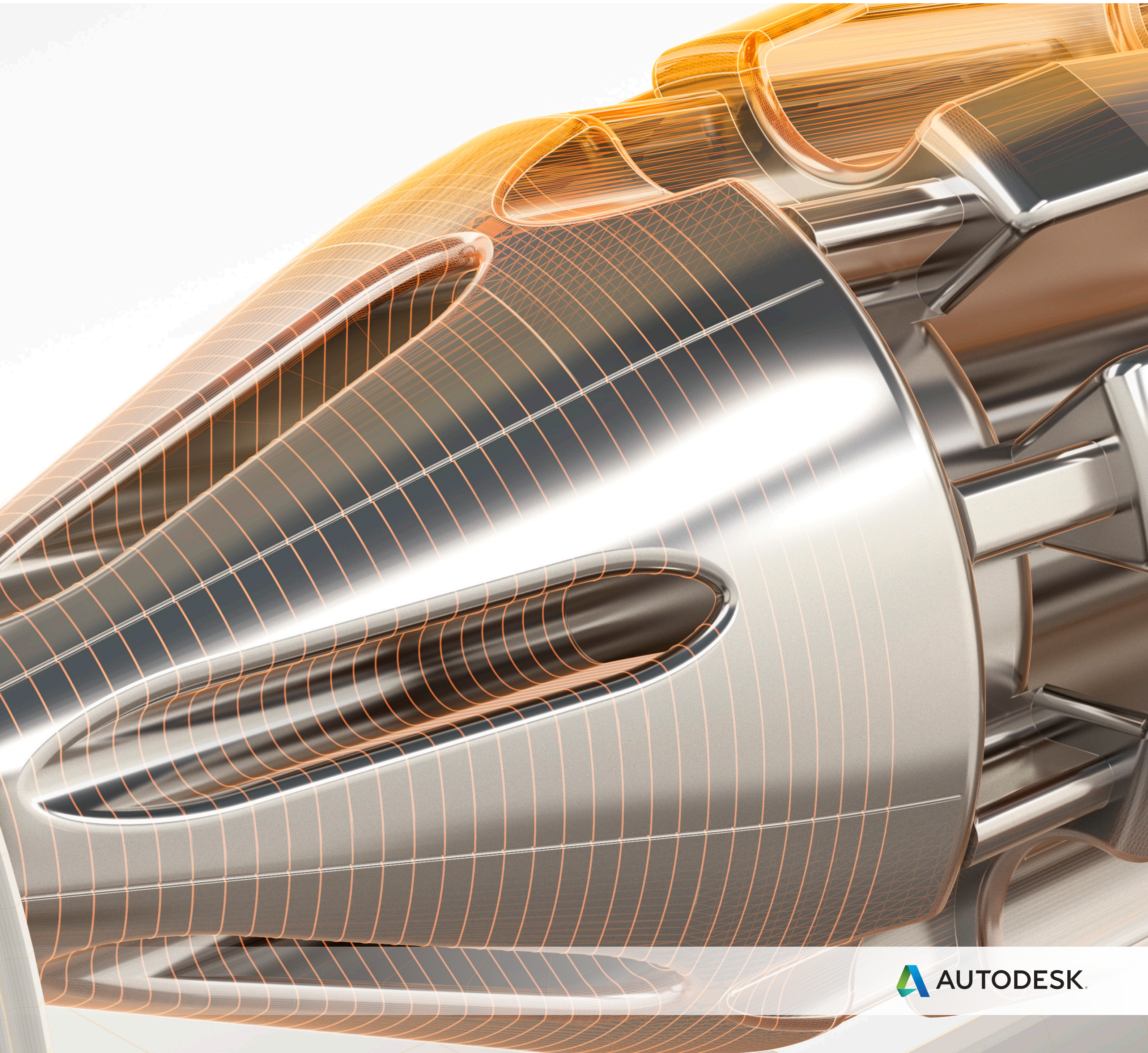
AUTODESK® FEATURECAM®

Automated to make parts faster

Training Course



AUTODESK.

Contents

FeatureCAM 3D Machining Feature Comparison

AUTODESK® FEATURECAM®

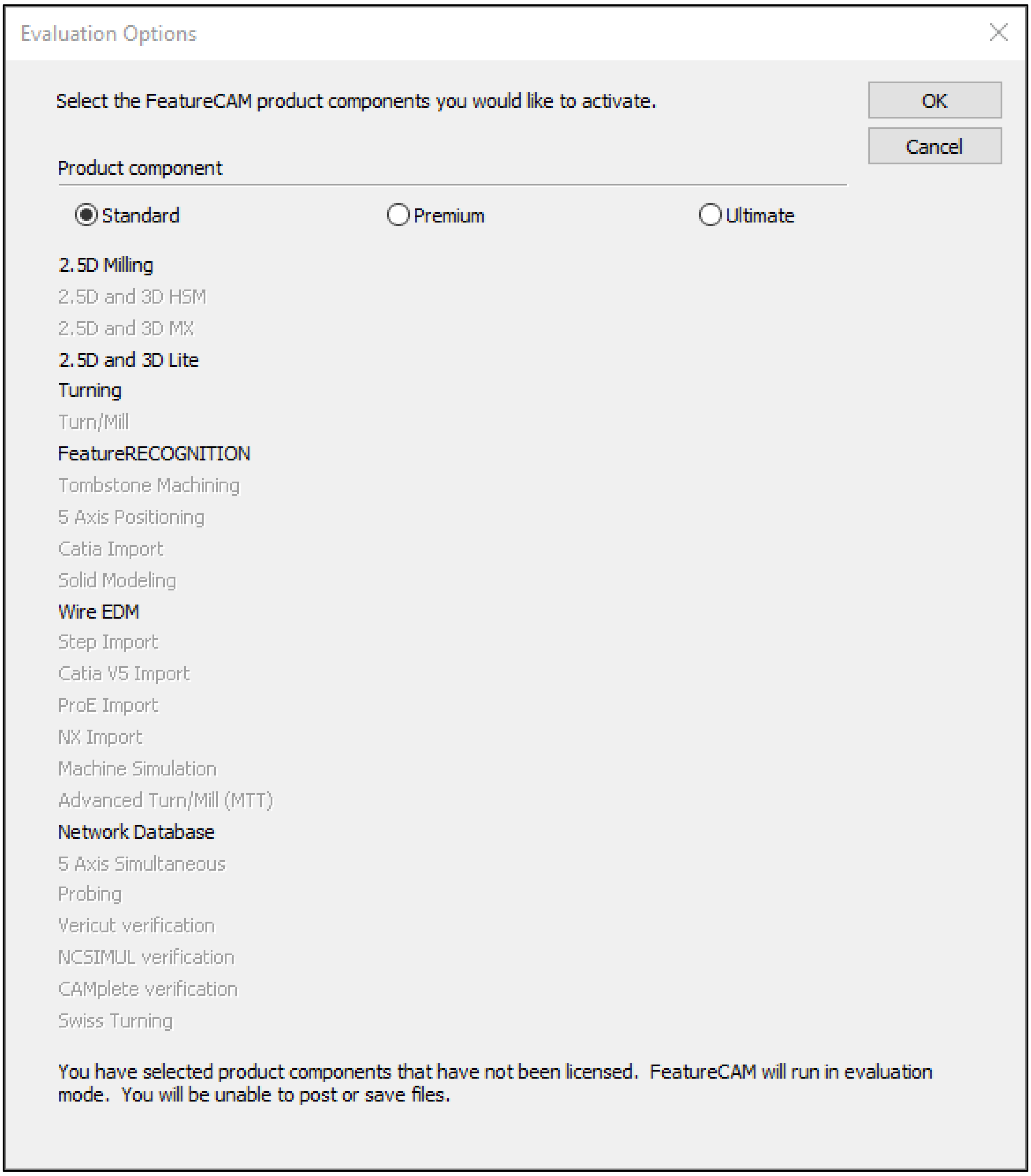
Feature Comparison

Feature Description	FeatureCAM Standard	FeatureCAM Premium	FeatureCAM Ultimate
Automated Programming			
Feature recognition	✓	✓	✓
Built-in intelligence	✓	✓	✓
Programming control	✓	✓	✓
Solid modeling		✓	✓
Simulation and Safety			
Collision avoidance	✓	✓	✓
Clamps & fixtures	✓	✓	✓
Stock models	✓	✓	✓
Machine simulation		✓	✓
Machine limits		✓	✓
Vericut, NCSIMUL & CAMplete			✓
CNC Milling Applications			
2.5-axis milling & 3-axis lite	✓	✓	✓
2-axis turning	✓	✓	✓
2-axis & 4-axis wire	✓	✓	✓
3-axis milling & 3+2 positioning		✓	✓
Tombstone machining		✓	✓
Probing			✓
Turn-mill		✓	✓
Advanced turn-mill			✓
Swiss-type lathes			✓
5-axis milling			✓
PartMaker CAM for Swiss Lathes			
Swiss machining			✓
Cycle time efficiency			✓
CNC compatibility			✓
NC code			✓

FeatureCAM *has now been split into* ***three levels****.* ***Standard, Premium*** *and* ***Ultimate****. The next few images show the available options for all modules.*

Standard

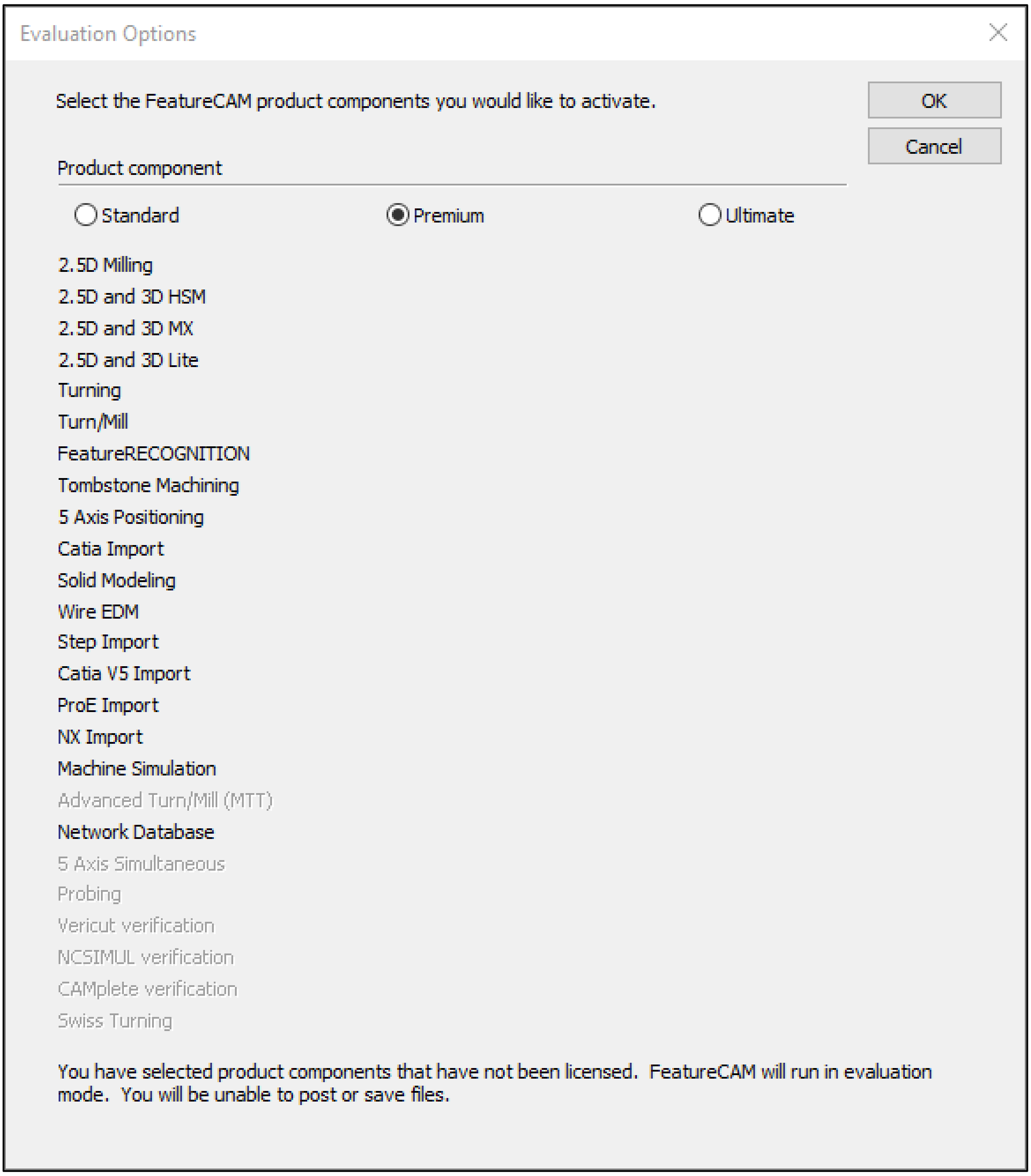
*The image below show the available options in **Standard FeatureCAM***



It is here that you will see the activated products that you have purchased.

Premium

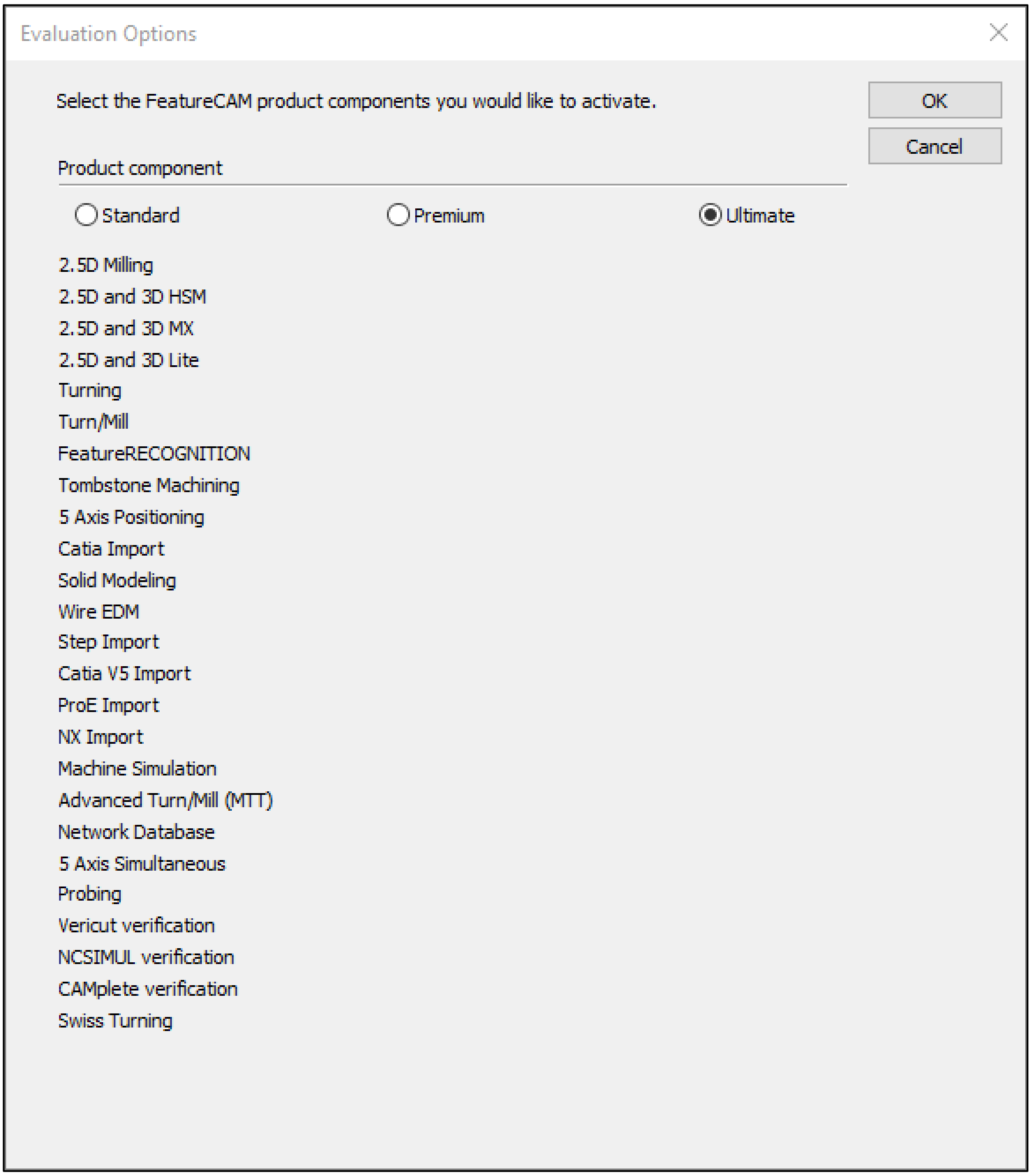
*The image below show the available options in **Premium FeatureCAM**.*



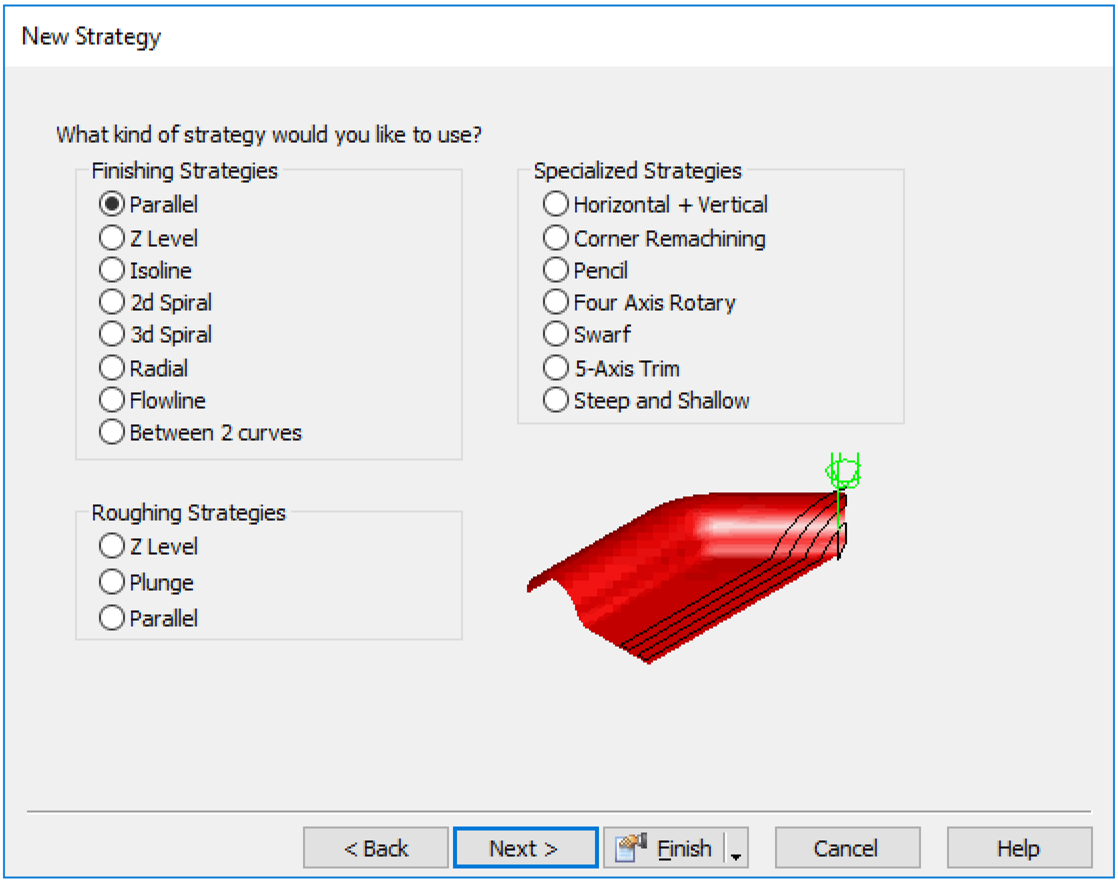
It is here that you will see the activated products that you have purchased.

Ultimate

*The image below show the available options in **Ultimate FeatureCAM**.*



It is here that you will see the activated products that you have purchased.



All 3D Strategies available for Ultimate machining

Contents

3D Premium and Ultimate Machining

Introduction

The following training notes will cover 3D Machining strategies. We will start with a model that will give you the ability to try out different Roughing and Finishing strategies. This will give you an understanding of the capabilities within each strategy. By the end of the course you will be able to machine complex 3D Components.

Finishing Basics

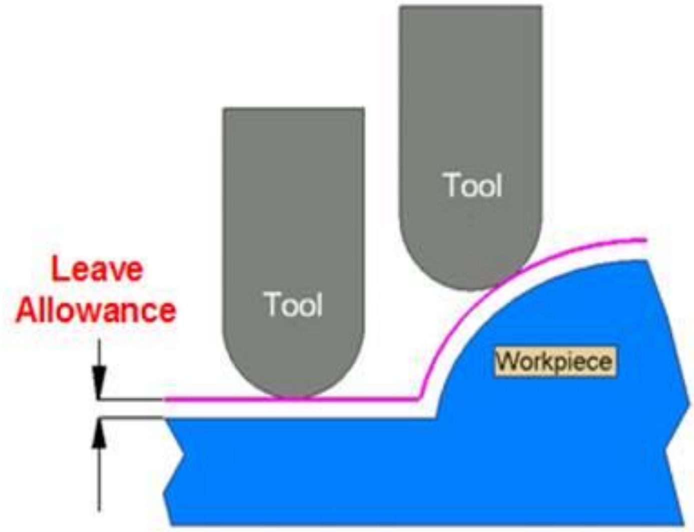
FeatureCAM has wide variety of methods for finish machining parts. The strategy that is chosen will depend upon the part geometry, the type of stock and the available cutting tools. By choosing the most appropriate combination of strategy and tooling the user can greatly reduce the time taken to finish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

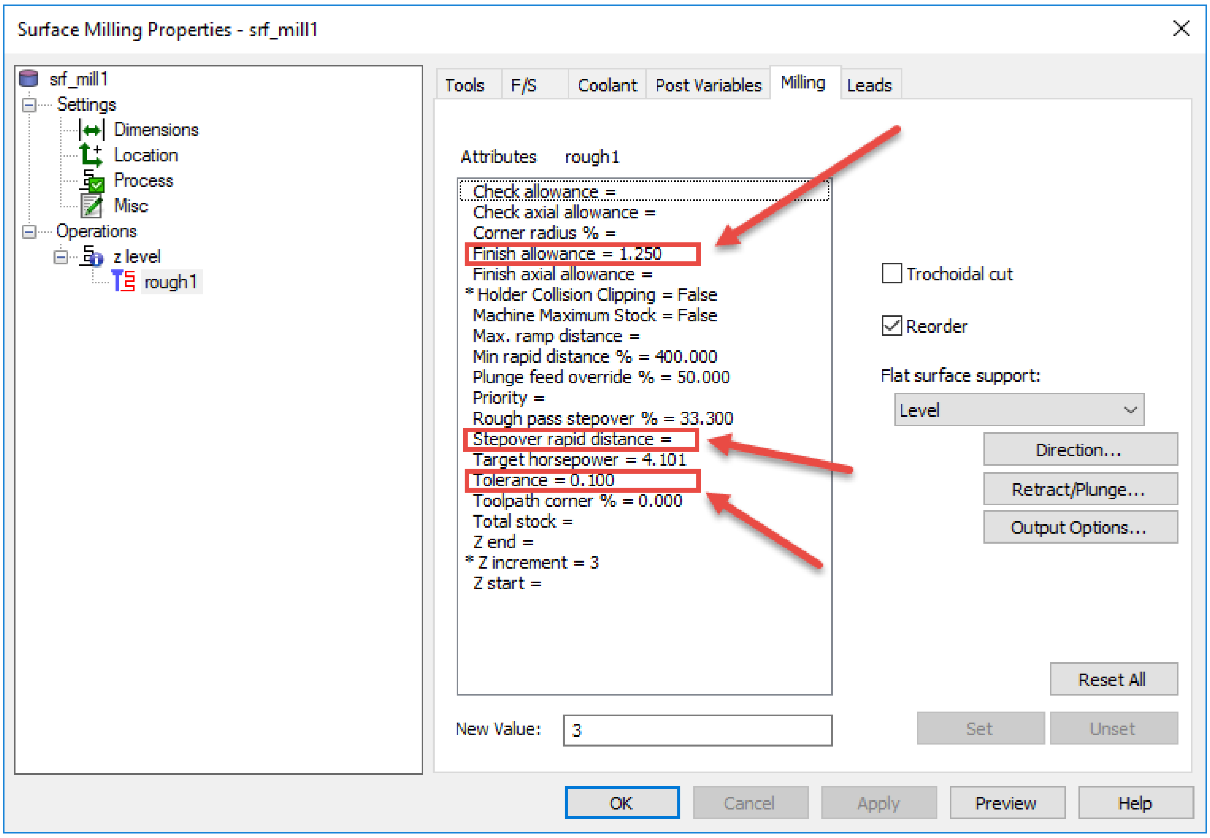
Leave Allowance, Tolerance and Stepover

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, 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. The leave allowance can also be negative this will cut the part undersize.

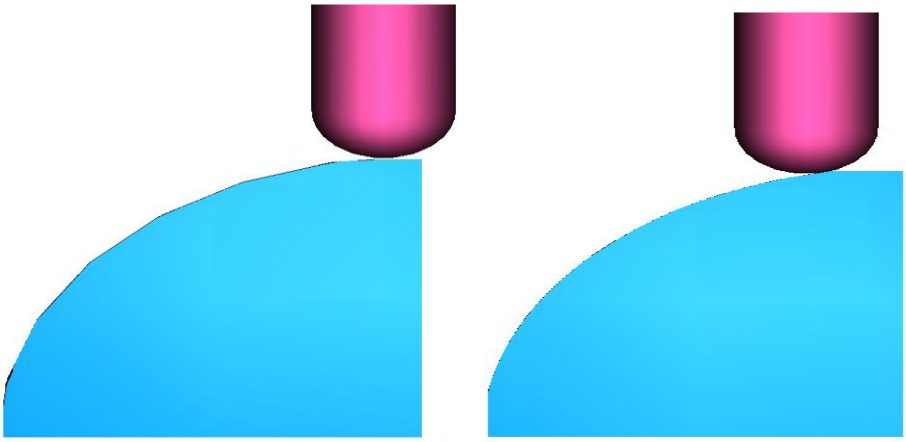




Tolerance

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 0.125mm means that the tool will not deviate from the surface by more than 0.125mm along the direction in which it is cutting. On a convex surface this will be the maximum amount of the permissible cut into the surface, on a concave surface it will be the maximum amount of material that will be left on the surfaces being cut.

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



Large Tolerance **Small Tolerance**

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

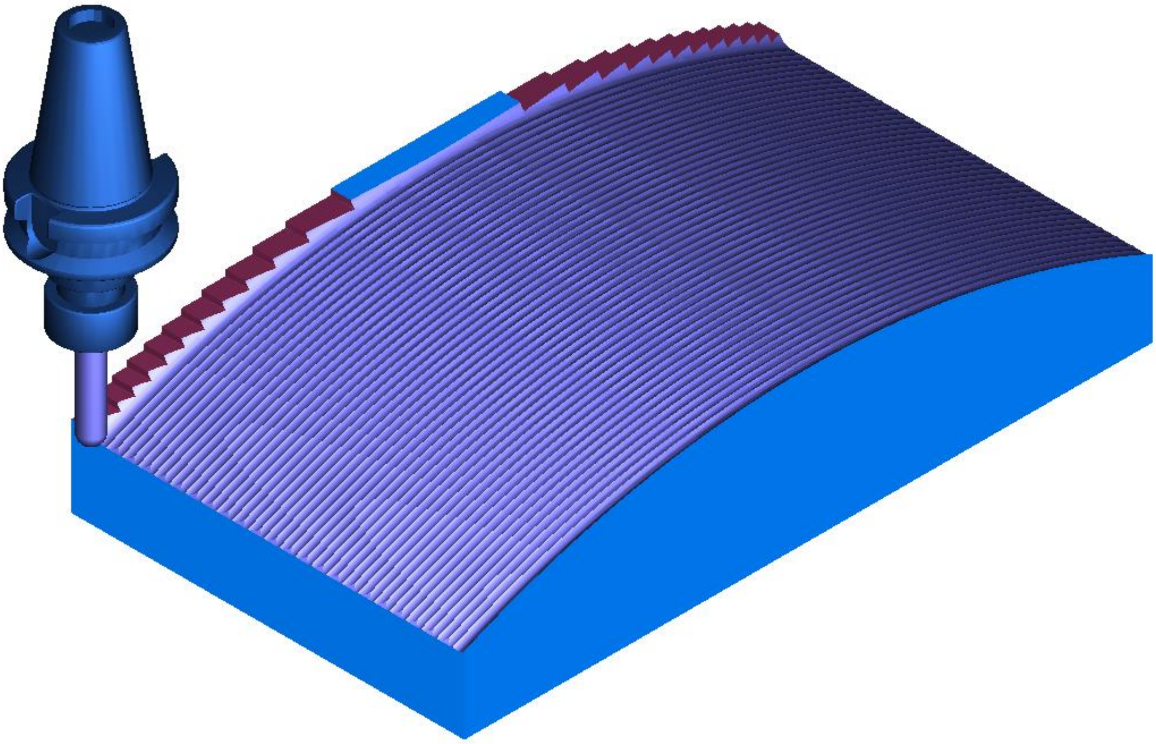
If a very fine tolerance is chosen for a roughing operation, it may take a long time to calculate the toolpath to an unnecessary accuracy. The optimum Tolerance value will depend upon the user balancing the required accuracy against the time taken to calculate the toolpath.

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.

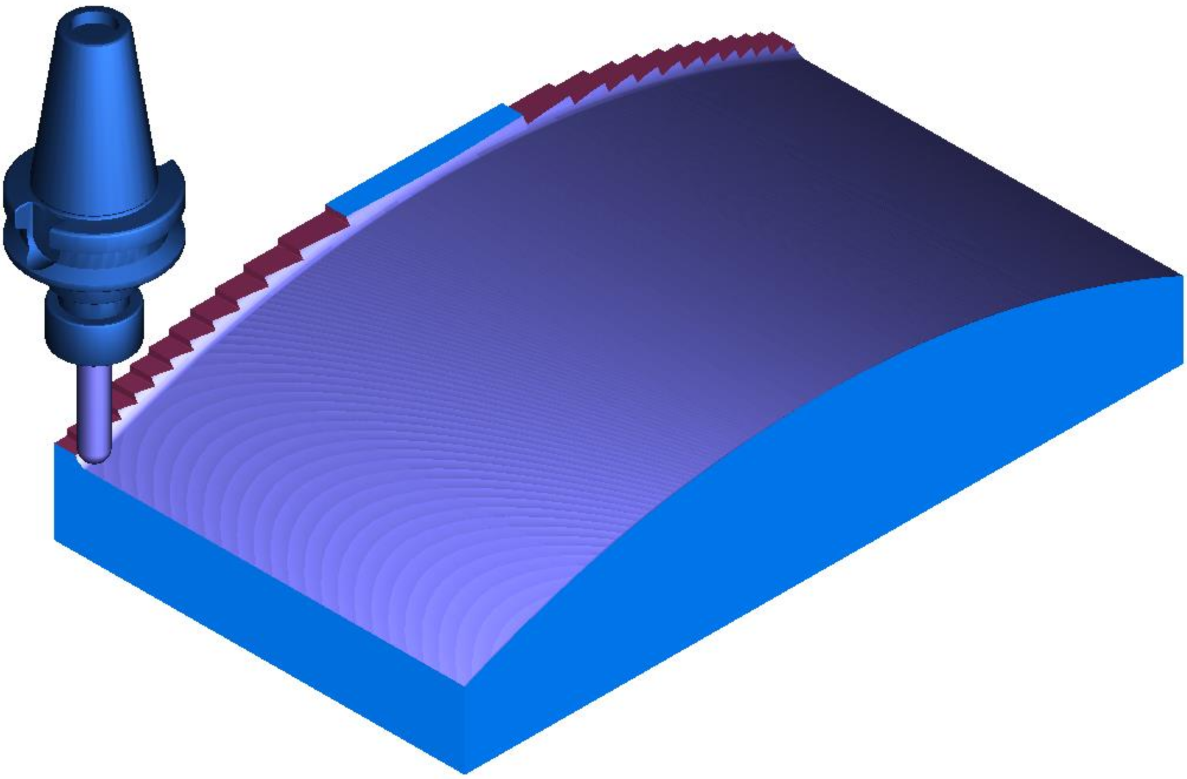
Stepover

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.

Large Stepover



Small Stepover

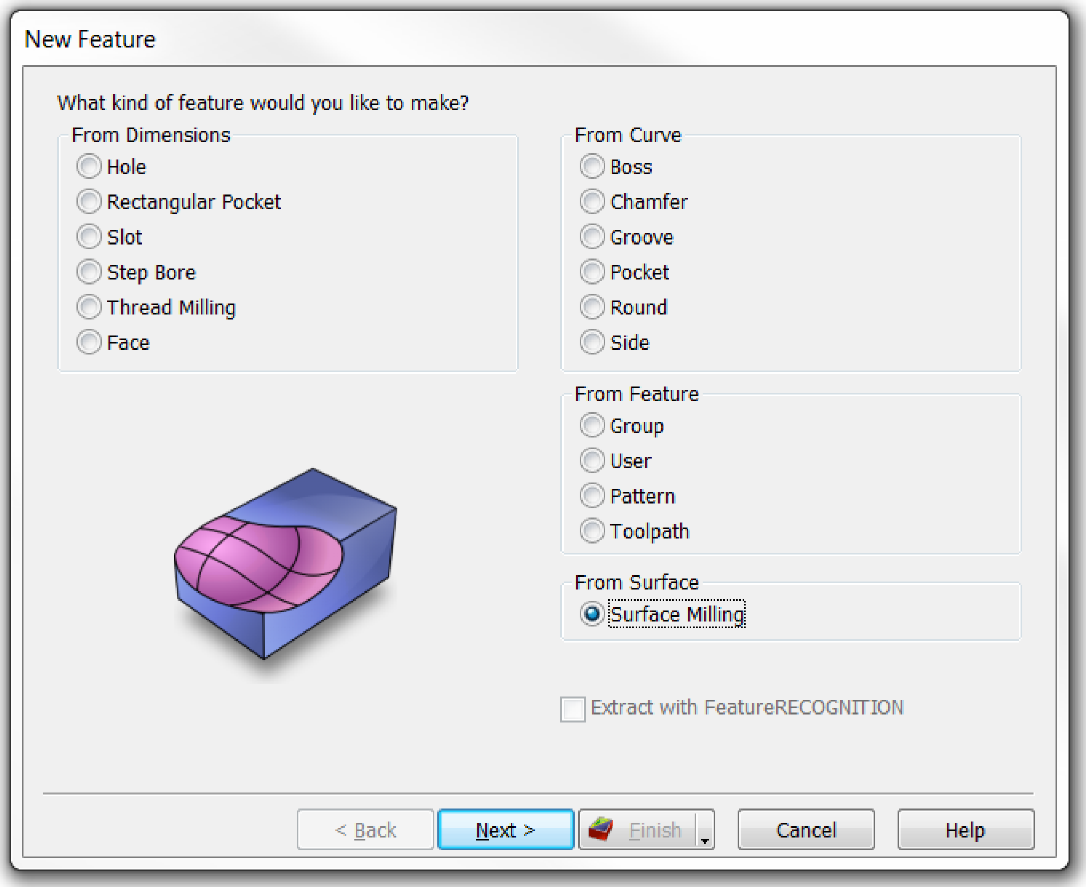


It is common practice to calculate toolpaths with a large tolerance and stepover while experimenting with different strategies. Once the strategy looks correct, the user can then tighten up the tolerance and stepover for the final manufacturing calculation. This can save a great deal of time when working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

Finishing Strategies

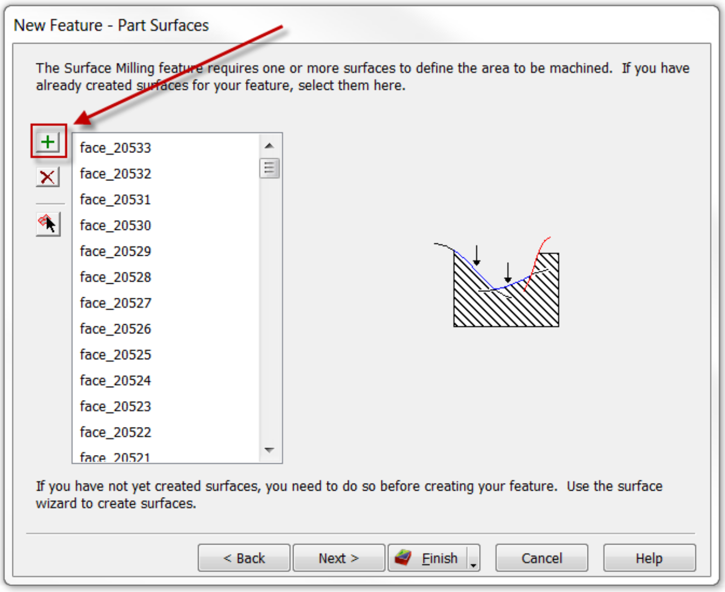
1 **Import** your solid Model

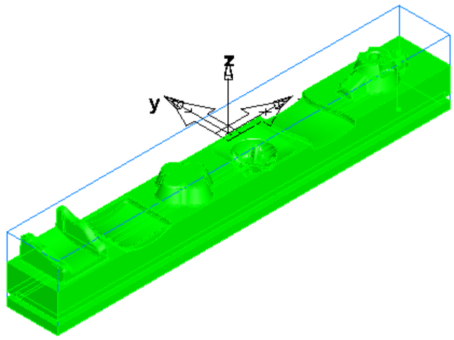
2 Select **Ctrl+R** to activate a New Feature. Select **Surface Milling**.



3 Select **Next**.

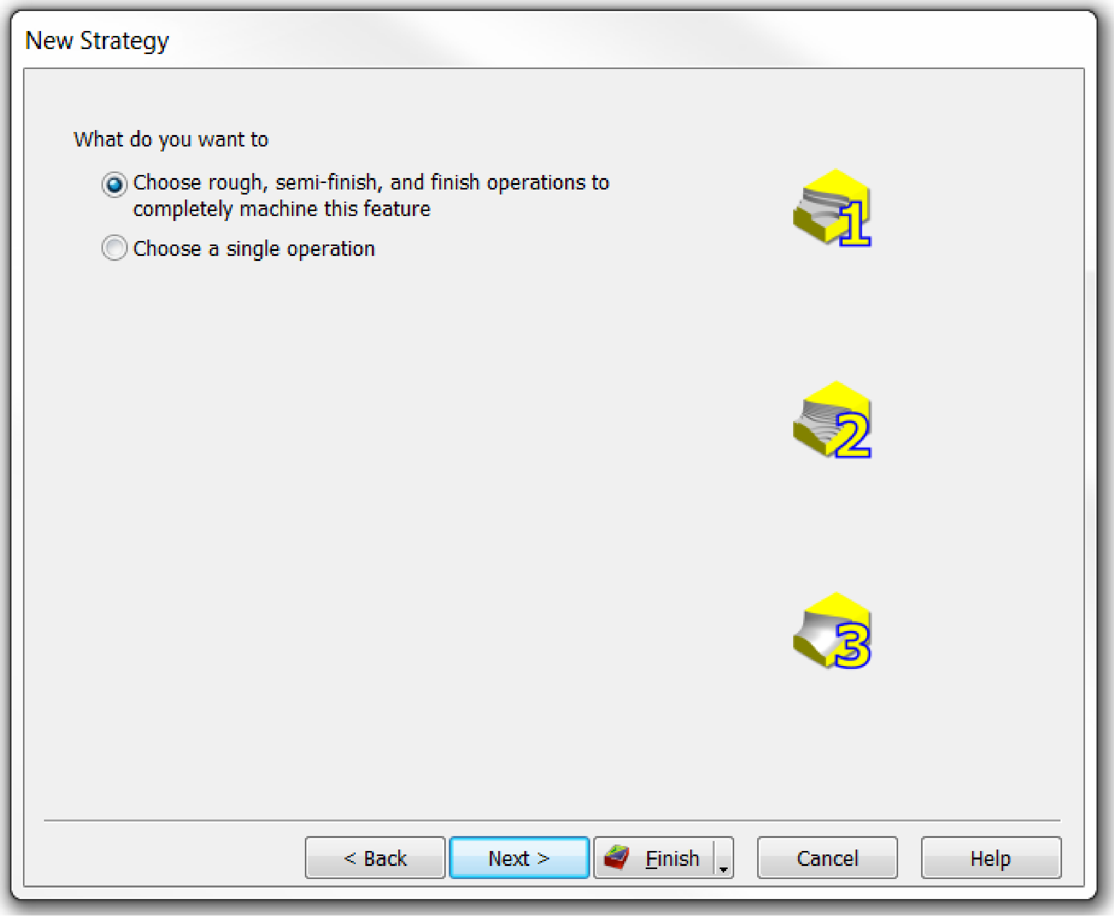
4 Manually select the solid model by windowing the part and then select the Green + key to select all of the surfaces on the solid model. As shown on the next page.





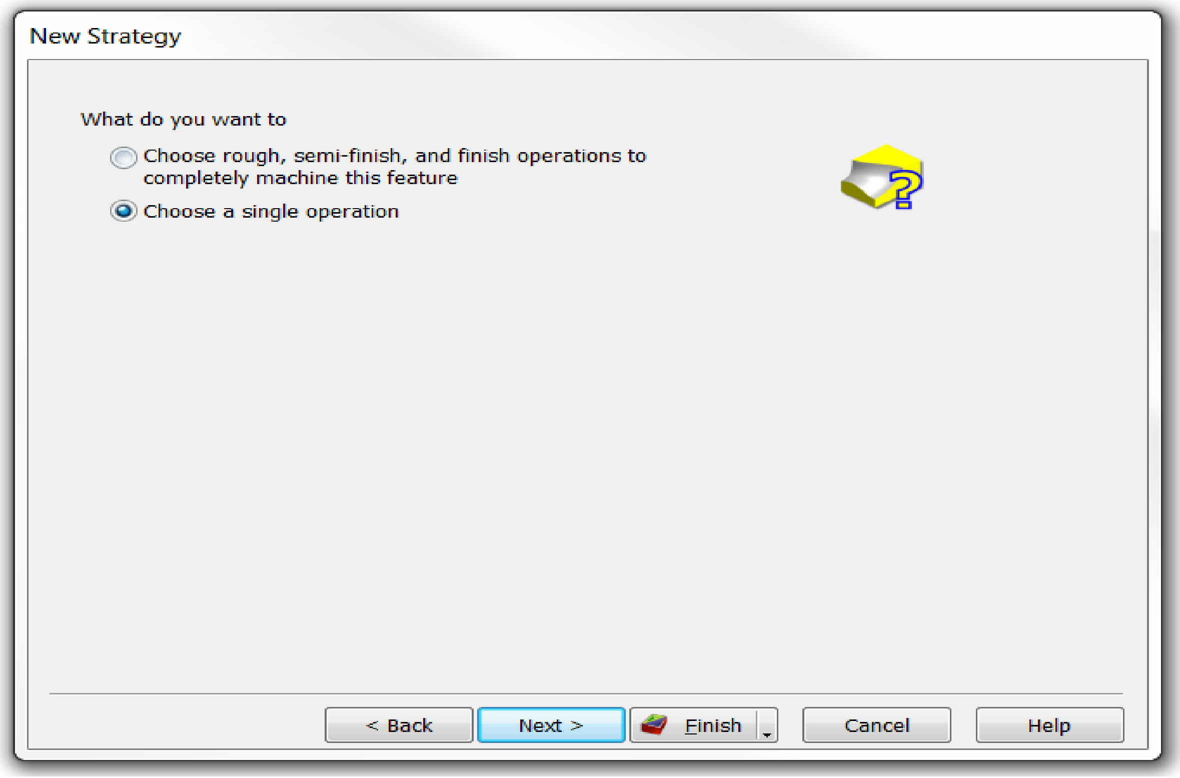
5 Select **Next** and you will be given **two** options.

6 Option **One**. This will allow you to machine automatically any part using Roughing and Finishing strategies. We will cover this later on in the chapter.

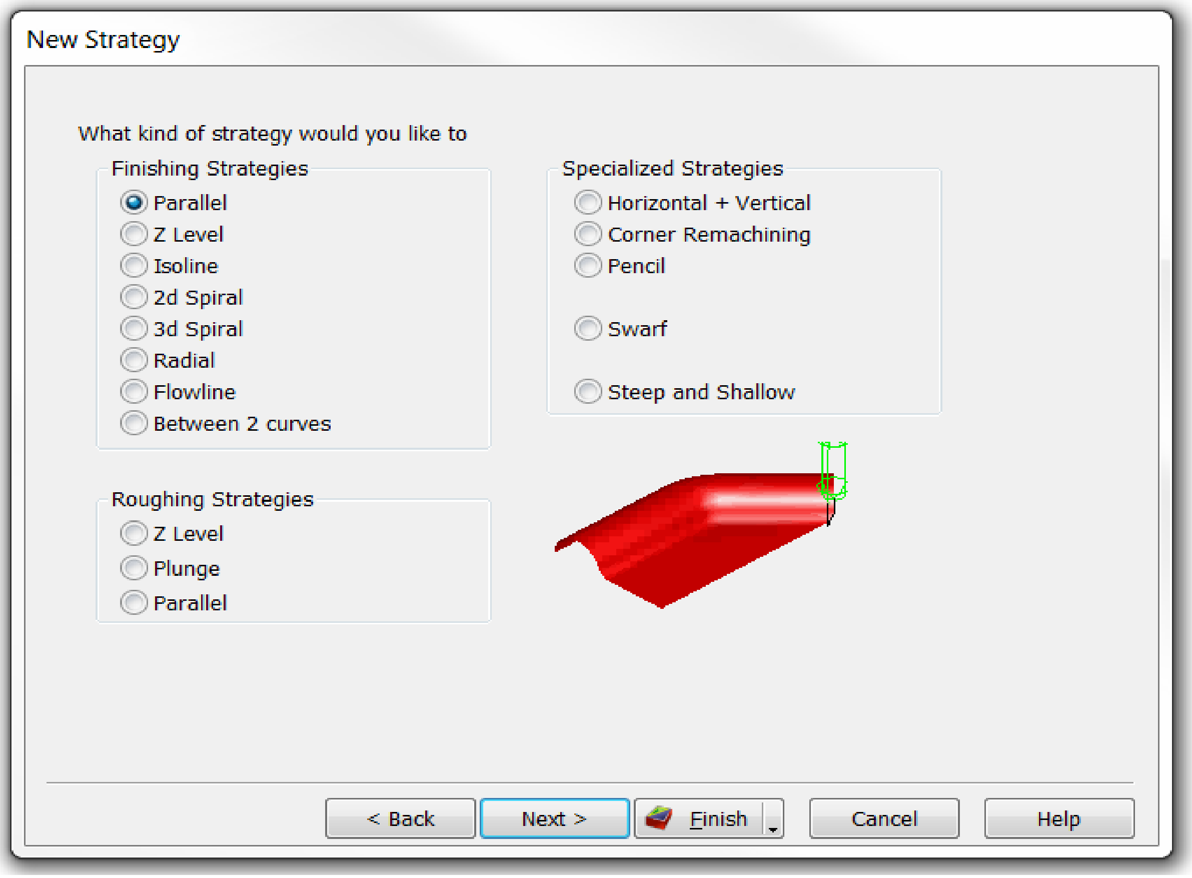


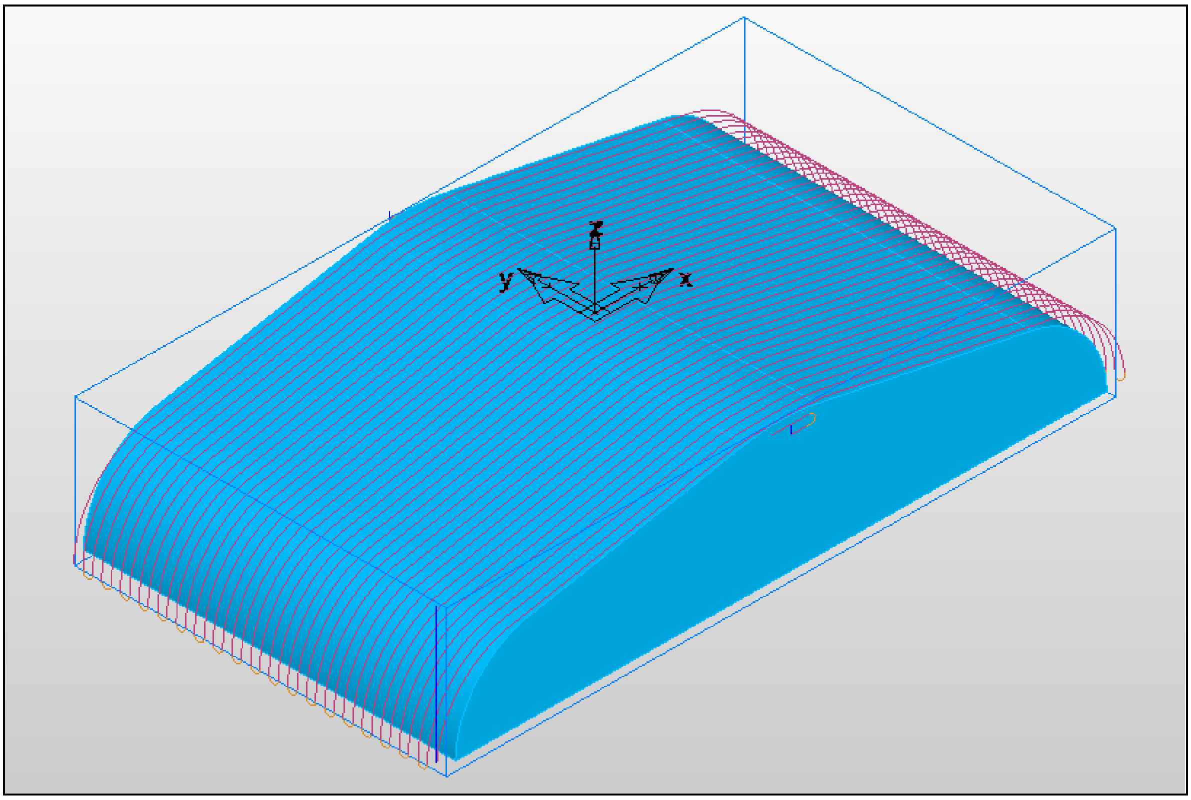
7 Option **Two**. Choose a single operation allows you to choose a strategy from another menu. Please select this option. As shown on the next page. Select **choose a single operation** option. One strategy at a time.

*The **3D Machining strategies** will be explained in detail.*



Parallel





- **Parallel** - Projection milling techniques are a robust and easily understood method of generating 3D toolpaths. They work by taking a pattern of curves and projecting points from these curves onto the surfaces of the part.

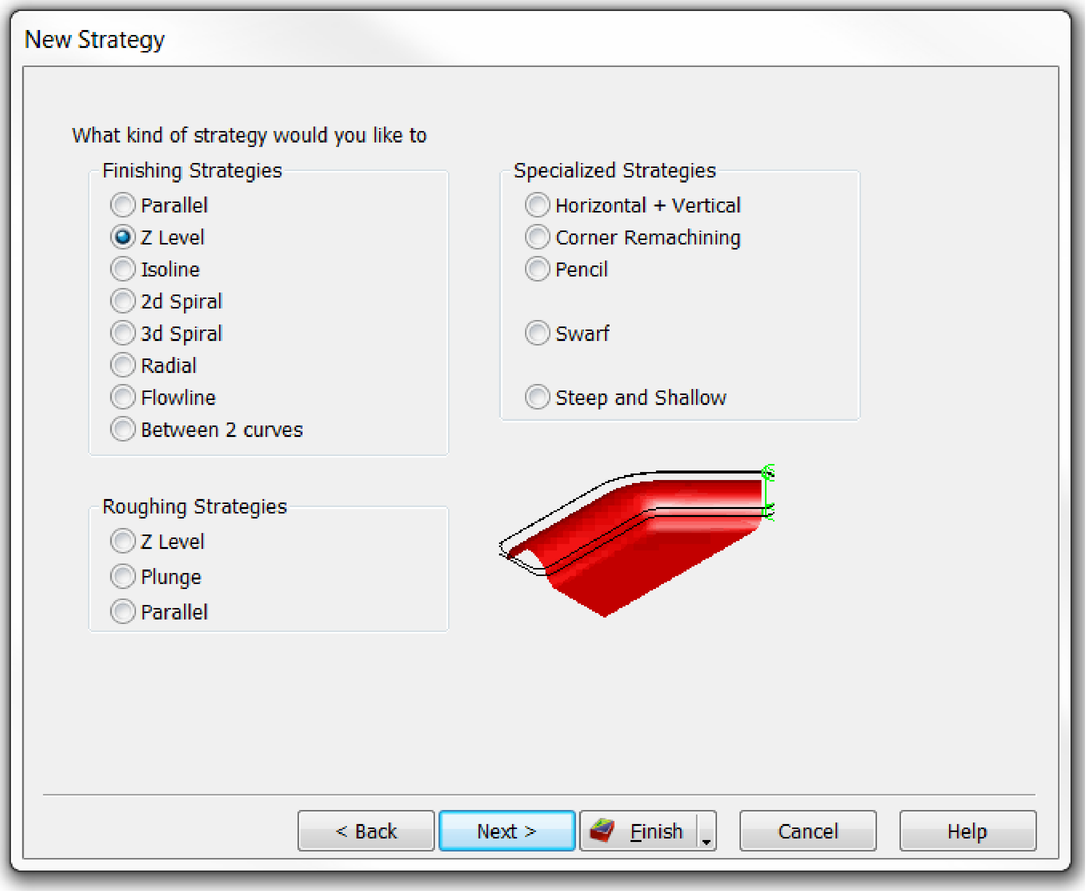
  Advantages

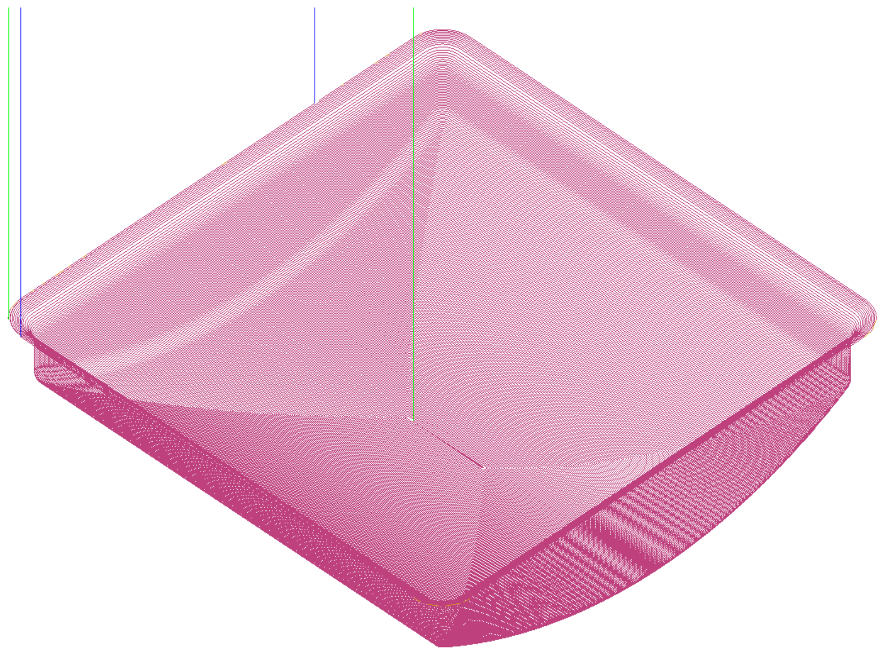
- The strategy is very robust.
- The strategy handles overlapping surfaces well.
- It is good for multiple surface manufacturing.
- It lets you specify planar stepover distances or scallop heights for finishing.
- Surface normals are not considered for the manufacturing computation.

  Disadvantages

- It has poor handling of near-vertical surfaces.
- It does not handle vertical surfaces because they are invisible from above.

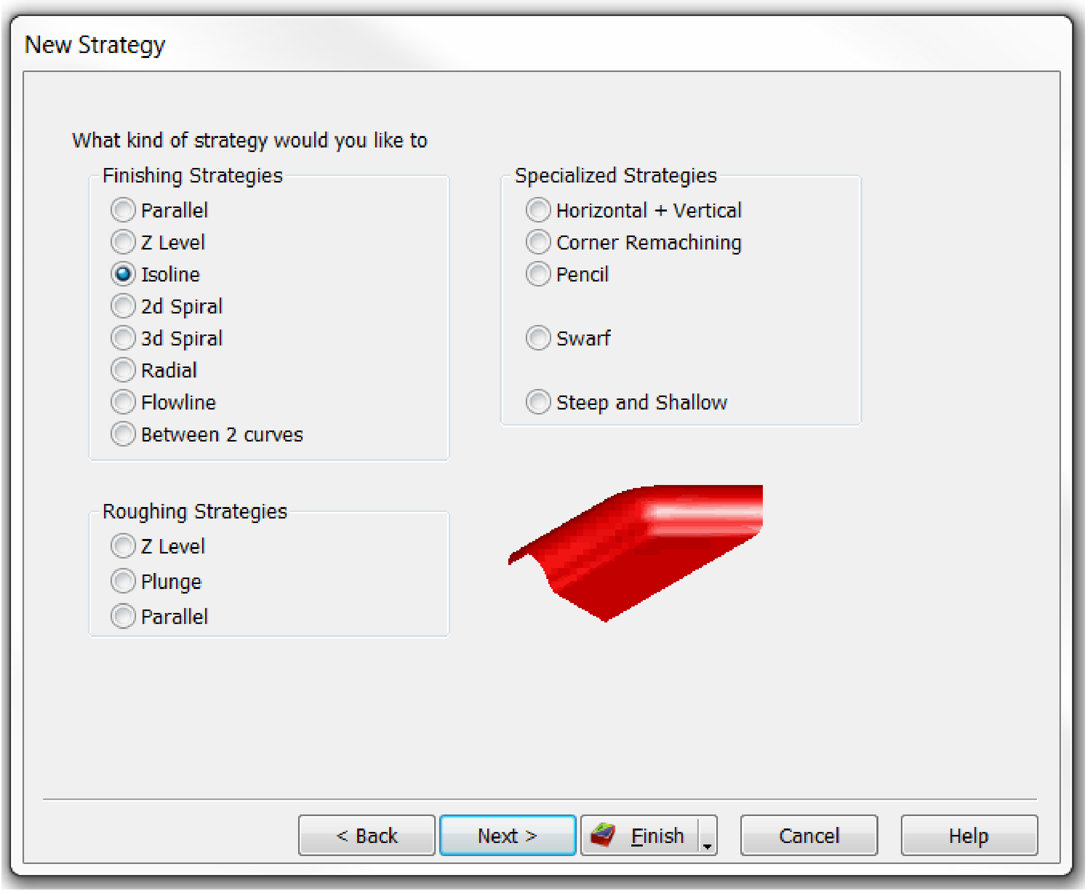
Z Level Finishing

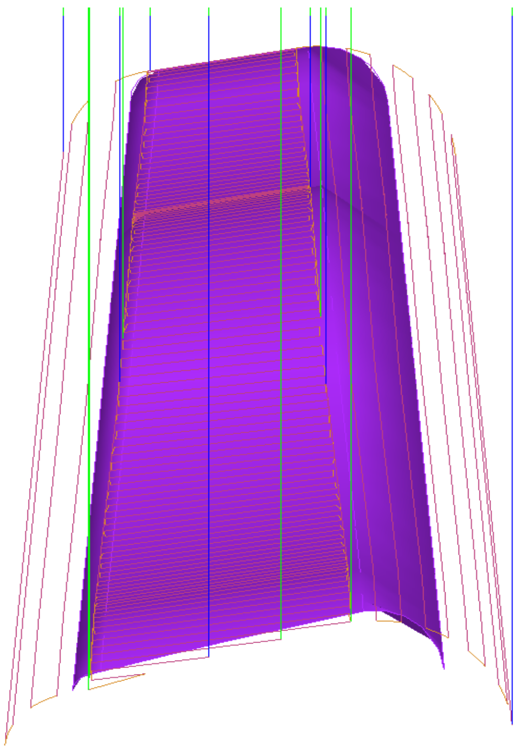




- **Z Level Finishing** - The Z Level Finishing method slices the feature at various depths, and then mills each slice. This is a good technique for finishing steep walls or when you require a consistent depth of cut. This technique is sometimes called waterline milling. Just like Z Level Rough, it starts with slicing the model and then creating toolpaths from the slices.

Isoline

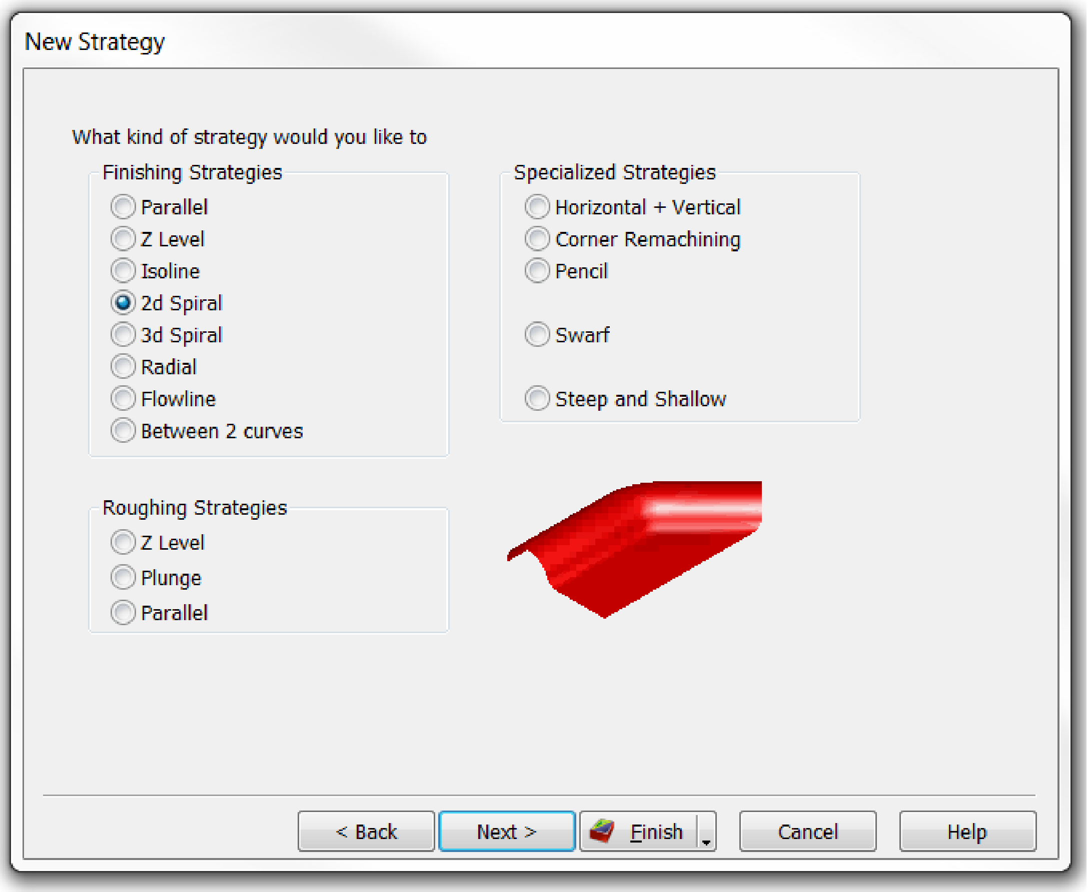


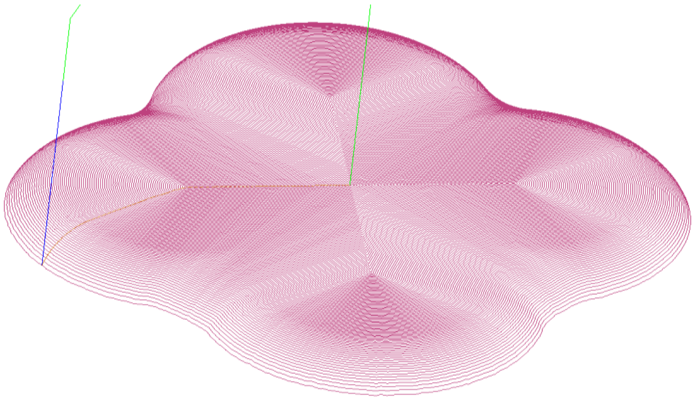


- **Isoline -** Uses the isoline curves of a surface to mill the surface. These curves can be in the **row** direction or **column** direction.

- **Advantages -** Uniform finish with scallop height control. The toolpaths are spaced based on the distance along the surface. Nearly-vertical walls handled well. Because this is not a projection technique, nearly vertical walls are cut correctly.
- **Disadvantages -** Toolpaths generated on a surface-by-surface basis. Restrictions of Isoline milling.
- **Isoline -** milling works on a surface-by-surface basis. This can result in numerous retracts. The orientation of the surfaces matters. Toolpaths are generated for surfaces whose normals point up. Surfaces are 'auto-flipped' where possible, but for vertical and some other cases, you must specify the machining side. Select the surface in the list and click Switch machining side. Isoline milling may mill on the wrong side of the surface or if certain flags are set, it may skip the surface.

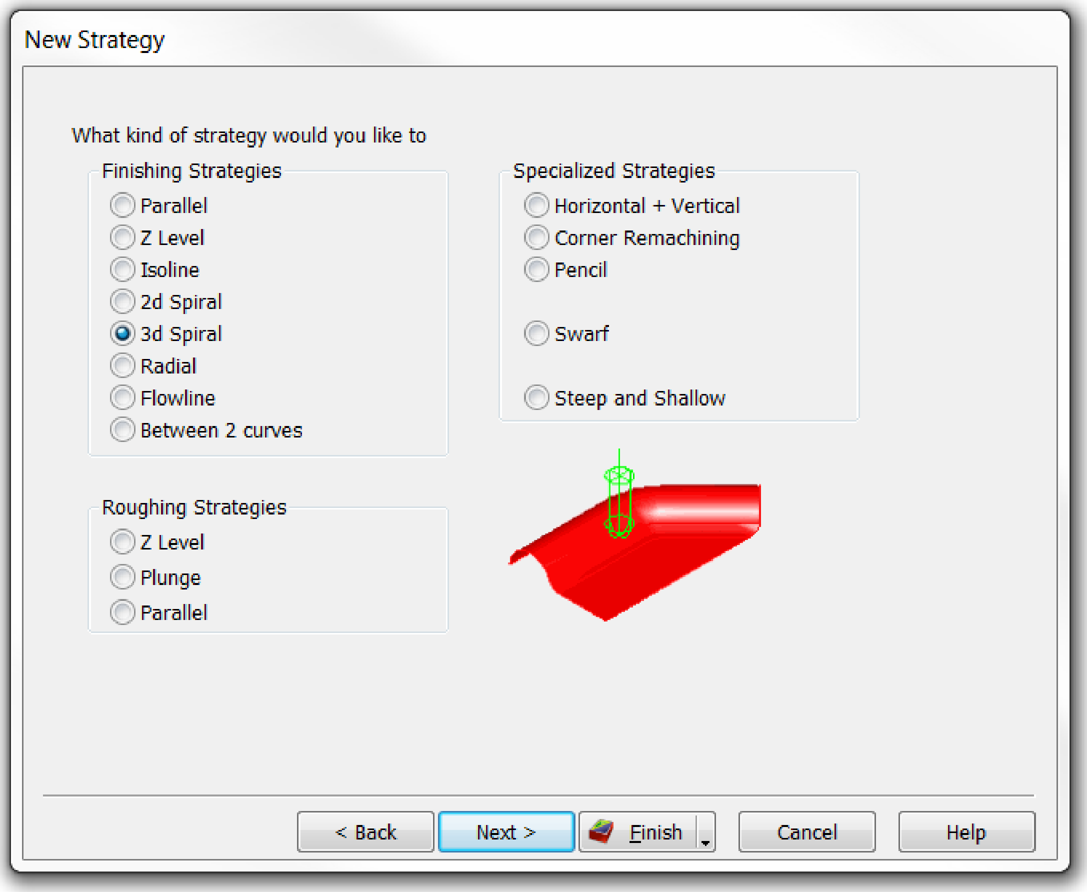
2d Spiral

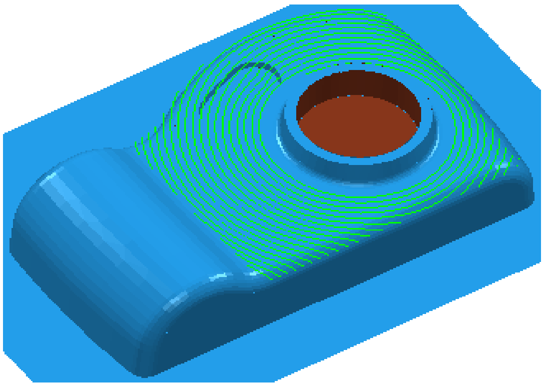




- **2d Spiral** - Will mill a feature with a pattern of offset toolpaths. The pattern is obtained by taking the stock boundary, the feature boundary or the curve specified on the Stock tab and offsetting this curve toward the center of the part. The pattern can be cut either towards the feature center or away from the feature centre.
- The steps between the passes are calculated in 2D. For spiral toolpaths that use a 3D stepover use the 3D spiral technique. To use the stock boundary, select Use stock dimensions on the Stock tab. This results in a square shape to the toolpaths.

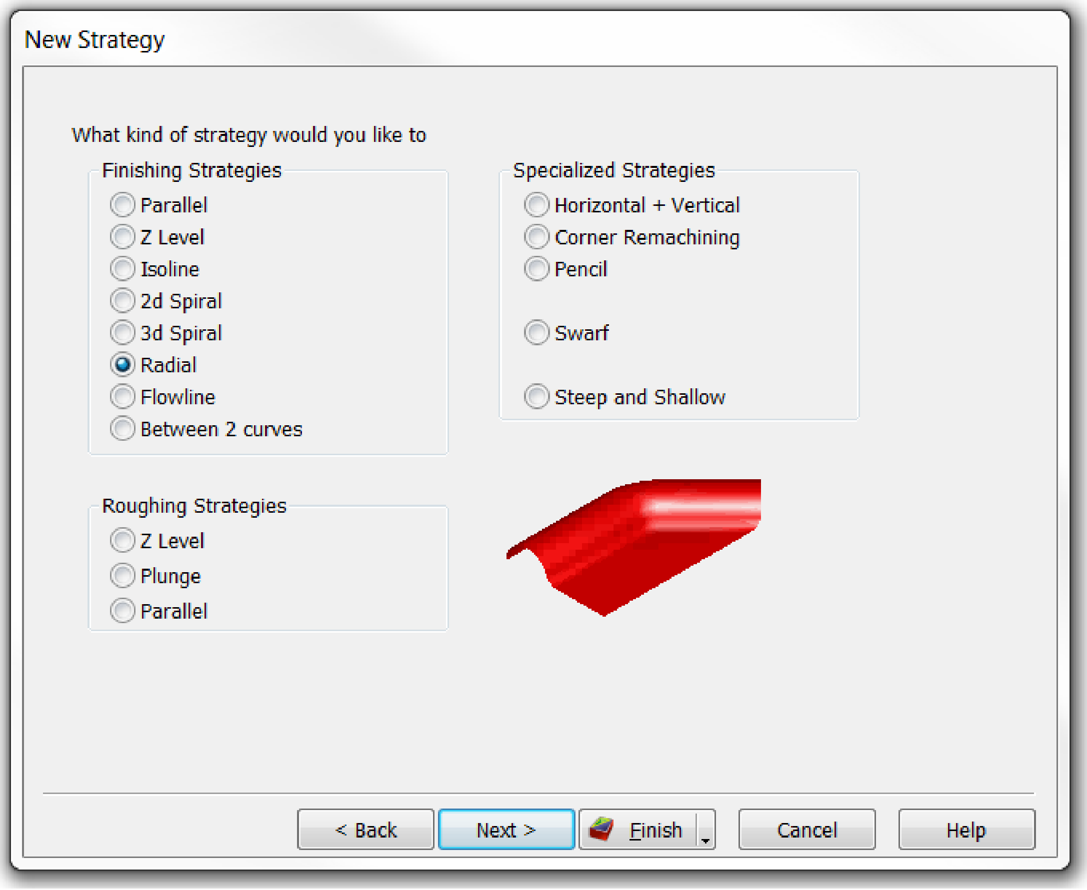
3d Spiral

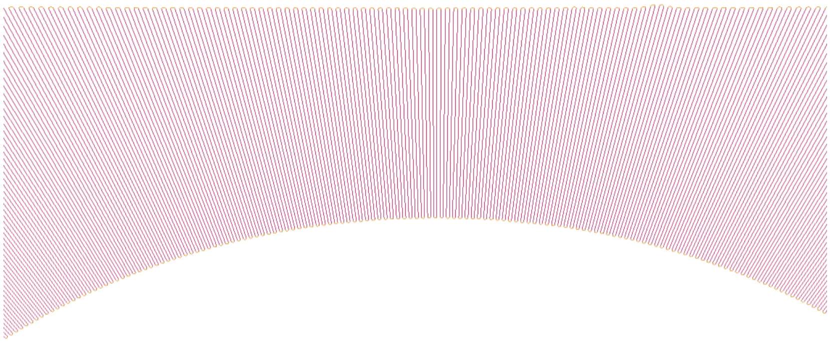




- **3d Spiral** - This finishing technique is best suited to machining areas that need a constant stepover and works well on near-horizontal faces.
- Spiral toolpaths mill a feature in a series of offsets towards the feature centre. The initial pattern is specified by taking the stock boundary, the feature boundary, or the curve specified on the Stock tab.

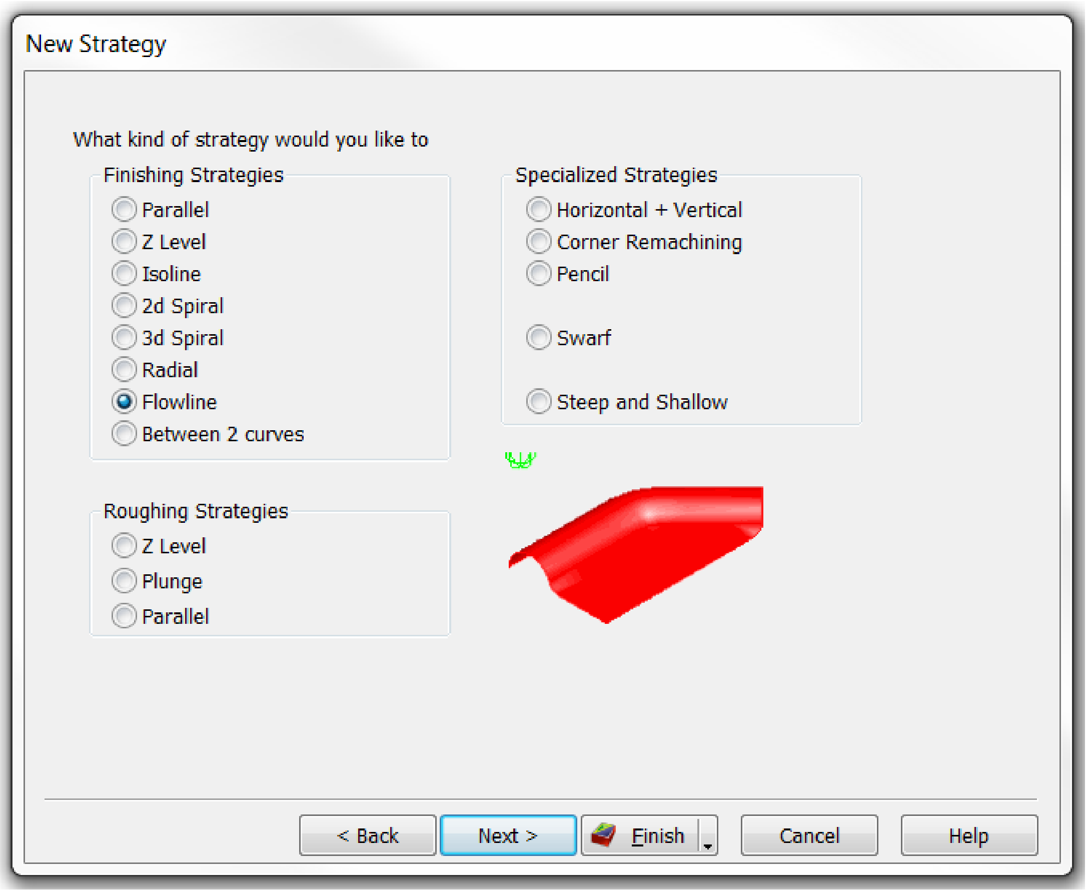
Radial

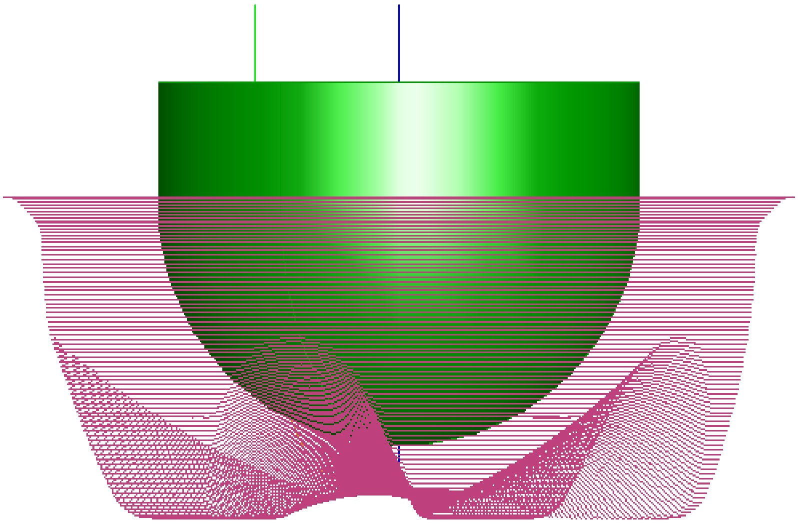




- **Radial -** This strategy creates a radial pattern within a boundary and projects it onto the model.
- **Centre point**. The centre of the radial pattern is calculated automatically unless you set a **Centre point**. This point is projected down onto the surface to become the centre of the radial pattern.

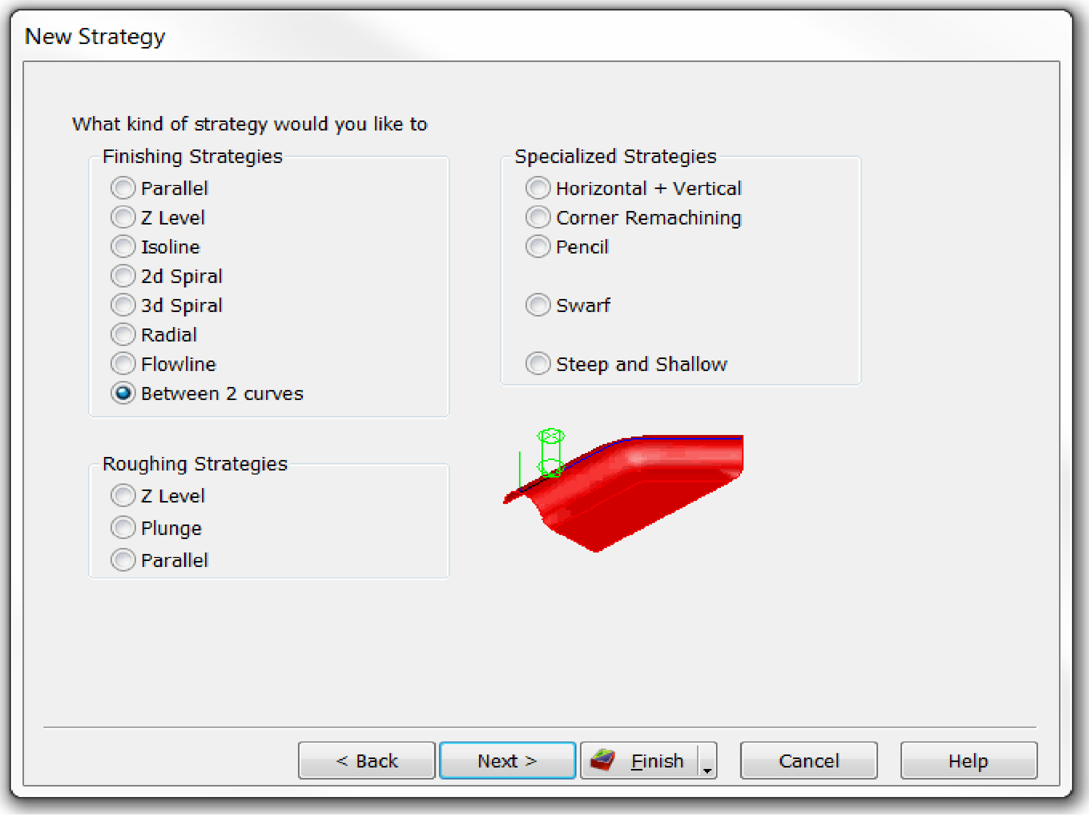
Flowline

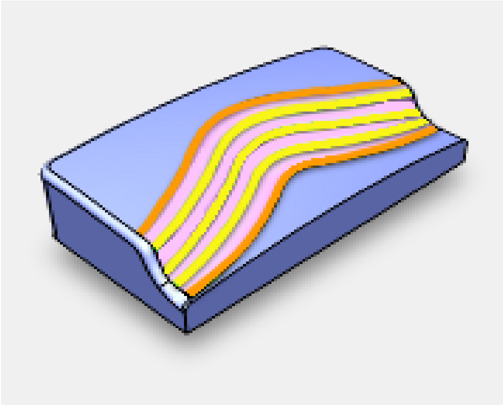


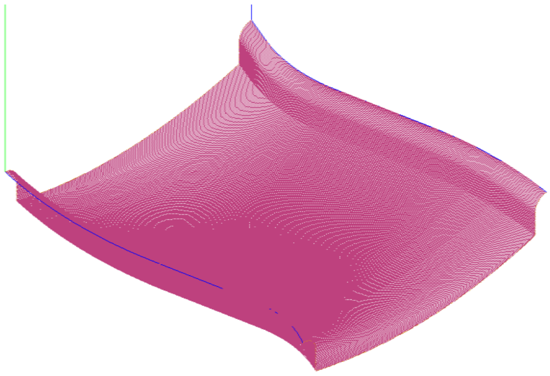


- **Flowline** - This technique projects the isolines from one surface onto the surfaces of the feature. The isolines are projected in the direction of the surface normal.

Between 2 curves

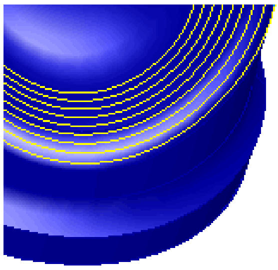




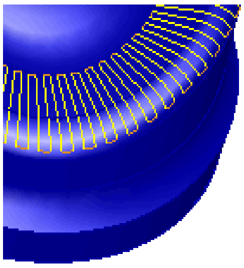


- **Between two curves** – This calculates toolpaths that are bounded by two curves. This finishing operation is used to control the shape of the toolpath over multiple surfaces by restricting the machining of the surface to be between the two specified curves. The toolpaths mill a feature in a series of offsets starting from the first or start curve towards the second or end curve.

Along curves — The toolpath is similar to an offset toolpath radiating out from the start curve to the end curve.

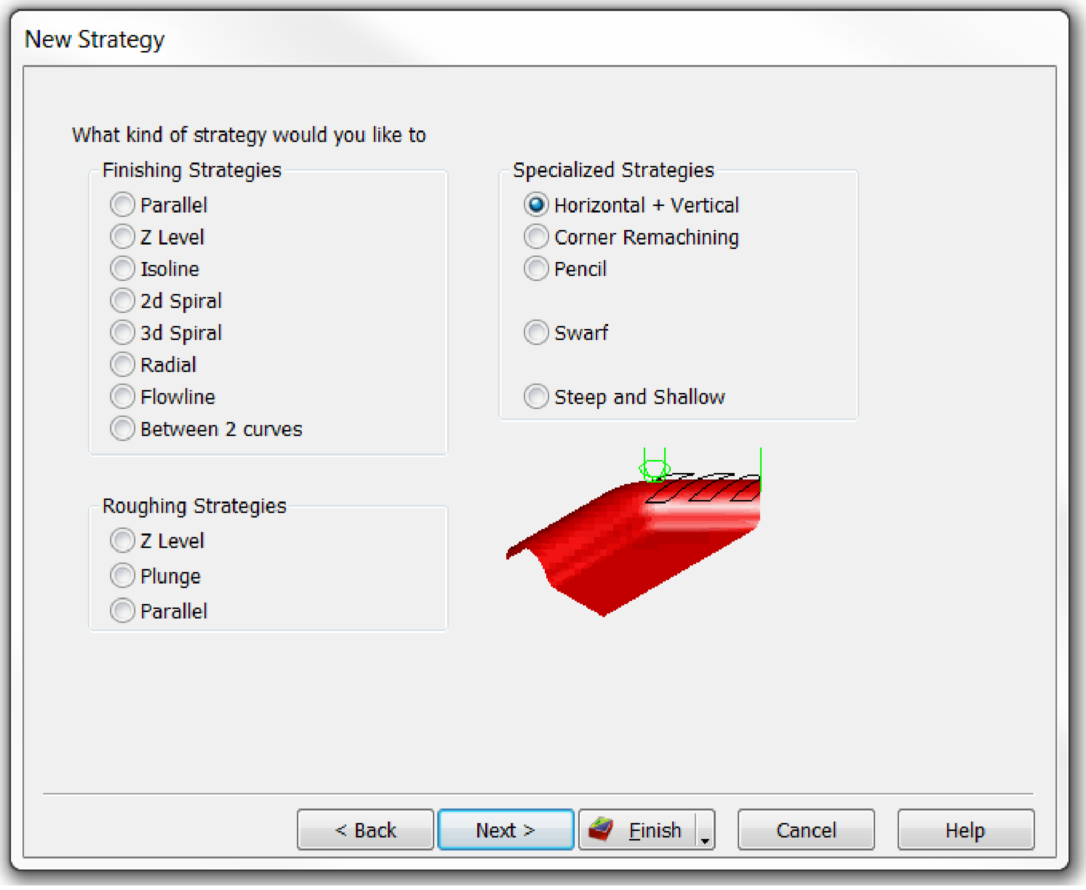


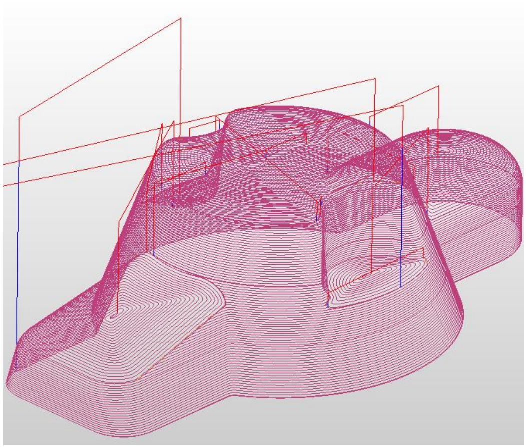
Across curves — The toolpath goes from a point on the start curve to a point on the second curve.



Specialised Strategies

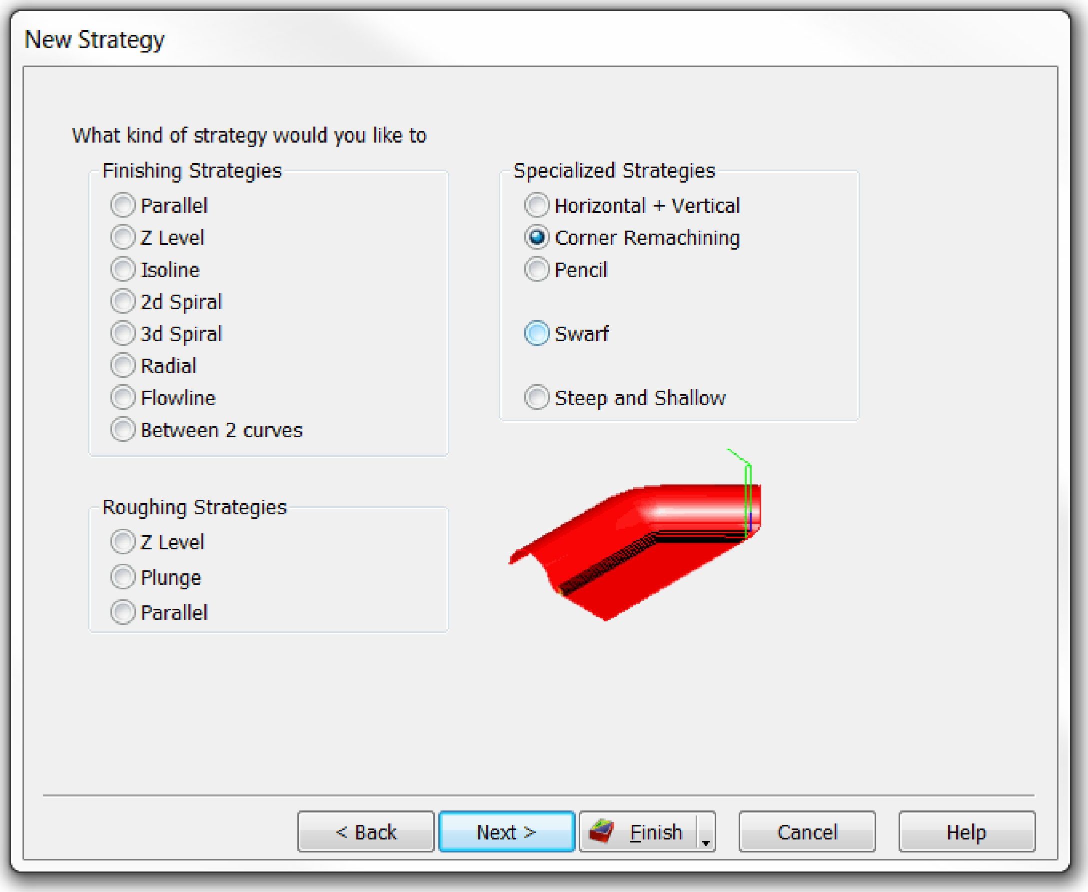
Horizontal + Vertical

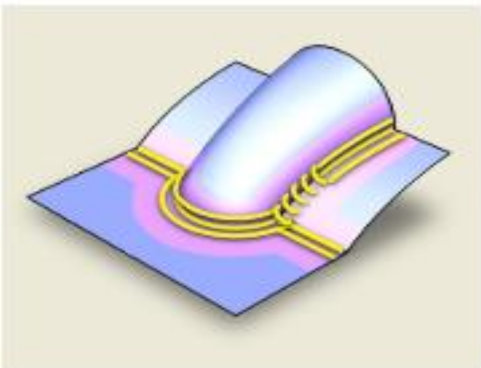




- **Horizontal + Vertical** - This strategy combines two different toolpath operations, one for finishing shallow portions of the part and one for finishing the steep regions.
- An X parallel or spiral toolpath is applied to the shallow regions, and a Z level finishing operation cuts the steep regions.
- The attributes available on the Milling tab are different for each operation:

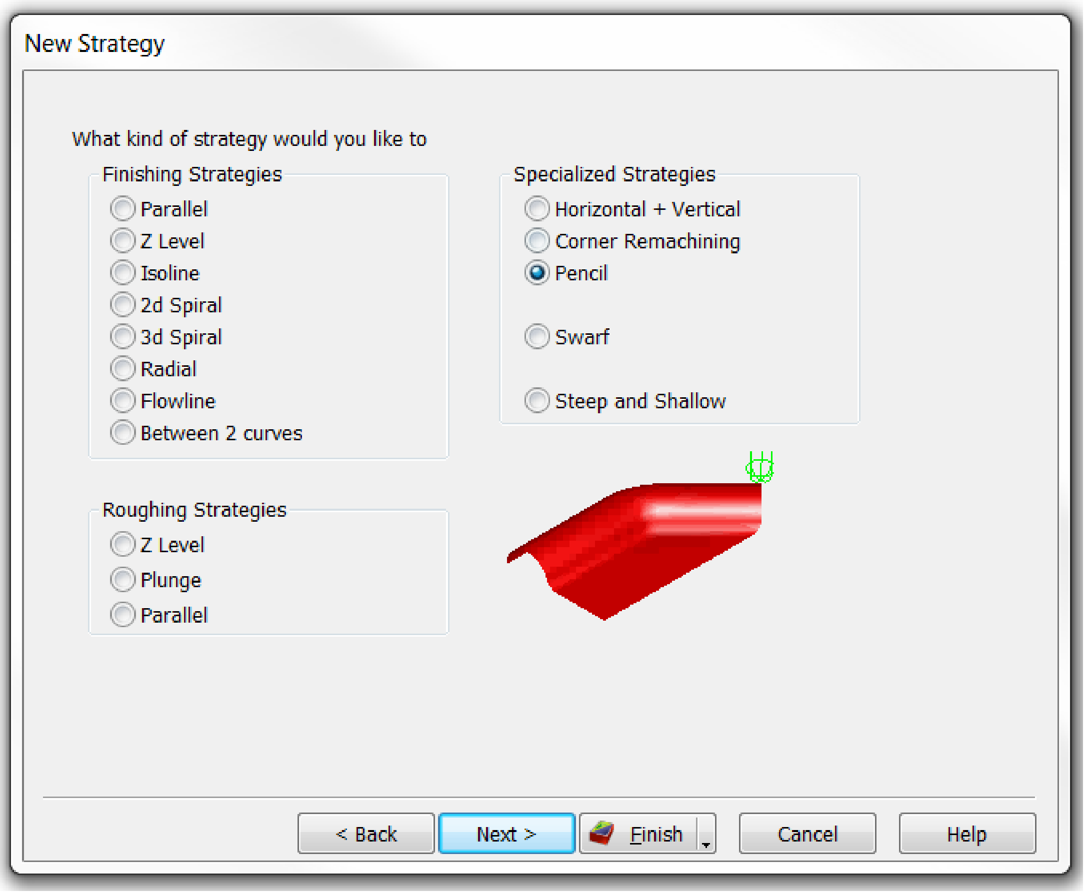
Corner Re-Machining

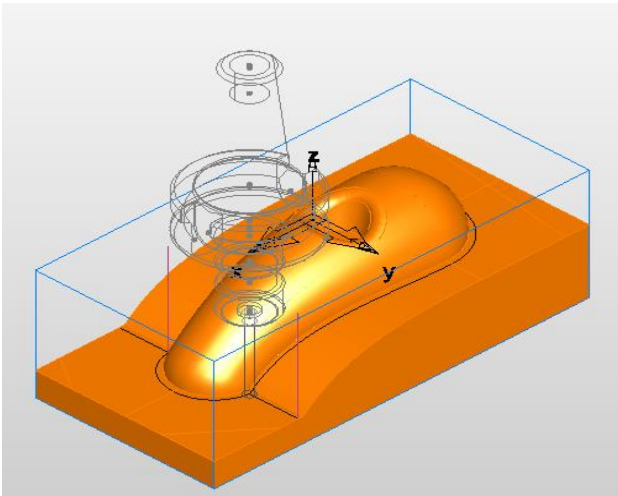




- **Corner Remachining** - This is used to automatically mill regions that were not cut by previous operations. You provide the diameter of the previous tool that was used to cut the part and **FeatureCAM** automatically determines the uncut regions and applies a toolpath to them.
- Corner Remachining is used to clean up corners that occur between non-tangential surfaces. Each corner edge is called a trace line. By using the options on the strategy page, you may cut in various directions relative to the trace lines.
- Corner Remachining is available in **four** different styles: **Along**, **Across**, **Combo along and across**, and **Multi pencil**.

Pencil



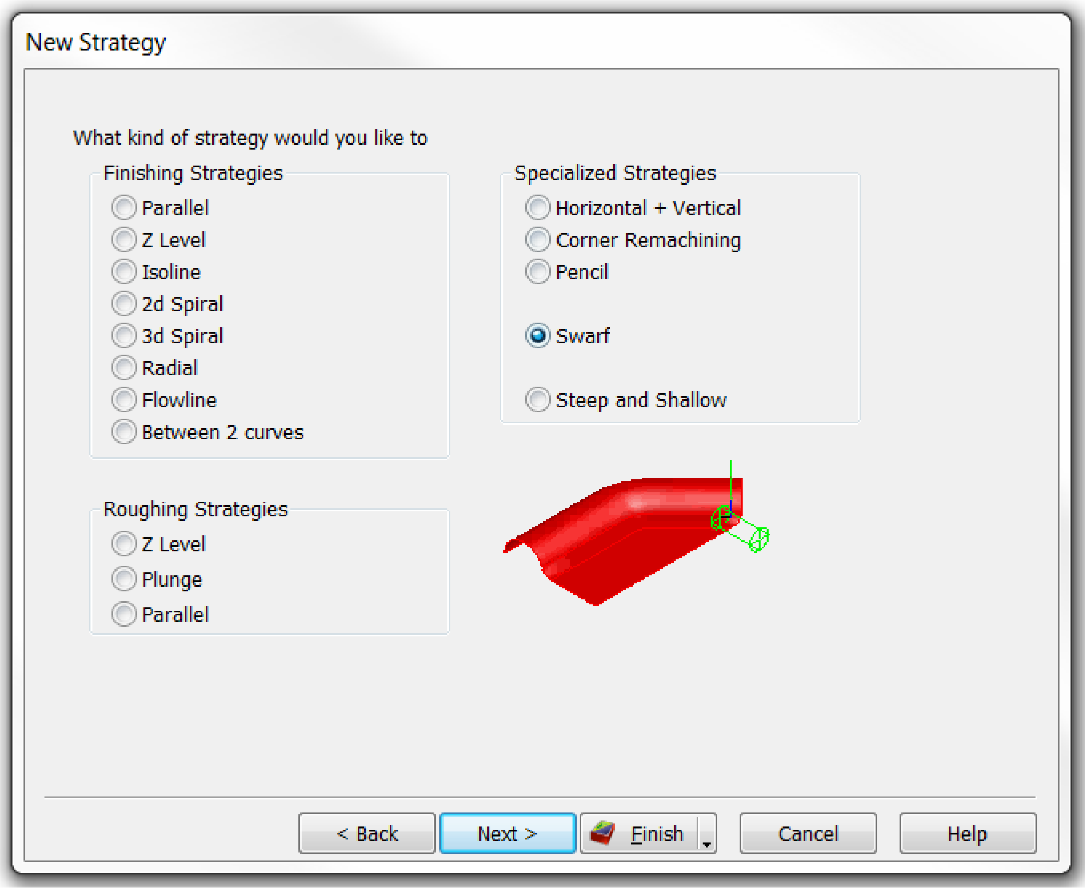


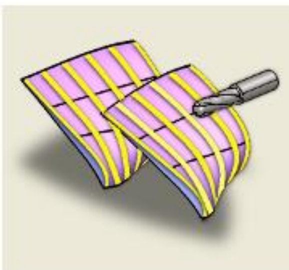
- **Pencil** - This strategy creates a single trace corner toolpath. It is used to clean up corners that occur between non-tangential surfaces. They are automatically calculated inside any existing boundary.

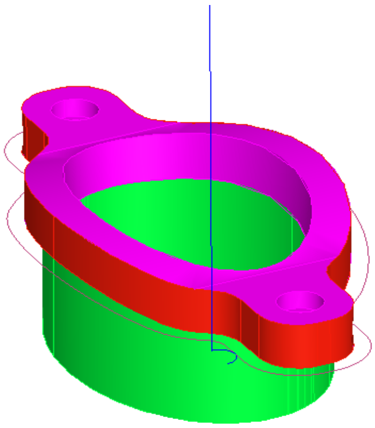
Applications for pencil milling include:

- Finishing fillets in a part with a single toolpath.
- Cleaning up sharp concave corners.
- Pre-relieving corners before high-speed finishing the part with a small tool.
- Roughing fillets by using the Finish allowance.

Swarf



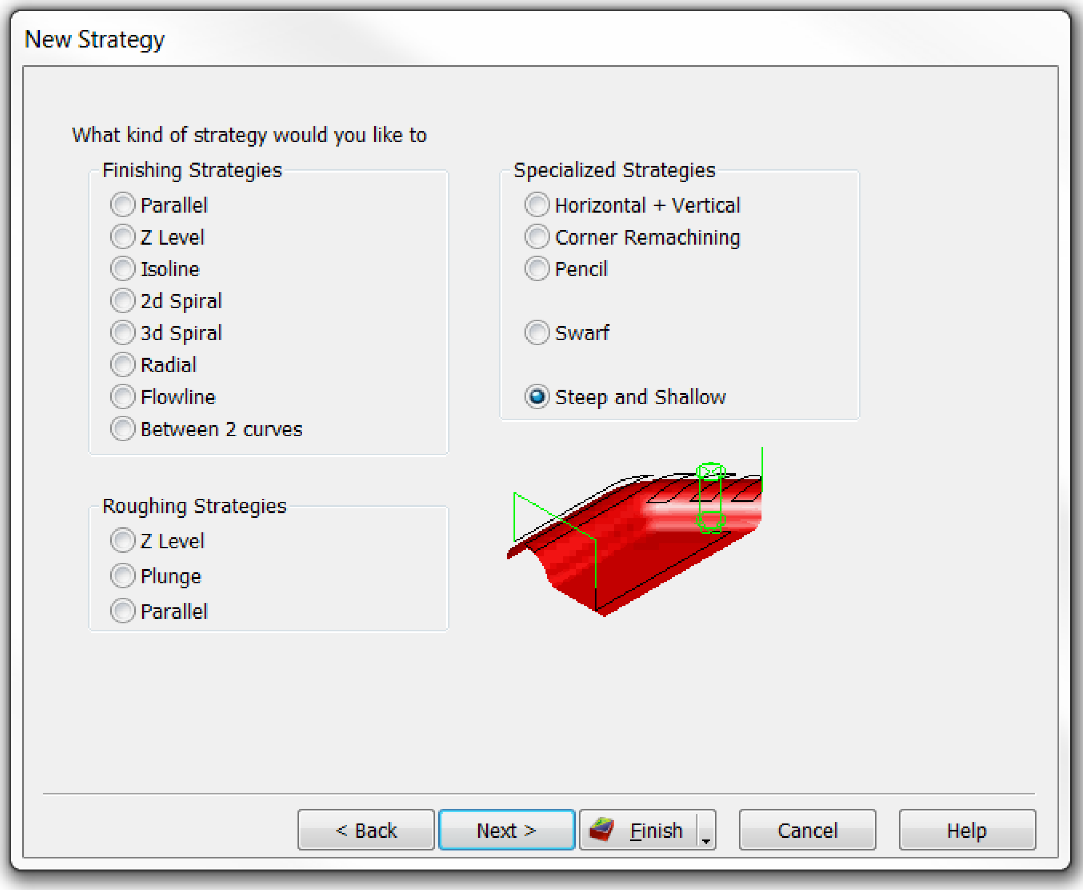


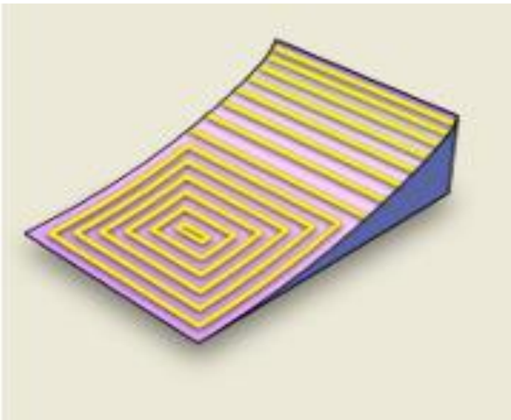


- **Swarf** – Swarf cutting calculates toolpaths that cut with the side of the tool and works only on developable surfaces because the tool needs to be in contact with the surface for the whole cutting depth.
- For a tool to Swarf cut, it has to be able to have contact with the surface at all points along the cutting edge of the tool. For a non-developable surface, **FeatureCAM** always leaves material on or produces fragmented toolpaths (rather than gouge). This means that you need to look at parts carefully before trying to Swarf cut them. It may be that rotating the part (cutting from the side rather than the top of an aerofoil blade) produces the desired result.

