Delcam Advanced Manufacturing Solutions

FeatureCAM 2012

Training Course

FeatureCAM 2012

Training Course

FeatureMILL 3D 5axis simultaneous

FeatureCAM

Acknowledgements

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

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

Contents

FeatureCAM overview

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

FeatureCAM includes five stand-alone modules:

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

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

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

The following add-on modules are also available:

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

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

Why creating toolpaths is so fast

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

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

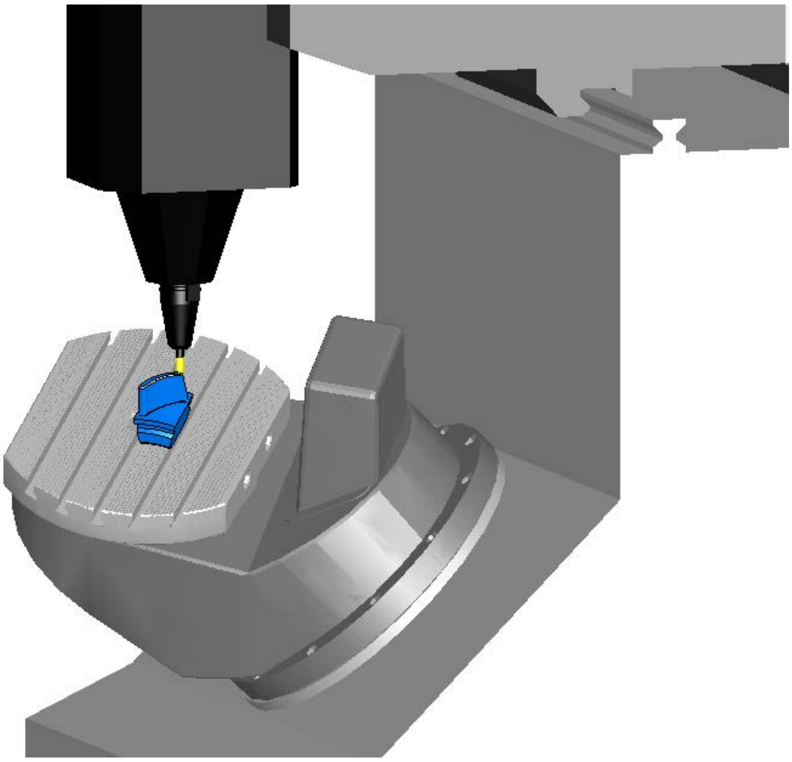
- Selects the most appropriate tools and operations;
- Recommends machining strategies;
- Calculates speeds and feeds;
- Generates toolpaths and creates the NC code.

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

5-Axis Introduction

Introduction

This module will provide the user with an introduction to the 5-axis tool orientations available in FeatureCAM. In 5-axis positioning machining, the user can position the tool along a fixed alternative axis giving a number of benefits. With 5-Axis simultaneous machining, the orientation of the tool axis can be continuously varied which allows much greater flexibility. By changing the tool axis we can avoid collisions with the part allowing the use of shorter, more rigid tooling giving a better surface finish.



The use of 5-Axis machining gives a number of benefits:

A part can be cut in a single setup including undercuts which would normally require the part to be dismounted from the machine and set up in a different orientation. This reduces the setup time and errors that can be introduced by having to do multiple setups.

By allowing the tool to be tilted relative to the surfaces being machined, the contact point of the tool on the surface can be changed giving better cutting conditions.

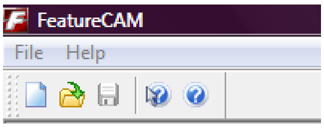
Using shorter tools increases the rigidity of the tool, reducing deflection and chatter giving improved surface finish and accuracy.

Turning on 5-Axis

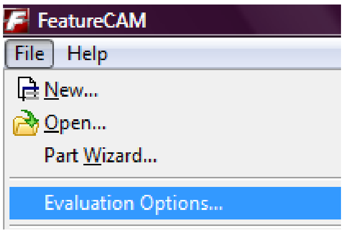
By default in FeatureCAM, the tool is oriented along the setup Z axis. In order to use 5-Axis simultaneous machining, we first need to enable 5-Axis simultaneous in the Evaluation options. Once this is enabled, we also need to change the stock properties to allow 5-axis indexing.

- Open FeatureCAM but **DO NOT open an FM document**

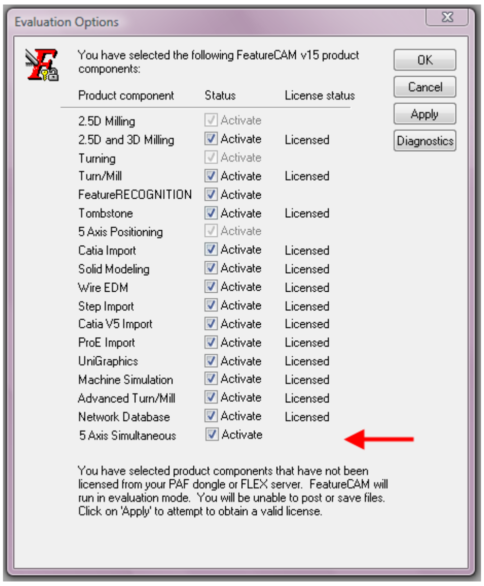
On the main toolbar you should only have two options, **File** and **Help**.



- Click on **File** and then select **Evaluation Options**



- Check **5 Axis Simultaneous**



FeatureCAM will give a warning that the product component has not been licensed. This just means that it has not yet checked the license file and dongle to see if you have the rights to use 5 Axis Simultaneous positioning.

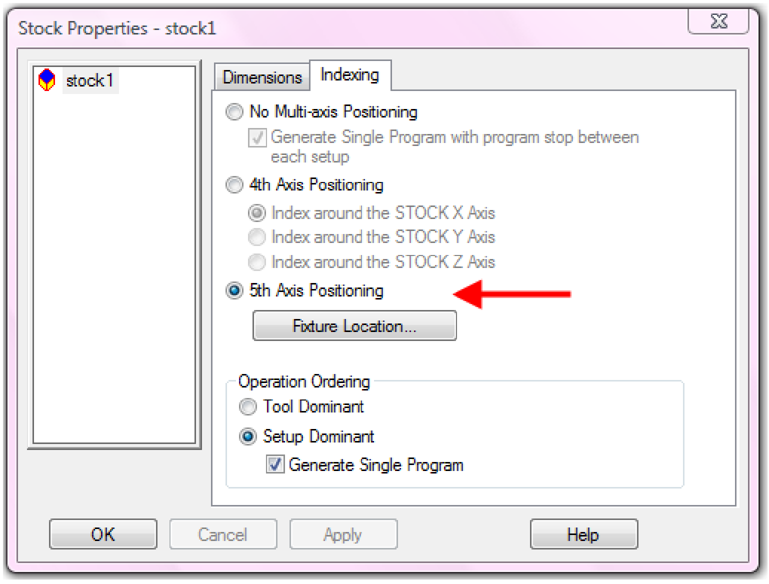
- Click **Apply**

If you have the correct options in your license file, you will then see the word **Licensed** next to the **5 Axis Simultaneous option**. If you do not see this, then your license file is not set up to license this option on the dongle you are using. In this case, you can either continue to use FeatureCAM in Evaluation mode which does not allow you to save, or you can contact your dealer to arrange a license.

- Click **OK** to close the Evaluation Options form

FeatureCAM now has the 5-Axis Simultaneous option enabled. In order to create a 5-Axis program, we also need to set up 5-Axis indexing in the stock properties.

- Create a new **Inch Milling document**
- On the **Dimensions** form click **Finish**
- Select the **Indexing tab**
- Check **5th Axis Positioning**



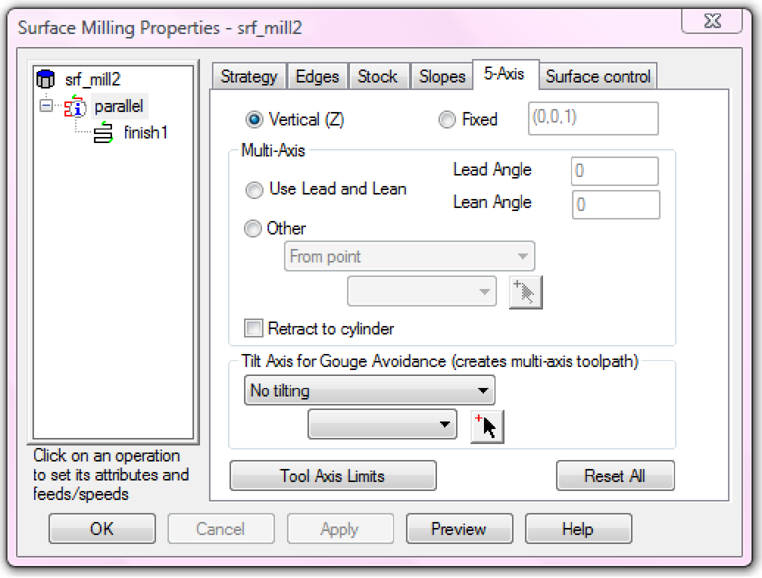
- Click **Apply** and then **OK**

The new document is now ready for programming of a 5-Axis part. There are a number of other things that need to be set up in order to position the part relative to the axes of rotation of the machine so that the program will produce the part correctly. This is rather more involved that the setup for a 3-Axis part, and differs depending upon the machine configuration and includes some variables which are machine specific. We will look into the machine specific setup variables later in this module.

Now that we have FeatureCAM set up to use 5-Axis positioning we will briefly look at the options that are available before we continue to look at each in detail.

- **Import** the part **Phone.igs**
- **Resize** the stock to the part
- Create a **Parallel toolpath** using the default values

- Click **Finish**
- Click on the **Parallel** operation



There is now an extra tab in the feature properties called **5-Axis**; this allows us to select which tool axis orientation we want to use. Each of the options will be examined in detail in separate modules later on, for now a brief description follows:

Vertical
The vertical tool axis option will align the axis of the tool with the Z axis of the active setup and so is identical to normal 3-axis machining.

Fixed
This option allows the user to define the tool axis direction as a vector.

Lead and Lean
The tool can be tilted at a specified angle relative to the direction of tool travel. The **Lead** angle tilts the tool forwards or back along the travel direction, **Lean** tilts the tool to the left or right of the travel direction.

Other
This activates a pulldown menu giving access to further tool axis options:

From Point
With this option, the centerline of the tool will pass through a user defined point with the tool axis directed from the specified point towards the surfaces to be machined.

To Point
With this option, the centerline of the tool will be directed towards a user defined point with the tool axis passing through the surfaces to be machined.

From Line
With this option, the centerline of the tool will pass through a user defined line with the tool axis directed from the specified line towards the surfaces to be machined.

To Line
With this option, the centerline of the tool will pass through a user defined line with the tool axis directed towards the specified line through the surfaces to be machined.

From Curve
The tool axis passes through a user defined curve towards the surfaces to be machined; this allows complex 3D control of the tool axis.

To Curve
With this option, the centerline of the tool will pass through the surfaces to be machined and towards a user defined curve.

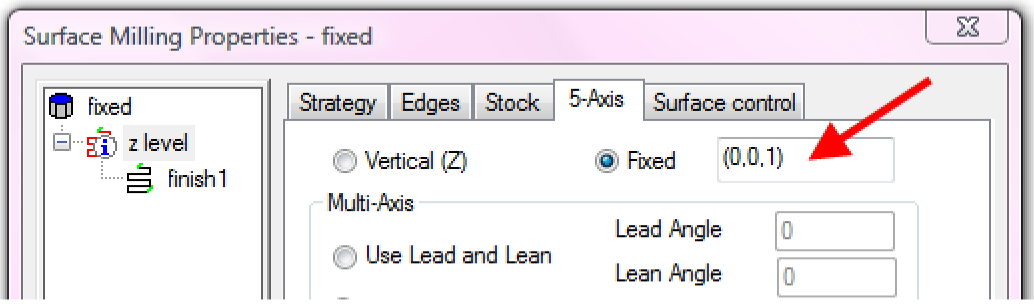
Not all machining strategies support all 5-Axis Simultaneous tool axis options. For instance, no roughing strategies are 5-Axis at present. The matrix below shows which tool axis options are available for each machining strategy.

	Lead/Lean	From Point	To Point	From Line	To Line	From Curve	To Curve	Fixed	Automatic	Avoid	Limits
Parallel	Y	Y	Y	Y	Y	Y	Y	Y	N	Y	Y
Z Level	Y	Y	Y	Y	Y	Y	Y	Y	N	Y	Y
Isoline	Y	Y	Y	Y	Y	Y	Y	Y	N	Y	Y
2D Spiral	N	N	N	N	N	N	N	N	N	N	N
3D Spiral	Y	Y	Y	Y	Y	Y	Y	Y	N	Y	Y
Radial	N	N	N	N	N	N	N	N	N	N	N
Flowline	Y	Y	Y	Y	Y	Y	Y	Y	N	Y	Y
Between 2 Curves	N	N	N	N	N	N	N	N	N	N	N
Horizontal & Vertical	Y	Y	Y	Y	Y	Y	Y	Y	N	Y	Y
Corner Finishing	N	N	N	N	N	N	N	N	N	N	N
Pencil	N	N	N	N	N	N	N	N	N	N	N
Swarf	Y	N	N	N	N	N	N	Y	Y	N	Y
5-axis Trim	Y	Y	Y	Y	Y	Y	Y	Y	N	N	Y
Z Level Rough	N	N	N	N	N	N	N	N	N	N	N
Plunge	N	N	N	N	N	N	N	N	N	N	N
Parallel	N	N	N	N	N	N	N	N	N	N	N

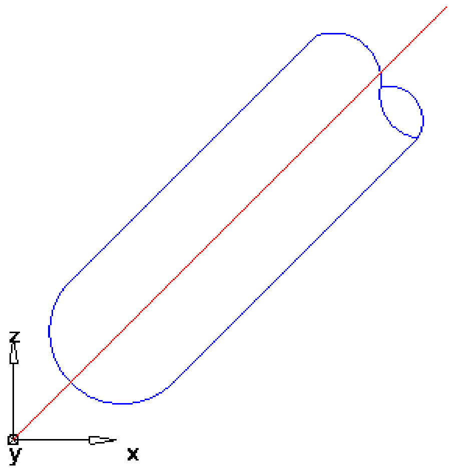
Fixed Axis Tool Alignment

Introduction

Fixed Axis tool alignment allows the tool to be aligned to a specified vector. This gives a 3+2 machining toolpath without the necessity of creating multiple setups. The vector is the direction from the tool tip back up the spindle of the machine. The vector is expressed as a value in X, Y, and Z separated by commas. For example, the vector (0,0,1) shown below indicates that the tool is pointing vertically down in the Z direction and is equivalent to a normal 3-Axis orientation.



To point the tool downwards at 45 degrees in the XZ plane as shown below the vector would be (1,0,1).



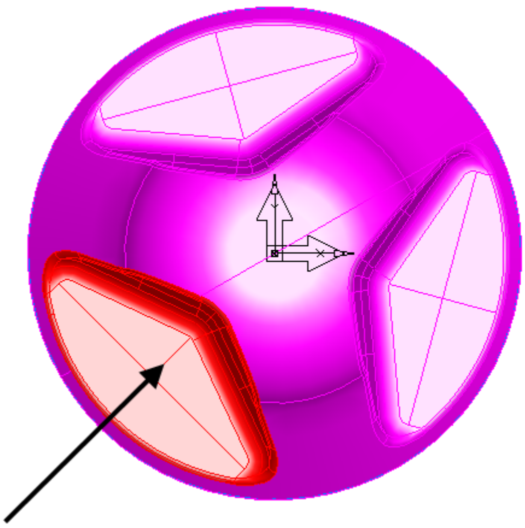
We will now look at an example where a short tool will be used to finish machine a round part. In order to avoid collisions with the model, the tool will need to be tilted over away from the part.

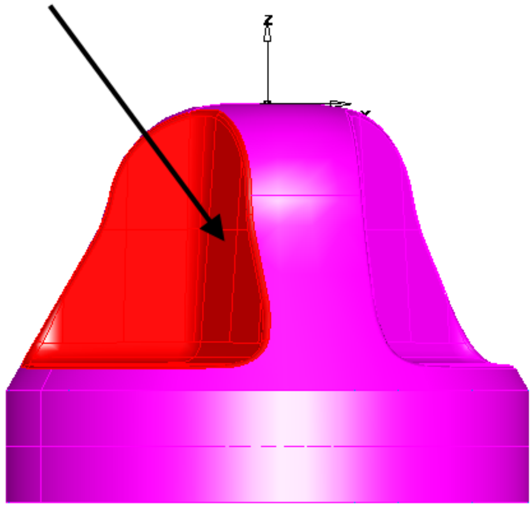
- Open the file **Fixed.fm**
- Select the tool crib **Fixed.fm_tools_from_last_save**
- Run a **3D simulation**

The roughing operation works OK, but on the finishing operation the tool is too short so that the holder collides with the part.

- **Eject** the simulation and select a **Top View**
- Select a **shaded view**

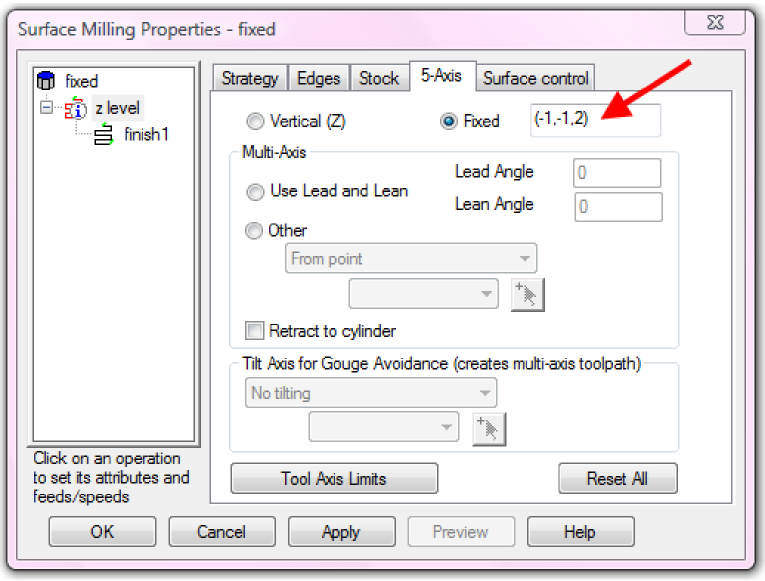
We want the finishing tool to approach the part from the lower left quadrant (-X, -Y) and from above (+Z) as indicated by the arrows in the images below.





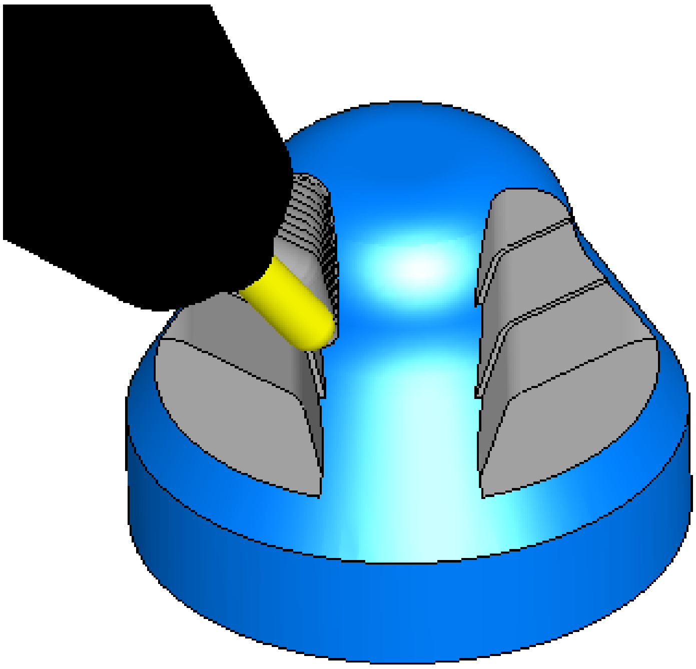
To make the tool come in at 45 degrees in the XY plane and from a steeper angle from above we shall now set a fixed tool axis vector of (-1,-1,2).

- **Edit** the properties of the **fixed** feature
- **Click** on the **Z Level** operation and select the **5-Axis tab**
- Check **Fixed** and enter the vector **(-1,-1,2)** as shown



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

- Run a **3D Simulation**



The tool is now tilted away from the part so that the surfaces can be machined without the holder colliding with the model.

- As an exercise, now machine the remaining two recesses on the part using a similar strategy

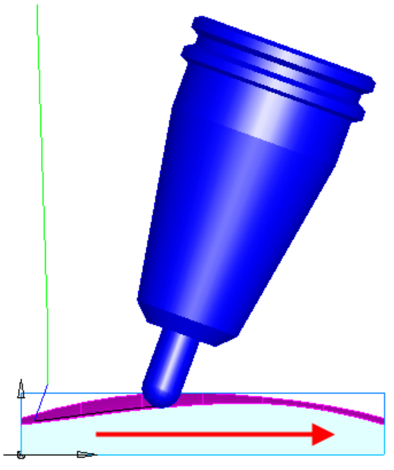
  Hint: Looking from above, the fixed axis vectors required are **(-0.25,1,2)** for the upper left recess and **(1,-0.25 2)** for the lower right.

Lead and Lean Tool Alignment

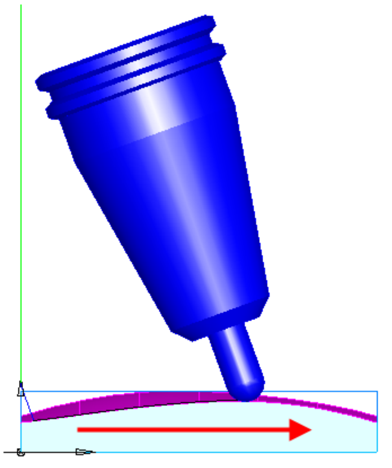
Introduction

The **Lead and Lean** tool axis option enables the user to tilt the tool over at an angle relative to its direction of travel along the tool path. The Lead and Lean angles can be set separately to achieve the desired tool orientation.

The **Lead** angle tilts the tool either **forward** or **backward along** the direction that it is travelling. A **positive lead** angle tilts the tool **forward,** and a **negative** angle will tilt it **backward**.

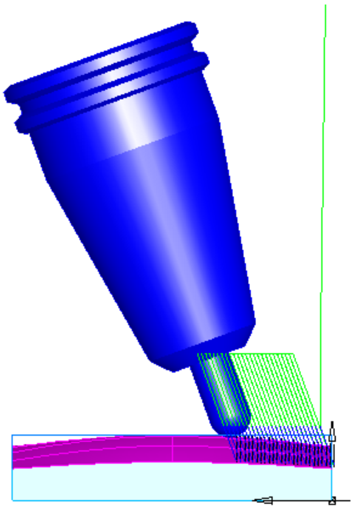


Positive Lead Angle

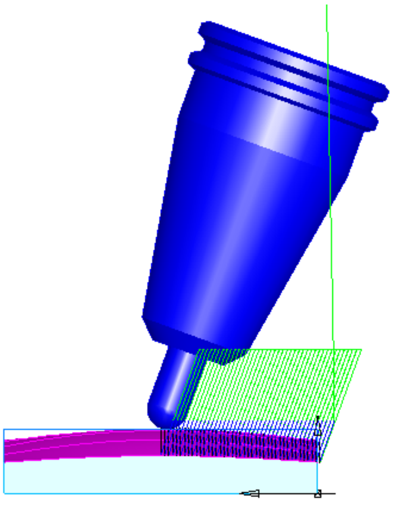


Negative Lead Angle

The **Lean** angle tilts the tool either **left** or **right** across the direction that it is travelling. A **positive lean** angle tilts the tool towards the **left,** and a **negative** angle will tilt it to the **right**. In the images below, the tool is travelling away from the viewer.



Positive Lean Angle



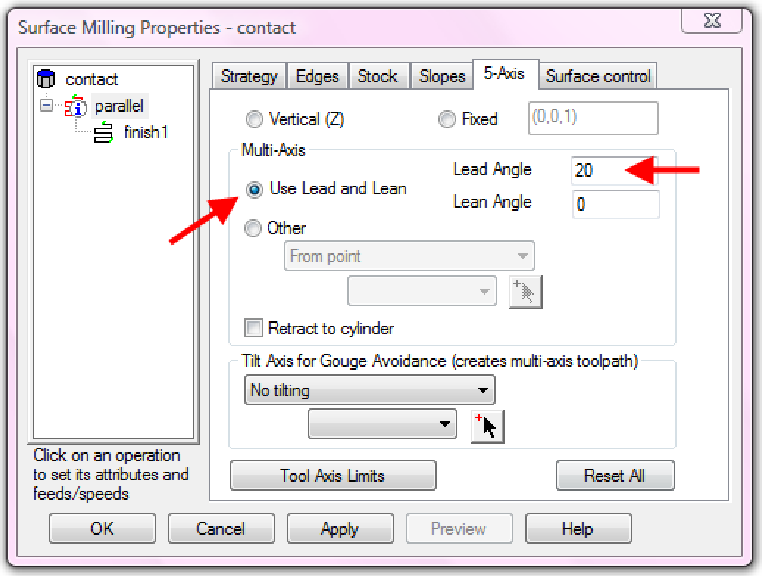
Negative Lean Angle

For the majority of toolpaths, the Lead and Lean angles are calculated relative to the plane between setup Z axis and the tool travel direction. The exceptions are Isoline and Flowline toolpaths. These will be discussed later in this module.

- Open the file **Lead_Lean.fm**
- Select the tool crib **Lead_Lean.fm_tools_from_last_save**
- Check the feature **contact**
- Run a **centerline simulation**

The tool is oriented along the Z axis by default. If this was a finishing operation removing a small amount of stock then the surface cutting speed at the point of contact would be very low as it is near to the centre of rotation of the tool. By tipping the tool over we can change the contact point of the tool on the surface so that the cutting conditions are more favorable.

- Open the feature properties, click on **Parallel** and select the **5-Axis tab**



- Check **Use Lead and Lean** and enter a **Lead** angle of **20** degrees

Note: A lead and lean angle of zero means that the tool will be aligned with the setup Z axis and is therefore the same as using a normal 3-axis orientation.

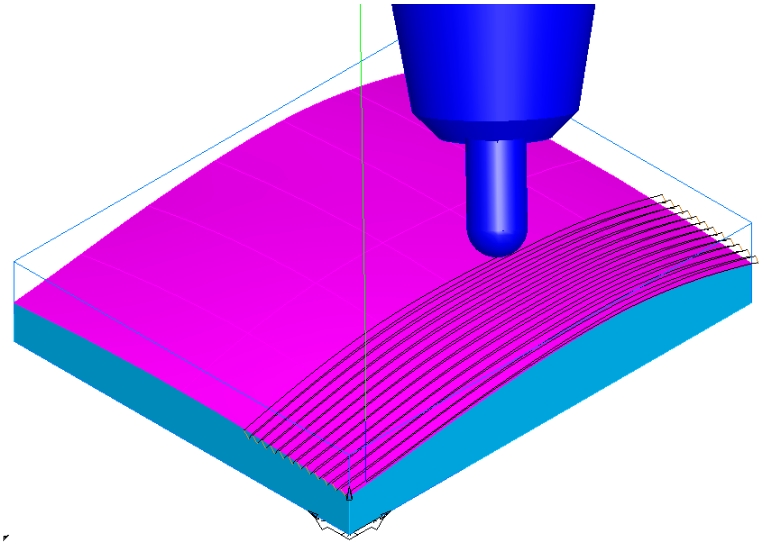
- **Run a centerline simulation**

You will see that the tool is now tilting forwards at an angle of 20 degrees to the vertical as it moves across the part.

