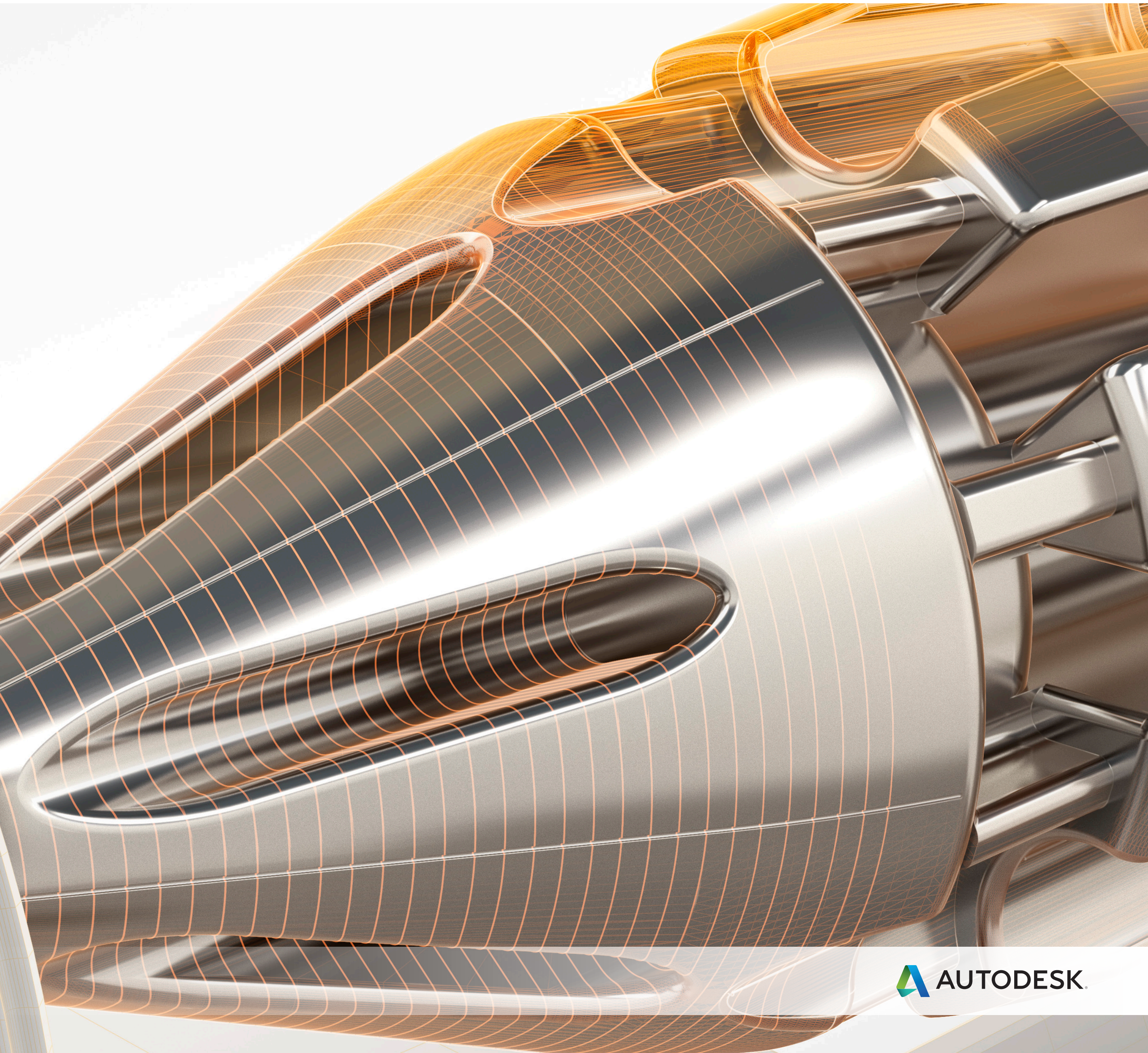
AUTODESK® FEATURECAM®

Automated to make parts faster

Training Course



AUTODESK.

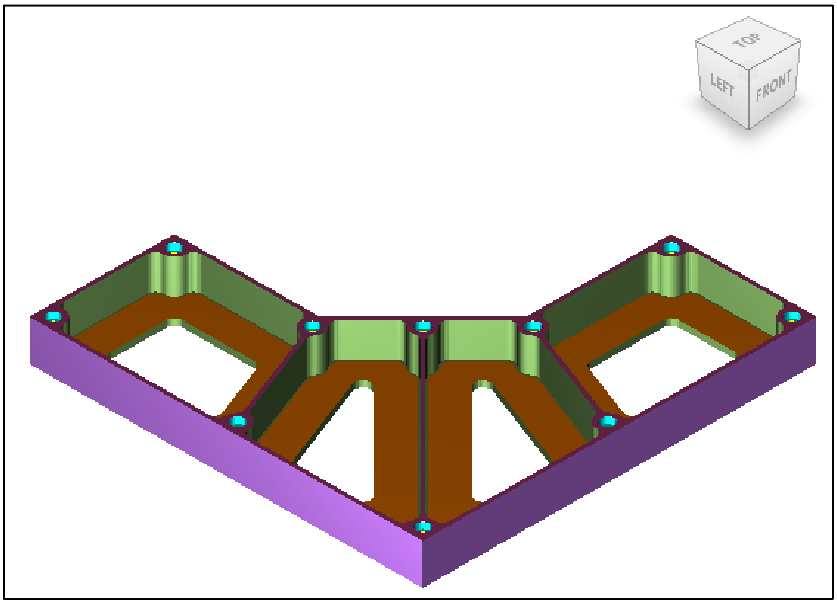
Contents

AUTODESK® FEATURECAM®

FeatureCAM 2.5D Machining

Introduction

- This module will familiarize you with **Autodesk FeatureCAM's** history, describes what **FeatureCAM** is and show how it is advantageous over other **CAM** systems.
- Increased shop productivity is the primary goal of all **FeatureCAM** products. Traditional **CAM** systems are **Operations Based** and require you to program every operation, one at a time, to create your part. All the manufacturing details are left to the machinist. That means that you must specify the details for every **Spot Drill**, **Drill**, **Ream** and **Roughing** pass of your part.
- With **FeatureCAM** you generate your part using **Features** such as a **Tapped Hole, Boss,** or a **Turned Groove** and the operations are automatically created. **FeatureCAM** also manages the details of the manufacturing process such as Tool selection, Speed and Feed Rates, and Toolpaths. To modify any element of the part program, just change a few settings on a feature and a whole new set of operations are generated to reflect your changes.
- **Ease of use** is another guiding principal of **FeatureCAM** products. Our system was originally developed as a graphical Windows-based program. **FeatureCAM** programs are full of graphical feedback, **Step-by-Step Wizards** and tutorial-style animations that run directly in the program dialog boxes. Ease of use has always been a **FeatureCAM** advantage and we continue to study ways to make our system even easier to learn and use.



*The image above shows a part that can be quickly machined in **FeatureCAM***

CAM Software Strategies

There are three main different types of **CAM** software strategies for creating toolpaths in today's modern **CAM** software. These strategies are **Operation based**, **Processed based** and **Feature Based**. Some of these strategies use **Knowledge Based Machining (KBM).** **KBM** is when machining intelligence is built right into the **CAM** software. **KBM** is used at various levels and degrees in different **CAM** systems.

Operations-based CAM systems require the user to go through multiple steps to machine each part. The user must select which type of operation to use, select the machining boundary and the type of toolpath, then manually select tools, feeds and speeds and multiple machining options such as step over distance and the incremental step in Z. These multiple steps must be repeated one at a time for every operation on a part, making it a time-consuming process that is prone to error. These systems offer little or no **KBM**.

Process-based CAM systems attempt to reduce these steps by grouping operations in standard processes. **KBM** is used as the program remembers the used processes and repeats them for future operations. However, this method only works when a shop uses the same processes over and over for the same types of parts.

When a process is applied to different types of parts, or parts made of different materials, the **CNC programmer** must still verify that the correct operations were used and that the correct tools, step over and speeds and feeds were selected because a static machining process does not automatically adapt to every part.

Below is a chart showing a comparison of the three different CAM strategies:

Feature Based CAM ✓	**Operations-Based CAM**	**Process-Based CAM**
1. Draw or Import Your Part **2.** Identify Features **3.** Click Simulation **You Are Done!** **FeatureCAM Automatically:** • Determines Rough and Finish Operations • Selects ALL Tools Sizes • Calculates Feeds & Speeds • Determines Stopover & Z Increments • Generates Toolpath • Creates NC Code	**1.** Import or draw part. **2.** Select a rough operation. **3.** Select rough boundary. **4.** Select type of toolpath. **5.** Select roughing tool. **6.** Select feeds and speeds **7.** Select step over and Z steps. **8.** Select a finish operation. **9.** Select finish boundary **10.**Select type of toolpath **11.** Select a finishing tool. **12.** Select step over and Z steps. **13.** Select feeds and speeds. **14.** Click simulation. **15.** Create NC code	**1.** Import or draw part. **2.** Identify features. **3.** Select machining processes. **4.** Click simulation. **5.** Create NC code. **???Are You Done? No.** You must analyse and verify: Does the process have the correct operations? Does the process use the correct size tools? Are the correct feeds and speeds used? Does the process have the correct step over and Z increments?

Feature-Based Cam Systems

Feature-based CAM systems use a set of interrelated machinable features to describe a complete part. The beauty of features is that they not only describe the shape, but these also are made up of one or more associative operations that describe the preferred method for cutting that shape at the **NC machine.**

A predefined set of machining rules and user preferences are applied to a part. **KBM** in these systems streamlines the manufacturing process by building the highest level of machining intelligence right into the **CAM system**. 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 Centre drill and preferred machining strategies for roughing and finishing. The **CAM system** evaluates the part geometry and part material, selects the most appropriate tools and operations, recommends machining strategies, calculates feeds and speeds, then automatically generates the **NC code**.

Feature-Based Machining has many advantages such as:

- Dramatically shorter part programming times.
- Part revisions do not require reprogramming.
- NC code is consistent and predictable regardless of the CNC programmer.
- Tools are optimized and tool changes reduced.
- Toolpath is automatically optimized for faster machining.

FeatureCAM is a Feature Based programming system with advantages listed above. Additionally **FeatureCAM** is very easy to use. A programmer new to **FeatureCAM** or new to programming can quickly learn the product and create G-code with the appropriate tools, feeds and speeds.

Below is a list of key points that make **FeatureCAM** very easy to use.

- **User Friendly Interface *(Steps & Wizards).***
- **Automatic Tool Selection *(2D & 3D).***
- **Automatic Feature RECOGNITION *(AFR).***
- **Interactive Feature RECOGNITION *(IFR).***
- **Standard Databases *(Tools, Materials, Feeds & Speeds).***
- **Customizable *Post Processors.***
- **Outstanding Technical Support**.

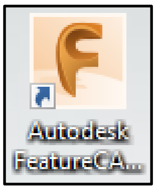
Built in intelligence

A major advantage of **FeatureCAM** is the users' ability to define their own set of machining preferences. **FeatureCAM** comes with predefined machining rules programmed into the CAM system which are based on a standard set of material properties and standard tool cribs. You can then customize these settings for specific materials that you use and the tooling that you actually have available in your shop. Once you have these defined, you simply select a material for your work piece and the tool crib you want to work with and **FeatureCAM** will then make selections of tooling for a feature automatically. If you change the tool crib or the material then **FeatureCAM** automatically updates all of the toolpaths in the part. This means that a part may be programmed for one machine with the tools available there and then shifted to another machine simply by changing the tool crib. Similarly if you wish to cut a test part in Aluminium and then cut the final part in Stainless Steel you program the part with Aluminium set as the material. Once you have cut the test piece you change the material to be Stainless Steel and **FeatureCAM** will update the toolpaths by changing feeds, speeds, depth of cut, step over etc. without any further intervention from the operator.

By providing a set of standard machining rules already in place, it's easier for you to customize the software to reflect the machining preferences of your Machine shop.

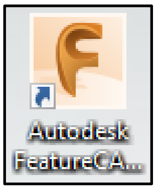
Setting up your own machining preferences and rules in **FeatureCAM** not only reduces the time it takes to generate programs, but it also streamlines production because the approach to machining every part is predictable and consistent. Machine operators can rely on the fact that each job will be machined in a predictable manner regardless of who did the programming.

Starting FeatureCAM

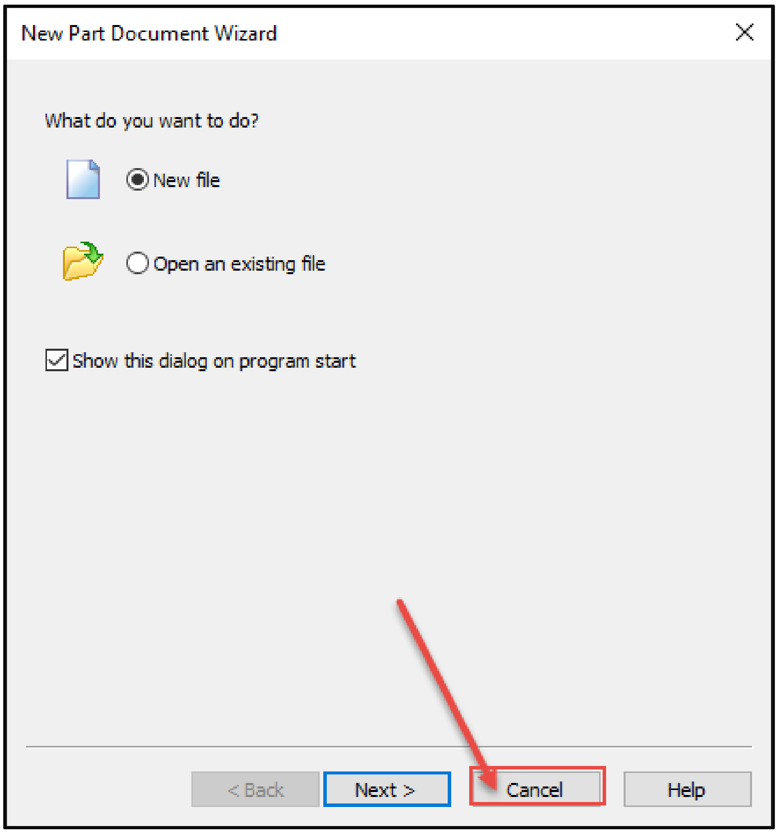
- Select the FeatureCAM icon 

from your desktop and select **New File** and **Milling Setup** and **Millimetre**. Select **Finish** twice and then **Ok** to close the Stock menus. The stock menu will be covered later on in the training notes.

Evaluation Options

1. Click on the shortcut 

that is on the Desktop. This will start FeatureCAM.
2. When it has finished **cancel** the new **Part document wizard** that is on the screen.



3 Go up to the top left of the screen and go to the **File** menu**.** Then Select **Evaluation Options.**

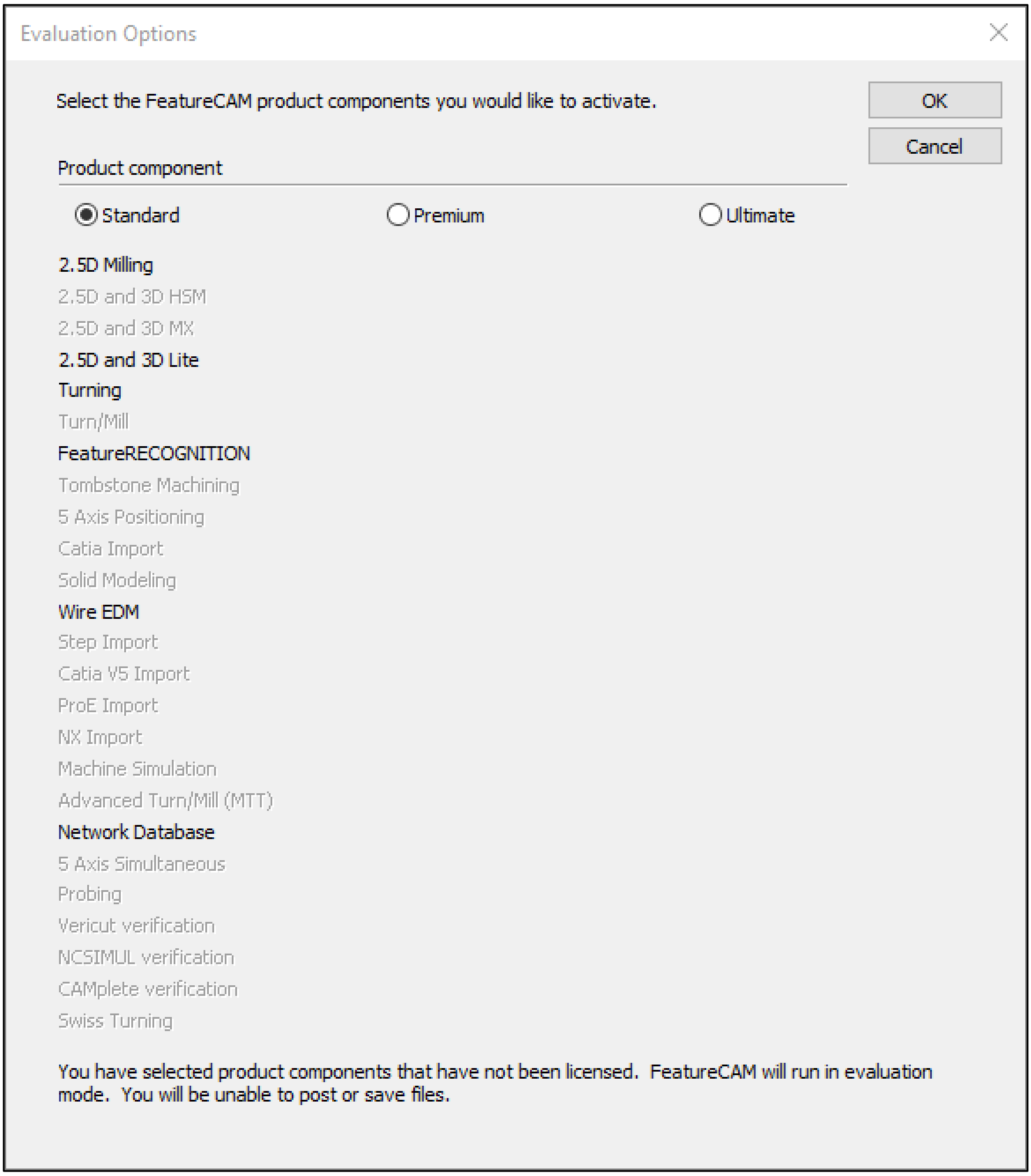
AUTODESK® **FEATURECAM®**

Feature Comparison

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

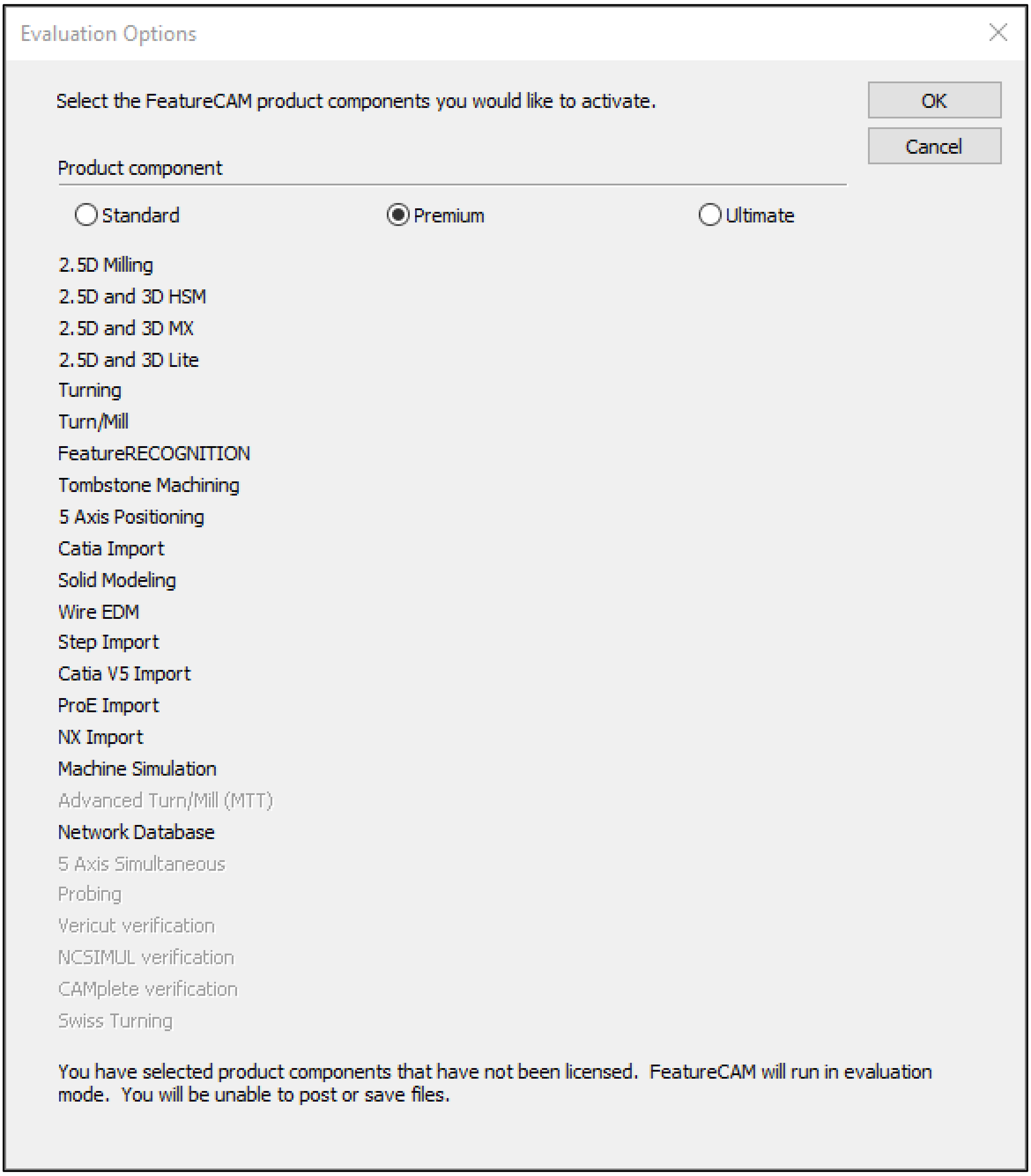
***FeatureCAM** has now been split into **three levels**. **Standard, Premium** and **Ultimate**. The next few images show the available options for all modules.*

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



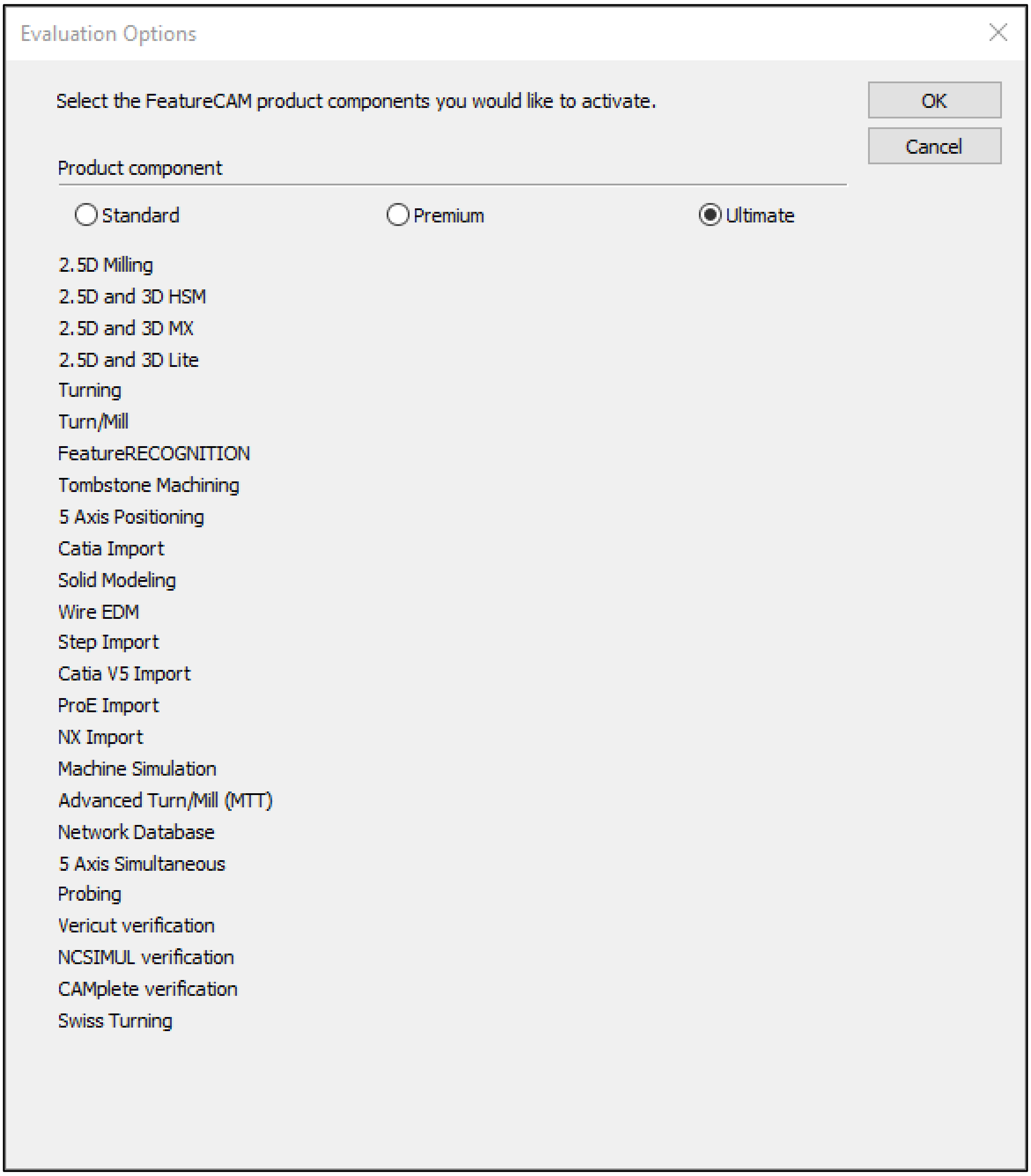
It is here that you will see the activated products that you have purchased.

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



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

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



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

Desktop Subscription

This is what you get when you subscribe:-

- Individuals, teams and enterprises have more choices than ever before Regardless of the size of your business - small, mid-size or enterprise.
- Autodesk offers the flexibility to choose the subscription options that best fit your business needs.
- Try out new tools without making a big up-front investment.
- Better manage changing software and budget needs.

Access options

When subscribing, select the option for access and use of your software:

- **Single-user access** - best for when software is needed for one person
- **Multi-user access** - ideal for situations in which you need more than one person to share software licences

Flexible term lengths

With monthly, quarterly, annual and multi-year options, you can get exactly the software you need for as long as you need it.

Access to the latest software

When you subscribe, you have instant access to the latest product releases and enhancements.

- Stay current on the latest software rather than waiting for major product releases
- Update software straightaway or choose a time that is right for your entire team

More about updates and releases

- Software when you need it, where you need it

Use your software in more places with access to both current and preceding versions.

- Access preceding versions :-

  You will be able to download and use preceding releases of most Autodesk software.
  More about preceding version rights :-
- Use your software at home - Access both current and preceding versions of your software on your home computer.
  More about home use rights
- Use your software when travelling :-

- Use your software and services globally when travelling outside of your home country. More about global travel rights

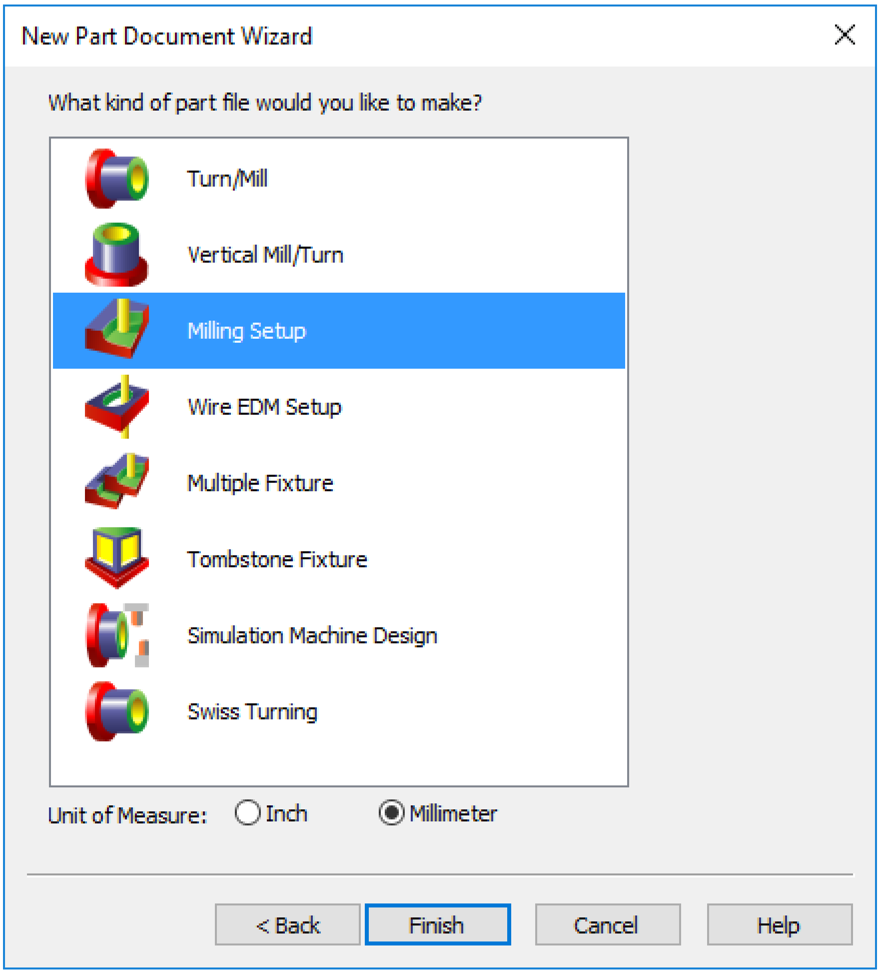
Autodesk support

Let us help you to resolve issues quickly. You will have access to:

- Senior Autodesk support specialists.
- Moderated community support forums.
- E-learning opportunities and exclusive training web casts New Part Document.

Starting a new FeatureCAM part document.

- Select **File**>**New** from the menu or select **Ctrl + N** keys.

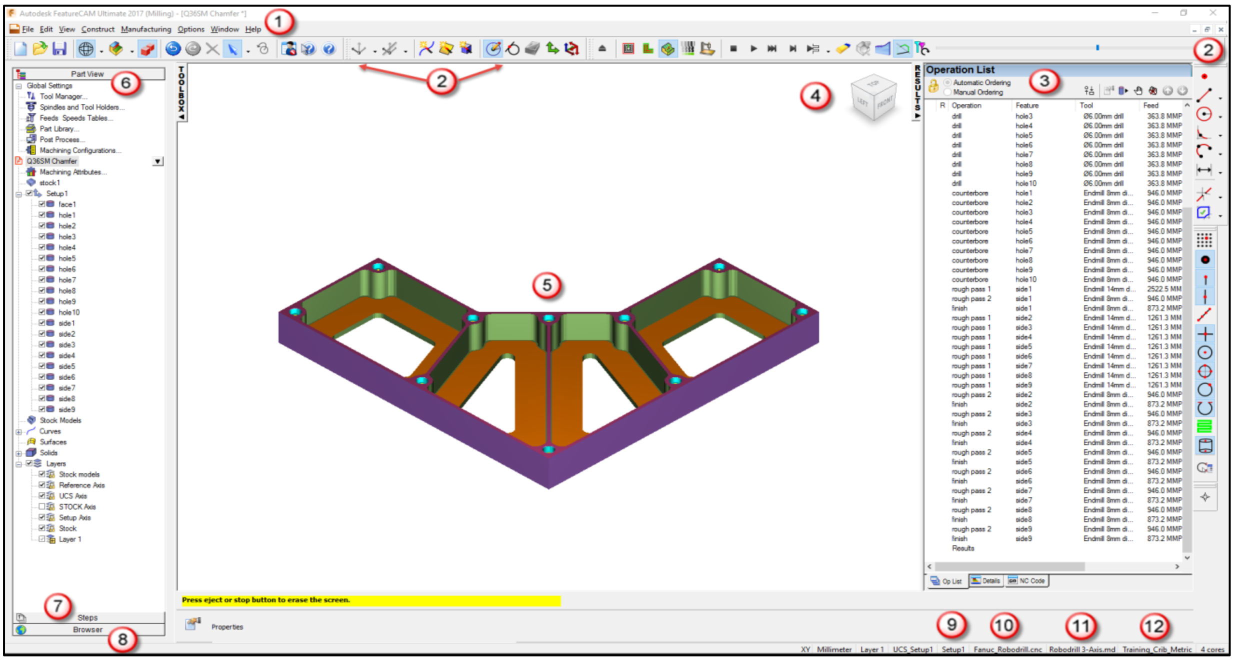


- Select your **Units of Measure** either **Inch** or **Millimeter**. Select the correct Module of your choice.
- The Modules you have purchased will be visible.

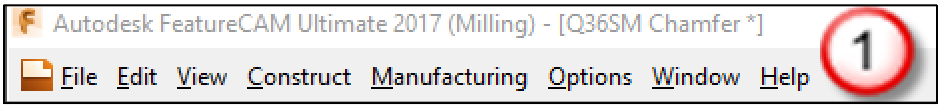
User Interface

- **FeatureCAM** is a windows-based product and has been a windows based product since its inception unlike other **CAM** products. Many of the functions and keystroke shortcuts that you are familiar with in windows are available inside **FeatureCAM**. Examples are **Ctrl+C** to copy, **Ctrl+V** to paste, **Ctrl+Z** Undo last operation. etc.

Screen Layout

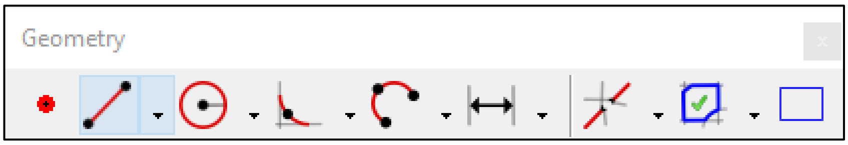


1 **Menu Bar** - Located at the top. Like many other Windows based software, packages **FeatureCAM** includes a series of drop down menus. These menus are unique to **FeatureCAM** and provide one or more functions used to facilitate the constructing, manufacturing, viewing and editing of your part program.

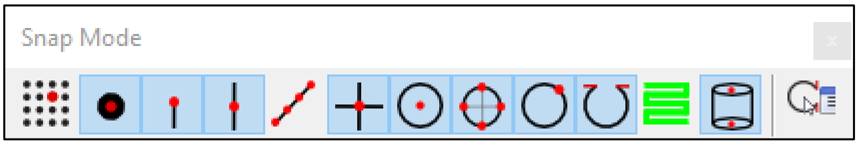


2 **Tool Bars -** usually located at the top but, may be docked in other locations. Can consist of one or more buttons and used to create objects in the Graphics Window such as the Geometry constructors for drawing the shape of your part.

2 **Geometry** - Use geometry constructors to create 2D geometry. Snapping is a CAD concept that helps you position lines, points or shapes as you construct geometry for the part.



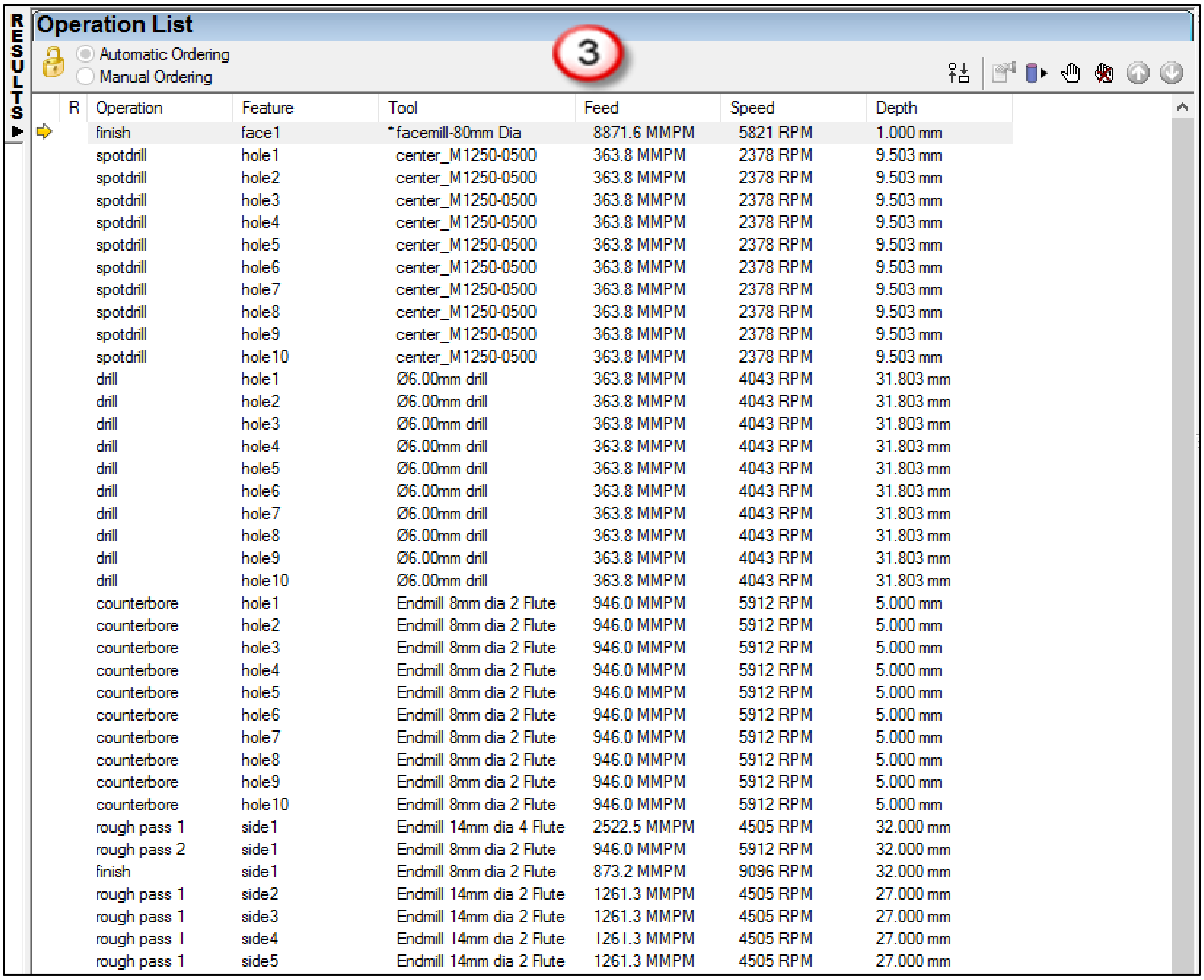
2 **Snap Icons**:



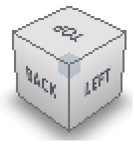
Entering points by coordinates is exact but not always convenient or feasible. Picking points with the mouse is not always precise enough.

The bridge between the methods of point selection is snaps. From left to right, the buttons set the snap to:

3 **Results** - Located on the right side. It contains the **Operation list**. Whereas the **Part View** contains all of the features, the **Operation List** contains all of the machining history. It also provides the details of each operation, including a tool list. The **NC code** is also here. All of the information residing in this area may be printed.



R	Operation	Feature	Tool	Feed	Speed	Depth
	finish	face1	facemill-80mm Dia	8871.6 MMPM	5821 RPM	1.000 mm
	spotdrill	hole1	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole2	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole3	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole4	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole5	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole6	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole7	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole8	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole9	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	spotdrill	hole10	center_M1250-0500	363.8 MMPM	2378 RPM	9.503 mm
	drill	hole1	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole2	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole3	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole4	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole5	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole6	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole7	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole8	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole9	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	drill	hole10	Ø6.00mm drill	363.8 MMPM	4043 RPM	31.803 mm
	counterbore	hole1	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole2	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole3	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole4	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole5	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole6	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole7	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole8	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole9	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	hole10	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	5.000 mm
	rough pass 1	side1	Endmill 14mm dia 4 Flute	2522.5 MMPM	4505 RPM	32.000 mm
	rough pass 2	side1	Endmill 8mm dia 2 Flute	946.0 MMPM	5912 RPM	32.000 mm
	finish	side1	Endmill 8mm dia 2 Flute	873.2 MMPM	9096 RPM	32.000 mm
	rough pass 1	side2	Endmill 14mm dia 2 Flute	1261.3 MMPM	4505 RPM	27.000 mm
	rough pass 1	side3	Endmill 14mm dia 2 Flute	1261.3 MMPM	4505 RPM	27.000 mm
	rough pass 1	side4	Endmill 14mm dia 2 Flute	1261.3 MMPM	4505 RPM	27.000 mm
	rough pass 1	side5	Endmill 14mm dia 2 Flute	1261.3 MMPM	4505 RPM	27.000 mm

4 ViewCube 

The ViewCube is a new feature in FeatureCAM, which enables you to change and identify the viewpoint of the Graphics window. By clicking the ViewCube's corners, faces, edges, and icons, you can use it to directly manipulate the view in the Graphics window. In addition, when you re-orient the view using a cube option, toolbar button, or keyboard shortcut, the ViewCube automatically reflects the new viewpoint.

The ViewCube is displayed in the upper-right corner of the Graphics window. Click and drag the cube to re-orient the view in any direction. Alternatively, click:

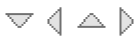
- A face to show it as an orthogonal view.
- An edge to show the adjacent faces.
- A corner to show the three adjacent faces.

In addition, when you move the cursor near the cube, the Home icon is displayed. Click the icon to show the Home view.

Manipulating orthogonal views

When you select a single-face view and move the cursor near the cube, control icons are displayed:

Click:

 To show the view of an adjacent face.

 To rotate the view clockwise through 90 degrees.

 To rotate the view counter-clockwise through 90 degrees.

Configuring the ViewCube

To control the behaviour and appearance of the ViewCube, right-click the cube and choose a menu option. Select:

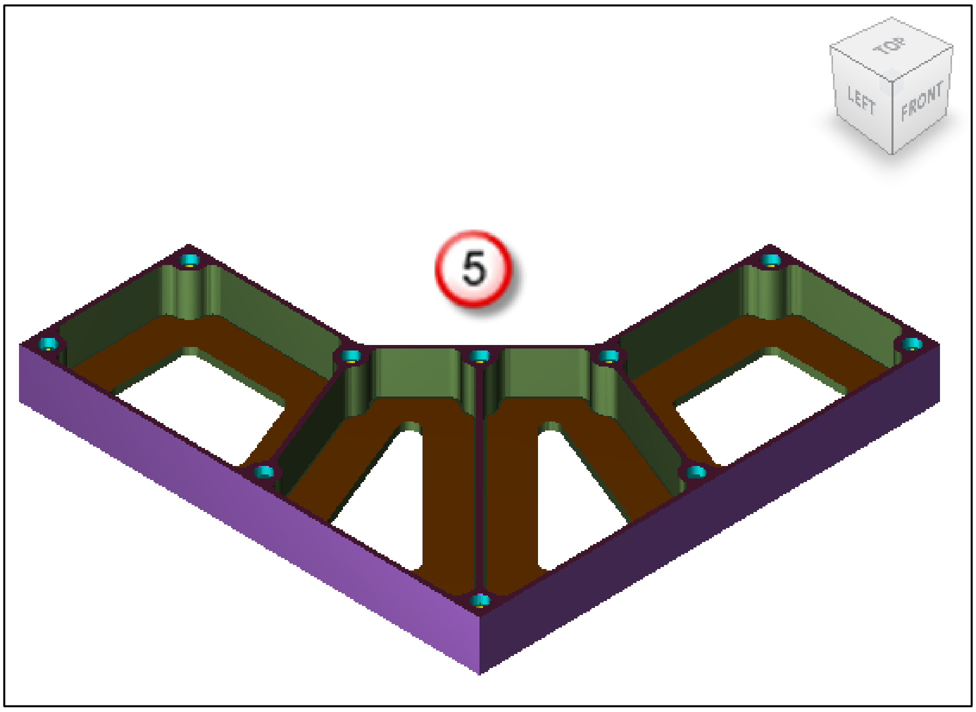
- **Set current view as home and keep scale** to save the current orientation and magnification of the model as the Home view.
- **Set current view as home and scale to fit** to save the current orientation of the model as the Home view, and scale it to fit the Graphics window.
- **Options** to display and change the settings of the ViewCube. The **ViewCube** tab of the **Viewing Options** dialog is displayed.

Setting ViewCube options

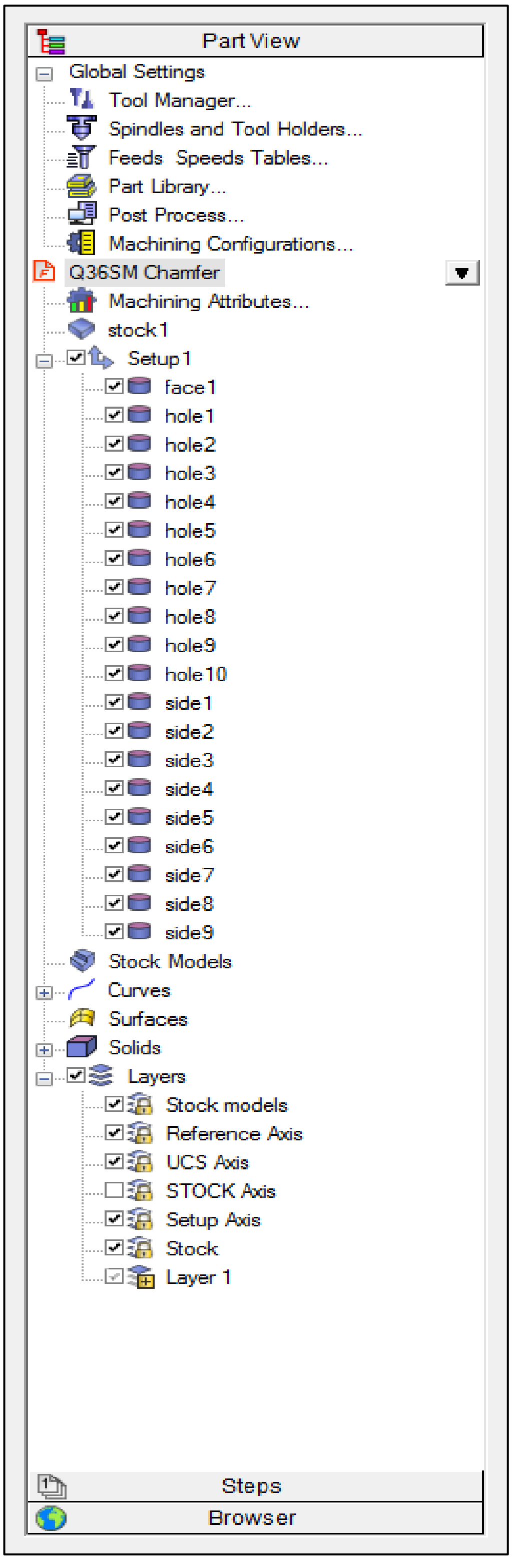
To modify the behaviour and appearance of the ViewCube, set these options on the **ViewCube** tab of the **Viewing Options** dialog:

- **Show the ViewCube** — Select this check box to show the ViewCube in the Graphics window. Deselect the check box to hide it.
- **ViewCube size** — Select an option from the list to choose the size of the ViewCube in the Graphics window. Alternatively, select **Automatic** to resize the cube when you resize the Graphics window.
- **Keep model upright** — Select this check box to prevent the view from being inverted. If you click an edge, corner, or face that would leave the view upside down, the Graphics window switches to the selected view and then rotates to leave the view upright.

5 Graphics Window - Located in the Centre. This is where most of the activity is taking place, creating Stock, Importing Drawings, Creating Solid and Surface models, constructing Geometry, Chaining with Curves, creating Features and Simulating the tool path in 3D or using the Centreline Simulation.



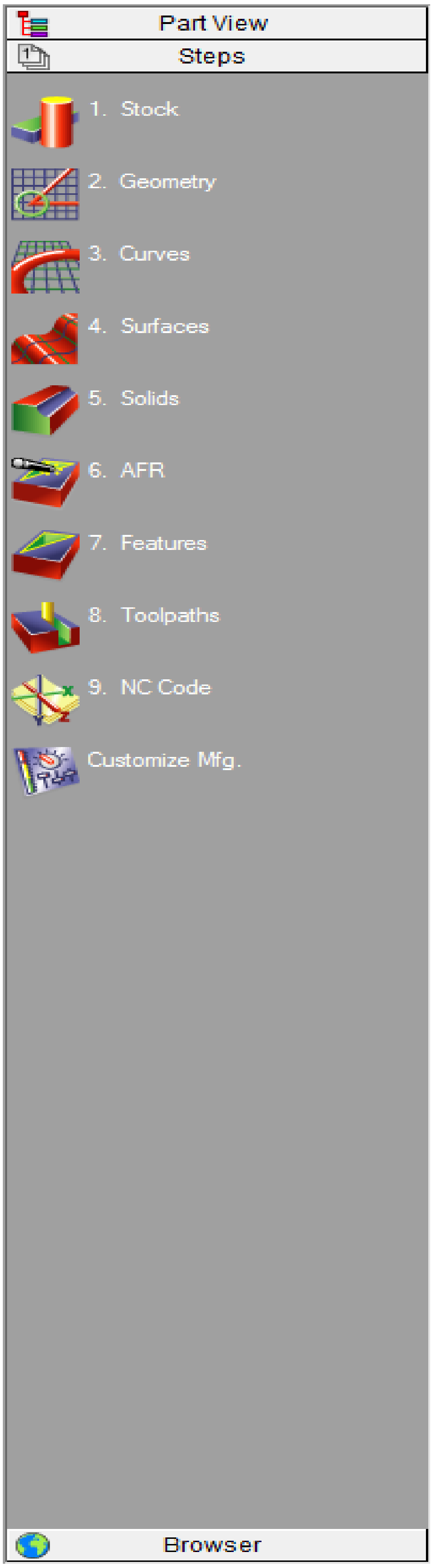
⑥ Part View - Located behind the **Steps** menu. This area shows the **Stock**, **Setup**, **Features**, **Curves**, **Surfaces and Solids when available.** This provides access to their respective Properties.



- In the Part View Tool Box this contains the names of the objects that are created on the screen. Objects that can be contained in this area are Stock, Setups, Curves, Surfaces, and Solids, which can include Stl triangulated models.

7 **Steps -** Located on the left side. A systematic sequence to guide, especially the beginner, as to the order of which needs to be completed first, second, etc.

Steps Menu



*The **Steps** toolbar is an outline for the flow of the program. The Steps are numbered sequentially in the order they should be executed and can be a guide for creating your part.*

1 **Stock Wizard** steps you through entering the shape and dimensions of the stock, the stock material, part program zero and the coordinate system for modeling.

2 **Geometry.** This gives you the ability to create part shapes using a series of Geometry commands for example Points, Lines, Arcs, Clip, Fillet etc. These are used to define the overall shape of parts. Many different geometry tools are available. Geometry can also be imported from other CAD systems.

3 **Curves.** Shapes that involve more than a single line or arc are described as curves.

4 **Surfaces.** This opens the **Surface Wizard.**

5 **Solids.** This opens the **Solid Wizard.**

6 **AFR. (Automatic Feature Recognition**) This creates 2.5D geometry from Solid Models ready for machining. **IFR** is also available.

7 **Features.** This relates to common shop terms like **Hole**, **Pocket**, **Slot** or **Thread Milling** etc. They are created from curves or dimensions. These objects are created in 2.5D and are used to generate toolpaths.

8 **Toolpaths.** Toolpaths are generated from collections of features. You can simulate them in **FeatureCAM** using toolpath Centerlines, 2D shaded or 3D solid shaded simulations. Plus full **Machine Tool Simulation**.

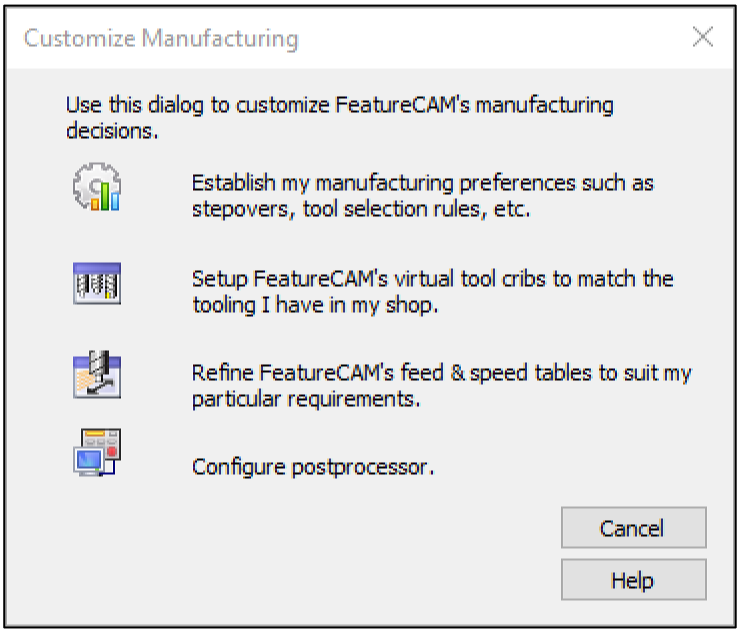
9 **NC Code.** Machine-specific G-codes are generated from the toolpaths. Translators are provided for many different NC controls and include a program for creating new translators.

Customize MFG

This menu gives you the ability to change your configuration file, change or add a tool crib, create or alter feed and speeds tables or configure posts processors.

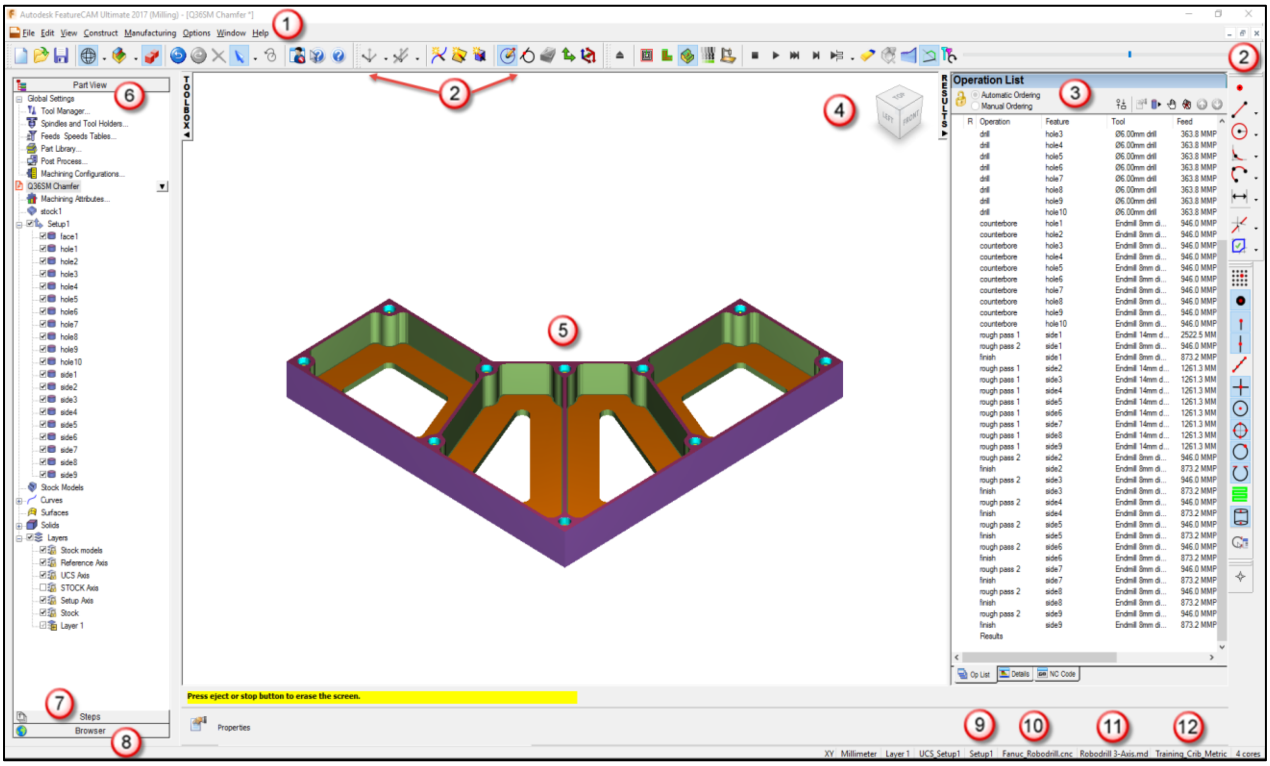
Some Steps are displayed or not displayed depending on which options you have purchased or have turned on.

Customize Manufacturing.

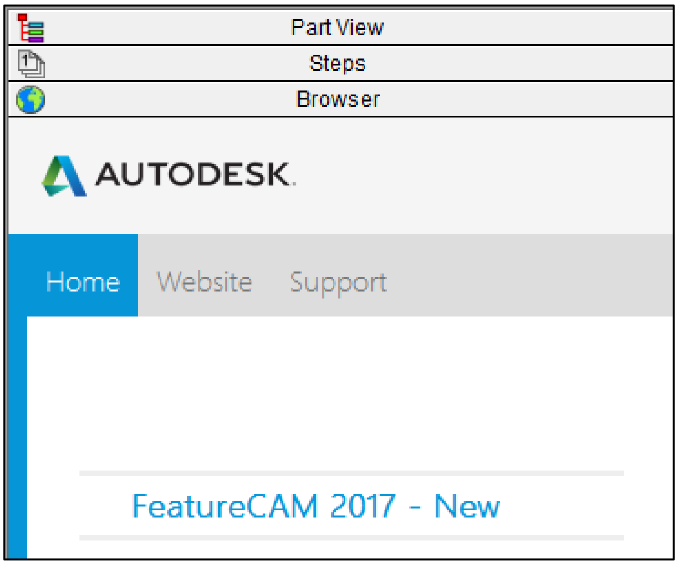


- **FeatureCAM** automates the entire part programming process; you can customize all of the System Settings including Feed/Speed Tables, Tooling Databases or Feature settings.

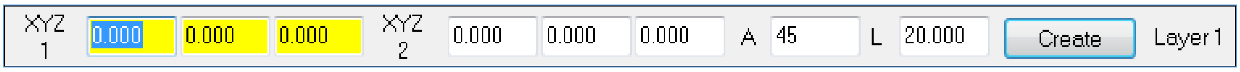
Screen Layout



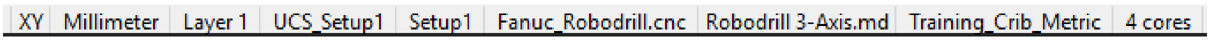
8 **Browser -** Located on the Left hand side **Toolbox** below **Steps**. Your computer must be connected to the Internet. The Browser provides a link to the Internet. Learn about the Browser by participating in the various sections with downloadable examples on the opening page, then click on the links and explore the contents.



Geometry Bar - Located near the bottom. This area is constantly used and is where the dimension boxes are used to locate the Geometry on your stock material in the Graphics Window.



9 **Status bar -** Metric or Inch. This is Located at the very bottom of **FeatureCAM**. It contains the plane you are working in **(XY–XZ-YZ),** the unit of measure (**Inch** or **Metric**), the layer you are working in, the Setup, Post Processor and the Tool Crib. Normally: - **BasicMetric** for **Metric Tools**. We also show the number of Cores Used.



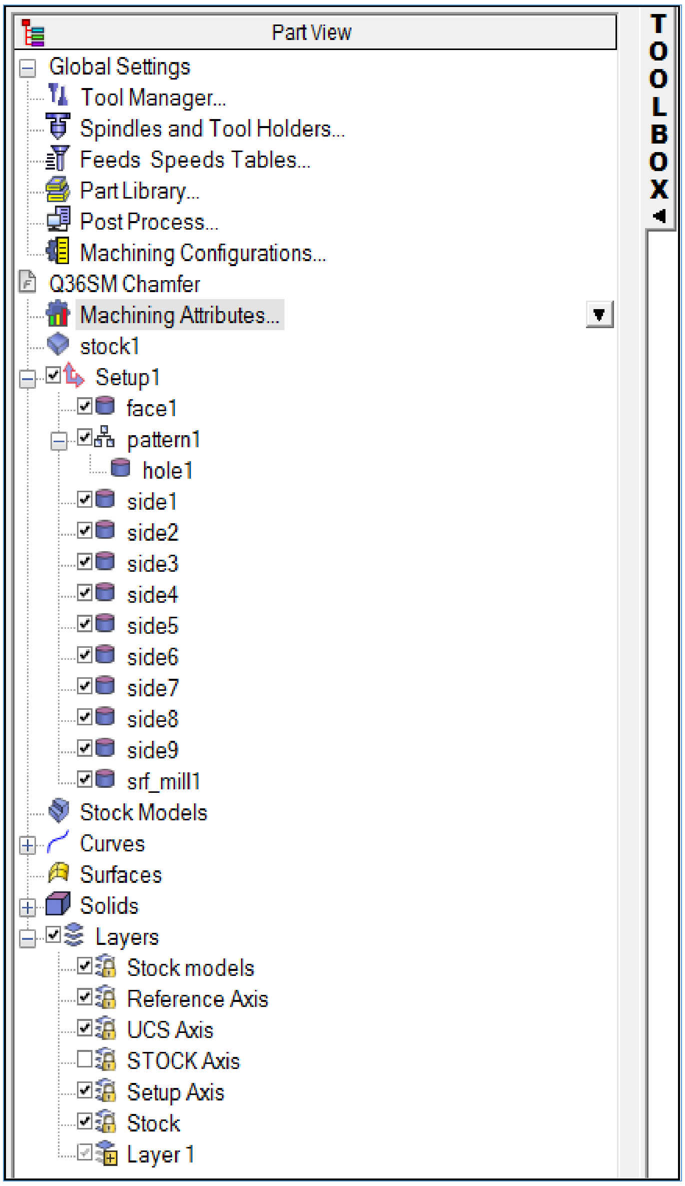
*Analyzing the above screenshot from the **Status Bar**.*

- XY is the current drawing plane being used. **Inch** or **Millimetres** are the type of units being used e.g. length ***4.000".*** **100mm. Layer1** is the current active Layer. **UCS_Setup1** is the current **UCS (User Coordinated System)** being used. **Setup1** is the current active setup in which features are created.

10 **Fanuc Robodrill. CNC** is the current **Post Processor** being used.

11 **Fanuc Robodrill.MD** is the current machine simulation file.

12 **Training_Crib_Metric** is the current **Tool Crib** and we are using **4 cores** on this computer



Clicking on the objects in the **Part View** enables different functions. Left-click selects the object in the **Graphics Window**. Right-click reveals menu of options available for the object. Clicking on a **"+"** will expand the view to reveal objects that are **subordinate**.

In the Centre of the screen is the Graphics Window. All drawings and toolpath simulations happen in this window. On the right is the **Manufacturing Results Window**. The automatically generated documentation and part programs are shown in this window.

- At the bottom of the screen is the **Assistance Bars** for **Geometry** and **Status**. The **Assistance Bar** is where you are prompted when creating **Geometry**.
- The screen shots shown above and on the next page show the **Results Window.** This shows you the order in which the machining is carried out. The Feature in which it is associated too, plus the Tools used in that sequence and the corresponding Feeds and Speeds. At the end you have the depth that the Tool will go to for example **25mm**
- There are three tabs available on the results window at the bottom.

- Op List | Details | NC Code — These are **Op List, Details** and **NC Code**.

RESULTS

Operation List

- Automatic Ordering (selected)
- Manual Ordering

F	Operation	Feature	Tool	Feed	Speed	Depth
	finish	face1	facemillM32...	5715.0 MMPM	10000 RPM	1.000 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	spotdrill	pattern1.hole1	center_M12...	363.8 MMPM	2293 RPM	9.830 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	drill	pattern1.hole1	TD_M0600:J	363.8 MMPM	4043 RPM	26.803 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	counterbore	pattern1.hole1	endmillM08...	946.0 MMPM	5912 RPM	5.000 mm
	rough pass 1	side5	endmillM20...	1261.3 MMPM	3153 RPM	25.000 mm
	rough pass 1	side5	endmillM20...	1261.3 MMPM	3153 RPM	25.000 mm
	rough pass 1	side4	endmillM20...	1261.3 MMPM	3153 RPM	25.000 mm
	rough pass 1	side4	endmillM20...	1261.3 MMPM	3153 RPM	25.000 mm
	rough pass 1	side3	endmillM20...	1261.3 MMPM	3153 RPM	25.000 mm
	rough pass 1	side3	endmillM20...	1261.3 MMPM	3153 RPM	25.000 mm
	rough pass 1	side2	endmillM20...	1261.3 MMPM	3153 RPM	25.000 mm
	rough pass 1	side2	endmillM20...	1261.3 MMPM	3153 RPM	25.000 mm
	rough pass 1	side1	endmillM20...	1261.3 MMPM	3153 RPM	30.000 mm
	rough pass 2	side1	endmillM12...	2522.5 MMPM	5255 RPM	30.000 mm
	finish	side1	endmillM12...	2328.5 MMPM	8085 RPM	30.000 mm

Manufacturing Details

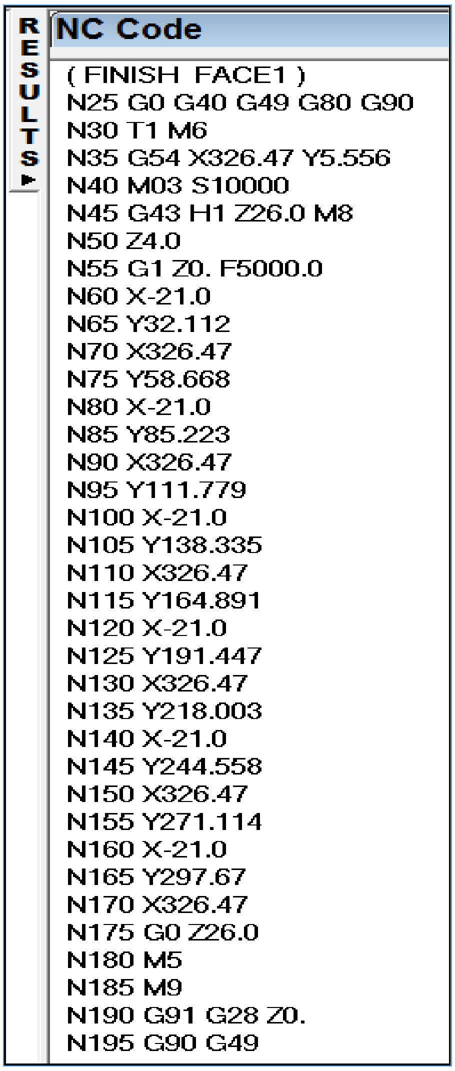
- Operation List
- Tool List (selected)

MANUFACTURING TOOL DETAIL SHEET

Part: Q36SM Chamfer
Setup: Setup1 (1 of 1)
Date: Tuesday, August 30, 2016 08:45:54

Crib: Training_Crib_Metric
Summary:

Slot 1:	facemill-32mm Dia	D 32.000 mm	H 30.000 mm		
Slot 2:	center_M1250-0500	D 5.000 mm	L 5.000 mm		
Slot 3:	Ø6.00mm drill	D 6.000 mm	L 60.000 mm		
Slot 4:	Endmill 8mm dia 2 Flute	D 8.000 mm	L 40.000 mm	F 2	T 0.000 mm
Slot 5:	Endmill 14mm dia 4 Flute	D 14.000 mm	L 35.000 mm	F 4	T 0.000 mm
Slot 6:	Endmill 14mm dia 2 Flute	D 14.000 mm	L 30.000 mm	F 2	T 0.000 mm


RESULTS

NC Code

```
( FINISH  FACE1 )
N25 G0 G40 G49 G80 G90
N30 T1 M6
N35 G54 X326.47 Y5.556
N40 M03 S10000
N45 G43 H1 Z26.0 M8
N50 Z4.0
N55 G1 Z0. F5000.0
N60 X-21.0
N65 Y32.112
N70 X326.47
N75 Y58.668
N80 X-21.0
N85 Y85.223
N90 X326.47
N95 Y111.779
N100 X-21.0
N105 Y138.335
N110 X326.47
N115 Y164.891
N120 X-21.0
N125 Y191.447
N130 X326.47
N135 Y218.003
N140 X-21.0
N145 Y244.558
N150 X326.47
N155 Y271.114
N160 X-21.0
N165 Y297.67
N170 X326.47
N175 G0 Z26.0
N180 M5
N185 M9
N190 G91 G28 Z0.
N195 G90 G49
```

- **Operation List Tab: -** This contains a table of different sequences in order of operation.
- **Details,** gives you a breakdown of the sequence of operations.
- **Tool List.** This gives a breakdown of the tools used.
- **NC Code.** CNC code that will be output from the currently **selected Post Processor**.

*Please note you have to run the **Simulation** before the **NC Code** is made available for output.*

How to create a part

Drawing a part in FeatureCAM

1. **Start FeatureCAM**.
2. **Draw geometry** in FeatureCAM.
3. **Resize Stock** to fit part.
4. **Create** and **move Setup** to desired location.
5. **Create** curves.

6. **Create** Features.
7. **Select Crib**, **Select Post**, and **Generate NC code**.

Importing a file into FeatureCAM

1. **Start FeatureCAM**.
2. **Import 2D** or **3D** files into FeatureCAM.
3. Use **Wizard** to **align part** and **create Setup**.
4. For Solid models use Automatic Feature Recognition **(AFR)** or use Interactive Feature Recognition **(IFR)** to create features which creates the toolpaths.
5. **Select Crib**, **Select Post**, and **Generate NC code**.

*Another example of the steps to take. Suppose you have a 2D drawing on your desk and you would like to create a solid model and a **CAM** program. To do this, we would **follow the steps sequentially**:*

1. **Specify type of stock** and stock dimensions,
2. **Draw Geometry.**
3. **Create Curves** from Geometry.
4. **Create Surfaces**.
5. **Create Solid from Curves**. (Extra Cost Option)
6. Use **AFR (Automatic Feature Recognition)** or **IFR (Interactive Feature Recognition).**
7. **Create Features** which **creates the toolpaths**.
8. **Select Crib**, **Select Post**, and **Generate NC code**.

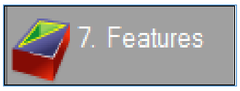
*You can access the majority of **FeatureCAM's** functions through these buttons. A summary of each of the **Steps** function is listed below. **Tip:** We recommend that if you are just starting out with the program or demonstrating it to others that you use the **steps toolbox** instead of toolbar buttons in other areas of the interface to keep the interface simple*

What is a Feature?

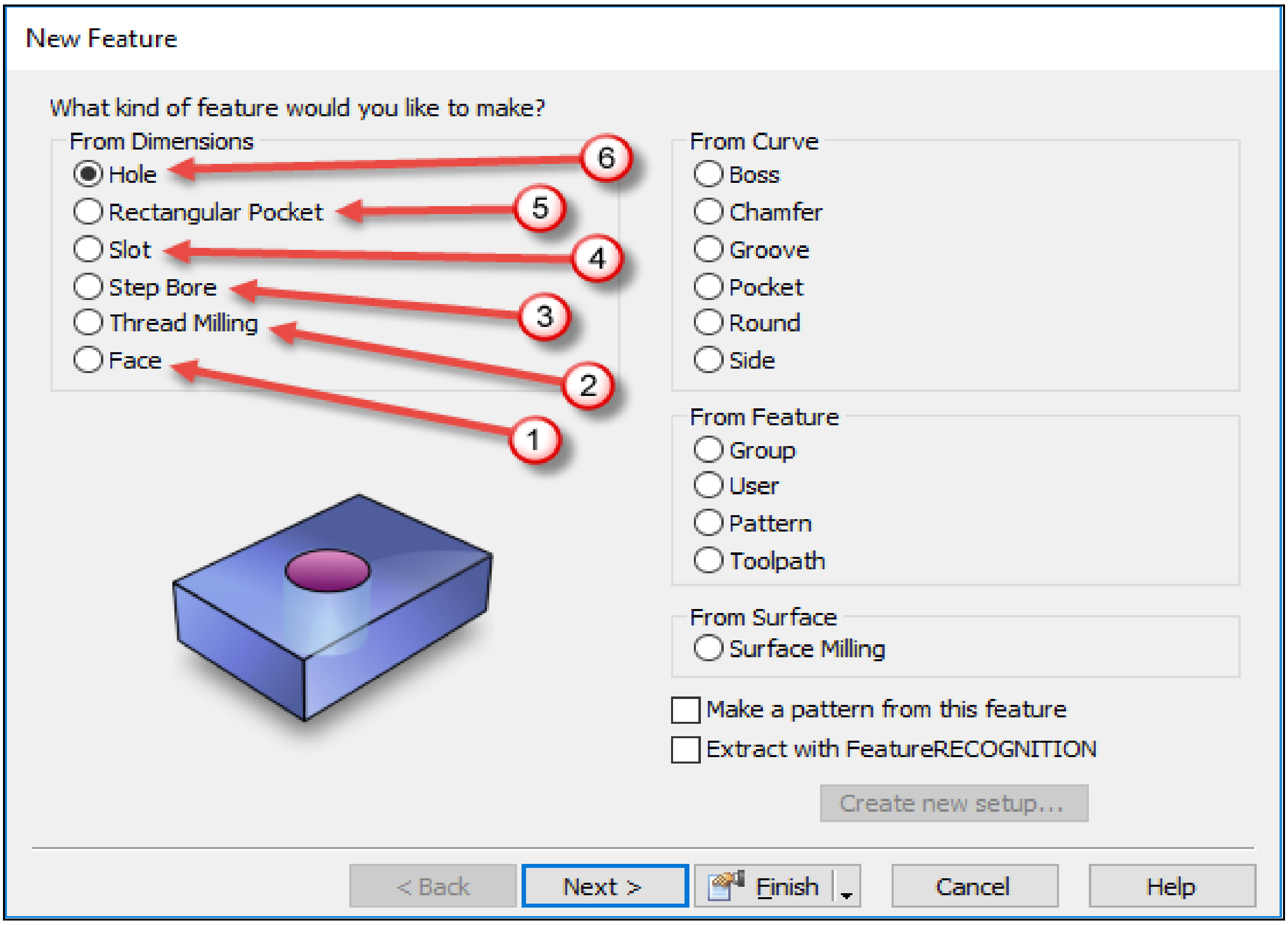
- If we look at a Part like the one shown on the next page and consider how we would manufacture it we can see that it can be broken down into a series of **Machining Operations**. For example we would probably want to start by **Facing** the top of the part, then machine the **Rectangular Pocket** and **Step Bore** and then drill the **Holes** in the part and finish of with **Thread milling** the **Step Bore**. Each of these areas of the part **Face, Rectangular Pocket, Hole**, **Step Bore**, **Thread Milling** and so on, is a **Feature** of the part.

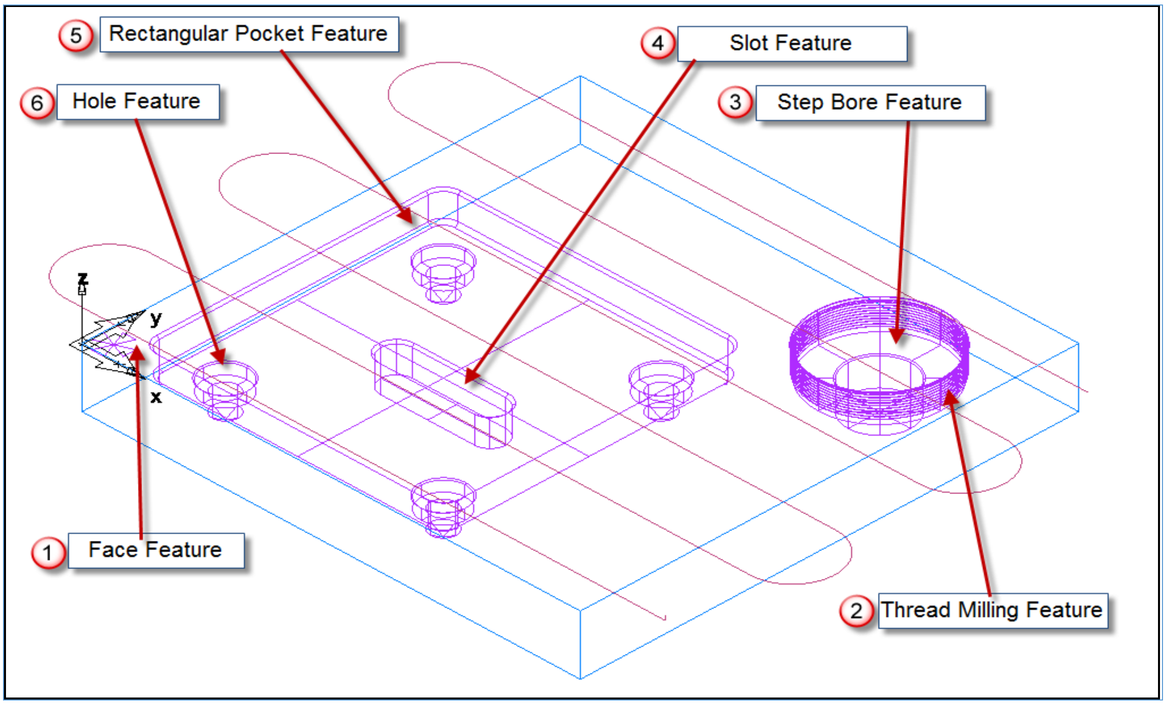
- **FeatureCAM** defines a part in terms of **Features** using common shop floor terminology for instance an area that looks like a **Pocket** can be machined using a **Pocket Feature**. When you define an area as a **Pocket**, **FeatureCAM** will then automatically decide what operations to use to machine that area. All you have to do is say "This is a **Pocket**" and **FeatureCAM** will select suitable **Roughing** and **Finishing Operations**, **Tooling**, **Speeds** and **Feeds**. You are then free to modify these default values to suit your particular situation. By automating the generation of operations the overall time taken to program a part can be drastically reduced.

*There are four different ways to activate the **New Features** menu.*

- Select **Features**  from the **Steps** menu.

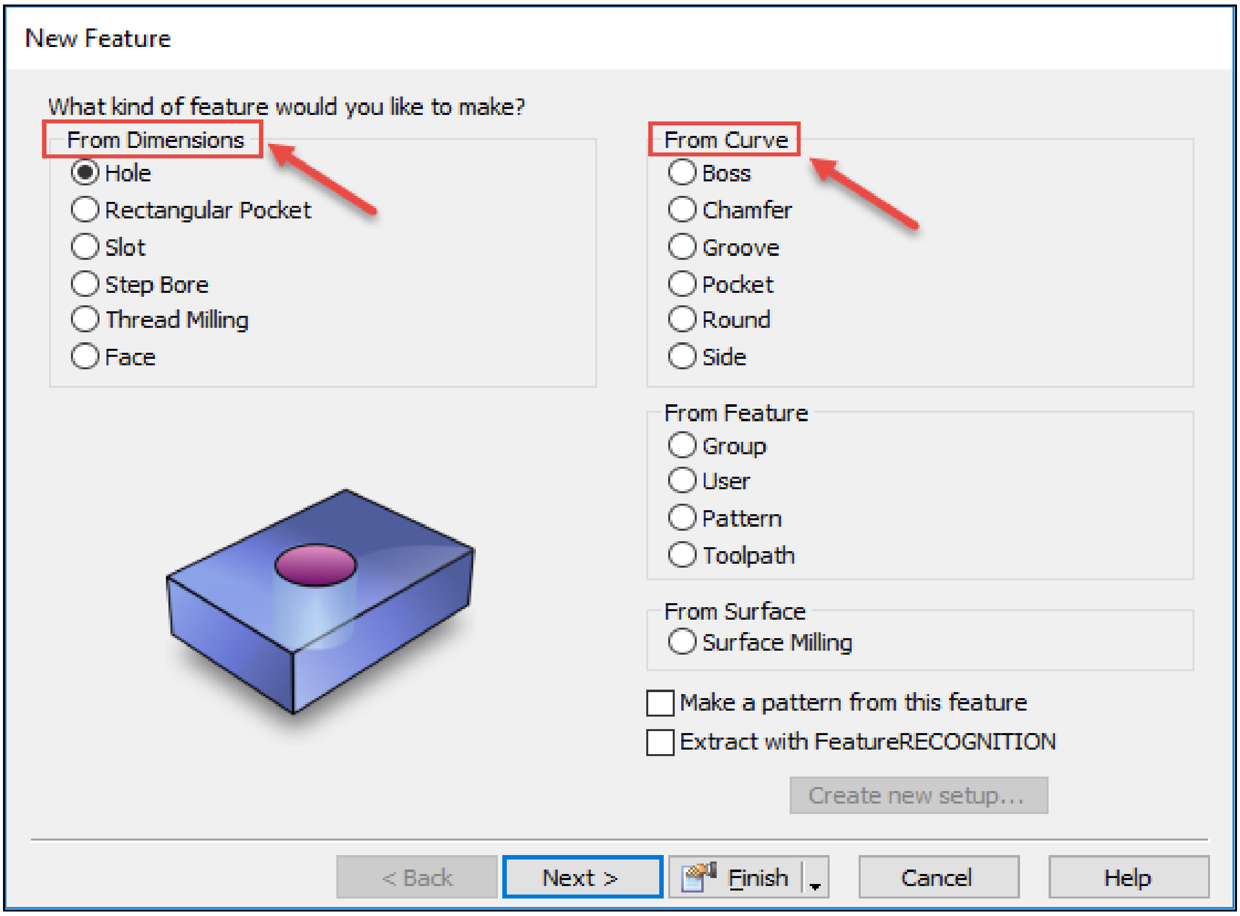
- Select **Ctrl + R New Feature.**
- Select **Construct>New Feature.**
- Select **New Feature** from the **Advanced Toolbar.**





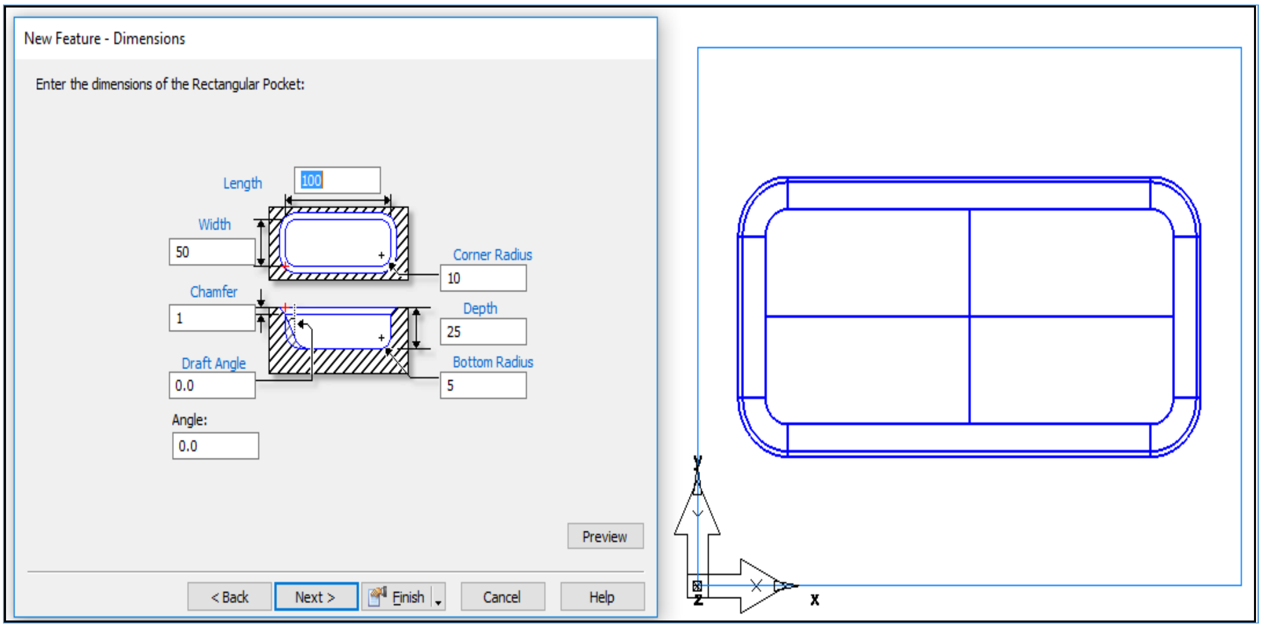
How is a Feature Created?

- There are a number of different ways in which **Features** can be created in **FeatureCAM**, which method you choose will depend upon the data available to you. In broad terms the methods are: - **From Dimensions** and **From Curve**.



From Dimensions

If you are working from a print (Paper Drawing) then often the quickest way of creating features is to pull the ***Dimensions*** *from the drawing and enter them directly into* ***FeatureCAM****. For example you can create a* ***Rectangular Pocket*** *by entering its* ***Length, Width, Depth*** *and* ***Location****. Additional attributes for the* ***Feature*** *can also be added such as a* ***Draft Angle****,* ***Fillet Radius*** *or* ***Chamfer****.*



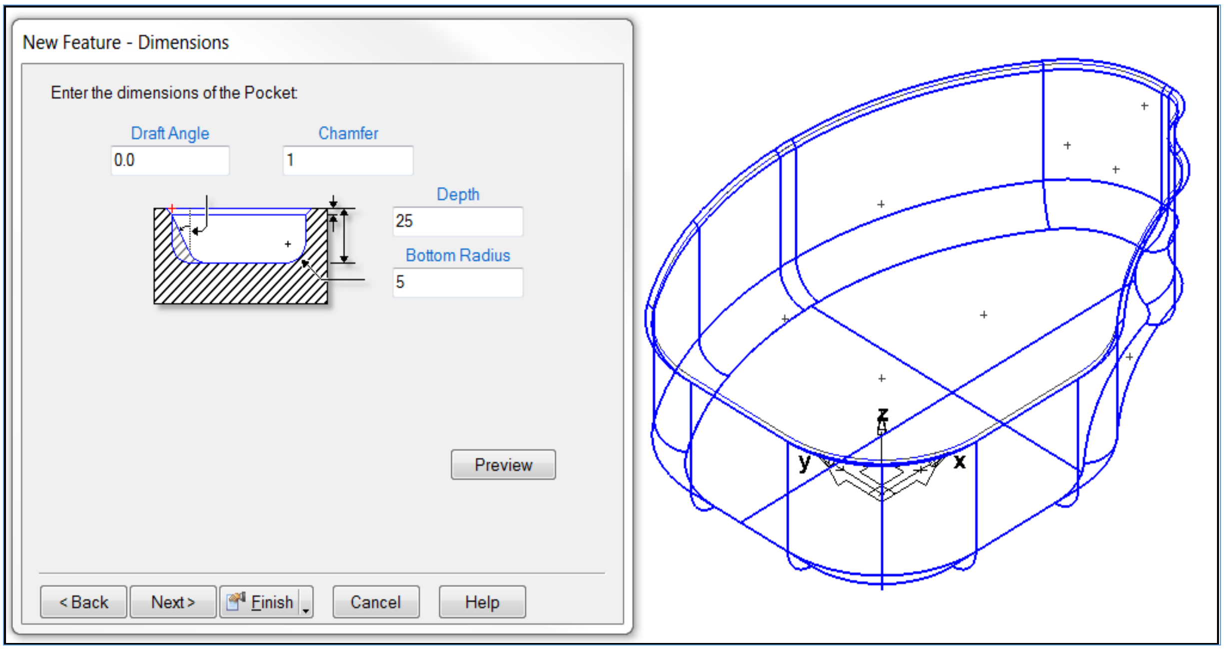
New Feature from dimension available.

- **Hole** Feature
- **Rectangular Pocket** Feature
- **Slot** Feature
- **Step Bore** feature
- **Thread Milling** feature
- **Face** feature

From Curve

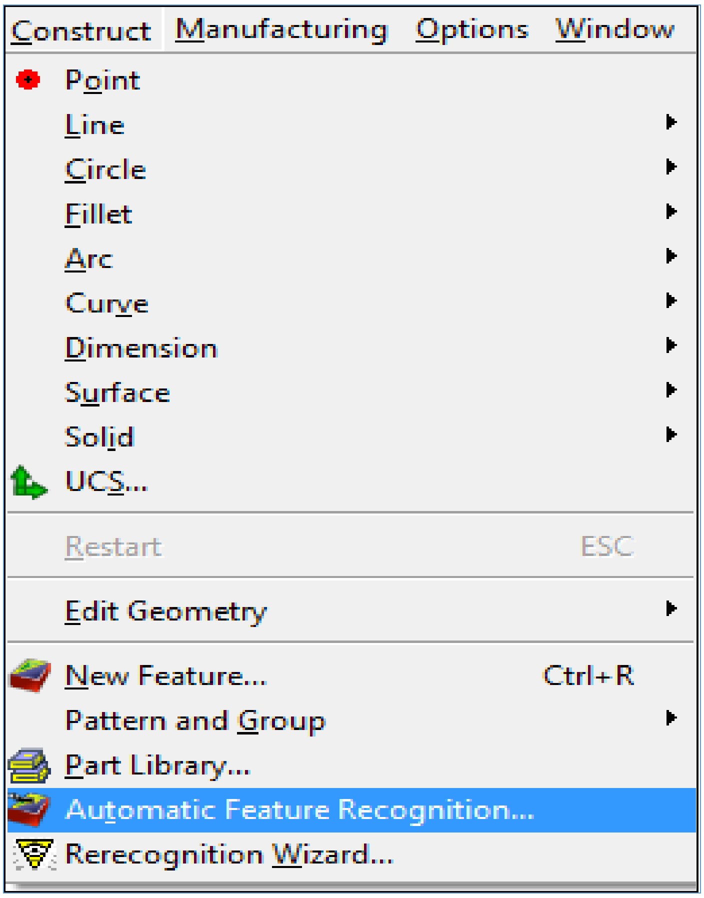
*If you have a **print** or a **2D drawing** of a part in electronic format for example **.dxf** or **.dwg** you can create **Curves** for more irregularly shaped parts and then create the **Feature** from the **Curve**. Just select **Closed Curve** select the **2D Geometry** and select the **Create** button. For open profiles use **Pick Curve Pieces** and create a **Side** feature.*

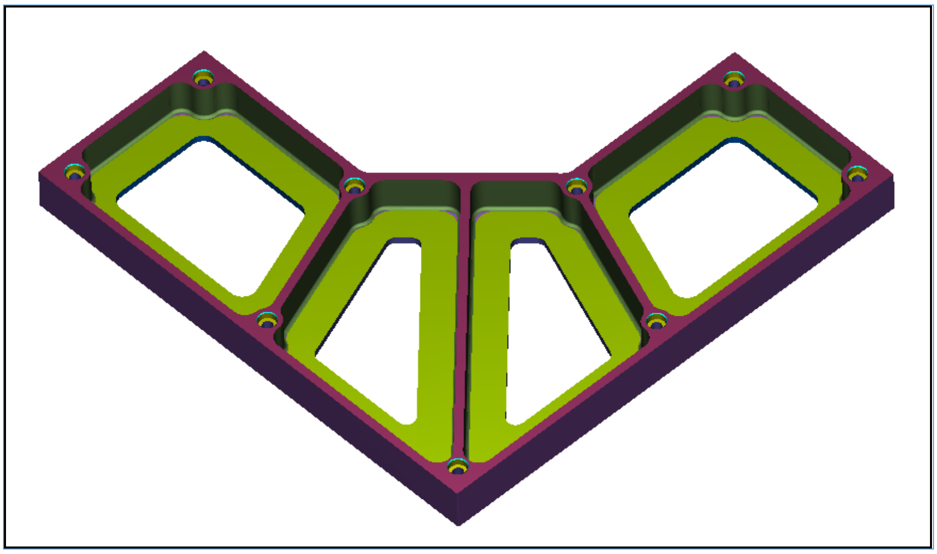
- **Boss** Feature
- **Chamfer** Feature
- **Groove** Feature
- **Pocket** Feature
- **Round** Feature
- **Side** Feature (For open and closed profiles)



Automatic Feature Recognition **(AFR)**

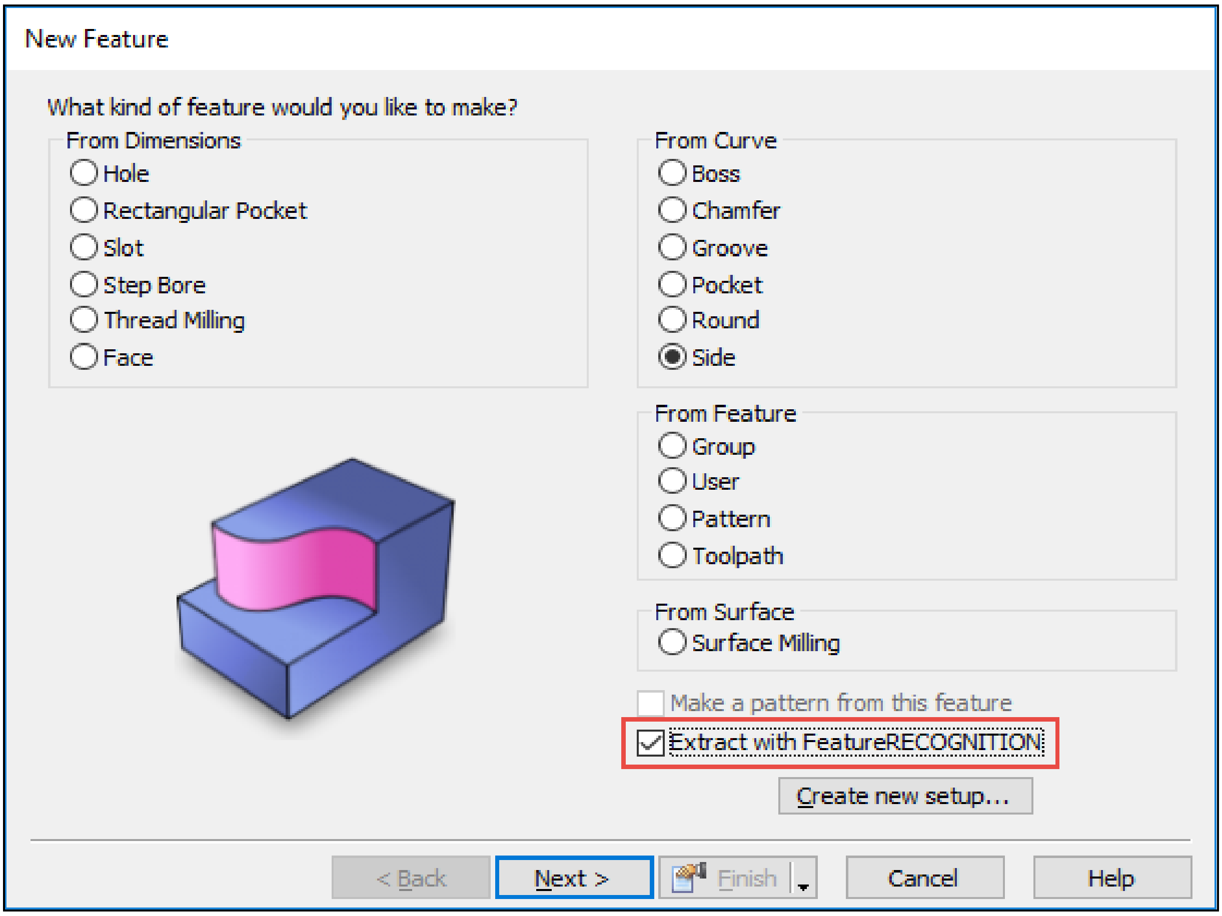
- If you are working from a **Solid Model** then **FeatureCAM** has the capability to analyse the part and identify **Features** automatically. Basically you say, "Here is the part, go and program it". **FeatureCAM** then looks for **Pockets, Bosses, Sides,** and **Holes** and so on and prepares the necessary **Machining Operations**, selects the **Tools and Feeds** & **Speeds**. This can work very well on simple parts but on more complex parts it may require some manual tweaking to get the optimal toolpaths. You could use **(IFR)** where you have more control over machining your part.





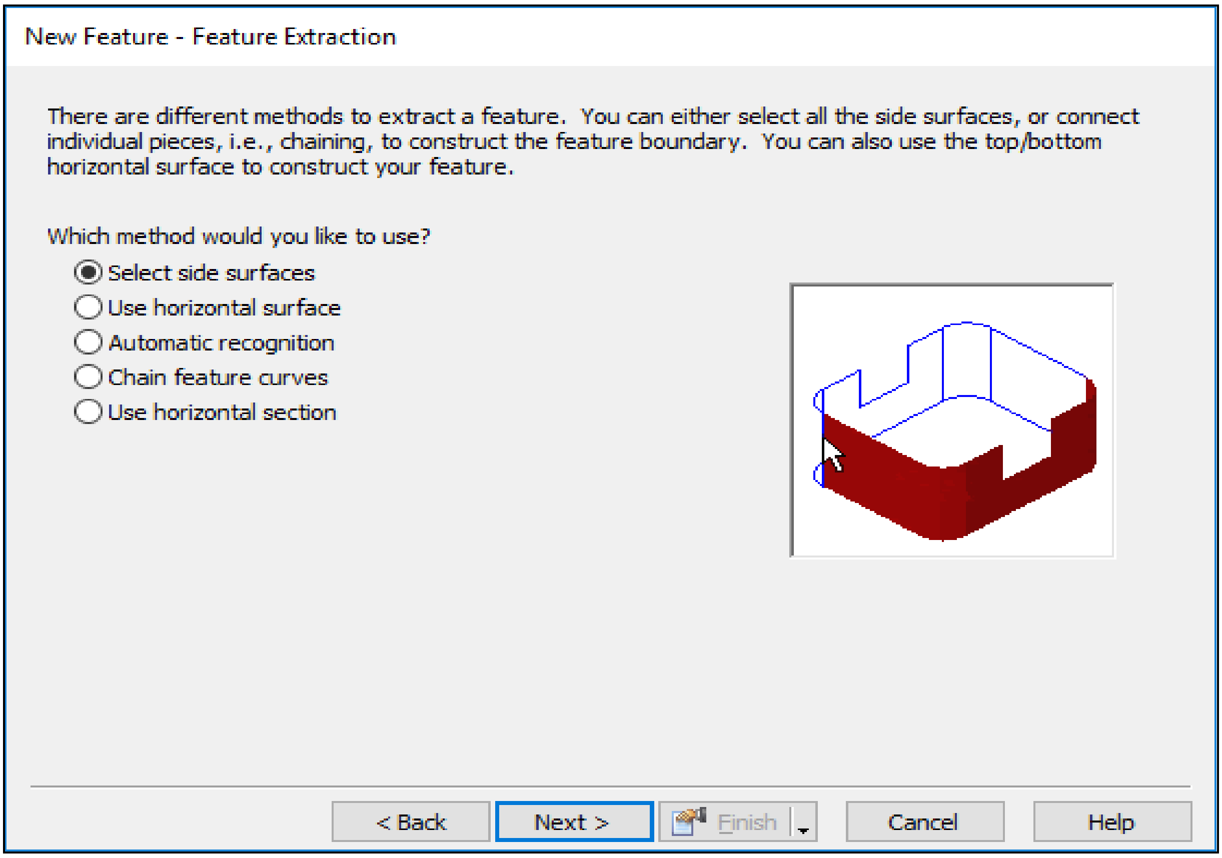
Interactive Feature Recognition **(IFR)**

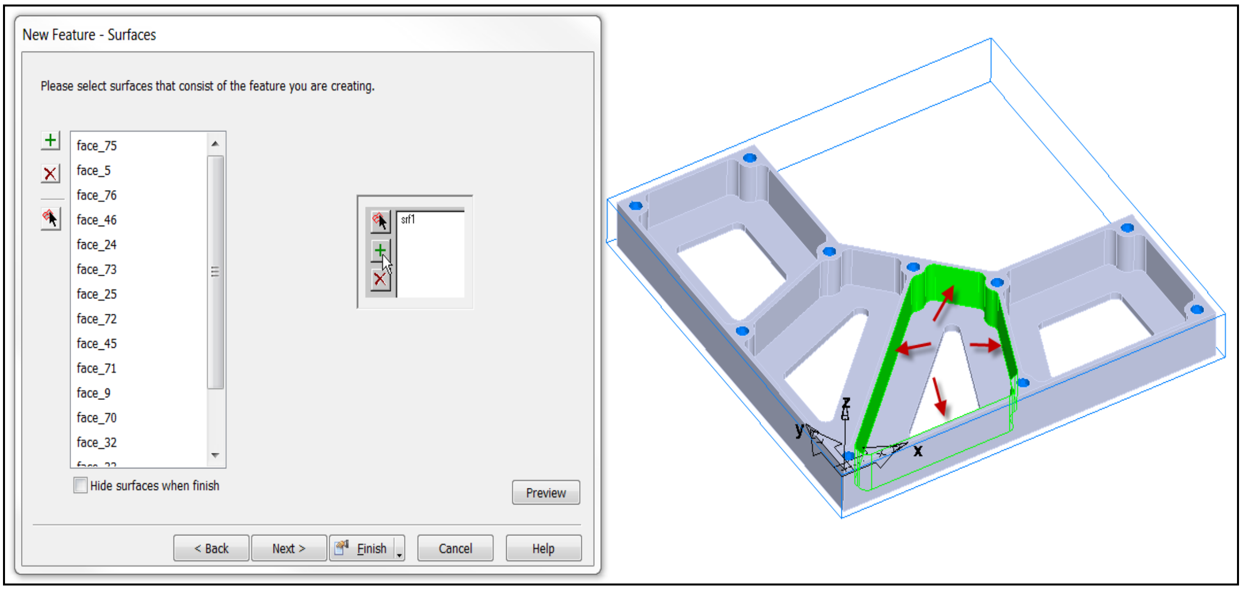
- This also works off a solid model in a similar manner to **AFR.** The difference is that you tell **FeatureCAM** what type of feature it is that you are looking for, for example "**Look only for Side features**" or **"Look only for holes"** with a diameter of less than **25mm.** This takes a little longer than **AFR** but gives you greater control over how the features are created and often results in a shorter cycle time for machining.



We also have different methods of extracting features from the model.

- **Select side surfaces** - Select the surfaces that represent the sides of the feature, such as the walls of a pocket.
- **Use horizontal surface** - Select a horizontal surface that represents the shape of the feature, such as the bottom of a pocket or the top of a boss.
- **Automatic recognition** - automatically recognize features.
- To create all recognition features at the same height, select **Force same Z height** and enter an **Elevation** to specify the height at which you want to create the features.
- **Chain feature curves** - Pick curves from a solid to define the shape of the feature.
- Enter a **Wall Angle** to angle the walls of the feature.
- Enter an **Elevation** to create the feature at a different height to the curves you select.
- **Use horizontal section -** Create features from a Z-slice through a solid.





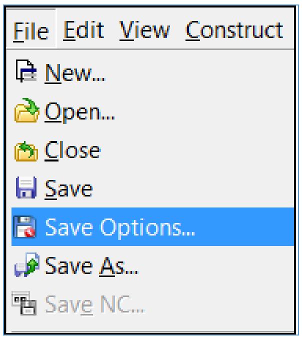
Saving your work

- **Question?** What is the difference between **Save** and **Save as**?
- **Answer:** When you are working on a document, it is a good idea to save your work every couple of minutes. In the **File** menu of the program, you will typically find the options "**Save**" and "**Save As**"
- Choosing "**Save**" simply saves your work by updating the last saved version of the file to match the current version you see on your screen.

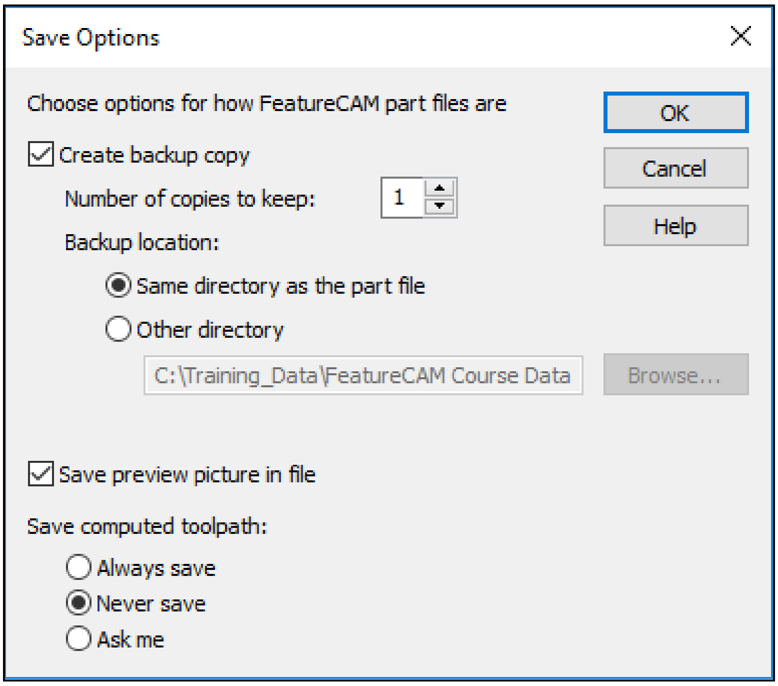
- Choosing "**Save As**" brings up a prompt to save your work as a file with a different name. For example, you might choose to save a document called "**FeatureCAM_opp1**". This way, you can save you file at different stages and keep multiple versions on your hard drive.
- Choosing "**Save**" and then "**Save As**" is also a good way to make a backup of your file. Just be sure to rename the file something new when you choose "**Save As**" or you will overwrite the current saved version, just like choosing the "**Save**" command would do.

Save Options

- In the **File** menu you have **save options.**



- See **Save Options** below. Adjust your computer to the same as the options shown.



- **Create backup copy: -** Select this option to save a number of previous versions of your part as you work. Enter the **Number of copies to keep** and set the **Backup location**. When you save a file, the previous version(s) are saved to disk using the name, but prefixed with **Backup of.** The latest version of the file is always saved using the name of your FeatureCAM part.
- **Save preview picture in file: -** Select this option to store an image of the part in the file. This image is displayed in the **Open** dialog.

- **Save computed toolpath: -** For certain parts, generating toolpaths can be time-consuming and you may want to save them for the next time you open a part. The setting of save computed toolpath controls the default behaviour for saving toolpaths. Select from **Always save**, **never save**, or **ask me** to be prompted each time you save a part. The toolpath is saved as an **.fmp** file.

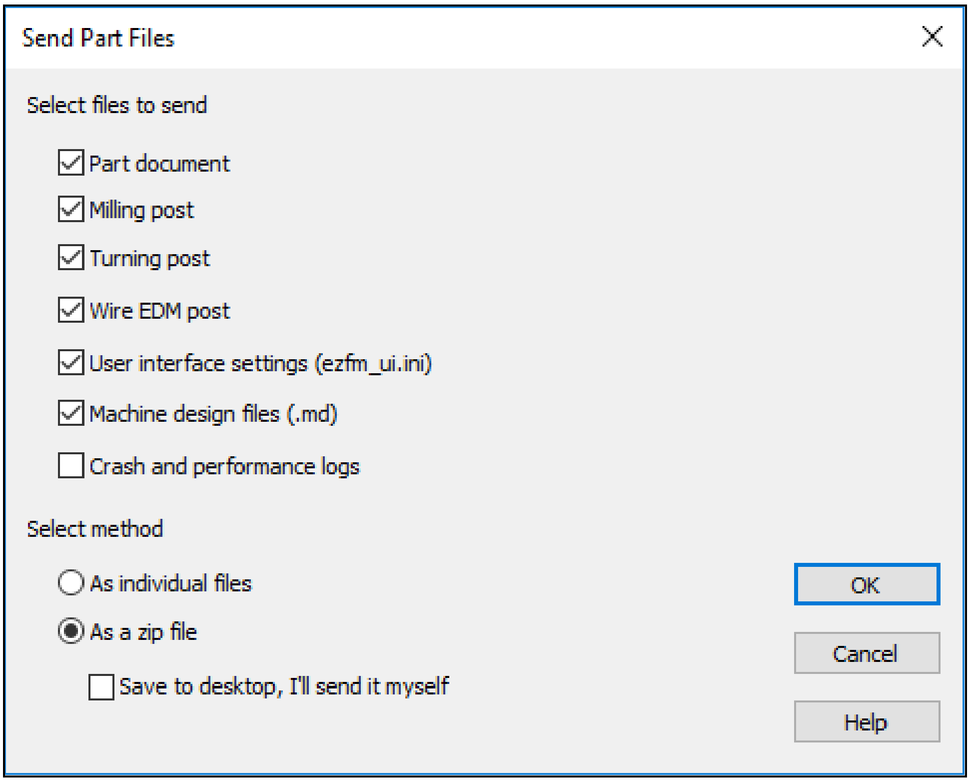
  This is different than saving the NC text file that the NC machine reads. Instead you are saving the FeatureCAM internal toolpath representation.

  *If you upgrade to a newer version of **FeatureCAM**, the toolpath is disregarded and recomputed.*

- Click the **OK** button to save your settings and close the dialog.
- **Cancel: -** Click the **Cancel** button to close the dialog without saving any changes.
- **Help: -** Click the **Help** button to open this Help topic.

Send Part Files dialog

*To display the **Send Part Files** dialog, select **File > Send** from the menu.*



Select the files you want to send from:

- **Part document: -** The current **.fm** file.
- **Milling post: -** The current milling **.cnc** file.
- **Turning post**: **-** The current turning **.cnc** file.
- **Wire EDM post: -** The current Wire EDM **.cnc** file.

- **User interface settings**: **-** The **.ini** file containing your user interface preferences.
- **Machine design files (.md): -** the current **.md** file.
- **Crash and performance logs**: **-** Select this option to create a separate .**zip file** of your system's crash and performance logs.
- Select how you want to send the files:
- **As individual files**: - The individual files are attached to a new email.
- **As a zip file**: - The files are zipped and the .zip file is attached to a new email.
- **Save to desktop**: - The files are zipped and the .zip file is saved to your desktop.

Size confirmation

- If you are sending **.zip files**, a Size Confirmation warning dialog is displayed:
- The average email server limits attachments to **10 MB**.
- The total size of the attachment(s) is: N (where N is the size of your attachment.
- Would you like to continue sending?
- Click **Send** to attach the files to an email.
- Click **Don't Send** to keep the files on your desktop.
- Click **Cance**l to discard the files.

FeatureCAM file types

You can save several different types of files for your part.

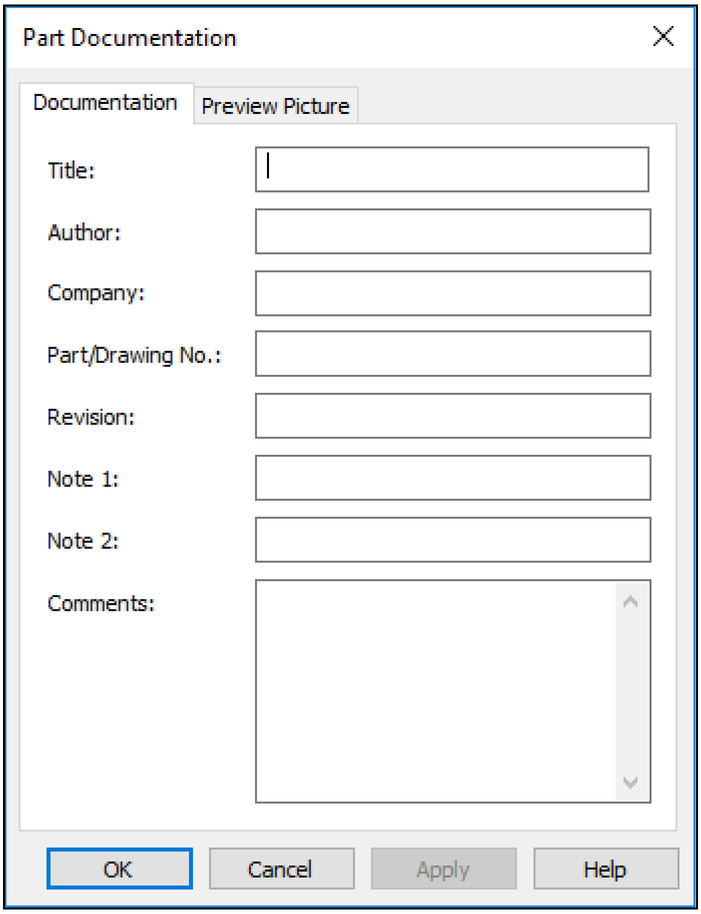
- **.op** is the Manufacturing Operation Sheet and is the same information shown on the **Details** tab when you select **Operation List**.
- **.tl** is the Manufacturing Tool Detail Sheet and is the same information shown on the **Details** tab when you select **Tool List**.
- **.txt** is a text file containing the NC code for the particular part file.
- **.tdb** is a FeatureCAM tooling database that contains just the tools you used to create the part.
- **.fdb** is a FeatureCAM material database that contains the feed and speed tables used for the part.
- **.cdb** is a FeatureCAM machine configuration database that contains the settings for default attributes. Part Documentation

The Part Documentation dialog enables you to add comments to the printed documentation and set a permanent preview picture for the part.

- To display the Part Documentation dialog, select **File > Part Documentation** from the menu.
- The **Part Documentation** dialog contains two tabs: **Documentation and Preview Picture**

Documentation tab

- On the Documentation tab of the Part Documentation dialog, optionally enter a **Title, Author, Company, Part/Drawing No., Revision, Note 1, Note 2, and Comments**.
- To print these values along with the documentation, select Comments in the **Printing Options** dialog.



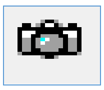
*If you use the custom setup sheet add-in, these values are copied to the **Setup Sheet Options** dialog.*

Preview Picture tab

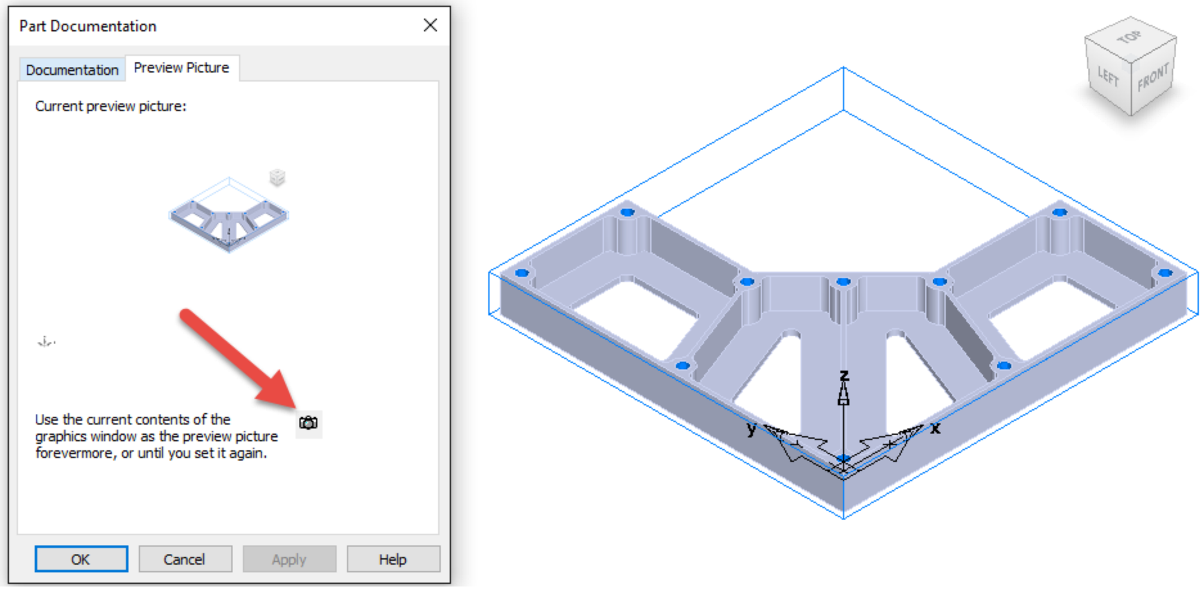
On the **Preview Picture** tab of the **Part Documentation** (see page 95) dialog, you can set a permanent preview image for the part. This image is displayed in the preview pane in the **File > Open** dialog when you select a file.

This image is normally updated each time you save the part. If you want to store a permanent image with the part:

1 Create the view of the part you want to store.

2 Click Update preview picture 

The current view in the graphics window is displayed as the ***Current preview picture****, for example:*



3 Click **OK** to save the preview picture or **Cancel** to close the dialog without saving the preview picture.

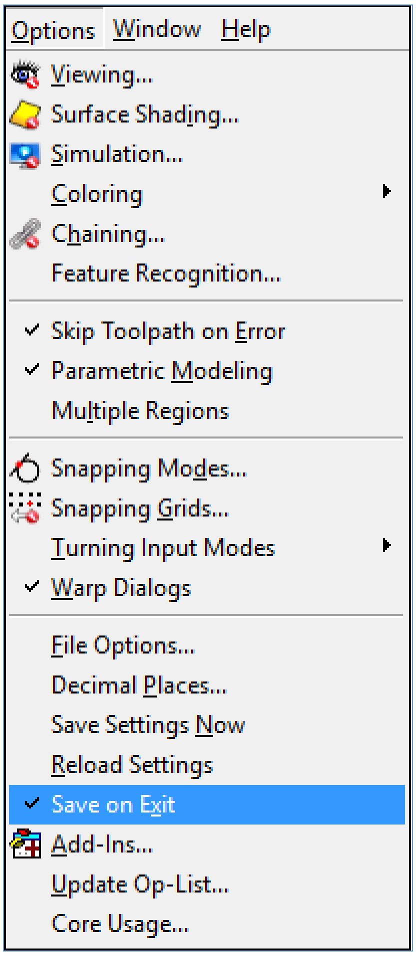
Saving your settings

- FeatureCAM uses two **.ini** files to store your settings:
- **Ezfm_ui.ini** contains toolbars, dialog locations, graphics settings, colours, and other user settings.
- **Ezfm_mfg.ini** contains manufacturing defaults and **.cfg** settings.

The **ezfm_mfg.ini** file contains default values for both inch and metric attributes. For example, there is an entry for **zrapid** in inches and also an entry for **zrapid_mm** in mm. In general, the name for the metric attributes is the same as the inch name, with a suffix of **_mm**.

There are three items from the ***Options*** *menu affect the communication with the* ***ezfm_ui.ini*** *and* ***ezfm_mfg.ini*** *files:*

- **Options > Save Settings Now**: **-** Writes the current settings to the files.
- **Options > Reload Settings**: **-** Reads the settings contained in the files into the program.
- **Options > Save on Exit**: **-** Saves the current settings when you exit the program. If this option is not selected, the settings for your current session are not saved to the files when you exit.



Import/Export

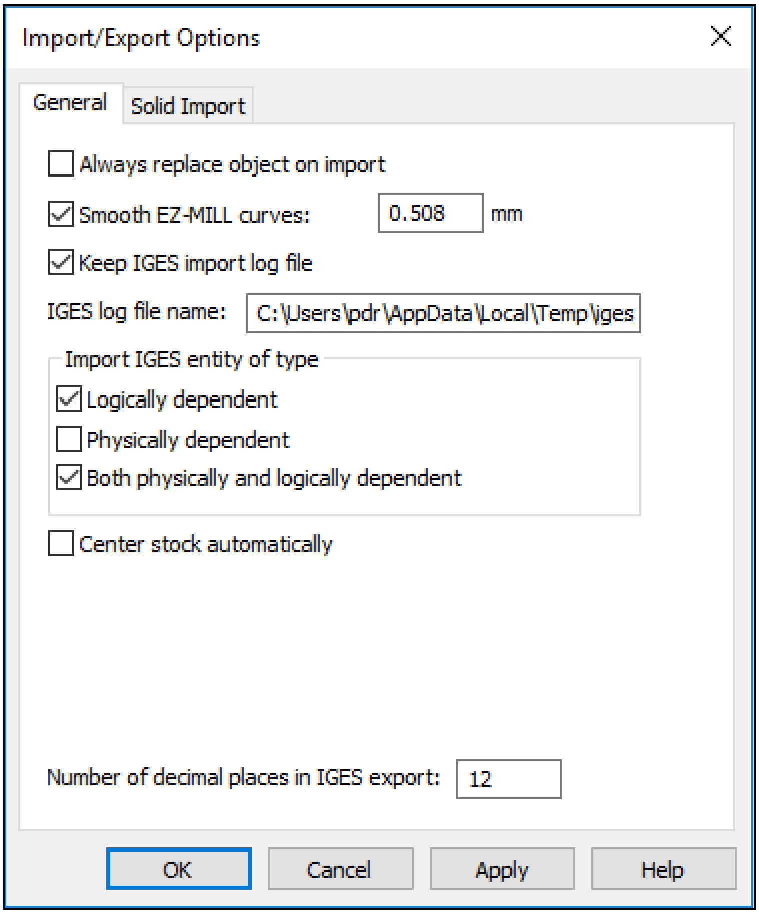
- You can import and export various file types into and from FeatureCAM.
- The settings for import and export are in the **Import/Export Options**
- To open this dialog select **File > Import/Export Options** from the menu.

Import/Export Options

- The **Import/Export Options** dialog contains settings for importing and exporting.
- To display the **Import/Export Options** dialog select **File > Import/Export Options** from the menu.
- The dialog has three tabs:
- **General**
- **Solid Import**
- **Digitized Data**

General tab

- The General tab controls how FeatureCAM **Imports** or **Exports** files. It contains the following options:

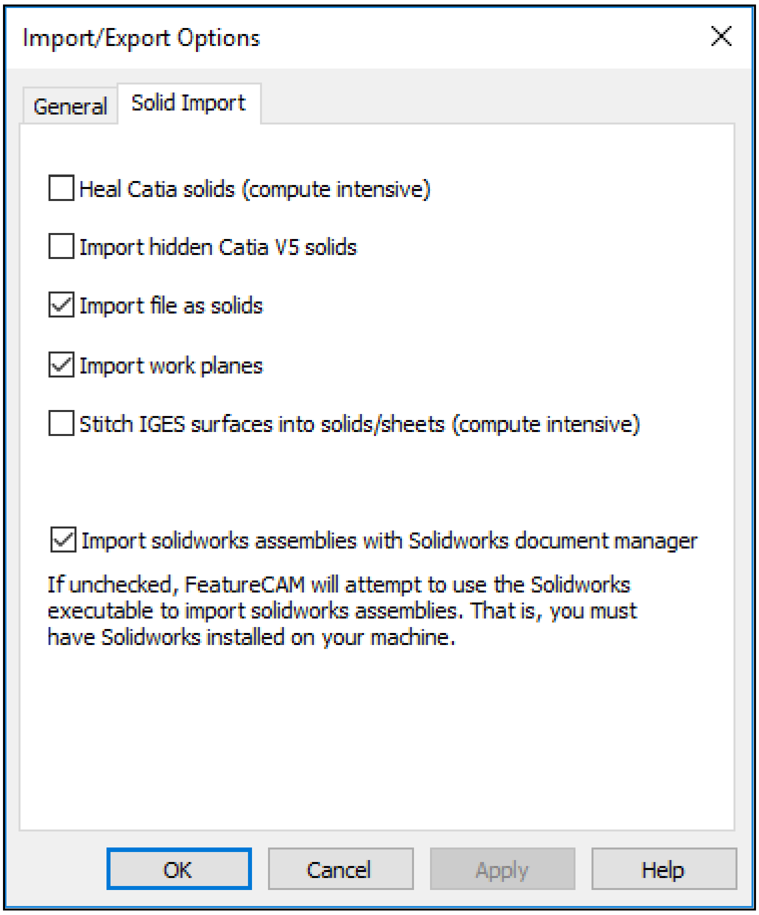


- **Always replace object on import**: **-** Select this option to overwrite objects of the same name when you import. Deselected prompts for every replacement. You can't have two objects with the same name.
- **Smooth EZ-MILL curves**: **-** Select this option to improve EZ-MILL curves on import because FeatureCAM has a higher resolution curve format.
- **Keep IGES import log file**: **-** Select this option to keep a log of the import process for later review or troubleshooting.
- **IGES log file name**: **-** Enter the path and filename for saving a record of the import process.
- **Import IGES entity of type**:
- **Logically dependent**: **-** can exist by itself but is also referenced by another entity (case of groups or grouping situation).
- **Physically dependent**: **-** generally, you should deselect this option to indicate you do not want physically dependent entities to be imported. But if you are having trouble with your IGES file and it is not importing properly, try selecting this option and re-importing. Entities in an IGES file are marked to be either physically dependent or not. Those that are marked to be physically dependent are entities that are used in the construction of other entities. For example, a trimming loop is physically dependent upon the trimmed surface that uses it. The trimming loop is not important all by itself, but is instead a building block of something else. The trimmed surface is in turn marked physically dependent to indicate that it is a building block of a parent entity, a solid

- By selecting **physically dependent**, you cause FeatureCAM to show everything in the IGES file - not just the top-level entities that are usually shown, but also all of the lower-level building blocks. This is generally not what you want. Generally you want only the top-level entity, for example the solid.
- **Both physically and logically dependent**: **-** meets both logically and physically dependent criteria (is referenced by at least two other entities: **-** the parent for the logical link cannot be the parent for the physical as well).
- **Centre stock automatically**: **-** Select this option to automatically size and position the stock so that it covers the imported data.
- **Number of decimal places in IGES export**: **-** determines how finely data is exported to the IGES format.

Solid import tab

The **Solid Import** tab contains the following options:



- **Heal Catia solids**: **-** Select this option to try to repair the faces or surfaces contained in the **CATIA .mod** file. This option can be time-consuming because it attempts to:
- Retrim the surfaces/faces against each other.
- Force edges of surfaces to actually lie on the surface.
- Simplify surfaces like converting a general surface into a cylinder.
- **Import hidden Catia V5 solids**: **-** Select this option to import all parts contained in the CATIA file even if those parts are hidden.

- **Import file as solids**: **-** Select this option to import files as solid models. If this option is deselected, the models are imported as surface models. We recommend that you import models as solids. If this option is selected and the solid fails to import properly, you are asked if you would like to attempt to heal the solid to try and fix the import problem.
- **Import work planes**: **-** When selected, planar surfaces in solids are imported. When deselected planar surfaces in solids are ignored.
- **Stitch IGES surfaces into solids/sheets**: **-** Select this option to try to create one or more solids or sheets from an IGES file. If an IGES file contains more than one solid or sheet, this is the most efficient way to create the multiple objects from the file.

Importing Files

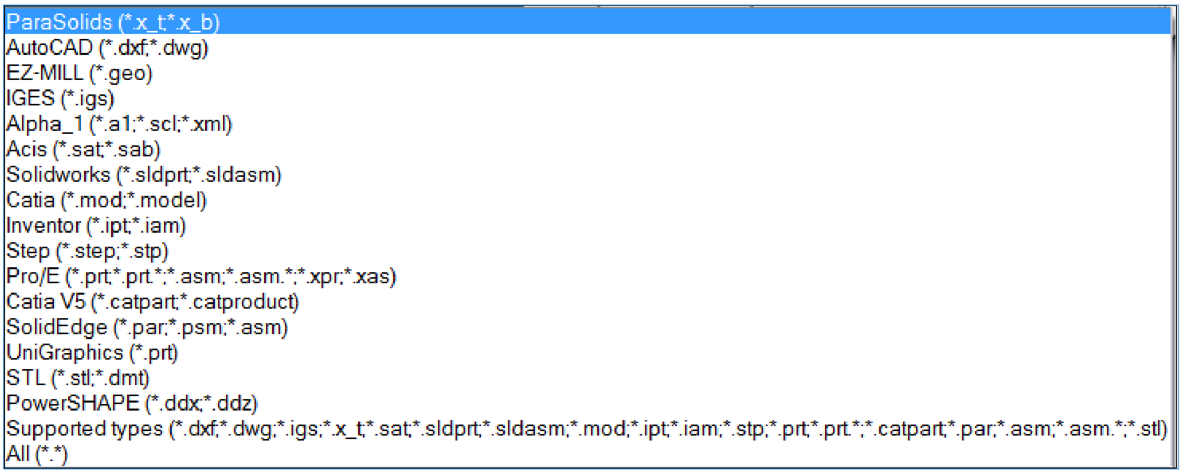
Use the **File > Import** menu option to load a CAD model from file.

To import a CAD model from a file:

1. Open a new or existing part file. You must have a part open to import geometry.
2. If you want to set or change the import options, select **File > Import/Export Options**.
3. Select **File > Import** from the menu. The **Import** dialog is displayed.
4. Select the file you want to import, and click **Open**.

   *A message asks if you want to review the log file: click **Yes** to display the results, or click **No** to continue. The **Import Results** wizard is displayed.*

5. Follow the instruction in the wizard to complete the process.



Please note that some of the file types are extra cost options

*Unless you have specifically ordered the FeatureCAM **.step** or **Stp** file import, you might get an error to say that this option needs to be licenced.*

*If this is the case then use the **Import using exchange** where the .step file import option is **free**.*

*Wherever possible avoid importing **.iges** files.*

*Please import solid models, where possible. The best option is Parasolid which has a file extension of **x_t** or **x_b** which is a binary file which is compressed. Another file format is **.SLDPRT** which is a **SolidWorks** file.*

Mouse buttons

- Each of the three mouse buttons performs a different dynamic operation in **FeatureCAM**.

Left mouse button Picking and selecting.

- This button is used for **Selecting Items** of the pull down menus, options within **forms**, and **entities** in the **graphics area**.

Middle mouse button Dynamics.

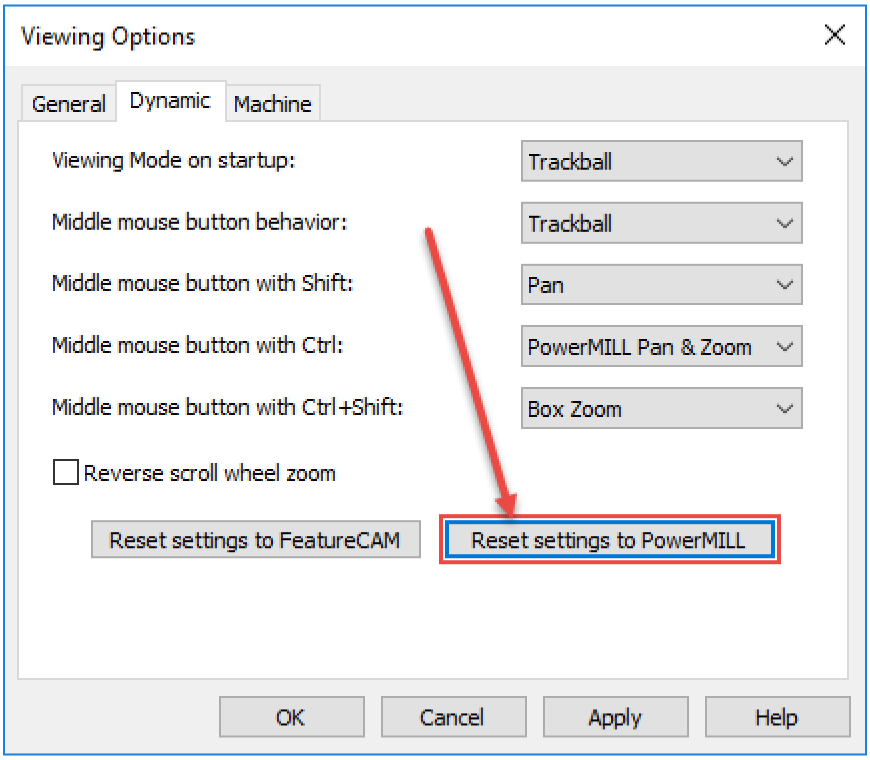
- In these options, you can set the behaviours to be specific viewing modes such as **Trackball, Rotate, Pan, Zoom, Pan** and **Zoom**, and so on, or choose the **Current Viewing Mode** option that allows the middle-mouse button (along with key combinations) to perform the viewing mode that is currently set in the toolbar.

- The most commonly used mouse feature is to hold the **Centre wheel down** and moving the mouse, this will rotate your part. Hold the shift key down at the same time and you can **Pan** anywhere on the screen. Hold the **shift key** down and where ever the mouse is it will **zoom** to that point by using the **Wheel**. This is excellent for looking closely at your component.
- **Zooming in and out: -** Hold down the **Shift** key and **mouse button 2**. Move the mouse up and down to zoom in and out.
- **Pan around the model: -**Click once with mouse button 2 to position in the Centre of the screen. Click and hold to zoom and then move the mouse in the required direction.
- **Zoom Box –** Hold down **mouse buttons 1** and then **button 2**, drag a box around the area to zoom. Release both buttons at the same time. **Rotate mode:** Hold down the **Ctrl** key and **mouse button 2**. Move the mouse up and down to zoom in and out.

Right mouse button Special Menus & FeatureCAM Options.

- When this button is pressed it brings up a local menu relating to whatever the Mouse Pointer is over, such as a named item in the **FeatureCAM Part View** or a physical entity in the graphics area. If nothing specific is selected the **View** menu appears.

Viewing Options



- For those users who are already familiar with other **Autodesk ®** products **DON'T PANIC!** Just do the following.
- **View** changes the way you interact with the view of the part. Selecting any of the options from the **View** menu puts you in *view mode.*
- Dynamic Viewing Options.
- **Dynamic Viewing Options** tab, you have two choices to reset the settings to:

  Reset to FeatureCAM Settings

  This button resets all the options in the dialog to **FeatureCAM** defaults, which are:
- **Dynamic Viewing Choice** upon FeatureCAM Start-Up - Pan and Zoom
- Middle-mouse button behaviour - **Current Viewing Mode**
- Middle-mouse button with Shift - **Pan and Zoom**
- Middle-mouse button with Ctrl - **Rotate**
- Middle-mouse button with Ctrl+Shift – **Zoom**

Reset to PowerMILL Settings

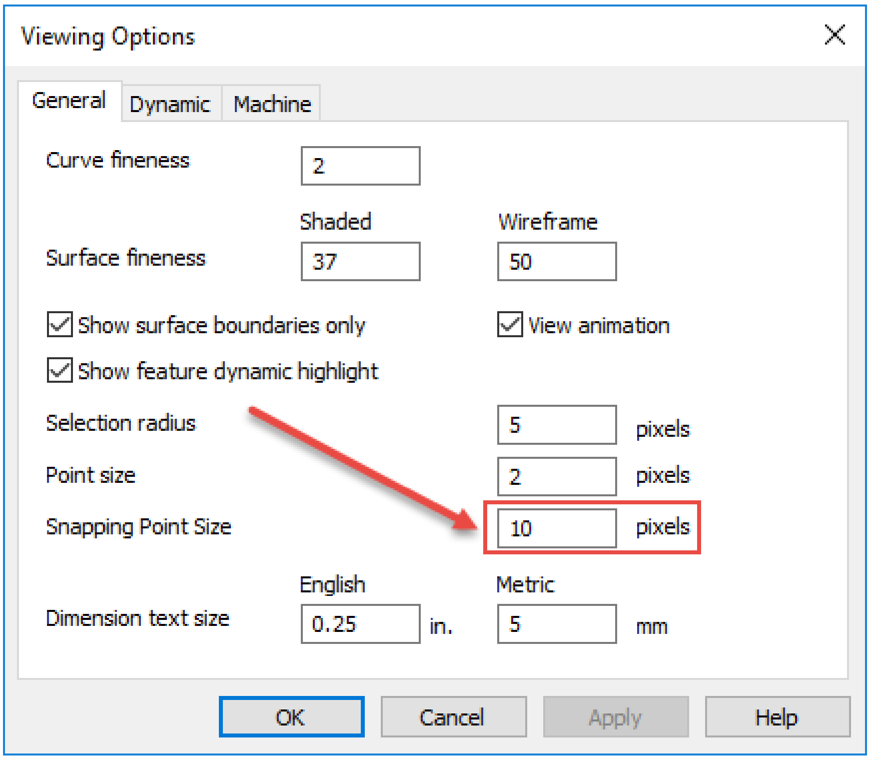
- This button resets all the options in the dialog to **PowerMILL** defaults, which are:

  Dynamic Viewing Choice upon **FeatureCAM** Start-Up - **Trackball**
- Middle-mouse button behaviour - **Trackball**
- Middle-mouse button with Shift - **Pan**
- Middle-mouse button with Ctrl - **PowerMILL Pan & Zoom**

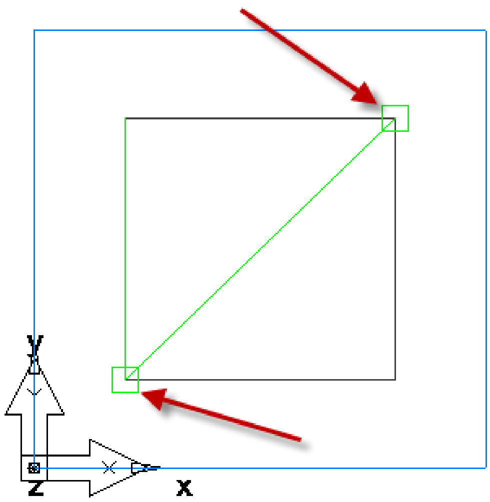
- Middle-mouse button with Ctrl+Shift - **Box Zoom**
- This menu controls the default viewing mode when **FeatureCAM** first opens. For example, if you select **Trackball**, the next time **FeatureCAM** starts up, the viewing mode is **Trackball.**

Viewing Options General

In the General Viewing Options we now have the ability to change the size of the Snapping point. By default this size is normally 4 pixels. In order to create larger points, change this to 8 pixels.

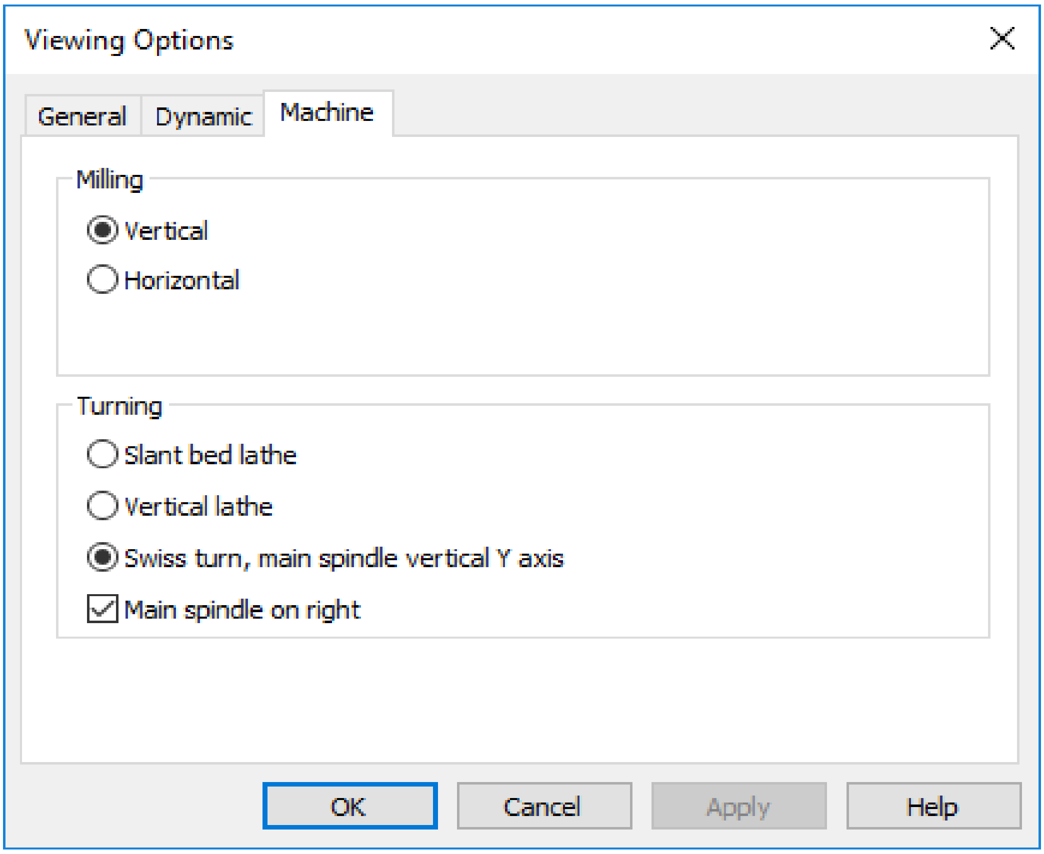


An example of this is shown below



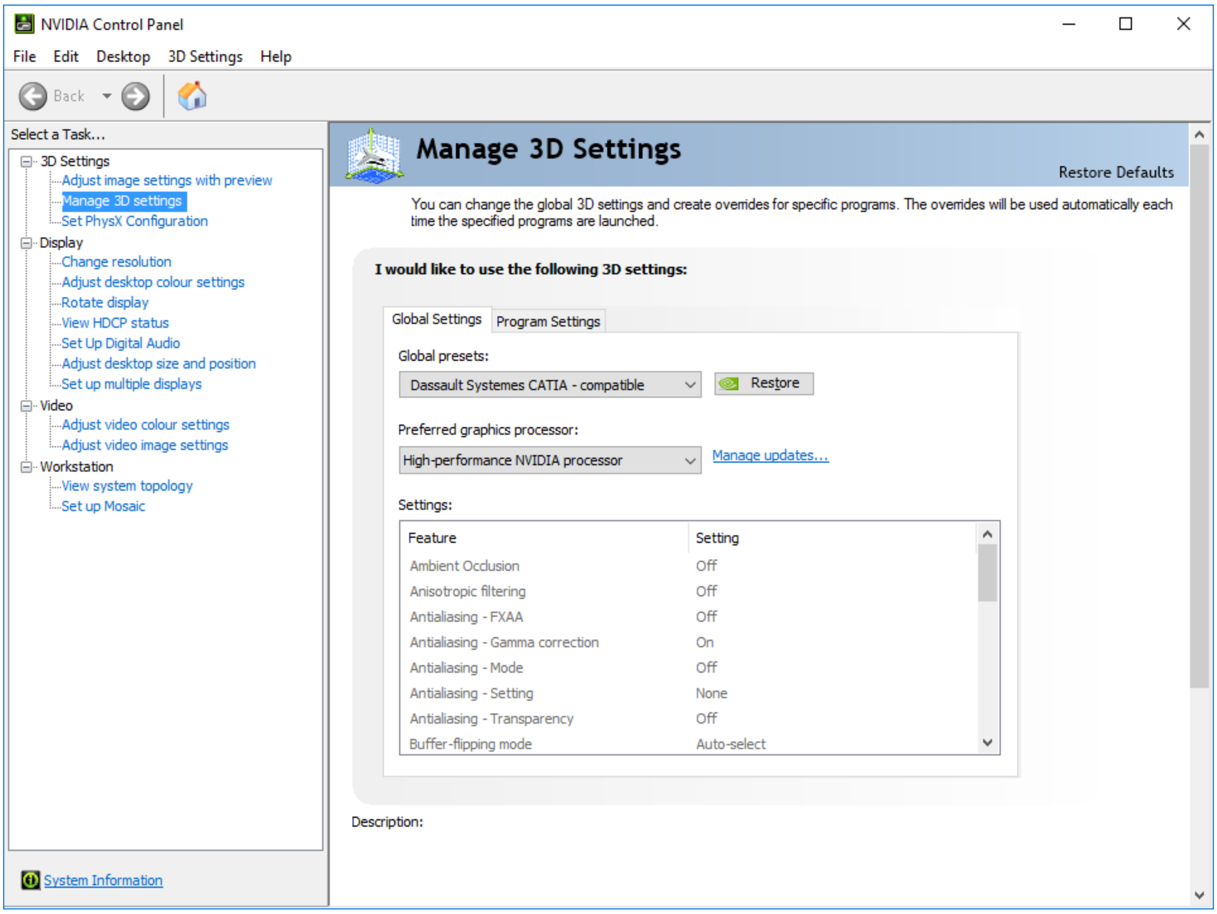
Viewing Options Machine

- When viewing machine axis, by default the view is shown vertical: If you have a Horizontal machine or a Mill/Turn machine (Vertical turret lathe) use this option to change the view.
- **Milling** — Select the type of milling machine from **Vertical** and **Horizontal**, to improve viewing during machine simulation and tool previews.
- **Turning** — Select the type of turning machine from **Slant bed lathe** and **Vertical turret lathe**, to improve viewing during machine simulation and tool previews.
- **Swiss turn, main spindle vertical y axis** – This setting is used when using the FeatureCAM Swiss module.



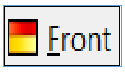
Display options

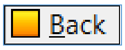
- For the best **Display Options** for your computer set the **NVidia ® Control Panel** options to the following.

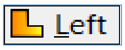


*All of the viewing options are available via **Right mouse button**. Click in the Main Graphics Area*

Principle views

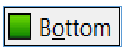
- 

Changes to a view of the part from the front with no other surfaces visible.
- 

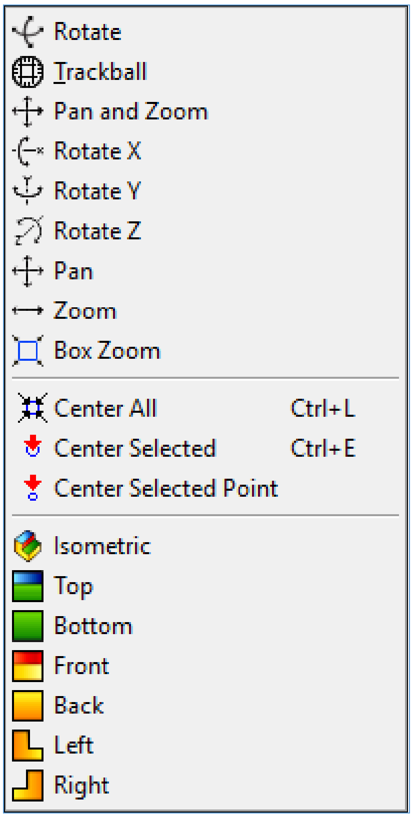
Changes to a view of the part from the back with no other surfaces visible.
- 

Changes to a view of the part from the left side with no other surfaces visible.
- 

Changes to a view of the part from the Right side with no other surfaces visible.
- 

Changes to a view of the part from the top only. Useful for drawing geometry, but harder to see the wireframe model of the part.
- 

Changes the view to the bottom of the part. No sides of the part are visible from this perspective.



Right Click in the graphics area to show this menu.

- All the viewing modes are available, plus an option called **Mode from Last Session.** This option keeps track of the viewing mode last set before **FeatureCAM** was closed, and uses that viewing mode the next time **FeatureCAM** starts up. For example, if the option is set to **Mode from Last Session**, and you had set the viewing mode to **rotate** before you closed **FeatureCAM, Rotate** is the viewing mode that is set in the toolbar the next time you open **FeatureCAM**.
- **Isometric** - changes the view to a three quarter view of the part showing the top and two sides with the current UCS near the bottom of the viewing area.
- **Perspective** - If selected, the view is a perspective view. If deselected, the view is an orthographic view.
- **As on setup** - If selected, the view is relative to the current setup. If deselected, the view is relative to the world coordinate system.
- **As on UCS** - changes your view to that of the current **UCS**.
- **As on world** - changes your view to that of the current **UCS**.

Keyboard shortcuts

- You can use the keyboard to navigate around the menus. To do this, press and hold the **Alt** key, and then press the letters on your keyboard that correspond to the letters underlined on the **Menu** bar and in the menu options. For example, to rotate your part, press and hold the **Alt** key, press **V** for **View** menu, press **V** for Viewing Modes and press **R** for **Rotate.**

Key sequence	Action
Alt+click-and-drag on any toolbar button	Move button around in toolbar.
Alt+**1** (**2**,**3**,**4**)	User View 1 (2,3,4)
Alt+**Enter**	Properties
Alt+**F1**	Centreline simulation.
Alt+**F2**	Play/pause simulation.
Alt+**F3**	Single-step simulation.
Alt+**L**	Last view.
Alt+**R**	Refresh
Alt+**Shift**+**V**	Save view.
Ctrl+click a column of any list box that does sorting	Activates the second level of sorting.
Ctrl+click a viewing mode button	Stay in that viewing mode.
Ctrl+click the **3D Simulation** button. Release key and click the **Play** button.	Run 3D simulation in hidden line mode and do continuous looping.
Ctrl+click **Fast Forward to End**	Stops the simulation when it encounters the next rapid (in addition to the next operation).
Ctrl+click the **Machine Simulation** button. Release key and click the **Play** button.	Run machine simulation in hidden line mode and do continuous looping.
Ctrl+**A**	Select all.
Ctrl+**C**	Copy
Ctrl+**E**	Centre selected.
Ctrl+**F**	Find (text in a text window). The cursor must be in a text window.
Ctrl+**H**	Replace (text in a text window). The cursor must be in a text window.
Ctrl+**L**	Centre all.
Ctrl+**N**	New file
Ctrl+**O**	Open file
Ctrl+**P**	Print
Ctrl+**R**	Opens the **New Feature** wizard.
Ctrl+**S**	Save file
Ctrl+**V**	Paste
Ctrl+**X**	Cut
Ctrl+**Y**	Redo
Ctrl+**Z**	Undo
Ctrl+click the **Pick Curve** button	Causes the warp status for that dialog to toggle.
Ctrl+click the **Pick Curve** button while a **Key sequence**	Toggles between simulation of the selection feature and the entire setup.
Ctrl+click the **Play** button	Run simulation in hidden line mode.
Hold **Ctrl** while toolpath is being computed	Temporarily turns off Toolpath Computation Minimization. Toolpaths will be recomputed for all features even if that feature did not change.

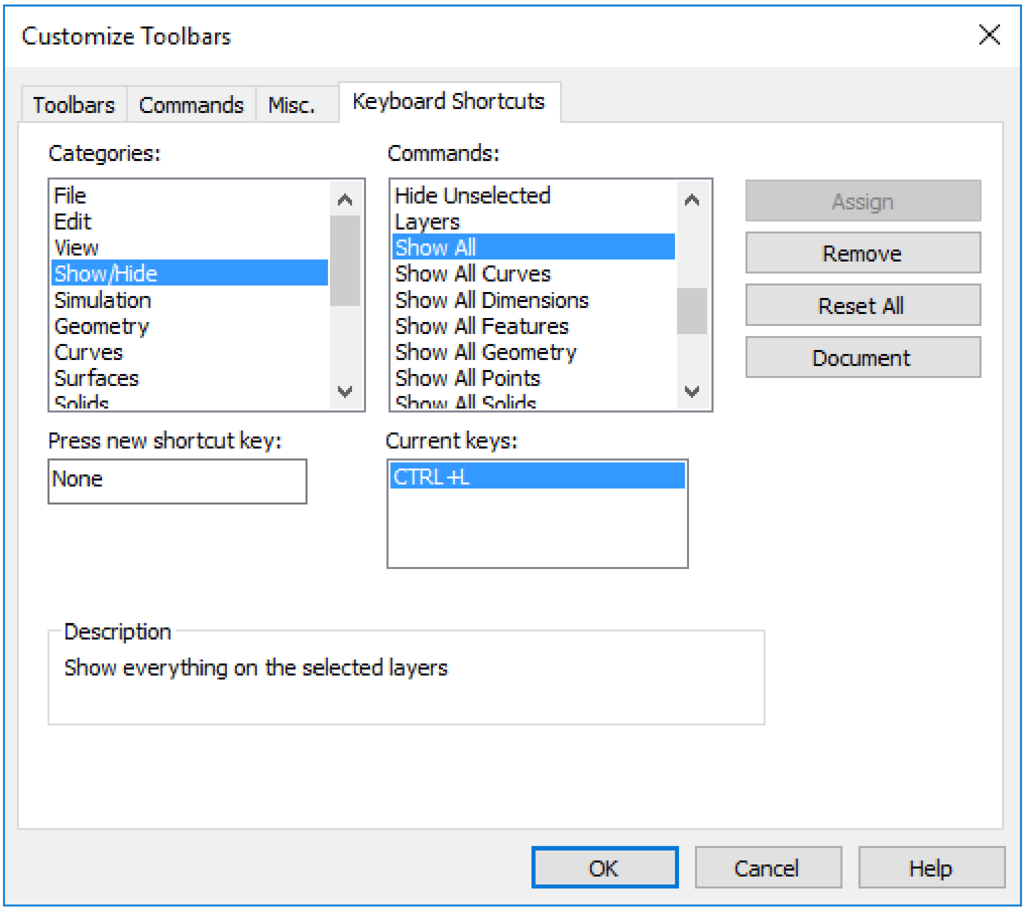
Ctrl+**Alt**+click-and-drag on any toolbar button	Duplicate and move button around in toolbar.
Ctrl+**Shift**+click the **3D Simulation** button. Release keys and click the **Play** button.	Continuous loop simulation for 3D simulation.
Ctrl+**Shift**+click the **Machine Simulation** button. Release keys and click the **Play** button.	Continuous loop simulation for machine simulation.
Ctrl+**Shift**+**A**	Shade selected.
Ctrl+**Shift**+**C**	Unshade all.
Ctrl+**Shift**+**N**	Unshade selected.
Ctrl+**Shift**+**P**	Toggles perspective.
Delete	Delete the selected object
Double-click a simulation button	Activate that simulation without clicking the play button.
Esc	Stop current simulation.
F1	Context-sensitive help.
Middle-click-and-drag in the graphics window	Performs viewing based on the current viewing mode. The mouse wheel-click-and-drag also performs this function.
Mouse scroll wheel	Zoom
Shift+click an edge when filleting	Selects all edges of the face and adds them to the list.
Shift+click the **NC Code** tab	Show ACL instead of NC code.
Shift+click while clipping when **Multiple Region** is **On**	Removes the entire picked region instead of to the nearest intersection.
Shift+right-click in graphics window	Dynamic viewing using current view mode.
Shift+click the **Show** button in the **SCL** dialog	Brings up a dialog that shows the attributes of the model selected in SCL dialog.

Commonly used Ctrl Keys

- **New Document - Ctrl + N**
- **New Feature Menu** or **Ctrl + R**
- **Isometric View** or **Ctrl + 1**
- **Top View** or **Ctrl + 5**
- **Refresh - Alt + R**

User interface: Keyboard shortcuts

- There is a new tab in the **Customize Toolbars** dialog, **Keyboard Shortcuts**:



*This tab lets you assign your own shortcut keys to the **FeatureCAM** menu commands. To set a new shortcut key:*

Select a category from the **Categories** list.

Select a command from the **Commands** list.

*If the command already has a shortcut, it is listed under **Current keys**.*

To assign a new key, select the **Press new shortcut key** field and press the keys that you want to use as the shortcut.

Press the keys on the keyboard, for example, press and hold down the **Alt** key and press the **N** key; do not type **ALT + N**.

A warning displays if the shortcut you entered is already assigned to a command.

Click the **Assign** button to save the new shortcut.

If you click **Assign** after receiving a warning that the shortcut is already assigned to a command, the shortcut is removed from the existing command and assigned to the new command.

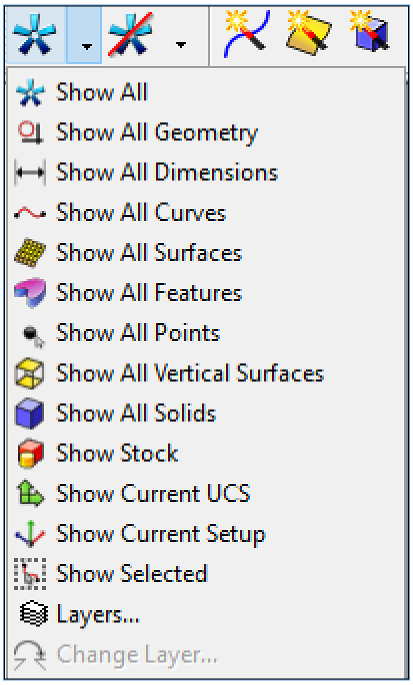
- **Remove** — Click this button to remove the shortcut assigned to the selected command.
- **Reset All** — Click this button to reset all shortcuts to the **FeatureCAM** defaults.

Blanking model entities

- Use the **Blank Selected** (**Ctrl + J**) option to temporarily hide all visible components.
- **(Ctrl K)** Hide Selected components
- To re-display all of the components. Right-click in the graphics area and select **Unblank (Ctrl L)**. The blanked model components are displayed. Please note this option will only work if you have set this up as explained in the previous chapter.

Show menu

- Show  functions help control what is displayed. This is useful as you place and model intricate features in a complex part.
- You access the Show functions by selecting **View >** Show from the menu or using the Show **Menu** button on the **Advanced** toolbar:

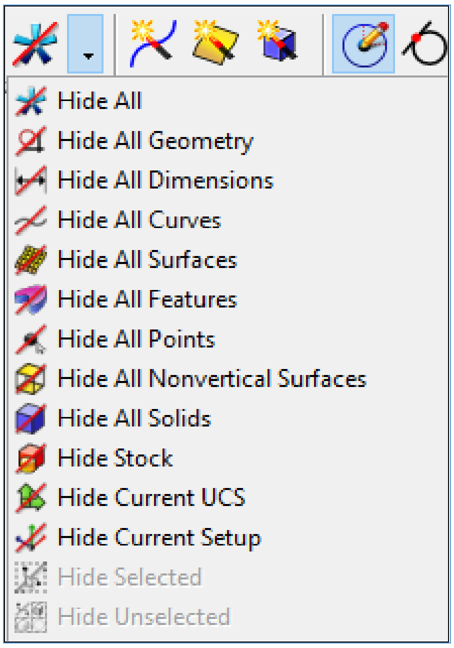


- Show **All** — Use this menu option to Show everything in the part model.
- Show **All Geometry** — Use this menu option to Show all geometry (points, lines, arcs, and circles).
- Show **All Dimensions** — Use this menu option to Show all dimension information added with the Dimensions Tool.
- Show **All Curves** — Use this menu option to Show all Curves.
- Show **All Surfaces** — Use this menu option to Show all surfaces (available only in **FeatureCAM 3D**).

- Show **All Features** — Use this menu option to Show all features.
- Show **All Solids** — Use this menu option to Show all solids
- Show **All Vertical Surfaces** — Use this menu option to Show the vertical surfaces in the model. This is useful for identifying surfaces that are part of 2.5D features like holes or pockets contained in a surface or solid model.
- Show **Stock** — Use this menu option to Show the stock outline.
- Show **Current UCS** — Use this menu option to Show the current UCS icon.
- Show **Current Setup** — Use this menu option to Show only the features and drawing elements that are in the current Setup.
- Show **Selected** — Use this menu option to Show only the selected elements

Hide Menu

- Hide controls what is displayed at any given time. This is useful as you place and model intricate features in a complex part. Besides the display factors, you can't snap, select or build curves from hidden entities. The Hide functions are not exclusive. You can click different buttons sequentially, hiding different entities until only the ones you want are still in view.
- You access the Hide functions by selecting **View >** Hide from the menu or using the Hide **Menu** button on the **Advanced** toolbar:



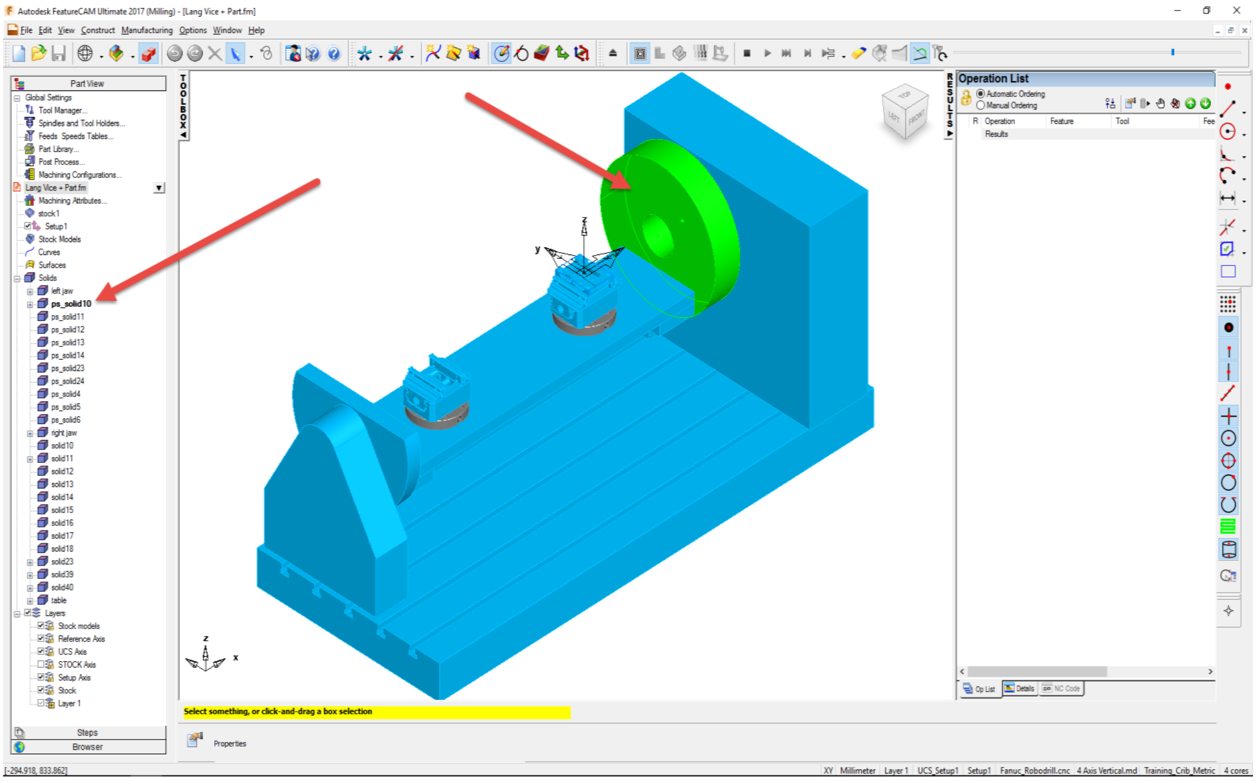
- Hide **All** — Use this menu option to hide all geometry, curves, Features. The stock and axis icon remain visible. A common procedure is to Hide **All**, then **Show** only one type of entity, for example features.
- Hide **All Geometry** — Use this menu option to hide all geometry. Other entities remain visible.
- Hide **All Dimensions** — Use this menu option to hide all dimension information added with the FeatureCAM Dimension tools.
- Hide **All Curves** — Use this menu option to hide all curves. Other entities remain visible.

- Hide **All Surfaces** — Use this menu option to hide all surfaces in the part model. Only available in the 3D version.
- Hide **All Features** — Use this menu option to hide all features.
- Hide **All Points** — Use this menu option to hide all points.
- Hide **All Non-Vertical Surfaces** — Use this menu option to hide all surfaces that are not vertical relative to the current setup. This is helpful if you want to isolate the surfaces that are part of **2.5D** features in a surface or solid model.
- Hide **All Solids** — Use this menu option to hide all of the solid models.
- Hide **Stock** — Use this menu option to hide the stock outline. All other entities remain visible.
- Hide **Current UCS** — Use this menu option to hide the current user coordinate system.
- Hide **Current Setup** — Use this menu option to hide the axis of the current Setup. All other entities remain visible.
- Hide **Selected** — Use this menu option to hide all selected entities. Non selected entities are still visible.
- Hide **Unselected** — Use this menu option to hide all entities other than the selected ones.

Highlighting objects from **Part View**

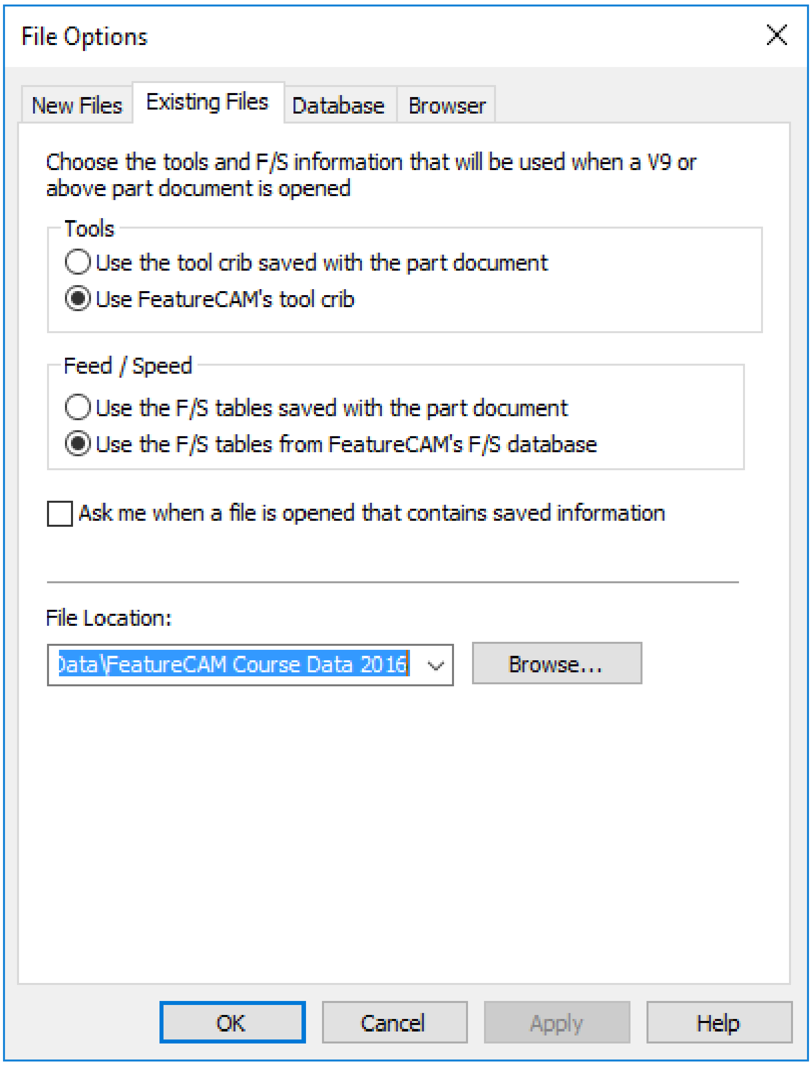
You can now highlight objects in the graphics window by moving the cursor over the object's name in the **Part View**. This enables you to find features quickly and improves the work-flow in complex documents.

For example, move the cursor over an object in the **Part View**:



Setting file location options

It is a good idea to set the file location for where you will save your files. Please select ***Options/File Options/Existing Files****.*

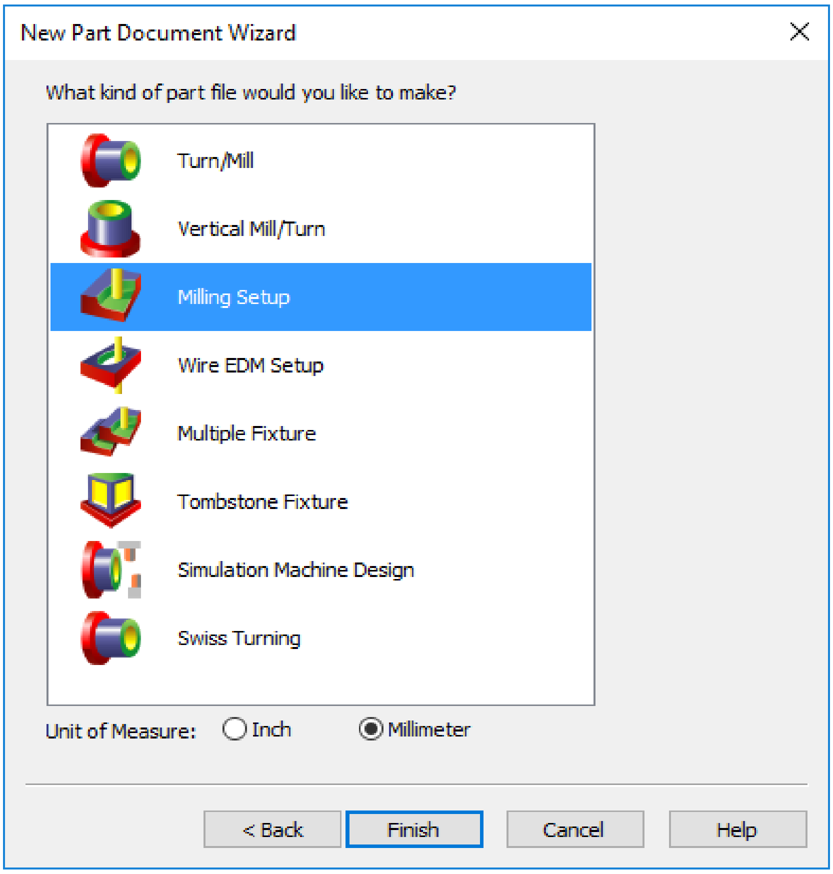


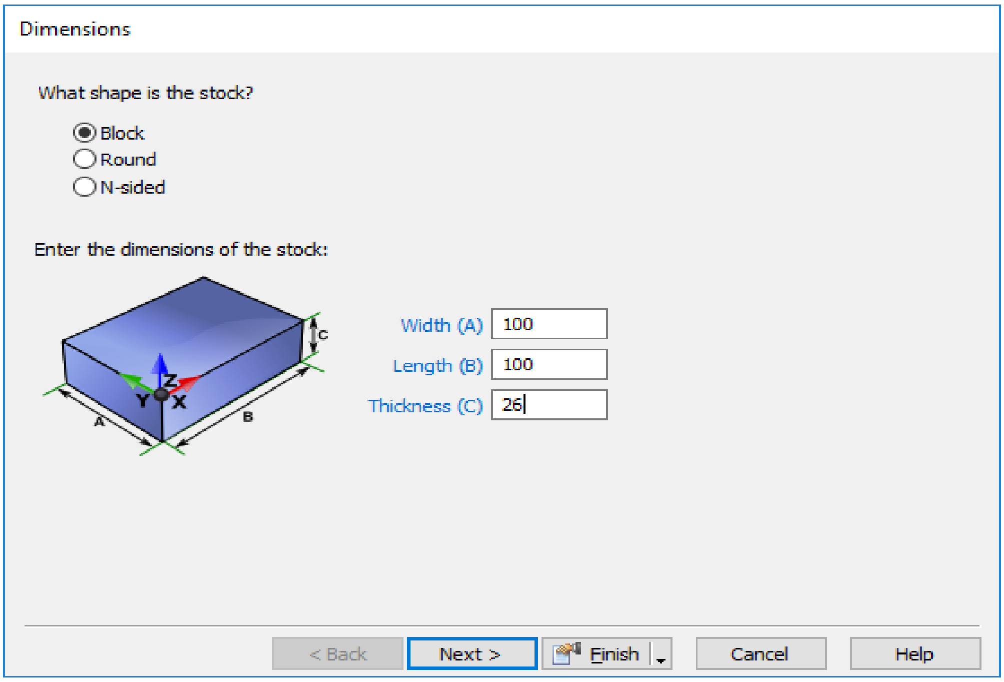
1. Select Browse. Then select the following location. **C:\Training_Data\FeatureCAM Course Data 2017**.
2. Select **Apply** and **Ok** to close the form. When we save or import files **FeatureCAM** will take us to this location.
3. Please save all files in **Coursework**.
4. Tool Crib = **BasicMetric** or **Training_Crib_Metric**
5. Post processor used = **Fanuc_Robodrill.cnc**

Stock dimensions

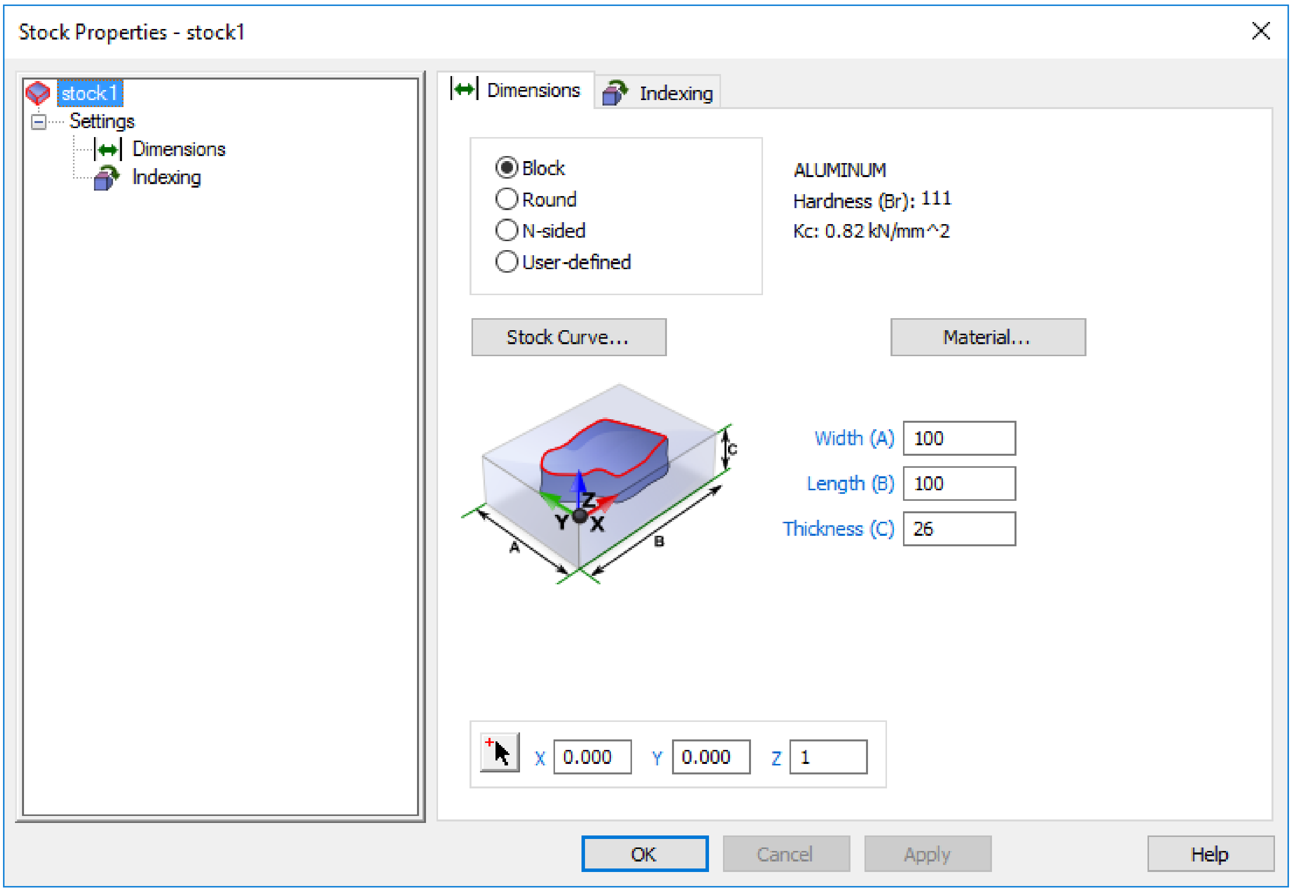
1 Open **FeatureCAM** by double clicking the **FeatureCAM** shortcut

2 Select **New File**. **Milling Setup**. Select **Finish**.

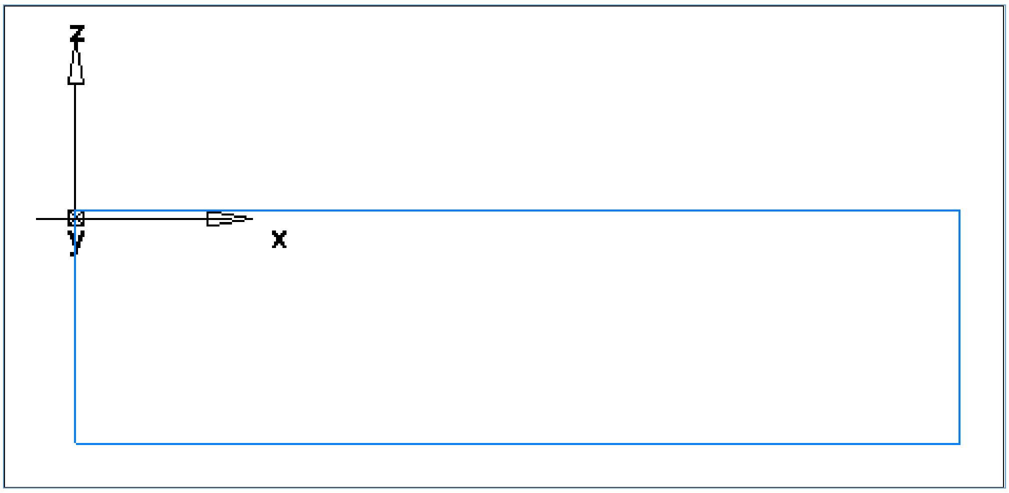




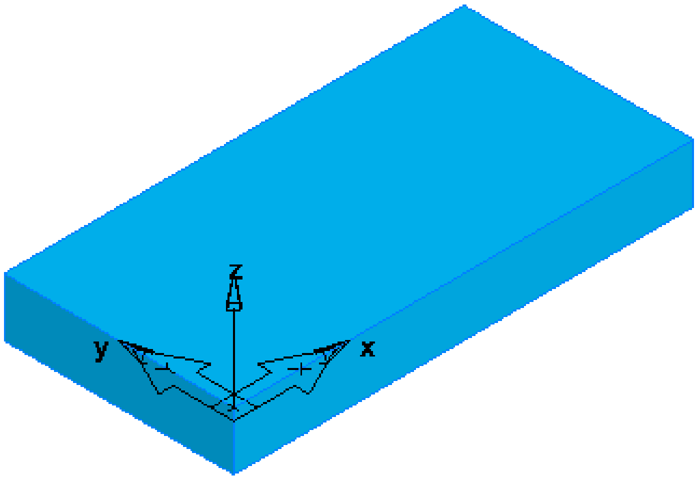
3 Enter the dimensions of the stock. Stock Size = **100mm x 100mm x 26mm thick** into the dimensions menu. Select **Finish** and the following menu will appear. Enter 1mm for Z to allow material above the datum. Select **Finish**.



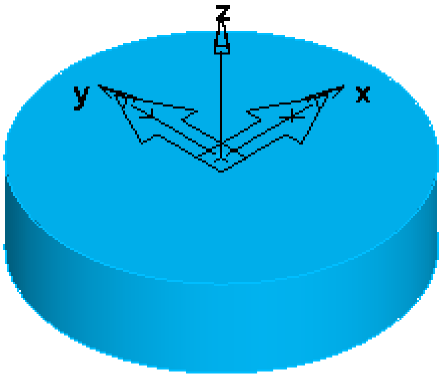
*Notice the **1mm** in the **Z** Menu. This adds 1mm above the Z Datum for machining off later when creating a Face Feature.*



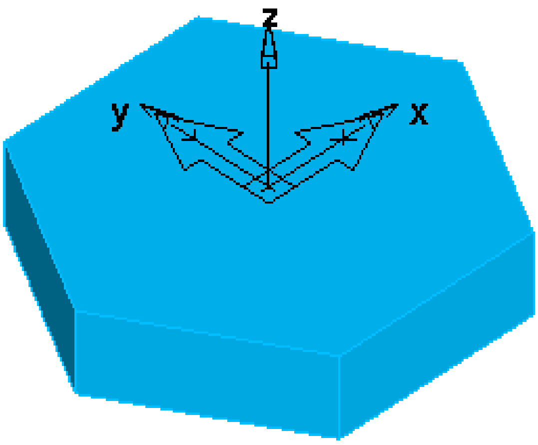
Stock Types available (Information Only)



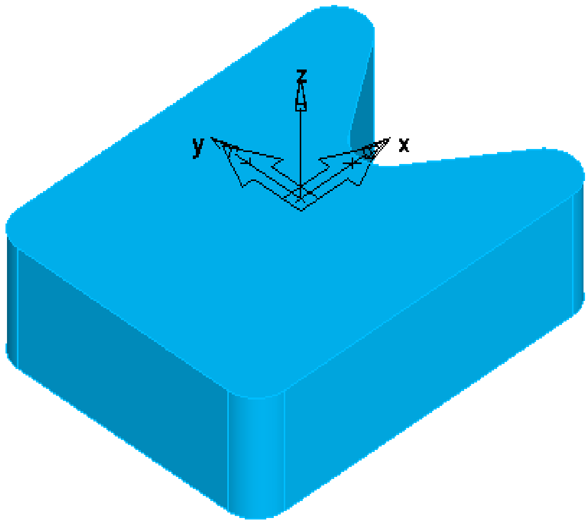
Block



Round

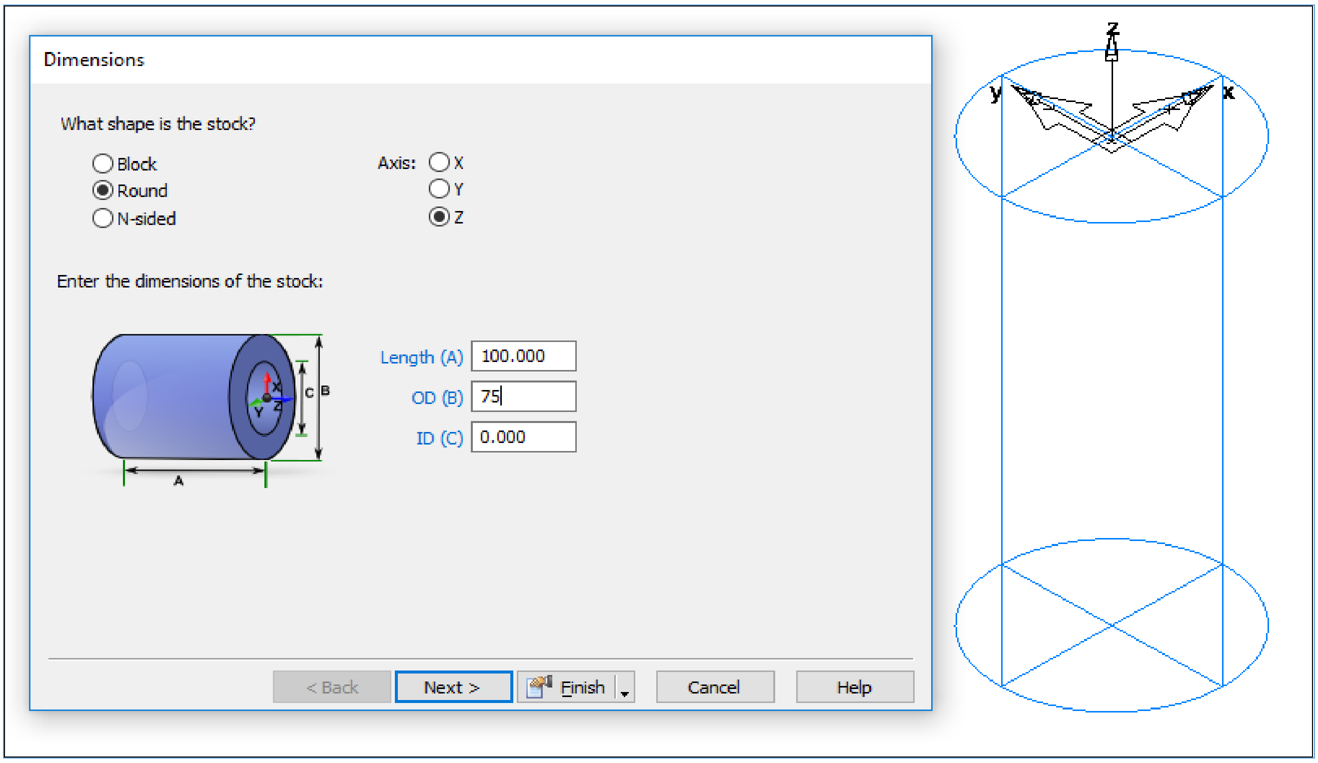


N-sided



User-defined

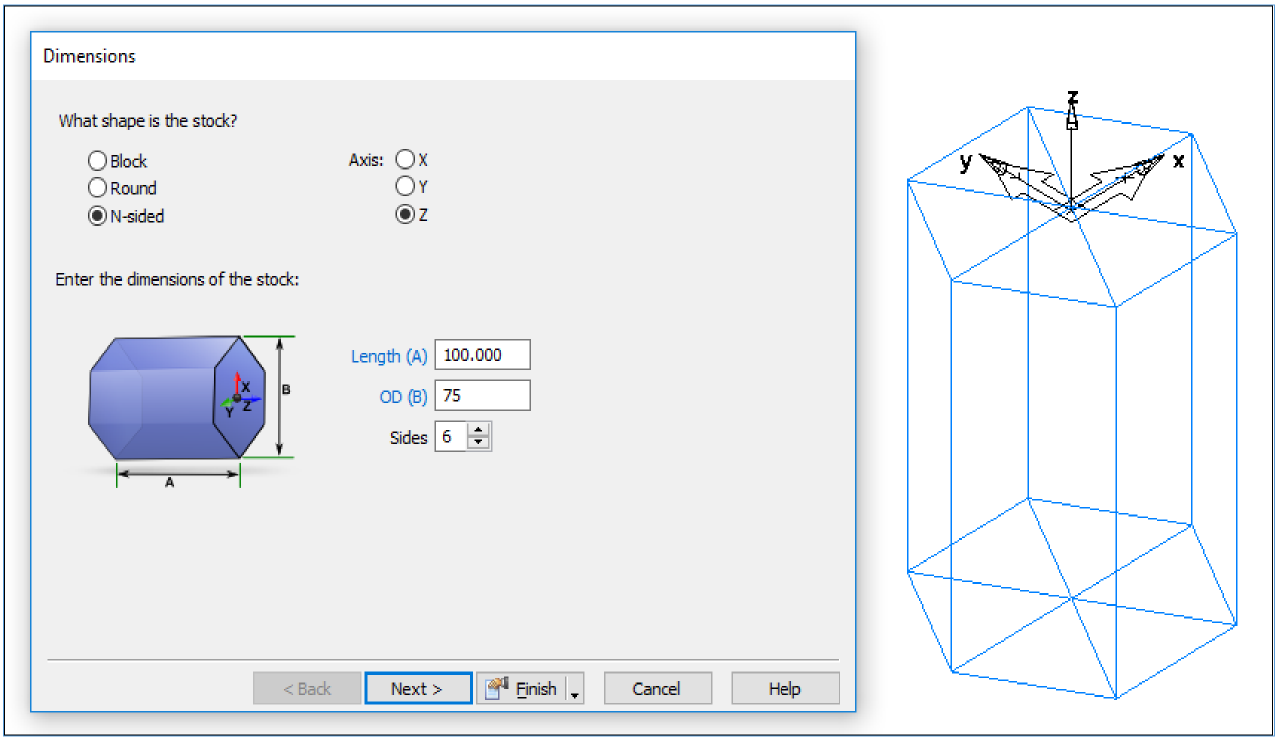
- Inside a **FeatureCAM** milling document there are several different **Stock types**. The stock can be defined as a block material, a round material or N-sided and user defined material. We have already gone through this procedure with our second exercise.dxf file and resized the material around the part.
- Selecting the **Round** Radio button you have an option to create **Round Stock** as shown below.
- Please note: - You can define this stock as a tube by typing in a value for the **ID** as well as the **OD.**



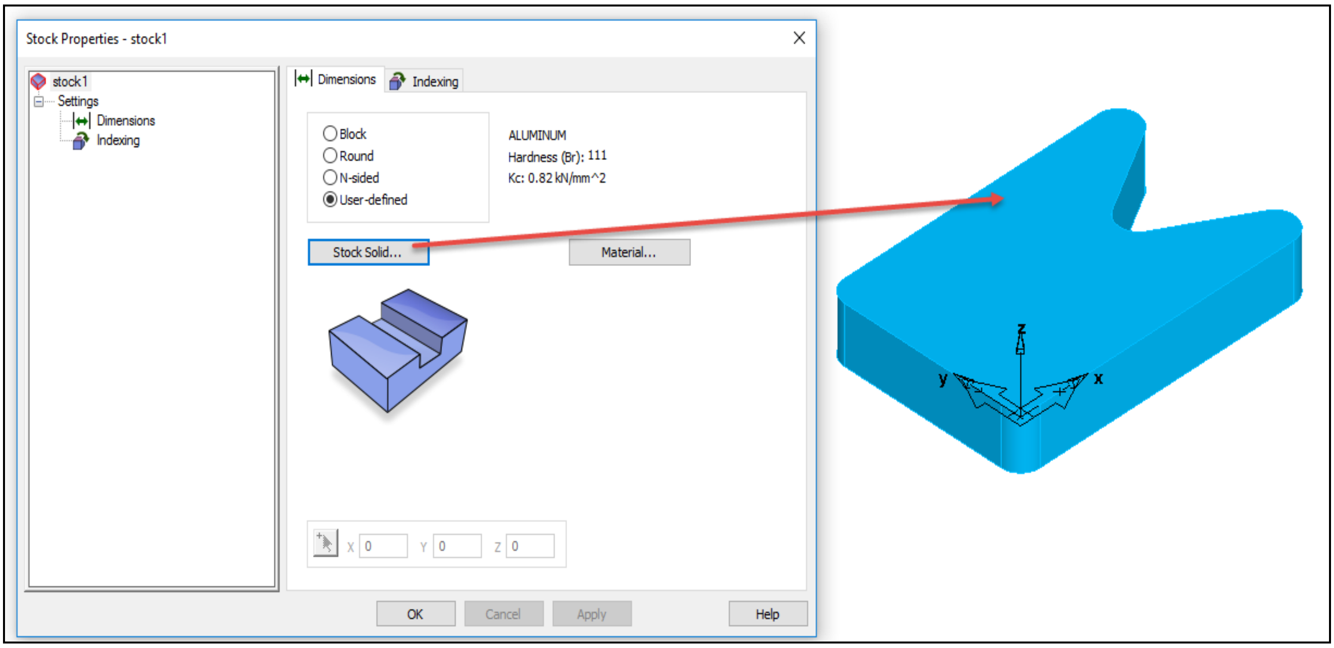
This option is ideal for work held in a **3** or **4 jaw chuck** on a **3** or **4 Axis** vertical machining Centre.

N-Sided

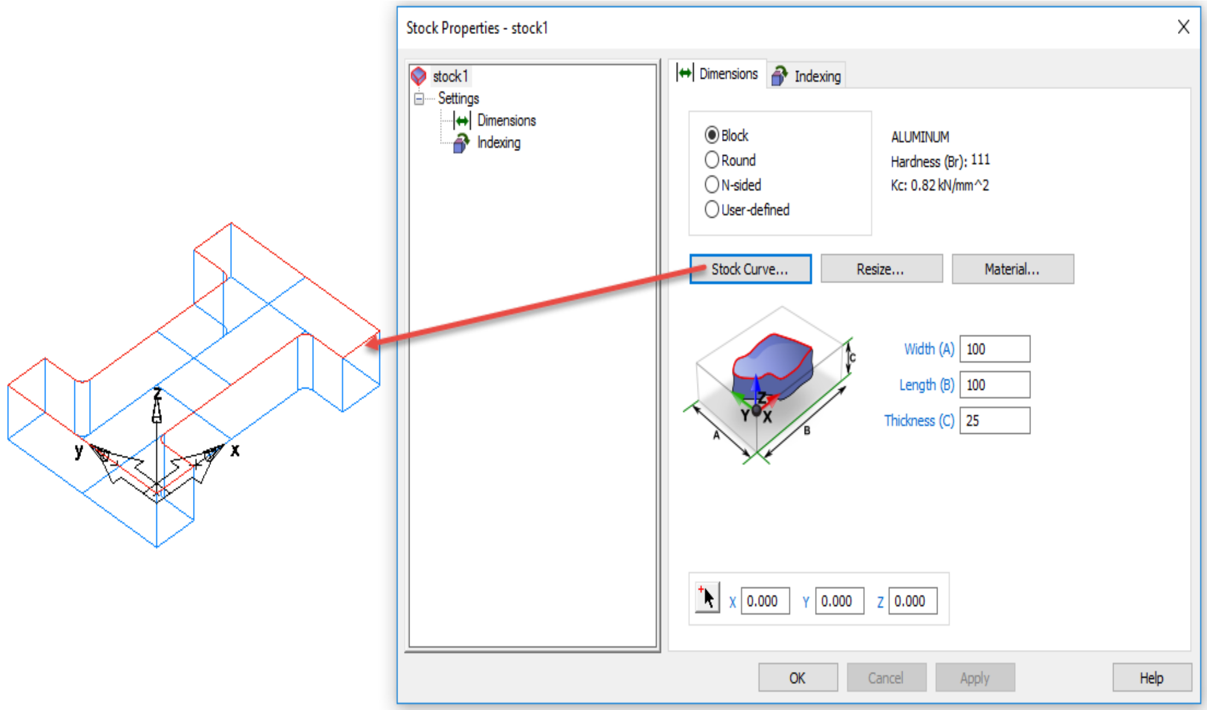
*Select the **N-Sided** Radio button. When this button is pressed the stock can be defined along a specific axis with a multiple number of sides. The dimensions shown are just an example only.*



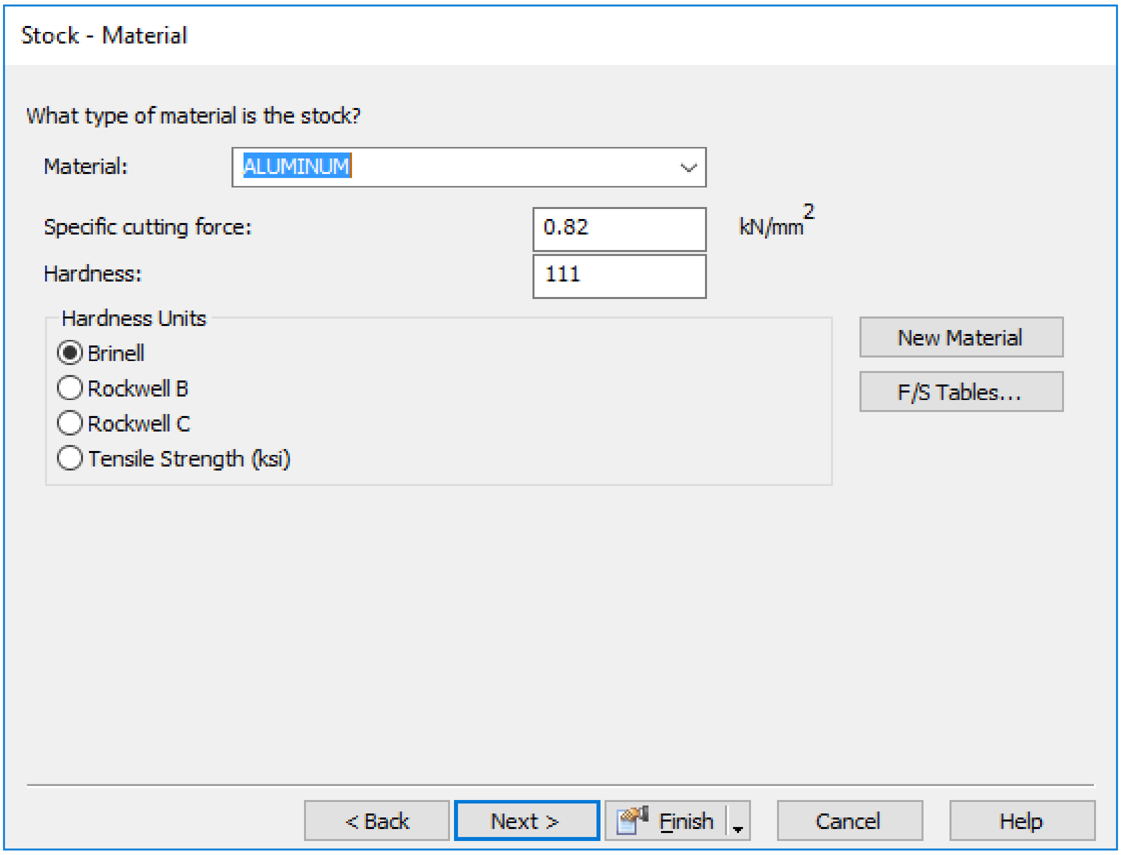
User Defined – Stock Solid



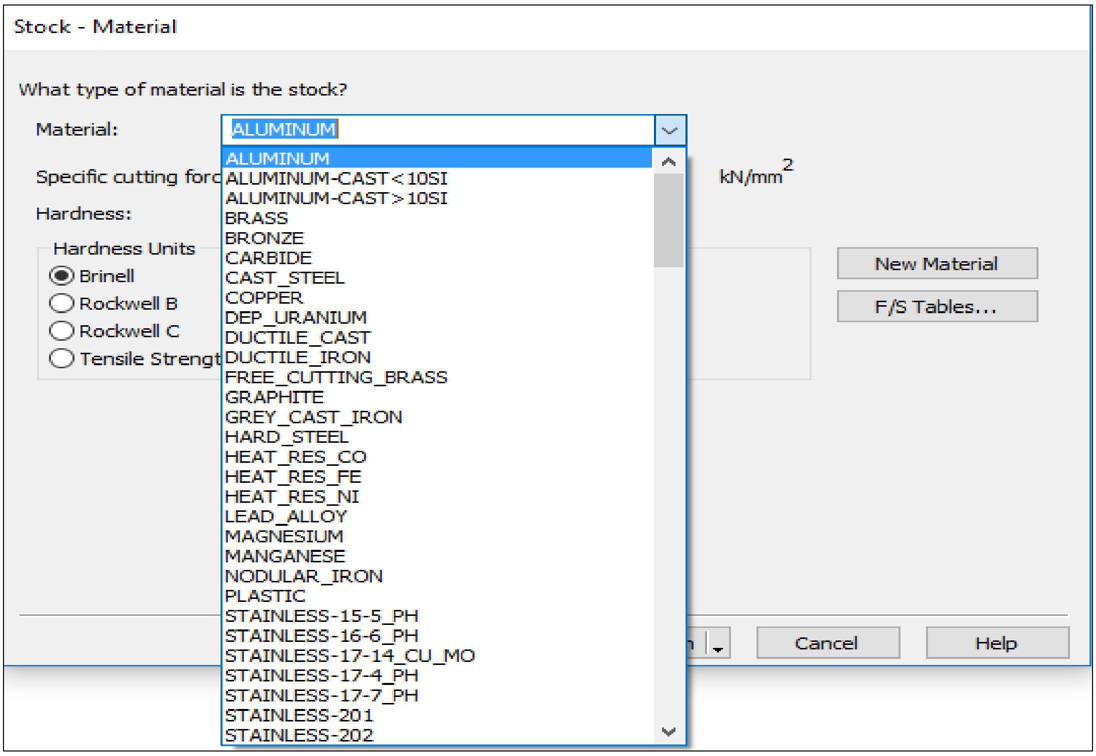
Stock Curve



Stock – Material

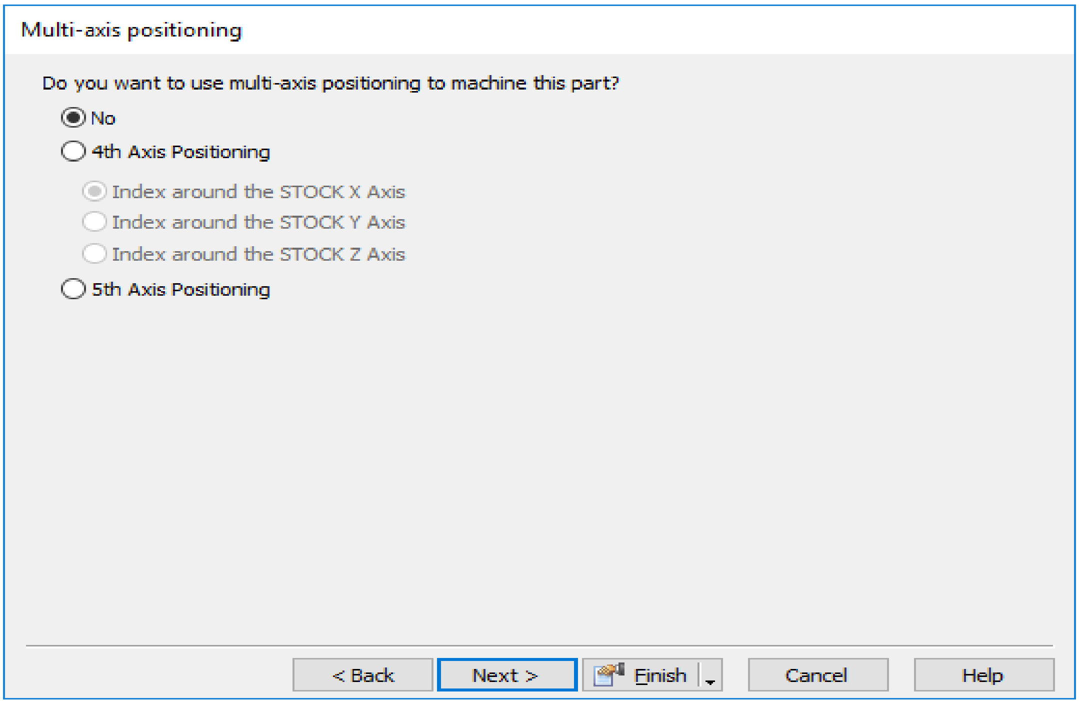


4 From the **Stock wizard** you can select the **Material**, **Specific Cutting Forces**, **Hardness** and **Hardness Units**. You can also access the existing **Feed** and **Speed tables** for the material or create a **new Material**.



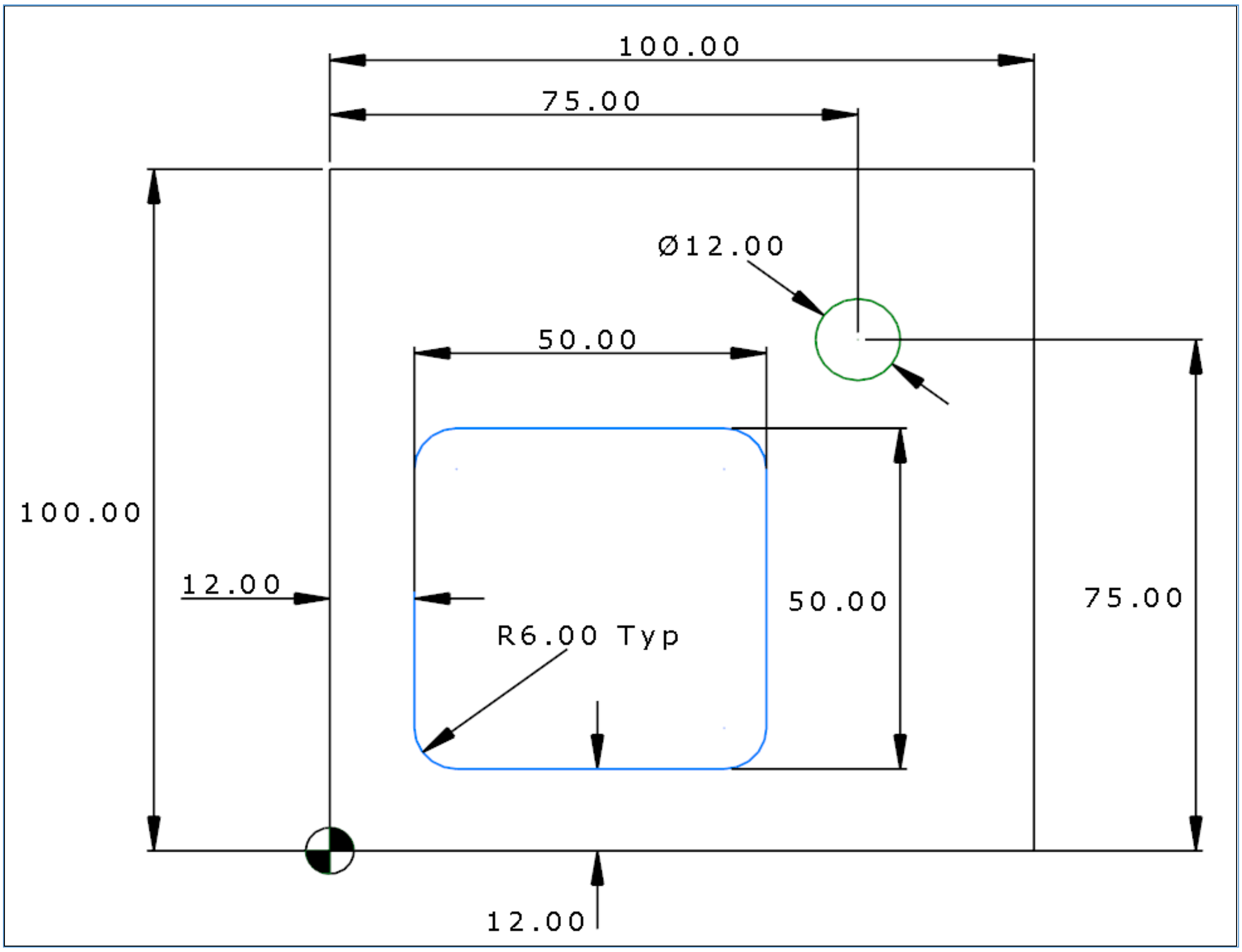
*On this page of the **Stock wizard** you can select the **Material**, **Specific Cutting Forces, Hardness and Hardness Units**. You can also access the existing **Feed and Speed tables** for the material or create a new Material*

*As shown by selecting **Next** the multi-axis positioning options will appear.*



In the next example we will only be using 3 axis. Select No to Multi-axis positioning to machine this part.