- Swarf machining makes every attempt to machine the selected surfaces but you may have to consider running more commands including ones with different options for Swarf milling to obtain the best machining results.
- It is possible to identify approximately whether a surface is developable and ruled by shading and also displaying its wire frame geometry within **FeatureCAM**. You can then orientate the view to be roughly down the expected tool axis vector.

Steep and Shallow

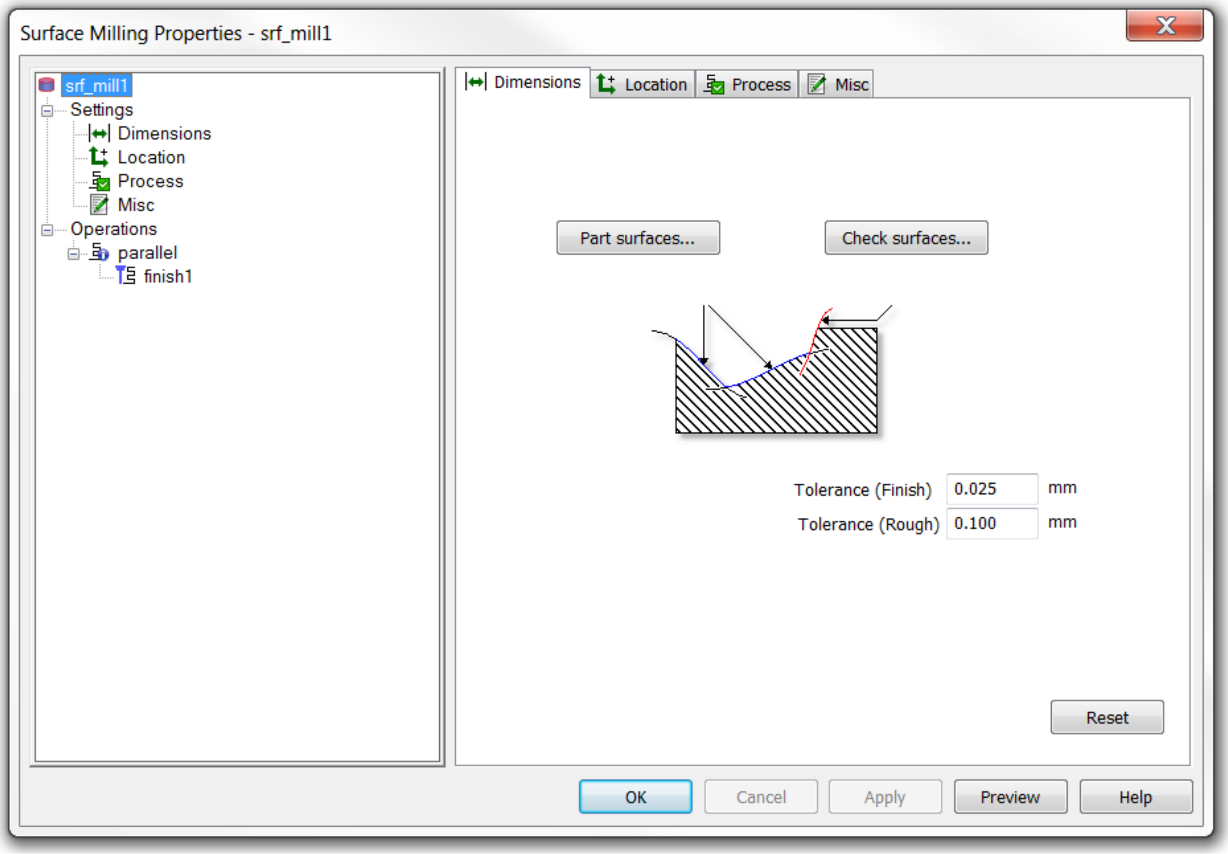




- **Steep and Shallow** - This strategy calculates a shallow boundary, and then creates a Z-level toolpath in the steep areas of the model and a Parallel or 3D Spiral toolpath in the shallow areas.

Surface Milling Properties

*You can use the **Surface Milling Properties** dialog to edit the properties of a **3D Surface Milling feature**.*



Feature-level tabs

The top level of the tree view is the feature level. Select the feature name at the top of the tree view to access these tabs:

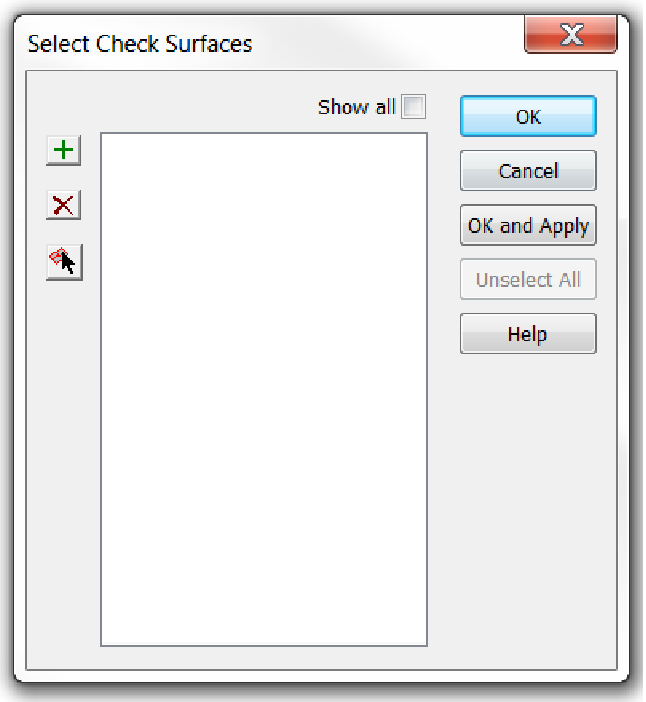
Dimensions — Use this tab to specify the **Part** and **Check** surfaces.

Select Part Surfaces dialog

- You can use the 3D Part Surfaces dialog to pick surfaces you want to include in your 3D part feature.
- Select the surface(s) in the list or click the Pick surface button and select a surface with the mouse. To pick additional surfaces, click the Pick surface button again before selecting each additional surface.
- Click **OK** to return to the Feature Properties dialog.
- Click **OK** and **Apply** to apply your surface selection to the feature and return to the Feature Properties dialog.
- You need to consider the following when specifying part surfaces:
- You cannot manufacture undercut surfaces using 3-axis machining, so it is a good idea to use only surfaces in the feature that can be cut from the setup.

- Some surfaces may be cut from multiple setups to manufacture all parts of the surface. In such situations, a Stock Curve is helpful in limiting the machining area to just those spots that need it.

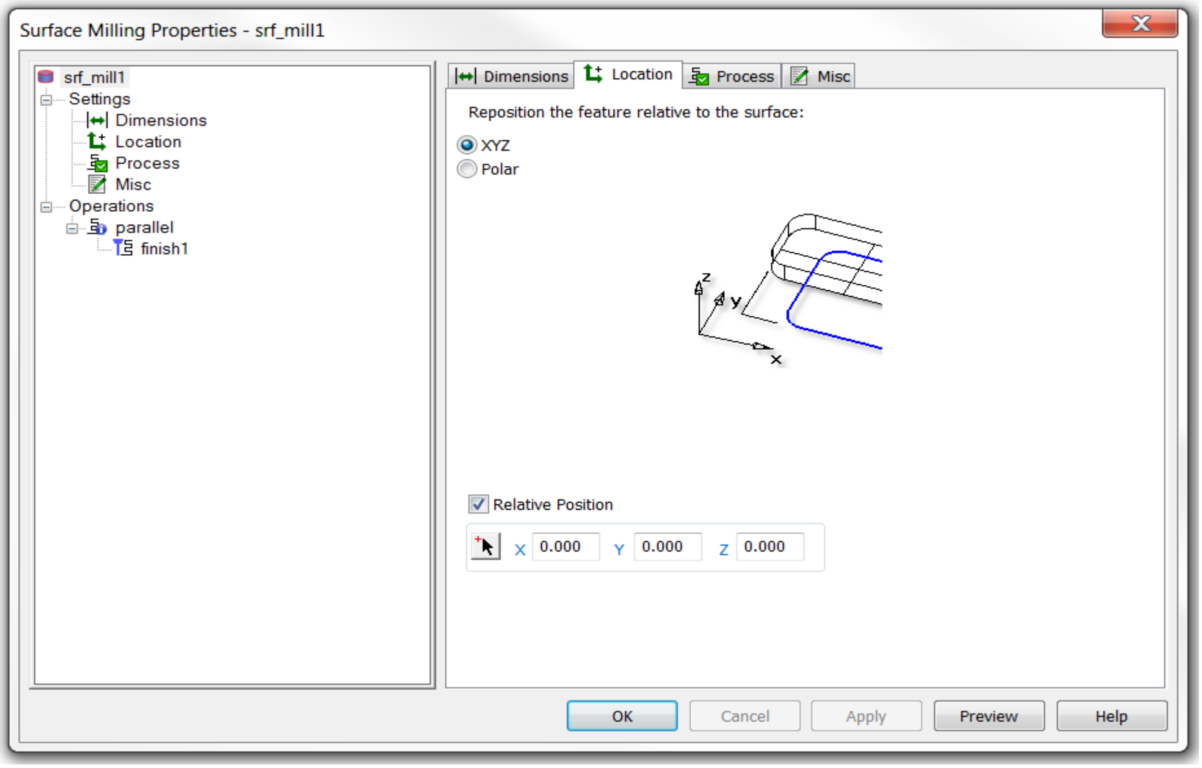
Select Check Surfaces dialog



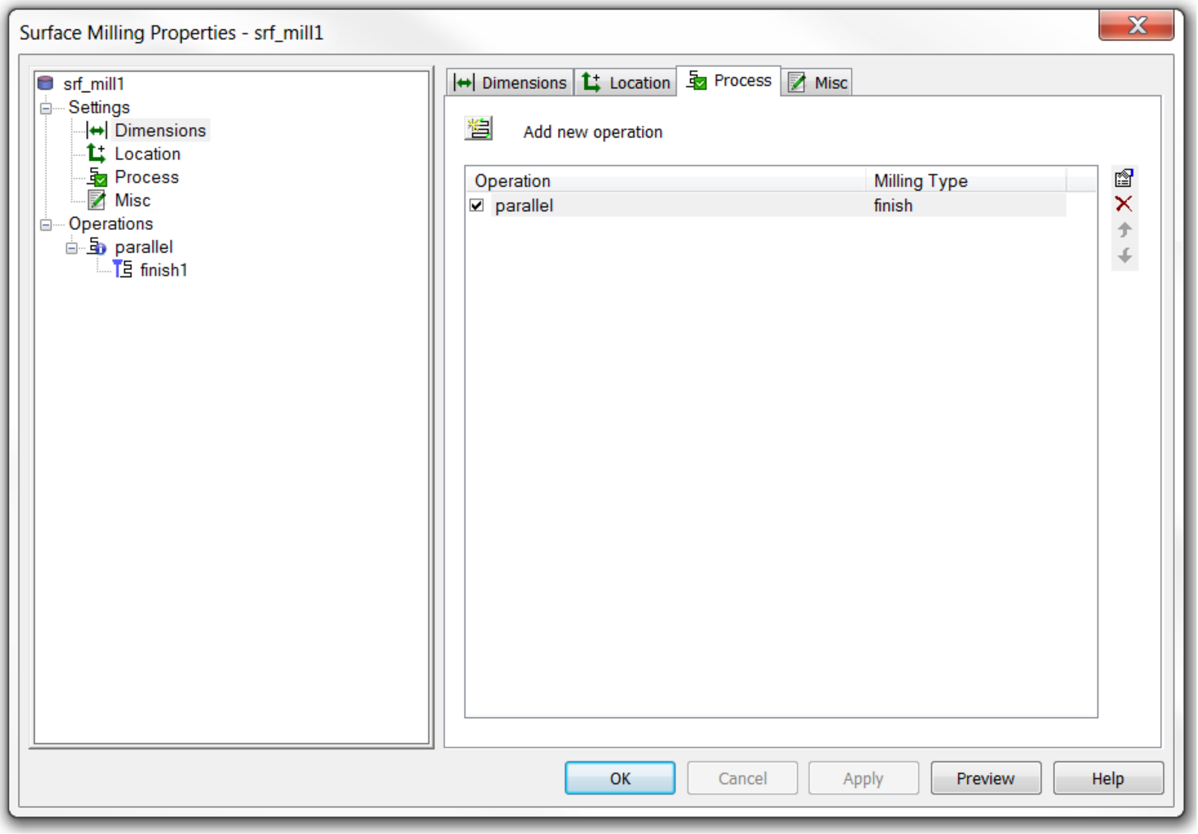
- Use this dialog to select surfaces that you want to use to limit machining.
- Check surfaces are surfaces that denote areas that you do not want to machine. Select surfaces that are more horizontal than vertical. A check surface acts as a boundary up to which milling occurs.
- If you select a vertical check surface the milling may resume on the other side of it if the surface to be milled extends beyond the check surface.
- Select the surface(s) in the list box or click Pick and select a surface with the mouse. To pick additional surfaces, click Pick again before selecting each additional surface.
- Click **OK** to return to the Feature Properties dialog.
- Click **OK** and **Apply** to apply your surface selection to the feature and return to the Feature Properties dialog.

Location

- **Location** — Use this tab to reposition the feature relative to the surface.



Process — Use this tab to create, delete and reorder the operations of the feature. New Operations can be added using the existing selected surfaces.



Machining Side — Use this tab to control which side of surfaces to machine

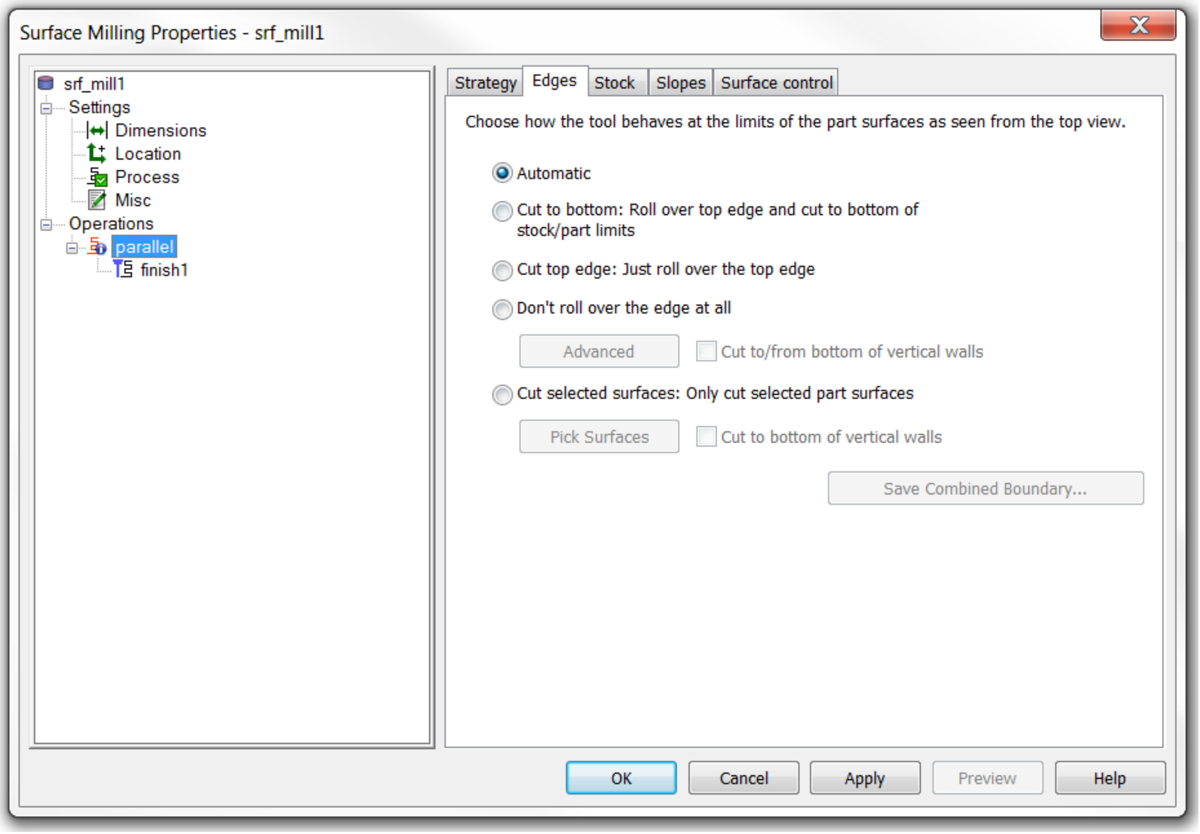
Misc — Use this tab for a variety of feature-level attributes

Strategy-level tabs

The next level is the strategy level. Select a strategy in the tree view to access the following tabs:

Strategy — Use this tab for rough/finish classification, edge protection, and re-machining

Edges — Use this tab to choose how the tool behaves at the limits of the part surfaces as seen from the top view.



The Edges tab is not available for Z-level roughing or Swarf finishing operations.

Stock — Use this tab to choose the clipping curves for the material to be removed

Slopes — Use this tab to set slope angle limits for restricting toolpaths

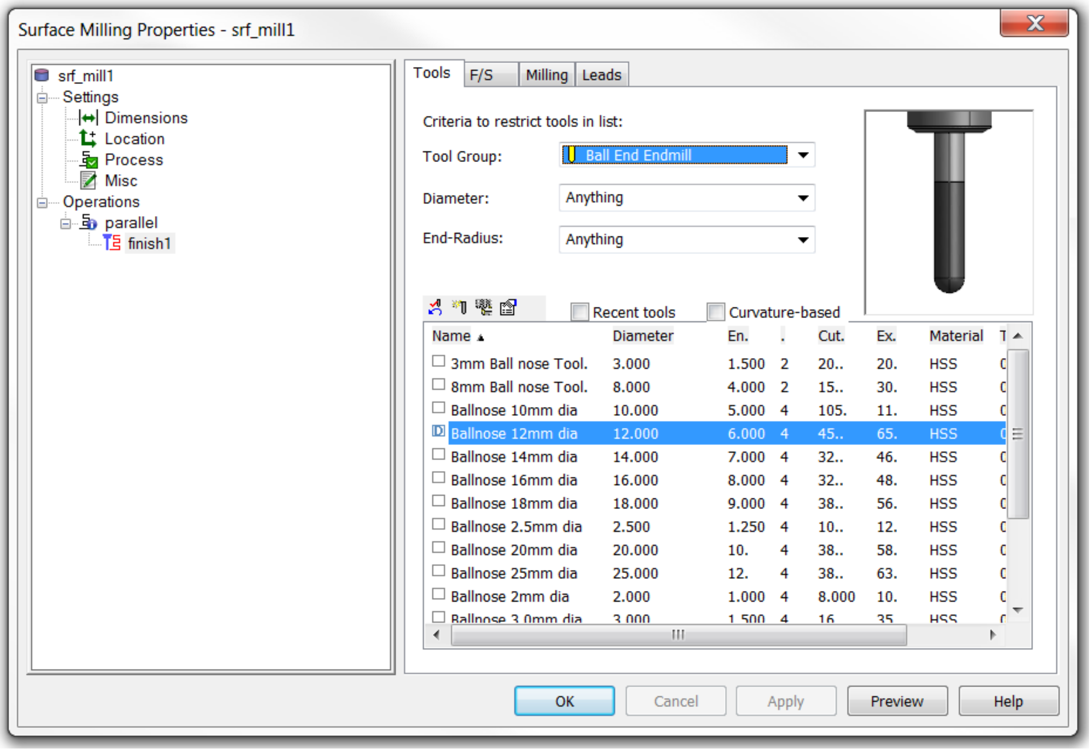
The **Slopes** tab is not available for **Z-level roughing**, **Plunge roughing**, **Isoline finishing**, **Flowline finishing**, **horizontal + vertical** and **Swarf**. Strategies.

Surface Control — Use this tab to exclude feature surfaces for specific operations

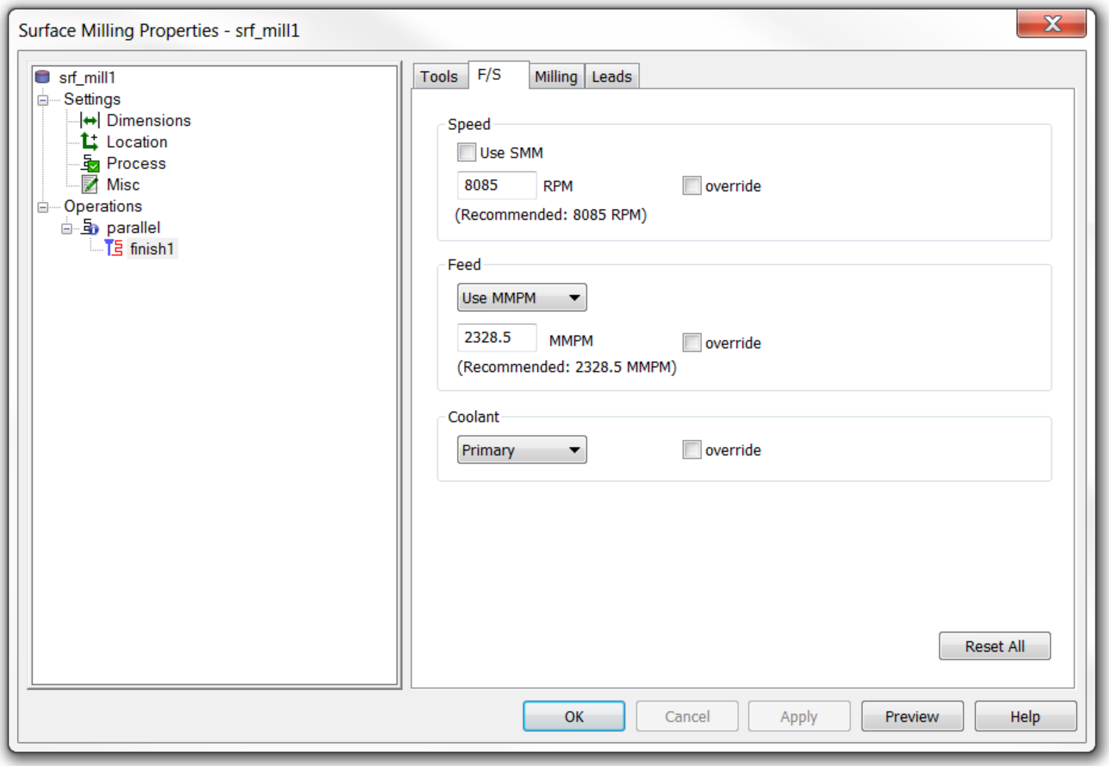
Operation-level tabs

The third level is the operation level. Select an operation in the tree view to access the following tabs:

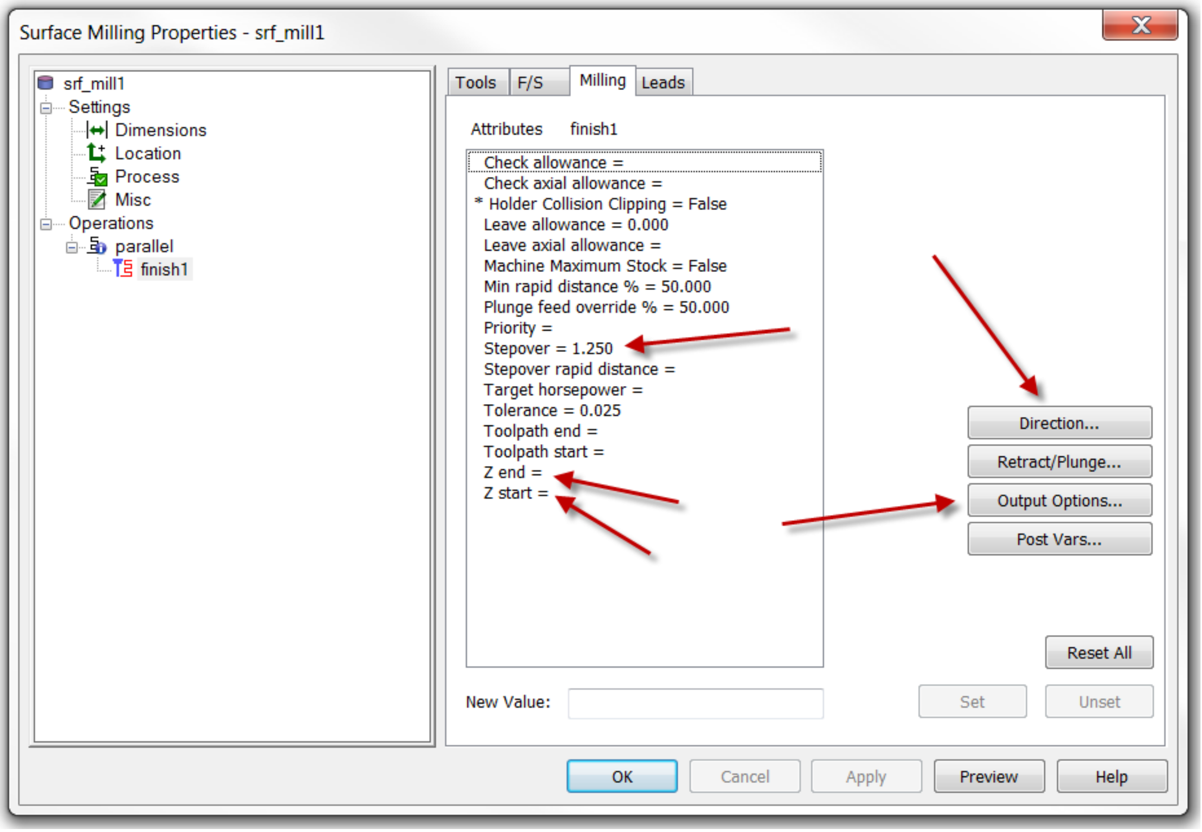
Tools — Use this tab to view selected tool or change to a different one.



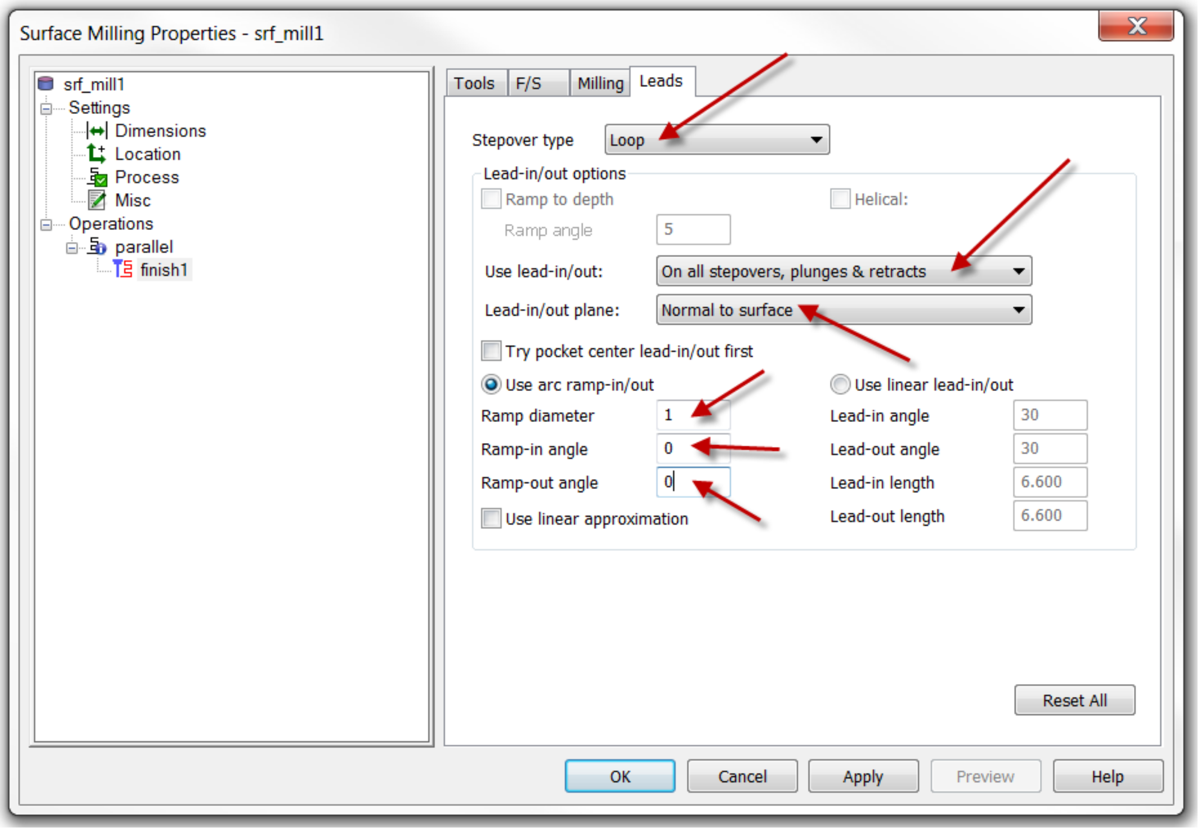
F/S — Use this tab to view automatically calculated feed or speed or change feed or speeds



Milling — Use this tab to set operation-level attributes.



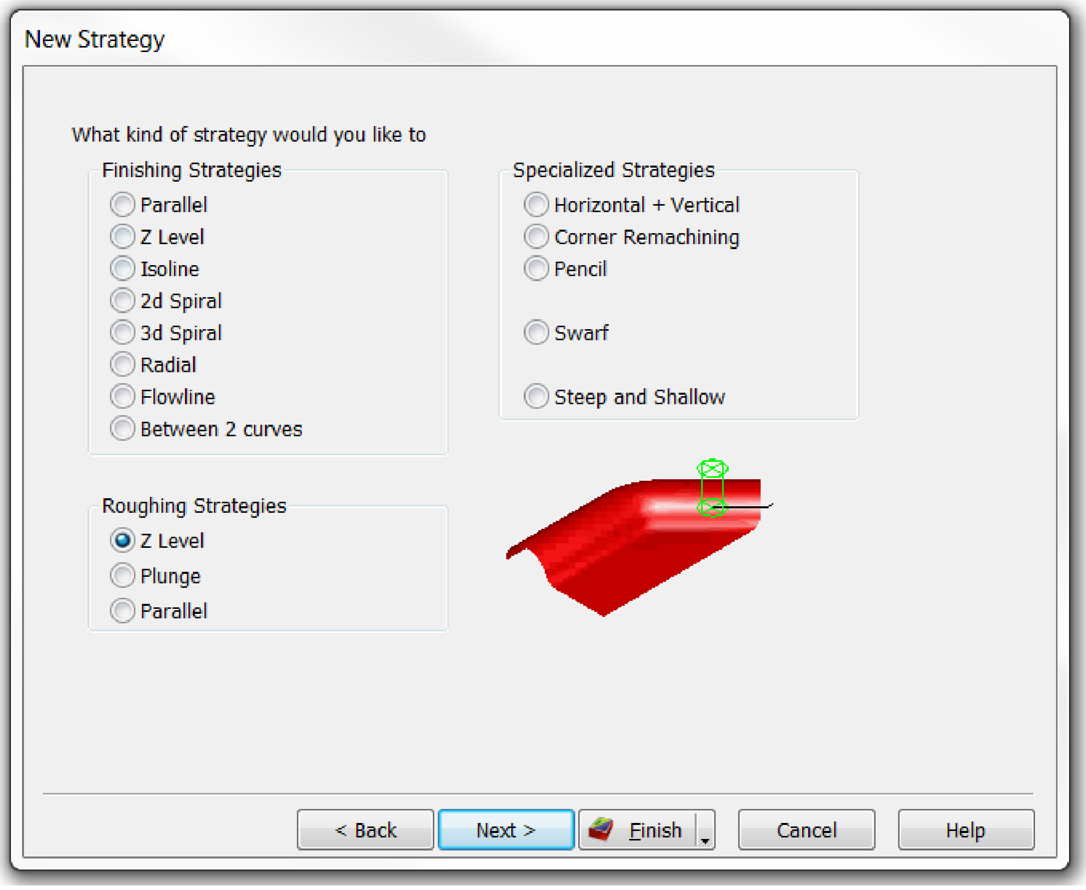
Leads — Use this tab to control leads and ramps.



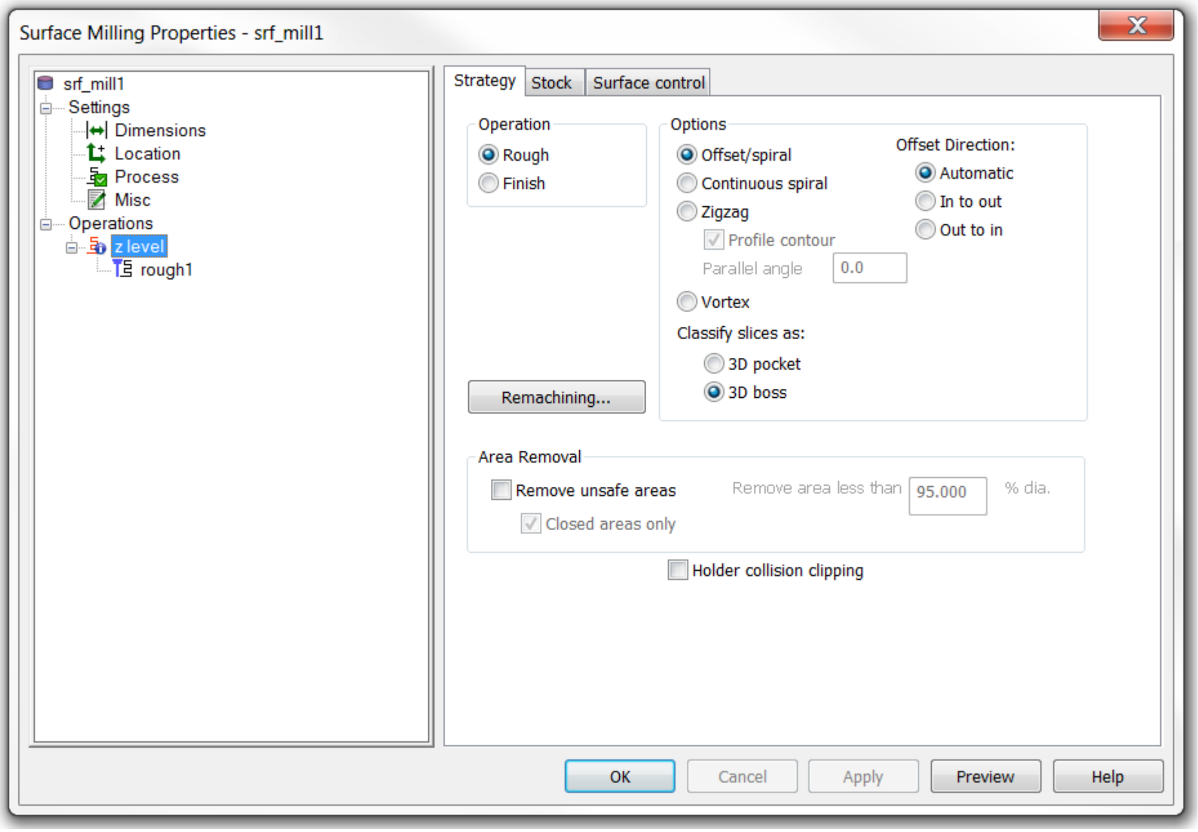
Roughing Strategies

FeatureCAM *has a number of different methods for Rough 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 machine a part for finish machining. In the roughing course module we shall look at the three main Roughing toolpath types in* ***FeatureCAM****. These are* ***Z Level Roughing****,* ***Plunge Roughing*** *and* ***Parallel Roughing****. We shall also consider different stock types and finally take a look at high speed and high efficiency Roughing techniques.*

Z Level Roughing



Z Level Roughing *– This is commonly known as waterline machining. In* ***Z Level*** *and* ***Parallel Roughing*** *a series of horizontal slices is taken through the part. At each level the profile of the part and the outline of the stock define the area of material to be the chosen strategy and tooling.*

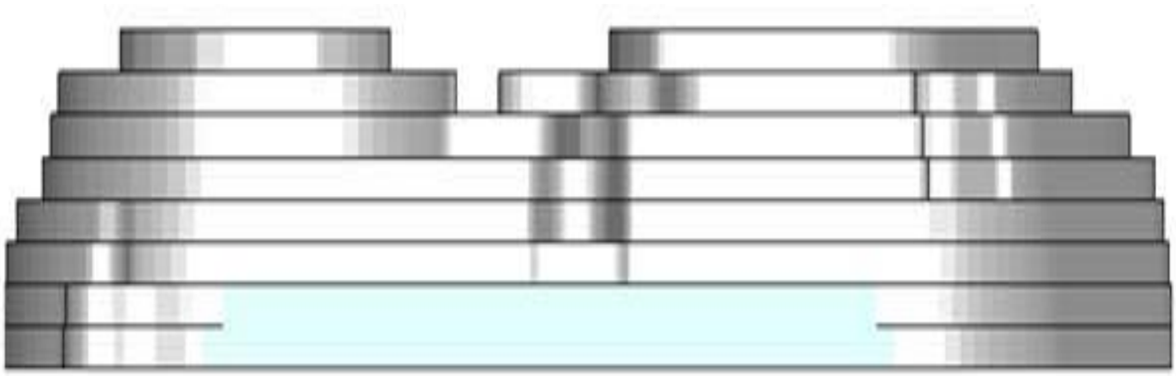


- You can rough your part as a **3D Pocket**, or as a **3D Boss**. If you select **3D Pocket**, the tool plunges or ramps onto the part and cuts from the inside of the part out toward the boundary of the part. A **3D Boss** typically plunges off the part and cuts from the outside toward the centre.
- Even if the surfaces of your part create a boss shape (meaning it protrudes instead of being a cavity); you can still select **3D Pocket**.
- Specify the style of toolpath. If you select **Zig-zag**, this roughs the part raster-style with an optional profile around each Z-slice (optionally select the Profile contour option).
- The value can be anywhere from -360 to 360 degrees, the default is 0.0. A positive value rotates counter-clockwise from the principle axis, and a negative value rotates clockwise from the axis.
- Setting the angle to 90 on an X-parallel operation causes it to effectively become a Y-parallel operation.
- Setting the angle to 180 causes the toolpaths to be cut from the opposite side of the part. For example, an X-parallel operation with the angle set to 0 starts at the minimum Y coordinate. With the angle set to 180, the toolpaths start at the maximum Y coordinate.
- **Continuous spiral** - This option enables the tool to move in a continuous smooth spiral motion and reduces tool load.
- **Vortex** (3D HSM) - An offset toolpath, which is machined at the specified cutting feed rate almost all of the time. The optimum tool engagement angle is never exceeded, by replacing difficult toolpath segments with trochoids. This works well for solid carbide tools.

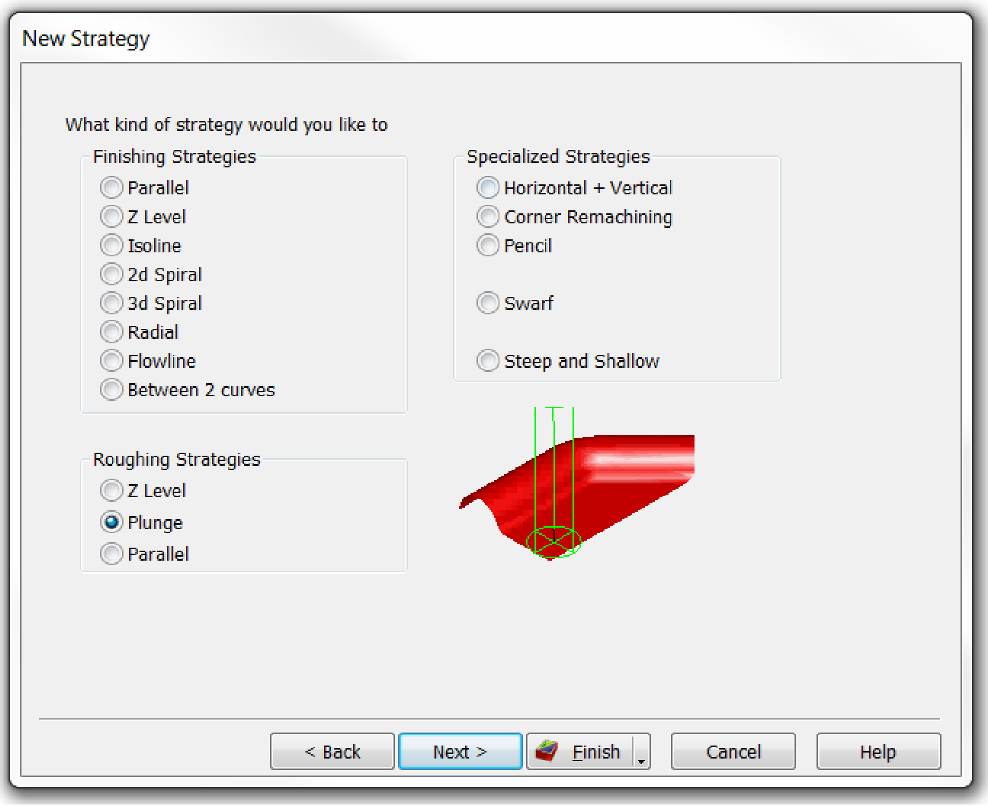
Classify slices as

Offset Direction options.

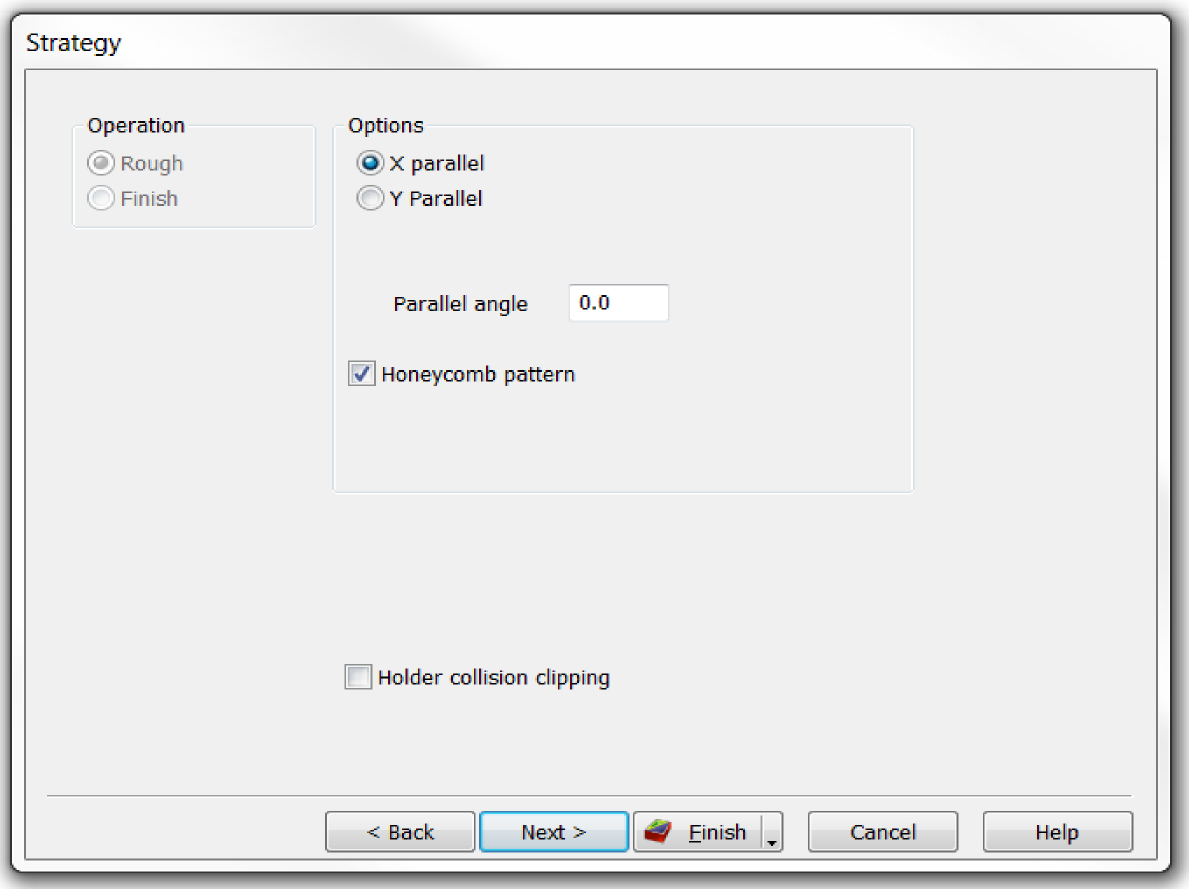
- **Automatic -** The tool cuts from the outside to the inside of the stock in a continuous radial movement. This is the default option.
- **In to out -** The tool plunges into the stock and cuts outwards.
- **Out to in -** The tool cuts from the outside to the inside of the stock. The tool outlines the surfaces and cuts large sections last.
- **Remachining** — click this button to open the Remachining dialog.
- **Area Removal (3D HSM)**
- Remove unsafe areas - This removes small toolpath segments to prevent tool damage when using non-centre cutting tools. When machining into small pockets, removing small segments stops the central, non-cutting underside of the tool from hitting non-machinable material. Unsafe segment removal filters out the machining of confined areas with small movement of the cutting tool.
- Remove area less than -This removes segments that are smaller than the entered percentage of the tool's diameter, unless they surround a Boss feature.
- Closed areas only - Select this option to remove segments, in enclosed areas, that are smaller than the Threshold value.



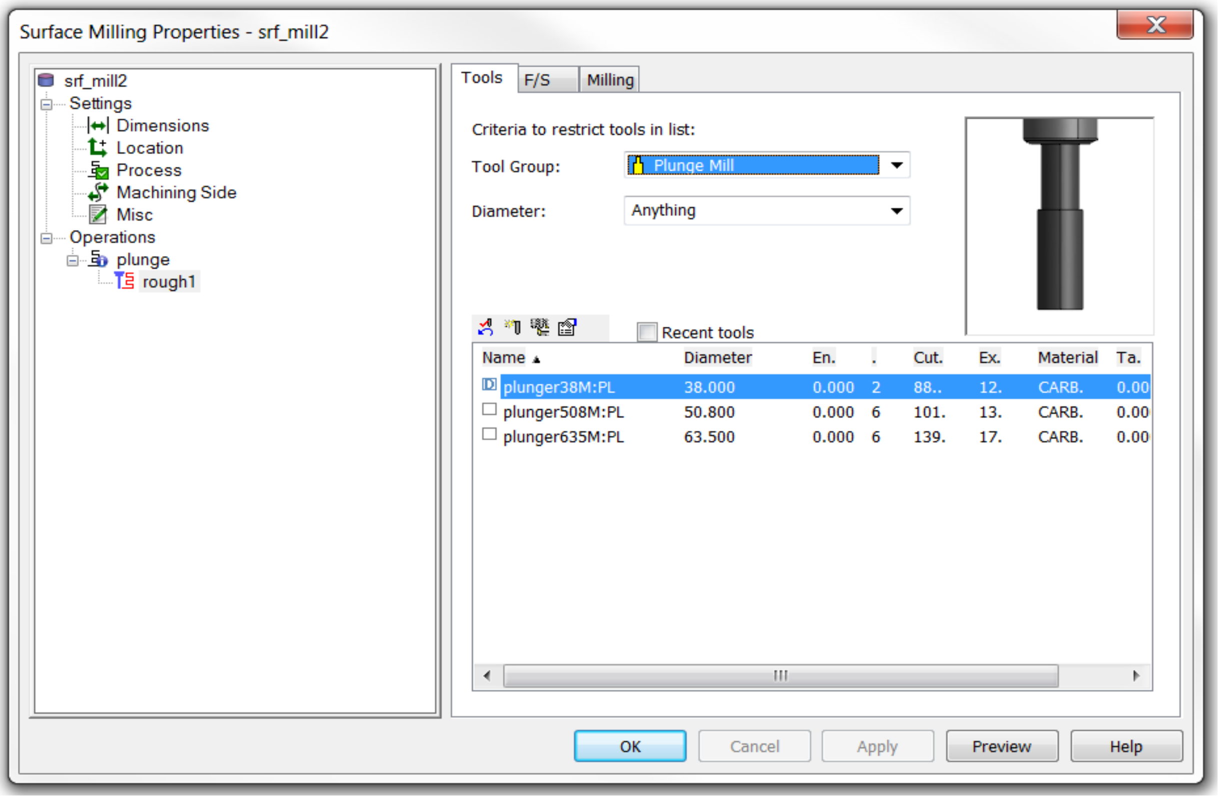
Plunge Roughing



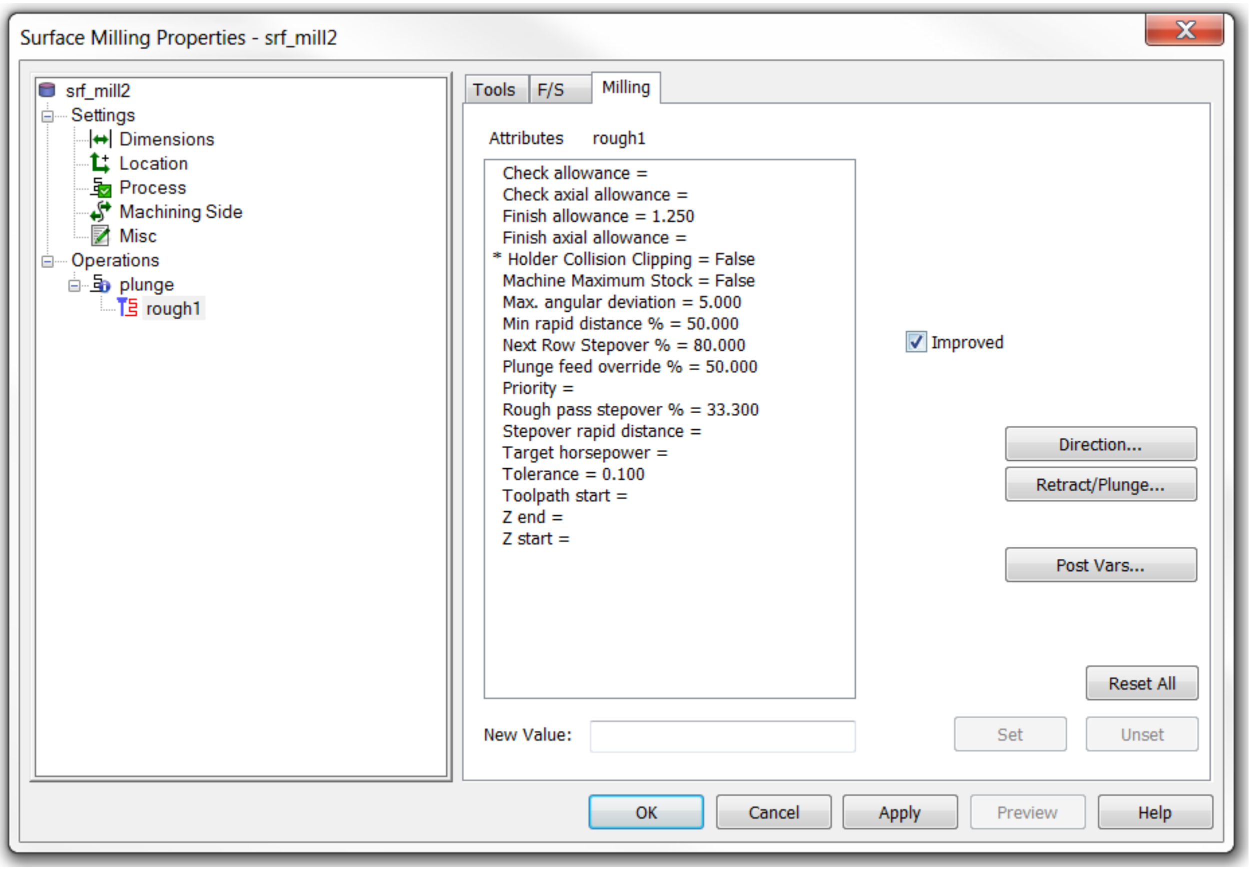
1 Select **Next** to show the next menu.



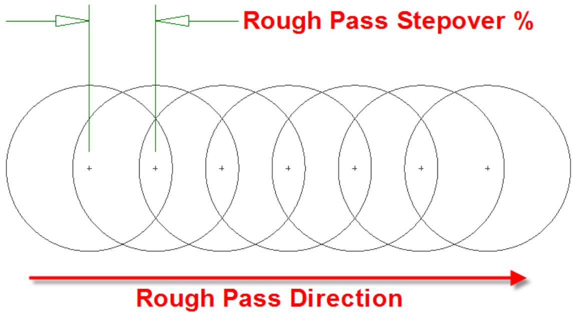
2 Select **Next** to show the menu below.



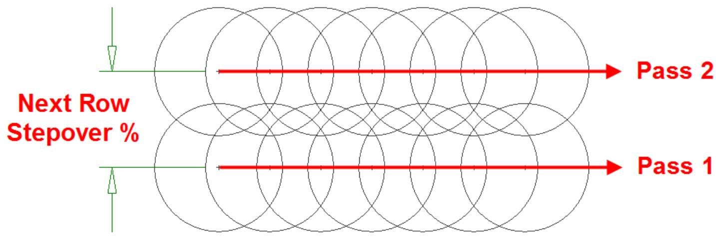
Milling Menu



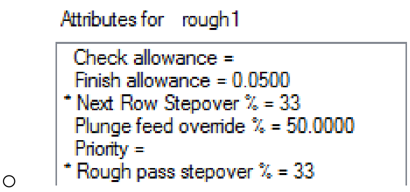
- The tool rapidly removes the material from the part by a series of drilling operations along parallel lines in the **X direction**. The distance between the parallel passes and the shape of the part determines the amount of material left on the part after plunge roughing. We will now make the distance between the passes smaller to reduce the steps in the remaining stock.
- There are two attributes which control the way that the plunge roughing tool steps over along a single roughing pass and between each pass. We shall now look at the effect of each.
- The first is the **Rough pass stepover %.**
- This sets the spacing between each drilling operation along a roughing pass. The stepover is a percentage of the tool diameter.



- The second attribute is the **Next row stepover %** this is the distance between each pass. The default value is 80%; with our 50mm diameter plunge rougher this is giving a stepover of 40.6mm leaving large steps.



- When setting these two attributes, the user should take care to ensure that only the cutting portion of the tool is plunging into the stock. Where an insert tool is not centre cutting it is very important to ensure that this will not be plunged into the stock. Failure to do this can result in damage to the tool, the part, and also the machine tool spindle.
- Set the **Next row stepover** and **Rough pass stepover** both to **33%**.

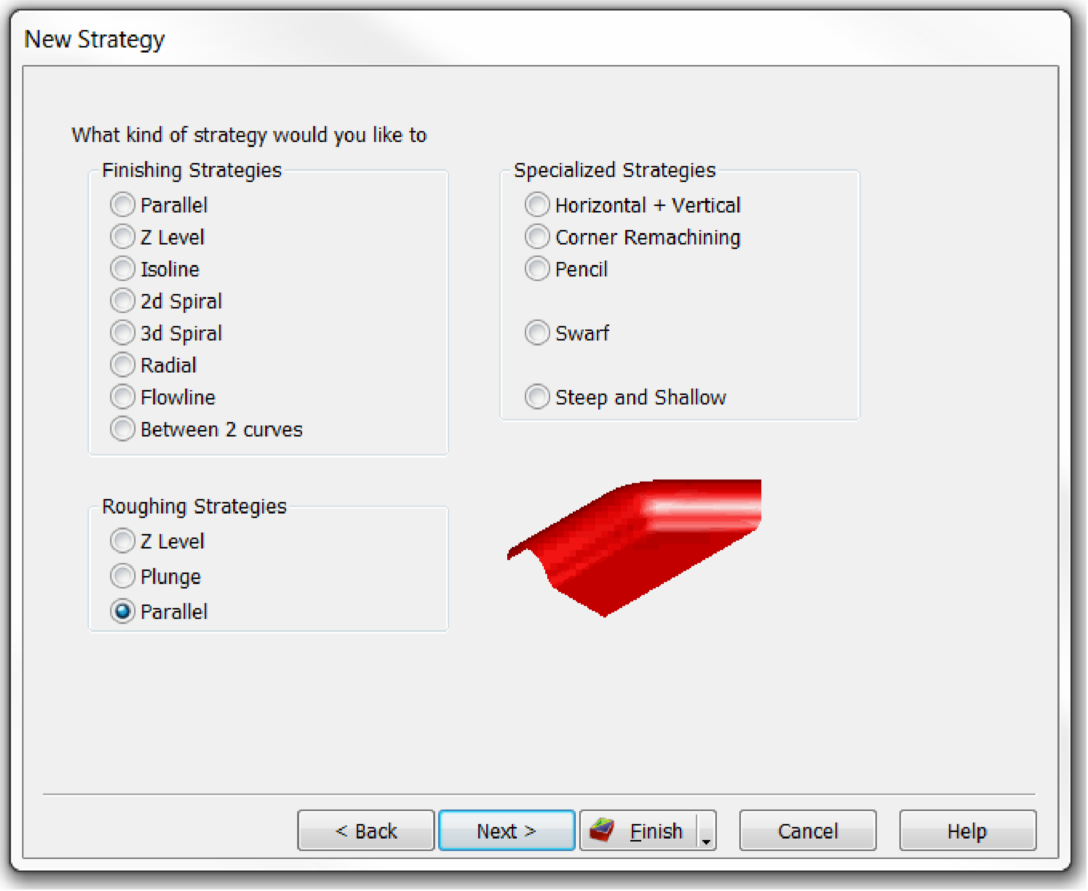


3 Set the **Z start** to **-2.5mm.**

- By putting the Z Start below the top of the material we are ensuring that we do not waste time by plunging into material at the top of the part that is very thin.

- **Plunge Roughing** provides an alternative form of roughing. With this technique, roughing is performed with a series of overlapping holes. The advantage of using this technique is that parts can be roughed quickly because the force of the operation is directly up the spindle.
- This figure shows a sphere that is being plunge roughed with a flat end tool.
- The pattern of drilling operations is parallel to the X or Y axes just like parallel milling. Neighbouring rows of the pattern can be offset horizontally to better cover the part.
- Plunge roughing is performed in either a straight rectangular pattern, as shown below:
- or a honeycomb pattern in which each row is offset horizontally by half the rough pass stepover amount:
- The honeycomb pattern is usually preferable. The Honeycomb pattern attribute controls the pattern type.
- The Rough pass stepover % attribute controls how far the holes in the same row are spaced. It is specified as a percentage of the tool's diameter.

Parallel Roughing



Introduction

Parallel roughing is similar to **Z Level Zigzag** in that the toolpaths are parallel to an axis, either X, Y or at an arbitrary angle. Unlike Zigzag the user does not have the option to make a profiling pass at each level. Instead of profile passes the strategy makes a 3D Parallel semi-finish toolpath over the whole part to remove the cusps left by the initial roughing operation. Due to the nature of the toolpath the default tool chosen is a ball nosed cutter. We will now rough machine the engine casing using a parallel roughing strategy.

1 Click on **Features** in the Step view or select Ctrl + R

2 Check Surface Milling and press **Next** twice.

3 On the New strategy form check **Choose a single operation**.

4 Click **Next**.

5 Check **Parallel** in the Roughing Strategies part of the form.

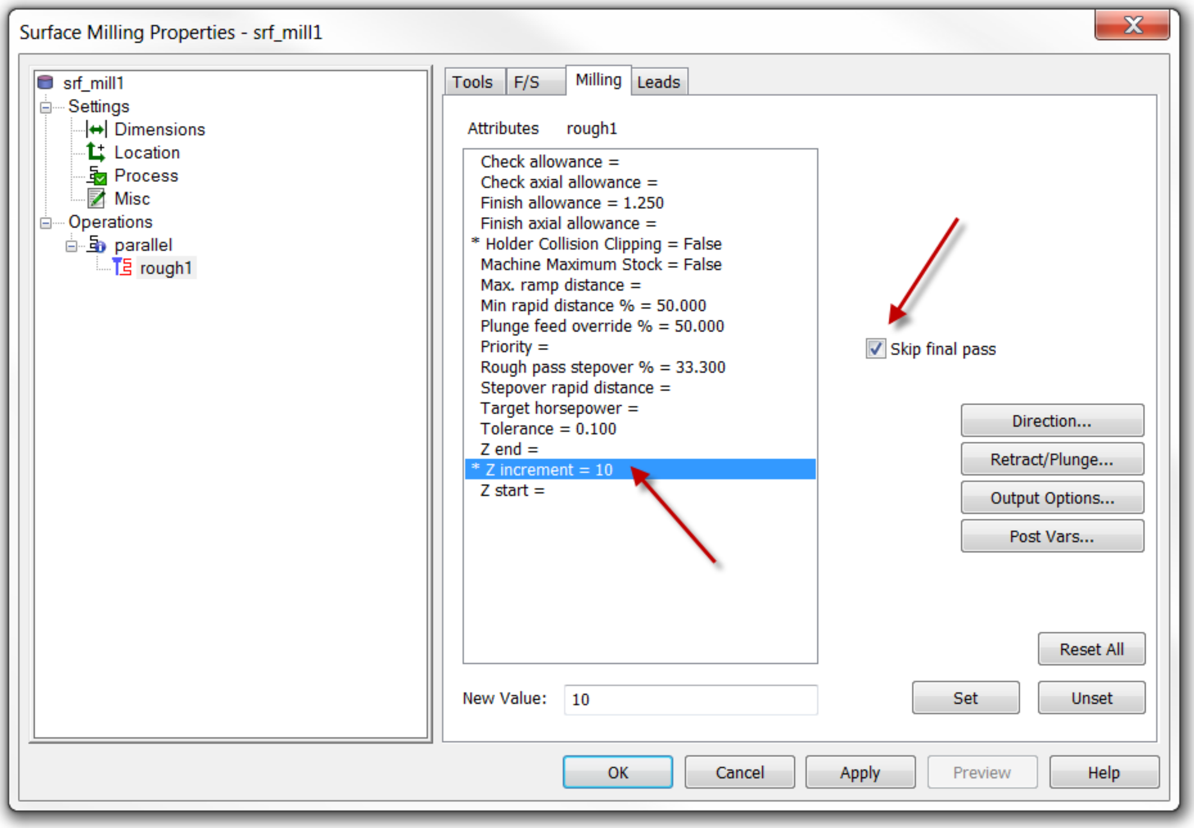
6 Press **Finish**.

7 Edit the feature by double clicking on it in the **Part View** window.

8 Click on **rough1** in the tree view.

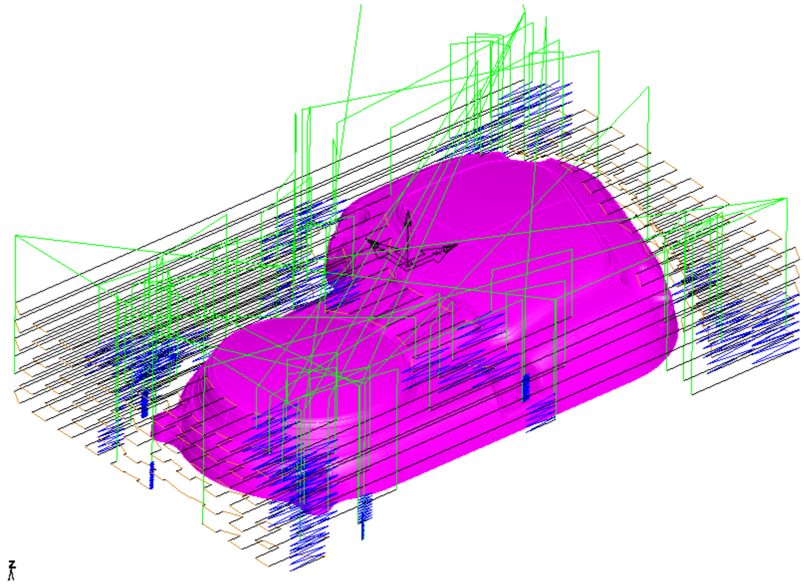
9 Select the **Milling** tab and set the **Z increment** to **10mm** as below.

10 Check **Skip final Pass**, click **Apply** then **OK**.



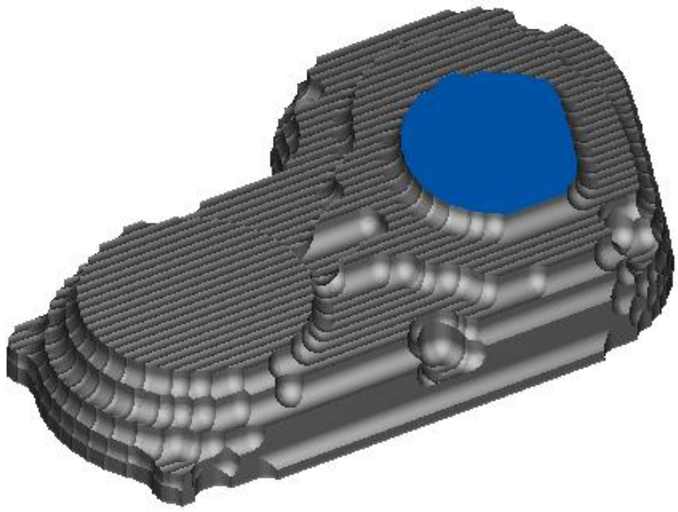
11 Click on the Tools tab and select a **25mm Ball Endmill**.

12 Run a **Centreline simulation**.



The tool path runs parallel with the x-axis at the Z-increments specified. Note the ramping approach moves, these are needed where a non-centre cutting tool is approaching the stock from above. We will deal with ramp moves in detail further on in these notes.

13 Run a **3D Rapid Cut Simulation** 



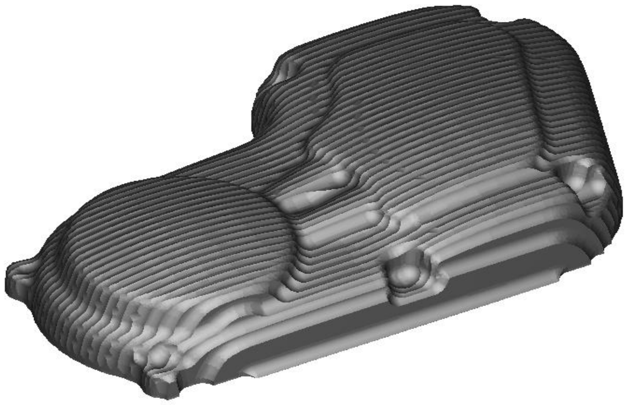
Note the large cusps left between each pass along the X axis. We will now reduce these by adding a semi-finish pass.

14 Click on **rough1** in the tree view.

15 Select the **Milling** tab and uncheck **Skip final pass**.

16 Click **Apply** and then **OK**.

17 Run a **3D Rapid Cut** simulation 

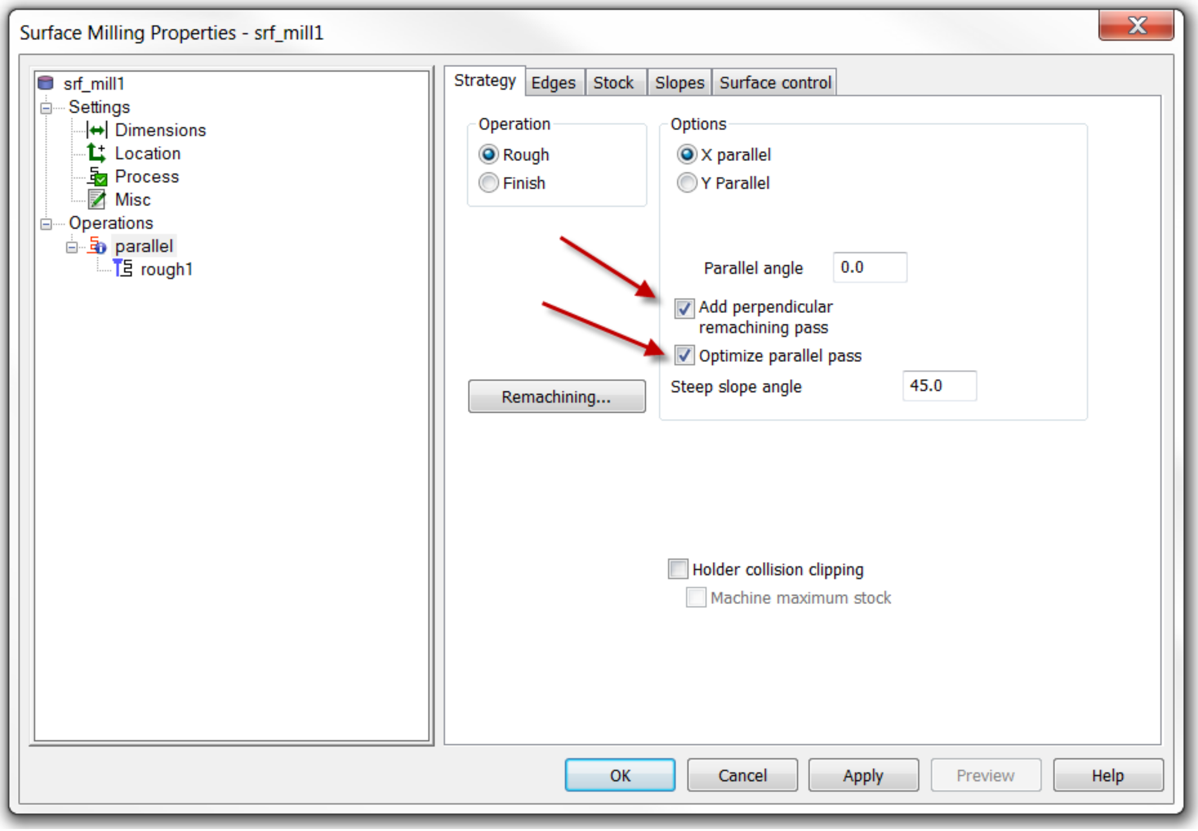


The tool path now has a final pass that is parallel to the x axis that changes in z according to the contour of the part. Note how the very top of the part is not machined. In order to ensure that the semi-finish pass goes all the way across the part we need to put additional stock on top to a thickness greater than the ***Finish Allowance****. You will also notice that where the part forms is steep in a direction perpendicular to the cutting direction the stepover leave large cusps. We will now fix these two problems.*

18 Edit the feature, select **parallel** in the tree view.

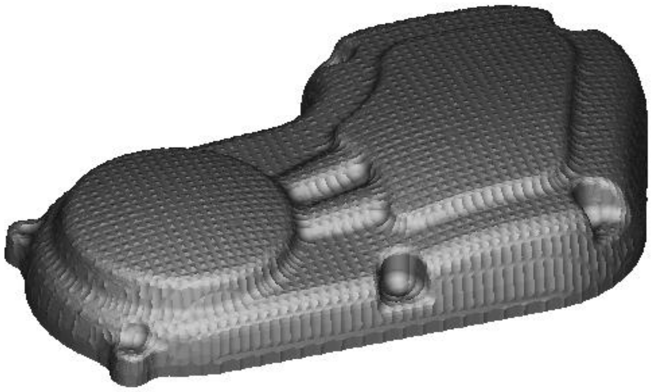
19 Check Add perpendicular re-machining pass.

20 Set the **Steep slope angle** to **Zero** degrees.



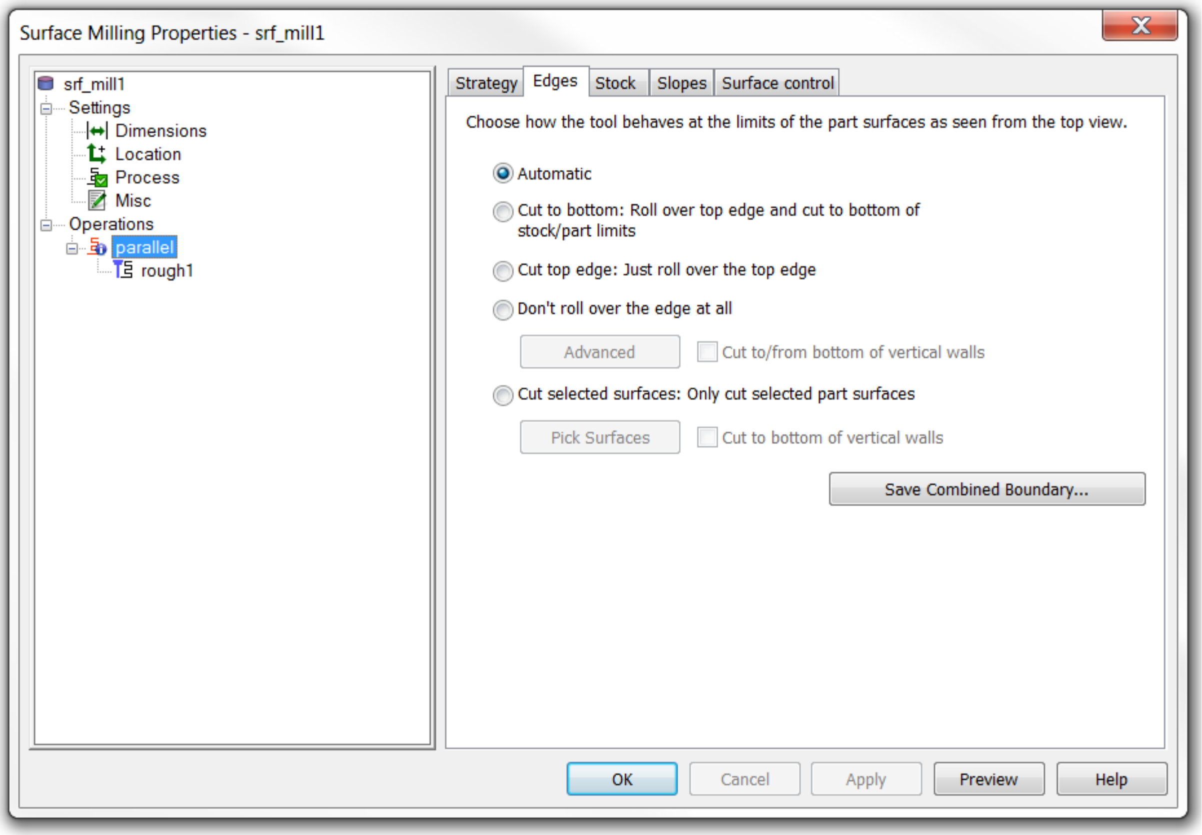
21 Click **Apply** and then **OK**.

22 Run a **3D Rapid Cut simulation**.



*By setting the steep slope angle to zero we are forcing **FeatureCAM** to semi-finish in two passes one along the X direction and the second along the Y across the whole part. This technique can be useful when cutting very large parts with a ball nose cutter. If a larger Steep slope angle is sent then only that portion of the part that is steeper that the specified angle perpendicular to the X direction will be re-machined.*

Edges



This tab gives you options to set how the toolpaths are generated near surface edges.

***Automatic** — a set of rules that chooses between **don't roll** and **Cut to bottom** to get the best performance.*

- **Cut to bottom: Roll over top edge and cut to bottom of stock/part limits** — this does not set any boundary. In this case, **FeatureCAM** uses only the boundary specified by the curve options in the **Stock** tab.
- **Cut top edge: Just roll over the top edge** — Sets a pocket-like curve boundary to the silhouette of the part offset by a tool radius. The tool rolls over the edges by a tool radius. In PowerMILL this is known as a silhouette boundary. Edge boundary uses a silhouette boundary of the part, but it uses the top (highest) Z values where the silhouettes are at a vertical surface.
- **Don't roll over the edge at all**
- Sets a pocket-like curve boundary to the silhouette of the part. The tool contacts the surface boundary. In **PowerMILL** this is known as a contact point boundary. This prevents any cutting of sharp corners at the external edges of the part. If the part has a vertical surface at the outside of the part and has a sharp corner at the top of the vertical surface, it prevents cutting of the vertical surface because it is coincident with the sharp corner at the top.
- You can set advanced options on the part boundary by clicking the advanced button.
- If you have a vertical surface and you want to cut to the bottom, select **cut to/from bottom of vertical walls**.
- **Cut selected surfaces: Only cut selected part surfaces** — **FeatureCAM** only machines the selected surfaces and does not cut any of the neighbouring or unselected surfaces. If there are no neighbouring surfaces, the tool rolls over the edges by a tool radius.
- If you have a vertical surface and you want to cut to the bottom, select **cut to bottom of vertical walls**.
- **Save Combined Boundary** — this preview button shows both the part boundary (in blue) and the tool centre (in red) curves on your model and enables you to save it for future use if you want to.

Contents

Stock Types and Boundaries

Introduction

So far in this module we have only considered roughing from a rectangular block of material. Often in practice we may have material that is round or multisided, has been pre-machined, or may be a fabrication or a casting. We will now examine the different options available for defining the type of stock we are using. The first one we will look at is Round Stock.

Close down all documents in **FeatureCAM**.

In Windows Explorer navigate to the Data folder.

Drag and Drop the file **Round.x_t** into the **FeatureCAM** graphics area.

Select a Metric Milling document and click OK.

Select "**Use the Wizard**" and Uncheck **Launch AFR**.

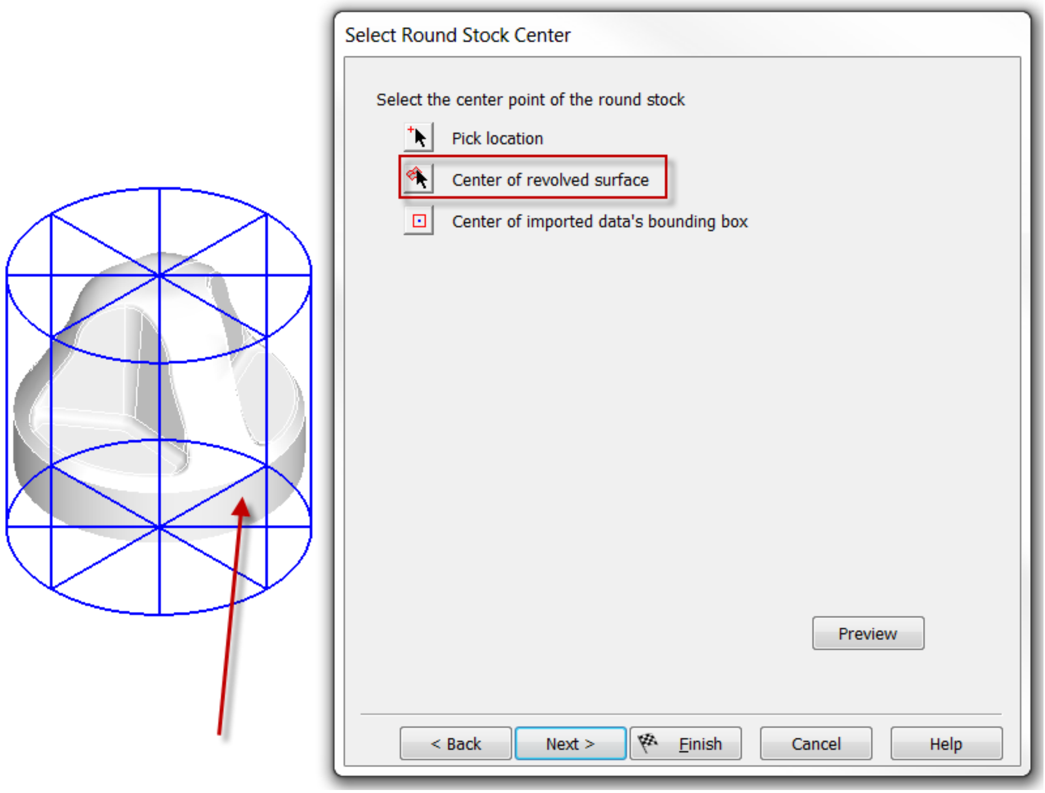
Click **Next** three times until you reach the **Stock Type** page.

Check **Round** and **Z** then click **Next**.

In this case our part is round and has its axis pointing up the Z axis. We now need to ensure that the round stock is centred on our part.

Click on **Centre of revolved surface** button.

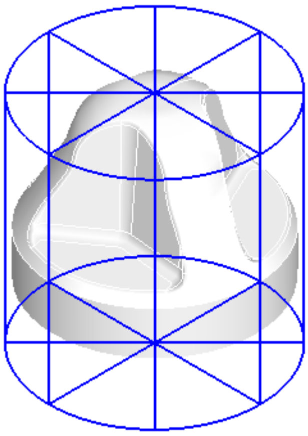
Click on the surface shown on the next page. **Shade** the solid if necessary.

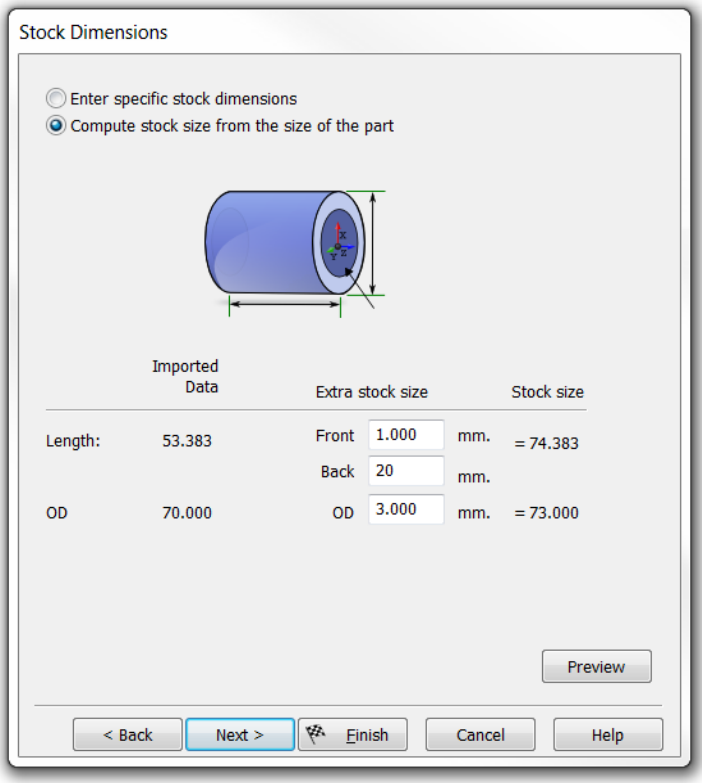


***FeatureCAM** finds the central axis of the selected surface and aligns the stock to it.*

Click **Next**.

Fill in the stock dimensions as shown and then click **Next**. The extra stock on the bottom is to hold in a chuck.





The next step is to position the setup at the required position. In this case we want it to be at the center of the upper end of the stock.

Click on the **Hand** to the **Right** of the stock on the form.

Click **Finish**.

The part is now correctly set up within the resized stock. We will now create a Z Level Offset roughing path to clear the excess material from around the part.

Click on **Features** in the **Steps** view.

Check **Surface Milling** and press **Next**.

Box select all of the surfaces and then press the **Add from selected items button** then click **Next**.

On the New strategy form check **Choose a single operation**.

Click **Next**.

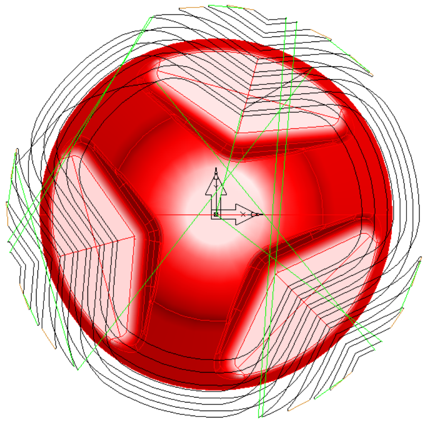
Check **Z Level** in the Roughing Strategies part of the form.

Click **Next**.

Fill in the Strategy form exactly as below and then press **Finish** and **OK**.

Run a **Centreline simulation**.

Change to a **Top View**.



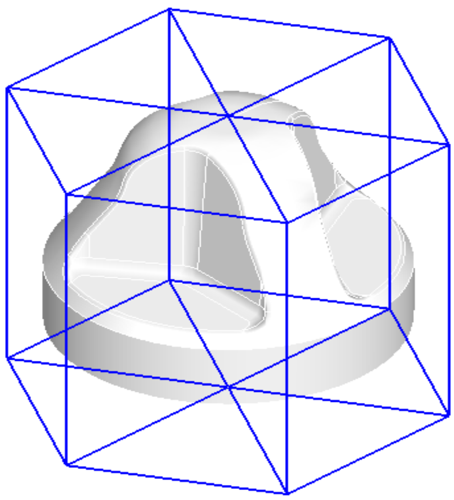
Note how the roughing passes are now confined to a cylindrical area around the part.

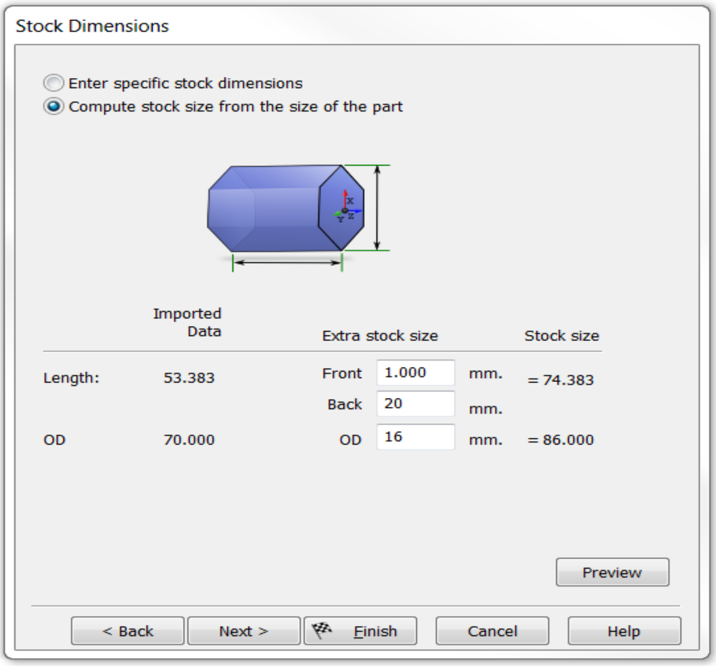
N-Sided Stock

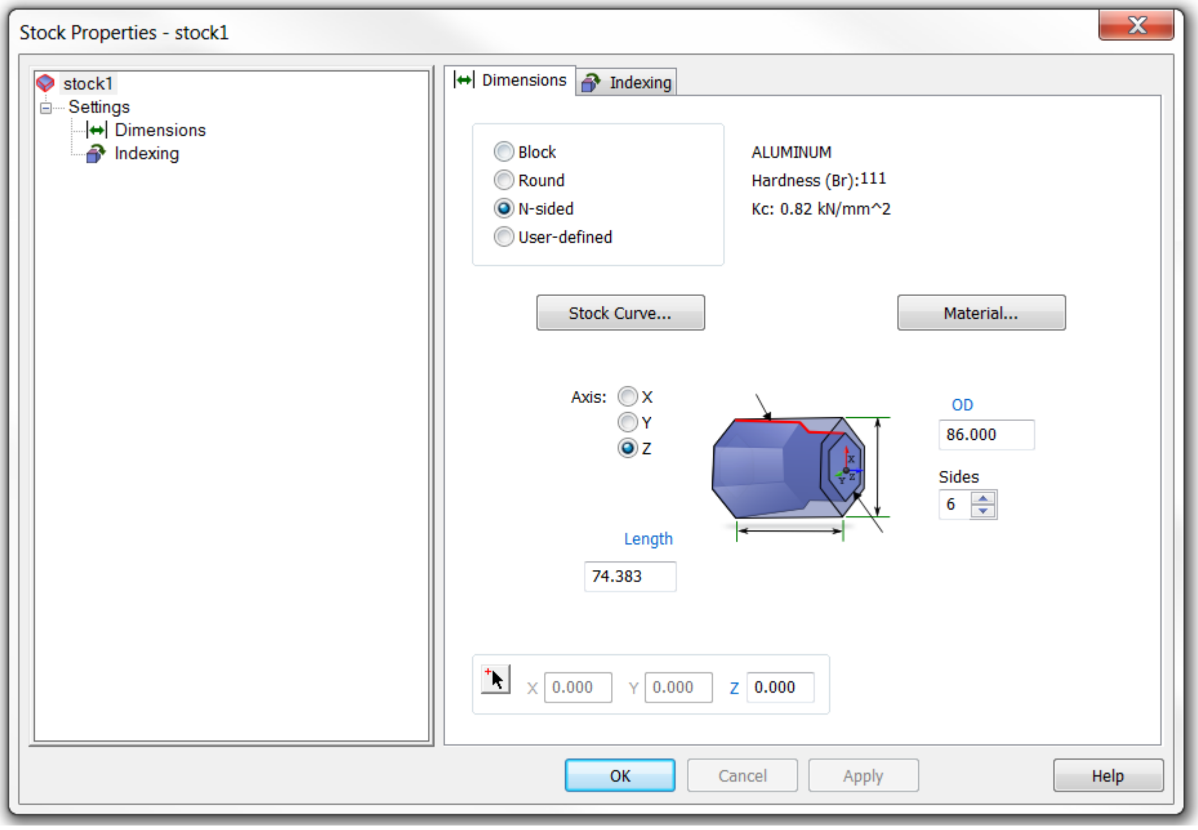
FeatureCAM can also create stock with a regular polygonal form such as hexagonal bar stock. The stock size is defined by its number of sides and outer diameter (circumscribing circle).

Double click on the **Stock** in **Part View** and fill in the properties as shown.

Apply and **OK**.

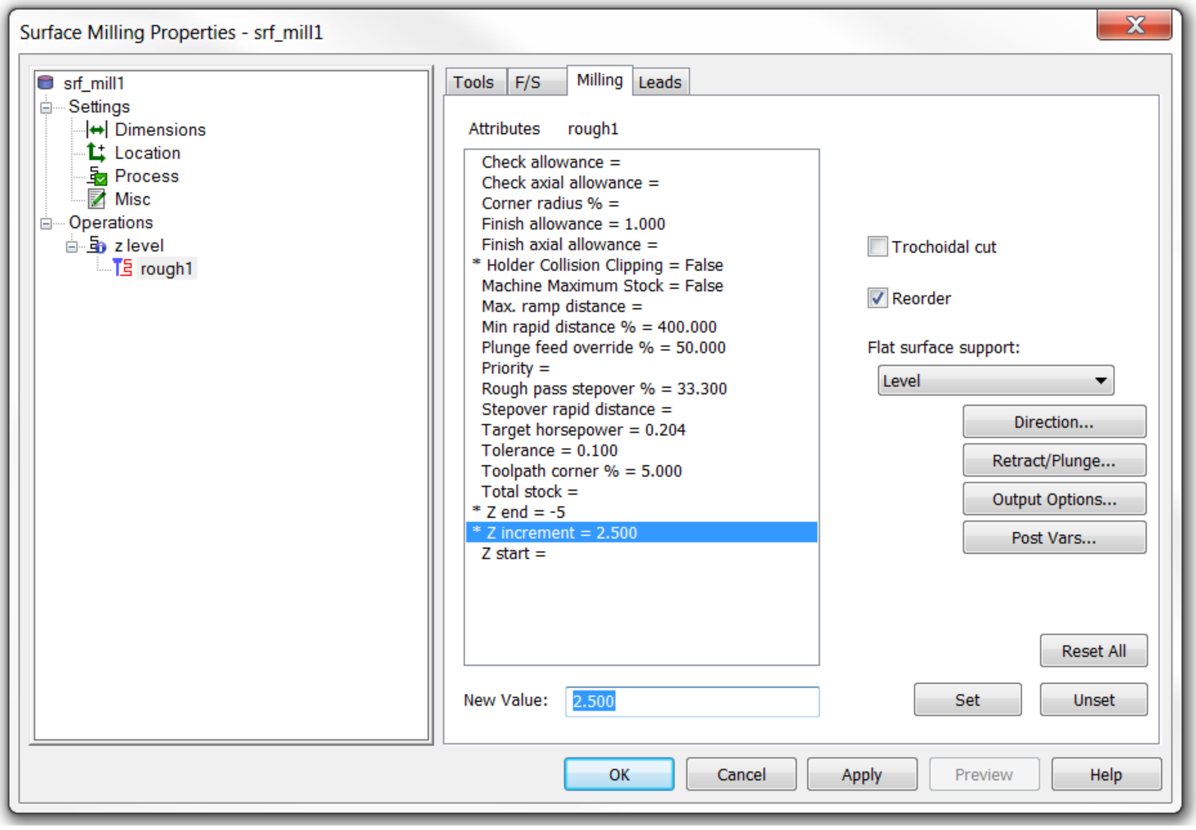






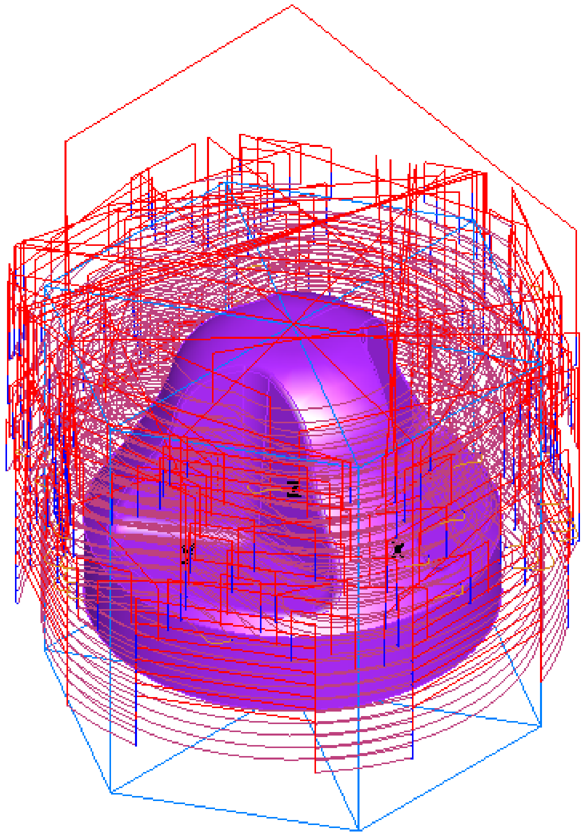
Edit the surface mill feature, select **rough1** and then the **Milling** tab.

Set the **Z increment** to **2.5mm** and **Z End** to **-5mm** as shown. **Apply** and **OK.**



Run a **Centreline simulation**.

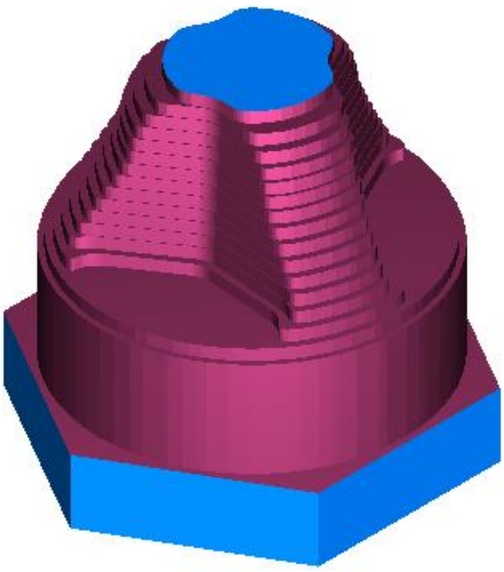
Change to a **Top View**.



The roughing operation is now confined to a hexagonal region around the part.

Run a **3D simulation**.

Change to as **Isometric View**.

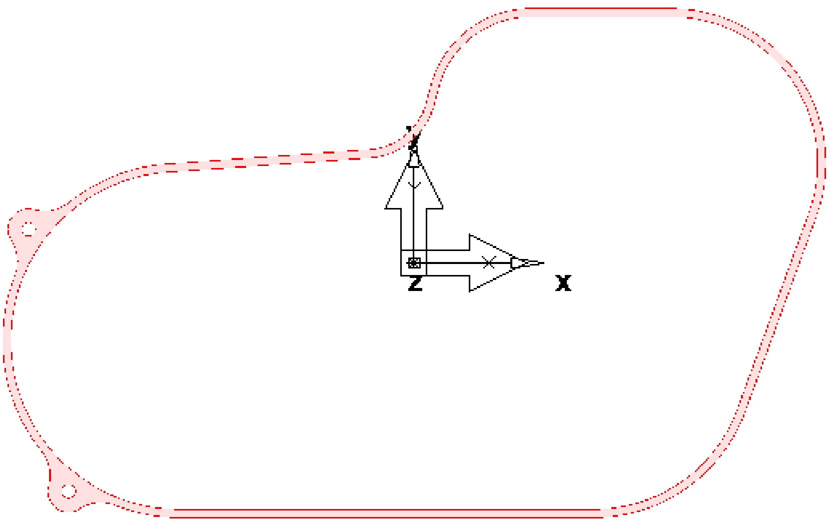


***Note:** how the hexagonal stock is accurately represented in the simulation.*

Stock Curve

Often a part has to be machined from a billet that has been pre-machined into an irregular prismatic form. For example the outline of the part may have been cut out using a band saw or machined using a 2.5D feature such as a Side or a Boss. It would be very inefficient in such a case to prepare a roughing toolpath based on a rectangular or round stock when most of the material has already been removed. **A Stock Curve** allows you to define the outline of the remaining material and then rough just within that area.

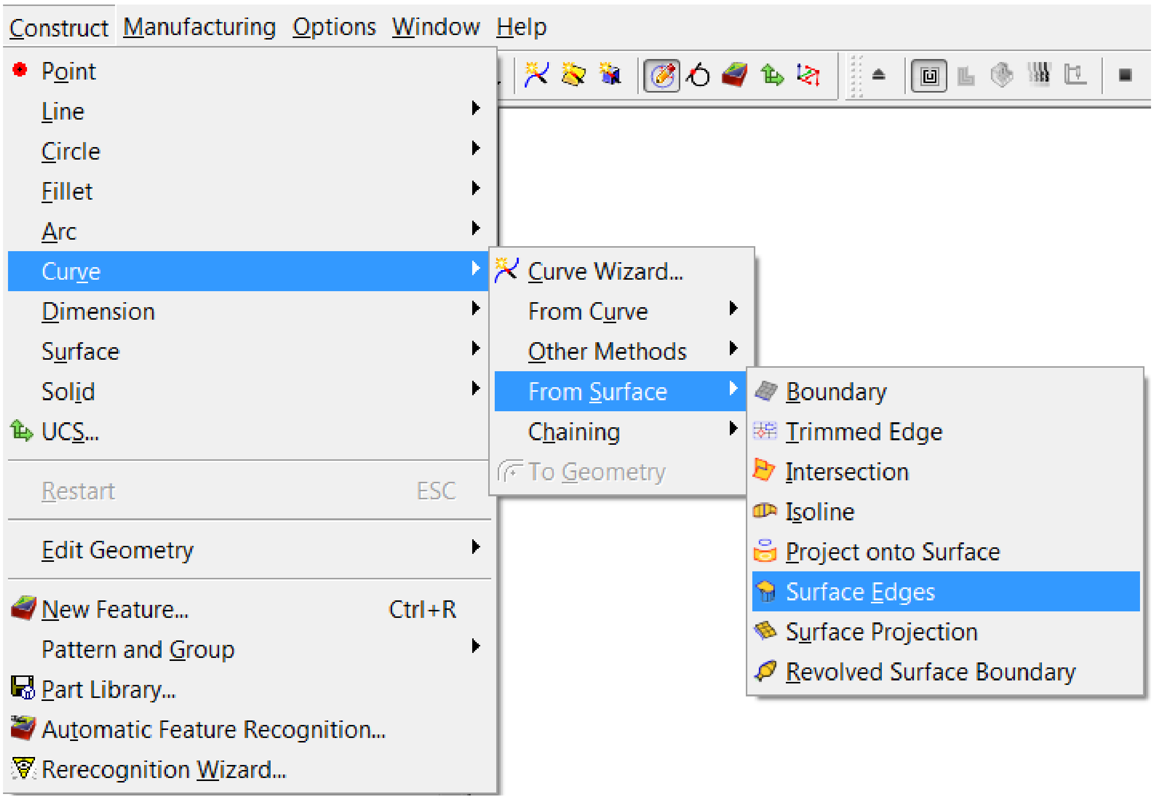
1. Import the part **Primary_Case.x_t**
2. Use the **import Wizard** to orientate the part and create **Setup1** and setup the **Stock** as described in the first chapter Roughing Strategies.
3. Select the bottom flat face of the solid model using your cursor.
4. Then select **Ctrl + K**. This hides everything apart from the selected surface.



We will create a Curve from the outer edges of this selected surface.

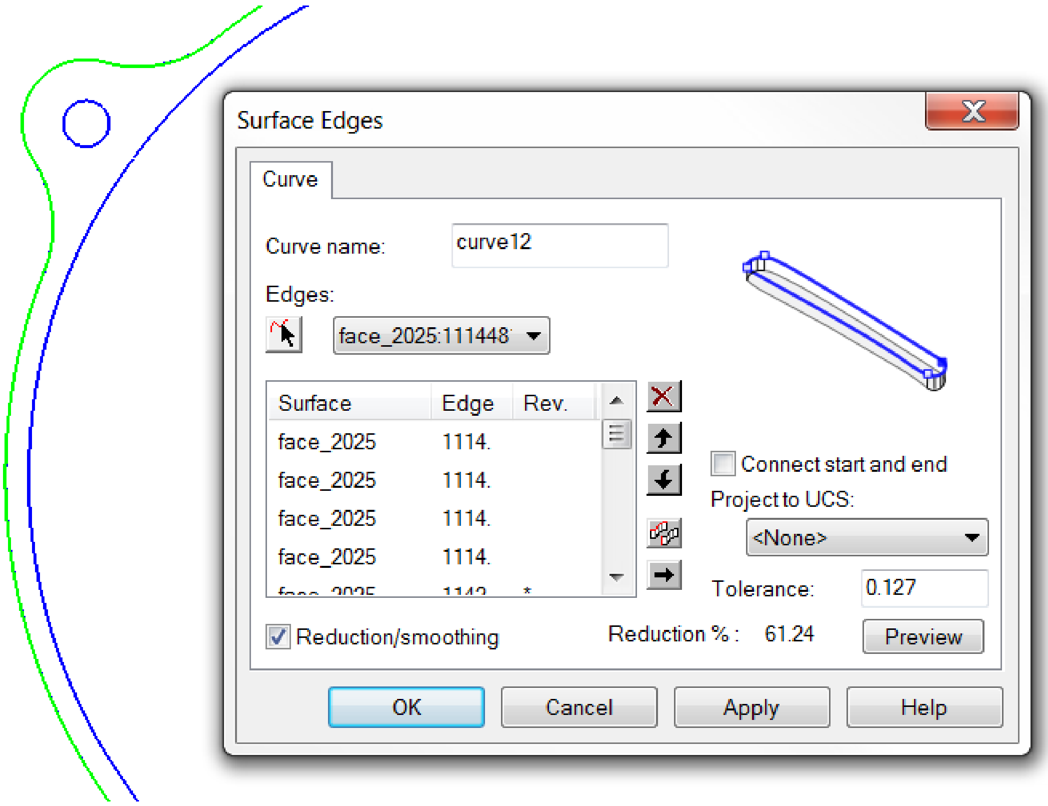
5. Select Construct>Curve>From Surface>Surface Edges.

See image on the next page.



