



FeatureCAM 2012 Training Course

FeatureCAM 2012

Training Course

FeatureMILL 3D Finishing



FeatureCAM

Copyright © 1995-2011 Delcam plc. All rights reserved.

Delcam plc has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of software.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence.

Delcam plc grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

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	7
Finishing Basics	9
Leave Allowance	9
Tolerance	
Stepover	11
Parallel Finish Machining	12
Overview	12
3D Spiral & Z Level Finishing	18
Introduction	18
Z Level machining	
3D Spiral machining	
Interleaved Z Level machining Horizontal and Vertical machining	
Corner & Pencil Finishing	36
Introduction	
Pencil Finishing	
Corner Along Finishing Corner Across Finishing	
Combo Along and Across Corner Finishing	
Corner Multi Pencil Finishing	51
Isoline Finish Machining	52
Introduction	52
Radial and 2D Spiral Finishing	65
Introduction	
Radial Finish Machining	
2D Spiral Finishing	
Flowline Finish Machining	77
Introduction	
Flowline Example	
Undercut Flowline Machining	

Between 2 Curves Finishing	90
Introduction	
Between 2 Curves Finish Machining	
3-Axis Swarf Finishing	94
Introduction	
3-Axis Swarf Finishing	
3-Axis Swarf – Multiple passes	103
4-Axis Rotary Finish Machining	107
Overview	107
4-Axis Finishing	
Wall tolerance for Z-Level	116
Overview	

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 3axis 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.

Õ

Finishing Basics

Introduction

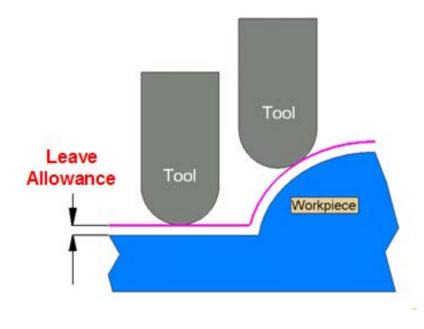
FeatureCAM has wide variety of methods for finish machining parts. The strategy that is chosen will depend upon the part geometry, the type of stock and the available cutting tools. By choosing the most appropriate combination of strategy and tooling the user can greatly reduce the time taken to finish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

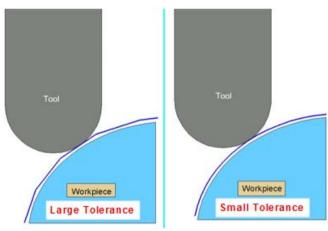
The Leave Allowance is the amount of material left on the part by the finish machining process. By default this is zero, i.e. the part will be finished to size. If the user wishes a positive leave allowance can be given, this will leave additional material on the part for subsequent processes - for example 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.



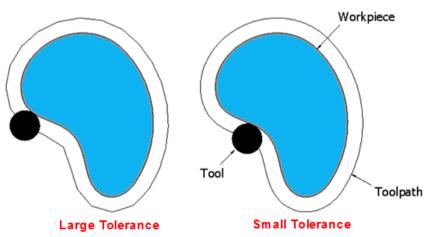
Tolerance

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 workpiece. 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).



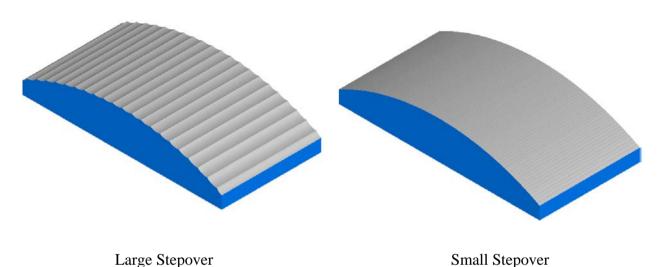
The effect of tolerance on a toolpath in a horizontal plane (Z Level toolpath).

<u>Note</u> 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).



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.

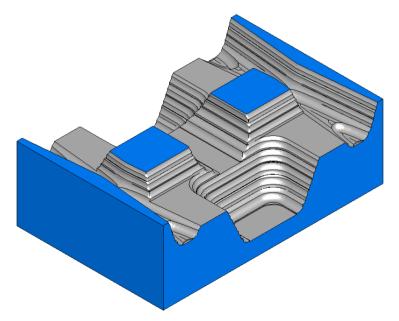
Parallel Finish Machining

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.

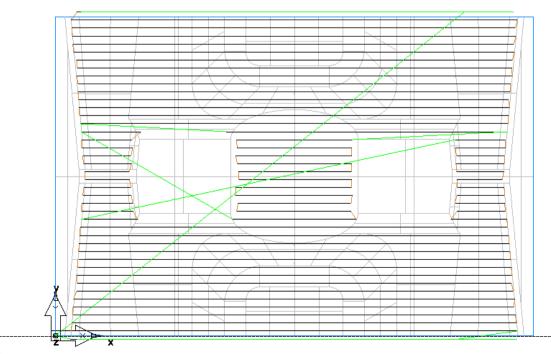
- Eject the simulation
- Edit the feature srf_mill1
- Select the Process tab
- Uncheck the Z Level rough operation

- 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

🖃 🛐 parallel	Tools F/S Milling Leads	
inish6	Attributes for finish6 Check allowance = * Leave allowance = 0.0250 Plunge feed override % = 50.0000	
	Priority = * Stepover = 0.2000 Stepover rapid distance = Target horsepower =	
	* Tolerance = 0.0100 Toolpath end = Toolpath start = Z end = Z start =	Direction
		Retract/Plunge Output Options
	New Value: .01	Post Vars
Click on an operation to set ts attributes and eeds/speeds	Set Unset	Reset All

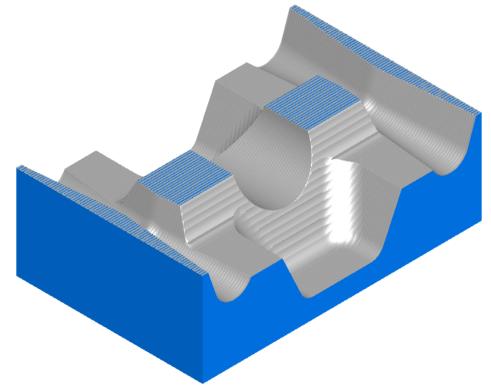
The **leave allowance** is telling FeatureCAM to leave 0.025" of material on the part for finishing. As this is a semi-finish operation, the surface finish is not important; this is why the stepover and tolerance have been increased to reduce the calculation time. As long as the tolerance is less than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

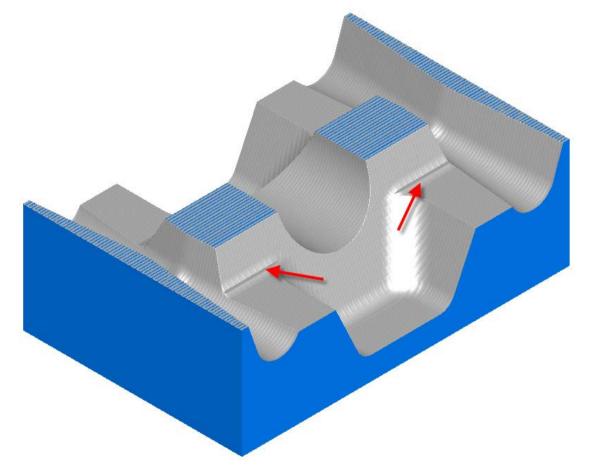


Note how the semi-finish operation removes the steps from the roughing. This provides a more constant tool loading for the subsequent finishing operations which will give a better surface finish.

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.



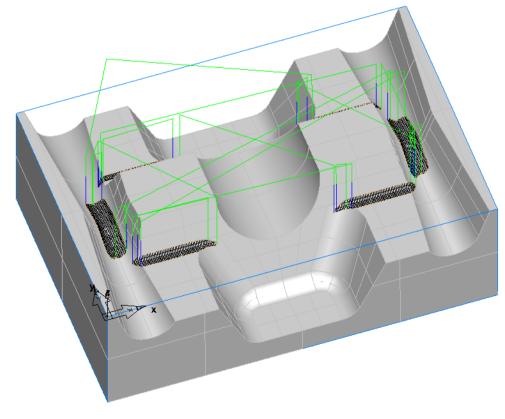
It can be seen however, that there are areas where the 1" ball nose cutter is not able to finish into the corners. We will now add another Parallel finishing pass with a smaller cutter which will re-machine only those areas where the larger ball could not reach.

- 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
---	---------	-----	------	---------	----	-------	-------

9 sf_mil1 - 글 rough1 - 글 rough5 - 글 parallel - 글 finish6 - 글 finish7	Operation Rough Finish	Options Ox parallel Or Y Parallel Parallel angle Parallel angle O.0 Add perpendicular remachining pass
	Remachining Previous tool diameter Overcut percent Minimum rest material	Steep slope angle 0.0 1.0000 5.00 0.0000 0.0000
lick on an operation to set s attributes and eeds/speeds	OK Cance	Apply Preview Help

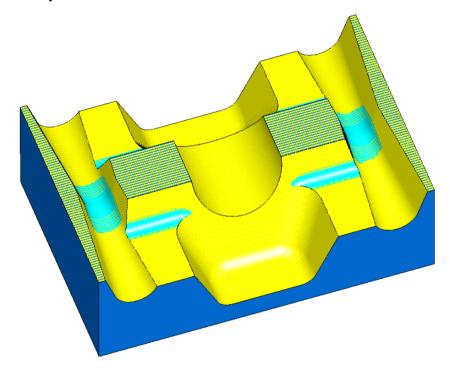
- •
- Click on **Apply** and then **Preview Play** the **Centreline simulation** •



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 stepover to machine the same part to the same surface finish.

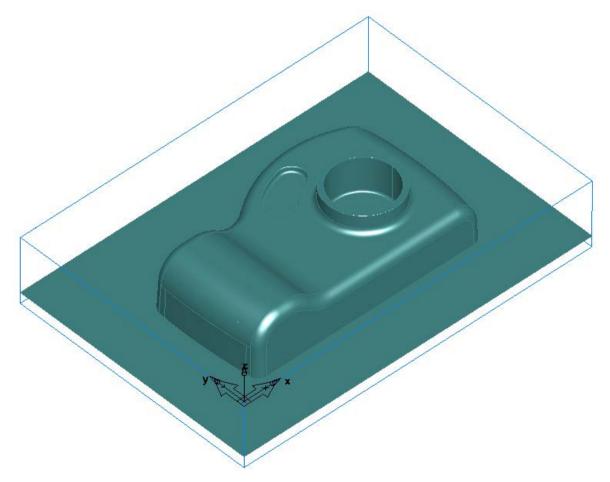
• Eject the simulation On the File menu select Close, DO NOT save the file

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.

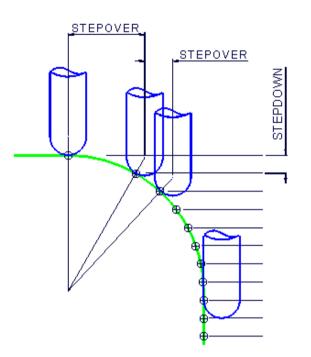
- Open the file camera.fm
- Select an isometric view



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.



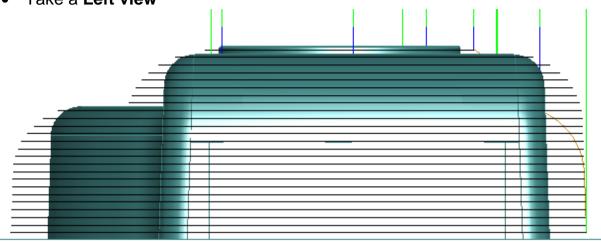
In Z level machining, a constant stepdown is used in the Z direction between each pass. As the component surface becomes shallow, the actual tool step over increases until it becomes non-existent on flat areas. For this reason it is a good idea to limit the toolpath to a slope boundary so that only the 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

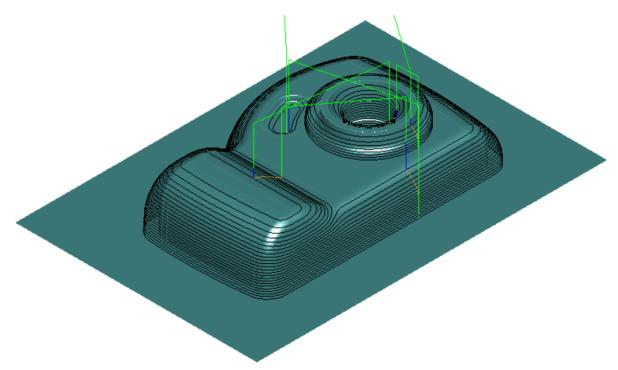
New Strategy What kind of strategy would y Finishing Strategies Parallel 2 Level Isoline 2 d Spiral 3 d Spiral Radial Flowline Between 2 curves Roughing Strategies Z Level Plunge Parallel	rou like to use? Specialized Strategies Horizontal + Vertical Comer Remachining Pencil Swarf 5-Axis Trim
< Back Next >	inish 🔪 Cancel Help

• 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.



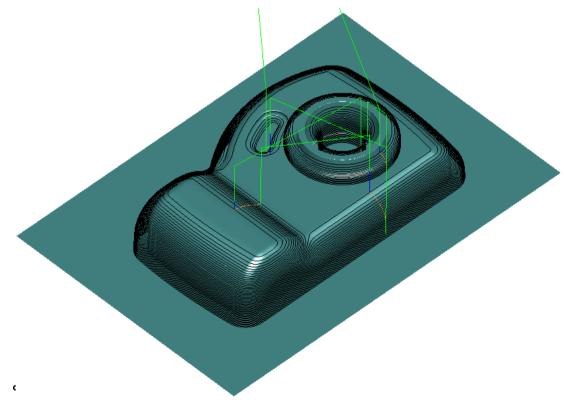
There are a number of ways in which we can remedy this problem. The first is simply to reduce the distance that the tool steps down between passes.

- **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

🗊 srf_mill1	Tools F/S Milling Leads	
i zievel i inish2	Attributes for finish2 Check allowance = Leave allowance = 0.0000 Plunge feed override % = 50.0000 Priority = Stepover rapid distance = Target horsepower = Tolerance = 0.0010 Z end = * Z increment = 0.0250 Z start = New Value: .025	 Scallop height Reorder Spiral Corner Correction: None Direction Direction Retract/Plunge Output Options Post Vars
Click on an operation to set its attributes and feeds/speeds	Set Unset	Reset All

• Run a Centerline simulation



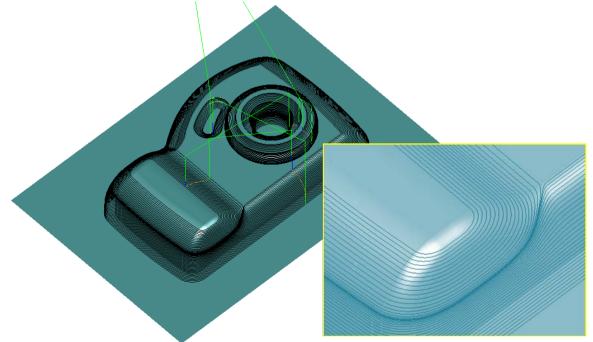
Note the improved surface finish in the shallow areas. However we are now making unnecessary extra passes on the steeper areas which will waste time.