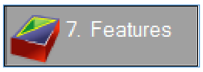
First class exercise (From Dimensions)



Additional information

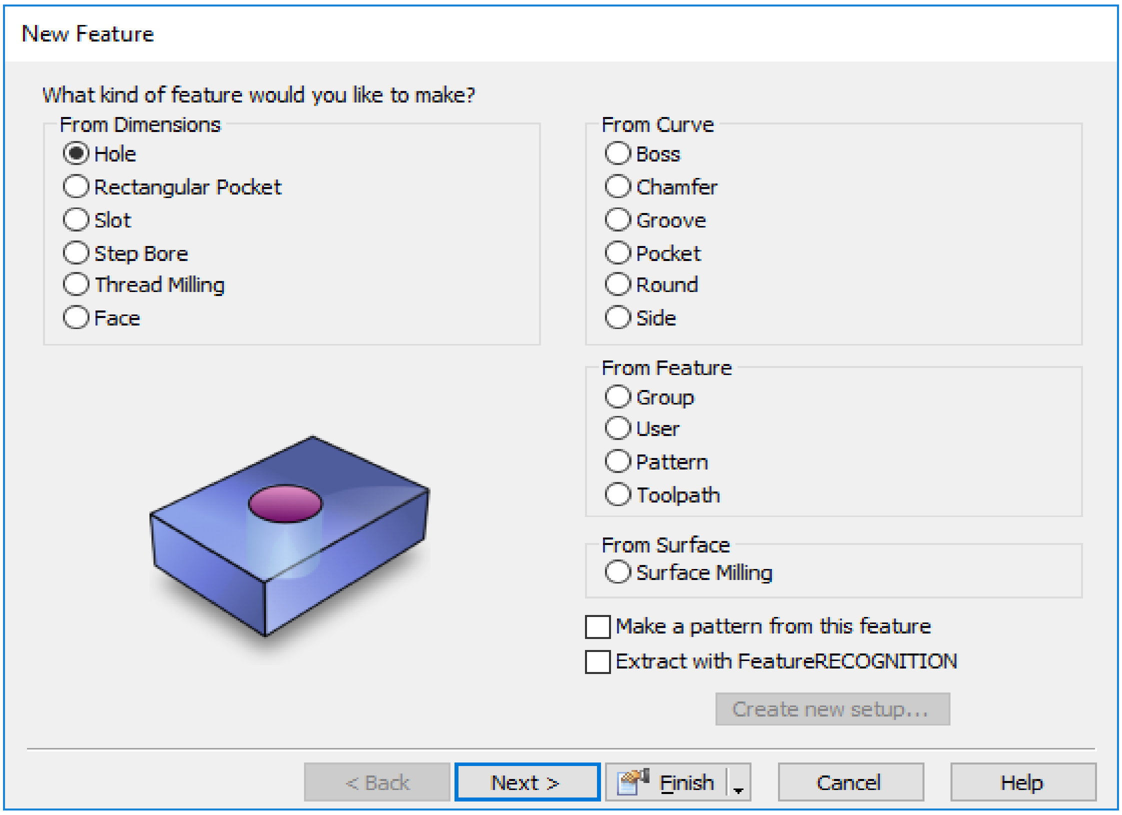
- Product Thickness **25mm**
- Pocket depth **12mm**
- Chamfer for Hole and Pocket features **0.5mm** to **1mm**

*There are four different ways to activate the **New Feature** menu.*

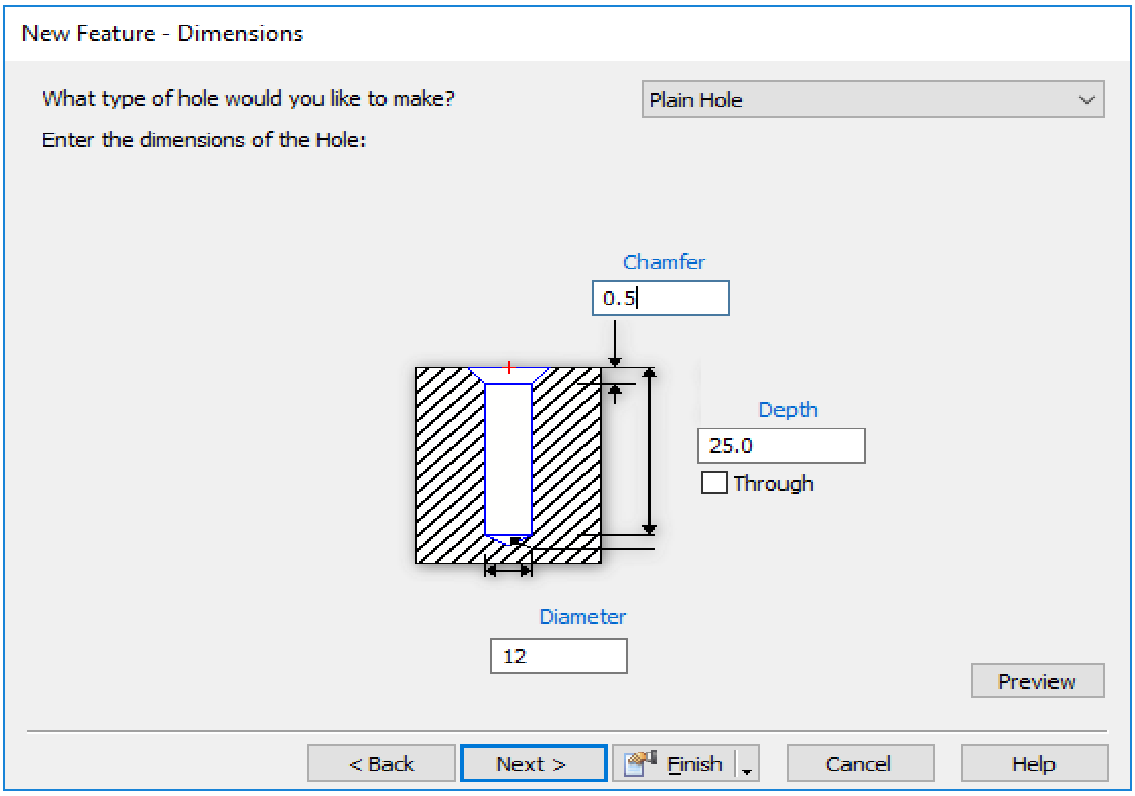
- Select **Features**  from the **Steps** menu.

- Select **Ctrl + R New Feature.**
- Select **Construct>New Feature.**
- Select **New Feature** from the **Advanced Toolbar.**

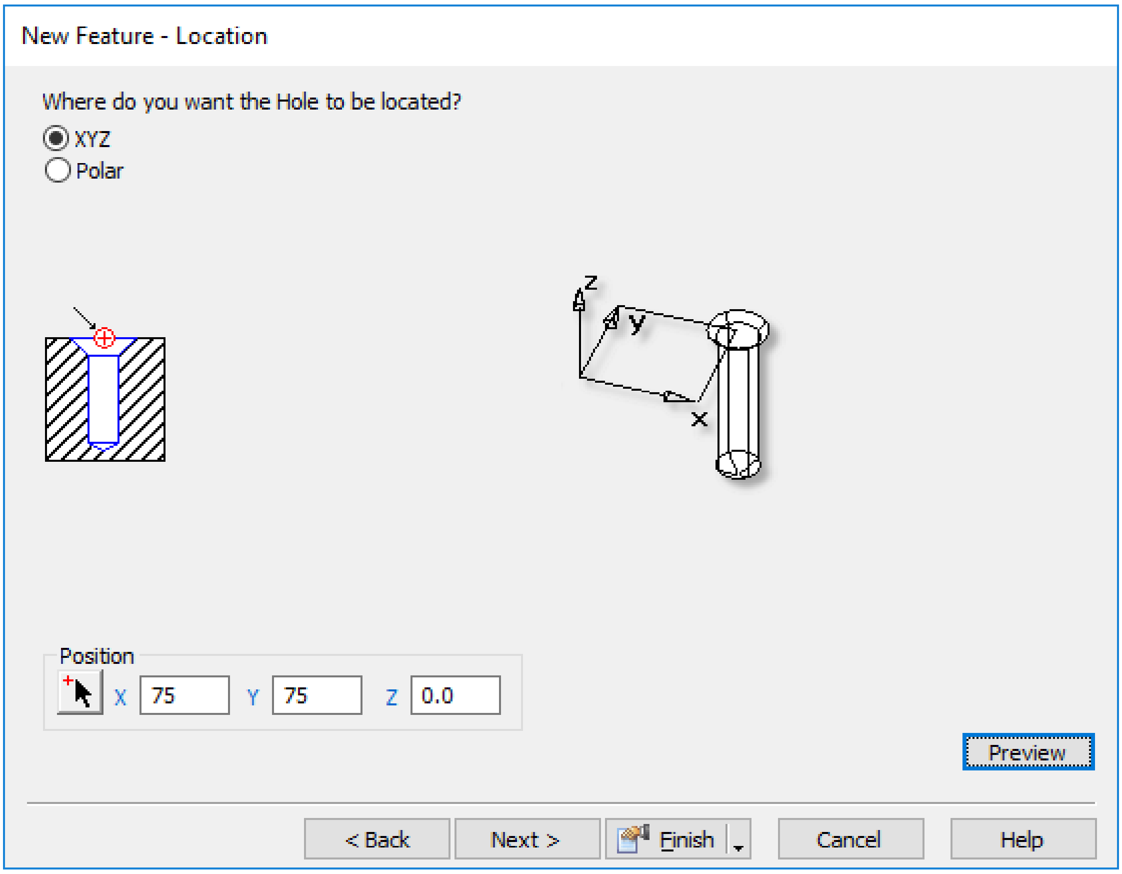
5 The following menu will appear. Select **Hole** and then **Next.**



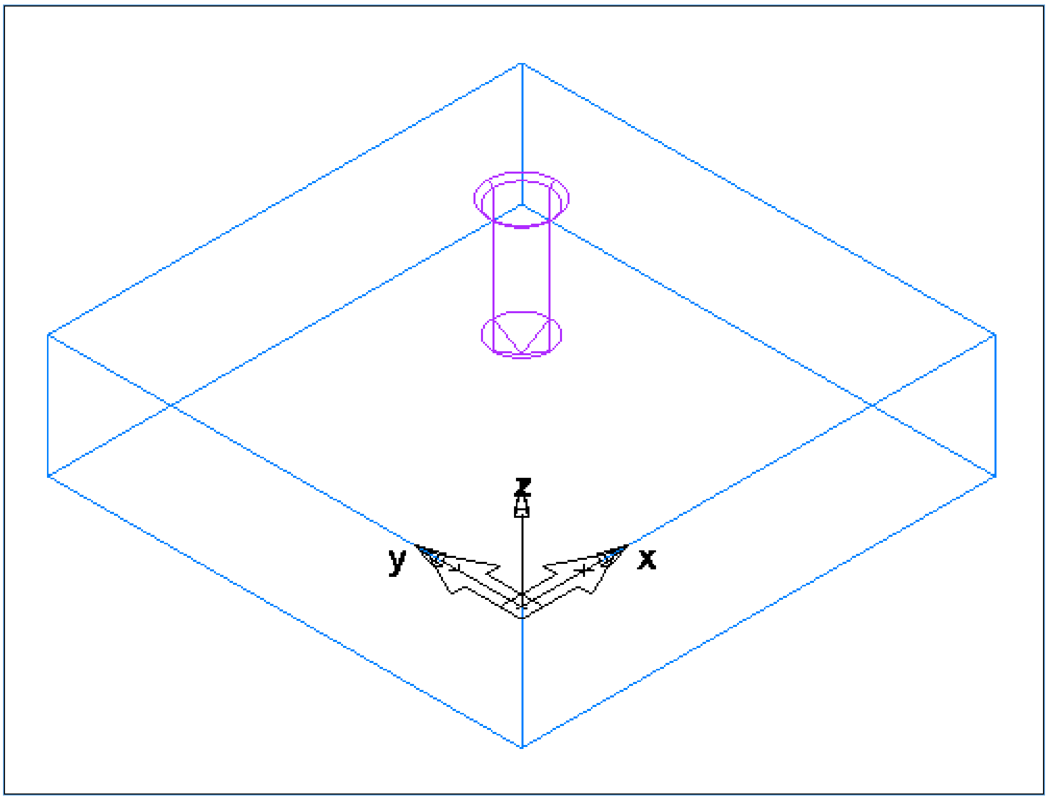
Enter the following information ***Chamfer 0.5mm Depth 25mm, Diameter 12mm****. Select* ***Next.***

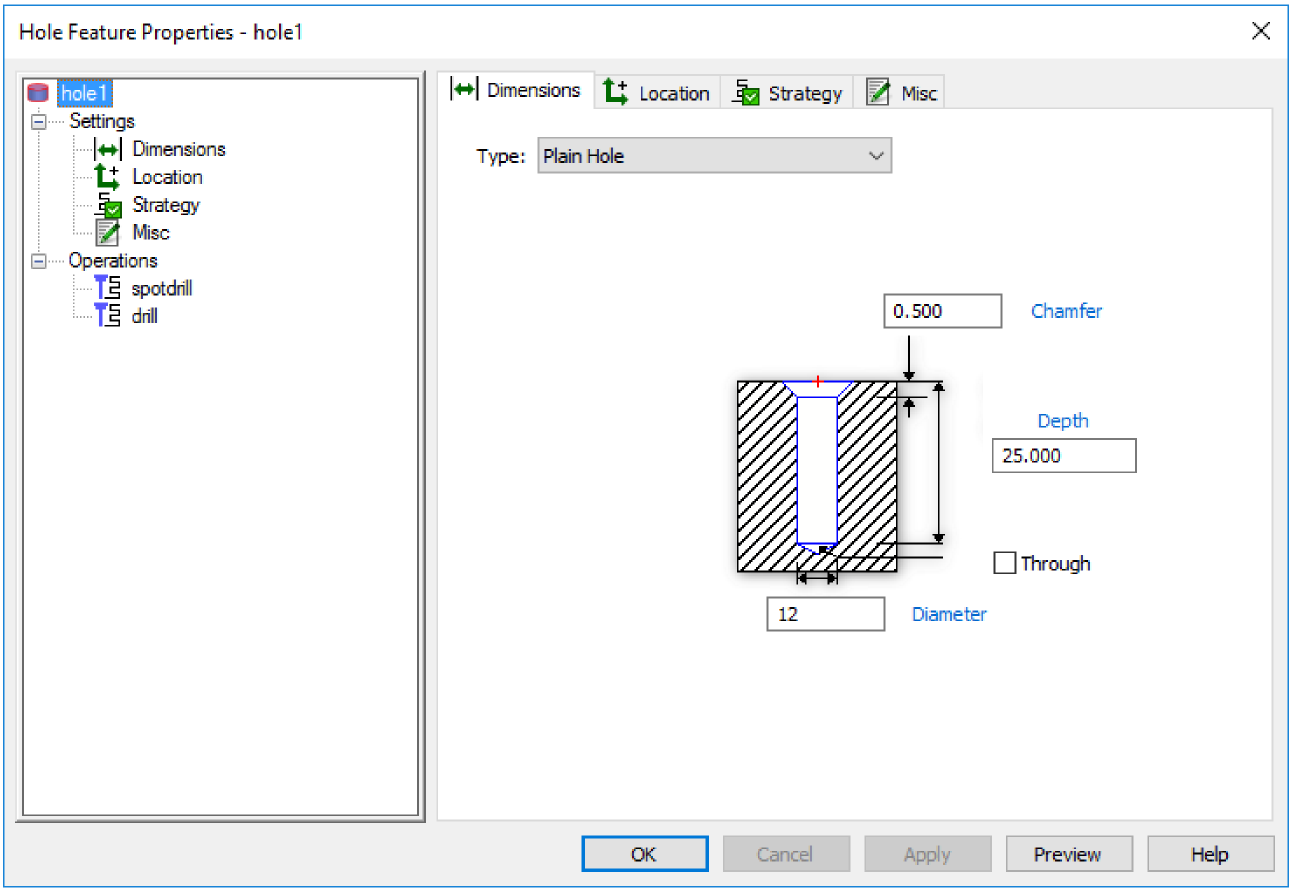


6 We now need to position the hole at **X75mm** and **Y75mm,** then Select **Finish** and then **OK** to close the form.

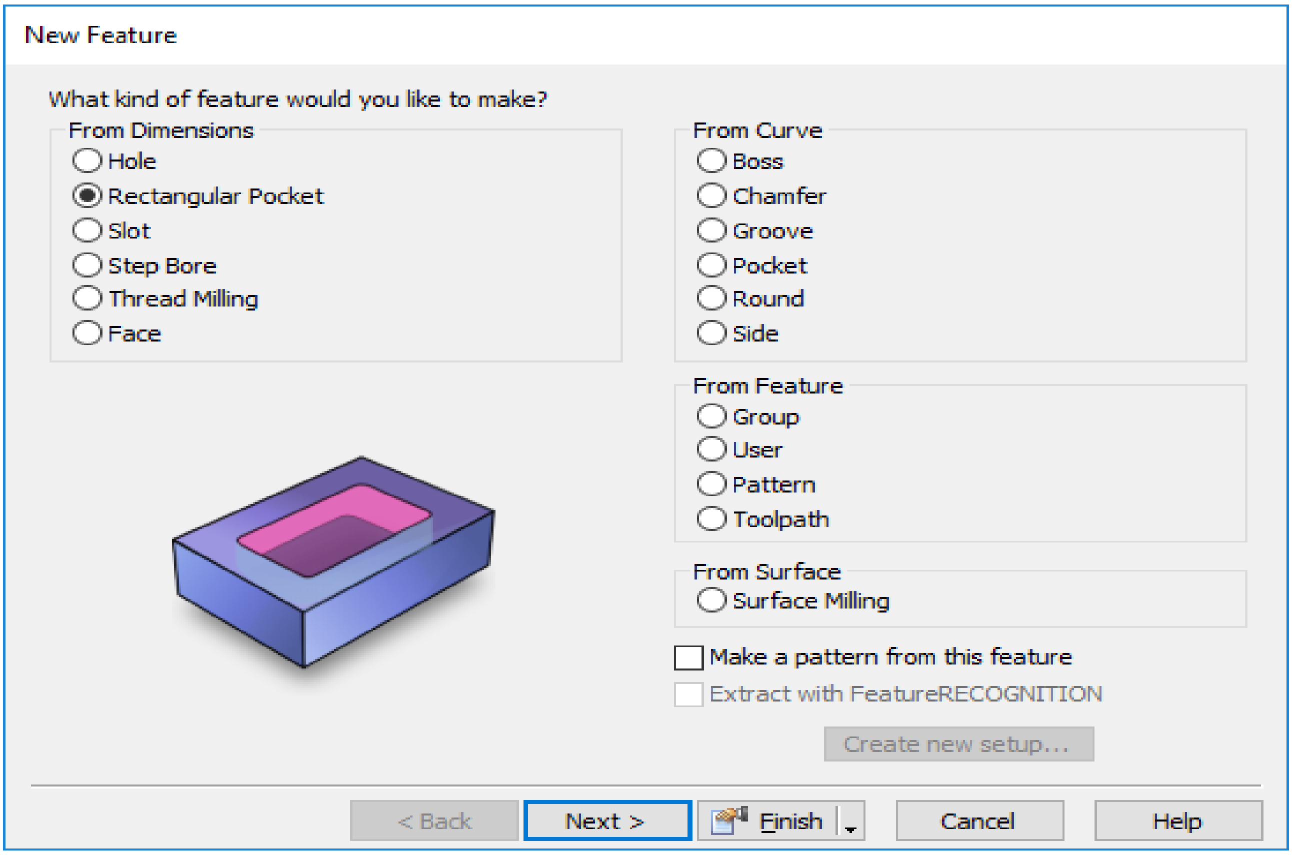


Hole positioned in stock.

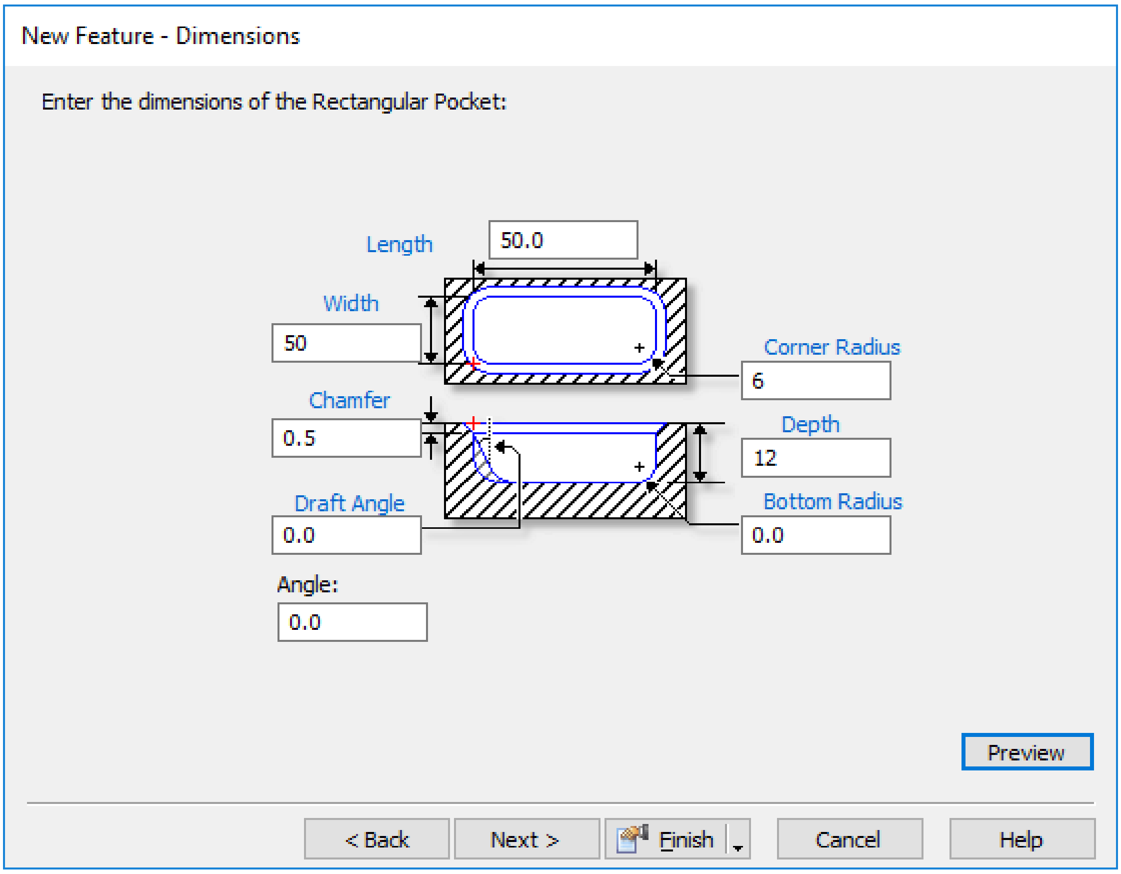


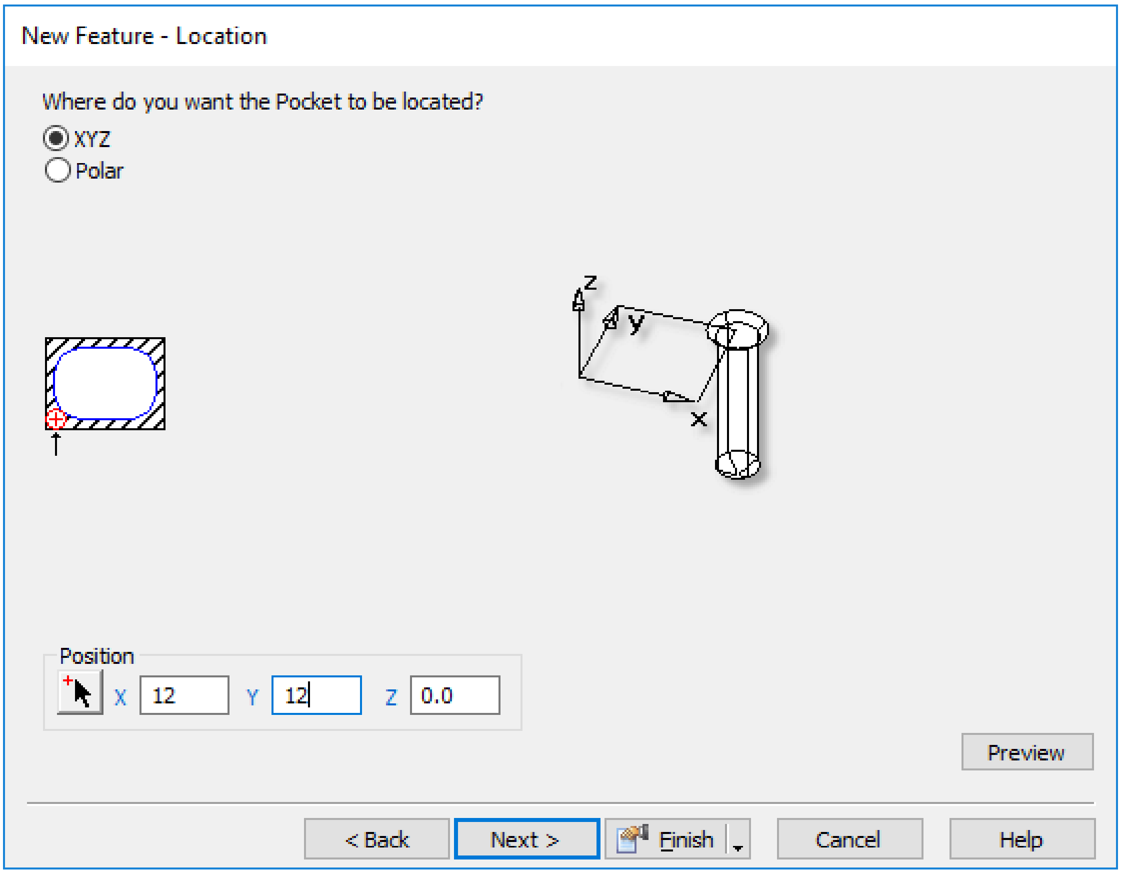


7 Select **Ctrl + R** or any of the 3 other ways to activate the **New Feature** Menu and Select **Rectangular Pocket** from **Dimensions,** then select **Next.**

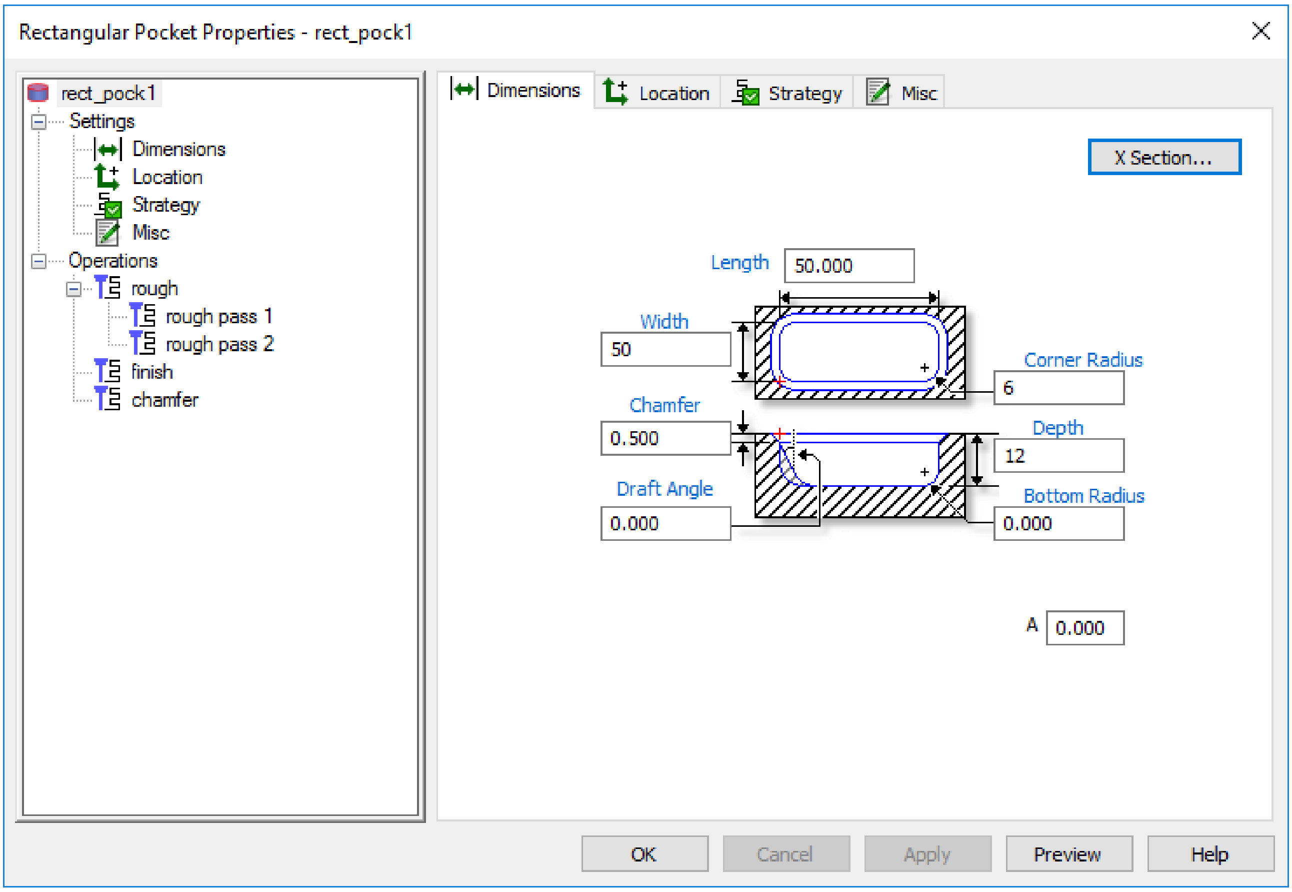


8 Enter Length **50mm,** Width **50mm,** corner radius **6mm,** Chamfer **1mm**, Depth **12mm**, select **Next**. Enter X**12mm** Y**12mm** for location**.**

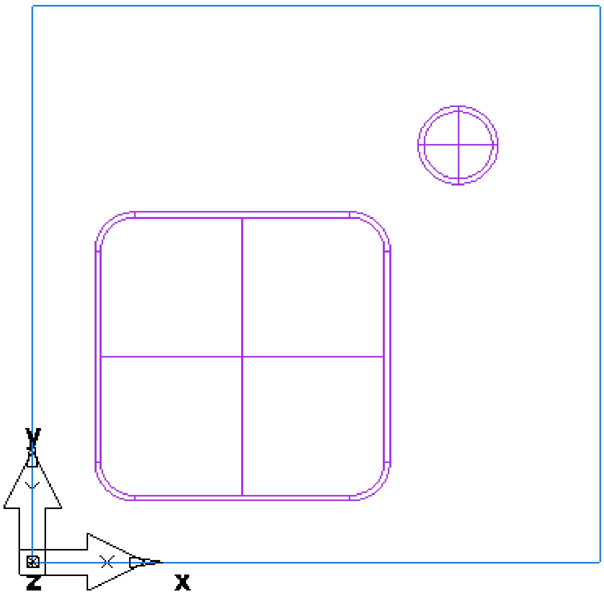




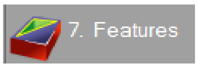
Rectangular Pocket Properties



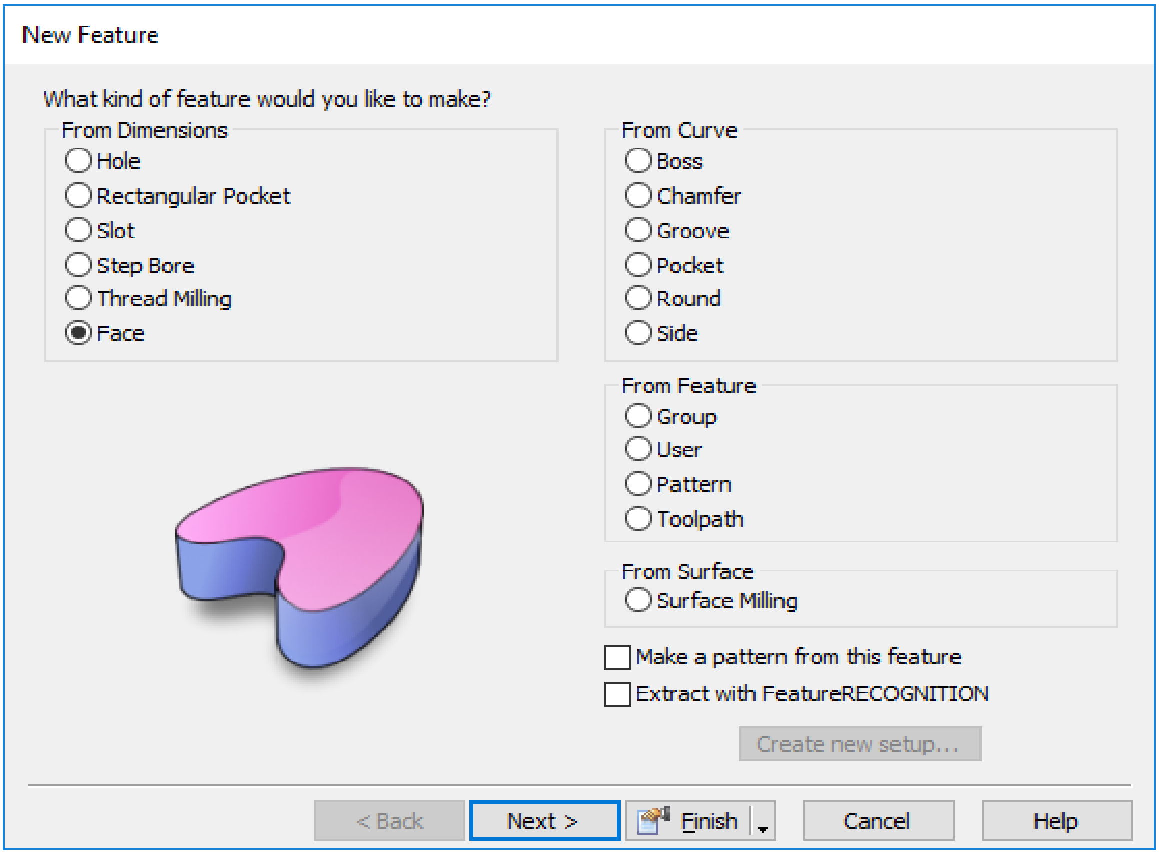
Rectangle from dimension in Position.



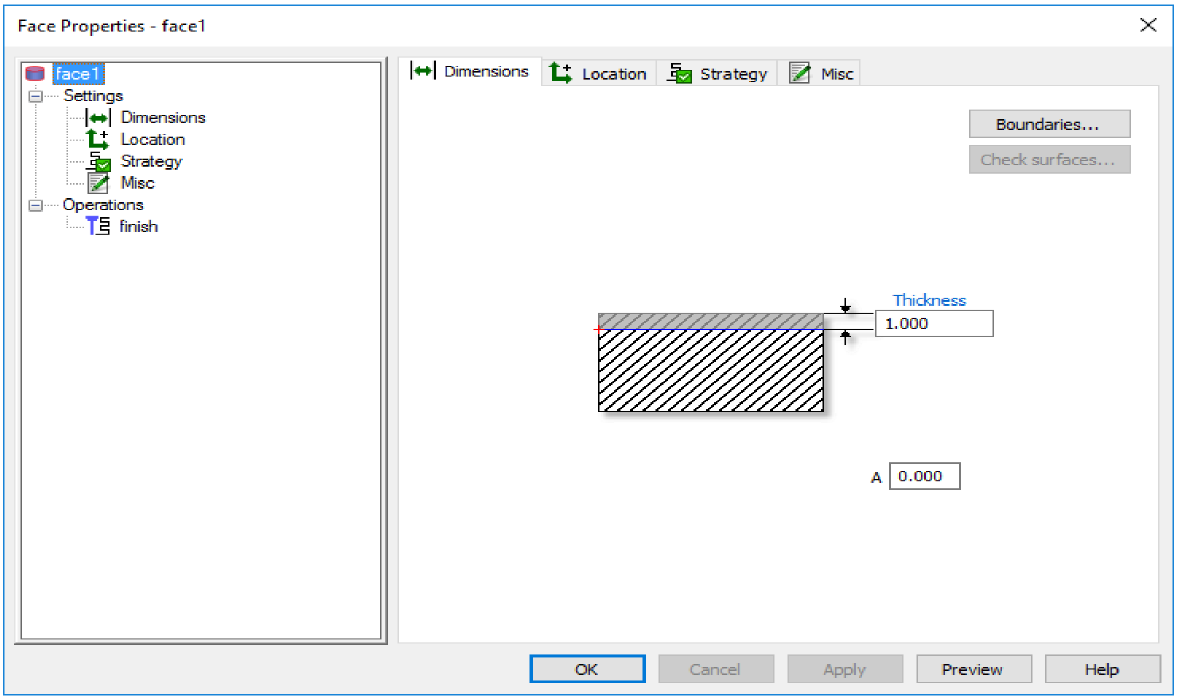
*We have excess material on the top face. We need to create a **Facing** Operation.*

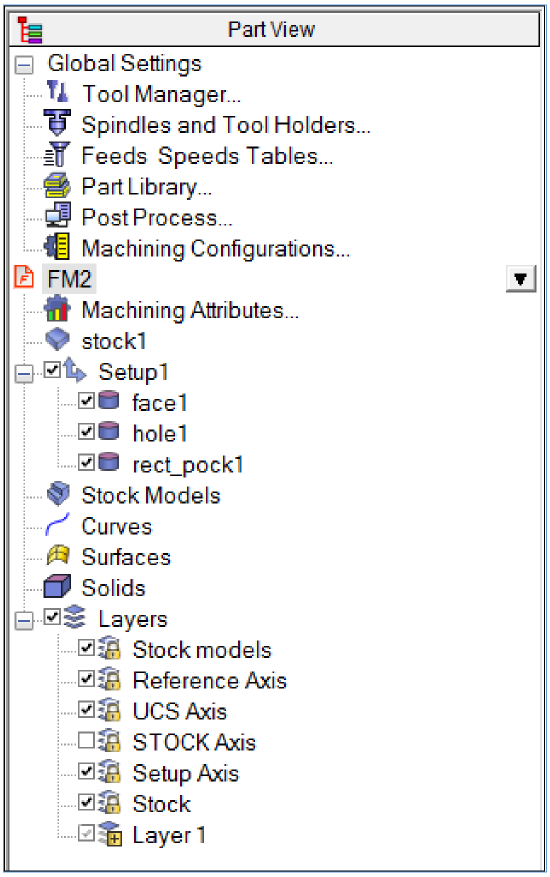
9 Select **Ctrl+R New Feature** or from **Steps** select 

10 Select **Face**. Remember to save the file as **Mill Exercise1.fm**



11 Select **Finish**. **FeatureCAM** will automatically select an appropriate Facemill and machine the top face to Zero. And re-order the Operations list, so that the **Face** operation jumps to the beginning of the operation list, because **Face** has a higher **Base Priority**.

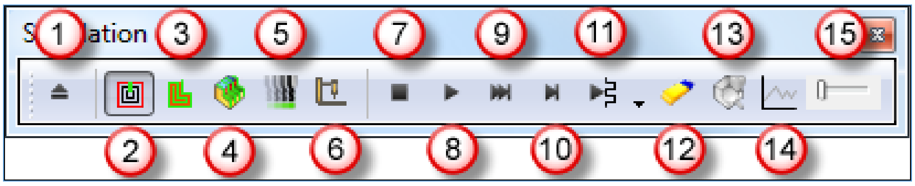




*That is it the job is finished; **FeatureCAM** has automatically created the operations.*

*Select **3D Simulation** and then press **Play*** 8 *on the **Simulation Toolbar**.*

Examine the diagram below of the Simulation Toolbar; Read the purpose of each button



1 **Eject**: Erases the Simulation from the Graphics window.

2 **Show Centreline**: A line drawn at the Centre of the tip of the tool is displayed.

3 **2D Simulation**: A two-dimensional Simulation showing the regions cut by each operation is displayed. The view is changed to the top view automatically.

4 **3D Simulation**: A 3D solid Simulation is displayed where the tool is animated through all of its moves.

5 **3D Rapid Cut**: In this mode a 3D Simulation is performed but the tool is not animated. Only the final result is displayed. For most parts, the simulation takes only a few seconds to complete.

This type of Simulation is only available in ***FeatureMILL3D****.*

6 **Machine Simulation**: A 3D solid Simulation is displayed where the tool is animated through all of its moves along with the machine tool

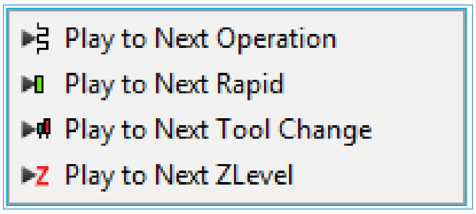
7 **Stop**: Cancels a Simulation.

8 **Play:** Starts the selected Simulation (Centreline, 2D, 3D or Rapid Cut), or restarts a paused Simulation.

9 **Fast Forward to end:** Skips to the end of the animation

10 **Single step:** Moves the Simulation ahead one tool move. The keyboard accelerator for this button is **ALT+F3** or **ALT+Right Arrow**

11 **Play to Next Operation:** continues to simulate until the next operation. This button is actually a fly-out menu. By clicking on the triangle to the right of the button the following additional options are revealed.



Next Rapid simulates until the next rapid tool move.

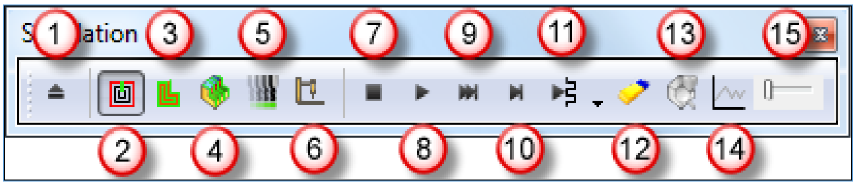
Next Tool Change simulates until the next tool change.

Next ZLevel will simulate the next Z of a Z level toolpath. For other toolpaths it will play the entire next operation

12 **Clear Toolpath:** Erases any Centreline toolpaths on the screen.

13 **Region of interest** limits the portion of the part that is simulated.

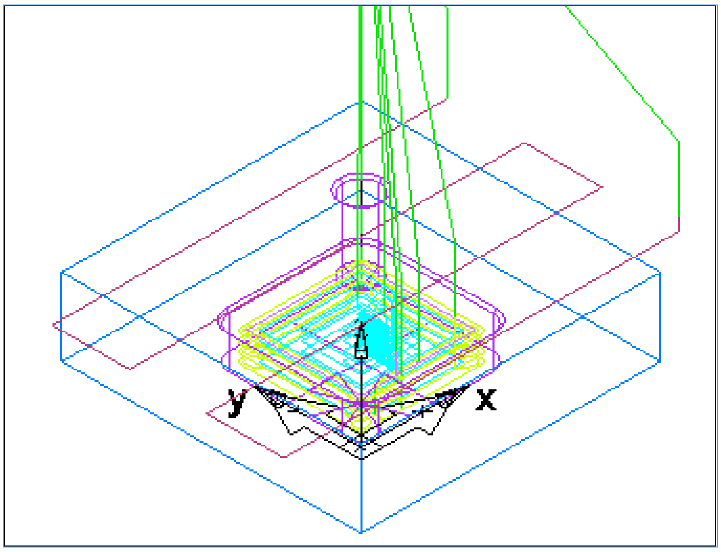
14 **Show tool load** indicates whether or not to display a graph of the tool load when the next 3D Simulation is performed.



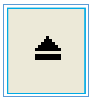
15 **Sim Speed:** To adjust the speed of a **Centreline**, **2D** or **3D Simulation**, use the **Slider** on the right-hand side of the controls.

*Slide to the right to speed up, and move to the left to slow down the **Simulation**. The Slider on the **Simulation Toolbar** also affects the display for **Rapid Cut Simulation**. If the Slider is all the way to the right, only the final **Simulation** result is displayed. Position the slider bar further to the left to see slower Rapid results.*

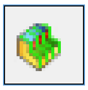
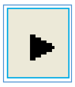
You should have a result similar to that shown on the next page.



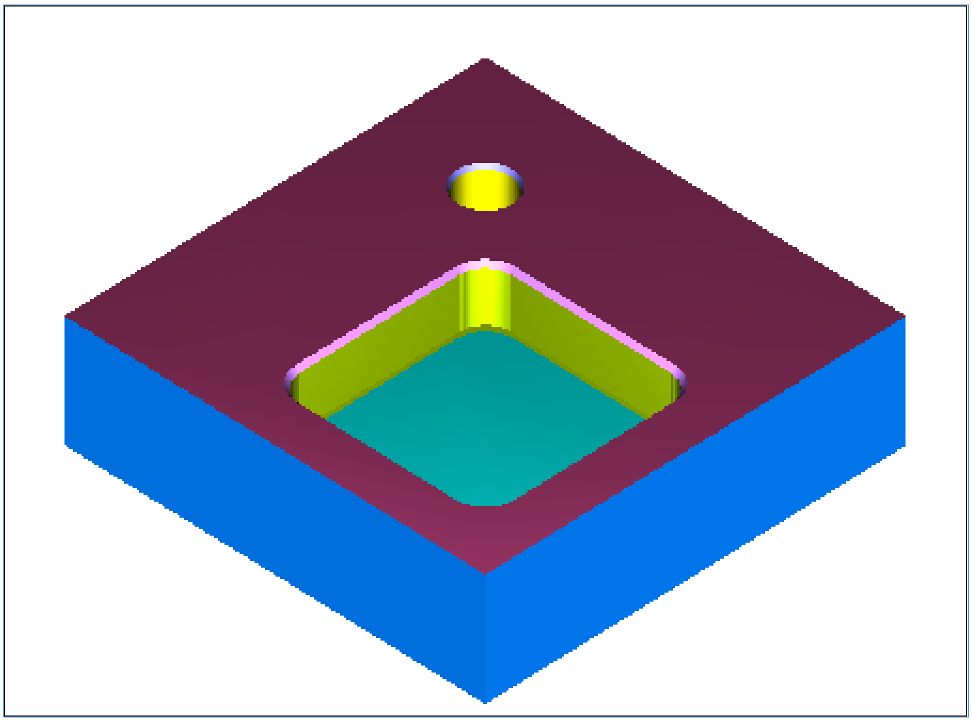
*A **Centreline** display of the toolpath is shown above where the lines displayed represent the centre of the bottom of the tools movement. Green lines represent rapid movement where blue lines show feed moves.*

*Press the **Eject** button*  *to close the simulation.*

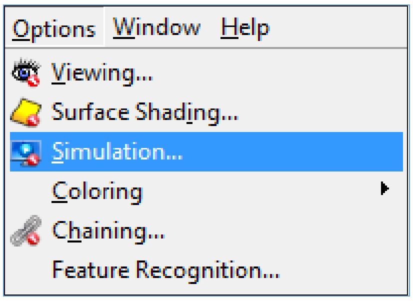
*The image shown is an **isometric View** of the **Toolpath Simulation** with different **colours** representing each operation.*

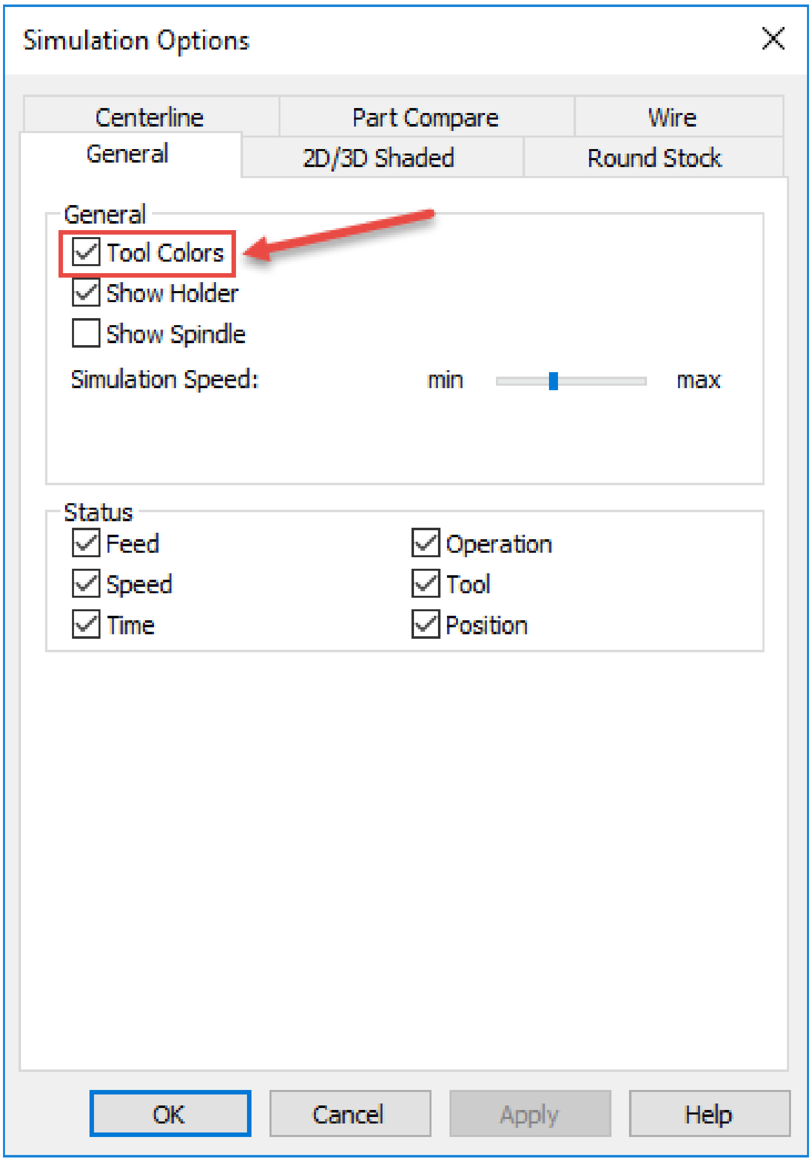
Select the **3D Simulation** button  and press **Play**  to show the image shown on the previous page.

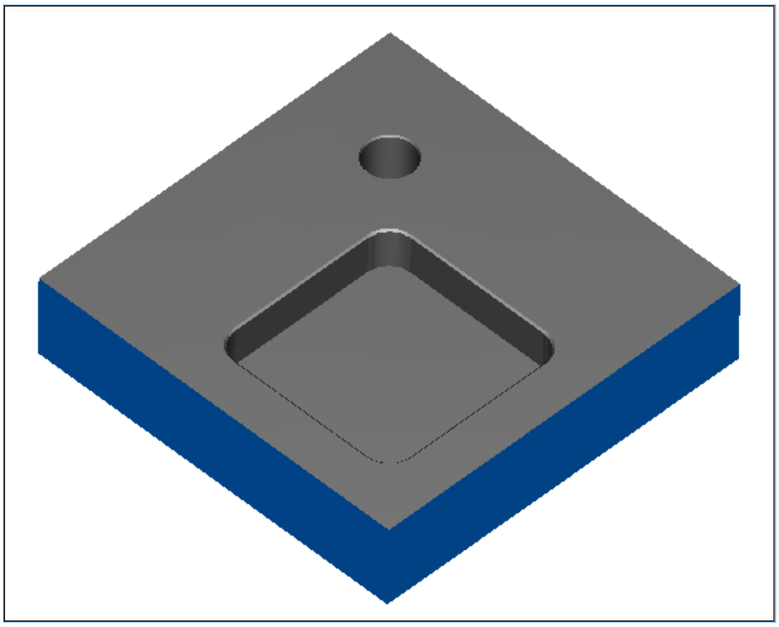
Code can only be output after the Simulation has finished. If the simulation fails. NC code will not be output.



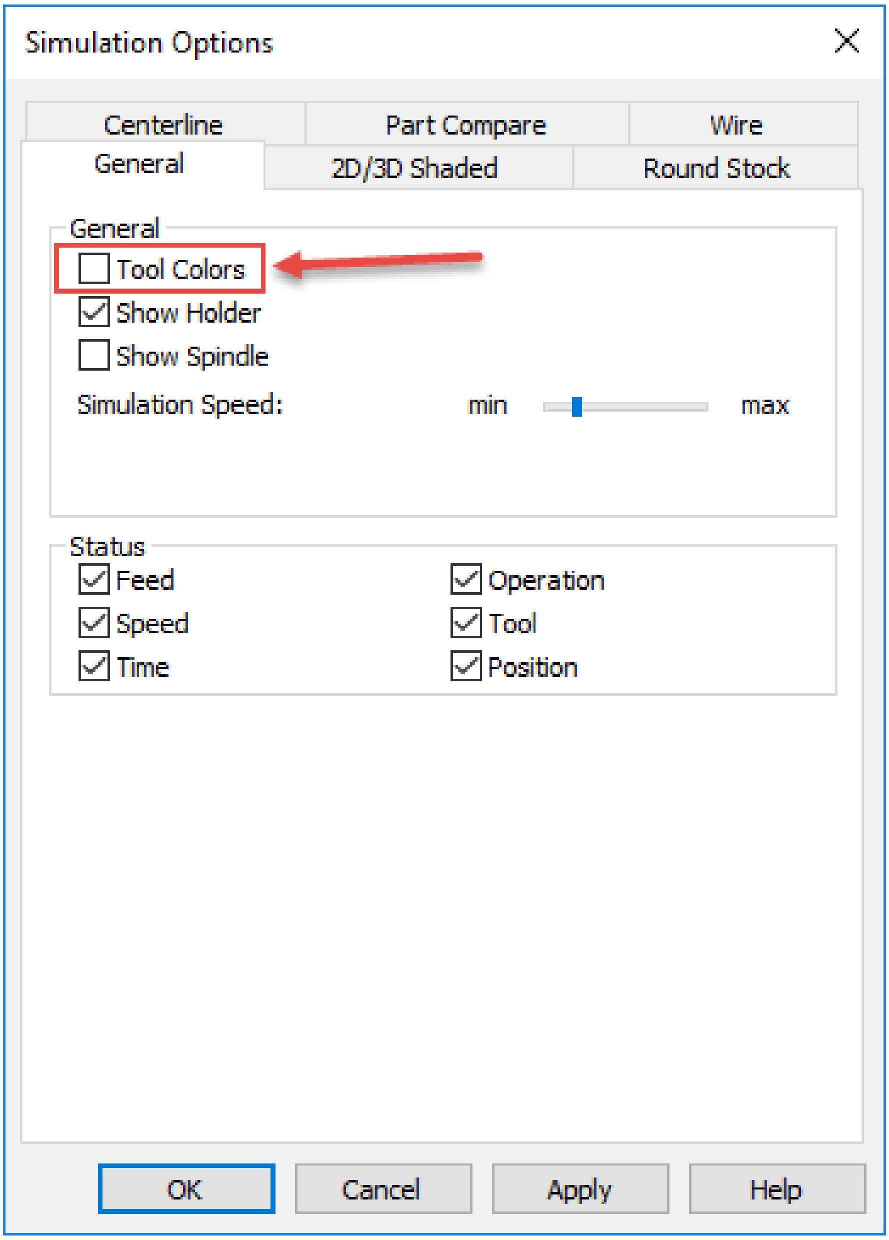
***Tip**: The image shown on the previous page shows **Tool Colours** switched on. **Colours** can be configured to suit your own preferences for all movements. Select **Options**>**Simulation>Simulation Options**.*



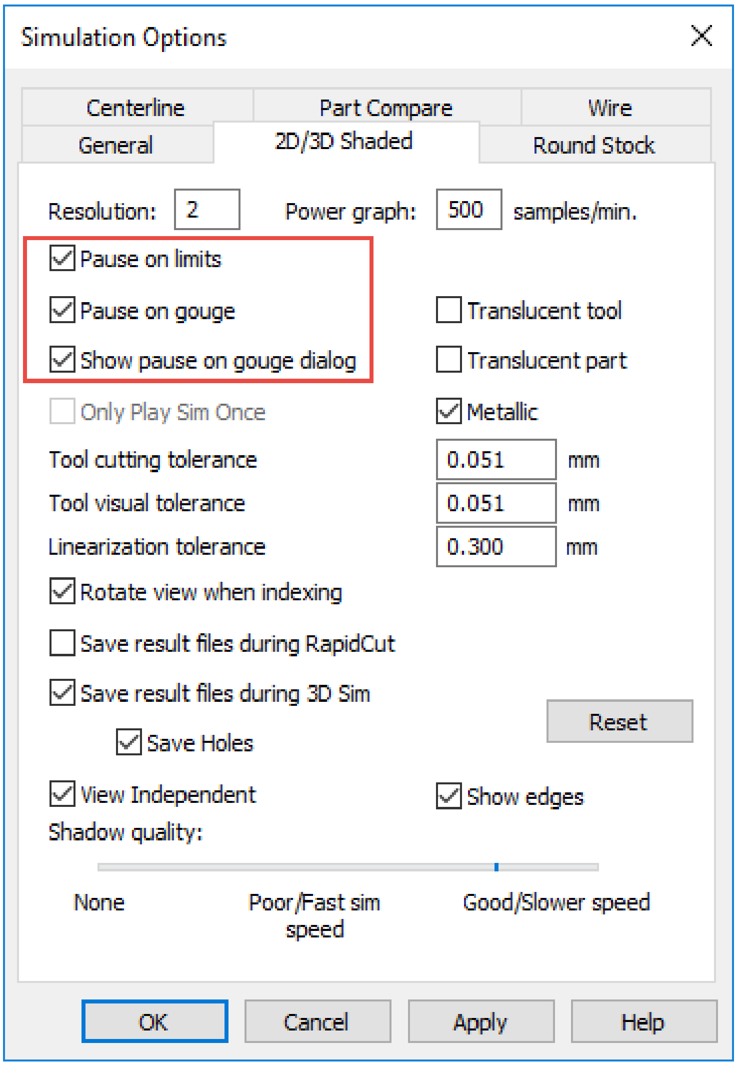

Simulation Options
Centerline
Part Compare
Wire
General
2D/3D Shaded
Round Stock
General
Tool Colors
Show Holder
Show Spindle
Simulation Speed:
min
max
Status
Feed
Speed
Time
Operation
Tool
Position
OK
Cancel
Apply
Help



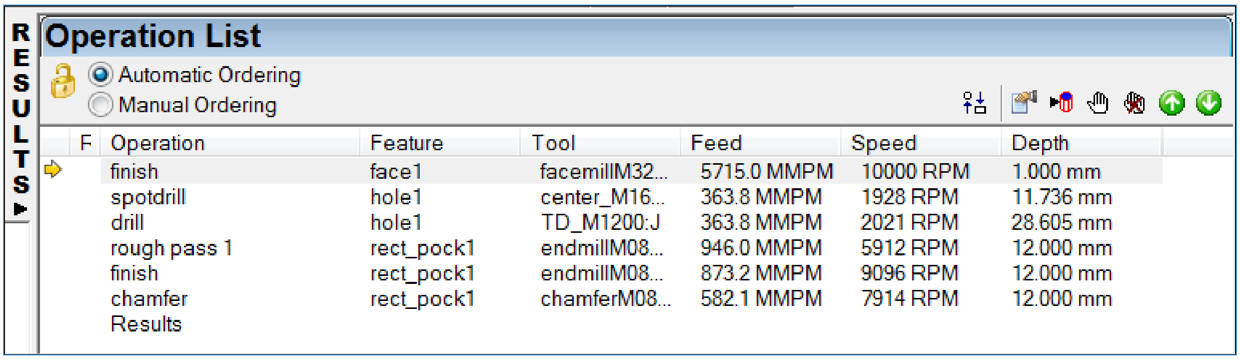
Tool colours switched off



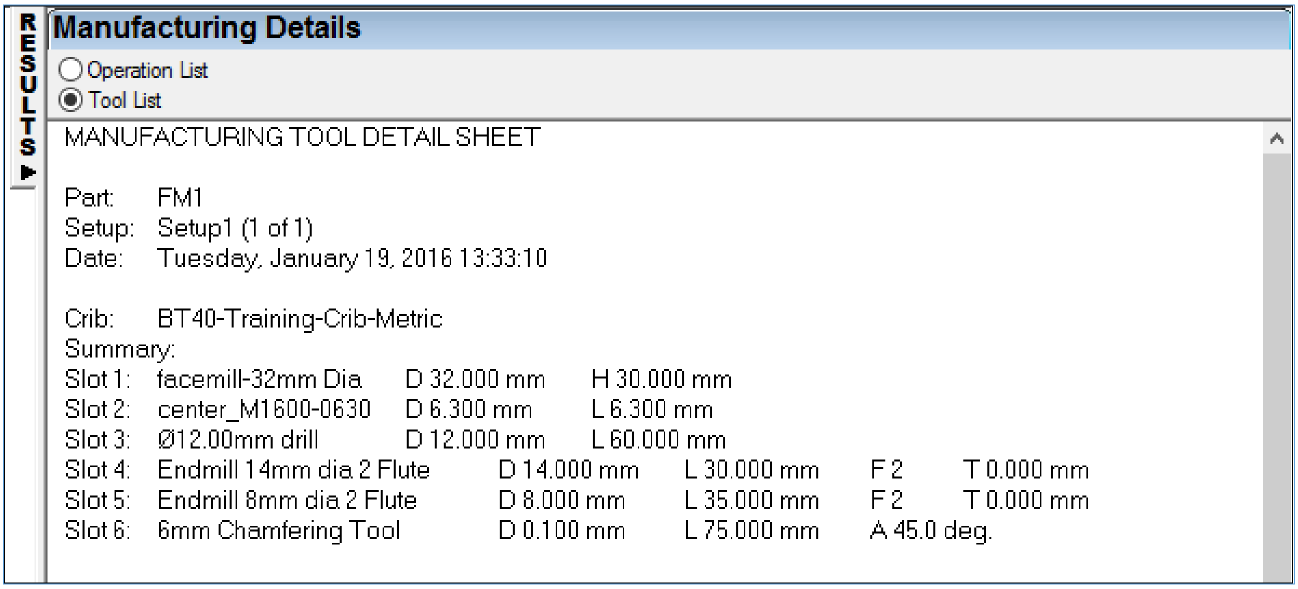
*In **Options>Simulation>2D/3D Shaded** from the top menu, Select **Pause on Gouge**, this will not only show the holder for the milling tool, but reveal any tool striking the stock in rapid traverse, or if the holder touches any part of the stock. These gouges will show as pink in colour, the simulation will stop, and a pop up menu will appear telling you of a possible gouge.*



*In the **Manufacturing results** window on the right hand side of the screen **click** the **Operation List tab**.*



*Click on the **details** Tab.*



12 Click on the **NC code tab.**

13 Post Selected is **Fanuc_Robodrill.cnc** set to **Metric**. Tool change position **X0, Y0, Z100.**

14 Tool Crib is **Training_Crib_Metric**.

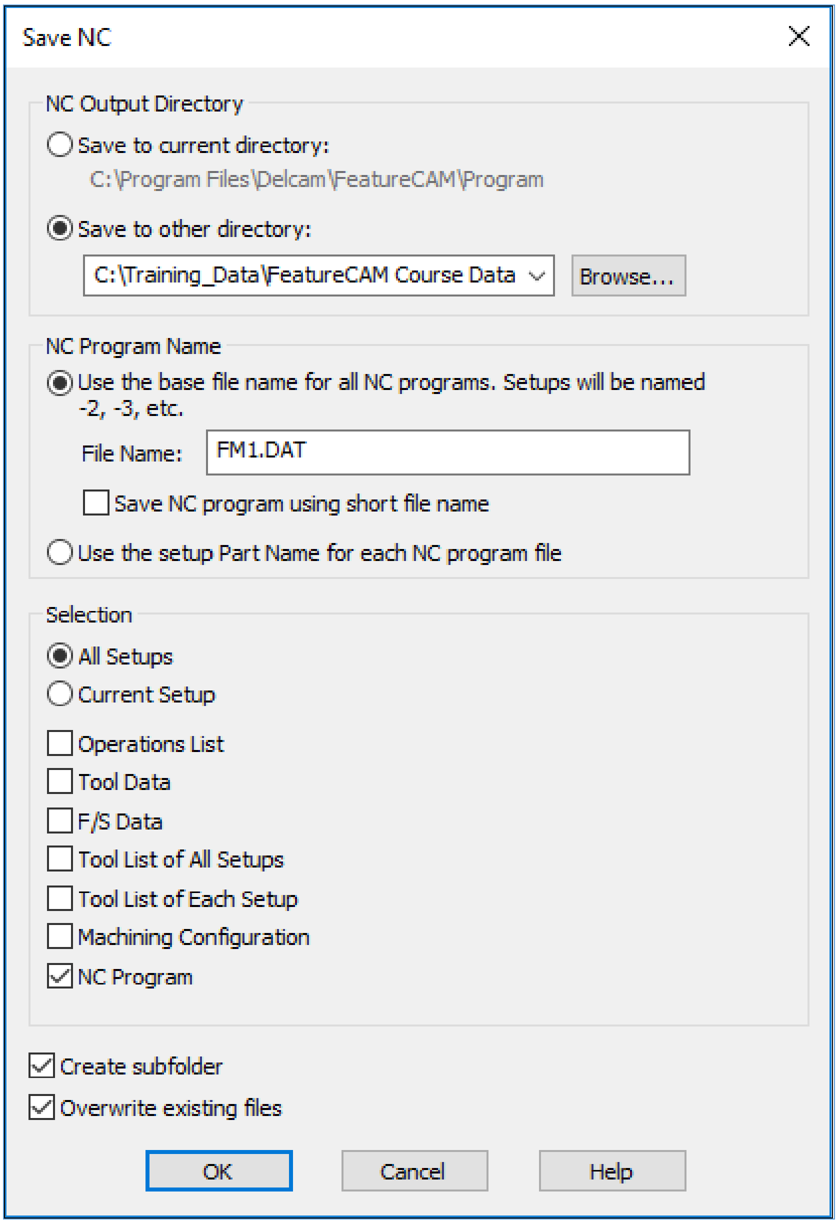
15 **NC code** is generated so it may be output and read on the machine tool. The NC Code shown has been generated for the Face operation and Hole Feature only.

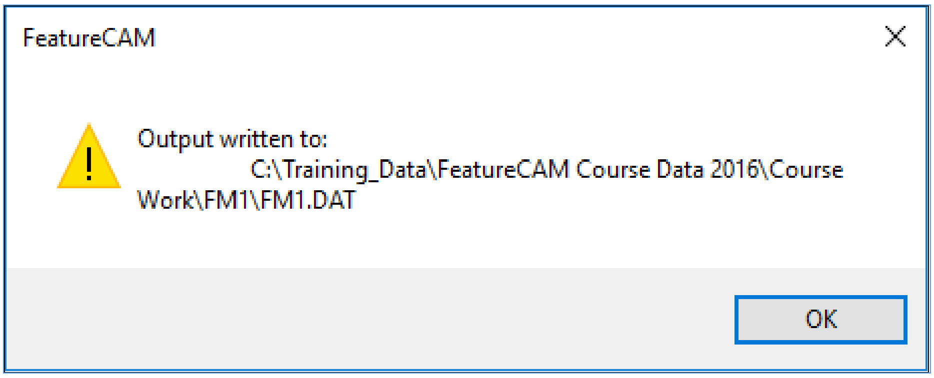
This is an example of the first two tools showing the code that has been generated using the ***Fanuc_Robodrill.cnc*** *Post Processor.*

```
%
O0001
( FINISH  FACE1 )
N25 G0 G40 G49 G80 G90
N30 T1 M6
N35 G54 X117.0 Y9.8
N40 M03 S10000
N45 G43 H1 Z26.0 M8
N50 Z4.0
N55 G1 Z0. F5000.0
N60 X-17.0
N65 Y35.6
N70 X117.0
N75 Y61.4
N80 X-17.0
N85 Y87.2
N90 X117.0
```

```
N95 G0 Z26.0
N100 M5
N105 M9
N110 G91 G28 Z0.
N115 G90 G49
(   HOLE1 )
N125 G0 G40 G49 G80 G90
N130 T2 M6
N135 G54 X75.0 Y75.0
N140 M03 S1843
N145 G43 H2 Z26.0 M8
N150 G98 G81 R3.0 Z-12.236 F363.66
N155 G80
N170 M5
N175 M9
N170 G91 G28 Z0.
N175 G90 G49
(   HOLE1 )
N185 G0 G40 G49 G80 G90
N190 T3 M6
N195 G54 X75.0 Y75.0
N200 M03 S2021
N205 G43 H3 Z26.0 M8
N210 G98 G83 R3.0 Z-28.605 Q3.0 F363.78
N215 G80
N220 M5
N225 M9
N230 G91 G28 Z0.
N235 G90 G49
N240 M30
%
```

16 Select **File** from the main menu and select **Save NC**. Click the radio button **Save to other Directory** and choose a directory that you're Instructor has chosen to save your files to. Select the following options.



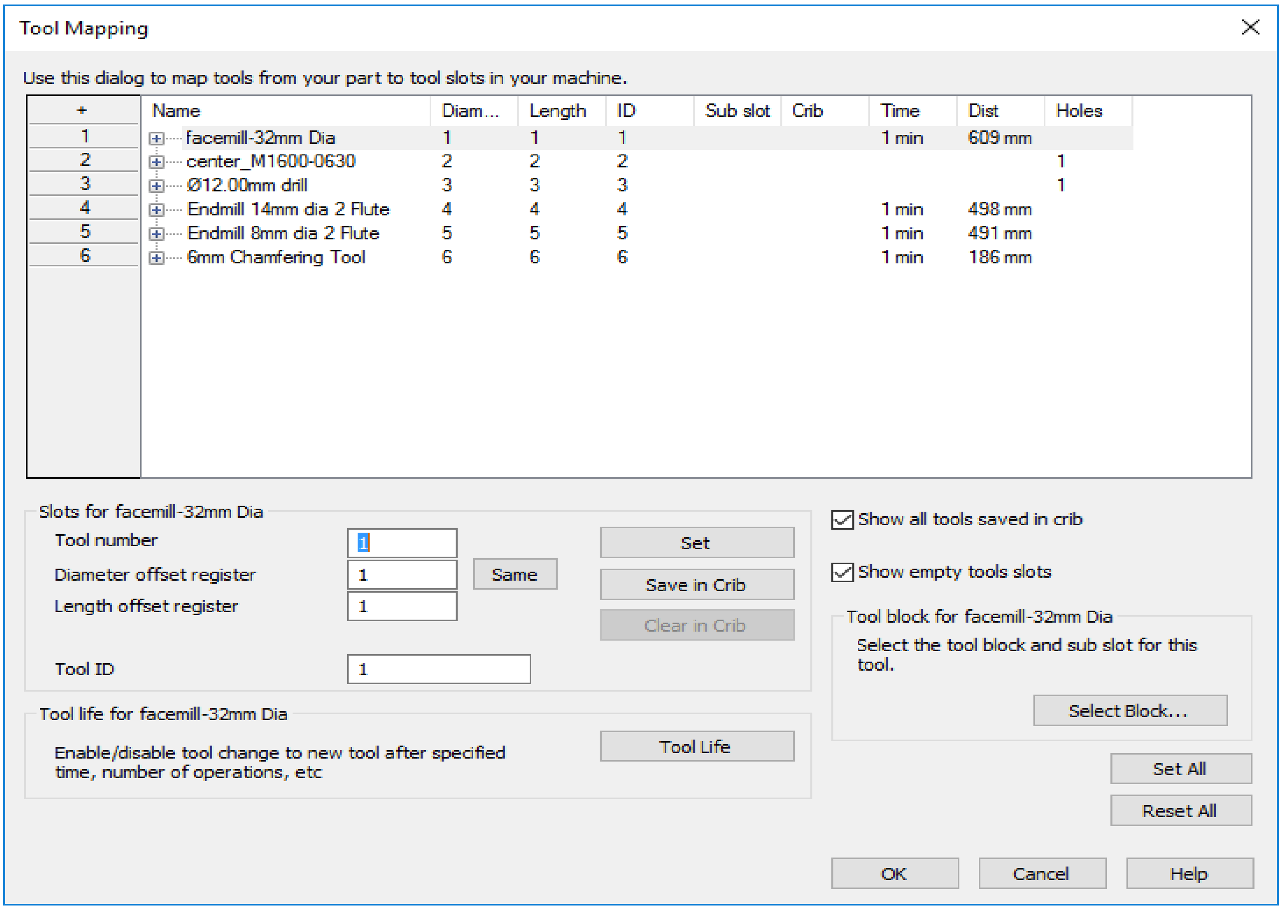


*Now that the **NC code** is **saved** it can be downloaded to the machine and the part with the simple pocket and hole can be machined.*

Tool Mapping

You can open the Tool Mapping dialog in one of these ways:

- Select **Manufacturing > Tool Mapping** from the menu.
- Click NC Code in the Steps panel, then click Re-map the tools to new tool slots in the NC Code dialog.



- If the order of the tools is changed you will have to re-run the simulation and re-output the code to the same folder.
- The **Tool Mapping** dialog is where you change the tool slot assigned to the selected tool. You can change the **Cutter comp**. **offset register** for any tool here too.
- The dialog has a table at the top. Each row of the table represents a tool. Select a tool to edit its values in the fields below the table.
- Double-click on a tool name, or click the **+** to the left of the tool name to see the list of operations that use that tool.
- Click the **Add tool slots** + button at the top left of the table to open the Number of tool slots dialog. It enables you to increase the number of tool slots listed; you cannot reduce this number.

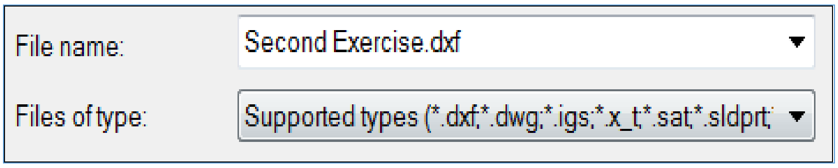
From Curve second exercise (Class exercise)

1 Open **FeatureCAM** by double clicking the **FeatureCAM** shortcut icon.

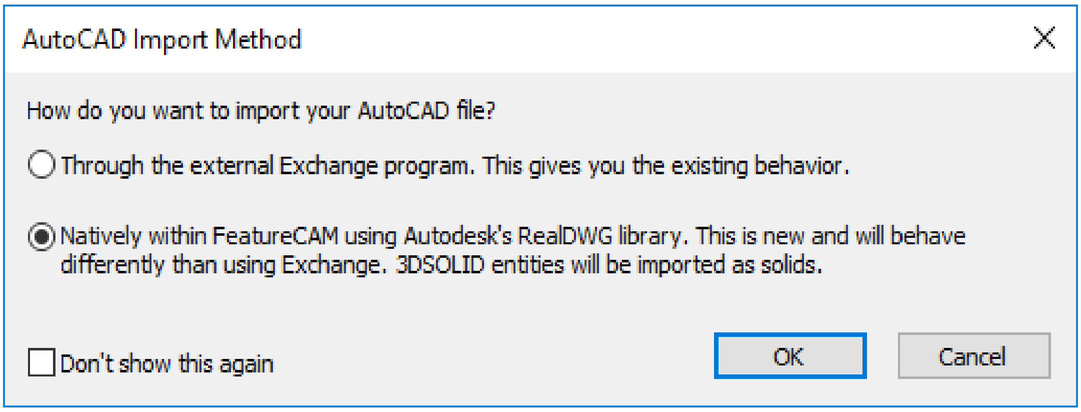
2 Select **New file** then **Next.** Then select **Milling Setup** then select **Millimetres.** Select **Finish.** You will be presented with a Dimensions menu just select **Cancel** for now and **Hide** the **Stock** on the screen. Select the **boundary** of the Stock then **Right click** and select **Hide Stock**. **The stock** will then disappear from view.

3 Go to **File** and select **Import** and at the bottom of the form select **files of type** as **.dxf (*.dxf).**

*You will find the file in. **C: /Training_Data/FeatureCAM Course Data 2017/Milling Files to import.***



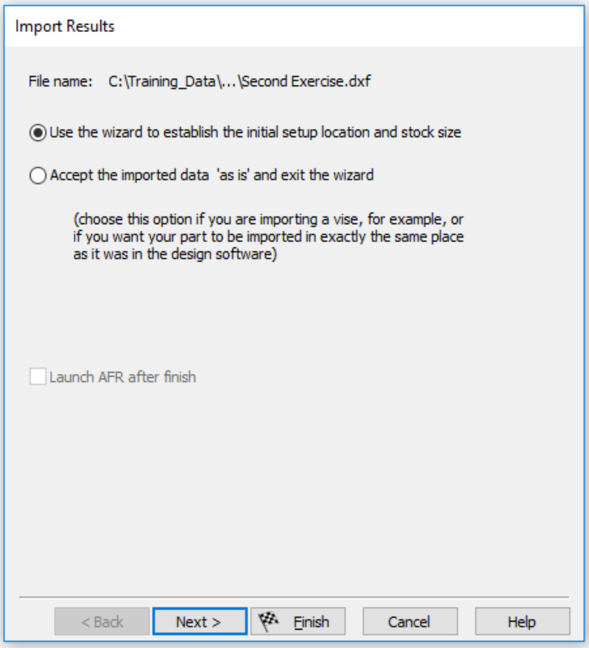
4 Select **Second Exercise.dxf** Select **Open**.

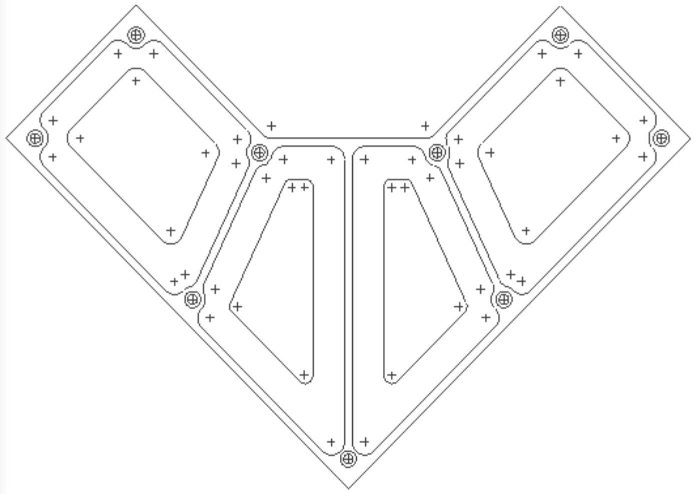


*Use the default option for file import. This uses the native **Autodesk's RealDWG file import**.*

5 You will be presented with a form which shows **Import Results**.

6 This will default to: - **Use the Wizard to establish the initial setup location and Stock size.**



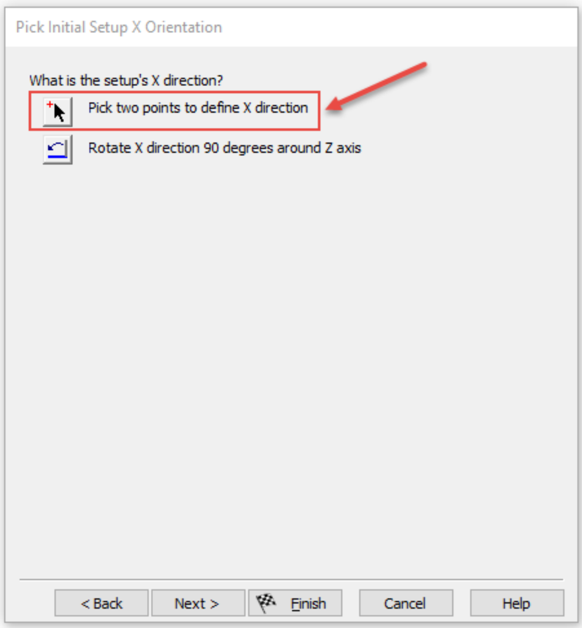


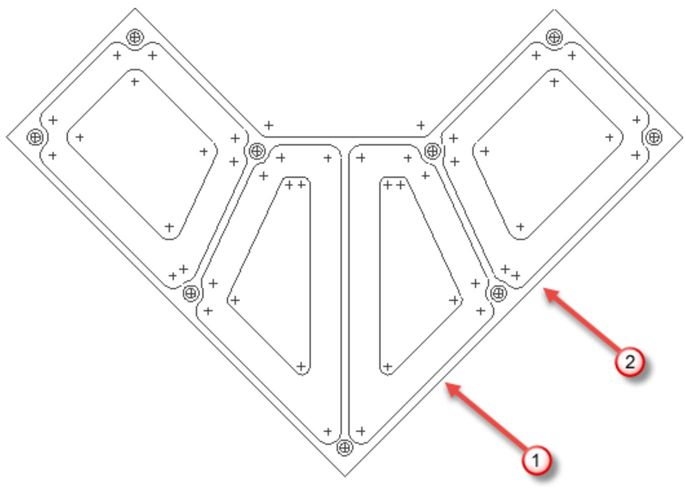
7 Select **Next** twice**.**

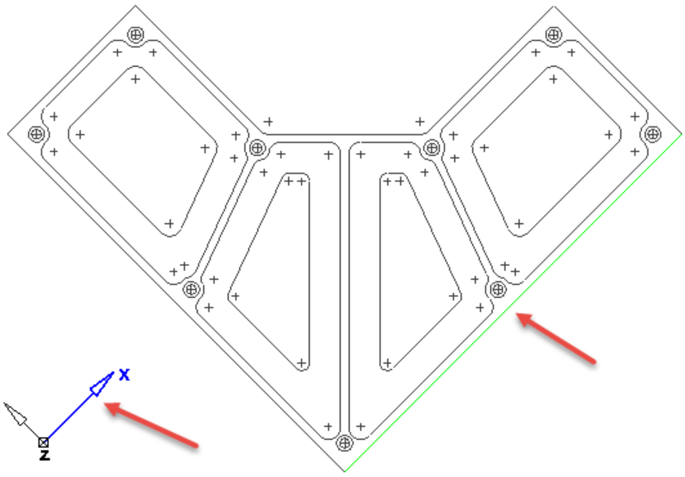
The part is already aligned in the Z axis.

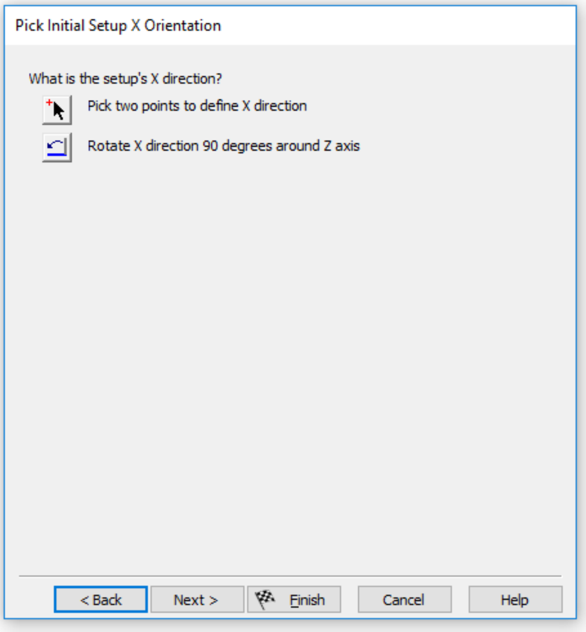
To reduce the amount of material needed to machine this part we need to orientate the part so the right hand long edge is aligned to the ***X axis****.*

8 Select the **Pick two points to define X direction** icon. Then select the edge in two points as shown on the next page.







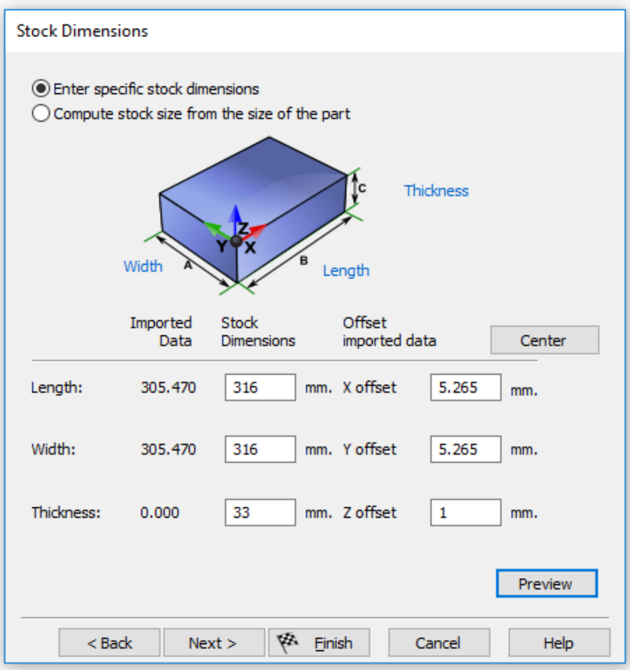


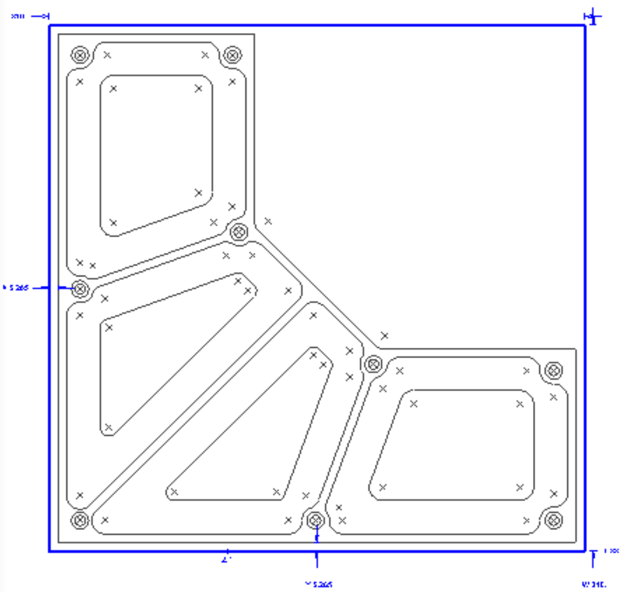
Notice the X axis is aligned parallel to the right hand edge.

9 Select **Next**. Use **Stock type** = **Block**. Select **Next**.

You will be presented with a form that shows ***Stock Dimensions*** *where we have two options.*

Option 1 - Enter specific stock dimensions or Option 2 - Compute stock size from the size of the part. *Other stock options are available but we will discuss this later on. Enter the following values.*

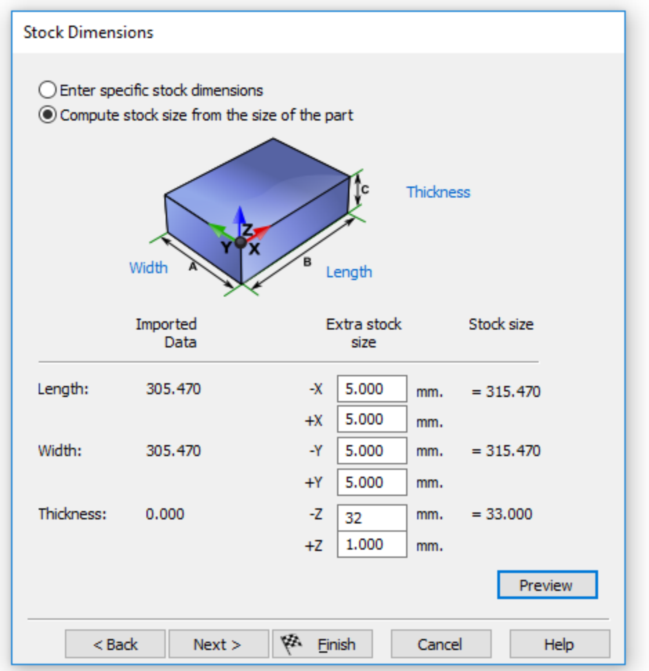


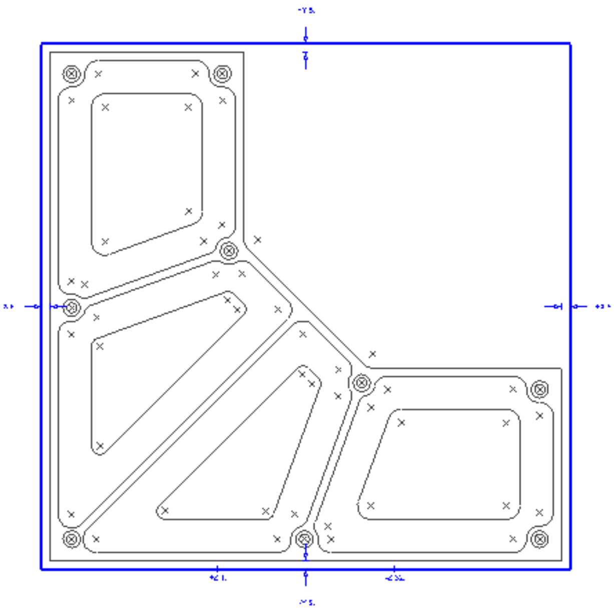


Fill in the dimensions shown above. Do not click ***Next*** *or* ***Finish?***

10 Please select **Compute Stock from the size of the part**.

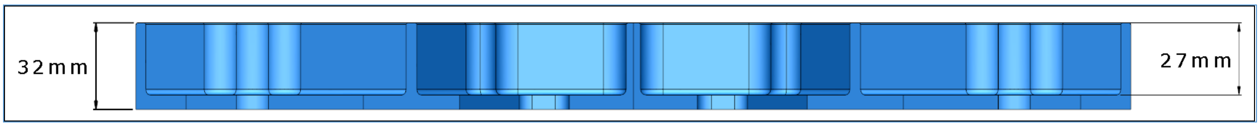
Metric example Option 2 - Compute stock size from the size of the part Metric By entering known values for example **5mm** into the extra stock size fields the Stock dimensions Wizard will automatically Centre the part. Remember to add a value in the **Thickness** field. The thickness will start at **33mm** with **1mm** extra stock size. The finished thickness of the part will finish at **32mm** after facing off the **1mm**.





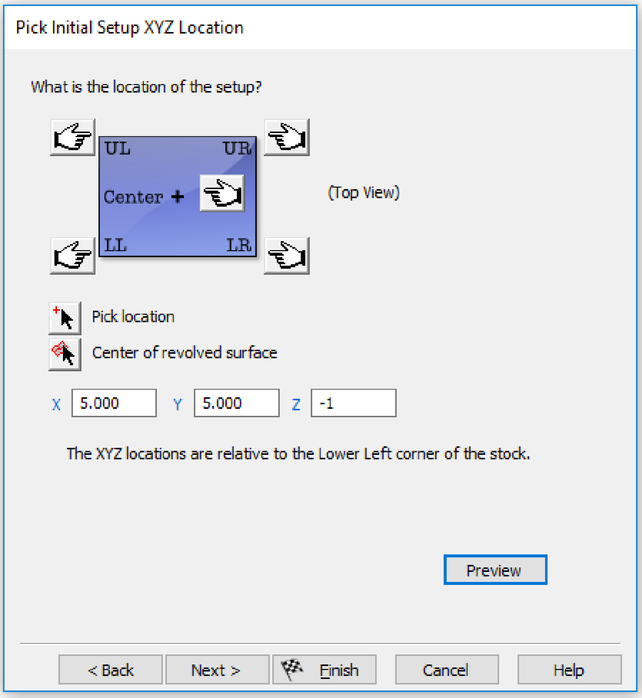
11 Select **Next**.

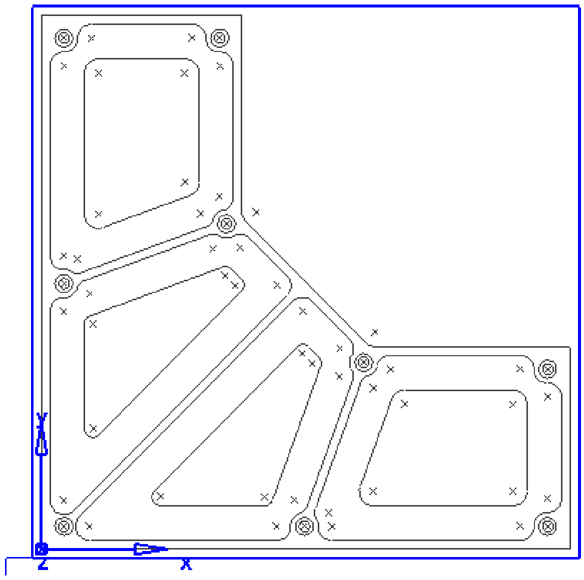
Sectional View



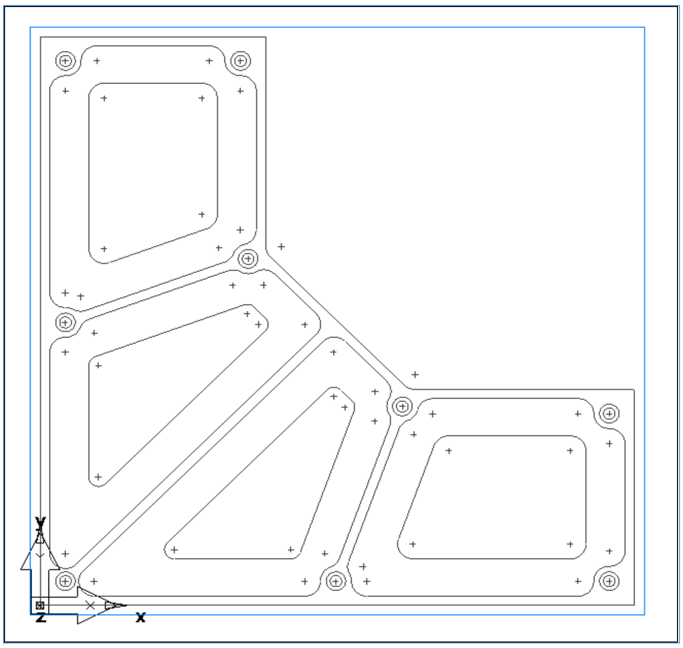
Setups

By selecting **Next** you will be given an option to **Pick Initial Setup XYZ Location**. Select **LL Lower Left,** in **X** enter **5** in **Y** enter **5** and **Z** enter **-1**. This will move the datum by these values; we will then have Stock to machine off when we run a facing operation.



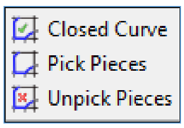


12 Select **Finish** which will close the Wizard.



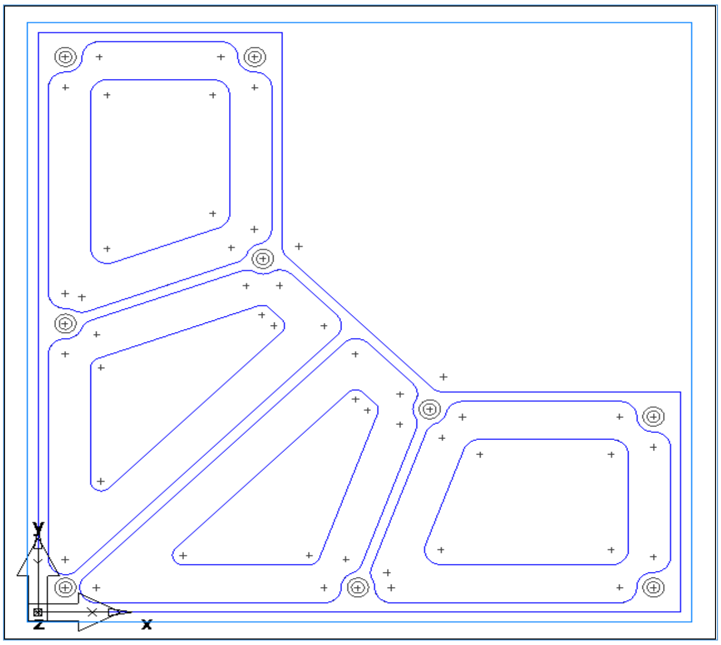
*With Guidance from your instructor we will now create Curves from the geometry and then **Face** the **top** of the **block**.*

*Select **TOP** view or **Ctrl+5.** Select **Ctrl+Q** to Centre the part on the screen.*

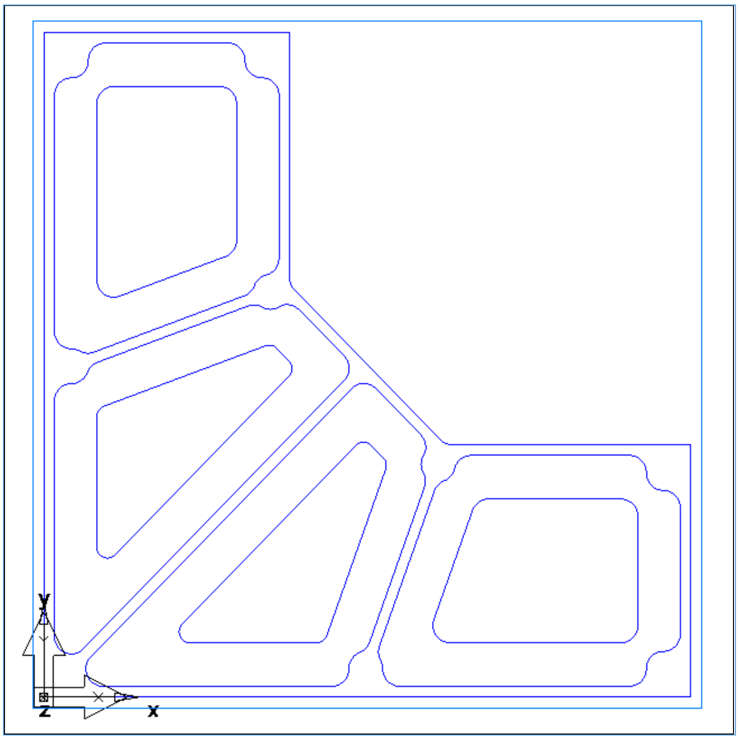


13 Select **Closed Curve** from the **Geometry Menu**

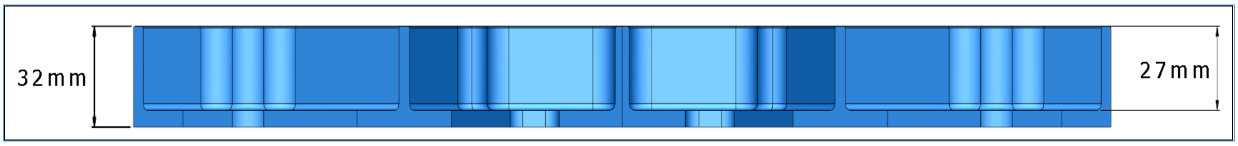
14 Single click the **Geometry** and select **Create**, FeatureCAM will create a **closed curve** around the profile selected. Do this for all profiles. Leave the holes these will be selected from the **Edit** menu using **Select Circles**.



15 Please select **Hide all Geometry** from the **View** Menu as shown.

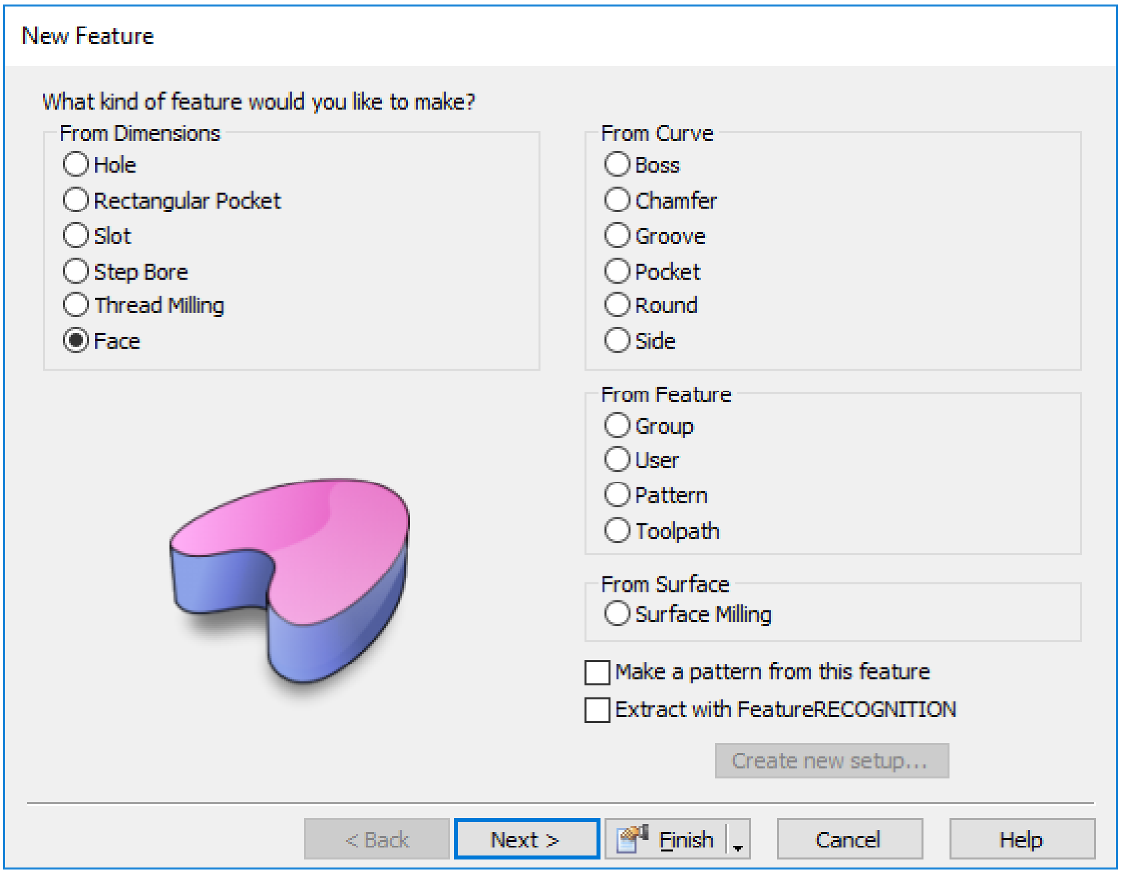


Sectional view of part



16 Select **Ctrl + R** or select **Steps** and **Features**. The **New Feature** dialog will appear.

17 Select **Face**.



18 Select **Finish** and **OK** to close the menu.

We are now going to use the outer curve to limit the face machining that will now just machine to the outer boundary.

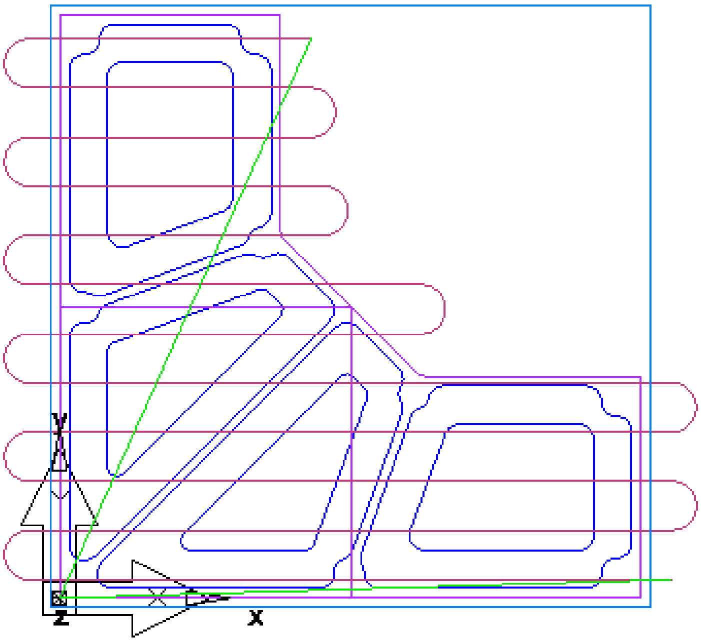
19 Double click **Face** in **PartView**.

20 From the **Dimensions** Tab Select **Boundaries**.

21 Select **Use a Curve as a stock boundary**. Use the **Cursor Icon** to select the curve.

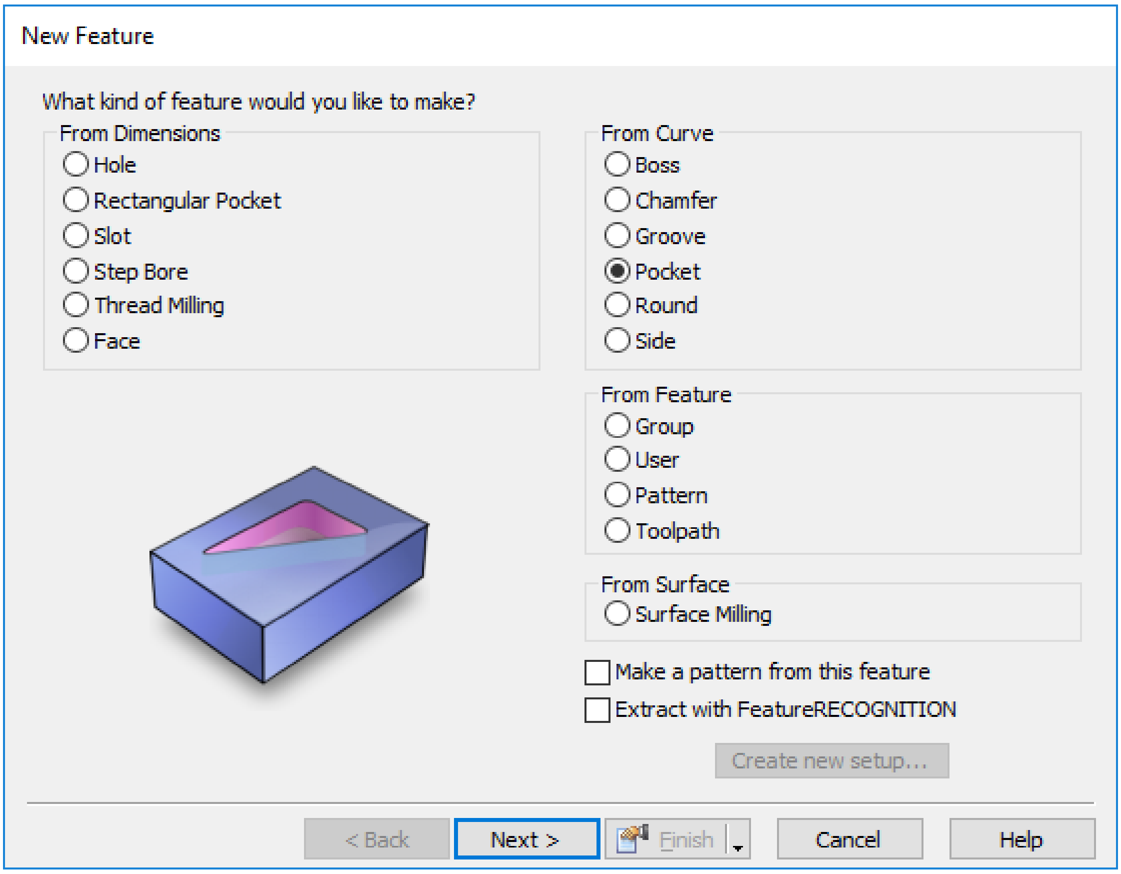
22 Select **OK and Apply**. From the **Strategy tab** select **Connect Step overs with Arc.**

23 Run a **Centreline Simulation** to see the changes.



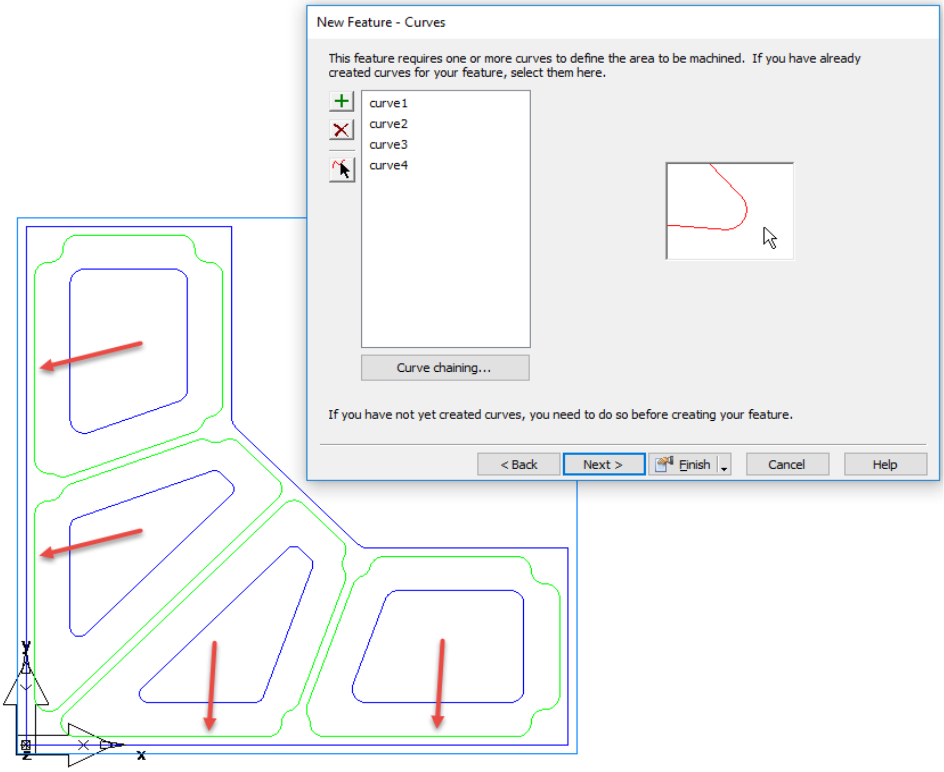
24 Select **Ctrl + R**. The New Feature dialog will appear.

25 Select **Pocket** from Curve.



26 Select **Next**.

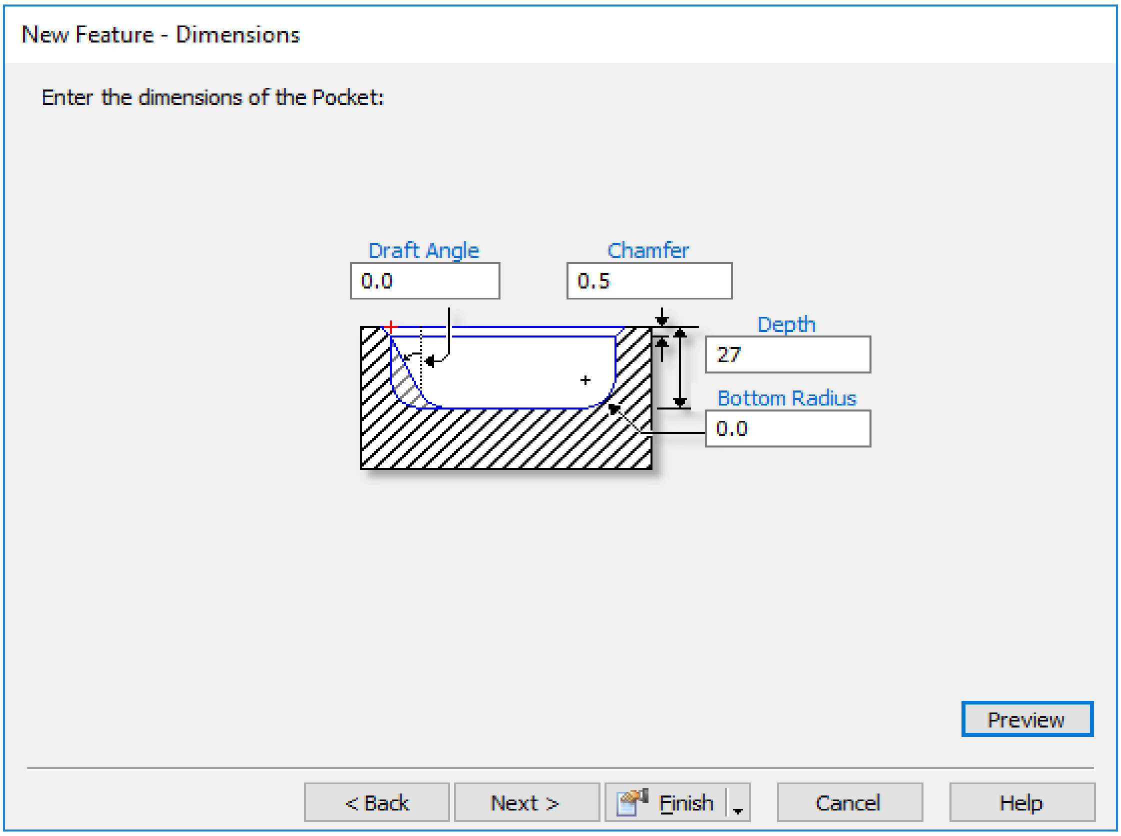
27 Using the **Pick Curve or Geometry** icon select the **4** larger pockets.



28 Select **Next**.

29 In this instance the **Offset from Curve Z Location** is zero.

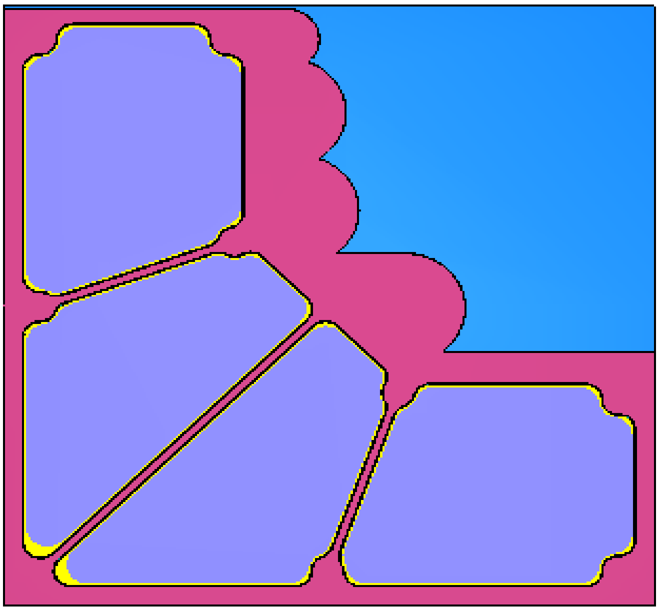
30 Select **Next**. You will now see the **Dimensions** tab. Enter the following values.



31 Select **Finish**.

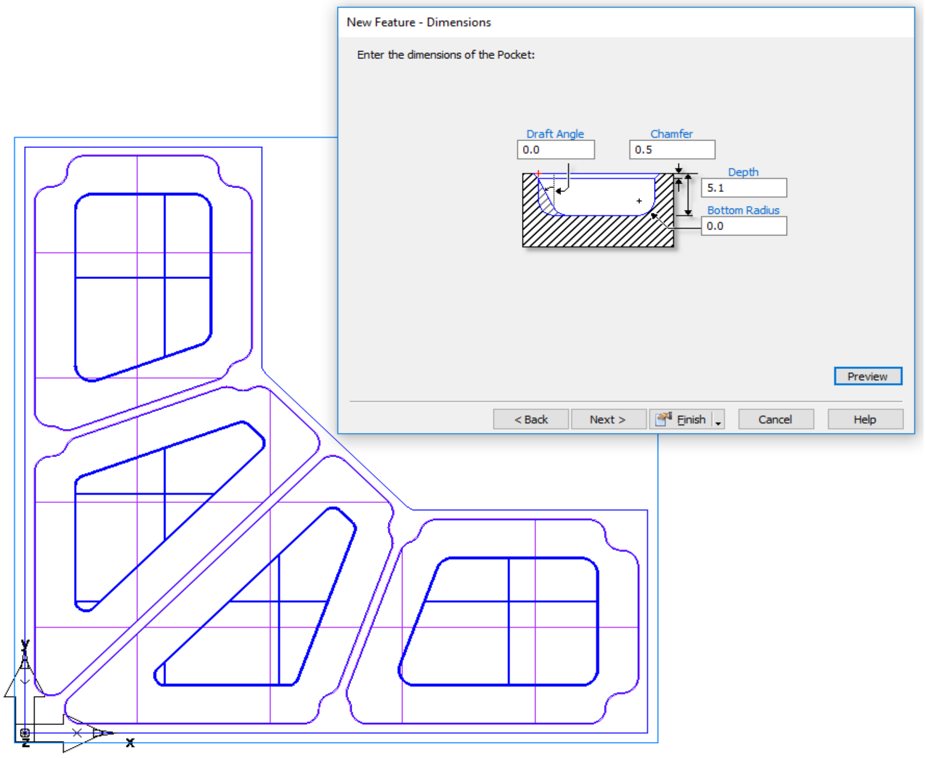
32 Under the **Pocket Properties** tab select **Rough** and you will see **Mult. Rough diameter(s) enter 20, 12** select **Set** and **Apply**.

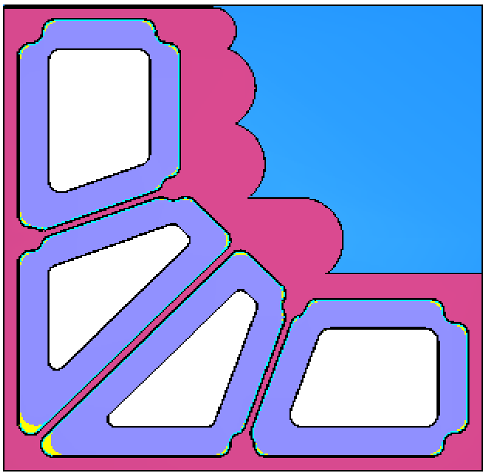
33 This will then rough out using a **20mm** tool and then semi finish the corners with the **12mm cutter**. Set the **Finish** diameter to **8mm**.



*Create another Pocket from curve operation the same as before for the four smaller pockets. The only difference is you will have to set **Offset from Curve Z Location** to **-27** and the **Feature thickness** to **5.1***

34 Under the **Pocket Properties tab** select **Rough** and you will see **Mult. Rough diameter(s)** enter **20, 12** select **Set** and **Apply**. Set the **Finish** diameter to **8mm**.

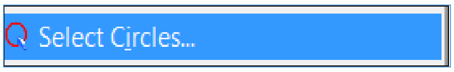




*Press **ESC** to cancel the simulation.*

35 Select **View** from the main menu. Then select **Show all Geometry**.

36 Select **Edit** from the main menu. Then select, **Select Circles**.

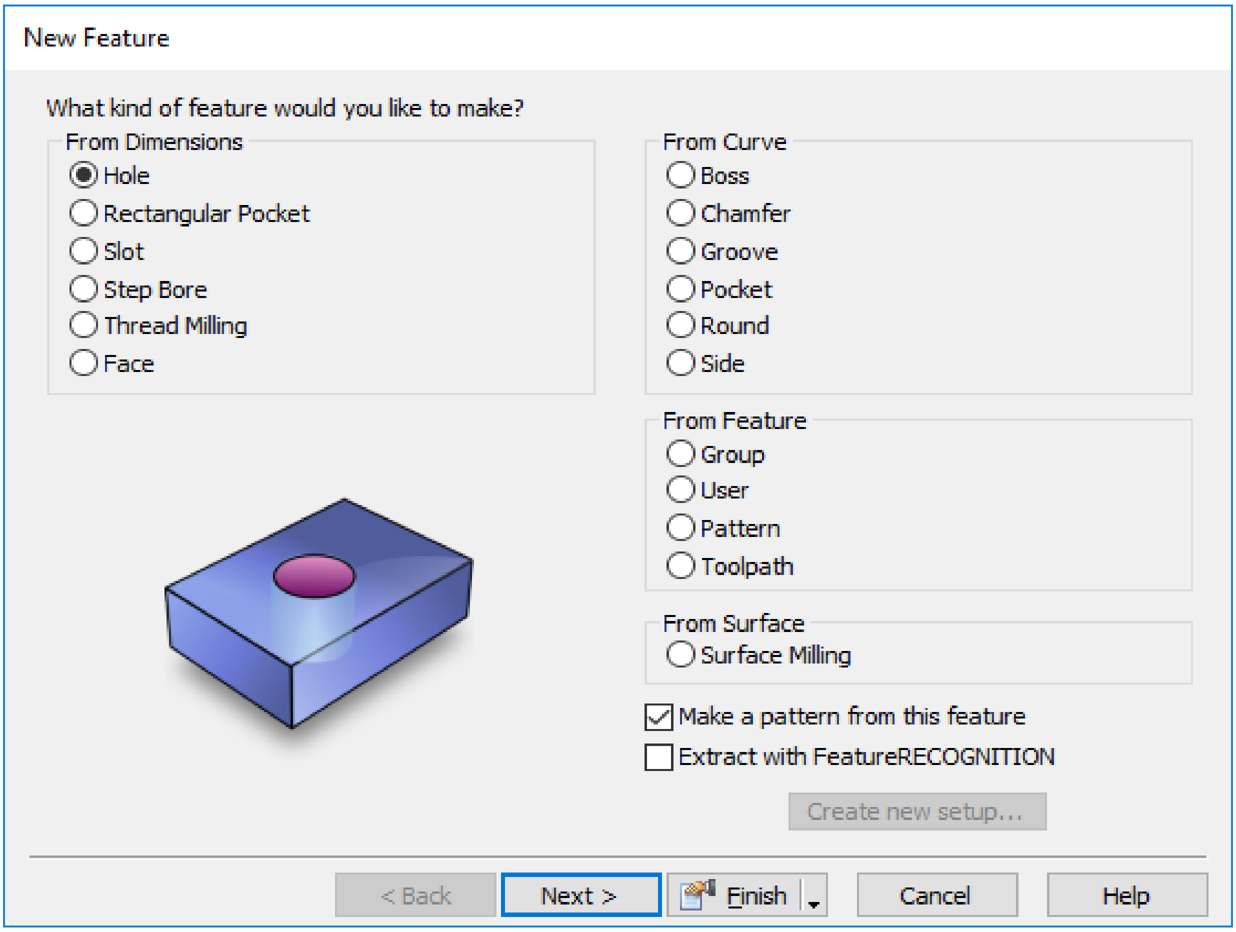


Click on the blue Hyperlink Radius.

37 Then select one of the **6mm** diameter smaller circles. Then select **OK**.

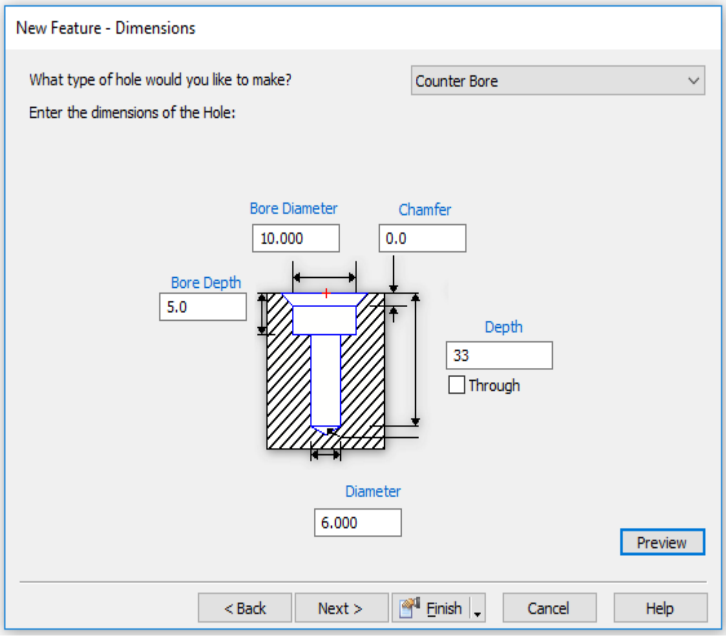
*All of the smaller **6mm** diameter circles will highlight.*

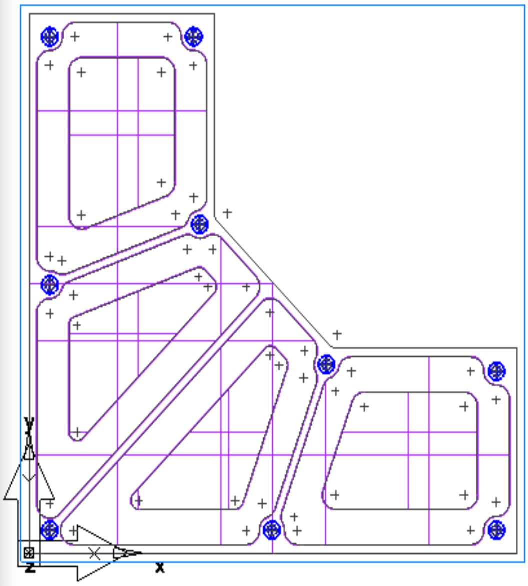
38 Select **Ctrl+R New Feature**.



39 **FeatureCAM** will default to **Hole** and automatically select. **Make a pattern from this Feature.**

40 Select **Next**. Then select **Counter Bore** from the drop down menu.

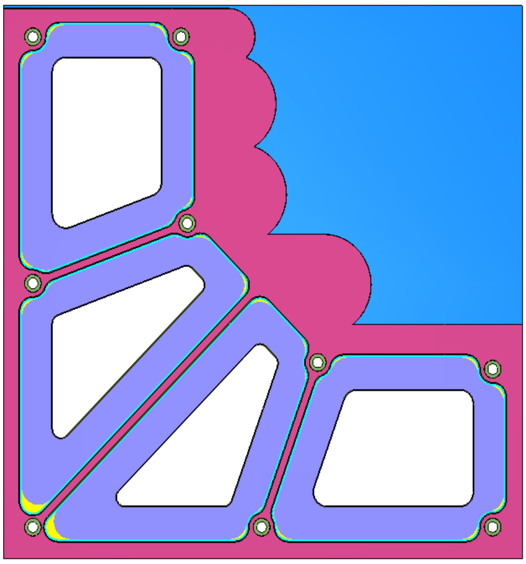




41 Enter **10mm** for the bore diameter and **33mm** for depth.

42 Select **Finish** and **Ok.**

43 **Run the 3D Simulation.**

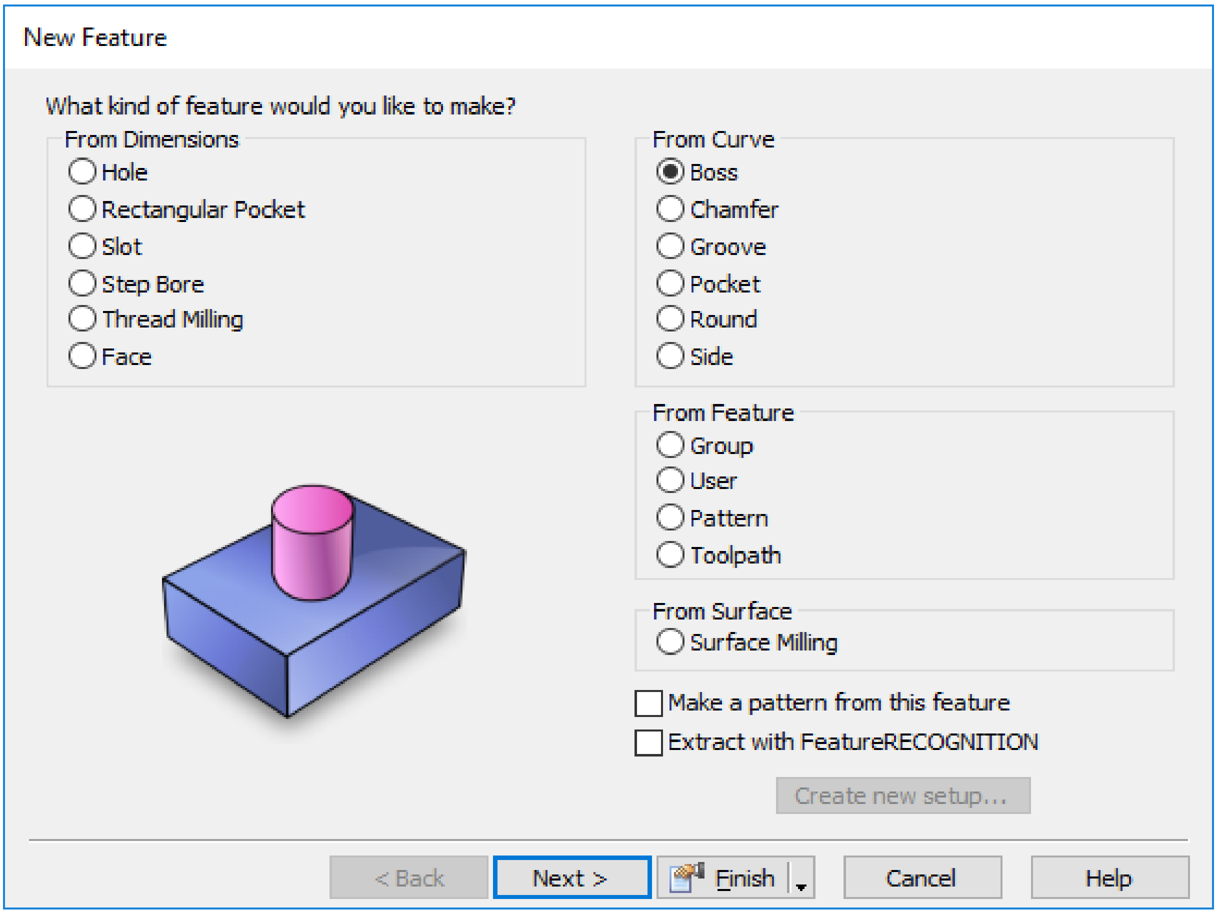


44 Press **ESC** to cancel the simulation.

45 **Hide all Geometry.**

*Create a **Boss** Feature to machine all around the outside of the part.*

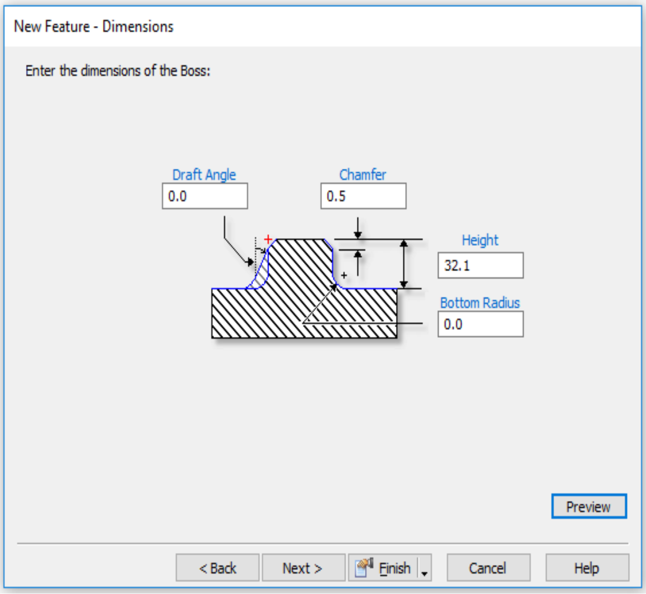
46 Select **Ctrl+R New Feature.** Select **Boss** from **Curve.**

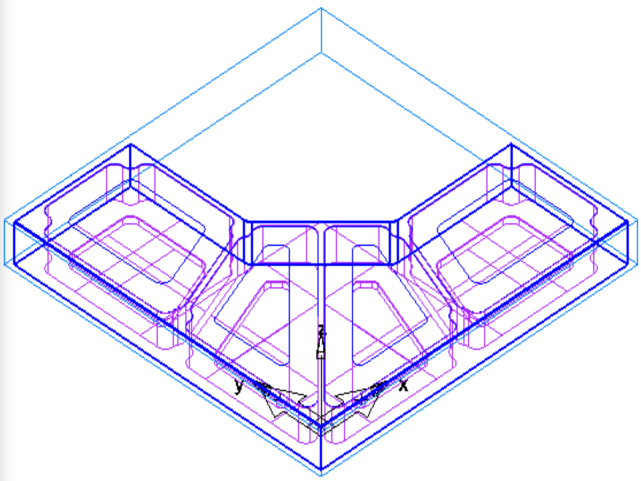


47 Select **Next**.

48 Select the **outer Curve** and then select **Next**. Then set the **Offset from curve Z Location** to **zero**.

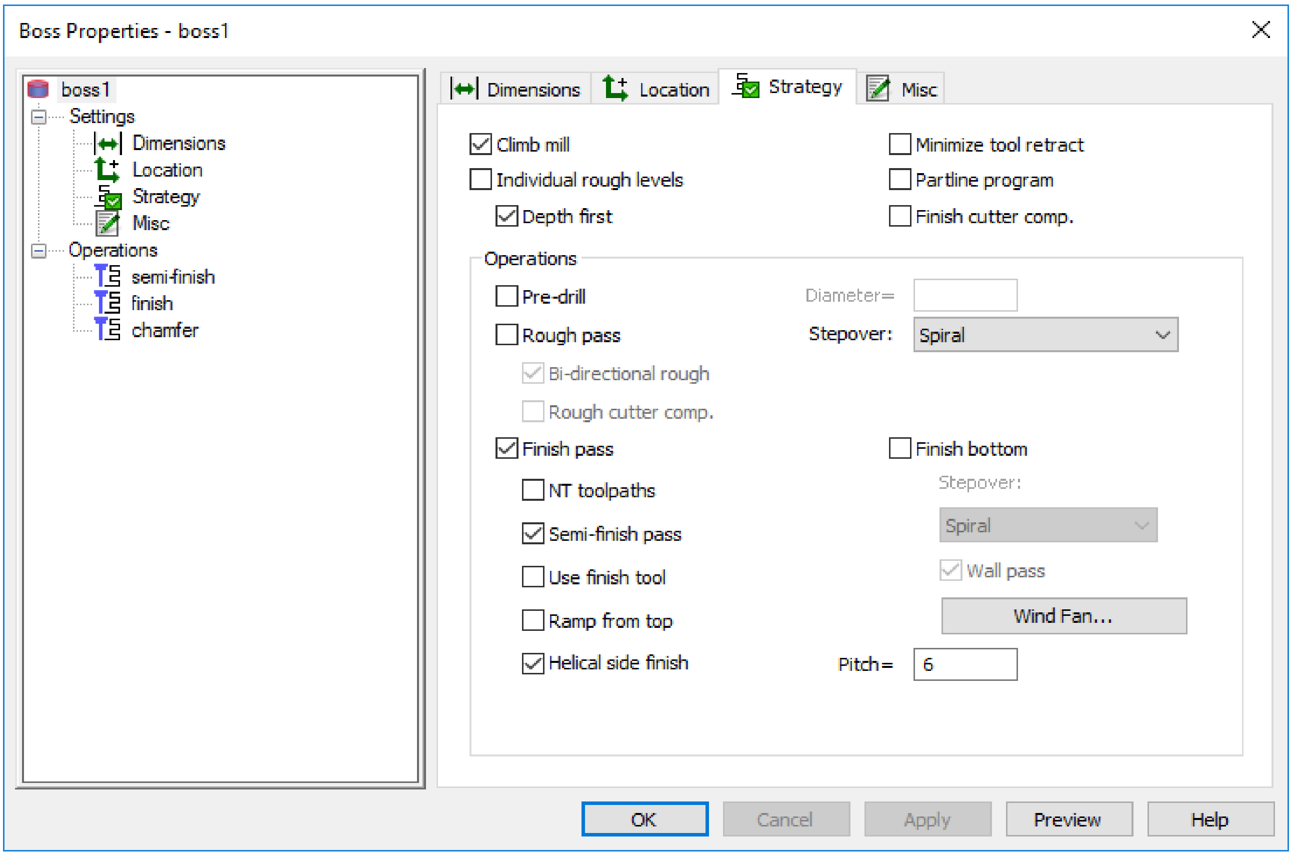
49 Select **Next** and set the following dimensions in the **Dimensions** tab.



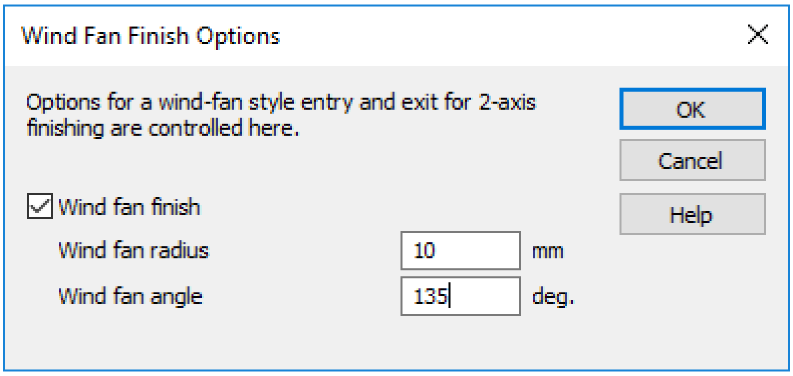


50 Select **Finish**.

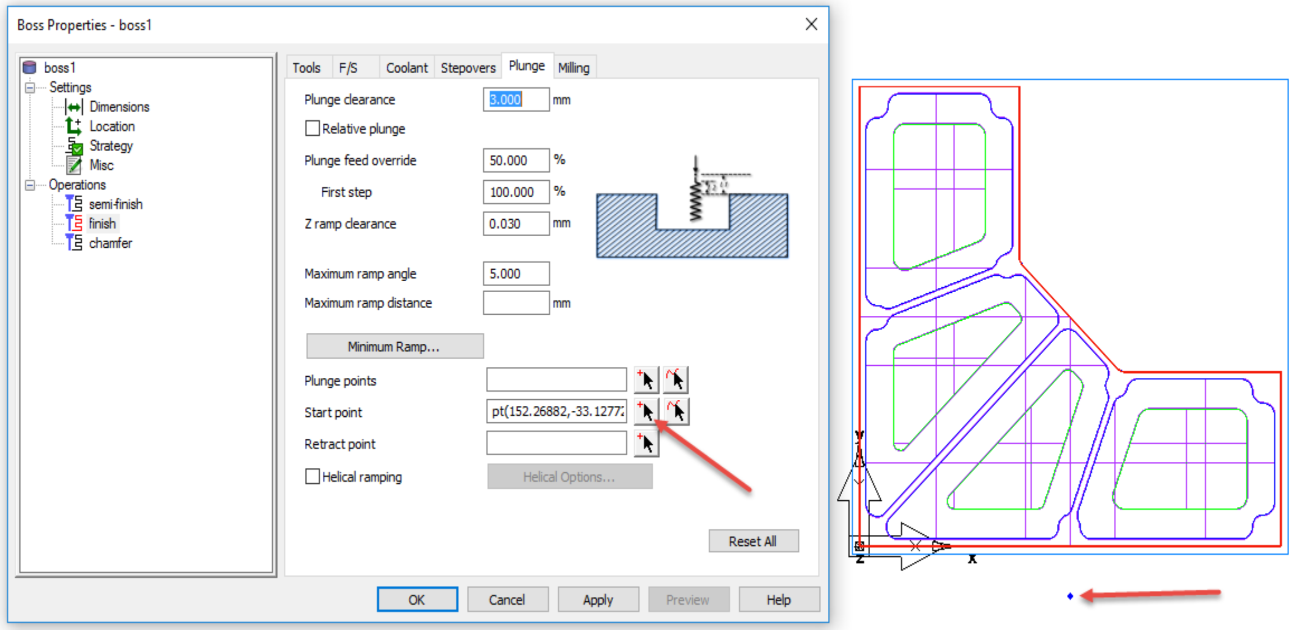
Change the following settings.



51 Under the **Strategy** tab select **Wind Fan.** Enter the following values.

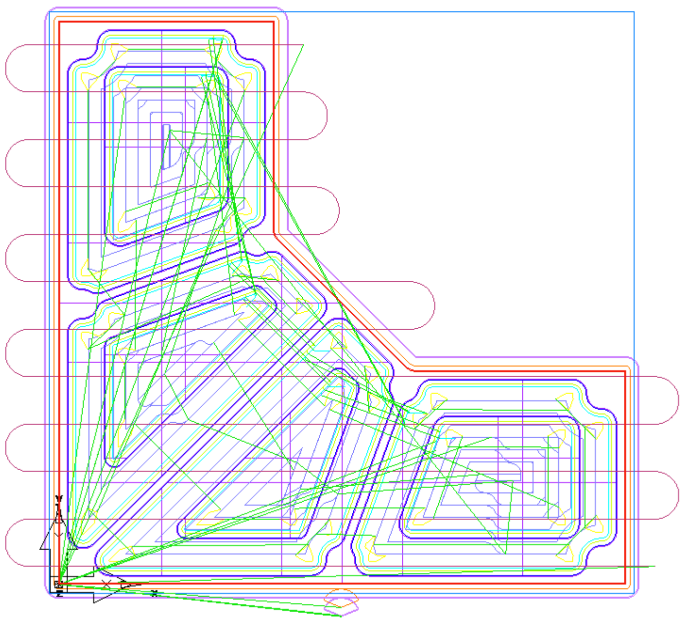


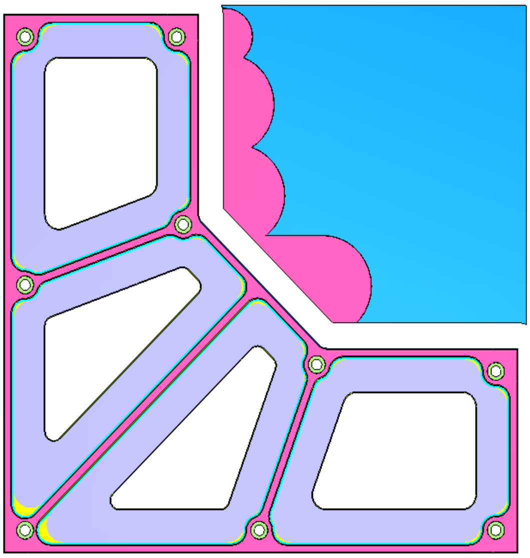
52 To change the start point select the **Plunge** tab. Select **Finish** under Operations. Pick the position as shown for the **start point**.



Run a 3D Simulation of the finished part.

53 Output the code as described in the previous chapter.

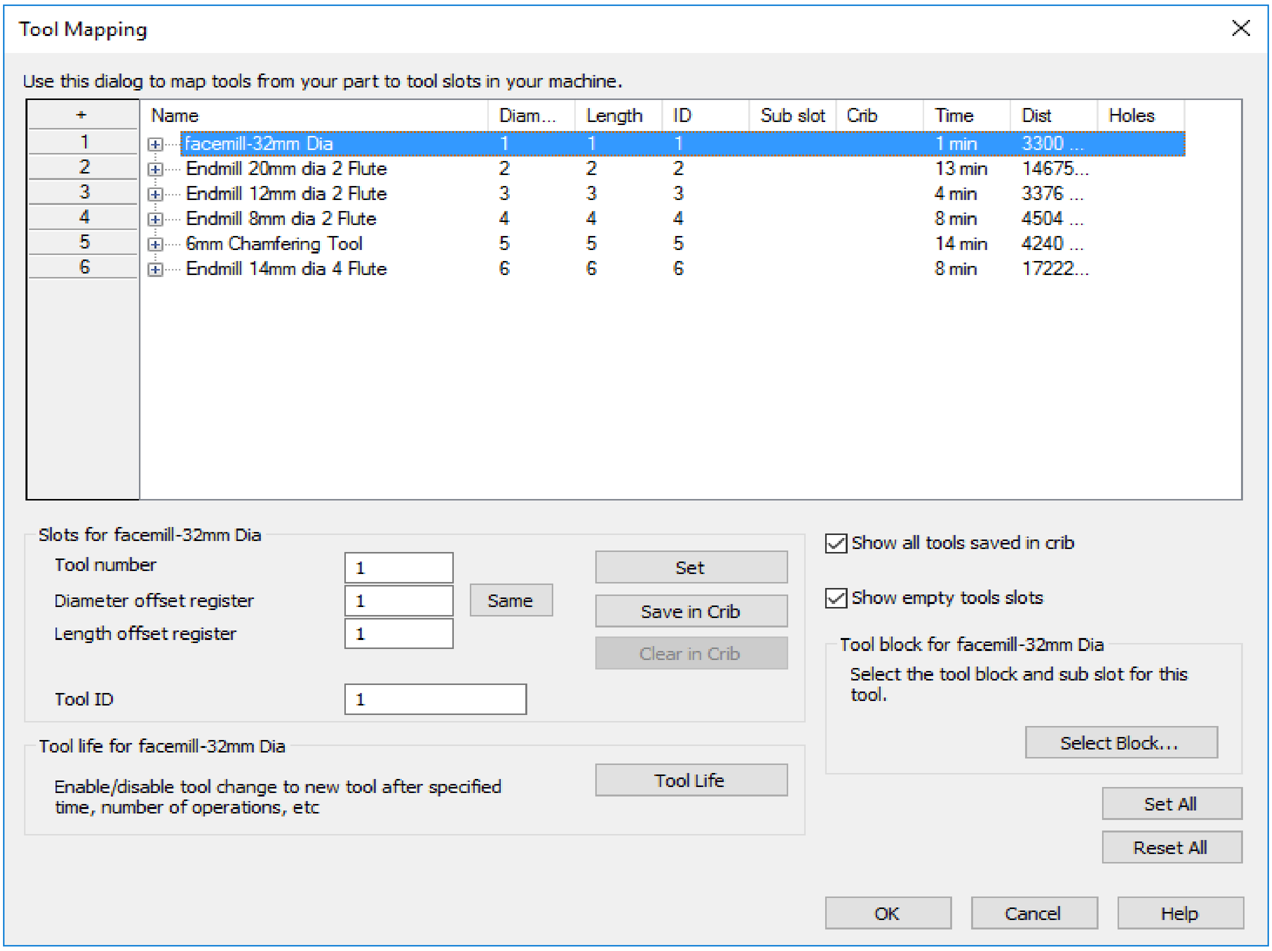




Tool Mapping

You can open the Tool Mapping dialog in one of these ways:

- Select **Manufacturing > Tool Mapping** from the menu.
- Click NC Code  in the Steps panel, then click Re-map the tools to new tool slots in the NC Code dialog.

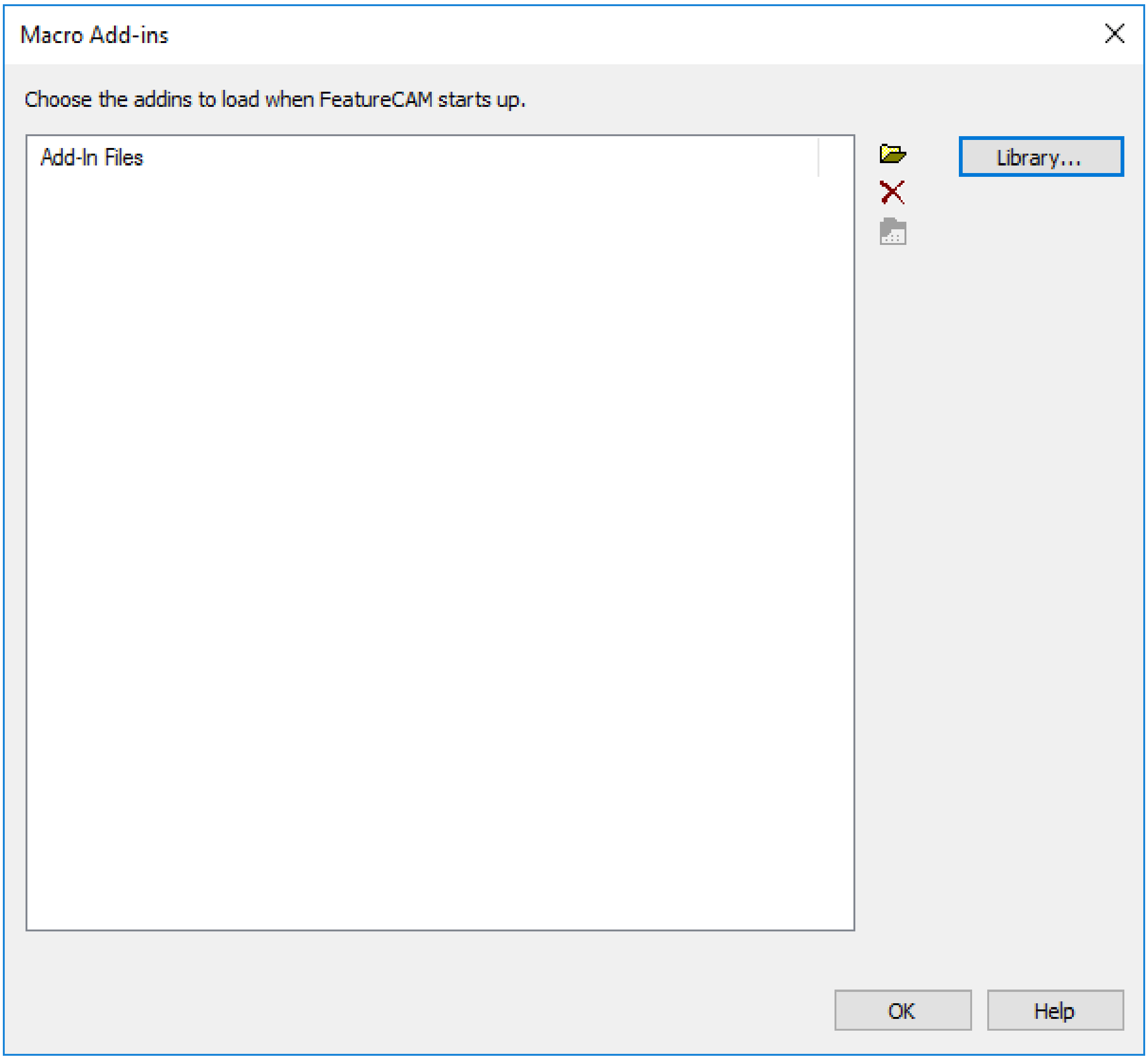


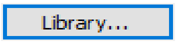
Custom Setup Sheet Add-In

- You can create custom setup sheets for milling, turning, and turn/mill parts using the **SetupSheet.dll** add-in.
- The add-in extracts information from the **Part Documentation** dialog and enables you to take images for each Setup.
- You can use setup sheets to give information to the machine operator about the manufacturing, tooling, and toolpaths of a part.

To load and run the add-in:

1 Select **Options > Add-Ins** from the menu. The **Macro Add-ins** dialog is displayed.

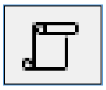


2 Click the 

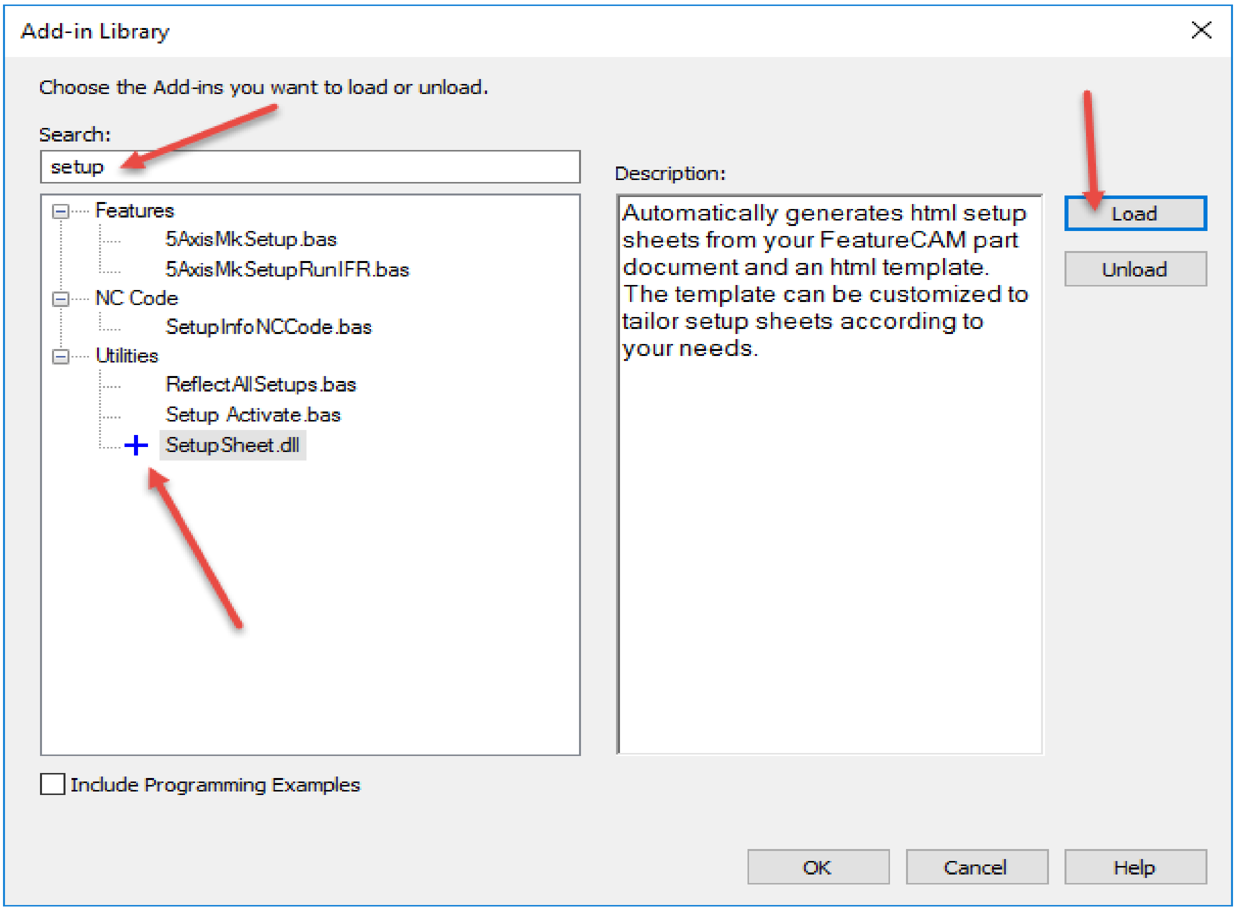
button and browse to the SetupSheet.dll file. If you installed FeatureCAM in the default location, the file is at C:\Program Files\Autodesk\FeatureCAM\Add ins\Setup Sheet\SetupSheet.dll.

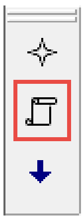
3 In the Macro Add-ins dialog, in the Add-In Files list, ensure that the check box to the left of the SetupSheet.dll file address is selected.

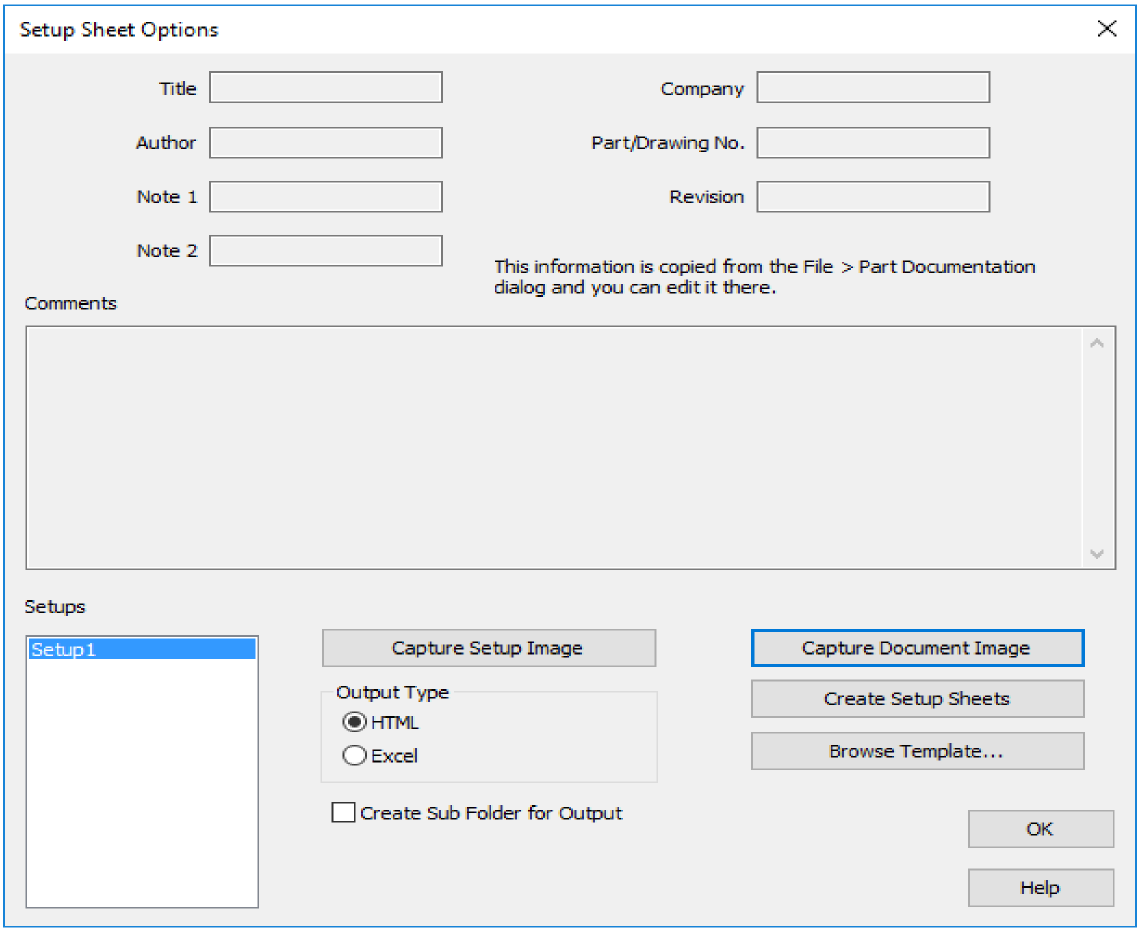
4 Click **OK**. The Utilities toolbar is displayed, containing the Setup Sheet button



5 Click the Setup Sheet button to run the add-in. The Setup Sheet Options dialog is displayed:







*Remember to set the Part Document information in **File>Part Documentation.***

- The **Title, Author, Note 1, Note 2, Company, Part/Drawing No., Revision, and Comments values** are copied from the Documentation tab of the Part Documentation dialog and you can edit them there.
- You can use this dialog to capture an image for each Setup in the current document, as well as an image to represent the whole document.

6 To capture an image for a Setup, select the Setup name in the **Setups** list and click the **Capture Setup Image** button to capture the current contents of the graphics window.

*First run a simulation only for the Setup you want to capture by deselecting other Setups in the **Part View**; then adjust the view to show a good orientation of the Setup, and open the **Setup Sheet Options** dialog to capture the image in the graphics window.*

You must use a template to create the setup sheets. Click the Browse Template button to find and set the template you want to use. There is a template, SetupSheetTemplate.html, in the Add ins\Setup Sheet folder.

7 To create the setup sheets, click the **Create Setup Sheets** button. The part is simulated to generate toolpaths and the setup sheet is displayed in your web browser. You can save the HTML file from your browser.

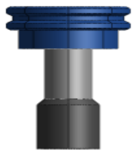
8 To open the setup sheet in the FeatureCAM Browser, select **File > Open** from the menu and browse to the setup sheet HTML file.

~Second Exercise.fm	FeatureCAM SetupSheet
Title / Company Author / Part/Drawing No. Note 1 / Revision Note 2	Comments Info

Stock Info

Type	Material	X	Y	Z
Block	ALUMINUM	315.470000	315.470000	33.000000

Tool List

Tool Number	Tool Name	Tool Diameter	Tool Exposed Length	Tool Length*	Tool Details	Tool Image
1	facemill-32mm Dia	32.0000	68.0000	- - -	Tool Name: facemill-32mm Dia Tool Material: CARBIDE Exposed Length: 68.0000 Tool Finish: BRIGHT Comment: - - - Holder: CAT 40, EndMill Maximum Depth: 1.0000 Maximum Setup Depth: 1.0000 Maximum Height: 0.0000 Coolant Override: No Override Tool Diameter: 32.0000 Effective Diameter: 0.0000 Number of Teeth: 3 Tip Radius: 0.0000 : 30.0000	
2	Endmill 20mm dia 2 Flute	20.0000	120.0000	42.0000	Tool Name: Endmill 20mm dia 2 Flute Tool Material: HSS Exposed Length: 120.0000 Tool Finish: BRIGHT Comment: - - - Holder: BT40 ER40X80 Maximum Depth: 27.0000 Maximum Setup Depth: 32.1000 Maximum Height: 0.0000 Coolant Override: No Override Tool Diameter: 20.0000 Shank Diameter: 19.8000 Flutes: 2 Taper: 0.0000 Length: 42.0000 Overall Length: 150.0000 End Radius: 0.0000	

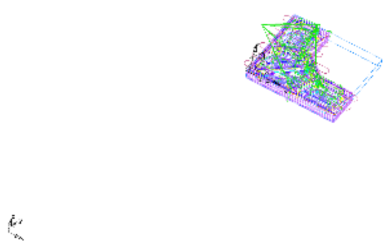
Setup : Setup1

Setup Origin: 5.0000, 5.0000, -1.0000

Feature Name	Feature Type	Operation	Tool Number	Tool Name	Operation Feed	Operation Speed	Operation Time	Operation Comment
face1	Face	Finish	1	facemill-32mm Dia	5715.0000	10000.0000	0:00:57.4	
pocket1	Pocket	Rough	2	Endmill 20mm dia 2 Flute	1261.2711	3153.1777	0:12:21.5	
pocket2	Pocket	Rough	2	Endmill 20mm dia 2 Flute	1261.2711	3153.1777	0:02:05.7	
pocket1	Pocket	Rough	3	Endmill 12mm dia 2 Flute	1261.2711	5255.2962	0:04:37.0	
pocket2	Pocket	Rough	3	Endmill 12mm dia 2 Flute	1261.2711	5255.2962	0:01:03.6	
pocket1	Pocket	Finish	4	Endmill 8mm dia 2 Flute	873.1877	9095.7050	0:05:36.4	
pocket2	Pocket	Finish	4	Endmill 8mm dia 2 Flute	873.1877	9095.7050	0:02:20.3	
pocket1	Pocket	Finish	5	6mm Chamfering Tool	330.0000	10000.0000	0:06:11.3	
pocket2	Pocket	Finish	5	6mm Chamfering Tool	330.0000	10000.0000	0:04:02.8	
boss1	Boss	Semi Finish	6	Endmill 14mm dia 4 Flute	2522.5421	4504.5396	0:03:57.8	
boss1	Boss	Finish	6	Endmill 14mm dia 4 Flute	2328.5005	6930.0610	0:03:59.9	
boss1	Boss	Finish	5	6mm Chamfering Tool	330.0000	10000.0000	0:04:02.7	

Total Setup Time: 00:51:16

~Second Exercise_Setup1.png



Procedure for machining a 2D part in from geometry

1. **Create** a new Part document in **FeatureCAM**.
2. **Select** the correct module that you will be working with.
3. **Cancel** the material stock.
4. **Import** a file, typical **2D** formats to import are **.dxf** or **.dwg** or **draw** your **Geometry** using **FeatureCAM** geometry construction tools.
5. If importing a file, use the **Wizard** to establish the Part orientation and **Setup1** position and **Stock Material**.
6. If you have drawn the geometry establish your **Material** around the part using the **Resize** button in the **Stock1 Properties.** Adding **depth** to the required level. Add extra **material** to the depth if you are holding the part in a vice. Change the position of the **Setup1** if required.
7. **Create** the **Curves** from your **Geometry** using **Closed Curve**. Remember to select **Pick pieces** for open profiles.
8. **Create Features** from the **Curves** by creating a **New Feature (Ctrl+R)**.
9. **Add tooling** if necessary to your **Crib** to machine your part.
10. **Re-order** your process if required in the **Operation list**. Selecting the **Automatic ordering options** for example select **Cut Higher operations first**.
11. **Run** the **3D Simulation** to prove out your machining at each stage of the process.
12. **Select** your **Post** and **Crib** and **Material type**. This can be done at any stage. If changed you will have to re-run the **3D Simulation** again.
13. Once you are happy with your machining process **save your part document**. The file extension will be an **.fm (FeatureCAM)** file.
14. **Select File>Save NC**. Save the file to a known directory so you can upload this to your CNC machine.

*You have now completed your part in **FeatureCAM**.*

Contents

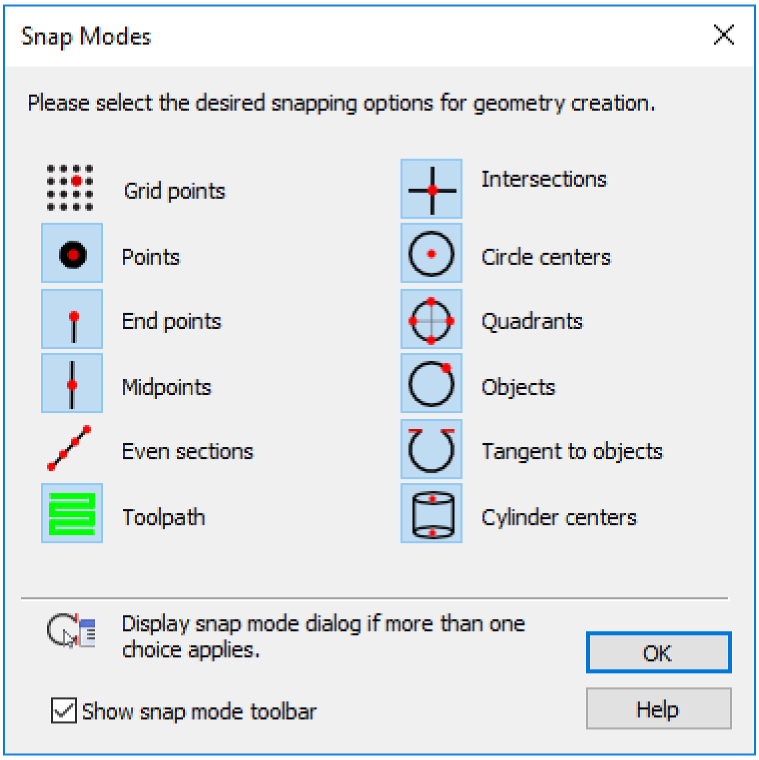
Geometry Creation Techniques

Introduction

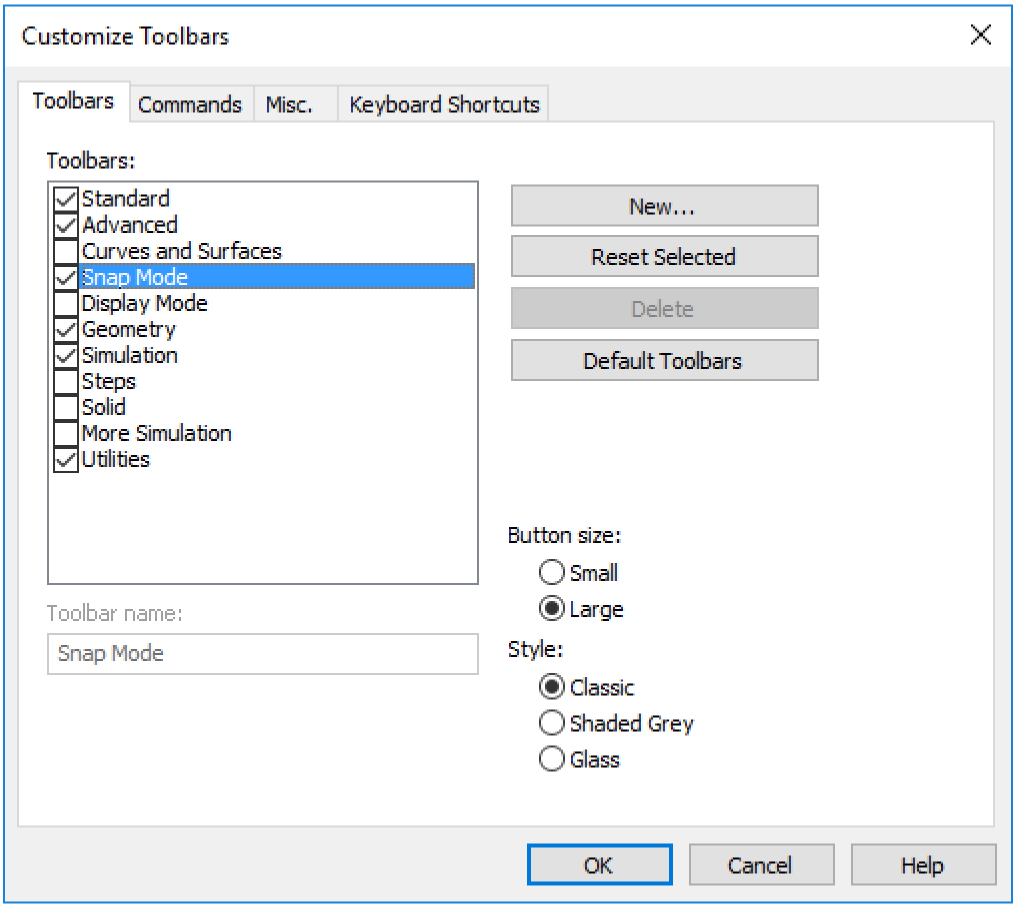
This module will provide an understanding of how different **Snapping Modes** work and how **Geometry** is created within **FeatureCAM**. By the end of the module the user will be able to construct **Geometry** and understand how each **Snapping Mode** functions.

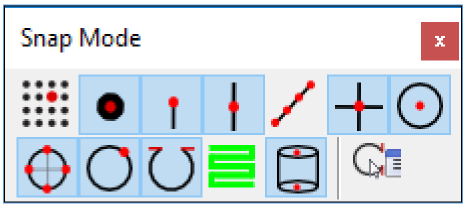
Snapping

- **Snapping Mode** commands can be accessed in two different areas of the user interface. Select the **Options** menu then **Snapping Modes**.

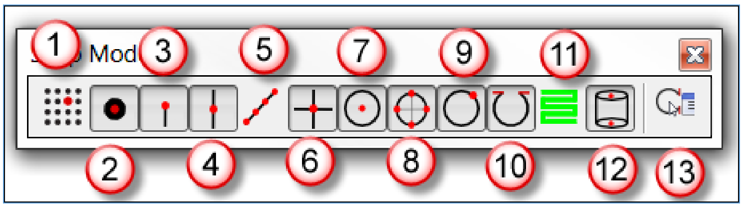


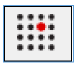
- **Snapping Modes** may also be accessed through the **Snap Mode Toolbar** (to show the **Snapping Toolbar**. Go to the **View** menu then **Toolbars,** then check the **Snap Mode** checkbox and select **OK**) (See image on the next page).

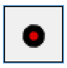




Looking at each icon from left to right we will give a short summary of each item's function:



1 The **Snap to Grid Points**  button displays a grid and enables snapping to the grid. Selecting snapping grids from the options menu controls the spacing of the grid. Grid snaps to a point on a coordinate system that is laid out on the **stock**. The grid size can be modified from the **Options** menu.

2 The **Snap to Point**  button snaps to a point object.

3 The **Snap to Endpoint** Snaps to the ends of lines or arcs. **Endpoint** also applies to the corners of the stock and verticals of a Solid.

4 **Snap to Midpoint** snaps to the middle points of lines and arcs.

5 **Snap to Section** snaps equal intervals of a finite line. The number of sections is controlled by the Sections parameter of the **Snapping Grids** dialog box.

6 **Snap to Intersection** snaps to the intersection of lines, arcs and circles.

7 **Snap to Centre** snaps to arc and circle centres. This setting also controls the display of circle and arc centre points.

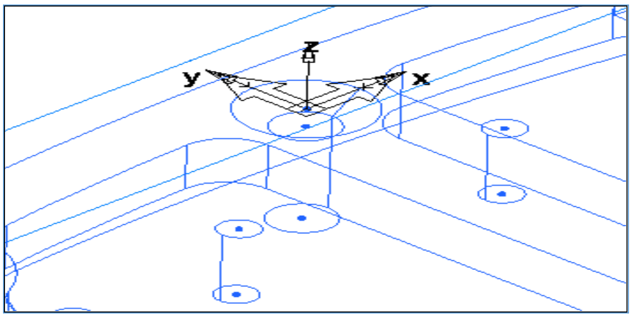
8 **Snap to Quadrant** snaps to the four points on a circle corresponding to 0°, 90°, 180° and 270°.

9 **Snap to object** snaps to a point on another object. This includes lines of surfaces and is convenient for snapping points to locations on a surface or Solid model.

10 **Snap to Tangent** snaps the point so that the object you are creating will be tangent to the object you snapped to.

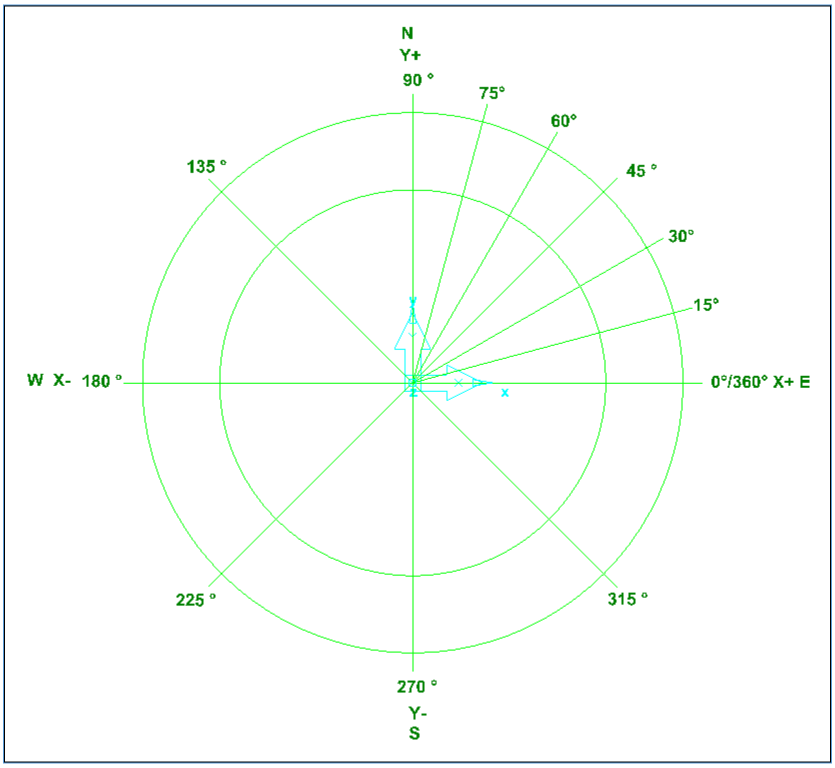
11 **Snap to Toolpath** snaps to toolpath lines.

12 **Snap to Cylinder** snaps to Cylinder centres, top and bottom points.



13 The **Snap Discrimination** dialog button displays the Snap Discrimination dialog box whenever there is an ambiguous pick. This dialog box will list the possible points to snap to and ask you to pick the one you want.

Drawing conventions when using Autodesk products.

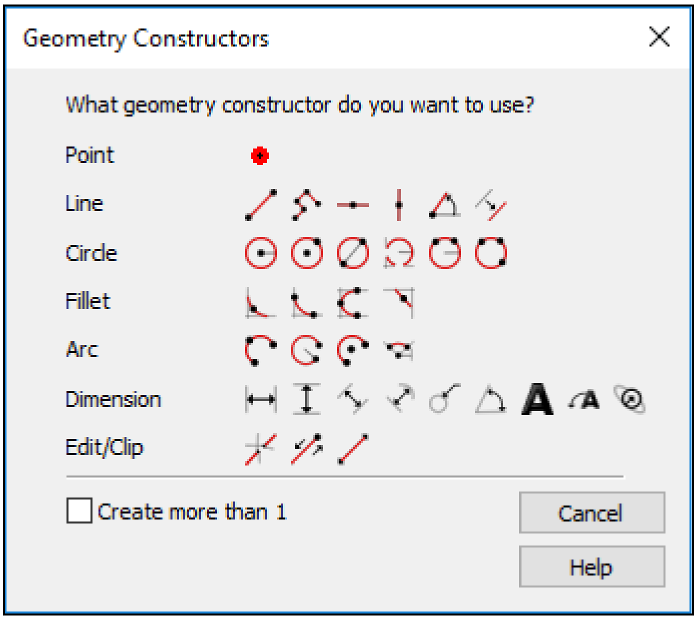


- When you start using **Autodesk** products and start to construct Geometry, the illustration above clearly shows you the directions of the cardinal drawing angles and positive and negative directions. This should help when creating the exercises in this chapter.

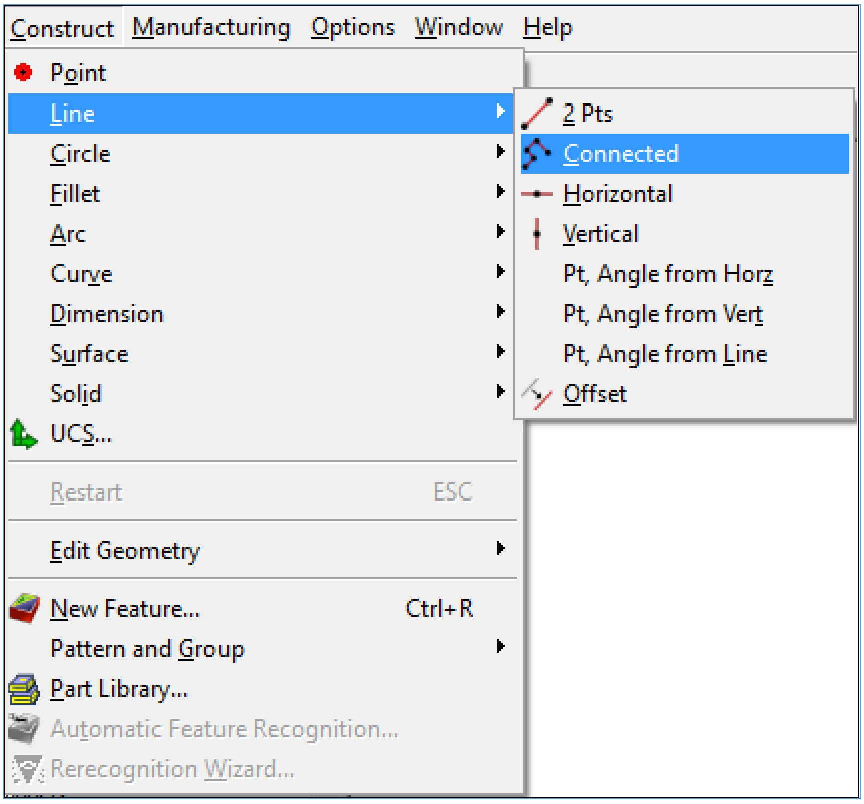
User Interface

Geometry constructor commands can be accessed in three different areas of the user interface.

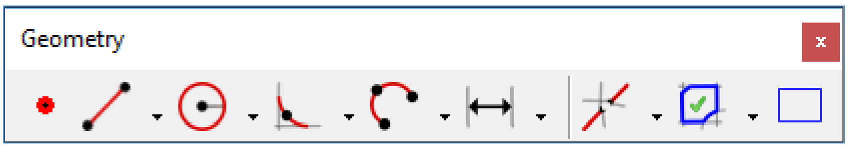
Commands can be accessed by clicking the **Geometry** 2. Geometry icon in **Steps**:



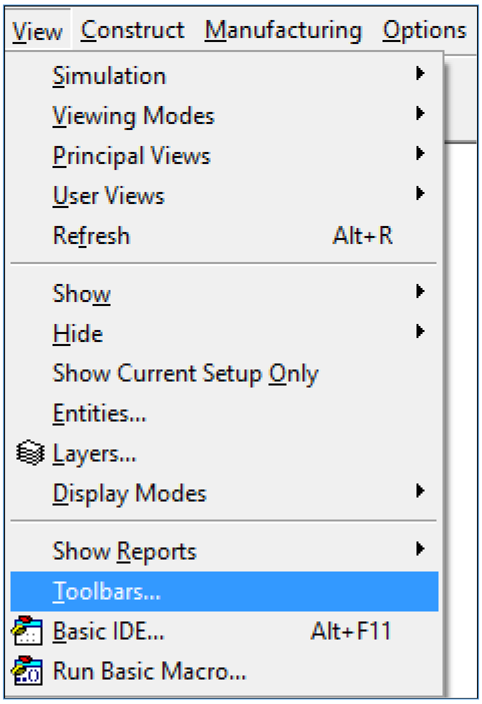
- This can also be accessed through the **Construct** Menu.

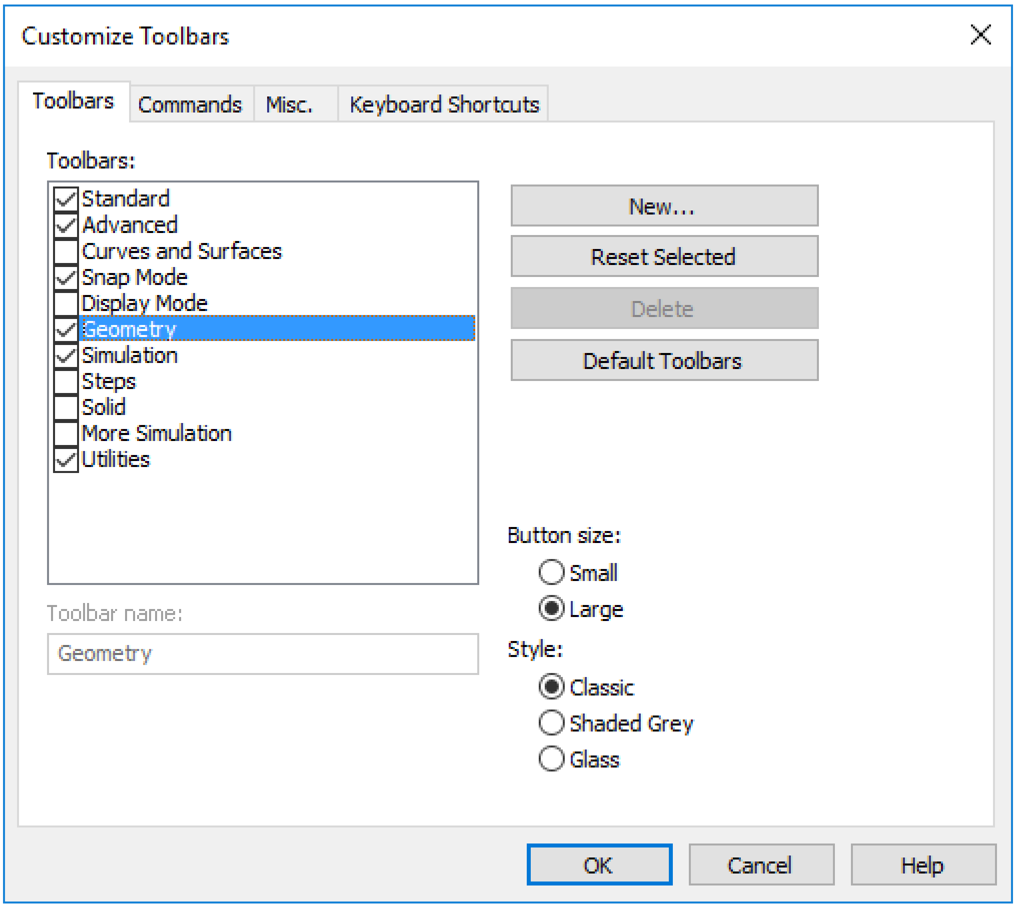


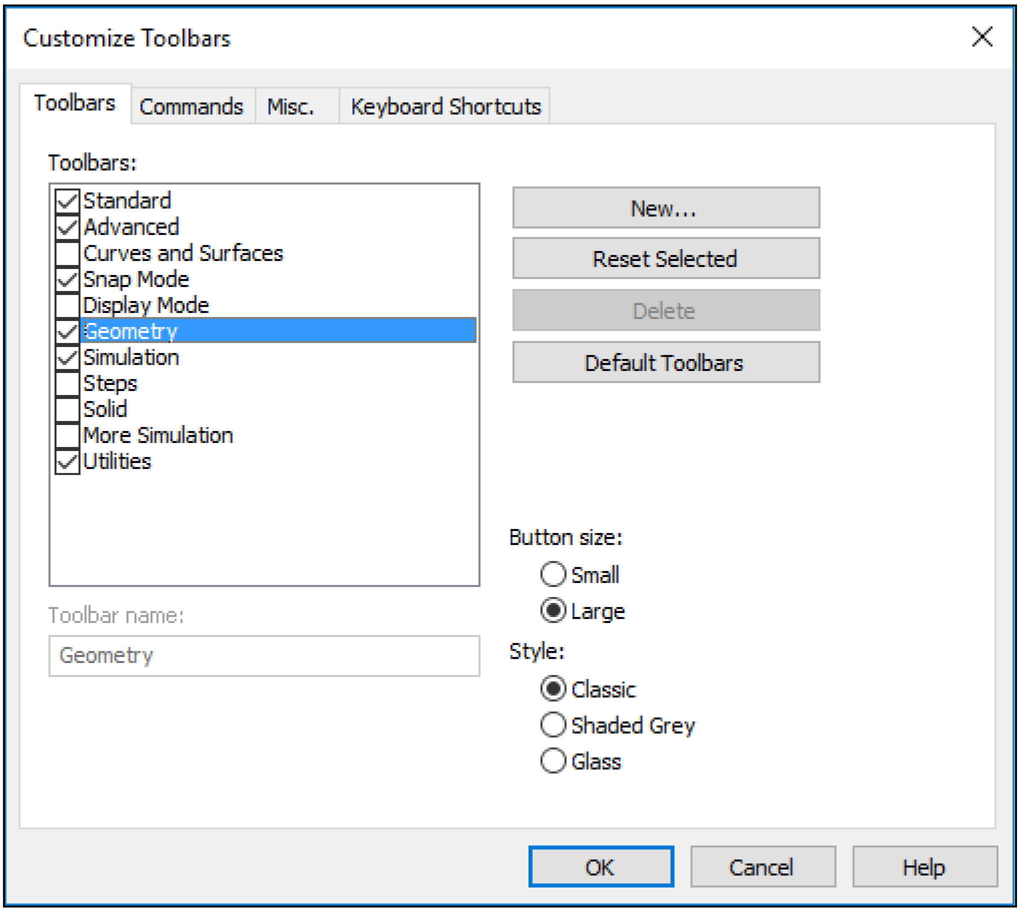
*Or may be accessed by the **Geometry Toolbar***



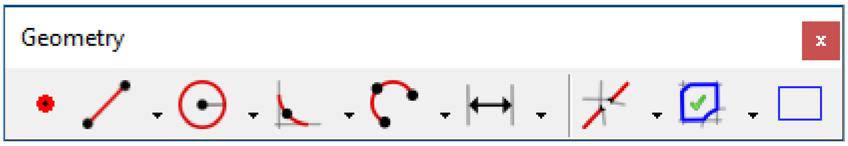
*To show the geometry toolbar click on **View**>**Toolbars,** then check the **Geometry checkbox** and press **OK***

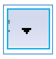

View
Construct
Manufacturing
Options
Simulation
Viewing Modes
Principal Views
User Views
Refresh Alt+R
Show
Hide
Show Current Setup Only
Entities...
Layers...
Display Modes
Show Reports
Toolbars...
Basic IDE... Alt+F11
Run Basic Macro...


Customize Toolbars
Toolbars
Commands
Misc.
Keyboard Shortcuts
Toolbars:
Standard
Advanced
Curves and Surfaces
Snap Mode
Display Mode
Geometry
Simulation
Steps
Solid
More Simulation
Utilities
New...
Reset Selected
Delete
Default Toolbars
Button size:
Small
Large
Style:
Classic
Shaded Grey
Glass
Toolbar name:
Geometry
OK
Cancel
Help



The geometry toolbar uses flyout menus. The last item previously selected is displayed as an icon in the tool bar.

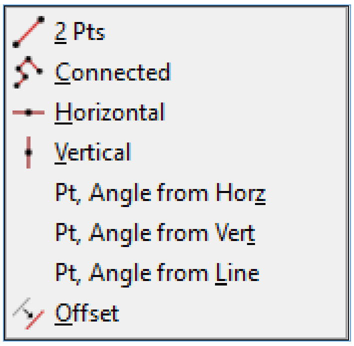


- Click on the **triangle**  to reveal the flyout menu.

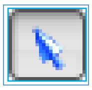
*These flyout menus are used throughout **FeatureCAM** and are useful for easy access to many program functions.*

- The following Menus show the Geometry creation options available.

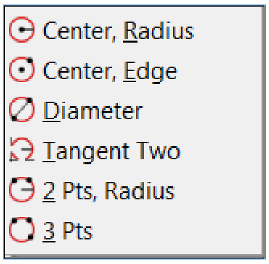
Line



- **2 Pts: -** Line created with two endpoints.
- **Connected: -** Multiple lines created in succession. Endpoint of one line becomes start point of next line.
- **Horizontal: -** Infinite horizontal line created through a point.
- **Vertical: -** Infinite vertical line created through a point.
- **Pt. Angle: -** Infinite line created through a point at an angle specified in degrees.
- **Offset: -** Line, Circle or Arc created offset from another. New lines have the same length as original

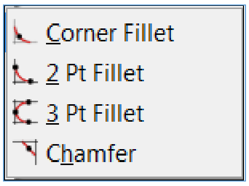
- Click on the Select  icon at the top of **FeatureCAM** on the Standard Toolbar to cancel the line drawing. Use this button at any time to exit selection mode. Or press the **Esc** key at any time.

Circle



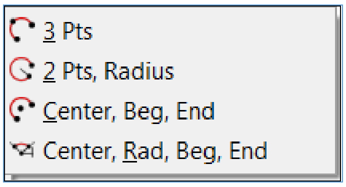
- **Centre, Radius:-** Creates a circle by clicking on the centre and specifying the radius or dragging the mouse to a radius
- **Centre, Edge:-** Creates a circle by clicking once on the centre then again on the edge
- **Diameter:-** Creates a circle with two points with the diameter being the distance between the two points
- **Tangent Two:-** Circle created by snapping the circle's edge tangent to two objects
- **2 Pts, Radius:-** Creates a circle by specifying two points and a radius
- **3 Pts:-** Creates a circle from three points

Fillet



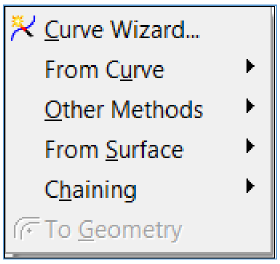
- **Corner Fillet: -** Creates a Fillet in a corner originally defined by an intersection of lines or arcs.
- **Two point Fillet: -** Creates a Fillet in a corner originally defined by an intersection of lines or arcs.
- **Three points: -** Defines a fillet by selecting three points, similar to the three point circle.
- **Chamfer: -** Creates a Chamfer in a corner originally defined by an intersection of lines or arcs.

Arc



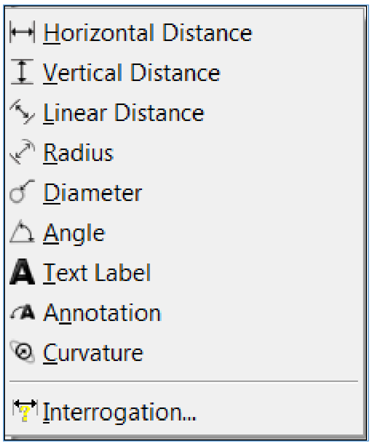
- **Three points: -** Constructs an arc through a start point, edge point, and a finish point.
- **Two points, radius: -** Constructs an arc through two points with a specific radius.
- **Centre, Beg, End: -** Constructs an arc from a centre point and given beginning and end points.
- **Centre, Rad, Beg, End: -** This selection constructs an arc with a specific centre and radius with the starting and ending points determined by angles.

Curve



- **Curves** are paths in **2D** or **3D** space. You can create them in **FeatureCAM** or Import them from a CAD system. Curves can be open, or closed. Open curves have end points that do not meet. You can use open curves in the following Features Side, Groove, Rounds and Chamfer Features.

Dimension



- Horizontal: - creates dimension information based on the horizontal axis of the part.
- Vertical: - creates dimension information based on the vertical axis of the part.
- Linear: - creates dimension information based on the absolute distance between two points regardless of the angle of the dimensioned space.
- Radius: - creates dimension information for the radius of the selected object.

  Diameter: - creates dimension information for the diameter of the selected object.

  Angle: - creates dimension information for the angle between two selected lines.

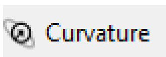
  Text label: - creates a text label entered in the dialogue bar

  Annotation: - places explanatory text, entered in the dialogue with an arrow of what is being explained

  Curvature: - samples the surface and computes the curvature in two directions to describe how the surface behaves at the point shown in the dialog bar. Unlike the other dimensioning tools, Curvature is a real-time rubber-banding effect where you traverse the surface to find the point with the smallest curvature radius. Knowing the smallest radius tells you the smallest tool to use to manufacture the surface.

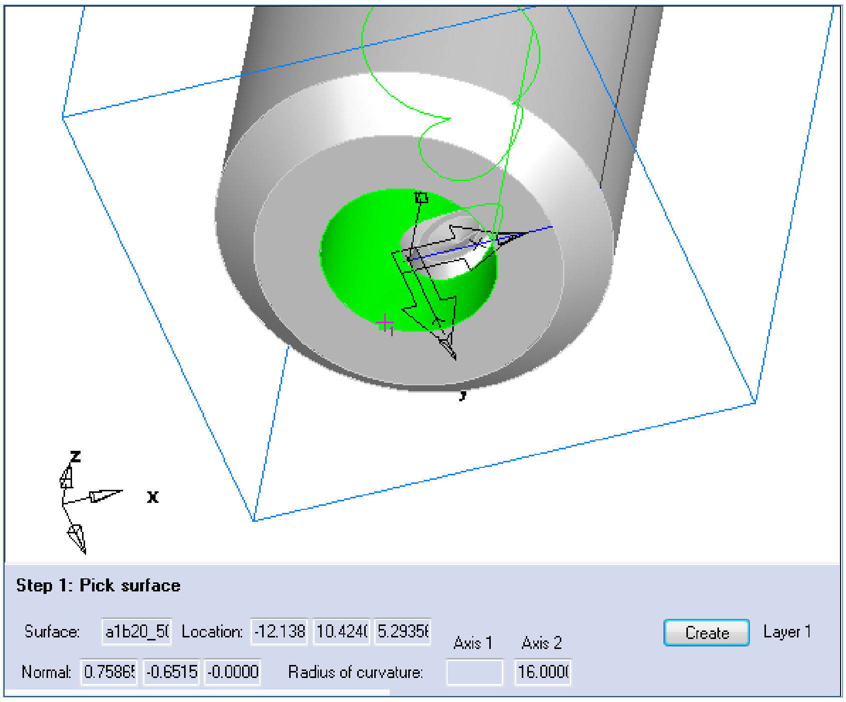
  Interrogation: - This dialog helps you extract numbers from the graphics window using snap modes and pick filters. You can then cut and paste these values into other dialogs.

Curvature

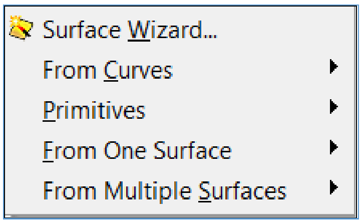


- Curvature samples the surface/Solid and computes the Curvature in two directions to describe how the surface behaves at the point shown in the dialog bar. Unlike the other dimensioning tools, Curvature is a real-time rubber-banding effect where you traverse the surface to find the point with the smallest Curvature radius. Knowing the smallest radius tells you the smallest tool to use to manufacture the surface.
- Select **Construct** > **Dimension** > **Curvature** from the menu.

- Move the cursor over the surface, particularly in the tight constrained areas of the surface or joints.
- Note the smallest value shown for **Curvature**. That is the smallest tool end radius you need to accurately machine the surface.
- Set up rough and finish passes for the surface feature based on this knowledge, and make sure the tool is available for production.

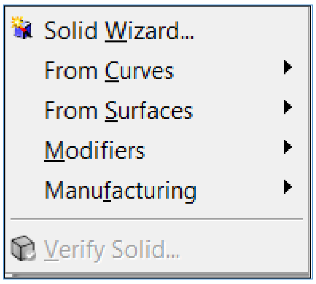


Surface



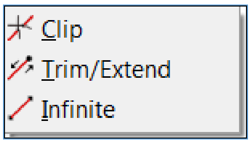
- To build and use **3D surfaces** in **FeatureCAM**, you need to understand how **FeatureCAM** defines a surface. A surface is defined by irregular sets of points and a description of how the surface behaves at each point on the surface which determines the smoothness of the surface. The set of points determines the shape, of the surface.

Solid



- The **FeatureCAM** support of solids lets you work with imported CAD designs from solid modelling systems and create 3D solid models using **FeatureCAM's** Solid Modelling Tools. Solids are a convenient representation for 3D parts because they group collections of surfaces into 3D volumes. By working with a solid instead of all of the individual surfaces, you are provided with a more convenient representation and more powerful modelling tools.
- After you have created your Solid Model you can then use **AFR** (Automatic Feature Recognition) or **IFR** (Interactive Feature Recognition) to create your toolpaths.

Edit Geometry



- **Clip**: - Clip removes a region of a line, arc, circle or curve. A region is defined as a portion of an object between two intersection points.
- **Trim/Extend:** - Changes the length of a line or an arc. Trim/extend can be used to lengthen or shorten lines and arcs as follows:
- **Infinite**: - Changes the length of a line or an arc.

Geometry Creation Exercise 1 (Class Exercise)

Introduction

*The exercise that follows will review most of the **Snapping Modes** on the **Snapping Toolbar** in no particular order, when complete it will provide a basic overview of the most commonly used snapping mode functions.*

1 Please **start** a **New** Part document in **FeatureCAM.**

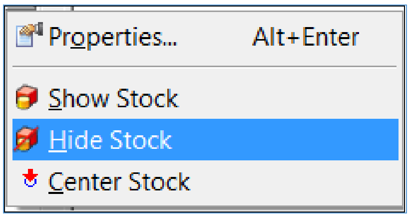
2 Select the **FeatureCAM** Icon on your **desktop** or **Taskbar**.

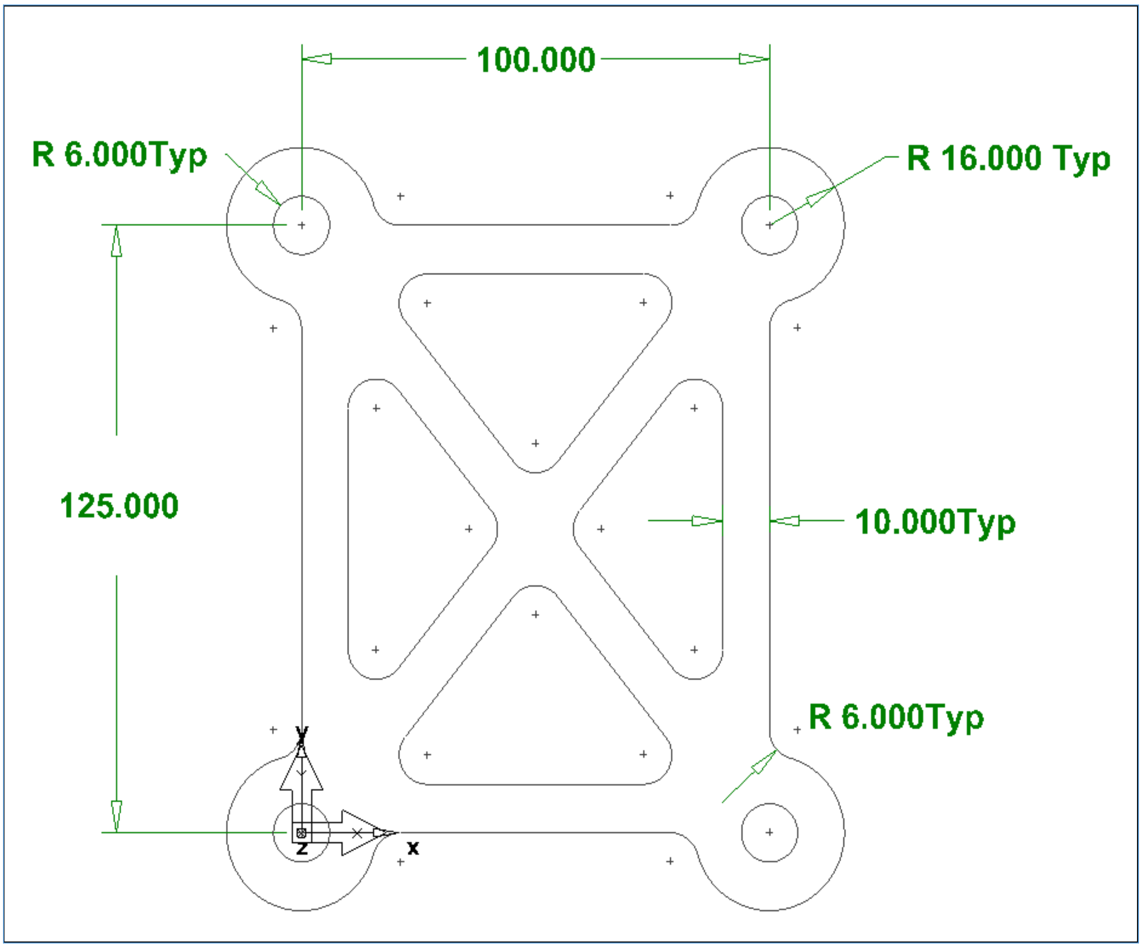
3 Select **New File** then **Milling Setup** and select **Millimeters**.

Typ** means Typical for all dimensions that are the same unless otherwise stated. **If in doubt ask

Do not worry about the stock size at this stage as we will resize this later. We are concentrating on Geometry creation techniques only.

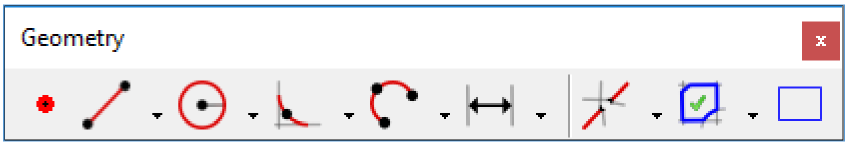
4 **Hide** the **Stock**, Right Click on the **Stock boundary** and select **Hide Stock.**





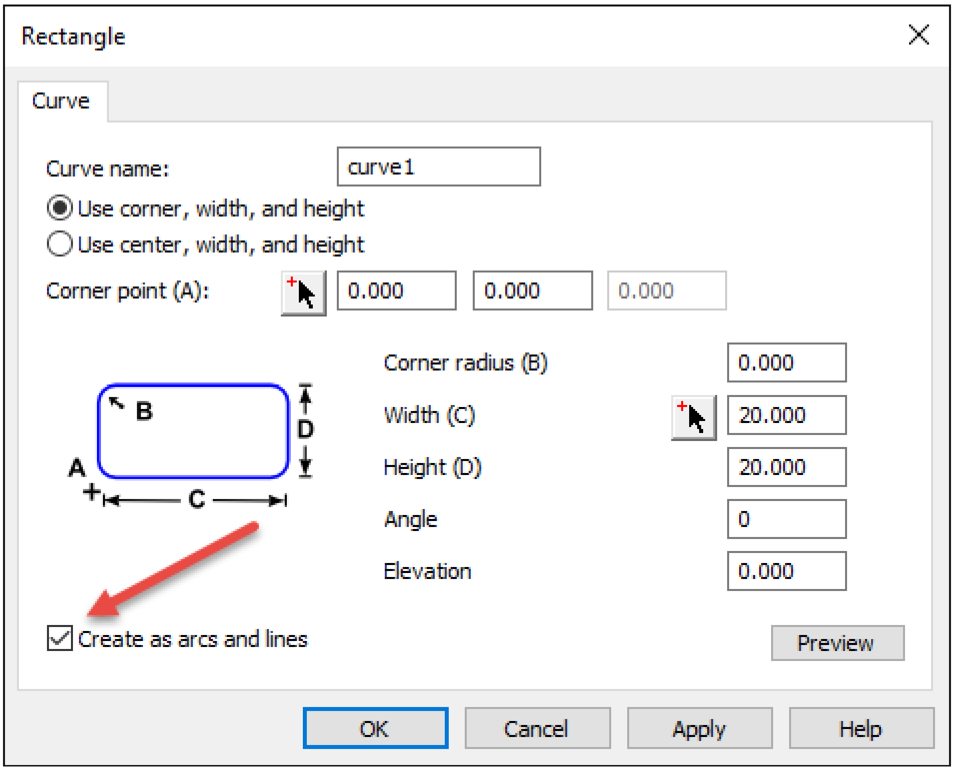
*We will create a **Rectangle** first.*

5 Go to **Construct>Curve>Other Methods>Rectangle.**



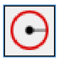
*Enter the following values **Width 100mm (c) Height 125mm (D)***

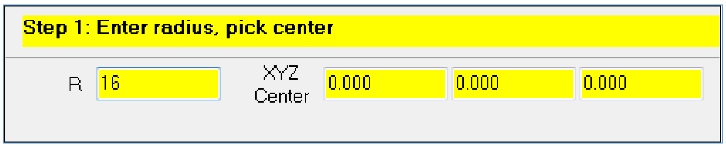
***Remember** to select **Create as arcs and Lines.** See image on the next page.*



This exercise will go through the most commonly used snapping modes to create the part.

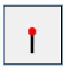
We will now create four Circles by snapping a single circle to each corner of the Rectangle.