6 Select the Pick Curve on surface Icon and double click on the outer shape

*Wait a few seconds and **FeatureCAM** will then create a **Curve** from the outer edge all the way around.*

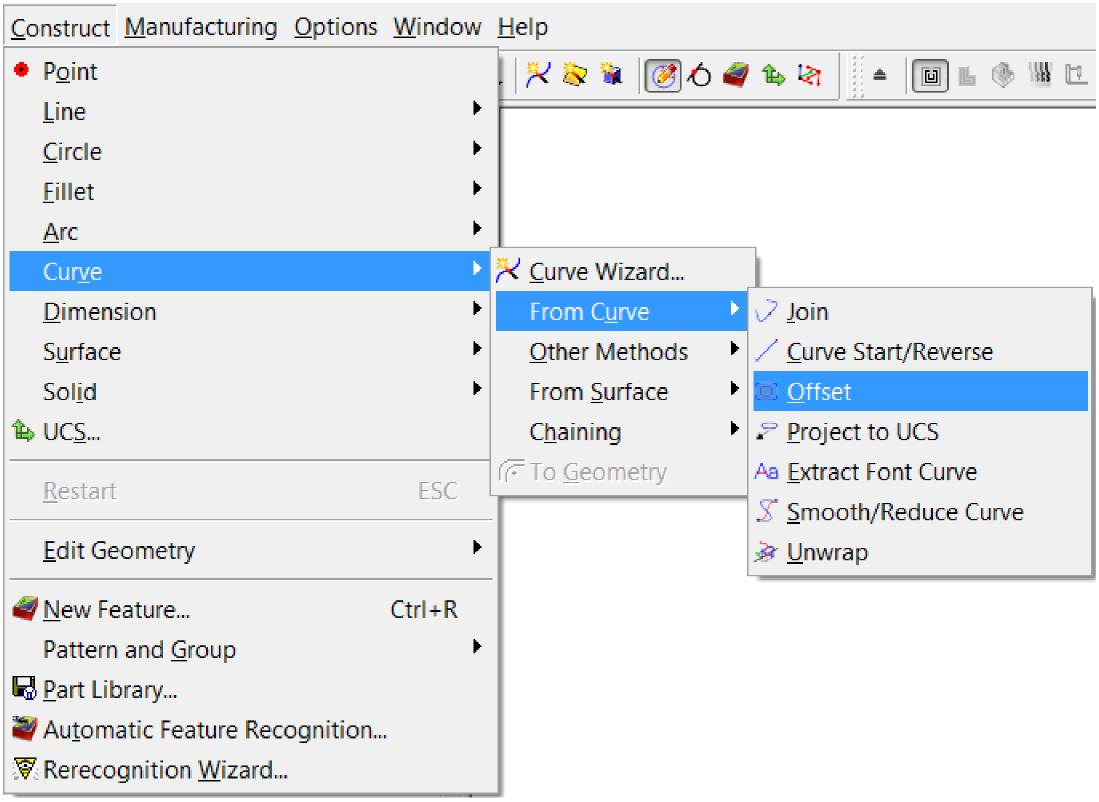


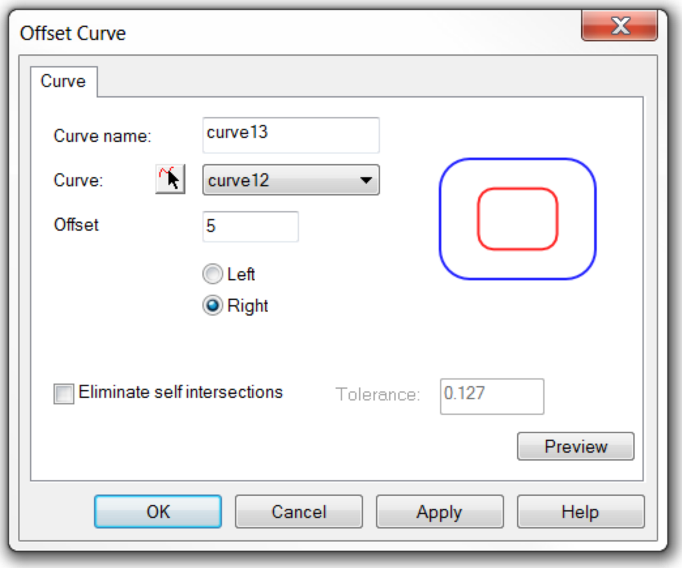
7 Click **Apply** and **OK**.

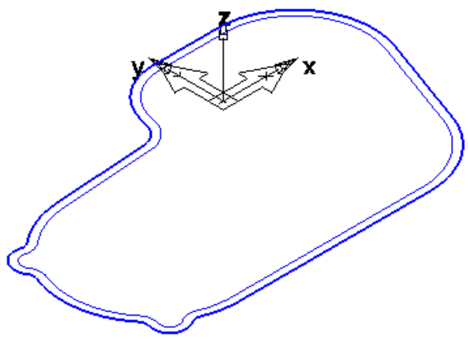
8 Change to an **Isometric View**.

9 Offset the Curve by 5mm.

10 Select Construct>Curve>From Curve>Offset.

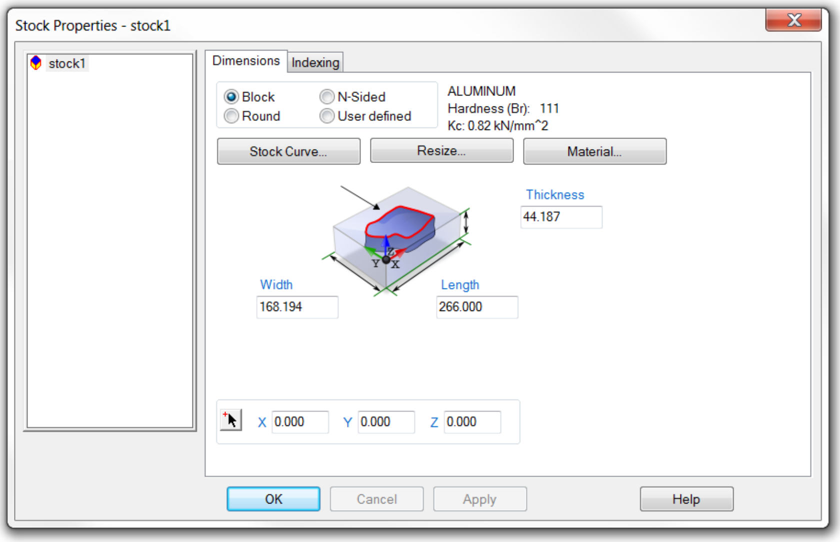


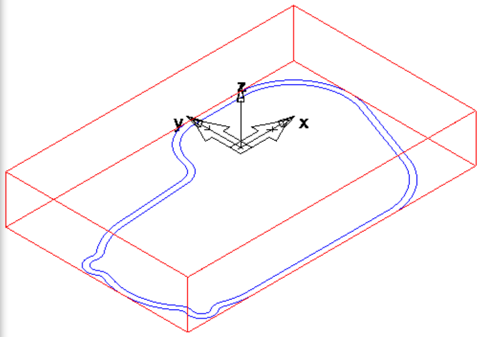




11 Double click on **Stock** in **PartView**.

This will show you the Stock Menu as shown on the next page.



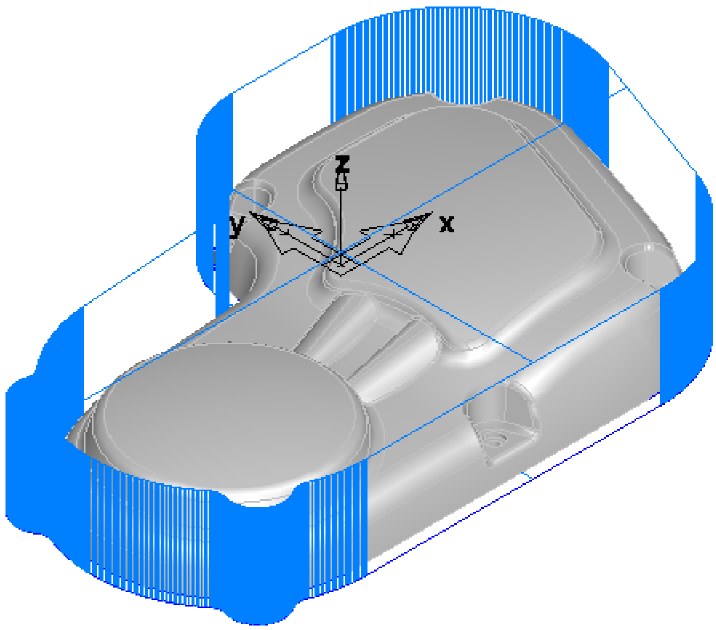


12 Select **Stock Curve** and pick the Curve you have just created using Offset.

13 Select **Apply** and **OK.**

14 In **PartView** select **Stock** and Right click then select **Show Stock**.

15 Or Select **Ctrl** + **L** this will **Show All**.



*You will see that **FeatureCAM** is now using the curve to define the stock as an extrusion of the outline of the remaining material. The thickness of the material defaults to the previous thickness of the rectangular block but can be changed if required.*

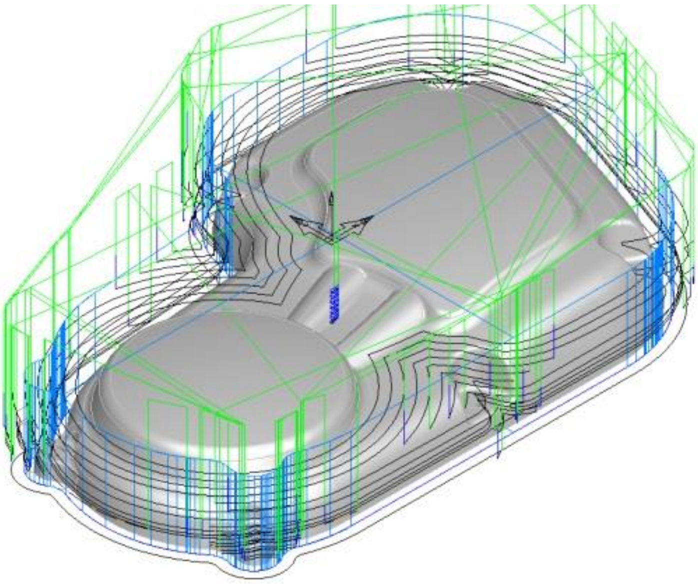
16 Box select all the surfaces.

17 Create a **Z Level Rough**. Use **Offset/Spiral** and **Boss** in the strategy.

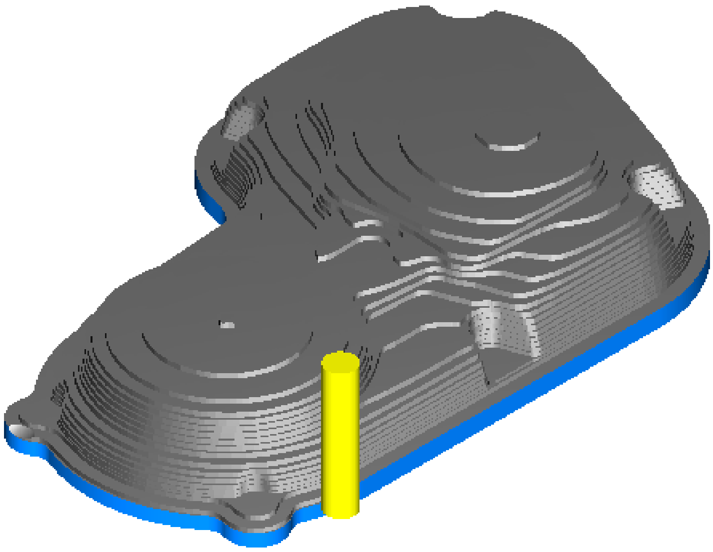
18 **Finish** and **OK**

19 Run a **Centreline Simulation**.

Only the area within the stock curve is roughed out.



20 Run a **3D Simulation**.



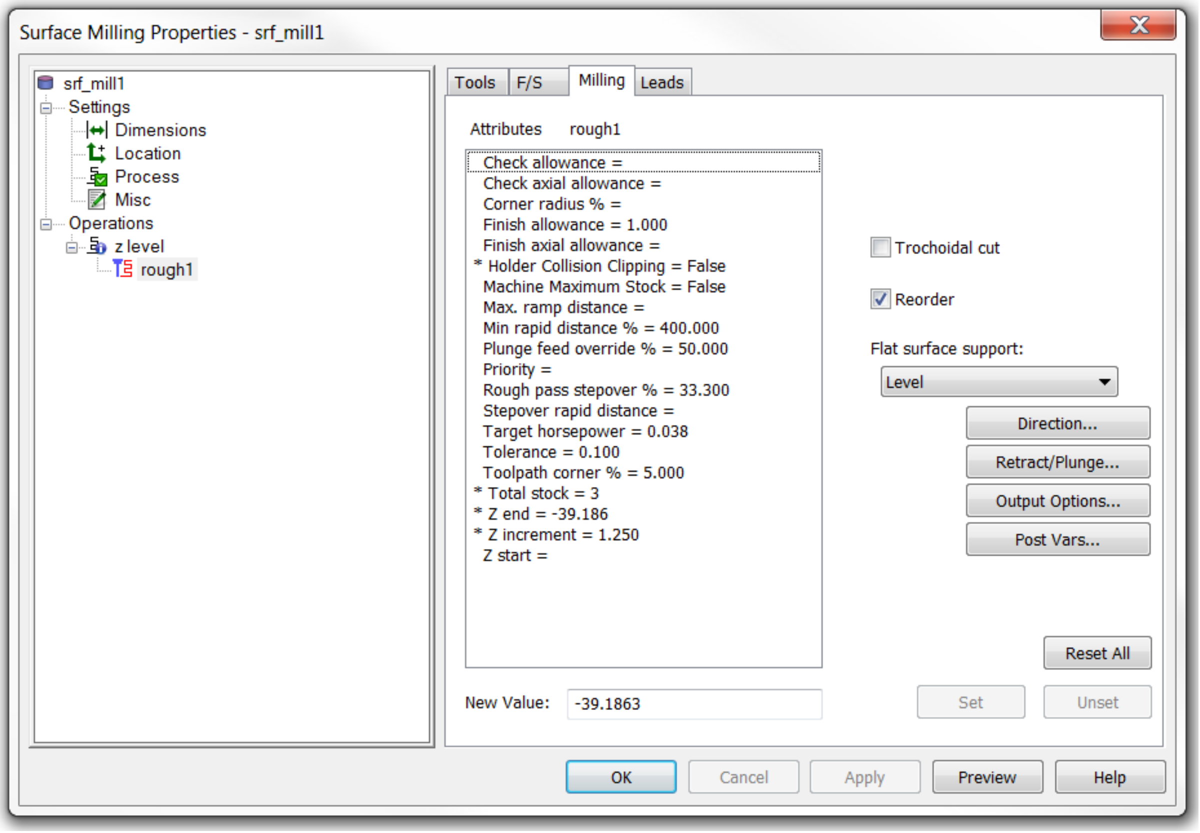
Simulation paused to show Material.

Total Stock

If the part is being machined from a casting or fabrication it may have an even thickness of material left on all of the surfaces. In such a case using a stock curve will result in the roughing making unnecessary air cutting moves. Fortunately in **FeatureCAM** we can allow for this by using the **Total Stock** attribute. By setting a total stock value we are telling **FeatureCAM** explicitly how much material is left on the part as a uniform thickness across the whole model.

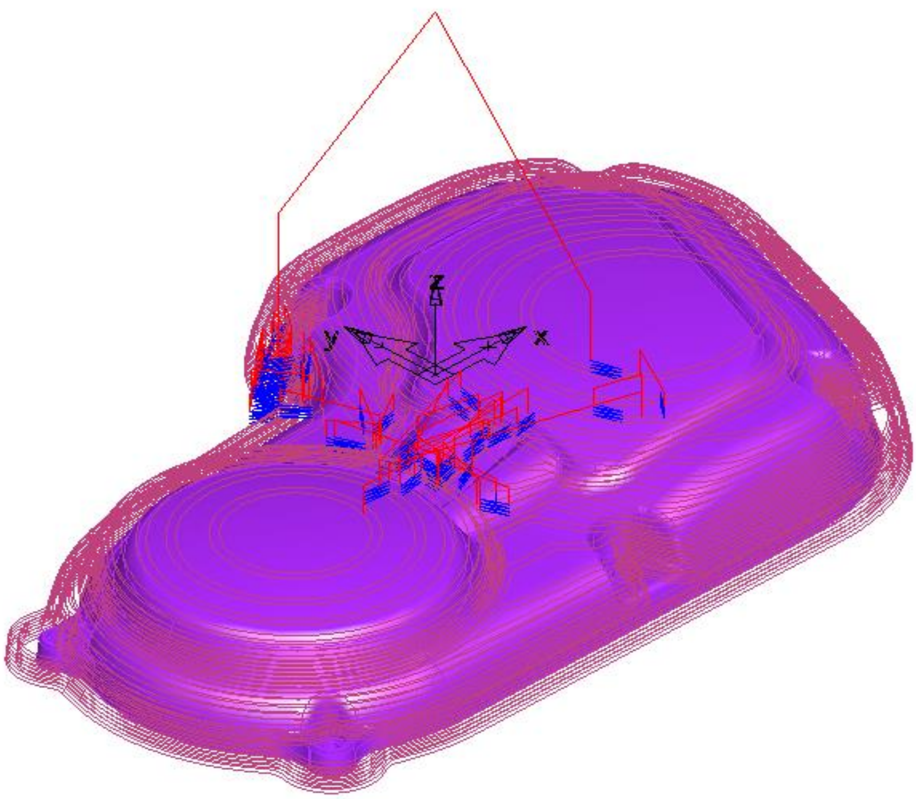
21 Edit the surface mill feature, select **rough1** and then the **Milling** tab.

22 Set the **Total Stock** to **3mm** and Z Increment to **1.25** as shown.



23 **Apply** and **OK**.

24 Run a **Centreline Simulation.**



The tool is now making a single roughing pass at each level to clear the remaining stock.

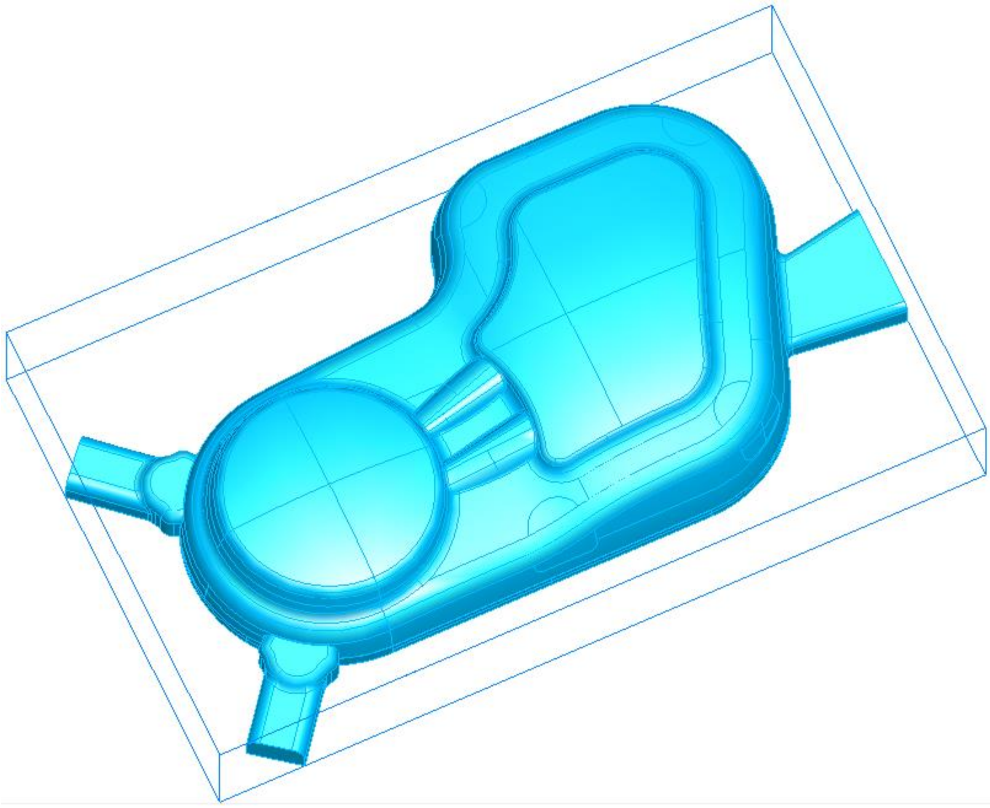
Stock Solid

When using **Total Stock** the remaining material is defined as an even thickness over the whole model. In practice however, the stock form is often only a simplified version of the finished model, and may include runners, risers and other features required by the manufacturing process. **FeatureCAM** allows the user to represent the stock using a solid model. Where a solid is used as the stock, the roughing process will only remove material between the solid stock and the part being machined.

1 Open the part **Primary_Case_Stock_Model.fm**

In this part there are two solids, one (part model) represents the part to be roughed, and the other (casting) is to be used as the stock material.

2 **Show all** and then hide the solid **part_model**. **Shade** if necessary.



This solid (casting) is the same as the part model except for certain changes; the recesses where the screw holes were have been filled in and casting sprues have been added. The aim of the following roughing process is to clear the area around the screw holes and remove the sprues using the minimum amount of roughing moves. Note the runner (to the right) and the risers (to the left). We will now prepare a roughing program to remove these from the part model and to rough out the recesses

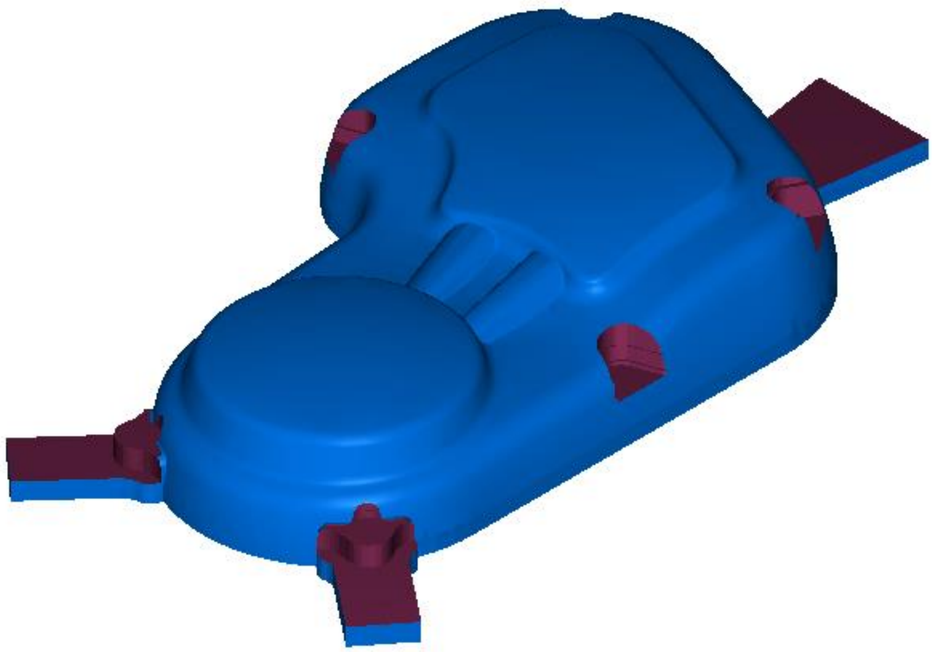
Double click on the stock.

3 Change the **Stock type** from **Block** to **User defined**.

4 Click on **Stock Solid**.

5 Check the solid **casting** and then click the **OK and Apply** button. **OK**.

6 Run a **Centreline Simulation** then a **3D Rapid Cut** simulation.



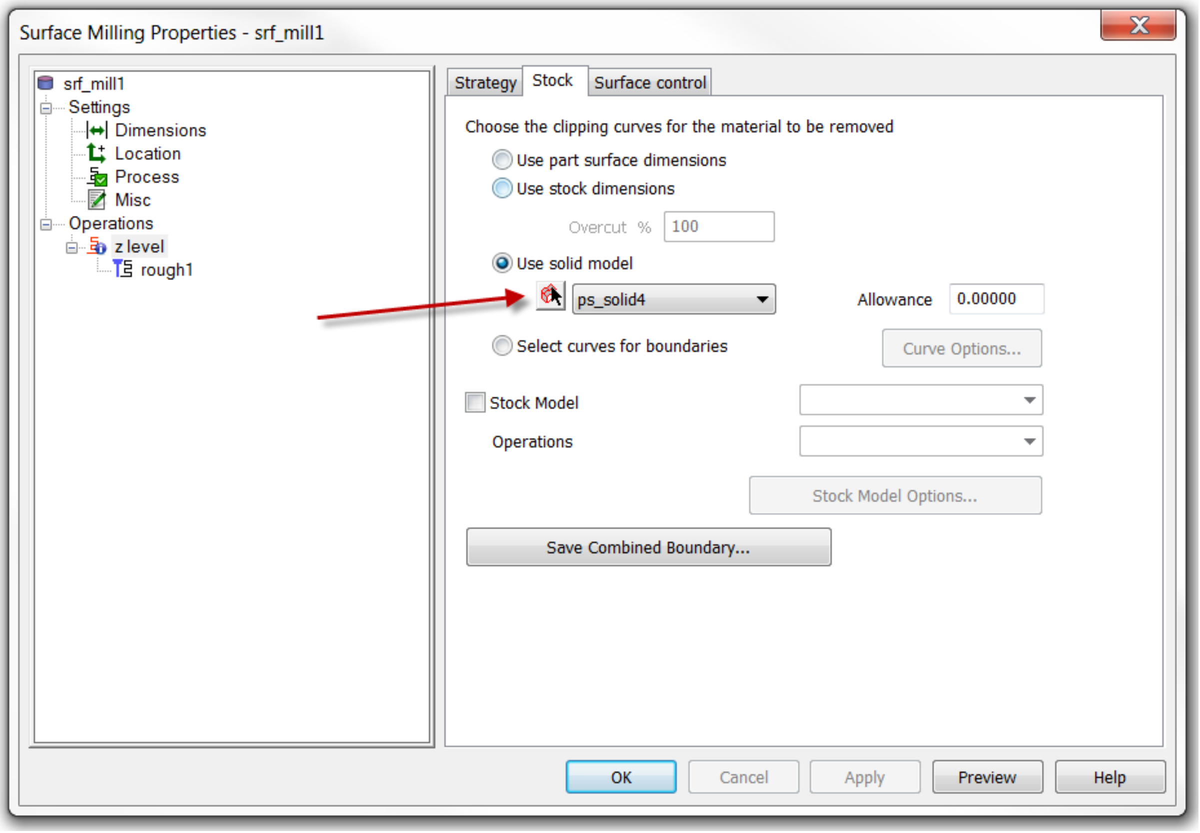
*You will see that the stock in the 3D simulation is the solid "casting" but that the actual roughing toolpaths are still sized to the previously defined block. We need to make one more change to allow **FeatureCAM** to calculate the roughing paths from the stock solid.*

7 **Eject** the simulation.

8 Edit the feature **srf_mill1.**

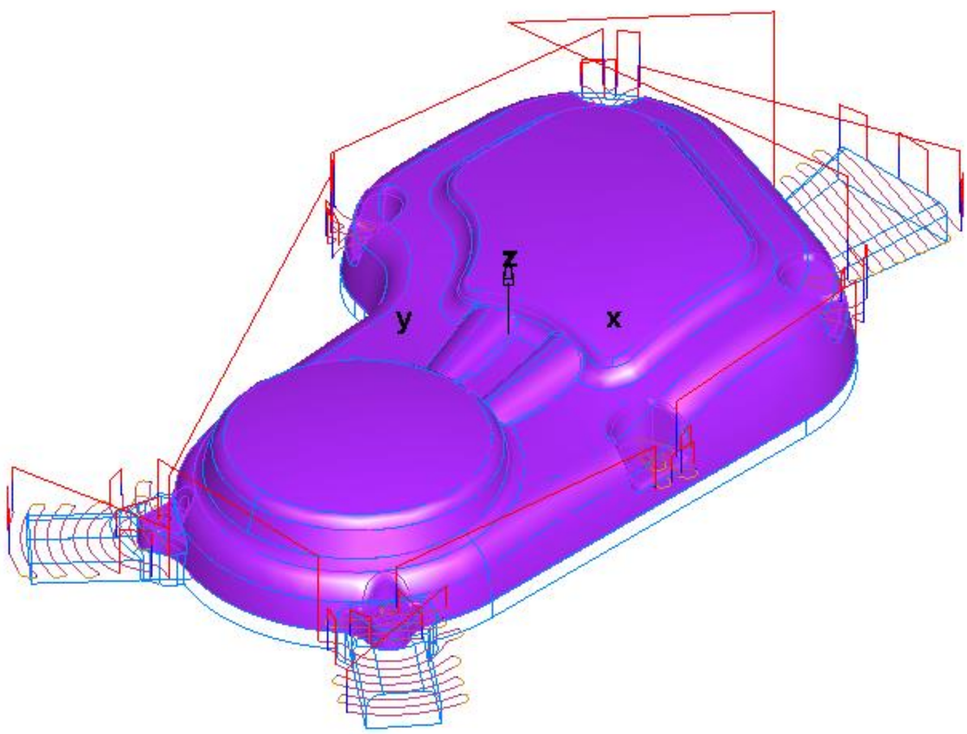
9 Select the **z level** feature and then click on the **Stock** tab.

10 Check **Use solid model** then select **casting** from the pull-down menu.



11 **Apply** and **OK**.

12 Run a **Centreline Simulation** then a 3D RapidCut simulation.



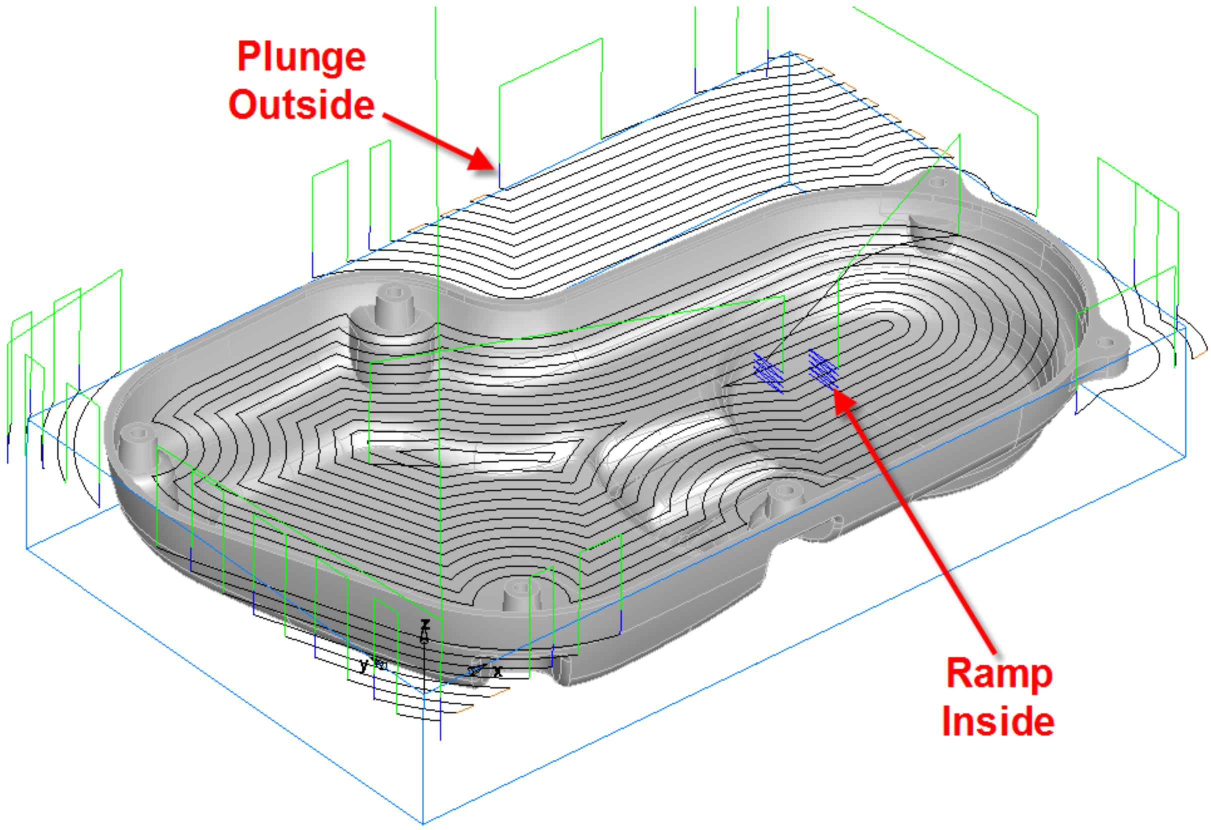
Note how the roughing now machines only those areas between the finished part and the stock solid. This eliminates a lot of air cutting.

Pocket/Boss

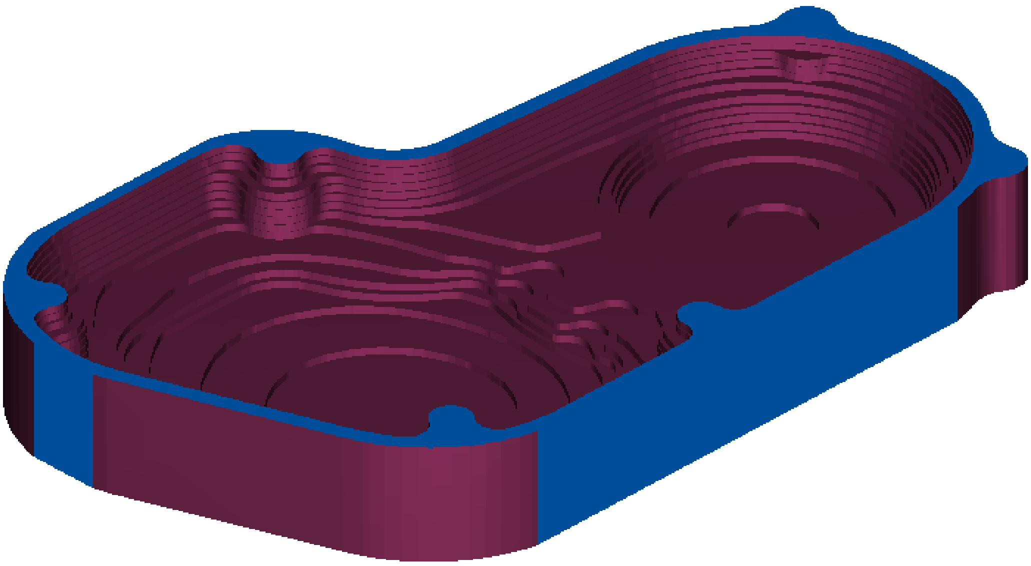
In all of the roughing operations so far we have used the default **3D Boss** option on the Strategy tab. We will now take a look at the **3D Pocket** option and how it differs from the Boss. The main difference between these two options is that with a Boss the tool will approach the part horizontally from outside of the stock if possible. If the feature is defined as a Pocket the tool will approach the part from above either by a plunging move or by ramping.

13 Open the part **Primary_Case_Pocket_Boss.fm**

14 Run a Centerline Simulation and then a 3D Simulation.



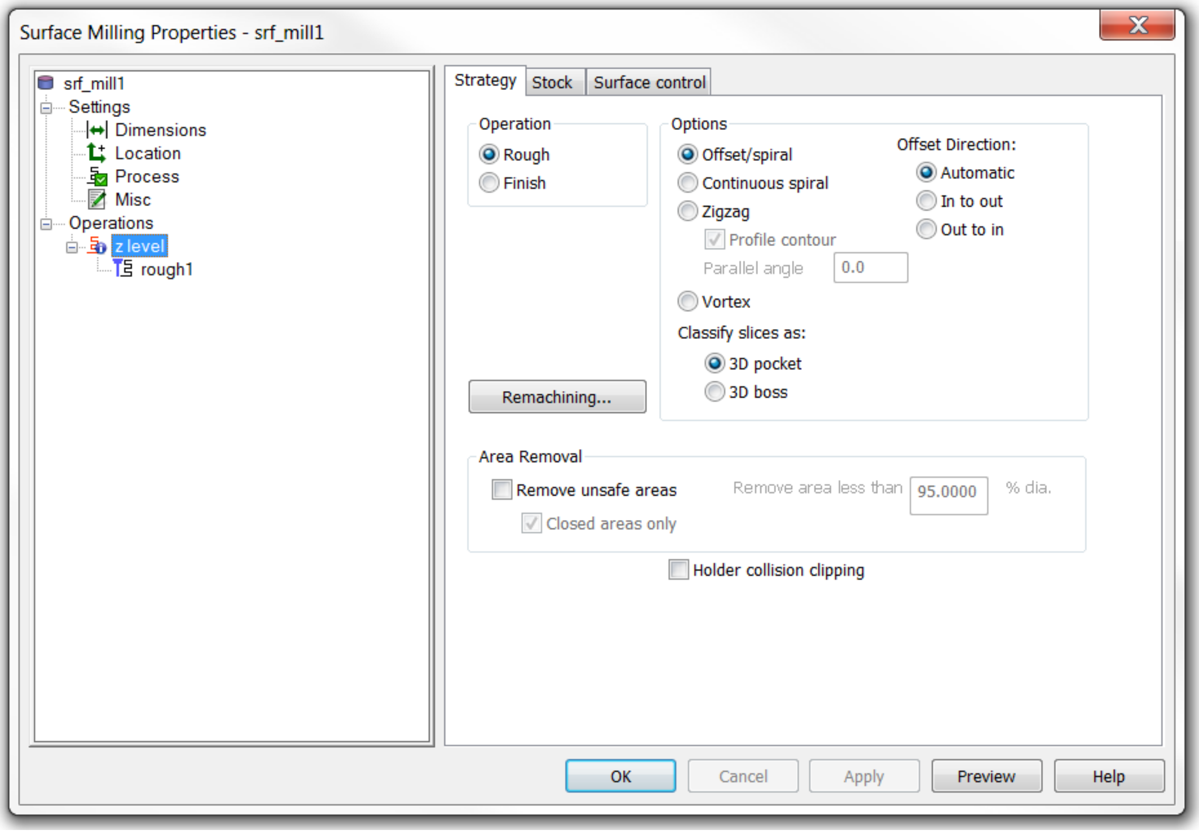
You will see that as the part is roughed out of the block stock the tool approaches the outer areas by plunging outside of the block, the inner, enclosed areas are approached by a ramping (blue zigzag) move from above.



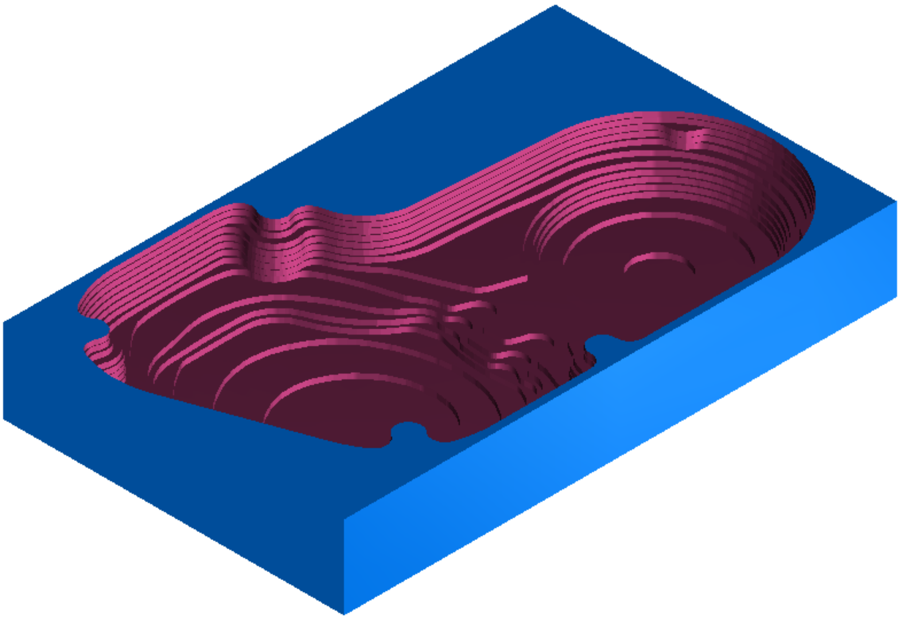
Often when roughing such a part the stock may be clamped on the outside, or the material may have been roughed earlier by cutting from the opposite side of the block. In such a situation the user would probably want to machine only the center pocket of the part and leave the outer area. We can do this by switching from the 3D Boss option to a 3D Pocket.

15 Edit the feature **srf_mill1** and click on **z level**.

16 Click on the **Strategy** tab and check **3D pocket**. **Apply** and **OK.**



17 Run a 3D Simulation.



***FeatureCAM** is now looking for enclosed pockets so the area outside of the part is not machined. This only happens because there are **NO Part Surfaces** selected **outside** of the area to be machined. If any surfaces outside of the pocket area are selected then they would be machined also, however, the selected areas outside of the pocket would be approached from above by ramping moves and the tool would not go outside of the block. This can be useful when machining a part that is held in soft jaws with the minimum amount of damage to the jaws.*

We will now add in extra surfaces to the Part surface list to force FeatureCAM to rough outside of the part as well as inside while restraining the toolpath to the stock outline.

18 Right click on the solid Base in the **Part view** and **Show selected**.

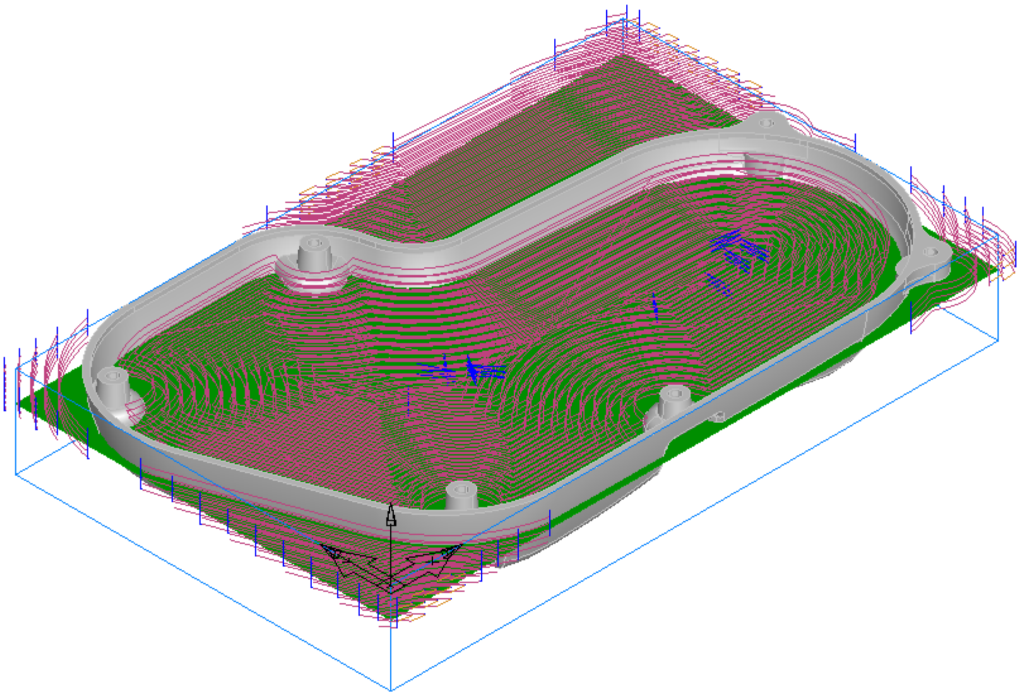
19 Edit the feature **srf_mill1** and click on **Part surfaces**.

20 Select the face **face_267** and **add** it to the part surfaces list.

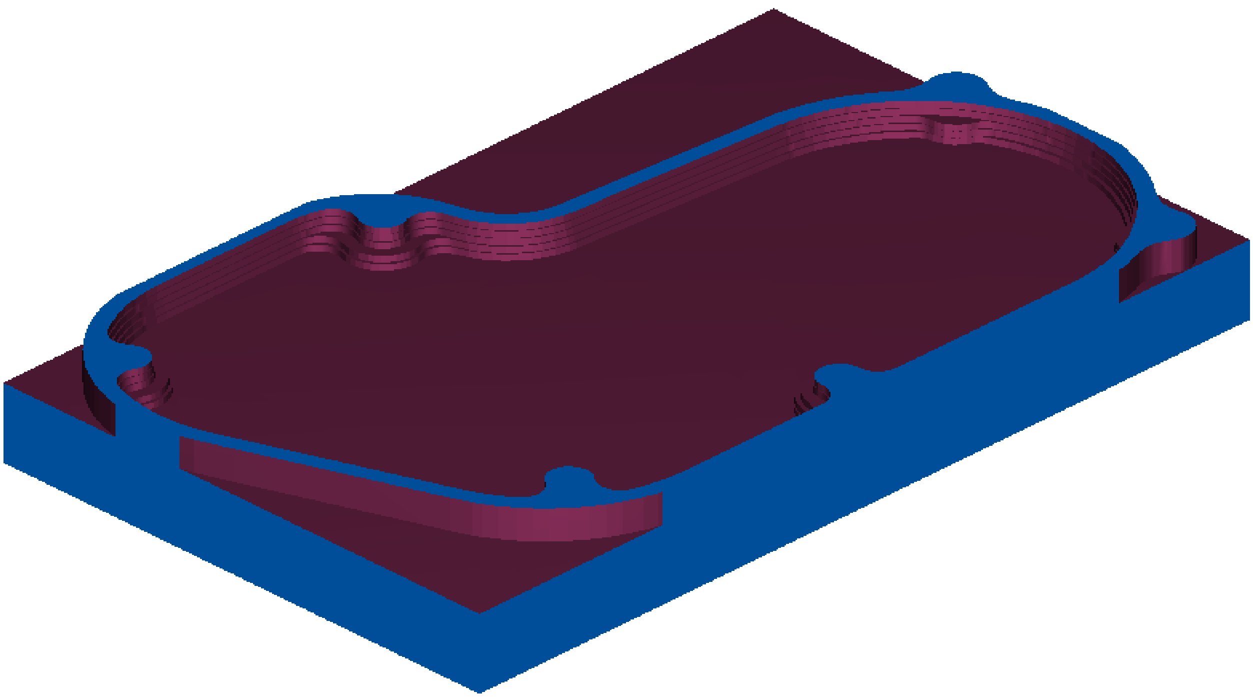
21 Click **OK** and **Apply** then **OK.**

*Note: **face_267** is the upper face of the solid called **base**.*

22 Run a Centreline Simulation and then a 3D Simulation.



Notice how the toolpath now remains inside the outline of the stock and approaches by a ramping move from above in all cases. The final roughing pass is above the surface ***face_267*** *by a distance equal to the* ***finish allowance plus*** *the* ***tolerance****.*



Use of Boundary Curves

It is often convenient to limit the area to be machined using a ***Boundary****. A boundary is a curve which is used to define an area of the part that is to be machined. The area to be machined may be either within the curve (a boundary) or outside of the curve (an island). Multiple curves may be used to define "nested" roughing areas as long as the curves do not cross. In this first example a simple curve will be used to limit roughing of the upper side of the primary case model.*

23 Open the part **Primary_Case_Boundaries.fm - This is an Inch Document.**

24 Run a **3D simulation**.

A 2.5D Boss feature has been used to rough out the billet. We will now rough out the remaining material using a boundary to minimize air cutting.

25 Box select all the surfaces.

26 Create **a Z level surface mill** feature. **Offset/spiral** and **3D Boss. Finish.**

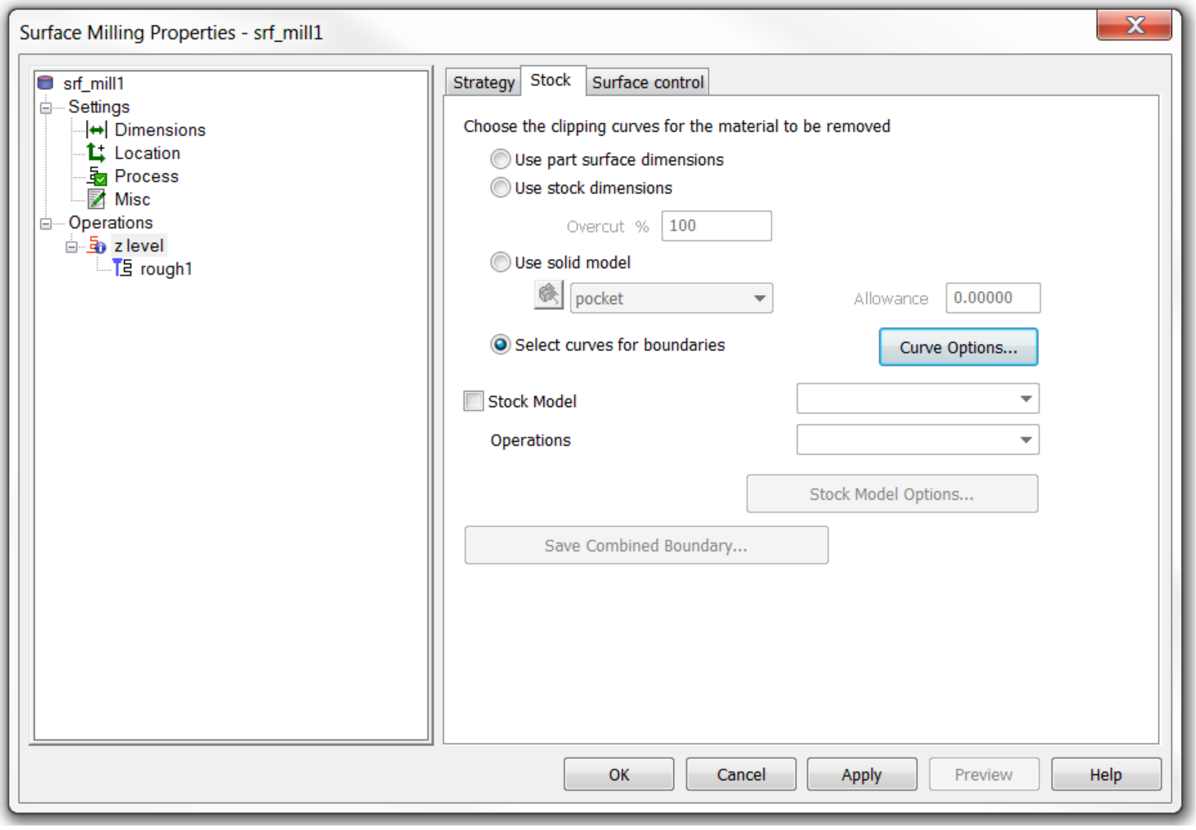
27 Select the **Milling tab (rough1)** in the **Surface Milling Properties** dialog.

28 Set the **Z Increment** to **0.1"**.

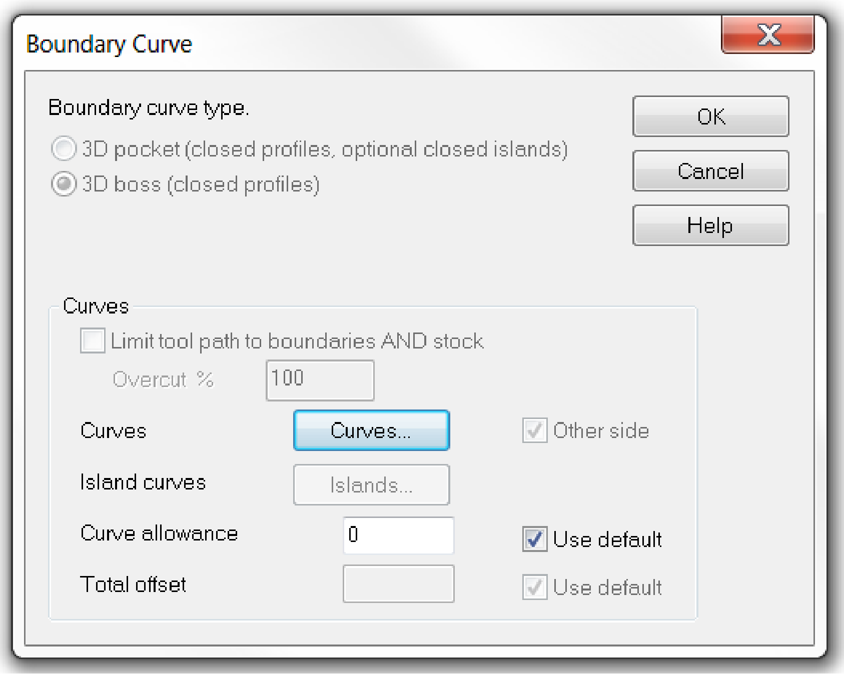
29 Select the Stock tab (pick z level in the tree).

30 Click on Select curves for boundaries.

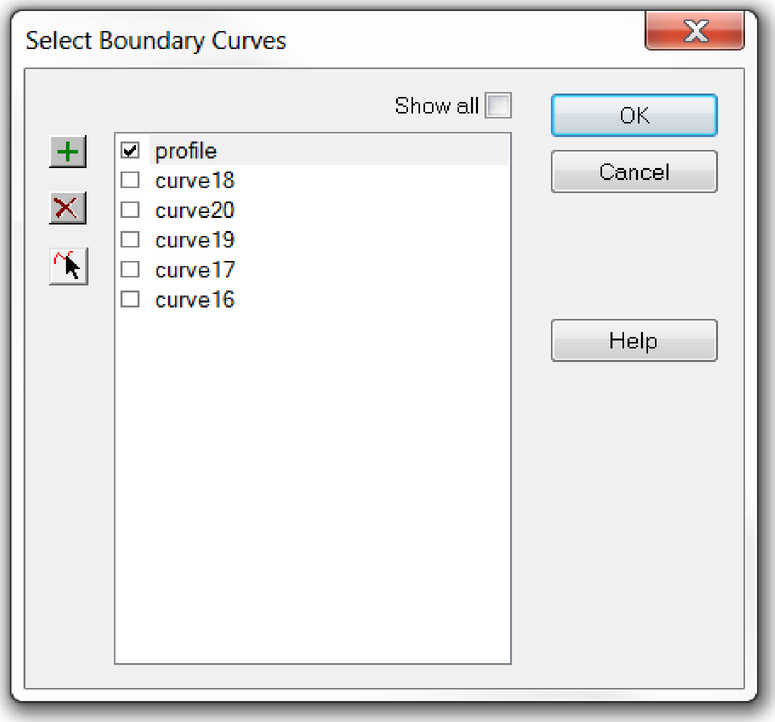
31 Click on Curve Options button.



32 Click on the **Curves**... button.



33 Select the curve called **profile**.



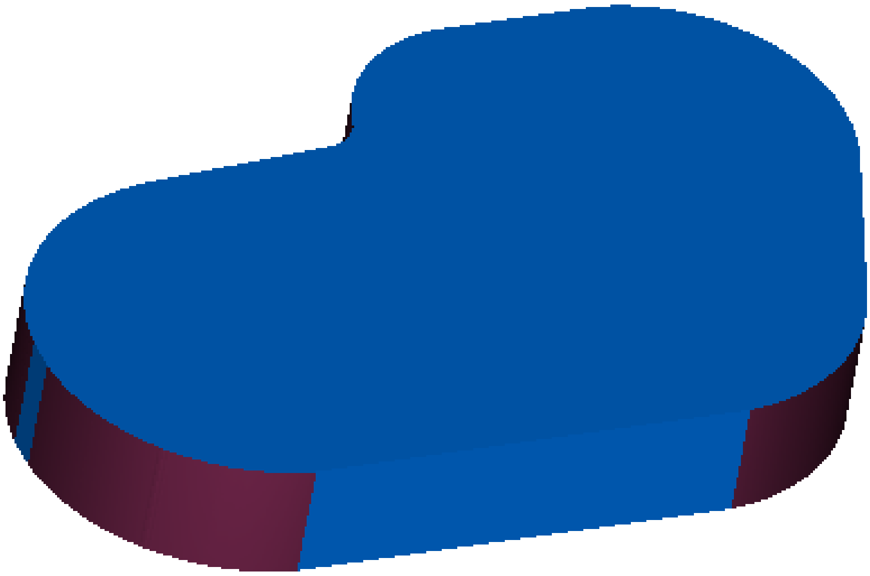
34 Click **OK**, **OK** then **Apply and OK**.

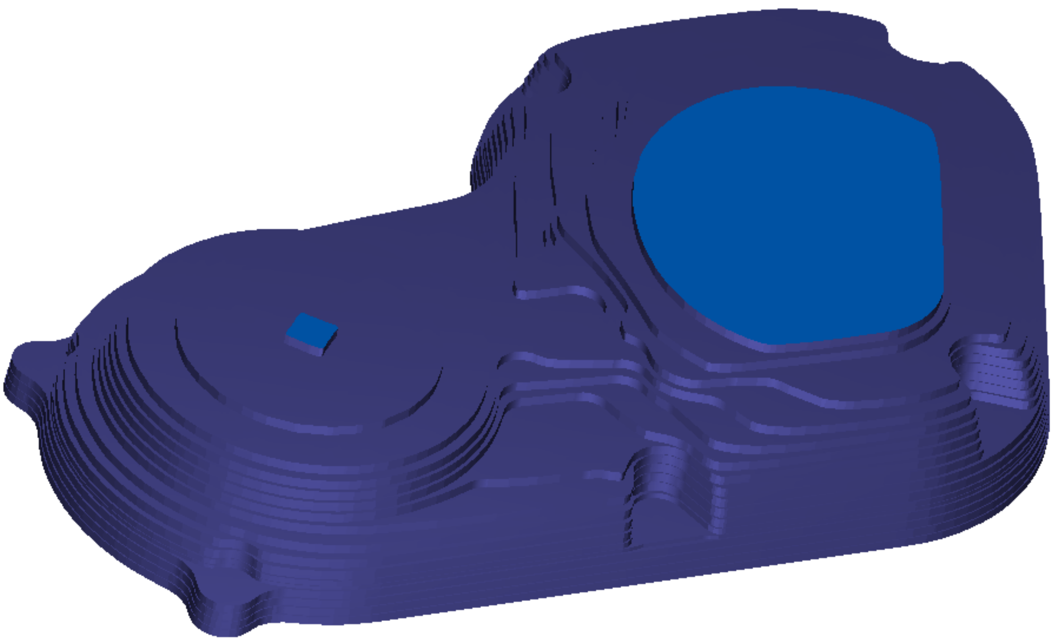
35 Run a Centreline simulation.

36 Run a 3D simulation.

The 3D roughing now only machines the material left after the initial 2.5 Boss roughing operation. This gives a very efficient toolpath removing the maximum amount of stock in the minimum time.

The first simulation shows the Boss operation and the finished simulation shows the profile boundary used in the Boss Operation. This is also used as a boundary in the Z level machining strategy.





Part Surfaces and Check Surfaces

Select Part Surfaces dialog

You can use the **3D Part Surfaces** dialog to pick surfaces you want to include in your 3D part feature.

Select the surface(s) in the list or click the **Pick surface** button and select a surface with the mouse. To pick additional surfaces, click the **Pick surface** button again before selecting each additional surface.

1 Click **OK** to return to the **Feature Properties** dialog.

2 Click **OK** and **Apply** to apply your surface selection to the feature and return to the **Feature Properties** dialog.

You need to consider the following when specifying part surfaces:

You cannot manufacture undercut surfaces using 3-axis machining, so it is a good idea to use only surfaces in the feature that can be cut from the setup.

Some surfaces may be cut from multiple setups to manufacture all parts of the surface. In such situations, a **Stock Curve** is helpful in limiting the machining area to just those spots that need it.

Check Surfaces

Check surfaces are surfaces that denote areas that are not milled away. Use this dialog to select surfaces you want to use to limit machining in a 3D feature.

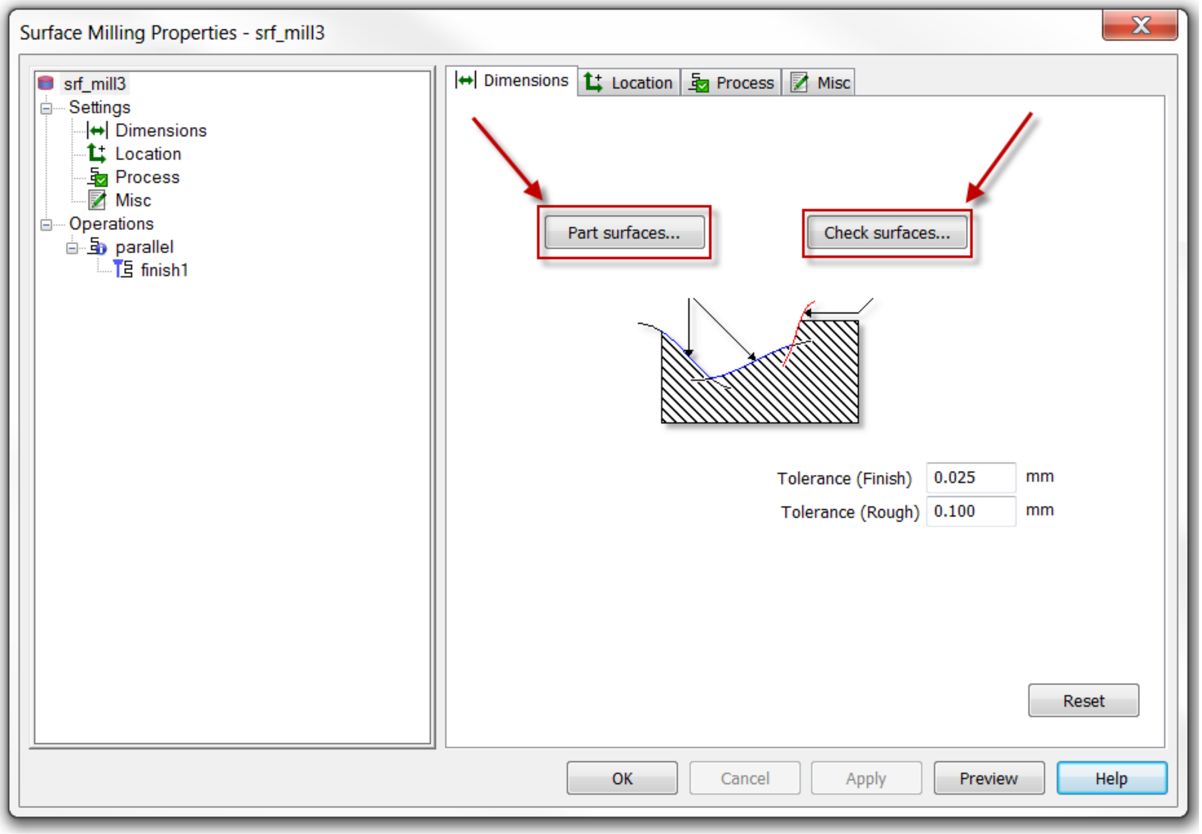
Select the surface(s) in the list box or click Pick and select a surface with the mouse. To pick additional surfaces, click Pick again before selecting each additional surface.

3 Click OK to return to the Feature Properties dialog.

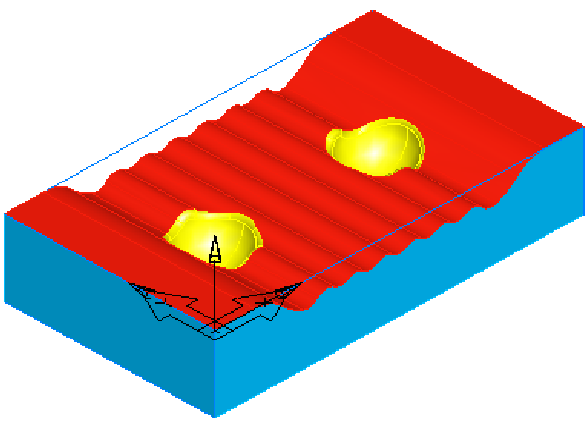
4 Click OK and Apply to apply your surface selection to the feature and return to the Feature Properties dialog.

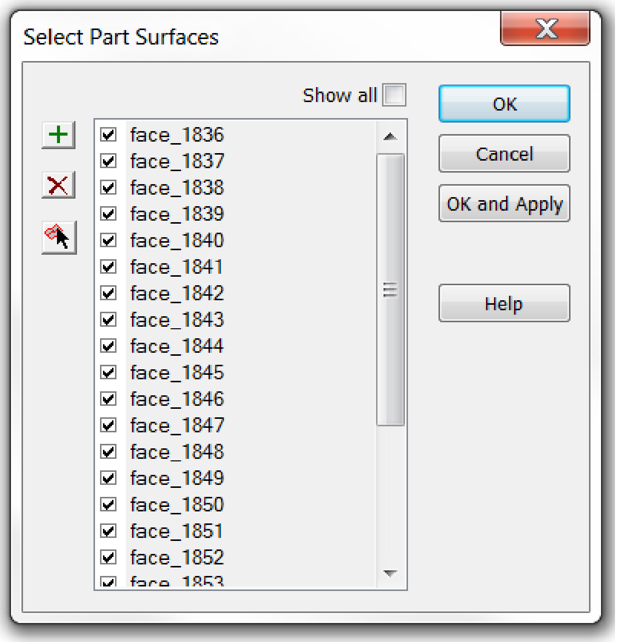
5 If you select any Part surfaces, they are allowed to be cut to the depth of the chamfer (in other words, the allowance is equal to - Depth).

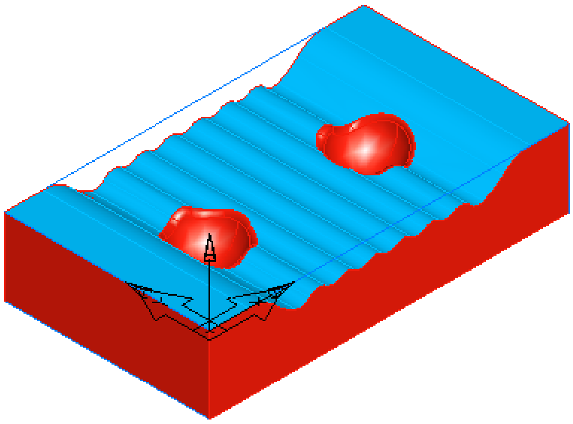
If you select any Check surfaces, there is a Check allowance attribute on the Milling tab (the default is 0).

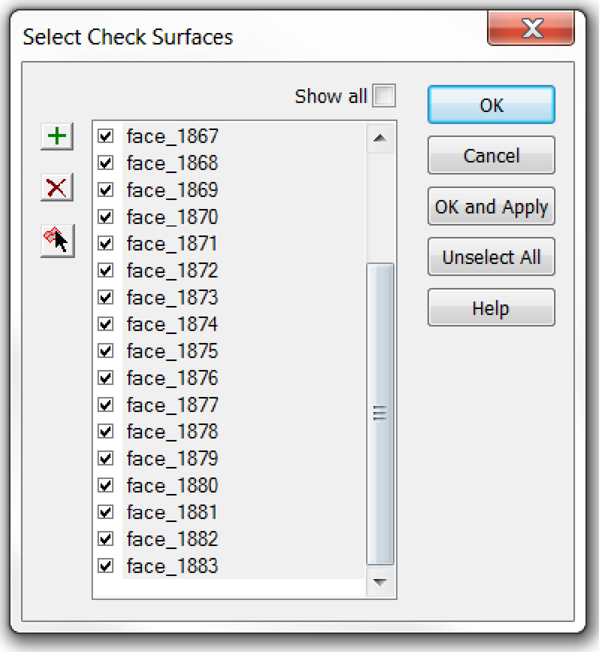


- The following example shows the difference between **Part Surfaces** and **Check Surfaces**.

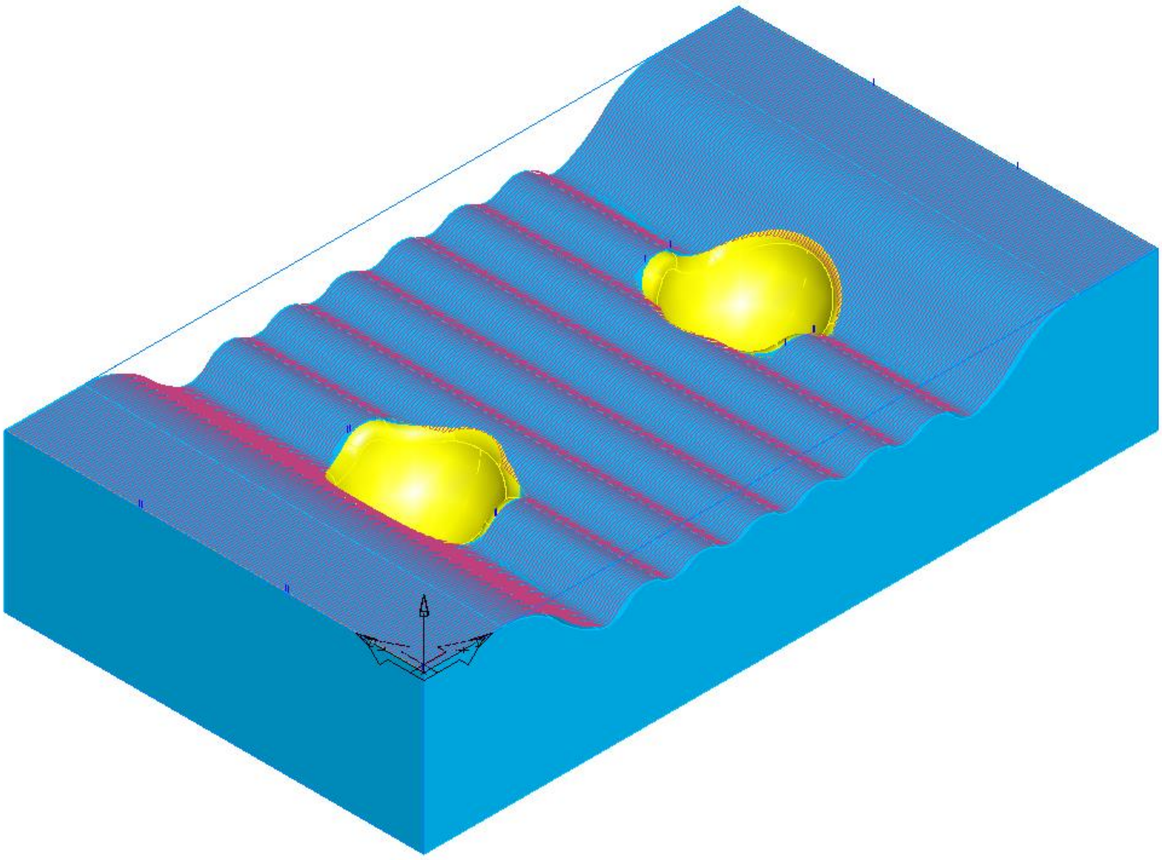








- The following Toolpath has been created using Part and Check Surfaces.



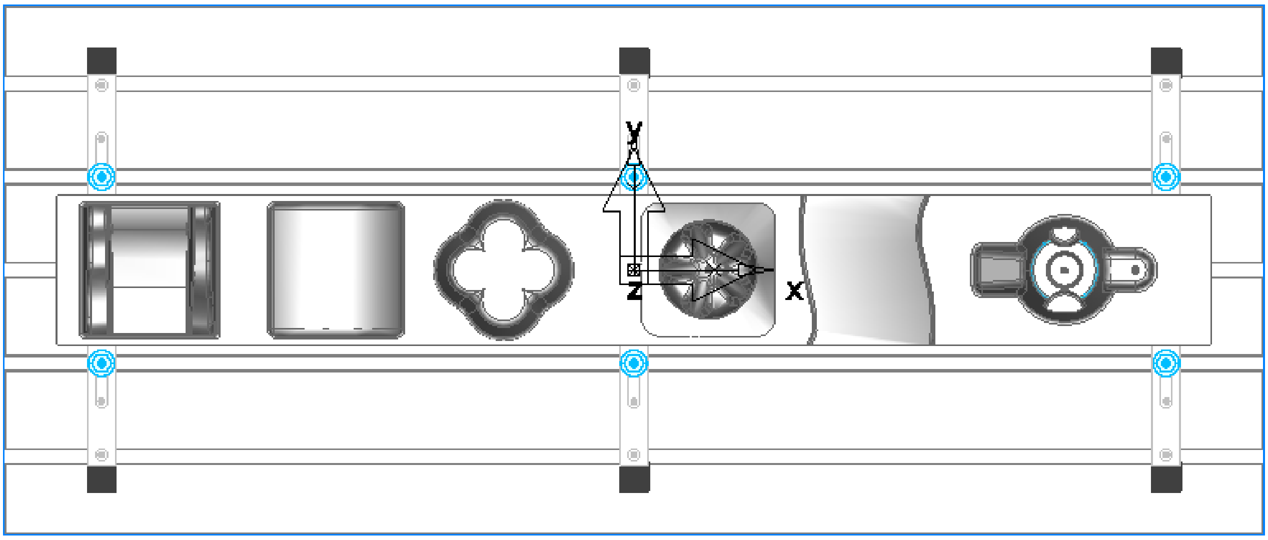
Contents

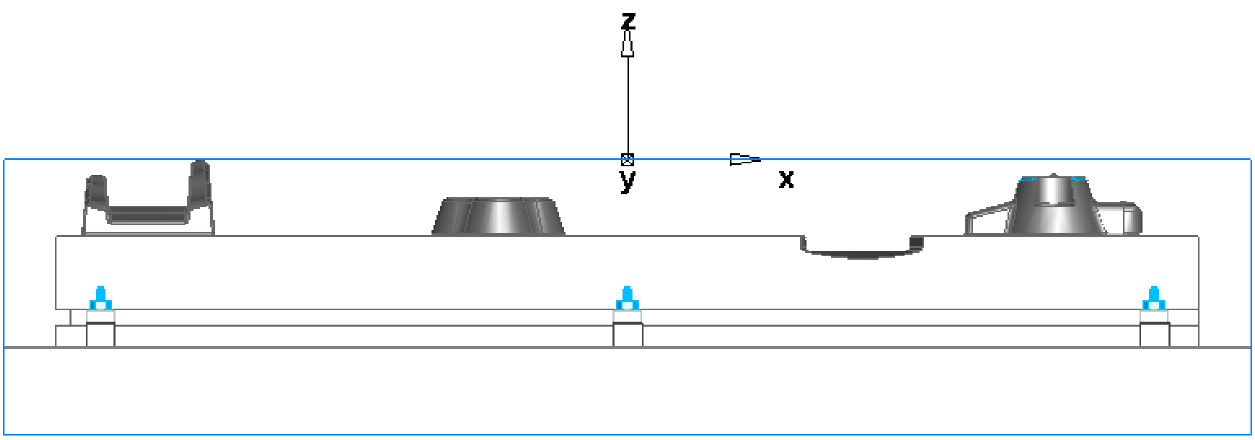
All Strategies Start

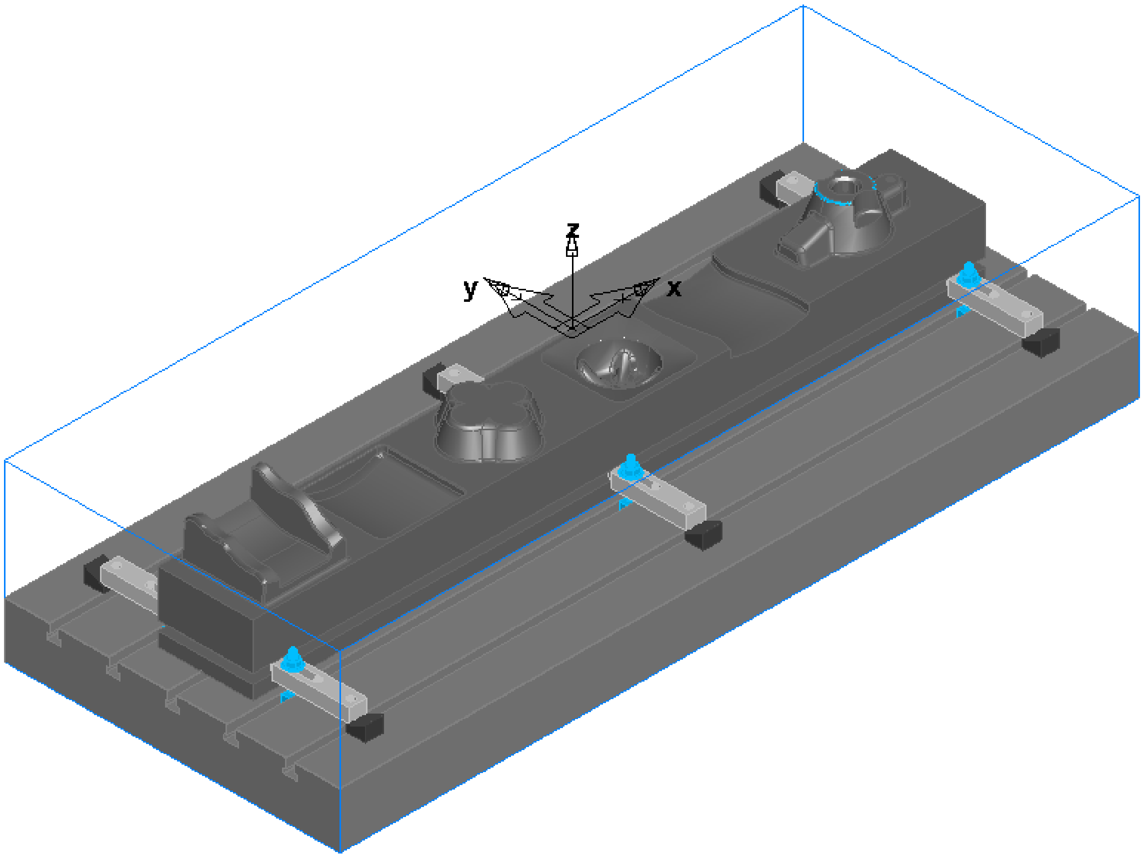
*We will start by importing a solid model into **FeatureCAM** which is in Parasolid format. You will find this in the FeatureCAM training folder **C:\Training_Data\FeatureCAM Course Data 2017\3D Machining\ All Strategies Part-1.x_t***

1 Go to **File>Import** and navigate to the above location.

*The image should look like the image shown below. Use the wizard to establish the part orientation and stock size. Leave as defaults. When you come to the stock just set the values to **zero** and select apply and then come out of the **wizard**.*





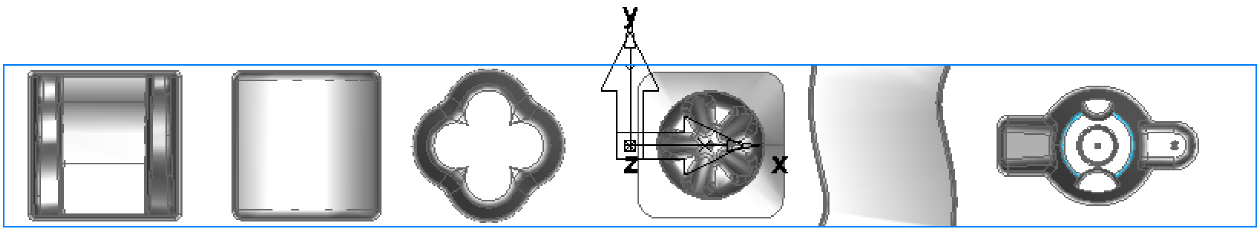


2 We will now change the stock size to the main solid model part.

3 Select the solid model in **PartView** and rename this to **Part**.

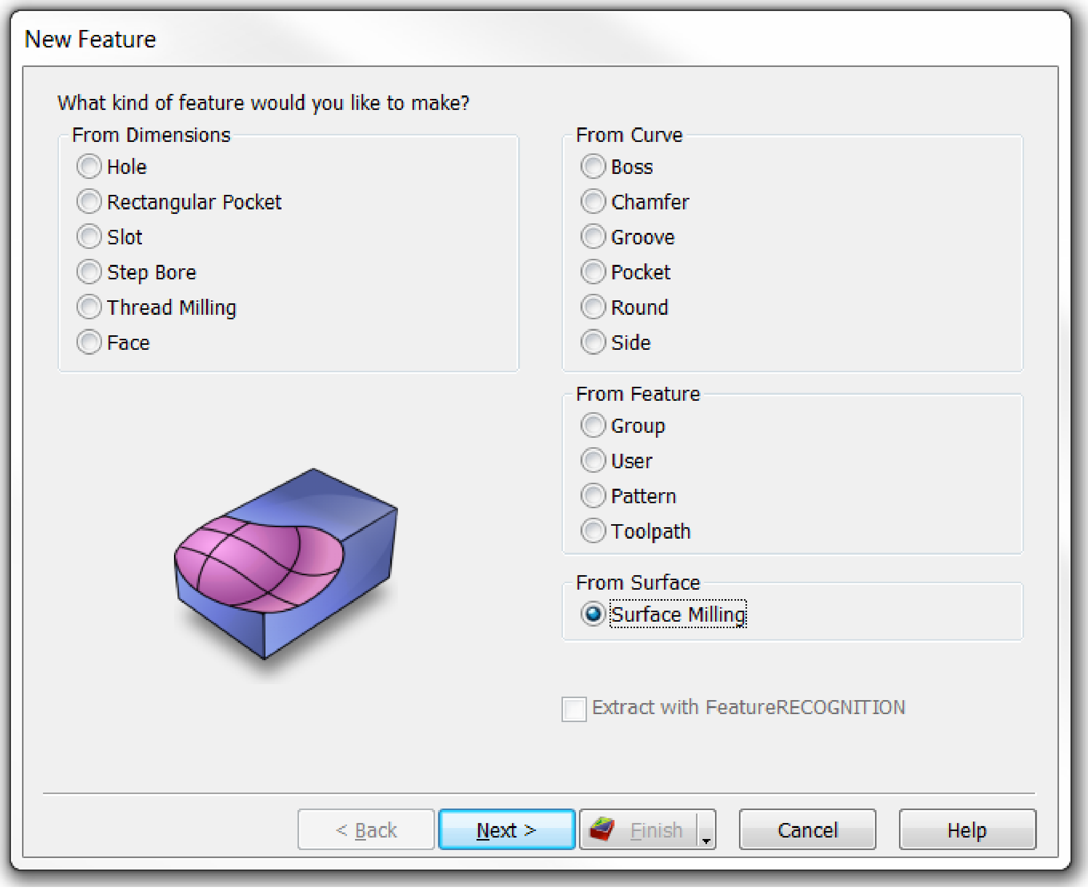
4 Select **Part** in **PartView** and then go to **Hide>Unselected**. This will then only show the main solid model.

5 Then double click on the **Stock** and select **Resize** and make all values **Zero**. The image below is how it should look.



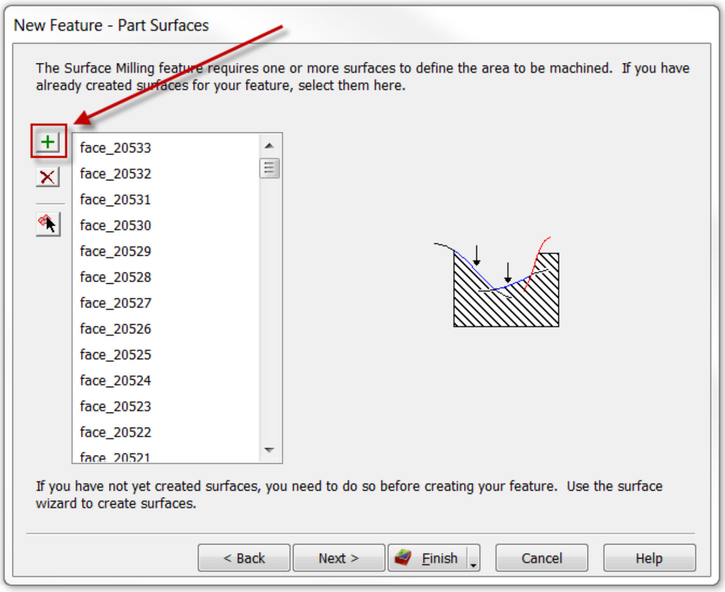
6 We are now ready to machine our part and will choose a **Z-Level Roughing** strategy to completely rough out our part.

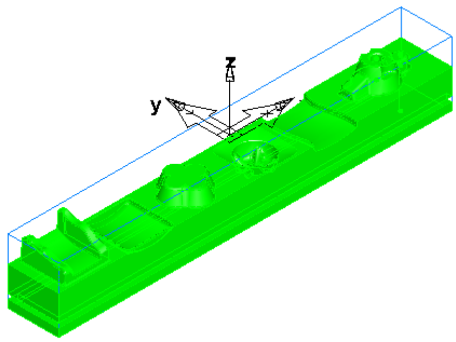
7 Select **Ctrl+R** to activate the **New Feature** menu. Select **Surface Milling**.



8 Select **Next**.

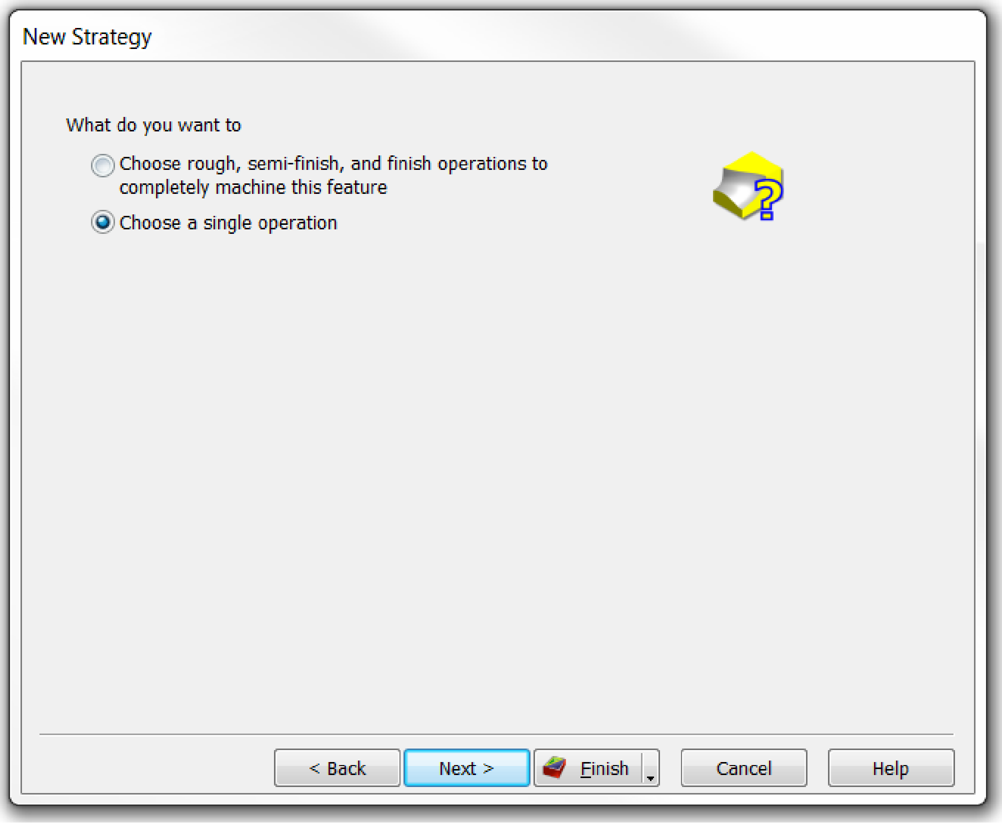
9 Manually select the solid model by windowing the part and then select the Green + key to select all of the surfaces on the solid model. As shown below.



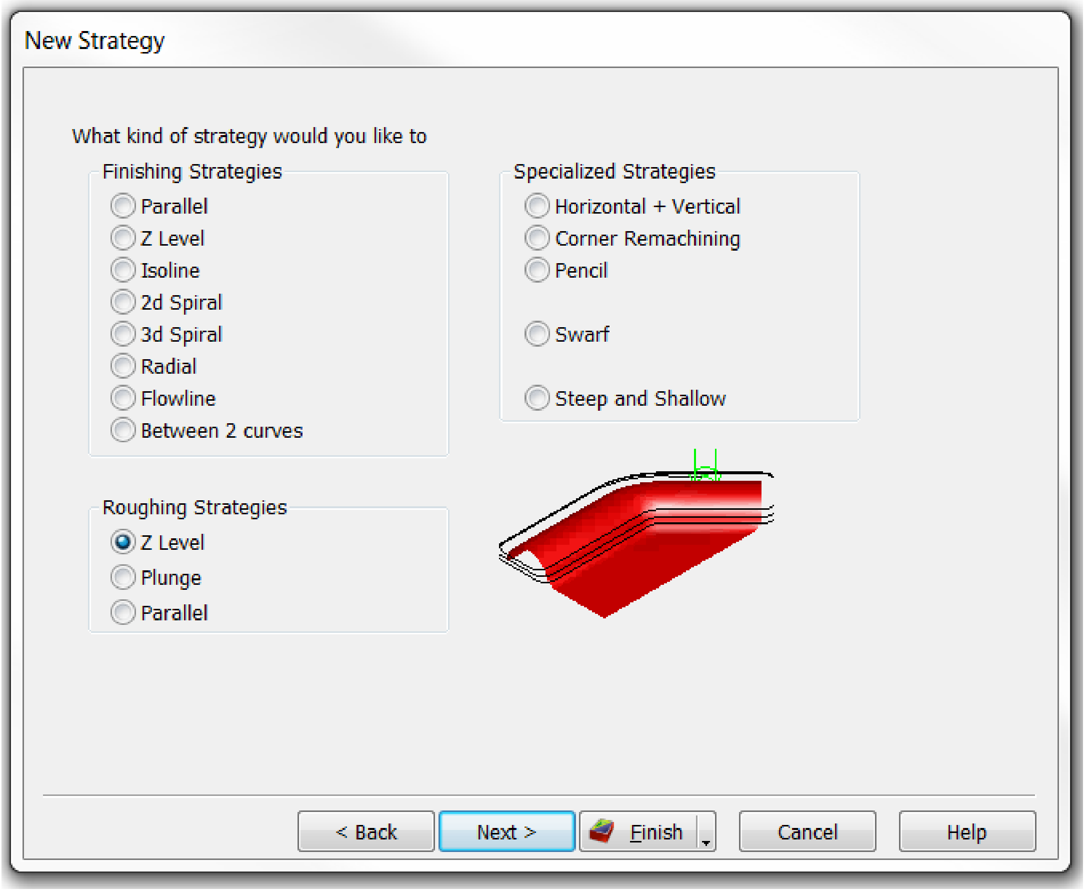


10 Select **Next** and you will be given **two options**.

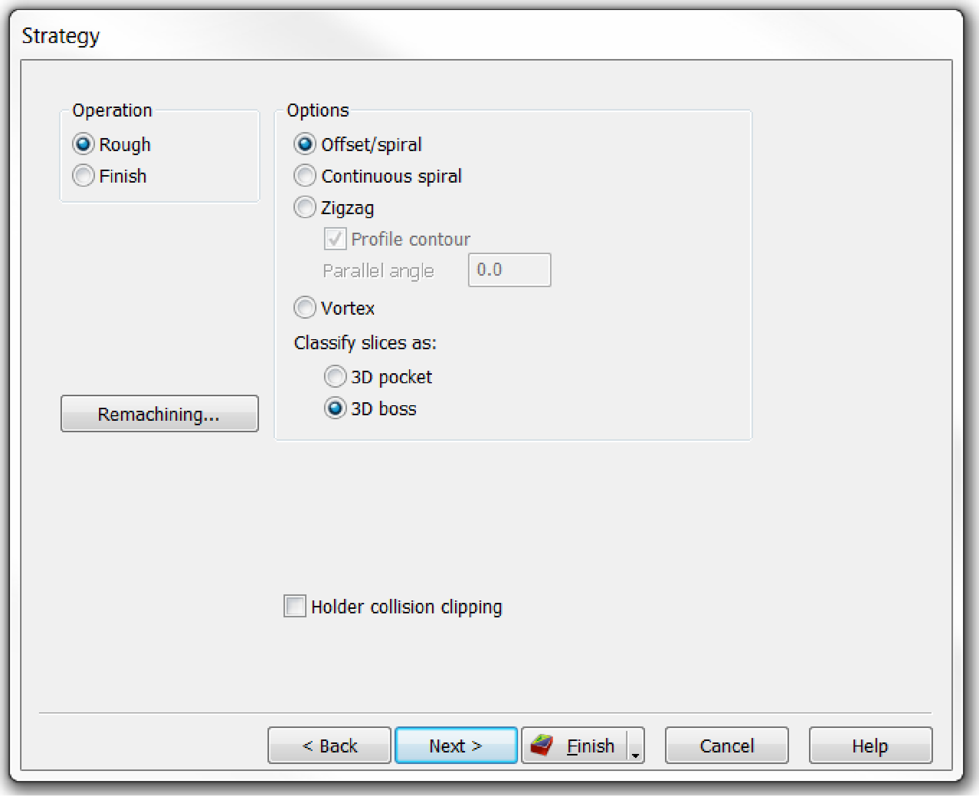
11 Select **Choose a single operation** which allows you to manually choose a strategy from another menu. Please select this option. As shown on the next page. Select **Next** to activate the **Choose a single operation** menu.



12 Select **Next**. Then select **Z Level Roughing**.

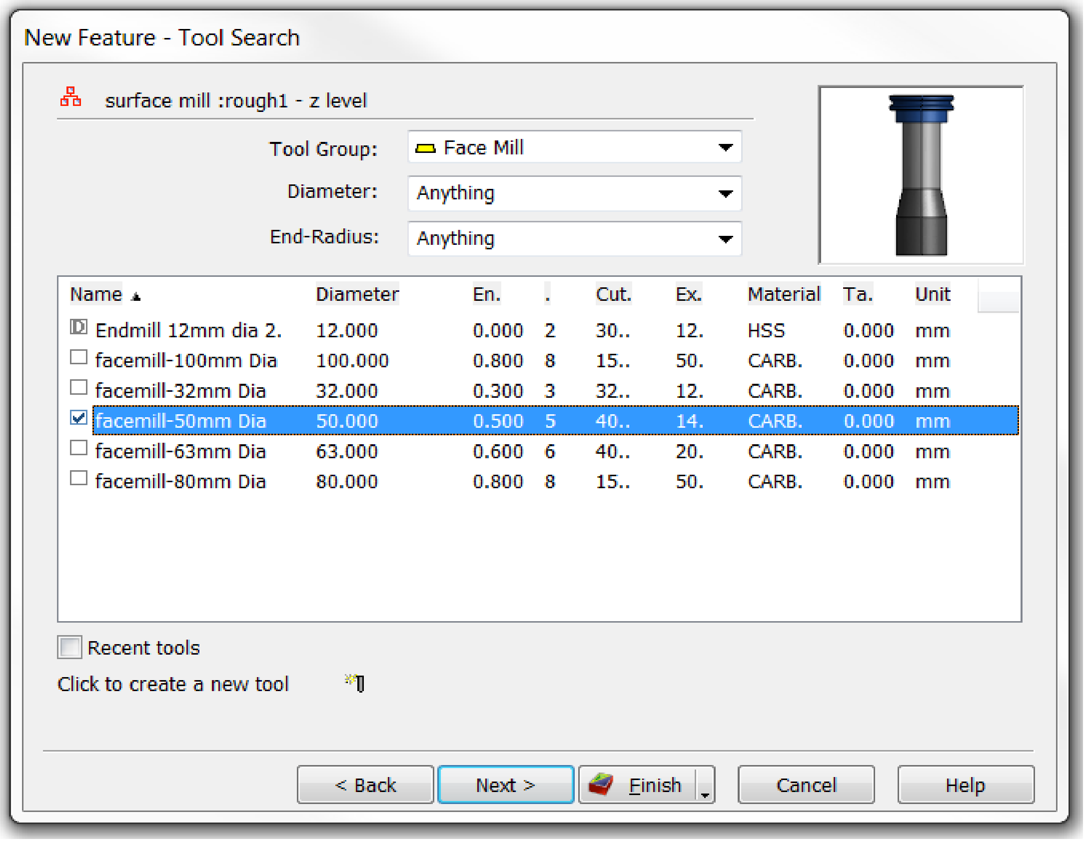


13 Select **Next**.

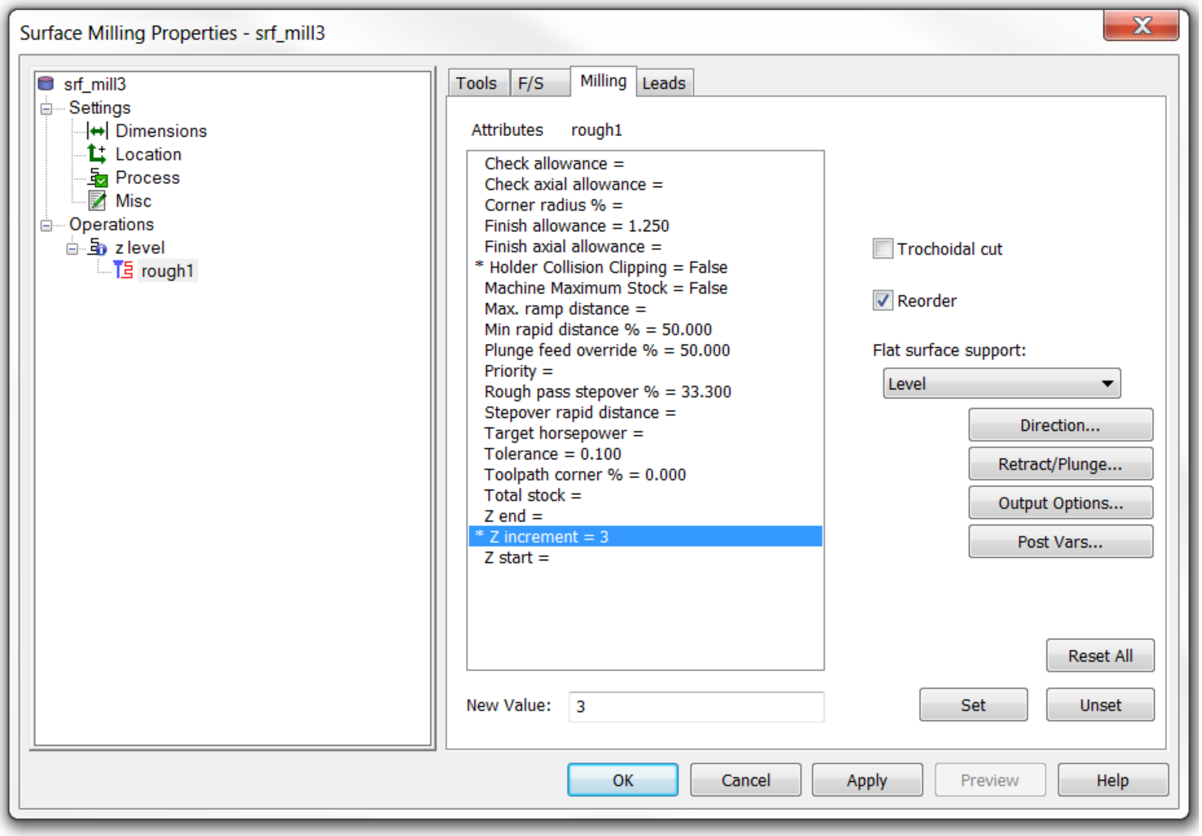


*By selecting **3D Boss** (Default) the toolpaths will be working from the **outside in.***

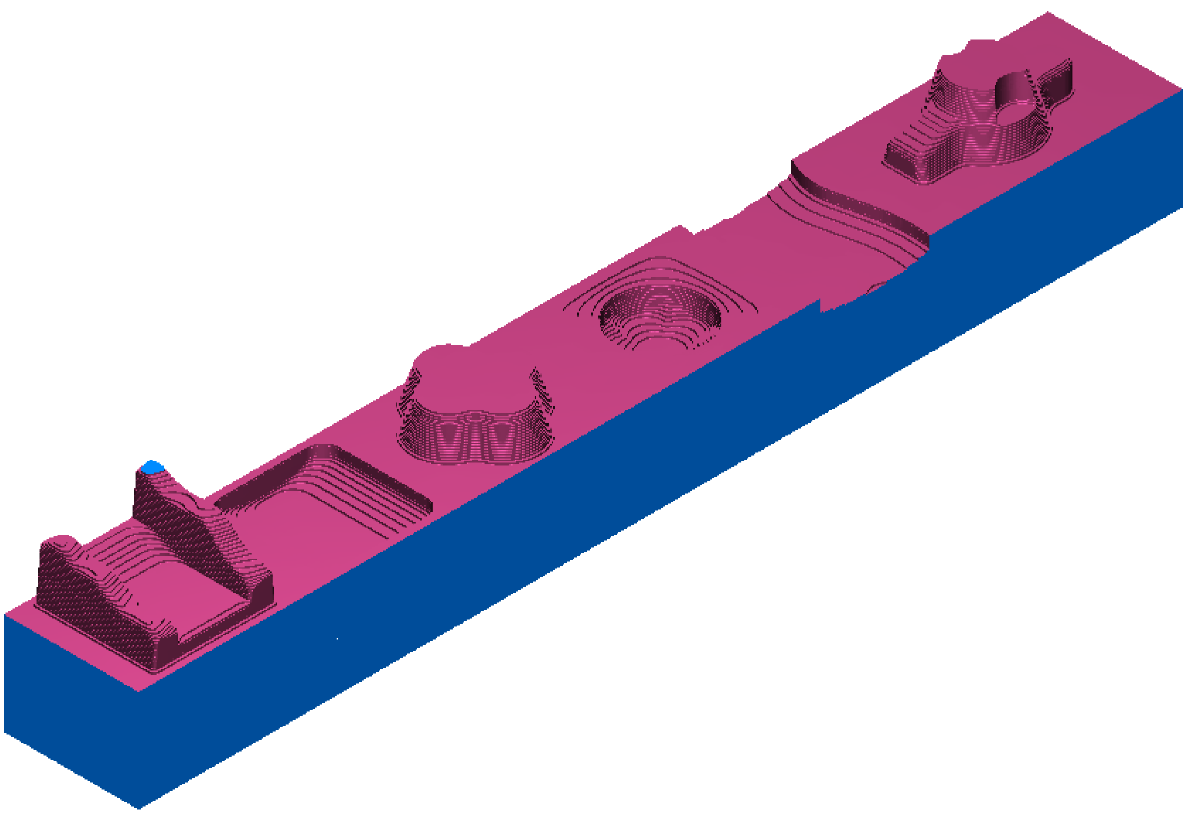
14 Select **Next**. Change the **Tool group** to **Face Mill**. Then select a **50mm Facemill** from the drop down menu. You may have to make this longer.



15 Select **Finish**. Then select the **Milling Tab** and change the **Z increment** to **3mm.** Select **Apply** and **Ok** to close the form.

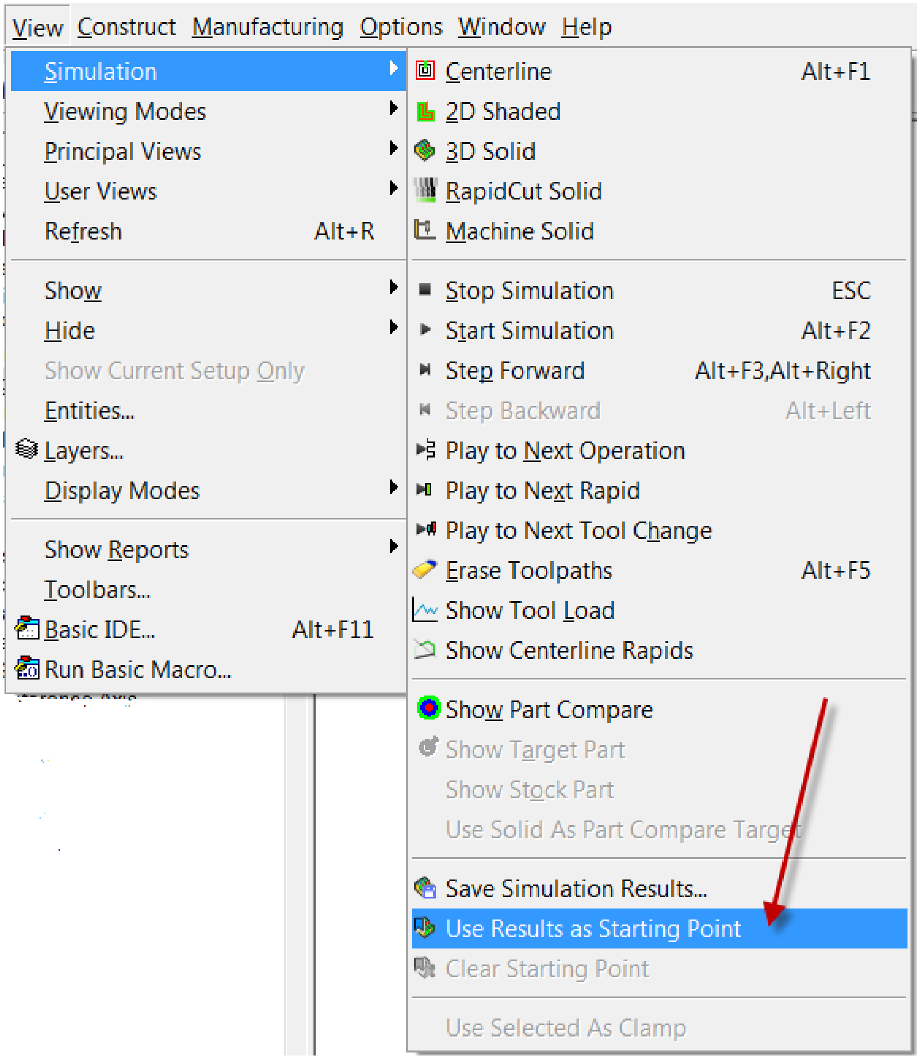


16 Run a **3D Simulation** to see the roughed out part. See below.



17 We now need to set the **3D Simulation** shown above as a starting point, to save simulating again up to this point.

18 Whilst the simulation is on the screen go to **View>Simulation>Use Results as a Starting Point.** Please select this option.