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

Surface Milling Properties - sr		2 ×
srf_mill1 sī z level	Tools F/S Milling Leads Attributes for finish Eave allowance = Eave	Scallop height Reorder Spiral Corner Correction: None Direction Retract/Plunge Output Options Post Vars
Click on an operation to set its attributes and feeds/speeds	New Value: .025	ResetAll
	OK Cancel Apply	Preview Help

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



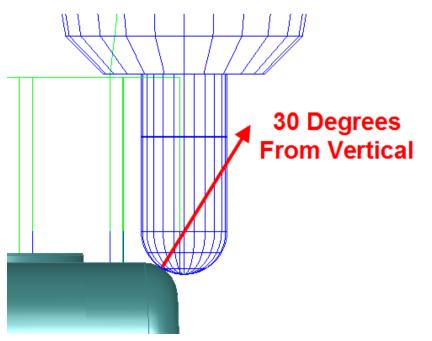
Note how the stepdown now varies to maintain the maximum scallop height. It should also be noted that where a part goes from a shallow to a steep area in the same height region the

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

Surface Milling Propert	ies - srf_mill1	_ X _
finish1 □ sf_mill1 □ sf_rinish1 □ sf_mill1 □ sf_nish1	Strategy Edges Stock Slopes 5. Slope limitations Image: None Image: Horizontal only Maximum surface slope (0.90) Image: Maximum surface slope (0.90) Image: Vertical only Minimum surface slope (0.90) Image: Minimum surface slope (0.90) Image: Vertical only Minimum surface slope (0.90)	Axis Surface control 60 30 Reset All
Click on an operation to set its attributes and feeds/speeds		
ОК	Cancel Apply Preview	Help

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.

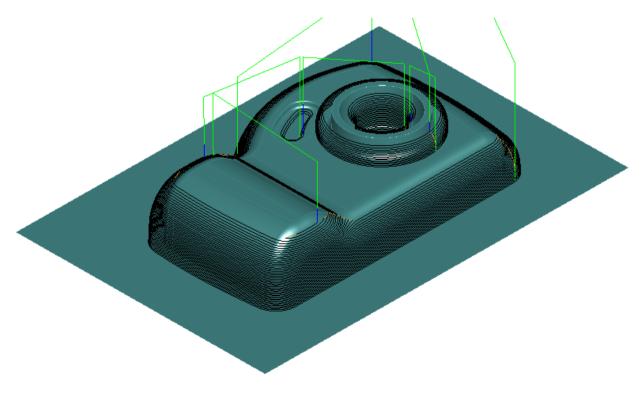


Setting a Steep angle will also have the additional effect of causing FeatureCAM to see the edges of the part as steep areas. To avoid the tool dipping down the side of the block we also need to tell FeatureCAM not to take the tool outside of the stock.

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

Surface Milling Properties - s	rf_mill1
Click on an operation to set	Strategy Edges Stock Slopes Surface control Choose the clipping curves for the material to be removed Use part surface dimensions Image: Use stock dimensions Image: Use solid model Image: Use solid model Image: Use solid center Image: Curve Options Contact point Tolerance 0.00100 Show Combined Boundary Stock Model
its attributes and feeds/speeds	OK Cancel Apply Preview Help

- Click Apply and then OK
- Run a Centerline simulation

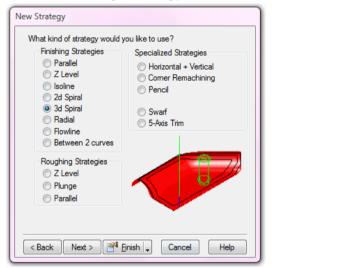


The shallow areas are no longer being machined. This is eliminating those parts of the model where a poor surface finish would have resulted and also reduced the length of time taken for the machining process. We can now select a different method to finish those areas to give a good finish over the whole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



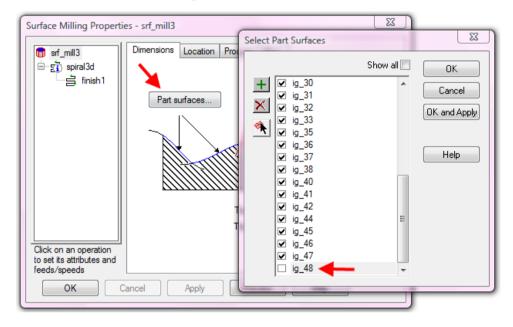
- Click Finish and then OK
- Run a Centerline simulation



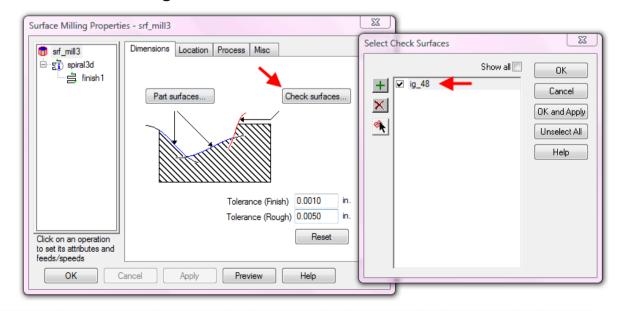
The first pass of the toolpath is following the rectangular outline of the flat surface. This pass is then offset inwards to produce the finished toolpath.

The default tool is a ¹/₂" 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



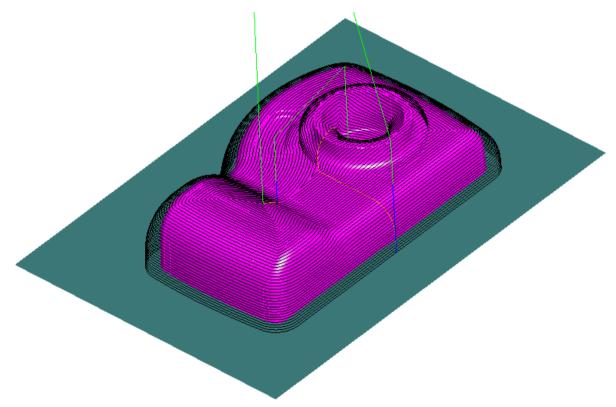
- Click OK and Apply
- Deselect the surface ig_48
- Click on Check surfaces
- Select the surface ig_48



• 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

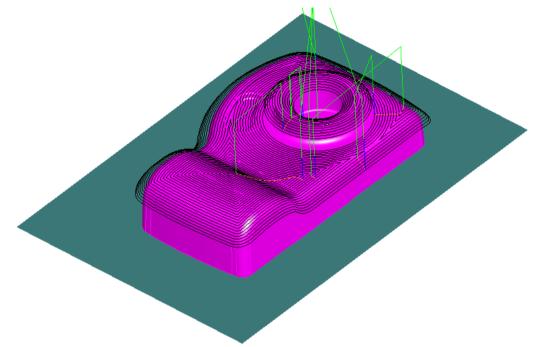


The toolpath is now starting at the outline of the selected surfaces and not wasting time cutting the flat area. This is still not a good toolpath, as it is starting at the bottom of the steep areas on the outside and cutting upwards. Notice also the way in which the toolpath rolls over the edge of the 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

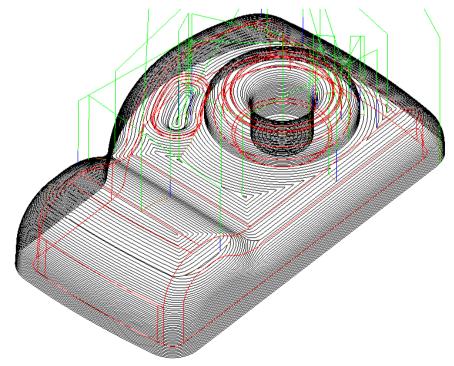
We have now told FeatureCAM that we only wish to machine those surfaces whose surface normal at the point of contact of the tool is inclined at less than 30 degrees to the vertical. This will coincide with the boundary we set earlier for the Z Level finishing toolpath.

• Run a **Centerline simulation**



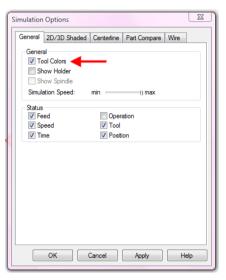
The toolpath is now confined to the shallow areas of the part.

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

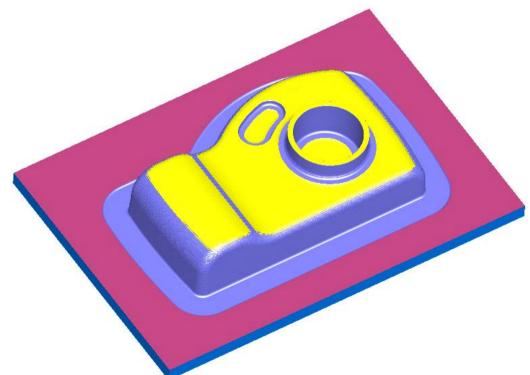


The two toolpaths now match at their edges finishing both the steep and shallow areas of the part. Due to the radius of the tool being used for the Z Level finishing toolpath it is not able to finish all of the way down the steep walls of the part. We will now change this.

- Eject the **simulation**
- Edit the **Z Level finish** operation
- Select the Tools tab and pick a 1/2" 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**

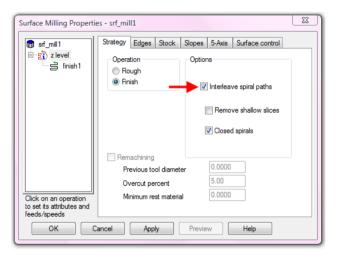


• As an exercise use a 2.5D feature to finish the lower, flat face of the part

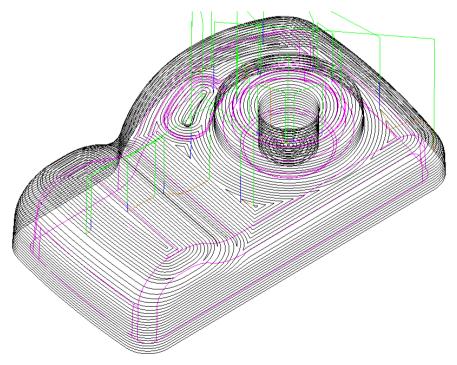
Interleaved Z Level machining

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

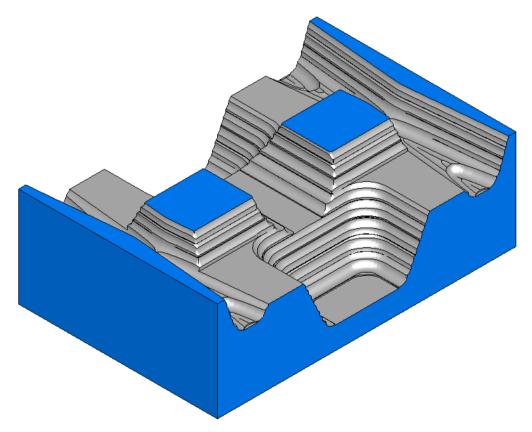


Note how the 3D spiral and Z Level parts of the toolpath are combined to reduce air moves. The toolpaths are reordered so that the part is cut from the top down reducing the risk of user error which can occur with manual boundary creation.

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

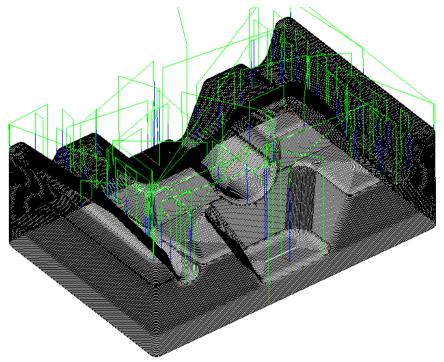
New Strategy	
What kind of strategy would s Finishing Strategies Parallel Soline 2 Level 3 d Spiral Radial Flowline Between 2 curves Roughing Strategies Z Level Plunge Parallel	you like to use? Specialized Strategies Ormer Remachining Pencil Swarf
< Back Next > Fin	ish Cancel Help

Click Finish

Surface Milling Propertie	es - srf_mill1		
🗊 srf_mill 1	Strategy Edges Stock	Surface control	
⊡ਾਸ਼੍ਰੇ horizontal + ve ਂਛੇ finish7 ਂਛੇ finish8	Operation Rough Finish	Options Spiral in Spiral out Parallel	
		Slope boundary Slope overlap	60.0 0.0
Click on an operation to set its attributes and feeds/speeds			
ОКС	ancel Apply	Preview	Help

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**



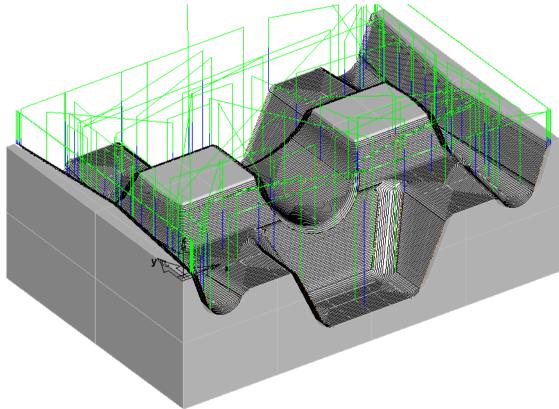
The part has been finished, however there are a number of improvements that could be made to the toolpath. We do not want to machine the upper flat surfaces; we can fix this by lowering the **Z Start** value for the 3D Spiral toolpath. The Z Level finish is machining down the side of the part; we can 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

া srf_mill1 লি হট horizontal + vertical লি চা finish7	Tools F/S Milling Leads Attributes for finish7	
â finish8	Check allowance = Leave allowance = 0.0000 Plunge feed override % = 50.0000 Priority = Stepover = 0.0500 Stepover rapid distance = Target horsepower = Tolerance = 0.0010 Z end = * Z start = -0.0100 New Value:	Direction Retract/Plunge Output Options Post Vars
Click on an operation to set its attributes and feeds/speeds	Set Unset	Reset All
	OK Cancel Apply	Preview Help

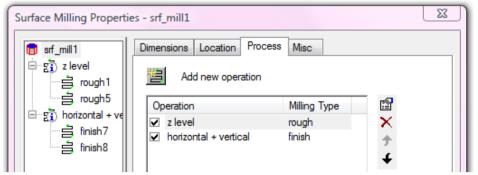
Issue FeatureMILL3D 2012

- Click on the Horizontal & Vertical operation
- Select the Stock tab
- Set the stock Overcut % to Zero then click Apply Surface Milling Properties - srf_mill1 X Strategy Edges Stock Surface control 🗊 srf_mill1 horizontal + vertical Choose the clipping curves for the material to be removed 음 finish7 음 finish8 Ouse part surface dimensions Use stock dimensions Overcut % 0 OUse solid model 🙊 ps_solid1 -Allowance 0.00000 -Limit-Select curves for boundaries Tool center Curve Options Contact point Tolerance 0.00100 Show Combined Boundary -Stock Model Click on an operation to set its attributes and feeds/speeds ОК Cancel Apply Preview Help
- Click **Apply** and then **OK**
- Run a Centerline simulation



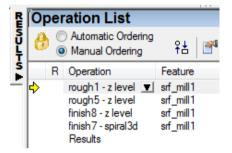
The 3D Spiral toolpath is now avoiding the upper surfaces and the Z Level does not finish the outside of the block. We will now turn on the roughing and reorder the toolpath sequence.