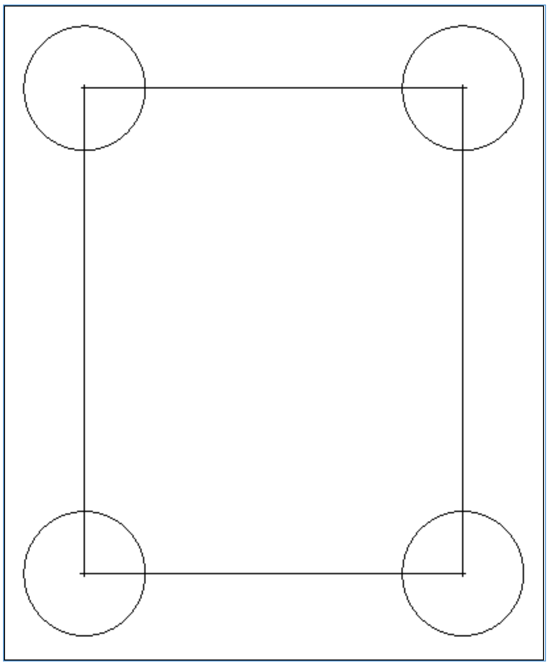
6 Select **Construct** then **Circle**, **Circle, Radius**  or select the same Icon from the **Geometry Toolbar**. Enter the following values **16mm.**



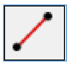
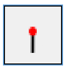
Metric

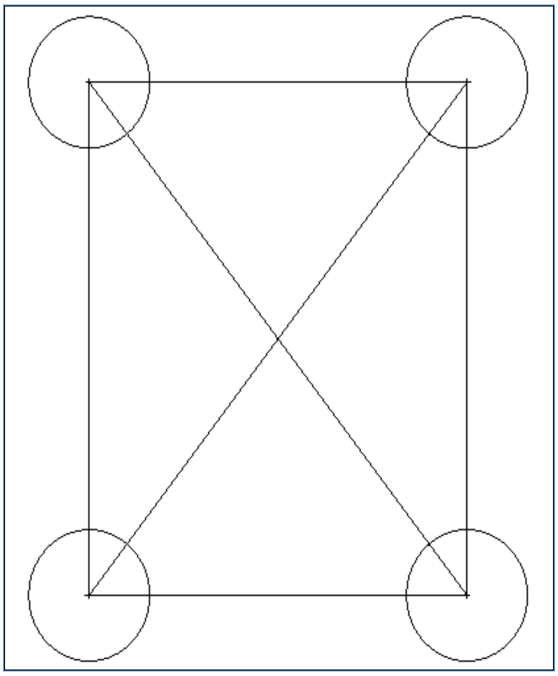
Please note that all Snapping mode functions can be switched on and off at any time. All of the snapping modes can be switched on together or used individually.

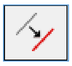
7 We will use **Snap to End Point**  as the most efficient snap mode in this instance. **Remember to snap to the four corners of the rectangle**. Your image should look like the one on the next page.

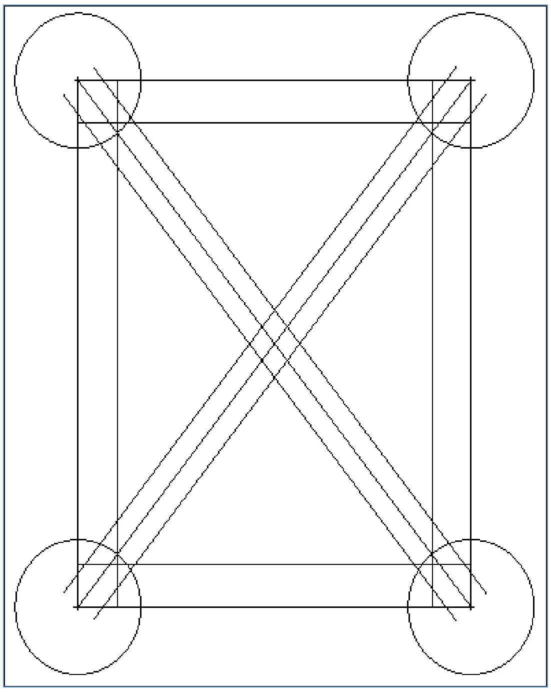


8 We will now draw two lines across the diagonally opposite corners.

9 Select **Line from 2 Points**  from the **Geometry** Toolbar. Use **Snap to End Point**  as the most efficient snap mode in this instance.

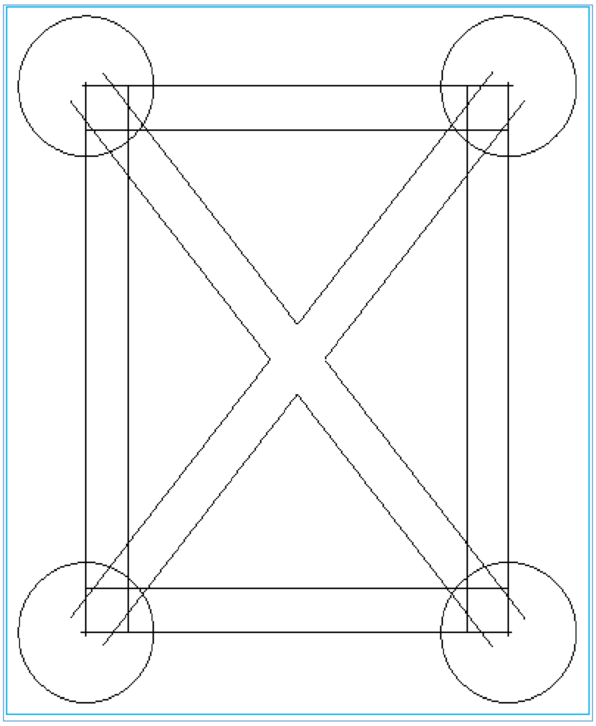


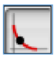
10 Using  Offset we will offset the outer lines only **10mm** towards the centre. Then whilst the Offset command is still active. Offset the diagonal lines **5mm** either side of the lines as shown below.

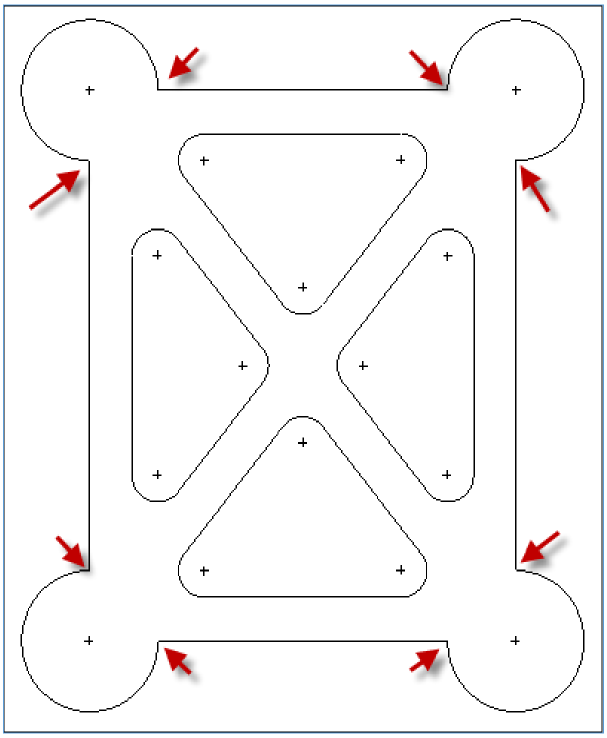


11 **Delete** the **2 centre diagonal lines**. To do this just select the lines and then

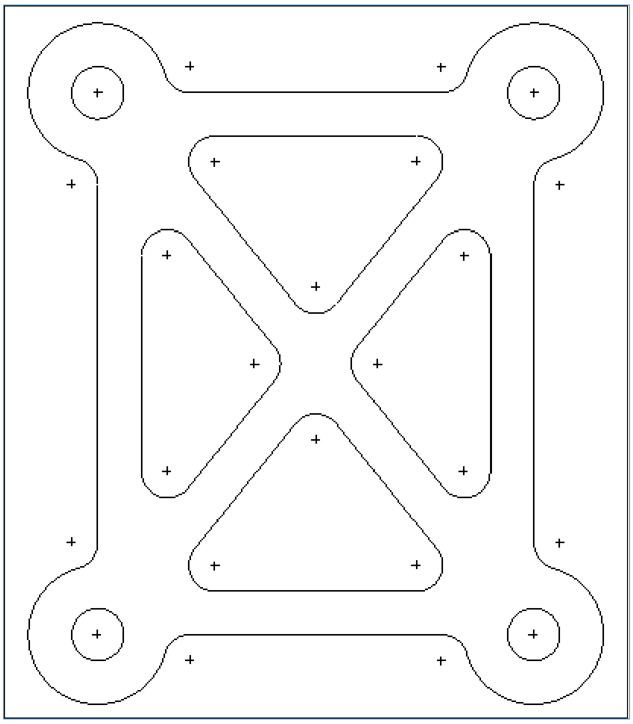
12 Select **Delete** on your **Keyboard**. Then  **Clip** the **middle diamond shape**. This will allow you to Fillet the four pockets. The Fillet command will automatically **Clip** the profile. An image up to this stage is shown below.



13 We will now **Fillet** all pocket corners **6mm** Select the **Corner Fillet**  Icon from the **Geometry** Toolbar and enter the values above. Present the cursor to each corner of the pockets until all corners are filleted. **Clip** the inner 4 corner radius and the four corners. Press **Esc** to finish the command. See image up to this stage below.



14 Fillet the corners **6mm** shown above. Draw 4 circles in the corners of the part. Circle radius is **6mm.** Snap to the circle centres**.** The finished image is shown below



 ***If in doubt ask.***

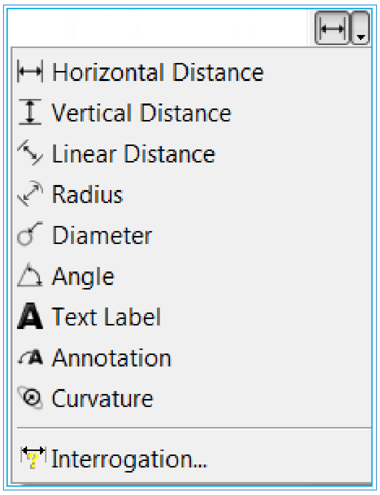
Dimensioning the part (Class Exercise)

We will change the Setup when we come to machine this later.

Next Select **Dimension** from the **Construct** menu, or the dimensioning flyout menu from the geometry toolbar. We will have a go at the different commands for example, **Horizontal Distance**, **Vertical Distance** etc.

The following text and images are a summary of what each item's function is on the Dimensioning flyout menu.

Typ means Typical for all dimensions that are the same unless otherwise stated. **If in doubt ask**.



Horizontal

Creates dimension information based on the horizontal axis of the part.

Vertical

Creates dimension information based on the vertical axis of the part.

Linear

Creates dimension information based on the absolute distance between Two points regardless of the angle of the dimensioned space.

Radius

Creates dimension information for the radius of the selected object.

Diameter

Creates dimension information for the diameter of the selected object.

Angle

Creates dimension information for the angle between two selected lines.

Text label

Creates a text label entered in the dialogue bar.

Annotation

Places explanatory text, entered in the dialogue with an arrow of what is Being explained.

Curvature

samples the surface and computes the curvature in two directions to Describe how the surface behaves at the point shown in the dialog bar. Unlike the other dimensioning tools, Curvature is a real-time rubber-banding effect where you traverse the surface to find the point with the Smallest curvature radius. Knowing the smallest radius tells you the Smallest tool to use to manufacture the surface.

Interrogation

This dialog helps you extract numbers from the graphics window using snap Modes and pick filters. You can then cut and paste these values into other dialogs

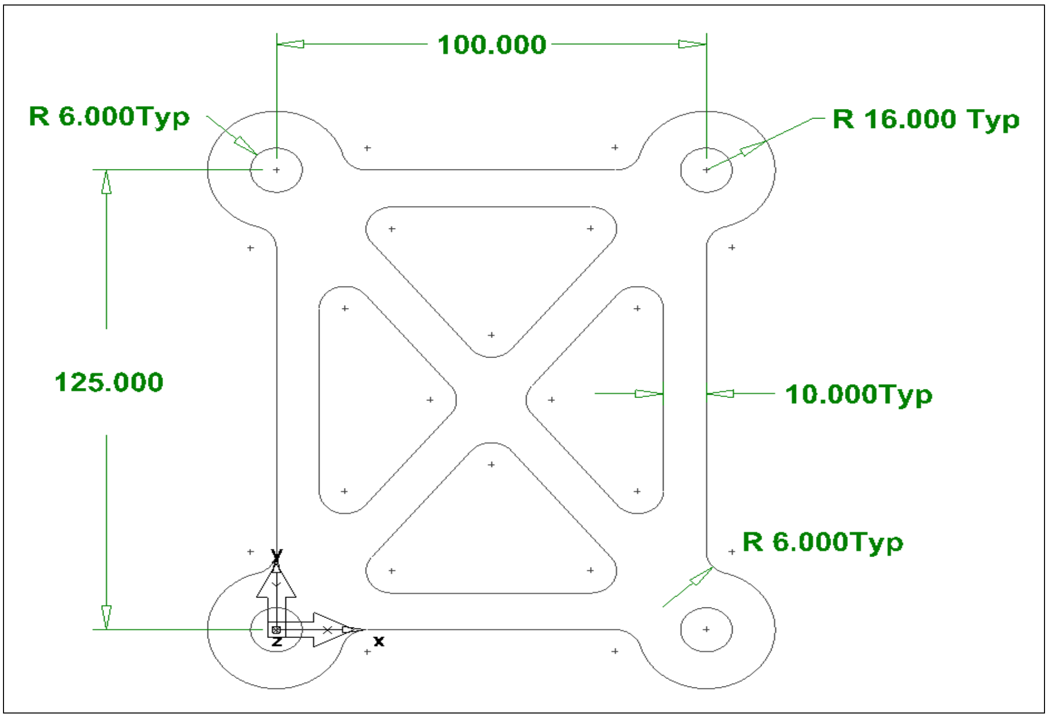
When creating dimensions. Do not worry if you make a mistake.

*Select undo or select **Ctrl + Z** then have another try.*

*Remember to **save** the part. We will machine this later on in the course.*

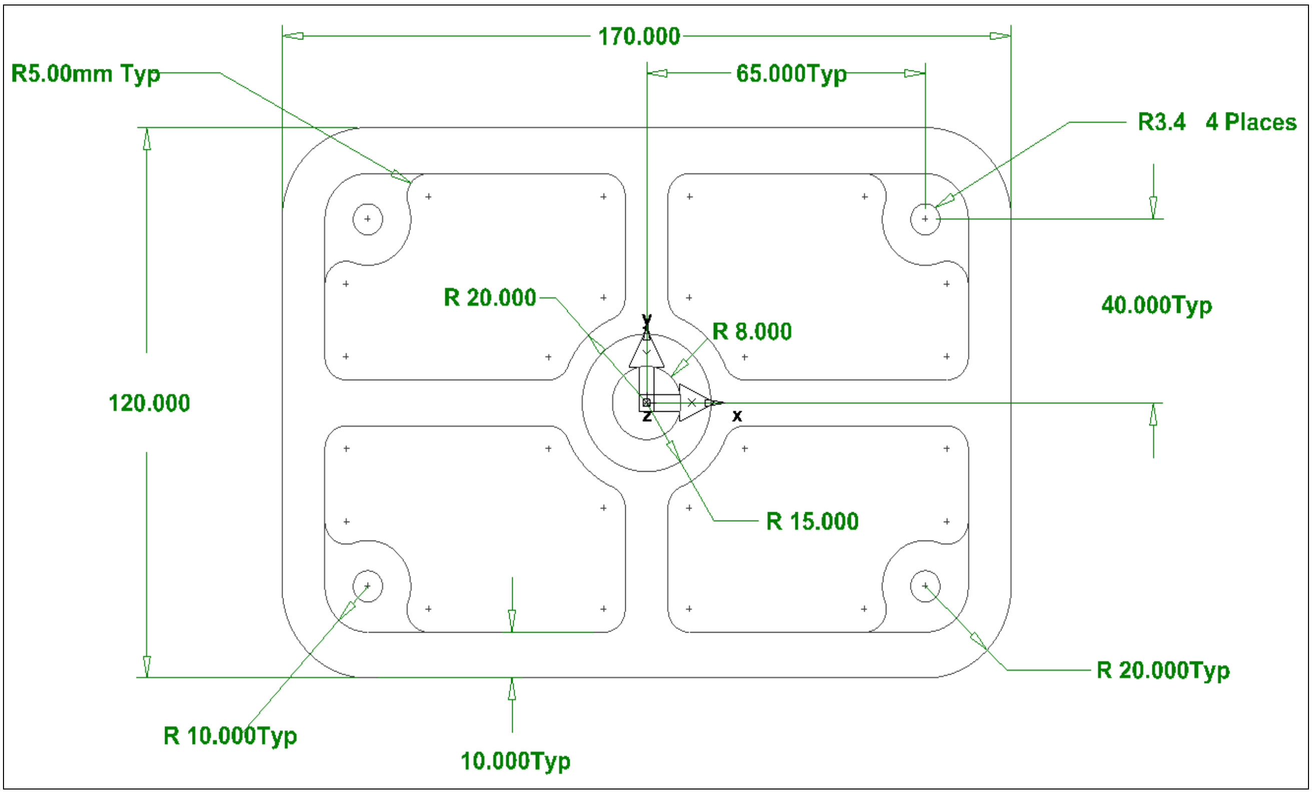
*Save the file as **"Construction Ex1"** in the Course Work Folder.*

C:\Training_Data\FeatureCAM Course Data 2017\Course work



*If you want to machine this part later on the thickness=**12mm.** Pockets are **6mm** deep and all holes are through.*

Construction Exercise 2 (Class Exercise)



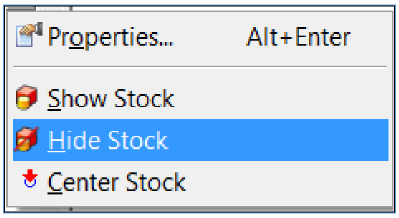
Please note that there is more than one way to create this example. For example you may decide to draw one quarter of the geometry and then reflect this about X0, Y0, this is the way we will create this example.

We will be creating this as a class exercise to give you the necessary skills to take back to your work place. The following pages give a step by step guide to creating this component.

1 Open a new Milling document and choose Millimeters.

2 Select Cancel to accept default Stock settings and then

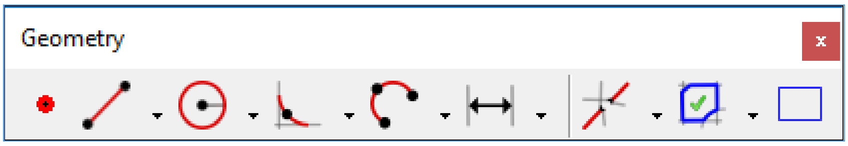
3 **Hide Stock**. In **PartView** select Stock by using the left Mouse button. Then Right click Stock again and then choose **Hide Stock**.



4 All that should be on the screen is **Setup1**. We will define the **Stock** when we have finished the geometry creation.

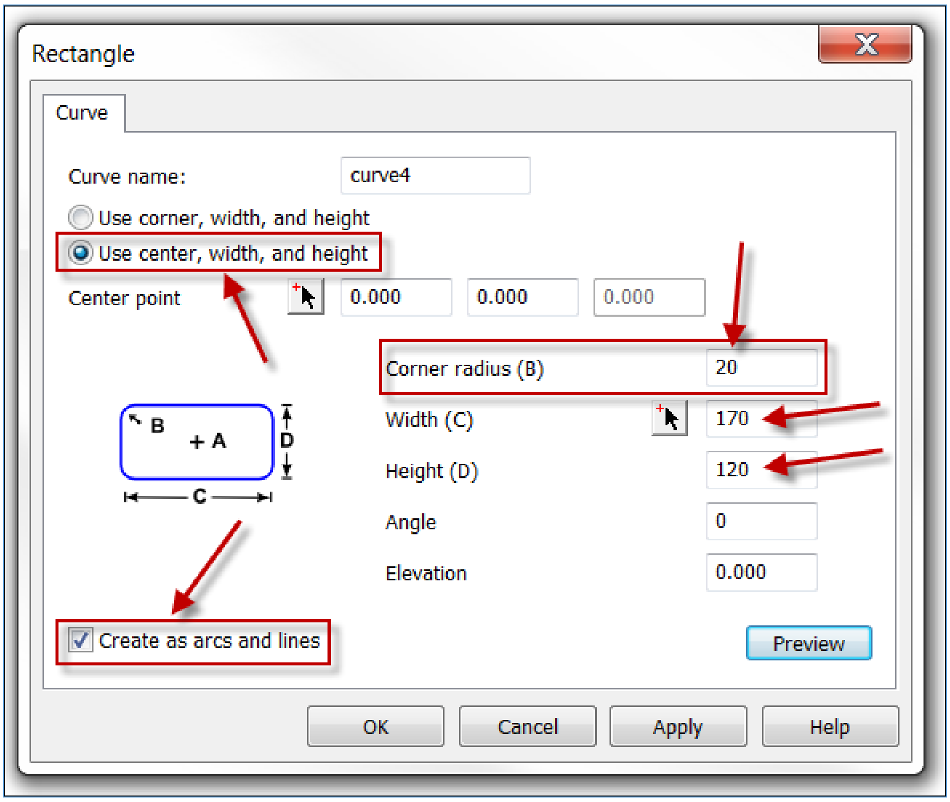
Drawing Stages

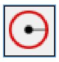
5 Construct a **Rectangle** using the **Construct**>**Curve**>**Other methods** >**Rectangle** Menu.

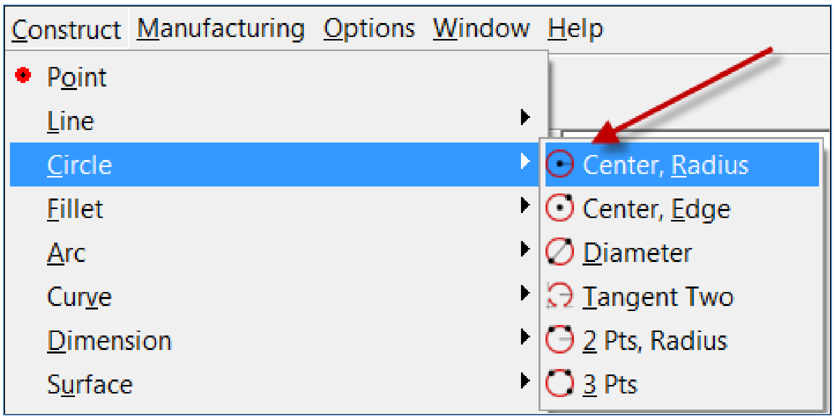


6 **Use Centre, Width and Height** with the Centre Point of **X0,Y0,Z0.** Rectangle. Width **170mm** Height **120mm** Radius **20mm.**

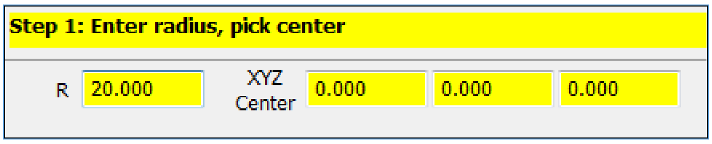

*Please Note: - Remember to select **create as arcs and Lines**.*



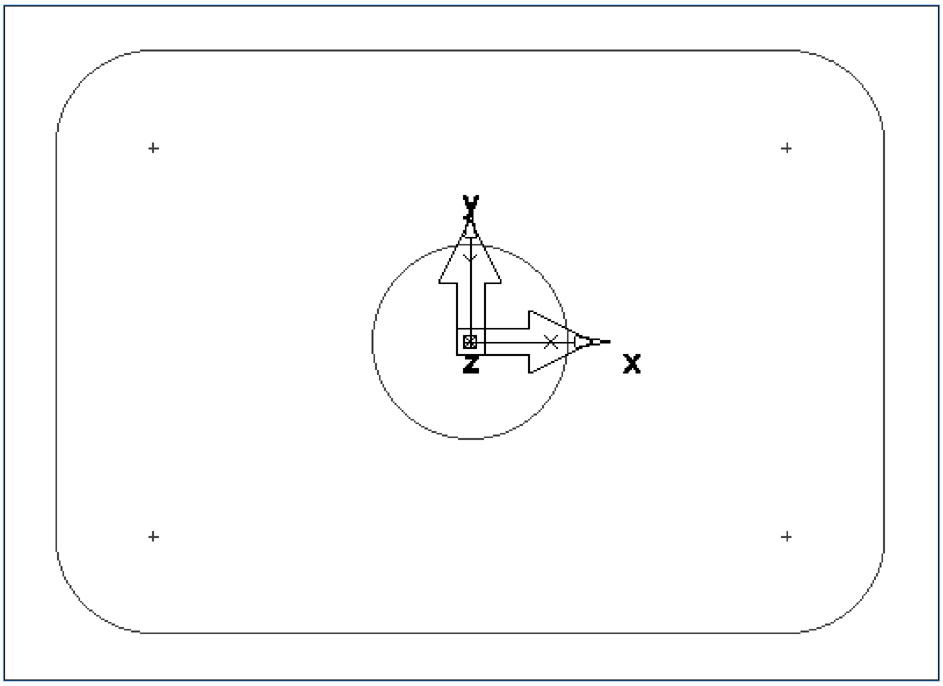
7 We will now construct a **Circle** using **Centre, Radius** 



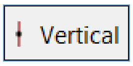
8 Type in the following **Radius** value **20mm** at **X0, Y0, Z0.** Select **Enter.**



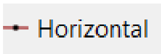
*Remember to select **Esc** to cancel the construction mode.*



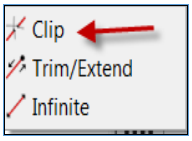
9 We will now create a Vertical and Horizontal line at X0, Y0, Z0.

From the Line menu select 

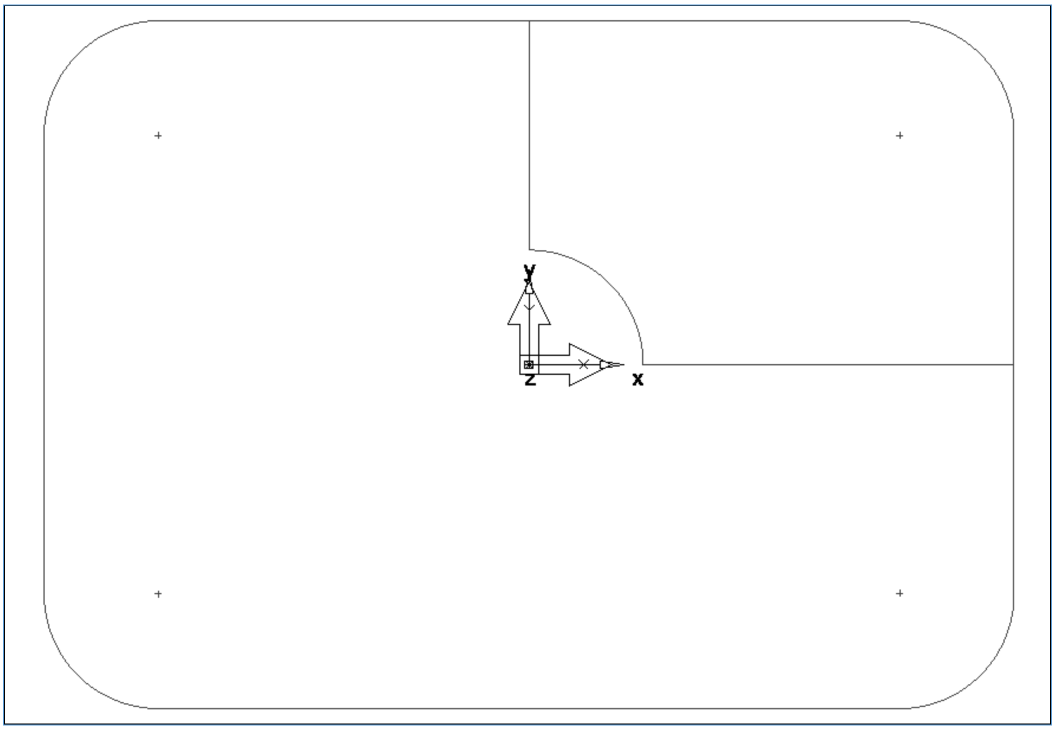
Vertical Line select Enter.

From the Line menu select 

Horizontal Line select Enter. This will Automatically default to X0, Y0, Z0.

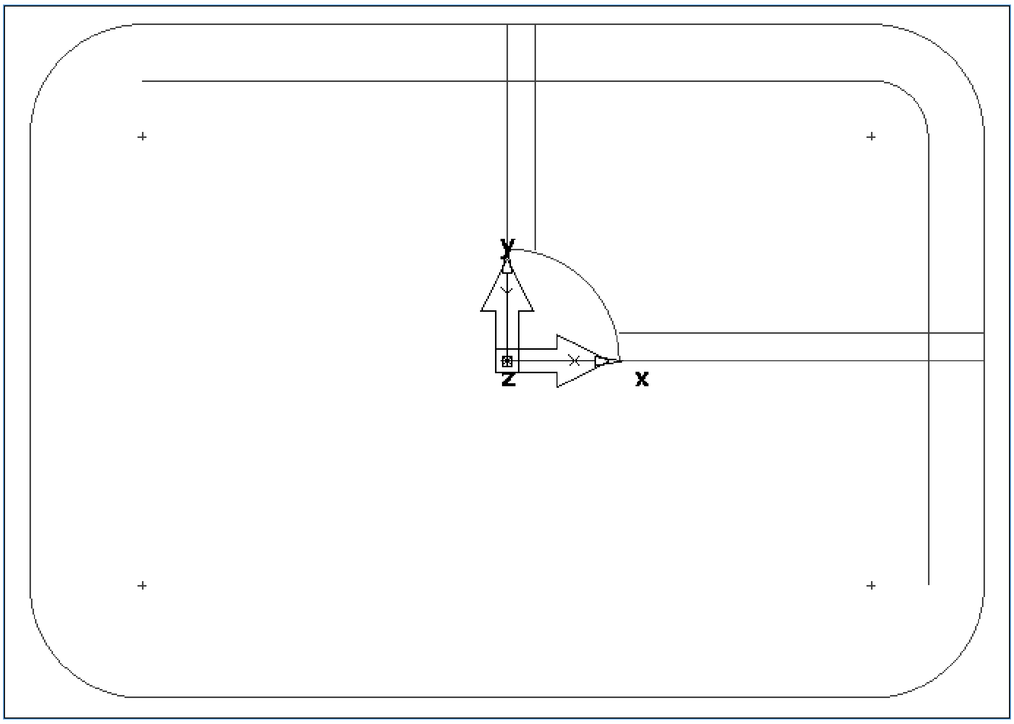
10 We will now **Clip** 

the unwanted geometry so it looks like the image on the next page.

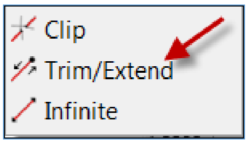


11 We will now **offset** the outer right hand corner shape by **10mm** inwards and the centre H**orizontal** and **Vertical Lines** by **5mm**.

12 Select Offset Offset enter each value and pick the correct side of the line to offset the geometry as shown below.



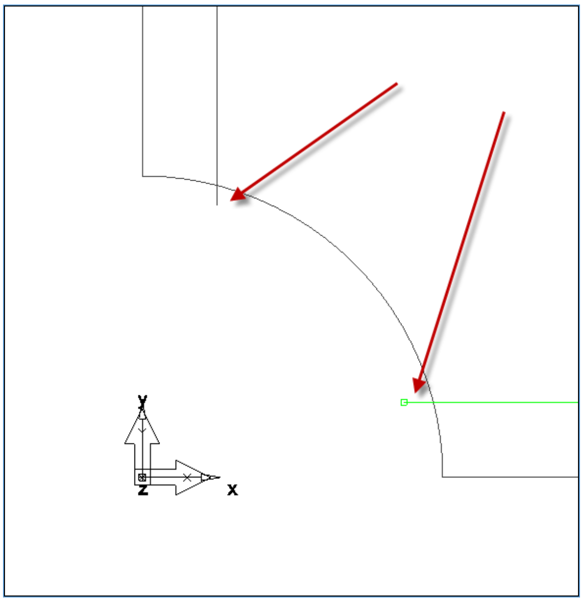
We now have a problem, because the two lines do not intersect the centre radius we cannot Clip the geometry. So we will have to extend the geometry.

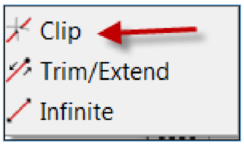
13 Select 

Trim/Extend from the menu.

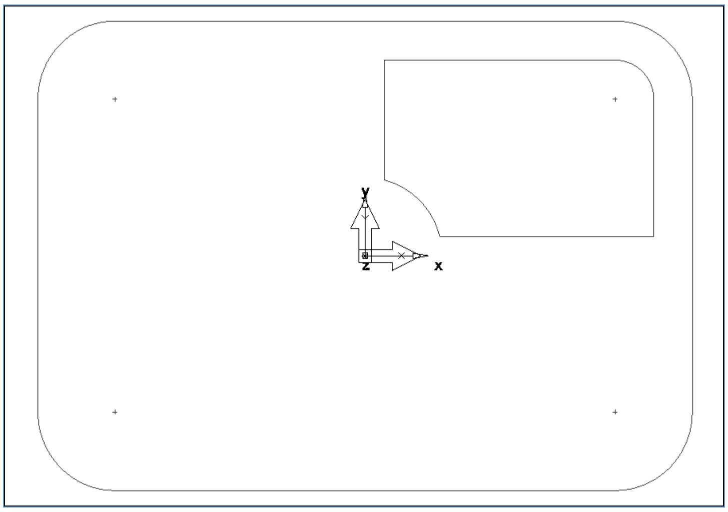
14 Zoom into the centre area. To do this position the cursor in the centre and use the scroll button on the mouse.

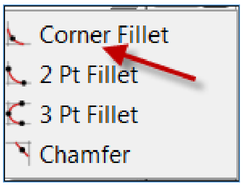
15 Select the **line** you want to extend, this will highlight green. Hold the left mouse button down and then move the pointer so it goes past the radius as shown in two places.



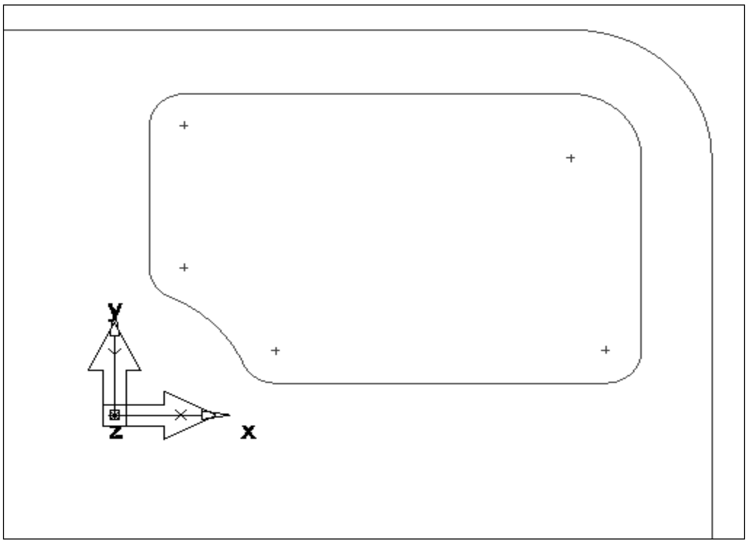
16 We will now **Clip** 

the unwanted geometry so it looks like the image below.

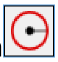


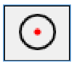
17 We will now **Fillet** the four corners by **5mm**. Select 

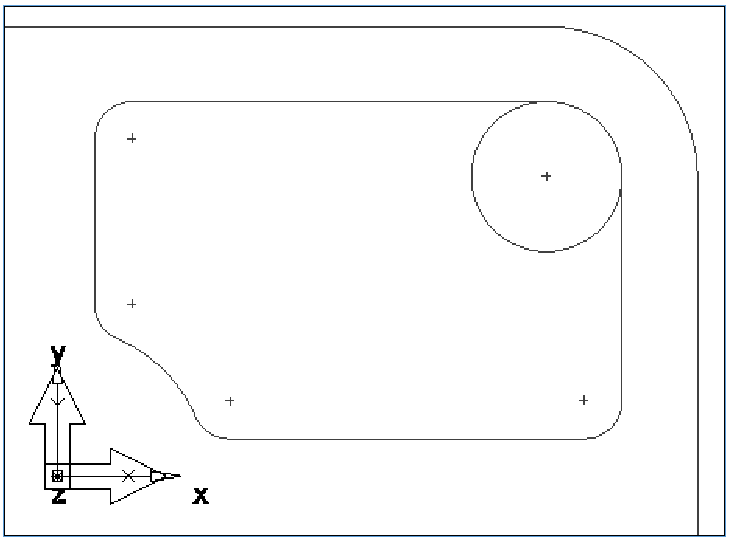
Corner Fillet. The image on the next page shows the fillets in the drawing.



18 **Next** we will Construct a Circle using **Centre, Radius** select the **Centre, Radius**

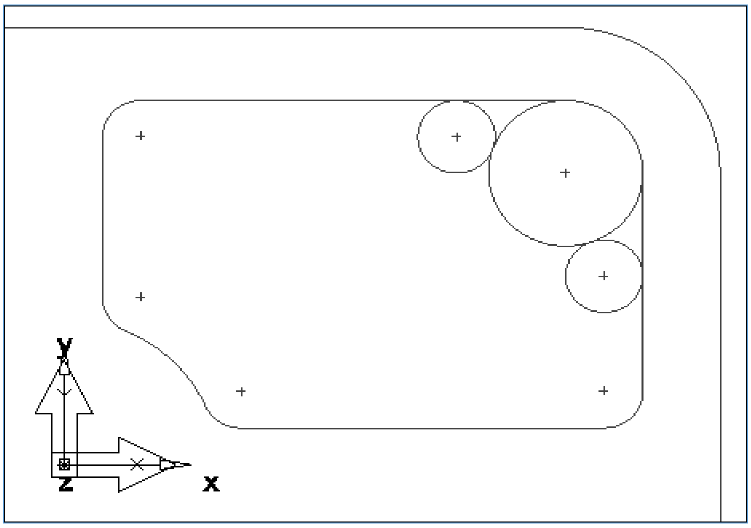
19 Icon  Type in the following values **10mm** and snap to the inner top Right

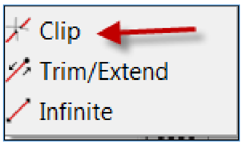
20 corner, using **Snap to Centre** 



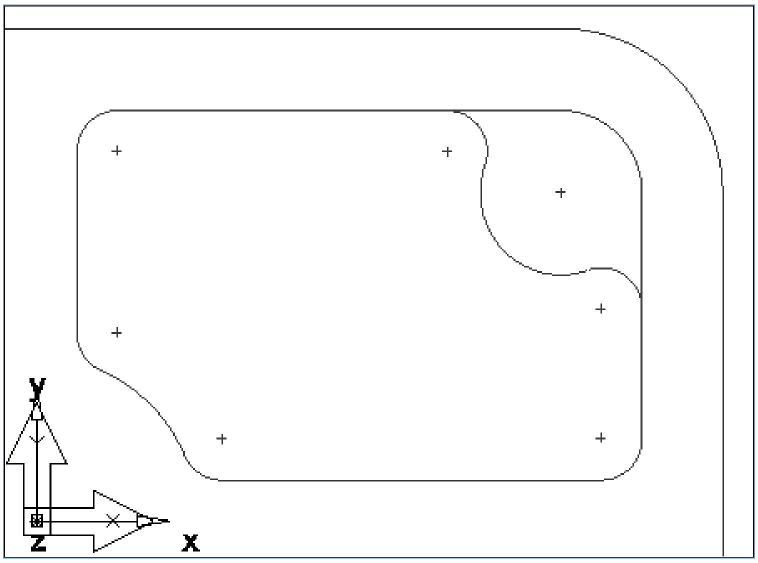
21 **Create 2 circles with radius 5mm** using **Tangent two,** 

 *It does not matter if you touch the line or circle first.*

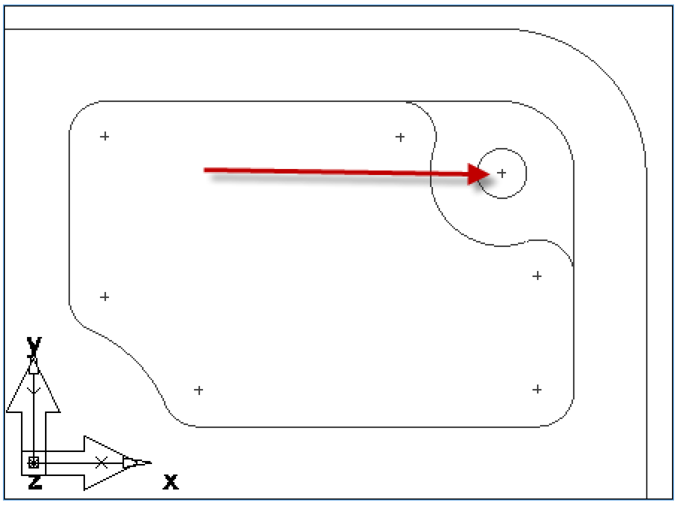


22 We now have to **Clip** 

the unwanted geometry. Please note we have geometry on top of geometry. Make sure you remove this geometry. See image below.

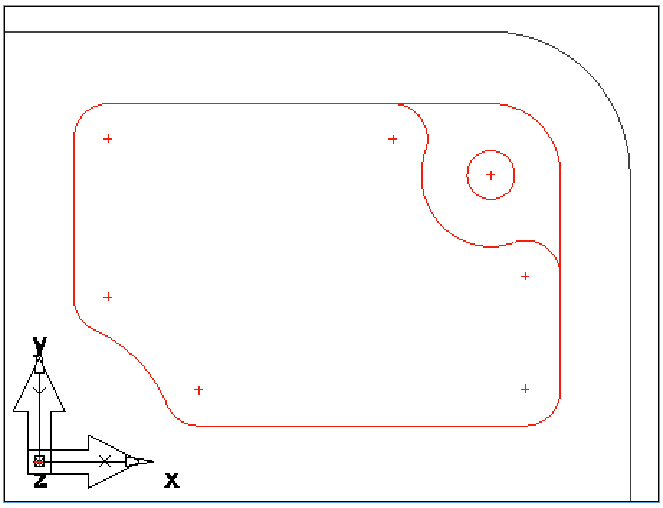


23 We now have to draw a circle with a **radius** of **3.4mm** and position this in the top right hand corner as shown.



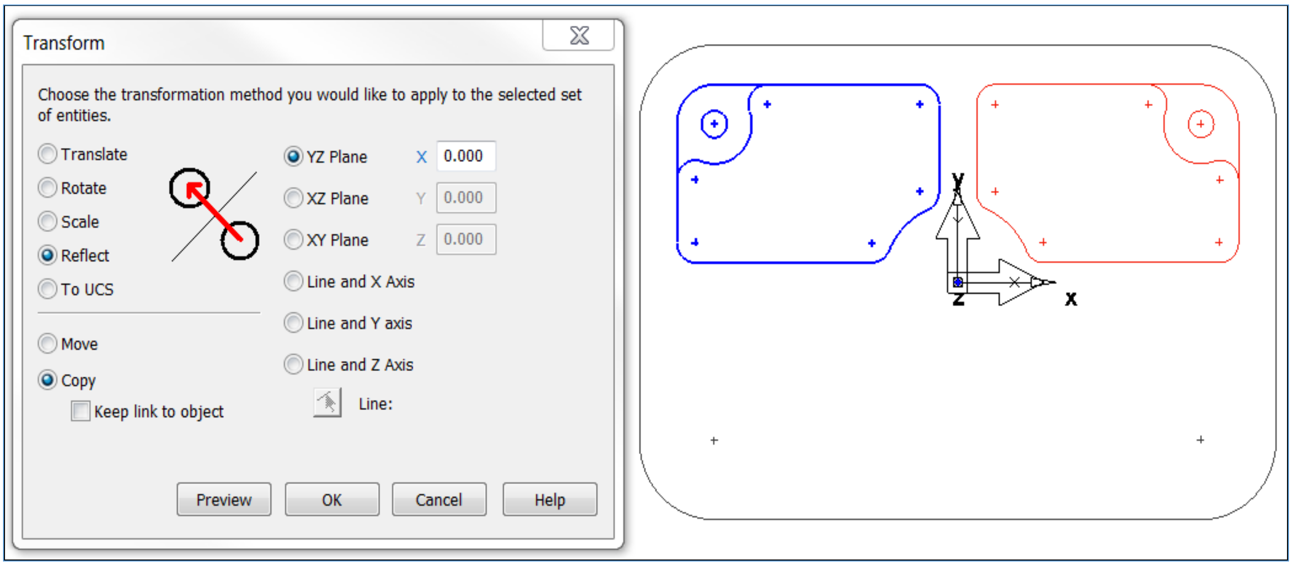
24 We will now **Copy** and **Reflect** this corner pocket first in **X** and then in **Y**.

25 Highlight all of the pocket and the geometry inside only. **Selection** is Red.



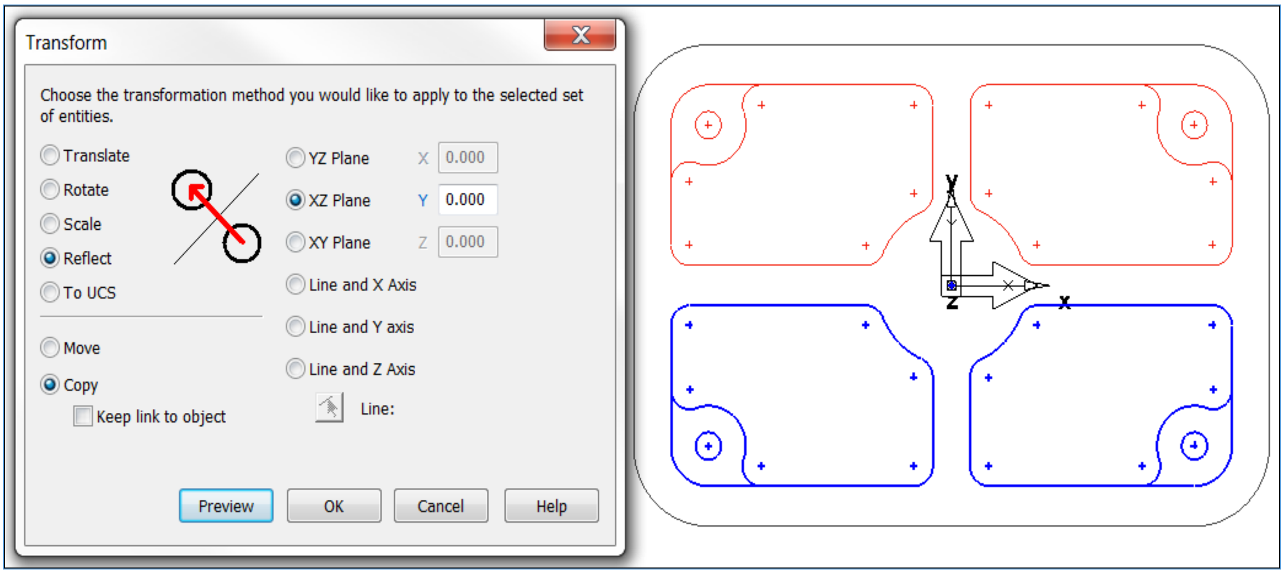
26 Select **Edit** from the main menu. Then select **Transform** select **YZ**.

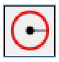
27 Select **Ok**.



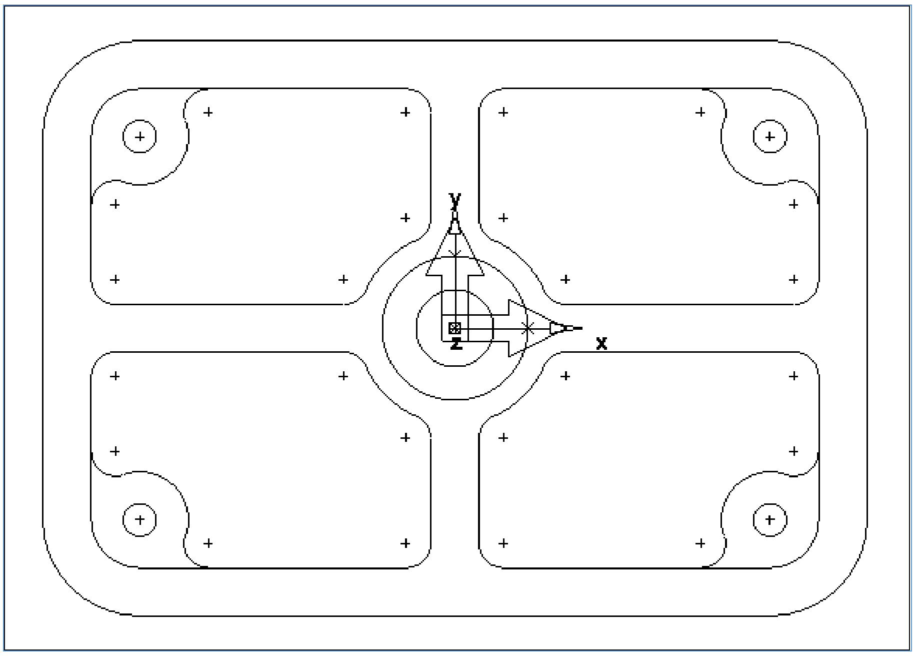
28 Then select **Edit** again and Transform. Then select **Transform** select **XZ**.

29 Select **OK**.

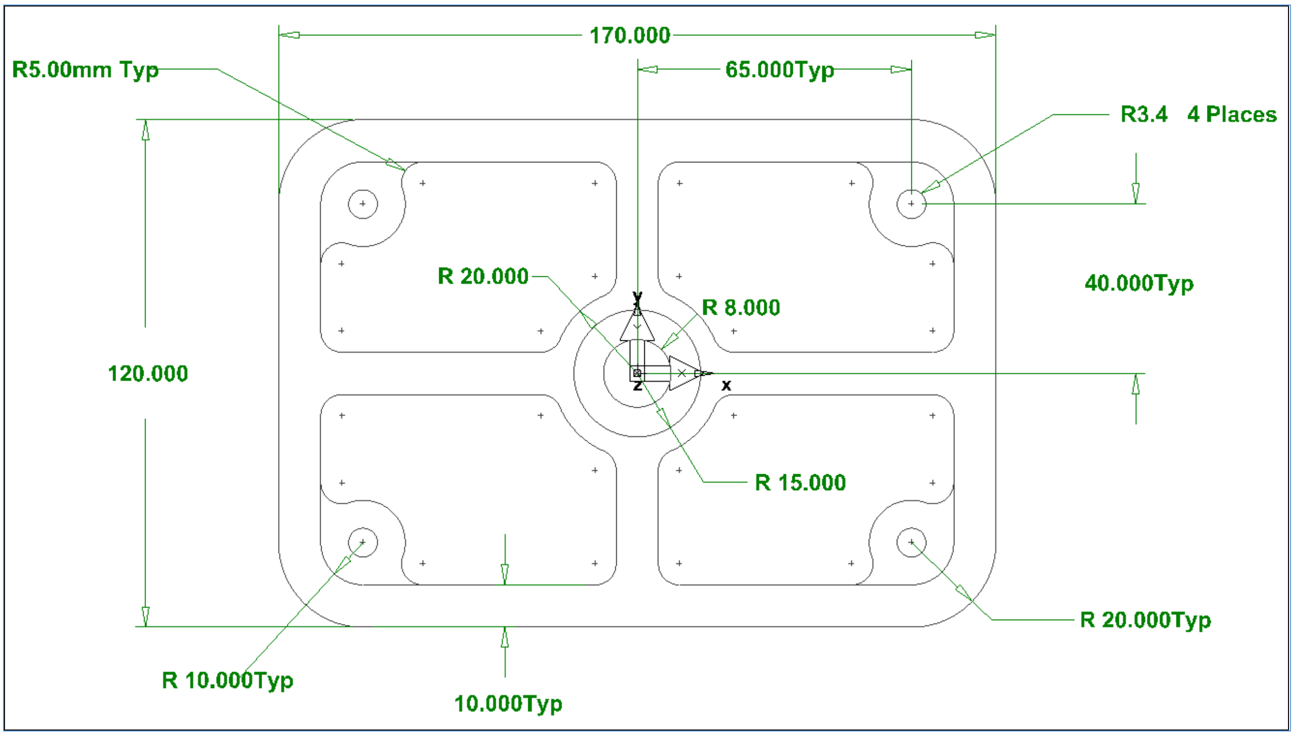


30 Select Centre Radius  and create a circle with a Radius of 15mm and another circle with a Radius of 8mm both at X0, Y0, Z0. The geometry is now complete. See completed image on the next page.

Completed image



*If you are ahead of the class, then practice **dimensioning** this component yourself. Ask your instructor for help if needed.*



31 Go to **Construct Dimensioning** and pick one at a time the available options.

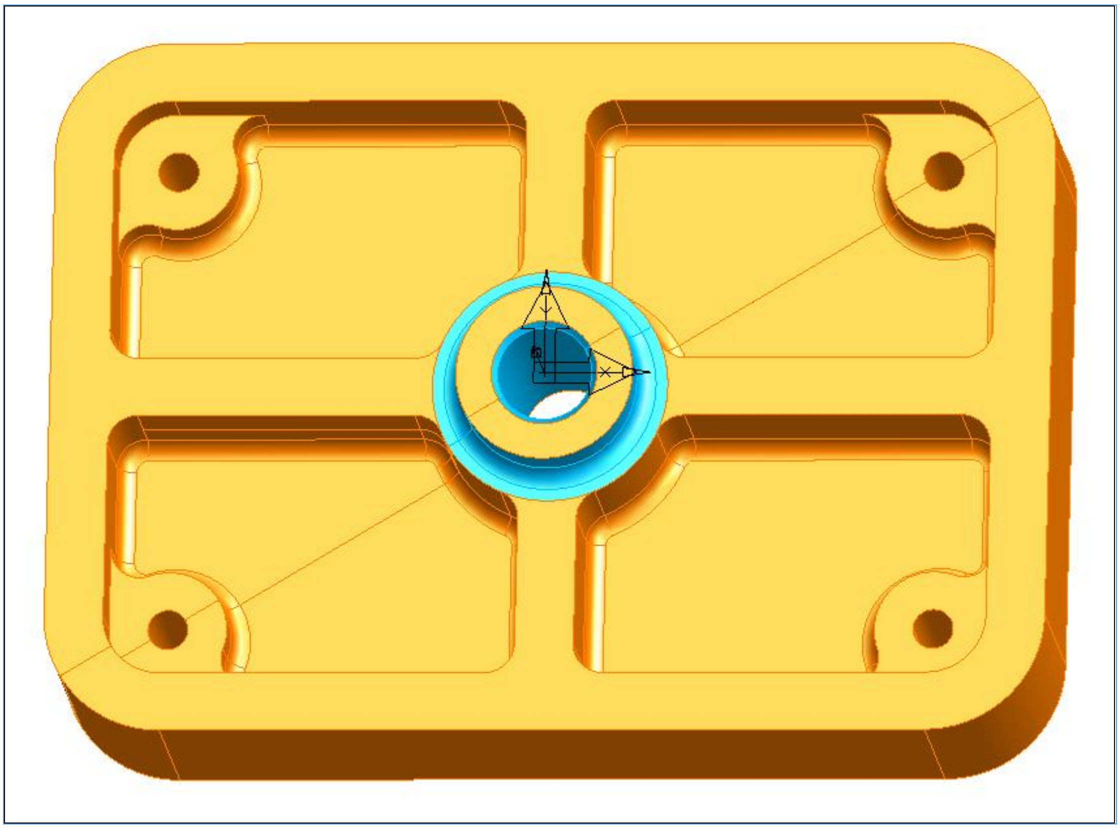
*After you have completed this exercise **save** the file.*

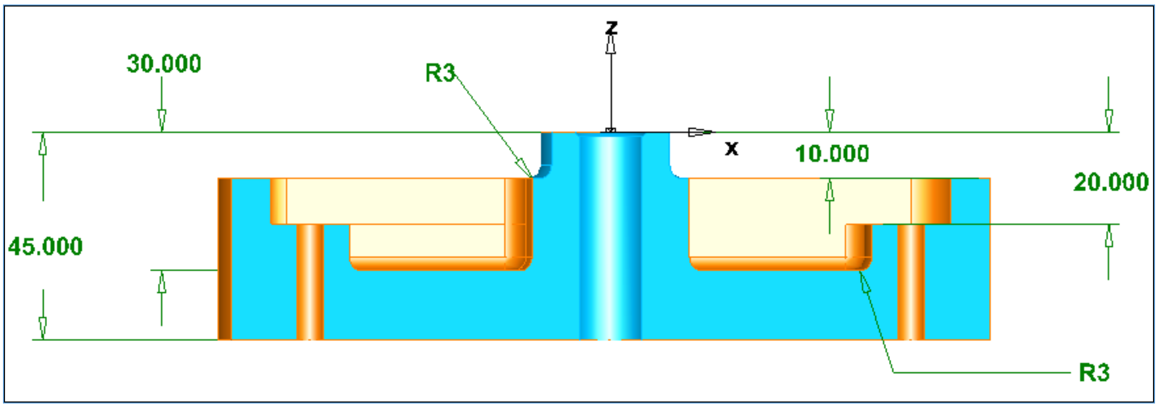
*Save the file as **Construction Exercise 2** in the following Folder.*

C:\Training_Data\FeatureCAM Course Data 2017\Course Work

*We will be machining this on **day 2** of the course. Just concentrate on geometry creation techniques.*

Have a go at the exercises on the following pages. If you need help, ask your instructor as you work through the exercises.





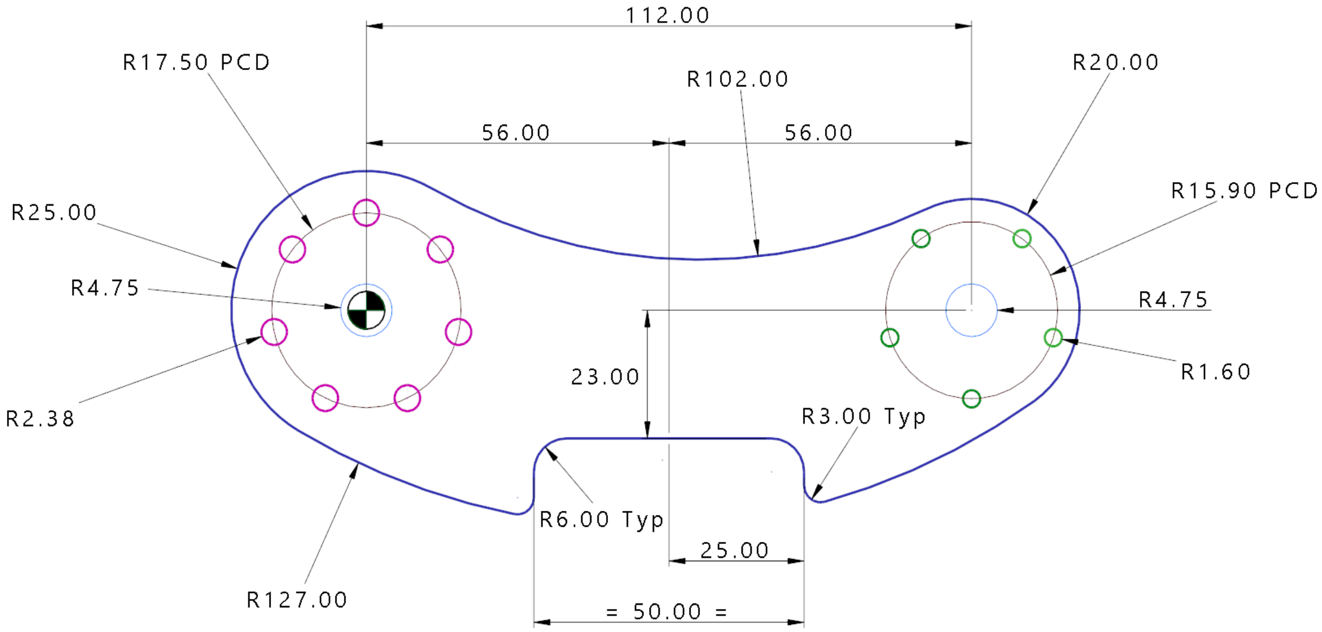
Additional machining information

- Product thickness **45mm**
- Centre hole **16mm** Dia
- **6.8mm** Dia holes are **through**.
- **Top Boss Radius 15mm Depth 10mm**
- Bottom Corner Radius values for **Pockets** and **Boss** are **3mm**

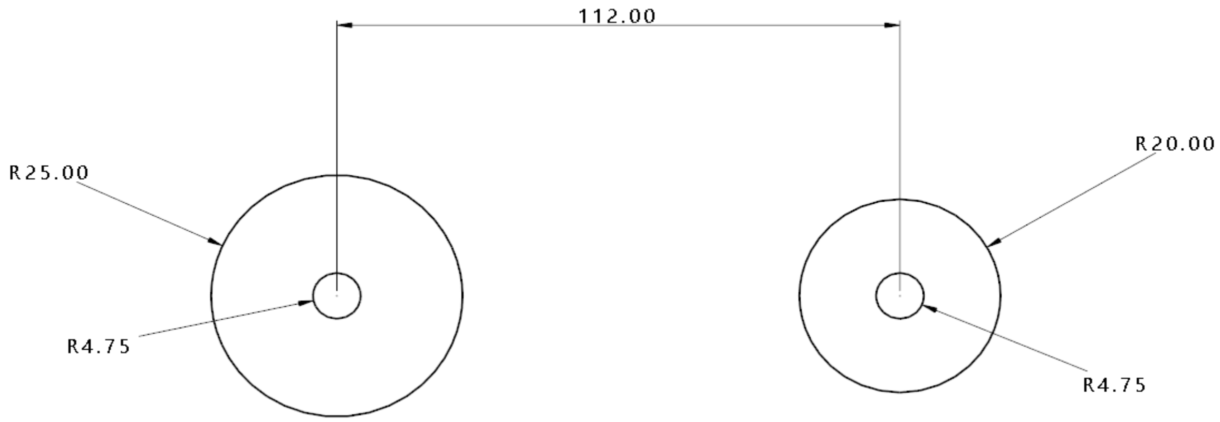
If in doubt ask.

Construction Exercise 3

Have a go yourself at the following exercise. Please ask your instructor for help if required. See further Help on how to start this example shown below.



- Before you start drawing this component spend time assimilating the information and decide what building blocks to draw first. A suggested process might be to start with the **R25mm** and **R20mm** Circles first and build the drawing from there. See the image below for a hint on what it should look like.



We will machine these components later on just concentrate on the geometry.

Additional machining information

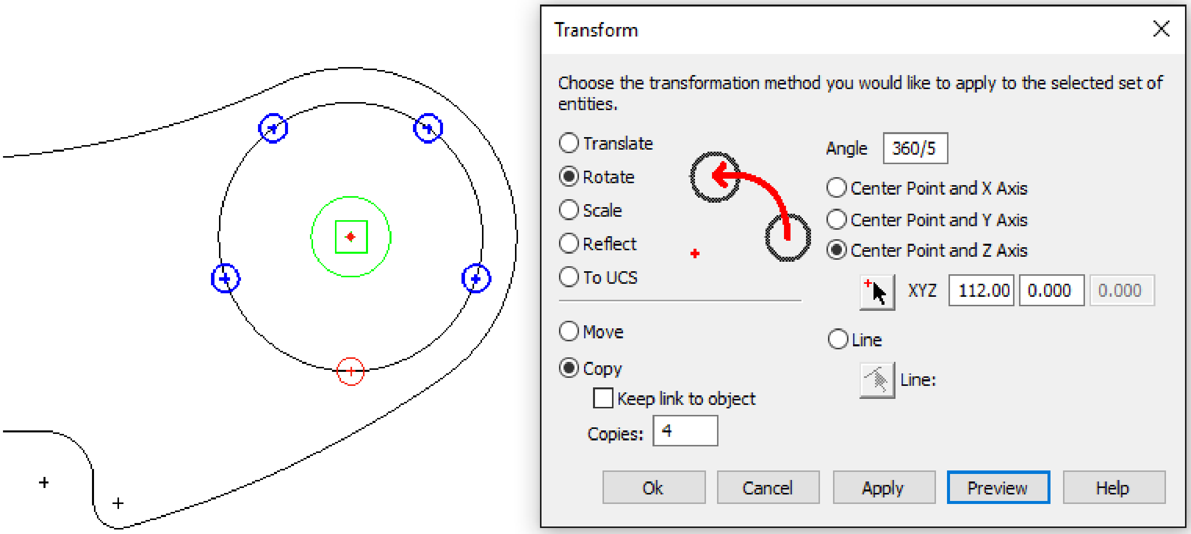
- Product thickness is **25mm**
- **9.5mm** Diameter holes are through

- **4.75mm** Diameter and **3.2mm** Diameter holes are **20mm** deep.

*A tip on creating this drawing is to create a single hole on the centre line of the **9.5mm** diameter holes on the respective pitch circle diameter and then using Transform, copy and rotate it about a specific point. (Shown below) select the hole you want to rotate, go to the **Edit** menu select **Transform** and then **Rotate** and **Copy**. In the Angle field type in for example **360/5 = 72** and then you will need **4 copies** which **(Excludes the original).** Remember to select the Rotation **centre point**. If you specify 5 copies you will get another circle on top of the original. When you come to drill the holes, you will get the same hole drilled twice.*

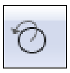
Using Transform

- Use the **Transform** dialog to **Move** or **Copy** objects:



1 Open the **Transform** dialog in one of these ways:

2 Select at least one object and click the **Transform** button in the **Standard**

3 Toolbar.  Or select **Edit > Transform** from the menu.

4 Or right-click on an object and select **Transform** from the context menu.

5 You can transform **Geometry**, **Curves**, **Features**, **Surfaces**, or **Solids**.

*Features cannot be transformed using the **Scale** or **To UCS** methods.*

***Translate** selected entity elements to a new location. You can move an absolute distance as specified in XYZ vectors, or you can move from point to point.*

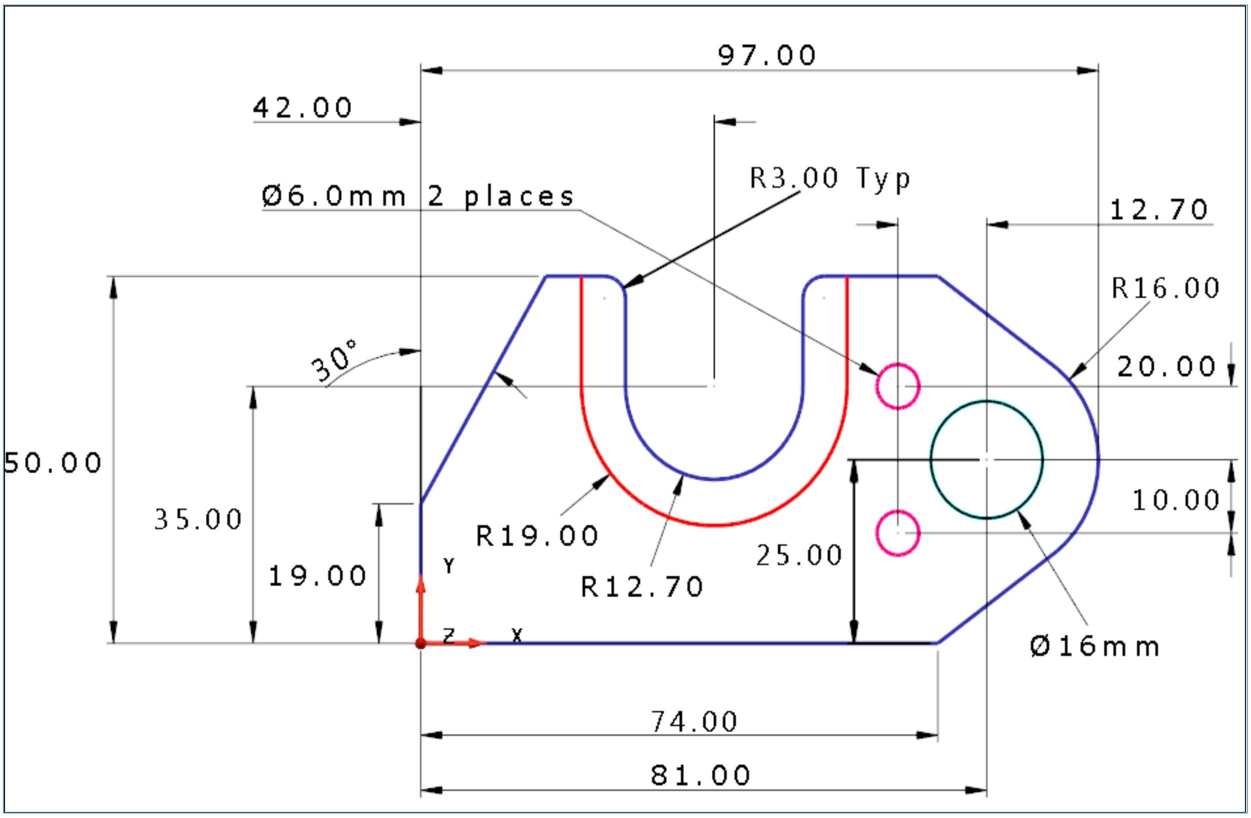
***Rotate** selected elements about a selected location to a specified angle, referenced from the positive X axis.*

Scale *proportionally reduces or expands selected elements about a specified point. This option is not available for features.*

Reflect *mirrors the element about a line. The line can be an existing axis, or any other line including one created just for reflecting around. The object can be flipped top to bottom, left to right or even both depending on the line you choose to reflect around. Set whether you want to* ***Move*** *or* ***Copy*** *the original element. If you are copying, more fields appear for setting the number of copies you want in the new location.*

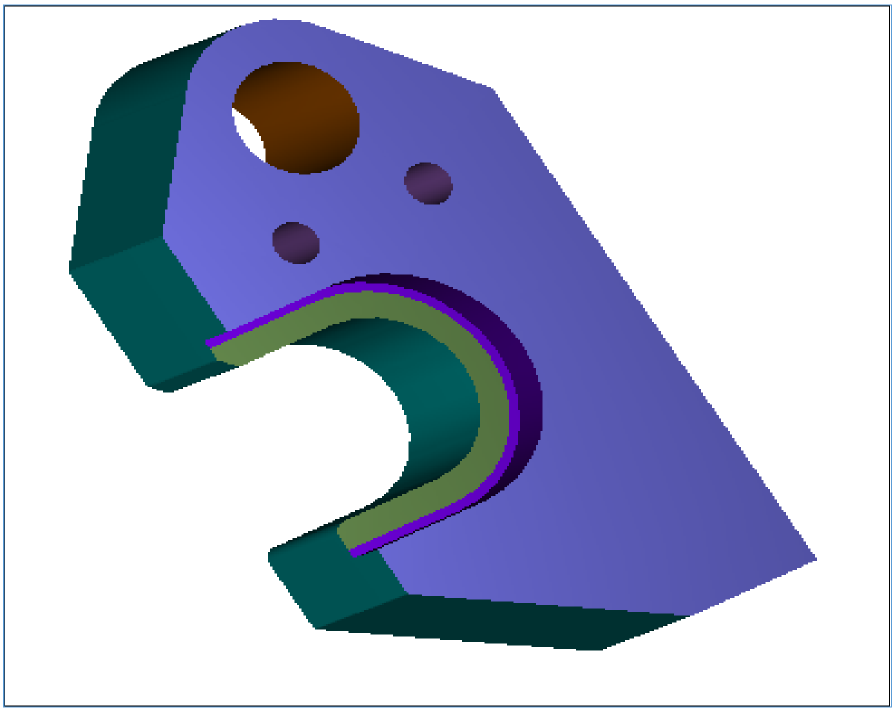
There are additional optional exercises that follow. This will give you more practice drawing and using tools you have learned up to this point.

Construction Exercise 4 (Optional Exercise)



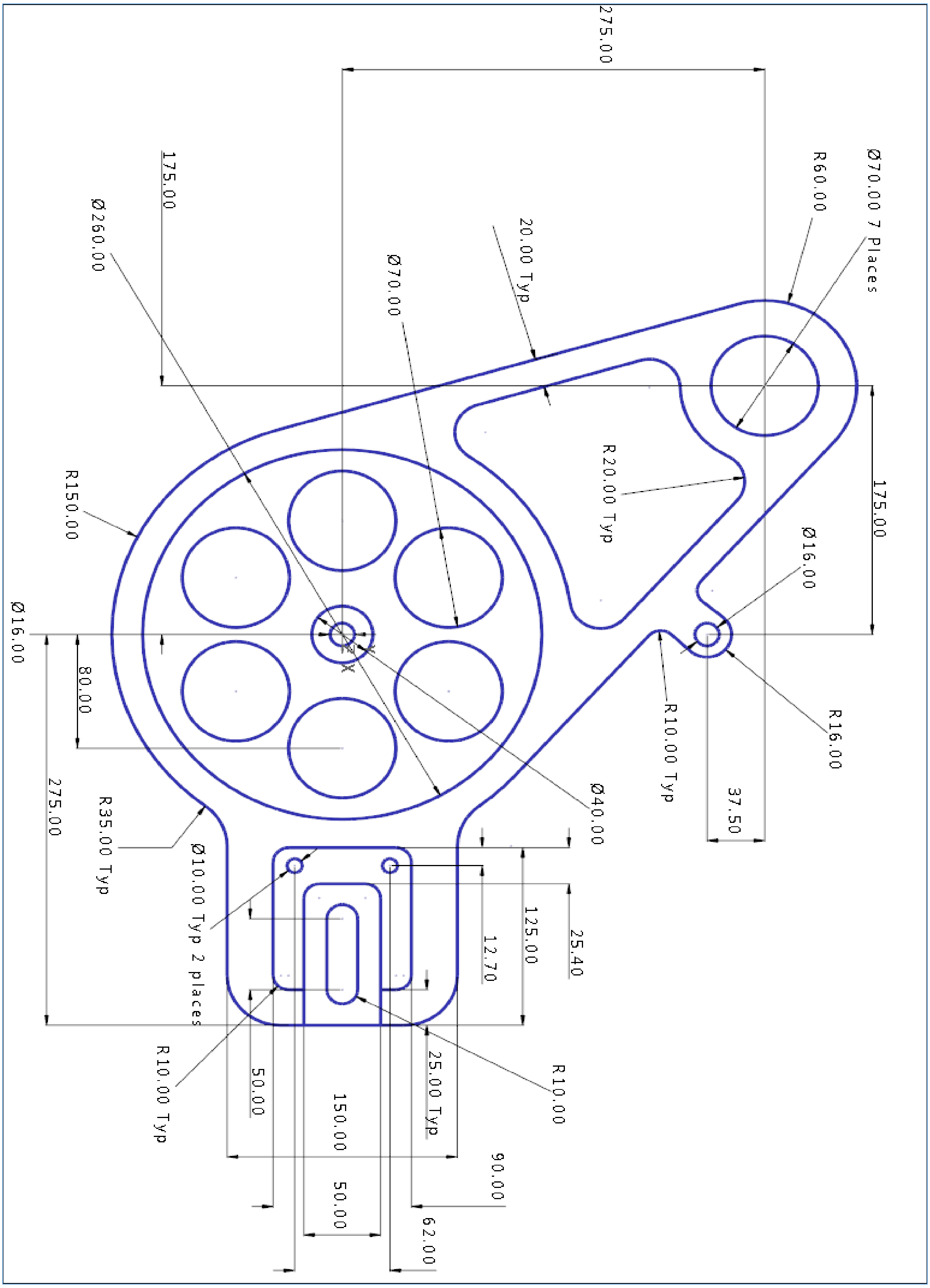
Additional machining information

- Product thickness is **25mm**
- Large **16mm** hole is through.
- **6mm** holes are **20mm** deep.
- Counter bore on open profile is **5mm** deep from top.
- **R12.7mm** & **R19mm** Radius centres are the same as the top **Ø6mm** diameter hole.

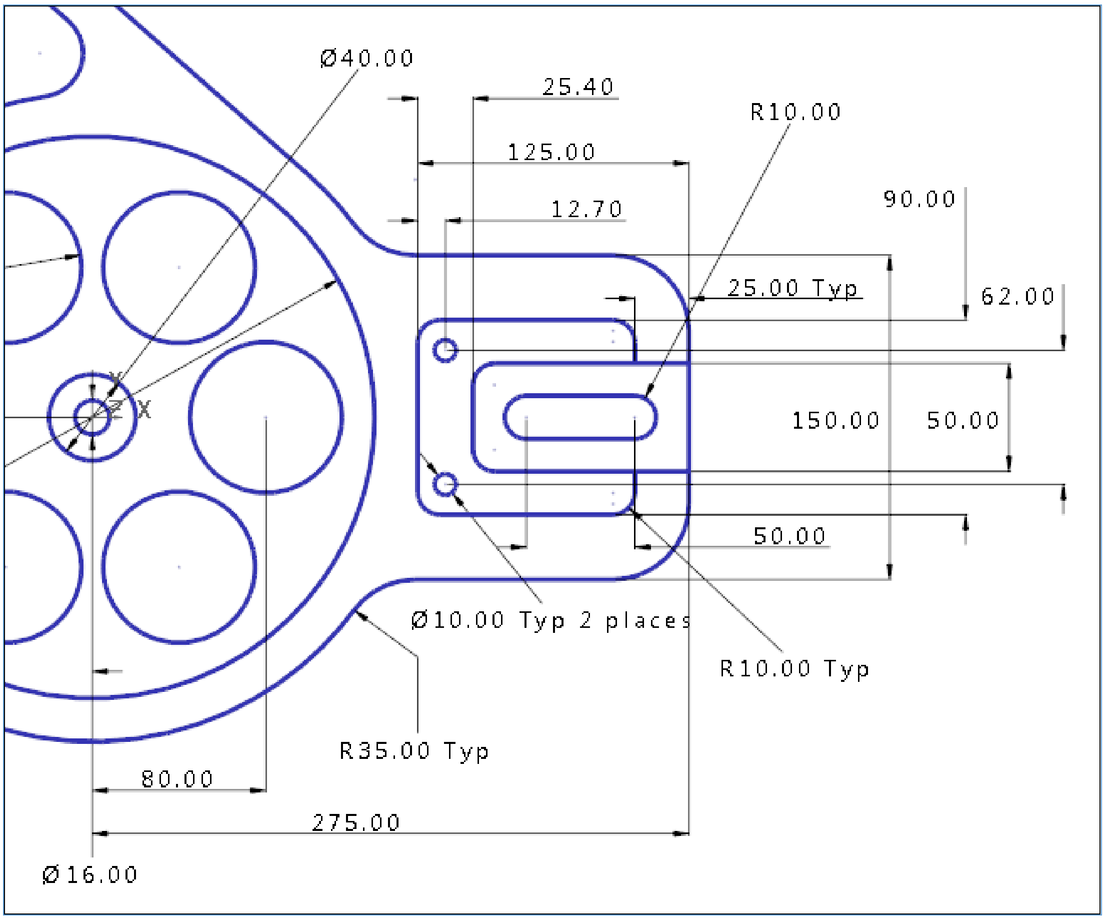


If in doubt ask.

Construction Exercise 5 (Optional Exercise)

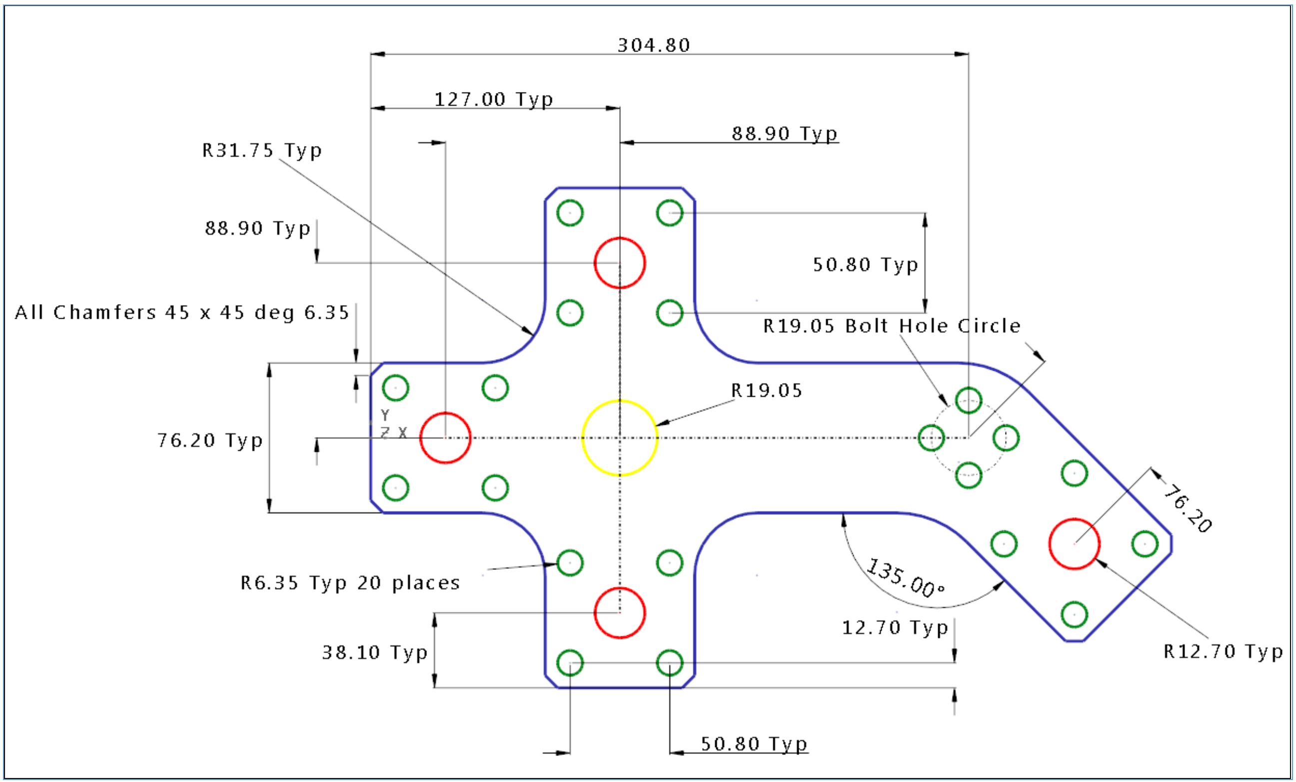


If in doubt ask.



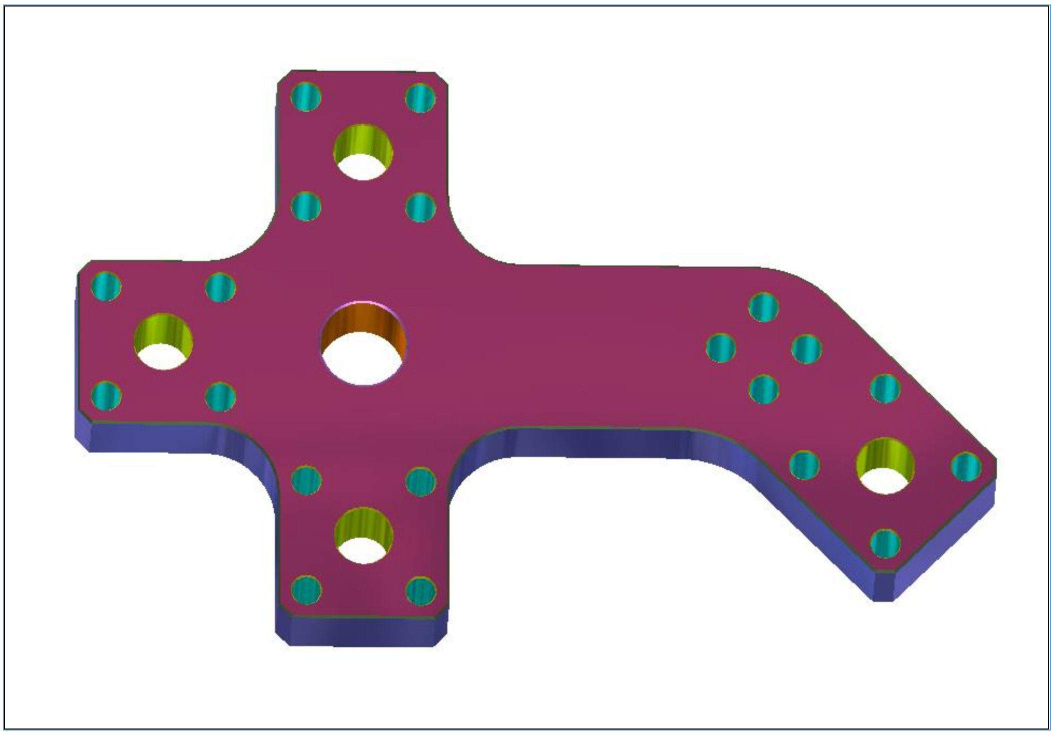
- Product thickness is **40mm**
- **260mm** diameter bore is **17.5mm** deep
- Large closed pocket is **16mm** deep.
- **70mm** bores are through.
- **100mm** x **90mm** Pocket is **10mm** deep.
- **50mm** pocket is **16mm** deep.

Construction Exercise 6 (Optional Exercise)

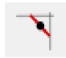


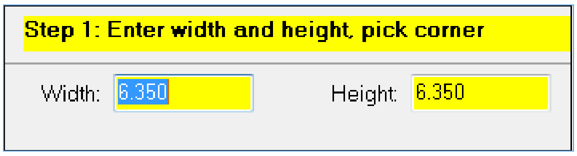
Additional machining information

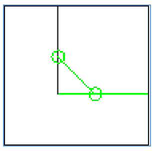
- Product thickness is **35mm. All holes are through**.
- 20 holes **12.7mm Dia**
- **Chamfer all edges.**



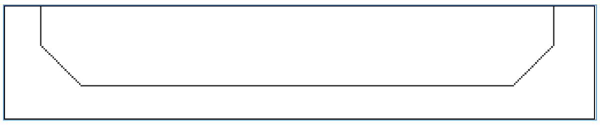
Tricks & Tips

- When you want to Chamfer the corners of this example use **Chamfer** 
- Enter Width **6.35mm** and Height **6.35mm**

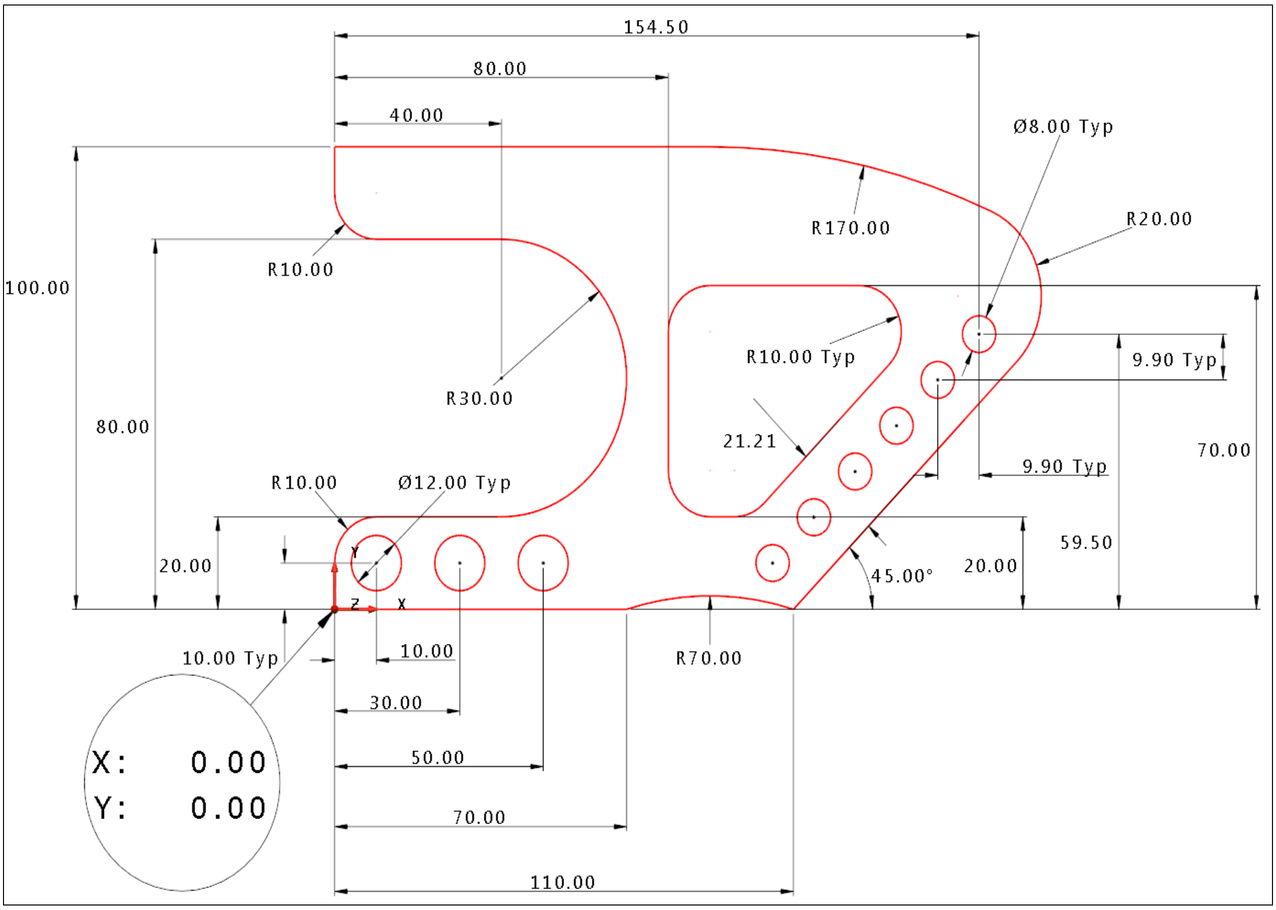




- Then use the **mouse pointer** and move it to the **corner** as shown
- Repeat this process for all chamfers.



See instructor for help with Chamfers at 135 degrees Construction Exercise 7 (Optional Exercise)

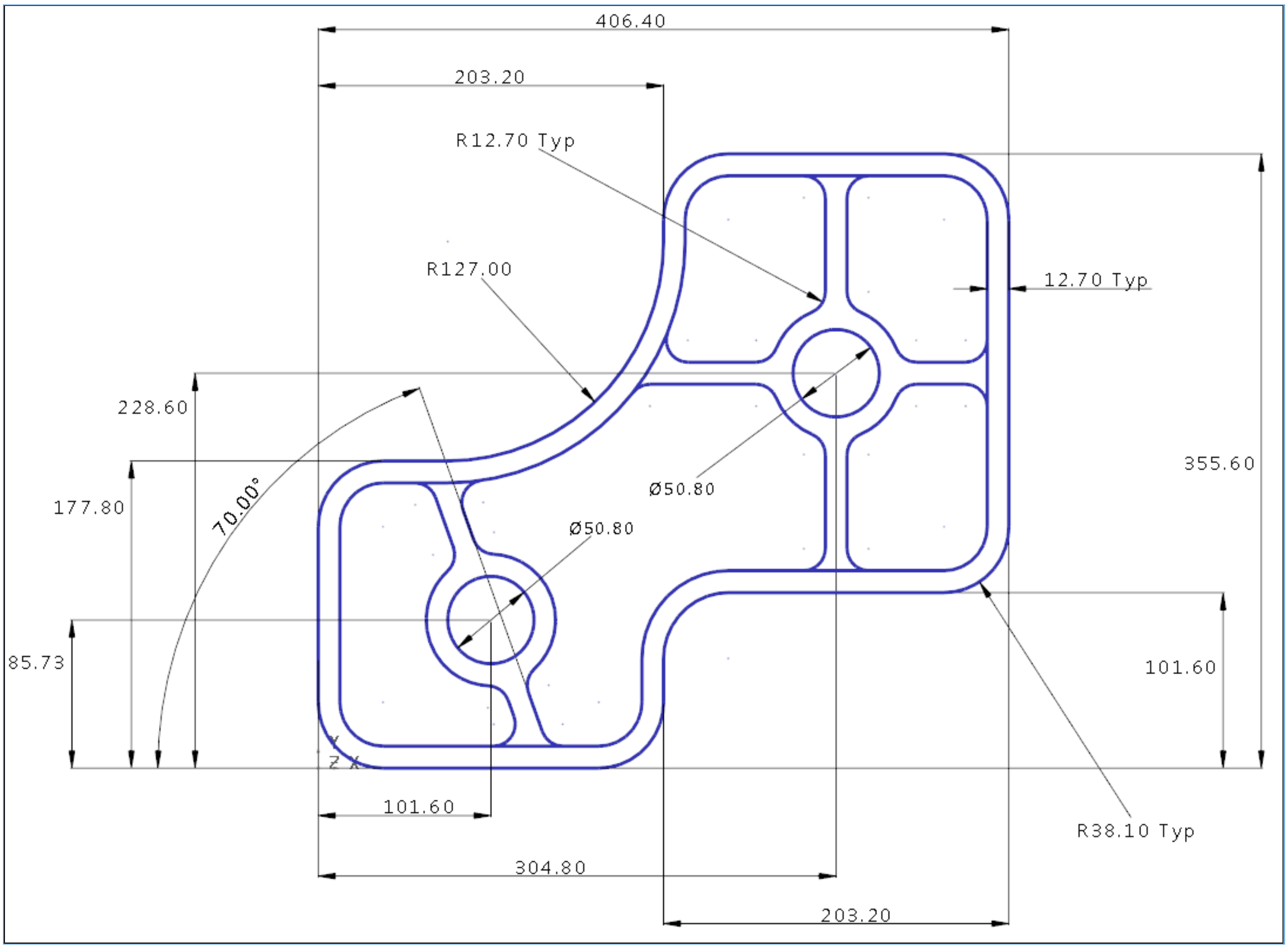


 ***If in doubt ask.***

Additional machining information

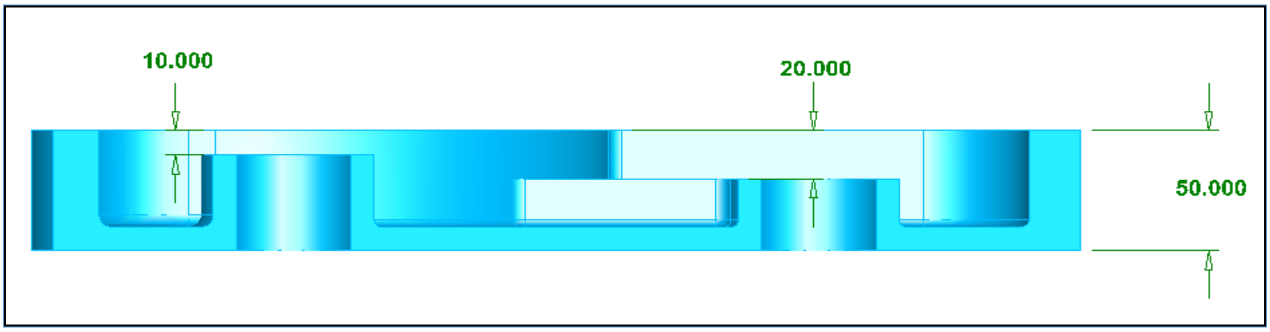
- Product thickness is **45mm**
- All Holes are through.
- **170mm** Radius centre is as follows **X80 Y-70 Radius 170mm**

Construction Exercise 8 (Optional Exercise)

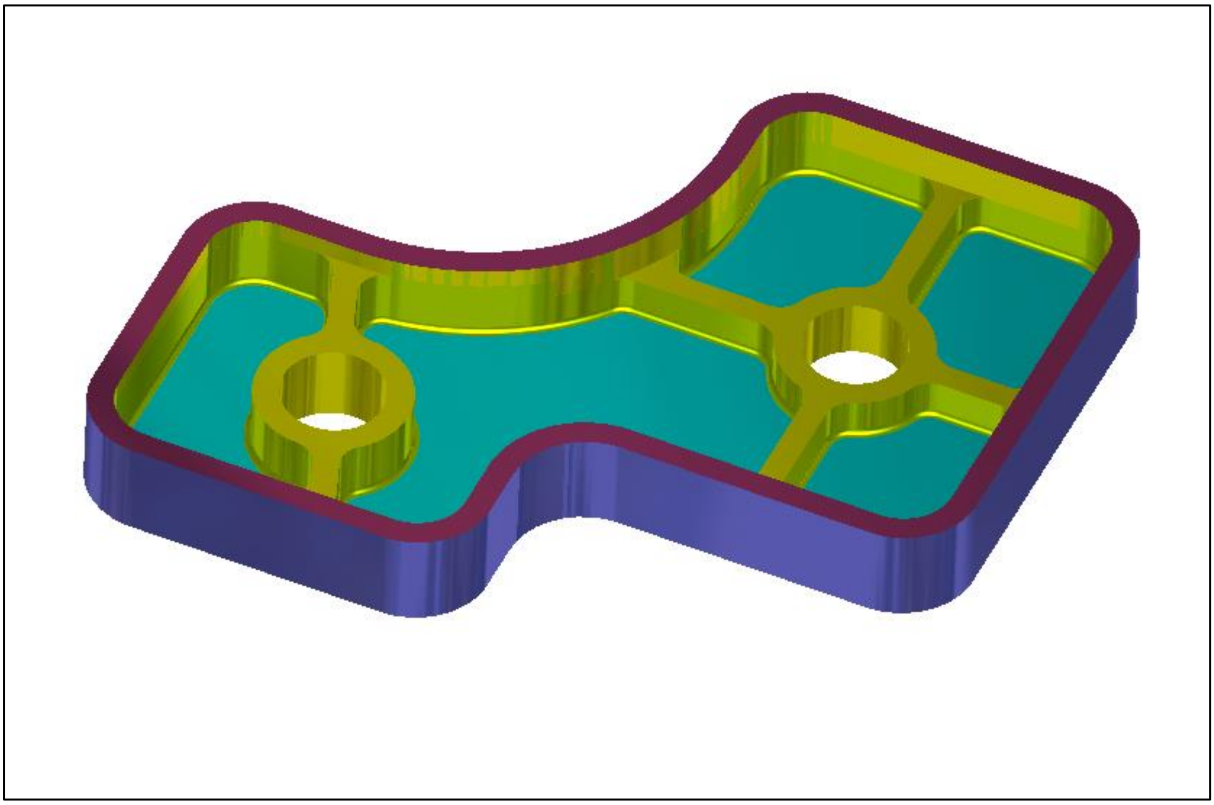


Additional machining information

- Datum point Lower Left corner.
- Product thickness 50mm.
- Section A – A machine 10mm deep,
- Section B – B 20mm deep from the top surface.
- 50.8mm Diameter bores are through.



Completed Machined image



If in doubt ask.

This page is intentionally left blank

Contents

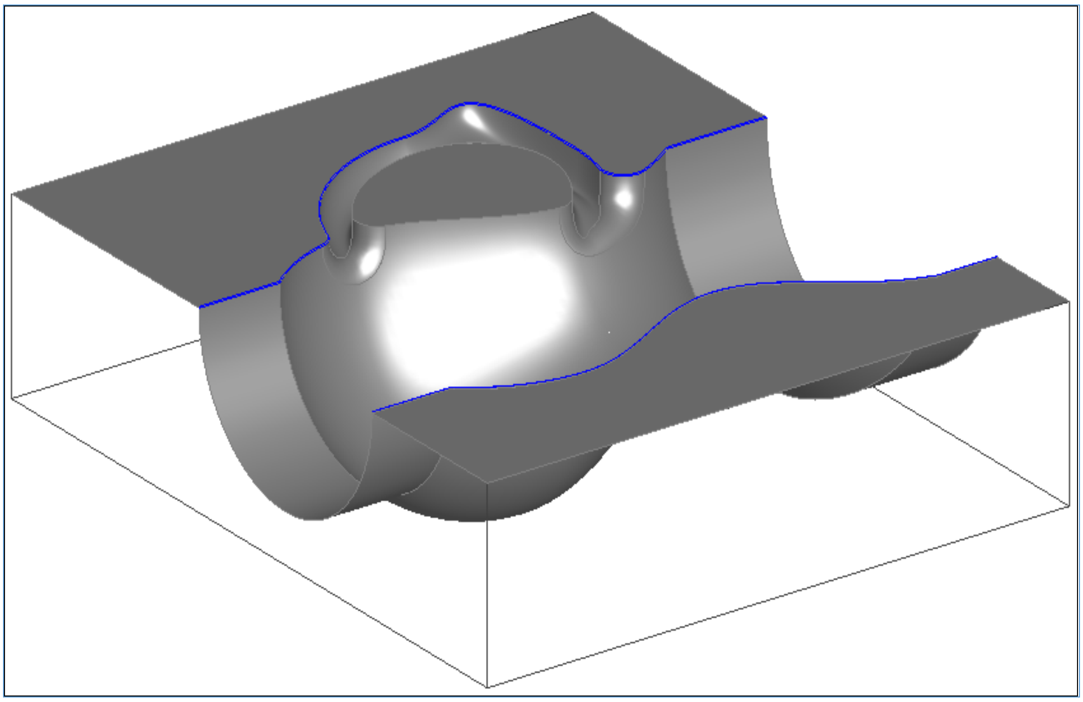
Curve Creation

Introduction

- This module will provide a basic understanding of what curves are and how to create them within **FeatureCAM**. By the end of the module the user will be able to construct both open and closed curves. There are more advanced Curve creation options available. These will be covered later on as we progress through the training course.

What is a Curve

- Curves are paths in 2D or 3D space. You can create them in **FeatureCAM** or import them from other CAD systems. Curves can be **open** or **closed**. You can create **Curves** from **Geometry lines** and **arcs**, from other **Curves**, from **Points**, from **CAM dimensions**, from **Windows™ fonts**, or from **Surface** and **Solid Models.**

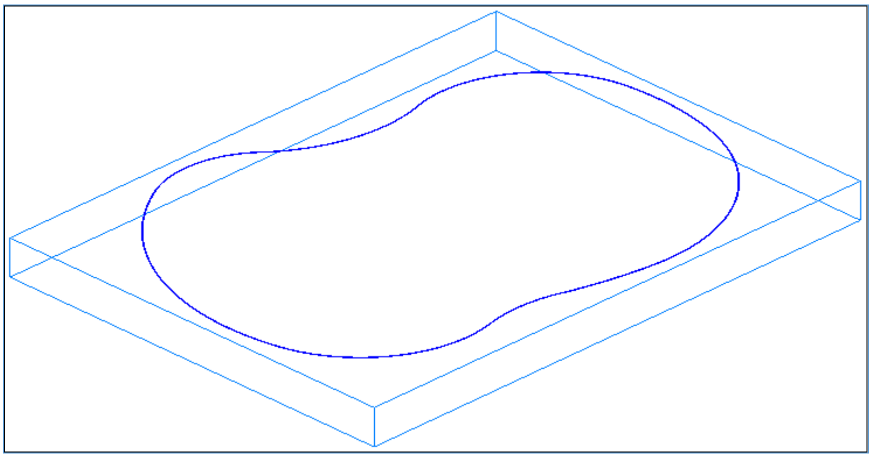


Chaining

- **Lines**, **circles** and **arcs** typically represent the shape of a part. To use a sequence of lines and arcs as the shape of a feature, you must **Chain** them into a **Curve**. To make a **Feature** from more than a single line, arc or circle, you must **connect** the geometry into a **Curve**. **Chaining** is the primary way of creating **Curves** by connecting pieces of geometry. In many cases you do not need to trim away pieces of geometry. **Chaining** will automatically prefer smooth, tangent-continuous paths since these paths are more conducive to manufacturing.

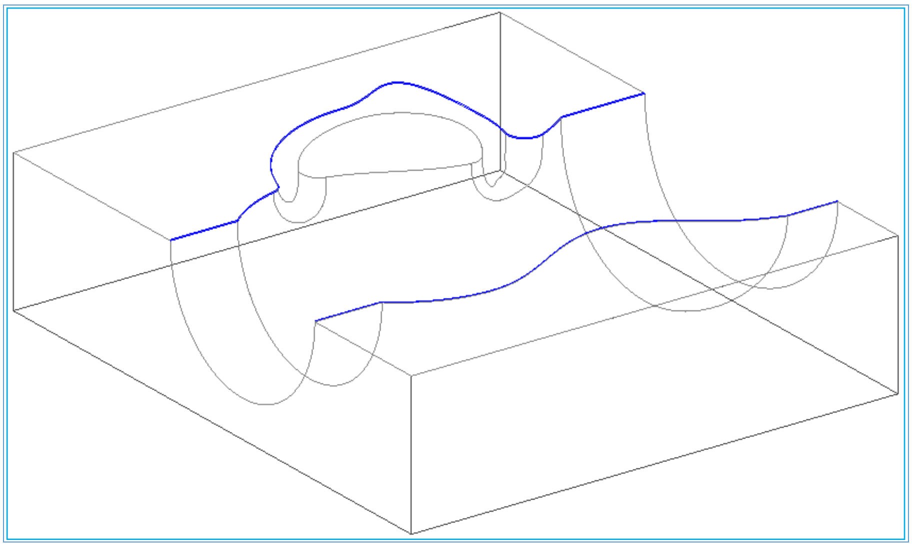
Closed Curves

- **Closed Curves** have **start** and **end points** in the **same location**, and at least one other point (not in that location) included in the curve. The **Closed Curve** clearly defines an area as the interior of the curve and completely separates this area from the exterior of the curve. Any ambiguities, such as overlapping curve links or intersecting curve links, cause failures and unpredictable results in the machining routines in **FeatureCAM**. To **Chain Open curves** the **closed curve** command may be used by simply **double-clicking** the **closed** geometry using the **Pick Pieces** command.



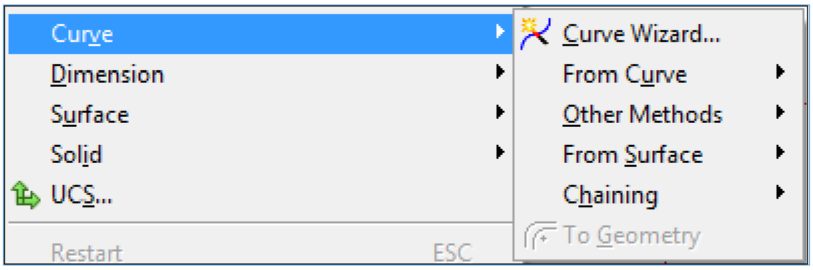
Open Curves

- **Open Curves** have end points that do not meet. You can only use open curves in **Side** and **Groove Features**. To chain open curves we use the **Pick Pieces** command. The image below shows two **Open Curves** on a part.



Curve Constructors

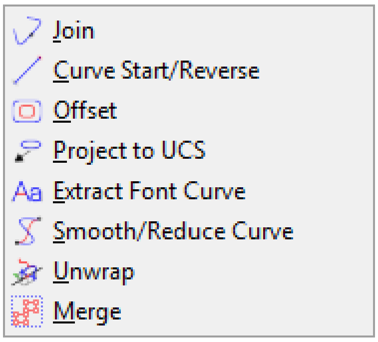
- Curve constructors are used to create/modify curves using different methods. In this section there will be a short description of how each function works.
- From the **File** menu click on **Construct** then on **Curve**. Let's review these.



There are four different methods for creating curves: ***From Curve****,* ***Other methods****,* ***From Surface*** *and* ***Chaining.***

From Curve

- From the **File** menu click on **Construct** then **Curve** then **From Curve.**



The **Curve from Curve** menu has functions that create new curves based on existing curves.

Curve Join: - Connects a collection of curves, arcs or lines into a single curve.

Curve Start/Reverse: - changes direction and starting point of the curve

Curve Offset: - offsets a curve in the direction you pick by the amount you set.

Project to UCS: - projects flat curve or 3D curve to current UCS

Extract Font Curve: - extracts curve segment from a font curve

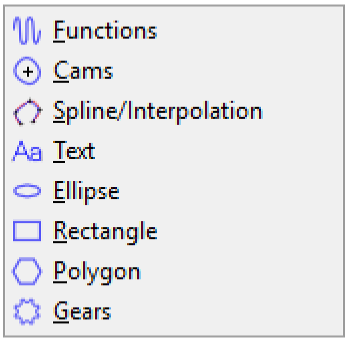
Smooth/Reduce Curve: - reduces the amount of data in a curve and

Smooths out any bumps within a user specified tolerance.

Unwrap: - unwraps a curve that has been wrapped around a cylinder

Other Methods

- From the **File** menu **click on**: - **Construct>Curve>Other Methods**.
- Curve from other methods gives you the ability to create curves using other types of constructors.



Function: - creates user-defined mathematical relationships to generate a graphical figure.

Cams: - creates the geometric profile of various reciprocating cams or cylindrical cams.

Spline/Interpolation: - creates a continuous smooth curve between points.

Text: - creates text curves from any TrueType installed for windows.

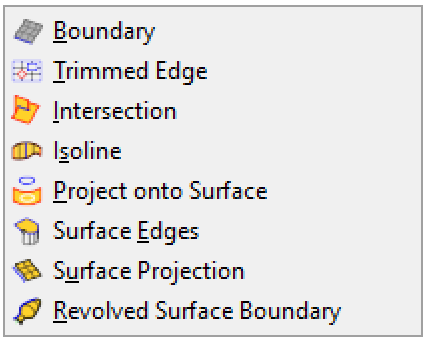
Ellipse: - curve tool creates an elliptically-shaped curve in the plane of the current UCS.

Rectangle: - creates a rectangular shaped curve in the plane of current UCS.

Gear: - creates a 2D gear profile.

From Surface

- From the **File** menu **click on**: - **Construct>Curve>From Surface**.
- The Curve from Surface menu has functions that create new curves from surfaces.



Boundary - extracts the curve from a surface's boundary

Trimmed Edge - extracts the trimmed edges of a surface

Intersection - extracts curves from a surface-surface intersection

Isoline - extracts surface row or column isoline

Project onto Surface - project existing curve onto a surface

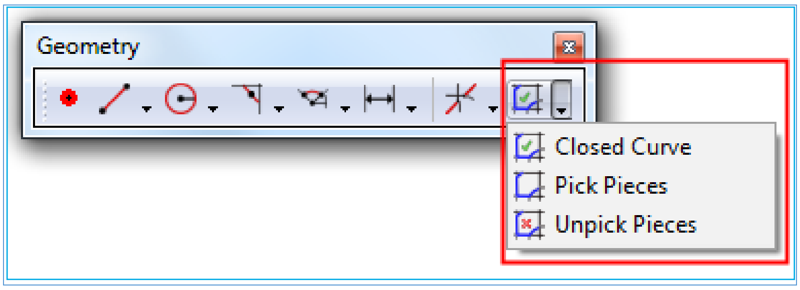
Surface Edges - extracts selected surface edges

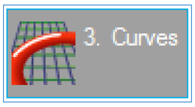
Surface Projection - extracts curves from all vertical walls

Revolved Surface Boundary - extracts a revolved profile about the part's index axis

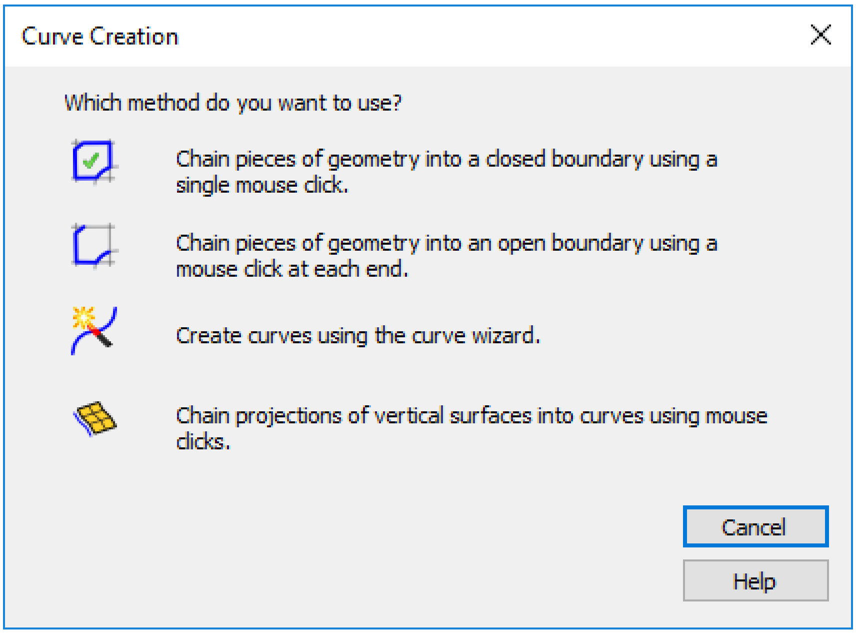
Chaining

- Curve chaining commands can be accessed from three different areas of the user interface:
- Curve chaining commands may be accessed through the Geometry toolbar.



- May be accessed from the **File menu** by clicking on **Construct>Curve> Chaining.**
- Or may be accessed by clicking on **Curves** 

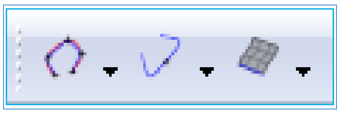
from the **Steps Toolbox**.



Curve Wizard

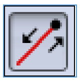
Curve Constructor commands can be accessed in three different areas of the user interface:

- Curve constructor commands are available through the **Steps** menu.
- They may also be accessed through the File menu by clicking on **Construct** and **Curve**
- May be accessed through the **curve toolbar.**



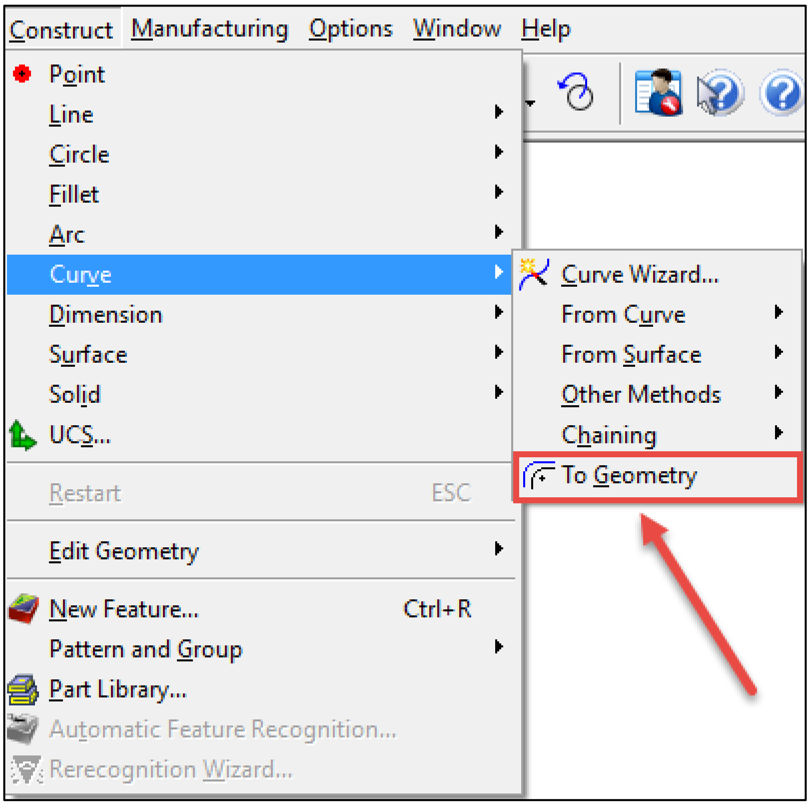
To show the curve toolbar, from the file menu click on View then Toolbars then check the Curves and Surfaces checkbox and press OK

Tricks and Tips

Curves can be extended just the same as **Geometry** when using **Trim /Extend**  this is useful on **Curves** that have been extracted from the **Solid Model** and you need to machine past a point.

Changing a Curve - To Geometry

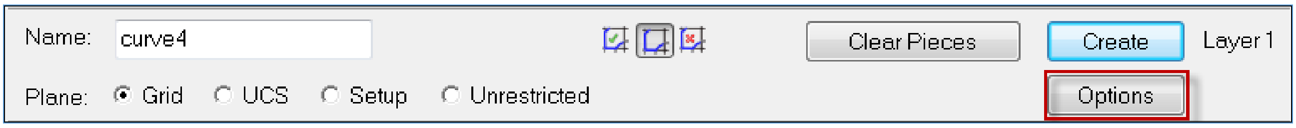
*Whenever you want to change a **Curve** back to **Geometry** just select the **Curve** and then go to **Construct** >**Curve** and select **To Geometry**.*



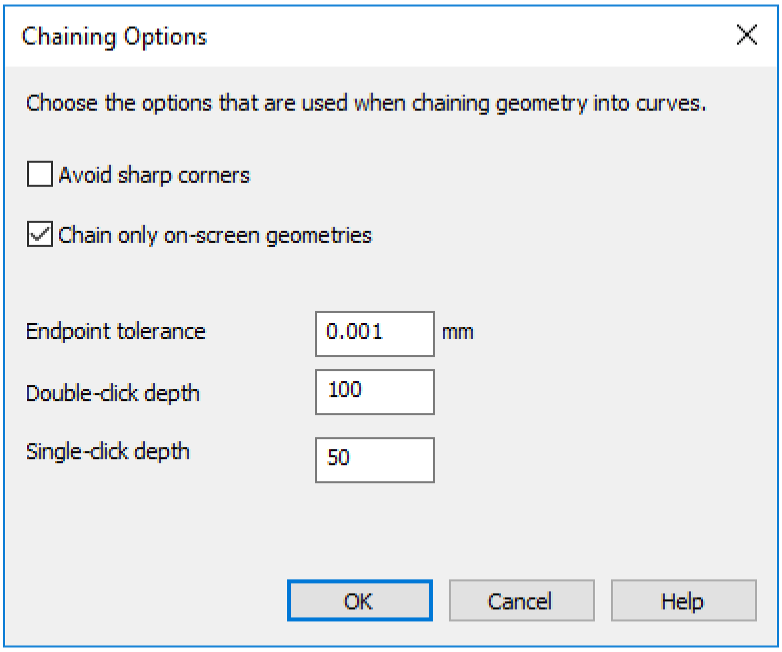
- This will then create the **Geometry** so you can then manipulate it as normal Geometry.

The curve will still be there in its original form.

- Another good tip to think about if you are having trouble creating curves.
- Select the **Options** tab as shown below.

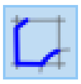


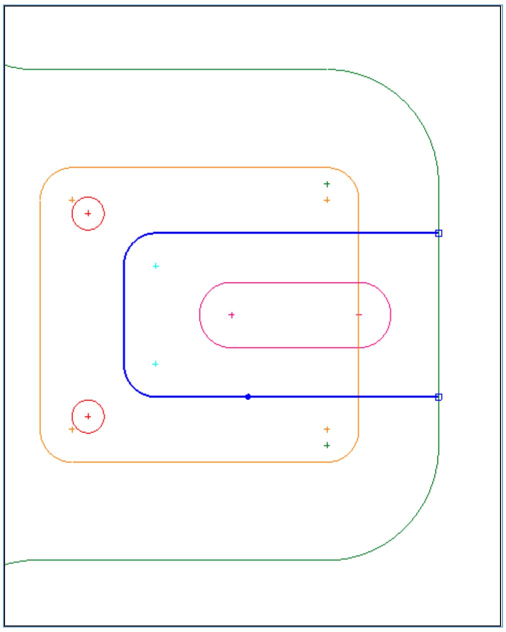
The following menu will appear.



- Change the Endpoint Tolerance to **0.1mm** this will help.
- Also the Double click Depth, change this to a higher number. This is ideal if you are having trouble creating curves from Splined circles which are faceted.

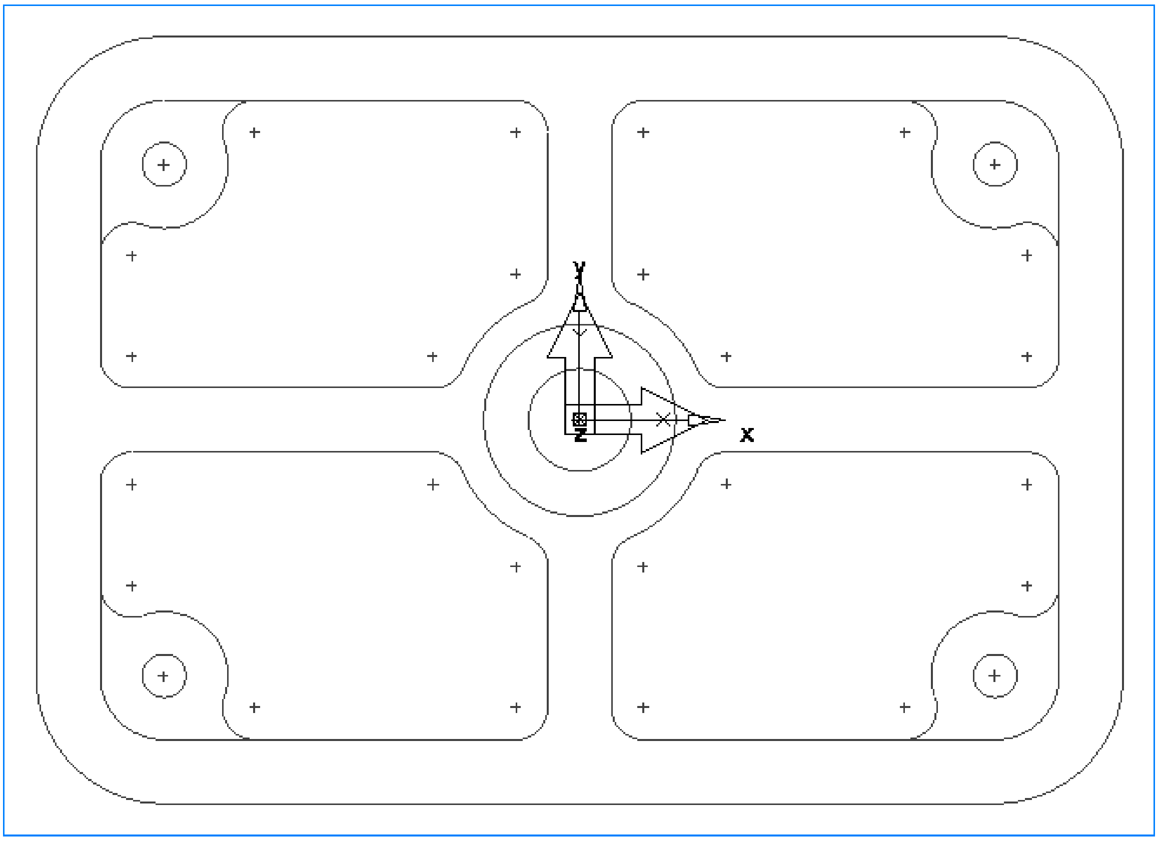
Open Curves

- Remember to use **Pick Pieces**  on **Open Curves.**
- For example the image below is an **Open Curve**. Exercise 4 has an open Curve.

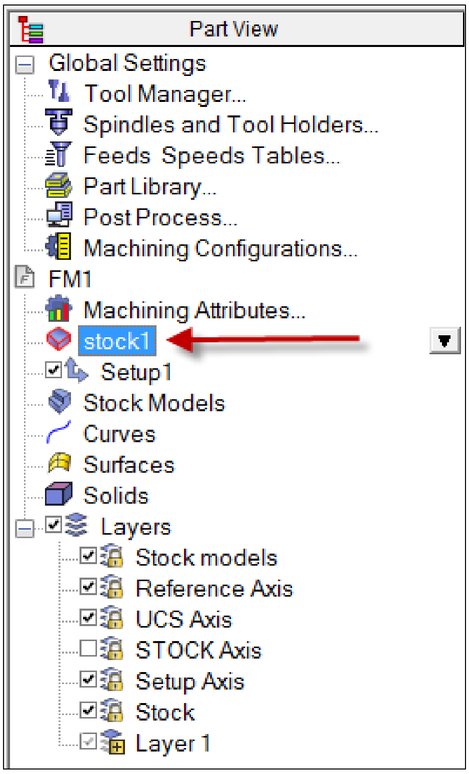


Creating Curves (Class Exercise)

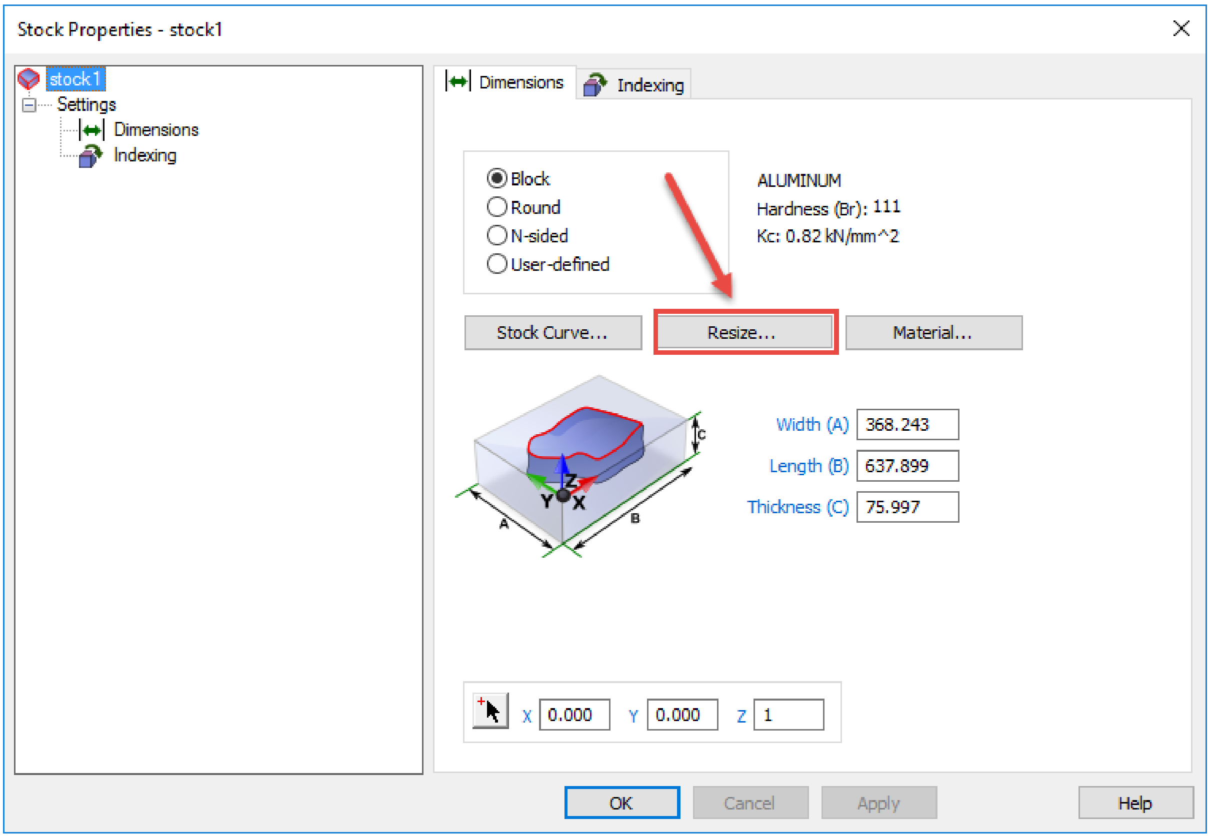
1. **Open** the file **Construction Exercise 2.**
2. Go to **File** and select **Open** and select the following Folder location.
3. **C:\Training_Data\FeatureCAM Course Data 2017\Course Work.** Within the **Folder** there will be the exercise you created on the first day called **Construction Exercise 2.** Open this file.



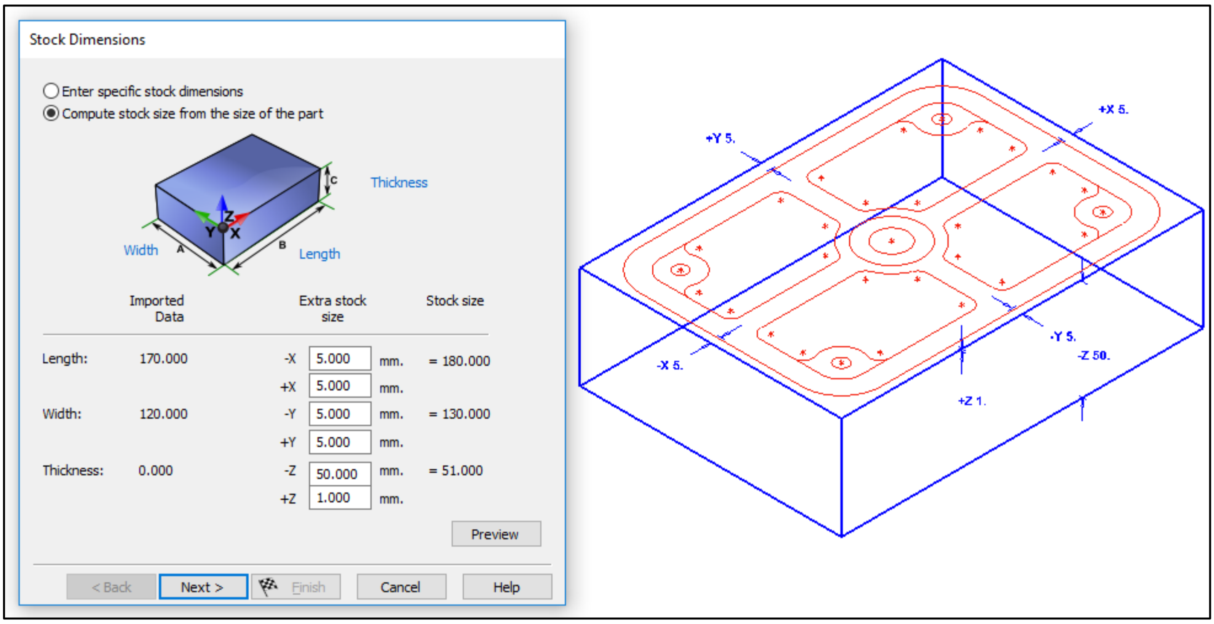
4 The image above shows Stock material around the Part. In **PartView** double click **Stock1.**



The image below shows the size of the current material. ***Select Resize.***



5 Select the option **Compute Stock size from the size of the part.**



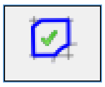
6 Enter the Metric values shown above. Select **Next** and then **Finish**.

7 In **Part View** Select **Stock1** and then Right click and select **Show Stock**.

Curve Creation

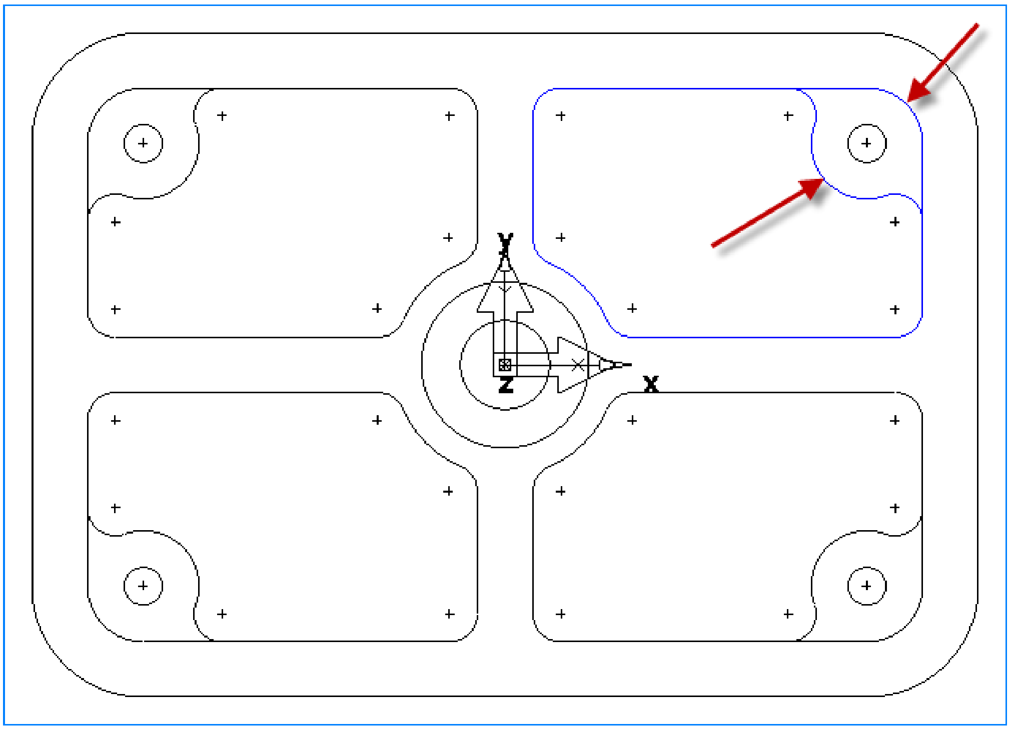
*We have the ability to give curves a unique name; this can be beneficial if you have the time. As we progress through this exercise we will give the Curves relevant names for each **Pocket** or **Boss**.*

*The best view to look at our part is **Top View** or **Ctrl + 5** please select this view.*

8 Please select the **Closed Curve Icon**  from the **Geometry Menu**.

Tricks and Tips

Please see the example below showing a selection point. This is a good point to create a closed Curve in our example. Apply this technique to all pockets.



9 We will now select the Outer profile of our Part. A single click will now create the Curve. Rename the **Curve** as **Outer_Large_Boss**.

10 Then Select the Larger center Circle and call this **Inner_Small_Boss**.

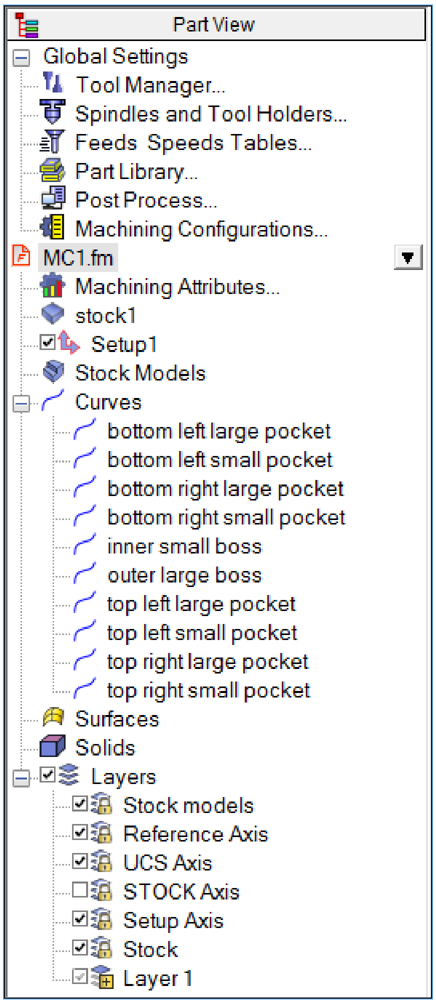
11 If you look in Part View you will see the names of the curves you have just created.

12 The next Curve we will create is from the larger top right pocket.

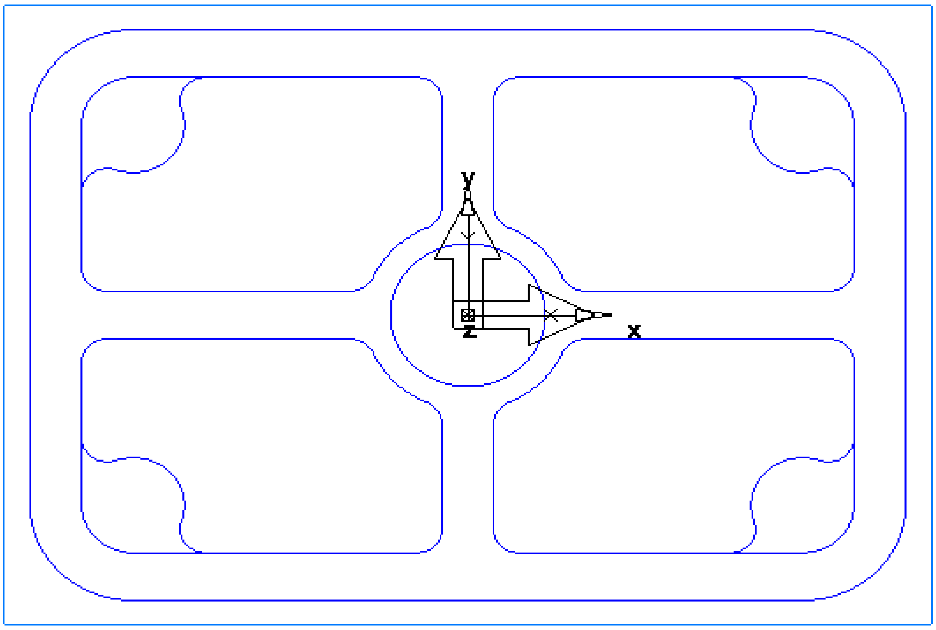
13 Please select this pocket and then move in a counter clockwise direction selecting all of the similar pockets.

14 The larger top right pocket will be called **Top_right_Large_pocket**

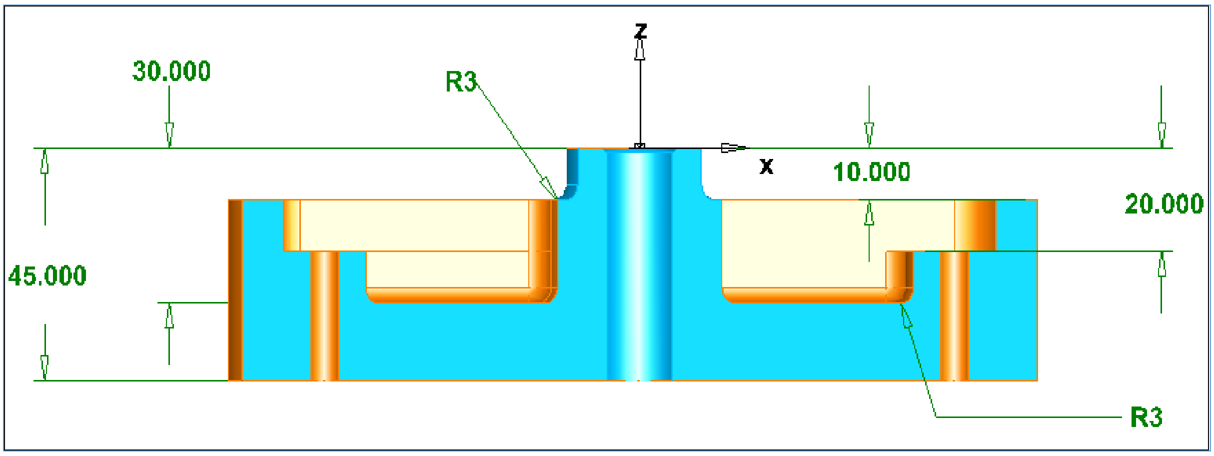
15 Work your way through all of the profiles. The naming convention should look like the image shown on the next page **3.13**



*The image below shows all of the curves we have just created. This has been achieved by **Hiding All Geometry** from the **View Menu.***



*The image below shows a cross section view of **Construction Exercise 2***



*Bottom Radius of the deepest pocket is **3mm***

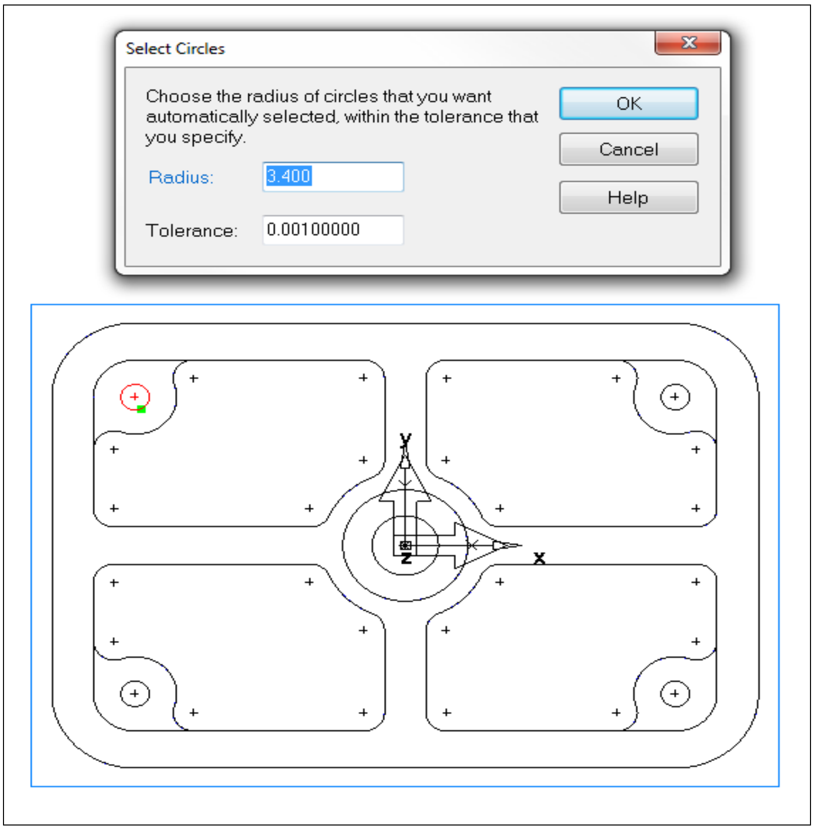
*We will now move the smaller **4 circles** in **Z** to **Z-20mm***

16 Select **Show** and then **Show All** from the **View Menu**.

17 Hold down the **Ctrl key** and select the four circles one at a time as shown.

18 Or go to **Edit/Select Circles**. Select the **Hyperlink** in Blue and pick one of the smaller circles.

19 Select **Ok** and all four circles will highlight.



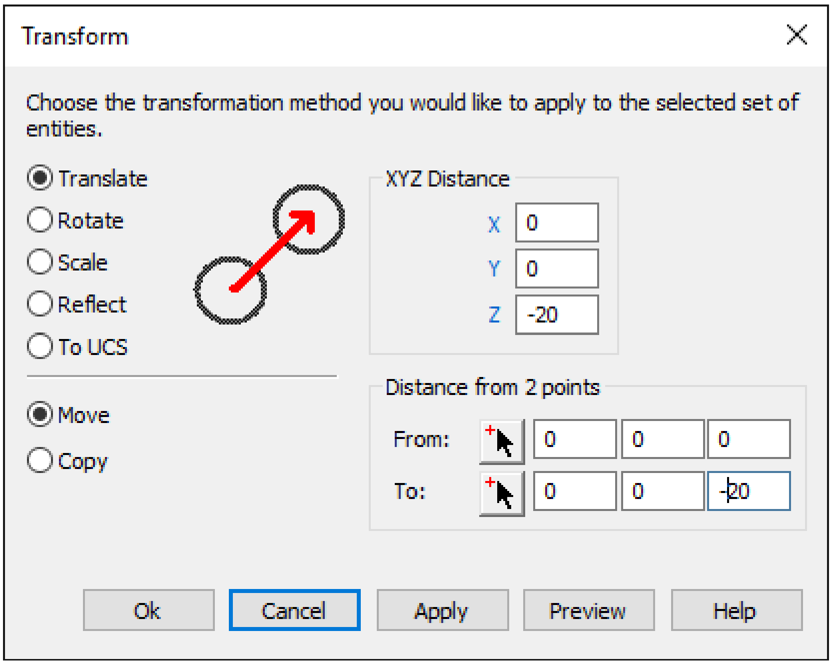
20 **Go to** Edit/Transform.

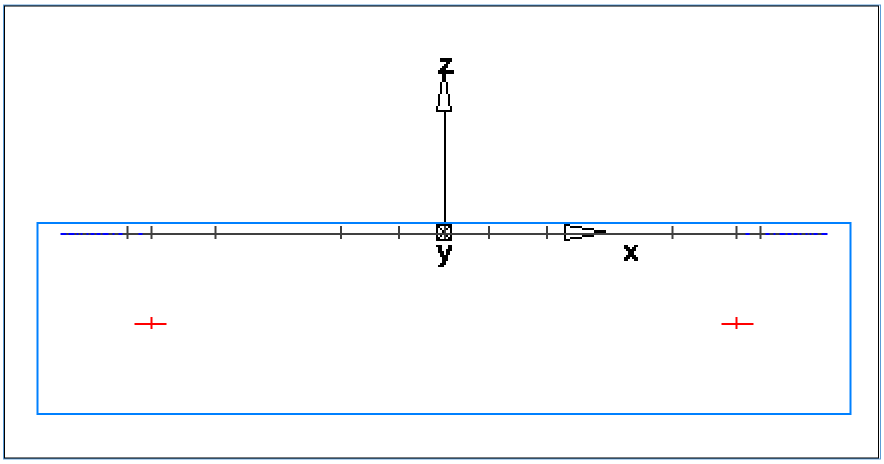
21 Then select **Move** and **Translate**.

22 We will be moving all four circles in **Z Only** to **Z-20mm**

*By translating the geometry when we come to drill the holes this will automatically start from **Z-20mm**.*

*See the **Transform** menu below.*





We are now ready for machining. This will be covered in the next chapter.

Save the File as **MC1.fm** in your instructors chosen location.

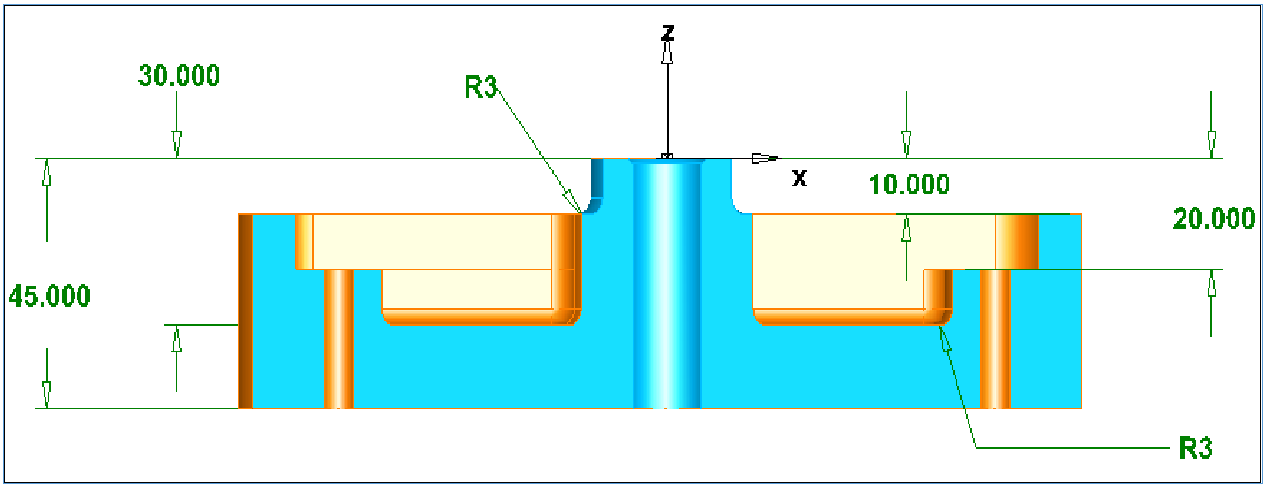
This page is intentionally left blank

Contents

Group exercise from Curves (Class Exercise)

Open File **MC1.fm** with Curves that we created in the Basic Curve Creation chapter.

*The image below shows the depth dimensions for our part. Bottom pocket radius is **3mm** and **Boss bottom** radius is **3mm.***



*We will machine the component using default settings showing each step as we work through the example. Tool **Crib** is **Training_Crib_Metric.** We will select a Post of our choice at the end of this example.*

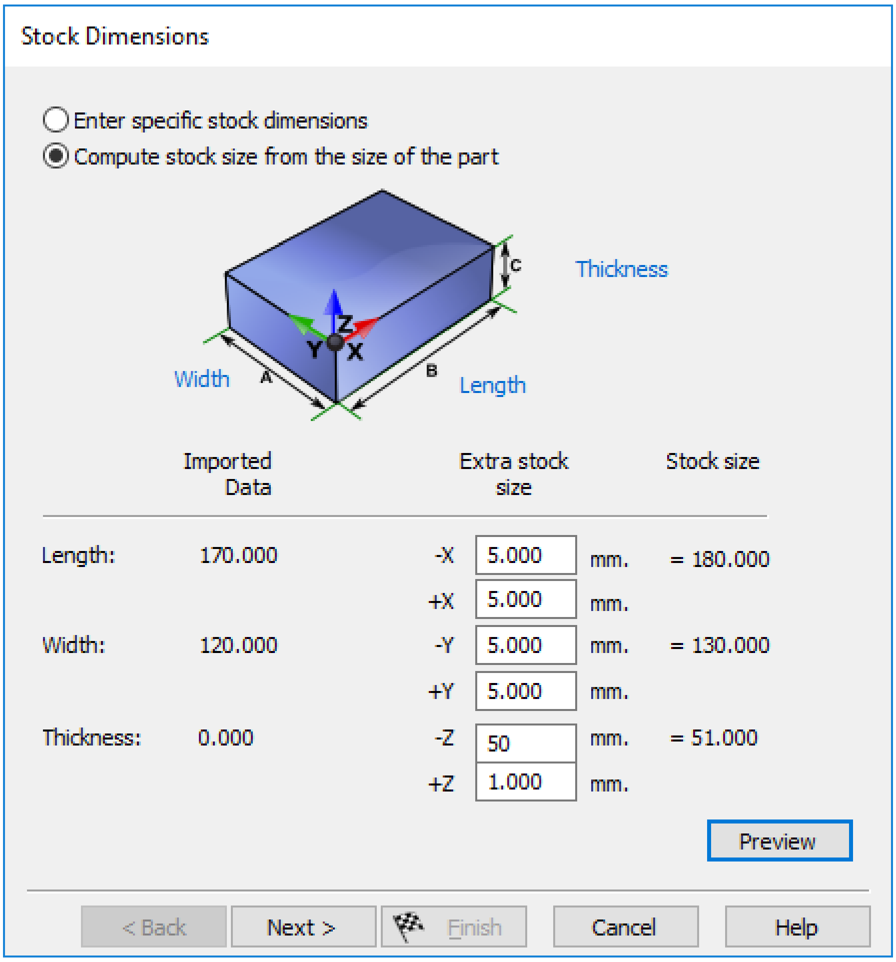
Please be aware that there is more than one way to machine this component. Time permitting your instructor will go through these other methods.

Defining Stock Parameters

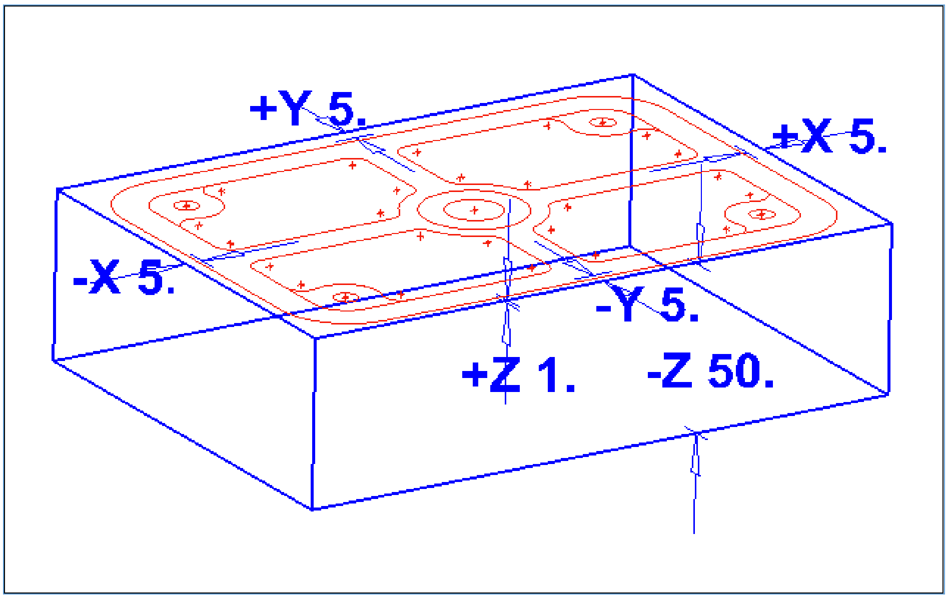
Before we can start creating our machining sequences we need to define our stock around our part. This may have been created in our previous chapter.

If not then please follow these instructions.

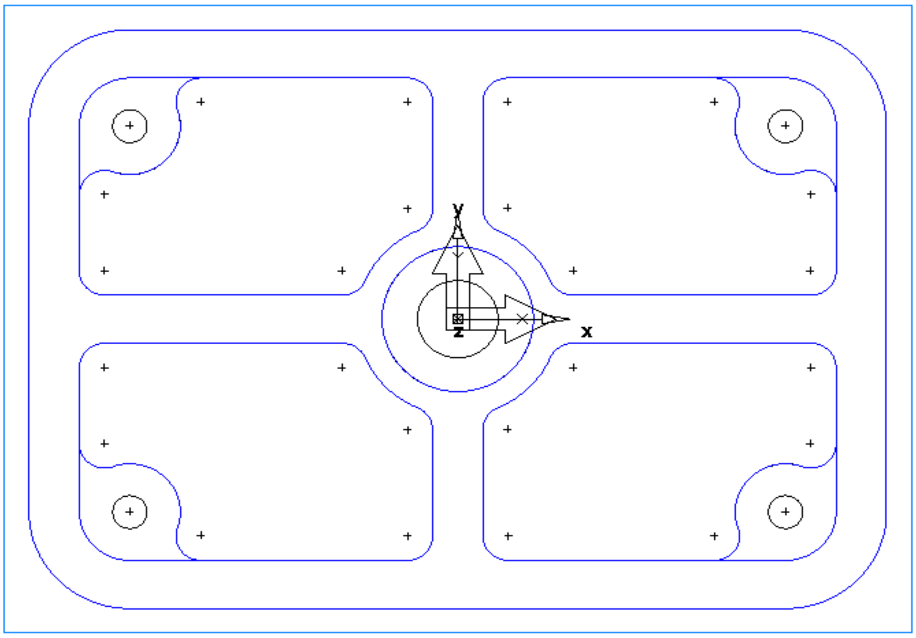
1 Show **Stock** from the **PartView** menu by right clicking and selecting **Show Stock.** Double click on the **Stock** and the following menu will appear. Please select **Compute stock from the size of the part.**



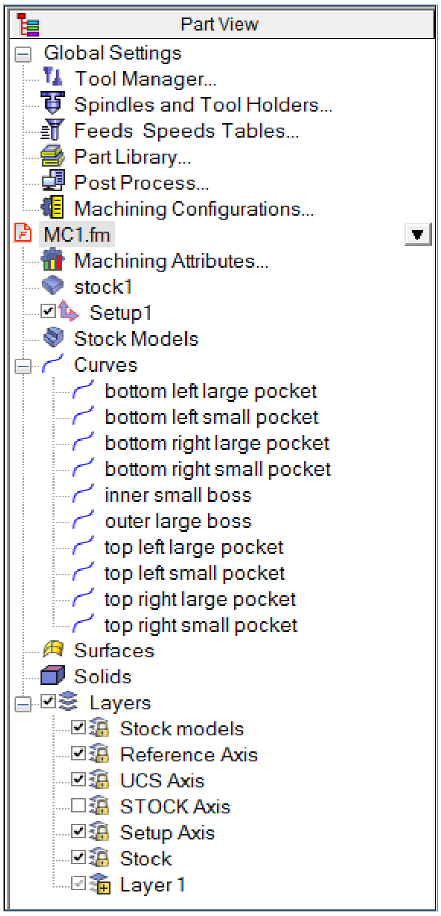
2 Enter the following dimensions into the respective fields on your computer. The reason for the extra thickness is so we can hold the part in a vice on the extra **5mm** of material.



The image below shows the ***Curves*** *we created in the previous chapter. Plus* ***Stock*****.** ***Setup1*** *is from* ***Centre + Top*****.**

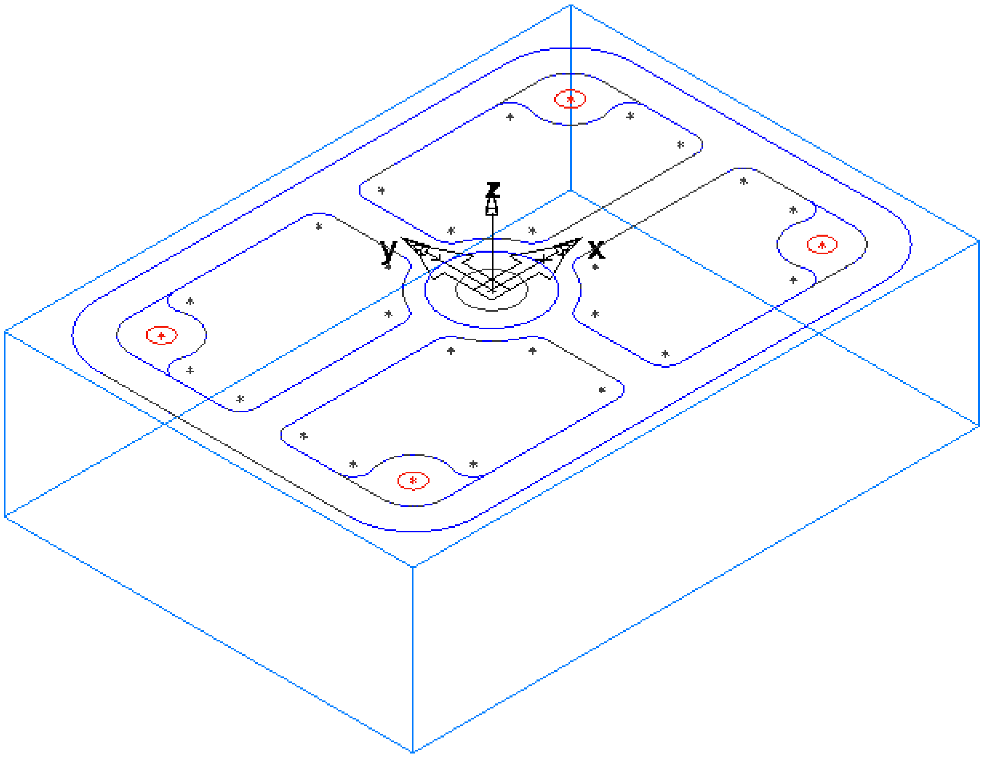


We have all the ***Curves*** *listed in* ***Part View*** *as shown below. If not please create your curves.*

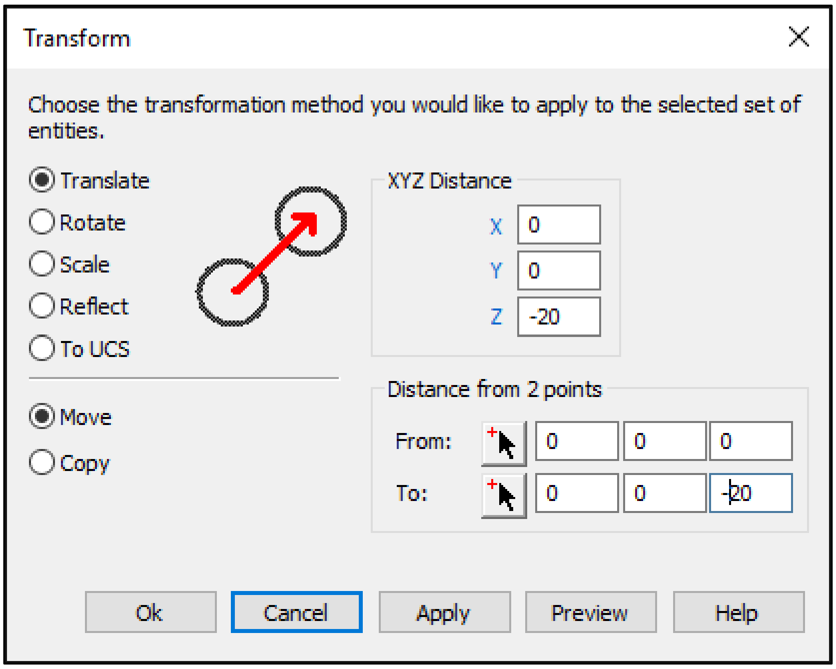


*We will have to move the 4 corner holes (6.8mm dia holes) in Z to **Z-20mm**.*

3 Select **Edit>Select Circles.** Select Radius which is a **hyperlink** and select one of the **6.8mm dia holes circle diameters**. Select **OK**. By selecting **OK** all of the same diameter holes will then be selected ready to create our **Hole** feature.

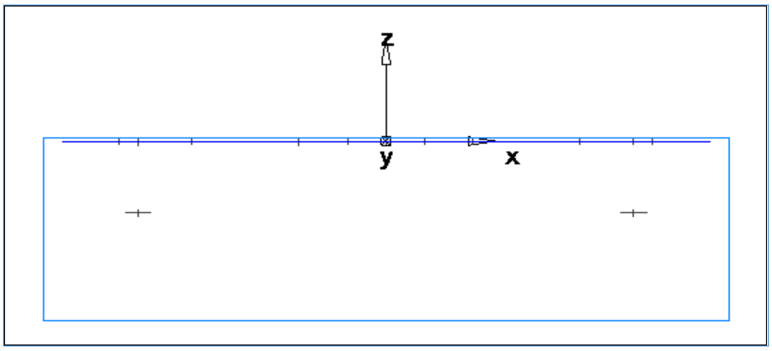


4 Select **Edit>Translate.**



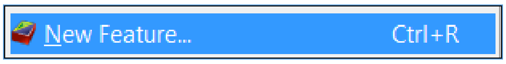
*You can see from the image below that the **6.8mm diameter** holes have moved in **Z to Z-20mm**.*

5 Please select **Hide all Geometry**.



*The first operation will be to machine the top **Face**. There are four ways in which to activate the **New feature** menu.*

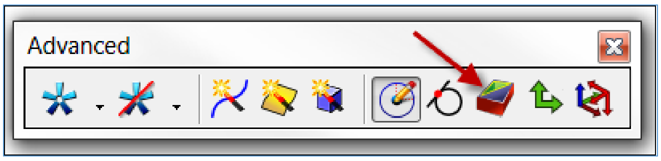
6 Select **Construct>New Feature**



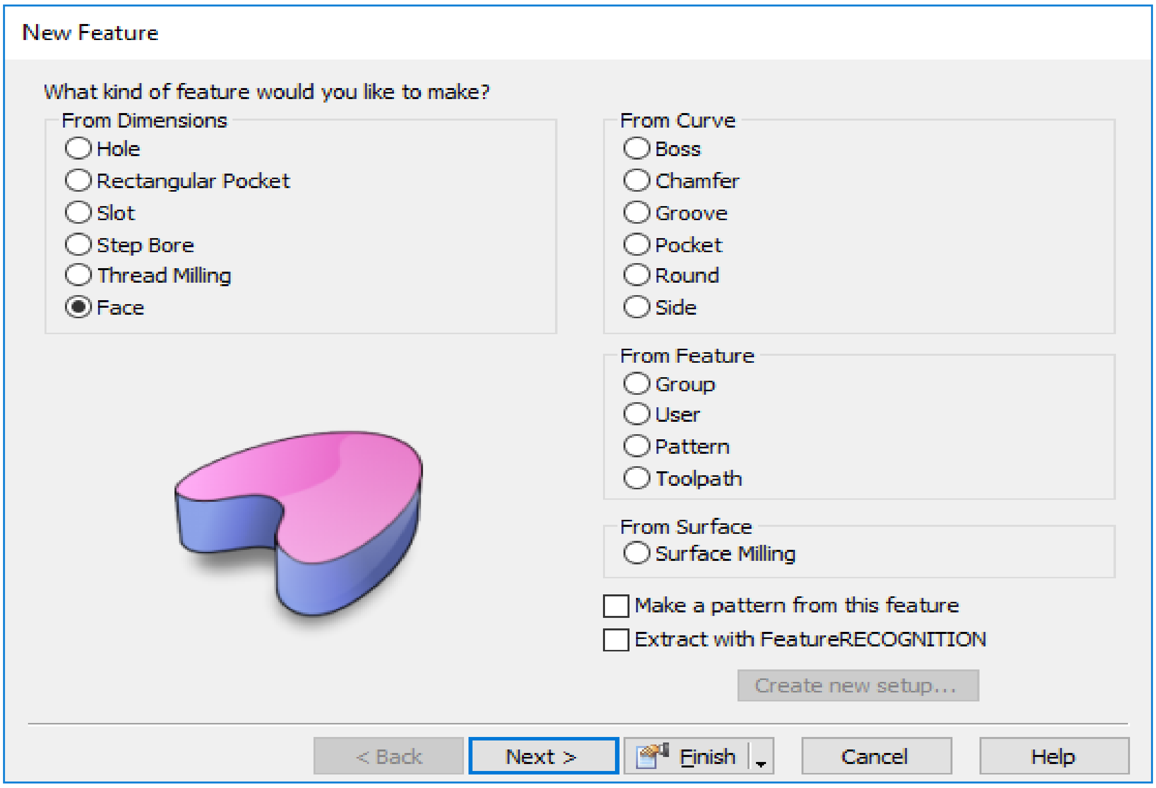
7 Select **Ctrl + R**

8 Select **Steps** and **Features**

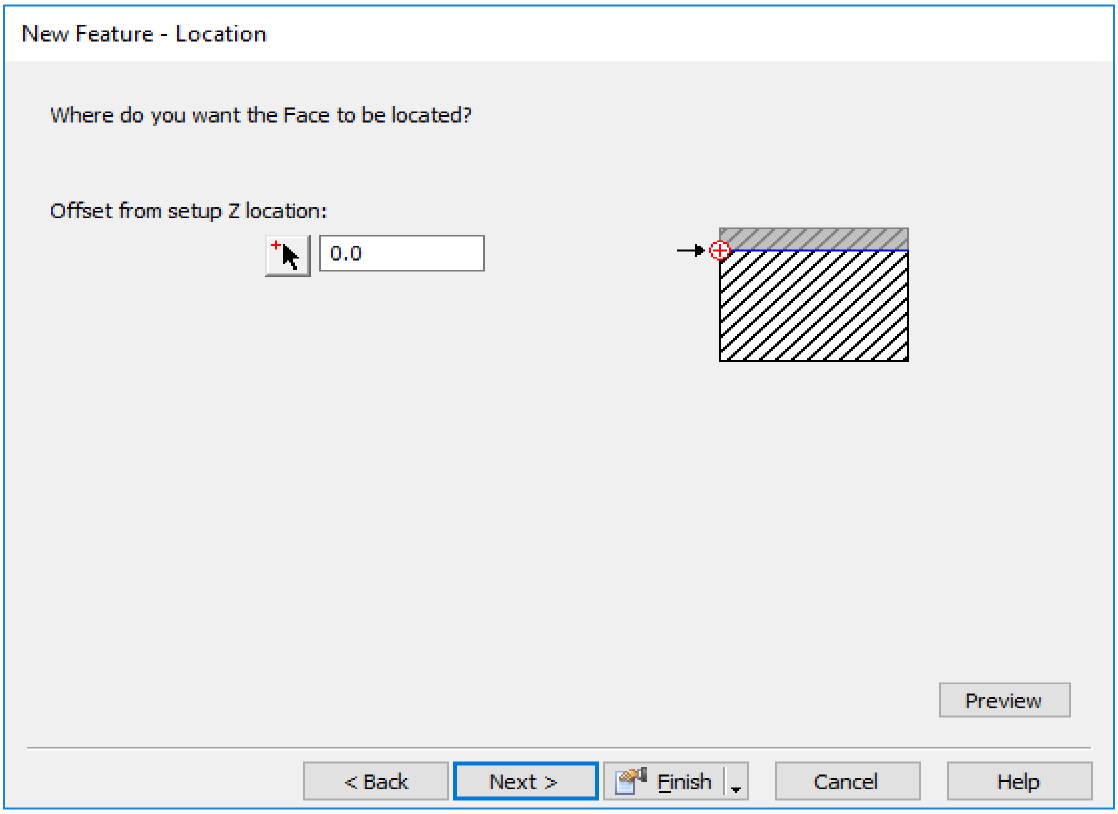
9 Select **New Feature** from the **Advanced Tool bar** to create a **New Feature**.



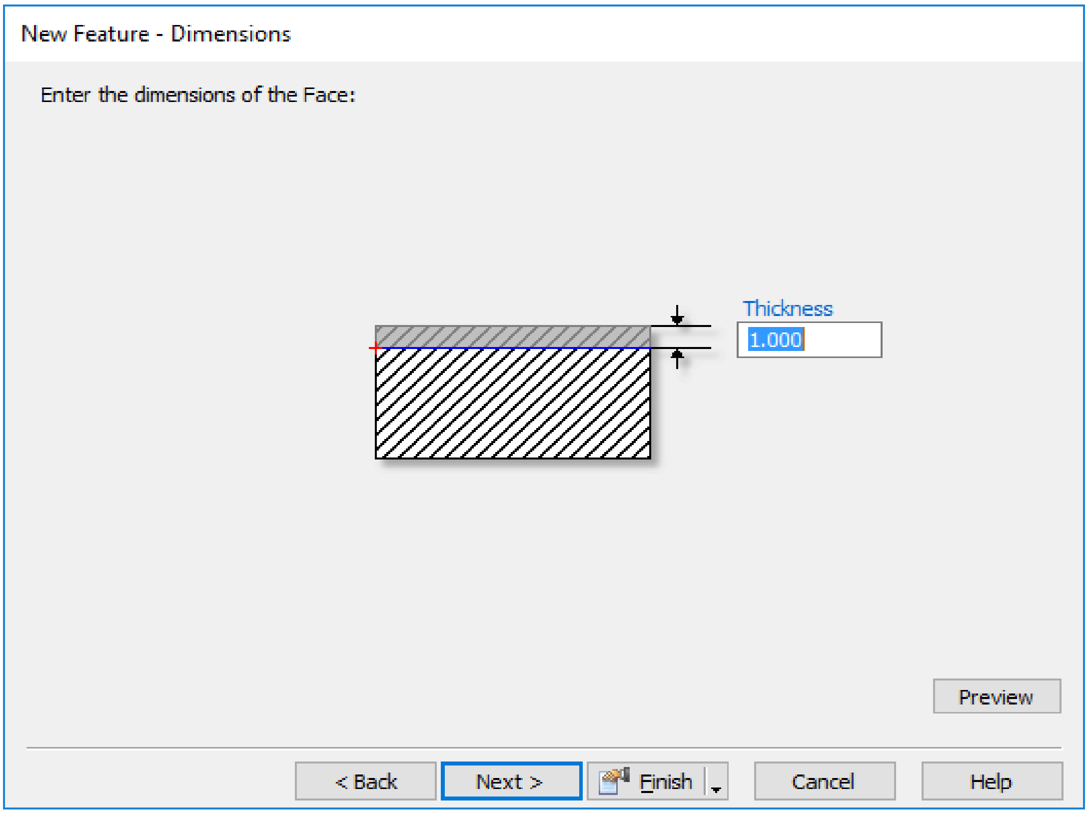
10 Select **Face.**



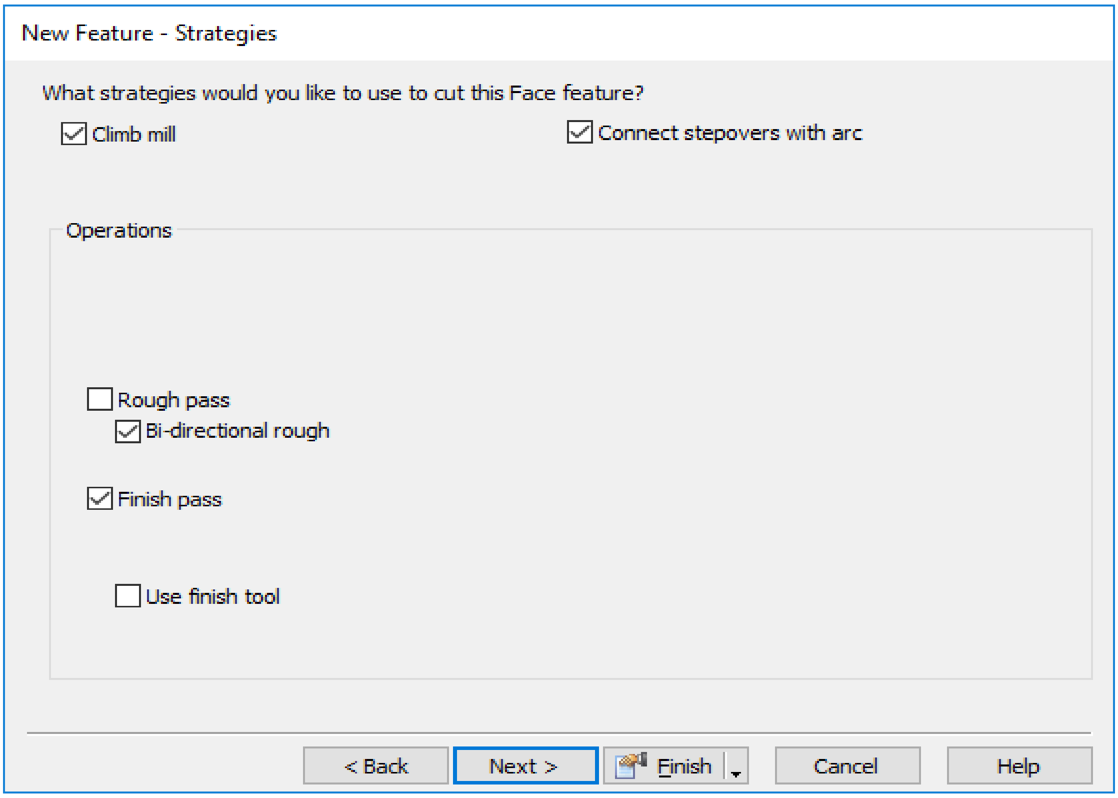
11 Select **Next** to show the **New Feature>Location** menu.



12 Select **Next** to show the **New Feature>Dimensions** Menu.

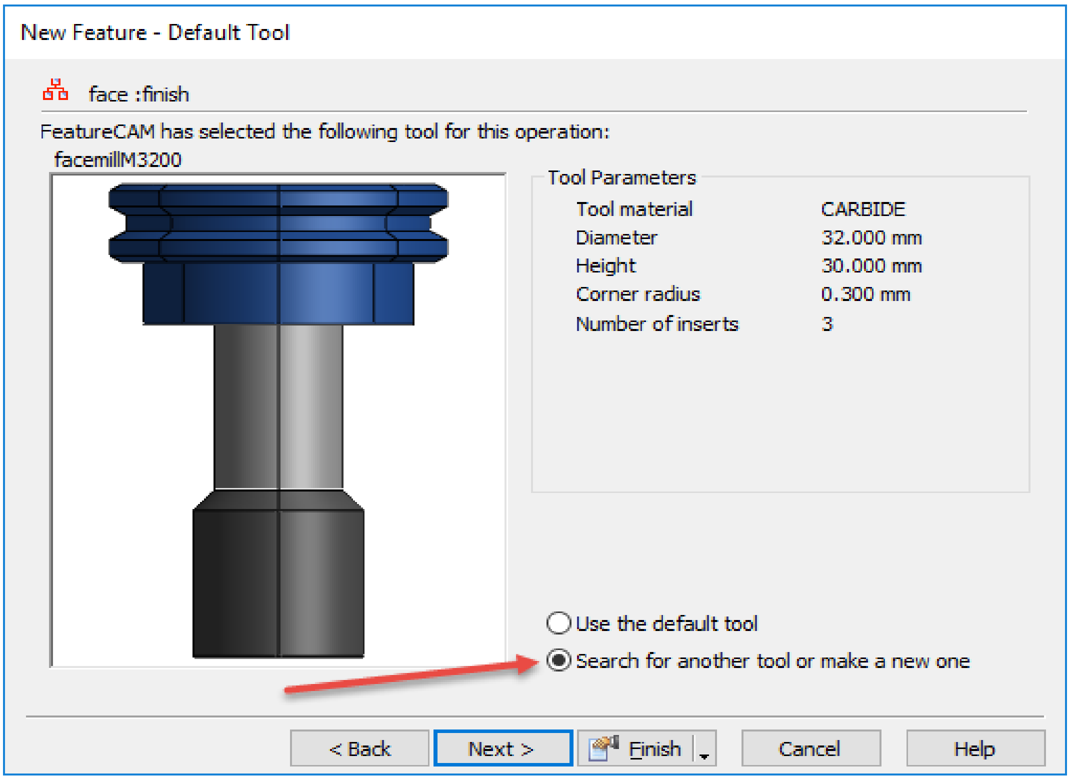


*Accept the current values. Select **Next.** The following menu will appear.*



Here we have the option to ***Connect Stepovers with arc****. This will increase the machining time but give a smoother transition on and off the work piece. We also have an option to select* ***Rough pass****. This is ideal if you have a lot of material to take off. For our example just leave it as shown.*

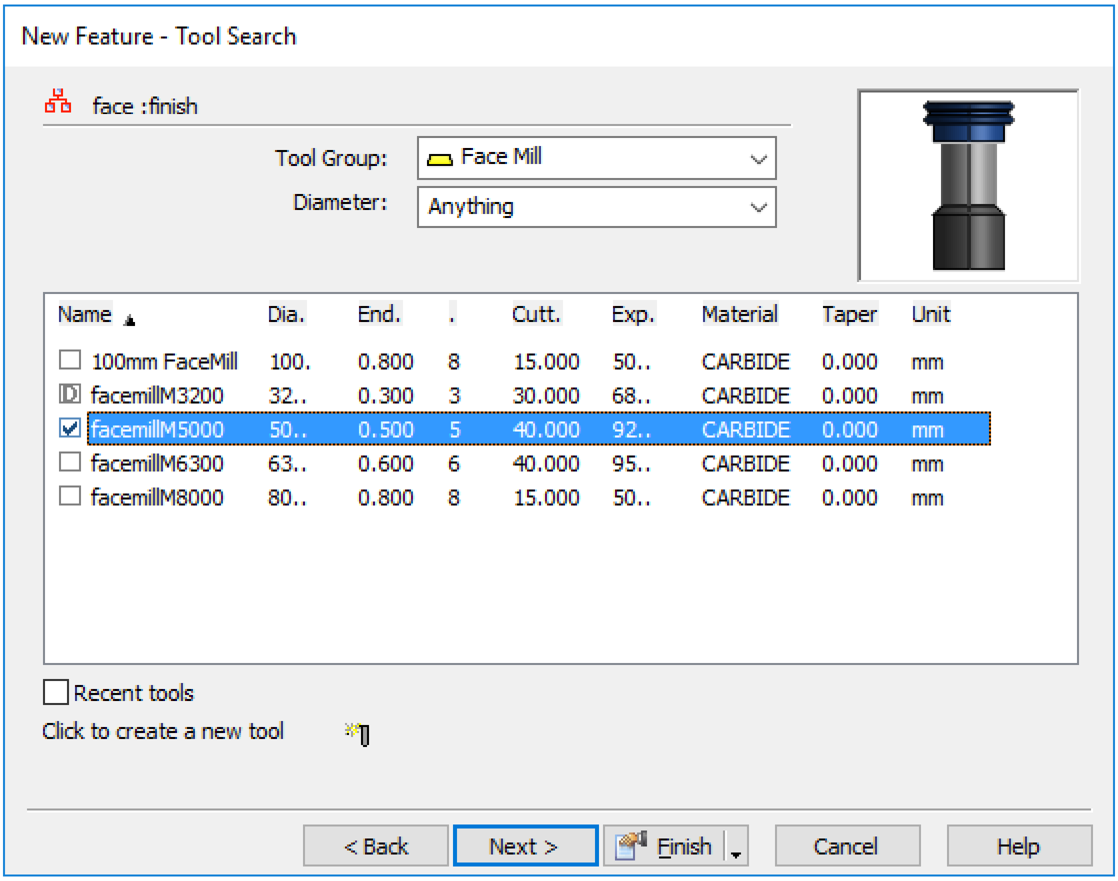
13 Select **Next** twice.



***FeatureCAM** will give you the option to keep the tool selected or **Search for another tool** or **make a new one**.*

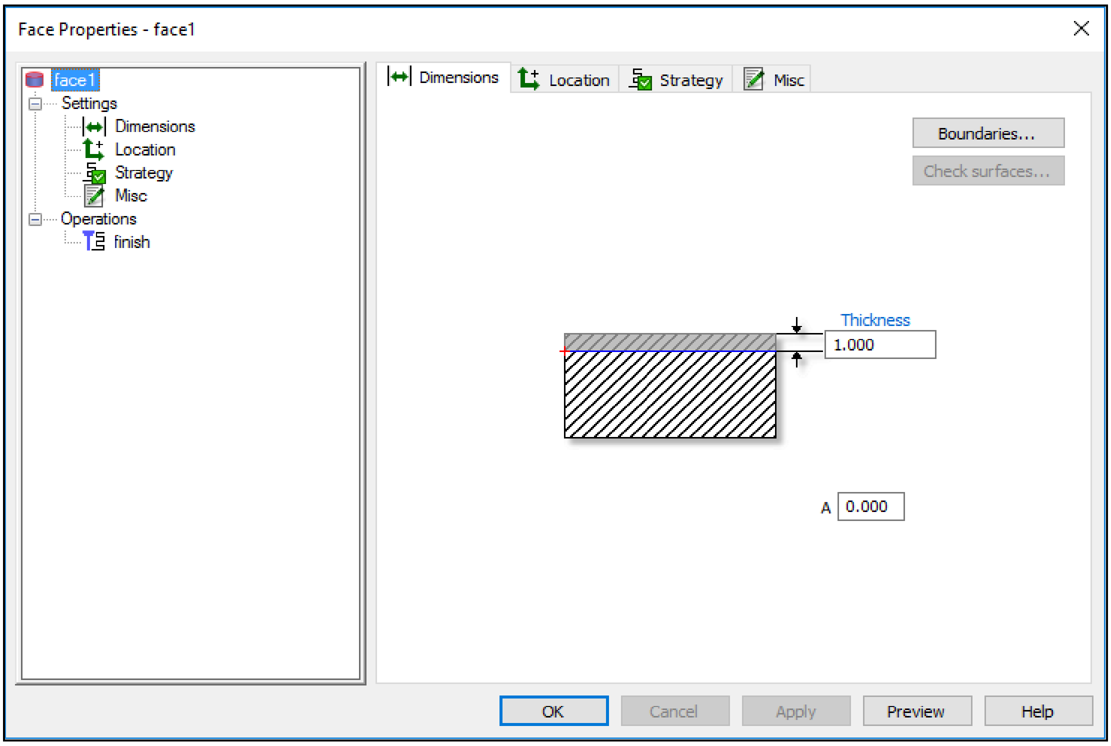
*Select the lower option. **Search for another tool or make a new one**. Then select **Next**.*

***FeatureCAM** will give you the option to select a tool from the list.*



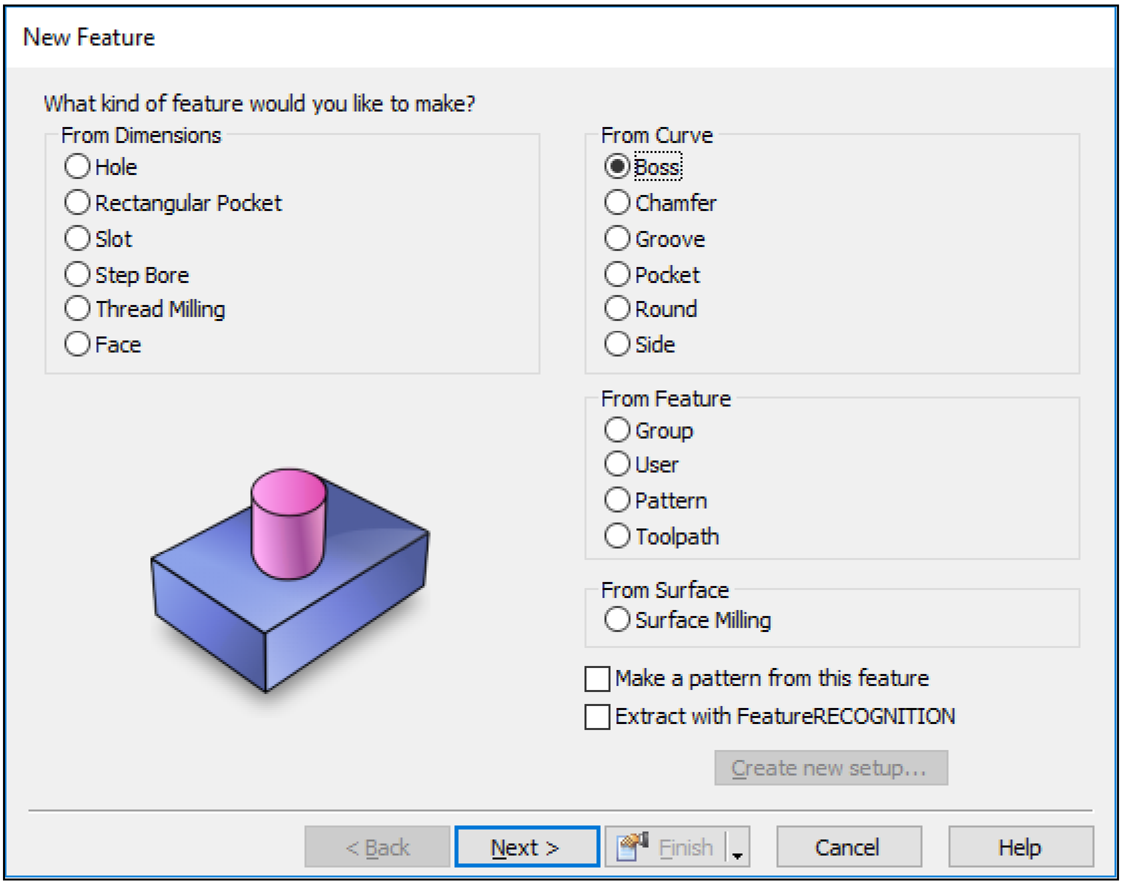
14 Choose a **Facemill** of your choice from the list.

15 Select **Finish** and **OK** to close the menu. D = Default selection.



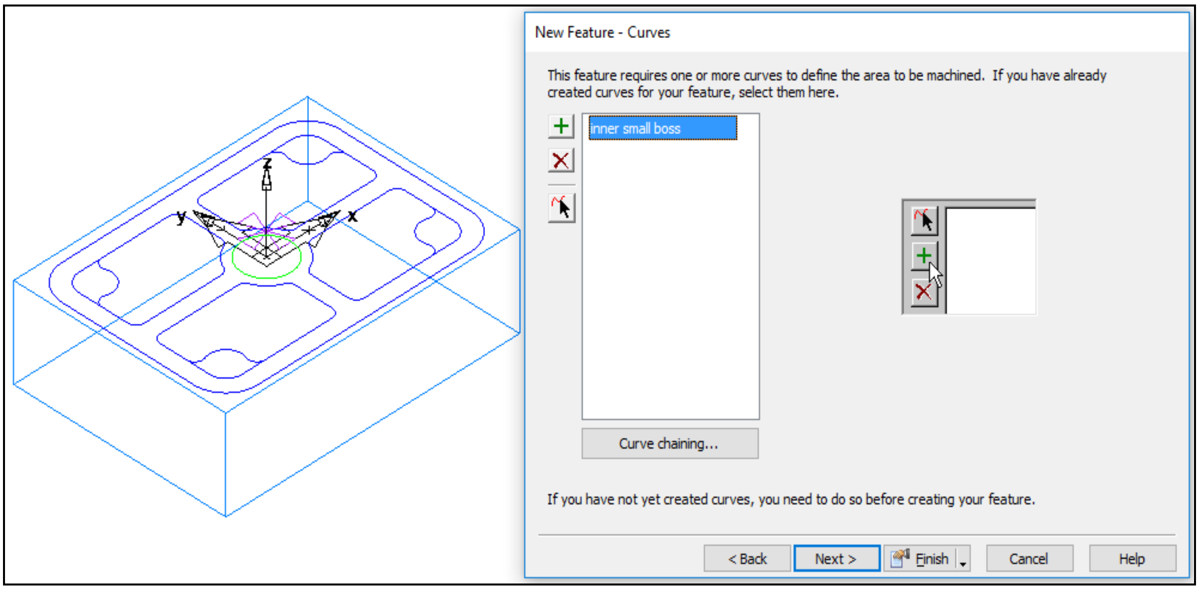
*The next operation will be to machine the **Small Centre Boss feature**.*

16 Select **Ctrl + R** or select **Steps** and **Features** to create a **New Boss Feature**.

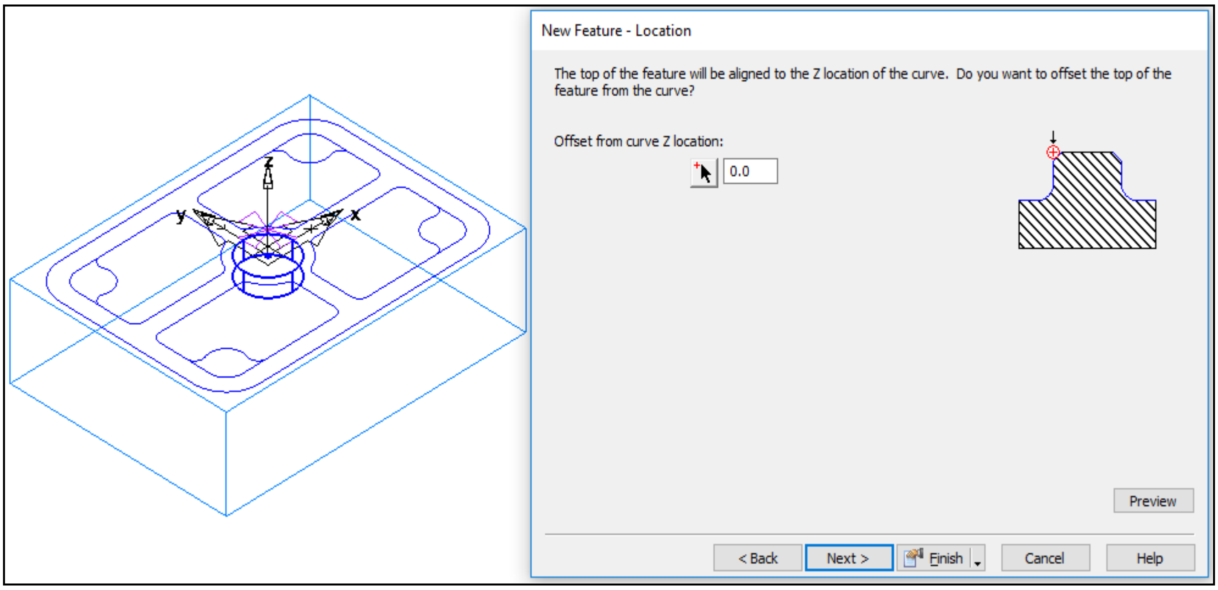


17 Select **Boss From Curve** and select **Next.**

18 Now select the **Curve** by either picking the **Curve** from the Screen using the  Pick Arrow icon. Or by selecting the **inner_small_boss** curve from **Part View.**



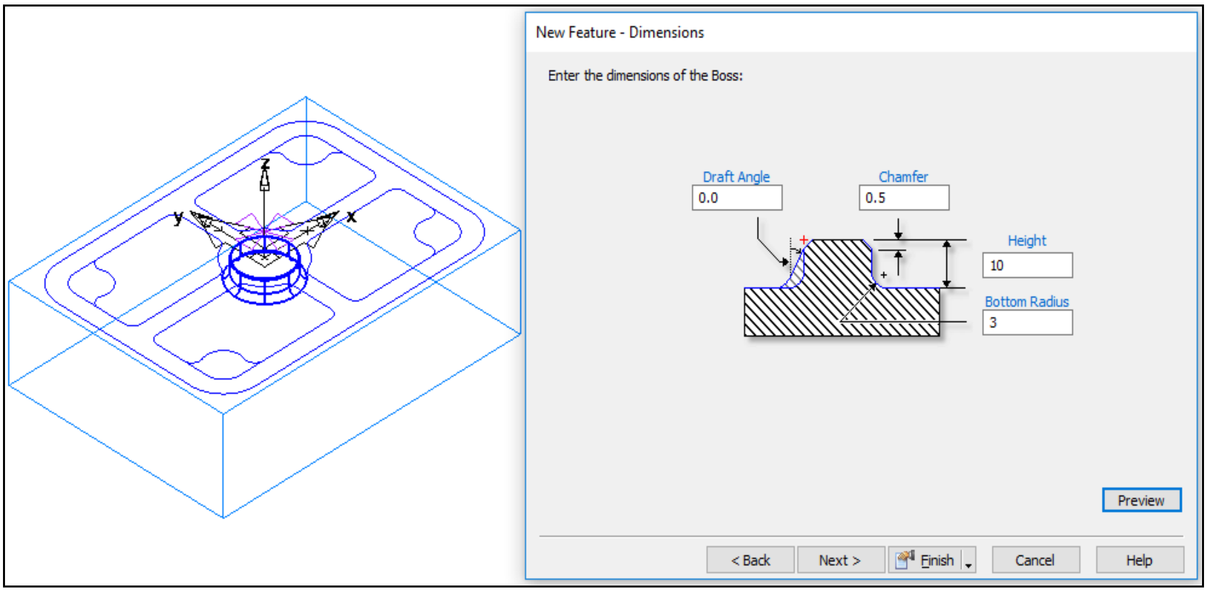
19 Select **Next.** You will be presented with a **New Feature - Location** Menu.



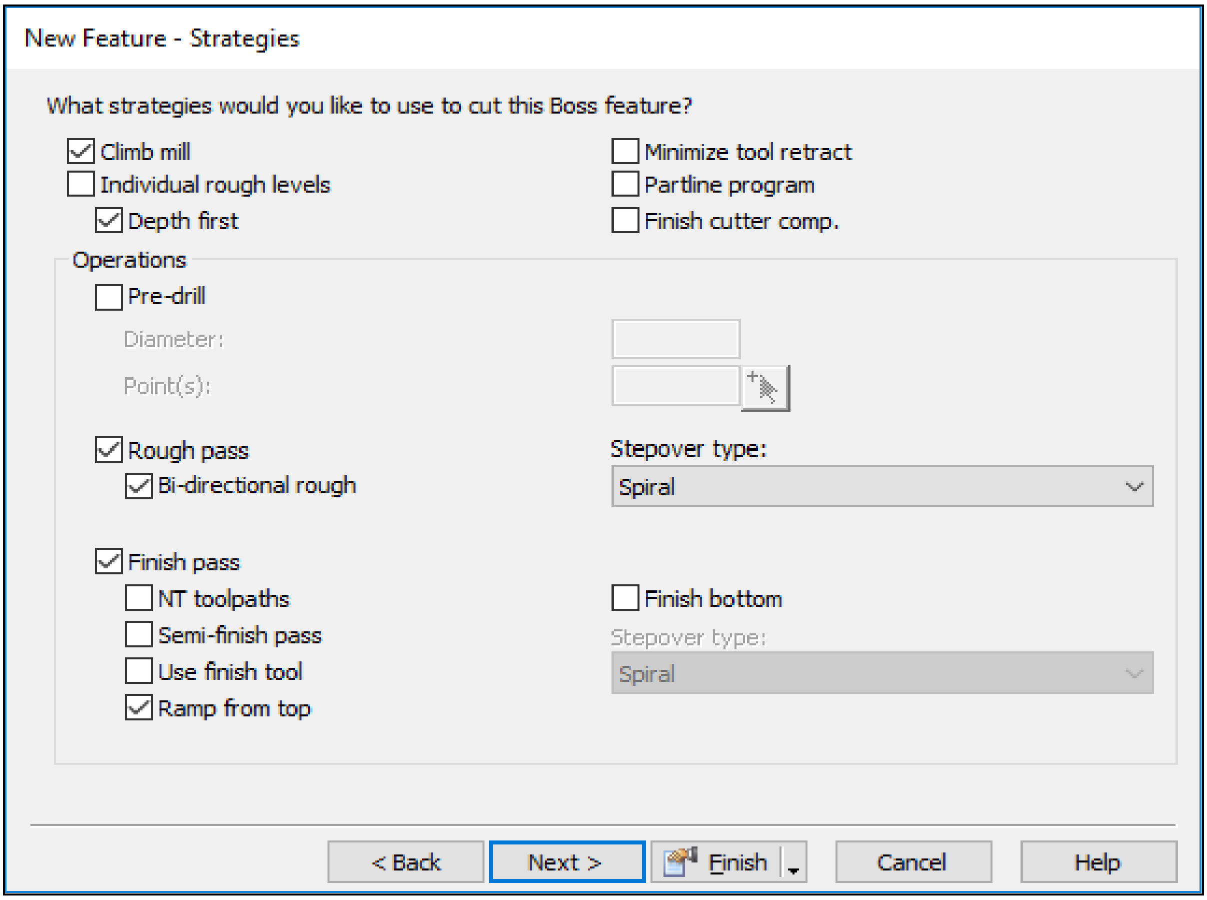
20 Because we are working from the top face in this instance the **Offset from Curve Z location** will be at **Zero.**

21 Select **Next.**

*The Boss information is as follows **Chamfer 0.5mm Height 10mm**. Bottom Radius **3mm***



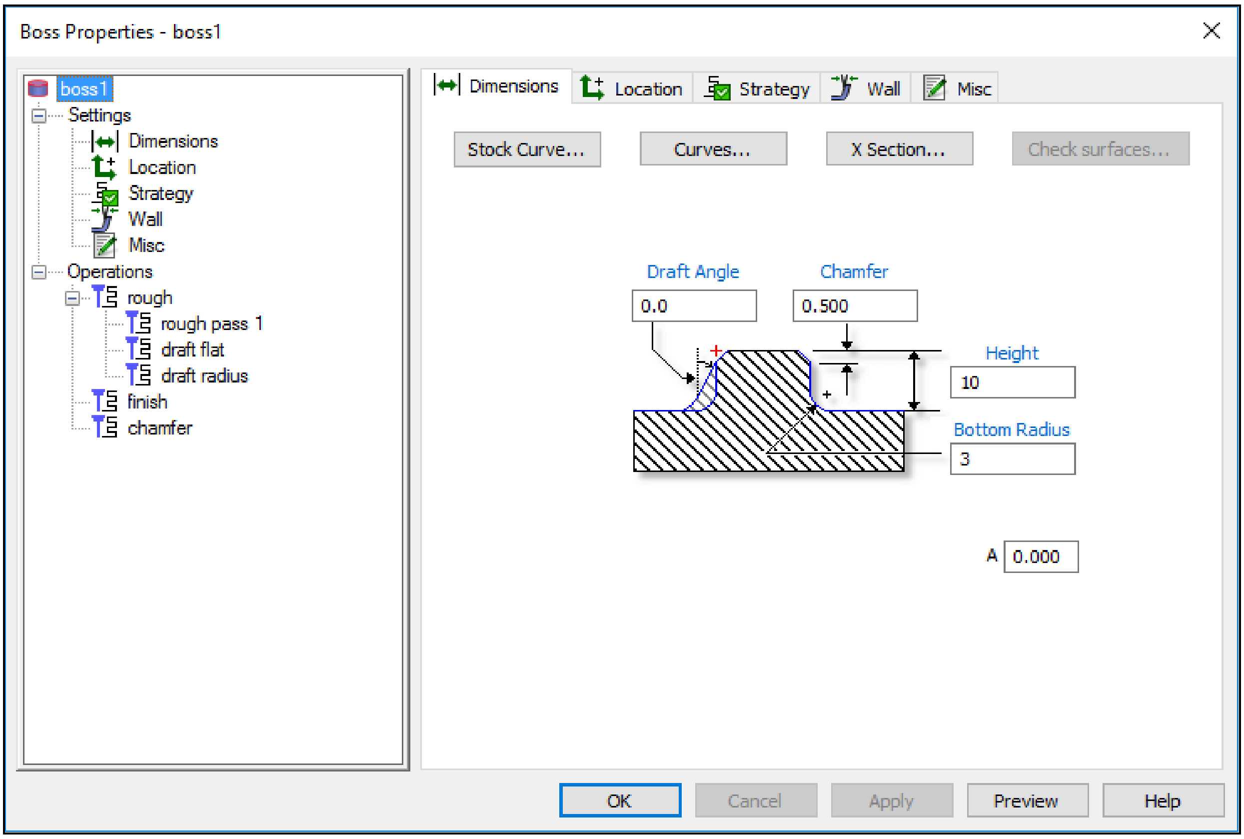
22 Select **Next** and the following menu will appear.



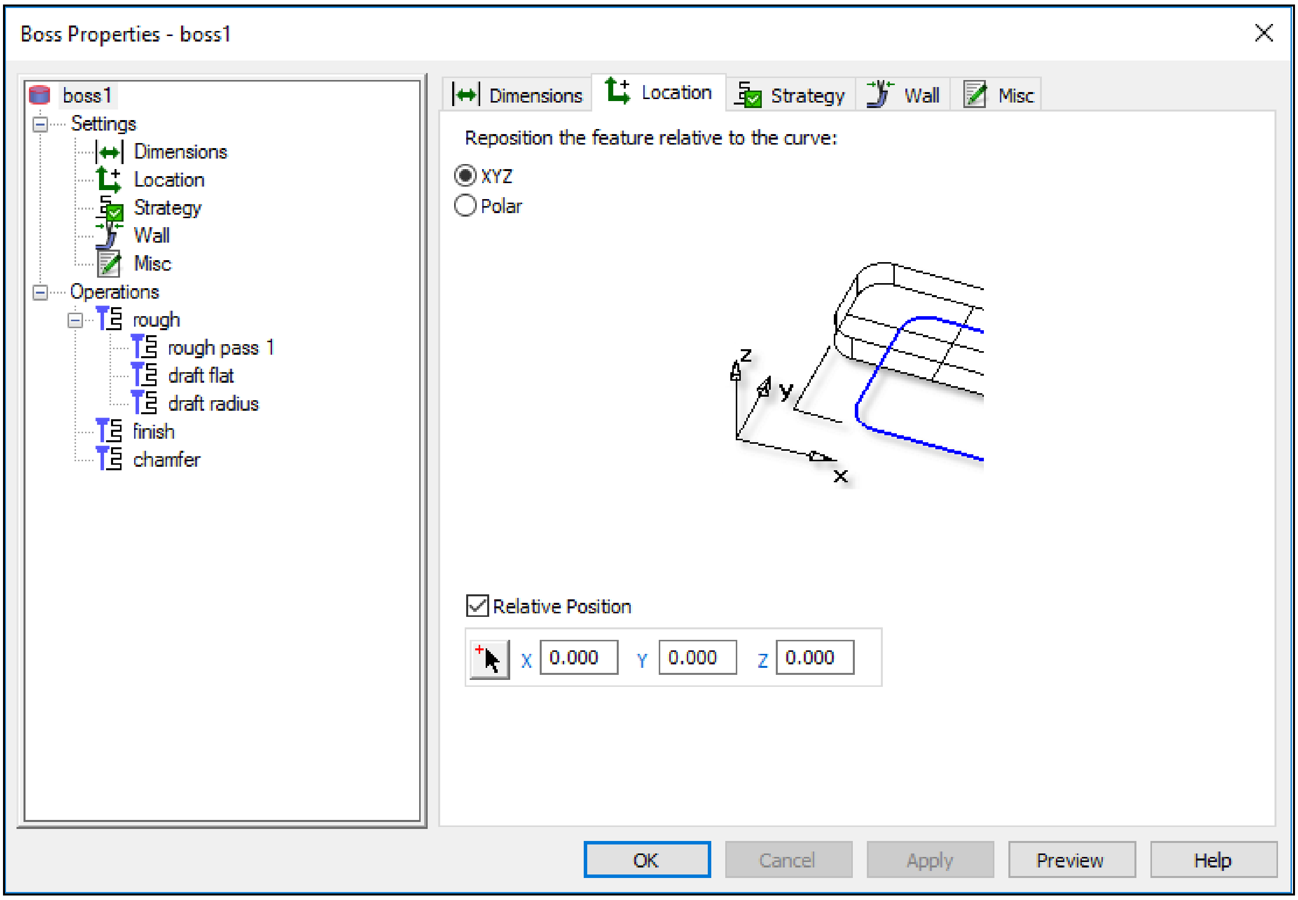
New Feature – Strategies gives you the options for machining the feature in different ways.

Use default settings in this instance.

