



Advanced  
Manufacturing  
Solutions

# FeatureCAM 2012

Training Course

---

**FeatureCAM 2012**

# **Training Course**

**FeatureMILL 3D Roughing**



## **FeatureCAM**

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

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

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

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

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

### **Acknowledgements**

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

# Contents

<b>FeatureCAM overview</b>	<b>7</b>
<b>Roughing Basics</b>	<b>9</b>
Introduction.....	9
Roughing Basics .....	9
Finish Allowance & Tolerance.....	12
<b>Roughing Z Level Zigzag</b>	<b>13</b>
Introduction .....	13
Setting Up .....	14
Creating a roughing Feature .....	16
Stepdown – Z Increment .....	20
Profile Pass .....	21
Stepover % .....	22
Arc Line Approximation .....	24
Parallel Angle .....	25
<b>Rough – Z Level Offset/Spiral</b>	<b>27</b>
Introduction.....	27
Multiple Rough Diameters.....	29
Z End.....	34
<b>Parallel Roughing</b>	<b>35</b>
Introduction.....	35
<b>Plunge Roughing</b>	<b>39</b>
Introduction.....	39
<b>Stock Types, Total Stock &amp; Boundaries</b>	<b>44</b>
Introduction.....	44
N-Sided Stock .....	48
Stock Curve .....	50
Total Stock .....	52
Stock Solid .....	53
Pocket/Boss .....	56
Use of Boundary Curves.....	59

<b>Stepovers &amp; Leads</b>	<b>62</b>
Introduction.....	62
Stepover types .....	62
Ramping moves .....	64
Ramp angle and Maximum ramp distance .....	64
Plunging to Pre-drilled holes.....	68
 <b>Roughing Flats</b>	 <b>69</b>
Introduction.....	69
 <b>High Speed and High Efficiency Roughing</b>	 <b>73</b>
Introduction.....	73
High Speed Roughing.....	74
High Efficiency Roughing .....	77
Limiting tool overload .....	78
 <b>Feedrate Optimization</b>	 <b>81</b>



# FeatureCAM overview

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

FeatureCAM includes five stand-alone modules:

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



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

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

The following add-on modules are also available:

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

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

---

## Why creating toolpaths is so fast

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

Traditional CAM systems are *operations-based* and require you to program every operation, one at a time, to create your part.

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

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



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



---

# Roughing Basics

---

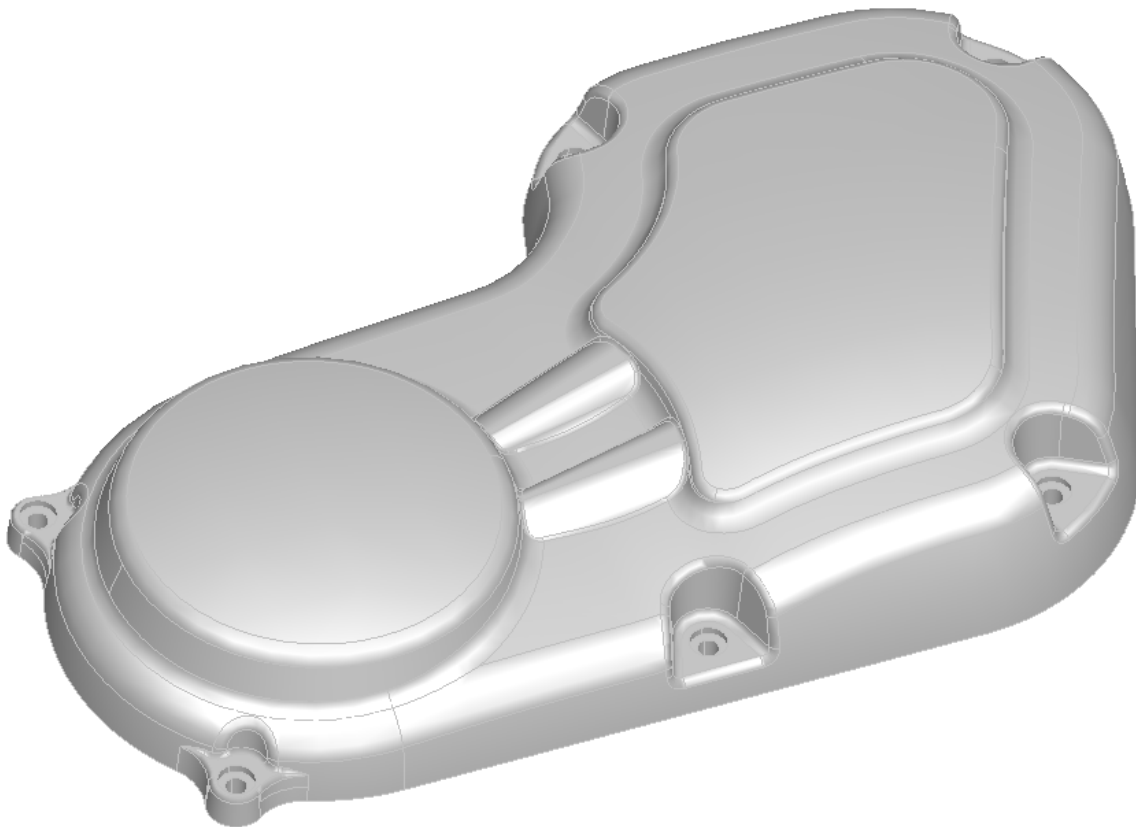
## Introduction

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 prepare 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**, **Parallel Roughing** and **Plunge Roughing**. We shall also consider different stock types and finally take a look at high speed and high efficiency roughing techniques.

---

## Roughing Basics

In this section we will be rough machining the engine casing pictured below. The part has an assortment of shallow and steep areas, areas with varying radius of curvature and has an irregular outline.

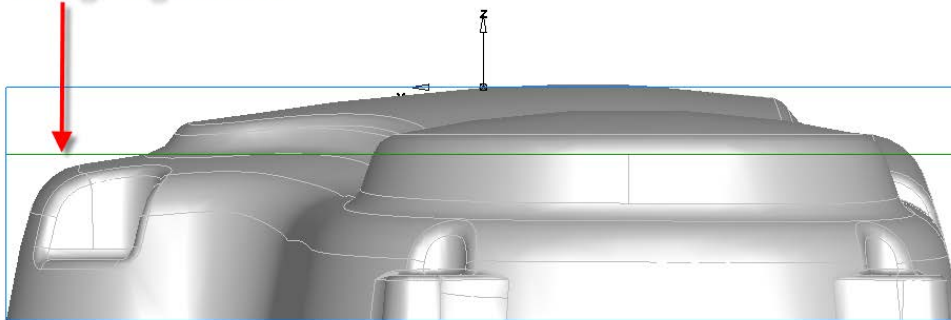


Each of these properties of the part affects the roughing process, selection of tooling and so on. By the end of this module you should know how to make choices to enable FeatureCAM to rough the part efficiently leaving the minimum amount of material on the part in the shortest possible time.

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.

In the picture below a single roughing **Z level** or **Slice** is represented by a horizontal plane intersecting the part. In practice multiple Z levels will be machined successively from the top of the stock downwards at regular increments to remove the excess material.

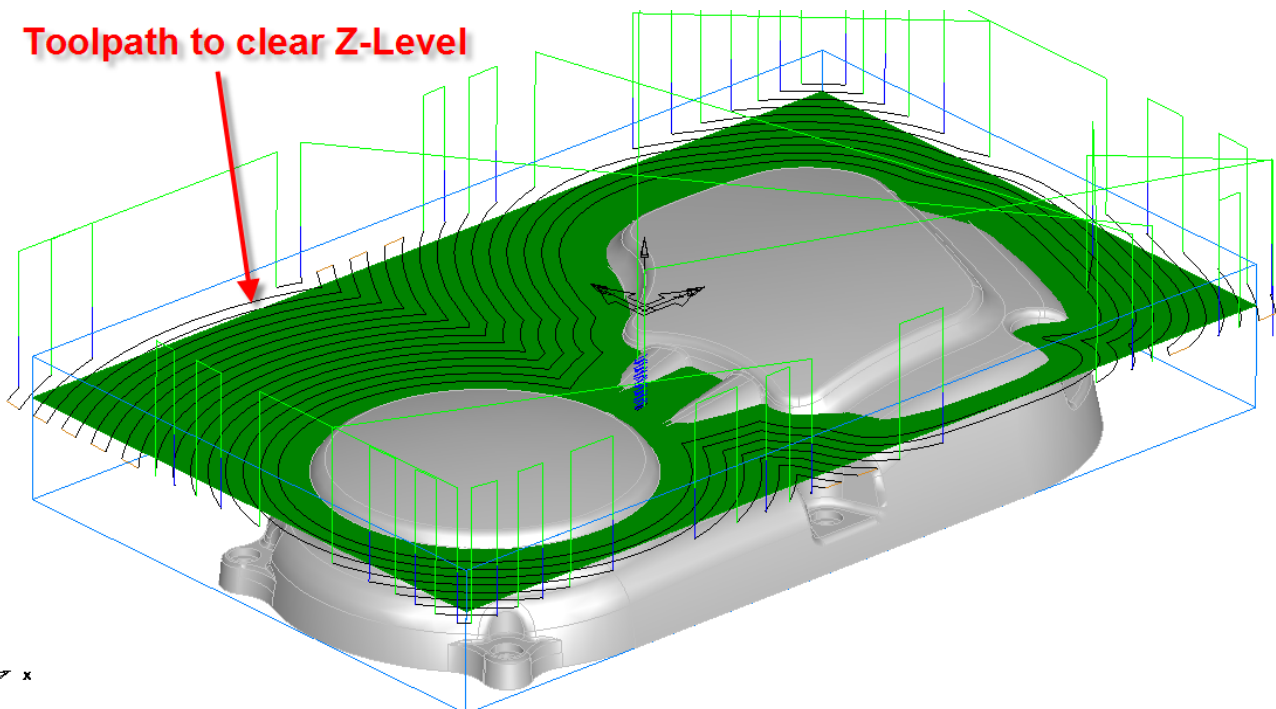
**Roughing Z Level**



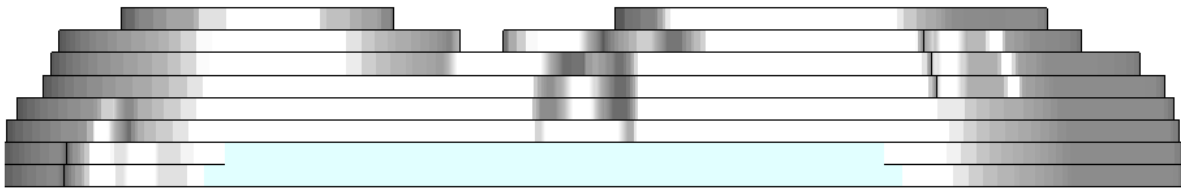
Another name for Z level roughing is “**Waterline roughing**”, the profiles to be cleared may be visualized as the waterline of the part when it is partially submerged in a tank of water. As the water is drained from the tank each level produces a distinct profile through the part.

The image below shows a toolpath clearing a single Z level of the part. The tool is making a series of passes starting at the outside of the block and gradually stepping inwards in a horizontal direction until it meets the waterline of the part with an allowance for later finishing processes.

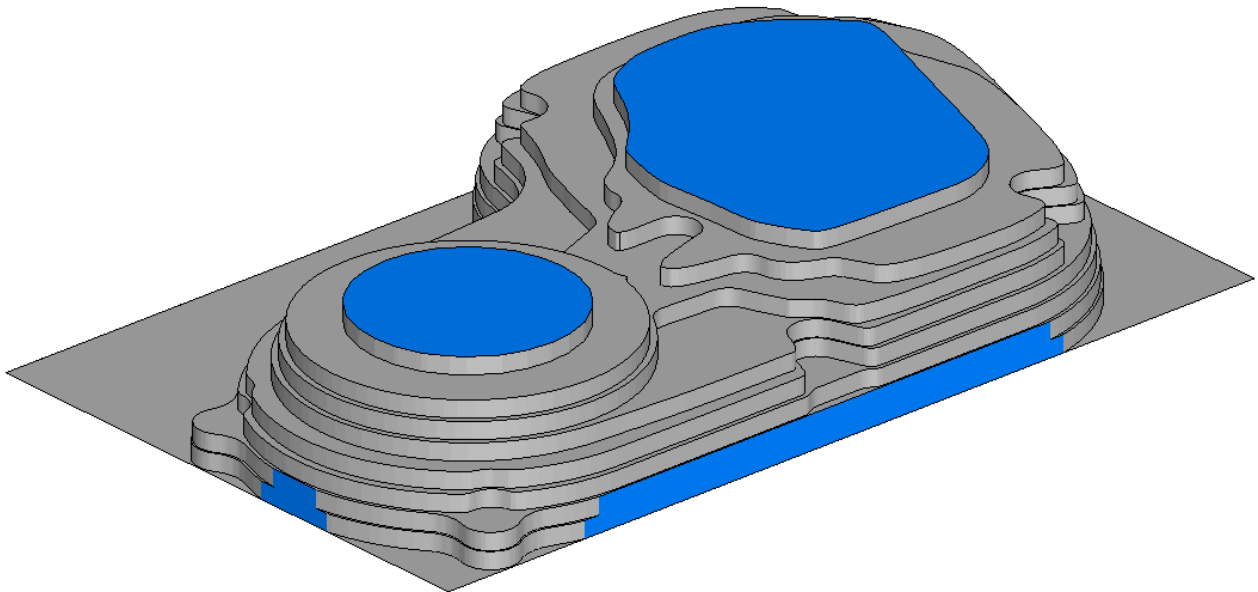
**Toolpath to clear Z-Level**



The part is roughed by machining a series of Z levels resulting in a stepped form where the majority of material has been cleared out around the model as shown in the pictures below.



A side view of the roughed part showing the roughing Z levels.



An Isometric view of the roughed part.

Depending upon the tooling selected, machining attributes and the strategy chosen the part may require subsequent rest roughing and semi-finishing processes before finishing can be carried out.

For some components it is desirable to use a secondary area clearance strategy. This is done using the **Multiple Rough** option where a series of tools are specified, each reducing in size from the previous. This will locally remove pockets of excess material inaccessible to the previous tool. This will reduce the degree of tool overload and provide a more consistent material removal rate for any subsequent finishing operations.

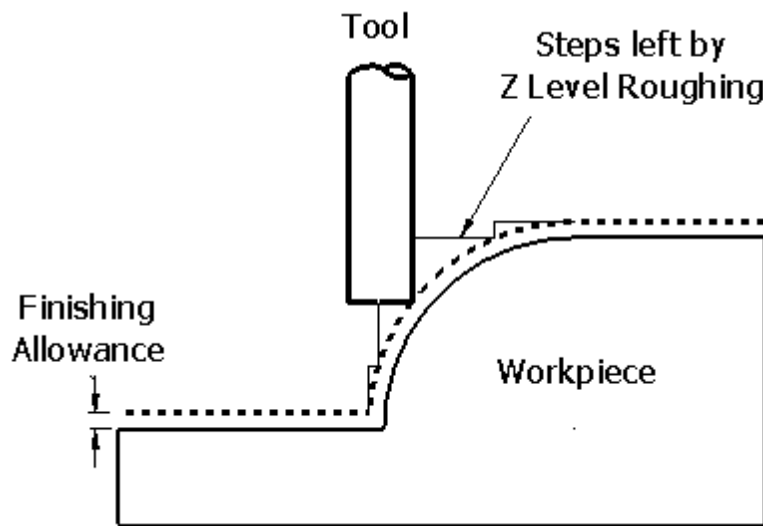
If the original material is in the form of a casting or fabrication then it may be necessary to use a different form of stock so that the roughing toolpath does not end up “cutting air”.

For now we will only consider roughing a part from a simple rectangular stock. In the following example we shall import and orient the casing model and apply a simple **Offset Area Clearance** strategy. First we need to know about the basic machining attributes that affect the generation of rough machining toolpaths.

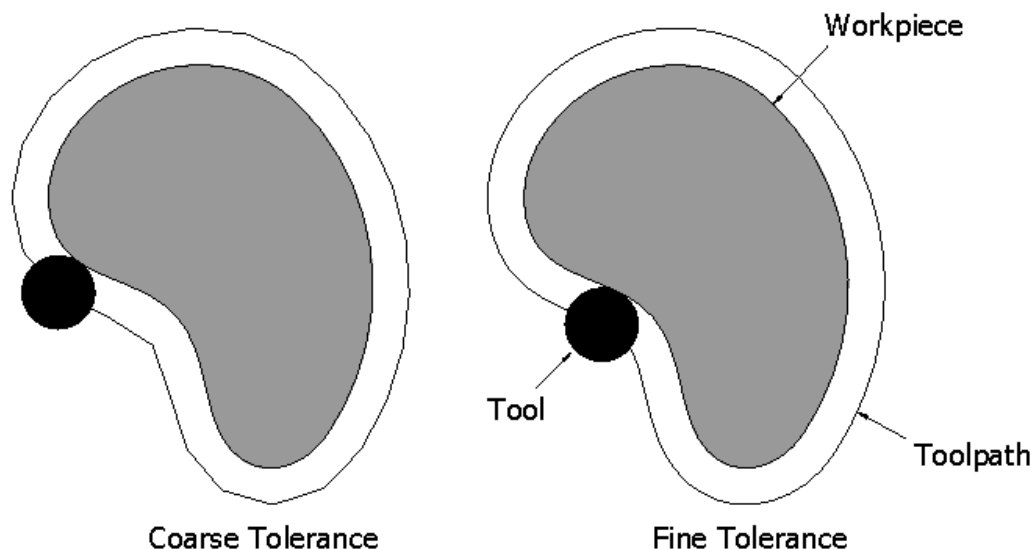
## Basic Roughing Attributes – Finish Allowance & Tolerance

Suitable values are required to control the accuracy and amount of excess material to be left on a component by a toolpath. The parameters used for this purpose are the **Finish Allowance** and **Tolerance**.

The **Finish Allowance** specifies the amount of extra material remaining on the work-piece after machining. It should be noted that this will be the **minimum** amount of material left on the part after roughing as the roughing Z level steps are superimposed upon this allowance.



The **Tolerance** controls the accuracy to which the toolpath follows the shape of the work-piece. The illustration shows an individual roughing Z level pass viewed from above. With a coarse tolerance the toolpath is more faceted as fewer moves are calculated for each profiling move.



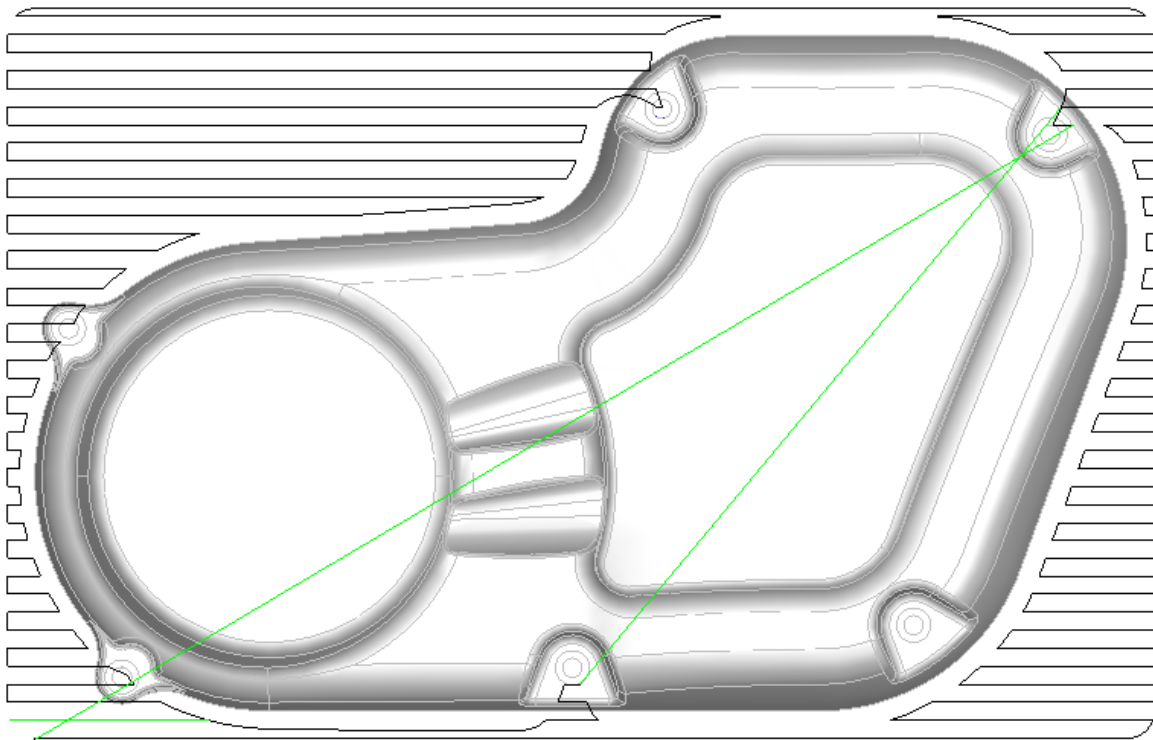
**Note** where the **Finish Allowance** value is greater than 0 it should always be more than the **Tolerance** value otherwise the part may be gouged. If a very fine tolerance is chosen for a roughing operation it may take a long time to calculate the toolpath to an unnecessary accuracy. The optimum Tolerance value will depend upon the user balancing the required accuracy against the time taken to calculate the toolpath.

# Roughing Z Level Zigzag

## Introduction

In this section the part will be roughed using the Z Level **Zigzag** strategy. Zigzag removes the material from the model using a series of parallel passes at each level on the part as shown below. The Zigzag area clearance is also known as Raster area clearance.

In the image below a single Z level of a Zigzag toolpath is shown, the tool is clearing out the material from the level by making a series of parallel passes along the X axis, at the end of each pass it steps over in the Y direction.

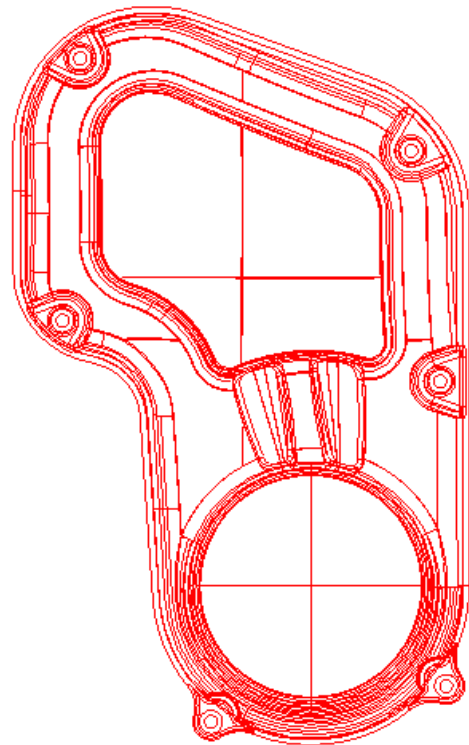
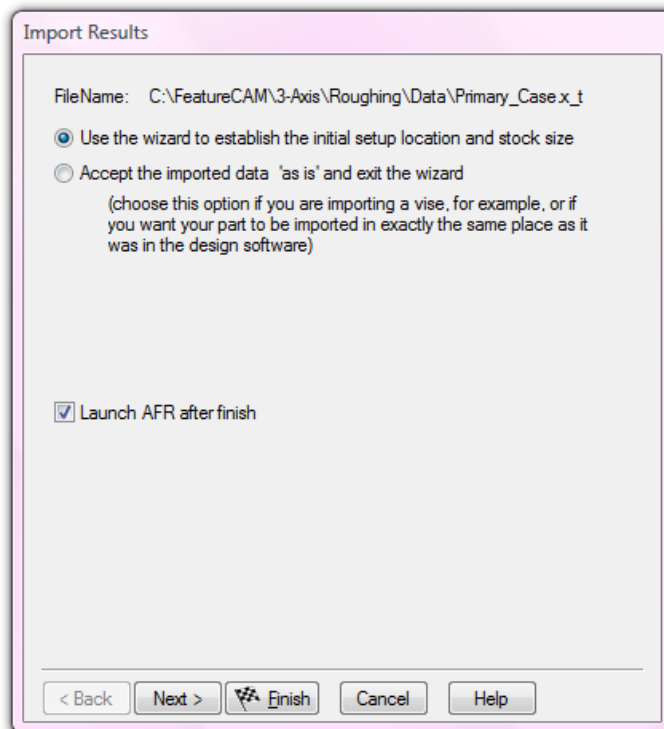


First a model will be imported and positioned using the stock wizard.

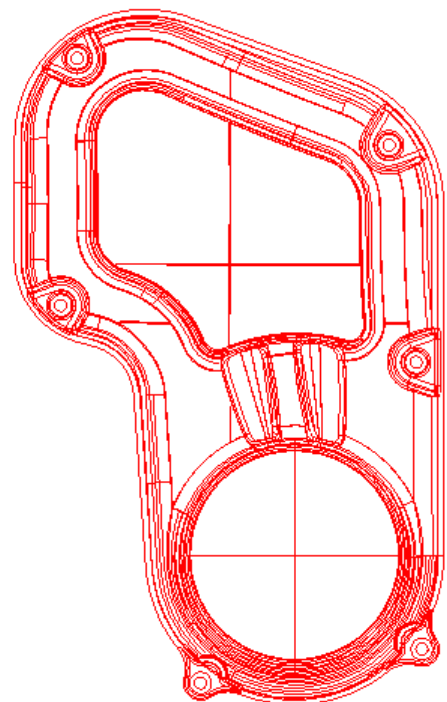
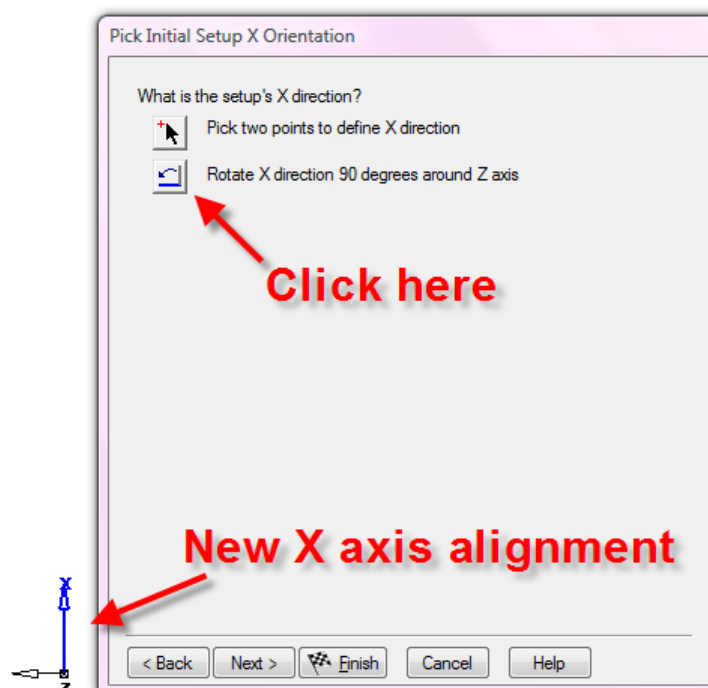
- Open a new **Inch milling document**.
- Press **Finish** on the **Dimensions** form and then click **OK**.
- **Import** the model **Primary\_Case.x\_t**
- On the import form click **Use the wizard to establish the initial setup location and stock size**. **Uncheck** the “Launch AFR after finish” checkbox.

## Setting Up

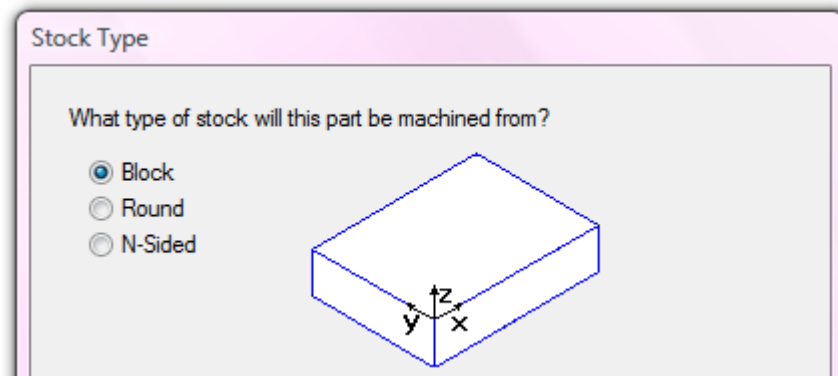
The long axis of the imported part is aligned with the Y axis. We need to turn the part around so that it is aligned with the X axis.



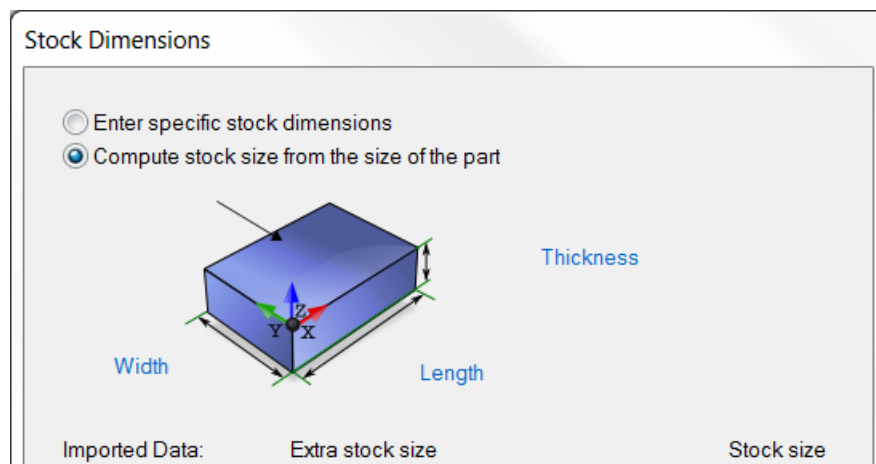
- Click **Next** until you reach the **Pick Initial Setup X Orientation** page.
- Click on **Rotate X direction 90 degrees around Z axis**



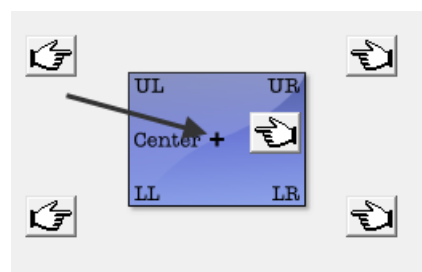
- Click **Next** until you reach the Stock Type page.
- Select a stock type **Block** and click **Next**.



- Check **Compute stock size from the size of the part** and click **Next**.

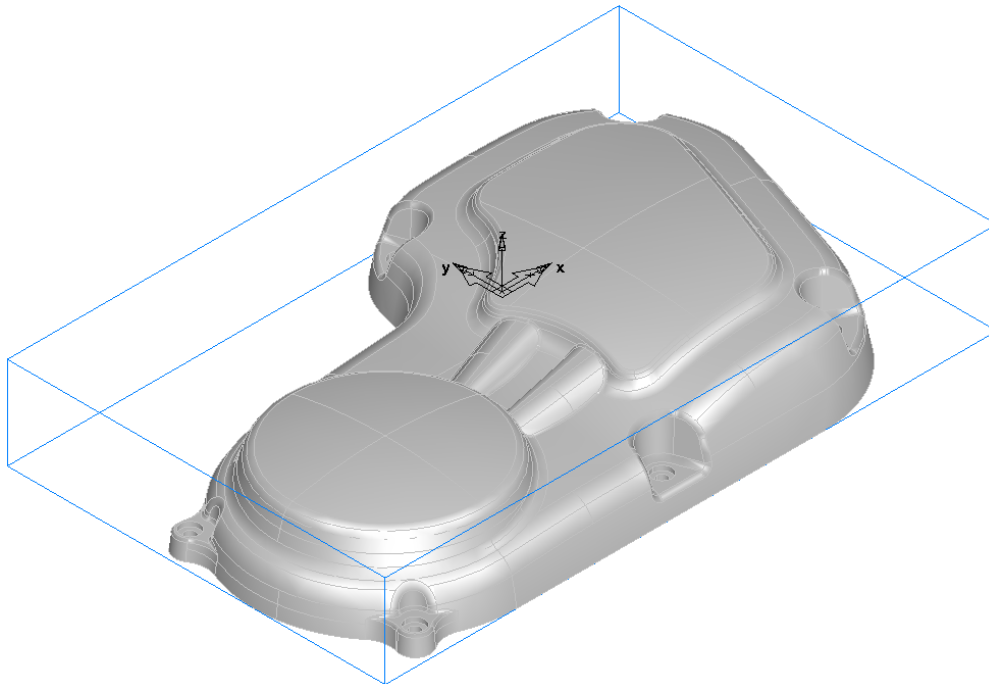


- Select the required setup location (stock center) by clicking on the hand icon indicated



- Click **Finish** and **Cancel**.
- Select an **Isometric** view.