- Open the feature properties, click on **Parallel** and select the **5-Axis tab**
- Enter a **Lead** *and* **Lean** angle of **20 degrees**
- Run a **centerline simulation**

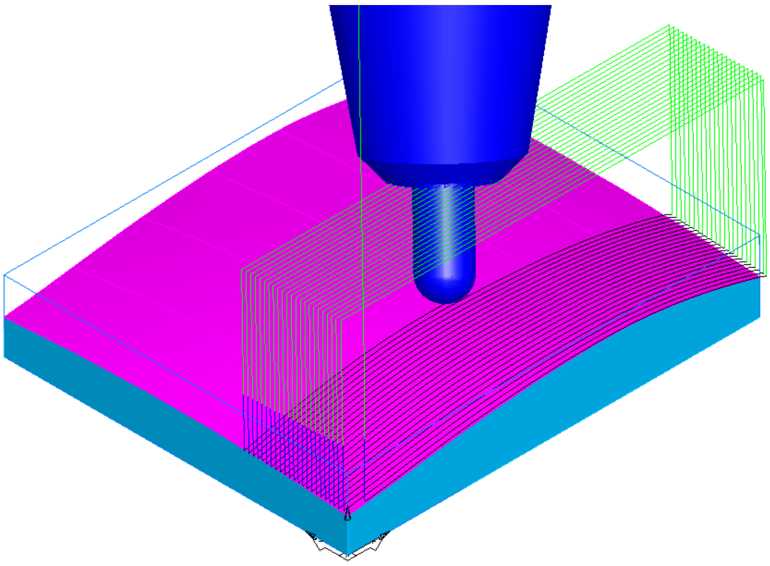
You will notice that the resulting toolpath looks different to a normal parallel toolpath. It looks as though the passes are closer on every second pass. This is because the centerline toolpath is showing the position of the centre of the tool rather than the point of contact on the surface. The tool tilts over in opposite directions on each pass so the position of the centre of the tool shifts on each pass.

Note how the tool flips its direction at the end of each pass. This will cause excessive movement on the machine. When using lead and lean it is best to use unidirectional cutting.



- Open the feature and click on the **finish1** operation
- Select the **Milling tab**
- Click on **Direction** and then check **Unidirectional**
- Run a **centerline simulation**

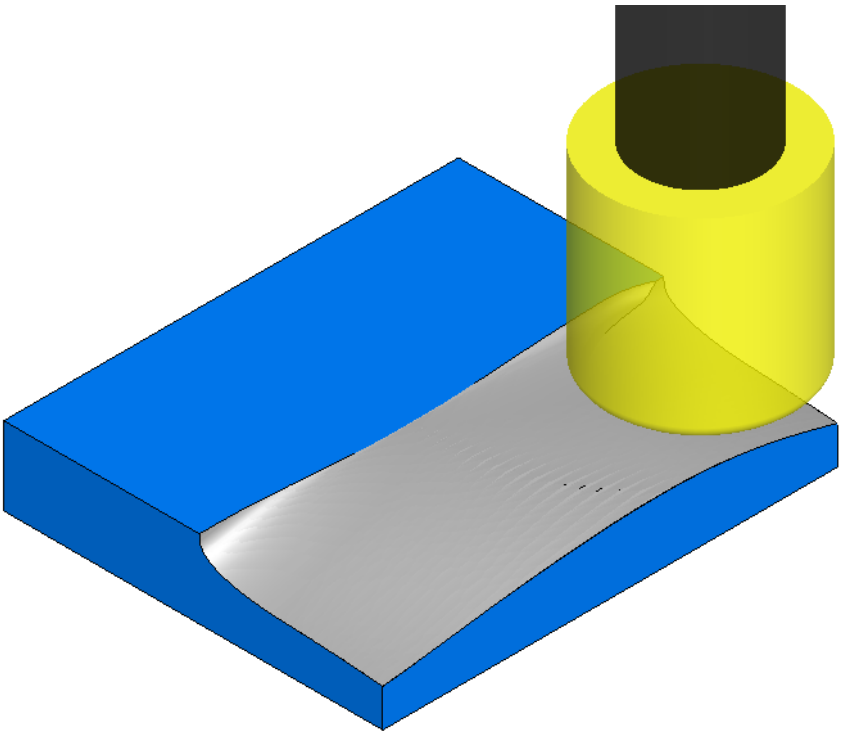
The tool now stays in the same orientation from pass to pass. The toolpath is now "cutting air" more than before but the more consistent motion of the tool axis will mean that it can be run at a higher feedrate.



- Open the feature properties, click on **Parallel** and select the **5-Axis tab**
- Experiment with changing the lead and lean angles to different positive and negative values to see the effect

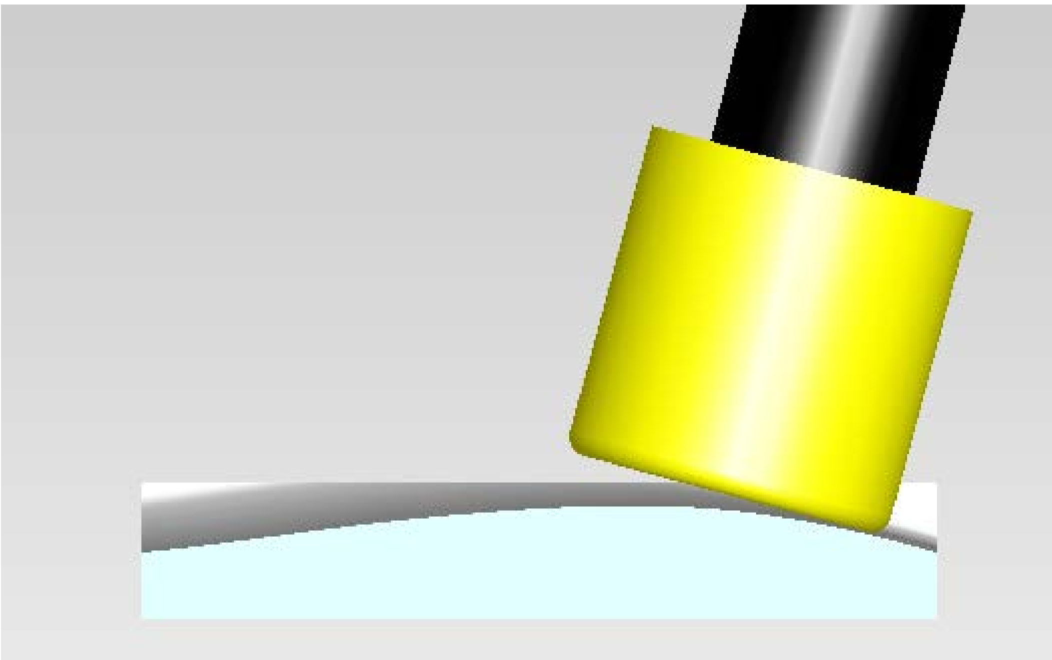
We will now look at another use of Lead/Lean. We will machine the same part but using a large bull nose end mill. Using this cutter, we can make a large stepover but still get an acceptable scallop height, however, we do not want to cut on the back edge of the cutter on the "downhill" parts of the toolpath. We can do this by tilting the tool forwards so that we always cut on its front edge.

- Uncheck the operation **contact** and check **front edge**
- Run a **3D simulation**



As the tool passes across the highest point on the part it has to switch from cutting on its front edge to the back. This may lead to a difference in surface finish and possibly a witness mark at the changeover point. If there were any recesses on the part they would not be machined as the flat of the tool could not fit in. We will now tilt the tool over at an angle so that it always cuts on the leading edge.

- Open the feature and click on the **Parallel** operation
- Open the **5-axis** tab
- Check **Use Lead and Lean** and enter a Lead angle of **15** degrees
- Run a **3D simulation**



You will see that the cutter is always cutting on its front edge now. Note that the surface finish is good even though the stepover is large. Tilting the cutter means that the profile which contacts the part has an elliptical section, this gives it a much larger effective diameter, so that a small bullnosed cutter or endmill can be equivalent to a very large ball nosed tool. This is a useful trick which is used to get a good surface finish on surfaces which have a very shallow curvature.

For most toolpaths, the lead and lean angle are relative to the setup Z axis and the direction of travel. For isoline toolpaths, the angles are relative to the surface normal.

- Uncheck the operation **front edge** and check **isolead**
- **Single step** a **centerline simulation** and note how the tool remains normal to the surface that is being machined

Note: In the case of flowline machining, the lead and lean angles are measured relative to the normal of the reference surface.

The **normal** of the toolpath is the direction along which it was originally projected onto the surface data during creation. For **Pattern finishing** this will always be vertical and for **Projection Finishing** it will vary depending on the defined projection, directional options.

From Point Tool Alignment

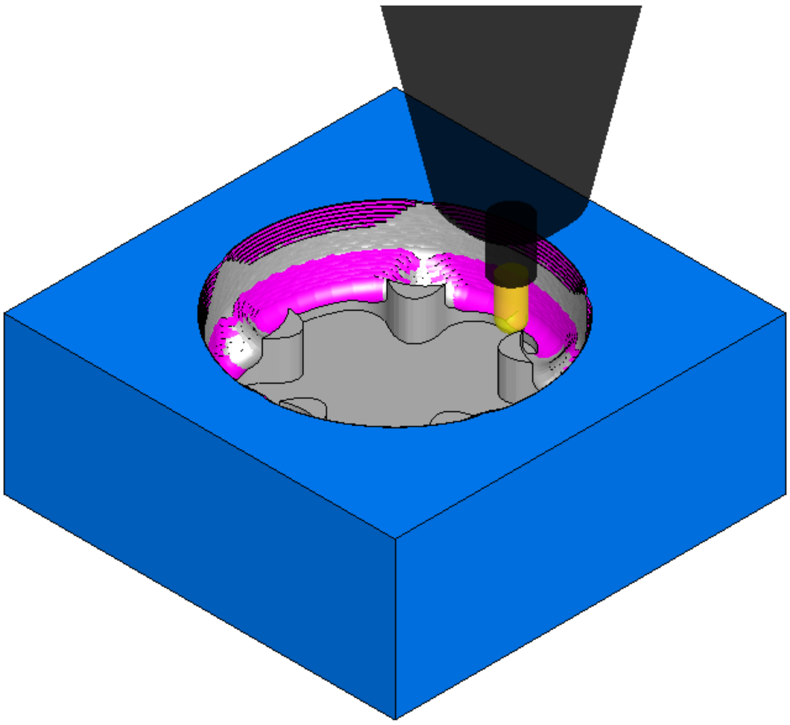
Introduction

In **From Point** alignment, the tool axis is constrained to always pass through a single point. The tool axis then effectively radiates out from that point towards the surfaces being machined. This is most useful when machining inside of an approximately hemispherical cavity or along a curved inside corner on a part.

In this example, we will machine across multiple surfaces using a Flowline strategy. The part is the bottom of a blow mold for a soda bottle. We shall be using the **From Point** axis alignment to prevent the tool holder from colliding with the part.

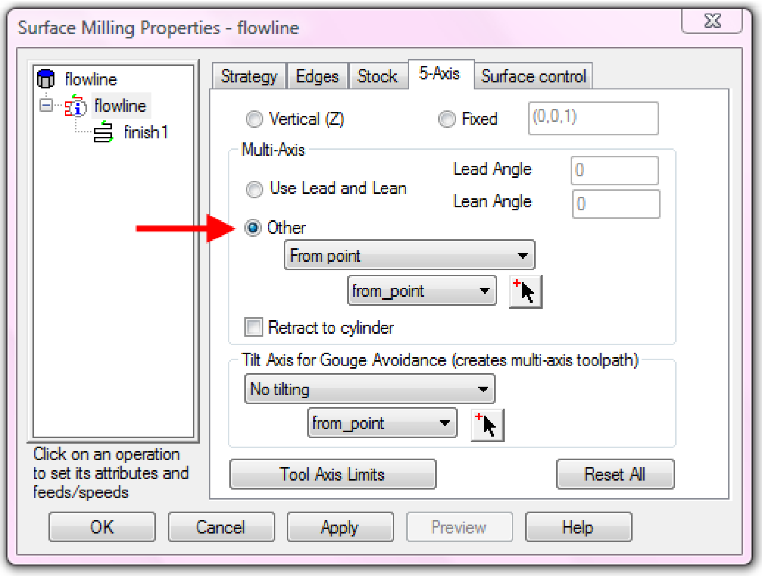
- Open the file **From_Point.fm**
- Select the tool crib **From_Point.fm_tools_from_last_save**
- Run a **3D simulation**

You will see that the holder collides with the part during the finish operation. This can be fixed by making the tool axis radiate from a point above the center of the part thus tilting the holder away from the stock.

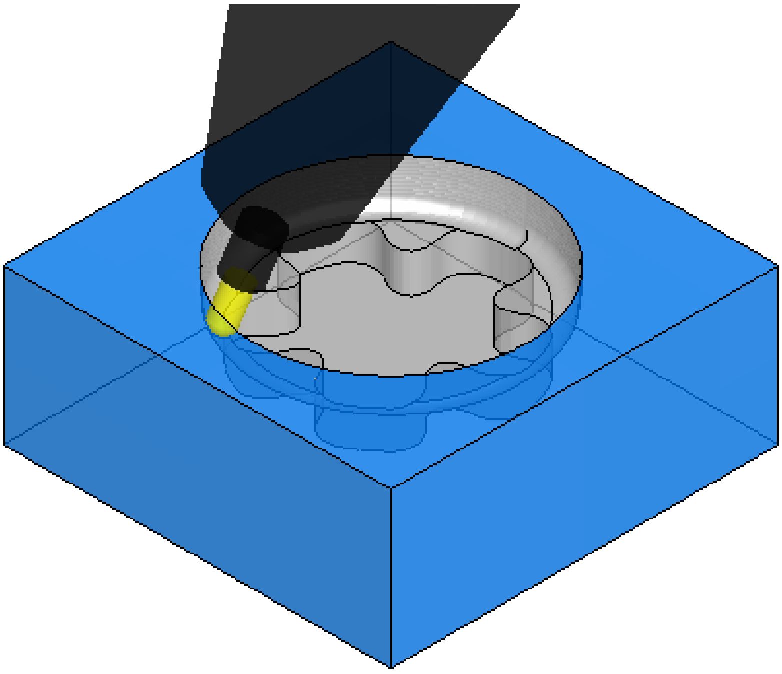


- **Eject** the simulation
- Open the properties of the **flowline** operation
- Select the **5-axis** tab
- Check **Other** and then select **From Point** from the pull-down menu

- Use the **Pick Location** button or the pull down menu to select the point “**from_point**” which is positioned 2” above the center of the part



- Click **Apply**
- Run a **3D simulation**



You will see that the tool axis now passes through the specified point which tilts the tool away from the stock preventing gouging. In this case, the point is close to the part which means that the spindle will be leaning over by a large angle at the start of the toolpath. We will now create another point which is higher above the part to reduce the angle that the part tilts whilst still ensuring that the tool and holder do not collide with the part.

- **Eject** the simulation
- From the main toolbar select **Construct** and then **Point**
- At the bottom of the screen enter the coordinates: 0,0,3

- Click **Create**
- Open the properties of the **flowline** operation
- Select the **5-axis** tab
- Check **Other** and then select **From Point** from the pull-down menu
- Use the **Pick Location** button to select the new point which is located 3" above the center of the part
- Click **Apply**
- Run a **3D simulation**

The tool axis is now passing through a point which is further away from the part. This means that the angle through which it moves from the top to the bottom of the part is reduced. In turn, this will reduce the angle through which the machine must move. You should remember that in 5-Axis simultaneous machining a small linear movement on the part may translate into a very large movement of the machine axes, whatever you can do to reduce these movements will allow you to run the toolpath faster.

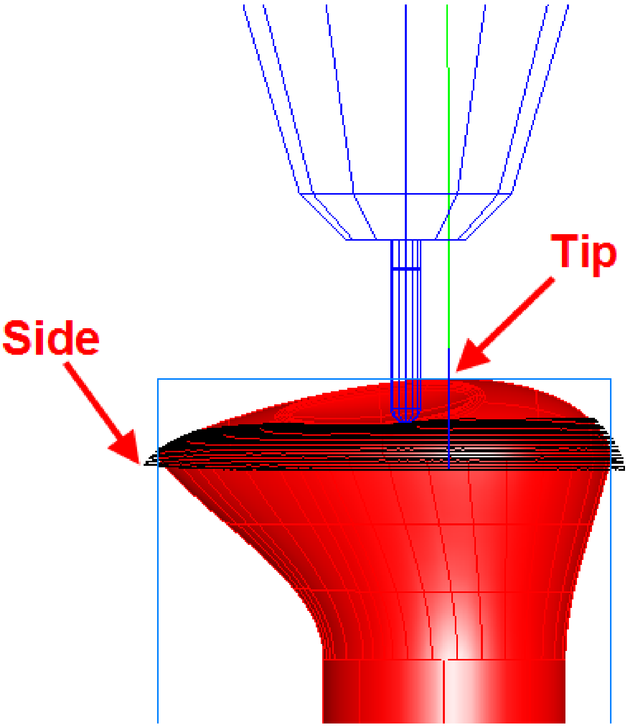
- Close the part

To Point Tool Alignment

Introduction

In **To Point** alignment the tool axis is focused towards a single point through the surfaces being machined. In this example we will machine the upper part of a model using a To Point alignment.

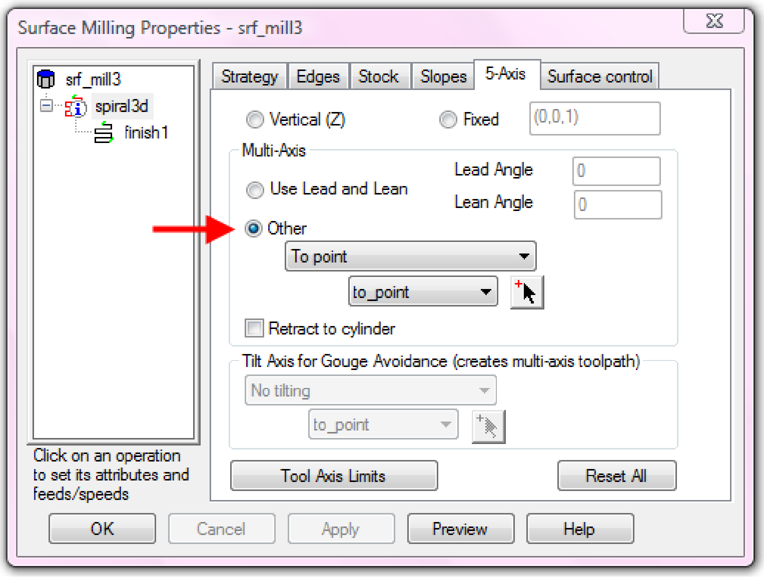
- Open the part **To_Point.fm**
- Run a **centerline simulation**



You will see that the point of contact of the tool is changing. At the lower edge of the part, the tool is touching on its side. When it reaches the top of the part, it is touching on its tip. This means that the surface cutting speed and tool loading are varying by a large amount over the part. The variation in cutting conditions will give a different surface finish as the tool moves from the bottom of the part to the top. By changing the tool axis direction, we can give a more constant cutting condition resulting in a more consistent surface finish.

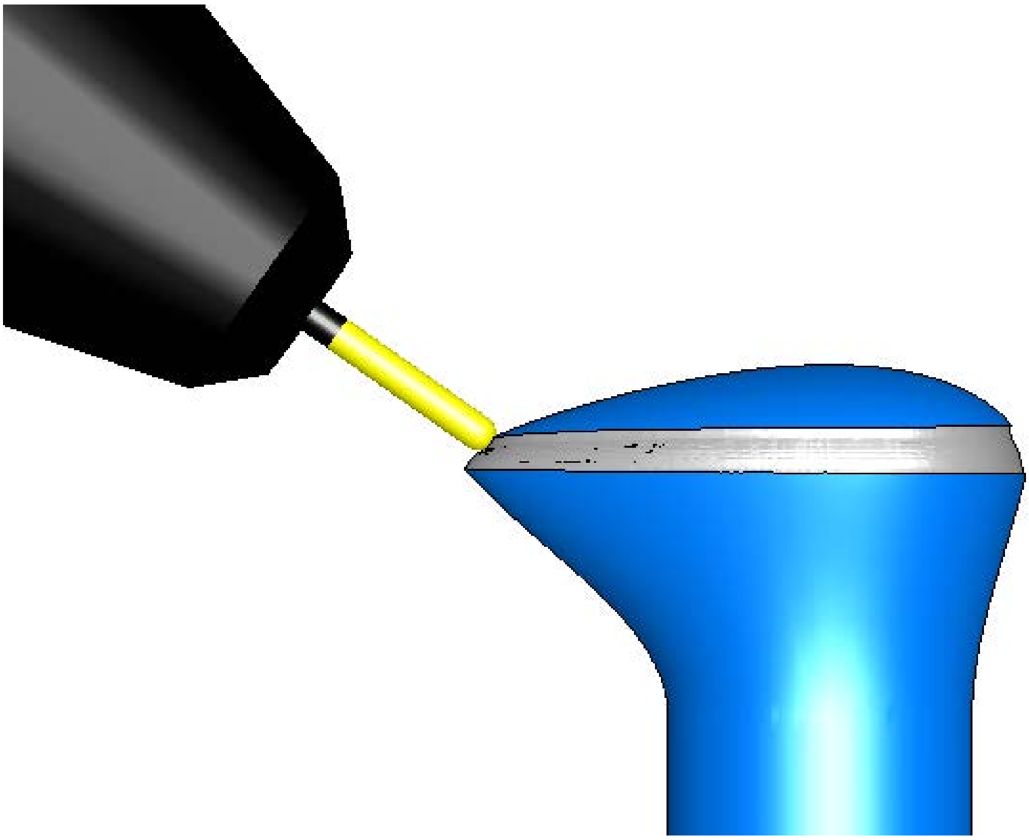
- Open the properties of **srf_mill3**
- Click on the **spiral** operation and then the **5-axis** tab

- Check **Other** and then select **To Point** from the pull-down menu
- Use the **Pick Location** button or the pull down menu to select the point "**to_point**" which is positioned 2" below the top of the part

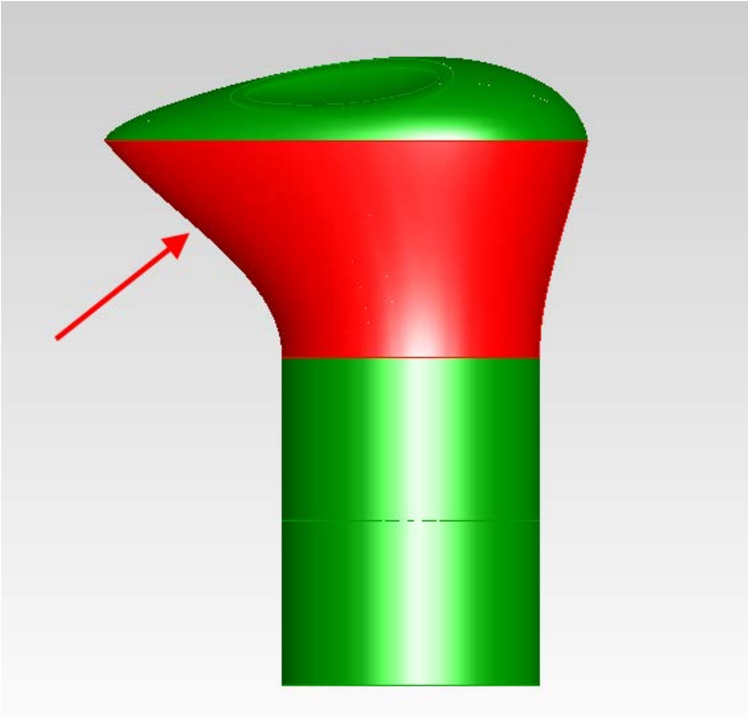


- Click **Apply**
- Run a **3D simulation**

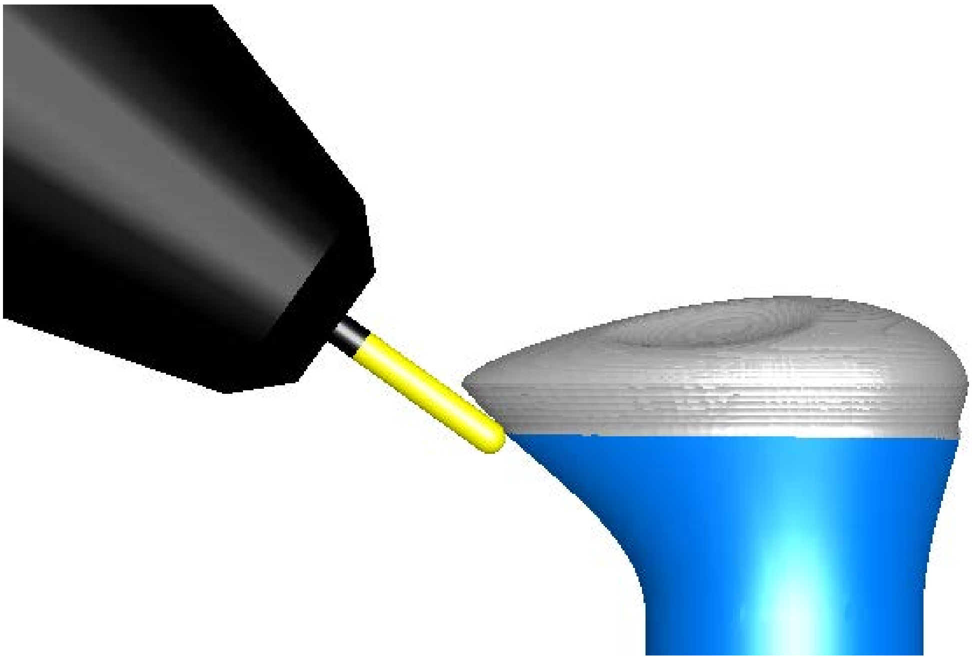
Note how the tool axis always points towards the point during the cutting operation.



- As an exercise machine the undercut surface indicated below. Use the same tool as the upper surface. You will need to use an Isoline toolpath and create a “to point” at (0.2,0.-0.5)



By selecting an appropriate “To Point” we are able to machine the undercut and also ensure that the tool axis does not dip so low that the machine head would collide with the table. If the point is too low, we cannot reach the entire undercut. If it is too high, then the head will tilt over too far. Try experimenting with the point Z position to see the effect.



Hint: If you like a little help with this exercise take a look at the part To_Point_Fin.fm.

