Delcam Advanced Manufacturing Solutions

FeatureCAM 2012

Training Course

FeatureCAM 2012

Training Course

FeatureMILL 3D Finishing

FeatureCAM

Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

FeatureCAM Version: 2012 Date: 03 August 2011 11:10

Contents

FeatureCAM overview

FeatureCAM is a CAD/CAM software suite that automates machining and minimizes programming times for parts on mills, lathes, and wire EDM. Unlike operations-based CAM systems, FeatureCAM generates toolpaths based on the features of the part, and automatically selects appropriate tools, determines roughing and finishing passes, and calculates feeds and speeds. The selections made can be based on the built-in machining knowledge that Delcam supplies 'out-of-the-box' with FeatureCAM, or from experience captured from your company, project or individual users' preferences.

FeatureCAM includes five stand-alone modules:

- **2.5D Milling** - 2.5D design and toolpath generation for 2- and 3-axis mills.
- **3D Milling** - 3D surface modeling and 3-axis toolpath generation.
- **3D Lite** - a limited version of **3D milling**.

  3D Lite *lets you mill only one surface per feature, but you can create multiple features. The strategies available in* ***3D Lite*** *are Z-level rough, Parallel rough, Parallel finish, Isoline, and 2D spiral.*

- **Turning** - 2-axis design and toolpath generation for 2-axis lathes.
- **Turn/Mill** - Supports lathes with C and Y-axis milling capabilities.
- **Wire EDM** - 2- and 4-axis wire EDM toolpath creation.

The following add-on modules are also available:

- **RECOGNITION** - 3D surface and solid import and the recognition of 2.5D features from solid models. Accelerates making 2.5D and turned parts from solid models.
- **Tombstone** - Multiple part manufacturing for horizontal or vertical milling machines with indexers.

- **Solid Modeling** - Solid modeling and tools for creating molds from solid models.
- **5-Axis Positioning** - Manufacture 2.5D features from 5-axis orientations.
- **Native Import Modules** - Native data can be read directly from SolidWorks, SolidWorks Assemblies, Autodesk Inventor, SolidEdge, Catia, NX, Pro-Engineer, and Step files.
- **Machine Simulation** - Modeling and simulation of a CNC machine.
- **Advanced Turn/Mill (MTT)** - Includes support for Turn/Mill in addition to support for B-axis (5-axis positioning) and multiple turret synchronization.
- **Network Database and Licensing** - Flexible product licensing allows sharing FeatureCAM licenses across a network.
- **5-Axis Simultaneous** - Manufacture 3D features while changing the tool axis.

Why creating toolpaths is so fast

FeatureCAM has the unique ability to generate toolpaths and create NC code to run the machines with a minimum amount of user input.

Traditional CAM systems are *operations-based* and require you to program every operation, one at a time, to create your part. FeatureCAM is *feature-based*; this means the part is created using features that describe that part, from simple holes, to complex pockets, to turned grooves. Machinable features contain information and rules describing how and where material removal should occur, cutting depths, whether to use climb cutting, whether to spot drill or center drill, and preferred machining strategies for roughing and finishing. This means that after you import or draw the part and identify its features, FeatureCAM automatically:

- Selects the most appropriate tools and operations;
- Recommends machining strategies;
- Calculates speeds and feeds;
- Generates toolpaths and creates the NC code.

You can customize this built-in 'intelligence' to your own style of cutting.

Finishing Basics

Introduction

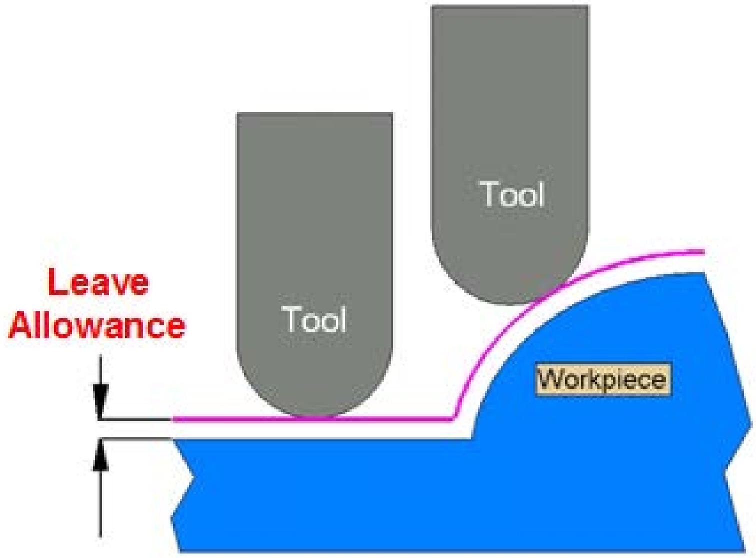
FeatureCAM has wide variety of methods for finish machining parts. The strategy that is chosen will depend upon the part geometry, the type of stock and the available cutting tools. By choosing the most appropriate combination of strategy and tooling the user can greatly reduce the time taken to finish machine a part. The strategy chosen can also have a large influence on the surface finish achieved.

Leave Allowance, Tolerance and Stepover

When FeatureCAM is finishing a part, the main considerations are accuracy, surface finish, and speed of calculation. These are generally governed by the choice of tool, strategy, tolerance, and stepover. The tool and strategy choices largely depend upon the shape of the part that is being machined and will be covered in later modules. For now we will consider the leave allowance, tolerance, and stepover.

Leave Allowance

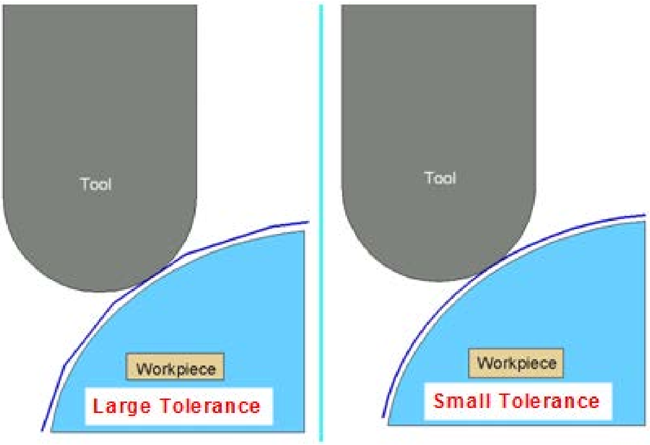
The Leave Allowance is the amount of material left on the part by the finish machining process. By default this is zero, i.e. the part will be finished to size. If the user wishes a positive leave allowance can be given, this will leave additional material on the part for subsequent processes - for example polishing or EDM (spark erosion). The leave allowance can also be negative; this will cut the part undersize. A negative leave allowance is usually used when machining electrodes for EDM, in such a case the leave allowance will be set to the required spark gap.



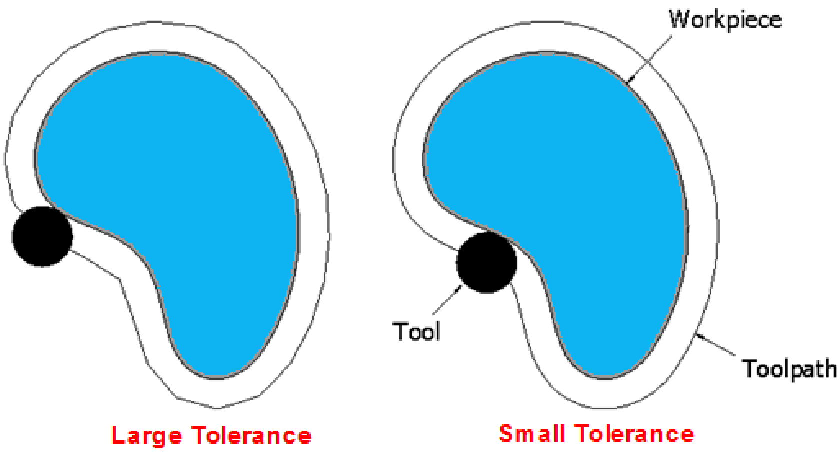
Tolerance

The toolpaths produced by FeatureCAM consist of a number of straight line moves which move the tool across the surfaces to be machined. The length of the moves produced depends upon the tolerance value and the curvature of the surface being cut. The **Tolerance** is the maximum deviation of the tool from the surfaces being cut, for example a tolerance of 0.005" means that the tool will not deviate from the surface by more than 0.005" along the direction in which it is cutting. On a convex surface this will be the maximum amount of the permissible cut into the surface, on a concave surface it will be the maximum amount of material that will be left on the surfaces being cut.

The **Tolerance** controls the accuracy to which the toolpath follows the shape of the work-piece. With a coarse tolerance the toolpath is more faceted, as fewer moves are calculated for each profiling move.



The effect of tolerance on a toolpath in a vertical plane (Parallel toolpath).



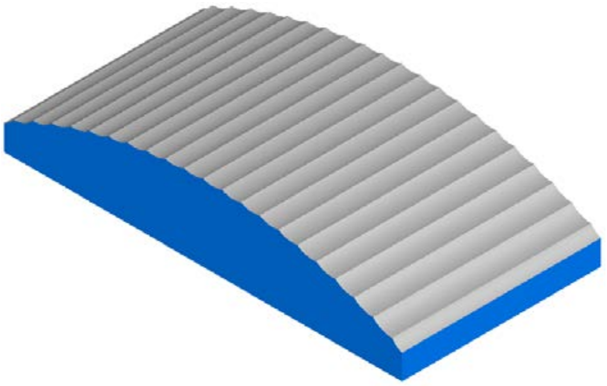
The effect of tolerance on a toolpath in a horizontal plane (Z Level toolpath).

Note If a very fine tolerance is chosen for a roughing operation, it may take a long time to calculate the toolpath to an unnecessary accuracy. The optimum Tolerance value will depend upon the user balancing the required accuracy against the time taken to calculate the toolpath.

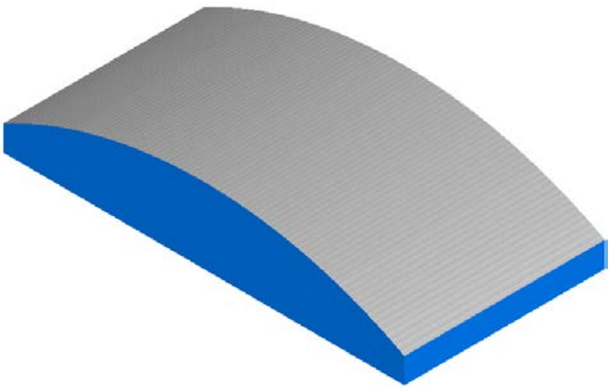
It should also be noted that tolerance required will also depend upon the curvature of the part being machined. Where a surface has a large radius of curvature a smaller tolerance will be required to prevent faceting of the part.

Stepover

The tolerance controls the accuracy with which the part is machined along the direction of travel of the tool. The distance by which the tool steps over between each tool pass will control the surface finish. In FeatureCAM, there are a number of ways in which the distance between passes can be controlled. It may be an explicit stepover either in the XY plane, along the surfaces, a vertical stepdown, or Z increment. Certain toolpath types allow the user to specify a scallop height. This is the height of the ridge in between the furrows left by the tool (equivalent to an Ra value).



Large Stepover



Small Stepover

It is common practice to calculate toolpaths with a large tolerance and stepover while experimenting with different strategies. Once the strategy "looks right", the user can then tighten up the tolerance and stepover for the final "manufacturing" calculation. This can save a great deal of time when working on large parts which will need to be finished to a high accuracy and surface finish.

Parallel Finish Machining

Overview

In this module, you will be introduced to the parallel finishing strategy. With this strategy a wide range of parts can be finish machined.

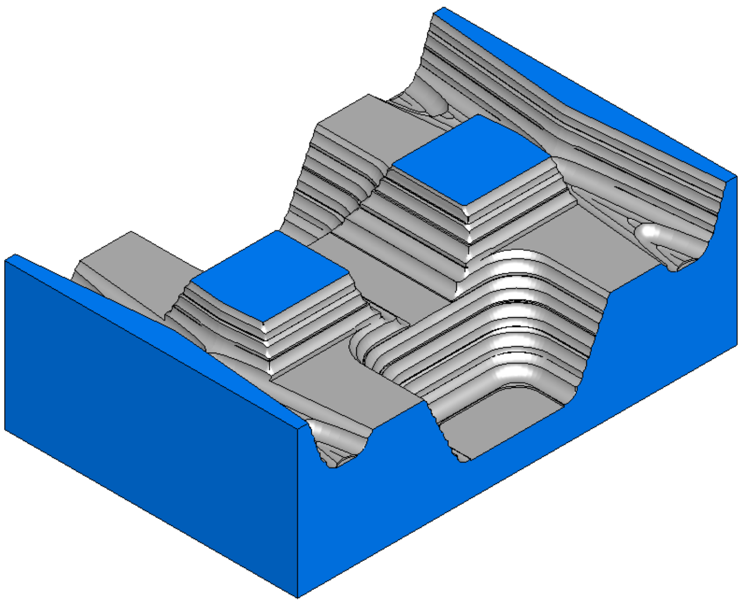
Parallel Finish Machining

In parallel finishing, the toolpath consists of a series of straight line passes across the part. The passes are equally spaced and parallel to each other at a specified angle to the X or Y axis. FeatureCAM varies the Z height of the tool to produce the finished part. Parallel finishing is also known as *Raster* finishing.

- Open the part:

C:\FeatureCAM\3-Axis\Finishing\Data\Die_Block.fm

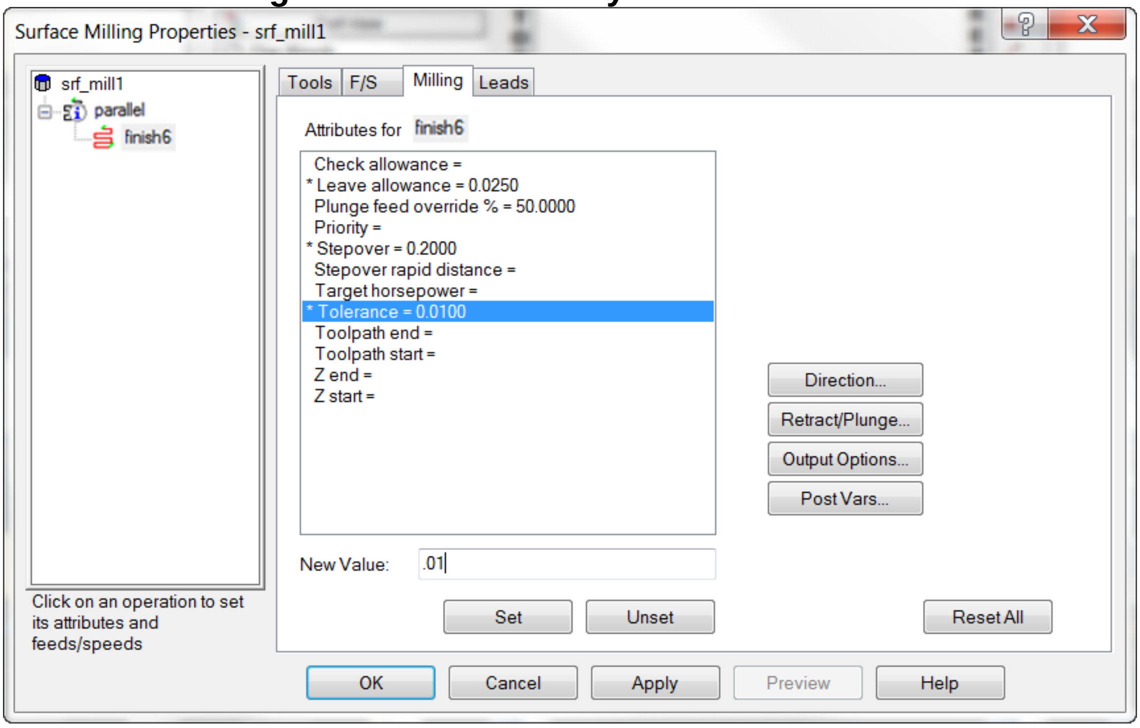
- Select the tool crib **Die_Block_Start.fm_Tools_from_last_save**
- Select an **Isometric view**
- Run a **3D simulation**



The part has been roughed out using a 1” endmill followed by a 0.5” ball nosed cutter. We will now generate an X Parallel finishing toolpath; this will remove the steps left by the roughing. This is known as a semi-finish operation.

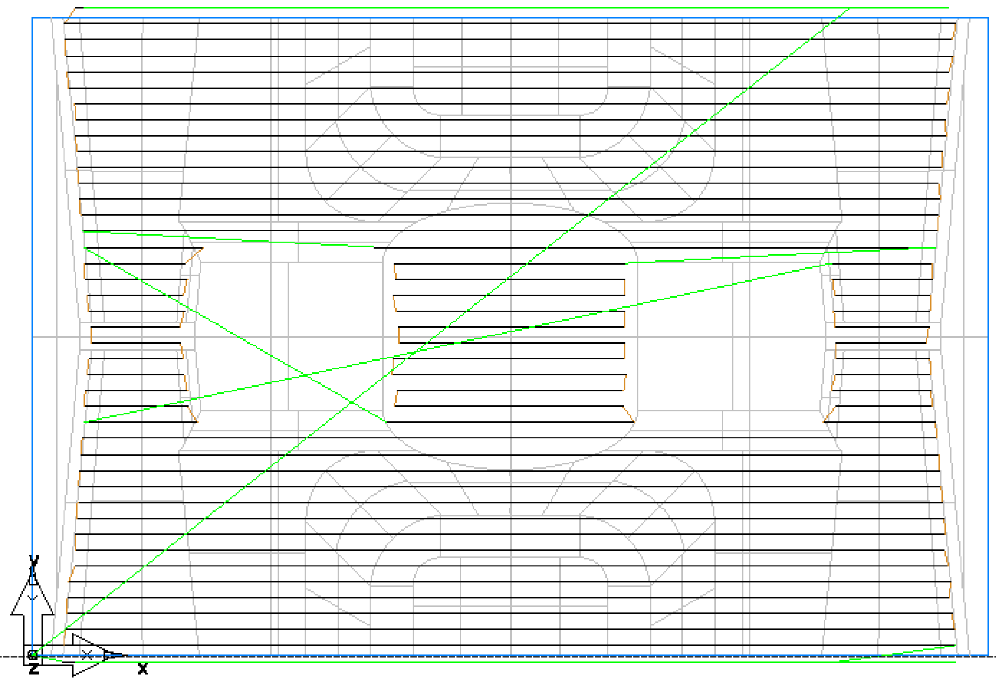
- **Eject** the simulation
- **Edit** the feature **srf_mill1**
- Select the **Process tab**
- **Uncheck** the **Z Level rough** operation

- Click on **Add new operation** 
- Under **Finishing Strategies** select **Parallel** and click **Finish**
- Edit the **finish operation** and select the **Tools** tab
- Select a **1” Ball nose** cutter, click **Apply**
- Select the **Milling tab** and fill in **exactly** as below



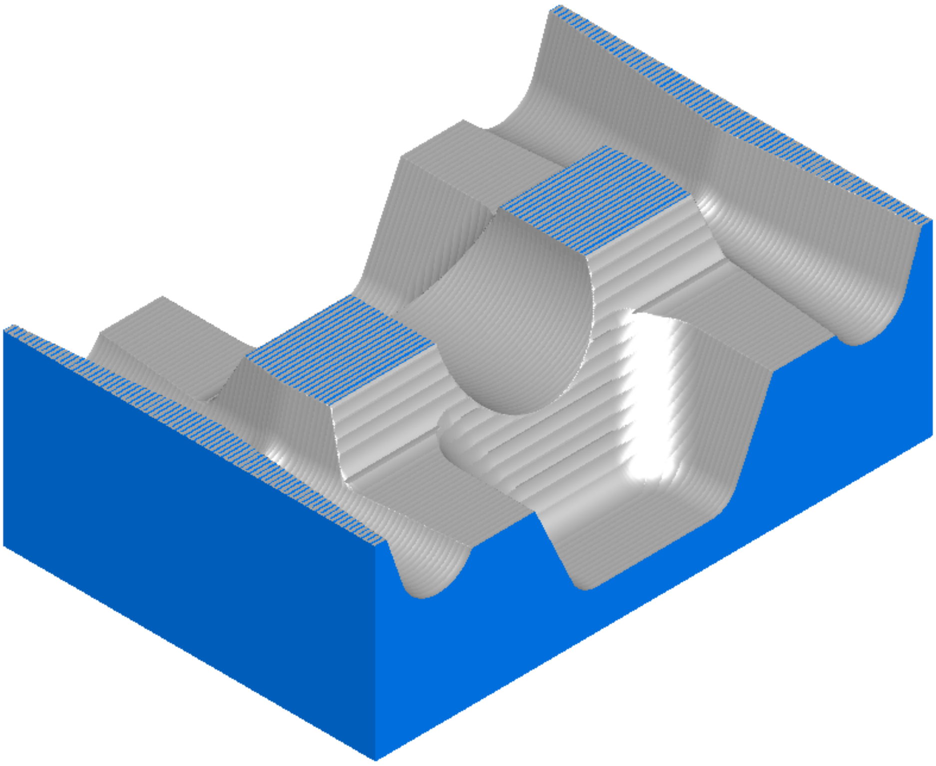
The **leave allowance** is telling FeatureCAM to leave 0.025” of material on the part for finishing. As this is a semi-finish operation, the surface finish is not important; this is why the stepover and tolerance have been increased to reduce the calculation time. As long as the tolerance is less than the finish allowance, the toolpath will not gouge the part.

- Select a **Top view** and run a **Centreline simulation**



The toolpath should appear as shown above. The cutter is making parallel passes along X. We will now add a finishing pass to the operation.

- Select the **Process tab**
- **Check** the **Z Level rough** operation
- Click on **Add new operation**
- Under **Finishing Strategies** select **Parallel** and click **Finish**
- Edit the new **Finish operation** and select the **Tools** tab
- Select a **1" Ball nose** cutter, click **Apply**
- Select an **Isometric view**
- Run a **3D RapidCut simulation** using Play to next operation

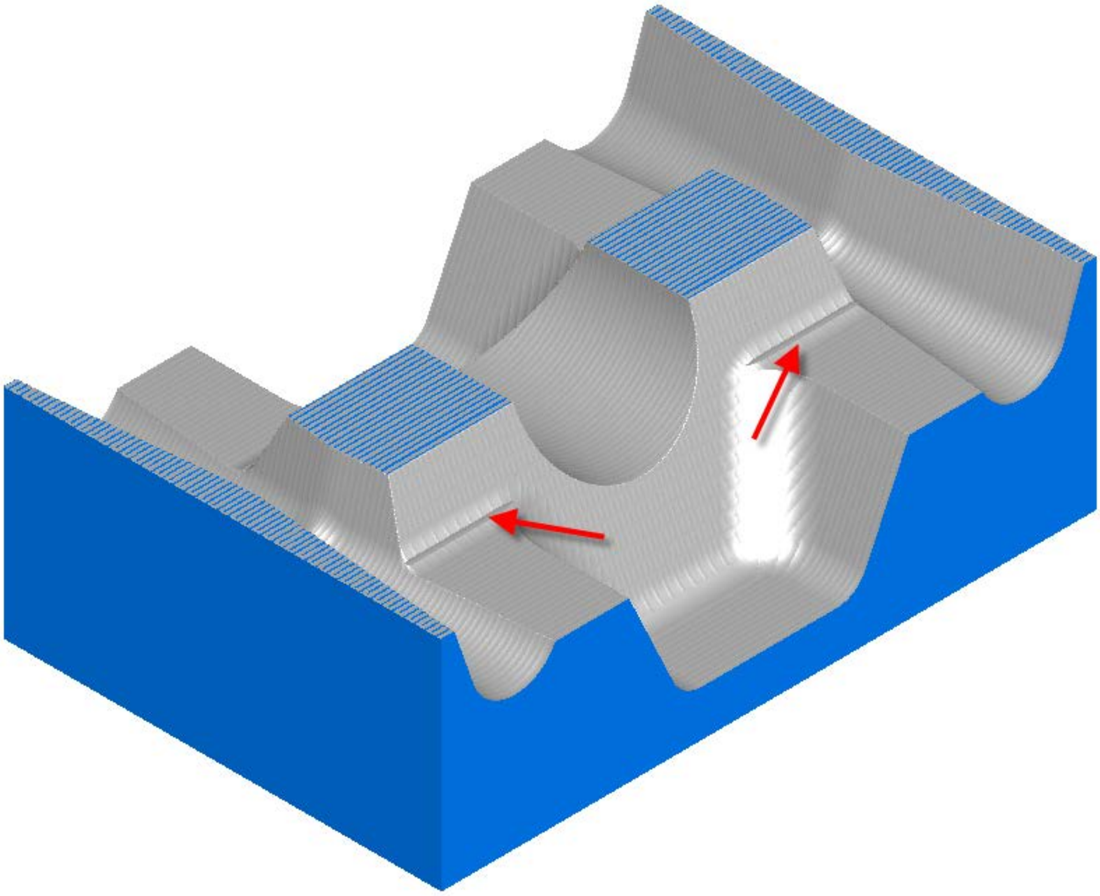


Note how the semi-finish operation removes the steps from the roughing. This provides a more constant tool loading for the subsequent finishing operations which will give a better surface finish.

You will see how the finishing pass in steep areas parallel to the cutting direction is giving a poor surface finish. We will now modify the finishing strategy to resolve this problem.

- In the Part view double click on **srf_mill 1**
- Click on the second **Parallel** in the tree view
- Check **Add perp. remach. pass**
- Click **Apply** and then **OK**
- Run a **3D RapidCut simulation** as before

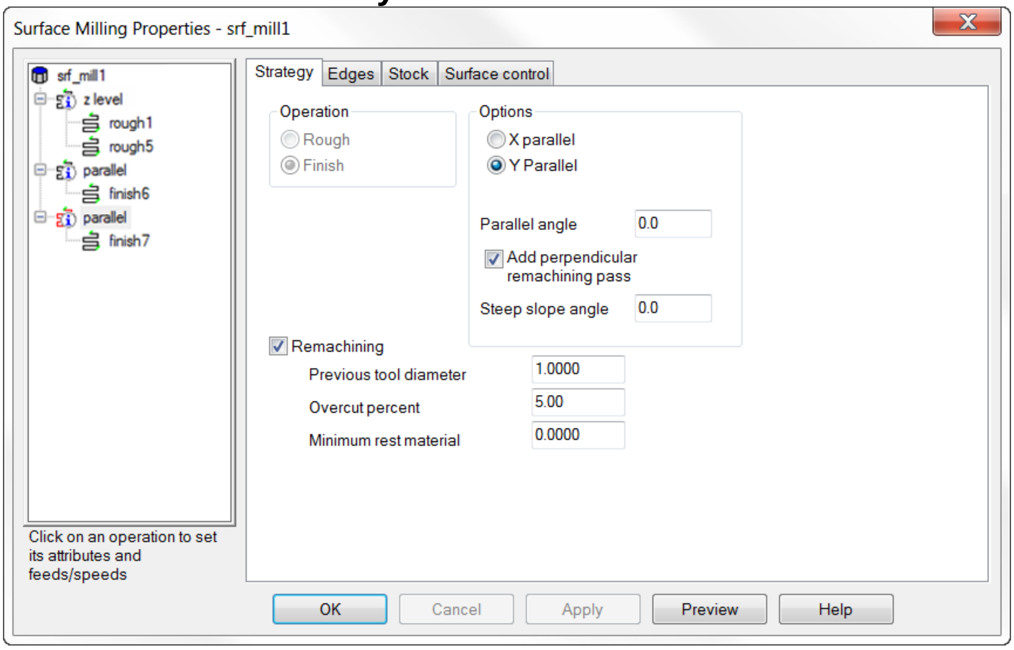
The area that was poorly machined before has now been re-machined with a perpendicular pass to give an improved surface finish.



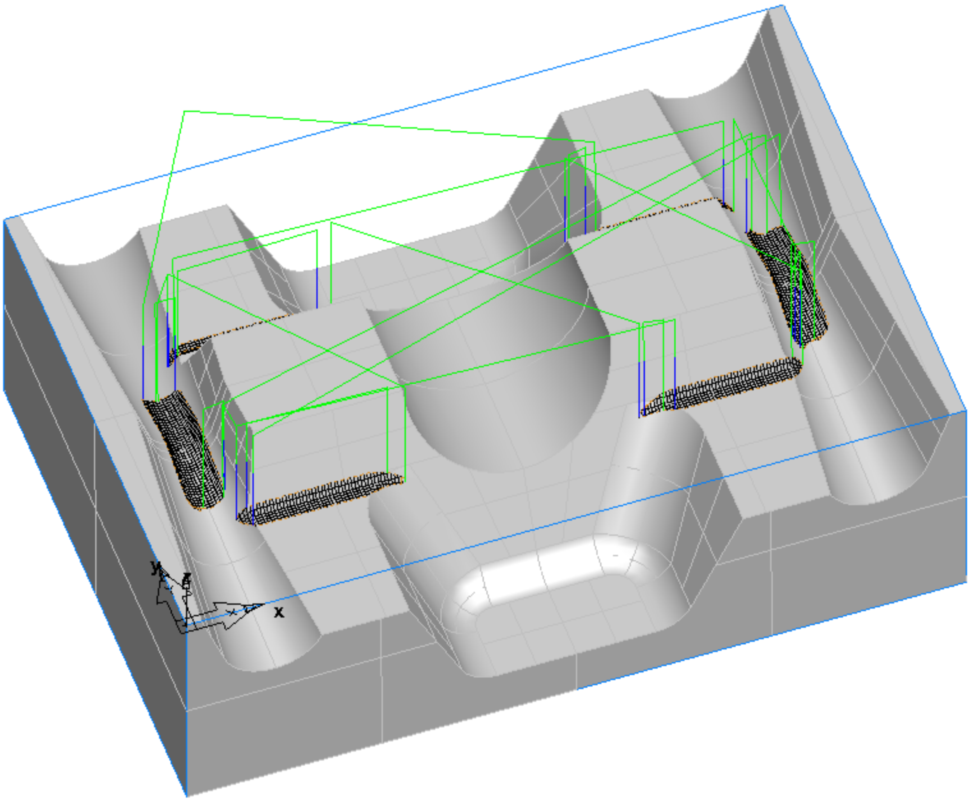
It can be seen however, that there are areas where the 1" ball nose cutter is not able to finish into the corners. We will now add another Parallel finishing pass with a smaller cutter which will re-machine only those areas where the larger ball could not reach.

- **Eject** the simulation
- **Edit** the feature **srf_mill1**
- Select the **Process tab**
- **Uncheck** the **Z Level rough** operation
- Click on **Add new operation** 
- Under **Finishing Strategies** select **Parallel** and click **Finish**
- Edit the **Finish operation** and select the **Tools tab**
- Select a **0.5" Ball nose** cutter, click **Apply**
- Click on the **Parallel operation**
- Select the **Strategy tab**

- Fill in the form **exactly** as shown below



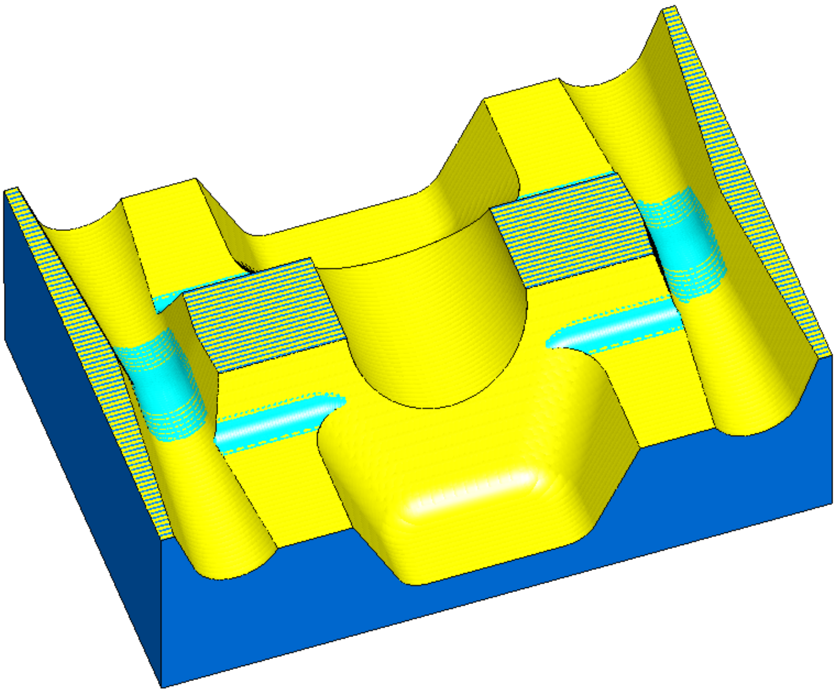
- Click on **Apply** and then **Preview**
- **Play** the **Centreline simulation**



The inside corner areas of the part have be **re-machined** with the smaller cutter, only the area where the 1” cutter could not reach are machined. The initial pass is along the Y direction with a second pass being along X. By setting the **Steep slope angle** to **zero,** we force FeatureCAM to machine the entire area in both directions.

We will now look at a 3D simulation with tool colors turned on so we can clearly see the results of this additional process.

- **Eject** the **simulation**
- On the **Options** menu select **Simulation**
- On the **General tab** check **Tool colors**
- Click on **Apply** and **OK**
- **Run** a **3D RapidCut simulation**



The use of re-machining allows us to finish machine the bulk of the part using a relatively large cutter and then just finish selected areas with a smaller cutter. This provides a tremendous time saving over the alternative of using the smaller cutter with a small step-over to machine the same part to the same surface finish.

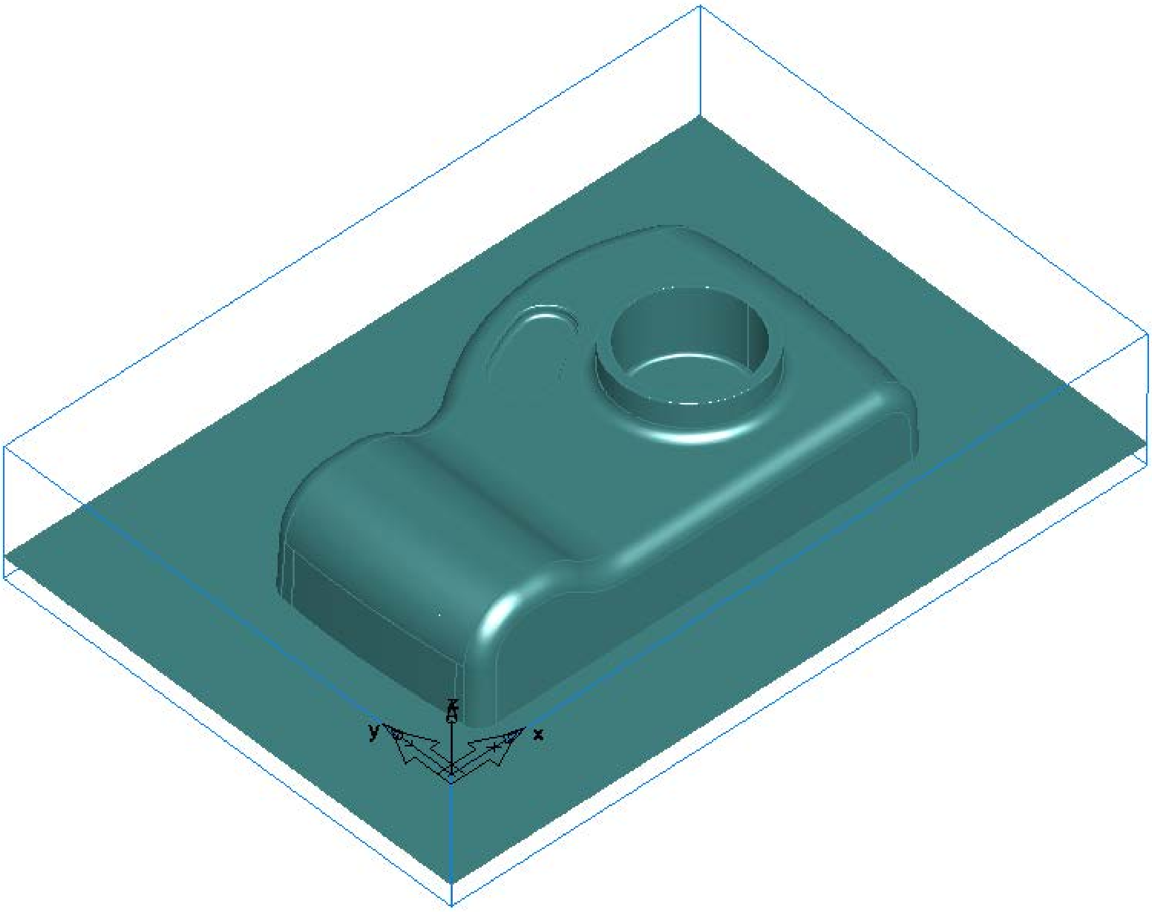
- **Eject** the **simulation**
  On the **File menu** select **Close**, **DO NOT save** the file

3D Spiral & Z Level Finishing

Introduction

In this section, **Z Level** and **3D Spiral** finishing strategies will be applied to a model that consists of a combination of flat and steep areas, plus a pocket with vertical walls. In this exercise, applying either toolpath to the complete model without using would not be recommended. In general, **Z Level** finishing is more suited to parts with steeply inclined walls and **3D Spiral** is better at gently curved shallow areas. By using **Boundaries,** we will limit the Z Level toolpath to the steep areas leaving the remaining shallow areas to be machined using the **3D Spiral** strategy.

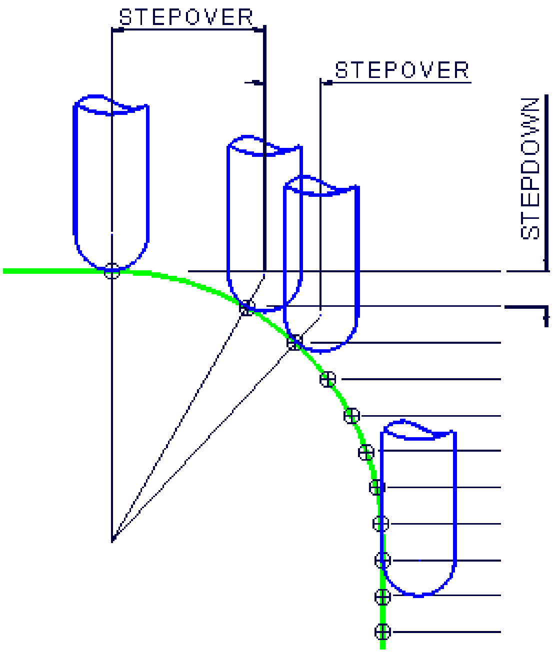
- **Open the** file **camera.fm**
- Select an **isometric view**



We will first machine the steep areas of the part using a Z Level finishing toolpath. Generally, it is better to finish steep areas first and then go back to do the shallower areas. Otherwise, you risk making a full depth cut using the side of the tool when you approach a very steep or vertical wall.

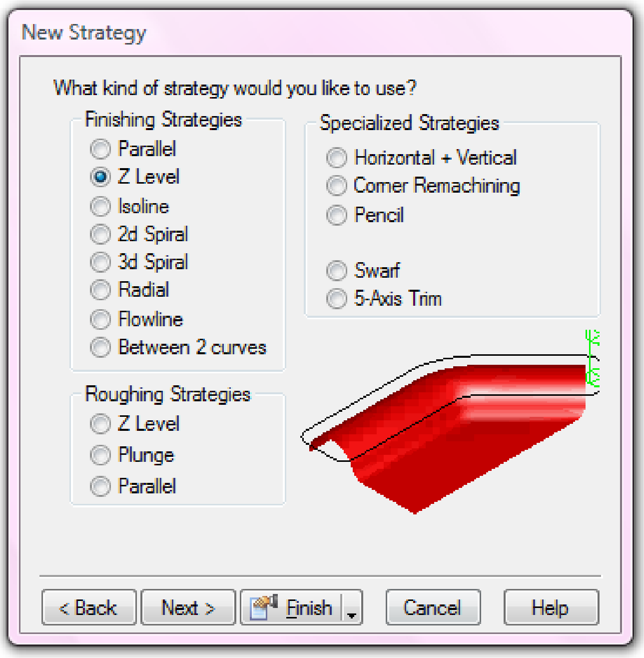
Z level machining

Z Level machining projects each tool track horizontally onto the component at fixed heights defined by the **Stepdown;** this also known as *waterline* machining.



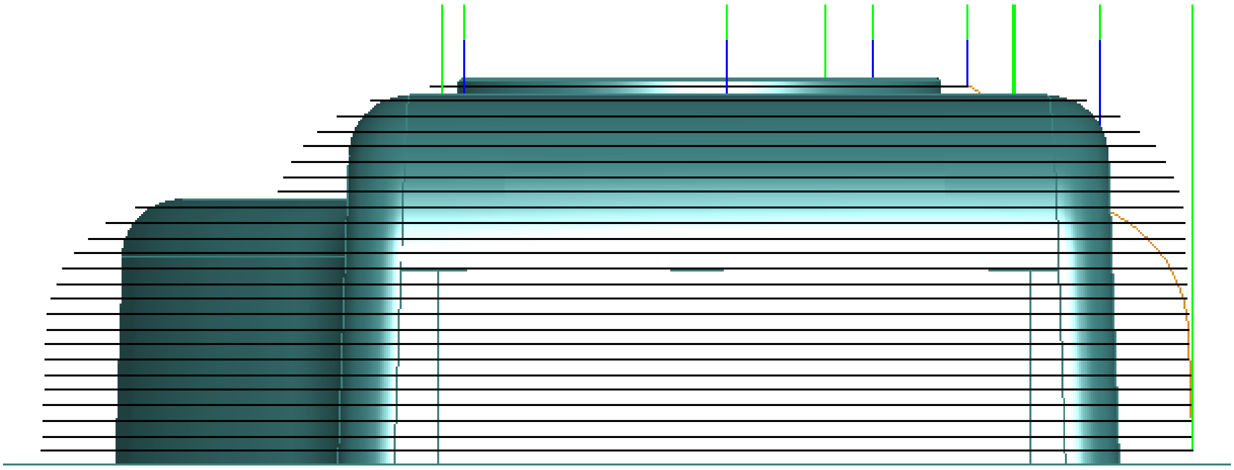
In Z level machining, a constant stepdown is used in the Z direction between each pass. As the component surface becomes shallow, the actual tool step over increases until it becomes non-existent on flat areas. For this reason it is a good idea to limit the toolpath to a slope boundary so that only the areas with a good surface finish are retained. By using a combination of 3D Spiral and Z level machining together with appropriate slope boundaries, the whole part can be machined to a good surface finish.

- **Select all** of the **surfaces**
- Create a new **Surface milling** feature
- Click **Next** until you reach the **New strategy page**
- **Choose a single operation** and click **Next**
- Select a **Z Level** finishing strategy

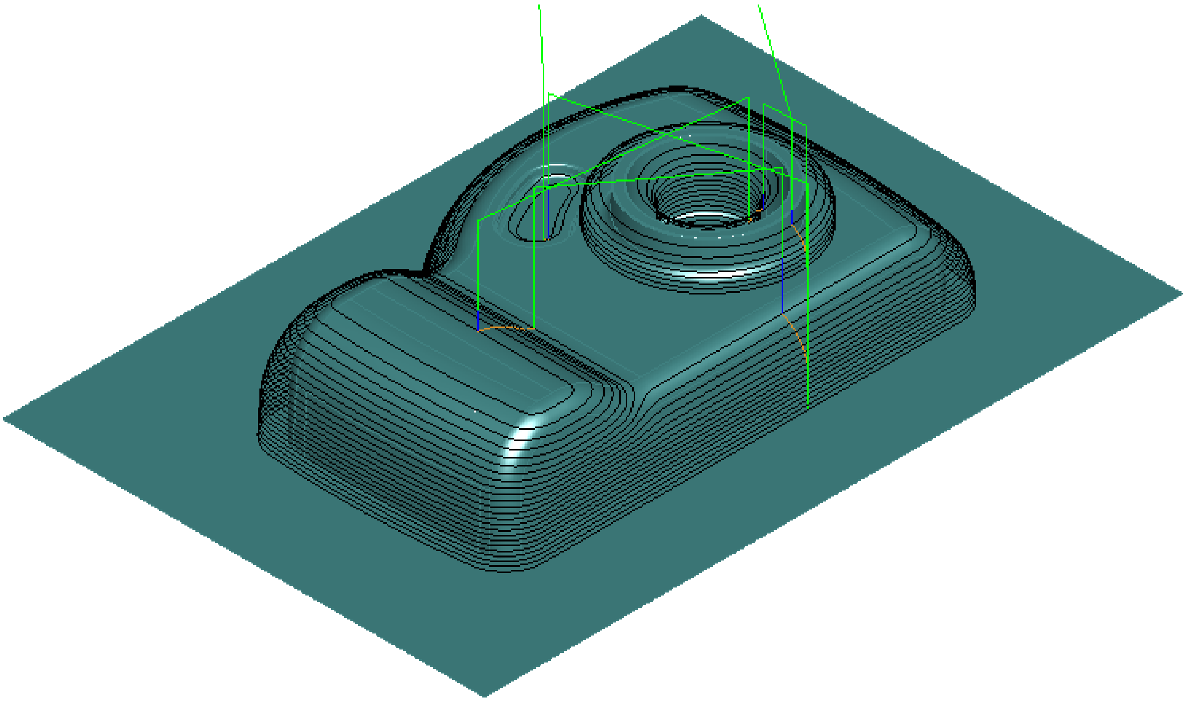


- Click **Finish** and then **OK**

- **Uncheck** the **roughing** operation in the Part view
- Run a **Centerline simulation**
- Take a **Left view**



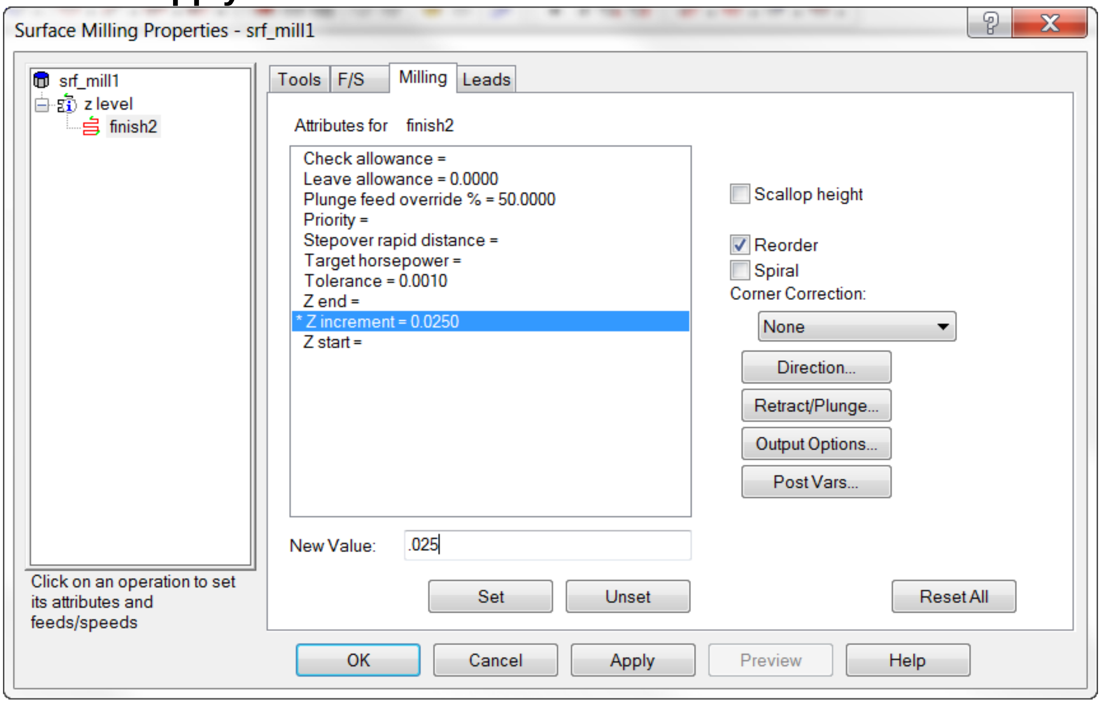
The part is finished by a series of horizontal passes around the part which are spaced equally in the Z direction. As you can see in the image below this gives a varying surface finish on the part which is dependent upon how shallow the surface is.



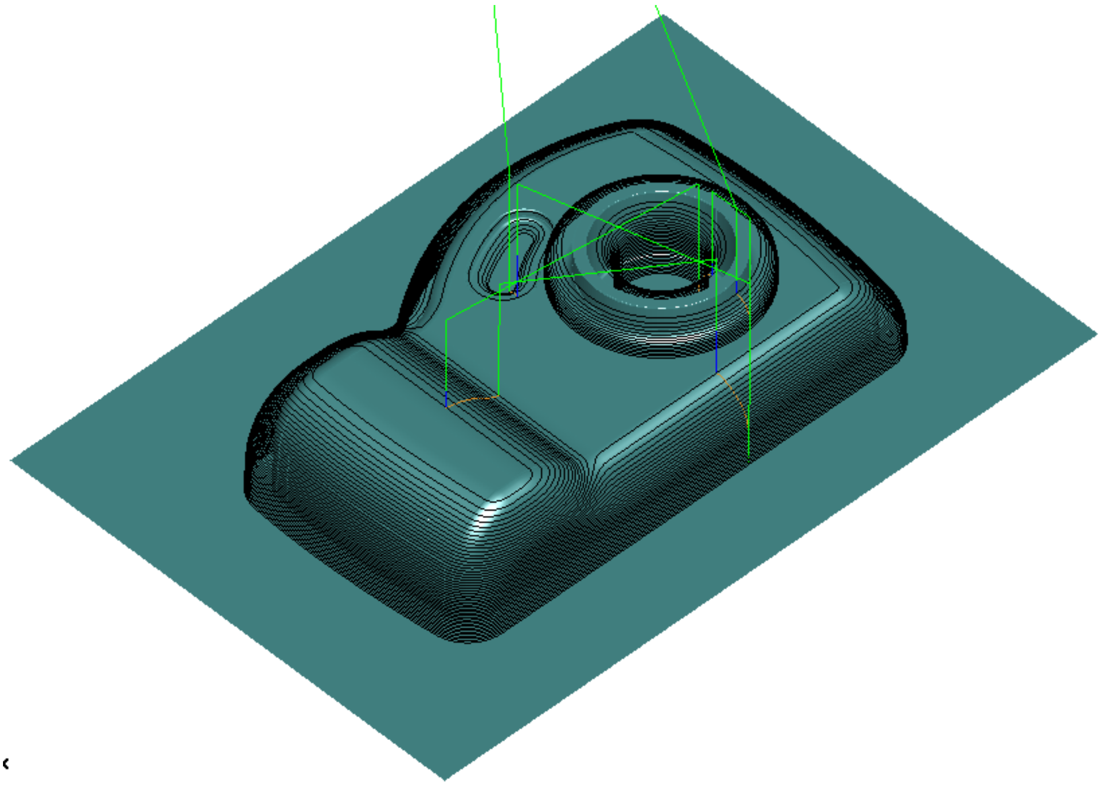
There are a number of ways in which we can remedy this problem. The first is simply to reduce the distance that the tool steps down between passes.