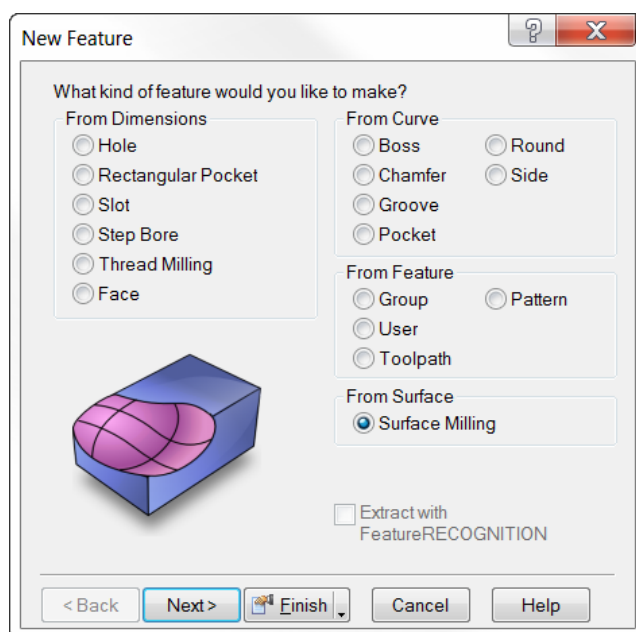
The part is now aligned correctly, the block stock has been sized to the part and the setup located correctly at the center and top of the stock.




## Creating a roughing Feature

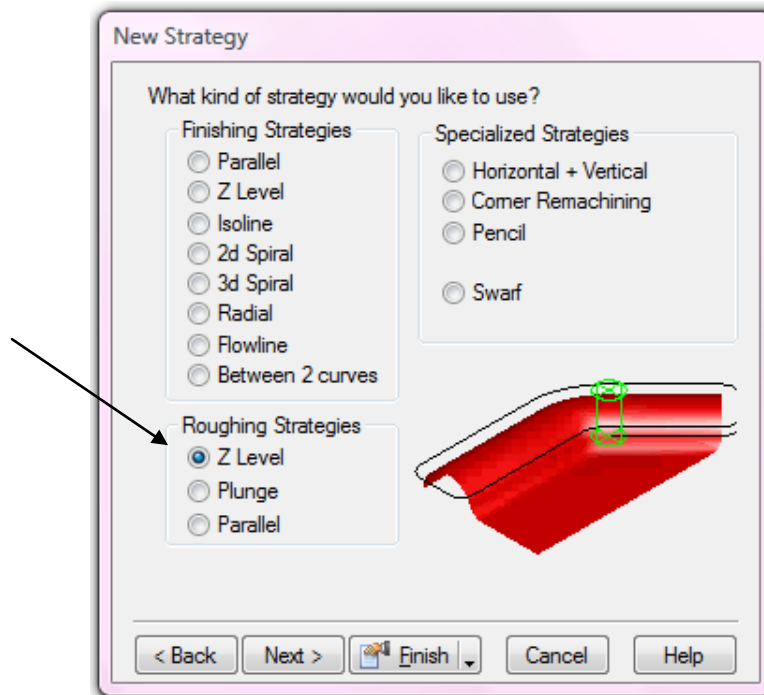
Now that the part has been imported and the setup completed we are ready to begin the roughing operation.

- Click on **Features** in the **Steps view**.
- Check **Surface Milling** and press **Next**.

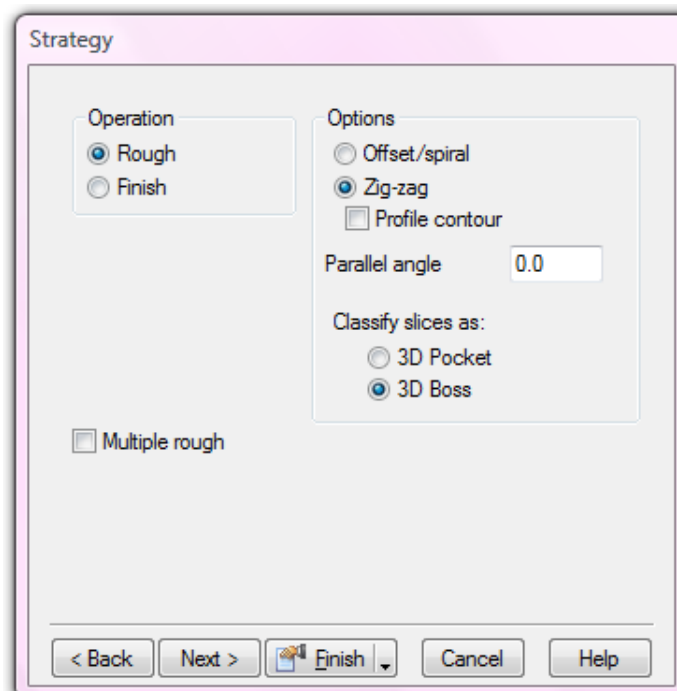




- **Box select** all of the surfaces and then press the **Add from selected items** button  then click next. If the faces already appear, click next.
- On the New Strategy page check **Choose a single operation**.
- Click **Next**.
- Check **Z Level** in the Roughing Strategies part of the form.

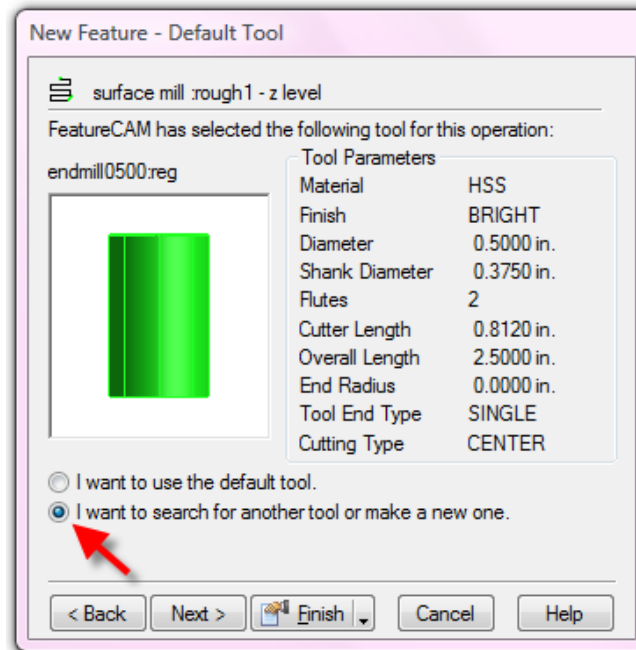


- Click **Next**.
- Fill in the Strategy page exactly as shown below and press **Next twice**.

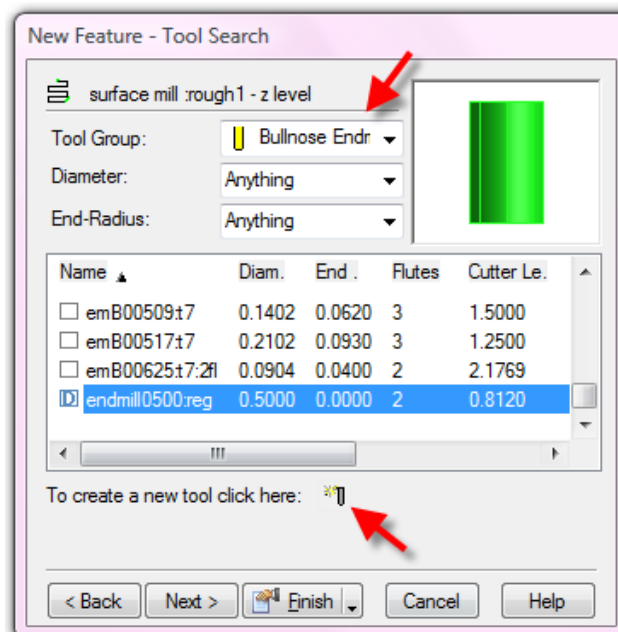


The default tool selected from the crib is a ½” diameter end mill. As this is a relatively large part it would be better to use a 1” diameter bull nosed cutter with a 0.125” tip radius.

- In the **New feature – Default Tool** page check “I want to search for another tool or make a new one” then click **Next**.

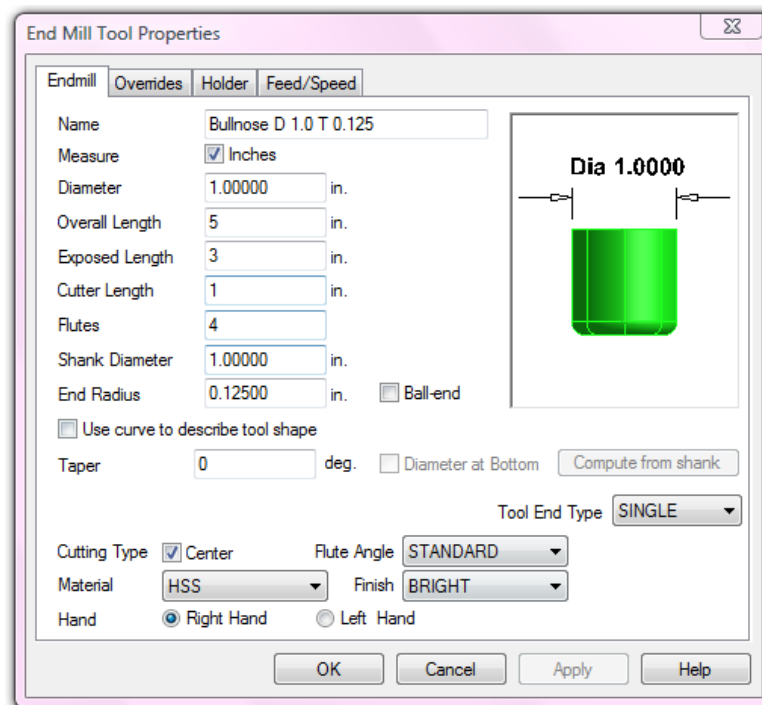


- On the **New Feature – Tool Search** page change the **Tool Group** to **Bullnose Endmill**.

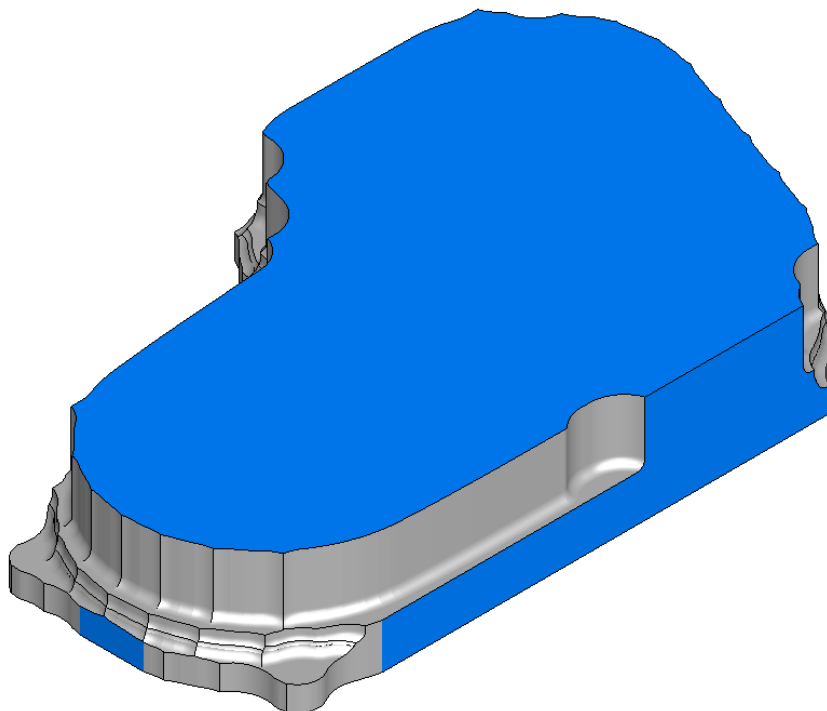


- Click on the **New tool** button  at the bottom of the form.

- Fill in the page exactly as shown below.



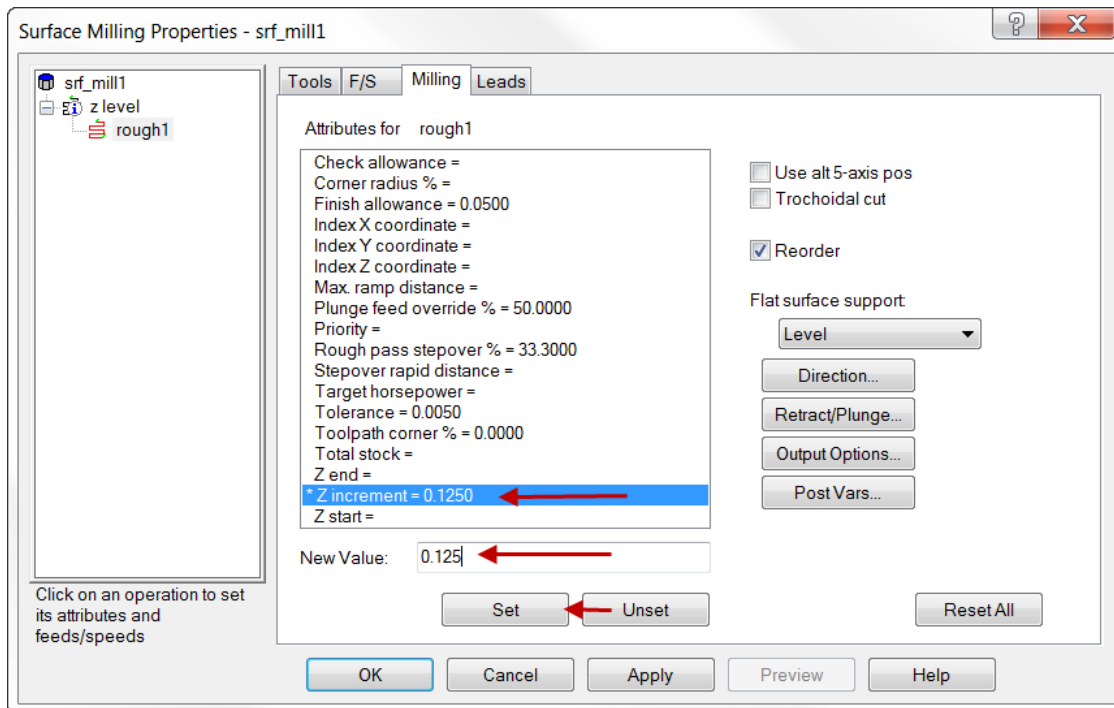
- Press **Apply** and then **OK**.
- At the prompt “**Set the new tool as override?**” Press **Yes**.
- Press **Finish** and **OK**.
- Run a **3D simulation**.



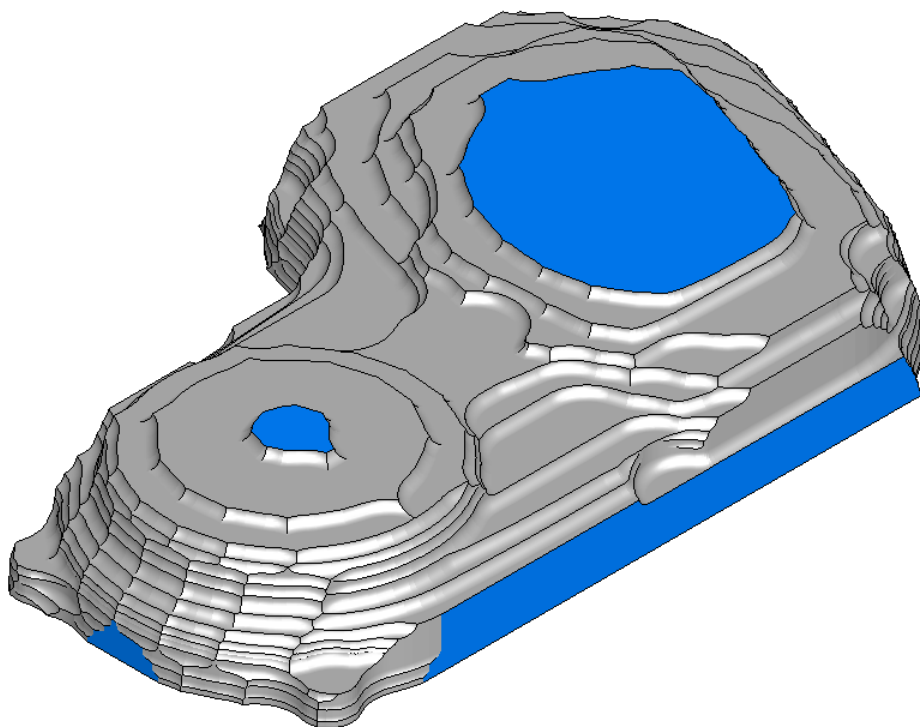
FeatureCAM chooses a stepdown automatically based upon the cutter diameter. In this case the stepdown is leaving excessive material on the part. We will now manually change the stepdown to a more suitable value.

## Stepdown – Z Increment

- Edit the feature (double click on **srf\_mill 1** in the Part view).
- Click on **rough1** and select the **Milling** tab.
- Select **Z increment**, enter a new value of 0.125 and click **Set**.



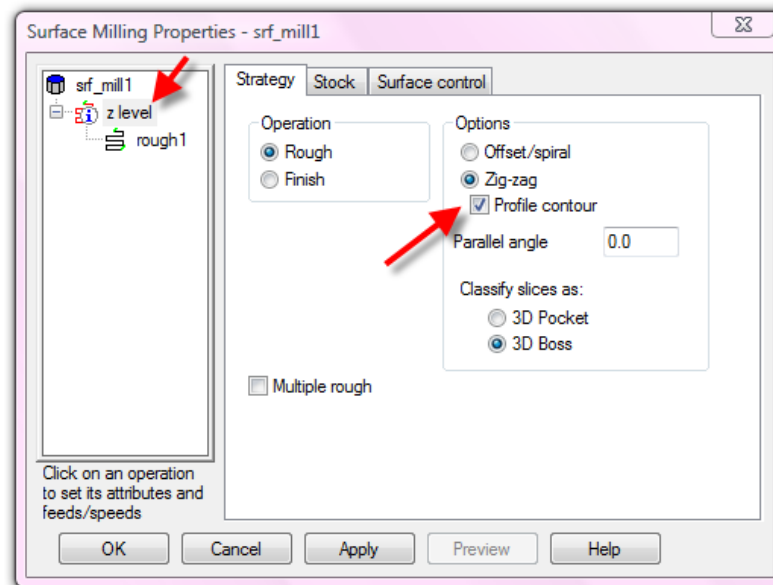
- Click **Apply**, **OK** and then run a **3D Simulation**.



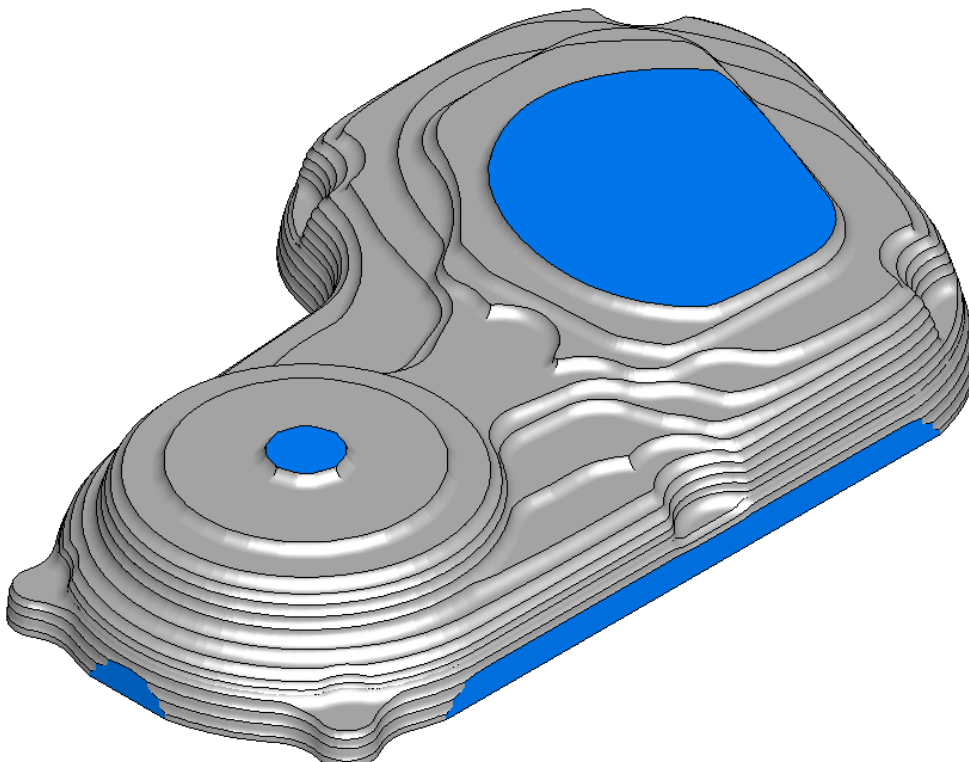
The steps are now much smaller and more closely approximate the shape of the finished part. Notice how the roughing operation has left cusps where the tool is changing direction at the end of every other pass. We will now modify the feature to rectify this.

## Profile Pass

- Edit the feature (double click on **srf\_mill 1** in the Part view).
- Click on Z level in the tree view.
- Select the **Strategy** tab and check **Profile contour**.
- Click **Apply** and then **OK**.



- Run a **3D simulation**.



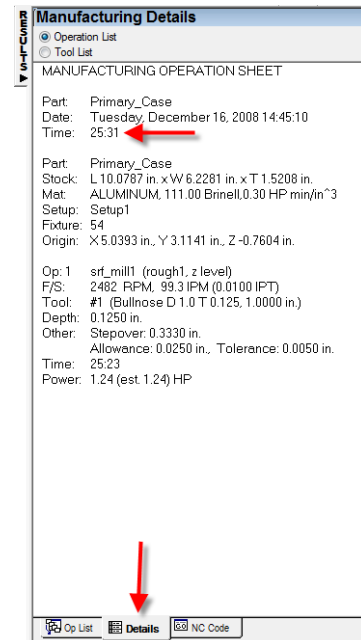
The tool is now profiling the part after roughing each Z level. This will increase the cutting time but will give more consistent tool loading in subsequent operations.

## Stepover %

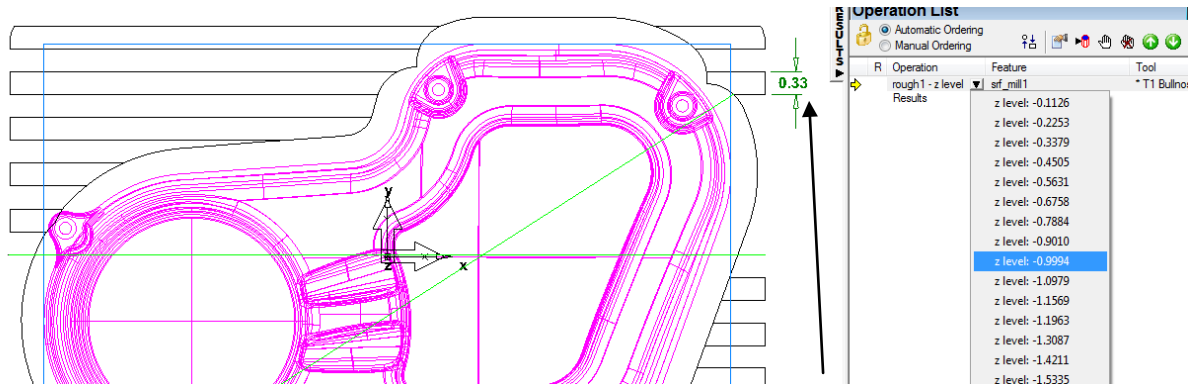
We can check the time required to complete the roughing operation by looking on the Details tab of the results pane.

- Open the **Results window** if it is not already open.
- Click on the **Details tab**.

You will see that there is a current time taken to rough machine the part. **Make a note of the time shown in your results window.** This estimation is dependent upon the distance the tool has to travel which is in turn a function of the total path length. We can shorten the toolpath by increasing the stepover between passes. This will increase the tool loading but as we have already reduced the depth of cut this will be OK.

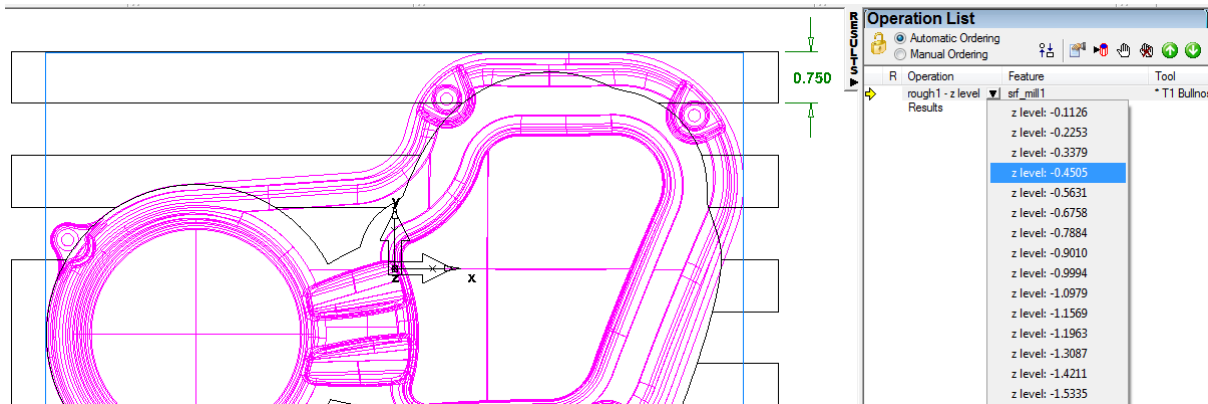


- Run a **Centerline simulation**.
- Using the **Op List** tab, click on the **pulldown arrow** next to **rough1**.
- Select the **Z level** at **Z -0.9994** (or thereabouts) as shown below.



In the **graphics window**, this displays just the single **Z level** that is selected. This is a useful trick when you want to see how the roughing toolpaths are behaving at a particular depth. The default stepover is 33.3% of the tool diameter, as the tool is 1" the stepover is 0.333". Follow the **arrow** pointing out the dimension at the **upper right** of the part in the image above to see the stepover value.

- Edit the feature. (double click srf\_mill1 in the part view again)
- Click on **rough1** in the tree view.
- Select the **Milling** tab.
- Click on **Rough pass stepover %** and set it to **75%**. **Apply** and **OK**.
- Select a **Top View** and run a **Centerline simulation**.

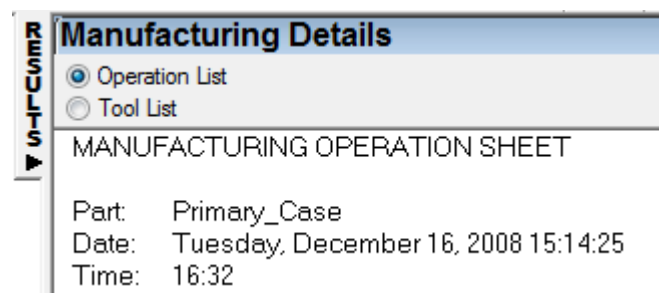


- Select the **Z level** at **Z-0.4505** (or thereabouts).

The stepover has now increased to 0.75” instead of the previous 0.333”

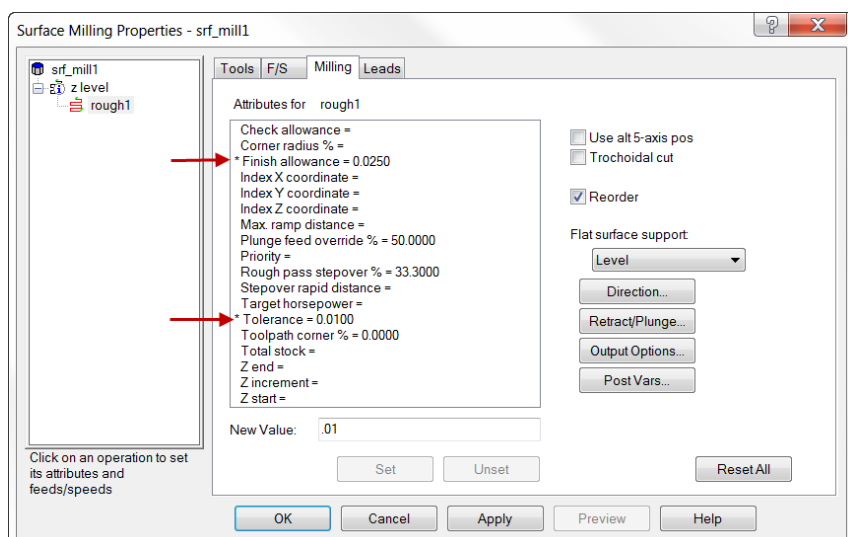
- Open the **Results** window.
- Click on the **Details** tab.

It can be seen that the time taken to rough the part has reduced from around 25 minutes to a much quicker time. Compare your original time to the time shown now.



- Edit the Feature. Select **Rough** and **Milling**.
- Click on **Finish Allowance**.
- Enter a new value of **0.025”**
- Change the **Tolerance** to **0.01”**.

This has changed the **minimum** amount of material left on the part to be **0.025”**. Please note that



the actual amount of material left on the job will depend upon the Finish Allowance, the Z increment (the step down between roughing levels) and the shape of the surfaces being roughed.

Reducing the tolerance results in a less smooth toolpath but reduces the time taken to calculate the moves. For a roughing operation normally the finish is unimportant. On a large part, changing the tolerance may make a huge difference to the toolpath calculation time.

**Important Note:** The **Tolerance** should always be **LESS** than the **Finish Allowance** to avoid gouging the part.

### Arc Line Approximation

If a part has a prismatic form, and a coarse machining tolerance is used, this can result in facets appearing on the machined part. If these are too pronounced this may lead to an uneven amount of material for the following operations causing uneven tool loading. If the part is large it may be impractical to tighten up the tolerance. Luckily FeatureCAM has a work around in such situations. Rather than outputting the profiling operation as a series of straight line moves you can choose to output as a series of arc (G2 & G3) moves within tolerance. This will get rid of any faceting and also reduce the overall size of the post processed program.

- Run a **Centerline simulation**.
- Click on the **NC code** tab.
- Scroll down to the end of the program.

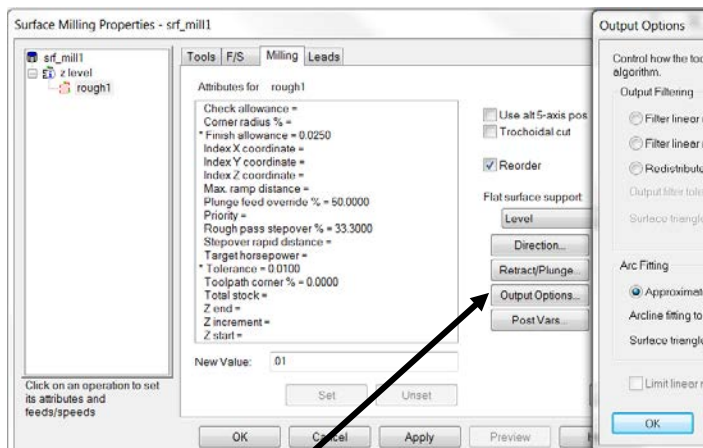
You will see that the final profiling pass is being carried out by a series of short, straight line moves. There are 6,885 lines of code in this picture. **Make note of how many you see in your NC code.** The post processor being used here is the standard HAASVF.cnc which is shipped with FeatureCAM.