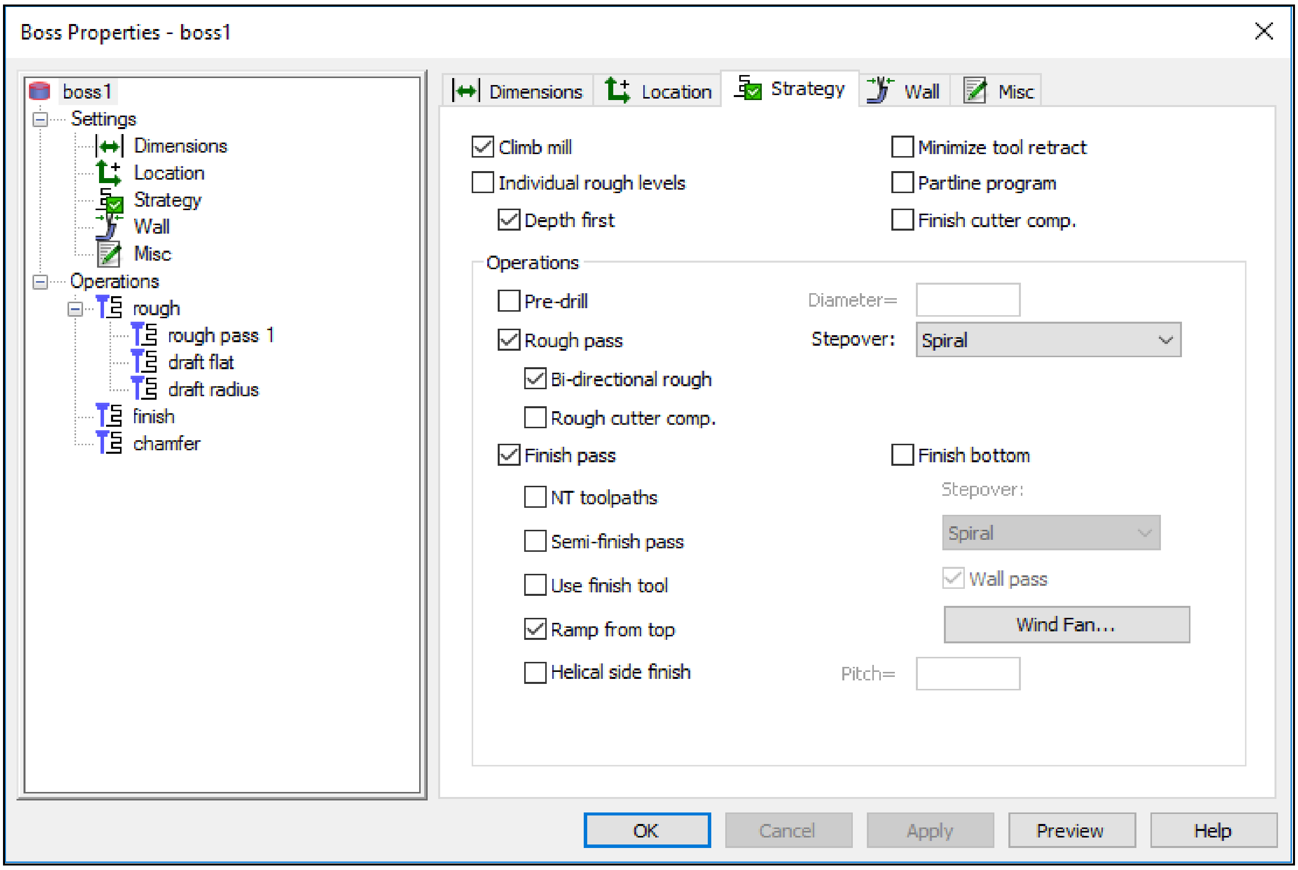
23 Select **Finish** and the **Boss Properties** menu will appear.



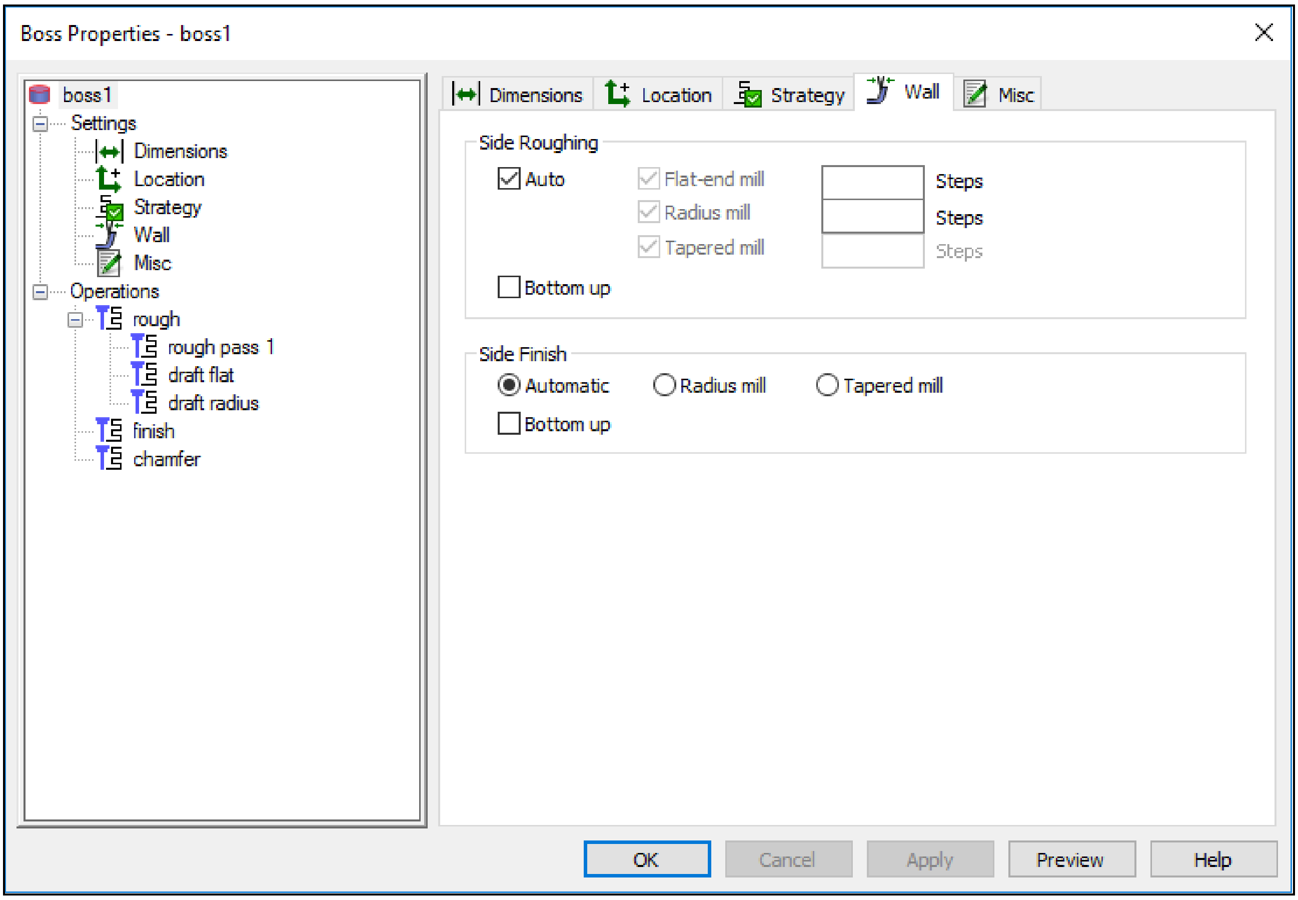
*By selecting the top menus this will give you more options. For example select **Location** and the following menu will appear.*



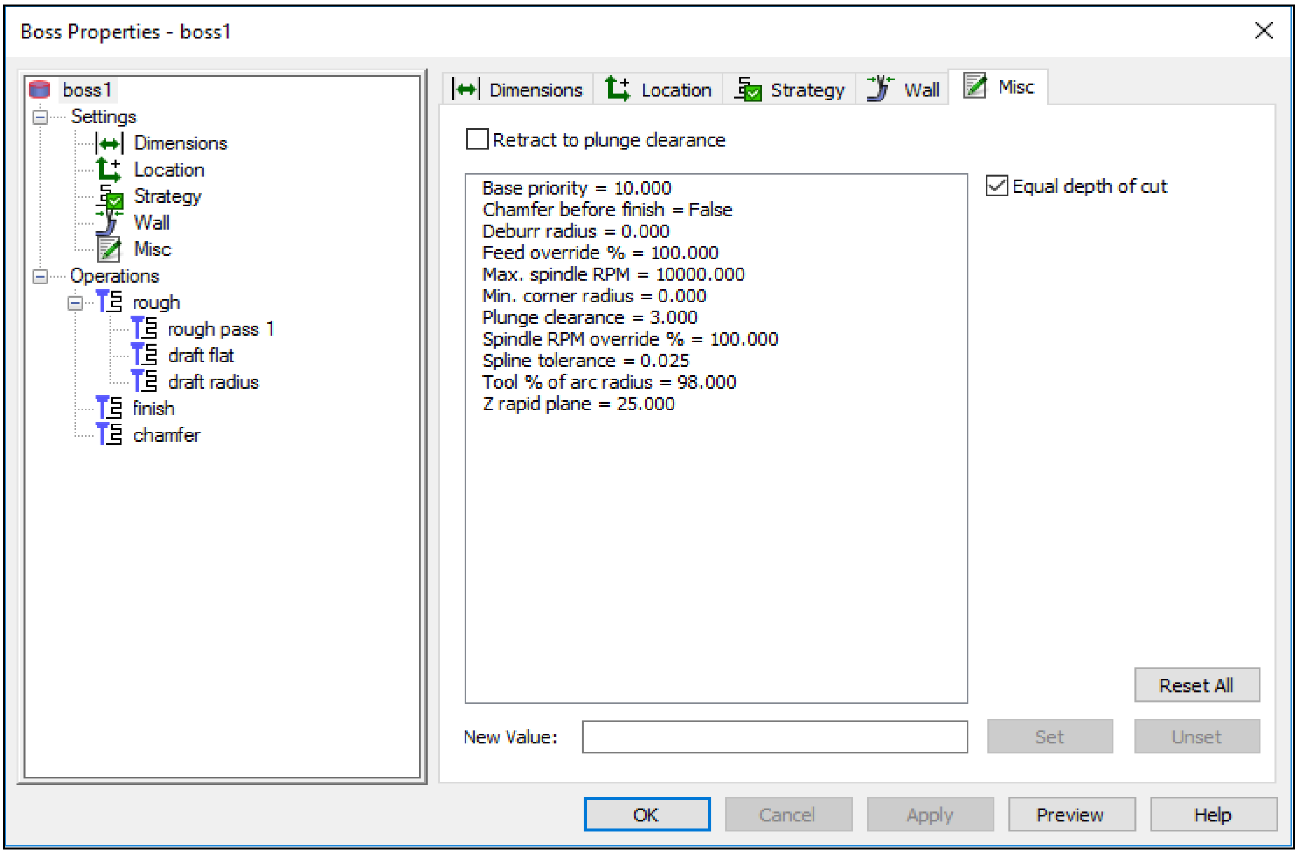
24 Select **Strategy** and the following menu will appear.



25 Select **Wall** and the following menu will appear.



26 Select **Misc** and the following menu will appear.



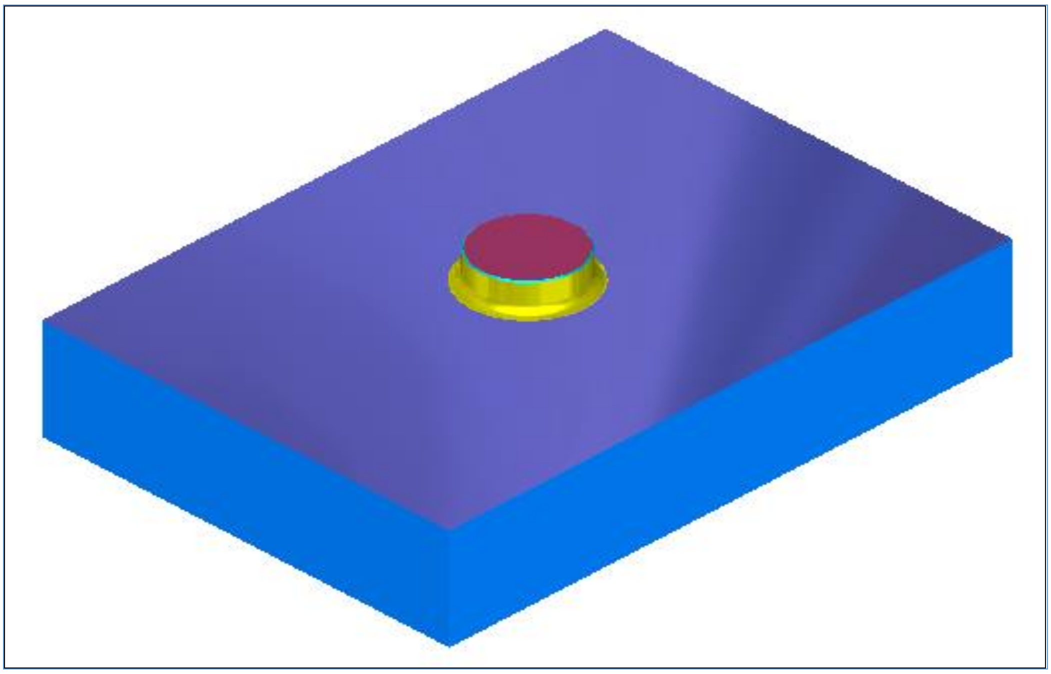
*This is a quick overview of the options in **Misc**.*

Base Priority. Enter the priority that the operation takes in the document. The lower the number, the higher priority the operation takes.

Plunge clearance. Enter the distance above the operation at which the tool feeds

Z Rapid Plane. Enter the minimum safe distance in Z above your part.

27 Select **OK** to hide the menu. Then run **3D Simulation** to view the results.

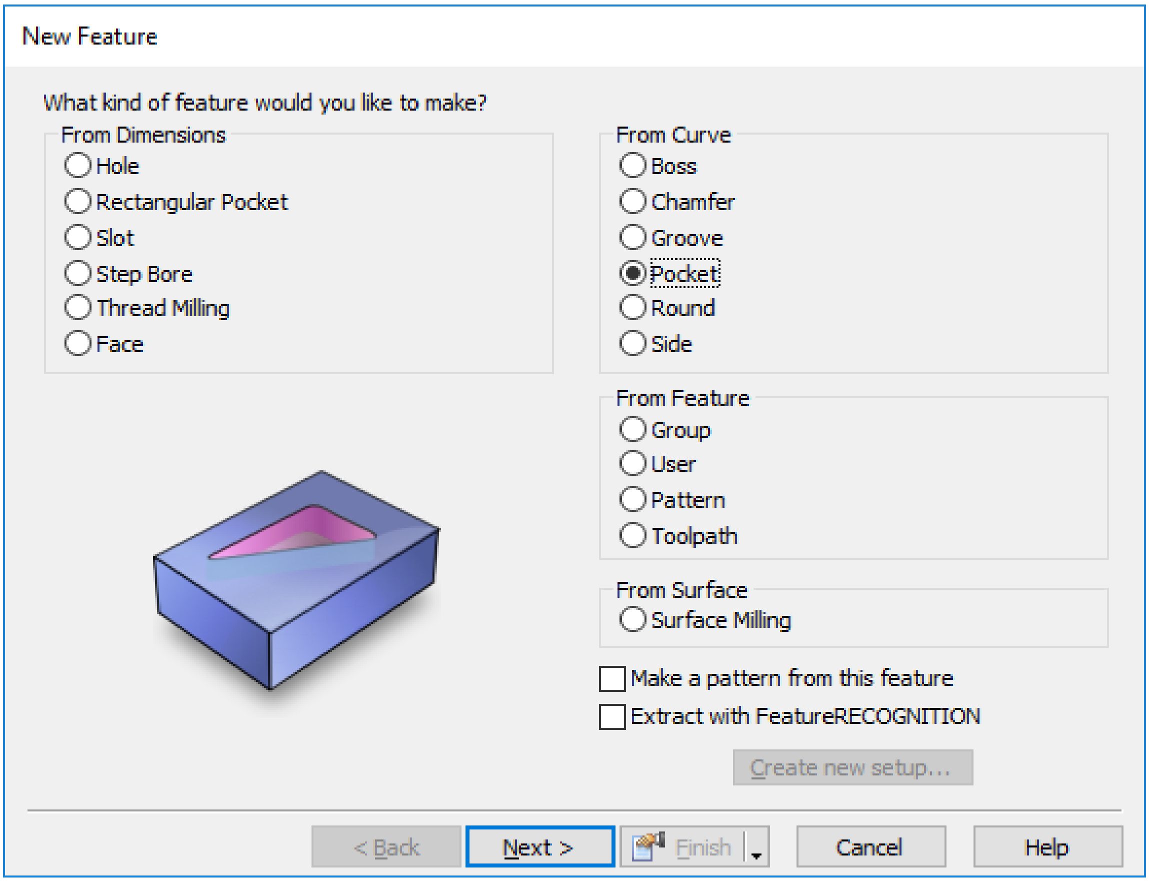


28 Eject the Simulation.

The next operation is to machine the larger 4 pockets

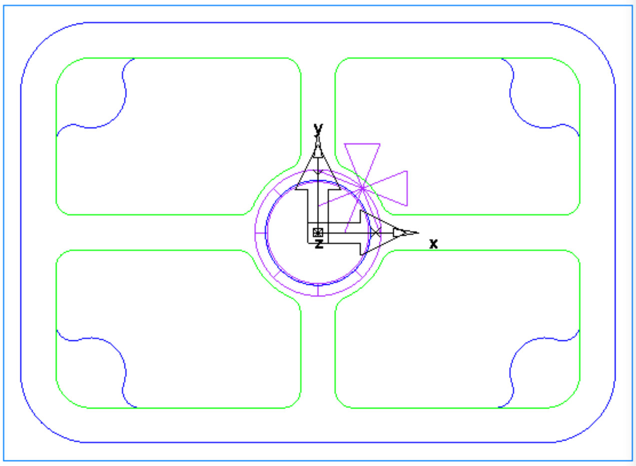
29 Select **Ctrl + R** or select **Steps** and **Features** to create a **New Feature**.

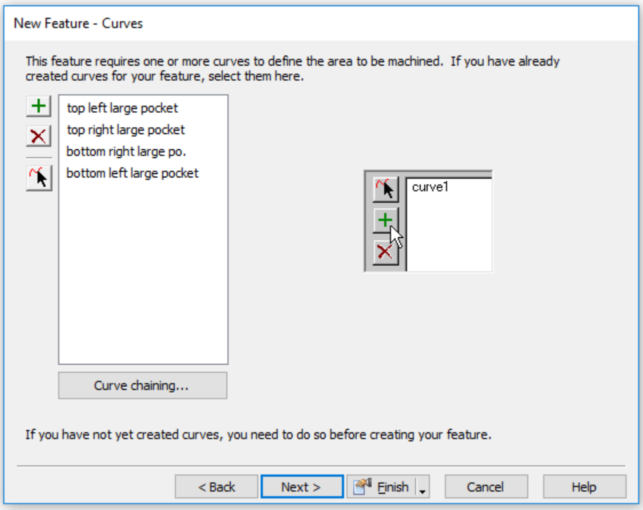
30 Select **Pocket** from Curve.



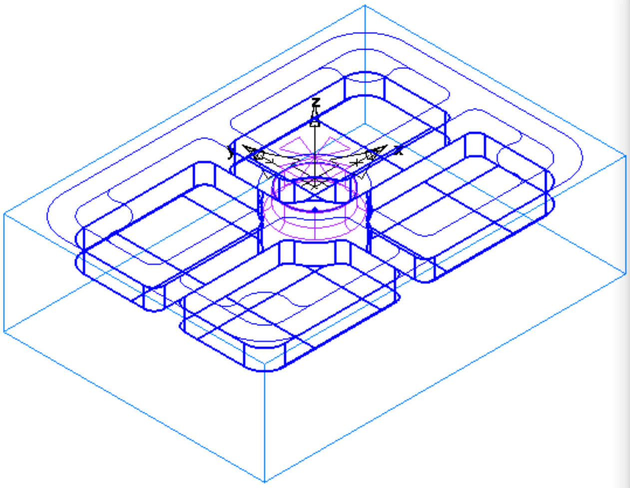
31 Select **Next**.

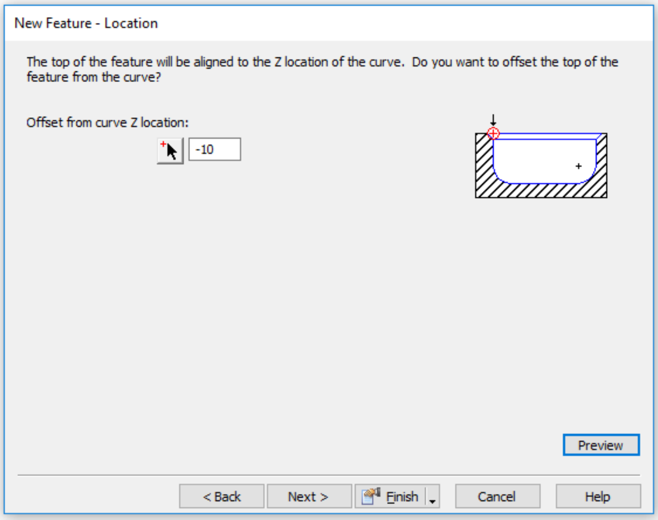
We need to select the four larger pockets. The next screen shot shows all four pockets selected. Showing the names of the pockets we created the day before. You can select them from ***Part View*** *or we can select the Curves manually first selecting them one at a time. For manual selection we use the Pick Arrow.*



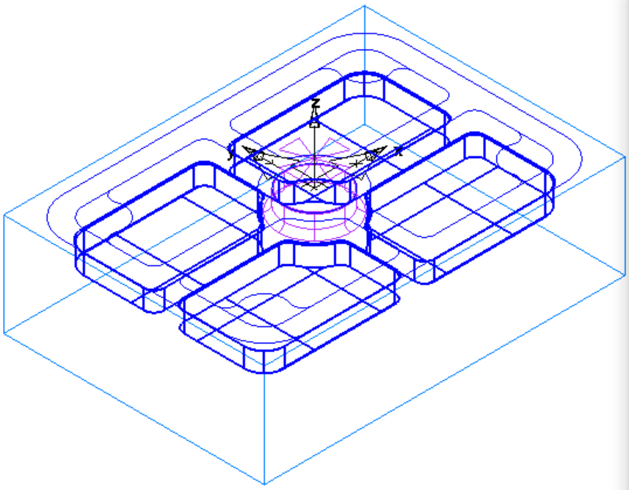


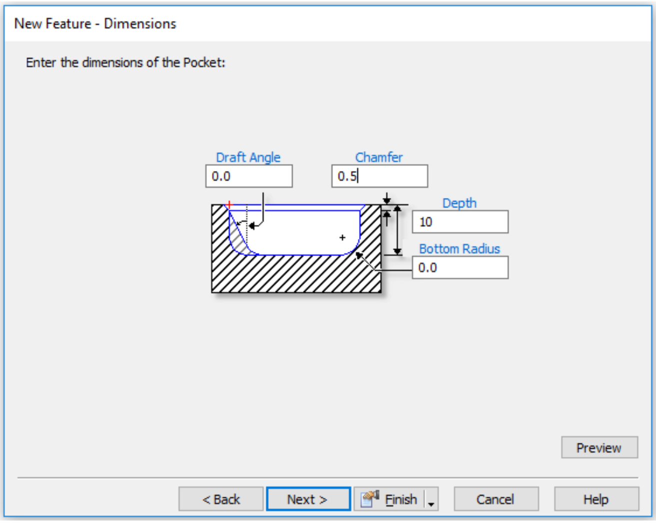
32 Select **Next**. This will show you a menu where you change the **Offset from Curve Z Location** to **-10mm**





33 Select **Next.**





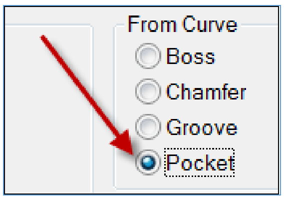
*Pocket information. **Chamfer 0.5mm Depth 10mm***

34 Select **Finish.** This will use default tools from the tool Crib **Training_Crib_Metric.**

35 The next operation will machine the 4 smaller pockets.

36 Select **Ctrl + R** or select **Steps** and **Features** to create a **New Feature.**

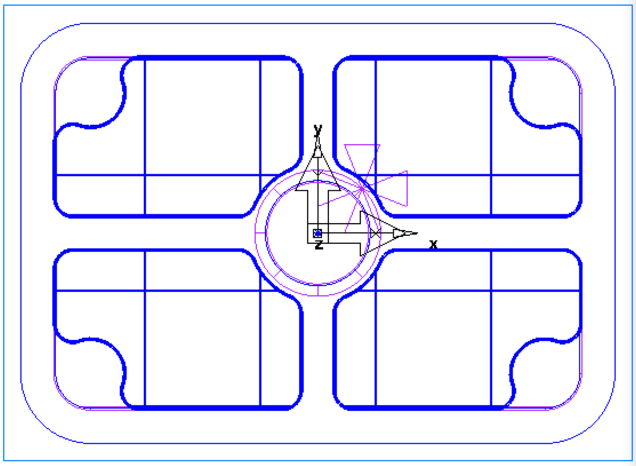
37 Select **Pocket** from Curve

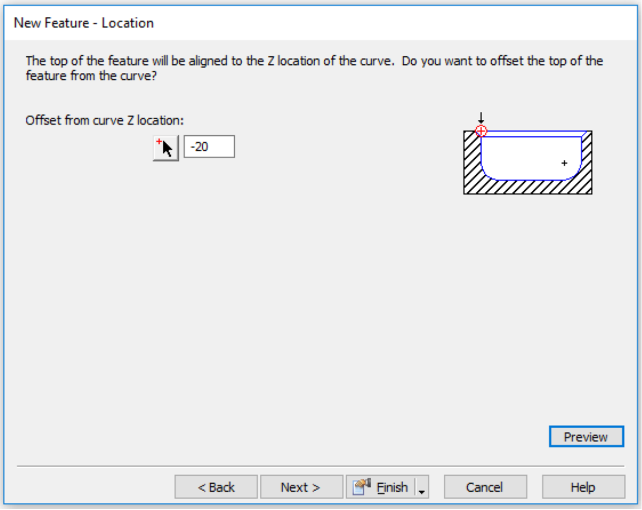


38 Select **Next.**

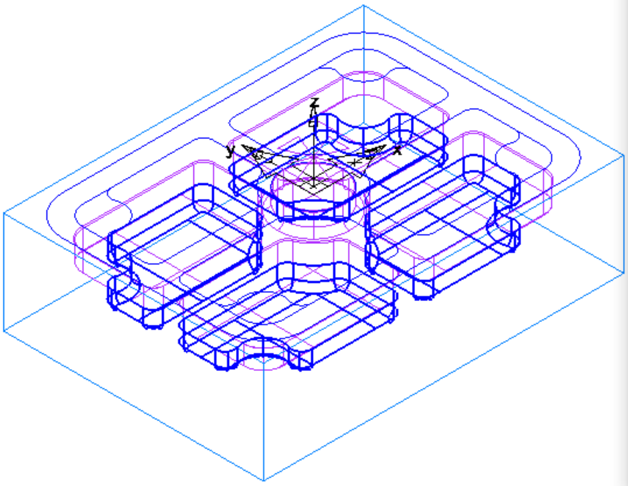
We need to select the four smaller pockets. The next screen shot shows all four pockets selected.

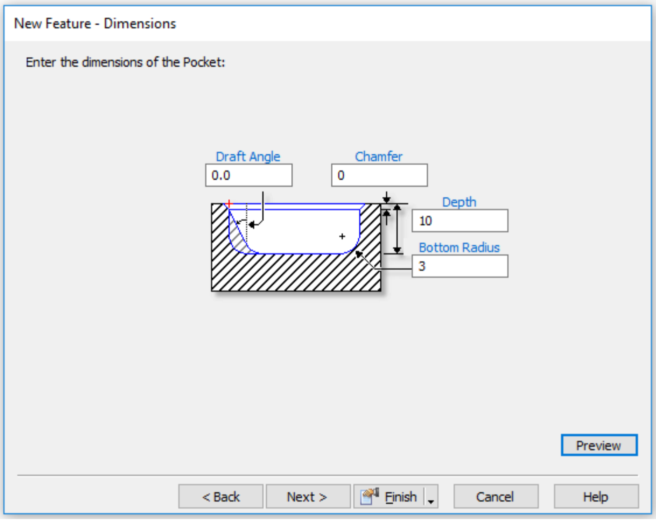
39 Select **Next.** This will show you a menu where you can change the **Offset from Curve Z Location to -20mm.**





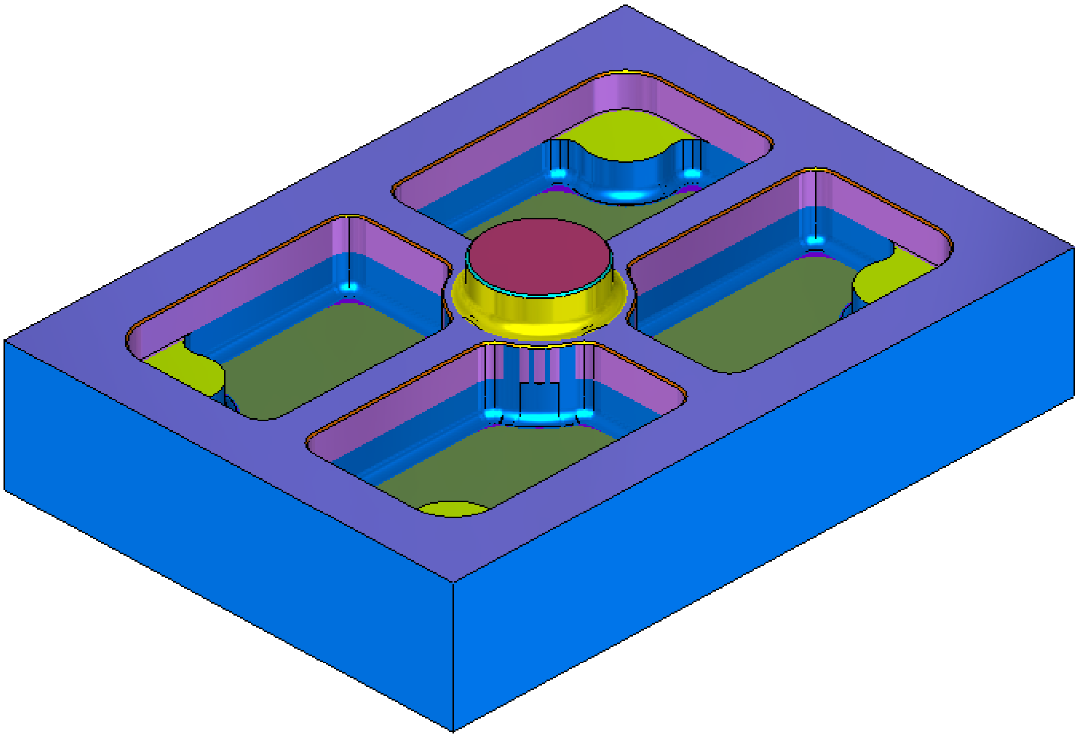
Pocket information Depth 10mm Bottom Radius 3mm





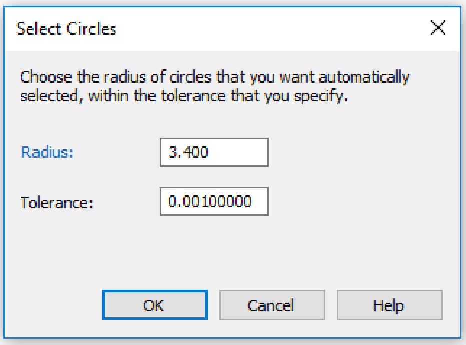
40 Select **Finish.** Run **3D Simulation**. See **3D Simulation** of the part below.

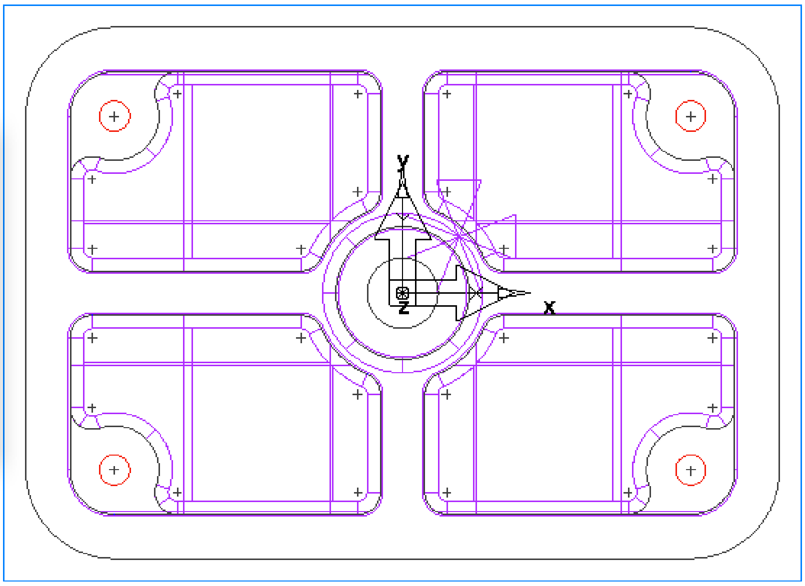
41 Select **Eject**.



*We now have to drill the four holes. One in each corner plus the 16mm diameter hole at **X0,Y0**.*

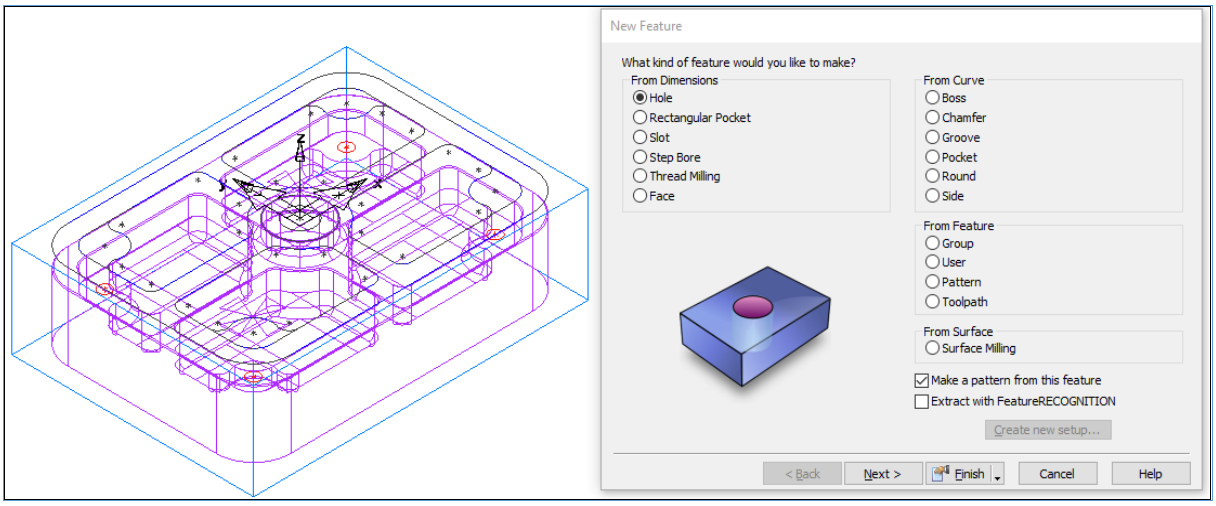
42 Select the **Edit** menu and then select **Circles**. Then Select the Blue Hyperlink Radius. Once you have selected this please select one **3.4mm** radius circles on the screen. Select **OK**. All four circles of the same diameter will be highlighted.



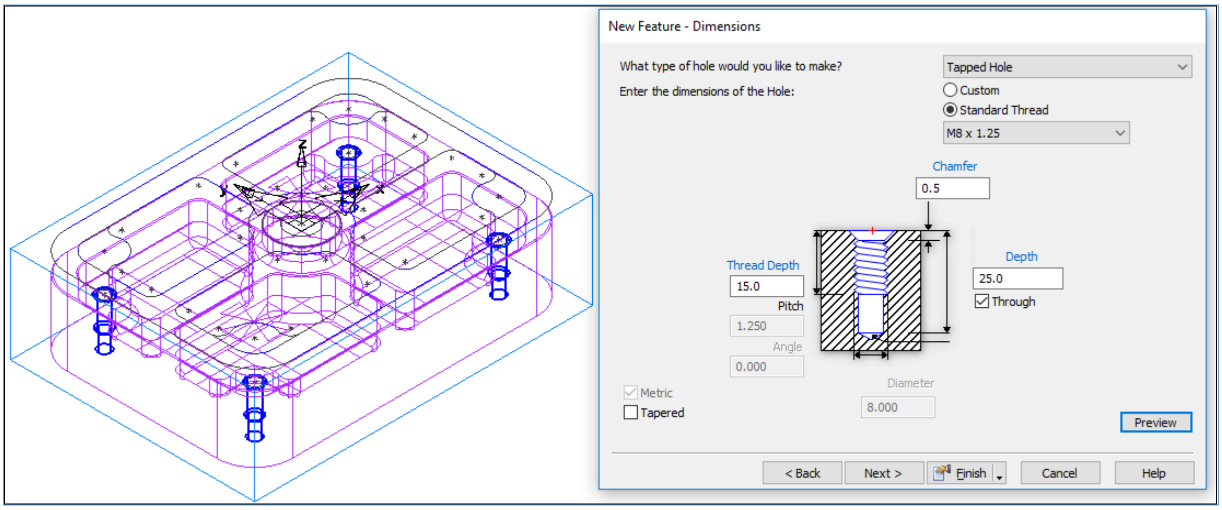


43 Then Select **Ctrl + R** or select **Steps** and **Features** to create a **New Feature**.

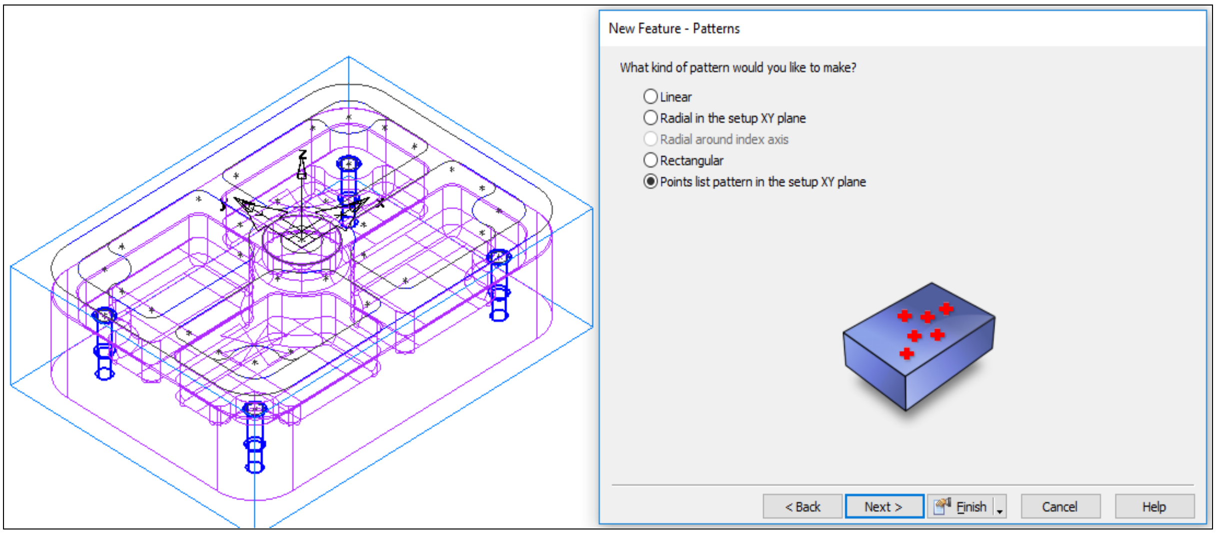
44 Select **Hole.** Also select **Make a Pattern from this Feature.**



45 Select **Next**, Please select **Tapped Hole** and enter the following values.

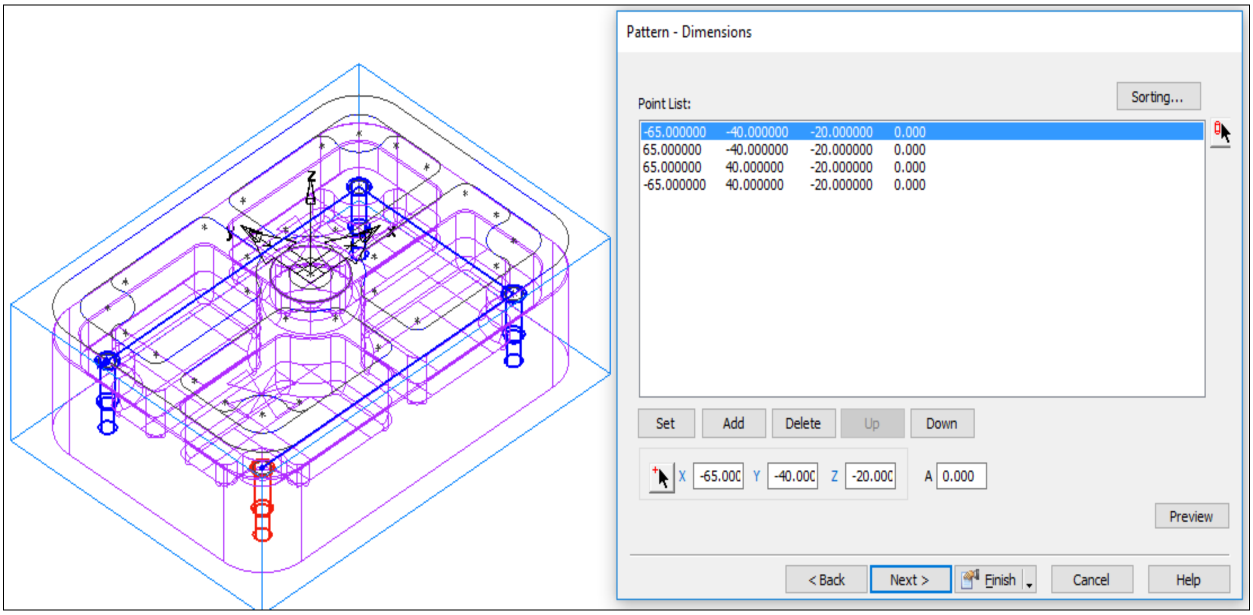


46 Select **Next**.

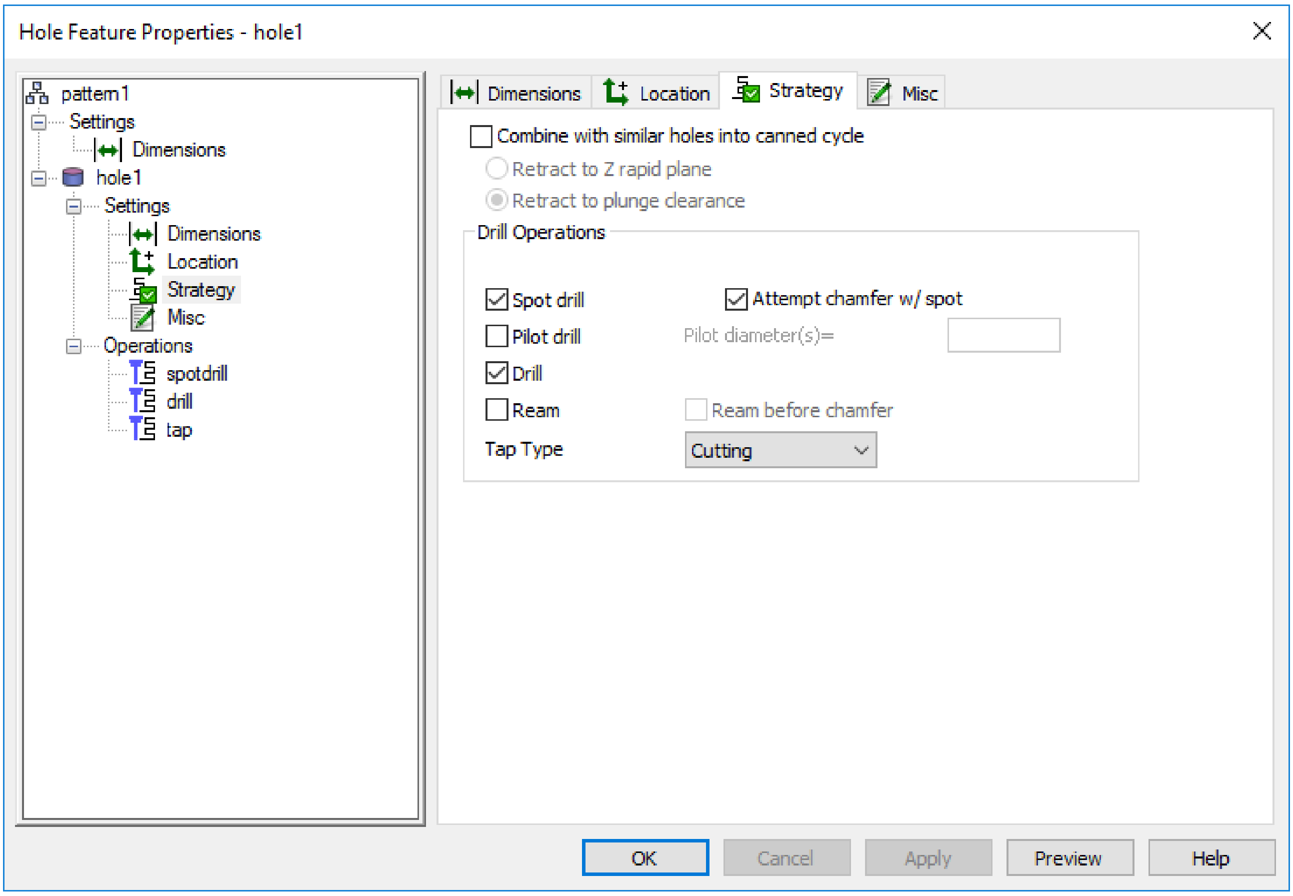


47 Select **Next**.

***FeatureCAM** shows the Dimensions for each Hole.*



48 Select **Finish** then select the **Strategies Tab.**



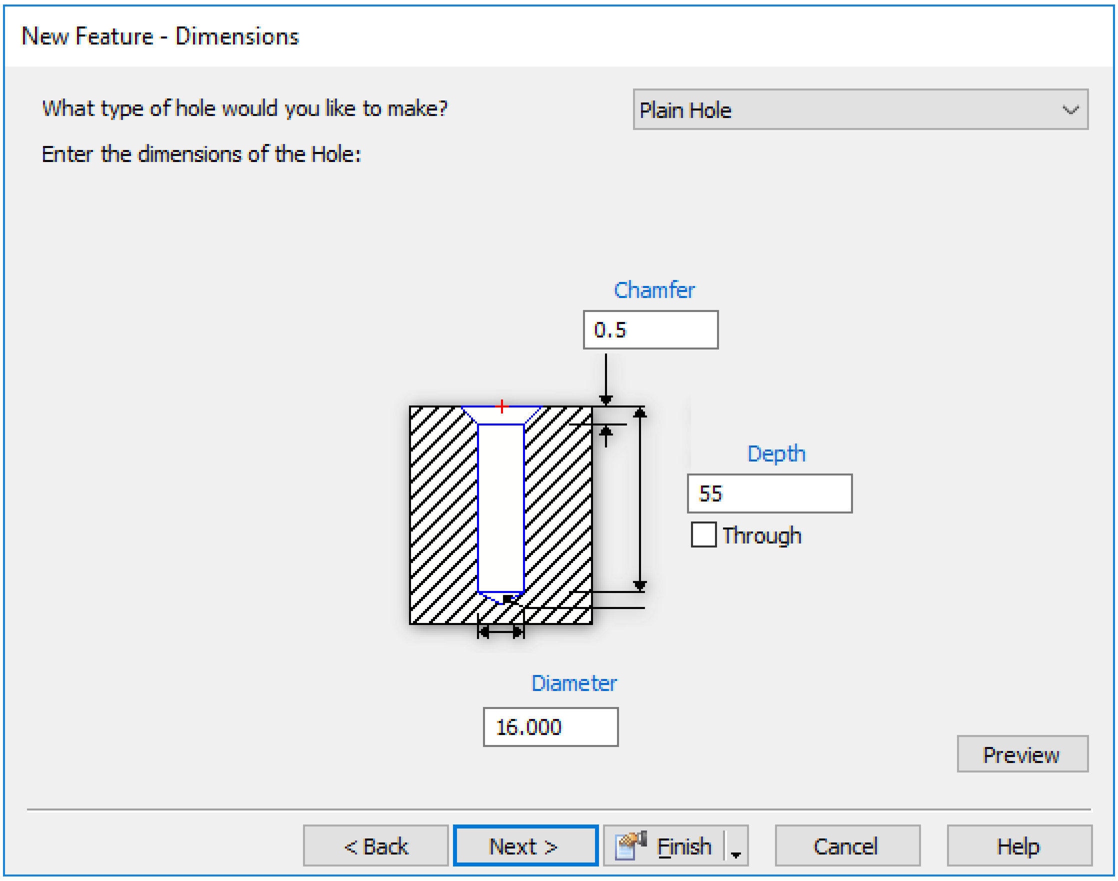
*Change the options to suit your preferred machining style. For example if you do not want to **Spot drill** all holes just **untick** the option in the menu.*

49 Select **Finish** accepting all default tooling. All of the selected holes will now be machined.

The next operation will be to machine the Centre hole ***16mm*** *Diameter plus chamfer.*

50 Select **Ctrl + R** or select **Steps** and **Features** to create a **New Feature**.

51 Select **Hole**. Select the single hole by selecting the geometry with your cursor (Left hand mouse button). Select **Next**.

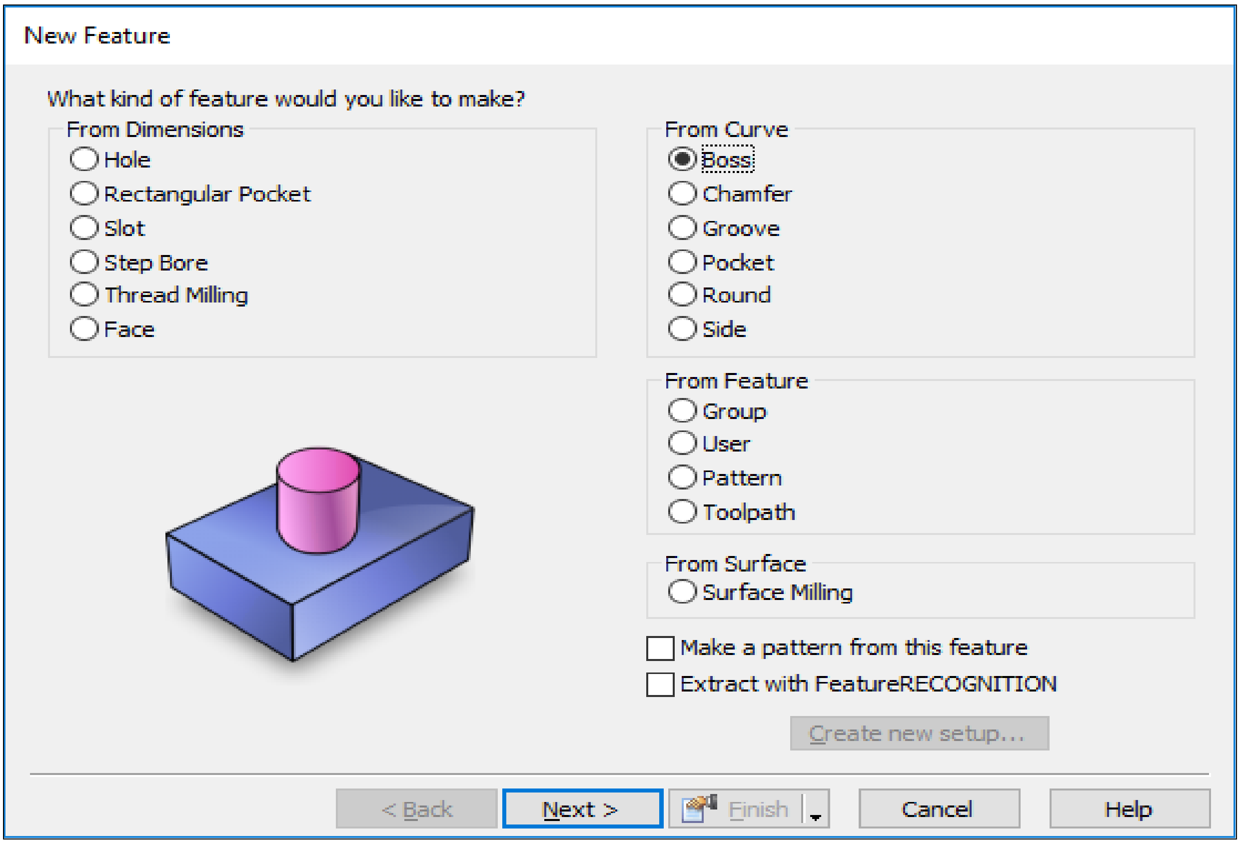


52 Select **Finish** to accept all default tooling from your selected **Crib**.

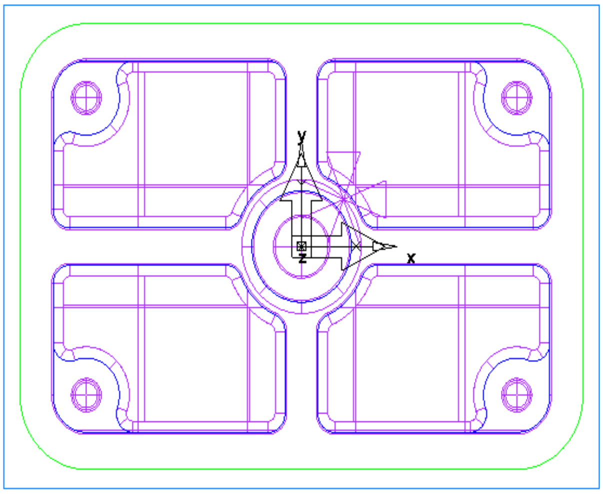
53 All we have to do now is machine the outside shape using **Boss**.

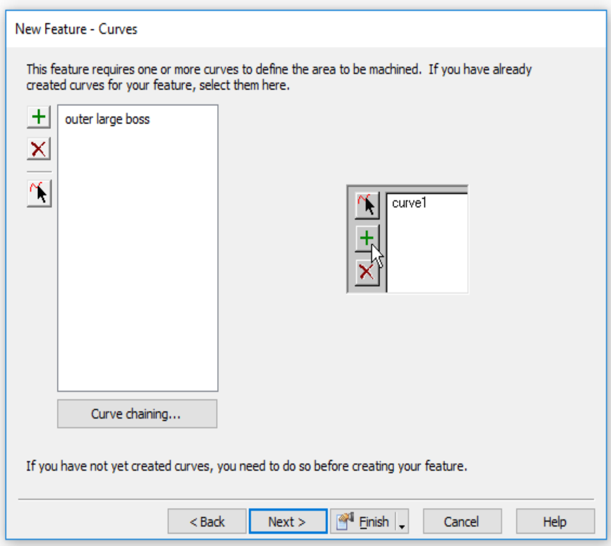
54 Select **Ctrl + R** or select **Steps** and **Features** to create a **New Feature**.

55 Select **Boss.** Select **Next.**

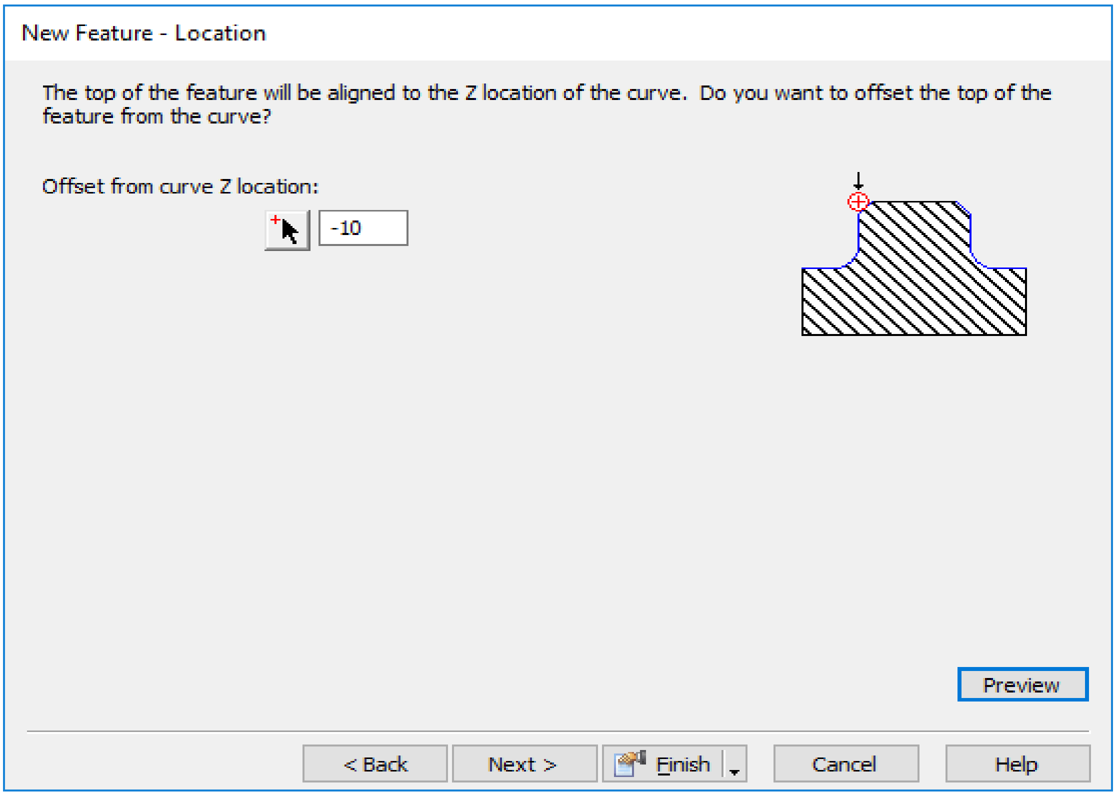


56 Select the outer Curve called **Outer_Large_Boss.**



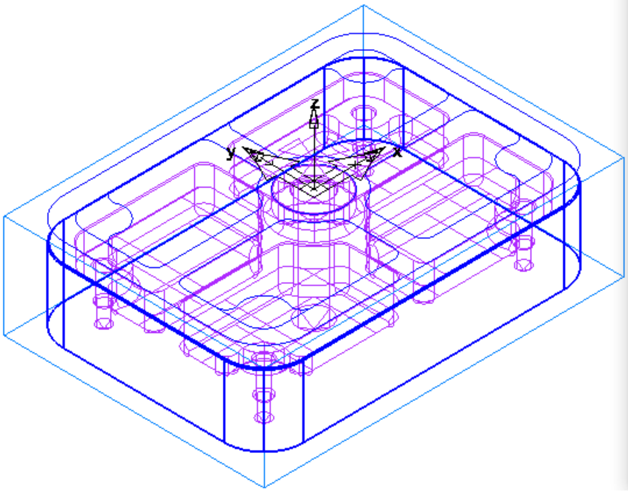


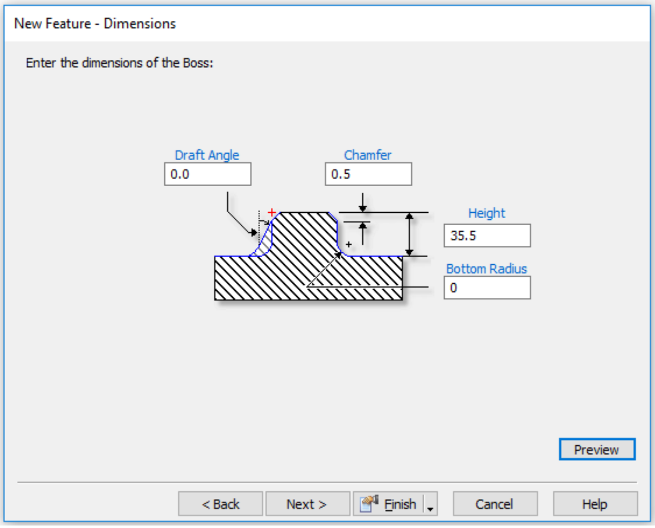
57 Select **Next**.



*The **Offset from Curve Z Location** is **-10mm***

58 Select **Next**.

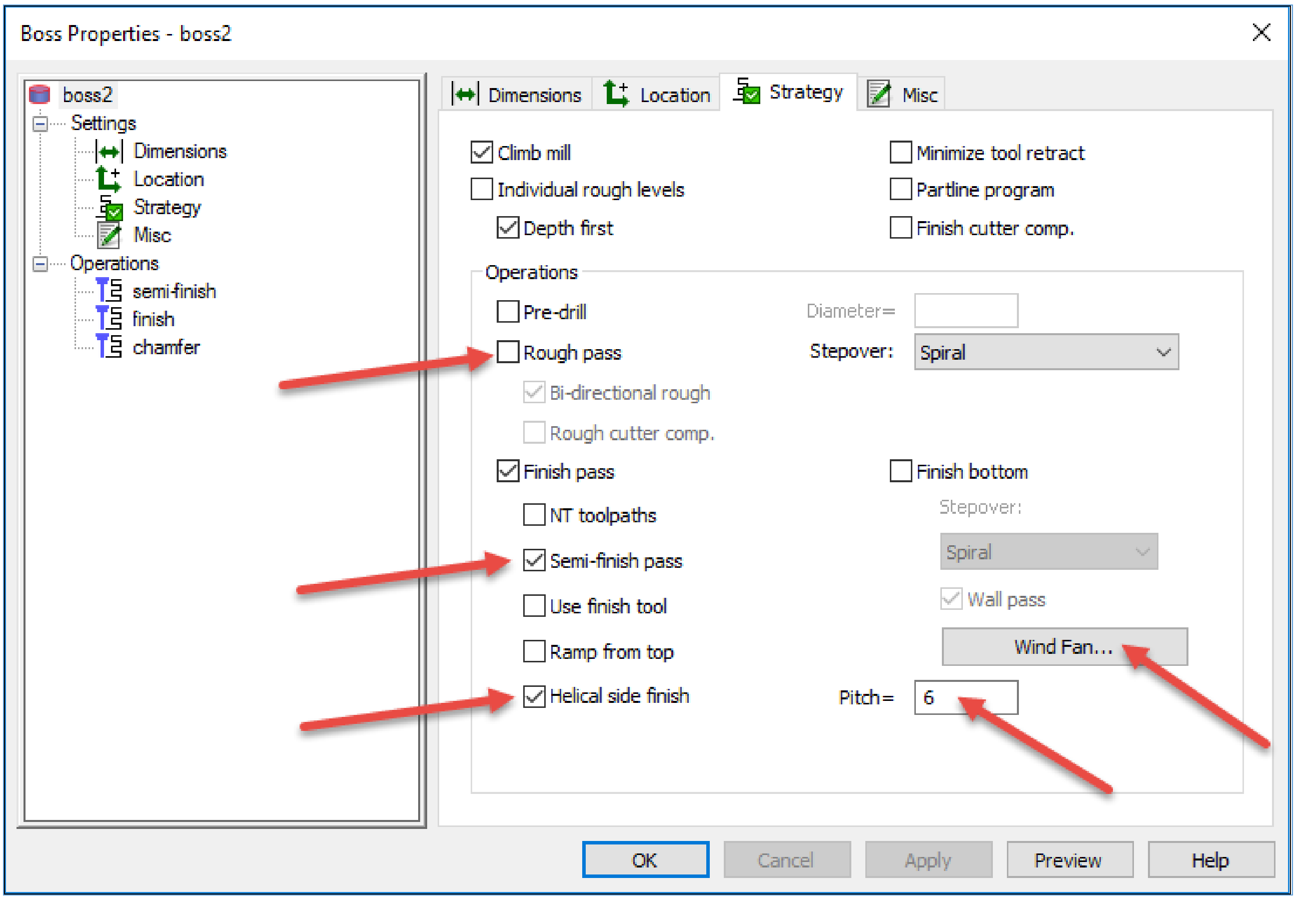




59 Enter the **Height** as **35.5mm**

60 Select **Next**.

61 Select **Finish**. Then select the **strategy** tab. Please select the following options.

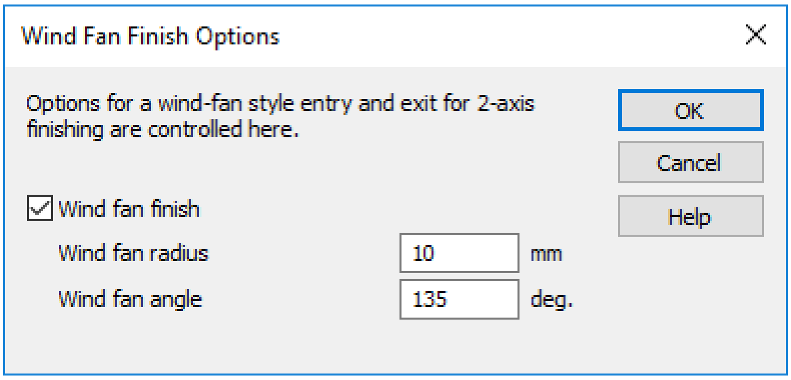


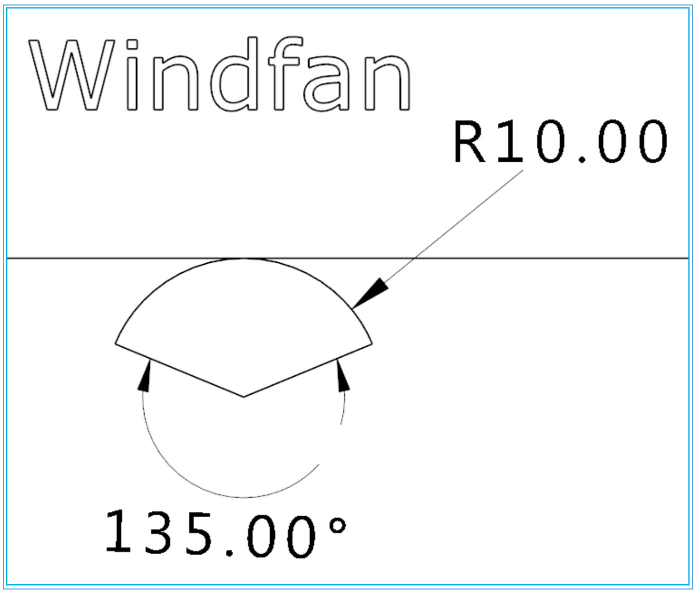
62 **Untick Rough pass and Ramp from top**. Then select **Semi-finish pass** and **helical side finish**. With a **helical pitch** of **6mm.**

63 Select **Wind Fan**.

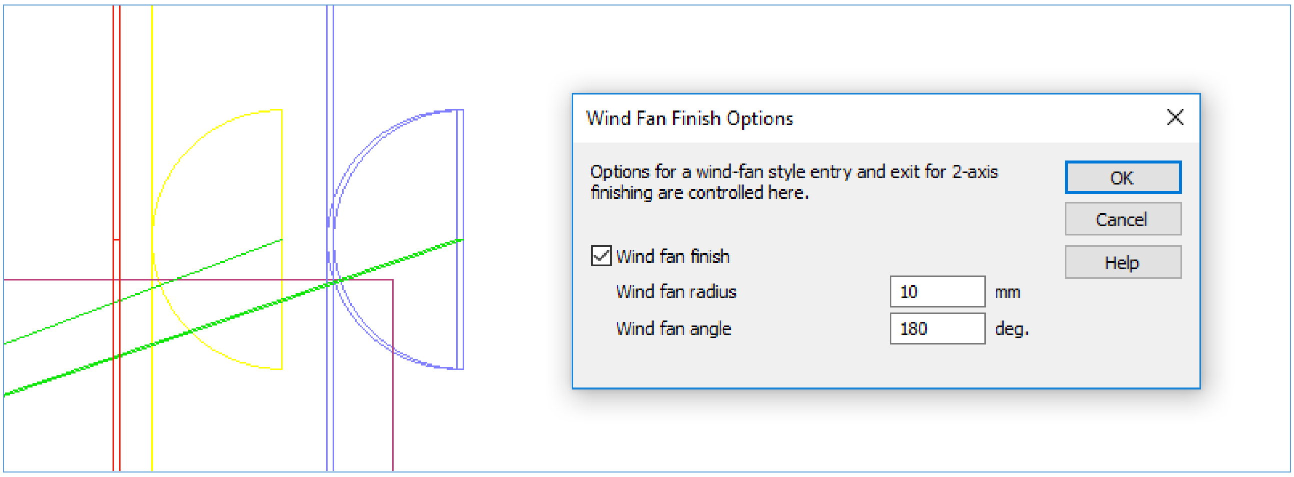
*This is the **Leadin/out** movement for this operation.*

***Select** the **WindFan** option in the menu, which shows a sub-menu as shown below. This produces the following toolpath*

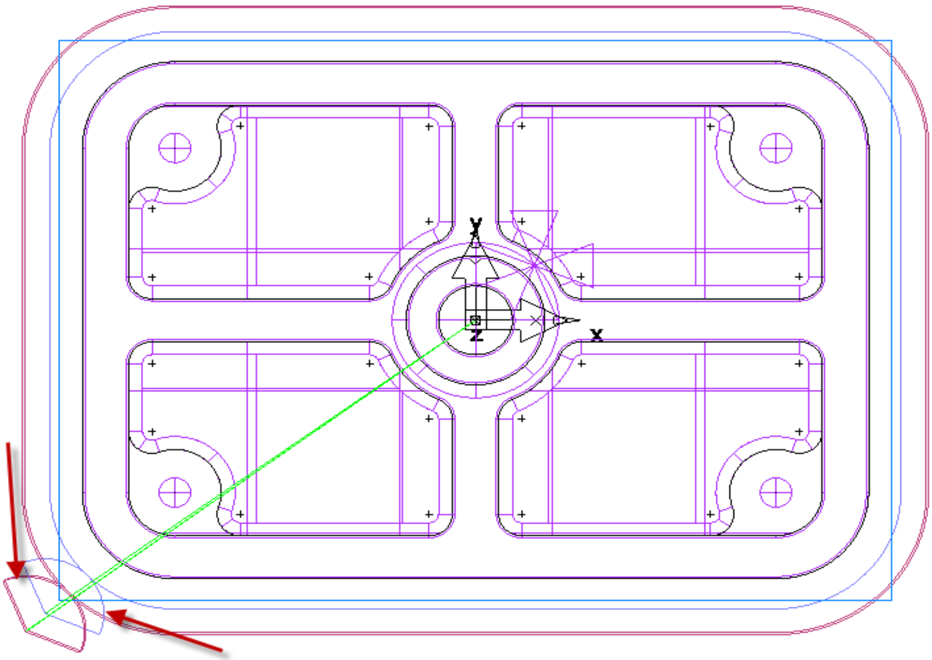




Windfan at 135 degrees and 10mm Radius



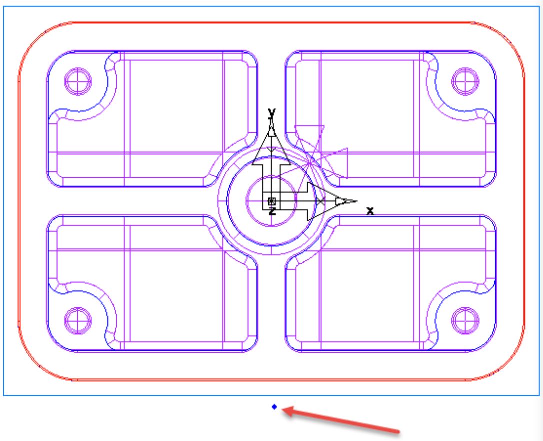
Windfan at 180 degrees and 10mm Radius

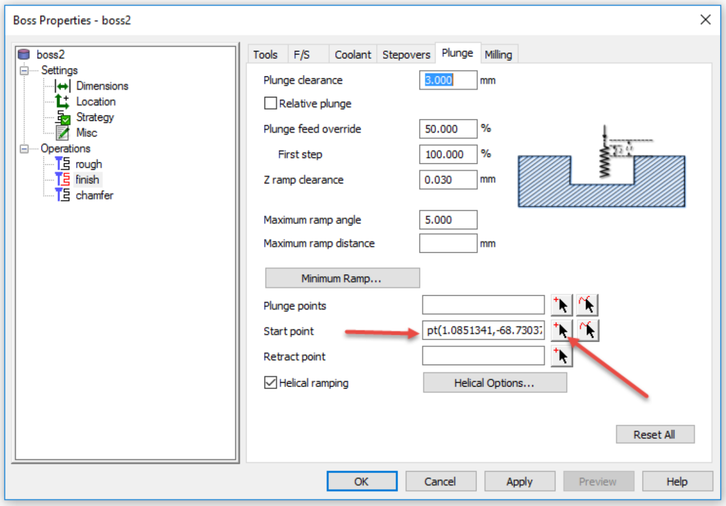


The position of the Leadin/out is not in the best position. The next menu gives you the option to change the start and end points.

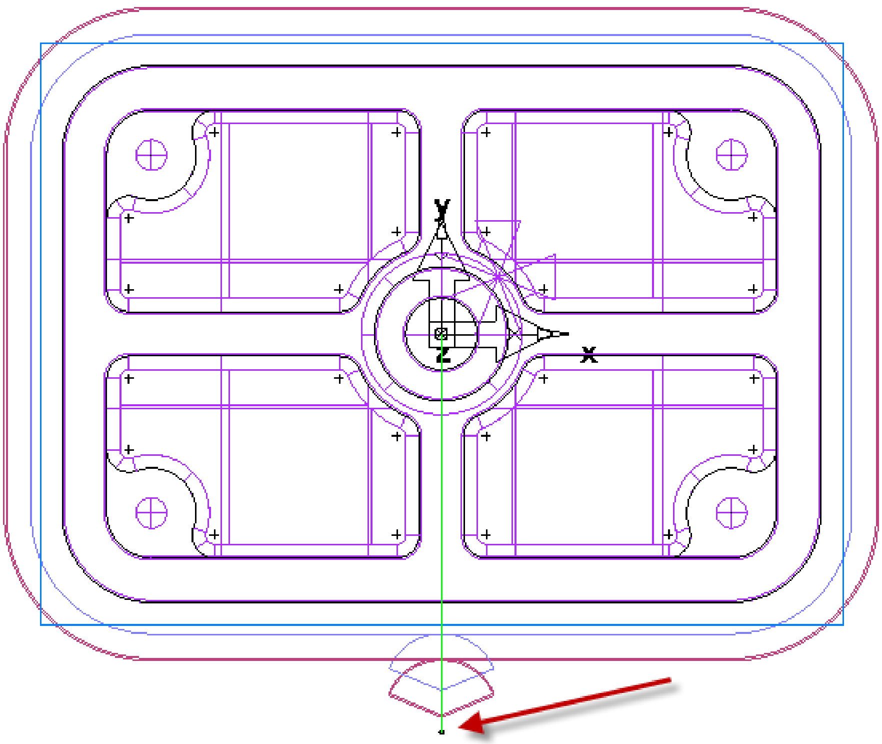
*We have created a **Point** as shown on the next page. This enables you to snap to this point to give a **Start point** and **retract point**.*

64 Select the **Boss2** feature in **PartView**. Then select **semi-finish** and then select **plunge**.

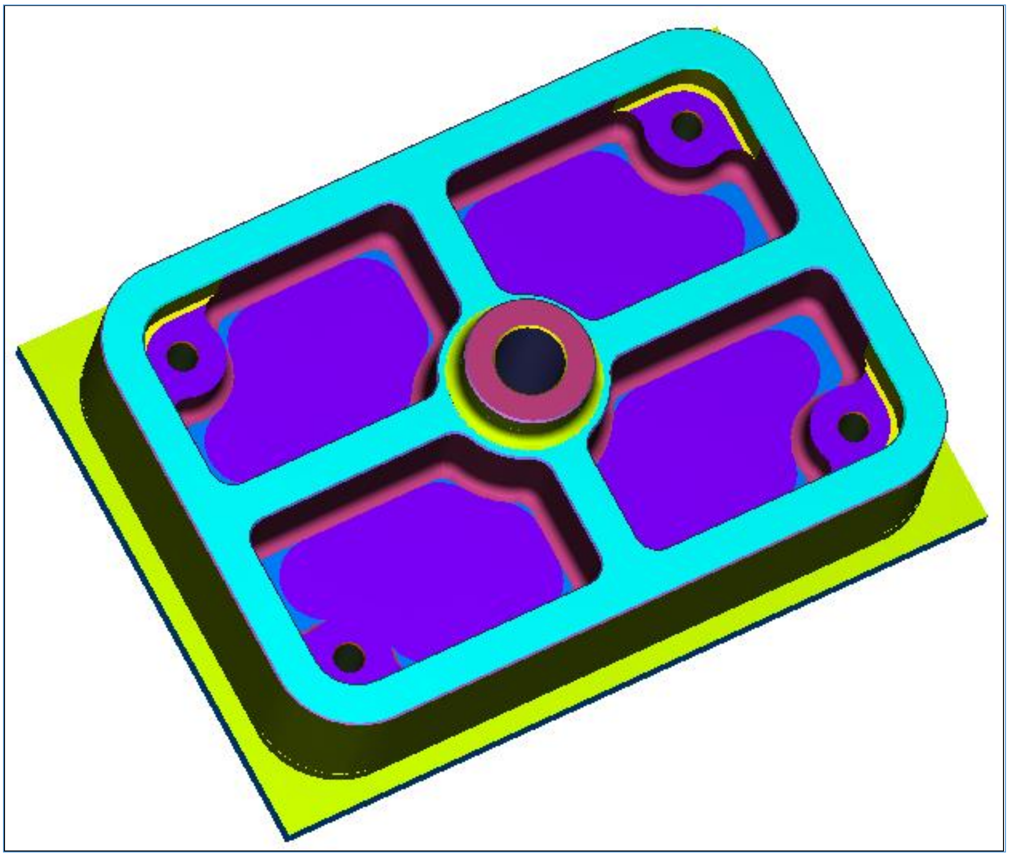




*This will update all **Start points** and **Retract points**. See following image.*



*The part is now complete for **side 1** See **3D Simulation***



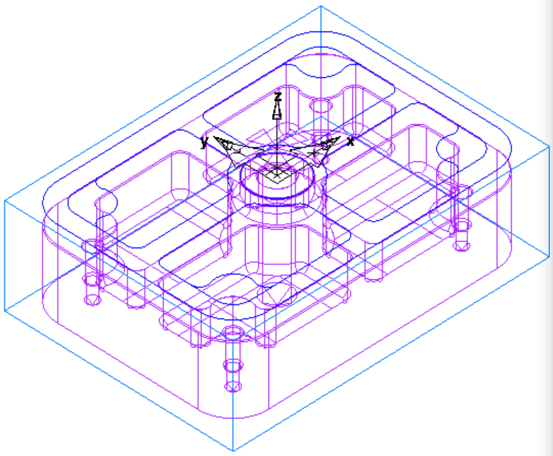
65 **Save the file** as **Exercise MC1 Method one** in your instructors preferred location.

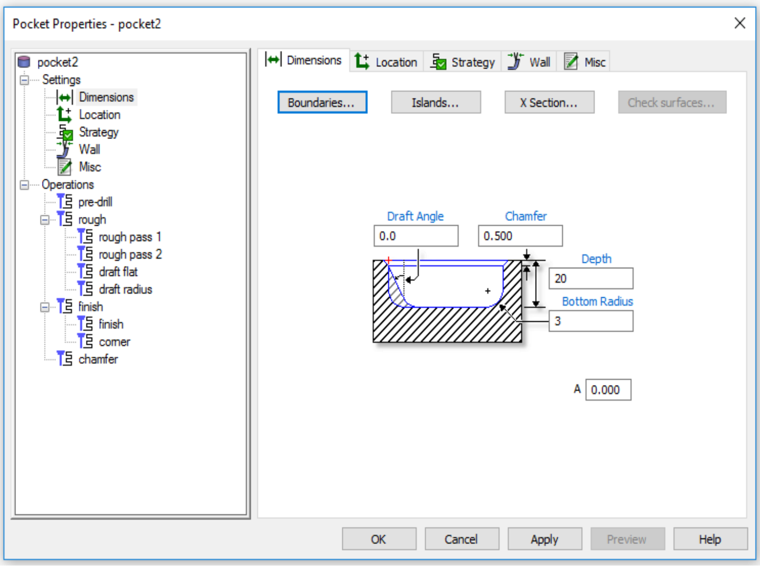
66 **Save the file again and call it MC2** Method two in the same location.

67 **Delete** the first pocketing Feature.

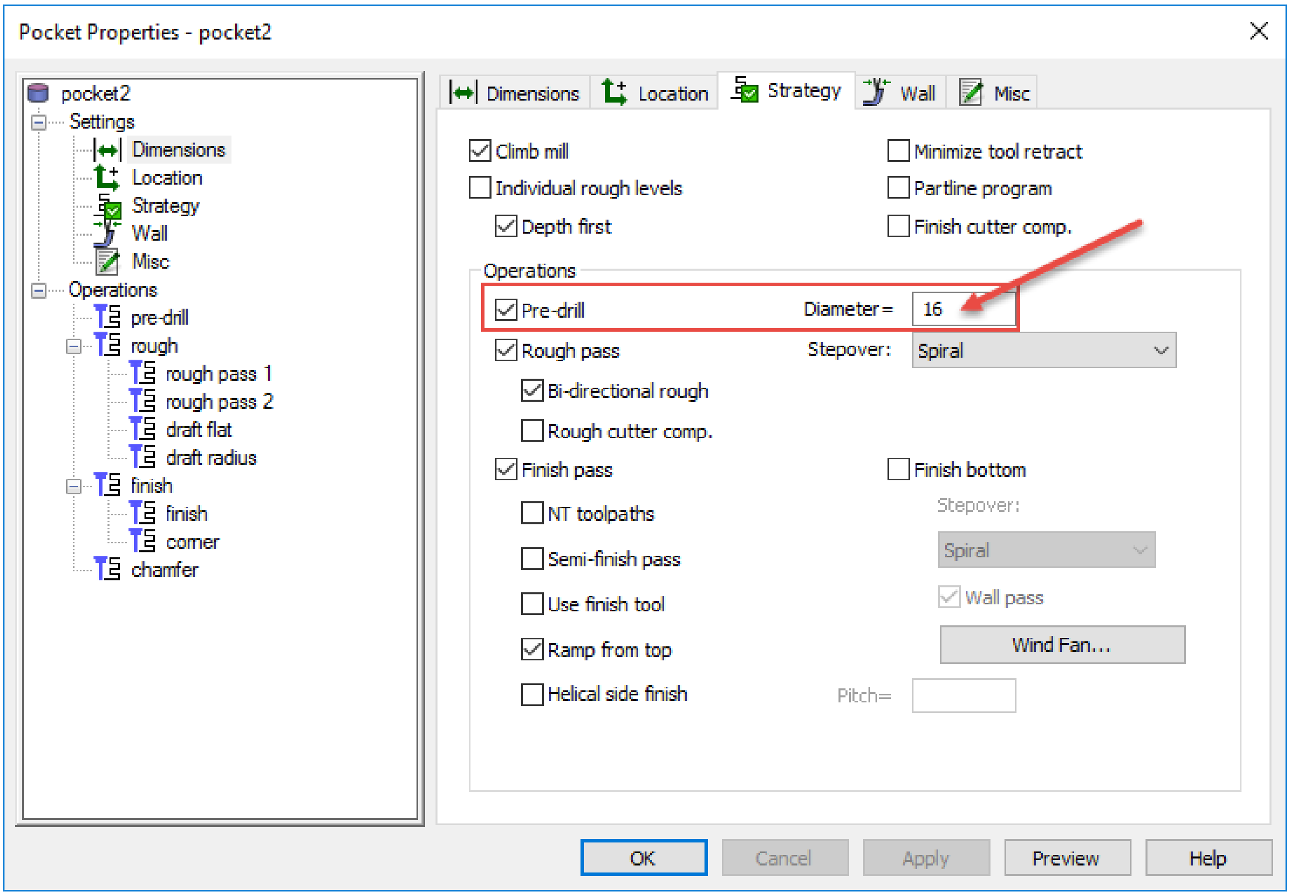
68 Double click on **Pocket2** and **Set Offset from curve Z Location** to -**10mm** then select **Next**.

69 Set the following values in the **Dimensions** tab.





70 Select **Next**. Then from the **Strategie**s page, select **Pre-drill** and enter **16mm**



71 Select **Finish**.

*Before we run a **3D Simulation**. We need to define 4 small curves for the corner details.*

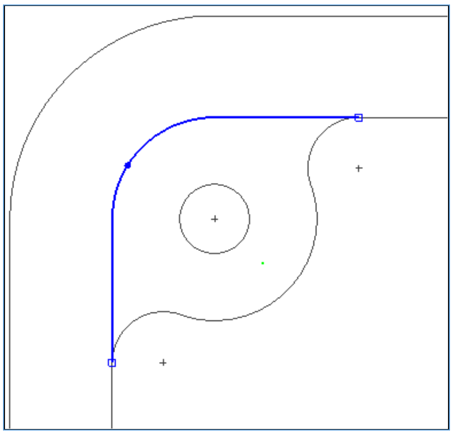
72 Select **Ctrl+5 Top View.**

73 Select **Hide All**.

74 Then select **Show all Geometry**.

75 Create an **Open Curve** by using **Pick pieces**.

76 Select the three elements shown and then apply this method to all four corners.



77 Select **Enter** for each created **Curve**.

78 Select **Esc** to **Finish**.

79 Select **Hide all Geometry**. All that will be visible will be the four Curves that we have just created.

80 Select the Four Curves on the screen. Use the **Ctrl** Key for group select.

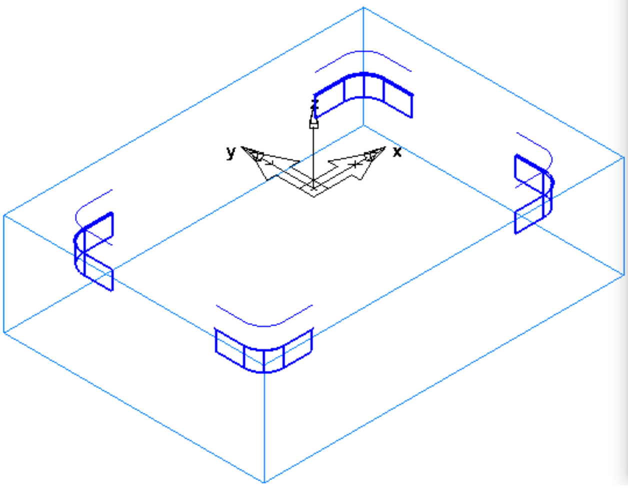
81 Select **Ctrl+R New Feature**. Select **Side**.

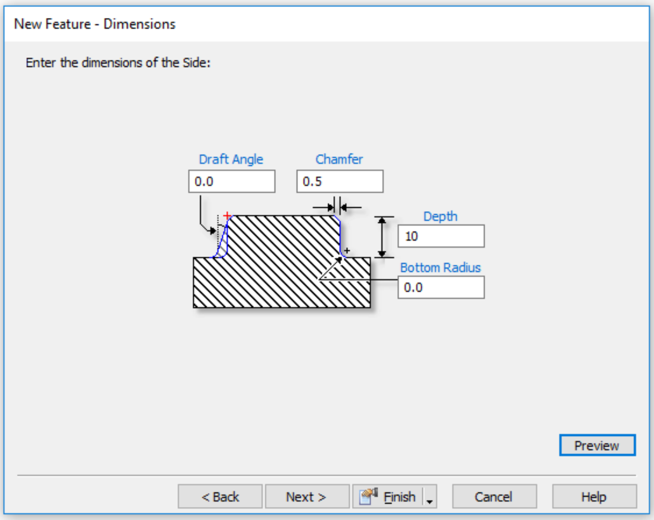
82 Select **Next** Twice.

83 Select the correct machining side for each Curve.

84 Select **Next** and then set the **Offset from Curve Z Location** to **-10mm**

85 Select **Next** and enter the following values.





86 Select **Finish**.

*You will have to change the **Minimum Rapid Distance** from **400%** to **50%** this will allow the cutter to Rapid to the safe level height between moves.*

***Min. Rapid distance%** - Enter the minimum distance, as a percentage of the tool diameter that the tool can use a rapid move for. Moves smaller than this distance use a feed move.*

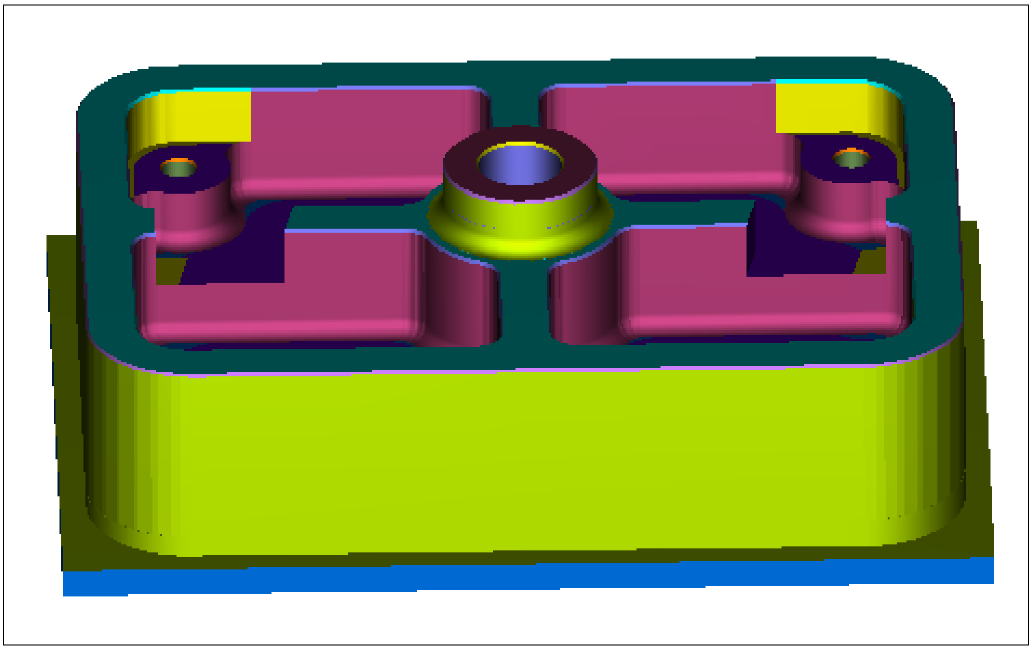
*You can set the default value of this attribute for the current document in the **Machining Attributes** dialog. Set this in the **Misc**. tab.*

***NT toolpaths** do not have the **Min. rapid distance %** attribute. They use **Stepover rapid distance** instead.*

Stepover rapid distance** — this option controls when to retract and plunge on Boss Stepovers for NT toolpaths. **Enter an absolute distance.

*Traditional toolpaths (**Spiral** and **Zigzag**) do not have the **Stepover rapid distance** attribute. They use **Min rapid distance %** instead.*

*If you get a **collision** on the part this means that the **Minimum rapid distance** is set to **400%.***



87 Go to Machining Attributes (Local Setting) and set the **Minimum rapid distance to <u>between 50% to 100%.</u>** This will allow the cutter to retract between moves.

It would be a good idea to set this Parameter in your main Configuration. (Global Settings) This will prevent any further problems. Remember to set this value on all computers.

88 Set the **Base Priority** for the following Features.

Base Priority *= Enter the priority that the operation takes in the document. The lower the number, the higher priority the operation takes.*

Face = Base Priority **1**

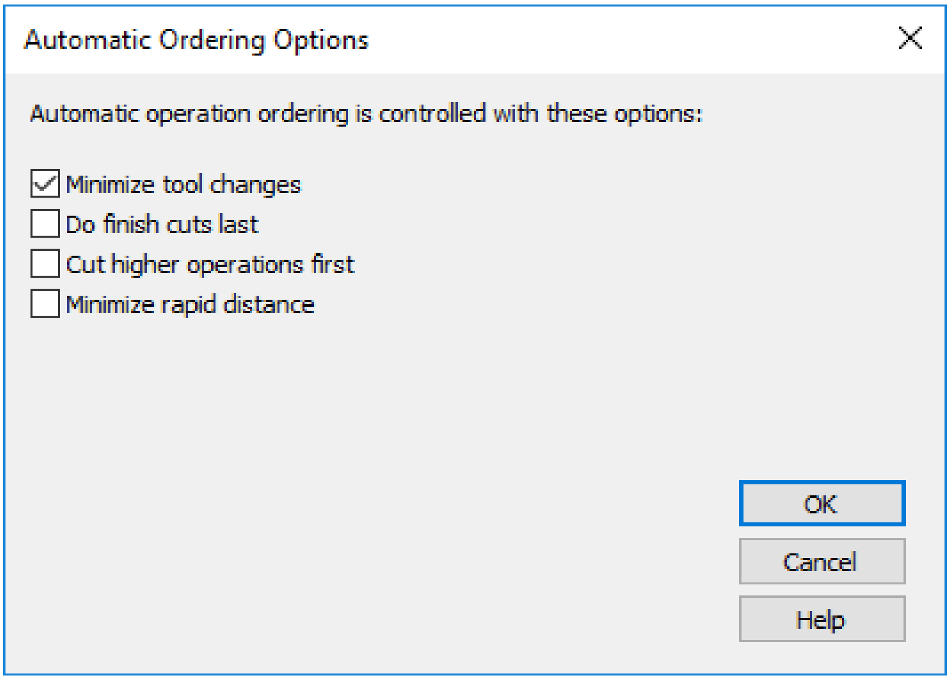
Boss = Base Priority **2**

Pockets = Base Priority **3**

Base priority

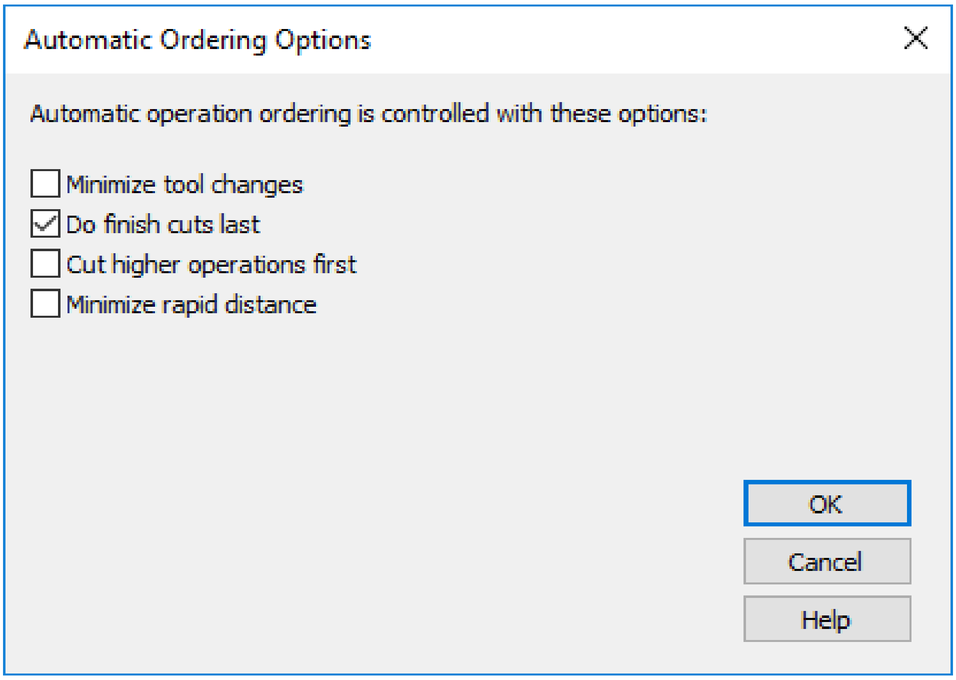
- The default priority for features. Features are sorted by their **priority** to determine the order in which they are manufactured.
- Set the **Priority** attribute on each feature to override the **Base priority** and order the features for manufacturing. For features that have the same priority value, FeatureCAM uses minimization of tool changes and other criteria to determine a manufacturing order.

Minimize Tool Changes



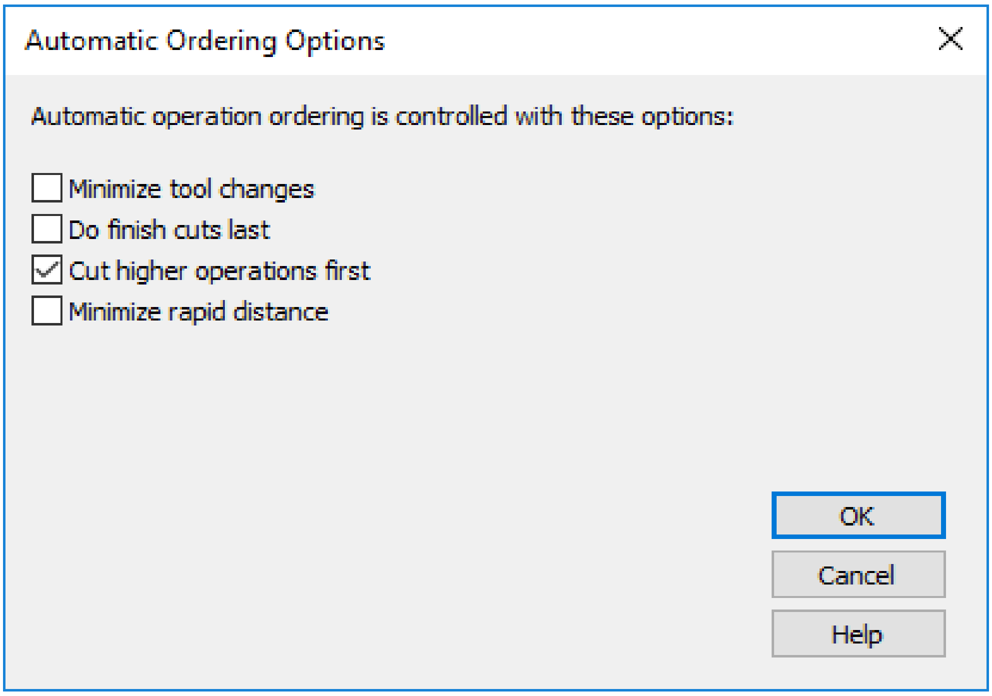
Minimize tool changes *groups operations together that use the same tool. This saves time for you by eliminating or* ***reducing needless tool changes****.*

Do Finish Cuts Last



Do finish cuts last *moves the finish milling operations to the end of the setup without altering the order of the finishing operations? If you want to perform all rough milling operations before finish milling operations, select the* ***Do finish cuts last*** *attribute*

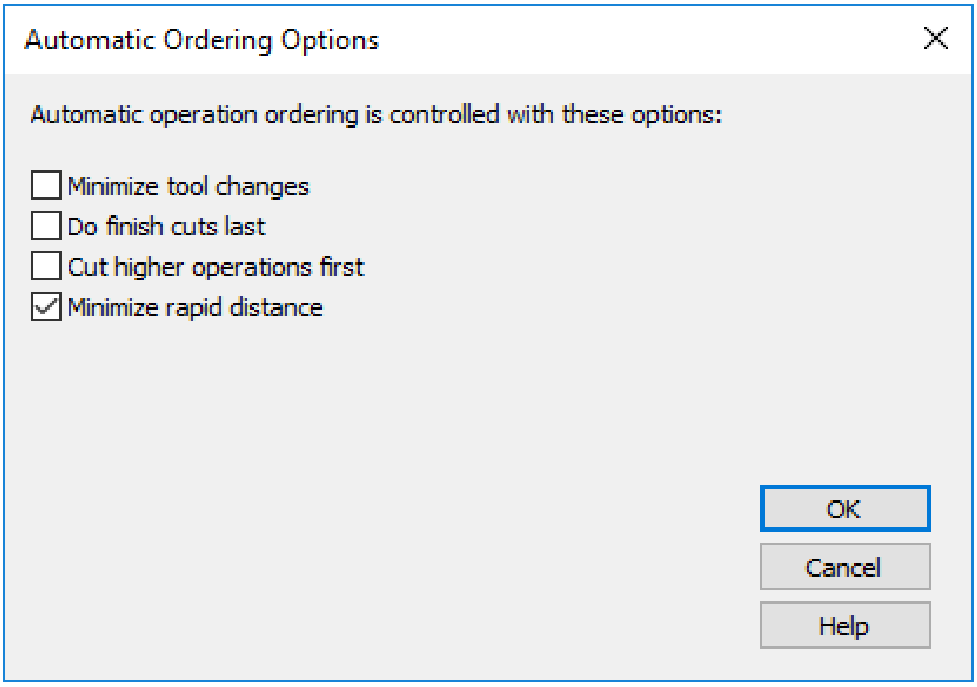
Cut higher operations first



This attribute only affects milling setups. Select this check box to mill the features from the top of the stock first and work toward the bottom.

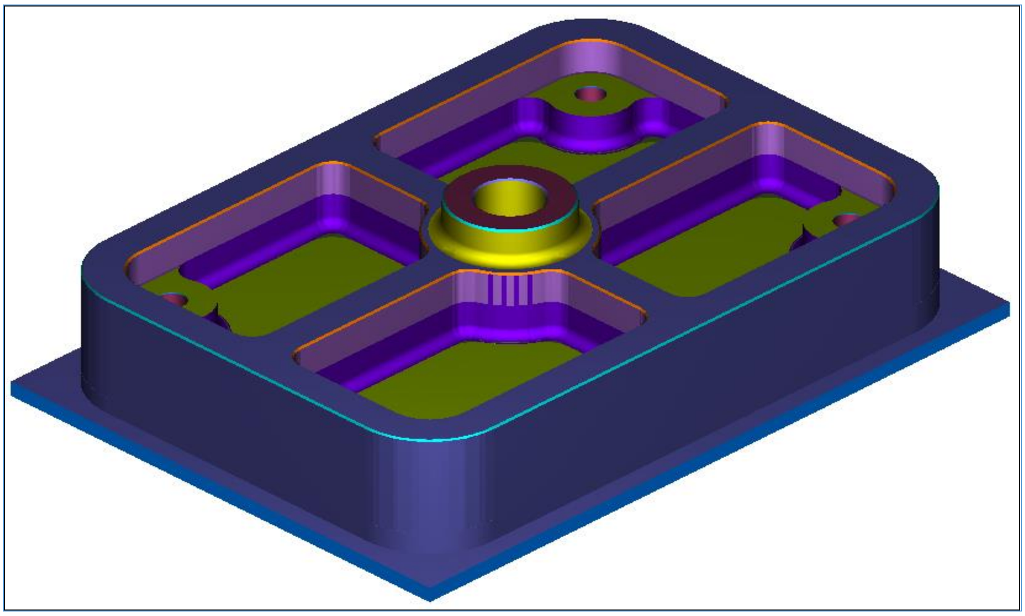
If you deselect this attribute, you should carefully graphically verify the toolpath before cutting your part.

Minimize Rapid Distance



This attribute only affects milling setups and is the only ordering option that will change the order of features specified in the part view. ***Minimize Rapid Distance*** *moves to the next closest feature that uses the same tool as the last operation.*

89 Run a **3D Simulation** of the finished Part.

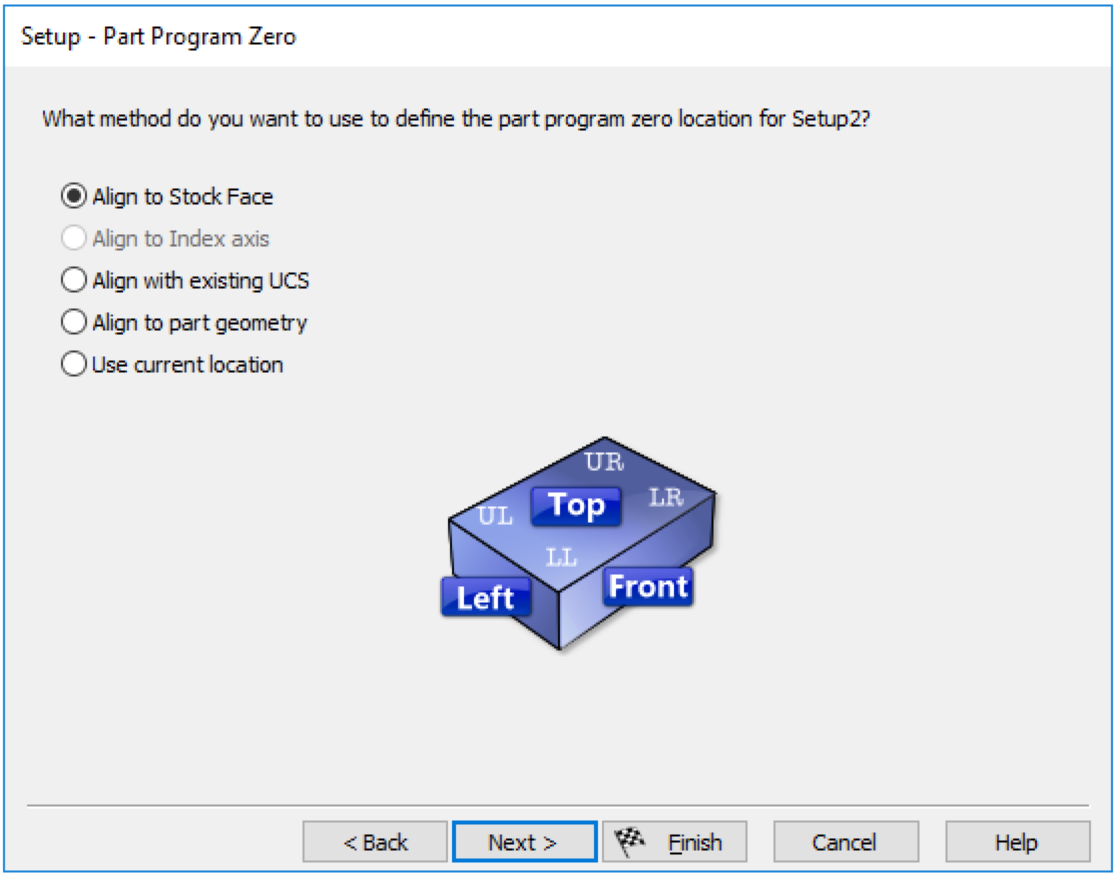


90 **Save** the File as **MC2 Method two**. **Save** the files in your **Coursework** folder.

*We need to **Face** the opposite side of the part.*

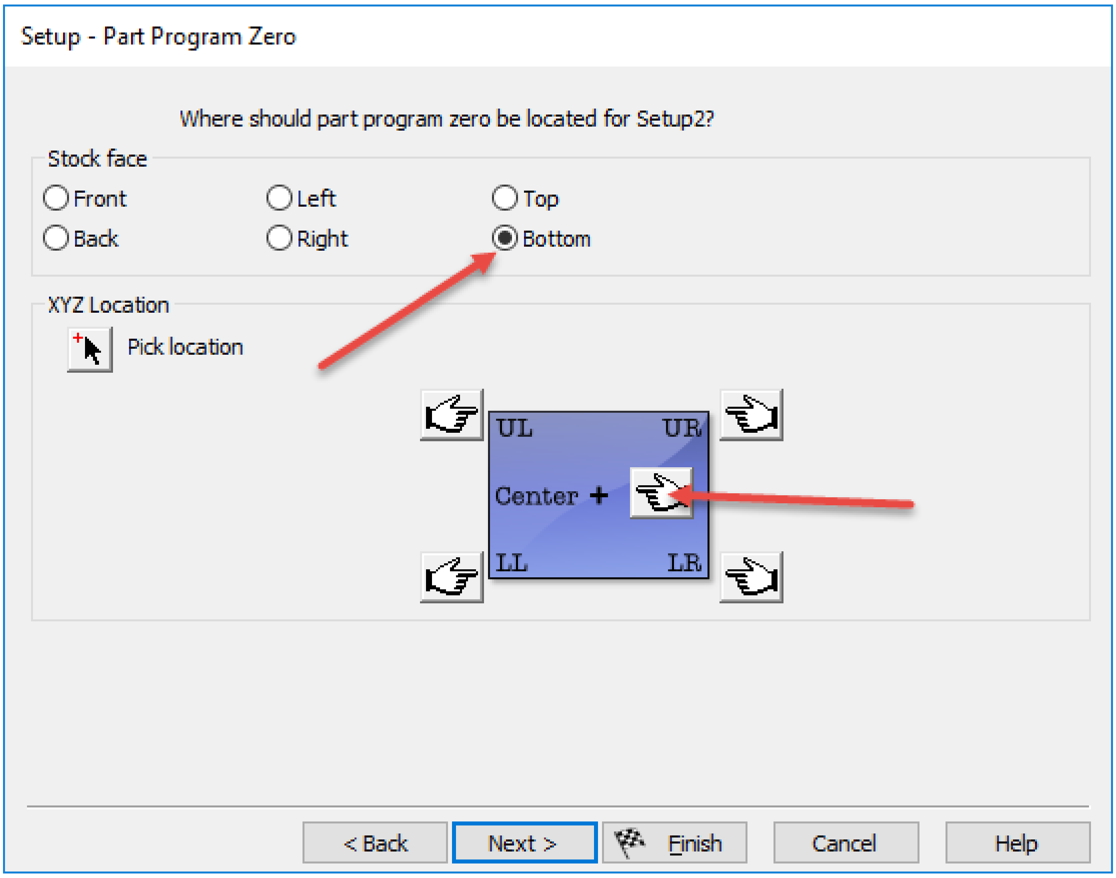
91 To do this we need to create **Setup2**.

92 Double click on **Setup1** in **PartView**. Then select **Align to Stock Face.**

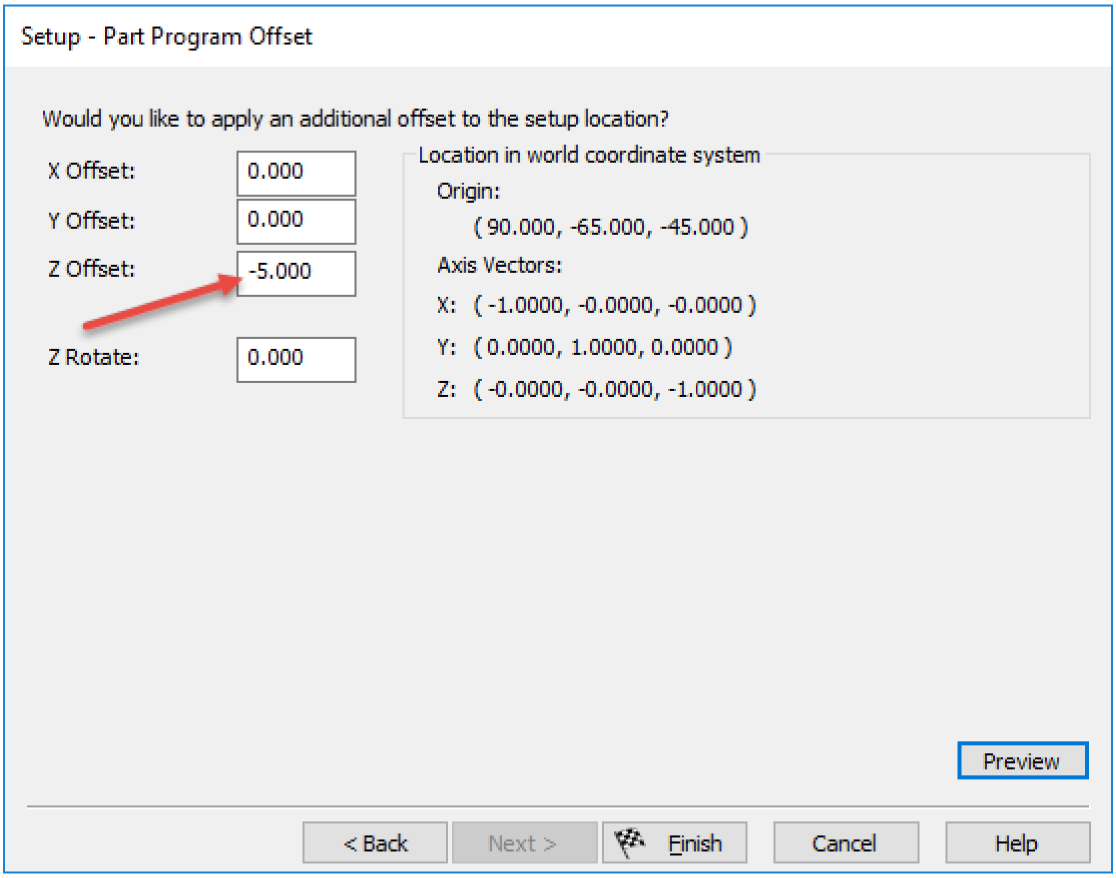


93 Select **Next**.

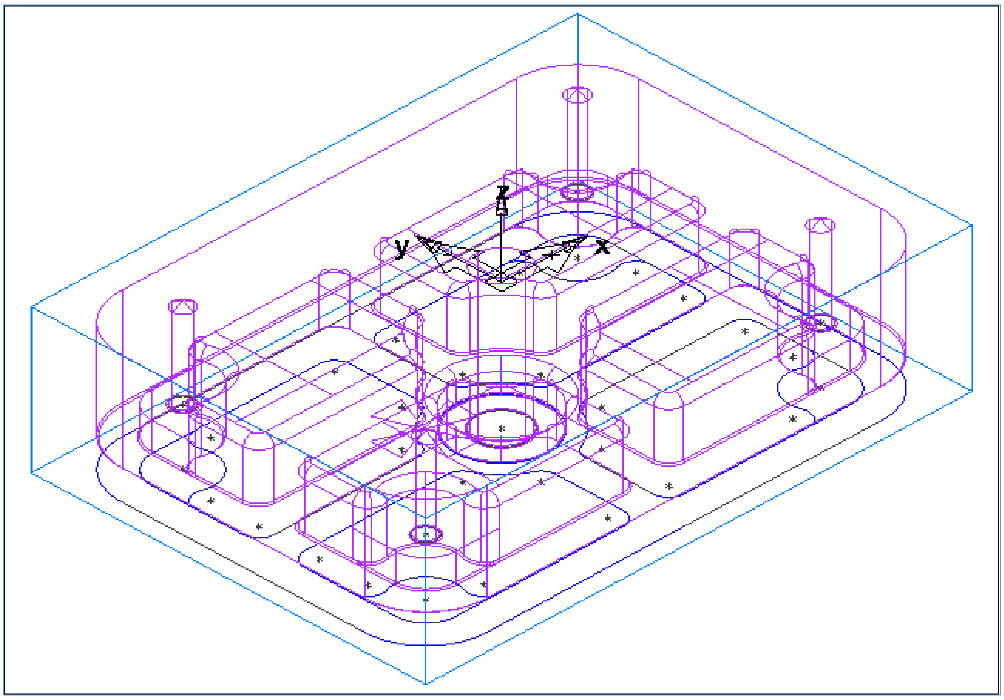
94 Select **Bottom** and **Center**.



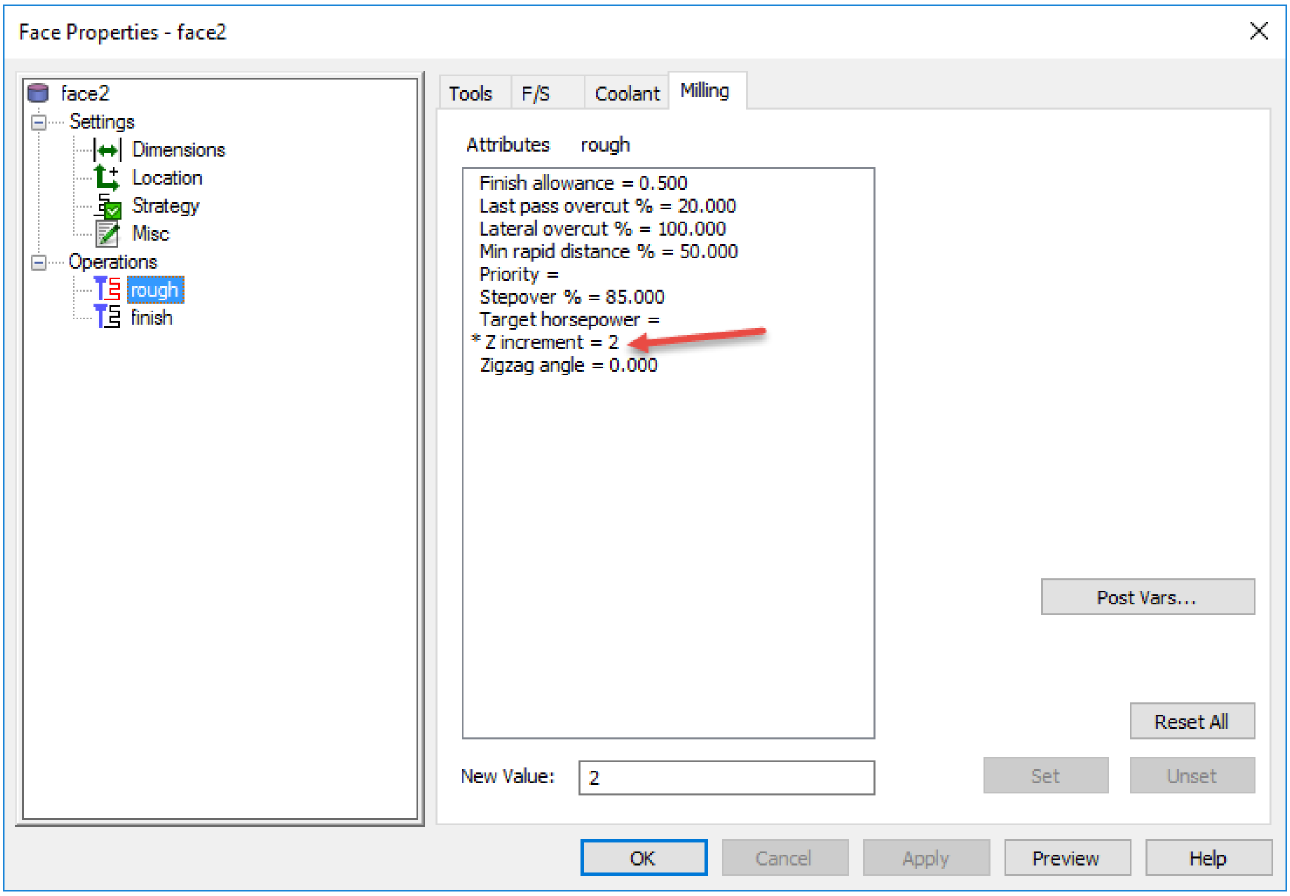
95 Select **Next**. Then set the Z Offset to **-5.00mm** then select **Finish.**



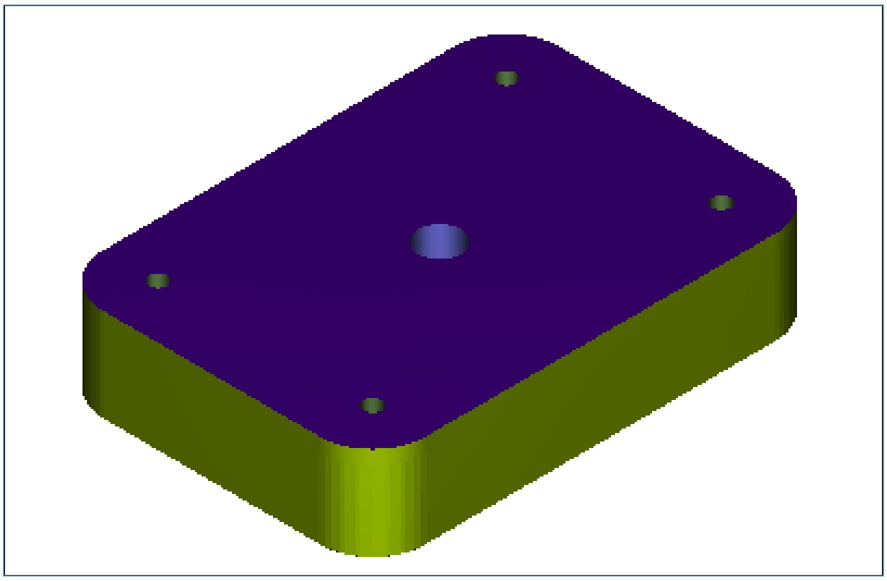
*To have a look at the new **Setup2** change the view to **isometric**.*



96 Create a **Facing** operation on **Setup2**. Remember to select the **Rough option** and set the **Z Increment** in the **Milling tab** to **2mm**.

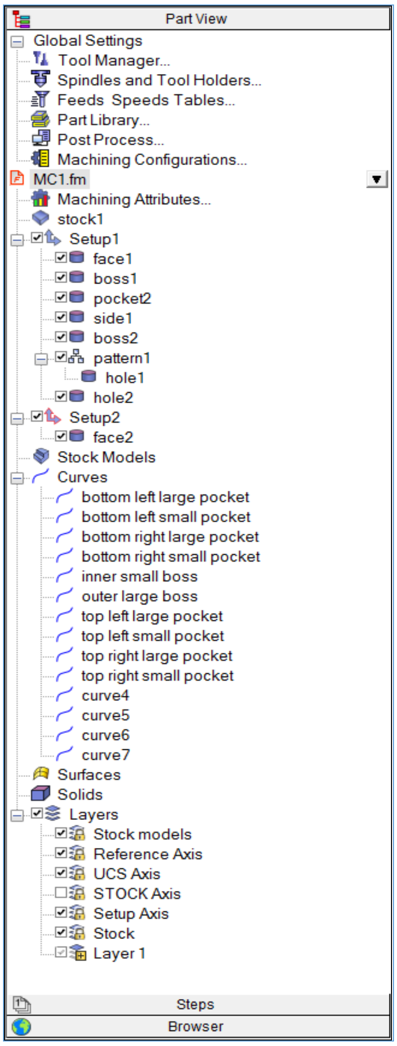


*Run a **3D Simulation** to verify the machining in this setup.*



97 Select **Eject** or **Esc** to close the **3D Simulation**.

This is the completed list in **PartView**.

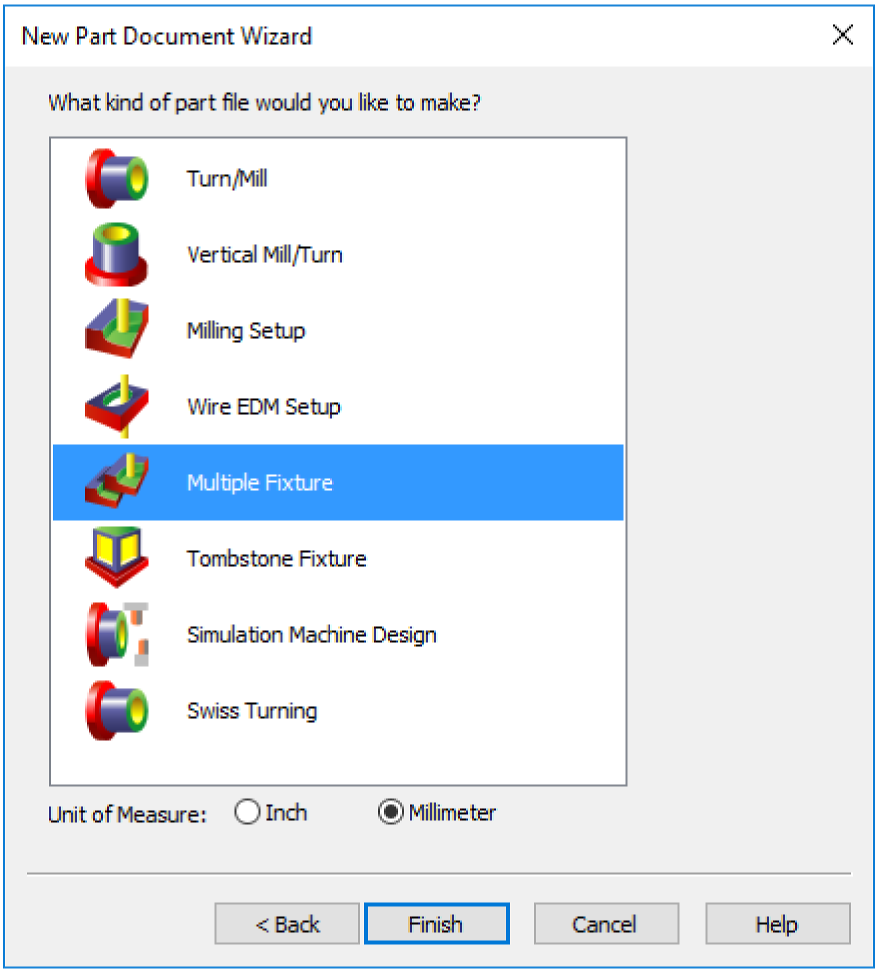


***Save** the File as **MC2 Method two**. **Save** the files in your **Coursework** folder.*

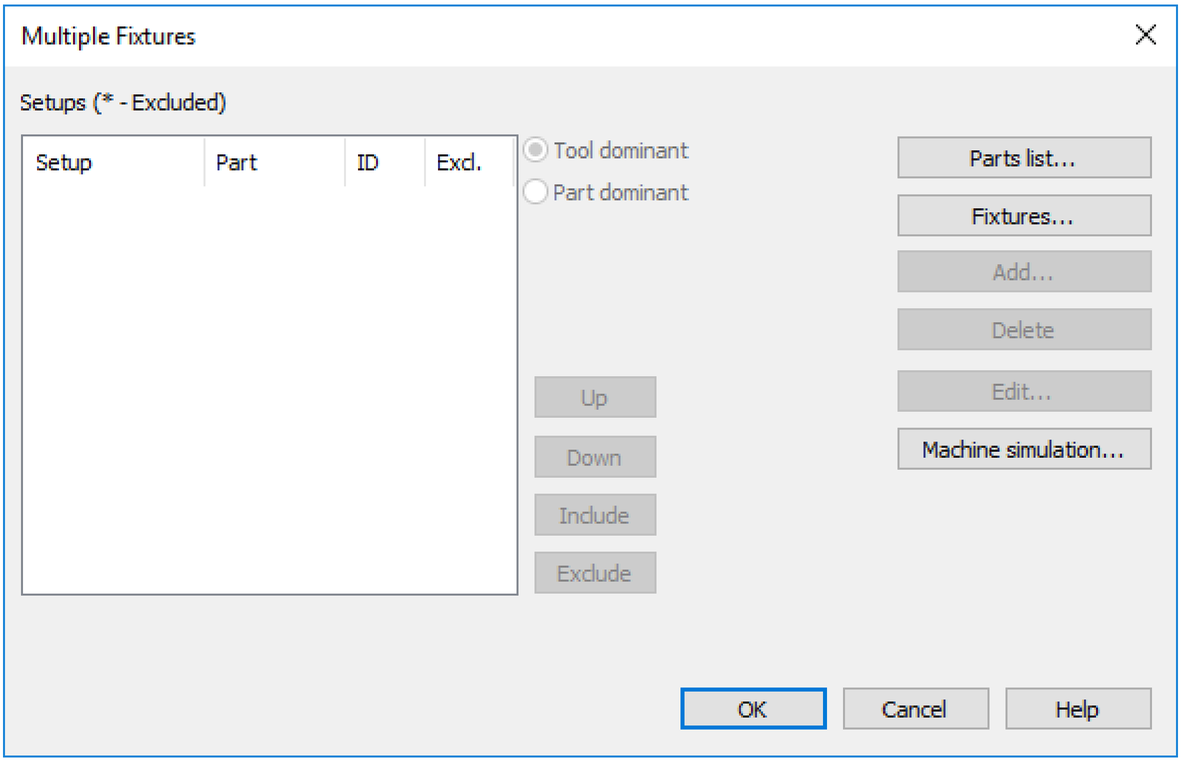
*Close this document and create a **New file.***

Multiple Fixture Document

1 Open a New **Multiple Fixture Document**.

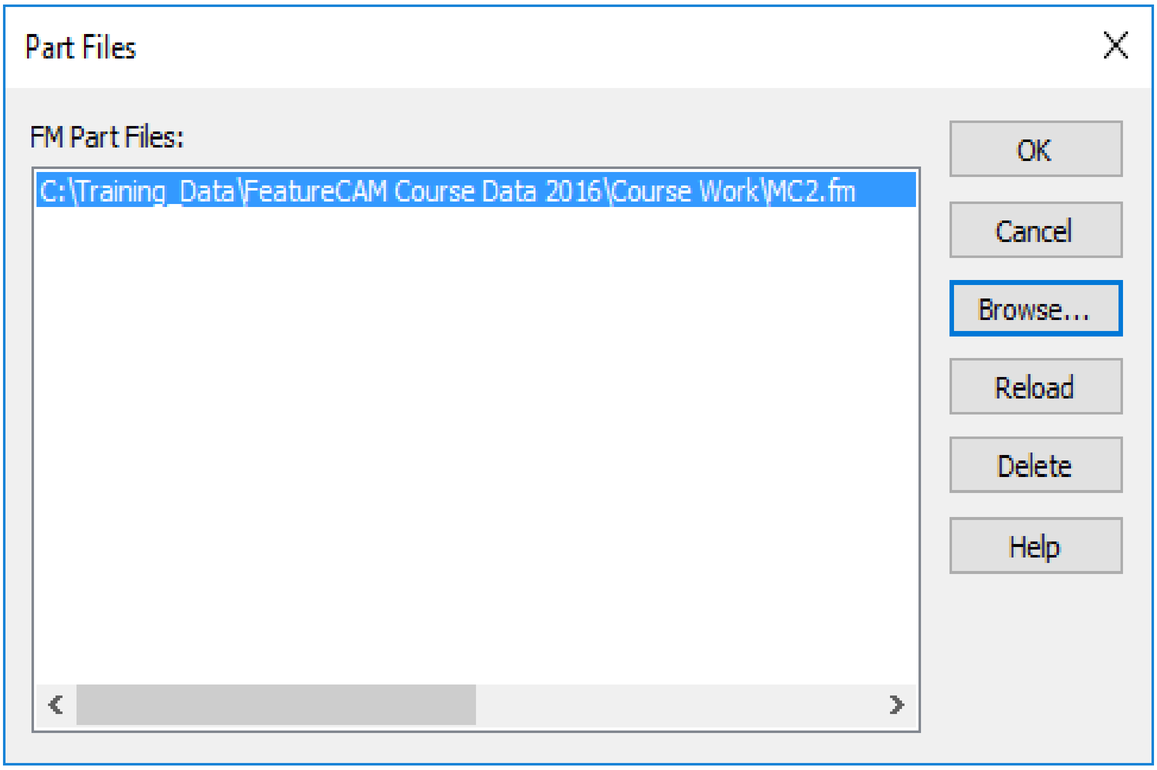


*From **Multiple fixture documents** the following menu will appear.*

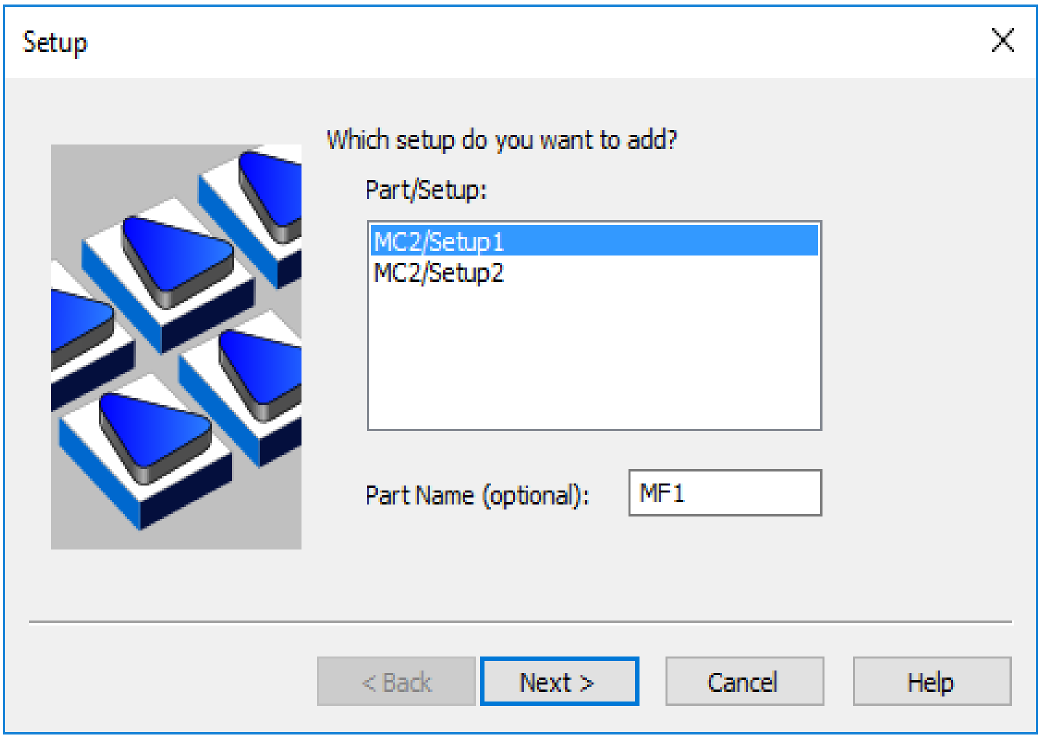


2 Select **Parts List**.

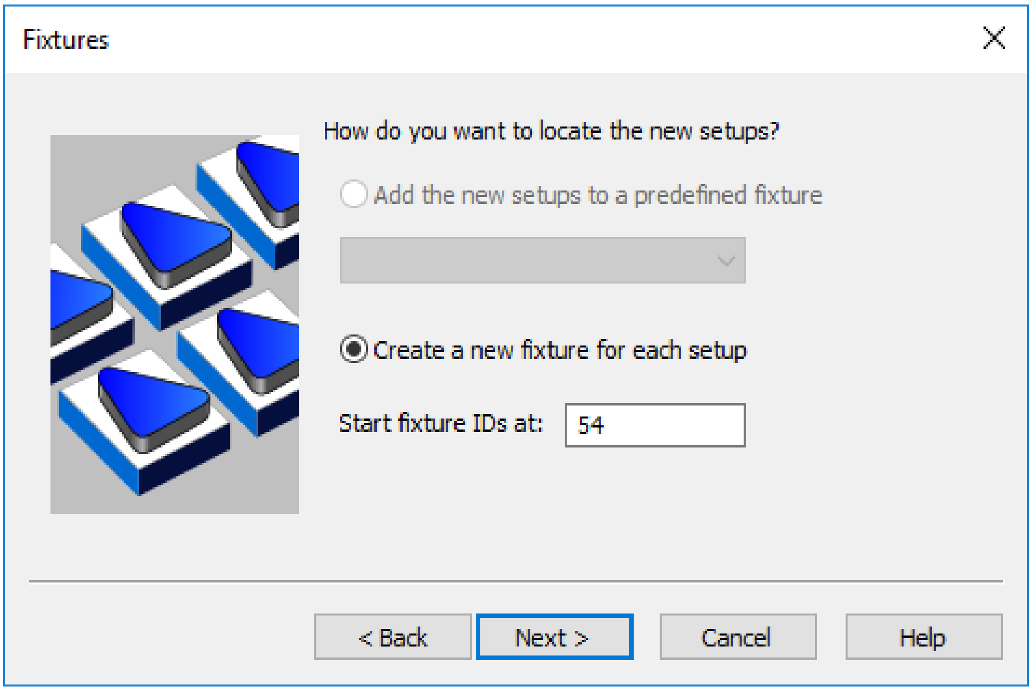
3 Select **Browse** and **add** the two files shown.



4 Select **OK** and then select **Add**.

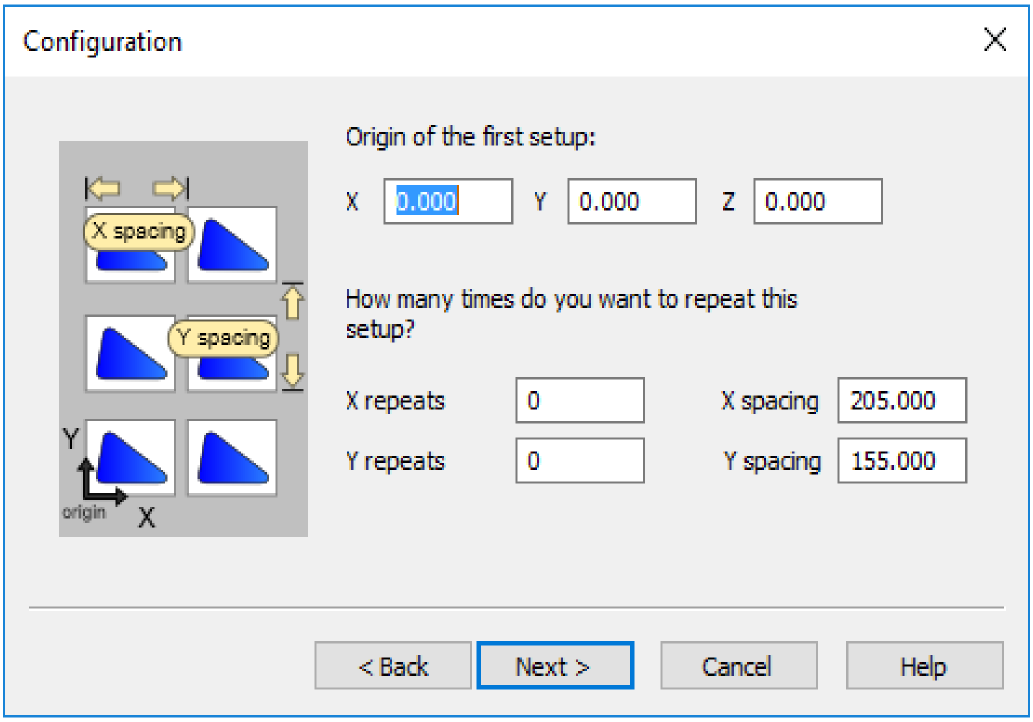


5 Select **Next**. Create a new **Fixture ID** for each Setup.



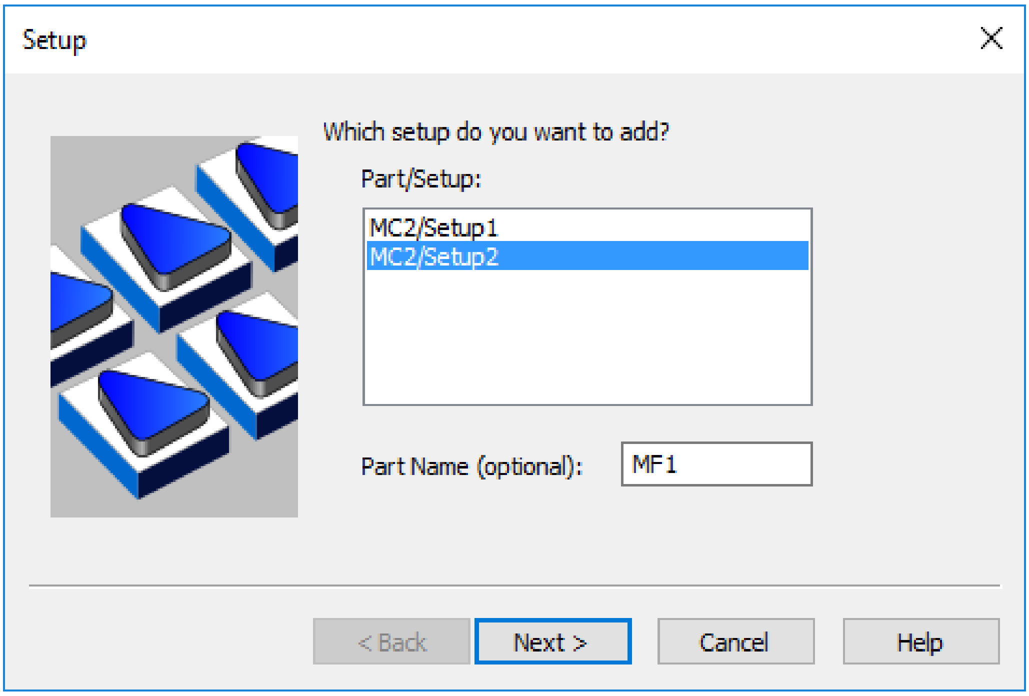
6 Select **Next**.

We will leave the first Datum at Zero in ***X0,Y0,Z0***

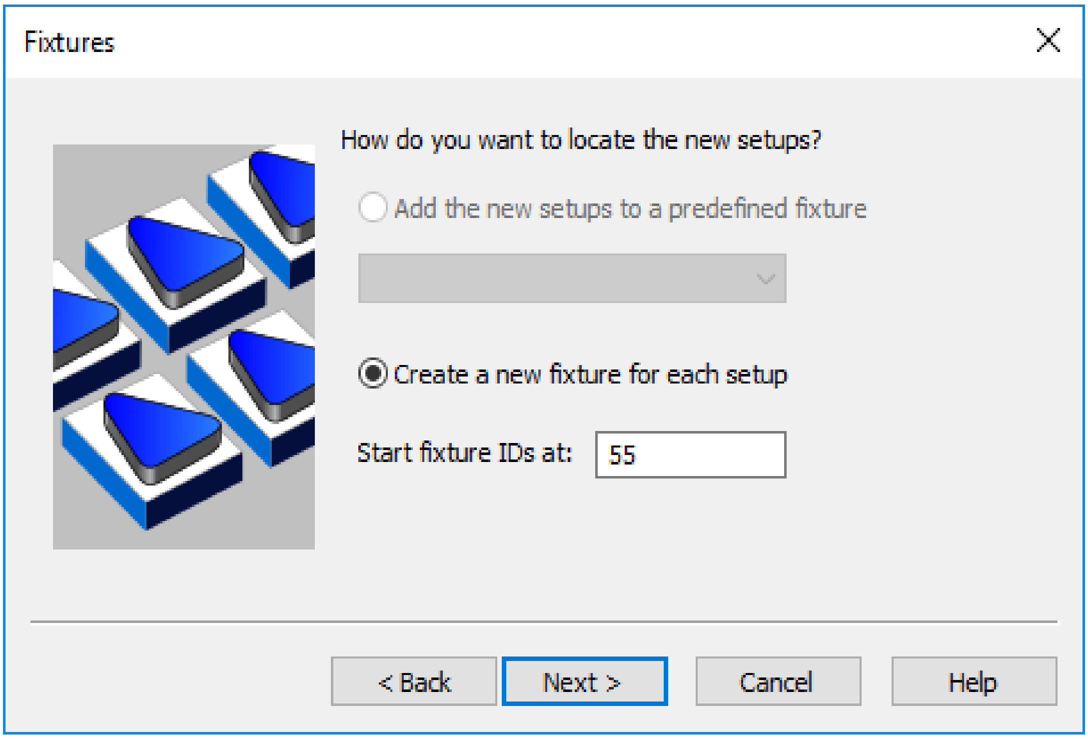


7 Select **Next**. Then select **Finish**.

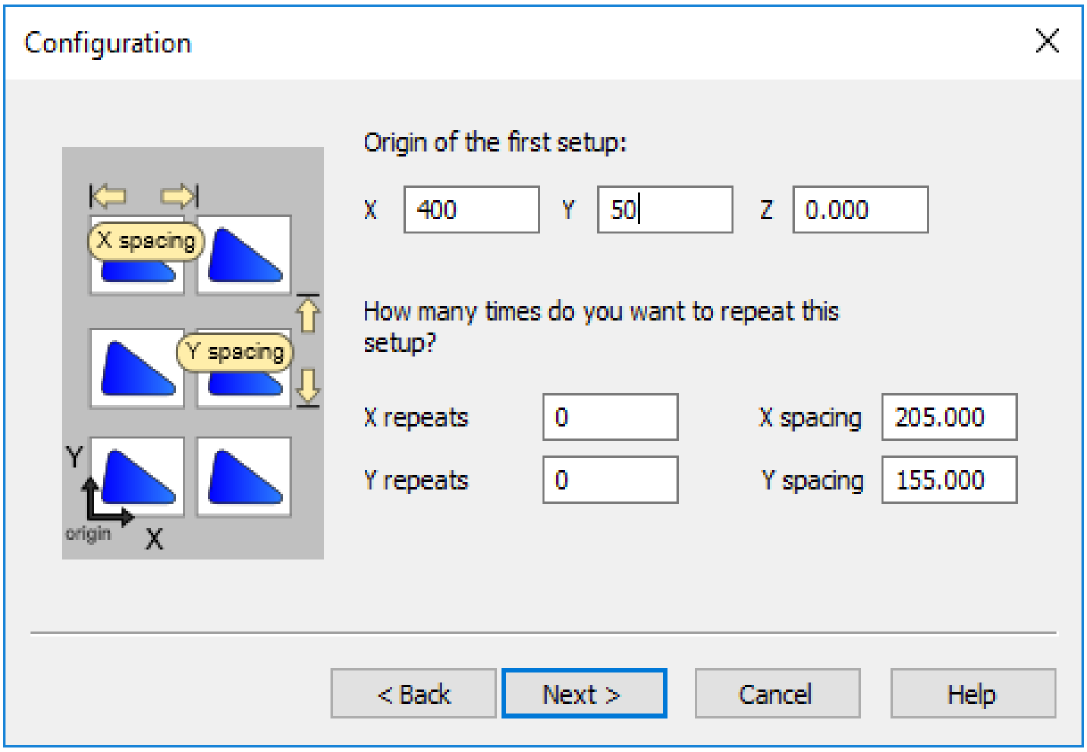
8 Then Select **Add** and then select **MC2**.



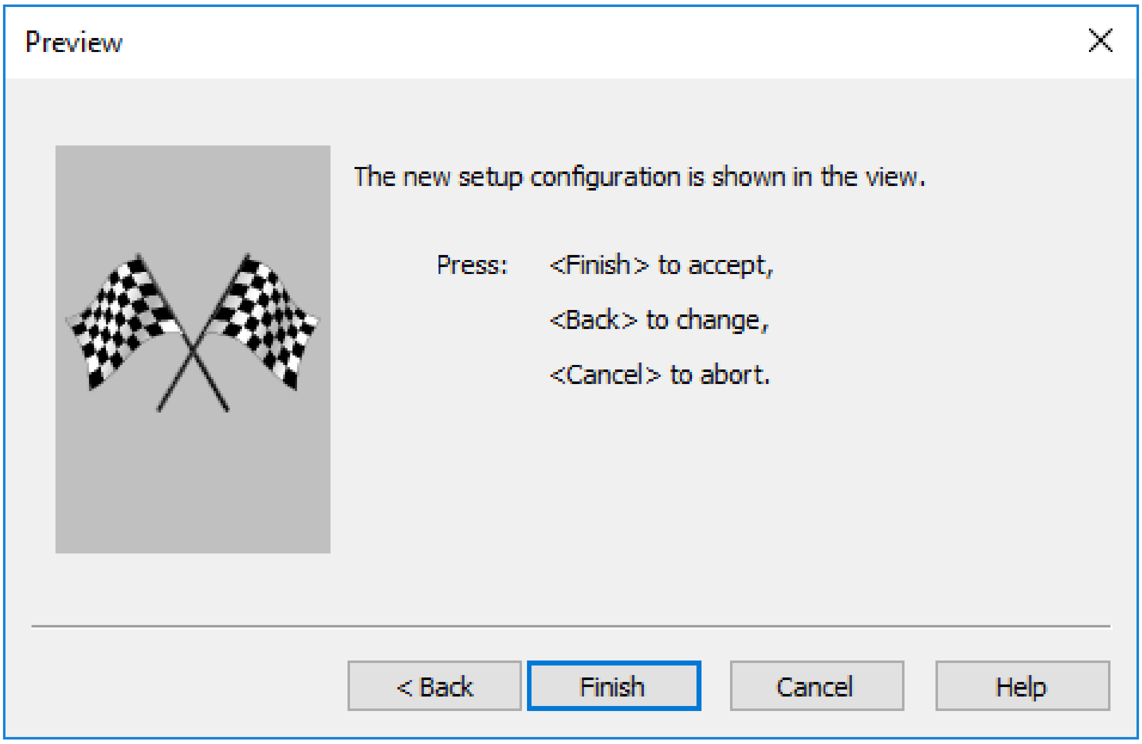
9 Select **Next**.

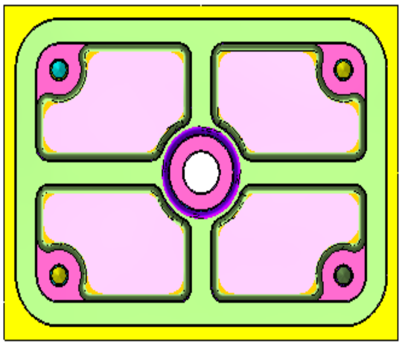


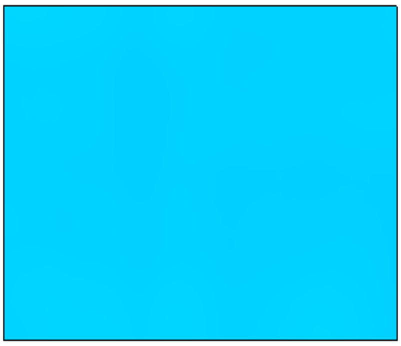
10 Select **Next**. Then change the **X** value to **400** and the **Y** value to **50**



11 Select **Next** then **Finish** to accept. Run a **3D Simulation** to see the results.





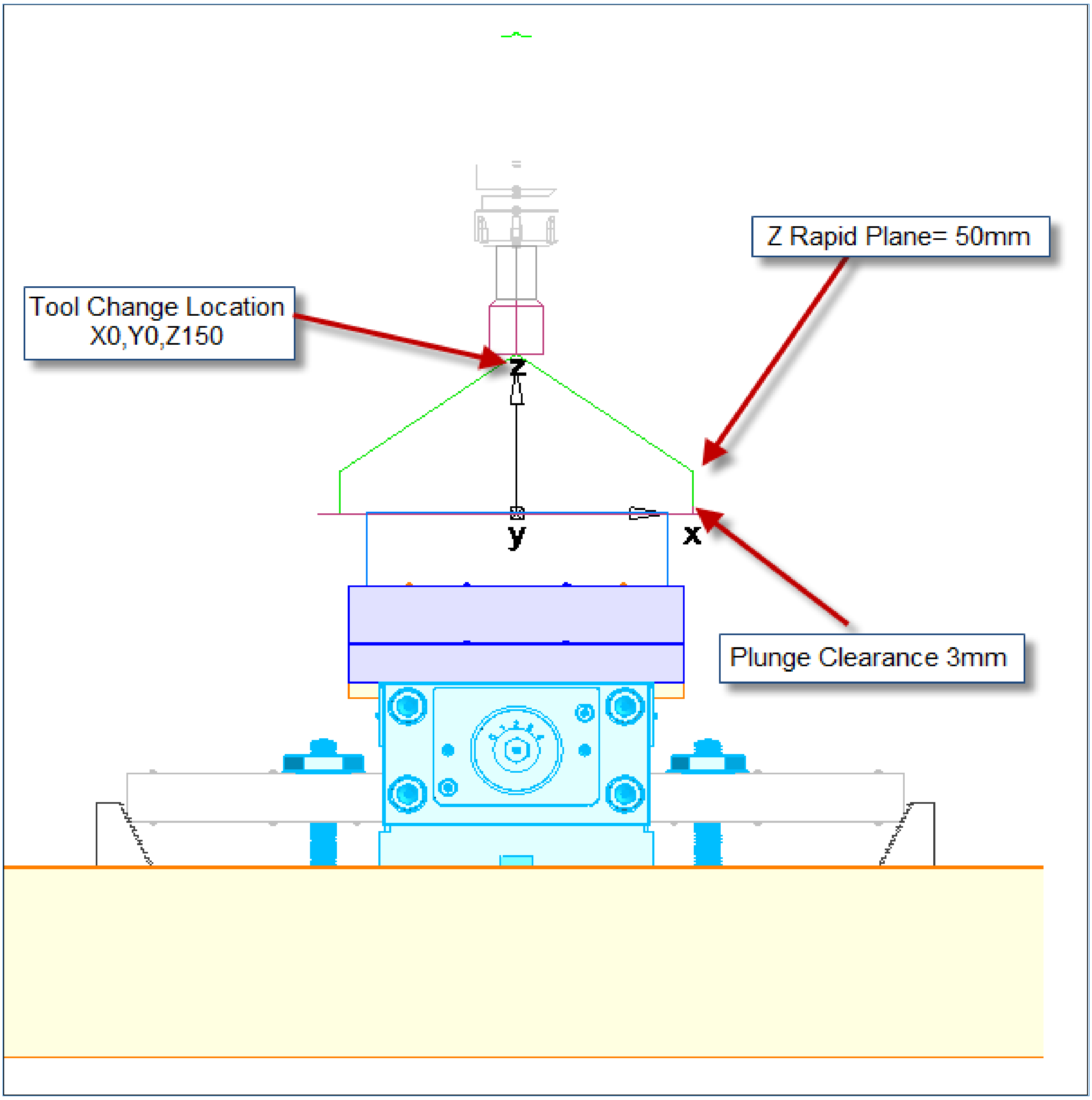


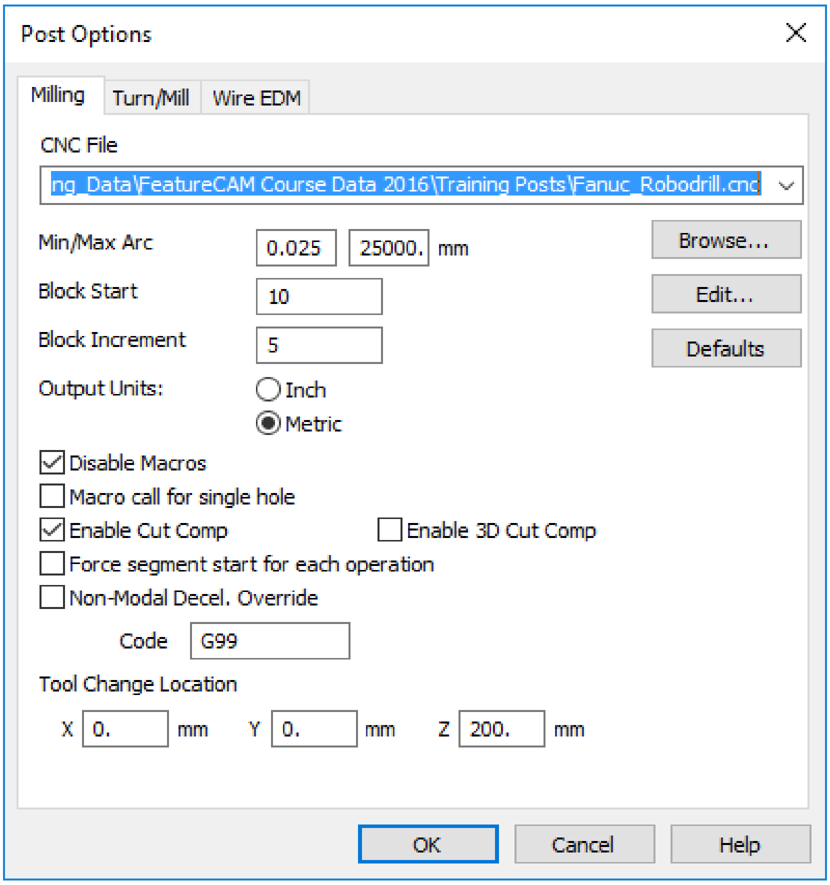
Basic Toolpath Terminology

Z Rapid Plane = 50mm (Default) *Enter the minimum safe distance in **Z** above your part which could include clamps and fixtures.*

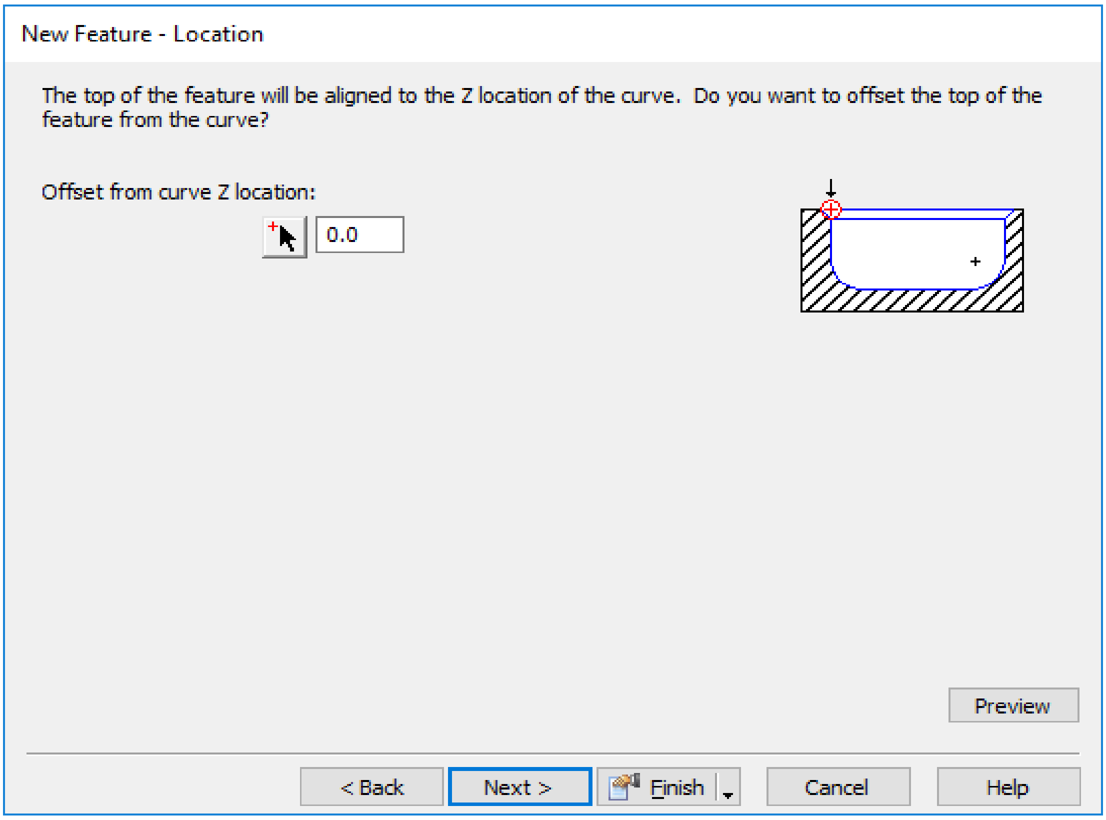
Plunge Clearance = 3mm (Default) *Enter the distance above the operation at which the tool feeds.*

Tool Change Location = *this is the point where the tip of the tool moves to prior to a tool change. (Post Options)*

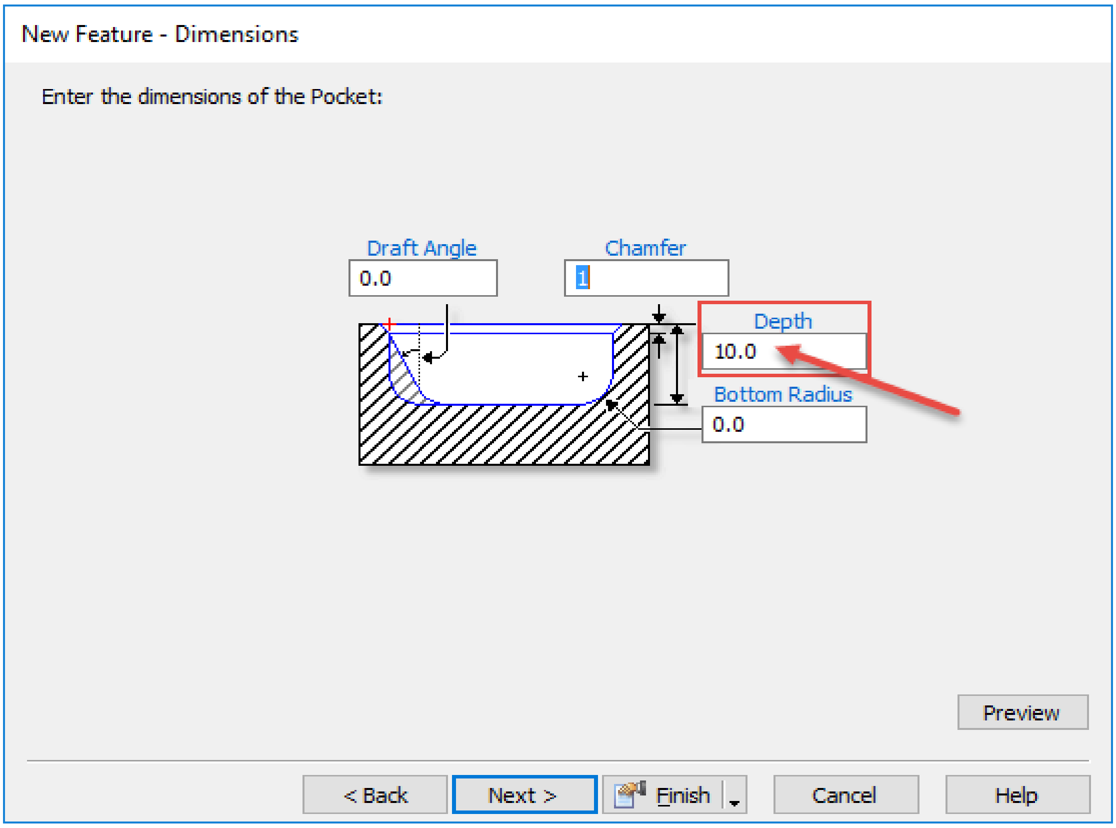




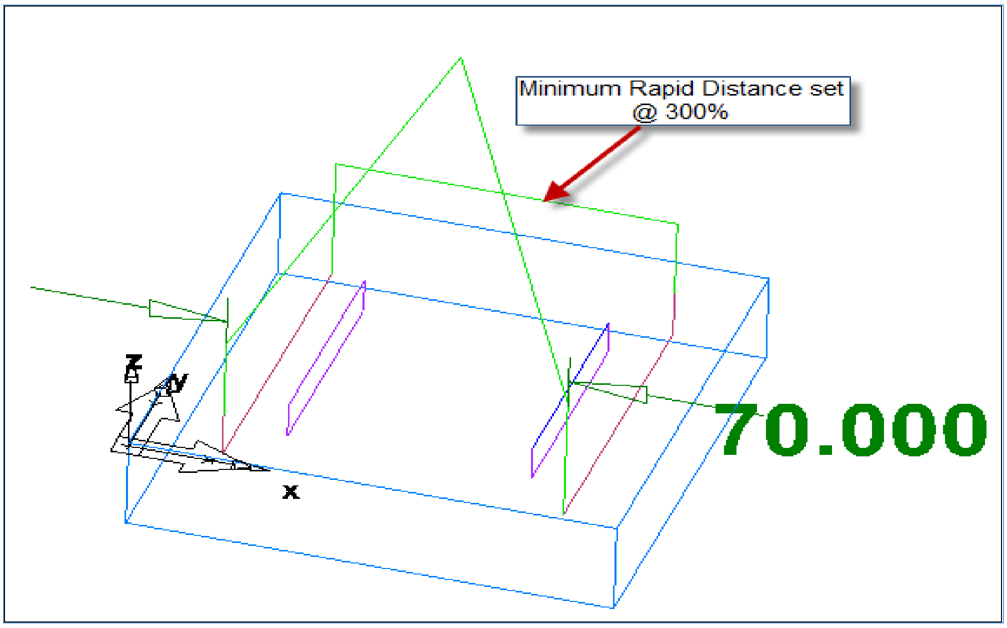
***Offset from curve Z location** = If you want to change the top of the feature, enter a value for **Offset from curve Z location**. This can be a positive or negative number.*



***Depth* = *Enter the distance cut into the material in* *Z*.**

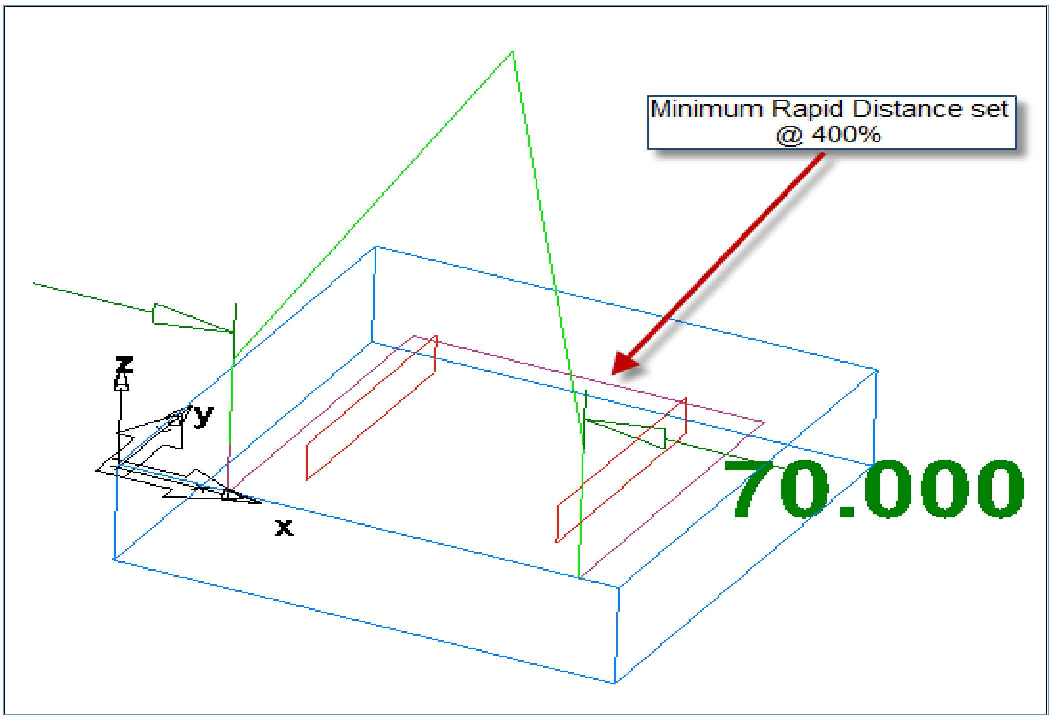


- Minimum Rapid Distance % = Enter the minimum distance, as a percentage of the tool diameter, that the tool can use a rapid move for. Moves smaller than this distance use a feed move. Please set this to **100%.** Can be set in **Machine Attributes** or **Machine configuration**. If the next rapid move is greater than the diameter of the cutter as a percentage % then the tool will lift up to the Rapid plane clear of the work piece.
- For example if you have a **20mm** dia tool and the **minimum rapid distance** is set to **300%** this will equate to **20mm x 3** = **60mm** minimum rapid distance.



*You can see that the tool lifts up between cuts set at **300%***

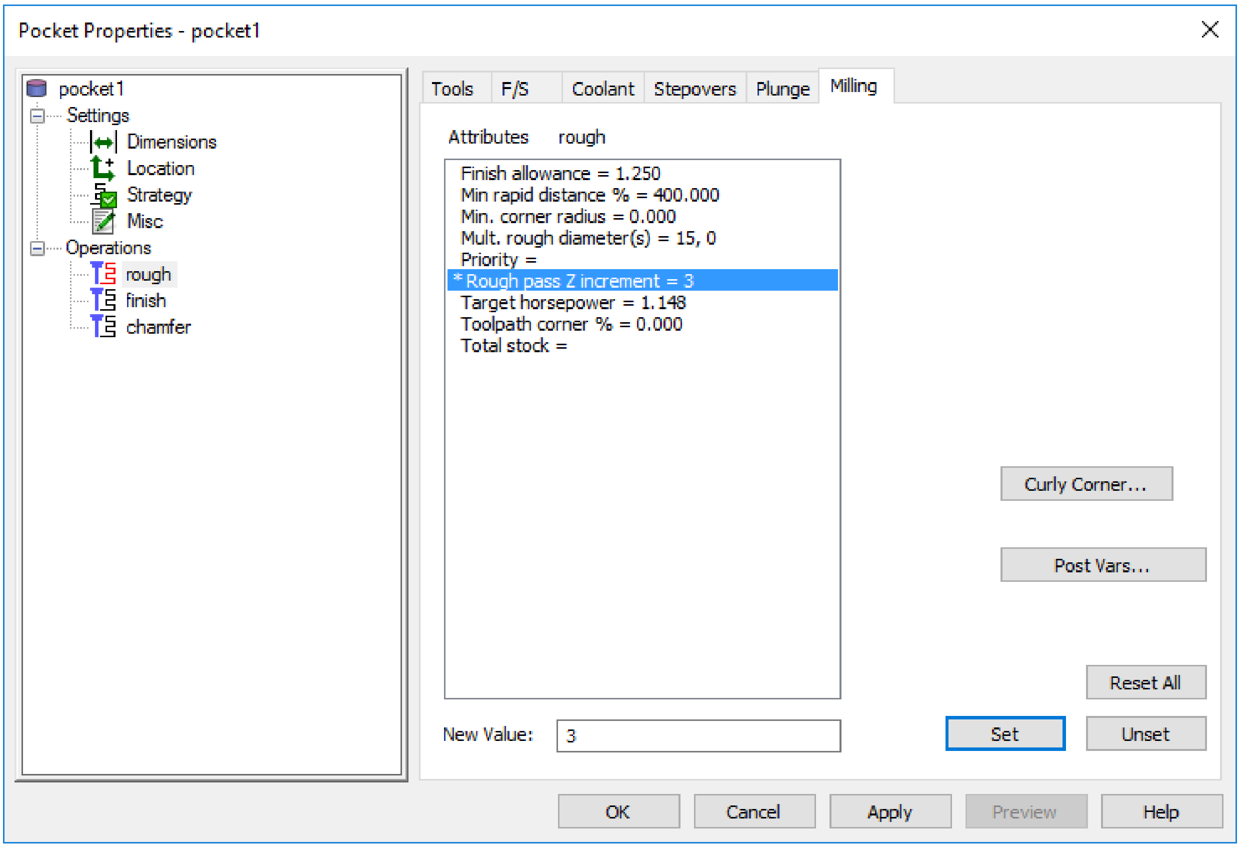
*For example if you have a **20mm** dia tool and the minimum rapid distance is set to **400%** this will equate to **20mm x 4** = **80mm** minimum rapid distance. As a guide I would always set the **Minimum Rapid distance** between **50%** and **100%. (Default = 400%)***



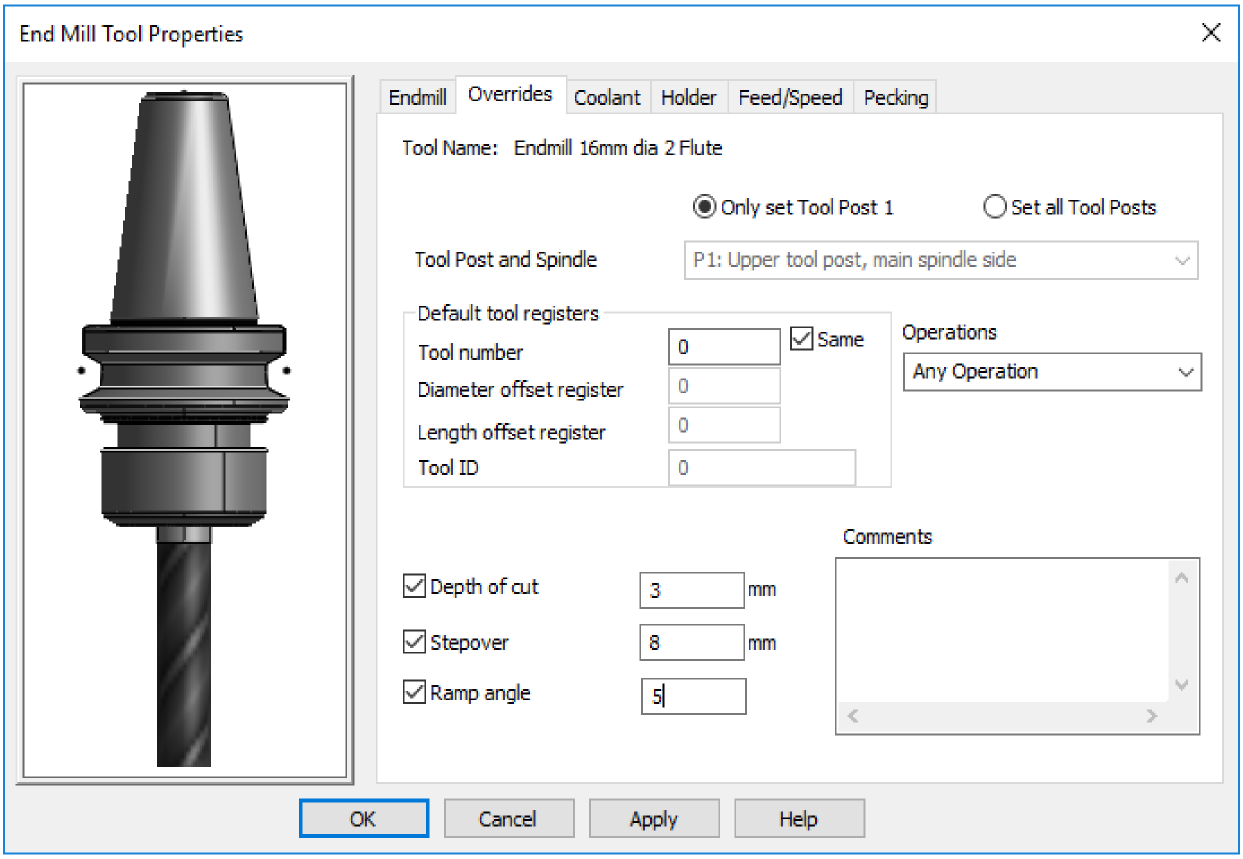
Please note that the tool will remain down and move at the current feedrate (G01).

***Rough pass Z increment: -** This sets the depth of cut for the Rough pass.*

*Enter a depth Z increment for each pass that the roughing routine performs on the part**.** You can set the depth of cut in several places. Also **Finish pass Z increment** is available under the **Finish** Tab.*



***Depth of cut** = **3mm** (Endmill Tool Properties)*



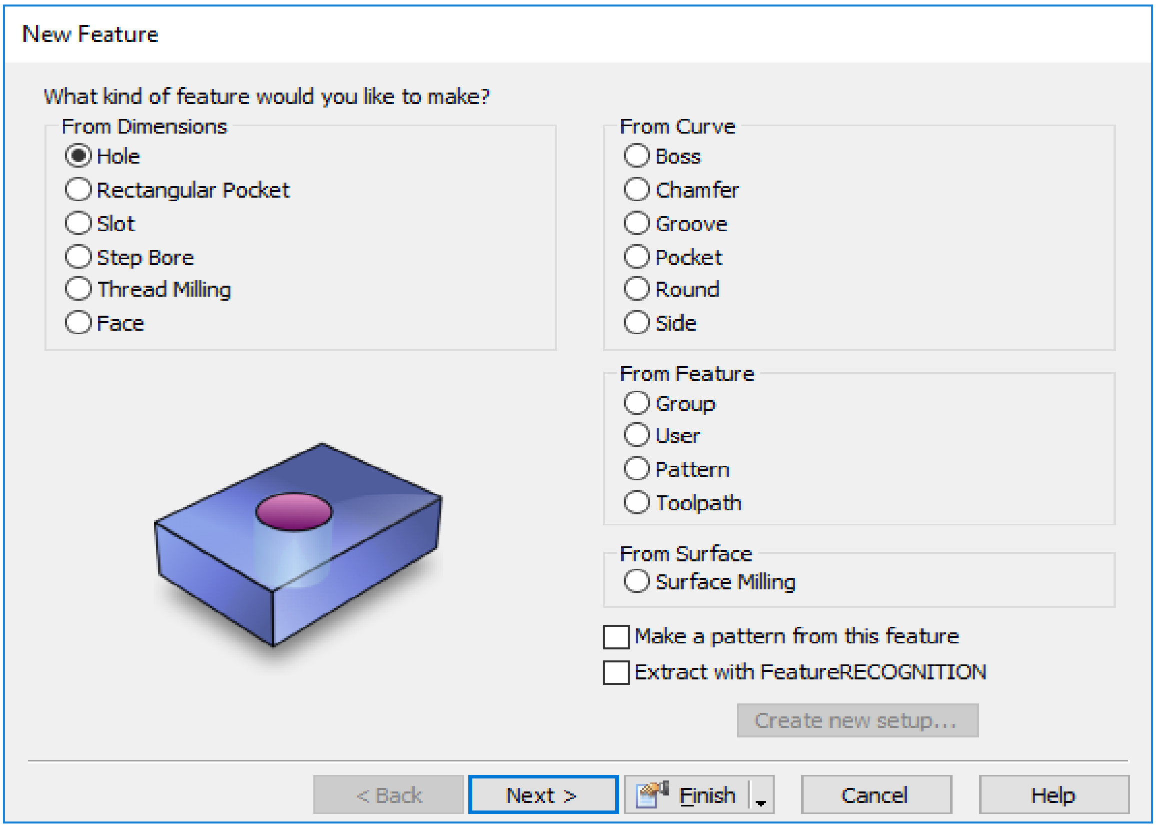
Understanding 2.5D Features

What are Features?

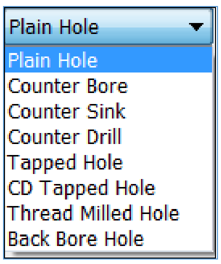
The easiest way to think of **Features** is as individual machining operations that the user would have to perform on a machine tool. For example, if the user wanted to face off the top of a block and then drill some holes at various points on that block. They would first need to put a Face mill into the machine tool and machine across the top of the block. This would be defined as a **Face Feature**. If they then wanted to drill some **10mm** diameter holes, they would need to change the tool and then maybe centre-drill and drill the holes that were **10mm** in diameter. This would be **Hole Feature 1**. If there were any more holes of a different size, these would need to be defined as another separate **Feature**, so the next **Feature** would be **Hole Feature 2** and so, on until the job is complete.

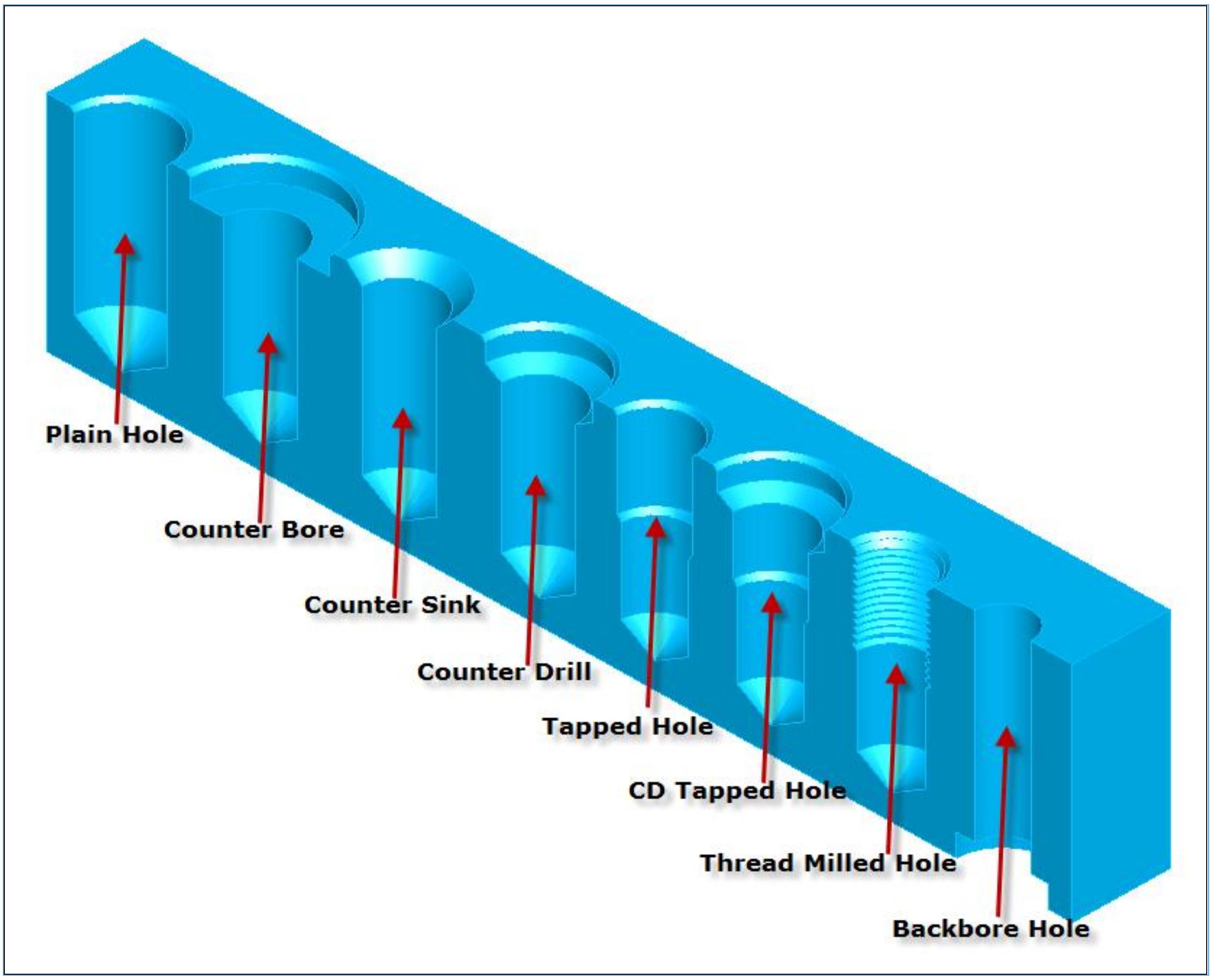
Feature Types

FeatureCAM uses various Features to define any **2.5D** shape and they are found on the New Features form which can be accessed via the Steps Toolbox.

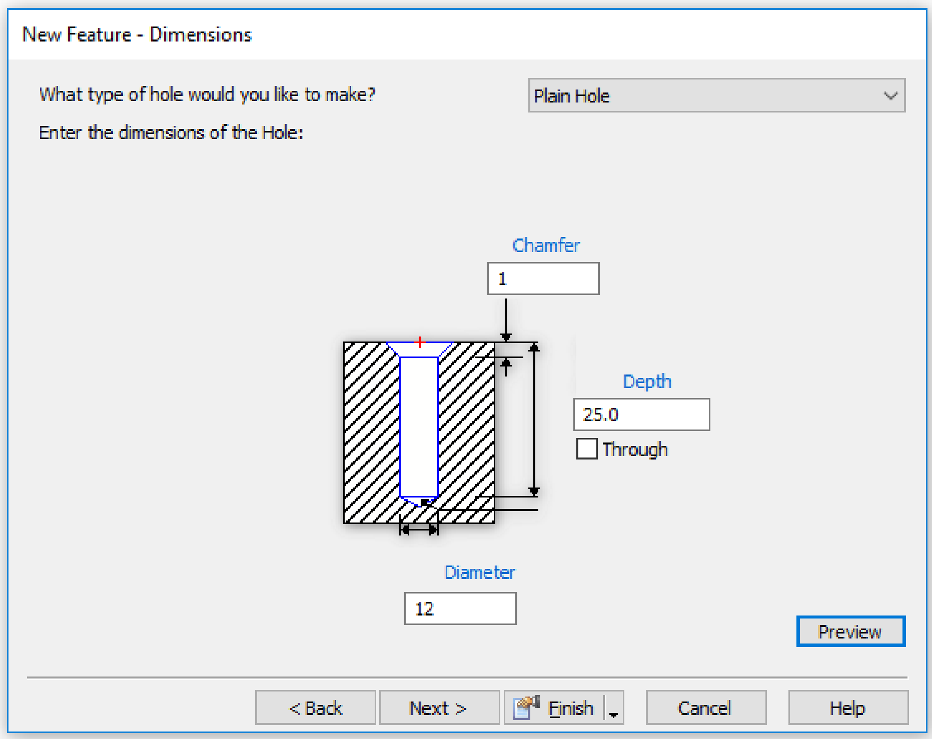


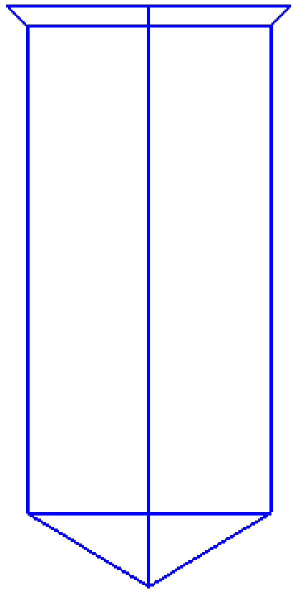
- They are separated into **Groups**, depending upon how the user wishes to define the **Features**.
- There is a **From Dimensions** group which uses dimensions that the user might take from a drawing to describe the shape. Or **From Curve** group which uses **Curves** to define the shape of the **Feature**.
- All **Features** created inside of **FeatureCAM** are initially created using the **New Feature Wizard**. The **New Feature Wizard** can be accessed in several different areas of the user interface for example. **Advanced Toolbar**, **Steps Toolbar** or **File Menu**.
- Within the **From Dimensions** Group are the following categories.
- **Hole**, **Rectangular Pocket**, **Slot**, **Step Bore**, **Thread Milling** and **Face**.
- We will work through all of Features listed and explain how they function.
- **Holes** are created by **Drilling** or **Boring** and may have other characteristics such as a **Chamfer** or **Tapped threads**. They are manufactured using canned drilling cycles.



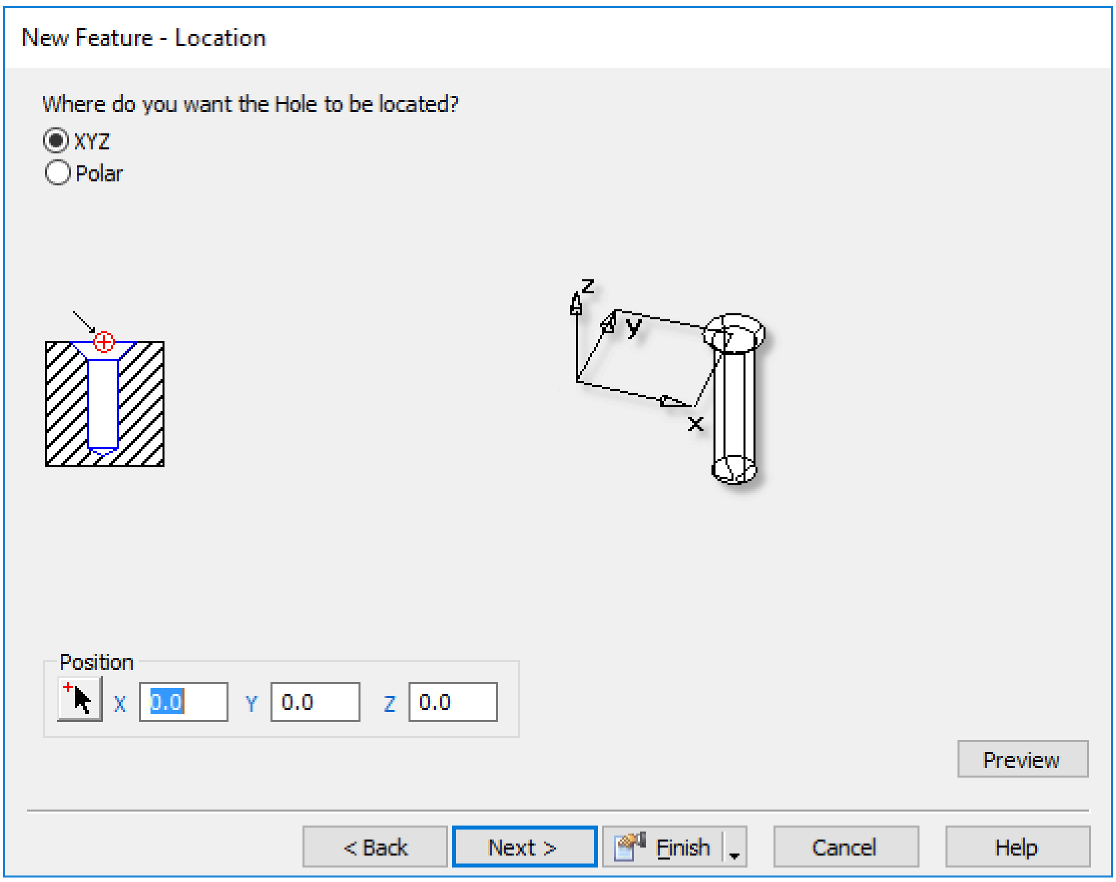

Plain Hole
Counter Bore
Counter Sink
Counter Drill
Tapped Hole
CD Tapped Hole
Thread Milled Hole
Backbore Hole

*We will start with the **Plain Hole**.*

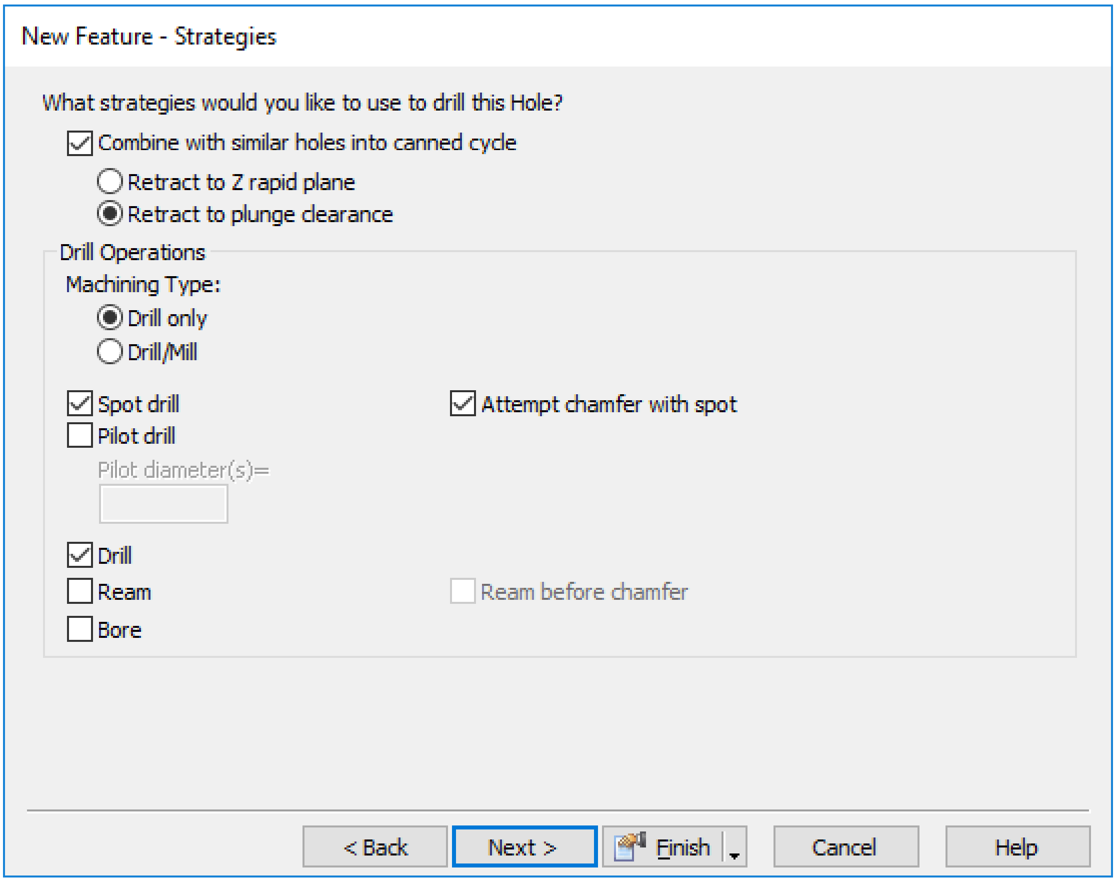




- Select **Next** and you will be presented with a location menu



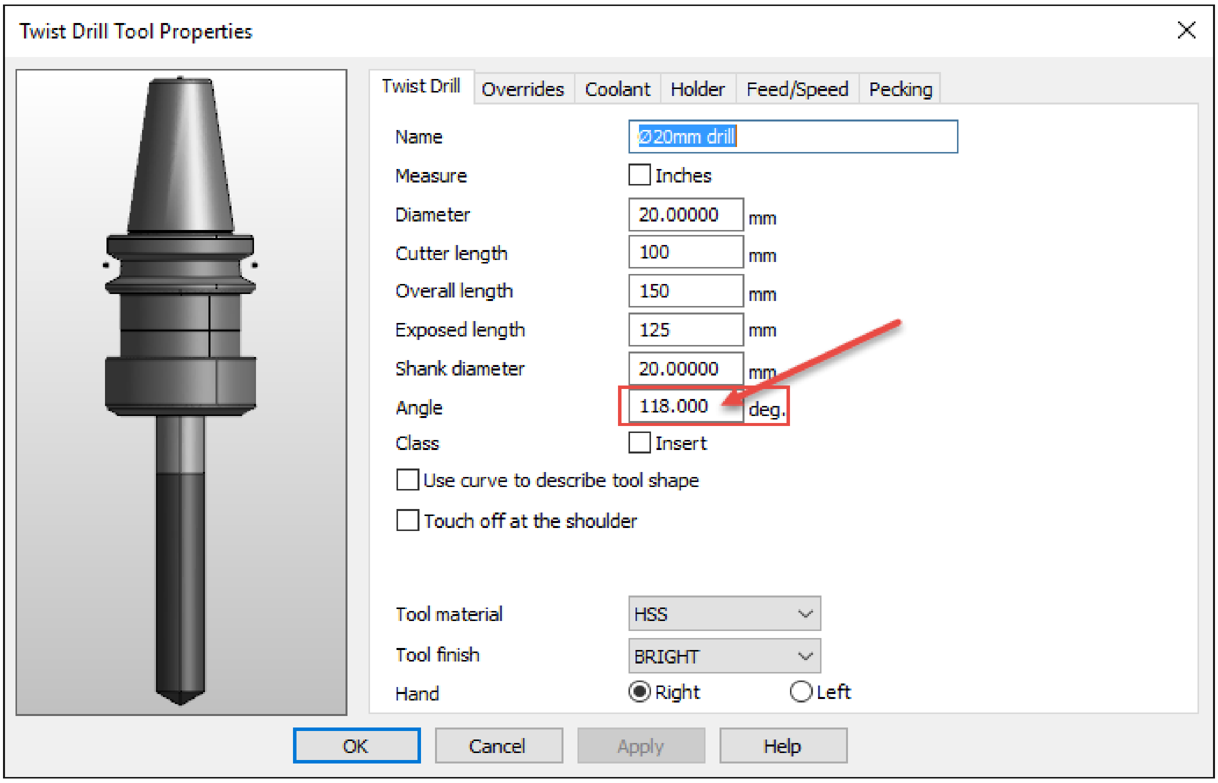
*Enter the coordinates for your hole. After the location menu has been displayed select **Next** and you will get the **strategies menu** which will allow you to customise the drilling sequence.*



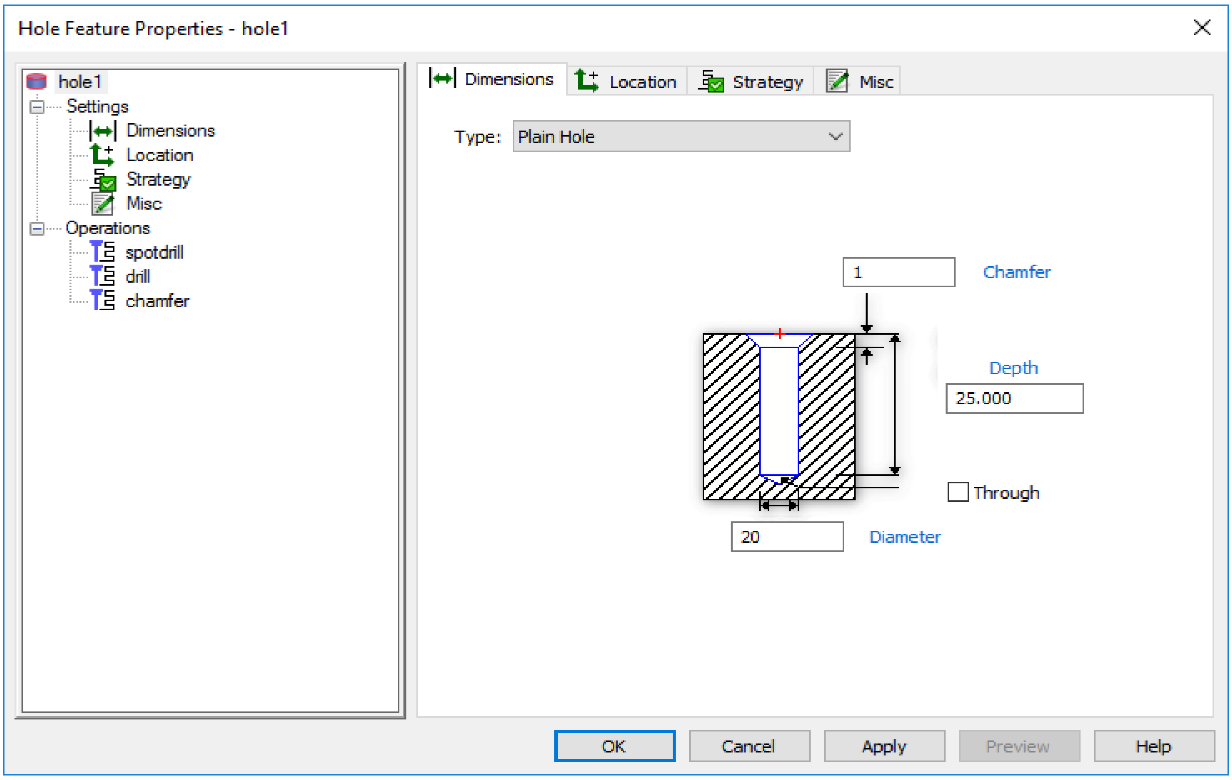
*Select **Finish.** Your **Hole Feature** has been created and machined.*

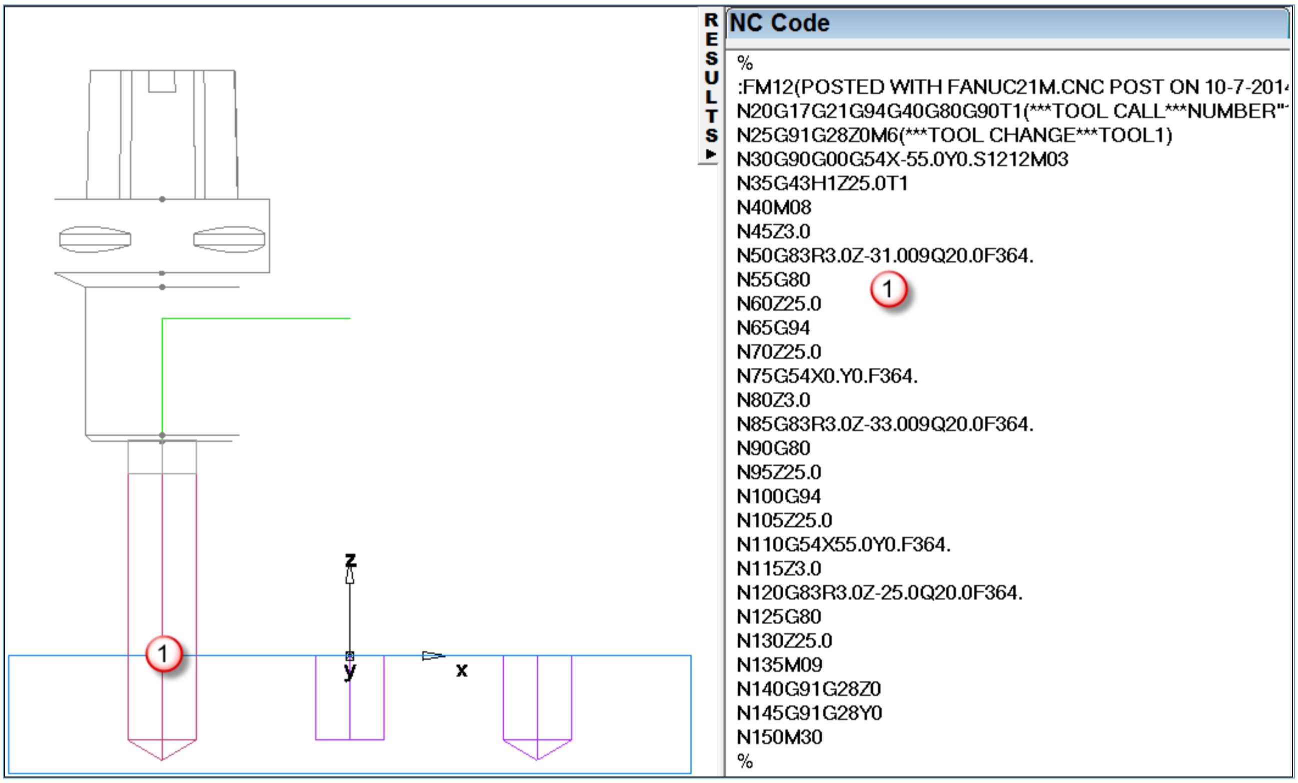
Hole: Drill to depth (Information Only)

- The actual depth of the twist drill operation is determined as follows:
- **Drilled Depth = depth + (diameter / 2) / tan (Angle of drill / 2)**
- For example, using a **20mm** drill with **118 degree Rake angle**. The depth is set to **25mm**. Allowing for **Drilled Depth = depth + (diameter / 2.0) / tan (Angle of drill / 2.0)** the actual drill depth will go to **Z-31.009mm**

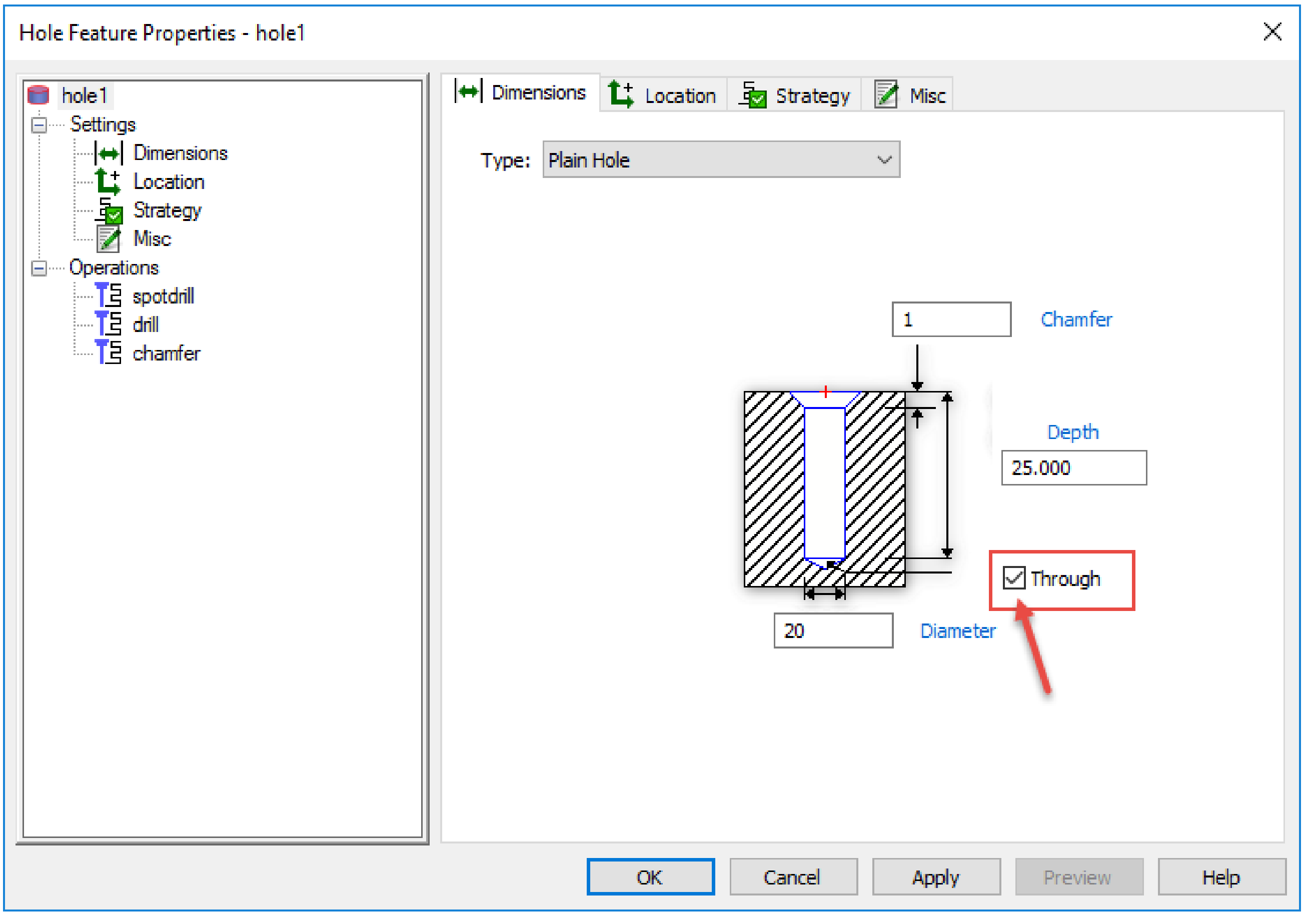


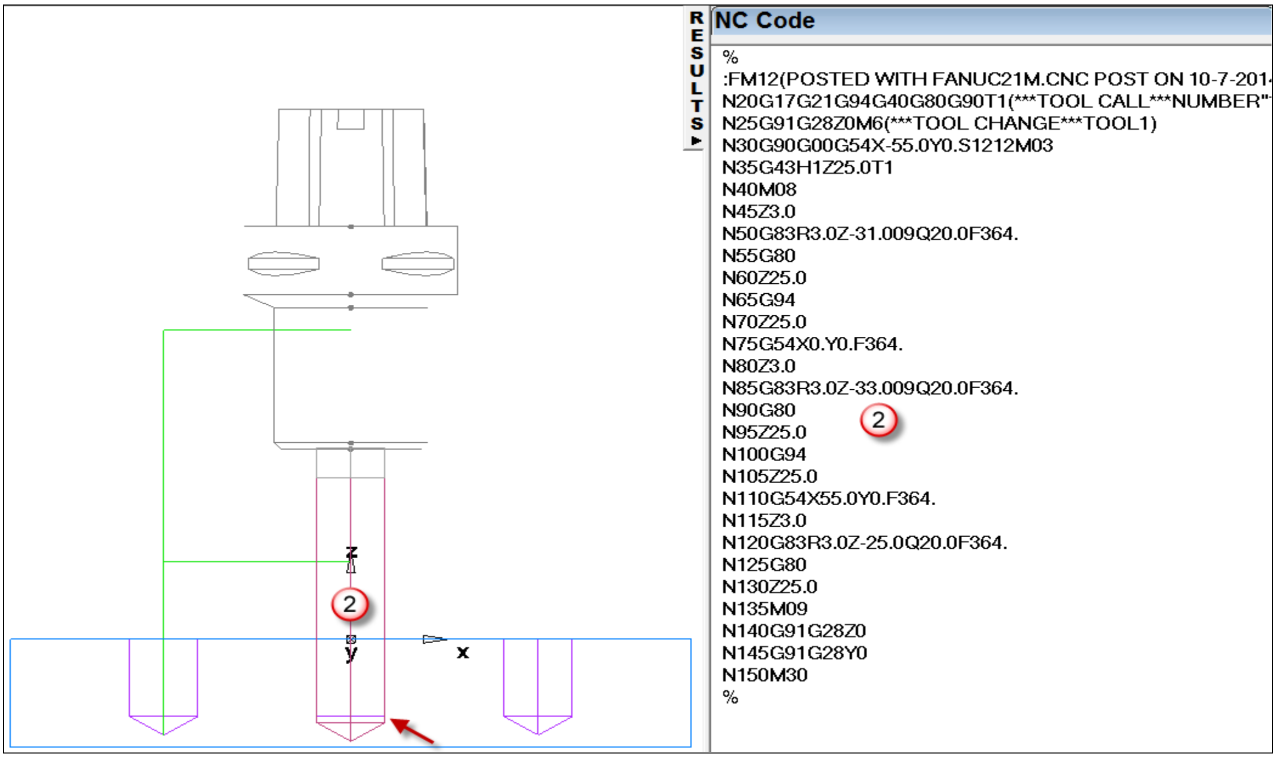
Example Drill to Depth Hole 1





Example Drill to Depth Hole 2

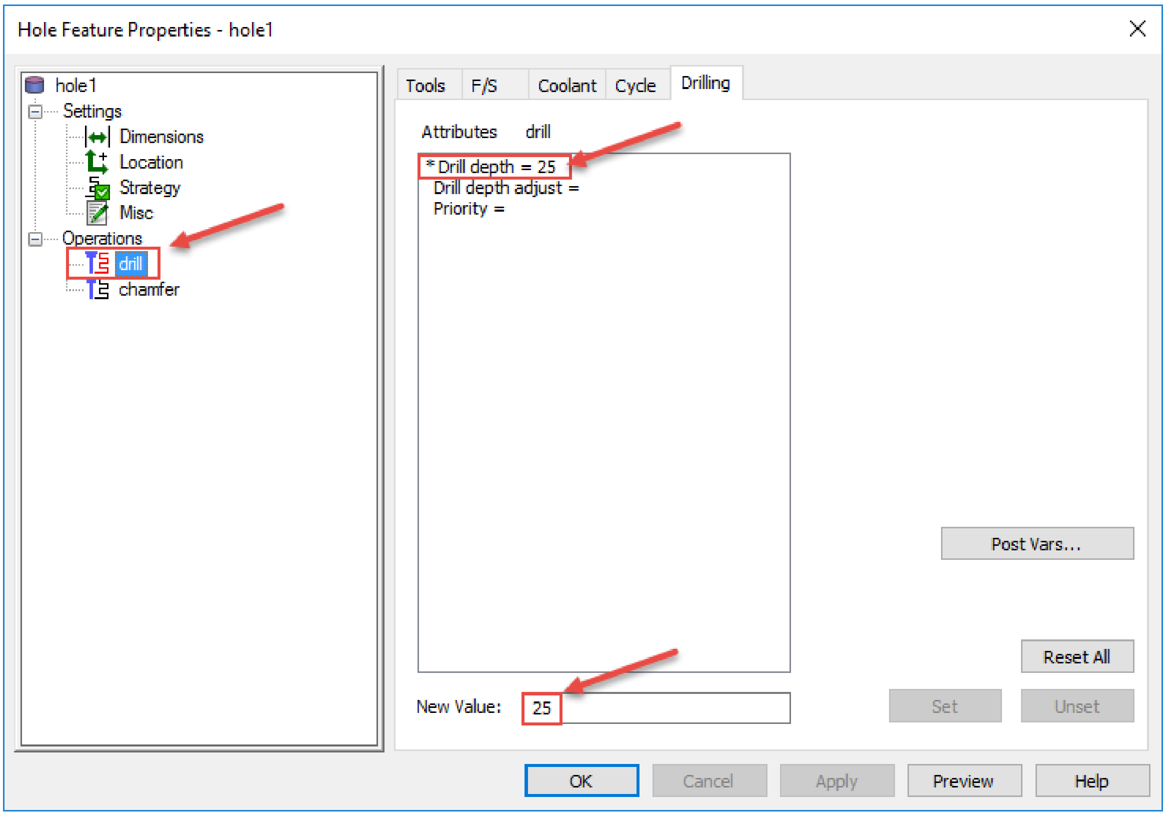


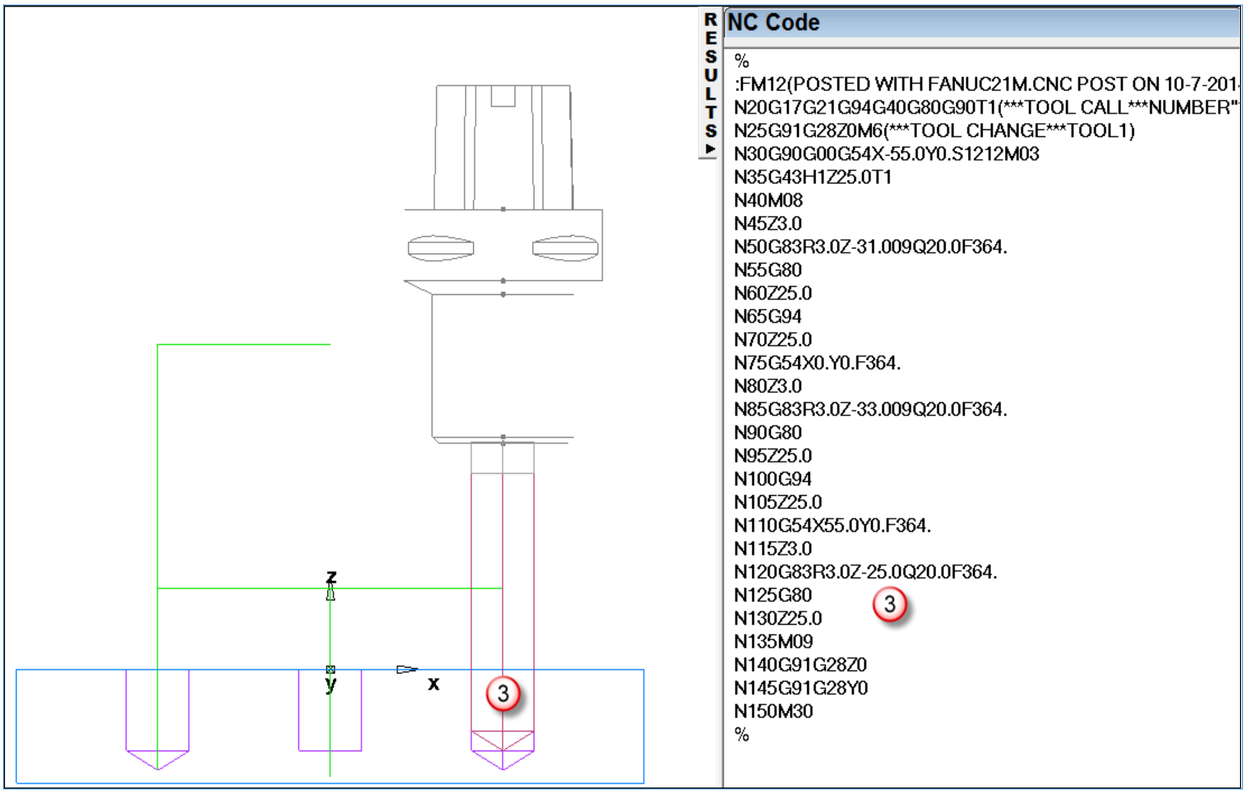


- *With **Through*** selected then this adds **0.1*Diameter** to the depth. The actual depth of the drill with **Through** selected is **Z-33.009mm**

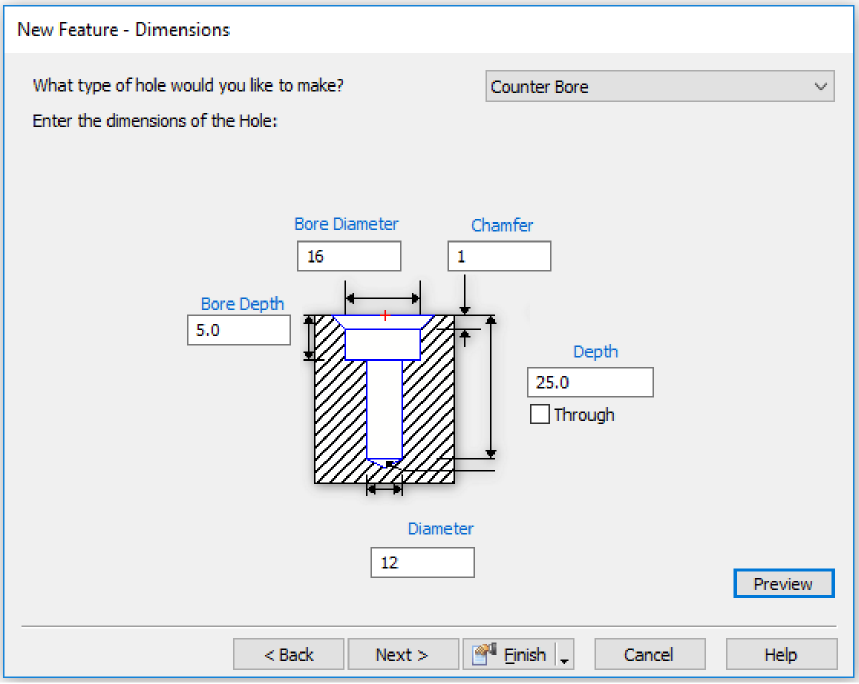
Example Drill to Depth Hole 3

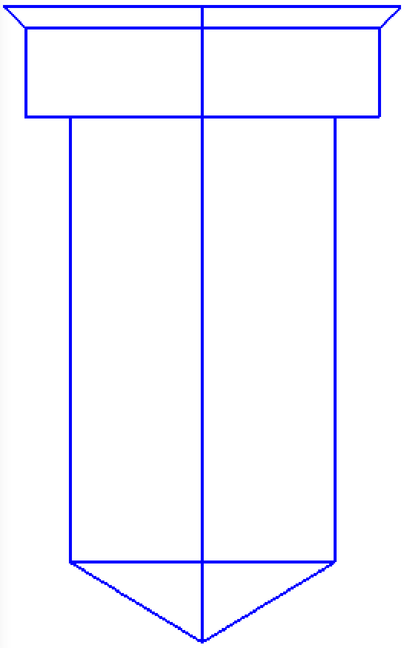
*If you want to force the depth from the drill point then select the **Drilling** Tab and enter the depth that you want to drill to from the drill point.*





*Other **Hole** types are available for example **Counter Bore.***

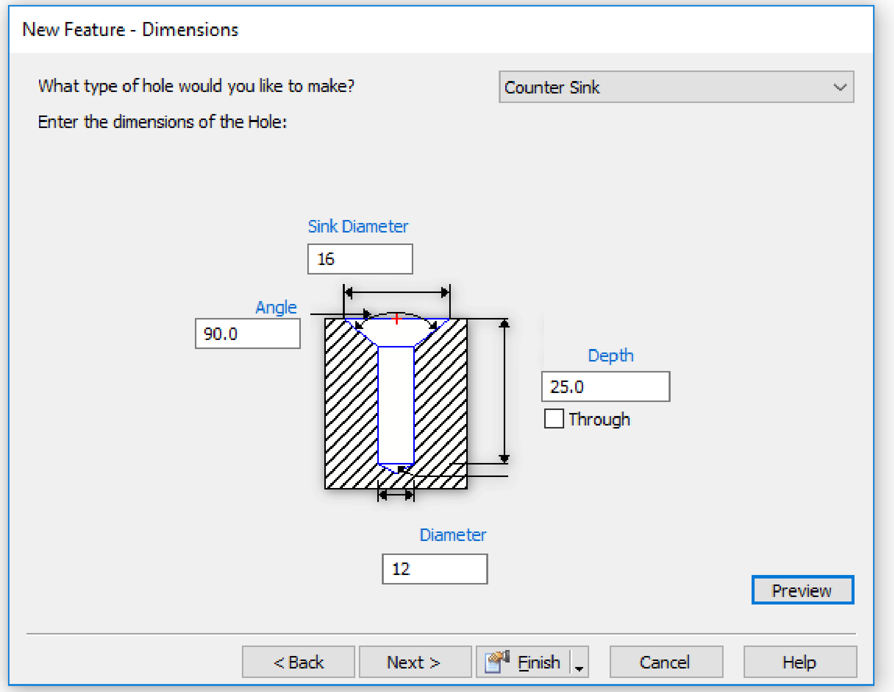


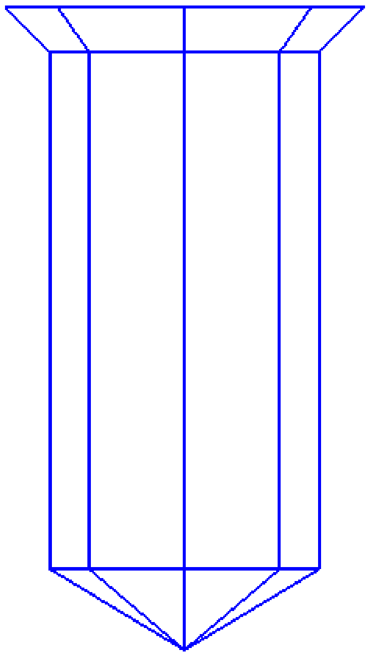


- Enter a **Diameter** value.