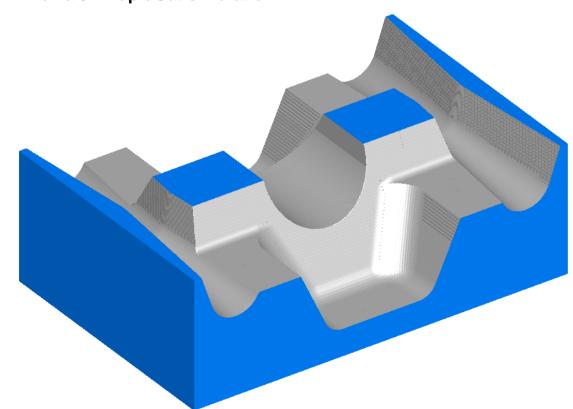
- Click on srf_mill1 and select the Process tab
- Check the **Z Level** operation



- 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**

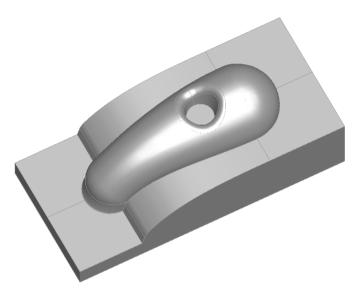
Corner & Pencil Finishing

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.

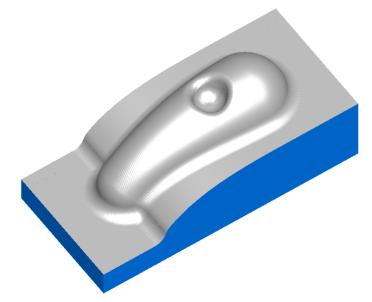
- Open the file Cowling.FM
- Select an Isometric view

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

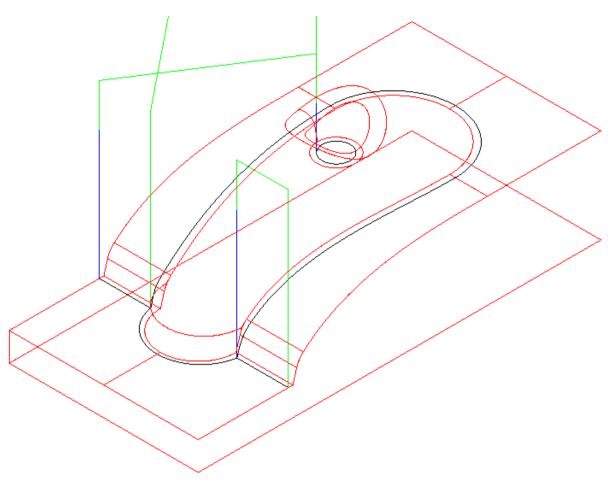
Finishing Strategies Parallel Z Level Isoline 2d Spiral 3d Spiral Radial Flowline Between 2 curves	Specialized Strategies Horizontal + Vertical Corner Remachining Pencil Swarf
Roughing Strategies Z Level Plunge Parallel	

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

Surface Milling Propertie	es - srf_mill2			
🗊 srf_mill2	Tools F/S Millin	g Leads		
ि⊡ि∑ो pencil िििहो finish1	Criteria to restrict tools	s in list:		
	Tool Group:	U Ball End Endi	-	
	Diameter:	Anything	-	
	End-Radius:	Anything	-	
	🛃 🦄 🦉 😭	Curvature-base	d	
	Name 🛓	Diam. End .	Flutes Cu	itter L 🔺
	endmillB00625:.	0.0625 0.0313	4 0.	5000 ≡
	endmillB009375			5000
	endmillB0250:4r			7500 0000
	endmillB0375:4r	0.3750 0.1875		7500 +
Click on an operation to set its attributes and	<			•
feeds/speeds				
ОК С	ancel Apply	Preview	Help	

• 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.

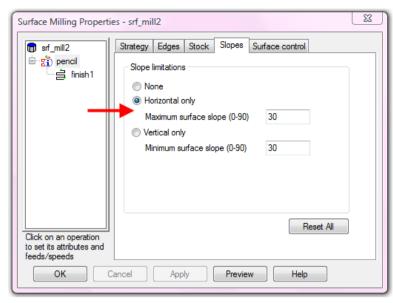
Horizontal only limits the cutting to regions with a slope less than the user defined maximum surface slope.

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

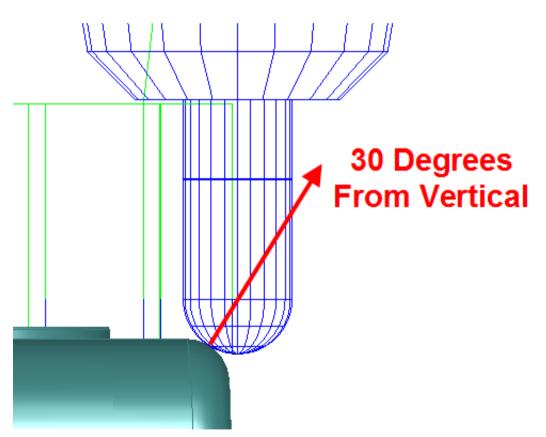
Limiting the regions where an operation is performed can solve the potential problems associated with the tool running up or down steep slopes. For instance the user could apply a stitch strategy to track across the steep areas and parallel strategy for shallow areas. Also a higher feed rate could be 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

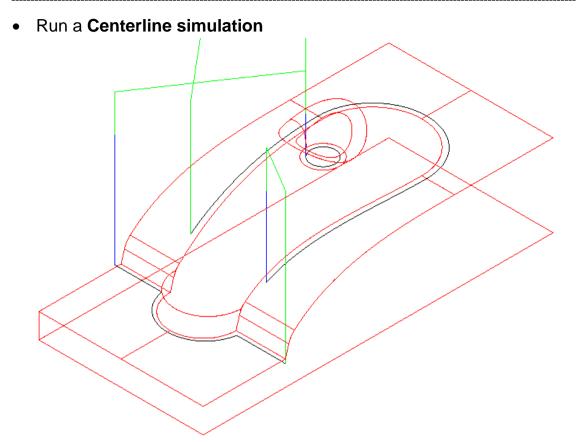
• Set the Maximum surface slope to 30



• 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.

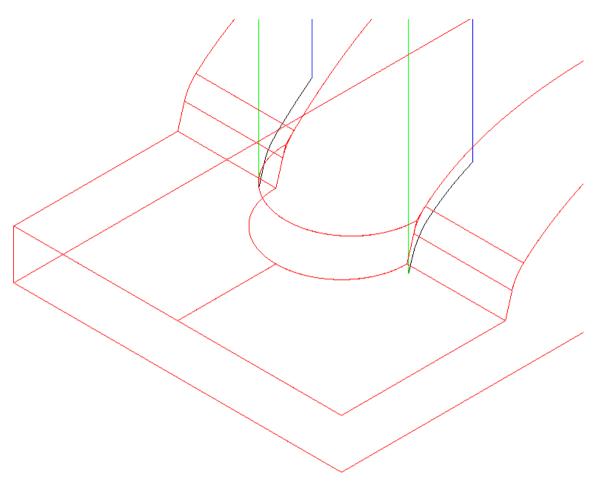


The toolpath is now only cutting the shallow areas of the model and avoiding the steep areas.

- 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

Surface Milling Propert	ies - srf_mill2	
n sf_mil2 ⊟-n pencil 금 finish1	Strategy Edges Stock Slopes Su Slope limitations None Horizontal only Maximum surface slope (0-90)	iface control
	 Vertical only 	
Click on an operation to set its attributes and feeds/speeds	Minimum surface slope (0-90)	30 Reset All
	Cancel Apply Preview	Help

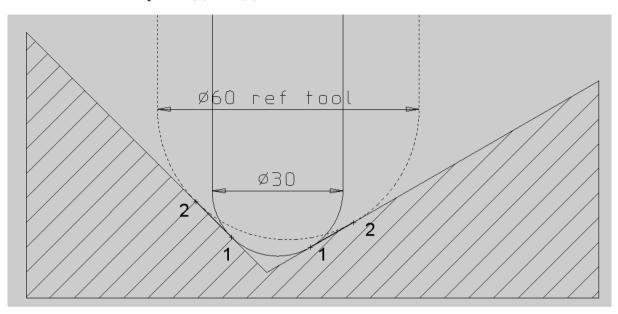
• Run a Centerline simulation



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.

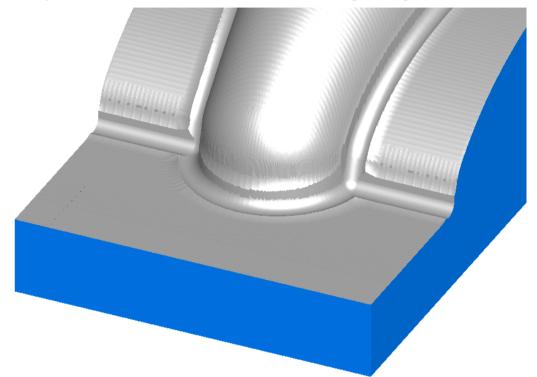


- Check the surface milling feature named srf_mill1 in the Part View
- Edit the pencil milling feature
- On the Slopes tab set Slope limitations to None
- Click Apply and then OK

Surface Milling Propert	ies - srf_mill2
Click on an operation to set its attributes and feeds/speeds	Strategy Edges Stock Slopes Surface control Slope limitations None Horizontal only Maximum surface slope (0-90) 30 Vertical only Minimum surface slope (0-90) 30 Reset All Reset All
ОК	Cancel Apply Preview Help

• 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

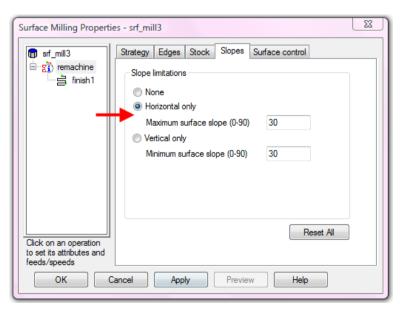
New Strategy What kind of strategy would y	rou like to use?
Finishing Strategies Parallel Z Level Isoline 2d Spiral 3d Spiral Radial Radial Between 2 curves	Specialized Strategies Horizontal + Vertical Comer Remachining Pencil Swarf
Roughing Strategies Z Level Plunge Parallel < Back Next >	Pinish Cancel Help

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

Strategy	
Operation Rough Finish	Options Multi pencil Along Across Combo Along and Across
✓ Remachining	Detection angle 165.0
Previous tool diameter	0.750000
Overcut percent	5.00
Minimum rest material	0.0000
< Back Next > Market Parket Pa	Finish 🔪 Cancel Help

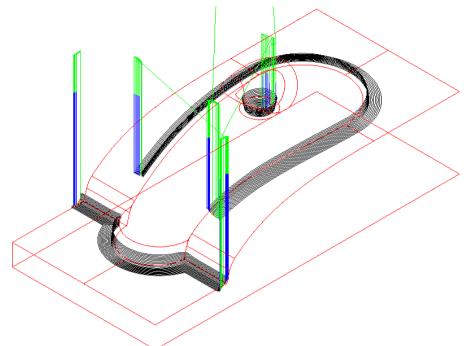
Nb. The Overcut percent figure defines the overlap of the Remachining operation over the original finishing operation. By default this is 5% which will overlap the two toolpaths sufficiently to eliminate any witness marks where the toolpaths meet.

- 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



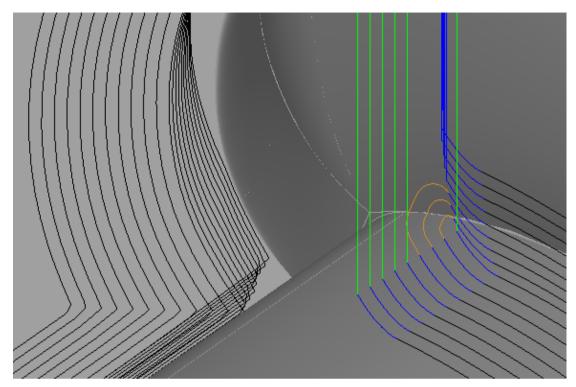
The shallow areas are being cleared of excess material from the outside inwards in a climb milling direction. There are multiple lift offs resulting in a lot of "air time". The tool is also plunging directly into the remaining material. We can remedy both of these problems by modifying the leads and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

Surface Milling Propertie	s - srf_mill3			
ft sf_mill3 ⊡ - gj) remachine ⊡ ≓ finish1	Tools F/S Milli Stepover type Lead-in/out option Ramp to depth Ramp angle 5	Loop	• Helical:	
	Use lead-in/out: Lead-in/out plane:	Oi Ve	n all stepovers, plunges ertical O Use linear lead	
	Ramp diameter	0.2500	Lead-in angle	30
	Ramp-in angle	60	Lead-out angle	30
	Ramp-out angle	60	Lead-in length	0.1375
	Use linear appro	ximation	Lead-out length	0.1375
Click on an operation to set its attributes and feeds/speeds				Reset All
ОК Са	ncel Apply	Pr	eview Help	

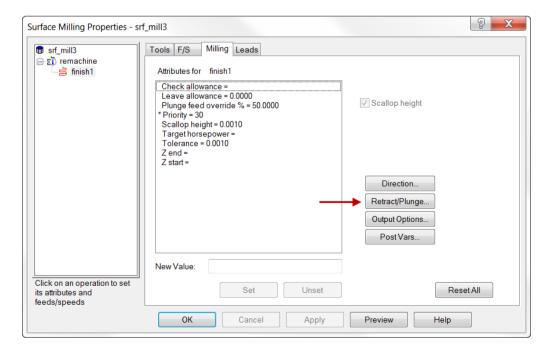
• Click **Apply** and then **OK**

• Run a Centerline simulation

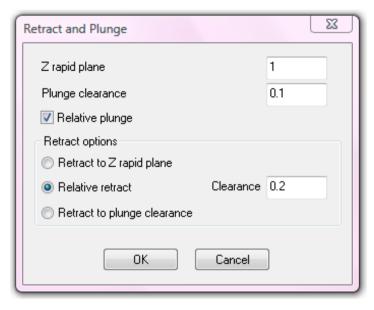


The tool is now arcing on and off the part smoothly. Short links are joined by loops. Longer links are still lifting off the job. We will now reduce the amount of time spent in the air by switching to Relative Plunge/Retract.