From Line Tool Alignment

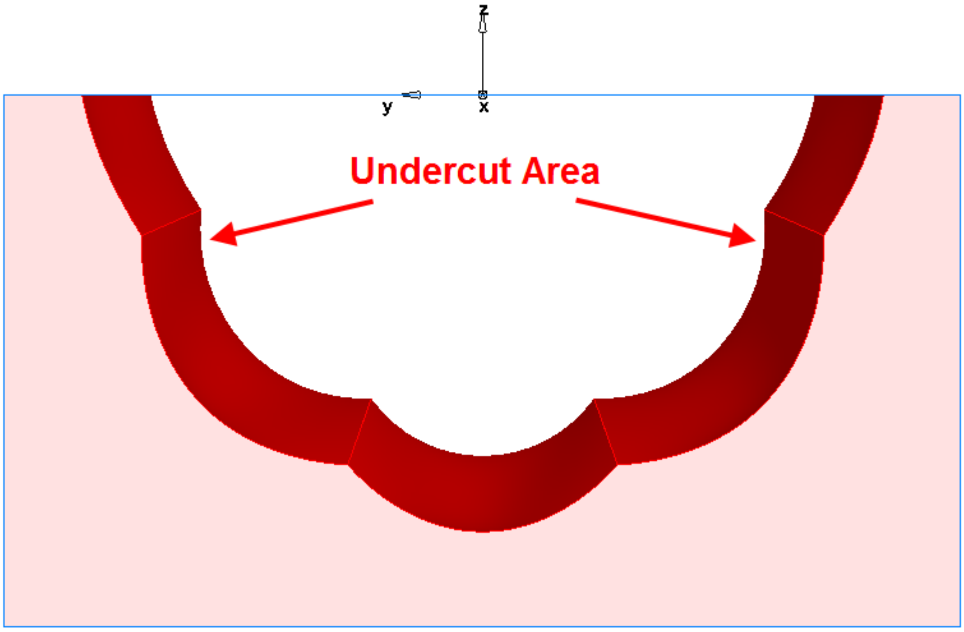
Introduction

With a From Line alignment the axis of the tool passes through a selected line. The line may be created using the geometry constructors within FeatureCAM or may be imported from an external CAD system such as PowerSHAPE.

The line used to control the tool axis can be horizontal, vertical or at any angle in between.

The technique is useful when machining open ended cavities with undercuts as in the following example.

- Open the part **From_Line.fm**
- **Shade** the model
- Take a **Left View**

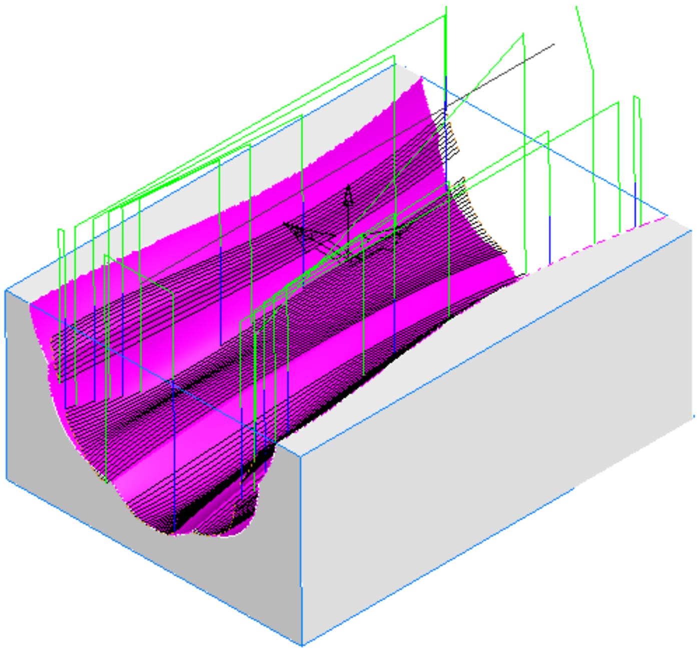


The part is to be finish machined using an Isoline strategy. There are two undercut areas on the part which a ball nosed tool will not be able to reach when aligned vertically. This will have two effects; the tool will leave material in the undercut areas and the point of contact will drop abruptly as the tool passes over the edge of the undercut.

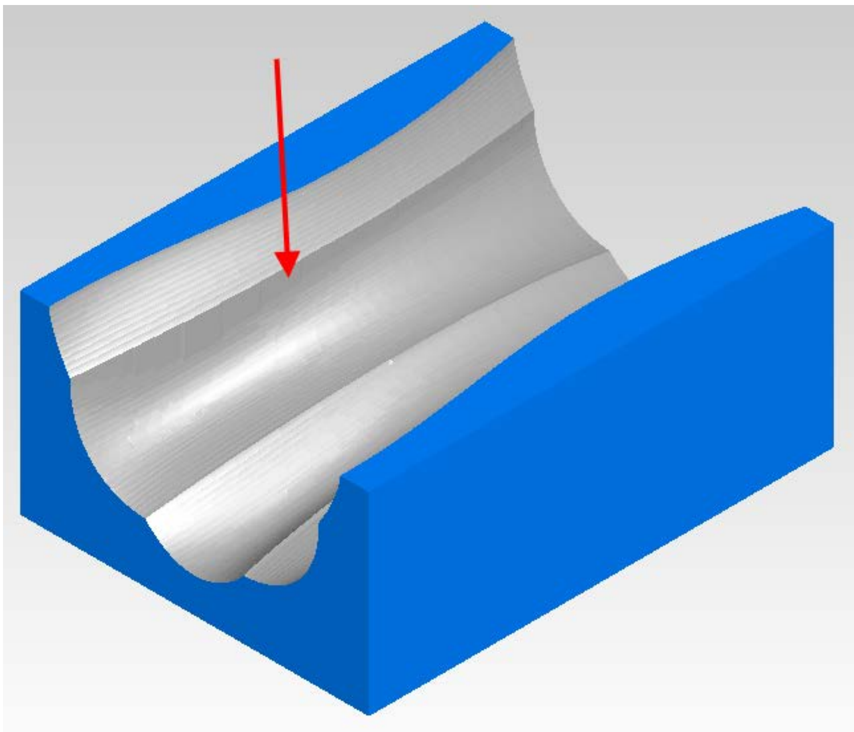
- Select an **Isometric view**
- **Uncheck** the **Roughing** operation

- Run a **Centerline simulation**

As the tool cannot "see" the point of contact on the upper edge of the lower surface consistently there will be many "lift-offs".



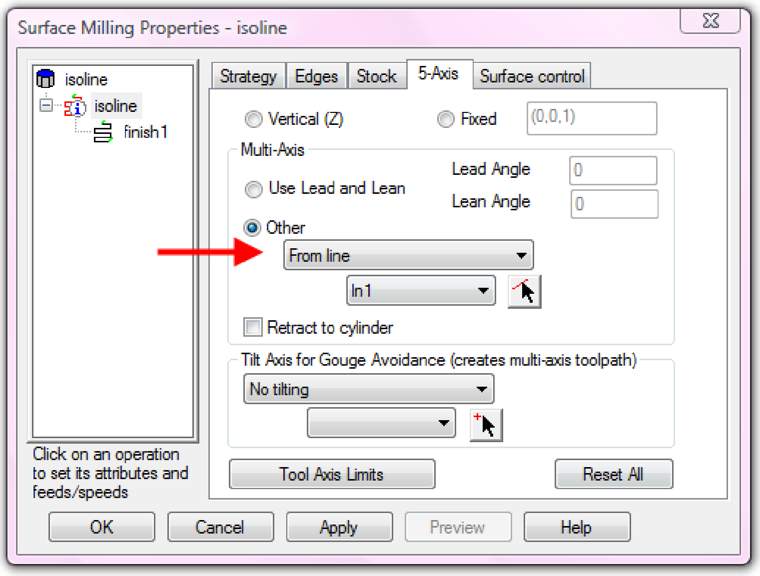
- Run a **3D simulation**



It can be seen that the undercuts in the area indicated below have not been finished and there are some very heavy cuts as the tool drops over the edge of the upper surface.

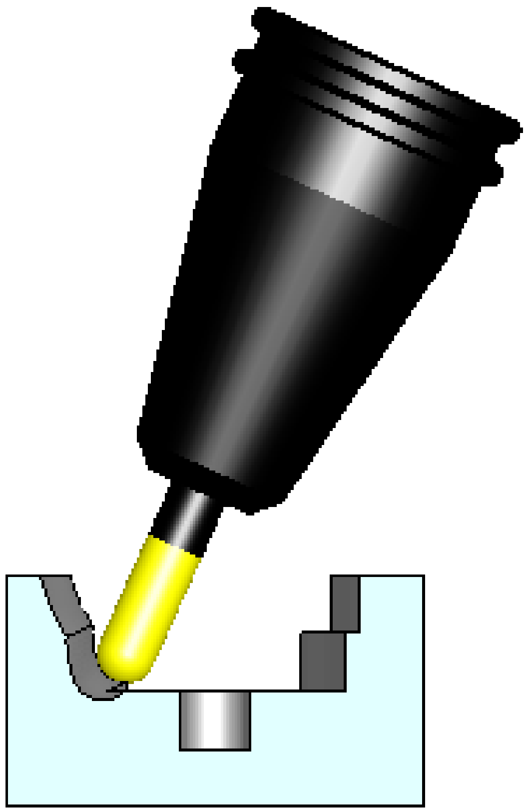
- Open the properties of **isoline**
- Click on the **isoline** operation and then the **5-axis** tab

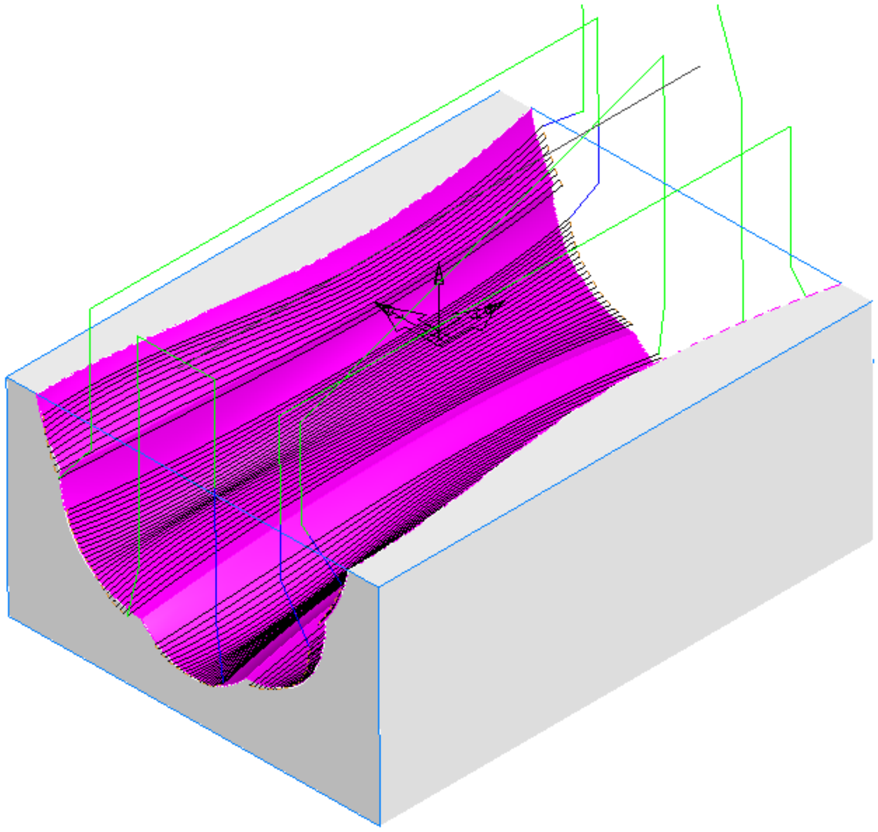
- Check **Other** and then select **From Line** from the pull-down menu
- Use the **Pick Location** button or the pull down menu to select the line "**ln_1**"

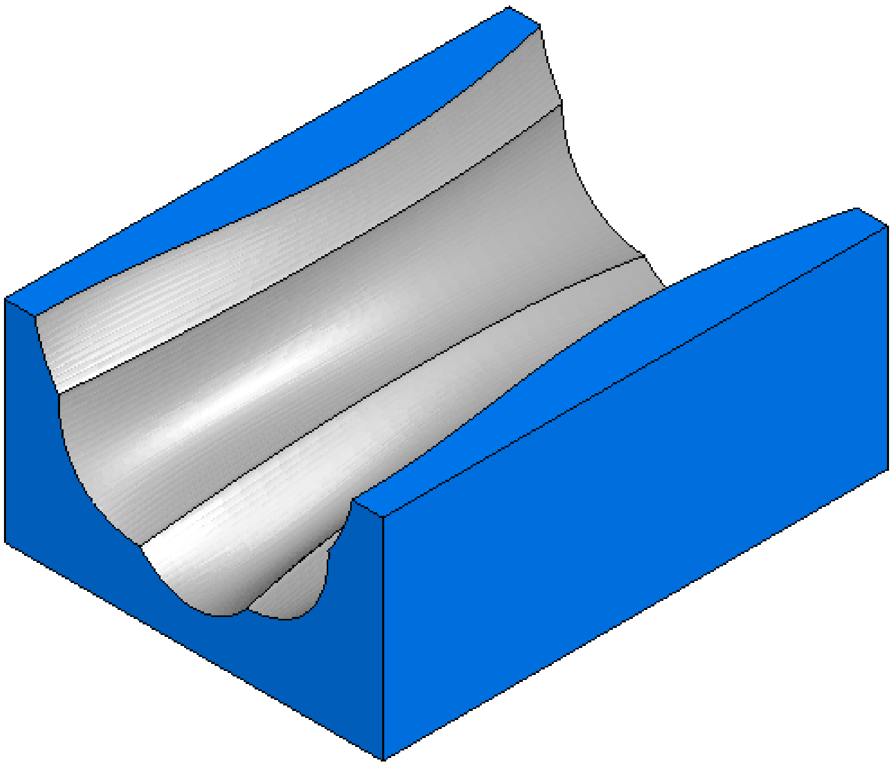


- Click **Apply**
- Run a **3D simulation**

The tool now tilts over so that its axis passes through the selected line. This allows it to reach the undercut areas of the lower surface and maintain a constant stepover. This reduces the tool loading, gives an improved surface finish, and also means that the tool is lifting off less frequently.







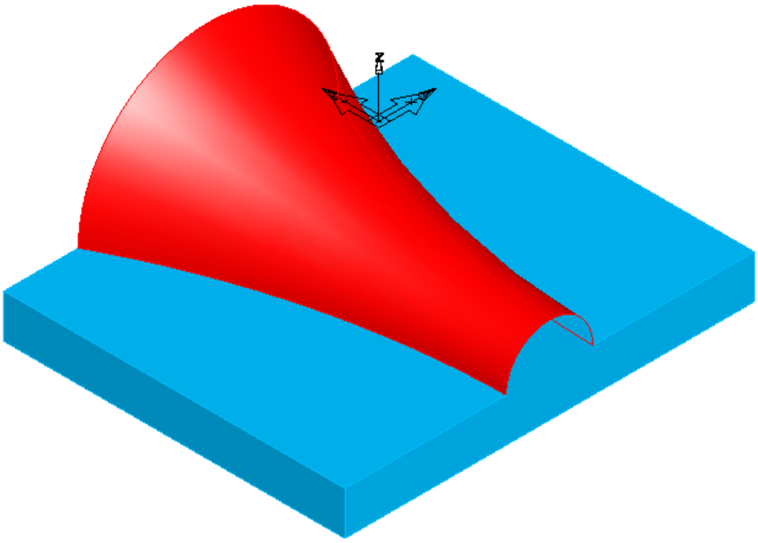
- As an exercise, create a new line which is 0.5” lower than the existing one. Use this to control the tool axis and see the difference in the orientation of the tool to the part

To Line Tool Alignment

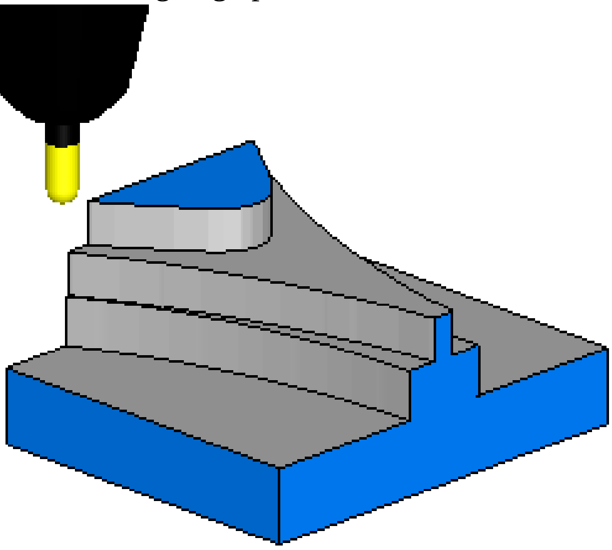
Introduction

With a **To Line** alignment, the axis of the tool points towards the selected line. The line may be horizontal, vertical, or inclined at an angle. In the first example, we will use a To Line alignment to prevent the holder colliding with the part.

- Open the part **To_Line.fm**

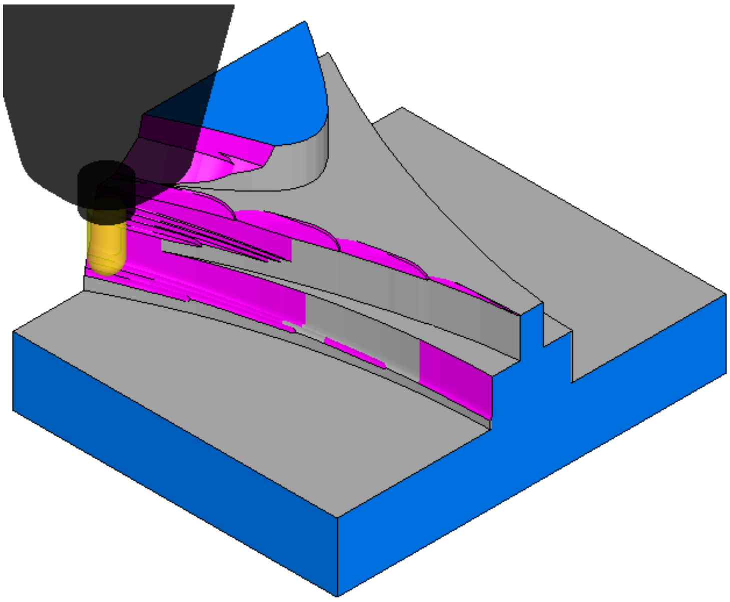


The part is to be roughed out with a Z level rough strategy and the flat areas finished with a 2.5D Face feature. The curved surface will then be finished with an isoline strategy using a short ball nosed cutter. As the roughing stepdown is large and the finishing cutter is short, there may be a problem with the shank or holder of the finishing cutter colliding with the remaining material from the roughing operation.



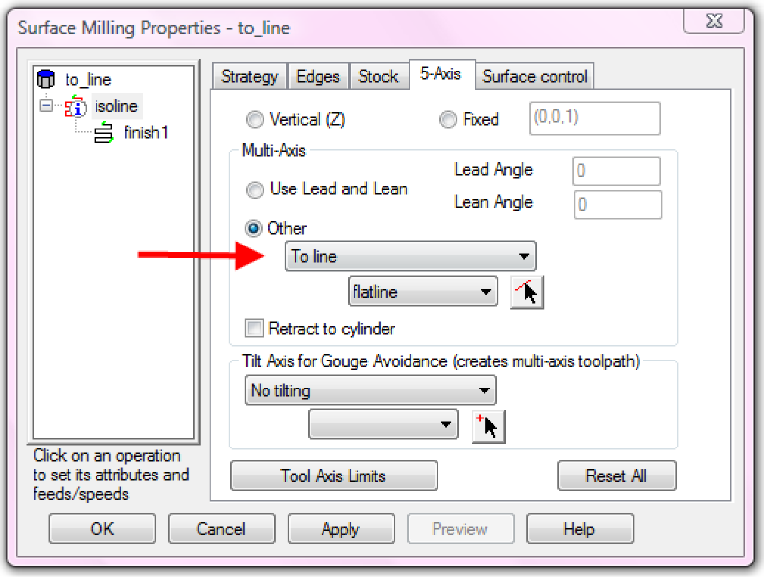
- Select the tool crib **To_Line_Tools_from_last_save**

- Run a **3D simulation**



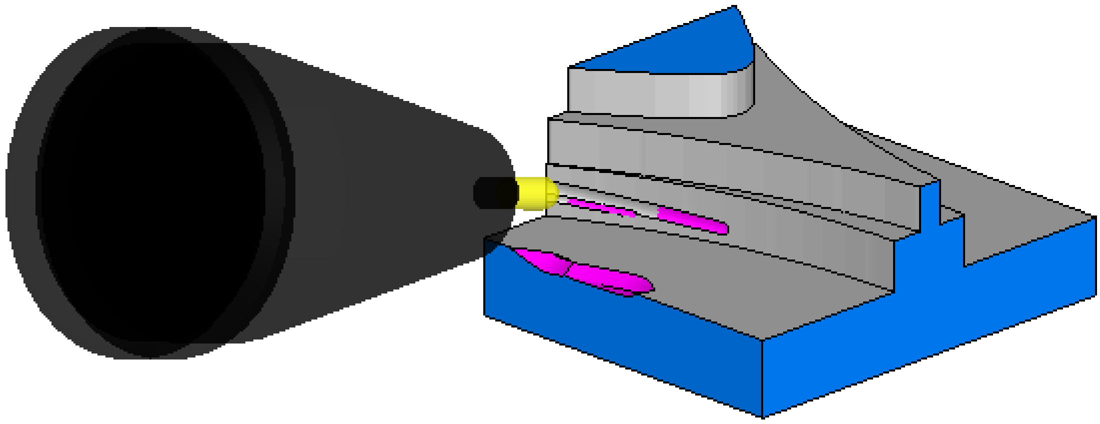
Note that the tool holder is colliding with the remaining stock from the roughing operation. We need to tilt the tool away from the part to prevent this happening. We will now change the tool alignment to be **towards a line** to prevent the collisions. Initially we shall use a horizontal line for the focus of the tool axis.

- Open the properties of **isoline**
- Click on the **isoline** operation and then the **5-axis** tab
- Check **Other** and then select **To Line** from the pull-down menu



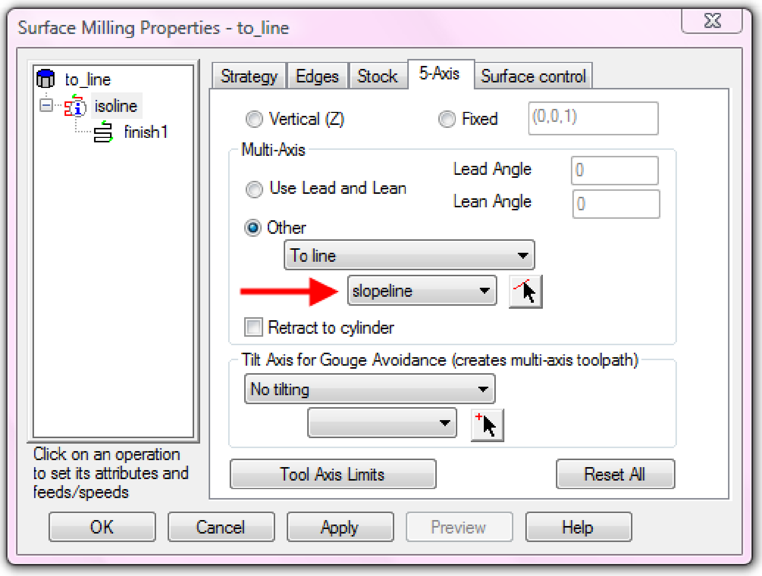
- Use the pull down menu to select the line **Flatline**
- Click **Apply** and then **OK**

- Run a **3D simulation**



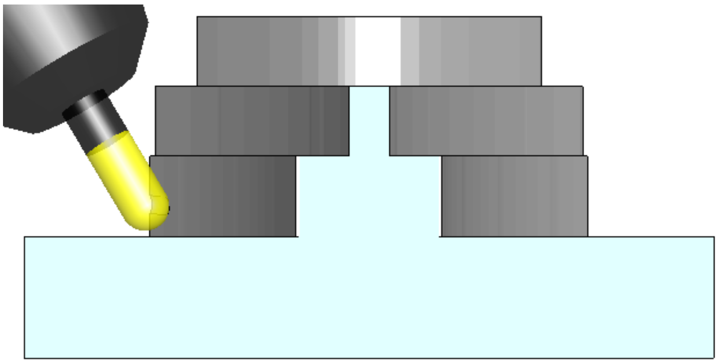
You will now see that the tool inclines so that its axis passes through the horizontal line. This is tilting the holder clear of the steps left by the roughing operation. Unfortunately, the angle of tilt is too steep and the holder is now hitting the lower flat area of the model instead. As the line drops in Z the angle of the tool tilting becomes shallower. By positioning the line lower, we can avoid this problem. A better solution however, is to use a sloping line so that we can give a more constant angle of contact between the tool and the part.

- Open the properties of **isoline**
- Click on the **isoline** operation and then the **5-axis** tab

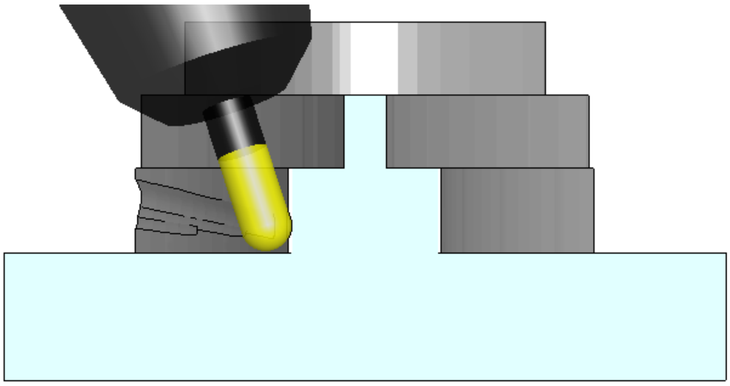


- Use the pull down menu to select the line **Slopeline**
- Click **Apply** and then **OK**

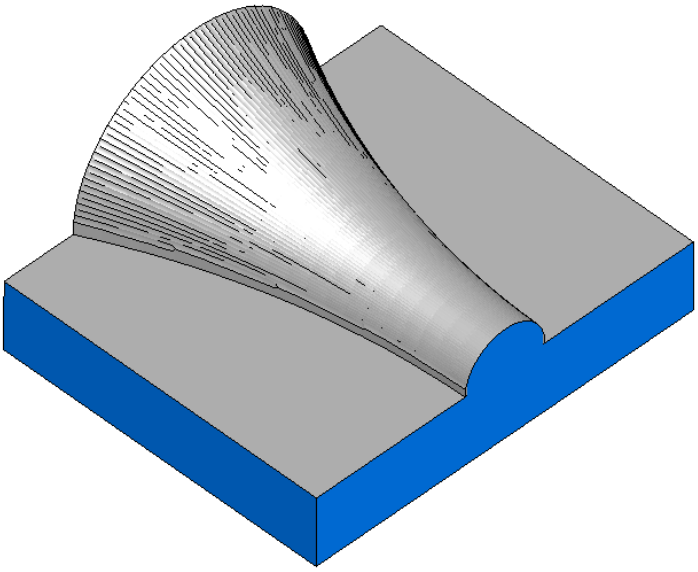
- Run a **3D simulation**



At the wide end of the part the tool is pointing to the lower end of the line which makes the tool axis more vertical than before.