- **Eject** the simulation
- **Edit** the feature **srf_mill1**

- Select the **Milling tab** and set the **Z Increment** to **0.025"**
- Click **Apply** and then **OK**



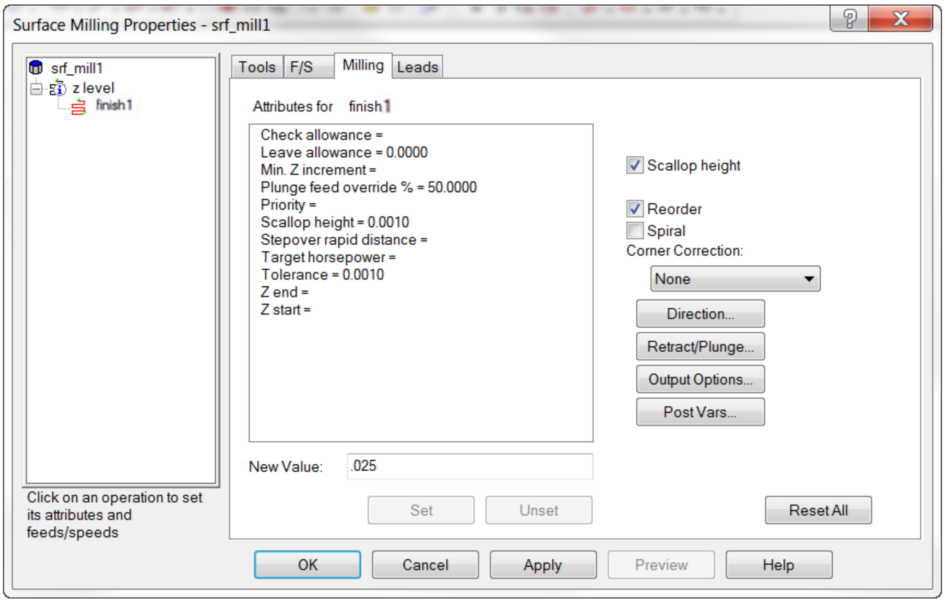
- Run a **Centerline simulation**



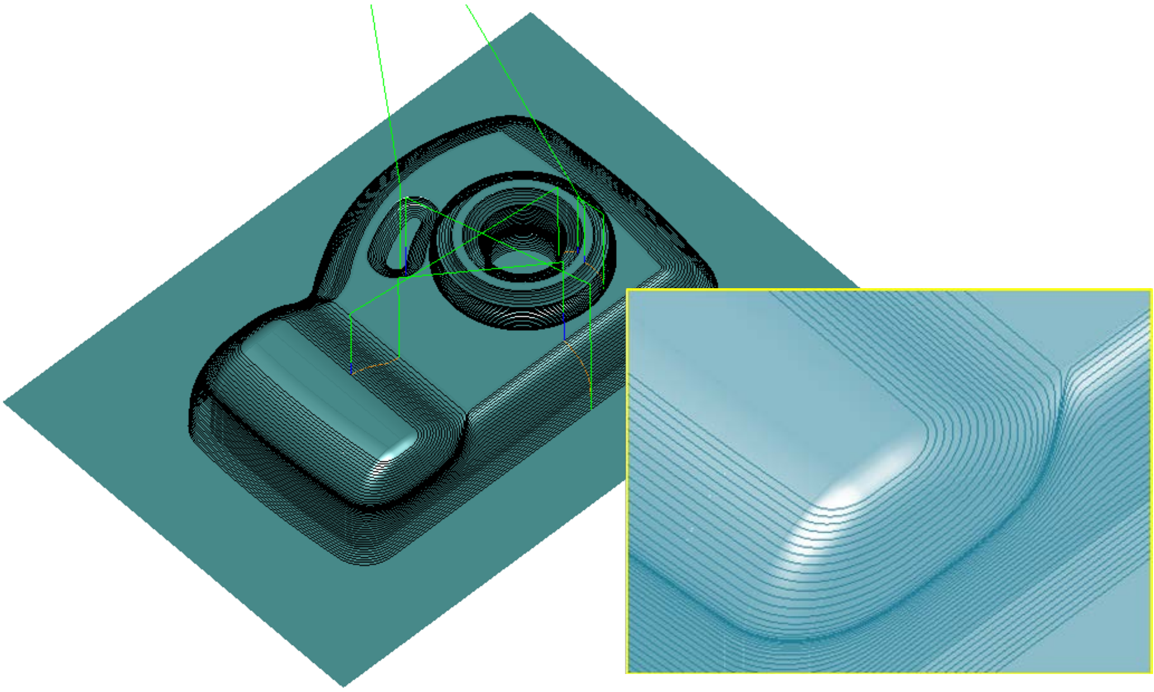
Note the improved surface finish in the shallow areas. However we are now making unnecessary extra passes on the steeper areas which will waste time.

Another alternative is to use a variable stepdown, which is calculated based upon the desired surface finish. This is specified as a **Scallop height**. The scallop height is the maximum permitted height of the ridges or scallops between adjacent passes.

- **Eject** the simulation
- **Edit** the feature **srf_mill1**
- Select the **Milling tab** and **check Scallop height**

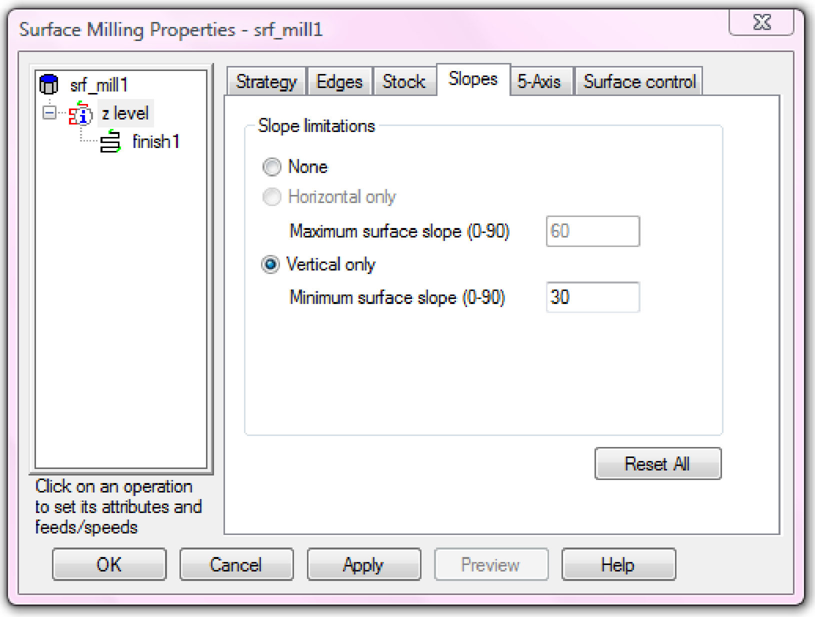


- Click **Apply** and then **OK**
- Run a **Centerline simulation**

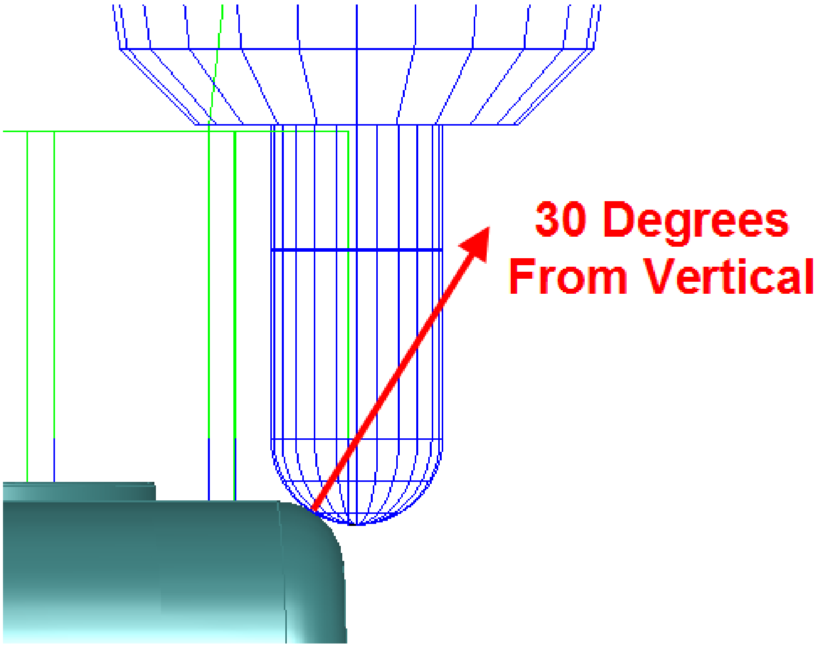


Note how the stepdown now varies to maintain the maximum scallop height. It should also be noted that where a part goes from a shallow to a steep area in the same height region the passes bunch together. We will now look at another, better way of controlling surface finish on surfaces with a wide variation in steepness.

- **Eject** the simulation
- **Edit** the feature **srf_mill1**
- Select the **Milling tab** and **uncheck Scallop height**
- Click on **Z Level** and then select the **Slopes tab**
- Check **Vertical only**

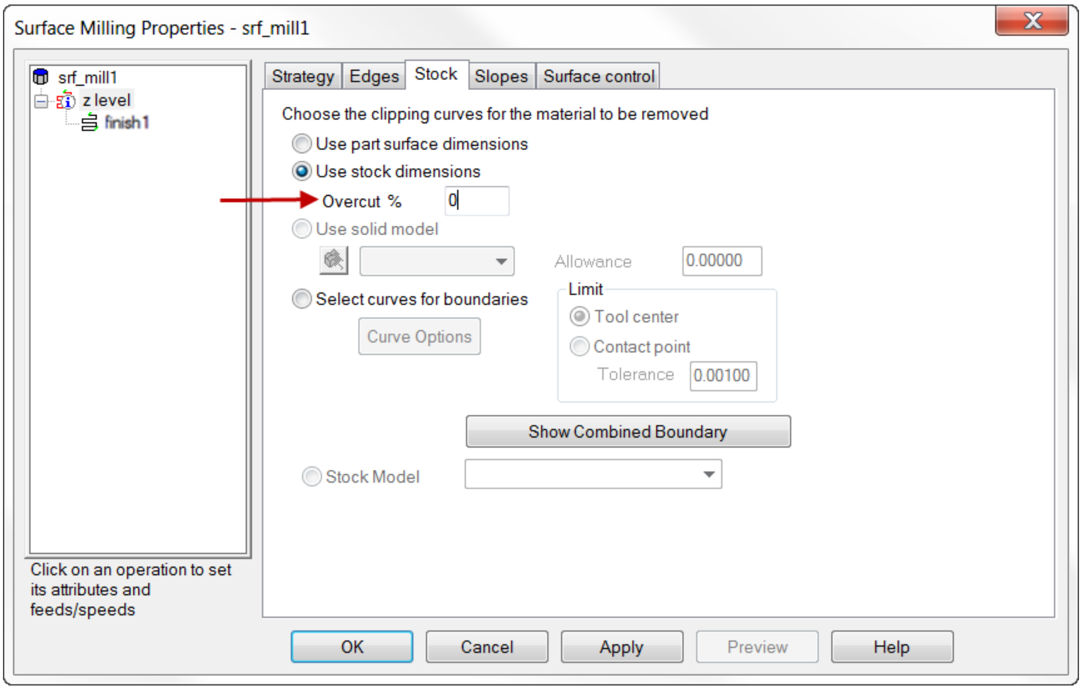


We have now told FeatureCAM that we do not want to machine the shallow areas of the part. We can vary the area to be machined by changing the Minimum surface slope angle. By default this is set to 30 degrees indicating that any area of the part whose surface normal at the point of contact of the tool is inclined at less than 30 degrees to the vertical will not be machined.

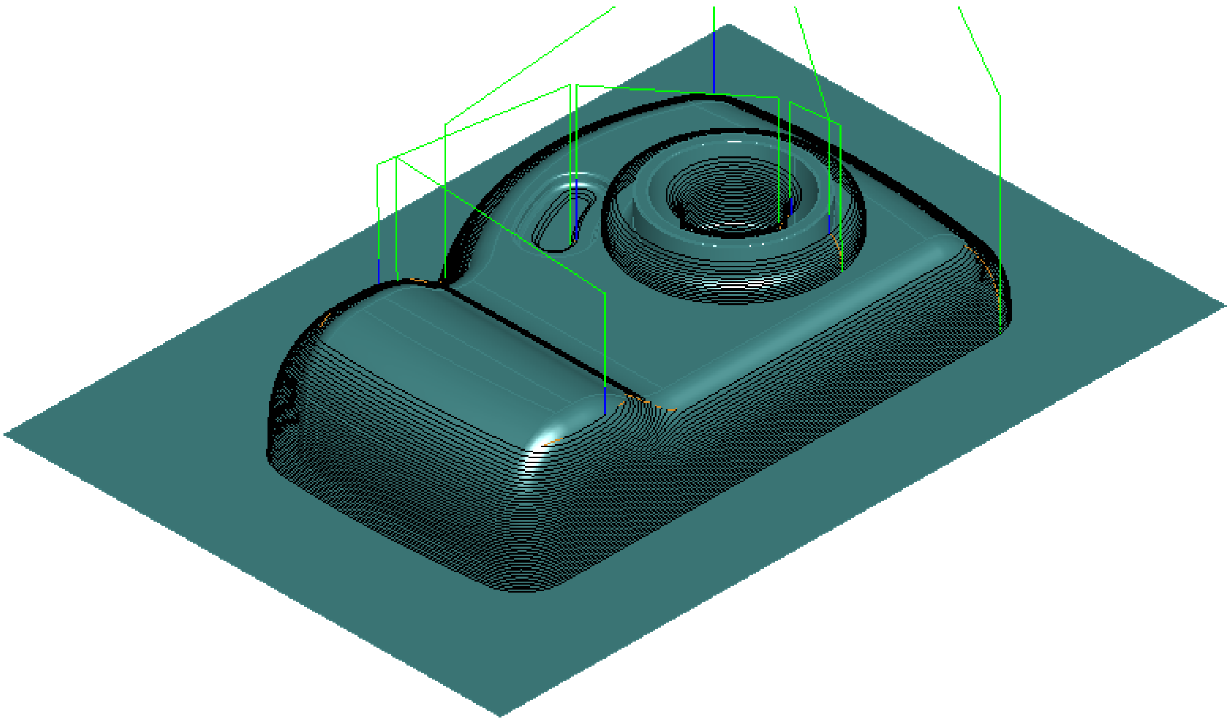


Setting a Steep angle will also have the additional effect of causing FeatureCAM to see the edges of the part as steep areas. To avoid the tool dipping down the side of the block we also need to tell FeatureCAM not to take the tool outside of the stock.

- Select the Stock tab and check Use stock dimensions
- Set the Overcut % to Zero



- Click **Apply** and then **OK**
- Run a **Centerline simulation**

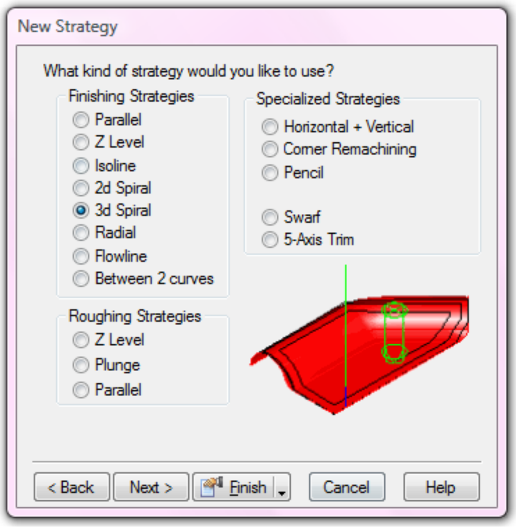


The shallow areas are no longer being machined. This is eliminating those parts of the model where a poor surface finish would have resulted and also reduced the length of time taken for the machining process. We can now select a different method to finish those areas to give a good finish over the whole part.

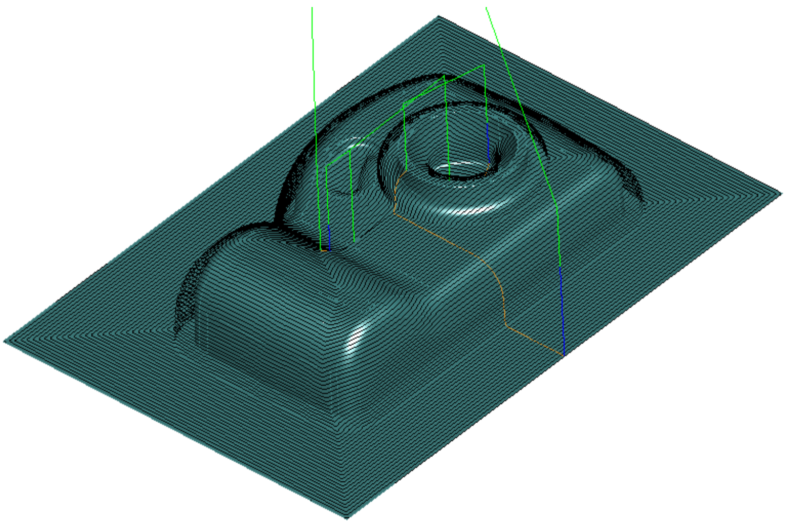
3D Spiral machining

In 3D Spiral machining FeatureCAM initially calculates a boundary around the edges area to be machined. The toolpaths are then generated by offsetting the tool contact point on the boundary by a fixed **Stepover** across the surfaces to be machined. FeatureCAM defines the tool Stepover relative to the 3D surface shape providing consistency over both flat areas and steep sidewalls. The stepover is calculated in such a way that each successive pass of the tool is equi-spaced from the previous pass.

- **Uncheck** the operation **srf_mill1** in the part view
- **Select all** of the **surfaces**
- Create a new **Surface milling** feature
- Click **Next** until you reach the **New strategy page**
- **Choose a single operation**
- Select a **3D Spiral** finishing strategy



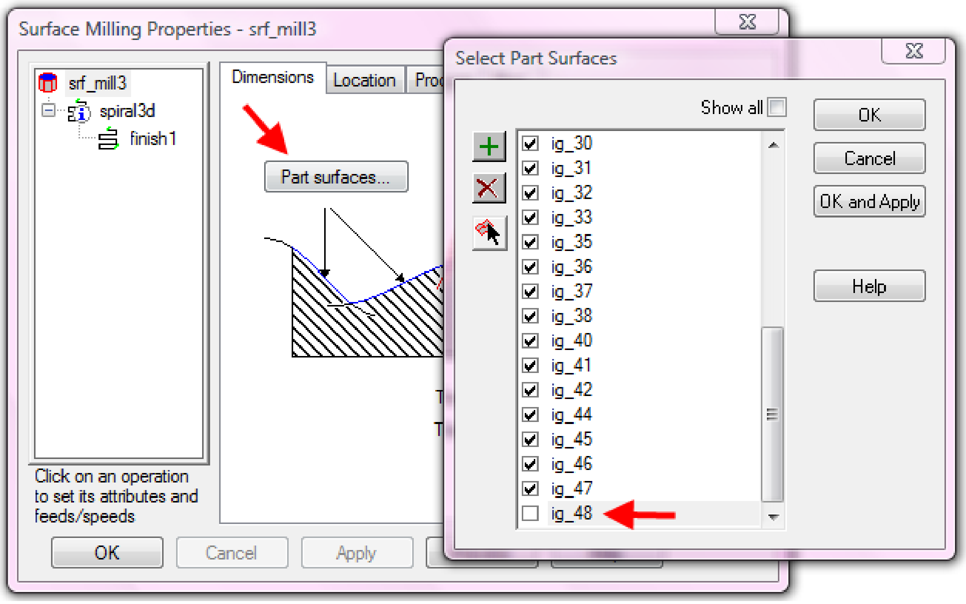
- Click **Finish** and then **OK**
- Run a **Centerline simulation**



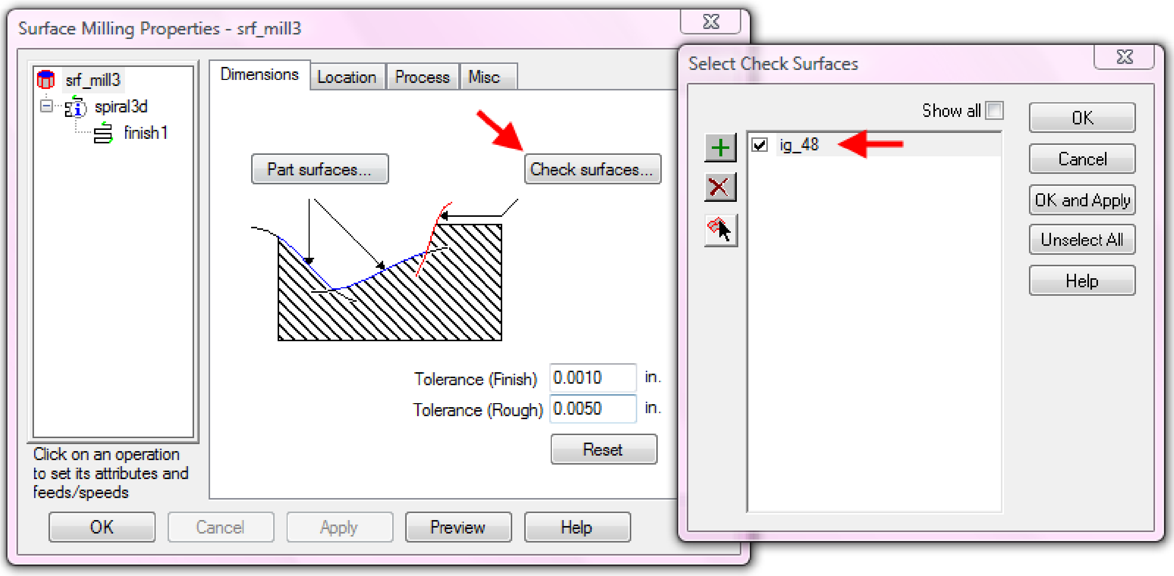
The first pass of the toolpath is following the rectangular outline of the flat surface. This pass is then offset inwards to produce the finished toolpath.

The default tool is a ½" ball nosed cutter, this toolpath would be an inefficient way of finishing the flat area of the part. We will now modify the toolpath so that the outline of the first pass is not taken from the rectangular surface.

- Eject the **simulation**
- Edit the feature and click **Part surfaces**
- Deselect the surface **ig_48**



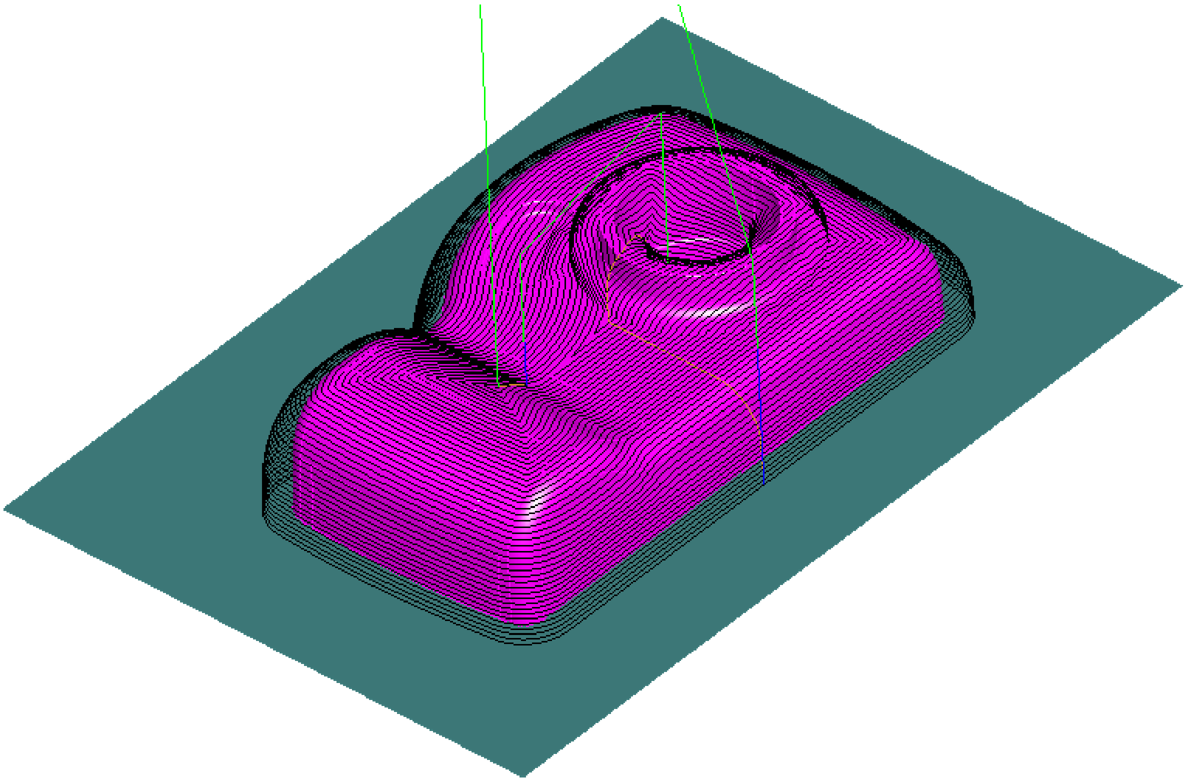
- Click **OK and Apply**
- Deselect the surface **ig_48**
- Click on Check surfaces
- Select the surface **ig_48**



- Click **OK and Apply** then **OK**

We have selected this lower surface as a check surface to prevent the tool from gouging it. Otherwise the toolpath would go past the bottom of the edges of the steep surfaces by the radius of the cutter.

- Run a **Centerline simulation**

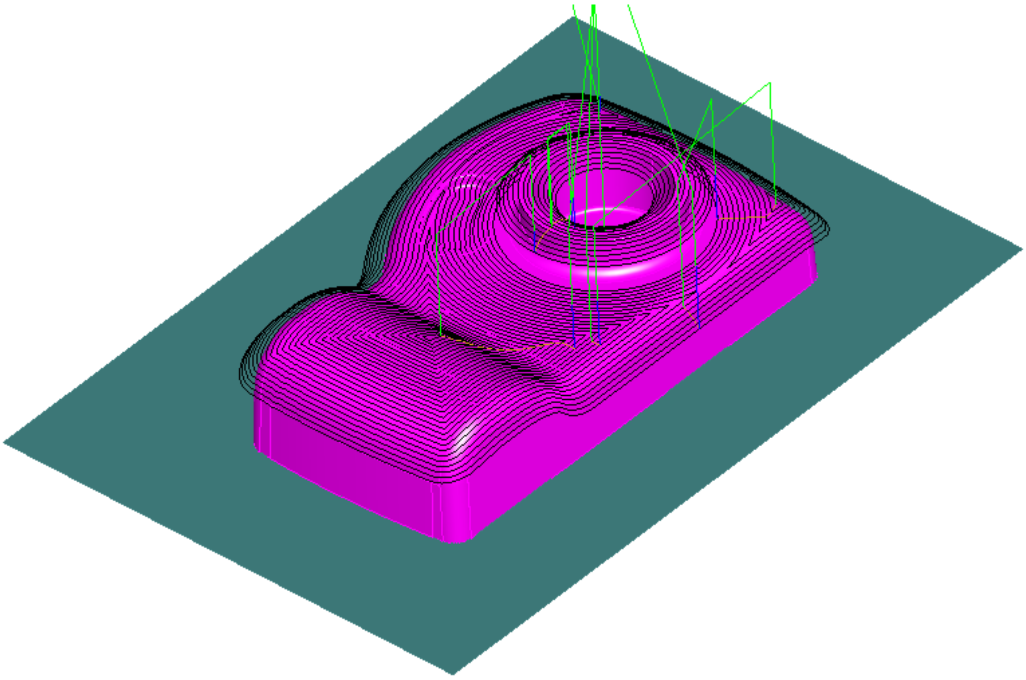


The toolpath is now starting at the outline of the selected surfaces and not wasting time cutting the flat area. This is still not a good toolpath, as it is starting at the bottom of the steep areas on the outside and cutting upwards. Notice also the way in which the toolpath rolls over the edge of the circular center pocket. We will now limit the toolpath so that it just cuts the shallow areas of the part. In this way we can use the previous Z Level toolpath for the steep areas.

- Eject the **simulation**
- Edit the feature and select the operation **spiral3d**
- Select the **Slopes tab**
- Check **Horizontal only**
- Set the **Maximum surface slope** angle to **30 degrees**
- Click **Apply** and then **OK**

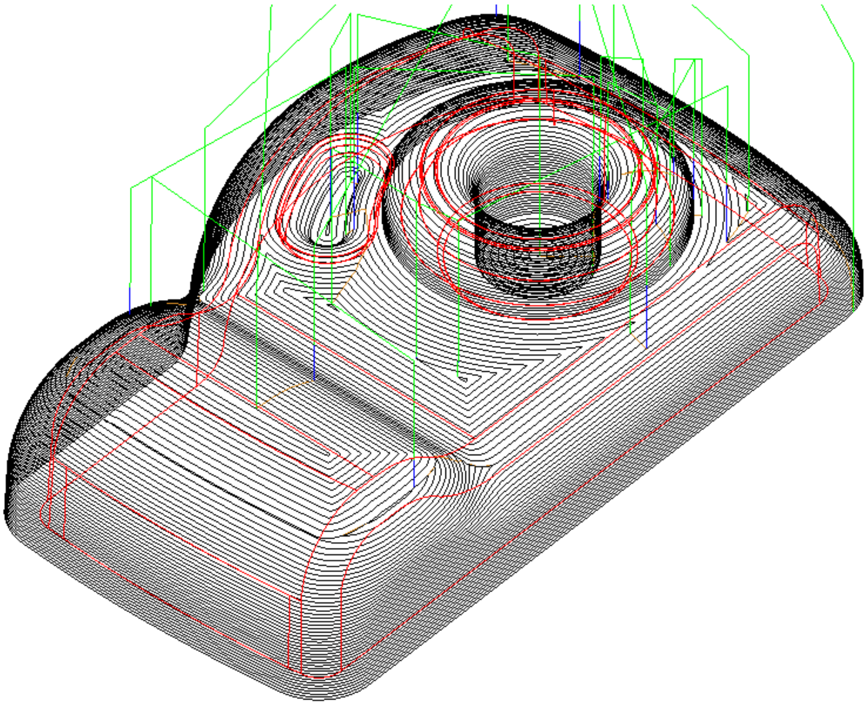
We have now told FeatureCAM that we only wish to machine those surfaces whose surface normal at the point of contact of the tool is inclined at less than 30 degrees to the vertical. This will coincide with the boundary we set earlier for the Z Level finishing toolpath.

- Run a **Centerline simulation**



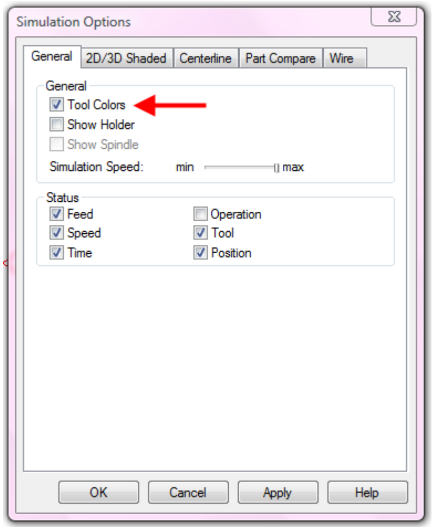
The toolpath is now confined to the shallow areas of the part.

- Eject the **simulation**
- Turn on the **Z Level finish** operation
- Run a **Centerline simulation**

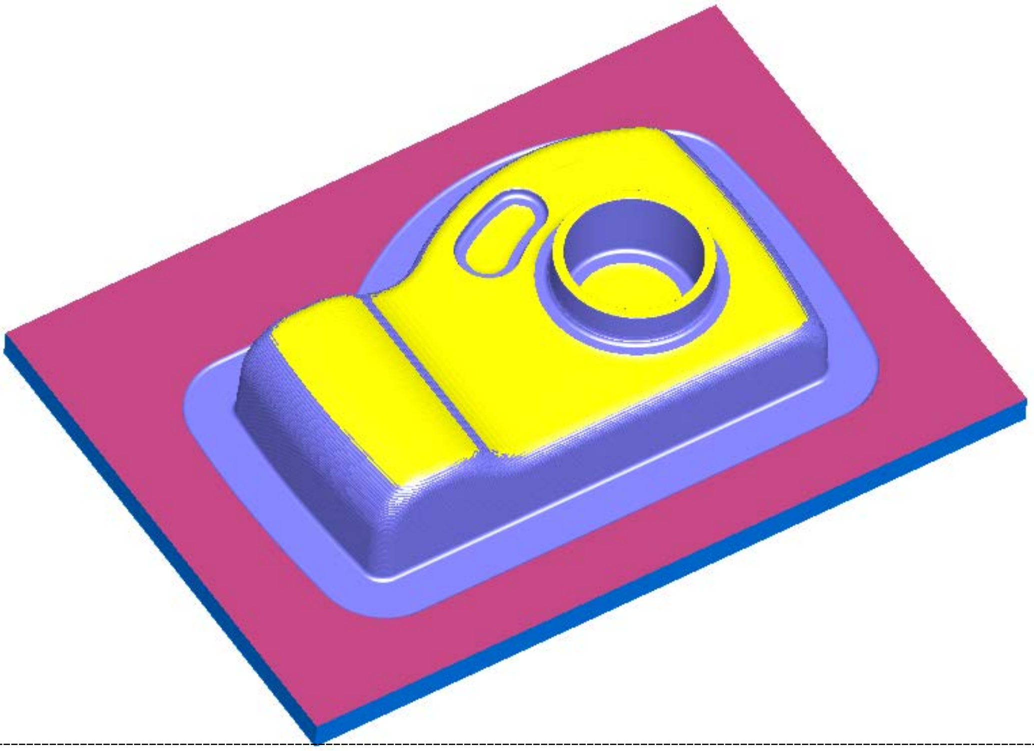


The two toolpaths now match at their edges finishing both the steep and shallow areas of the part. Due to the radius of the tool being used for the Z Level finishing toolpath it is not able to finish all of the way down the steep walls of the part. We will now change this.

- Eject the **simulation**
- Edit the **Z Level finish** operation
- Select the **Tools tab** and pick a **½" Flat end mill**
- Change the **End radius** to **0.05"**
- Click **Apply** and then **OK**
- Click **Apply** and then **OK** again
- From the **Options** menu select **Simulation**
- Check **Tool colors**



- Turn on all of the operations
- Run a **3D simulation**

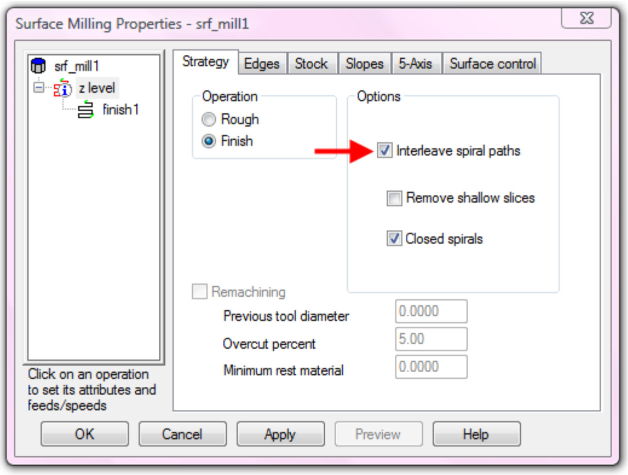


- As an exercise use a 2.5D feature to finish the lower, flat face of the part

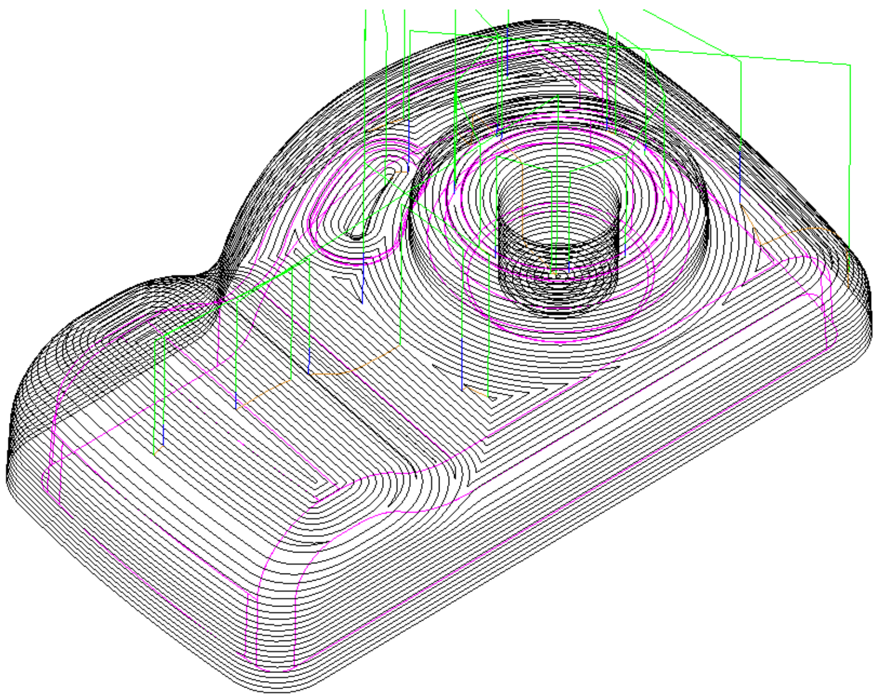
Interleaved Z Level machining

This strategy is a mixture of Z Level and 3D Spiral machining. Where the model is steep, Constant Z is used and for other areas, 3D offset is used. The method uses a default slope angle of 30 degrees to separate the steep and shallow areas. The two tool path passes are combined into one and reordered to reduce the number of times the tool lifts off the part.

- **Uncheck** the **3D Spiral** & **Roughing** operations
- **Edit** the **Z Level** operation **srf_mill1**
- Select **Z Level** and then the **Strategy tab**
- Check **Interleave spiral paths**



- Run a **Centerline simulation**

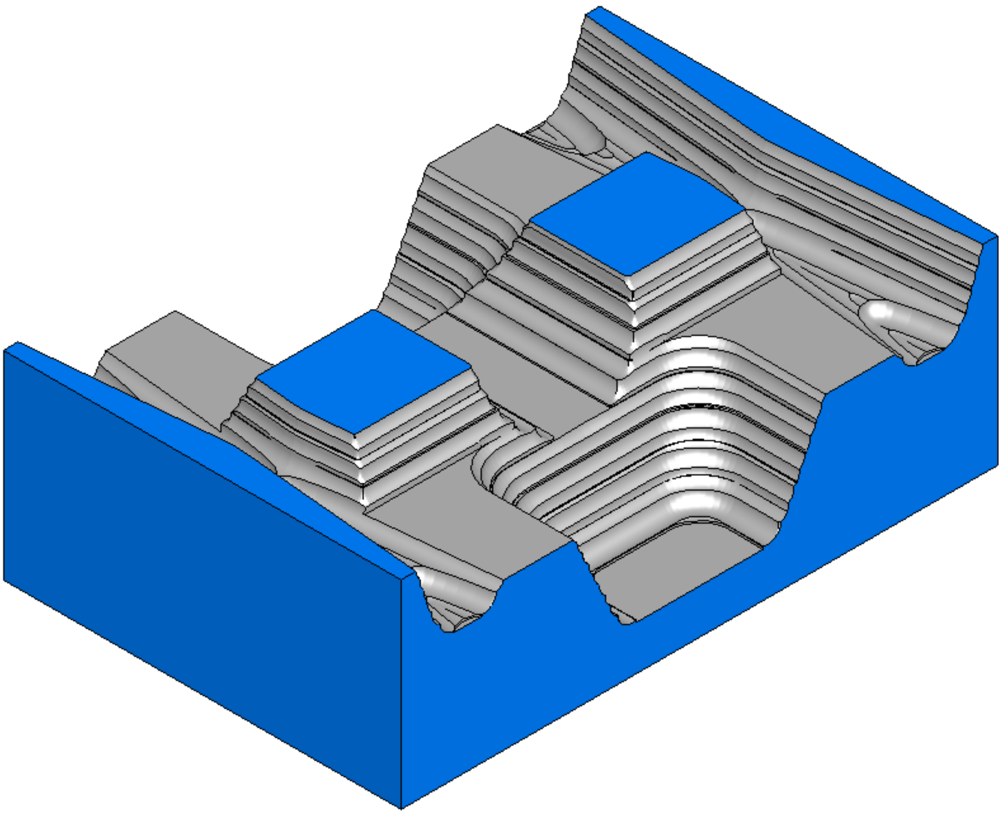


Note how the 3D spiral and Z Level parts of the toolpath are combined to reduce air moves. The toolpaths are reordered so that the part is cut from the top down reducing the risk of user error which can occur with manual boundary creation.

Horizontal and Vertical machining

This strategy is also mixture of Z Level and 3D Spiral machining. Where the model is steep, Constant Z is used and for other areas, 3D offset is used. The method uses a default slope angle of 60 degrees to separate the steep and shallow areas. Unlike Interleaved Z Level, the two tool path passes are NOT combined into one. The resulting toolpath calculates more quickly than an Interleaved Z Level as FeatureCAM does not have to go through the combining and reordering processes. The finished toolpath should be checked carefully to ensure that the ordering is safe, particularly where a part contains vertical walls. The strategy works well on large parts with shallow walls where calculation time is critical.

- Open the part **Die_Block_Start.fm**
- Select the tool crib **Die_Block_Start.fm_Tools_from_last_save**
- Select an **Isometric view**
- Run a **3D simulation**

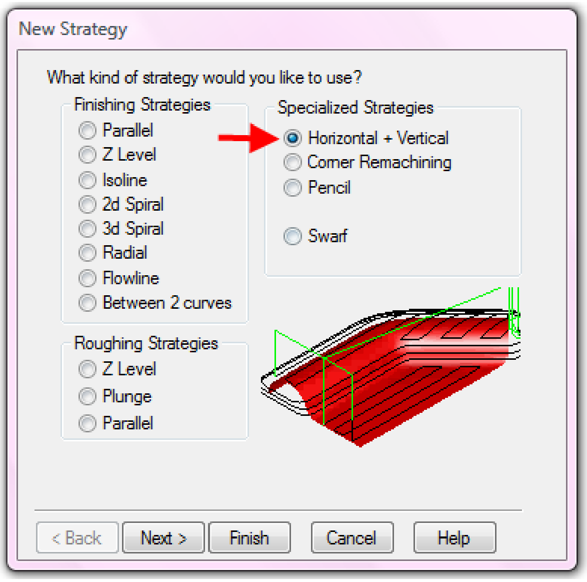


The part has been roughed out with multiple rough tools. We will now use Horizontal and Vertical finishing to complete the part.

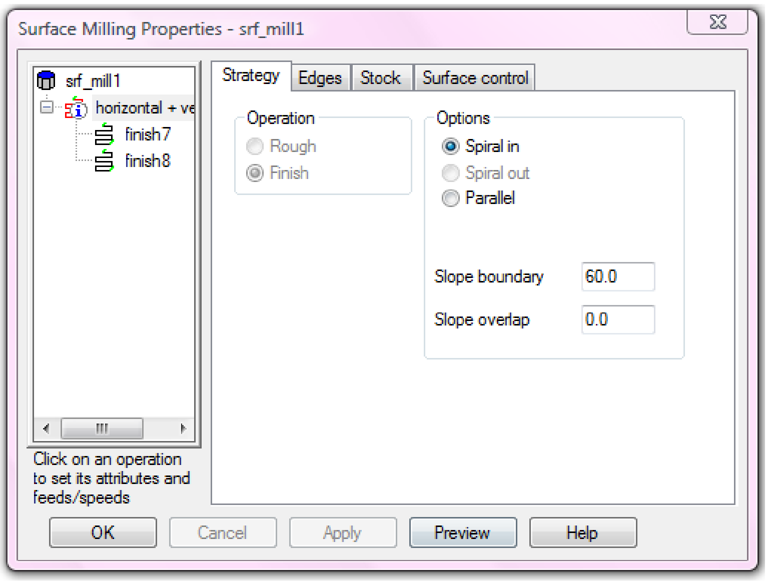
- **Eject** the simulation
- **Edit** the feature **srf_mill1**

- Select the **Process tab**
- **Uncheck** the **Z Level rough** operation
- Click on **Add new operation** 

- Under **New Strategy** select **Horizontal and Vertical**

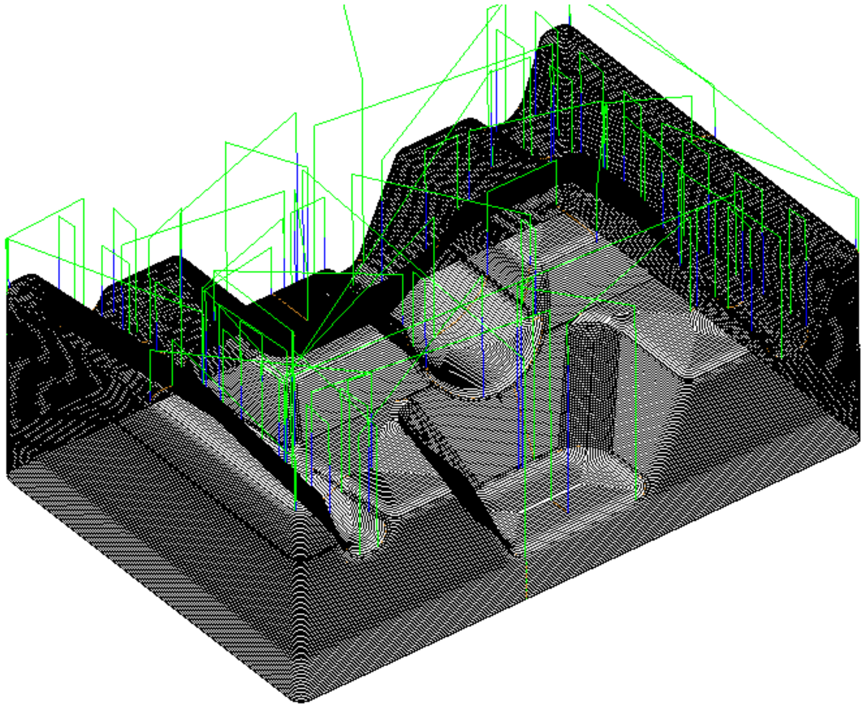


- Click **Finish**



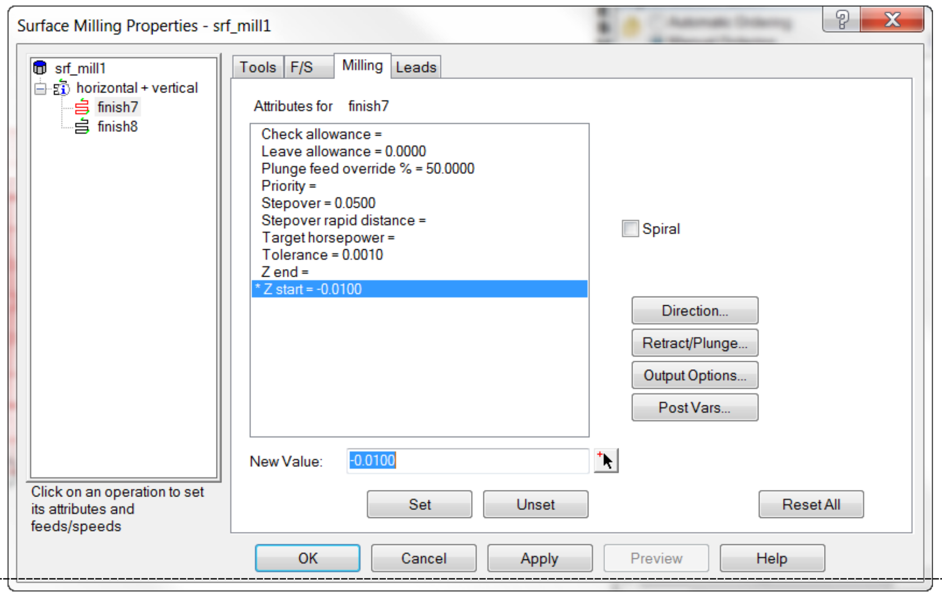
FeatureCAM has made two new machining operations under the Horizontal + Vertical strategy. The first is the horizontal operation which will machine the shallow areas of the part with a 3D Spiral strategy. The second operation is a Z Level finish which will machine the steeper areas of the part. The Slope boundary number defines the cutoff between the steep and shallow areas. The Slope overlap allows the user to overlap the areas machined by the two operations to remove any witness marks at their meeting point.

- Set the **Slope overlap** to **0.5 degrees**
- Click **Apply** and then **OK**
- Run a **Centerline simulation**

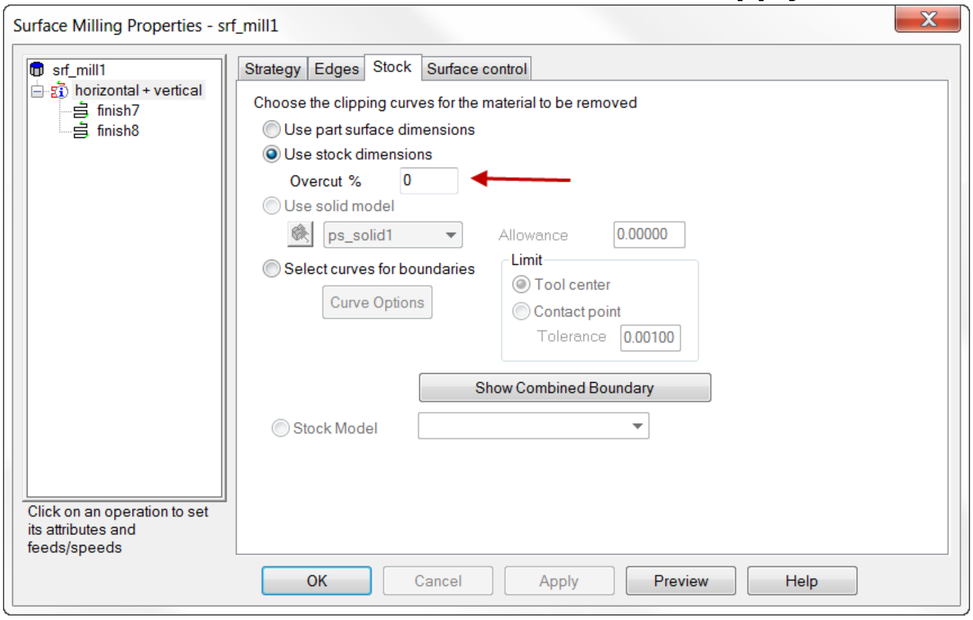


The part has been finished, however there are a number of improvements that could be made to the toolpath. We do not want to machine the upper flat surfaces; we can fix this by lowering the **Z Start** value for the 3D Spiral toolpath. The Z Level finish is machining down the side of the part; we can stop this using the stock **Overcut %**. Finally it would be better to machine the steep surfaces first then the shallow; we can do this just by changing the operation order in the Op list.

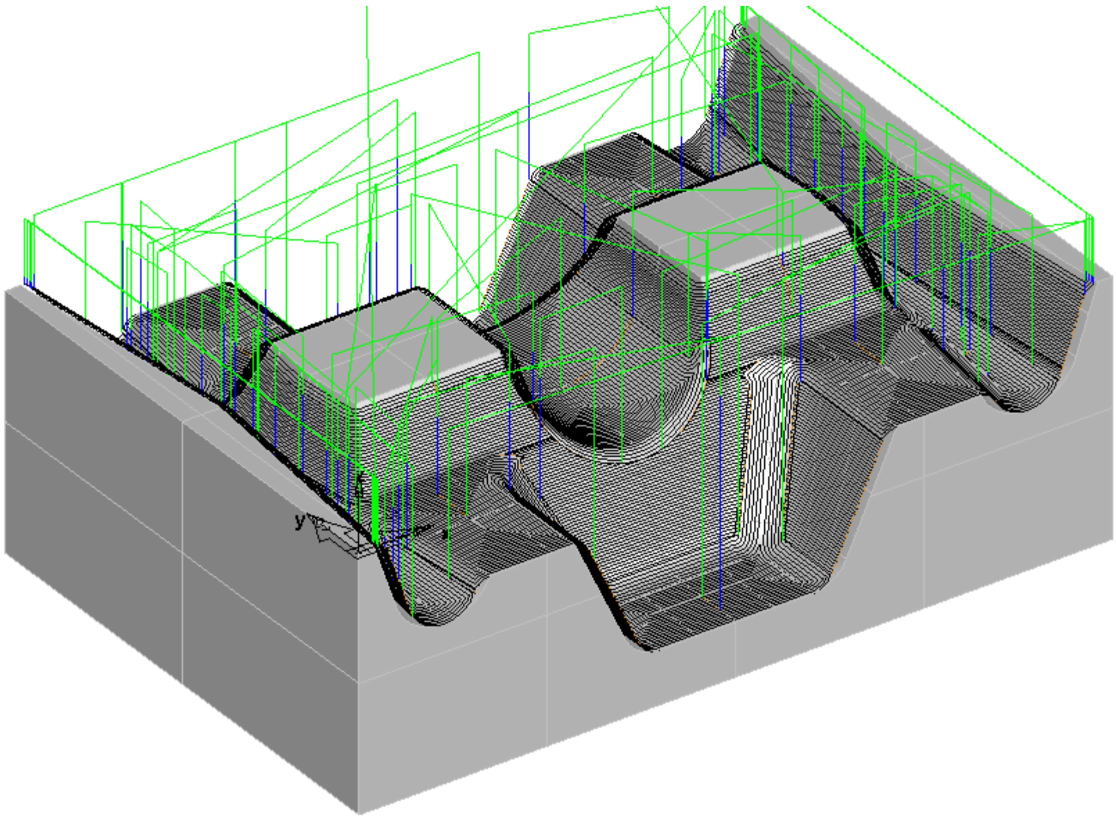
- **Eject** the **simulation**
- Select the first (**3D Spiral**) finish operation
- Select the **Milling tab**
- Set the **Z Start** value to **-0.01”** and then click **Apply**



- Click on the **Horizontal & Vertic**al operation
- Select the **Stock tab**
- Set the stock **Overcut %** to **Zero** then click **Apply**

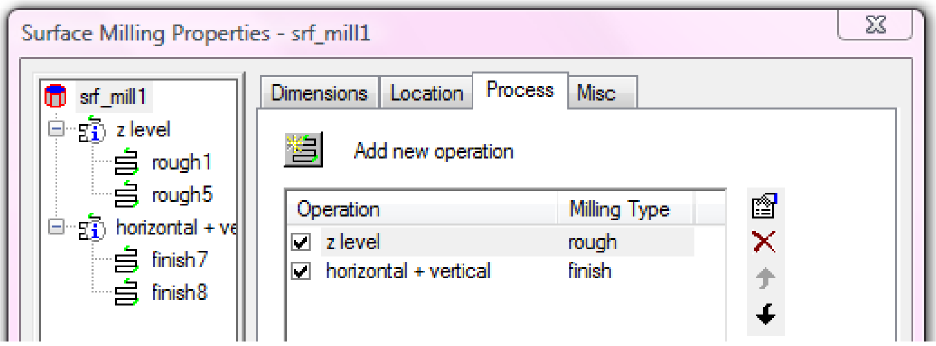


- Click **Apply** and then **OK**
- Run a **Centerline simulation**



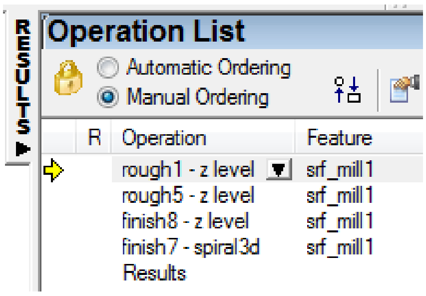
The 3D Spiral toolpath is now avoiding the upper surfaces and the Z Level does not finish the outside of the block. We will now turn on the roughing and reorder the toolpath sequence.