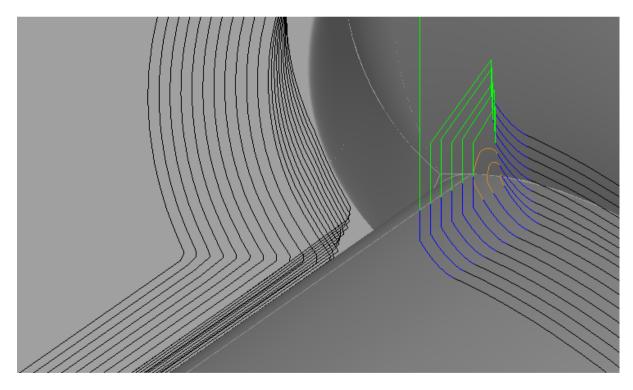
- Eject the simulation and Edit the feature
- Click on **Finish1** in the tree view and click on the **Milling tab** Note: The surface finish is controlled by the scallop height value
- Click on **Retract/Plunge**



• 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.

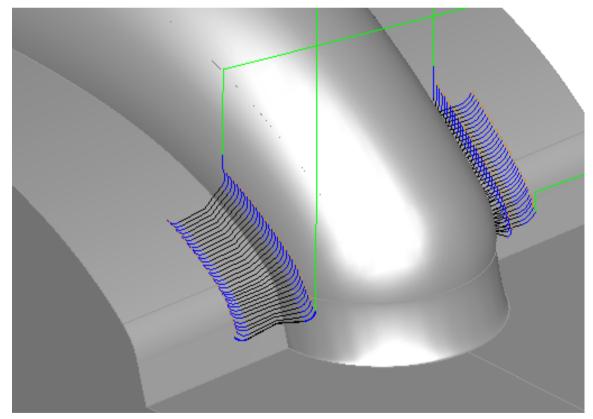
Relative retract instruct FeatureCAM to retract by only enough to maintain the specified clearance from the part when moving at rapid from one toolpath pass to the next. Another

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



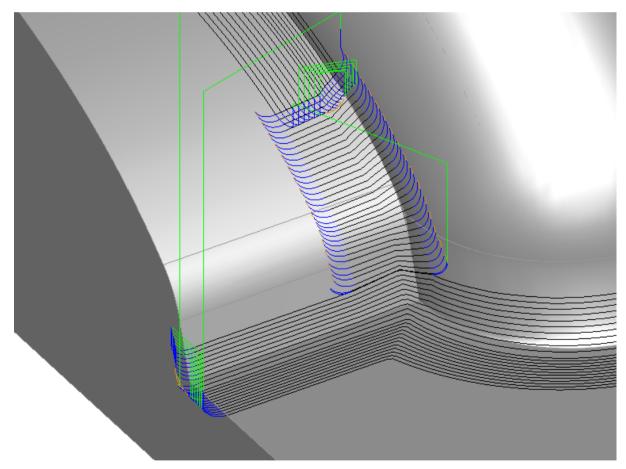
The tool stitches across the corner in such a way that each pass follows the surface normals of the intersecting surfaces at the point of contact of the tool. On a horizontal corner the place of the toolpath passes would be vertical, when the corner is vertical the toll will make passes in

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

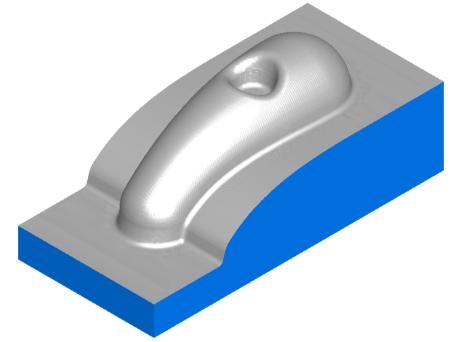
In the examples above we have seen how steep and shallow corners can be machined using Along and Across strategies respectively. By creating two operations and choosing appropriate slope angles we can machine all of the corners on a part. In order to make this easier FeatureCAM has another strategy which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

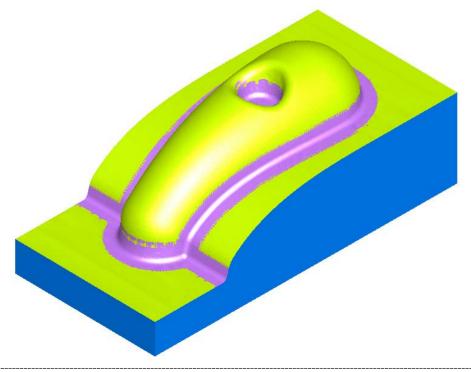


This has machined the shallow areas using an Along strategy and the steep areas using Across. This gives the optimum surface finish whilst protecting the tool in steep areas. The two parts of the toolpath meet exactly at the point where the slope angle changes from steep to shallow. Where the smoothness 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**

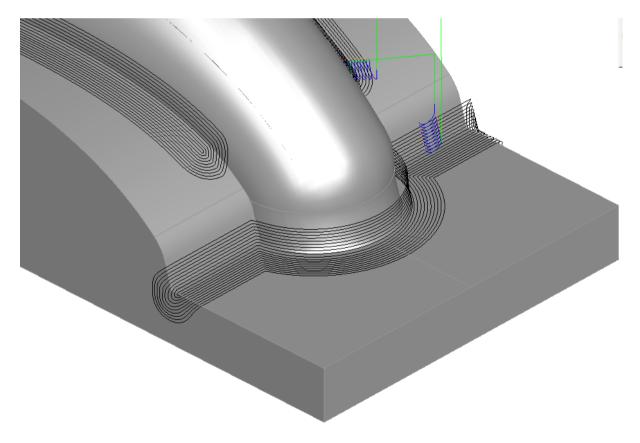


You can now clearly see the area where the Remachining has removed the excess stock.

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



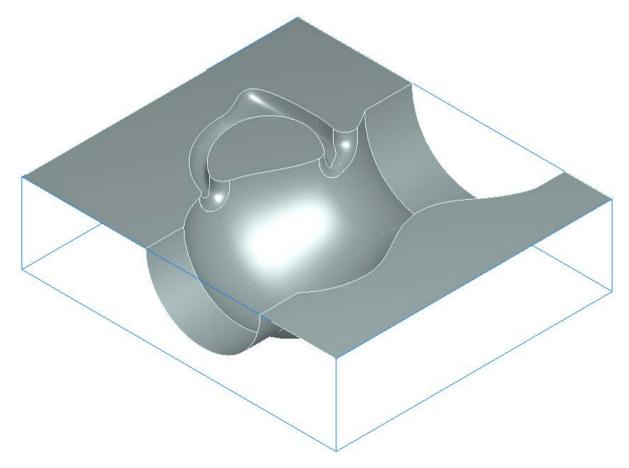
This toolpath is similar to the **Along** strategy except that it doesn't lift off the job as much. The only disadvantage is that it alternates between climb and conventional milling.

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.

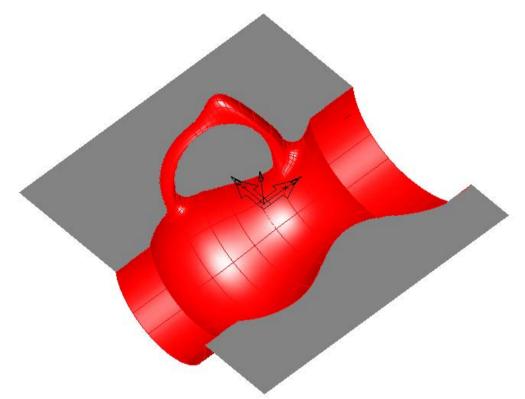
- Open the part Isoline.fm
- Load the tool crib Isoline.fm_Tools_from_last_save
- Select an Isometric view



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

Swarf

Click Finish

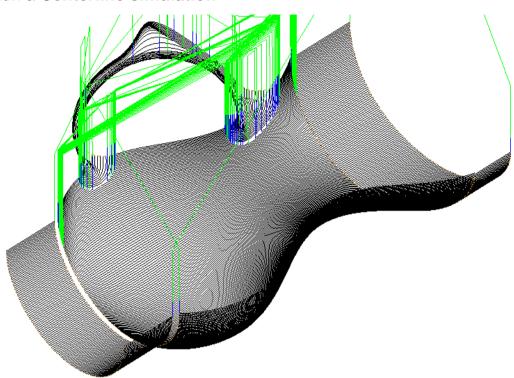
• Click on finish1 and select the Tools tab

Surface Milling Propertie	s - srf_mill2 Tools F/S Milling	Leads		23
⊡ :: isoline ∴ : finish1	Criteria to restrict tools i			
	Tool Group:	📙 Ball End Endi 👻	-	
	Diameter: A	nything 🗸	Ī	
	End-Radius: A	nything 🗸	-T	
	🧏 🖑 🦉 😭	Curvature-based		
	Name 🛓	Diam. End .	Flutes Cutter L 🔺	
	endmillB0750:4r.	0.7500 0.3750	4 1.6250	
	endmillB1000:4r.	1.0000 0.5000	4 2.0000	
	endmillB1250:4r.	0.1250 0.0625	4 0.7500	
	endmillB1875:4r.	0.1875 0.0938	4 0.7500	
	endmillB4375:4r.	0.4375 0.2188	4 1.2500 👻	
Click on an operation			Þ	
to set its attributes and feeds/speeds				
OK Ca	ancel Apply	Preview	Help	

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

Surface Milling Properties - srf_mill2	₽ X
Image: String isoline Image: String isoline <t< td=""><td>Spiral Direction Retract/Plunge Output Options Post Vars Reset All Preview Help</td></t<>	Spiral Direction Retract/Plunge Output Options Post Vars Reset All Preview Help

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



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

Surface Milling Propertie	es - srf_mill2			23
finil2 ⊡ isoline inish1	Strategy Edges Attributes for iso Surface	Stock	control	
	ig_shell2 ✓ ig_shell4 ✓ ig_shell6 ✓ ig_shell7 ✓ ig_shell8 ✓ ig_shell9	First Row First Row First Row	Increasing Increasing Increasing Increasing Increasing Increasing	
Click on an operation to set its attributes and feeds/speeds OK C	ancel Appl	y Previe	ew Help	

• Run a Centerline simulation

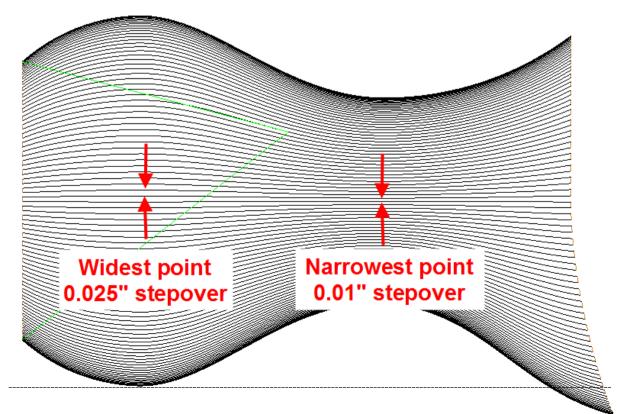
The surfaces are machined in the order in which they appear in the surface control list. Surfaces may be moved up or down the list by selecting them and then using the up and down arrows to the right of the list.

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

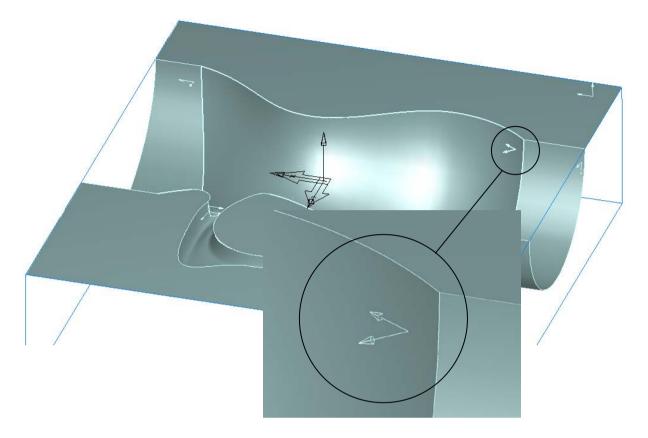
for srf_mill2 ⊡…∰ isoline finish1	Strategy Edges Attributes for is	otoor	e control
	Surface	Start Curve First Row First Row First Row First Row First Row First Row	Cut direction Increasing Increasing Increasing Increasing Increasing Increasing

- 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".