We will now compare this NC code with that output when Arc/Line Approximation is being used.

```
N13700 X-4.9512 Y-3.0942
N13705 X-4.8537 Y-3.2661
N13710 X-4.8132 Y-3.3156
N13715 X-4.6413 Y-3.4594
N13720 X-4.4694 Y-3.538
N13725 X-4.2975 Y-3.5715
N13730 X-4.1256 Y-3.5659
N13735 X-3.9537 Y-3.5202
N13740 X-3.8347 Y-3.4622
N13745 X-3.438 Y-3.5793
N13750 X-3.1685 Y-3.6191
N13755 G0 Z1.0
N13760 G53 G49 Z0. M9
N13765 G53 Y0.
N13770 M30
%
```

- Edit the feature.
- Click on the **Milling** tab.
- Select **Output Options...** Select **Approx. linear moves with arcs and lines** (leave the values in the fields as in the picture below)
- Run a **Centerline simulation**.





```

N5750 X1.1792 Y3.644
N5755 Z-1.4335
N5760 G1 Z-1.6459 F49.6
N5765 Y3.6191 F99.3
N5770 G3 X-0.1496 Y2.6887 I0.2341 J-1.7485
N5775 X-0.3652 Y2.0679 I2.1385 J-1.0906
N5780 G2 X-0.5486 Y1.9144 I-0.2415 J0.1021
N5785 G1 X-3.0423 Y1.7365
N5790 G3 X-4.5404 Y1.1633 I0.1697 J-2.688
N5795 X-5.3866 Y0.0124 I-0.1865 J-0.7495
N5800 X-5.5444 Y-0.6377 I2.3943 J-0.9252
N5805 G0 Z1.0
N5810 G53 G49 Z0. M9
N5815 G53 Y0.
N5820 M30
%
```

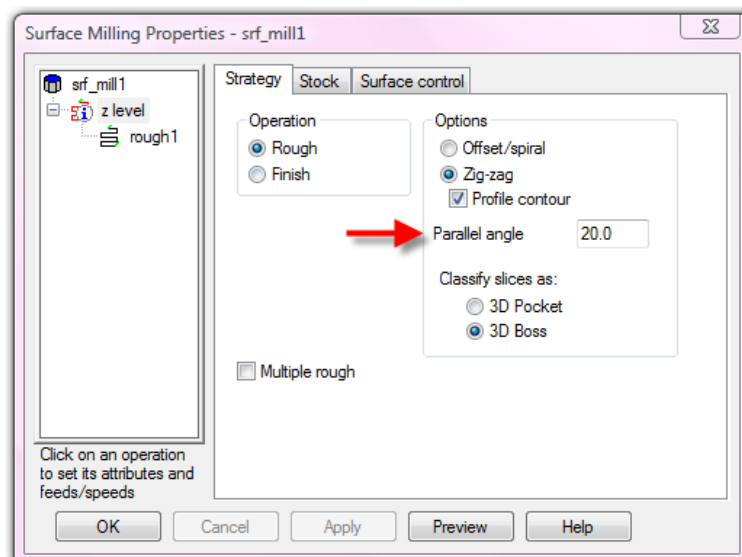
- Click on the **NC code** tab.
- Scroll down to the end of the program.

You should now see FeatureCAM using a mixture of straight line and arc moves as shown on the right. The length of the program (in this picture) is now only 2,910 lines. **Compare your original Code length to what you see now.** The code is about 60% less than before. For older controllers with limited data storage or slow baud rates this can be extremely useful.

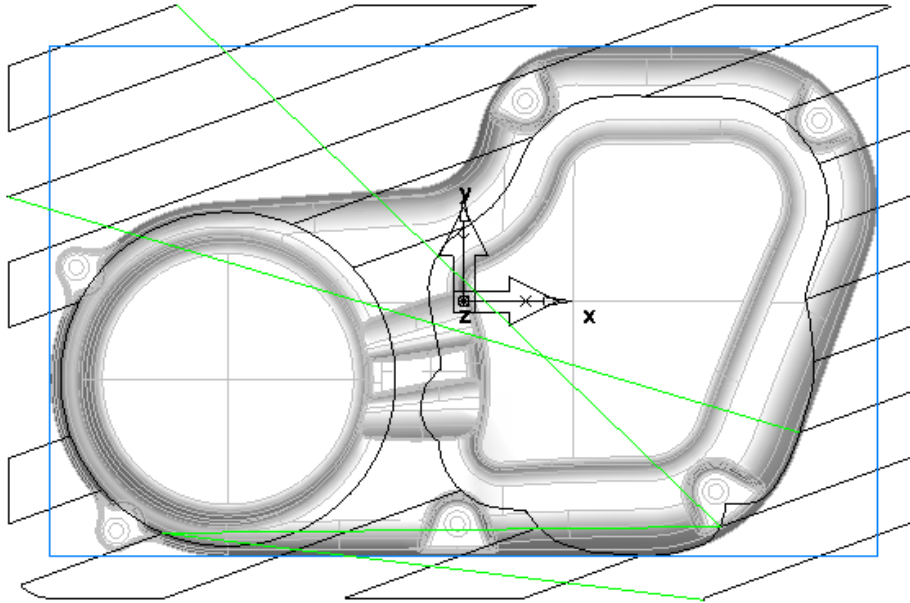
## Parallel Angle

By default a Zigzag roughing operation will machine up and down the X axis, stepping over in the Y direction. Depending upon the shape of the part this may or may not be an efficient strategy. If a part was longer in the Y direction than the X it would make more sense to make cutting passes along the Y axis rather than the X. FeatureCAM allows the user to specify any angle they like for the pass direction. This allows the user complete control to optimize the pass direction for a given part.

- Edit the feature.
- Click on **z level** and select the **Strategy** tab.
- Change the **Parallel Angle** to **20** degrees. **Apply** and **OK**.



- Run a Centerline sim and then display just the pass around Z-0.350
- Select **Top View**



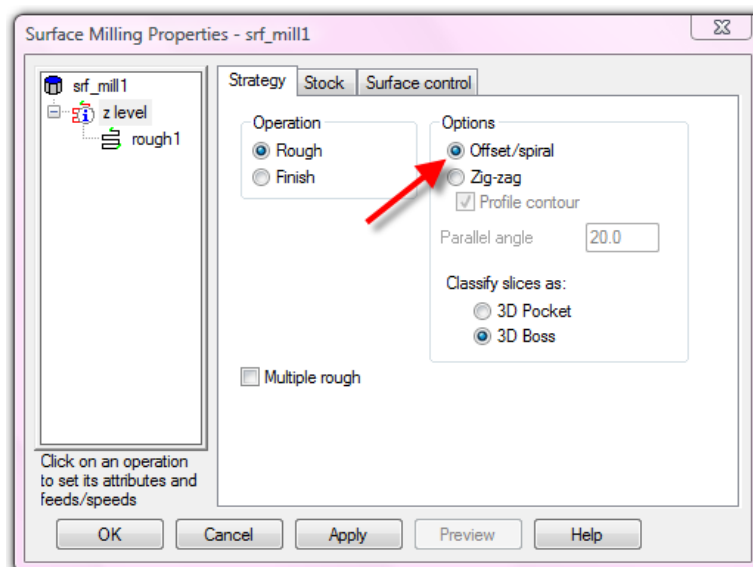
Note how the roughing passes are now going at 20 degrees to the X axis. Selecting an appropriate roughing angle can greatly reduce the time taken to rough machine a part.

# Rough – Z Level Offset/Spiral

## Introduction

We will now look at Offset/spiral roughing. In this method the passes follow the profile of the part at each Z level rather than being parallel to an axis direction. An Offset/Spiral toolpath is often shorter than a Zigzag toolpath with the same settings. Given the speed with which FeatureCAM calculates toolpaths it is worth experimenting with settings to get the shortest possible cutting time on the machine.

- Open the part **Primary\_Case.fm**
- **Box select** all of the surfaces
- Click on **Features** in the **Step view** of the **TOOLBOX**.
- Check **Surface Milling** and press **Next twice**.
- On the New strategy page check **Choose a single operation**.
- Click **Next**.
- Check **Z Level** in the Strategy part of the page. Click **Finish**.
- Edit the Feature. (double click the srf\_mill feature in the part view)
- Select **Z level** and the **Strategy tab** and check **Offset/Spiral**.
- Select the **Rough** operation and the **Milling tab**.
- Set the **Z increment** to .125
- Click on **Apply/OK**.

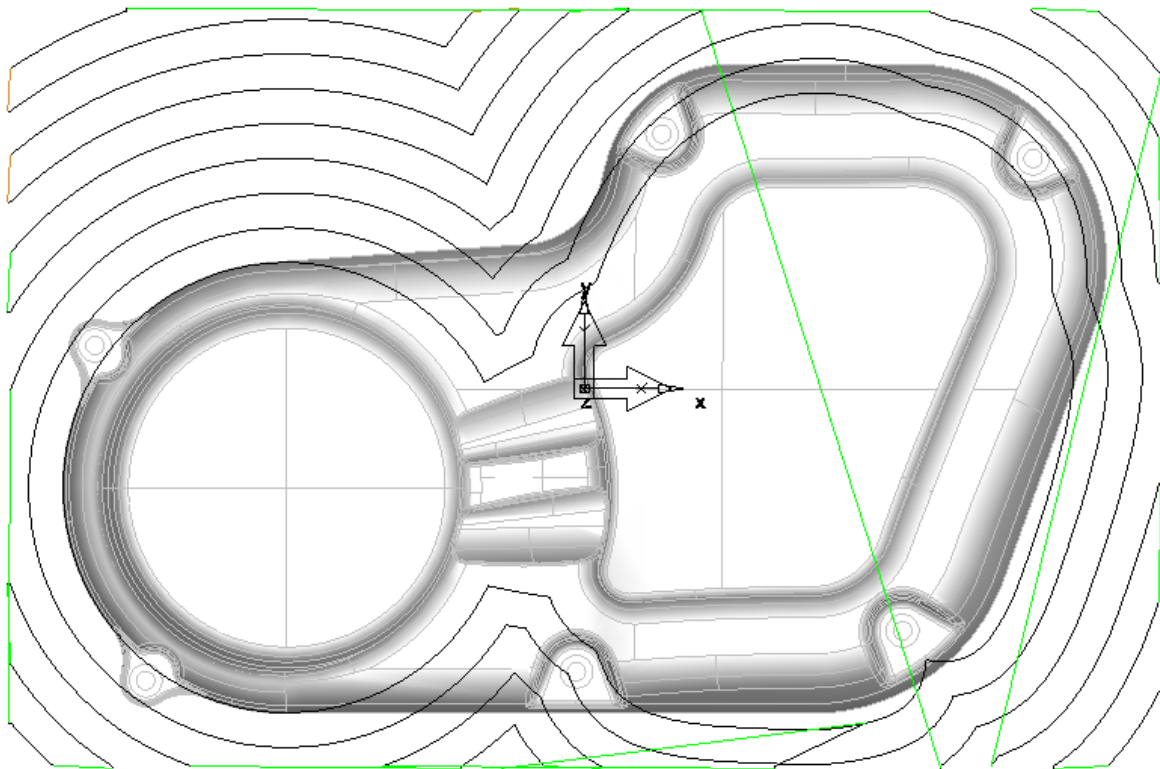


- Run a **Centerline simulation**.

- Select a **Top view**
- Open the **Results** window and the **Op List Tab**.
- Select the **down arrow** to the right of “rough1 – z level”



- Display just the roughing near **Z-0.4505**.

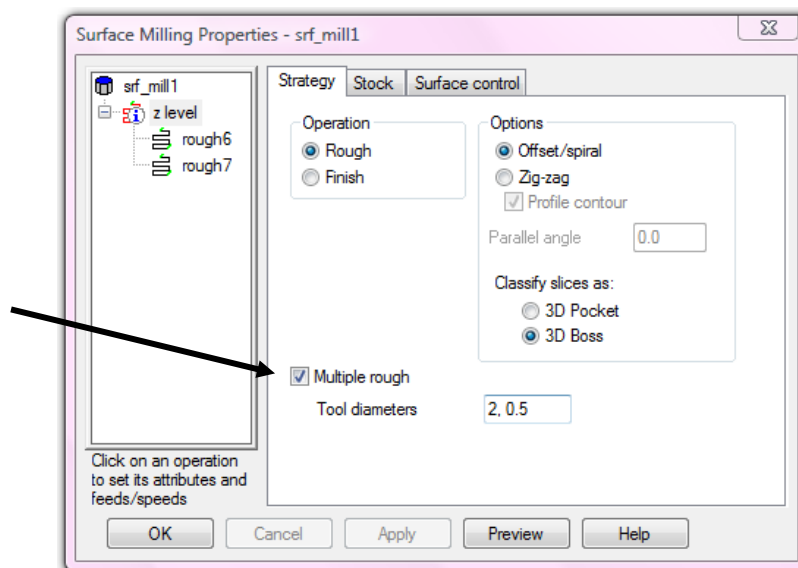


The tool profiles smoothly around the part with fewer sudden changes in direction than the Zigzag strategy.

## Multiple Rough Diameters

So far we have roughed the part using a 1” diameter bull nosed mill with a stepdown of 0.125”. This is giving us a roughing toolpath that closely approximates the finished part. However, because of the small stepdown, we are spending more time clearing the rough material than necessary. Also there are areas where the 1” tool cannot fully enter, particularly in the recesses above the bolt holes resulting in excess stock being left on the part. We will now look at a technique that allows us to use a series of progressively smaller tools to clear out the rough material more efficiently.

- Edit the feature.
- Click on **z level** and select the **Strategy** tab.
- Check the **Multiple rough** checkbox.
- Enter the values **exactly** as shown below. **Apply** and **OK**.

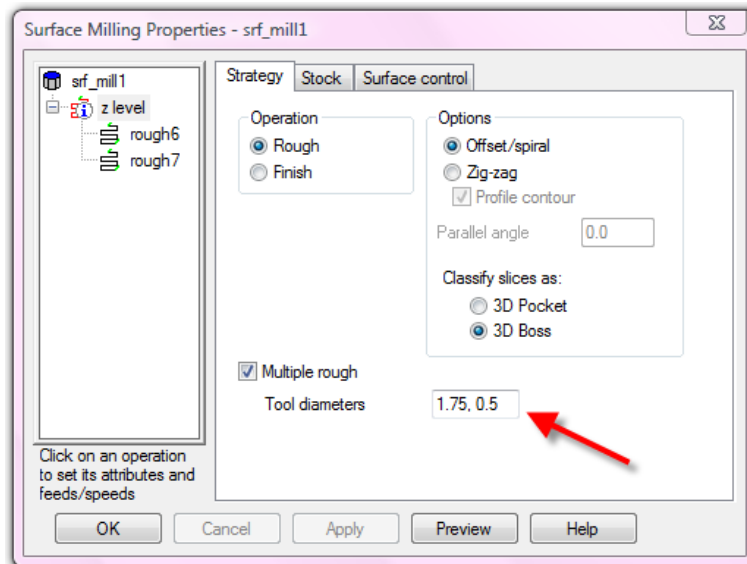


- Run a **Centerline** simulation.

Notice that there are **two separate tools** cutting material away. **Click** on the **rough toolpaths** in the **Op List** of the **Results window** one at a time to compare the two toolpaths.

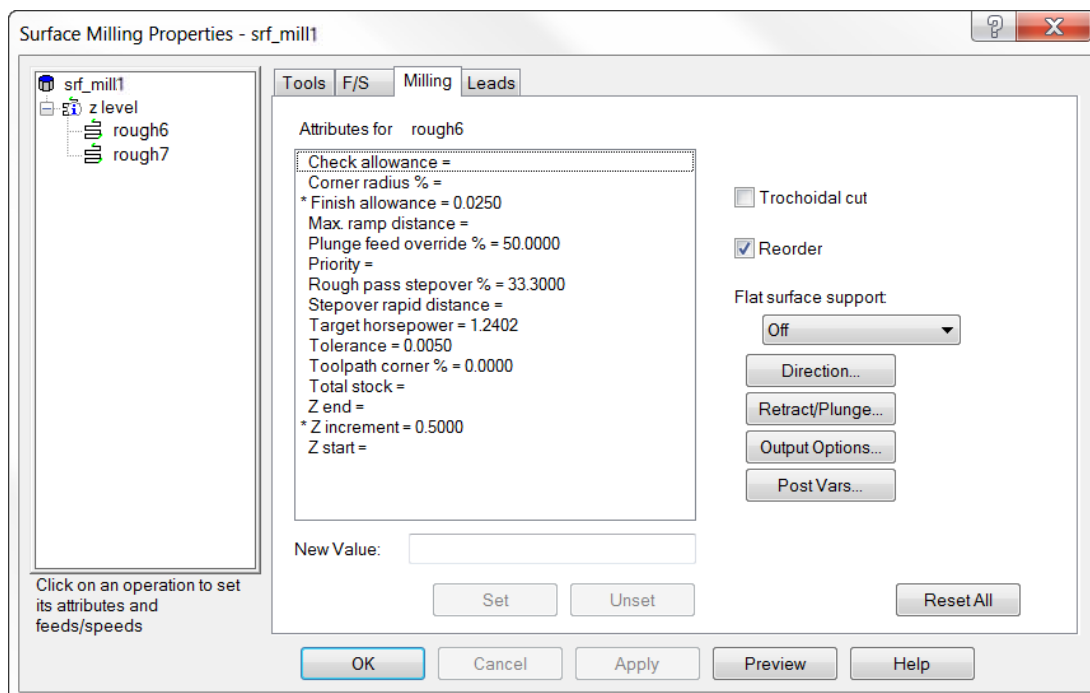
We will now choose different tooling and modify the strategy to remove the material in a more consistent manner. This will provide consistent tool load for subsequent operations.

- Edit the feature. (double click the srf\_mill1 feature in the part view)
- Click on **z Level** and select the **Strategy** tab.
- Enter the tool diameters to be used as shown below. **(1.75,0.5)**.



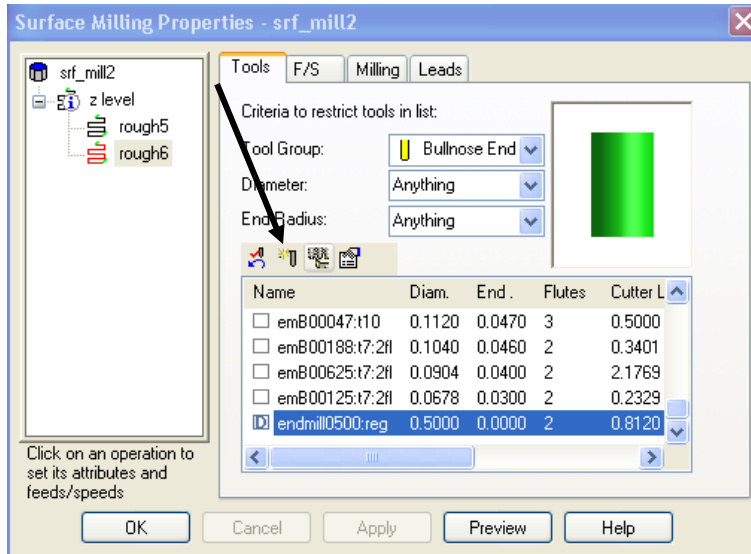
- Click **Apply**.

You should now see two roughing operations as in the above image. When FeatureCAM has been told to multiple rough it uses the first tool to machine the part in the normal manner. The second smaller tool is used only to machine the material left by the first. In this way we can reduce the size of the steps left by the first, larger tool and then pick out the inside corners where the big tool could not fit. This process is also called **Rest Roughing**. We will now modify the attributes of the two roughing operations to optimise the roughing process.

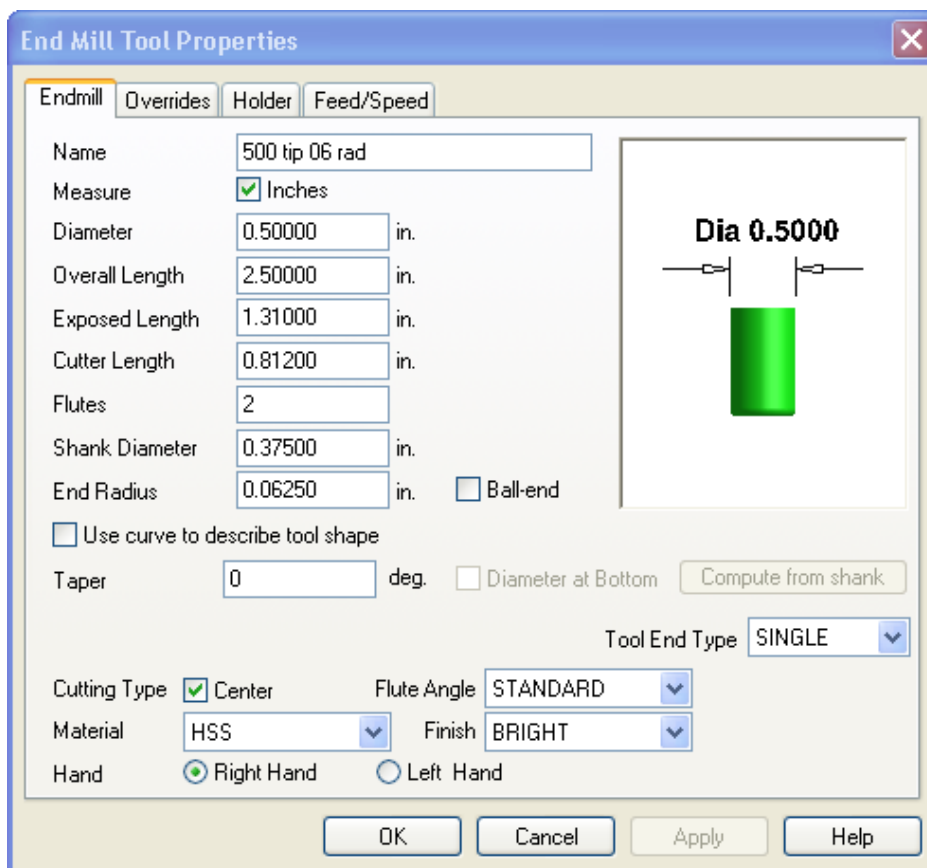


- Select the first roughing operation (**rough6** above).
- Select the **milling** tab.
- **Set** the machining attributes **exactly** as shown above.

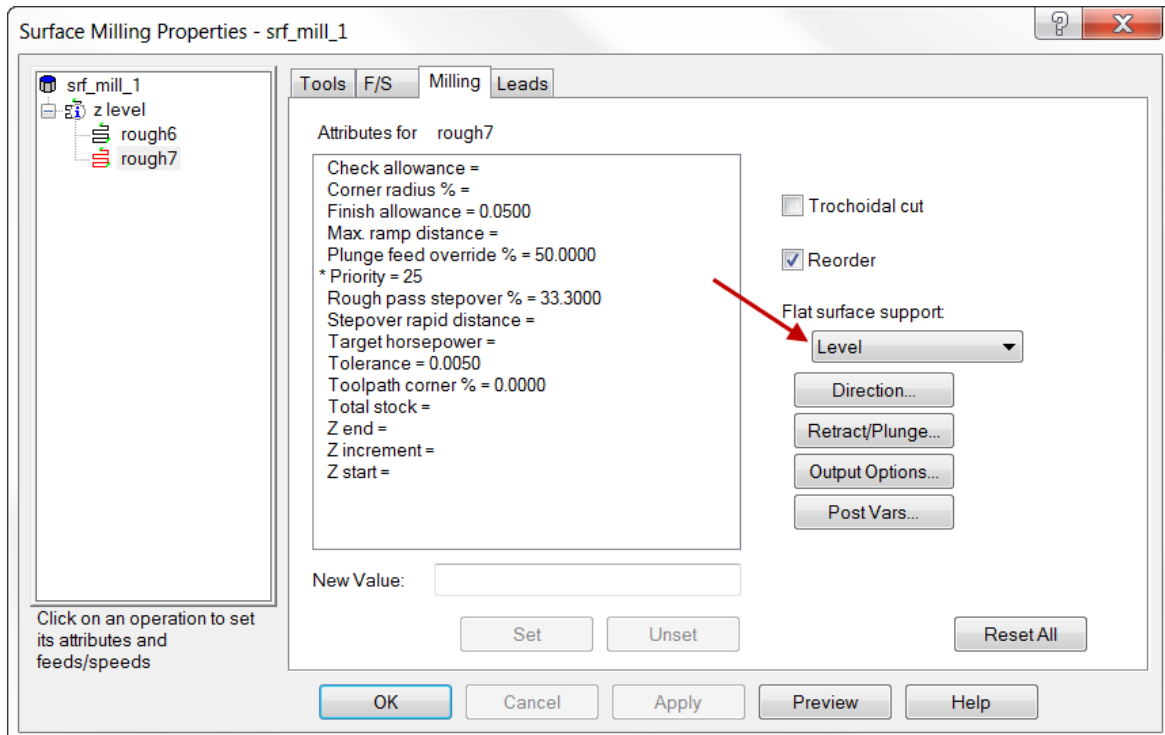
- Select the second rough operation.
- Select the **Tools** tab.
- **Select the Tool Group “Bullnose Endmill”.**
- Create a new tool by clicking the **New Tool** icon shown below.



- Name the tool **500 tip 06 rad.**
- Enter the values as shown below.

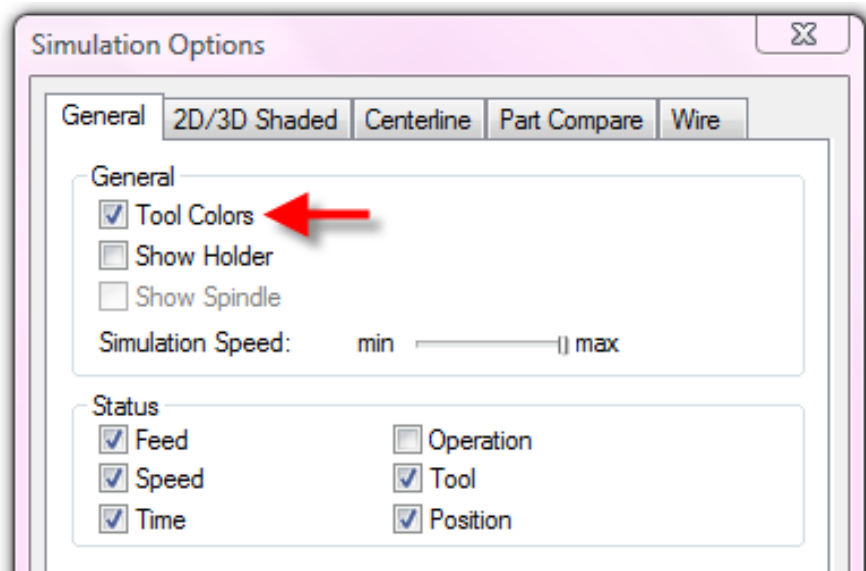


- Click **Apply** and **OK**.
- Click **Yes** on the “**Set the new tool as override?**” page. Click **OK**.
- Edit the Feature. (double click on the srf\_mill1 feature in the part view)
- Select the second rough operation (**rough7** below)
- Select the **milling** tab.
- Set the machining attributes **exactly** as shown.

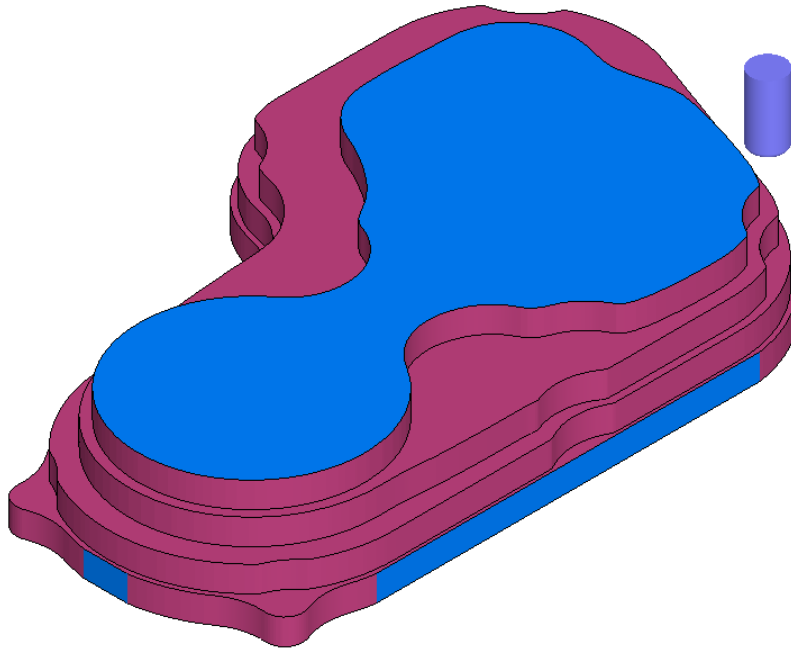


- Click **Apply** and then **OK**.
- Go to the **Options** menu and select **Simulation**.
- Check **Tool Colors** on the **General** tab. **Apply** and **OK**.

This will display the area cut by each tool in the simulation as a different color.



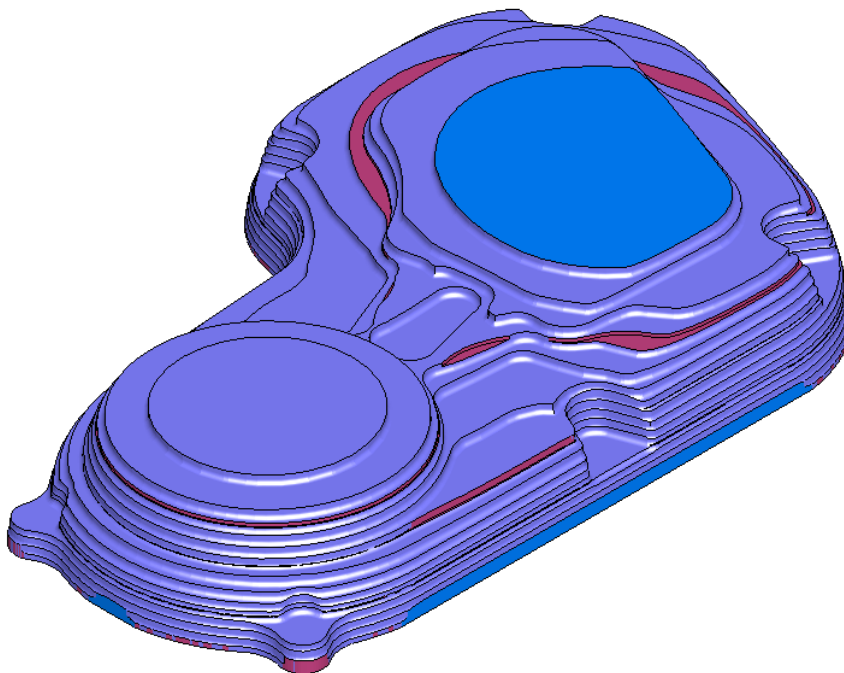




Select a **3D simulation** and click **Play to next operation**.

The larger 1.75" tool and the 0.5" stepdown are now removing the bulk of the rough stock much more quickly. As flat surface support has been turned off the stepdown is exactly 0.5" regardless of the shape of the part.

- Click **Play to next operation**.



The second, smaller tool is now only machining the remaining material. This will result in a more efficient toolpath that is leaving a more consistent material thickness for following tools.

## Z End

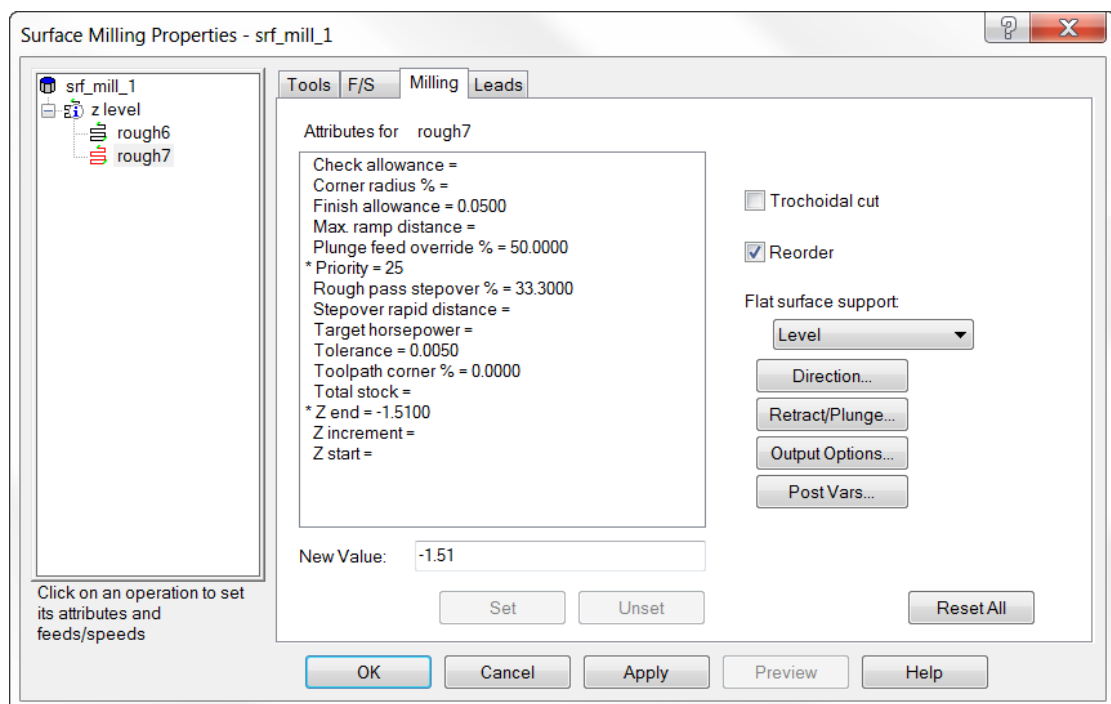
You may have noticed that the final pass of the second tool machined the flat area at the bottom of the stock. You can see this clearly if you slow down the simulation when running the second pass. FeatureCAM assumes that the bottom of the stock requires milling off unless we tell it otherwise. We can do this by applying a Z end value above the bottom of the stock.