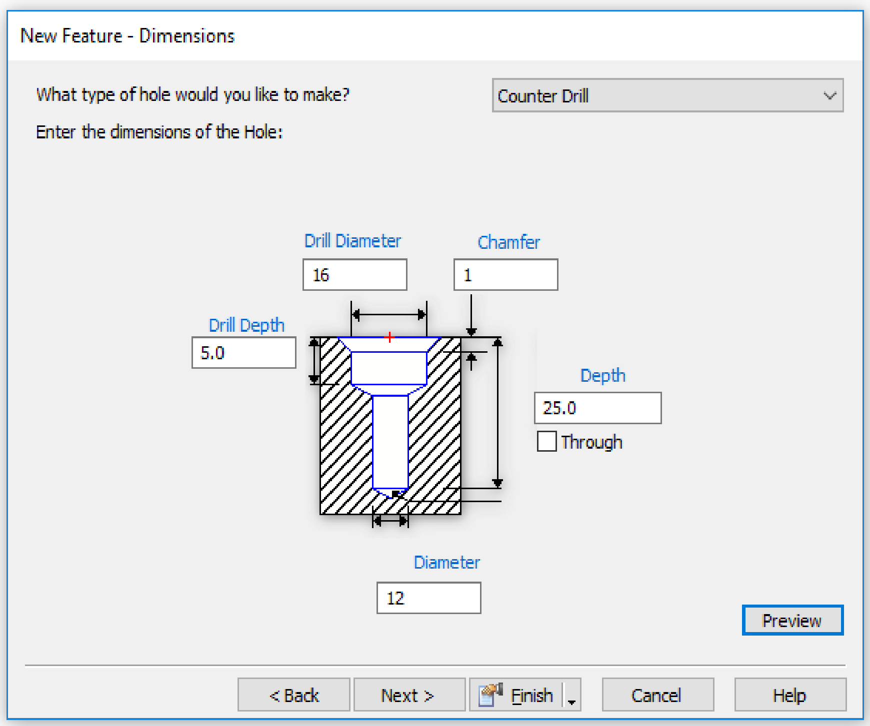
- If you are building holes from circles, select the circle before opening the wizard to pre-populate this field.
- Enter how **deep** the hole is in the **Depth** field.
- Depending on the type of hole you selected, you may have other dimensions to fill in such as **Chamfer size** and **Drill Depth**. For tapped holes, you can click the **Standard Threads button** and select a **thread type**. Each thread type sets the Thread depth, TPI, and Diameter dimensions.
- Select **Next** to open the **New Feature - Location** page. Select whether you want to enter the location as XYZ or Polar.
- For XYZ, enter the X, Y, and Z coordinates or pick the location in the Graphics window.
- For Polar, enter the Radius (the distance along the X axis), the Angle, and the Z location.
- Click Next to open the New Feature - **Strategies** page. The options on this page are the same as those on the Strategy tab of the Hole Feature Properties dialog.
- Click **Next** to open the **New Feature - Operations page**.
- Click **Next** to open the **New Feature – Default Tool page**.
- Click **Next** to open the **New Feature – Feeds/Speeds page**.
- Click **Next** to open the **New Feature – Summary page**.
- Select **Finish**, to create the feature and exit the wizard or click **Back** to return to previous pages.

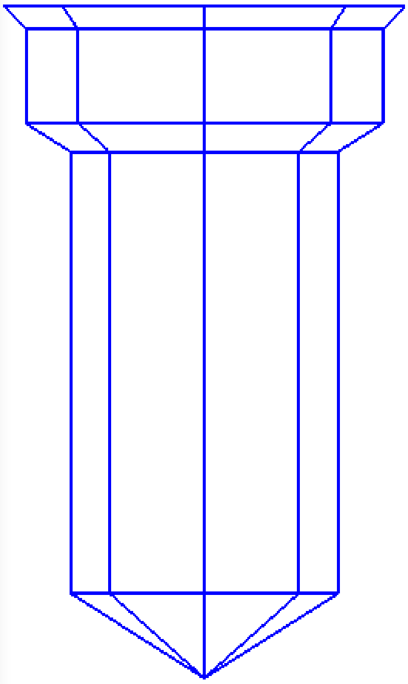
Counter Sink (Information Only)



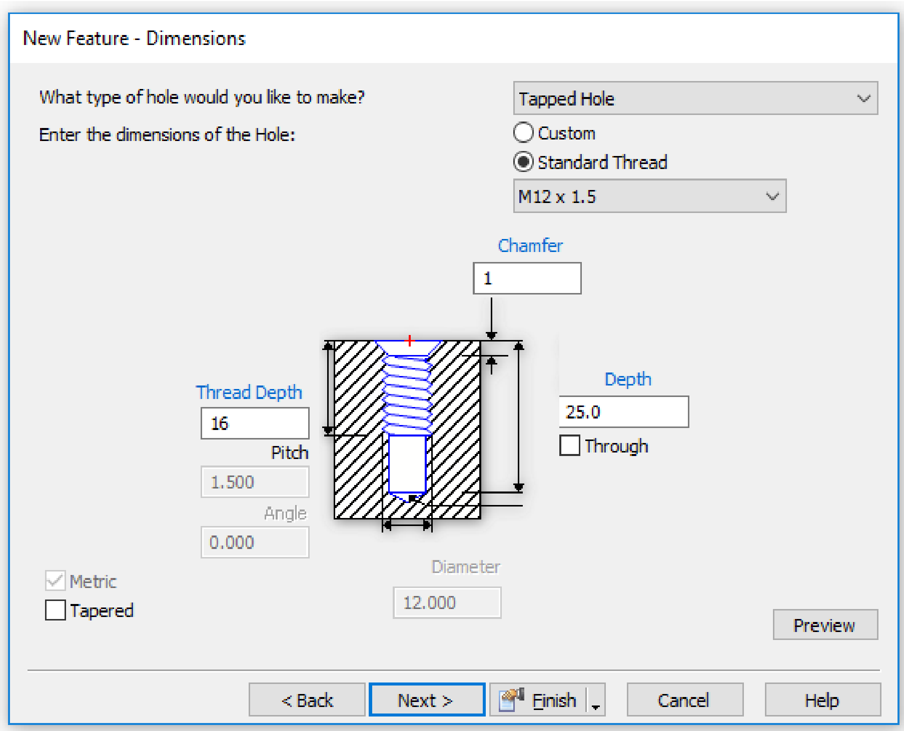


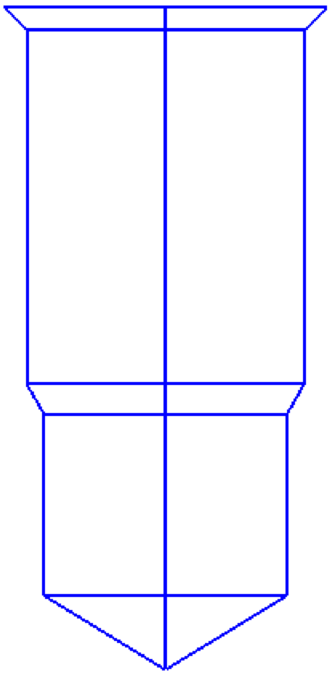
Counter Drill (Information Only)



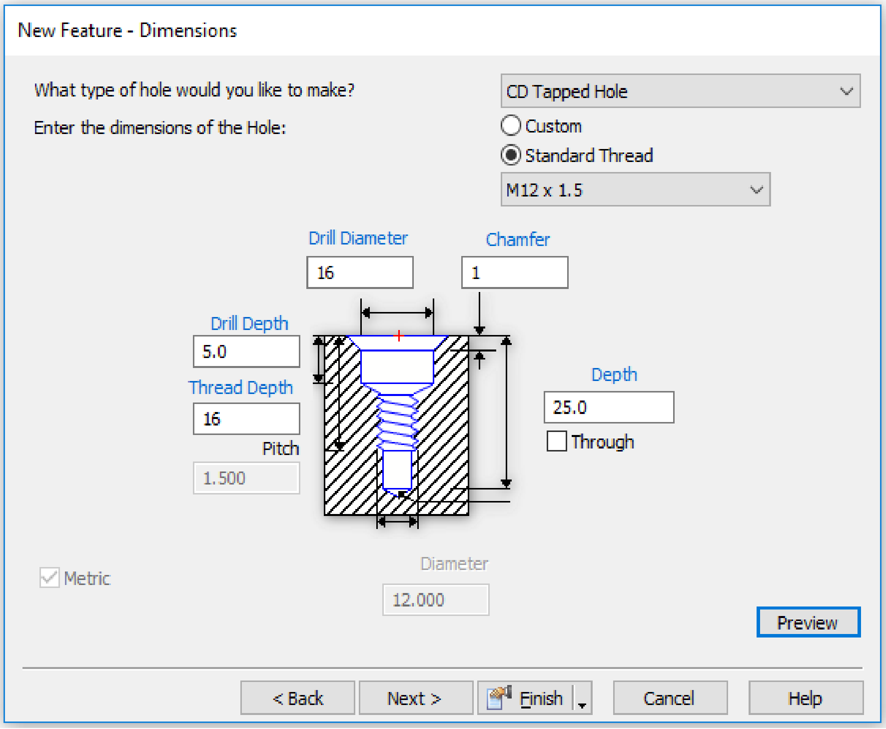


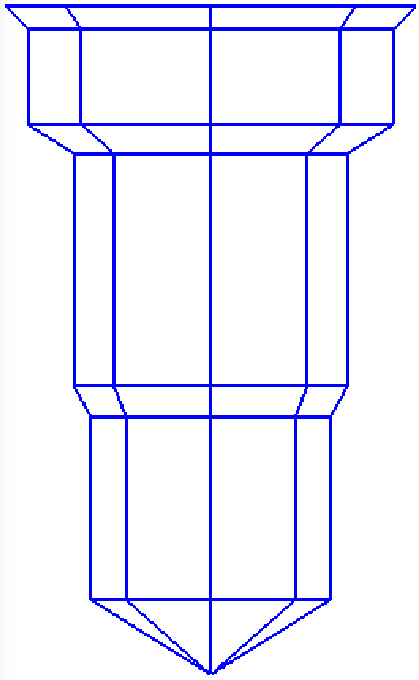
Tapped Hole (Information Only)



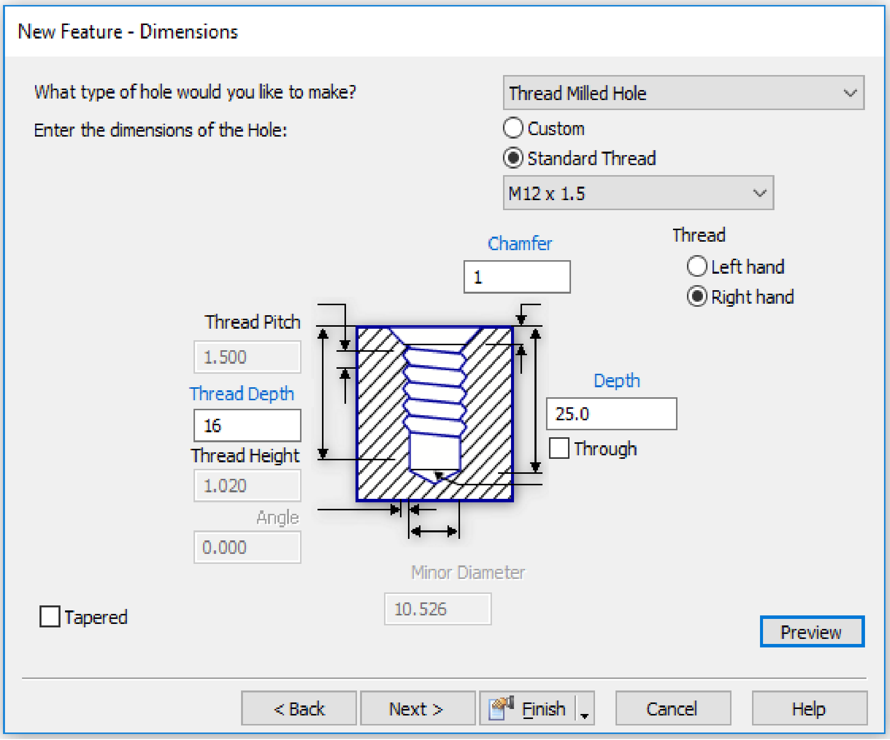


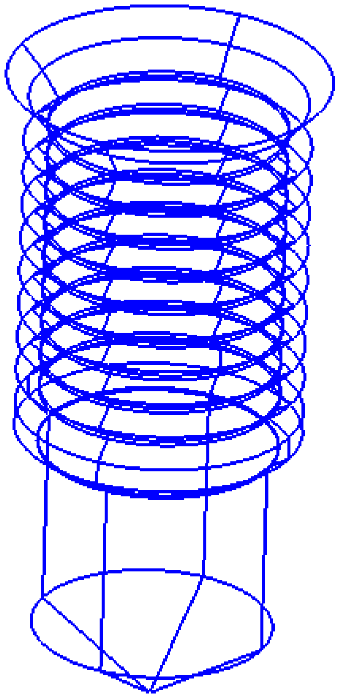
CD Tapped Hole (Information Only)

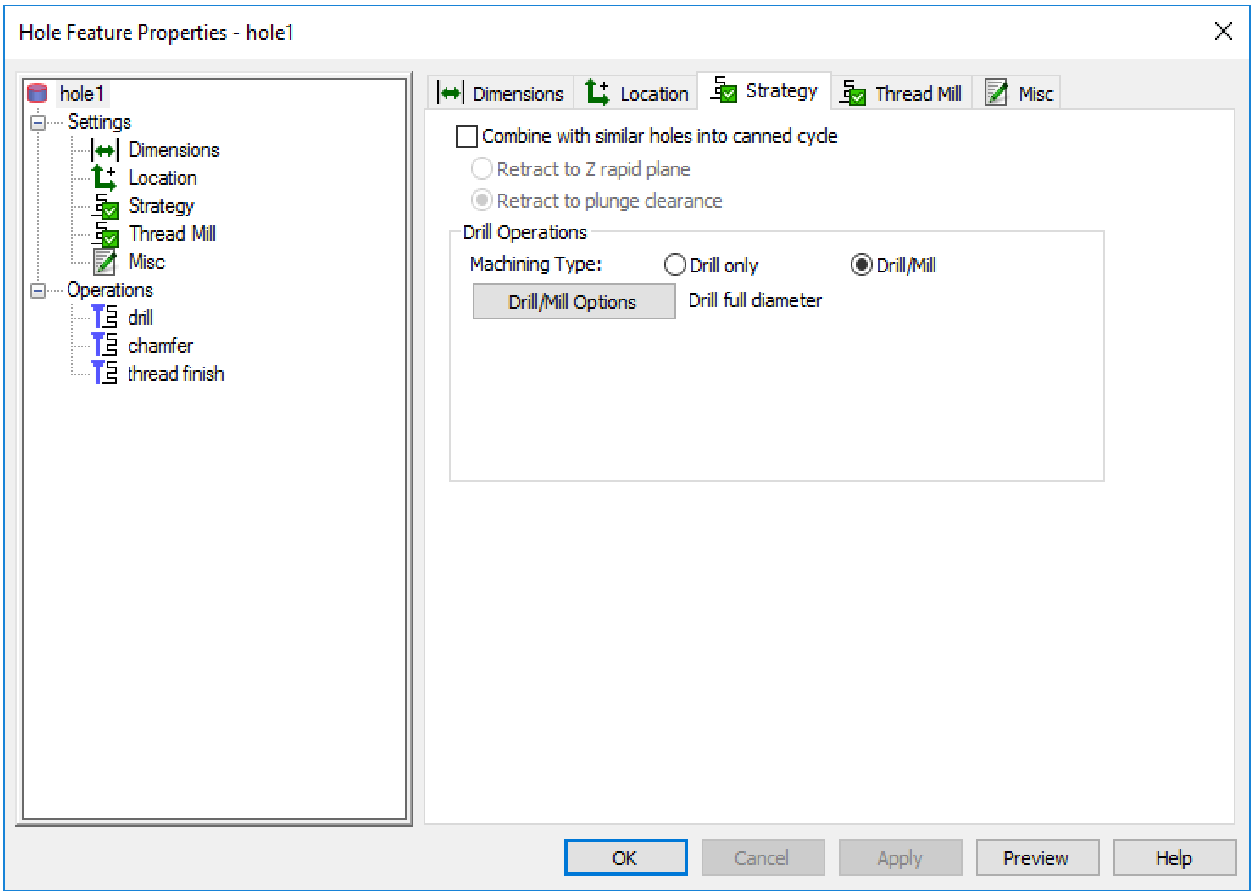




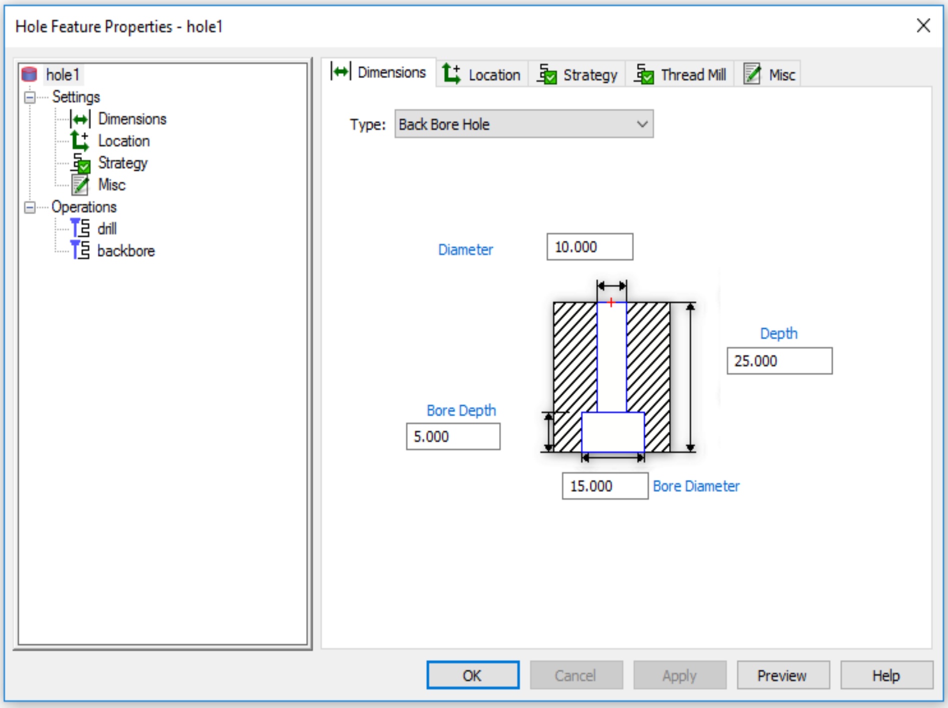
Thread Milled Holes

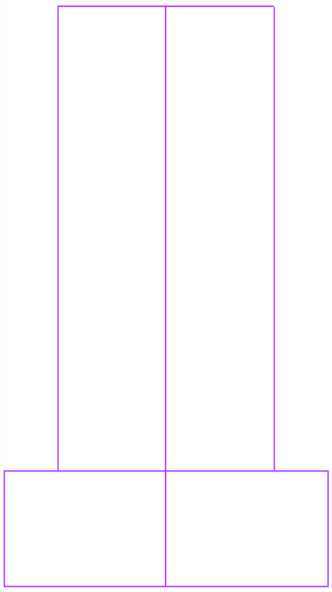






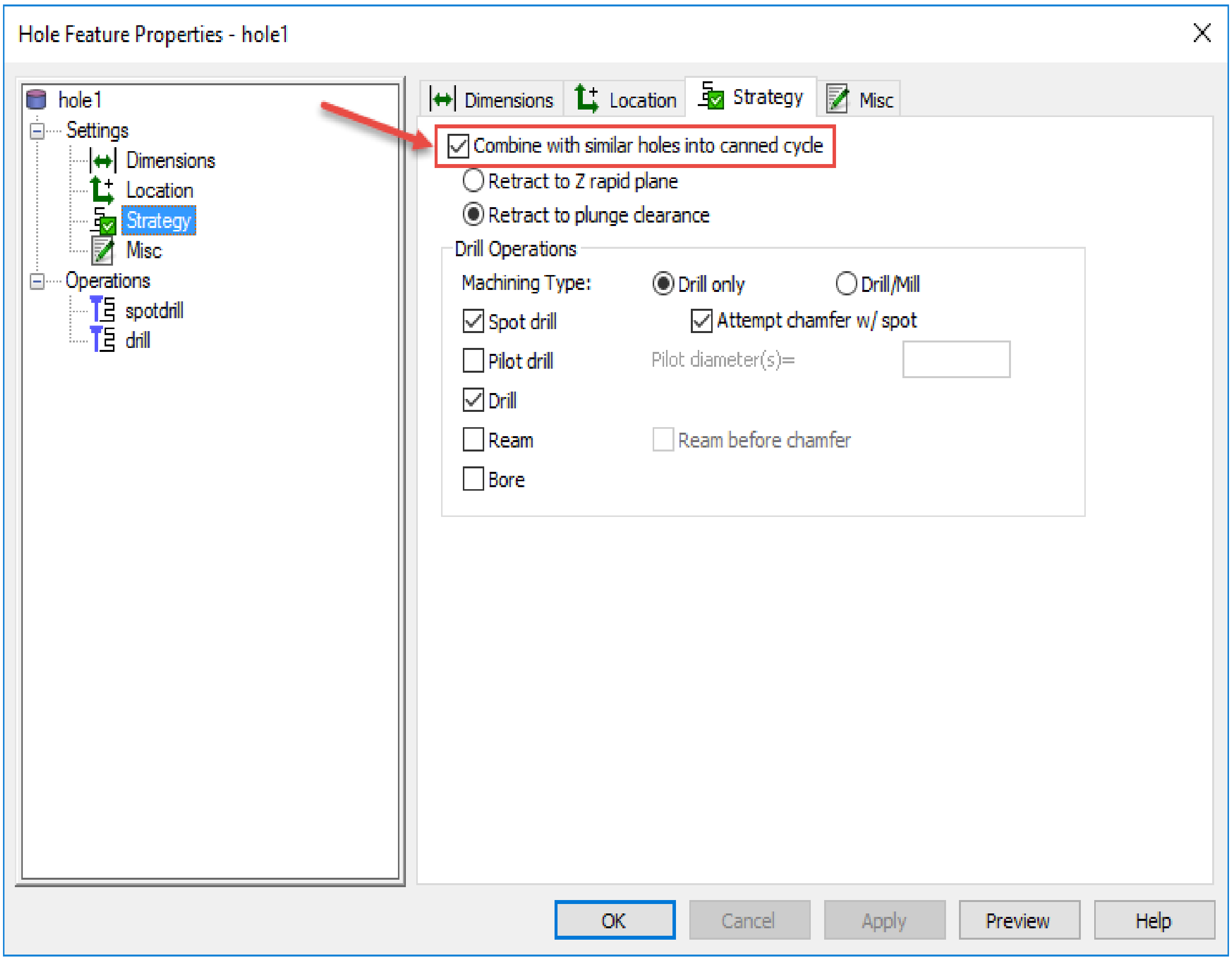
Back Bore Hole (Information Only)





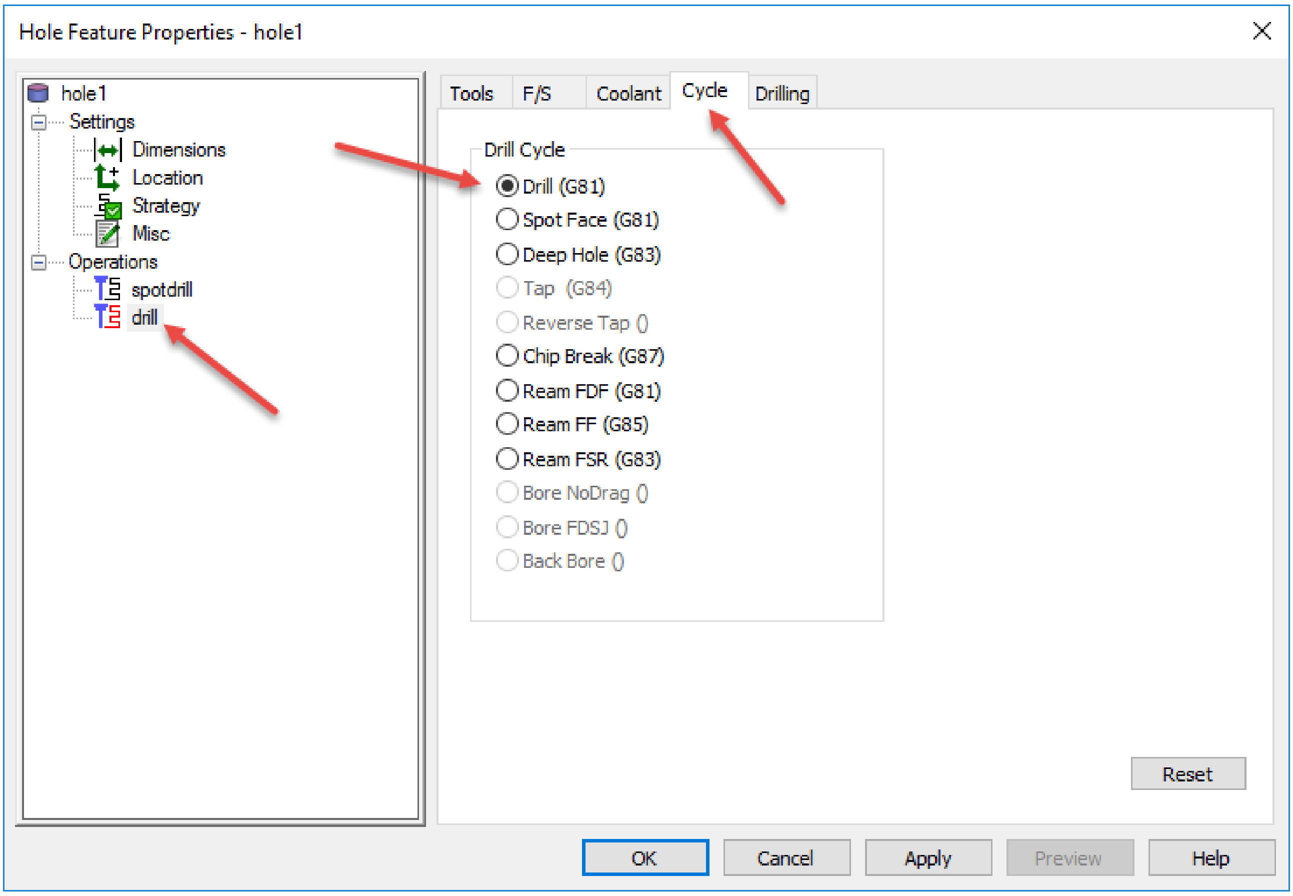
Combine similar holes into canned cycle

*By selecting the **Strategy** Tab you have the option to select **Combine similar holes into canned cycles** which apply to drilling operations.*



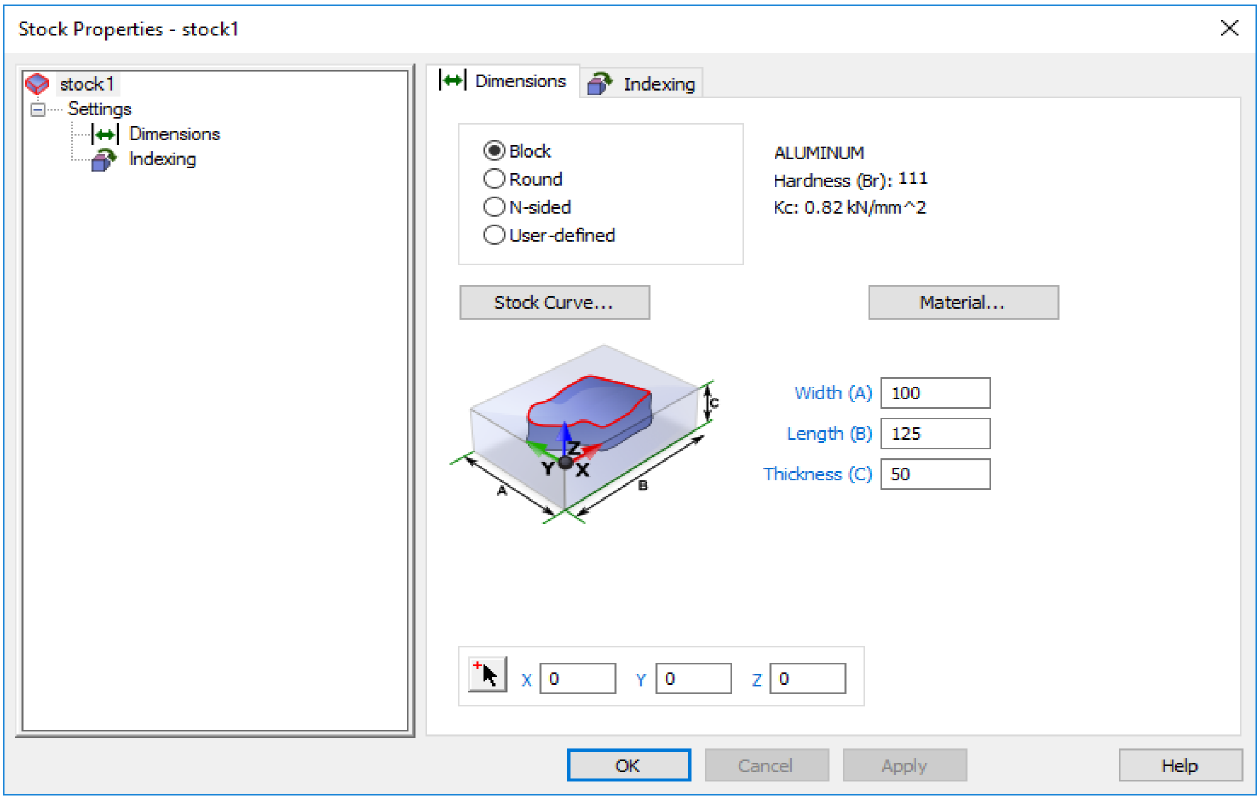
- By default **G83** is selected when drilling holes. This is a safety feature. If you want to change the **Cycle** to **G81** select the **Hole** feature in **PartView.** Expand the Feature tree by selecting **+** next to **Hole**. Select **Drill** then in the Menu select **Cycle** and change the **Drill Cycle** to **G81**. (See image below)

Change Canned Cycle



Rectangular Pocket (Class Exercise)

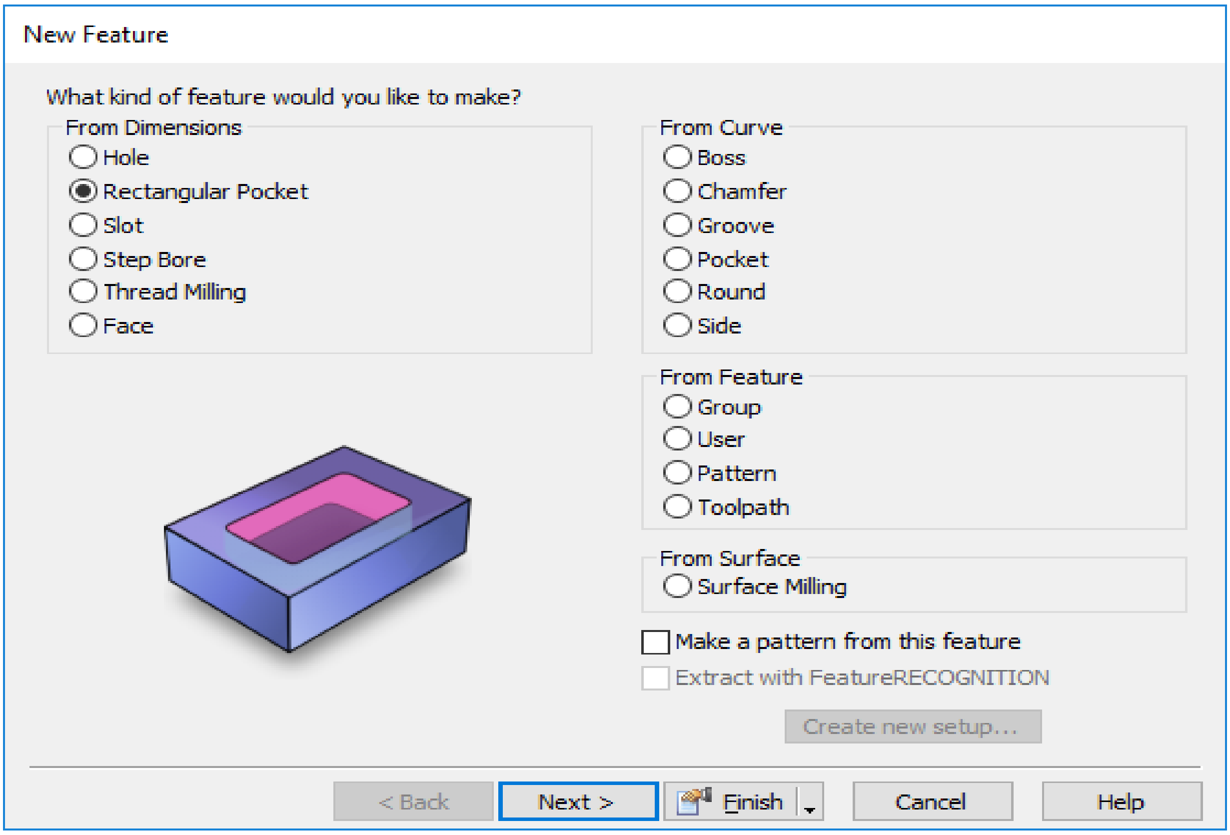
1. Create a **New Part Document** using the **Wizard.**
2. Select **Millimeters** and Select **Finish**.
3. A stock dimensions form will appear, enter the dimensions of the **Stock**.
4. **Width (A) 100**
5. **Length (B) 125**
6. Thickness (C) 50
7. After entering your dimensions select **Apply** and **OK** to close the form.



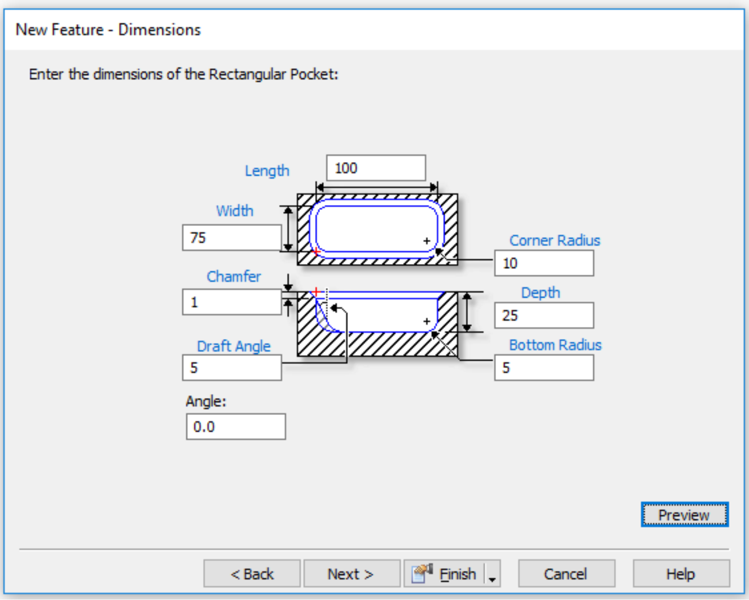
8 Select **OK** and then **Apply**.

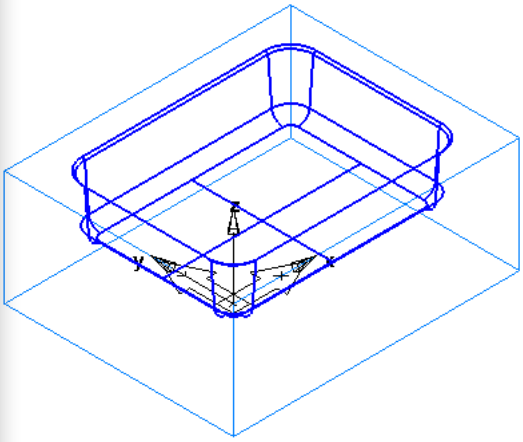
9 Select **Ctrl + 1** or Right click on the view screen and Select **Isometric**.

10 Select **Ctrl + R** or select **Features** from **Steps**. Select **Rectangular Pocket** This machines a Rectangular Pocket with rounded corners. No Curve is needed for this Rectangular Pocket. Select **Next**.



11 Select **Next.**





12 Enter the **Length=100**

13 Enter the **Width=75**

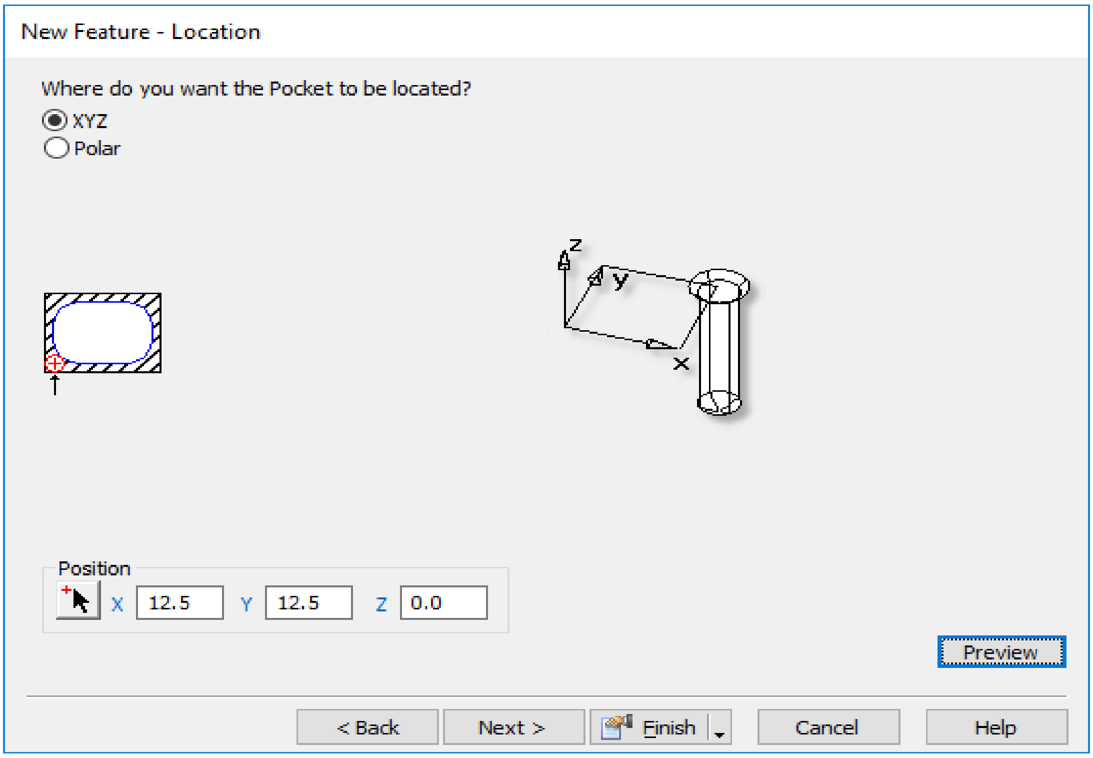
14 Enter the **Depth=25**, the distance cut into the **Material**.

15 **Rectangular Pockets** have a **Corner Radius** that defines the four corners of the **Pocket**. Enter a value of **10mm** for the **Corner Radius**.

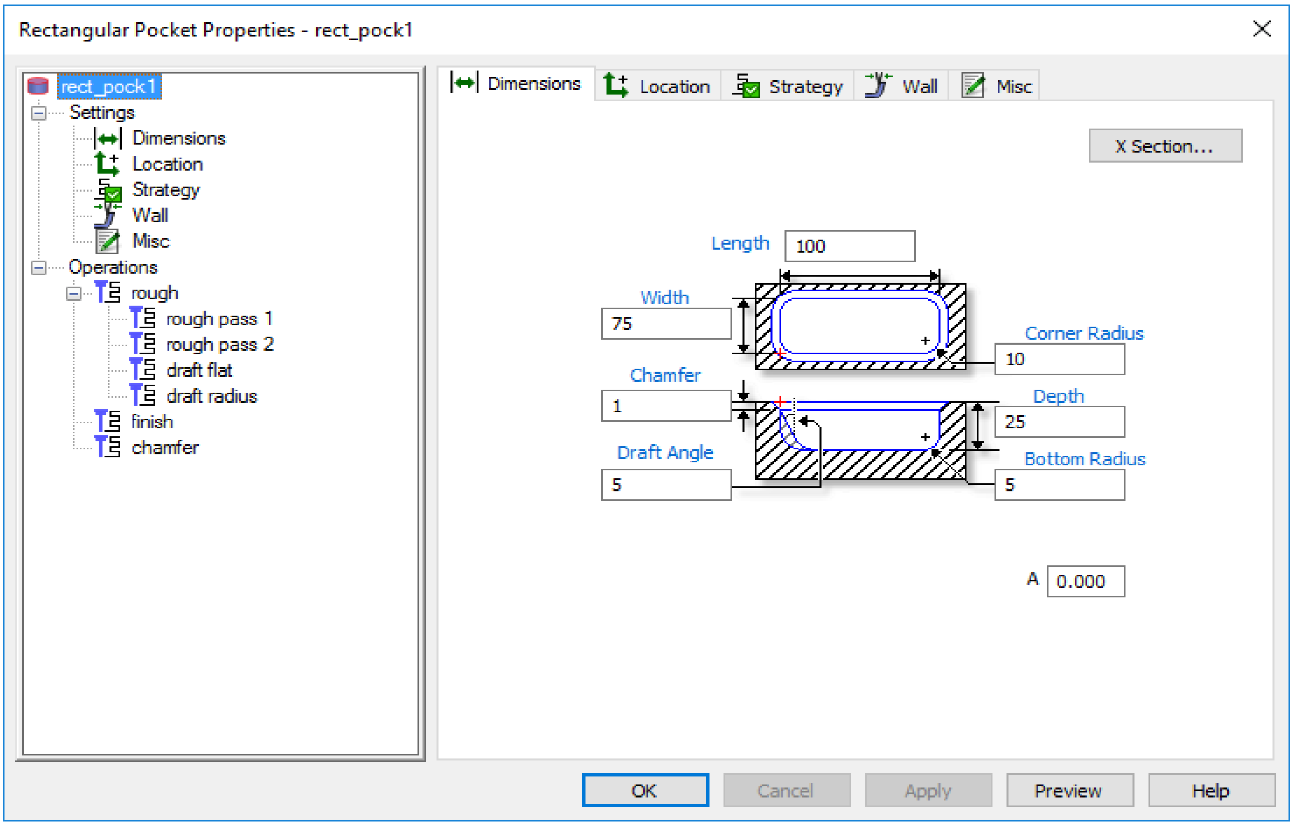
16 **Set** the **Chamfer** to **1mm**. Default angle is **45 degrees** for the **Chamfer** cut at the top edge of the **Feature**.

17 Enter a **Bottom Radius** of **5mm** Enter a **Draft Angle** of **5 Degrees**.

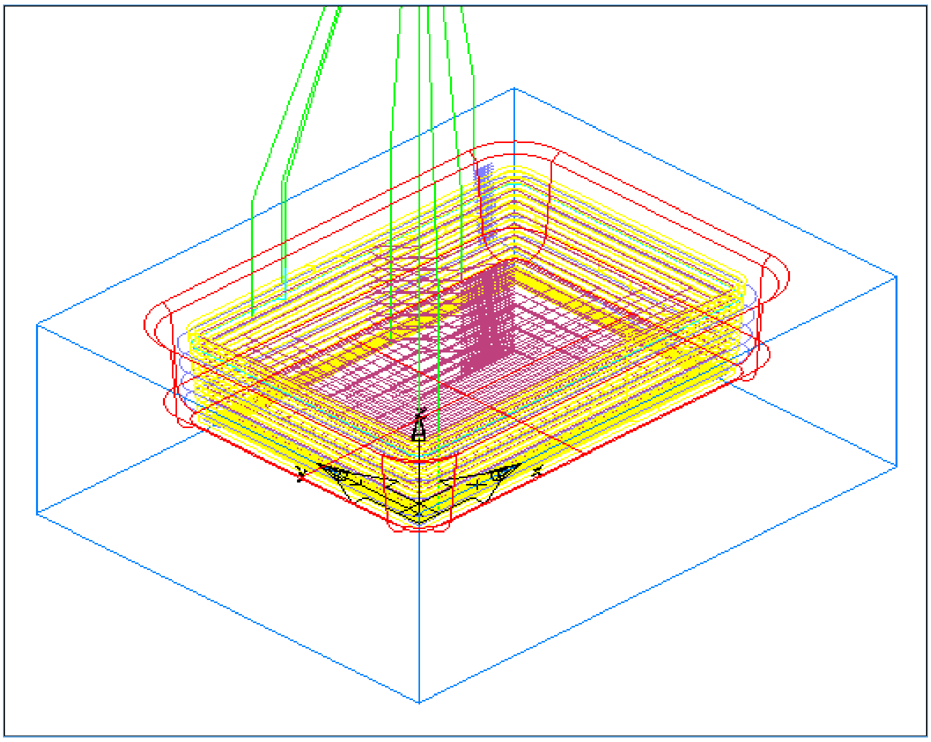
18 Select **Next.** We need to position the pocket central to the **Stock. A location form will appear. Enter X12.5mm Y12.5mm Z0**



19 Select **Finish.**



Run a **Centreline Simulation**.



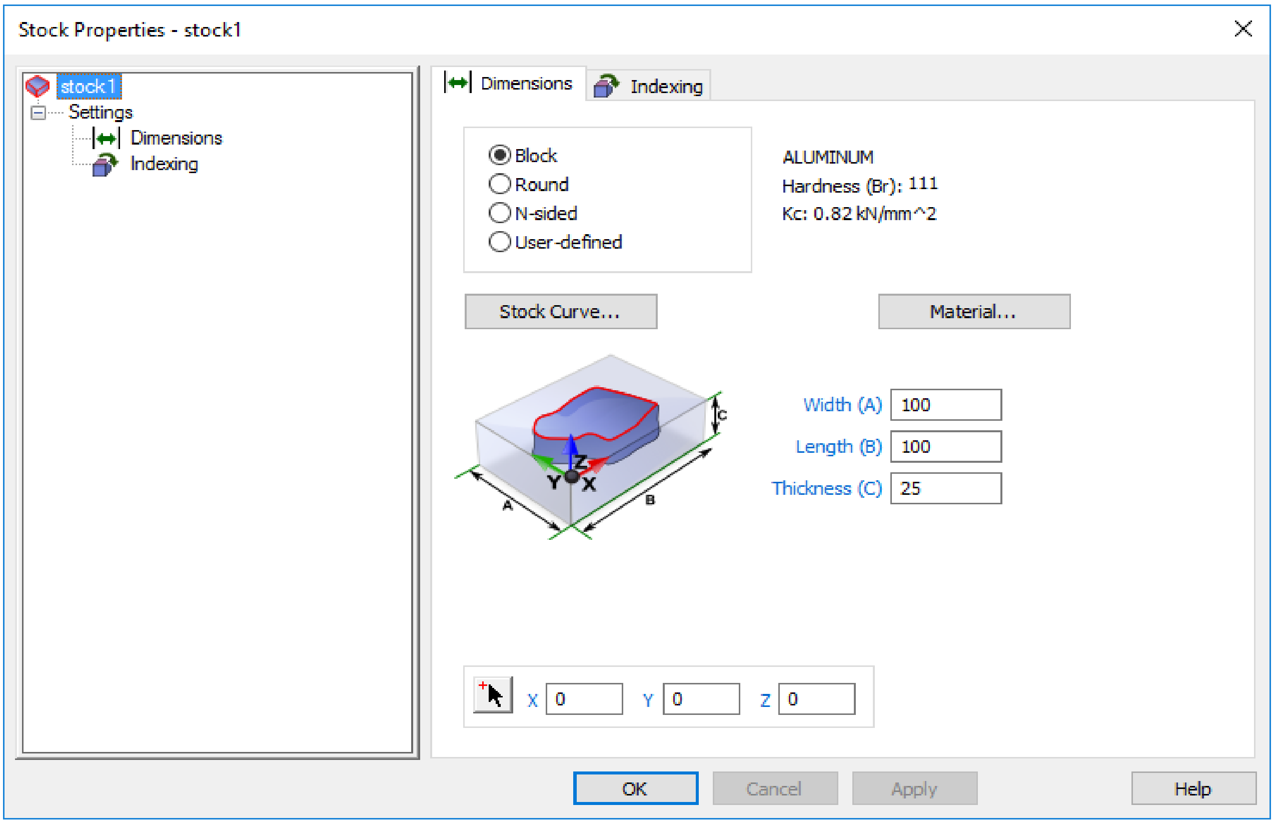
20 **Save** the file in your instructors chosen location.

Slot (Information Only)

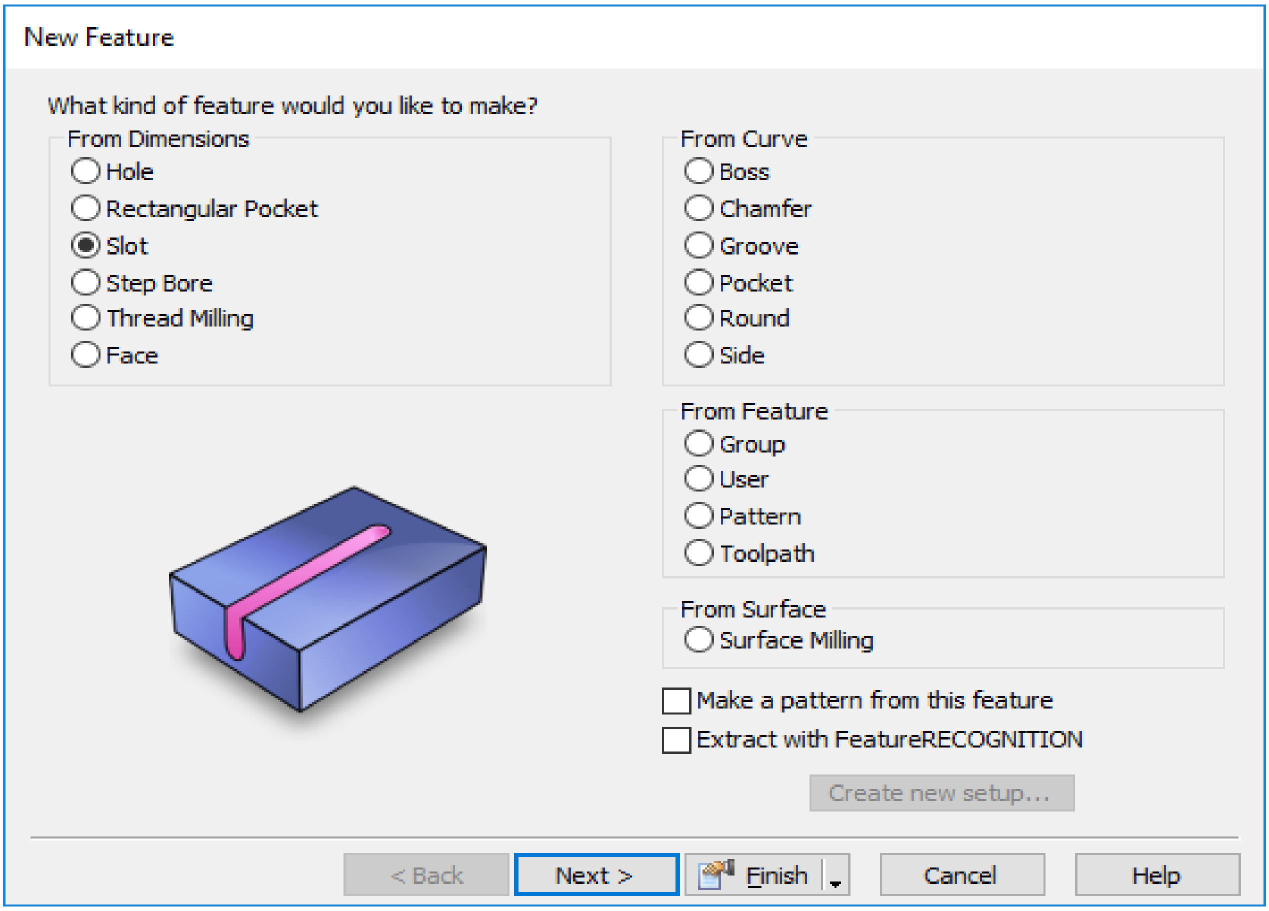
***Slot** is a straight slot with rounded ends. No **Curve** is needed for a **Slot**.*

1 Open a **New Part Document**.

2 Create a new block Stock with Properties of **100mm x 100mm x 25mm.**



3 Select **Ctrl + R** or select **Features** from **Steps**. Select **Slot from Dimensions**.



*The **Simple** option simplifies the manufacturing strategy for the **Slot**. If it is selected, the slot is manufactured by making a single pass down the centre of the **Slot** with a tool whose radius is equal to the width of the slot.*

4 Enter the slot's **Length**. The width of a slot does not have to match the diameter of a standard available Endmill, unless you are making a **Simple Slot**. If an exact match cannot be found, then a smaller tool is selected and multiple horizontal passes are performed.

5 **Enter the** slot's **Width**. Enter the **Depth**, the distance cut into the material.

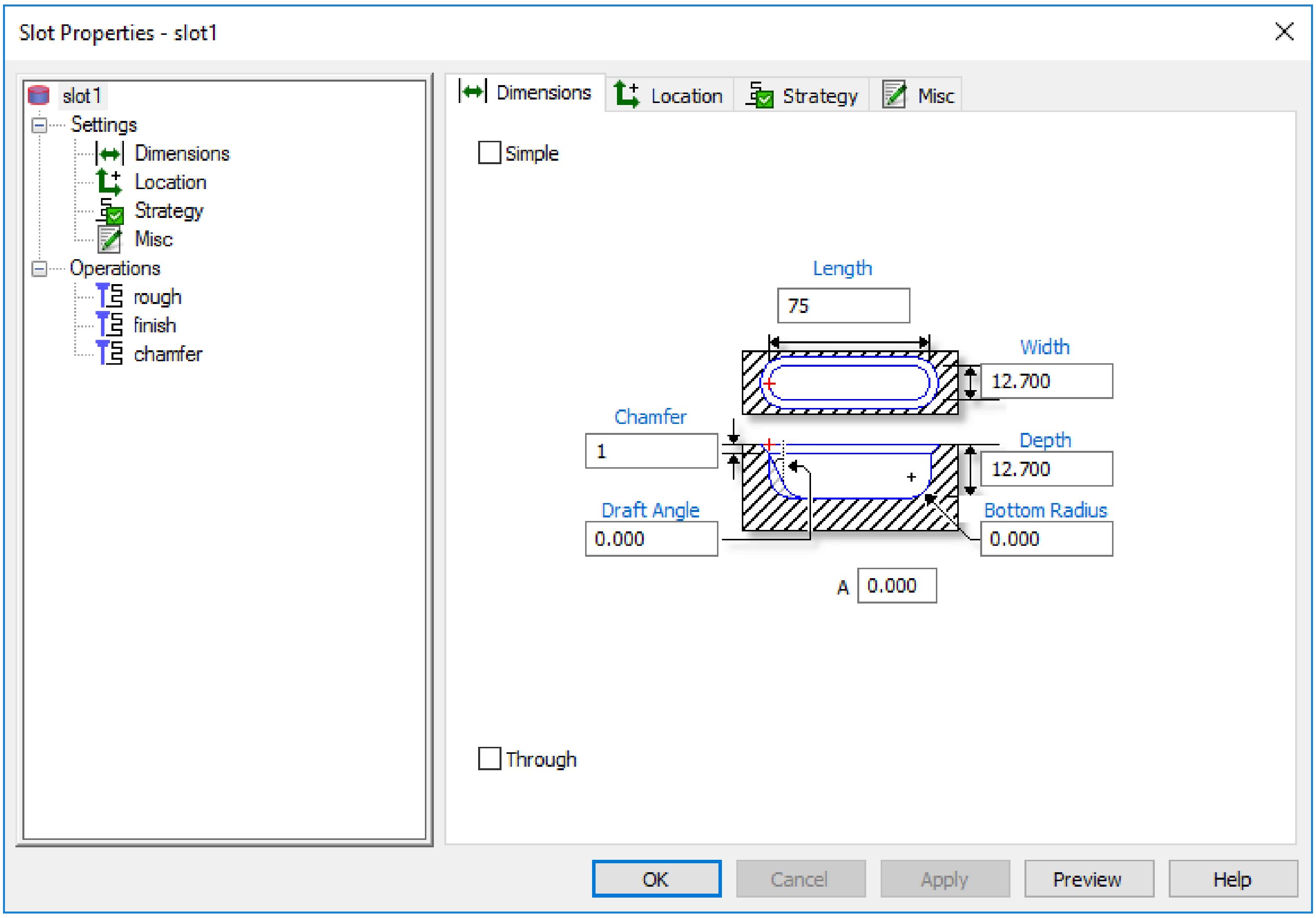
6 Chamfer sets the depth of a 45 Chamfer cut at the top edge of the Feature. Leave this value at 0, the default, for No Chamfer.

7 Optionally enter a Bottom Radius. Optionally enter a Draft Angle.

8 Select Next. Enter the following dimensions into your menu. Untick **Simple. Length 75mm Width 12.7mm Chamfer 1mm Depth 12.7mm Bottom Radius 1mm.** See image below.

9 Select **Next**. Then enter Slot location details as **X12.5mm, Y50mm.**

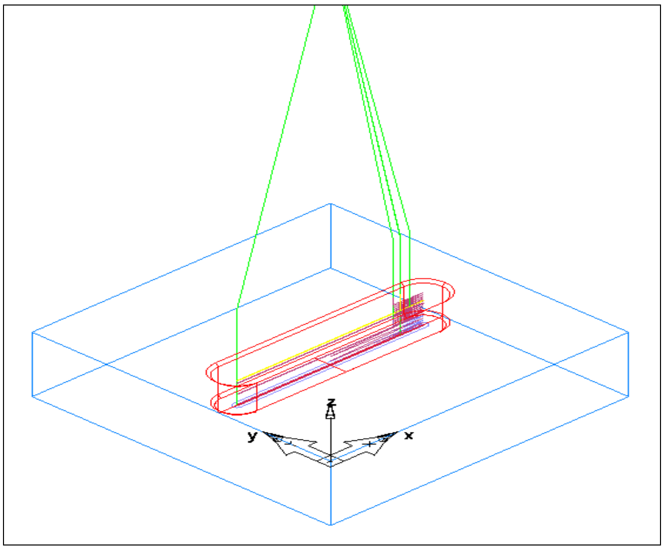
10 Select **Finish** and **OK** to close the form**.**



Please select **Show Centreline Simulation**.

11 **Save** the file in your instructors chosen location.

See image below.



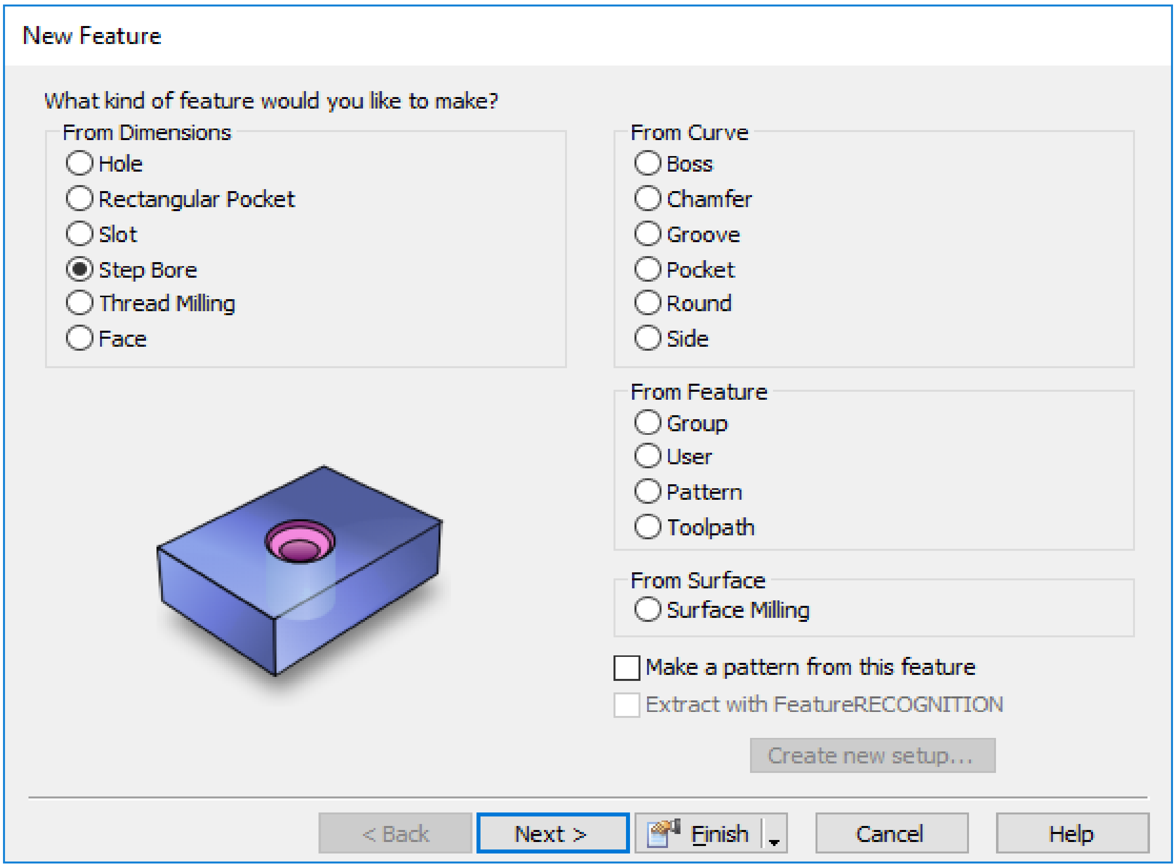
Step Bore (Optional Exercise)

1 Open a **New Part Document**. **Millimeters.**

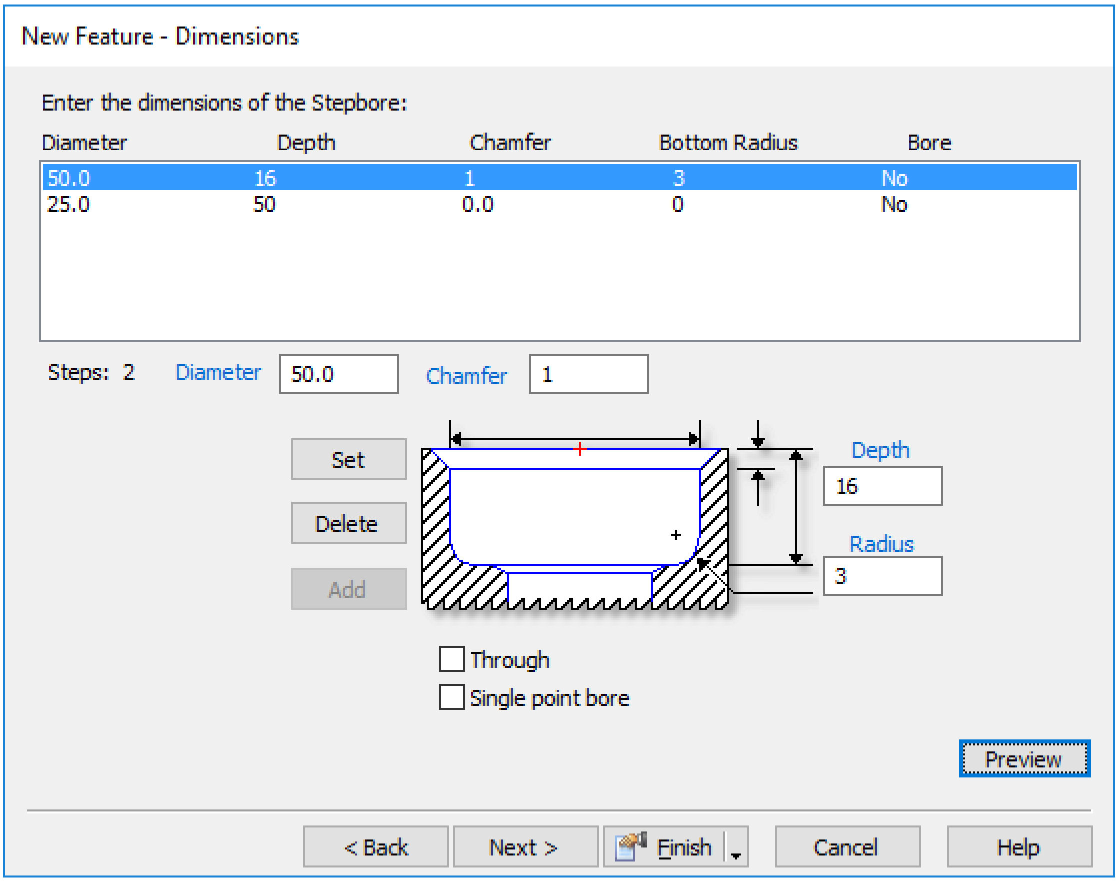
2 Create your Stock Material size to **X100mm Y100mm** Depth **50mm.**

*A **Step Bore** is a series of Round Pockets with a common Centre. No Curve is needed for a Step Bore.*

3 Select **Ctrl + R** or select **Features** from **Steps** and select **Step Bore.**

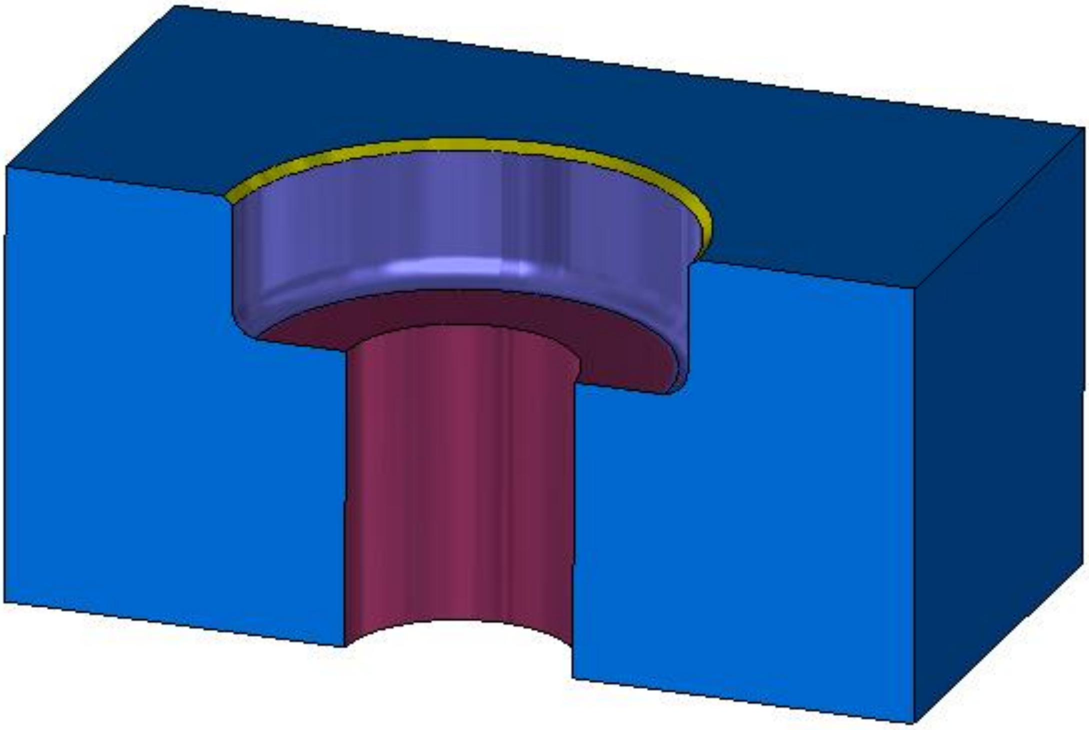


4 Select **Next** and then enter the following **Metric** values as shown on the next page. Then select **Finish.**



5 Select **Next** and then enter the location information **X50mm, Y50mm.**

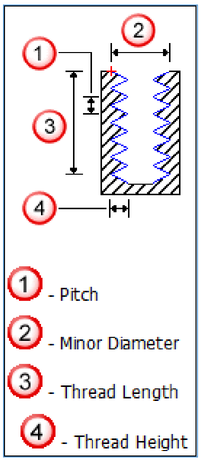
The image shows a cross section of the part.



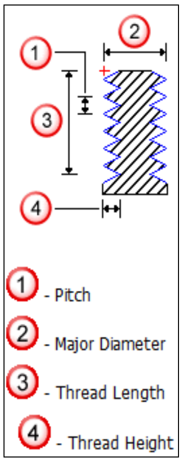
6 **Run** the **3D Simulation** to see your finished part.

7 **Save** the file in your instructors chosen location.

Thread Milling (Information Only)

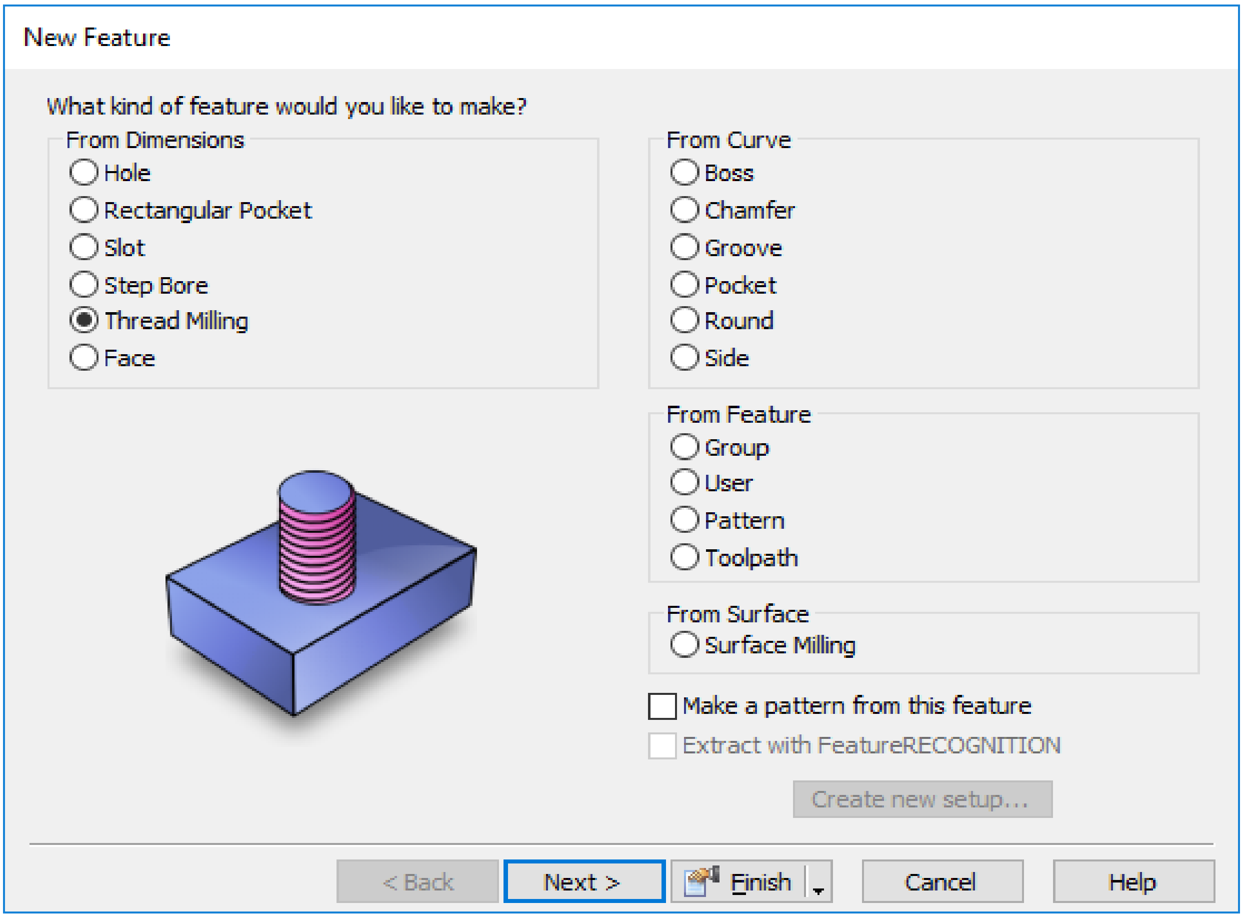


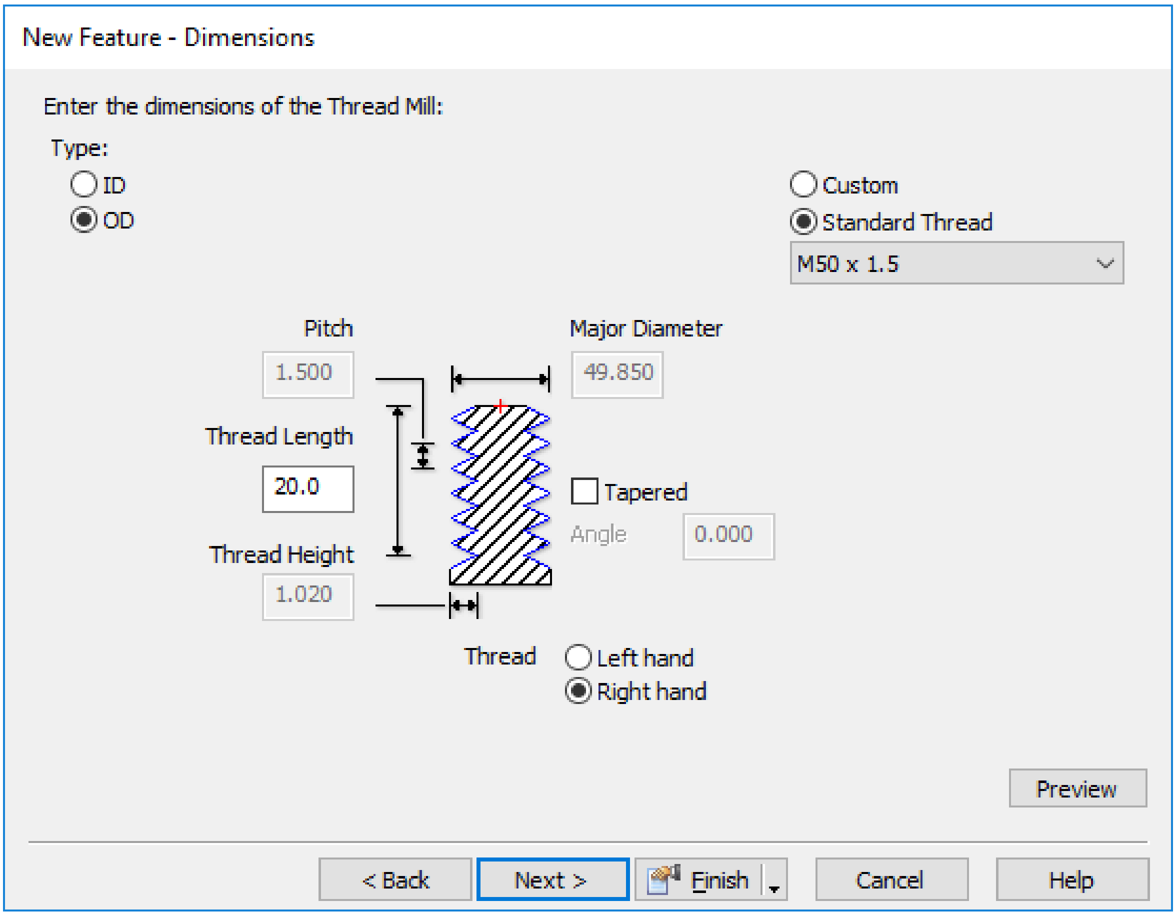
ID Thread



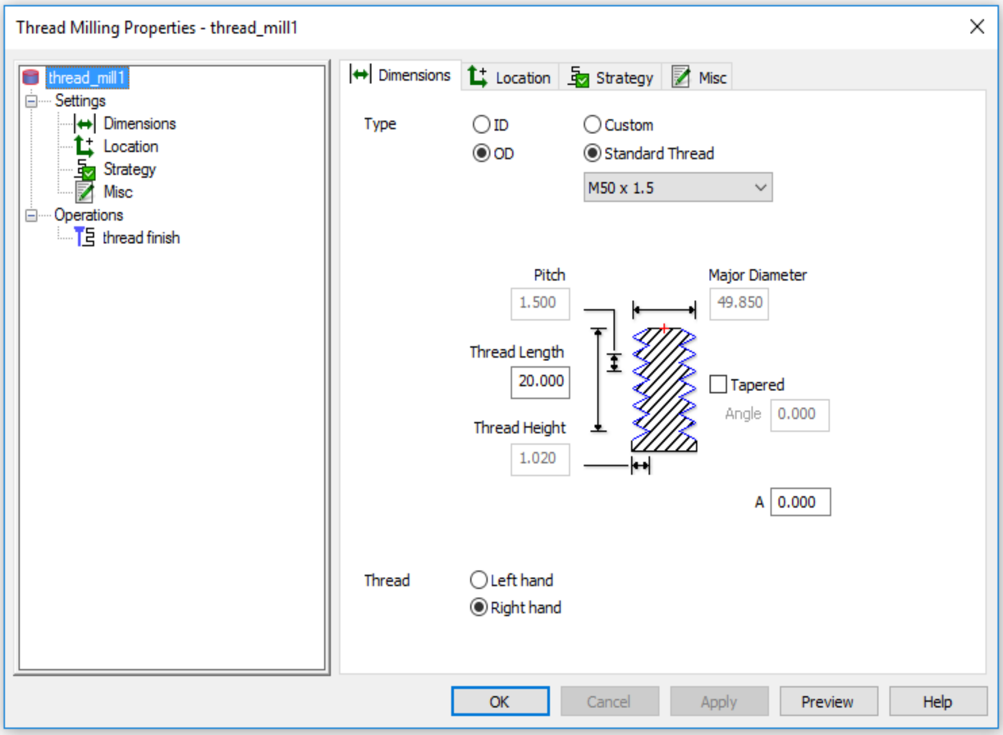
OD Thread

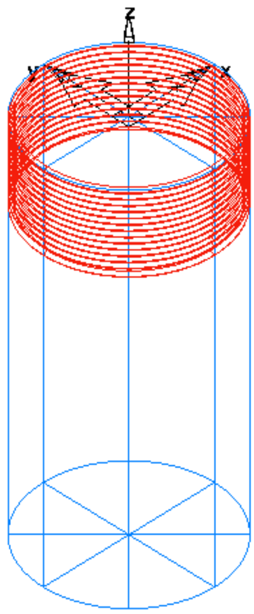
- **FeatureCAM** follows this process to create a single **Thread Milling Feature**:
- An appropriate tool is selected. The tool selected by default and has the same **Pitch** as the **Thread**.
- The **Internal**/**External** classification matches the **Feature**.





Enter the values and then select Finish.





Metric Thread Milling Exercise (Optional Exercise)

1 Open a **New Part Document**. **Millimeters.**

2 Create a **Round Stock Material** of **50mm Diameter, 75mm** long in the Z Axis

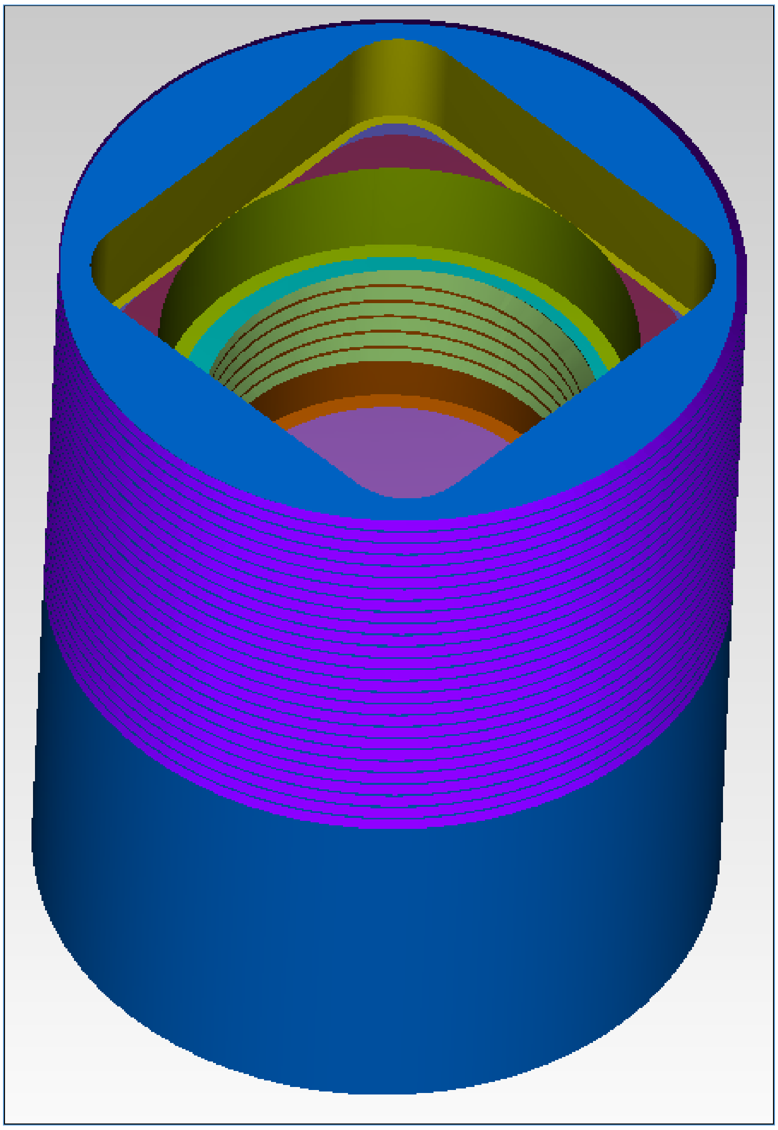
3 Create a **Rectangular Pocket 35mm X 35mm R5mm** Corners. Depth **10mm** Location **X-17.5mm Y-17.5mm Z0**

4 Create a **Step Bore** Top Dia **35mm Depth 10mm** Bottom Dia **28mm Depth 25mm**.

5 Create a **Thread Milling Feature 50mm** OD Right hand **40mm** Long **1.5mm** Pitch.

6 Create an ID **30mm** Thread **1.5mm** Pitch Depth **20mm** Location **Z-10mm.**

See Example below. Save the file as **Metric Thread Milling Exercise.**



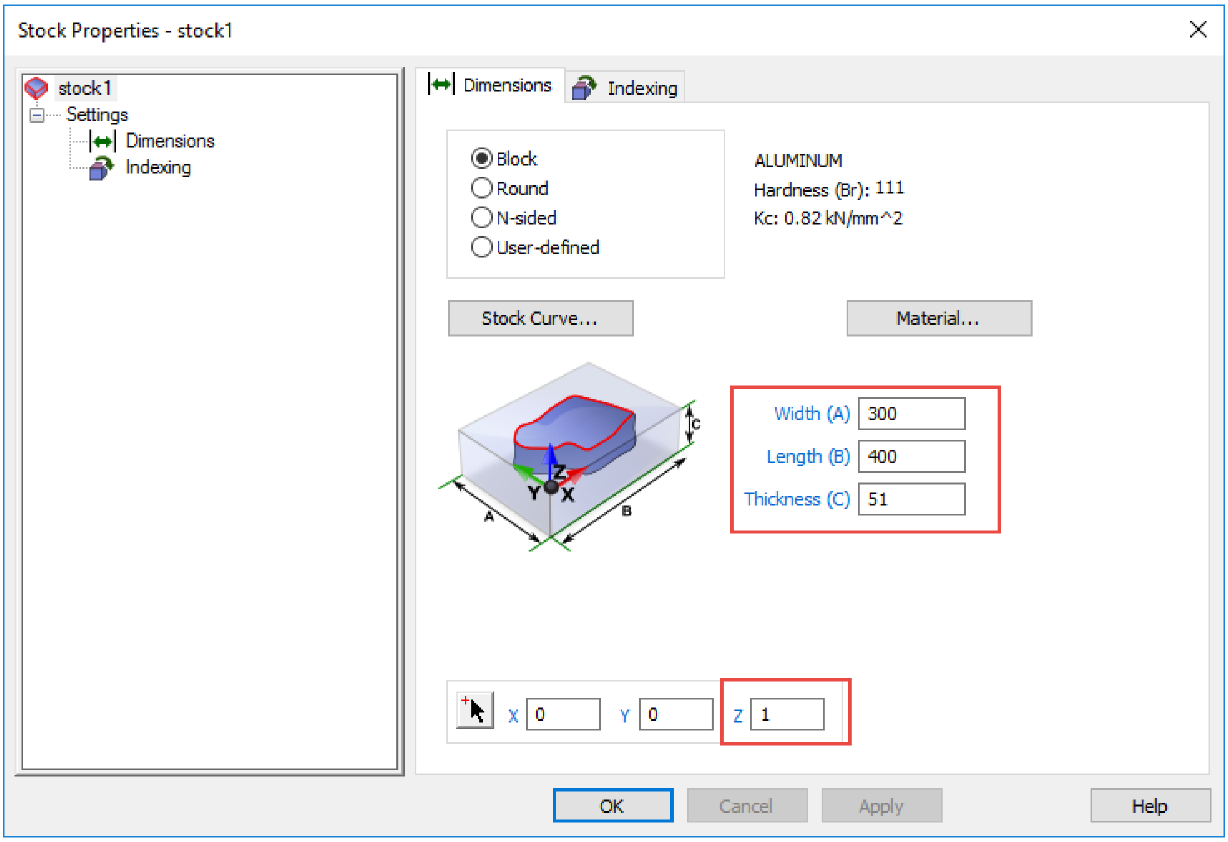
7 Run **3D Simulation.**

8 **Save** the file in your instructors chosen location.

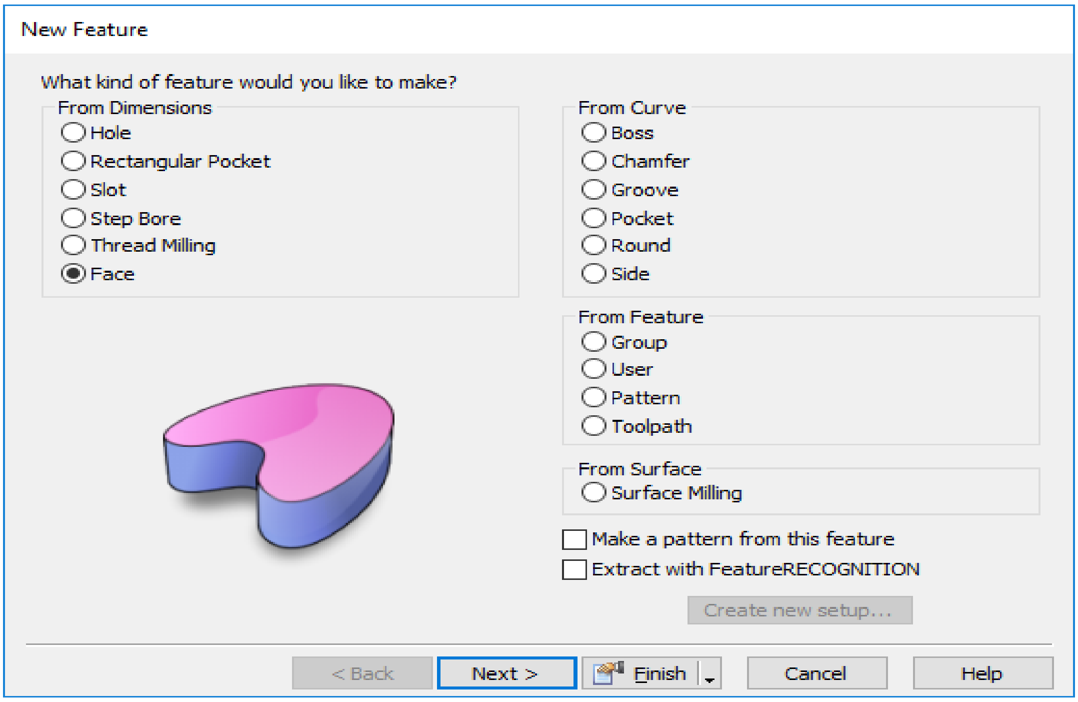
Face Milling (Information Only)

***Face** is a milling operation to machine a face of the stock and to cut the **Stock** to an exact dimension.*

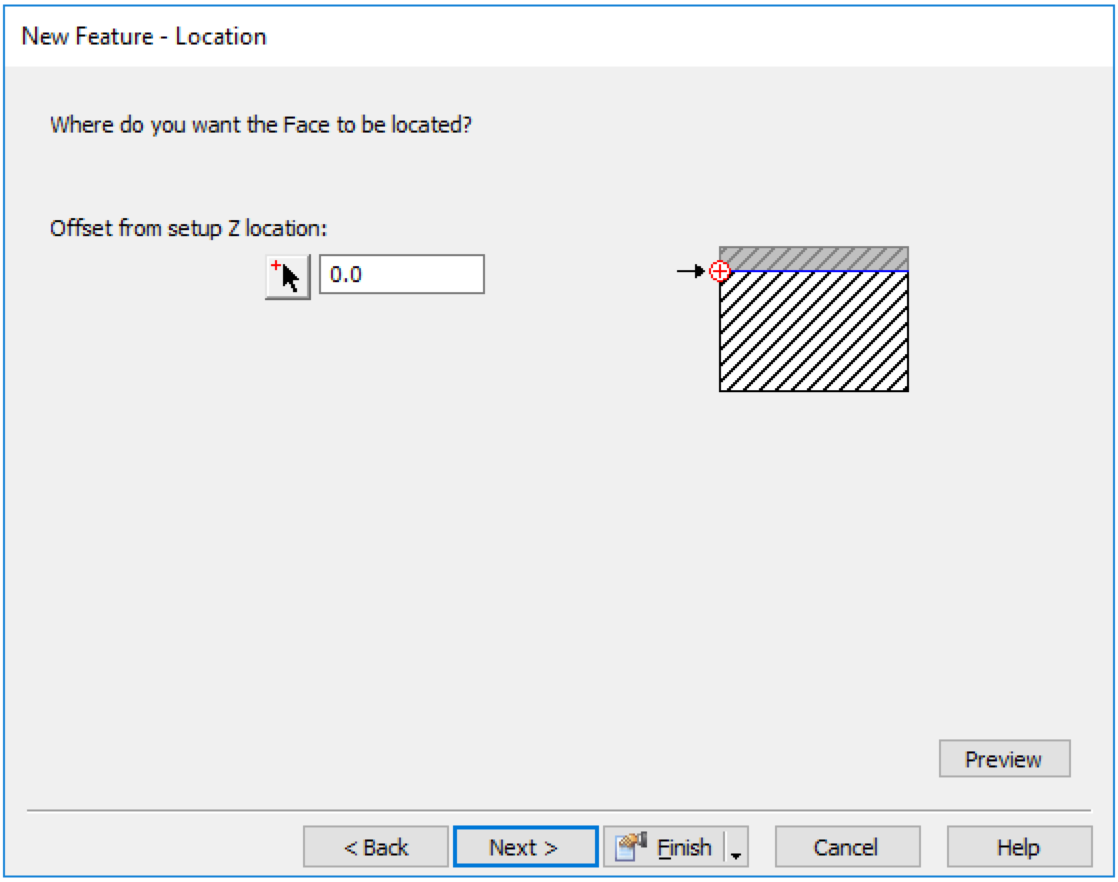
1 Open New Part Document Millimeters**.** Create a new block Stock with Properties of **300mm x 400mm x 51mm.**



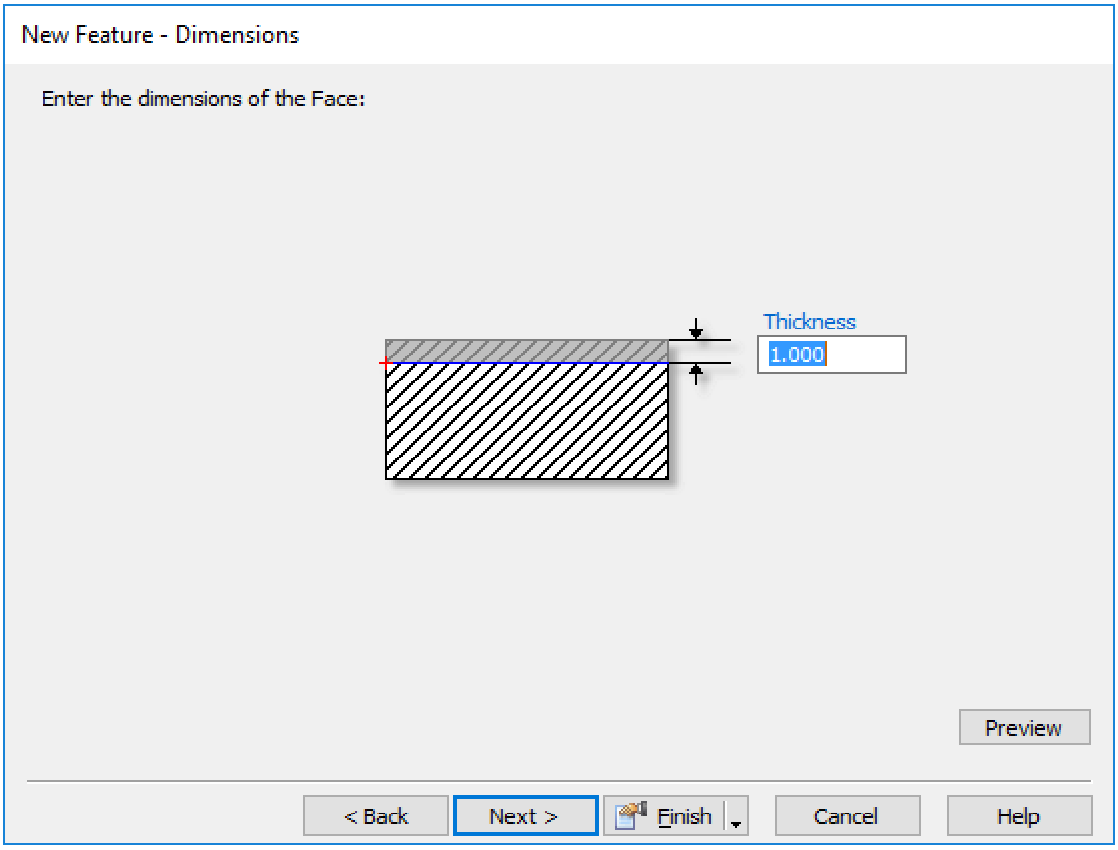
2 Select **Ctrl + R** or **Feature** from **Steps** and select **Face.**



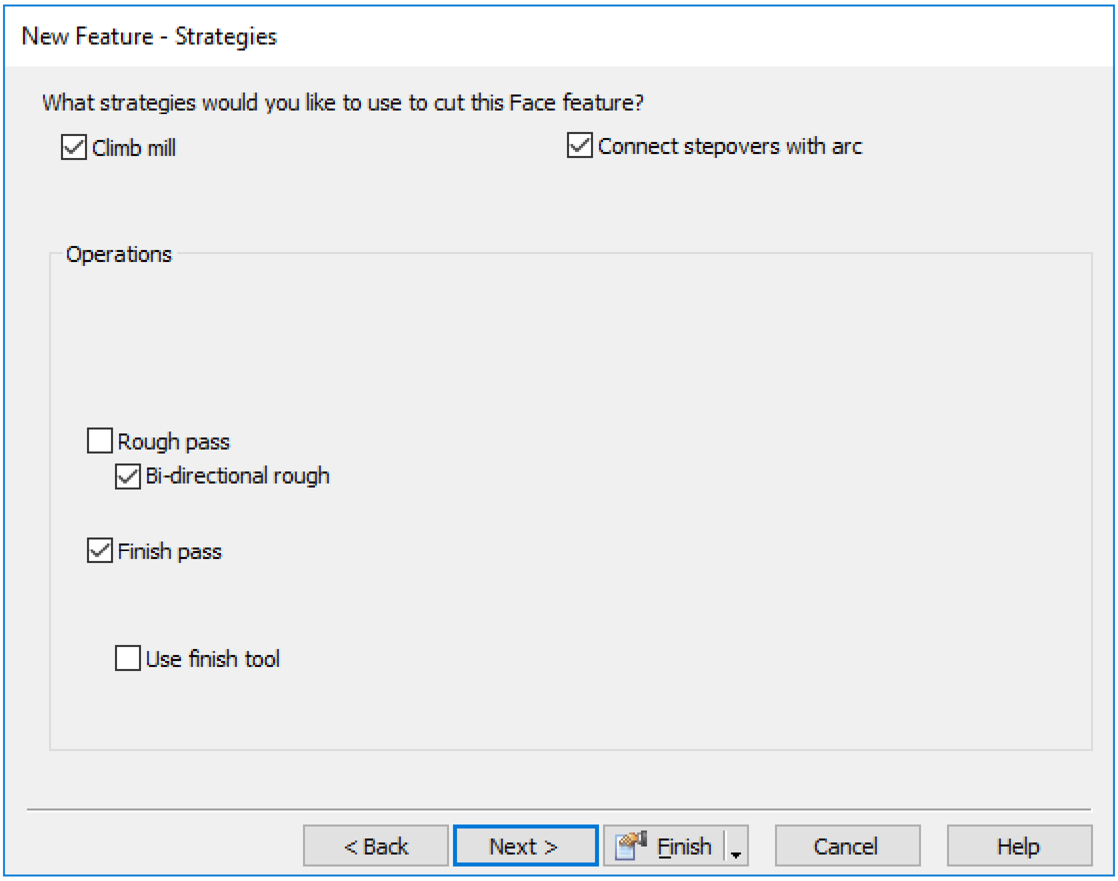
3 Select **Next.**



4 Select **Next**



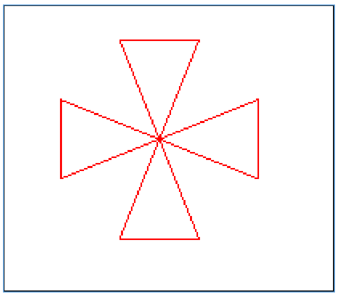
5 Select **Next**



*You will notice that **Connect Stepovers with arc** has been ticked; this will link the toolpaths with an arc producing a smoother transition at high speed between cuts.*

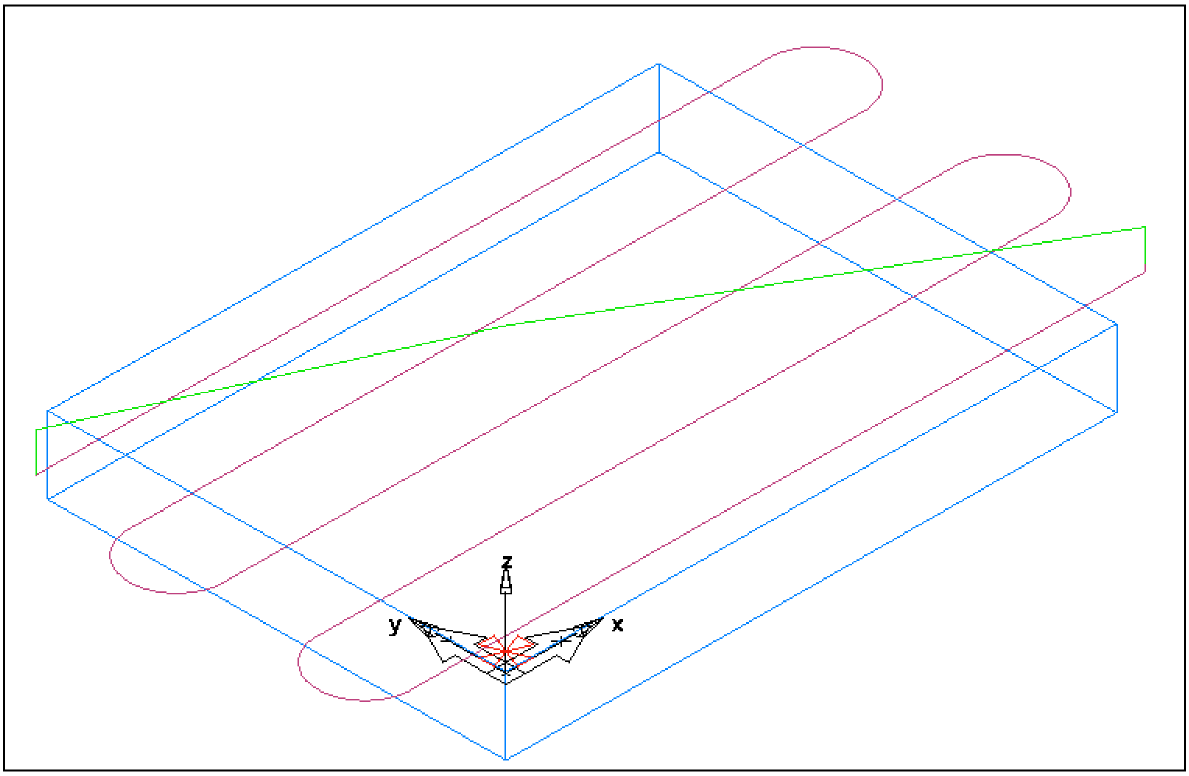
*When you are using **Face** the operation will be re-positioned to the start of the machining in the **Operations list** as this has high priority.*

*You will notice the **Face** feature which denotes that the top is now finished to size.*

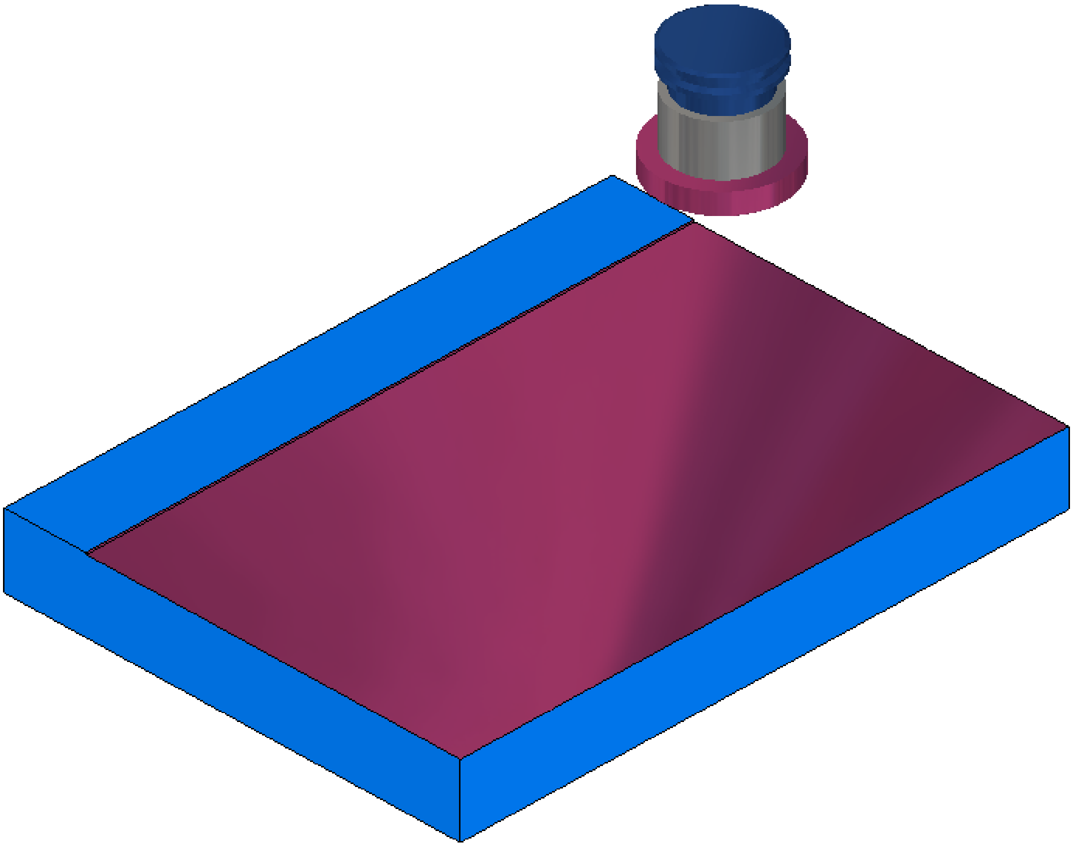


Face Milling shown as 2D toolpath.

Please note we are using a 80mm Facemill instead of the default 32mm

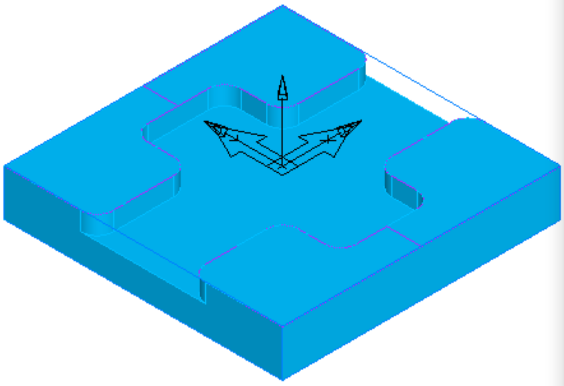


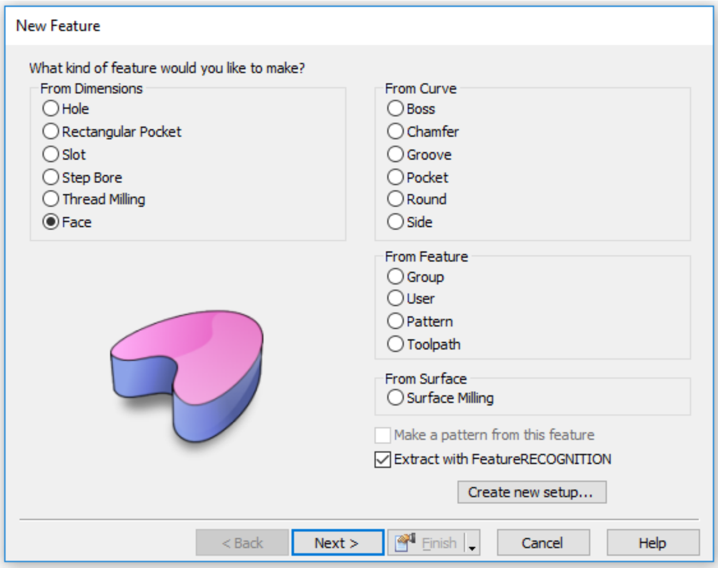
Face Milling shown in ***3D Simulation***.



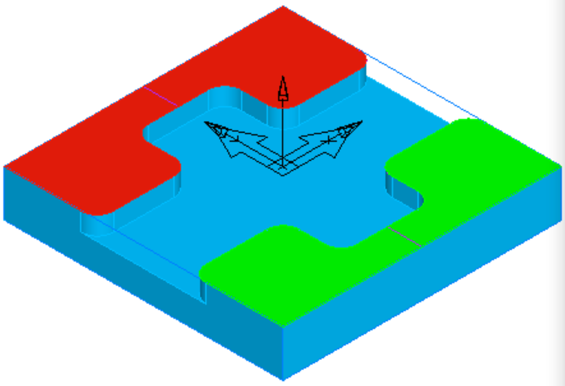
Face milling a multiface feature (Information Only)

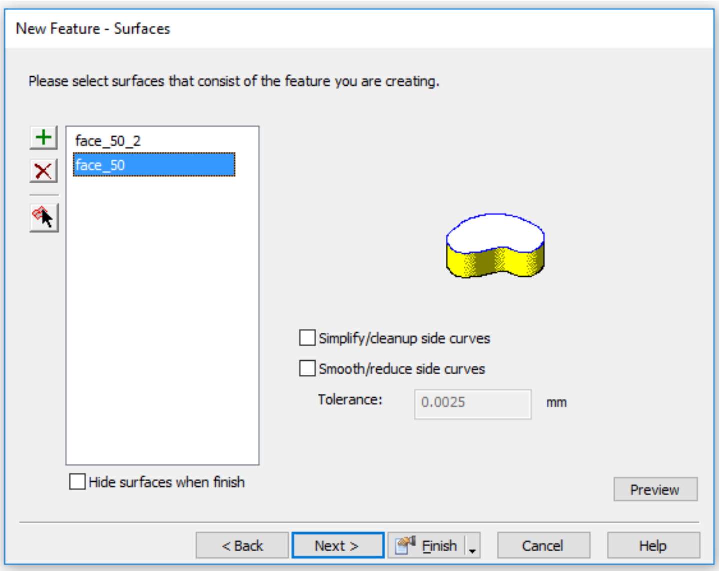
New to FeatureCAM 2016, we now have the ability to machine a multiface feature. Make sure you select Extract with Feature Recognition.



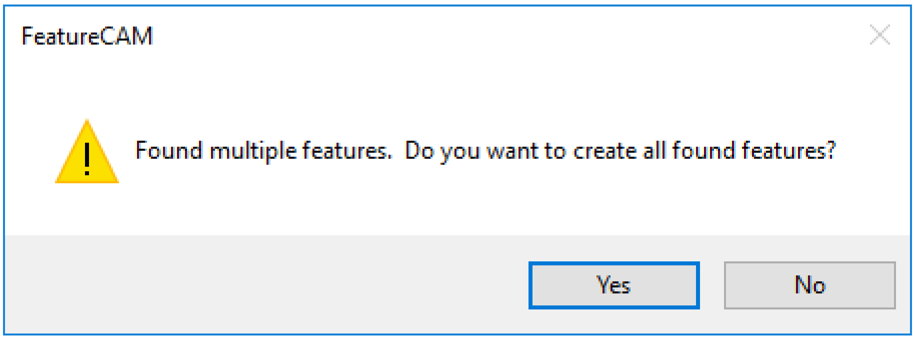


1 Select **Next.** Then select your **face** features.





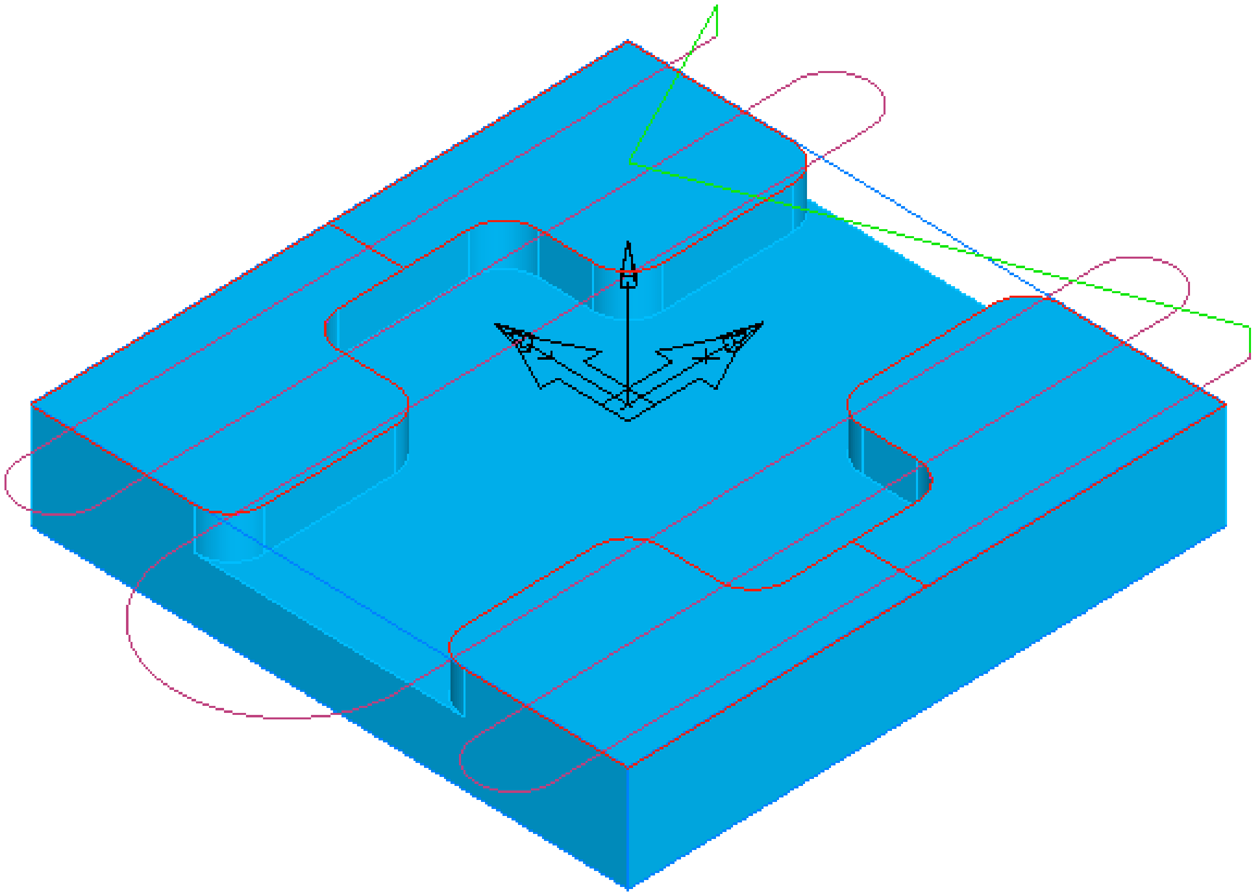
FeatureCAM states that it has found multiple features.



2 Select **yes** to create all found features. Select **Finish**.

See the results in the image below.

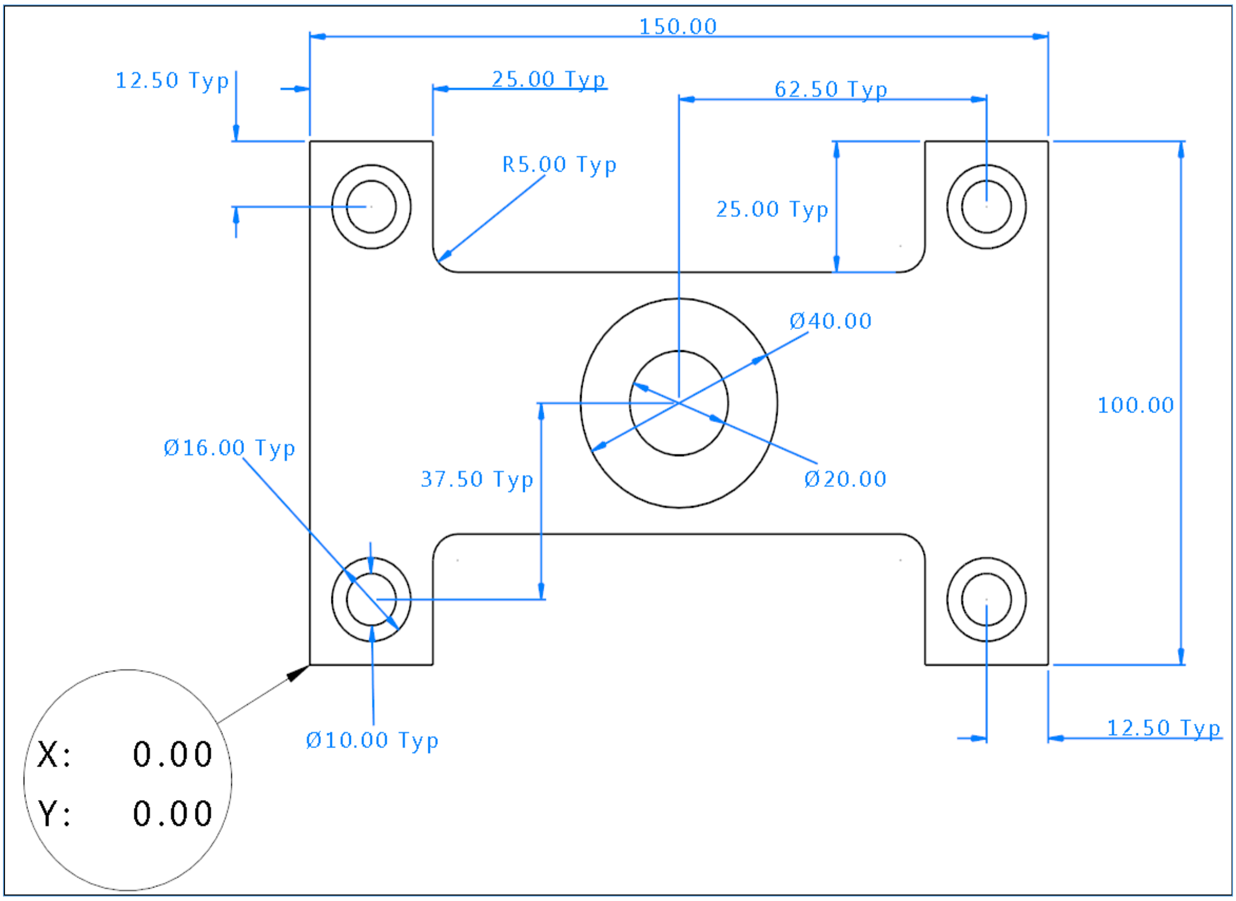
*Please note we have selected a **100mm** Facemill. Note the large toolpath offset moving from face to face.*



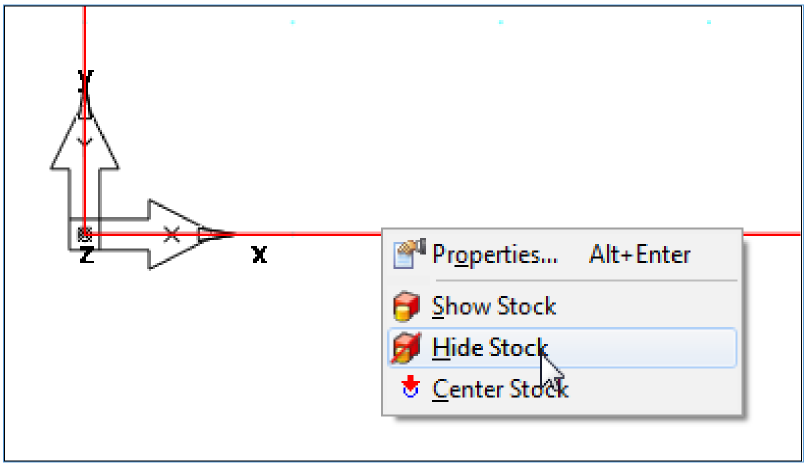
Run a Centreline Simulation.

User Defined Stock Exercise (Class Exercise)

Please note there is always more than one way to create the following example.



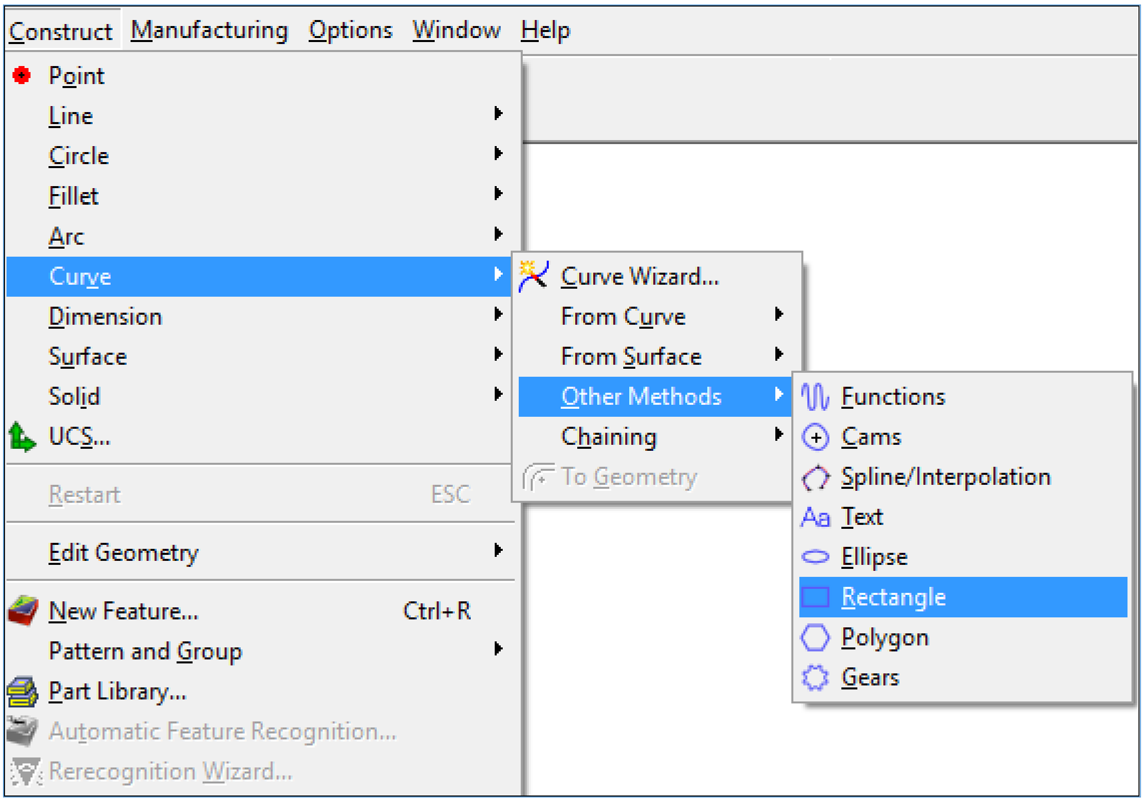
1 Use the **mouse pointer** and select the **Stock line,** use the **Left** mouse button **Click** to select.

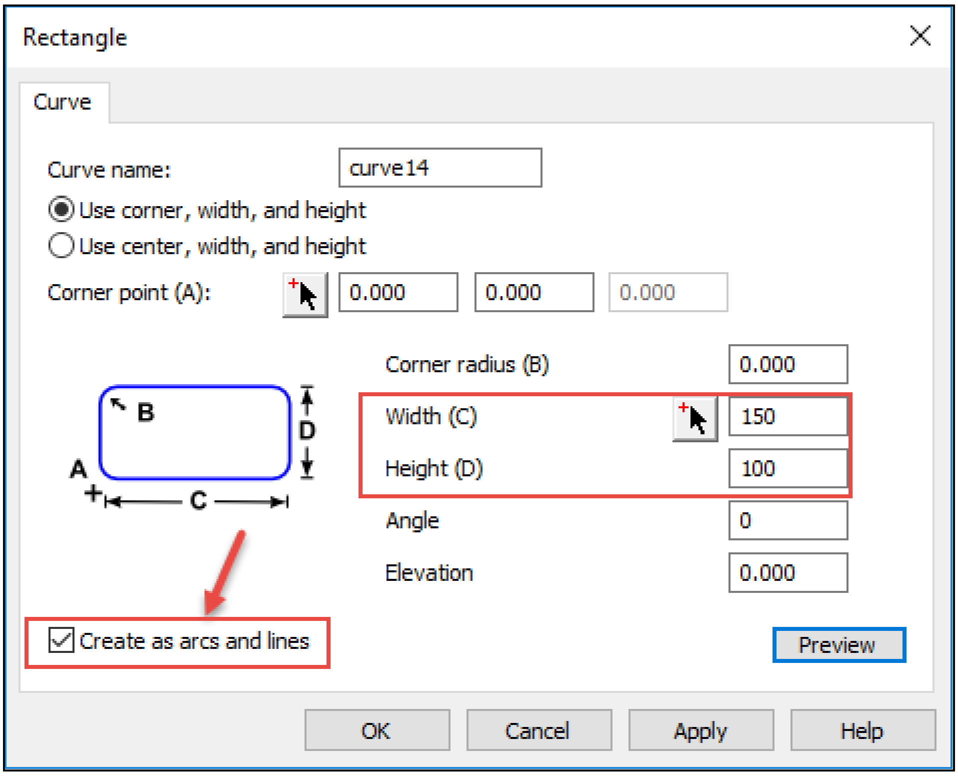


Right click to show the following menu. Then select **Hide Stock.**

Alternatively you can select **Ctrl + J** this will **Hide All** if shortcuts have been configured on your computer.

2 Go to **Construct>Curve>Other Methods>Rectangle.**





*Use **Top View** or **Ctrl+5** to orientate to the **XY** plane.*

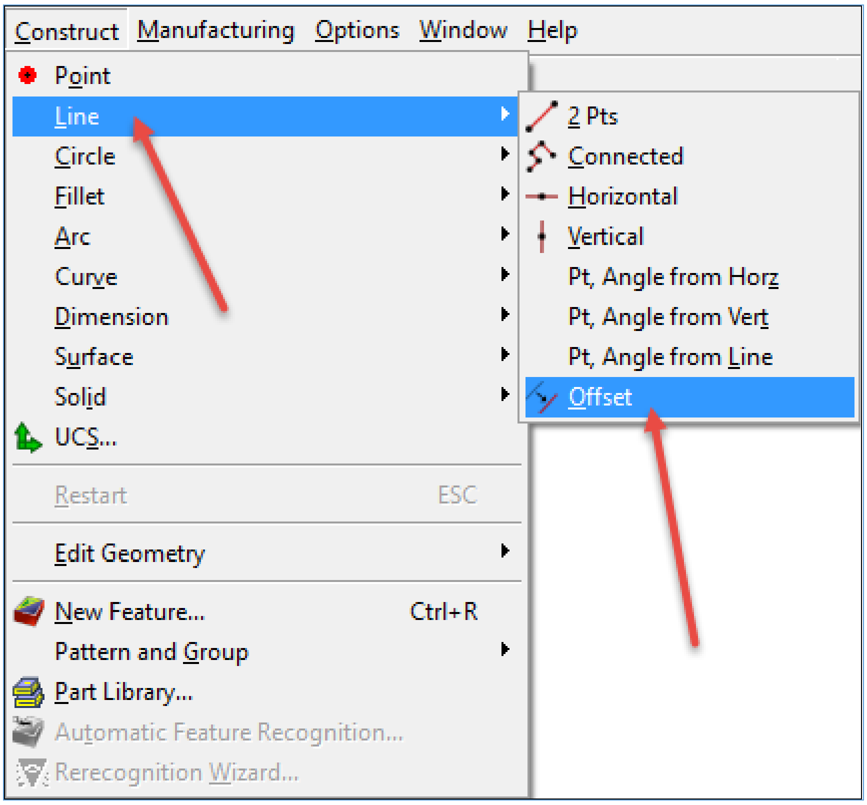
3 Enter the following Co-ordinates **Width 150mm Height 100mm**

4 Remember to select create as arcs and lines.

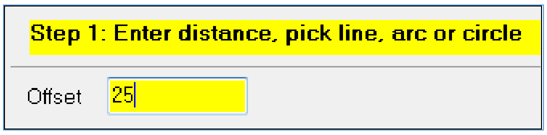
*We now have a **Rectangle** in the **X Y** plane that we can use to create our part*

5 The next step is to use **Offset** and offset the four edges inwards by **25mm**

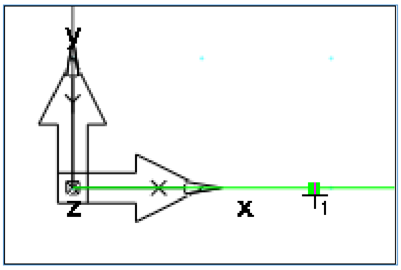
6 Select **Offset** from the **Construct** menu as shown below.



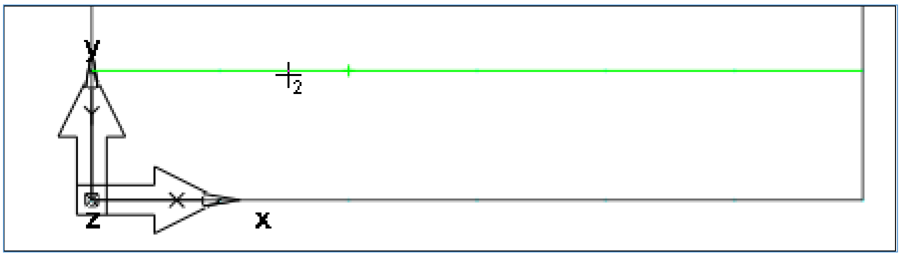
7 We will now set an **Offset** value of **25mm** as shown.



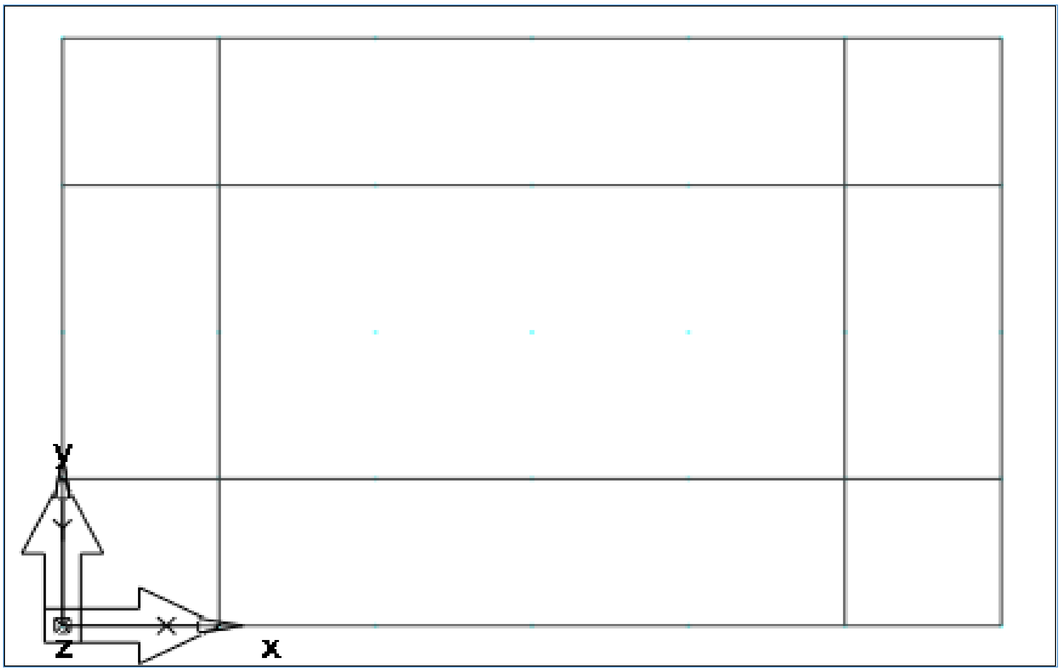
8 Use the **Offset** command to create parallel shapes of **Lines**, **Circles**, **Arcs**, and **Ellipses** etc. A copy of a selected entity is placed at a specified distance from the original entity. The original entity stays in position. Set the position by selecting either top or bottom left or right of the profile.



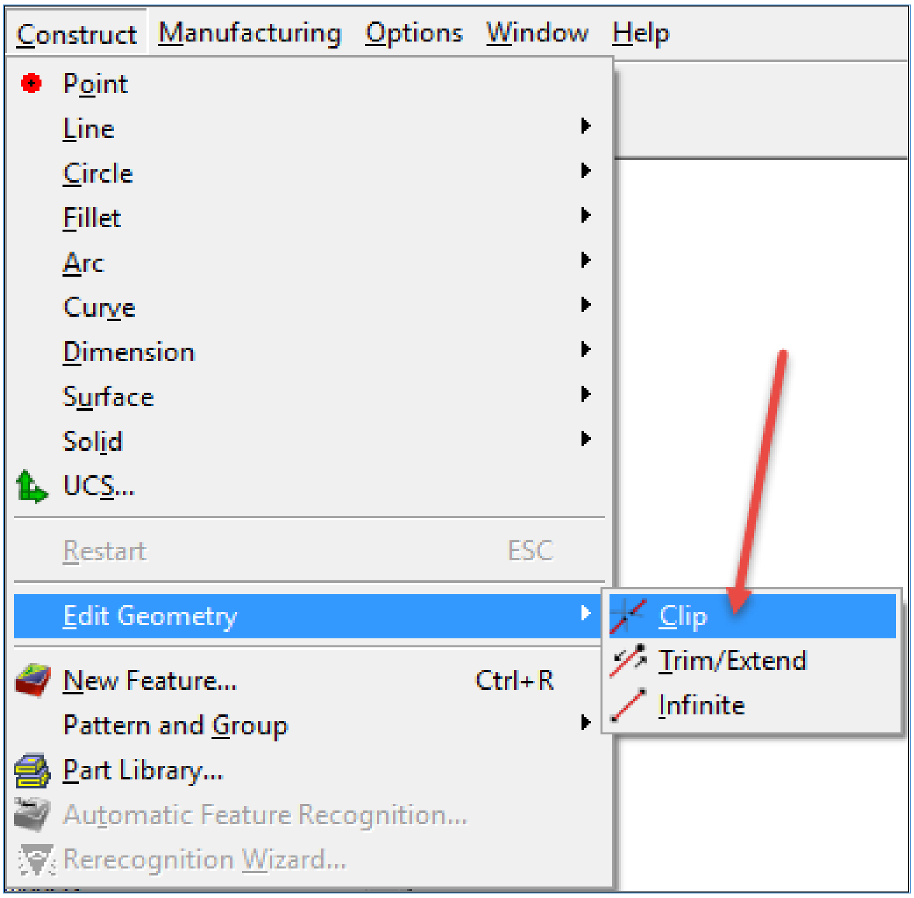
9 Put the cursor on the inside of the line to denote the side the offset line will be.



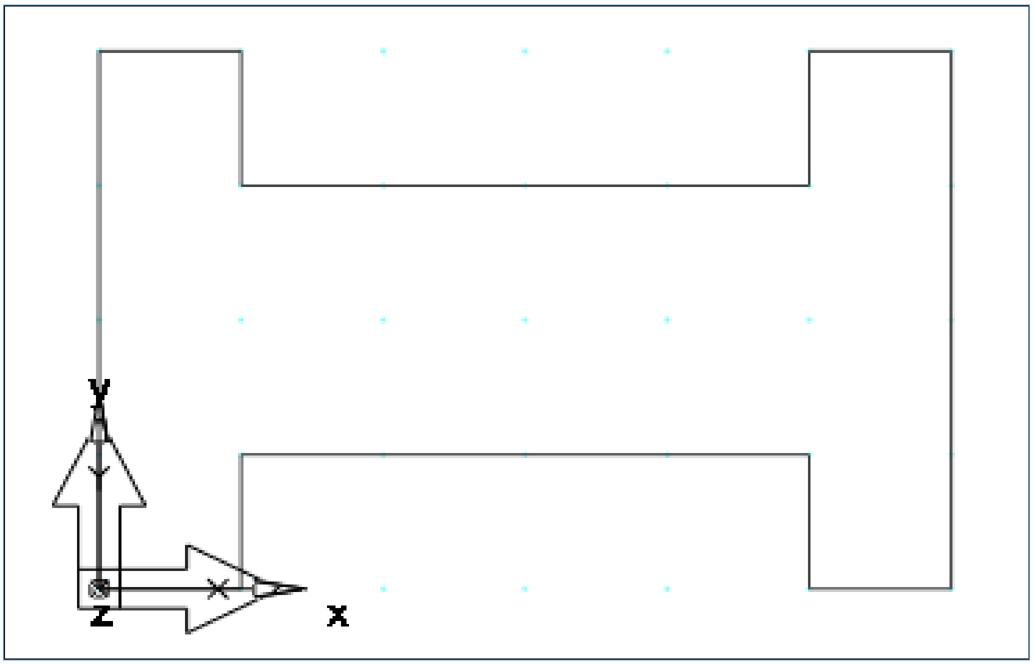
10 Apply this to the four edges as shown.



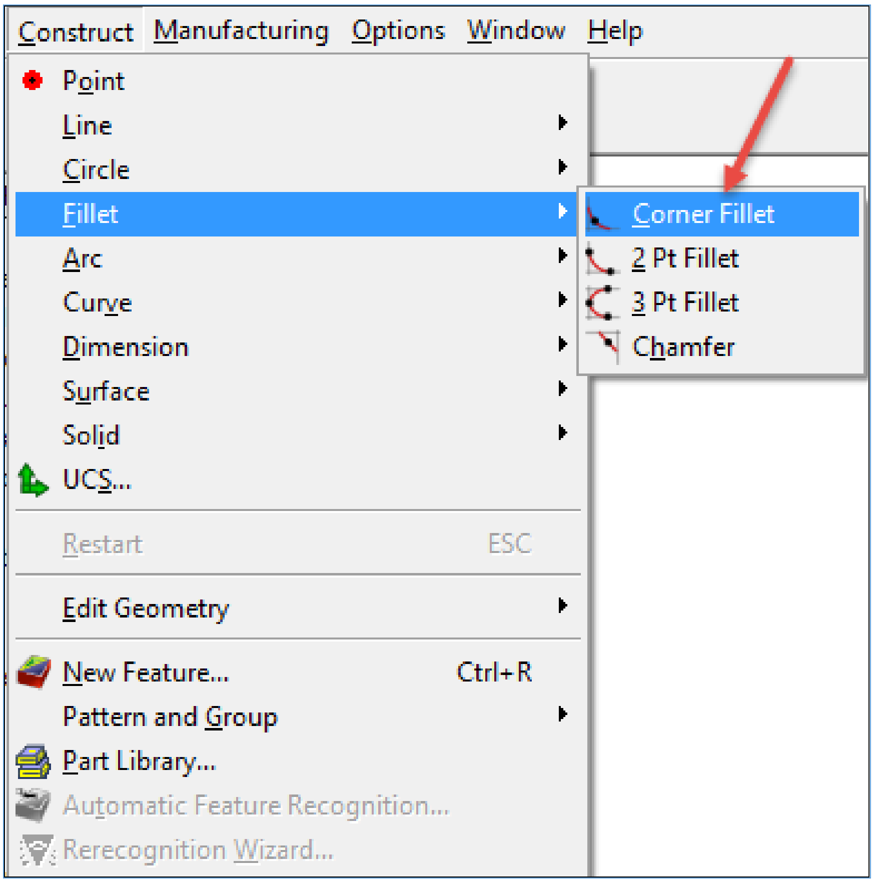
11 Now **Clip** the unwanted lines. Select the **Clip** command from the **Construct** / **Edit Geometry** Menu.



12 **Clip** the lines so the image is like the one shown below.

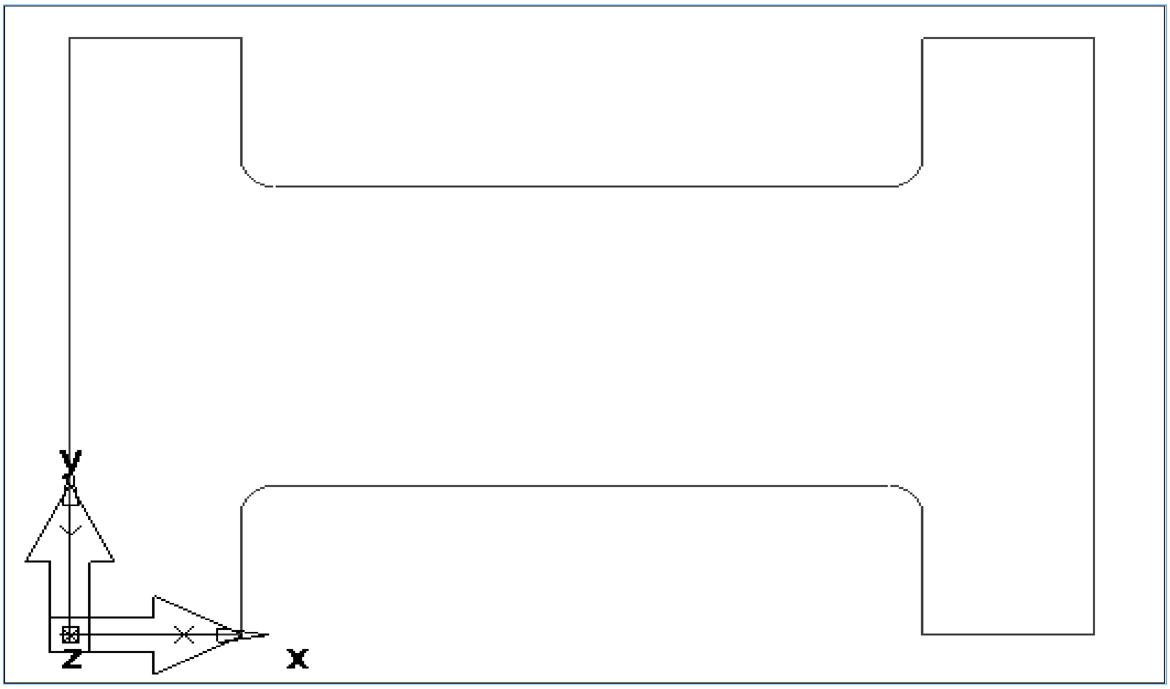


*We now need to **Fillet** the inner corners **5mm***

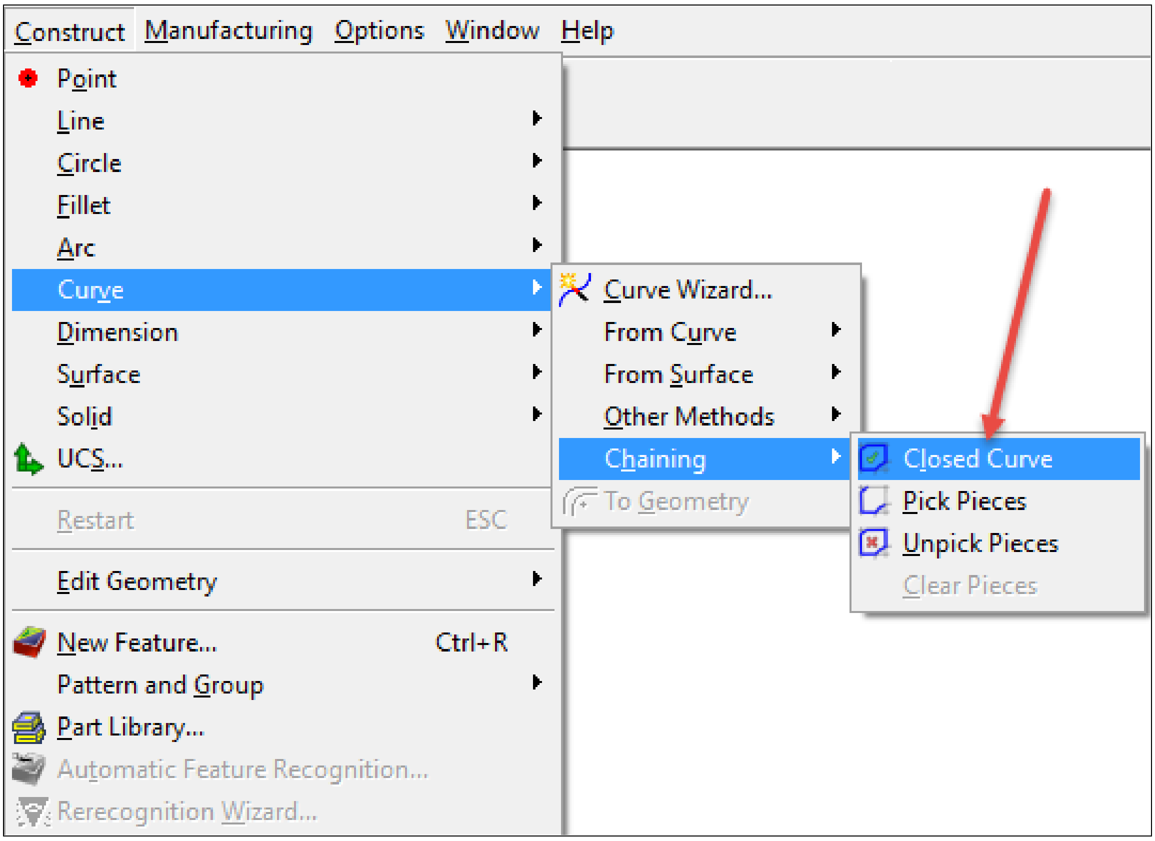


13 Select **Corner Fillet** Enter **5mm** select the corners shown.

14 It should look like the image shown below.

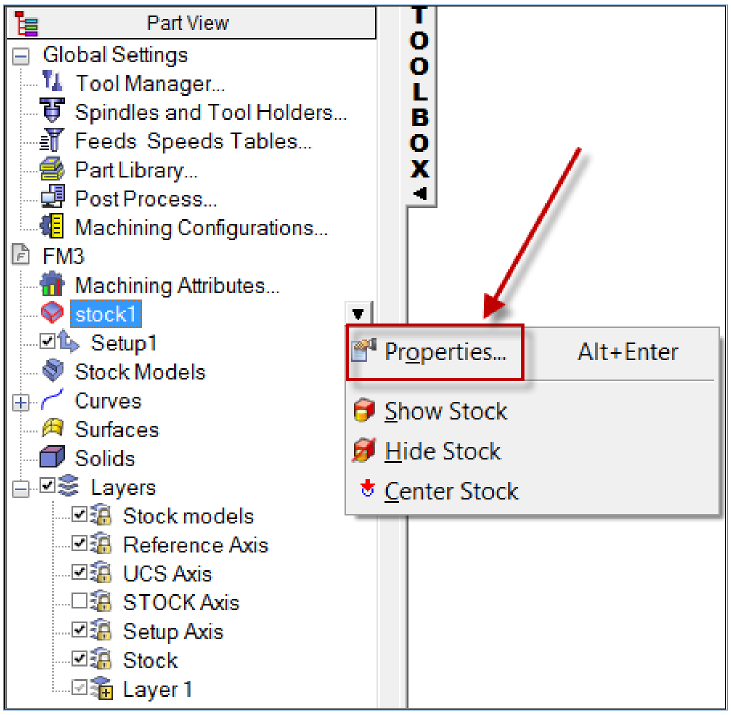


*Use the **Closed Curve** option to **Chain** the **geometry** into a **closed boundary** using a single mouse click. Rename the curve as **Stock Curve**.*

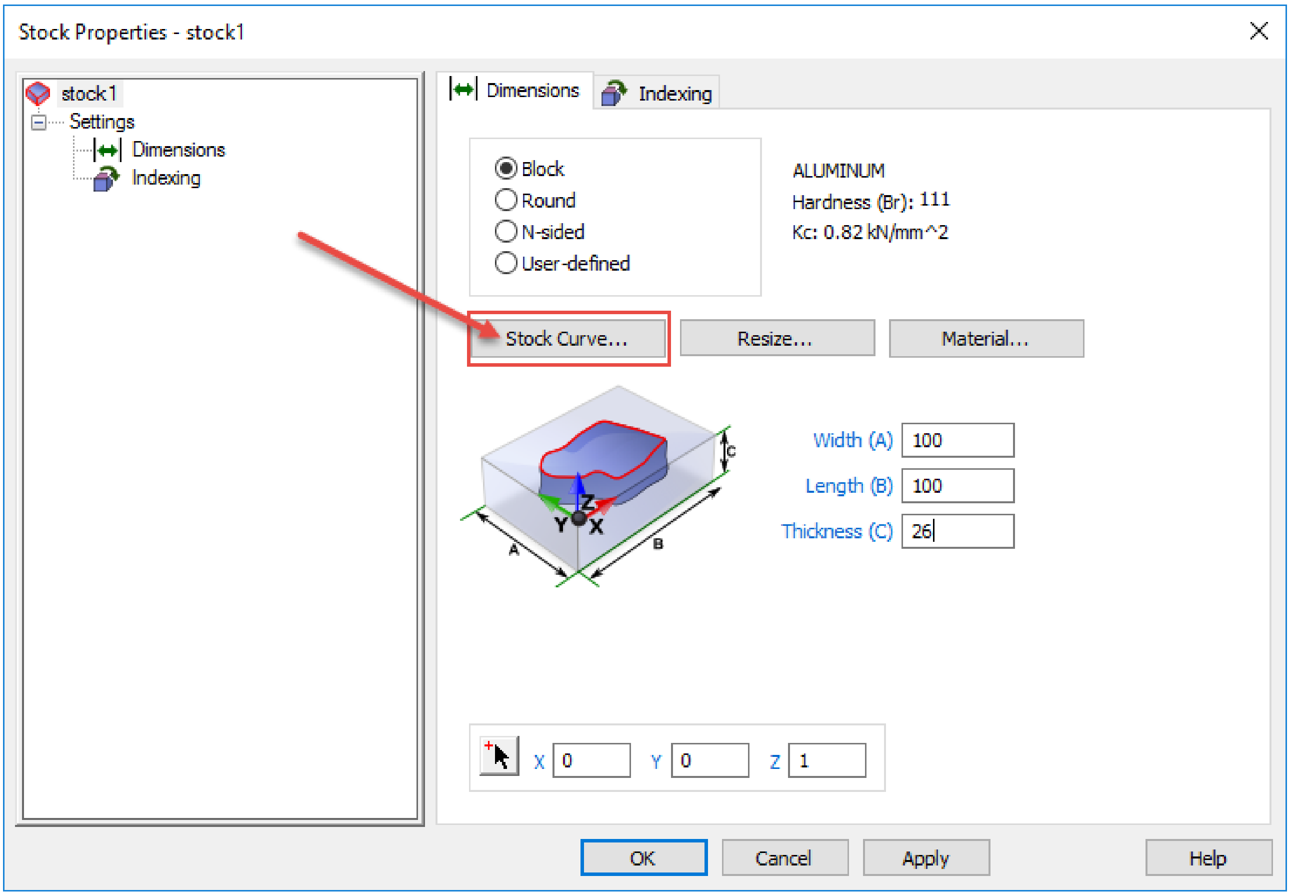


15 Select the **Geometry** and select **Create** to create the **Closed Curve**.

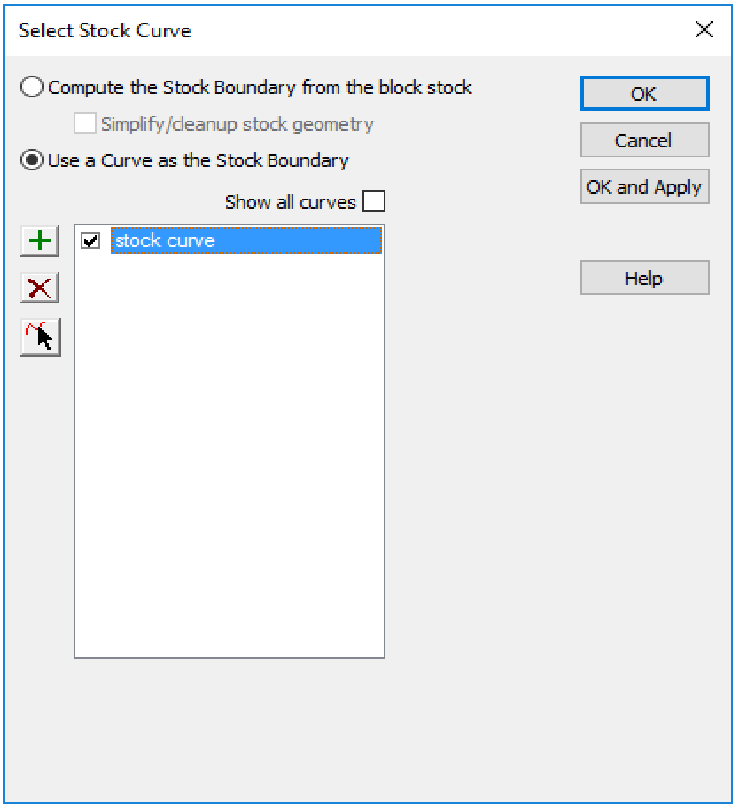
16 **Right-Click on the Stock** from the **Part View** in the **Toolbox** at the left hand side of the **FeatureCAM** interface and choose **Properties**.



17 Press the **Stock Curve** button as shown below.

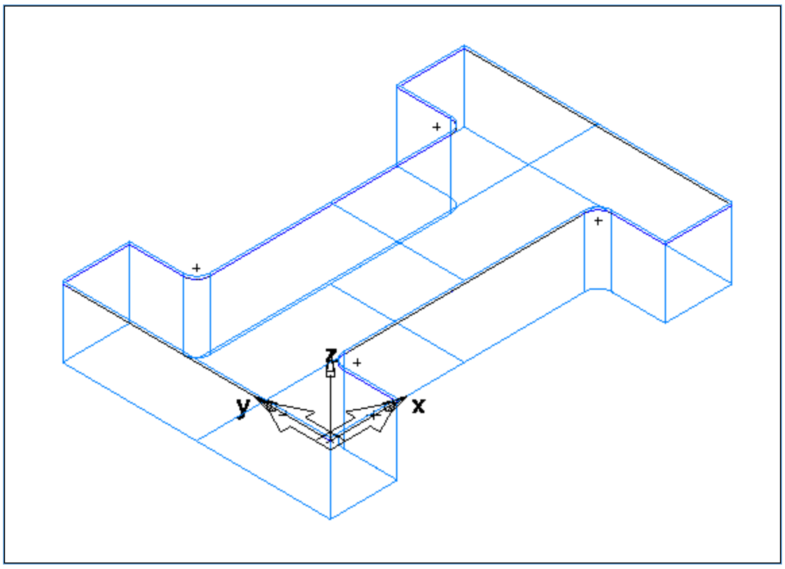


18 Select **Stock Curve** when prompted in my case it is **Curve1** but your Curve number may be different. Press **Apply** and **OK.**



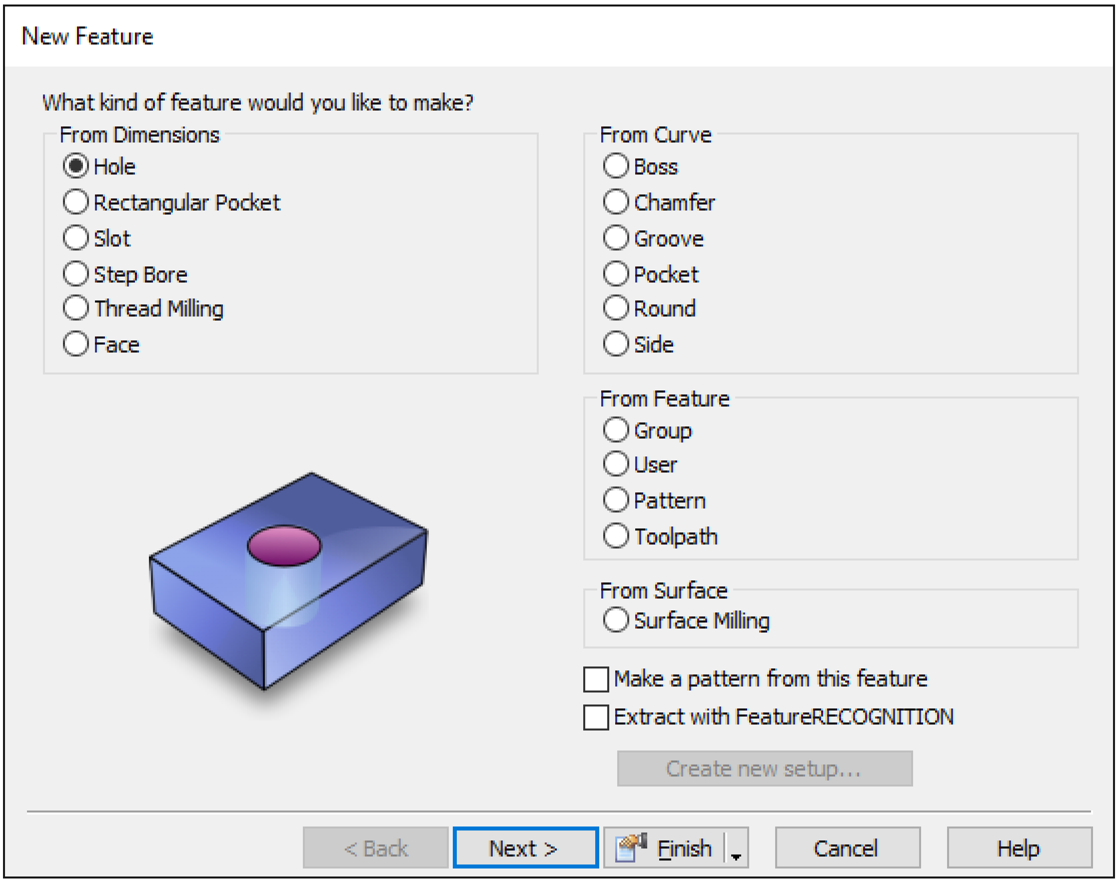
*Remember to **show stock**.*

The **Stock** should now look like the image shown below.

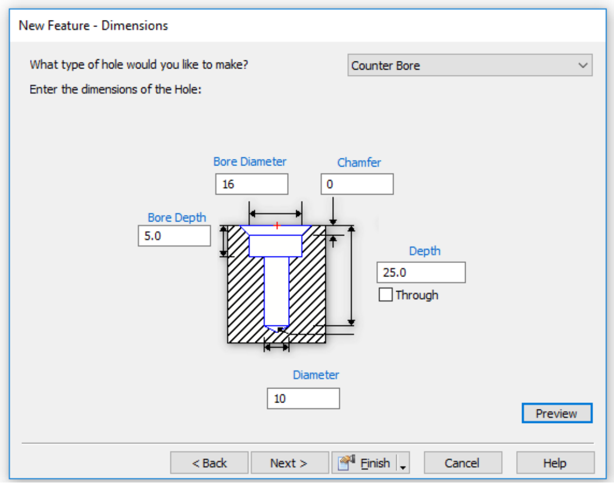


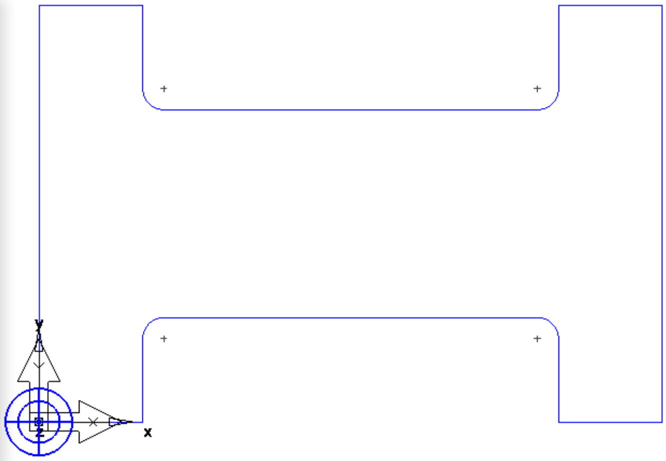
*We will now create the **5 Counter Bores** in this example*

19 Create the following **Counter bore** Features. Using the **Features Menu** press the following keys as a short cut, select **Ctrl + R** or select **Steps** and then **Feature** from the **Steps menu**. Create a **New Hole Feature**.



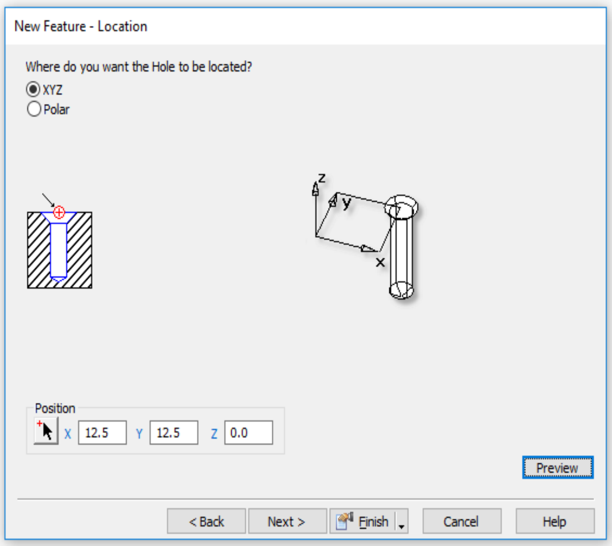
20 Select **Next** and select **Counter Bore**. All **Counter Bores** are **5mm** deep. Enter the values shown in the Menu on the next page.

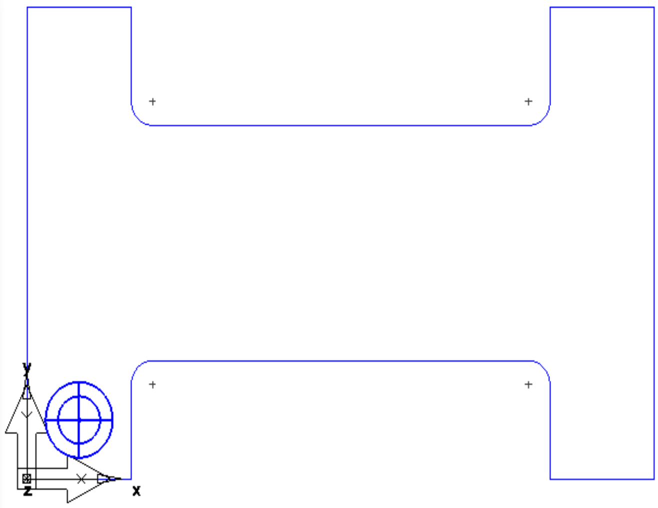




*Bore Dia **16mm** Bore Depth **5mm** Diameter **10mm** Depth **25mm** Select Next so we can position the Counter Bore.*

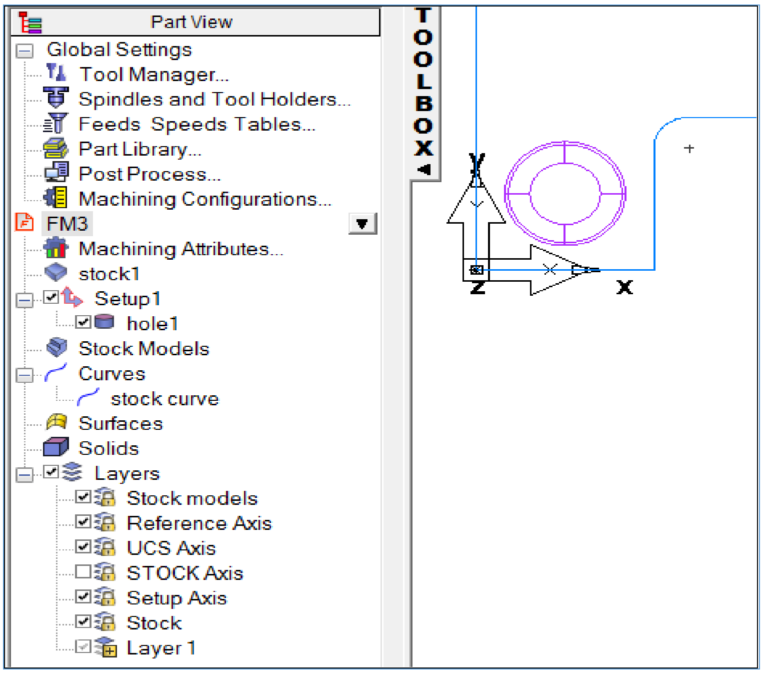
*The menu shown on the next page allows you to enter the positions for the Counter Bore. The positions are as follows **X12.5mm Y12.5mm***



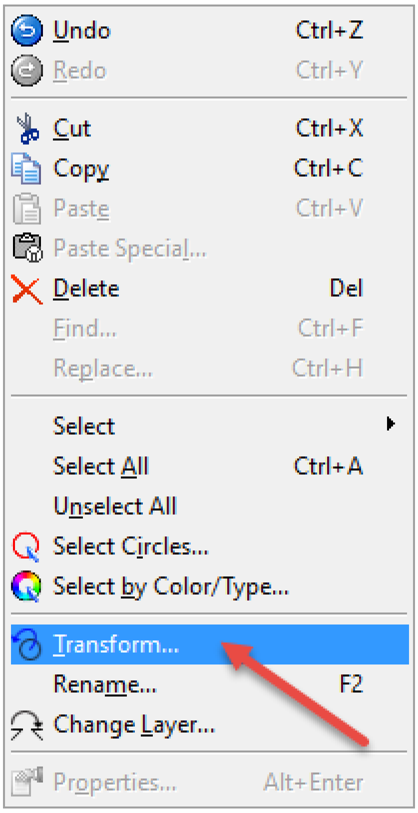


*We will now create all of the **Counter Bores** in their respective positions. We can either type in all of the co-ordinates manually or we can use the **reflect** command in **Transform**. This is because the hole positions are symmetrical about the **Centre line**.*

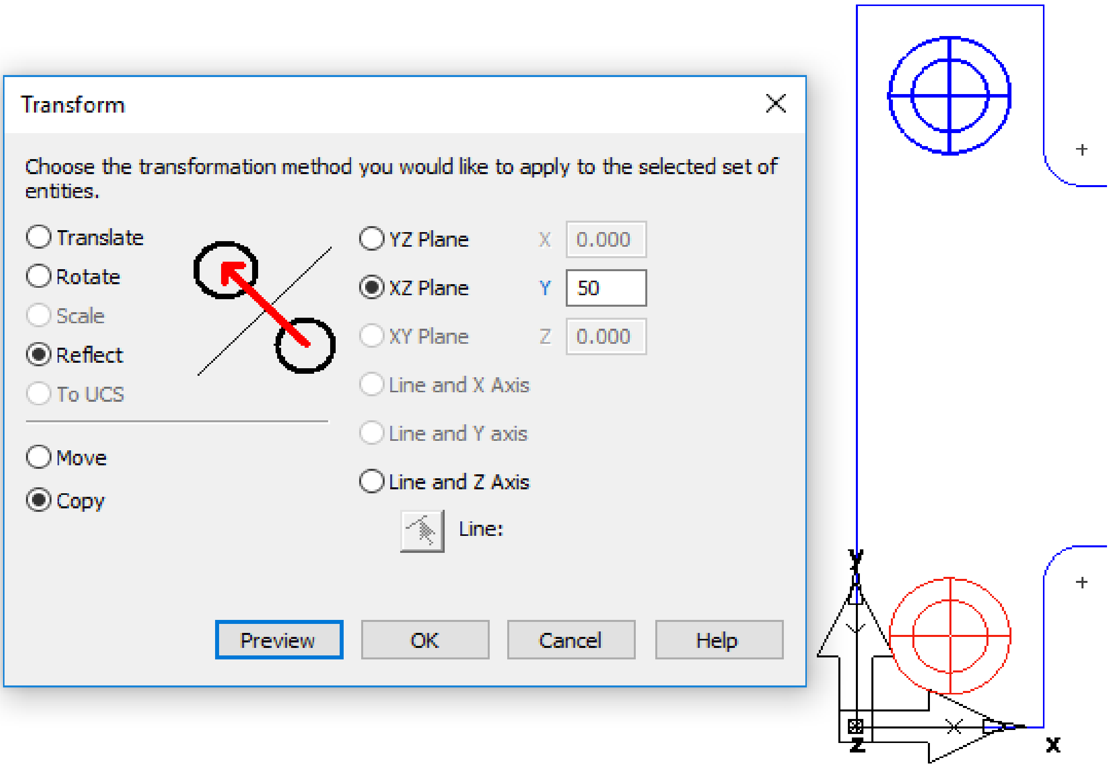
*Make sure you have highlighted the new **Counter Bore Feature**. The best way to do this is in the **Part View** Menu.*



21 Please select the **Edit** menu and select **Transform**.

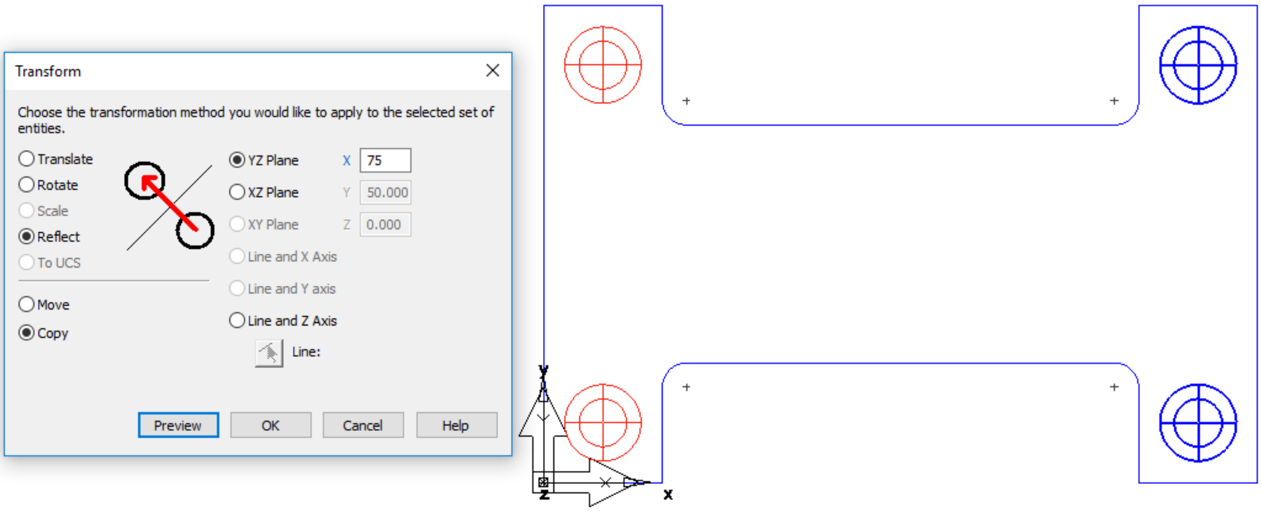


22 The following menu will appear.

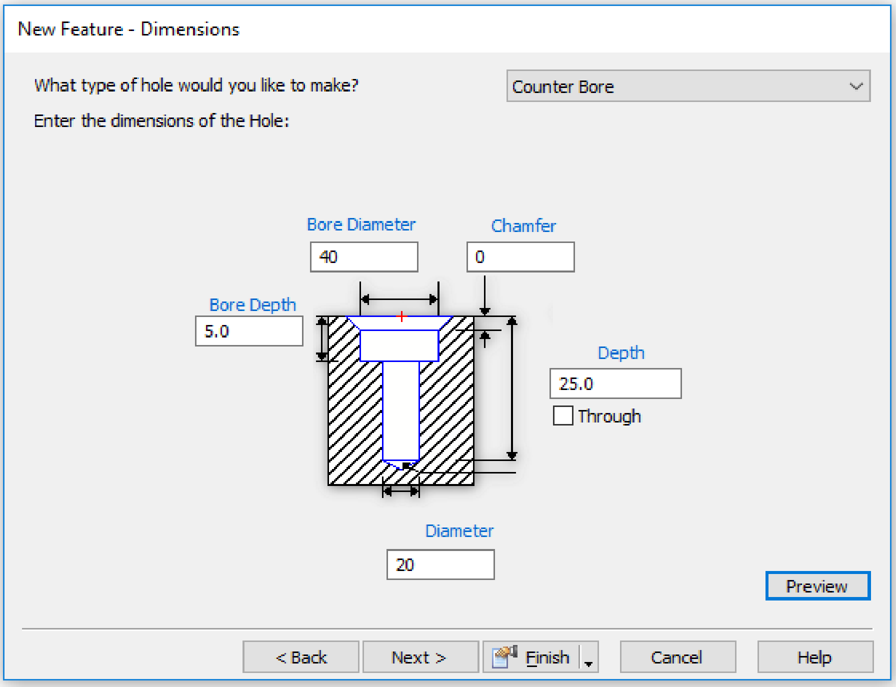


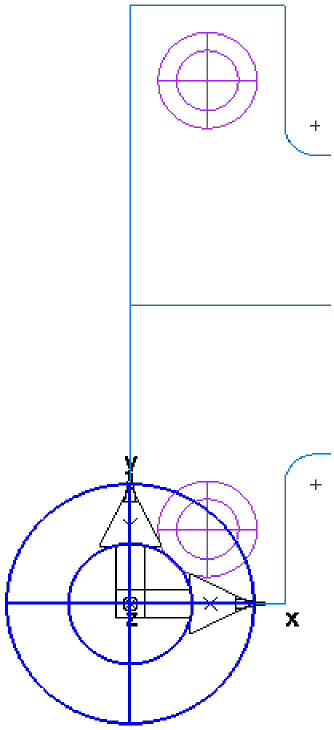
*You will notice that the **Reflect** position is in the **XZ** type in **50mm** this is the **Reflect** point in the Y direction. Enter your relative units and select **OK**.*

*We now need to **reflect** the two **Counter Bores** in the **YZ** Direction along the X direction. Highlight the two **Counter Bores**, select the first one and hold down the **Ctrl** Key, select the other one in the **Part View**. Both will be selected. Then select **Edit** then **Transform** as shown. Remember to select **Copy** and **Reflect**. Set the **YZ** Plane and enter the following values. **X75mm**. Select Preview if required, then select **OK**.*



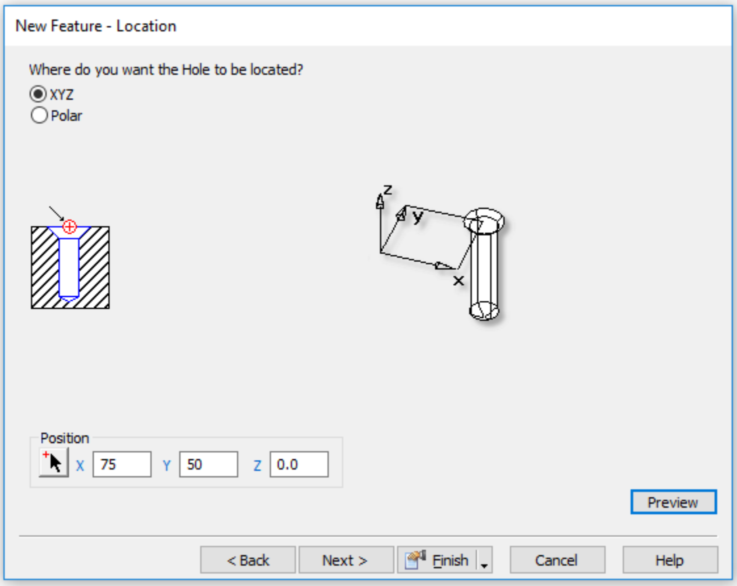
23 We just need to create the **Counter Bore** in the middle at **X75mm Y50mm** Select **Ctrl+R** or select **Feature** from the **Steps** menu. Select **Hole** as the **New Feature** and **Counter Bore,** and then enter **40mm** for the top bore and **20mm** for the hole through.

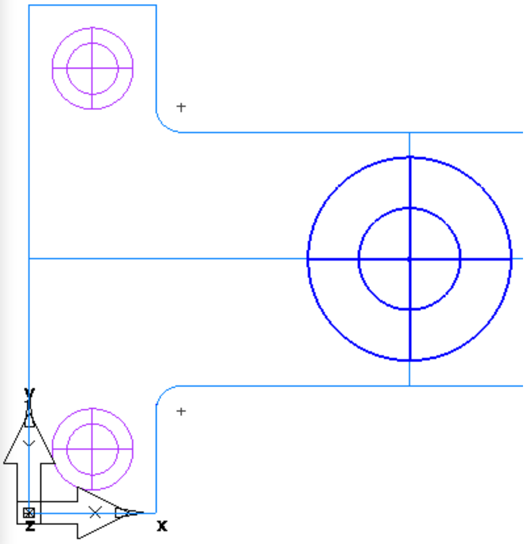




24 After entering the values select **Next**.

25 Now position the **Counter Bore** at **X75mm Y50mm.** Select **Next**.





26 Select **Finish.**

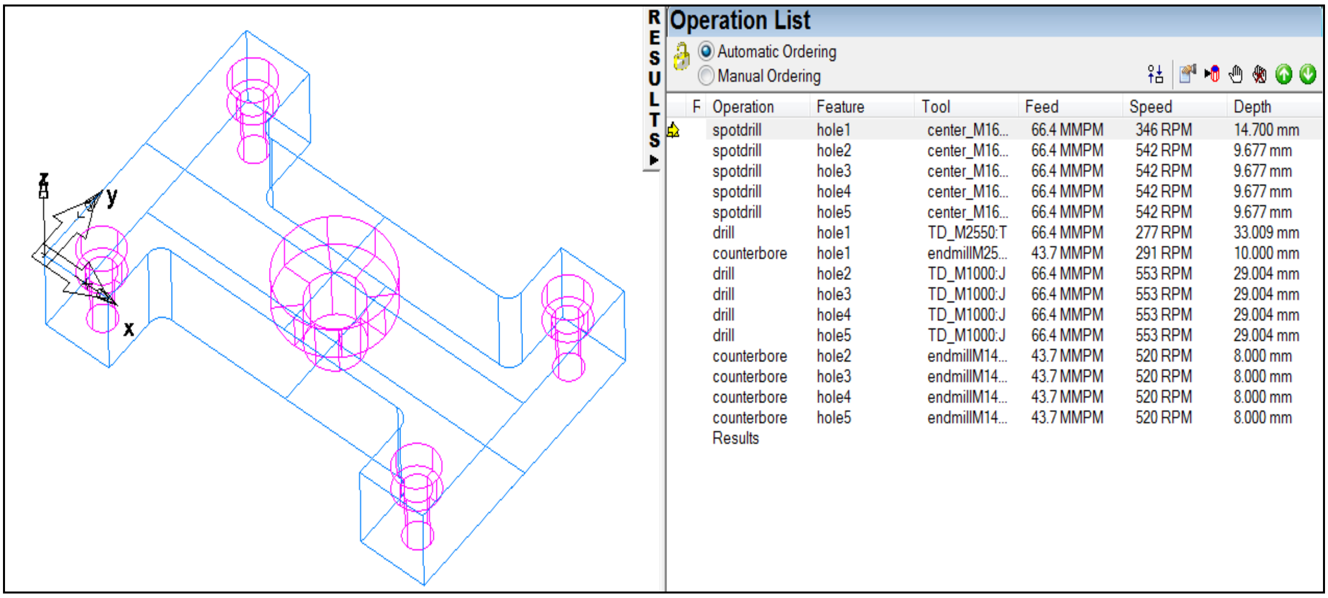
*The job is now **finished**.*

27 Run the **3D Simulation** as shown below.

*Have a look at the Wireframe image plus **Operation List**.*

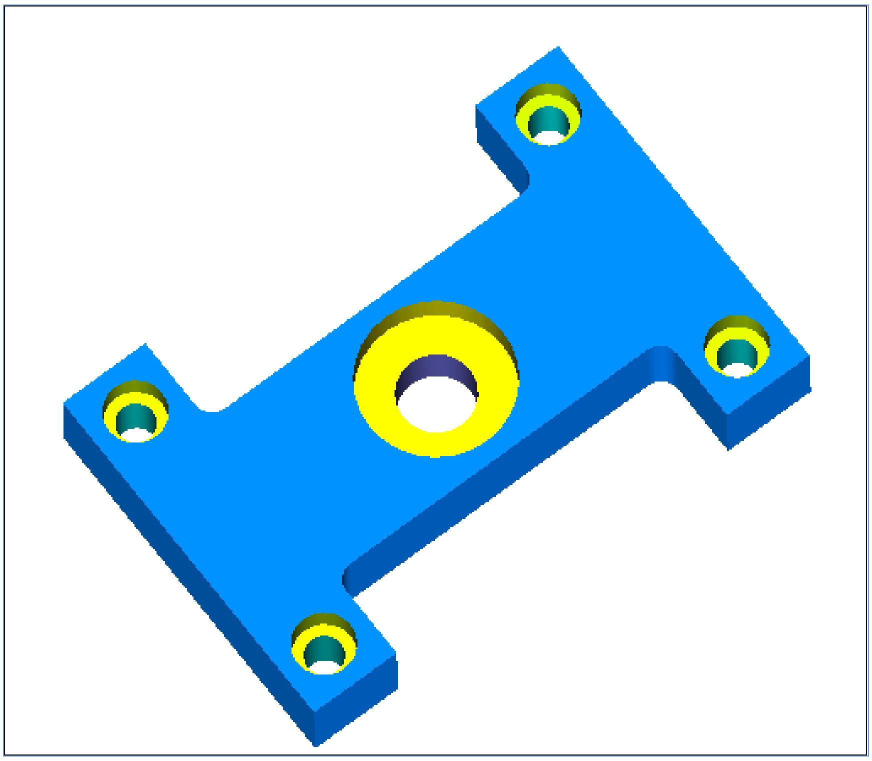
28 Select **File**>**Save NC** to Output CNC Code to a directory of your choice.

29 Remember to **Save** the file in your Instructors preferred location.



F	Operation	Feature	Tool	Feed	Speed	Depth
	spotdrill	hole1	center_M16...	66.4 MMPM	346 RPM	14.700 mm
	spotdrill	hole2	center_M16...	66.4 MMPM	542 RPM	9.677 mm
	spotdrill	hole3	center_M16...	66.4 MMPM	542 RPM	9.677 mm
	spotdrill	hole4	center_M16...	66.4 MMPM	542 RPM	9.677 mm
	spotdrill	hole5	center_M16...	66.4 MMPM	542 RPM	9.677 mm
	drill	hole1	TD_M2550:T	66.4 MMPM	277 RPM	33.009 mm
	counterbore	hole1	endmillM25...	43.7 MMPM	291 RPM	10.000 mm
	drill	hole2	TD_M1000:J	66.4 MMPM	553 RPM	29.004 mm
	drill	hole3	TD_M1000:J	66.4 MMPM	553 RPM	29.004 mm
	drill	hole4	TD_M1000:J	66.4 MMPM	553 RPM	29.004 mm
	drill	hole5	TD_M1000:J	66.4 MMPM	553 RPM	29.004 mm
	counterbore	hole2	endmillM14...	43.7 MMPM	520 RPM	8.000 mm
	counterbore	hole3	endmillM14...	43.7 MMPM	520 RPM	8.000 mm
	counterbore	hole4	endmillM14...	43.7 MMPM	520 RPM	8.000 mm
	counterbore	hole5	endmillM14...	43.7 MMPM	520 RPM	8.000 mm
	Results					

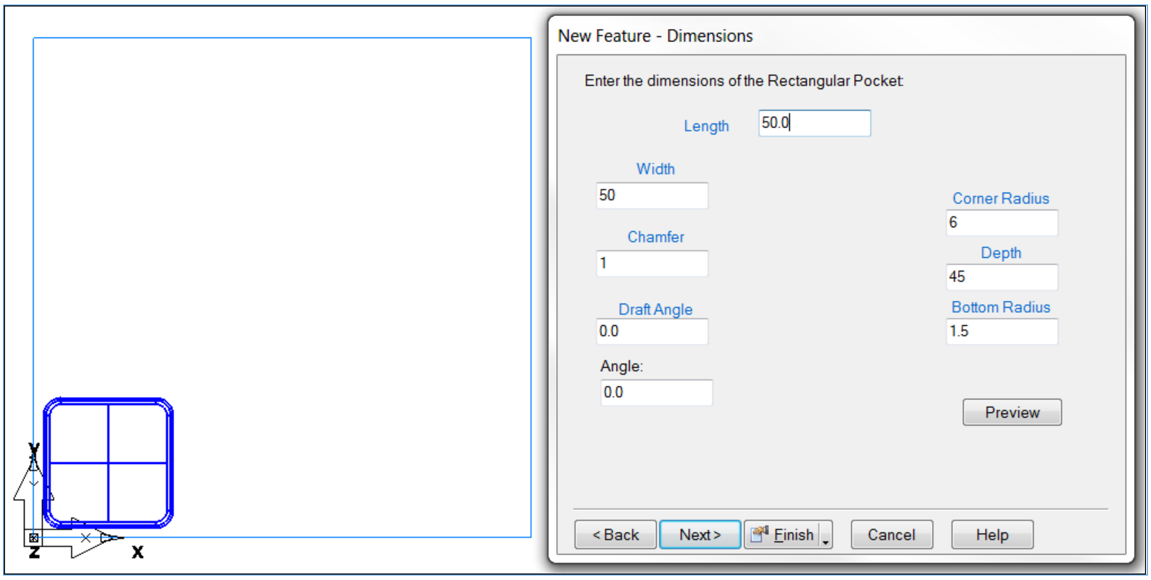
***FeatureCAM** has created the toolpaths automatically from the **Features** you have created. With **FeatureCAM** you generate your part using **Features** such as a Tapped hole etc., operations are automatically created. **FeatureCAM** also manages the details of the manufacturing process such as **Tool selection**, **Speed** and **Feed** rates, and toolpaths. To modify any element of the part program, just change a few settings on a **Feature** and a whole new set of operations are generated to reflect your changes.*



Pocket and Pattern Exercise (Optional Exercise)

A Pocket Feature is easily changed into a Pattern

1. Open a **New Part Document**. **Millimeters**
2. Define a Stock block of **200mm** x **200mm** x **50mm**. Location **X0,Y0,Z0.**
3. Select **Top View** or **Ctrl + 5** to orientate the view into the X, Y, Plane.
4. Select Ctrl+R New Feature.
5. Create a Rectangular Pocket Feature from dimensions.

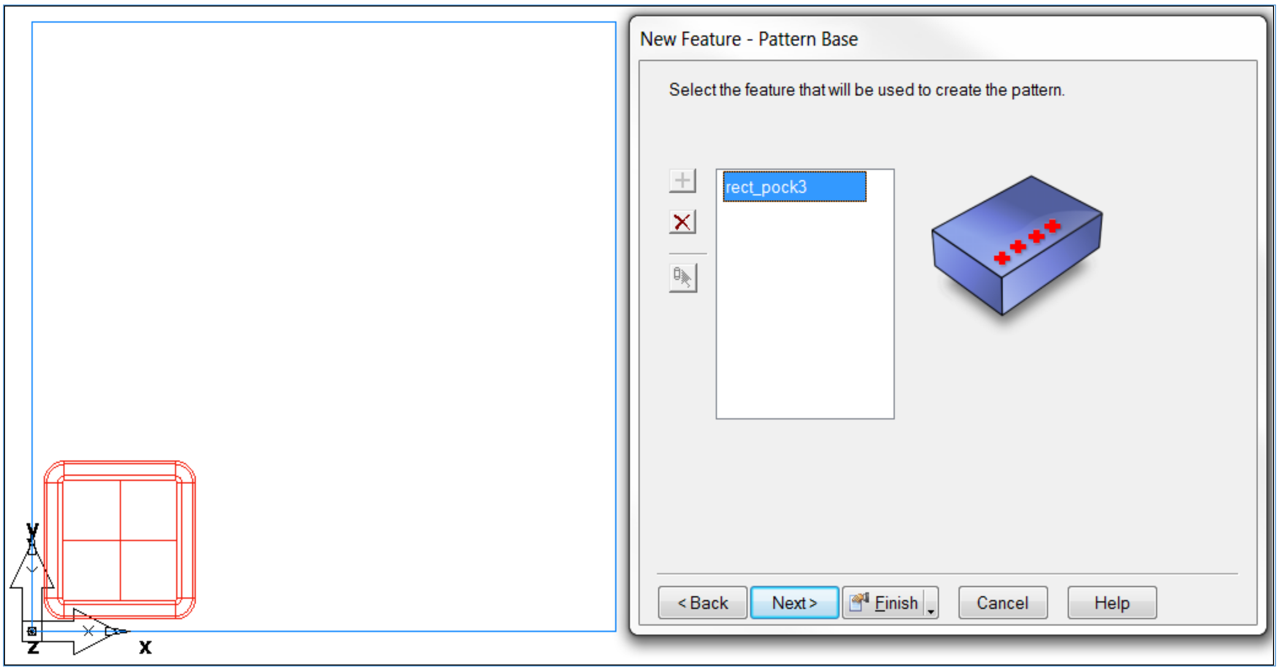


6. Enter the above values into the Menu. Then select **Next.**
7. Location. **X5mm Y5mm**

8 Select **Finish**.

9 Create a **Pattern**. Select **Ctrl+R** select **Pattern.**

10 In **Part View** select the **Rectangular Pocket** we have just created.



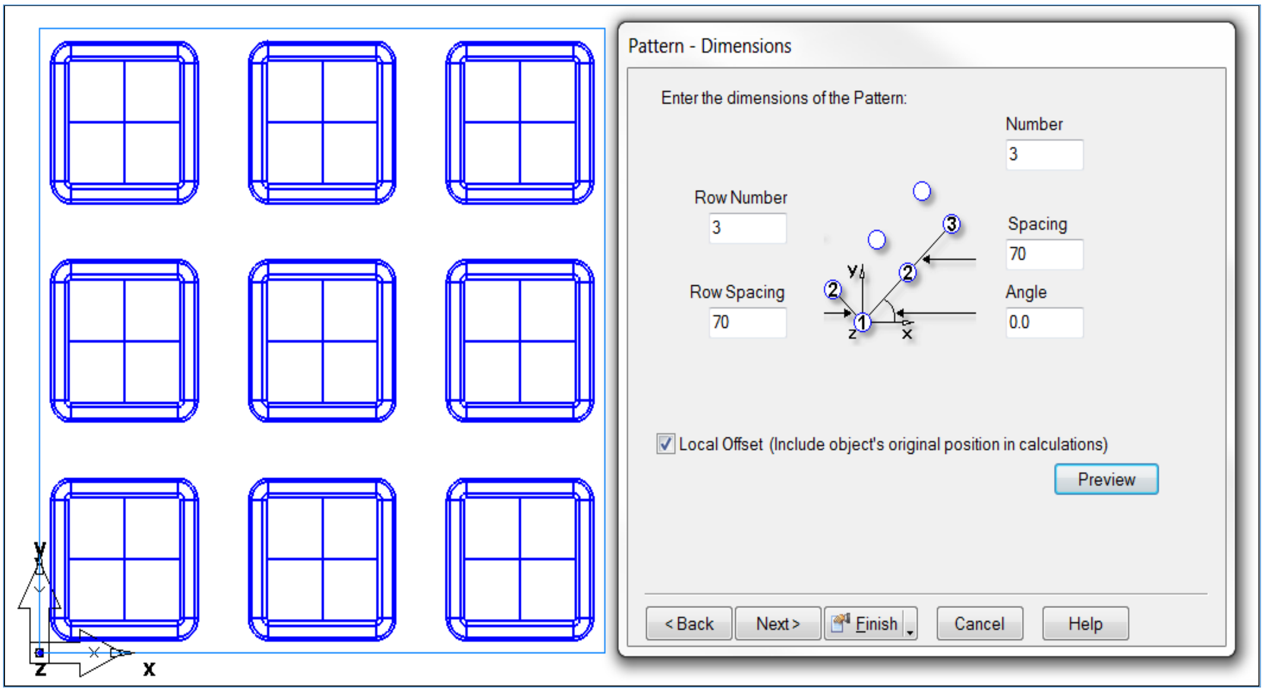
11 Select **Next**.

12 We need a **Rectangular** Pattern.

13 Select **Next**.

14 Select **Local Offset**.

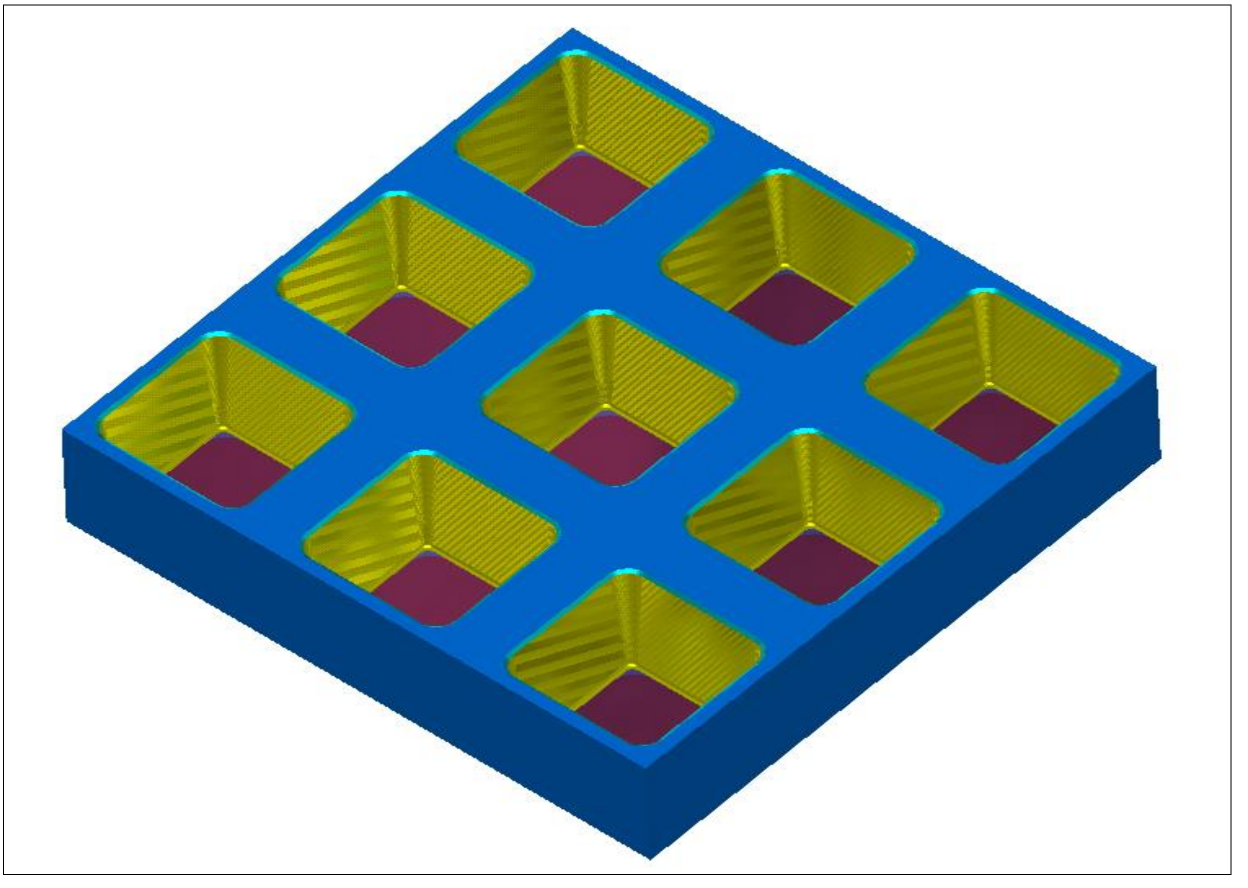
15 Enter the values as shown.