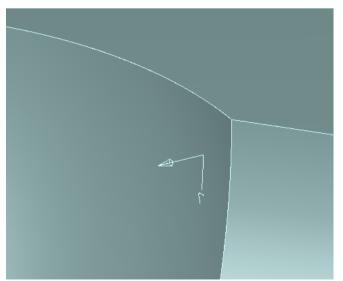
- Edit the feature
- Click on Isoline in the tree view and select the Surface control tab
- Click on ig_shell9 in the surface control list



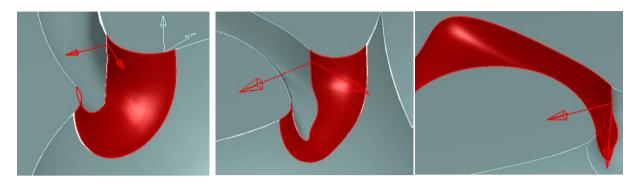
The pair of arrows on each of the surfaces shows the surface normal and the direction of the first cut. The surface normal arrow must point towards the side of the surface that you want to be machined. The direction arrow indicates whether the first cut is to be along the surface or across and which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

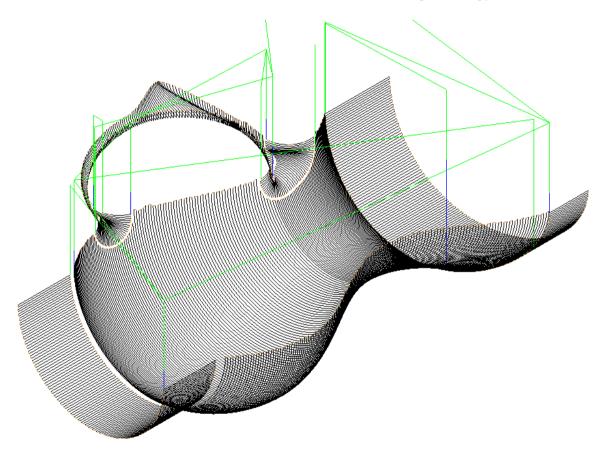


ig_shell6

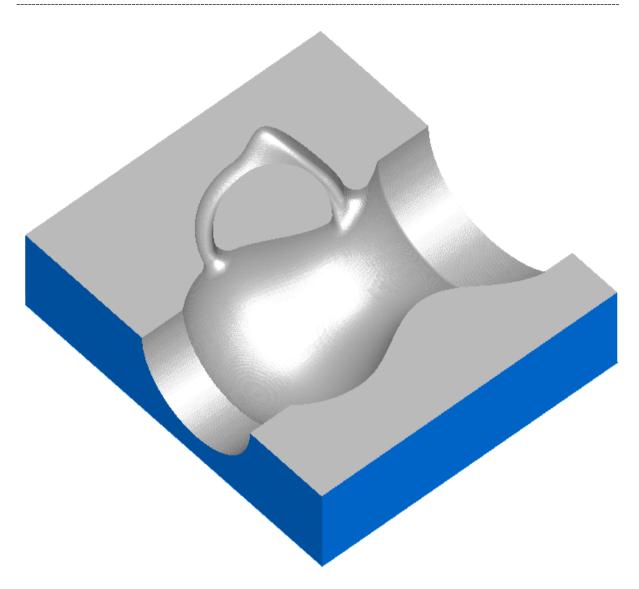


ig_shell8

- 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

Next>

< Back

Cancel

• Check the **Advanced** toolbar

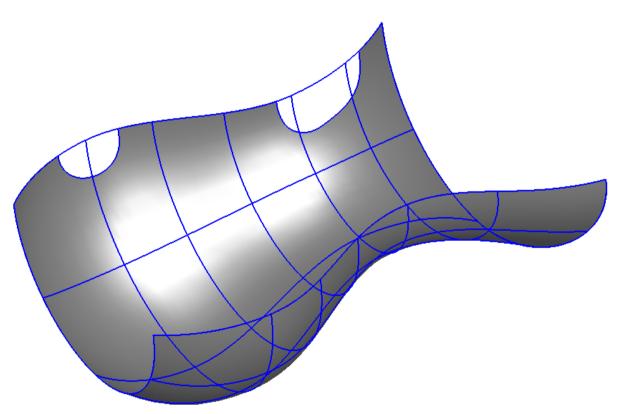
olbars Commands Misc.				
oolbars:				
✓ Standard ✓ Advanced	New		Surface	\ <u>\</u>
Curves and Surfaces	Reset Selected		Surface	VVIZ
✔ Snap Mode ✔ Display Mode	Delete			
✓ Geometry ✓ Simulation	Default Toolbars			
✓ Steps ✓ Solid		ļ		
More Simulation				
	Button size:	Small	This mines the Advenced	4 11-
	Buildin Size.	C Large	This raises the Advanced	
	Style:	 Classic 	from which you can acces	ss the
oolbar name:		Shaded Grey	surface wizard.	
Advanced		Glass		
	OK Canc	el Help		
	OK Canc	Help		
Click OK	OK Canc	Help		
		eel Help		
Click on the Su	rface wizard	Help		
Click on the Su Check From or	rface wizard ne surface	eel Help		
Click on the Su Check From or Check Untrim s	rface wizard ne surface	el Help		
Click on the Su Check From or Check Untrim s	rface wizard ne surface	el Help		
Click on the Su Check From or Check Untrim s	rface wizard ne surface	el Help	×	
Click on the Su Check From or Check Untrim s Click Next	rface wizard ne surface	el Help	×	
Click on the Su Check From or Check Untrim s Click Next	rface wizard ne surface surface	eel Help	×	
Click on the Sur Check From or Check Untrim s Click Next Surface Wizard What method of constru-	rface wizard ne surface surface uction do you want?		×	
Click on the Sur Check From or Check Untrim s Click Next Surface Wizard What method of constru	rface wizard ne surface surface uction do you want?		×	
Click on the Sur Check From or Check Untrim s Click Next Surface Wizard What method of constru-	uction do you want?	e surface	×	
Click on the Sur Check From or Check Untrim s Click Next Surface Wizard What method of constru- © From curves © Primitive surface	rface wizard the surface surface uction do you want? () From one () From mul or do you want?	e surface	×	
What method of constru From curves Primitive surface What specific constructor	rface wizard ne surface surface uction do you want?	e surface		
Click on the Sur Check From or Check Untrim s Click Next Surface Wizard What method of constru- © From curves © Primitive surface What specific construct © Region from Sur	rface wizard ne surface surface uction do you want?	e surface		

Help

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

Untrim a Trimmed Surface	X
Surface name: srf3	 Create new surface(s) Modify existing surface(s)
Surface: 🔦 ig_shell9	•
Outrim all Untrim selected loop Loop Untrim edge(s) Edge	
	Preview
< Back Finish	Cancel Help

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.

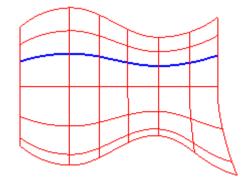
- Click on the Surface wizard
- Check From one surface
- Check Split surface

What method of construction of	
From curves	From one surface
Primitive surface	From multiple surfaces
What specific constructor do y	ou want?
Region from Surface	
Reverse Surface	
Offset Surface	
Extend Surface	
Trim Surface	
OUntrim Surface (Fill Hole	e)
Split Surface	

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

Split Surface	x
Surface name 1: srf1	
Surface name 2: srf1_1	
Surface: <u>ig_shell9</u>	
Row	
Column	
<back can<="" finish="" td=""><td>cel Help</td></back>	cel Help

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.

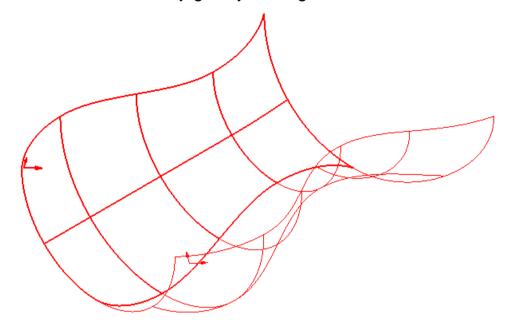
- Take a **Top view**
- Click on the **Pick point button** at the lower left of the form
- Pick the centerline of the jug as indicated

Split Surface	
Surface name 1: srf1 Surface name 2: srf1_1 Surface: ig_shell9 Isoline selection © Row Column 1.14773 -0.4921 -1.5098	
<back cancel="" finish="" help<="" td=""><td></td></back>	

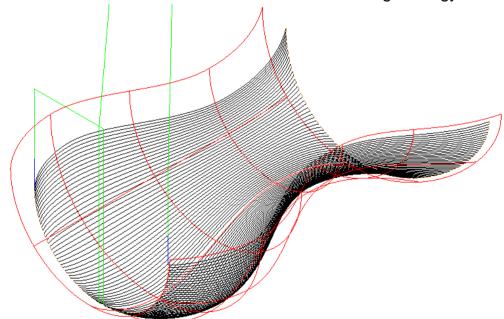
- Click Finish
- Hide the surface ig_shell9 and then select the two new surfaces
- Create a new surface milling feature
- Check Choose a single operation and click Next
- Check Isoline and click Finish
- Click on Isoline in the tree view and select the Surface control tab

Surface Milling Properties	- srf_mill2			×
	itrategy Edges Attributes for iso	Stock Surface o	control	
출 finish1	Surface	Start Curve	Cut direction	۵.
	✓ srf2_1	First Row	Increasing	
	✓ srf2	First Row	Increasing	<u>*</u>
				1
				Ŧ
				<u>*</u>
Click on an operation	I			_
to set its attributes and				
feeds/speeds				
OK Can	cel Appl	y Preview	v Help	

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

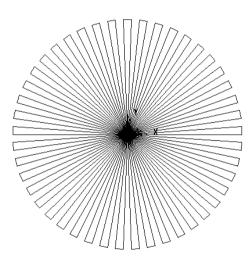
Radial and 2D Spiral Finishing

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.

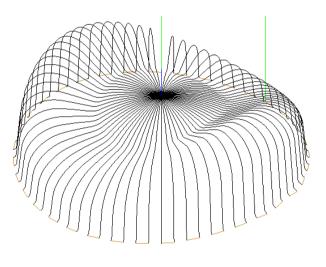
Radial Finish Machining

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

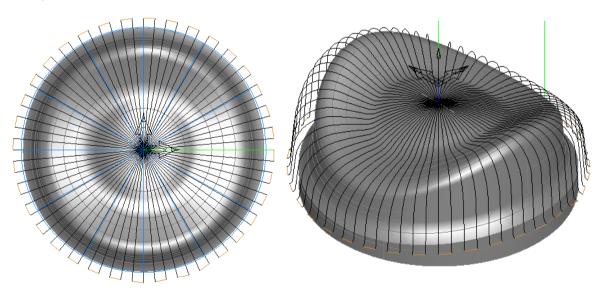
 Z Level Isoline 2d Spiral 3d Spiral Radial Flowline Between 2 curves Roughing Strategies Z Level Plunge Parallel 	Corner Remachining Pencil Swarf
--	---------------------------------

- 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).

Surface Milling Properties - srf	i_mill2	? ×
srf_mill2	Tools F/S Milling Leads	
⊖ ∰ finish1	Attributes for finish1 Angle end = Angle start = * Center point = pt(0.0.0) Check allowance = Leave allowance = 0.0000 Plunge feed override % = 50.0000 Priority = Radius start = Stepover angle = 5.0000 Target horsepower = Tolerance = 0.0010 Z end = Z start = Output Options Post Vars	
Click on an operation to set its attributes and	New Value: pt(0,0,0)	All
feeds/speeds	OK Cancel Apply Preview Help	

- Click Apply then Preview
- Play the simulation



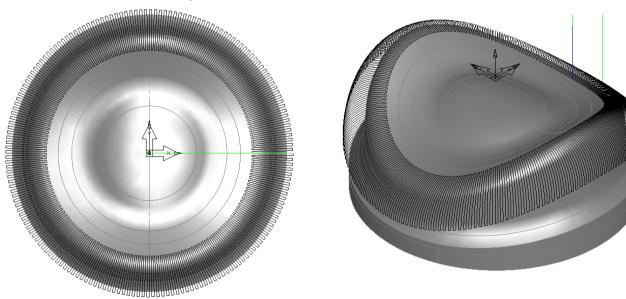
You will notice a number of things in the simulation. In the left image above, you can see that the toolpaths are far apart at the outside of the model and close together in the center. This would give a poor surface finish on the outside edges and possibly overmachine the center. In the right view, 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

Surface Milling Properties - srf	i_mill2	9 X
Surface Milling Properties - srf	Tools F/S Milling Leads Attributes for finish1 Angle end = Angle start = Angle start = * Center point = pt(0,0,0) Check allowance = Leave allowance = 0.0000 Plunge feed override % = 50.0000 Priority = * Radius end = 1.6750 * Radius start = 1.2000 * Stepover angle = 1 Target horsepower = Tolerance = 0.0010 Z end = Z start = Output Options Post Vars New Value:	
its attributes and feeds/speeds	Set Unset Reset	
	OK Cancel Apply Preview Help	

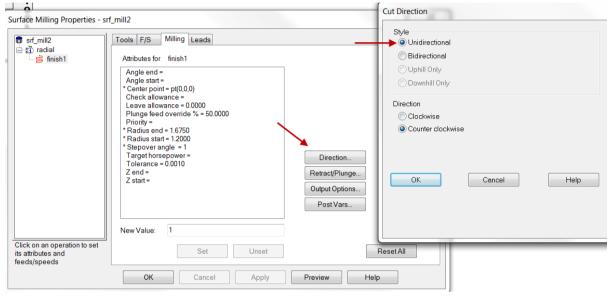
- Click **Apply** and then **OK**
- **Preview** these changes.



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.

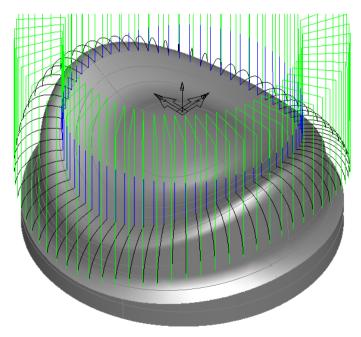
If the toolpath is set cut uni-directionally, this allows us to control whether the tool cuts outside in or inside out. For example, if the **Radius start** is **larger** than the **Radius end**, the tool will start at the **outside** of the part and cut **inwards**.

- 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**



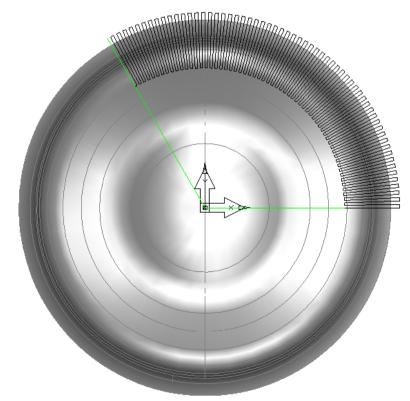
The tool is now cutting the part in a single direction, plunging at the **Radius start** and then cutting in an outward direction to the **Radius end**.