- Open the **Stock Properties** of **Stock1** (double click stock1 in part view)
- Set the **thickness** at 2"
- **Apply** and **OK**

We will use this extra material as if we were clamping our part in a vice to demonstrate how Z end works.

Run a **3D Simulation**. Notice that FeatureCAM will cut to the bottom of the stock.

- Edit the feature.
- Open the properties of **rough6** and select the **Milling** tab.
- **Set** the **Z End** attribute to **-1.510** as shown.
- Do the same for **rough7**
- **Apply** and **OK**.



- Run a **3D simulation**

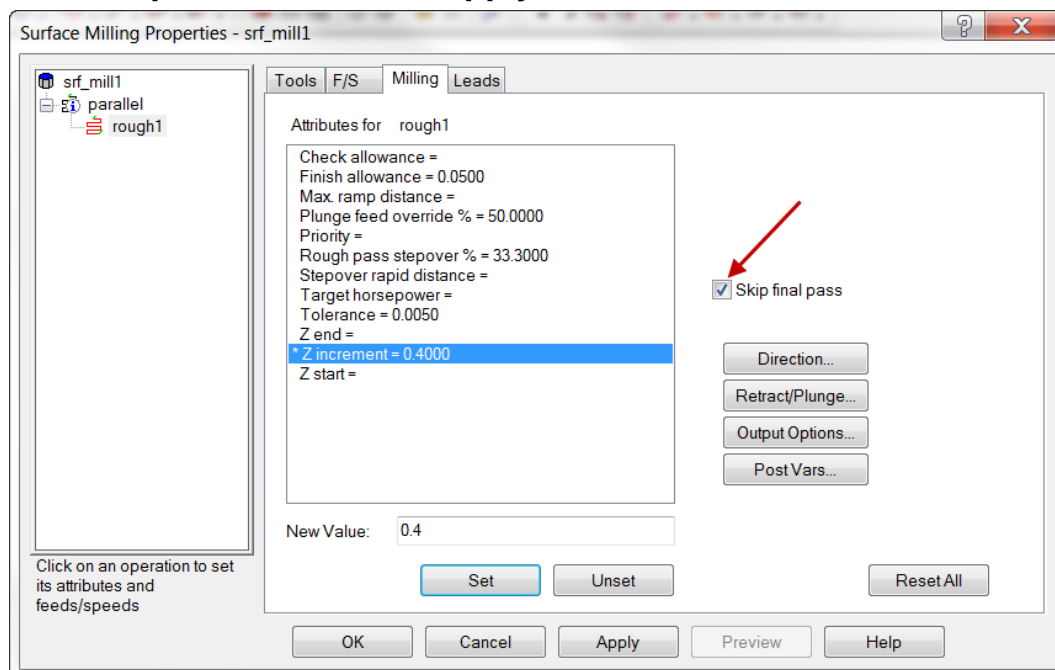
**Note that the pass at the bottom is not cutting all the stock all the way to the bottom as it was before.**

# Parallel Roughing

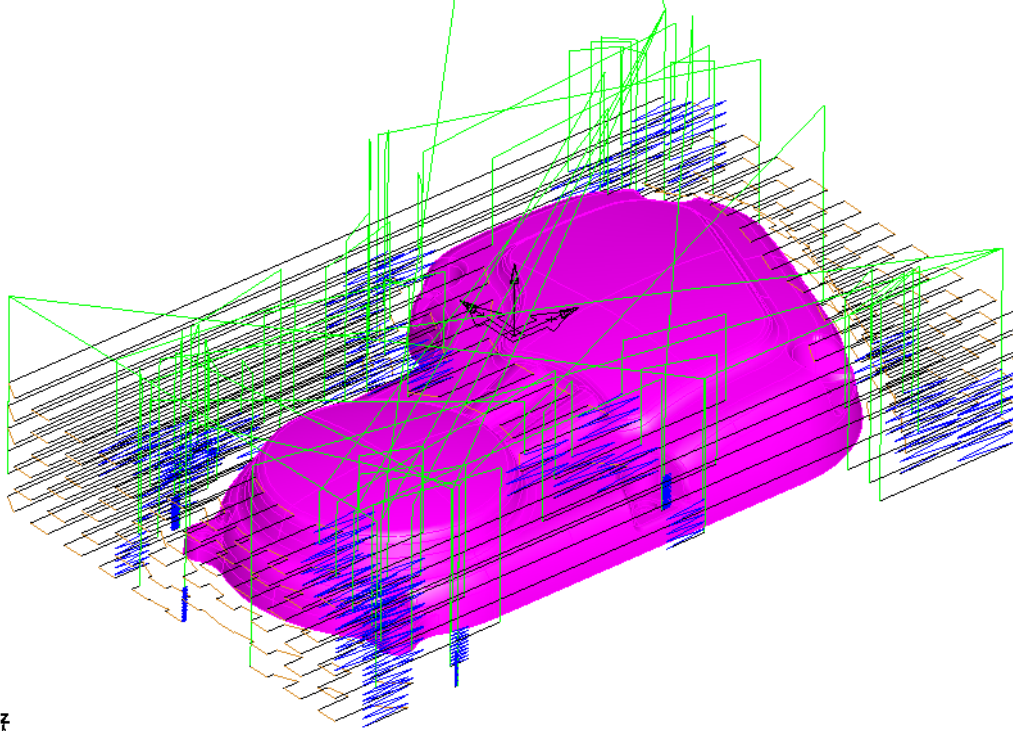
## Introduction

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.

- Open the part **Primary\_Case.fm**
- **Box select** all of the surfaces
- Click on **Features** in the **Step view**.
- Check **Surface Milling** and press **Next** twice.
- On the New strategy form check **Choose a single operation**.
- Click **Next**.
- Check **Parallel** in the Roughing Strategies part of the form.
- Press **Finish**.
- Edit the feature by double clicking on it in the **Part View** window.
- Click on **rough1** in the tree view.
- Select the **Milling** tab and set the **Z increment** to **0.4"** as below.
- Check **Skip final Pass**, click **Apply** then **OK**.

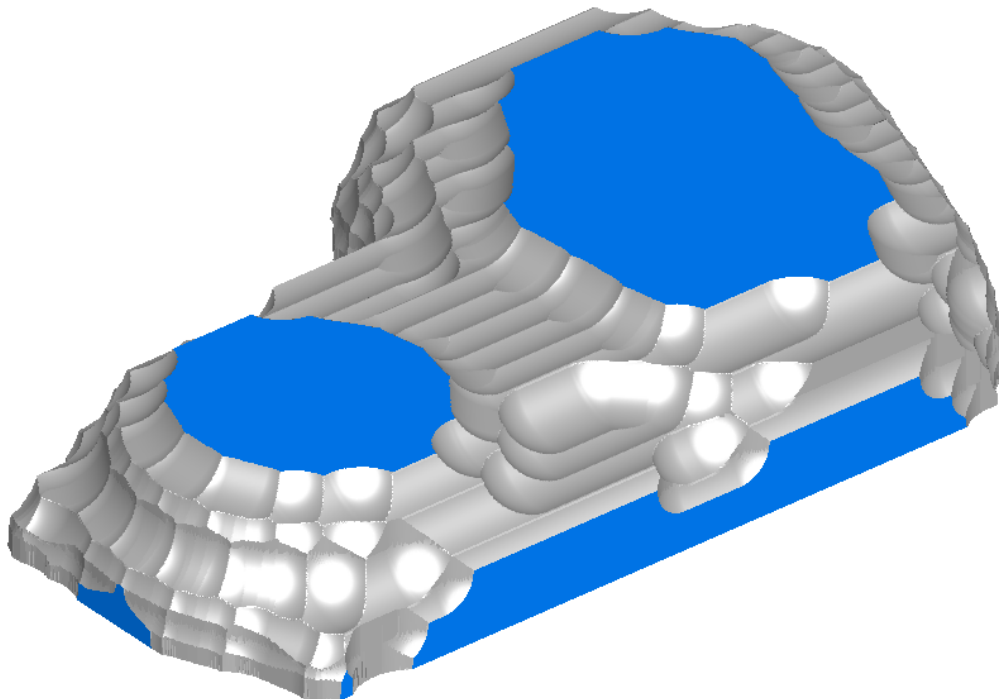


- Click on the **Tools** tab and select a **1" Ball Endmill**.
- Run a **Centerline simulation**.




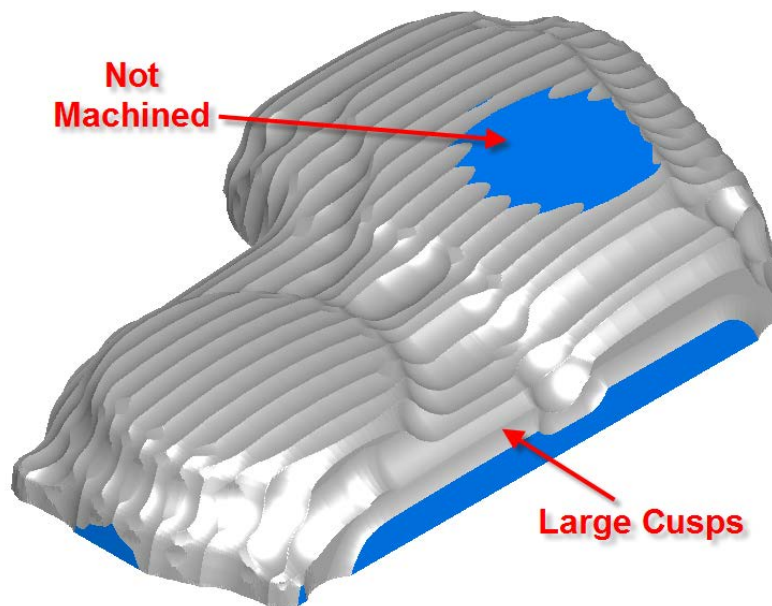
The tool path runs parallel with the x-axis at the z-increments specified. Note the ramping approach moves, these are needed where a non center cutting tool is approaching the stock from above. We will deal with ramp moves in detail further on in these notes.

- Run a **3D RapidCut Simulation** 



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

- Click on **rough1** in the tree view.
- Select the **Milling** tab and uncheck **Skip final pass**.
- Click **Apply** and then **OK**.
- Run a **3D RapidCut simulation** 

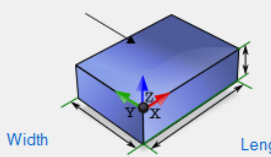


The tool path now has a final pass that is parallel to the x axis that changes in z according to the contour of the part. Note how the very top of the part is not machined. In order to ensure that the semi-finish pass goes all the way across the part we need to put additional stock on top to a thickness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

- Double click on the stock.
- Click on **Resize**.
- Add **0.1"** to the top of the stock.
- Click **Next** and then **Finish**.

Stock Dimensions

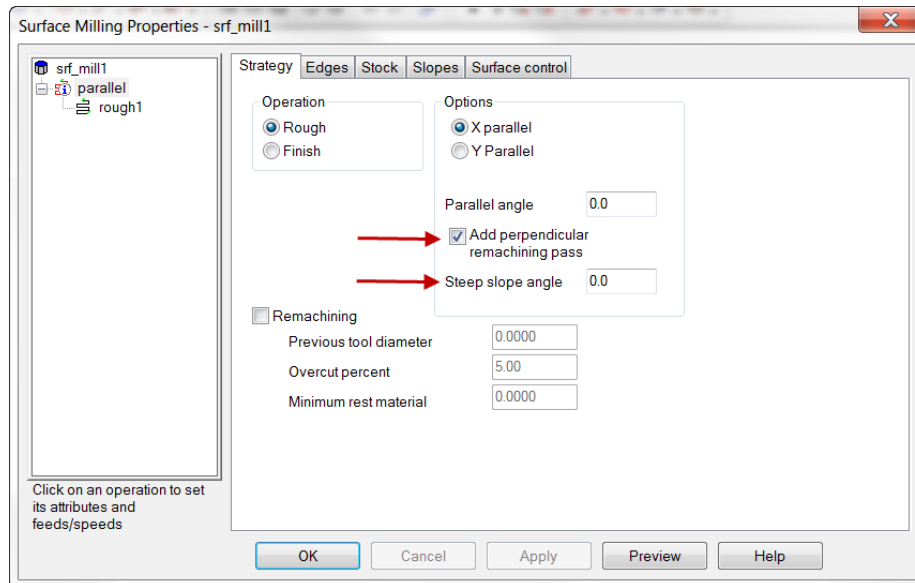
☐ Enter specific stock dimensions  
☒ Compute stock size from the size of the part



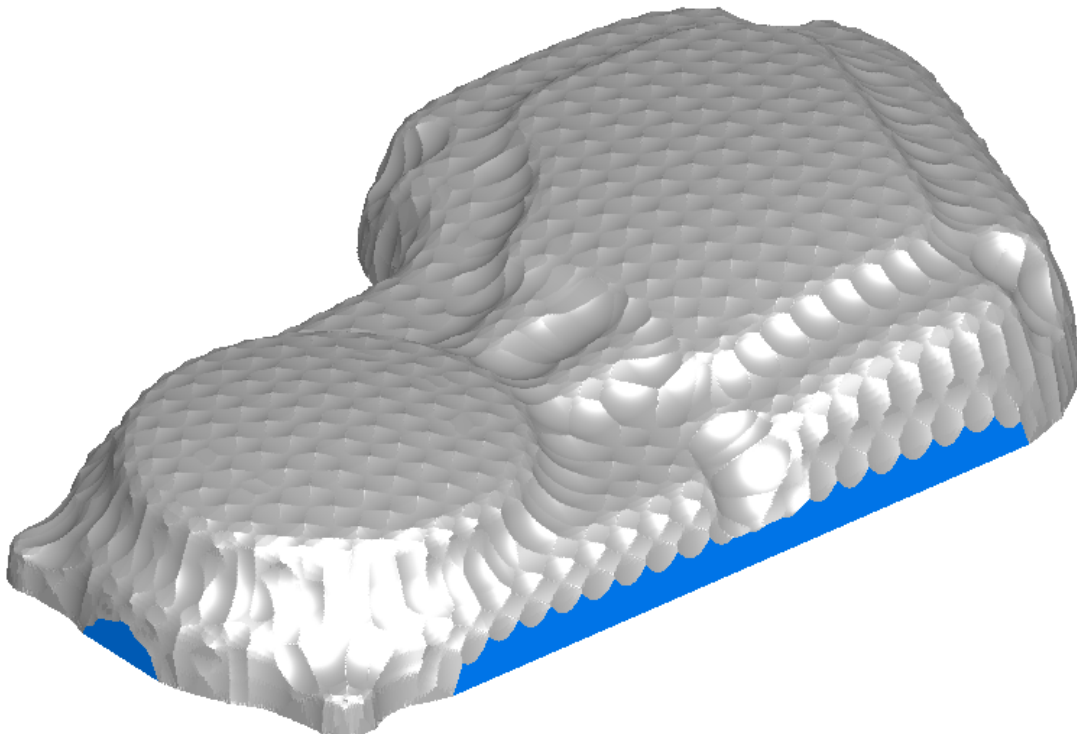
Imported Data:	Extra stock size		Stock size	
Length: 10.0787	+ Extra stock size (-X):	0.0000 in.	=	10.0787
	+ Extra stock size (+X):	0.0000 in.		
Width: 6.2281	+ Extra stock size (-Y):	0.0000 in.	=	6.2281
	+ Extra stock size (+Y):	0.0000 in.		
Thickness: 1.5391	+ Extra stock size (-Z):	0.0000 in.	=	1.6391
	+ Extra stock size (+Z):	0.1000 in.		

Preview

- Edit the feature, select **parallel** in the tree view.
- Check **Add perpendicular re-machining pass**.
- Set the **Steep slope angle** to **Zero** degrees.



- Click **Apply** and then **OK**.
- Run a **3D RapidCut 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 than the specified angle perpendicular to the X direction will be re-machined.

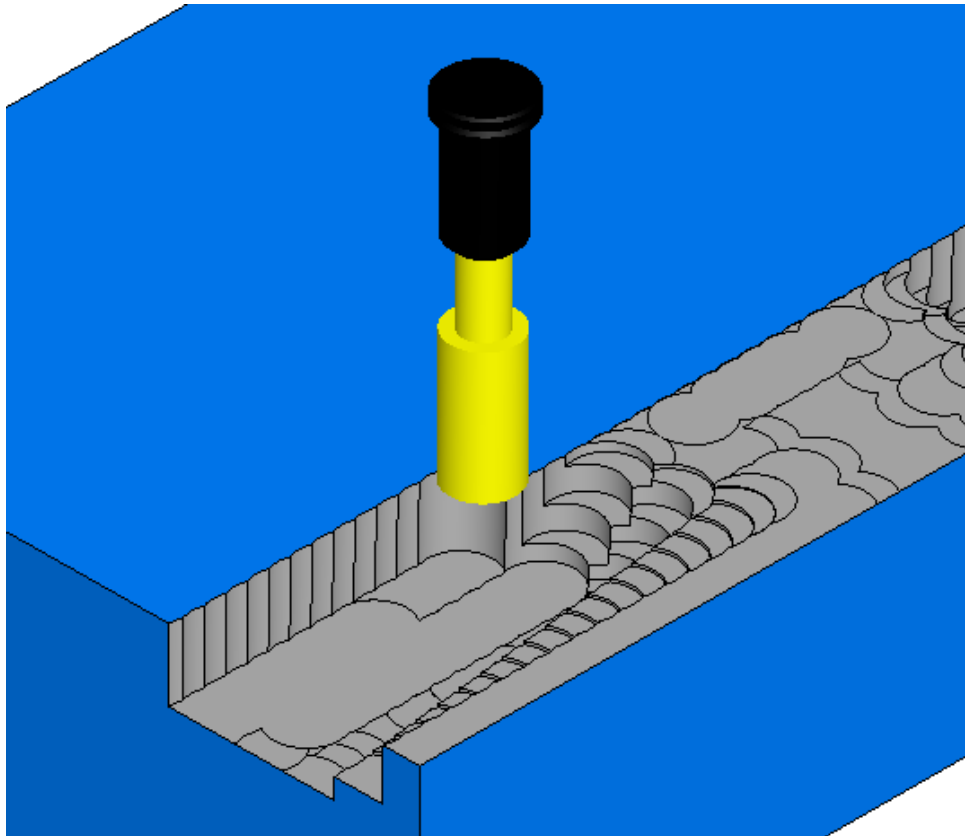
---

# Plunge Roughing

---

## Introduction

Plunge roughing is a technique for removing large amounts of stock very quickly by a series of drilling operations. Plunge roughing uses specialised tooling which resembles a long reach insert drill. Before attempting plunge roughing you should consult with your machine tool supplier to ensure that your machine spindle is capable of withstanding the large torque and axial loads involved.

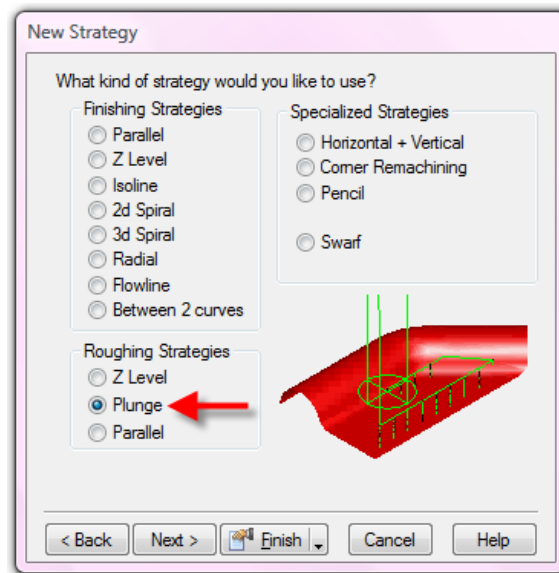


When applied correctly plunge roughing can give superior metal removal rates when compared to traditional Z level roughing, particularly where deep parts are being roughed. With normal roughing when working in a deep part, long reach tooling is required. As these longer tools are more easily deflected the cutting feed rates have to be slowed to reduce the side loading on the tool. With plunge roughing the main loading is along the axis of the tool so the length has a much smaller effect on tool deflection.

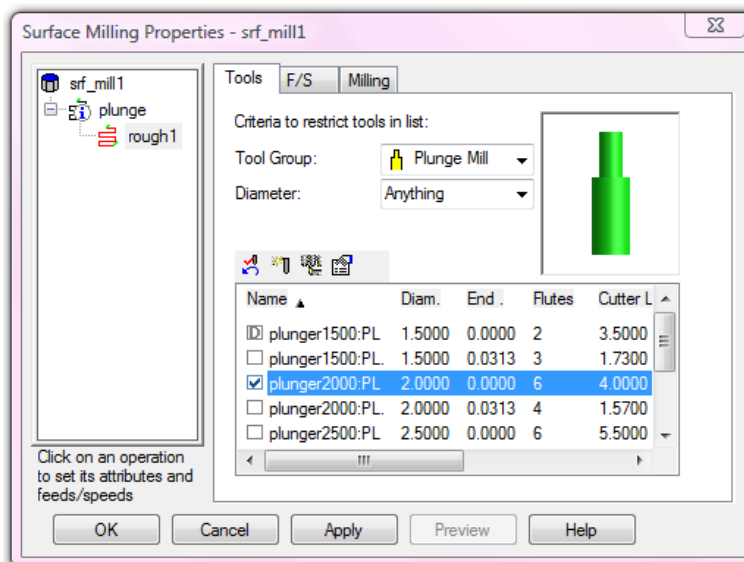
- **Open** the file **Die\_Block.fm**.
- **Box Select** all surfaces.
- Create a new **Surface Milling Feature**. **Next** twice.



- Choose a single operation. Next.
- On the **New Strategy** page select **Plunge Roughing**.



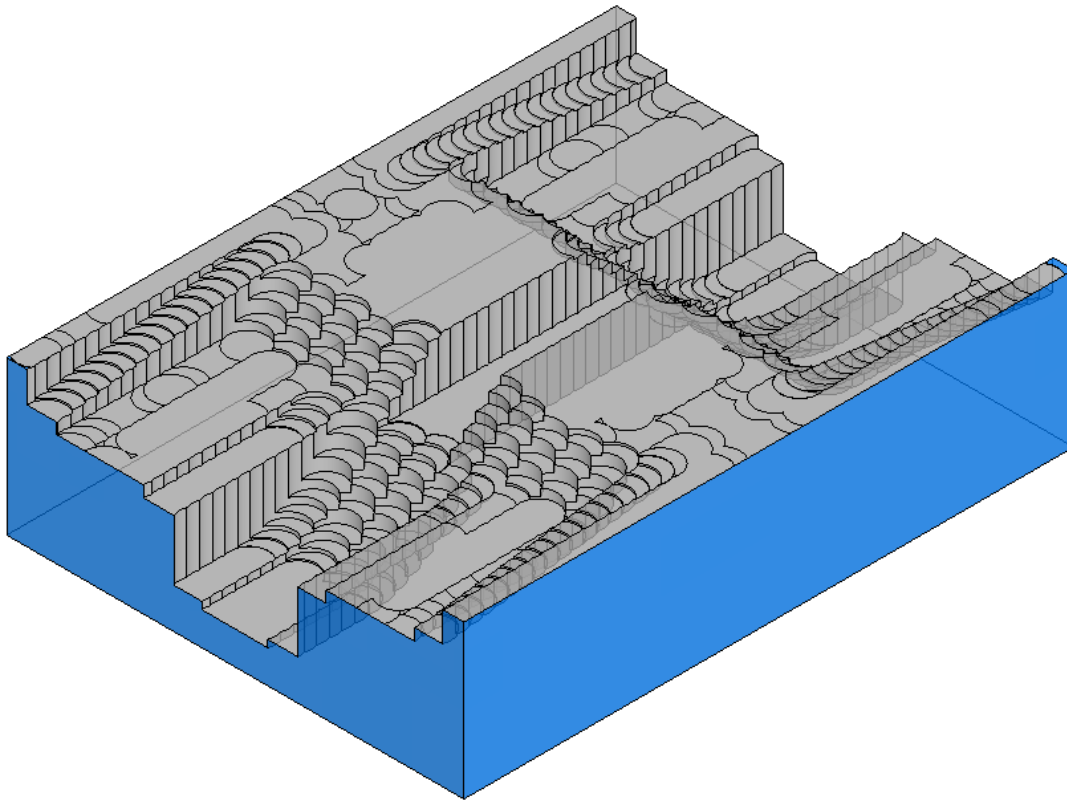
- Click **Finish**.
- Select **rough1** in the feature properties and click on the **Tools** tab.
- Select the **2" Plunge roughing tool** then click **Apply** and **OK**.



- From **Options** on the **Menu bar** select **Simulation**.
- Click on the **2D/3D Shaded** tab and check **Translucent part**.
- Click **Apply & OK**.



- Run a **3D Simulation**.



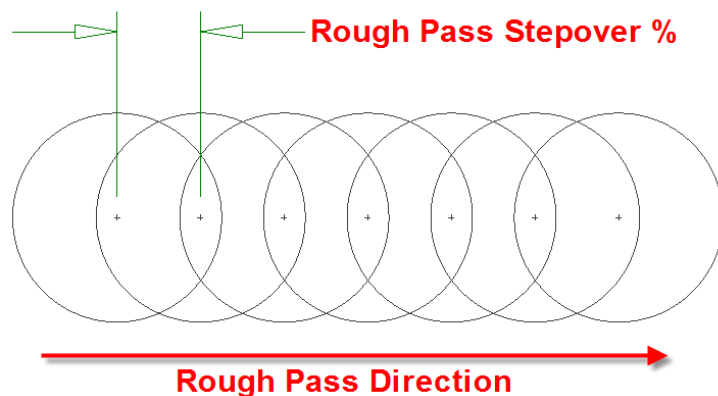
The tool rapidly removes the material from the part by a series of drilling operations along parallel lines in the X direction. The distance between the parallel passes and the shape of the part determines the amount of material left on the part after plunge roughing. We will now make the distance between the passes smaller to reduce the steps in the remaining stock.

- **Eject the Simulation**
- Edit the feature, click on **rough1** and select the **Milling tab**.

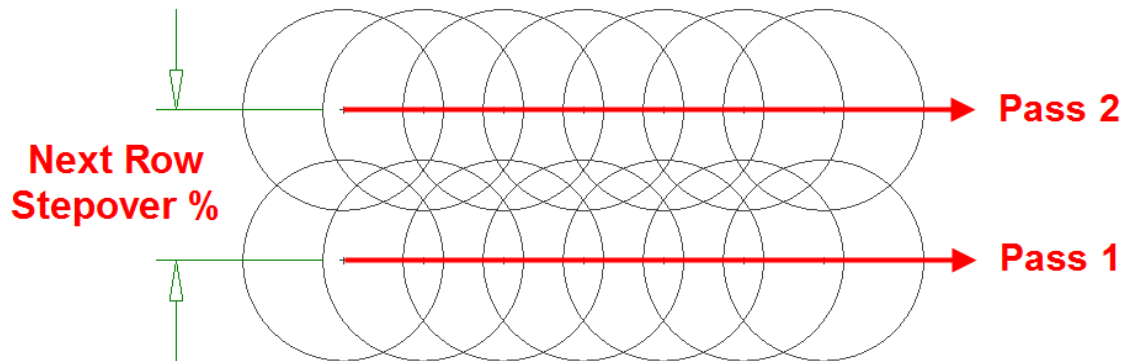
There are two attributes which control the way that the plunge roughing tool steps over along a single roughing pass and between each pass. We shall now look at the effect of each.

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 2" diameter plunge rougher this is giving a stepover of 1.6" 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 center 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%**.

Attributes for rough1

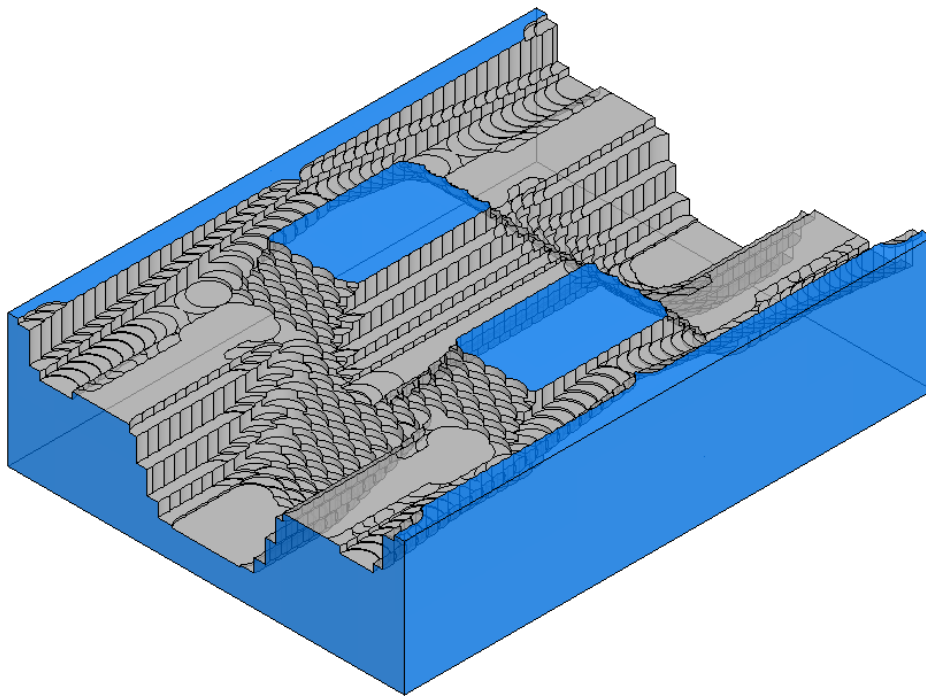
Check allowance =
Finish allowance = 0.0500
* Next Row Stepover % = 33
Plunge feed override % = 50.0000
Priority =
* Rough pass stepover % = 33

- Set the **Z start** to **-0.1"**.

By pulling 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.

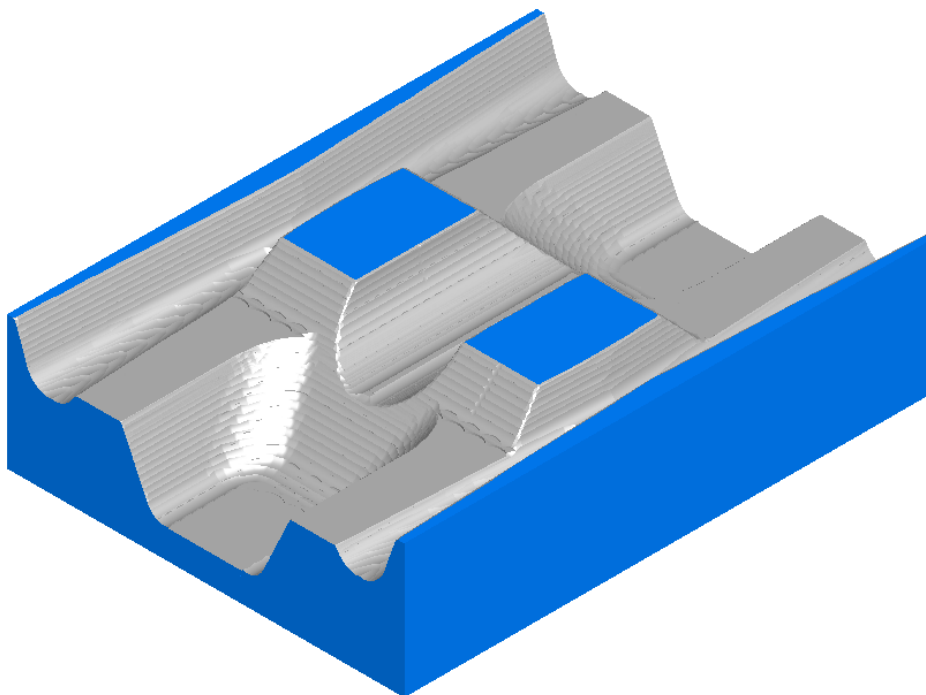
- Click **Apply** and then **OK**.