19 Select **Esc** or **Eject** to cancel the **Simulation**.

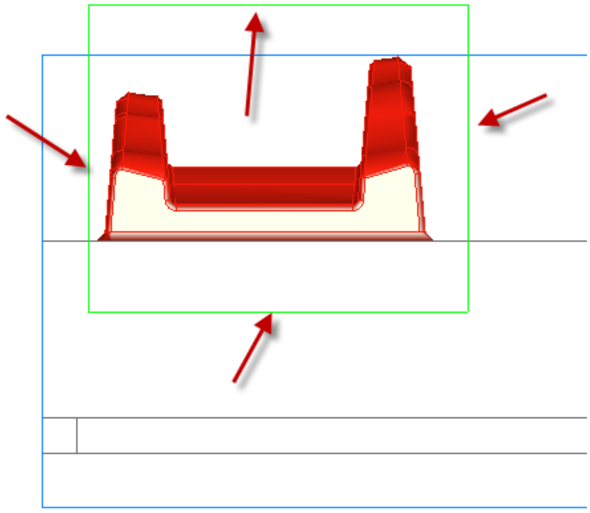
20 Rename the First machining operation to **Z Level Roughing** in **PartView.**

*We will now machine the model working from **Left** to **Right** starting with a **Parallel Finish** machining strategy. To hide the **3D Roughing Feature** please select the **Surf_Mill?** In **PartView** and then in the Right click menu select **Hide**.*

21 Create a New Feature **Ctrl+R** select **Surface Milling.**

22 Select **Ctrl+2** which is the **Front View**.

23 Only select the left model by windowing around the part. See Example on the next page.

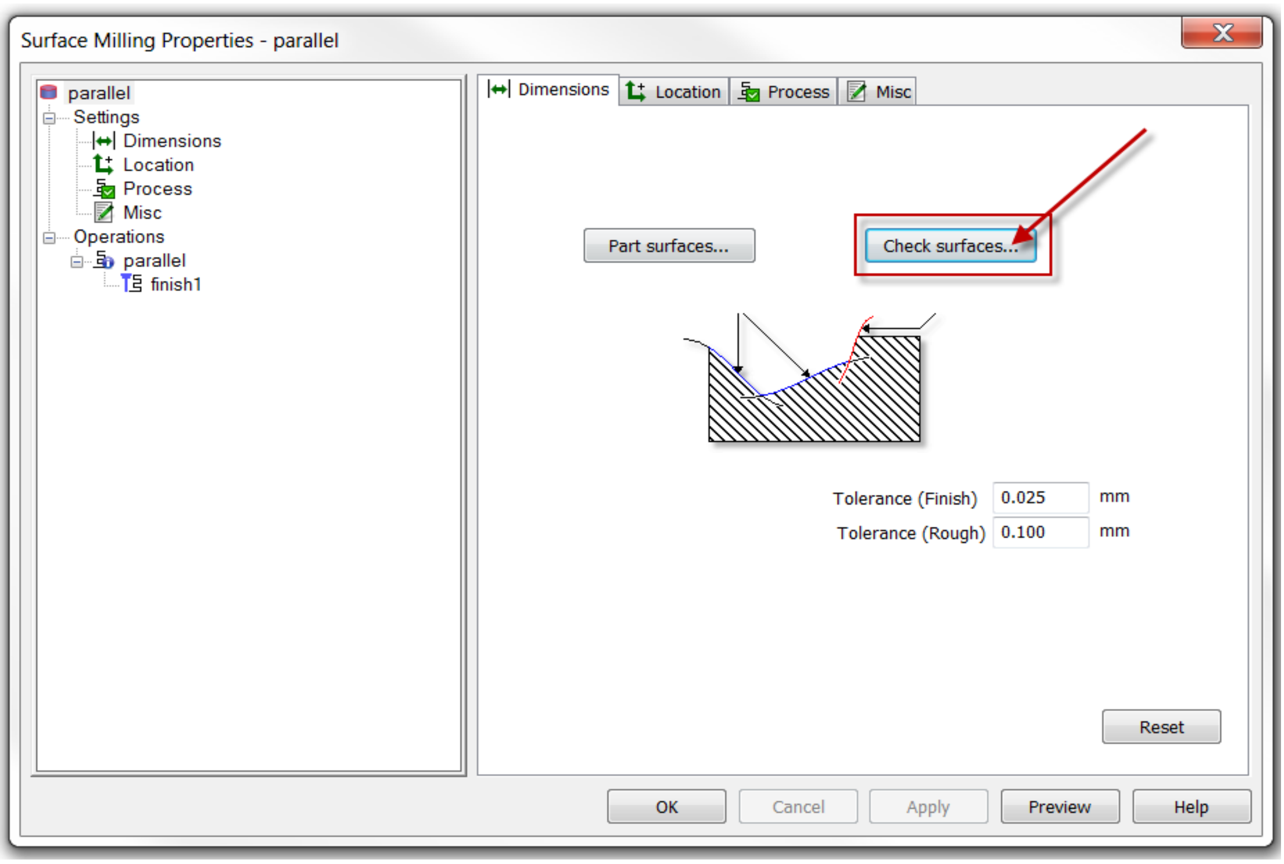


24 Select Next and then Choose a single Operation.

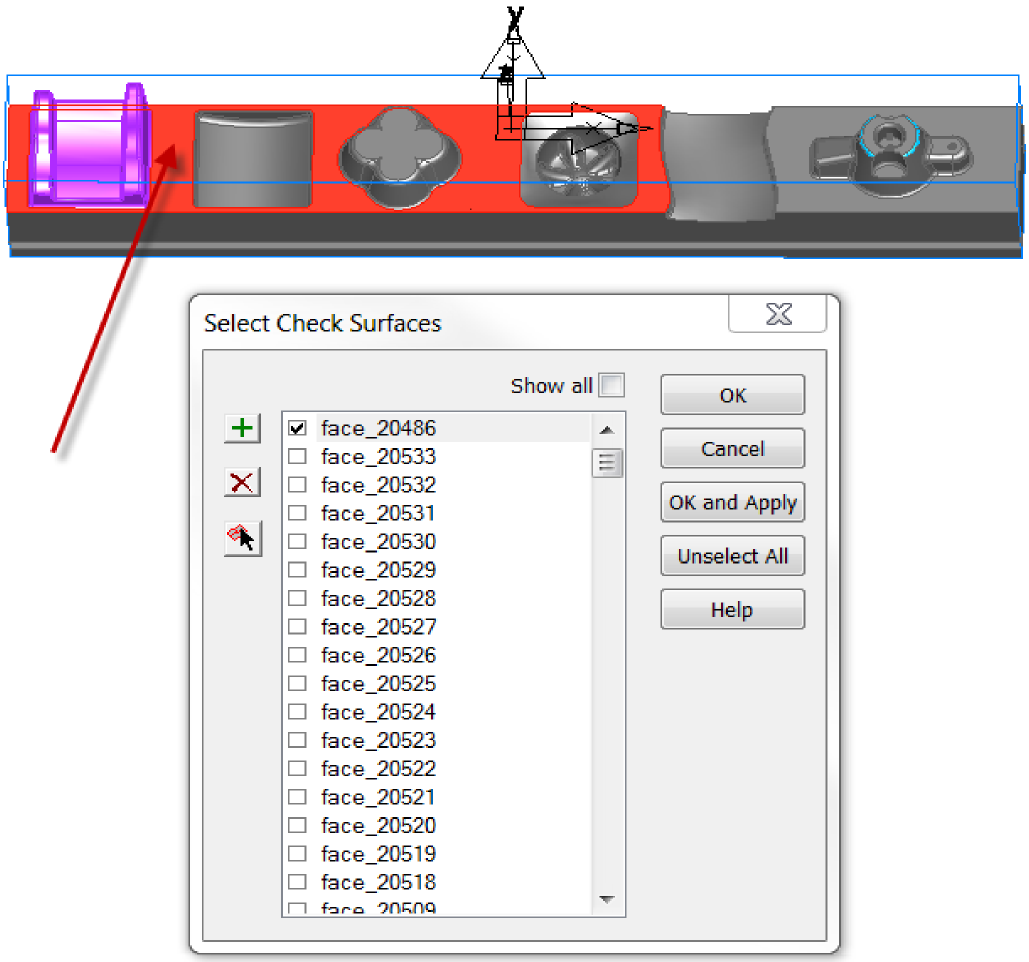
25 Select Next and then Parallel Finishing.

26 Select Finish. Then Double click on the new Surface Machining Feature in PartView. Rename to Parallel.

27 We could check the bottom surface as a way of preventing the toolpaths machining below this surface. Select Check Surfaces.

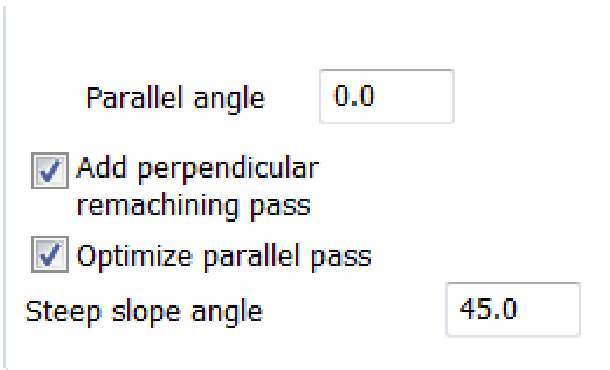


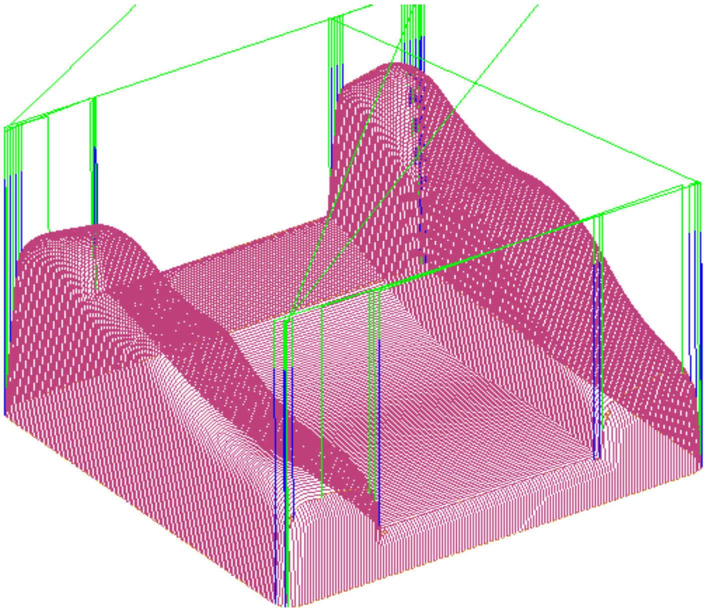
28 Pick the following surface as shown on the next page.



*Another useful option is to select **Add perpendicular Remachining pass** and **Optimise Parallel pass**. This will produce a better finished product.*

Please experiment with different angles to achieve different results.

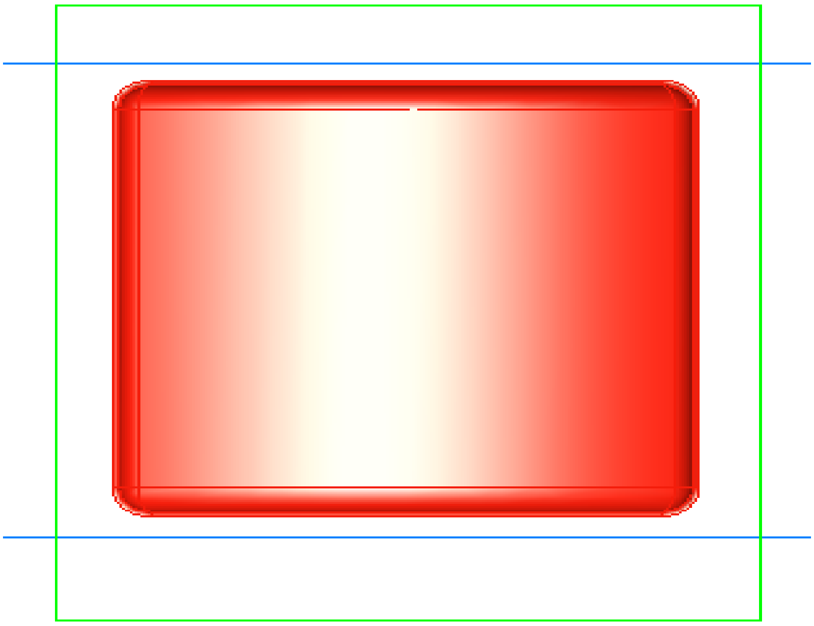




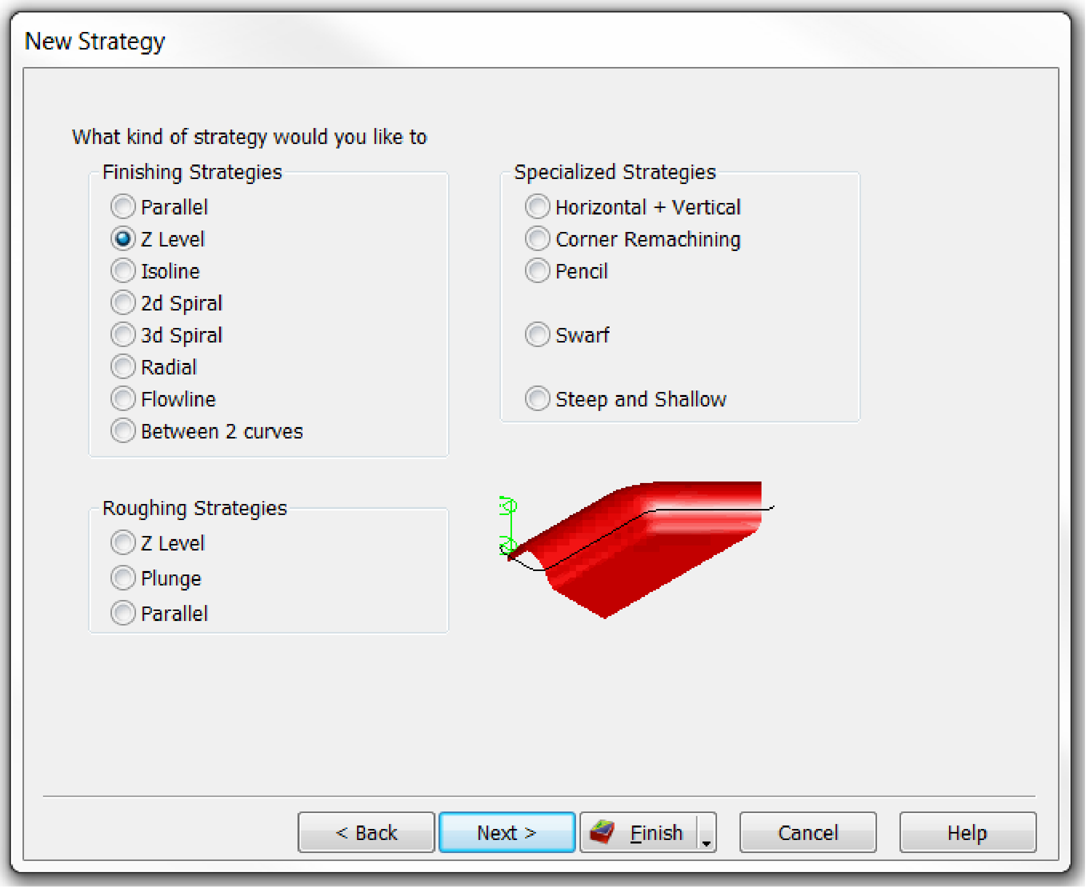
Z Level Finishing

*Working from left to right on our model we will now use **Z Level Finishing** to machine our part. You may decide to do a pre-roughing operation first to take away the material away in the corners first.*

1 First of all select the surface that you want to machine. You know how to do this by now. See the image below.



2 Create a new Feature Ctrl+R then Surface Milling. Select Next Twice then choose a Single Operation and then select Next. Select Z Level Finishing.

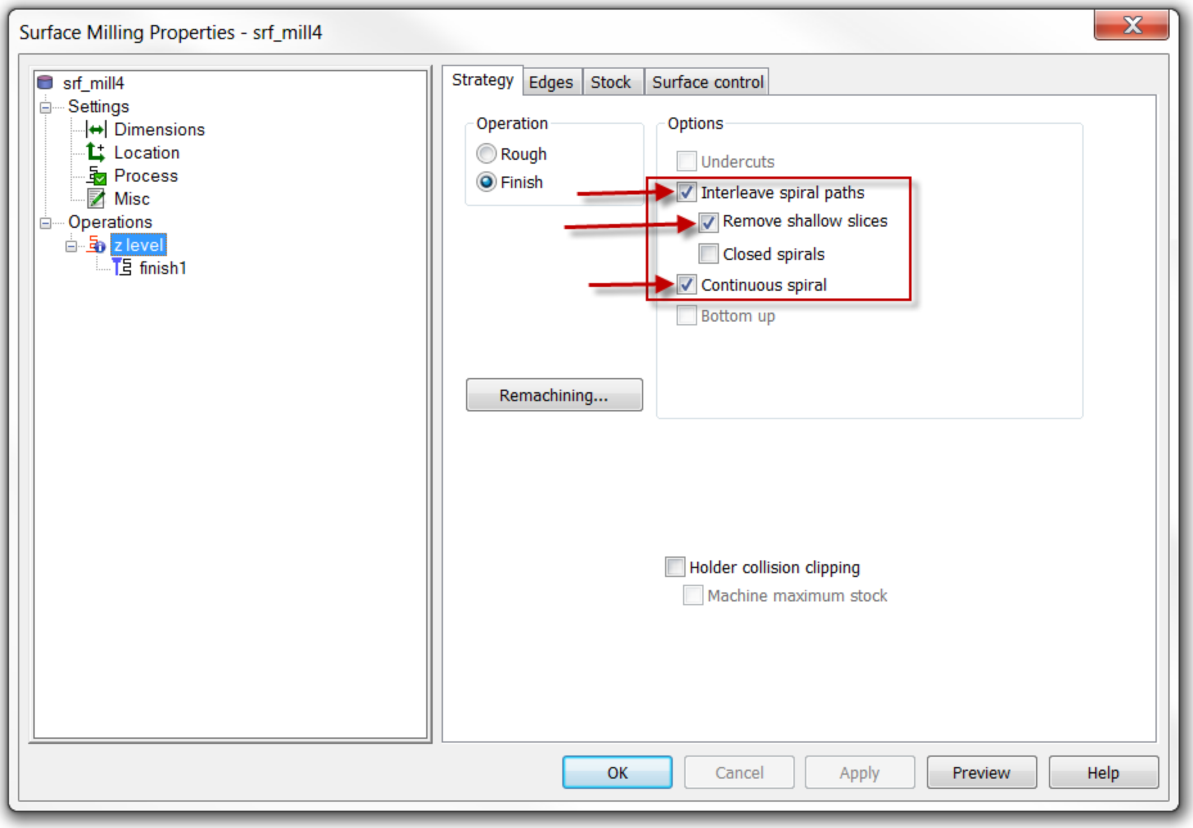


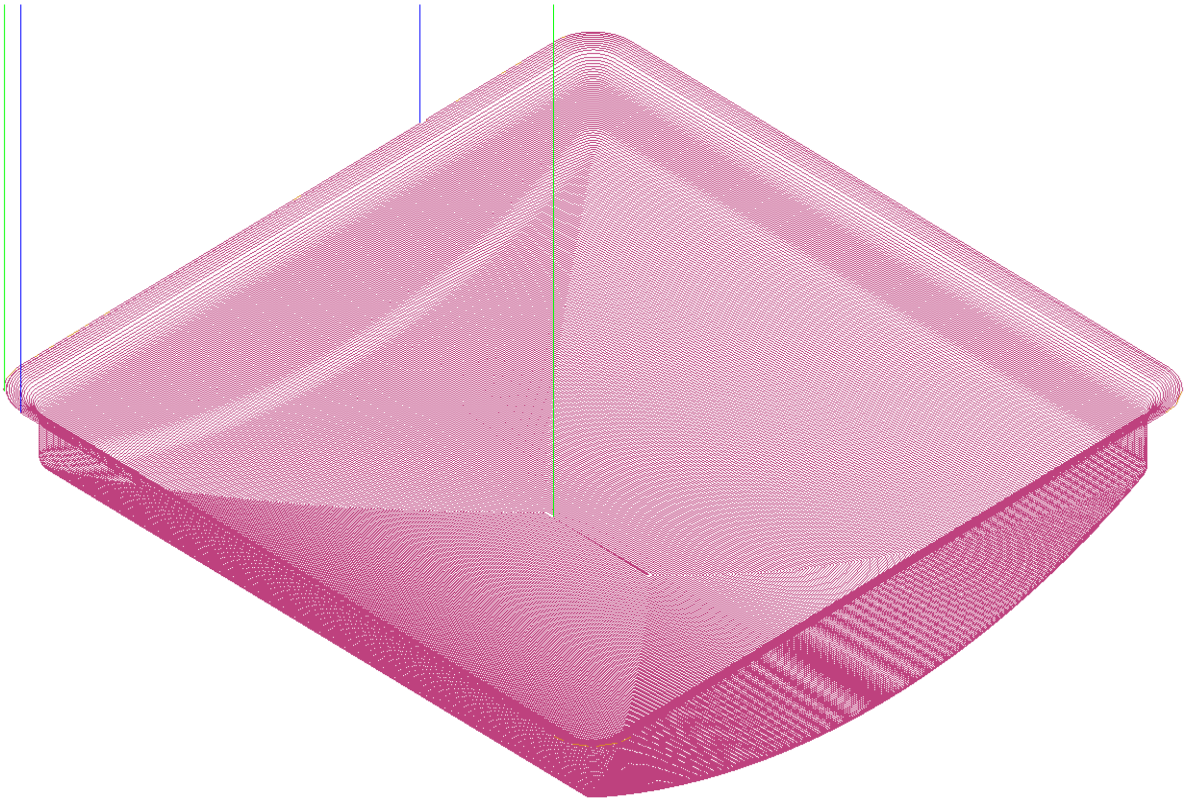
3 Select **Finish**.

4 Rename this operation to **Z Level Finishing**.

5 Double click on **Z Level Finishing** in **PartView.**

6 Then select **Z Level** and then select the following options. Select Apply and then see Results below.


Surface Milling Properties - srf_mill4
srf_mill4
Settings
Dimensions
Location
Process
Misc
Operations
z level
finish1
Strategy
Edges
Stock
Surface control
Operation
Rough
Finish
Options
Undercuts
Interleave spiral paths
Remove shallow slices
Closed spirals
Continuous spiral
Bottom up
Remachining...
Holder collision clipping
Machine maximum stock
OK
Cancel
Apply
Preview
Help

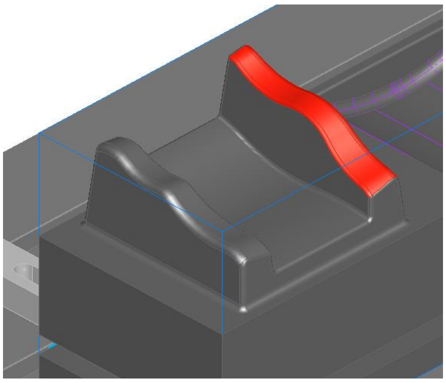


Isoline

Isoline machining works on single surfaces. Even if selected as part of a group of surfaces. Direction is set to ***None.*** *This will give a continuous toolpath. Please set either Row or Column for cut direction for each surface selected.*

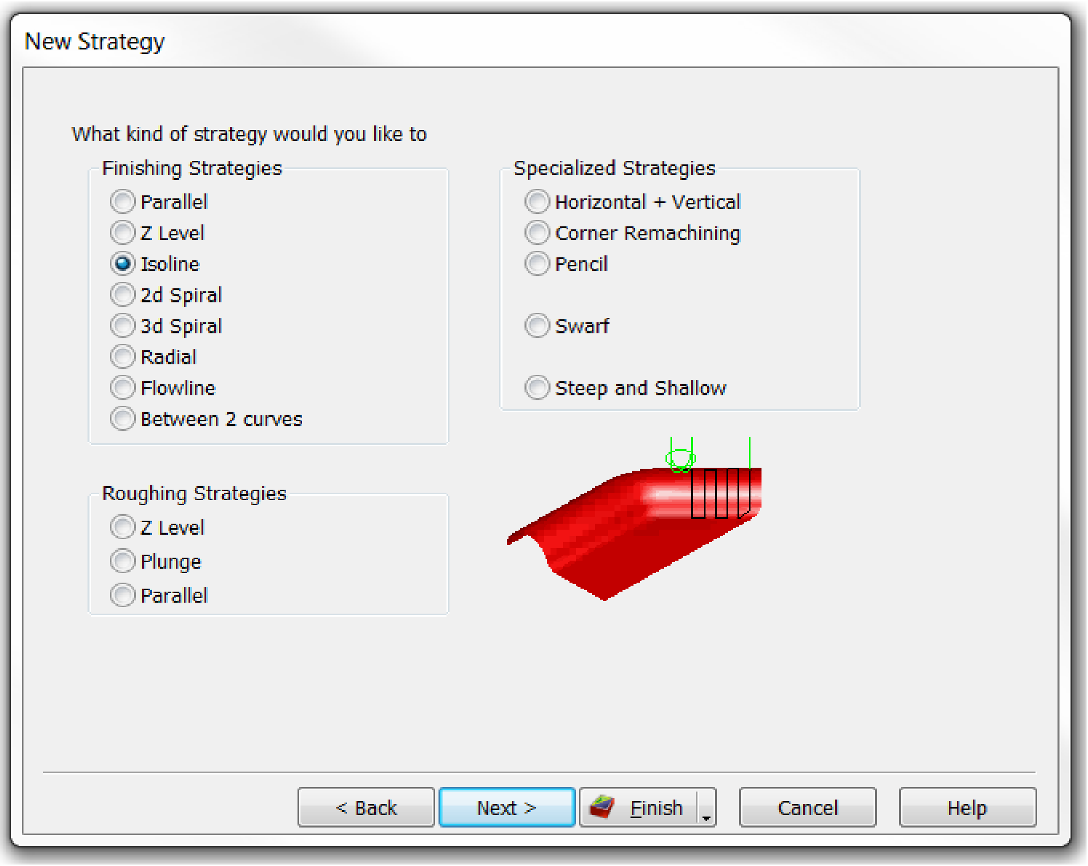
The following is an example of machining selected surfaces using the Isoline surface machining strategy. It is more productive to have surfaces as one complete surface rather than individual surfaces.

Please use Powershape to make individual surfaces into one surface. Please select the following ***3*** *surfaces.*

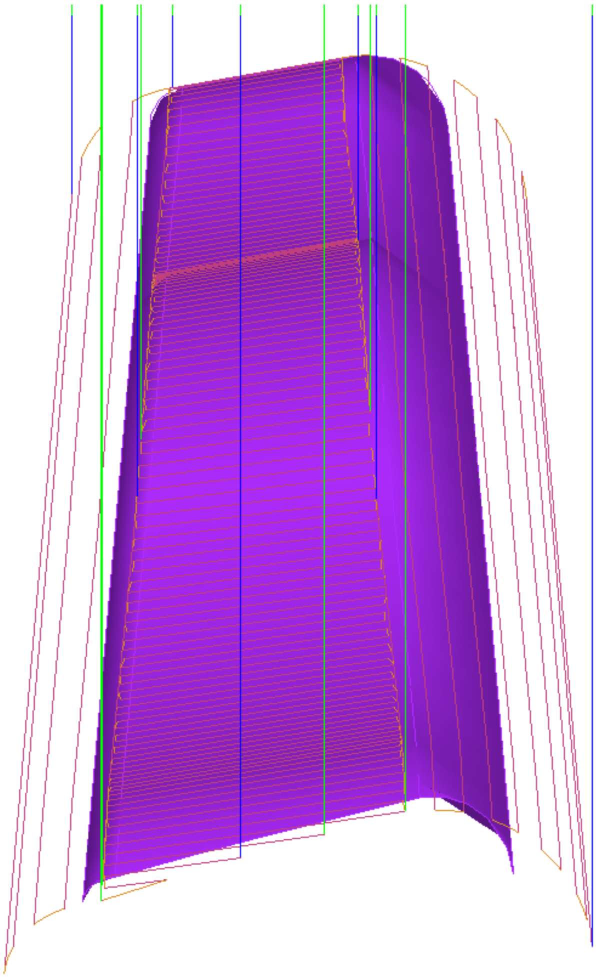


7 Create a new **Surface Milling** Feature. You know how to do this by now.

8 Select **Isoline**. Then select **Finish**. Rename operation to **Isoline**.

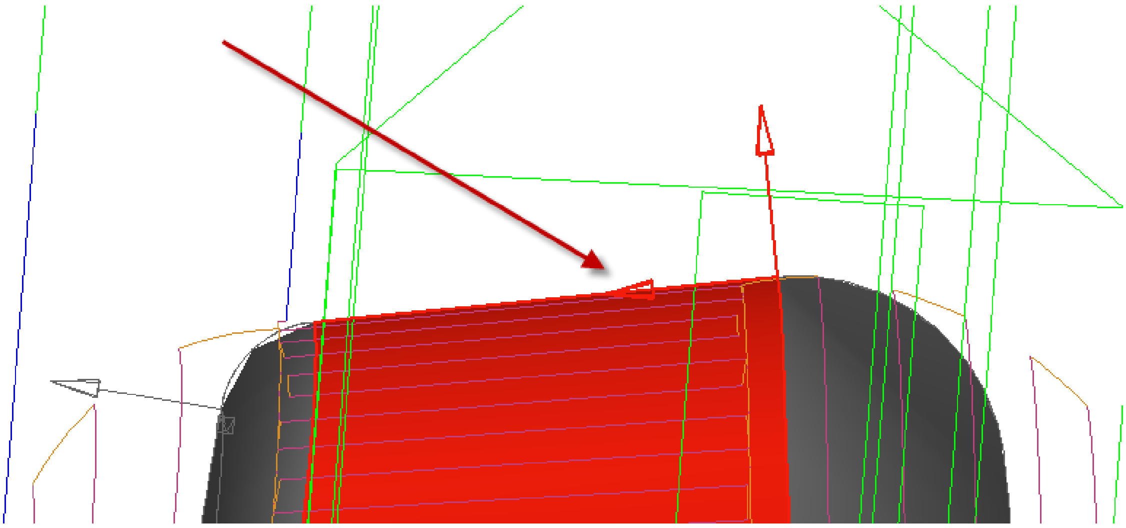


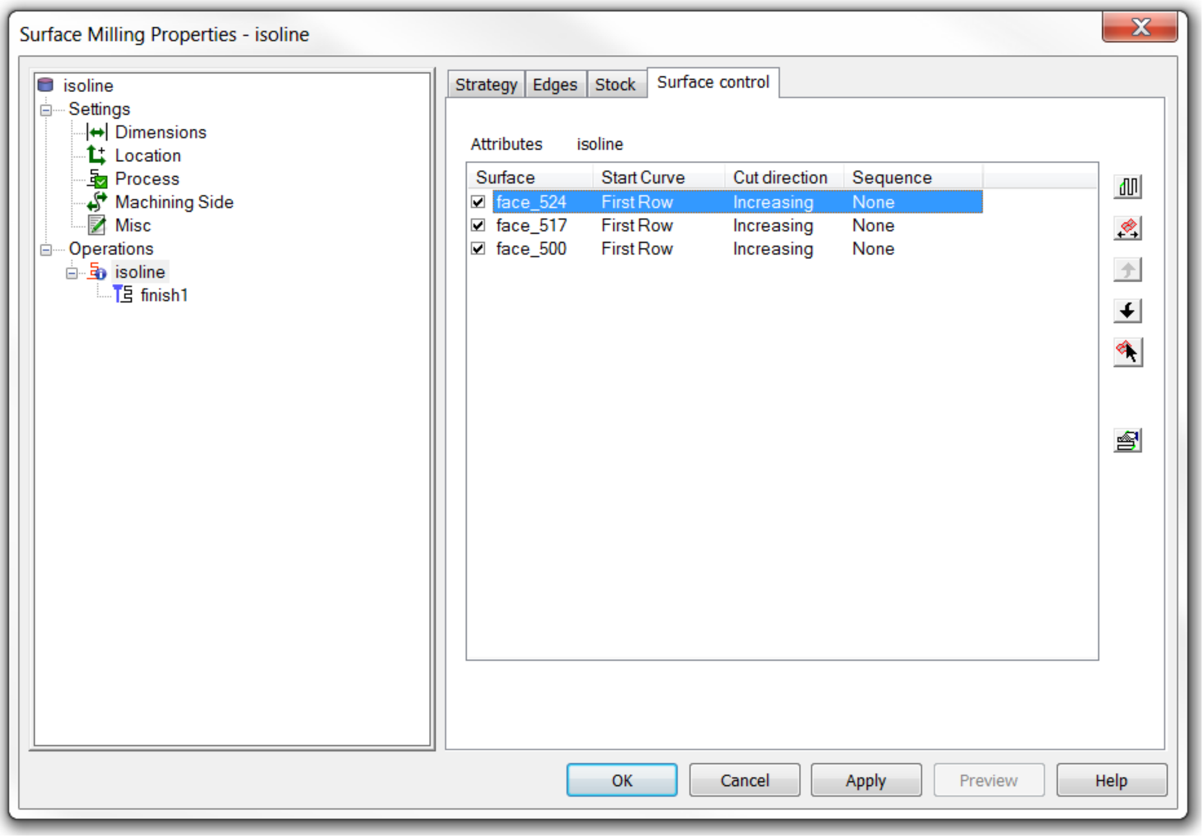
The following image shows the directions of cut for each surface. In order to create a uniform toolpath we need to change the direction of cut on the main surface.



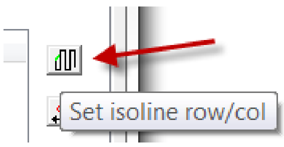
9 Double click on **Isoline** in **PartView**. Select Isoline and Surface Control.

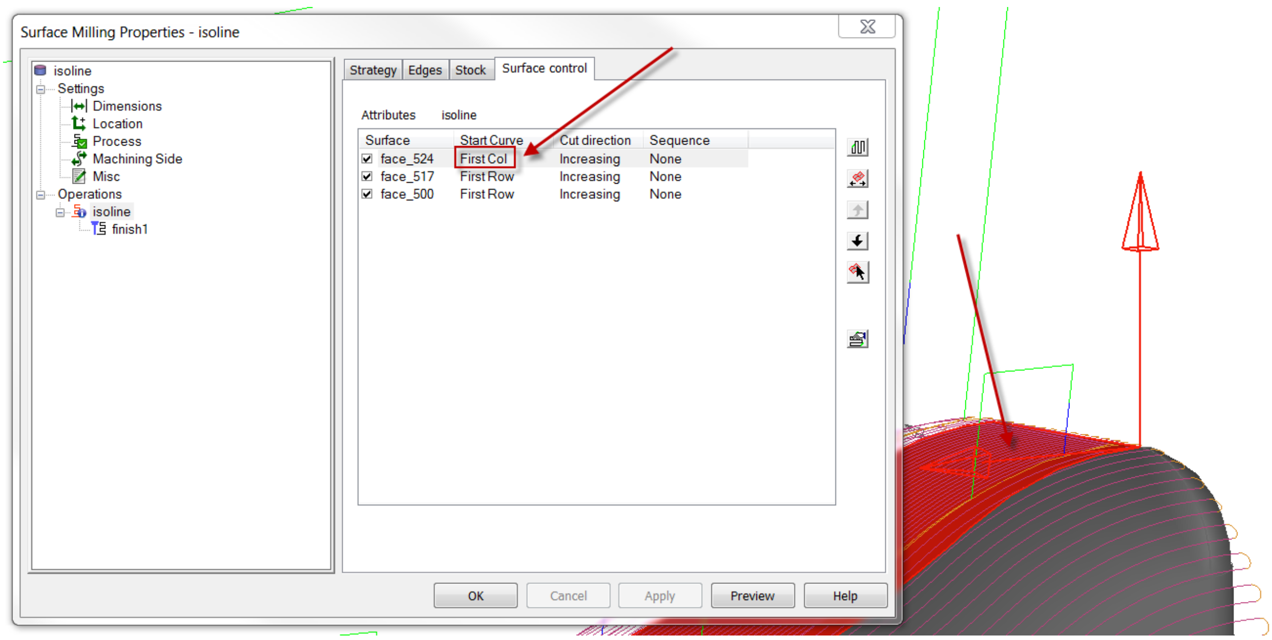
10 You will see the direction of cut as shown below.





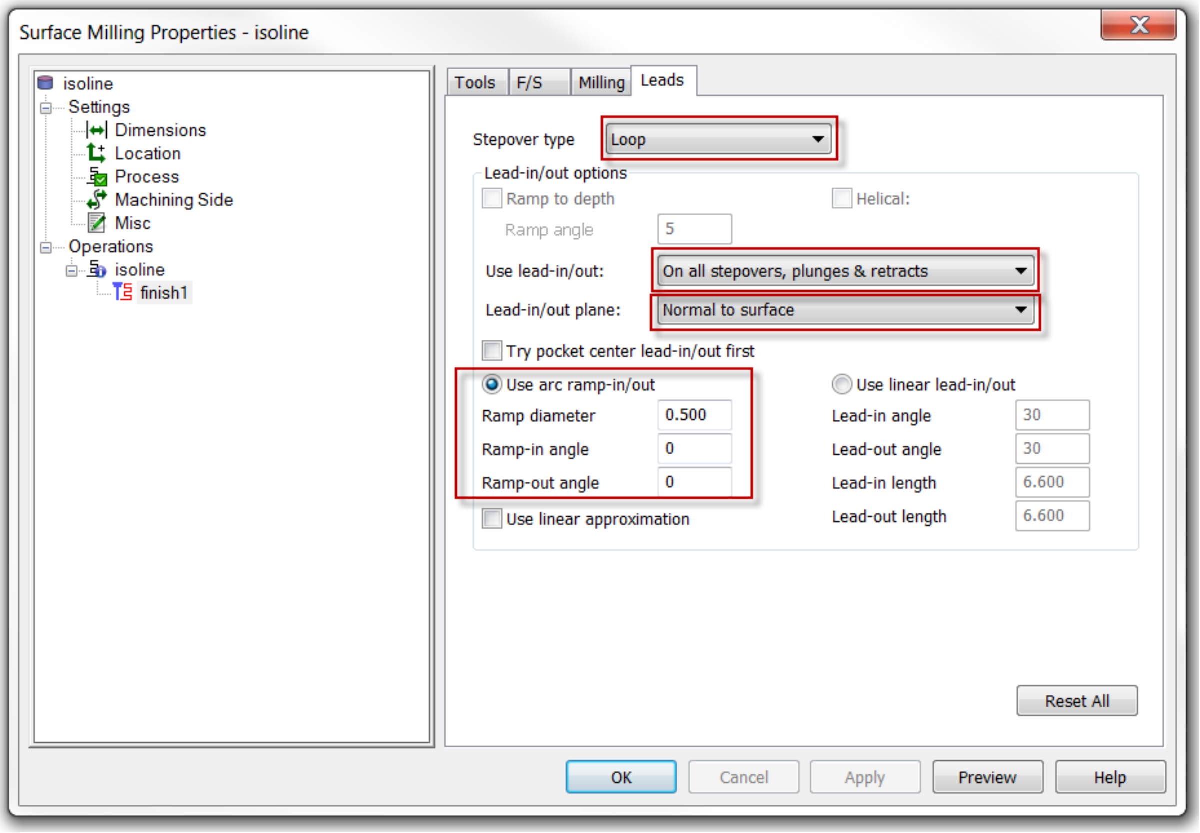
11 Change the direction by selecting the **Set isoline row/col** icon.

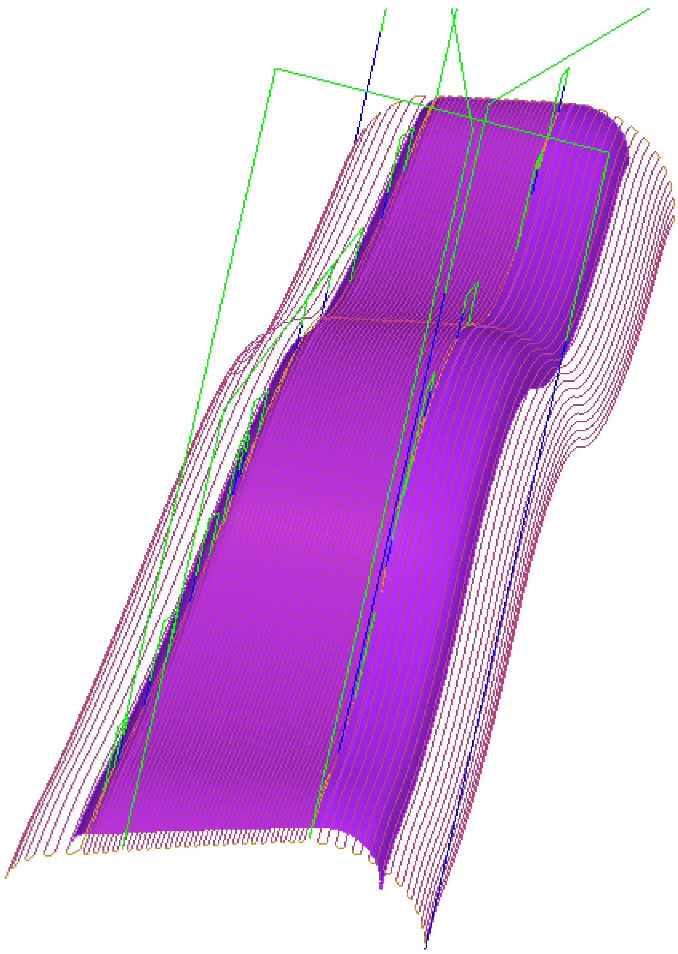




12 We also need to set **Stepover to 0.5mm** under the **Milling** tab and **Leads** to the following as shown on the next page.

The image shown on the next page shows the final toolpath.

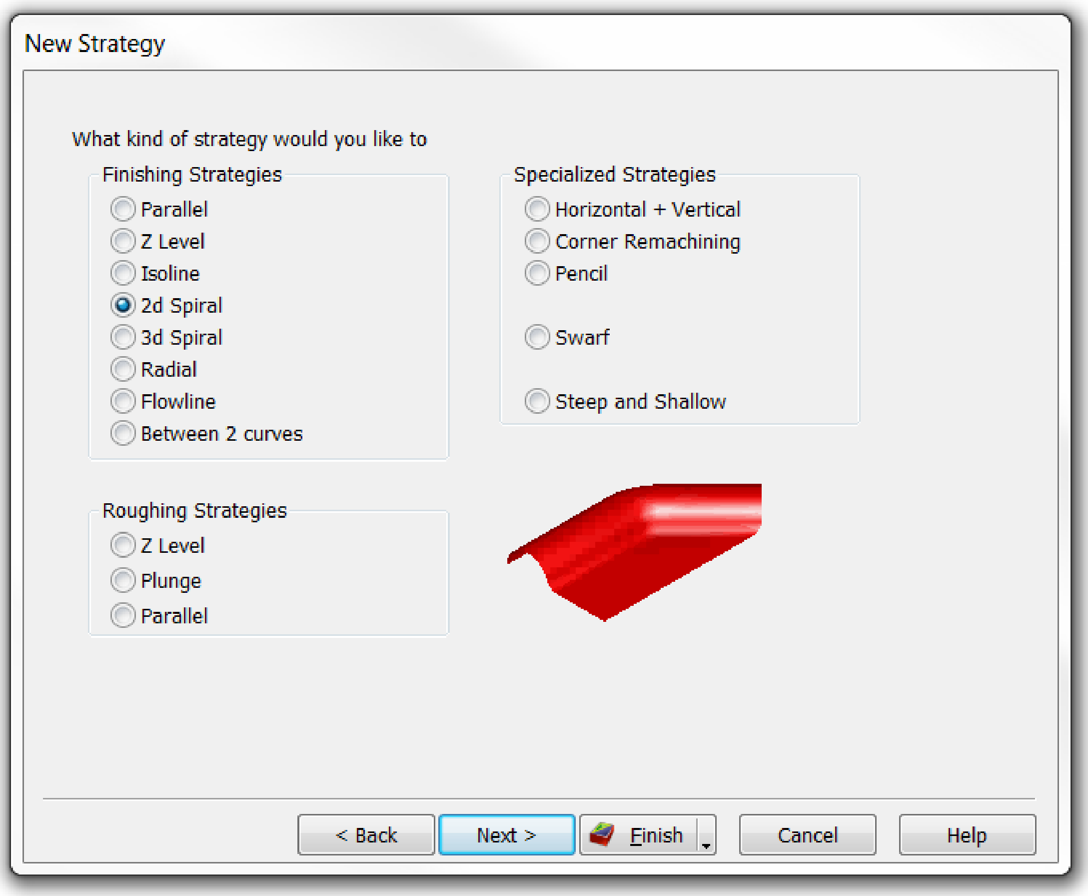

Surface Milling Properties - isoline
isoline
Settings
Dimensions
Location
Process
Machining Side
Misc
Operations
isoline
finish1
Tools
F/S
Milling
Leads
Stepover type
Loop
Lead-in/out options
Ramp to depth
Helical:
Ramp angle
5
Use lead-in/out:
On all stepovers, plunges & retracts
Lead-in/out plane:
Normal to surface
Try pocket center lead-in/out first
Use arc ramp-in/out
Use linear lead-in/out
Ramp diameter
0.500
Ramp-in angle
0
Ramp-out angle
0
Lead-in angle
30
Lead-out angle
30
Lead-in length
6.600
Lead-out length
6.600
Use linear approximation
Reset All
OK
Cancel
Apply
Preview
Help

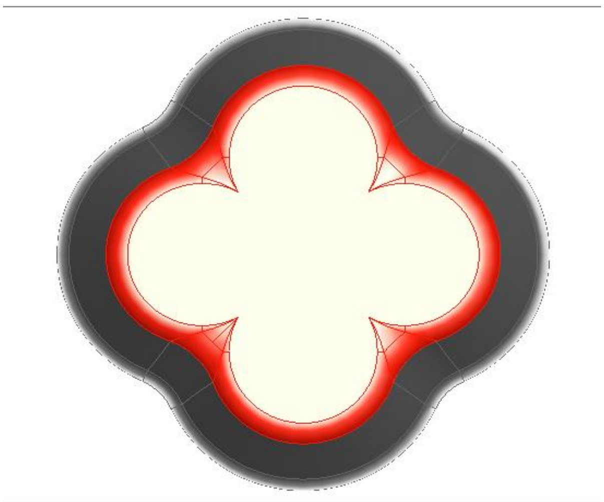


2d Spiral

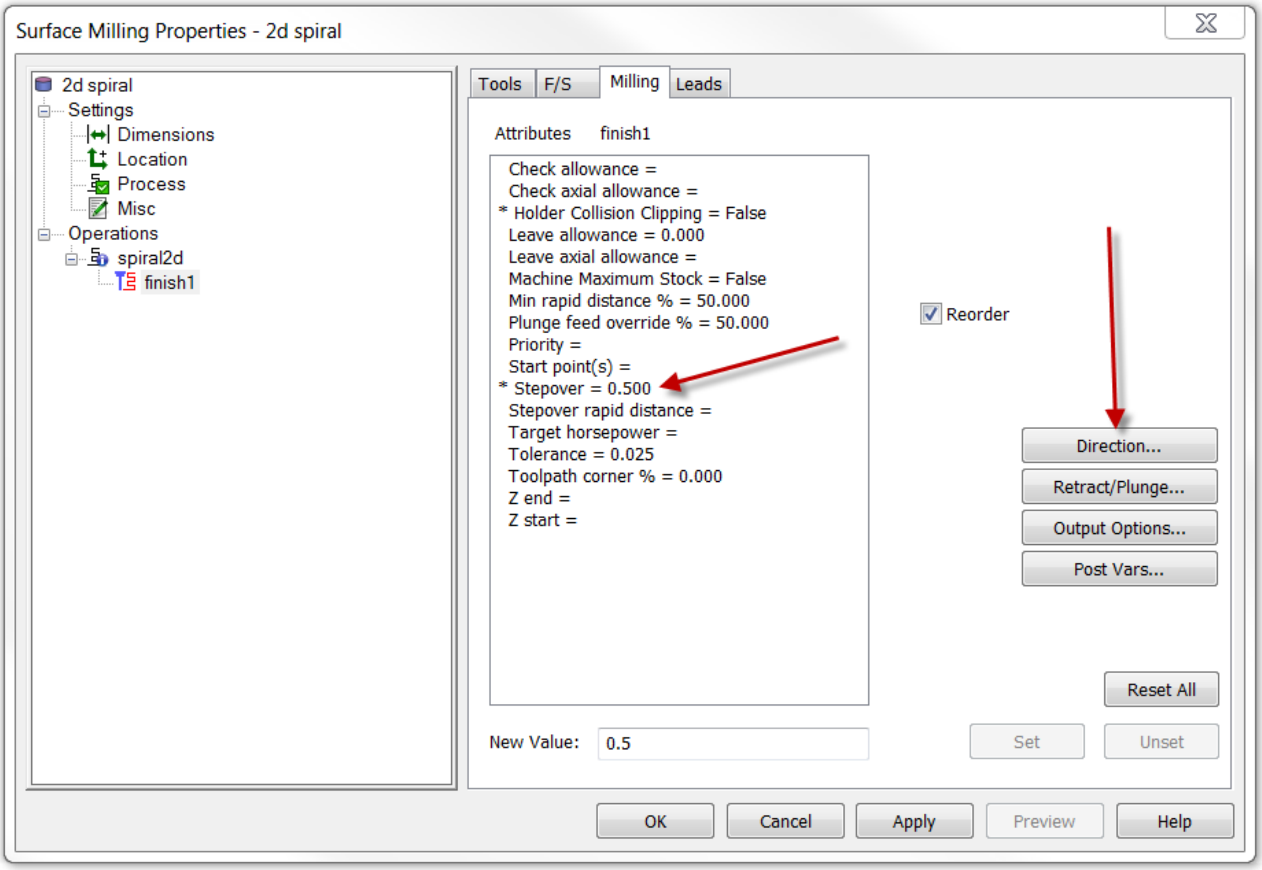
2d Spiral *will mill a feature with a pattern of offset toolpaths. The pattern is obtained by taking the stock boundary, the feature boundary or the curve specified on the Stock tab and offsetting this curve toward the centre of the part. The pattern can be cut either towards the feature centre or away from the feature centre.*

1 Please select the following surfaces. Then create a new feature. Surface milling. Select **2d Spiral** and select **Finish**. Accepting all defaults.

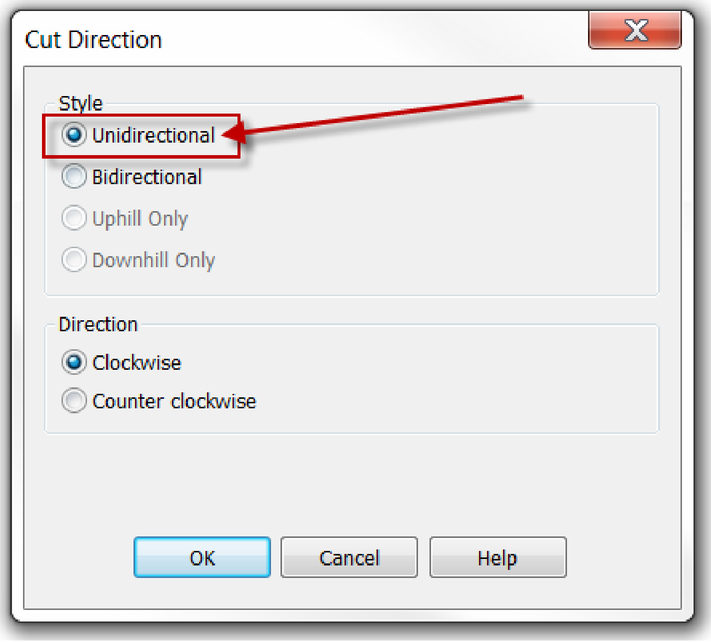


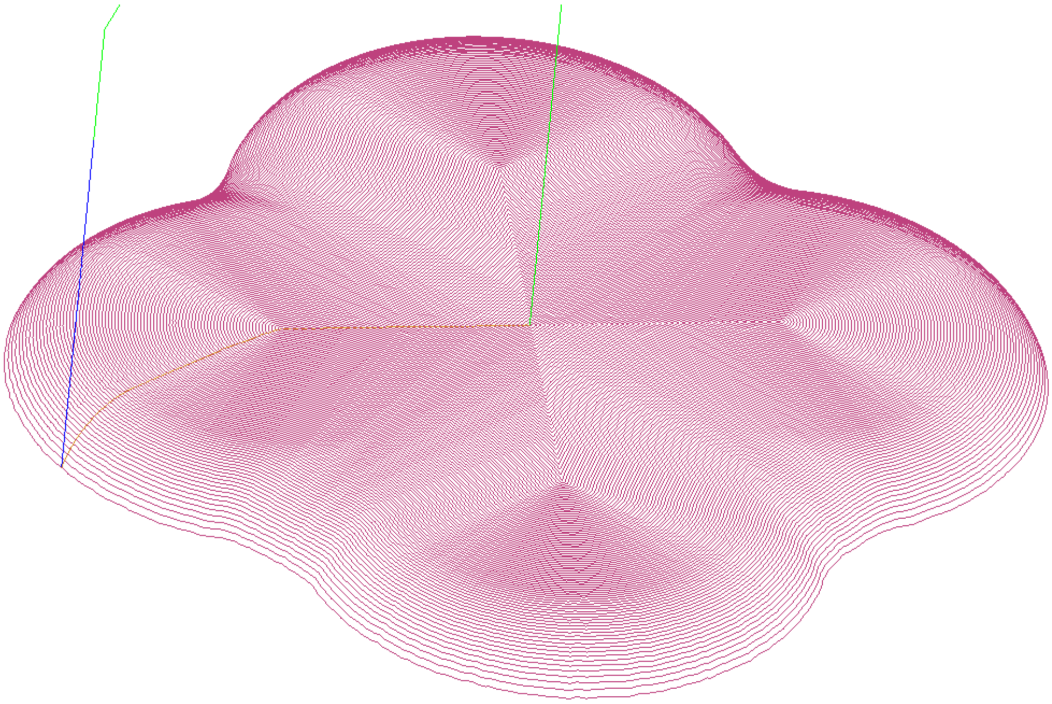


2 **Rename** the operation to **2d Spiral**. Double click on **2d Spiral** and change the **Stepover** to **0.5mm** under the **milling tab** and then select **Direction**.



3 Change **Direction** to **Unidirectional**. See results below.



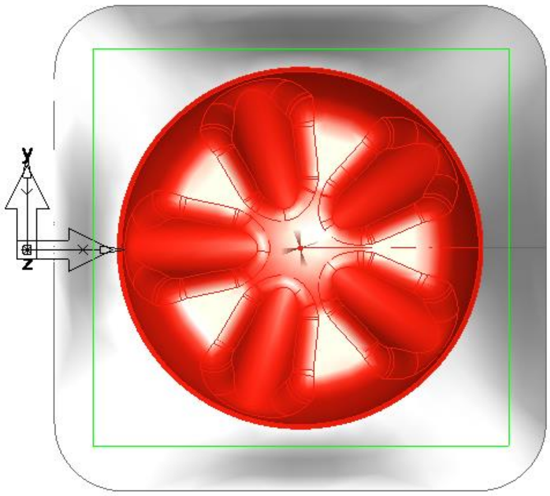


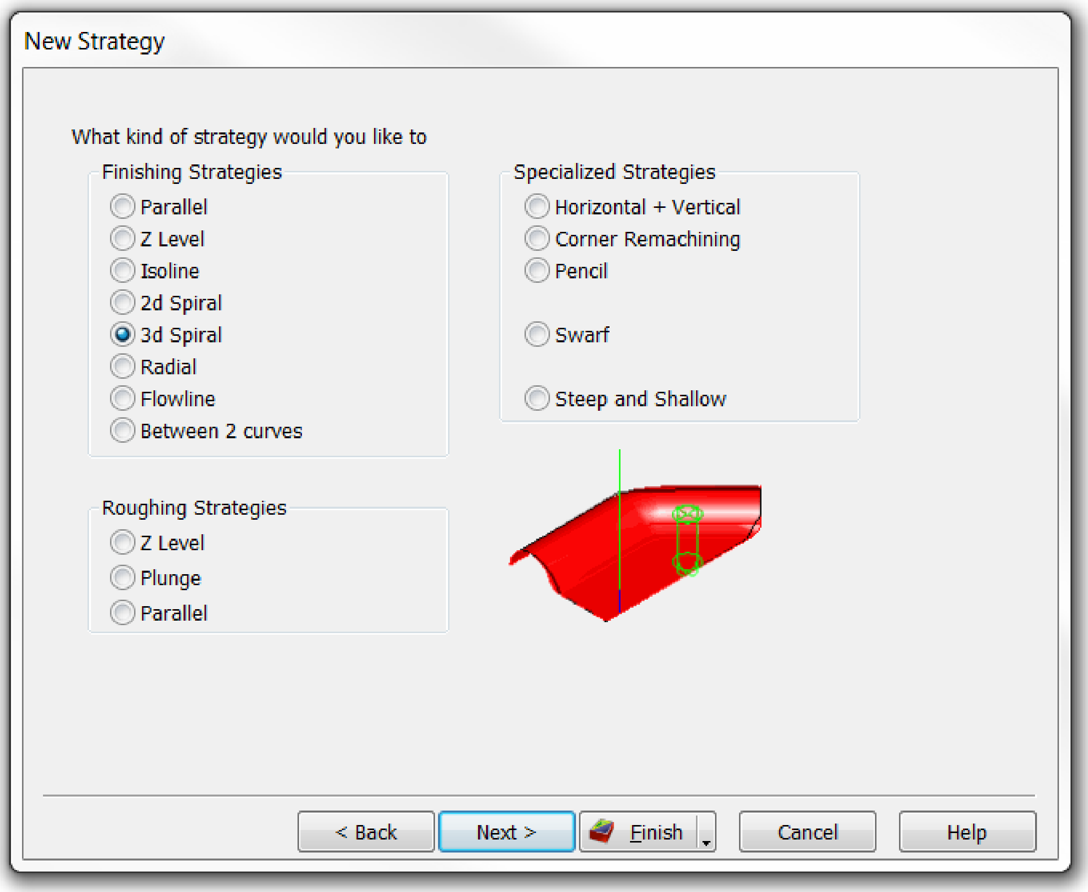
This will make the toolpath a continuous toolpath.

3d Spiral

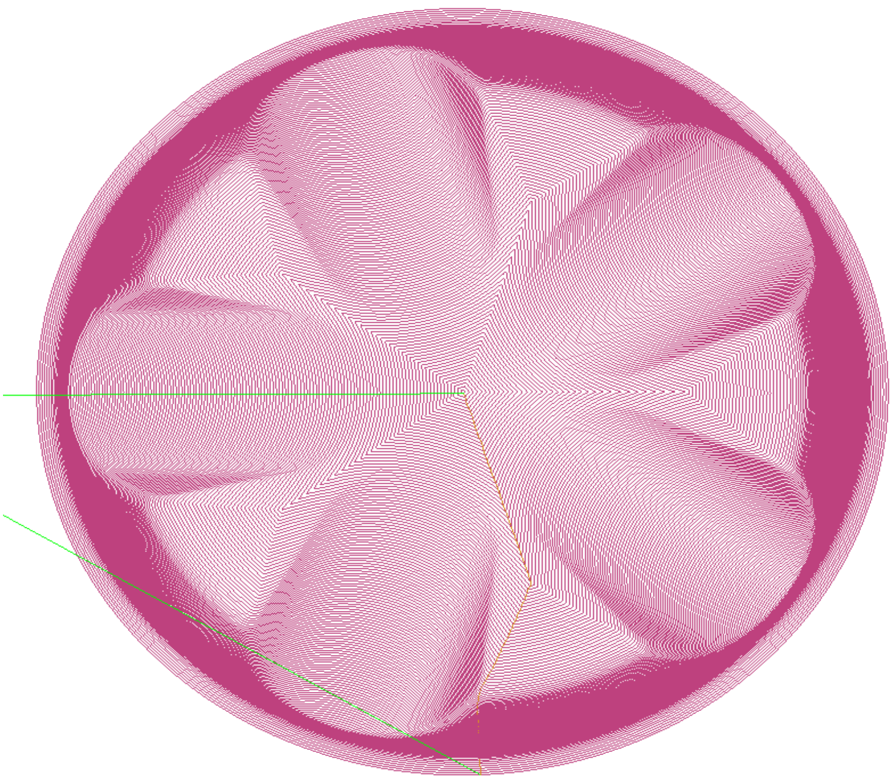
3d Spiral This finishing technique is best suited to machining areas that need a constant stepover and works well on near-horizontal faces. Spiral toolpaths mill a feature in a series of offsets towards the feature centre. The initial pattern is specified by taking the stock boundary, the feature boundary, or the curve specified on the Stock tab.

1 Please select the following surfaces. Then create a new **Surface Milling feature.** Then select **3d Spiral.** Select **Finish.**





Edit the toolpath and change the stepover to 0.5mm

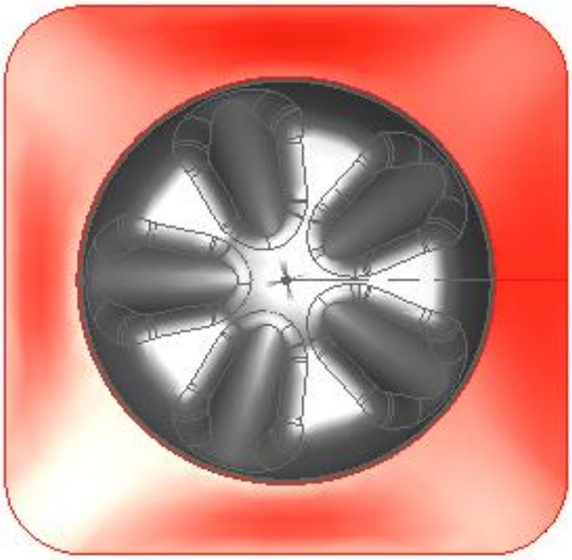


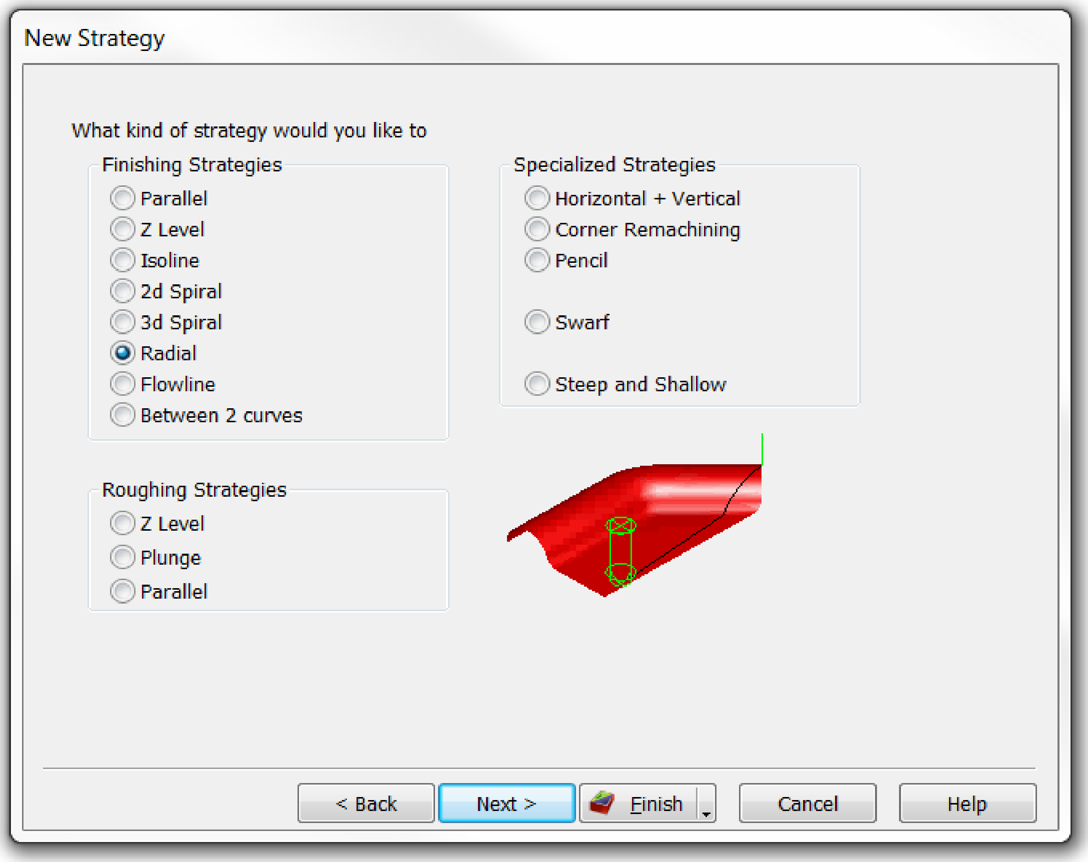
Radial

***Radial: -** This strategy creates a radial pattern within a boundary and projects it onto the model.*

***Centre point**. The centre of the radial pattern is calculated automatically unless you set a **Centre point**. This point is projected down onto the surface to become the centre of the radial pattern.*

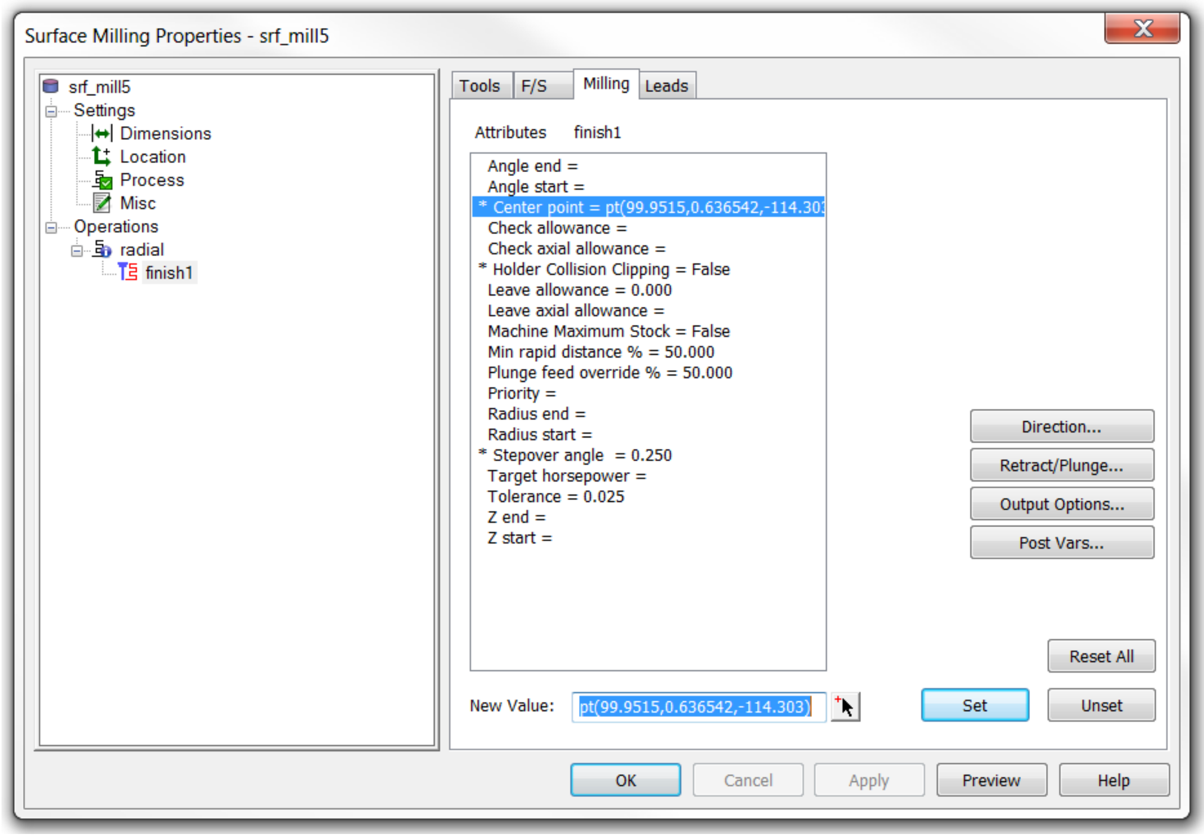
1 Select the following Surface. Then create a new **Surface Milling Feature**.



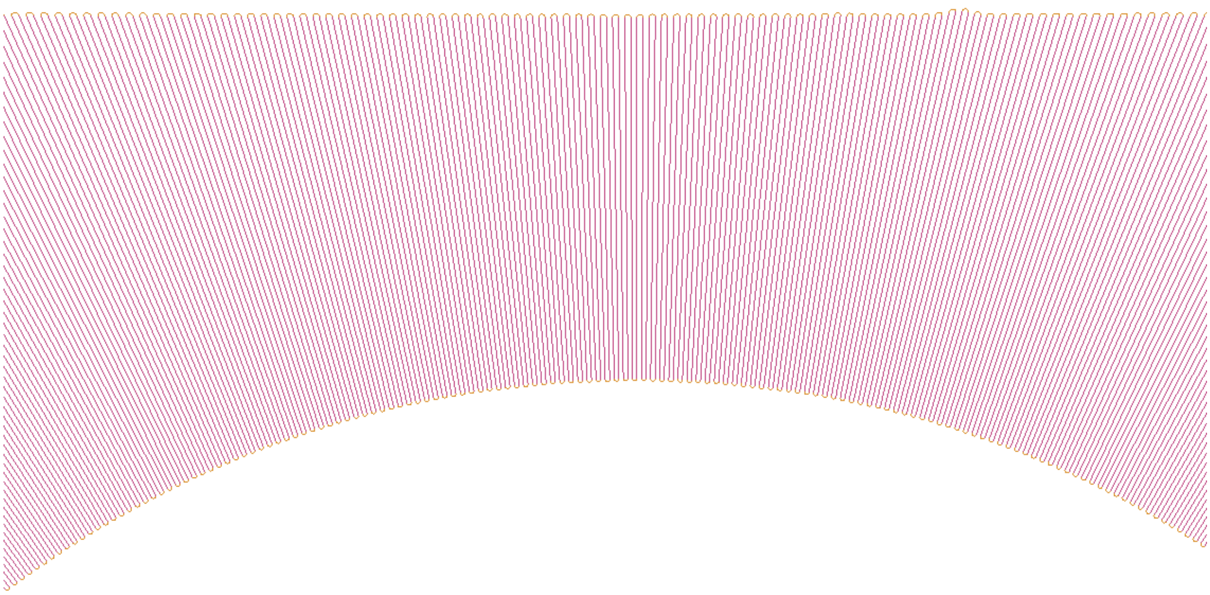


2 Select **Finish**.

3 Please select the **Centre point** of the radial toolpath by snapping onto the cylinder centre point of the radial feature on the solid model.

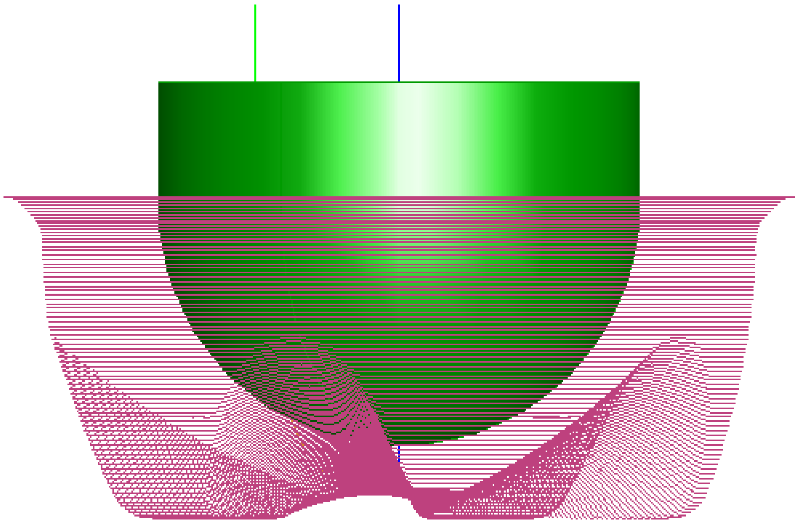


4 Change the **Stepover Angle to 0.25** degrees. See a small section of the toolpath result shown below. Change the Leadin/out to the same as previous settings.

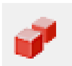


Flowline

The Flowline technique projects the Isolines from one surface onto the surfaces of the feature. The Isolines are projected in the direction of the surface normal.



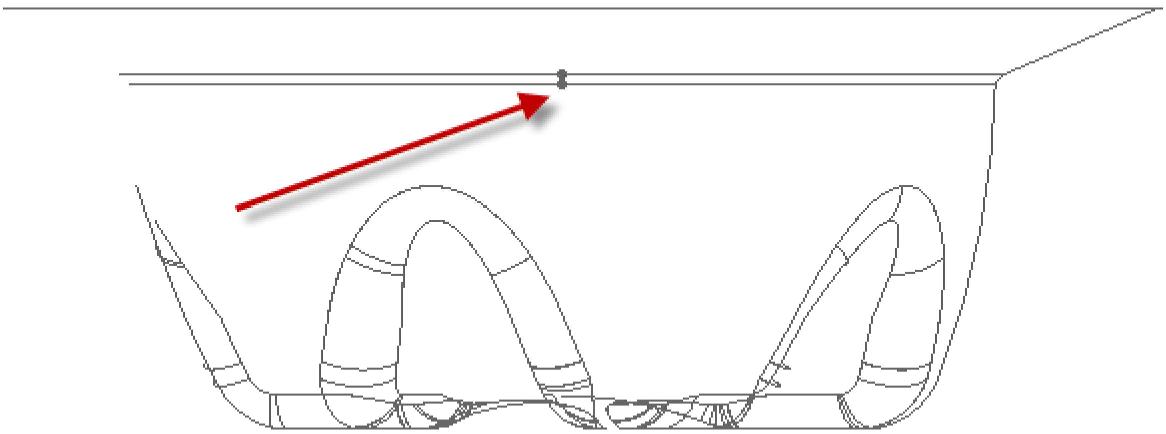
5 Create an arc at the Feature centre for example radius 40mm. Create a vertical geometry line so we can use this for surface of revolution.

6 Select the shade Surfaces Icon  and toggle this so the solid model is transparent and all you will be able to see is the outline of the solid model.

7 Hide the Stock.

8 Zoom into the above feature.

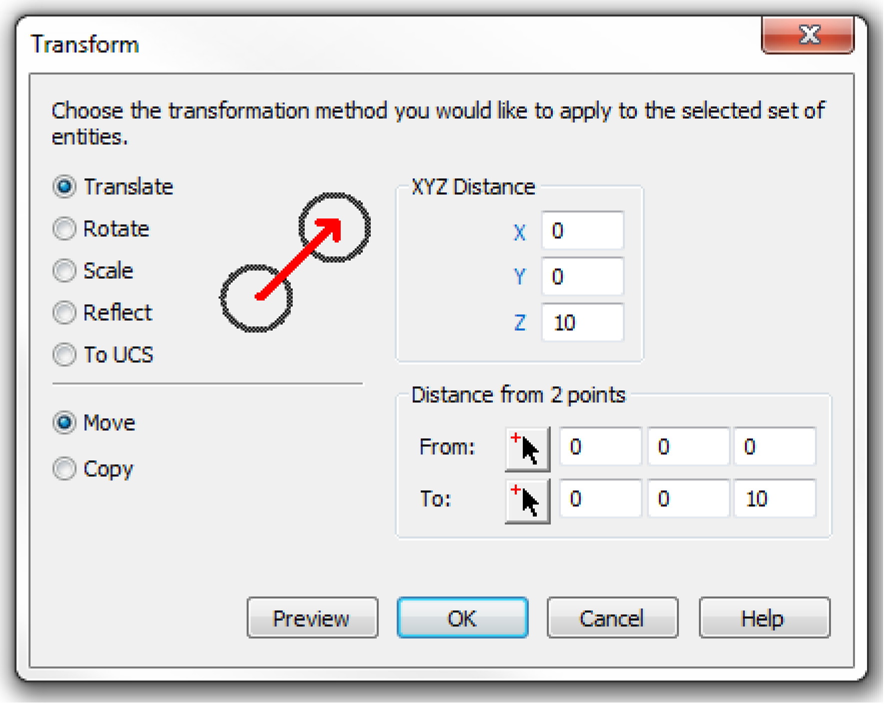
9 Select the Front View.



10 Draw a **2Pt line** between these two points as shown above. You may have to orientate the model slightly to snap to the points. Select Front view again.

11 Select the line and change the length to **50mm**. Select **Modify** to action the change.

12 Select the Line and then right click and select **Transform.** Move the line **10mm** in **Z** only. (1)



13 Select **OK** to action the command.

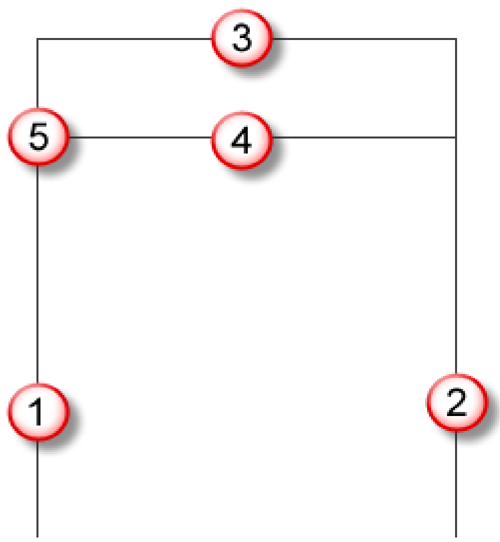
14 Hide the solid model to make things clearer.

15 Offset the line you have just created by **40mm** to the right (2)

16 Then draw a line at the top of the two vertical lines (3)

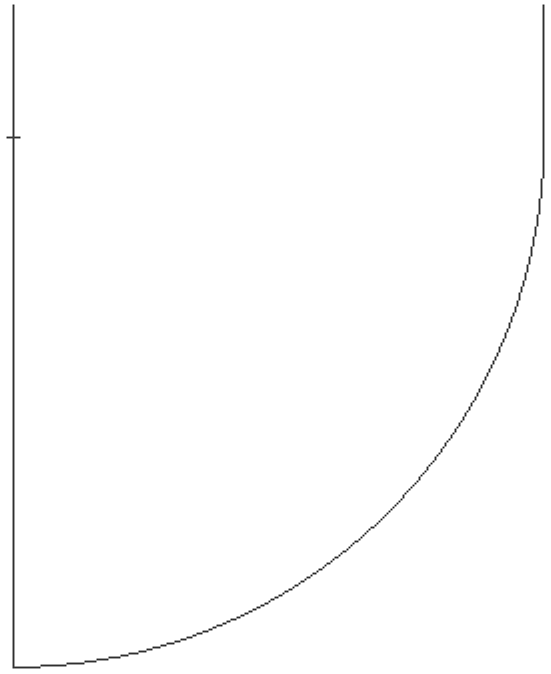
17 Offset this line **10mm** down (4)

The image should look like one shown below.



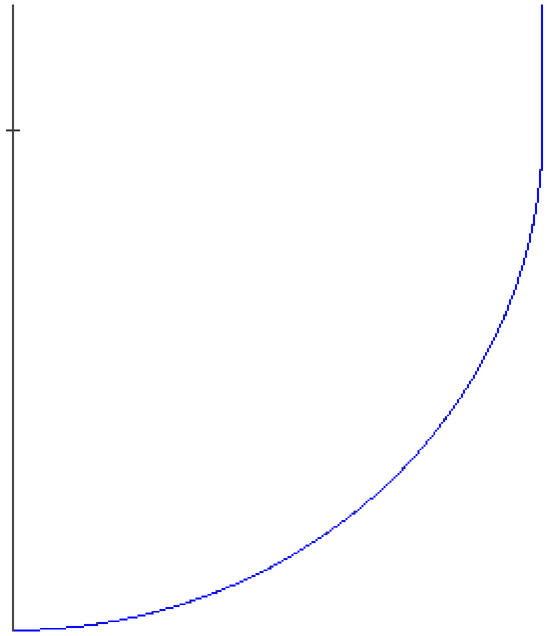
18 Draw a circle with a radius of **40mm** and snap the centre to point (5)

19 Then clip the circle so it looks like the image below.



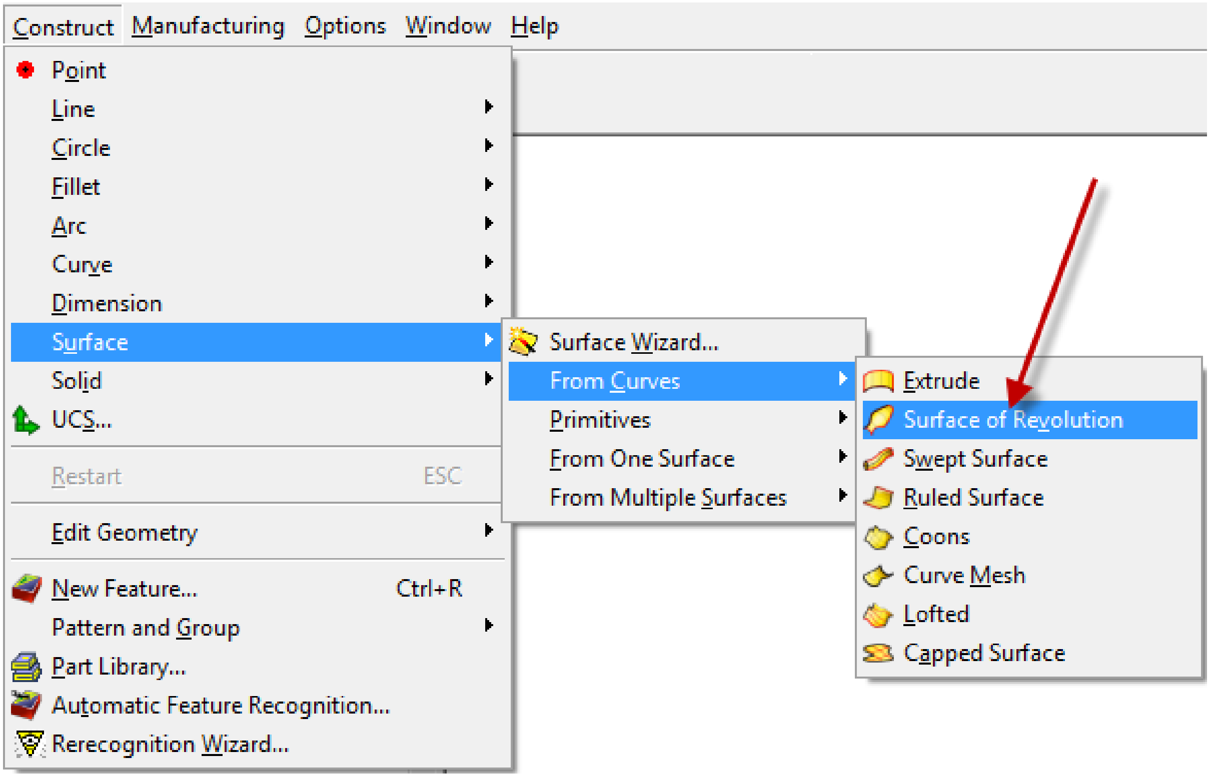
20 Create a **Curve** using **Pick Pieces**.

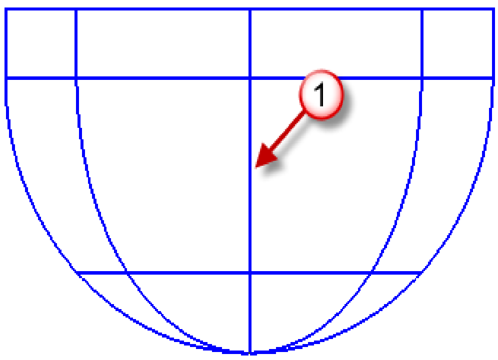
21 Select the line working from top to bottom.

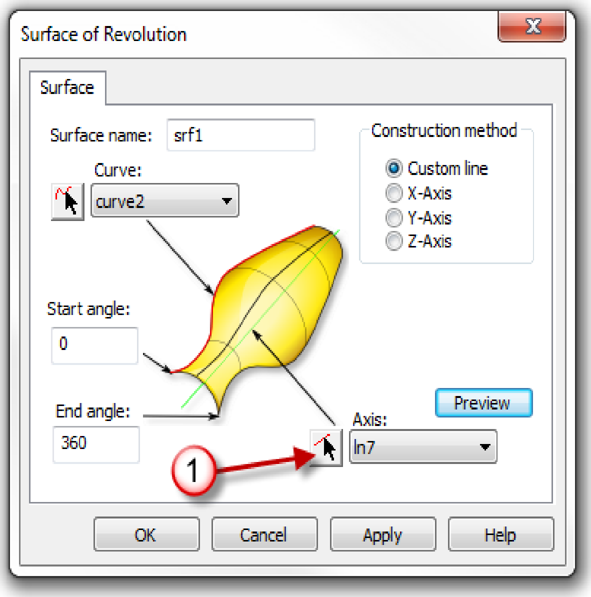


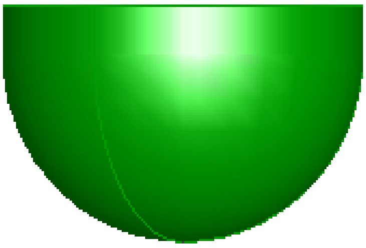
22 Create a Surface of Revolution using the vertical line as a rotation point.

23 Select **Construct>Surface>From Curves>Surface of Revolution**



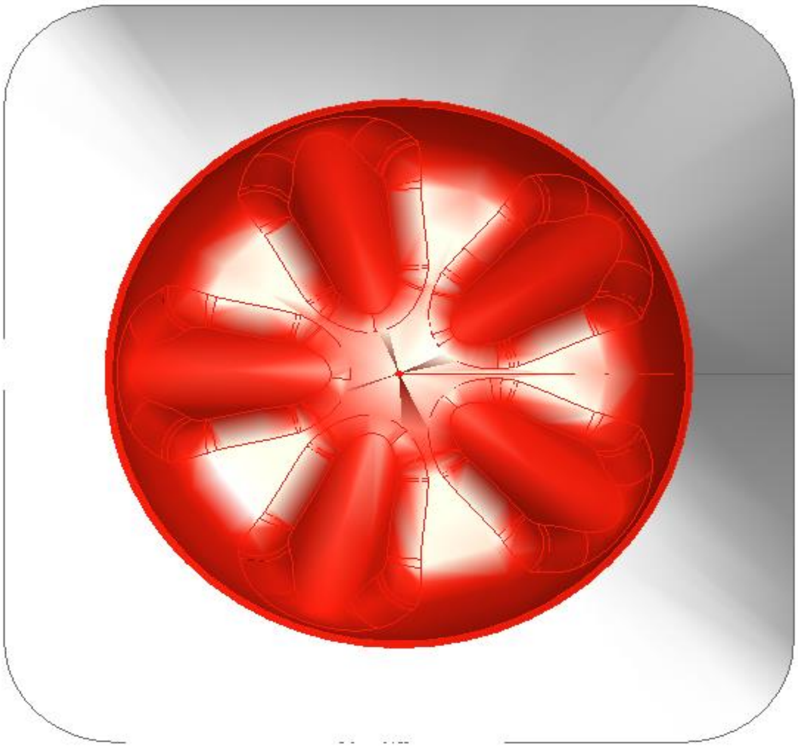




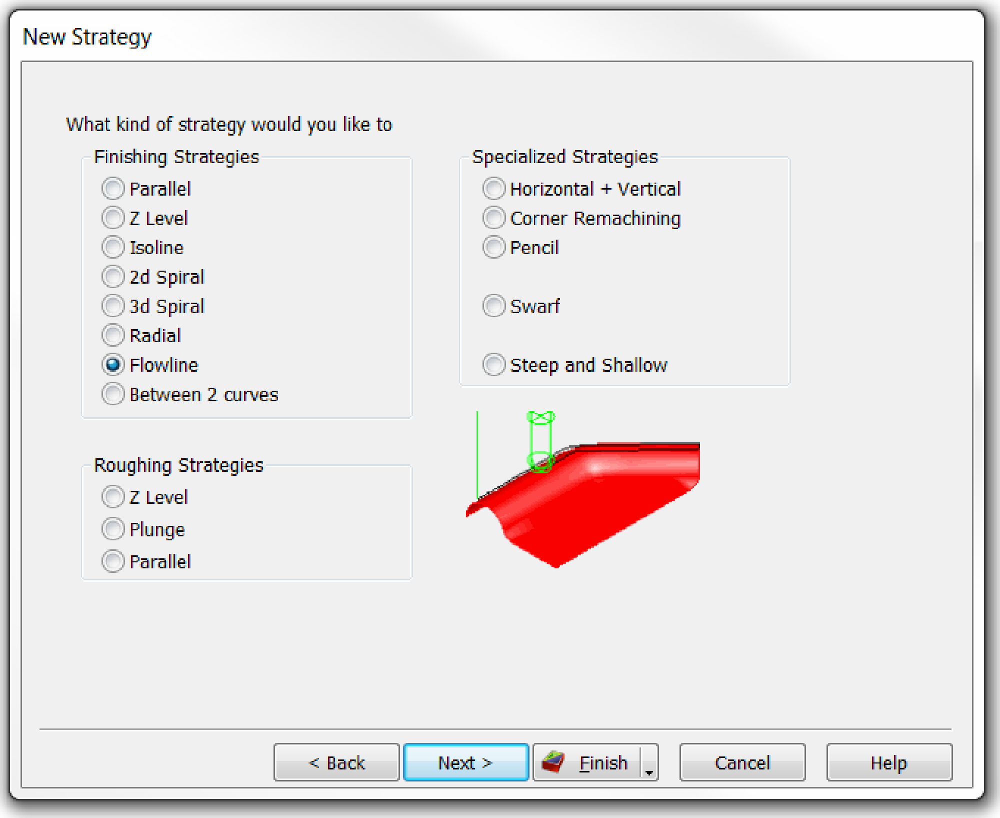


24 Create a Surface as shown. This will be used to project the toolpath onto the selected feature. Rename the surface and call this **Flowline Surface**.

25 Select the features you want to machine.



26 Create a new Surface Milling Feature. Select **Flowline**.

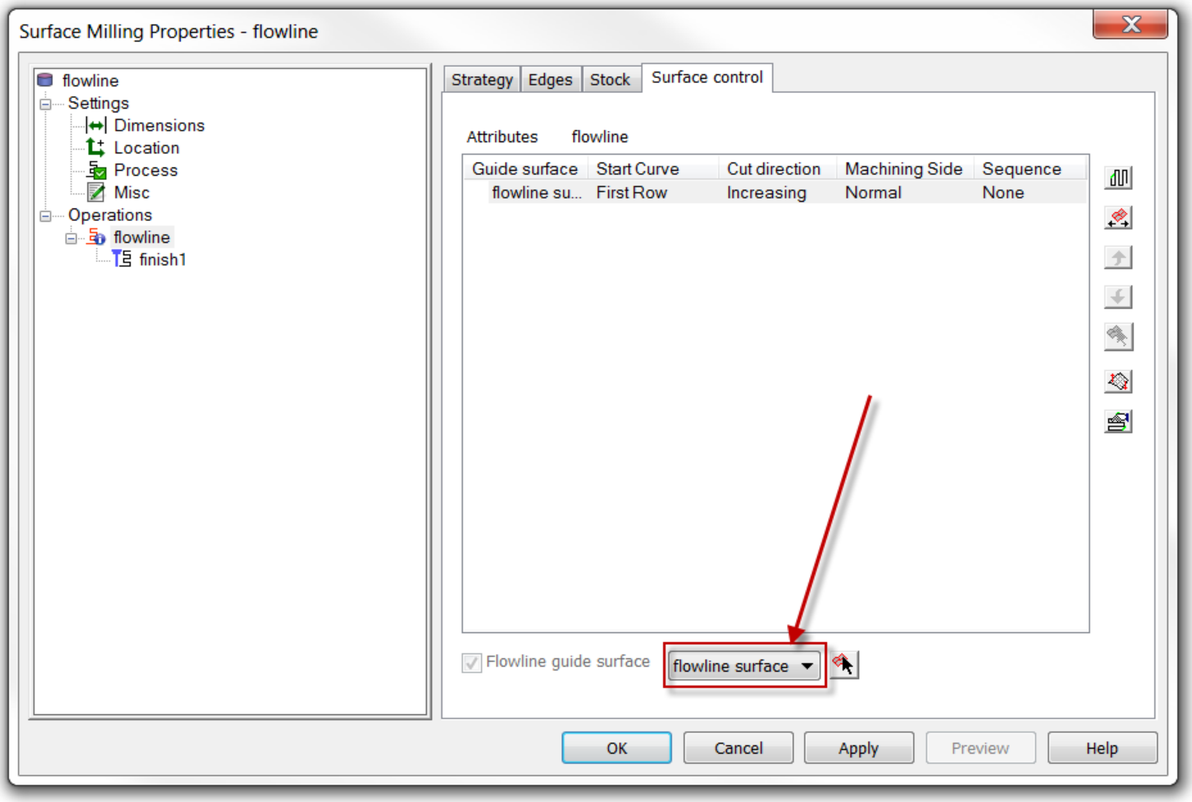


27 Select **Finish**.

28 Rename Feature to **Flowline**.

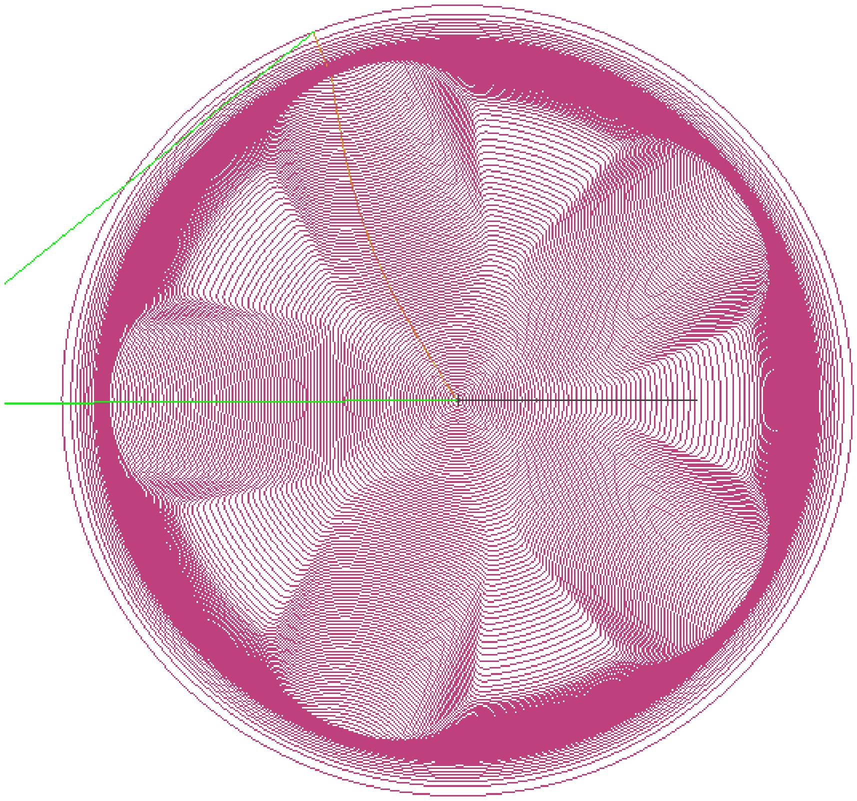
29 Double click on **Flowline** in **PartView**.

30 Navigate to **Surface Control** and pick the **Flowline surface** as shown.



31 Under Milling change the stepdown to **0.5mm**

32 See finished Toolpath below.



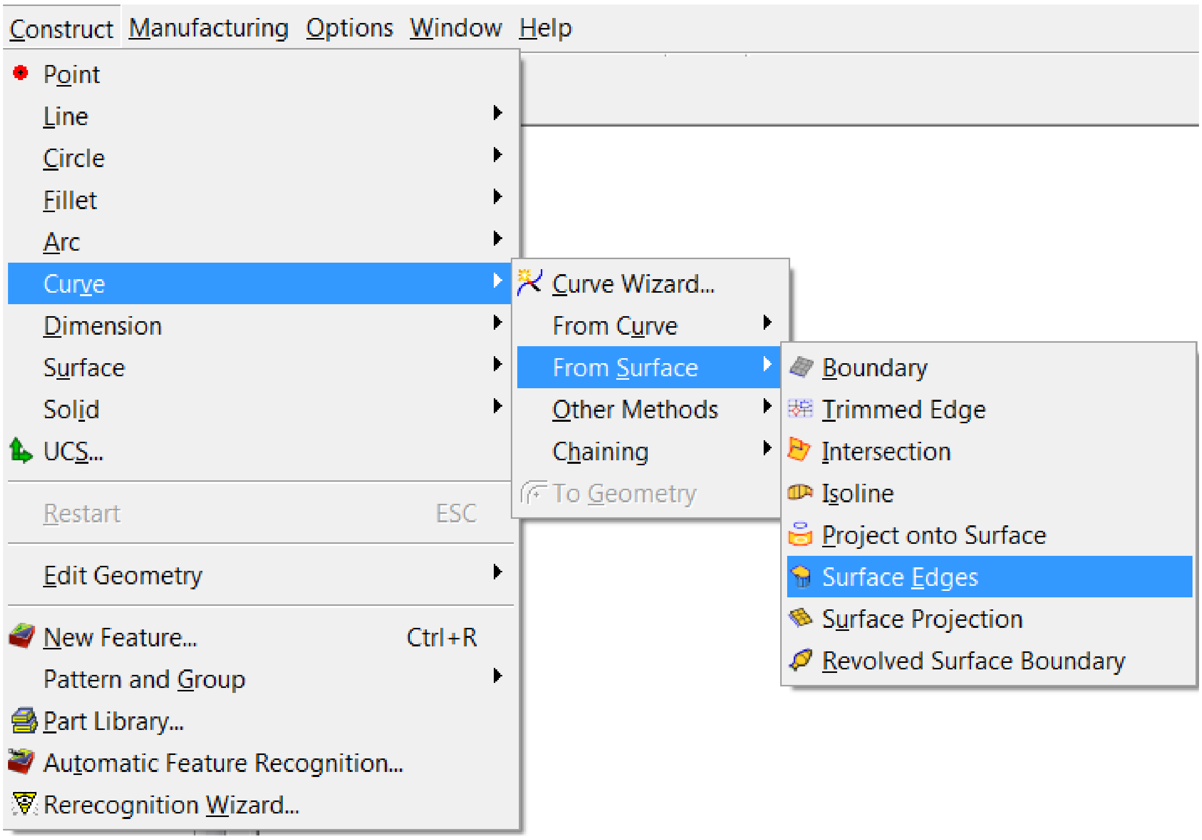
Between 2 Curves

Between two curves calculates toolpaths that are bounded by two curves. This finishing operation is used to control the shape of the toolpath over multiple surfaces by restricting the machining of the surface to be between the two specified curves. The toolpaths mill a feature in a series of offsets starting from the first or start curve towards the second or end curve.

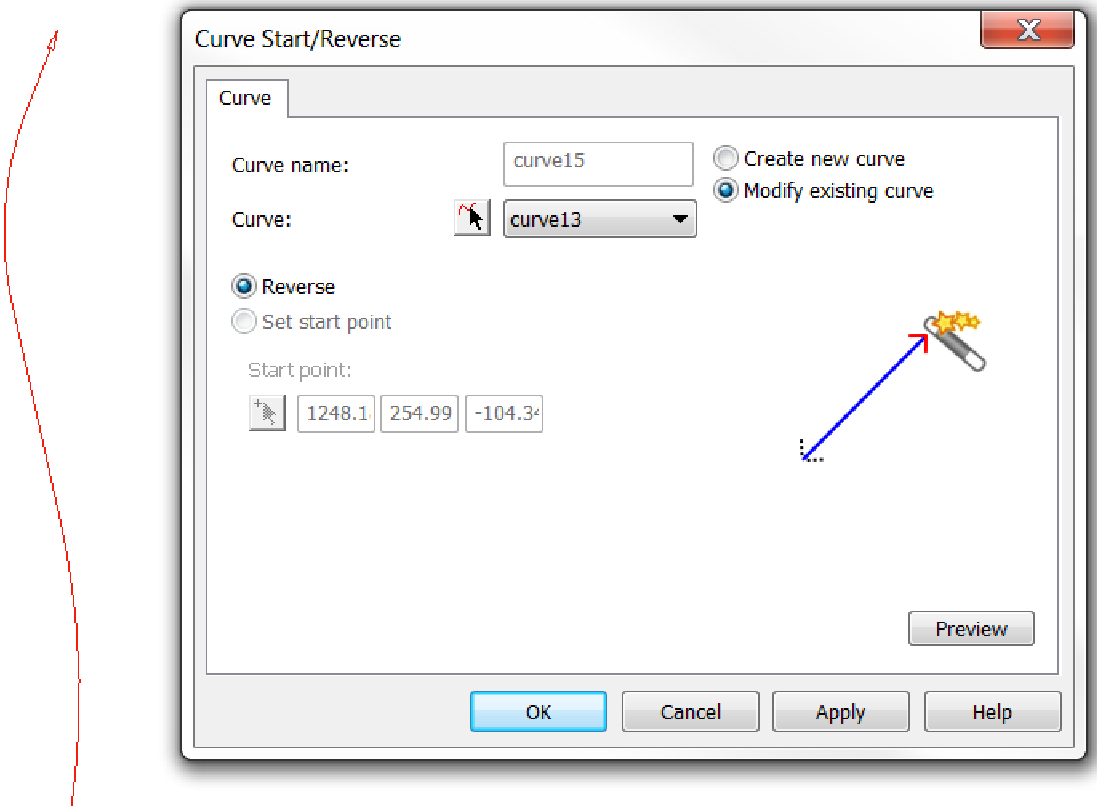
1 Create two curves using surface edges.