At the narrow end of the part the line is higher which tends to tilt the tool over further. The net result is to maintain the tool angle to closer the vertical within tighter limits defined by the slope of the line. The whole part can now be machined without hitting either the remaining stock from the roughing operation or the model itself.



Try editing the positions of the ends of the line **Slopeline** to see the effect on the tool axis during the finishing operation.

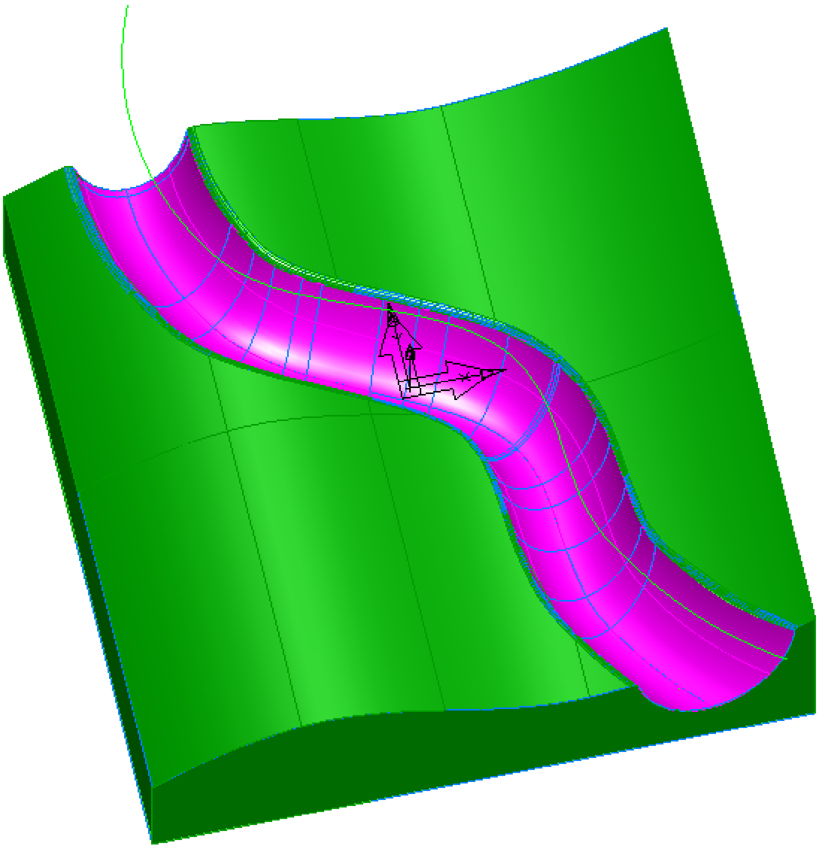
From Curve Tool Alignment

Introduction

FeatureCAM has the ability to align the tool axis from a curve. If this option is chosen, the tool is aligned so that its axis passes through the chosen curve while maintaining the intended contact point on the surfaces to be machined. This gives the user a very flexible way of precisely controlling the way that the tool is inclined as it passes over the part.

In this example, we will be machining the curved slot in the part shown below using an Isoline strategy.

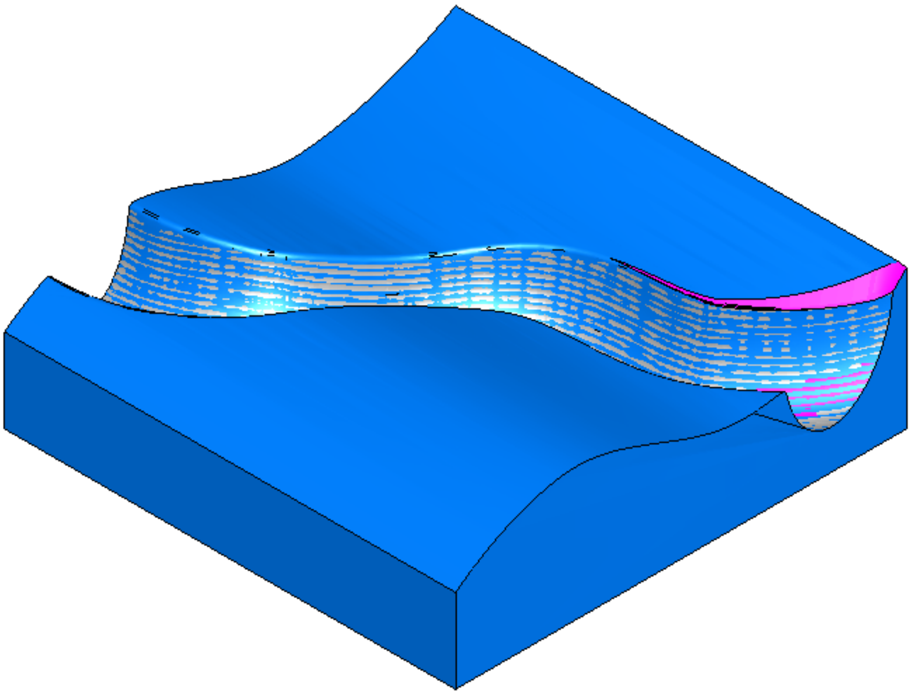
- Open the file **From_Curve.fm**
- Select the tool crib **From_Curve.fm_tools_from_last_save**
- Select a **shaded view**



The slot is deep relative to the tool length. As the sides of the slot are nearly vertical it would not be possible to use a Lead & Lean orientation. Equally From Point and From Line would be unsuitable as the slot has several changes in direction.

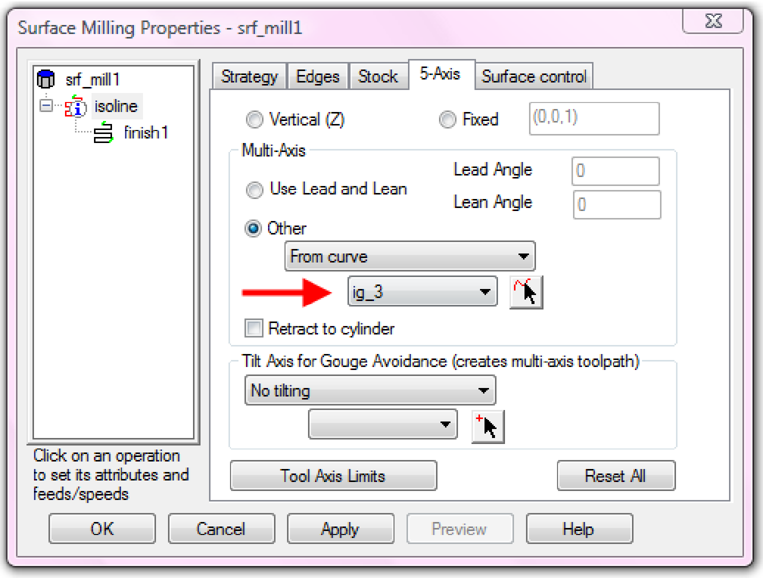
- Select an **Isometric View**

- In the **Options** menu select **Simulation**
- Check **Show Holder**
- Run a **3D simulation**



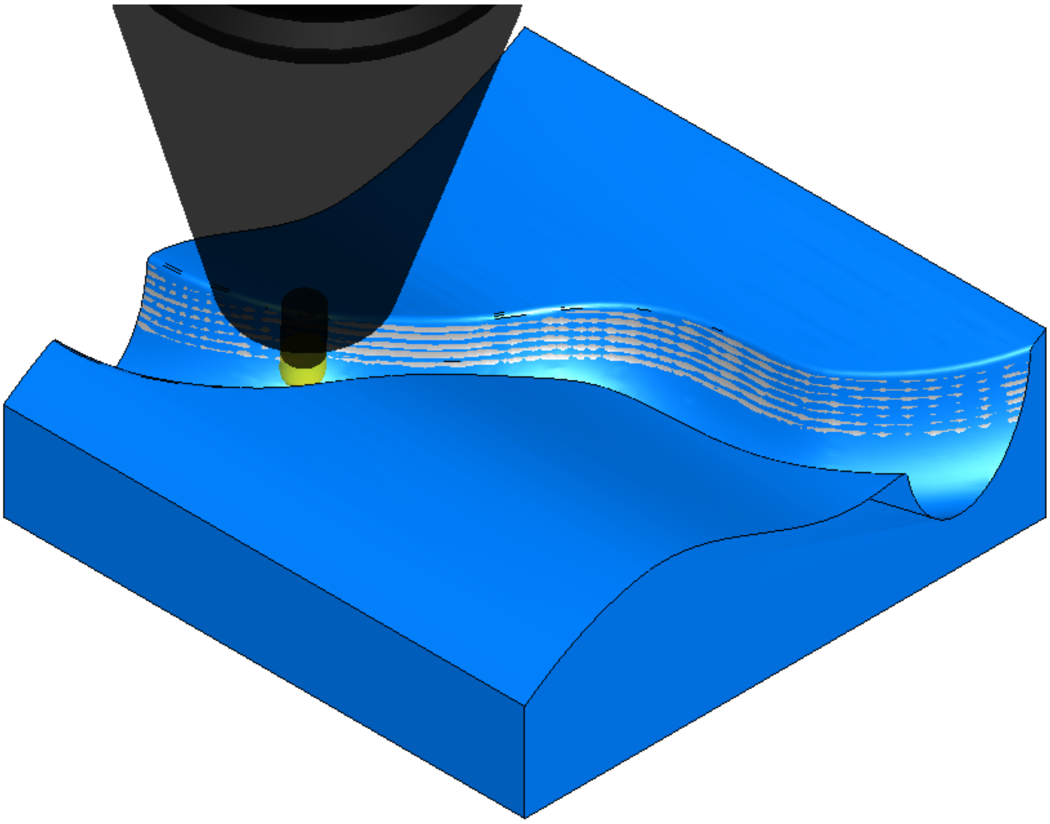
The holder is colliding with the model in the area on the right of the view above. We will now use a From Curve tool axis orientation to tilt the tool away from the part to remove this problem.

- Open the properties of **srf_mill1**
- Click on the **isoline** operation and then the **5-axis** tab
- Check **Other** and then select **From Curve** from the pull-down menu



- Use the pull down menu to select the curve **igs3**

- Click **Apply** and then **OK**
- Run a **3D simulation**



The axis of the tool is now following the line from the selected curve to the surfaces being machined. Because the curve is above the centerline of the slot, the tool is tilted away from both sides of the part removing all gouges.

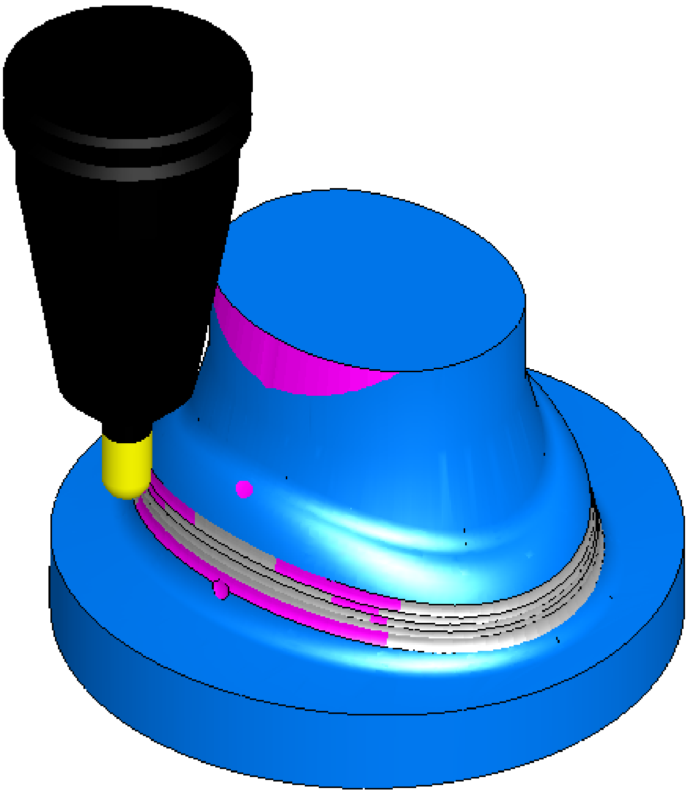
To Curve Tool Alignment

Introduction

The **To Curve** tool axis alignment aligns the centerline of the tool so that it passes through a user-defined curve. This gives the user much greater flexibility than other tool axis alignment options; by modifying the shape of the curve the tool axis can be controlled very precisely over a given part of the toolpath.

In this example, we will be machining a single surface using an Isoline toolpath. We need to constrain the toolpath so that the holder does not collide with the central boss of the part or the machine table.

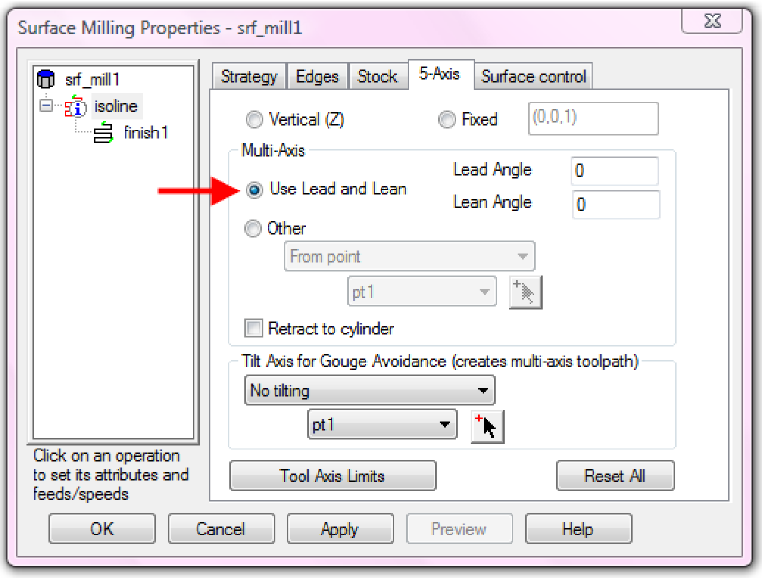
- Open the file **To_Curve.fm**
- Select the tool crib **To_Curve.fm_tools_from_last_save**
- Run a **3D simulation**



The tool is oriented along the Z axis by default. This is causing the holder to collide with the central boss on the part. We will now try using a Lean angle to tilt the tool away from the part.

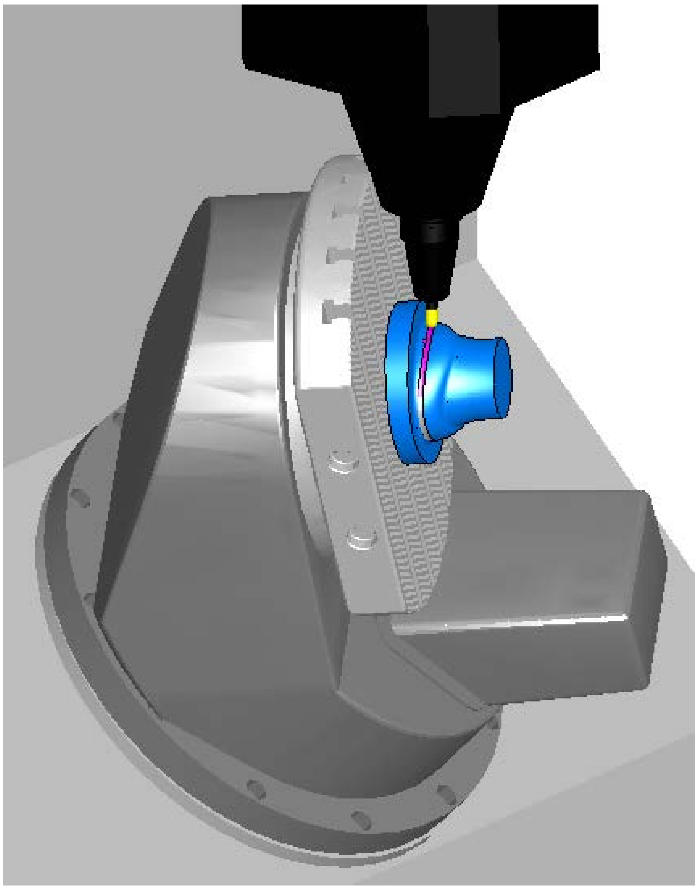
- **Eject** the simulation

- Open the feature properties, click on **Isoline** and select the **5-Axis tab**
- Set a **Lead** and **Lean** angle of **Zero degrees**



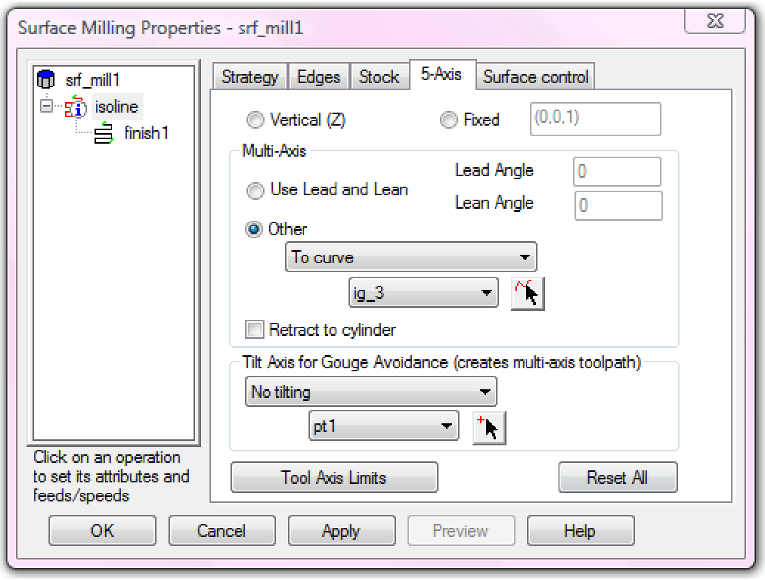
- Click **Apply** and then **OK**
- Select the **DMG Evo post** and the matching **MD file**
- Run a **Machine simulation**

When the tool is leaning over to be perpendicular to the bottom of the surface being machined, the spindle is striking the table of the machine.

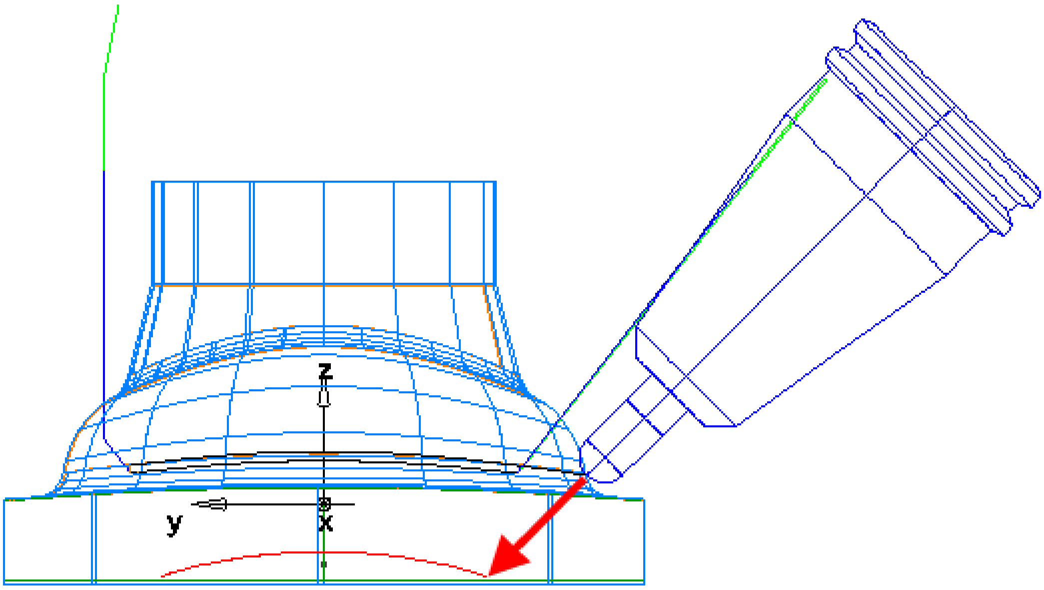


We could use a different lean angle to avoid the problem, but this would need us to change the cutting to unidirectional. This would mean that the C-axis of the machine would continuously cut in the same direction. With this particular machine, that would not be a problem, however, some machines cannot “wind up”. That is, there is a limit to the number of rotations an axis can make. We will now look at an alternative method of controlling the tool axis.

- **Eject** the simulation
- Open the feature properties, click on **Isoline** and select the **5-Axis tab**
- Check **Other** and then select **To Curve** from the pull down menu
- Select the curve **ig_3**



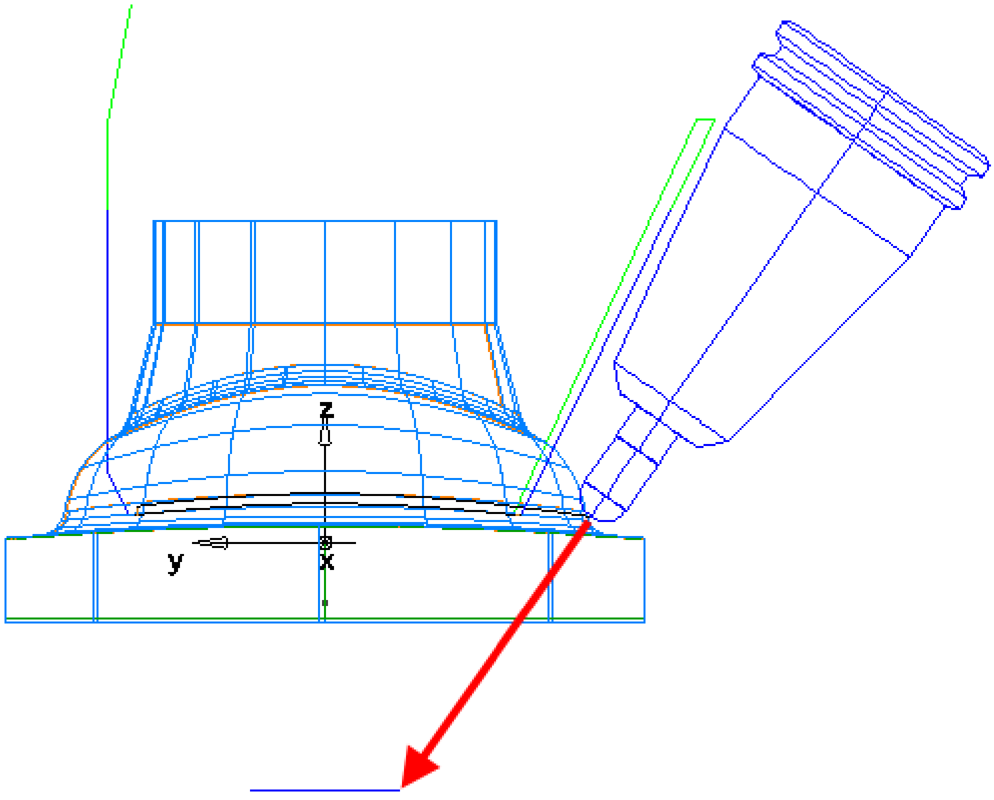
- Click **Apply** and then **OK**
- Select a **Left view**
- **Turn off Shaded view**
- **Step** through a **Centerline simulation**



As you step the tool around the job, you will see that the axis of the tool is always directed to the selected curve.

- **Eject** the simulation

- Open the feature properties, click on **Isoline** and select the **5-Axis tab**
- Check **Other** and then select **To Curve** from the pull down menu
- Select the **curve3**
- Click **Apply** and then **OK**
- **Step** through a **Centerline simulation**



As the curve is smaller and positioned lower, the tool axis is modified. It is steeper at the lower edge of the part leaning it further away from the table.

- As an exercise, try moving and changing the size of the curves to see the effect on the tool axis orientation.

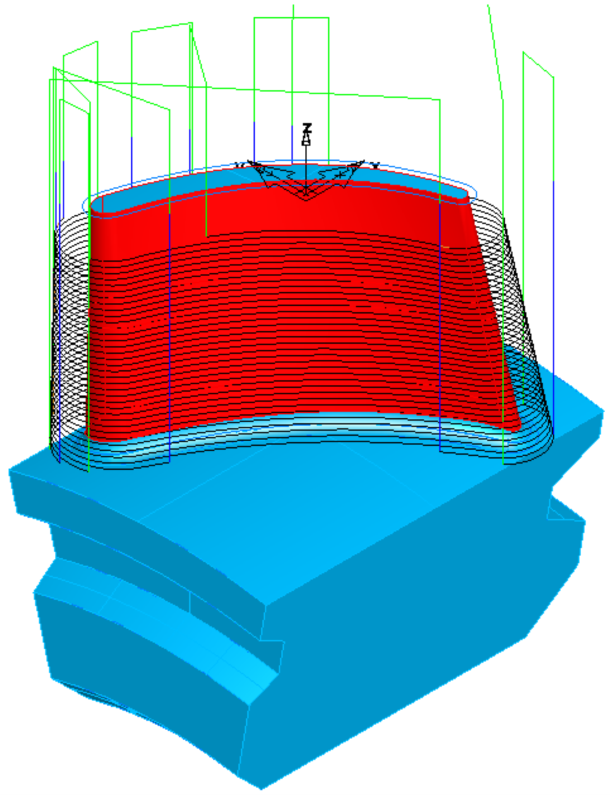
5-axis Swarf

Introduction

In swarf machining, the part is cut using the side of the tool rather than the tip. Not all surfaces can be cut with a single pass swarf toolpath. In order for it to work, the surface must be singly curved (a section through the surface must be a straight line in one direction) and developable (a surface which could be opened out flat without deforming it other than by unwrapping).

In this first example, we will machine a turbine blade using a swarf machining operation.

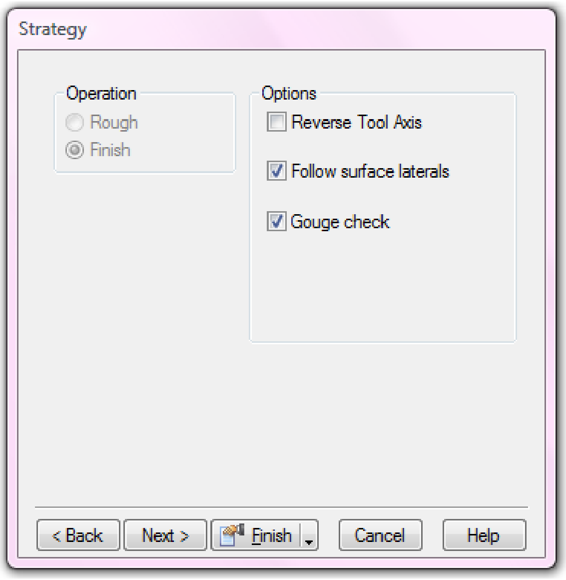
- Open the file **Swarf.fm**
- Select the tool crib **Swarf.fm_tools_from_last_save**
- Run a **centerline simulation**



The part is being finished using a Z Level finish toolpath. Note how the toolpath is fragmented at the top and bottom edges of the surface being machined. This results in multiple lift-offs and a poor finish on these edges. To get a good surface finish, the toolpath has to make many passes thus increasing the machining time.