- Run a **3D Simulation**.



It can be seen that the reduced **Next row stepover %** has removed a lot more material from the part giving a closer approximation to the finished part. In most cases plunge roughing will be followed by a semi-finishing operation to even out the steps in the remaining material on the part.

- **Open** the file **Die\_Block\_Fin.fm**
- **Options, Simulation, 2D/3D Shaded tab**, turn off **Translucent part**.
- Run a **3D Simulation**.



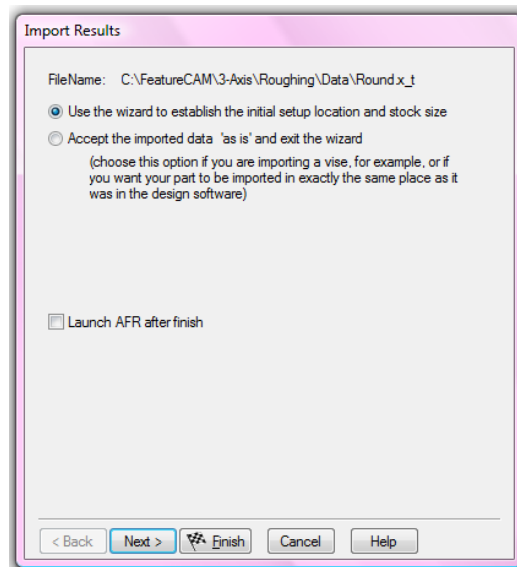
A **Z-Level finish** operation with a ball nosed cutter has been used with a leave allowance to semi-finish the part by removing the steps left by the plunge roughing operation.

# Stock Types, Total Stock & Boundaries

## Introduction

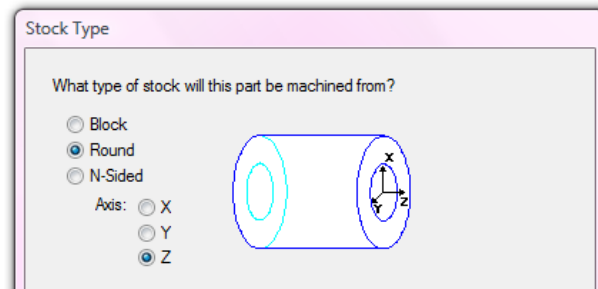
So far in this module we have only considered roughing from a rectangular block of material. Often in practice we may have material that is round or multi-sided, has been pre-machined, or may be a fabrication or a casting. We will now examine the different options available for defining the type of stock 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 an **Inch 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**.

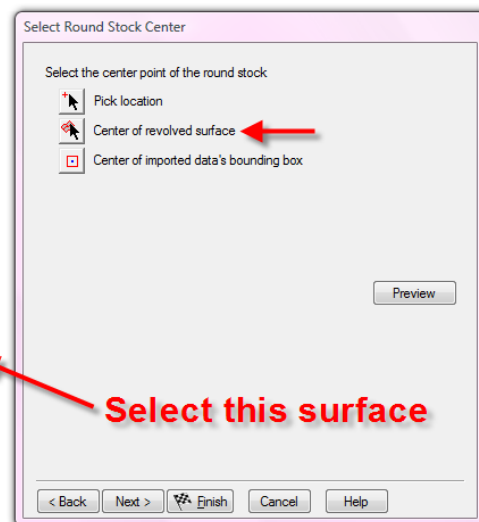
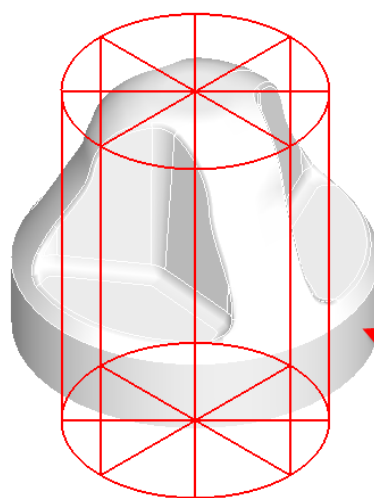


In this case our part is round and has its axis pointing up the Z axis. We now need to ensure that the round stock is centered on our part.

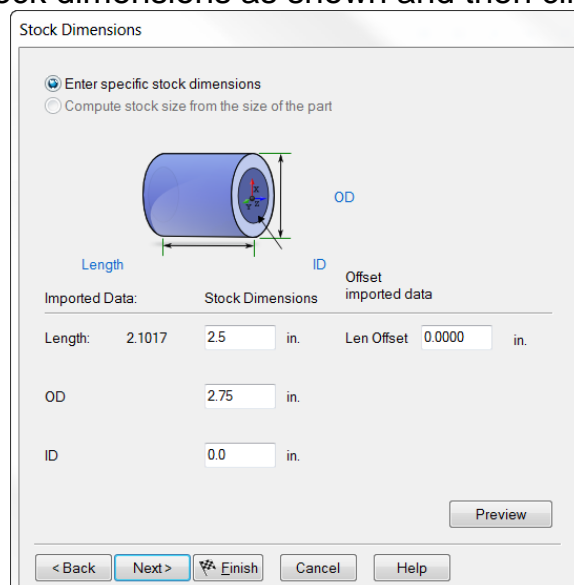
- Click on **Center of revolved surface** button.
- Click on the surface shown. **Shade** the solid if necessary.

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

- Click **Next**.

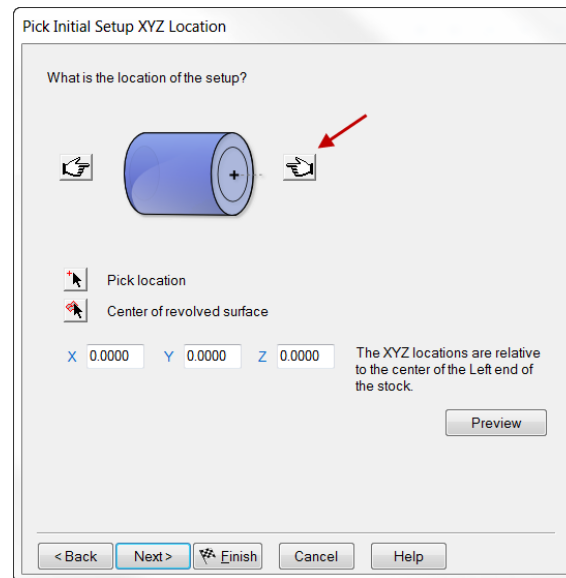


- Fill in the stock dimensions as shown and then click **Next**.



The 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.

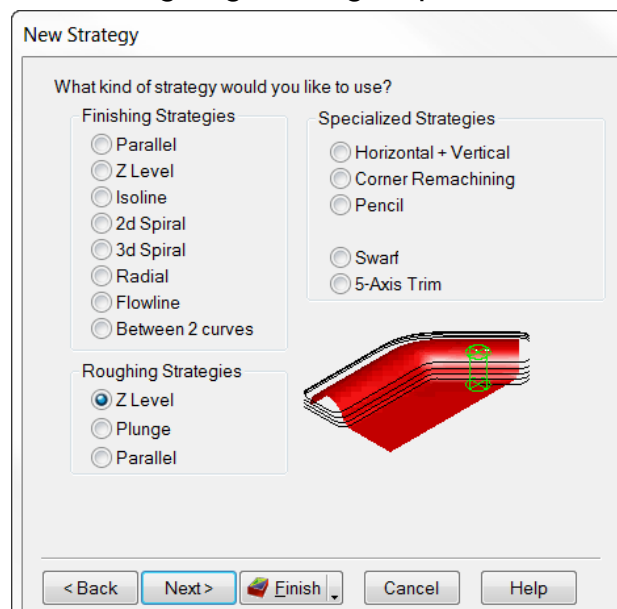


- Click **Finish**.

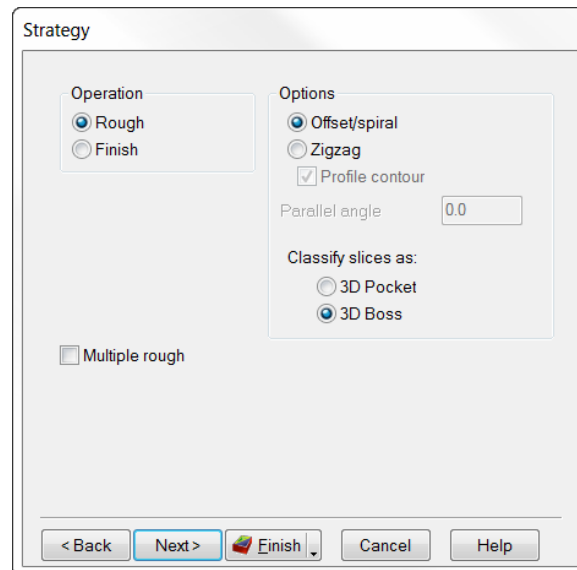
The part is now correctly set up within the resized stock. We will now create a Z Level Offset roughing path to clear the excess material from around the part.

- Click on **Features** in the **Steps view**.
- Check **Surface Milling** and press **Next**.

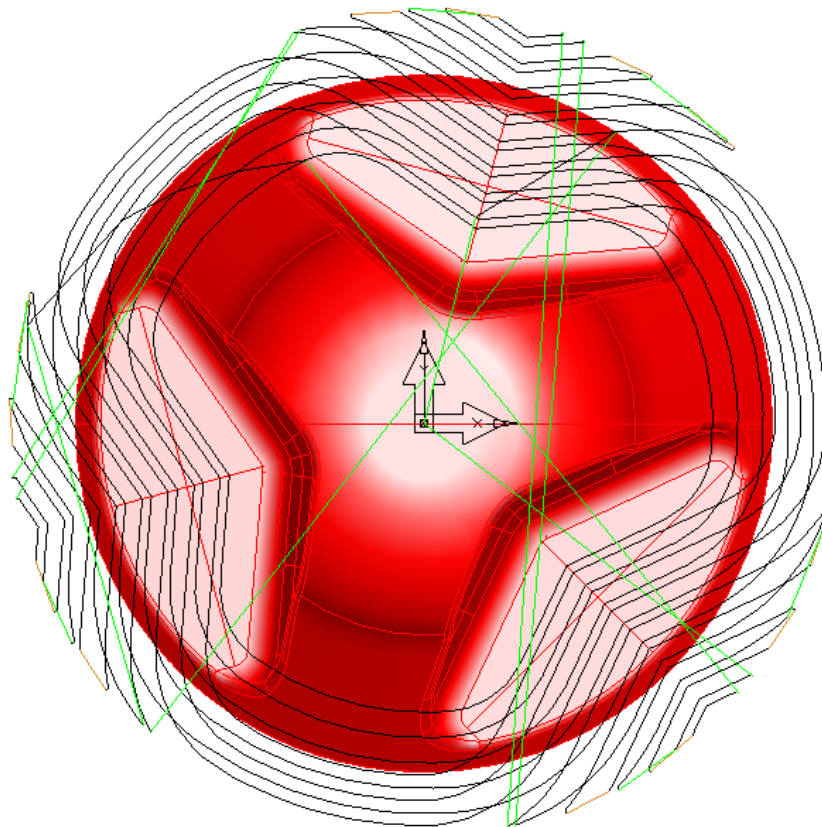
- Box select** all of the surfaces and then press the **Add from selected items button**  then click next.
- On the New strategy form check **Choose a single operation**.
- Click **Next**.
- Check **Z Level** in the Roughing Strategies part of the form.



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



- Run a **Centerline** simulation.
- Change to a **Top View**.

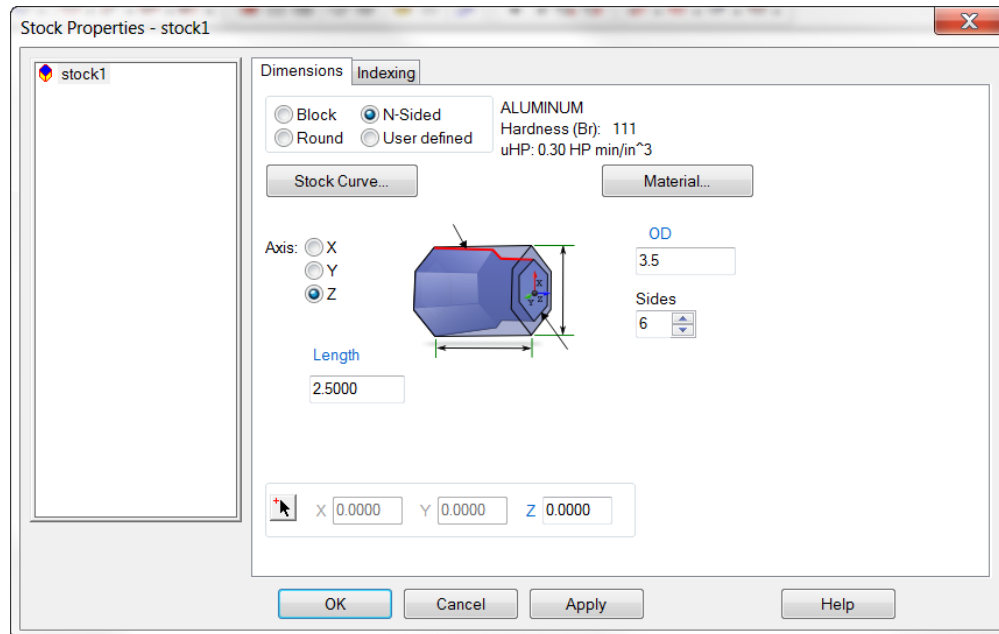


Note how the roughing passes are now confined to a cylindrical area around the part.

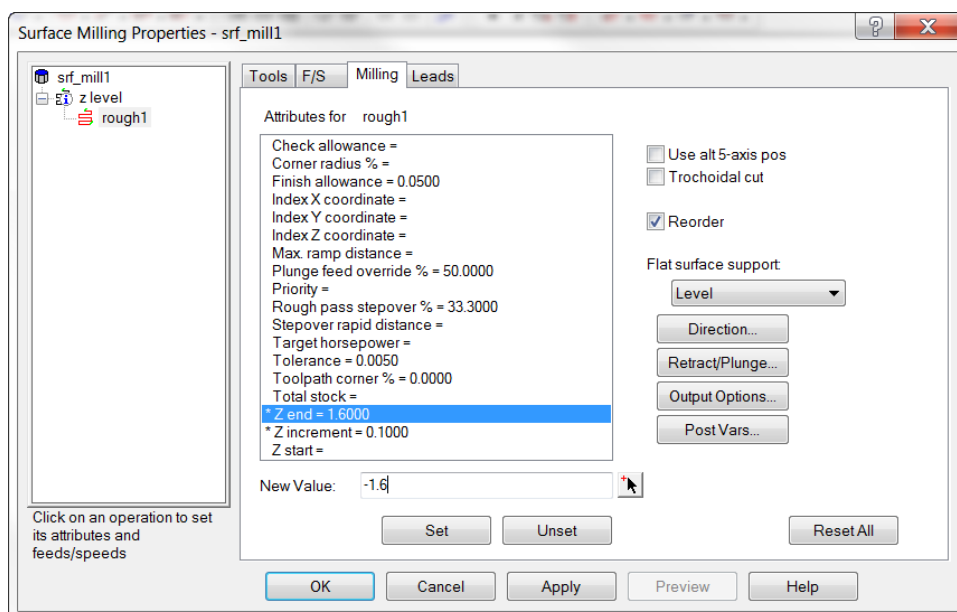
## N-Sided Stock

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**.

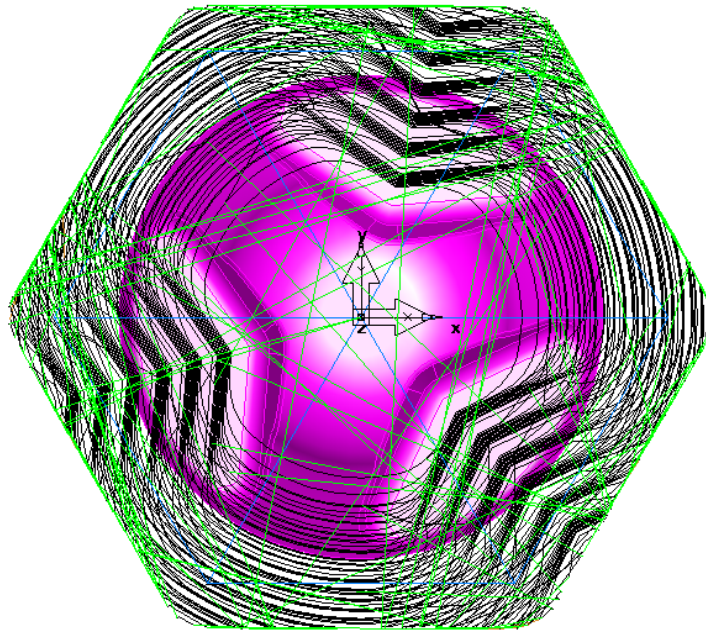


- Edit the surface mill feature, select **rough1** and then the **Milling** tab.
- Set the **Z increment** to **0.1** and **Z End** to **-1.6** as shown. **Apply** and **OK**.



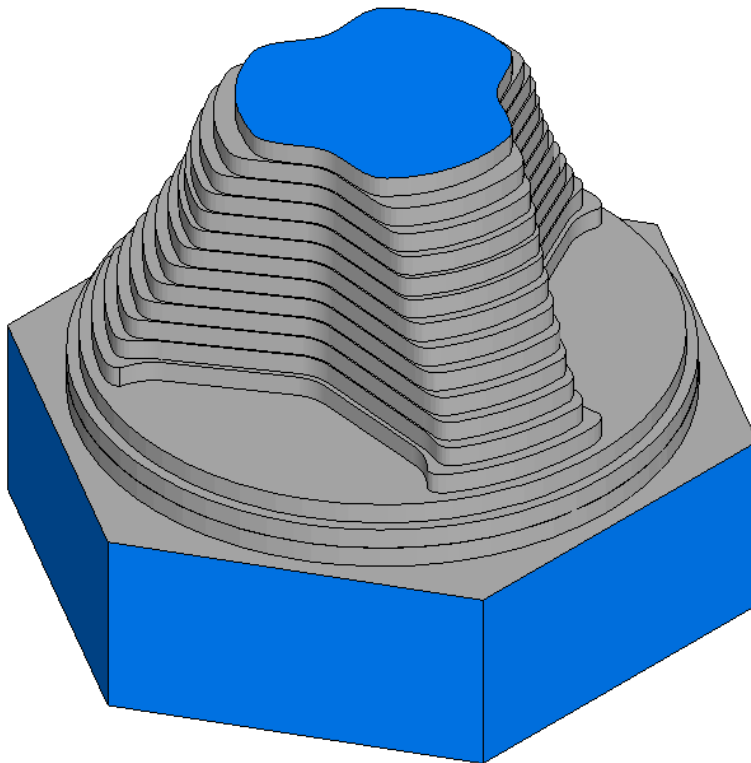
- Run a **Centerline** simulation.
- Change to a **Top View**.





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.

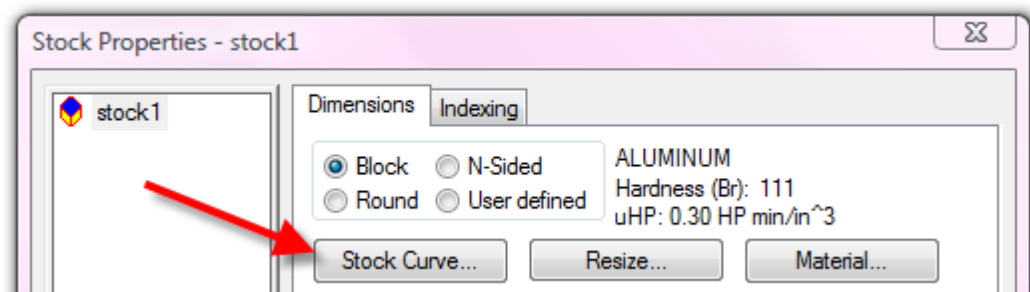
## Stock Curve

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.

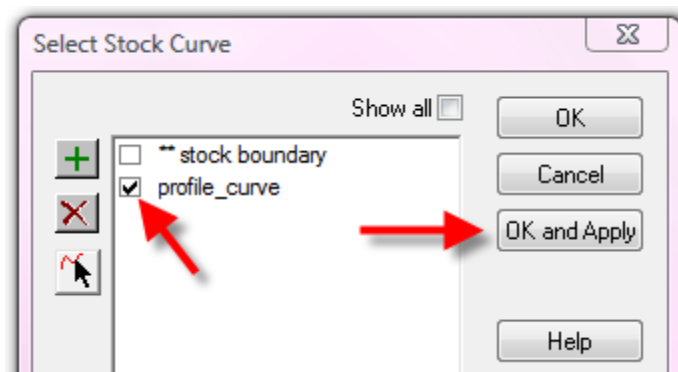
- Open the part **Primary\_Case\_Stock\_Curve.fm**

In the **Stock Properties** dialog you will see that there is a curve in the part called “**curve3**” which is offset by 0.125” from the outline of the model. We will now use this to define the profile of the roughing stock.

- Double click on the **stock1** to edit its properties.
- On the **Stock Properties** form click **Stock Curve**.

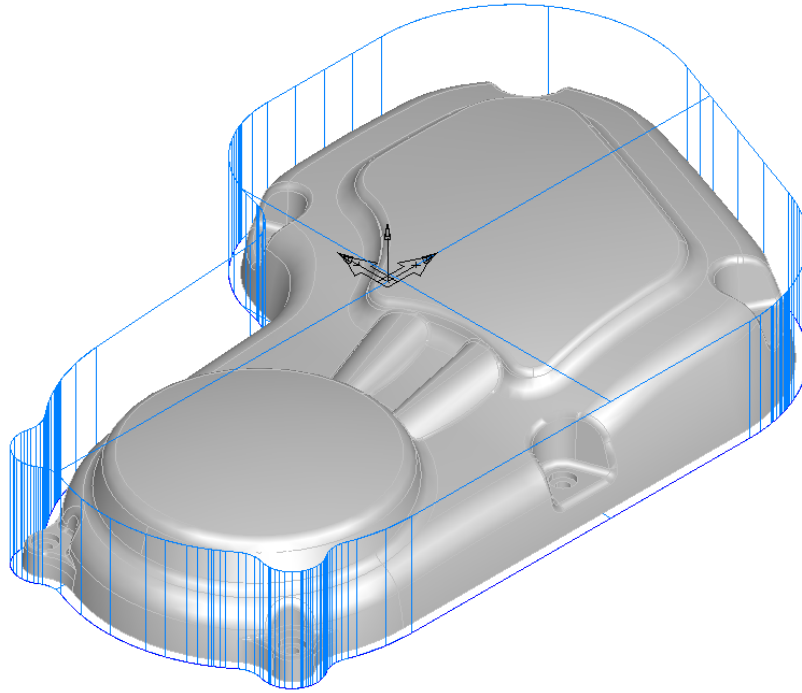


- Check **curve3** and then click the **OK & Apply** button.



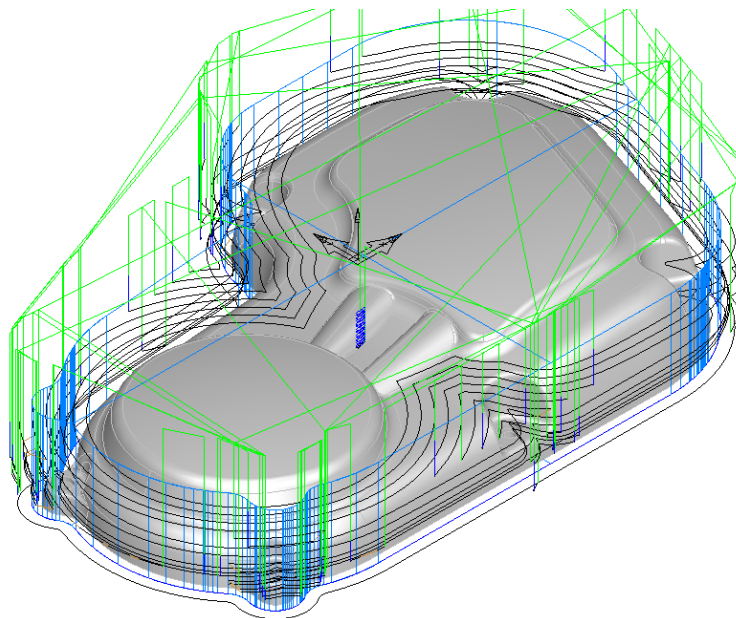
- Click **OK**.
- Change to an **Isometric View**.

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.



- **Box select** all the surfaces.
- Create a **Z Level Rough**. Use **Offset/Spiral** and **Boss** in the strategy.
- **Finish** and **OK**
- Run a **Centerline Simulation**.

Only the area within the stock curve is roughed out.

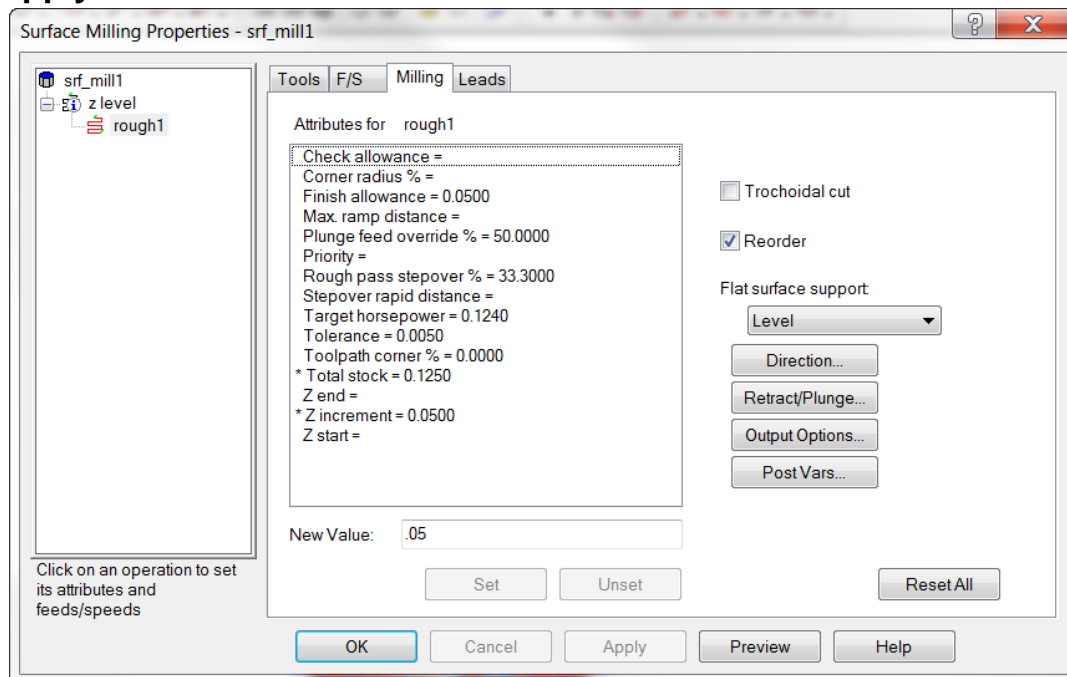


- Run a **3D Simulation**.

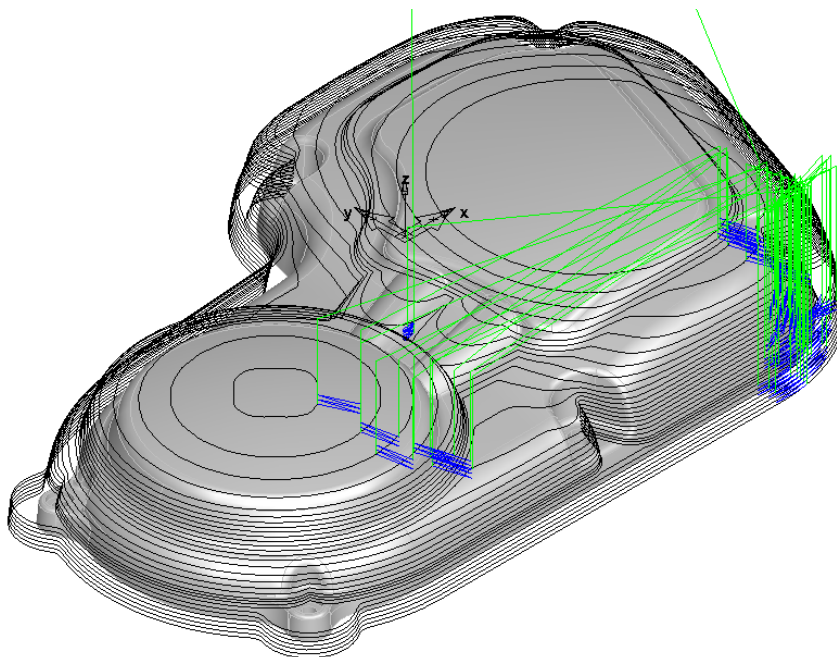
## Total Stock

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.

- Edit the surface mill feature, select **rough1** and then the **Milling tab**.
- Set the **Total Stock** to **0.125"** and **Z Increment** to **0.05"** as shown.
- **Apply** and **OK**.



- Run a **Centerline Simulation**.



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

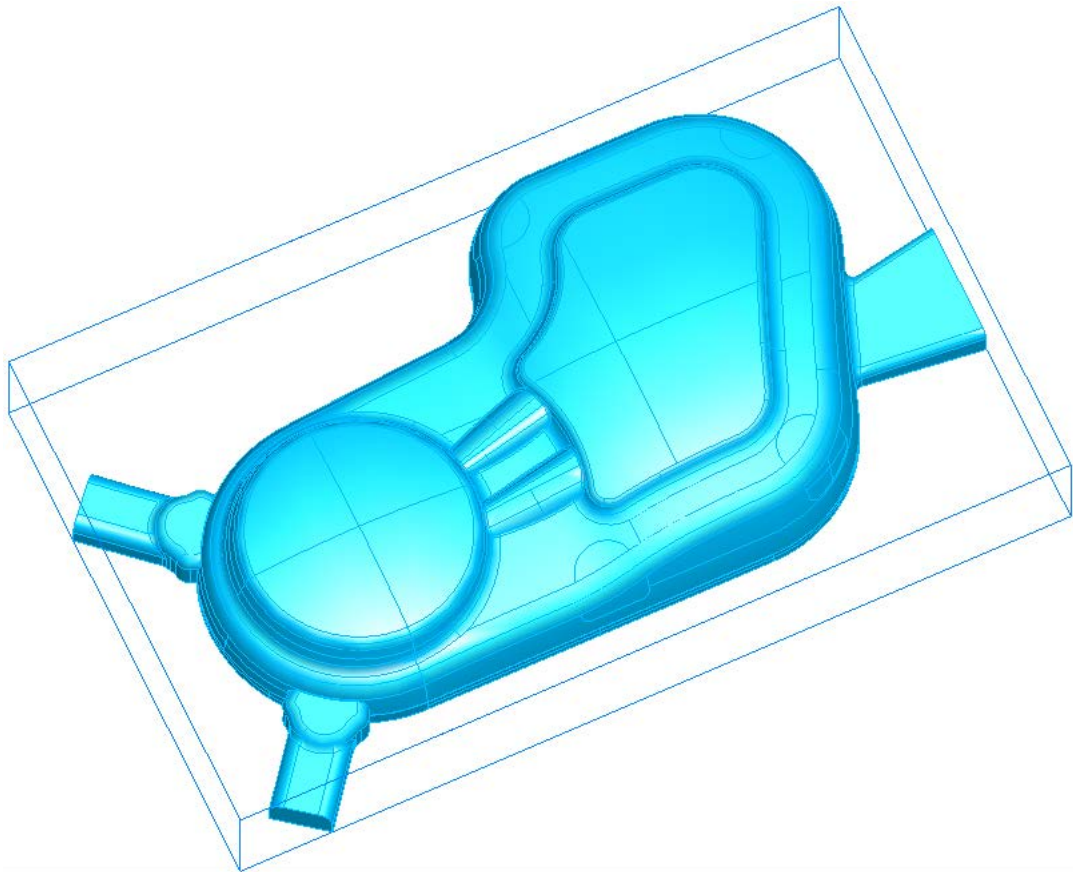
### **Stock Solid**

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.