- Click on **srf_mill1** and select the **Process tab**
- Check the **Z Level** operation

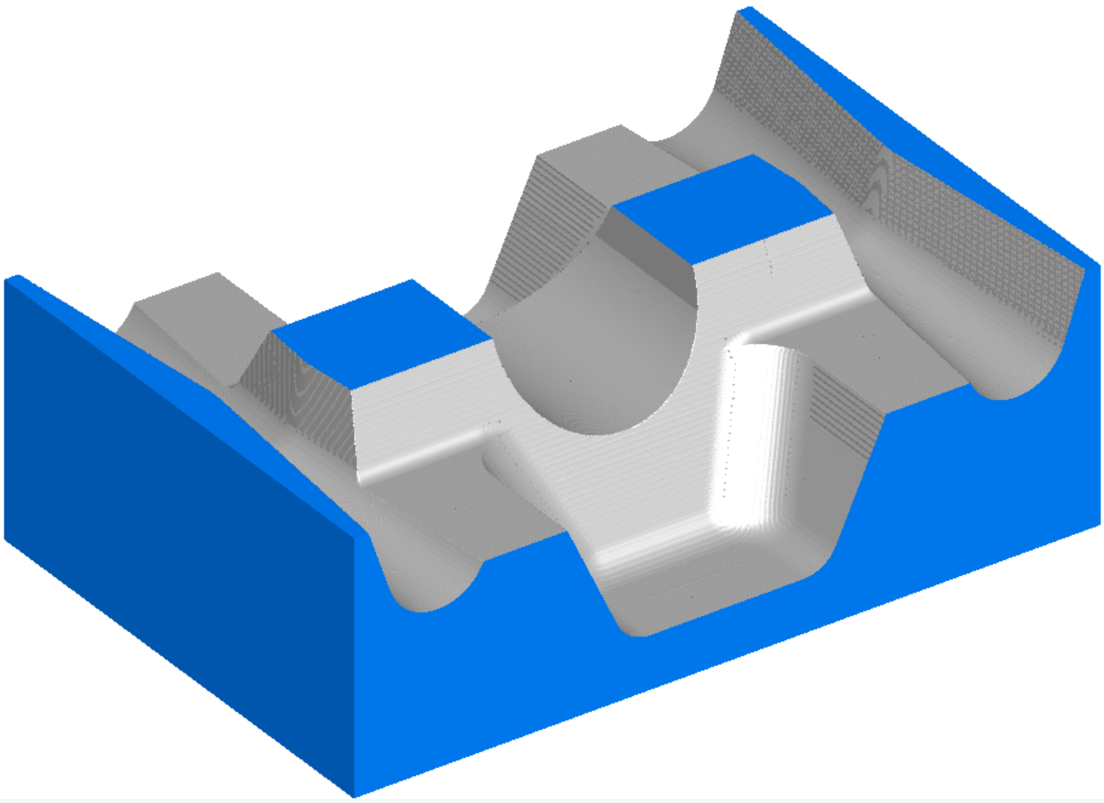


- Click **Apply** then **OK**
- In the Operation list drag the operations into order

The two Roughing operations should be first, followed by the Z Level and then finally the 3D Spiral finishing.



- Run a **3D RapidCut simulation**



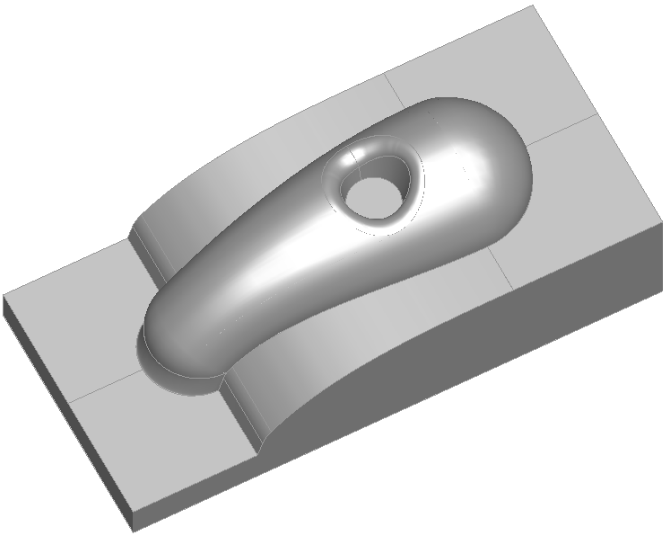
Corner & Pencil Finishing

Introduction

Corner Machining includes 5 different types of strategy: **Pencil, Multi Pencil, Along, Across and Combo Along** and **Across**. Pencil milling creates "single pass" machining along sharp internal corners while corner machining performs local re-machining of areas inaccessible to a larger tool.

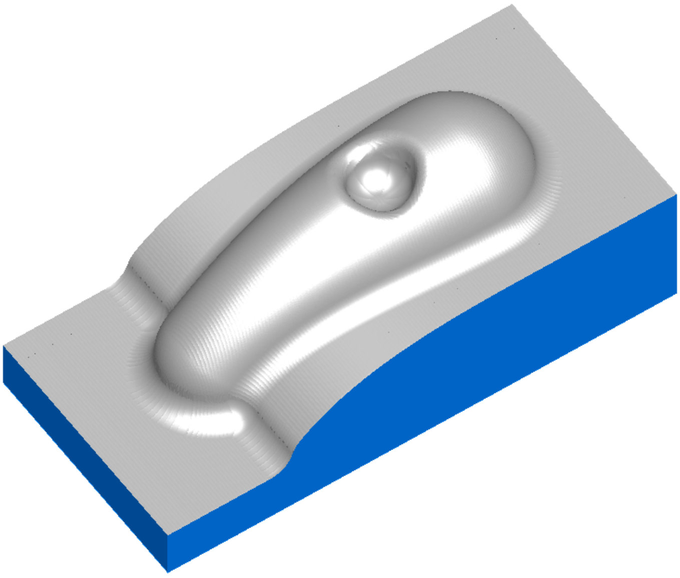
- **Open** the file **Cowling.FM**
- Select an Isometric view

This model contains a number of sharp inside corners. During this module we will see how to finish these with a smooth radius without having to model the radii into the part. On large complex models using the machining process to produce the radii on the model can give very significant reductions in the time taken for the designer to produce the model.



By using a large tool to finish the bulk of the part and then picking out the corners later with a smaller tool, we can also make significant time savings in the finishing process.

- Run a **3D RapidCut simulation**

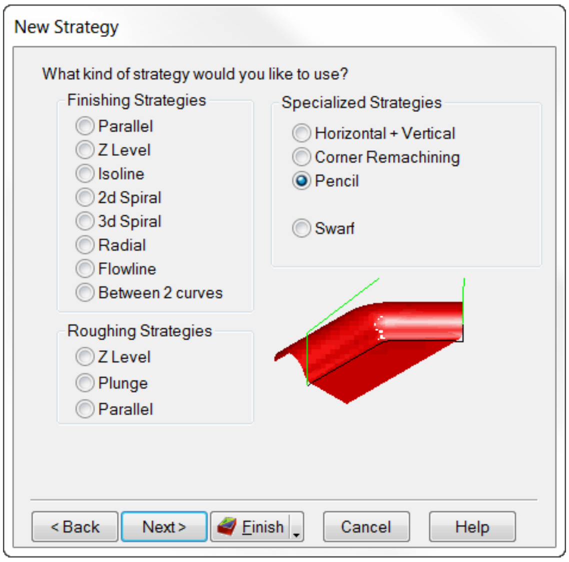


The part has been roughed out and then finished using a 0.75" ball nosed cutter. This has been done so that a large finishing stepover can be used to finish the majority of the areas while still leaving enough material in the corners to produce 0.125" fillet radii in the sharp corners on the model at the end of the machining process.

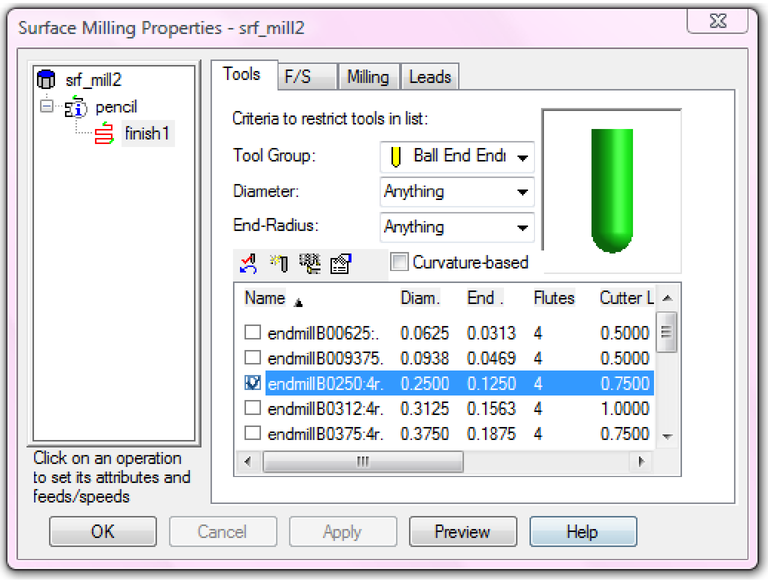
Pencil Finishing

This operation produces single pass tool tracks along the intersection between sharp internal corners of the component surface. We will now create a pencil milling feature with the default values.

- **Eject** the **simulation**
- **Uncheck** the operation **srf_mill1** in the part view
- Select all of the part surfaces
- Create a new **Surface milling** operation
- Click **Choose a single operation** and click **Next**
- Select **Pencil** strategy and click **Finish**

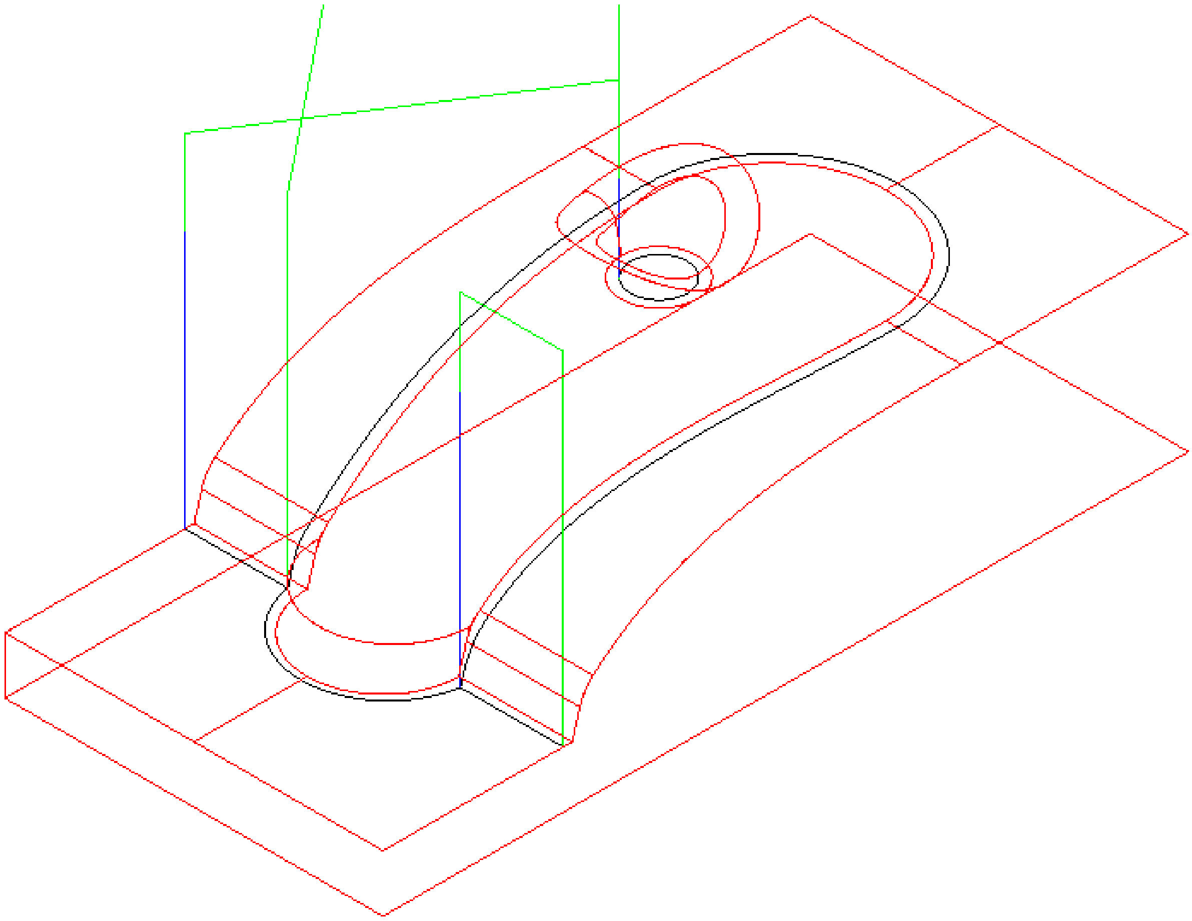


- **Edit** the **feature**
- Click on **Finish1** in the tree view
- Select the **Tools tab** and pick a **0.25" ball endmill**



- Click **Apply** and then **OK**

- **Unshade** the part
- Run a **Centerline simulation**



The tool is making a single pass around all of the inside corners on the part. Both **Corner Finishing** and **Pencil Milling** strategies allow slope limits to be specified on the Slopes tab of the operation.

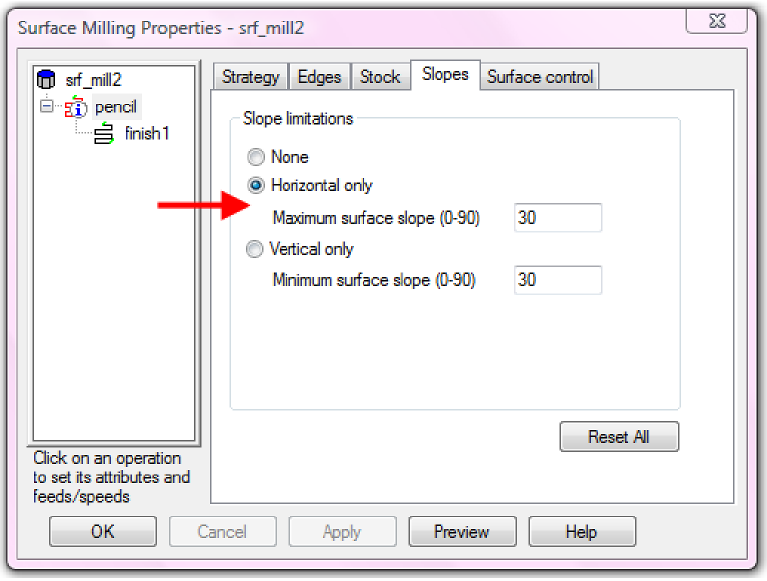
Horizontal only limits the cutting to regions with a slope less than the user defined maximum surface slope.

Vertical only limits the cutting to steep regions with a slope greater than user defined minimum surface slope.

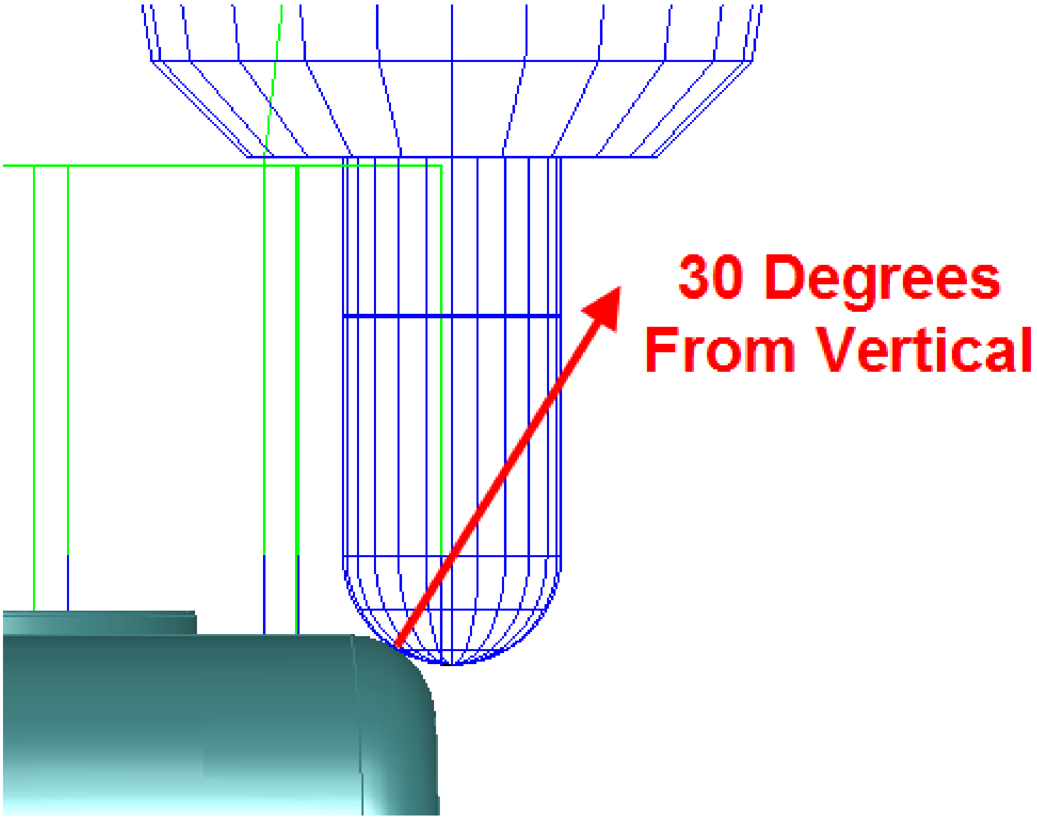
Limiting the regions where an operation is performed can solve the potential problems associated with the tool running up or down steep slopes. For instance the user could apply a stitch strategy to track across the steep areas and parallel strategy for shallow areas. Also a higher feed rate could be used on the shallow areas and a lower feed rate on the steep areas, which if applied first is more likely to be taking a heavy cut.

- **Eject** the **simulation**
- **Edit** the **feature**
- Click on **Pencil** in the tree view and click on the **Slopes** tab
- On the **Slopes tab**, Click **Horizontal only**

- Set the **Maximum surface slope** to **30**

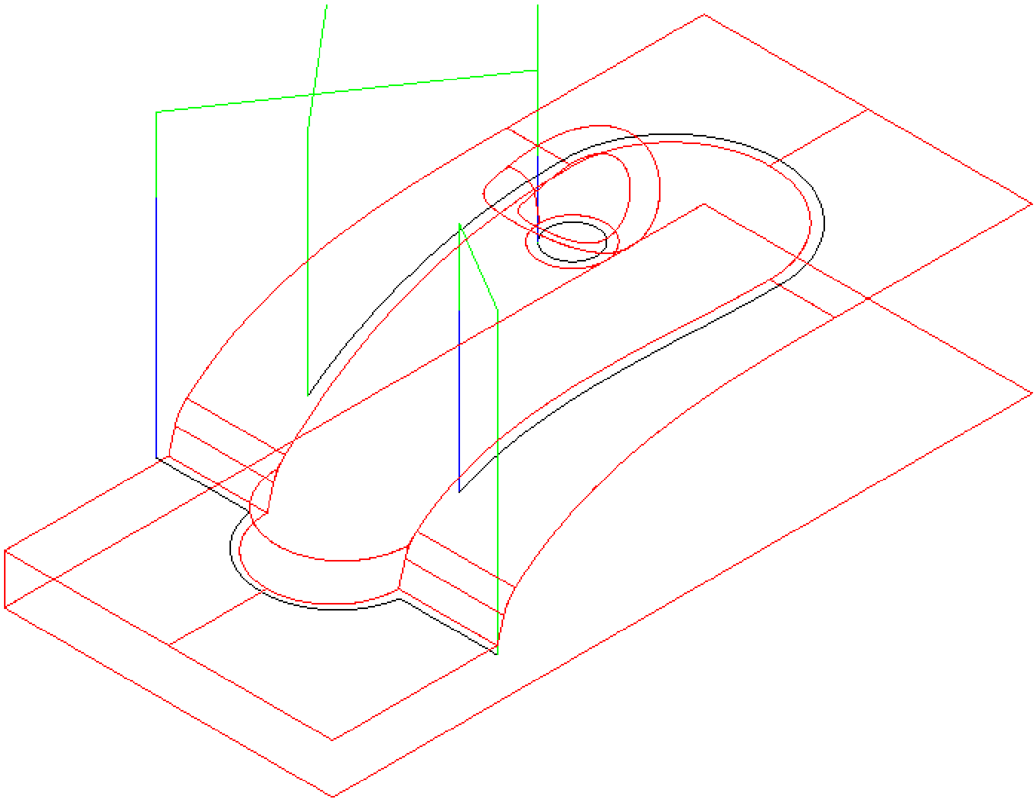


- Click **Apply** and then **OK**



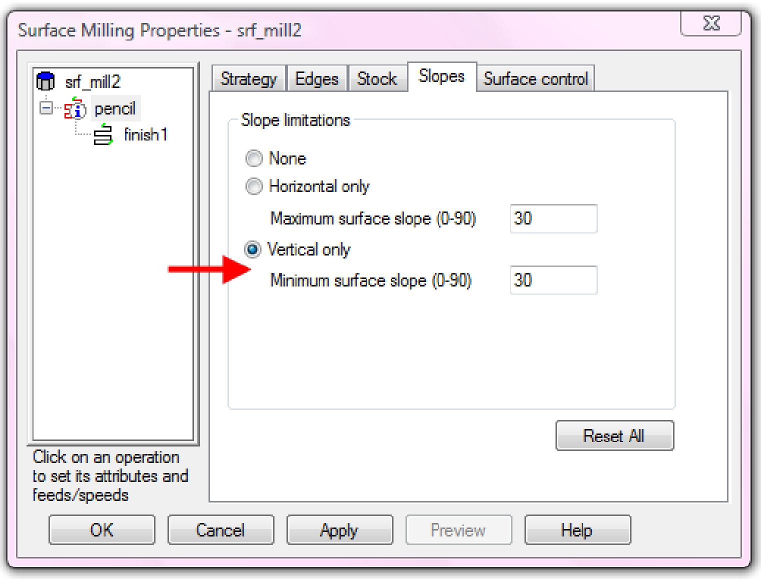
We are restricting the toolpath so that it only cuts those areas on the model where the surface normal is inclined at less than 30 degrees to the vertical.

- Run a **Centerline simulation**

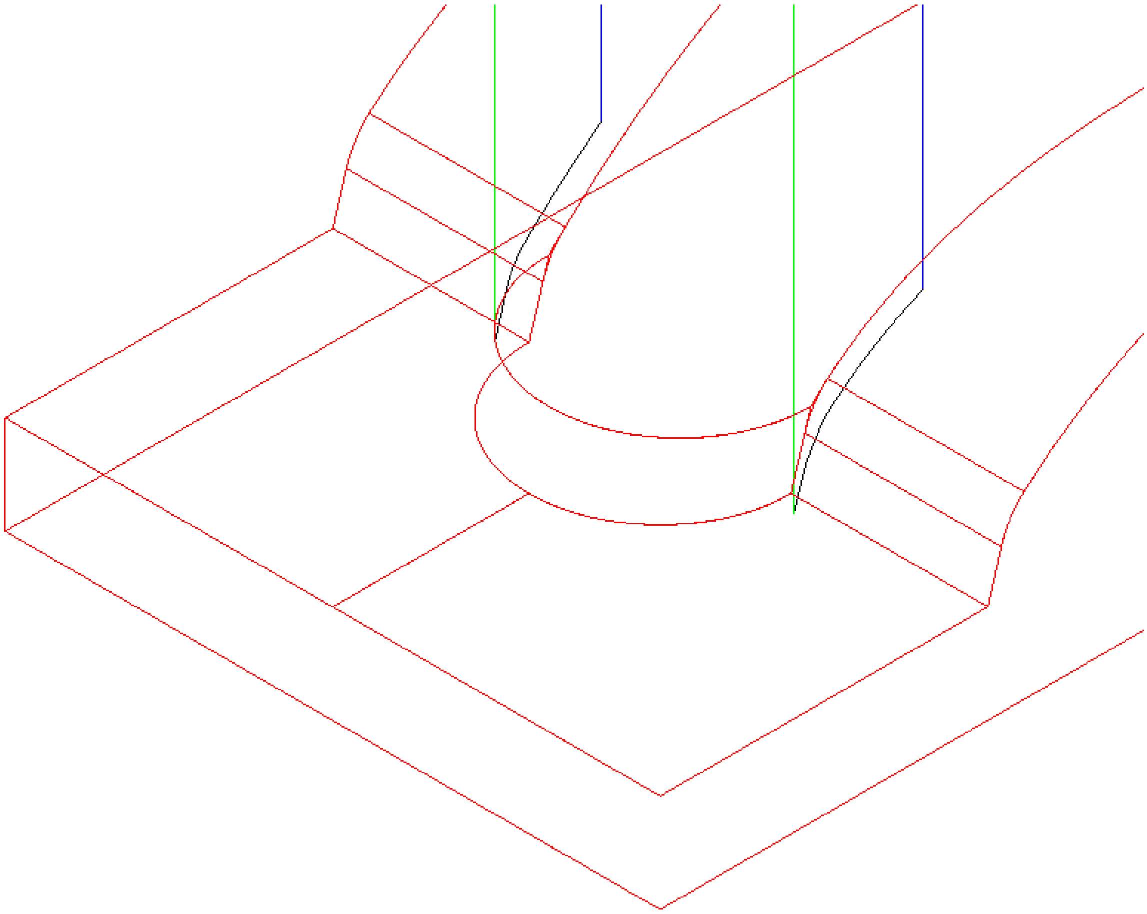


The toolpath is now only cutting the shallow areas of the model and avoiding the steep areas.

- **Eject** the **simulation**
- **Edit** the **feature**
- Click on **Pencil** in the tree view and click on the **Slopes** tab
- Click **Vertical only** and set the **Minimum surface slope** to **30**
- Click **Apply** and then **OK**



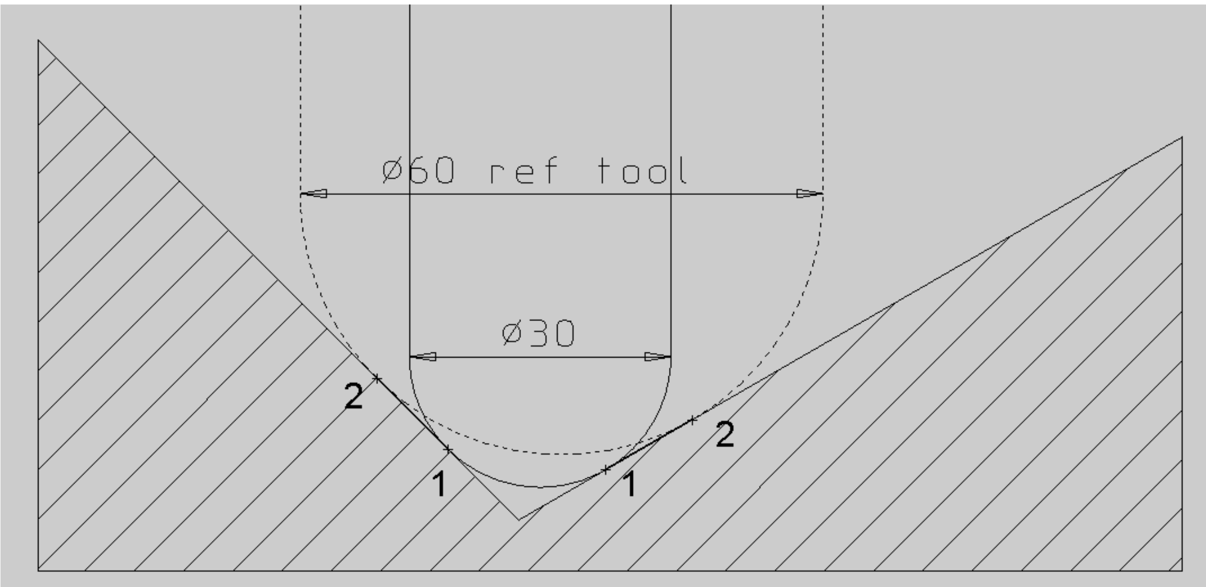
- Run a **Centerline simulation**



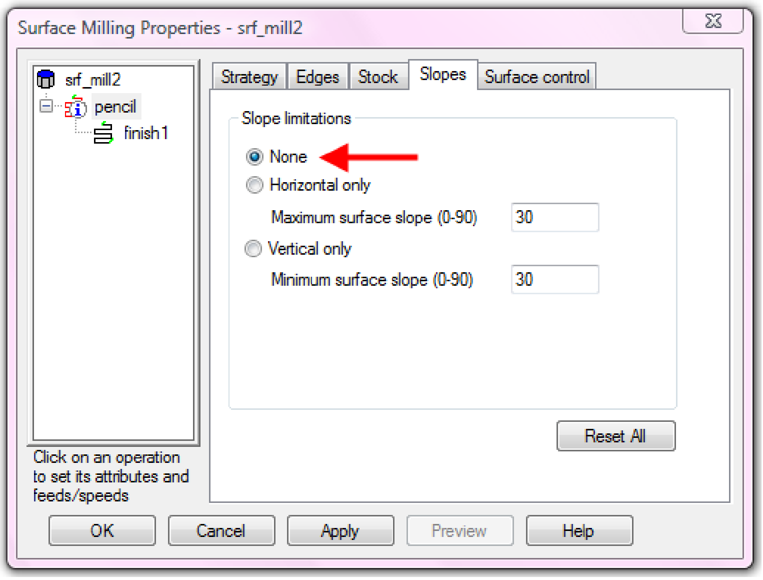
The toolpath is now only cutting the steep areas of the model and avoiding the shallow areas.

Corner Along Finishing

All the **Corner Finishing** strategies, (except for **Pencil**) are corner re-machining techniques. In the illustration below a 60mm diameter tool would have left material in the corner between the points marked (2). The 30mm diameter corner finishing tool needs to remove the extra material between the points (1) and (2) on each side of the corner.

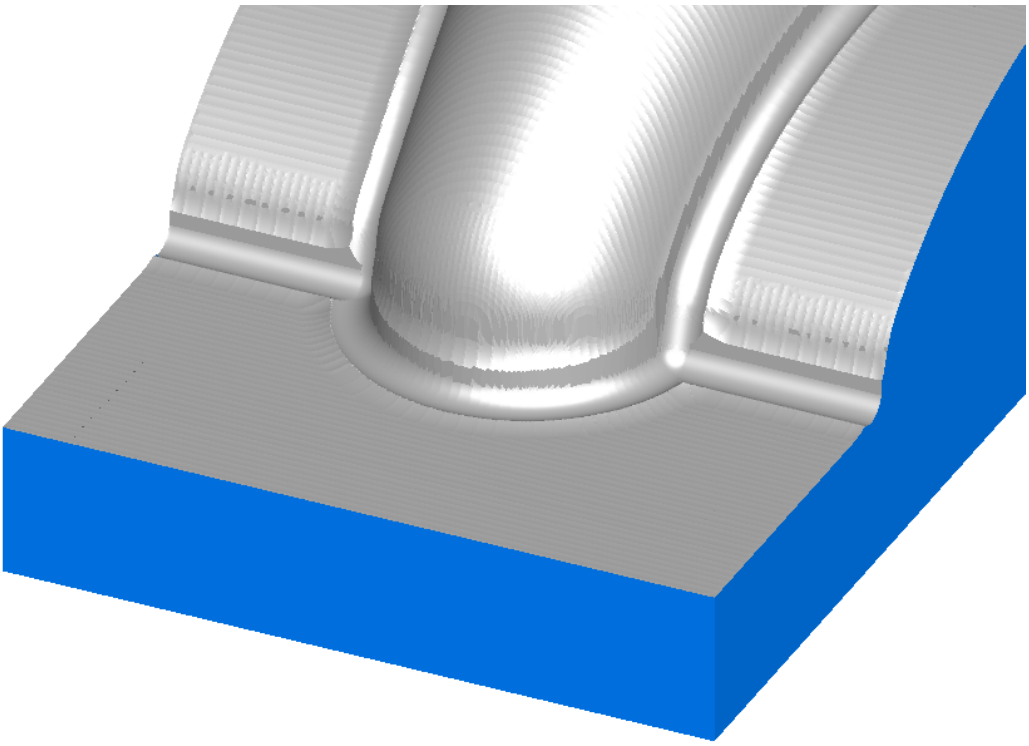


- **Check** the surface milling feature named **srf_mill1** in the **Part View**
- **Edit** the **pencil milling** feature
- On the **Slopes tab** set **Slope limitations** to **None**
- Click **Apply** and then **OK**

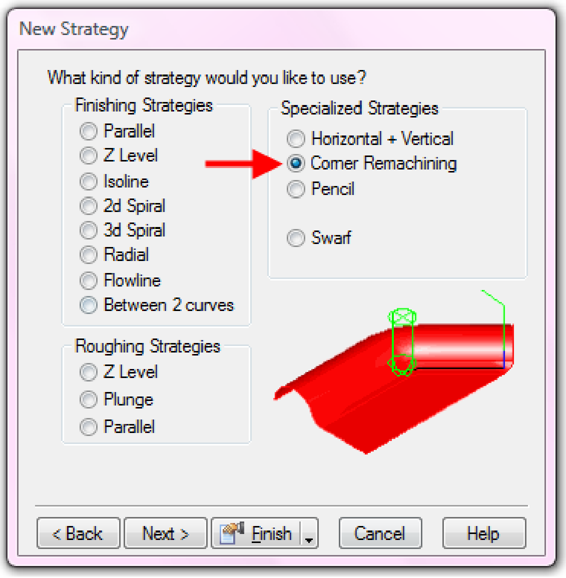


- Run a **3D RapidCut simulation**

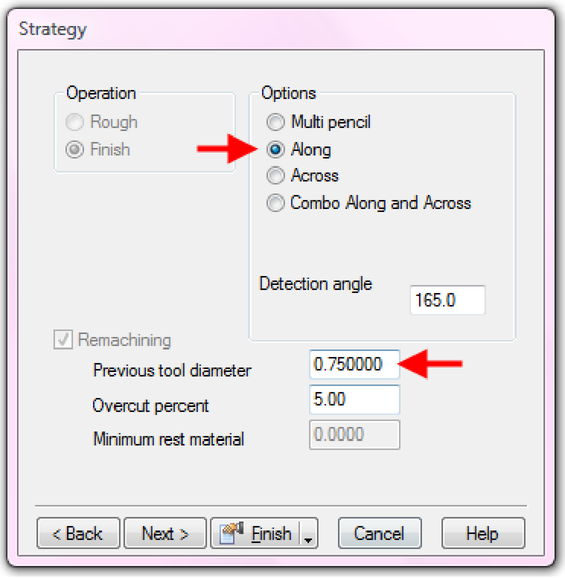
You can clearly see in the inside corners that there is material remaining between the areas finished by the 0.75” cutter and the 0.25” tool used in the pencil operation.



- **Eject** the **simulation**
- **Uncheck srf_mill1** and **srf_mill2** in the Parts view
- Select all of the part surfaces
- Create a new **Surface milling** operation
- Click **Choose a single operation** and click **Next**
- On the New strategy tab select Corner Remachining

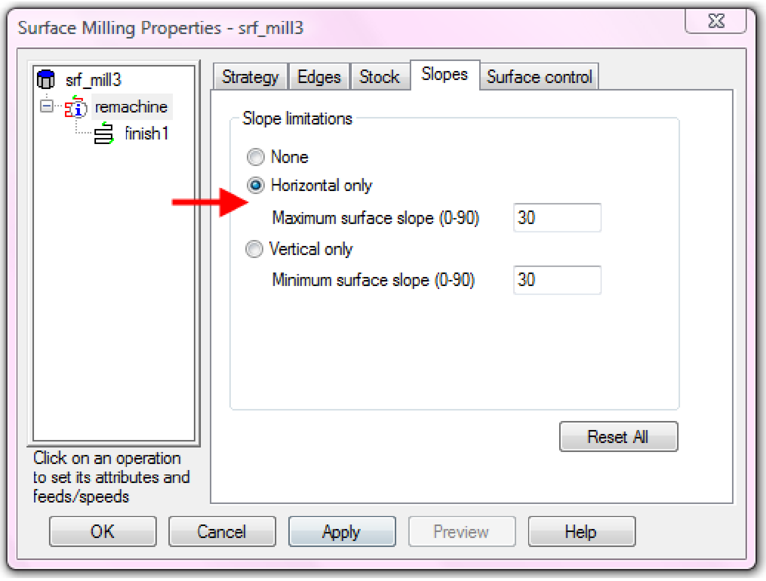


- Click **Next**
- Select **Along** and set the **Previous tool diameter** to **0.75"**



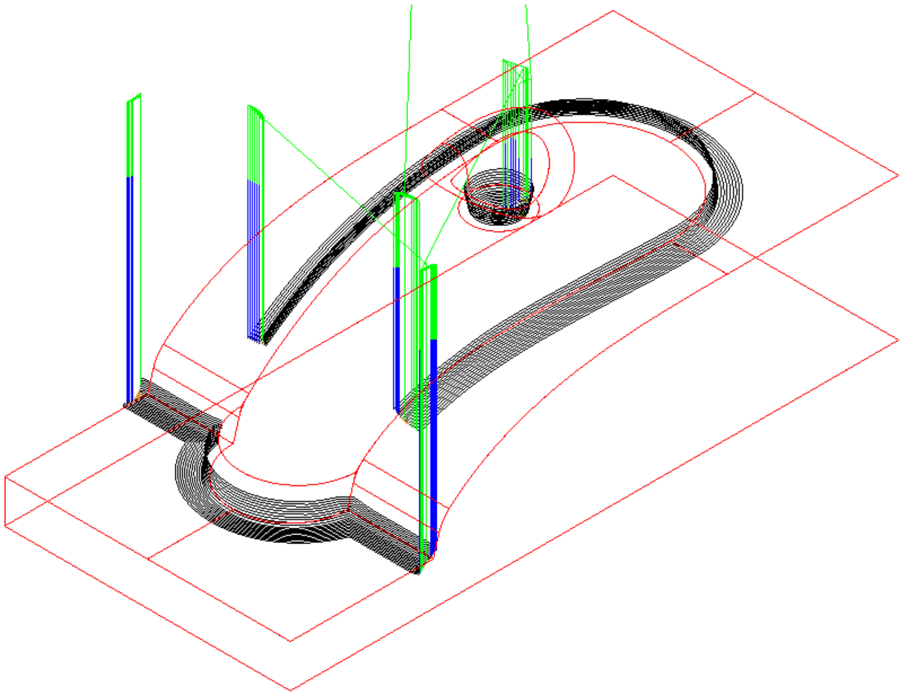
Nb. The Overcut percent figure defines the overlap of the Remachining operation over the original finishing operation. By default this is 5% which will overlap the two toolpaths sufficiently to eliminate any witness marks where the toolpaths meet.

- Click **Finish**
- Edit the feature. Click on **Remachine** in the tree view
- On the Slopes tab, Click **Horizontal only** and set the **Maximum surface slope** to **30**



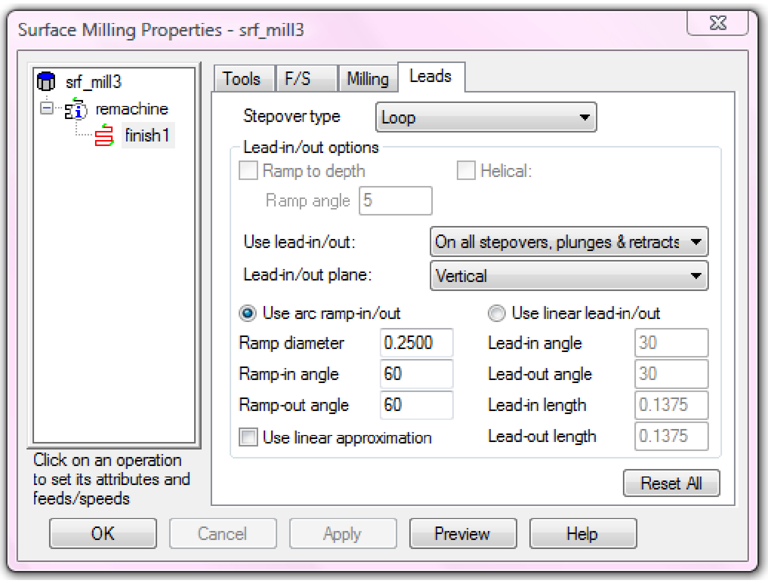
- **Click** on **finish1** in the tree view

- Click on the **Tools** tab and select a **0.25” ball endmill**
- Click **Apply** and then **OK**
- Run a **Centerline simulation**



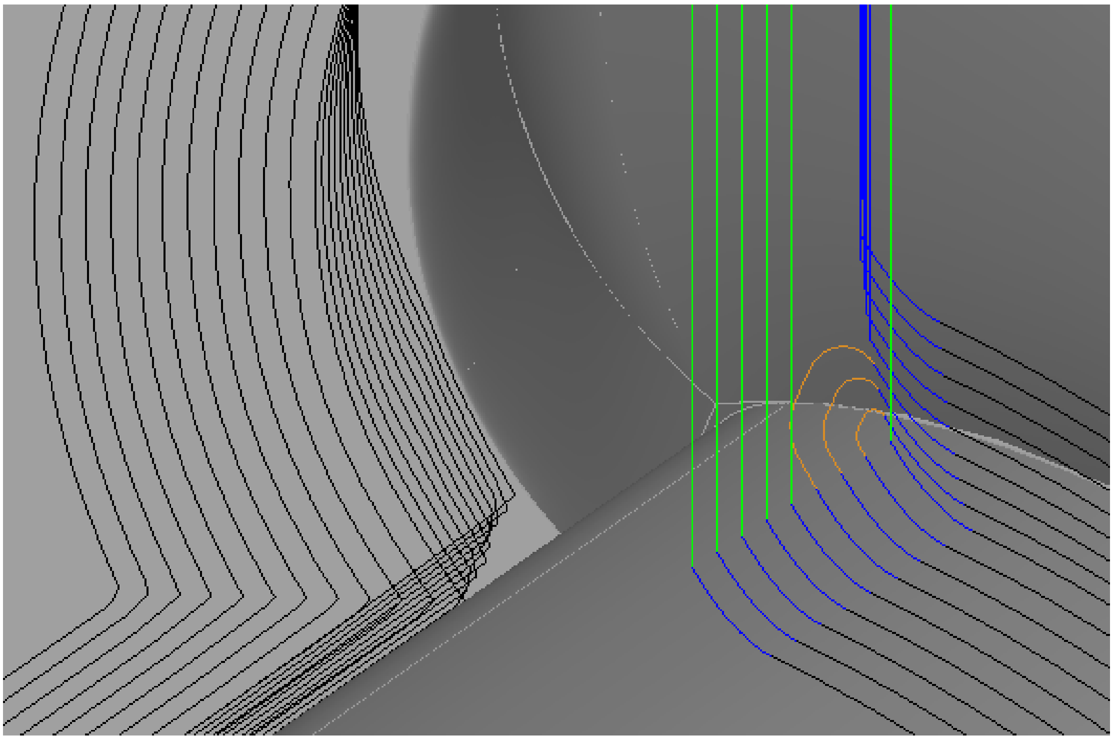
The shallow areas are being cleared of excess material from the outside inwards in a climb milling direction. There are multiple lift offs resulting in a lot of “air time”. The tool is also plunging directly into the remaining material. We can remedy both of these problems by modifying the leads and links.

- **Eject** the **simulation** and **Edit** the **feature**
- Click on **Finish1** in the tree view and click on the **Leads tab**
- Fill in the form **exactly** as shown



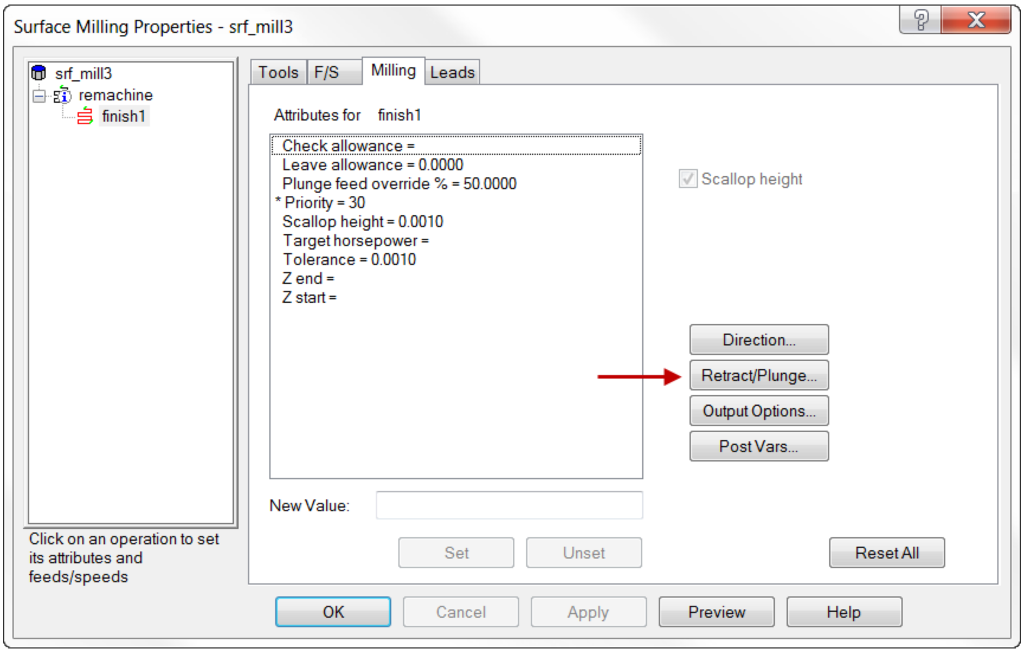
- Click **Apply** and then **OK**

- Run a **Centerline simulation**

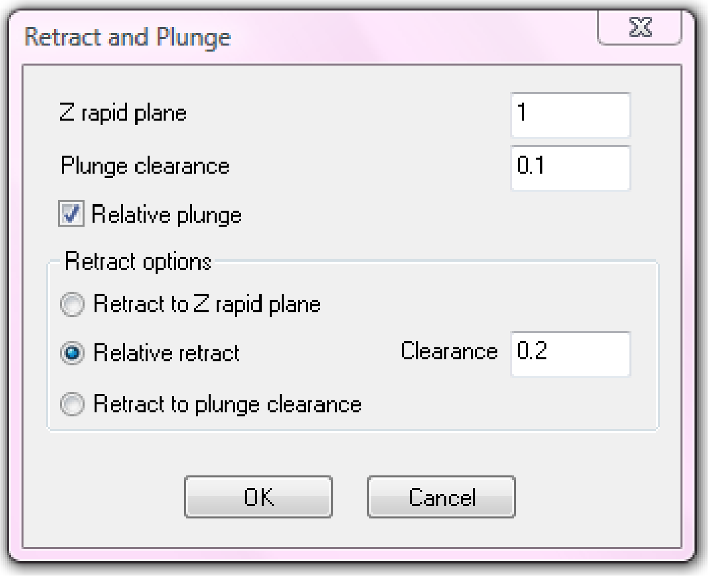


The tool is now arcing on and off the part smoothly. Short links are joined by loops. Longer links are still lifting off the job. We will now reduce the amount of time spent in the air by switching to Relative Plunge/Retract.

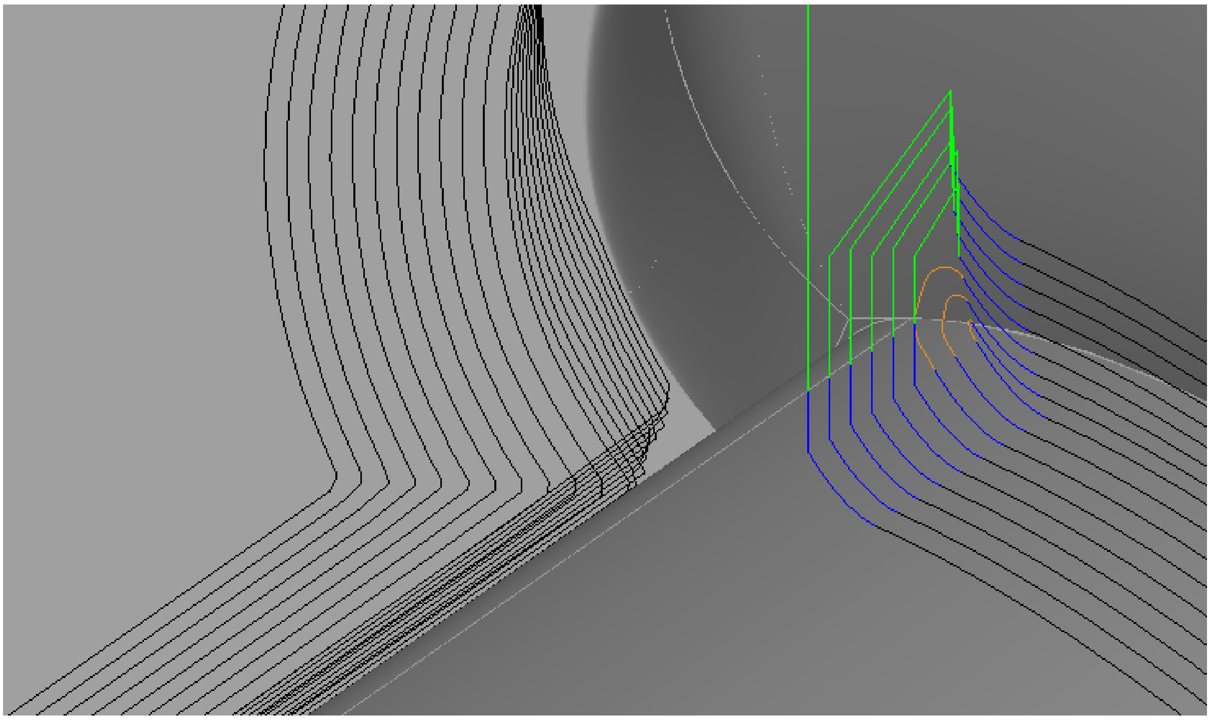
- **Eject** the **simulation** and **Edit** the **feature**
- Click on **Finish1** in the tree view and click on the **Milling tab**
  Note: The surface finish is controlled by the scallop height value
- Click on **Retract/Plunge**



- Fill in the form **exactly** as shown and click **OK**



- Click **Apply** and then **OK**
- Run a **Centerline simulation**



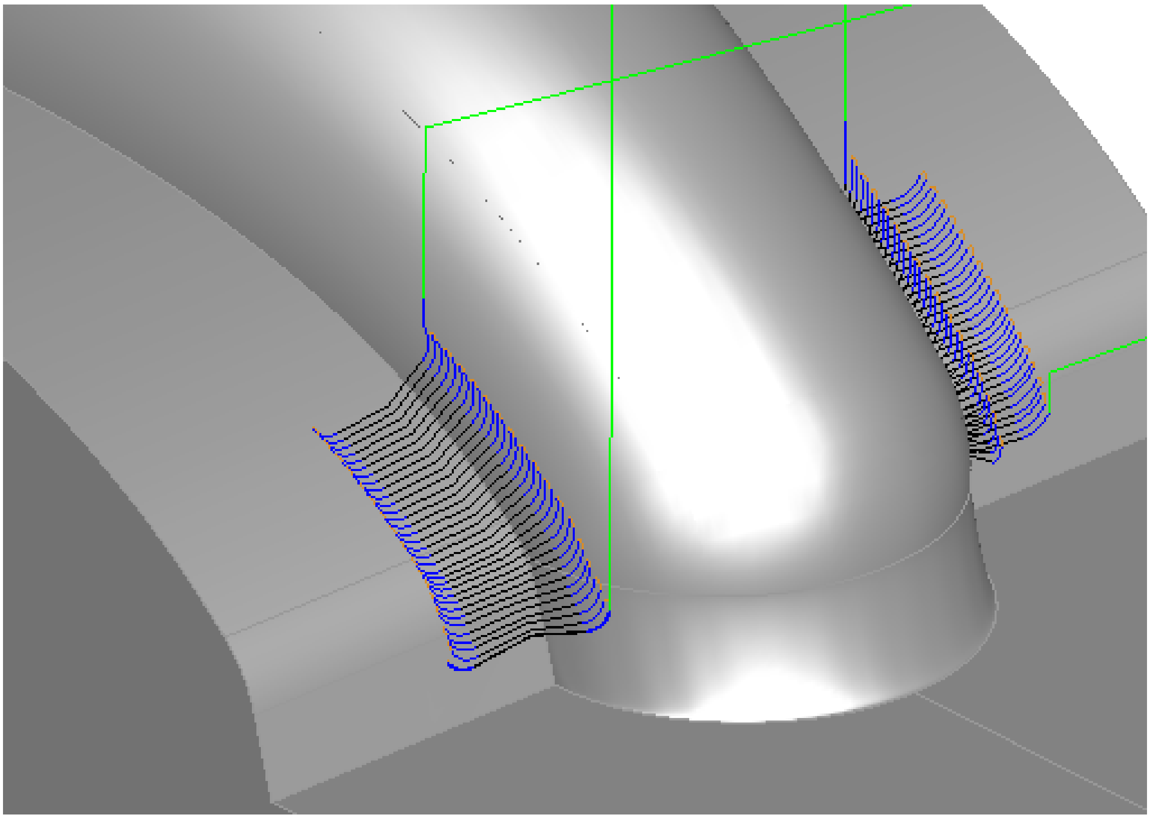
Relative plunge tells FeatureCAM that you wish to plunge at rapid down to the specified Plunge clearance above the next point to be cut rather than plunging at a reduced feedrate all the way from the Z rapid plane. This reduces the length of time spent on slow plunge moves.