2 Select the following option as shown on the next page.

3 Then create curves from the edges shown.

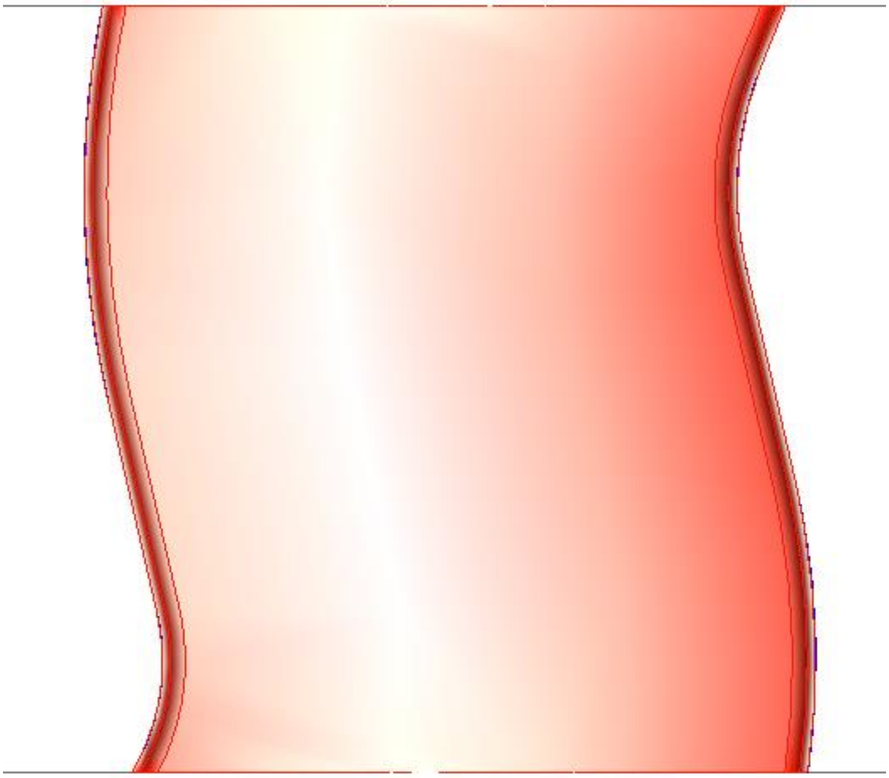


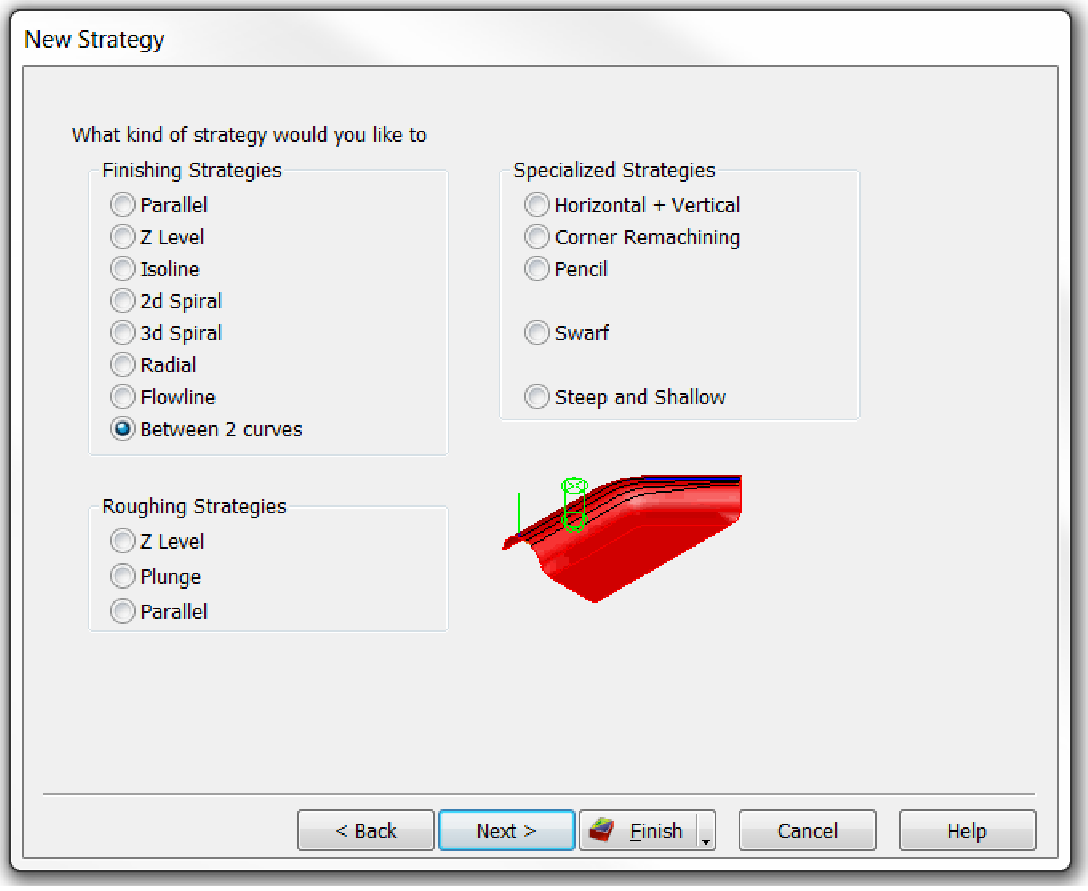
4 Once you have created your curves make sure that the directions of both curves are running in the same direction. See example of curve direction below. Reverse if necessary.



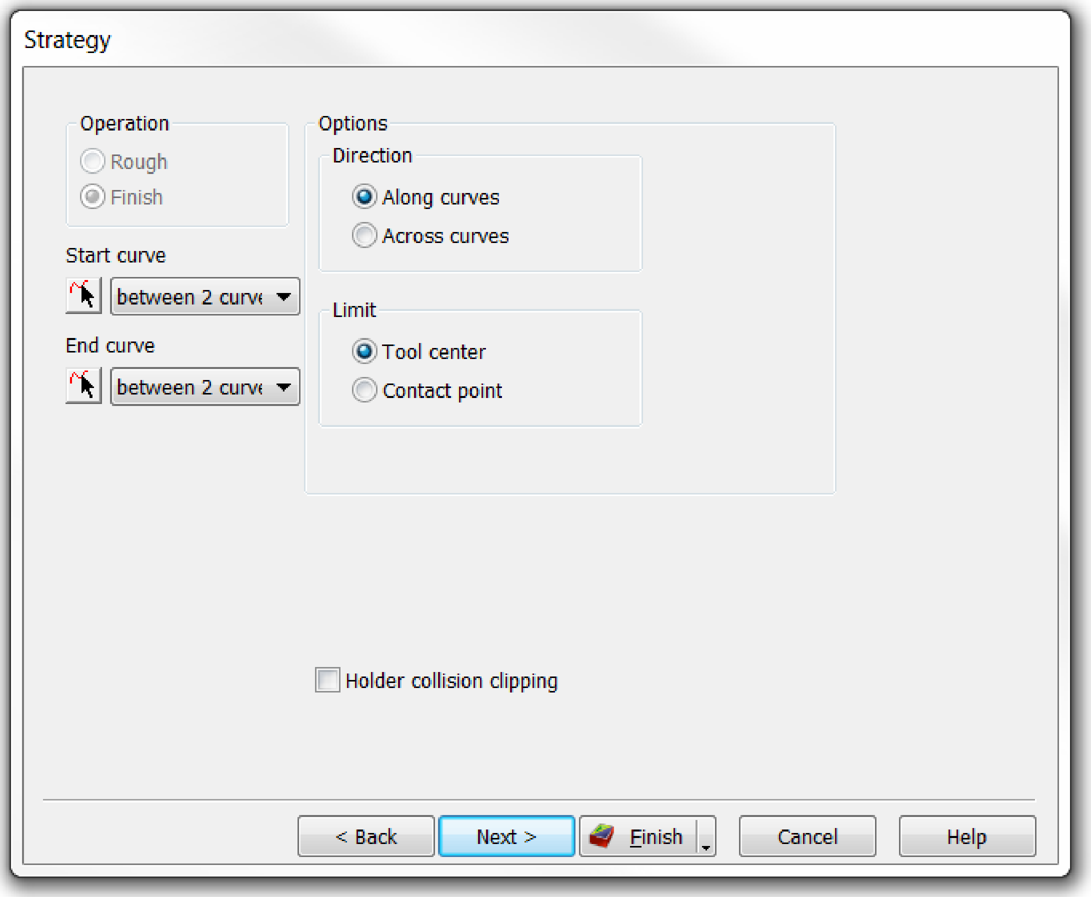
5 Create a new Surface Milling Feature

6 Select Surfaces on model.

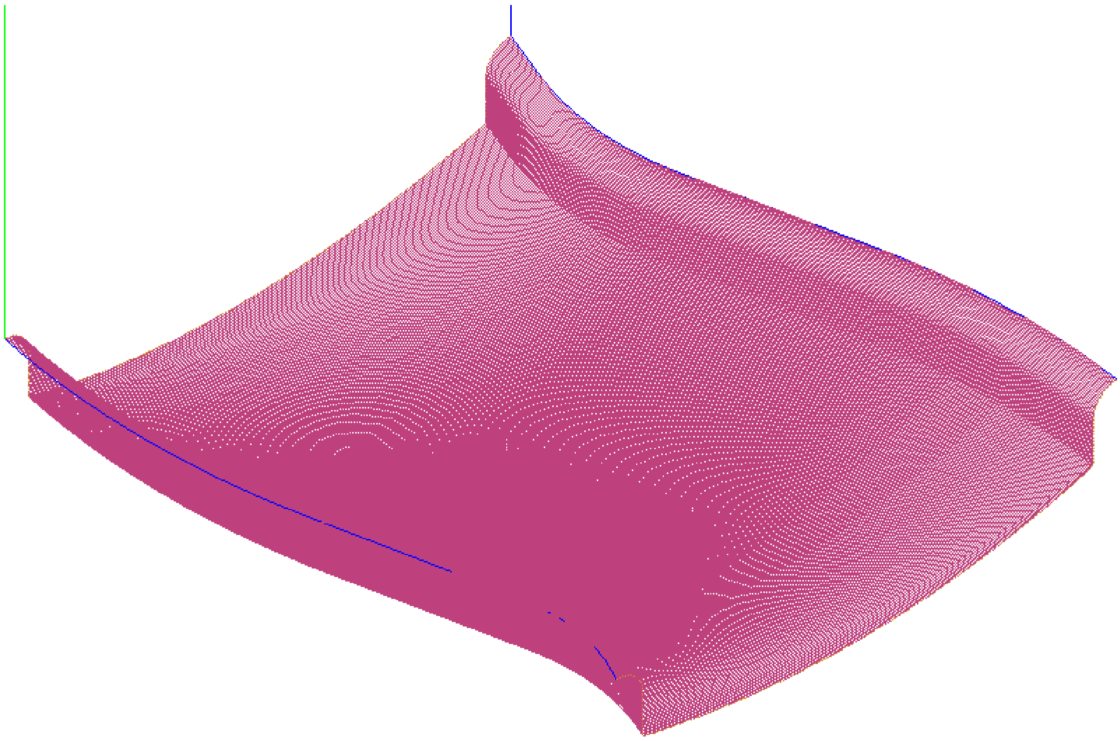




7 Select **Next**. Pick the two curves as shown.

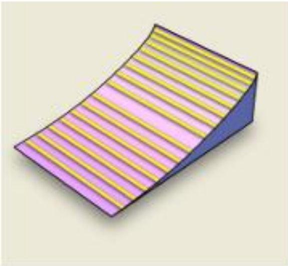


8 Finished Toolpath.



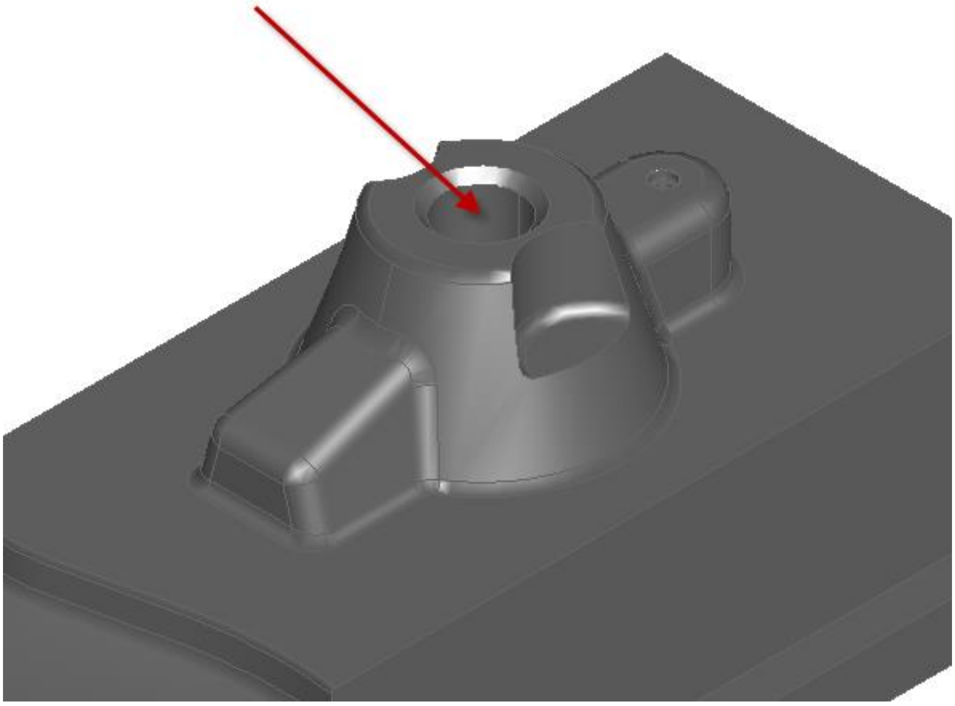
Specialized Strategies

Horizontal + Vertical

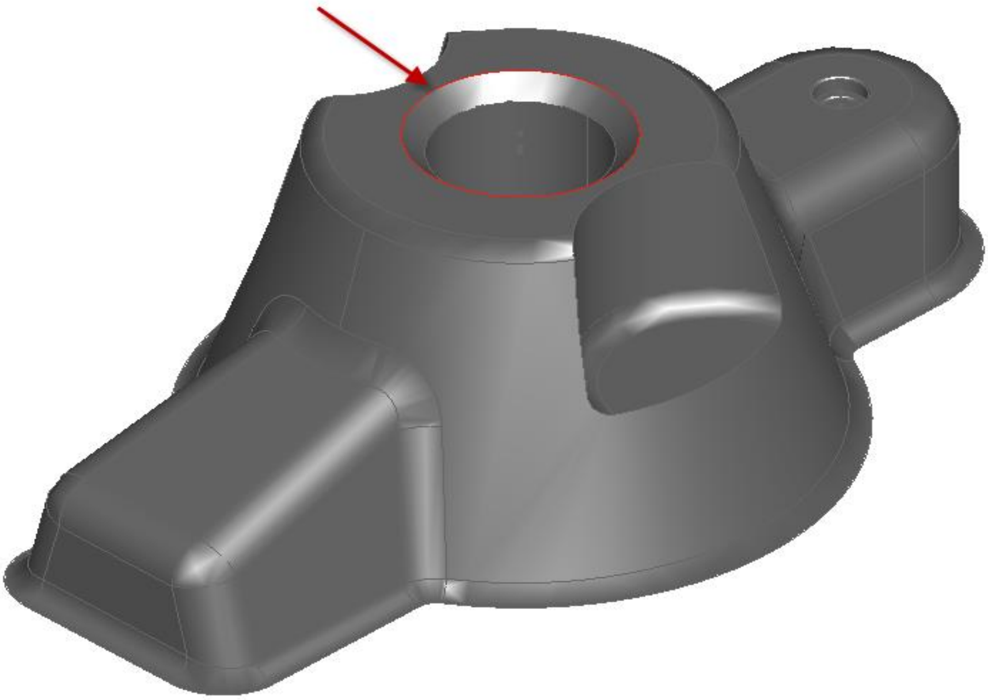


This strategy combines two different toolpath operations, one for finishing shallow portions of the part and one for finishing the steep regions. An X parallel or spiral toolpath is applied to the shallow regions, and a Z level finishing operation cuts the steep regions.

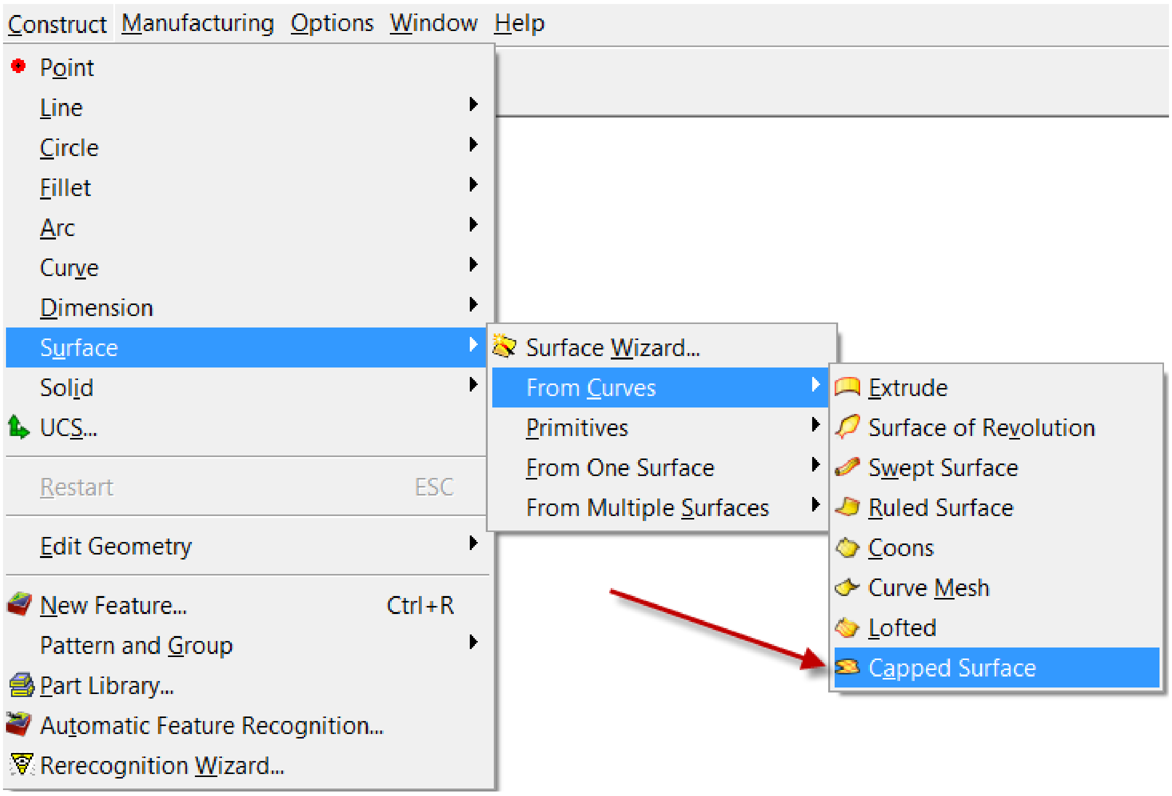
*We have a problem with this model. The tool will go down the hole when machining. To prevent this from happening we need to create a Curve and then create a surface to cover the hole or what's known as a **capped surface**.*



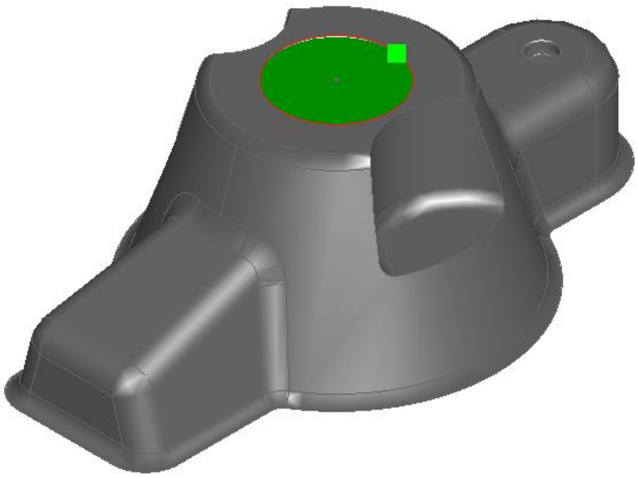
1 Create a Curve using Create a Curve from Surface - Surface edges. Pick the edge shown.

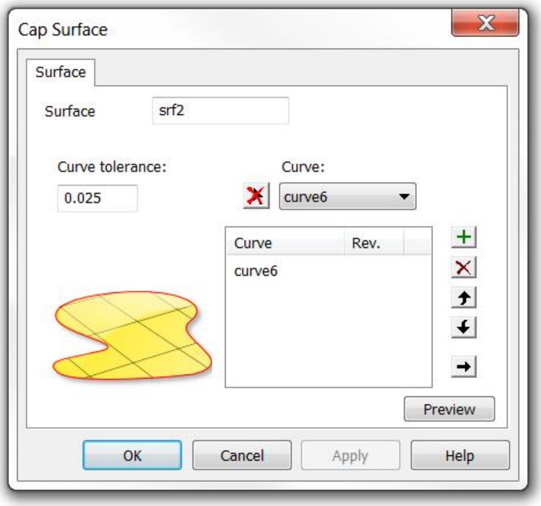


2 Now cap the surface using the curve you have just created.

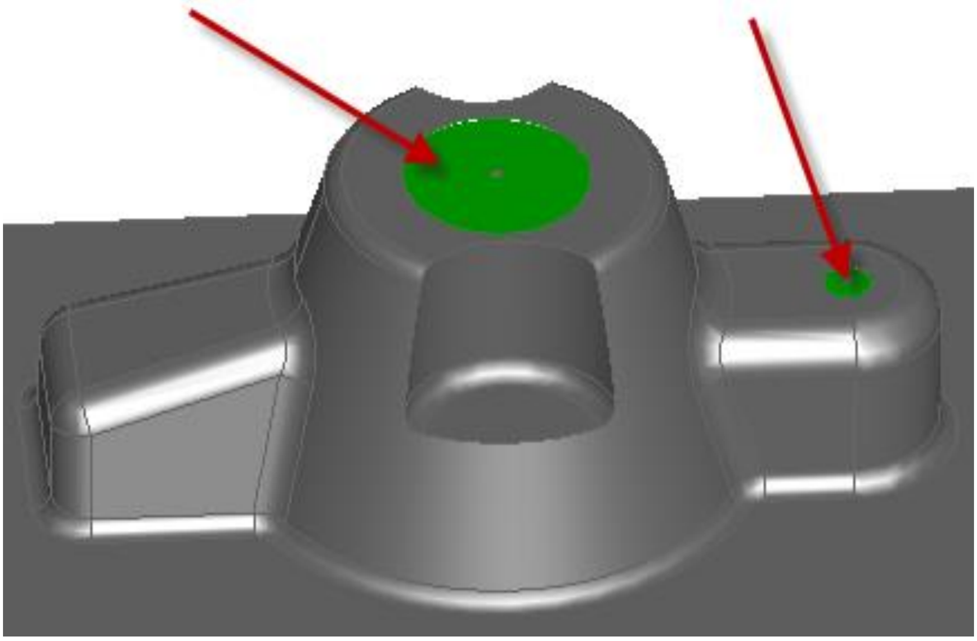


3 Select the curve as shown on the next page this will then create a surface that will cover the hole. Make sure that this surface is also selected when machining this part.





Do the same for the smaller hole. This will prevent the tool from diving down into the hole when we apply our machining strategy.



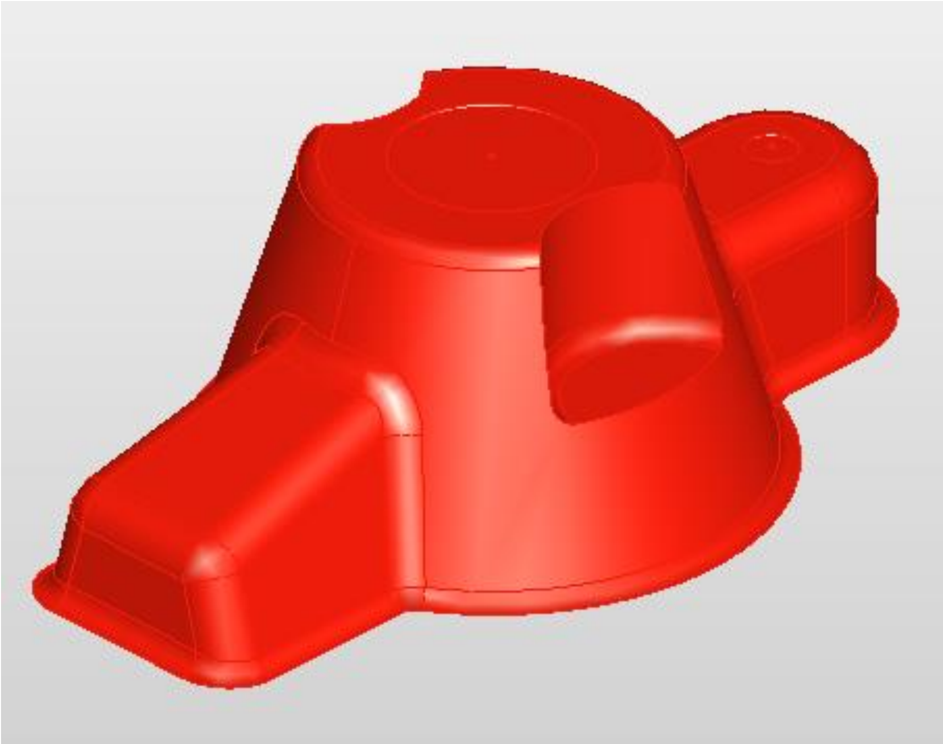
4 We will now create a **Horizontal + Vertical** machining strategy.

5 Create a new Feature **Ctrl+R**

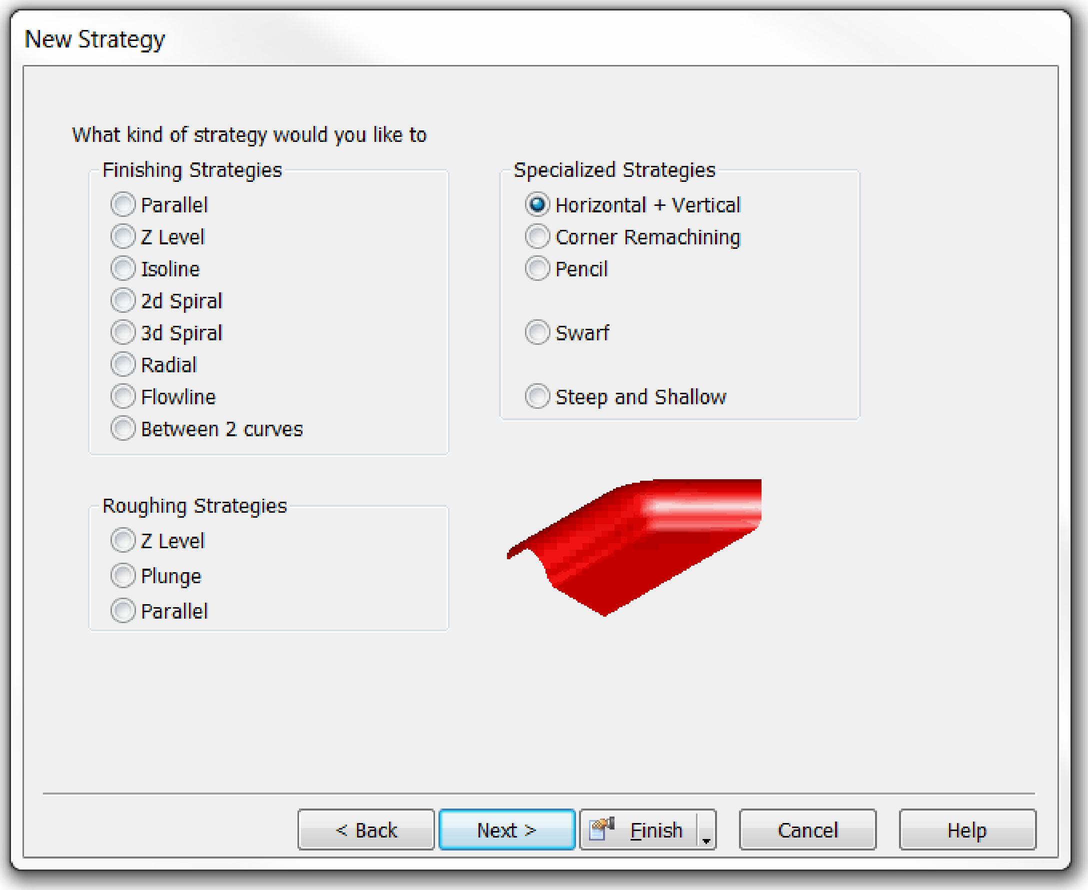
6 Select **Surface Milling** and then select **Next**.

7 Select the Part Surfaces by windowing over the selected surfaces.

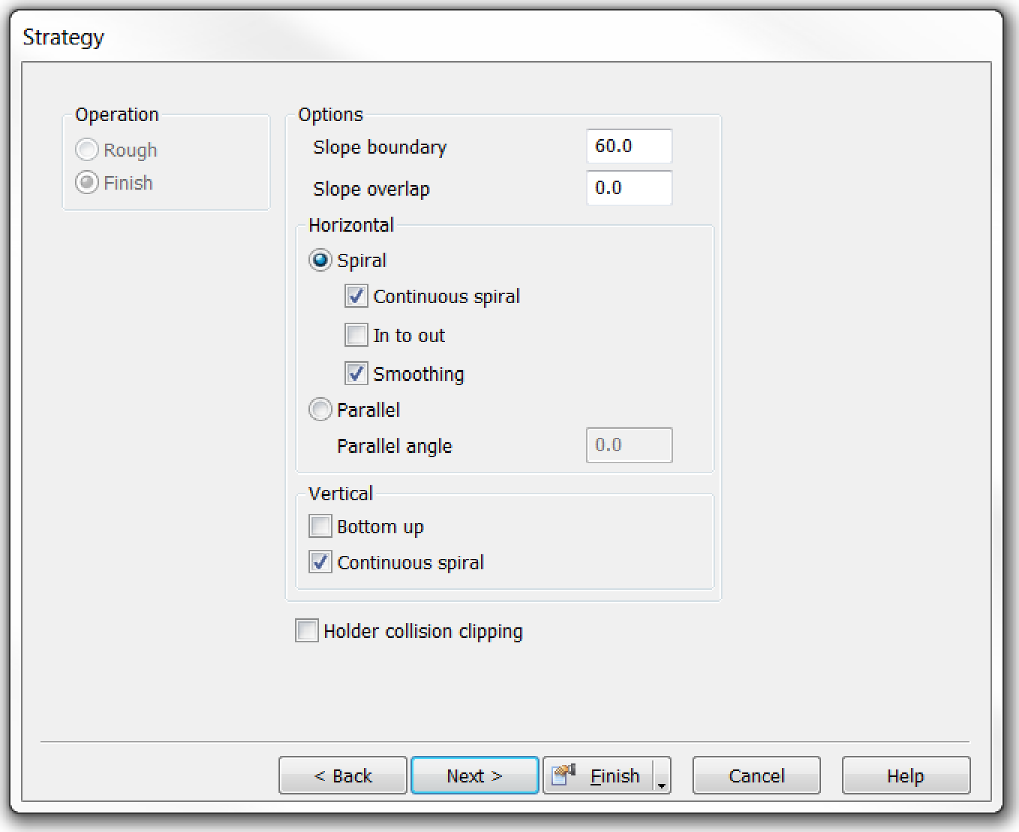
8 Select all the surfaces shown below. You may have to select the bottom surface as a check surface to prevent the toolpaths from going below the surface.



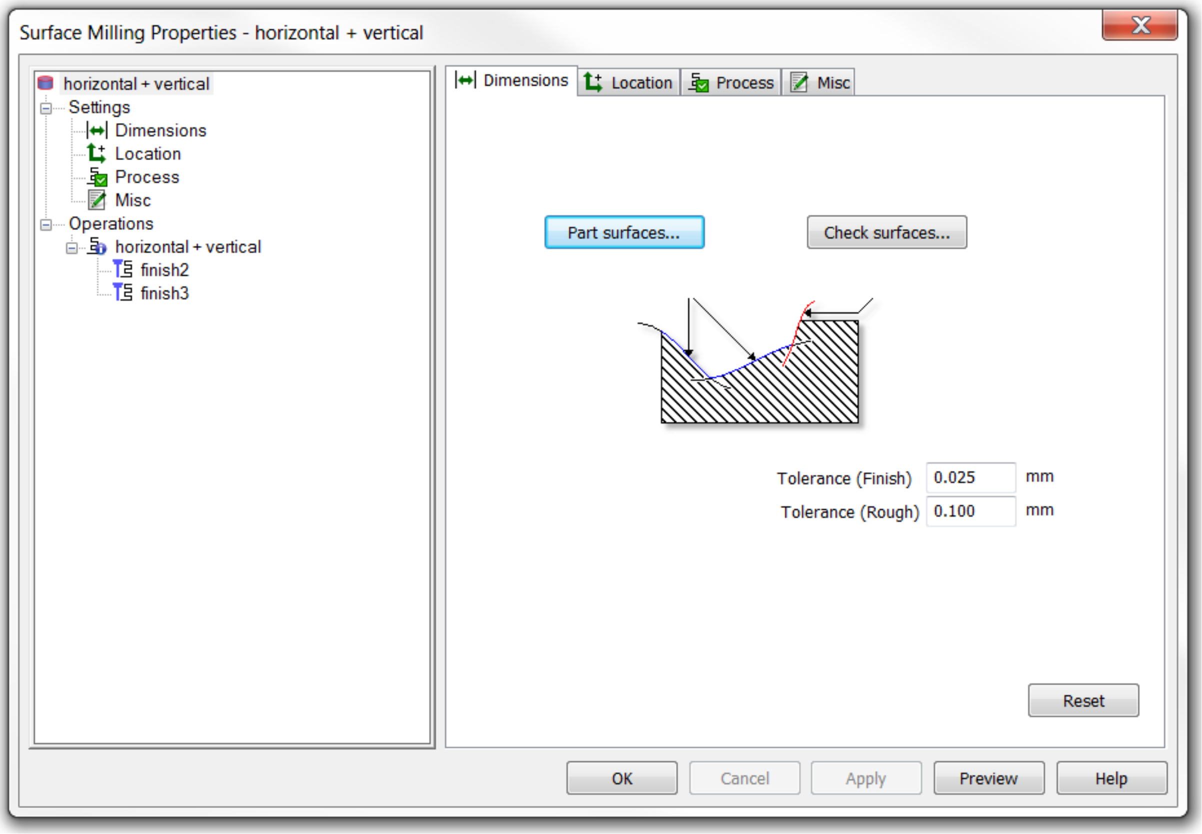
9 Please select. Choose a single operation. Select Next.

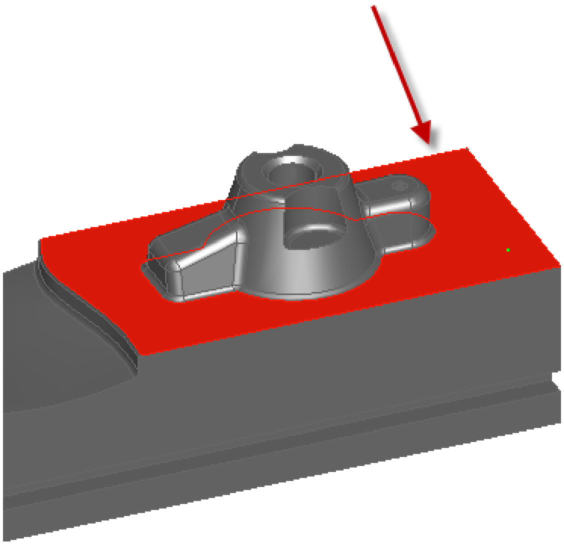


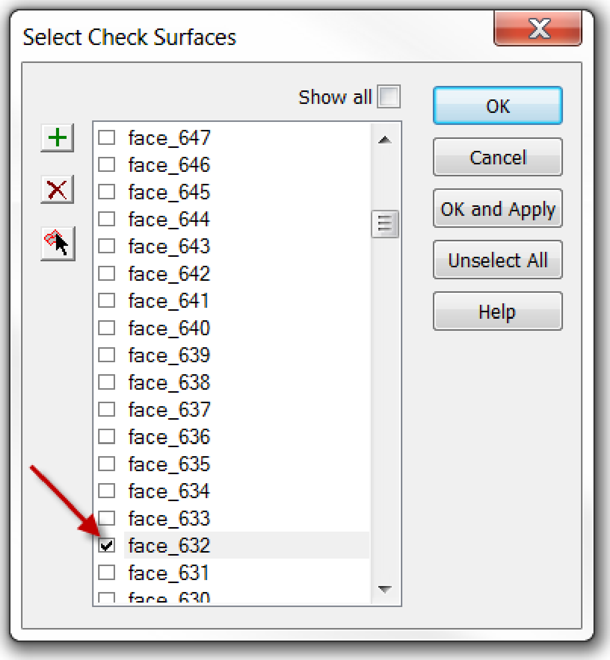
10 Then select the **Horizontal + Vertical** machining strategy. Select **Next** and then select the following options.



11 Select **Finish** for the default tools and settings. Now select **Check surfaces** and select the surface shown on the second image.

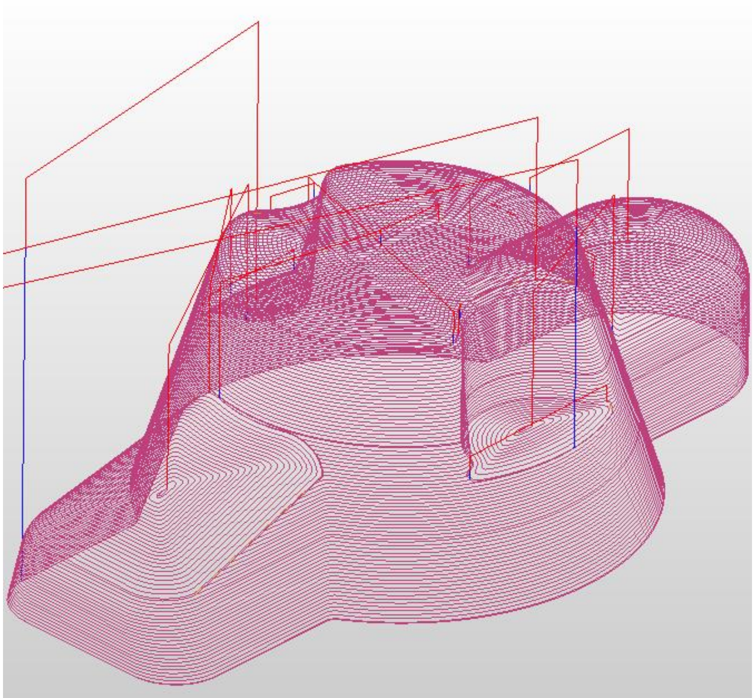




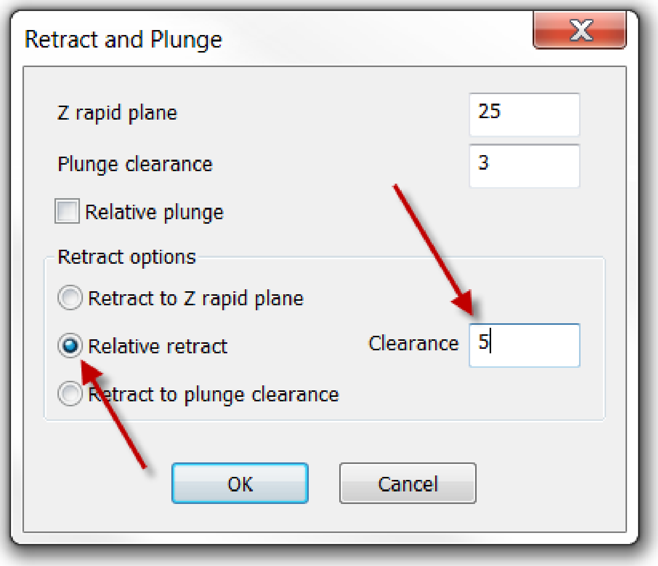


12 Select **Ok** and **Apply.**

13 Remember to rename the strategy to **horizontal + vertical.** The results of the toolpath are shown below.



A suggested tip would be to make the toolpaths more efficient by changing the Rapids to ***Relative retract****. To select this double click on the Surface milling properties, then select the milling tab and then select* ***Retract/Plunge****. Select the following options. The image above shows the results of changing this option making the toolpath more efficient.*

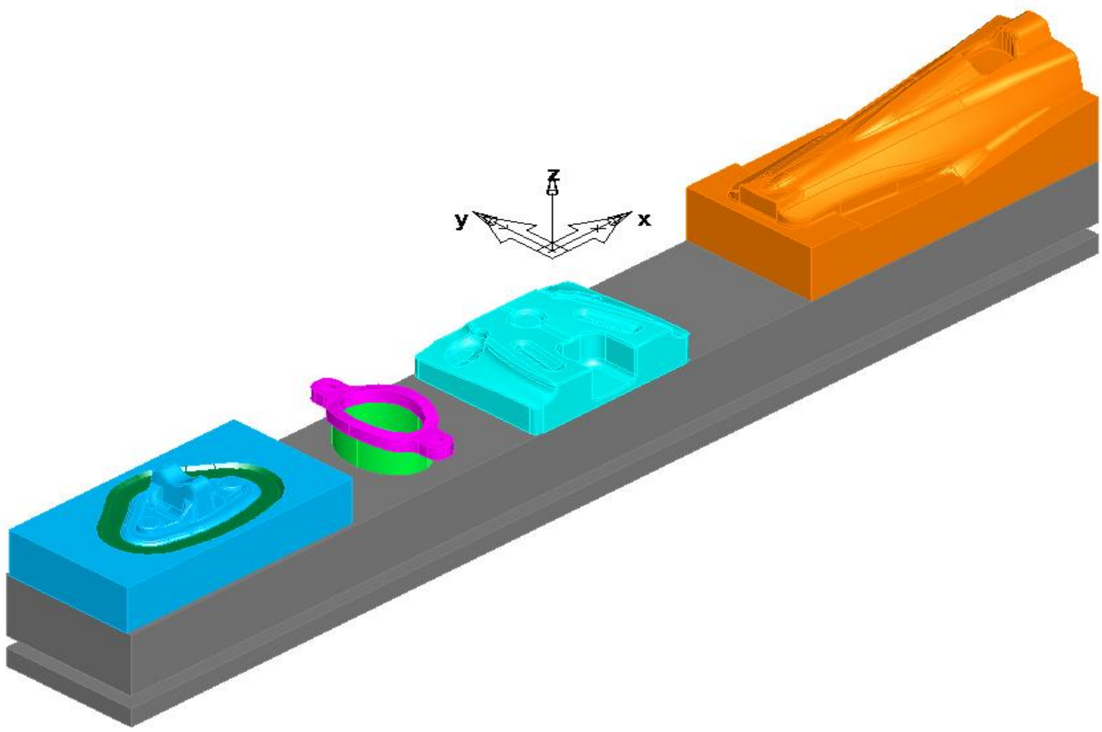


***Relative retract** - When you retract and rapid to a new position, using the **Relative retract** option, the tool only retracts as high as it needs to go plus the **Clearance** amount that you enter.*

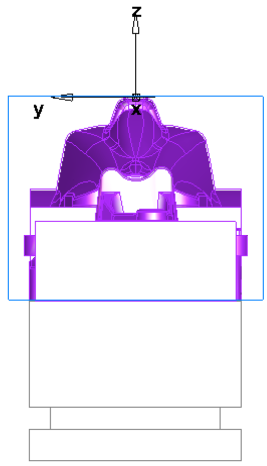
This concludes the use of this Model

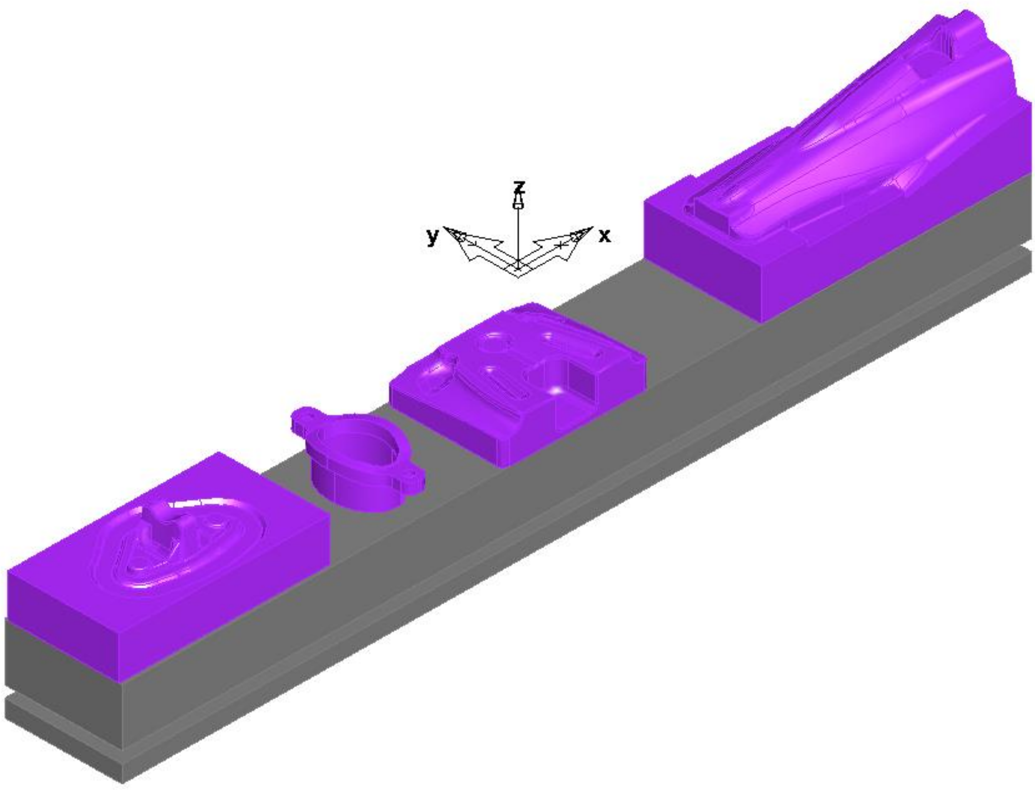
Corner Remachining

- We will start by importing a solid model in Parasolid format into **FeatureCAM** You will find this in the FeatureCAM training folder **C:\Training_Data\FeatureCAM Course Data 2017\3D Machining\ All Strategies Part-2. x_t**



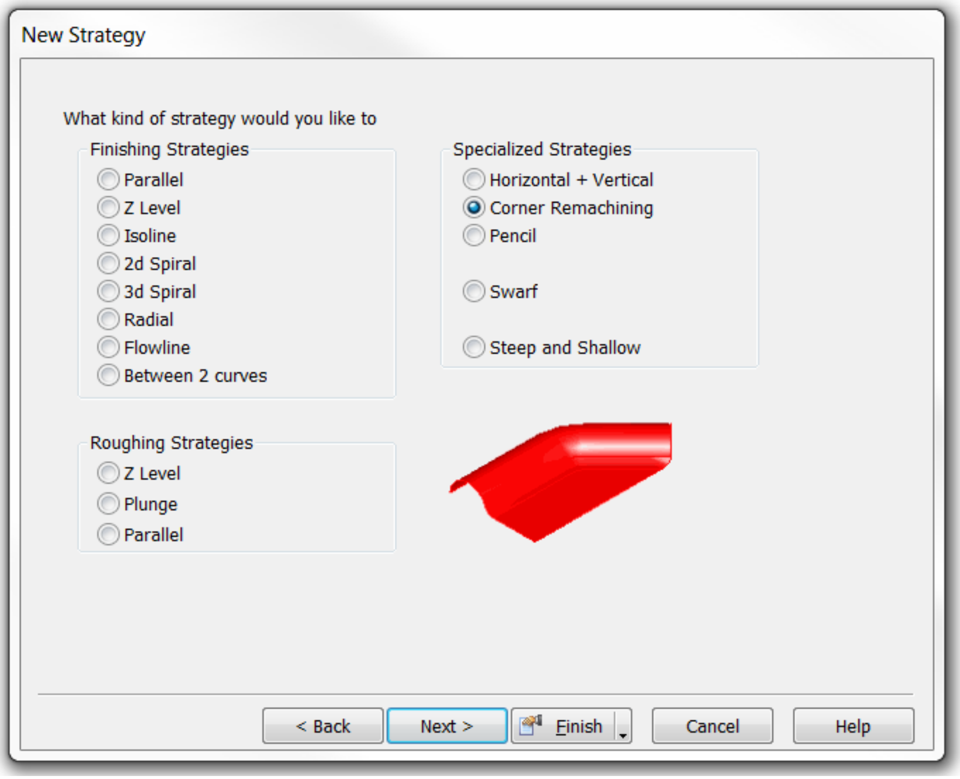
1 Create a Stock boundary to make sure you machine the complete part. The pink solid model shown above is larger than the block it is supported on. Make the height just above the bottom surface of the four models. Select a **Z Level Roughing** strategy to rough out our part. Use a 50mm Facemill to rough the part out with a Z Level step down of 3mm.



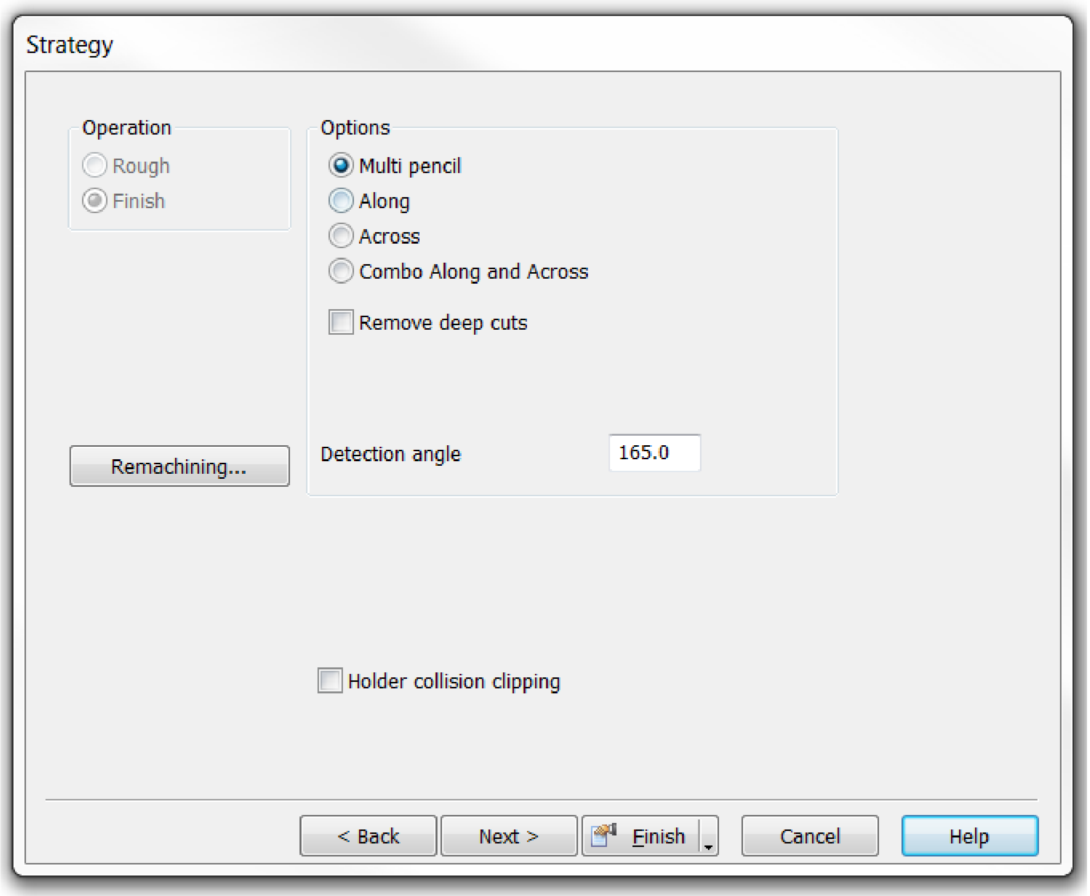


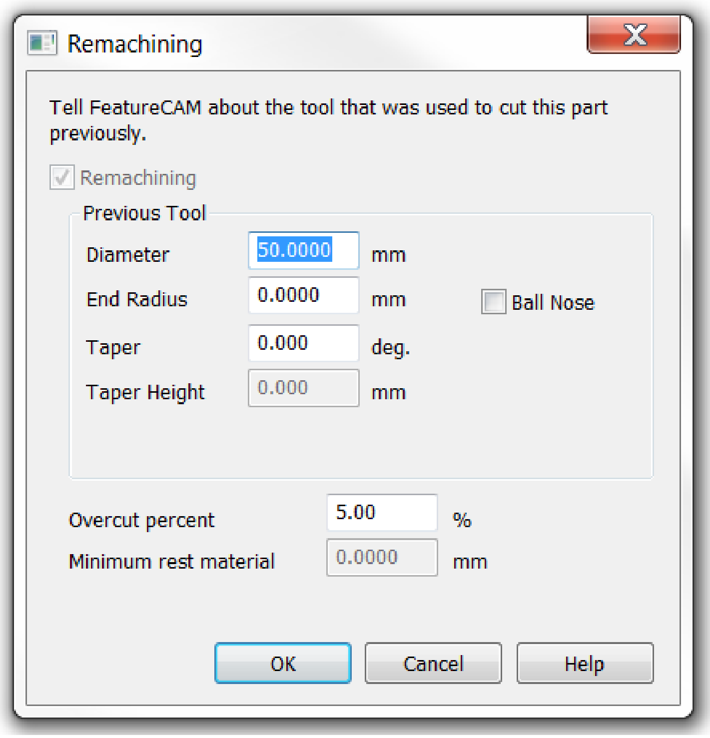
Corner Remachining

Corner Remachining is used to clean up corners that occur between non-tangential surfaces. Each corner edge is called a trace line. By using the options on the strategy page, you may cut in various directions relative to the trace lines.

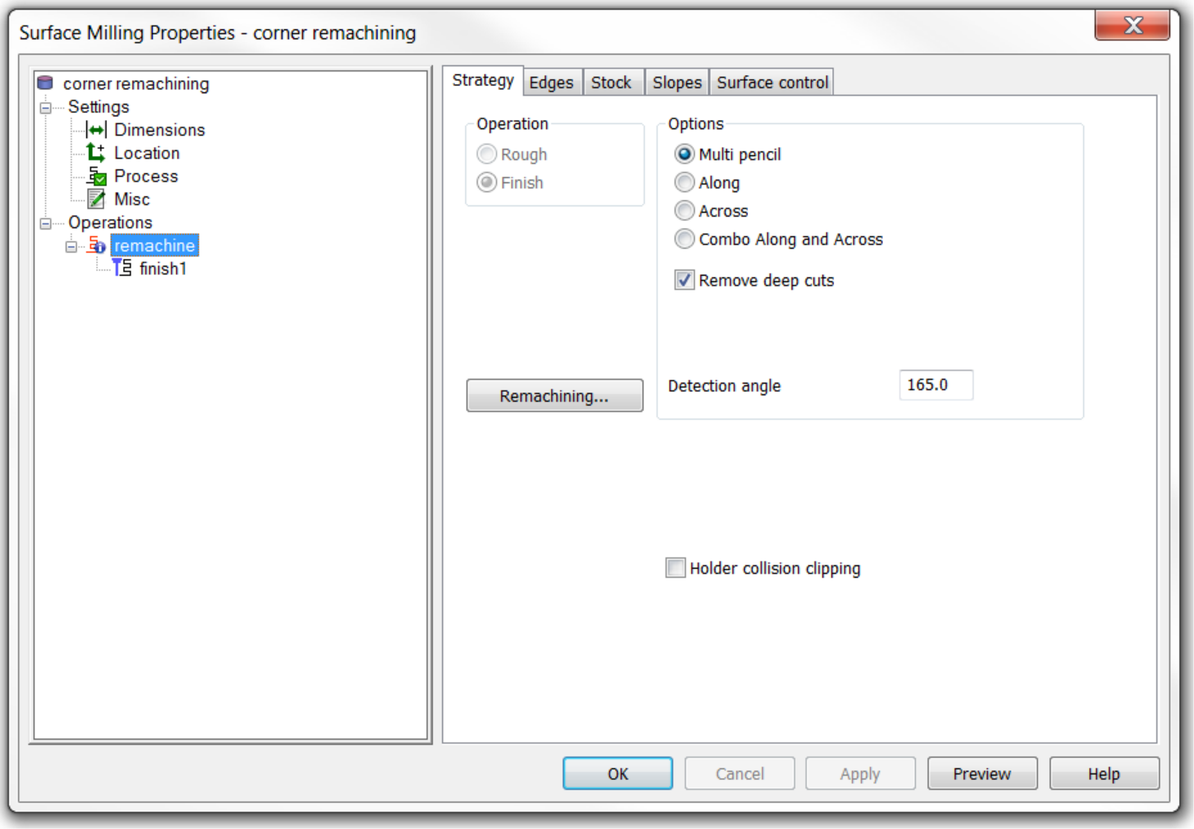


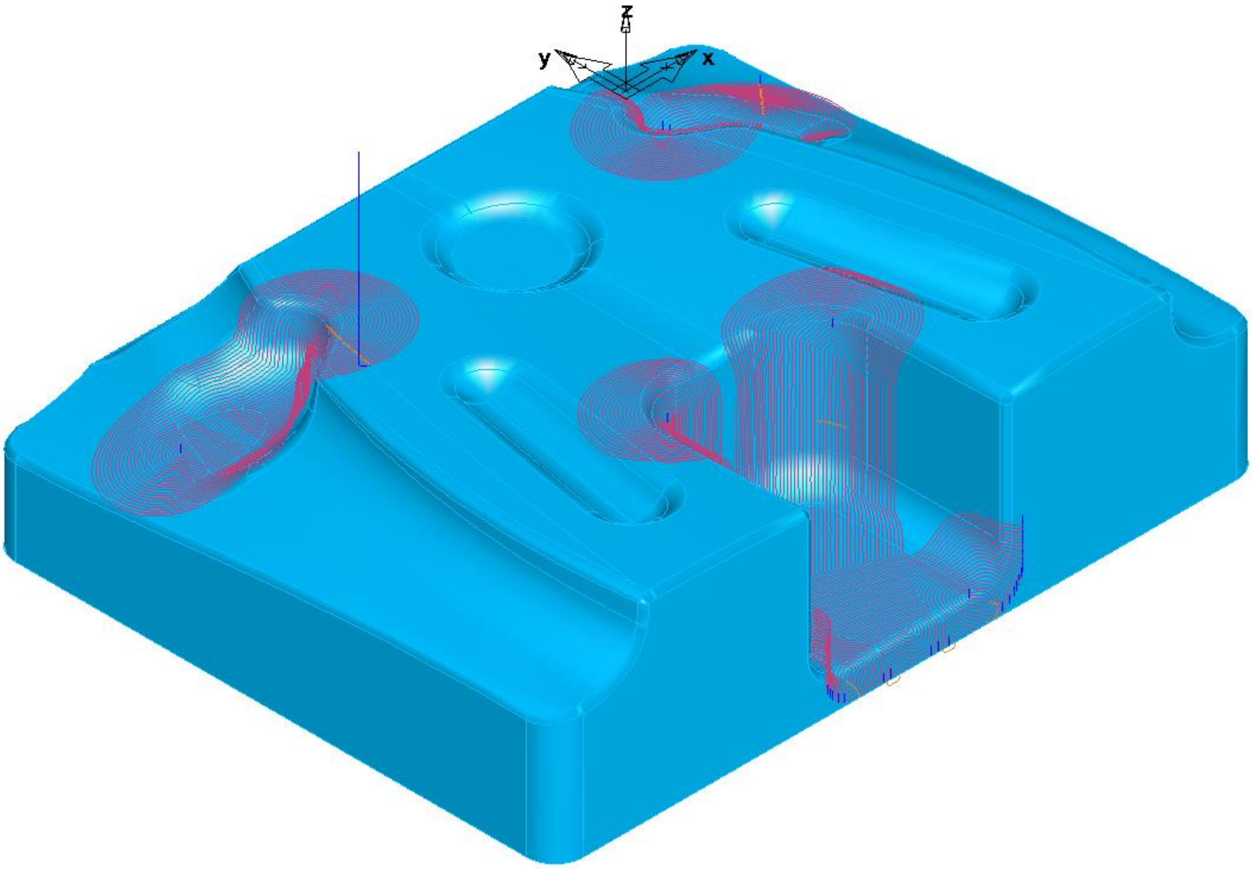
- Corner Remachining is available in **four** different styles: **Multi pencil, Along**, **Across**, **Combo along and across**, with remove deep cuts as an option.
- Remachining is used to automatically mill regions that were not cut by previous operations. You provide the diameter of the previous tool that was used to cut the part and **FeatureCAM** automatically determines the uncut regions and applies a toolpath to them.
- Corner remachining is used to clean up corners that occur between non-tangential surfaces. Each corner edge is called a trace line. By using the options on the strategy page, you may cut in various directions relative to the trace lines.
- Remember to select the surfaces you want to machine.
- Select **Finish**, and then access the **Remachining** Tab and input the **50mm Facemill.**



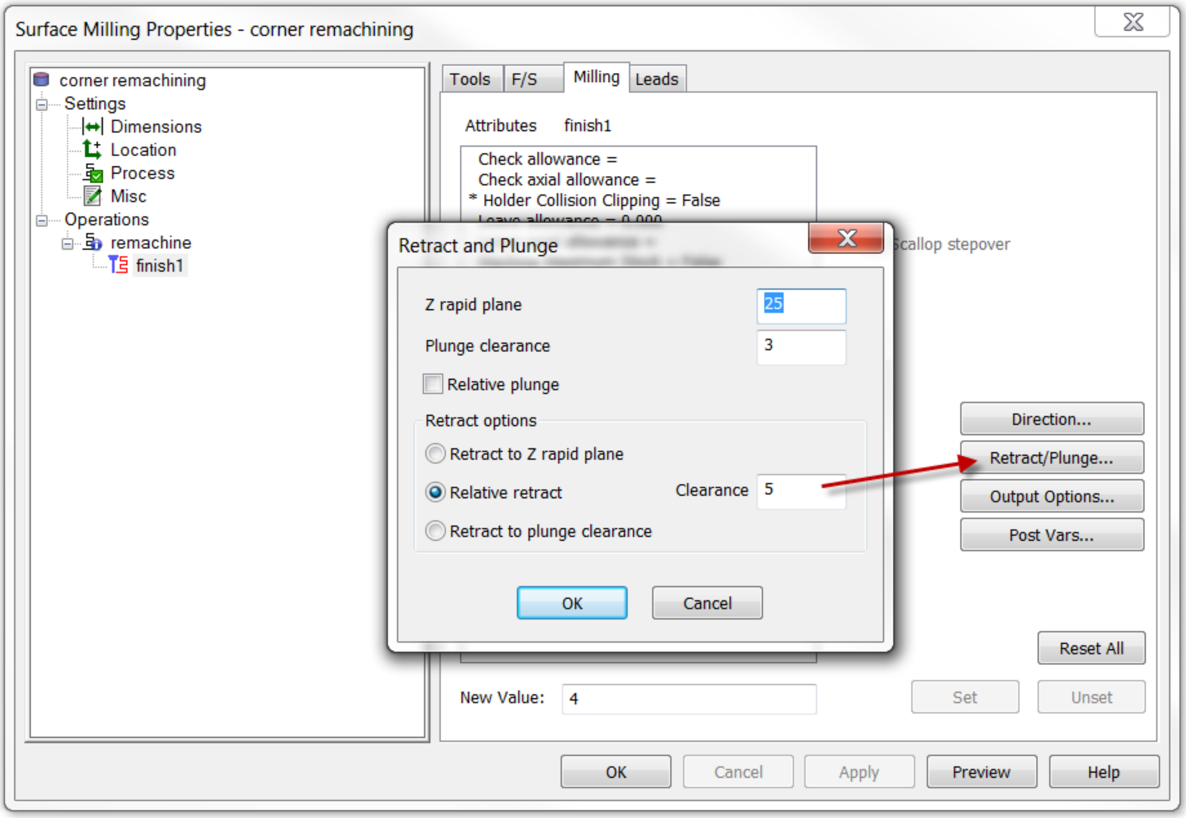


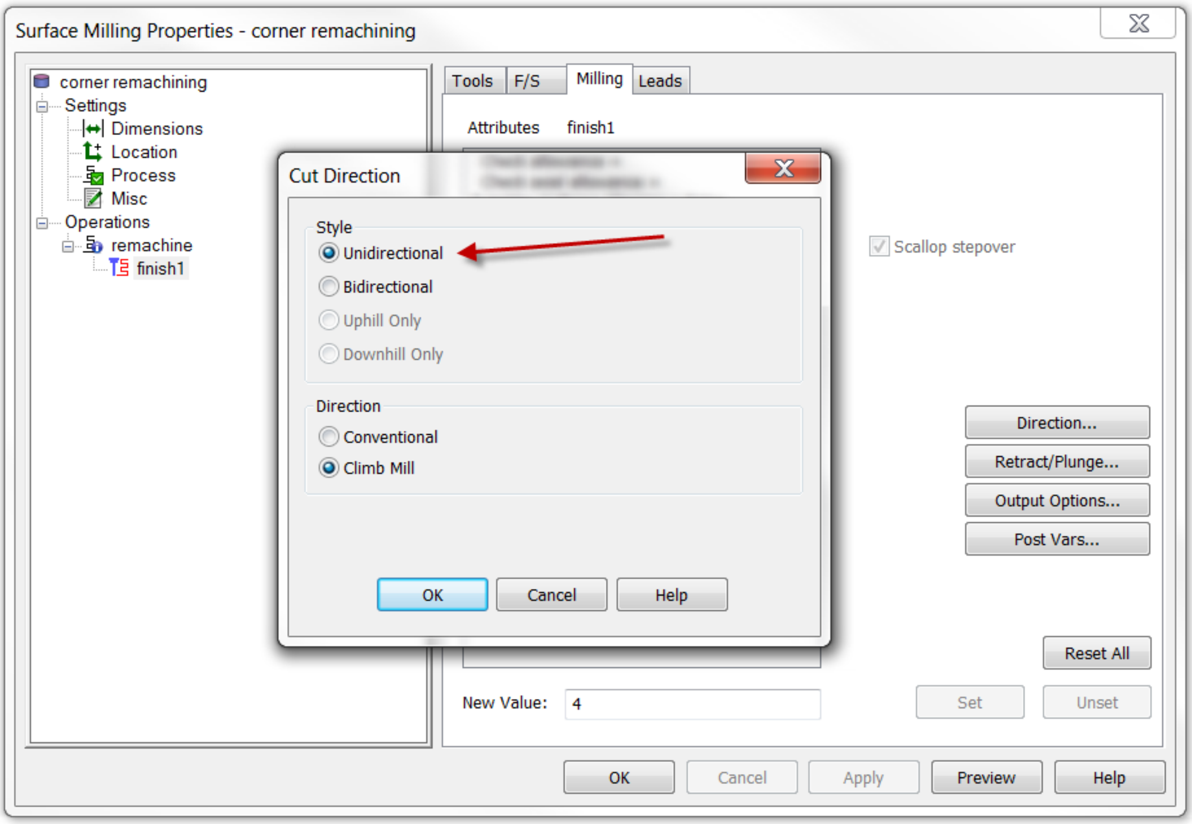
***Multi Pencil -** creates a corner toolpath which follows 'along' the trace lines. This is basically a different ordering of the **along** tool path.*


Surface Milling Properties - corner remachining
corner remachining
Settings
Dimensions
Location
Process
Misc
Operations
remachine
finish1
Strategy
Edges
Stock
Slopes
Surface control
Operation
Rough
Finish
Options
Multi pencil
Along
Across
Combo Along and Across
Remove deep cuts
Remachining...
Detection angle
165.0
Holder collision clipping
OK
Cancel
Apply
Preview
Help

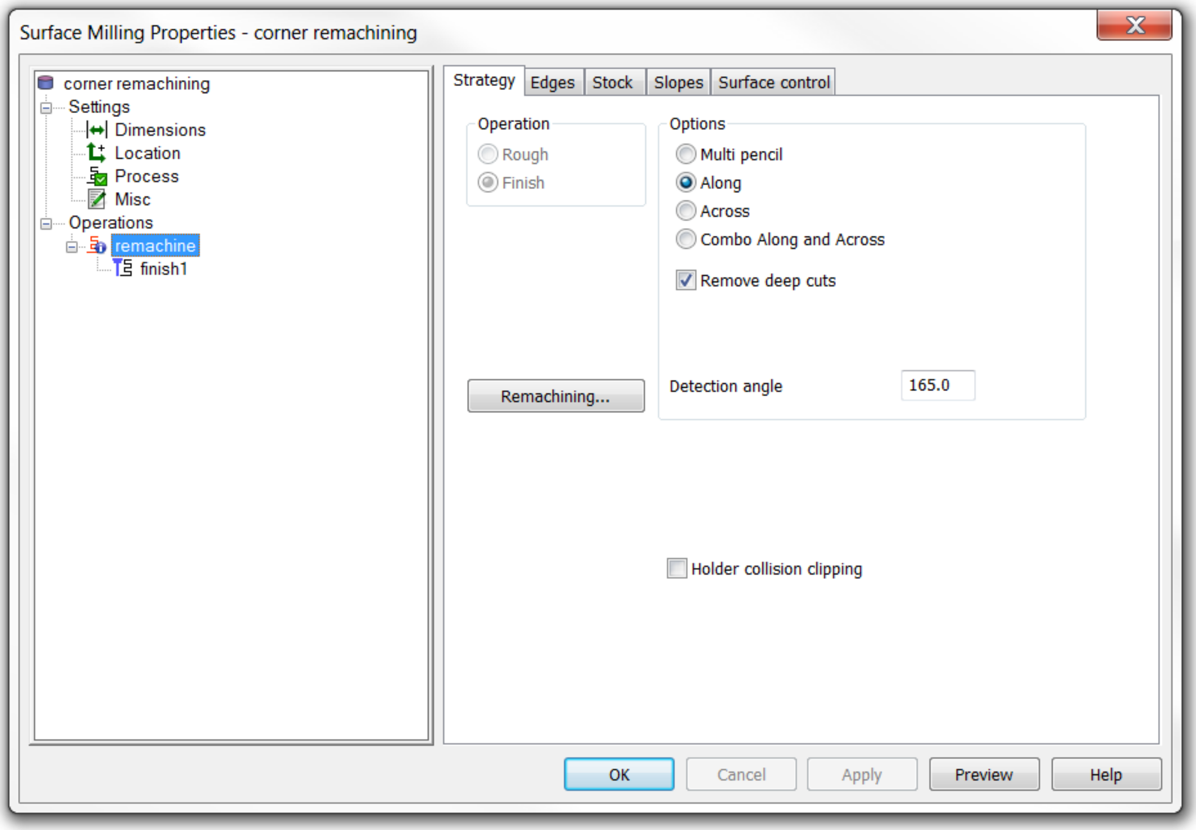

z
y
x

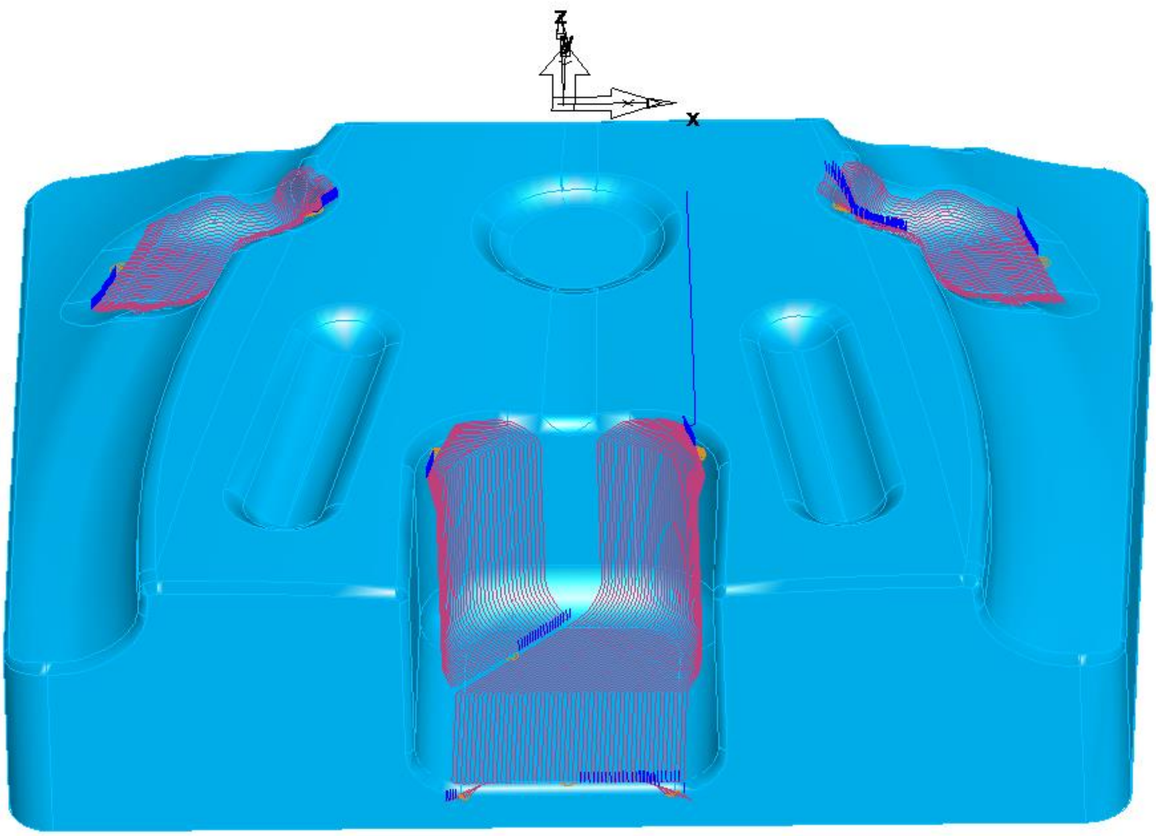
To achieve the results shown above the following parameters have been changed.



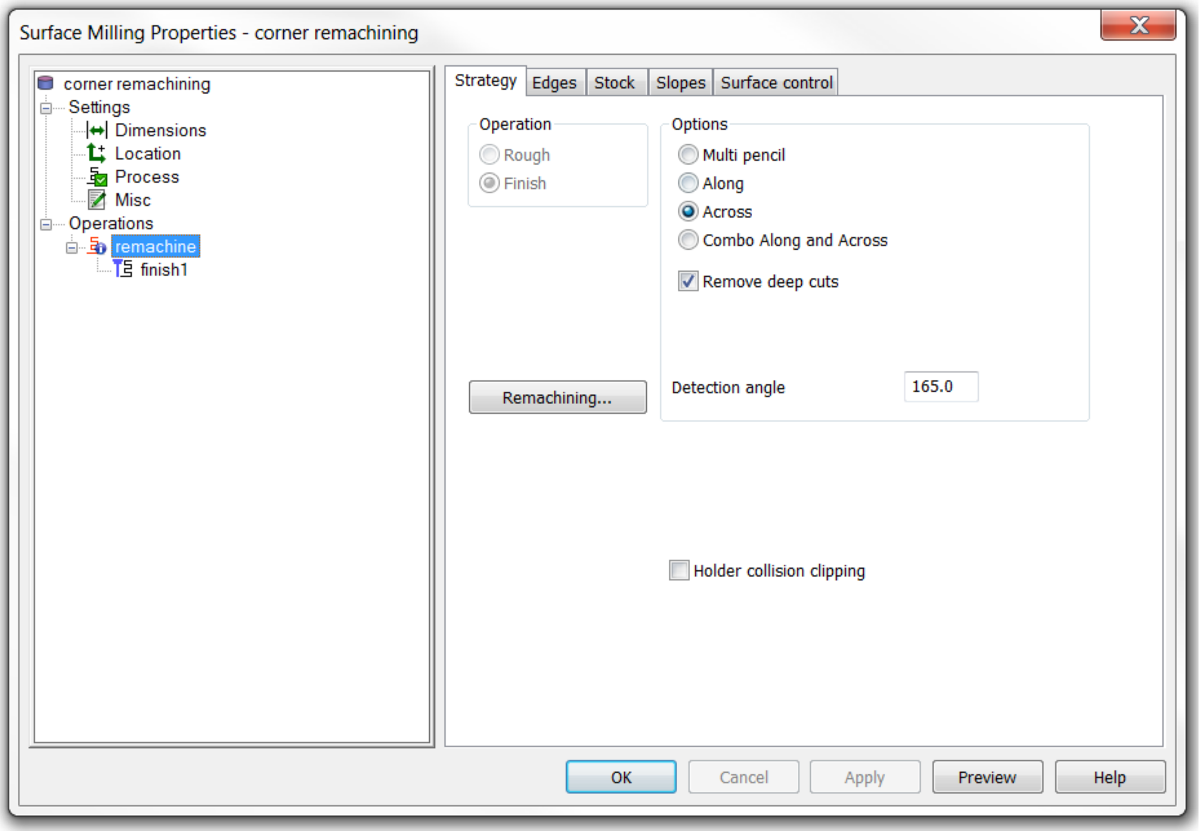


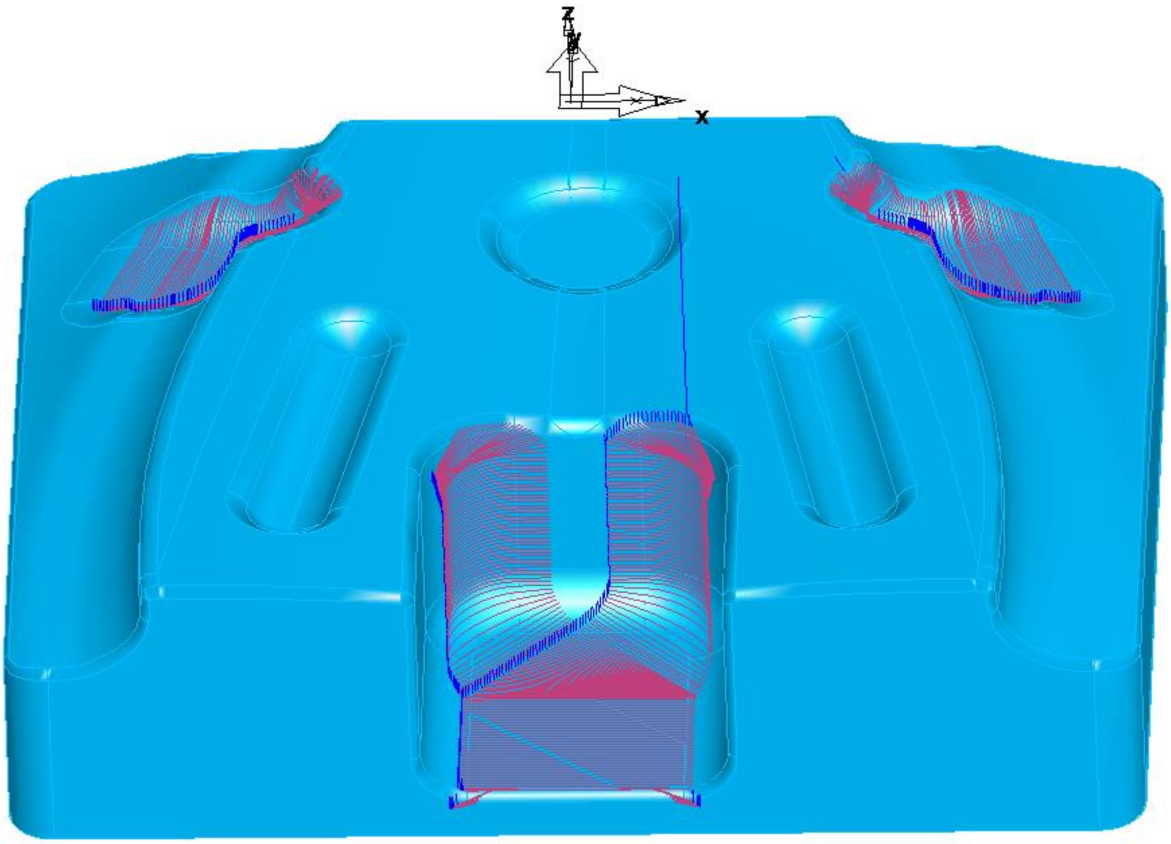
***Along** - This style of Remachining creates a corner toolpath which follows the trace lines.*





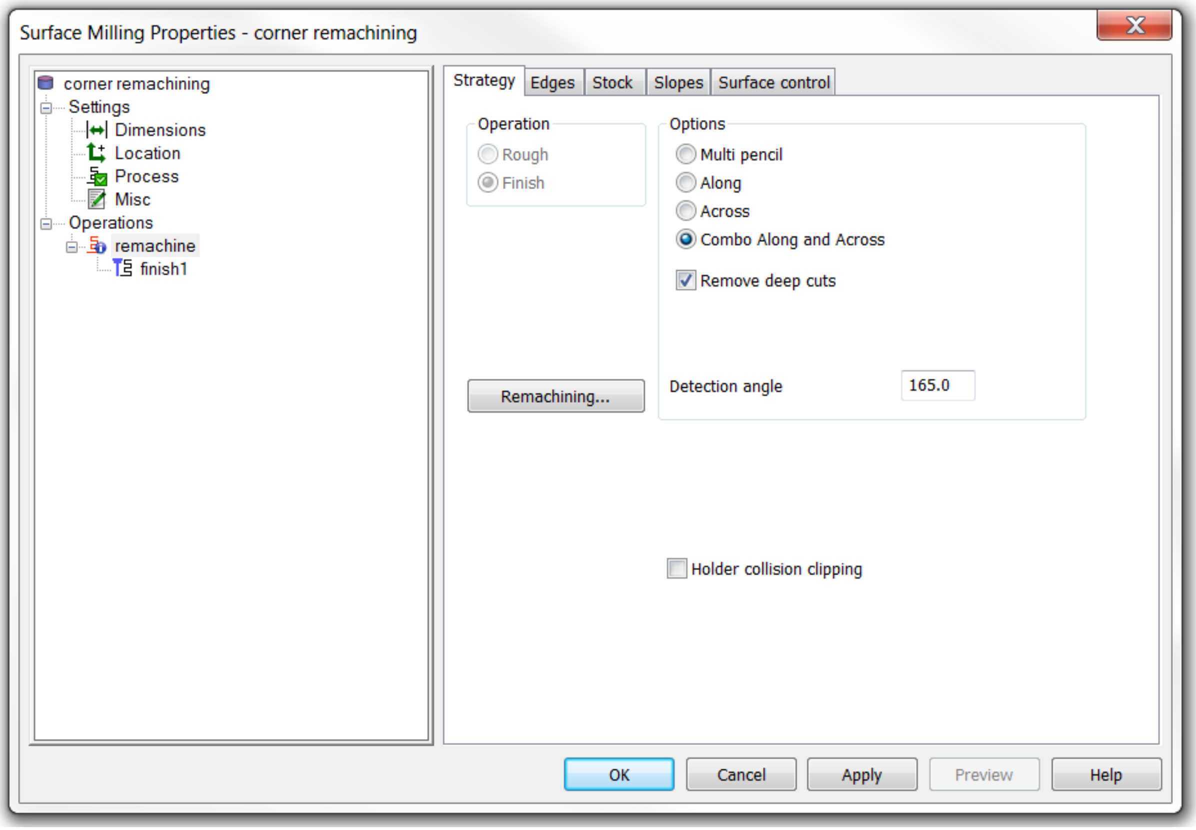
***Across -** Remachining creates corner toolpaths that **zigzag** across the trace lines.*

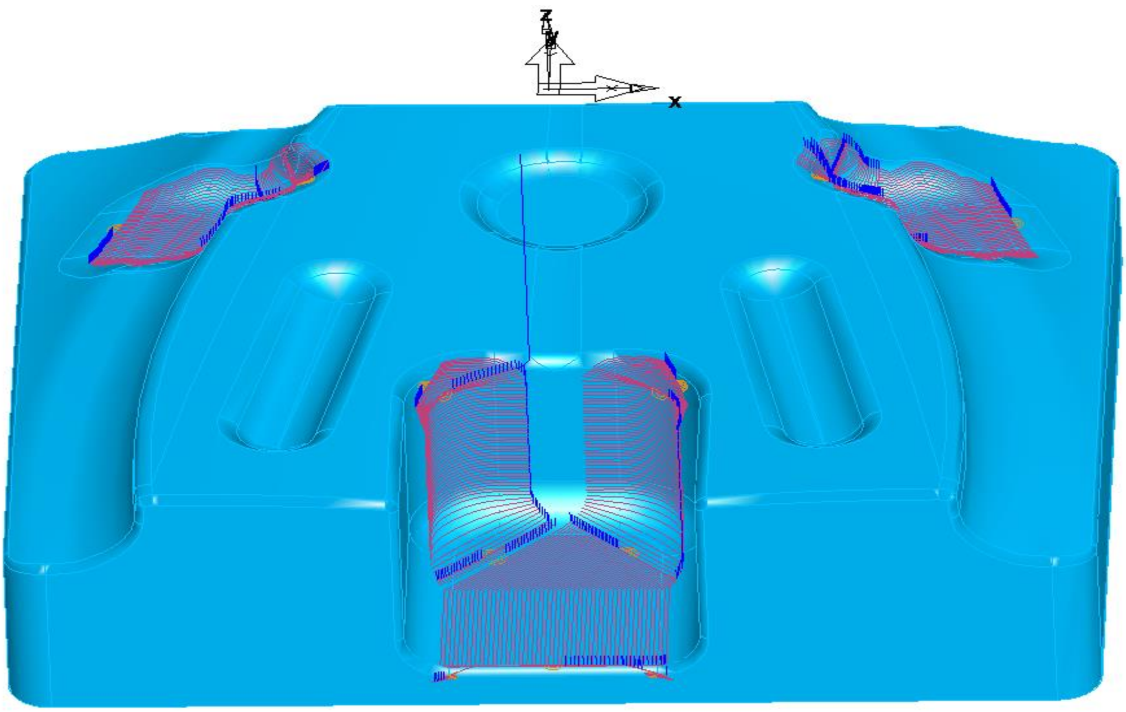




Combo Along and across - *The combo corner toolpath creates a corner toolpath which produces* ***across*** *toolpaths on the steep areas of the trace line and* ***along*** *toolpaths on the shallow areas of the trace line.*

Remove deep cuts - *removes segments of the toolpath where the depth of cut is too large, to prevent tool damage.*





Steep and Shallow + Pencil

Steep and Shallow: - *This strategy calculates a shallow boundary, and then creates a Z-level toolpath in the steep areas of the model and a Parallel or 3D Spiral toolpath in the shallow areas.*

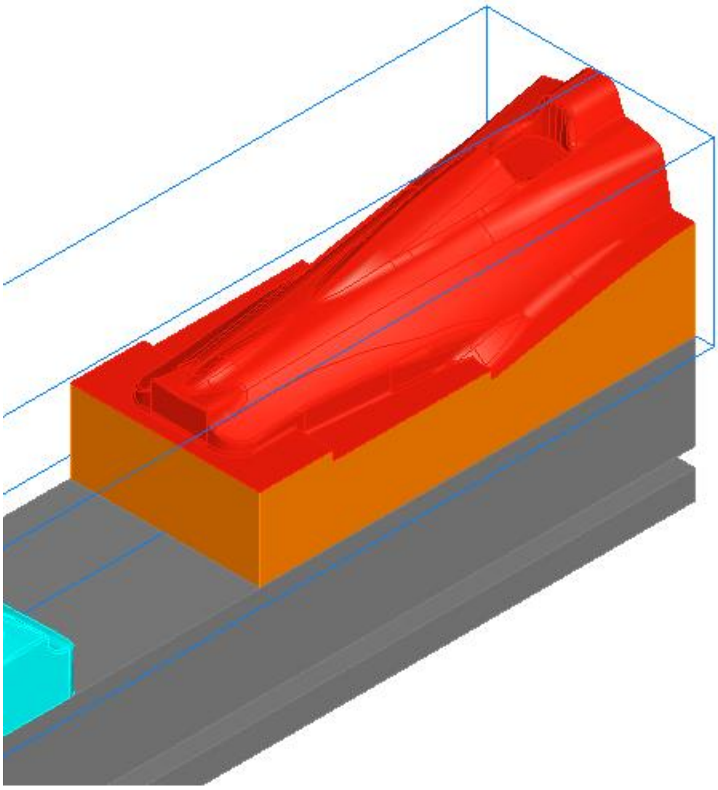
Pencil: - *This strategy creates a single trace corner toolpath. It is used to clean up corners that occur between non-tangential surfaces. They are automatically calculated inside any existing boundary.*

Applications of pencil milling include:

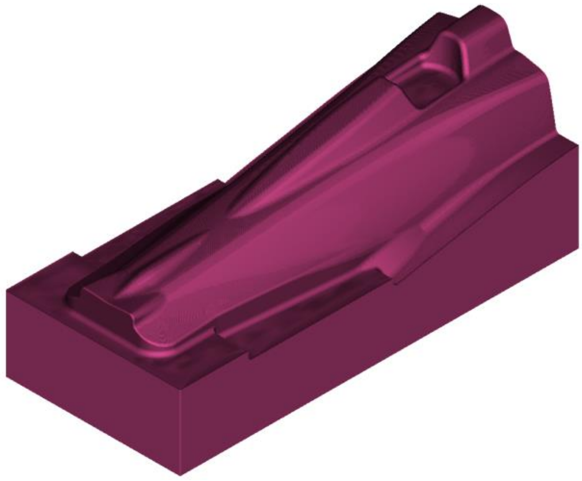
- Finishing fillets in a part with a single toolpath.
- Cleaning up sharp concave corners.
- Pre-relieving corners before high-speed finishing the part with a small tool.
- Roughing fillets by using the Finish allowance.

*Both **Steep and Shallow + Pencil** strategies can be added together in the **Process** tab using the same selected surfaces and Check surfaces.*

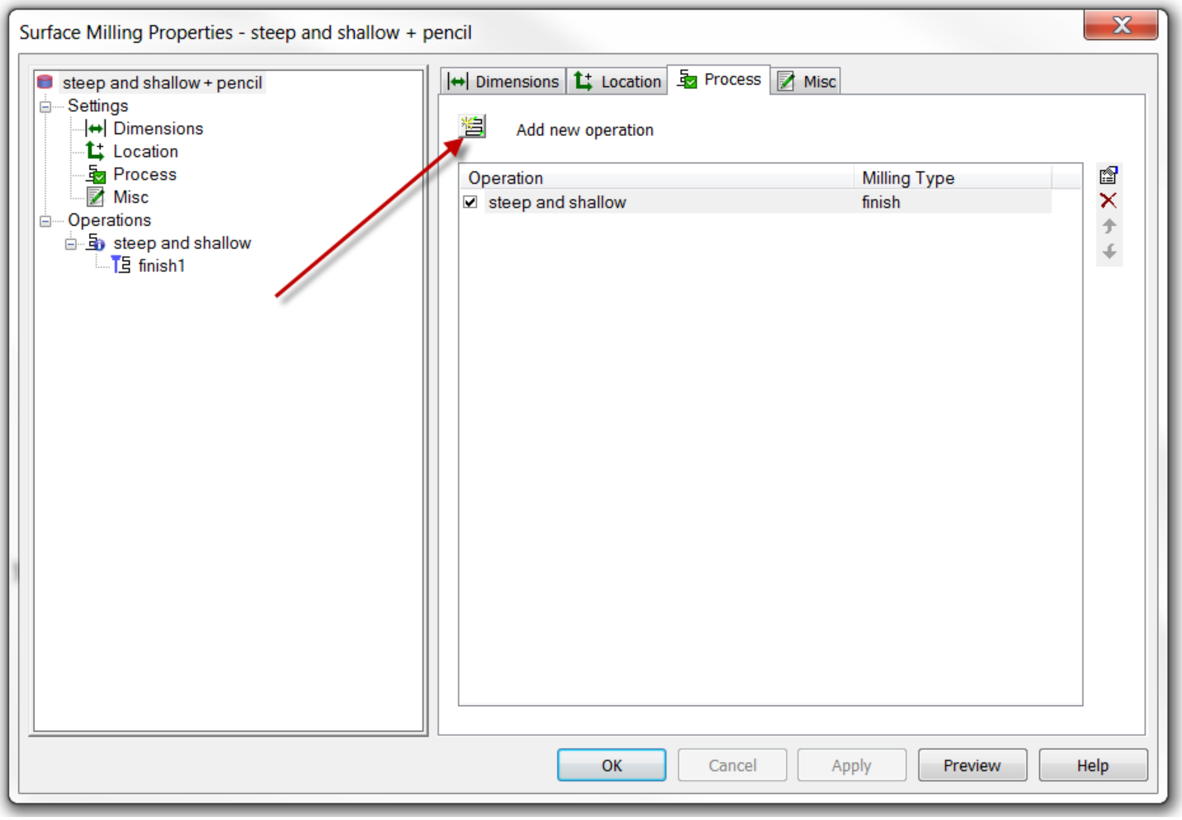
1 Please select the following surfaces. I have used the side surfaces as check surfaces to prevent the cutter from diving down the side walls.

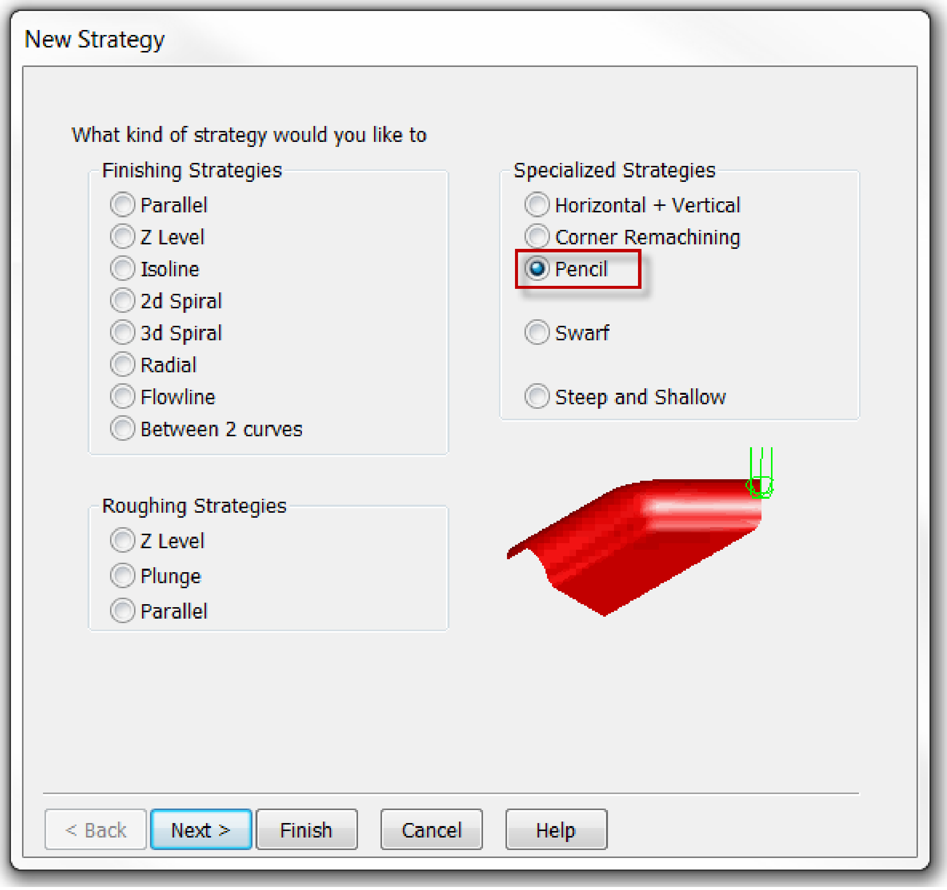


2 Create a New Surface Milling Feature. Select **Ctrl+R** to create a new feature. Use a **10mm Ballend** tool. Change the **Stepover** for **Steep** and **Shallow** to **0.5mm**. Change the retract to **Relative retract**.

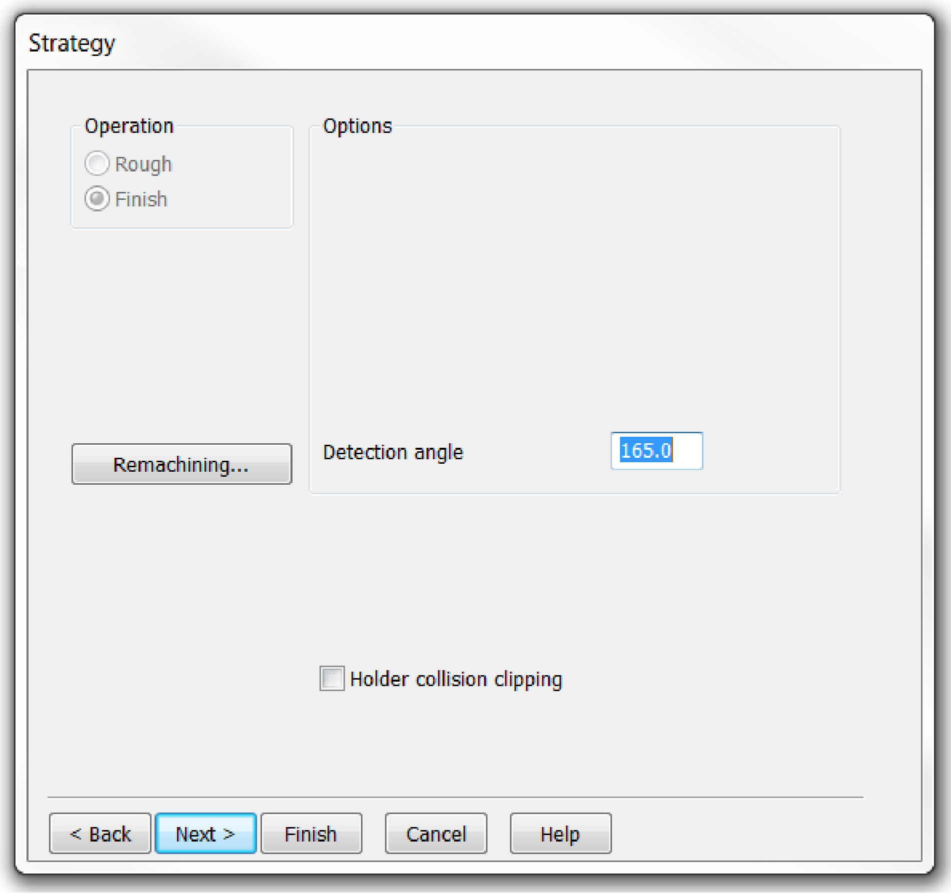


3 We can now add to the same process a **Pencil** strategy. Select the **Add new operation** Icon as shown below.

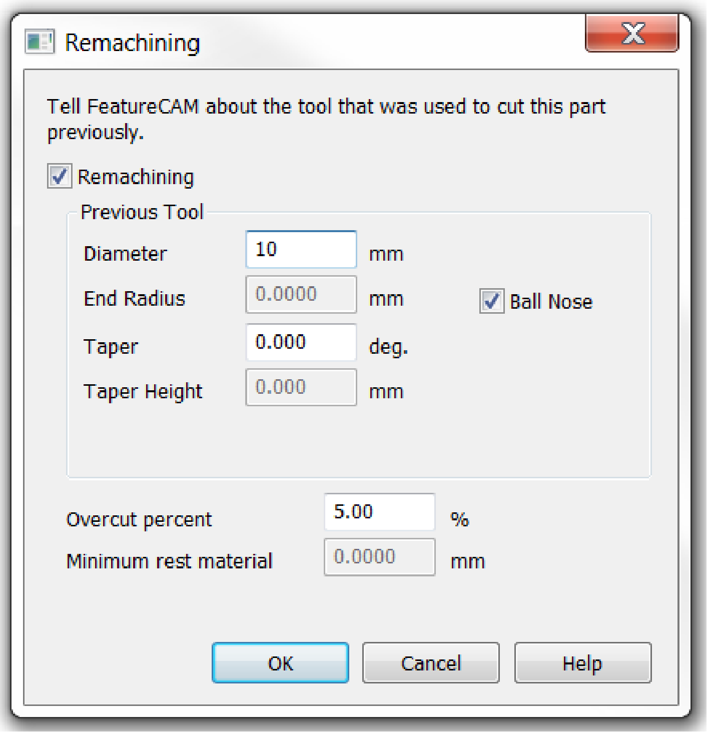




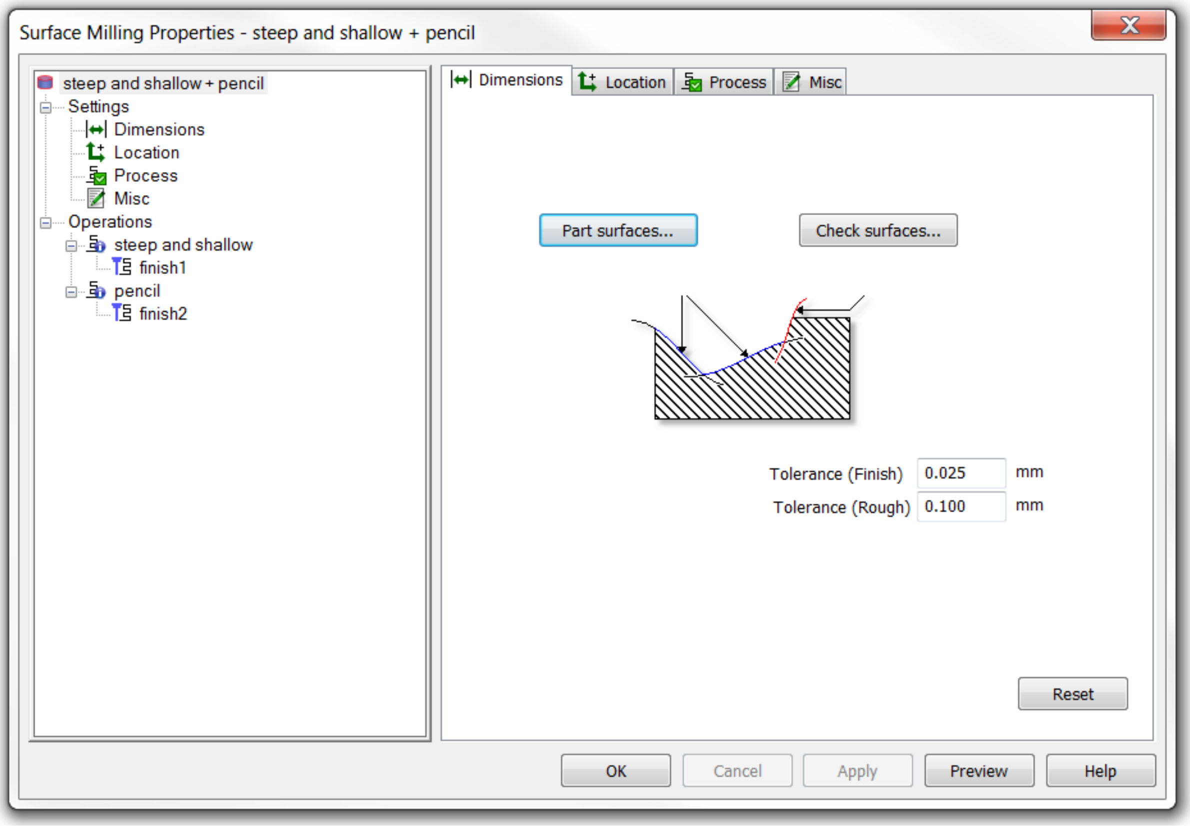
4 Select **Next**.



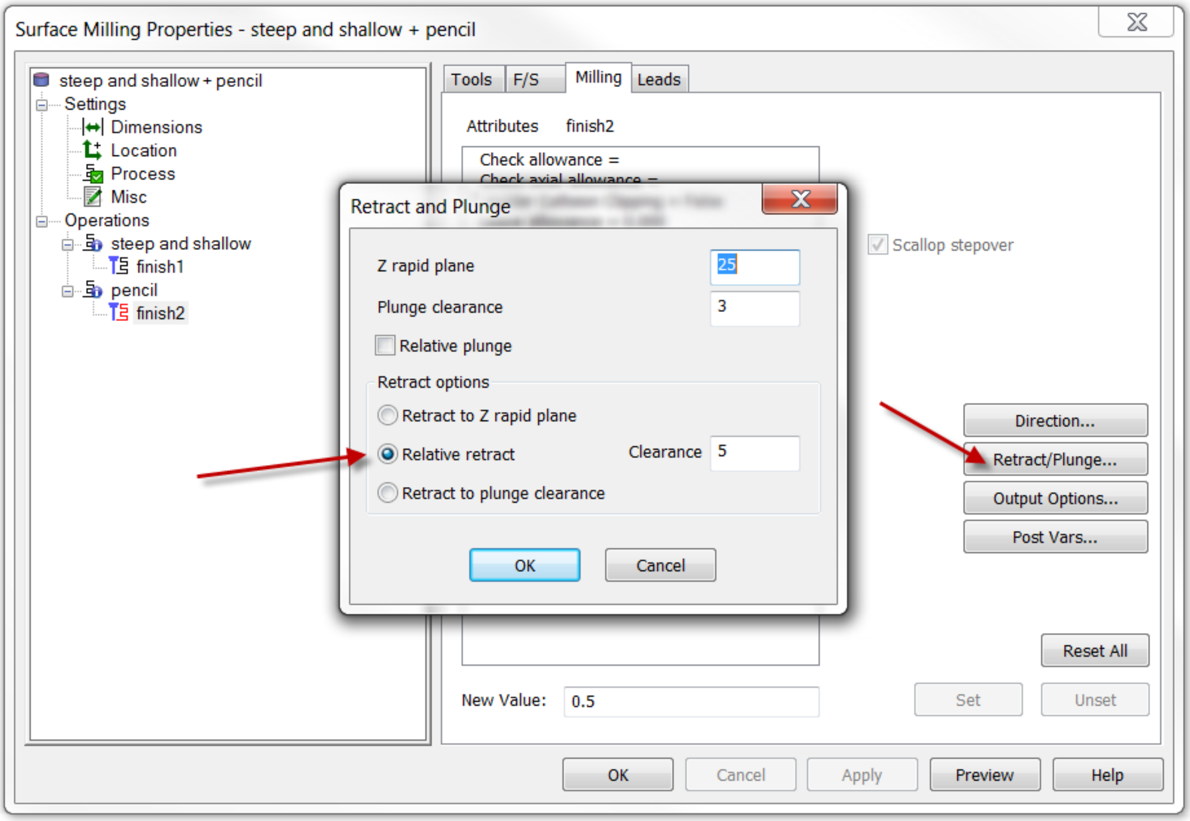
5 We need to activate the **Remachining** tab to let **FeatureCAM** know the size of the previous tool used. This is a **10mm Ball Nose** tool.