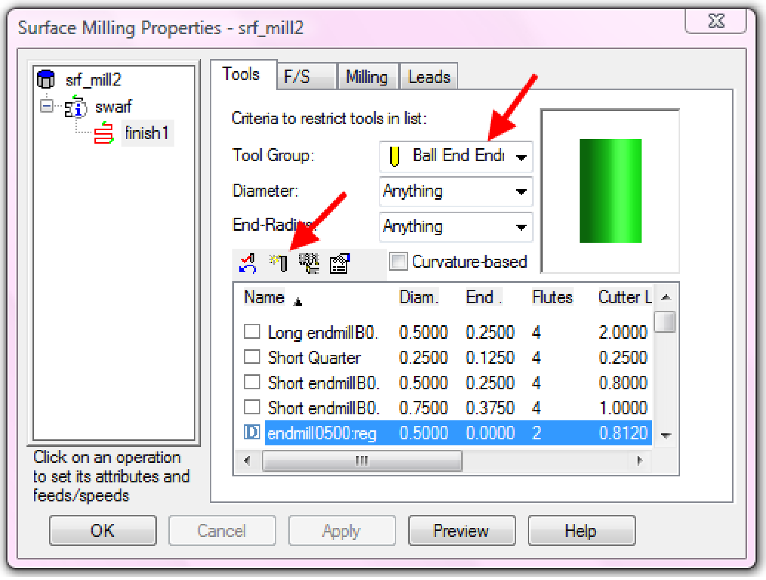
- Uncheck the feature **srfmill_1**
- Select the surface “**face_172**”
- Create a new **surface milling** feature
- Create a **Single operation**

- On the **New Strategy** page select **Swarf** then click **Next**
- Make sure that **Gouge check** and **Follow surface laterals** are checked



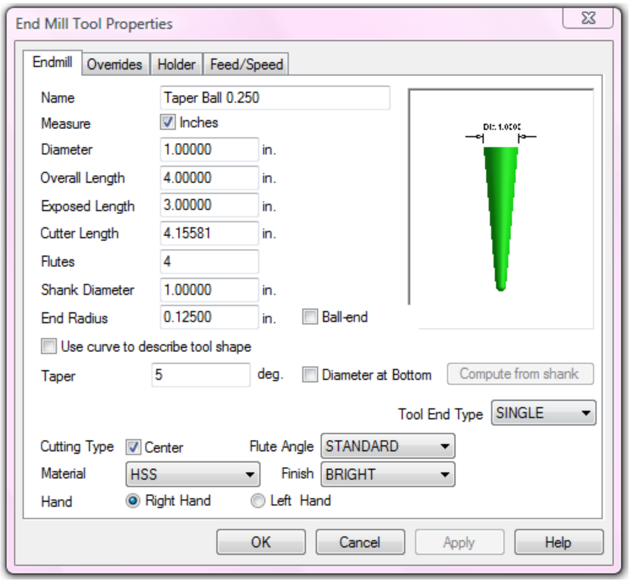
- Click **Finish**

In order to make the best use of the Swarf operation, we need to select or create a tool that is capable of finishing the whole surface in a single pass.

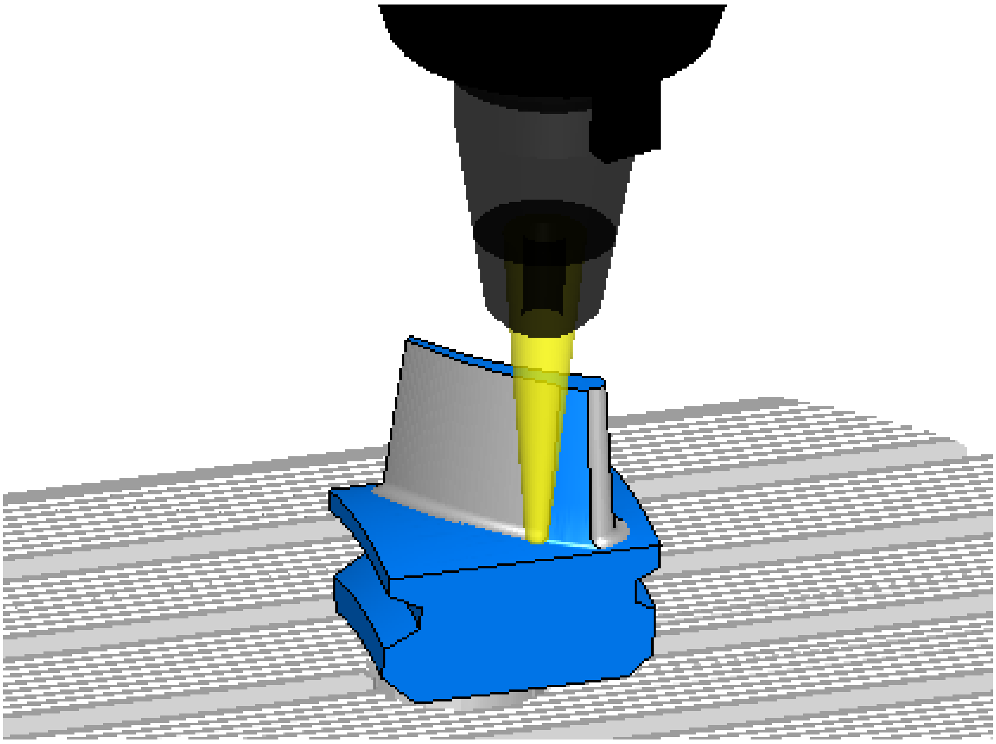


- Click on **finish1** and select the **Tools tab**
- Select the **Tool Group - Ball End Endmill**
- Select the tool **Long Endmill B0500** and then click on **New tool**

- Fill in the form exactly as shown below



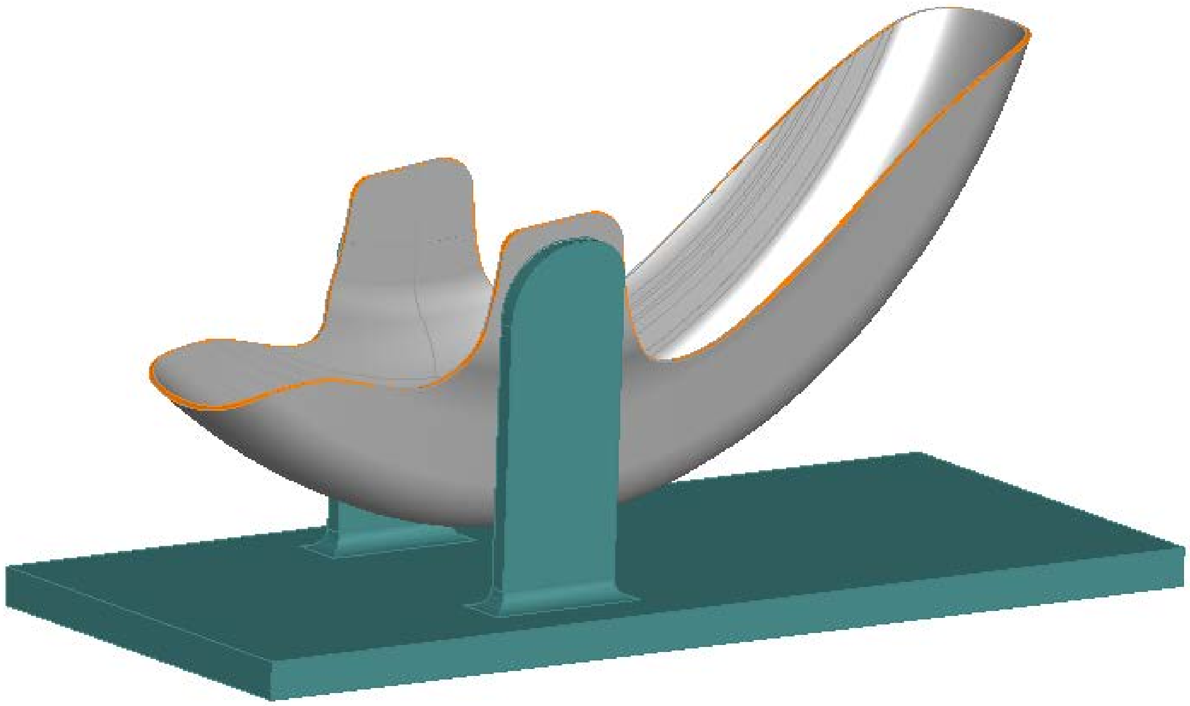
- Click on **Apply** and then **OK**
- Select the **DMU Evo post processor** and **machine design** file
- Run a **machine simulation**



The part is finished to the full depth with a single pass of the tapered tool. This both saves time and gives an improved surface finish as there are no scallops produced on the surface.

Swarf machining can also be used to finish the edges of manufactured parts. In this type of application, the side of the tool can be used to remove burrs from machined parts, flash from molded, cast or forged part and excess material from composite lay-ups. In this example, we will use a swarf toolpath to cut off excess material from a molding of a motorcycle fender.

- Open the file **Fender_Swarf.fm**

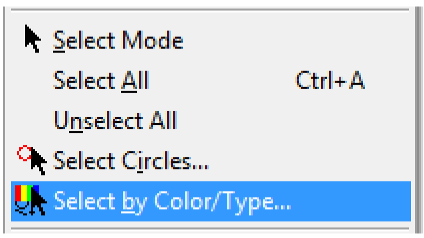


The model is mounted in a jig ready for the edges to be trimmed. We will trim the excess material from the part using a tool tilted over at 90 degrees to the edge.

- **DO NOT select** any surfaces
- Create a new **surface milling** feature and click **Next**

We need to select just the edge faces of the part. As they have a different color, it is a simple matter to use a filter to select only the orange surfaces that we wish to machine.

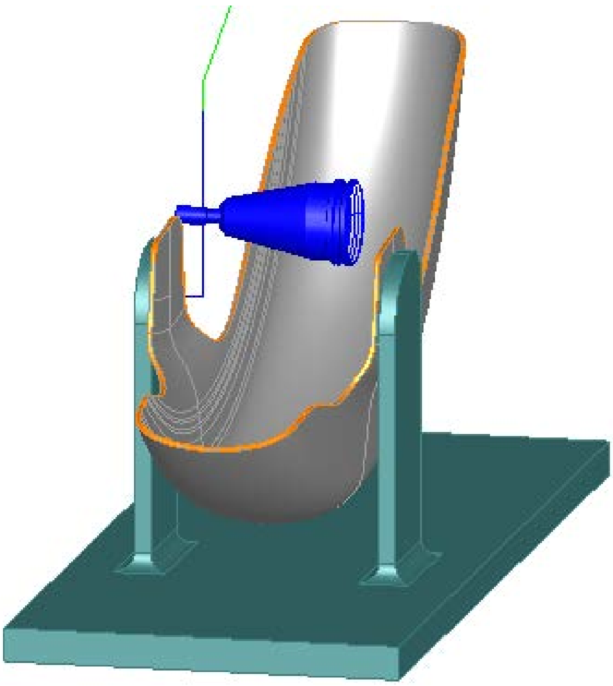
- From the **main tool bar** select **Edit**
- Choose **Select by Color/Type**



- From the **pulldown menu** select the **Orange color** then click **OK**
- On the **New Feature – Part Surfaces** form click the **green cross** to add the selected surfaces into the feature

- Click **Next** and select **Choose a single operation** then **Next**
- Select **Swarf** and click **Finish** then **OK**
- **Single step** a **centerline simulation**

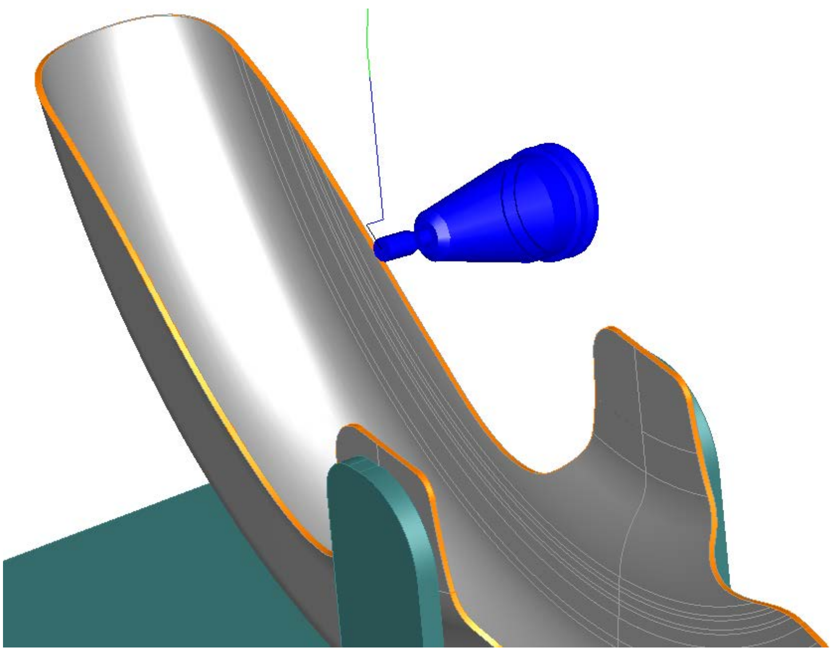
By default FeatureCAM chooses to approach the part from the side that is nearest to the setup. In this case we want to bring the cutter in from outside of the part so we need to flip over the tool. We can control this from the Strategy tab of the feature.



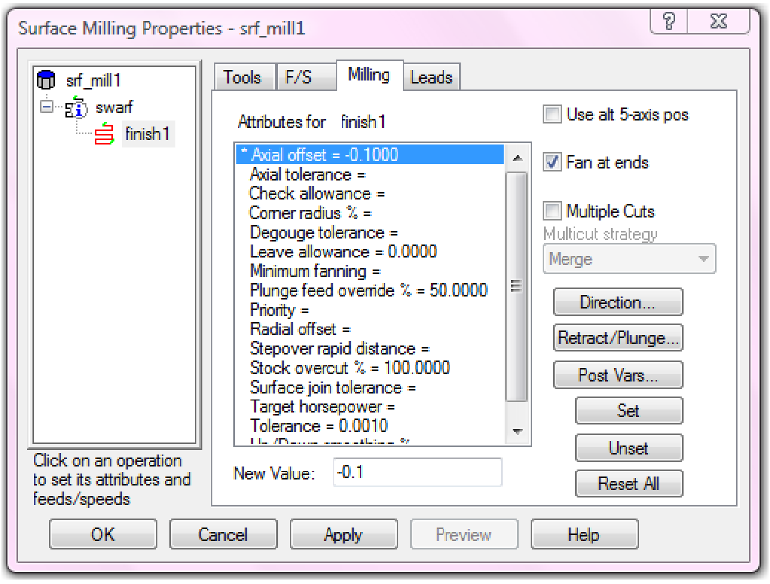
- **Eject** the simulation
- **Edit** the feature and select the **Strategy tab**
- Check **Reverse tool axis**
- Click **Apply** and then **OK**
- **Single step** a **centerline simulation**

The tool is now approaching from the correct direction. If you zoom up on the tool, you will see that it is machining exactly to the inside edge of the part.

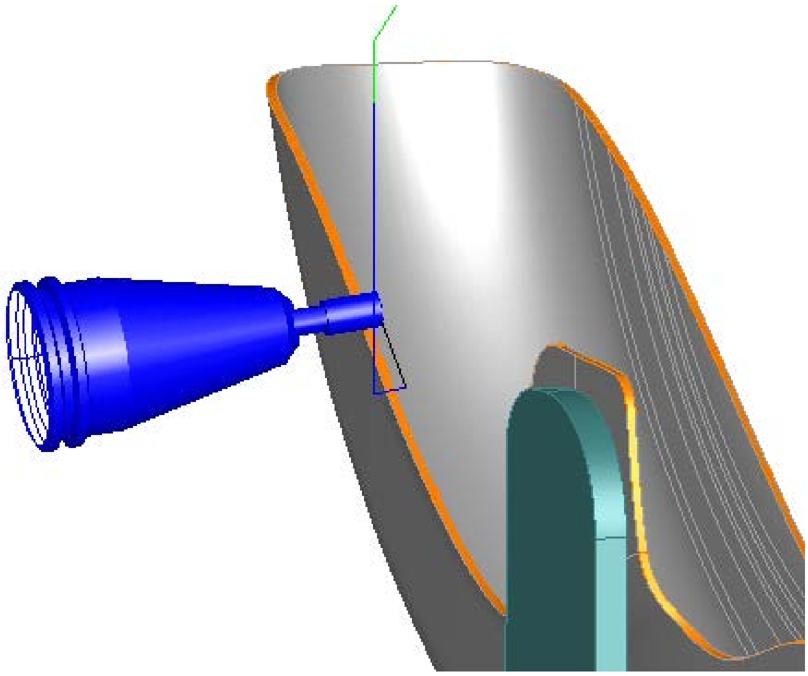
When trimming flexible workpieces such as this, it is often better to take the tool past the edge of the part to ensure that the edge is fully machined.



- **Edit** the feature and select the **Milling tab**
- Set the **Axial offset** to **-0.1"**



- Click **Apply** and then **OK**
- **Single step** a **centerline simulation**



By applying a negative Axial offset, we are forcing the tool to machine with its tip past the edge of the surface to be finished. This will ensure that there is a good finish on the edge.

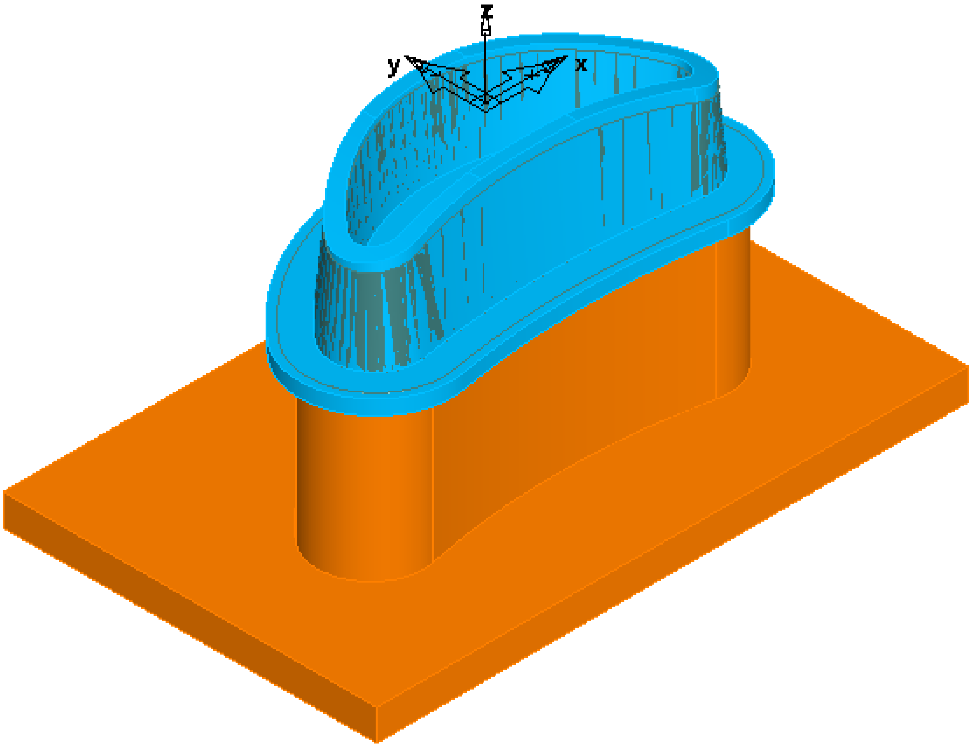
5-axis Trim

Introduction

The 5-Axis trim toolpath is used for trimming up the edge of moldings, castings, lay-ups, etc. The tool makes two passes around the edge of the selected surfaces to trim off excess material. Either the face of the surface or its outer edges can be trimmed. If this toolpath is used on the outside edge of a part with a negative offset, it can be used for de-burring.

In this example, we will trim up the edge of a die-casting with an end mill. The tool axis will be set to be normal to the surfaces being machine by using a Lead and Lean of zero.

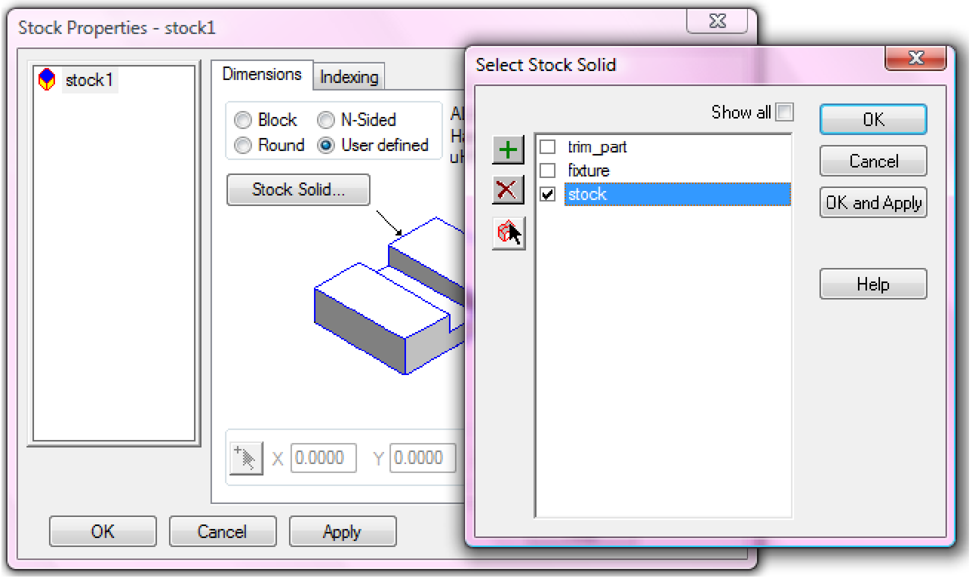
- Open the part **Trimming.fm**
- From the **view menu** select **Show all**



The part contains three solids; the silver one is the part to be machined, the blue one is the stock, and the orange is the fixture on which the part will be mounted. We will now set up ready for machining.

- Select the **tool crib basic**

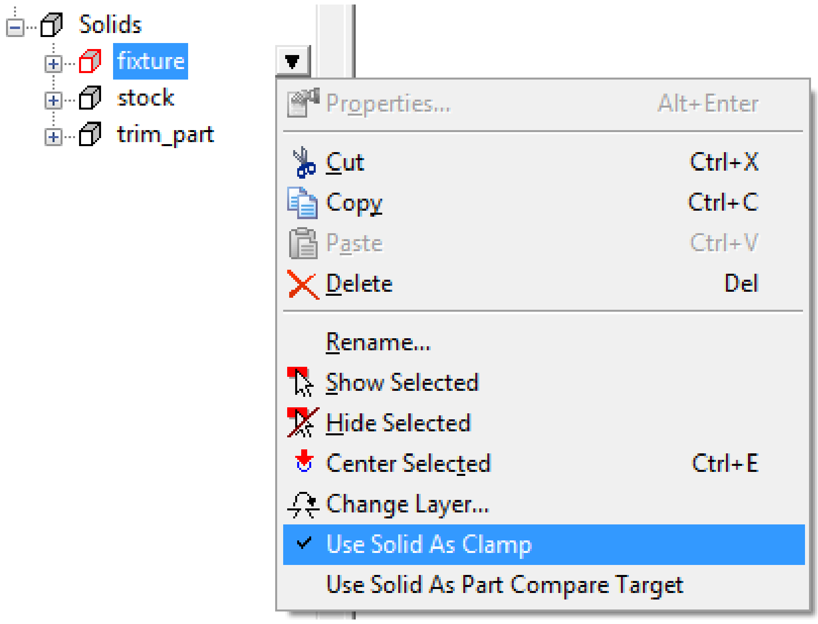
- Open the **stock properties**, select **User defined**
- Choose the solid **stock** from the list as shown



- Click **OK and Apply**
- Click **OK** to close the stock properties form
- In the **Part view,** right click on the **solid** called **stock** and **Hide selected**

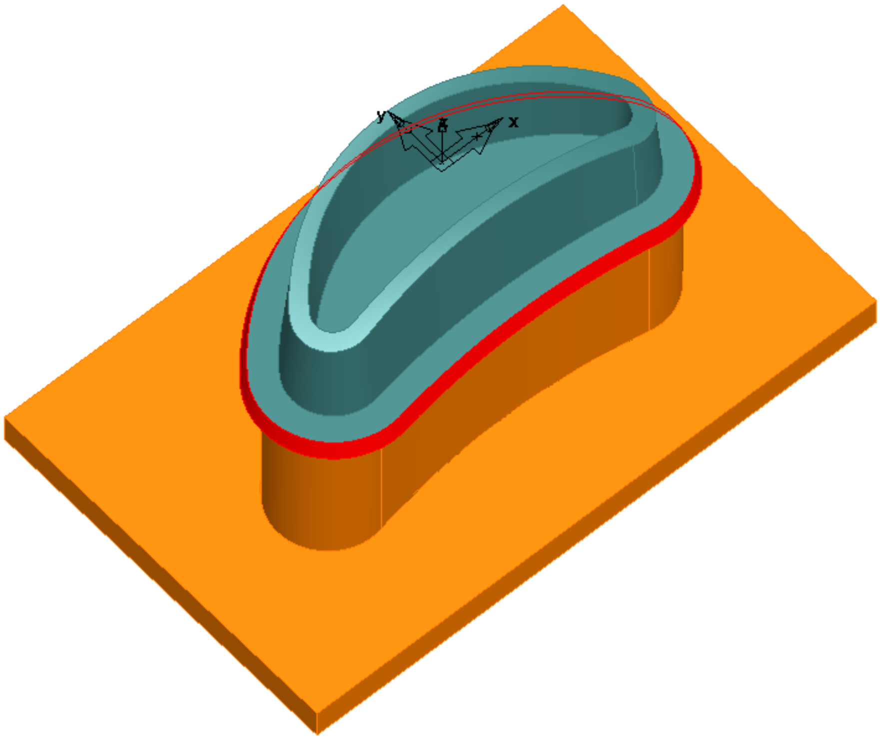
We will now register the solid fixture as a clamp so that it appears in the simulation.