Relative retract instruct FeatureCAM to retract by only enough to maintain the specified clearance from the part when moving at rapid from one toolpath pass to the next. Another name for this is Skimming as the tool skims above any obstacles rather than retracting all the way to the z rapid plane.

Corner Across Finishing

Along finishing works well on shallow areas of a part, however, where a corner runs down a steep area the strategy is less suitable as it will either be machining down the corner or when cutting it will be bringing the tool sideways into the remaining material at the bottom of the pass. In such cases it is better to use a Corner Across strategy where the tool clears the remaining material by beginning at the top of the steep area and “stitches” to and fro across the corner. The tool loading is thus reduced giving a good surface finish with a smaller risk of tool damage.

- **Eject** the **simulation**
- **Edit** the **srf_mill3** feature
- Click on **Remachine** in the part view and select **Across** as the **Option**
- Click on **Slopes** tab and Select **Vertical only** and enter **a Minimum surface slope** of **30**
- On the **Stock tab** change the **Overcut %** to **Zero**
- Click **Apply** and then **OK**
- Run a **Centerline simulation**

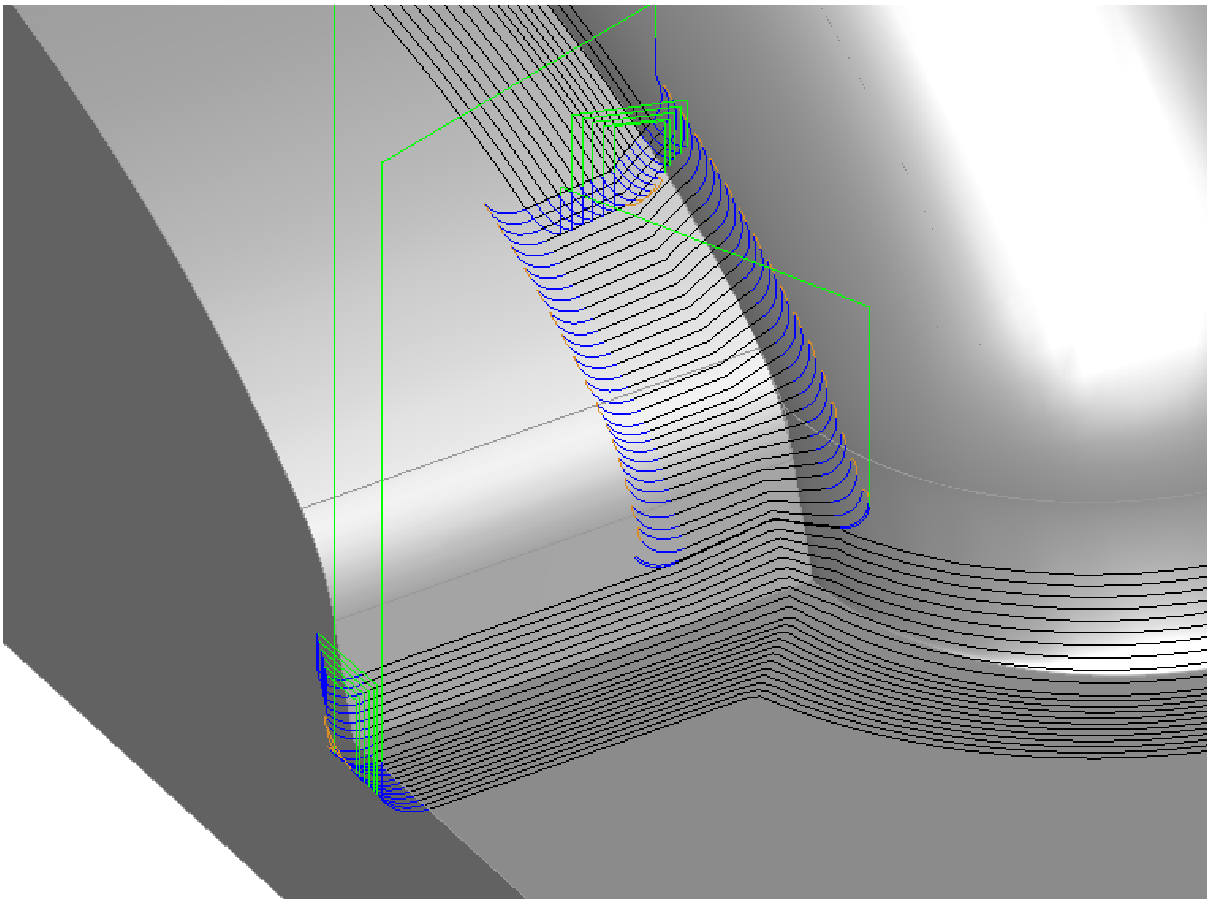


The tool stitches across the corner in such a way that each pass follows the surface normals of the intersecting surfaces at the point of contact of the tool. On a horizontal corner the place of the toolpath passes would be vertical, when the corner is vertical the toll will make passes in the horizontal plane. Note how the plane of the tool passes varies smoothly with the changing angle of the surfaces being machined.

Combo Along and Across Corner Finishing

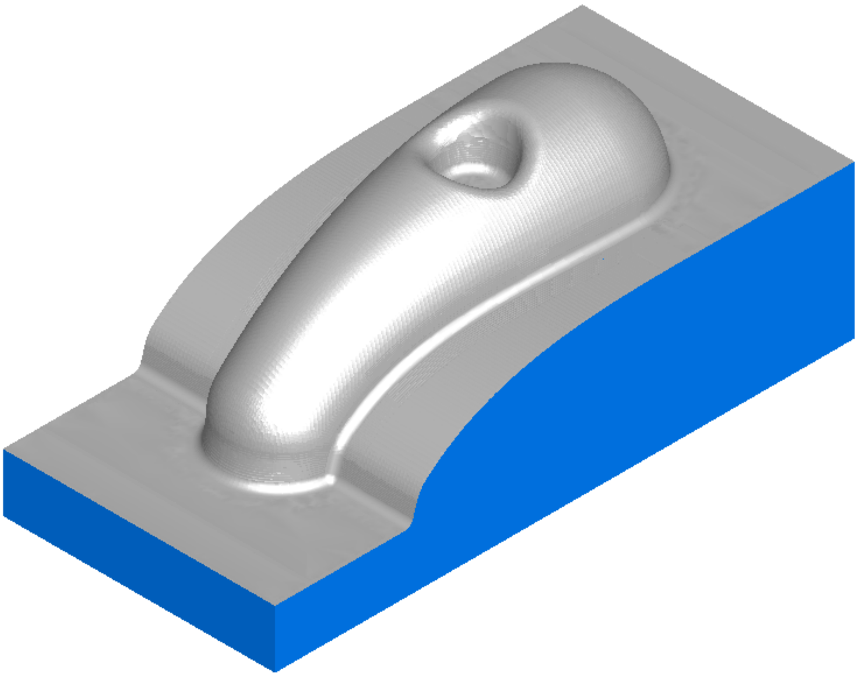
In the examples above we have seen how steep and shallow corners can be machined using Along and Across strategies respectively. By creating two operations and choosing appropriate slope angles we can machine all of the corners on a part. In order to make this easier FeatureCAM has another strategy which automatically combines the along and across strategies in a single toolpath.

- **Eject** the **simulation**
- **Edit** the **srf_mill2** feature
- Click on **Remachine**
- On the **Strategy tab** select **Combo Along and Across**
- On the Slopes tab, Click **None**
- Click **OK** and **simulate** the toolpaths

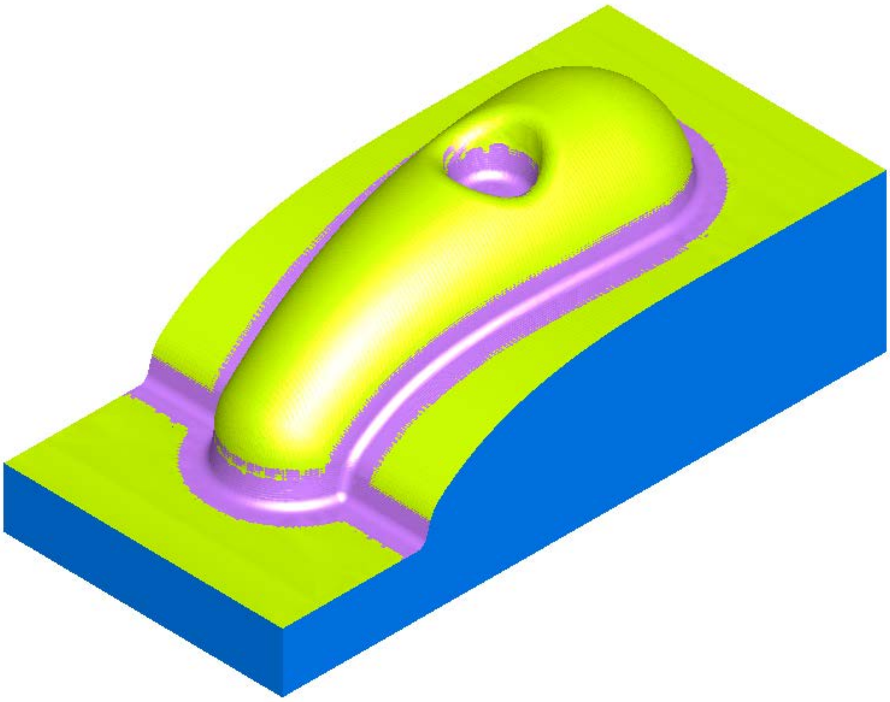


This has machined the shallow areas using an Along strategy and the steep areas using Across. This gives the optimum surface finish whilst protecting the tool in steep areas. The two parts of the toolpath meet exactly at the point where the slope angle changes from steep to shallow. Where the smoothness of the fillets are important it may be desirable to follow an Across or Combo toolpath with a Pencil operation to remove the scallops left where the tool has stitched across the corner.

- Turn on the features **srf_mill1** and **srf_mill2** in the part view
- **Edit** the feature **srf_mill2**
- On the **Slopes tab** select **None**
- Click **Apply** and then **OK**
- Reorder the toolpaths in the operation view so the pencil is last
- Run a **3D simulation**



- Turn on **Tool colors** in the **Simulations options** - **General tab**
- Run a **3D simulation**

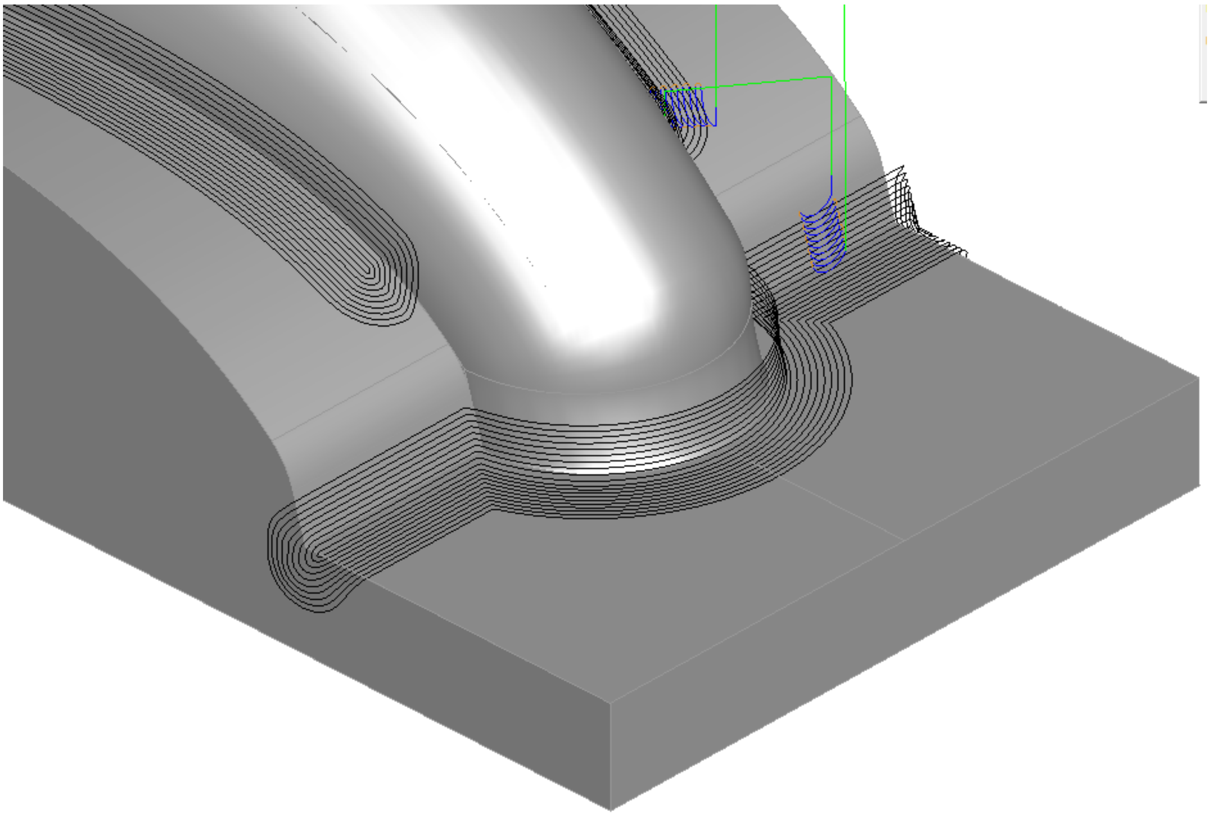


You can now clearly see the area where the Remachining has removed the excess stock.

Corner Multi Pencil Finishing

Multi Pencil finishing is a variation on Corner Along machining. The main difference is that the toolpaths are ordered in such a way as to minimize the number of times the tool lifts off the job. Successive passes are joined by smooth looping moves. The toolpath is best suited to parts with shallow corner areas and materials/tooling where the cutting direction is not important as it is not possible to control the cutting direction.

- **Eject** the **simulation**
- **Uncheck srf_mill1** and **srf_mill2** in the Part view
- **Edit** the **srf_mill3** feature
- Click on **Remachine** in the part view and select **Multi pencil** as the Option
- On the **Slopes** tab, Click **Horizontal only** and set the **Maximum surface slope** to **30**
- On the **Stock tab** set the **Overcut %** to **100**
- Click **Apply** and then **OK**
- Run a **Centreline** simulation



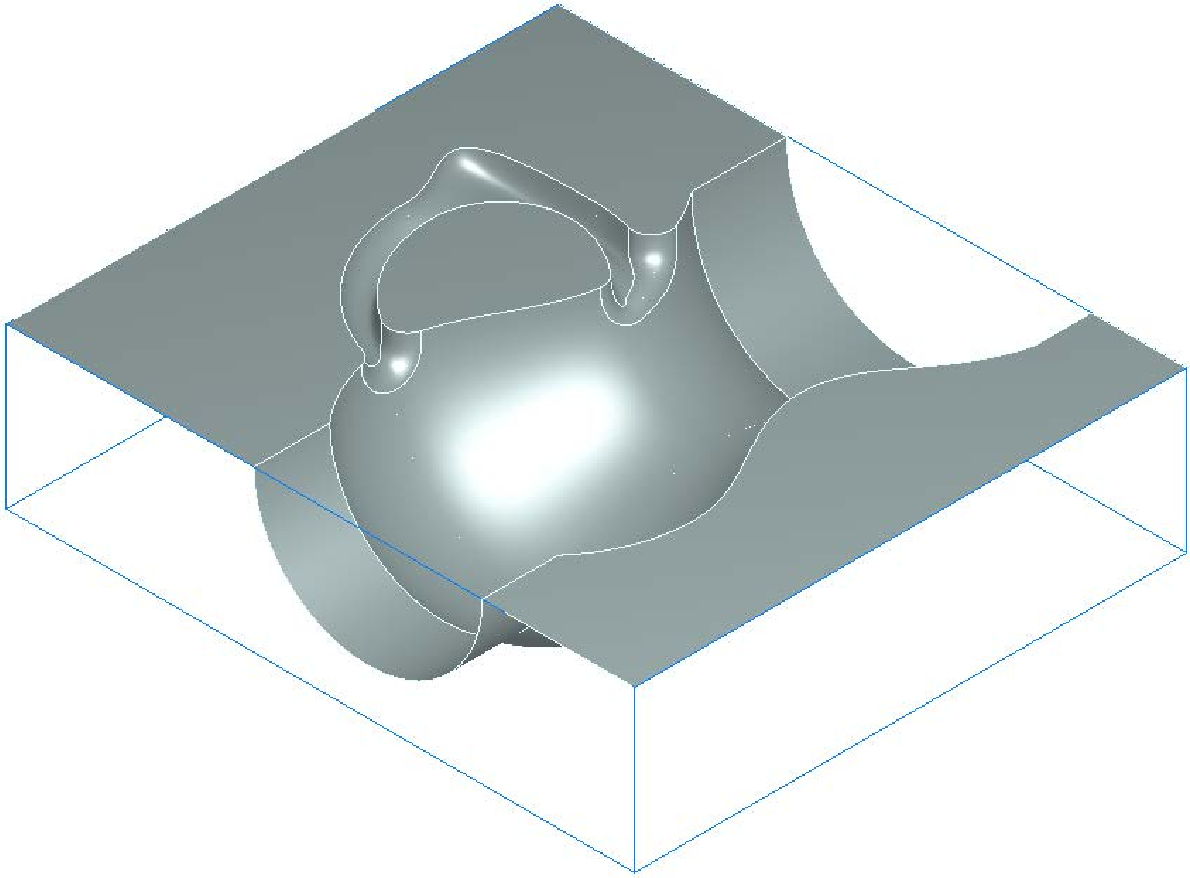
This toolpath is similar to the **Along** strategy except that it doesn't lift off the job as much. The only disadvantage is that it alternates between climb and conventional milling.

Isoline Finish Machining

Introduction

Isoline machining is a finishing technique that machines each surface in a part individually. The tooling passes are aligned with the internal curves (isolines) that make up the surfaces. This often gives the user a greater degree of control over the cutting direction and surface finish than other methods. In this example, we shall machine a die for a jug.

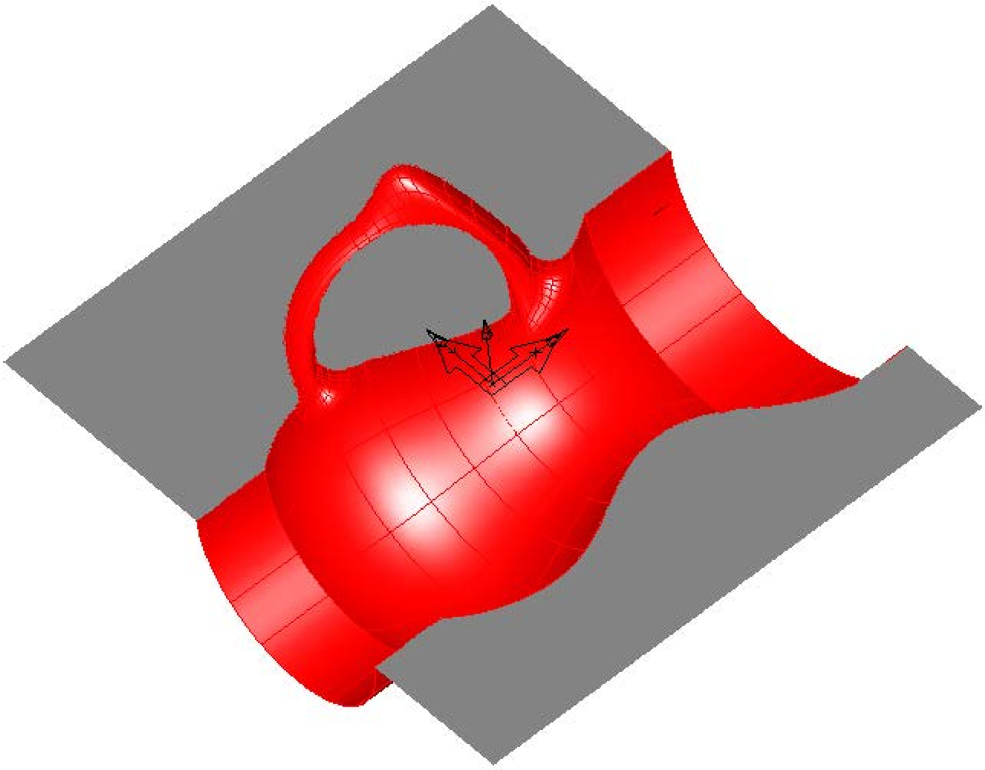
- Open the part **Isoline.fm**
- Load the tool crib **Isoline.fm_Tools_from_last_save**
- Select an **Isometric view**



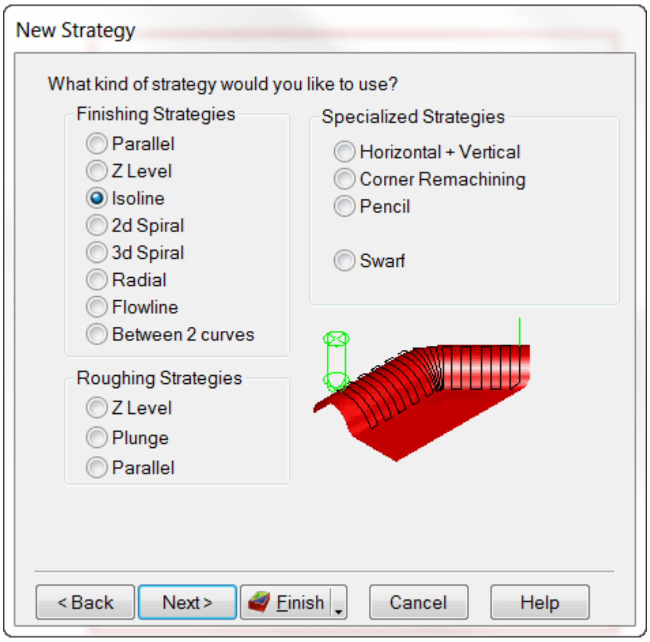
This part contains nine surfaces. Roughing strategies to clear out the bulk of the stock have already been set up. We will now create an isoline finishing strategy for each of the surfaces.

In this case we only want to finish the curved surfaces. The flat top of the stock has been finished by a facing operation. We therefore do not need to include these upper flat surfaces in the list of surfaces to be machined.

- Select all of the surfaces except for the three flat surfaces

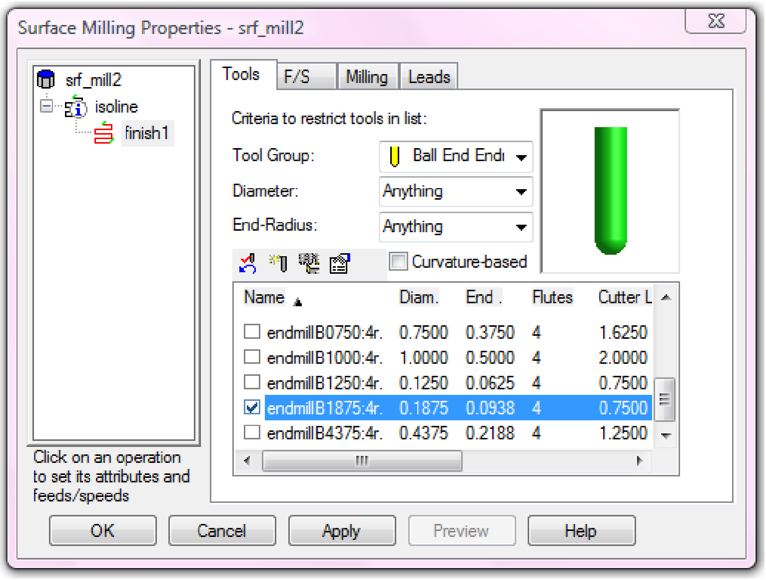


- Create a new **surface milling feature**
- Check **Choose a single operation**
- Click **Next** until you reach the **New strategy page**
- Select **Isoline**

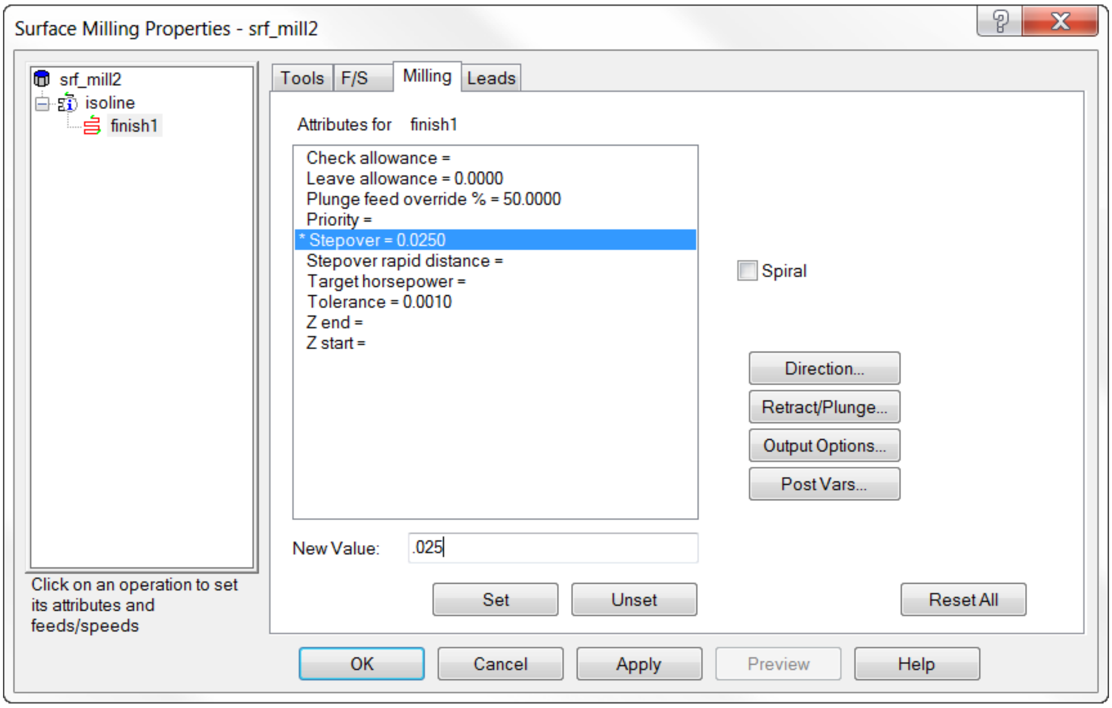


- Click **Finish**

- Click on **finish1** and select the **Tools tab**

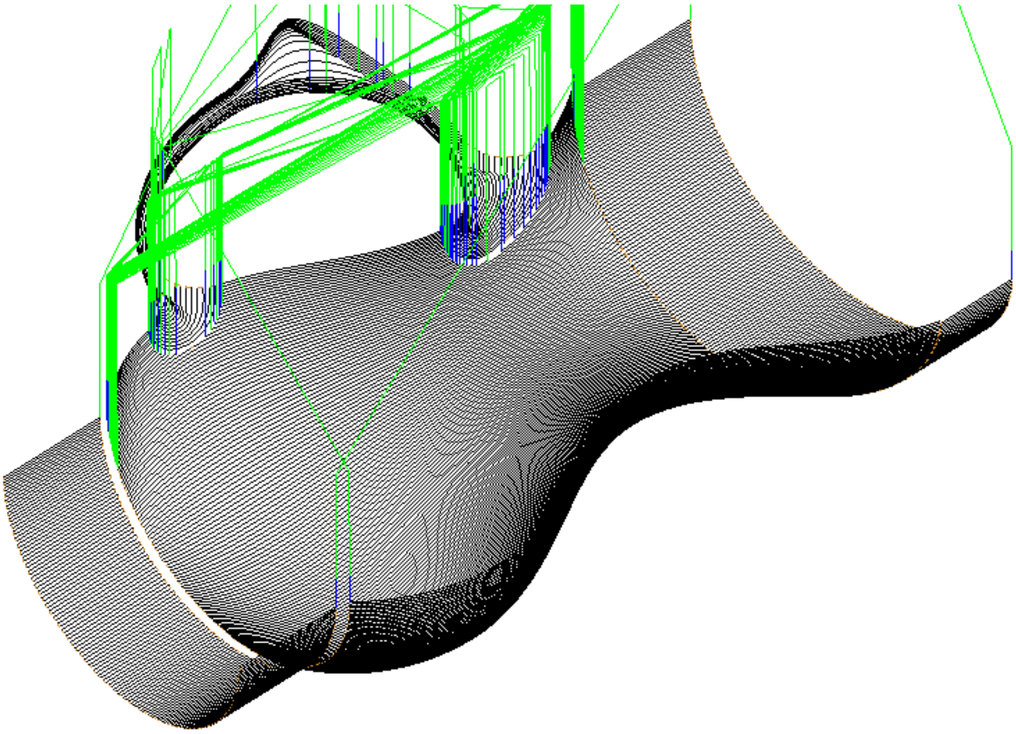


- Select a **0.1875" diameter ball nosed tool** and press **Apply**
- Click on the **Milling tab** and set the **stepover** to **0.025"**



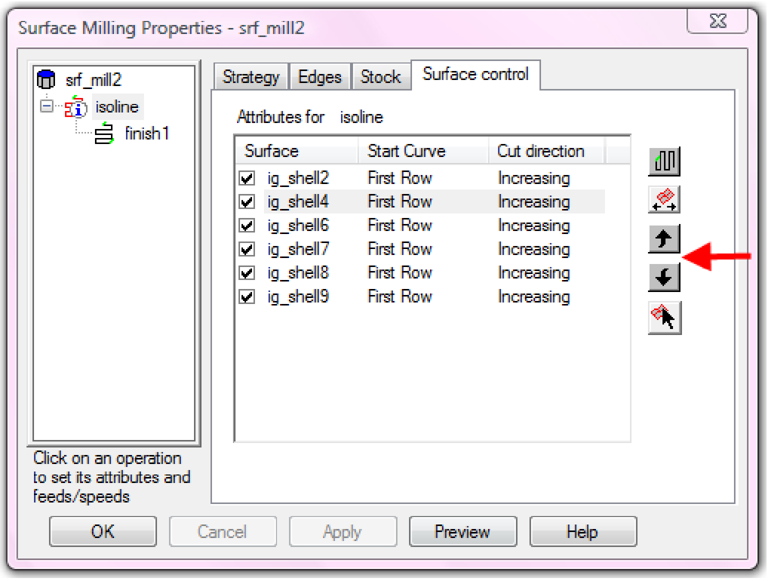
- Click **Apply**
- Press **Apply** and then **OK**
- **Uncheck** the operations **face1** and **srf_mill1** in the parts view

- Run a **Centerline simulation**



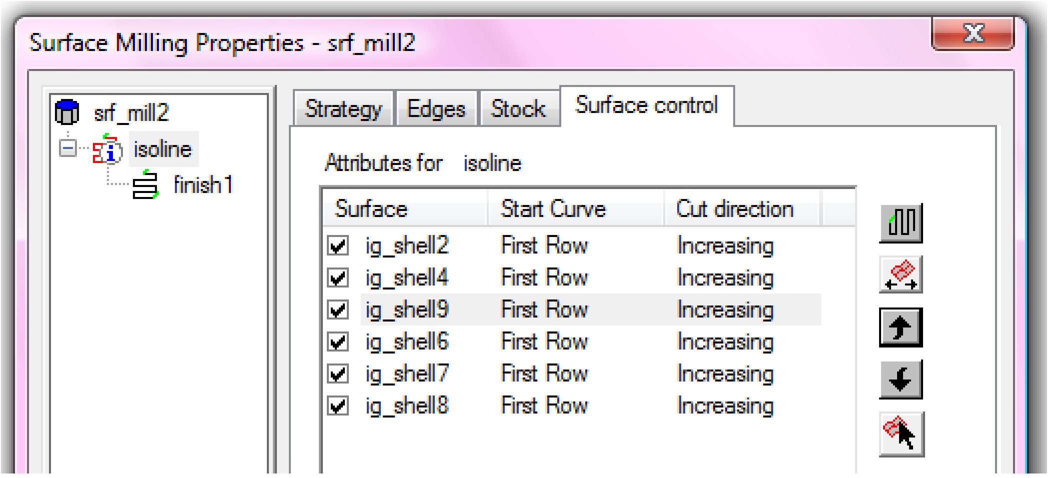
The toolpaths should appear as in the picture above. Note how each toolpath follow the shape of its individual surface. We will now modify the order in which the surfaces are machined and direction in which they are cut. This is being done in order to give better cutting conditions in the narrow sections and a better surface finish overall.

- Edit the feature
- Click on **Isoline** in the tree view and select the **Surface control tab**



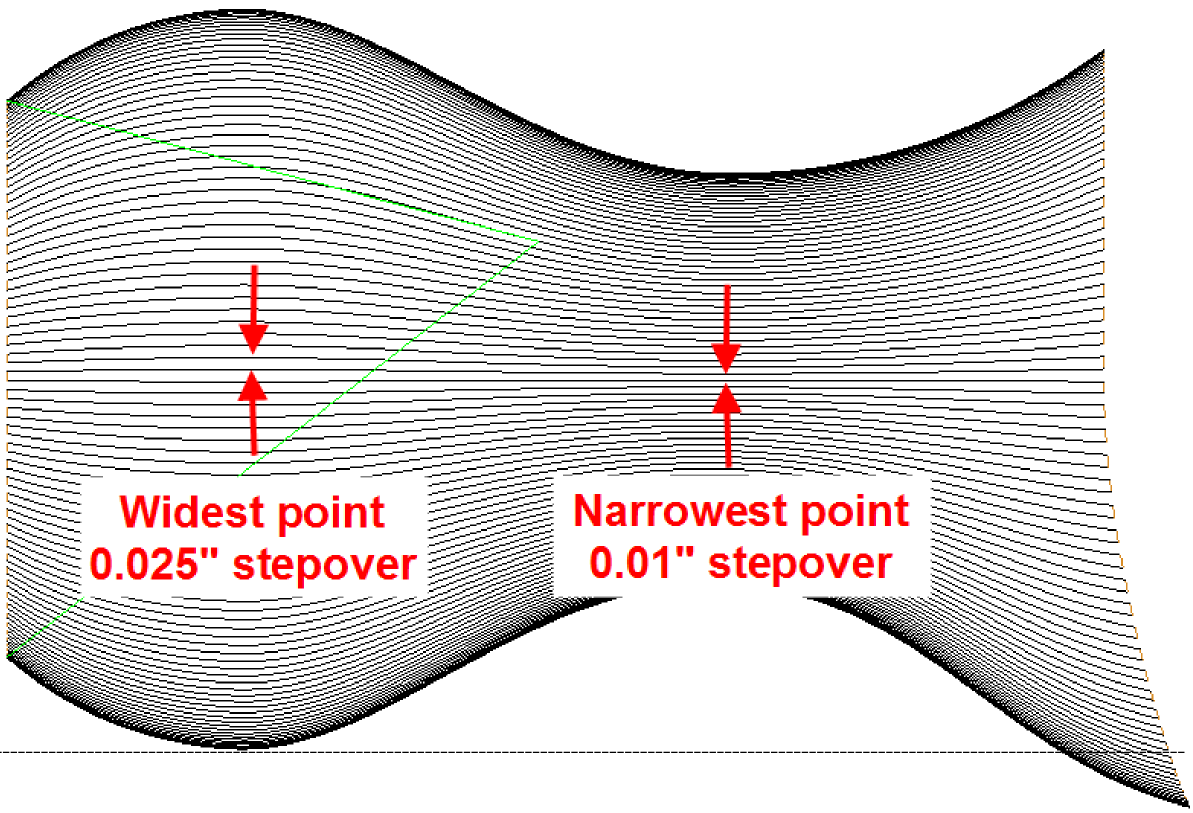
The surfaces are machined in the order in which they appear in the surface control list. Surfaces may be moved up or down the list by selecting them and then using the up and down arrows to the right of the list.

- Using this method, place the surfaces into the order shown below

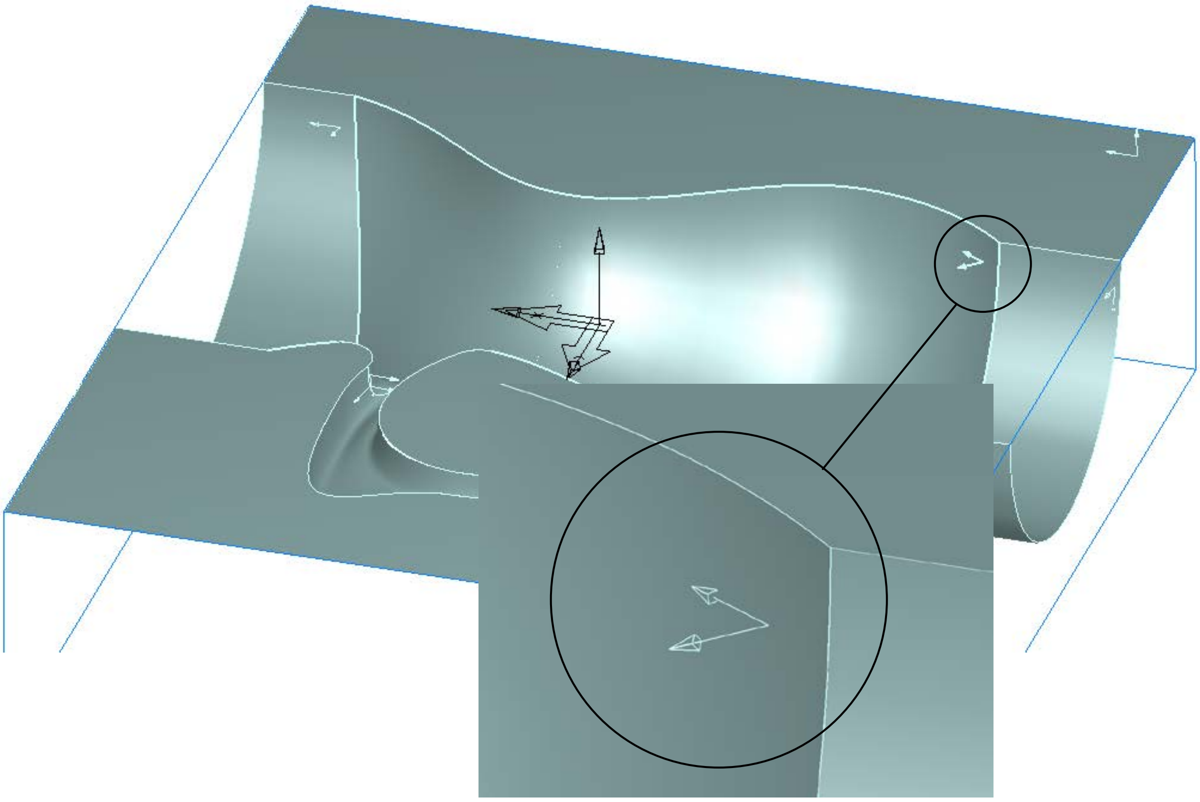


- Click **Apply** and then **OK**
- Run a **Centerline simulation** to see how the order in which the surfaces are machined has been changed

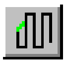
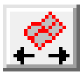
We will now consider how to get the best surface finish and shortest machining time by changing the cutting directions from the defaults. The **stepover** set in **Isoline** machining is the **maximum stepover**. Where a surface is getting wider and narrower, the stepover will apply at the widest point on the surface. As the surface narrows, so will the stepover. This may result in a longer than necessary toolpath and over machining where the isolines are very close together. For example on the jug body the stepover is varying between approximately 0.025” and 0.010”.

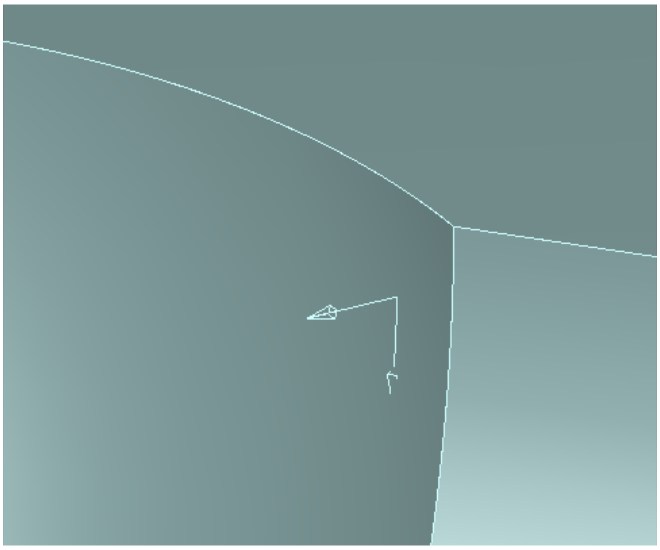


- Edit the feature
- Click on **Isoline** in the tree view and select the **Surface control tab**
- Click on **ig_shell9** in the surface control list



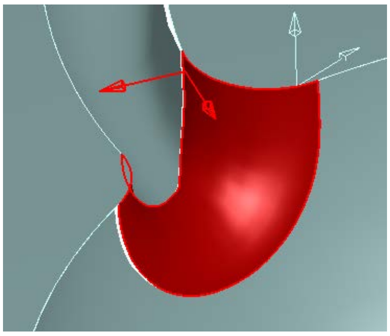
The pair of arrows on each of the surfaces shows the surface normal and the direction of the first cut. The surface normal arrow must point towards the side of the surface that you want to be machined. The direction arrow indicates whether the first cut is to be along the surface or across and which corner the pass will start at.

- Click on the **Set isoline row/col** button  and the Cut direction button  until the arrows on the jug body are aligned as shown below

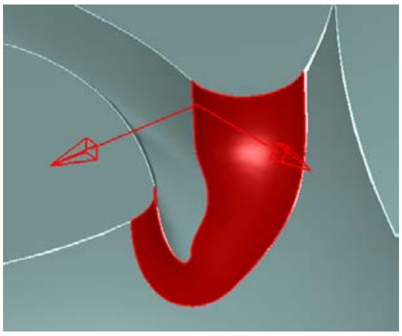


The first pass will now start in this corner and cut around the body of the jug instead of along. This will give a constant stepover.

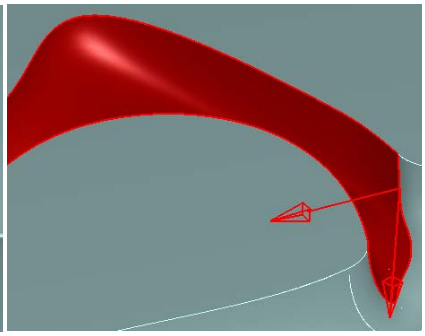
- Repeat this process for ig_shell6, ig_shell7 and ig_shell8 aligning the cutting directions as shown below



ig_shell6

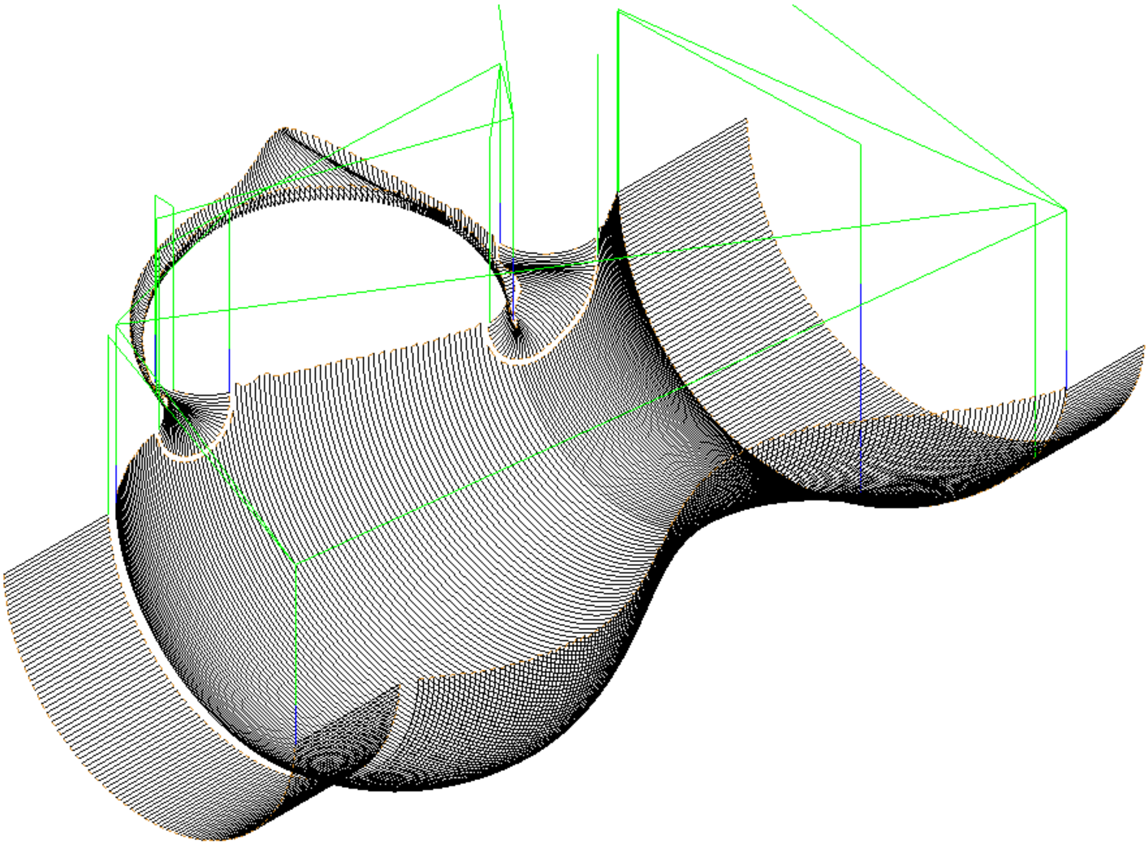


ig_shell7

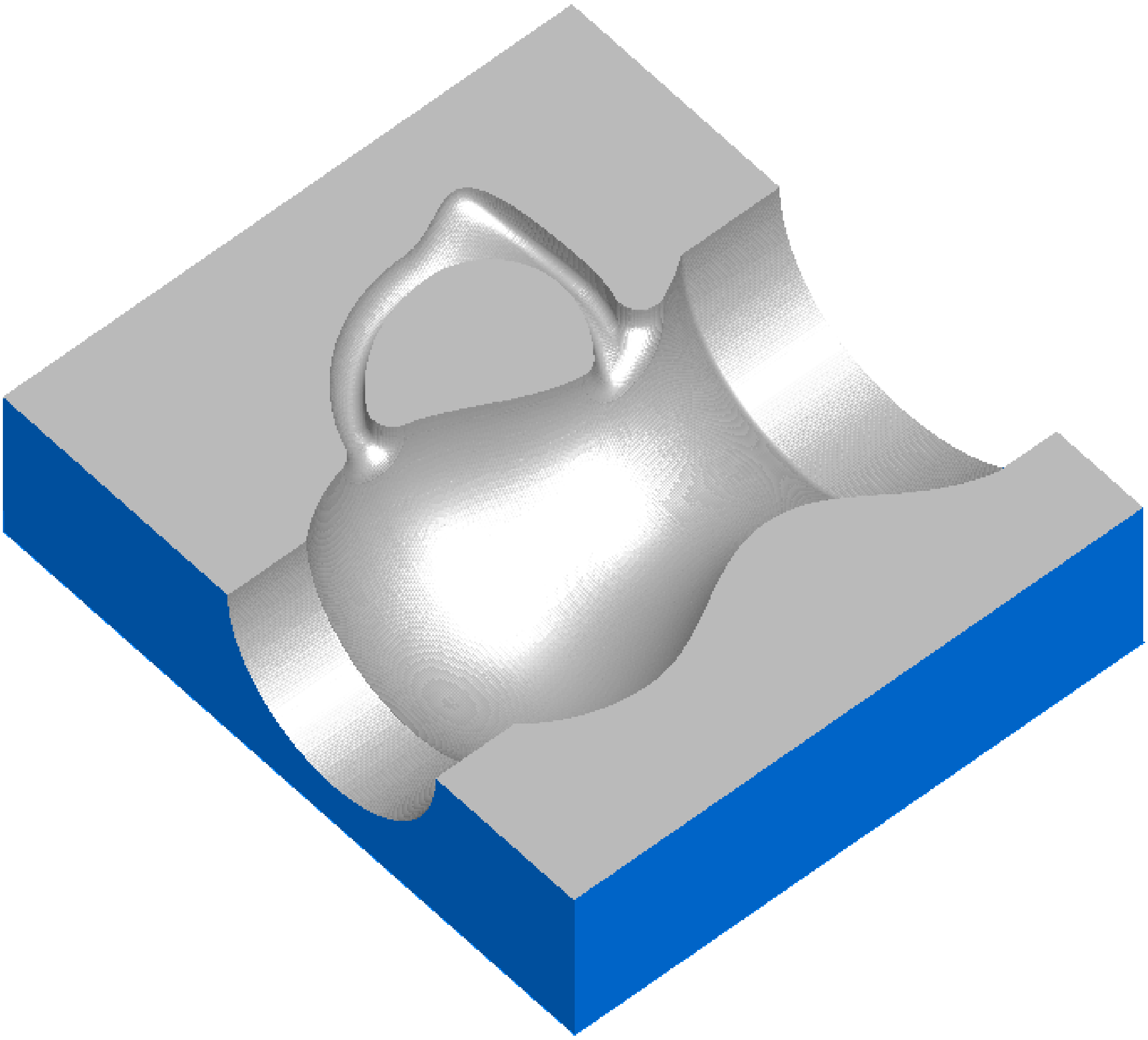


ig_shell8

- Press **Apply** and then **OK**
- Run a **Centerline simulation** to see the new cutting strategy



- Check **face1** and **srf_mill 1** in the part view
- Run a **3D simulation** to see the surface finish

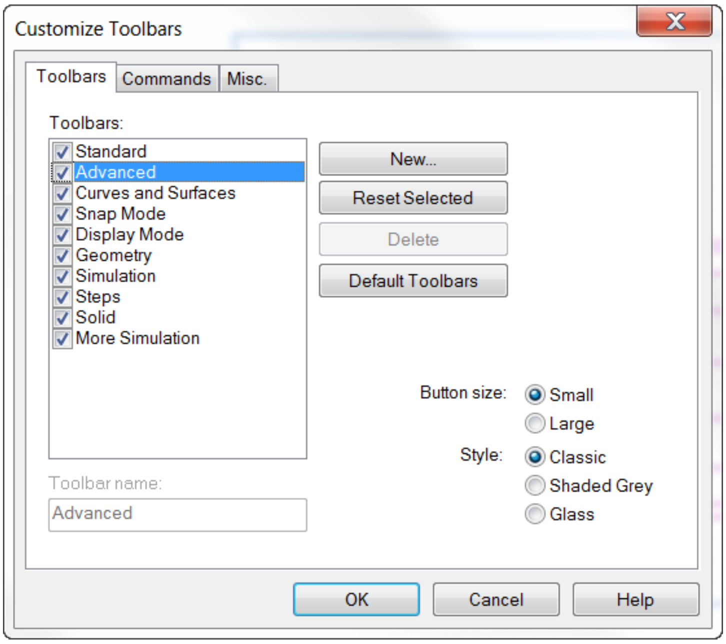


The surfaces are now being cut in the specified order and direction to give a good surface finish. You will note however that in some areas of the model the tool is cutting from the bottom up towards steep or vertical surfaces. We can remedy this by modifying the model by un-trimming and breaking surfaces into smaller pieces to improve the cutting conditions.