- 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

Surface Milling Properties - srf	mill2	
srf_mill2	Tools F/S Milling Leads	
iradial ∟ġ finish1	Attributes for finish1 * Angle end = 120 * Angle start = 0 * Center point = pt(0.0.0) Check allowance = Leave allowance = 0.0000 Plunge feed override % = 50.0000 Priority = * Radius end = 1.6750 * Radius start = 1.2000 * Stepover angle = 1 Target horsepower = Tolerance = 0.0010 Z end = Z start = Direction Post Vars Post Vars	
	New Value:	
Click on an operation to set its attributes and feeds/speeds	Set Unset Reset All	
	OK Cancel Apply Preview Help	 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

Surface Milling Propertie	s - srf_mill2		8
n srf_mill2 □	Dimensions Location Proce	Misc	
	Operation	Milling Type	
	✓ radial	Milling Type	
		÷	
Click on an operation to set its attributes and feeds/speeds			
ОК	Cancel Apply	Preview Help	

• Select a 2D Spiral strategy and click Finish

Finishing Strategies Parallel Z Level Isoline 2 d Spiral 3 d Spiral Radial Flowline Between 2 curves	Specialized Strategies Horizontal + Vertical Corner Remachining Pencil Swarf
Roughing Strategies Z Level Plunge Parallel	

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

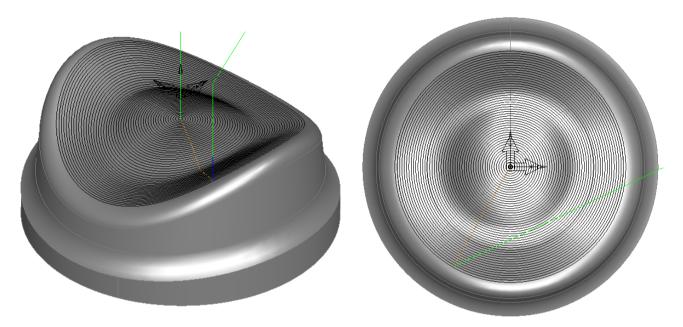
🗊 srf_mill2	Strategy Edges Stock Slopes Surface control
⊖ ਨੂੰ radial ਨੇ finish1 ਨੇ spiral2d ਨੇ finish2	Choose the drive curve for spiral machining Cuse part surface dimensions Use stock dimensions Overcut % 100 Use solid model Solid1 Allowance O.00000 Limit O Tool center Curve Options Curve Options Curve Options O Curve O C
Click on an operation to s its attributes and feeds/speeds	Stock Model

• Check 3D pocket then click on Boundaries

Boundary Curve			×	1		
Boundary curve type. 3D pocket (closed 3D boss (closed profile) 3D side (open profile) Wall only (single patholic) Curves Limit tool path too Overcut % [1] Boundary curves	ofiles) les) iss, open or closed p boundaries AND st	orofiles) ock	OK Cancel Help			
Island curves	Islands	ĺ	Select Boundary	Curves		23
Curve allowance Total offset		♥ L ♥ L	+ ⊂ circ4		Show all	OK Cancel
						Help

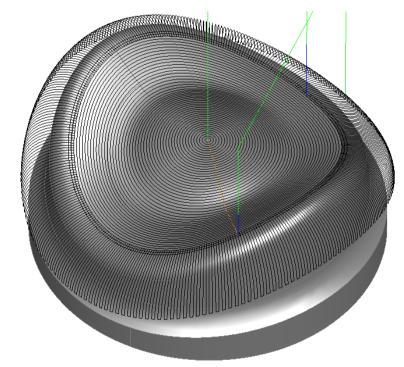
• Select the circle **circ4** and then click **OK**, then **OK** again (your circle number could be different)

- 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



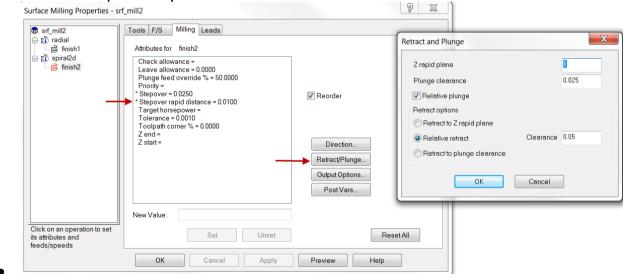
The two toolpaths overlap by 0.05", we set this by the Start radius of the radial toolpath and the size of the circle we used for the 2D Spiral.

- 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.

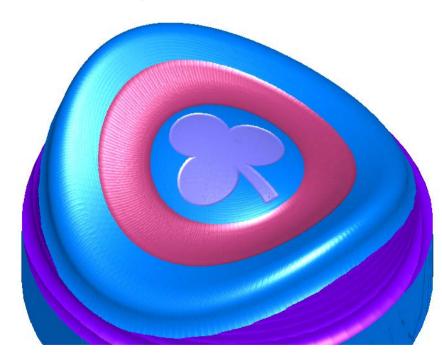
- As an exercise, create a new radial toolpath with a start radius of 0.6, end radius 0.95, angle stepover 1.5 degrees using a 0.25" ball nose cutter.
- Run a **3D RapidCut simulation**

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

Issue FeatureMILL3D 2012

Flowline Finish Machining

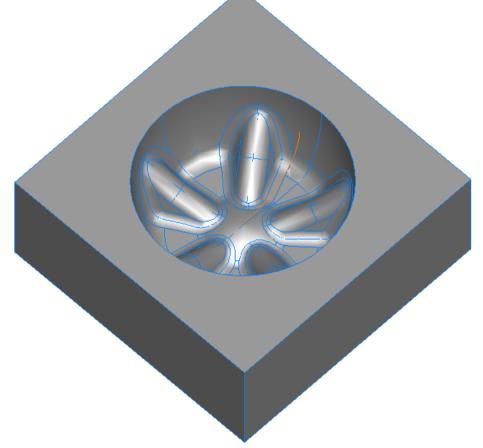
Introduction

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

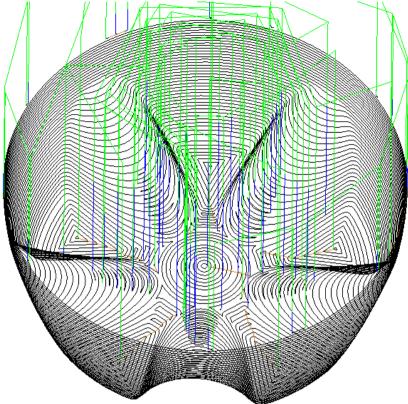
This model shown below is the bottom of a blow mold for a soda bottle.

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



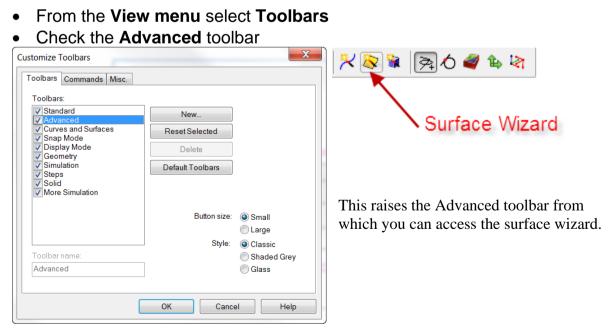
The inner part of the mold contains many surfaces. We want to create a toolpath which will smoothly machine over all of them.

• 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.



• Click OK

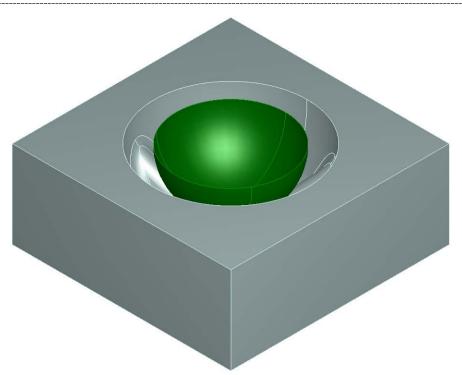
- Click on the Surface wizard
- Check From Curve
- Check Surface of revolution

What method of construction d	o you want?	
▶ ● From curves	From one surface	
Primitive surface	From multiple surfaces	
What specific constructor do yo	ou want?	
Extrude		
Surface of Revolution		
Swept Surface		
Ruled Surface		
Coons		
Curve Mesh	\sim	
Lofted Surface		
Cap Surface		

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

Surface of Revolution	on			X
Surface name:	srf1	C	Construction method	
Curve:	•		Custom line X-Axis Y-Axis Z-Axis	
Start angle: 0	X)		
End angle: 360		Axis	Previev	/
< Back	Finish C	ancel	Help	

Click Finish



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

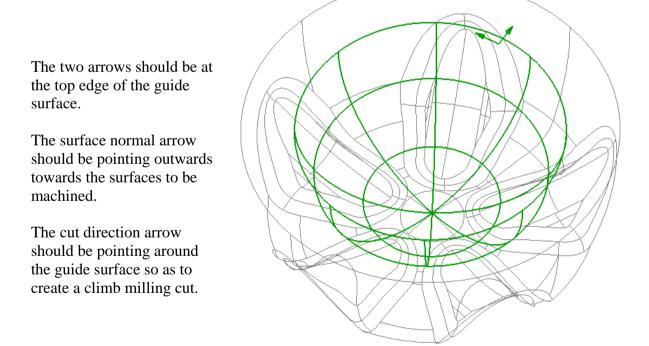
- Edit the feature srf_mill1 and select the Process tab
- Uncheck the Z Level operation
- Click Add new operation

Surface Milling Properties - srf_mill1 Surface Milling Properties - srf_mill1 Surface Milling Properties - srf_mill1 Dimensions Location T Add new operation C z level	Tool Axis Process Misc
Select Flowline Click Finish	New Strategy What kind of strategies Finishing Strategies Parallel Z Level Isoline 2d Spiral 3d Spiral Radial Powline Between 2 curves Roughing Strategies Z Level Plunge Parallel

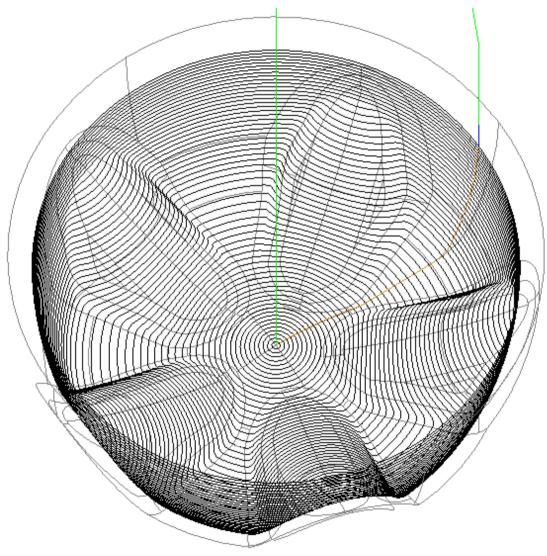
- Click on Flowline and select Surface Control
- From the pulldown select guide

Surface Milling Properties		tock 4-Axis S	urface control		<u> </u>
⊡ 🛐 flowline 二 🚊 finish2	Attributes for flowli	ine			
	Guide surface S	Start Curve	Cut direction	Ma du	
	guide F	First Row	Increasing	Nc	
				<u>*</u>	
				- Aller	
				-	
				٠	
				2	
	4			•	
Click on an operation					
to set its attributes and feeds/speeds	√ Flowline guide su	uface guide	▼	•	
OK Ca	Apply	Preview	Help		

• 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)



- 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.

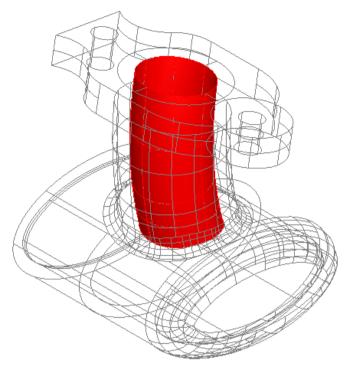
- Raise the Advanced toolbar (as described earlier)
- Click on the Surface Wizard
- On the surface wizard check From curves and Swept surface

rface Wizard	×
What method of construction of	lo you want?
From curves	From one surface
Primitive surface	From multiple surfaces
What specific constructor do ye	ou want?
Extrude	
Surface of Revolution	
> Swept Surface	
Ruled Surface	
Coons	
Curve Mesh	
Conted Surface	
Cap Surface	
<back next=""></back>	Cancel Help
Next>	Gancer

- 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

Swept Surface	J
Surface name: control_surface	
Axis:	
Preview Cross section: Sweep from other end	
<pre>< Back Finish Cancel Help</pre>	
Press Preview	
ne preview surface should appear down the enter 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

What kind of strategy would yo Finishing Strategies Parallel Z Level Isoline 2d Spiral Ad Spiral Flowline Between 2 curves Roughing Strategies Z Level Plunge Parallel	u like to use? Specialized Strategies Horizontal + Vertical Corner Remachining Pencil Swarf
< Back Next > 🧳 Ei	nish 🗸 Cancel Help

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

		Surface Milling Properties - srf_mill1	22
		sf_mill1 Strategy Edges Stock Surface control	
		Guide surface Start Curve Cut direction	
		control_surface First Row Increasing	
		· · · · · · · · · · · · · · · · · · ·	
_			
		to set its attributes and feeds/speeds	
		OK Cancel Apply Preview Help	
\searrow			
		/	
		nnl	a i
• Use the	Isoline set re	ow/column 🔟 and Switch machining side 🔯	t.
		chining direction arrows pointing down and out fro	
the conti	rol surface as	shown above left	
Click on	-		
	ne Tool tab		
	ne tooling gro	•	
	ne tool 3/16_1	ollipop	
Click Ne	w tool		
ſ	Surface Milling Propertie	es - srf_mill1	
	srf_mill1	Tools F/S Milling Leads	
	en strain flowline finish1	Criteria to restrict tools in list:	
		Tool Group: Side Mill	
		Diameter: Anything	
		End-Radius Anything	
		メ 御職 III Curvature-based Name ▲ Diam. End. Flutes Cutter L ▲	
		□ 1/8_Lollipop 0.1250 0.0000 4 0.1250 □	
		□ 1/8_SLITTING 3.0000 0.0000 30 0.1250 □ 1_T_SLOT 1.8430 0.0000 8 0.8280	
		204_KEYSEAT 0.5000 0.0000 10 0.0625	
	Click on an operation	☑ 3/16_Lollipop 0.3750 0.0000 4 0.1875 - <	
	to set its attributes and feeds/speeds		

Preview

Help

Apply

ОК

Cancel

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

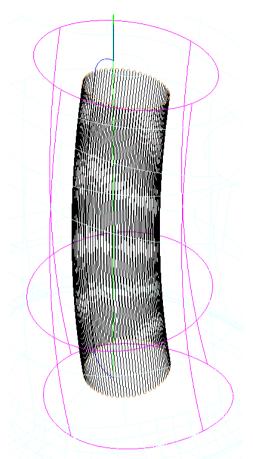
Side Mill Tool Propert	ties		23
Side Mill Overrides	Holder Feed	/Speed	
Name	3/8 Side Mill		
Туре	Slitting Sav	N	Dia 0.3750
Measure	Inches		
Diameter	0.37500	in.	
Cutter Width	0.18750	in.	
Tip Radius	0.09375	in.	
Overall Length	3.00000	in.	L
Exposed Length	2.50000	in.	
Shank Diameter	0.18750	in.	
Neck Diameter	0.12500	in.	
Arbor Tip Length	0.00000	in.	
Teeth	4		
Material HSS	•	Finish BRIGHT	•
Use curve to de	scribe tool shape	e	
Hand 💿 R	ight Hand	○ Left Hand	
		OK Cancel	Apply Help

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

Surface Milling Propertie	s - srf_mill1					
ரை srf_mill1 ்தி flowline ்தி finish1	Tools F/S Milling Leads Stepover type Loop Lead-in/out options Ramp to depth Helical:					
	Ramp angle 5 Use lead-in/out: On first plunge/last retract					
	Lead-in/out plane: Horizontal O Use arc ramp-in/out O Use linear lead-in/out					
	Ramp diameter	0.3750	Lead-in angle	30		
	Ramp-in angle 45 Lead-out angle 30 Ramp-out angle 45 Lead-in length 0.2063					
Click on an operation	Use linear appro	ximation	Lead-out length	0.2063		
to set its attributes and feeds/speeds				Reset All		
OK Ca	ancel Apply	Pr	review Help			

• Click Apply and then OK

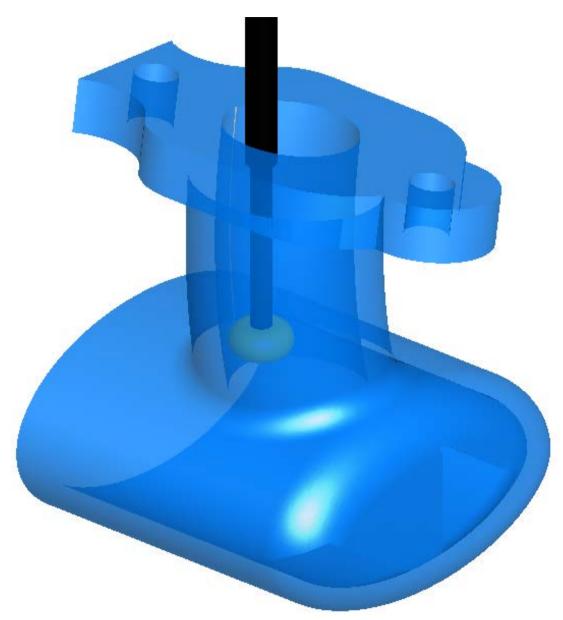
• Run a Centerline simulation



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.