16 Select **Finish** and **OK**.

17 Select Minimize Rapid Distances in the **Automatic Ordering Options**.

18 Run **3D Simulation.** See image on the next page.



Pockets with Islands (Optional Exercise)

1 Open a **New Part Document**. **Millimeters**

2 Create **a Material Block X400mm Y400mm Z40mm.** *Hide the material.*

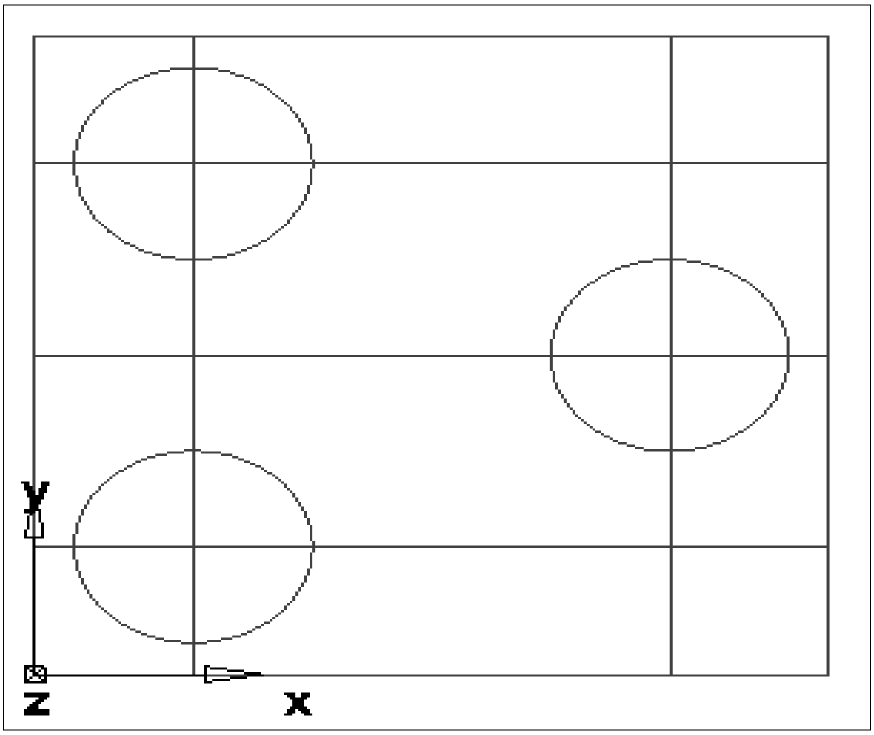
3 Use **Top View** or **Ctrl + 5** to set the correct View.

4 Create a Rectangle **X400mm Y400mm.** Select **Construct>Curve>Other Methods>Rectangle.**

Remember to Create as Arcs and Lines.

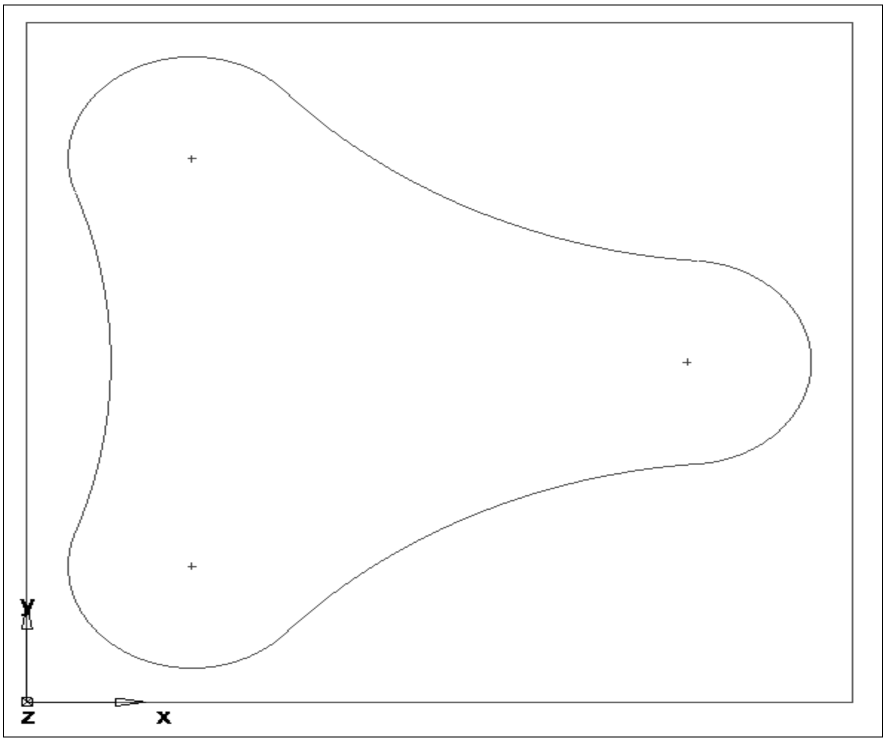
5 **Offset** the lines by **80mm** to make an inner square. Make a Horizontal Line in the middle of the square at **200mm.**

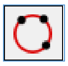
6 Create Three Circles Radius **60mm** as shown below.

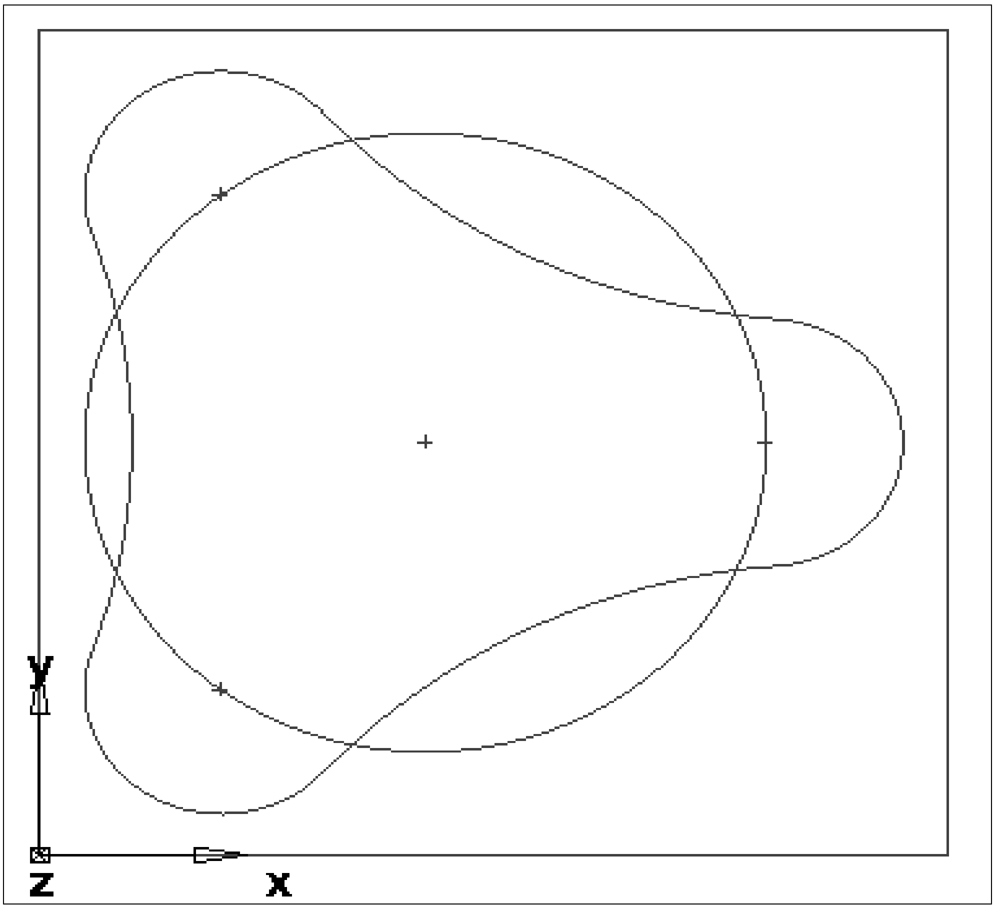


7 Create a circle **Tangent two** with a radius of **300mm** and snap to all circles. Delete all **Vertical** or **Horizontal Lines**, and then **Clip** to trim back the unwanted Geometry.

8 The Geometry should now look like this below.

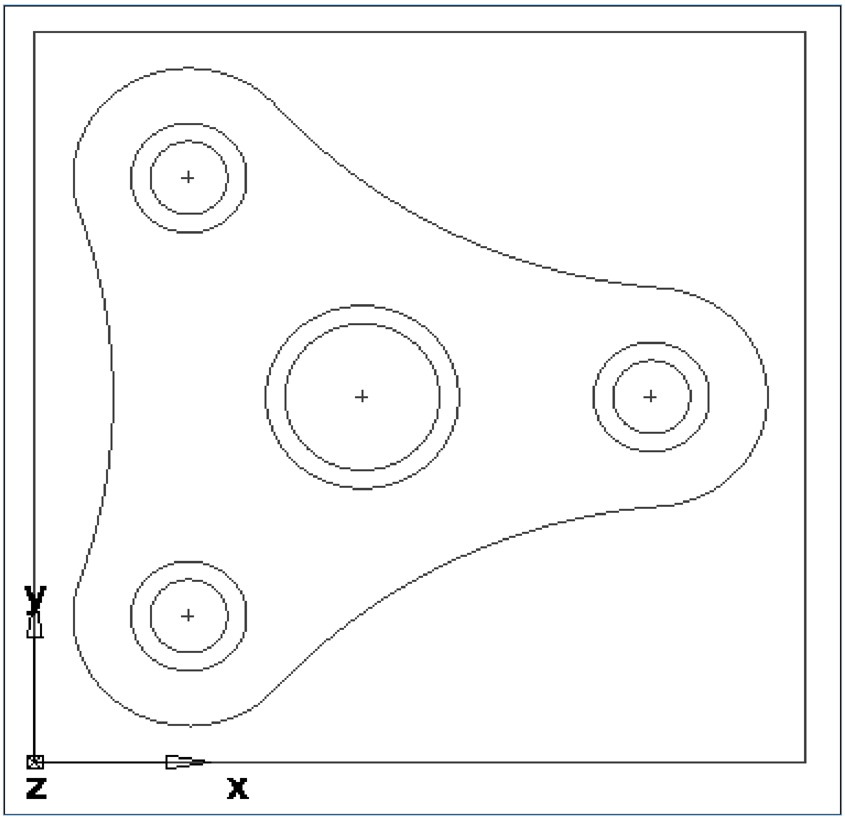


9 To find the Centre of the defined shape create a circle from 3 points  snapping to each of the larger radius points in turn.



*This will create a centre point. We will use this to create a circle with a radius of **50mm** then create 3 circles with a radius of **30mm** and snap to the three points as shown below.*

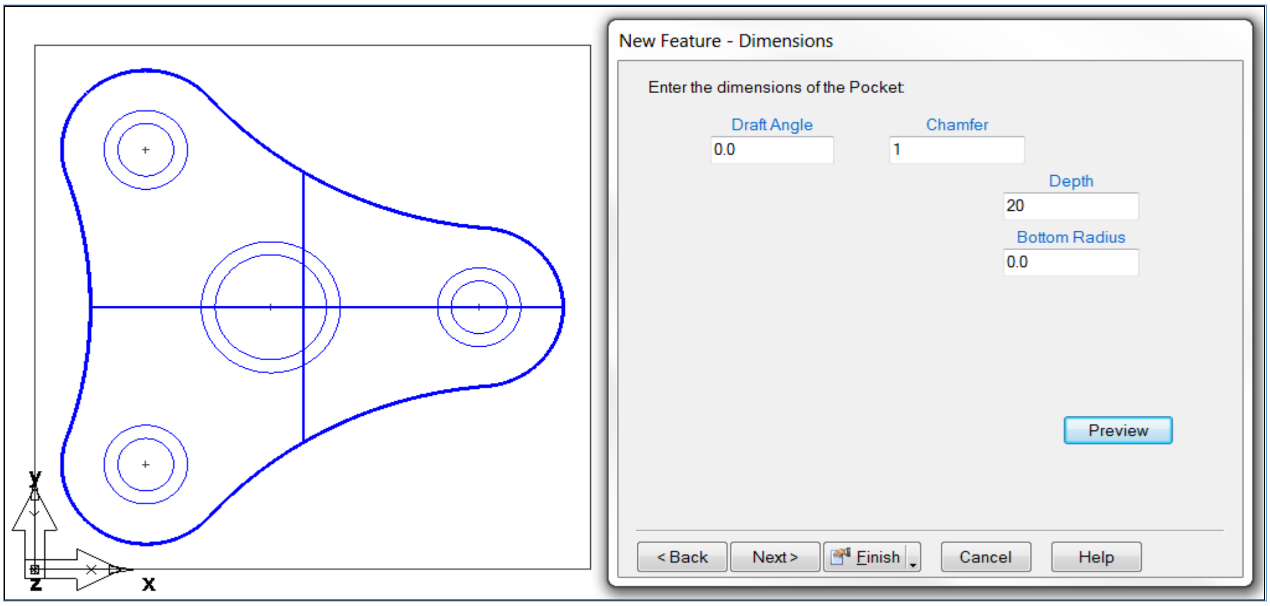
*Delete the unwanted circle in the middle. **Offset***  *all inner Geometry by **10mm** to the **inside** of all the circles. Your drawing should look like this below.*



10 Create a Closed Curve for all of the geometries.

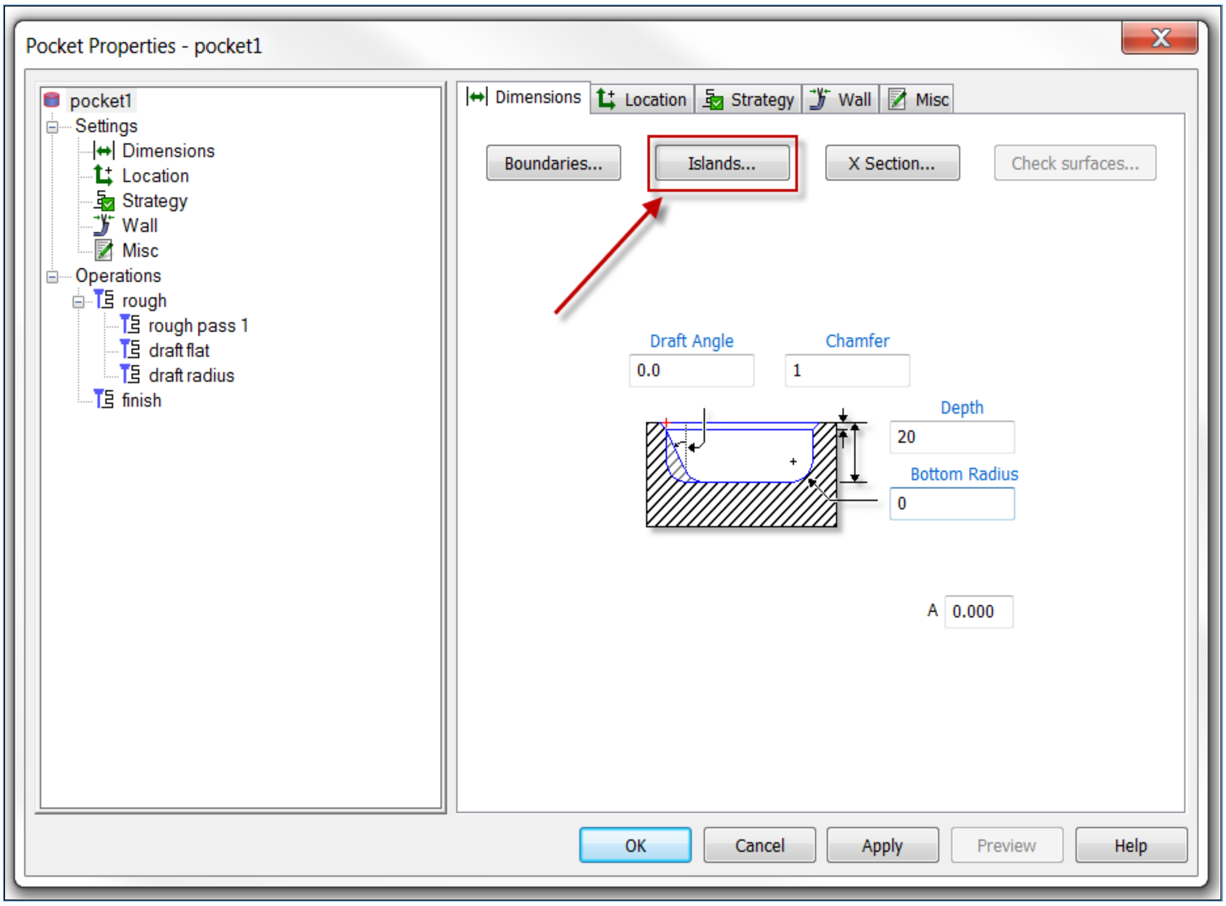
11 Select **CTRL + R** to create a **New Feature**. Select **Pocket from Curve**.

12 Select the Outer shape as the main profile. Select **next** then **next** and enter the following information.

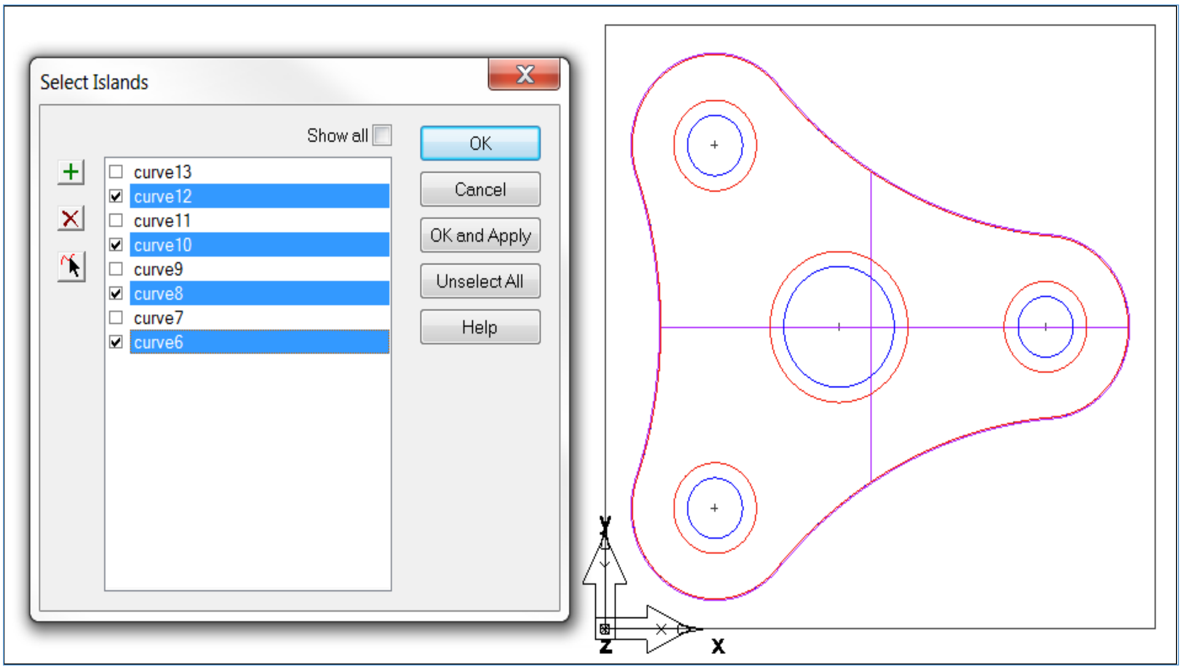


13 Select **Finish** and you will be presented with a new menu.

14 Select **Islands** from the menu.

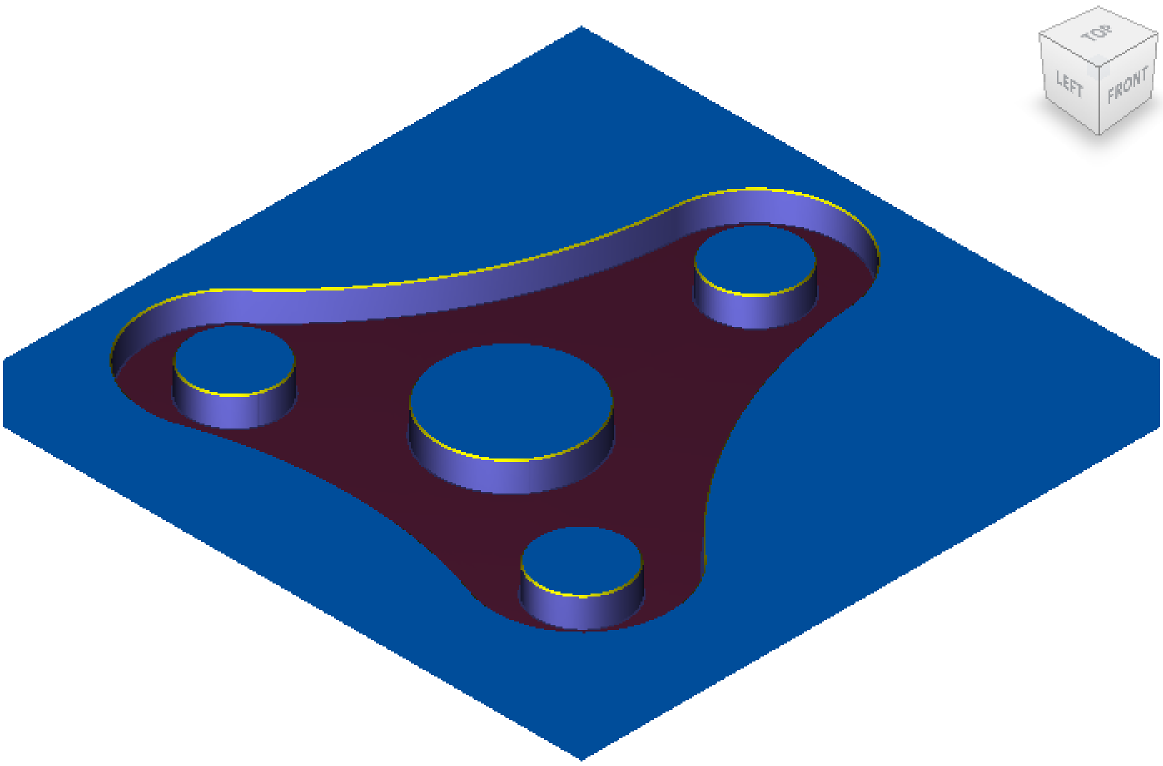


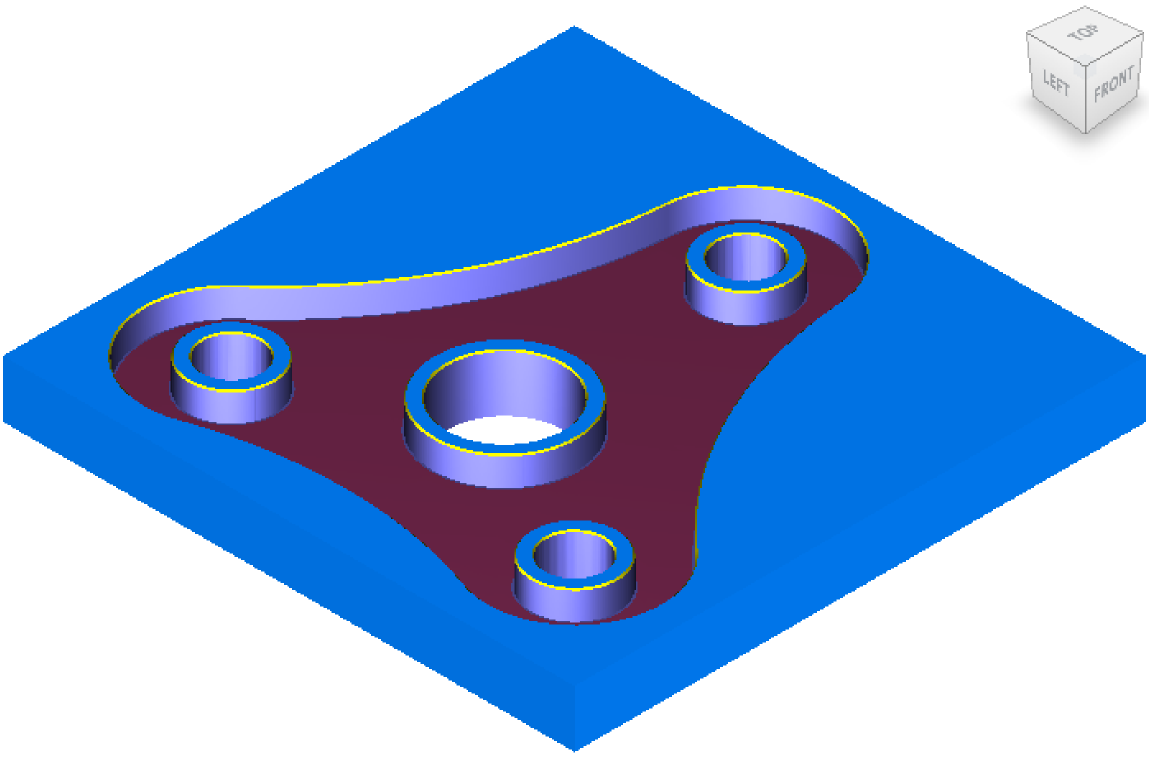
15 Select each of the outer curves for each Boss. Select OK and Apply



16 **Run** the **3D Simulation** to see the results.

We need to finish off by creating the pockets through the four bosses .Create another pocket feature and select all the curves that are left.



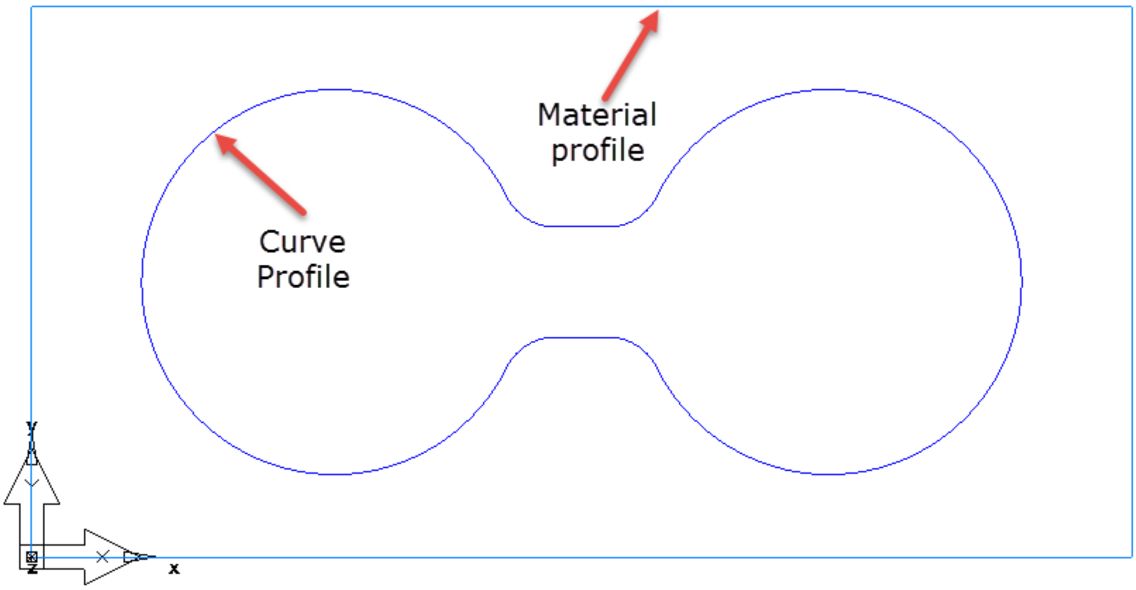


The job is now ***finished****.*

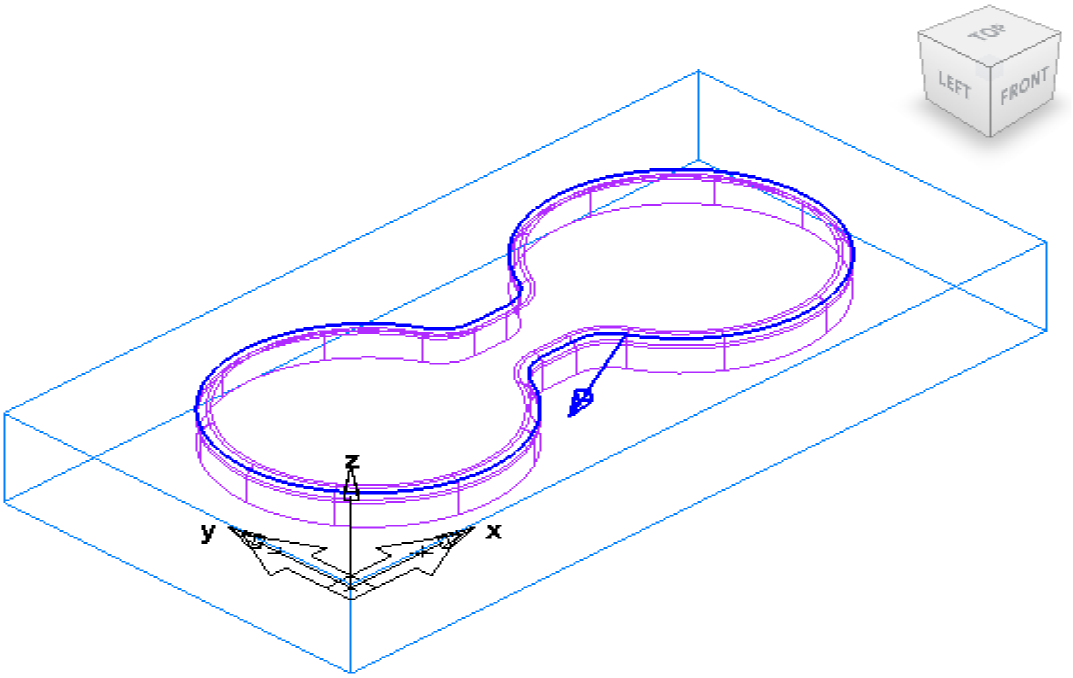
Round & Chamfer Exercise (Optional Exercise)

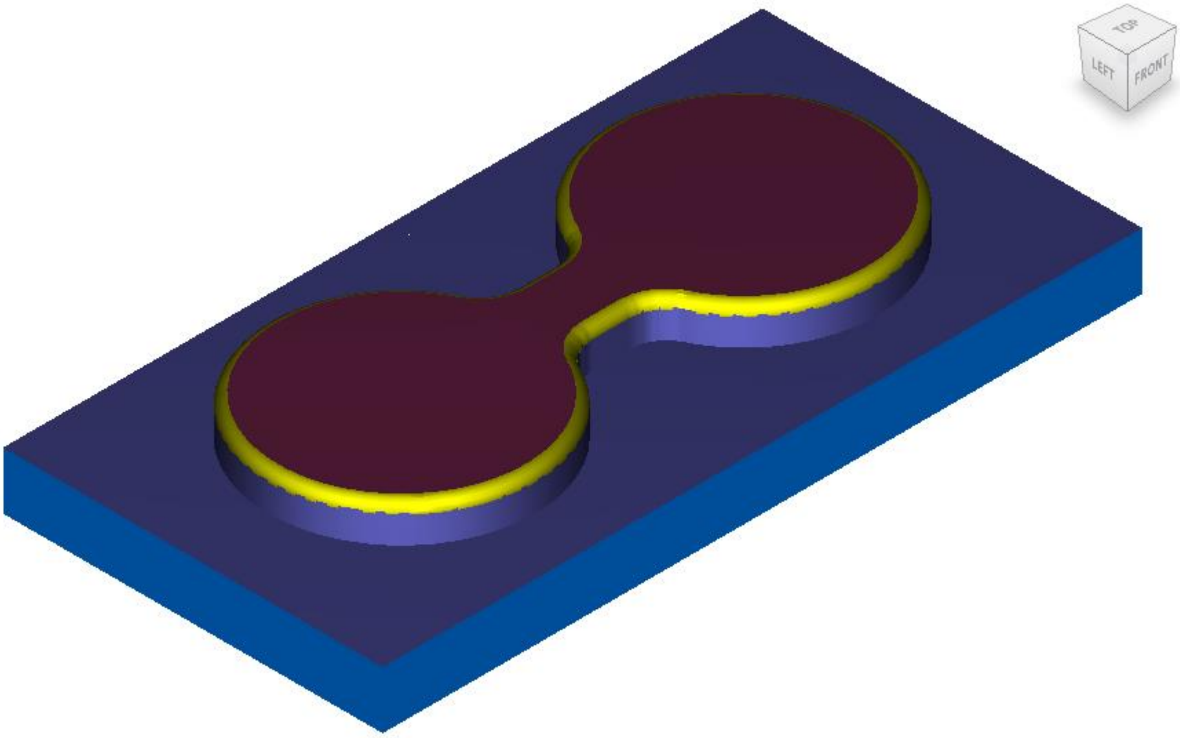
Round: - *This uses a user defined tool that has been pre-ground to a shape that will produce a radius on an edge. This will machine a rounding operation that follows a curve and is best described as a fillet radius around the top of a Boss or Side Feature.*

1. Open a **New Part Document**. **Millimeters**
2. Create a material block **100mm** wide x **200mm** long **30mm** thick.
3. Press **CTRL + 5** to orientate the view to Top or XY plane.
4. Draw a circle Radius **35mm X55mm Y50mm Z0**. Then draw another circle Radius **35mm X145mm Y50mm Z0**.
5. Draw a line snapping to the centres of the two circles.
6. Offset the line up and down by **16mm** and trim away the unwanted geometry using **Clip**.
7. Fillet the corners **16mm**. It should look like the image below.



8. Create a closed Curve of the Geometry and create a **Side** Feature **12mm** deep.
9. Run the **3D simulation** and then press **Eject**.
10. Create a New Feature **Round** Select the Curve and make sure the Arrow is pointing on the outside.

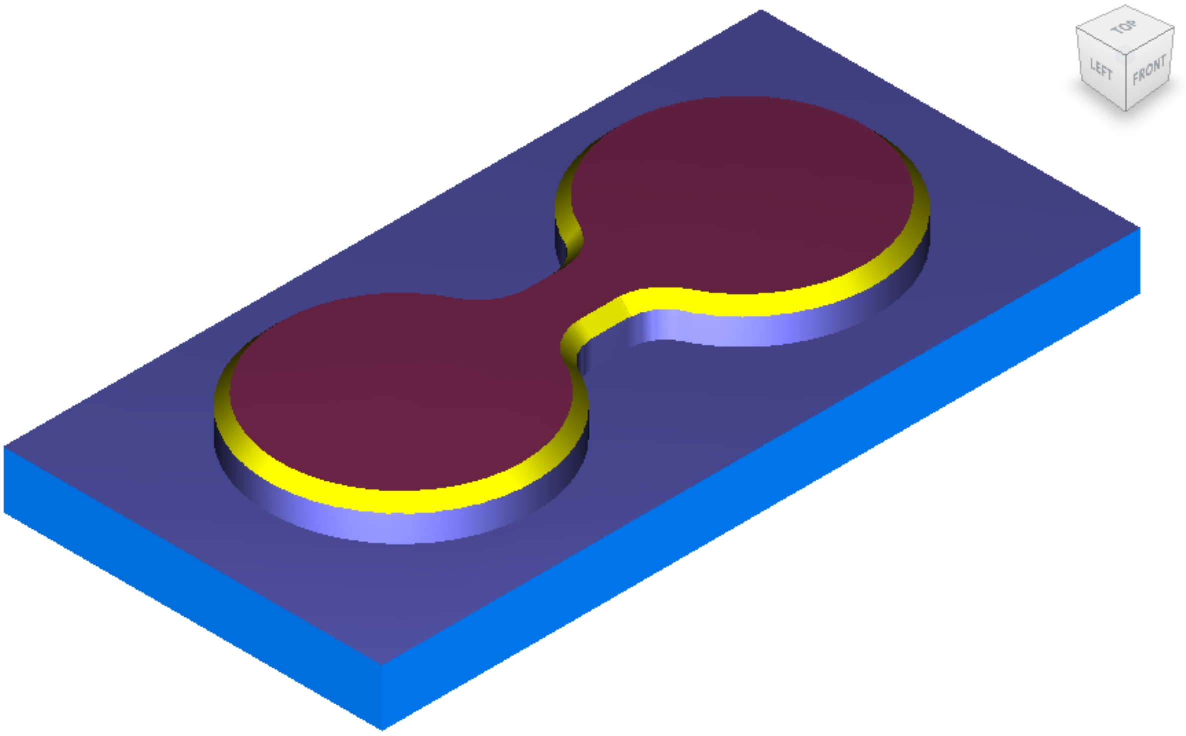




11 Accept the **3mm** radius and press **finish**. Your example should look like the one shown above.

12 Save the Round Example and then undo the round operation.

13 Create a new Feature **Chamfer 3mm.** Then save file as chamfer.



Groove (Simple) (Optional Exercise)

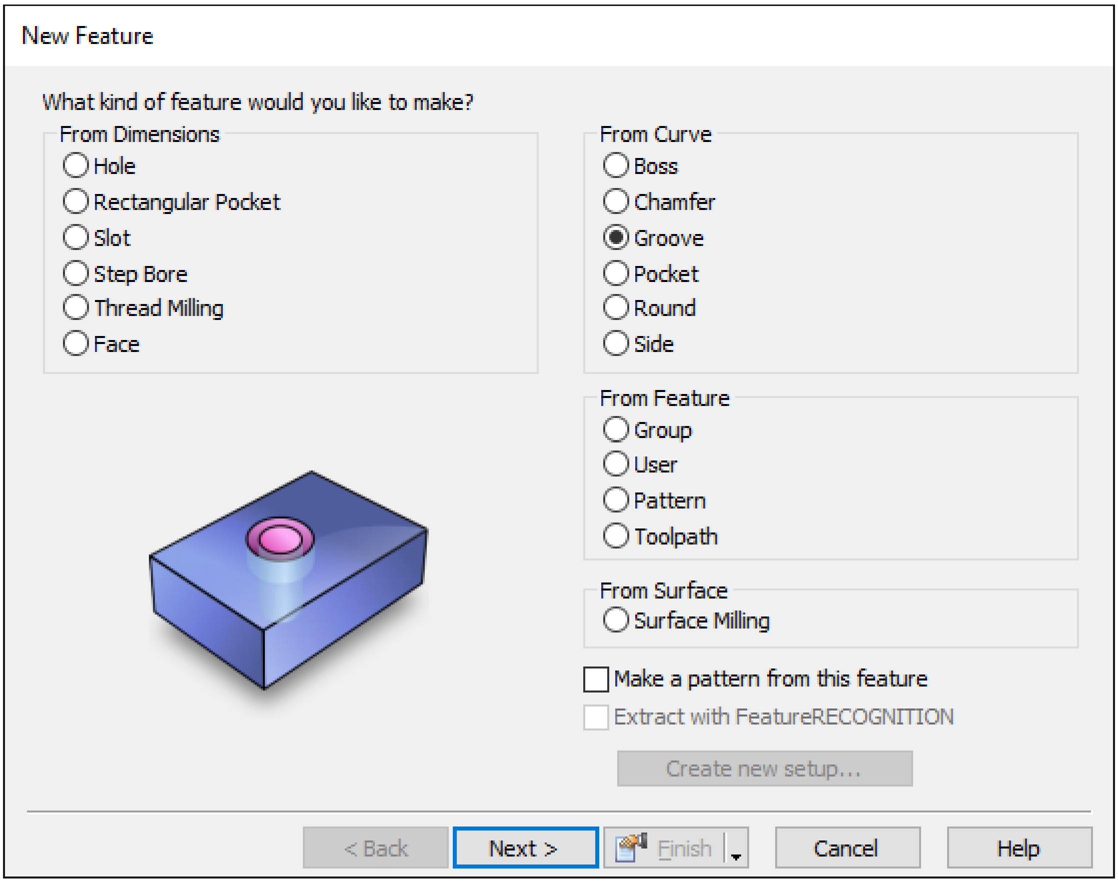
*The **Groove Feature** provides the ability to apply grooves to the face with a flat or ball end mill, to the side using a side cutting milling tool or to perform engraving using a flat end mill, ball end mill or a Bevel/pointed tool.*

1 Open a **New Part Document**. **Millimeters**

2 Create a material Block of **150mm x 150mm x 150mm** Draw a **circle 60mm** Radius at location **X75mm Y75mm**

3 Create a **Curve** from the circle, Use Closed Curve

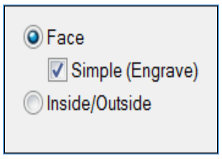
4 Press **CTRL + R** to activate the **New Feature** menu or select **Steps** and then **Feature**, select **Groove** and then **Next.**



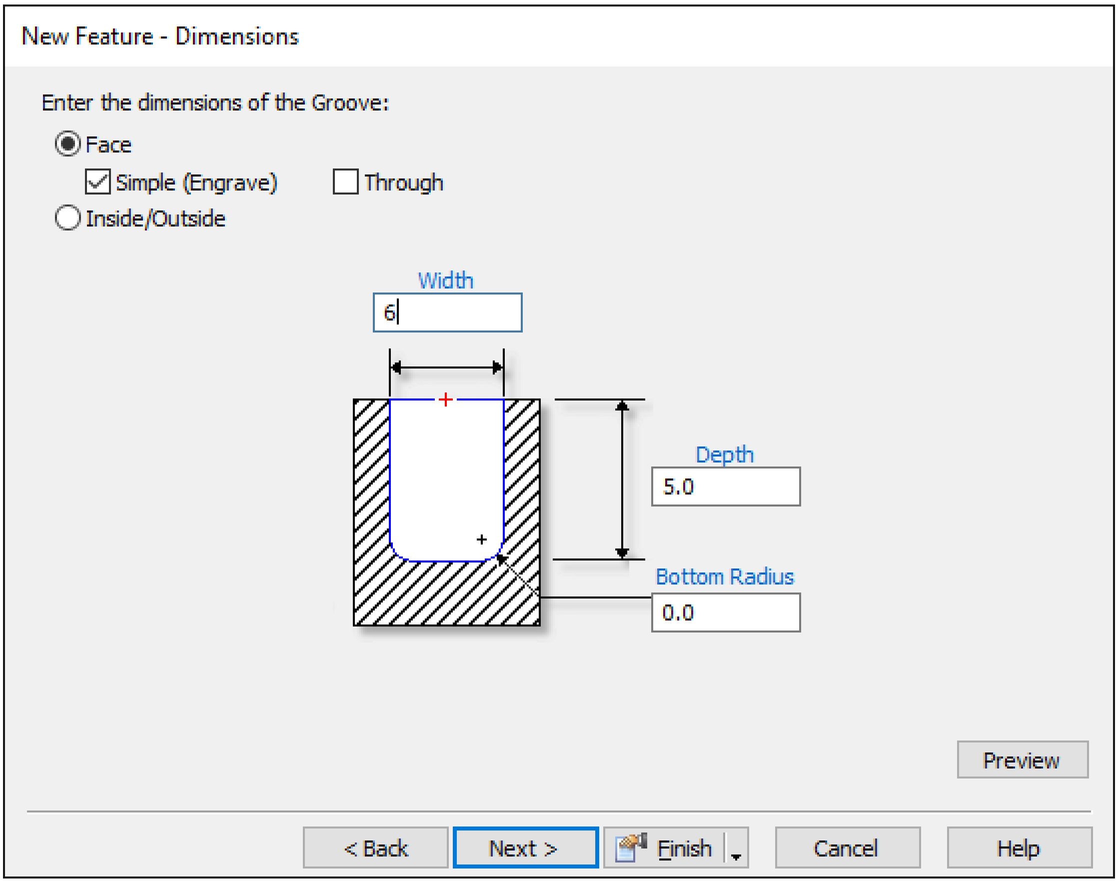
5 Select the **Curve**.

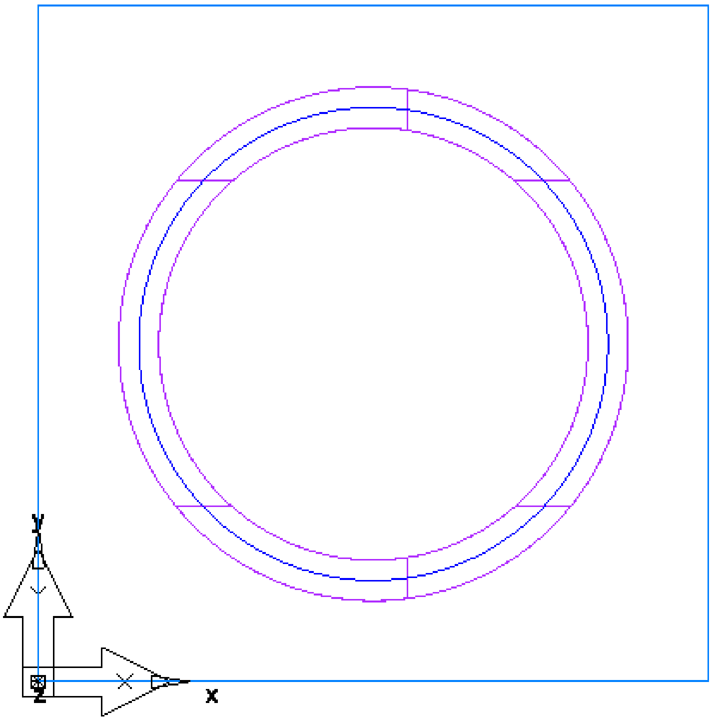
6 **FeatureCAM** will display another menu for location just leave this at zero

7 Leave the selection on **Face/ Simple (Engrave)**



8 Type in the following values **Width 6mm Depth 5mm bottom rad 0 (Zero)**

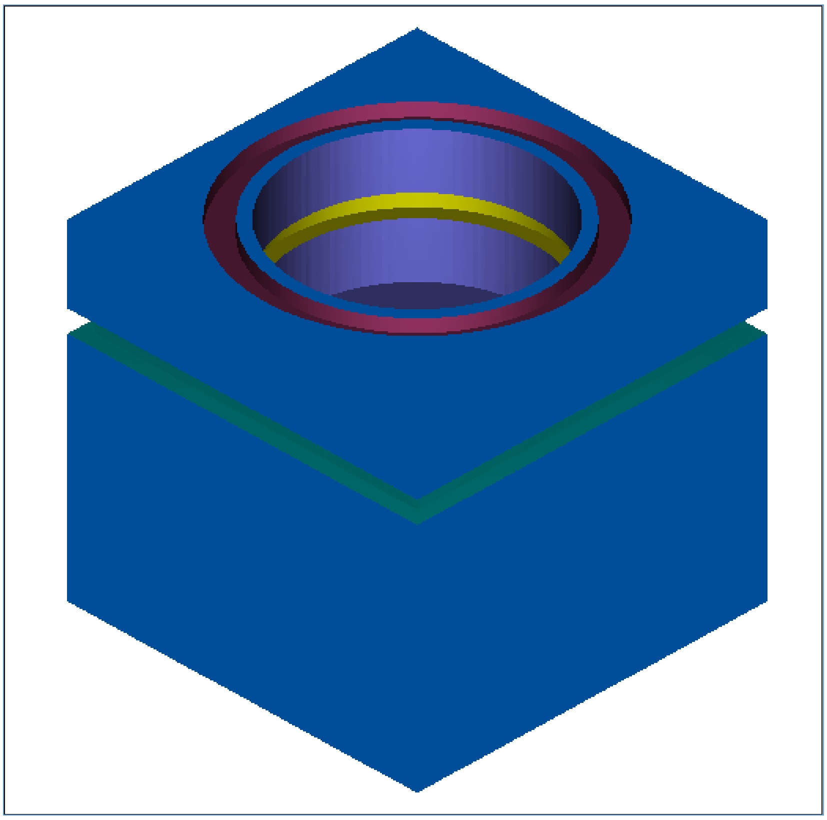




*Notice on the example above that the cutter path is on centre and uses a **6mm** cutter to finish the slot. If you need to rough the slot out first copy and paste the original **Groove Feature** and change the first cutter to **4mm** Diameter.*

Groove (Simple unchecked) (Optional Exercise)

*An **Inside/Outside** groove may be created on the outside of the part, or the inside of vertical walls using a side-cutting tool. When the tool is the same width as the groove, one rough and one finish pass is made. A single finish pass may also be used.*

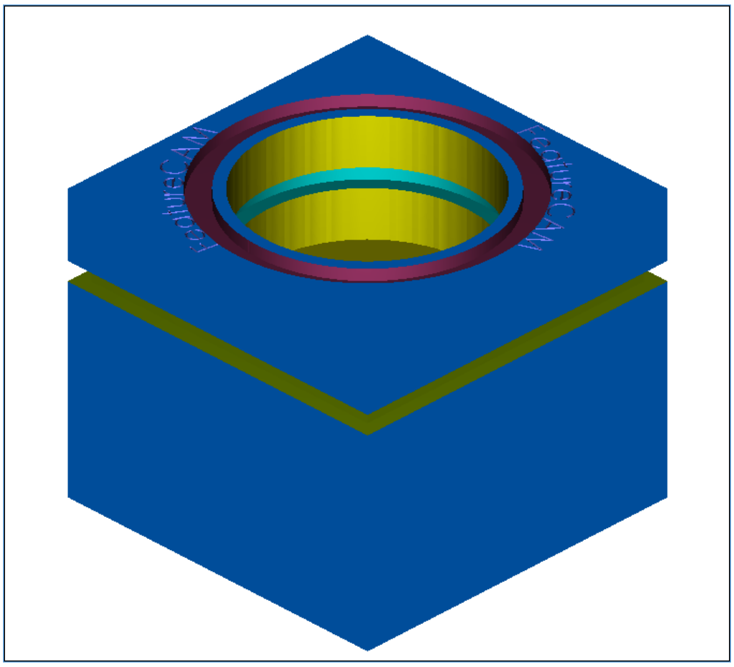


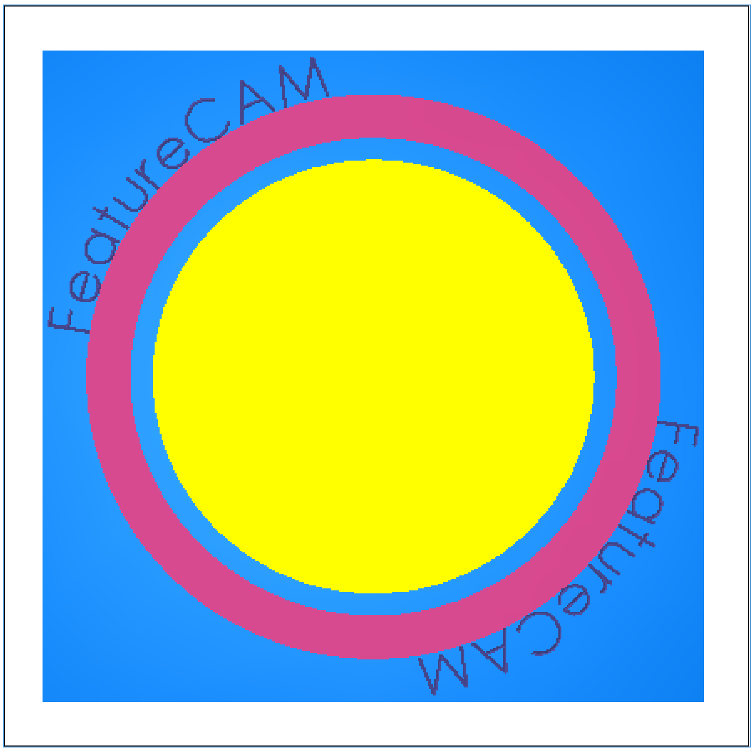
*The example above shows three grooves the first one as illustrated shows an internal groove of **60mm Dia** inside a bore of **50mm Dia 60mm deep**. The internal groove is **Z-30mm** the external groove is **Z-40mm**.*

Engraving (Optional Exercise)

*The engraving text was created using **Curve/Other Methods/Text** using **Machine Tool Gothic** size 24.*

1 Create a curve and align the text to the curve. Copy the Feature and rotate it **180 degrees** using **Translate**. Use **Simple groove** to machine the text and use the following parameters **Width 0.2 depth 0.4**.





Feature from Feature Group Pattern exercise 1

(Optional Exercise)

1 Open a New Part Document. **Millimeters.**

2 Create a material block Width **100mm** Length **125mm** and thickness **30mm.**

3 Create a hole from Dimensions **6mm** Diameter **25mm** deep **1mm** Chamfer.

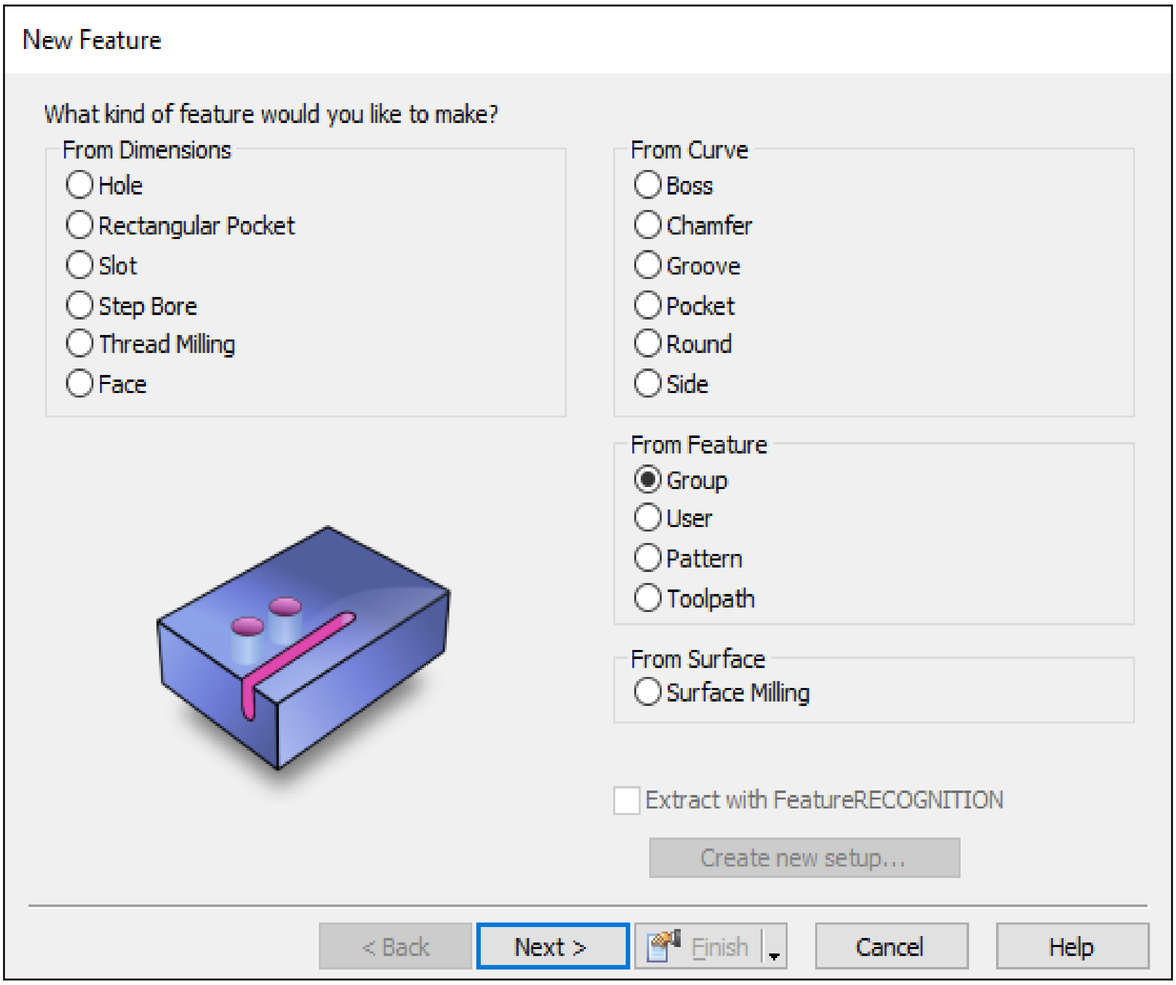
4 Location **X16mm Y16mm**

5 Create a **Rectangular Pocket 30mm** x **30mm 10mm** deep corner Rad **5mm** *chamfer* **1mm.**

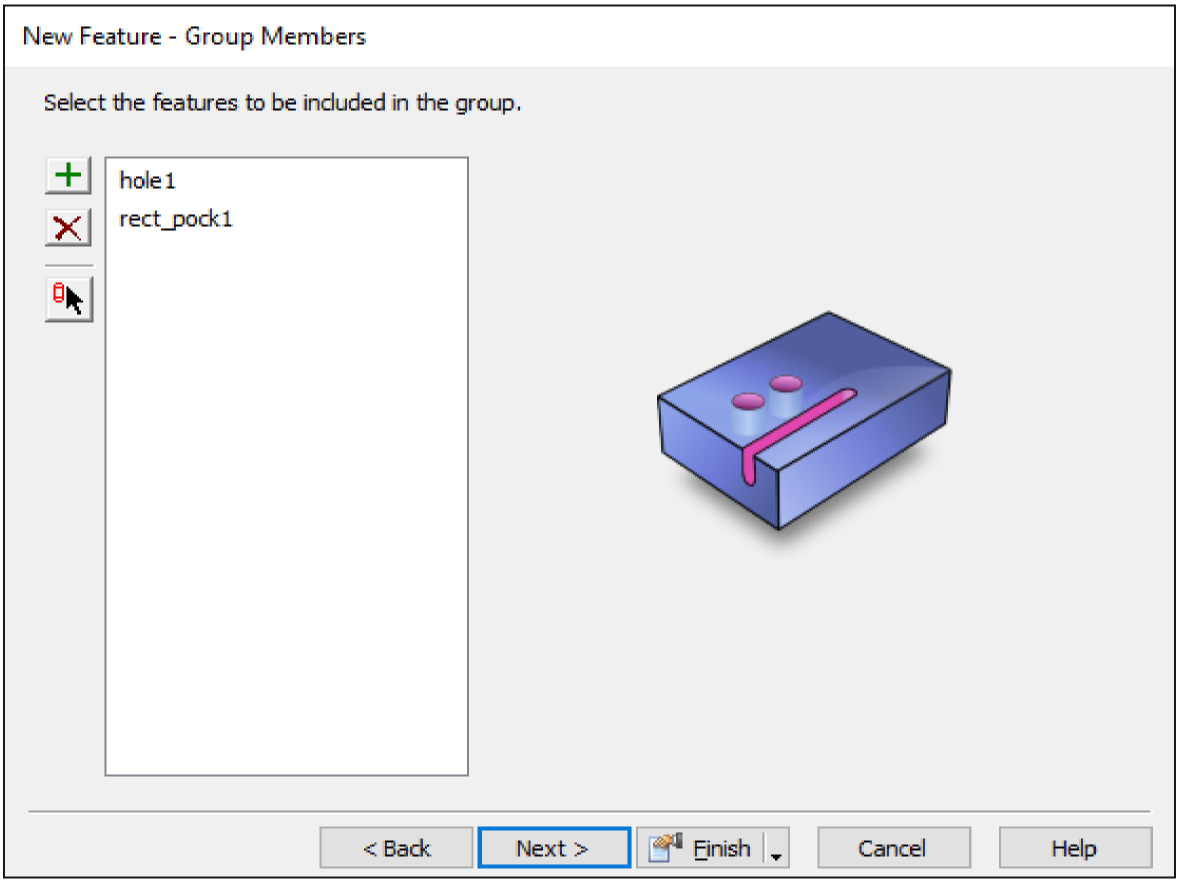
6 Location **X21mm Y21mm.**

7 Examine the features in **setup1**. Currently there is a single hole and a Rectangular Pocket. Say for example you would like to create a pattern by combining the hole & rectangular pocket together. You could create a pattern of a hole, then a pattern of the pocket or you could group the hole and the pocket together and create a pattern of the group.

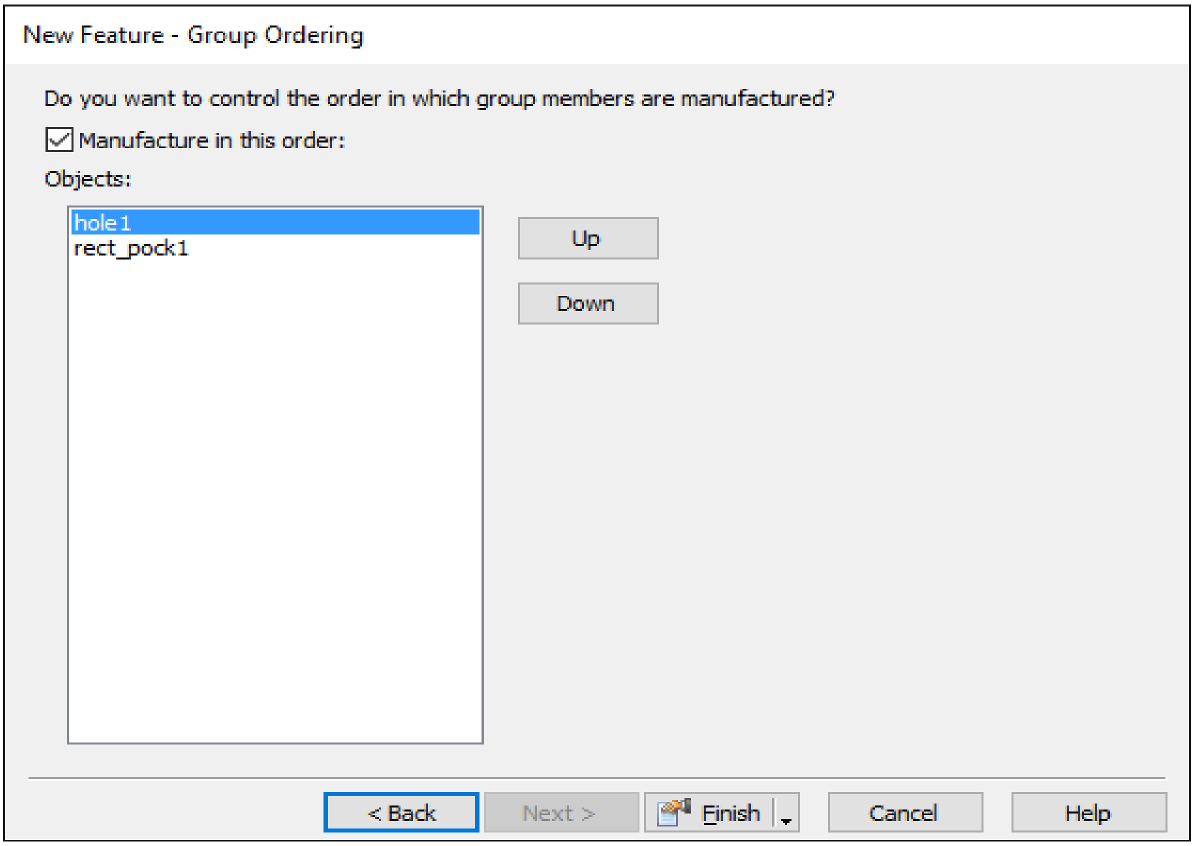
8 Open the **New Feature** Wizard in the **From Feature >Group** select **Group** then click **Next.**



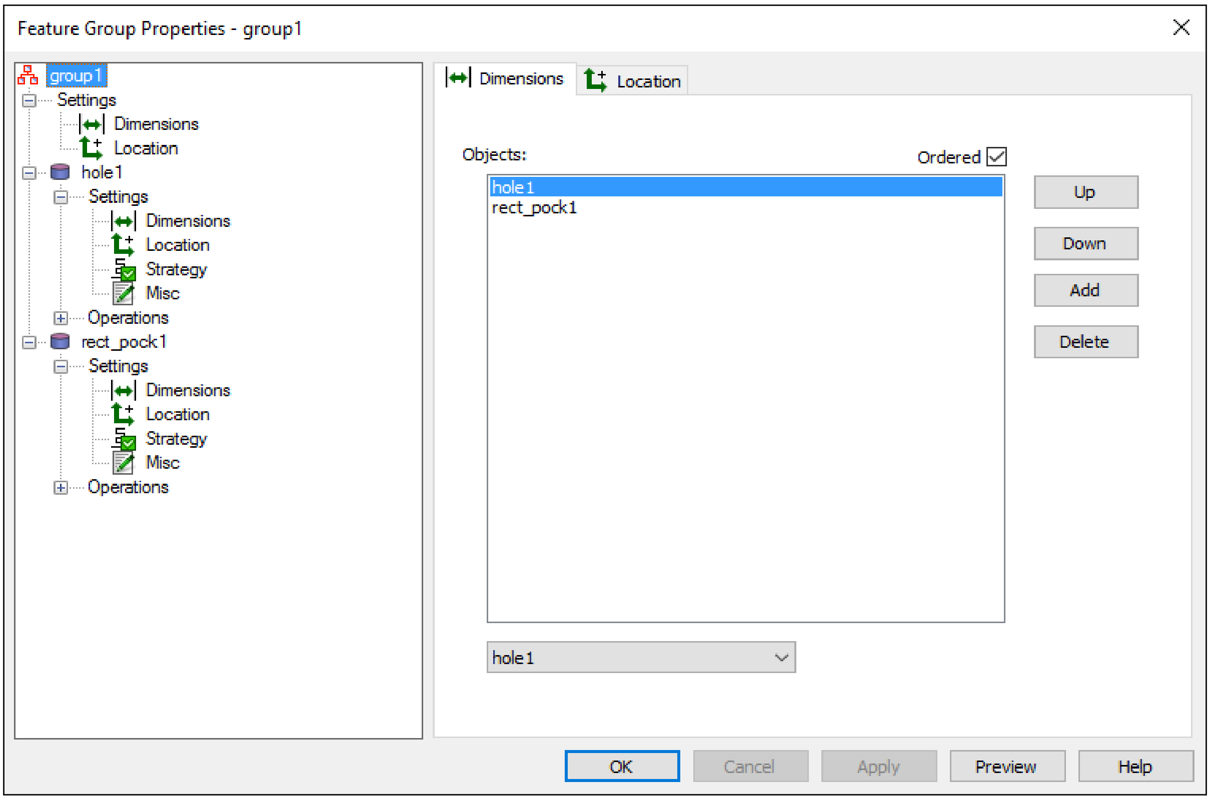
9 Select the **Features to be included in the group**. Then select **Next.**



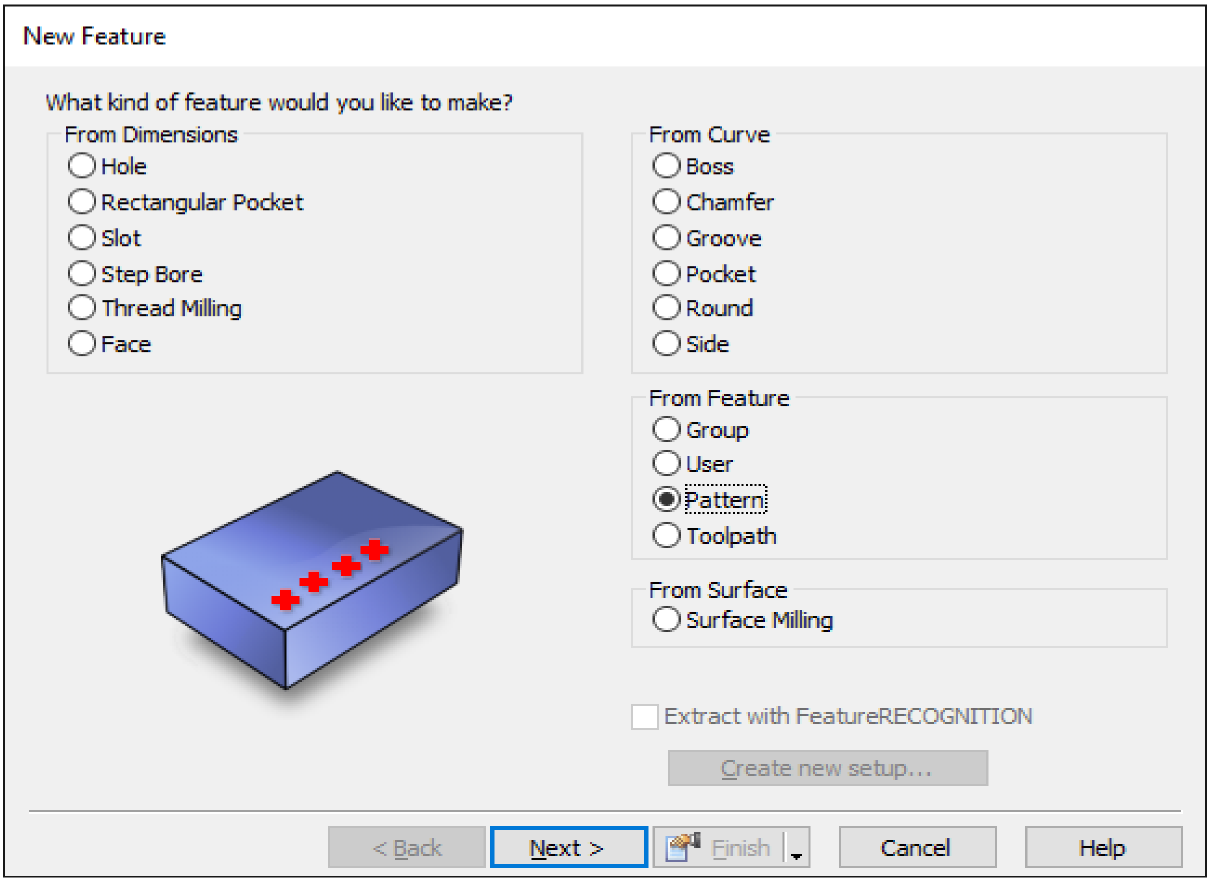
You have the ability to move the order of machining up or down see below.



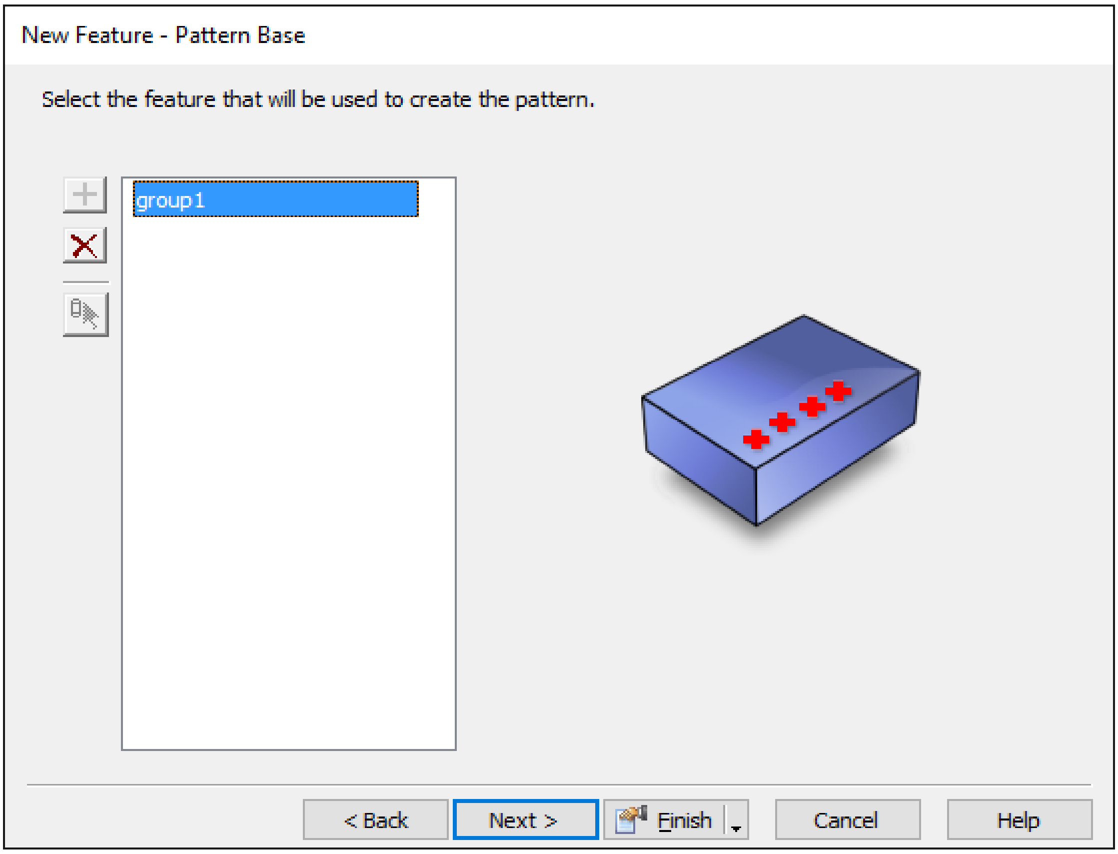
10 Select **Finish**



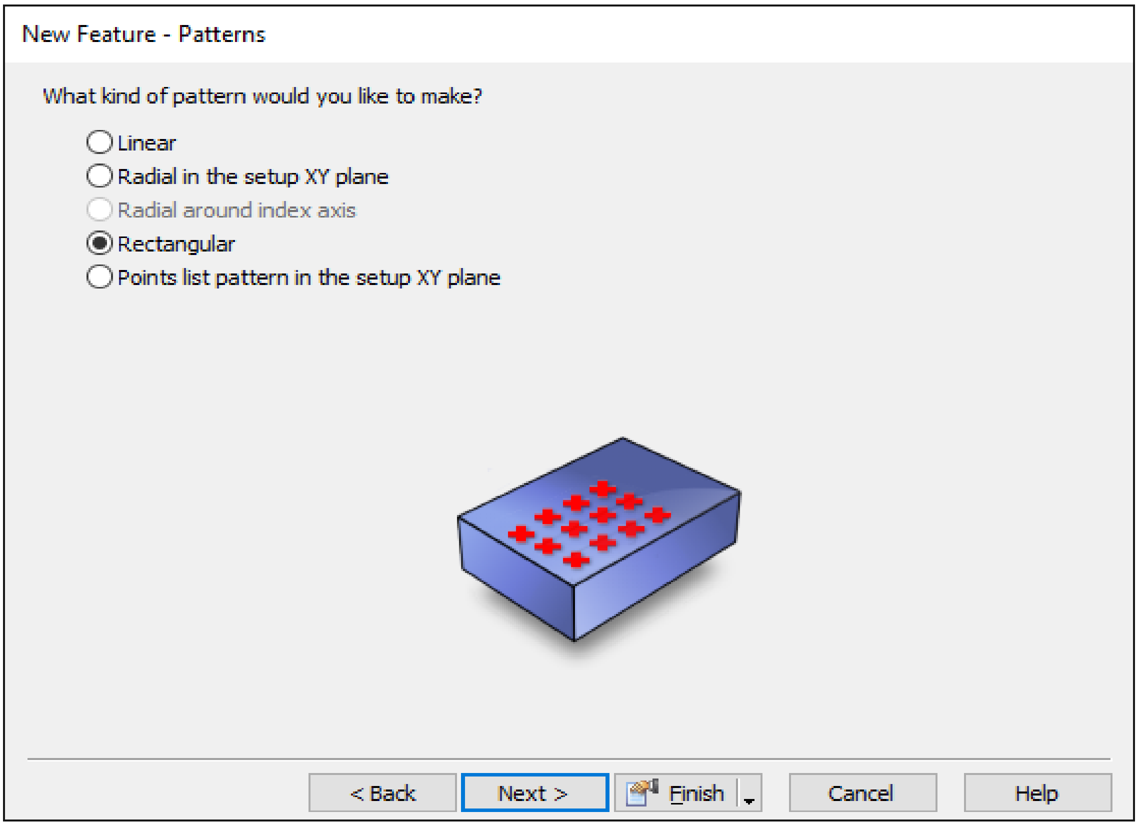
11 Create a new **Pattern** Feature then select **Next.**



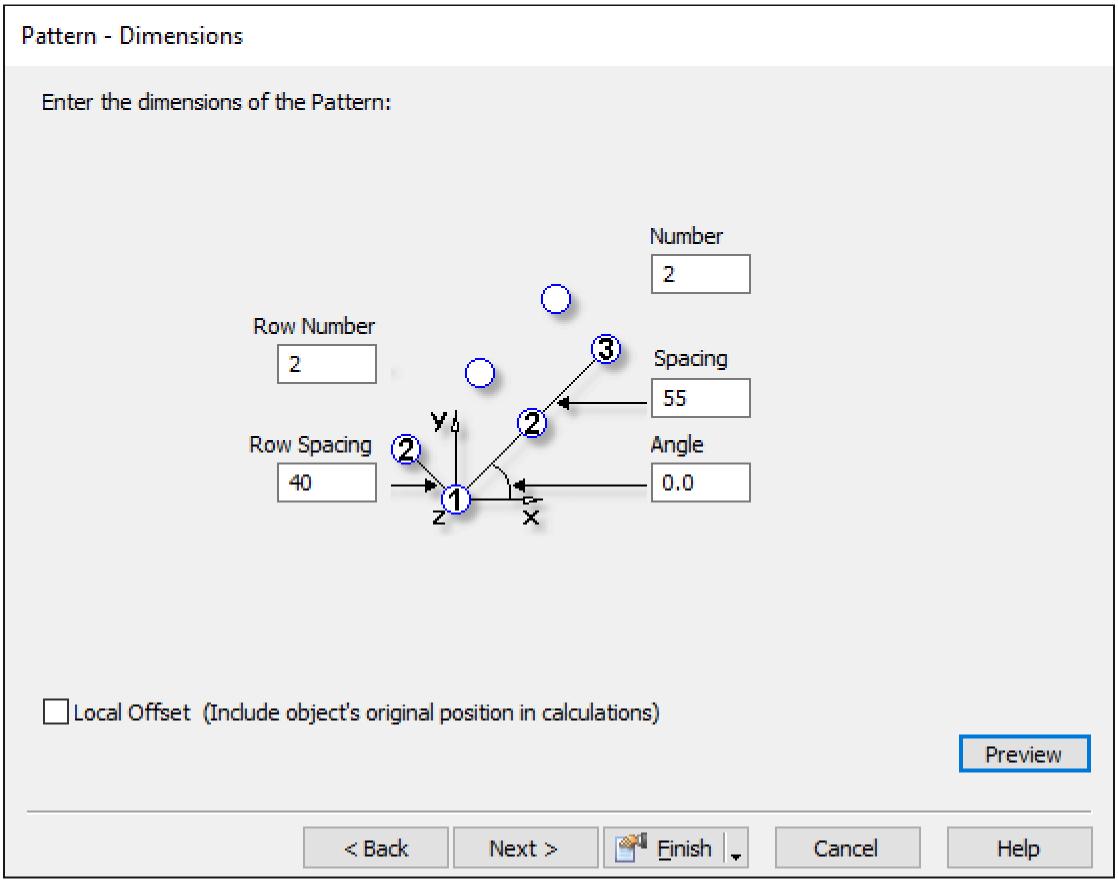
12 Select the **Hole** and **Pocket** this is the Pattern base known as **Group1**

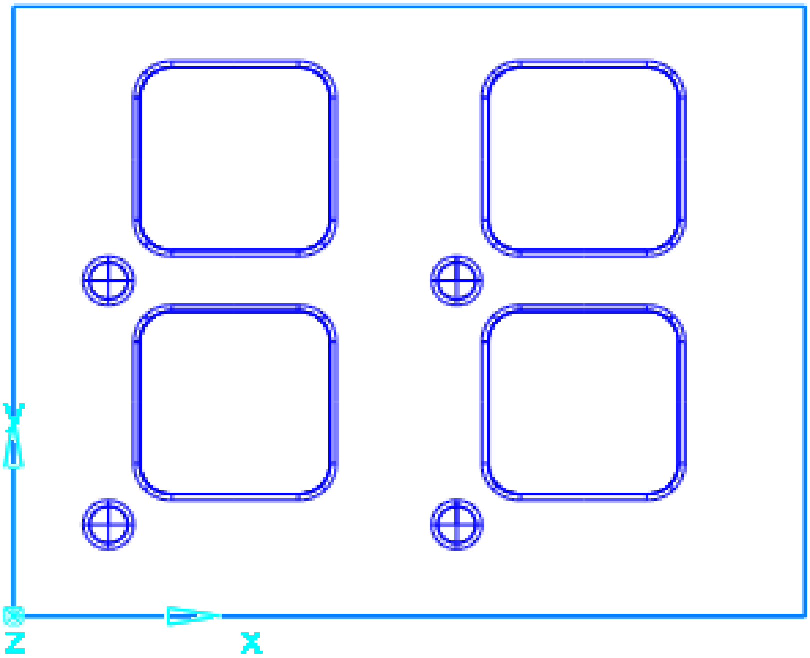


13 Select **Next** What Kind of Pattern would you like to make. Select **Rectangular** and then **Next**.

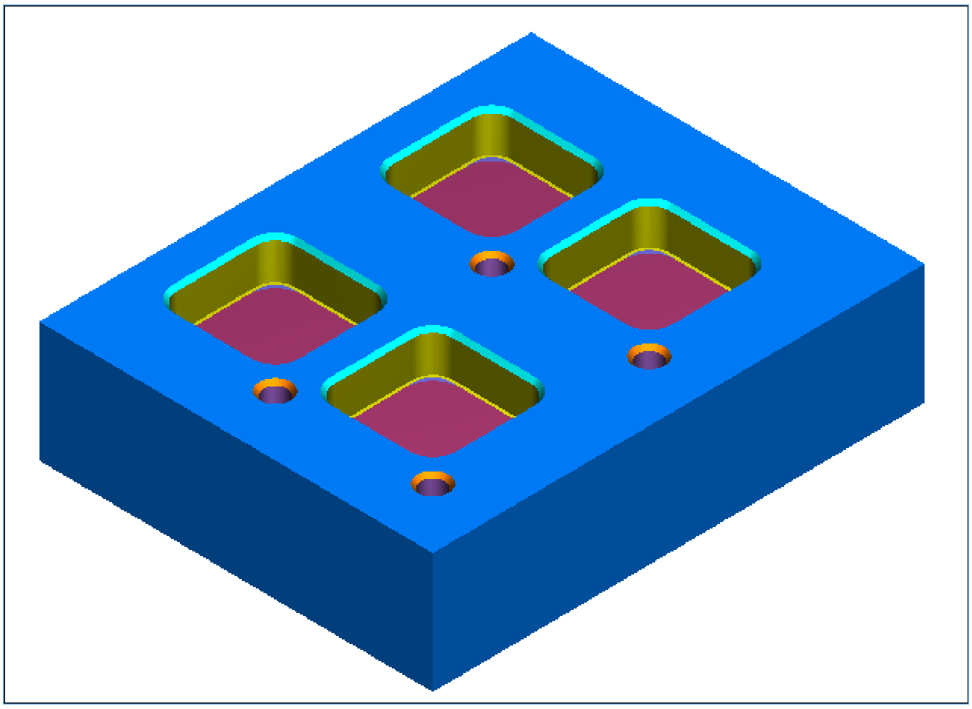


14 Enter the following figures into the Dimensions form.





Select **Finish.** Then run the **3D Simulation.**

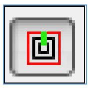


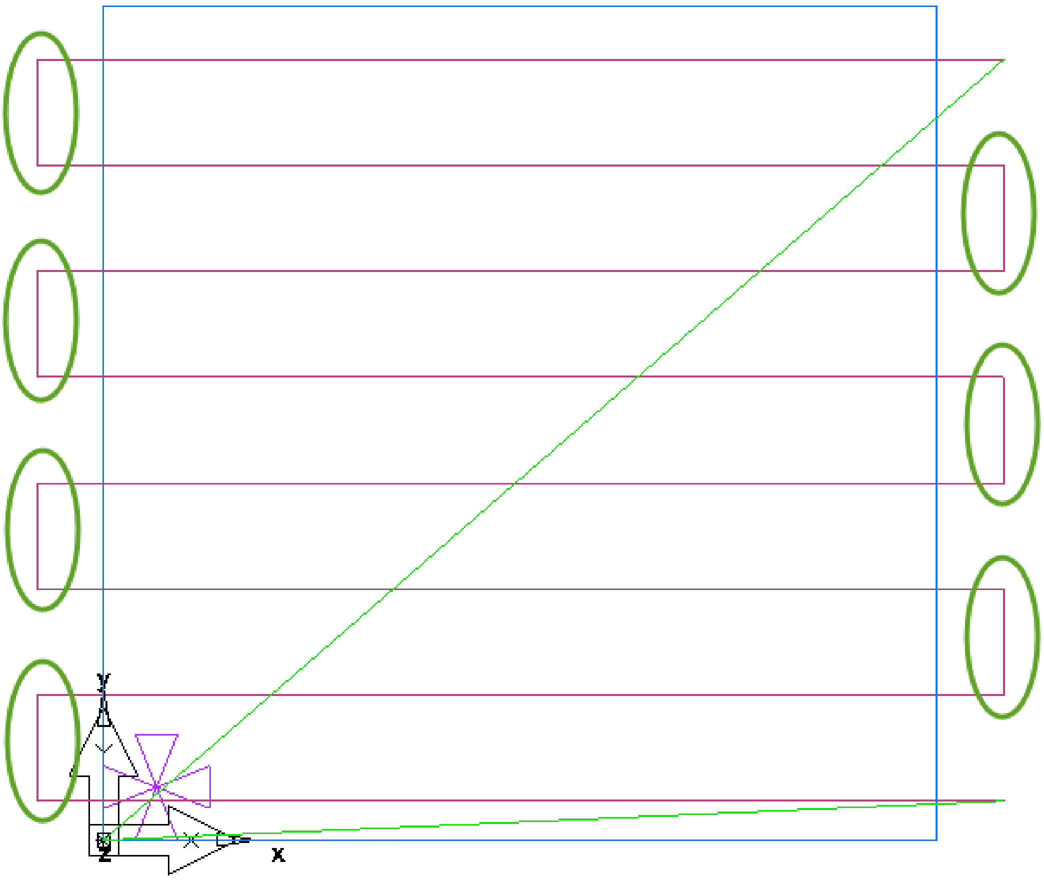
Feature from Feature Exercise 2 (Optional Exercise)

1 From the file menu select **File** then **Open** select **ToolpathEdit.fm** in the examples folder then press **Open**.

*Notice there is only a single face feature in this example located at **Z0***

2 Run a **Centreline Simulation** of the current toolpath. To do this press the

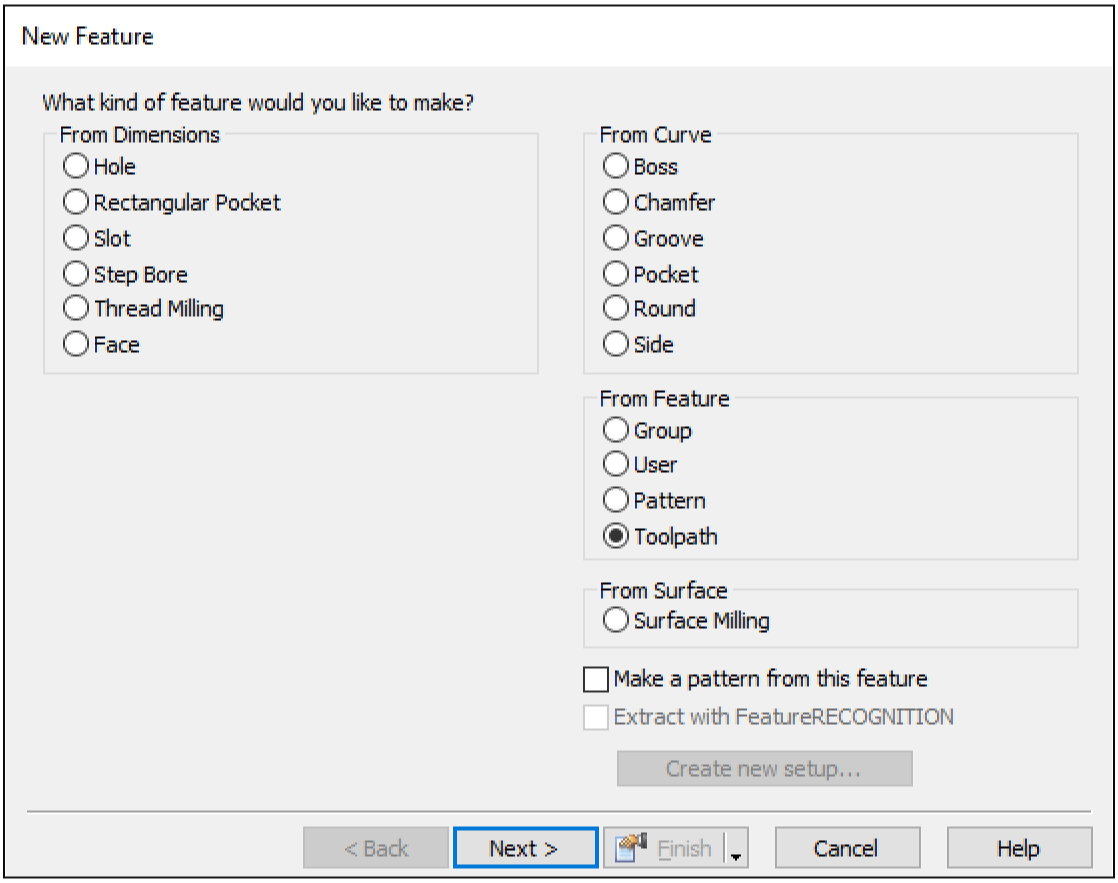
3 **Centreline Simulation** Icon  on the Toolbar then press the **Play** button

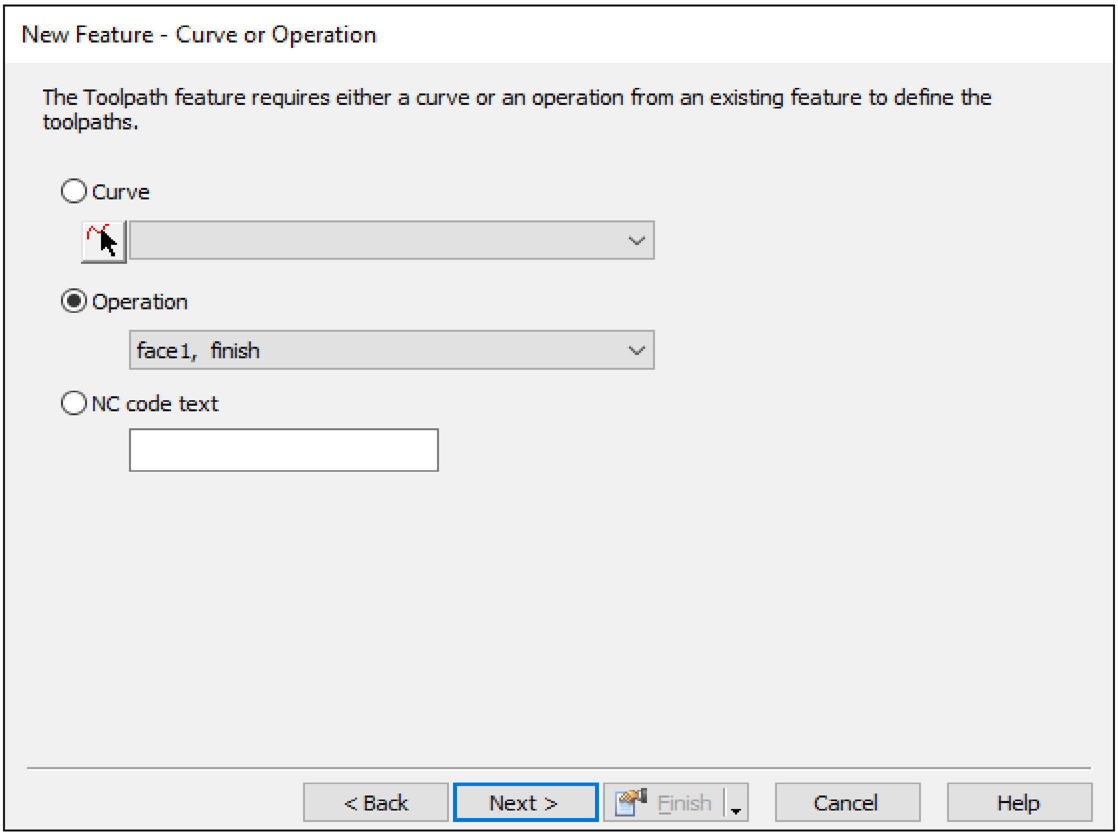


On this facing toolpath there are seven undesirable areas where we would like to change the toolpath. For the toolpath representation there are both feed moves and rapid moves. Pink toolpath segments represent the feed moves while the green segments represent rapid moves. Currently circled in Green are feed moves for the stepover between facing passes. Since the stepover of the facing tool is off the stock it is safe for us to change these to rapid moves.

4 Press **Eject** on the **simulation** toolbar

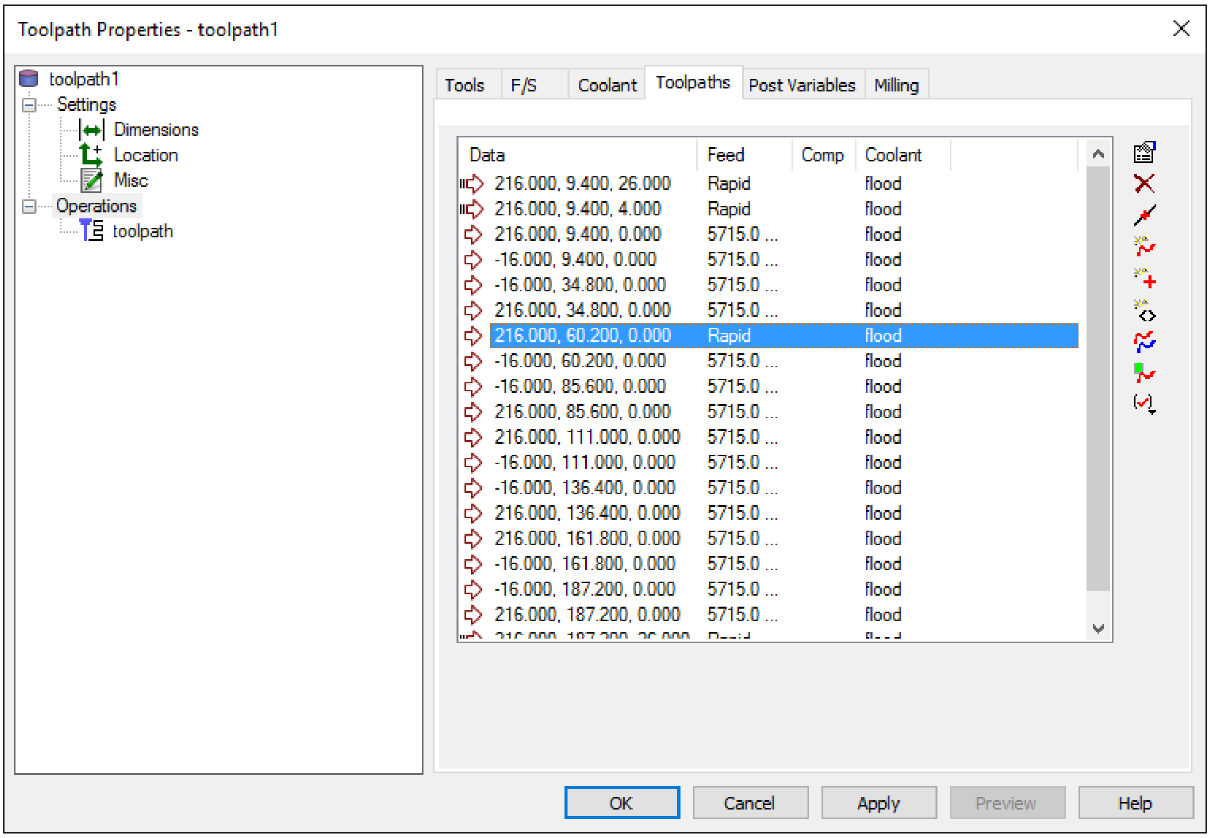
5 Open the **New Feature Wizard** and select Toolpath in the From Feature section then press **Next**.



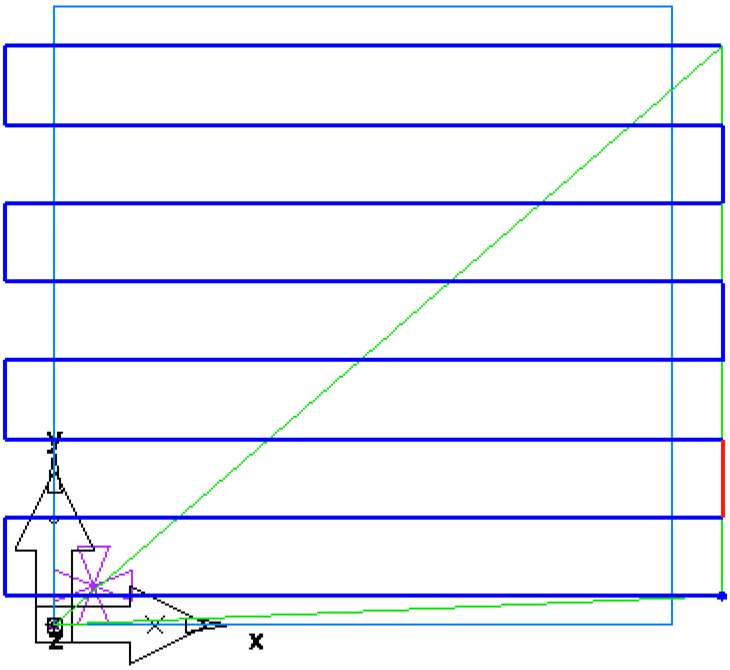


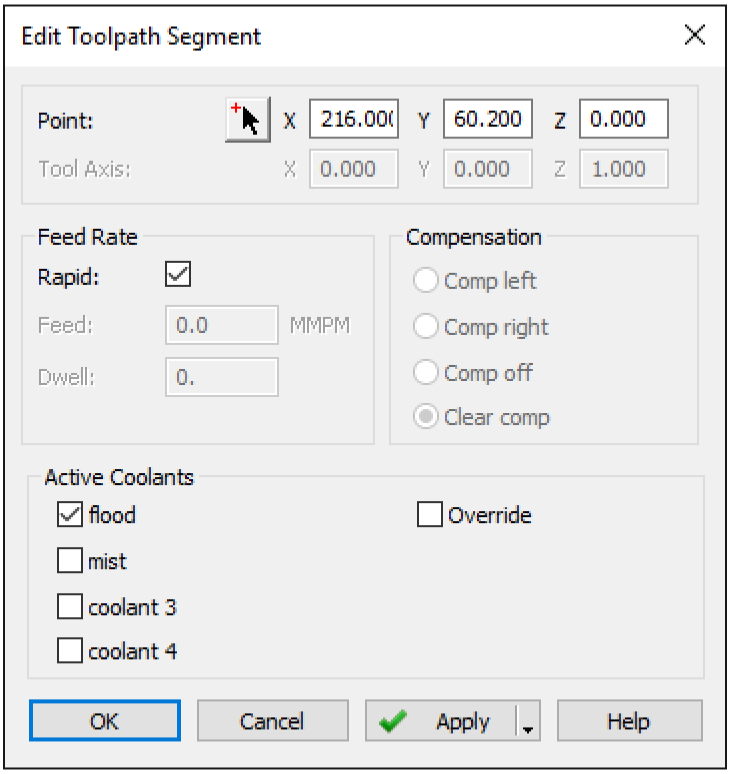
6 Press the Operation radio button select **Face1**, **finish** then press **Next**.

7 Select the first **Y feed move** from the list, and then press the **Edit segment button.**



8 Check the **Rapid checkbox** then press **OK.**

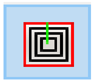


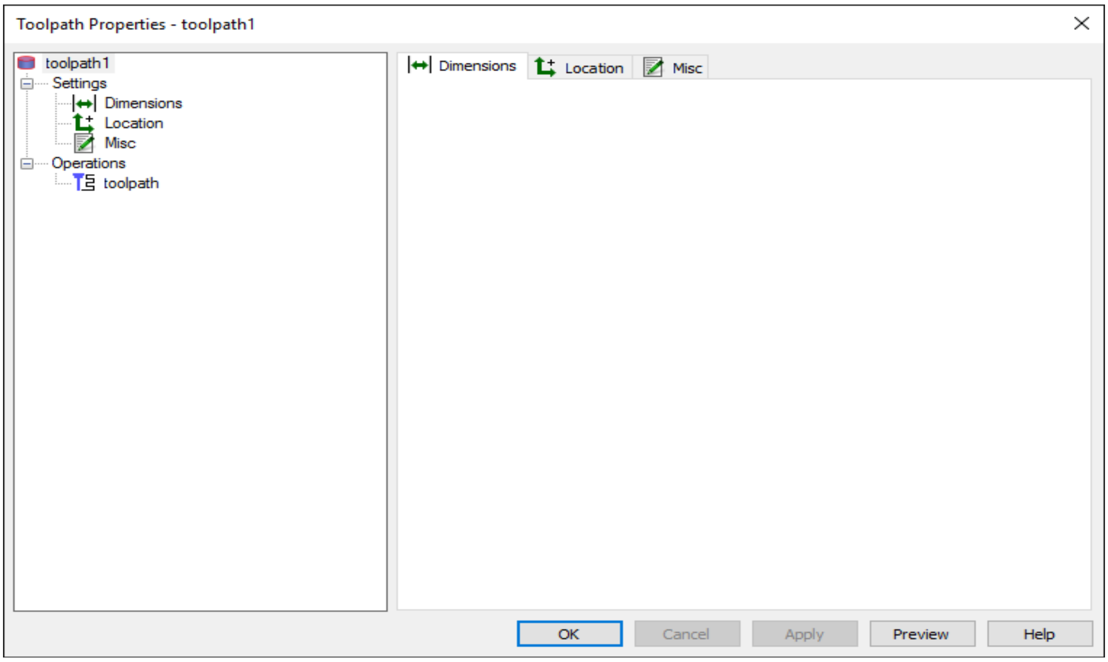


*Locate the remaining Y stepover feed moves and change them to **Rapid** also then press **Finish**.*

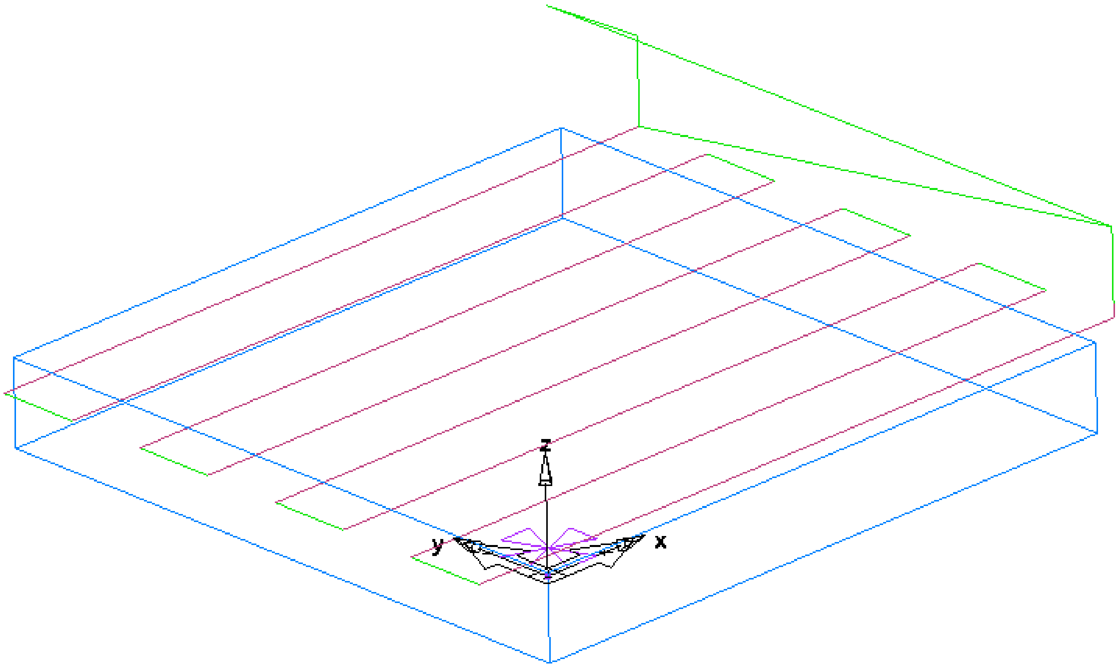
9 Observe the Features in the **Part View**. Now there should be a **Facing Feature** and a toolpath feature. The toolpath feature is simply a copy of the original features toolpaths with the edited move.

10 Uncheck **face 1** in the **Part View** (so there are not two facing operations)

11 Press **Play** on the **Centreline Simulation**  and observe the toolpath.



12 Now notice that the Stepover moves have been converted to **Rapid** moves that will reduce the cutting time of the part.



Contents

Advanced 2.5D Machining

Introduction

*In **2.5D** milling there are more Advanced Features than just milling and drilling, such as **X cross section**, **Work Planes**, **4axis**, etc. The user will learn how tapered walls and other shapes can be machined to emulate **3D milling surfaces** using a ball end mill, but are still **2.5D** machining.*

Cross Section (X section)

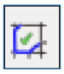
- Side, Pocket and Boss Features.
- When creating either a **Boss**, **Pocket** or **Side** features there is an option of selecting **X section**. This allows you to apply a shape, other than an angle, to the walls. **FeatureCAM** will even select a ball end mill where necessary. Surface or Solid models are **NOT** required, although the end result is similar to a **3D finished surface.**

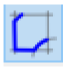
Rules for an X-Section Curve

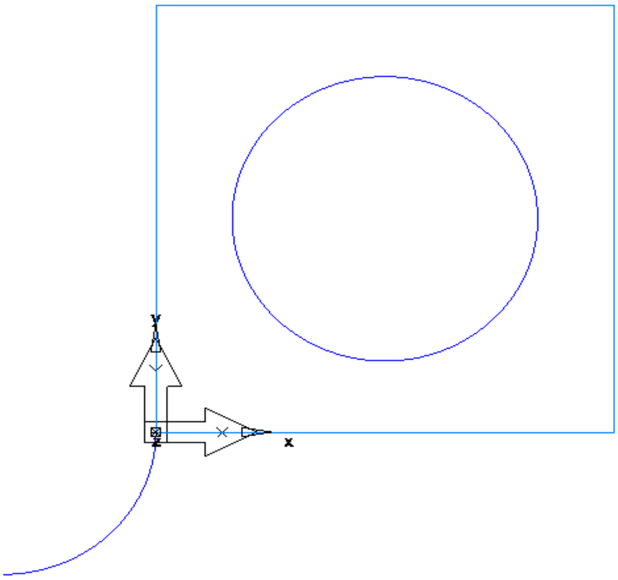
- When using geometry for your cross section shape it does not have to be round. It can be other shapes as well.
- Once you start drawing away from **XYZ=0** you may not draw back towards the start point in the X or Y direction. This would create an **undercut**.
- A straight line in the X or Y is ok as long as the end point continues away from **XY=0**. It must also remain planar.
- **Z elevation must stay at Z = 0**
- By drawing the shape, starting in the illustrated position **(XYZ=0)**, it will attach to the top of the pocket and form the pocket walls identical to the X-section curve.
- A Boss X-Section is the same, but if there are multiple bosses, and the bosses are **NOT** the same height, X Section **CANNOT** be used.

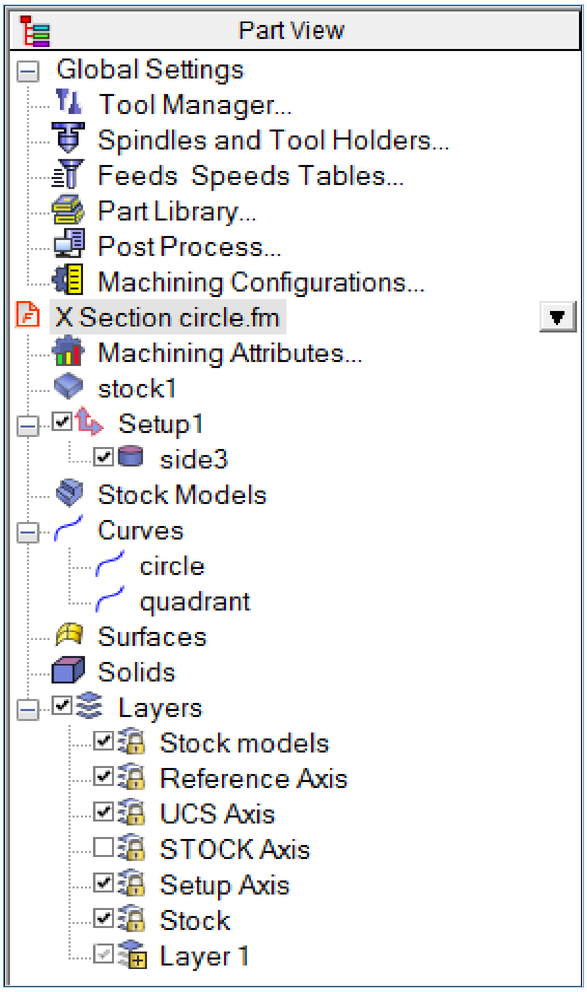
Cross Section (X section) (Optional Exercise)

1. Create a new document, Milling Setup, and Millimetres then Select Block and use Stock Dimensions of **150mm** x **150mm** x **150mm**. Location **X0,Y0,Z0**
2. Select Top View or Ctrl + 5
3. Create a Pocket by drawing a **50mm radius circle**, centred on **Top View** of the XY plane of the stock. Location **X75mm Y75mm**.
4. For use in Cross **(X Section)**, draw a **50mm** radius circle, using **Centre, Radius** from the **Geometry** toolbar. Enter **X-50.0mm** and **Y0**. Using geometry, draw a **Horizontal** and **Vertical** line through the **Centre** of this new circle. The lines will be used for trimming. Trim the geometry until you have the lower right quadrant of the circle remaining as shown.

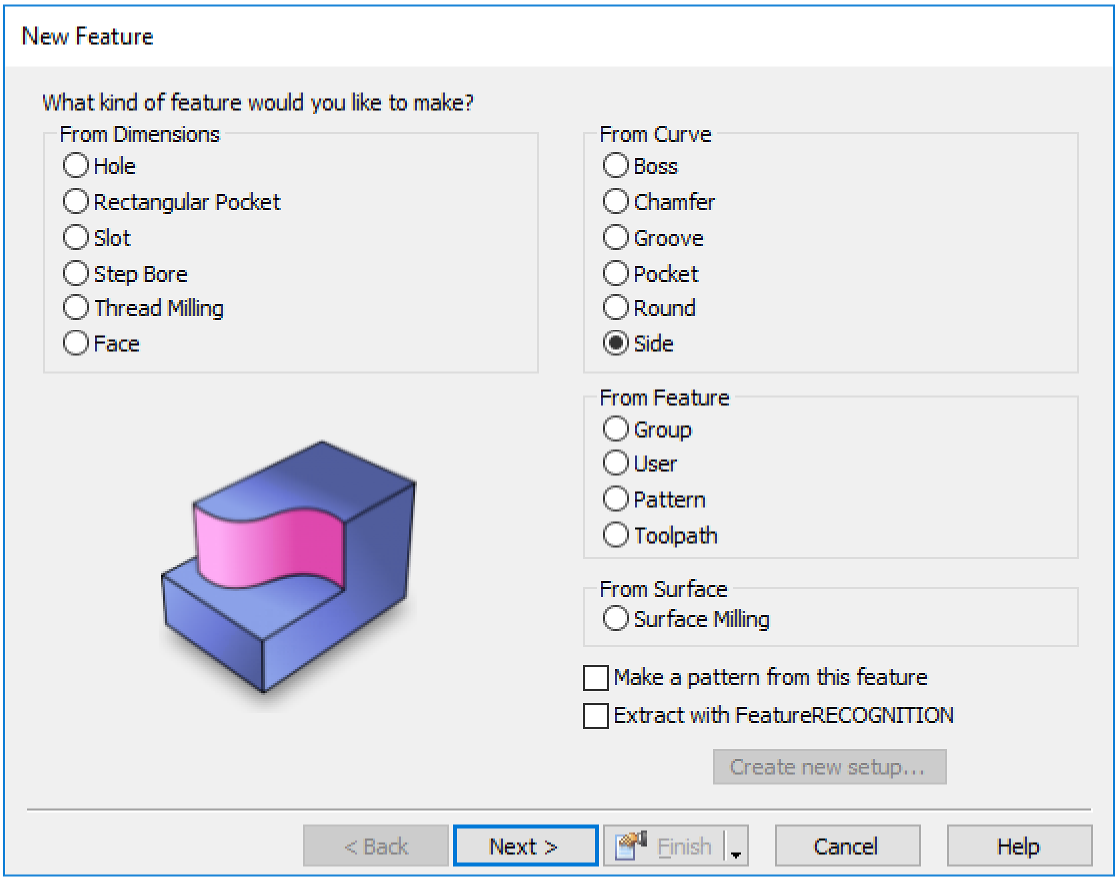
5 Chain both pieces of geometry. Use **Closed Curve**  chaining for the **Circle**

6 And Rename this to Circle in PartView. Use Pick Pieces  for the Quadrant and Rename the Curve to Quadrant.



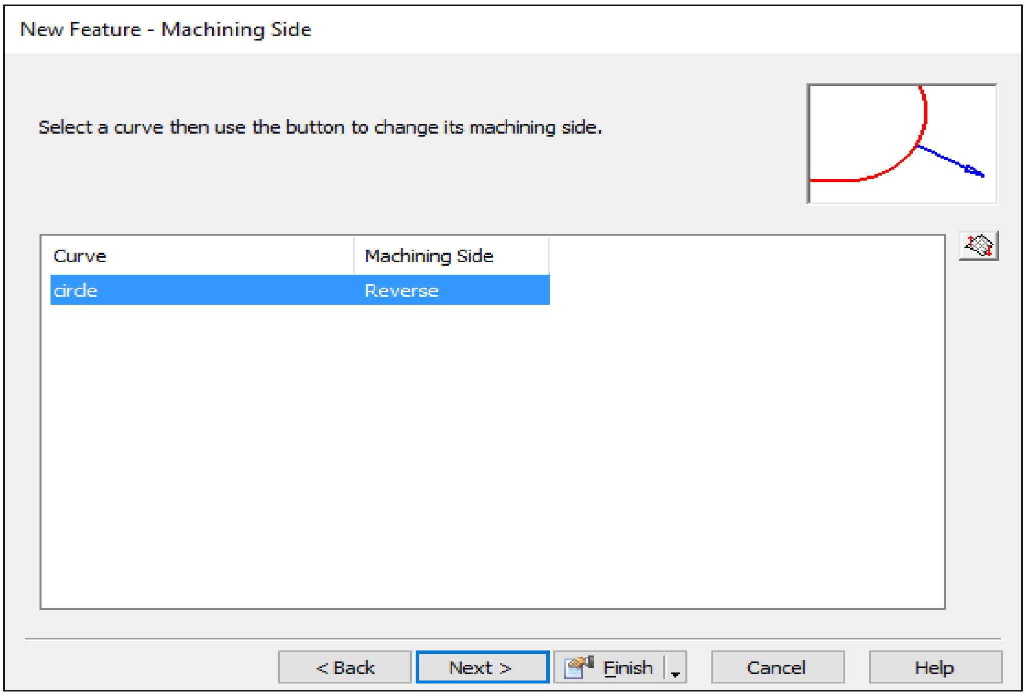


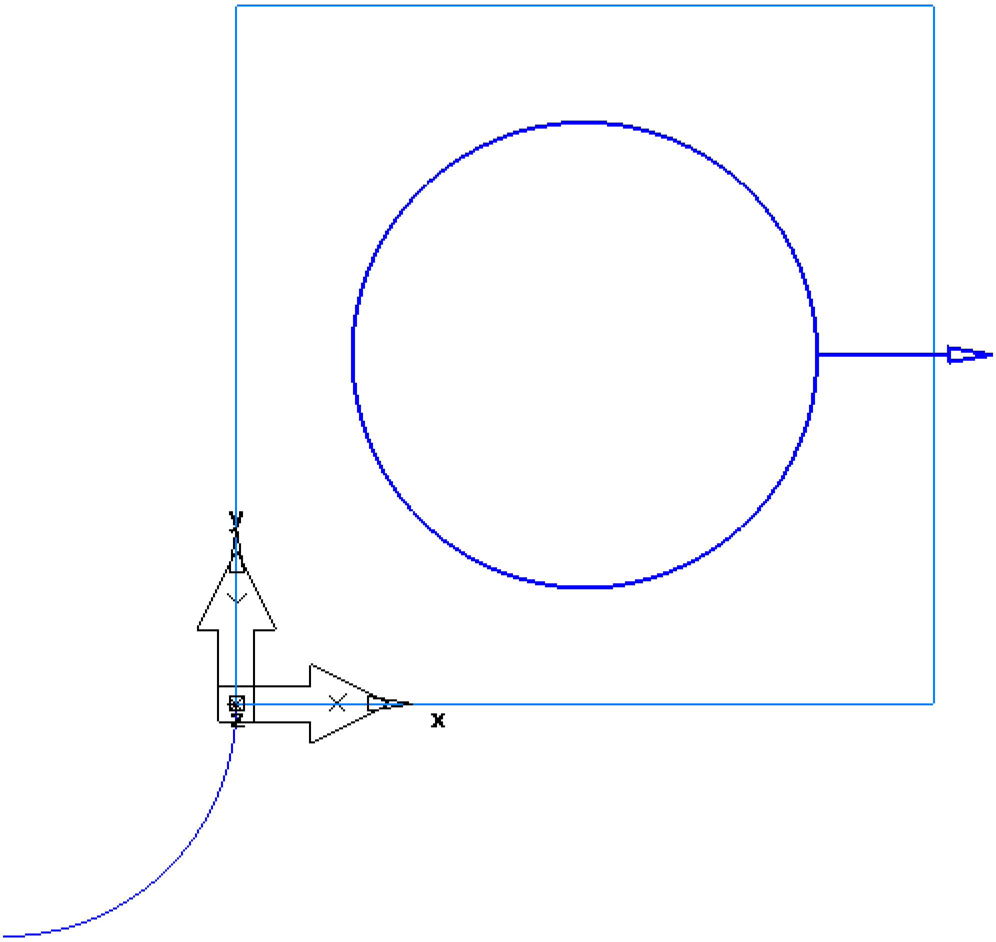
7 Press **Feature** in the **Steps** menu or press **Ctrl** + **R** to activate a **New Feature** Menu, select **Side** and then **Next.**

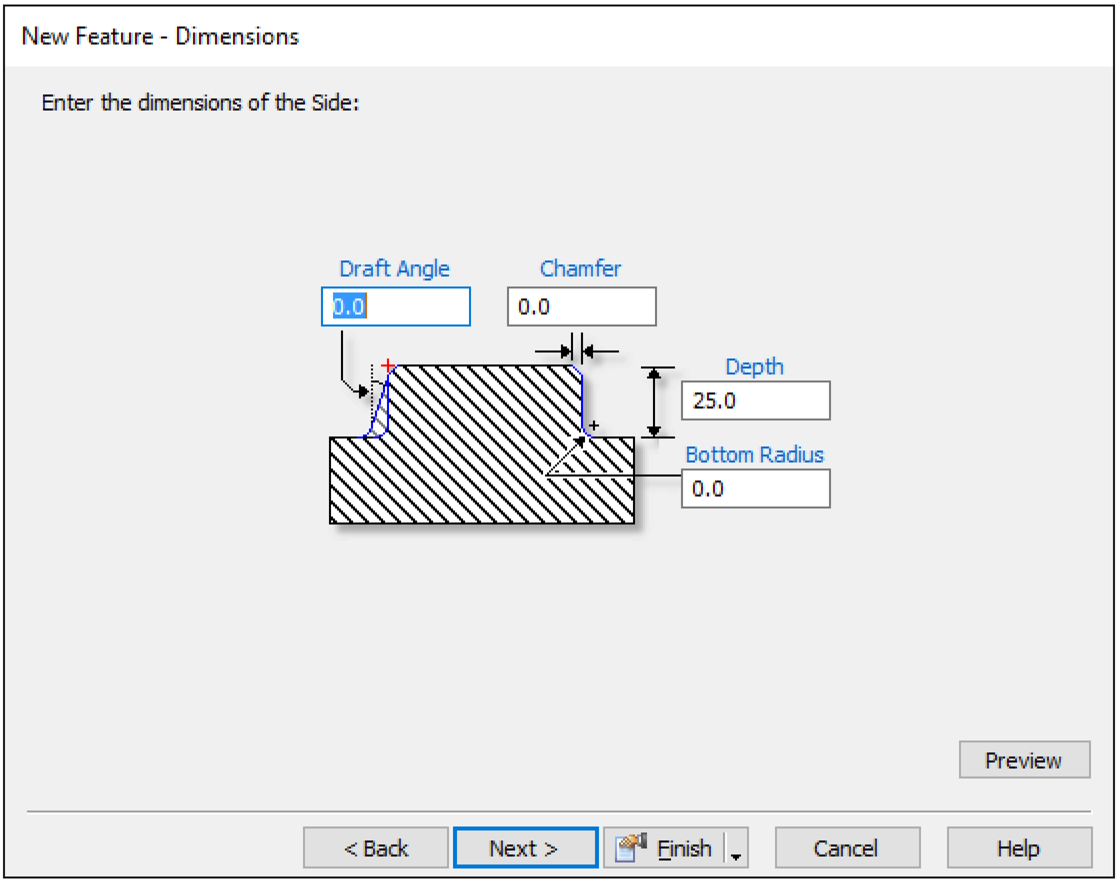


8 Select the **Circle Curve** first. Then **Next.** If you have selected **Side** make sure the arrow is pointing to the Outside. If you need to change the direction select

9 The **Switch Machining Side** Icon  then Select **Next** twice.

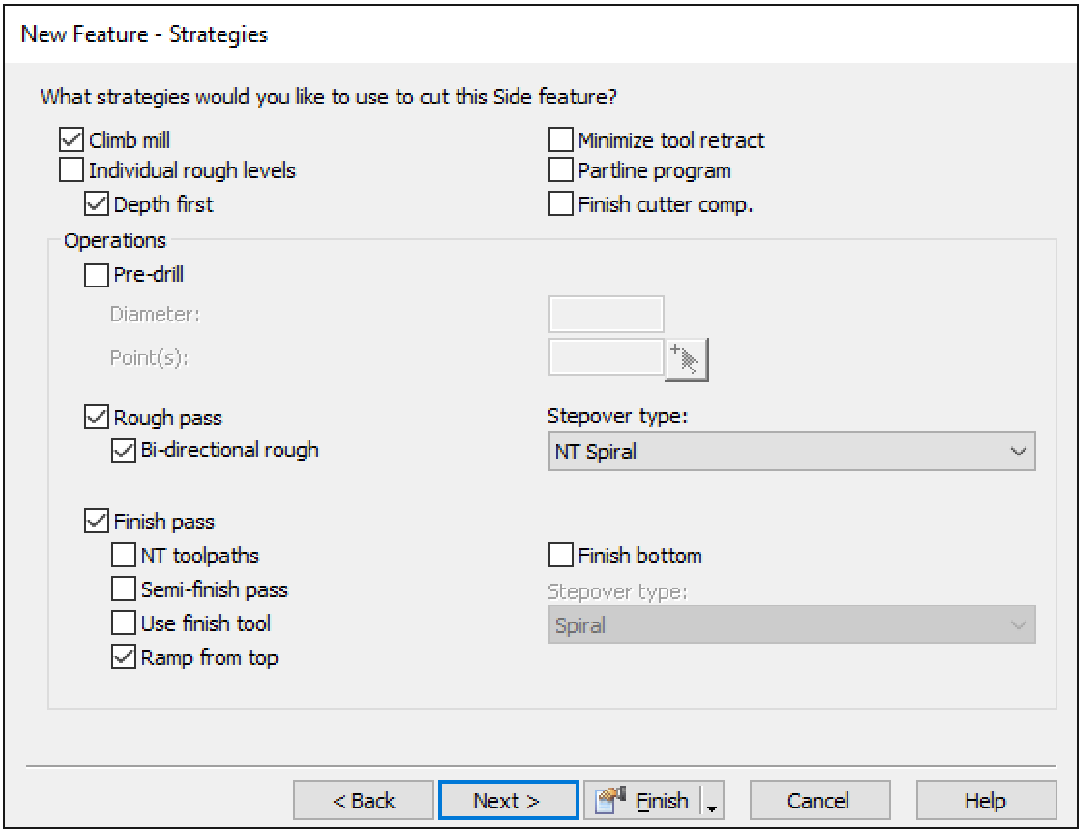



Y
X
Z

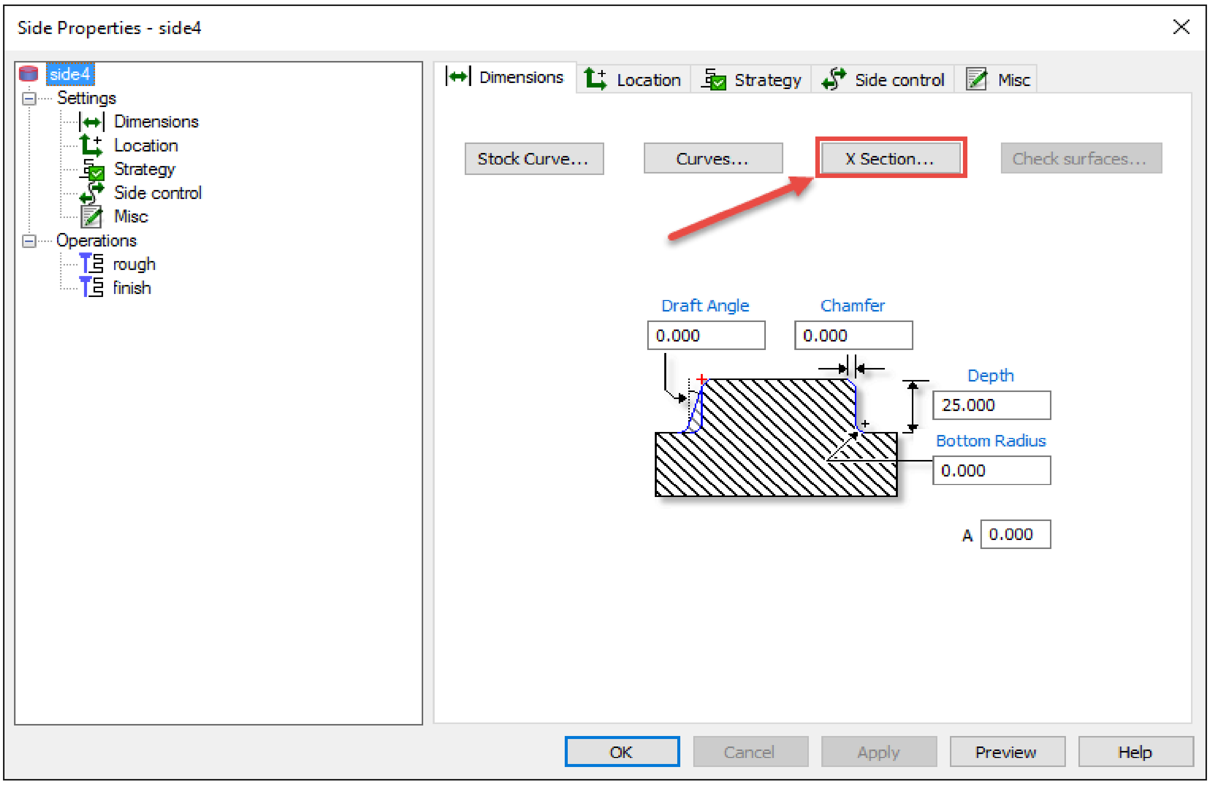

New Feature - Dimensions
Enter the dimensions of the Side:
Draft Angle
0.0
Chamfer
0.0
Depth
25.0
Bottom Radius
0.0
Preview
< Back
Next >
Finish
Cancel
Help

10 Enter **25mm** for the depth. Then **Next**.

11 You will be presented with the **New Feature - Strategies menu**. Select **NT Spiral**. **NT** stands for **New Technology** and uses the **PowerMill** machining Algorithms.

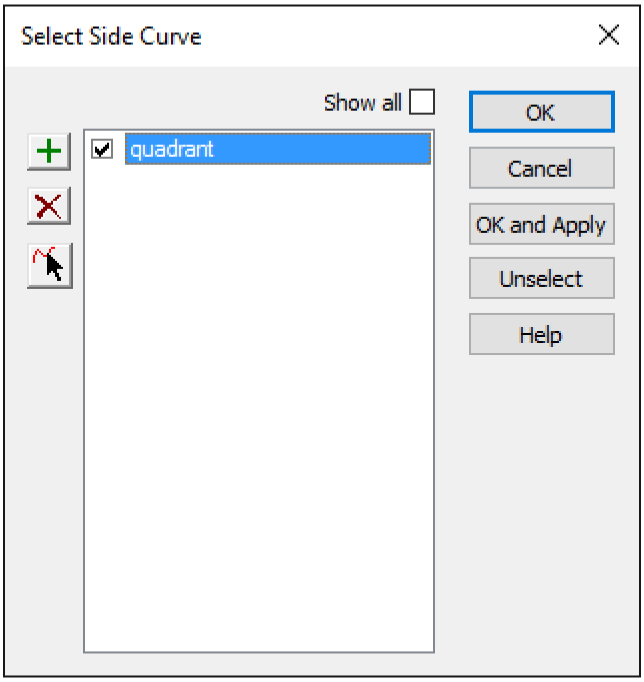


12 Select **Finish** and the following menu will appear.

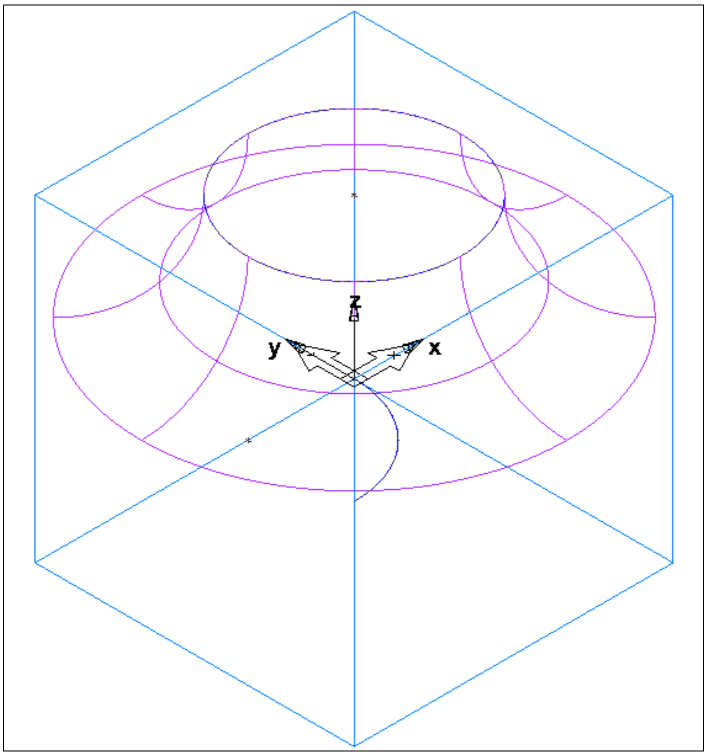


13 Select **X Section.**

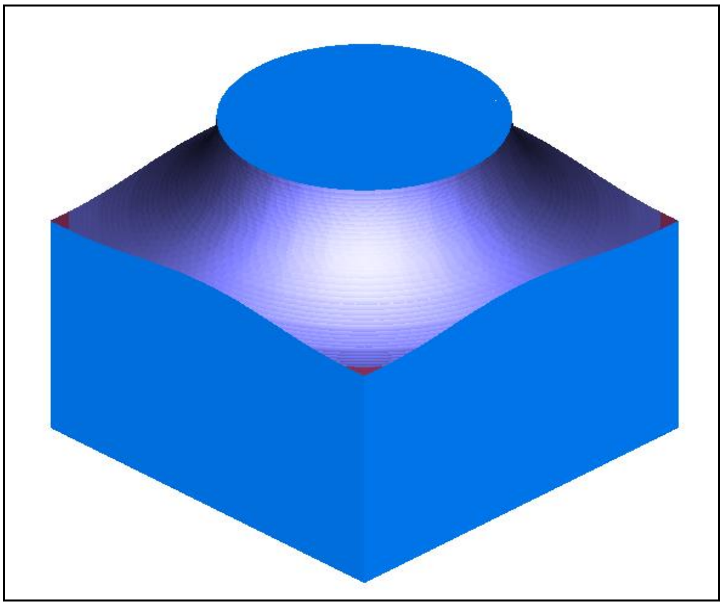
14 You will be presented with another form select **X-Section** and select the **Quadrant** Curve.



15 Select **OK** and **Apply.**

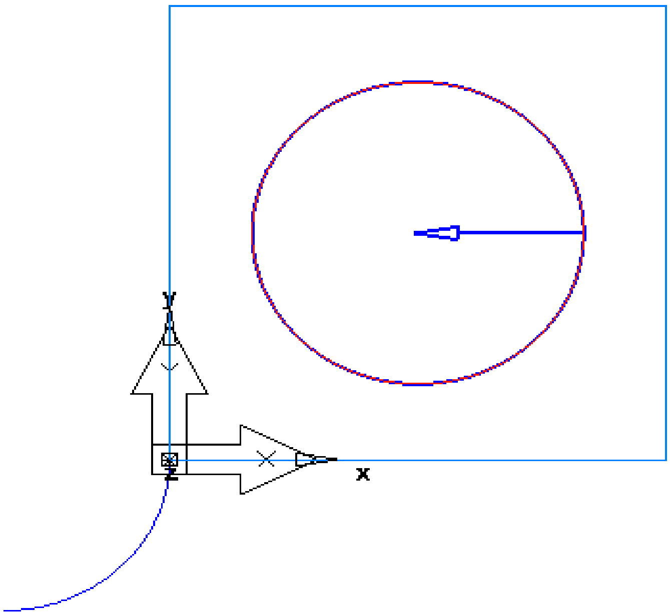


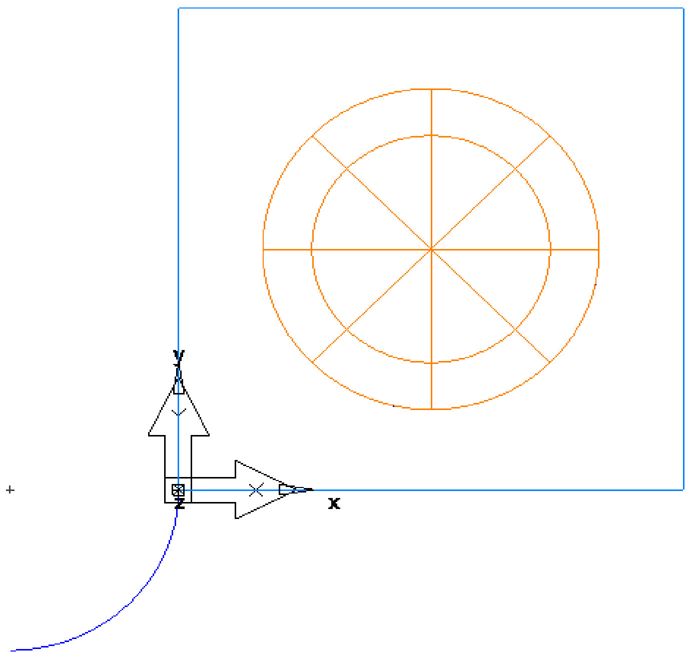
16 See finished image on the next page.

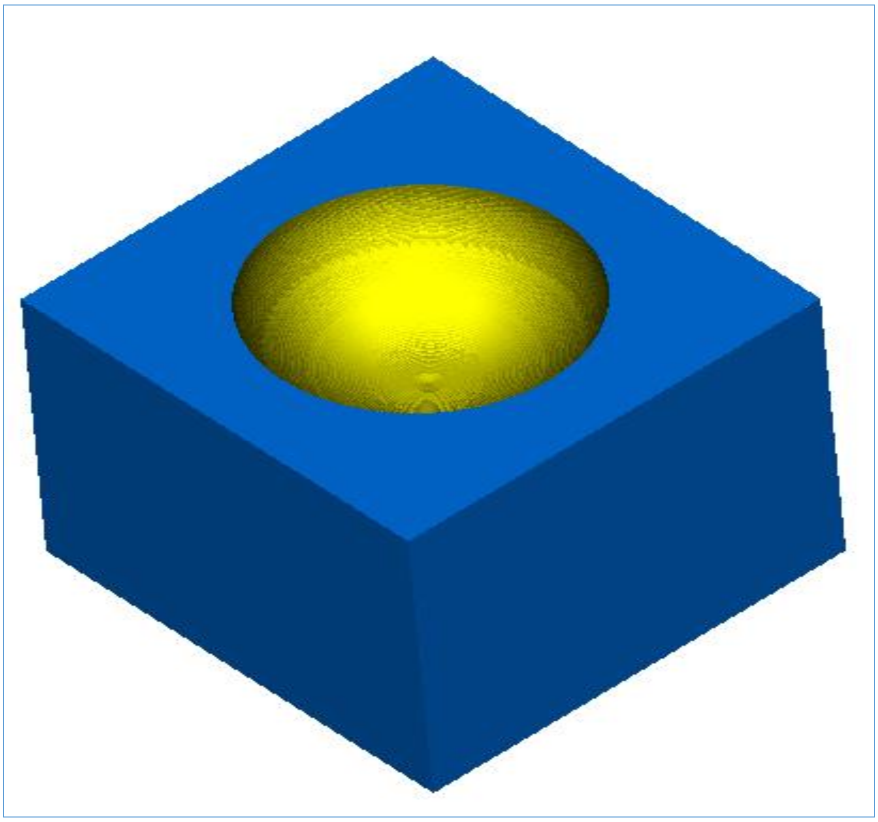


Creating a Concave Shape using Side (Optional Exercise)

1. Save the previous example and then press Ctrl + Z or  to undo the previous example until you get to the Curve stage. Then get up to the same point where you change the Curve called Circle so it points inwards. Use the same depth and Finish.
2. Select **X-Section** and **pick the Quadrant curve**.
3. Select **OK** and Apply, the result should be the same as shown below.




Y
Z
X



Straight line chamfer using X-Section

(Optional Exercise)

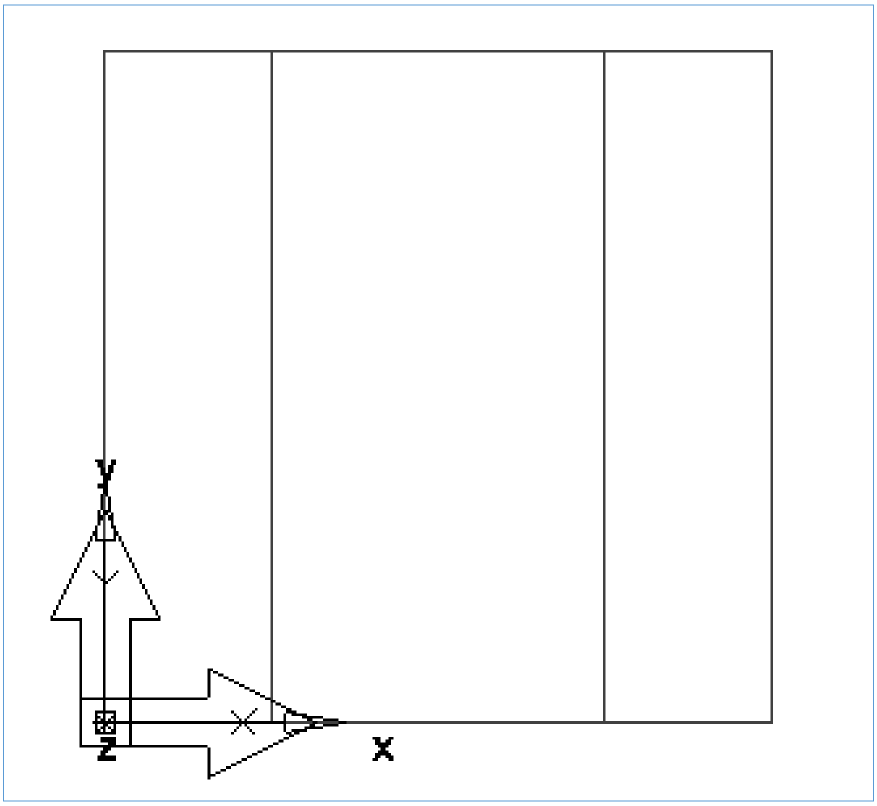
1 Create **a Stock Block 100mm x 100mm x 42mm deep**.

2 **Hide Stock** by selecting **View>Hide>Hide Stock**.

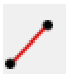
3 Then Construct a Rectangle from **Construct> Curve>other Methods>Rectangle**. **100mm x 100mm**.

Remember to select create as arcs and lines.

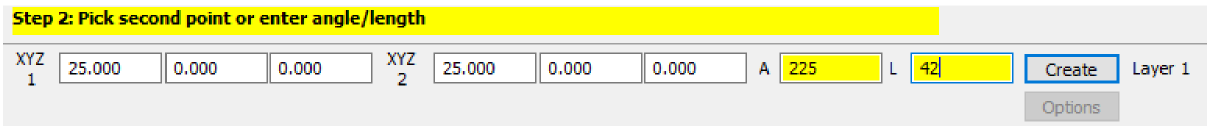
4 Offset the lines **inwards** using **Offset**  from the Geometry Toolbar **25mm** as shown below.



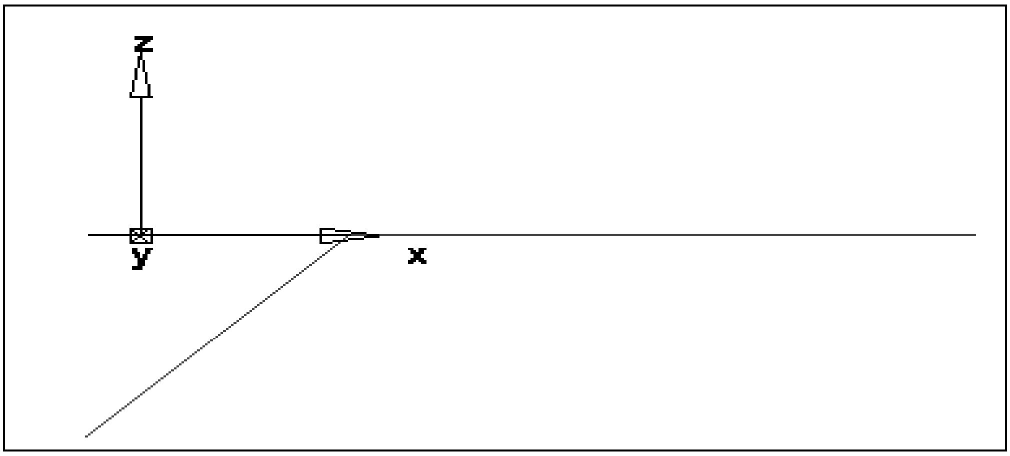
5 Right click in the **graphics area** and select **Front** or **Ctrl** + **2** you will be looking end on along the X Axis. Snap to the first point for the left line as your first point.

6 Select line from two points. 

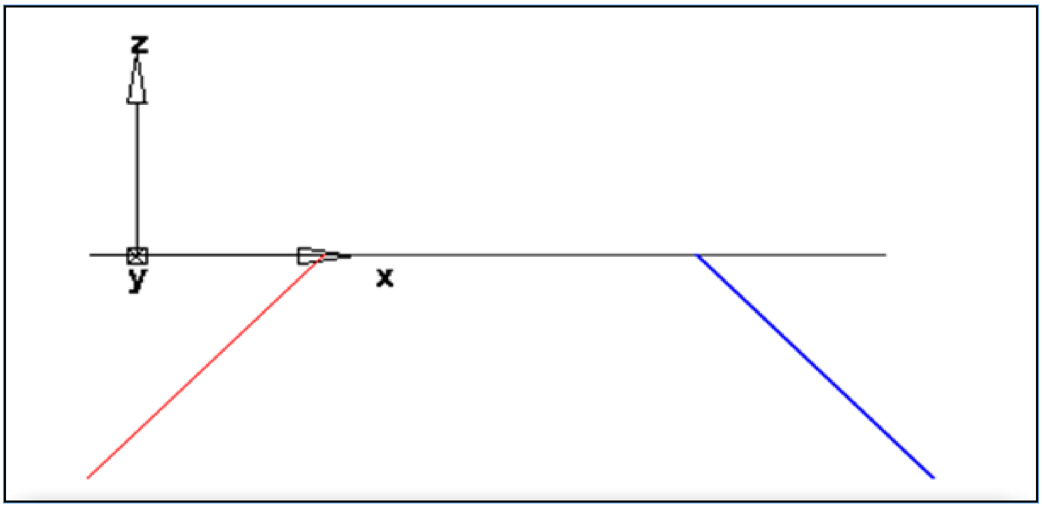
7 Type in the following XYZ2 **Metric** co-ordinates. **25** for X2 Tab to the **A** Angle field and enter **225** then Tab to L and enter **42** then press enter to action this process.

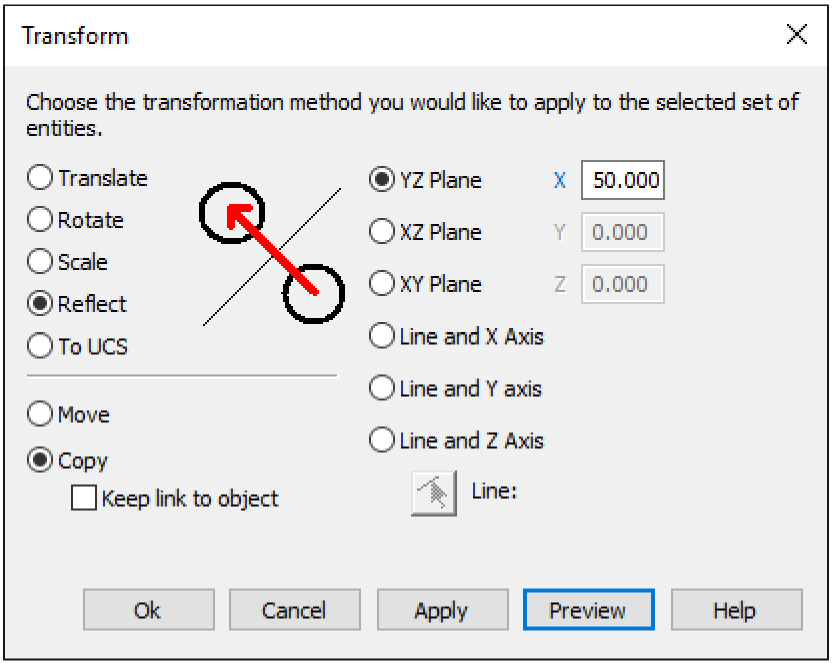


You should have the image as shown below. This is in ***Front View*** *or* ***Ctrl*** *+* ***2.***



8 Now we will select the **line** and then using **Transform** we will **reflect** the line about the centre line in X as shown.



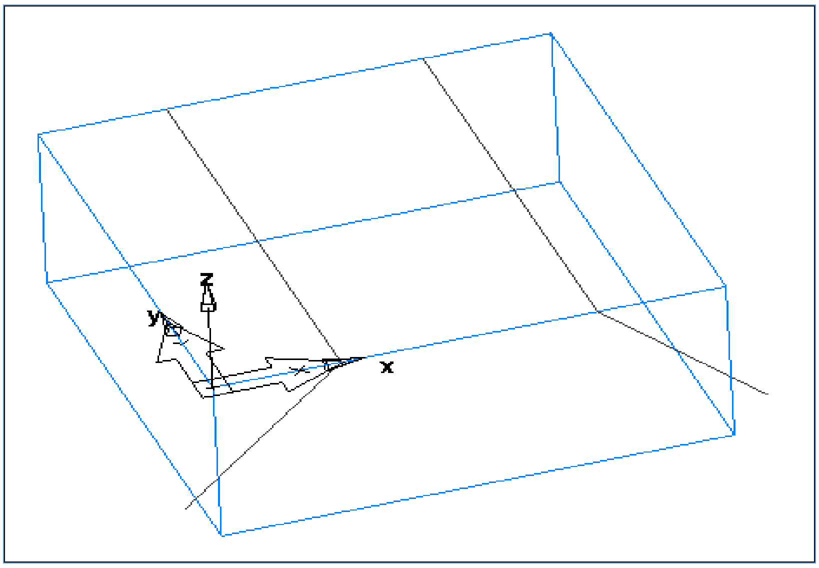


9 Select **Copy** and **YZ Plane** and enter **50mm.** Select **Apply.**

10 We will now use the lines in our **X-Section** machining example.

*Remember to **show stock**.*

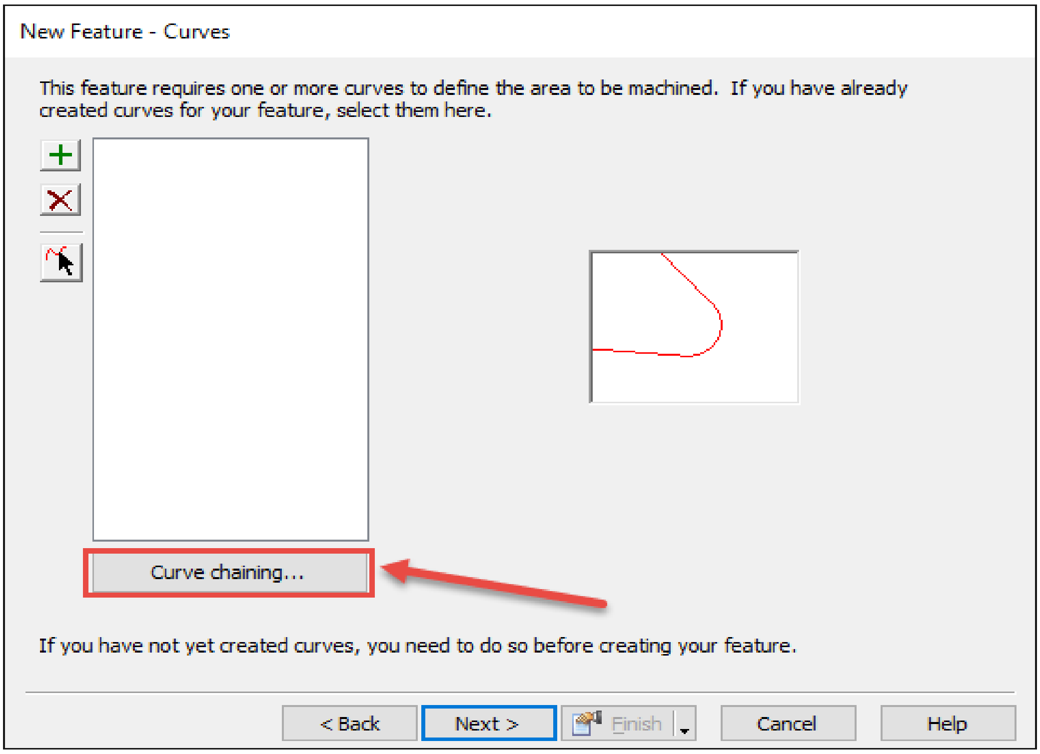
Your image should look like the one shown below before machining.



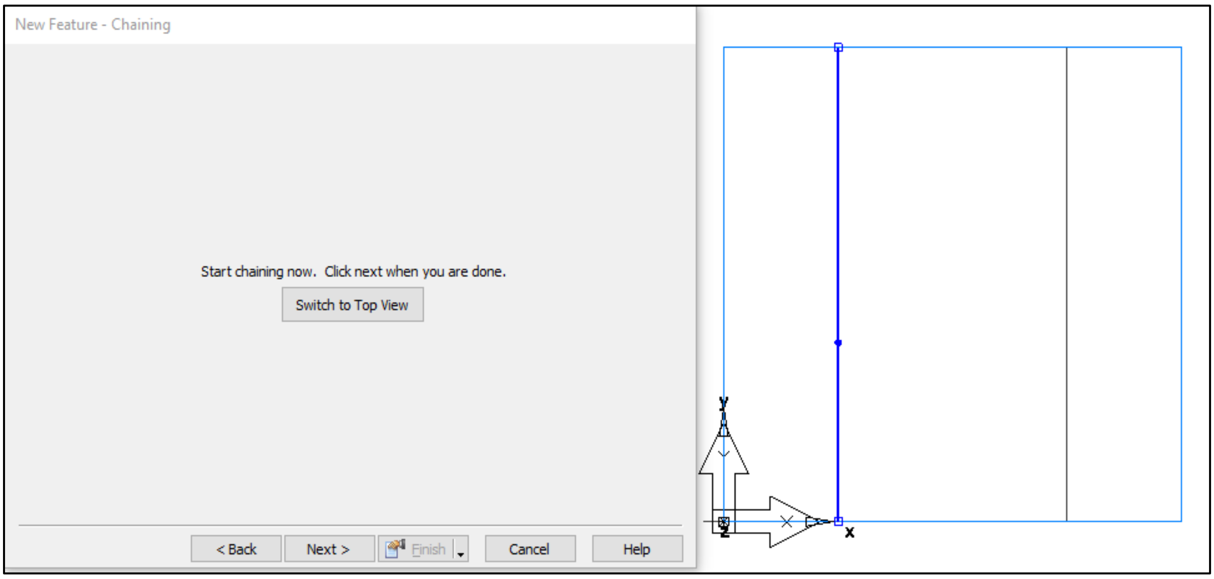
11 Select **Features** from the **Steps** Menu or press **Ctrl** + **R** and select **Side** then **Next.**

*You will be presented with a Menu **New Feature - Curves.** This is where you select your **Curves** or **Chain** new **Curves**.*

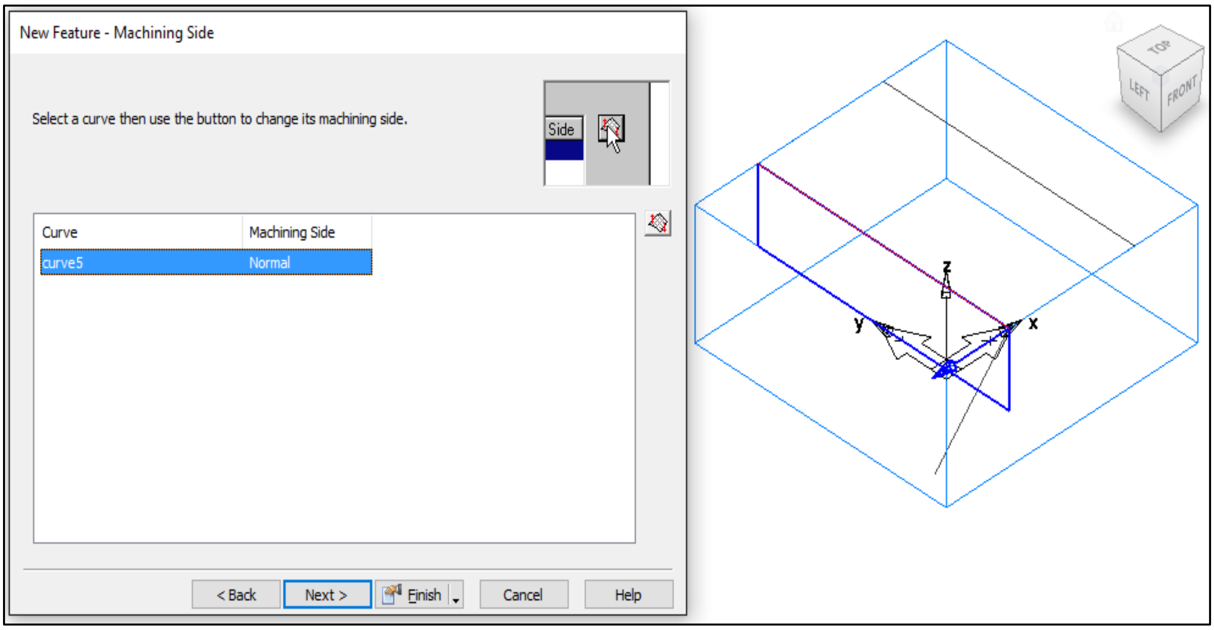
12 Select **Curve Chaining** as shown.

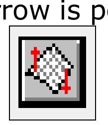


13 Select one of the longer top lines we created first.

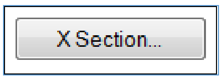


14 Select **Next.**

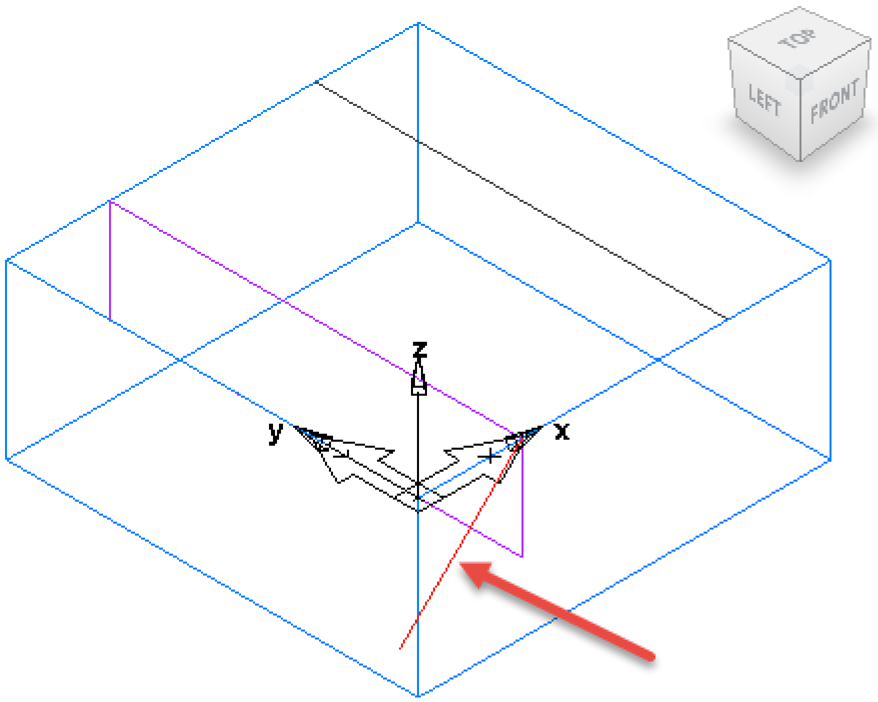


15 Make sure the arrow is pointing as shown. If not select the **Switch Machining Side**  Icon to change the direction. Select **Finish**.

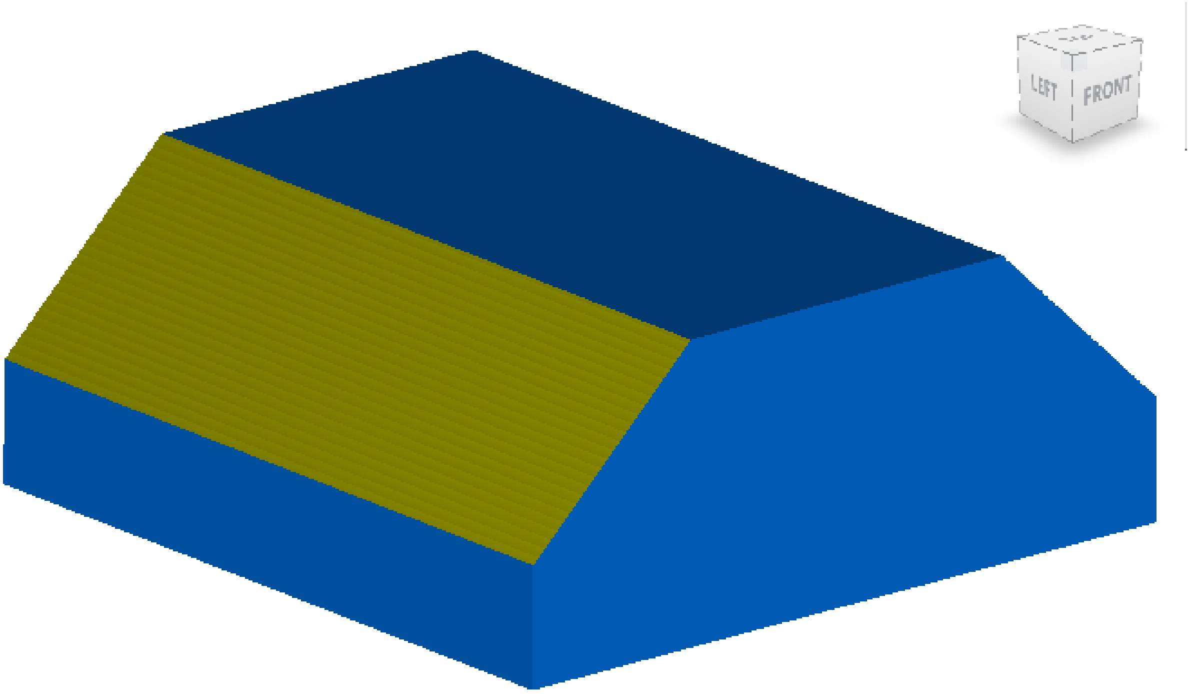
16 You will be presented with a menu showing **Side properties.**

One of the buttons shows **X-Section**. Select this button. 

17 Select **Side Curve**. Select **Ok and Apply.**



*You have now finished this exercise and the feature is now machined. Apply this method of machining to the other Geometry. Then **Run** the **3D Simulation**.*



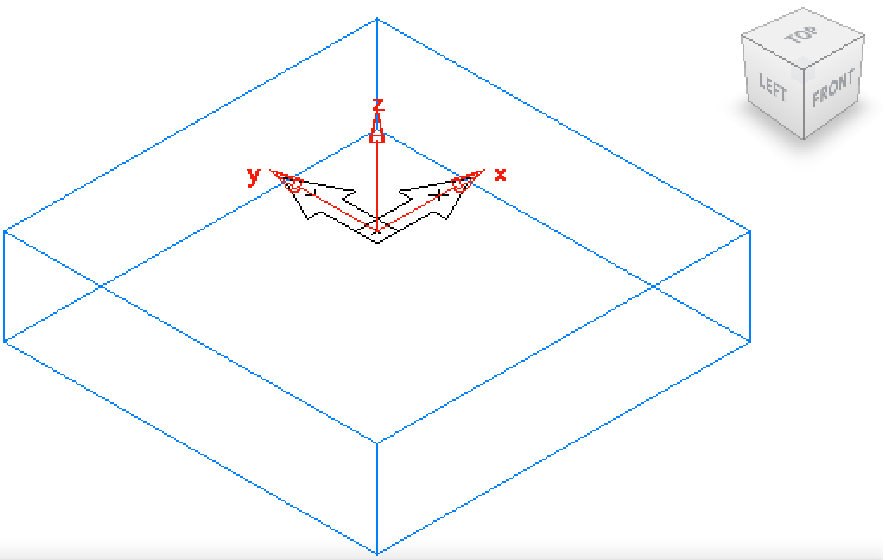
*Completed **3D Simulation** showing **X-Section** machining.*

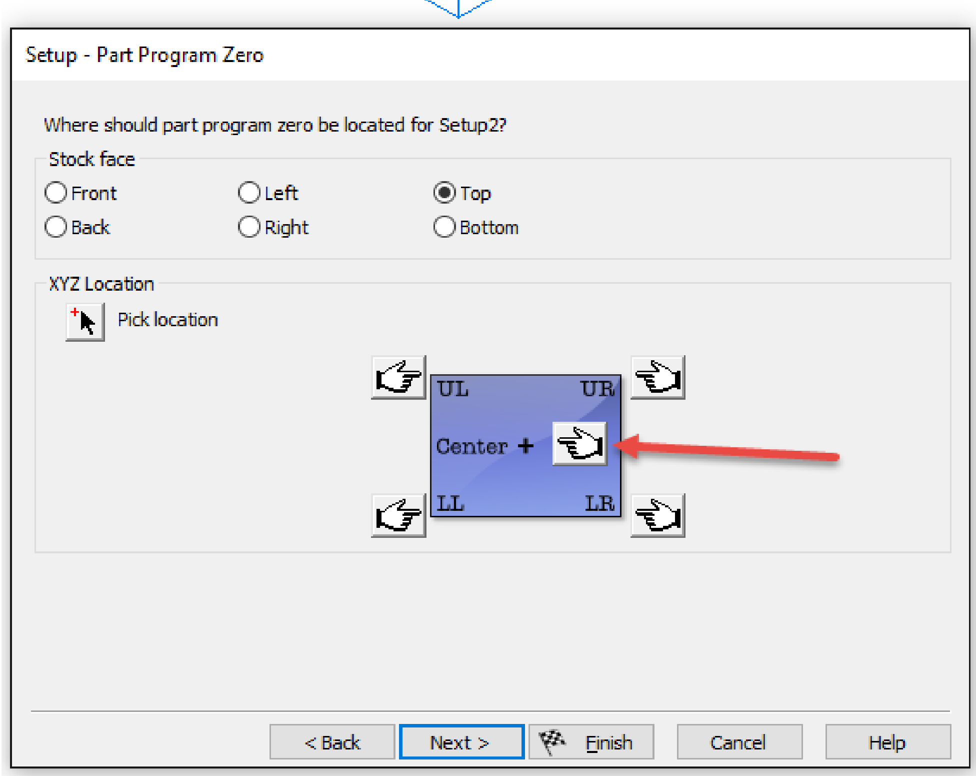
Creating Machining Set-Ups

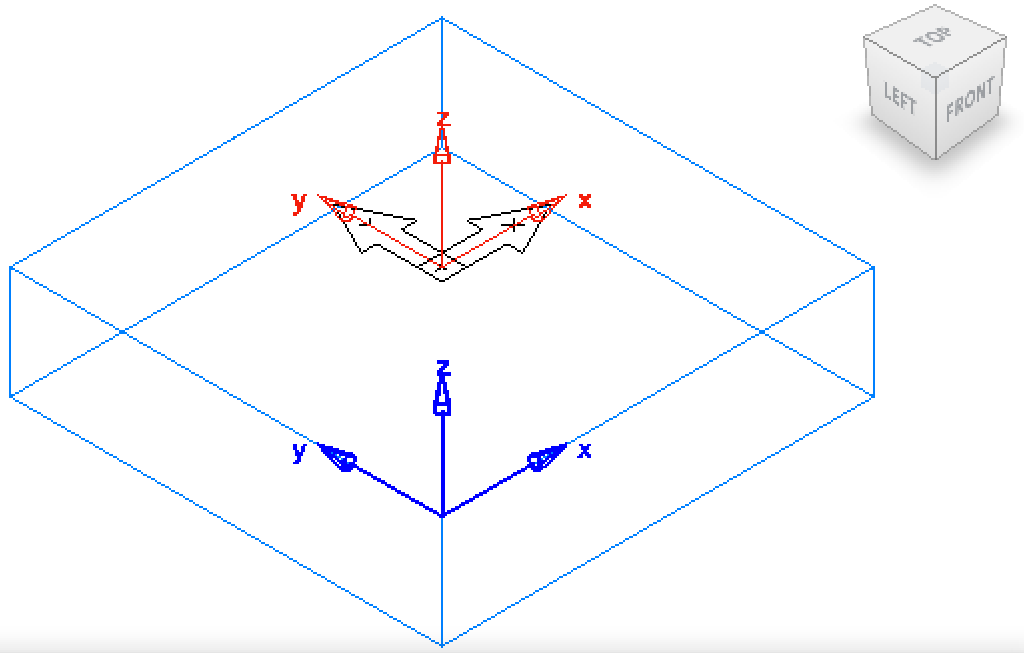
*Before starting any machining operation, it is necessary to set a **Machine Offset Zero** position from which to work from. In **FeatureCAM** these positions are called **Setups** and this can be defined by using a number of techniques.*

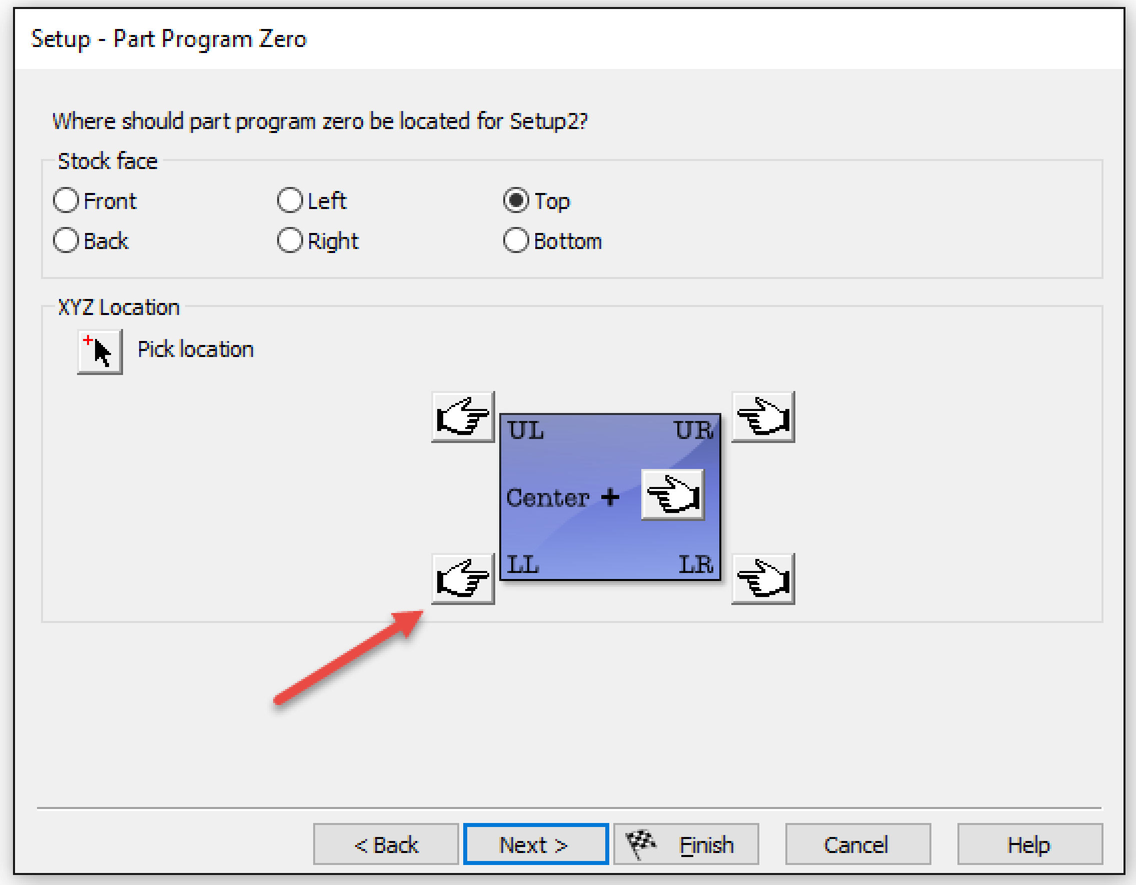
Creating Setups from Stock (Optional Exercise)

*The **Stock** represents the **Material** which is to be machined. The most common places that are used when defining the **Setup** on a standard Block are **Top Centre** of a **Face** or one of the **Corners** as shown below.*





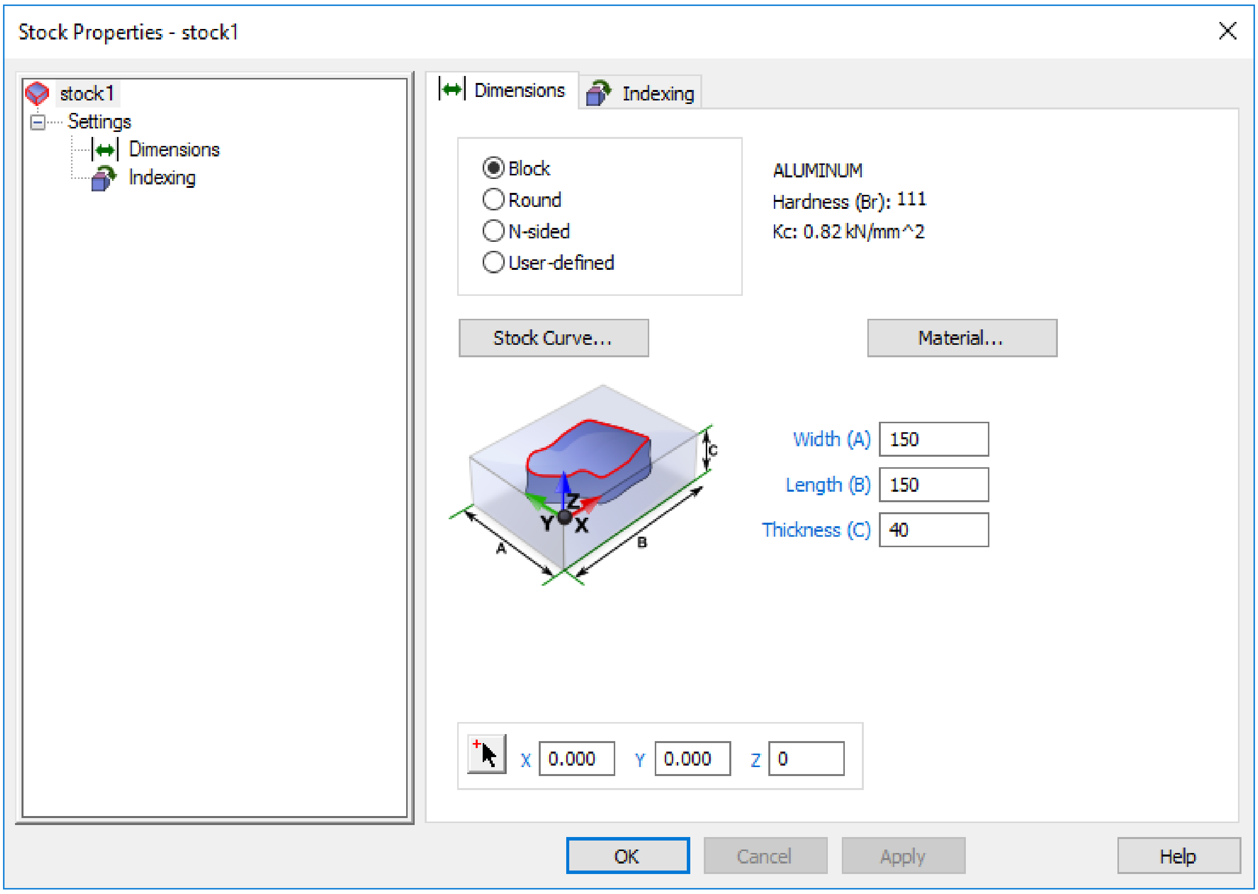




*When **FeatureCAM** opens a new **Document, Stock** is automatically displayed on the screen. Enter the values shown on the next page.*

18 Double click on the **stock** either in the **Part View Toolbox** or in the main **Graphics area**.

19 Fill in the Form as shown, **Length 150mm**, **Width 150mm** and **Thickness 40mm.**



*The **X**, **Y**, **Z** input fields at the bottom of the form are the position of the top face bottom left corner in the **World Coordinate System** as shown in the image above.*

1 Change the values in the **input fields** and watch how the Stock moves around.

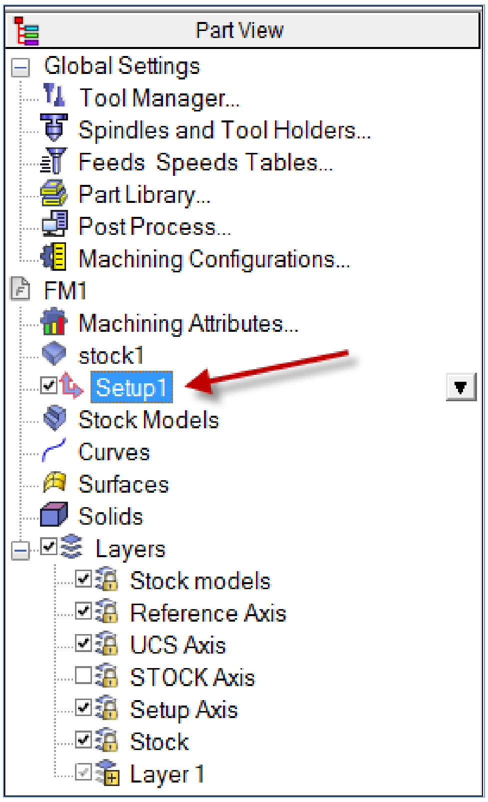
2 Reset the figures back to **X0, Y0, Z0**.

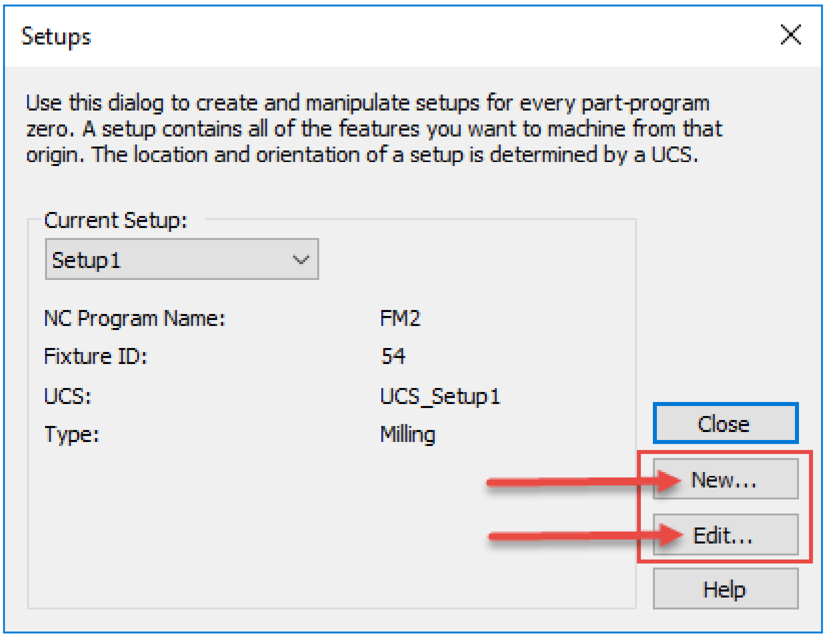
3 To place the **Setup** in the **Centre** of the **Block**, it is simply a case of moving the **Setup** to the **Centre** of a **Stock face** and this is achieved by changing the position, using the **Part View Toolbox**.

At the left of the screen are the **Toolboxes** and depending upon which **Toolbox** is selected; it will either show the **Part View** or the **Steps Toolbox**.

*If you can only see the **Part View Toolbox** and not the **Steps** options as shown. Try looking at the bottom of the screen. (When it is not in use, the **Steps Toolbox** drops down to the bottom, just click on the **Tab** to reactivate the **Toolbox**.)*

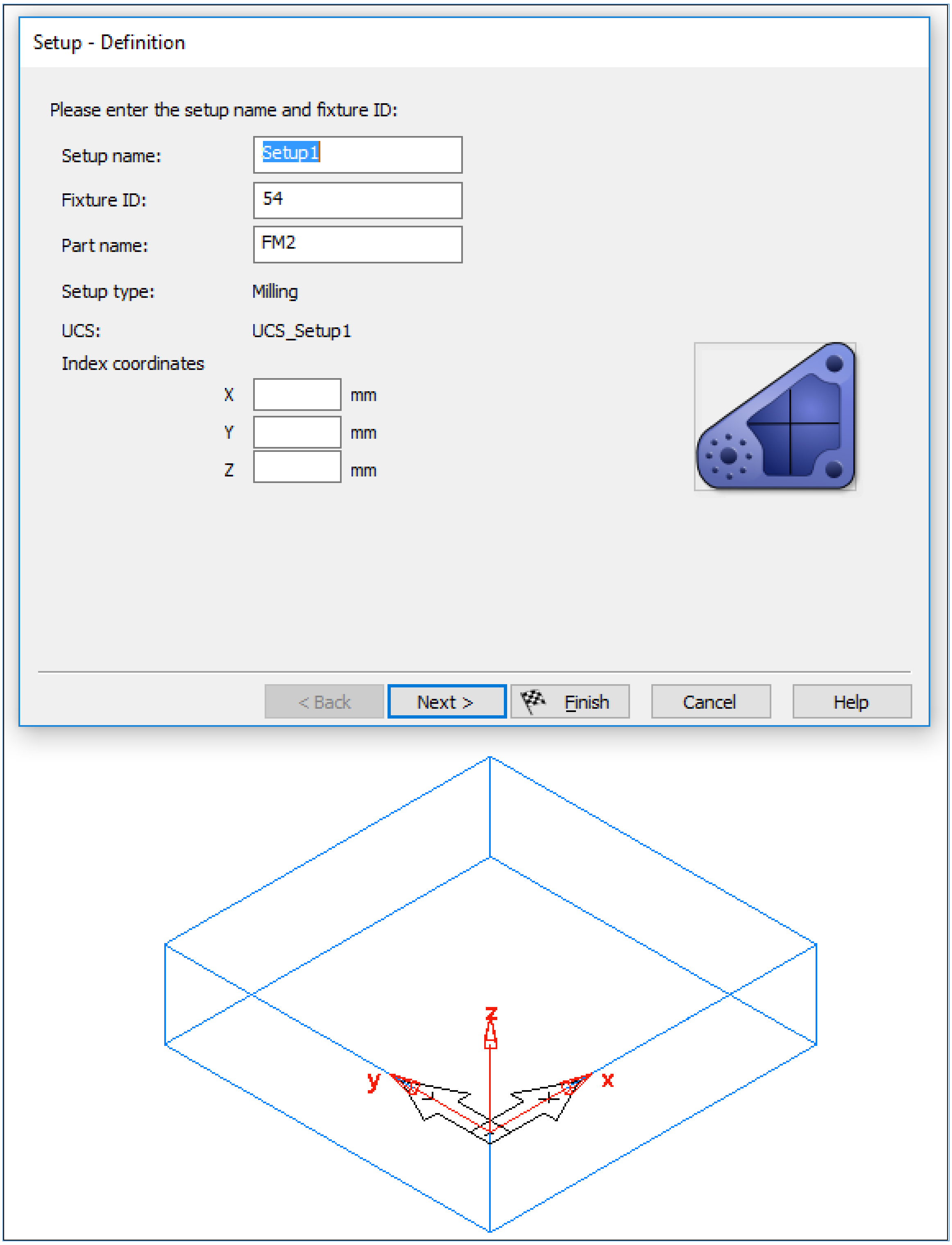
1 Double click on **Setup1** in the **Part View** and the menu appears as shown.



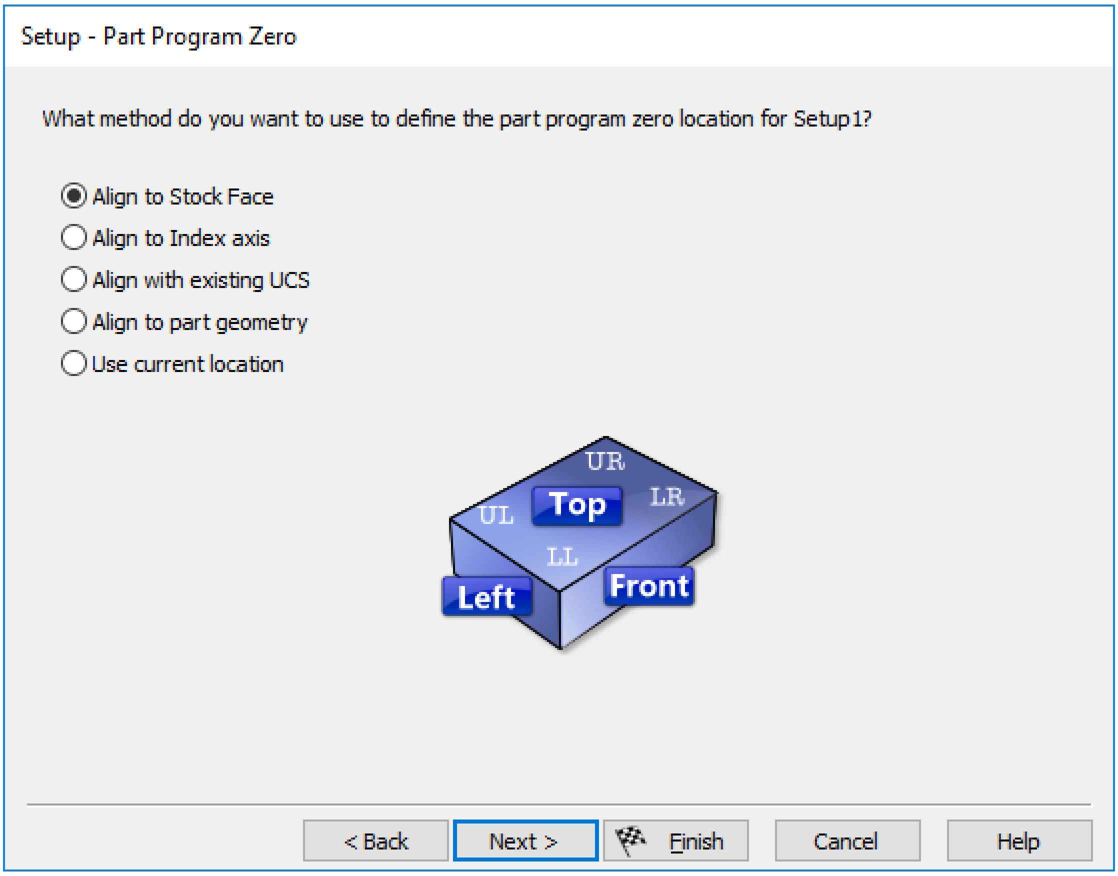


2 On this form, the user can choose to create a **New Setup** or to **Edit Change** an existing one.

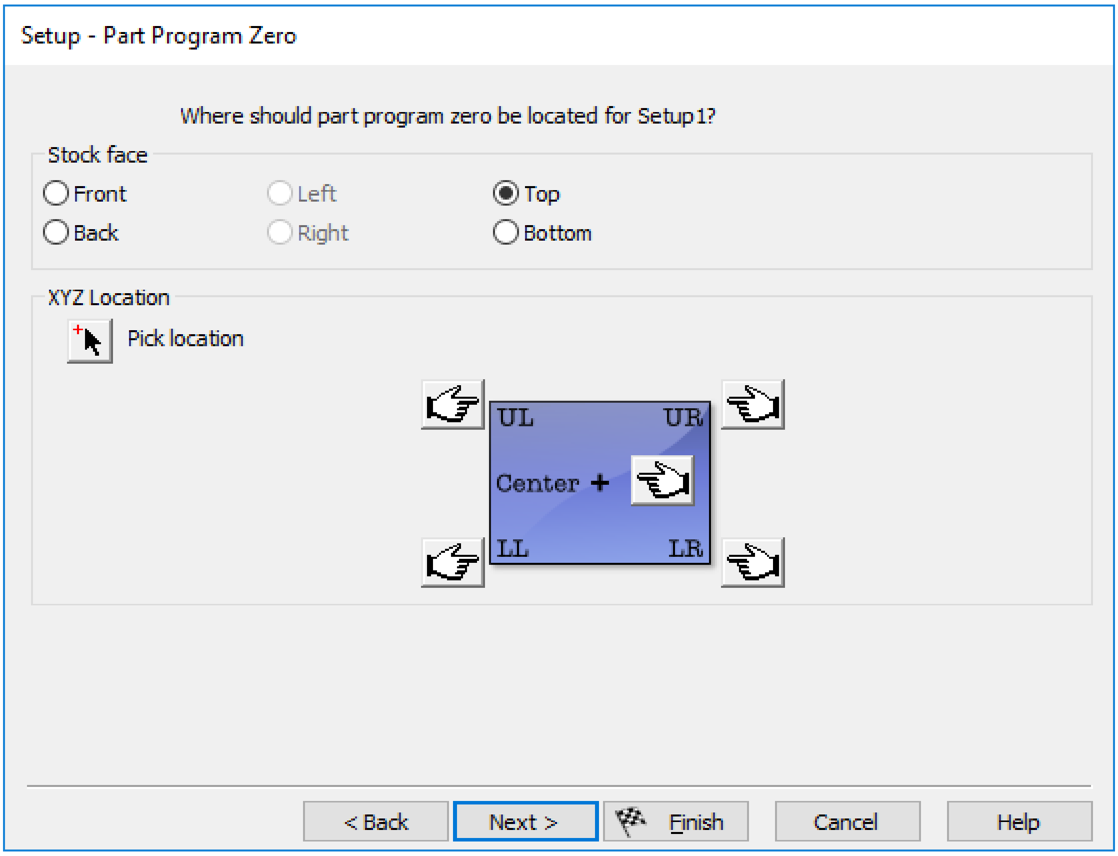
3 Click on **Edit**. Then **Next**.



4 Use **Align to Stock Face**.



*The form is divided into two areas and these are **Stock Face** and **XYZ** location.*

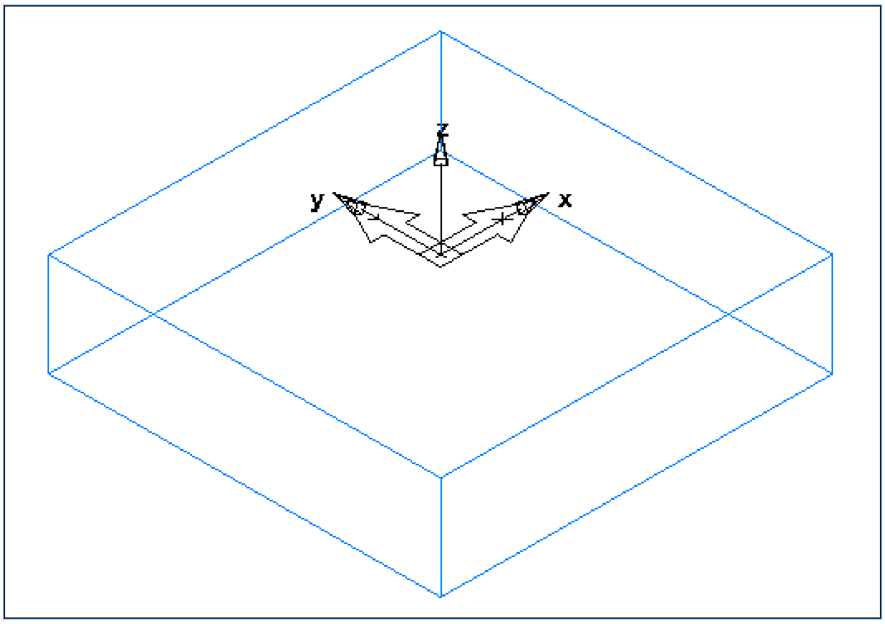


__Stock Face__ is used to tell __FeatureCAM__ on which Face to place the __Setup__.

__XYZ Location__ is used to position the Setup on that Face.

5 Select **Top** followed by **Centre**. Click **Finish** and then **Close**.

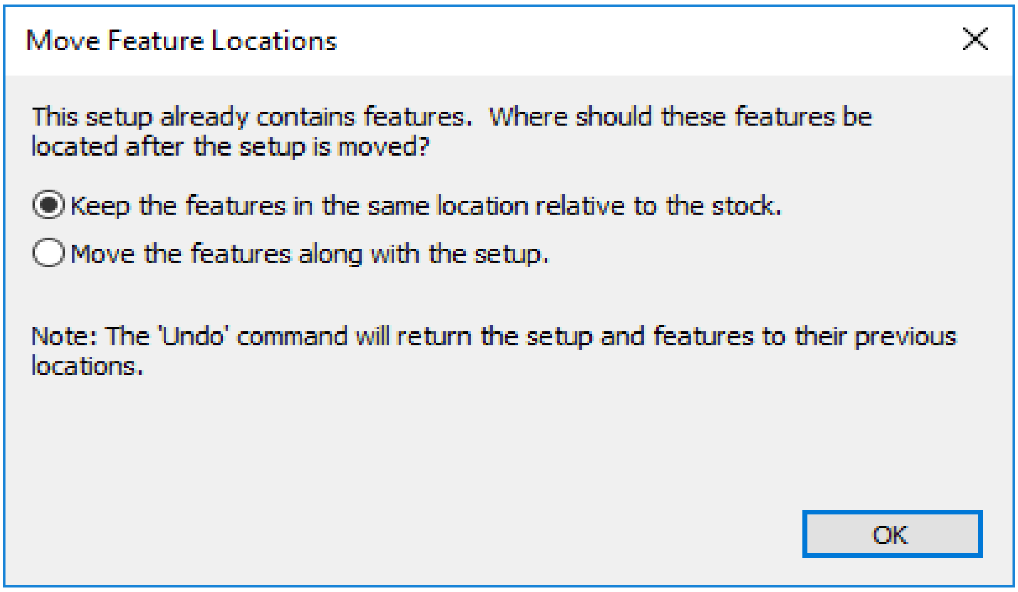
The screen should now look like the one shown below.



To get more familiar with creating new setups. Use the same method and create a new set up on the __Front Face__ at the __Upper Right Corner.__

Move Feature Locations

If you decide to change a Setup position after you have machined a Feature. The following menu will appear.



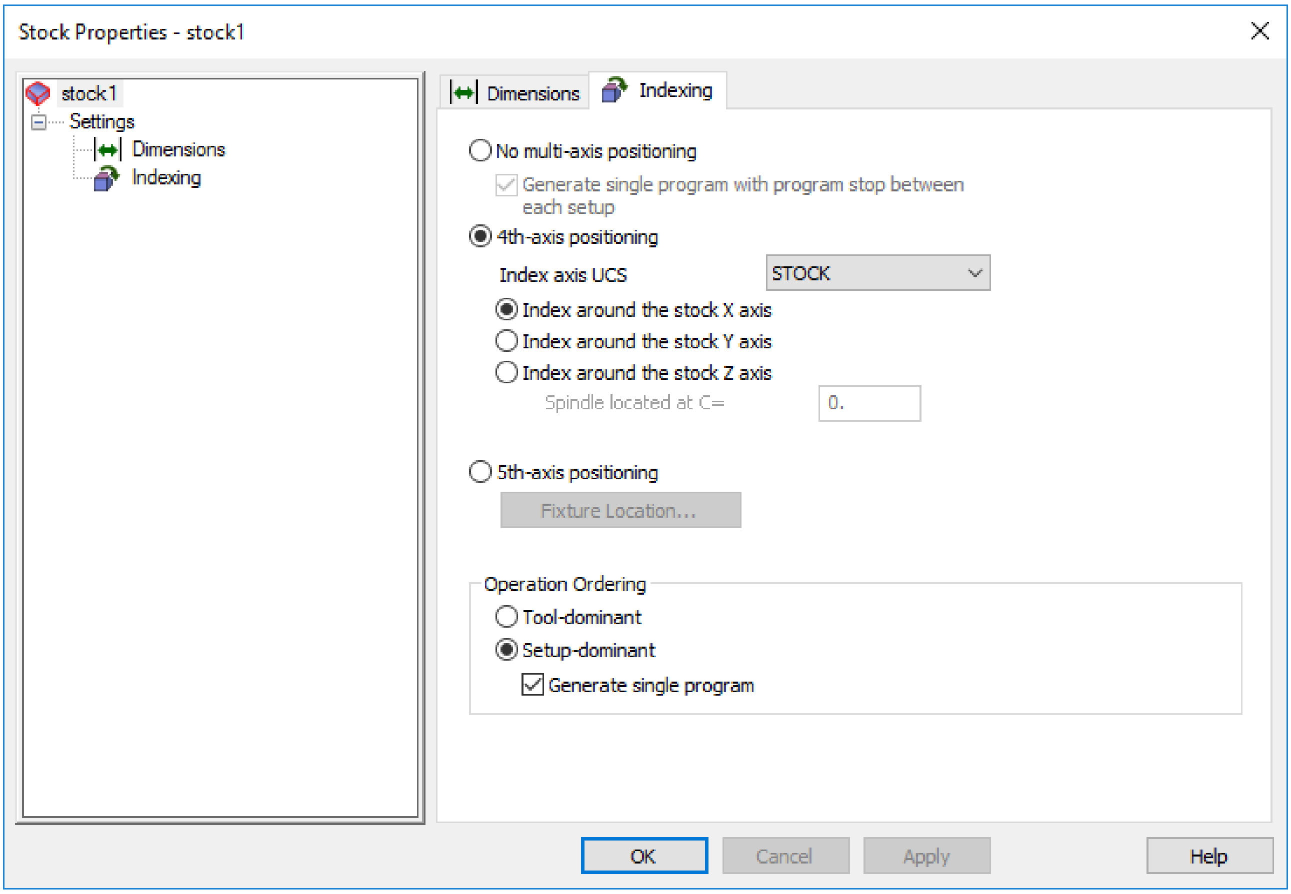
Keep the Features in the same location relative to the stock: - The Machine Offset Zero moves and everything else remains in the same position.

Move the Feature along with the setup*: - This will move the Feature relative to the old* ***Machine Offset Zero*** *and move this to the new* ***Machine Offset Zero****.*

4th Axis Indexing (Information Only)

In ***2.5D*** *milling,* ***4th Axis indexing*** *is standard, and can be accessed when opening a new document. The stock wizard allows you to choose* ***4th Axis****, as multi-axis positioning. It can be accessed any time in the stock properties under the indexing tab. Your machine must have four axis capabilities, and a rotary table, or indexer. Also a 4th axis post processor must be loaded in* ***FeatureCAM****.*

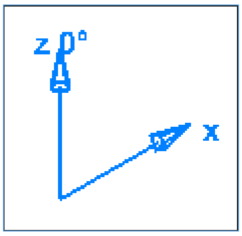
The part can be indexed around the X, Y or Z axis. The desired axis you wish to index around must be selected in the stock wizard (illustrated) while selecting the shape and size of your stock.



Your machine must have four axis capabilities, and a rotary table, or indexer. Also a 4 axis post processor must be loaded in ***FeatureCAM****.*

The part can be indexed around the X, Y or Z axis. The desired axis you wish to index around must be selected in the stock wizard (illustrated) while selecting the shape and size of your stock.

*The **Stock Axis** is **not** normally displayed but to view the **Stock Axis** click **View** and select **Show Stock Axis** from the **View** menu. It is displayed as two vectors (blue). One shows the axis of rotation (X or Y) and the other indicates the orientation of a 0 degree rotation (pointing at the spindle). You must align your part centre of rotation to the rotation axis of the **Stock Axis**.*



If the Stock Axis is not centred, any features that you will be adding to the part will be out of place. When a block stock is being used the UCS and Stock Axis are many times together but not always centred.

*The **Stock axis CANNOT be moved**, the stock must be positioned around it.*

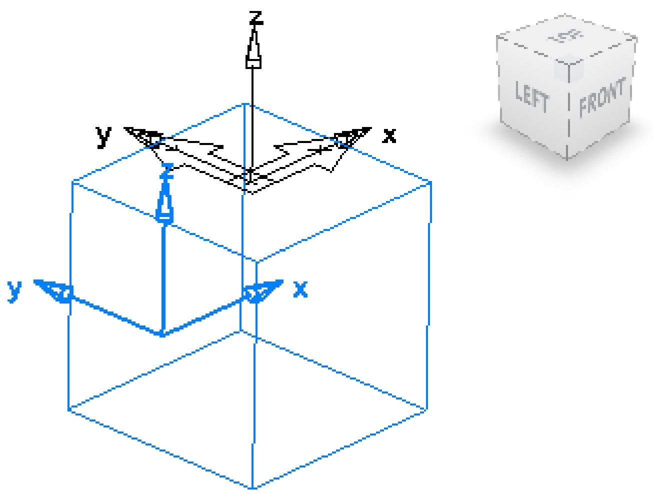
1 Click on **Options** then **Add-ins**.

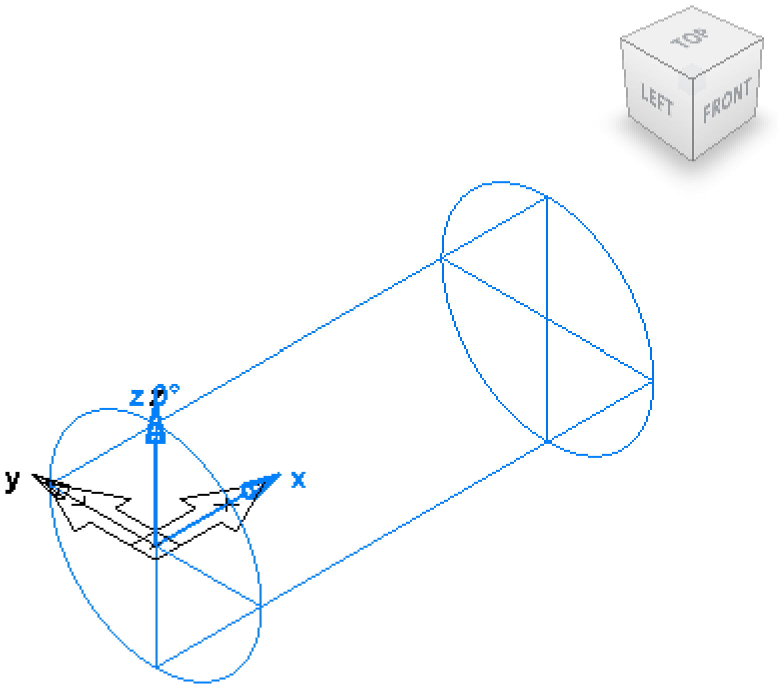
Check the box in front of the Macro called "**Centre Indexed Stock.bas**" and a little tool bar appears, usually by the **Part View**.

2 Drag the toolbar to dock it to an area next to the one of the other toolbars. Click this new button**.**

*This macro will **Move and Centre** the stock with the Stock Axis.*

*When a new document is opened, and round is selected, the **UCS** and **Stock Axis** will be merged and centred on the stock. Now the features can be placed around the indexing axis. The **UCS** may be moved for Feature creation, if desired.*





*When indexing a part that is not round, **FeatureCAM** calculates the corners of the Stock rotation and retracts a little extra to clear the corners of the stock. **DO NOT** use Retract to plunge clearance whenever there is a corner present on the part that will rotate under the tool, when indexed. An example of this is when there is a circular bolt pattern on more than one face of a block stock. In this case you will index to another face using the same tool. After drilling the last hole there must be a retraction to the Z Rapid Plane to clear the corners. If Retract to plunge clearance is set the part will hit and break the drill, or **CAUSE DAMAGE TO YOUR MACHINE.***

Once you have completed the prior steps, you may begin creating and placing features on your part.