We will now look briefly a one of the surfaces to see how this can be achieved.

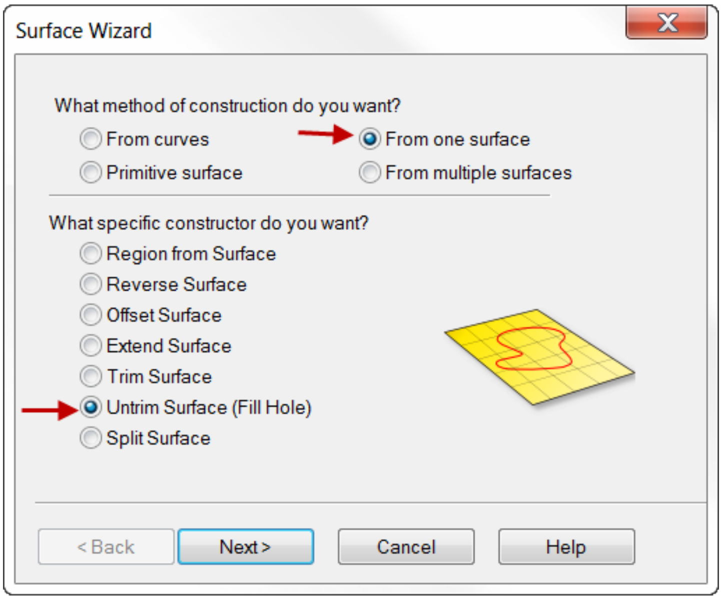
- Eject the simulation
- Delete the feature **srf_mill 2** in the part view

- Select the surface that makes up the body of the jug, **ig_shell9**
- Hide everything else (use the **Ctrl, Shift and U** keys together)
- From the **View menu** select **Toolbars**
- Check the **Advanced** toolbar

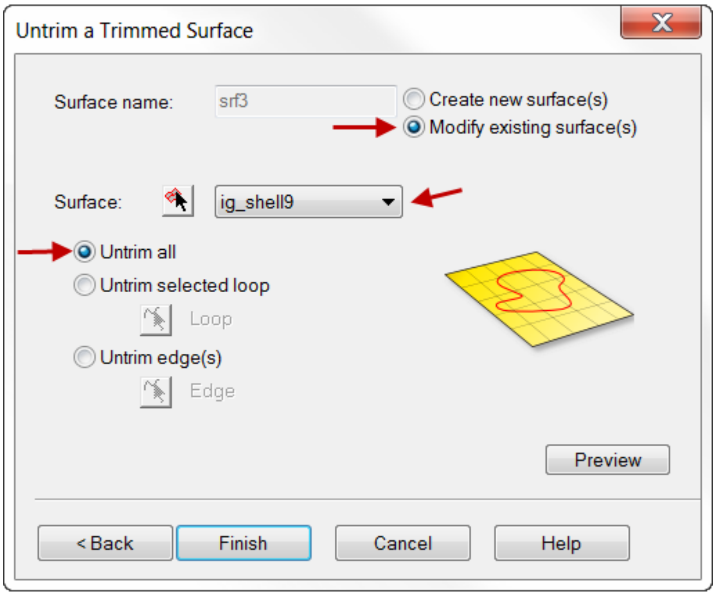


This raises the Advanced toolbar from which you can access the surface wizard.

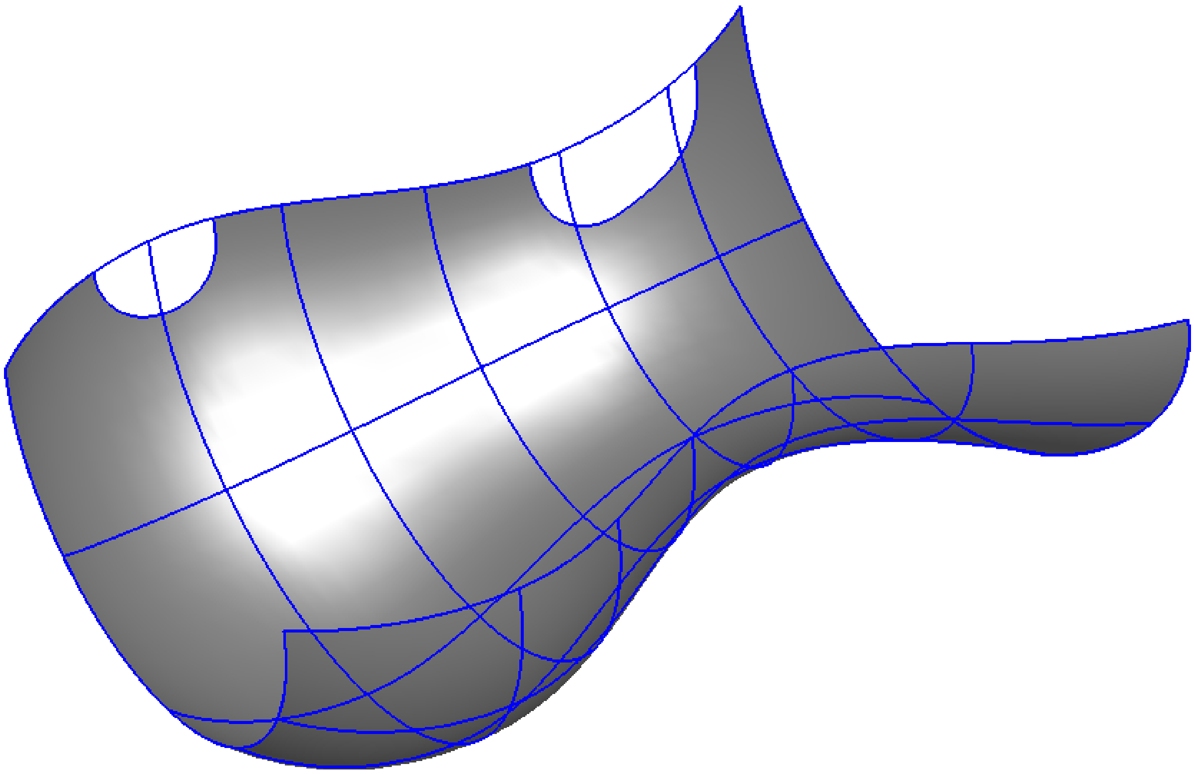
- Click **OK**
- Click on the **Surface wizard**
- Check **From one surface**
- Check **Untrim surface**
- Click **Next**



- Fill in the form as shown making sure that the correct surface is selected in the pulldown menu



- Click **Preview**

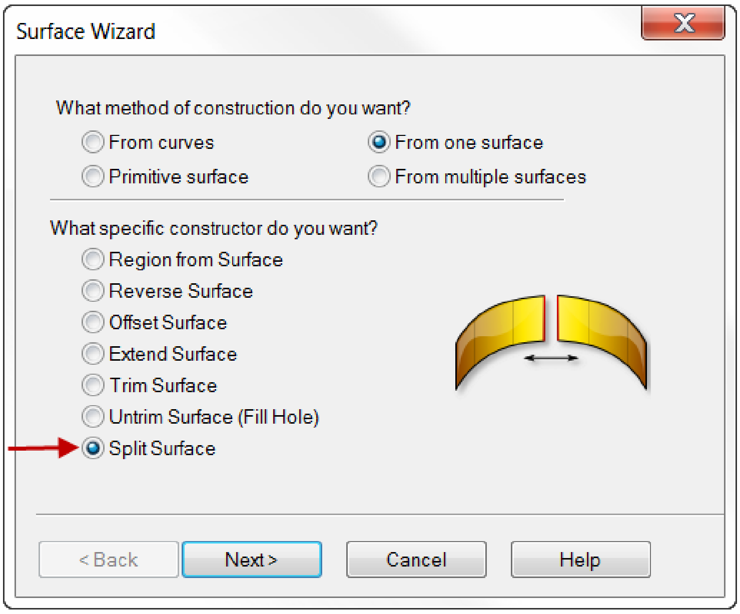


FeatureCAM shows you a wireframe of how the surface will look after the trimming is removed. Note how the two cutaways where the fillets meet the body of the jug will be filled in. This will allow us to machine along the jug without getting multiple retracts.

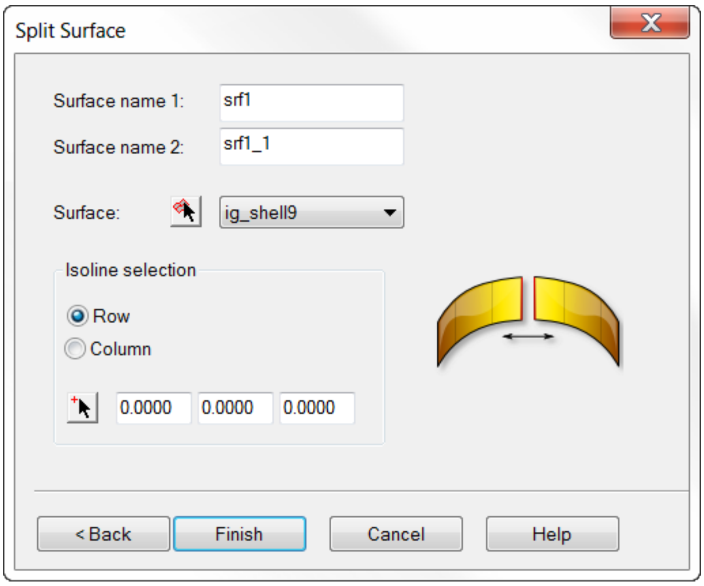
- Click **Finish**

Now that we have untrimmed the jug body the next step is to split it into two separate surfaces at its lowest point. This will allow us to machine each of the surfaces individually in a top down direction.

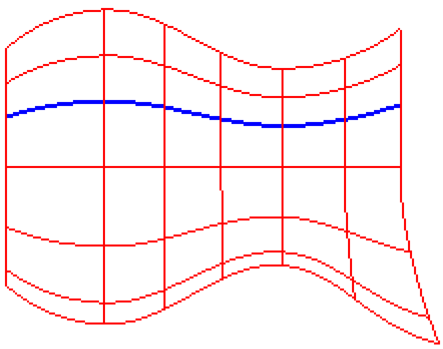
- Click on the **Surface wizard**
- Check **From one surface**
- Check **Split surface**



- Click **Next**
- Fill in the form as shown making sure that the correct surface is selected in the pulldown menu

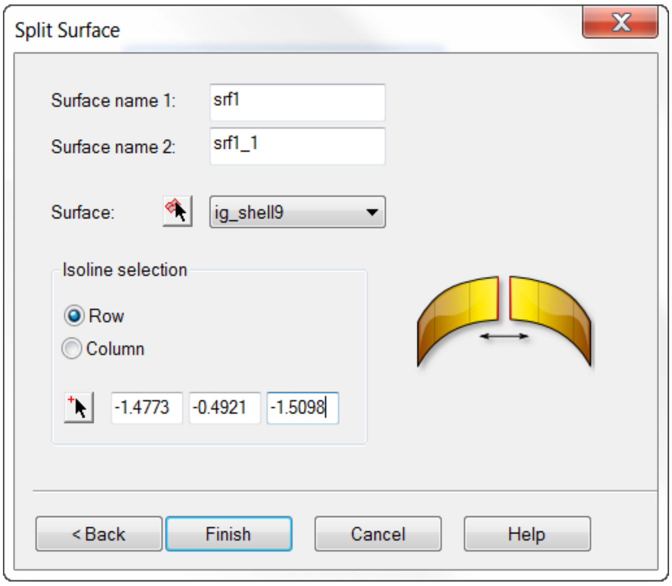


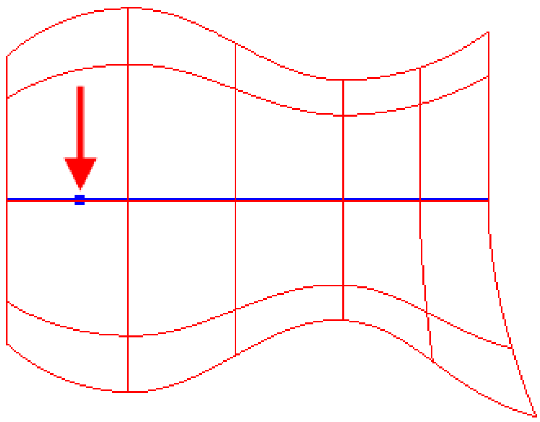
Note: as you check either Row or Column a blue curve will appear on the surface indicating the direction it will be split along as shown below.



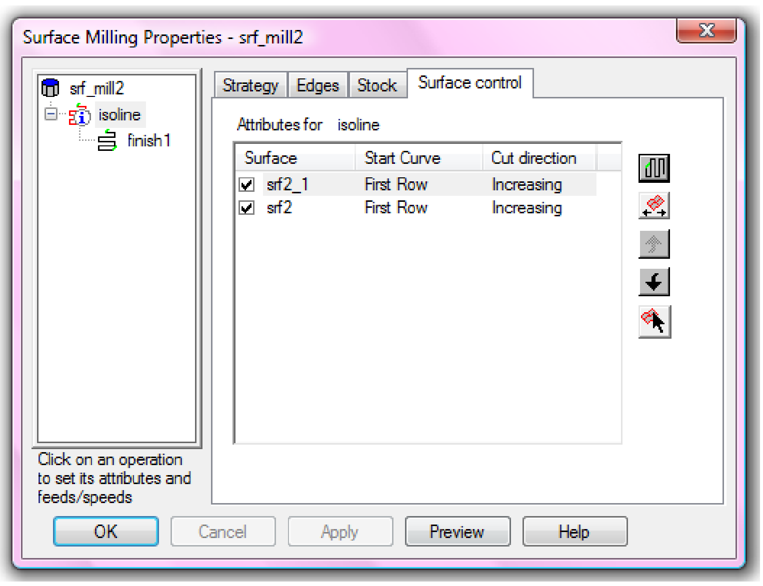
We now need to indicate the particular point at which we want to split the surface. We do this by selecting an isoline position. If we select an existing isoline then the surface will split there. If we select in between isolines then FeatureCAM will first insert a new interpolated isoline at the indicated position and then split the surface along the new curve.

- Take a **Top view**
- Click on the **Pick point button** at the lower left of the form
- Pick the centerline of the jug as indicated

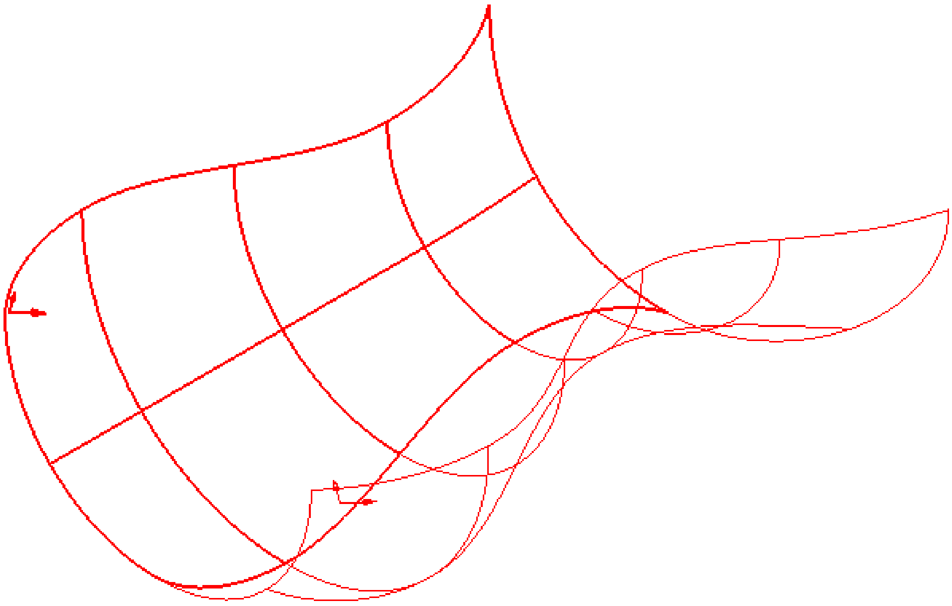




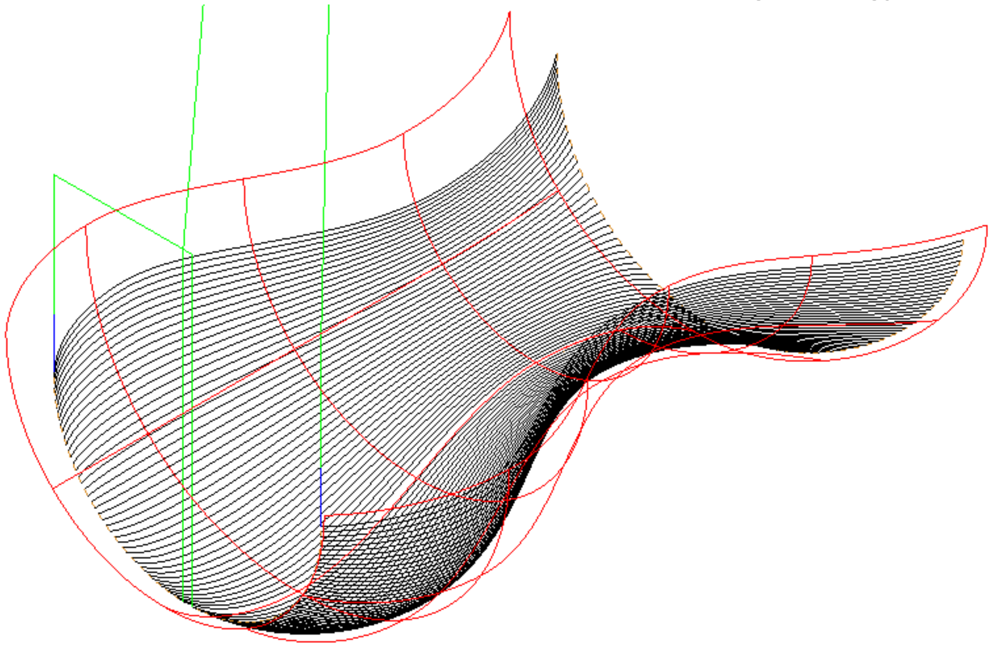
- Click **Finish**
- **Hide** the surface **ig_shell9** and then select the two new surfaces
- Create a new **surface milling feature**
- Check **Choose a single operation** and click **Next**
- Check **Isoline** and click **Finish**
- Click on **Isoline** in the tree view and select the **Surface control tab**



- Click on the **Set isoline row/col** button and the Cut direction button until the arrows on the jug body are aligned as shown below



- Press **Apply** and then **OK**
- Uncheck **face1** and **srf_mill 1** in the part view
- Run a **Centerline simulation** to see the new cutting strategy



The tool is now starting on the top edge of the jug body, cutting along the part and stepping over from the top towards the bottom.

- As an exercise use Untrim surface and Split surface to prepare the rest of the model for optimum cutting conditions and minimum number of retracts
- Once this is complete prepare further Isoline toolpaths to finish the part

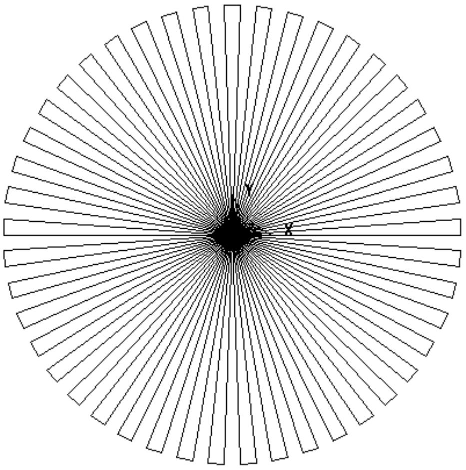
Radial and 2D Spiral Finishing

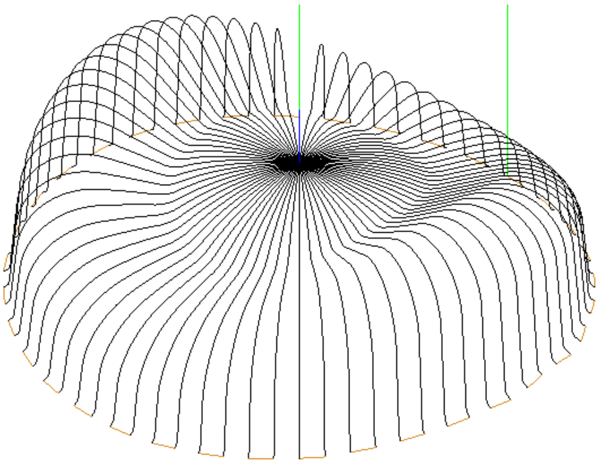
Introduction

FeatureCAM has three strategies which are created by the **downward projection of a pattern**; these are **Parallel, Radial, and Spiral**. As we have already looked at Parallel in some detail, this module will just deal with Radial and 2D Spiral.

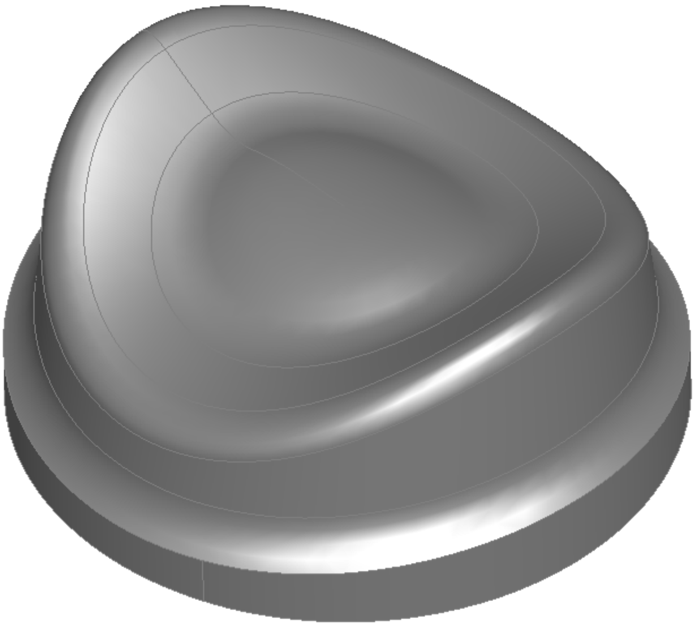
Radial Finish Machining

In radial finishing, the pattern from which the toolpath is created consists of a series of straight lines which radiate out from a center point. The passes are equally spaced at a user-defined angle. The image below shows a radial pattern viewed from above and the resulting toolpath after projection onto a set of surfaces.





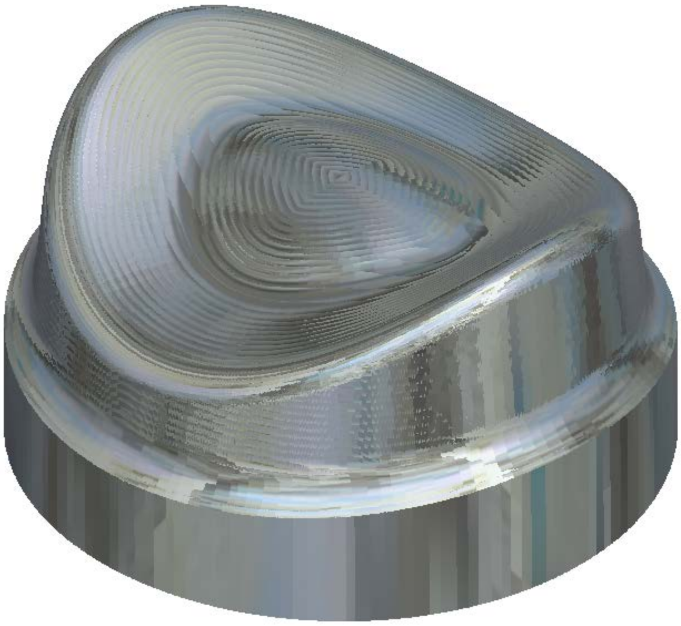
- **Open** the file **Doofa.fm**
- Select an **Isometric view**



One advantage of pattern machining strategies over others is that they are quicker to calculate. This is because FeatureCAM already knows the 2D (XY) path that the tool center is going to follow; it then just has to calculate the required Z height to keep the tool in contact with the part.

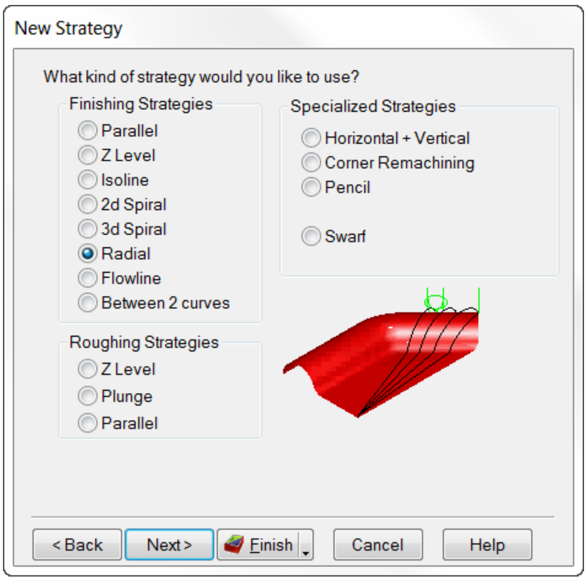
The file already contains a Z Level roughing and an Horizontal & Vertical finishing toolpath. We will now run a 3D RapidCut simulation to see the result.

- Run a **3D RapidCut simulation**



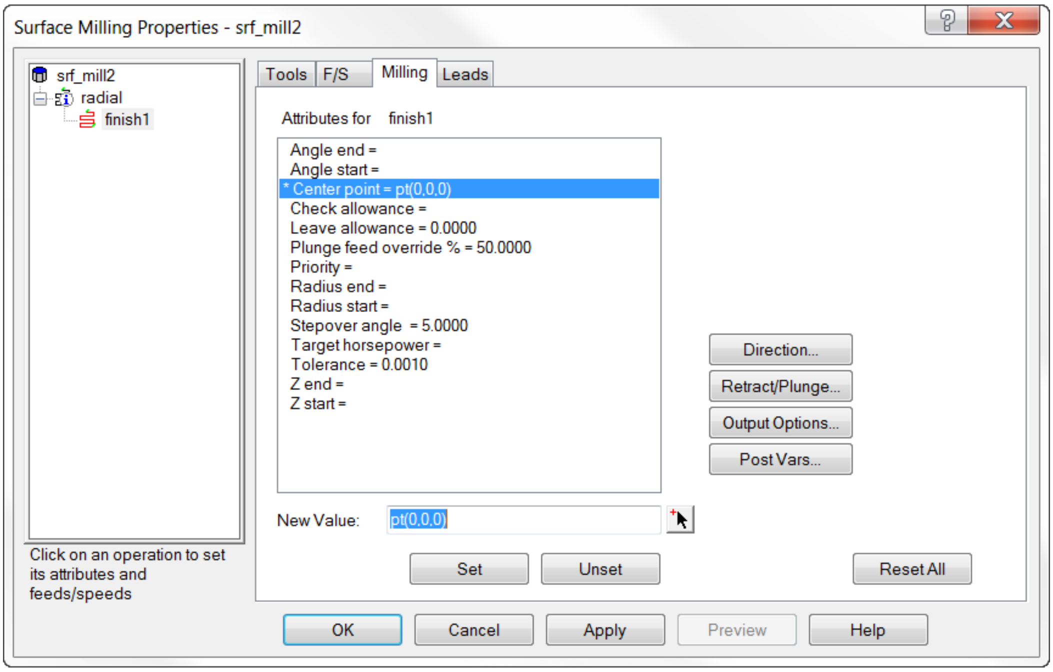
The part has been finished with a 0.025” stepover. It can be seen that the machining marks on the surface are fairly obvious so although the part has been machined accurately it may not be aesthetically pleasing. We will now look at using a radial toolpath to provide a better finish on the upper surfaces.

- **Eject** the **simulation**
- **Delete** the feature **srf_mill1**, or simply Uncheck it in the Part View.
- Box-select all the faces and create a new **surface milling feature**
- **Choose a single operation**
- On the New strategy page select **Radial**
- Click Finish

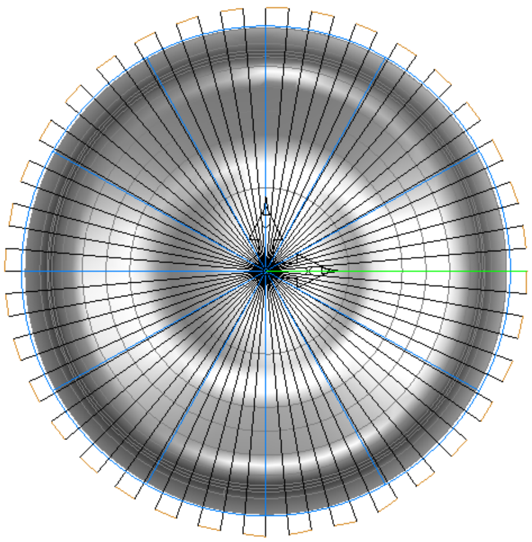


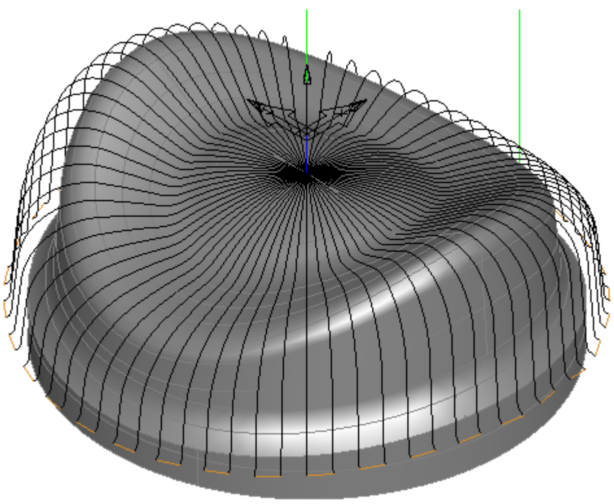
- Click on the **finish1** in the tree view and select the **Milling tab**
- Set the **Center point** to **pt(0,0,0)** as shown

(You can do this either by typing or using the pick button and picking off the model).



- Click **Apply** then **Preview**
- Play the simulation

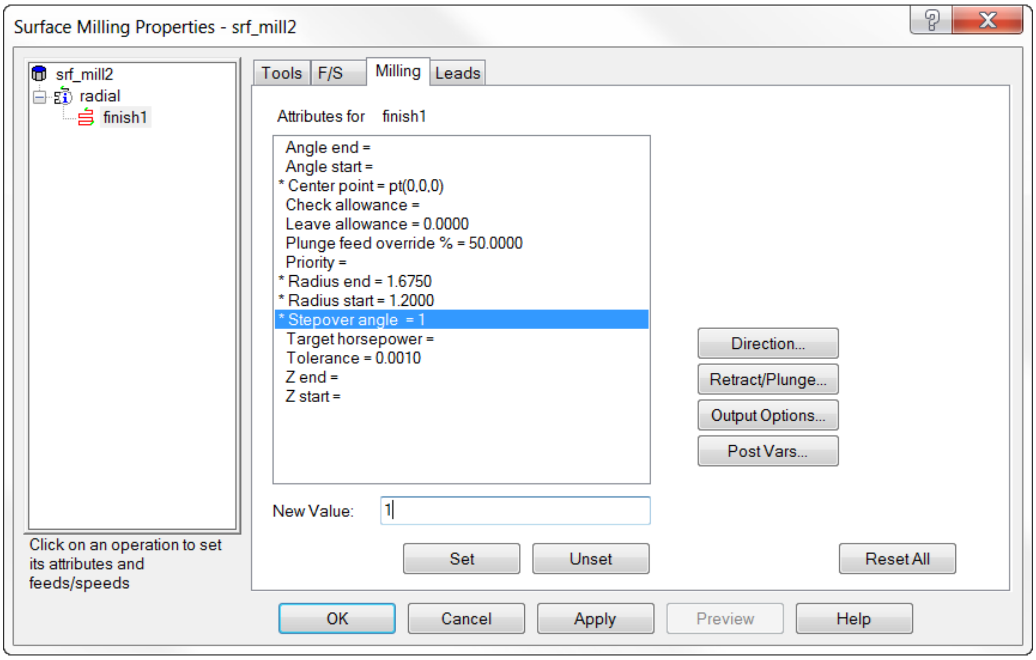




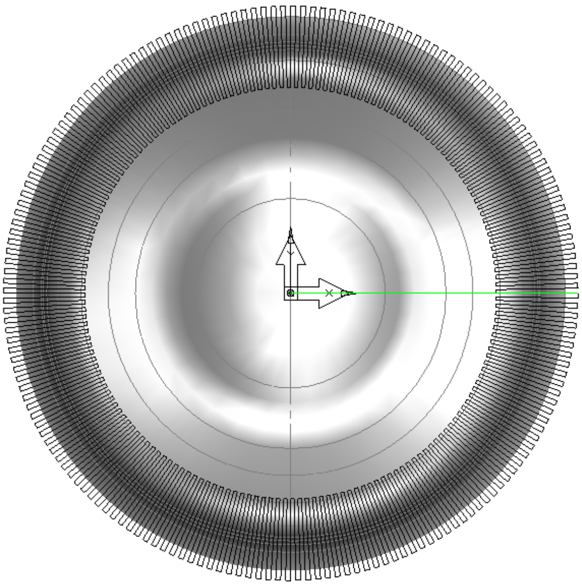
You will notice a number of things in the simulation. In the left image above, you can see that the toolpaths are far apart at the outside of the model and close together in the center. This would give a poor surface finish on the outside edges and possibly overmachine the center. In the right view, you can see that the tool is machining too far down the side of the part, we want to machine just the upper, less steep surfaces.

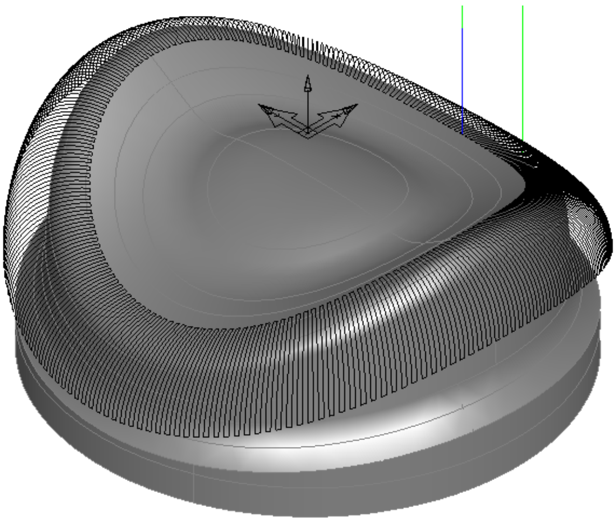
We will now limit the area to be machined by setting an inside and outside radius limit on the toolpath. This will leave an area in the center of the part which is not machined; we will come back and machine it using a different strategy later.

- **Eject** the **simulation**
- **Edit** the feature **srf_mill2**
- Click on **finish1** and select the **Milling tab**
- Set the **Radius start** to **1.2"** and the **Radius end** to **1.675"**
- Change the **Stepover angle** to **1 degree**



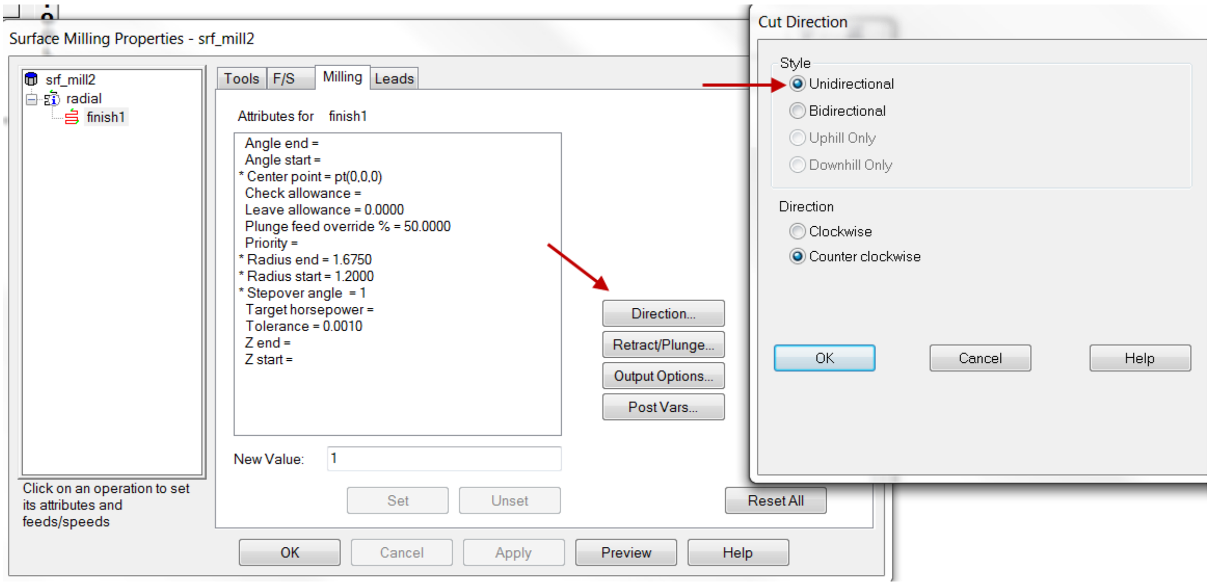
- Click **Apply** and then **OK**
- **Preview** these changes.





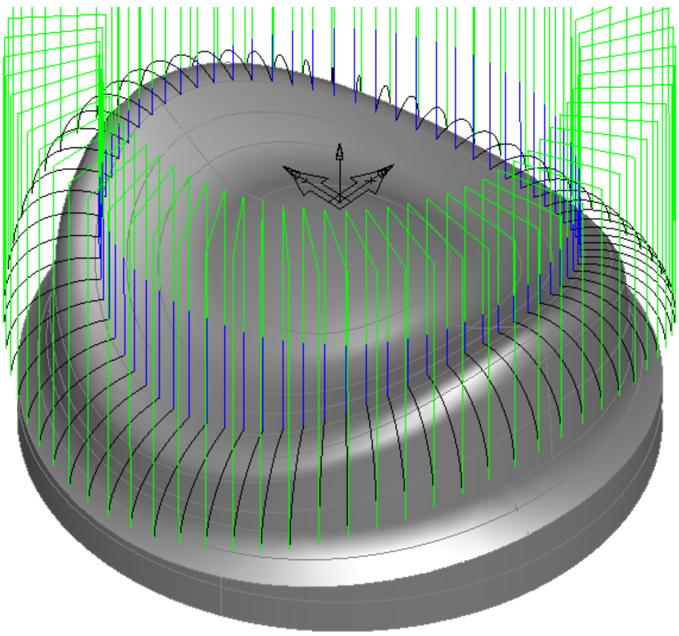
The toolpath is now restricted to the area between the Radius start and the Radius end. The first pass will start at the Radius start and cut towards the Radius end.
If the toolpath is set cut uni-directionally, this allows us to control whether the tool cuts outside in or inside out. For example, if the **Radius start** is **larger** than the **Radius end,** the tool will start at the **outside** of the part and cut **inwards**.

- **Eject** the **Simulation**
- Edit the feature
- On the **Milling** tab click **Direction**
- Select **Unidirectional**



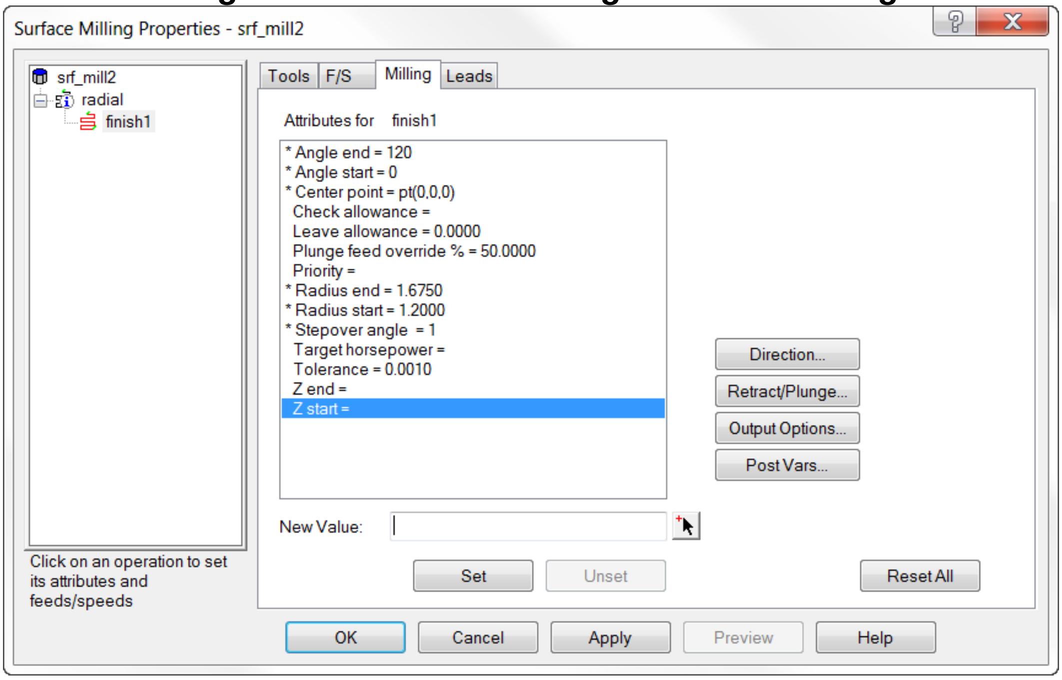
Note that you can also control whether the tool passes go around the part in a clockwise or counter clockwise direction

- Click **OK**, then **Apply** and then **OK**
- Run a **Centerline simulation**

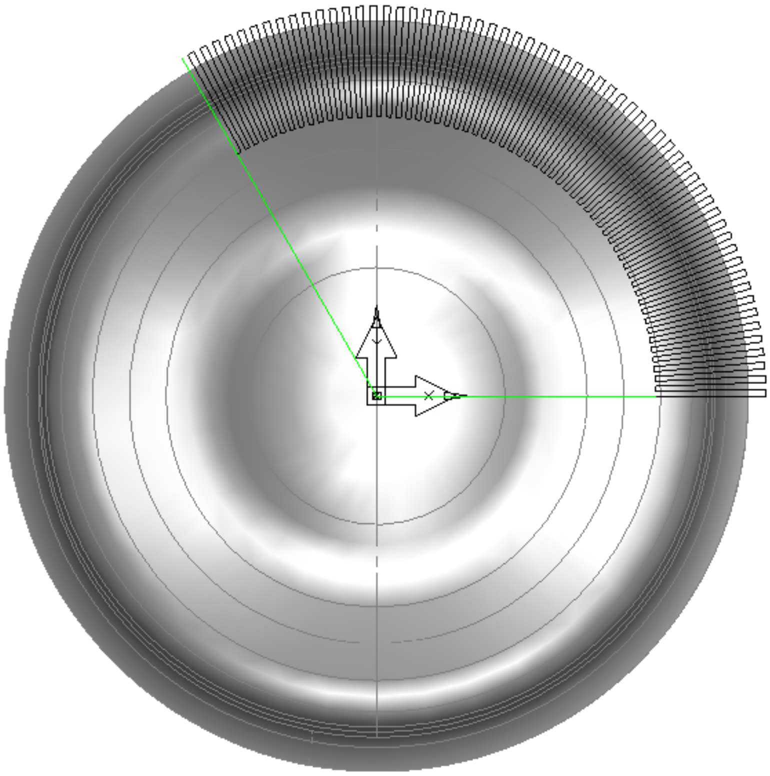


The tool is now cutting the part in a single direction, plunging at the **Radius start** and then cutting in an outward direction to the **Radius end**.

- **Eject** the **Simulation**
- Edit the feature
- On the **Milling** tab click **Direction**
- Select **Bidirectional** and click **OK**
- Set the **Angle start** to **0** and the **Angle end** to **120 degrees**



- Click **Apply** and then **OK**
- Run a **Centerline simulation**

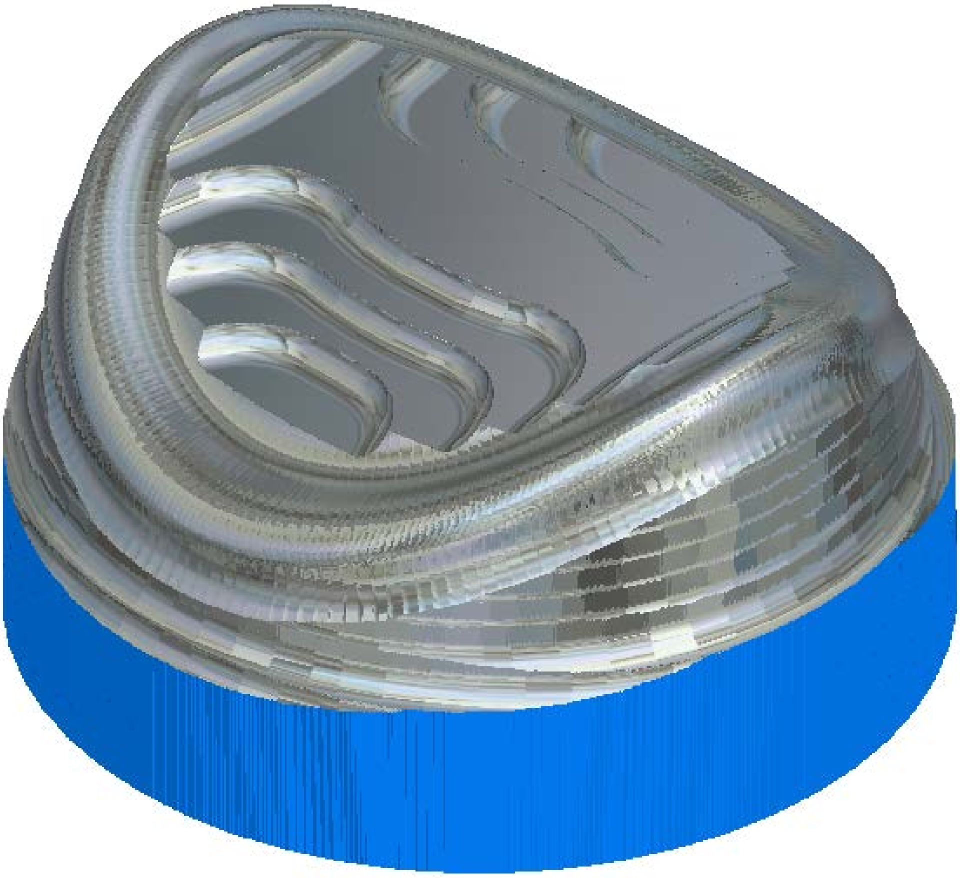


The tool is starting at an angle of zero (along the X axis) and cutting around the part until it reaches an angle of 120 degrees in the XY plane.

- **Eject** the **Simulation**
- Edit the feature
- Select the **Milling** tab
- Set the **Angle start** to **120 degrees** and the **Angle end** to **Zero**
- Click **Apply** and then **OK**
- Run a **Centerline simulation**

Note how the tool now starts at the 120 degree position and then works back to the X axis.

- **Eject** the **Simulation**
- Edit the feature
- Select the **Milling** tab
- Unset the **Angle start** and the **Angle end**
- Click **Apply** and then **OK**
- **Check** the **roughing** operation in the part view
- Run a **3D RapidCut simulation**

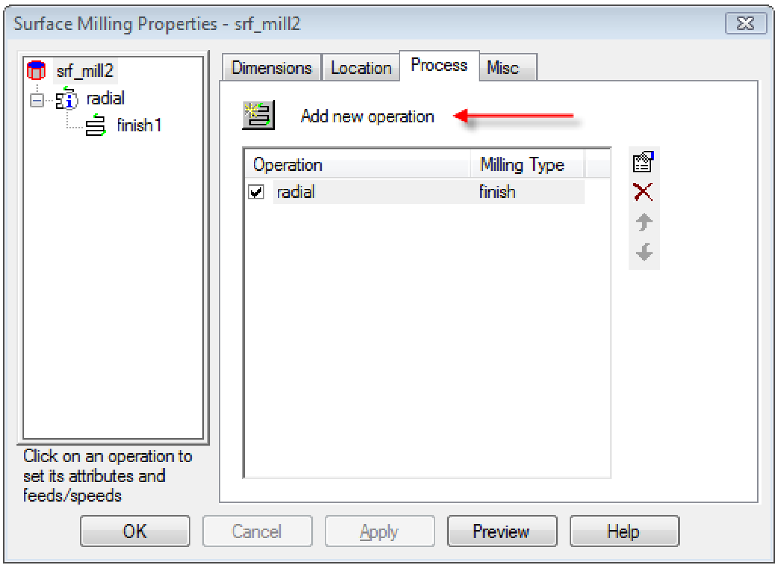


The toolpath is machining across the fillet and surrounding surfaces giving a superior surface finish to that we were originally getting with the Horizontal and Vertical finishing toolpath. We will now machine the center portion of the part using a different technique.

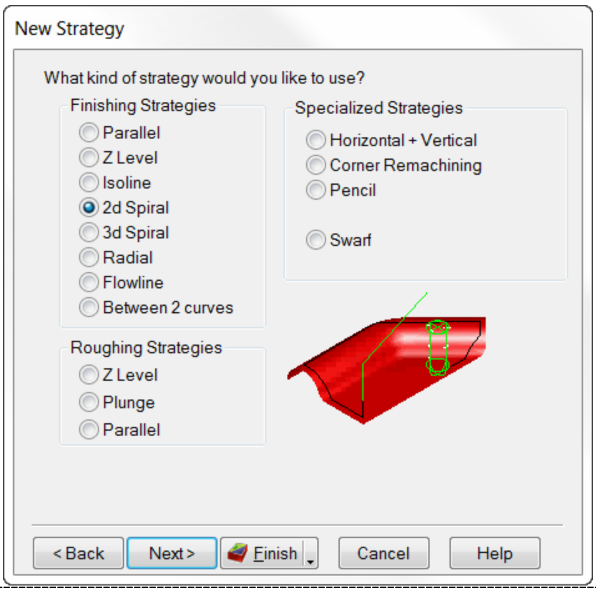
2D Spiral Finishing

We will be using a 2D spiral toolpath next. This is best suited to surfaces with a fairly constant slope as it uses a 2D stepover. On surfaces with a combination of steep and shallow surfaces 2D Spiral will give a poor finish on the steep areas. We will be using a circle as a boundary, this will be created next.