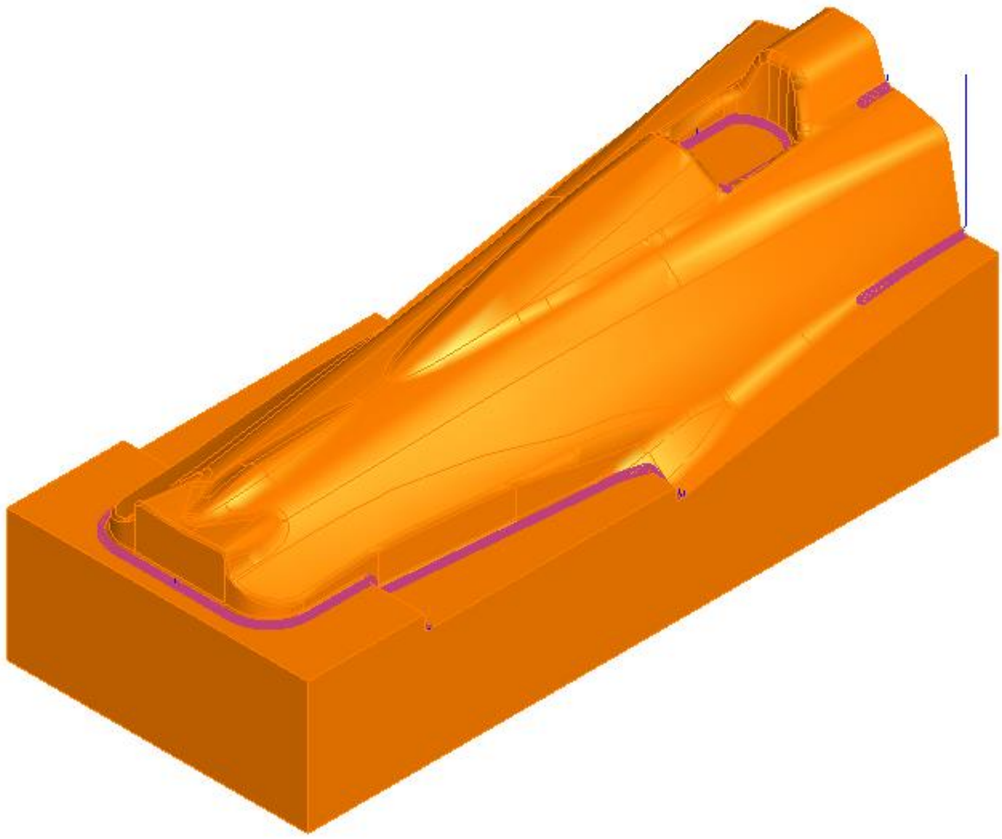


6 Select **Finish**.



7 Change to Relative retract under the **Finish2>Milling** tab.

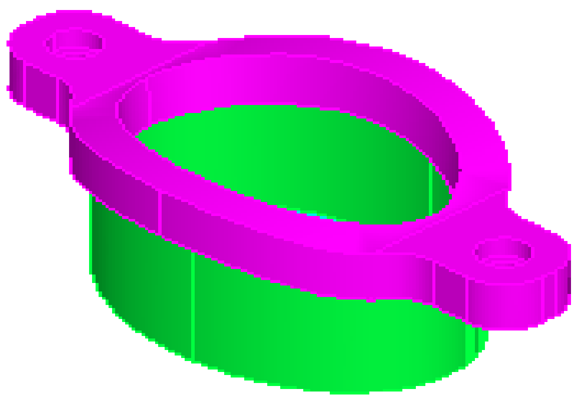




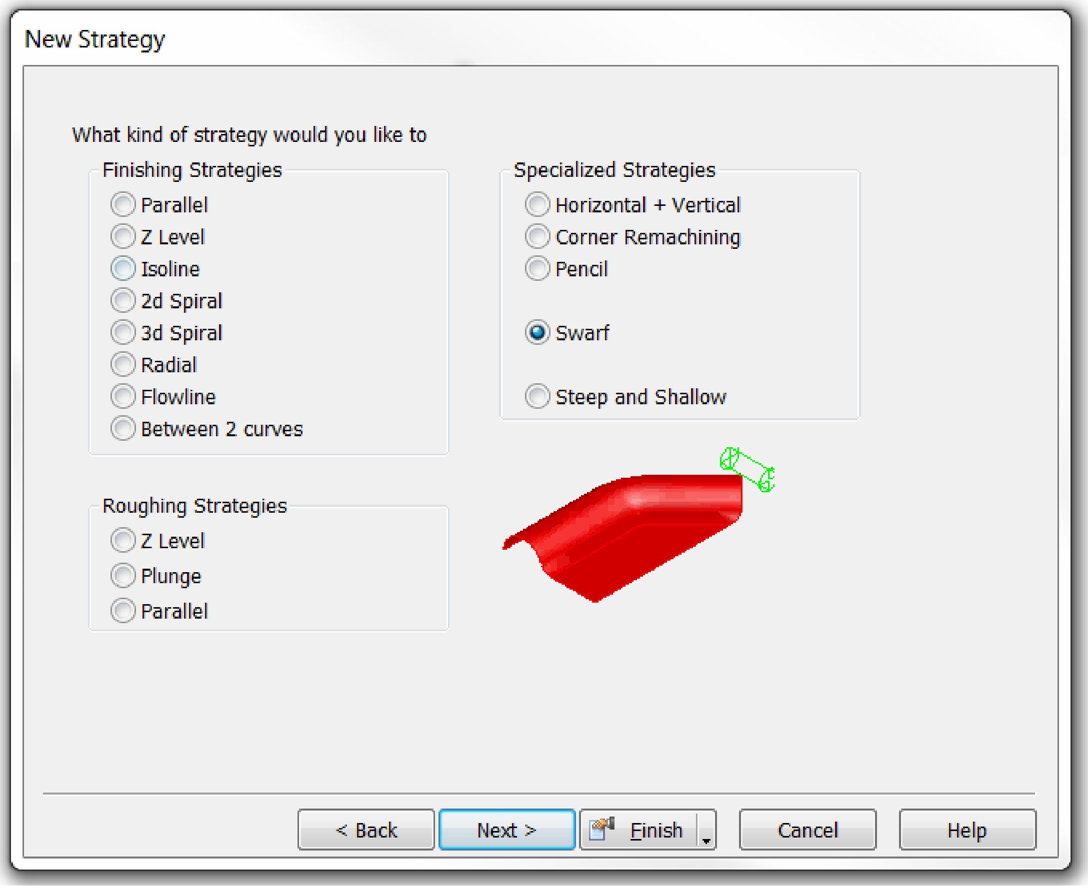
Swarf

Swarf cutting calculates toolpaths that cut with the side of the tool and works only on developable surfaces because the tool needs to be in contact with the surface for the whole cutting depth.

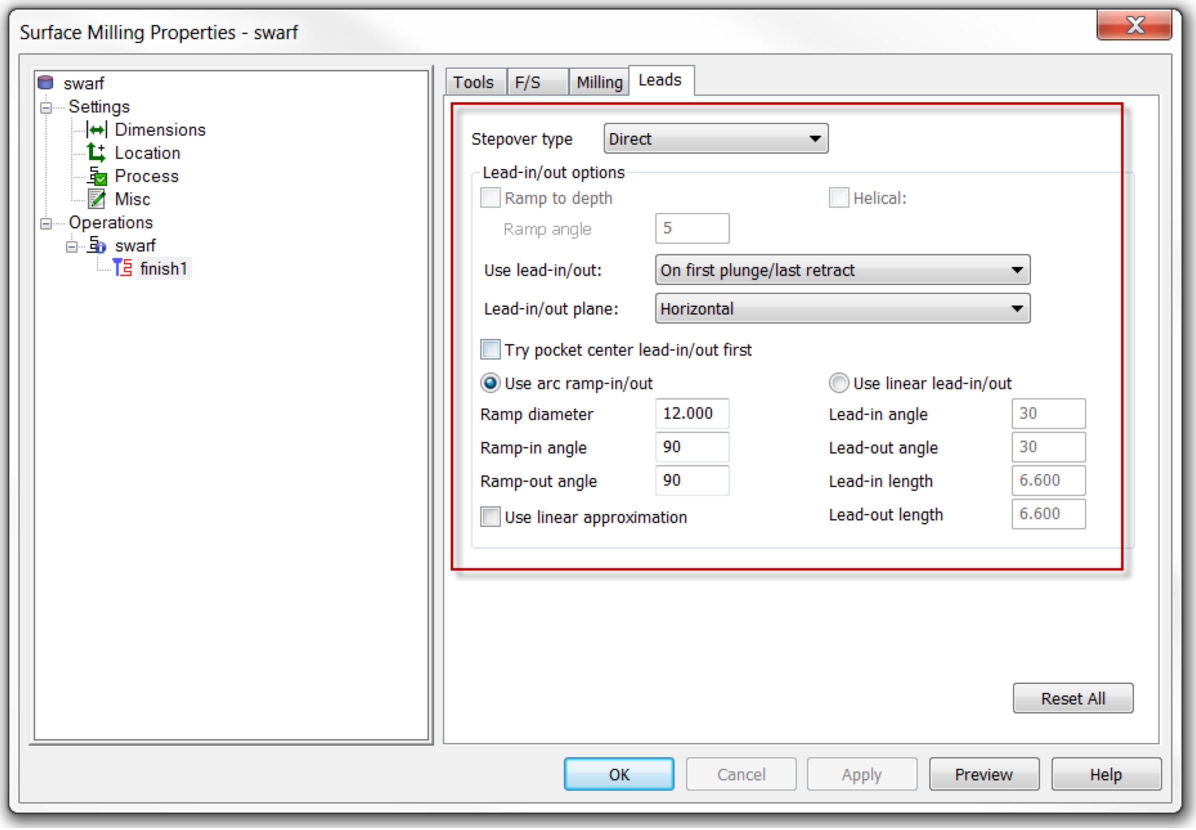
1 Select the following model. Select **Hide>Hide Unselected.**

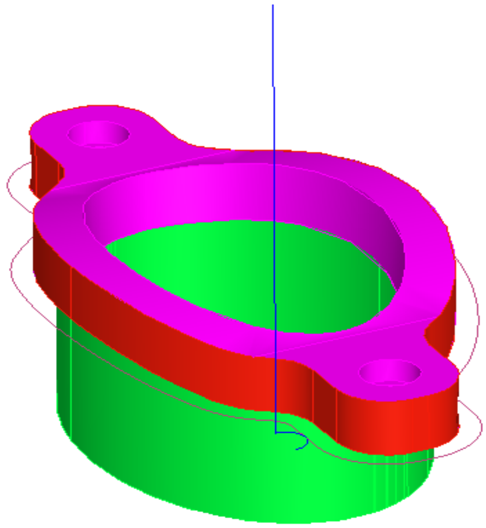


2 Please select the magenta outside surfaces. Then create a New Swarf Feature.



3 Please select **Finish**. Change the **Leads** to the following.



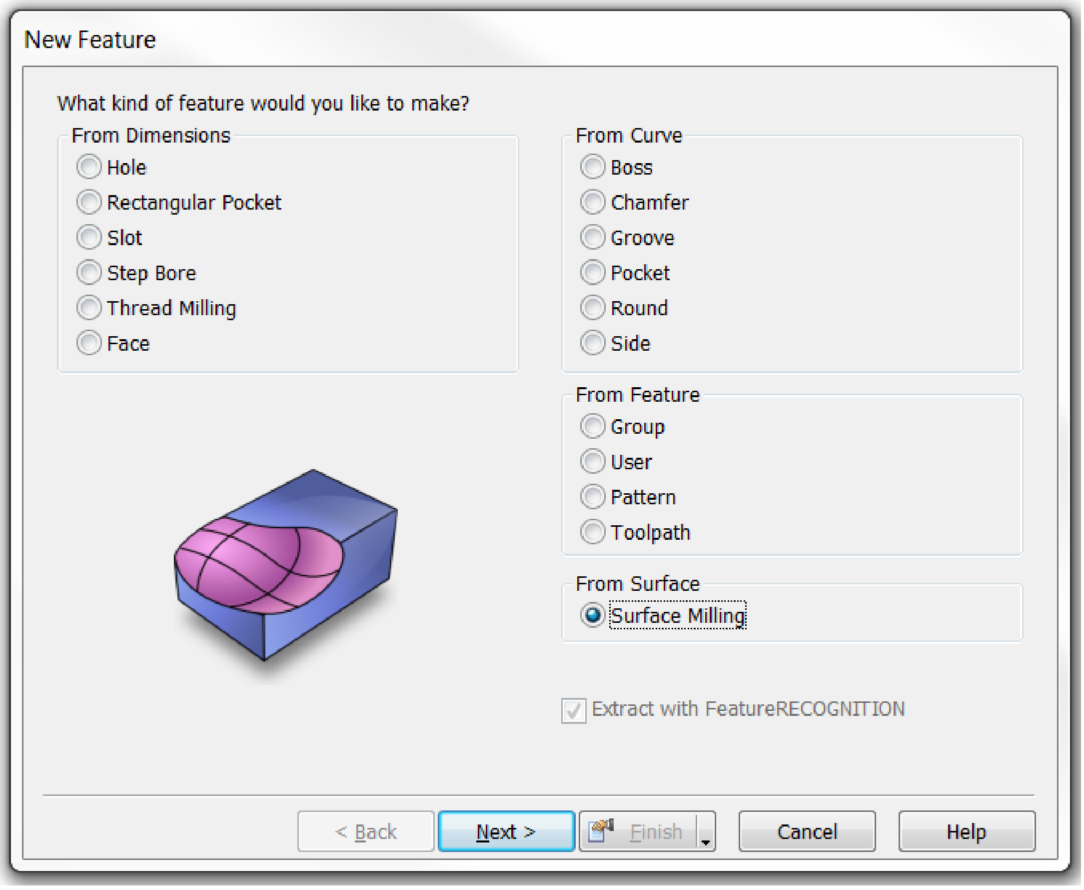


Steep and Shallow with Collision and Rest Machining

Steep and Shallow*: - This strategy calculates a shallow boundary, and then creates a* ***Z-level*** *toolpath in the steep areas of the model and a* ***Parallel*** *or* ***3D Spiral*** *toolpath in the shallow areas.*

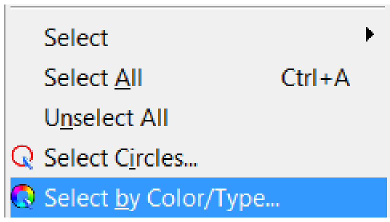
We will now machine the left hand solid model. You will see the area that needs to be machined, this is Yellow. The check surface is blue.

1 Create a new Feature **Ctrl+R**. **Surface Milling**.

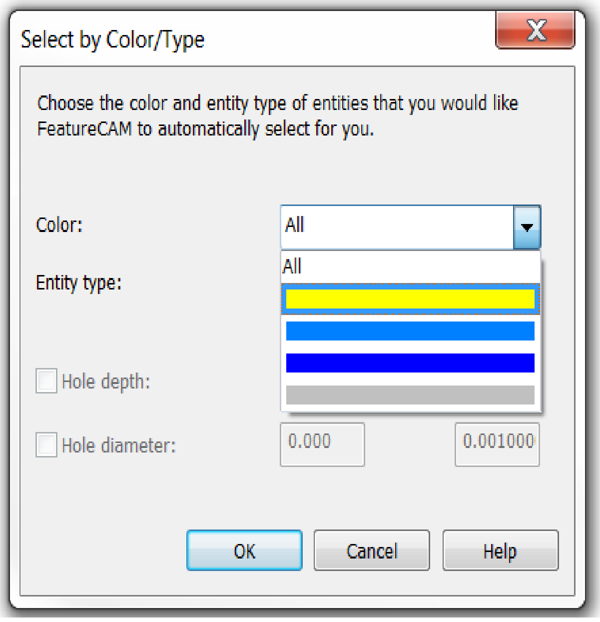


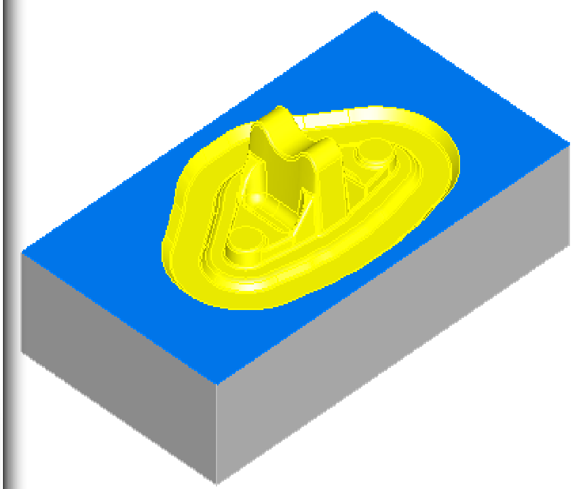
2 Select **Next**.

We are going to select the yellow surfaces. We can now select these by colour from the ***Edit*** *menu.*



3 Please select this option. See next page for the selection options available.





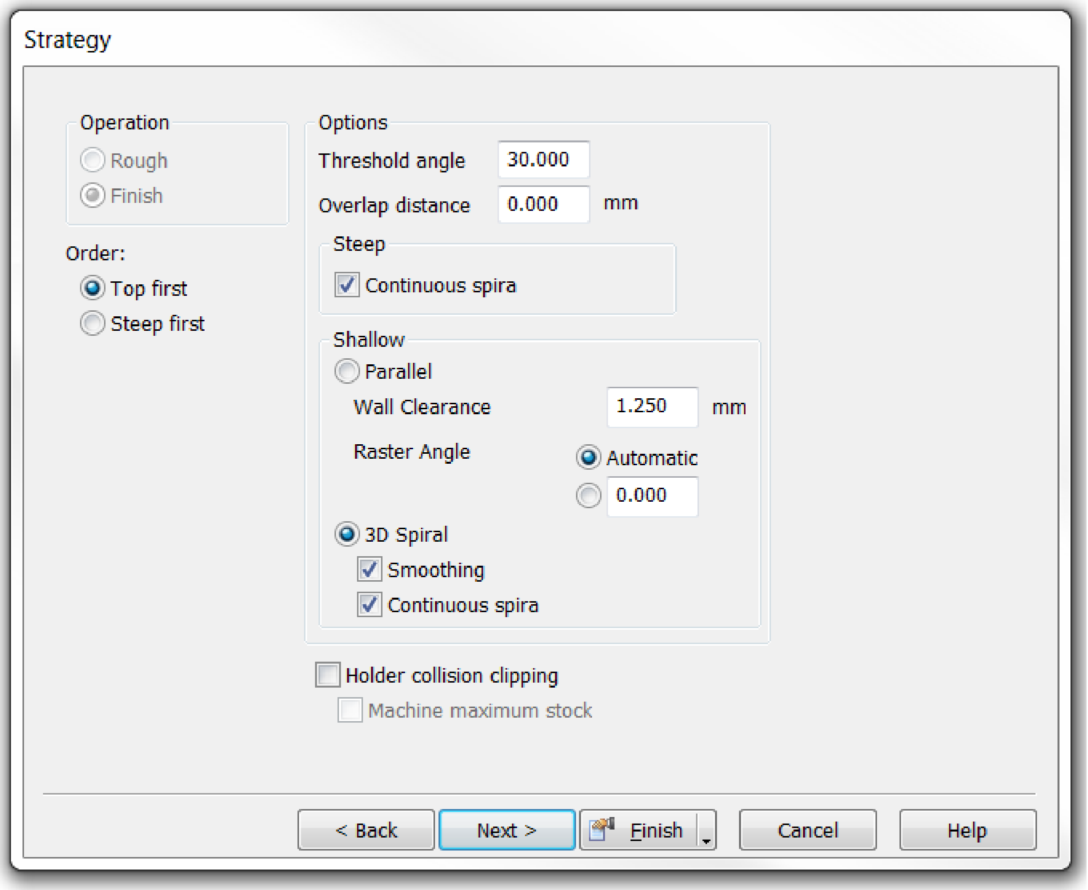
4 Select **Colour** and select **Yellow** from the colour drop down menu.

5 Select **Ok** to select. Then select the green + key to enter the selection.

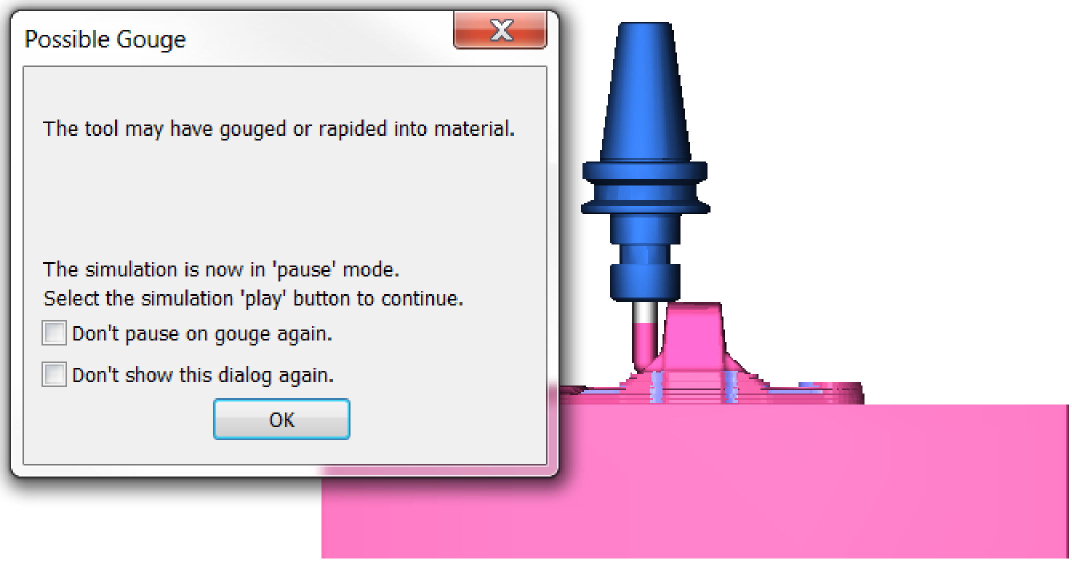
6 Select **Next** and then select **Choose a Single Operation**.

7 Select **Next** and choose **Steep and Shallow**. Select **Next**.

8 Please select the following options. (See menu)



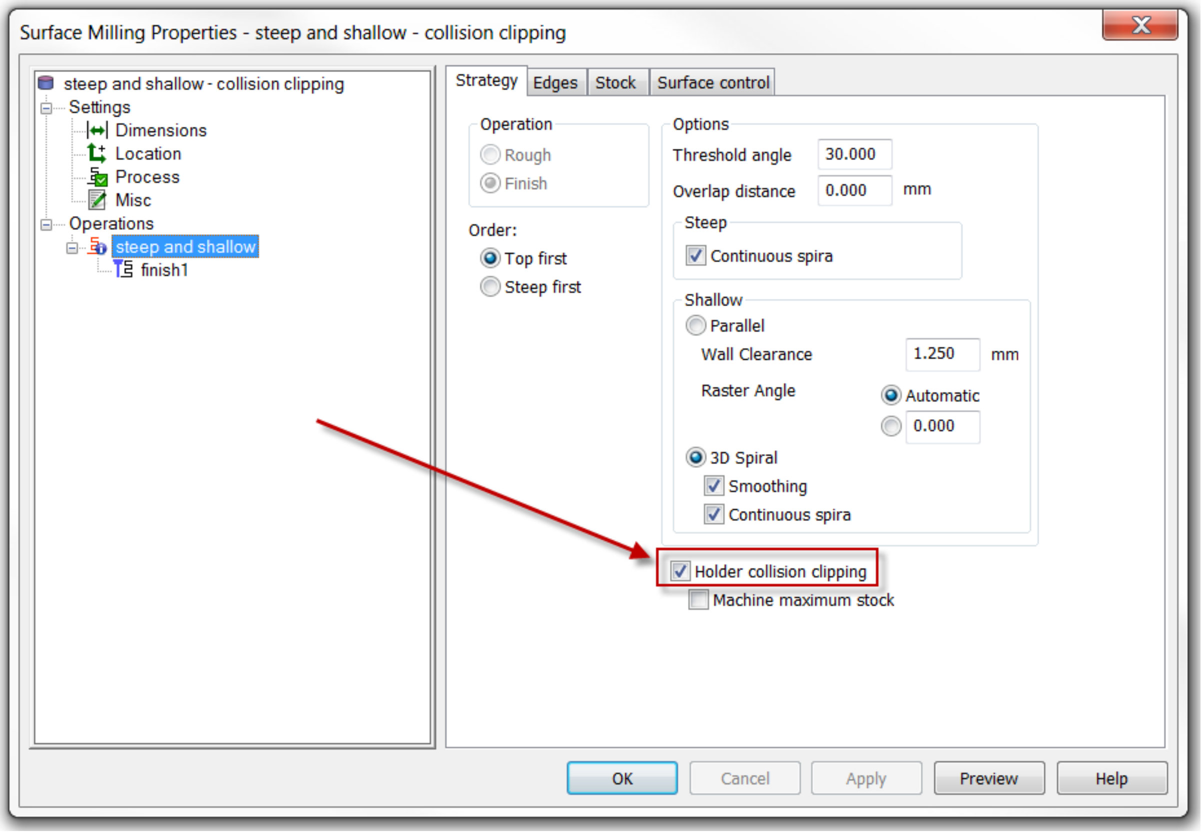
9 Select **Finish**. **FeatureCAM** should select a small short reach 12mm Ballend tool to finish the selected surfaces. We are not interested in the surface finish in this instance. This is to show you the tool gouging the part.



We have a number of options to avoid this.

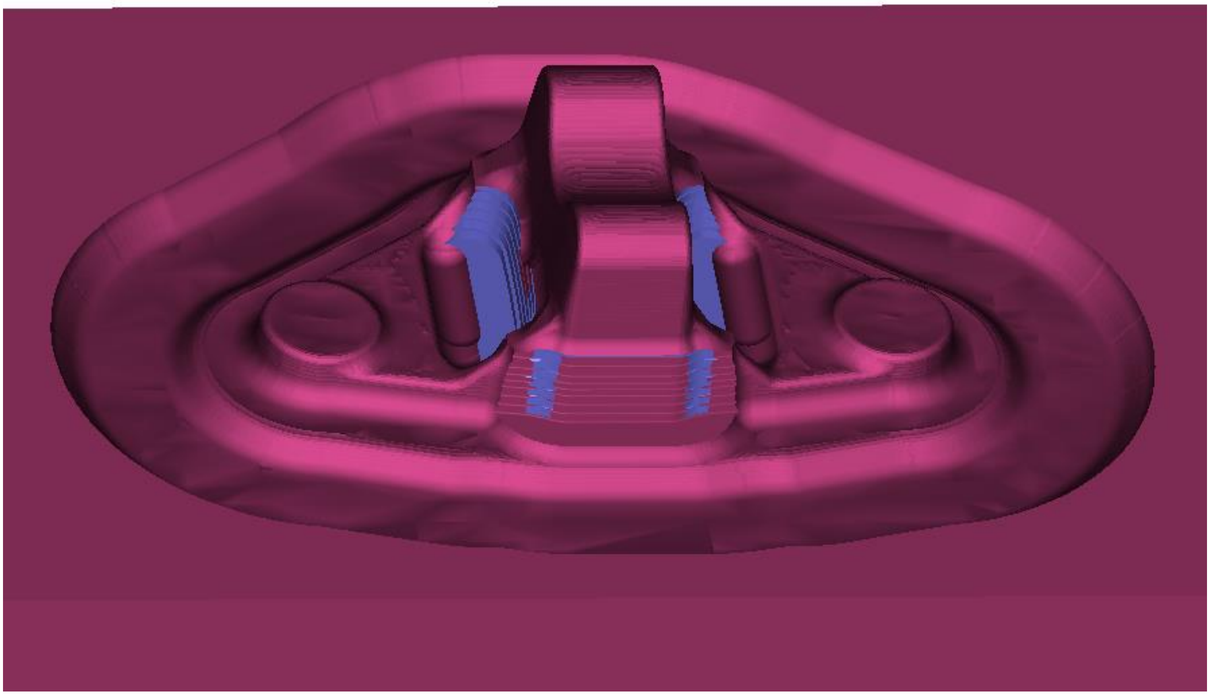
***1** – Use a longer tool.*

***2** – Use **Holder collision clipping**. See menu below.*



10 **Copy** File in **PartView**. Then paste the file. Rename the first Surface machining operation. **Steep and Shallow – original**. Then rename the copy and call this **Steep and shallow – Holder collision clipping**.

11 Edit the copy and select the option shown above. Run a **3D Simulation**.



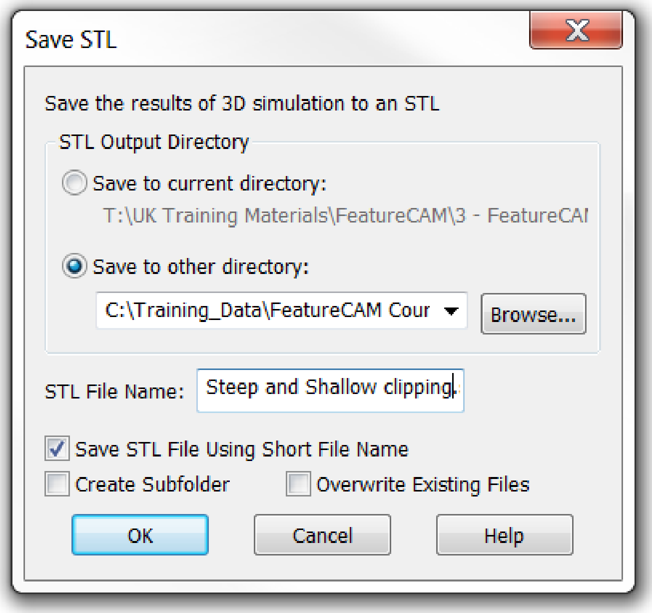
12 You can see that **FeatureCAM** has modified the toolpath to allow for the short tool.

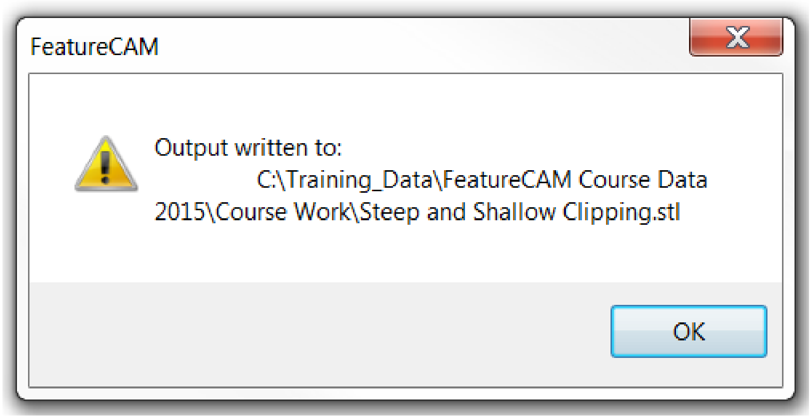
13 We will now create a Stock model of the toolpath just created. We can then just machine away the material that is left. This is known as rest machining.

14 Select **View>Simulation>Save simulation results.** This will create a .Stl file of the machine area we have just machined.

15 Call the file Steep and Shallow clipping.stl

16 Save the file in Course work in the FeatureCAM directory.

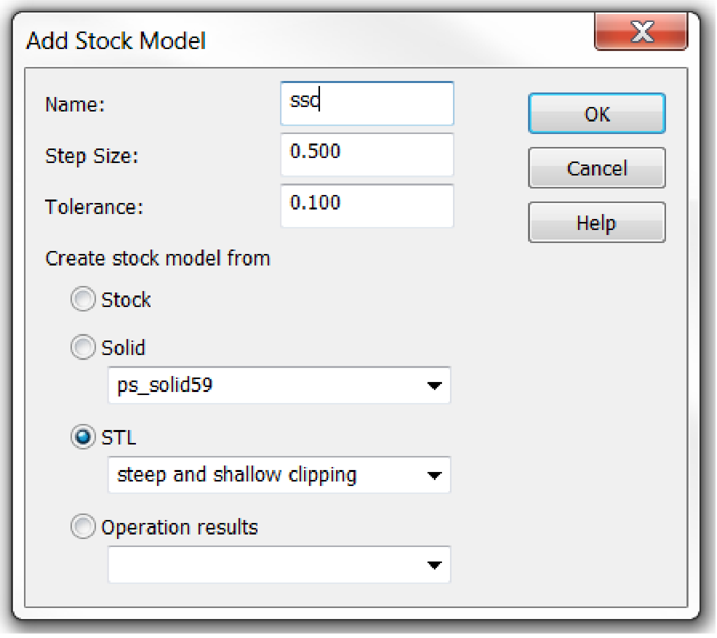




17 Import the .Stl file Steep and Shallow clipping.stl

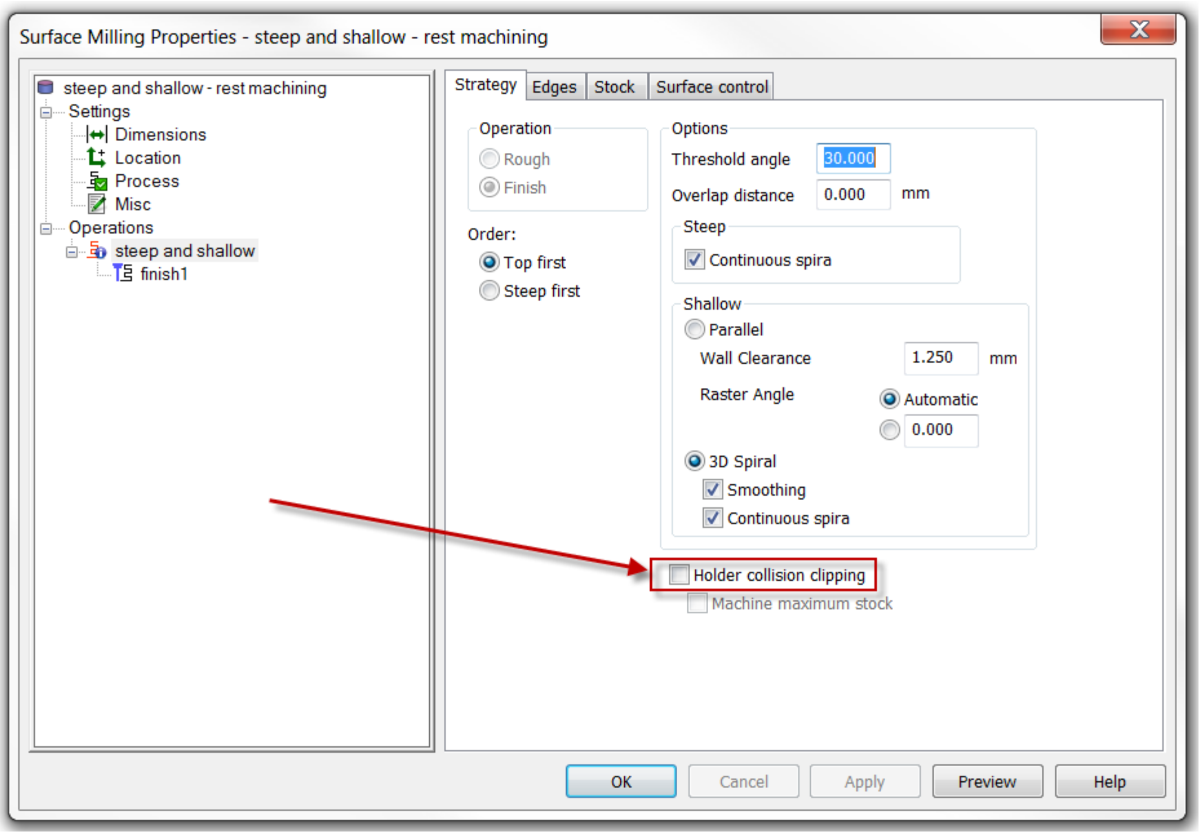
18 In PartView double click Stock Models and select Add.

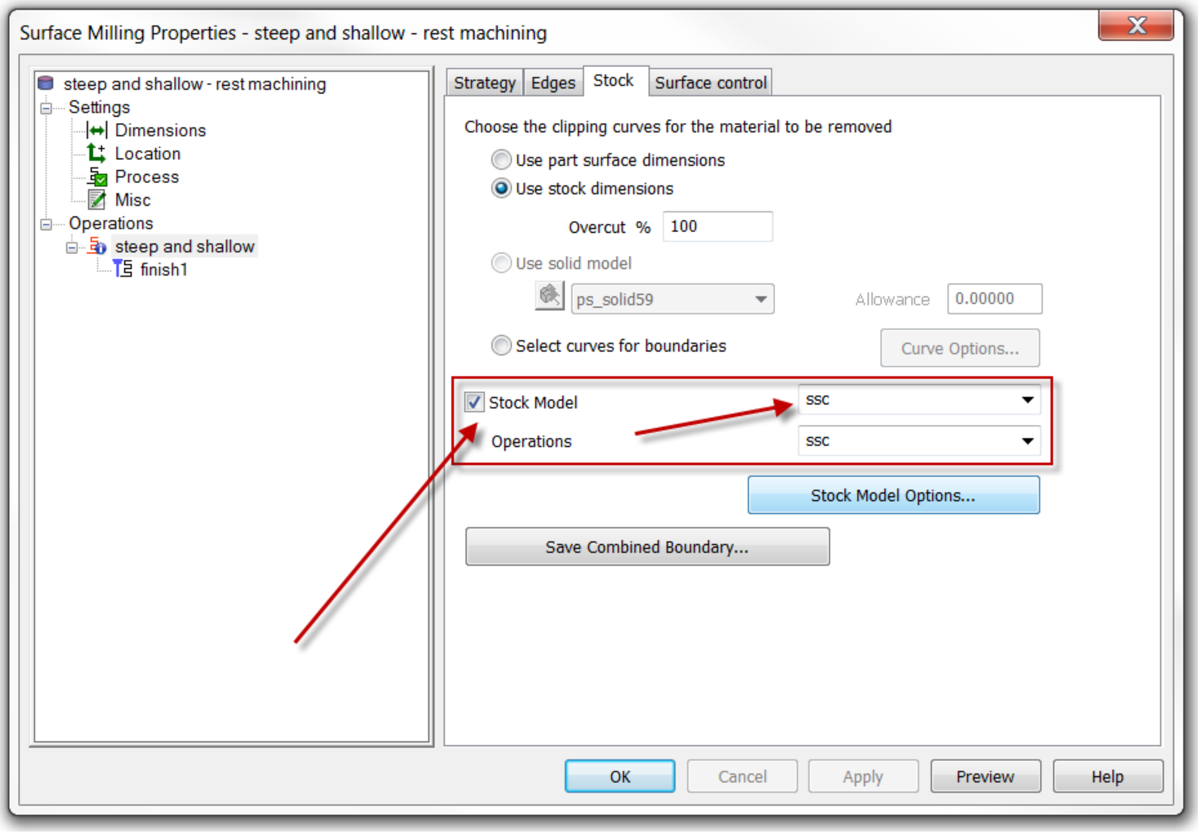
19 Select STL and select Steep and Shallow clipping. Name the Stock model as SSC.



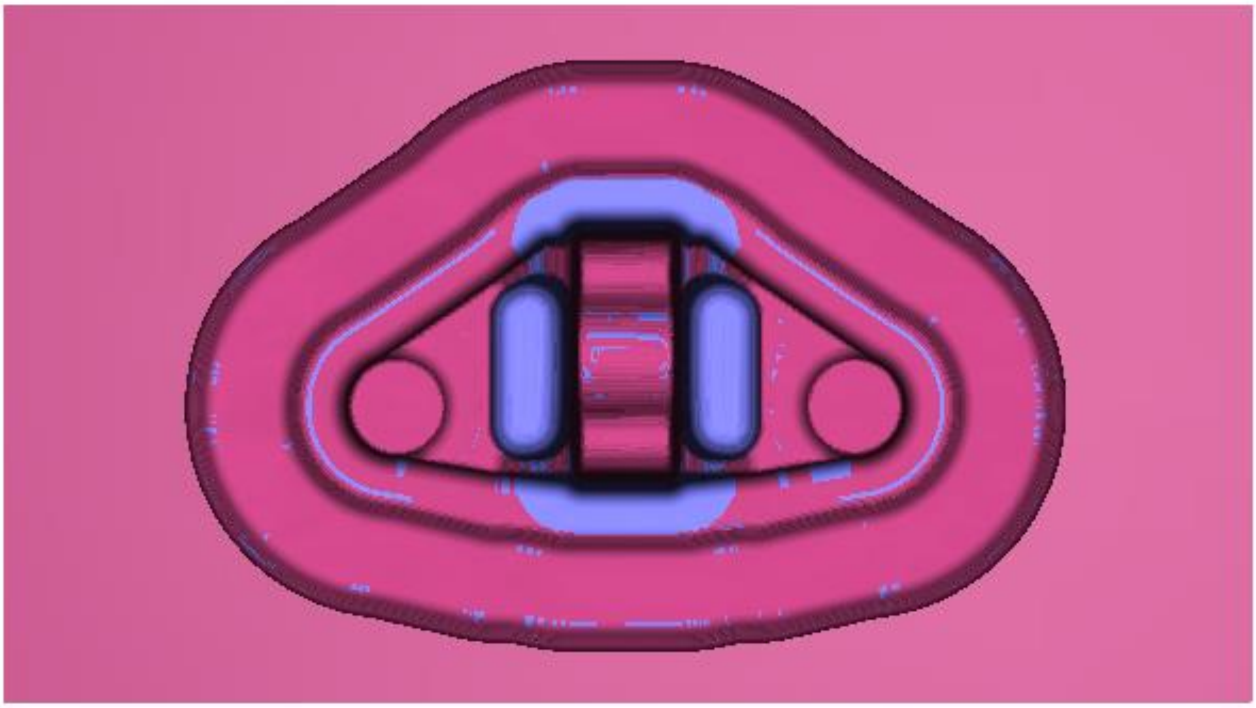
20 Copy the previous toolpath and call this **Steep and Shallow Rest Machining.**

21 Double click on the toolpath in PartView and change the following options on the next page.





22 Make sure you select a longer tool of the same diameter. If this does not exist please create one. Run a **3D Simulation** to see the following results.



*The next chapter covers **boundarie**s for selective **3D machining**.*

Contents

Stepovers, Leads & Ramps

Introduction

*During roughing operations, **FeatureCAM** gives the user a very high level of control over the way that the tool initially engages the part and how it steps over between each pass on a z level. These connecting moves are controlled from within the Leads tab of the roughing feature. We will now look at the options available for stepping over between passes.*

Stepover types

*When **FeatureCAM** clears a Z level it does so by means of a series of parallel passes, each separated from the next by the Rough pass stepover %. The way in which the passes are connected on a Z level is controlled by the Stepover type attribute on the Leads tab of the roughing feature. Stepover types that are available are Direct and Arc. By default the stepover type will be direct.*

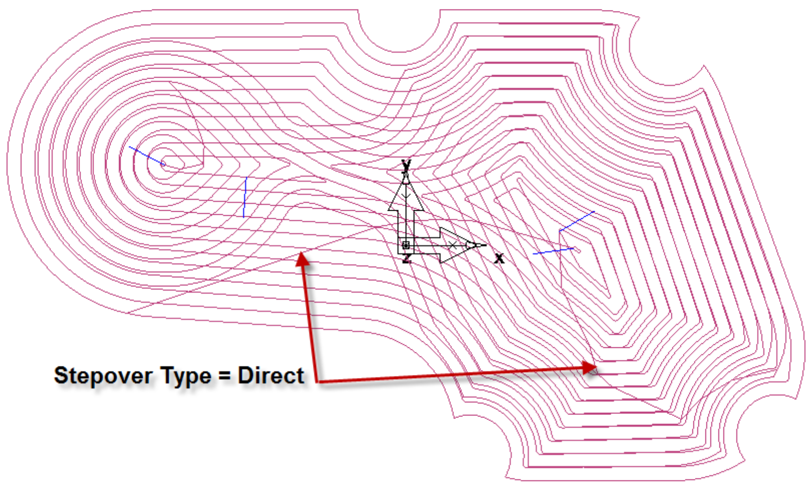
1 Open the part **Primary_Case_Pocket_Leads.fm**

2 Run a **Centreline** Simulation.

3 In the **Operation List** click on the **pull-down** next to **rough1 – Z level**.

4 Select the **first Z level**.

5 **Hide** all features and select a **Top view**.



*By default **FeatureCAM** makes a direct straight-line move from the end of one pass to the beginning of the next. This can lead to sudden changes in direction which may be undesirable, particularly during high speed machining.*

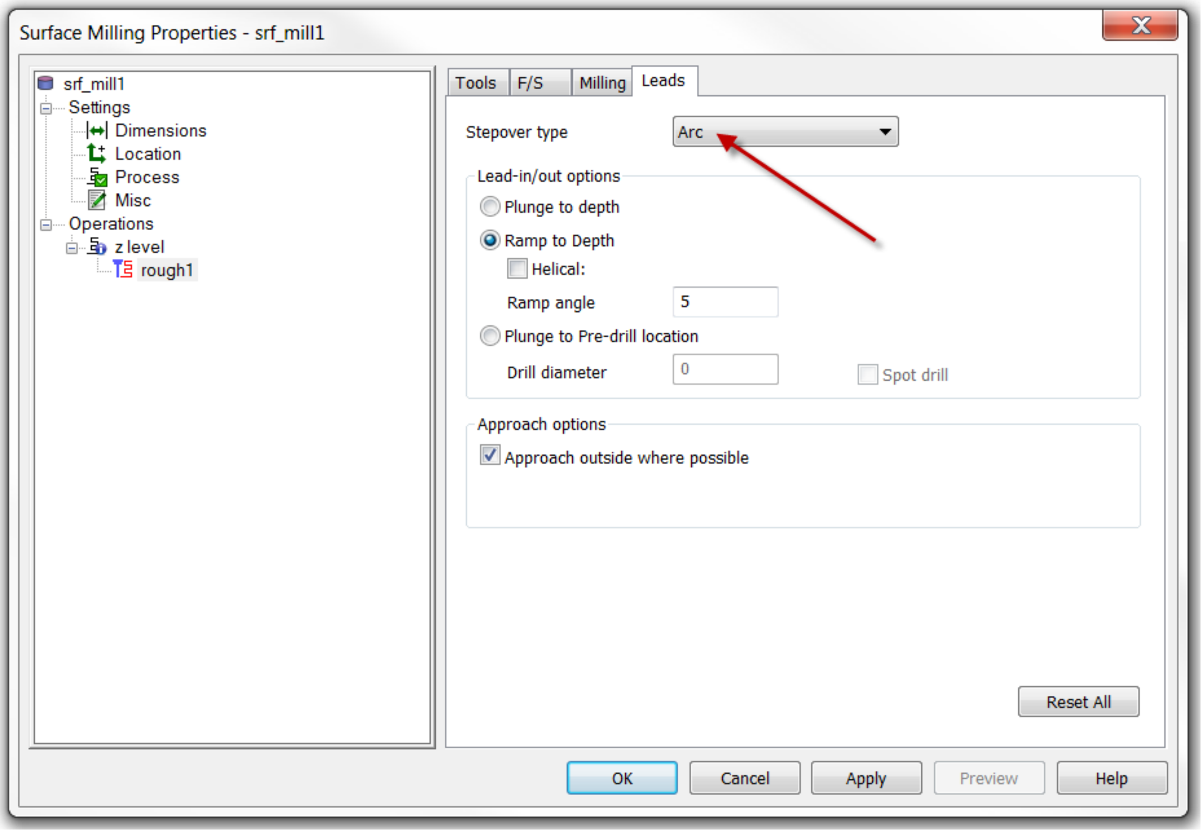
We will now change the stepover type from Direct to Arc. In an Arc stepover the tool makes a smooth circular arc move off one pass and then leads into the next pass with a second circular arc. This minimizes the rate of change in direction of the tool, and also reduces the changing of the tool load as it leaves one pass and engages the next.

6 **Eject** the simulation.

7 Edit the feature **srf_mill1** and click on **rough1**.

8 Select the **Leads** tab.

9 Change the **Stepover type** to **Arc**. **Apply** and **OK**.

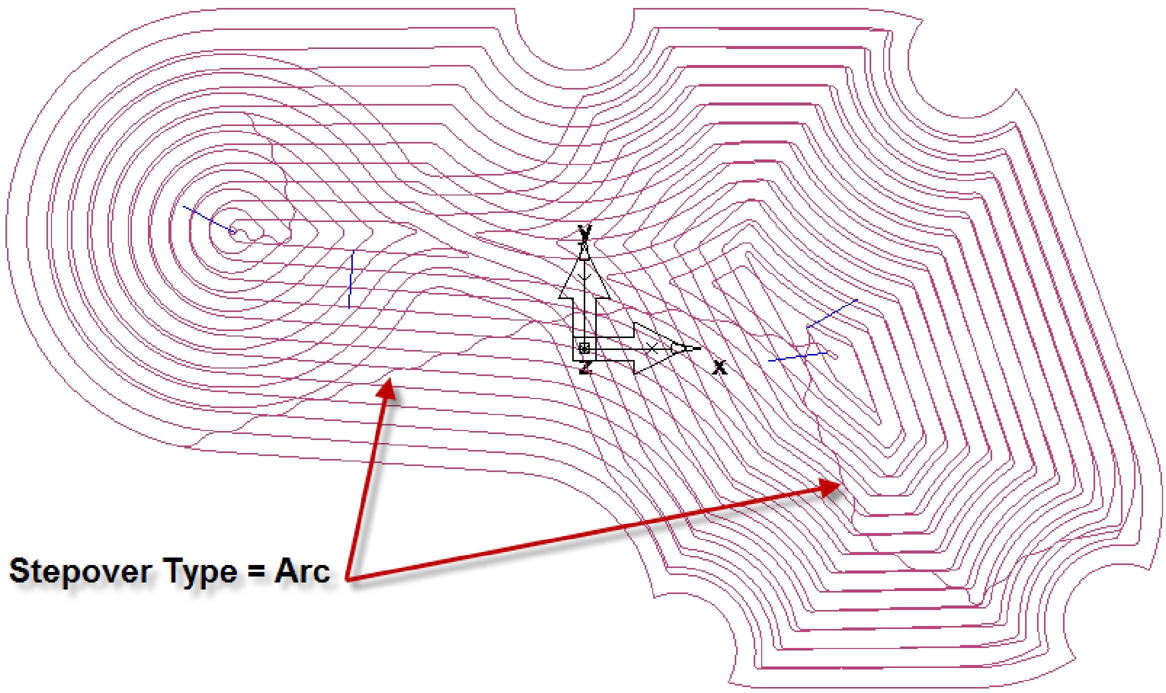


10 Run a **Centreline** Simulation.

11 In the **Operation List** click on the pull-down next to **rough1 – Z level**.

12 Select the first z level.

13 Hide all features and select a **Top** view.



The passes are now connected by a smooth S-shaped arc leading off from one pass and onto the next. This allows the machine to be run at a higher speed and minimizes shock loading on the tool, thus reducing chipping and wear.

Ramping moves

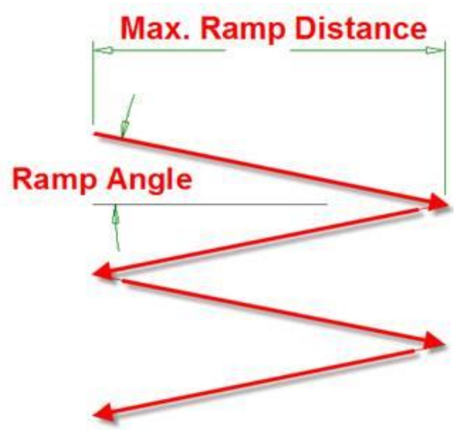
When **FeatureCAM** roughs out a pocket, by default, all approach moves to the part will be ramps. That is, instead of plunging the tool directly into the material the approach will be made by a series of zigzag moves which progressively move the tool down into the material. We will now examine how these ramping moves may be controlled.

Ramp angle and Maximum ramp distance

The two attributes that control linear (straight line) ramping are the **Ramp angle** and the **Maximum ramp distance**.

The **Ramp angle** controls the angle at which the tool approaches the part while moving downwards to the roughing z level. By default this is 5 degrees. In practice the angle that can be used will depend upon the type of tool being used and the material. For example with a centre cutting tool it may not be necessary to ramp at all whereas an insert cutter may require a shallow ramp angle.

The **Maximum ramp distance** controls the length of the Zigs and Zags in the ramp. If no value is explicitly set by the user then **FeatureCAM** will base the maximum ramp distance on the tool diameter. The user can also choose to enter a value based upon the cutter geometry so as to prevent build-up of swarf below the tool centre.



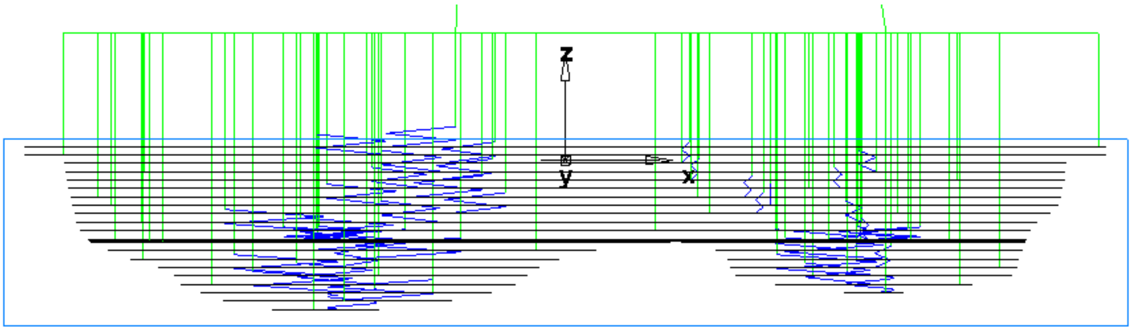
The diagram above shows a side view of a toolpath ramping into a part.

14 Run a **Centreline** Simulation.

15 Click on the pulldown next to **rough1** in the **operation list**.

16 In PartView Hide the Solid Part and change the simulation to Show Centreline

17 Change to **Front** view or **Ctrl** + **2**.



The tool approaches the material by making a series of zigzag moves at the default angle of five degrees to the horizontal plane. Form simulation purposes I have changed the Maximum Ramp Distance to 25mm.

18 **Eject** the simulation.

19 Edit the feature **srf_mill1** and click on **rough1.**

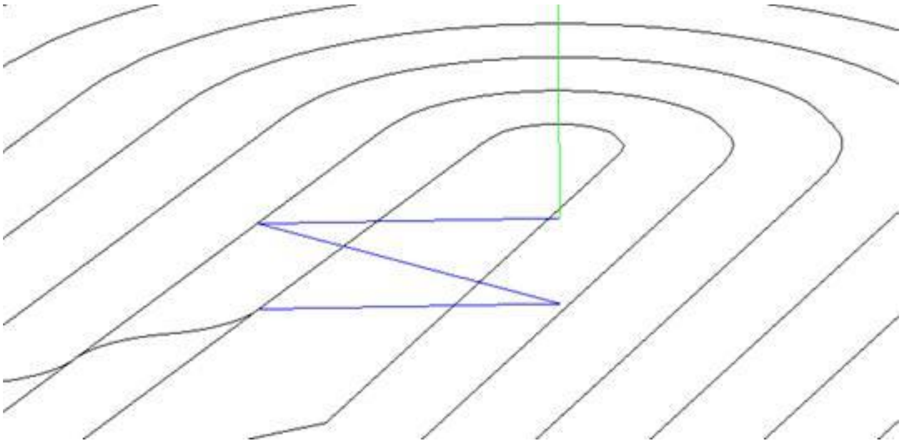
20 Select the **Leads** tab.

21 Set the **Ramp** angle to 15 degrees.

22 Run a Centreline simulation. **Apply** and **OK.**

23 Click on the pull-down next to **rough1** in the **operation list.**

24 Select the first roughing level.



The tool is now approaching the material at a steeper angle and making fewer moves on the way down. Approach moves are made at a slower feedrate than normal cutting moves, typically 50% slower. This means that the amount of time spent ramping can end up being a significant proportion of the overall machining time. By increasing the ramp angle we are reducing the total distance travelled in the approach moves and thus reducing the time taken to machine the part.

25 **Eject** the simulation.

26 Edit the feature **srf_mill1** and click on **rough1**.

27 Select the **Milling** tab.

28 Set the **Max. Ramp distance** to 50mm

29 Click **Apply** and **OK**.

30 Run a **Centreline** simulation.

31 Click on the pull-down next to **rough1** in the operation list.

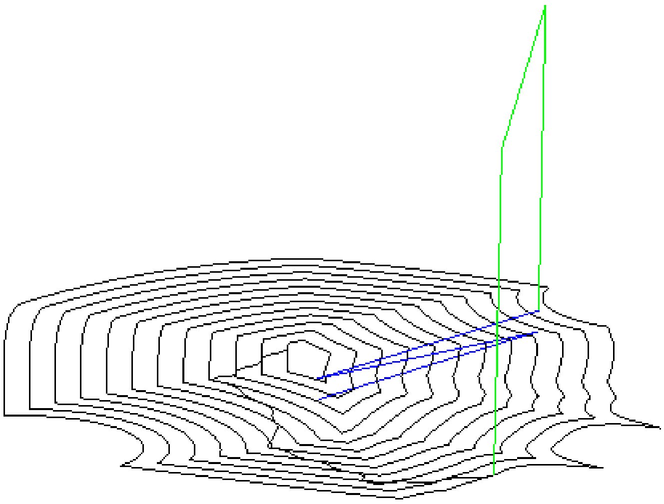
32 Select the First roughing level.

Note how the length of the ramping moves has increased from the default of the tool diameter to 50mm.

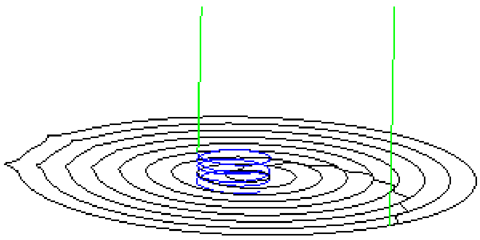
33 Click on the pull-down next to **rough1** in the **operation list**.

34 Select the roughing level near Z -21.324**.**

At this lower level the size of the part is smaller and consists of two separate pocket areas. ***FeatureCAM*** *uses the specified ramp distance wherever possible. In the smaller round pocket area it is not possible to fit in a ramp with the specified length of 50mm so* ***FeatureCAM*** *automatically switches to using a helical ramp. It is possible to force* ***FeatureCAM*** *to use a Helical ramp in all cases:*



Straight Ramp



Helical Ramp

35 **Eject** the Simulation.

36 Edit the feature **srf_mill1** and click on **rough1**.

37 Select the **Leads** tab.

38 Check the **Helical** check box below the **Ramp to depth wording**.

39 **Apply** and **OK**.

40 Run a **Centreline** simulation.

All of the ramping moves are now helical. It should be noted that the Maximum ramp distance also applies to helical ramping moves. Rather than defining the length of the move before the direction is changed for a helical ramp the distance defines the diameter of the helix.

41 **Eject** the simulation.

42 Edit the feature **srf_mill1** and click on **rough1**.

43 Select the **Milling tab.**

44 Change the **Max. Ramp distance** to **6mm**. **Apply** and **OK.**

45 Run a **Centreline** simulation.

Note how the diameter of the helical ramp is reduced.

Plunging to Pre-drilled holes

If a roughing cutter does not centre cut ramping moves can be used to allow the cutter to engage the material without causing damage to either the cutter or the part. The main disadvantage of ramps is that the cutter is moving at a reduced feedrate and making unproductive moves. Where a part has many roughing levels and a shallow ramp angle used the ramps can become a significant proportion of the overall cutting time. It is often more efficient to use a pre-drilling operation to provide a location where the roughing tool can plunge into free space and then engage the material in a horizontal cutting move.

46 Eject the simulation.

47 Edit the feature **srf_mill1** and click on **rough1**.

48 Select the **Leads** tab.

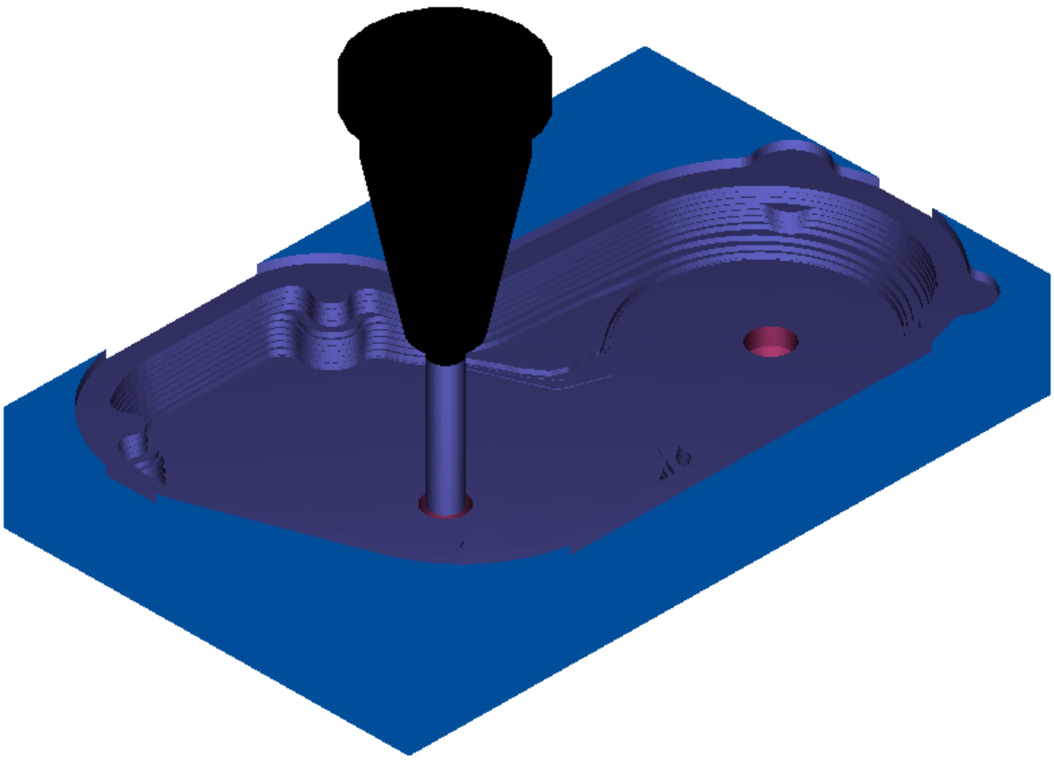
49 Check **Plunge to pre-drill location.**

50 Set the **Drill diameter** to **16mm. Apply** and **OK**.

51 It is also possible to add a Spot drill operation if required, in this case we will not.

52 Use **Play to next rapid** in a **3D Simulation**.

The tool plunges into the centre of the pre-drilled holes. The locations of the holes are automatically calculated to lie at the lowest point of each enclosed area of the part.



Contents

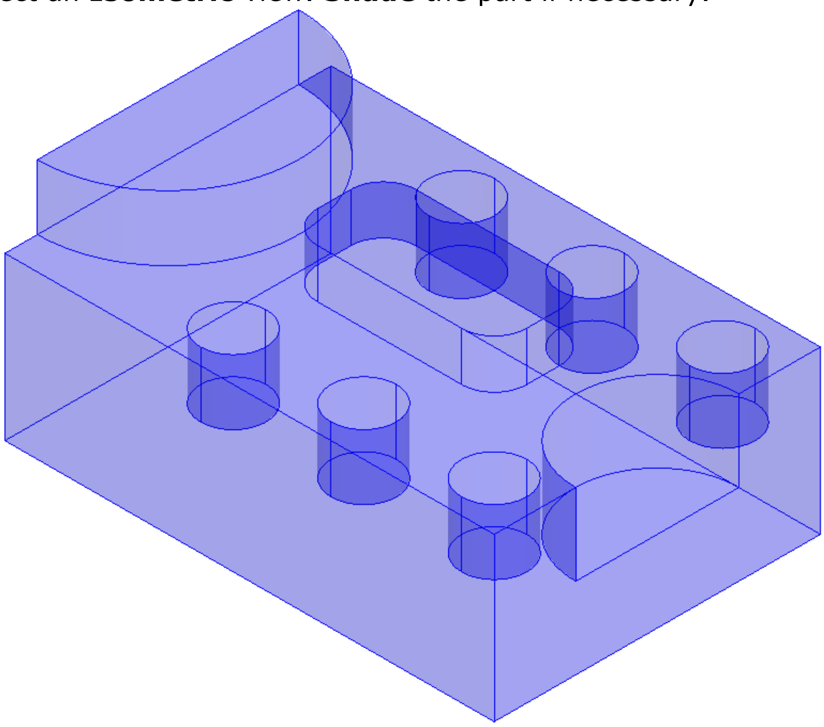
Roughing Flat Areas

Introduction

The area clearance strategies in FeatureMILL3D have an option that allows the user to control the way in which flat areas of the model are rough machined. There are cases in which the user will want to ignore flat areas leaving extra material in the vertical direction on those areas. Usually it is necessary to remove material from the horizontal areas to leave the same amount of stock there as on the rest of the part.

1 Open the file **Flats.fm**.

2 Select an **Isometric** view. **Shade** the part if necessary.



This model contains several horizontal flat areas. If these areas are not machined there may be an excess of stock remaining for the semi-finishing process to work well. We will now look at how to control the machining of flats.

3 Click on **Features** in the Step view.

4 Check **Surface Milling** and press **Next**.

5 **Box select** all of the surfaces and then press the **Add from selected items button**  then click next.

6 On the **New Strategy** dialog check **Choose a single operation**.

7 Click **Next**.

8 Check **Z Level** in the **Roughing Strategies** part of the dialog.

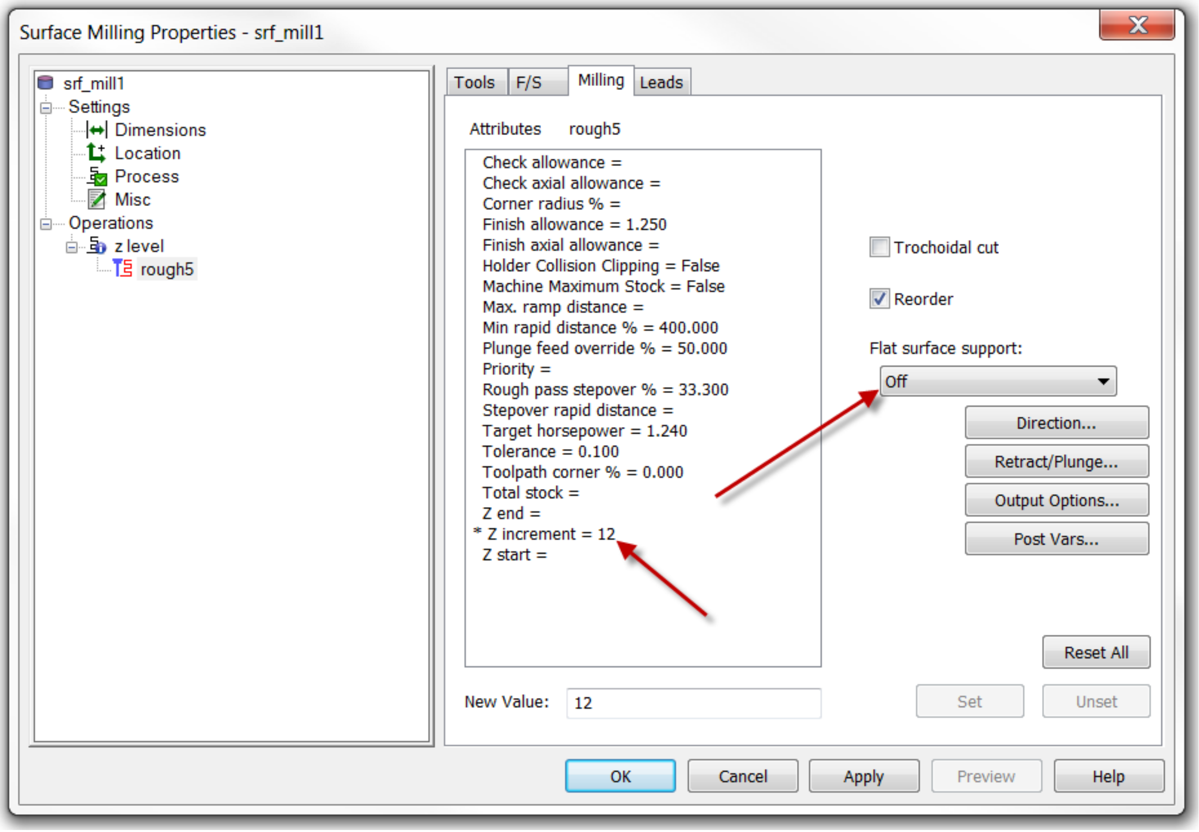
9 Press **Finish**.

10 Edit the feature.

11 Click on **rough1** in the tree view.

12 Select the **Milling tab**.

*Fill in the form exactly as shown below, setting **Flat surface support** to **Off** and setting the **Z increment** to **12mm**.*



13 Press **Apply** and **OK**.

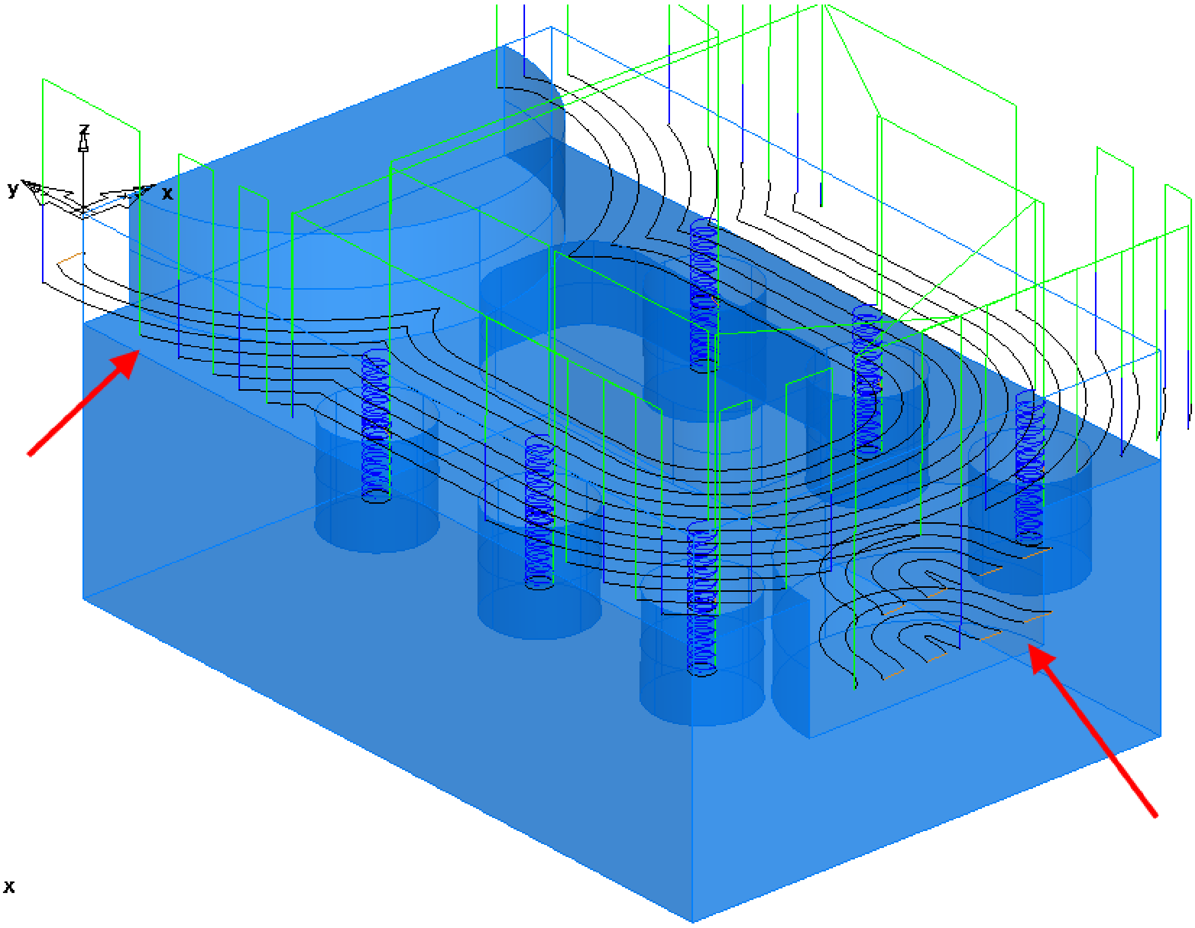
14 Click **Manufacturing** on the main menu.

15 Choose **Post Process**.

16 Set the **Tool change location Z** to be **100mm**. Click **OK.**

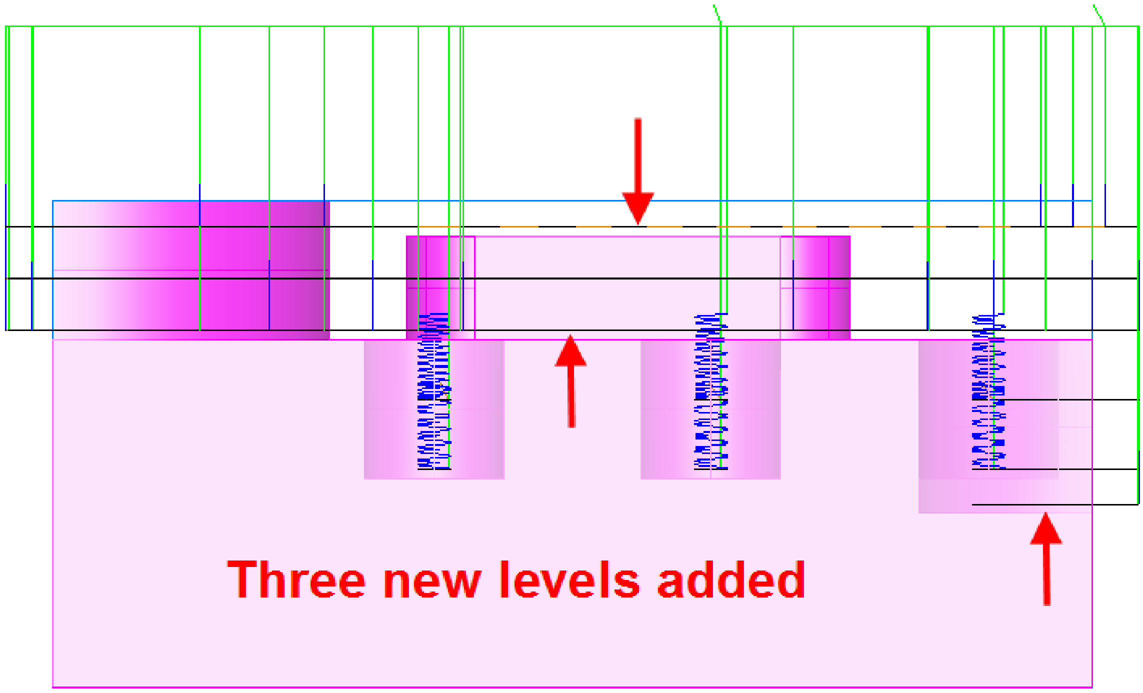
17 Run a **Centerline simulation**.

*As flat surface support is turned off, FeatureMILL3D has ignored all of the Flat Surfaces and cut only at the specified Z levels **(every 12mm down from the top of the stock).***



It can be seen that a lot of stock remains in the areas indicated. This could cause problems for subsequent operations. We will now look at the alternatives.

18 Edit the feature.
19 Click on **rough 1** in the tree view.
20 Select the **Milling tab**.
21 Change **Flat surface support** to **Level**.
22 Click **Apply** and **OK**.
23 Run a **Centerline simulation**.
24 Select a view from the **Left**.



Three extra roughing levels have been added. These are offset from the flat surfaces by the Finish Allowance plus the Rough Tolerance. The levels extend all the way to the edge of the block thus increasing the machining time. If a part has many flats the machining time may become very long.

25 Select an **Isometric** view.

26 Run a **3D Simulation** to see how the flat levels are cleared all the way to the edge of the stock.

We will now look at the most efficient way of machining the flat areas on the part.

27 Edit the feature.

28 Click on **rough 1** in the tree view.

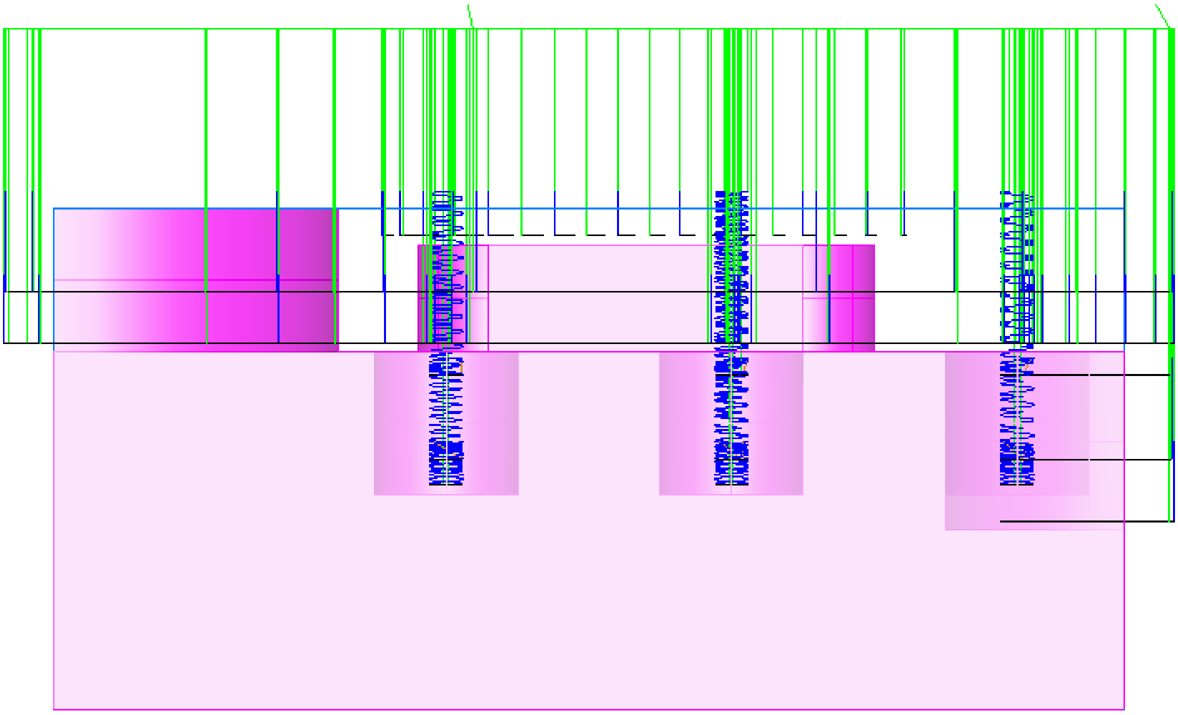
29 Select the **Milling tab**.

30 Change **Flat surface support** to **Area**.

31 Click **Apply** and **OK**.

32 Run a **Centerline simulation**.

33 Select a view from the **Left**.



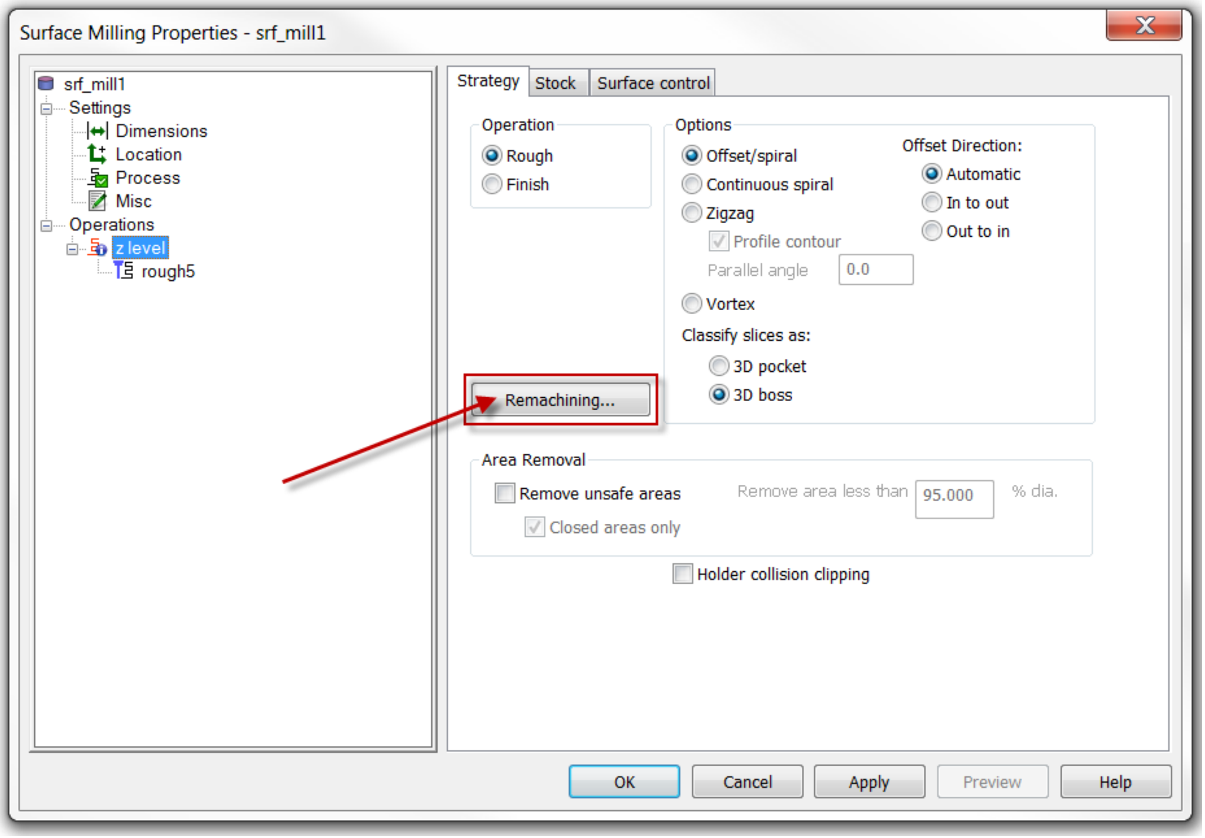
Note how the flat area clearance now only extends to the edge of each flat area. This eliminates unnecessary tool moves and will result in a shorter, more efficient tool path.

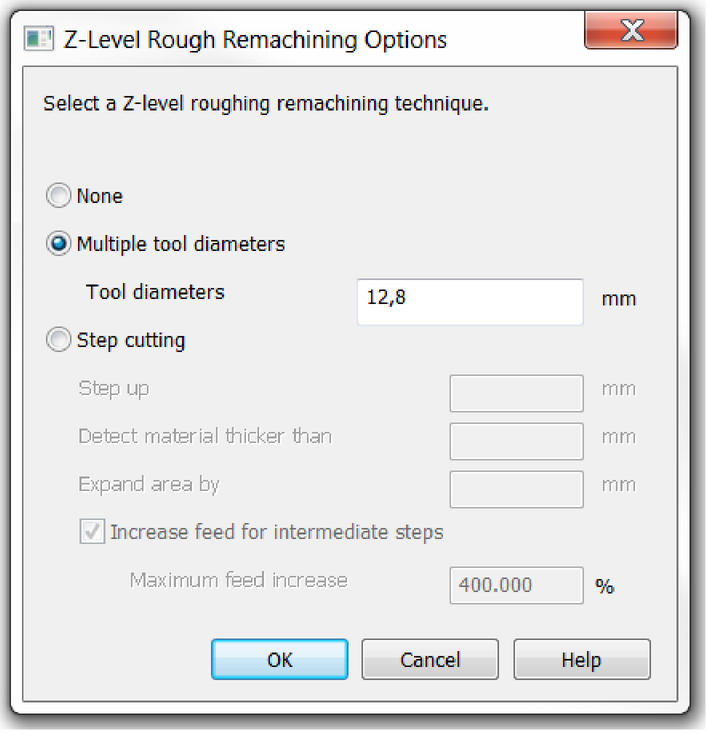
34 Select an **Isometric** view.

35 Run a **3D Simulation** to see how the flat levels are cleared only in the required areas.

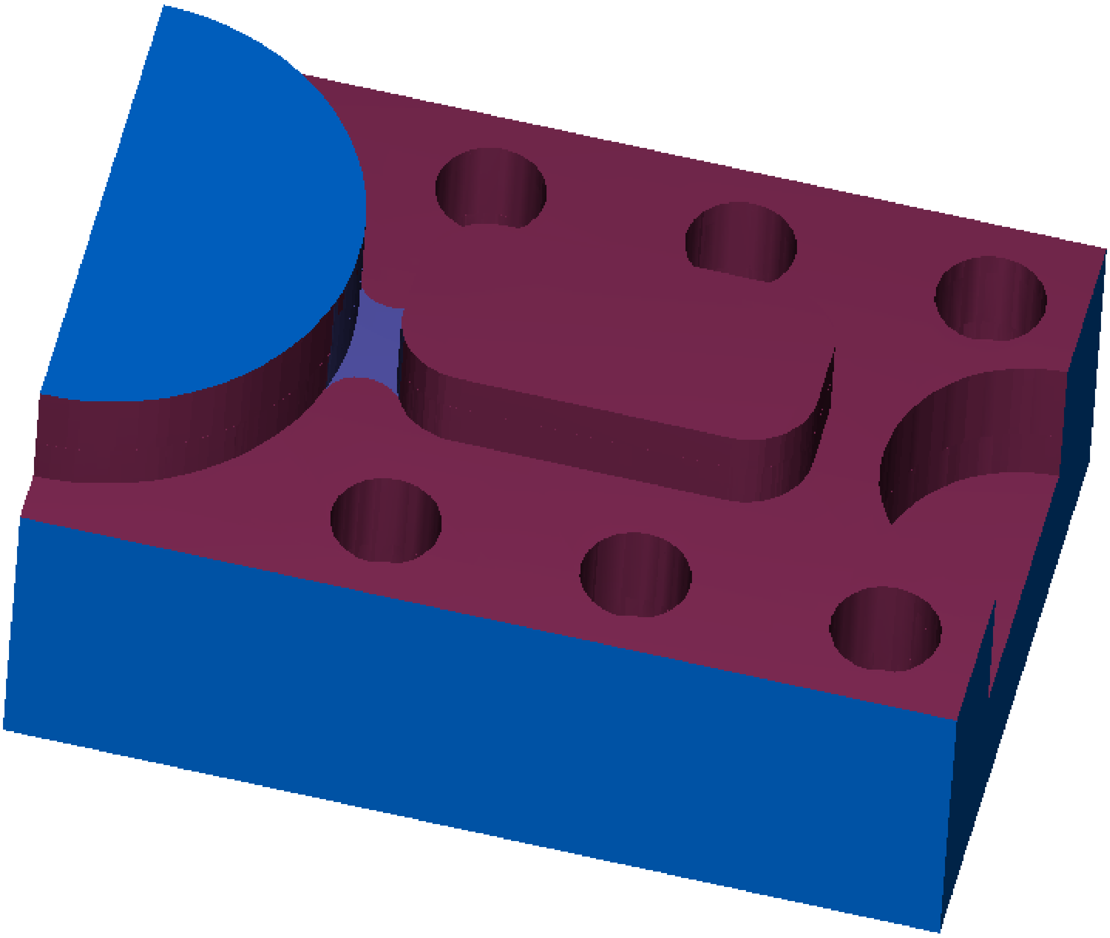
36 To add an extra tool to the process to remove material left by the 12mm cutter we can add a tool under the **Remachining** tab in **Z-Level**.

37 Select the Remachining tab and then select **Multiple tool diameters** and add **12,8** into the field as shown on the next page.





*Remember to change both cutters to **Area** under the milling tab and change the Z Increment to 12mm for the 8mm cutter. The following image shows the 8mm tool machining the area that was left by the 12mm cutter.*



Contents

High Speed and High Efficiency Roughing

Introduction

The aim of the roughing process is to remove as much material from the part in the shortest possible time. There are three main approaches to do this, High Efficiency roughing, High Speed roughing and Plunge roughing.

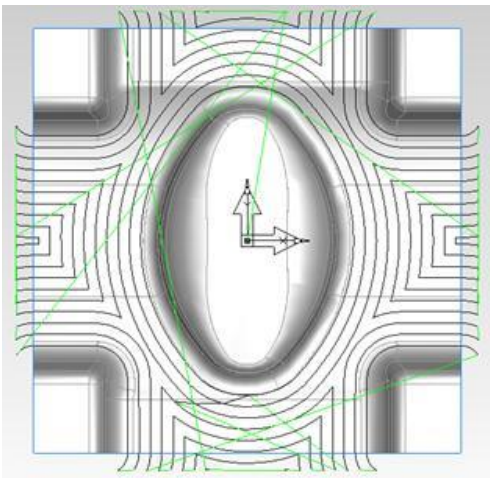
In high efficiency roughing special tooling is used to take very heavy cuts. This removes the maximum possible amount of material per pass. Due to the heavy cuts it is important to maintain a steady load on the tool. **FeatureCAM** uses the trochoidal milling method in areas where a tool overload condition would occur. Trochoidal milling moves the tool into the material in a series of loops whenever a potential overload is detected.

In high speed roughing a relatively light cut is taken but the cutting speed is made as fast as possible. While it is possible to output NC code with very high feed rates, in practice such a program would probably not run at the programmed speed on a machine tool. The controller slows down the feedrate each time a sharp change of direction is detected. The accelerations and decelerations involved can result in a program taking much longer to run than expected. To program a true high speed program it is important to minimize changes in direction and make the transitions between directions as smooth as possible. **FeatureCAM's** trochoidal milling and patented raceline machining processes are ideally suited to high speed roughing.

Plunge roughing roughs a part by a series of drilling moves using a large cutter. Specialized tooling is required. As the machine is only moving in Z it is more rigid and hence more aggressive feeds can be used. Plunge roughing is particularly suited to deep pockets. With conventional cutting the feed rate has to be reduced the longer the tool gets. With plunging the same feedrate applies all the way down the pocket. Plunge roughing is covered in a separate document.

High Speed Roughing

1. Import the file **Fast_Rough.x_t**.
2. Create Stock and Setup
3. Box select all the surfaces.
4. Create a **Z Level** roughing feature. Use **Offset/Spiral** in the **Strategy**.
5. Click **Finish**.
6. Click on **rough1** and select the **Milling** tab.
7. Change the **Rough pass stepover %** to **15%**. **Apply** and **OK**.
8. Run a **Centreline** simulation.
9. In the **Operations List,** click on the down arrow next to the roughing operation and select the second roughing level.
10. Select a **top** view.



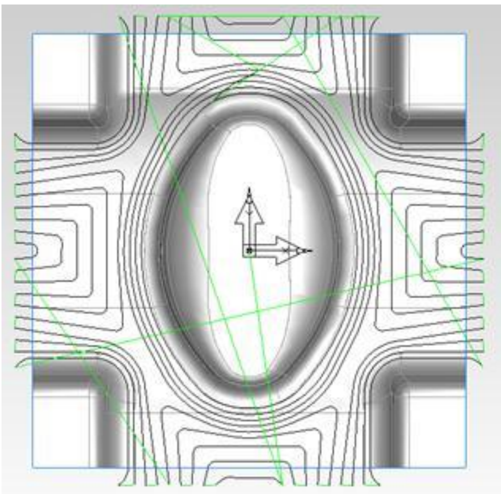
This has created a Z Level roughing operation with the default settings. Note the sudden changes in direction; this path is not suitable for high speed machining. We will now modify so that it can be run at high speed.

11 **Eject** the simulation

12 Select the **Milling** tab and change the **Toolpath corner %** to **50**.

13 Click **Apply**, then **Preview** and then **Play selected**

14 In the **operations list** click on the down arrow next to the roughing operation and select the second roughing level.



Note how the toolpath now has radii where it changes direction also the way that the paths are smoothed as they get further from the profile. This path has some cuts which engage a large part of the cutter diameter however as we are going to use a small depth of cut this is not important. Although the cutting portion of the toolpath now is much smoother the Stepovers still make sudden changes of direction. We will now change this.

15 **Eject** the simulation

16 Double click on the feature to open its properties.

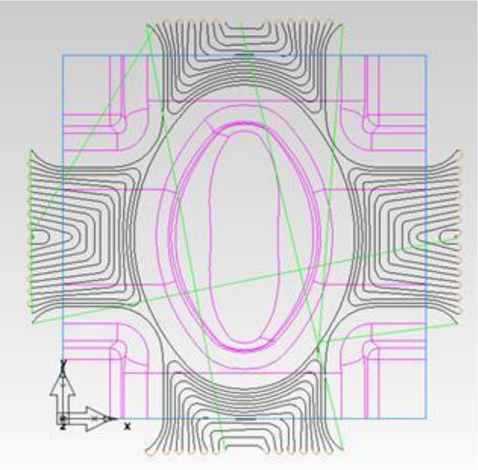
17 Select the **Leads** tab and change the **Stepover Type** to **Arc**.

18 Select the **Milling** Tab.

19 Set the **Z increment** to **1mm** and **Rough pass stepover** to **10%**.

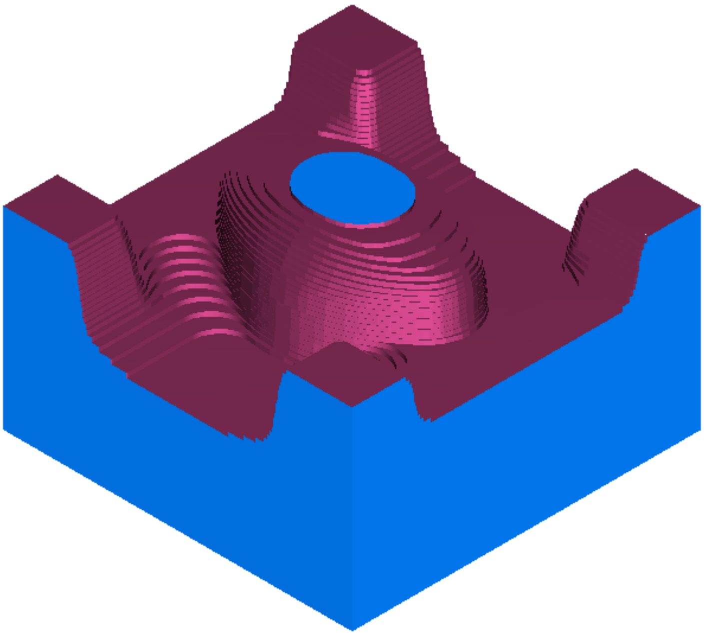
20 Run a **Centreline** simulation.

21 In the **operations list** click on the down arrow next to the roughing operation and select the roughing level at Z -2.040 or thereabouts.



Note: *the links now make a smooth arc between the toolpath segments.*

22 Select an **Isometric** view and run a 3D simulation.



The material is roughed out in a series of shallow steps with a small stepover. The smooth shape of the toolpath and the small amount of material being removed per pass means that the cutter can run much faster giving an overall high rate of metal removal.

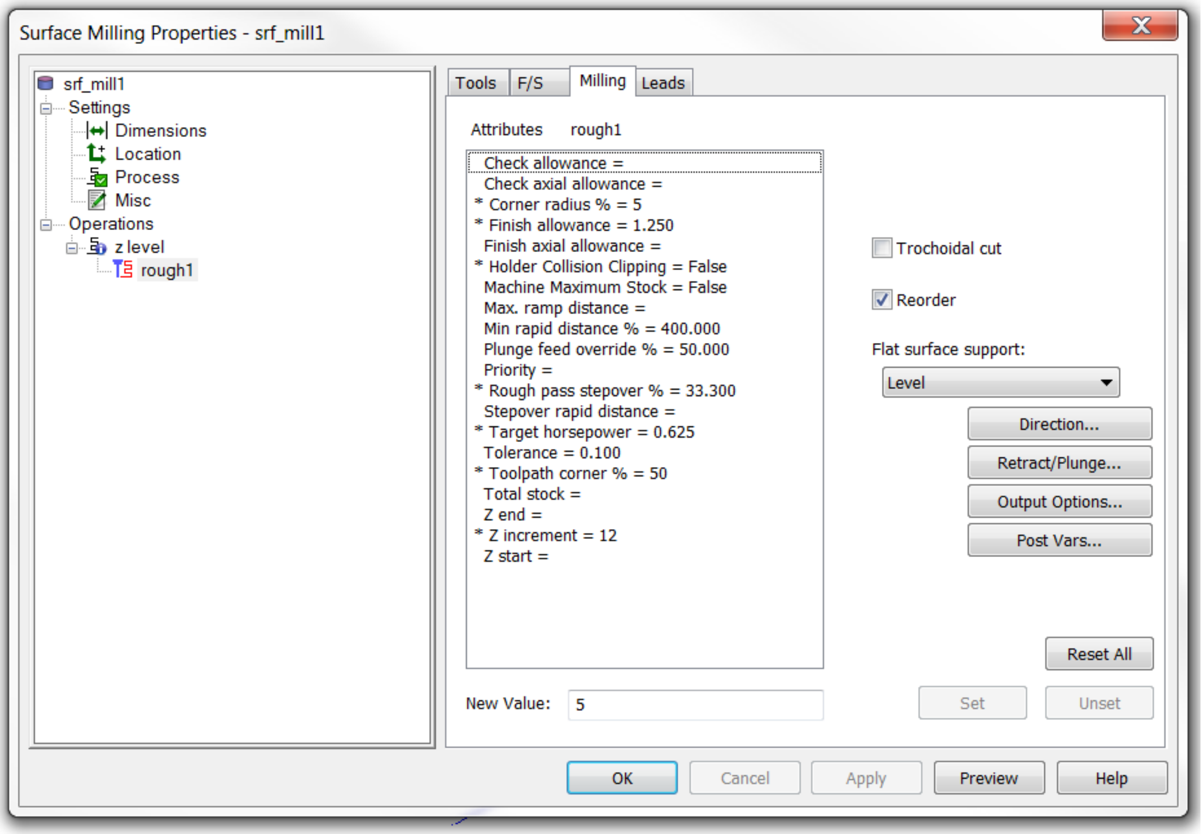
23 **Eject** the simulation

High Efficiency Roughing

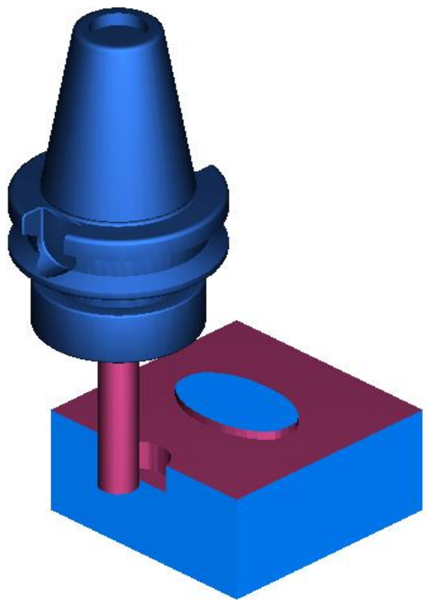
We will now take a look at high efficiency roughing where a large tool will be used with a deep cut and small stepover to increase the metal removal rate.

24 Double click on the feature to open its properties.

25 Select the **milling** tab and fill in **exactly** as below.



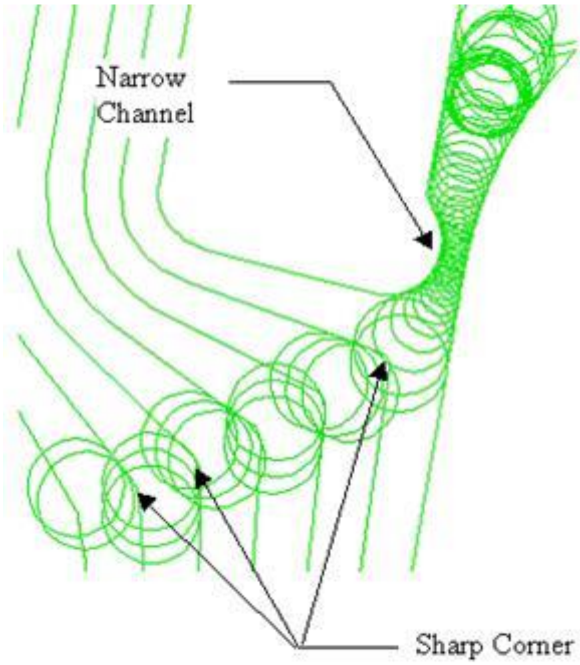
26 **Single step** a 3d simulation.



The initial moves into the material are taking a very heavy full width cut. This overload situation may cause damage to the tool and would require a reduced feedrate. We will now use Trochoidal roughing to reduce the tool loading.

Limiting tool overload

When roughing a part which has narrow slots, when entering a sharp corner or on the first pass in a pocket it is desirable to avoid overloading the tool. It is possible to do this by using trochoidal machining. This strategy detects when tool overload is likely and changes from normal machining to a toolpath which enters the overload area in a series of loops as shown below. This greatly reduces the tool loading and allows the toolpath to be run at a higher feedrate without damaging the tool.

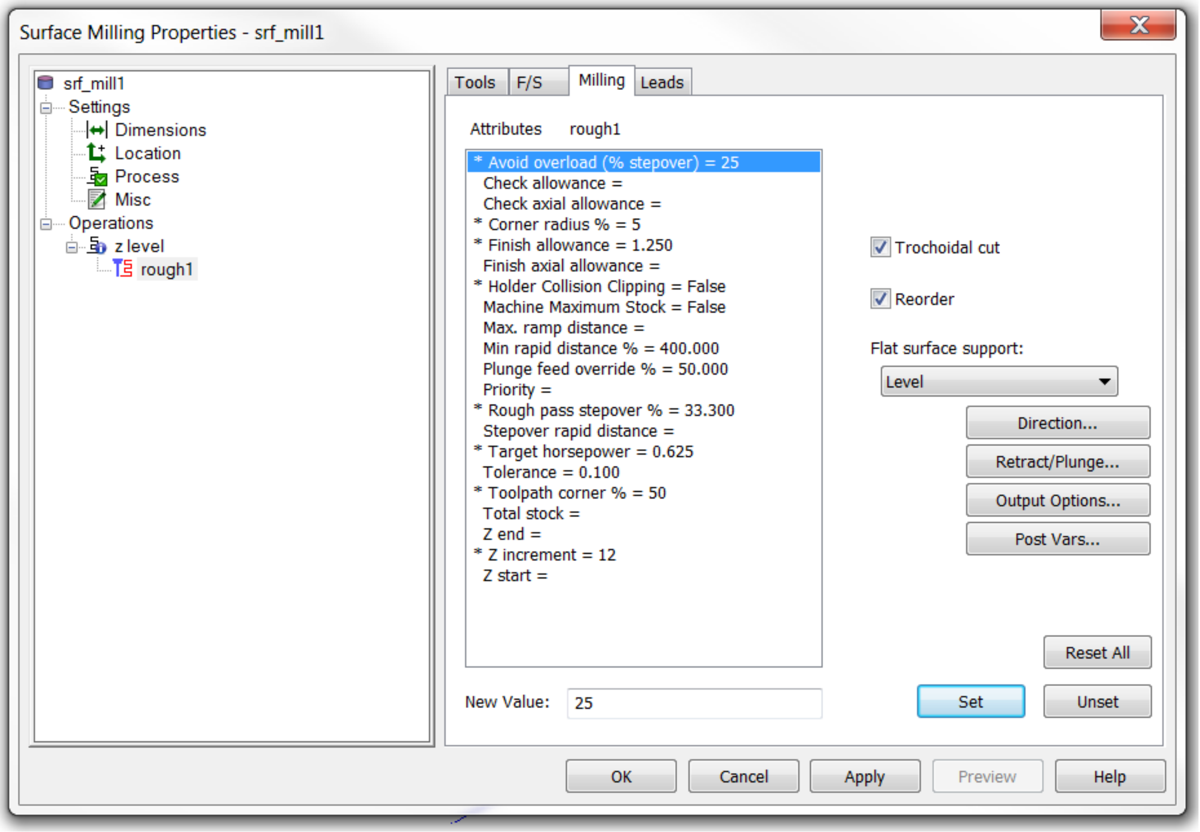


27 **Eject** the 3d simulation.