*Once you have completed the prior steps, you may begin creating and placing features on your part. All **2.5D**, and **3D**, features may be programmed on an indexed part. **FeatureCAM** can index from face to face and cut features, or machine a continuously wrapping feature. Geometry curves and features are applied in the same manner as in any **2.5D** part programs. They are placed in relationship to the UCS the same way as non-indexed parts.*

*Wherever the **UCS** is located, for example the centre of the part, the **Feature** may be created at **Z0**, but must be moved up to the desired Z elevation to place it in the proper location.*

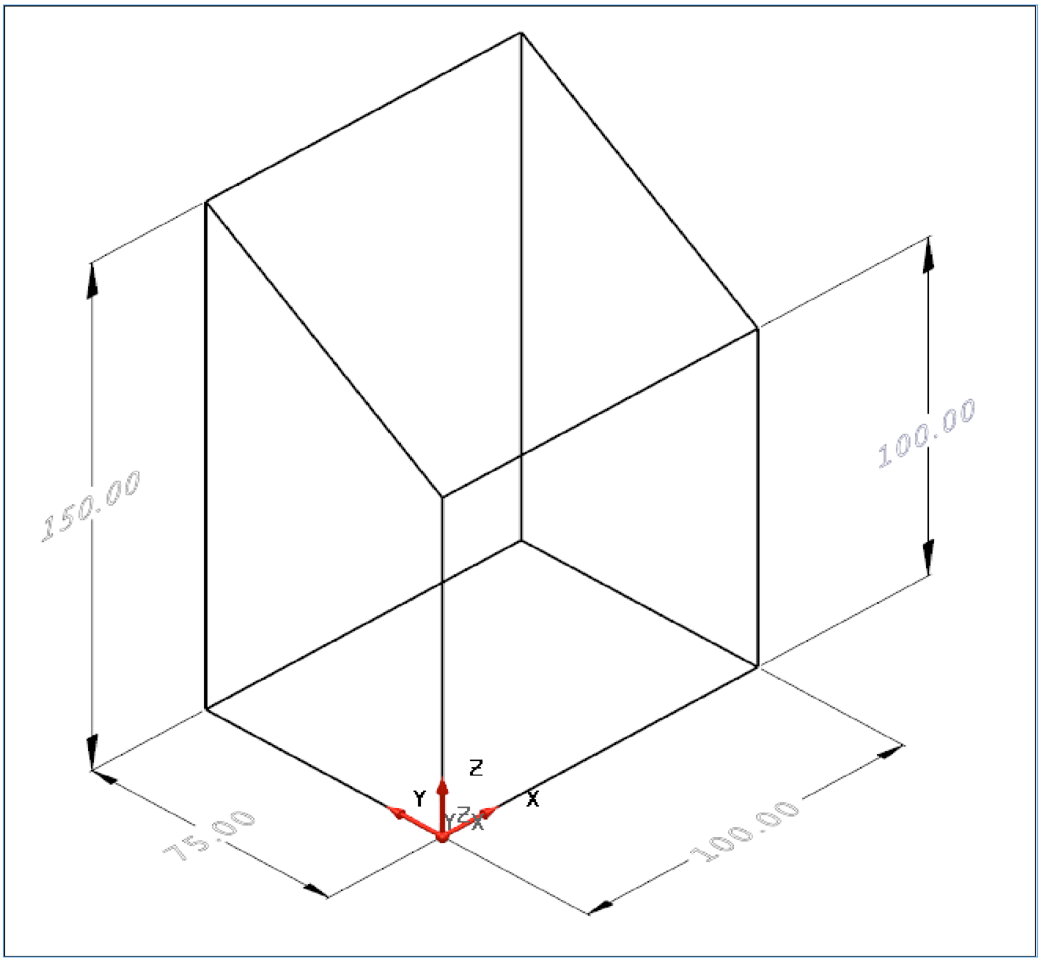
*Features **May NOT be Transformed/Rotated/Copied around an Indexing Axis** but **Geometry** and **Curves** can. You may also create patterns around an **Indexing Axis**. On block stock, the part is indexed to the desired face, and the features are applied to each face.*

*On the indexing tab you may choose **Tool Dominant** or **Setup Dominant**. **Tool Dominant** does all of the machining on every **Feature** that uses that particular tool before it changes the tool, more indexing, less tool changes. **Tool Dominant** can have just one **Setup** for all of the **Part Features**.*

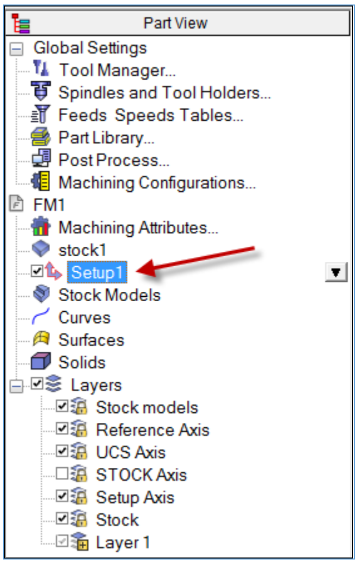
***Setup Dominant** requires a separate **Setup** for each face to be programmed and it completes all **Features** for each **Setup** before indexing to the next **Setup**, more tool changes, less indexing.*

Creating Setups Using Geometry (Class Exercise)

1 It is also possible to create **Setups** which are aligned to existing Geometry. Draw the wireframe geometry shown below.



2 Double click on **Setup1** in the **PartView** and the **Setups** menu will appear.

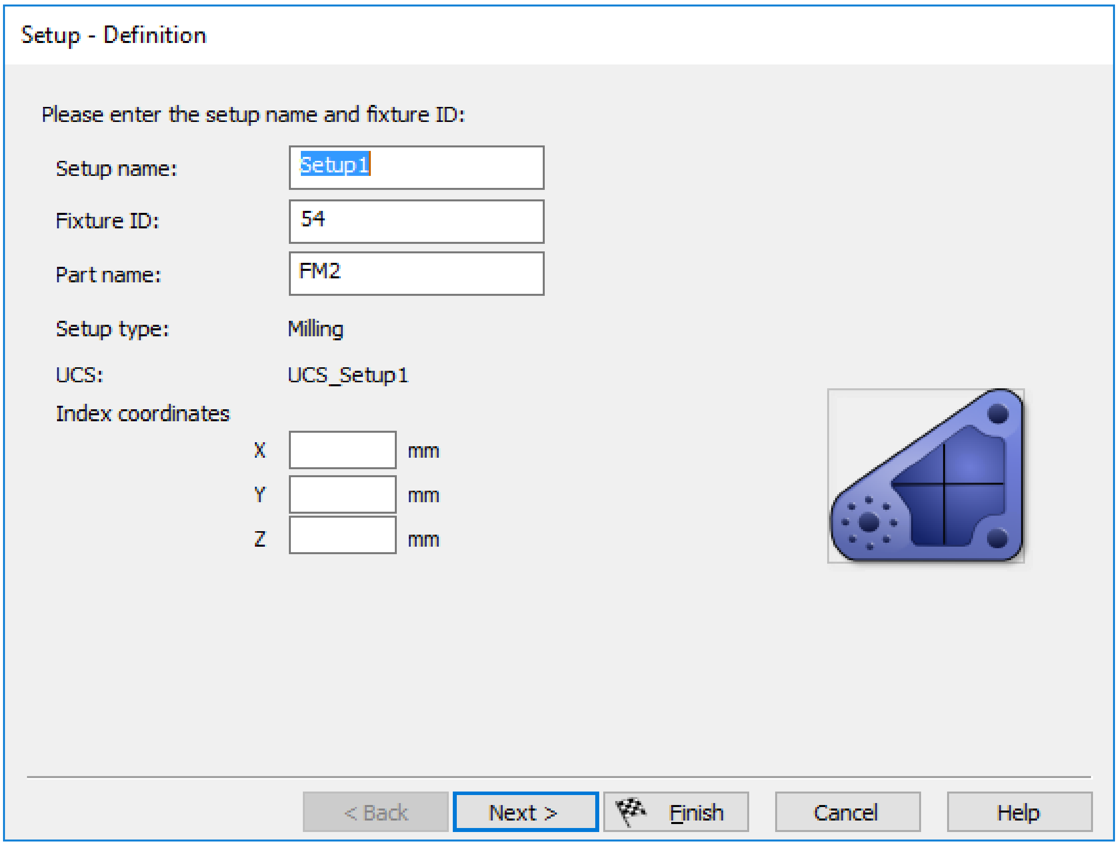


3 Select **Setup1** from the **drop down menu** select **Edit**.

This will open the Setup Definitions form.

4 **Rename** the **Setups** so that they are easier to manage.

5 Enter the name **Machine Offset Zero** into the **Setup Name:** field. This name will appear in the **Part View Toolbox**.



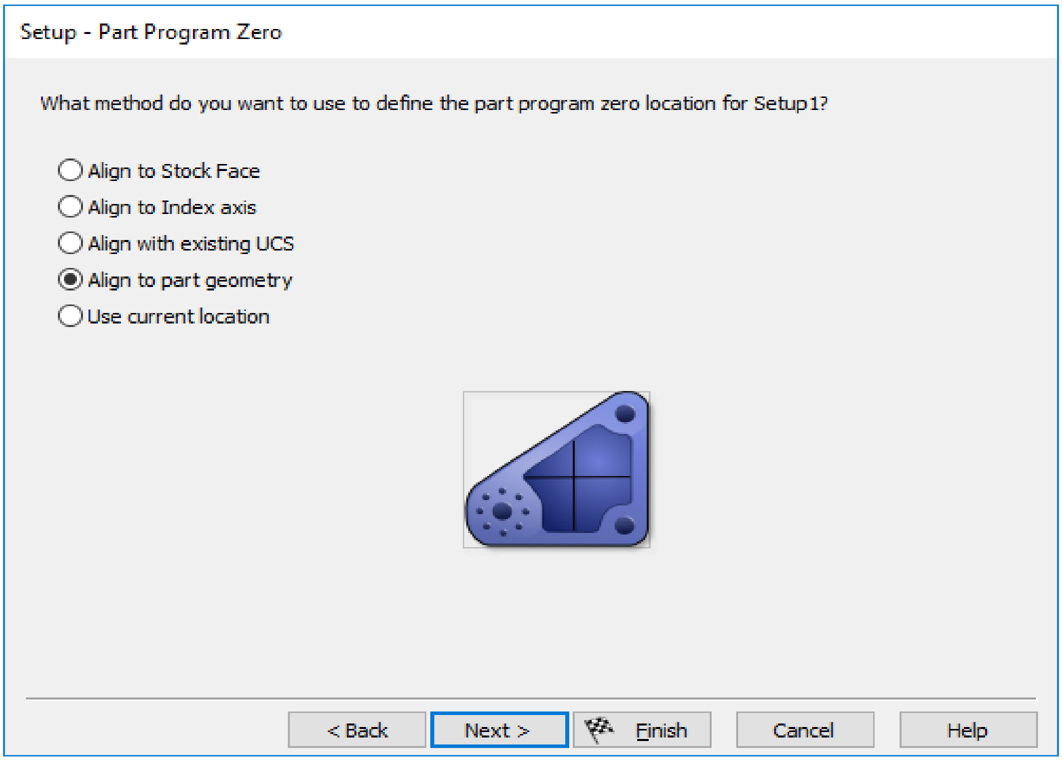
*The **Fixture ID** field is where the user tells **FeatureCAM** which machine offset to use.*

*For example: If the machine uses **G54, G55** then enter **54**. If the machine tool uses **P1, P2** etc. then enter **1.***

***FeatureCAM** will automatically insert the correct **Fixture ID** for the next new **Setup**.*

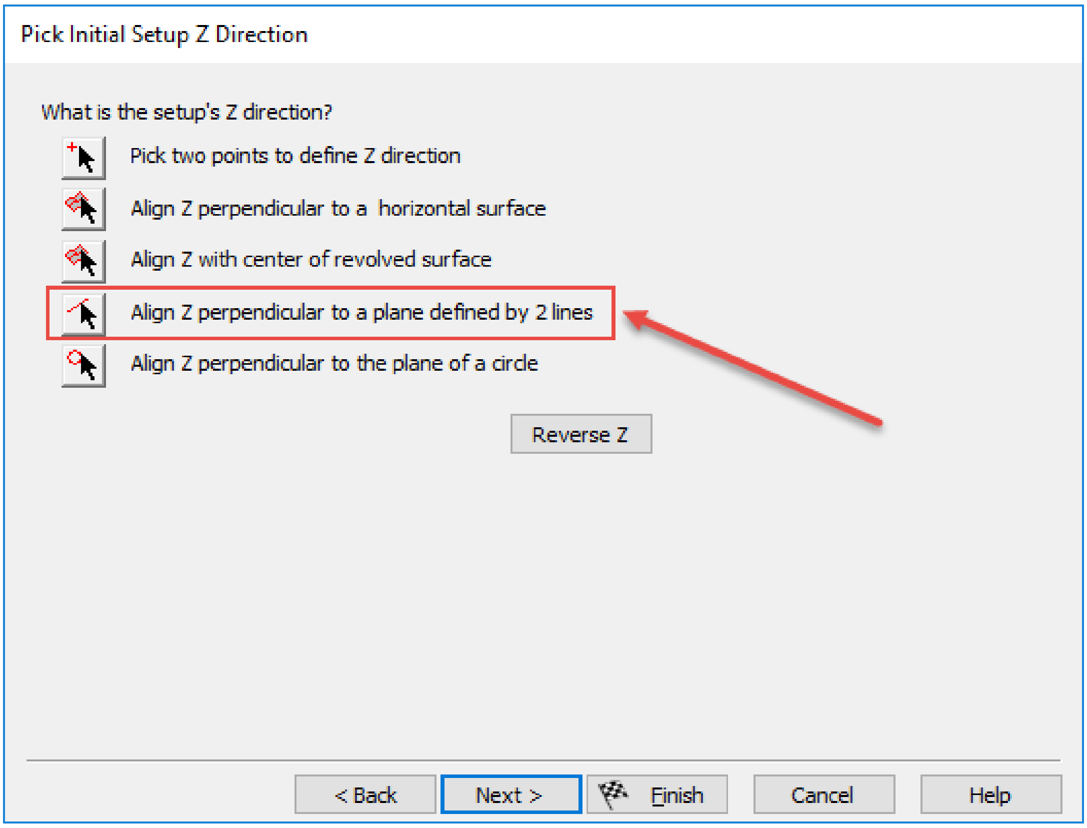
*It can be useful for the **Part / Product** name to appear in the **NC output file**; this can be inserted in the **Part Name:** field.*

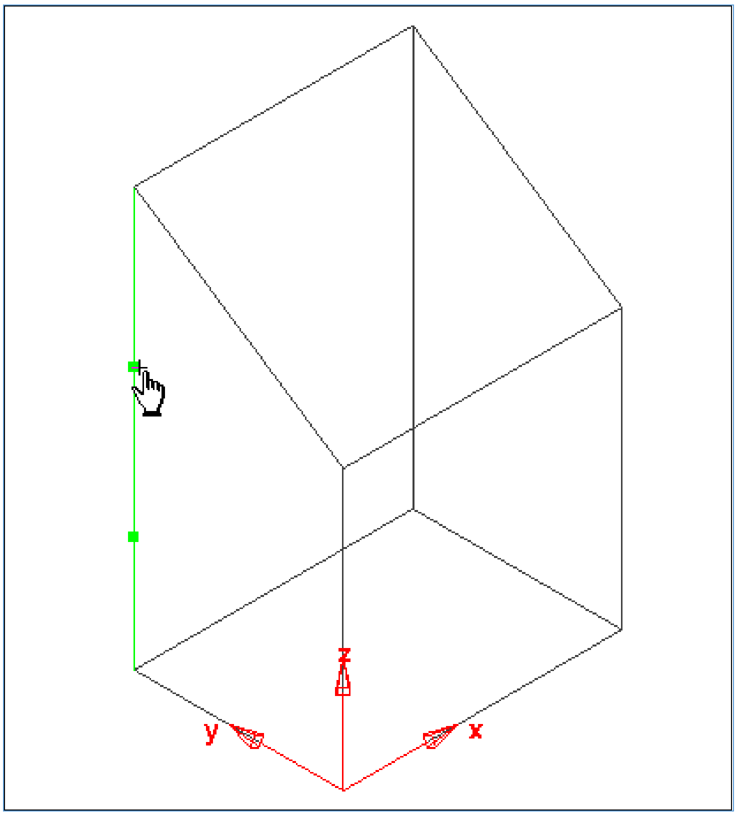
6 Click **Next** and select the **Align to Part Geometry** option.



*By selecting the **Align to Part Geometry** option, **FeatureCAM** knows that it has to provide the user with some tools to help them set up or align the New Work Plane. There are five options shown below and depending on the geometry available; the user can choose one of these to align the Z axis.*

7 Pick two lines to define Z direction.


Pick Initial Setup Z Direction
What is the setup's Z direction?
Pick two points to define Z direction
Align Z perpendicular to a horizontal surface
Align Z with center of revolved surface
Align Z perpendicular to a plane defined by 2 lines
Align Z perpendicular to the plane of a circle
Reverse Z
< Back
Next >
Finish
Cancel
Help

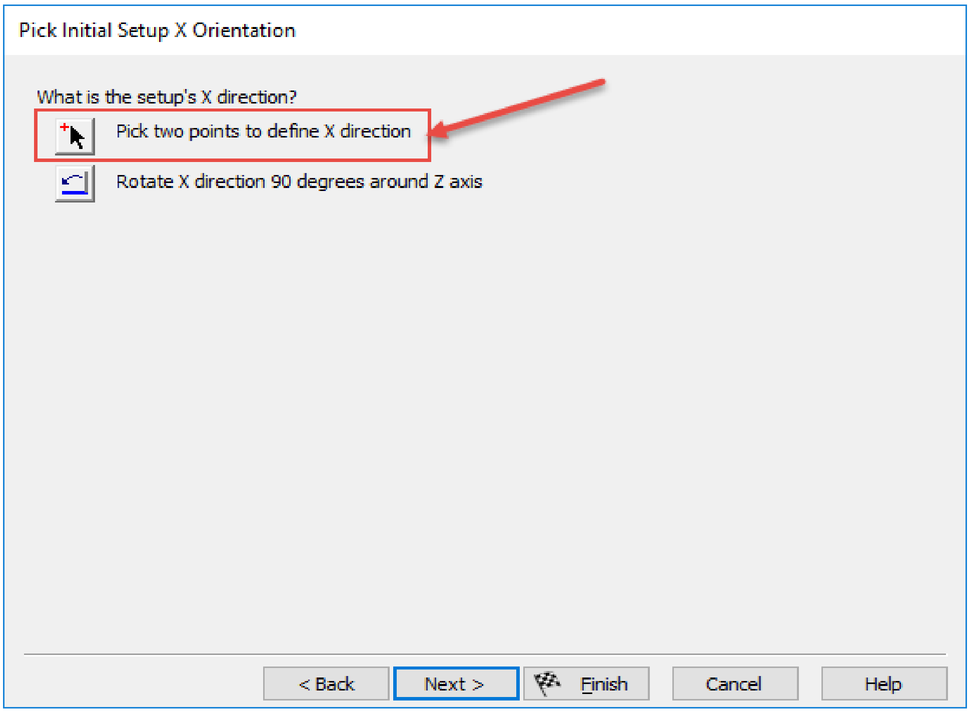

z
y
x

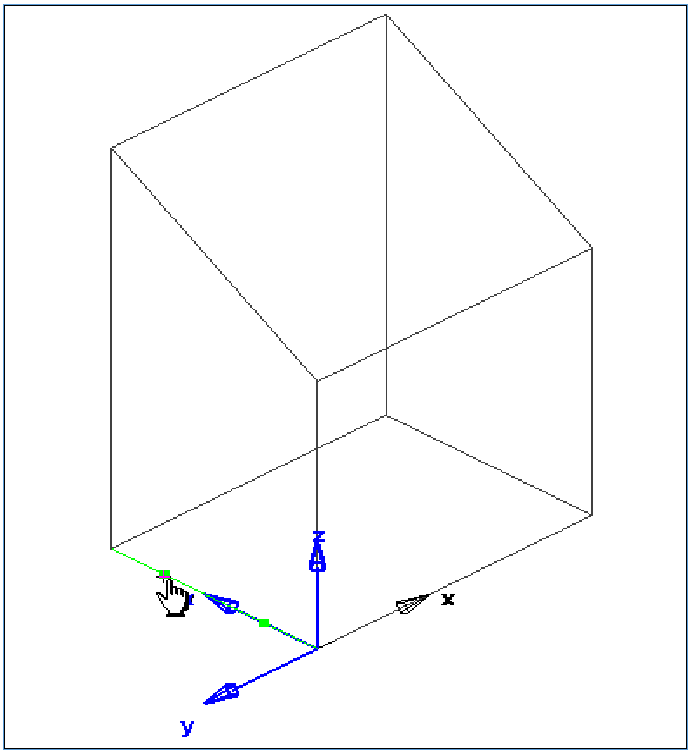
The form will shrink to one side of the screen to allow access to the Geometry.

Two mouse clicks are used to define the Z Axis direction.

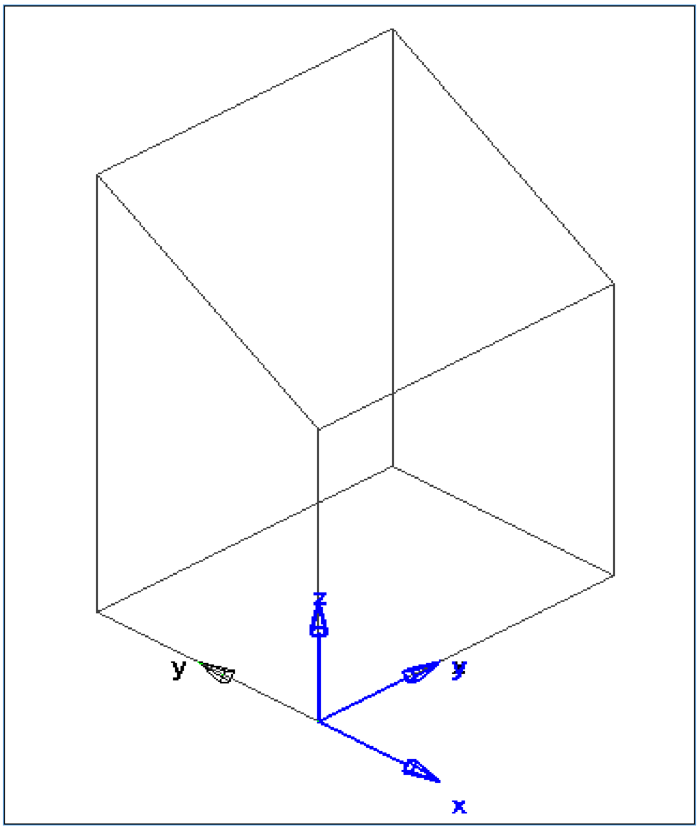
8 Select the bottom point as shown, followed by the top point. The position of the second click in relation to the first is what controls the direction of the Axis.

9 When the form returns click Next. Then select pick two points to define the X direction.



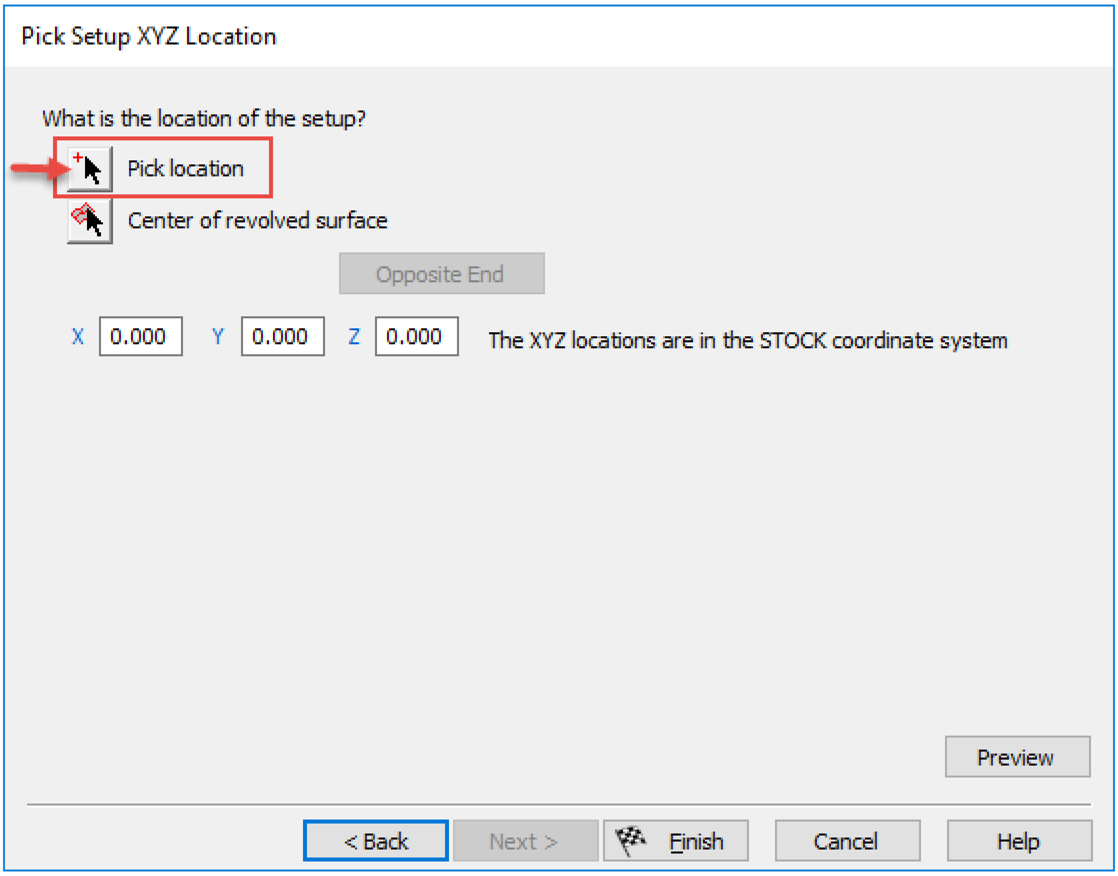


10 Or you can select Rotate X direction 90 degrees around Z Axis. See Below.

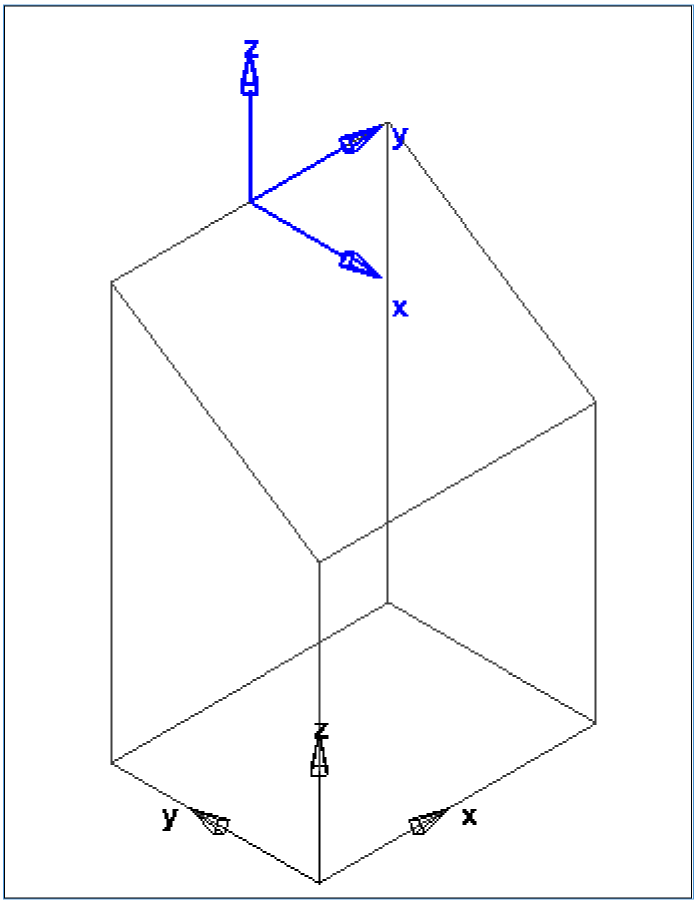


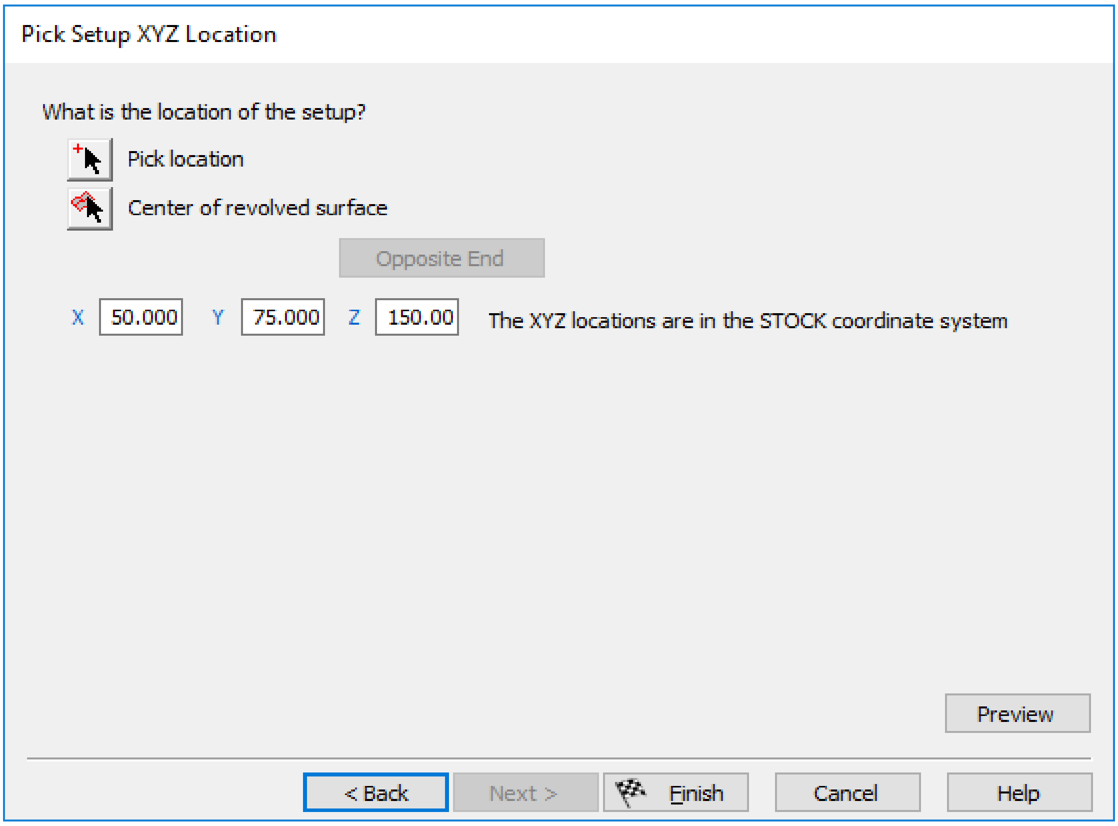
11 Select **Next.**

12 From the next Menu select **Pick location**.



13 Pick middle section top edge.

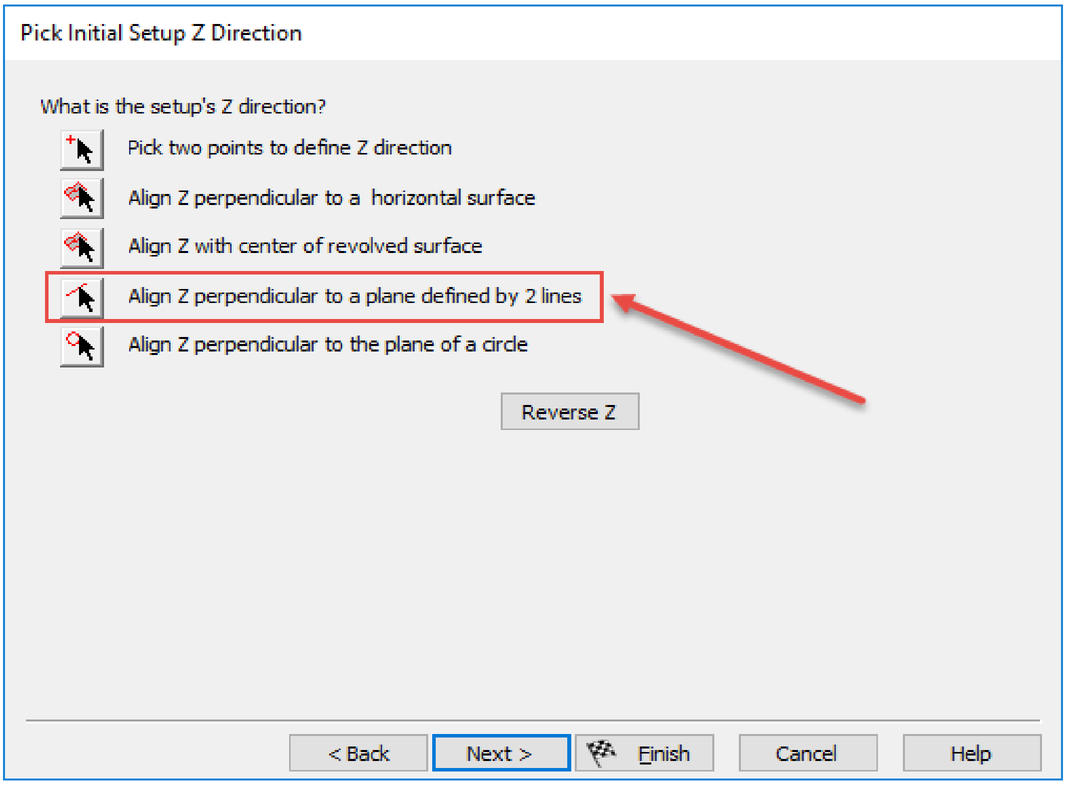


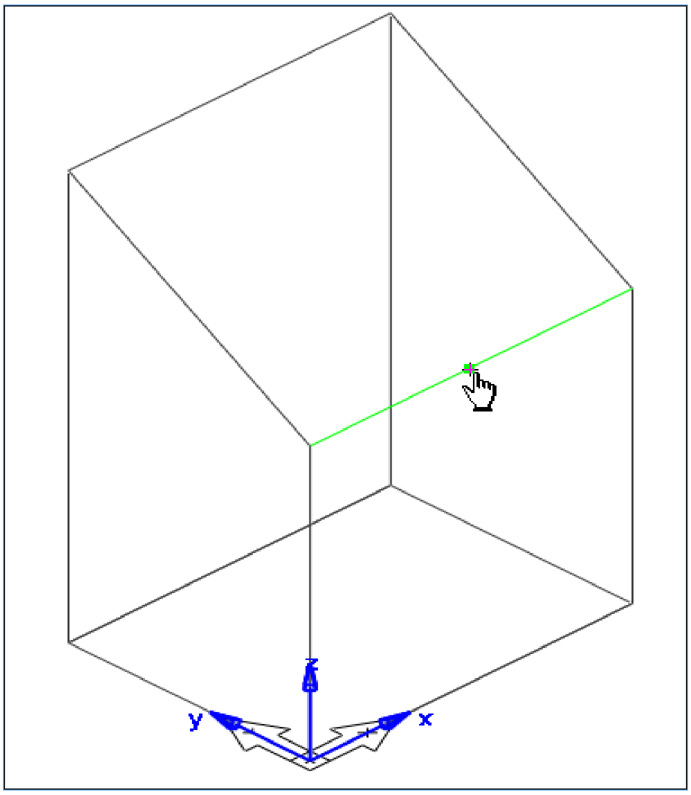


14 Select **Finish.** This will be the new position **Setup. Machine Offset Zero.**

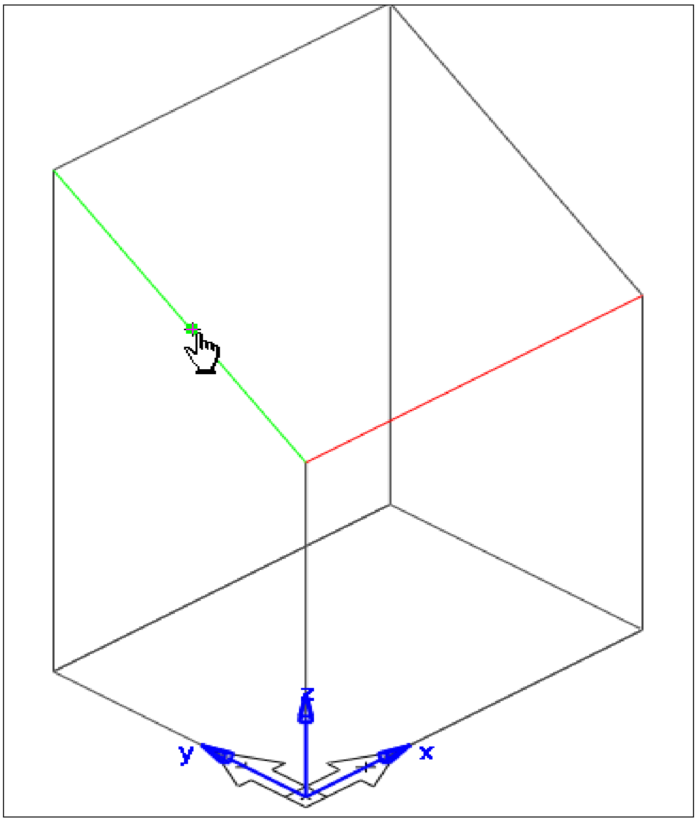
Create a new setup aligned to the angled face

1 Double click on Setup select New followed by Next and then Align to part Geometry. Select Next. And then pick Align Z Perpendicular to a plane defined by 2 lines.



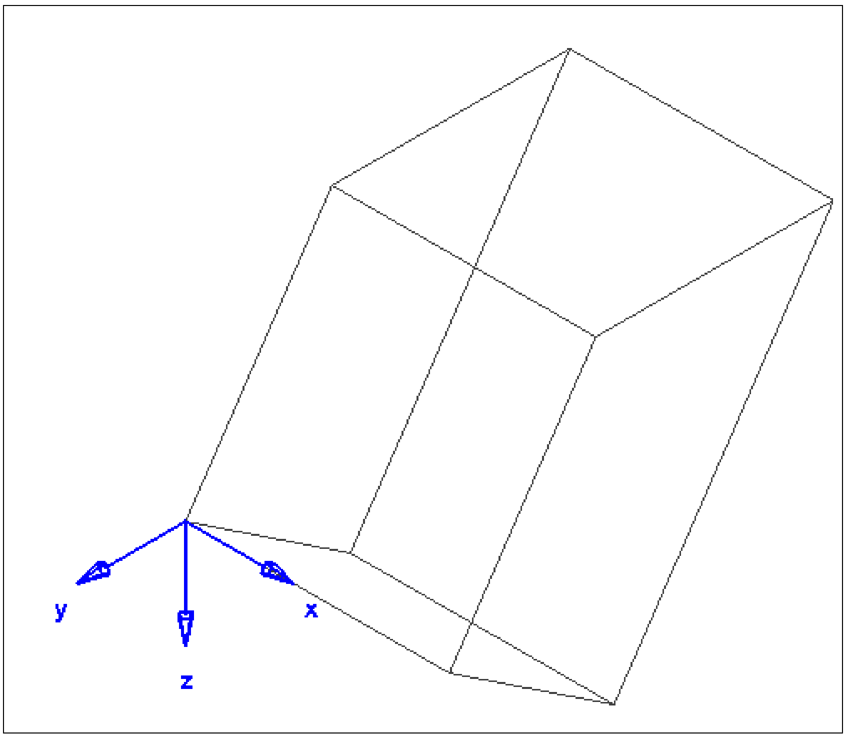


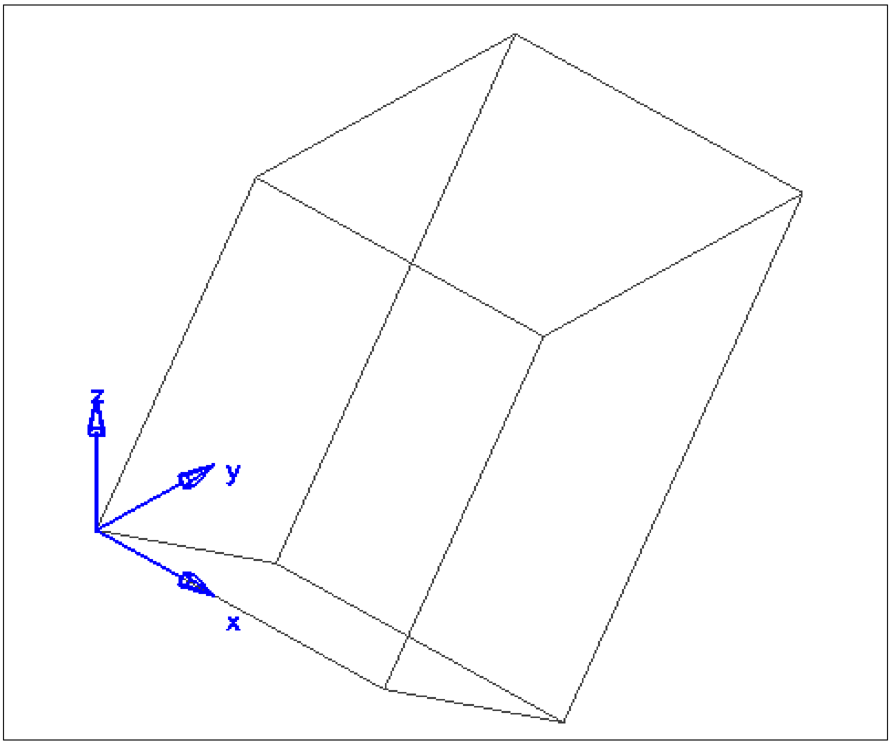
2 **First** Line selection.



3 **Second** Line Selection.

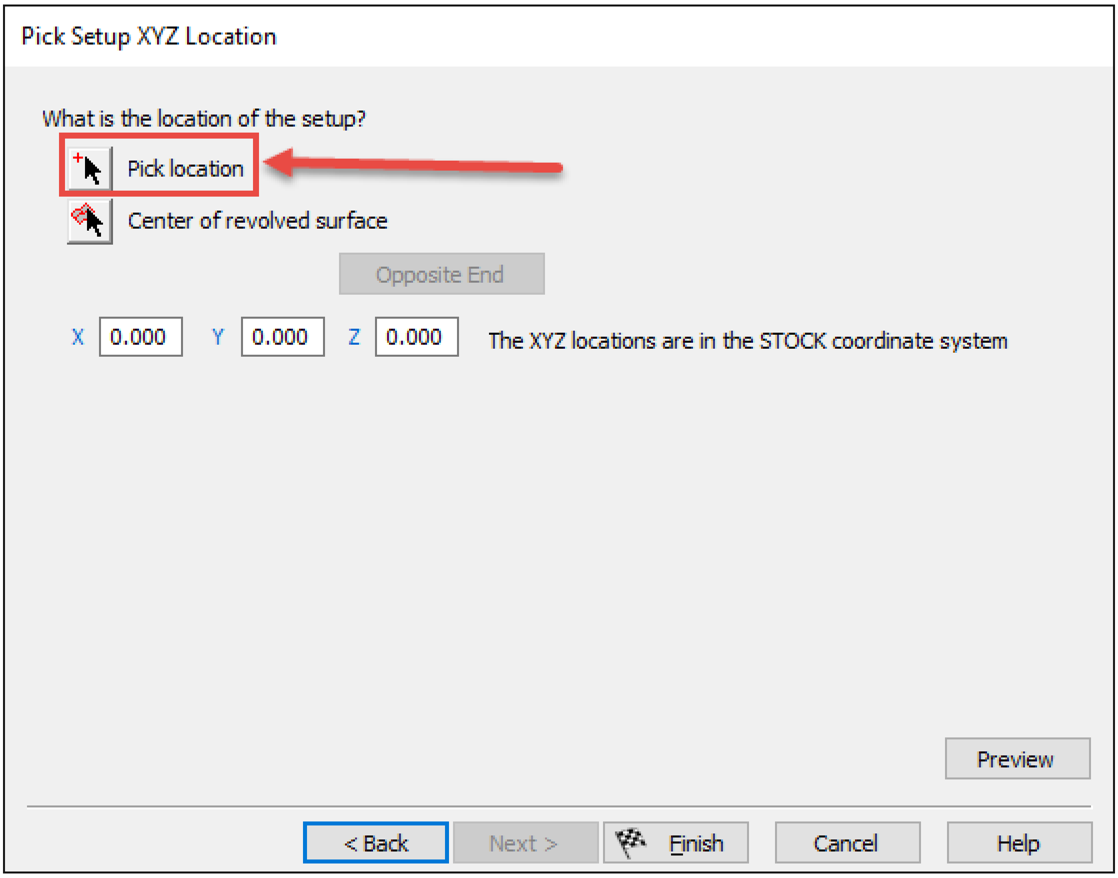
*You can see from the image below that the Z is pointing in the Wrong direction. Select **Reverse Z** on the same menu so that the **Z points** in the correct direction.*

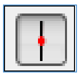


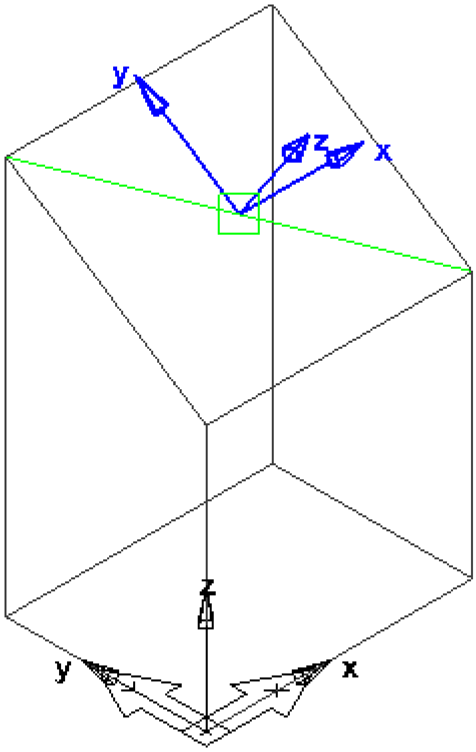


4 Select **Next.**

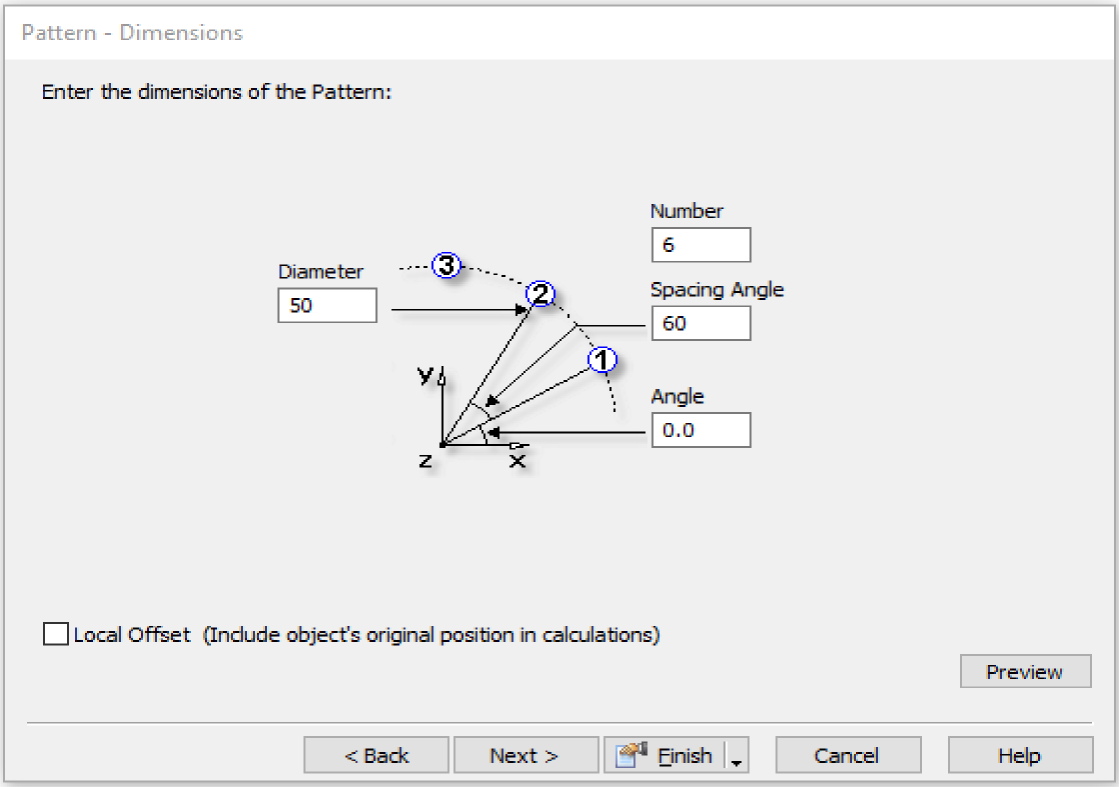
5 Draw a **Line** from a corner to the opposite corner of the angled face

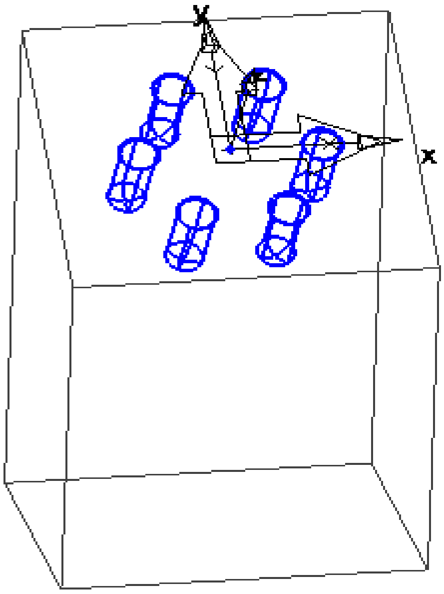


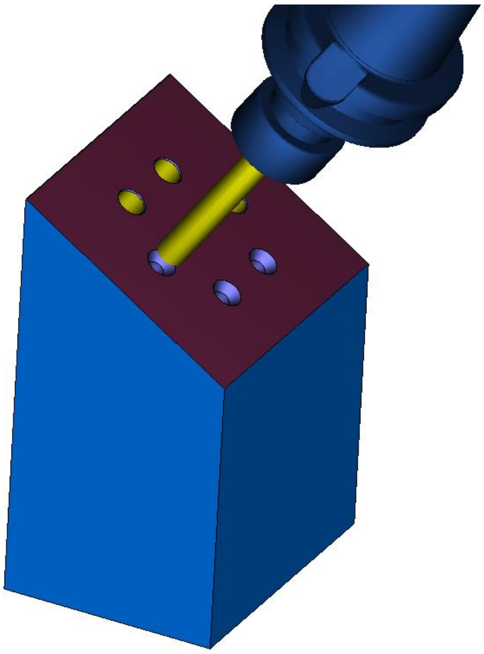
6 Pick **Snap to Midpoint**  snapping to the **Centre** of the new **Line**.



*Using the current **Setup**, use **Face** to machine off the surplus material. Remember to select Rough to machine off the extra material. Then create a **10mm** diameter **Hole Feature 25mm deep** in the middle at **X0, Y0.** Make sure you select **Make a pattern from this feature.** Select **Next.** Then select **Radial in the setup XY plane.** Select **Next.** Enter PCD **Diameter** to **50mm. Number=6, spacing =60.** Select preview to see the result. Select **Next**. Accept the position of **X0, Y0, Z0**. Run a **3D Simulation**. The image should appear like the one below.*


Pattern - Dimensions
Enter the dimensions of the Pattern:
Number
6
Diameter
50
Spacing Angle
60
Angle
0.0
Local Offset (Include object's original position in calculations)
Preview
< Back
Next >
Finish
Cancel
Help

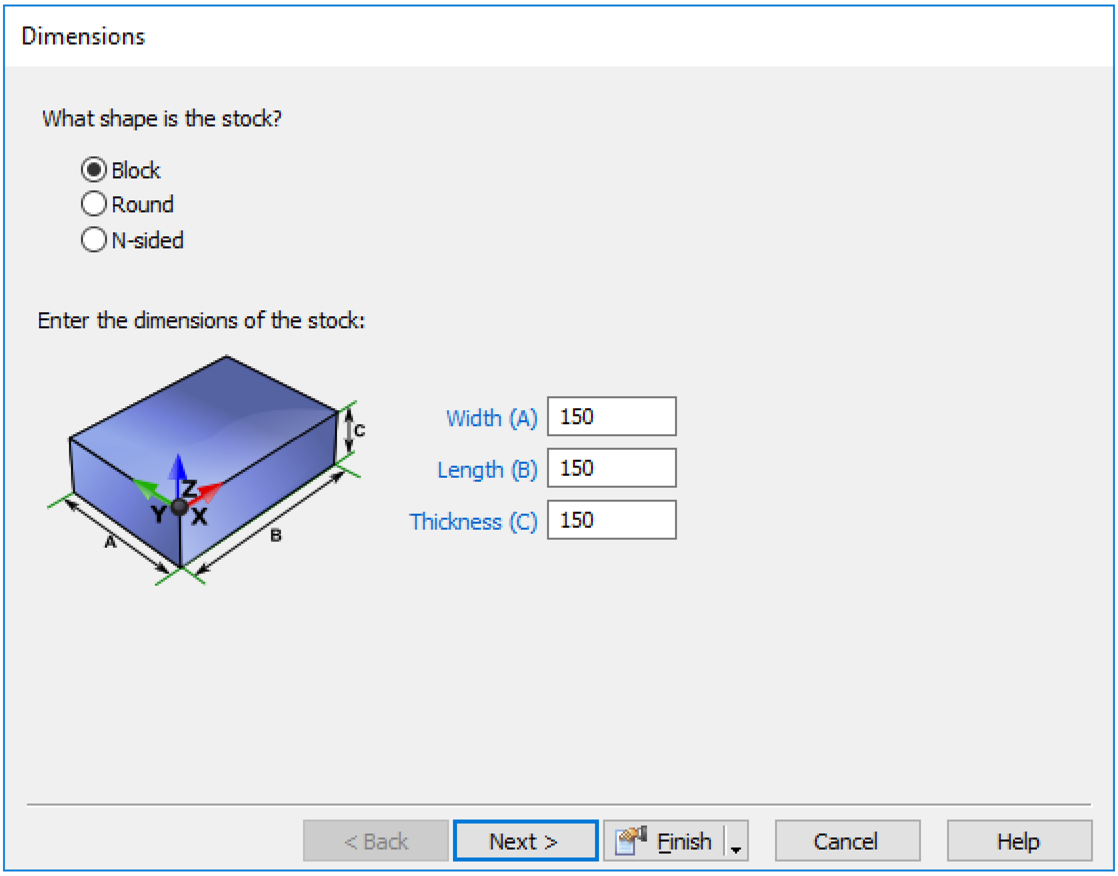




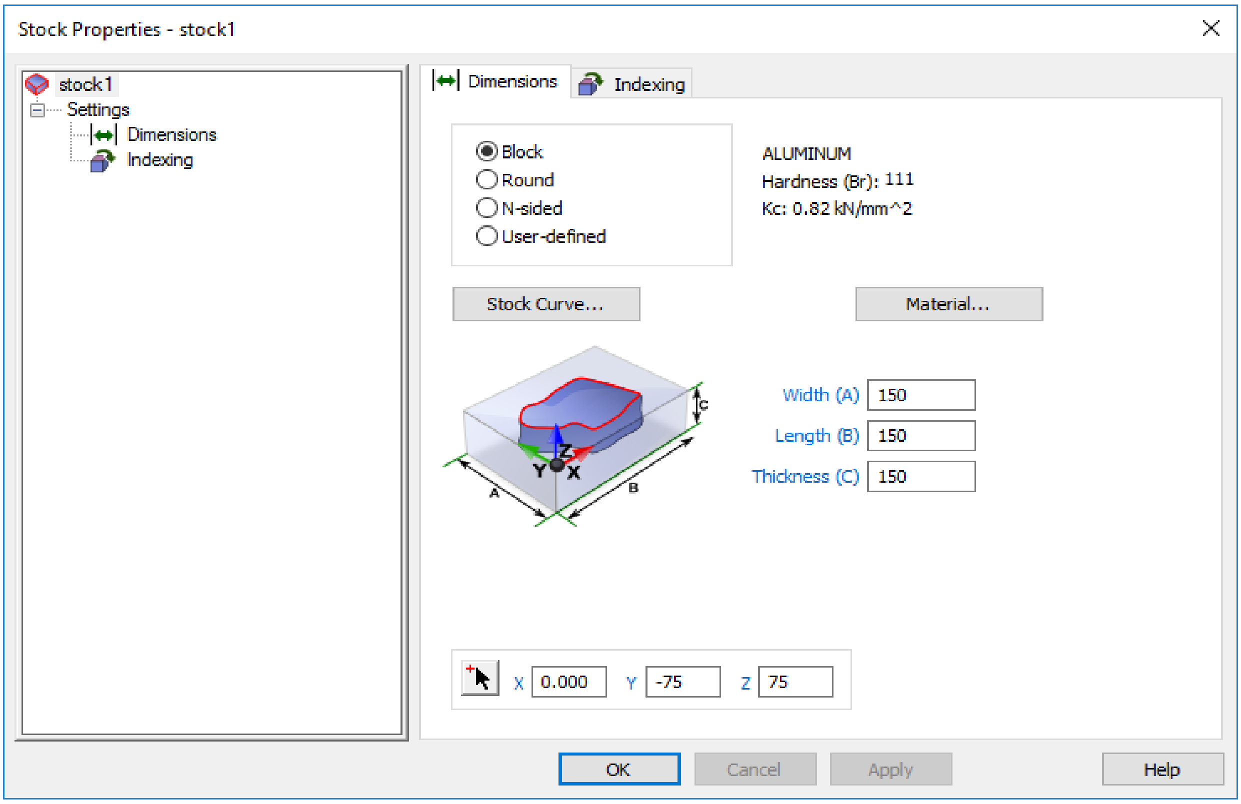
If you get a gouge after running the Simulation have a look at the Tool change position? Adjust the Z position so it will clear the part.

4 Axis index around the Stock Axis (Class exercise)

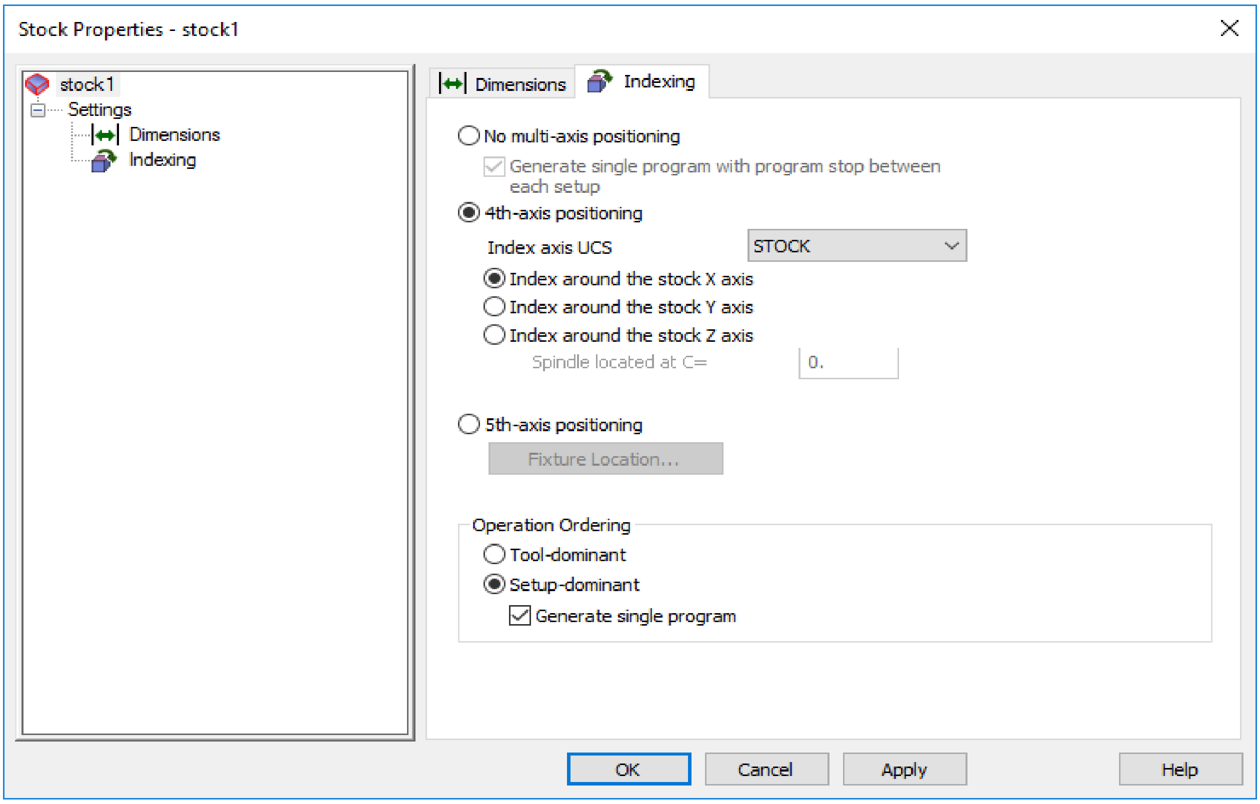
1 Open a **New** Document and create a **Stock** (Block) with the dimensions, **150mm Length**, **Width** and **Thickness**.



2 Select **Finish.** We now need to set the position of Rotation for the **Stock Axis** at **Y-75mm, Z75mm.**

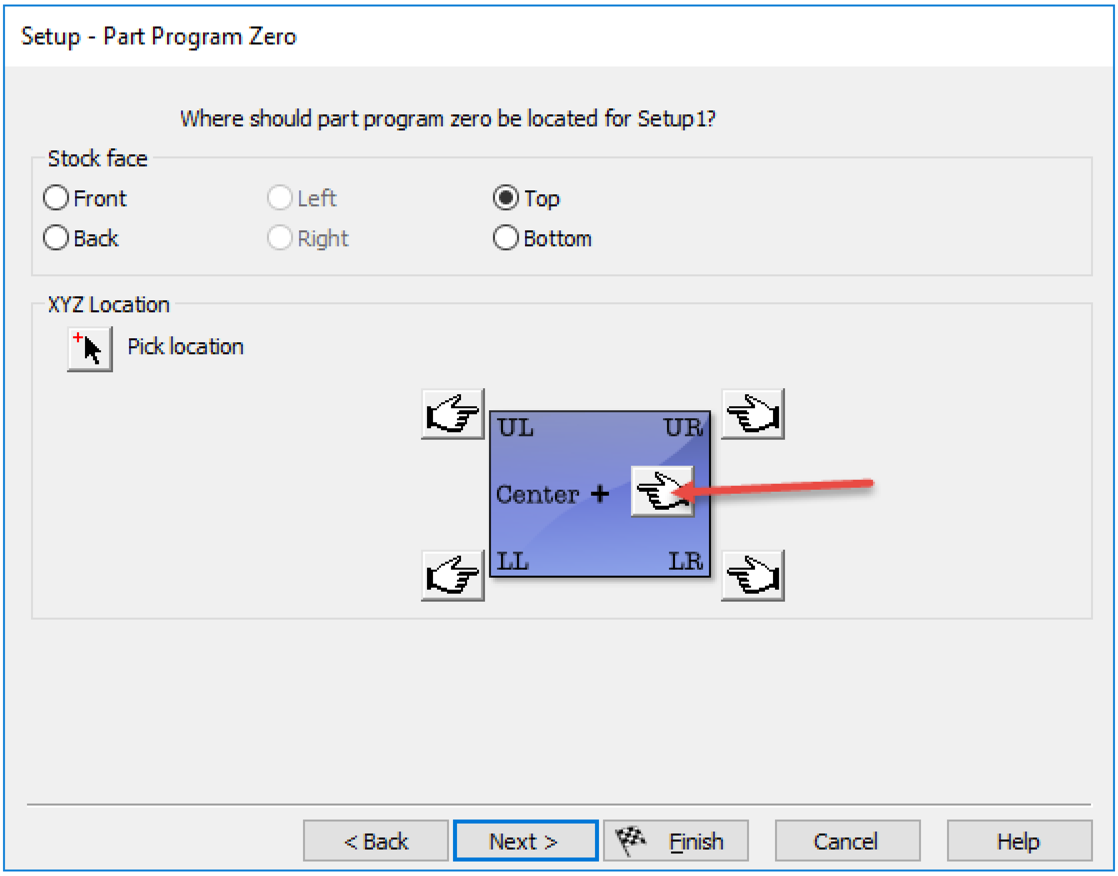


3 Click on the Indexing tab and select 4th axis positioning. Select Index around the Stock X axis and Tool Dominant. Select OK.

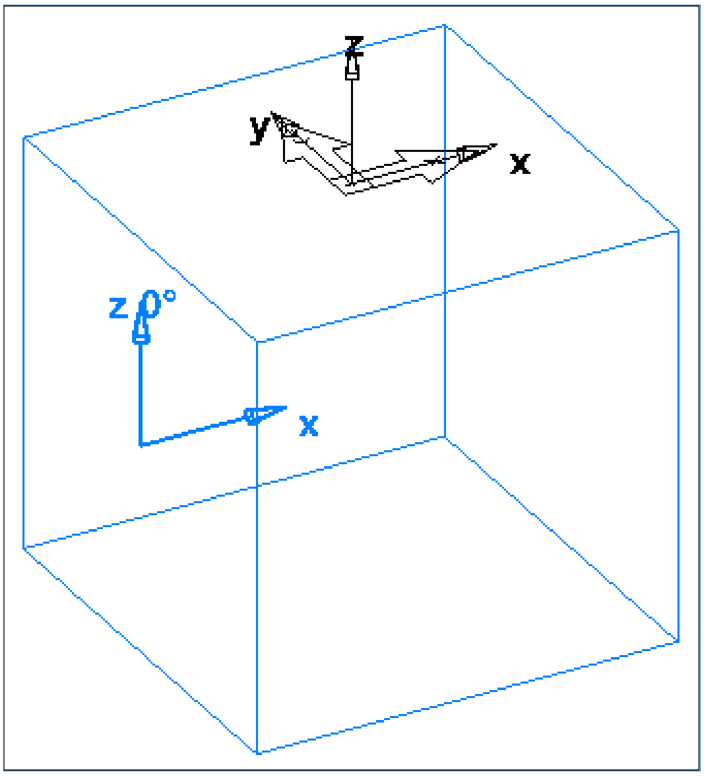


Double click on **Setup1** in **Part View** select **Edit** and then **Next.**

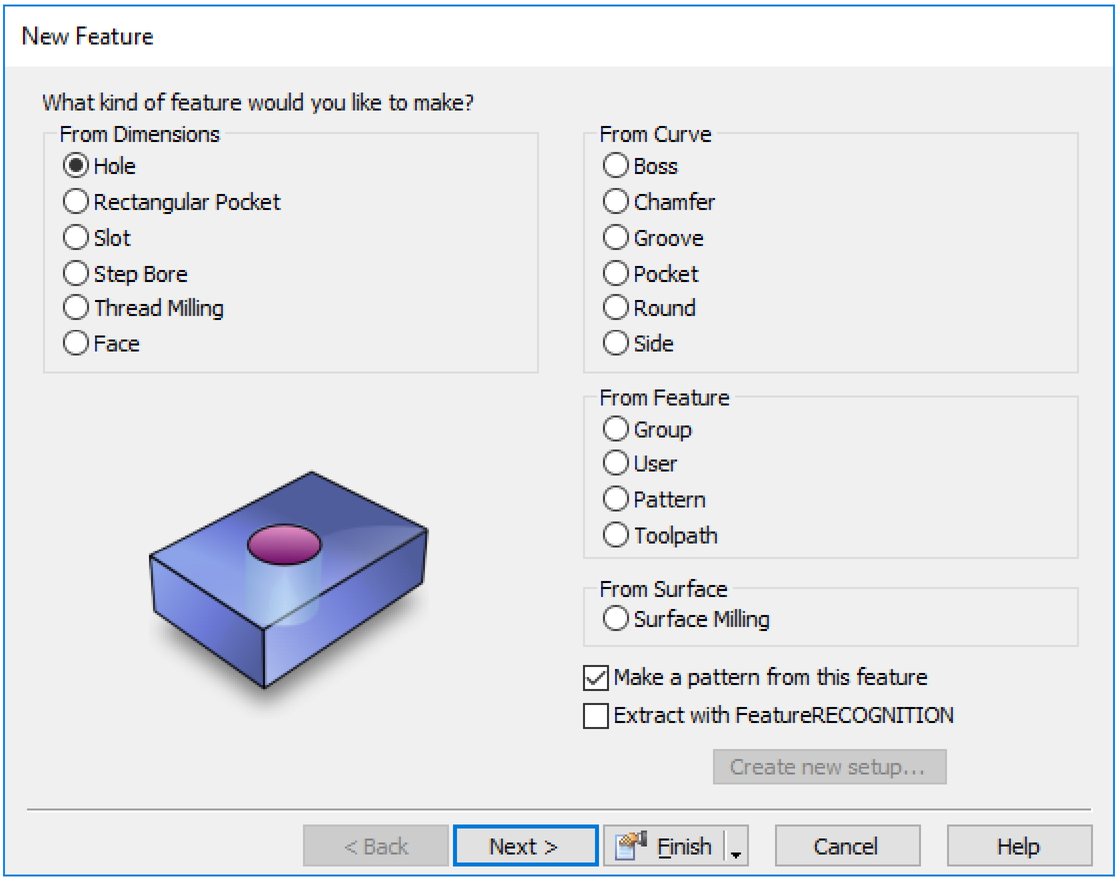
4 Select Align to Stock Face then select Next. Pick TOP and Centre +



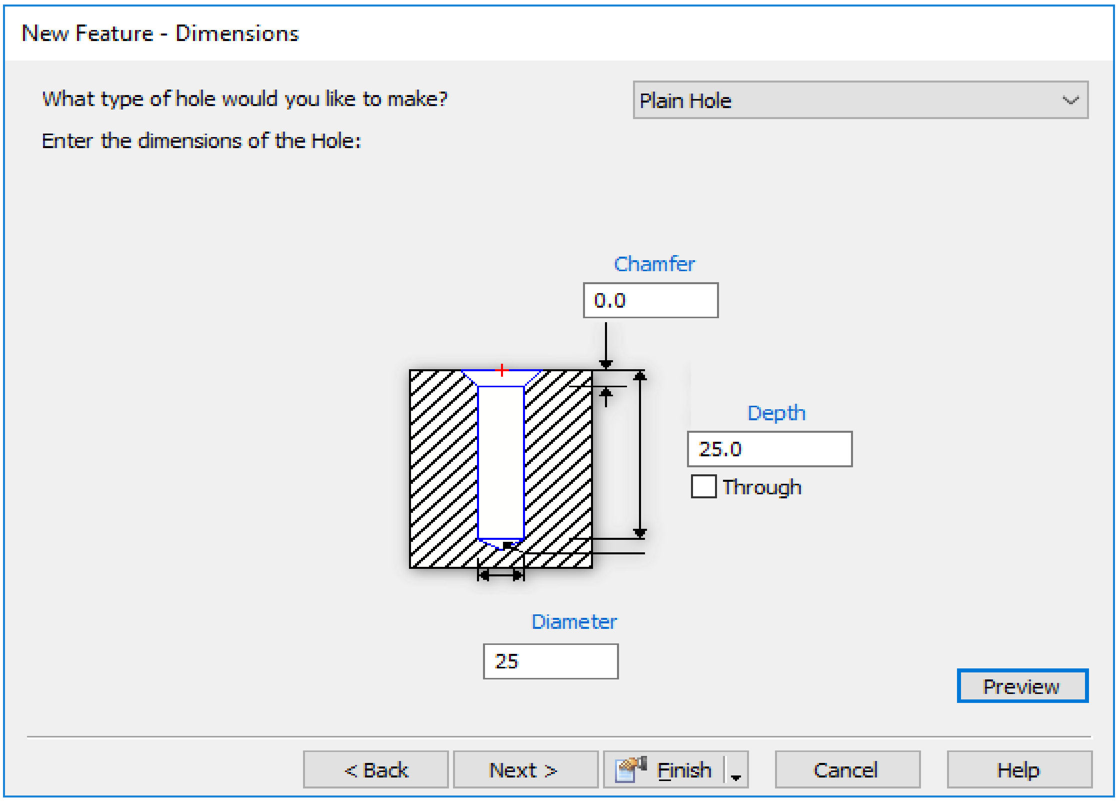
5 With your **Cursor** select **View>Show>Stock Axis**. From the pop down menu.



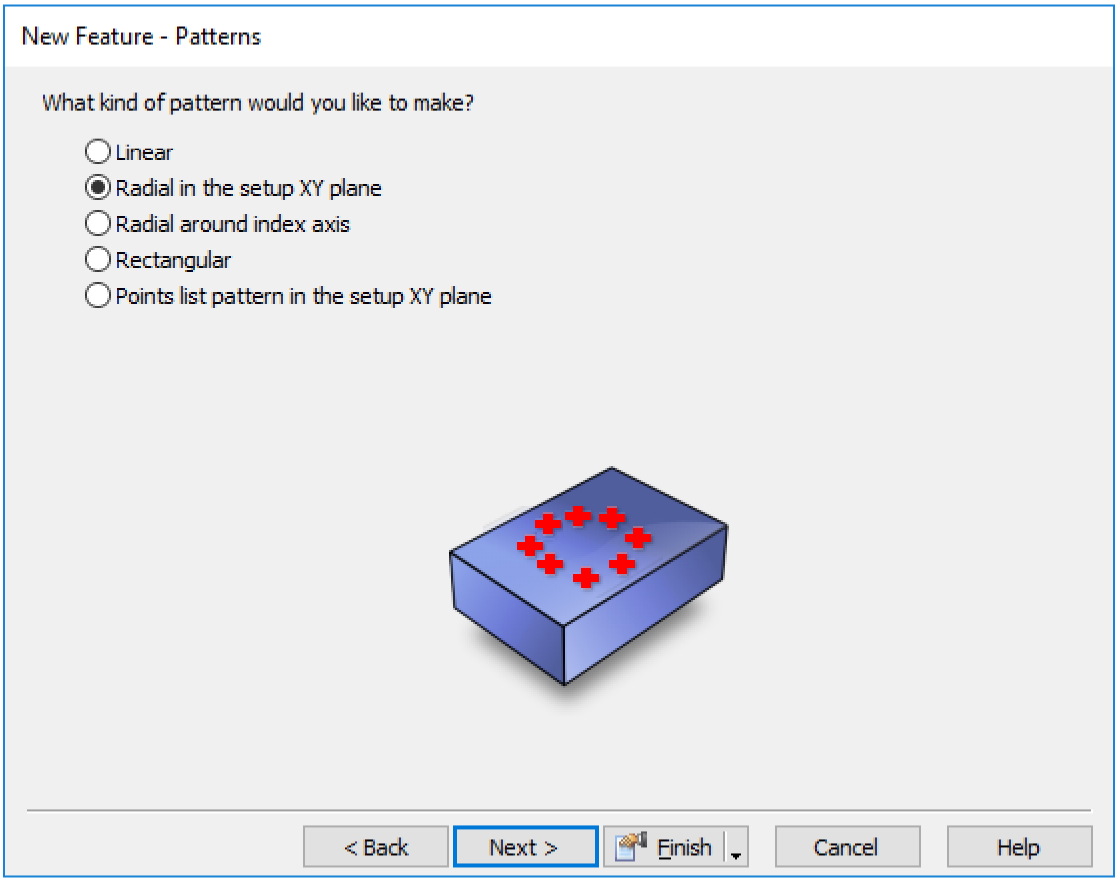
6 Create a **New Feature**, Select **Features** in **Steps** or Select **Ctrl** + **R**. Select **Hole** from Dimension and **Make a pattern from this feature**. Select **Next**.



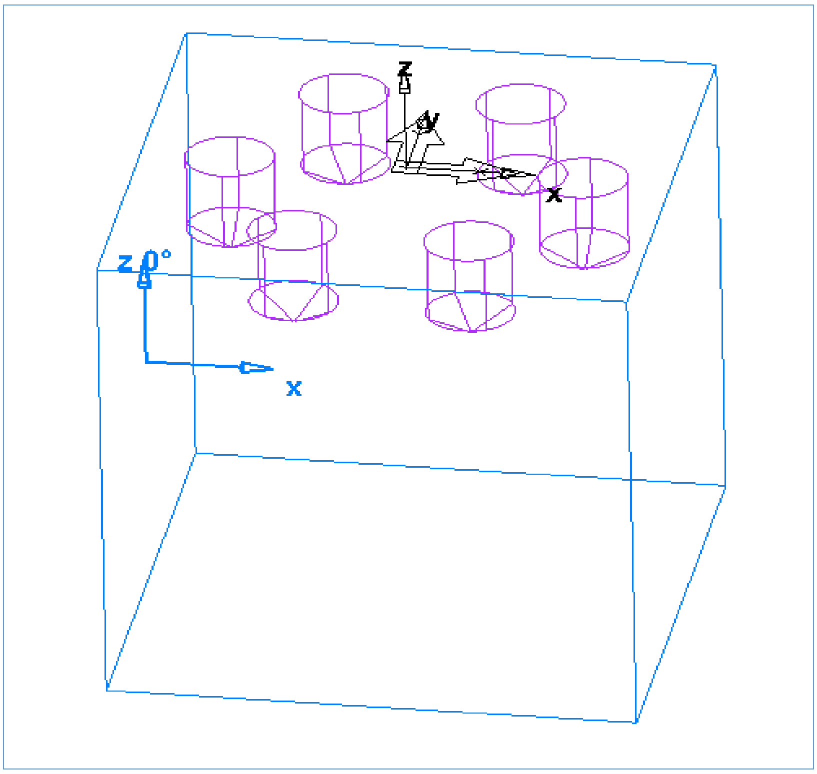
7 Enter **Plain Hole**, Diameter and Depth of **25mm** and check through. Click **Next**.



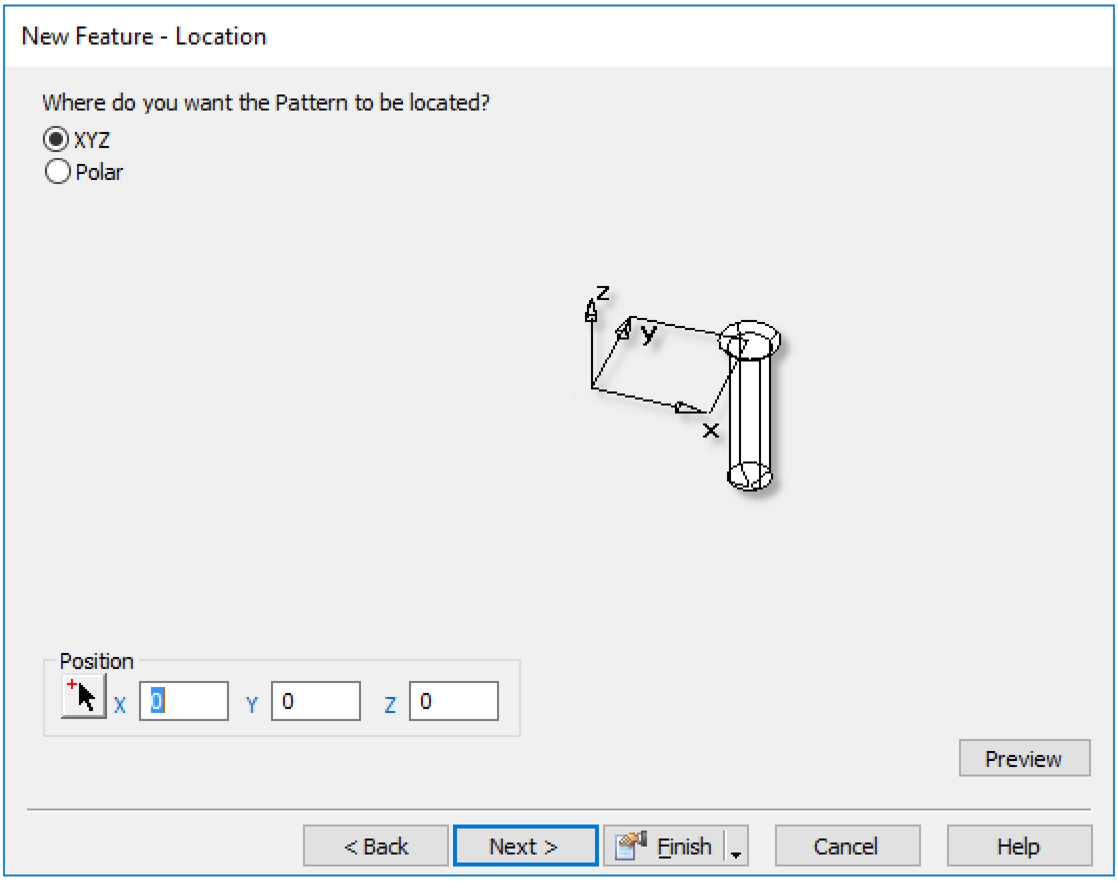
8 Pick Radial in the Setup XY plane. Select Next.



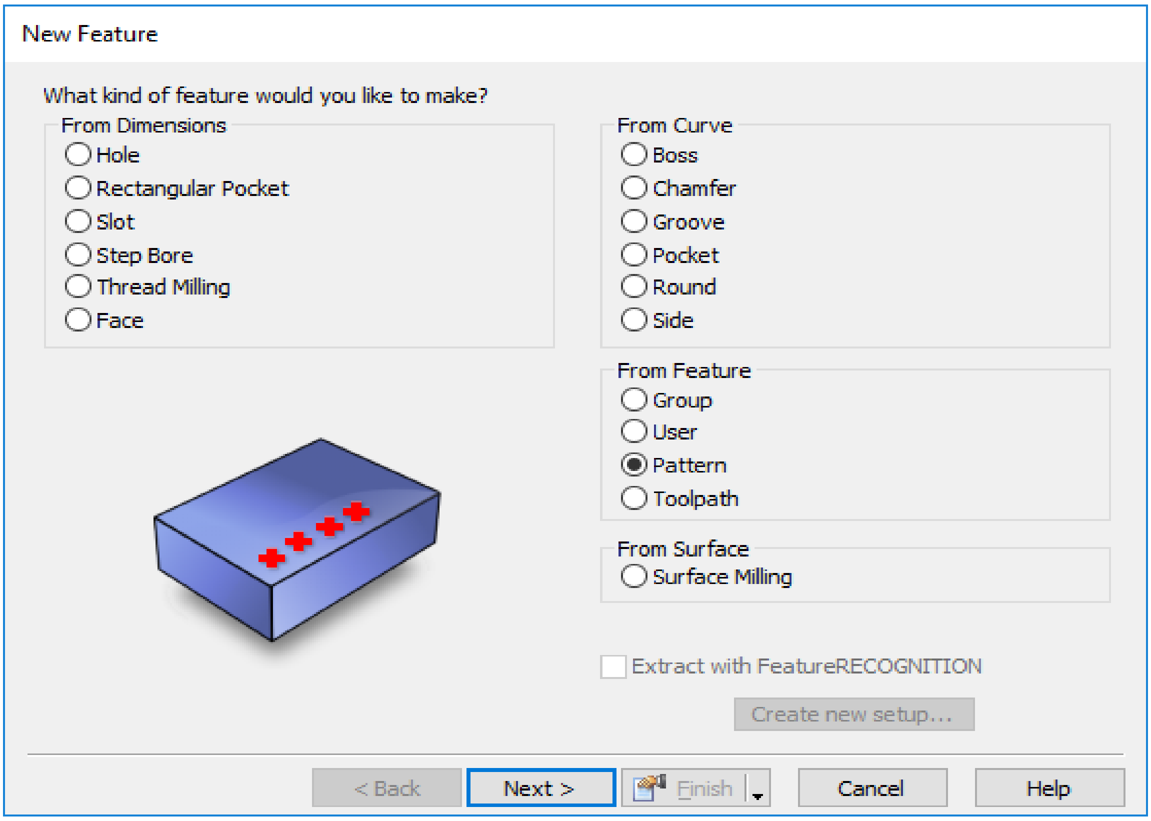
9 Set the following options: - Diameter **100mm,** Number **6**, Spacing **Angle 60**, **Angle 0**. This creates a pattern on one face and will be used to create the same pattern on the other 3 faces, a pattern within a pattern. Next.



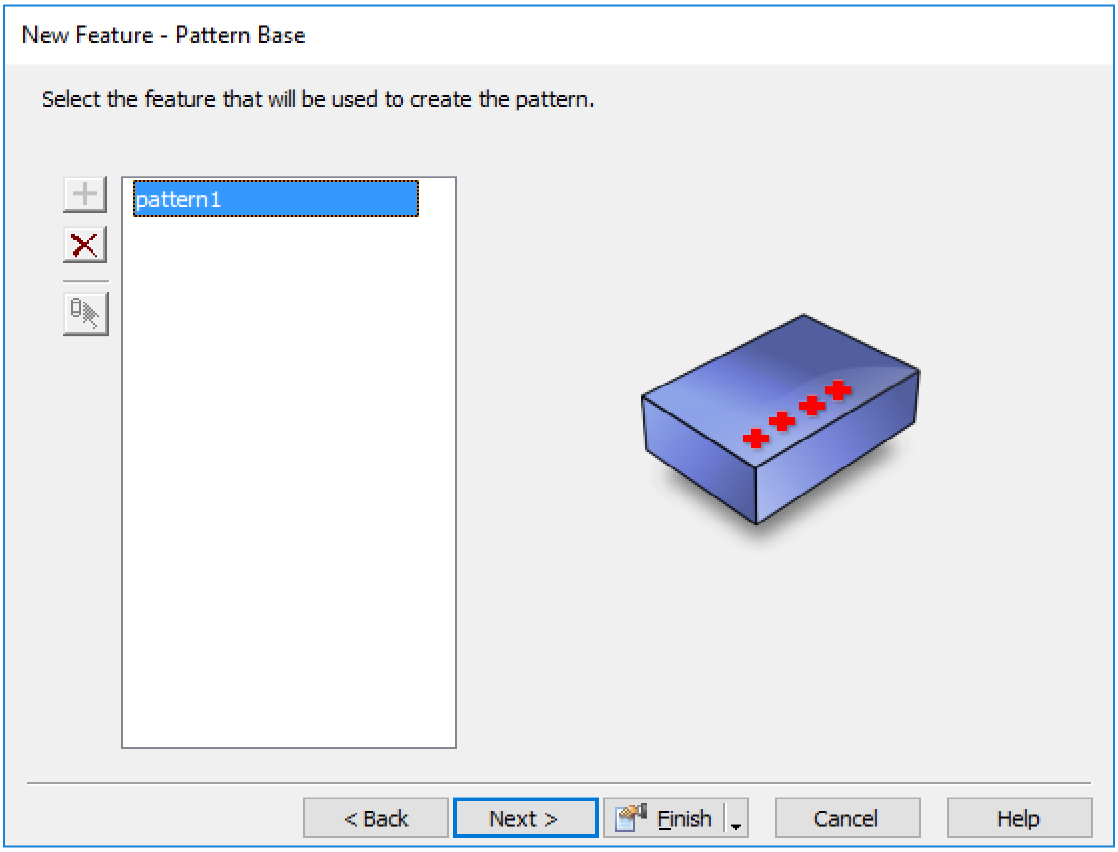
10 Location is set to **X0, Y0, Z0** Select **Finish**.



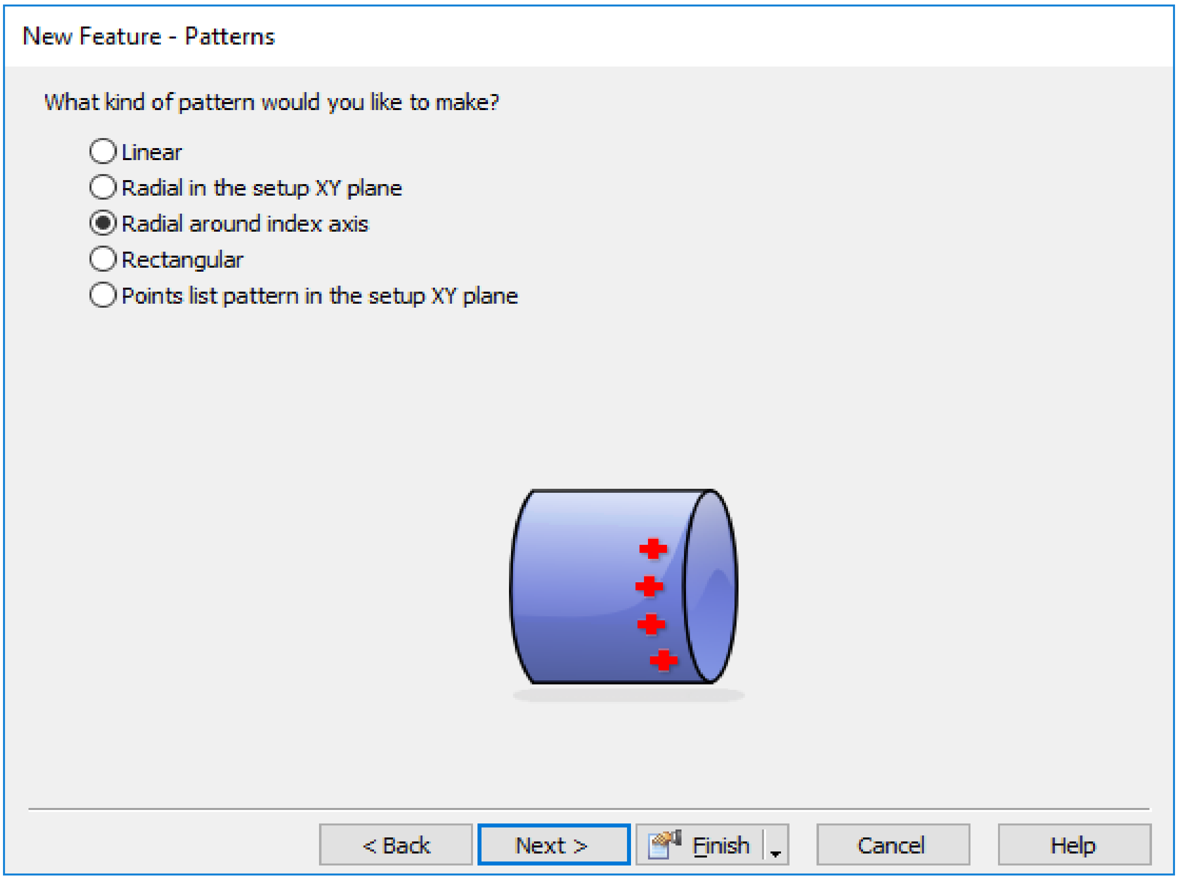
11 Create a **New Feature**. Select **Ctrl** + **R. Pattern** will be automatically selected. Select **Next.**



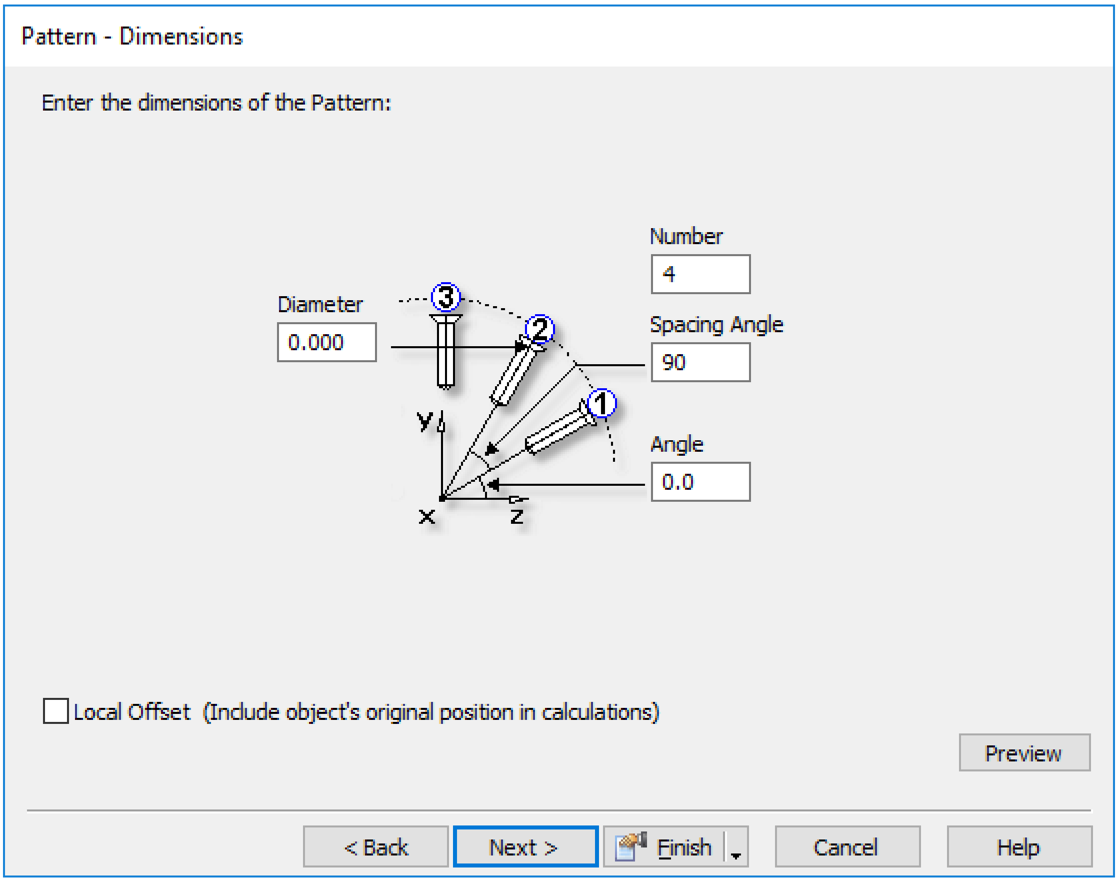
12 Select Pattern1



13 Select Radial Around the index axis. Select Next.

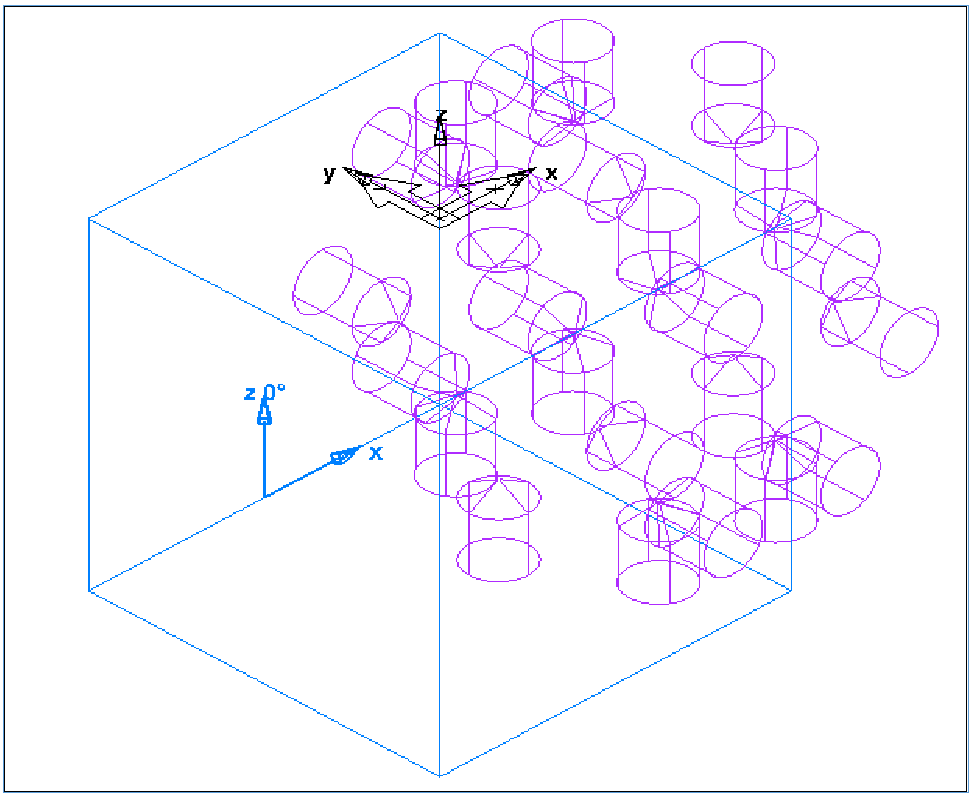


14 Number **4**, Spacing Angle **90**. Select **Next**.

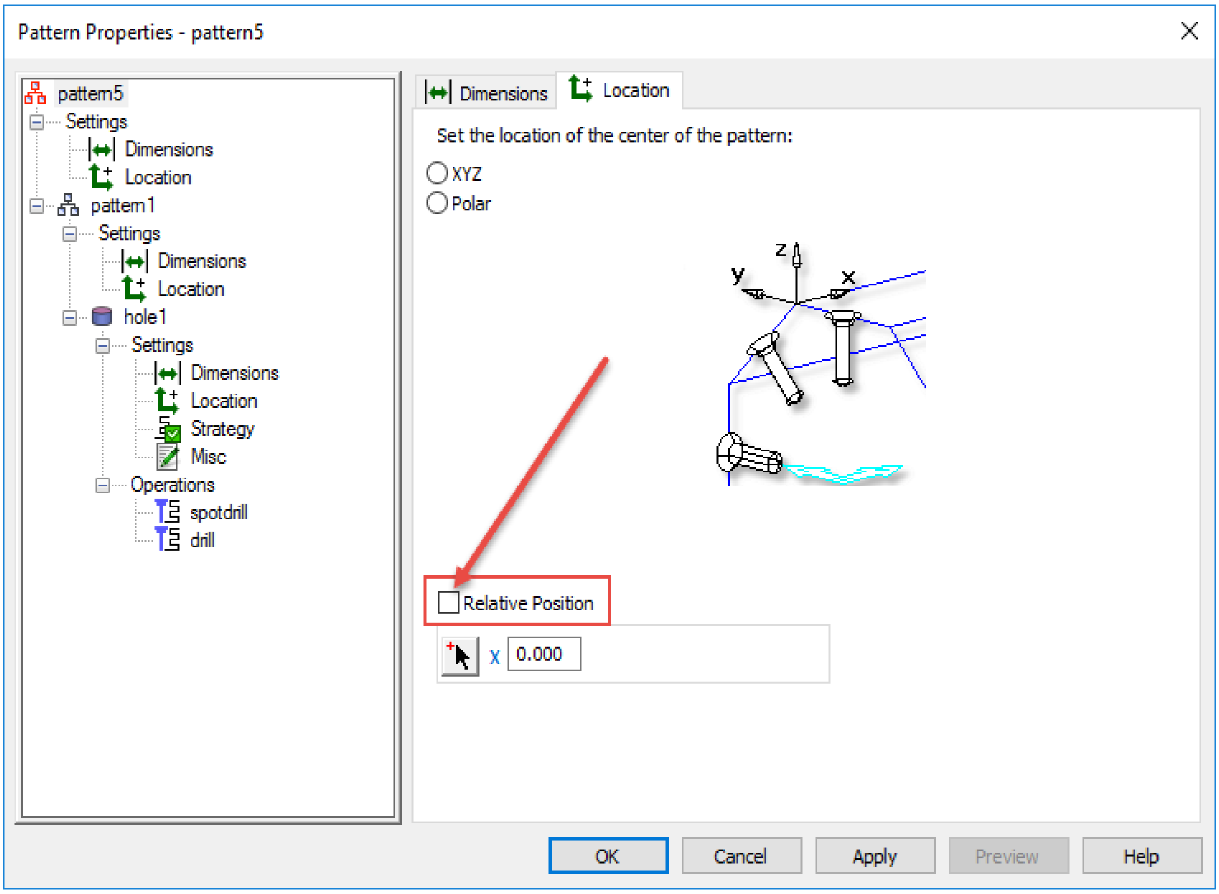


15 Select **Finish**.

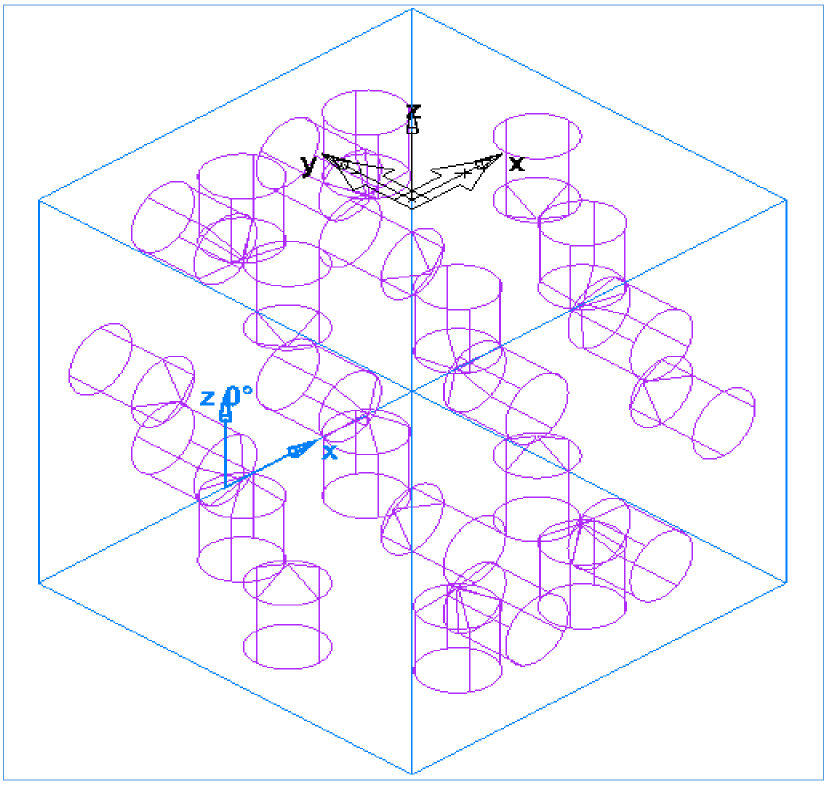
You will notice that the Pattern is out of position.



Select Location in the Pattern properties. Then **unselect** the **Relative position**.



16 This places the (6) Hole pattern on four faces positioned the same as the original pattern.

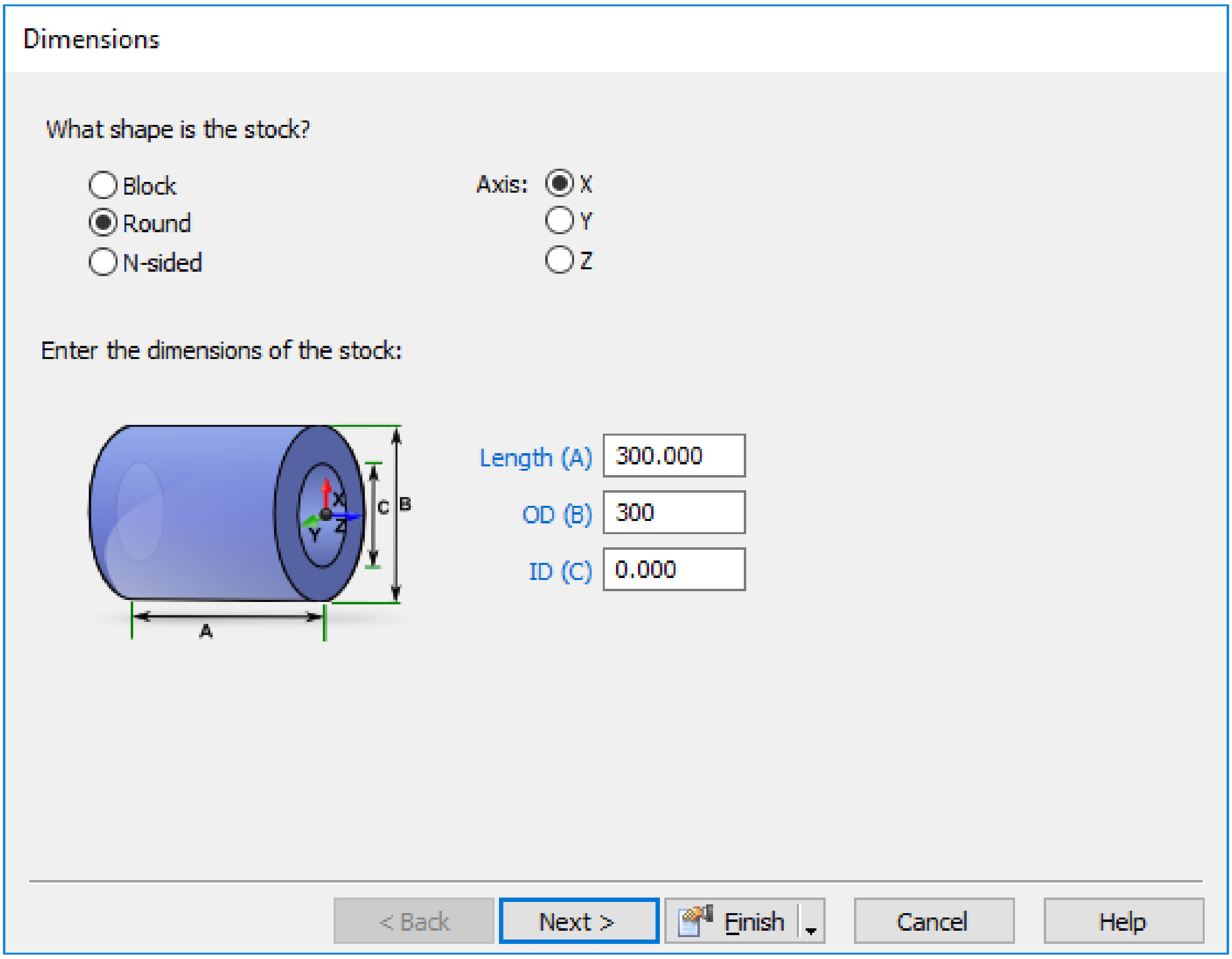


Round Stock

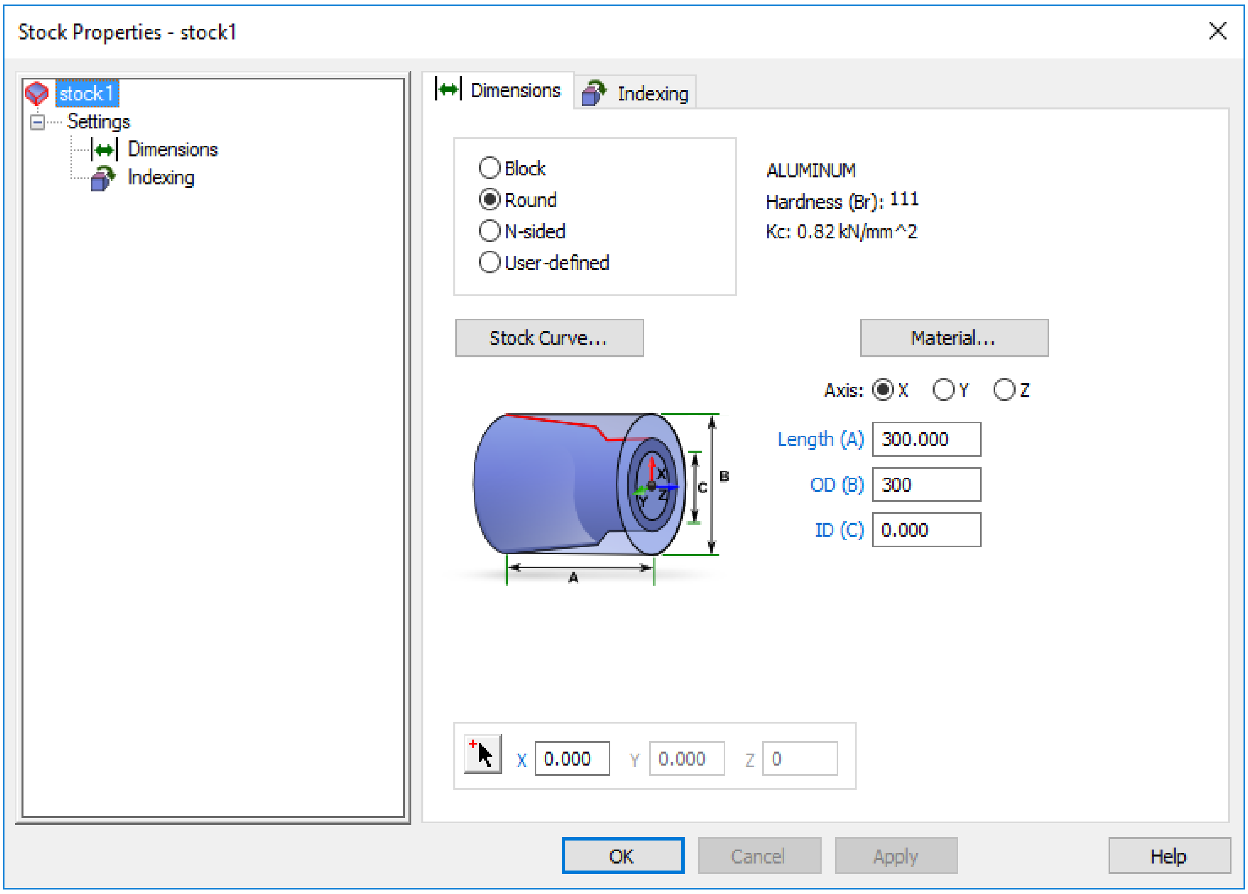
- The **Stock Axis** is in the centre of the part and so is the **Setup**. However, the **Setup** and **UCS** may be moved for programming purposes. The same rules apply to round parts as those of the block stock parts. Features **CANNOT** be **Transformed/rotate/copied** around the **Stock Axis**. One **Setup** is commonly used, but other **setups** may be added if needed. You may also **wrap** a **Pocket** and **Groove** features, including **Engraving**, around the **Stock Axis**.
- To **Wrap** a **Pocket** you must first create the **Pocket**, and if the **UCS** is at the centre of the part, you must **translate** the feature in the Z direction to put the top of the feature at the outer radius of the stock.
- On a round part the **Feature** can be applied as normal and then wrapped which is selected from within the feature. The "**Wrap feature around X axis**" check box becomes visible on the **Dimension** tab when **4th axis is turned on**. If it is not visible usually two things happen, either 4th axis is turned off (stock, indexing tab) or the feature was created at the centre of the part and you failed to move it up to the desired radius on the part, it cannot wrap around the centre.

Wrapping a Groove (Optional Exercise)

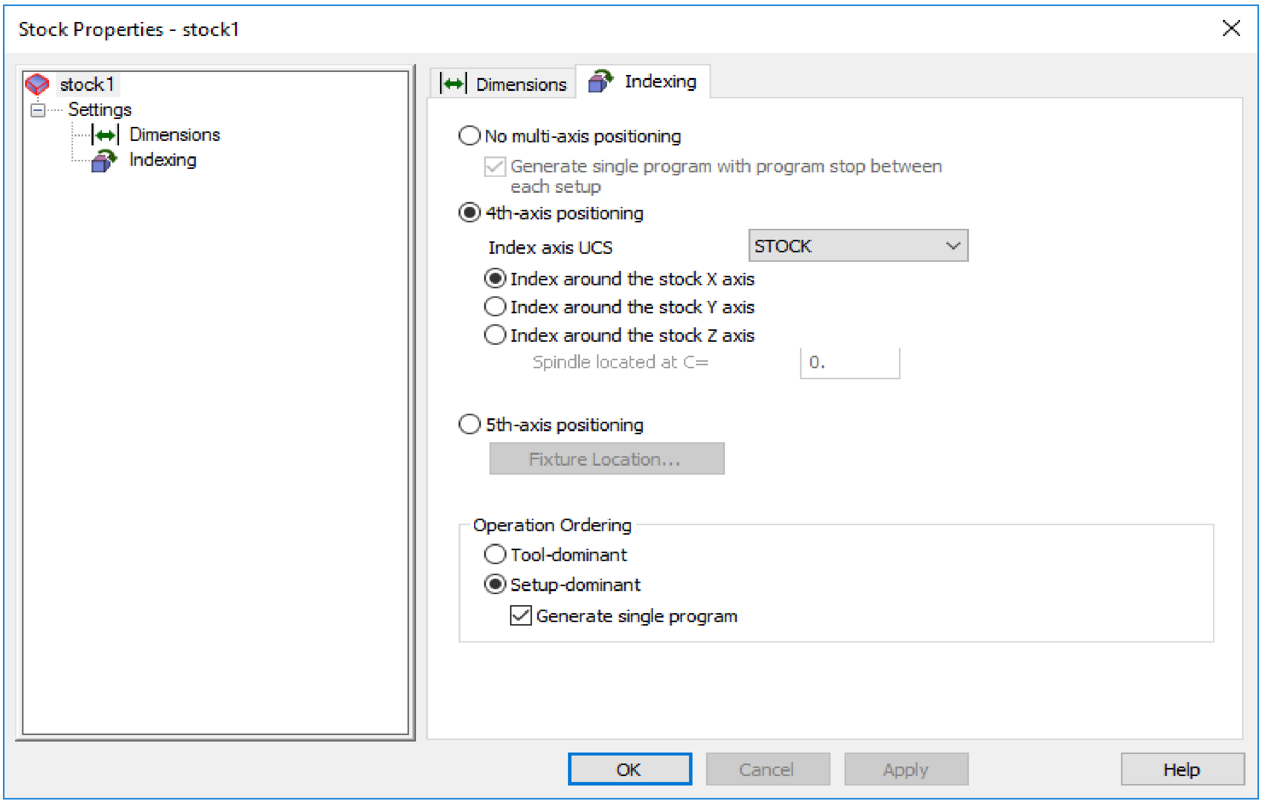
1 **Open** a New Part Document - Milling Setup Metric, **Round Stock, X axis, OD 300mm, Length 300mm, and ID 0.**



2 Select **Finish** and the following menu will appear.

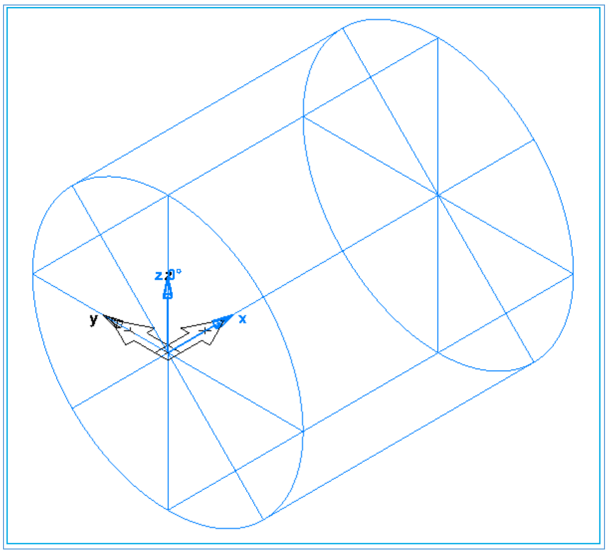


3 On the Indexing Tab select 4th axis Positioning and Index around the STOCK X Axis.



4 Select **OK** to close the form.

5 Click on **View** from the top Menu bar. **Show** > **Show Stock Axis.**

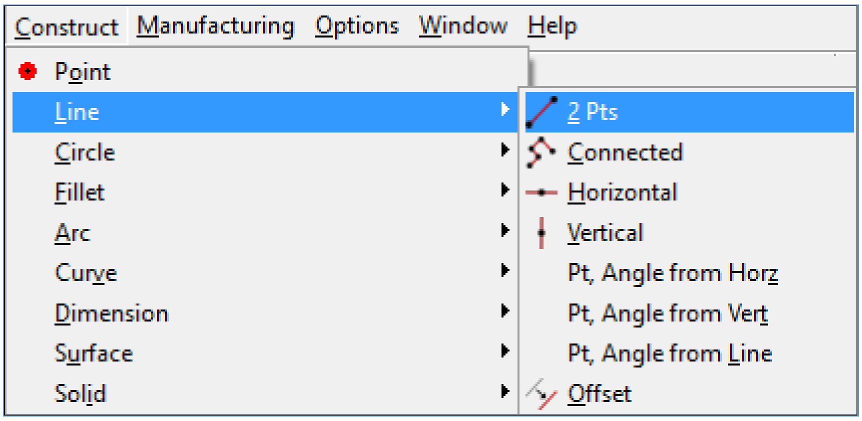


*The **Stock Axis** and the **Setup/UCS** are together in the centre of the part. The stock and length of the stock are aligned with the X axis.*

*When wrapping a groove you must first create the geometry for the grooving feature, and it **must** lie in the **XY Plane**.*

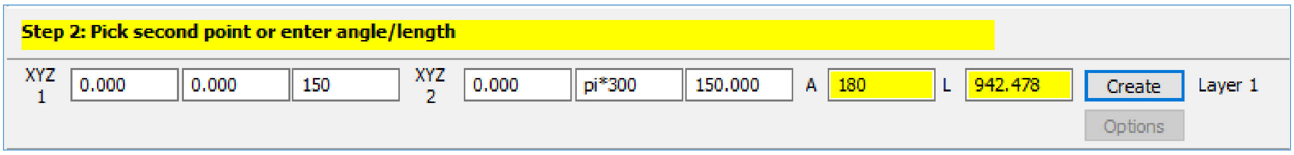
***Wrapping** a **Groove** is like wrapping a label around a tin can. The values entered as indicated below, will start the groove with the centre of the tool being on **X0** end with no movement in the X direction. The Y is **Pi*300** which will determine the length of the geometry and the part will then rotate **360 degrees**. The **Z150mm** puts the top of the groove on the outside radius of the **Stock**. The **Geometry** is the path for the **Groove**, but does not create the **Groove**.*

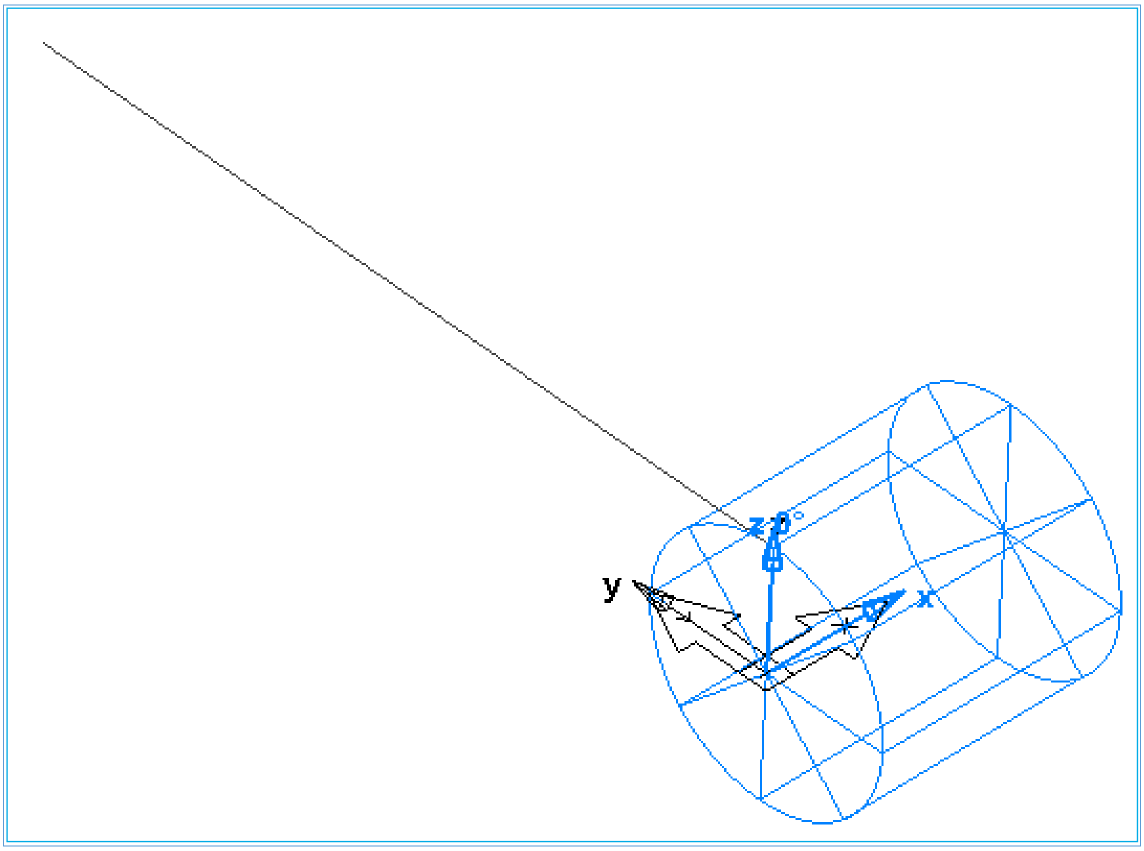
6 From **Construct** select **Line>2 pts** draw a line to the dimensions shown below.



7 Type in the following co-ordinates. XYZ 1 **X0.0 Y0.0 Z150.0**

XYZ 2 ***X0.0 Y Pi*300*** **Z150.0 A180** Select **Create.**



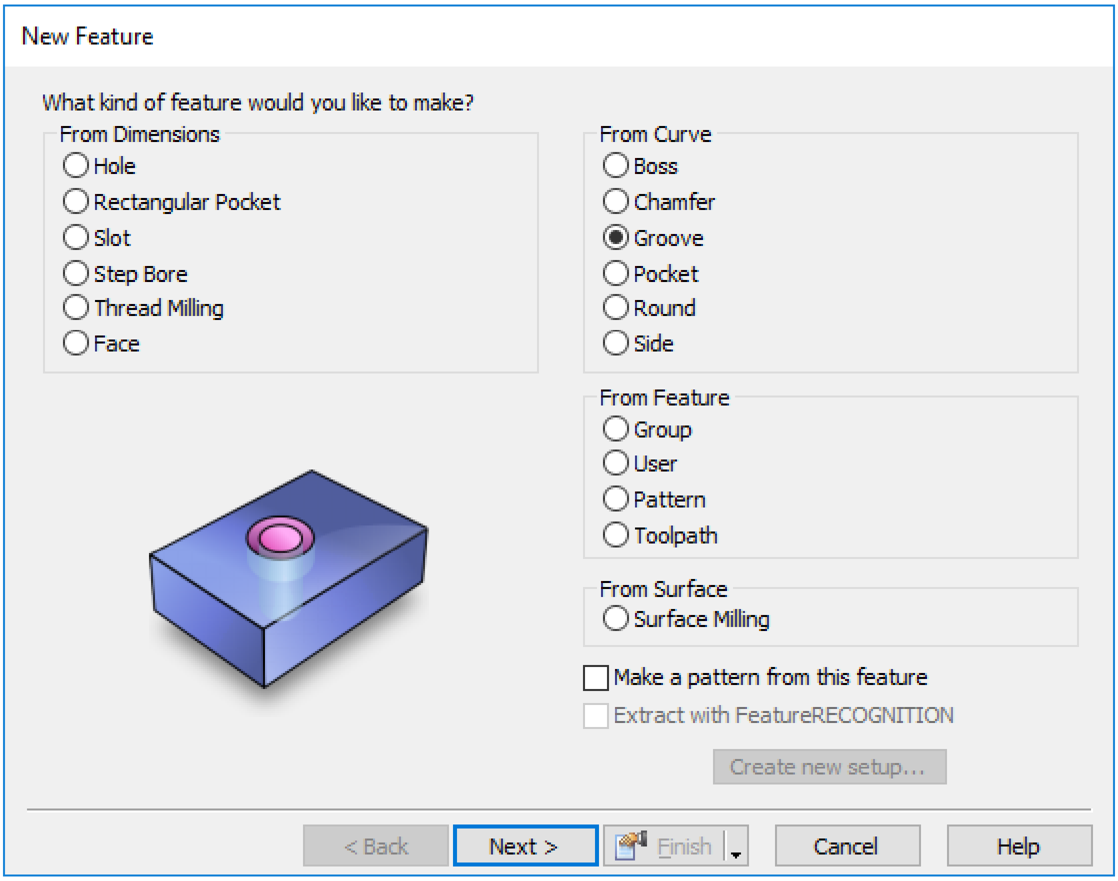


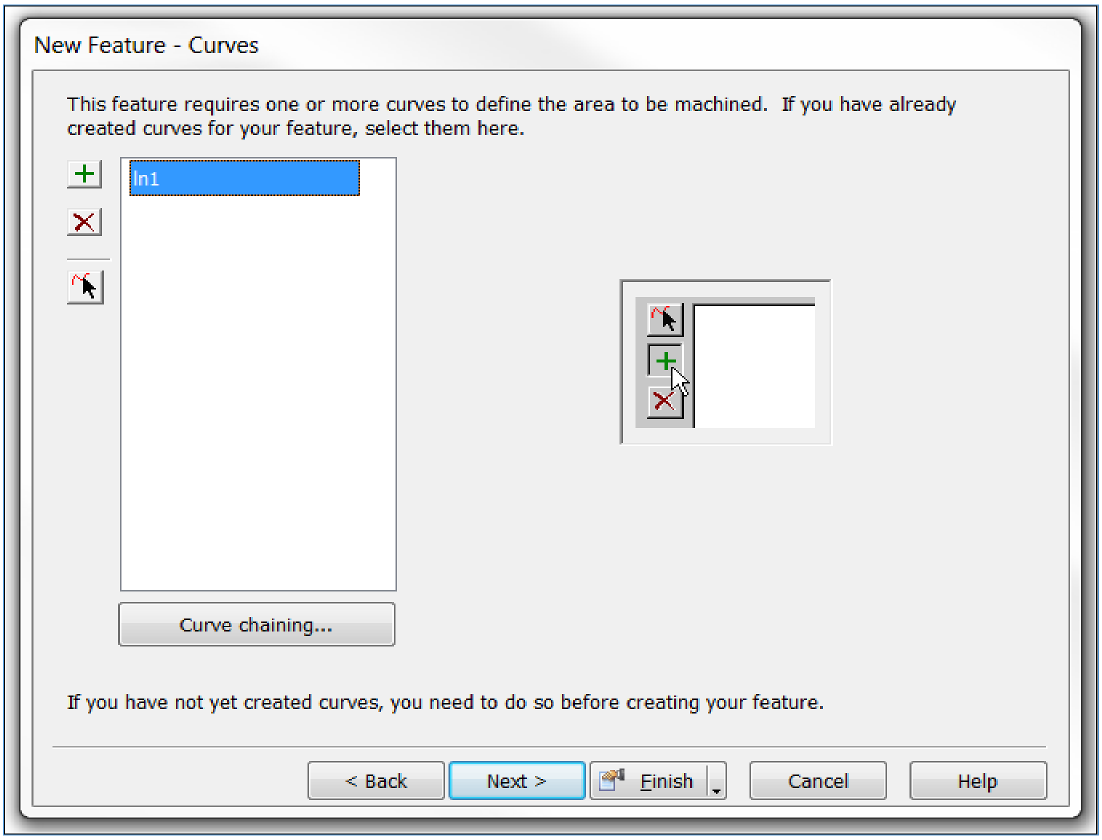
XYZ 1 *sets the start point of the groove tool at the top or radius of the stock centred on the end where the Setup X 0.0 is located. Y 0.0 means it starts on 0 in the Y direction.* ***Z150mm*** *means it starts at a* ***150mm*** *elevation in the Z direction above the centre on the* ***150mm*** *radius of the stock.*

XYZ 2 *is the end of the groove geometry and also determines whether it travels in the X and how many times it wraps around. X 0.0 – This means the tool will not travel in the X direction. If there is a different value in X then the tool will move to that distance in the X direction as the part rotates.* ***Y Pi*300*** *- This means the part will rotate once around the part. Notice that the value is* ***Pi*300****, this means* ***Pi times the diameter*** *which is the circumference of the* ***300mm*** *diameter stock and* ***FeatureCAM*** *understands* ***Pi*** *and its value and the *(asterisk) is* ***Multiplication****.*

Z150mm *- This means the tool ends up at the same* ***Z elevation*** *as when it started.*

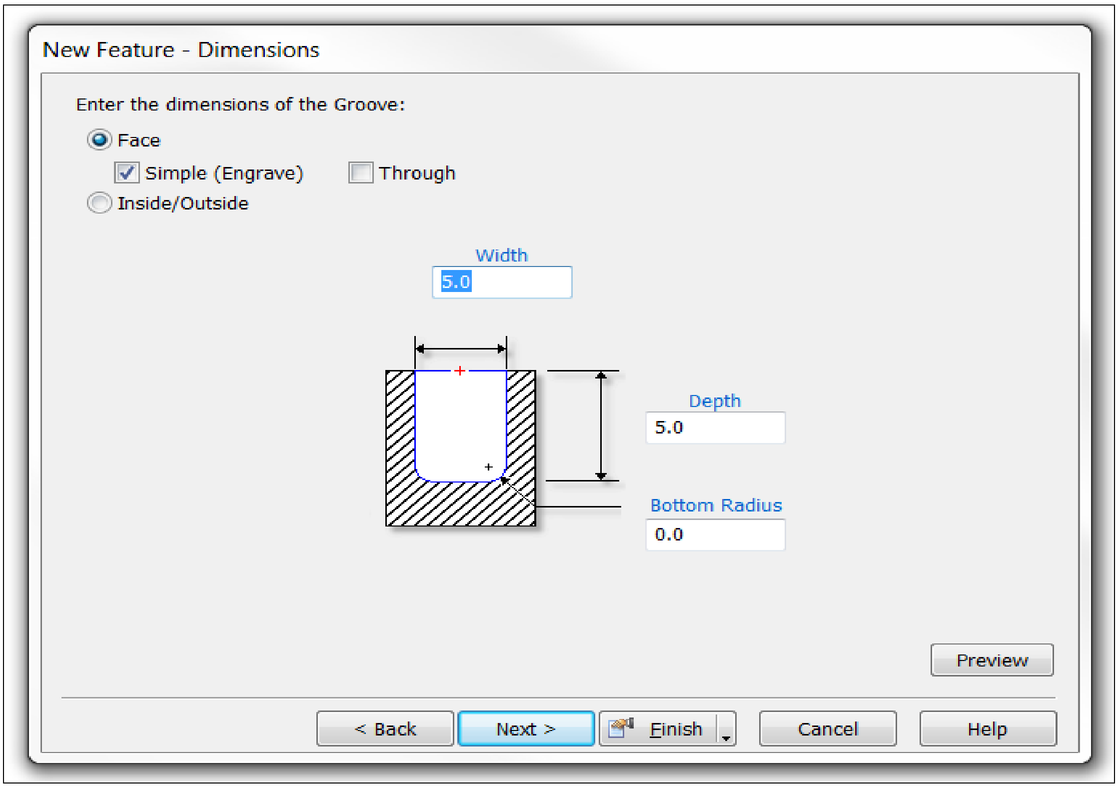
8 Select **Feature** from **Steps** or **CTRL+ R**, Select **Groove from Curve** and select the **Line**.

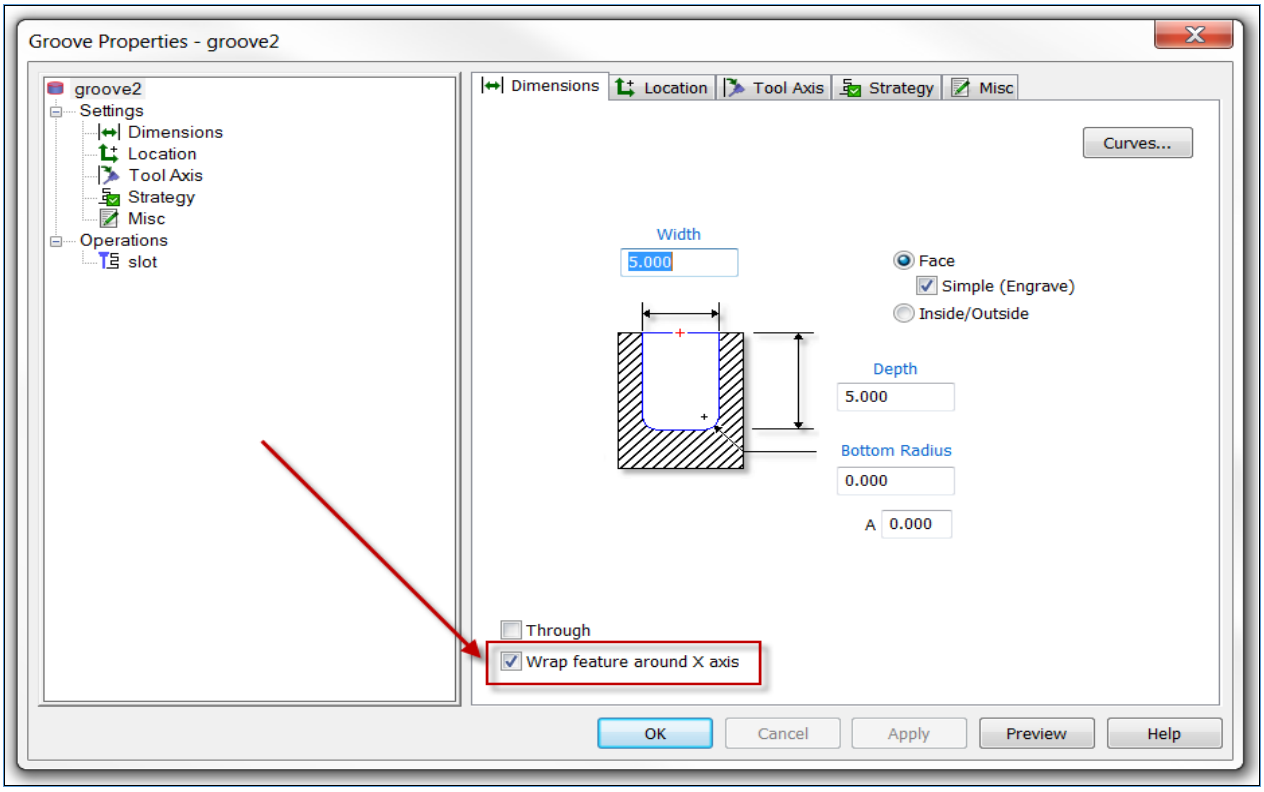


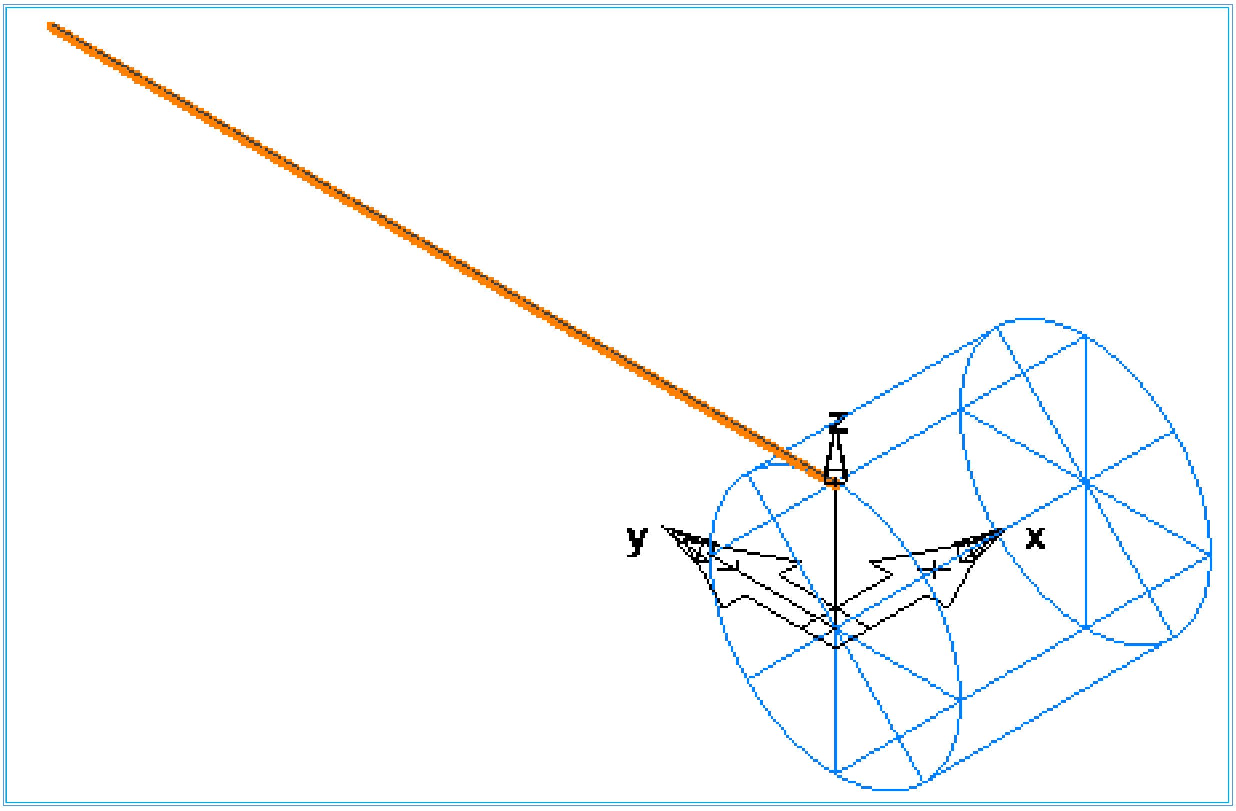


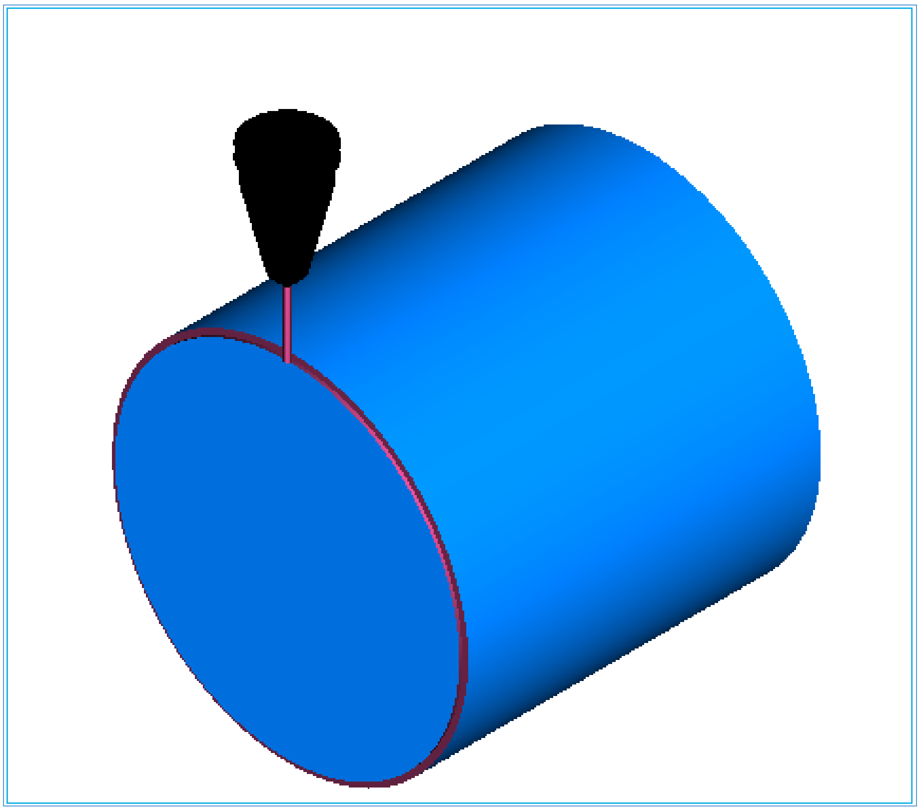
9 Select **Next** twice.

10 Double click the **Groove** feature in the **Part View** window. Check the **"Wrap feature around the X axis "**box at the bottom of the **Groove properties tab**. Click **OK** and then run a **3D simulation**. This will produce a **groove** that travels once around the circumference.

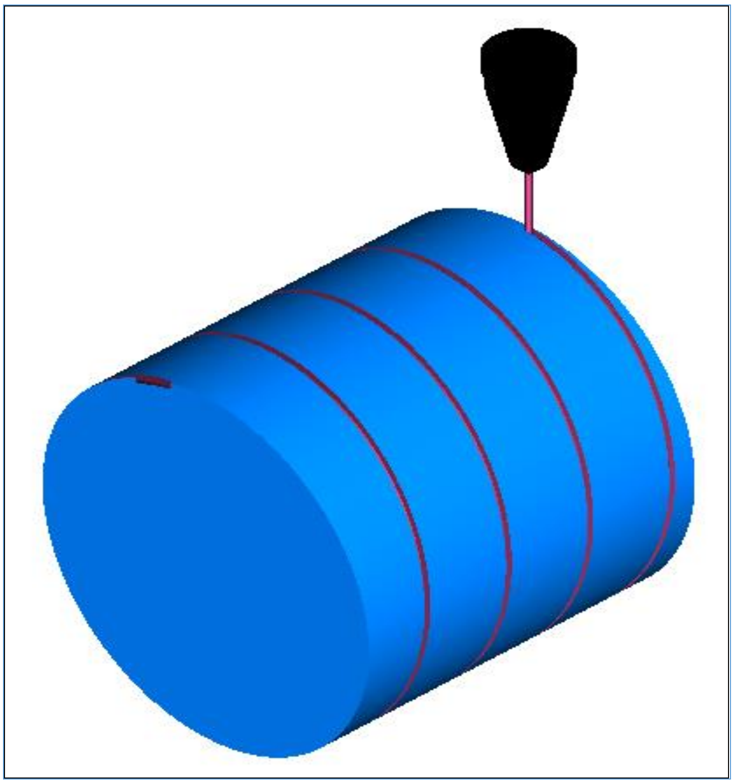





z
y
x



11 Double click onto the **2D** line and Change only the **end point XYZ2** in the **X to 300mm** = Length change the **Y to 4*Pi*300** (4 =Number of Rotations) and **Z150** is the depth of the slot, click **Modify**, **run a 3D simulation**.



Wrapping a Pocket (Optional Exercise)

The **Pocket** may be created by drawing or importing **Geometry** or from **Dimensions** as you would normally create this feature.

It can be created at the centre of the stock and given a value in the Location Z direction while creating the feature so as to place the top of the feature on the radius of the stock.

The feature when completed may be **Transformed/Translated** in the Z direction to locate the top at the radius of the stock.

The **UCS** can be **translated** in the Z direction to the radius of the stock and then create the feature, then the Location in Z remains at 0.

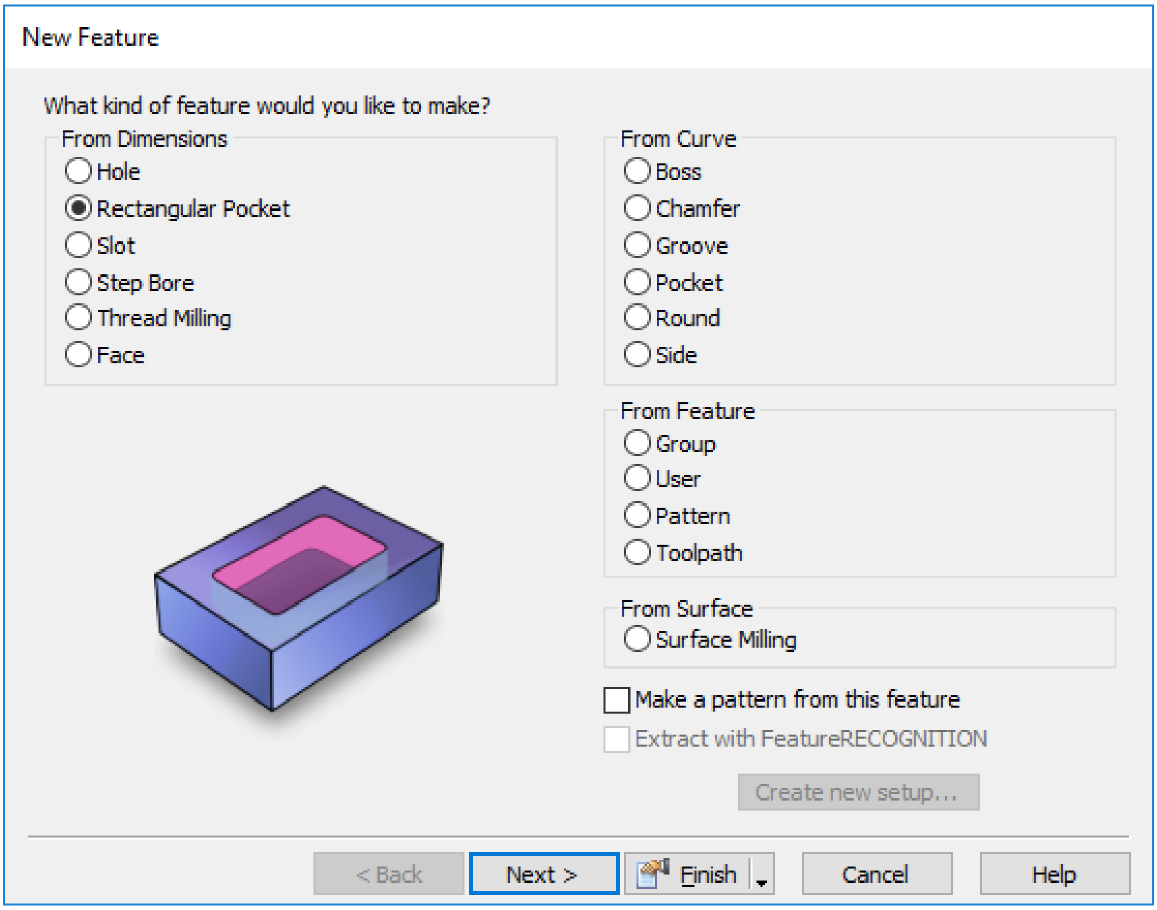
Remember if the 4th axis is not checked, on the **stock/indexing tab** or the feature is located down on the centre when trying to wrap the "Wrap feature around the indexing axis" on the dimension tab this will **not be visible.**

1 Open a **New Milling Setup**, **Millimetres, Stock** is Round, along the X axis.

2 Diameter 150mm, Length 300mm, ID 0, Next, Next, 4th axis indexing around the Stock X axis.

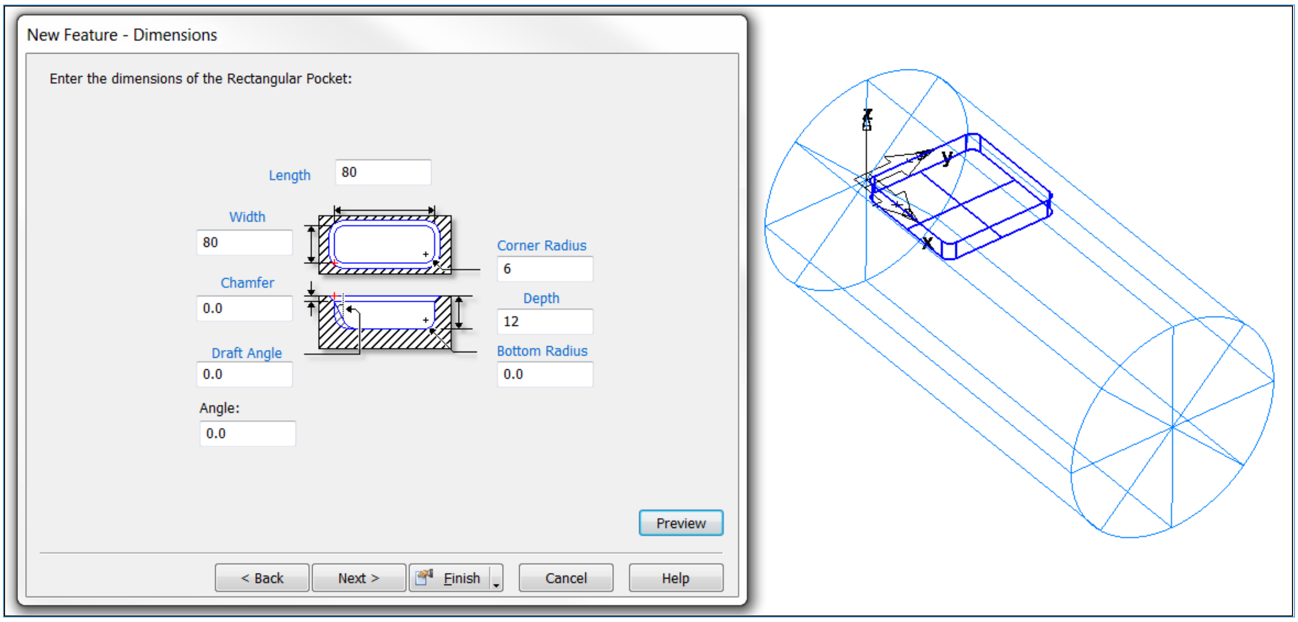
3 Rename **Setup** if you choose, observe that the fixture offset matches your machine requirement (**54**) and the **Part Name** if completed here we will place this information both in the code (Post Processor must be setup first) on the proper line to identify the program and also can be set when saving **NC code** so as to name the text file. Use current location for alignment, **Next** and **Finish**.

4 Create New Feature, Rectangular Pocket from Dimensions. Select Next.

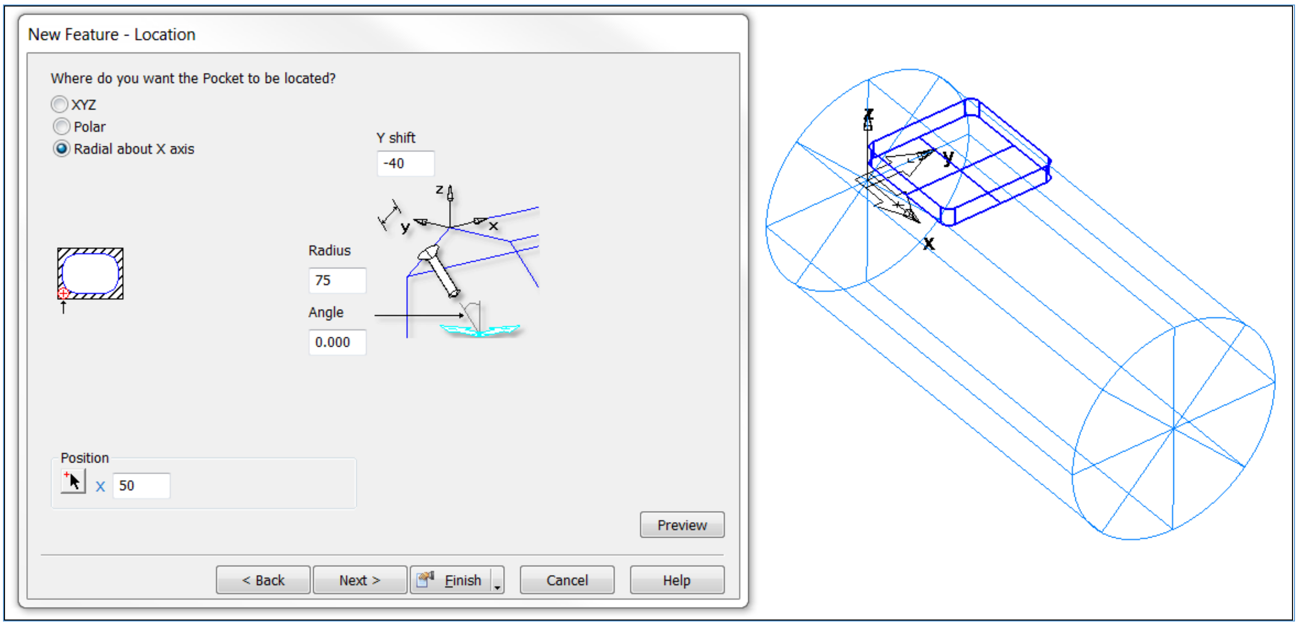


5 Enter Dimensions, Length **80mm,** Width **80mm,** Corner Radius **6mm** and Depth **12mm.**

6 Select **Next**.

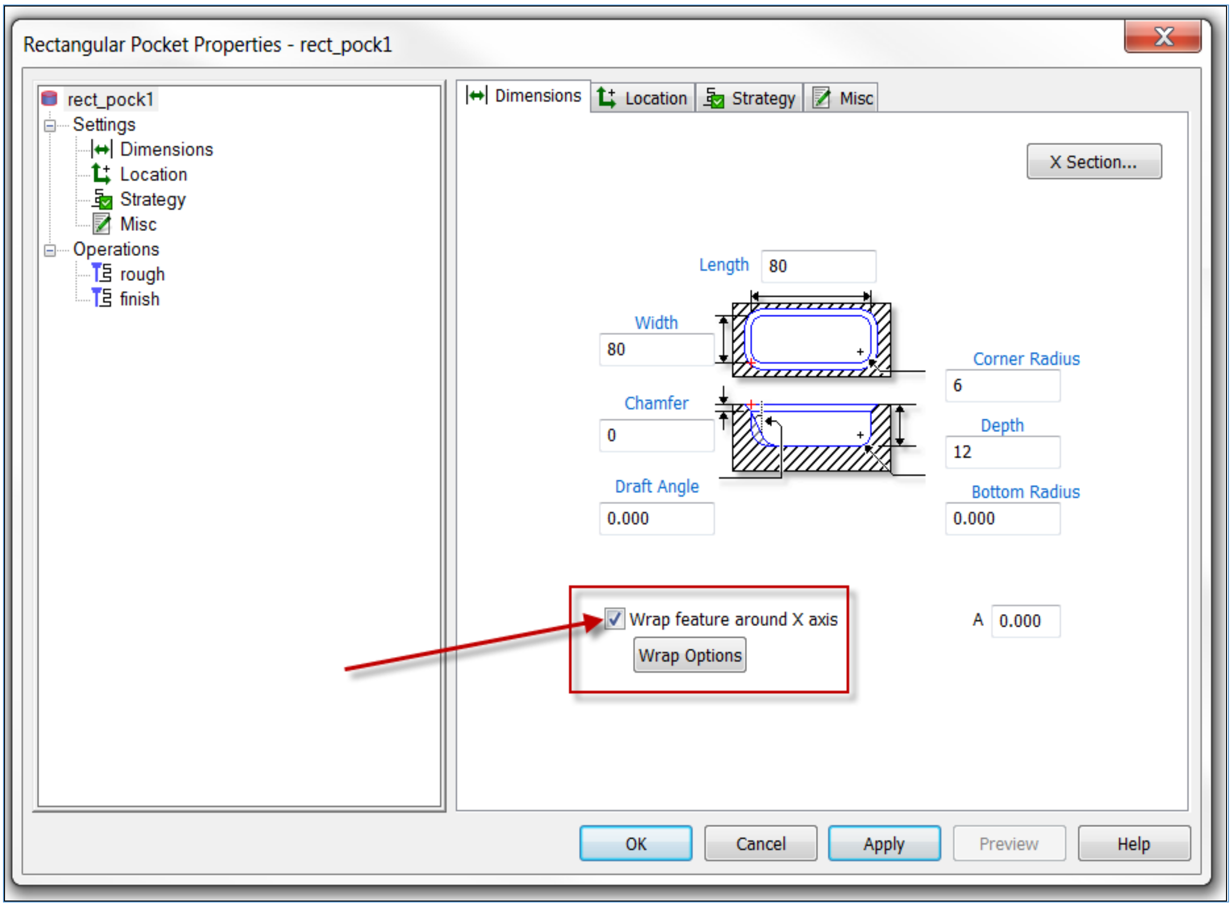


7 Select **Radial about X axis** and then enter the other information shown on the next page.

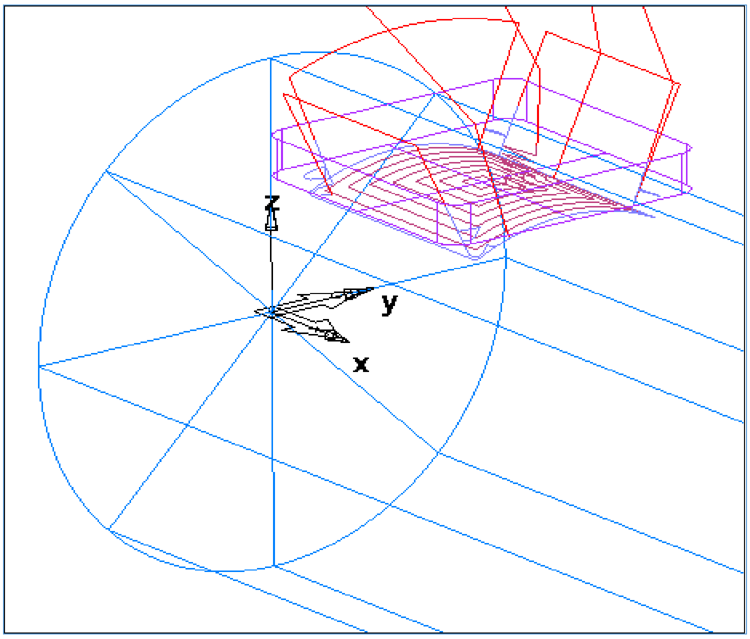


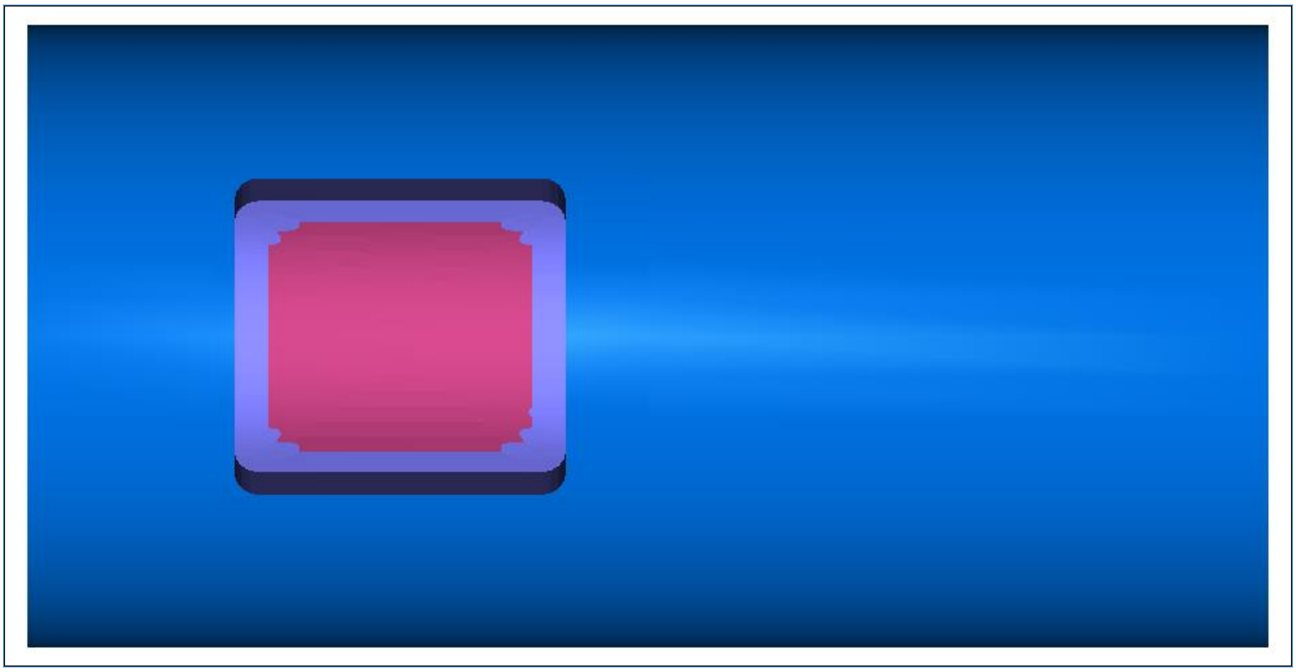
8 Enter **X50mm, Y-40mm, Z75mm,** this centres the pocket and places the top of the feature at the outside radius of the stock. Select **Finish.**

9 Select **Wrap feature around X axis**.



10 Run **3D Simulation**, See finished image on the next page. To make this more interesting, create a pattern from this feature. The details of which are shown after the first pocket Simulation.





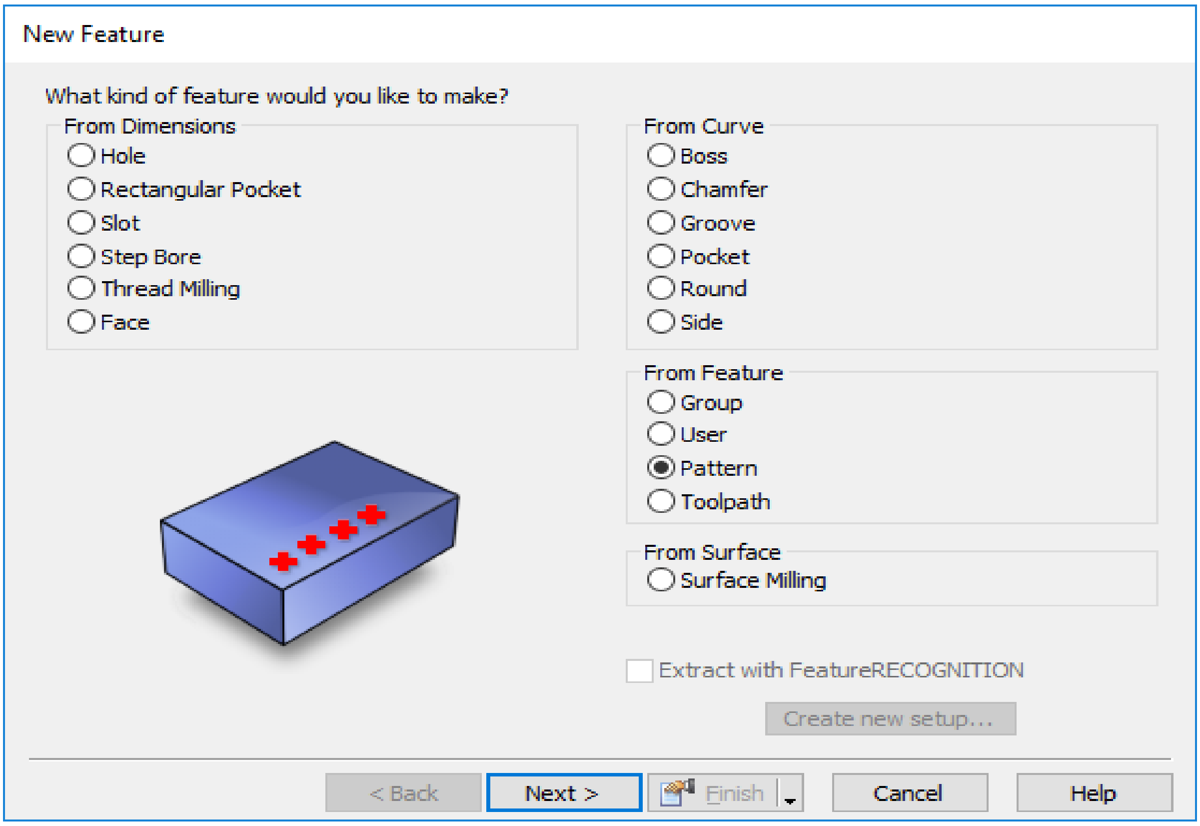
Pattern from the pocket Feature

11 First of all **eject** the **3D Simulation**.

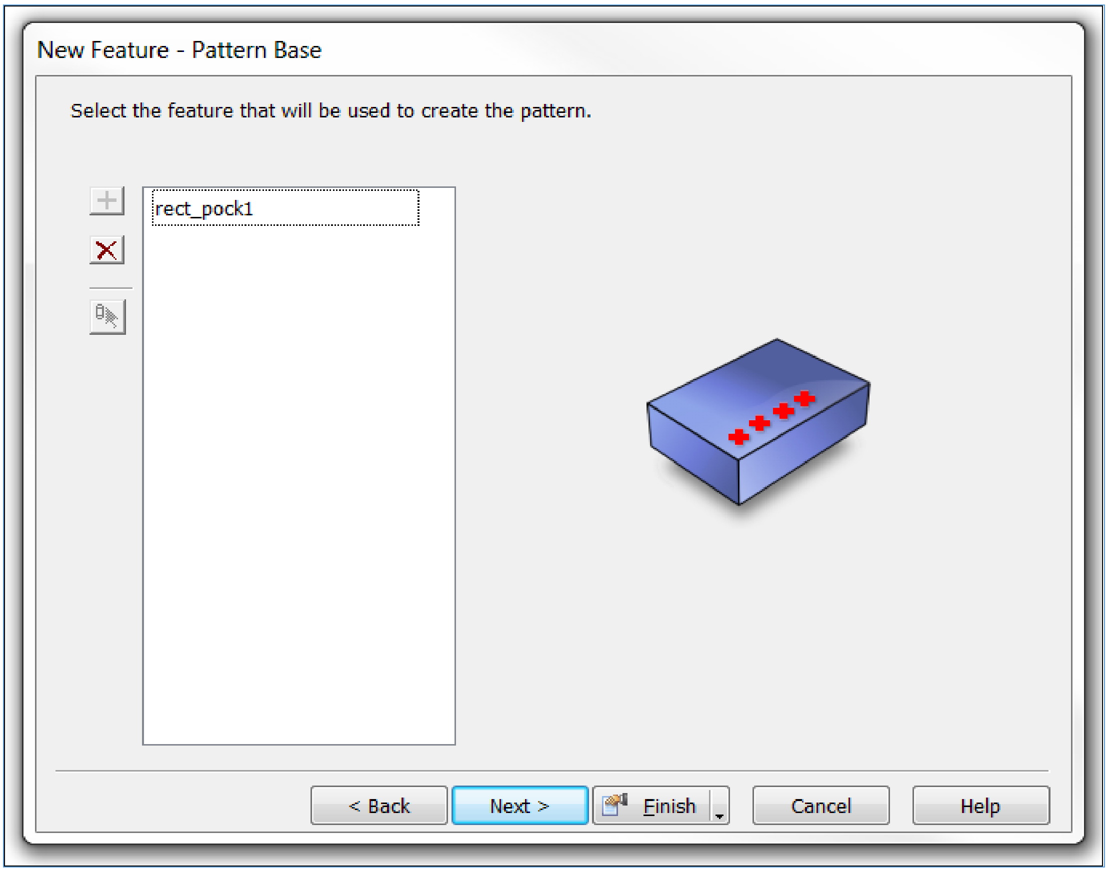
12 Then double click on **stock** and change the **ID** value to **120mm**. We are essentially creating a tube.

13 Select **Ctrl+R** or create a **New Feature** from **Steps** or the **Advance toolbar**.

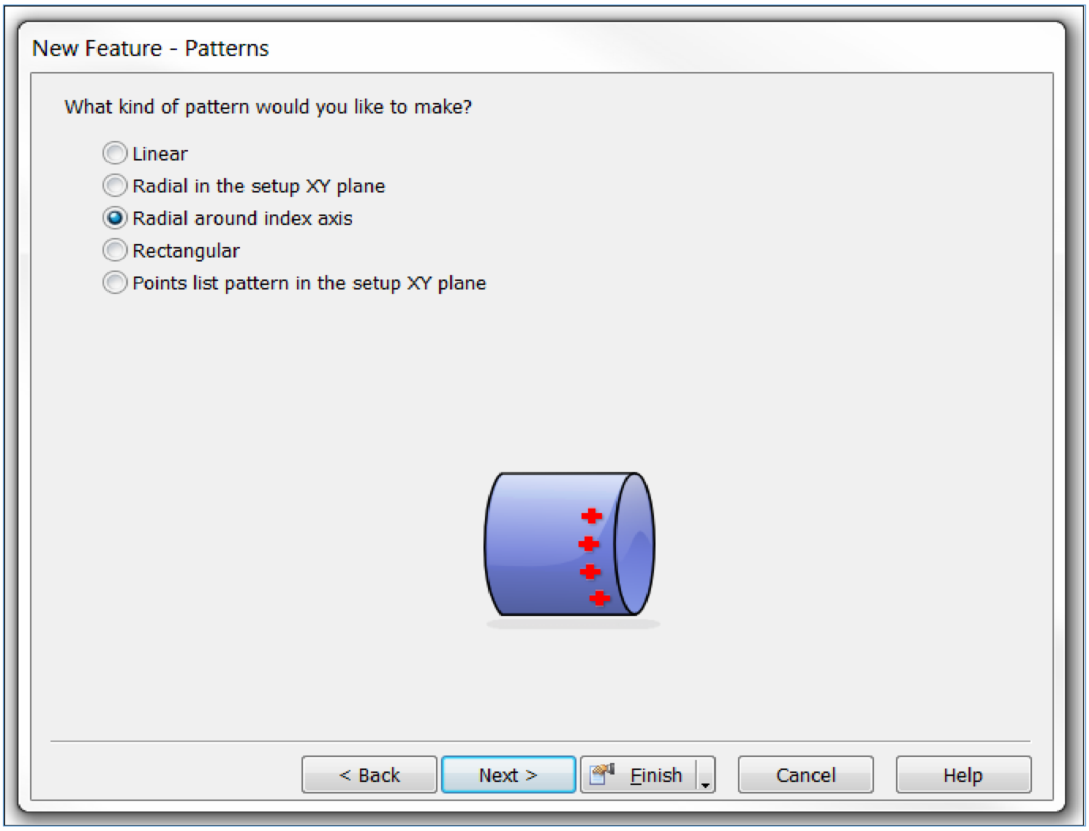
14 **Pattern** will be automatically selected. If not select **Pattern**.



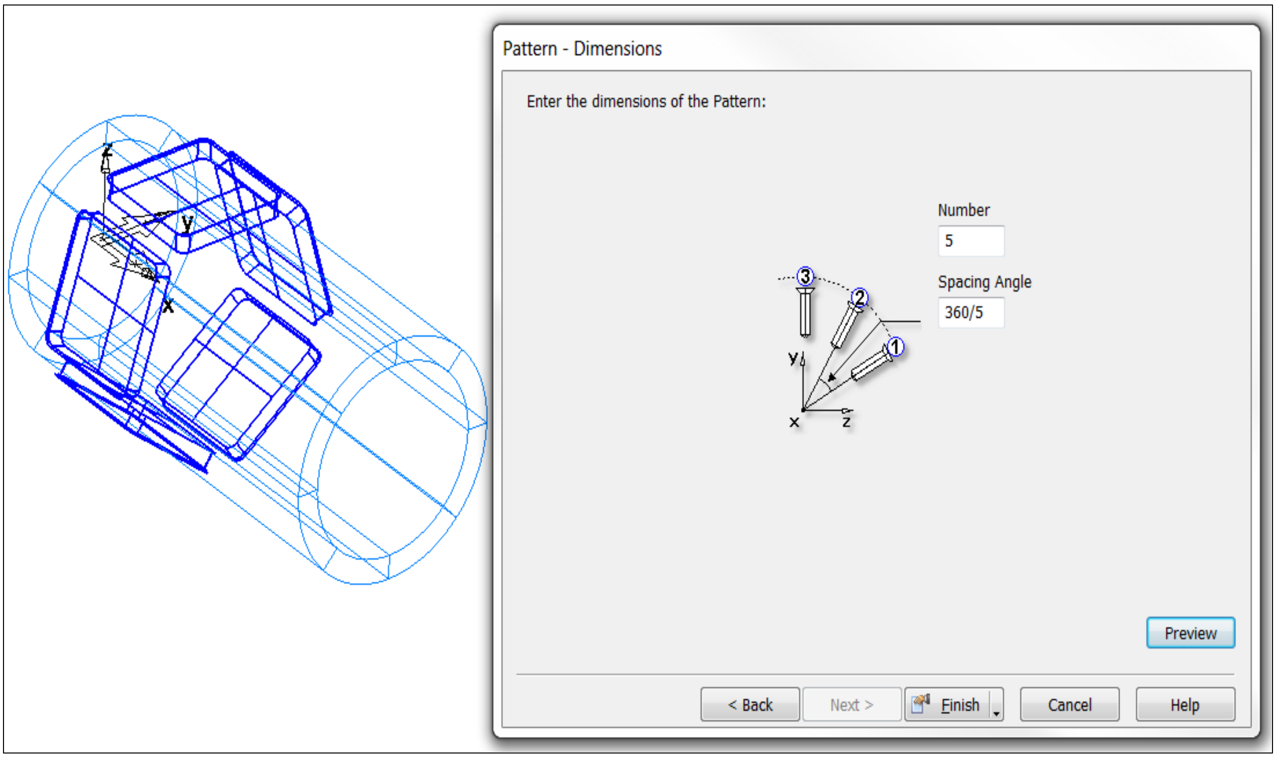
15 Select **Next**. You will be presented with **a New Feature – Pattern Base** menu. Select the **rect_pock1** feature from **PartView.**



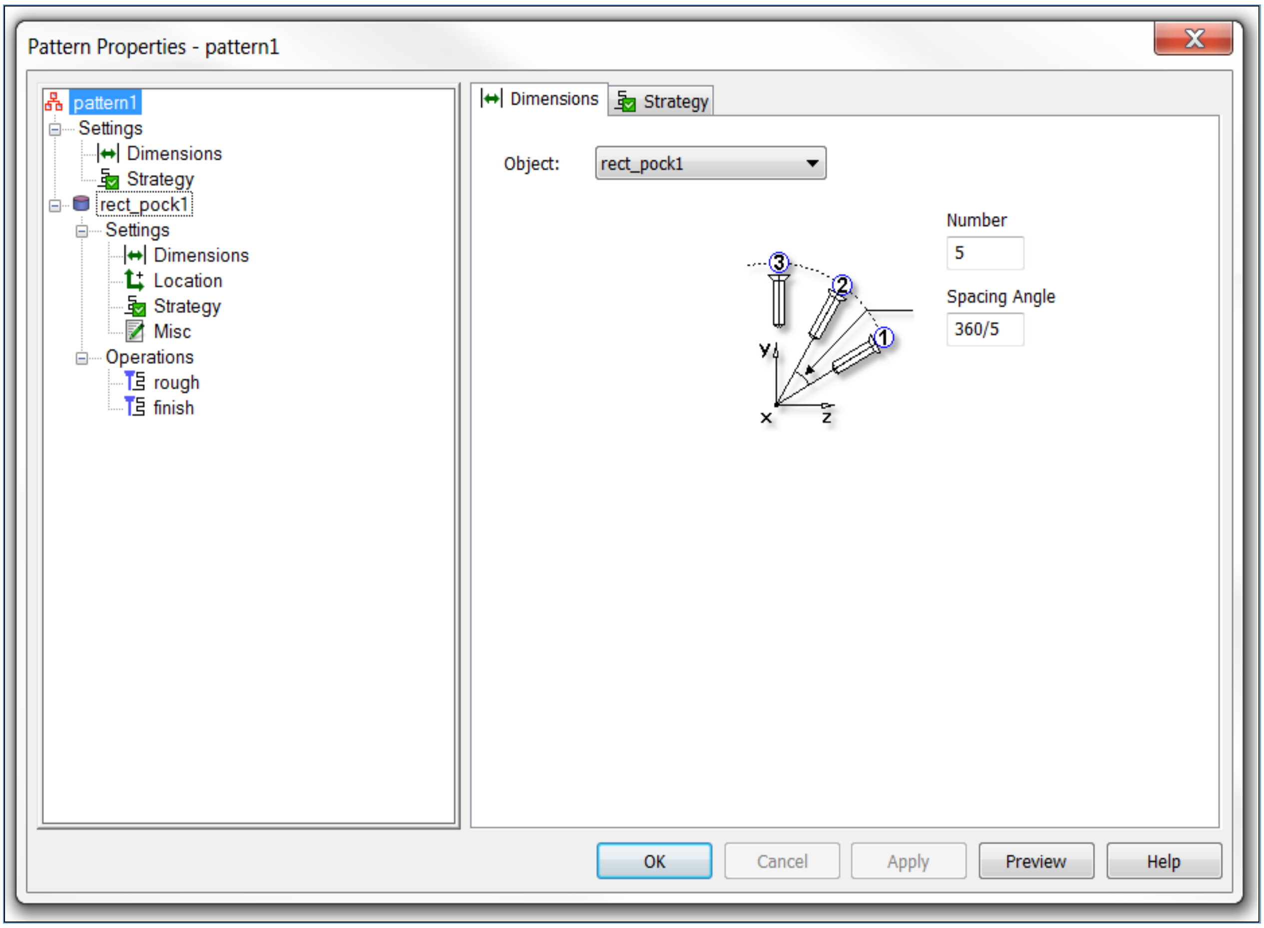
16 Select **Next**. Then select **Radial around index axis** as shown.

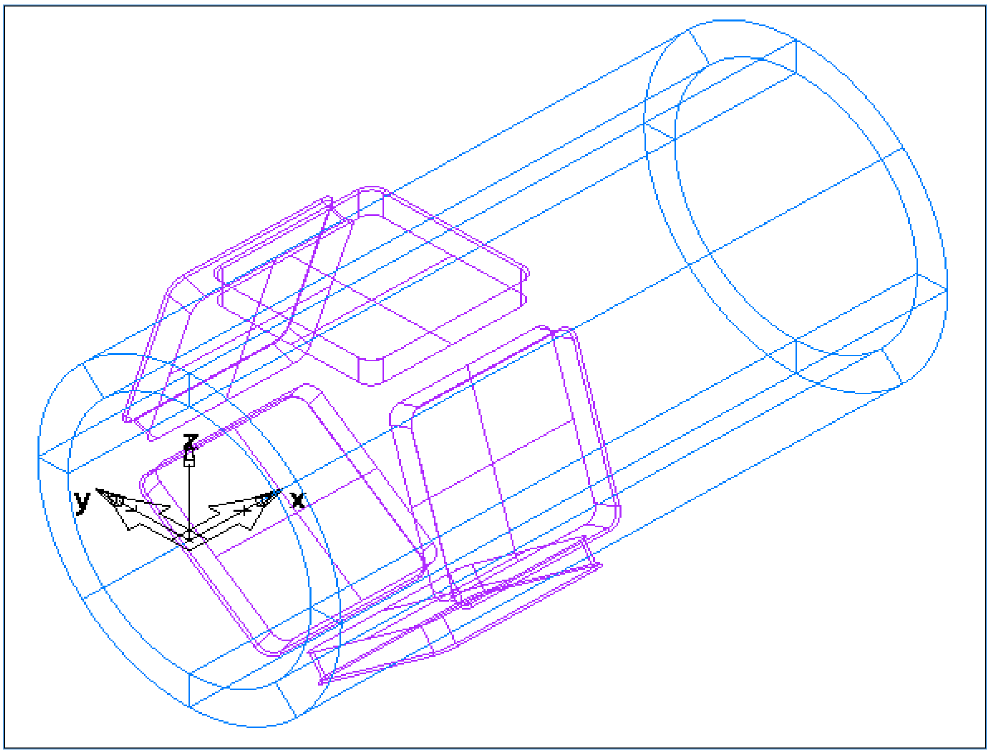


17 Select **Next**. Enter 5 for number then type in 360/5. **FeatureCAM** will calculate the angle.



18 Select **Finish**. Then select **OK** to close the form.

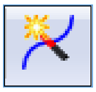

Pattern Properties - pattern1
pattern1
Settings
Dimensions
Strategy
rect_pock1
Settings
Dimensions
Location
Strategy
Misc
Operations
rough
finish
Dimensions
Strategy
Object:
rect_pock1
Number
5
Spacing Angle
360/5
OK
Cancel
Apply
Preview
Help

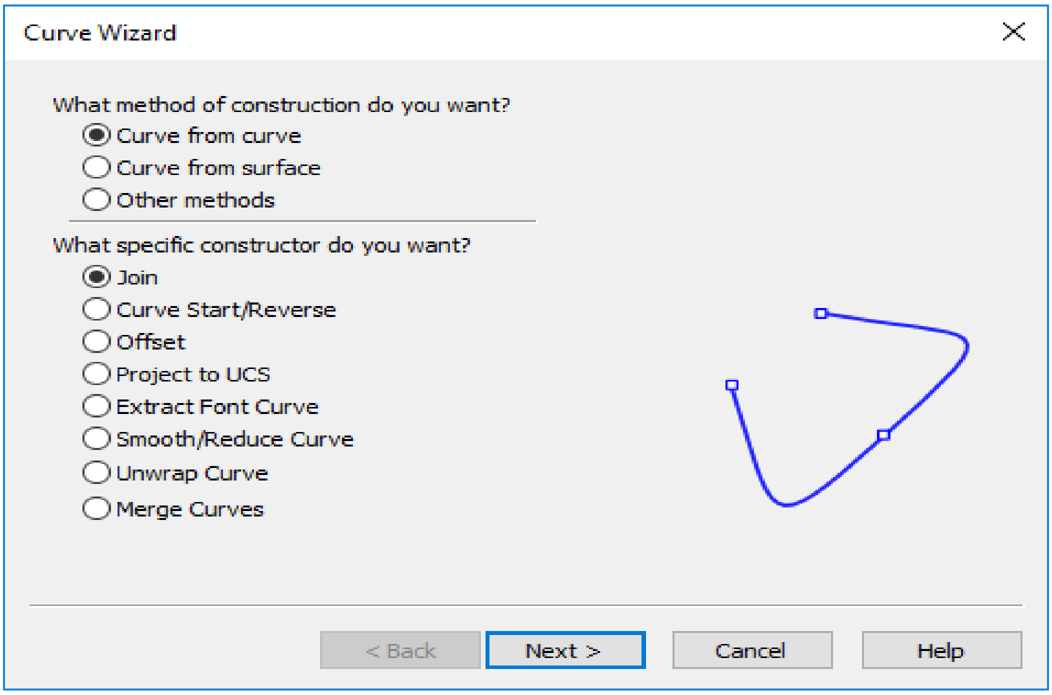


Wrapped Engraving

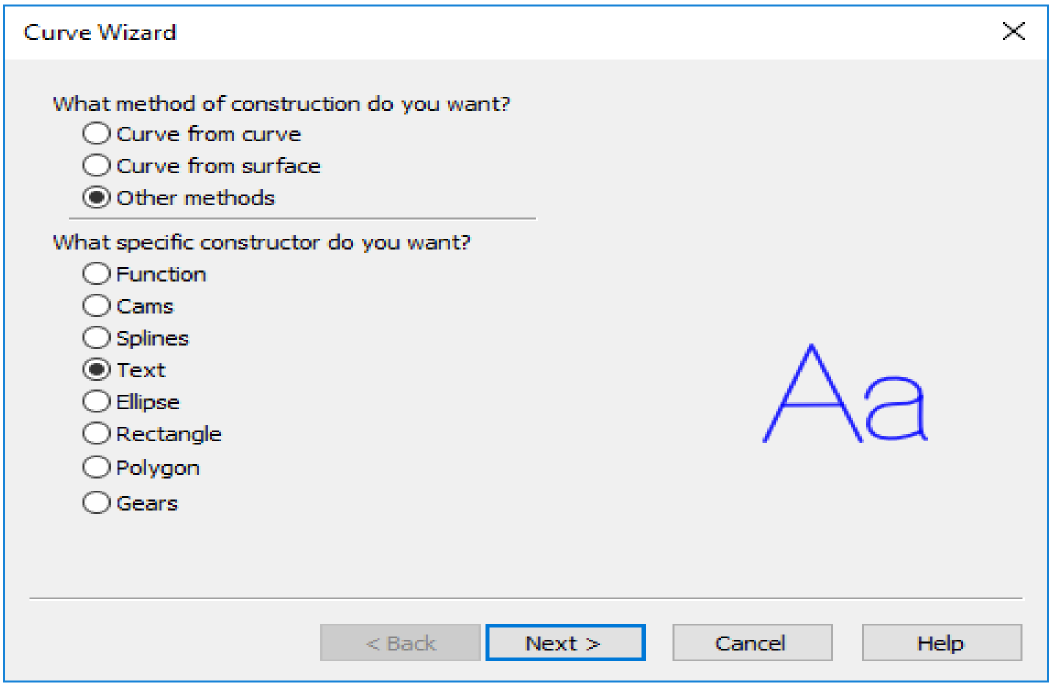
***Engraving** around a cylinder can be accomplished by first creating the text and then applying a **Grooving feature** to the text curve.*

1 Open a New Milling Document, Millimetres, Round Stock, X axis, 4th axis indexing, Diameter 300mm, Length 300mm, ID 0, setup centred.

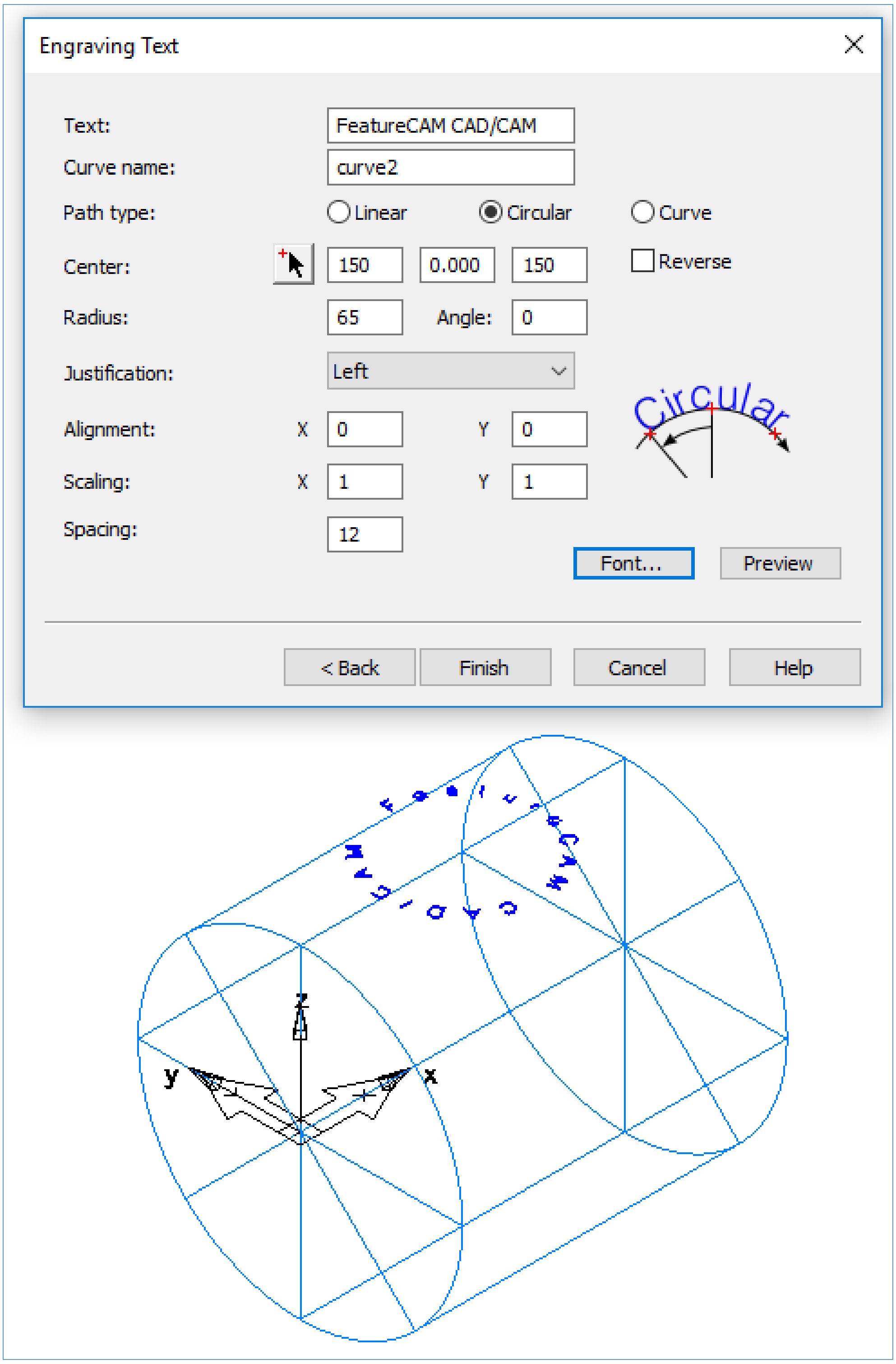
2 Change to **Top view** or **Ctrl** + **5**, Select **Curves**; click Create curves using the Curve Wizard. Select the **Curve Wizard Icon**. 



3 Select **Other methods** and **Text**. Select **Next**.

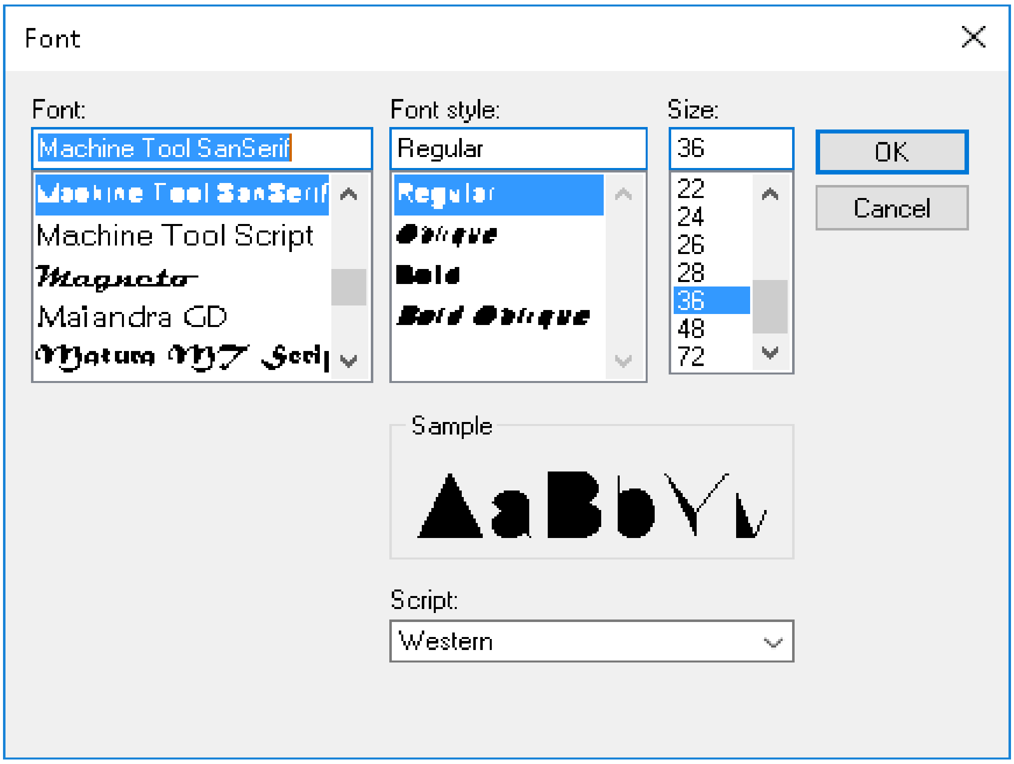


4 Type in the following information. Shown on the next page. The curve name may be different to your name.

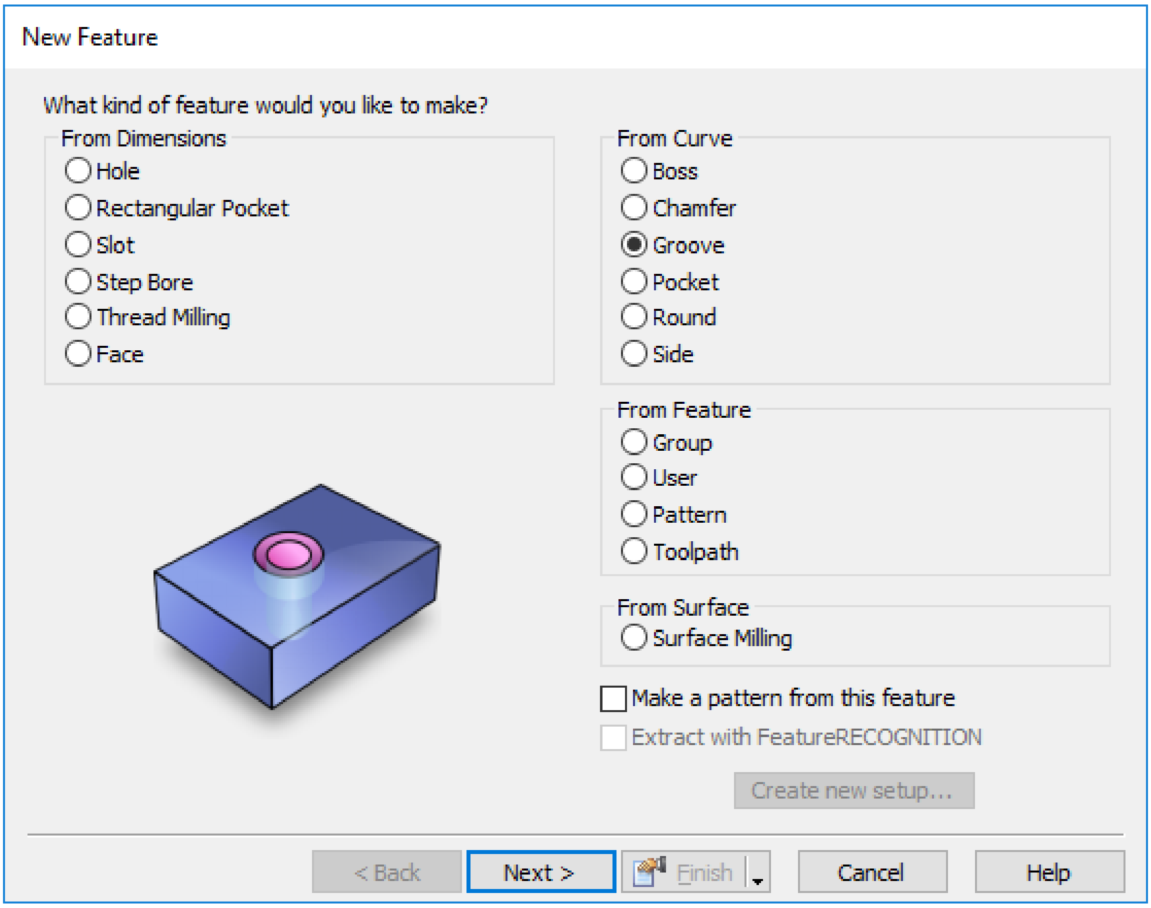


5 **Centre X150mm Y0 Z150mm Radius 65mm**.

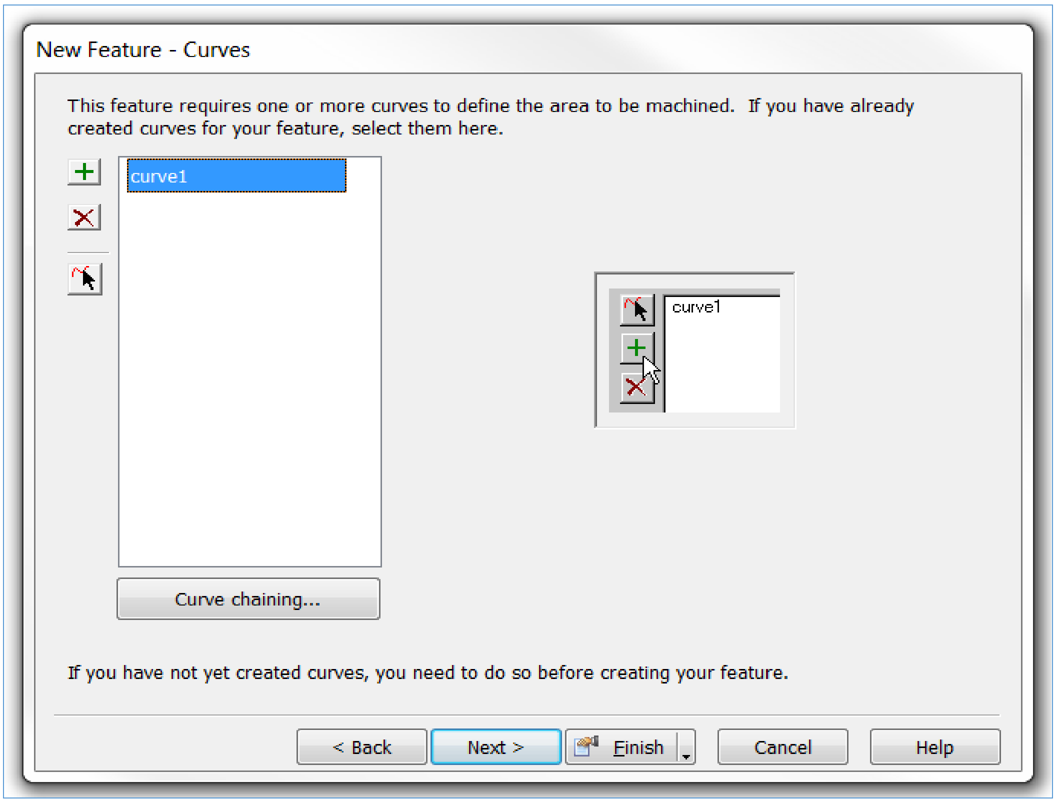
6 The **Font** we are going to use **is Machine Tool SanSerif. Size 36.**



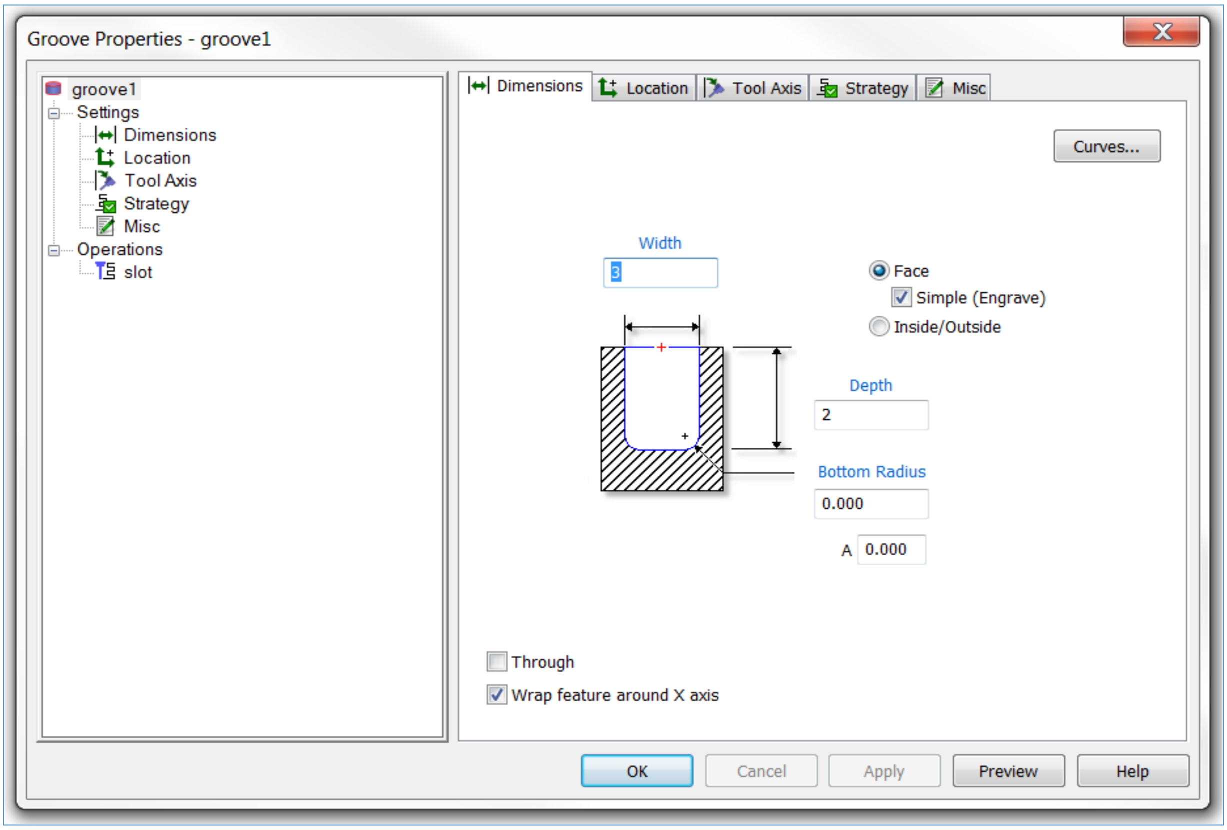
7 Select **New Feature** or **Ctrl+ R** New Feature. Select **Groove** from **Curve** and then select next and select the **Curve**.



8 Pick **Curve**



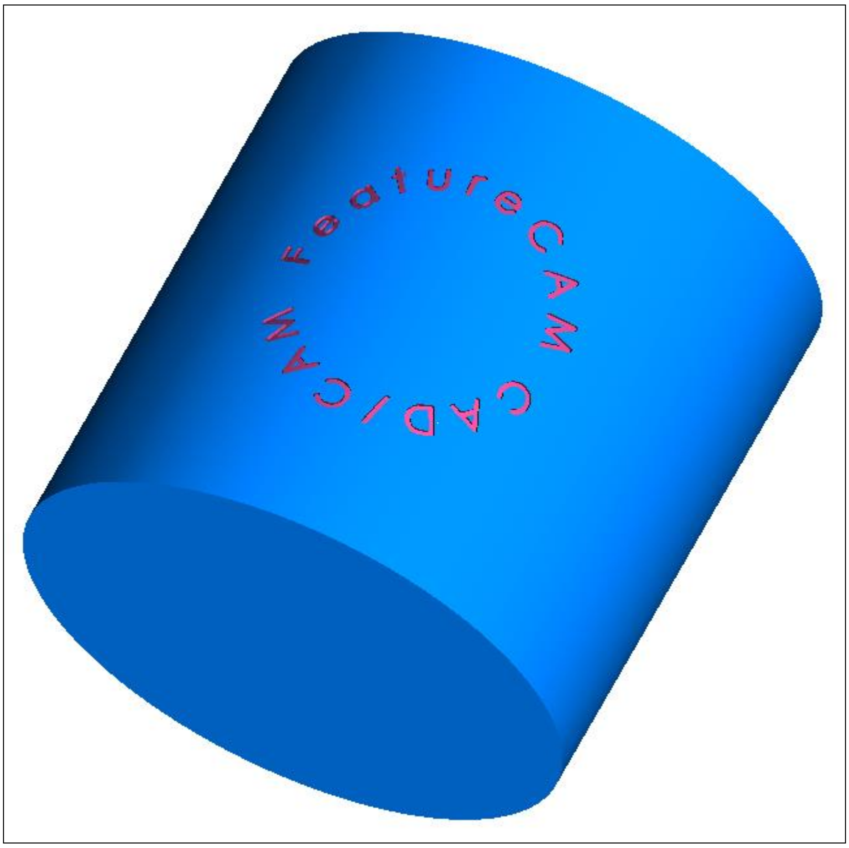
9 In this case my curve is **curve1**. Yours may be a different number. Select **Finish**.



10 Select Wrap feature around X axis.

11 change the depth to 2mm and width to 3mm

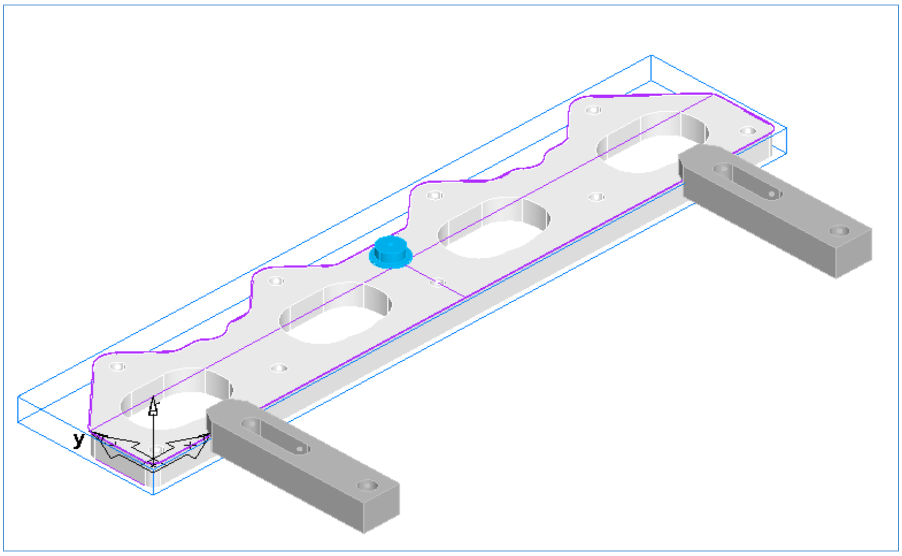
12 Select the 3D Simulation and press Play.



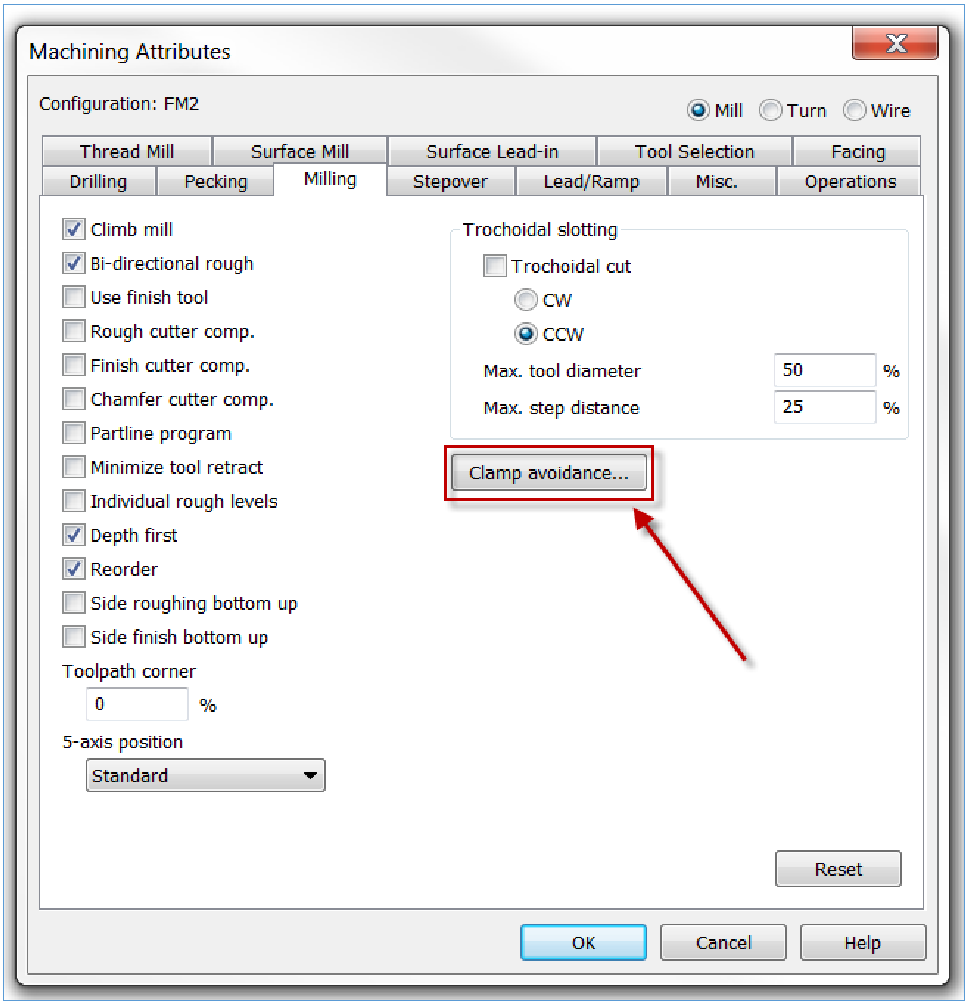
Automatic Clamp Avoidance

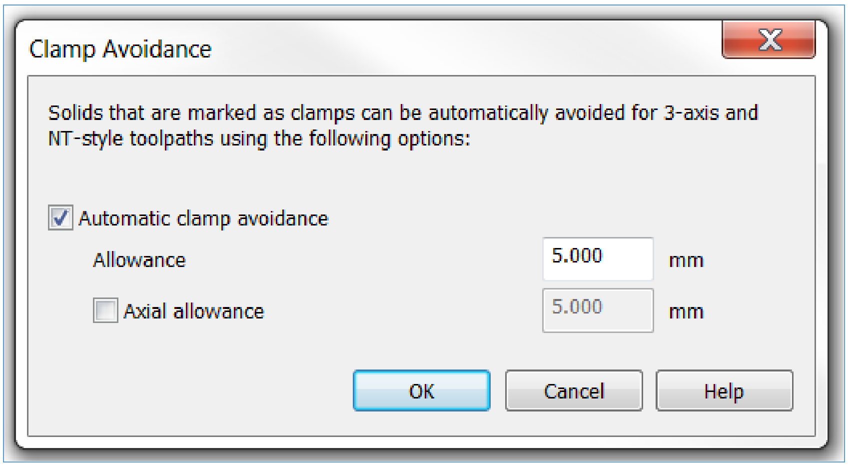
In FeatureCAM you can now automatically avoid solid model clamps when using 2.5D NT-style toolpaths and 3D toolpaths.