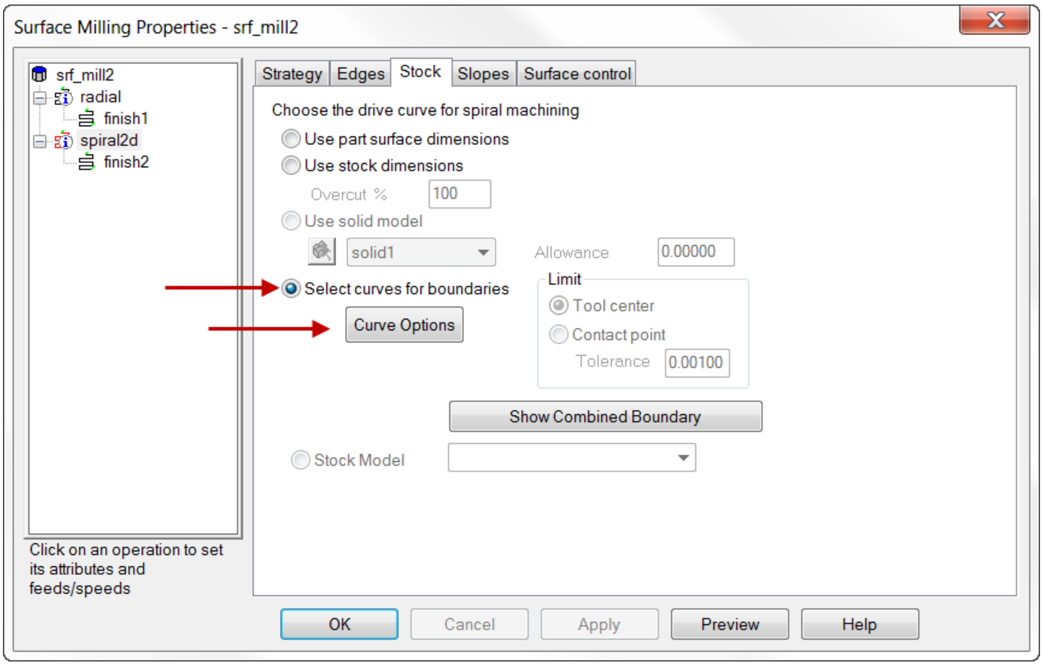
- Eject the simulation
- Create a **Circle** with a **diameter** of **1.25** inches at the origin **(0,0,0)**
- Double click on the feature **srf_mill2** to open its properties
- Select the **Process tab**
- Click **Add new operation**



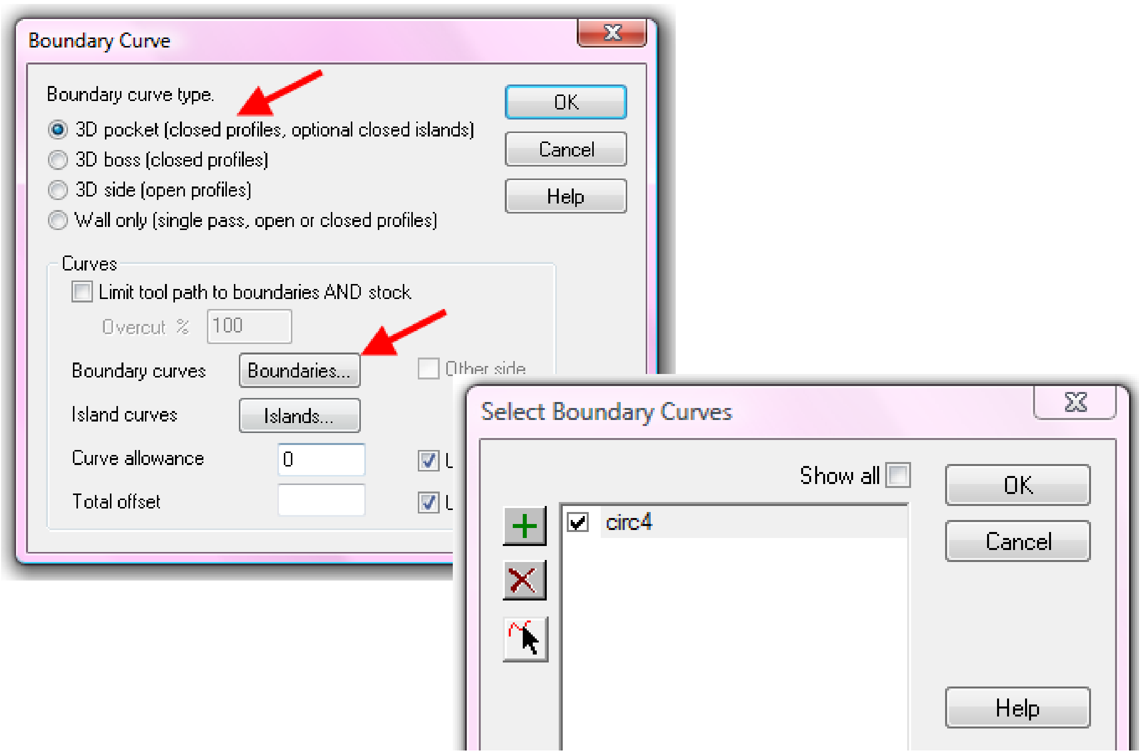
- Select a **2D Spiral** strategy and click **Finish**



- Select the **spiral2d operation** and then the **Stock tab**
- Check **Select curves for boundaries** then click **Curve options**

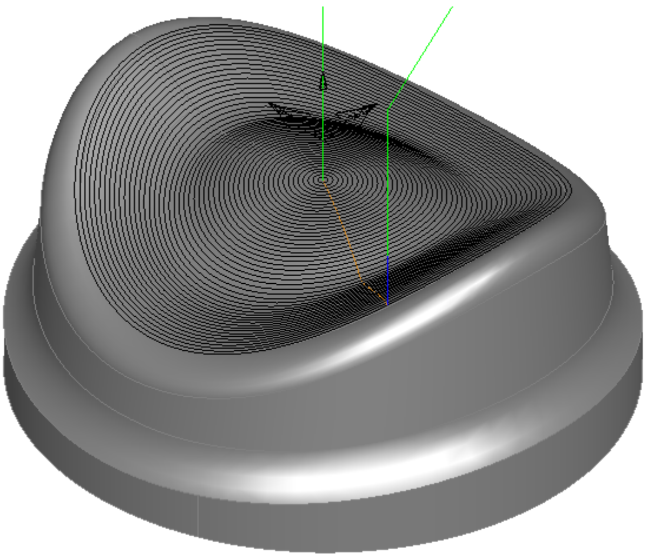


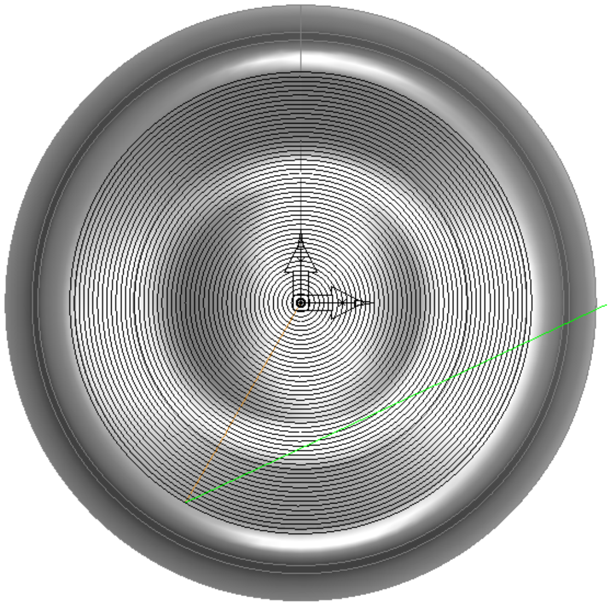
- Check **3D pocket** then click on **Boundaries**



- Select the circle **circ4** and then click **OK**, then **OK** again (your circle number could be different)

- Select **finish2** and then the **Milling tab**
- Change the **Stepover** to **0.025”**
- Click **Apply** and then **Preview**
- **Run** the **simulation**

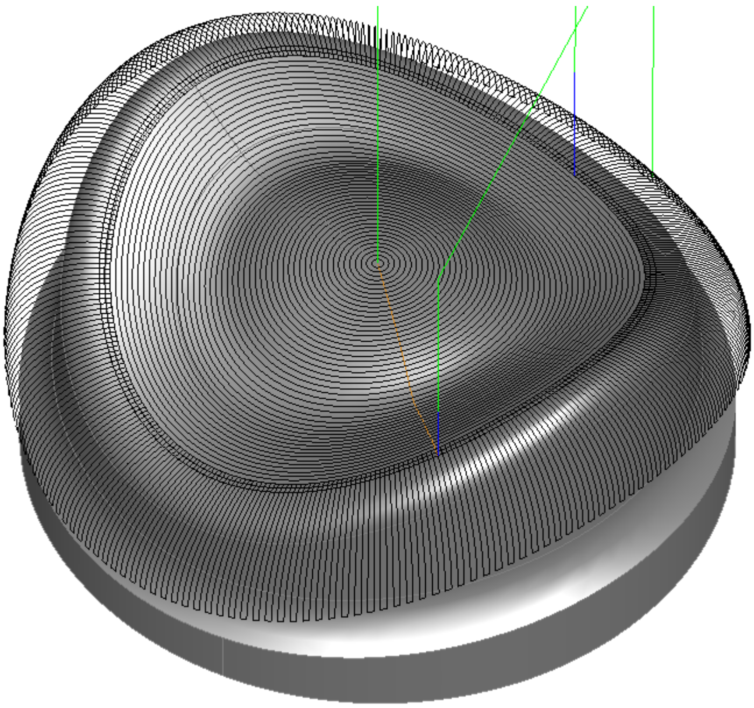




The circle is offset repeatedly by the specified 0.025” stepover to create a pattern of concentric circles. This pattern is then projected down on to the part to make the toolpath.

- **Eject** the **simulation**
- Run a **Centerline simulation**

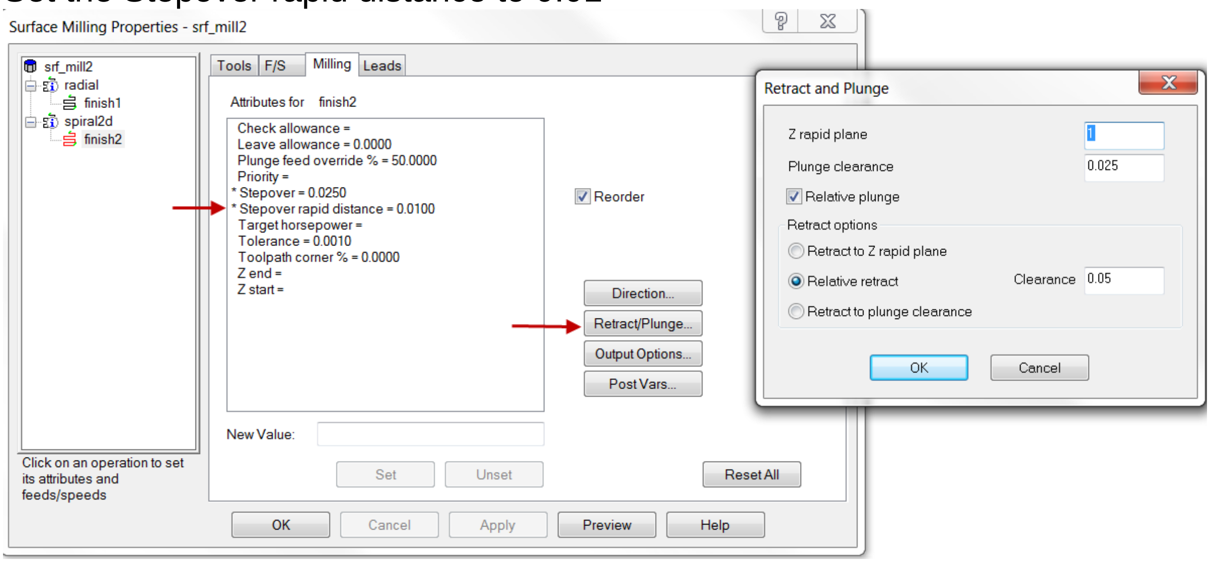
The two toolpaths overlap by 0.05”, we set this by the Start radius of the radial toolpath and the size of the circle we used for the 2D Spiral.



- **Turn on** the **roughing** operation
- Run a **3D RapidCut simulation**

The surface finish is generally better than that which we originally got with Horizontal and Vertical finishing. There are a couple of areas where we could make further improvements. Firstly, there is a witness mark where the tool is stepping between each pass of the 2D Spiral. Secondly, in the trough around the center, there are some undesirable marks where the contact point is changing from one side of the tool to the other.

- **Eject** the **simulation** and edit the **2D Spiral** feature
- Set the Stepover rapid distance to 0.01"



- Use a relative plunge and retract as shown

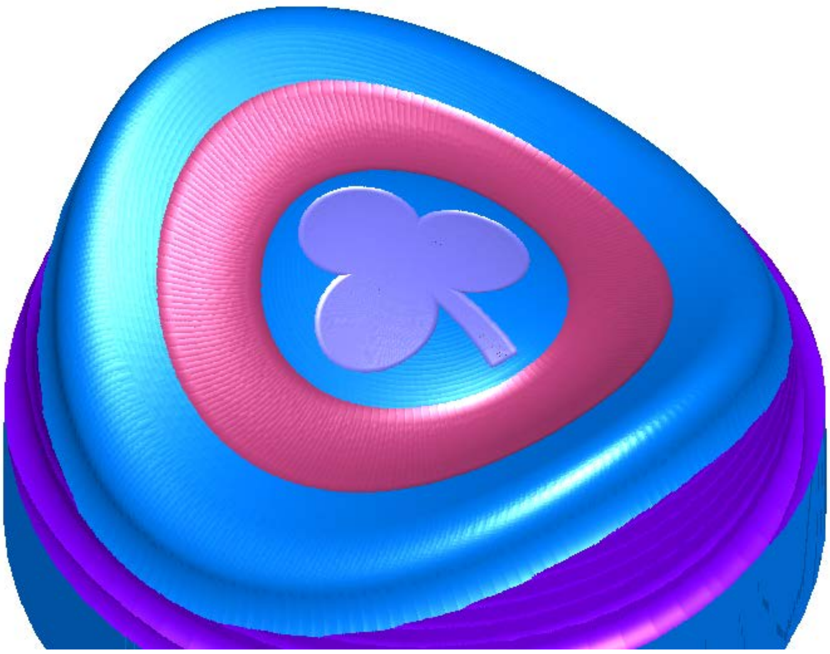
By setting the Stepover rapid distance smaller than the toolpath stepover we are forcing the tool to lift off at the end of every pass. The relative plunge and retract minimize the amount of time spent in the air.

- As an exercise, create a new radial toolpath with a start radius of 0.6, end radius 0.95, angle stepover 1.5 degrees using a 0.25" ball nose cutter.

- Run a **3D RapidCut simulation**

By causing the tool to lift off between passes we have removed the witness marks and the second radial toolpath has improved the finish in the trough area.

- As a final exercise, import the file **Clover.igs** and use it to engrave onto the centre of the part as shown with a recess around it



Hint: Use a 2D Spiral toolpath with a small cutter and a *negative leave allowance* for the clover leaf.

Modify the existing, second radial toolpath to produce the ditch.

If you get stuck take a look at the file:

Doofa_Finished.fm

Flowline Finish Machining

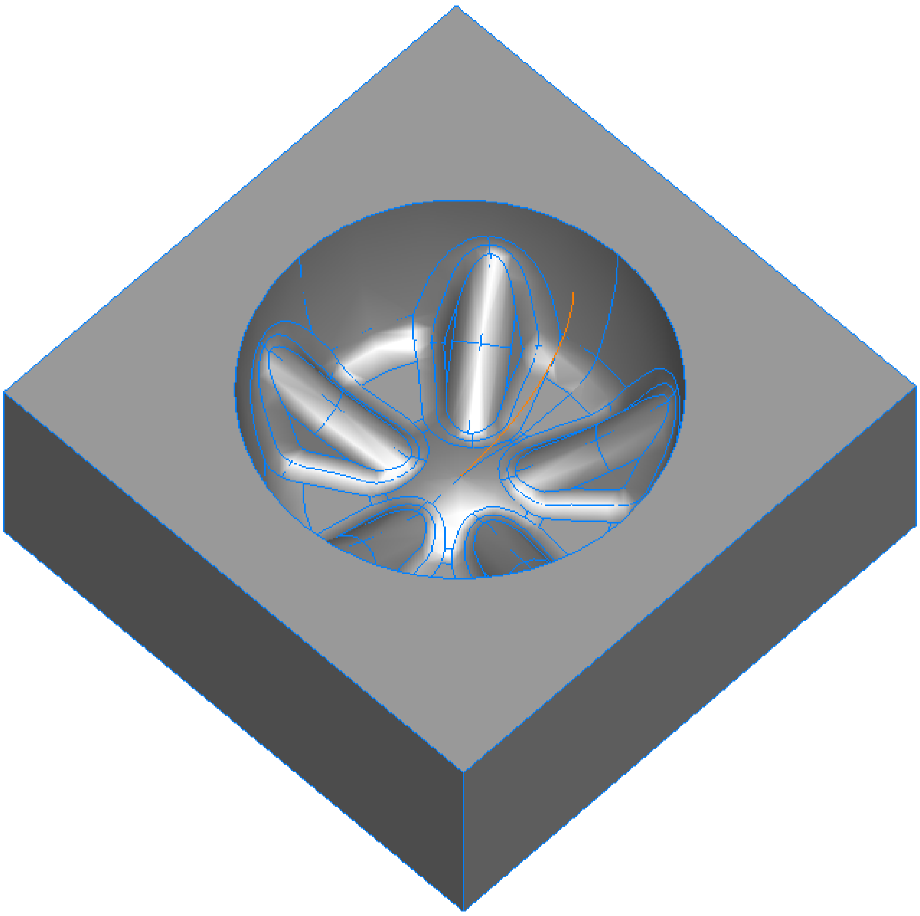
Introduction

In this module you will be introduced to the Flowline machining strategy. In flowline a guide surface is used to produce toolpaths across multiple surfaces. Internally FeatureCAM produces an Isoline toolpath on the guide surface, this it then projected along the guide surface normals onto the surfaces that are to be machined. The flowline technique has many applications. As long as you can create a single surface that mimics that shape of the toolpaths you desire, flowline toolpaths are applicable.

Flowline Example

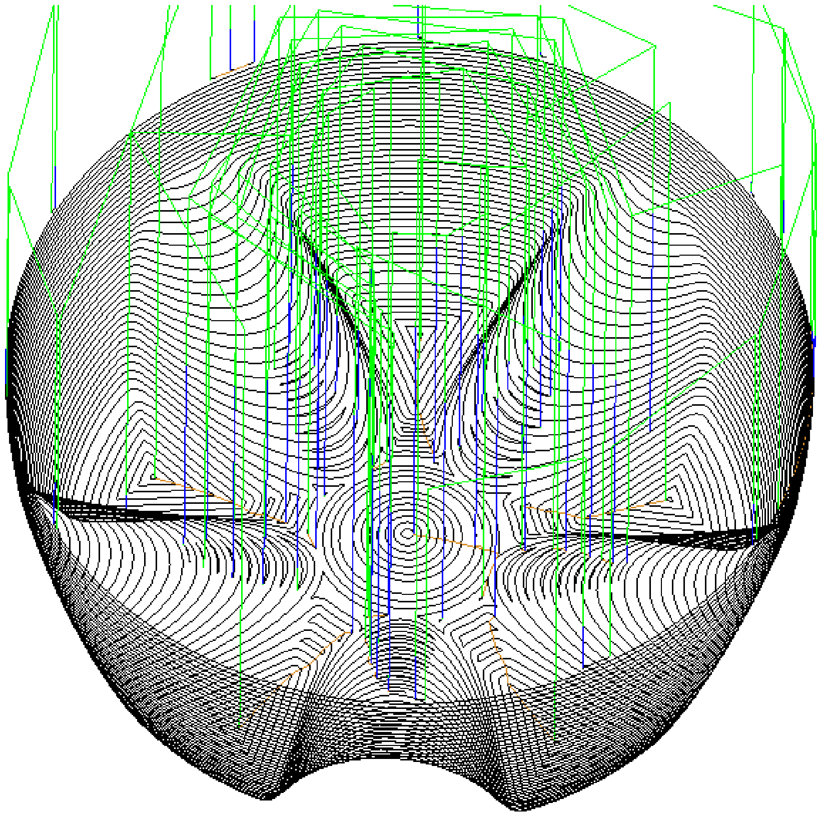
This model shown below is the bottom of a blow mold for a soda bottle.

- Open the part **Blow_Mold.fm**
- Select the tool crib **Blow_Mold.fm_Tools_from_last_save**
- Select an **Isometric view**



The inner part of the mold contains many surfaces. We want to create a toolpath which will smoothly machine over all of them.

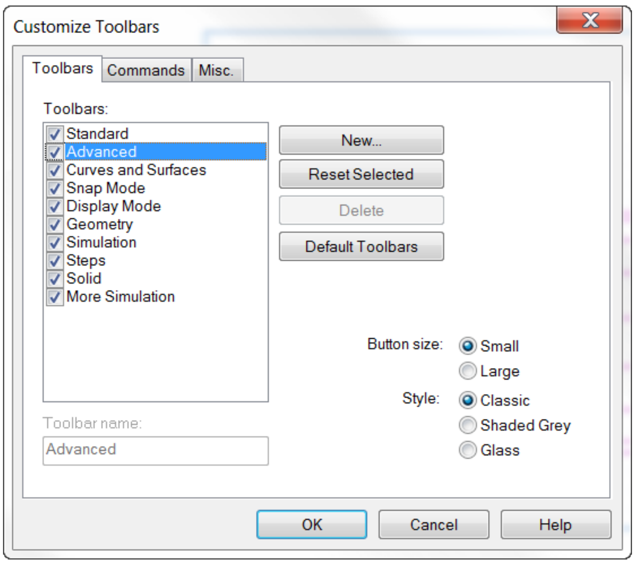
- Run a Centerline simulation



The present Z Level Interleaved finishing toolpath is fragmented with many retracts. The toolpath shape may lead to undesirable machining witness marks on the part. We will now see how a flowline toolpath can give a much better result.

We first need to create a surface to use as the flowline guide surface. We already have a curve in the center of the pocket. This will be spun around the Z axis to form a surface of revolution. We will now open up the Surface Wizard to model this surface.

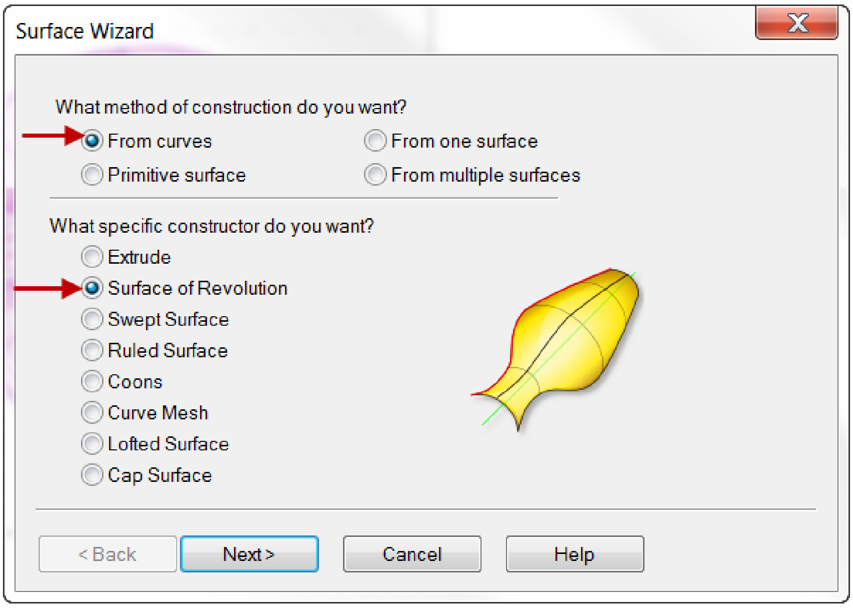
- From the **View menu** select **Toolbars**
- Check the **Advanced** toolbar



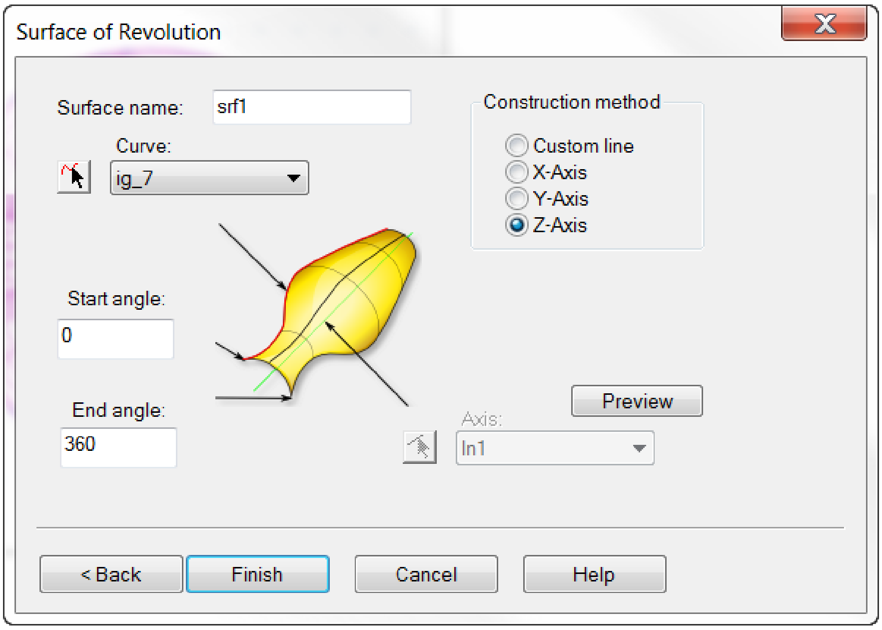
This raises the Advanced toolbar from which you can access the surface wizard.

- Click **OK**

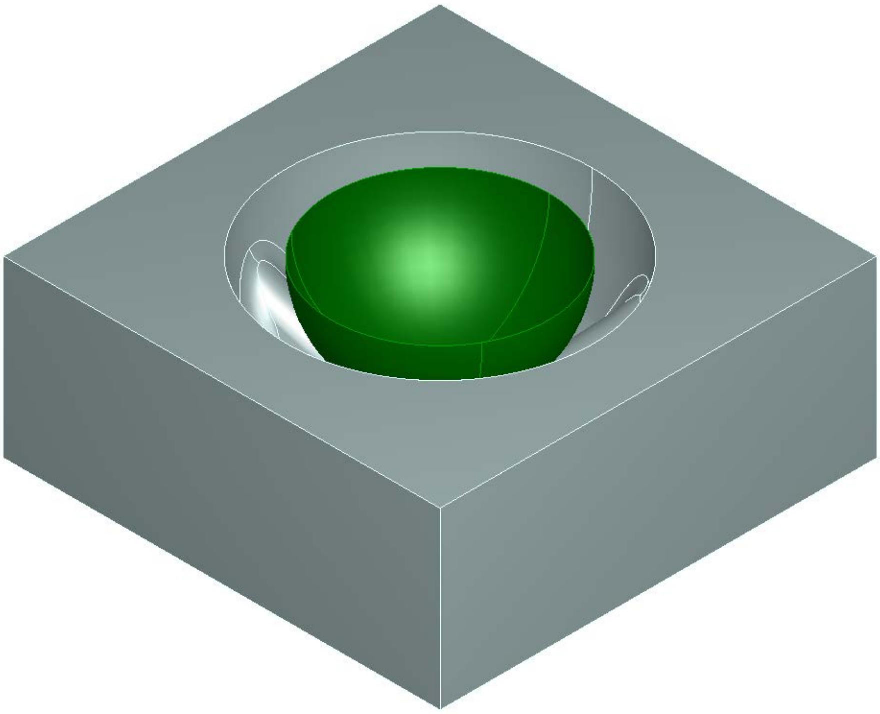
- Click on the **Surface wizard**
- Check **From Curve**
- Check **Surface of revolution**



- Click **Next**
- Fill in the form exactly as below (be sure to name the new surface "Guide")

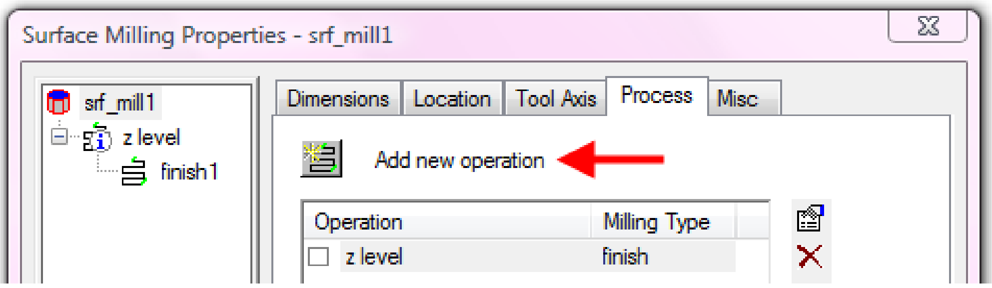


- Click **Finish**

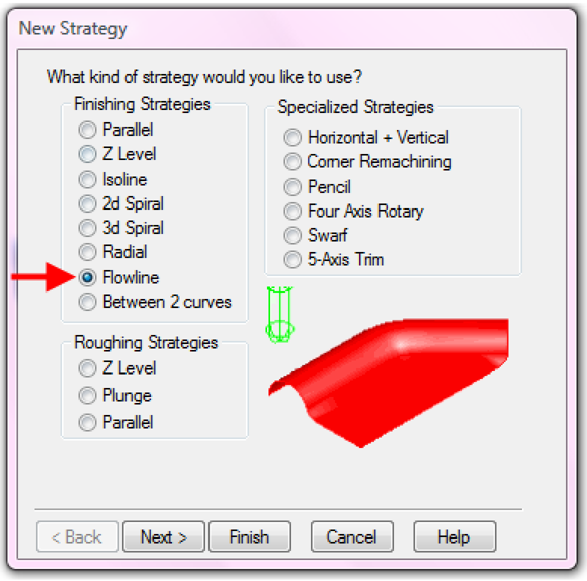


The new, hemispherical control surface is positioned in the center of the pocket.

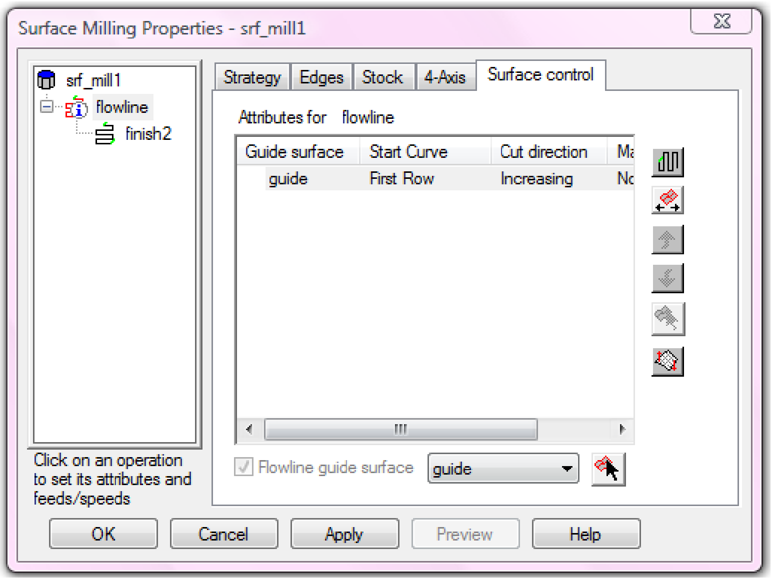
- **Edit** the feature **srf_mill1** and select the **Process tab**
- **Uncheck** the **Z Level** operation
- Click **Add new operation**



- Select **Flowline**
- Click **Finish**



- Click on **Flowline** and select **Surface Control**
- From the pulldown select **guide**

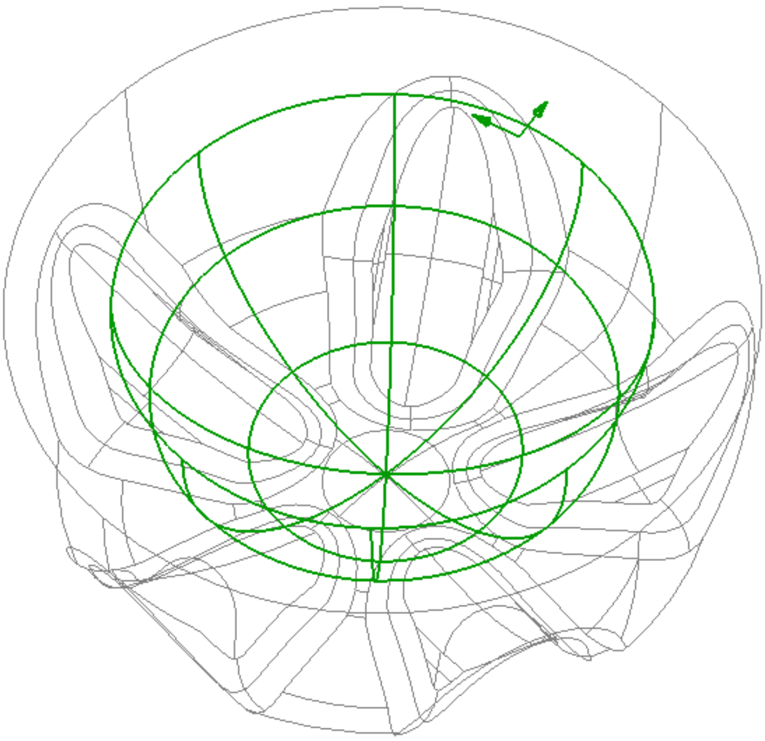


- Ensure that the machining direction arrows appear as shown below, if not use the **Set isoline row/column**, **Cut direction** and **Switch machining side** buttons (top two and bottommost buttons on the right of the form)

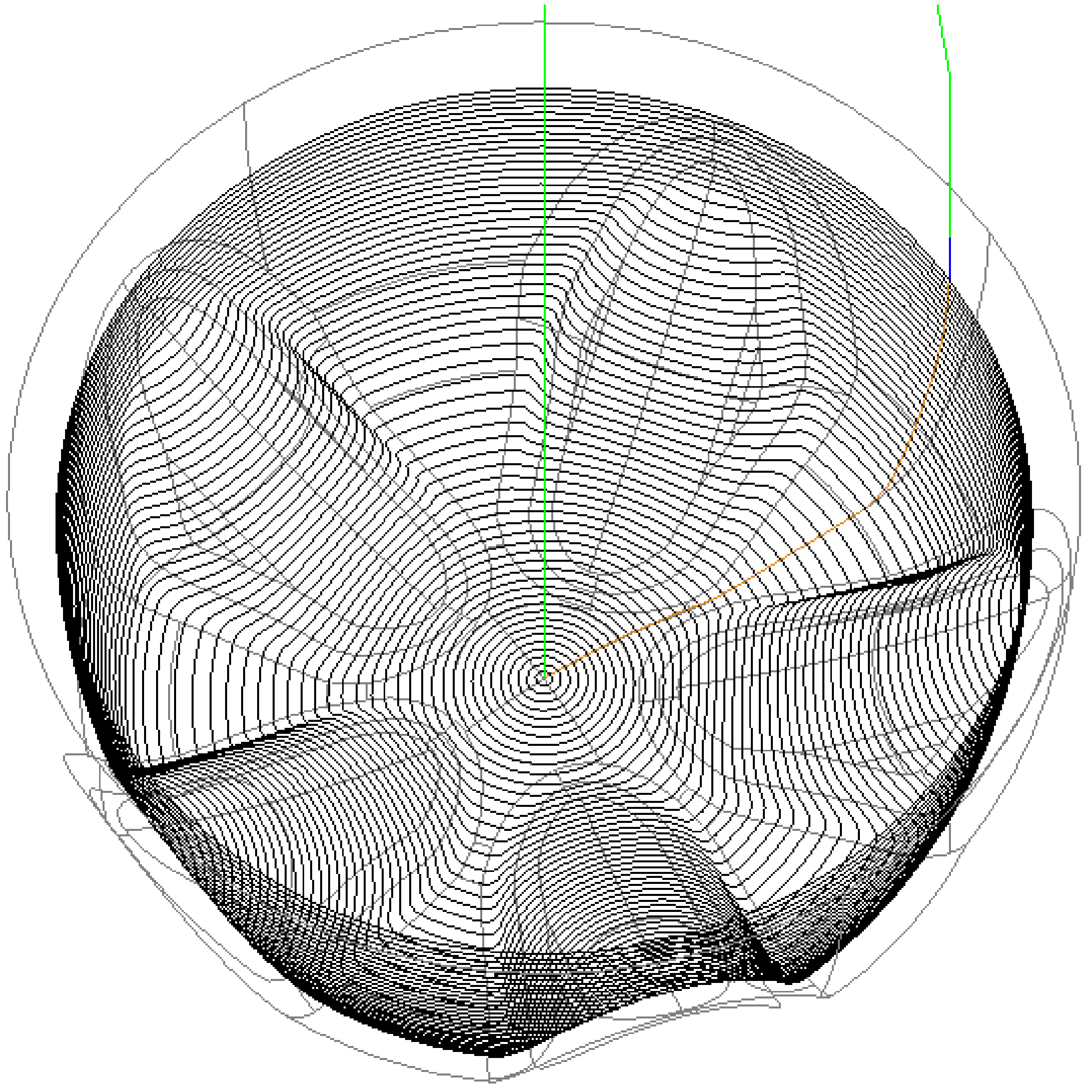
The two arrows should be at the top edge of the guide surface.

The surface normal arrow should be pointing outwards towards the surfaces to be machined.

The cut direction arrow should be pointing around the guide surface so as to create a climb milling cut.



- Click **Apply**
- Click on **finish2** and select the **Tools tab**
- Select a **0.25” Ball end mill**
- Select the **Milling** tab
- Set the **Stepover** to be **0.025”**
- Click **Apply** and **OK**
- **Hide** the **control surface**
- Run a **centerline simulation**

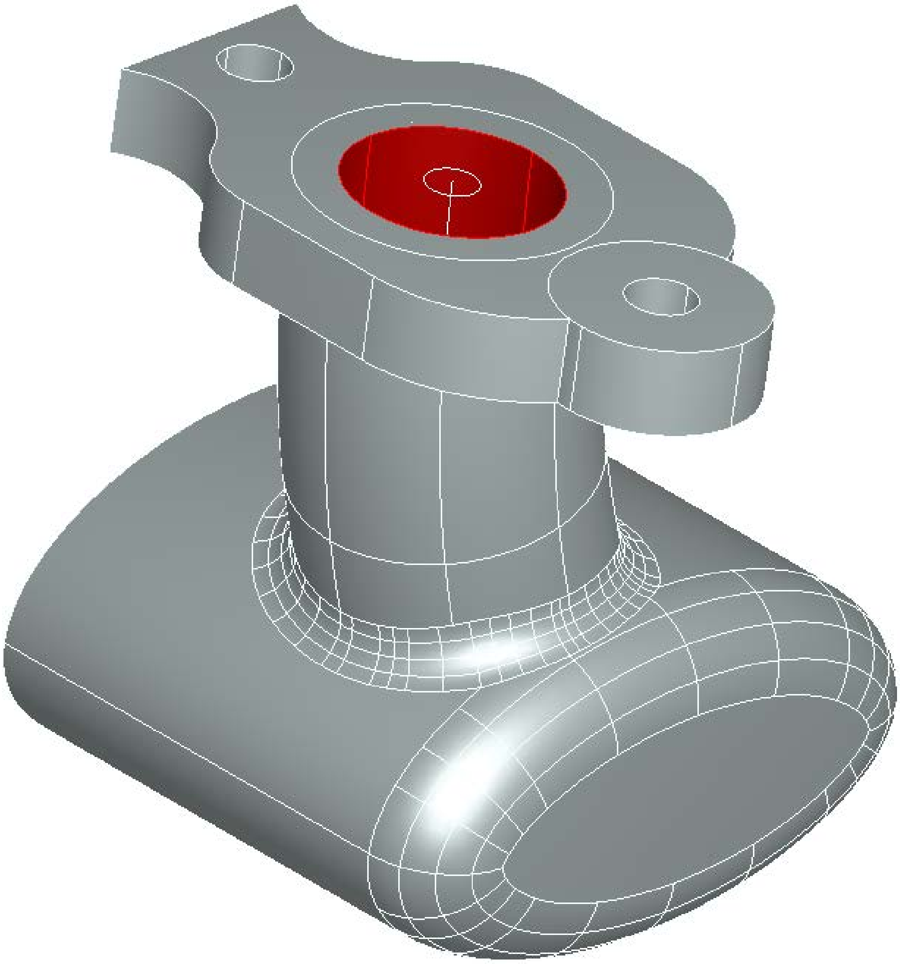


The flowline toolpath is machining across all of the surfaces with a single un-fragmented toolpath. There are no retracts during the cutting operation and the stepover is much more even and less dependent upon the slope of the surfaces being machined.

Undercut Flowline Machining

The use of flowline machining together with a lollipop or other side cutter allows the user to machine undercut areas without the use of 5 axis strategies. In this example flowline machining will be used to machine a port on a manifold, achieving an even surface finish this difficult undercut part. By machining along the port, the machining cusps are aligned with the gas flow direction giving better flow properties and ease of polishing.

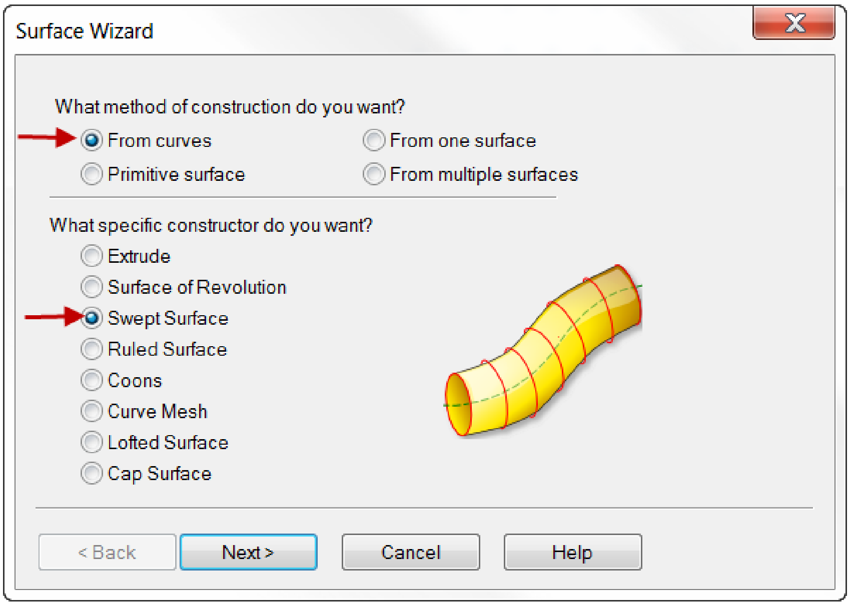
- Open the part **Manifold_Start.fm**
- Select the tool crib **Tools**
- Select an **Isometric view**



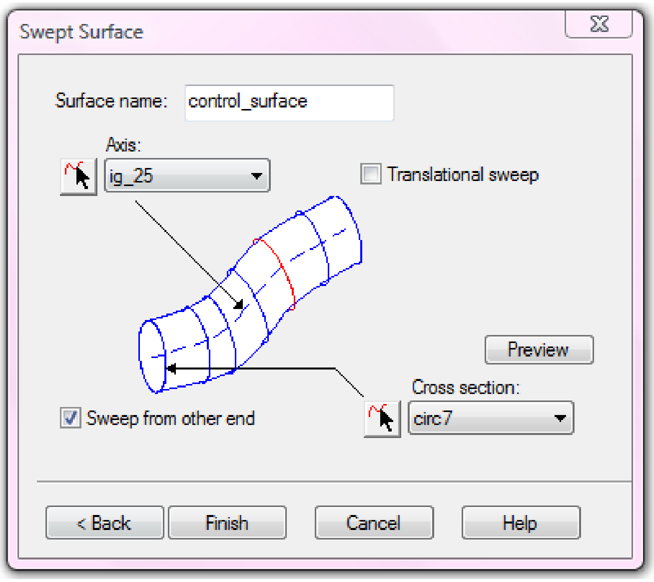
The part has been positioned so that the undercut is as small as possible. We are going to machine the port which is highlighted in the image above. The first step is to create the flowline control surface from the circle and centerline curve provided.

- Raise the **Advanced toolbar** (as described earlier)
- Click on the **Surface Wizard**

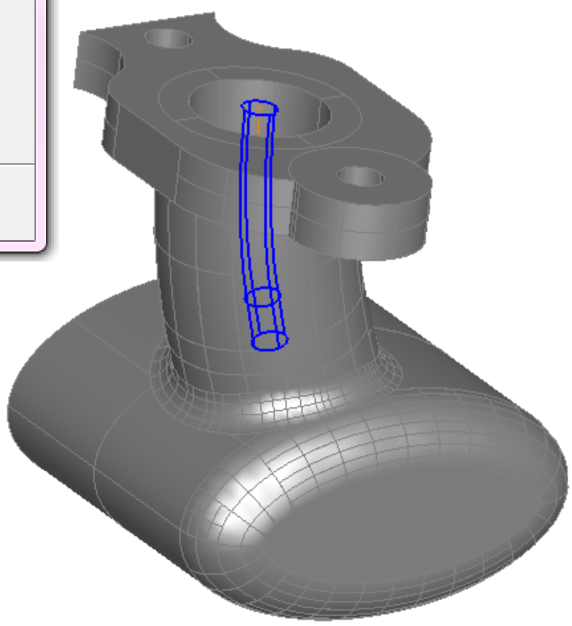
- On the surface wizard check **From curves** and **Swept surface**



- Press **Next**
- Change the **Surface name** to **Control_surface**
- Set the **Axis** to **ig_25**
- Click the **curve selector** next to **Cross section** 
- Select the circle at the top center of the port (circ7)
- Check **Sweep from other end**

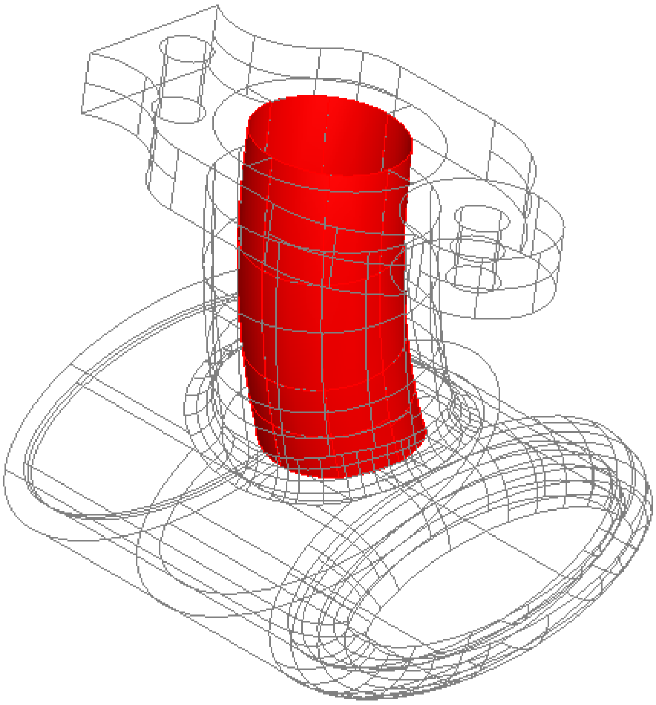


- Press **Preview**

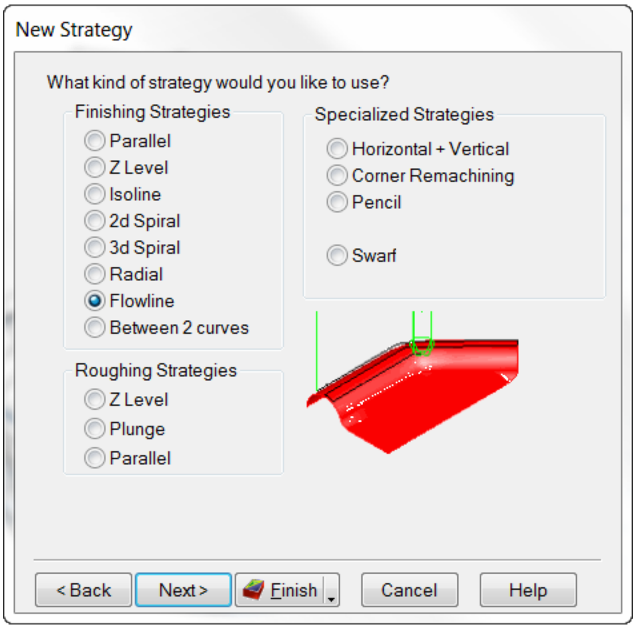


The preview surface should appear down the center of the port as shown.

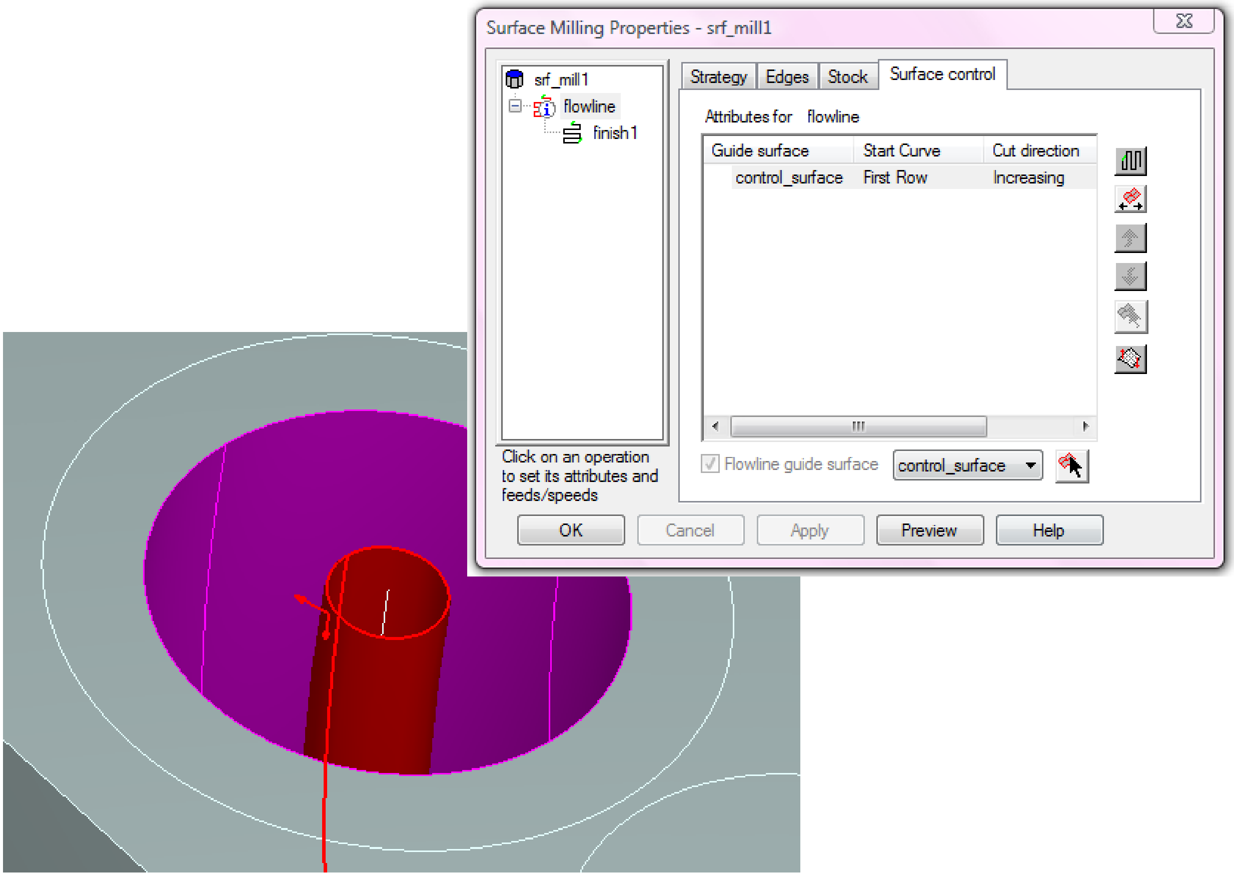
- Press **Finish**
- Select the **8 surfaces** that form the inside of the port



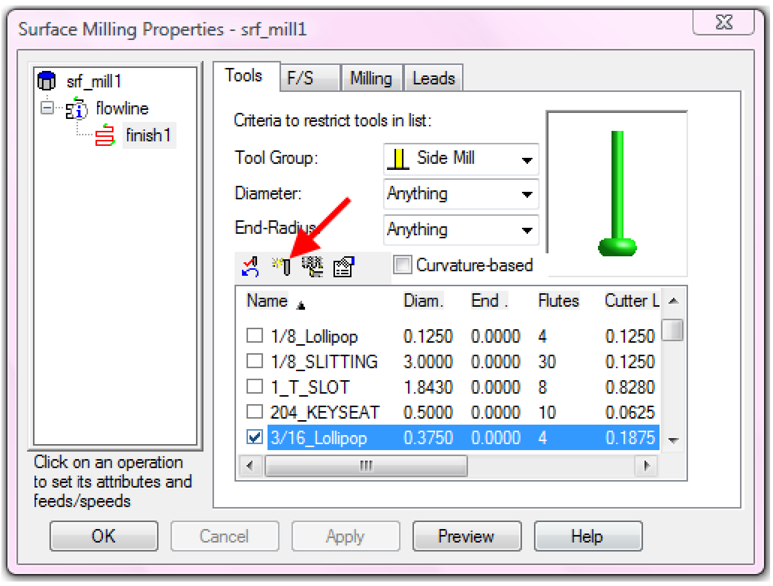
- Create a **Surface milling feature** and press **Next**
- Check **Choose a single operation** and press **Next**
- Check **Flowline** then press **Finish**



- **Edit** the **Feature**
- Click on **Flowline** and select the **Surface control** tab
- From the pulldown select the surface **Control_Surface**

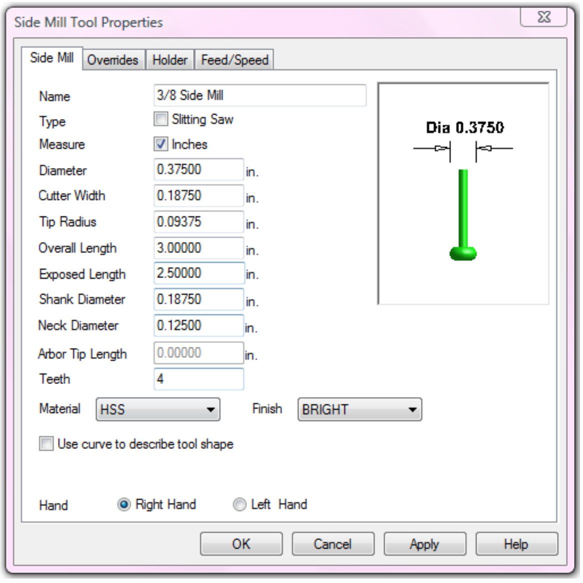


- Use the **Isoline set row/column** and **Switch machining side** buttons to get the machining direction **arrows** pointing **down** and **out** from the control surface as shown above left
- Click on **finish1**
- Select the **Tool tab**
- Select the tooling group **Side Mill**
- Select the tool **3/16_lollipop**
- Click **New tool**

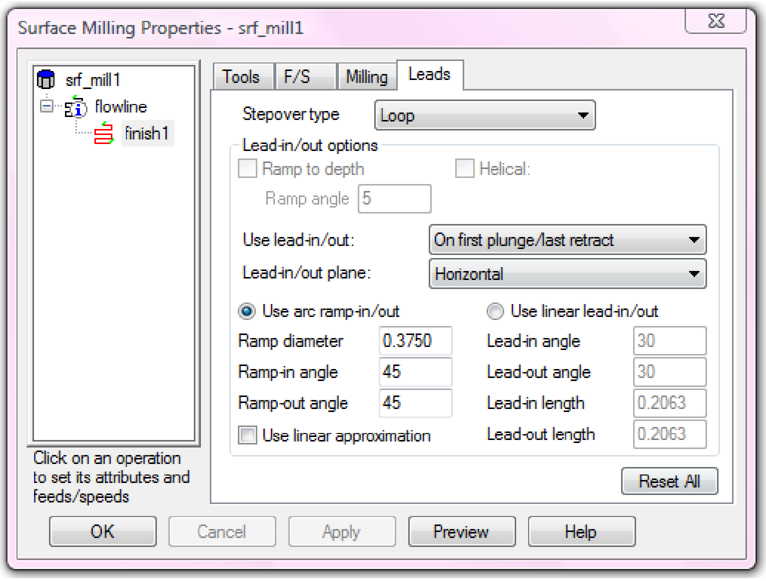


We need a larger diameter tool so that the neck and shank of the tool will clear the sides of the port.