28 Edit the feature and select the **Milling** tab.

29 Check **Trochoidal cut** and set **Avoid overload** to **25%**.

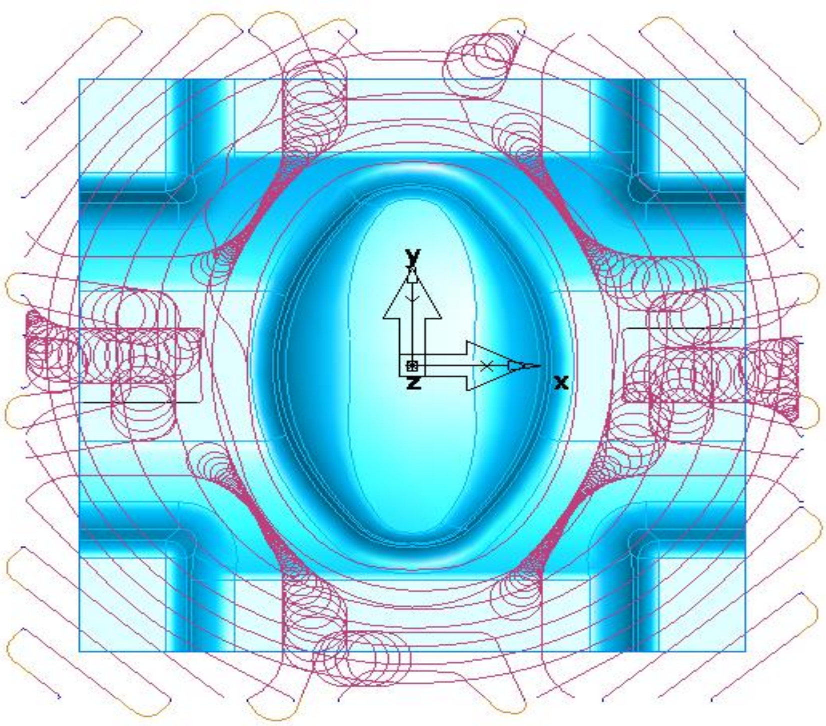
This is telling FeatureCAM that when the tool engagement reaches 25% of the tool diameter the cutting strategy should be switched to Trochoidal.



30 Click **Apply** and then **OK**.

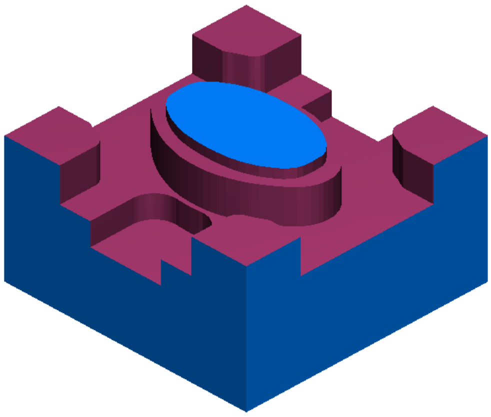
31 Run a **Centreline** simulation.

32 Take a **Top** view.



It can be seen that the toolpath has changed so that the tool overload situation is avoided by changing to Trochoidal moves. The tool loading is significantly reduced but the overall toolpath length is increased, this will mean that the roughing process will take longer to complete. This can be fixed by using ***FeatureCAM's*** *Feedrate Optimization capability. We will see how to do this at the end of this module.*

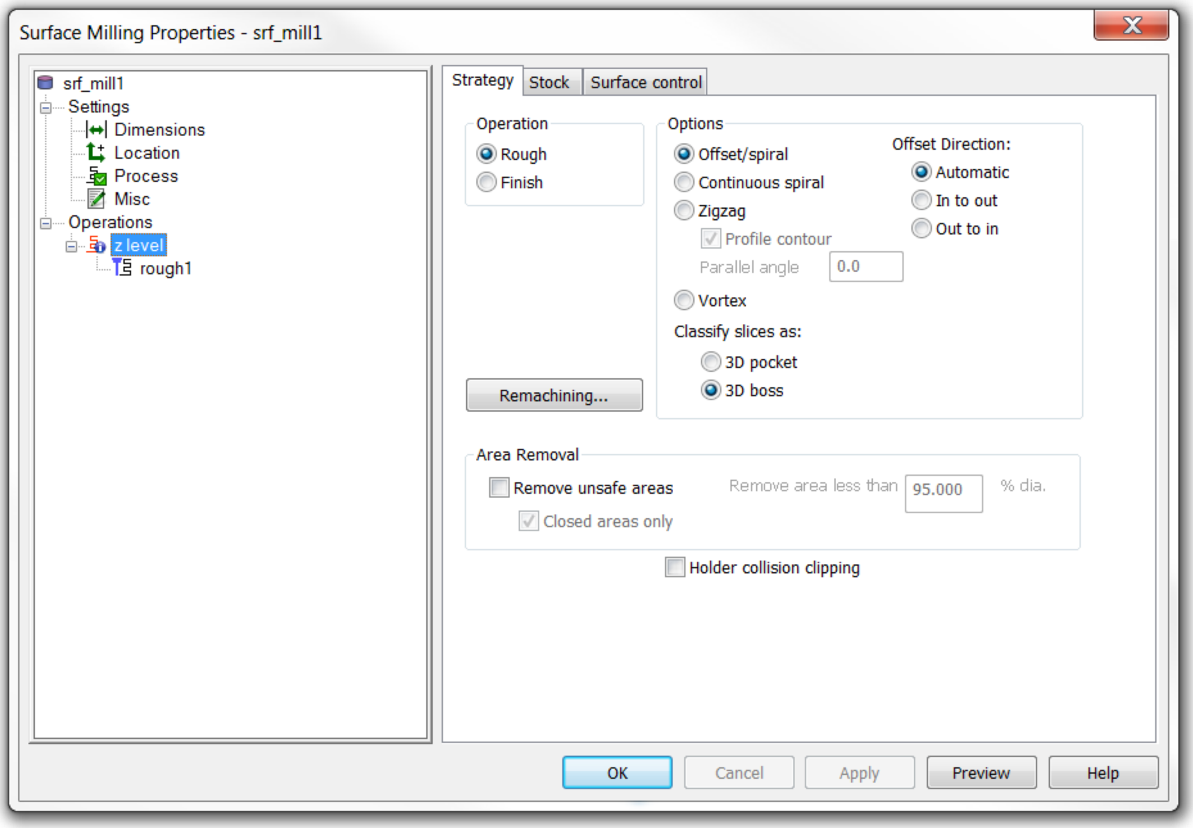
33 Run a 3D Simulation.

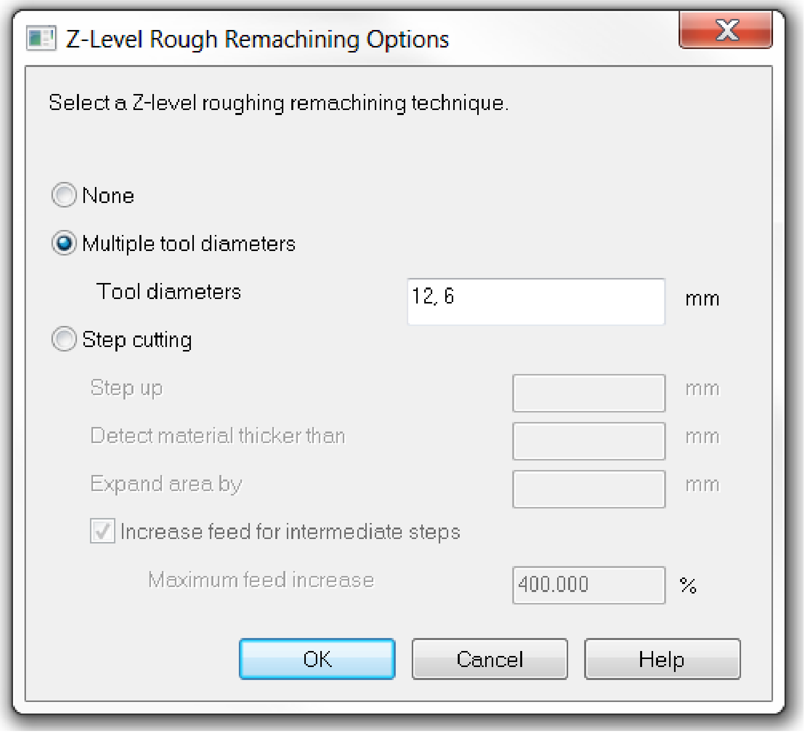


The bulk of the material is removed quickly but the remaining stock has large steps. Due to the large tool diameter a lot of material is left on the inside corners. A second roughing pass will be added to reduce the steps and remove more material from the corners.

34 Double click on the feature to open its properties.

35 Select **Z level** and fill in the form as below.

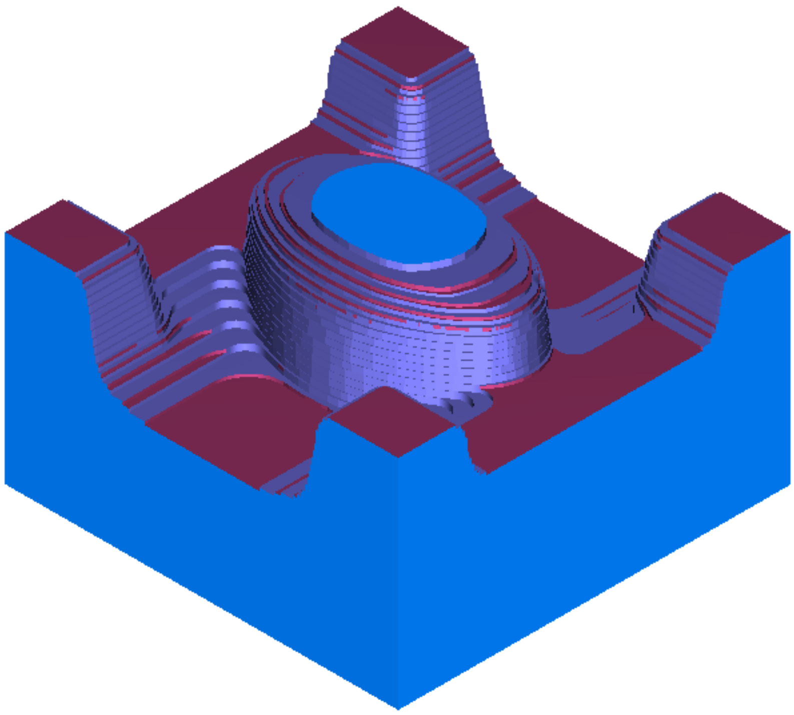




36 Click **Apply**.

37 Select the second roughing operation and set its **Z increment** to **1.25mm.**

38 Run a **3D Simulation**.



The material is quickly roughed out by the large tool; the steps and excess inside corner material are then reduced by the smaller tool in the second roughing operation.

Feedrate Optimization

*In the previous example trochoidal roughing was used to reduce the loading on the tool when first engaging the part and in confined areas. While this technique reduces the tool loading it also increases the overall length of the toolpath. Fortunately **FeatureCAM** has the ability to vary the feedrate on each block of NC code within a toolpath. The rate can be increased or decreased depending upon the actual amount of material being removed by the tool and a target horsepower which the user sets for each operation.*

39 Run the simulation and then click on the **NC Code tab**.

Just two machining feed rates are being used within this, program, the plunging feedrate and the cutting feedrate.
*We will now instruct **FeatureCAM** to calculate the feedrate such that it maintains an optimum loading on the tool, where the cut is lighter the tool will move faster, where the cut is heavy it will slow down.*

40 Edit the feature and select the **Milling** tab for the first roughing operation.

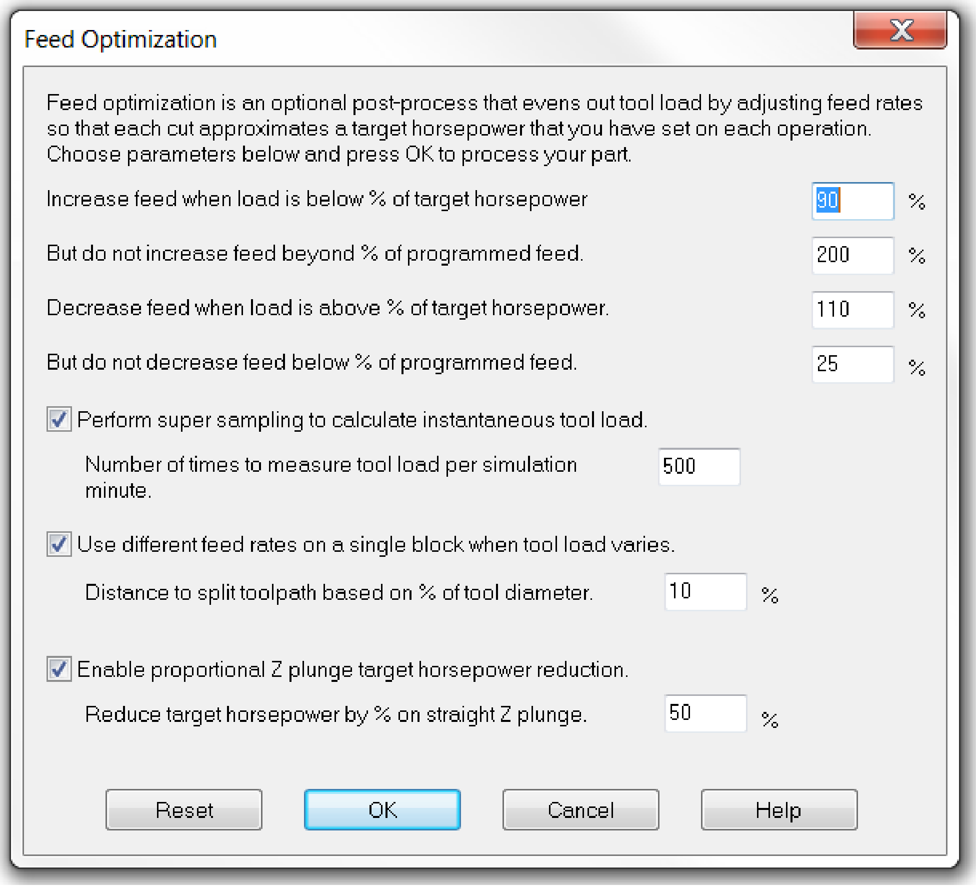
41 Set the **Target horsepower** to **0.2 HP**.

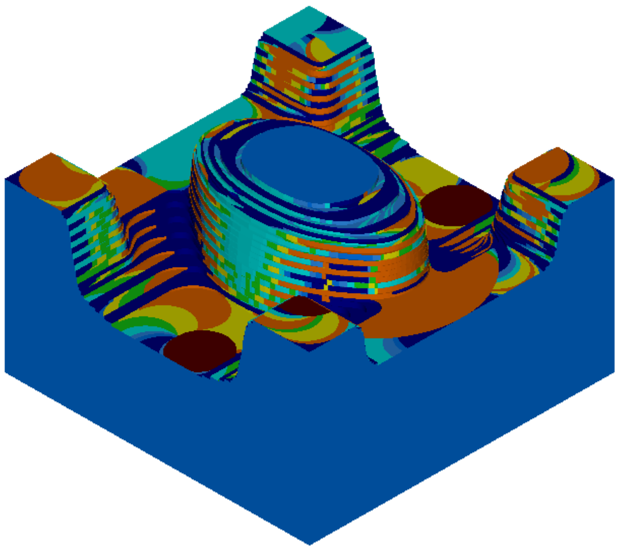
42 Click **Apply** and **OK**.

*Given the required tool loading (0.2 HP) **FeatureCAM** can then look at the spindle speed, number of flutes on the tool and the material properties and calculate the required feedrate to maintain the load on the tool.*

43 From the Manufacturing menu select Feed optimization.

Fill in the form as shown on the next page.





- The first item is telling **FeatureCAM** that if the load is less than 90% of that specified (0.2 Hp) then the feedrate should be increased until the calculated load is as required.
- The second item limits the feedrate so that if the cut is extremely light the feedrate does not go more than four times the originally programmed rate.

- The third item is limiting the overload on the tool by setting an upper limit for the horsepower, in this case 110% or 0.22HP.
  The fourth item sets a lower limit on the feedrate so that the tool does not move at an unreasonably slow speed when encountering heavy cutting conditions.

  The other items are fairly self-explanatory, for further information please see the help file.

44 Click **OK**.

- **FeatureCAM** now recalculates the toolpath. For each move it calculates the feedrate required to maintain the spindle loading within the limits specified above. The resulting NC code is shown below
- It can be seen that the feedrate is now varying. As the Trochoidal toolpath is reducing the loading the feed rates are being increased. This will result in the toolpath running in a shorter time on the machine.
  The more constant loading will also reduce rubbing in areas where the cut is light and prevent overloading in heavy cut areas. This will increase the tool life and give a better surface finish.

Contents

Corner & Pencil Finishing

Introduction

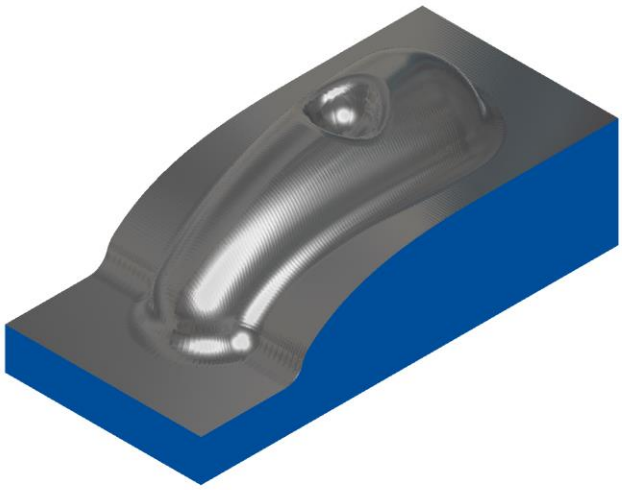
Corner Machining includes 5 different types of strategy: Pencil, Multi Pencil, Along, Across and Combo Along and across. Pencil milling creates "single pass" machining along sharp internal corners while corner machining performs local re-machining of areas inaccessible to a larger tool.

1 Open the file **Cowling.FM**

2 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.

3 Run a **3D Rapid Cut** simulation



The part has been roughed out and then finished using a 20mm 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 3mm 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.

4 **Eject** the simulation

5 Uncheck the operation **srf_mill1** in the part view

6 Select all of the part surfaces

7 Create a new **Surface milling** operation

8 Click **Choose a single operation** and click **Next**

9 Select **Pencil** strategy and click **Finish**

10 Edit the feature

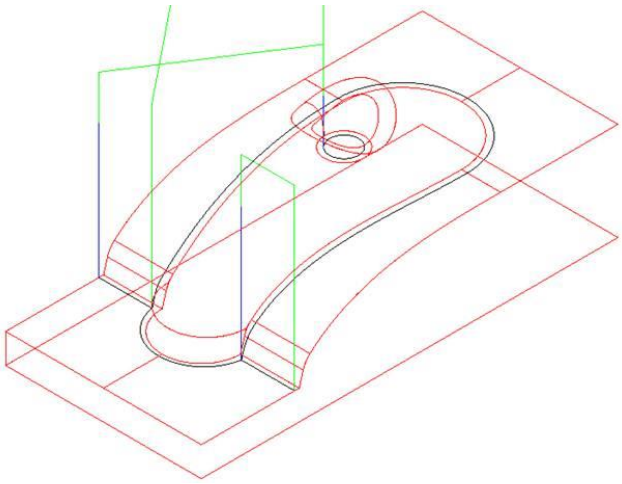
11 Click on **Finish1** in the tree view

12 Select the **Tools tab** and pick a **6mm ball endmill**

13 Click **Apply** and then **OK**

14 **Unshade** the part

15 Run a **Centreline** simulation



The tool is making a single pass around the entire 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.

16 **Eject** the simulation

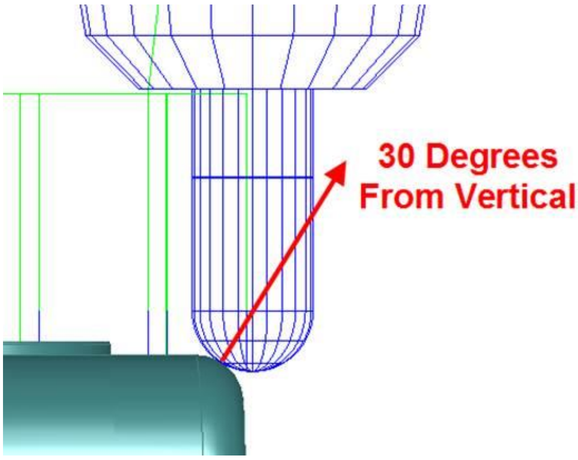
17 Edit the feature

18 Click on **Pencil** in the tree view and click on the **Slopes** tab

19 On the **Slopes** tab, Click **Horizontal** only

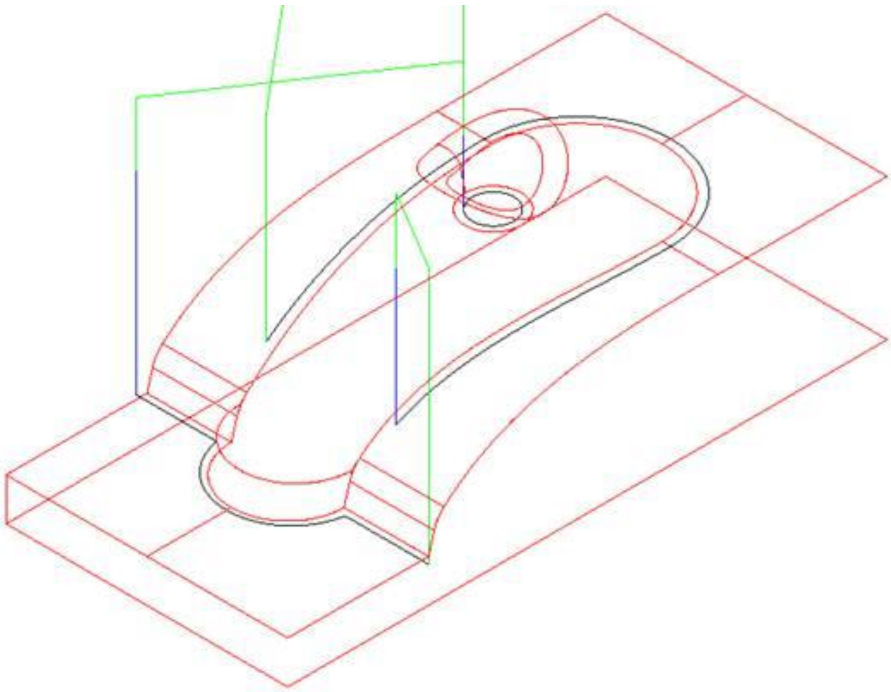
20 Set the **Maximum surface slope** to **30**

21 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.

22 Run a **Centreline** simulation



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

23 **Eject** the simulation

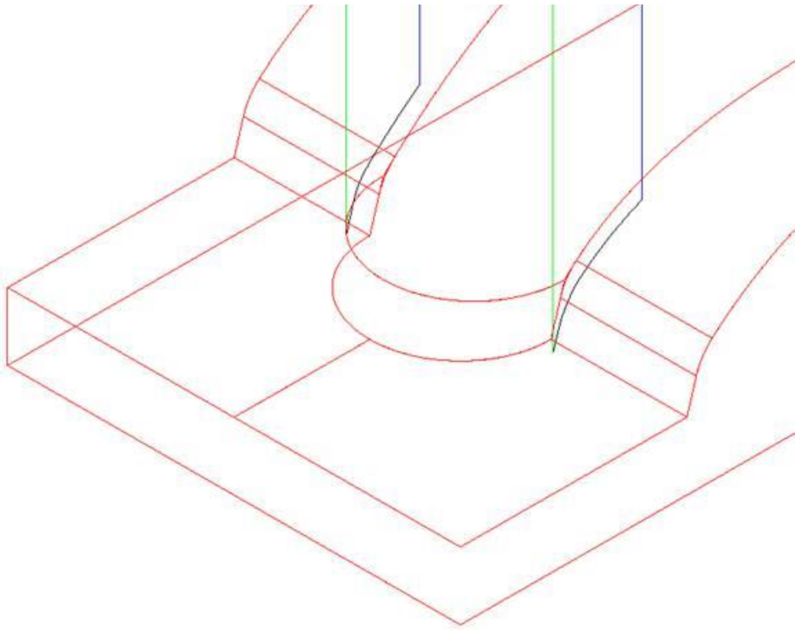
24 Edit the feature

25 Click on **Pencil** in the tree view and click on the **Slopes** tab

26 Click **Vertical only** and then set the **Minimum surface slope** to **30 degrees.**

27 Click **Apply** and then **OK**

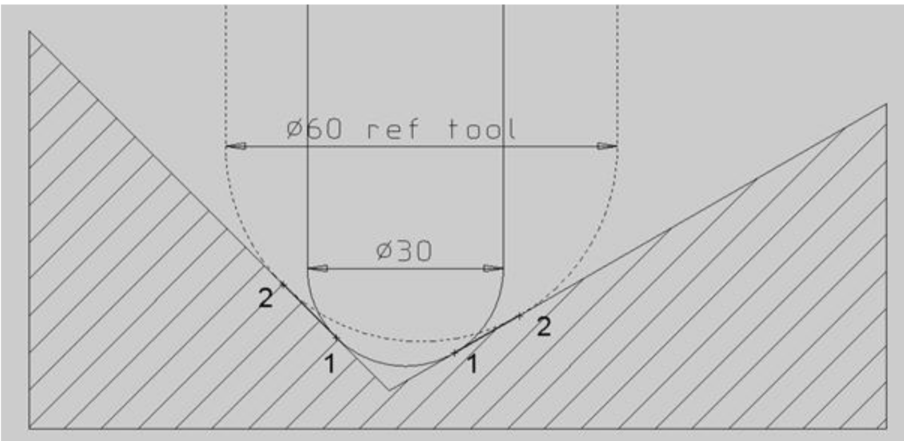
28 Run a **Centreline** simulation



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

Corner 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.



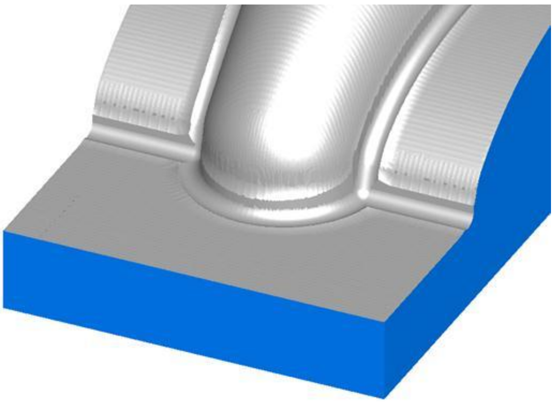
29 Check the surface milling feature named **srf_mill1** in the Part View

30 Edit the **pencil milling** feature

31 On the **Slopes** tab set **Slope limitations** to **None**

32 Click **Apply** and then **OK**

33 Run a **3D Rapid Cut** simulation



You can clearly see in the inside corners that there is material remaining between the areas finished by the 20mm cutter and the 6mm tool used in the pencil operation.

Eject the simulation

34 Uncheck **srf_mill1** and **srf_mill2** in the Parts view

35 Select all of the part surfaces

36 Create a new **Surface milling** operation

37 Click **Choose a single operation** and click **Next**

38 On the New strategy tab select **Corner Remachining**

39 Click **Next**

40 Select **Along** and set the **Previous tool diameter** to **20mm**

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.

41 Click **Finish**

42 Edit the feature**.** Click on **Remachining** in the tree view

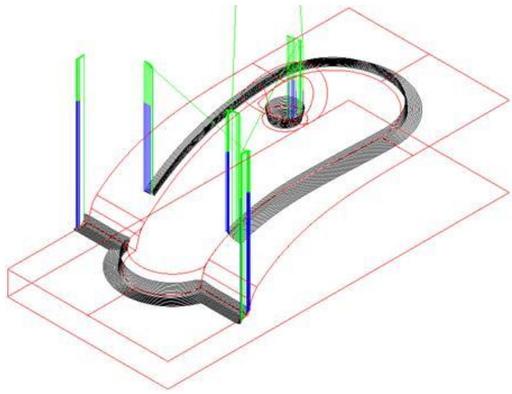
43 On the **Slopes** tab, Click **Horizontal only** and set the **Maximum surface slope** to **30**

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

45 Click on the **Tools** tab and select a **6mm ball endmill**

46 Click **Apply** and then **OK**

47 Run a **Centreline** simulation

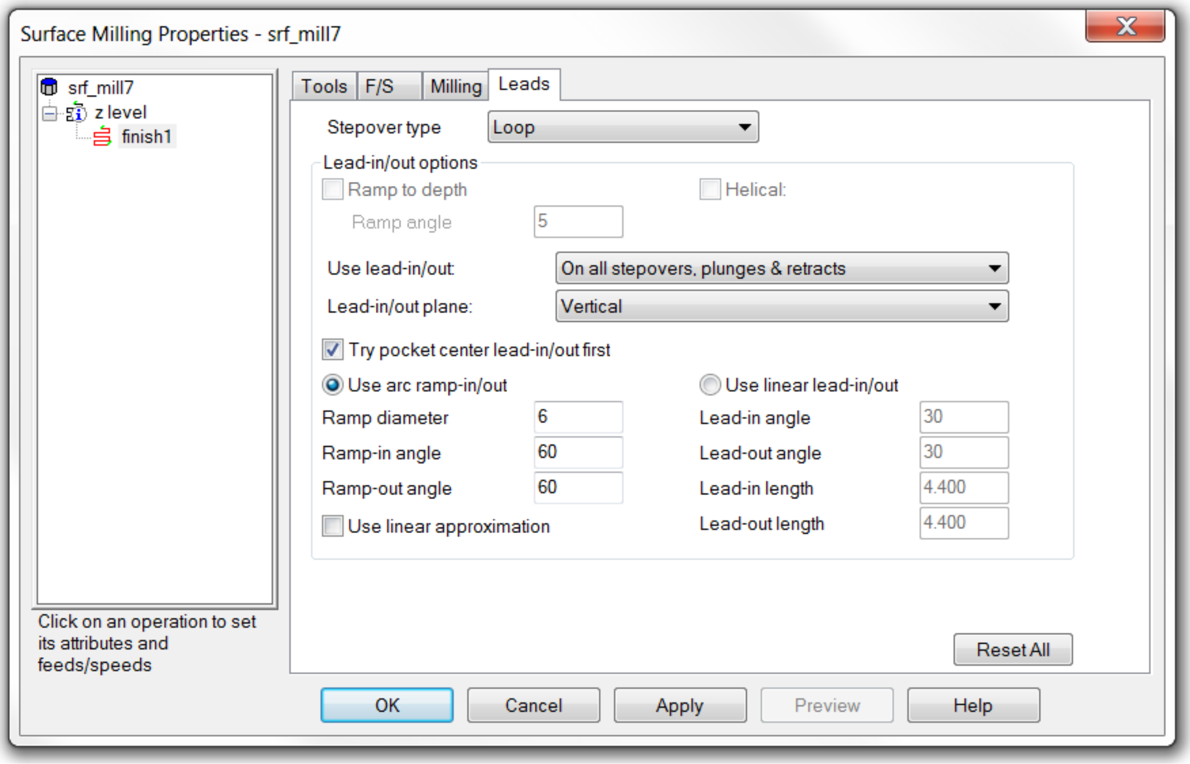


The shallow areas are being cleared of excess material from the outside inwards in a climb milling direction. There are multiple lift offs resulting in a lot of "air time". The tool is also plunging directly into the remaining material. We can remedy both of these problems by modifying the leads and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

48 **Eject** the simulation and Edit the feature

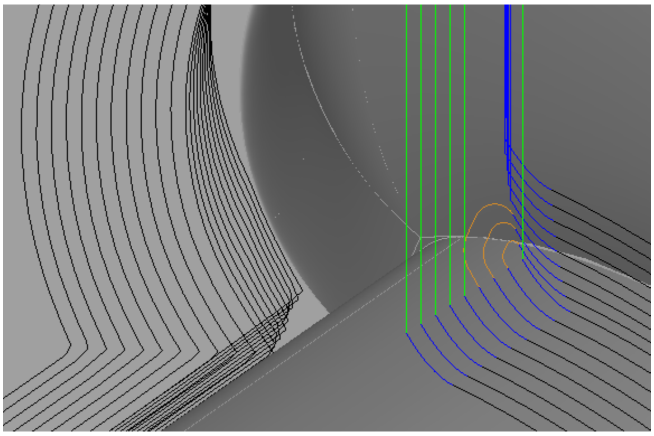
49 Click on **Finish1** in the tree view and click on the **Leads** tab

50 Fill in the form exactly as shown



51 Click **Apply** and then **OK**

52 Run a **Centreline** 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.

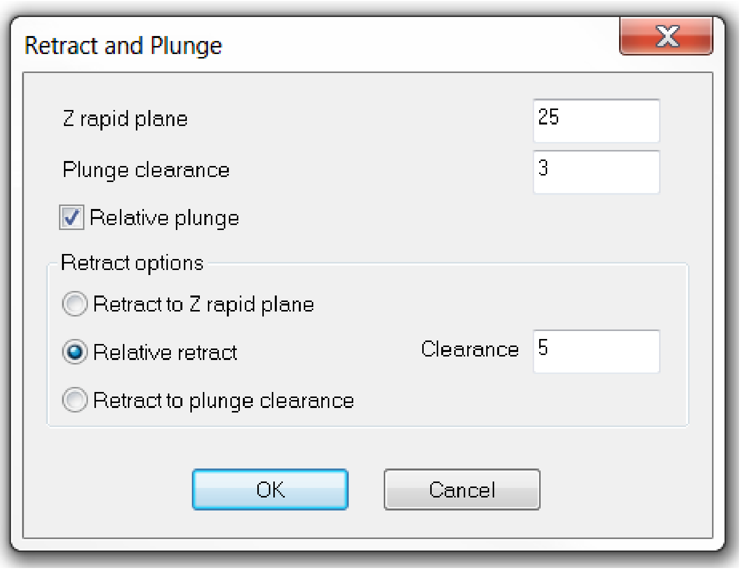
53 **Eject** the simulation and Edit the feature

54 Click on **Finish1** in the tree view and click on the **Milling** tab

55 Note: The surface finish is controlled by the scallop height value

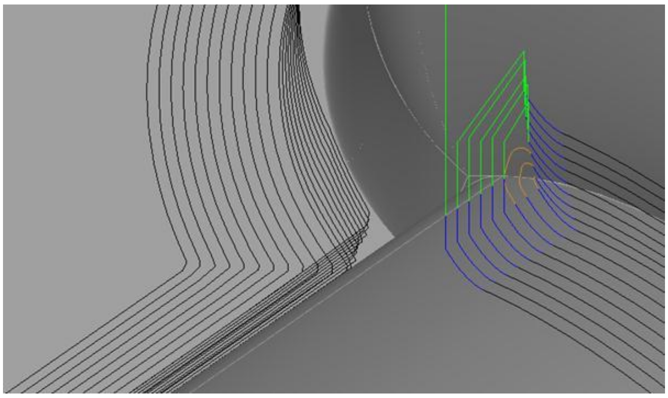
56 Click on **Retract/Plunge**

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



58 Click **Apply** and then **OK**

59 Run a **Centreline** simulation

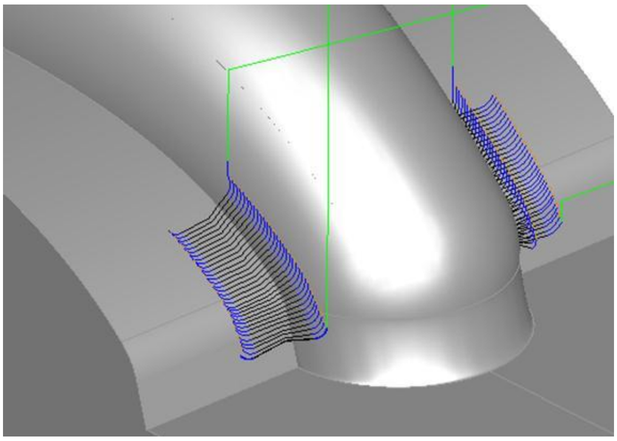


*Relative plunge tells **FeatureCAM** that you wish to plunge at rapid down to the specified Plunge clearance above the next point to be cut rather than plunging at a reduced feedrate all the way from the Z rapid plane. This reduces the length of time spent on slow plunge moves.*
*Relative retract instruct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

1. **Eject** the simulation
2. Edit the **srf_mill3** feature
3. Click on **Remachining** in the part view and select **Across** as the Option
4. Click on **Slopes** tab and Select **Vertical only** and enter a **Minimum surface slope** of **30**
5. On the **Stock** tab change the **Overcut %** to **Zero**
6. Click **Apply** and then **OK**
7. Run a **Centreline** 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.*

8 **Eject** the simulation

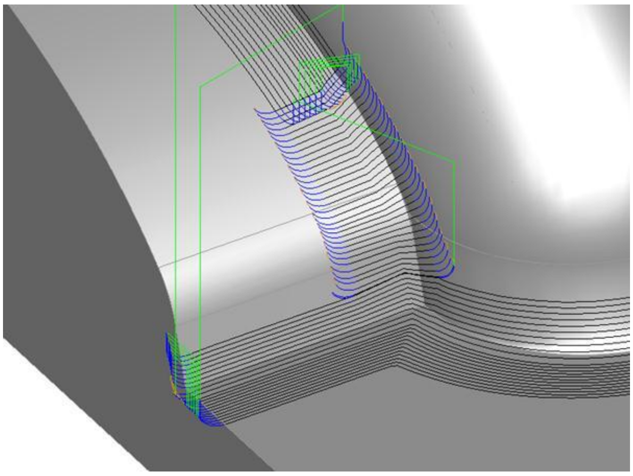
9 Edit the **srf_mill2** feature

10 Click on **Remachining**

11 On the **Strategy** tab select **Combo Along and Across**

12 On the **Slopes** tab, Click **None**

13 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.

14 Turn on the features **srf_mill1** and **srf_mill2** in the part view

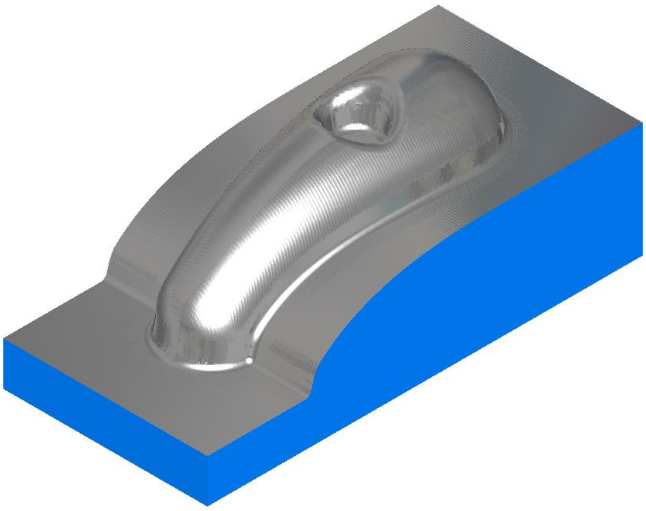
15 Edit the feature **srf_mill2**

16 On the **Slopes** tab select **None**

17 Click **Apply** and then **OK**

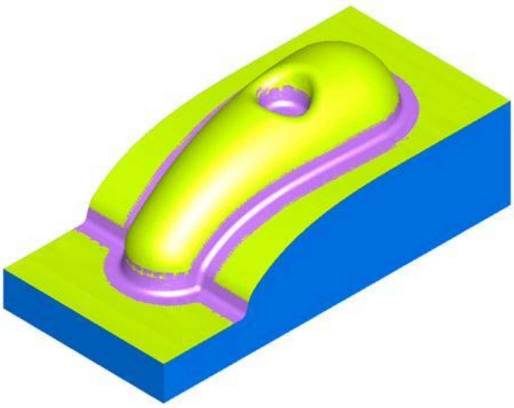
18 Reorder the toolpaths in the operation view so the pencil is last

19 Run a **3D** simulation



20 Turn on **Tool colours** in the **Simulations** options - **General** tab

21 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.

22 **Eject** the simulation

23 Uncheck **srf_mill1** and **srf_mill2** in the Part view

24 Edit the **srf_mill3** feature

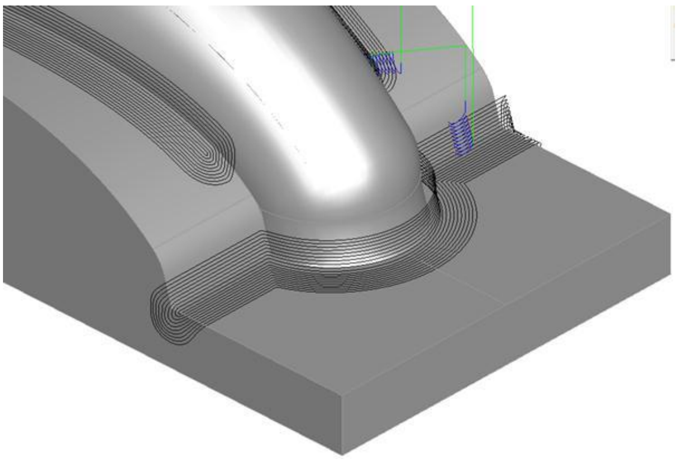
25 Click on **Remachining** in the part view and select **Multi pencil** as the Option

26 On the **Slopes** tab, Click **Horizontal only** and set the **Maximum surface slope** to **30**

27 On the **Stock** tab set the **Overcut % to 100**

28 Click **Apply** and then **OK**

29 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.

Contents

4-Axis Rotary Finish Machining

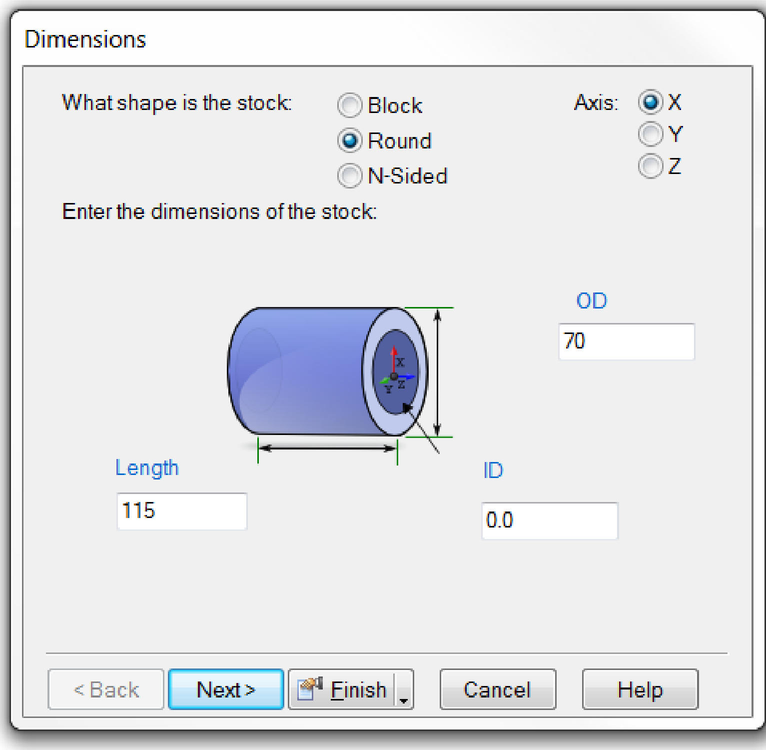
Overview

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

1 Open a new Metric milling document.

2 Fill in the stock Dimensions form as shown below

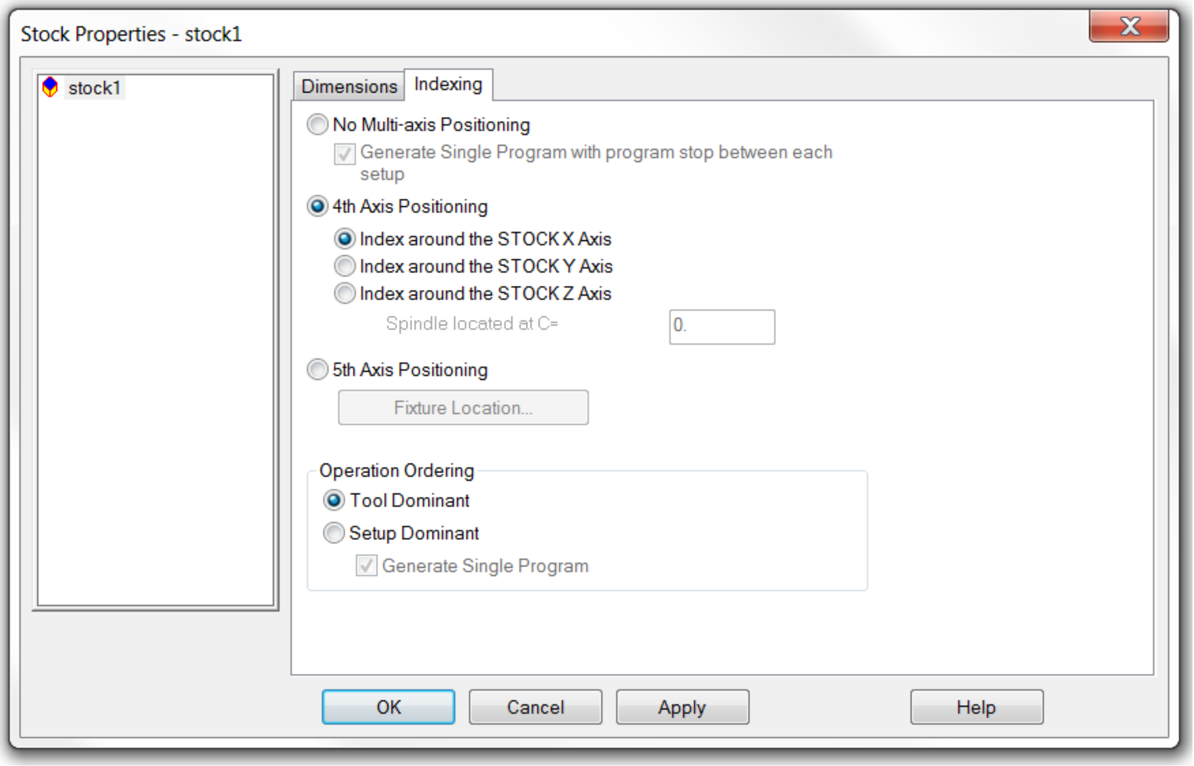


3 Press **Finish**

4 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.

5 Fill in the form as shown below



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

6 Press **Apply** and then **OK**

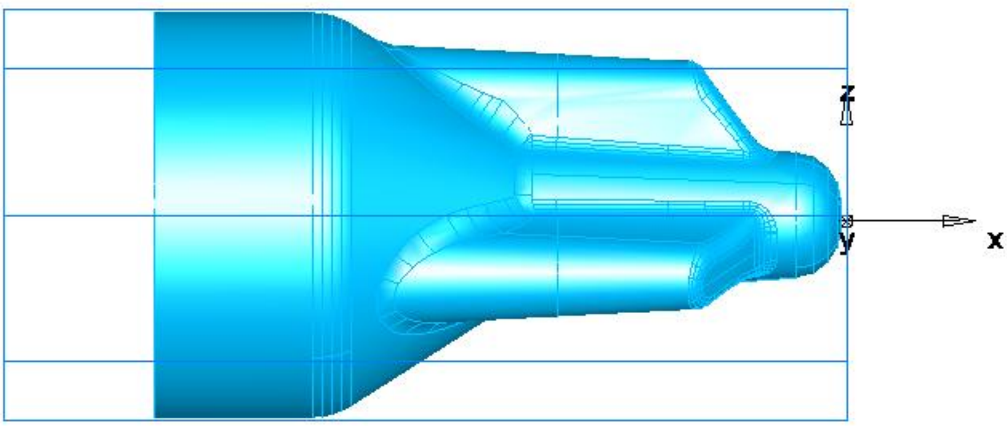
7 Import the model **Spindle.x_t**

8 Use the wizard to set the stock. Round and in the x axis.

9 Use **Compute stock from the size of the part**. Allow **1mm for front**, **25mm back** and **1mm OD**.

10 Have the Setup1 at the front. Select the right hand icon then select -1 Press **Finish**

11 Select the **Front** view. It should look like the image shown on the next page.



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.

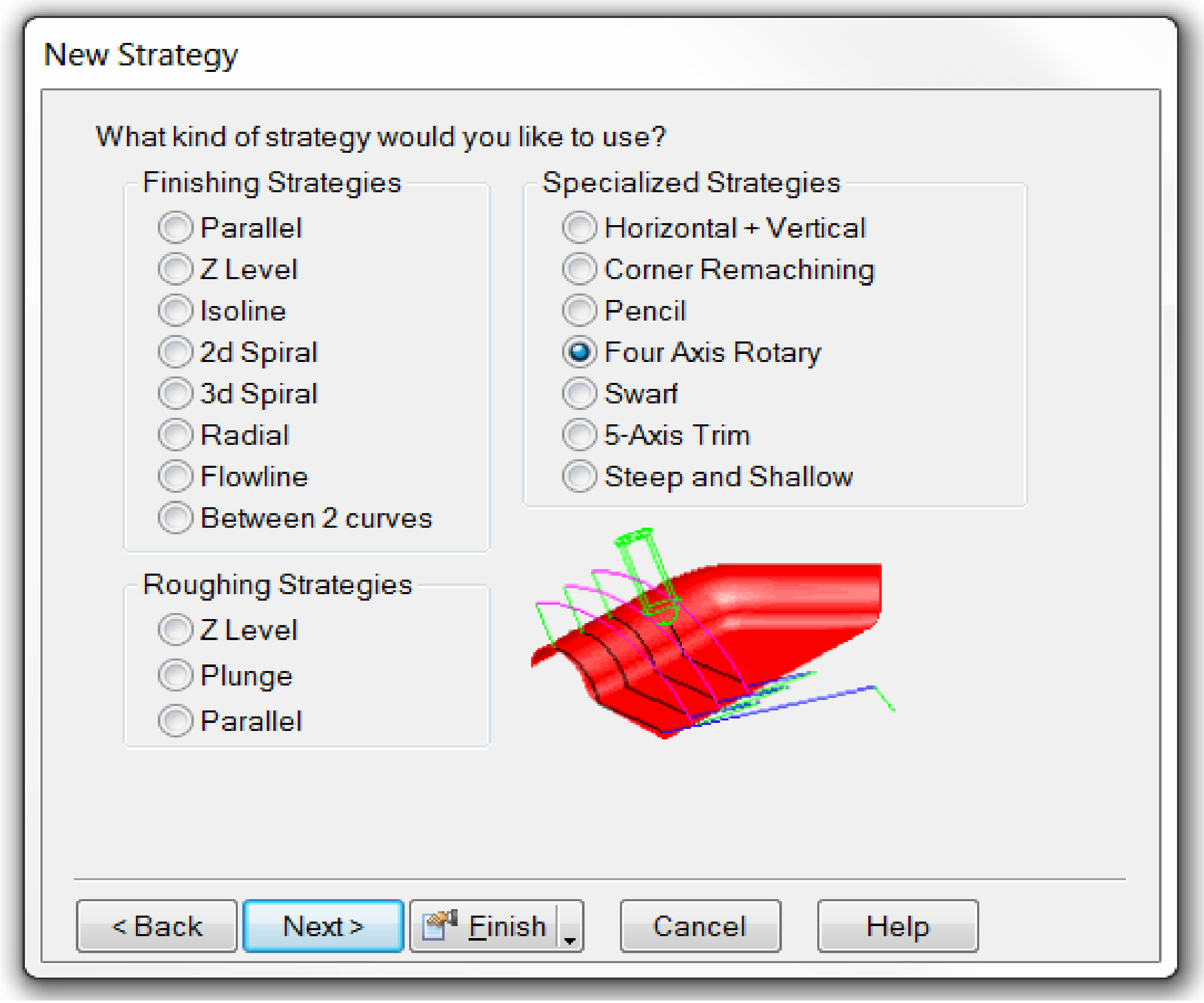
12 Select all of the surfaces

13 Create a new **surface milling** feature and **Next**

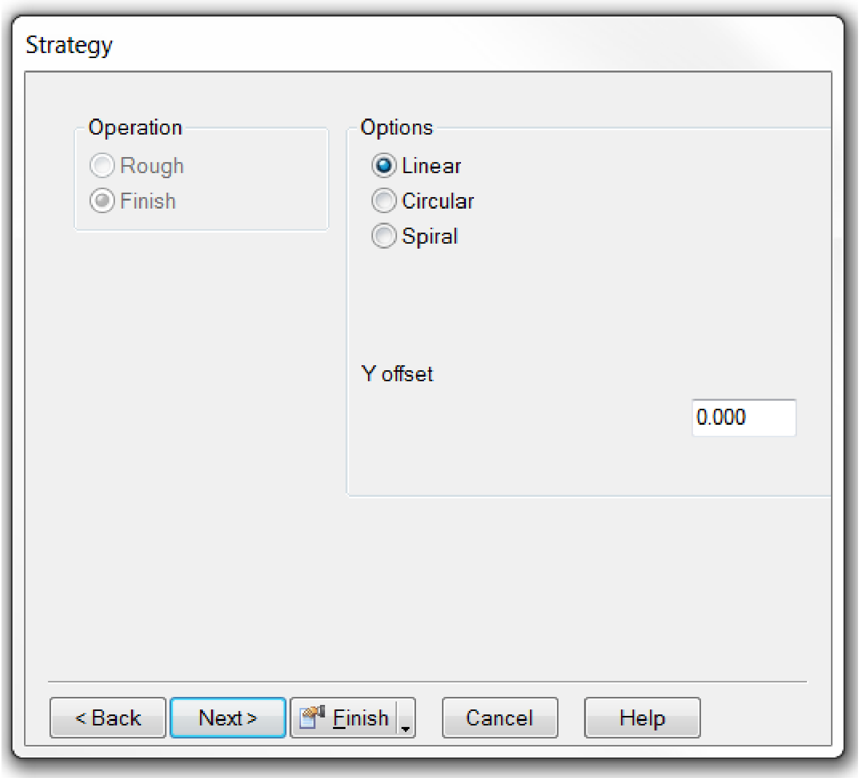
14 Check **Choose a single operation** and press **Next**

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

15 Check **Four Axis Rotary** and press **Next**



16 Fill in the form as shown on the next page.



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 centre cutting.

17 Press **Finish**

18 Press **OK**

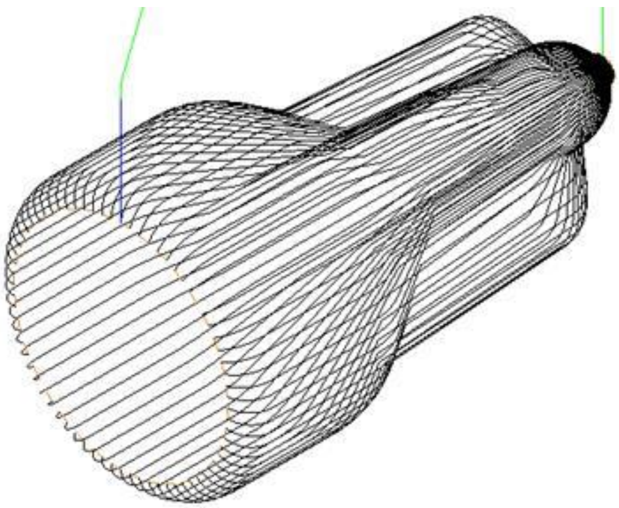
19 From the main menu select **Manufacturing** then **Post Process**

20 Set the **Tool change location Z** to **300mm**

21 Click on the Browse button and select the **HaasVF-4axis post**

22 Press **OK**

23 Run a **Centreline** 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.

24 **Eject** the simulation

25 Edit the feature

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

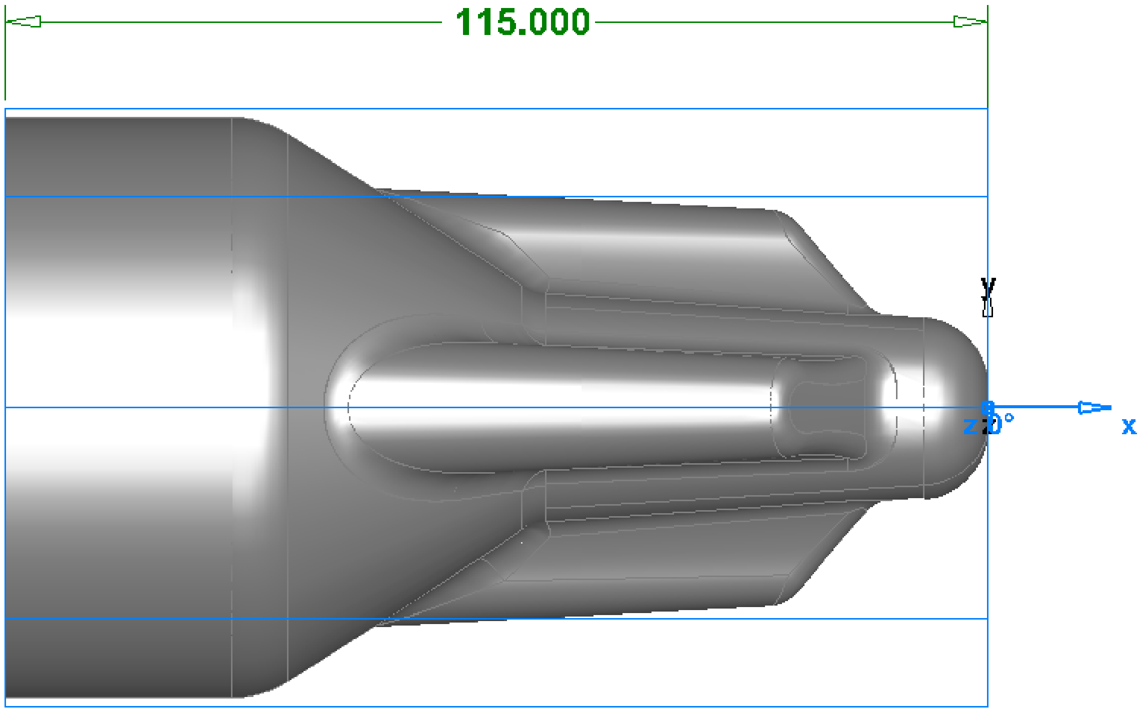
27 Select the **Milling** tab

28 Click **on Index start coordinate** and set the value to **0**

29 Click **on Index end coordinate** and set the value to **115mm**

30 Click **Apply** and then **OK**

31 Run a **centreline** simulation



The toolpath is now limited to the portion of the feature between X 0.0 and X-115mm. It is important to remember that the X values given are from the setup origin to the centre 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.

32 Edit the feature

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

34 Select the **Milling** tab

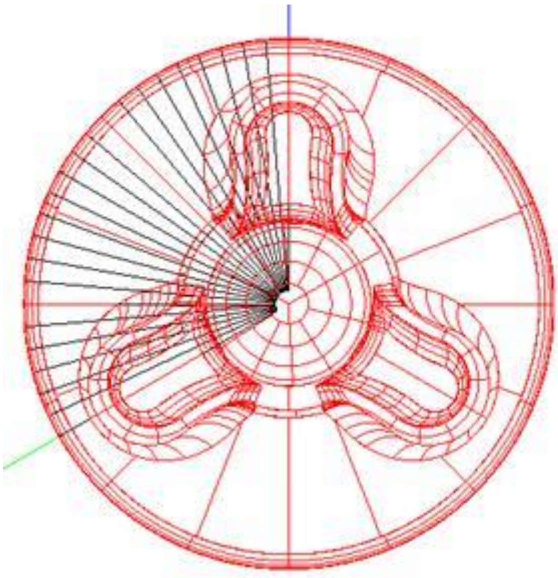
35 Click **on Angle start** and set the value to **0**

36 Click **on Angle end** and set the value to **120**

37 Click **Apply** and then **OK**

38 Run a **centreline** simulation

39 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.

40 Edit the feature

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

42 Select the **Milling** tab

43 Click **on Angle start** and then **Unset**

44 Click **on Angle end** and then **Unset** and **Apply**

45 Click on **four axis** in the tree view

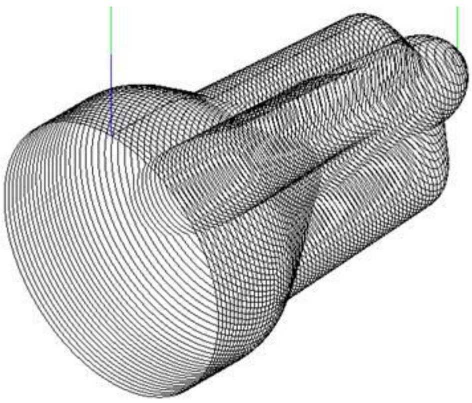
46 Check the **Circular** option on the **Strategy** tab

47 Click **Apply** and then **OK**

48 Select an **Isometric** view

49 Move the **Simulation speed slider** to the **left** to slow down the simulation

50 Run a **centreline** 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.

51 Edit the feature

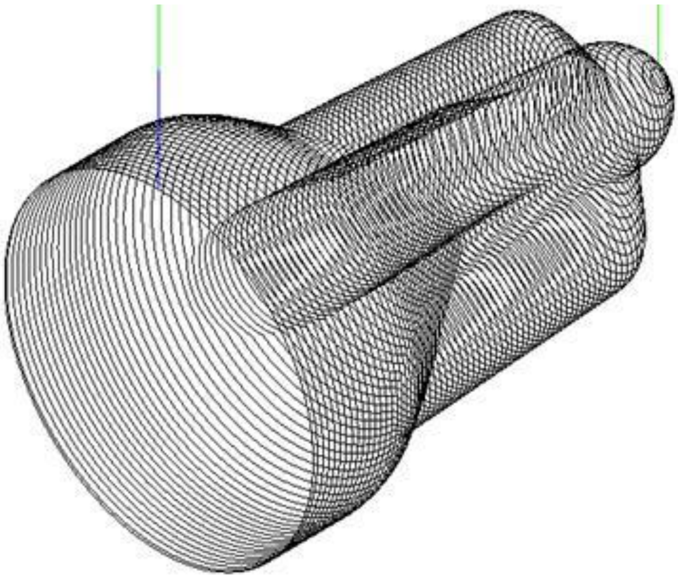
52 Click on **four axis** in the tree view

53 Check the **Spiral** option on the **Strategy** tab

54 Click **Apply** and then **OK**

55 Select an **Isometric** view

56 Run a **centreline** 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.

57 Edit the feature

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

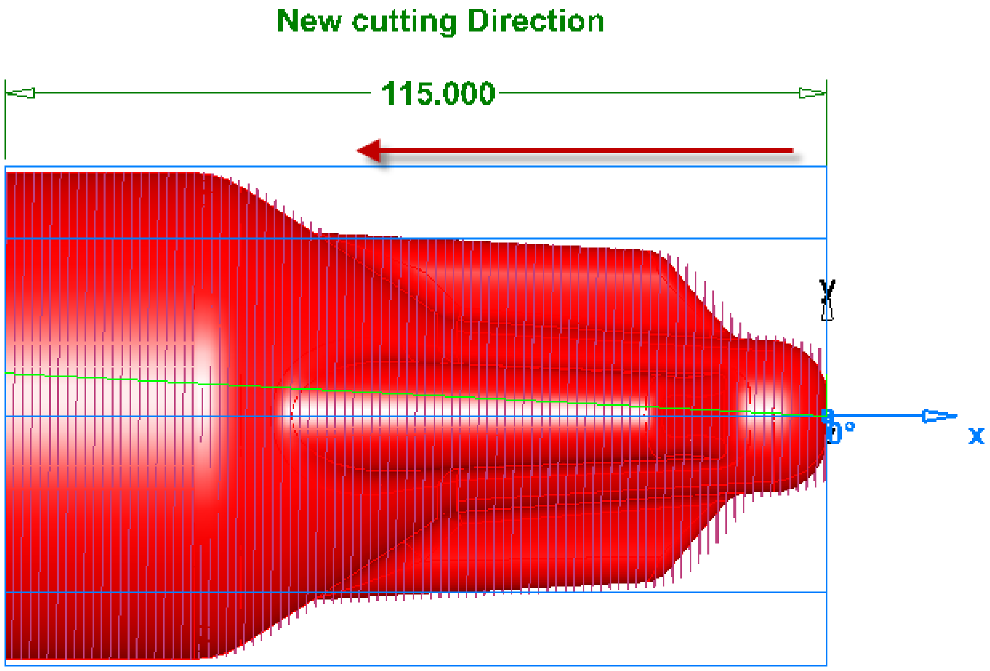
59 Select the **Milling** tab

60 Click **on Index start** and set the value to **0.0mm**

61 Click **on Index end** and set the value to **-115**

62 Click **Apply** and then **OK**

63 Run a **centreline** simulation



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

64 Edit the feature

65 Click on **four axis** in the tree view

66 Set the **Y Offset** to be **6mm**

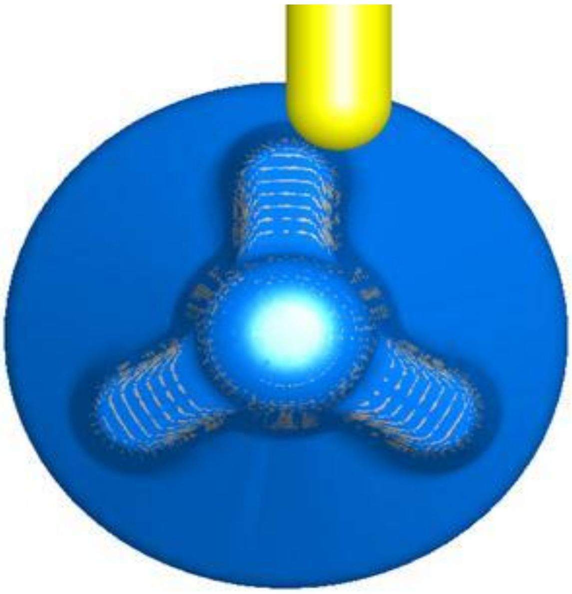
67 Click **Apply** and then **OK**

68 Edit the **stock properties** and change to a User defined stock

69 Select **solid1**

70 Select a **Right** view

71 Run a 3D simulation



It can be seen that the cutter is now offset from the rotary axis by 6mm in the Y direction. This means that the tool is not centre cutting. This will create better cutting conditions giving improved surface finish and tool life.

Contents

Steep & Shallow finishing with separate Shallow Step-over.

Introduction

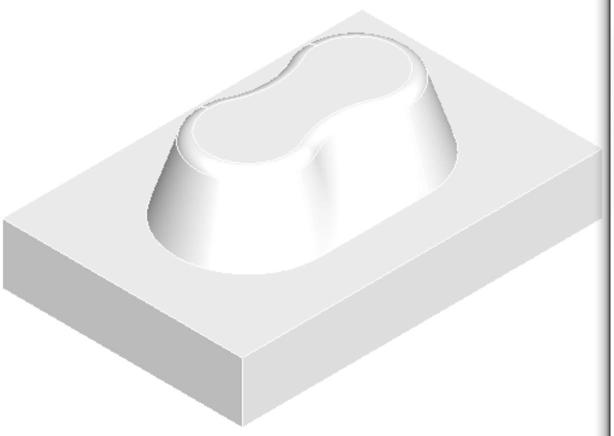
*Both **Optimised Constant Z** and **Steep and Shallow** strategies have an option to apply a separate, larger **Stepover** across shallow areas. This option is designed to enable a more efficient machining of shallow areas with a **Tip Radius** tool.*

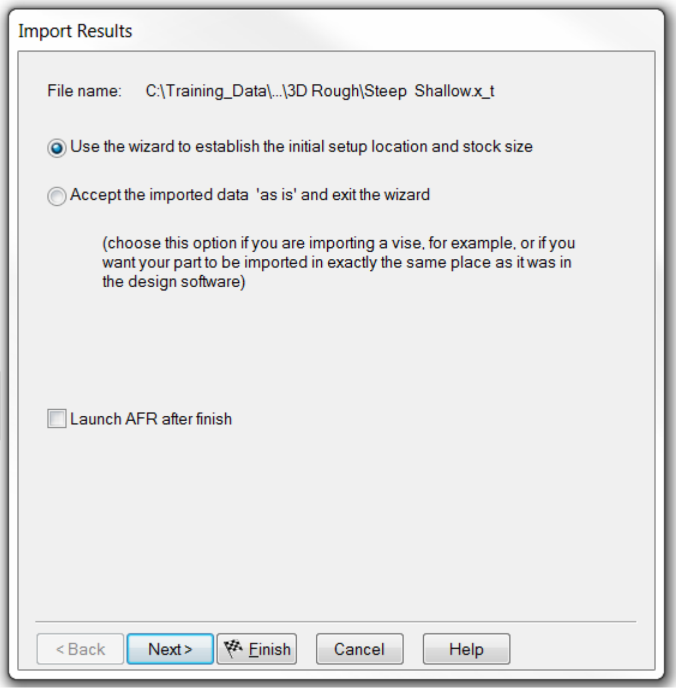
*The main advantage with the **Steep and Shallow** strategy is that the **Threshold Angle** is user defined. This can be set to **0** making the large stepover apply over flat areas only.*

1 Create a new **Metric** Document.

2 Import **Steep & Shallow. X_t** Model into **FeatureCAM**.

3 Go to **File**>**Import.** Browse to your instructors preferred location for **3D models.**



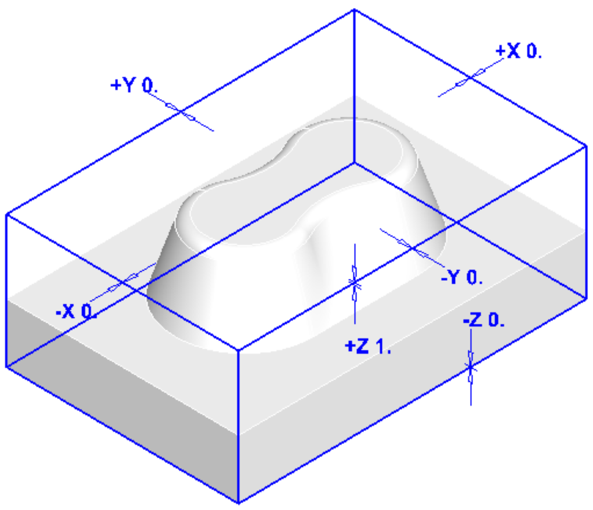


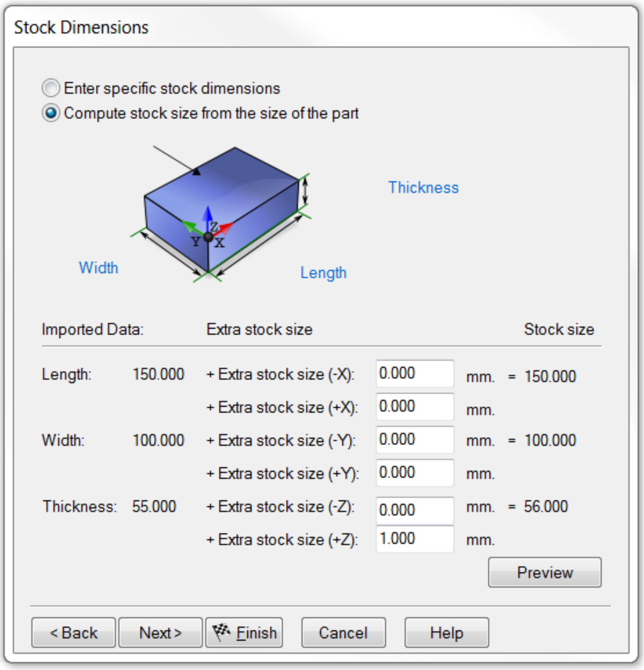
4 Untick Launch AFR after Finish.

5 Select Next 4 times until you get to Stock dimensions.

6 The model is already aligned.

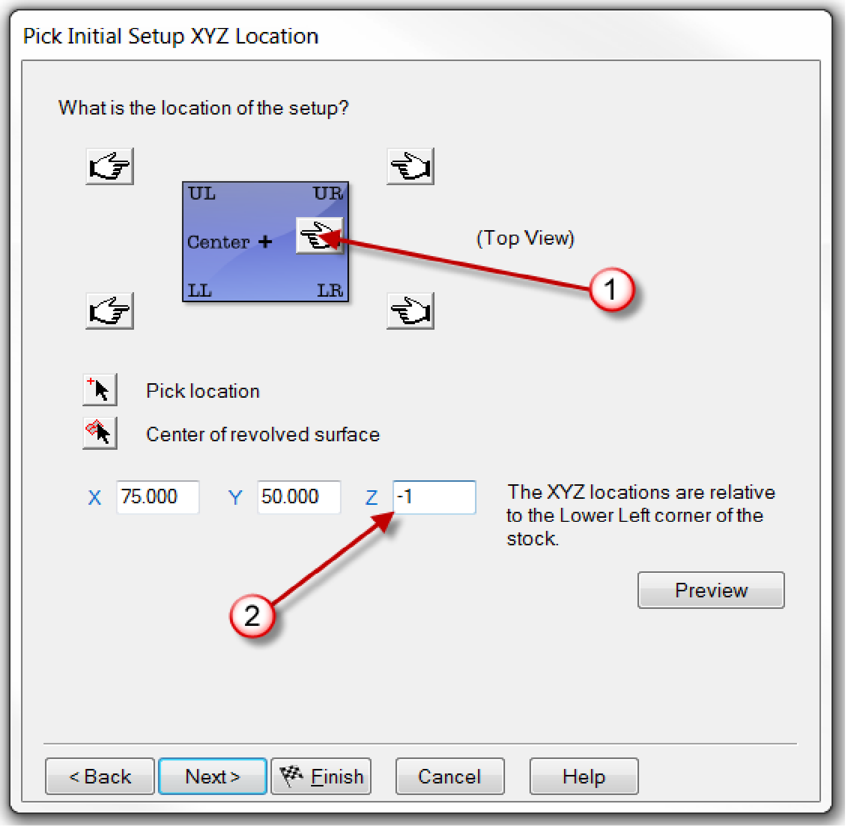
7 Enter the following **Metric** Stock dimensions.



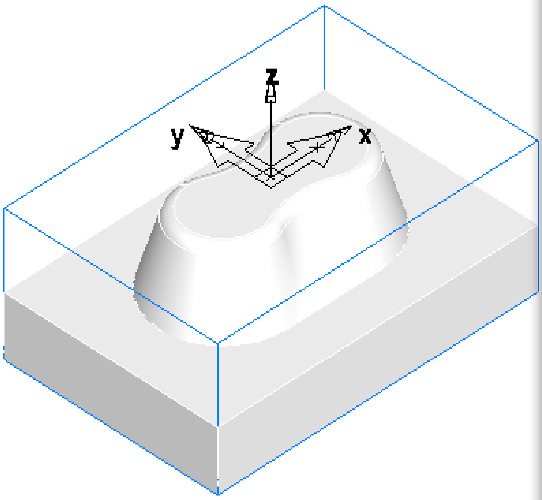


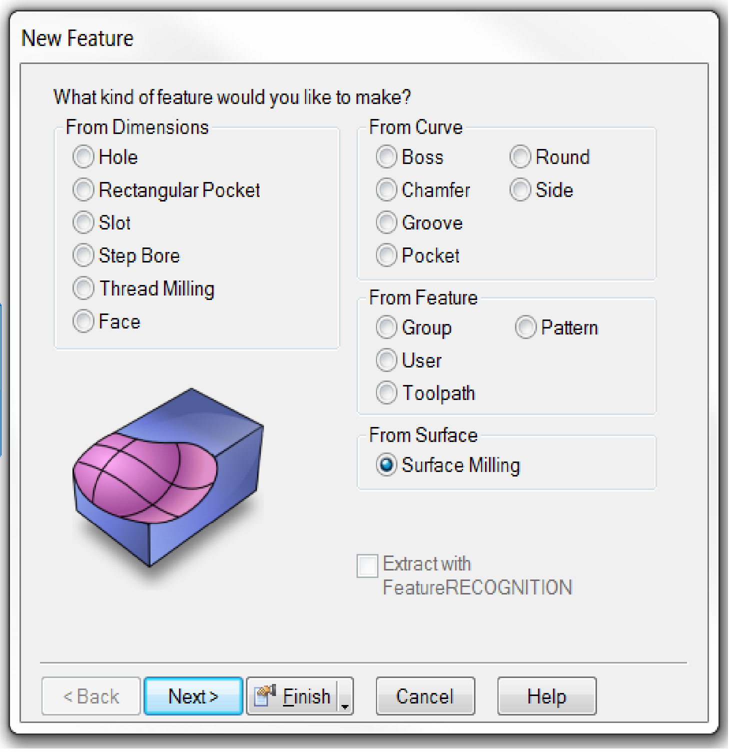
8 Select Next to set the Pick Initial Setup XYZ Location Metric.

9 Select the Centre + 1 button which will set the Z position to Zero. Then enter 2 *-1* which will set the Setup position to the top of the solid model.

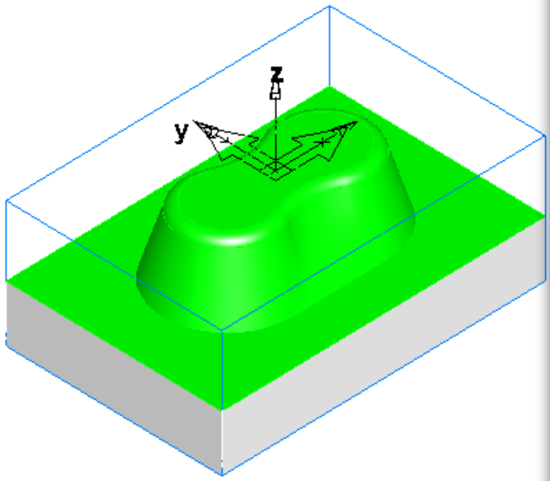


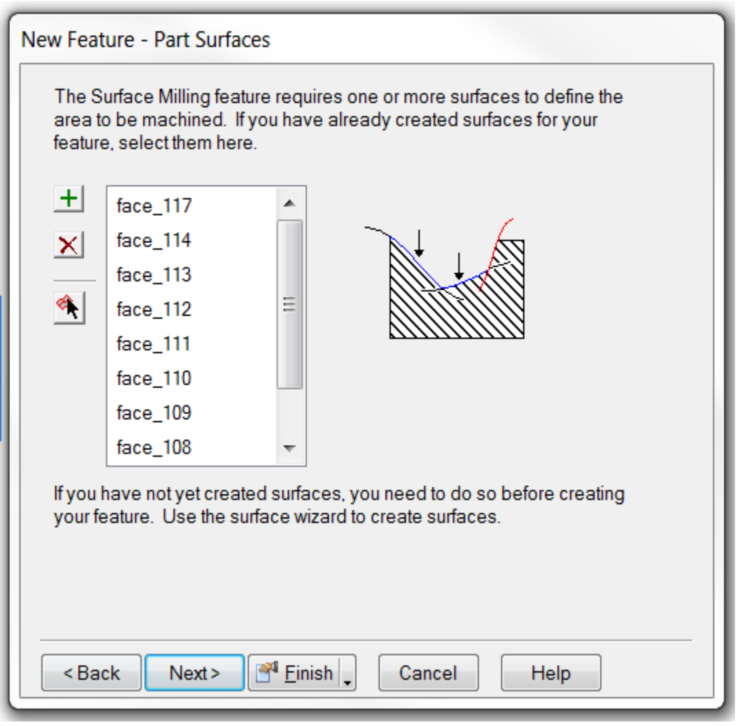
10 Create a New Feature, Surface Machining. Select **Ctrl+R** to activate the New Feature Menu.





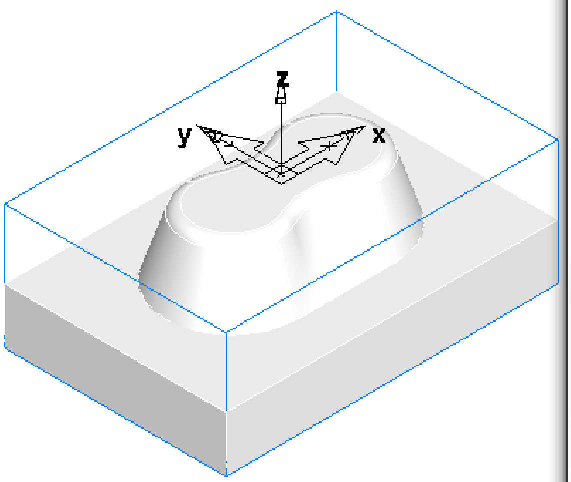
11 Select **Next**.

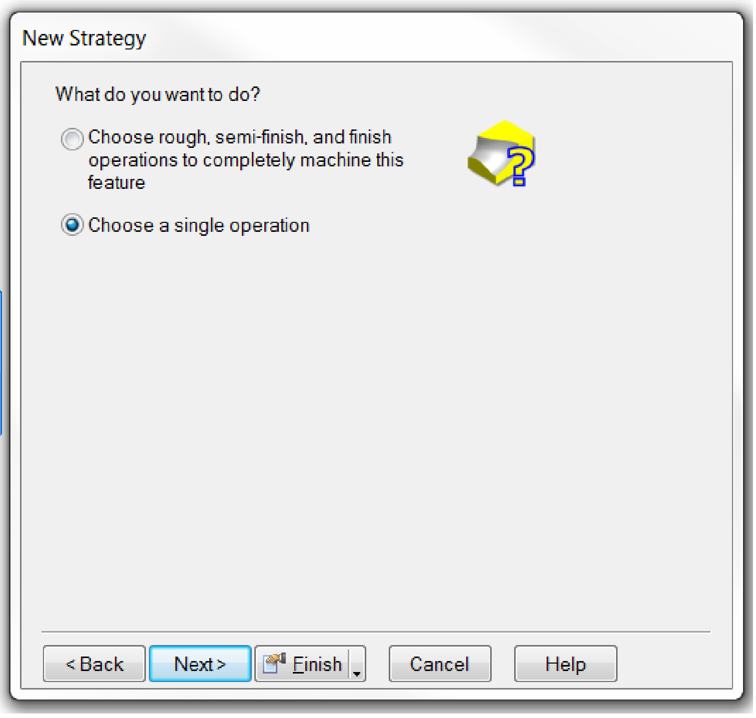




12 Select only the Surfaces shown above.

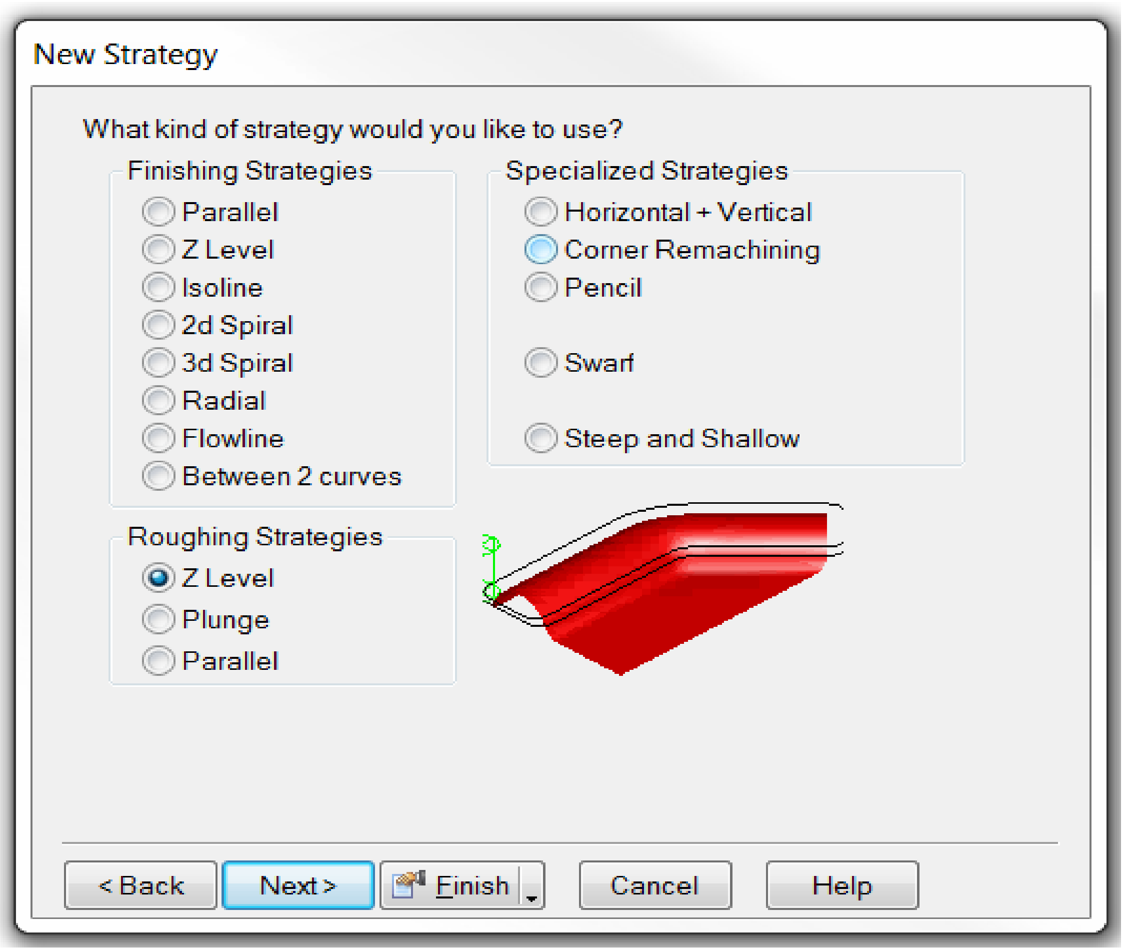
13 Select **Next**.





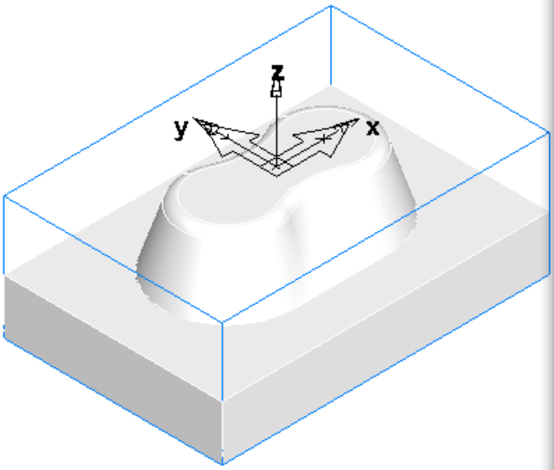
14 Select Choose a single operation.

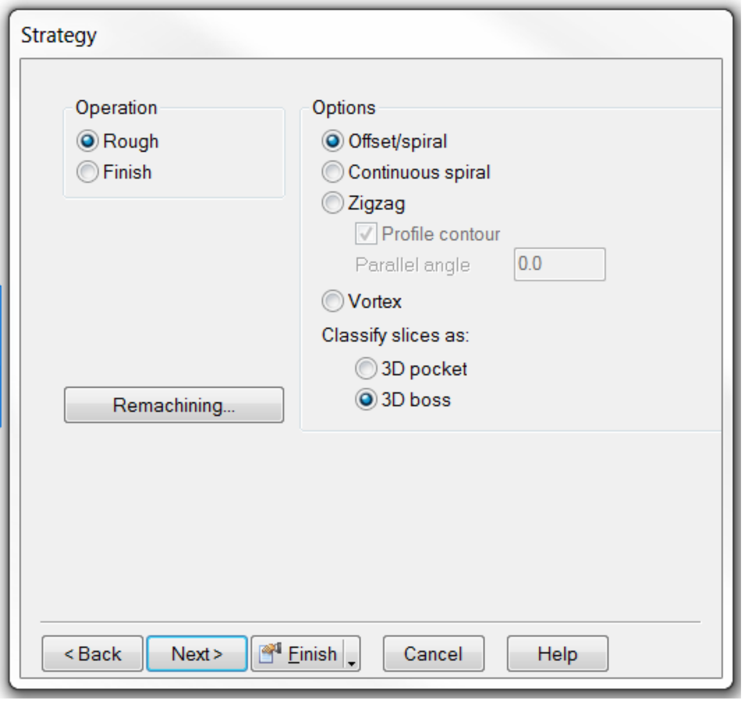
15 Select Next. Select Z Level.



16 Select **Next**.

The following menu will appear.

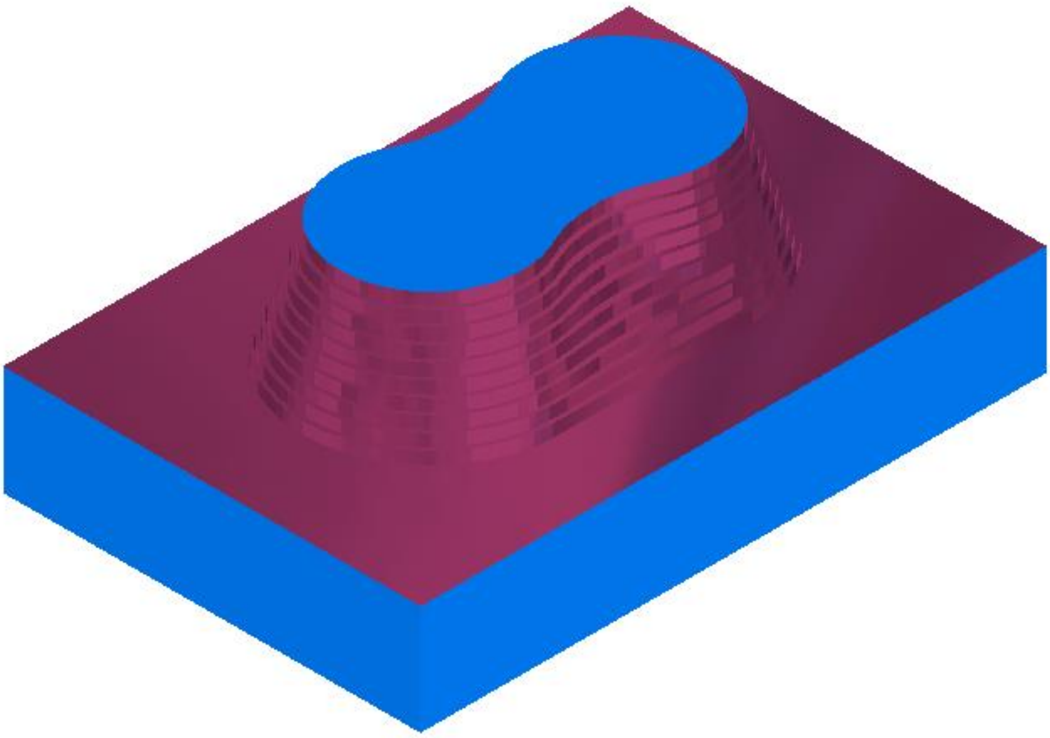




17 Select Rough>Offset/Spiral>3D Boss.

18 Select Finish.

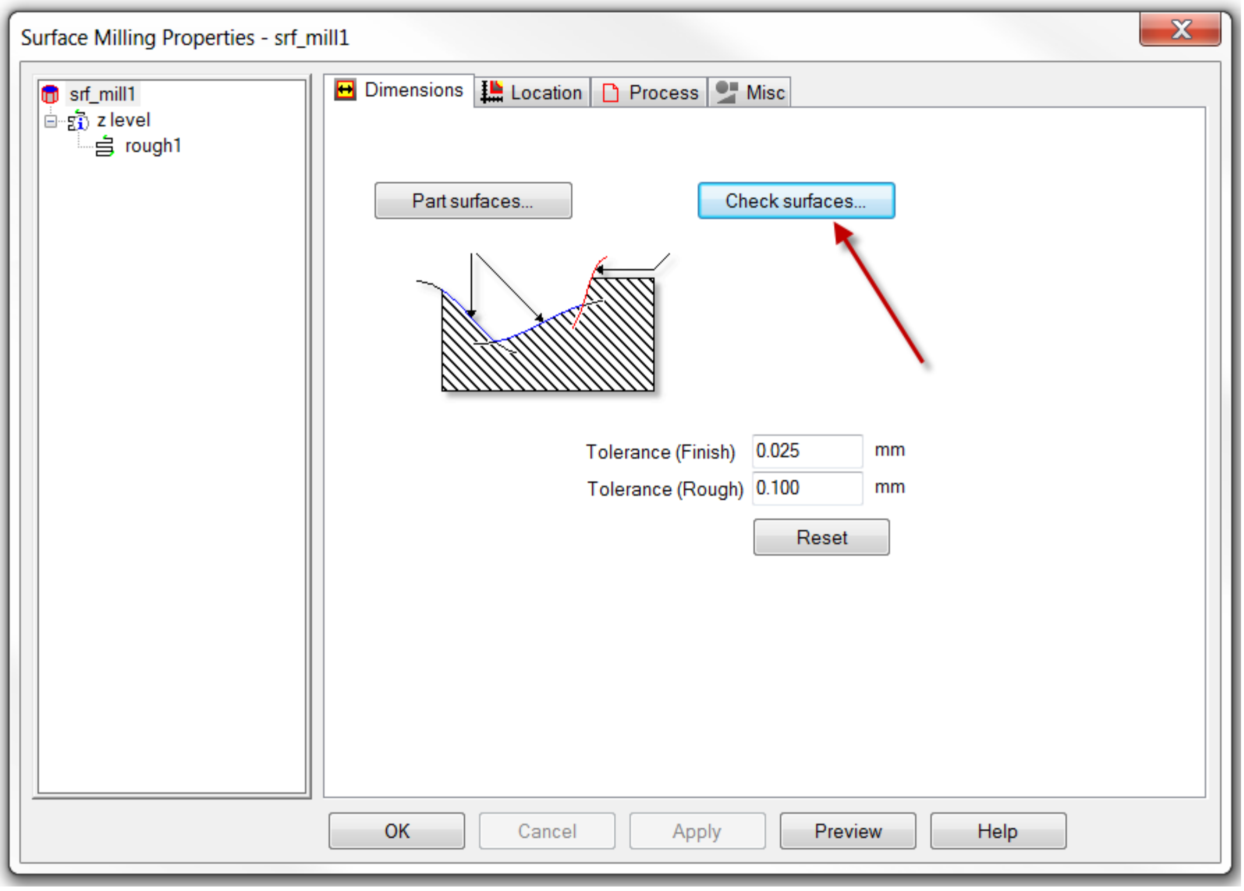
*This will Rough out our **Steep and Shallow** example.*



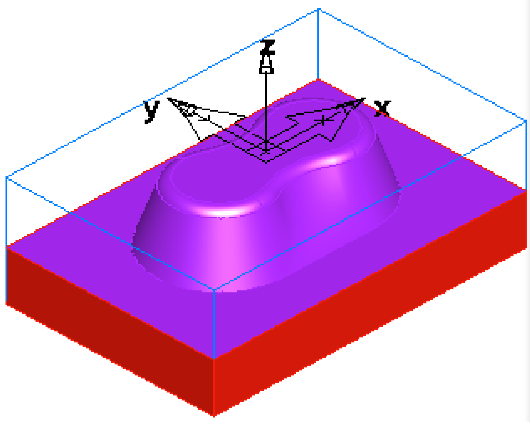
19 Change the Override Depth of cut on your tool, to change the depth of cut to a more suitable step down.