- In the Part view **right click** on the **solid fixture**
- Select **Use solid as a clamp**

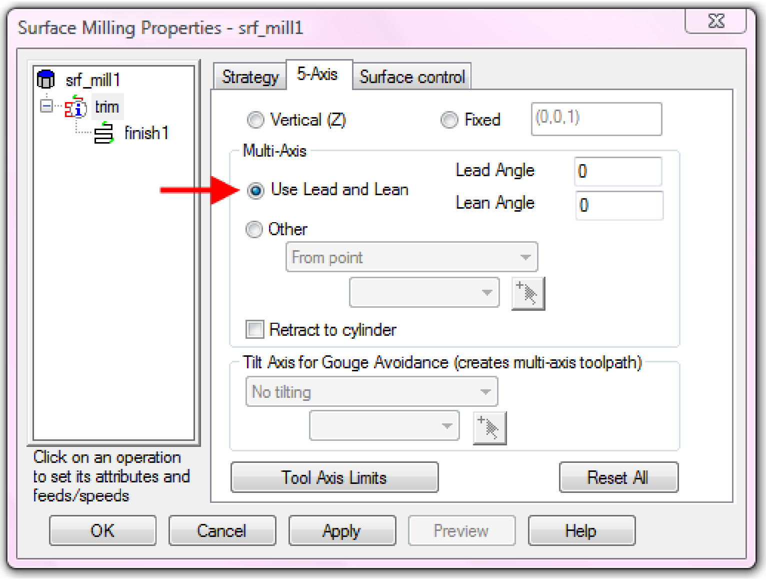


We are now ready to create the first trim toolpath. This will finish the lower edge of the part.

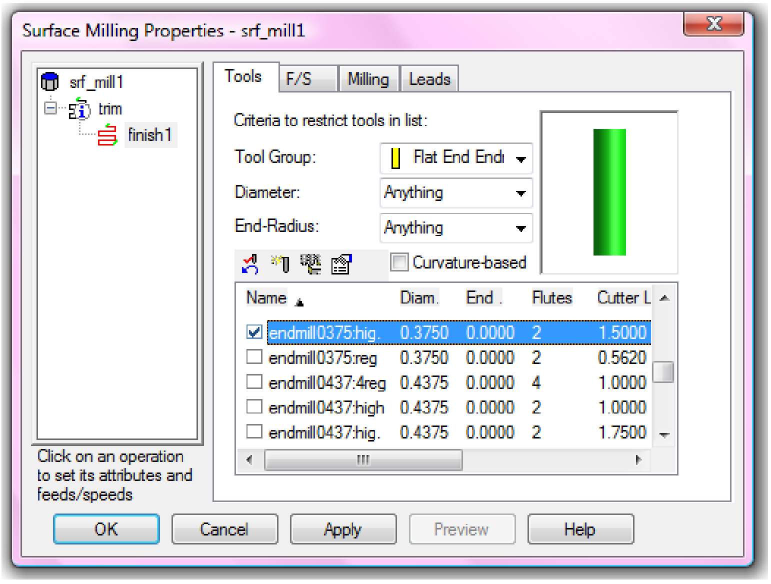
- Select the surface that makes up the lower edge of the part as shown



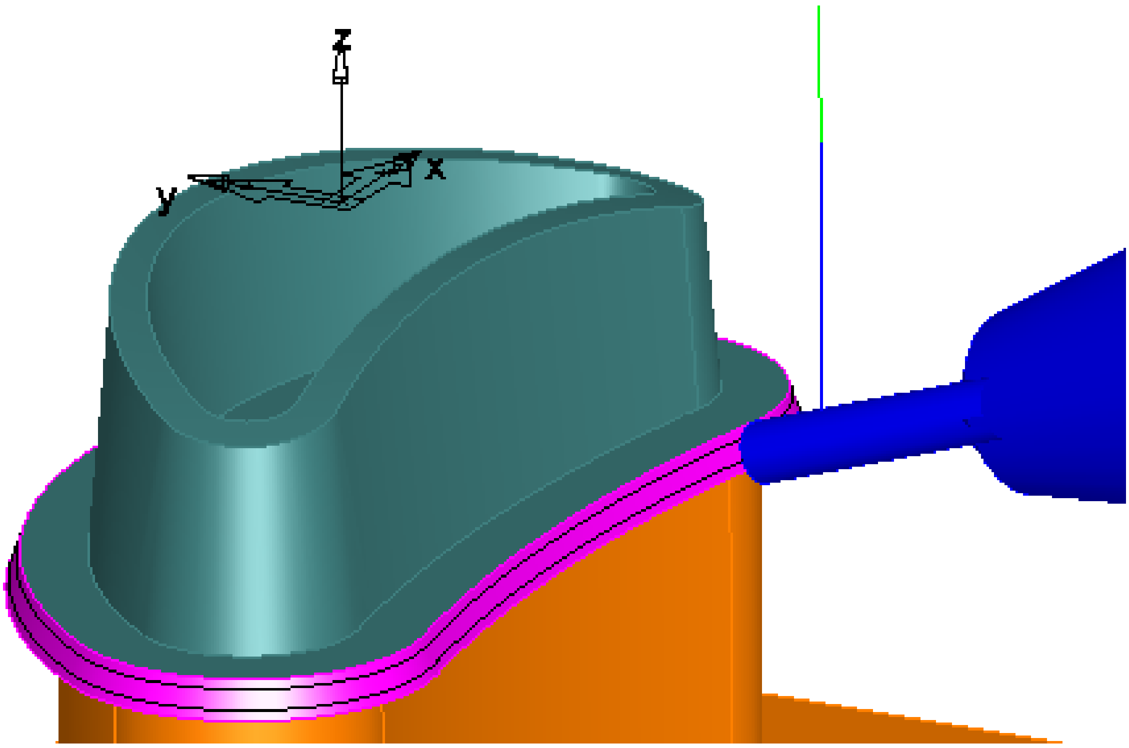
- Create a new **surface milling** feature
- Select **Choose a single operation**
- On the **New strategy** page select **5-Axis trim**
- Click **Finish**
- Select the 5-Axis tab
- Check **Use Lead and Lean** & set both **Lead** and **Lean** angles to **Zero**
- Click **Apply**



- Select **finish1** and then the **Tools tab**
- Select a **0.375" long reach Flat End Mill** as shown



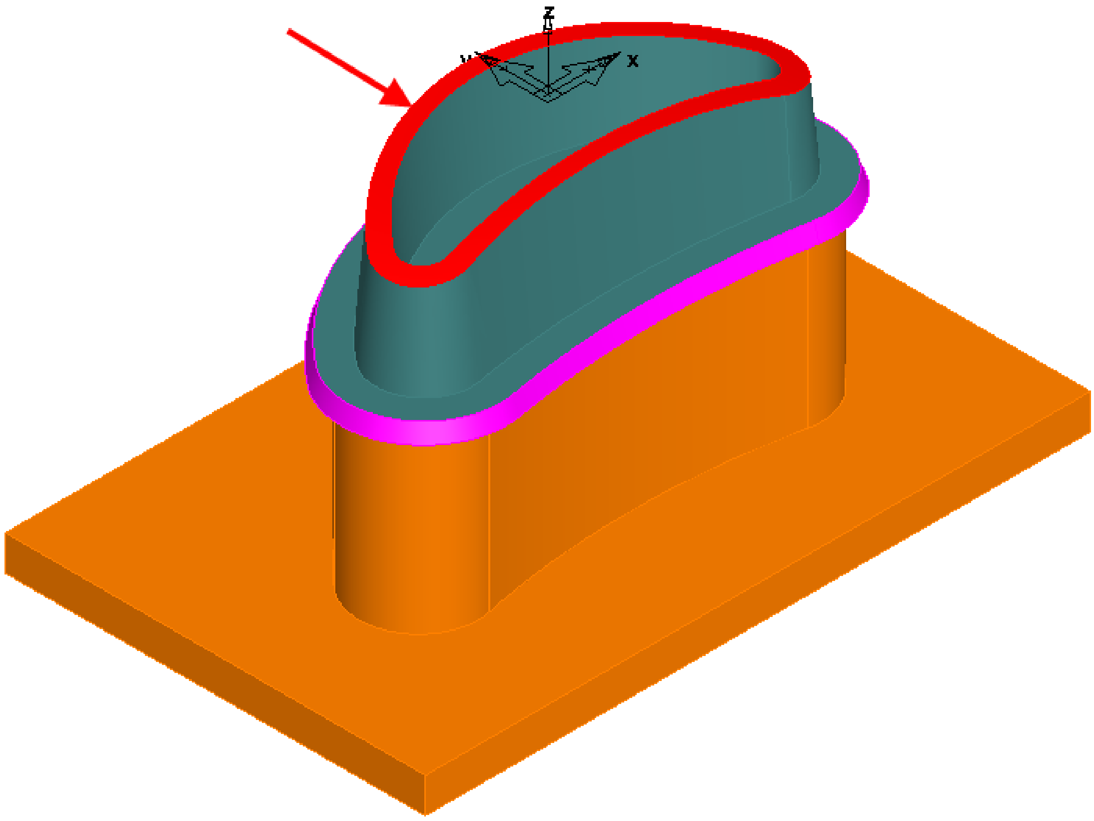
- Click **Apply** and then **OK**
- Run a **centerline simulation**



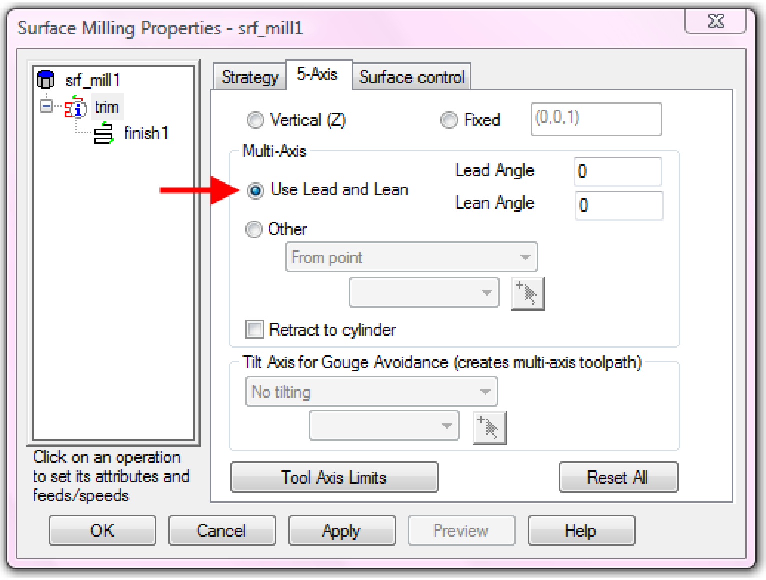
The tool makes two passes around the selected surface cleaning up the edge. As we have set lead and lean angles of zero, the tool tilts as it passes around the part maintaining its axis perpendicular to the surfaces being machine.

We will now repeat this process for the upper edge of the part.

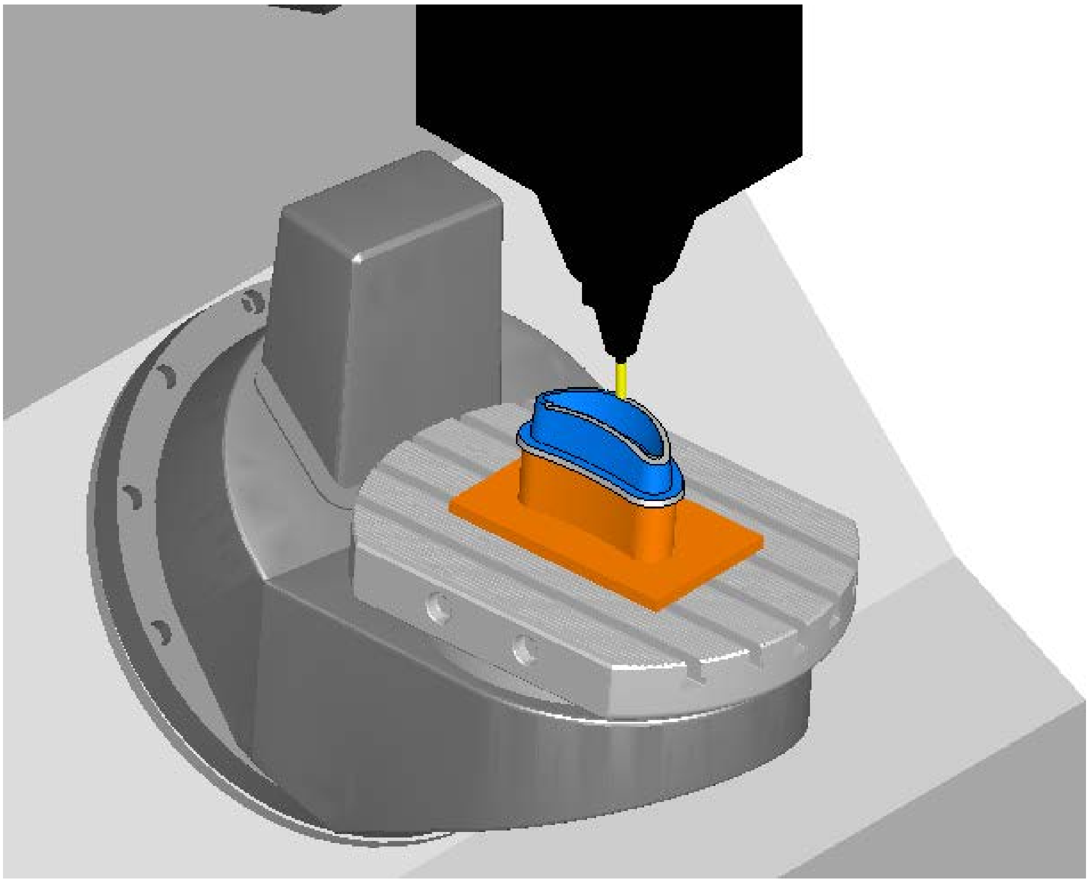
- Select the surface that makes up the upper edge of the part as shown



- Create a new **surface milling** feature
- Select **Choose a single operation**
- On the **New strategy** page select **5-Axis trim**
- Click **Finish**
- Select the 5-Axis tab
- Check **Use Lead and Lean** & set both **Lead** and **Lean** angles to **Zero**
- Click **Apply**

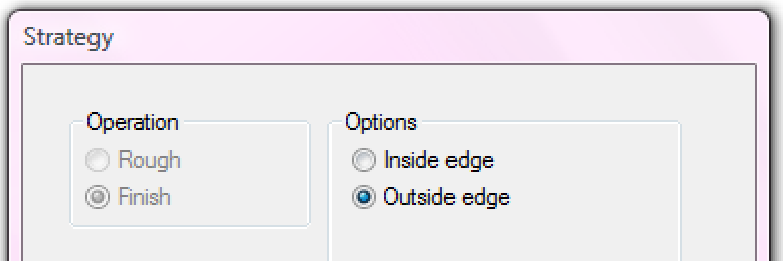


- Select **finish1** and then the **Tools tab**
- Select a **0.375" long reach Flat End Mill** as before
- Click **Apply** and then **OK**
- Select the **DMU Evo post** and ensure you have the **matching MD** file
- Run a **Machine simulation**

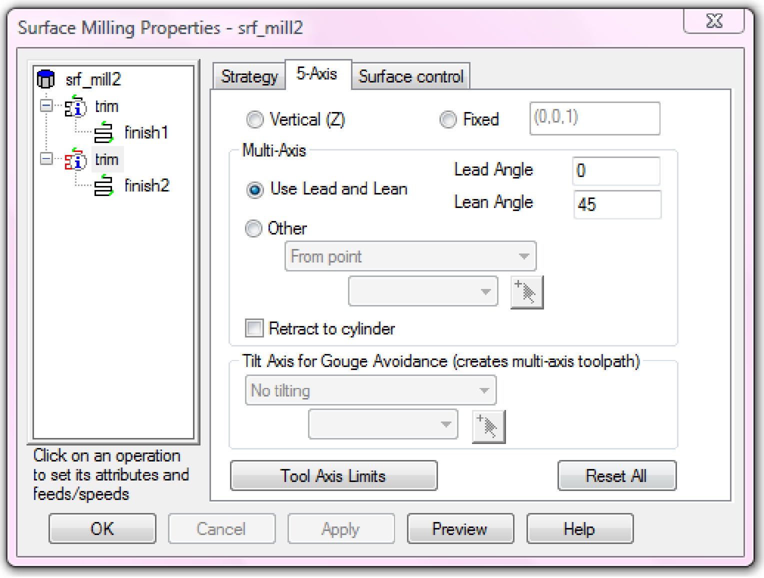


To finish off the part we will finally add an Outside edge trim operation to deburr the edges of the upper surface.

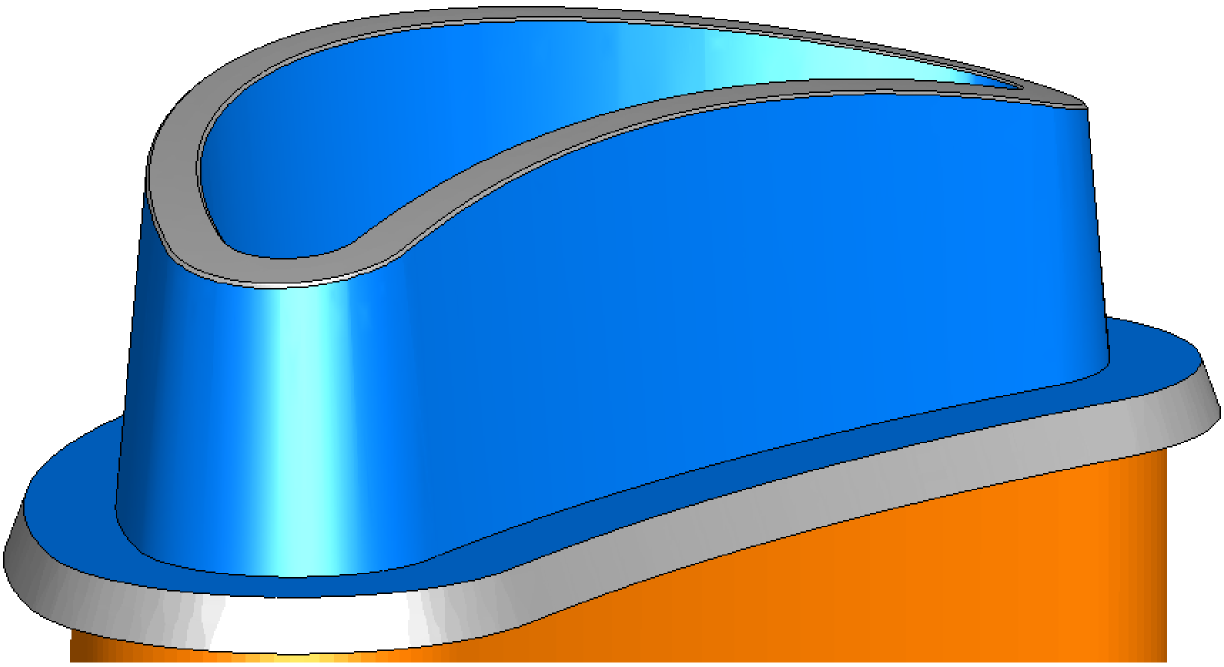
- **Eject** the simulation
- Open the **feature properties** of **srfmill_2**
- Select the **Process tab** and click on **Add new operation**
- Choose a **5-Axis trim** operation
- Click **Next** and then choose **Outside edge**
- Click **Finish**



- Click on the new **trim** operation and select the **5-Axis tab**
- Set the **Lead** to **Zero** and the **Lean** to **45 degrees**



- Click **Apply** and then **OK**
- Select the **finish2** operation and then the **Tools tab**
- Select a **0.25" long reach Ball Nose Tool**
- Select the **Milling tab** and set the **Leave allowance** to **-0.010"**
- Click **Apply** and then **OK**
- ***Zoom in*** on the upper edge of the part and run a **3D Simulation**



Note how the new 5-Axis trim toolpath has neatly deburred the sharp edges left by the previous operation.

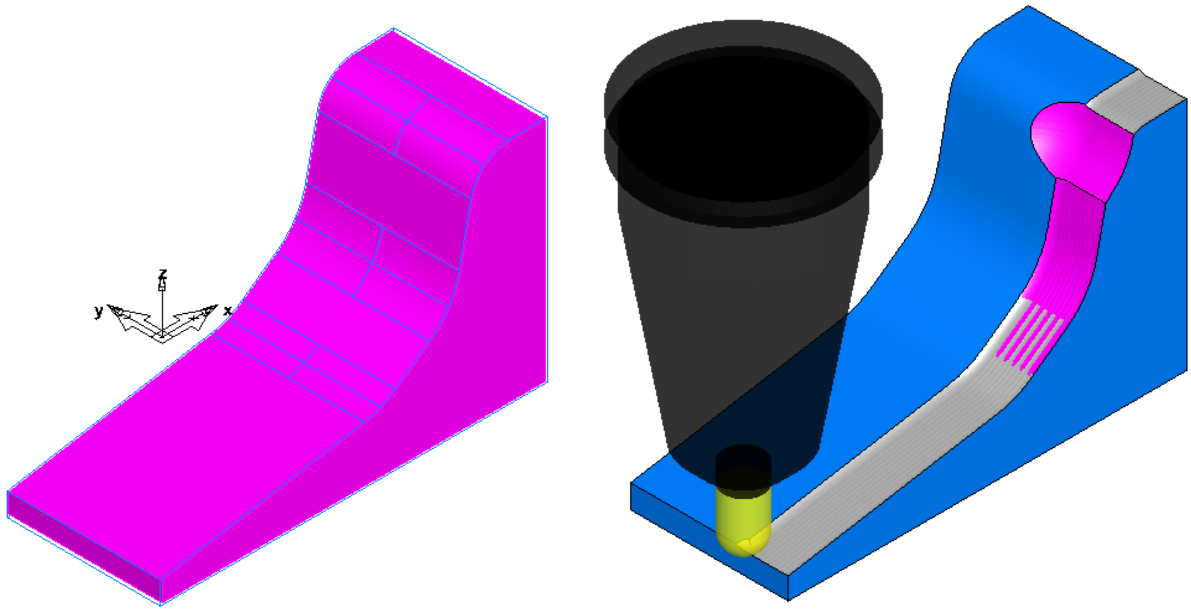
Collision Avoidance

Introduction

FeatureCAM is capable of detecting when the tool shank or holder will collide with a part. It is then able to take evasive action provided that the user tells it how it should tilt the tool to get out of trouble. Any 5-Axis tool alignment can be used as a collision avoidance strategy. In this module we will look at several ways in which collision avoidance can be applied.

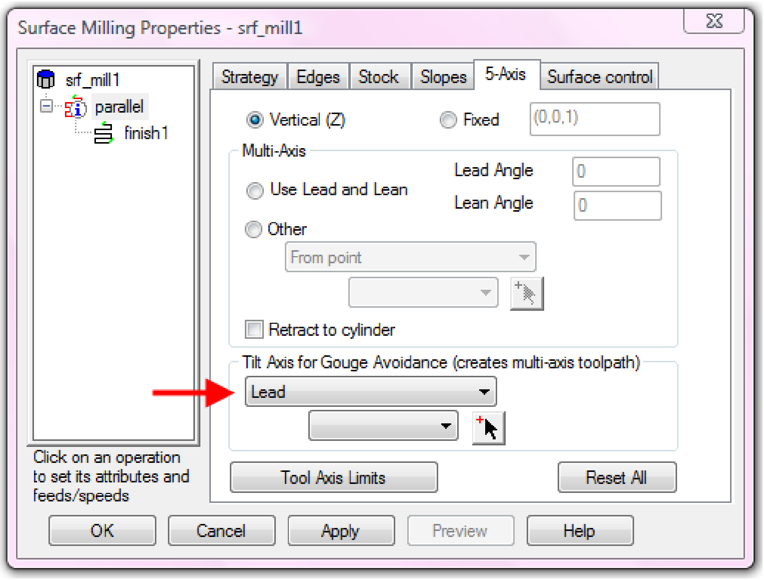
In this first example, a stepped part will be machined. The step is steep and would require a long reach tool to be machined safely with a 3-Axis toolpath. We will look at using a **Lead** and then a **Lean** avoidance tool axis to allow us to machine the part with a short reach cutter to finish the part.

- Open the file **Avoid_Lead_Lean.fm**
- Select the tool crib **Avoid_Lead_Lean.fm_tools_from_last_save**
- Run a **3D simulation**

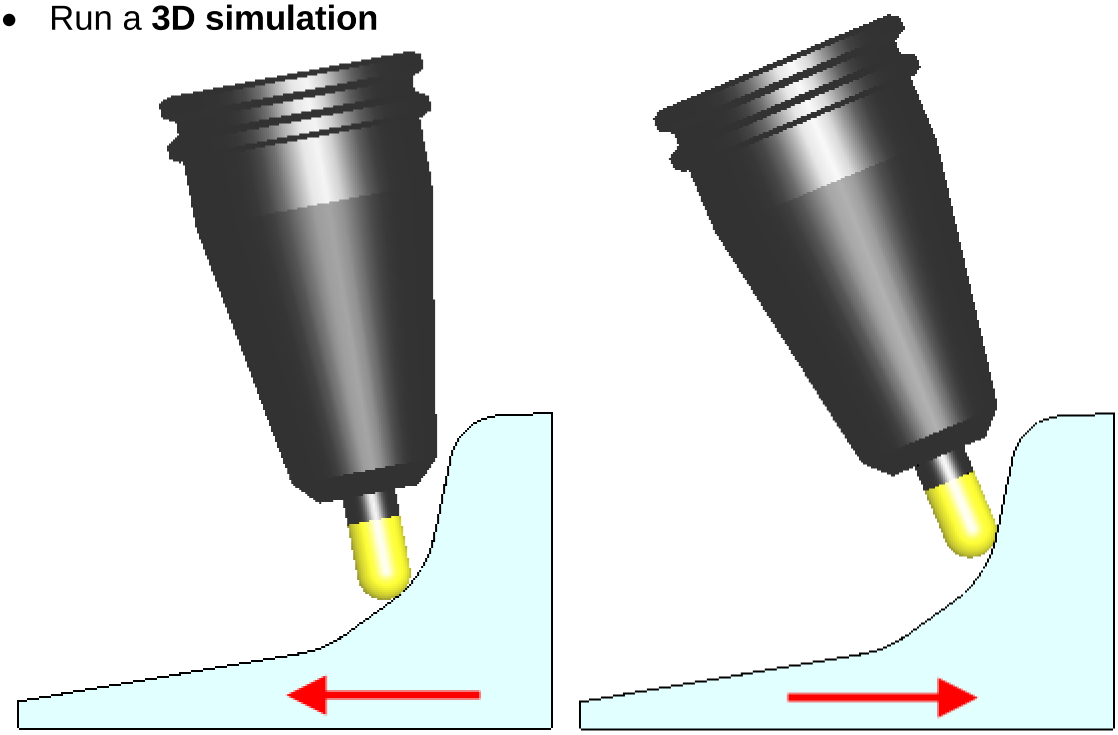


We are using a short cutter here to minimize chatter and deflection as this will give improved accuracy and surface finish. It is obvious however that the holder is colliding with the part. We will now fix the gouge by tilting the tool away from the part along its direction of travel, That is, we will use a Lead angle tool axis. We could apply a lead angle to the whole toolpath, however, we only wish to tilt the tool in the area where the collision would occur. In general, 5-Axis toolpaths take longer to calculate. By using a collision avoidance strategy we only tilt the tool where it is necessary thus reducing the overall length of time taken to calculate the toolpath.