- Fill in the form as shown then click **Apply** and **OK**

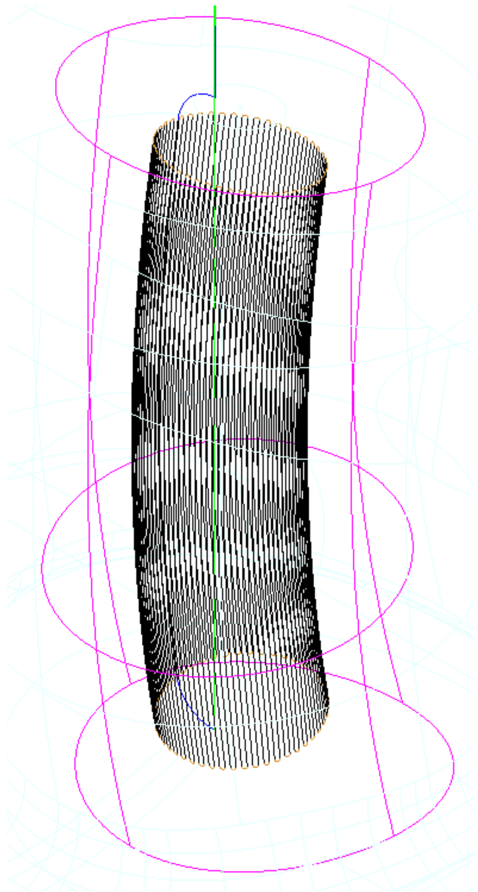


- Click on the **Milling** tab
- Set the **Stepover** to **0.010"**
- Click on the **Leads** tab
- Fill in the form as shown



- Click **Apply** and then **OK**

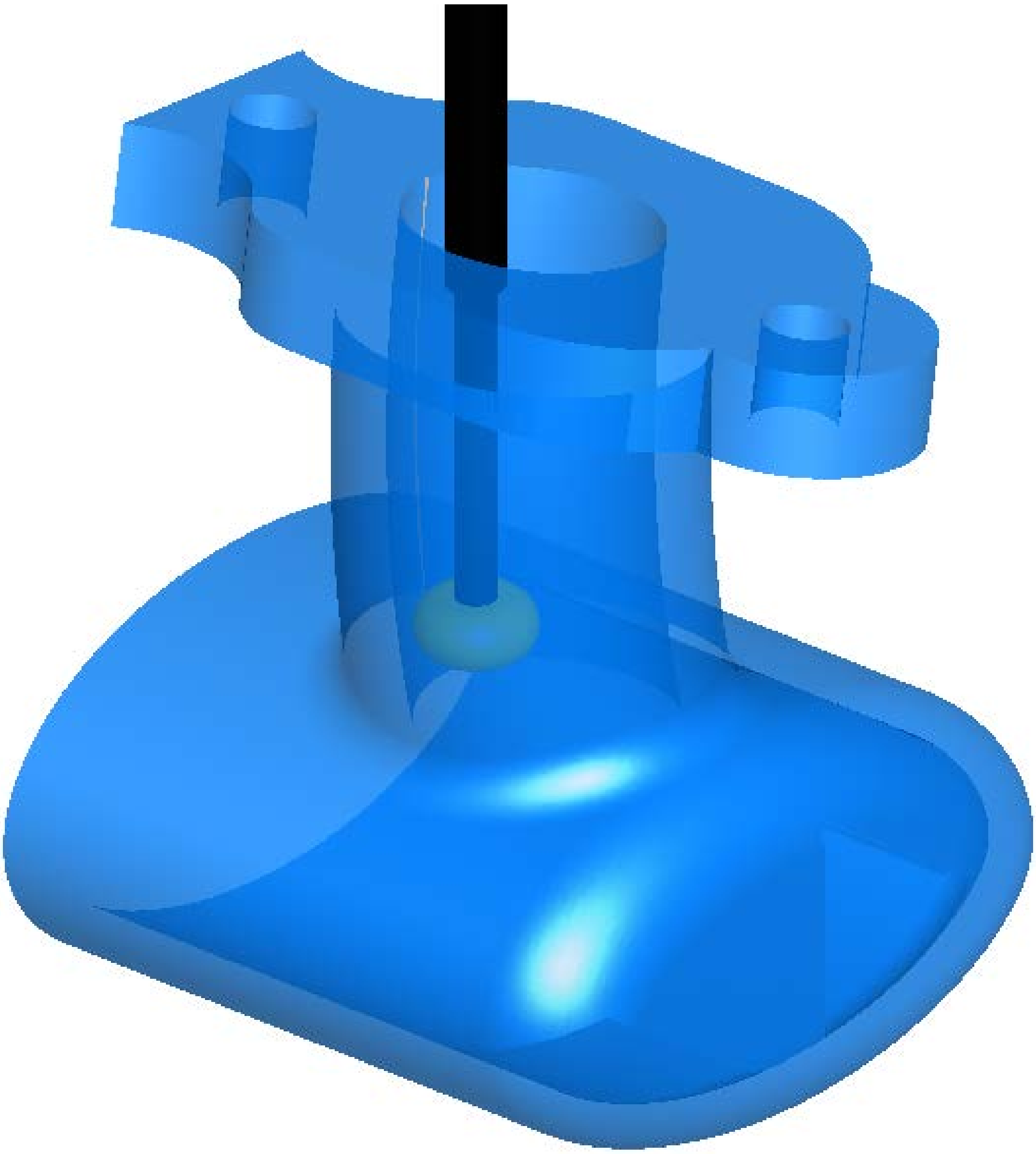
- Run a **Centerline simulation**



The port has now been finish machined. By using flowline machining with a control surface that follows the form of the port we have achieved an even stepover. Note how the lead-out move ensures that the tool clears the job on the retract move.

- **Eject** the **simulation**
- From the **Options** menu select **Simulation**
- On the **General tab** check **Show holder**
- Select the **2D/3D Shaded tab**
- Check **Translucent part**
- Click **Apply** and then **OK**

- Run a **3D simulation**



The tool neck and shank are missing the part while machining the undercut, however the tool and shank length may be giving too much flexibility causing deflection and chatter.

- As an exercise try cutting the part with the tool machining around the bore rather than up and down. Also try changing the tool shank and neck diameters and lengths to find values which give the optimum rigidity while not colliding with the part.

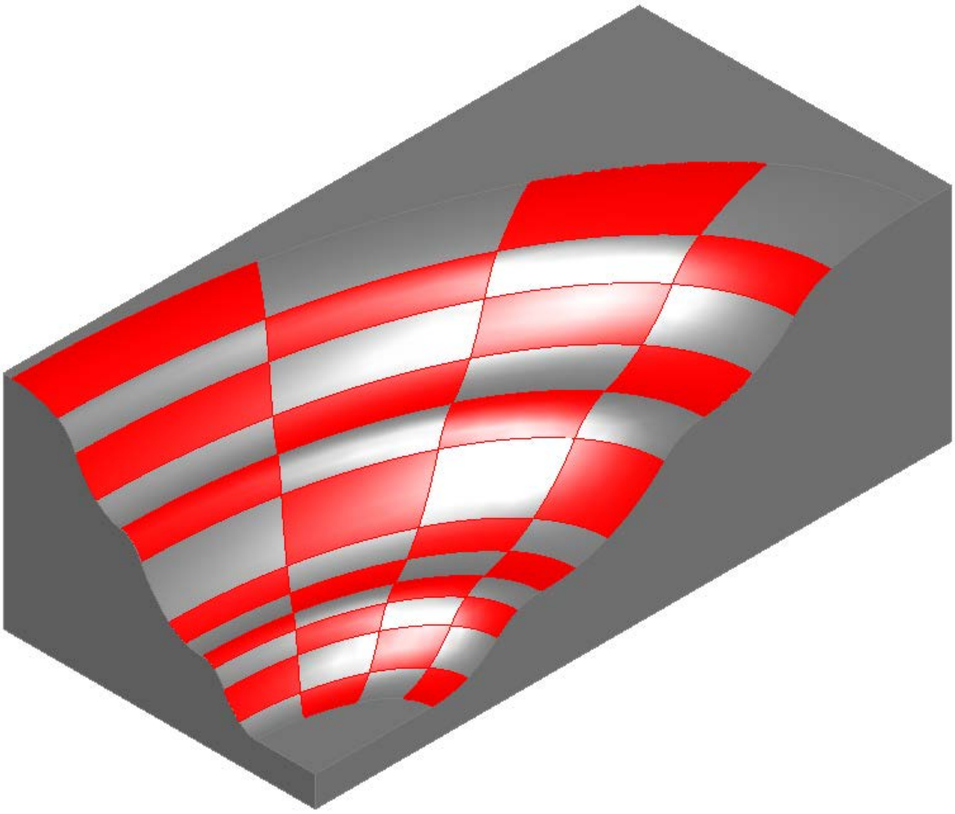
Between 2 Curves Finishing

Introduction

This strategy generates a toolpath between two user-defined curves. The stepover is along the surface in a similar manner to 3D Spiral machining. The curves can be used to define the location of the center of the tool or its point of contact on the part. The toolpath morphs from the shape of the first curve to the shape of the second with a user defined stepover. Both of the curves must be either open or closed; it is not possible to mix open and closed curves.

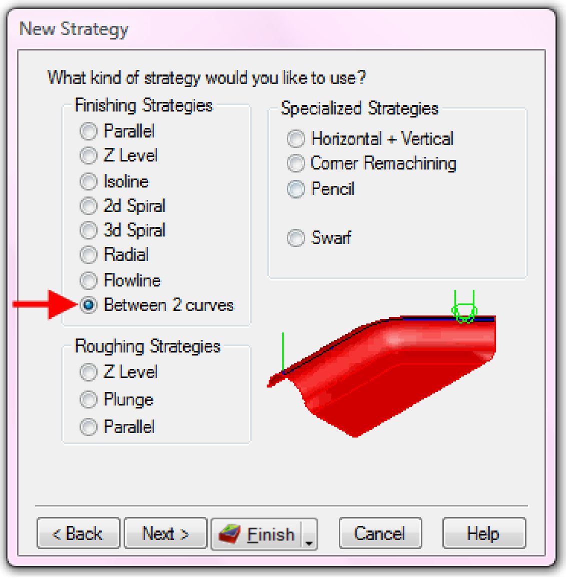
Between 2 Curves Finish Machining

- Open the part **Two_Curves.fm**
- Select an **Isometric view**

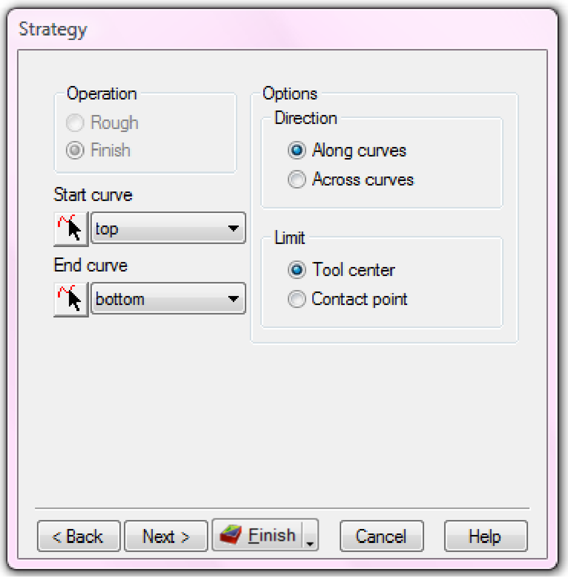


This part has a smooth, sweeping curved area. If this were a single surface, an Isoline strategy would be ideal. In this case however, the area is made up of many separate faces making it impractical to use Isoline. A similar result may be obtained by using the Between 2 curves strategy.
There are two curves in the part (top and bottom) which follow the upper and lower edges of the area to be machined. These are positioned above the top of the stock so they can be easily seen. In practice, the Z position of the curves is unimportant.

- Select all of the surfaces
- Create a new **Surface milling** operation
- Click **Choose a single operation** and click **Next**
- Select **Between 2 curves** and click **Next**

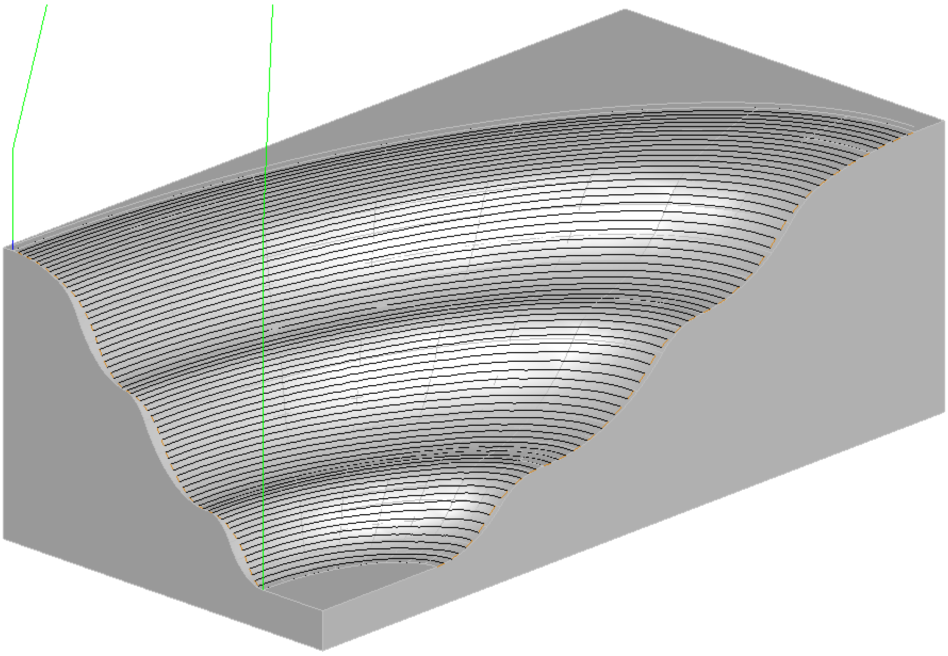


- Select the curves **top** and **bottom** in the pulldown menus as shown



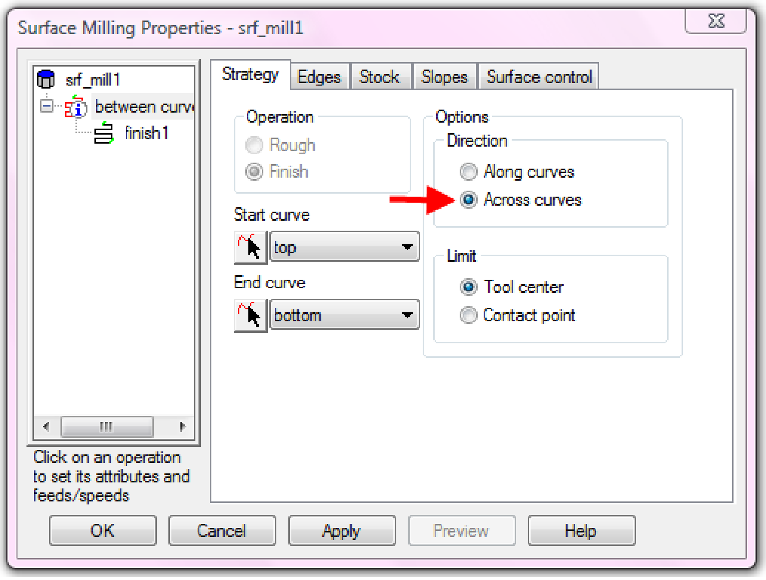
- Click **Finish**
- Click on **finish1** and select the **Milling tab**
- Set the **Stepover** to **0.1”**

- Run a **Centerline simulation**



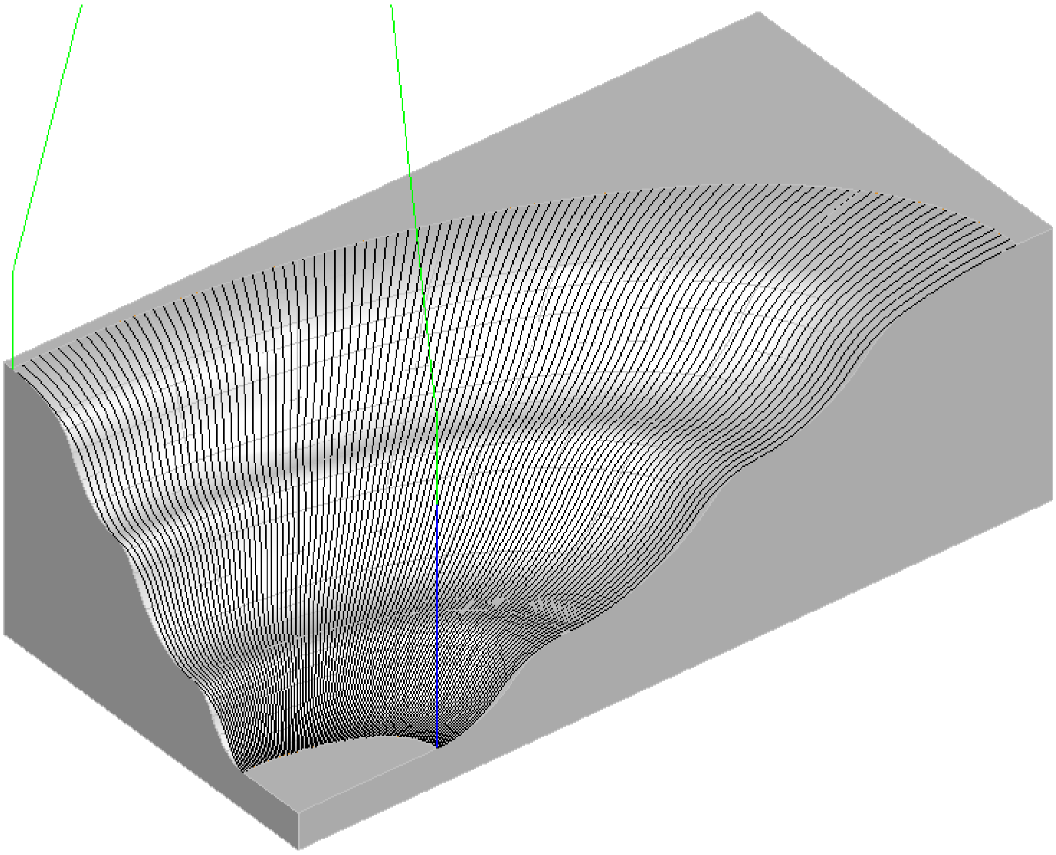
The tool is following a smooth flowing path which starts off by following the first curve (top) and then gradually transforms its shape until it meets the second curve (bottom).

- **Eject** the **simulation**
- Edit the feature
- Select the **strategy tab** and choose **Across curves**



- Click **Apply** and then **OK**

- Run a **Centerline simulation**



The tool is now “Stitching” to and fro between the two curves. As the length of the curves is very different the toolpaths are far apart at the upper edge and close together at the lower edge. Normally the “Across” option would only be used where the curves were of similar length and curvature.

- **Eject** the **simulation**

- As an exercise make a new curve part way between **top** and **bottom**. Use this to make **two** new **Between 2 curves** toolpaths using **across** for the **upper portion** and **along** for the **lower**

3-Axis Swarf Finishing

Introduction

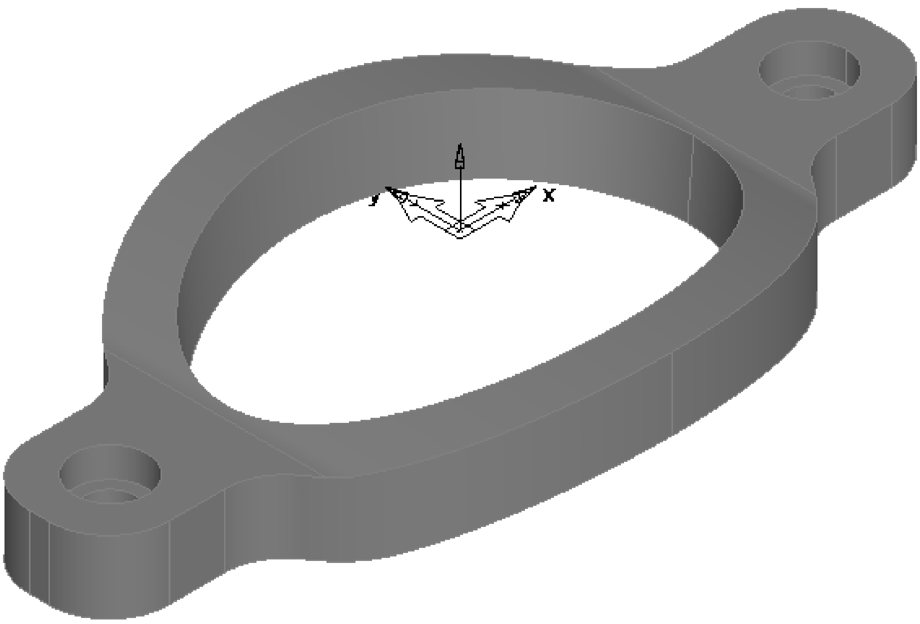
Swarf machining is normally a 5-Axis machining strategy where the part is finished using the side of the tool. The advantage is that a surface can be finished with a single pass giving a shorter machining time and a superior surface finish. Swarf machining may also be used as a 3-Axis machining strategy. Parts with a vertical wall or with a constant angle taper can be finished in a single pass by using an appropriate tool.

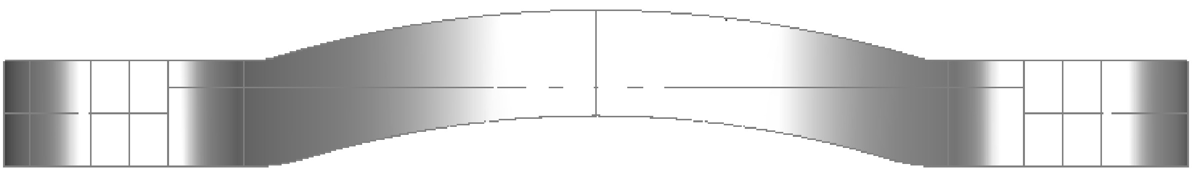
Where the side walls of the part have a variable taper it is possible to machine using a ball nosed cutter with multiple passes. In this module, we will first look at a part that can be machined with a single pass and then at a more complex part that requires multiple passes.

3-Axis Swarf

The part used in this example is a simple flange casting which requires the inside and outside edges finishing. The outside edge is vertical and the inside edge has a constant 5 degree taper. As the edge to be finished is a 3D profile, it would not be simple to create a 2.5D side feature to finish the part.

- Open the part **Swarf.fm**
- Select an **Isometric view**

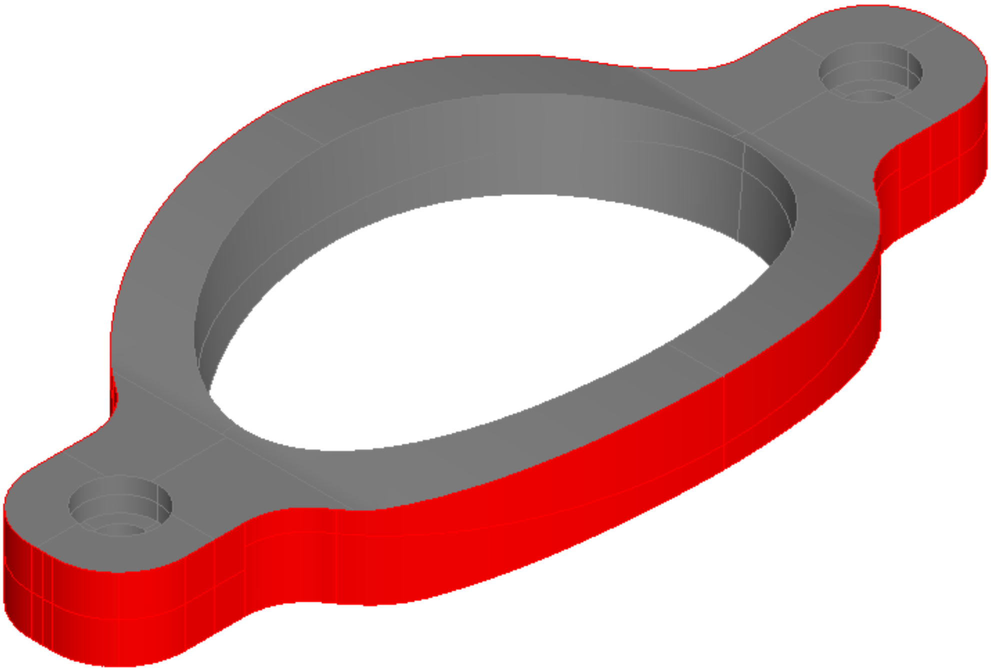




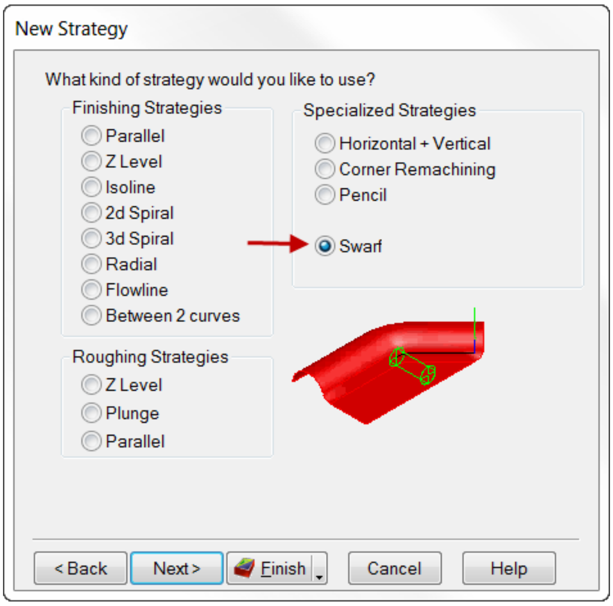
As you can see from the side view, the top and bottom edges of the part are curved.

First of all we shall machine the outer, vertical edge of the part using a flat endmill. This would be possible using a side feature but the tool would be going all the way to the bottom of the part throughout the side finish pass. This would require a long tool.

- Select the surfaces that make up the outer wall of the part

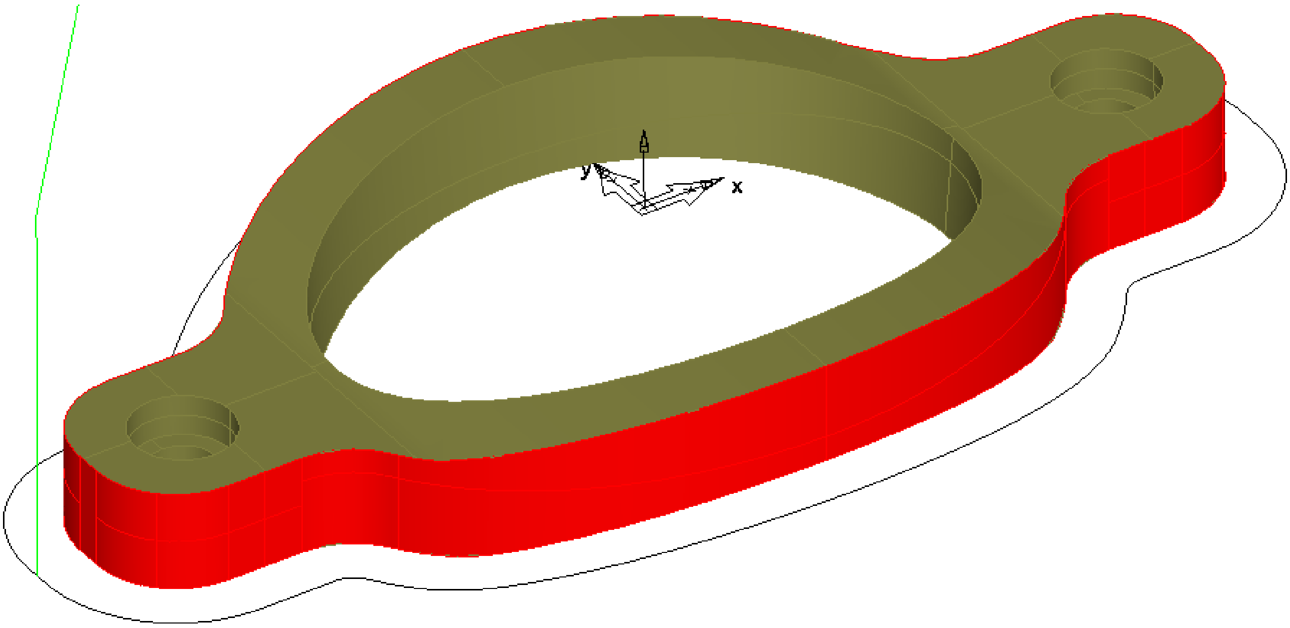


- Create a new **Surface milling** operation
- Click **Choose a single operation** and click **Next**
- Select **Swarf** and click **Finish**



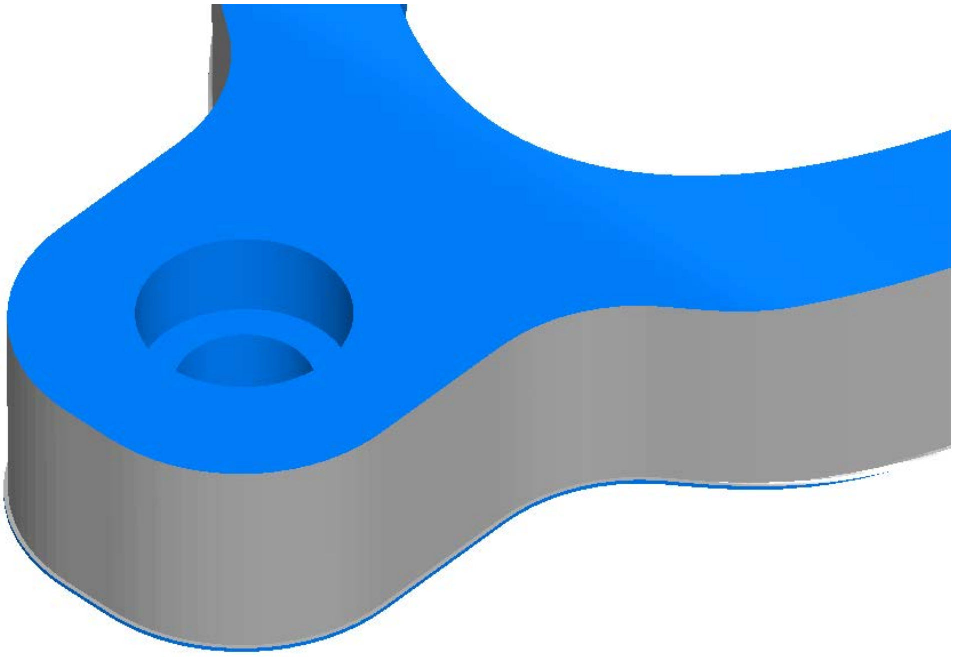
- Click **OK**
- Slow down the simulation speed

- Run a **Centerline simulation**



The tool is machining around the part following the lower edge of the selected surfaces.

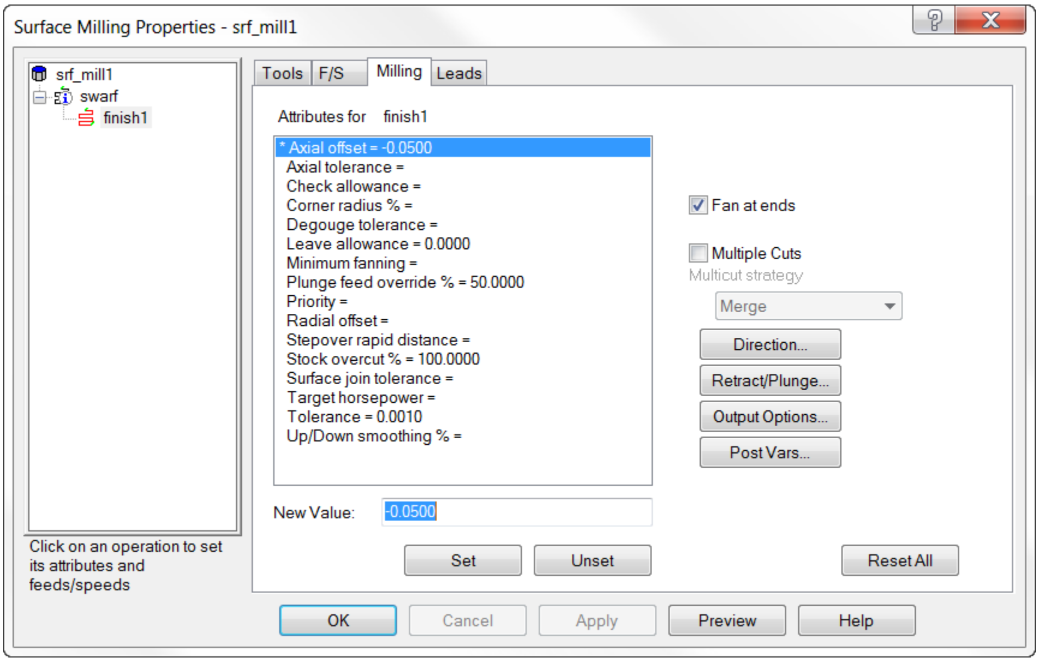
- Run a **3D simulation**



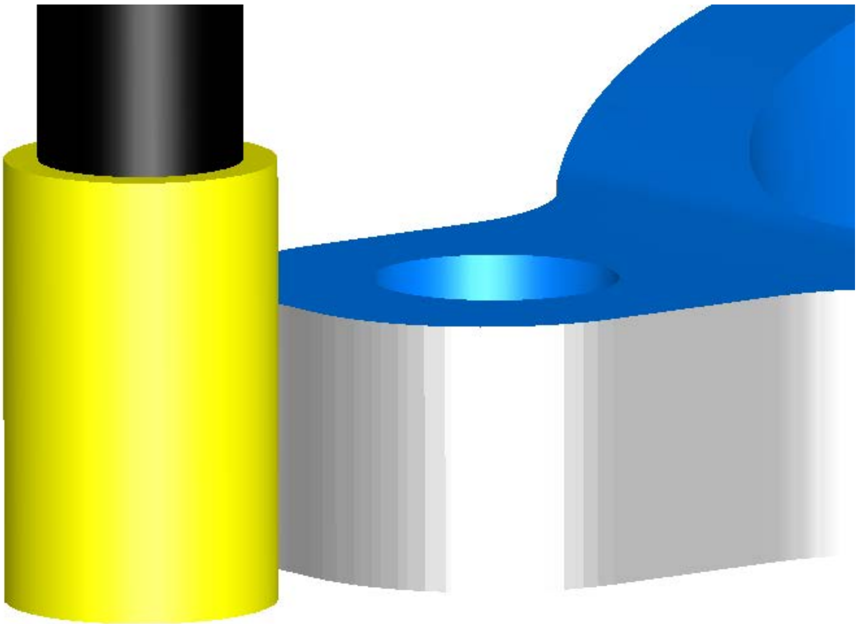
The casting is slightly larger than the finished part, this means that a small lip of excess material is felt after the finishing pass. We need to make the tool tip move slightly past the lower edge of the part to ensure that it cleans up the edge.

- **Eject** the **simulation**
- Edit the feature and select the **Milling tab**

- Set the **Axial Offset** to **-0.05"**. (This tells the tool to go past the edge by 0.05")



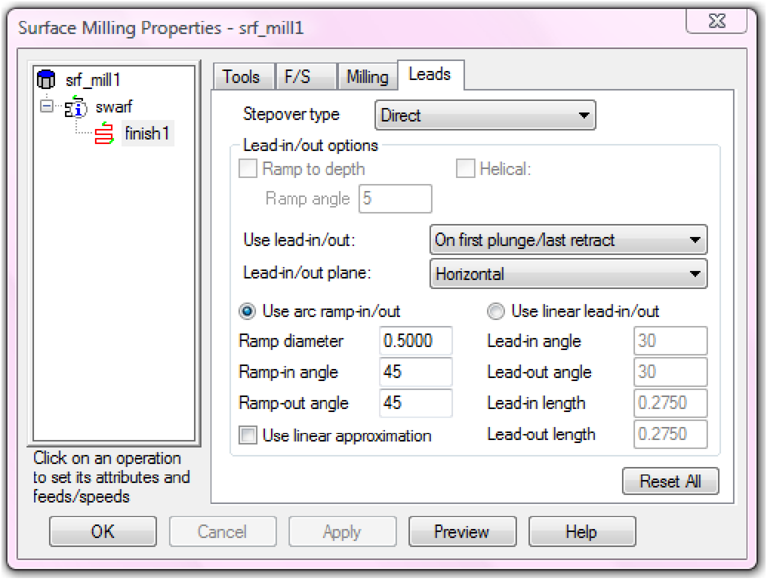
- Run a **3D simulation**



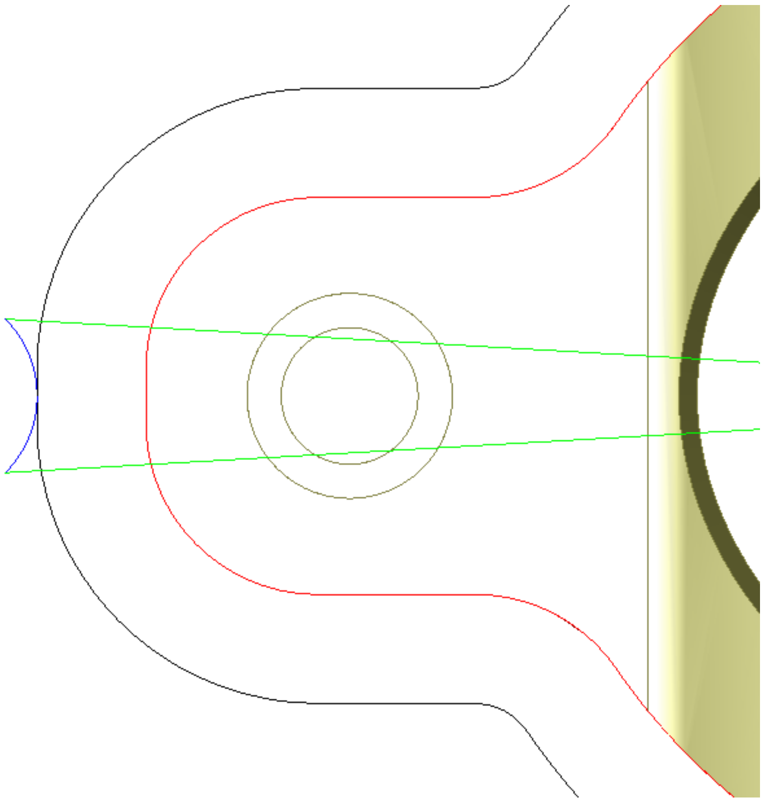
The tool now projects past the lower edges of the surfaces selected for machining by 0.05" so that a clean finish results. It will be noted that the tool is approaching the part by plunging vertically down the face that is to be finished. This could result in a witness mark. We will now modify the toolpath to ramp into the part with a horizontal arc.

- **Eject** the **simulation**
- Edit the feature and select the **Leads tab**

- Select a **Horizontal arc Lead in** and **Lead out** as shown below



- Click **Apply** and then **OK**
- Take a **Top View**
- Run a **Centerline simulation**

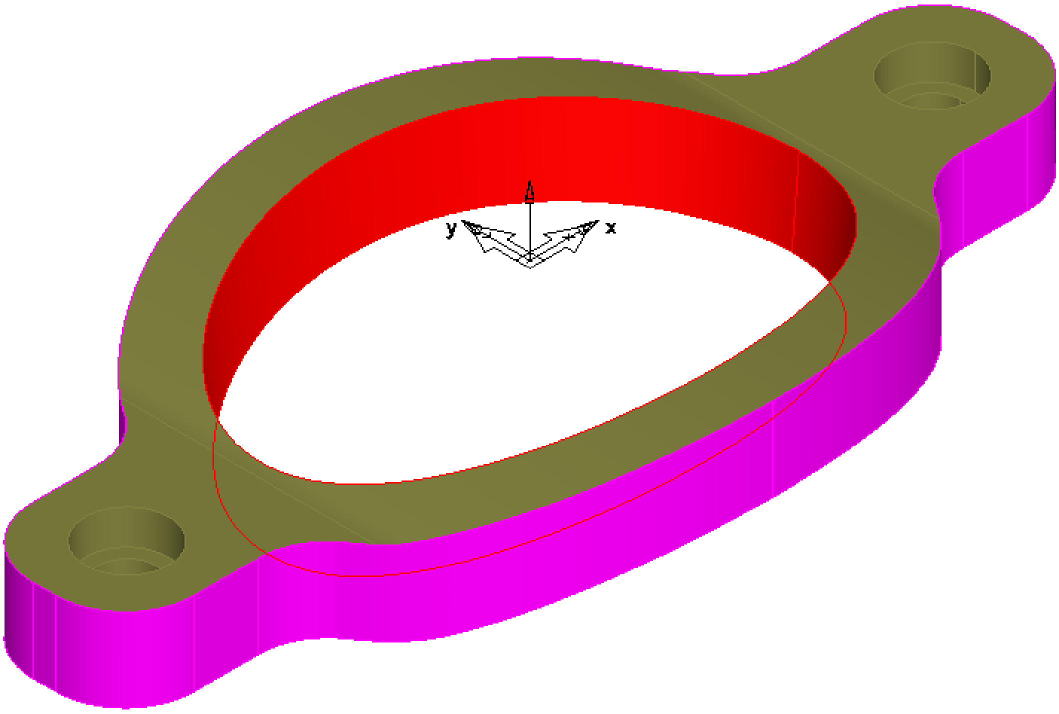


The tool is now ramping smoothly on and off the part.

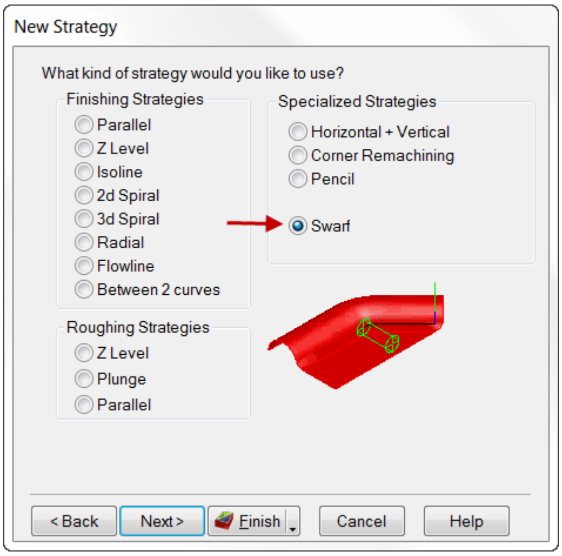
- **Eject** the **simulation**

We will now machine the inside edge of the part which has a 5 degree taper.

- Select the inside surface of the part

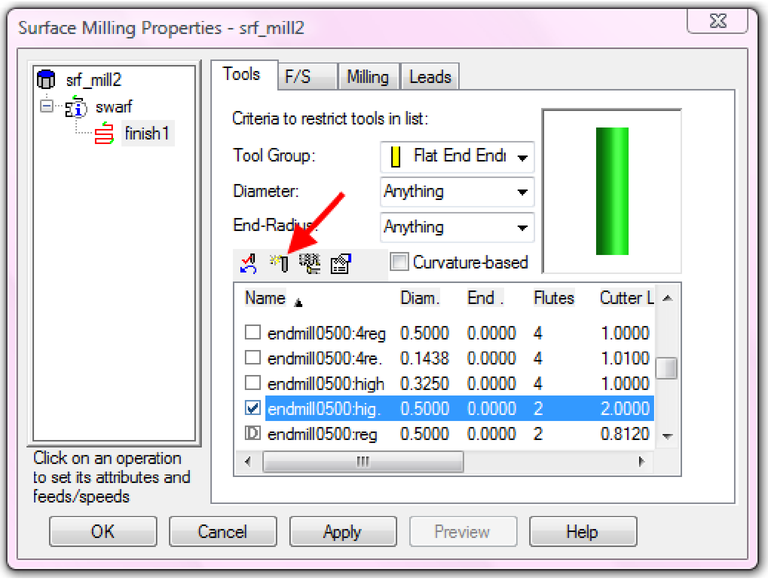


- Create a new **Surface milling** operation
- Click **Choose a single operation** and click **Next**
- Select **Swarf** and click **Finish**

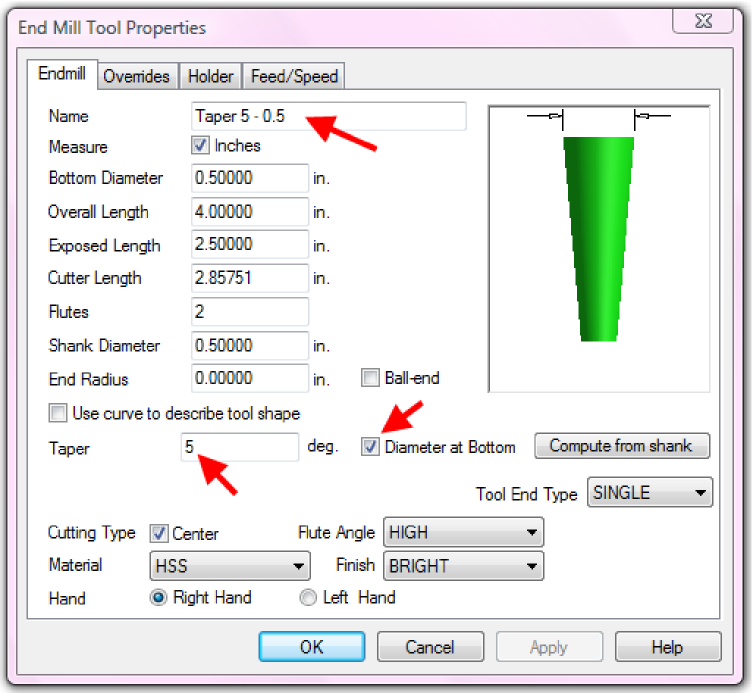


- Click on **finish1** and select the **Tools tab**

- Select a long reach **0.5" Flat endmill** and then select **New tool**

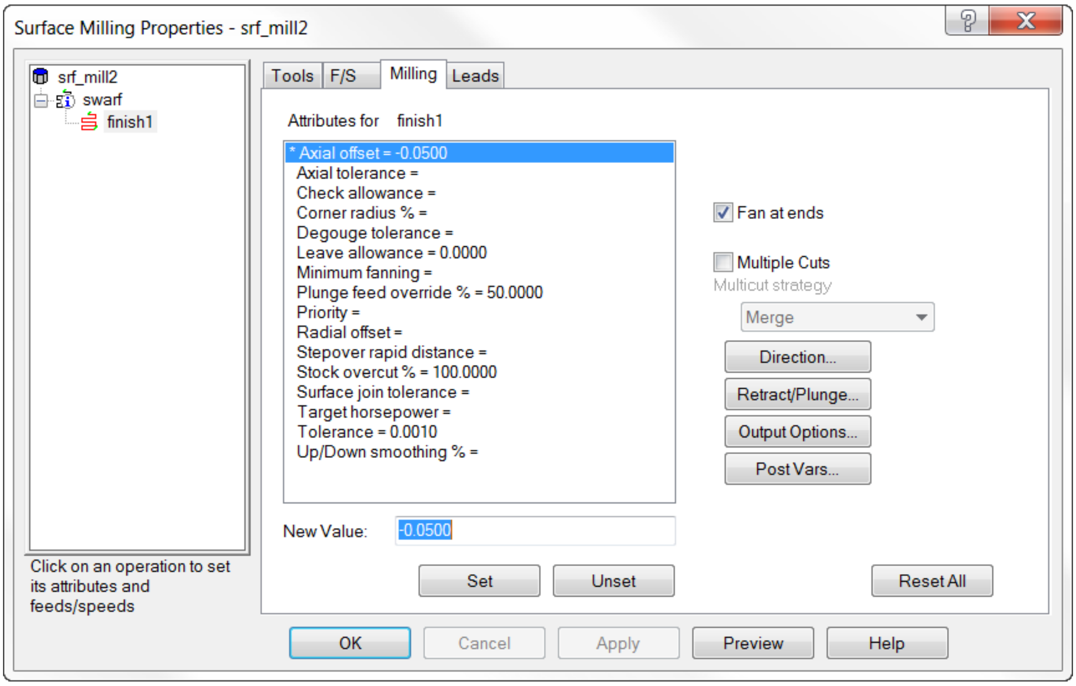


- Fill in the form exactly as shown below

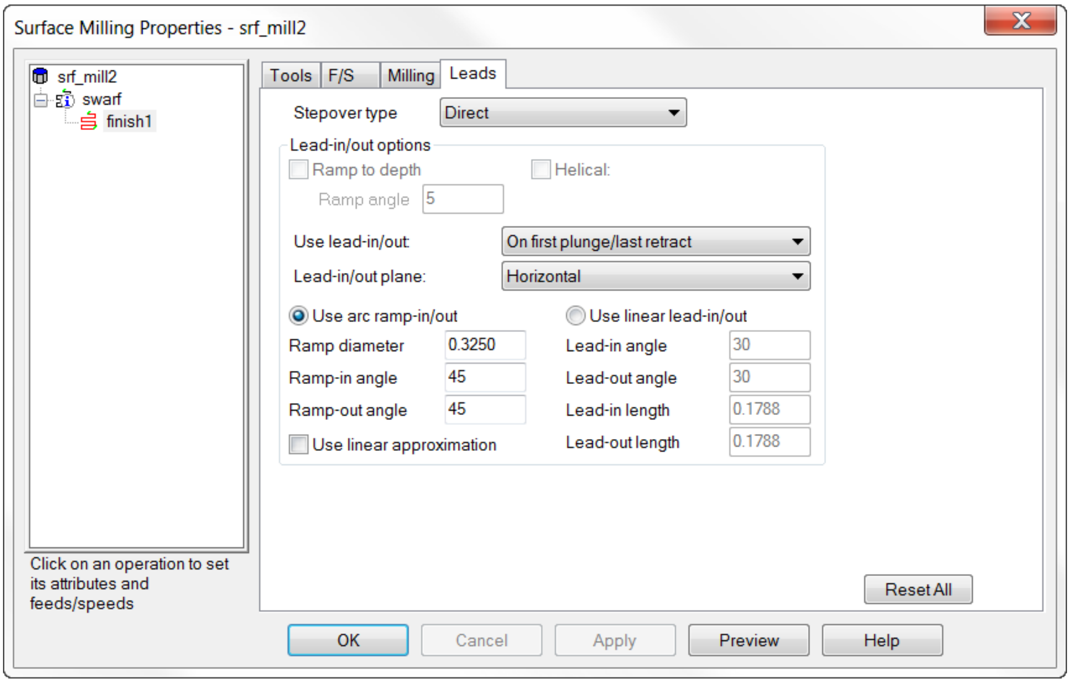


- Click **Apply** and **OK**

- Edit the feature and select the **Milling tab**
- Set the **Axial Offset** to **-0.05”** (This tells the tool to go past the edge by 0.05”)

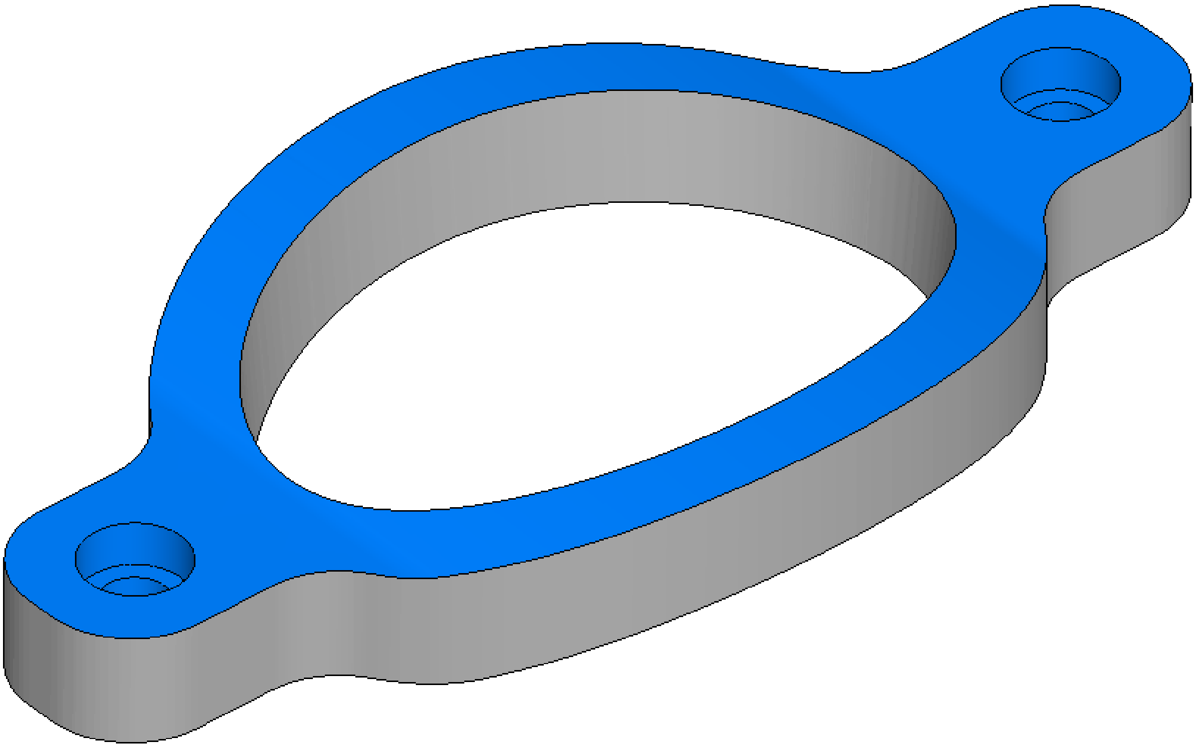


- Select the **Leads tab**
- Select a **Horizontal arc Lead in** and **Lead out** as shown below



- Click **Apply** and then **OK**

- Run a **3D Simulation**



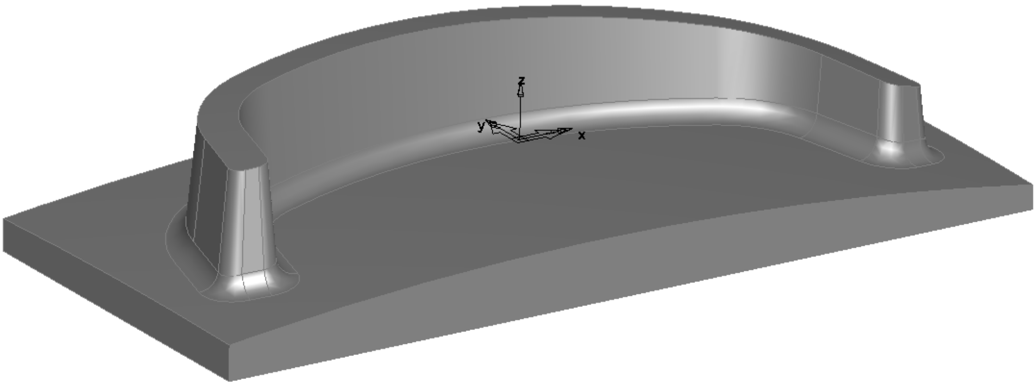
The part is now finished; note how the tool tracks up and down in Z while following the edge of the constant tapered part. This would not be possible with a 2.5D Side feature.