- Open the part **Primary\_Case\_Stock\_Model.fm**

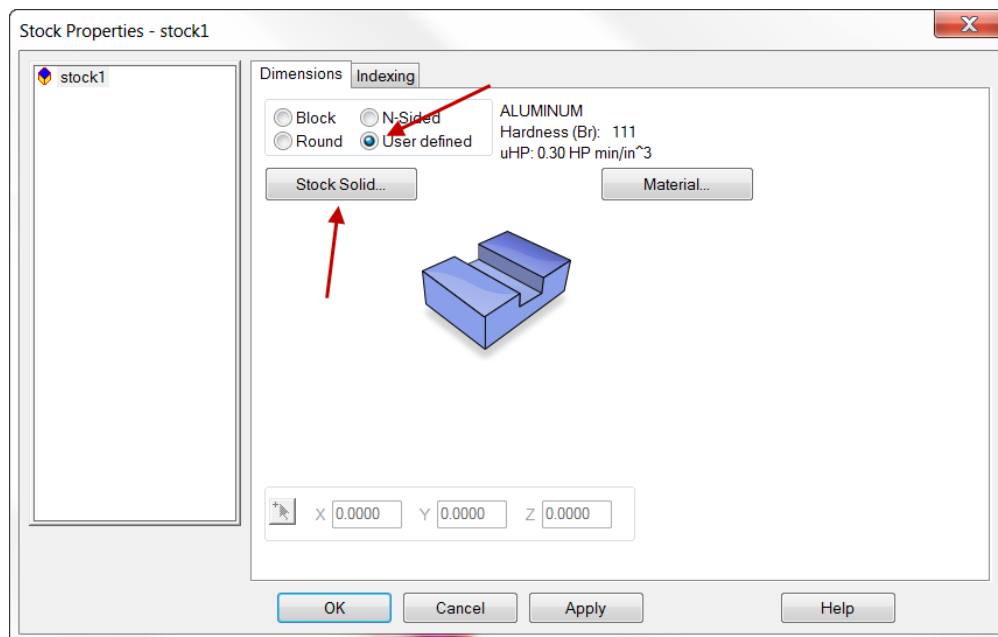
In this part there are two solids, one (part\_model) represents the part to be roughed, and the other (casting) is to be used as the stock material.

- **Show all** and then hide the solid **part\_model**. **Shade** if necessary.

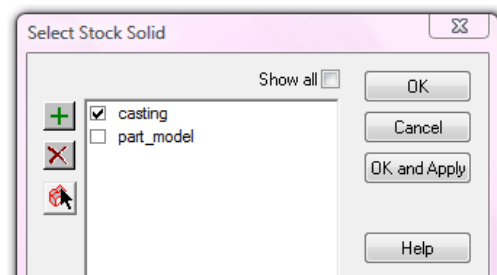


This solid (casting) is the same as the part model except for certain changes; the recesses where the screw holes were have been filled in and casting sprues have been added. The aim of the following roughing process is to clear the area around the screw holes and remove the sprues using the minimum 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

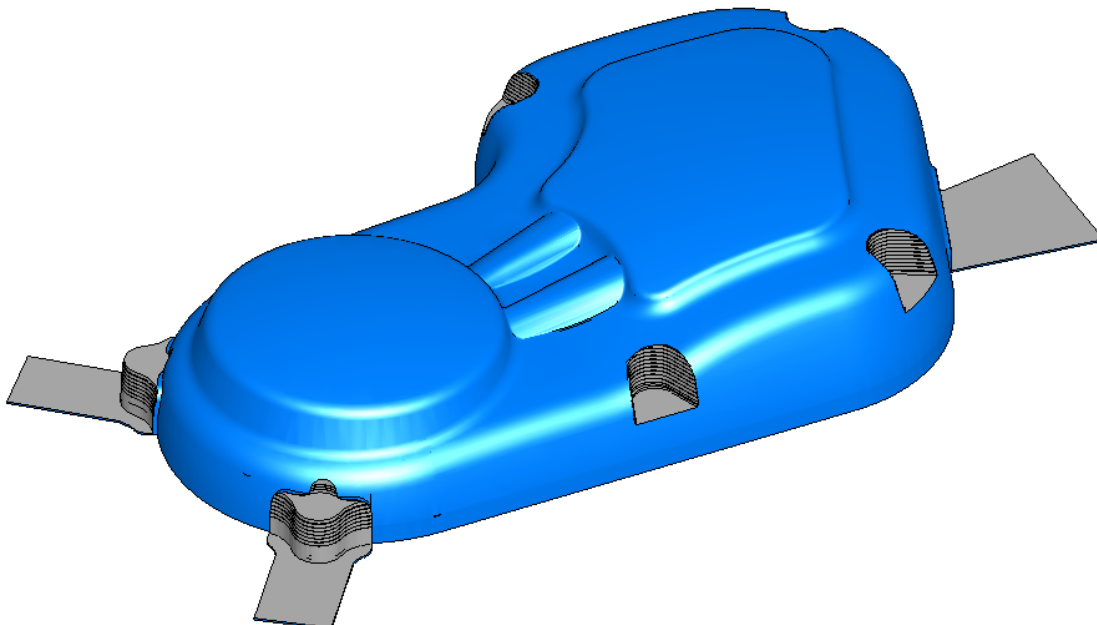
- **Double click** on the **stock**.
- Change the **Stock type** from **Block** to **User defined**.
- Click on **Stock Solid**.



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



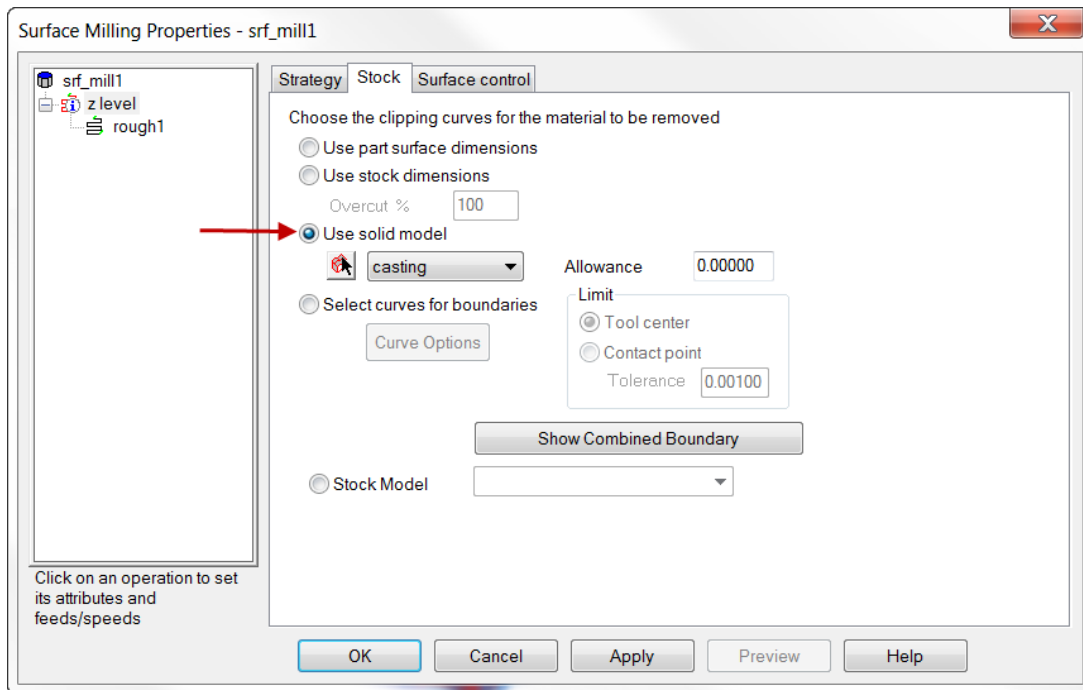
- Run a **Centerline Simulation** then a **3D RapidCut simulation**.



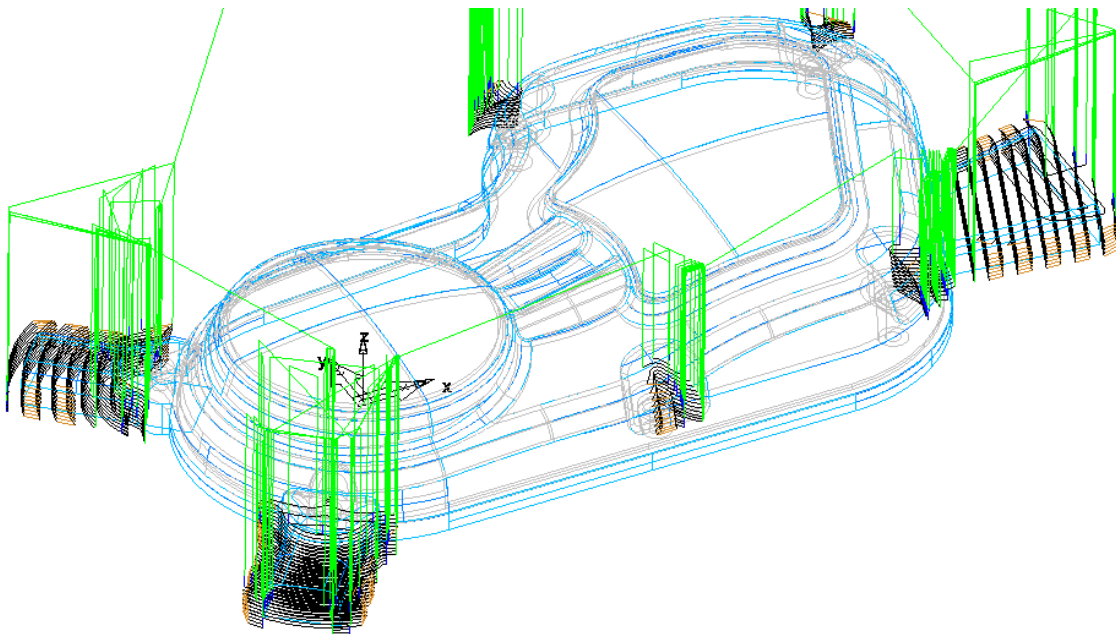


You will see that the stock in the 3D simulation is the solid “casting” but that the actual roughing toolpaths are still sized to the previously defined block. We need to make one more change to allow FeatureCAM to calculate the roughing paths from the stock solid.

- **Eject the simulation.**
- **Edit the feature srf\_mill1.**
- **Select the z level feature and then click on the **Stock** tab.**
- Check **Use solid model** then select **casting** from the pull-down menu.
- **Apply and OK.**



- Run a **Centerline Simulation** then a **3D RapidCut simulation**.

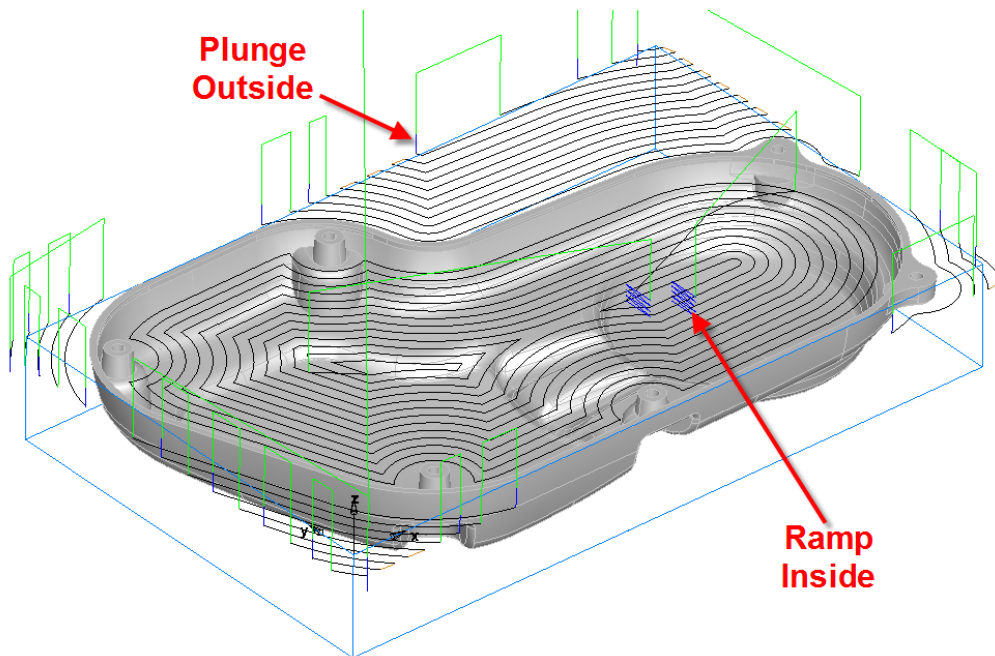


Note how the roughing now machines only those areas between the finished part and the stock solid. This eliminates a lot of air cutting.

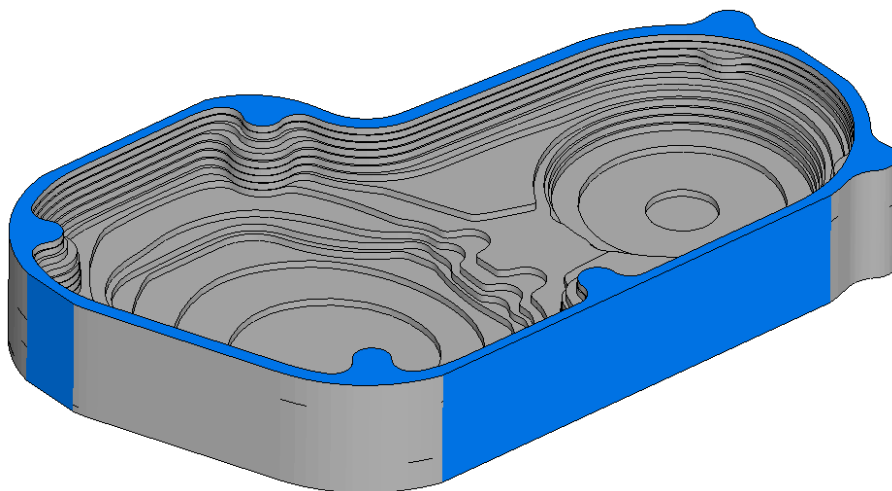
### Pocket/Boss

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.

- Open the part **Primary\_Case\_Pocket\_Boss.fm**
- 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.

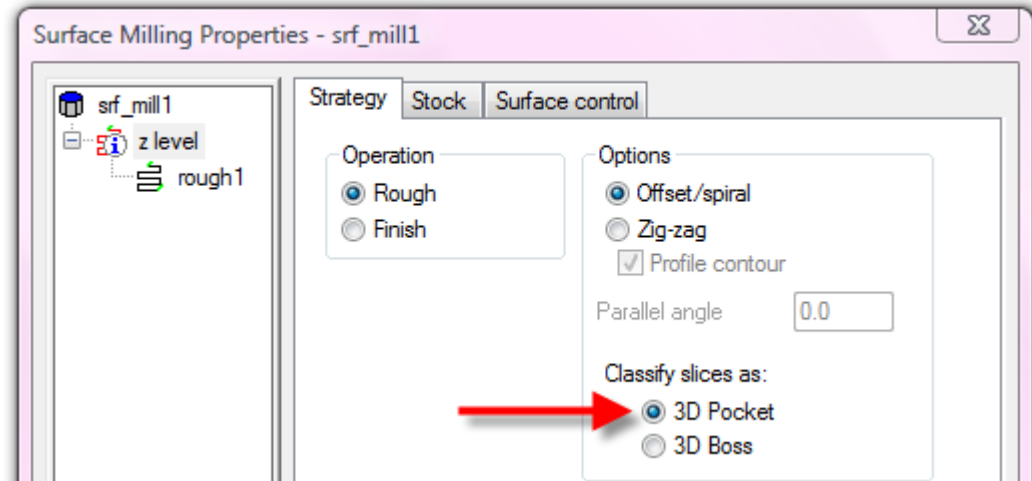


Often when roughing such a part the stock may be clamped on the outside, or the material may have been roughed earlier by cutting from the opposite side of the block. In such a

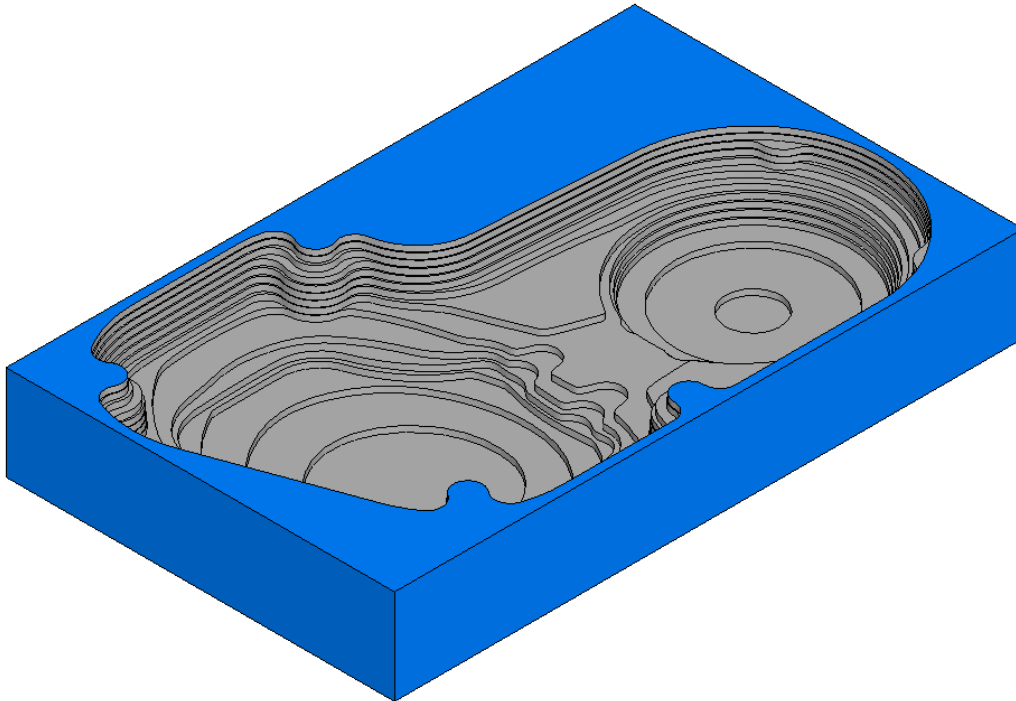


situation the user would probably want to machine only the center pocket of the part and leave the outer area. We can do this by switching from the 3D Boss option to a 3D Pocket.

- **Edit** the feature **srf\_mill1** and click on **z level**.
- Click on the **Strategy** tab and check **3D pocket**. **Apply** and **OK**.



- **Run a 3D Simulation.**



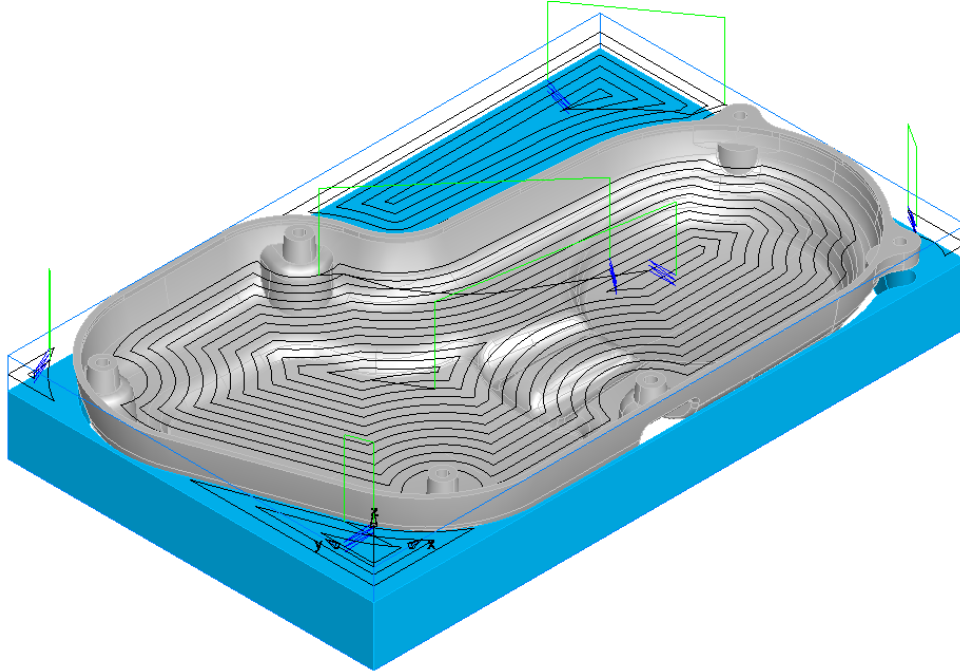
FeatureCAM is now looking for enclosed pockets so the area outside of the part is not machined. This only happens because there are **NO Part Surfaces** selected **outside** of the area to be machined. If any surfaces outside of the pocket area are selected then they would be machined also, however, 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.

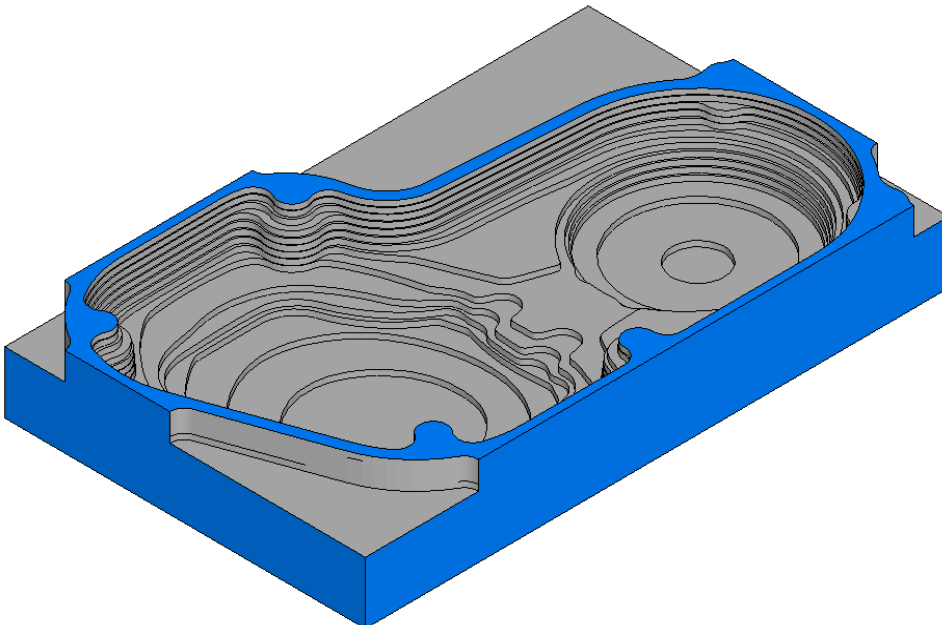
- **Right click** on the solid **Base** in the **Part view** and **Show selected**.
- **Edit** the feature **srf\_mill1** and click on **Part surfaces**.
- **Select** the face **face\_267** and **Add** it to the **part surfaces list**.
- Click **OK** and **Apply** then **OK**.

Note: **face\_267** is the **upper face** of the solid called **base**.

- Run a **Centerline 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**.



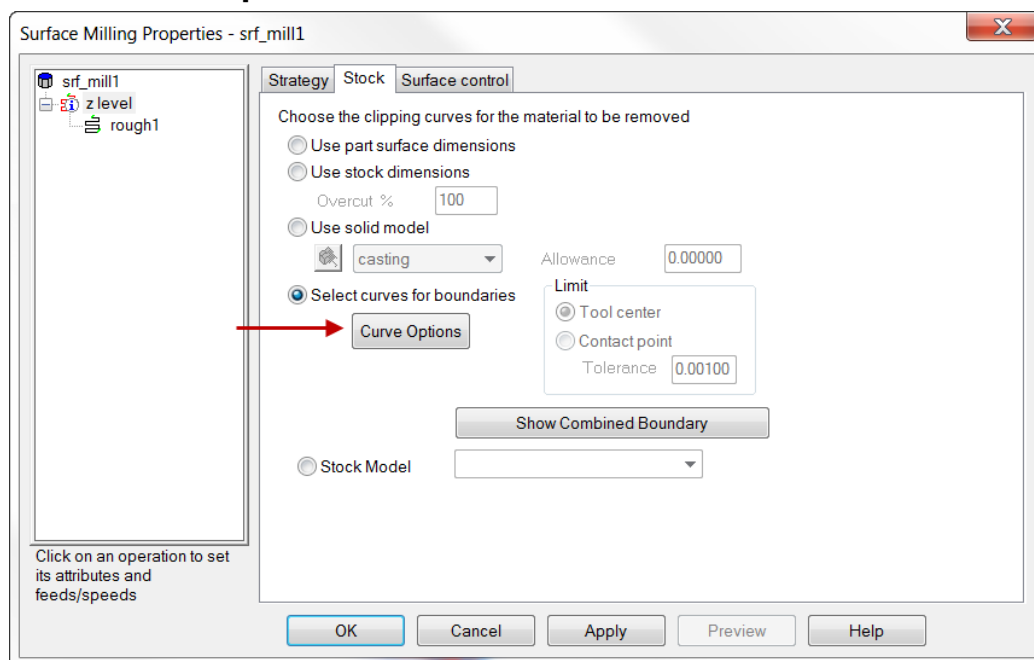
## Use of Boundary Curves

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.

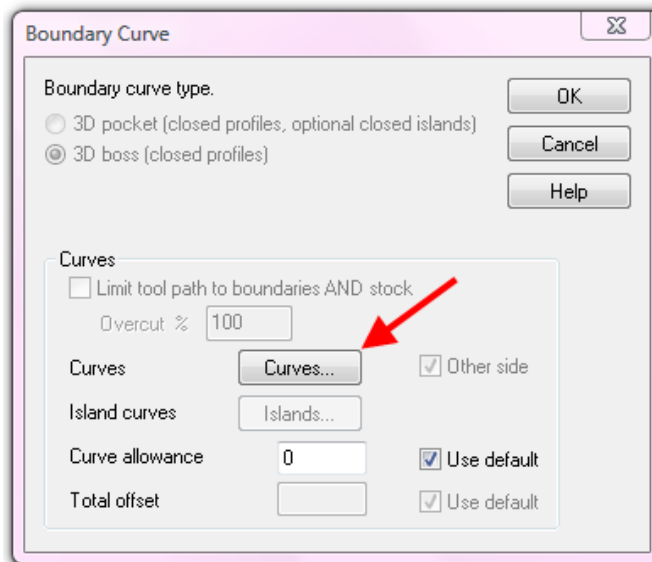
- **Open the part `Primary_Case_Boundaries.fm` from:**
- **Run a 3D simulation.**

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.

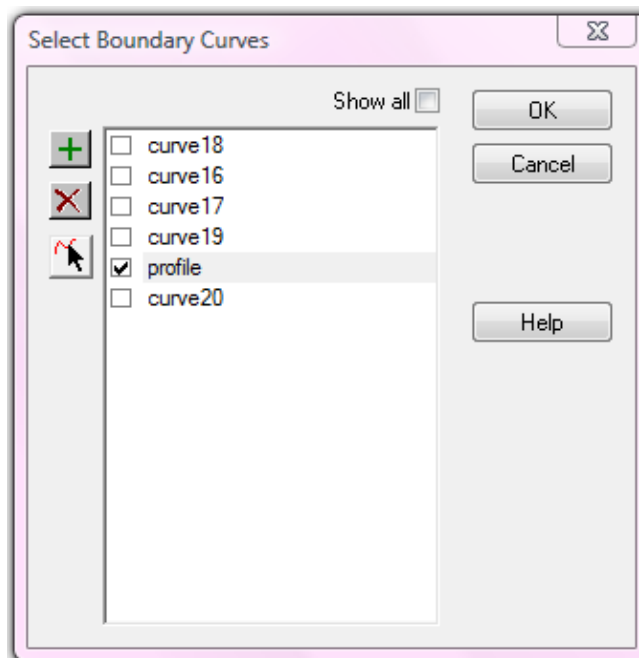
- **Box select** all the surfaces.
- Create a **Z level surface mill feature. Offset/spiral** and **3D Boss**
- **Finish.**
- Select the **Milling tab** (rough1) in the **Surface Milling Properties** dialog.
- Set the **Z Increment** to **0.1”**.
- Select the **Stock tab** (pick z level in the tree).
- Click on **Select curves for boundaries**.
- Click on **Curve Options** button.



- Click on the **Curves...** button.

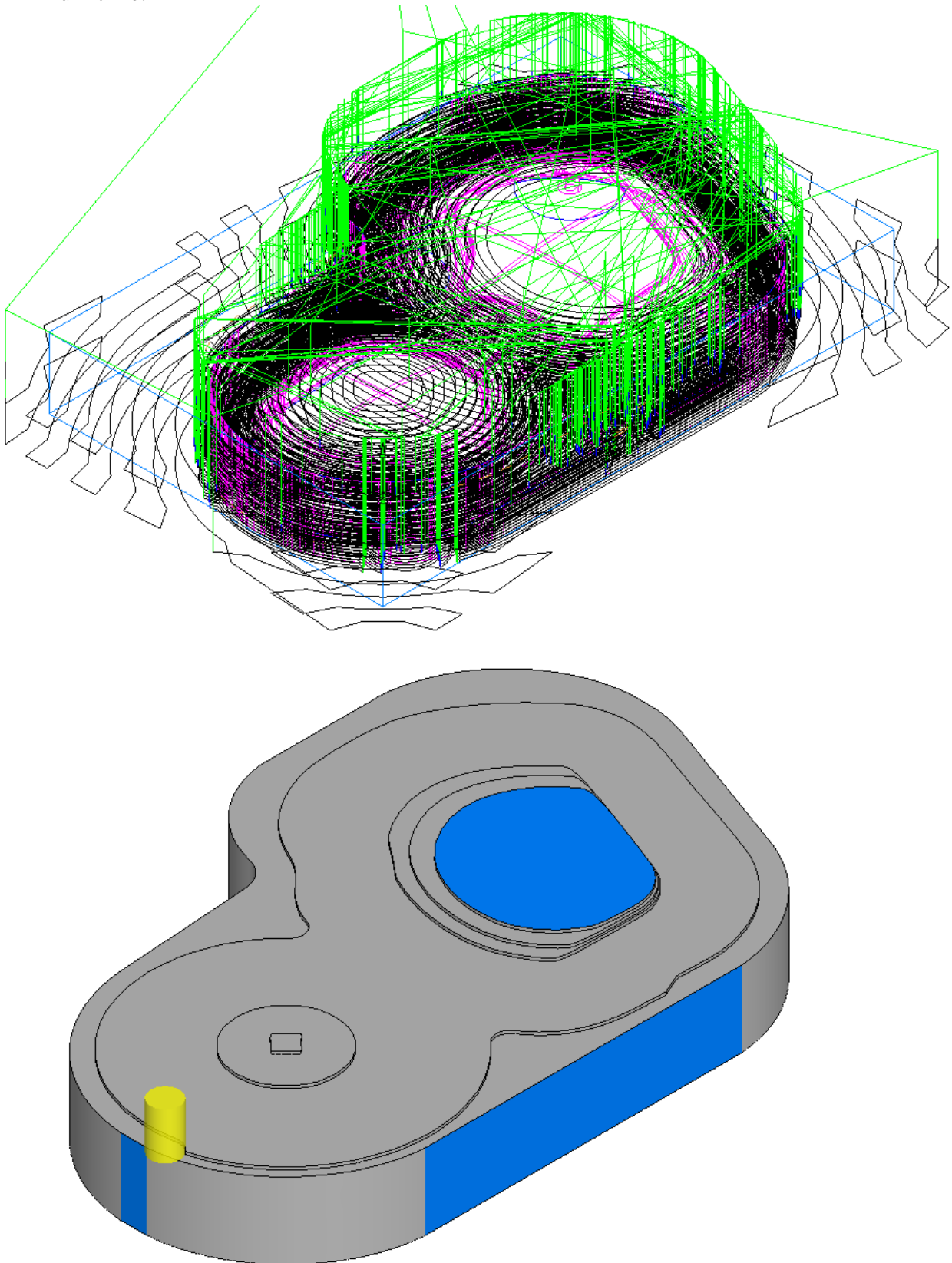


- Select the curve called **profile**.