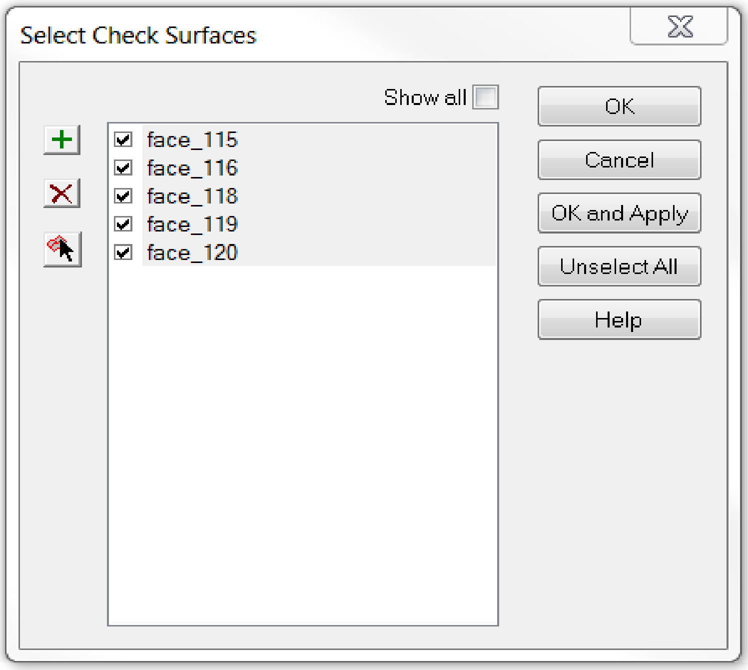
20 In **PartView** select **srf_mill1** by double clicking the **Feature.**

21 The following menu will appear.



22 Please select **Check Surfaces**.

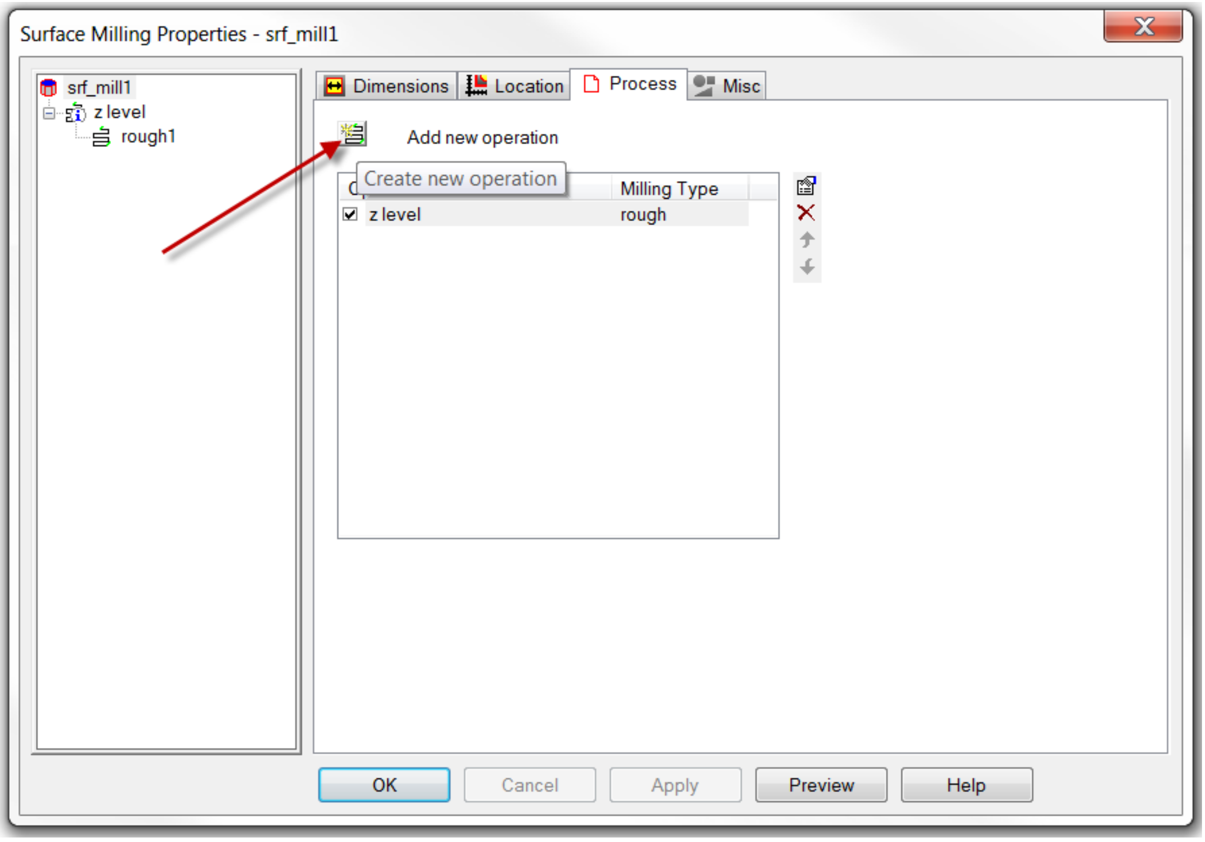




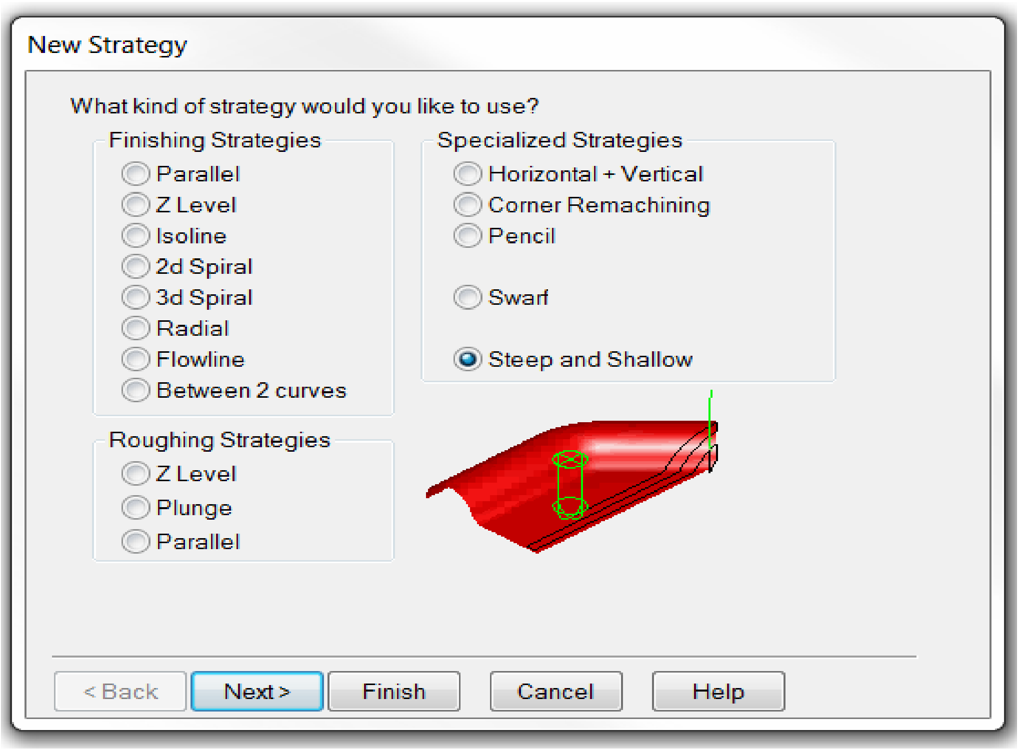
23 Pick the surfaces selected above. This will make sure that these surfaces are not machined. **(Check Surfaces).**

24 Select **OK** to close the menu.

25 Then select **Process** and then select the **Add new operation** icon.

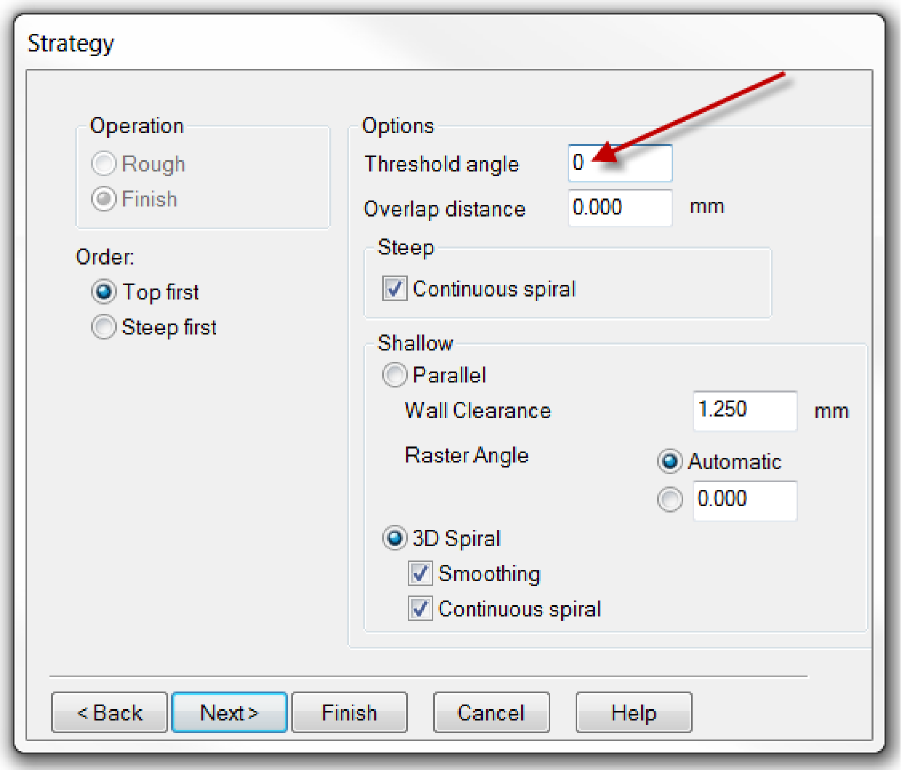


26 The following menu will appear. Select **Steep and Shallow**.



27 Select **Next**. The following menu will appear.

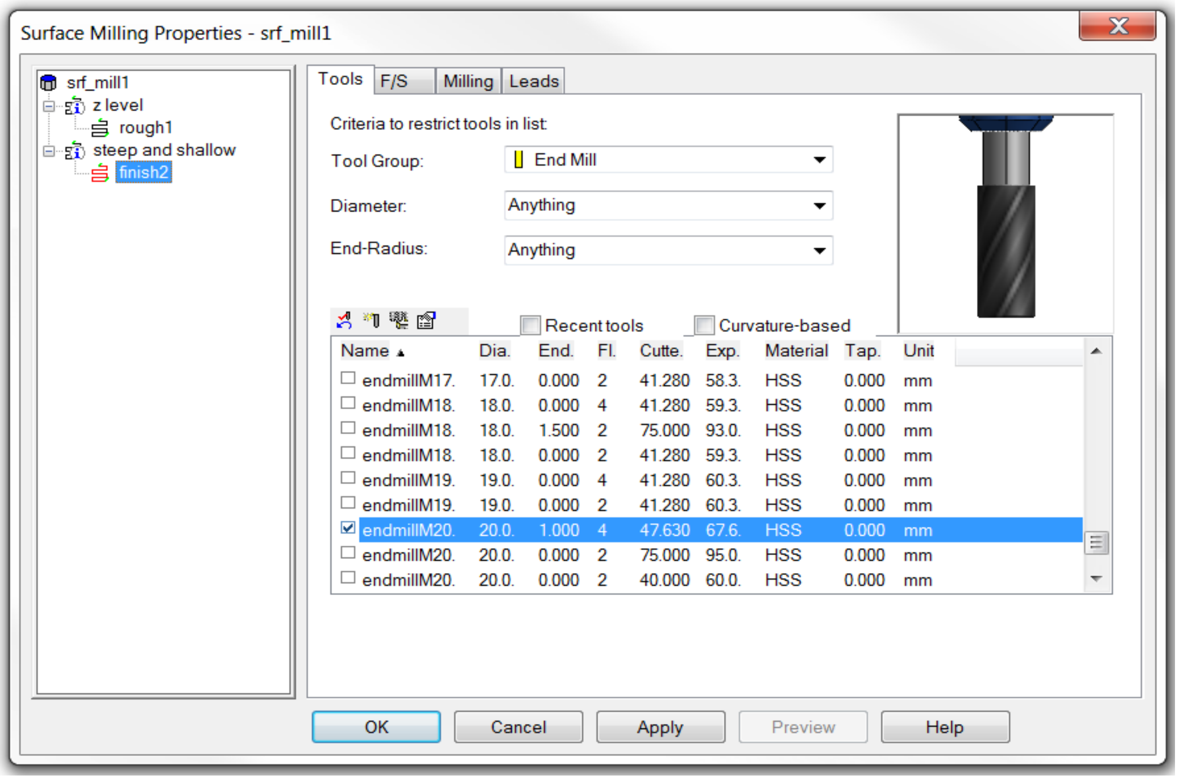
*By default the **Threshold angle** defaults to **30.00** degrees. Set this to **Zero**. Then select **Continuous spiral** for **Steep** and Smoothing and Continuous spiral for **Shallow.***



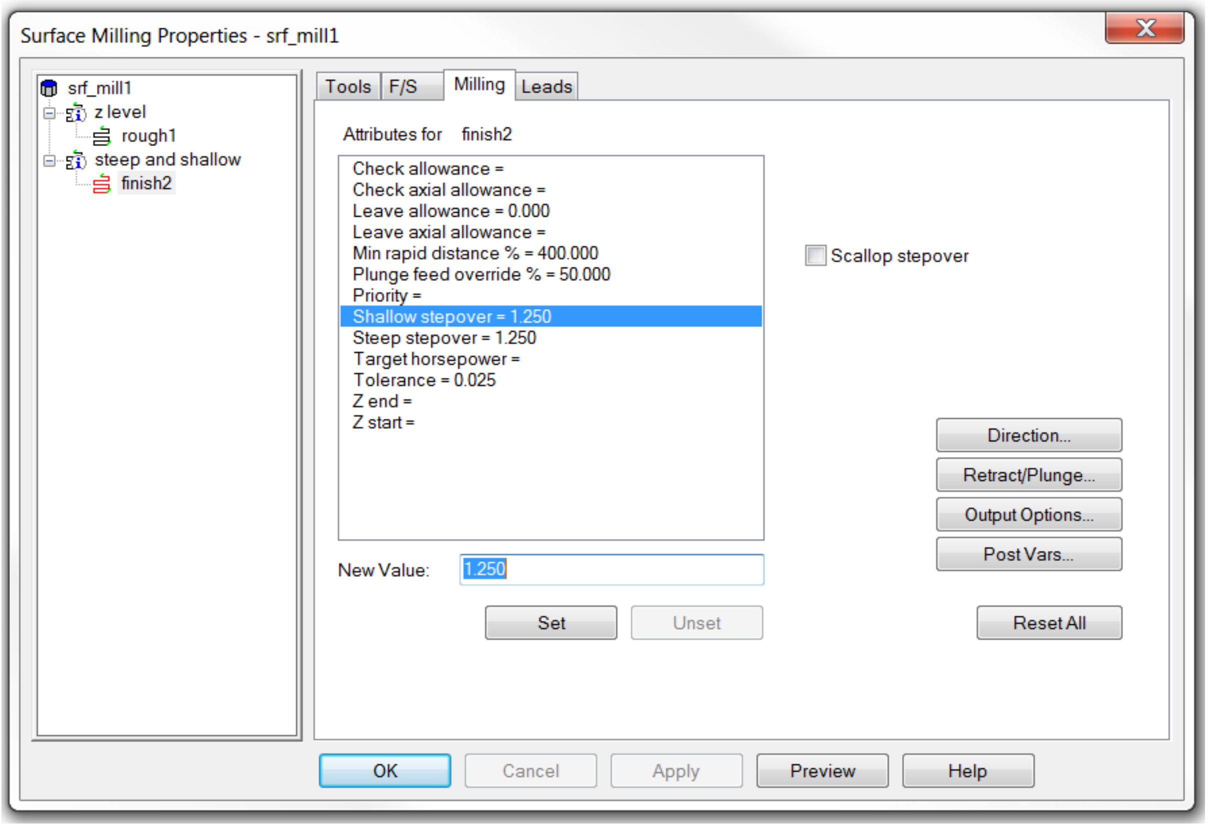
28 Select **Finish** accepting the default settings.

29 By default **FeatureCAM** will select a **12mm** Ballend tool.

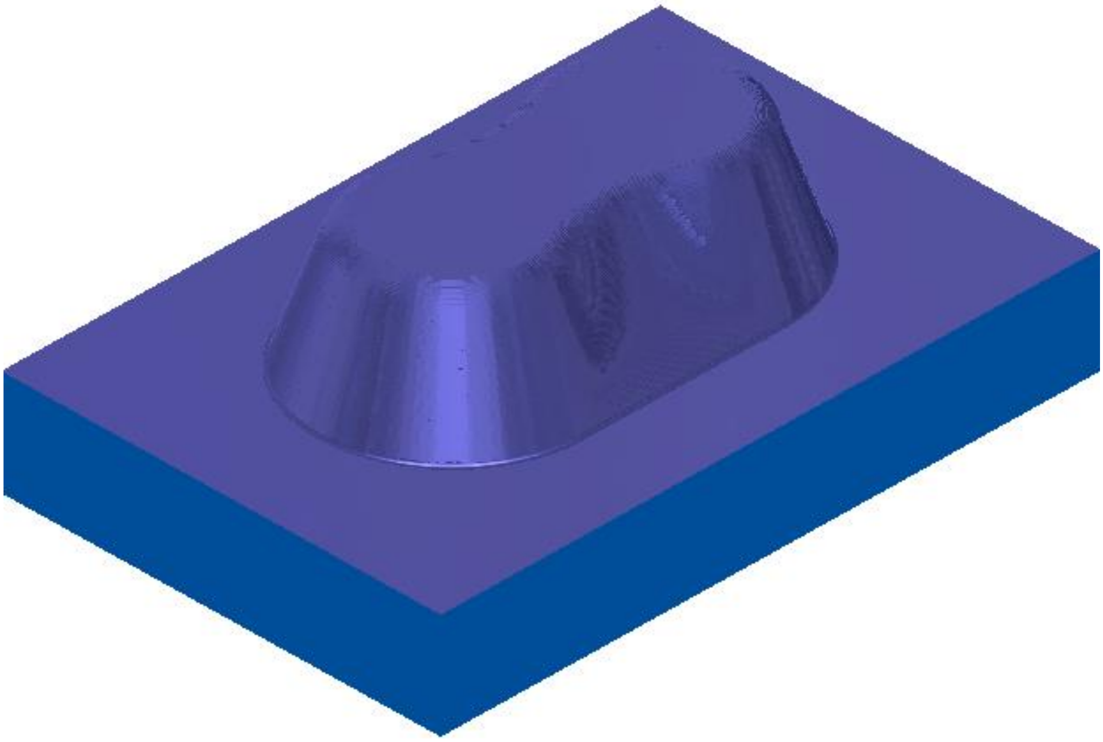
30 We need to change this to a Tip radius tool with a larger diameter and a small tip radius. For example **20mm R1.**

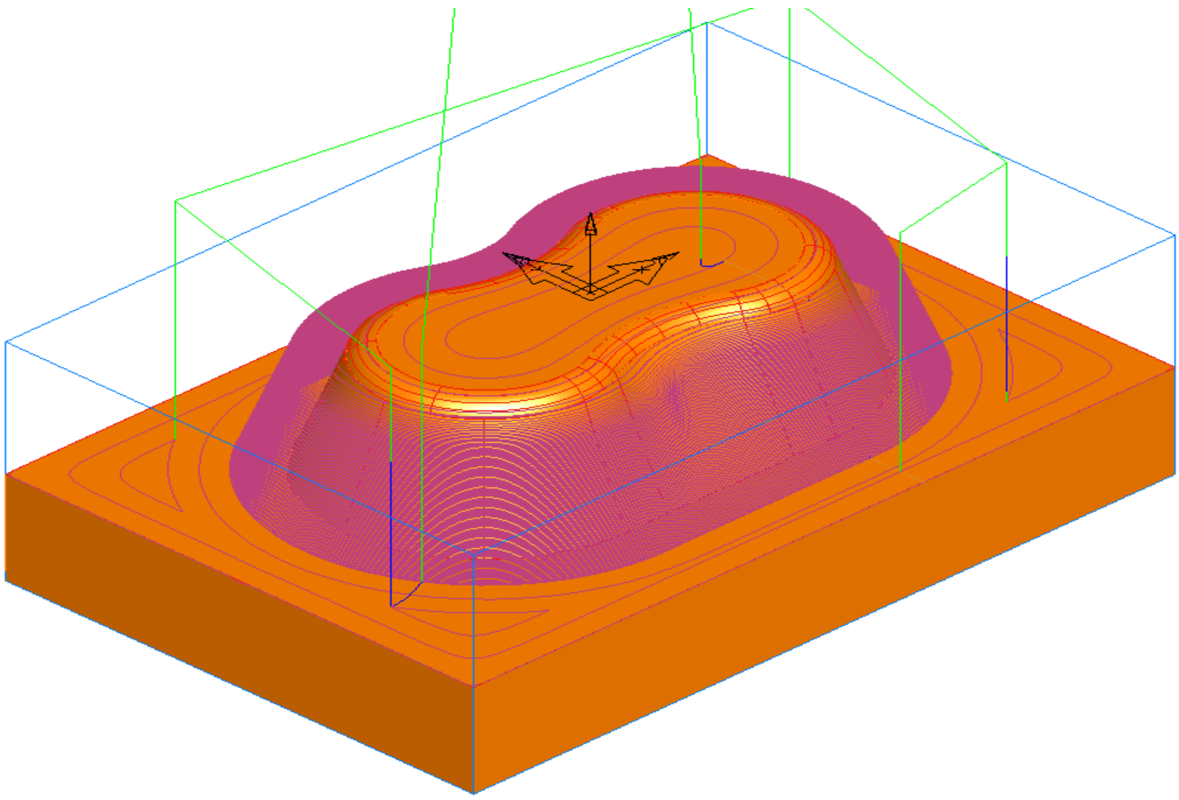


31 Please select the **Milling** Tab in the same menu. The following menu will appear with default settings.



*We need to set the following parameters. **Scallop stepover**. Please select this option. This will remove the **Steep stepover** and replace this with **Scallop Height**=**0.025** Change the Steep stepover = **8mm** Then change the **Check axial allowance** to **-8mm** The image below shows a **3D Simulation** using these values.*





Notice the larger step over on the flat surfaces.

32 Save the file in your Instructors preferred location.

Contents

Vortex 3D

If you have previously covered the 2.5D Milling Course and Vortex, then the first three exercises may be skipped.

- **Vortex** has been developed by Delcam specifically to gain the maximum benefit from solid carbide tooling, in particular those designs that can give deeper cuts by using the full flute length as the cutting surface.
- It can be used for 2 and 3 axis roughing, positional five-axis area clearance and for rest machining based on stock models or reference toolpaths.
- While **Vortex** will be an available strategy for 2.5 Axis features, a Feature RECOGNITION license will be required to utilize it.

What is Vortex and how does this toolpath strategy work?

- **Vortex** machining is an area clearance strategy that rapidly removes material from a 3D part while controlling tool load. **Vortex** is best suited to solid carbide tools and is frequently used in combination with step cutting (3D HSM license is required for Step Cutting).

***Vortex** is an offset-style toolpath and has these main features:*

- The engagement angle never exceeds more than 15%, which is produced by a straight line cut with a given stepover. This eliminates excessive tool load on all full-width cuts. This enables you to increase feed rates. For other area clearance toolpaths, the cutting values are based on the tool manufacturer's slot cutting parameters to ensure the tool can sustain full cutting engagement. As the tool approaches the maximum engagement angle for optimum machining, the toolpath changes to a trochoidal path to avoid tool overload.
- The machine tool almost always runs at the specified feed rate. With other area clearance toolpaths, the machine tool automatically slows down as it approaches a corner and the engagement angle increases. **Vortex** modifies the toolpath so the tool engagement angle is never exceeded and the machine tool achieves the specified feed rate. The only time the machine tool doesn't run at the specified feed rate is when the model geometry (a slot or corner) is smaller than the smallest radius that the machine can run at full speed.

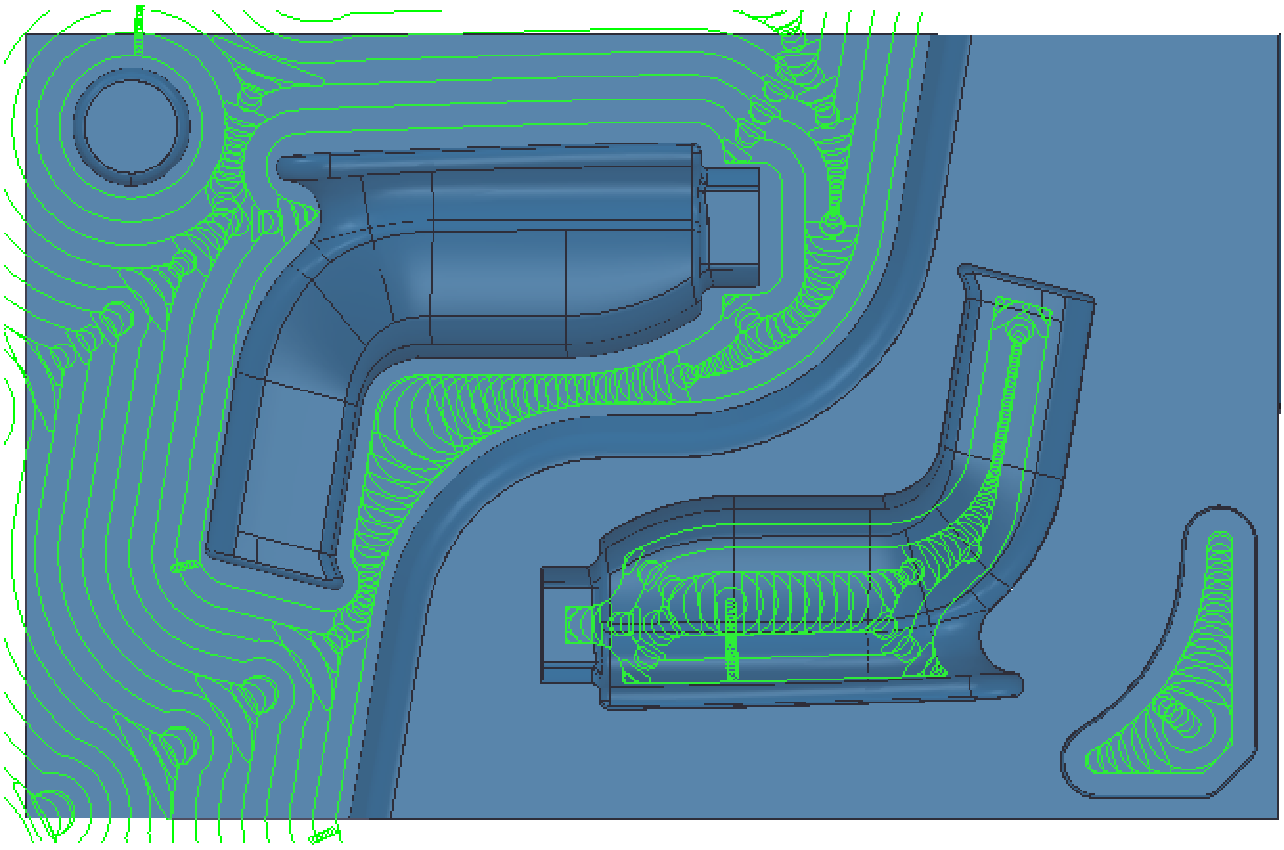
  ***Vortex** machining cuts with the side of the tool so it is designed for solid carbide tools, but you may be able to use other tools. Because FeatureCAM controls the tool engagement; you can increase the depth of cut, which minimizes machining time. **Vortex** machining is frequently used in combination with **Step cutting** to minimize terracing while maximizing the removal rate.*

  ***Vortex** toolpaths are automatically checked for safety. FeatureCAM checks for:*

- Plunges into stock.
- Excess tool engagement
- Excess depth of cut
- Small arc movements

*To maximize the benefits of **Vortex** machining:*

- Configure the **Vortex** parameters to suit each machine tool.
- Use step cutting (3D HSM only) to minimize terracing caused by the increased depth of cut.
- With optimum settings, **Vortex** machining greatly reduces machining times.



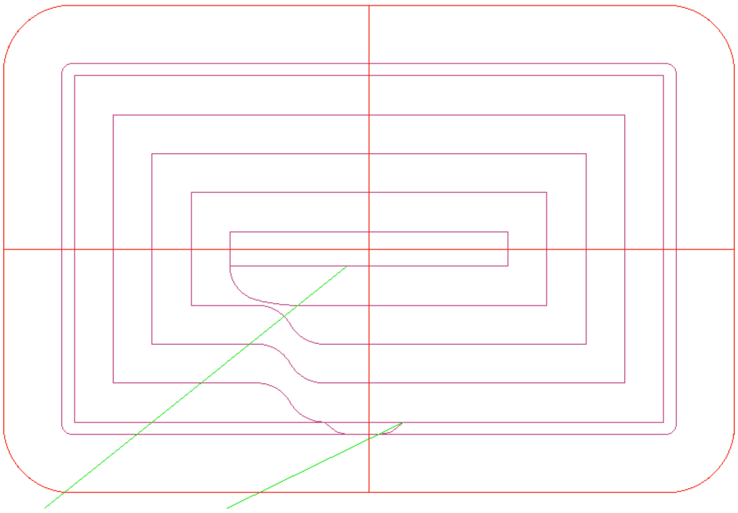
Vortex example #1

How to create a Vortex toolpath (2.5D)

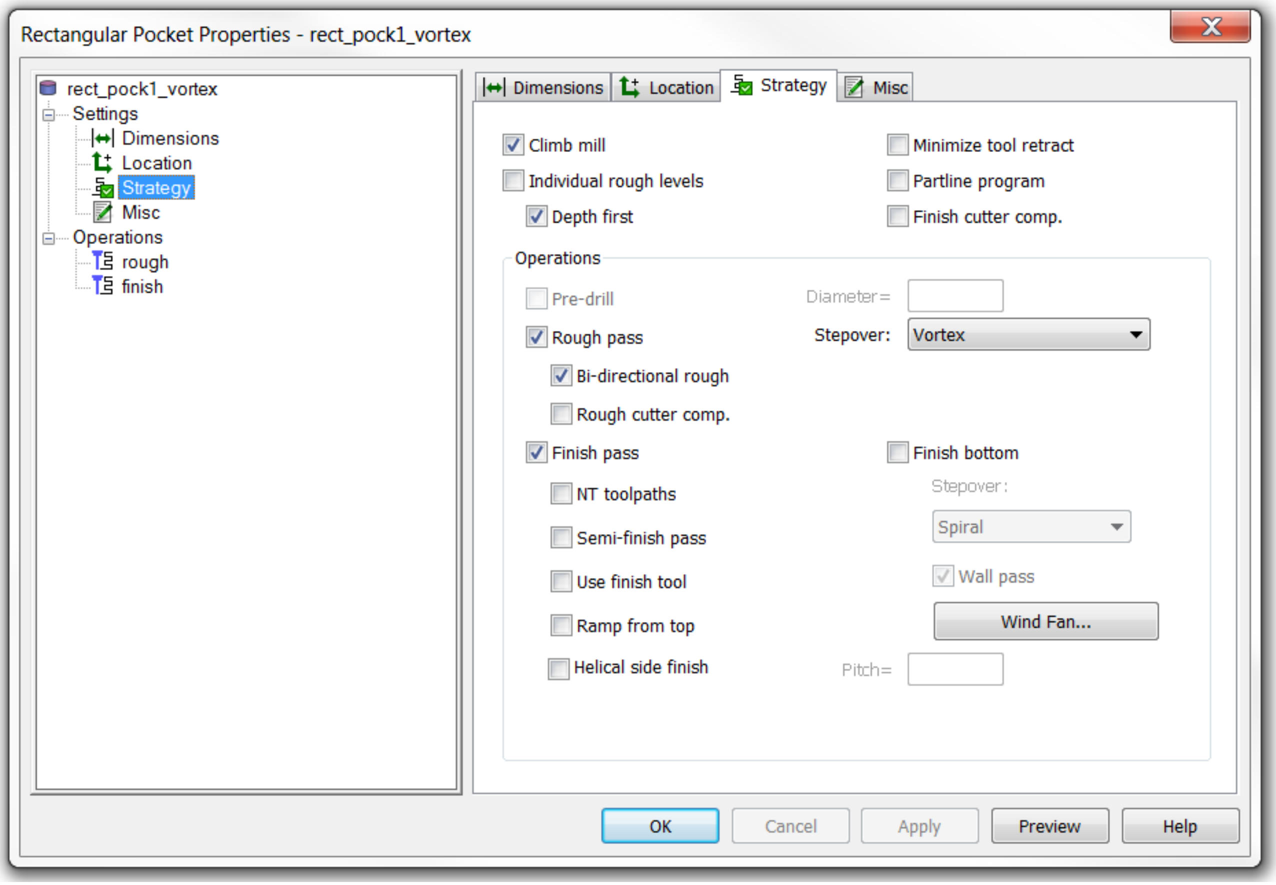
1. Open the file named **Vortex_example1.fm**

   There are already operations for rectangular pocket and Vortex in this file and it is using a 12mm endmill to machine the pocket.

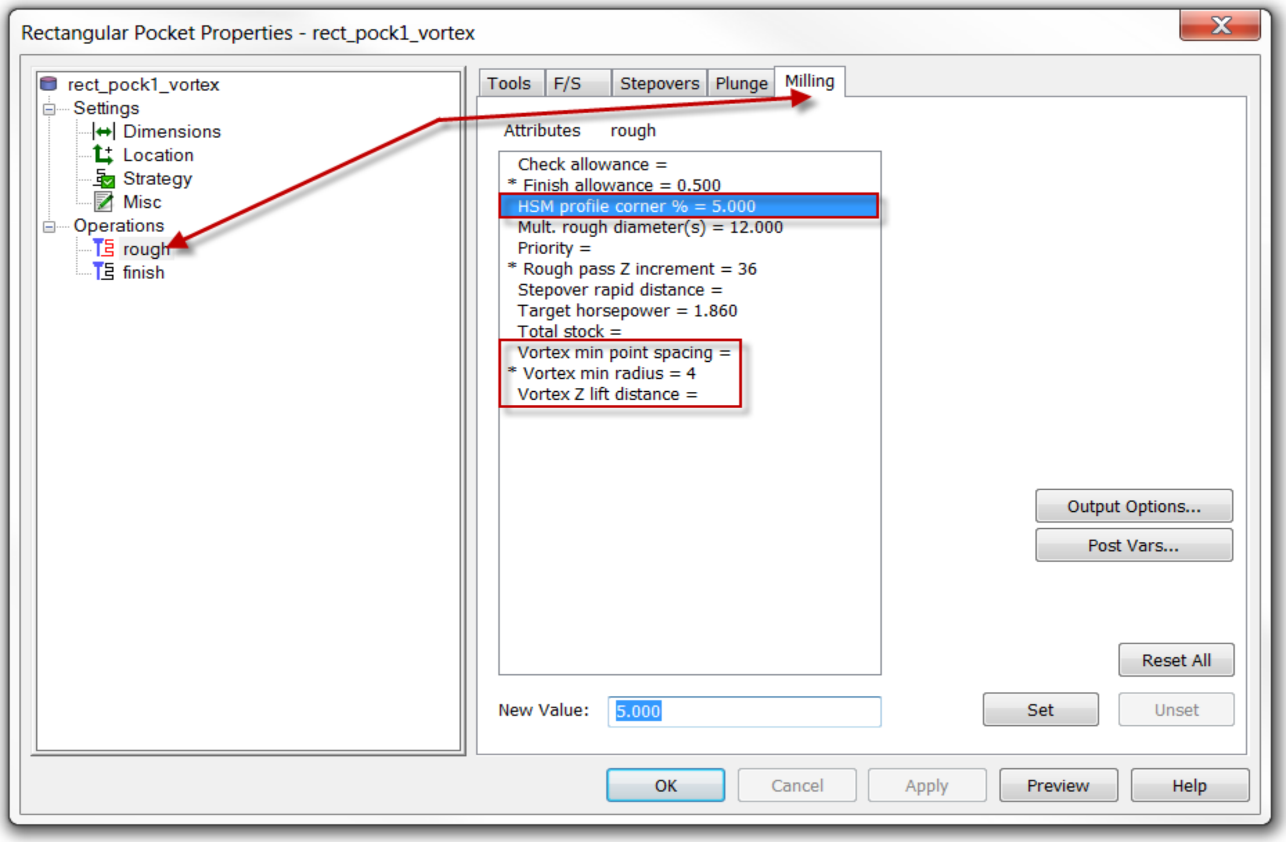
2. Run a **Centerline Simulation** of the Rectangular pocket operation only to see the toolpath similar to that shown below:



3. Check the time of this operation in details.
4. Unselect the Rectangular Pocket operation in PartView and then select the Vortex toolpath. We have set the Z Level increment to 36mm which is the depth of the Feature.
5. Run a **Centerline Simulation** and then look at the details and compare the time.
6. To familiarize yourself with the different parameters double click on the Vortex toolpath in PartView.



7 Click on the **Rough** operation in the feature properties window and select the **Milling tab**. Do not change anything right now.



*The user will need to determine the parameters they would prefer **Vortex** to use when calculating our toolpath. Let's examine these settings.*

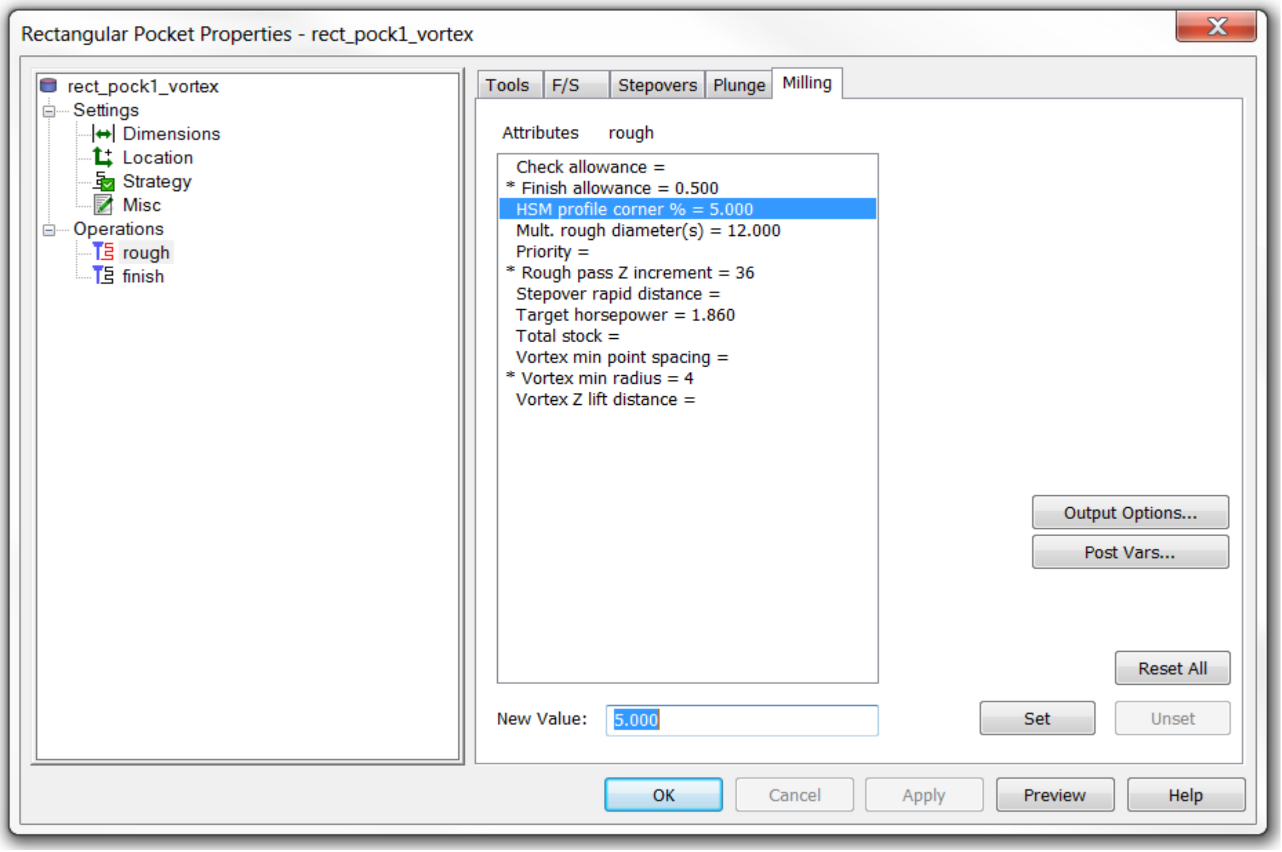
Vortex min point spacing: - Enter the minimum point spacing at which the machine tool can move at the specified feed rate. If the machine tool has too many points to process, it cannot sustain the specified feed rate.

Vortex min radius: - Enter the minimum radius of the internal trochoids. **Vortex** toolpaths use trochoidal moves to maintain a constant feed rate. Higher feed rates require a larger minimum radius.

Vortex Z lift distance: - Enter a Z distance to lift the tool during trochoidal moves to avoid contact between the tool and the surface.

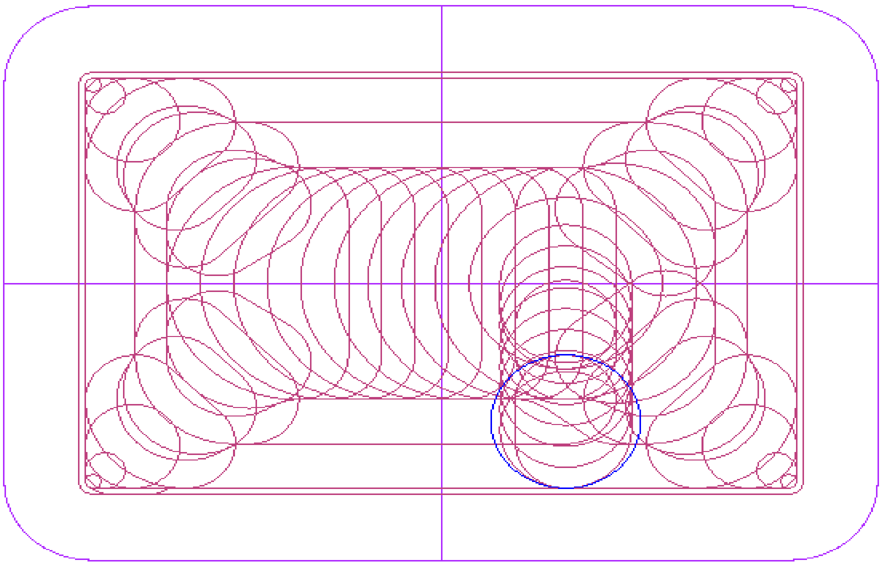
*There is also a setting called **HSM profile corner %**. For this setting enter the value as a percentage of the tool's diameter. **This enables arc fitting of profiles to avoid sharp changes of direction in internal toolpaths.** This attribute applies to NT Spiral, NT Continuous Spiral, NT Zigzag and **Vortex** toolpaths.*

*For now we will leave the settings at the default when we activated **Vortex**, you should only see the **HSM** profile corner% at **5.000** as shown below:*

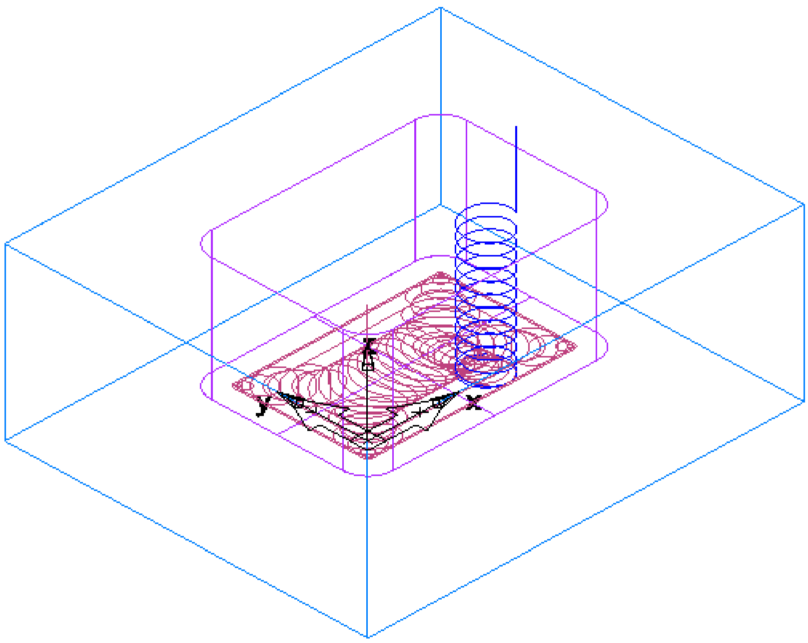


8 Select **OK**

9 Run a **Centerline Simulation** to see the altered toolpath.



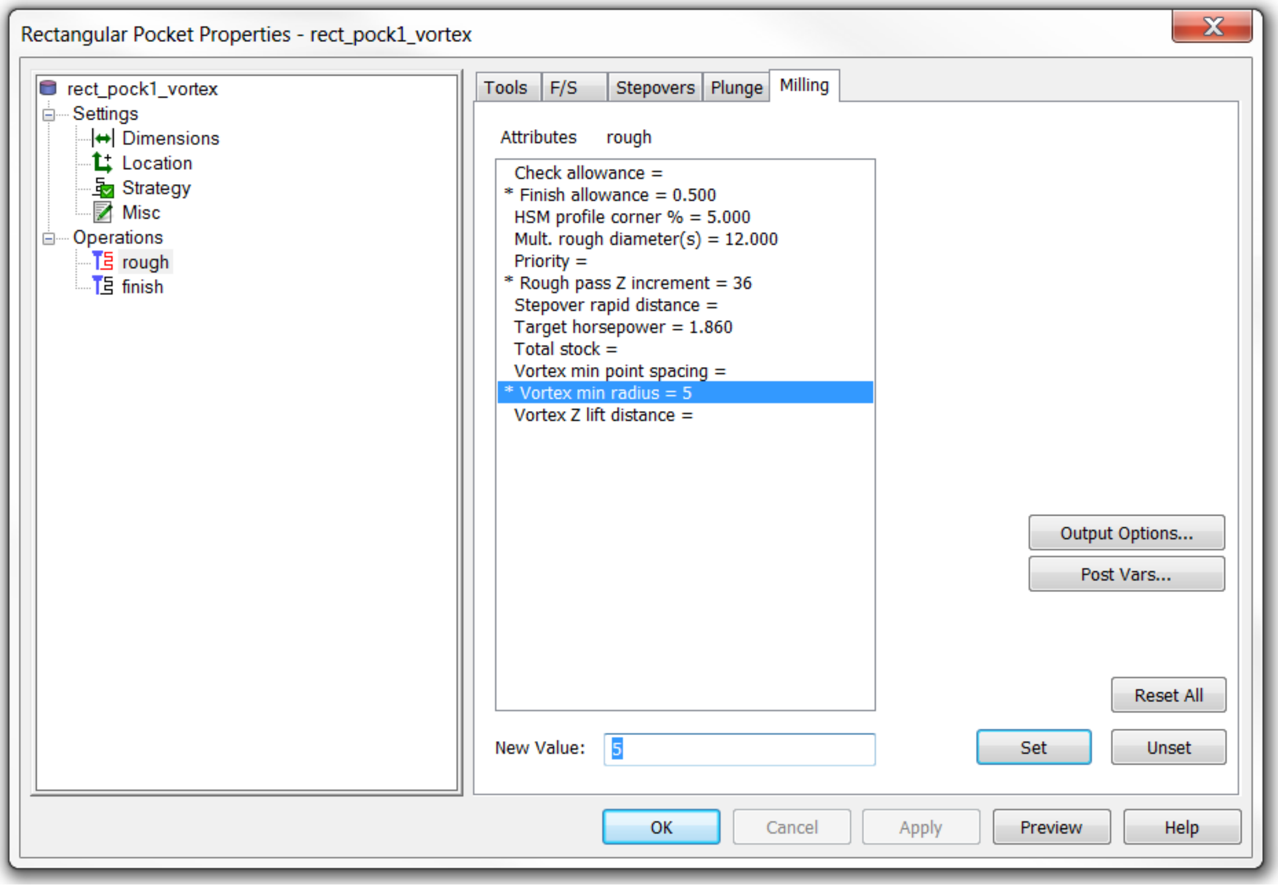
*You can see (in blue) the helical ramp of the tool and also the channel that **Vortex** creates to clear material out. Essentially what is taking place is that for pockets, the tool spirals down into the pocket before using trochoidal paths over the full-width cuts. On completion of the initial full-width cut, the trochoids are placed in the corners where the maximum tool engagement angle would otherwise be exceeded.*



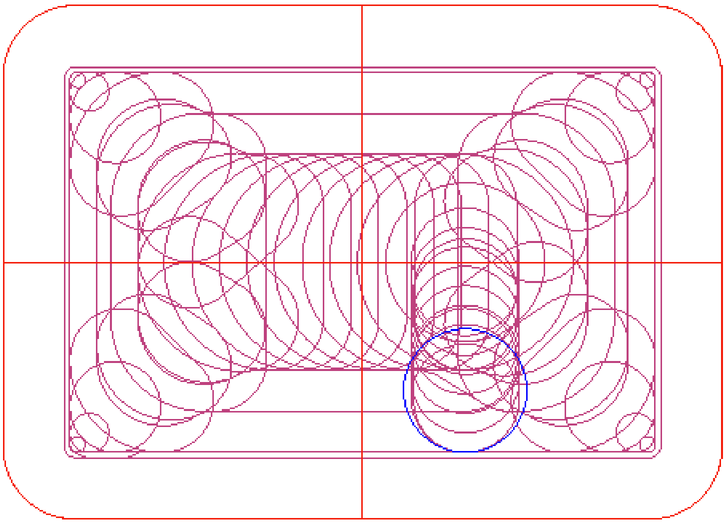
10 **Double-click** on **the Rectangular Pocket** in the **Part View** of the **Toolbox** on the left hand side of the FeatureCAM interface to open the Properties of the feature once again as before.

11 Navigate to the rectangular pocket feature's **Milling tab** for the **rough** operation again.

12 Set the **Vortex min radius** to **5mm** and run a **Centerline simulation** again.



*Notice the larger **Vortex** trochoids in the toolpath. Try using different values to see the change.*

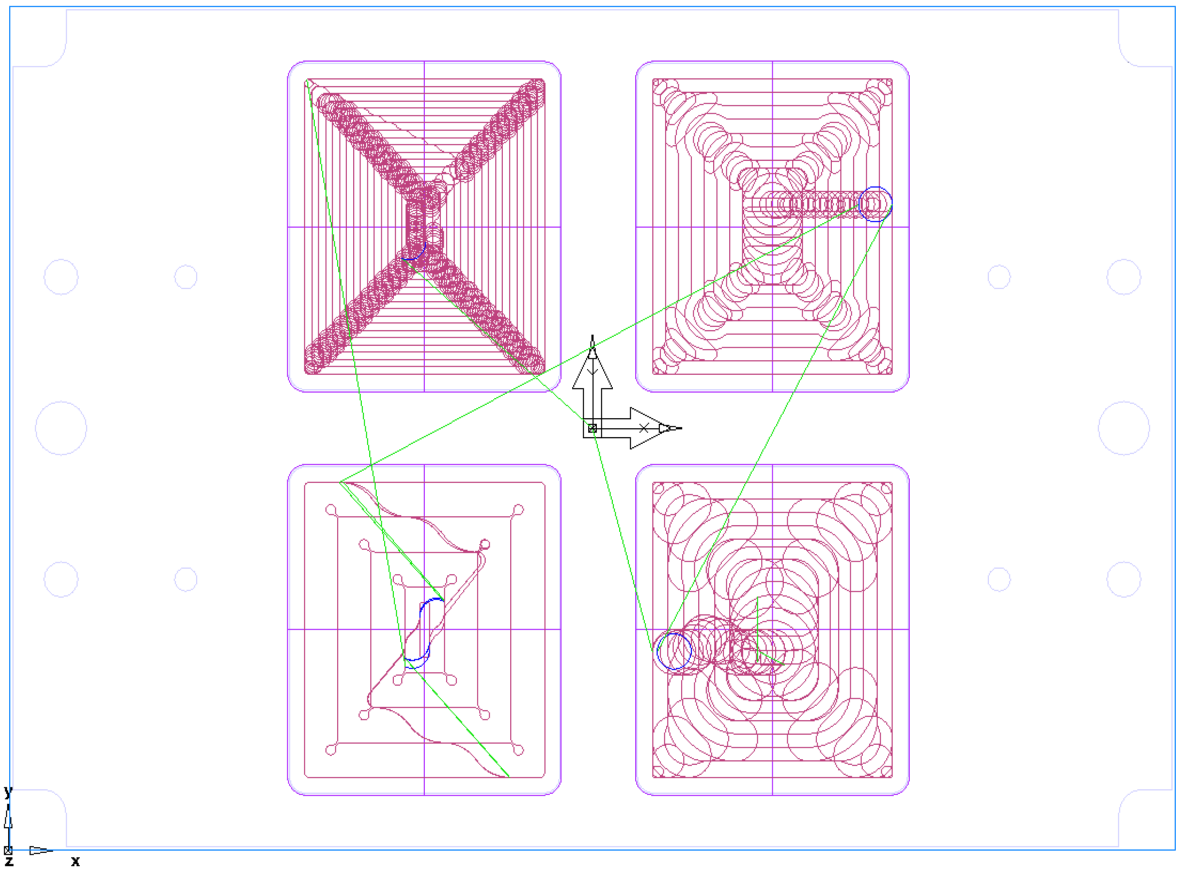


Remember, when using higher feed rates a larger minimum radius is required.

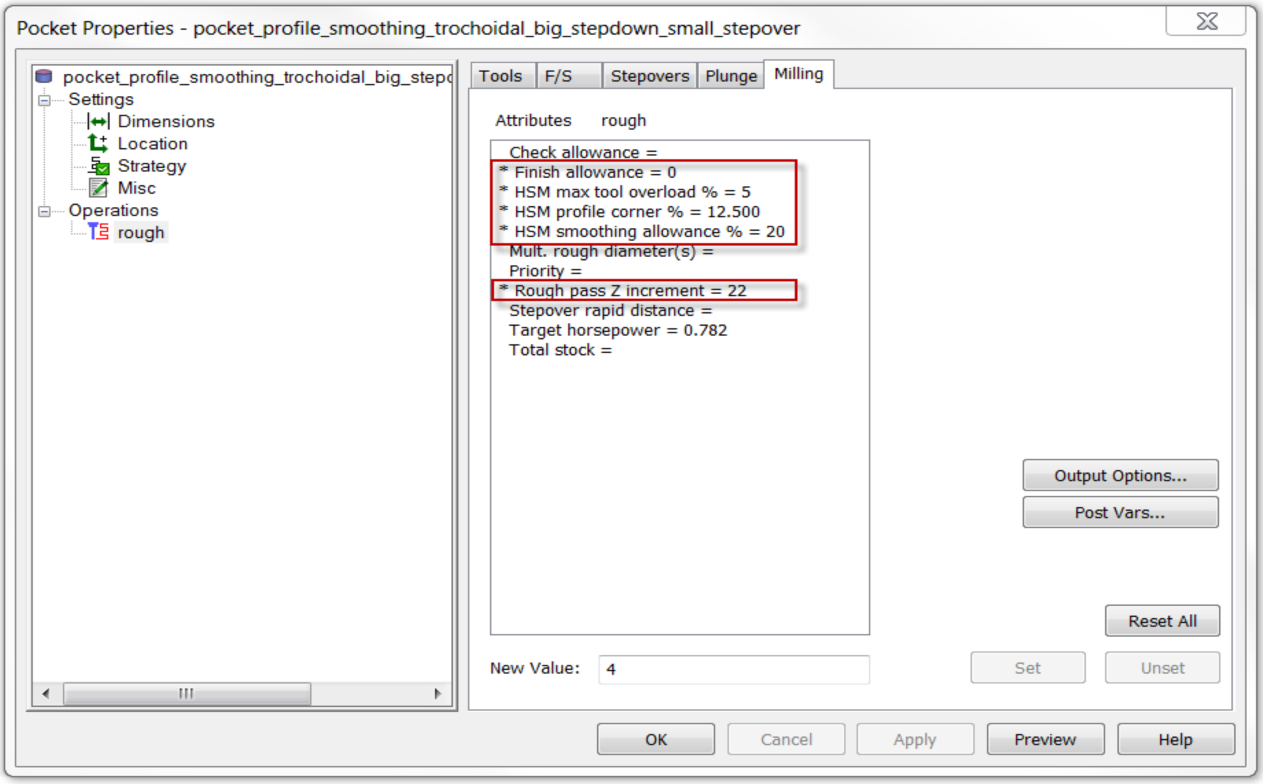
Vortex Example #2

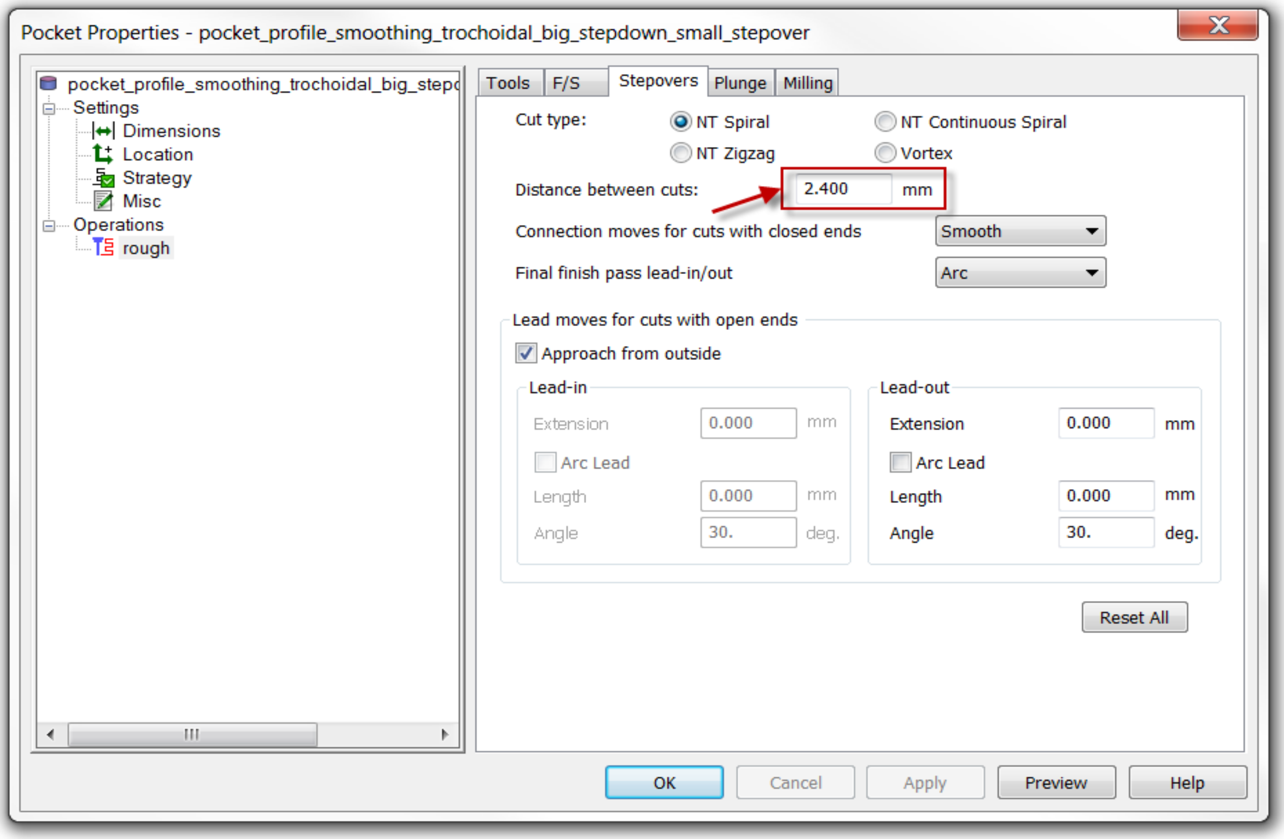
1 Open the file named **Vortex example2.fm**

2 Run a **Centerline simulation**

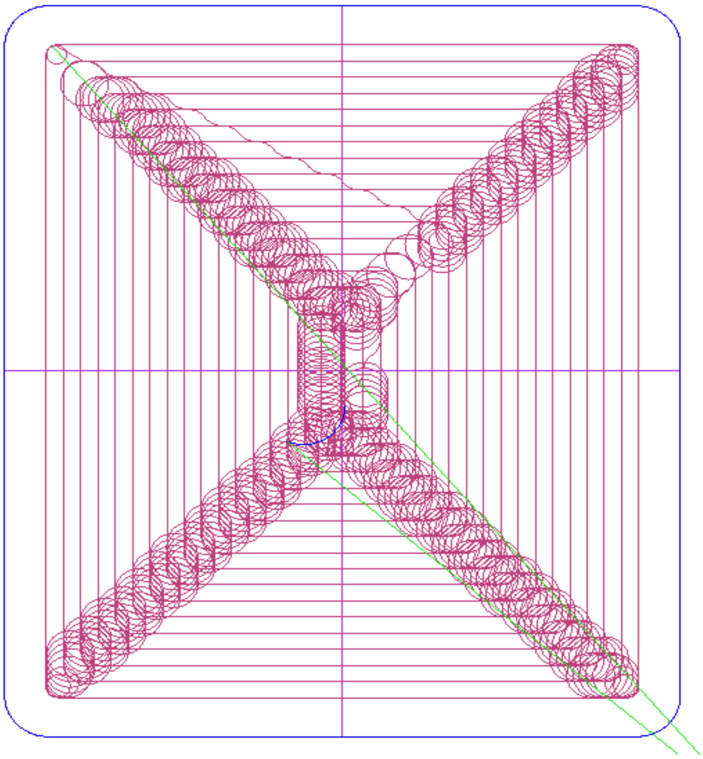


*Notice the f**our different pocket toolpaths**. The **Upper left** pocket (as seen above) is using **Profile Smoothing, Trochoidal** milling, a **large stepdown** of the tool, and a **small Stepover** of the tool.*

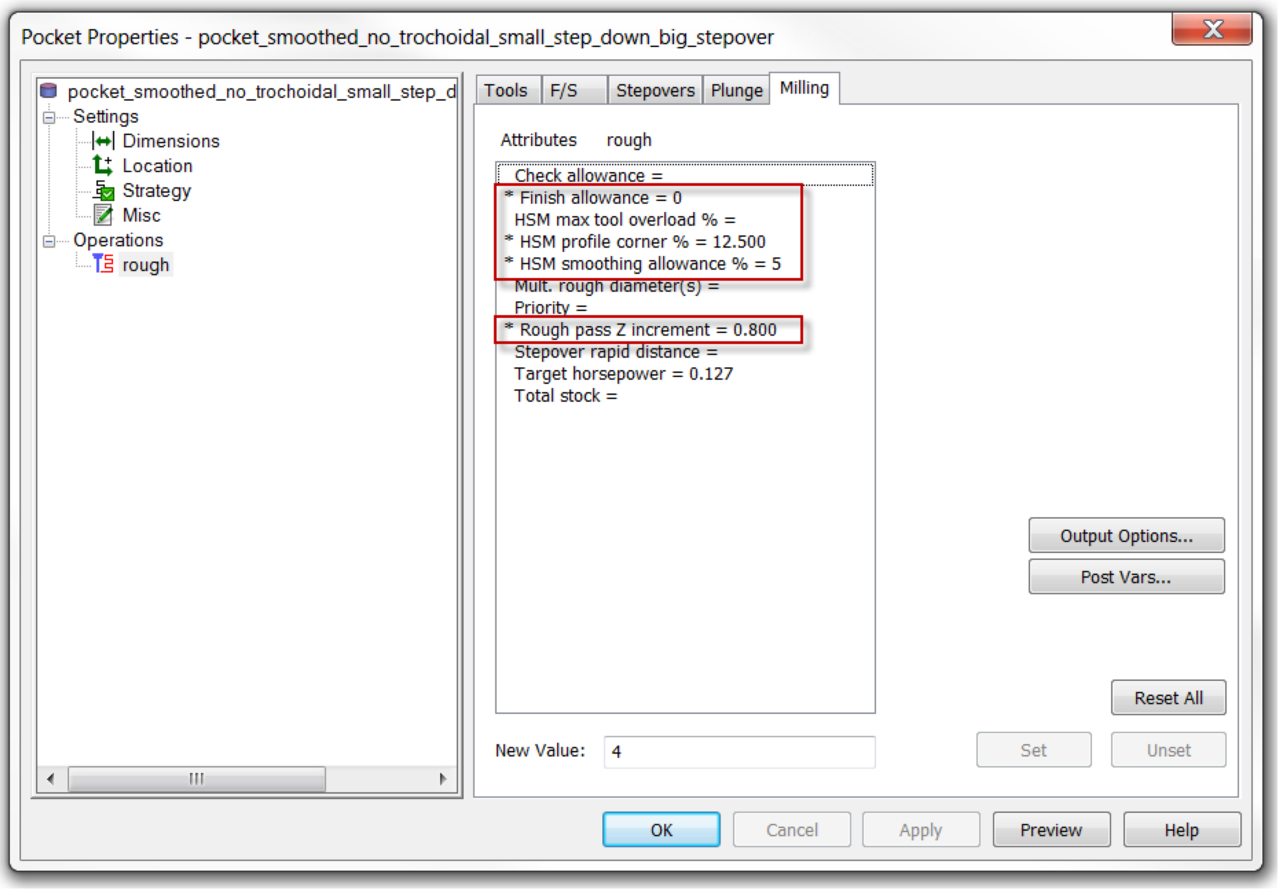


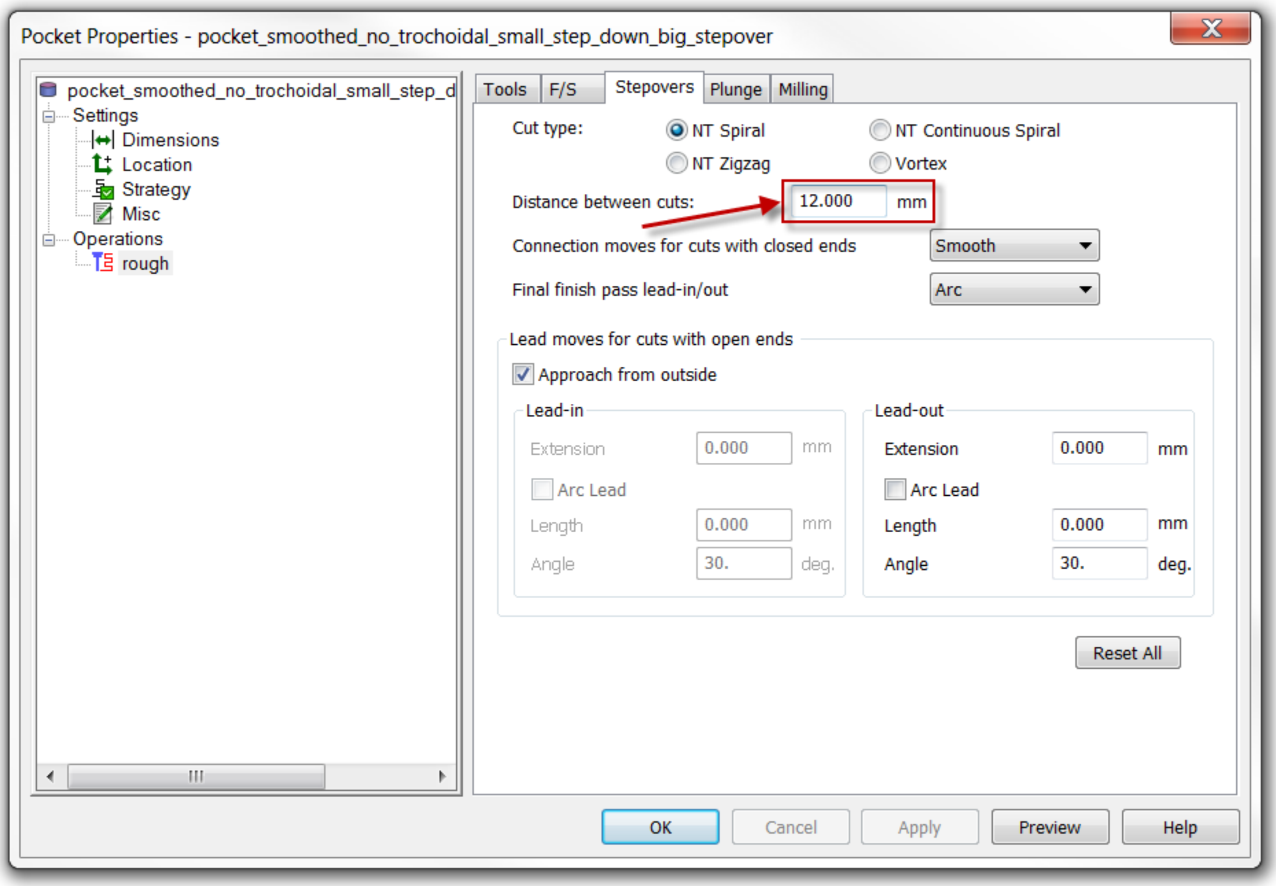


See results on the next page.



The **Lower Left** pocket is using **Profile Smoothing, no Trochoidal** milling, and a **small stepdown** of the tool, and a **big Stepover** of the tool.





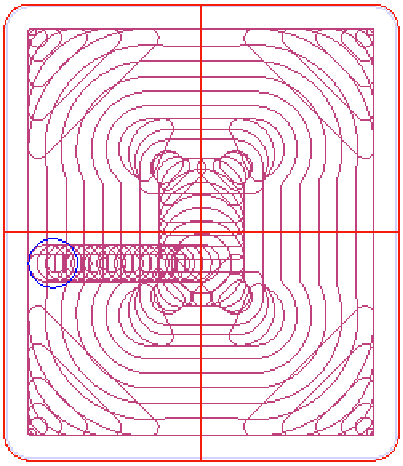
3 From the **Results window** open the **Details tab** and look at the **Operation list**. Looking at ***Op1 and Op2 for Estimated machine times*** you should see similar results as shown below.

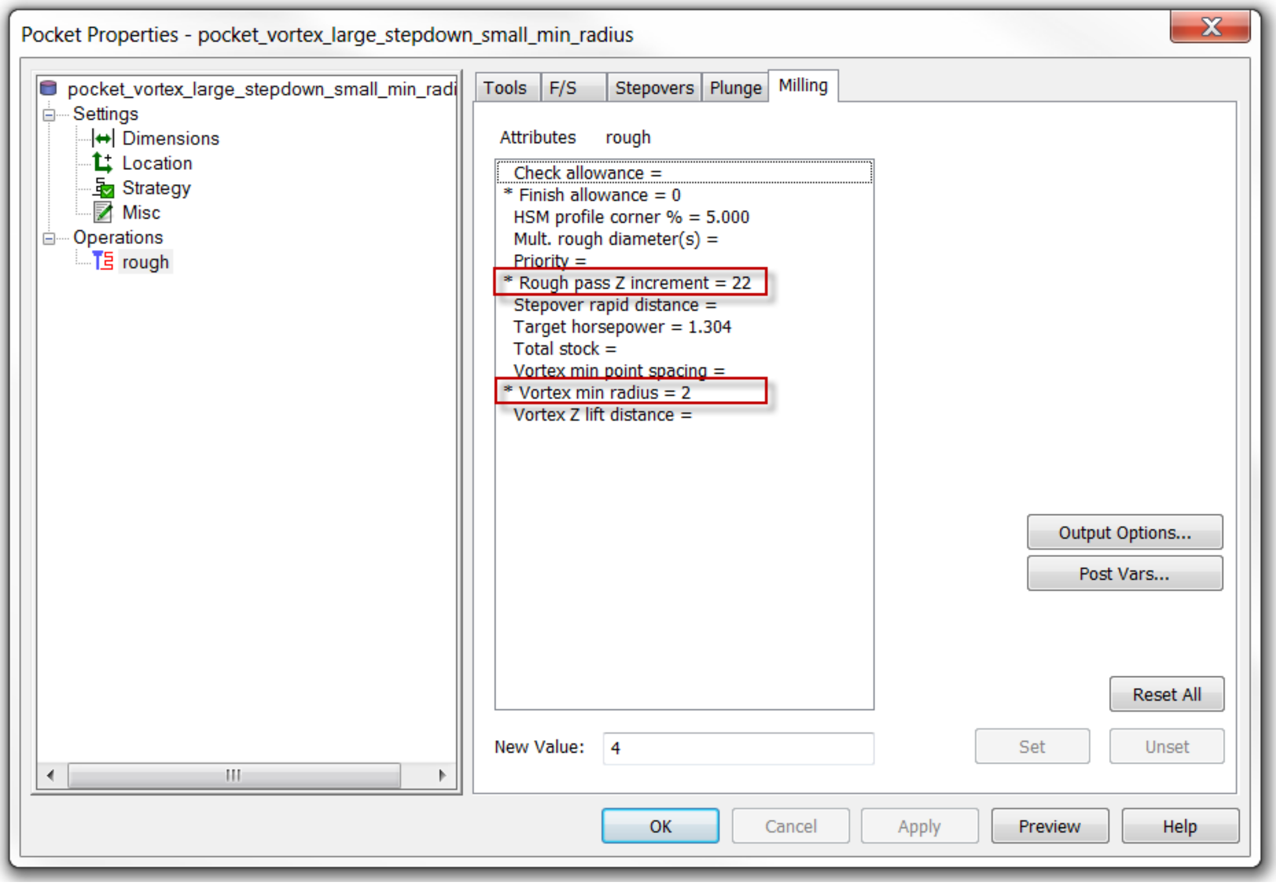
Op: 1 pocket_profile_smoothing_trochoidal_big_stepdown_small_stepover (rough1)
F/S: 2918 RPM, 1109 MMPM (0.190 MMPT)
Tool: #1 (endmillM1050:reg, 10.500 mm)
Depth: 21.500 mm
Other: Stepover: 2.400 mm
Time: 10:48.0
Power: 0.78 (est. 0.78) kW

Op: 2 pocket_smoothed_no_trochoidal_small_step_down_big_stepover (rough1)
F/S: 2918 RPM, 1109 MMPM (0.190 MMPT)
Tool: #1 (endmillM1050:reg, 10.500 mm)
Depth: 21.500 mm (in 27 steps, 0.796 mm each)
Other: Stepover: 12.000 mm
Time: 31:20.7
Power: 0.13 (est. 0.13) kW

*Now let's take a look at the **Vortex** toolpaths to see what improvement can be seen by implementing this strategy.*

*The Upper Right pocket is using **Vortex** with a **large Stepdown** and a **small min radius.***

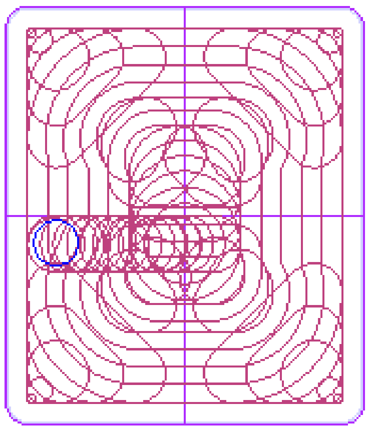


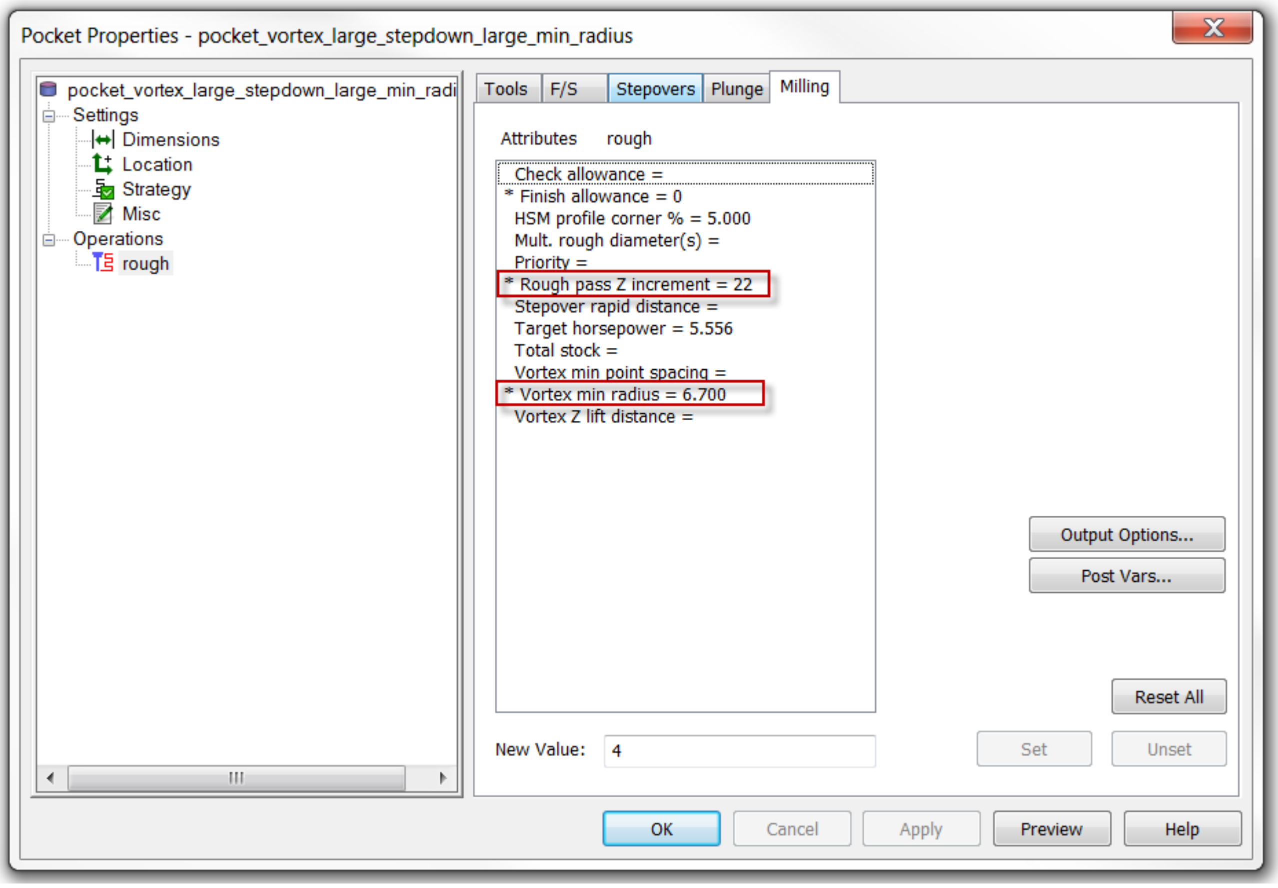


4 From the **Results window** open the **Details tab** and look at the **Operation list**. Looking at **Op3 *Estimated machine time*** you should see similar results as shown on the next page. As you can see this is a significant time saving.

Op: 3 pocket_vortex_large_stepdown_small_min_radius (rough1)
F/S: 2918 RPM, 1109 MMPM (0.190 MMPT)
Tool: #1 (endmillM1050:reg, 10.500 mm)
Depth: 21.500 mm
Other: Stepover: 4.000 mm
Time: 5:11.4
Power: 1.30 (est. 1.30) kW

The Lower Right pocket is using ***Vortex*** *with a* ***large Stepdown*** *and a* ***large min radius.***





From the **Results window** open the **Details tab** and look at the **Operation list**. Looking at **Op4 *Estimated machine time*** you should see similar results as shown below.

Op: 4 pocket_vortex_large_stepdown_large_min_radius (rough1)
F/S: 14589 RPM, 3939 MMPM (0.135 MMPT)
Tool: #1 (endmillM1050:reg, 10.500 mm)
Depth: 21.500 mm
Other: Stepover: 4.800 mm
Time: 3:57.3
Power: 5.56 (est. 5.56) kW

What are the benefits to using a Vortex toolpath?

Time savings/efficiency

- As you can see above there is a time **saving of around 60%** from the trochoidal strategies and almost another minute is saved in the second **Vortex** compared to the first **Vortex** toolpath.
- **Vortex** toolpaths are calculated to follow the shape of the part by creating offsets from the part profile. This minimizes air moves for more efficient machining of both pockets and cores.

Tool life

- Because **Vortex** toolpaths have a controlled engagement angle, tools will never be overloaded and so will achieve the maximum tool life. Shock loading caused by changes in the contact angle is eliminated, preventing chipping of the flutes.
- The stability of the cutting conditions gives constant edge temperatures prolonging the life of the tool coating and protecting against heat damage to the surface of the part.
- Remember, **Vortex** modifies the toolpath so **the tool engagement angle is never exceeded** and the machine tool achieves the specified feed rate. The only time the machine tool doesn't run at the specified feed rate is when the model geometry (a slot or corner) is smaller than the smallest radius that the machine can run at full speed. This should result in better tool life, and thus cost to machine a given part.

Maximizes your Material Removal Rate

- **Vortex** addresses the fundamental problem of conventional roughing strategies that can only achieve the optimum cutting conditions during a straight-line cut. Unlike other high speed roughing techniques that aim to maintain a constant theoretical metal removal rate, the **Vortex** strategy produces toolpaths with a controlled engagement angle for the complete operation.
- This allows you to achieve the optimum cutting conditions that would normally be possible only for the straight-line moves across the entire toolpath As a result, you can reduce machining times by up to 60%, with more consistent material removal rates and feed rate.

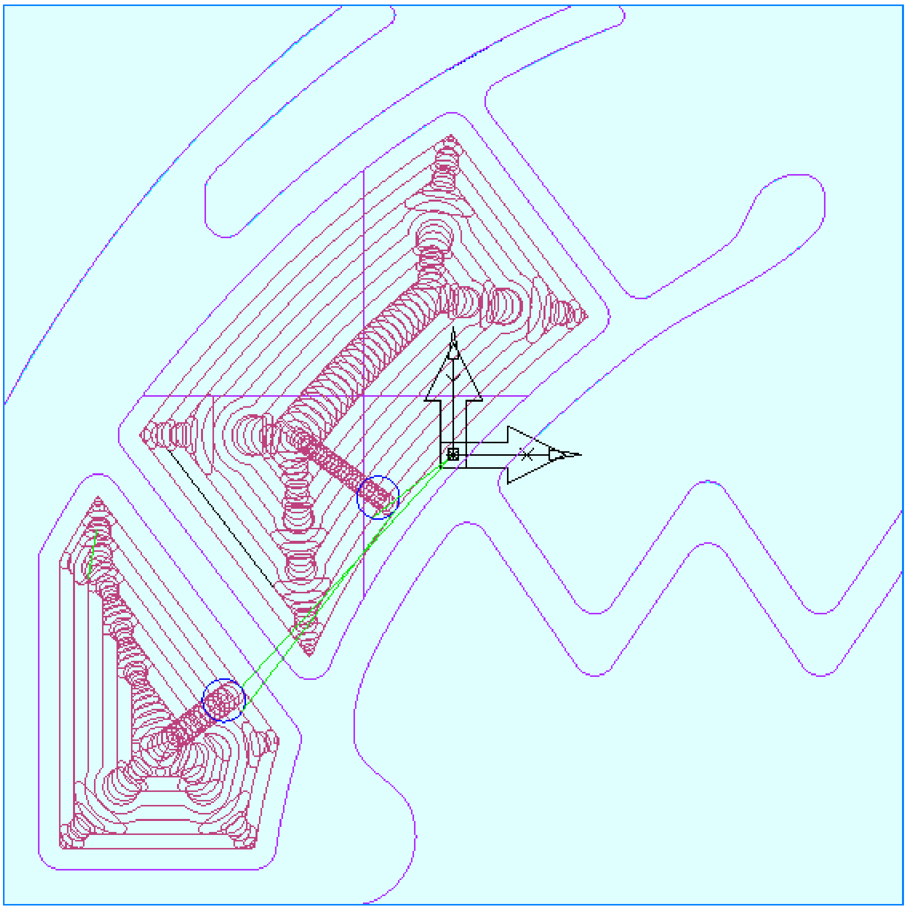
Vortex Example #3

Important considerations when creating a Vortex toolpath

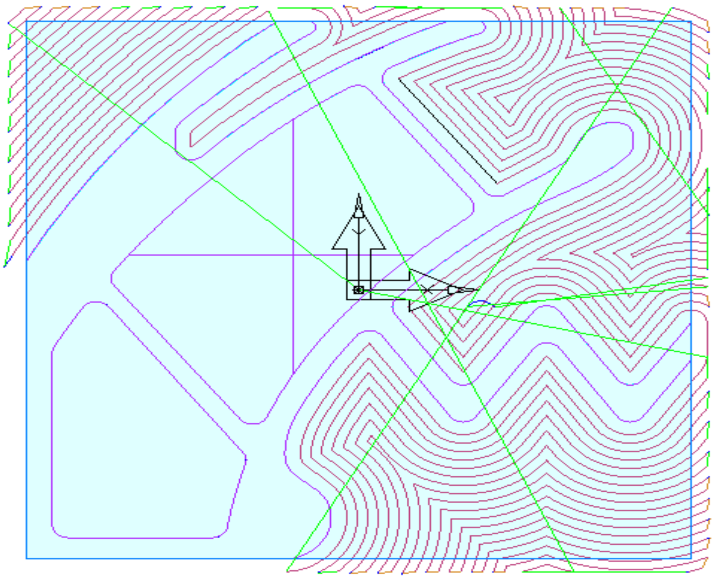
1 Open Vortex example3

2 Run a Centerline simulation

*In the image below you will see the pocket feature using **Vortex**. Notice the toolpath that we have now become accustomed to seeing.*

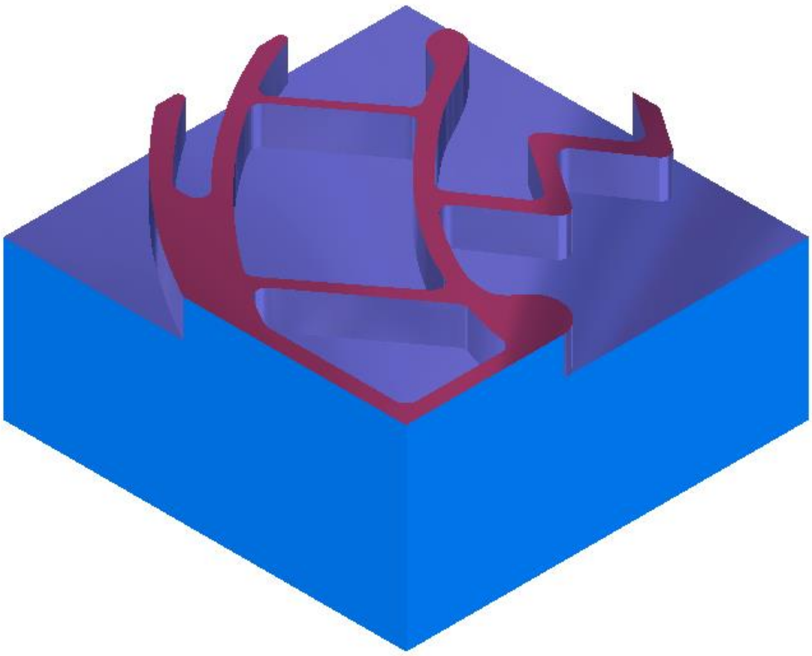


*Now look at the first side feature using **Vortex** and the toolpath as we have seen before.*



IMPORTANT: You can use Vortex to machine a Side feature only if the curve is closed. If the curve is open and you select a Vortex strategy, NT Spiral will be used instead.

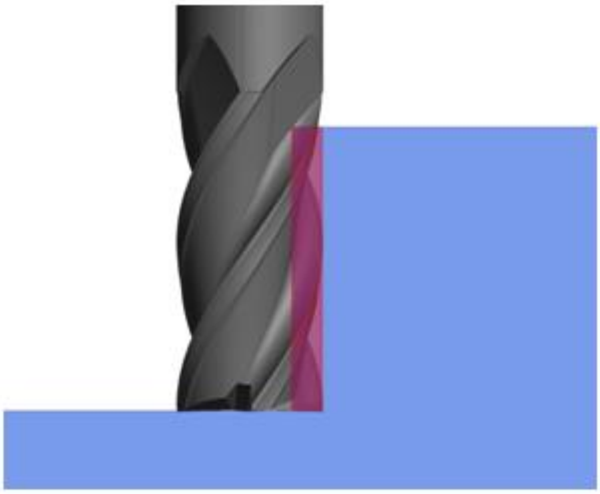
*Please Note: You will see in the remaining three side features, as shown above, that there is no **Vortex** toolpath generated even though it has been set in all three of these features. It is important that you remember this and set all your Feeds and Speeds, Stepovers, and Depth of Cut as you would have while not using **Vortex**. The alternative is to close the curve. Ask instructor if you are unsure.*



Vortex Example #4 (3D HSM)

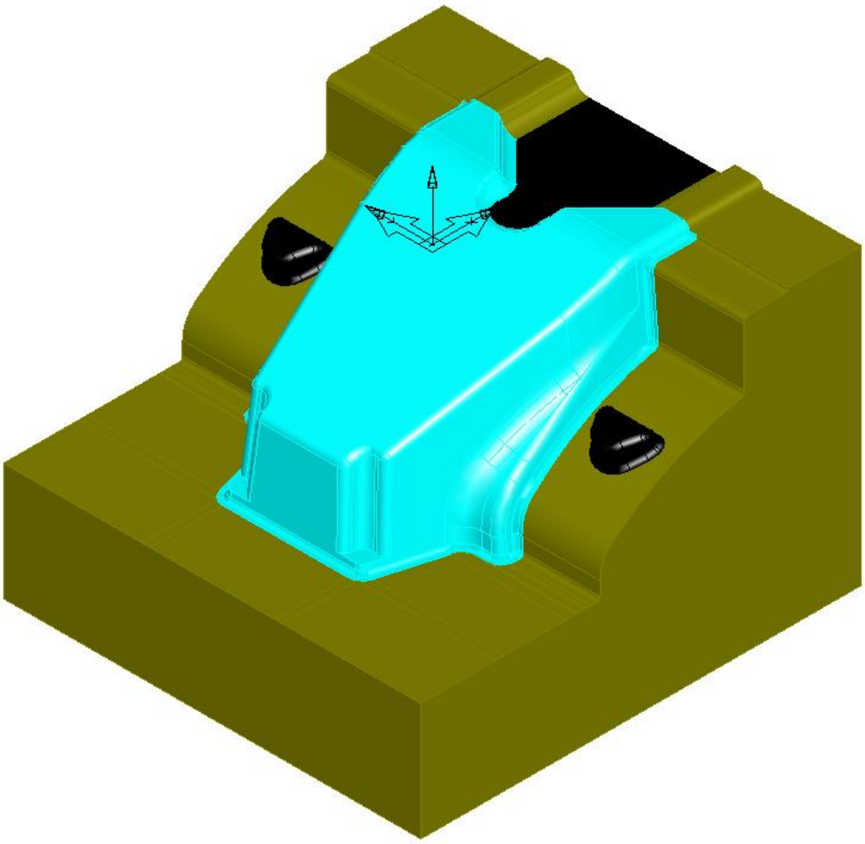
This example shows you how to combine Vortex machining with step cutting to rapidly remove material.

*Because **Vortex** machining cuts with the side of the tool, it is designed for solid carbide tools, but there may be other types of tools suitable for Vortex. These tools work best when taking deep cuts with a relatively small stepover.*

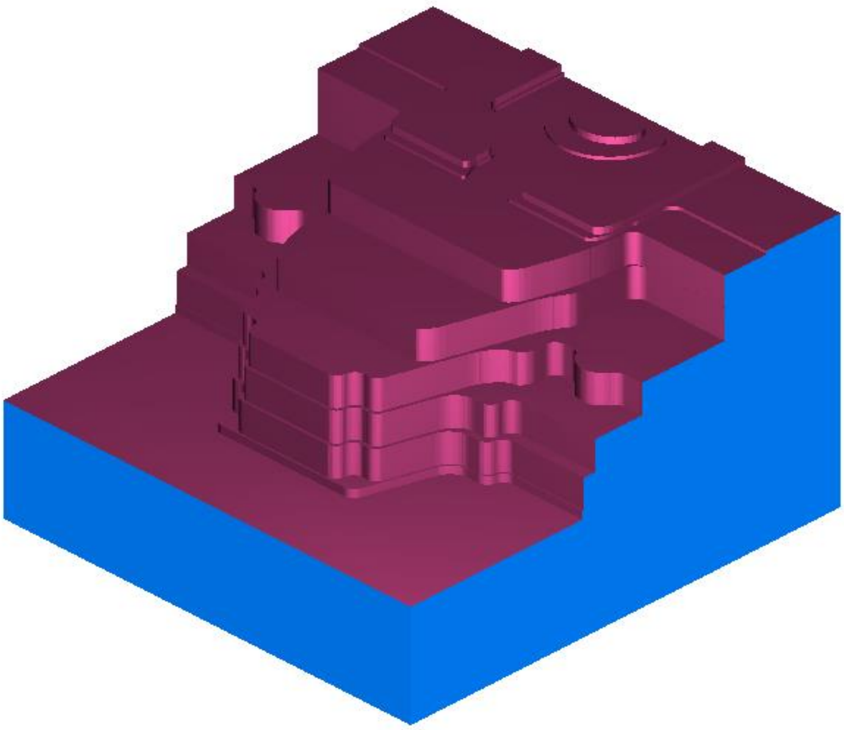


*To machine effectively when taking large depths of cut, you must ensure the tool engagement angle never exceeds the specified value. This eliminates excessive tool load and all full-width cuts. **FeatureCAM** achieves this by introducing trochoidal moves to prevent the tool from exceeding the maximum tool engagement value.*

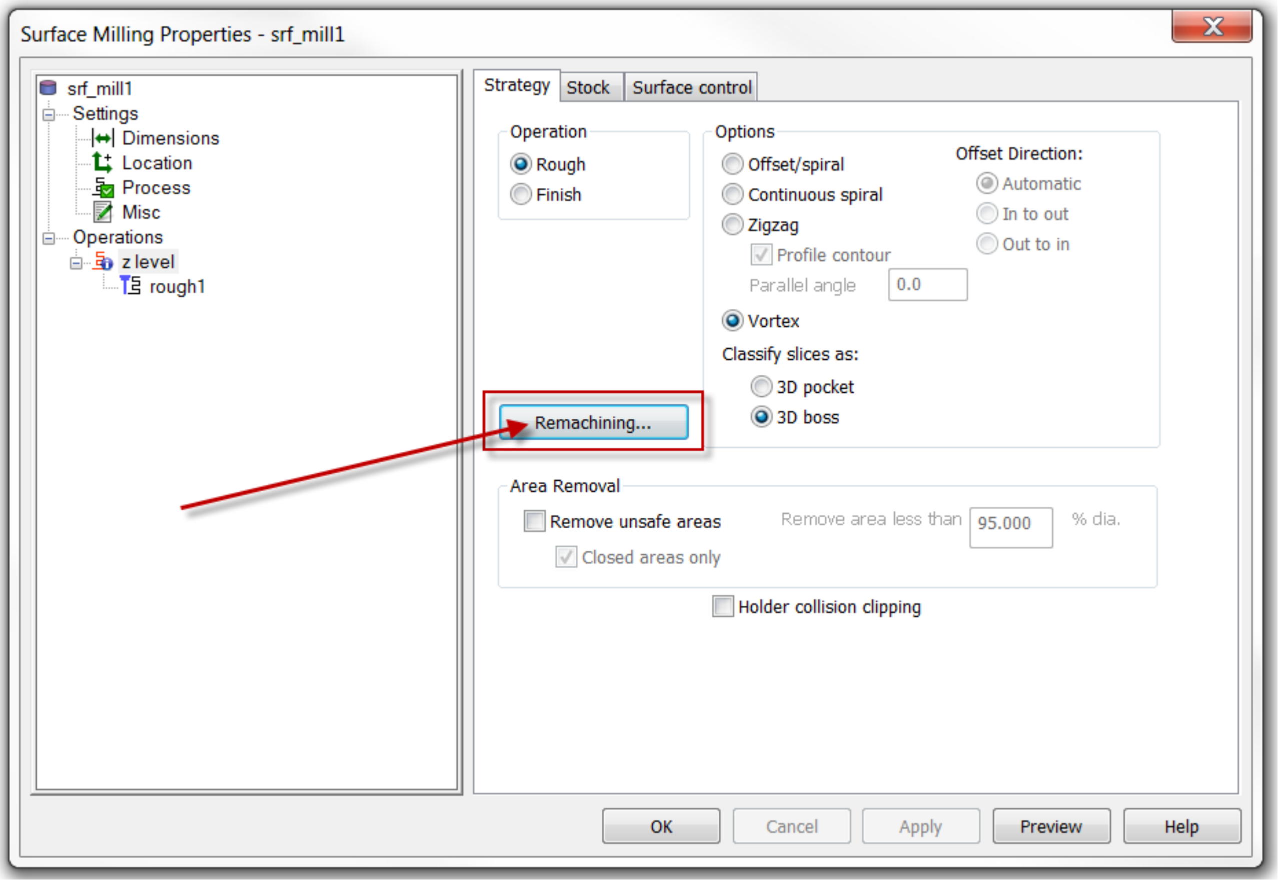
The following images are just for display purposes. Using a 3D Model:-

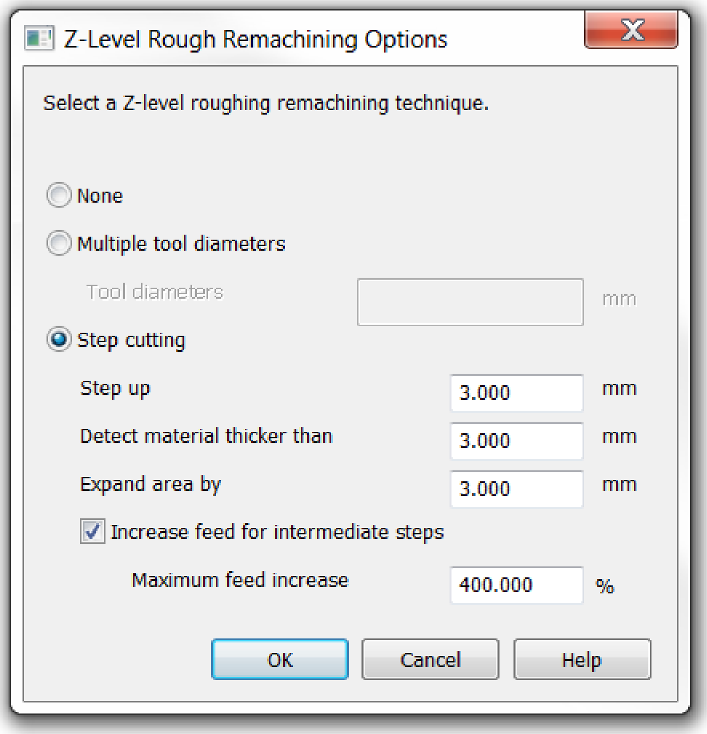


If you create a Vortex toolpath without step cutting it is possible to remove vast quantities of material quickly, but leaves large terraces of un-machined stock on the part as shown below.

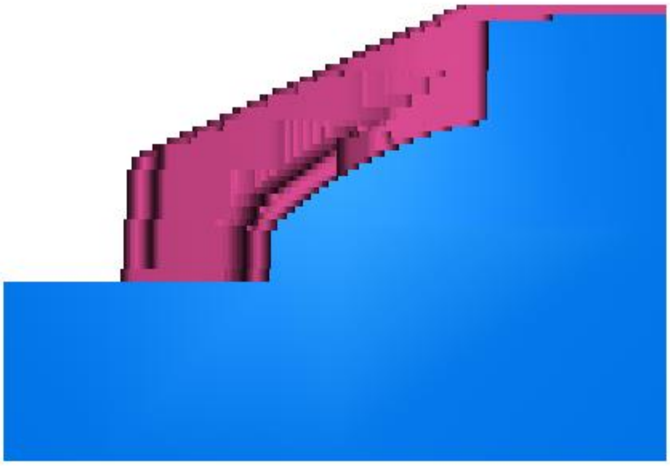


You can minimize the size of these terraces using the ***Step cutting*** *options in the Remachining settings of the feature.*



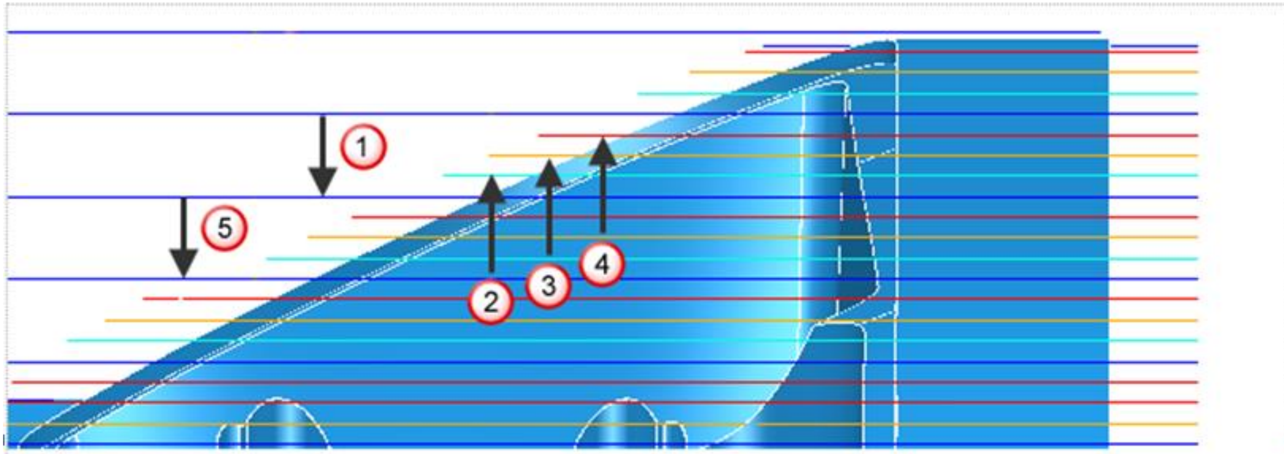


It also machines more of the excess stock.



To better understand these settings review the following information on the next page.

***Step Cutting**: This option machines the terraces that would otherwise remain during a Z-level rough toolpath with the same tool in the same toolpath. The terraces are machined from bottom to top. **FeatureCAM** increases the feed rate of the intermediate slices as the depth of cut gets progressively smaller, which enables a constant volume removal rate.*



(1) The first pass is a main slice.

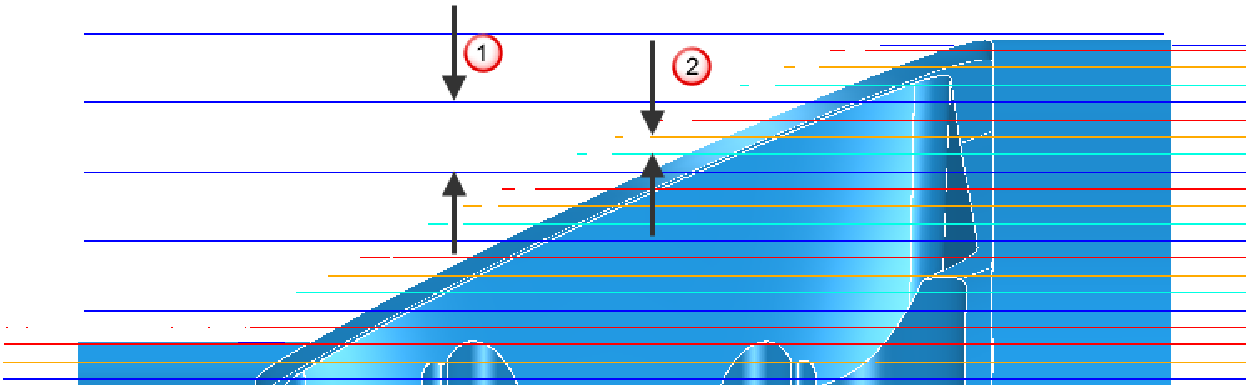
(2) The next pass is the lowest step-cutting slice.

(3) The next pass is the next to lowest step cutting slice and so on up the part (shown by (4)) until the previous main slice is reached.

(5) The next main slice.

The main slices are cut from top to bottom as normal. The intermediate slices are cut from bottom to top. This is possible because the depth of cut is always less than the depth of cut (or **Stepdown**) for the main slice. This is the most efficient way of cutting the part as it reduces the number of passes and increases the depth of cut of each pass.

Step up: Enter the distance between intermediate cutting levels.



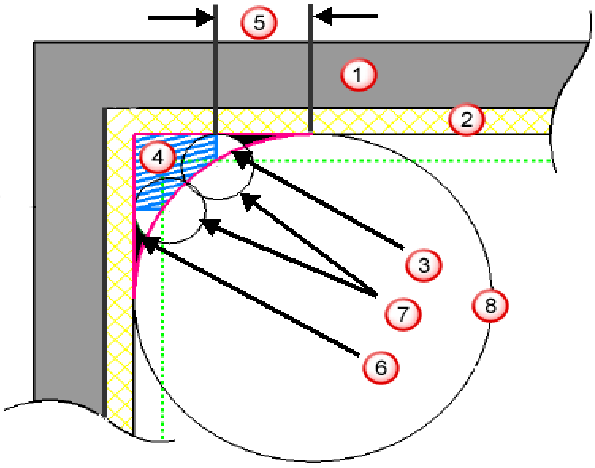
1 Stepdown of the main slice.

2 Stepup of the intermediate slice.

The main slice clears the majority of the material. The intermediate slices remove the terraces remaining after the main slice.

Detect material thicker than: Enter a threshold value. FeatureCAM ignores rest material that is thinner than the specificed threshold.

Expand area by: Enter the distance by which to expand the rest areas, measured along the surface. Use with the **Detect material thicker than** value to reduce the areas to be machined to the details (for examples, corners), and then to offset these areas slightly to ensure that all the detail (for example, on the corners) are machined.



1 — model

2 — thickness

3 — true rest material (outlined in pink)

4 — actual rest material detected (blue hatched area)

5 — amount you need to expand the area by to include all the rest material

6 — undetected material (black area)

7 — tool

8 — reference tool

Use the **Expand area by** option to increase the rest area (the blue hatched area) and eliminate the undetected area (the black area).

Increase feed for intermediate steps *— Select to increase the **Feed rate** of the intermediate slices. Because the stepdown of the intermediate slices is less than that of the main slice, you can increase the feed rate of the intermediate slices while maintaining the tool load. Each intermediate slice can have an increasing feed rate as the depth of cut gets progressively smaller.*

Maximum feed increase % *— Enter the maximum allowable cutting feed rate for the intermediate slices as a percentage of the normal feed rate. This value must be larger than 100%. A value of **300** means the cutting feed rate of the intermediate slices can be up to three times faster than for the main slices.*

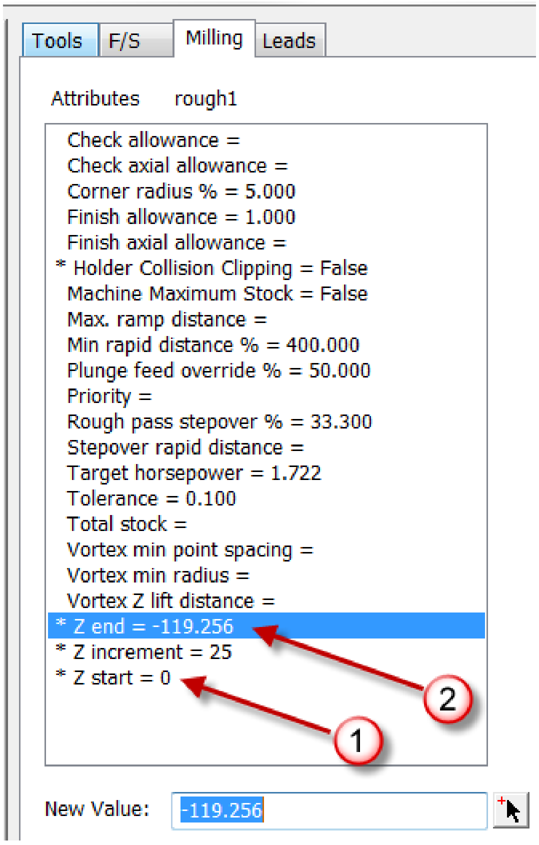
Z Start – Z End

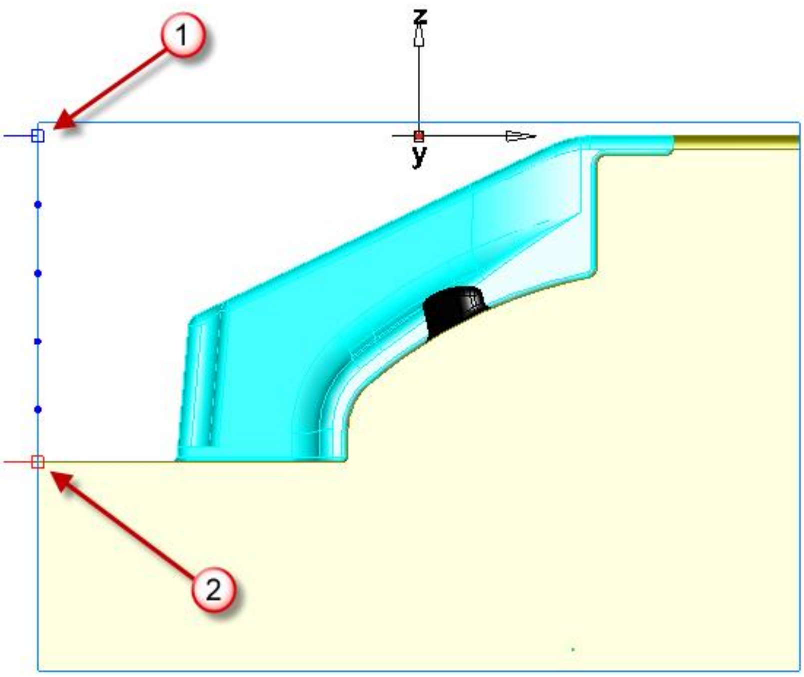
Z start — Enter the distance along the Z axis where the milling operation starts. You can use this to save time if the stock material has already been machined away in an earlier operation. Use the Pick Z location Icon to pick a location on the solid model.

Z End — Enter the distance along the Z axis where the milling operation finishes. You can use this when using short series endmill which are more rigid and can be run at higher feed rates.

Once the endmill length limit has been achieved then a longer endmill can be used and the Z end would be the starting point for the longer endmill with a new Z end point added. This process would then be repeated until the full depth of the component has been fully machined.

Check surface can also be used to prevent a tool going past a selected face.


Tools
F/S
Milling
Leads
Attributes rough1
Check allowance =
Check axial allowance =
Corner radius % = 5.000
Finish allowance = 1.000
Finish axial allowance =
* Holder Collision Clipping = False
Machine Maximum Stock = False
Max. ramp distance =
Min rapid distance % = 400.000
Plunge feed override % = 50.000
Priority =
Rough pass stepover % = 33.300
Stepover rapid distance =
Target horsepower = 1.722
Tolerance = 0.100
Total stock =
Vortex min point spacing =
Vortex min radius =
Vortex Z lift distance =
* Z end = -119.256
* Z increment = 25
* Z start = 0
2
1
New Value: -119.256


1
z
y
2