We shall now look at a part that requires multiple passes for finishing.

3-Axis Swarf – Multiple passes

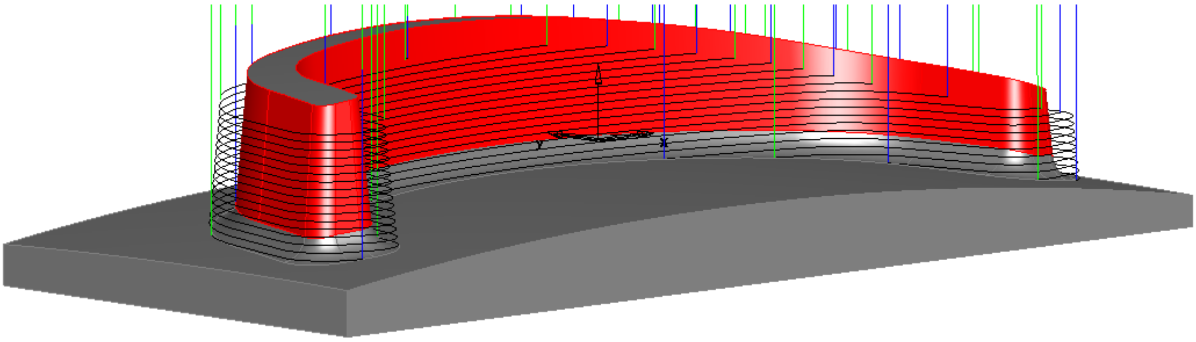
This part has a varying radius around the corners. It could be finished by a 5-Axis swarf toolpath in a single pass. If we use a 3-Axis swarf toolpath then multiple passes will be required.

- Open the part **Multi_Swarf.fm**
- Select the tool crib **Multi_Swarf.fm_Tools_from_last_save**
- Select an **Isometric view**

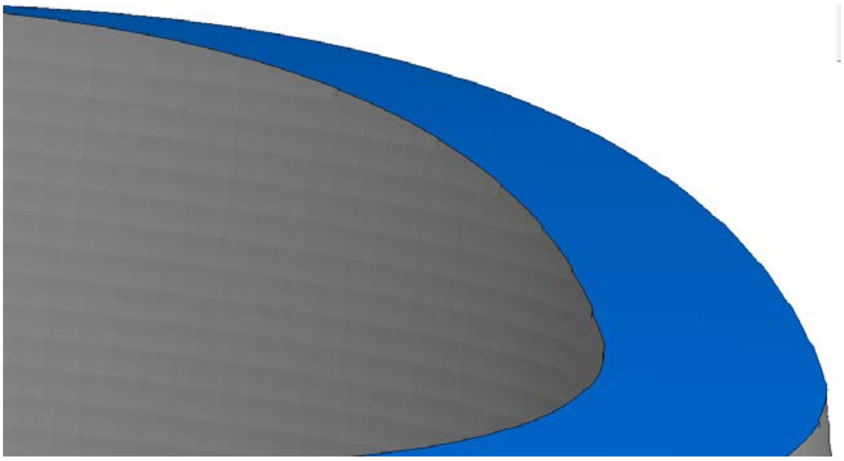


The part contains a Z Level finish toolpath for the tapered surfaces, we will now run a 3D simulation to see the result of this toolpath.

- Run a **Centerline simulation** then a **3D simulation**

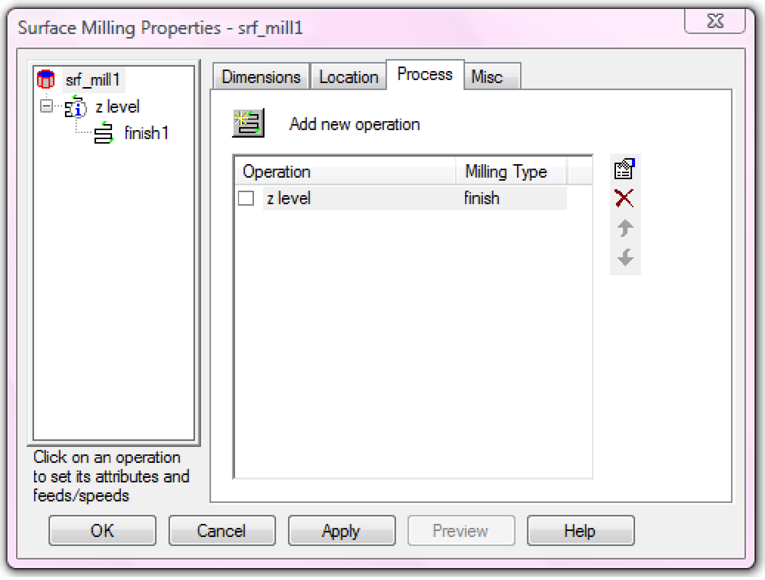


The shallow slopes at the top and bottom edges of the part are causing the tool to lift off many times. This not only results in wasted air moves but also gives a poor finish at both edges of the part.

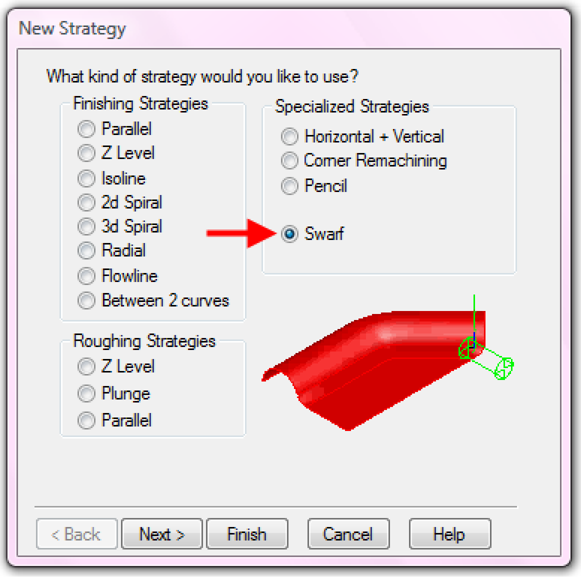


- **Eject** the **simulation**

- Edit the feature
- Select the **Process tab** and uncheck the **Z Level** operation
- Click on **Add new operation**

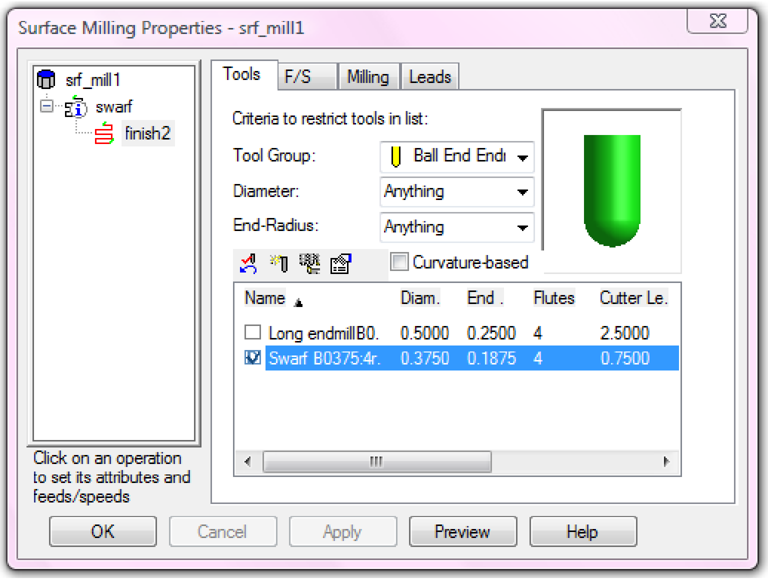


- Select a **Swarf operation** and click **Finish**

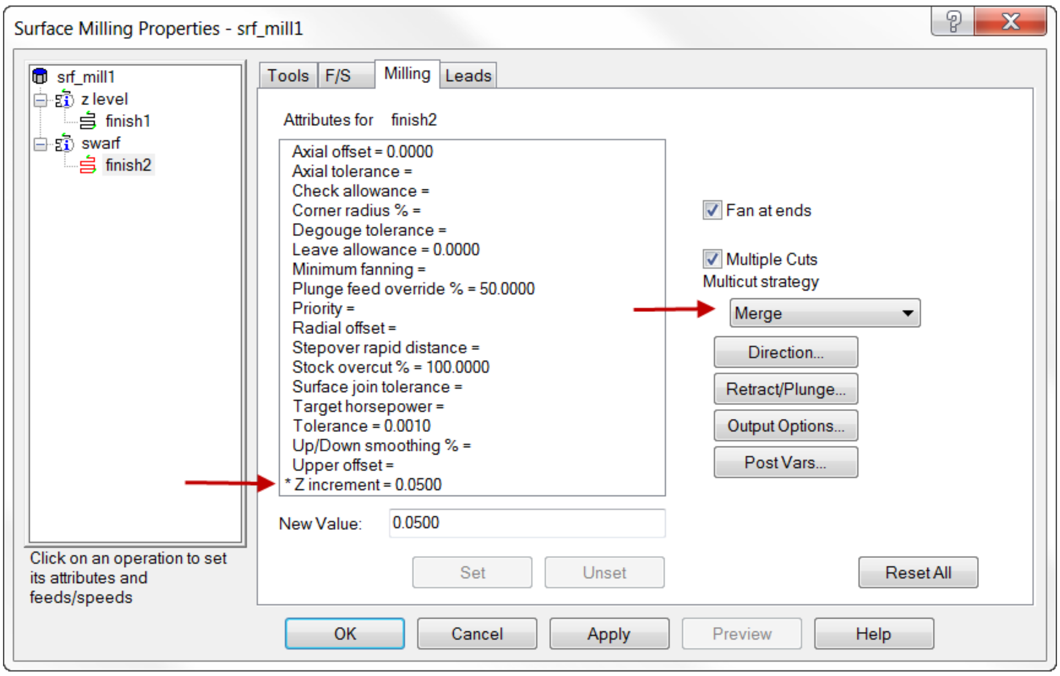


- Click on **finish2** and select the **Tools tab**

- Select a **0.375" Ball nosed endmill**

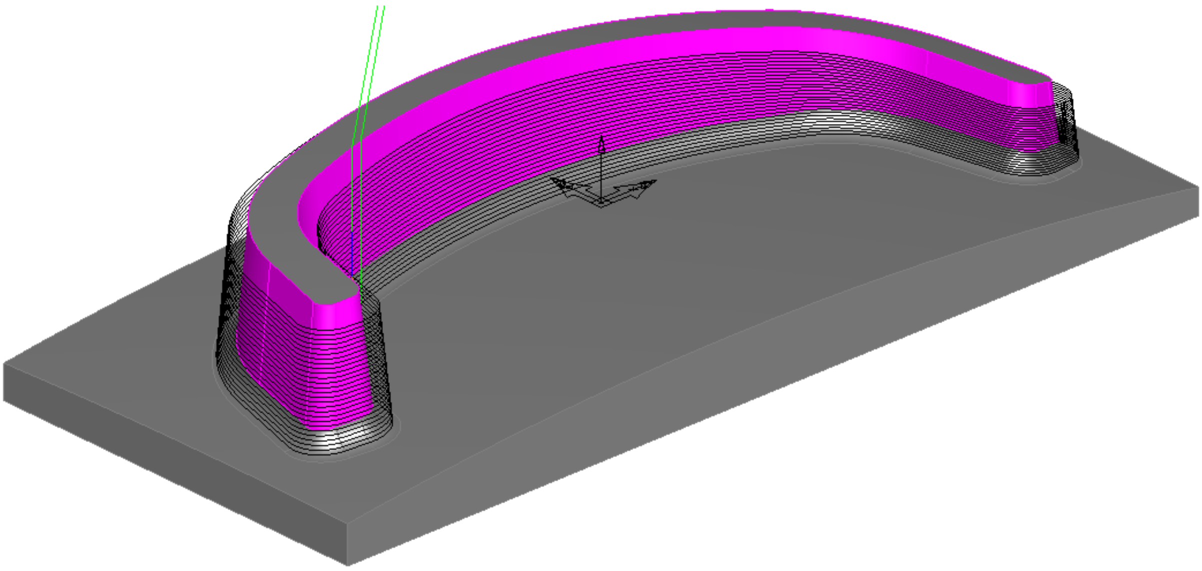


- Click **Apply** and then select the **Milling tab**
- Check **Multiple cuts**
- Click on the **Multicut strategy** pulldown
- Select **Merge**
- Set the **Z Increment** to **0.050"**

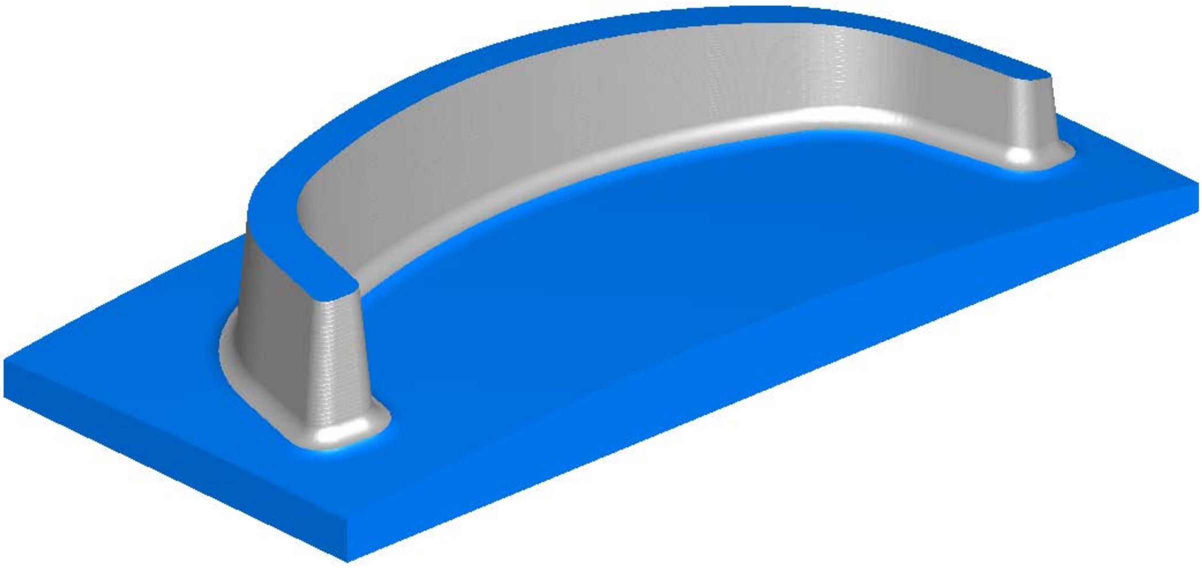


- Click **Apply** and then **OK**

- Run a **Centerline simulation** then a **3D simulation**



The toolpath now "Merges" from a single unbroken pass around the top edge of the part to a single pass around the bottom edge. The stepover is adjusted to give a whole number of passes. The specified stepover of 0.05" is applied at the widest part of the surfaces being machined and adjusted proportionately depending upon the width of the part at each point. If you look at the left end of the part where the surfaces are at their narrowest you will see that the toolpaths are bunching together at that point.



- As an exercise try the three different options for multiple cuts – **Merge**, **Offset top down** and **Offset bottom up**.

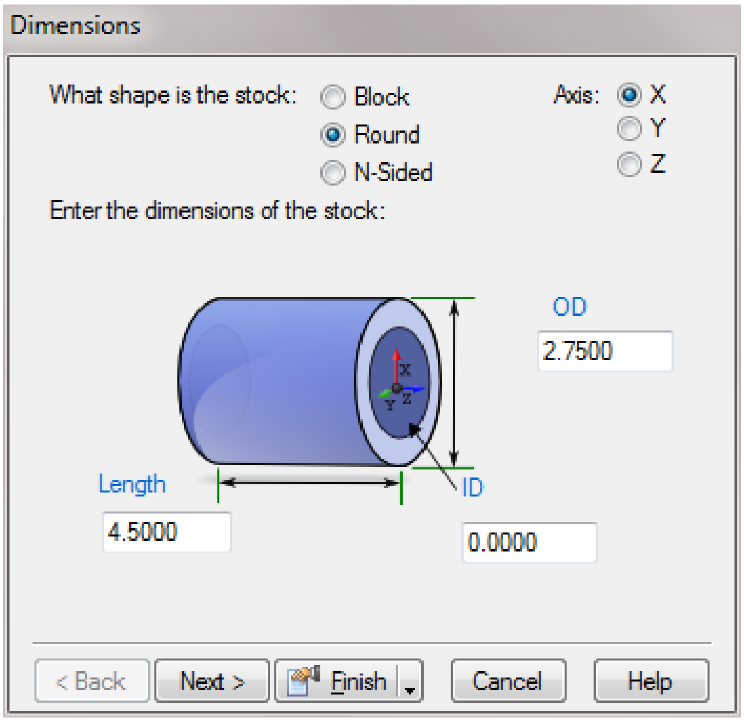
4-Axis Rotary Finish Machining

Overview

In 4th Axis or Rotary machining, a part is mounted in a rotary chuck which rotates to allow access to areas of the part not normally reachable by 3 axis machining. In this example, we will be finish machining a component which would be difficult to machine using a 3-axis setup due to its length and small inside radii.

4-Axis Finishing

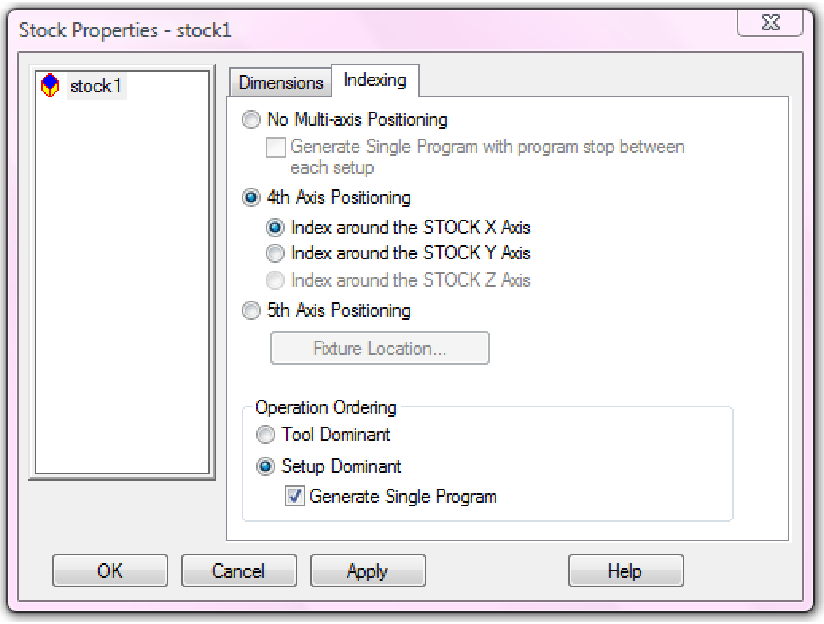
- Open a new **Inch milling document**
- Fill in the stock **Dimensions** form as shown below



- Press **Finish**
- On the **Stock properties** form select the **Indexing tab**

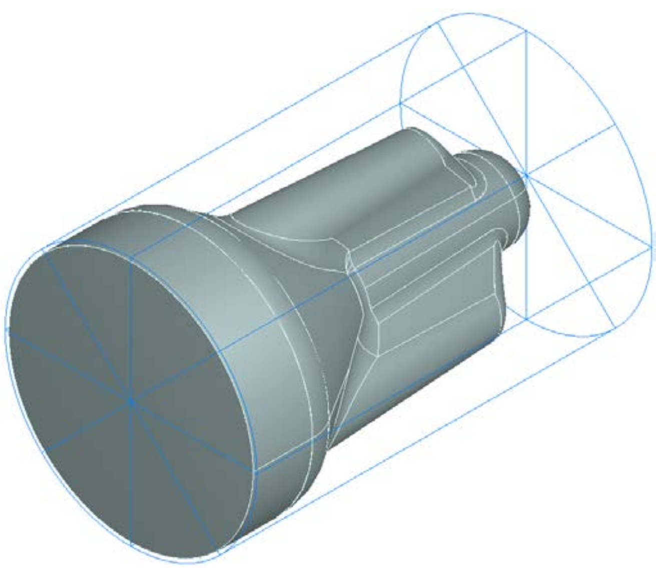
We need to tell FeatureCAM which axis our machine will be using for rotational moves. In this case we are setting up to rotate around the X axis. In general, there are three possible rotary axes. These are called the A, B and C axes. A 4-Axis machine will have the normal X,Y and Z linear movement axes plus one rotary axis. Depending upon the machine, this may be an A axis (rotates around X), a B axis (rotates around Y)or a C axis (rotates around Z). The most common 4-Axis machine configuration is XYZ plus A axes.

- Fill in the form as shown below



Here we have set up the stock to rotate about a rotary X or A axis.

- Press **Apply** and then **OK**
- Import the model **Spindle.igs**
- Check **Accept the imported data "as is" and exit the wizard**
- Press **Finish**
- Select an **Isometric** view

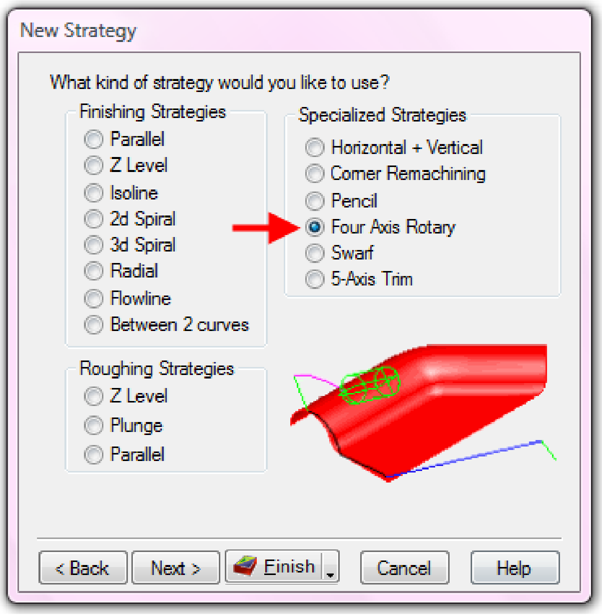


The stock and component are aligned with the X axis. In the stock properties form we told FeatureMILL3D that we were going to use 4th Axis Positioning and were going to rotate around the X axis. We now need to create a surface milling feature.

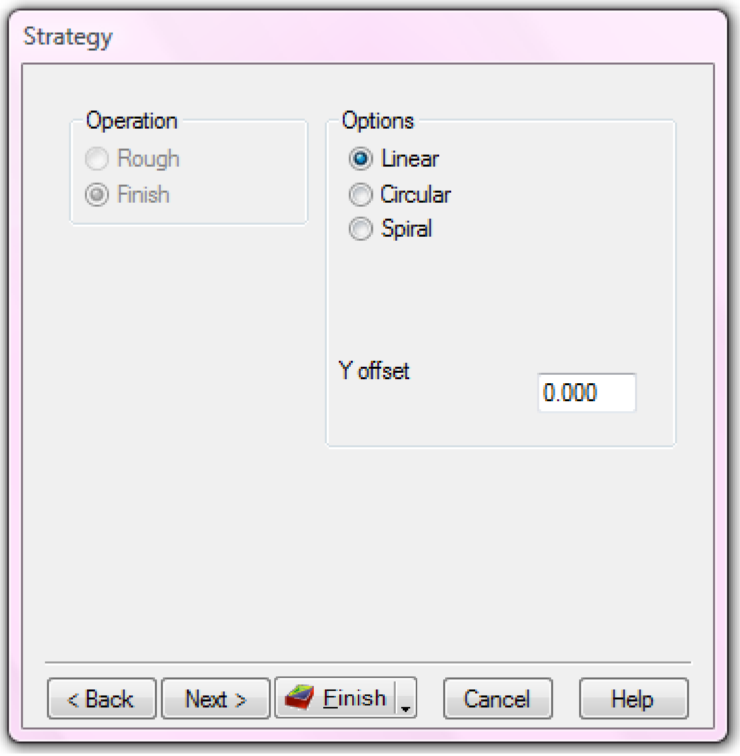
- Select all of the surfaces

- Create a new **surface milling feature** and **Next**
- Check **Choose a single operation** and press **Next**

As you will see there is now a new option on the New strategy form – Four Axis



- Check **Four Axis Rotary** and press **Next**
- Fill in the form as shown below



The Y offset allows the tool to be offset away from the default Y zero position. This allows the user to prevent the tool from center cutting.

- Press **Finish**
- Press **OK**

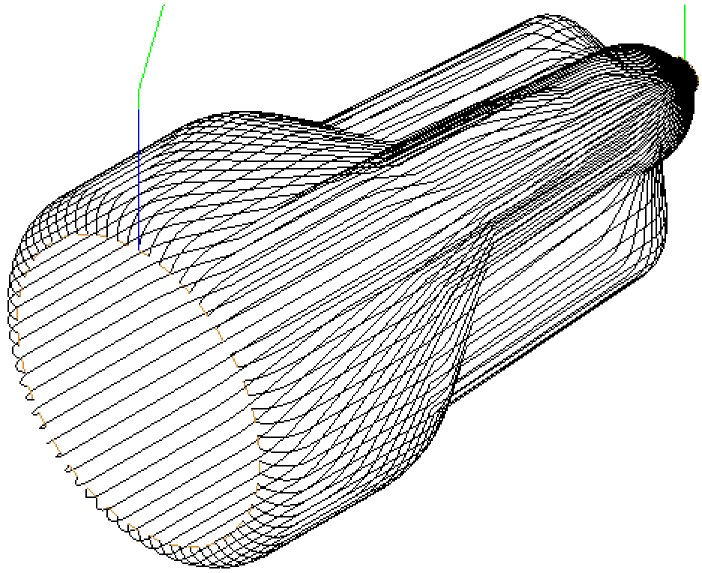
- From the main menu select **Manufacturing** then **Post Process**

- Set the **Tool change location Z** to **12"**
- Click on the Browse button and select the **HaasVF-4axis** post from:

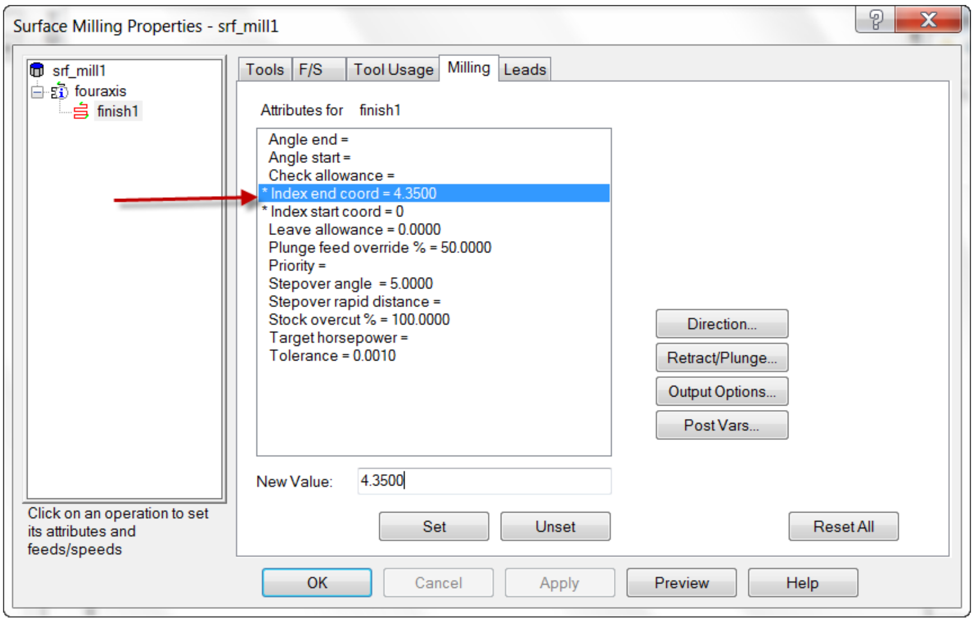
  C:/Program Files/FeatureCAM/M-Library/4thxs

- Press **OK**
- Run a **Centerline simulation**

It can be seen that the toolpath is rolling over the ends of the part. We need to limit the extents of the toolpath to give the desired result.

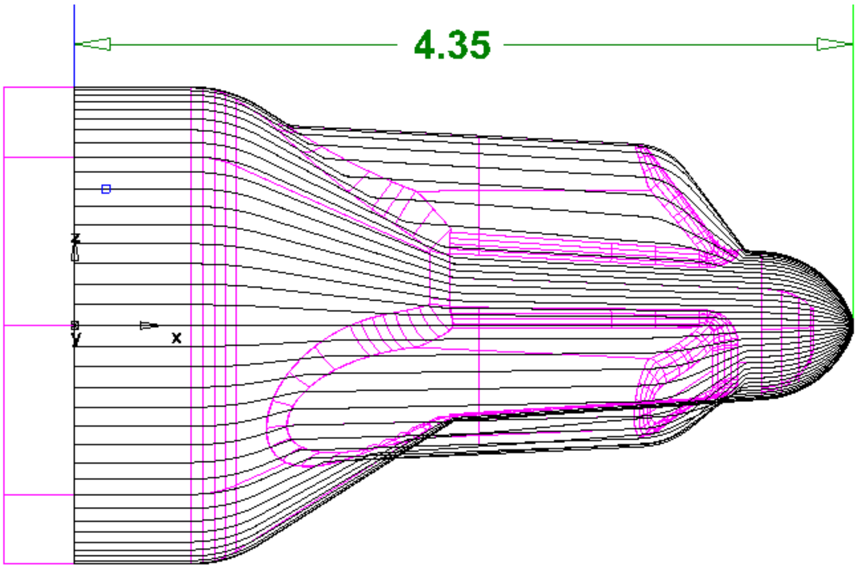


- **Eject** the **simulation**
- Edit the feature
- Click on **finish1** in the tree view
- Select the **Milling tab**
- Click on **Index start coordinate** and set the value to **0**
- Click on **Index end coordinate** and set the value to **4.35**



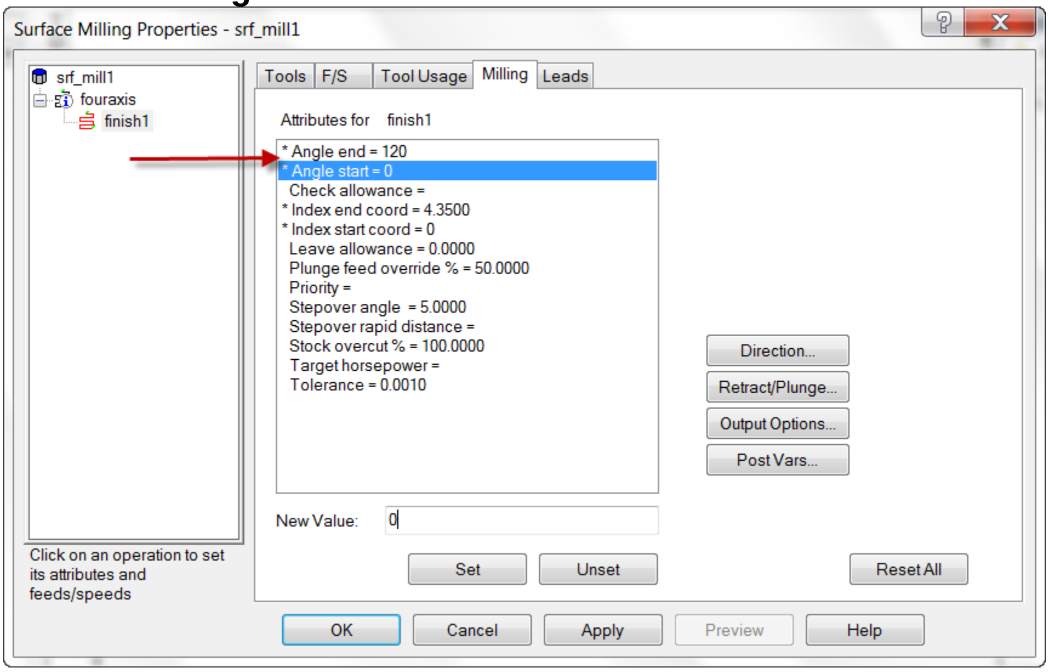
- Click **Apply** and then **OK**

- Run a **centerline simulation**



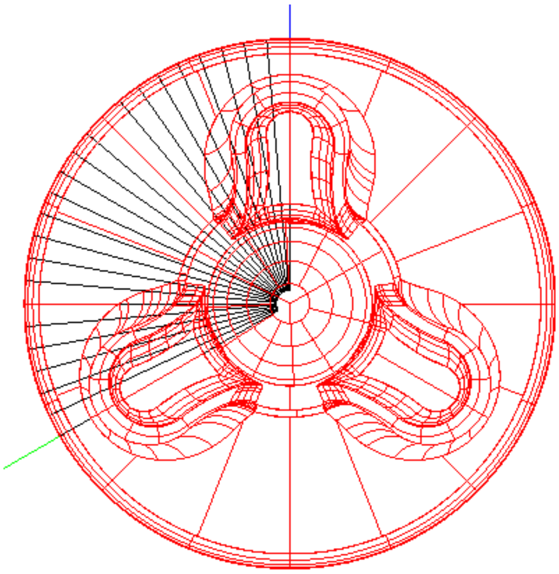
The toolpath is now limited to the portion of the feature between X 0.0" and X 4.35". It is important to remember that the X values given are from the setup origin to the **center** of the tool. It is also possible to apply an angular limit to the toolpath allowing the user to machine a selected area of the feature.

- Edit the feature
- Click on **finish1** in the tree view
- Select the **Milling tab**
- Click on **Angle start** and set the value to **0**
- Click on **Angle end** and set the value to **120**



- Click **Apply** and then **OK**

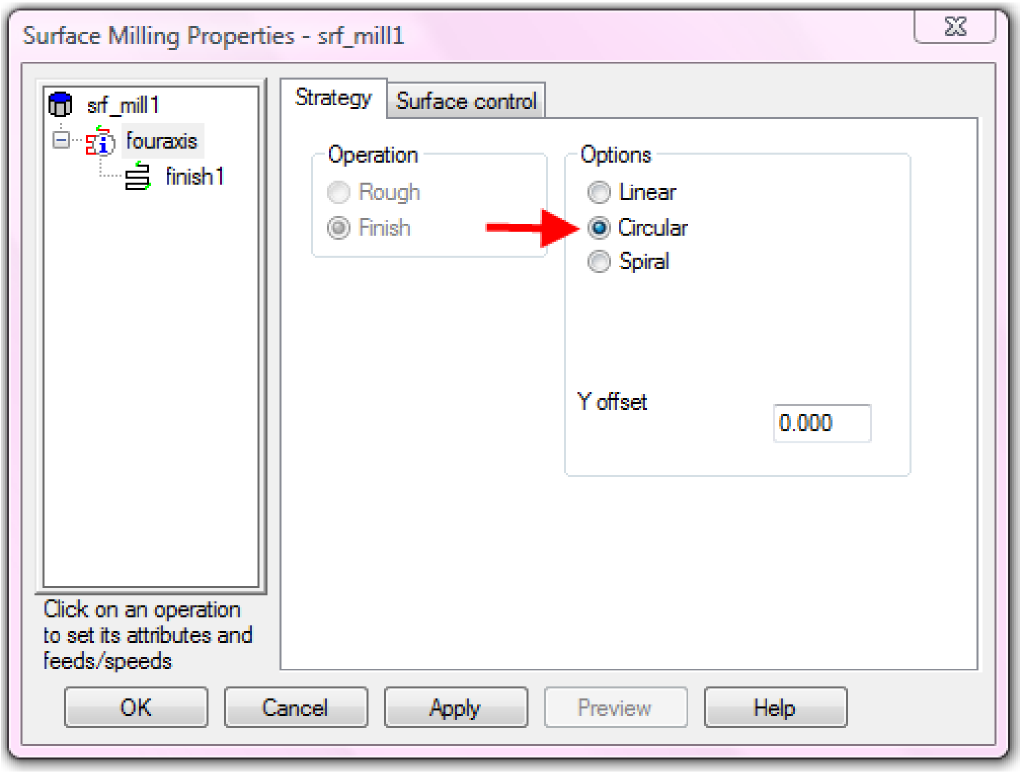
- Run a **centerline simulation**
- Select a **view** from the **Left**



It can be seen that only one third of the part has been machined. The start angle of 0 is aligned with the Z axis, and the end angle of 120 is in the clockwise direction around the X axis.

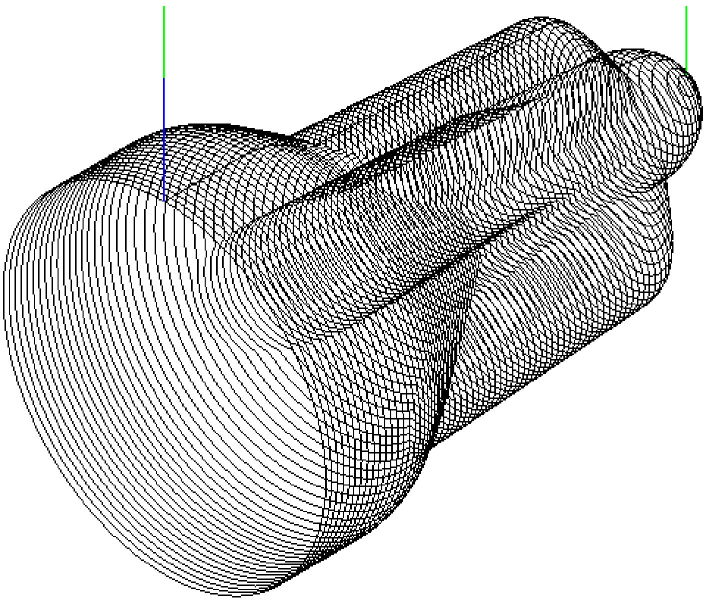
We will now look at the Circular and Spiral options of 4 axis machining.

- Edit the feature
- Click on **finish1** in the tree view
- Select the **Milling tab**
- Click on **Angle start** and then **Unset**
- Click on **Angle end** and then **Unset** and **Apply**
- Click on **fouraxis** in the tree view
- Check the **Circular** option on the **Strategy tab**



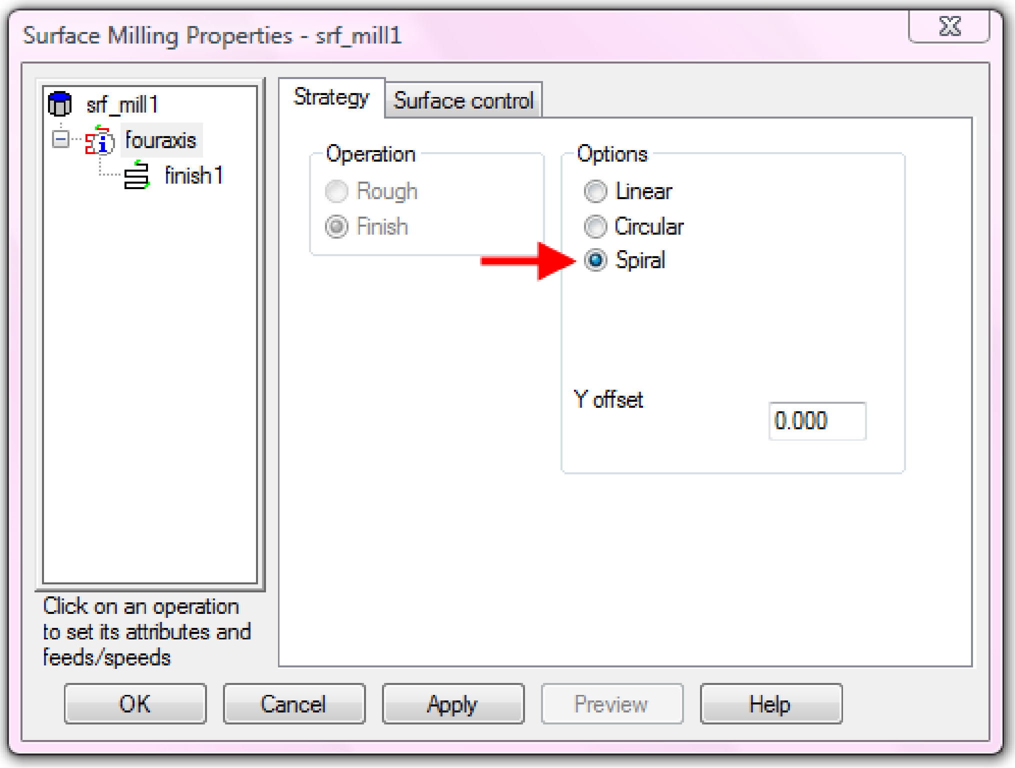
- Click **Apply** and then **OK**

- Select an **Isometric view**
- Move the **Simulation speed slider** to the **left** to slow down the simulation
- Run a **centerline simulation**

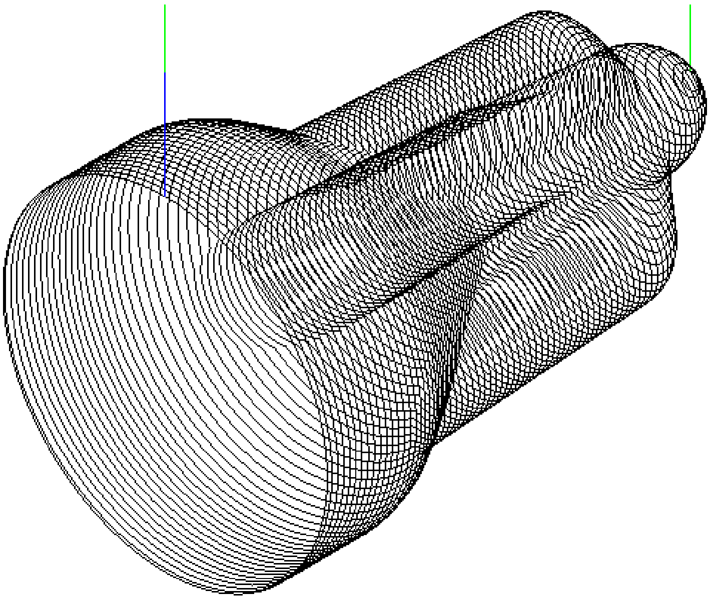


This time the toolpath runs around the part instead of along it. The direction of rotation reverses on each pass. The tool steps over on the surface of the part between passes.

- Edit the feature
- Click on **fouraxis** in the tree view
- Check the **Spiral** option on the **Strategy tab**



- Click **Apply** and then **OK**
- Select an **Isometric view**
- Run a **centerline simulation**

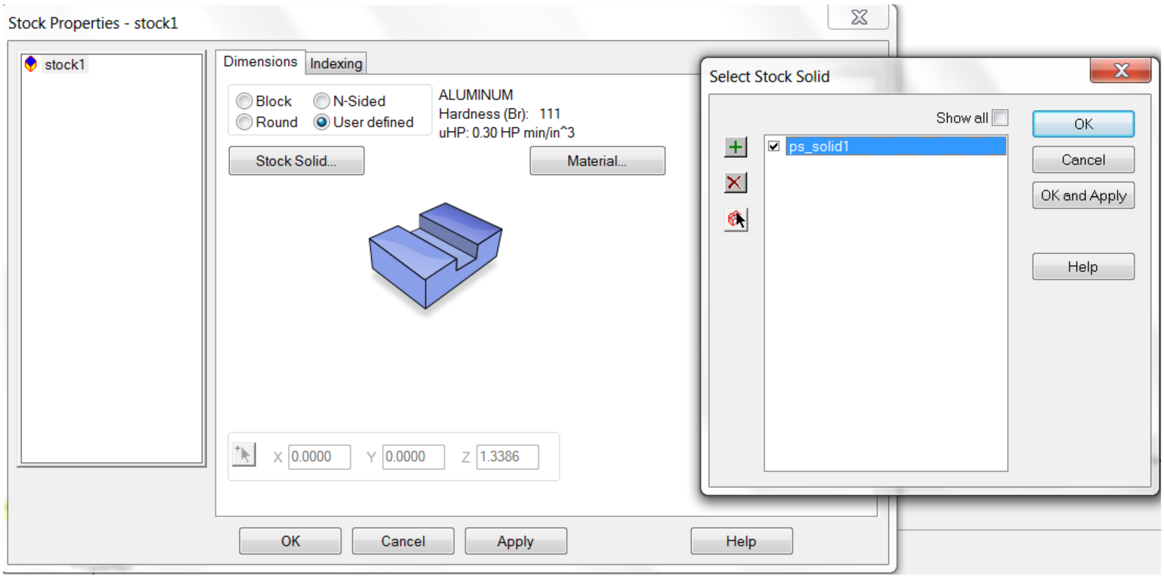


With the spiral option selected the toolpath follows a continuous spiral along the part. This will give a better surface finish than the circular option as there will be no dwell marks from stepover moves. The strategy will require a machine tool with a rotary axis which can rotate continually with no axis limit. We will now look at controlling the cutting direction.

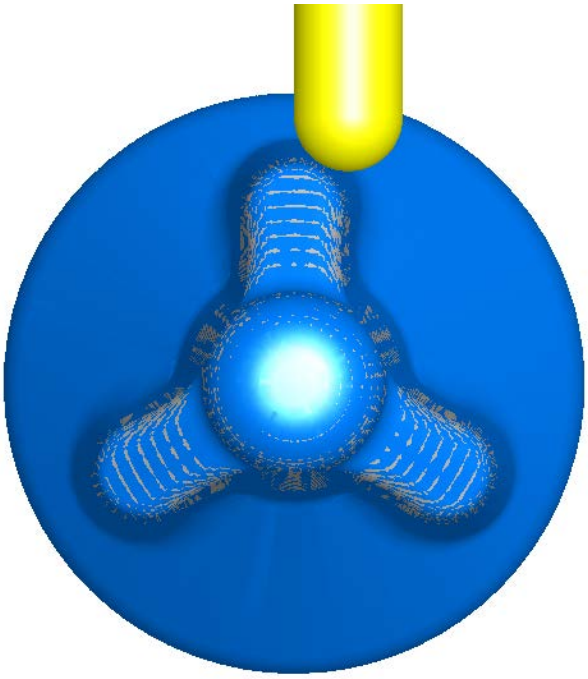
- Edit the feature
- Click on **finish1** in the tree view
- Select the **Milling tab**
- Click on **Index start** and set the value to **4.35"**
- Click on **Index end** and set the value to **0"**
- Click **Apply** and **OK**
- Run a **centerline simulation**

It can be seen that the X direction of cutting has now been reversed.

- Edit the feature
- Click on **fouraxis** in the tree view
- Set the **Y Offset** to be **0.25”**
- Click **Apply** and then **OK**
- Edit the stock properties and change to a User defined stock



- Select solid1
- Select a Right view
- Run a **3D simulation**



It can be seen that the cutter is now offset from the rotary axis by 0.25” in the Y direction. This means that the tool is not center cutting. This will create better cutting conditions giving improved surface finish and tool life.

Wall tolerance for Z-Level

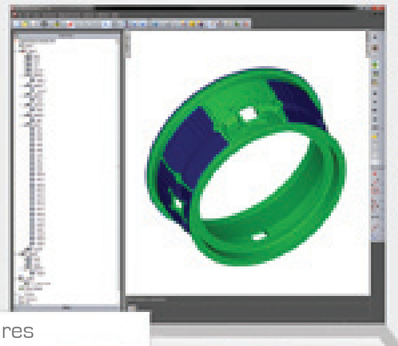
Overview

FeatureCAM decides whether a wall is vertical or not based on a tolerance. If the horizontal distance between the top and bottom edges of the wall is greater than the tolerance it is regarded as not being vertical and will be machined. There are two tolerances used, the "Part boundary tolerance" (PBT) and the "Wall tolerance". These are accessed through the Advanced button next to "Don't roll over edges at all" on the edges menu (please see attached image).

If "Use separate wall tolerance" is not checked then the verticality of the walls is based solely on the PBT. If the checkbox is marked then the PBT is used to assess the overall part boundary and the wall tolerance is used to assess whether walls are vertical or not. This allows the user to relax the tolerance for walls (so that more areas are seen as vertical) without affecting the overall quality of the part boundary.

FeatureCAM 2012

FeatureCAM 2012



new features

Delcam TV

www.delcam.tv

FeatureCAM Learning Zone

www.delcam.tv/lz

FeatureCAM Website

www.featurecam.com

THE QUEEN'S AWARDS FOR ENTERPRISE: INNOVATION 2010

THE QUEEN'S AWARDS FOR ENTERPRISE: INTERNATIONAL TRADE 2011

Powering your productivity

Delcam Headquarters | Small Heath Business Park | Birmingham | B10 0HJ | UK
+44 (0)121 766 5544 | marketing@delcam.com | www.delcam.com

To contact your local reseller, visit **www.delcam.com/resellers**