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



# Stepovers & Leads

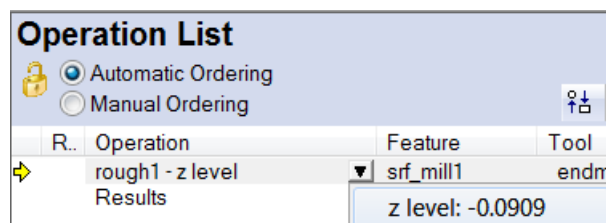
## Introduction

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

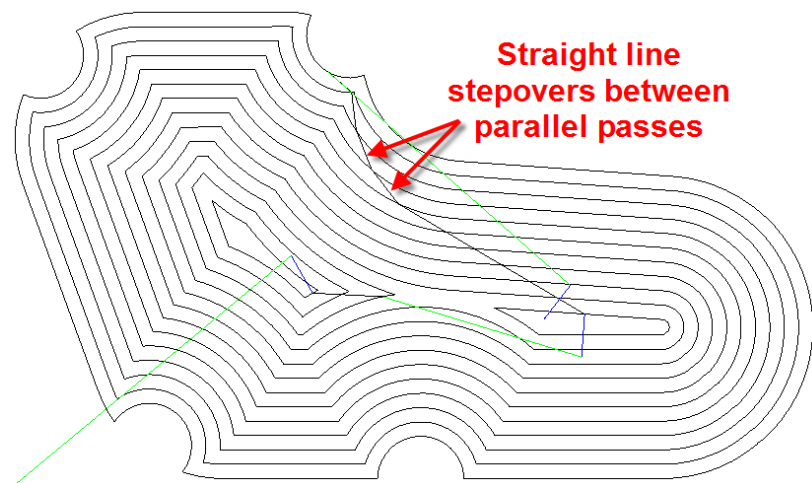
When FeatureCAM clears a **z level** it does so by means of a series of parallel passes, each separated from the next by the **Rough pass stepover %**. The way in which the passes are connected on a z level is controlled by the **Stepover type** attribute on the **Leads tab** of the roughing feature.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

- Open the part **Primary\_Case\_Pocket\_Leads.fm**
- Run a **Centerline Simulation**.
- In the **Operation List** click on the pulldown next to **rough1 – z level**.



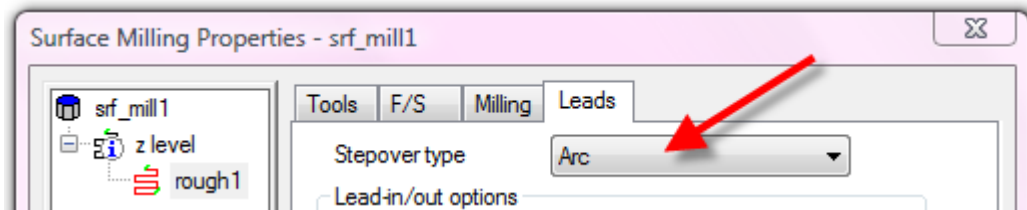
- Select the **first z level**.
- **Hide all features** and select a **Top view**.

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.

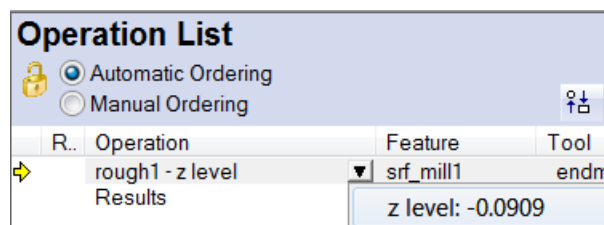


We will now change the stepover type from Direct to Arc. In an Arc stepover the tool makes a smooth circular arc move off one pass and then leads into the next pass with a second circular arc. This minimizes the rate of change in direction of the tool, and also reduces the changing of the tool load as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

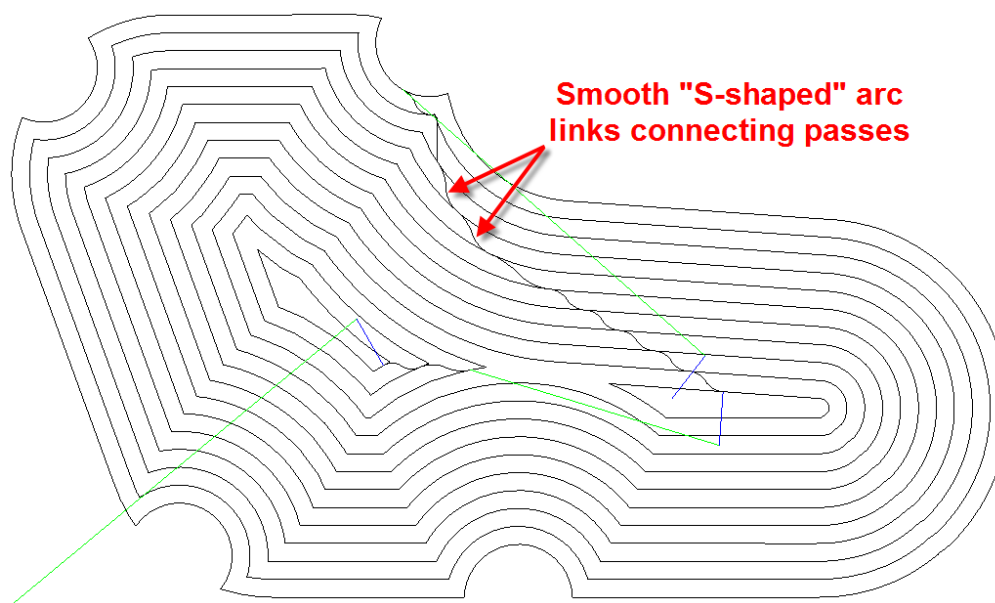
- **Eject the simulation.**
- **Edit the feature `srf_mill1` and click on `rough1`.**
- **Select the `Leads` tab.**



- Change the **Stepover type** to **Arc**. **Apply** and **OK**.
- Run a **Centerline Simulation**.
- In the **Operation List** click on the pulldown next to **rough1 – z level**.



- Select the **first z level**.
- **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

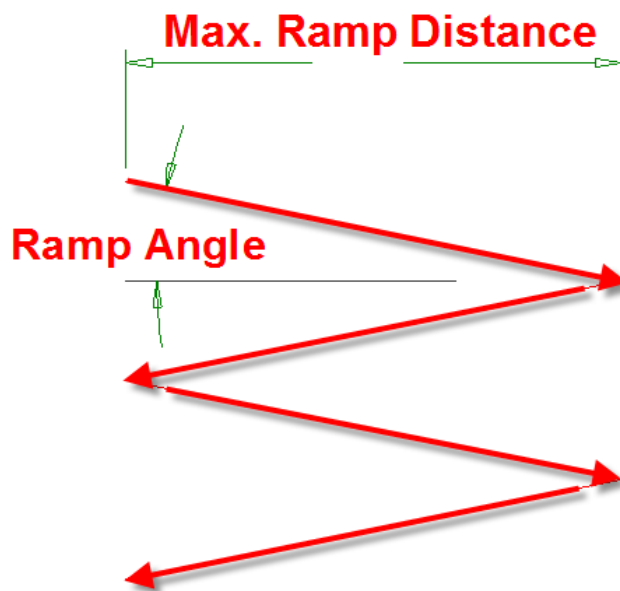
When FeatureCAM roughs out a pocket, by default, all approach moves to the part will be ramps. That is, instead of plunging the tool directly into the material the approach will be made by a series of zigzag moves which progressively move the tool down into the material. We will now examine how these 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 center cutting tool it may not be necessary to ramp at all whereas an insert cutter may require a shallow ramp angle.

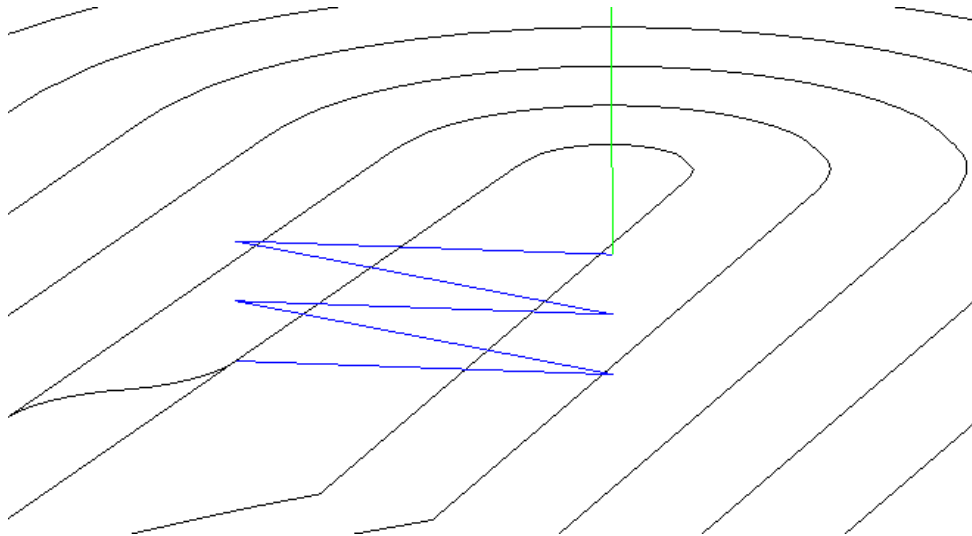
The **Maximum ramp distance** controls the length of the Zigs and Zags in the ramp. If no value is explicitly set by the user then FeatureCAM will base the maximum ramp distance on the tool diameter. The user can also choose to enter a value based upon the cutter geometry so as to prevent build-up of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



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

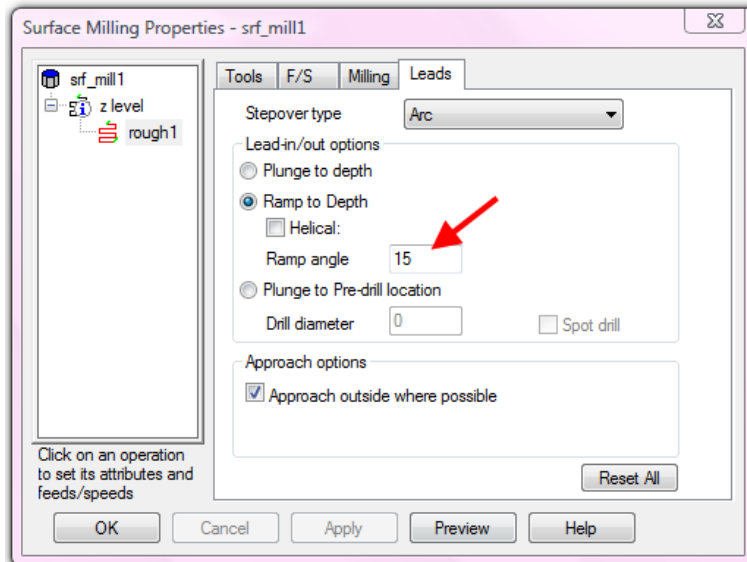
- Run a **Centerline simulation**.
- Click on the pulldown next to **rough1** in the **operation list**.
- Select the first roughing level. Change to an **Isometric** view.





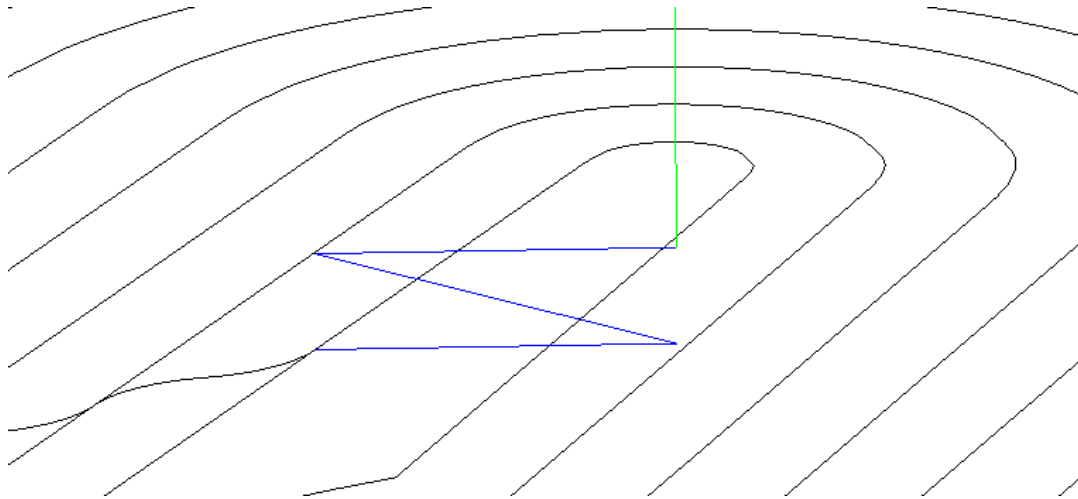
The tool approaches the material by making a series of zigzag moves at the default angle of five degrees to the horizontal plane.

- **Eject the simulation.**
- **Edit the feature `srf_mill1` and click on `rough1`.**
- **Select the `Leads` tab.**
- **Set the `Ramp angle` to `15 degrees`.**



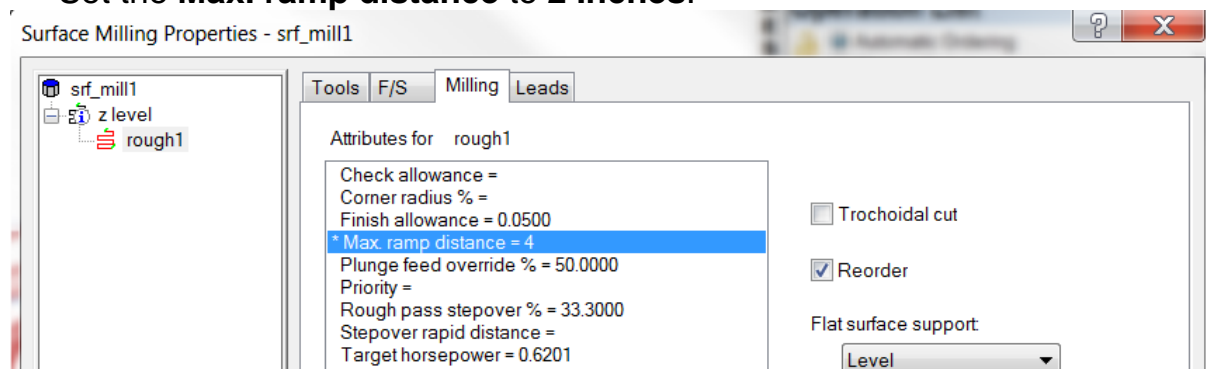
- **Run a `Centerline` simulation. `Apply` and `OK`.**

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

- **Eject the simulation.**
- **Edit the feature srf\_mill1** and click on **rough1**.
- Select the **Milling tab**.
- Set the **Max. ramp distance** to **2 Inches**.

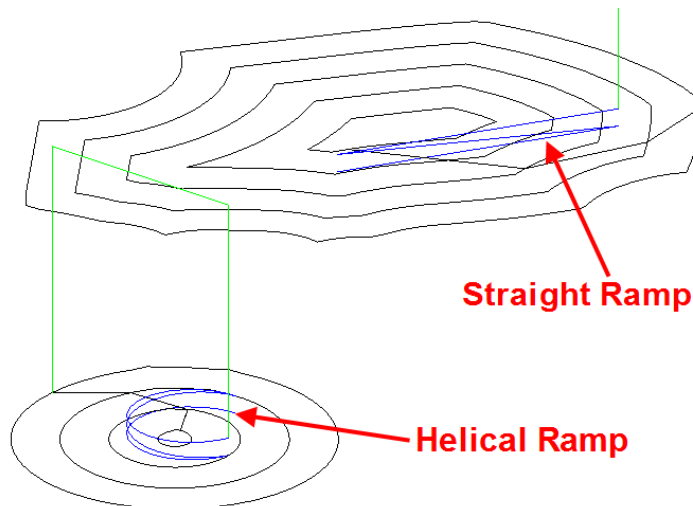


Click **Apply** and **OK**.

- Run a **Centerline simulation**.
- Click on the pulldown next to **rough1** in the **operation list**.
- Select the **First** roughing level.

Note how the length of the ramping moves has increased from the default of the tool diameter to two inches.

- Click on the pulldown next to **rough1** in the **operation list**.
- Select the roughing level near **Z -0.850**.



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 two inches so FeatureCAM automatically switches to using a helical ramp.

It is possible to force FeatureCAM to use a Helical ramp in all cases:

- **Eject the simulation.**
- **Edit** the feature **srf\_mill1** and click on **rough1**.
- Select the **Leads** tab.
- Check the **Helical** check box below the **Ramp to depth** wording.
- **Apply** and **OK**.
- Run a **Centerline** 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.

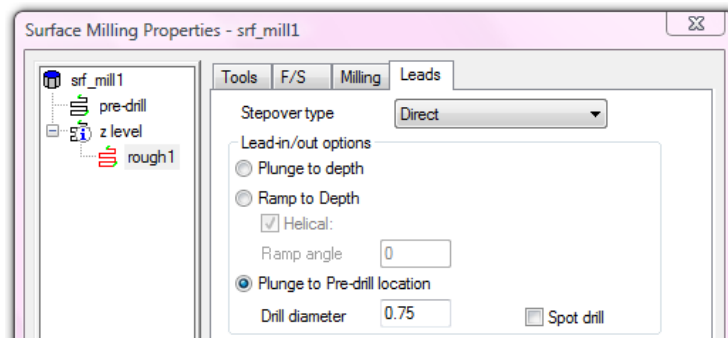
- **Eject the simulation.**
- **Edit** the feature **srf\_mill1** and click on **rough1**.
- Select the **Milling** tab.
- Change the **Max. ramp distance** to **0.25"**. **Apply** and **OK**.
- Run a **Centerline** simulation.

Note how the diameter of the helical ramp is reduced.

## Plunging to Pre-drilled holes

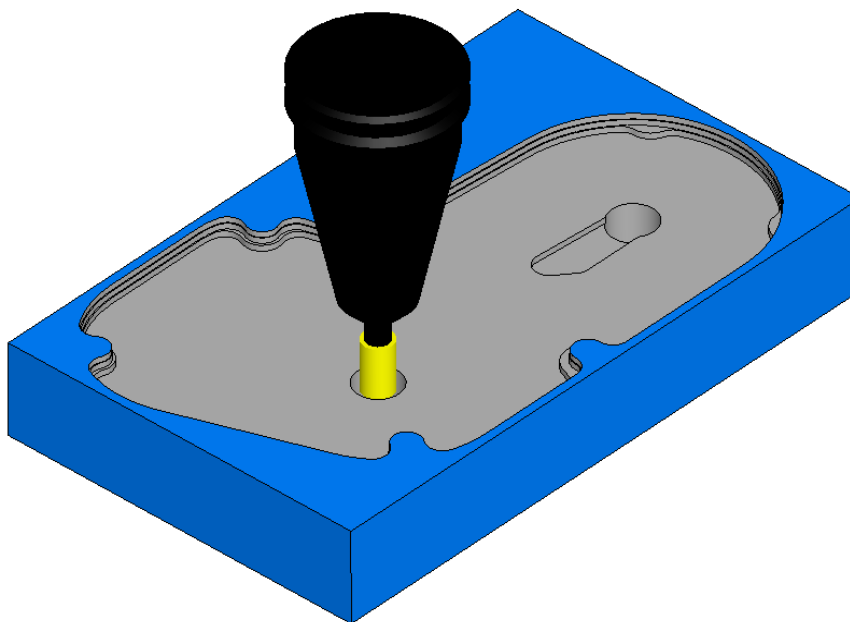
If a roughing cutter does not center cut ramping moves can be used to allow the cutter to engage the material without causing damage to either the cutter or the part. The main disadvantage of ramps is that the cutter is moving at a reduced feedrate and making unproductive moves. Where a part has many 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.

- **Eject the simulation.**
- **Edit the feature `srf_mill1` and click on `rough1`.**
- **Select the `Leads` tab.**
- **Check `Plunge to pre-drill location`.**
- **Set the `Drill diameter` to `0.75"`. **Apply** and **OK**.**



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

- **Use `Play to next rapid` in a 3D simulation.**



The tool plunges into the center 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.

---

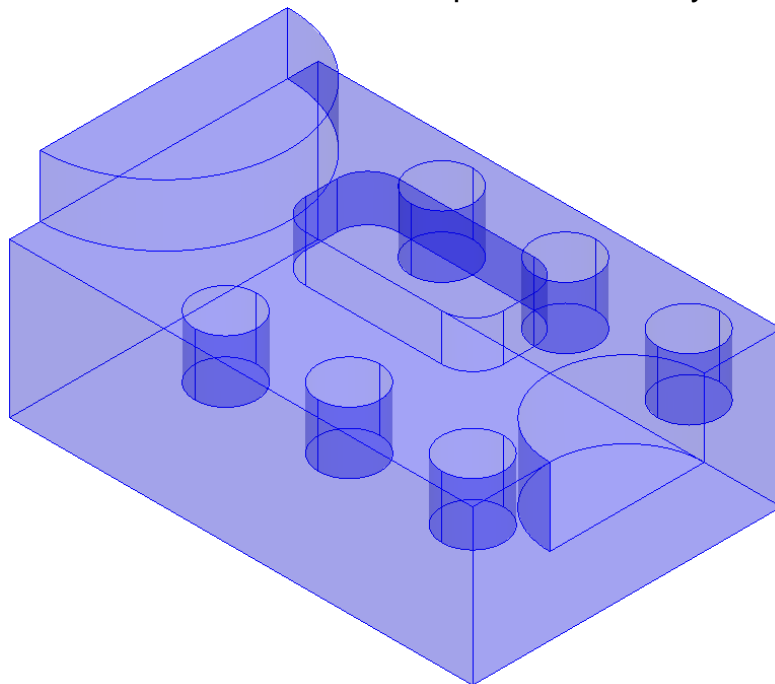
# Roughing Flats

---

## 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.

- Open the file **Flats.fm**.
- Select an **Isometric** view. **Shade** the part if necessary.



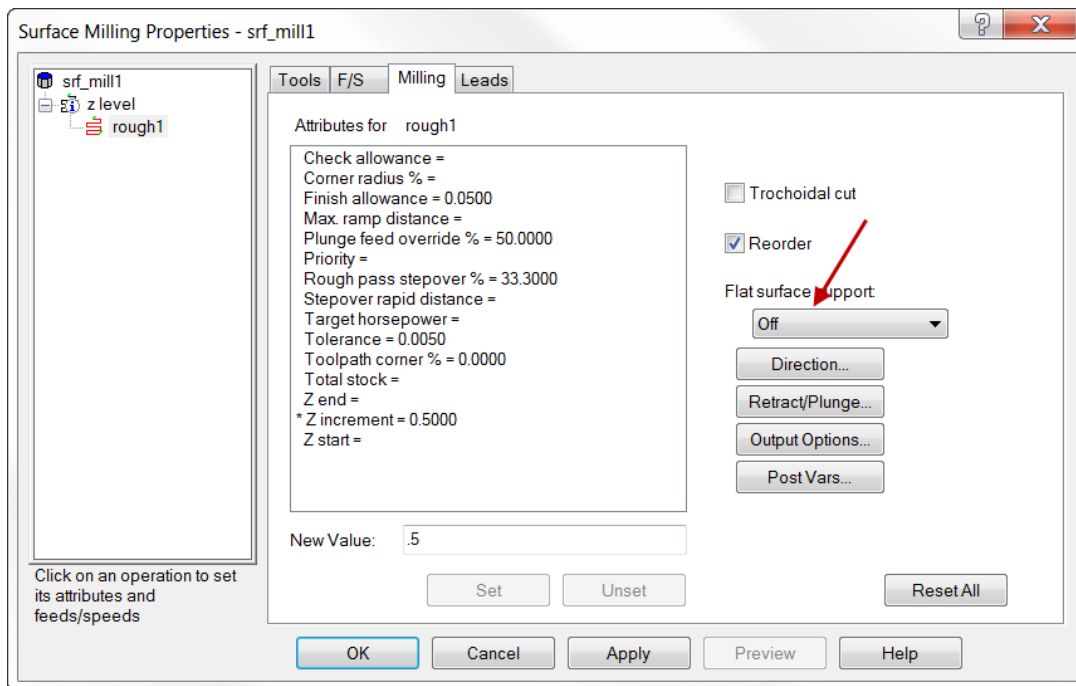
This model contains several horizontal flat areas. If these areas are not machined there may be an excess of stock remaining for semi-finishing process to work well. We will now look at how to control the machining of flats.

- Click on **Features** in the Step 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** dialog check **Choose a single operation**.
- Click **Next**.
- Check **Z Level** in the **Roughing Strategies** part of the dialog.
- Press **Finish**.
- Edit the feature.

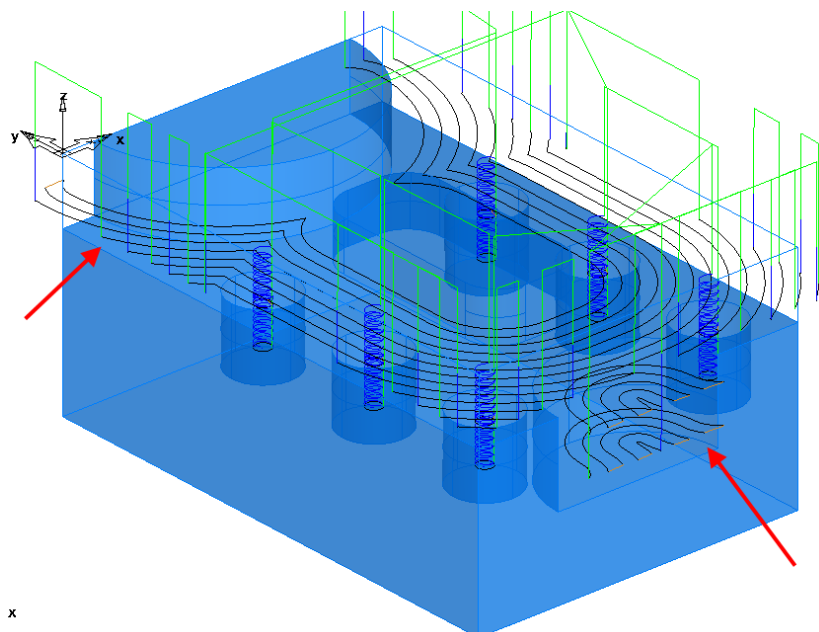
- Click on **rough1** in the tree view.
- Select the **Milling** tab.
- Fill in the form exactly as shown below, setting **Flat surface support** to **Off** and setting the **Z increment** to **0.5"**.



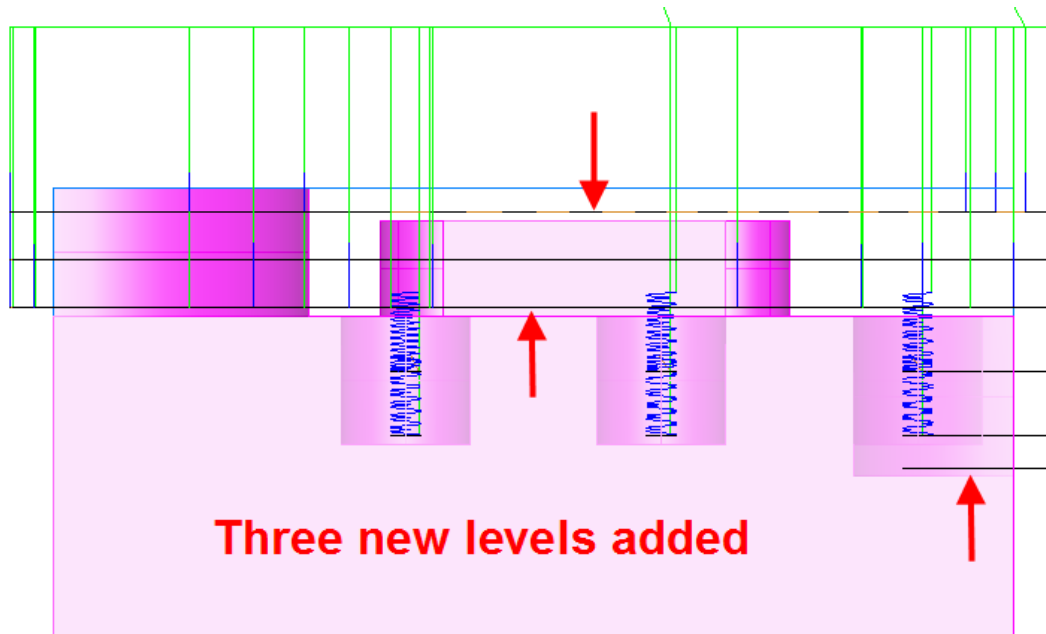
- Press **Apply** and **OK**.
- Click **Manufacturing** on the main menu.
- Choose **Post Process**.
- Set the **Tool change location Z** to be **4"**. Click **OK**.
- Run a **Centerline** simulation.

As Flat surface support is turned off, FeatureMILL3D has ignored all of the flat surfaces and cut only at the specified Z levels (every 0.5" 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.



- Edit the feature.
- Click on **rough 1** in the tree view.
- Select the **Milling tab**.
- Change **Flat surface support** to **Level**.
- Click **Apply** and **OK**.
- Run a **Centerline simulation**.
- Select a view from the **Left**.

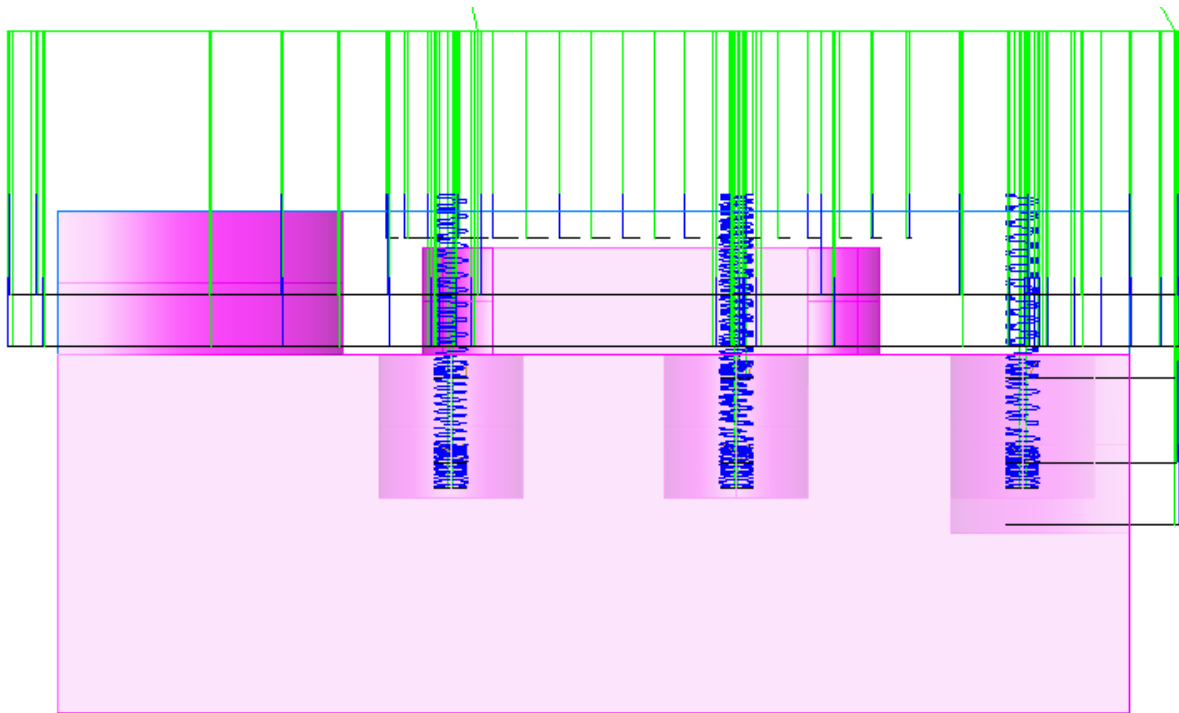


Three extra roughing levels have been added. These are offset from the flat surfaces by the Finish Allowance plus the Rough Tolerance. The levels extend all the way to the edge of the block thus increasing the machining time. If a part has many flats the machining time may become very long.

- Select an **Isometric** view.
- 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.

- Edit the feature.
- Click on **rough 1** in the tree view.
- Select the **Milling tab**.
- Change **Flat surface support** to **Area**.
- Click **Apply** and **OK**.
- Run a **Centerline simulation**.
- 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 toolpath.

- Select an **Isometric** view.
- Run a **3D Simulation** to see how the flat levels are cleared only in the required areas.



# 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 this, High Efficiency roughing, High Speed roughing and Plunge roughing.

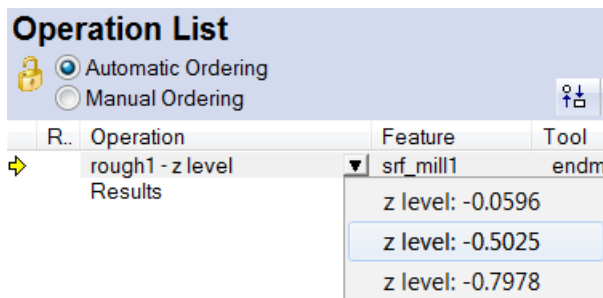
In high efficiency roughing special tooling is used to take very heavy cuts. This removes the maximum possible amount of material per pass. Due to the heavy cuts it is important to maintain a steady load on the tool. FeatureCAM uses the trochoidal milling method in areas where a tool overload condition 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 feedrates, 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.