- Open the properties of **srf_mill1**
- Click on the **Parallel** operation and then the **5-axis** tab



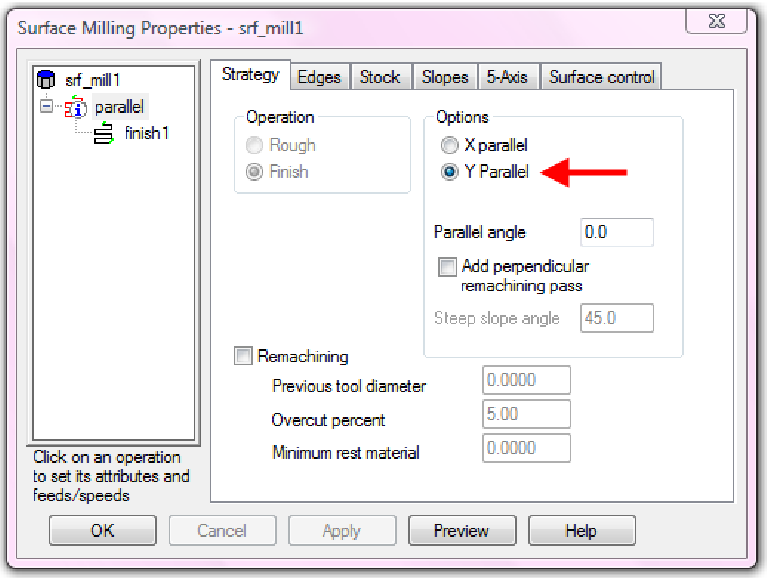
- Use the **Tilt Axis** pull down menu to select **Lead**
- Click **Apply** and then **OK**
- Run a **3D simulation**



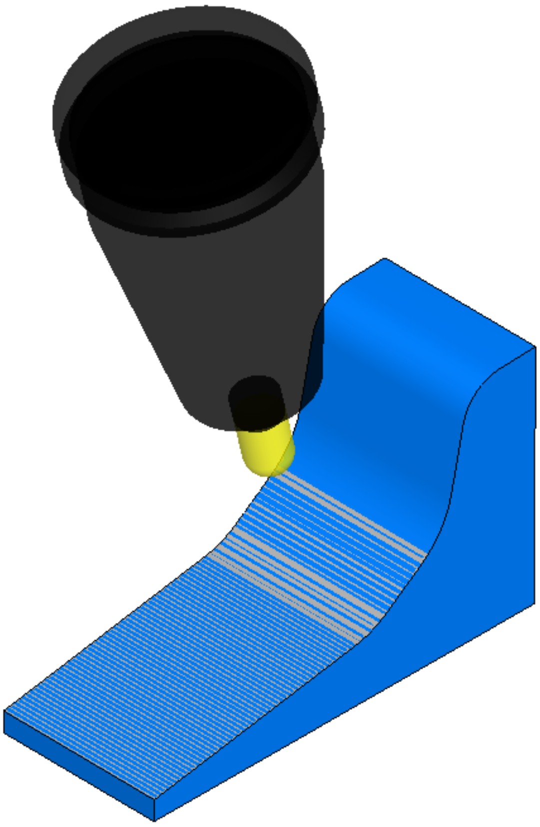
The tool tilts away from the part sufficiently to avoid a collision. It is important to note that the direction of the tilt is independent of the direction in which the tool is moving; this is unlike a normal Lead tool orientation. FeatureCAM is looking at the surfaces to be avoided and automatically deciding the Lead angle and whether it needs to be positive or negative.

We will now change to a Y Parallel toolpath and use a Lean angle to prevent gouging.

- Open the feature properties - click on **Parallel** and select check **Y Parallel**.



- Select the **5-Axis tab**
- Use the **Tilt Axis** pull down menu to select **Lean**
- Click **Apply** and then **OK**
- Select an **Isometric View**
- Run a **3D simulation**

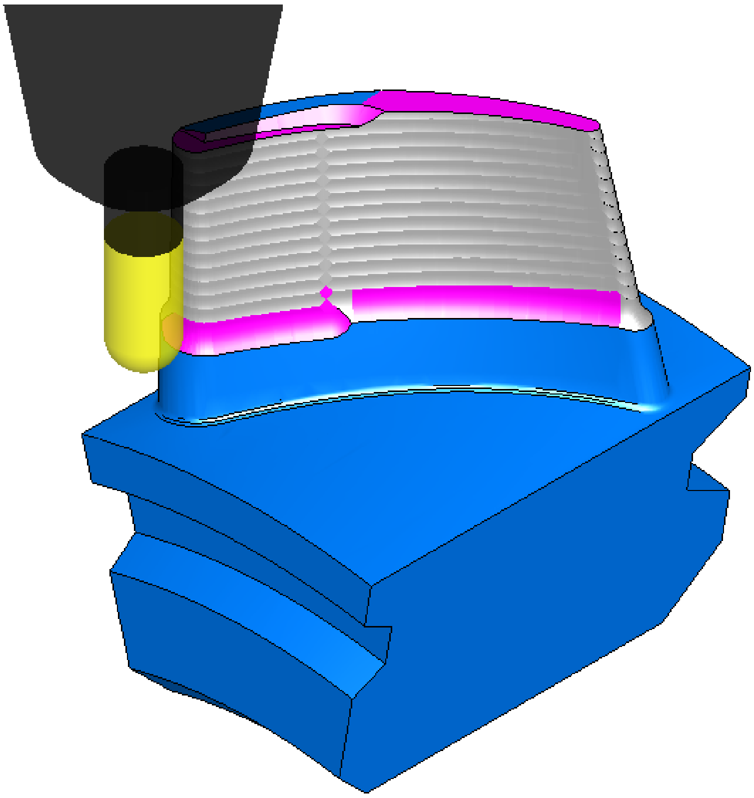


The tool is now cutting with a bi-directional parallel toolpath along the Y direction. The tool is leaning away from the part. As before, FeatureCAM is automatically deciding the amount of the lean angle and whether it needs to be positive or negative.

- As an exercise create a line from **(2,0,1)** to **(2,2,1)** and use it to provide a **From Line** tilt axis for collision avoidance

We will now use a Lean avoidance strategy with a Z-Level finish toolpath on a turbine blade model. In order to better visualize the motions of the machine, we will use a machine tool simulation.

- Open the file **Avoid.fm**
- Select the tool crib **Avoid.fm_tools_from_last_save**
- Run a **3D simulation**



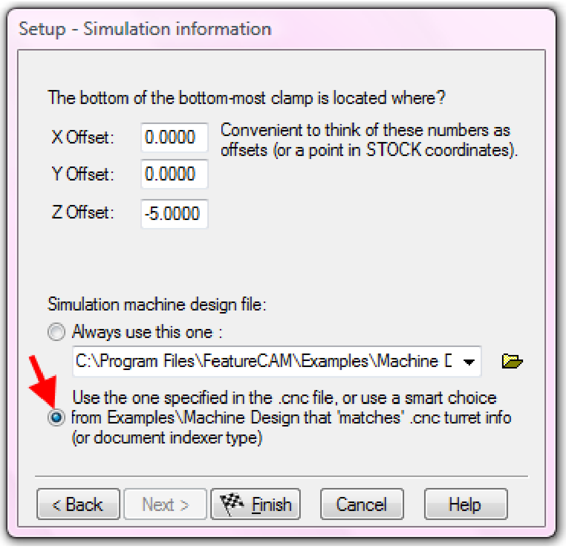
The first part of the toolpath is OK, but when the tool gets about three quarters of the way down the part, the holder collides with the top of the blade. We could fix this by changing the whole of the toolpath to use a Lean tool axis. The disadvantage of this approach is that the 5-Axis toolpath will take longer to calculate. By using collision avoidance we can calculate most of the toolpath quickly using a 3-Axis strategy and only switch to 5_Axis where it is needed to avoid hitting the part.

- **Eject** the simulation
- Open the feature properties, click on **Z Level**
- Select the **5-Axis tab**
- Use the **Tilt Axis** pull down menu to select **Toward Surface Normal**
- Click **Apply** and then **OK**

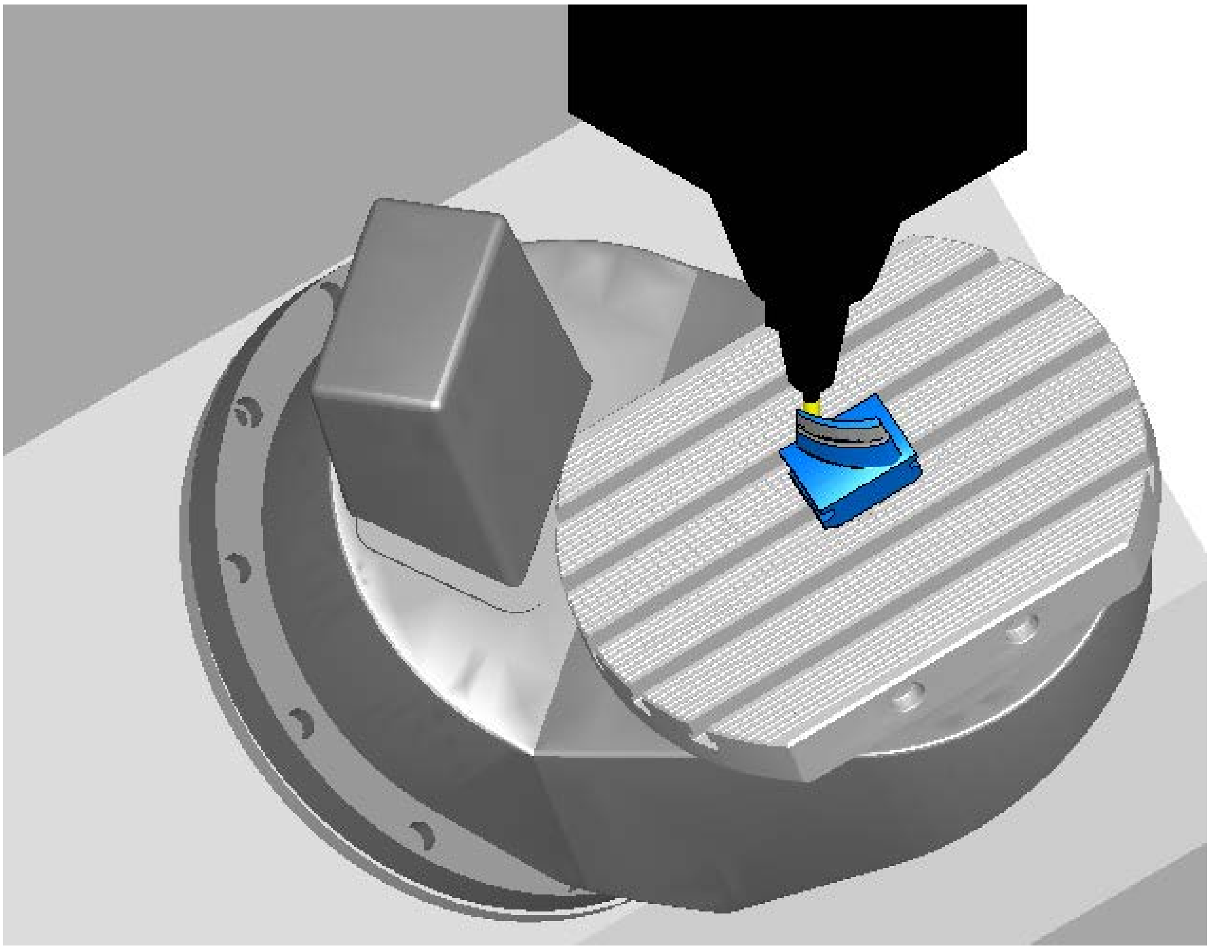
Lead and Lean are useful collision avoidance options when the obstacles to be avoided are all aligned in a single direction so that either a Lead or a Lean will allow FeatureCAM to prevent a gouge. When features on the part have many different orientations, it can be difficult to avoid a collision using a single tool axis option.

In addition to the normal tool axis alignments available through the "Other" pull down menu, FeatureCAM has an additional option for collision avoidance; this is Towards Surface Normal. When this option is selected, FeatureCAM tilts the tool away from surface which it is cutting. The direction of the tilt is towards the normal to the surface – that is, in a direction perpendicular to the surface at the point of contact. In most cases, this will tilt the tool in a safe direction away from the surfaces being machined.

- From the **Manufacturing** menu select **Post Process**
- Browse to select the **DMG Evo** post processor
- In the setup properties ensure that FeatureCAM is going to use the machine desiggn file that is specified in the post processor



- Select an **Isometric View**
- Run a **Machine Simulation**



The rotary axes of the machine remain stationary at the start of the toolpath. It is only when the tool shank or holder become too close to the part that the rotary axes are bought into play to avoid the collision.

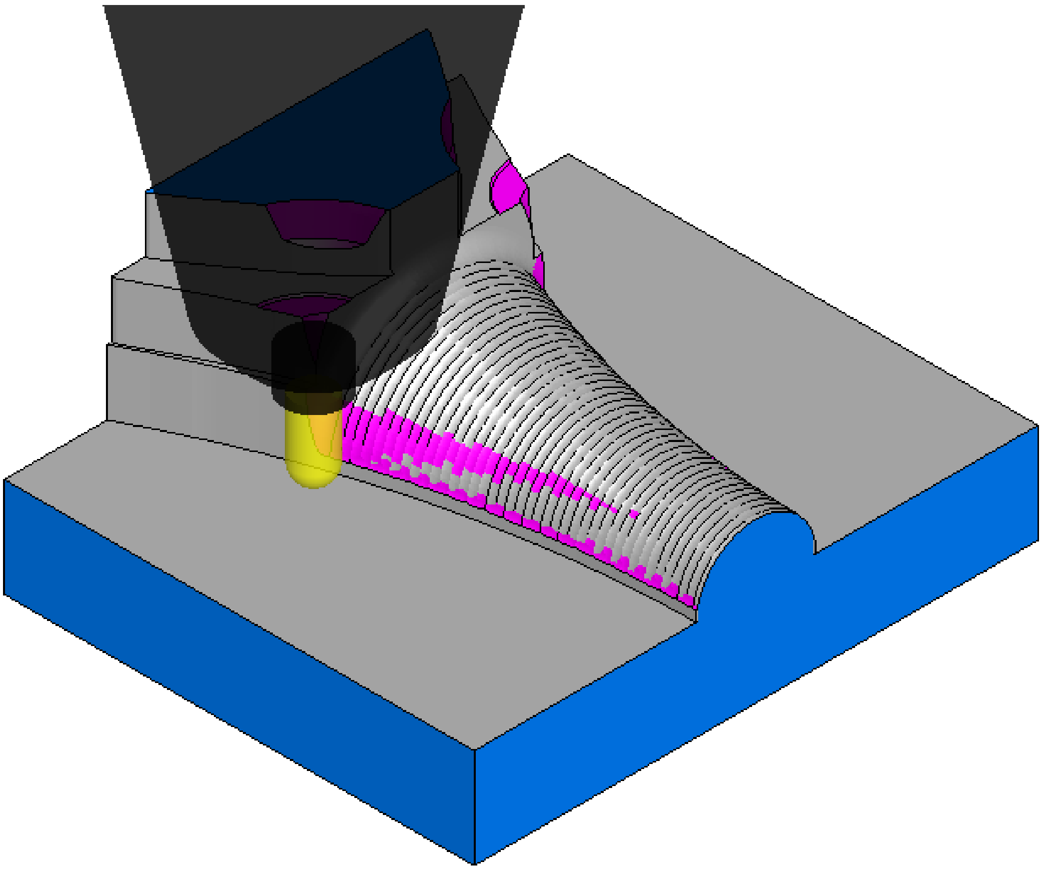
Tool Axis Limits

Introduction

In 5-Axis machining, it is often necessary to limit the motion of the tool relative to the part. This may be to avoid a collision between the tool holder and the part, work holder, or table, or to prevent the rotary axes from going over their limits.

In this example, we will be machining a curved surface using an Isoline strategy. In order to avoid collisions with the stock remaining after roughing, we shall use a Lead & Lean tool axis orientation. This will cause collisions with the part which we shall then resolve.

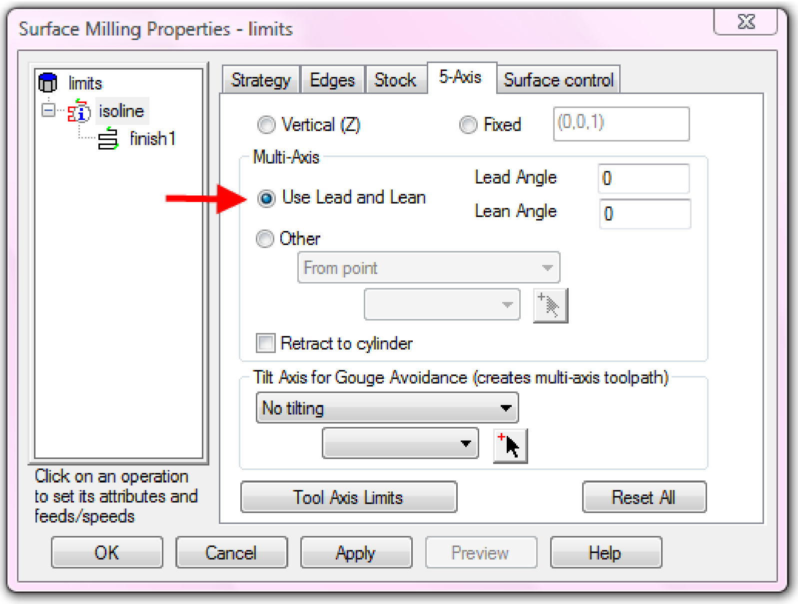
- Open the file **Limits.fm**
- Select the tool crib **Limits.fm_tools_from_last_save**
- Run a **3D simulation**



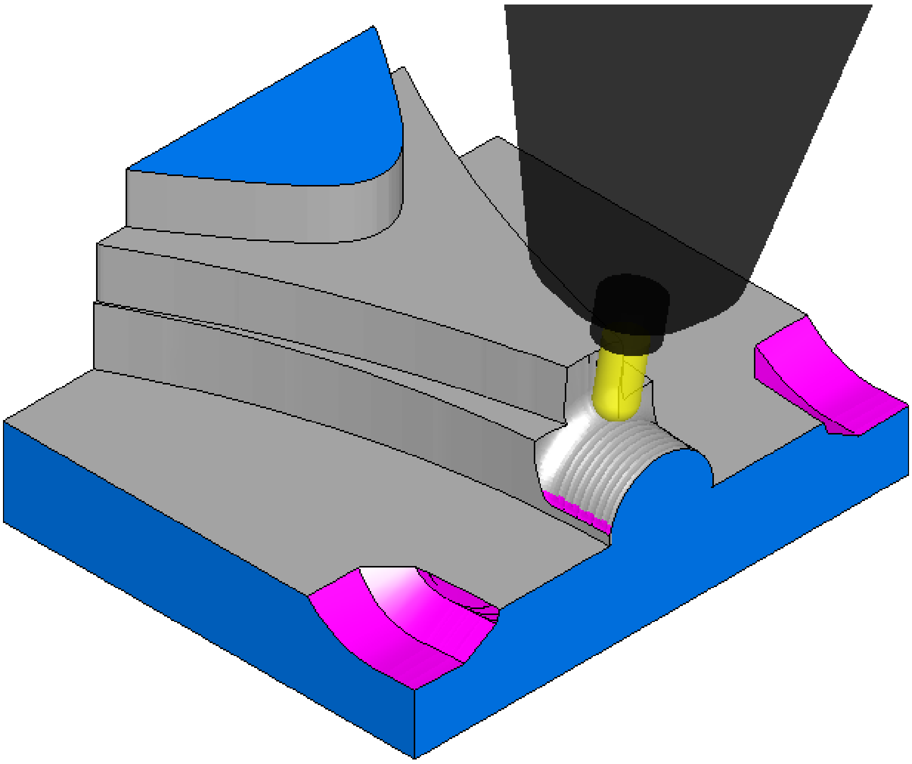
The tool holder is colliding with the steps left by the roughing operation. In order to prevent this, we could either use a longer tool or we could tilt the tool away from the part to avoid hitting the steps. In this case, we will try using a Lead & Lean tool axis with both angles set to zero, this will align the tool axis with the normal of the surface being machined.

- **Eject** the simulation

- Open the feature **Limits**, click on **Isoline** and select the **5-Axis tab**
- Select **Lead & Lean** and set the Lead and Lean angles to **Zero**

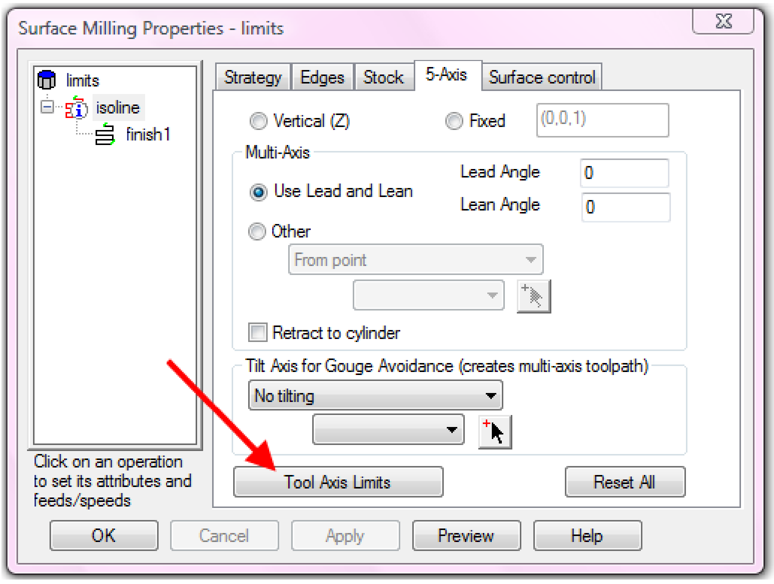


- Click **Apply** and then **OK**
- Run a **3D simulation**

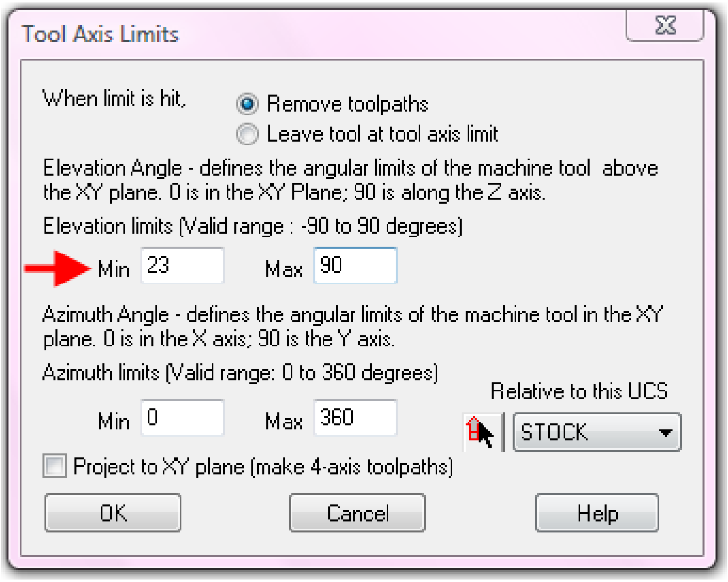


As you can see, the tool holder is no longer colliding with the steps left by the roughing. However, as the tool axis is leaning down almost to the horizontal the tool holder is now hitting the lower, flat areas of the part. We will now use Tool Axis Limits to fix this problem.

- **Eject** the simulation
- Open the feature **Limits**, click on **Isoline** and select the **5-Axis tab**
- Click on **Tool Axis Limits**



This opens up the Tool Axis Limits form. We will now use some of the options here to control the way that the tool is allowed to move in order to prevent collisions with the part.

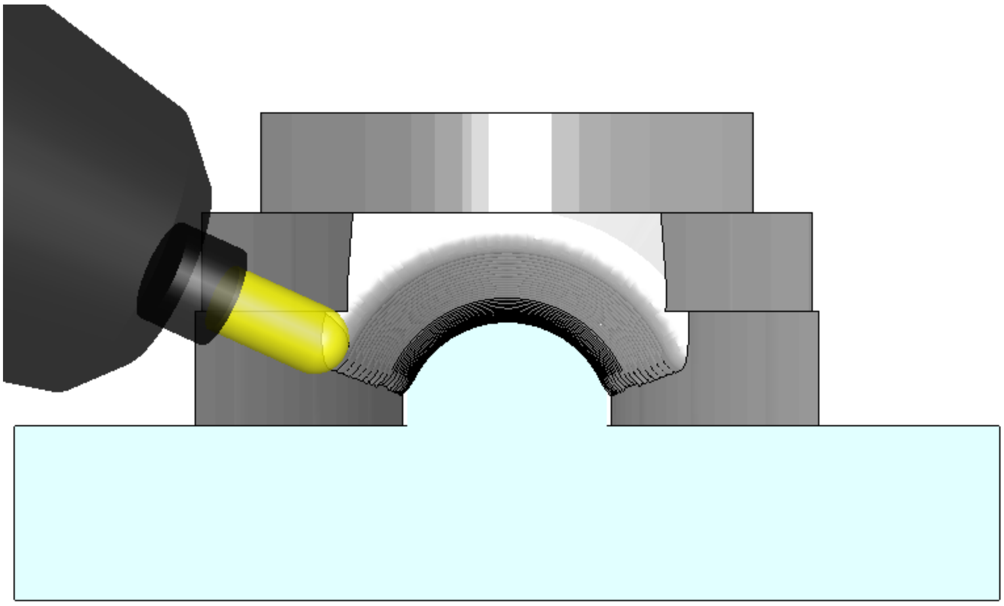


- Select **Remove toolpaths**
- Set the **Minimum elevation** to **23 degrees** and hit **OK**

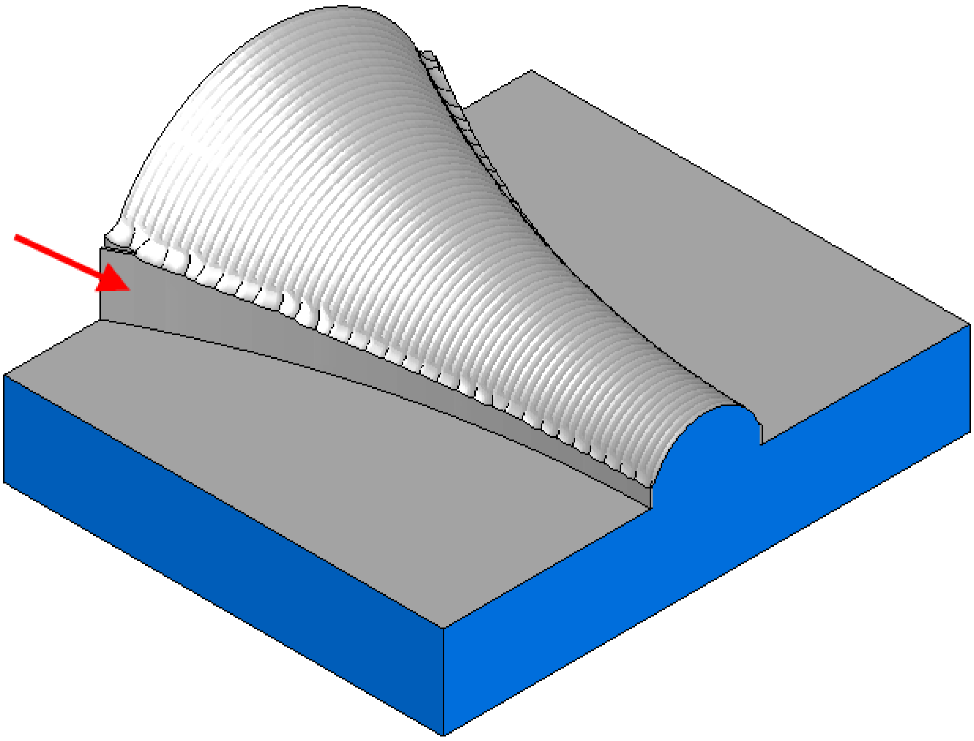
This is telling FeatureCAM that we want to allow the tool to go no lower than 23 degrees above the XY plane of the selected UCS (in this case the STOCK UCS). When the tool reaches this limit, any part of the toolpath below that angle will be removed.