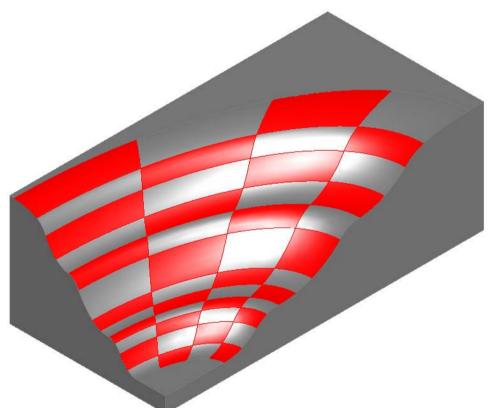
Between 2 Curves Finishing

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.

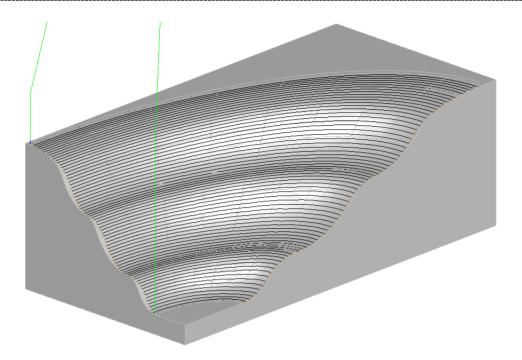
- Select all of the surfaces
- Create a new Surface milling operation
- Click Choose a single operation and click Next
- Select Between 2 curves and click Next

What kind of strategy would yo Finishing Strategies Parallel Z Level Isoline 2 d Spiral S ad Spiral Flowline Between 2 curves Roughing Strategies Z Level Plunge Parallel	Specialized Strategies Horizontal + Vertical Comer Remachining Pencil Swarf
---	---

• Select the curves top and bottom in the pulldown menus as shown

Strategy	
Operation Rough Finish Start curve	Options Direction
End curve	Limit
< Back Next > 🥥	Einish , Cancel Help

- Click Finish
- Click on finish1 and select the Milling tab
- Set the Stepover to 0.1"
- Run a Centerline simulation

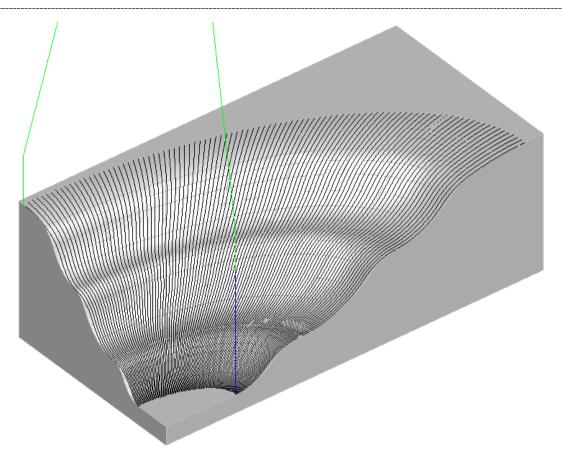


The tool is following a smooth flowing path which starts off by following the first curve (top) and then gradually transforms its shape until it meets the second curve (bottom).

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

Surface Milling Properties - srf_mill1					
📅 srf_mill 1	Strategy	Edges	Stock	Slopes	Surface control
e-:武) between curv	Operation Options Bruish Direction Finish Along curves Start curve Along curves Itop Limit End curve Tool center bottom Contact point		ction) Along curves) Across curves		
Click on an operation to set its attributes and feeds/speeds OK Ca	ancel	Арр	ły (Previe	w Help

- 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

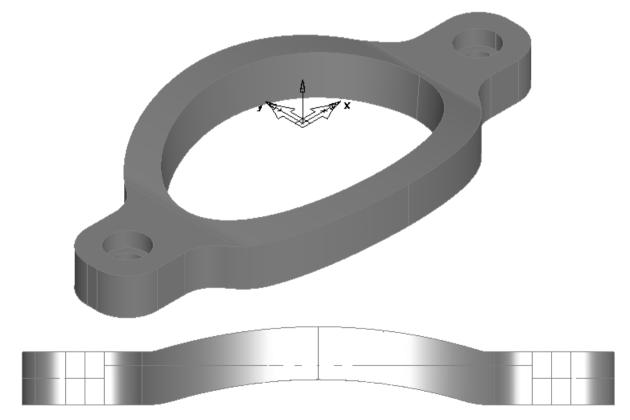
Swarf machining is normally a 5-Axis machining strategy where the part is finished using the side of the tool. The advantage is that a surface can be finished with a single pass giving a shorter machining time and a superior surface finish. Swarf machining may also be used as a 3-Axis machining strategy. 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.

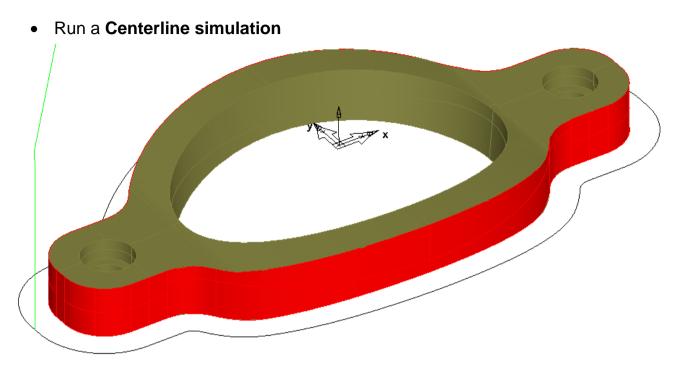
• Select the surfaces that make up the outer wall of the part



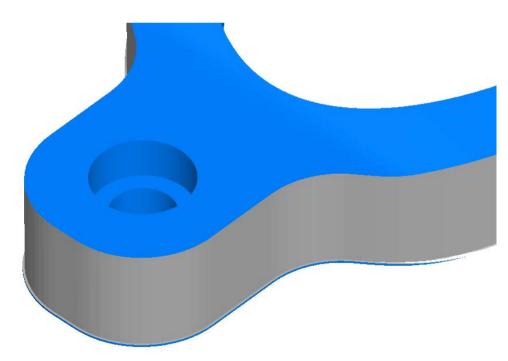
- Create a new Surface milling operation
- Click Choose a single operation and click Next
- Select Swarf and click Finish

Flowline Between 2 curves	Swarf
Roughing Strategies Z Level Plunge Parallel	* @

- Click OK
- Slow down the simulation speed



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



• Run a **3D simulation**

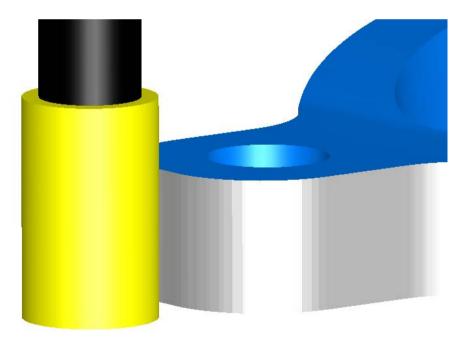
The casting is slightly larger than the finished part, this means that a small lip of excess material is felt after the finishing pass. We need to make the tool tip move slightly past the lower edge of the part to ensure that it cleans up the edge.

- 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")

Surface Milling Properties - srf	f_mill1	? ×
Click on an operation to set	Tools F/S Milling Leads Attributes for finish1 * Axial offset = -0.0500 Axial tolerance = Check allowance = Comer radius % = Degouge tolerance = Leave allowance = 0.0000 Minimum fanning = Plunge feed override % = 50.0000 Priority = Radial offset = Stepover rapid distance = Stock overcut % = 100.0000 Suface join tolerance = Target horsepower = Tolerance = 0.0010 Up/Down smoothing % = New Value: 00500	•
its attributes and feeds/speeds	Set Unset R	eset All
	OK Cancel Apply Preview Help	

• Run a **3D simulation**



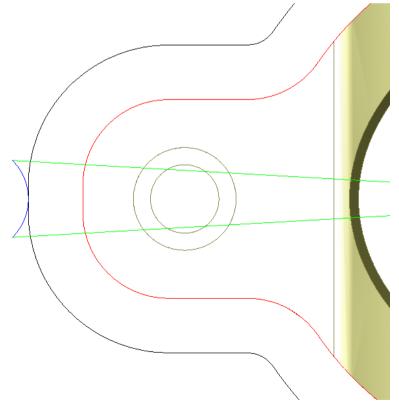
The tool now projects past the lower edges of the surfaces selected for machining by 0.05" so that a clean finish results. It will be noted that the tool is approaching the part by plunging vertically down the face that is to be finished. This could result in a witness mark. We will now modify the toolpath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

Surface Milling Properties - srf_mill1				
ft sf_mill1 ⊡_ff swaf ⊡_finish1	Tools F/S Milling Leads Stepover type Direct Lead-in/out options Ramp to depth Helical: Ramp angle 5			
	Use lead-in/out: On first plunge/last retract Lead-in/out plane: Horizontal			
	O Use arc ramp-in/out			in/out
	Ramp diameter 0.5000		Lead-in angle	30
	Ramp-in angle	45	Lead-out angle	30
	Ramp-out angle 45		Lead-in length	0.2750
	Use linear approximation		Lead-out length	0.2750
Click on an operation to set its attributes and feeds/speeds				Reset All
ОК Са	ancel Apply	P	review Help	

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

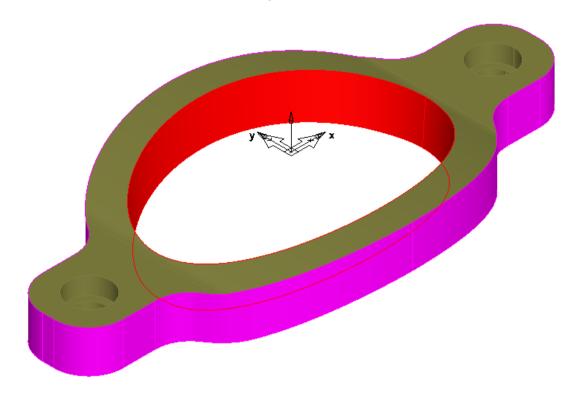


The tool is now ramping smoothly on and off the part.

• 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

Finishing Strategies Parallel Z Level Isoline 2d Spiral 3d Spiral	Specialized Strategies Horizontal + Vertical Corner Remachining Pencil Swarf
C Radial Flowline Between 2 curves Roughing Strategies Z Level Plunge	
Parallel < Back Next> # For the second se	inish Cancel Help

• Click on finish1 and select the Tools tab

- 23 Surface Milling Properties - srf_mill2 Tools F/S Milling Leads 🗊 srf_mill2 ່⊟∵ຊີ້ງ swarf Criteria to restrict tools in list: finish1 Tool Group: 📕 Flat End Endı 👻 Diameter: Anything Ŧ End-Radi Anything • 🛃 🐂 🎇 😭 Curvature-based Name 🛓 Diam. End . Flutes Cutter L 🔺 □ endmill0500:4reg 0.5000 0.0000 4 1.0000 1.0100 endmill0500:4re. 0.1438 0.0000 4 endmill0500:high 0.3250 0.0000 4 1.0000 🗹 endmill0500:hig. 0.5000 2.0000 D endmill0500:reg 0.5000 0.0000 2 0.8120 + Click on an operation to set its attributes and • Þ feeds/speeds Cancel Apply Preview Help OK
- Select a long reach 0.5" Flat endmill and then select New tool

• Fill in the form exactly as shown below

End Mill Tool Properties			
Endmill Overrides	Holder Feed/	Speed	
Name	Taper 5 - 0.5		
Measure	Inches		
Bottom Diameter	0.50000	in.	
Overall Length	4.00000	in.	
Exposed Length	2.50000	in.	
Cutter Length	2.85751	in.	
Flutes	2		
Shank Diameter	0.50000	in.	
End Radius	0.00000	in. 🔲 Ball-end	
Use curve to de	escribe tool shap	e 🖌	
Taper	5	deg. 📝 Diameter at Bottom Compute from shank	
	K	Tool End Type SINGLE	
Cutting Type 👿 (Center	Flute Angle HIGH	
Material HS	S	Finish BRIGHT	
Hand 💿 F	Right Hand	C Left Hand	
		OK Cancel Apply Help	

• 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")

Surface Milling Properties - srf	_mill2 2 X
⊡∄ swarf finish1	Attributes for finish1 Axial offset = -0.0500 Axial tolerance = Check allowance = Corner radius % = Degouge tolerance = Leave allowance = 0.0000 Minimum fanning = Plunge feed override % = 50.0000 Priority = Radial offset = Stepover rapid distance = Stock overcut % = 100.0000 Suiface join tolerance = Target horsepower = Tolerance = 0.0010 Up/Down smoothing % = New Value:
Click on an operation to set its attributes and feeds/speeds	Set Unset Reset All
	OK Cancel Apply Preview Help

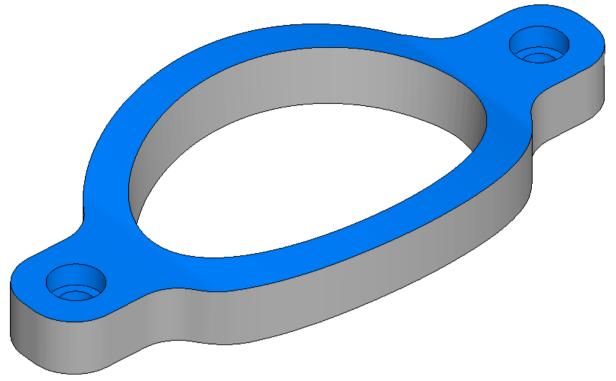
• Select the Leads tab

• Select a Horizontal arc Lead in and Lead out as shown below

srf_mill2	Tools F/S Milling I	Leads			
⊟-ট্রী swarf ≓≣ finish1	Stepover type	Direct	•		
	Lead-in/out options				
	Ramp to depth	Heli	cal:		
	Ramp angle 5				
	Use lead-in/out:	On first plur	nge/last retract	•	
	Lead-in/out plane:	Horizontal		•	
	Ose arc ramp-in/ou	t 🔘	Use linear lead-	in/out	
	Ramp diameter 0	0.3250 Lea	ad-in angle	30	
	Ramp-in angle 4	45 Lea	ad-out angle	30	
	Ramp-out angle 4	45 Lea	ad-in length	0.1788	
	Use linear approxim	mation Lea	ad-out length	0.1788	
Click on an operation to set its attributes and feeds/speeds					Reset All
its attributes and	ОК	Cancel	Apply	Preview	[

• Click **Apply** and then **OK**

• Run a **3D Simulation**



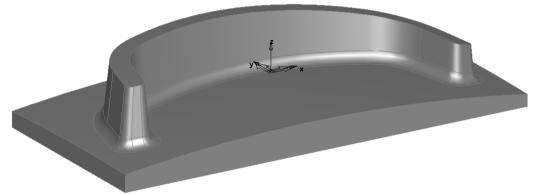
The part is now finished; note how the tool tracks up and down in Z while following the edge of the constant tapered part. This would not be possible with a 2.5D Side feature.