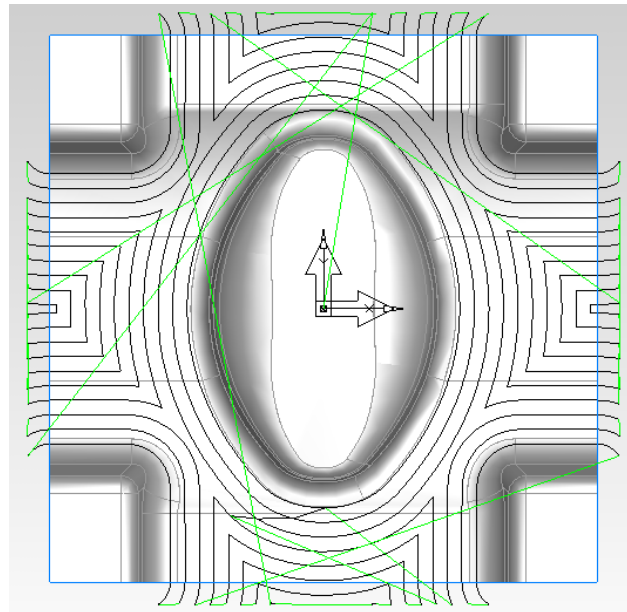
Plunge roughing roughs a part by a series of drilling moves using a large cutter. Specialized tooling is required. As the machine is only moving in Z it is more rigid and hence more aggressive feeds can be used. Plunge roughing is particularly suited to deep pockets. With conventional cutting the feed 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



- Open the file **Fast\_Rough.fm**.
- **Box select** all the surfaces.
- Create a **Z Level** roughing feature. Use **Offset/Spiral** in the **Strategy**.
- Click **Finish**.
- Click on **rough1** and select the **Milling** tab.
- Change the **Rough pass stepover %** to **15%**. **Apply** and **OK**.
- Run a **Centerline simulation**.
- In the **Operations List**, click on the down arrow next to the roughing operation and select the second roughing level.



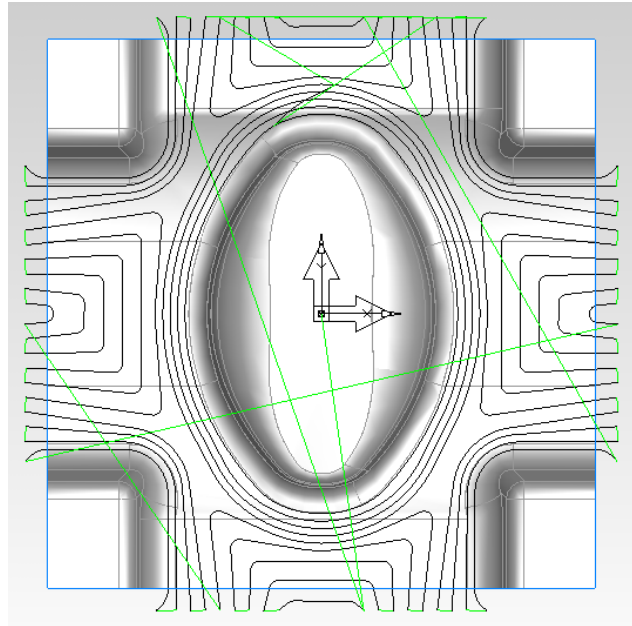
- Select a **top view**.




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.

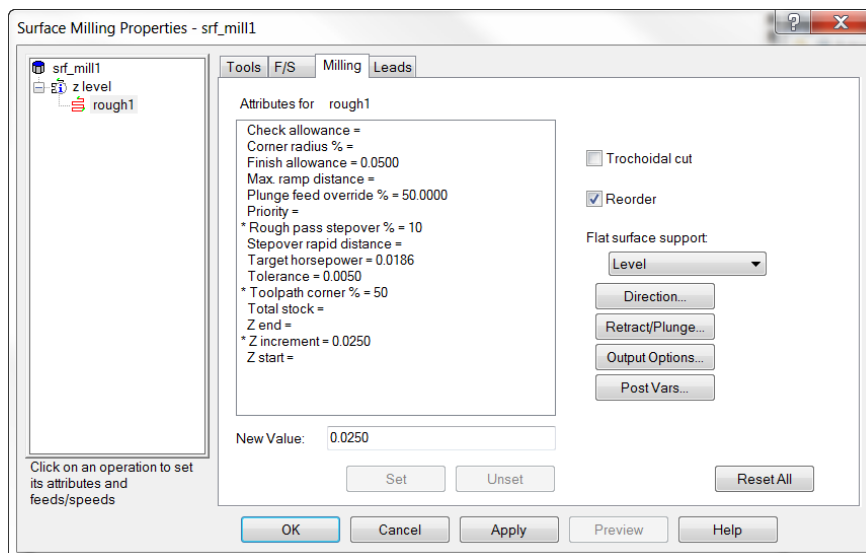
- **Eject** the simulation 
- Select the **Milling** tab and change the **Toolpath corner %** to **50**.
- Click **Apply**, then **Preview** and then **Play selected** 

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



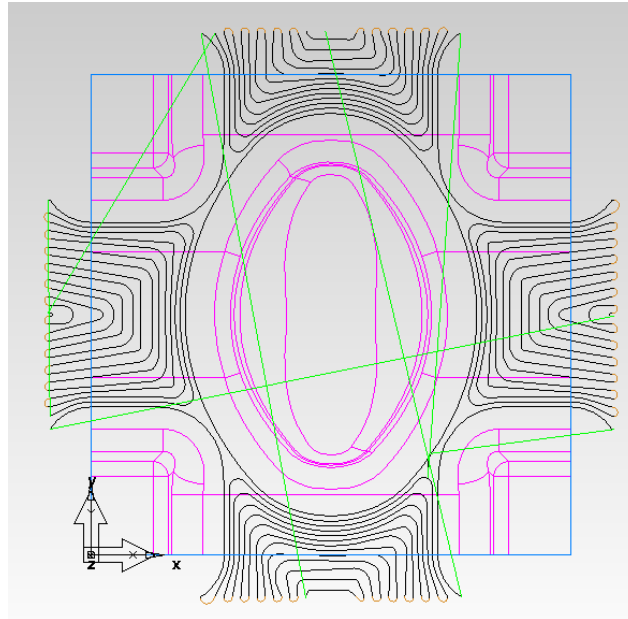
Note how the toolpath now has radii where it changes direction also the way that the paths are smoothed as they get further from the profile. This path has some cuts which engage a large part of the cutter diameter however as we are going to use a small depth of cut this is not important. Although the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

- **Eject** the simulation 
- Double click on the feature to open its properties.
- Select the **Leads** tab and change the **Stepover Type** to **Arc**.
- Select the **Milling** Tab.
- Set the **Z increment** to **0.025"** and **Rough pass stepover** to **10%**.



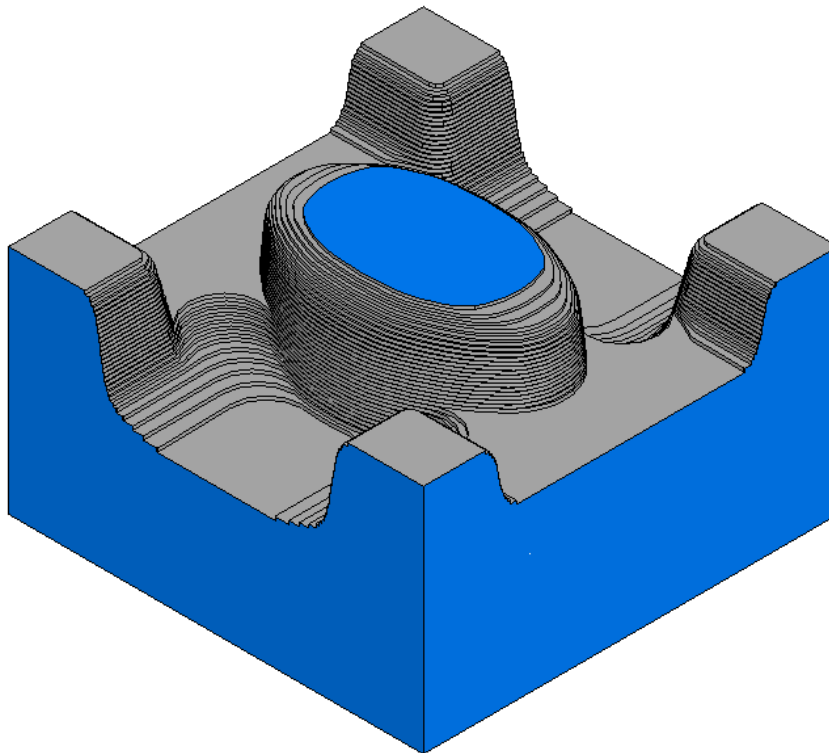
- Run a **Centerline** simulation.

- In the operations list click on the down arrow next to the roughing operation and select the roughing level at **Z -0.4053 or thereabouts**.




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

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



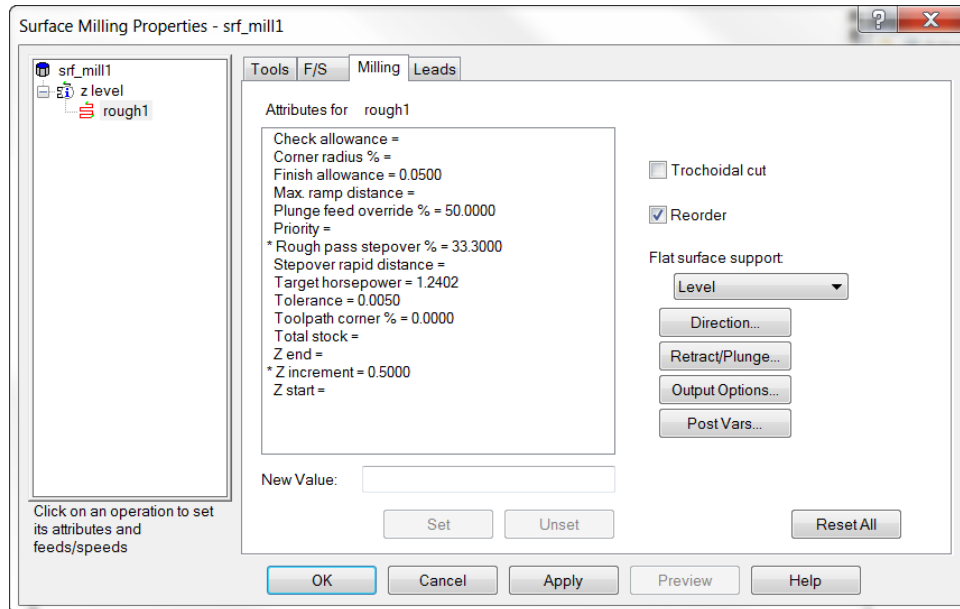
The material is roughed out in a series of shallow steps with a small stepover. The smooth shape of the toolpath and the small amount of material being removed per pass means that the cutter can run much faster giving an overall high rate of metal removal.

- **Eject** the simulation 

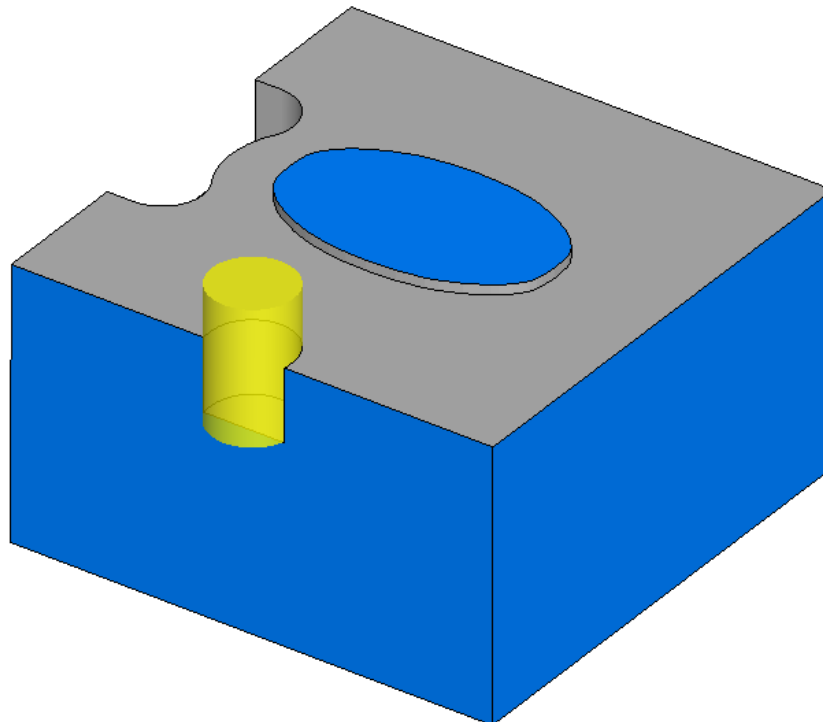
## High Efficiency Roughing

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.

- Double click on the feature to open its properties.
- Select the **milling tab** and fill in **exactly** as below.



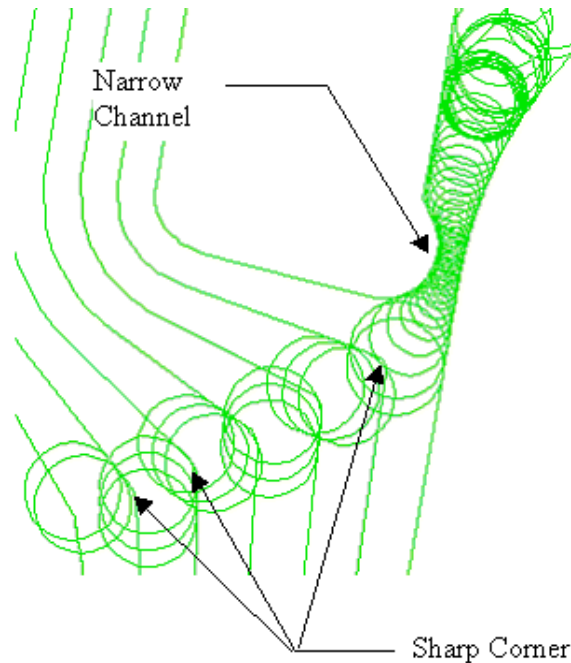
- **Single step a 3d simulation.**



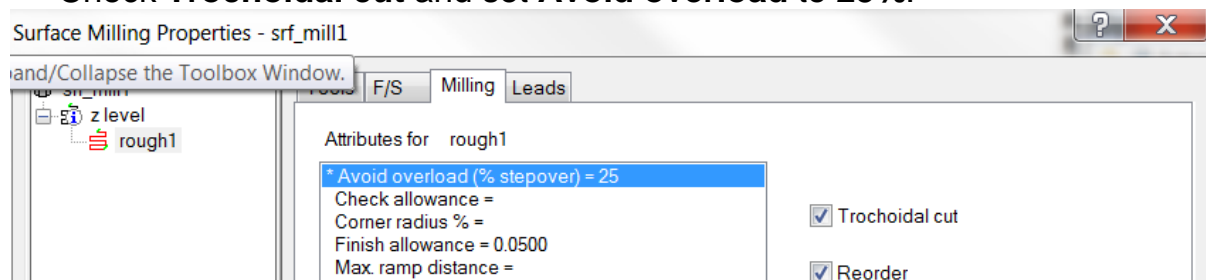
The initial moves into the material are taking a very heavy full width cut. This overload situation may cause damage to the tool and would require a reduced feedrate. We will now use Trochoidal roughing to reduce the tool loading.

## 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.



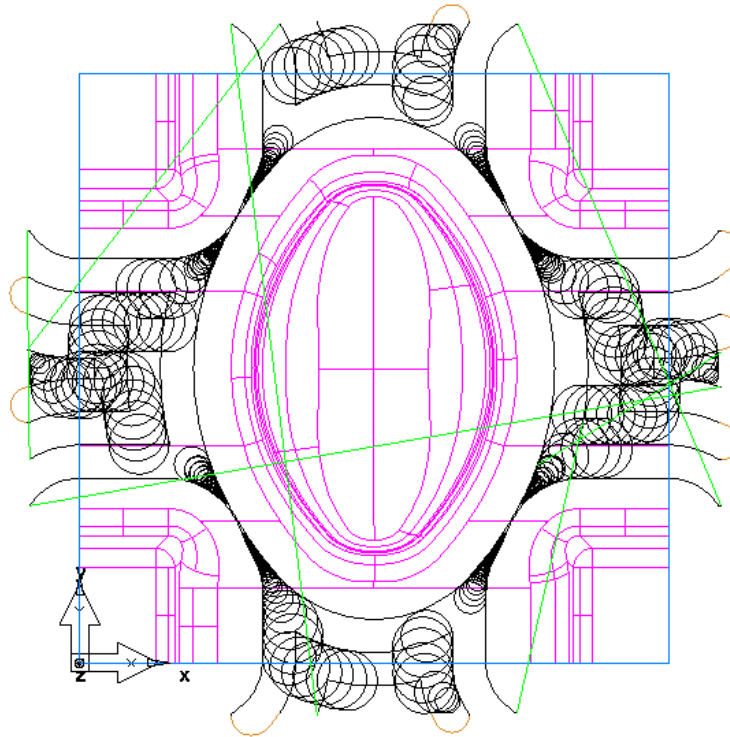
- **Eject the 3d simulation.**
- Edit the feature and select the **Milling** tab.
- Check **Trochoidal cut** and set **Avoid overload** to **25%**.



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

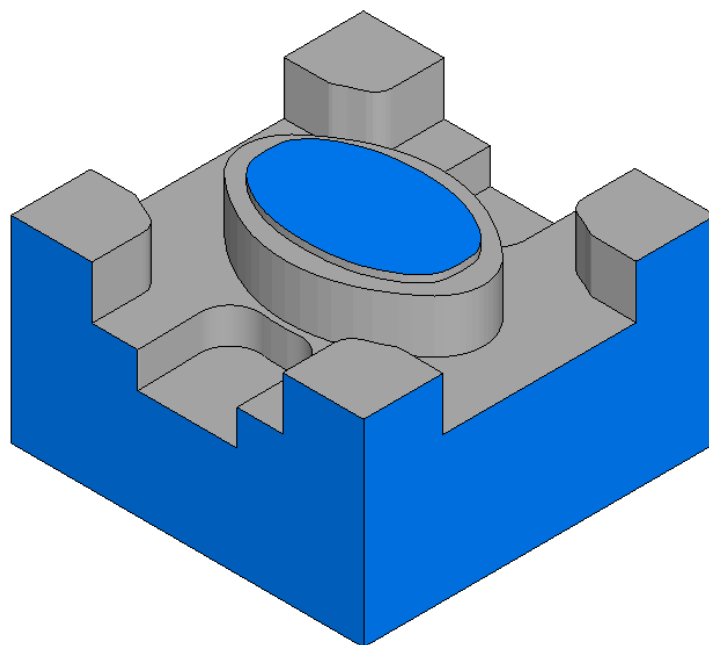
- Click **Apply** and then **OK**.
- Run a **Centerline simulation**.

- In the operations list click on the down arrow next to the roughing operation and select the roughing level at **Z -0.5037**.
- Take a Top view.



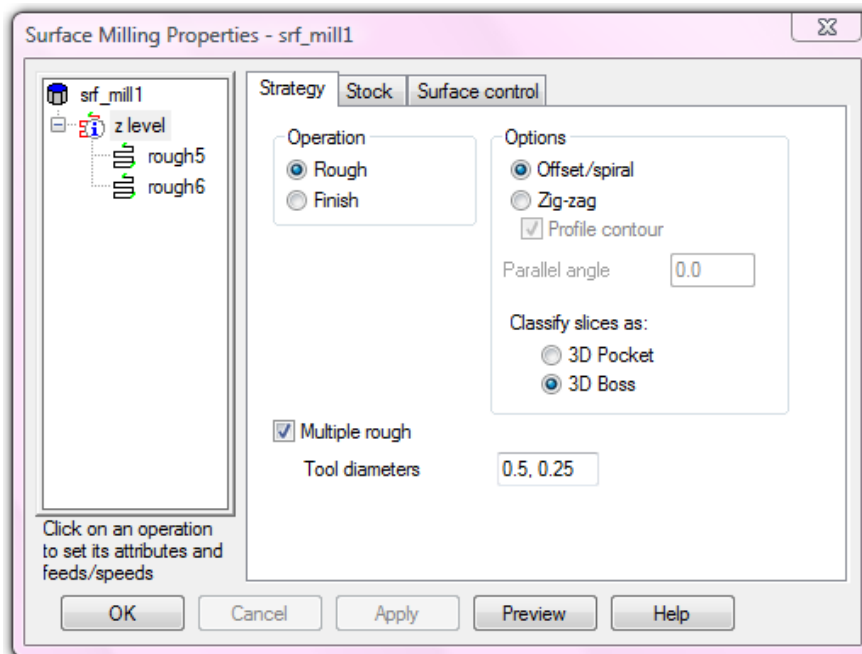
It can be seen that the toolpath has changed so that the tool overload situation is avoided by changing to Trochoidal moves. The tool loading is significantly reduced but the overall toolpath length is increased, this will mean that the roughing process will take longer to complete. This can be fixed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

- Run a **3D simulation**.

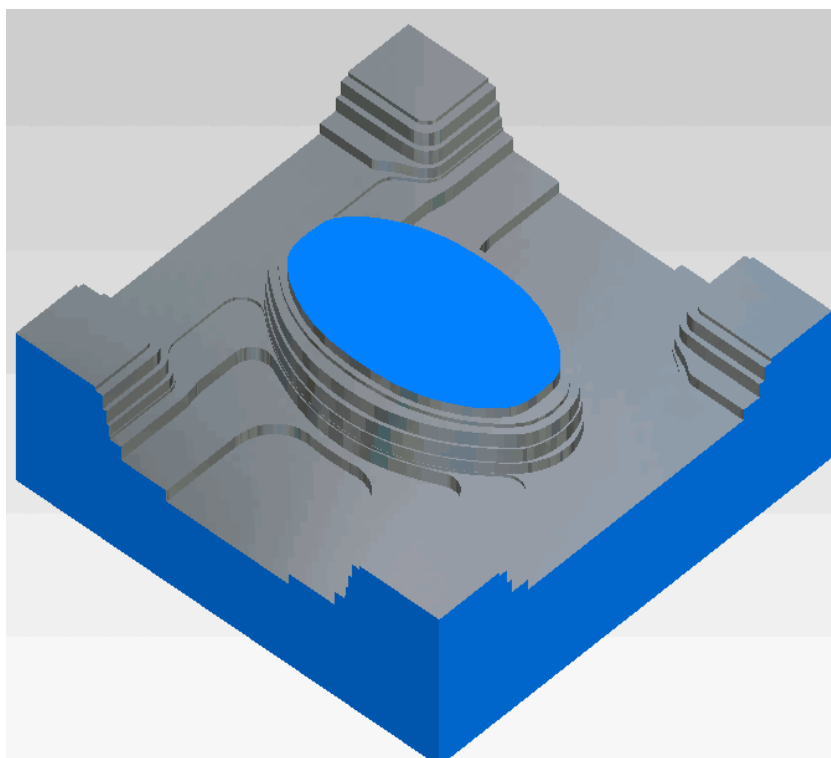


The bulk of the material is removed quickly but the remaining stock has large steps. Due to the large tool diameter a lot of material is left on the inside corners. A second roughing pass will be added to reduce the steps and remove more material from the corners.

- Double click on the feature to open its properties.
- Select **Z level** and fill in the form as below.



- Click **Apply**.
- Select the **second** roughing operation and set its **Z increment** to **0.05"**.
- 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.

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

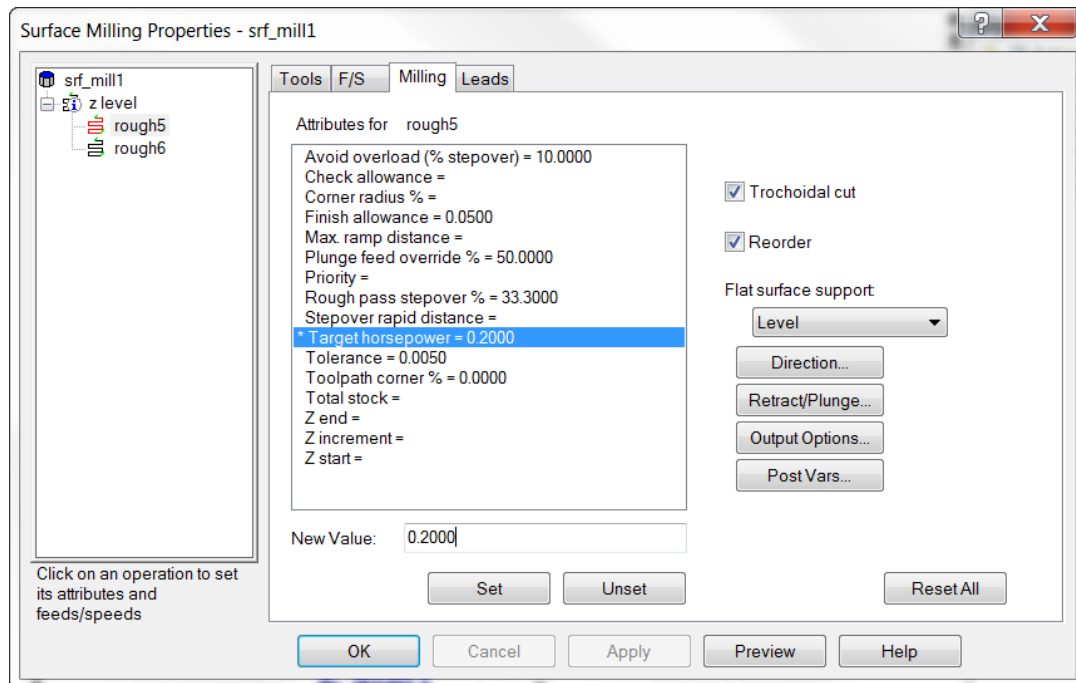
```
NC Code
%
O0001(FAST_ROUGH FIN)
( SETUP1 - 01/07/2009 - 11:32:15 )
( FEATURECAM - HAAS VF )
( MACHINE TIME = )

N35 (ROUGH5 Z LEVEL SRF_MILL1 )
N40 G0 G17 G20 G40
N45 T1 M6 (ENDMILL0500:REG 0.5 DIA. )
N50 G54 G90 X3.2177 Y0.0995 S4965 M3
N55 G43 H1 Z1.0 M8
N60 Z0.1
N65 G1 Z-0.0608 F24.8
N70 X3.2052 F49.6
N75 X3.183 Y0.0592
N80 X3.141 Y-0.0117
N85 X3.0947 Y-0.0798
N90 X3.0436 Y-0.1502
N95 X3.0038 Y-0.2023
N100 X2.9615 Y-0.2524
N105 X2.7334 Y-0.265
```

Just two machining feedrates are being used within this, program, the plunging feedrate and the cutting feedrate.

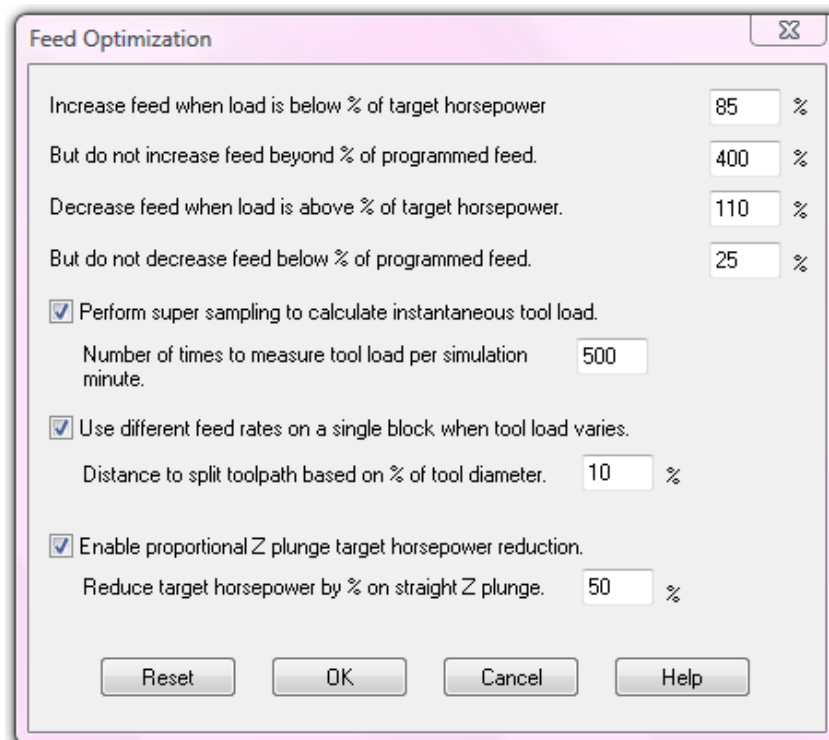
We will now instruct FeatureCAM to calculate the feedrate such that it maintains an optimum loading on the tool, where the cut is lighter the tool will move faster, where the cut is heavy it will slow down.

- Edit the feature and select the **Milling** tab for the first roughing operation.
- Set the **Target horsepower** to **0.2 HP**.
- 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.

- From the **Manufacturing** menu select **Feed optimization**.
- Fill in the form as shown below:



The first item is telling FeatureCAM that if the load is less than 85% 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.

- 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.

```

NC Code
%
O0001(FAST_ROUGH FIN)
( SETUP1 - 01/07/2009 - 11:34:26 )
( FEATURECAM - HAAS VF )
( MACHINE TIME = )

N35 ( ROUGH5 Z LEVEL SRF_MILL1 )
N40 G0 G17 G20 G40
N45 T1 M6 ( ENDMILL0500:REG 0.5 DIA. )
N50 G54 G90 X3.2177 Y0.0995 S4965 M3
N55 G43 H1 Z1.0 M8
N60 Z0.1
N65 G1 Z-0.0608 F99.3
N70 X3.2052 F198.6
N75 X3.183 Y0.0592 F184.7
N80 X3.162 Y0.0237 F157.4
N85 X3.141 Y-0.0117 F198.6
N90 X3.0947 Y-0.0798
N95 X3.0436 Y-0.1502
N100 X3.0038 Y-0.2023
N105 X2.9615 Y-0.2524
N110 X2.7334 Y-0.265
N115 Y-0.2525
N120 X2.7409 Y-0.2448
N125 X2.7482 Y-0.2371 F96.3
N130 X2.7817 Y-0.2014 F66.5
N135 X2.8152 Y-0.1656 F63.1
N140 X2.8638 Y-0.1105 F62.6
N145 X2.9089 Y-0.0524 F61.6
N150 X2.9344 Y-0.0172 F71.5
N155 X2.9599 Y0.018 F85.9
N160 X3.0004 Y0.0776 F105.8
N165 X3.0372 Y0.1396 F133.1
N170 X3.0631 Y0.1865 F171.8
N175 X3.1148 Y0.2803 F198.6
N180 X3.1454 Y0.3398

```

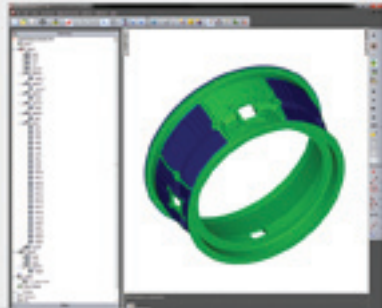
It can be seen that the feedrate is now varying. As the Trochoidal toolpath is reducing the loading the feedrates are being increased. This will result in the toolpath running in a shorter time on the machine.

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.



# FeatureCAM 2012

FeatureCAM 2012



new features

Delcam TV



[www.delcam.tv](http://www.delcam.tv)

FeatureCAM Learning Zone



[www.delcam.tv/lz](http://www.delcam.tv/lz)

FeatureCAM Website



[www.featurecam.com](http://www.featurecam.com)



THE QUEEN'S AWARDS  
FOR ENTERPRISE:  
INNOVATION  
2010



THE QUEEN'S AWARDS  
FOR ENTERPRISE:  
INTERNATIONAL TRADE  
2011

## Powering your productivity

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

To contact your local reseller, visit [www.delcam.com/resellers](http://www.delcam.com/resellers)