- Select a **Front view**

- Run a **3D Simulation**; pause when you reach the Isoline toolpath and single step through the program from there

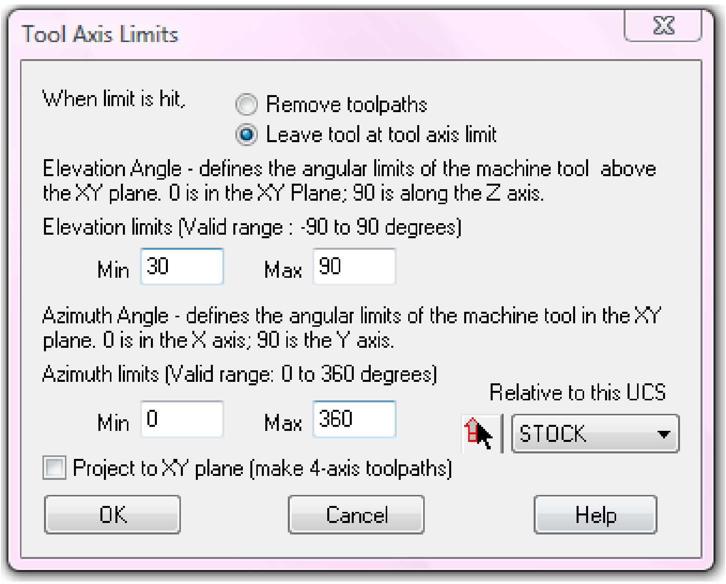


The tool is now only moving down the part until the normal of the surface being machine reaches an angle of 23 degrees to the horizontal. When it reaches this limit, it machines no lower.

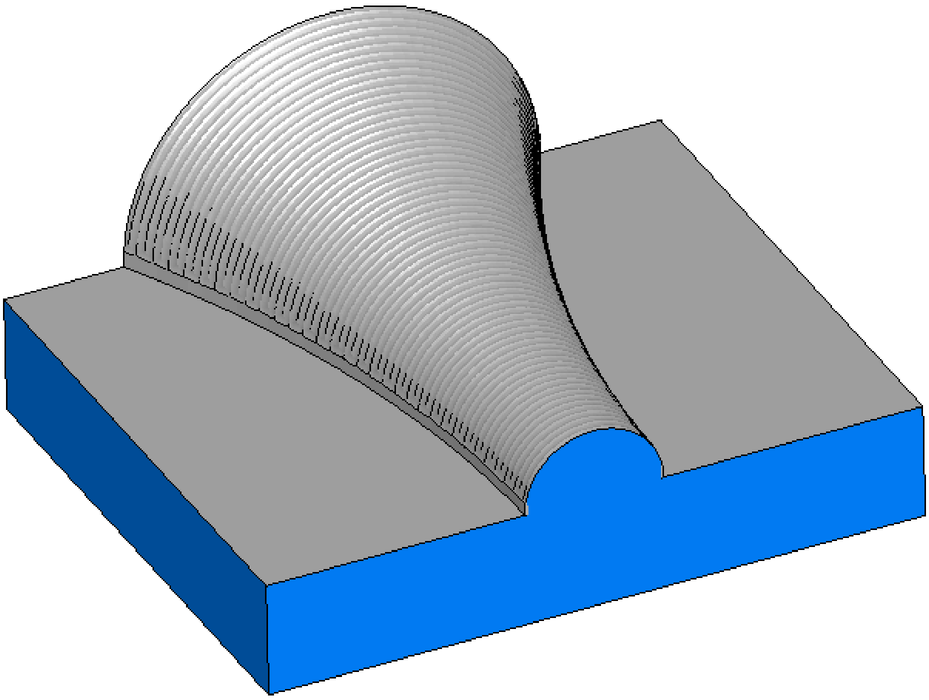


As the surface is tapered, the limit of the specified angle is moving up that part as it gets wider. This means that at the widest end of the part, the finish machining path is missing a large part of the surface to be machined. We can get around this by using another option on the Tool Axis Limits form: **Leave tool at axis limit**. When **Leave tool at axis limit** is selected, the tool will remain at the maximum angle but will continue to machine down to the end of the surface when the tool axis limit is reached.

- **Eject** the simulation
- Open the feature **Limits**, click on **Isoline** and select the **5-Axis tab**
- Click on **Tool Axis Limits**
- Select **Leave tool at axis limit**
- Set the **Minimum elevation** to **30 degrees** and then Click **OK**



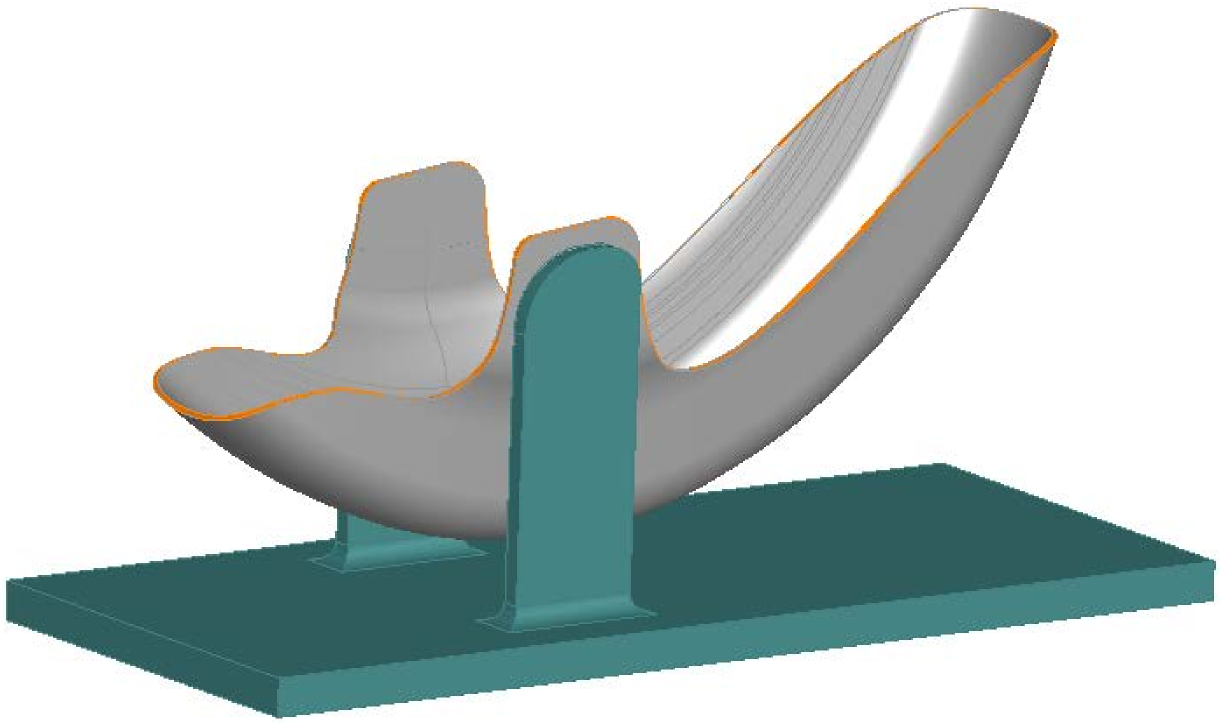
- Select a **Front view**
- Run a **3D Simulation**; pause when you reach the Isoline toolpath and single step through the program from there



By limiting the tool axis to a steeper angle and then maintaining the axis at that angle rather than removing the toolpath, we have now machined much more of the surface without colliding with the lower part of the model.

5-Axis Trim Exercise

- Open the file **Fender_Trim.fm**

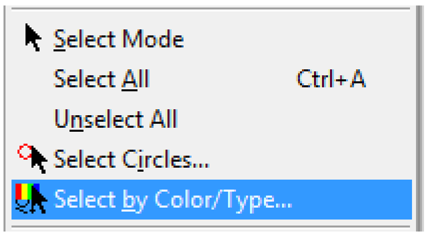


The model is mounted in a jig ready for the edges to be trimmed. We will trim the excess material from the part using a tool aligned perpendicular to the surfaces that make up the edge to be trimmed.

- **DO NOT select** any surfaces
- Create a new **surface milling** feature and click **Next**

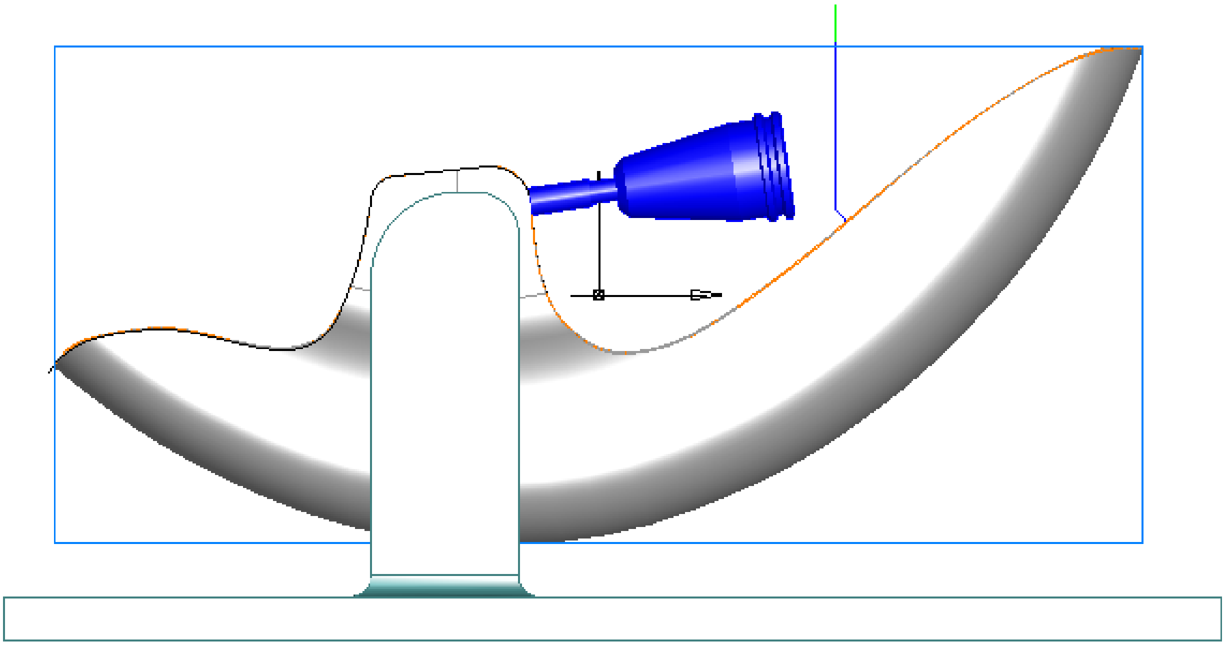
We need to select just the edge faces of the part, as there have a different color it is a simple matter to use a filter to select only the orange surfaces that we wish to machine.

- From the **main tool bar** select **Edit**
- Choose **Select by Color/Type**



- From the **pulldown menu** select the **Orange color** then click **OK**
- On the **New Feature – Part Surfaces** form click the **green cross** to add the selected surfaces into the feature
- Click **Next** and select **Choose a single operation** then **Next**
- Select **5-axis trim** and click **Finish**

- Select the **finish operation** and then the **Tools tab**
- Select a **0.25" Flat End Endmill**
- Click **Apply** and then **OK**
- Select a **Front view**
- **Single step** a **centerline simulation**



The toolpath is using a Lead and Lean angle of Zero. This would result in the head of the machine colliding with the part.

- Use **Tool axis limits** to modify the toolpath so it can trim the part safely

4-Axis Simultaneous

Introduction

FeatureCAM has three methods available for cutting 4-axis features:

Wrapped 2.5D Features	Requires FeatureMILL 2.5D
Wrapped Surface Milling Feature	Requires FeatureMILL 3D.
4-axis Simultaneous Features	Requires FeatureMILL 5-Axis Simultaneous.

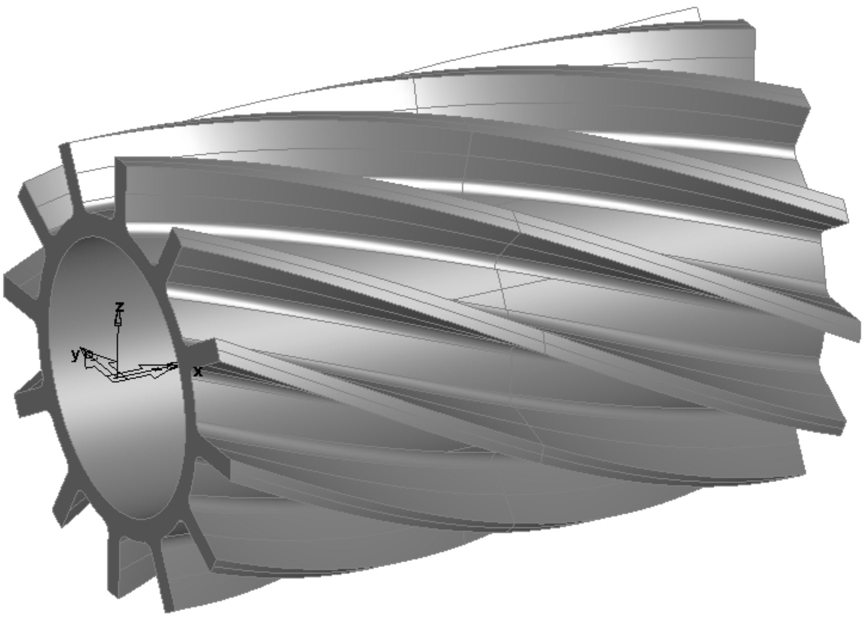
In this module, we will focus on 4-axis simultaneous features as 2.5D and 3D wrapped features are covered in separate modules.

FeatureCAM 4-Axis simultaneous is used where it is possible to cut a part by a 5-Axis strategy, but only a 4-axis machine is available. This gives the user access to a much wider range of strategies than would be possible using just 2.5D or 3D wrapped features. The method works by generating a full 5-Axis toolpath and then “locking out” one of the axes to give a 4-Axis toolpath. This may mean that the part is not cut fully, but this may be what is required as it is often very difficult to model a part exactly so that it can be machined with just four axes.

Impeller Example

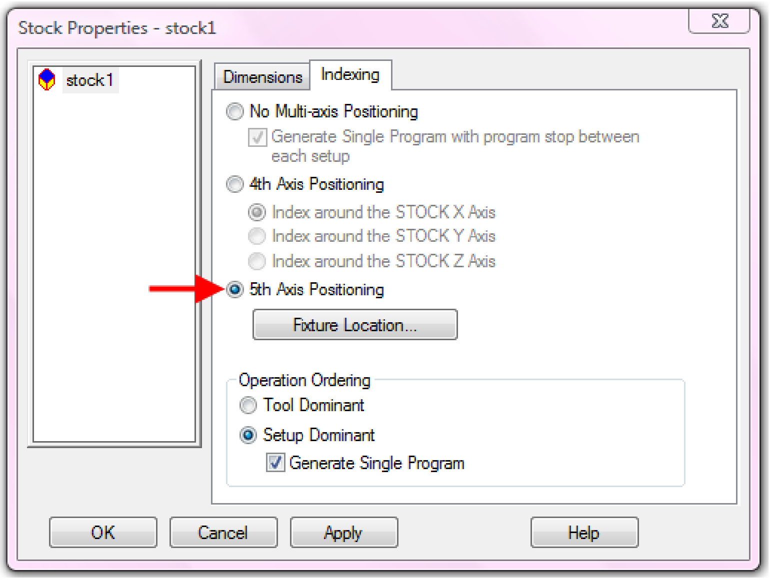
In this example, a part will be completed using the swarf and isoline 4-axis strategies. The part is roughed using swarf cutting at multiple levels, and then is finished using swarf and isoline.

- Open the part **Impeller_Start.fm**



At first glance it may appear that this part can be cut 4th axis, but the straight isolines on the surface are not perpendicular to the X-Axis. So, it is a 5-Axis part.

- Double click on the stock to edit its properties
- Select the **Indexing tab**
- Note that we have **5th Axis Positioning** selected



- Click **Cancel**
- Select the **Haas-5 axis.cnc** post processor from:

C:\Program Files\FeatureCAM\M-Library\5th Axis

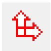
- Run a **Centerline simulation**
- Check the **NC code**

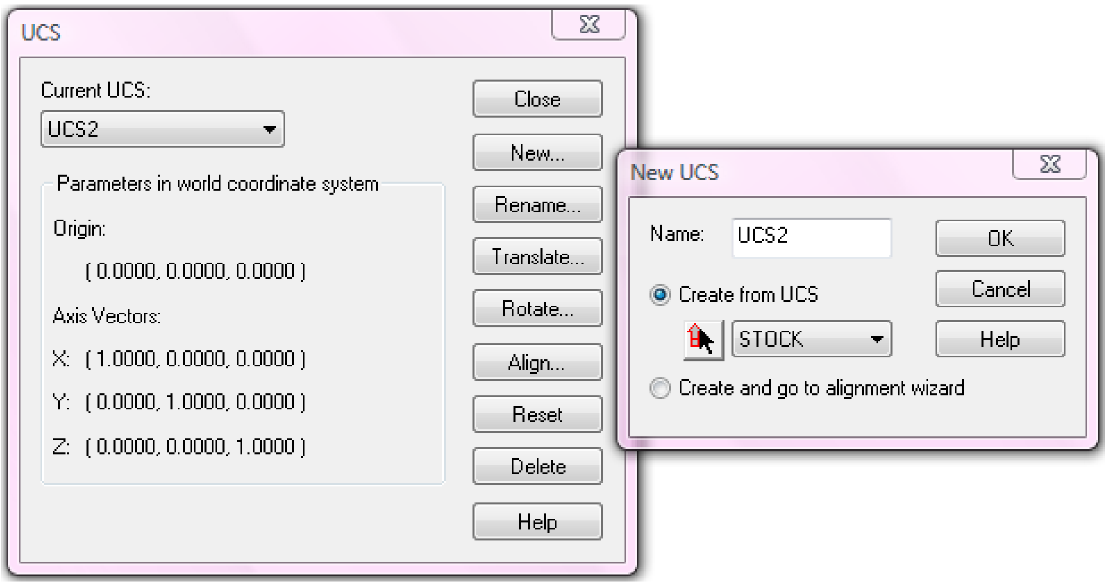
NC Code

```
N1850 Z7.2644
N1855 A-33.207 B3.185
N1860 X1.9656 Y2.5056
N1865 Z7.5959
N1870 A-25.135 B4.162
N1875 X1.407 Y1.8942
N1880 Z7.8353
N1885 A-17.076 B5.984
N1890 X0.8482 Y1.2499
N1895 Z7.9785
N1900 A-9.06 B10.903
N1905 X0.2847 Y0.6019
N1910 Z8.0234
N1915 A-1.817 B63.483
N1920 X-0.2174 Y0.313
N1925 G93
N1930 G01 Z6.809 F18.5263
N1935 X-0.2134 Y0.1756 Z6.8058 F327.2727
N1940 X-0.0719 Y0.0666 Z6.8056 A-2.748 B27.898 F23.075
N1945 X0.0219 Y0.0975 Z6.803 A-3.475 B18.492 F84.01
N1950 X0.1787 Y0.2553 Z6.7919 A-5.518 B11.782 F43.3466
N1955 X0.3187 Y0.421 Z6.7748 A-7.503 B8.772 F57.7134
```

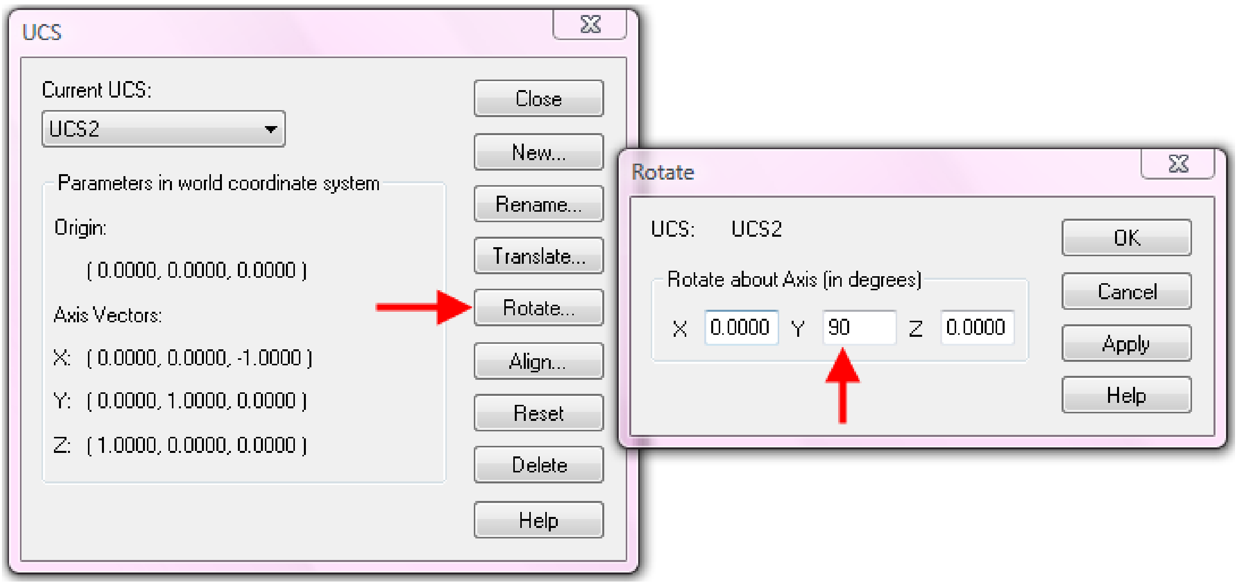
We have a mixture of A and B rotary moves. We want to restrict this to A only. FeatureCAM can do this by locking out the B axis.

In order to lock an axis, we have to restrict the tool moves to the XY plane of a UCS. In this case as we want to lock out the B axis, we need to make a new UCS with its Z axis aligned with the machine X axis.

- Eject the simulation
- On the main menu click on **UCS** 
- Check **Create from UCS**

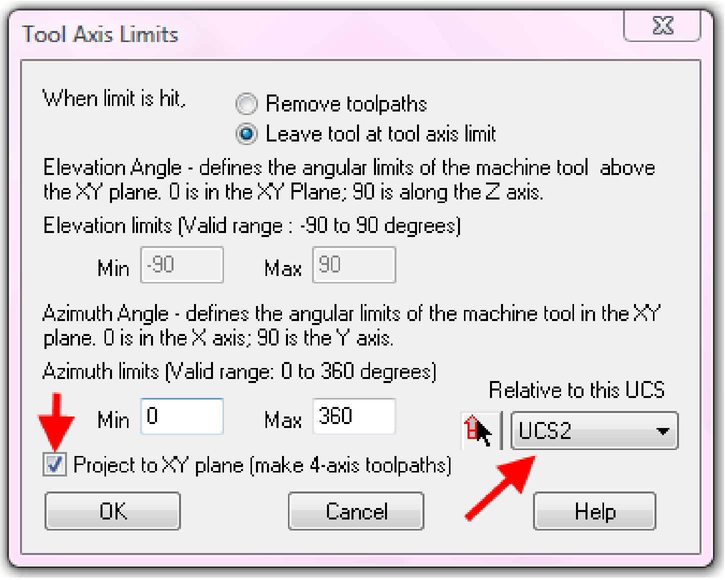


- Click **OK**
- Click Rotate and enter a Y angle of 90 degrees

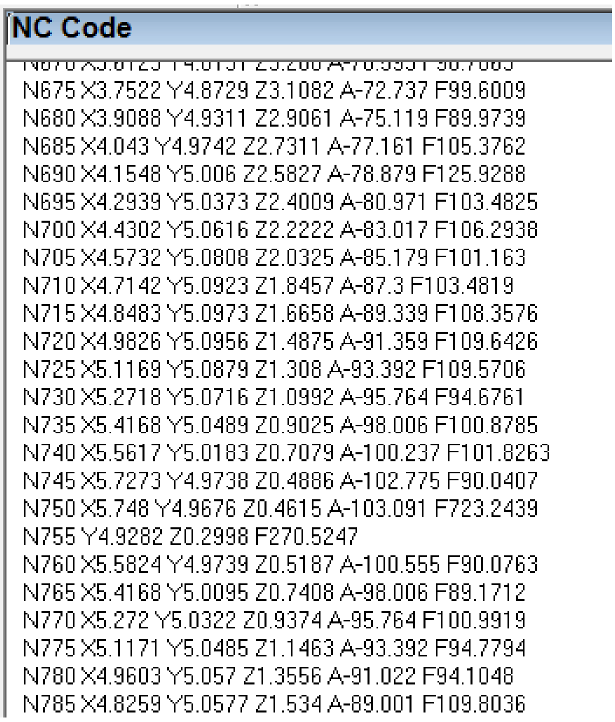


This has aligned the Z axis of the new UCS with the X axis of the machine. We now need to change the 5-Axis toolpaths to 4-Axis with the B axis locked out.

- Select the first **srf_mill1** feature in the **Parts view** and access the Properties
- Click on **Swarf**
- Select the **5-Axis tab**
- Click on **Tool Axis limits**
- Check **Project to XY Plane (make 4-axis toolpaths)**
- Select **UCS2** from the pulldown and then click **OK** then **OK** again



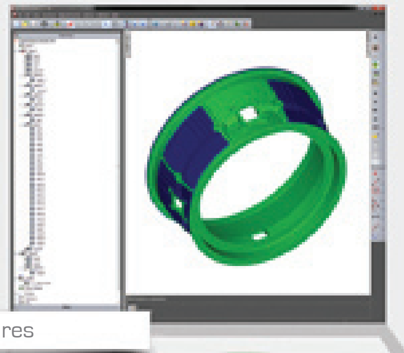
- Repeat the above procedure for the other features in the part
- Run a **Centerline simulation**
- Check the **NC code**



We now have a true 4-axis toolpath.

FeatureCAM 2012

FeatureCAM 2012



new features

Delcam TV

www.delcam.tv

FeatureCAM Learning Zone

www.delcam.tv/lz

FeatureCAM Website

www.featurecam.com

THE QUEEN'S AWARDS FOR ENTERPRISE: INNOVATION 2010

THE QUEEN'S AWARDS FOR ENTERPRISE: INTERNATIONAL TRADE 2011

Powering your productivity

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

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