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

3-Axis Swarf – Multiple passes

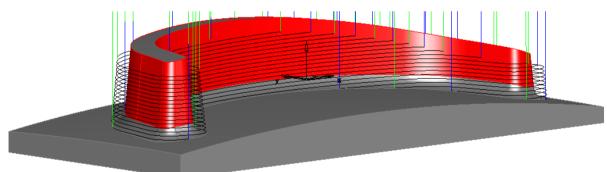
This part has a varying radius around the corners. It could be finished by a 5-Axis swarf toolpath in a single pass. If we use a 3-Axis swarf toolpath then multiple passes will be required.

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

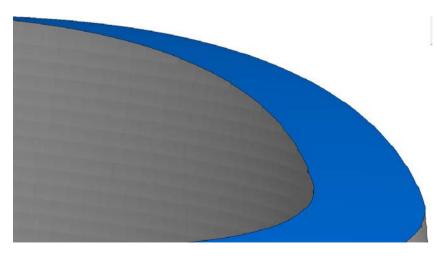


The part contains a Z Level finish toolpath for the tapered surfaces, we will now run a 3D simulation to see the result of this toolpath.

• 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

Surface Milling Properti	es - srf_mill1	
<mark>ा</mark> ि sf_mill1 ⊡_ट्रों) z level ं⊖ finish1	Dimensions Location Process Misc Image: Add new operation Add new operation Image: Add new opera	
	Operation Milling Typ z level finish	e
Click on an operation to set its attributes and feeds/speeds OK	Cancel Apply Preview	Help

• Select a Swarf operation and click Finish

New Strategy What kind of strategy would y Finishing Strategies Parallel Z Level Isoline 2 d Spiral	vou like to use? Specialized Strategies ⓒ Horizontal + Vertical ⓒ Comer Remachining ⓒ Pencil
 3d Spiral Radial Flowline Between 2 curves Roughing Strategies Z Level Plunge 	• Swarf
Parallel 	sh Cancel Help

• Click on finish2 and select the Tools tab

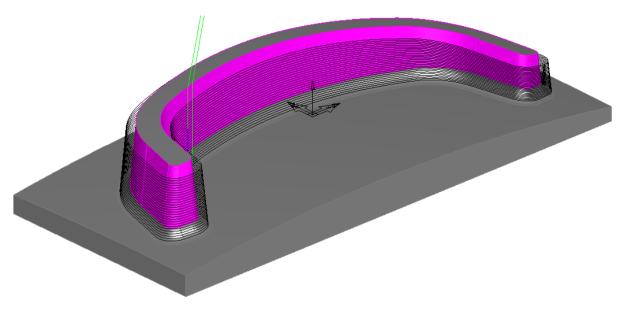
• Select a 0.375" Ball nosed endmill

Surface Milling Propertie	es - srf_mill1		X
srf_mill 1	Tools F/S Millin	ng Leads	
⊡ swarf ⊟ finish2	Criteria to restrict tool	s in list:	
	Tool Group:	📙 Ball End Endı 👻	
	Diameter:	Anything 👻	
	End-Radius:	Anything 👻	
	🫃 🤊 🦉 😭	Curvature-based	
	Name 🛓	Diam. End. Flutes	Cutter Le.
	Long endmillB0		2.5000
	Swarf B0375:4r	. 0.3750 0.1875 4	0.7500
Click on an operation		1	•
to set its attributes and feeds/speeds			,
	Cancel Apply	Preview	lelp

- 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"

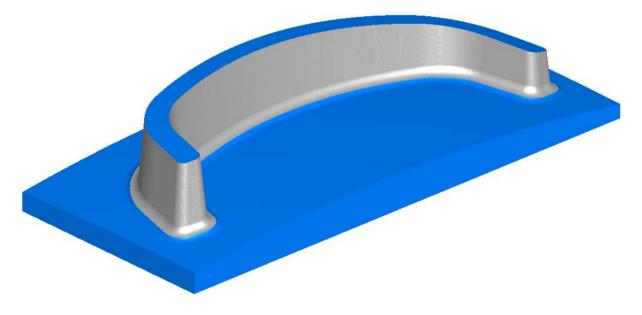
Surface Milling Properties - srf	mill1 Tools F/S Milling Leads Attributes for finish2 Axial offset = 0.0000 Axial tolerance = Comer radius % = Degouge tolerance = Leave allowance = 0.0000 Minimum fanning = Plunge feed override % = 50.0000 Priority = Radial offset = Stepover rapid distance = Stock overcut % = 100.0000 Surface join tolerance = Target horsepower = Tolerance = 0.0010 Up/Down smoothing % = Upper offset = * Z increment = 0.0500 New Value: 0.0500	 Fan at ends Multiple Cuts Multicut strategy Merge Direction Retract/Plunge Output Options Post Vars
Click on an operation to set its attributes and feeds/speeds	Set Unset	ResetAll
	OK Cancel Apply	Preview Help

• 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.

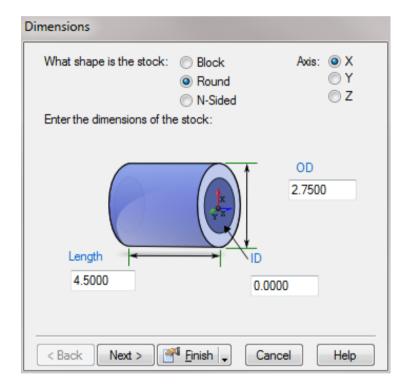
4-Axis Rotary Finish Machining

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

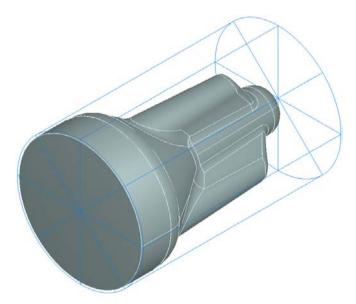
We need to tell FeatureCAM which axis our machine will be using for rotational moves. In this case we are setting up to rotate around the X axis. In general, there are three possible rotary axes. These are called the A, B and C axes. A 4-Axis machine will have the normal X,Y and Z linear movement axes 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

Stock Properties - stoc	k1	23
🔶 stock1	Dimensions Indexing	
	 No Multi-axis Positioning Generate Single Program with program stop between each setup 	
	4th Axis Positioning 4	
	 Index around the STOCK X Axis Index around the STOCK Y Axis Index around the STOCK Z Axis 	
	Sth Axis Positioning Fixture Location	
	Operation Ordering Tool Dominant	
	 Setup Dominant 	
ОК	Cancel Apply Help	

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

What kind of strategy would Finishing Strategies Parallel Z Level Isoline 2d Spiral 3d Spiral Radial Flowline Between 2 curves	Specialized Strategies Horizontal + Vertical Comer Remachining Pencil Four Axis Rotary Swarf 5-Axis Trim
Roughing Strategies Z Level Plunge Parallel < Back Next > 49	

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

Strategy		
Operation Rough Finish	Options Linear Circular Spiral 	
	Y offset 0.000	
< Back Next >	Zancel Help	

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.

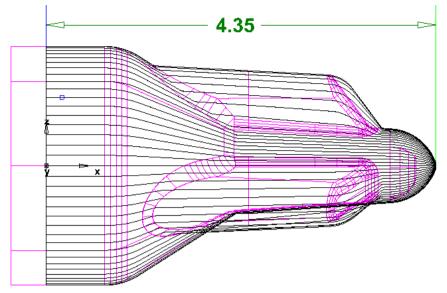
- Eject the simulation
- Edit the feature
- Click on finish1 in the tree view
- Select the Milling tab
- Click on **Index start coordinate** and set the value to **0**
- Click on Index end coordinate and set the value to 4.35

Surface Milling Properties - srf	_mill1
or_mill ⊟ ∰ fouraxis ∟ ⊜ finish1	Tools F/S Tool Usage Milling Attributes for finish1 Angle end = Angle start = Check allowance = - * Index start coord = 4.3500 * Index start coord = 4.3500 * Index start coord = 0 Leave allowance = 0.0000 Plunge feed override % = 50.0000 Priority = Stepover nagle = 5.0000 Stepover ragid distance = Stock overcut% = 100.0000 Target horsepower = Tolerance = 0.0010 Direction Retract/Plunge Output Options Post Vars
Click on an operation to set its attributes and feeds/speeds	New Value: 4.3500 Set Unset Reset All
	OK Cancel Apply Preview Help

• 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.

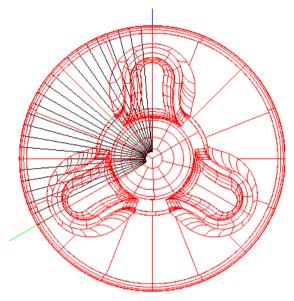
- Edit the feature
- Click on finish1 in the tree view
- Select the Milling tab
- Click on Angle start and set the value to 0
- Click on Angle end and set the value to 120

🗊 srf_mill1	Tools F/S Tool Usage Milling Leads	
ੇ ∰ fouraxis	Attributes for finish1 * Angle end = 120 Angle start= 0 Check allowance = Index end coord = 4.3500 * Index start coord = 0 Leave allowance = 0.0000 Plunge feed override % = 50.0000 Priority = Stepover angle = 5.0000 Stepover rapid distance = Stock overcut % = 100.0000 Target horsepower = Direction	
Click on an operation to set	Tolerance = 0.0010 Retract/Plunge Output Options Post Vars New Value: 0	5
its attributes and feeds/speeds	Set Unset	Reset All
	OK Cancel Apply Preview	Help

- 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.

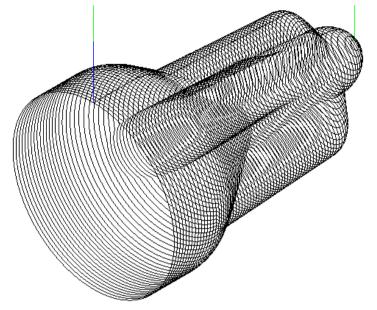


- Edit the feature
- Click on finish1 in the tree view
- Select the Milling tab
- Click on Angle start and then Unset
- Click on Angle end and then Unset and Apply
- Click on fouraxis in the tree view
- Check the Circular option on the Strategy tab

Surface Milling Properties - srf_mill1		<u> </u>		
finish1 Click on an operation	es - srf_mill1 Strategy Surface control Operation Rough Finish	Options Cinear Circular Spiral Y offset	0.000	
to set its attributes and feeds/speeds				
	ancel Apply	Preview	Help	

• 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.

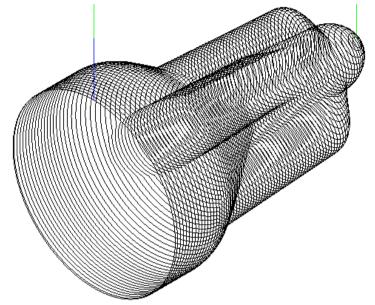
- Edit the feature
- Click on **fouraxis** in the tree view
- Check the Spiral option on the Strategy tab

Surface Milling Properti	es - srf_mill1	J
n srf_mill1 □	Strategy Surface control Operation Rough Finish Strategy Curcular Strategy Surface control Options Circular Strategy Spiral	
	Y offset 0.000	
Click on an operation to set its attributes and feeds/speeds OK	Cancel Apply Preview Help	

• Click Apply and then OK

FeatureMILL3D

- Select an Isometric view
- Run a centerline simulation



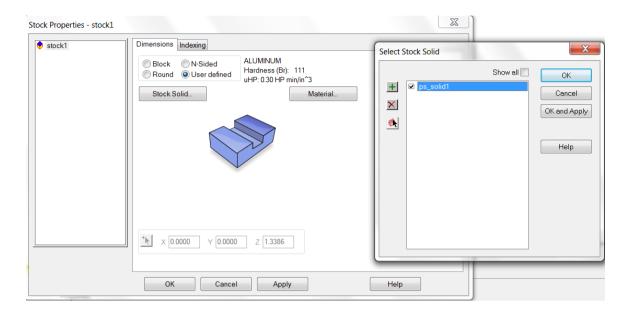
With the spiral option selected the toolpath follows a continuous spiral along the part. This will give a better surface finish than the circular option as there will be no dwell marks from stepover moves. The strategy will require a machine tool with a rotary axis which can rotate continually with no 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 then OK
- Run a centerline simulation

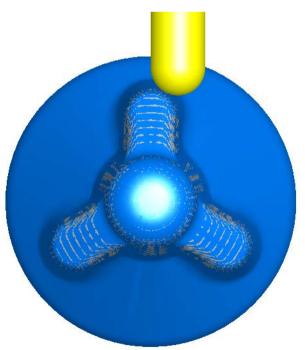


It can be seen that the X direction of cutting has now been reversed.

- 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







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

© Copyright Delcam plc 2011. All other trademarks are the property of their respective owners.