1 Open file **Manifold plate with clamps.fm**



2 Go to Manufacturing>Machining configuration and select the file Manifold plate with clamps.fm and select Edit. Select Clamp avoidance under Milling. Change the value to 5mm.

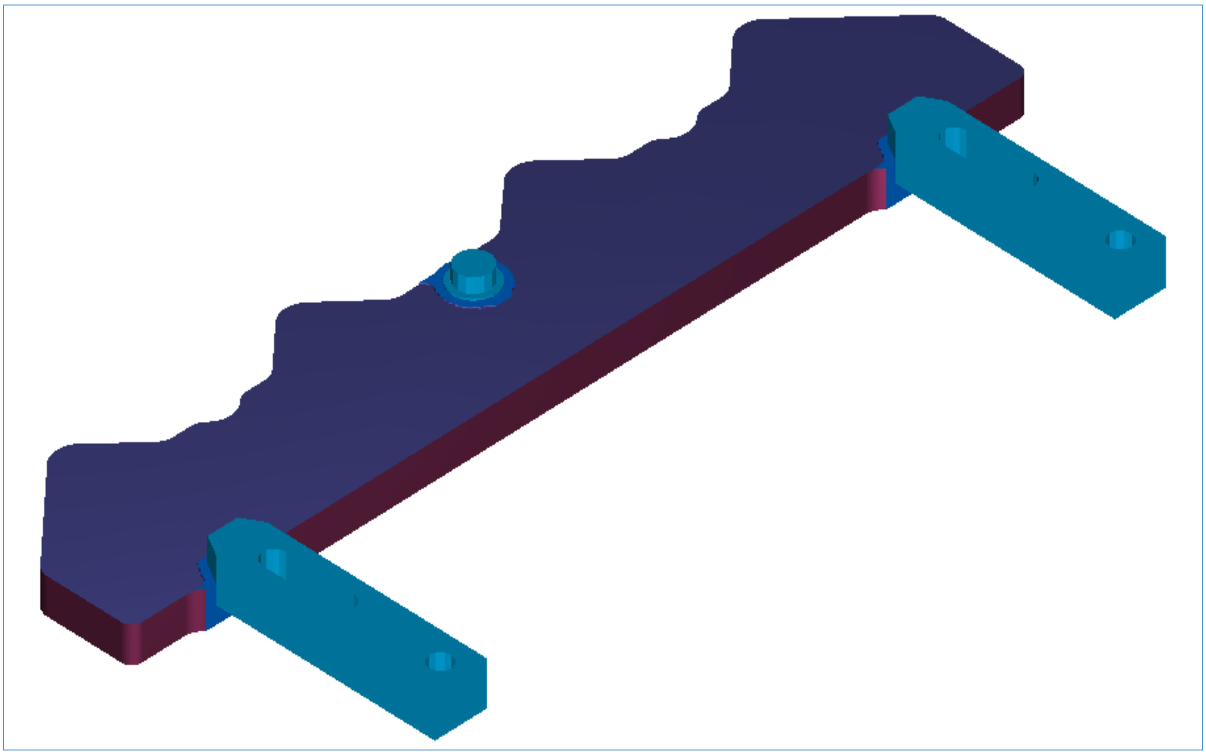




*By setting this Value to **5mm** all of the **NT toolpaths** will be re-calculated and will stand off any clamps by this amount.*

*To turn a solid model into a clamp right click on the solid model in **PartView** and select **Use Solid as a clamp**. FeatureCAM will then know to avoid this solid model. As shown on the next page.*

The top surface machining is using an NT pocketing strategy.



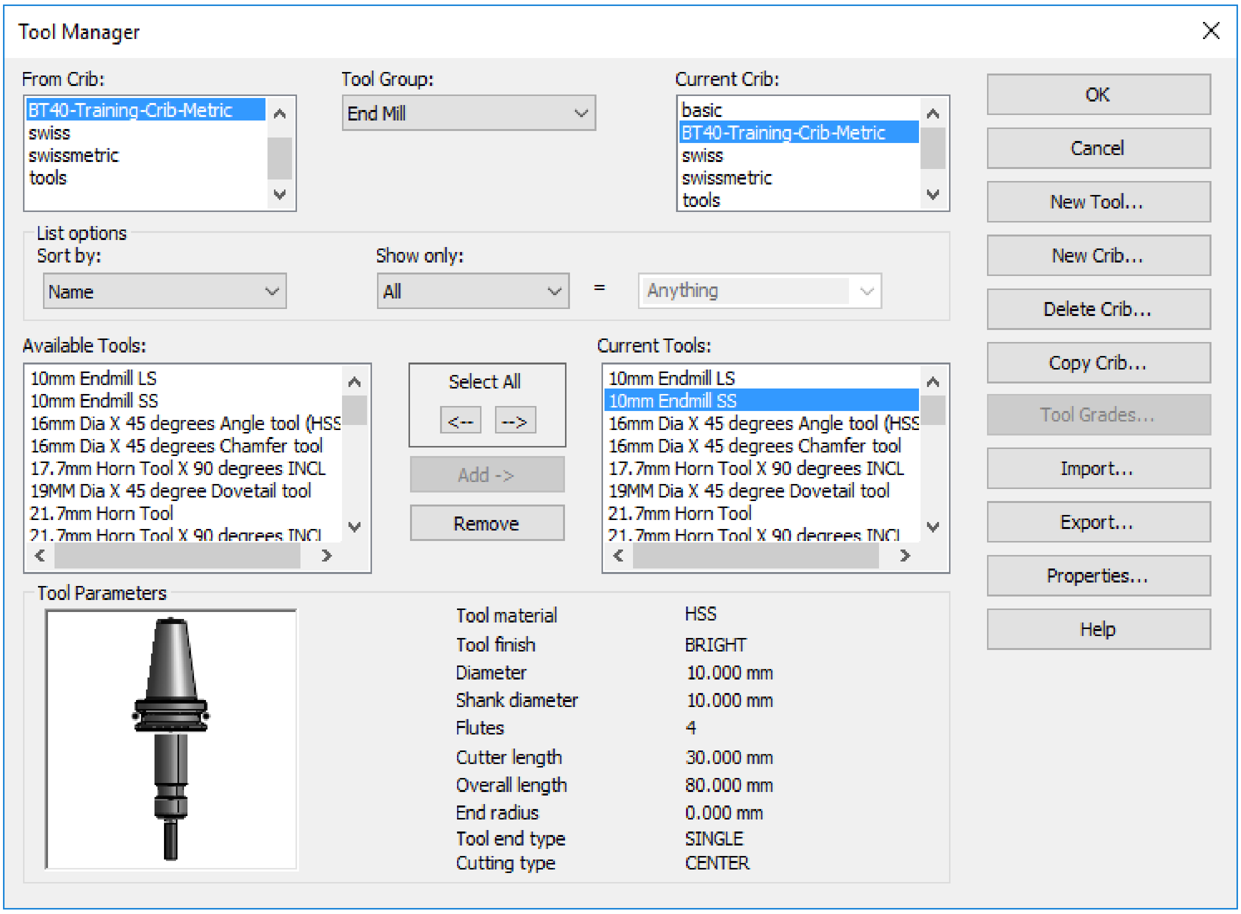
Contents

Tooling Reference Guide

*It is possible to create Tool libraries in **FeatureCAM**. These are called Tool **Cribs.** The main reason for creating Tool Cribs is because in any machine shop there is normally a finite amount of tooling available and by creating a custom Tool Crib, **FeatureCAM** will know what tooling is available to choose from. This can be machine specific or a generic crib for all machines.*

Tool Manager

*The **Tool Manager** is where information on your **Crib** is stored and allows you to edit and create new Cribs as well as creating new tools. The Tool Manager can be accessed by going to the **Manufacturing Menu** and down to **Tool Manager**. This will open the following form.*

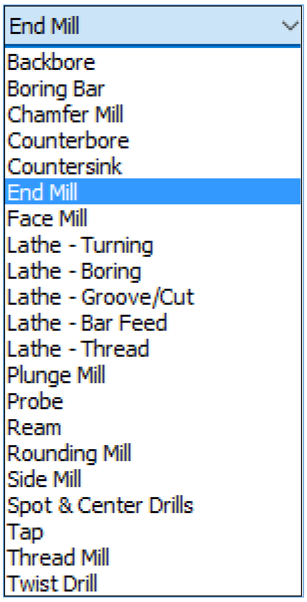


*There are three tool Cribs loaded as standard. **(Basic, Basicmetric and Tools)** and these can be used as a starting point when the user first starts to look at setting up their own Cribs.*

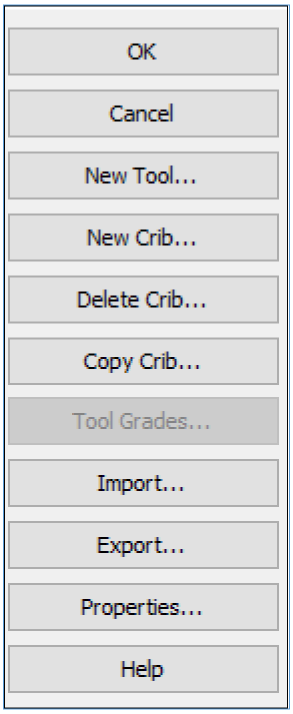
The window is divided into separate areas and at first glance can look a little intimidating, but by applying some simple rules it becomes much easier to work with.

If the user divides the form in two down the middle of the form. The left side is used to select which Crib to copy from whereas the right hand side of the Form relates to the Crib that is current or the working Crib

Tool Group*. The tools can be grouped together depending upon their type or use.*



Also listed down the right hand side of the form are a number of options. Among these options are ***New Tool****,* ***New Crib****,* ***Delete Crib*** *and* ***Copy Crib****.*



These options do exactly what they say.

***New Tool** is used to create a new tool type.*

***New Crib** will create a new crib, ready for the user to insert their selection of tools.*

***Delete Crib** will delete an existing crib*

***Copy Crib** Copy Crib will produce a copy of an existing Crib.*

***Tool Grades** is used to apply scaling to feeds/speeds to similar tool types that use different speeds/feeds*

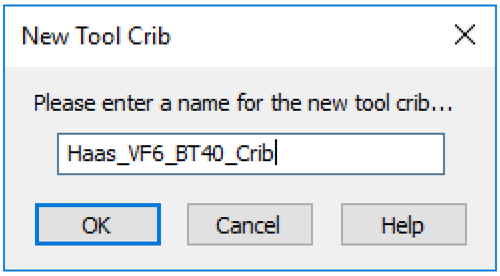
***Import / Export** allow the user, to either import or export tool cribs from one computer to another computer containing a copy of **FeatureCAM**.*

Tool Manager Exercise

1 Start a **New FeatureCAM** Part File ignoring the stock set-up.

2 Open the Tool Manager. Go to **Manufacturing>Tool Manager**.

3 By selecting the New Tool Crib option, the following form will appear. Enter the name as **Haas_VF6_BT40_Crib**, and then click **OK**.

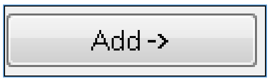


The New Tool Crib will now be listed in the Tool Manager on the left hand side The new Crib is empty so it is possible to copy tools from existing Cribs.

4 Select the **Basicmetric Crib** from the **From Crib** Menu and the **Haas_VF6_BT40_Crib** Crib from the Current Tools: Crib Menu.

5 In the Tool Group **select End Mill** and select **EndmillM2000 long**. This is a long series **20mm dia Endmill**.

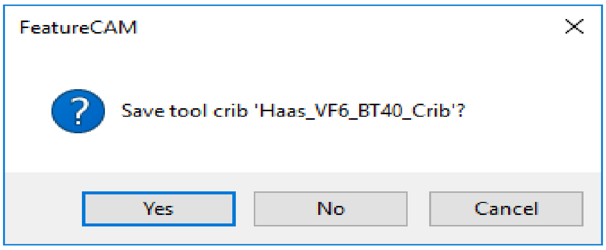
6 Click on the **Select All Button** and then on the **Add button**



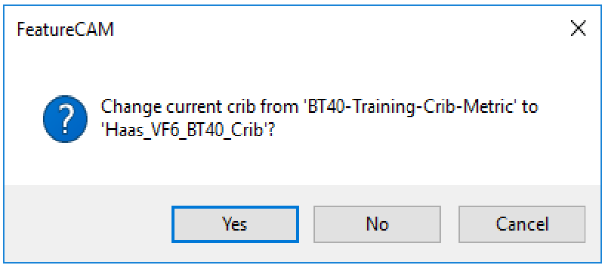
to transfer the selected tool over to **Haas_VF6_BT40_Crib**.

7 Repeat this procedure for the different tool groups and add some **6mm, 8mm, 10mm & 12mm** Twist Drills and some Spotting and Center Drills.

8 Click on the **OK button** and the following Menu will appear.



9 Select **Yes** and you will be presented with another menu.

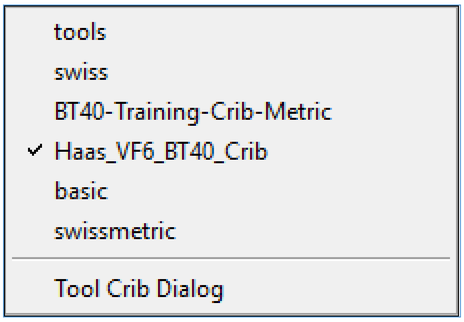


10 Select **Yes** to change from the **tools** Crib to the **Haas_VF6_BT40_Crib** Crib.

More tools can be added at a later date just by selecting them from the Tools Crib or defining New Tools based on Catalogue data and adding the tools as needed.

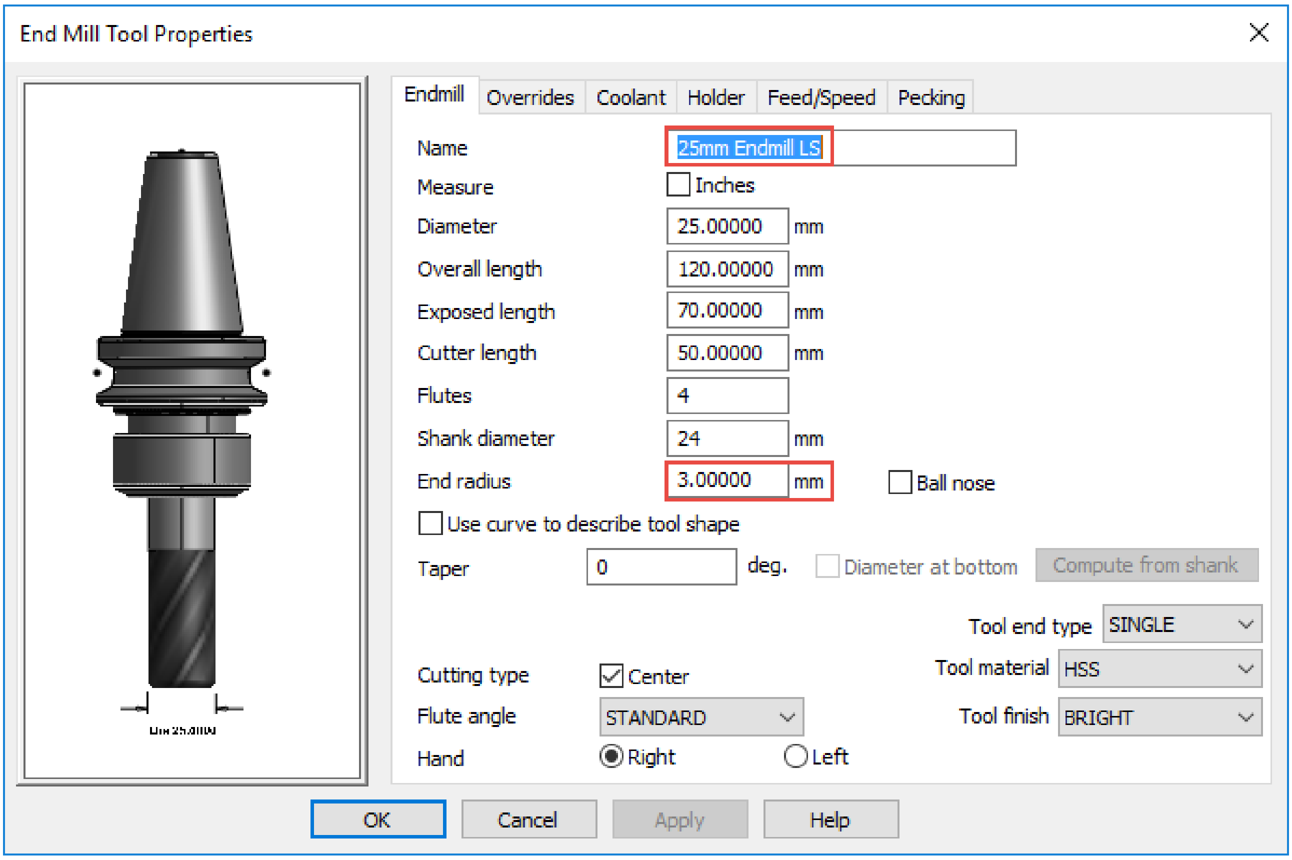
To set the Tool Crib so that it is used when calculating the tool path, go to Manufacturing, Set tool crib and select the crib from the list. Alternatively go to the bottom Right of the screen and click on the area indicated.

11 Select the **Crib required as shown**.



Editing Tools

*It is also possible to **Edit Tools** from within the Tool Manager. **Double click** on any **End Mill** to open the **End Mill Tool Properties** and change the name and End Radius to match the image below to create a new **Bullnose Tool**. This is the easiest way to create your custom tools.*

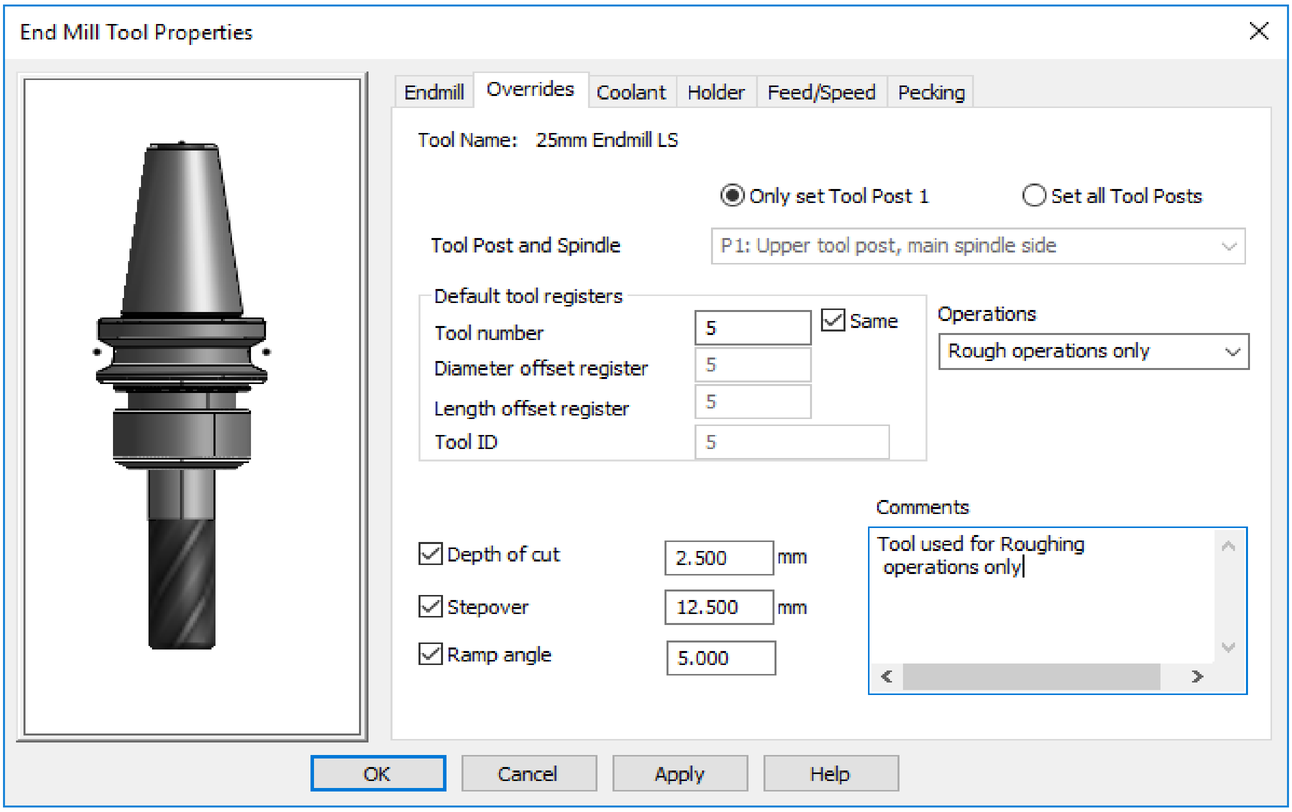


Setting Tool Overrides.

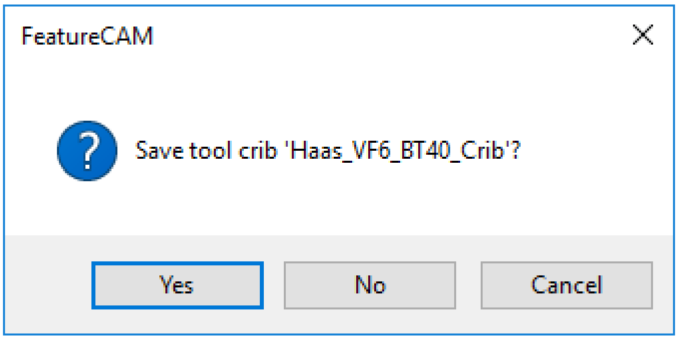
*From **Tool Properties** it is also possible to set what each tool will be used for as well as setting specific **Depths** of **Cut**, **Stepovers**, **Tool ID's** and other information. This is all done from the **Overrides** tab.*

12 With the Tool Manager still open select the **Overrides** tab and change the Operations drop down on the left hand side to be **Rough Operations Only**. When **FeatureCAM** now selects tools for toolpaths the Bullnose will only be used for roughing passes.

13 Also check the **depth of cut** and **stepover** checkboxes and enter values to match the image below. You can also set the **Tool ID** from here to match which tool position your tools are set to in your machine by editing the **Default Tool Registers**. Do this as well.



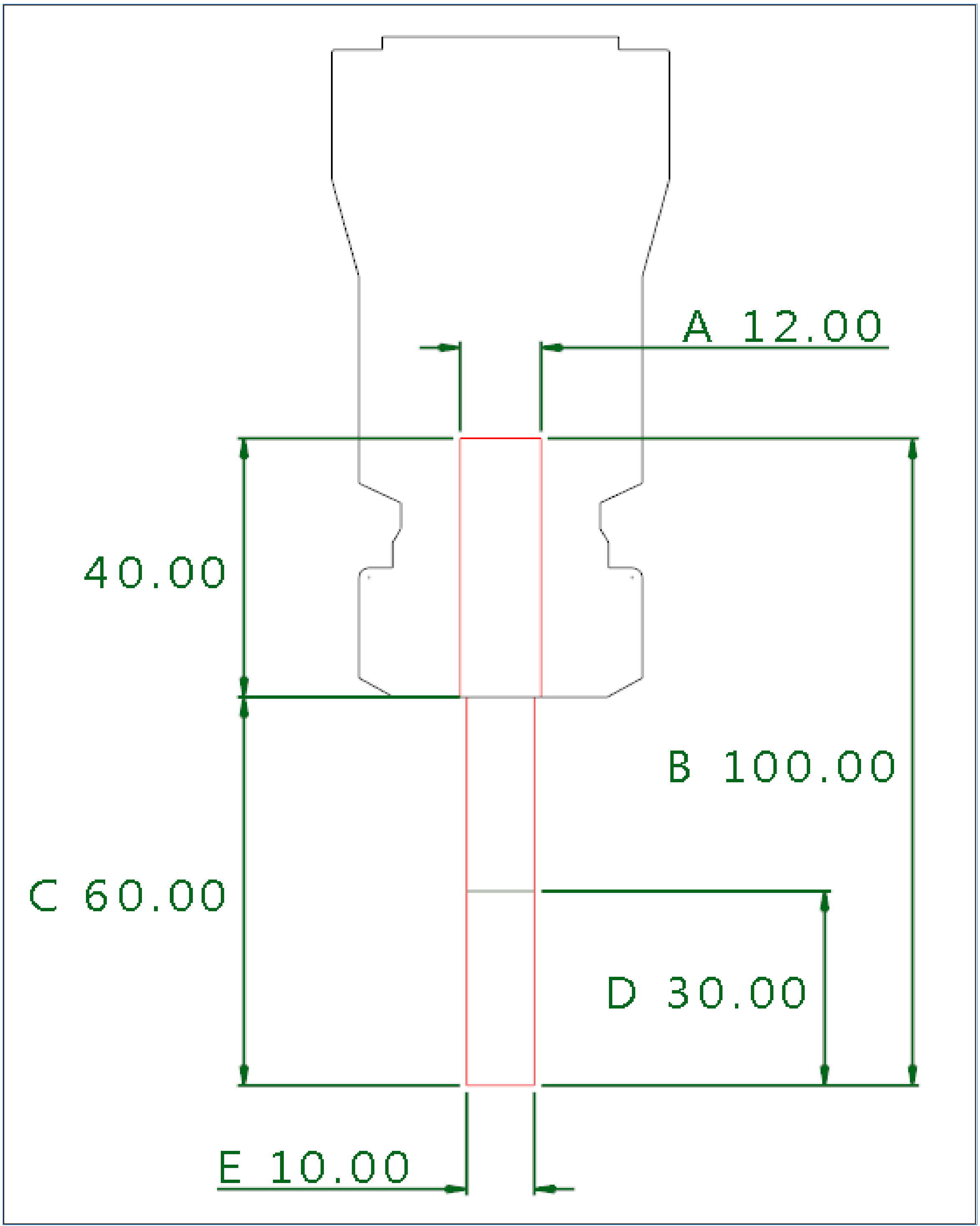
14 Click on the **OK button** and the following Menu will appear.



15 Select **Yes** to save the current Tool Crib after editing the tool.

End Mill Tool Properties

- End mill length and diameter properties.



A = Shank Diameter **12mm**

B = Overall Length **100mm**

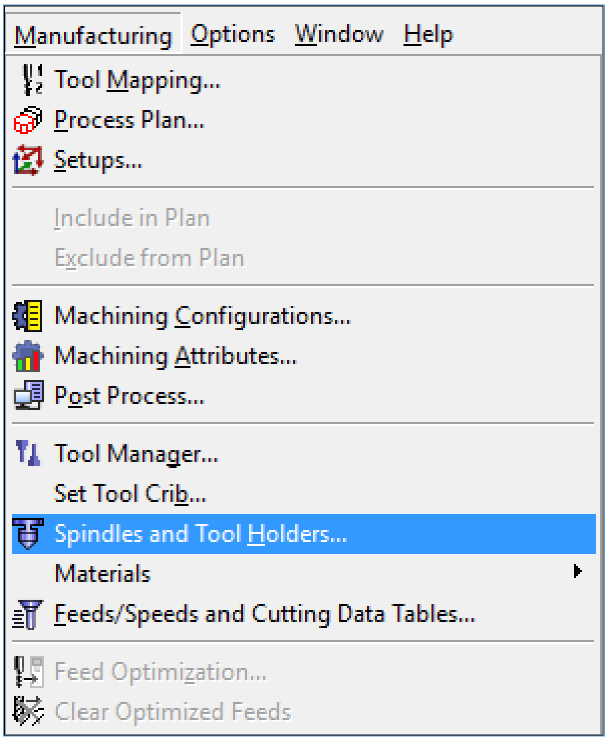
C = Exposed Length **60mm**

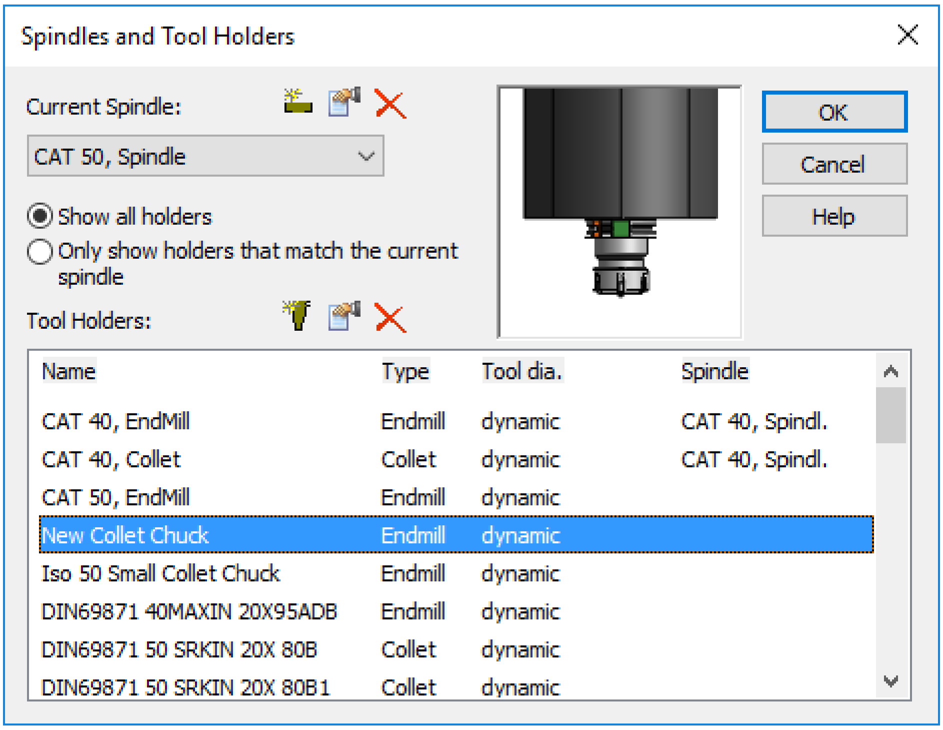
D = Cutter Length (Flute Length) **30mm**

E = Cutter Diameter **10mm**

Spindle and Tool Holders

- This can be accessed by going to **Manufacturing/Spindle and Tool Holders** and allows you to create and edit Tool Holder Properties.





Tool Holder Exercise

*One of the best ways to create a Holder in **FeatureCAM** is to import a Solid Model of the Tool Holder.*

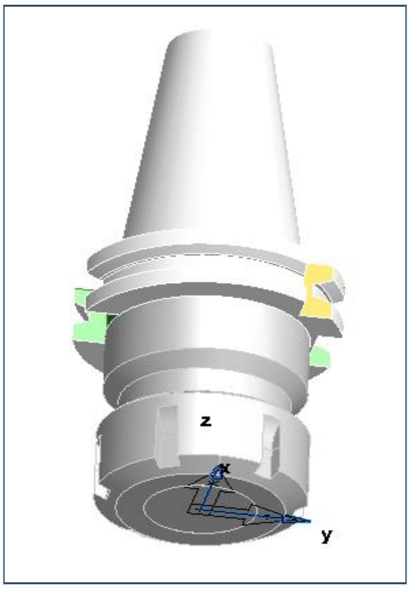
1 Import the file **Din 69871 Collet Chuck Holder.x_t** into you current **FeatureCAM** document.

2 **Cancel** the Import Wizard

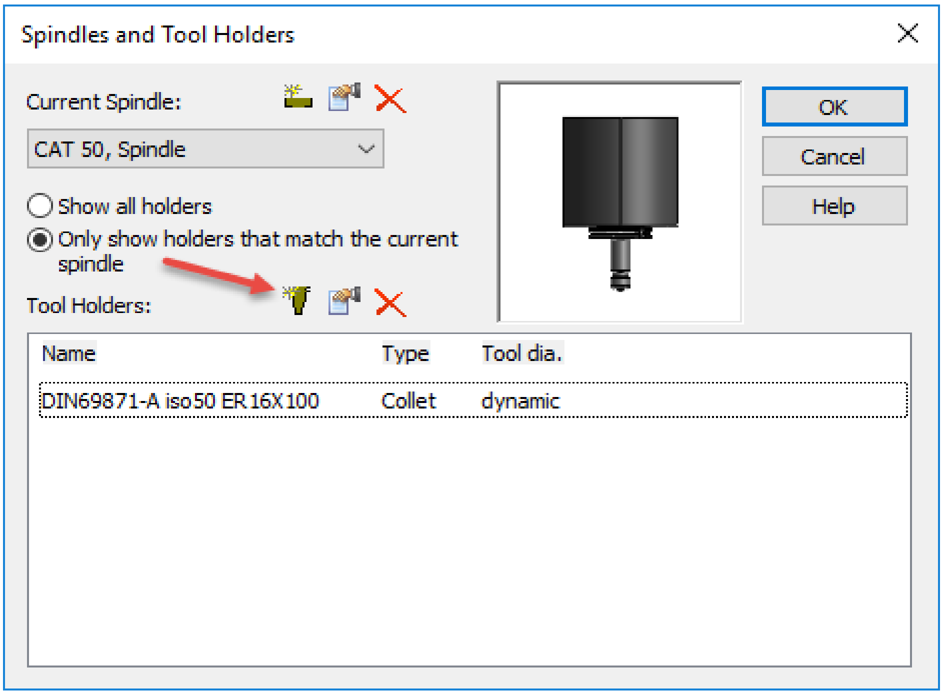
3 Hide **the Stock.**

4 The **Setup** and **Stock Axis** are in the correct position. At the Holder base point in the center of the Solid Model.

5 Right click and set the **View** to **Front** or **Ctrl+2**



6 Open the **Spindles and Tool Holders** form from the Manufacturing tab and click the **Create New Tool Holder** icon shown.



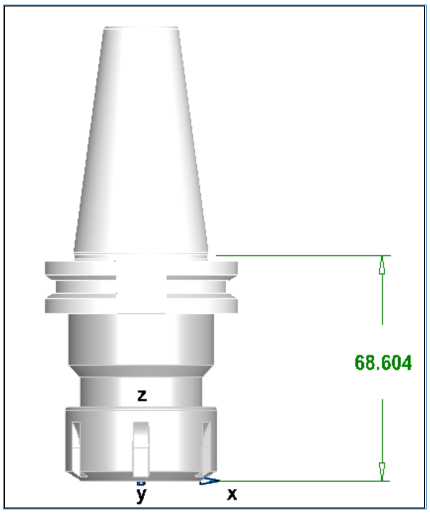
7 Create a New name for the Holder.

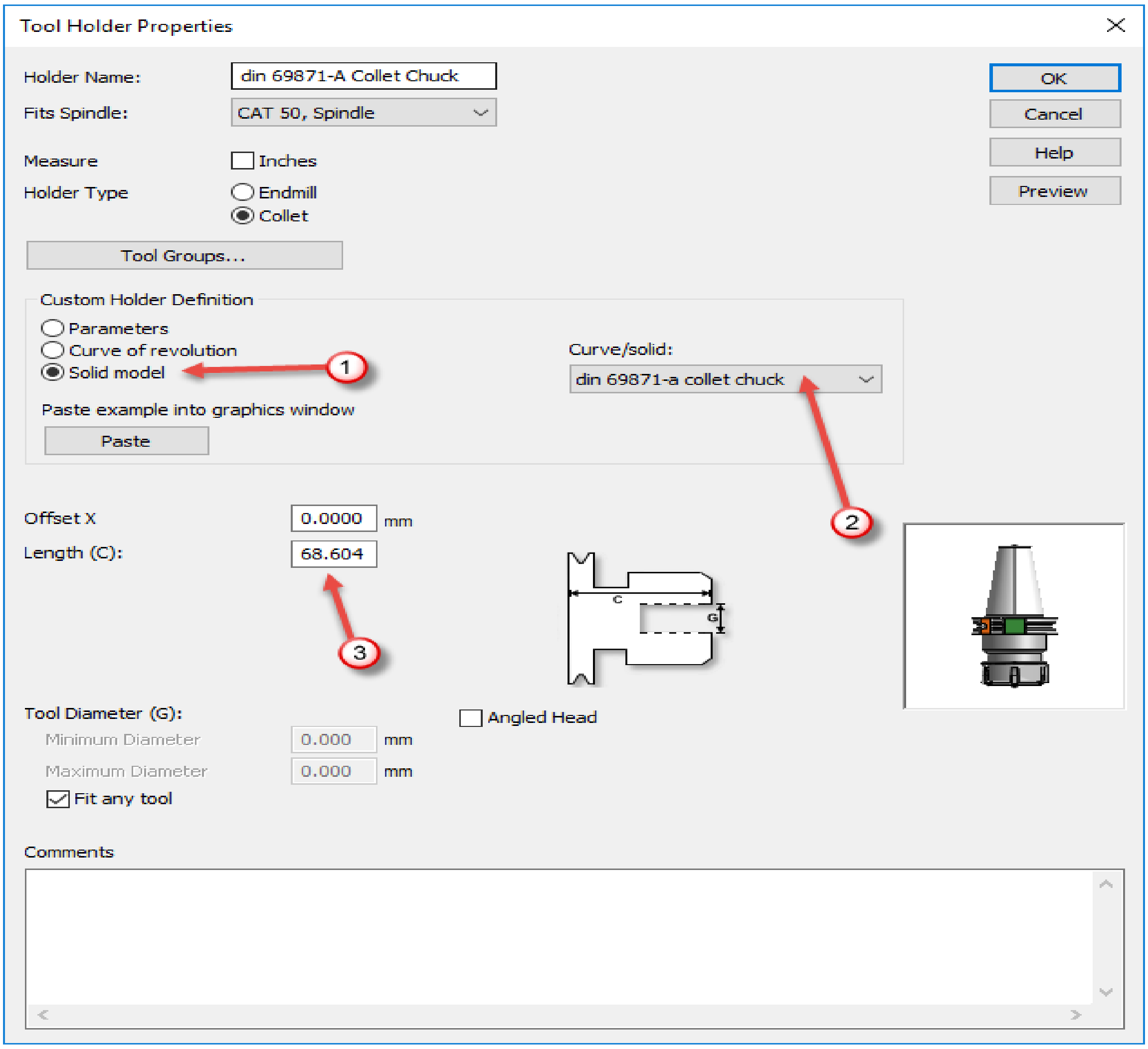
8 Select **Use Solid to describe holder shape.**

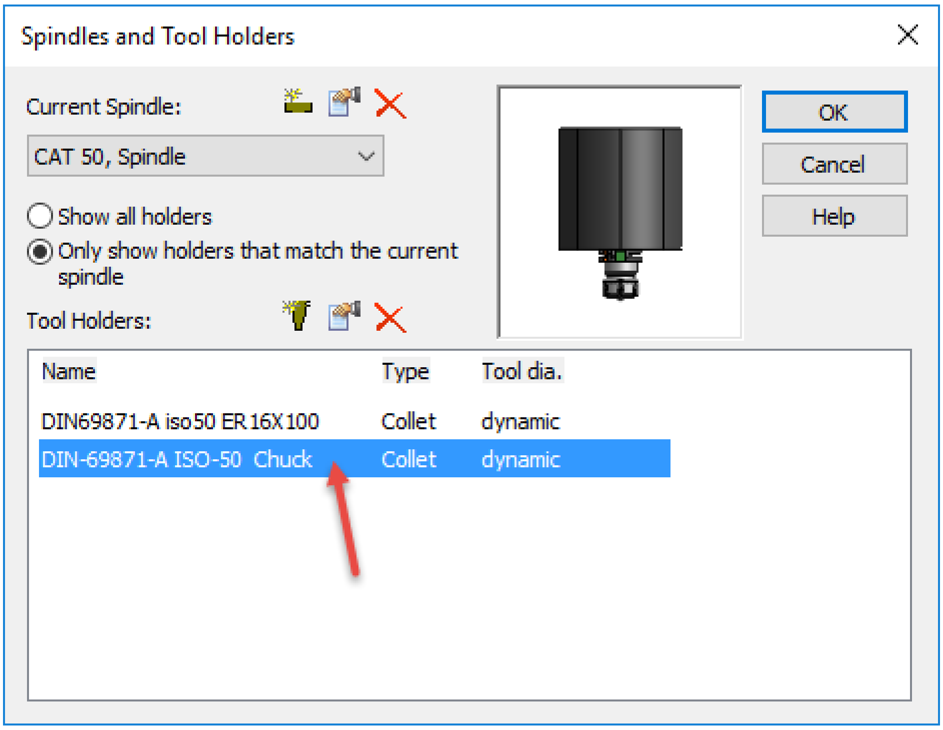
9 Select the holder solid.

10 Enter the Length from the Holder Gauge line to the Holder base center point. In our case the value is **68.604mm**

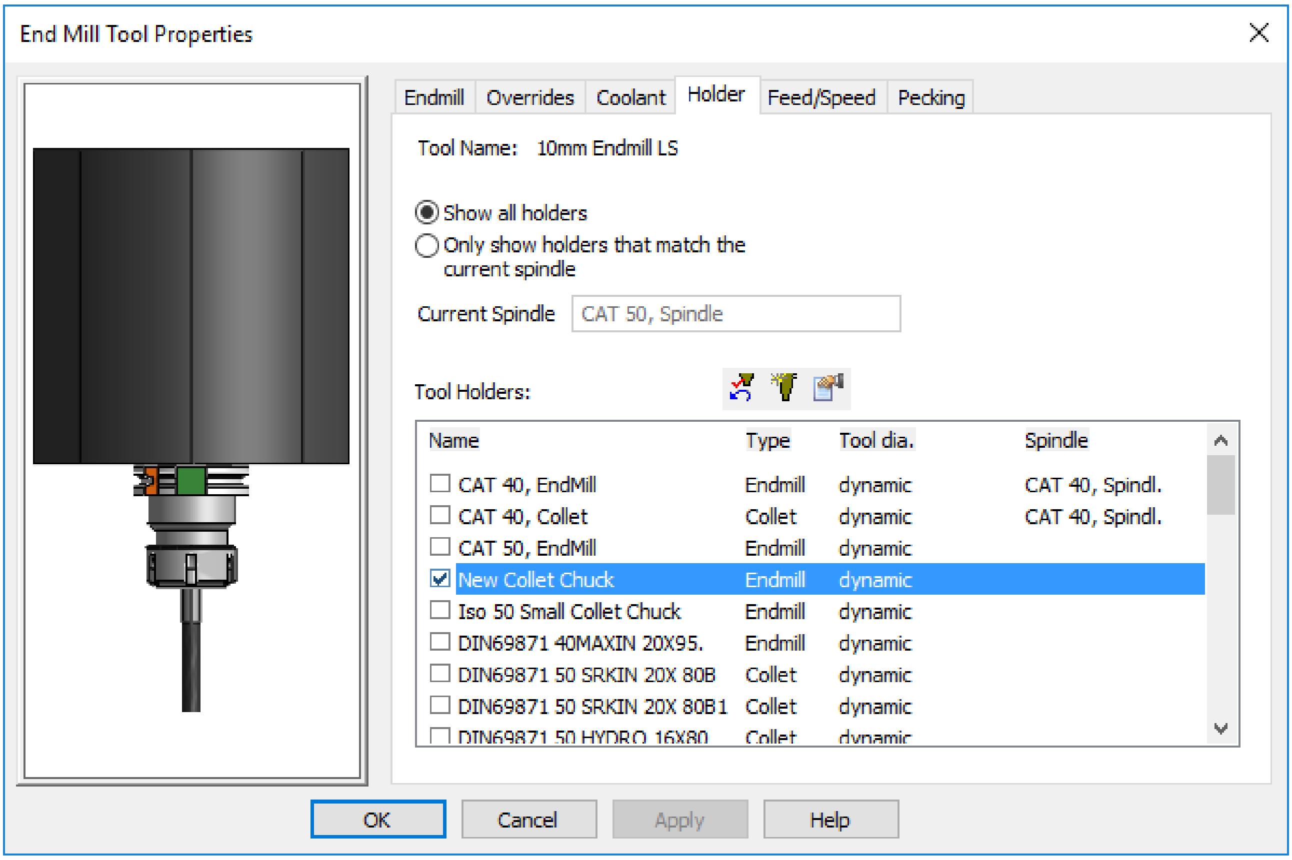
11 Your Holder is now in **FeatureCAM** and can be used for any available tool.



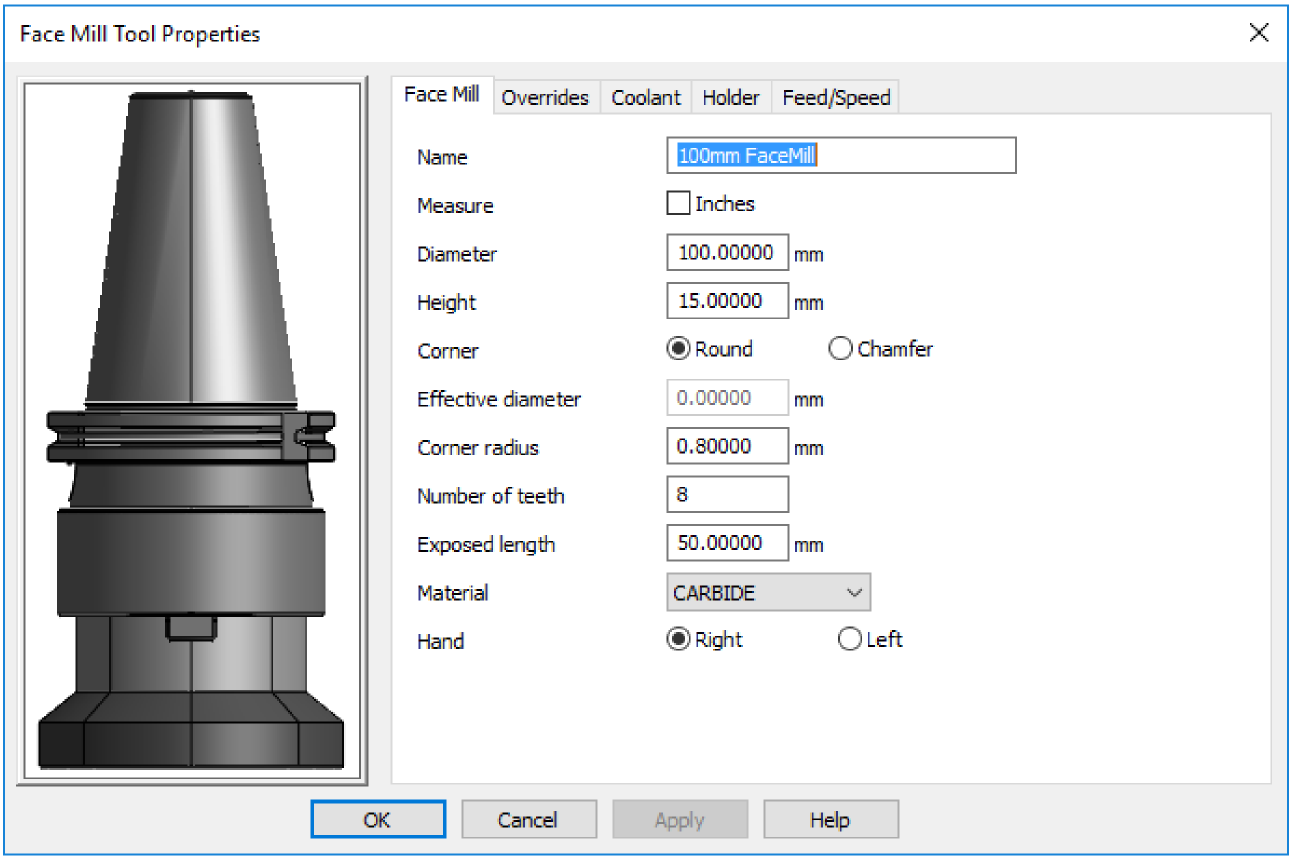




End mill - Properties / Holder / Mill



Face Mill - Tool Properties example.



There is a new option to specify a 45 degree chamfer for ***Face Mill*** *tools:*

This new option allows more accurate simulation and machining. Existing tools are classified as ***Round*** *by default. To set a 45 chamfer on a Face Mill tool:*

1 Select **Chamfer** as the **Corner** type.

2 Enter an **Effective Diameter** greater than zero.

3 Enter a **Tip Radius** greater than or equal to zero.

4 Click the **Apply** button.

User defined Tools (Form Tools)

*In this module the user will learn how to create Form Tools for milling with end mills and side cutting milling cutters. These tools may be used in Milling and Turn/MILL. At the end of the module the user will be able to draw the geometry for the shape of the tool and chain a curve to that shape for **FeatureCAM** to use to create a new tool in the tool crib and save it in a Tool Group called Form Tools that can be recalled at any time.*

Create a Chamfer Milling Tool

1 **Open** a New Milling Document, Metric called **Chamfer Form Tool.fm** for reference.

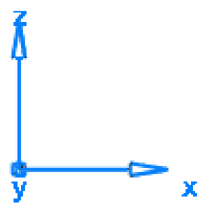
2 **Hide** Stock,

3 **Right Click** in graphics window and View from the front. The **XZ** plane.

4 Select **Front** or select **Ctrl +2** to view in the correct orientation

*It is very important that the **STOCK AXIS** is located at **XYZ 0**. If this is omitted, the curve will not be accessible when creating the form tool.*

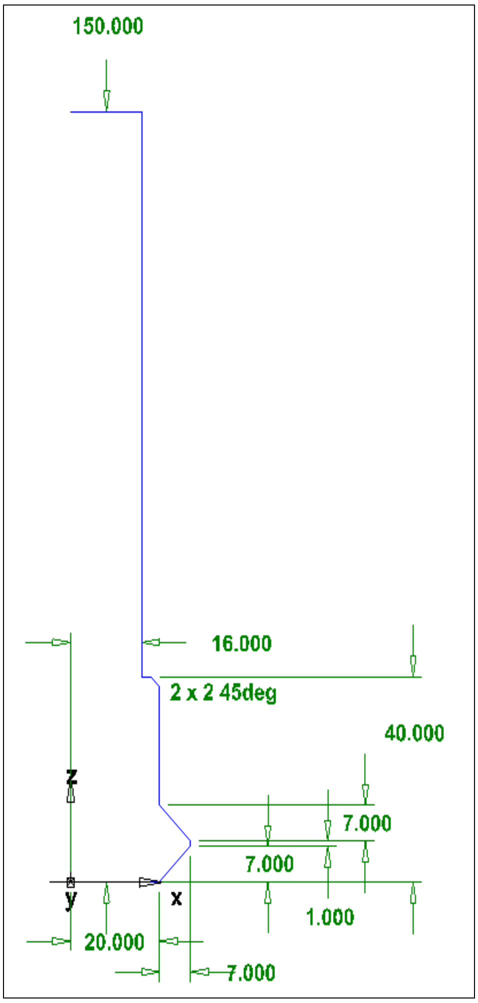
5 To view the **STOCK AXIS** Click View in top menu, select **View> Show**>**Show STOCK AXIS.**

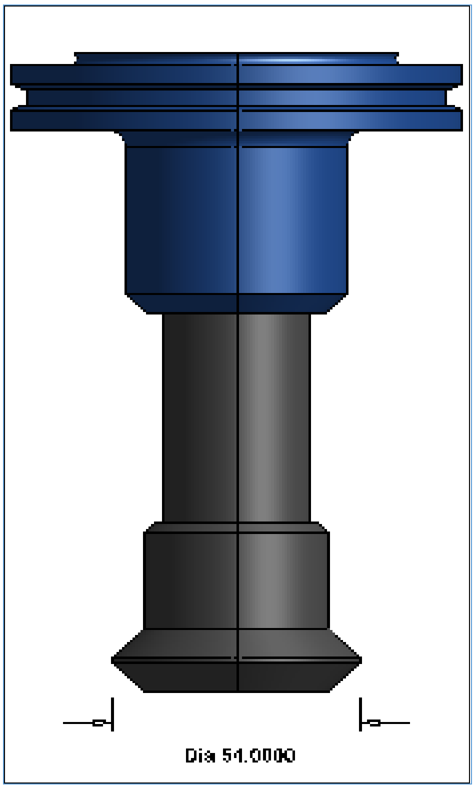


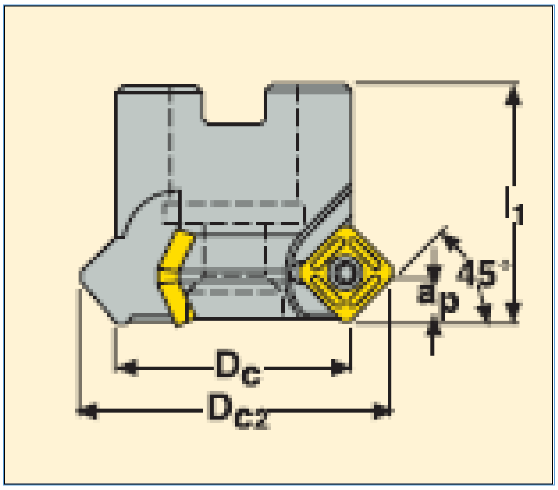
Another set of arrows (blue) will show up in the graphics window merged with the UCS.

*The Z0 will be the bottom of the **Chamfer Form Tool**. The X0 is the center of the tool and only half (radius) of the tool needs to be created (+ side of X). The finish curve will be an open curve.*

6 Using **the Geometry constructors** Draw this shape and **chain an open curve**. (no centreline)


150.000
16.000
2 x 2 45deg
40.000
z
7.000
7.000
y
x
1.000
20.000
7.000


Dia 51.0000



Create a side Milling Form Tool

1 Open a New Milling Document, Metric called Side Milling Form Tool.fm for reference.

2 Hide Stock.

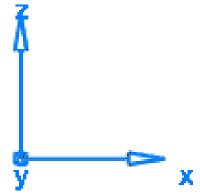
Remember to view the stock axis as shown in blue on the previous example.

3 Right Click in graphics window and View from the front. The XZ plane.

4 Select **Front** or select **Ctrl +2** to view in the correct orientation.

*It is very important that the **STOCK AXIS** is located at **XYZ 0**. If this is omitted, the curve will not be accessible when creating the form tool.*

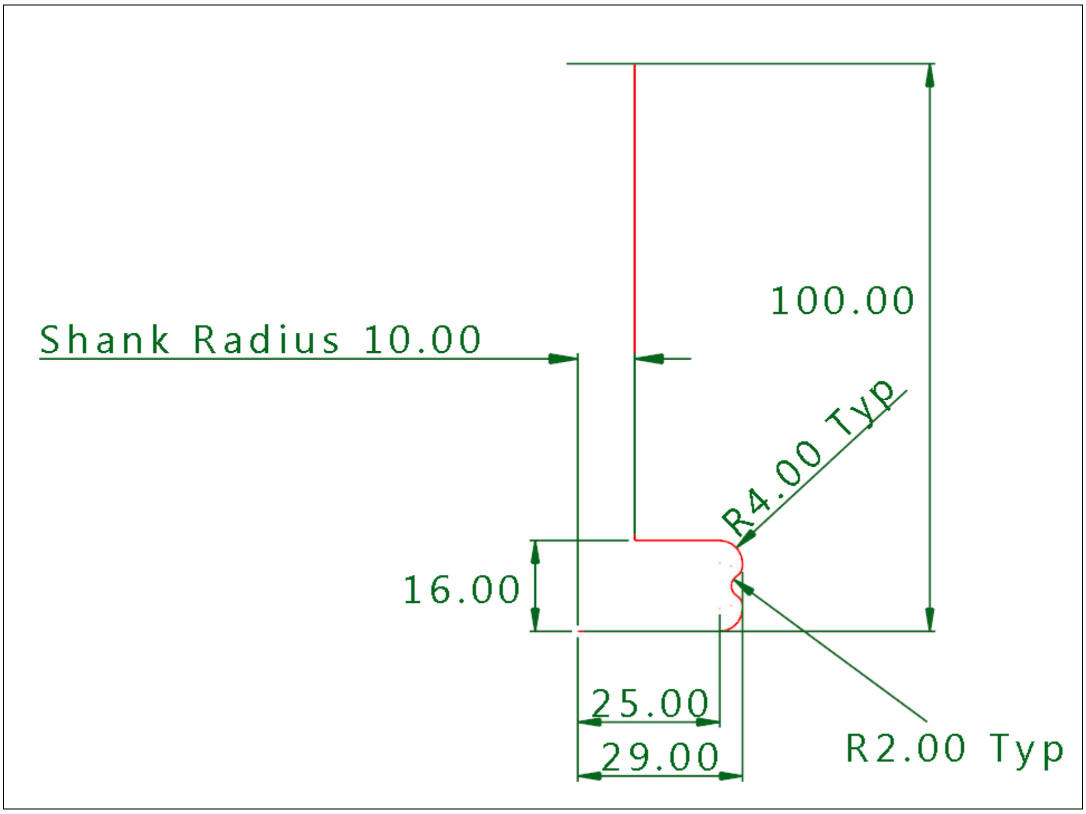
5 To view the **STOCK AXIS** Click View in top menu, select **View> Show**>**Show STOCK AXIS.**

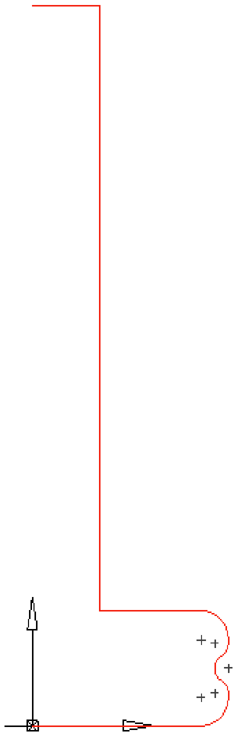


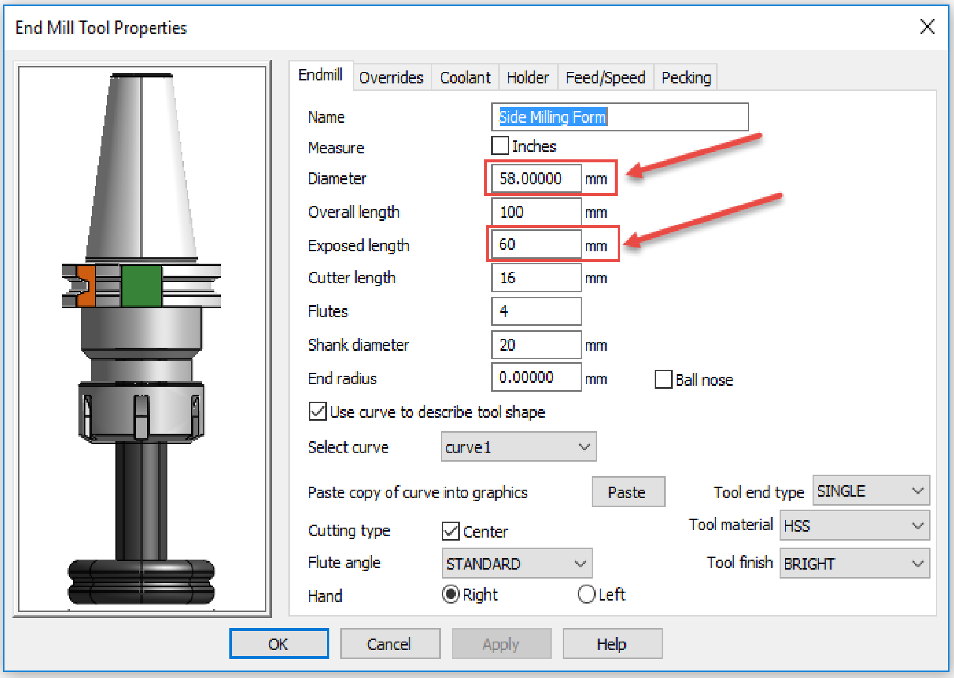
Another set of arrows (blue) will show up in the graphics window merged with the UCS.

*The Z0 will be the bottom of the **Chamfer Form Tool**. The X0 is the center of the tool and only half (radius) of the tool needs to be created (+ side of X). The finish curve will be an open curve.*

Using the Geometry constructors Draw this shape and chain an open curve. (no centre line)


100.00
Shank Radius 10.00
R4.00 Typ
16.00
25.00
29.00
R2.00 Typ

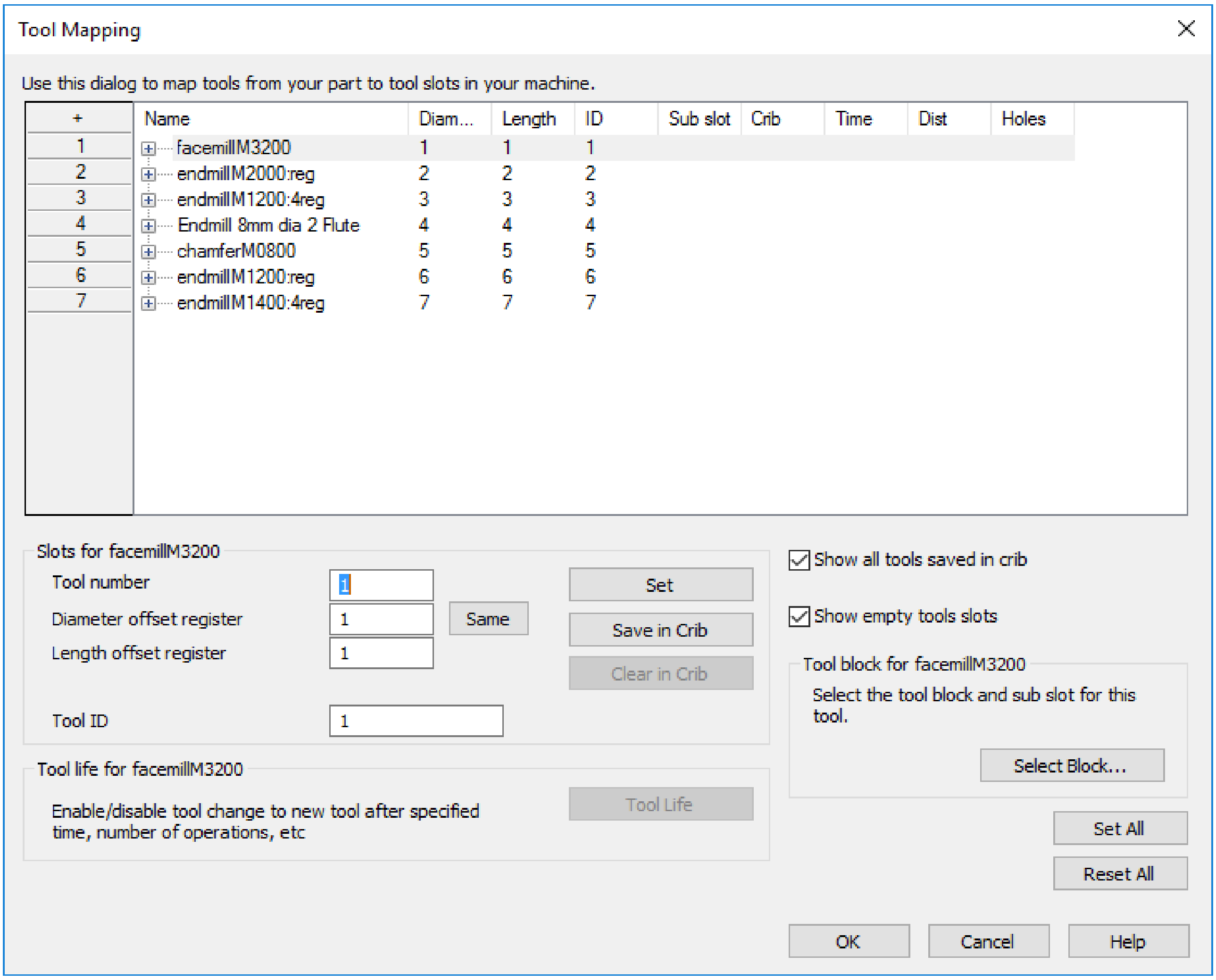



End Mill Tool Properties
Endmill
Overrides
Coolant
Holder
Feed/Speed
Pecking
Name
Side Milling Form
Measure
Inches
Diameter
58.00000
mm
Overall length
100
mm
Exposed length
60
mm
Cutter length
16
mm
Flutes
4
Shank diameter
20
mm
End radius
0.00000
mm
Ball nose
Use curve to describe tool shape
Select curve
curve1
Paste copy of curve into graphics
Paste
Tool end type
SINGLE
Cutting type
Center
Tool material
HSS
Flute angle
STANDARD
Tool finish
BRIGHT
Hand
Right
Left
OK
Cancel
Apply
Help

Tool Mapping

Tool mapping now supports 8-digit Diameter & Length offset registers.

- This ability now supports more machine types because you can now enter 8 digits for the tool Length offset register and Diameter offset register in the Tool Mapping dialog:



*The **Tool Mapping** dialog is where you change the tool slot assigned to the selected tool. You can change the **Cutter comp. offset register** for any tool here too. The dialog has a table at the top. Each row of the table represents a tool. Select a tool to edit its values in the fields below the table. Double-click on a tool name, or click the + to the left of the tool name to see the list of operations that use that tool. Click the Add tool slots button at the top left of the table to open the Number of tool slots dialog. It enables you to increase the number of tool slots listed; you cannot reduce this number. Tool number corresponds to the first (grey) column in the table and is the current tool slot number for that tool. To move a tool to a different slot tool slot, enter a new Tool number and click the Set button, or drag-and-drop the name of the tool in the table onto the tool slot number in the left column. More than one turning tool can occupy the same tool slot.*

Tool Numbering

FeatureCAM automatically selects tools from the active tool crib.

These tools are assigned a tool number (also referred to as a tool Slot or tool pocket) for an automatic tool changer. The numbering is assigned according to these rules:

Use the number assigned in the Tool Mapping dialog.

This numbering is in effect for the current part only.

If no number has been assigned via tool mapping, then the number assigned to the tool in the crib (the Tool number field for milling tools or the Tool slot for turning tools), is used as the tool number. If two tools have the same permanent number in the crib, the first tool used is assigned the pre-set number and the other tool is given a new number.

If no number has been assigned via tool mapping or in the crib, FeatureCAM assigns a tool number.

Tool Life

To open the **Tool Life** dialog, select the tool you want to manage in the table in the **Tool Mapping** dialog and click the **Tool Life** button. The **Tool Life** dialog has different options depending on if the tool is a drill or a milling tool.

See help file for more details

Contents

Machining Configurations / Attributes

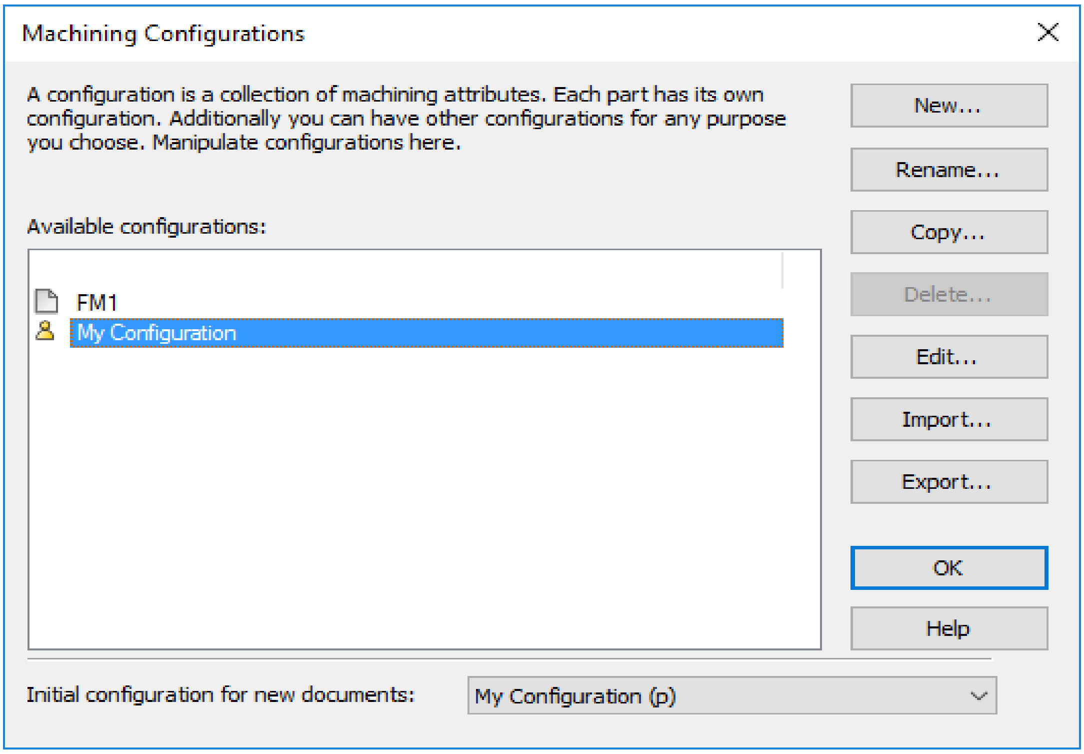
Introduction

This module will provide the user with an understanding of what attributes are, characteristics and features and where they reside Also covering when an attribute is chosen, where it shows up in ***FeatureCAM****, and how it affects the feature.*

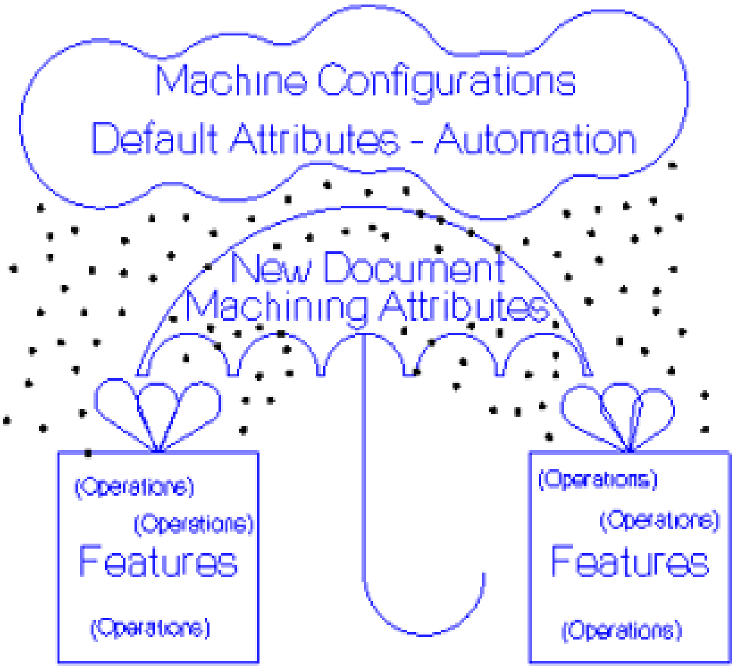
By the end of the module the user will be able to select, prior to creating the program, the machining configuration of attributes, as the defaults for the material and/or machine to be used. This saves time in programming by not having to make the same repetitive entries for the choices you wish to be defaults when that particular material, or machine, is chosen. It also eliminates having to remember the different settings.

Machining Configurations

- The **Machining Configurations** can be accessed from the **Manufacturing** drop down menu.
- Machining Configurations are predetermined default functions that are selected, and used in all features that are created in the document.
- A Machining Configuration contains approximately 150 settings.
- An unlimited number of Machining Configurations may be created.
- The user can name a Configuration to reference the material and/or machine to be used.



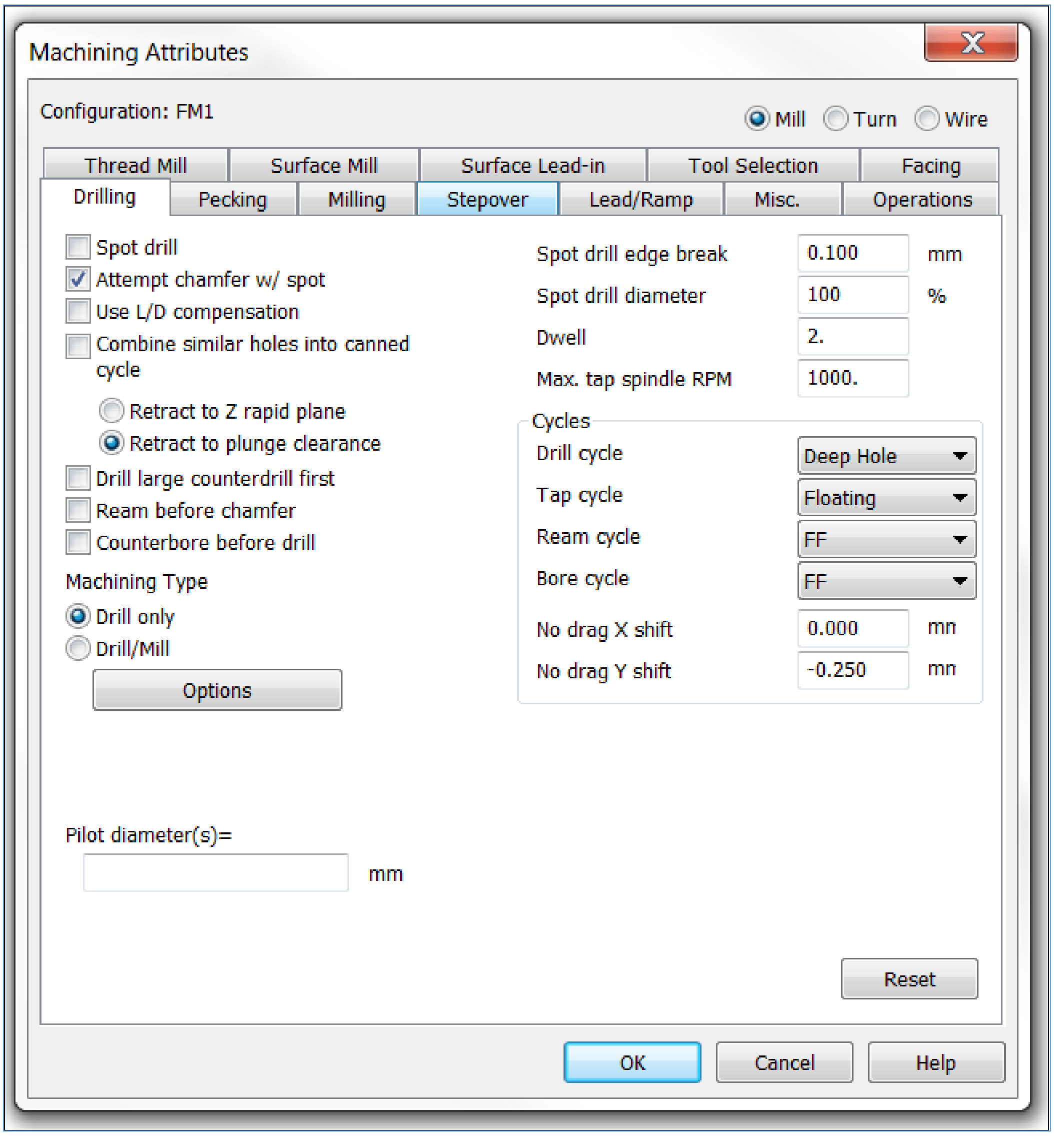
- "Initial Configuration for new documents", is in the drop down menu at the bottom.
- When a Machining Configuration is selected prior to programming, the attributes, so to speak, rain down into the new document just opened. These are used as the default settings. They are divorced from the Machining Configurations and become Machining Attributes, assigned to, and a permanent part of the new document.



These settings may be changed in the document. You can click Manufacturing and Machining Attributes at any time. Changing any attributes will assign them to this document only, but does not affect the original, or default, Machining Configuration.

In other words, these changes do not become the defaults and when a new document is opened the old defaults are active. These settings stay with the document and are active anytime this same .fm document is opened.

What are Machining Attributes?



*When an option in the Machining Attribute is selected, a box is checked or a value is entered, it becomes a default. This means that **FeatureCAM** uses this setting every time a feature requiring that particular setting is selected in a program. There is no input required from the user during programming, as it is now automatically set. However, it may be overridden and changed in the feature as needed.*

Tabs are used to categorize the different operation types and necessary attribute settings.

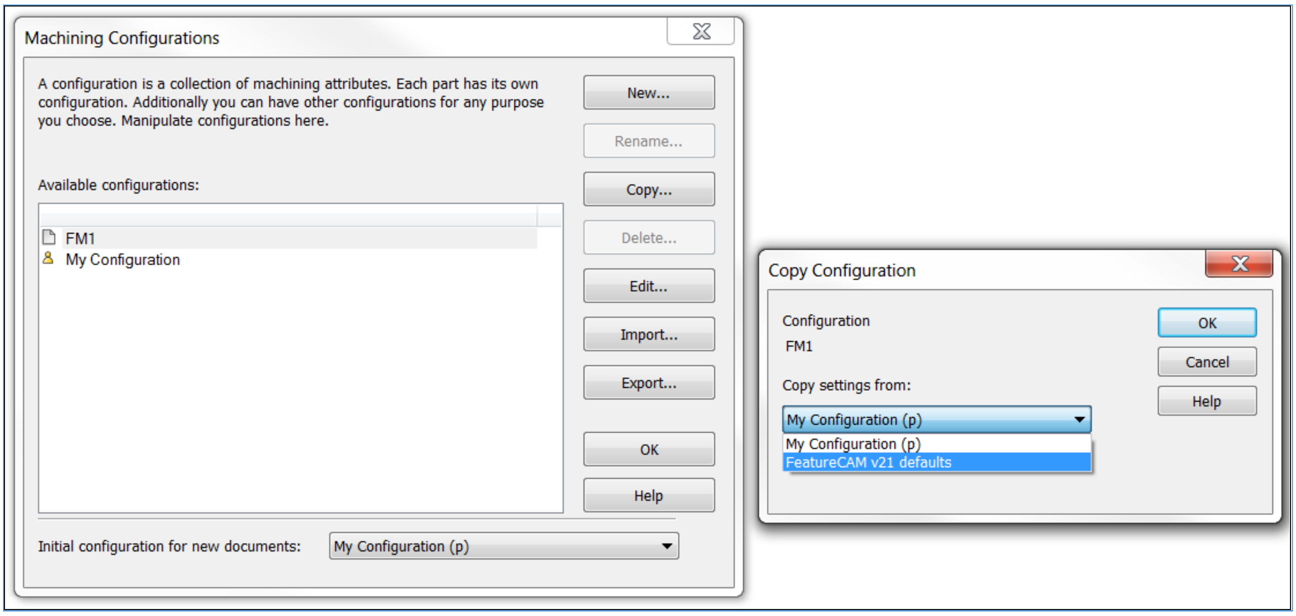
Some selections are check boxes, some are numerical values, and some are specific functions. The majority of these show up in the feature that they affect, but some settings do not. These settings may only be changed in the attributes. This is the reason it is so important to know where these settings reside, what their function is, and where they are located in the features.

To apply the changes you have made, in the Attributes for the document, to become the defaults for any new documents they must also be changed in the Machining Configuration that opened with the file. Once set, and the next time a new document is opened, those changes will then take effect as the default settings.

1 Open a New Document and click **Manufacturing** on the top menu and select **Machining Configurations**.

2 Select the **document you just opened**. Usually there is only one document open but multiple documents may be opened. In this case, you would see multiple documents in the list.

3 Click **Copy** and **select the desired configuration** from drop down menu. These defaults load into the new document.



To copy the changes from the document's configuration into another configuration to be set as the defaults:

4 Select the desired Configuration in the window. This is the configuration you want to copy the attributes to.

5 Click Copy and select the document from drop down menu in the Copy Configuration dialog.

6 Select OK to action Copy Configuration.

This reverses the copy and places the changes from the document Attributes into the configuration selected in the window of the Machining Configurations dialog.

Creating Machining Configurations

1 Open a New Milling Document. Click **Manufacturing** on Menu bar then Machining Configurations.

2 Click on New and type the word **Aluminium** in the field. **OK**

3 Repeat these **steps** to create a **Stainless** configuration. **OK**.

4 Double click on **Stainless** in the window. Select the **Stepover Tab** in the **Machining Attributes** dialog. Change the **Rough Pass depth** to **25%.**

*The Rough pass depth for milling was 100% of the diameter of the cutter. It is now 25%, so for a **25mm** endmill the rough pass depth is set to take cuts at a depth of **6.25mm.***

Note: You now have three separate Machining Configurations. Each has approximately 150 settings.

5 Select the Document you have open in the Machining Configurations dialog at the top of the window.

6 Copy and select **Aluminium** from the **Copy Configuration** dialog.

7 **OK** twice. These settings will be used in the new document.

8 Open an existing file named **Milling Attributes.fm**, from the 2.5D Advanced Data folder, and run a 3D simulation.

*The **12mm** diameter milling cutter cuts to the bottom of the **12mm** deep pocket in one pass because the default attribute specifies the depth of roughing passes is to be 100% of the diameter of the cutter.*

9 Click Manufacturing then Machining Configurations. Select the document in the window on the left named **Milling Attributes.fm**.

10 Press the **Copy** button. Select **Stainless** from the drop down list in the Copy Configuration dialog. Press **OK** twice.

11 Using the same feature and by only changing the Machining Configuration the Attributes automatically change to the Stainless settings.

12 Run a 3D simulation.

*The setting for Stainless is 25% of the diameter of the cutter so now the tool makes four passes at **3.175mm** depth of cut instead of one at **12.7mm** with no further input from the user.*

Machining Attributes Explained

- The rest of this document will be used to explain the attributes found in each of the tabs inside the Machining Attributes dialog. To follow along and compare, click Manufacturing then Machining Attributes to open the dialog.

Drilling Tab

- **Spot drill**: All holes that are drilled will receive a spot or center drill operation before the hole is drilled.
- **Attempt chamfer w/spot**: When a chamfer is entered in a hole feature, a spot drilling tool is selected that is large enough to cut the chamfer as it center drills for the hole feature. If the hole is too large, it will still be spot drilled but will give a warning that it was not able to achieve the chamfer and use a chamfer tool in addition to the center drill.
- **Use L/D Compensation**: As a hole is drilled deeper the feed and speed is adjusted to compensate for the additional depth. This is true for holes that have a ratio of hole depth (L) to hole diameter (D) of greater than 2.5. The greater this ratio, the greater the speed/feed reduction becomes.

- **Combine with similar holes into canned cycle**: This Attribute serves two functions. First it creates more efficient NC code by entering canned cycle mode only once. It also causes the tool to retract to the lower Plunge Clearance plane after drilling each hole, typically .100 above the hole instead of the Z Rapid Plane which is usually set much higher to clear clamps, etc. Retracting to the higher plane when necessary is possible on a per hole bases in the Results Window Op List by clicking on the green arrow of the selected hole.
- **Drill large counter drill first**: A counter drill hole has a second, larger hole drilled on top of a smaller hole. You may select to drill the larger hole first to avoid drilling the same material with the smaller tool that will be removed with the larger tool. The smaller tool will then rapid to the plunge clearance above where the first drill stopped drilling.
- **Ream before Chamfer**: Allows you to ream a hole before the chamfer to avoid pushing any kind of a burr or edge back up onto the chamfer in the event the chamfer is a sealing surface.
- **Pilot drill diameter**: Set the diameter for a frequently used pilot drill size as a default.
- **Spot drill edge break**: If 0 chamfer is entered in a hole feature, this setting will cause the center drill to break the edge by this amount, typically .005.
- **Spot drill diameter**: This percentage is used to select a spot drilling tool. A value of 100 specifies that the spot drill should be the same diameter as the hole. A smaller value will create only a starter hole.
- **Dwell**: The spot drill will dwell for this many seconds.
- **Max tap spindle RPM**: Sets the maximum spindle speed that **FeatureCAM** will select when tapping regardless of the size of the tap.
- **(Cycles)**
- **Drill cycle**: **Deep hole** will peck and retract to the plunge clearance and return to previous depth and **chip break** only stops feeding in order to break the chip, both perform their function until hole is to depth.
- **Tap Cycle**: **Floating** (floating & tension-compression holders), **Rigid** which is most commonly available on today's machine tools being sold, **Deep hole** and **Chip break** is the same as drilling.
- **Ream Cycle**: Ream cycle affects how a ream operation is performed. The choices are **Ream FDF** (feed-dwell-feed), Ream FF (feed-feed), and **Ream FSR** (feed-stop spindle-retract).

  If you select** Ream FF**, the cycle is posted using the Bore (F-F) format in XBUILD.** Ream FDF **will use the Bore (F-D-F) format, and Ream FSR uses the Bore (F-S-R) format.

  FDF, feed down-dwell-feed up, **FF**, feed down-feed up, **FSR**, feed-stop-retract.
- **Bore cycle:** **FDF**, feed down-dwell-feed up, **FF**, feed down-feed up, **FSR**, feed-stop-retract and **No Drag** shifts a designated amount in **No drag X** and **Y** shift so as not to touch the finished wall upon retract.

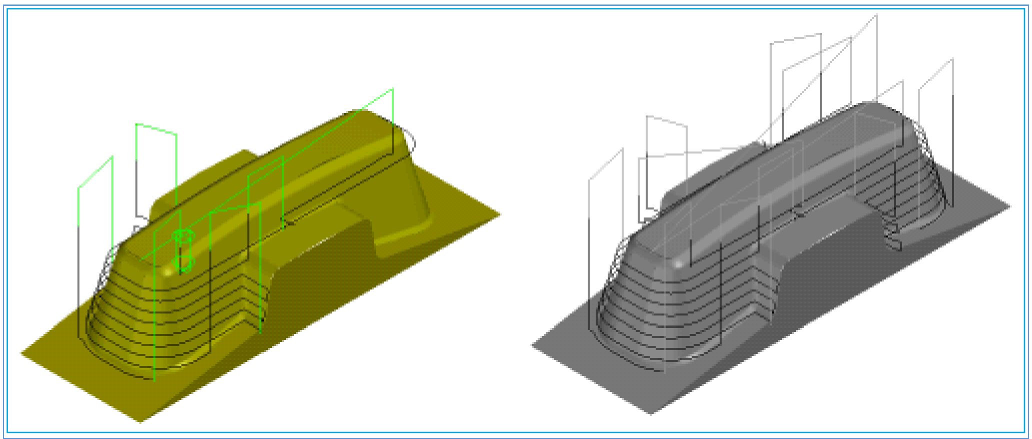
Pecking Tab

- **Drilling**: Depth of the First, Second and Minimum peck may be entered by the % of the drill diameter.
- **Tapping**: Depth of the First, Second and Minimum peck may be entered by the % of the tap diameter.

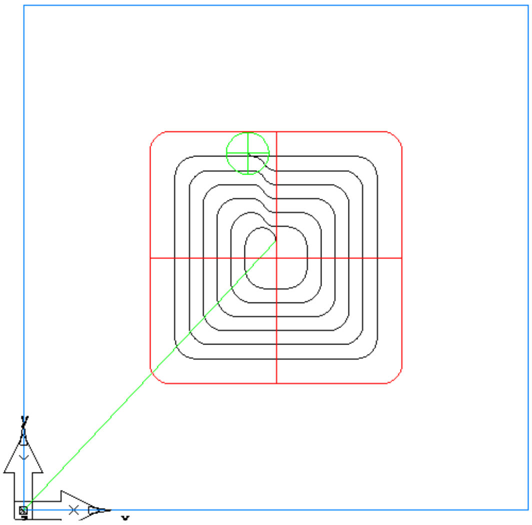
Milling Tab

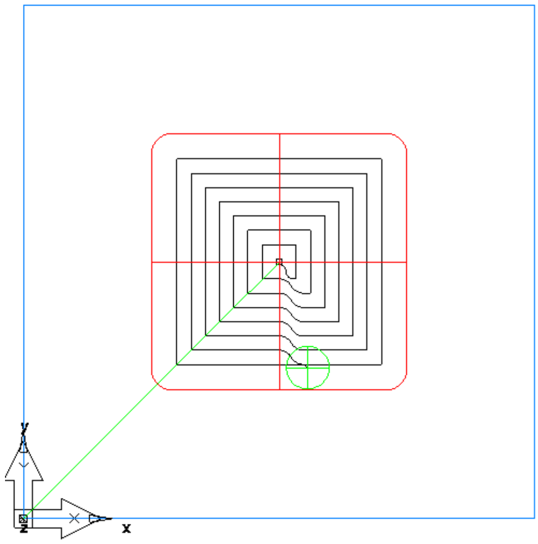
- **Climb mill**: With the milling cutter rotating clockwise if you follow behind the cutter in the direction of the travel of the tool and the cutter is on the left side of the curve it is climb milling and if it is on the right side it is conventional milling.
- **Bi-directional rough**: The rough passes alternate between climb milling and conventional milling. When cutting a Boss feature on a square block, the cutter cuts across the corners first. When climb milling is selected in the strategies, the first pass is a climb cut, and then the tool alternates with conventional milling until the corners are finished. The cutter finishes the roughing cuts using climb milling.
- **Use finish tool**: When rough and finish milling, **FeatureCAM** uses the same cutter to rough and finish. When Use finish tool is checked **FeatureCAM** will always add a second tool to do the finish operation.
- **Cutter Comp**: Whether cutter comp is selected or not **FeatureCAM** offsets the tool by the radius of the tool. By selecting Cutter comp, it adds the necessary G40, G41and G42 to the NC code so the tool size for holding tolerances may be adjusted with the tool offset in the control by the user.
- **Part line program**: Instead of the cutter center being offset by **FeatureCAM** the program aligns the center of the tool on the curve. The NC code reflects this with the print dimensions. All of the offsetting is done by the machine control.
- **Minimize tool retract**: For use when it is more desirable to feed the tool to another region of a feature instead of retracting and plunging. For example: in a pocket with islands, minimize tool retract will keep the tool from retracting and plunging.
- **Individual rough levels**: Many roughing cuts are performed at multiple Z levels due to the depth of the feature. If you select Individual levels on the Strategy page, you will be able to customize the manufacturing attributes of each level. If you have a feature that is roughed in four levels there will be four z-levels listed in the feature instead of one. Note that each pass is listed underneath the rough operation. By clicking any of the passes, you can set attributes in the Milling tab. If Individual level is not selected, then only the rough pass is listed in the tree view and you can only make changes to Milling tab attributes that will apply to all levels. Individual levels also controls clipping of boss and side features against the stock model including both STL and solid stock models. With this attribute selected, the toolpaths are clipped against the stock boundary at each Z level.

- **Depth-first machining**: Selecting the Depth first option will cut each region of the feature completely before moving on to another region. The toolpaths descend in Z. If this option is deselected, then all regions of a feature are cut at one Z level before descending to a deeper Z-level. If you are using multiple roughing tools or multiple finishing tools, to efficiently rough out tight corners, Depth first is also useful. The second tool that roughs the corners only will cut each corner to depth before retracting and moving to the next corner.
- **Reorder**: (Surface Milling) The Reorder attribute tells **FeatureCAM** to re-sequence the toolpaths to minimize retractions while trying to avoid full width cuts. Use Reorder when you have a part where several separate regions are cut. If you want the toolpaths to move directly across a surface without worrying about retractions, deselect Reorder.
- For Z-level operations (rough or finish), the Reorder attribute enables zone machining where the toolpaths descends in the Z (or -Z) direction. Use this if it is more efficient than cutting the entire part in complete Z levels. The phone handset example below shows that the toolpaths cut the top of the part in complete Z levels and then cut one side and the other.

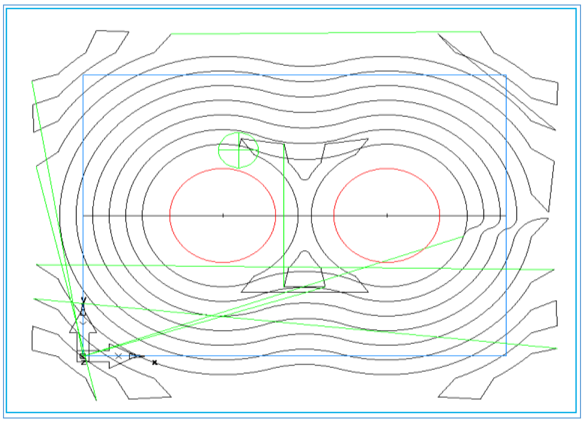


- **Side roughing bottom up: A 2.5D feature** with a Draft Angle may be roughed from the bottom up instead of the normal top down.
- **Side finishing bottom up**: **A 2.5D feature** with a Draft Angle may be finished from the bottom up instead of the normal top down.
- **Toolpath corner** (%): This attribute rounds the corners of milled roughing passes. It is specified as a percentage of the tool diameter. Rounding the sharp corners of the toolpaths provides a more constant tool velocity and reduces the tool load.
- It applies to all 2.5D milling features. The left figure shows a pocket without Toolpath corner % set and the right is set with Toolpath corner set to 98%:

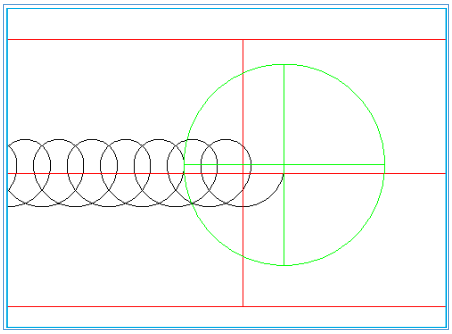




- **Use alternative 5 axis position**: Depending on the type of 5-Axis machine you are working with, there are often two ways of orienting the machine axes to address a particular face.
- **Hi speed machining technique**. The first rough Tool path corner at 500% flattens the tool path, usually faster feeds and speeds. The second rough Tool path corner at 25% cleans up only what the first rougher left. The 25% value helps smooth out some of the sharp corners, but must not be too large, or there might be an excessive amount for the finishing tool.



- **Trochoidal cut**: Simple grooves have an option of using a Trochoidal toolpath. Instead of a simple slotting cut, the tool uses a series of circles to clear away the metal, as shown. This toolpath has the advantage of maintaining a constant radial depth of cut which produces the same load on the tool at all times.



Step over Tab

- (**Rough Pass)**
- **Do rough pass creates a rough operation for ALL milling features.**
- **Depth** in the Z direction is a % of the diameter of the cutting tool for each pass.
- **Spiral** is the Tool motion offset from the curve shape and the tool steps over as it moves radially toward the curve, this step over is a % of the tool diameter. **(33% or less is recommended)**
- **Zig-zag** is how the tool travels parallel to X or Y or at an angle, the radial step over is a % of the diameter of cutter. (Up to 55% is recommended)
- **(Semi-finish Pass)**
- **Do semi-finish** pass is selected if an additional finish pass is needed before the final finish pass and will be applied to all milling features.
- **Allowance** is the amount of material left for the finish pass.
- **Bottom Allowance**, if required, is the amount left on the bottom for the finisher to remove.
- (**Finish Pass)**
- **Do finish pass** is selected if a finish pass is desired and will be applied to all milling features.
- **Allowance** is the amount left by the rougher for finishing.
- **No. of Passes** made by the finish tool, additional passes may be added, they are considered spring passes the tool does not move in any further then the finish curve.
- **Overlap** is the amount from where the tool ramps on for the finish cut and where it ramps off when complete.
- **Finish bottom** is selected if fishing the bottom of the feature is desired and Wall pass is checked if it is to be included.
- **Spiral** is the type of finish toolpath with a % of the tool diameter, radial step over.

- **Zig-zag** is the type of finish toolpath with a % of the tool diameter, radial step over.
- **Bottom allowance** is the amount of material left by the rougher when finish bottom is selected.

Lead/Ramp Tab

- **(Horizontal lead/ramp)**
- **Extension distance** pertains to Lead moves for cuts with open ends. In the milling feature on the stepover tab, there are settings to extend the lead onto the feature without having to extend or change the feature. An arc ramp may be added and a 90 degree approach that is perpendicular to the arc may also be included.
- **Lead distance** is the distance the tool plunges from the end of the curve and is measured in a % of the tool diameter.
- **Lead in angle** is the angle to the curve that the tool approaches the feature after plunging. A 90 degree Lead in is perpendicular to the curve.
- **Lead out angle** is the angle to the curve when the tool exits the feature before retracting. A 90 degree Lead out would be perpendicular to the curve.
- **All Stepover** applies the Lead in and Lead out to every step over.
- **Ramp type** is the shape of the transition move when the tool steps over radially. They include Direct (90 degrees) Arc, Line and S-shape. S-shape provides very smooth transition, good for hi-speed machining.
- **Ramp diameter** is measured as % of tool diameter and determines the arc size for the radial step over.
- **Minimum ramp distance** controls the length of the ramp.
- **Wind fan finish** plunges and retracts the finish tool in the same place and ramps on and off the finish curve in the same place with no overlap. The arc size for ramping may be entered along with the angle move to the arc. To watch this in centreline simulation, it looks like a fan you would hold in your hand.
- **(Vertical ramp**)
- **Minimum Z ramp** dist. is the allowable minimum distance (radially) when ramping has been selected. It may be 0 and if the tool doesn't fit when ramping FeatureCAM will plunge straight down with no ramping to attempt to make it fit.
- **Max ramp angle** is the maximum vertical angle the tool is allowed to plunge (rough op)
- **Max finish ramp angle** is the maximum vertical angle the tool plunges (finish op)
- **Helical Ramping** is a technique that plunges the tool into the material in a helical (circular, like a screw) motion and may be CW or CCW.
- **Linear Approx.** may be used when helical is not applicable. It uses short linear moves for the helical motions instead of arcs. (also thread milling)
- **Ramp diameter** is the radial size of the helical arc that is used. (thread. milling)
- (**Arc lead**)

- **Arc lead**, Use 90-deg Comp on/off and Distance are associated with **Extension distance** and pertains to Lead moves for cuts with open ends.

Misc. Tab

- **Z rapid plane** is the distance measured in the Z positive direction that the tool retracts to when either moving to the next hole for drilling or moving to another region to continue milling and usually considered a safe height for clearance of clamps, etc.
- **Plunge clearance** is the distance in the Z direction above the part that a tool will rapid to and then go into feed rate to plunge or drill to depth.
- **Tap plunge clearance** has the same purpose in a tap operation as for drilling and milling.
- **Z ramp clearance** is the distance above the part in the Z direction that a milling tool feeds straight down to after plunging to the plunge clearance when ramping is being used so it doesn't ramp above the part.
- **Spline Tolerance** parameter controls the maximum deviation of the generated tool path from the theoretical spline contour.
- **Z index clearance** is applicable on 4 axis indexing parts. When for example a square part is indexed on a 4th axis table FeatureCAM takes into account the corners being higher than the flats during index and retracts the tool to the Z index clearance height above the corners which is safe for the corner to index under any tool.
- **Wrap tolerance** also applicable to 4th axis parts. Certain features can be created flat as normal and then wrapped around a cylinder on a 4 axis rotary table and this tolerance setting determines how close.
- **Chamfer depth** is how far the point of the chamfer tool goes past the bottom of the chamfer on the part. This is to make sure the point of the tool uses the full length of the cutting edge and not give a burr along the bottom edge because the point of the tool is possibly not sharp to the end.
- **Deburr radius** is automatically applied to a sharp outside corner to remove the burr, for example on a boss feature where there is no radius on the corner. It can be any size and if there is any radius whatsoever on the curve of the feature then FeatureCAM ignores the Deburr radius and the curve radius is applied instead.
- **Minimize corner radius** is used to restrict the maximum size tool that FeatureCAM will pick when there is a square inside corner with no radius.
- **Minimum rapid distance** is the distance determined by a percentage of the tool diameter as to whether the tool will feed or retract and rapid to the next region to be machined. The smaller the setting the more retracting, the greater the number then it will feed to the next area to be machined.
- **Speed** is the percentage of programmed spindle speed that the spindle will run in the program.
- **Feed** is the percentage of programmed feed rate the tool will feed in the program.
- **Plunge feed** is the percentage of the programmed feed rate that the tool will feed when plunging.
- **Use IPR/IPT** will use inches per revolution or per tooth instead of inches per minute (IPM).

- **Proportional plunge feed** when selected determines the milling plunge feed rate according to the angle of ramping while plunging. The steeper the angle the slower the feed rate and the more shallow the angle the higher the feed rate.
- **Peripheral Feed** has various settings for inside and outside corners during milling to slow or increase the feed rates to help maintain a more consistent tool load and obtaining the optimum feed rate.
- **Post Variables p**rovides the opportunity to pass data directly to the NC code in designated locations as laid out by the post variables built into the post processor.

Operations Tab

(Ordering)

Automatic Options has four functions:

1. Minimize tool changes attempts to optimize tooling to do all the work for that tool on the entire part before changing tools, especially helpful when one tool can be used on several functions or features including 4th axis, more indexing and less tool changes.
2. Do finish cuts last allows all of the roughing of milling features to be completed first and then does the finish cuts to avoid any possible movement of the part after a finish cut has been completed.
3. Cut higher operations first completes for example a pocket with drilled holes in the bottom, pocket first then the drilled holes.
4. Minimize rapid distance helps to find the most efficient path for the tooling by examining the distance of each move and then picking the shortest distance.

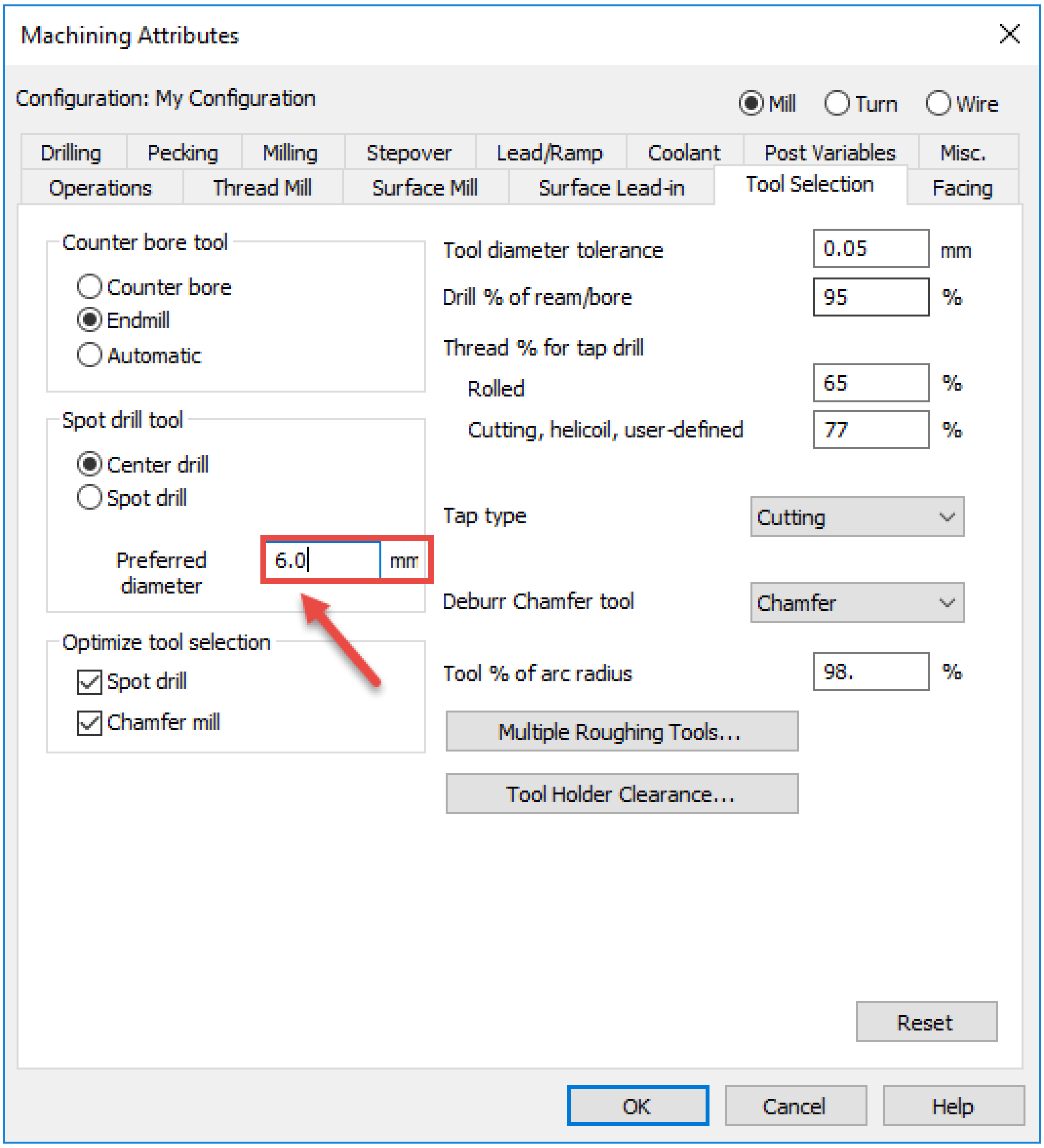
- **Base Priority** is when you want to ensure that an individual feature is cut before anything else, you can set its **Priority** attribute in the **Misc**. tab. All features have a Priority manufacturing attribute. By default, the value is 10. To make sure that a feature is manufactured first, set its priority to a lower value and to make a feature last, set its priority to a higher value. For example, if you set the **Priority** of a pocket to 8, its roughing pass is the first operation performed, its finish pass is second, and the rest of the operations are ordered according to the scheme described above.
- While you can specify the exact order of every feature by priority, you shouldn't do so casually because you lose the automatic optimization sequences built into FeatureMILL. It's harder to maintain or change the part too.
- **Time estimation** has six settings including indexing speed of the 4th axis rotary table. If care is taken to enter accurate figures, a very close estimation of the actual time it will take to run your part, excluding any operator intervention can be achieved. A stop watch may have to be used instead of accepting speeds from manuals.

Facing

- **Do rough pass** when roughing passes are desired.
- **Do finish pass** when a finish pass is desired? May be used without rougher.
- **Finish allowance** is the amount of material left by the roughing operation.

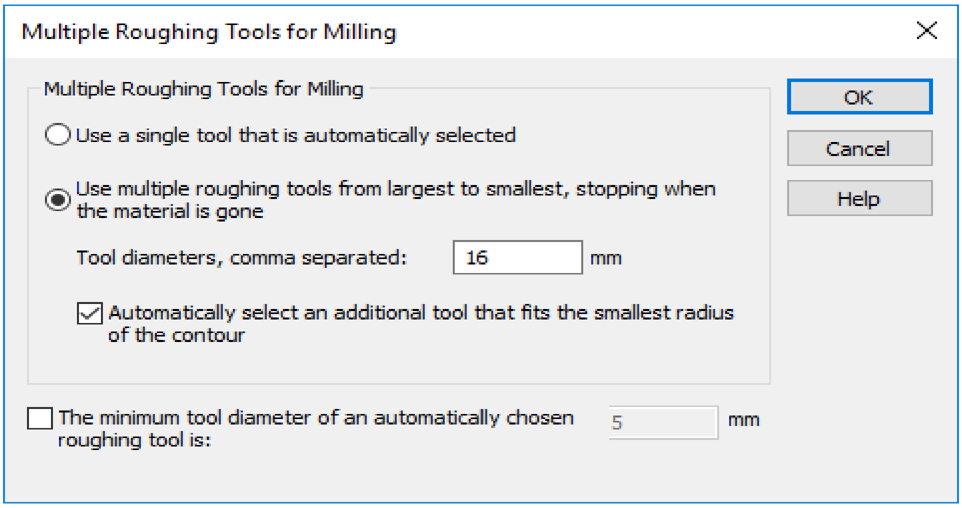
- **Facing stepover** is the radial step over perpendicular to the cut direction measured by a % of the cutter diameter.
- **Lateral overcut** is how far, measured by a % of the cutter diameter, that the tool goes off the end of the part parallel to the direction of the cut.
- **Last pass overcut** is how far by a % of the cutter diameter, that the tool goes off the part radially perpendicular to the cut direction on the final pass.
- **Max depth** of cut is the maximum Z increment depth the cutter will take in one pass.
- **Connect stepover** with arc provides a very smooth transition by changing the direct 90 degree stepover to an arc or loop. This is also a high speed machining technique.
- **Zigzag angle** is the angle that FeatureCAM uses to cut Face features.

Tool Selection



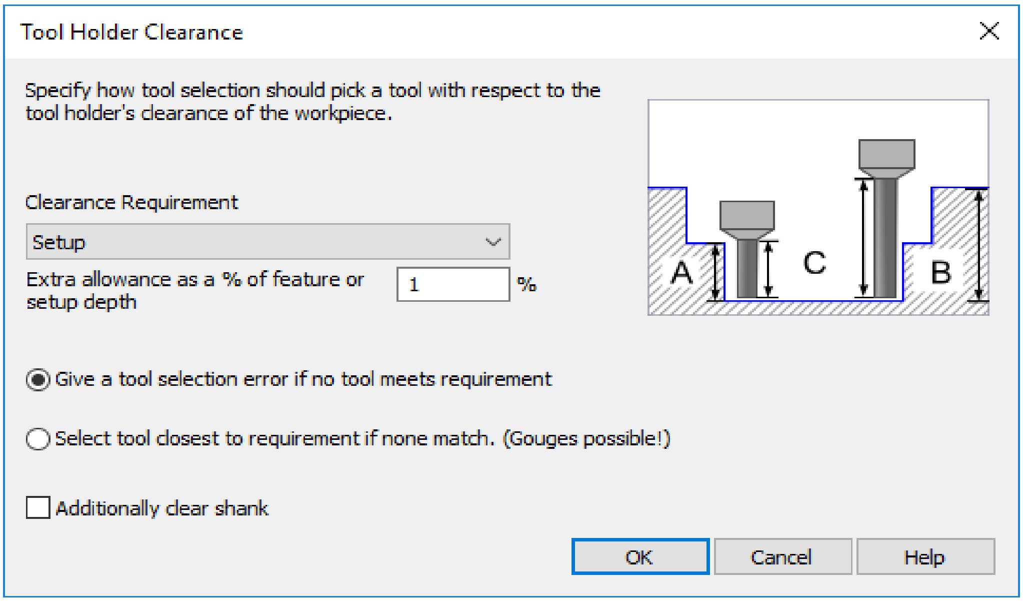
- **Counter bore: -** will select a specific diameter counter bore tool. An inventory of various sizes of counter bores is usually maintained. Counter Bore Eliminates an inventory of counter bore tools and can cut any size.
- **Automatic selects** Endmill when counter bore tools are being used and the desired counter bore size is not available.
- **Spot Drill: - Prefer spot drill** is used to cover a wider variety of hole sizes because it has one diameter and comes to a point.
- **Prefer center dril**l is usually selected for spotting a hole to be drilled, short and makes the initial plunge into the material with no wandering or movement.
- **Preferred spot drill** diameter allows a specific size spot drill to be selected on ALL holes. Size may be changed when needed.
- **Optimize spot drill** tool selection will attempt to use a tool that will satisfy All holes when there is a variety of sizes to be drilled. If attempting to chamfer holes when spot drilling one tool may not satisfy and an additional tool may be selected.
- **Optimize chamfer** tool selection will attempt to use a chamfer tool that will satisfy All holes when there is a variety of sizes to be chamfered.
- **Tool diameter tolerance** is a value not to exceed when selecting tools, for example, if the tolerance is set to .002 then if **FeatureCAM** doesn't find a tool plus or minus .002 of the needed size it will show a red flag next to the operation indicating it doesn't have a tool within the tolerance.
- **Drill % of ream/bore** determines the size of the drill to be used for a ream or bore operation.
- **Thread % for tap drill (cut)** determines the size of the drill to be used for the tapping operation to obtain a specific % of thread (example 77%)
- **Thread % for tap drill (rolled)** determines the size of the drill to be used for the tapping operation to obtain a specific % of thread (example 65%). Rolled threads drilled hole size is very important because the thread is formed and material is pushed down into the root of the tap and if the hole is to small it will break the tap and if too large there is not enough to form an accurate thread.
- **Tap type** lets you select the type of tapping you do most.
- **Tool %** of arc radius (also Toolpath corner % in milling tab) is set to 100 then a tool equal to the smallest corner radius is selected for a feature such as a pocket. With Toolpath corner% set to 100 the tool dwells in the corners as it changes direction. This can sometimes nick the part. To avoid this problem, set Toolpath corner % to a slightly smaller number, such as 98.

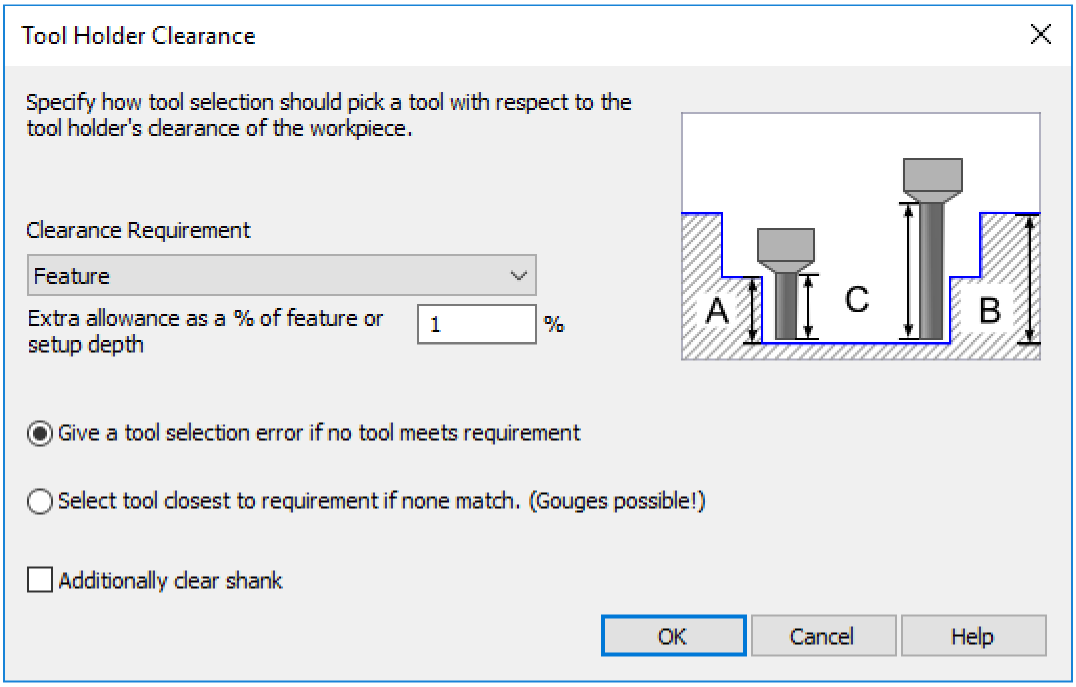
- **Multiple Roughing Tools** gives you the opportunity to allow **FeatureCAM** to select additional roughing tools when, for example, you have a pocket with small radius corners and **FeatureCAM** chooses to rough the entire pocket with the small cutter. When selecting Use multiple roughing tools from largest to smallest, stopping when the material is gone, you enter tools that you want **FeatureCAM** to select from when multiple cutters are necessary to make the operation efficient. Starting with the largest tool and adding smaller tools separated by a comma, **FeatureCAM** will start with the largest tool that will fit. It will also select a rougher that fits the corner so there will be the same amount of leave allowance in the corners as there is along the straights. You may also restrict the minimum size tool to use as a rougher. Remember though that if you are running small parts with small tools you may have a tool needed in your tool crib for a specific operation but it will never select it for roughing if it is smaller than the minimum tool diameter.



Holder Clearance

Use the **Tool Holder Clearance** dialog to specify an additional clearance for FeatureCAM's automatic tool selection to prevent tool holder gouges.





To specify a tool selection clearance:

*In the **Clearance Requirement** list, select the clearance you want between the tool holder and the part. Select from:*

- **None** — select this option to leave no additional clearance. Old part files select the same tools as before.
- **Feature** — Select this option to ensure the tool is long enough for the tool holder to clear the feature.
- **Setup** — Select this option to ensure the tool is long enough for the tool holder to clear the total depth below the setup.
- **Stock** — Select this option to ensure the tool is long enough for the tool holder to clear the total depth into the stock.

*Enter an **Extra allowance as a % of feature or setup depth** to leave extra clearance of the tool holder above the feature, setup, or stock*

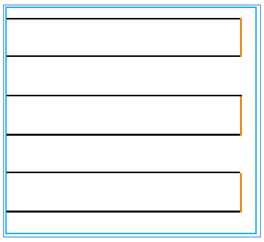
Select how tool selection is affected if no matching tool is found:

- **Give an error if no tool meets requirements** — FeatureCAM does not select a tool for the operation, so an error is shown during NC code generation. In the **Operation List**, a red exclamation point is displayed beside operations with no tool selected.
- **Select tool closest to requirements if none match** — this enables you to generate NC code, but it may result in tool holder gouges because a smaller tool may be used.

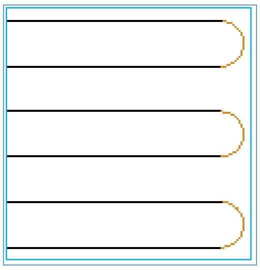
Surface Leadin

Stepover type is the transition move the tool makes radially to make the next pass (usually bi-directional) this is perpendicular to the tool path direction.

1 **Direct** is two 90 degree turns, stopping twice to change direction of 180 degrees.

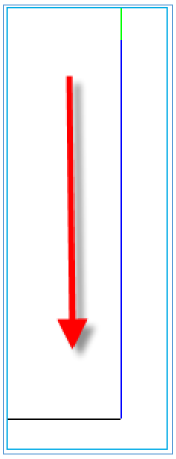


2 **Loop** is a smooth rounding transition which does not stop to change direction. Good hi-speed technique.

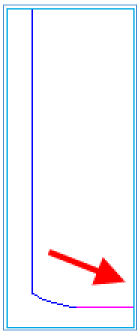


(**Lead-in/out parameters**)

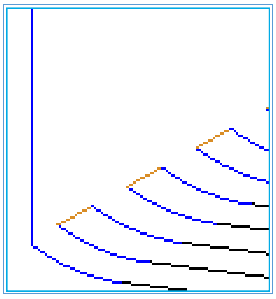
Use lead-in/out: *Never provides any leads, it uses plunge clearance and plunges to depth.*



On all plunges/retracts *or* ***on first plunge/last retrac****t applies the lead type, line or arc, to every plunge and retract or the first and last only.*



***On all stepover, plunges & retracts** applies the lead type to every transition move the tool makes.*



Surface normal applies the leads normal to the surface whether horizontal or vertical.

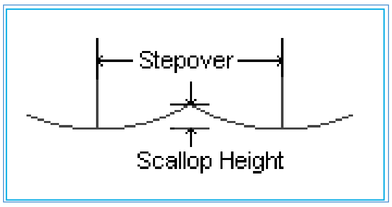
Use arc ramp-in/out allows you to choose the ramp diameter and the ramp in/out angle.

Use linear lead-in/out allows you to choose the lead-in/out angle and the lead-in/out length.

Surface Mill Tab

- **Tolerance (Rough .005) (Finish .001)** sets how close the milling will be to the mathematically ideal surface. This does not guarantee that your feature is machined to this tolerance in all locations if the tool you select is incapable of cutting within that tolerance in constrained areas. If your part shows a faceted appearance, set the tolerance to a lower value.
- **Scallop height stepover** is the height between passes of a Ball End Mill for isoline milling, projection milling finishing passes and Z-level finishing. This distance is measured along the surface and represents the maximum cusp height between neighbouring passes as shown here. The scallop height and stepover default values may also be entered but can be entered or changed in individual features also.
- **Parallel angle** refers to parallel tool path strategies. The value of 0 causes the tool to travel parallel to the X or to the Y axis.
- **Tool diameter** allows a default tool diameter to be selected for Surface Milling
- **Tool end radius** provides the opportunity to select the default for the cutter end, Ball, Flat or Bull Nose and also the diameter.
- **Finish allowance** is the amount of material left by the rougher for the finisher to remove.

- **Check allowance** is how far away you wish the tool to stay away from selected surfaces or faces.
- **(Slope limitation angles)**
- **Horizontal only** cuts horizontal surfaces up to a desired slope angle.
- **Vertical only** cuts vertical surfaces above a desired slope angle.
- **Swarf axial tolerance** is for a relatively rare number of geometries where the tool axis can waver slightly as it positions accurately on the surfaces to be machined. This can be due to small but significant changes in the geometry as the tool moves from one position to another. To allow a degree of latitude for the command, this tolerance can be set to a value larger than the machining tolerance to allow the tool axis to be stabilized as it moves across this geometrically varying region. As a result excess material may be left on the surface involved but the load on the tool may be reduced.



Edges contain four choices as to how you want the tool to react when it comes to the edge of a surface or face at the stock.

Automatic, depending on the toolpath strategy being used, does much like don't roll over the edges at all and therefore calculates a boundary but if a different boundary is used such as a curve, FeatureCAM won't calculate for both.

Don't roll over the edge at all cuts 100 percent cleanly to the edge but does not roll over the edge.

Cut top edge: Just roll over the top edge allows the ball only to roll over the edge.

Cut to bottom: Roll over the top edge and cut to bottom of stock/part basically turning off all boundaries and letting the tool cut until it runs out of part or stock.

Thread Mill

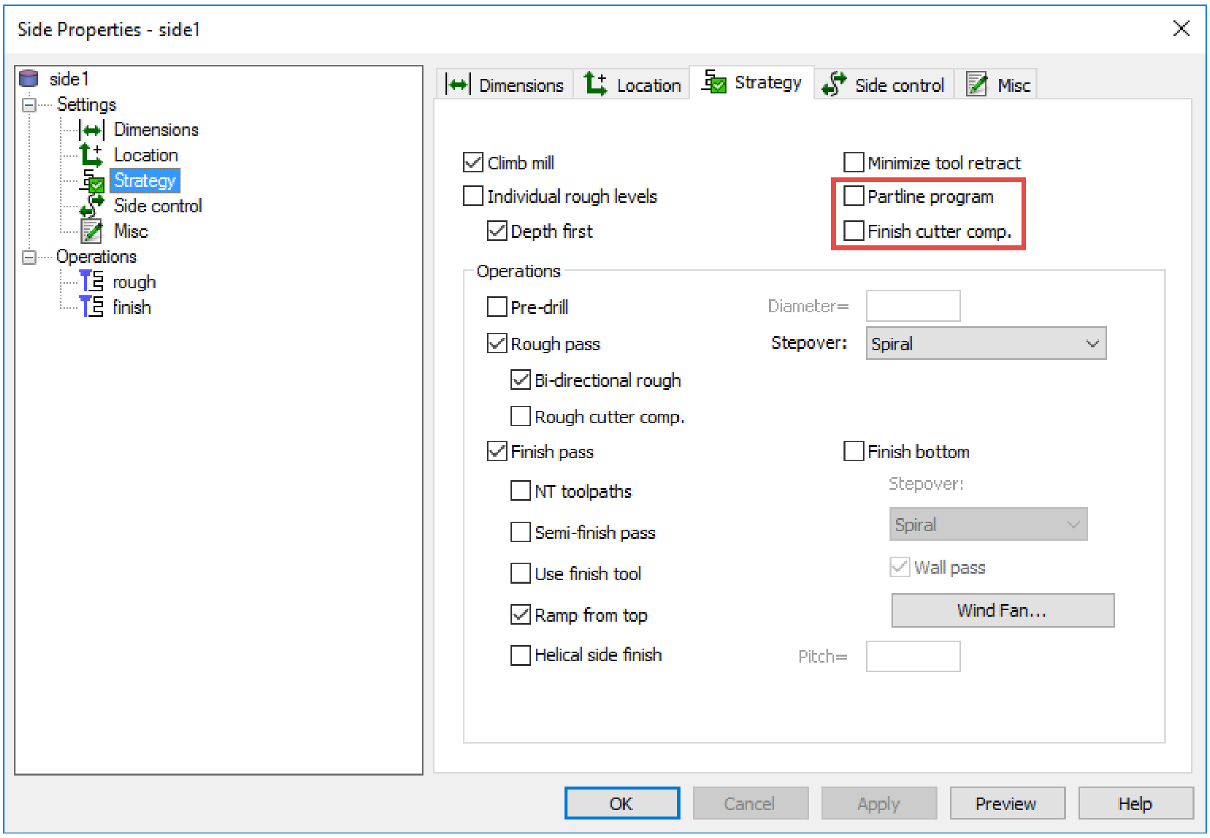
- **(Wind Fan)**
- Wind fan is the same as a milling feature, it can plunge and retract and feed on and off in the same location plus the angle of approach and the ramping radius.
- **Linear ramping** may also be applied using ramp distance, diameter and angle offset.
- **Feed direction** can be either Negative Z or Positive Z.
- **Feed overrides** for ramp in and ramp out.
- **Passes** for Rough and Finish with Rough Stepover, Finish allowance and Spring passes.
- **Starts** may be multiple if desired.
- **Start angle** can be entered if necessary.

- **Tooth overlap** attribute controls the amount that one revolution of a multi-thread tool will overlap the previous revolution. It is an integer that represents the number of threads. We recommend that you overlap at least one thread.
- **Tooth outside** is the number of teeth that will be above (if feeding in negative Z) or below (if feeding in positive Z) the thread mill feature for the first pass.
- **Taper approx. angle** is for tapered threads. The toolpath is increasing in diameter as well as moving in Z. These moves are approximated with 3D arcs. The Taper approx. angle is the angle around the thread that will be approximated by a single arc. A 360 must be evenly divisible by the Taper approx. angle. For example, if set to 90, a single revolution of the tool will be broken into 4 arcs.
- **Feed to depth** override may be set to a desired feed rate.
- **Linear Approx**. attribute is associated with Helical ramping and Thread mill features. If Linear approx. is selected, then the arc moves are approximated by linear moves. Helix linear approx. tolerance controls how accurate the approximation is relative to the theoretical spiral. Set this tolerance to a smaller number to more accurately approximate the spiral.
- **Cutter comp** to apply G40, G41 and G42 to the NC code so the control can adjust size with tool offset.
- **Part line program** puts the center of the tool on the finish curve and the NC code reads to print dimensions and all offsetting is done by the control.
- **Through** for thread milling is set if there is no material on the bottom of the thread. If Through is not set, the toolpaths will be generated to ensure that the tool will not cut past the end of the thread.
- Use **Finish Tool** allows a separate tool to be used as a finisher.
- **Plunge to center** plunges the tool on the center of the hole to assure clearance.

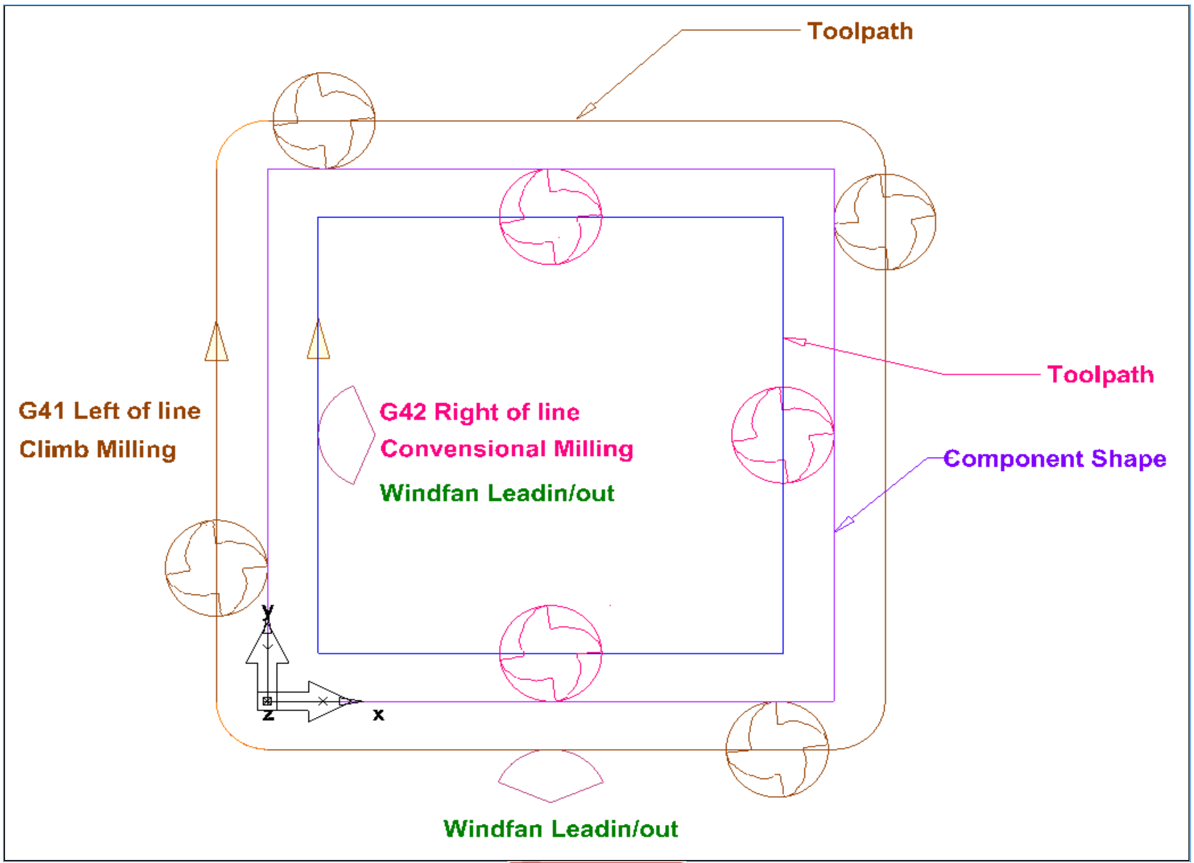
Cutter Compensation explained

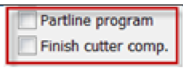
No Cutter Compensation (Information Only)

- The following examples will explain the three main differences when using Cutter compensation. To make this example easy to follow. We have created a **100mm x 100mm x 25mm** Block. We will be using an **18mm** diameter cutter. When we look at the code output it will be obvious which strategy is used by looking at the **cnc code.**
- The Post processor used is a **Fanuc Robodrill.cnc** to view the code output.
- The first example will have **No** Cutter compensation active. (see image)



- This code output **allows** for the diameter of the cutter in the toolpath with **No** cutter compensation **(G41/G42)**



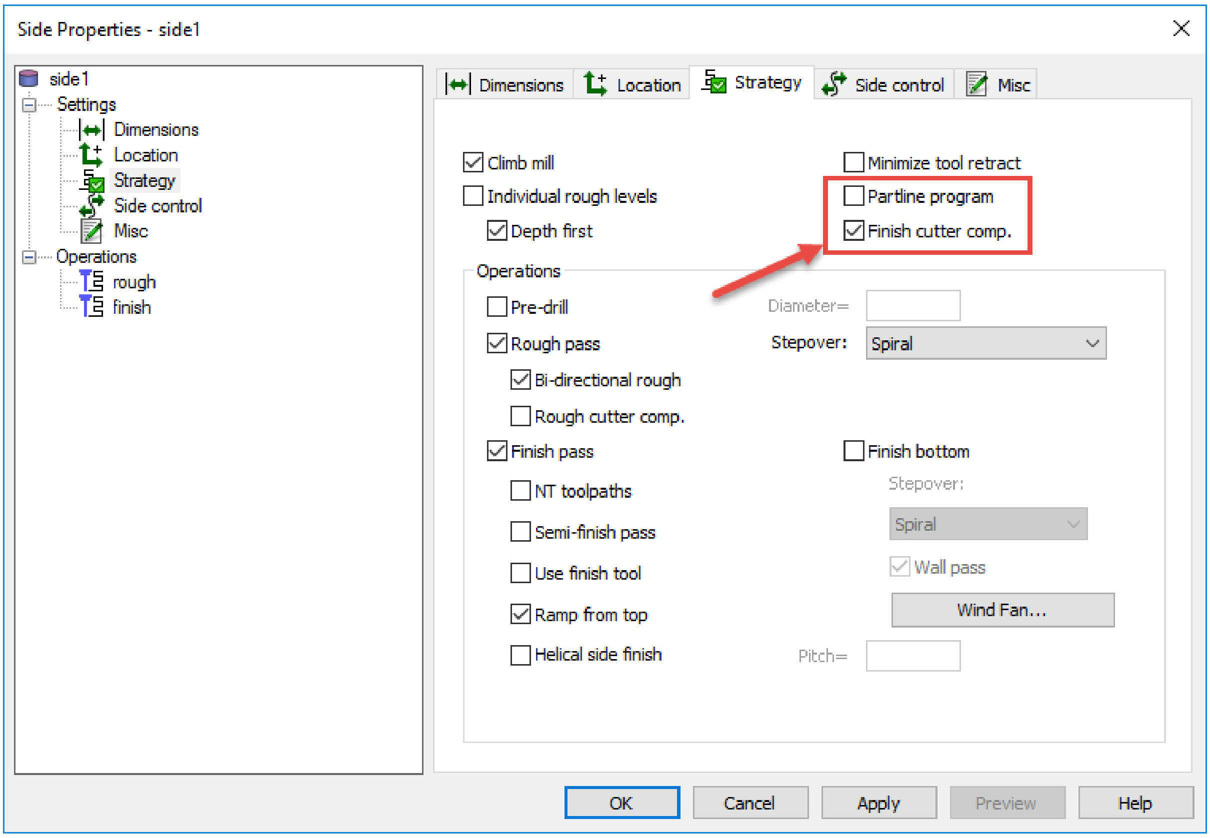


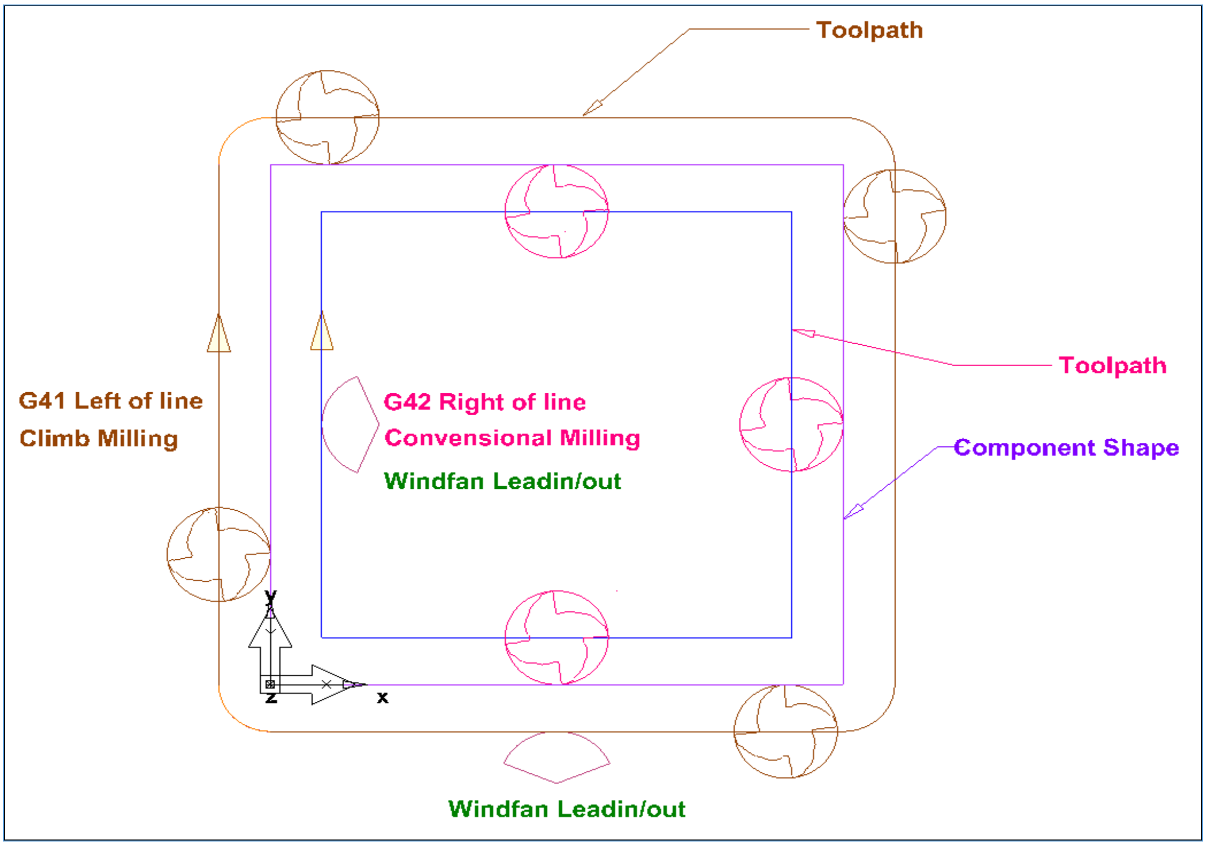
***Code output Millimeters**. This code output **allows** for the diameter of the cutter in the toolpath. Code is output with **No** cutter compensation.*

```
%
O0001
( FINISH  SIDE1 )
N25 G0 G40 G49 G80 G90
N30 T1 M6
N35 G54 X50.0 Y-19.0
N40 M03 S5390
N45 G43 H1 Z26.0 M8
N50 Z3.0
N55 G1 Z-25.0 F1164.24
N60 X59.239 Y-15.173 F2328.48
```

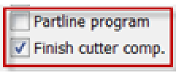
```
N65 G3 X50.0 Y-9.0 I-9.239 J-3.827 F1225.52
N70 G1 X0. F2328.48
N75 G2 X-9.0 Y0. I0. J9.0 F3492.72
N80 G1 Y100.0 F2328.48
N85 G2 X0. Y109.0 I9.0 J0. F3492.72
N90 G1 X100.0 F2328.48
N95 G2 X109.0 Y100.0 I0. J-9.0 F3492.72
N100 G1 Y0. F2328.48
N105 G2 X100.0 Y-9.0 I-9.0 J0. F3492.72
N110 G1 X50.0 F2328.48
N115 G3 X40.761 Y-15.173 I0. J-10.0 F1225.52
N120 G1 X50.0 Y-19.0 F2328.48
N125 G0 Z26.0
N130 X19.0 Y50.0
N135 Z3.0
N140 G1 Z-25.0 F1164.24
N145 X15.173 Y40.761 F2328.48
N150 G2 X9.0 Y50.0 I3.827 J9.239 F1225.52
N155 G1 Y91.0 F2328.48
N160 X91.0
N165 Y9.0
N170 X9.0
N175 Y50.0
N180 G2 X15.173 Y59.239 I10.0 J0. F1225.52
N185 G1 X19.0 Y50.0 F2328.48
N190 G0 Z26.0
N195 M5
N200 M9
N205 G91 G28 Z0.
N210 G90 G49
N215 M30
```

Finish Cutter Comp (Information Only)





Code output Millimeters**. This code output **allows** for the diameter of the cutter in the toolpath. Code is output **with** cutter compensation **(G41/G42)



%

O0001

(FINISH SIDE1)

N25 G0 G40 G49 G80 G90

N30 T1 M6

N35 G54 X50.0 Y-19.0

N40 M03 S5390

N45 G43 H1 Z26.0 M8

N50 Z3.0

N55 G1 Z-25.0 F1164.24

N60 **G41** D1 X59.239 Y-15.173 F2328.48

N65 G3 X50.0 Y-9.0 I-9.239 J-3.827 F1225.52

N70 G1 X0. F2328.48

N75 G2 X-9.0 Y0. I0. J9.0 F3492.72

N80 G1 Y100.0 F2328.48

N85 G2 X0. Y109.0 I9.0 J0. F3492.72

N90 G1 X100.0 F2328.48

N95 G2 X109.0 Y100.0 I0. J-9.0 F3492.72

N100 G1 Y0. F2328.48

N105 G2 X100.0 Y-9.0 I-9.0 J0. F3492.72

N110 G1 X50.0 F2328.48

N115 G3 X40.761 Y-15.173 I0. J-10.0 F1225.52

N120 G1 G40 X50.0 Y-19.0 F2328.48

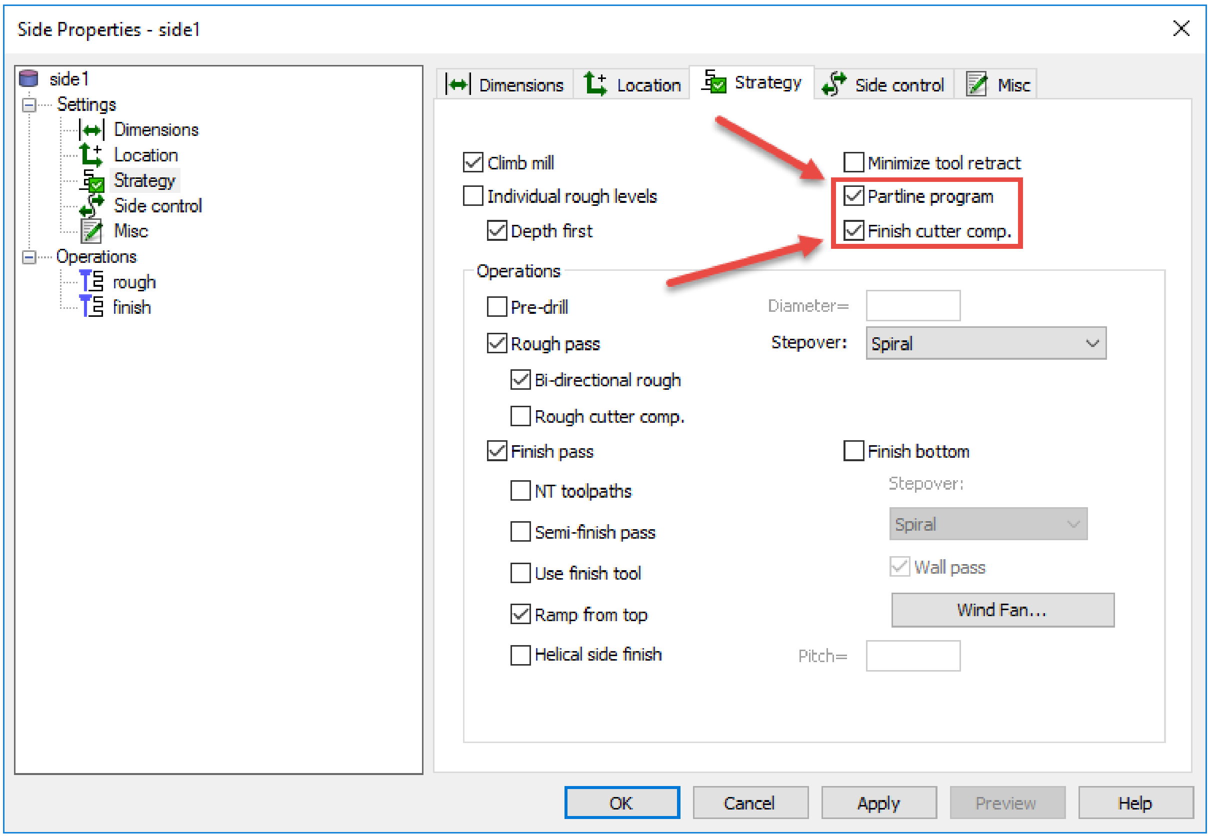
N125 G0 Z26.0

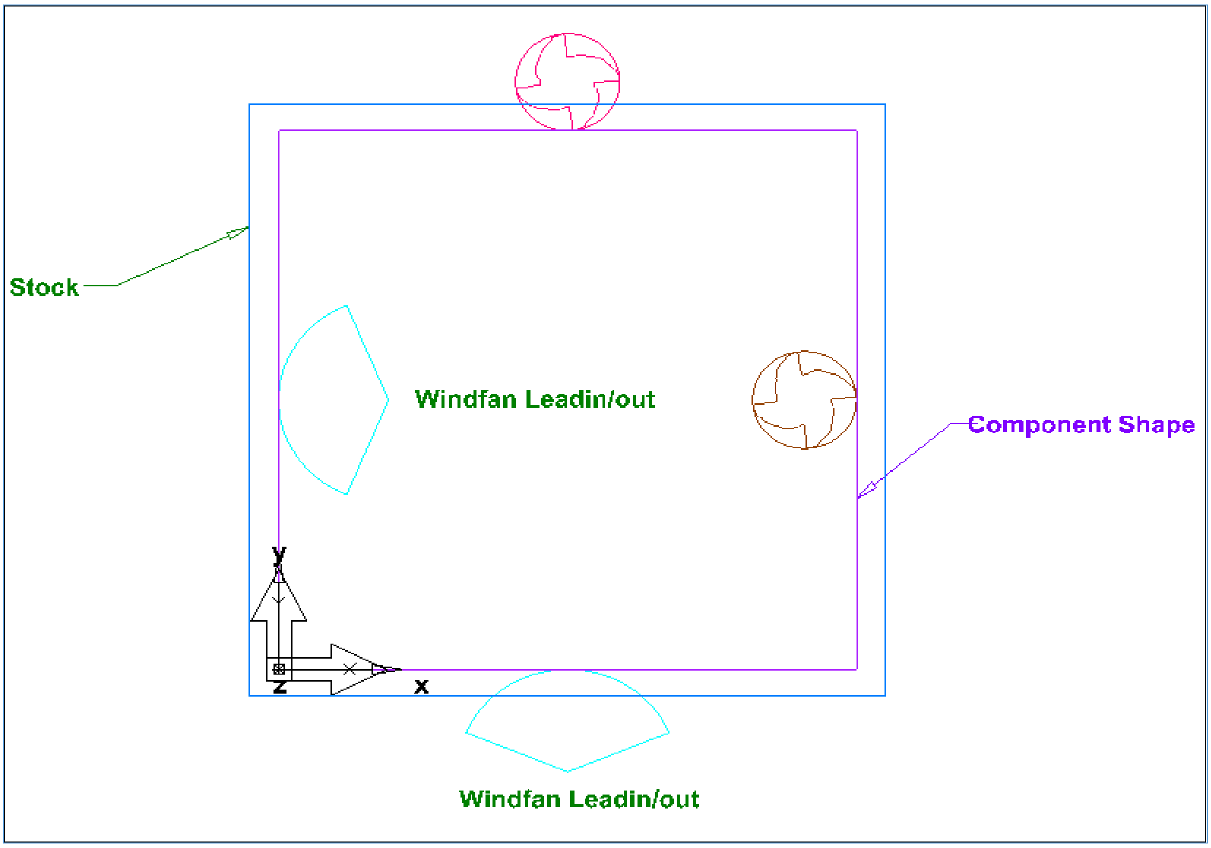
N130 X19.0 Y50.0

N135 Z3.0

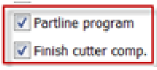
```
N140 G1 Z-25.0 F1164.24

N145 G42 D1 X15.173 Y40.761 F2328.48

N150 G2 X9.0 Y50.0 I3.827 J9.239 F1225.52

N155 G1 Y91.0 F2328.48

N160 X91.0

N165 Y9.0

N170 X9.0

N175 Y50.0

N180 G2 X15.173 Y59.239 I10.0 J0. F1225.52

N185 G1 G40 X19.0 Y50.0 F2328.48

N190 G0 Z26.0

N195 M5

N200 M9

N205 G91 G28 Z0.

N210 G90 G49

N215 M30

%
```

Partline program with Finish Cutter Comp





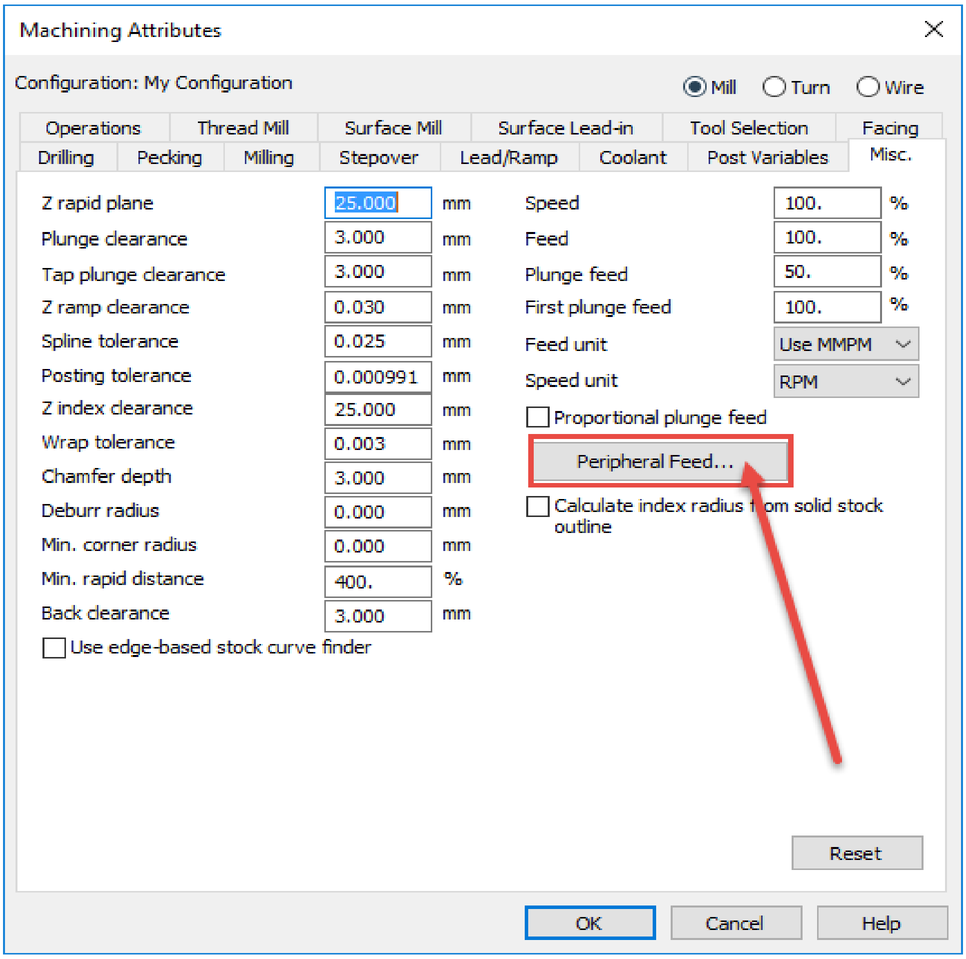
Code output Millimeters**. This code output **Does Not** allow for the diameter of the cutter in the toolpath The code is output **with** cutter compensation **(G41/G42)

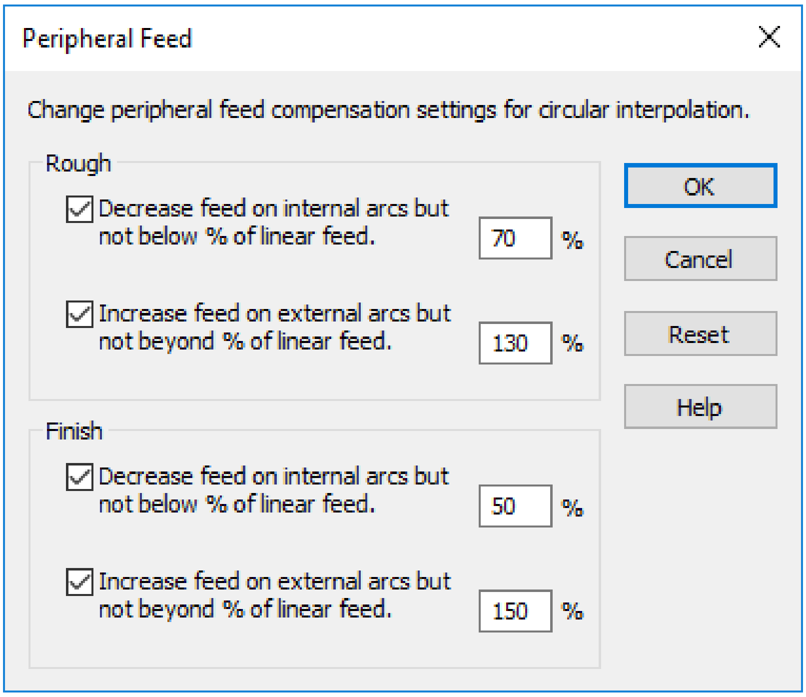



```
%
( FINISH  SIDE1 )
N25 G0 G40 G49 G80 G90
N30 T1 M6
N35 G54 X50.0 Y-19.0
N40 M03 S5390
N45 G43 H1 Z26.0 M8
N50 Z3.0
N55 G1 Z-25.0 F1164.24
N60 G41 D1 X67.554 Y-11.729 F2328.48
N65 G3 X50.0 Y0. I-17.554 J-7.271 F1225.52
N70 G1 X0. F2328.48
N75 Y100.0
N80 X100.0
N85 Y0.
N90 X50.0
N95 G3 X32.446 Y-11.729 I0. J-19.0 F1225.52
N100 G1 G40 X50.0 Y-19.0 F2328.48
N105 G0 Z26.0
N110 X19.0 Y50.0
N115 Z3.0
N120 G1 Z-25.0 F1164.24
N125 G42 D1 X11.729 Y32.446 F2328.48
N130 G2 X0. Y50.0 I7.271 J17.554 F1225.52
N135 G1 Y100.0 F2328.48
N140 X100.0
N145 Y0.
N150 X0.
N155 Y50.0
N160 G2 X11.729 Y67.554 I19.0 J0. F1225.52
N165 G1 G40 X19.0 Y50.0 F2328.48
N170 G0 Z26.0
N175 M5
N180 M9
N185 G91 G28 Z0.
N190 G90 G49
N195 M30
```

Speed Reduction on Corners.

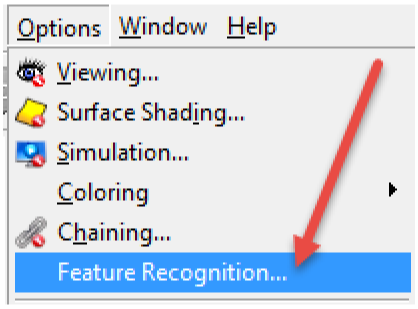
- Automatic speed reduction in corners is standard on all toolpaths. To access the settings for this option go to Machining **Configurations/Misc** then **Peripheral Feed.**



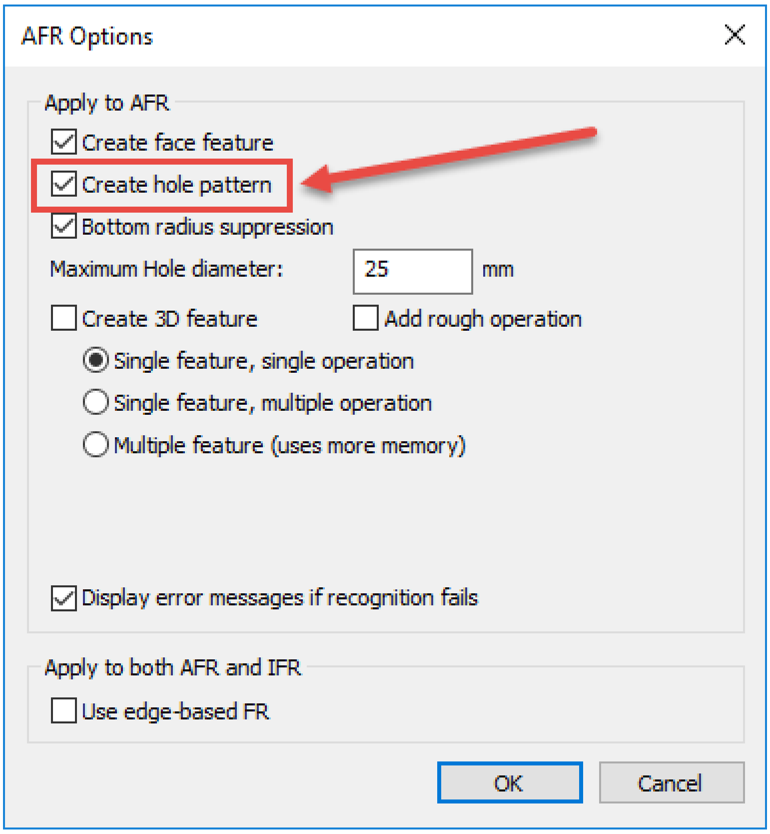


Options/Feature Recognition

- Considerations when using **Feature Recognition**.



Please select the following options. ***Create hole as pattern****. All similar holes will be grouped into a Pattern. This allows you to change one feature depth and all holes in that pattern are then changed.*

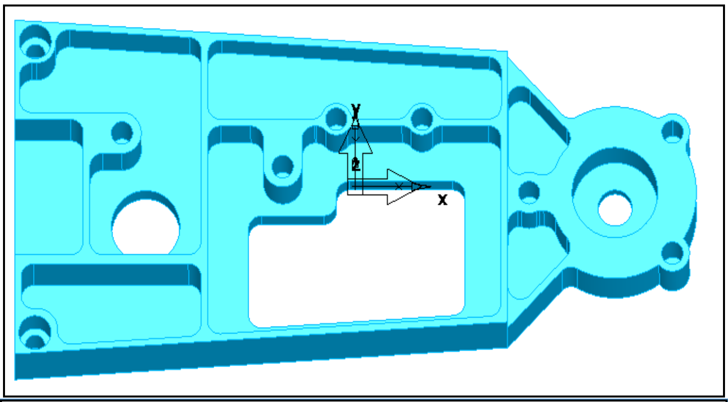


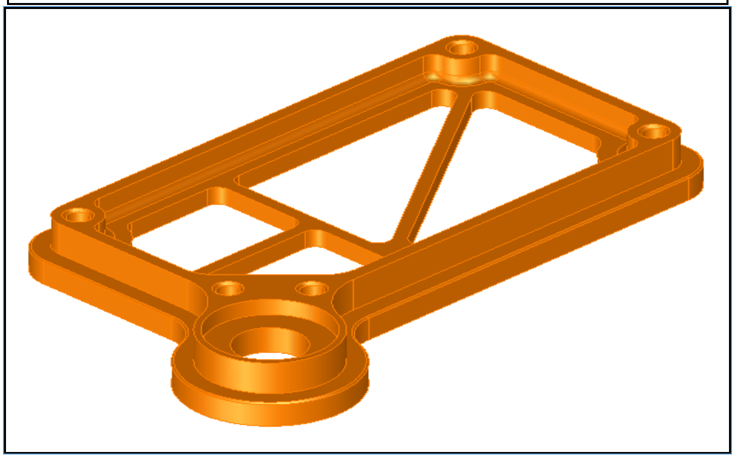
Contents

Feature Recognition for Milling

Introduction

*This Module will provide an introduction to **FeatureCAM's Feature Recognition** capabilities. When this module is complete you will know how to perform **Automatic Feature Recognition (AFR),** and **Interactive Feature Recognition (IFR)** and **Feature Re-recognition**.*





Important things to consider when using AFR

*The way **AFR** works is that it examines the model and looks for any flats on the part in Z axis. **AFR** creates features by dividing the model into horizontal slices at these flats and automatically determines side control.*

*Any features (excluding holes) remaining will be cut using surface milling (if checked in the **AFR Options**) Notice that **AFR** creates a set of features that will completely cut the solid, but it may create more features than you might create if you have modeled the features yourself.*

*The advantage of this method is that a part may be programmed quickly as it requires minimal user input. This method could be useful for quickly programming prototypes, or to help users new to programming in **FeatureCAM**. It can also be used by experienced users to create curves or to help get a better idea on how to cut the part*

The disadvantages of this method are that the most efficient toolpath may not be created and the features created may not be exactly the way the programmer is used to cutting the part.

*One last thing to remember is that a solid is required to run **AFR**. The reason for this is so that **AFR** is able to properly recognize side control and will cut on the correct side of the part.*

What is a Feature?

- Features are the intelligent core entities that a machinist would use to make a certain shape in the material they are cutting. Shapes such as a **Hole, Boss, Slot**, etc. Features are used to make toolpaths in **FeatureCAM** to create these shapes. Features may consist of several operations to create the final result or shape. These may consist of several operations including; **Spot Drill, Centre Drill, Drill, Pre-Drill, Tap, Roughing, Semi-Finish and Finish Passes. FeatureCAM** takes, for example, a tapped hole and combines the operations like the **centre Drill, Drill and Tap** and automatically selects the necessary **Tooling**, **Feeds**, **Speeds**, etc.

What is Feature Recognition? Why is it necessary?

***Feature Recognition** has the ability to extract manufacturing **Features** and associated operations from an existing **Solid** or **Surface** model. **Geometry**, **Curves**, and dimensional input are not required as this information is automatically extracted out of the design model.*

***Feature Recognition** is beneficial to production for several reasons, the first being that the **CAD** data available is just a collection of surfaces and faces. The **CAD** data (surfaces and faces) needs to be interpreted by the **CAM** software so useful features and subsequent operations may be Collected from this data to manufacture a part. For example **Holes** in a solid are just cylinders and a **Pocket** in a solid is just a collection of faces.*

*The second reason **Feature Recognition** is beneficial to production is that in modern manufacturing the majority of the design software is used for part design is solid or surface based. This valuable information contained in the model file has already been entered by the **CAD** engineer, why should the machinist re-enter information that is already available in the model? A review of the manual process is listed below, which can show the process to be quite lengthy:*

1. Create Geometry
2. Create Curves
3. Select Feature Type
4. Select Curve
5. Specify Depth
6. Enter manufacturing information
7. Obtain wireframe visual display
8. Obtain Feeds and Speeds, tools, toolpaths and NC code

*The third reason **Feature RECOGNITION** is of benefit is that re-entering the model data by the machinist is error prone as the model information needs to be completely re-entered.*

How does Feature Recognition Work?

- There are two different types of Feature Recognition **(FR) Automatic Feature Recognition (AFR)** and **Interactive Feature Recognition (IFR).** Both tools search the CAD data for **(Pockets, Sides, Holes, Surfaces and Faces)** and match them to **FeatureCAM's** features so manufacturing operations may be generated. These two techniques use different algorithms for identifying features. Descriptions of the algorithms used will be described below.

Requirements for FR

1. In order to use **Feature Recognition** a Solid Model is required. There are a wide variety of different formats that are supported:
2. **Igs, Acis, Parasolid, SolidWorks, Inventor, SolidEdge.**
3. Optional **(Extra Cost)** import plugins: **Catia V4 & V5, Step, Unigraphics** and **Creo.**
4. There are several ways that the model can be imported directly into **FeatureCAM**.
5. Click on **File> Import**.
6. Locate the file, **left click** on the file to import, Alternatively **hold the left mouse button down then drag and drop** the file into **FeatureCAM's** Graphics Window from any folder where the model resides **(except a zip folder).**

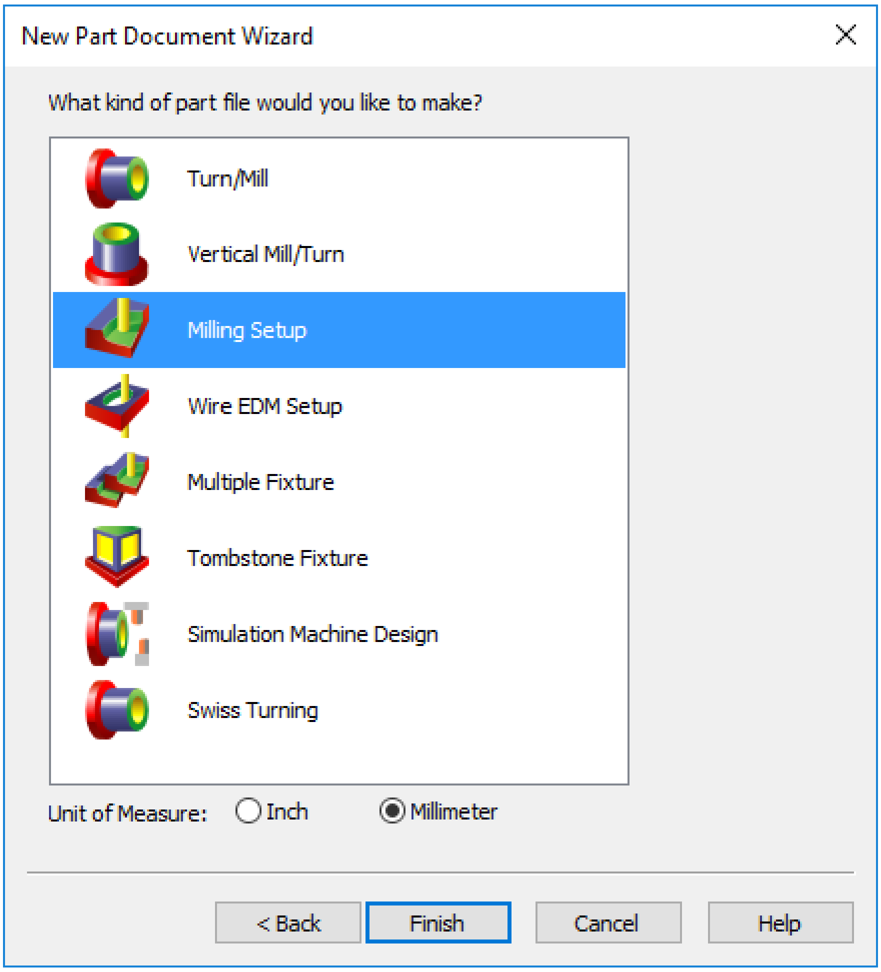
Importing a file one Setup. (Class Exercise)

*When you import a file, **FeatureCAM** steps you through its **Import wizard**. The **Import wizard** pops up automatically and assists you in importing the file into **FeatureCAM**. This wizard helps to:*

1. Import the file into **FeatureCAM**.
2. Sizes the stock.
3. Orientates the stock allowing for additional material if required.
4. Positions the part program zero point.
5. Sets up a milling part for indexing.
6. For some solid formats, it even helps you recognize and suppress part features.

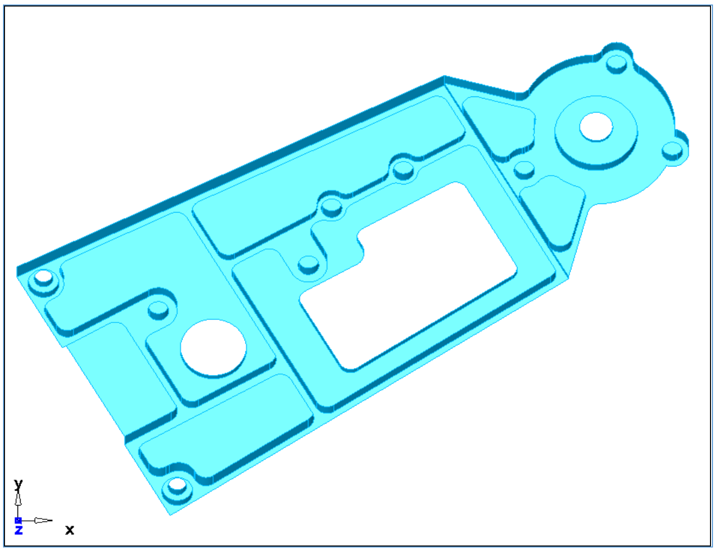
AFR Example #1 (Class Exercise)

1. Open a **New Part Document**.

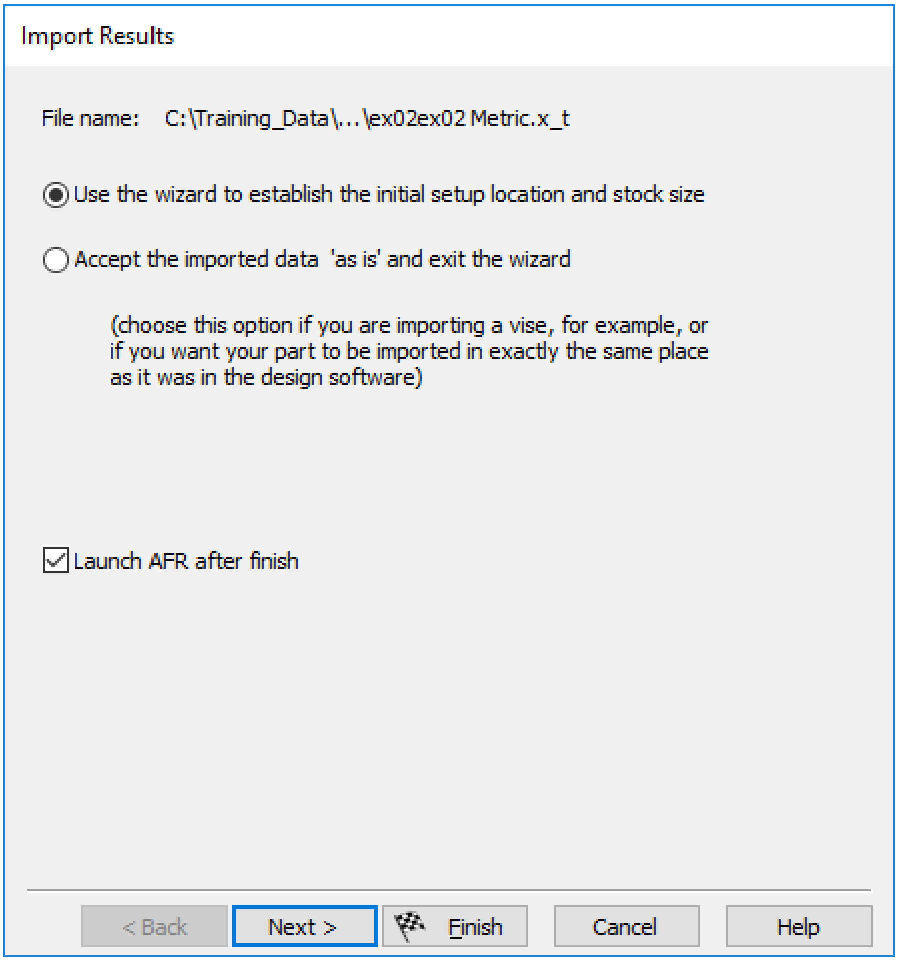


2. Select **Milling Setup** and select your Unit of Measurement in our case this is **Millimetre** then press **OK**. Select **Cancel** for **Stock** defaults.
3. **Click** on **File>Import**. Under Files of type at the bottom of the screen select **Parasolid (*.x_t, *.x_b)**. The file is called **ex02ex02 Metric. X_t.**

4 Find the file **in C:\Training_Data\FeatureCAM Course Data 2017\Milling Files to import.**



Import Results Wizard



1 Import Results - **Use the wizard to establish the initial setup location and stock size**, this will be checked automatically then check the "**Launch AFR after finish"**.

2 Select **Next**.

*To aid in aligning the model, when the model is first brought into the Graphics Window, right click and select **View Top** or **Isometric** and Shade the model so you can see how it is positioned in the window.*

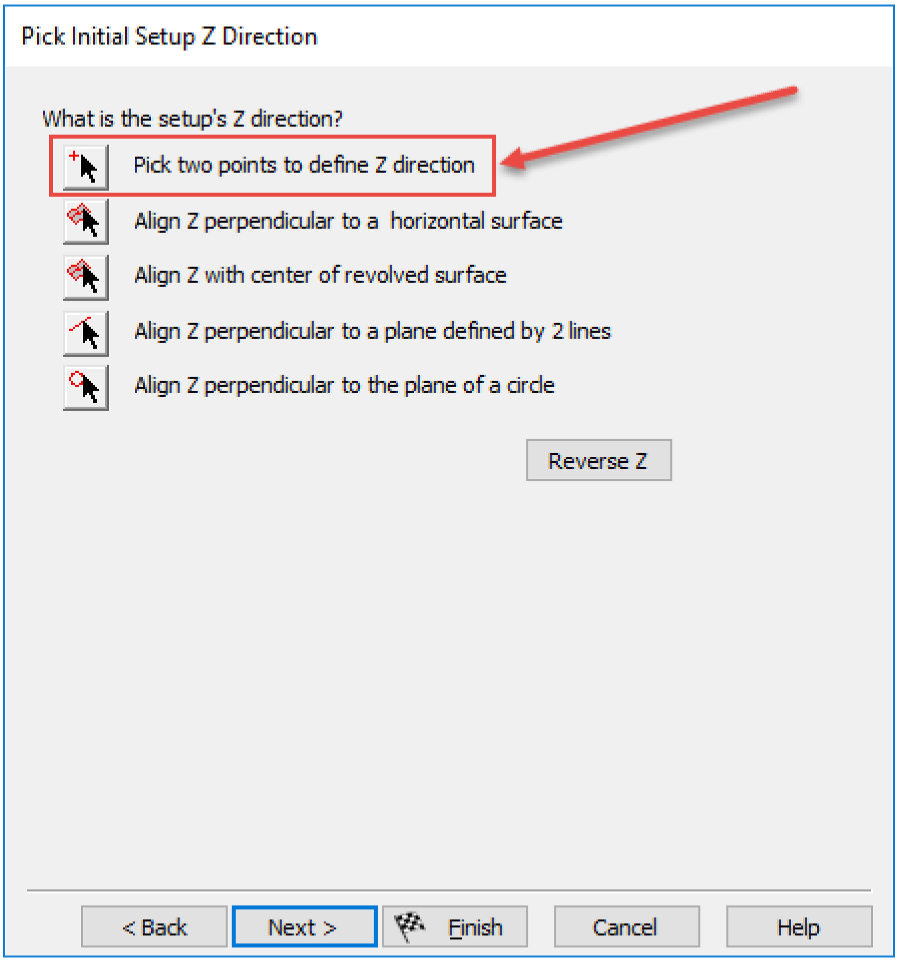
*(**Shade**-upper tool bar-button with two little **Red Barrels**)*

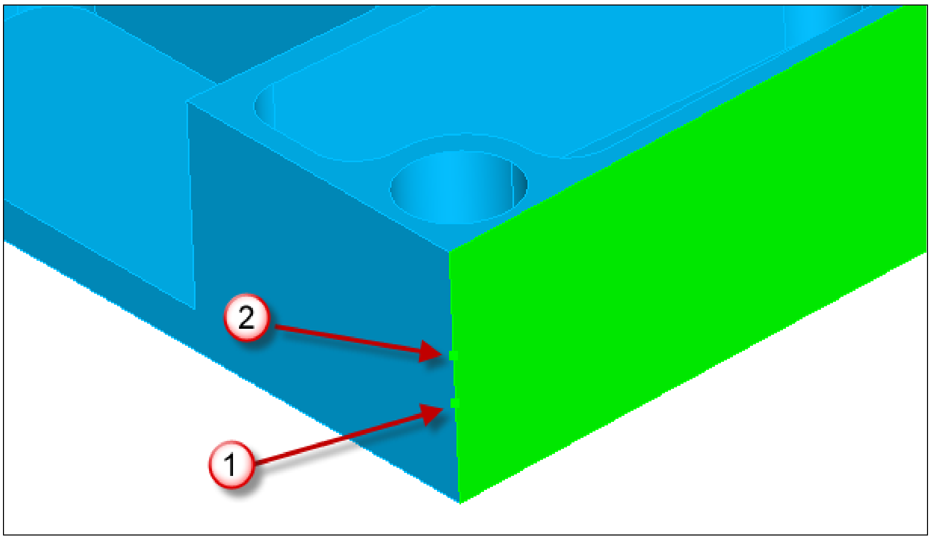
*Because the CAD model may have been saved in a position that is not aligned with your Z and X axis, **FeatureCAM** in the **Pick Initial setup Z Direction form** provides five tools to assist in aligning the model in the **Z** and **X** directions, which in turn aligns to the machine Spindle, Table Axis.*

*This model is not **aligned correctly** as you can see from the image on the previous page**.** The model has been rotated **30 degrees in Z** and **10 degrees in X**.*

3 Select **Next.**

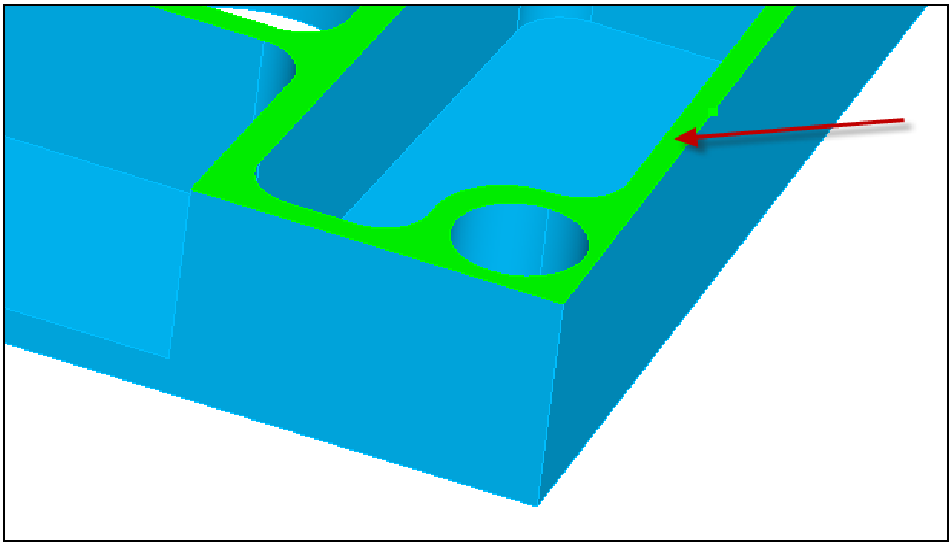
4 The model needs to be aligned. Select - **Pick two points to define Z direction**.





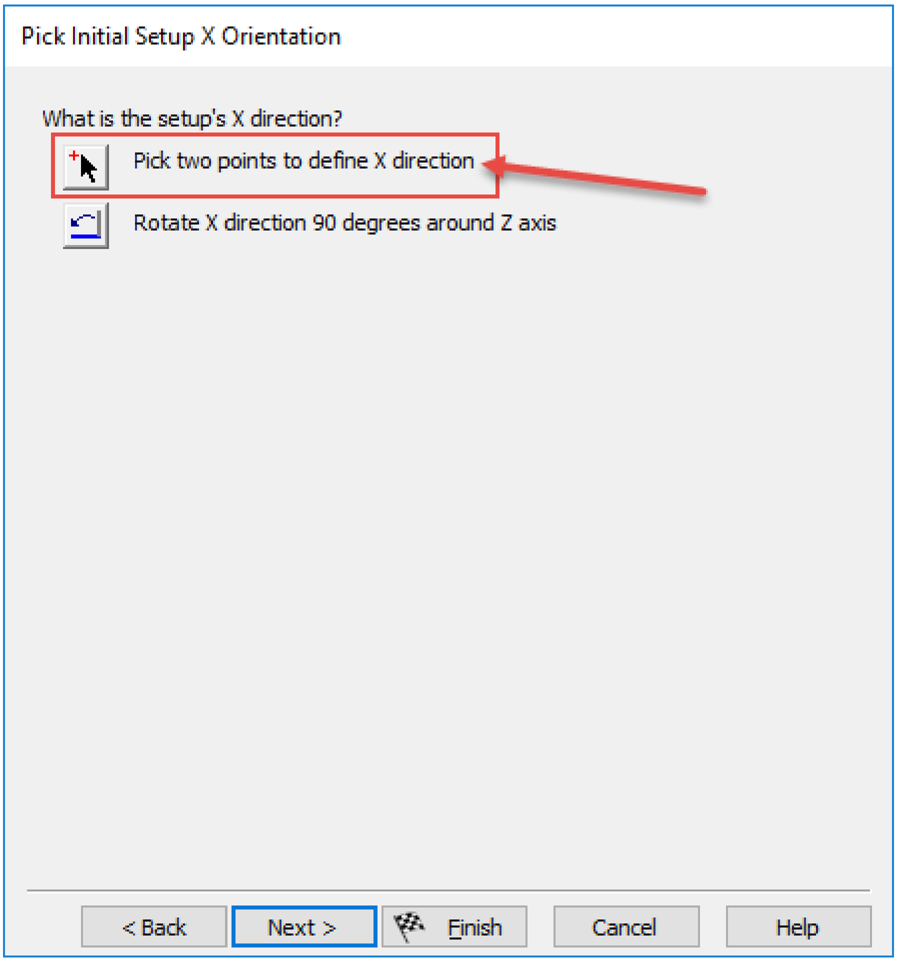
5 Click twice on a vertical surface edge, starting low 1 then the second click at 2 above will point to the spindle.

6 Or you could use **Align Z perpendicular to a horizontal surface**. Click on any known face that is flat and horizontal that you wish to be perpendicular to the spindle. See example below. **Use this method to align the Z Axis.**



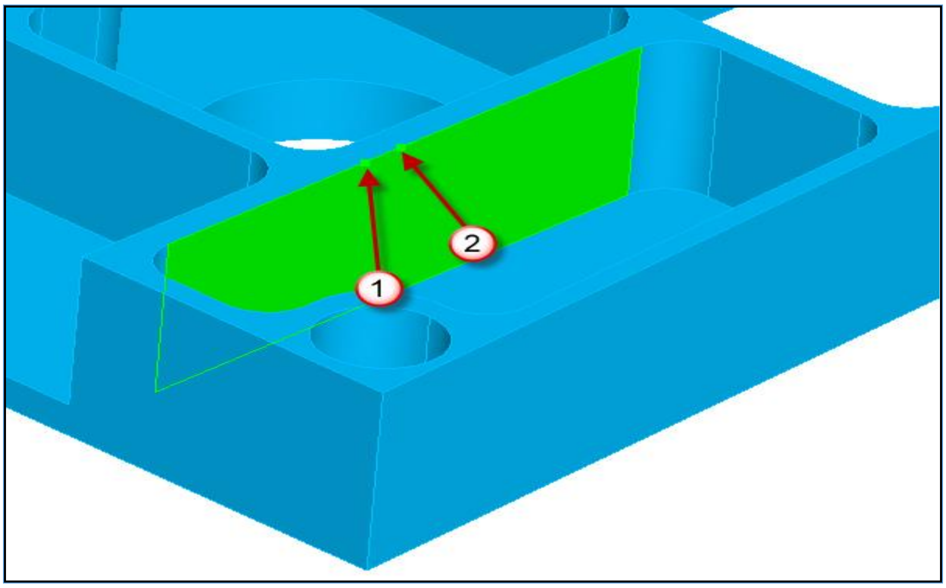
*Other **options** are available for example **Align with centre of a revolved Surface** such as a hole by selecting the arrow and clicking on the round surface. Can be used even when geometry alone is imported by clicking on the defined geometry.*

7 Select **Next**



8 **Pick Initial Setup X Orientation** offers assistance to align the setups X direction.

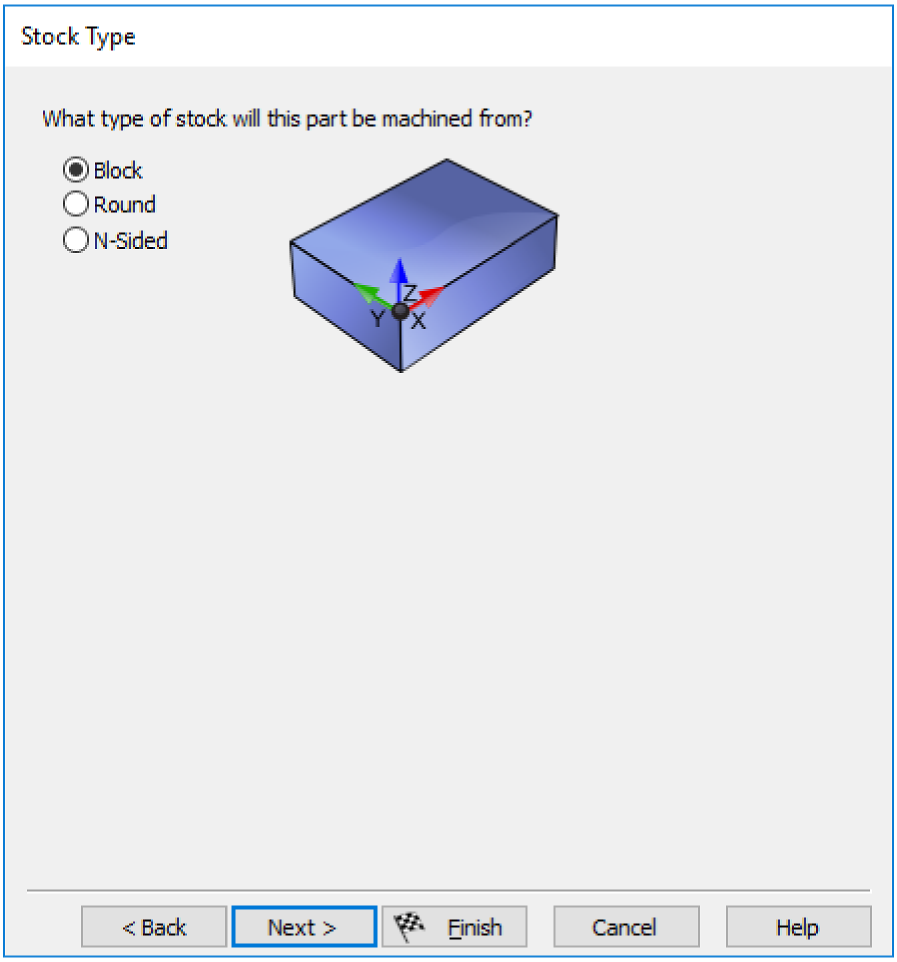
9 Please select **Pick two points to define X direction.** This works the same as the Z, except the edge you click on will be parallel to the table X axis.



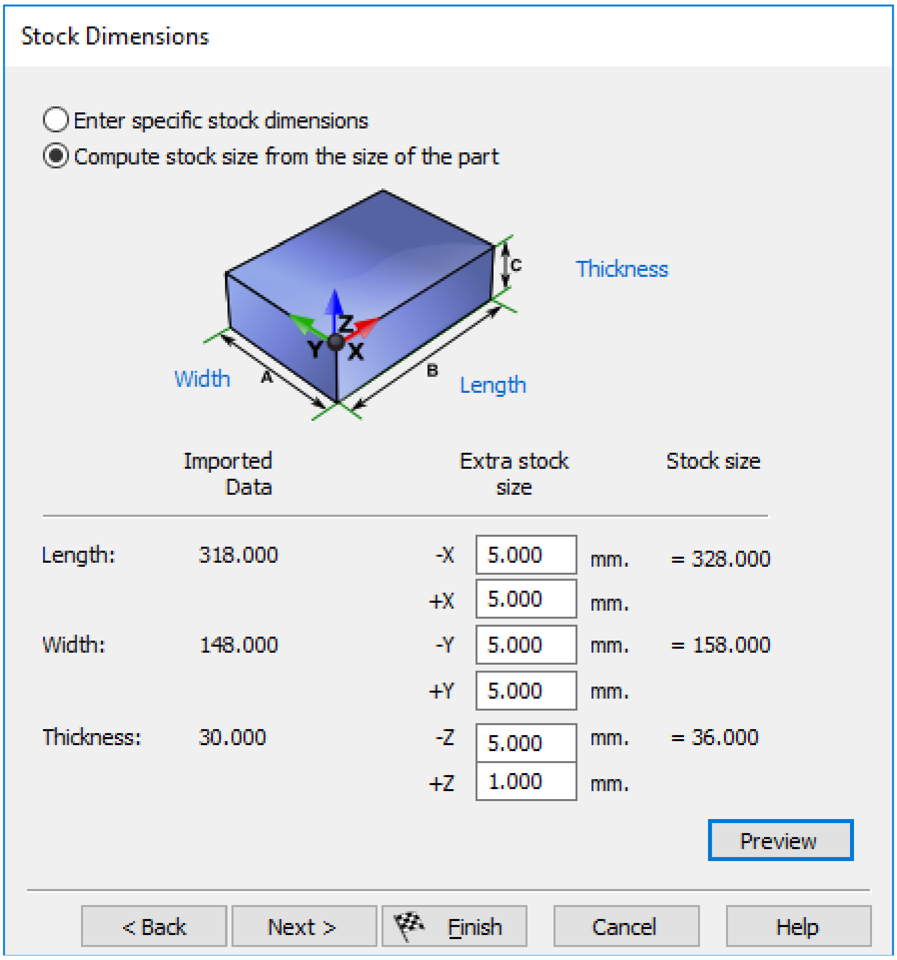
10 More **Orientation** options are available for example**; you can rotate the X direction 90 degrees around Z axis.** This will rotate the model 90 degrees for each click; three times will rotate **270 degrees**.

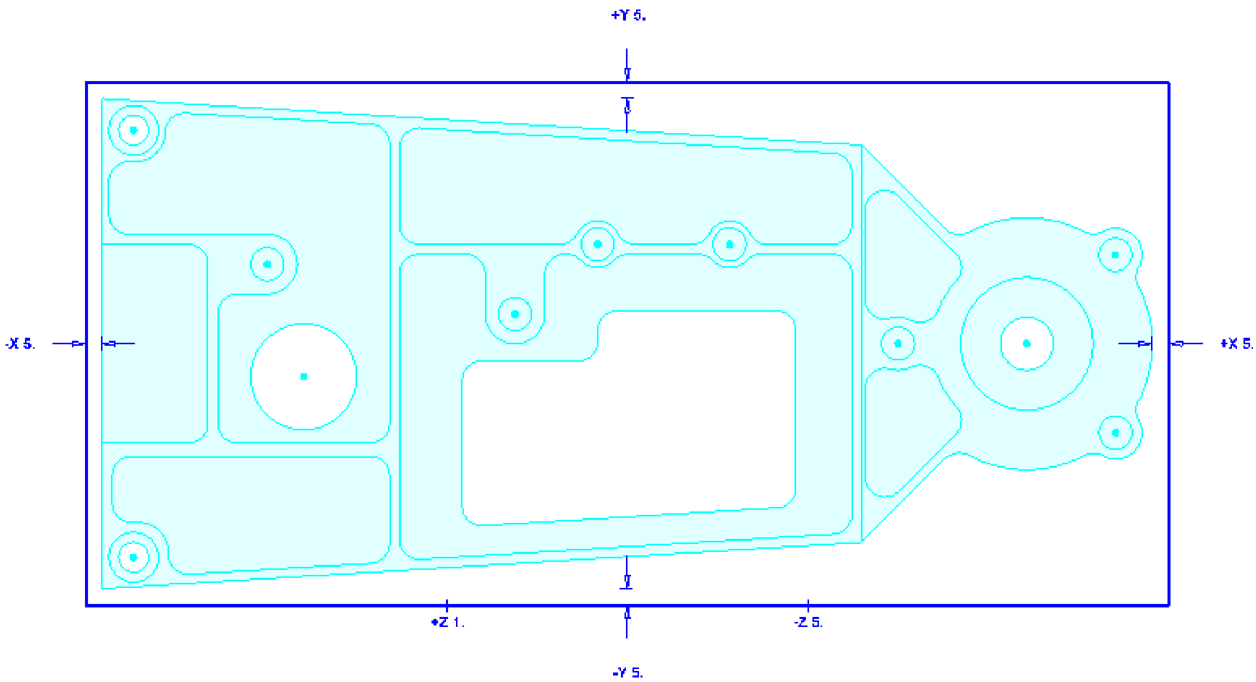
11 Select **Next**.

12 **Stock type** gives you the choice of **Block**, **Round** or **N-Sided** such as hexagonal stock. Select **Block** and select **Next**.



13 On the **Stock Dimensions** page check the radio button**, compute stock size from the size of the part**. Enter dimensions as shown below.

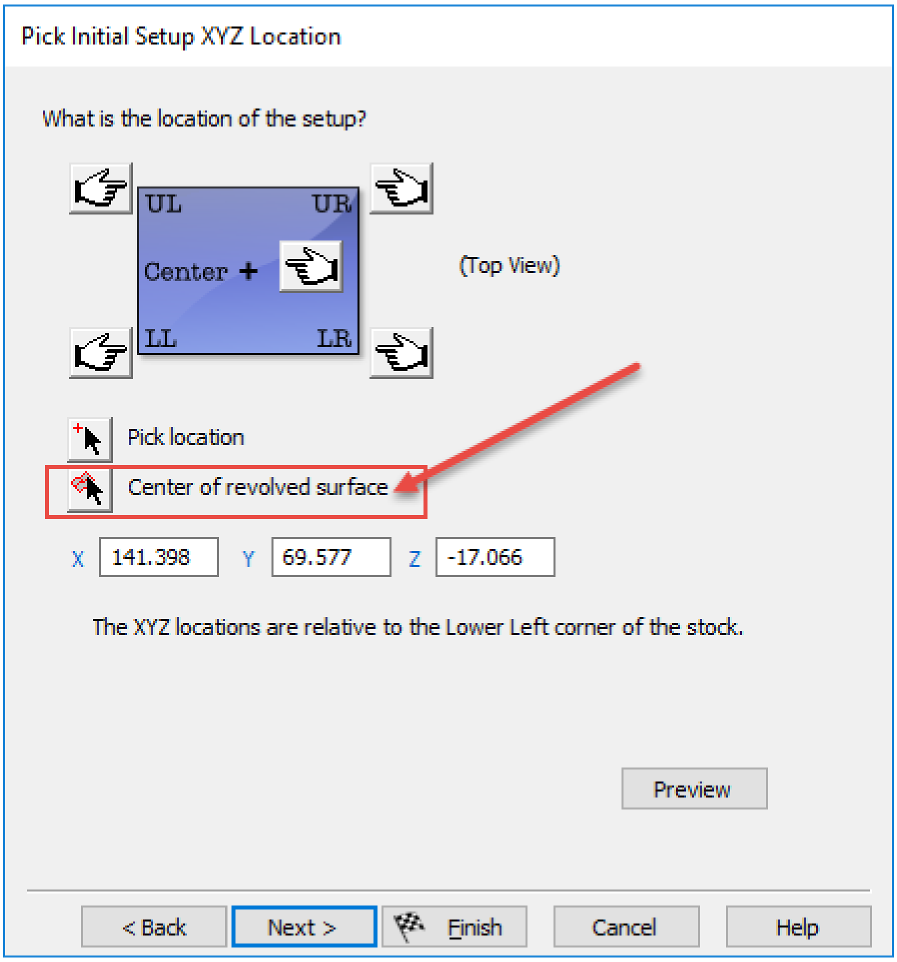


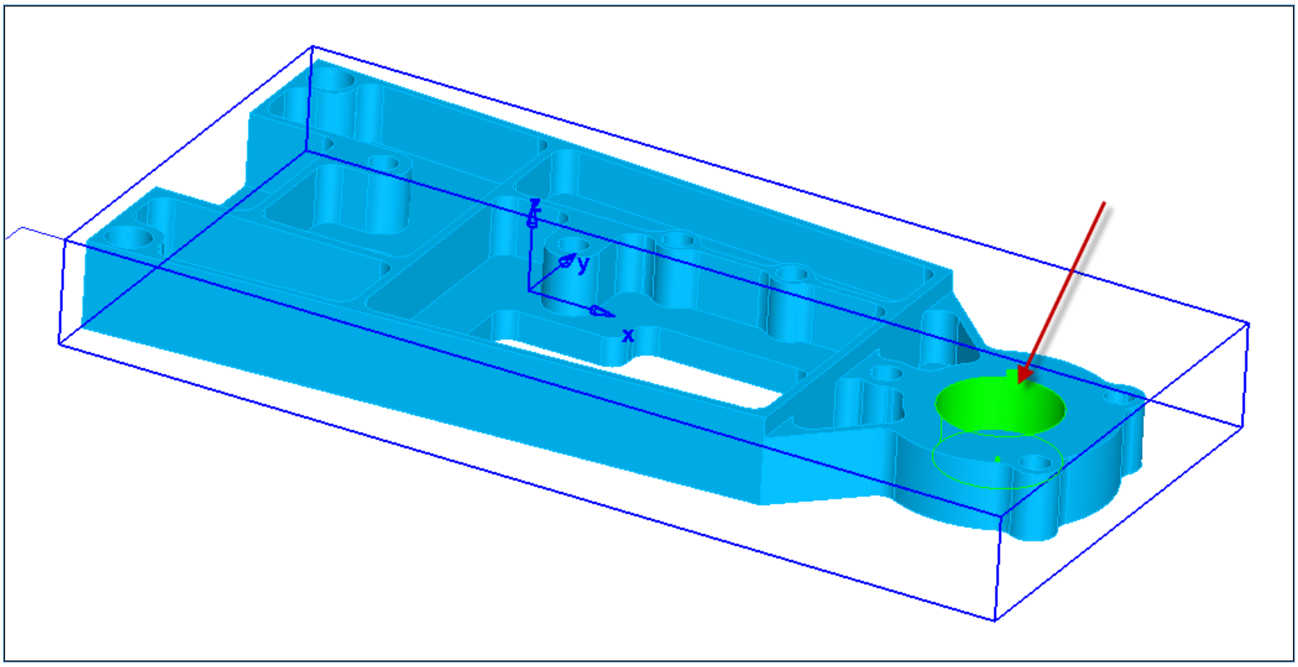


*The Values above are **Metric** we have allowed **5mm** for each side edge. We also have **1mm** for **Z+ Stock** and **5mm** for the **–Z** stock to hold in the Vice.*

14 Select **Next**.

15 Pick **initial setup XYZ Location** on the next form to appear. Please select Centre of revolved surface edge.

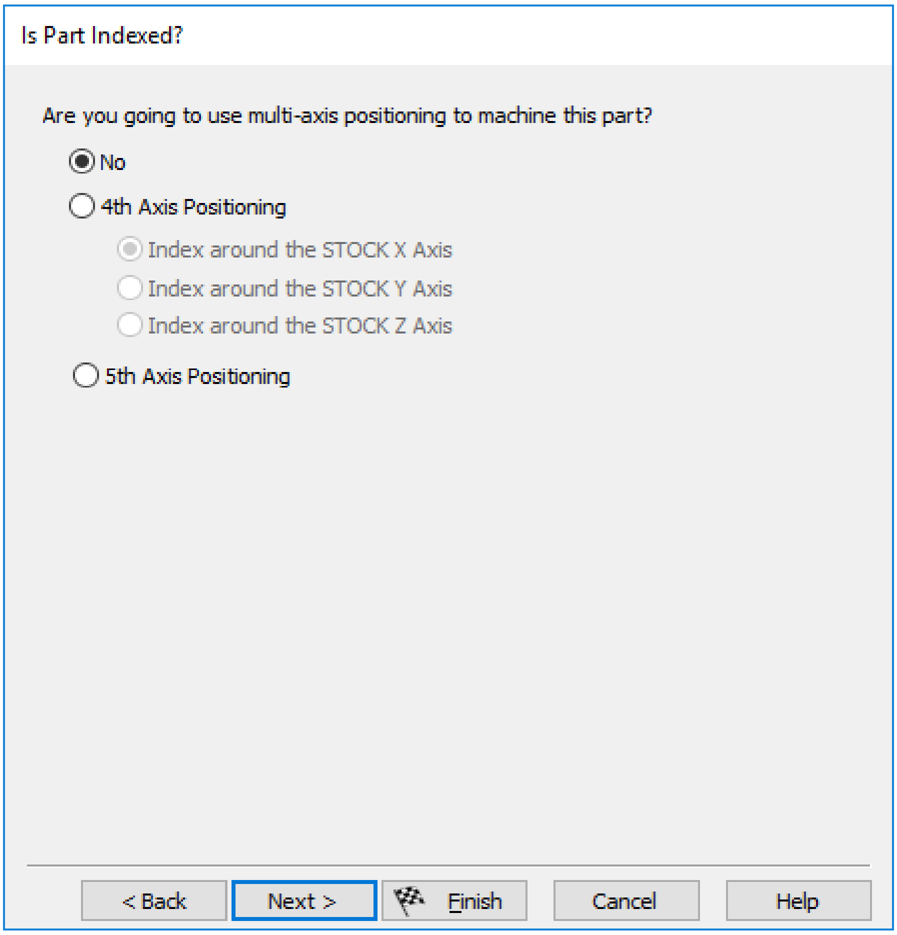




16 Change the **Z** value from **-17.066 mm** to **-1mm**.

17 Select **Next**.

18 On the **Is Part Indexed?** Page choose **No**. This means it is only a **3 Axis** part. Select **Finish**.

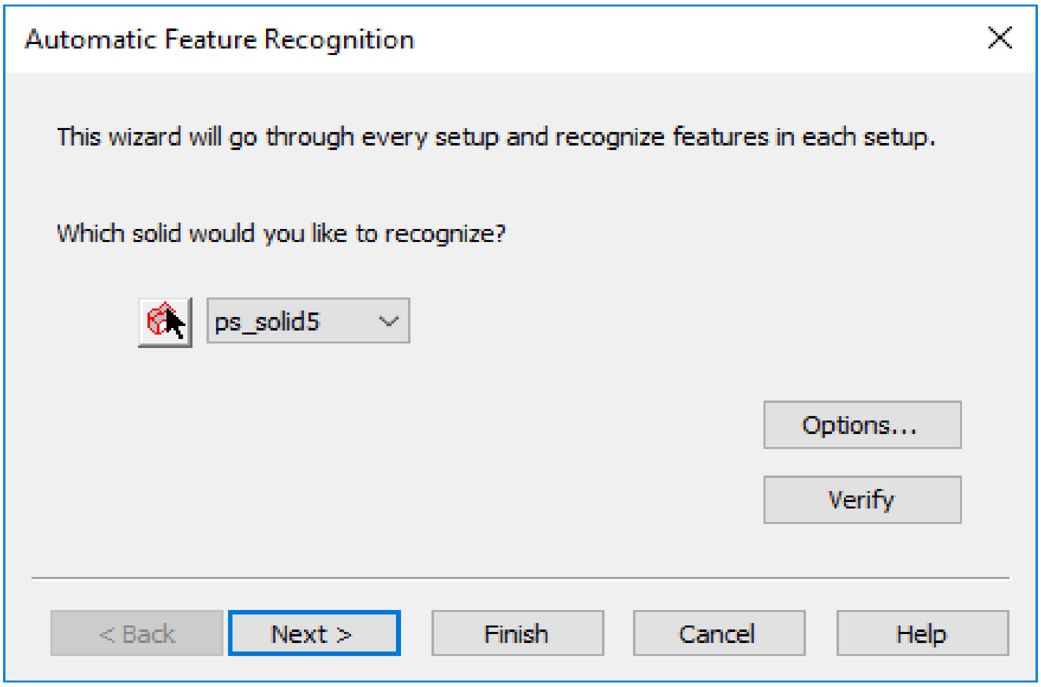


Because the ***"Launch AFR after Finish"*** *box was checked on the initial Import form, the* ***Automatic Feature Recognition*** *pops up. It then identifies the solid just imported.*

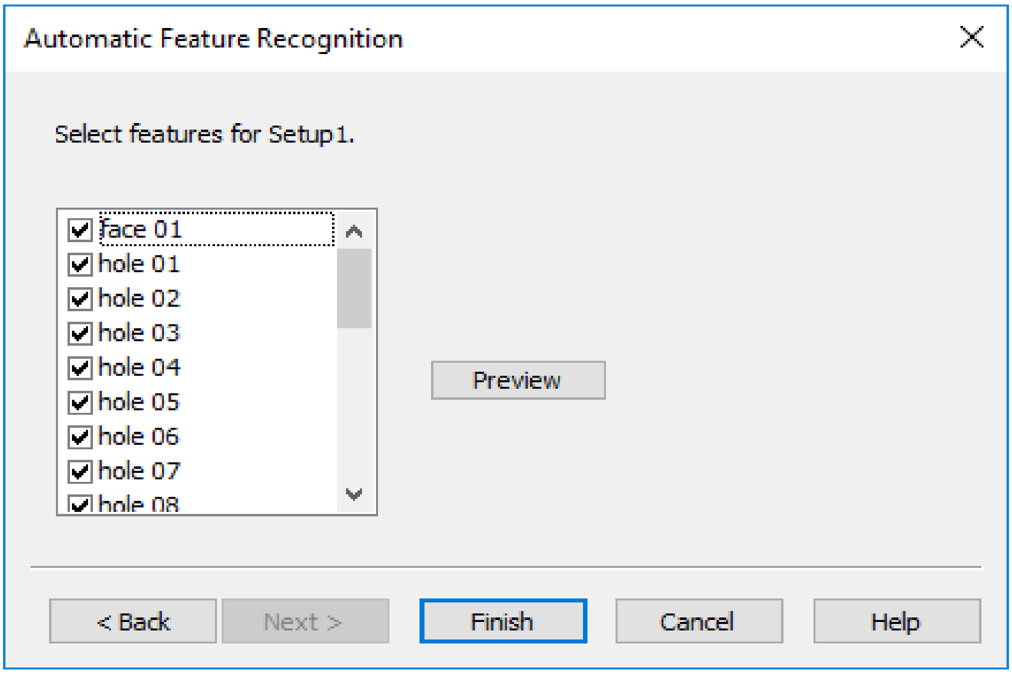
Please note we are only using ***One Setup*** *for this job as there are no features to be machined on the opposite side.*

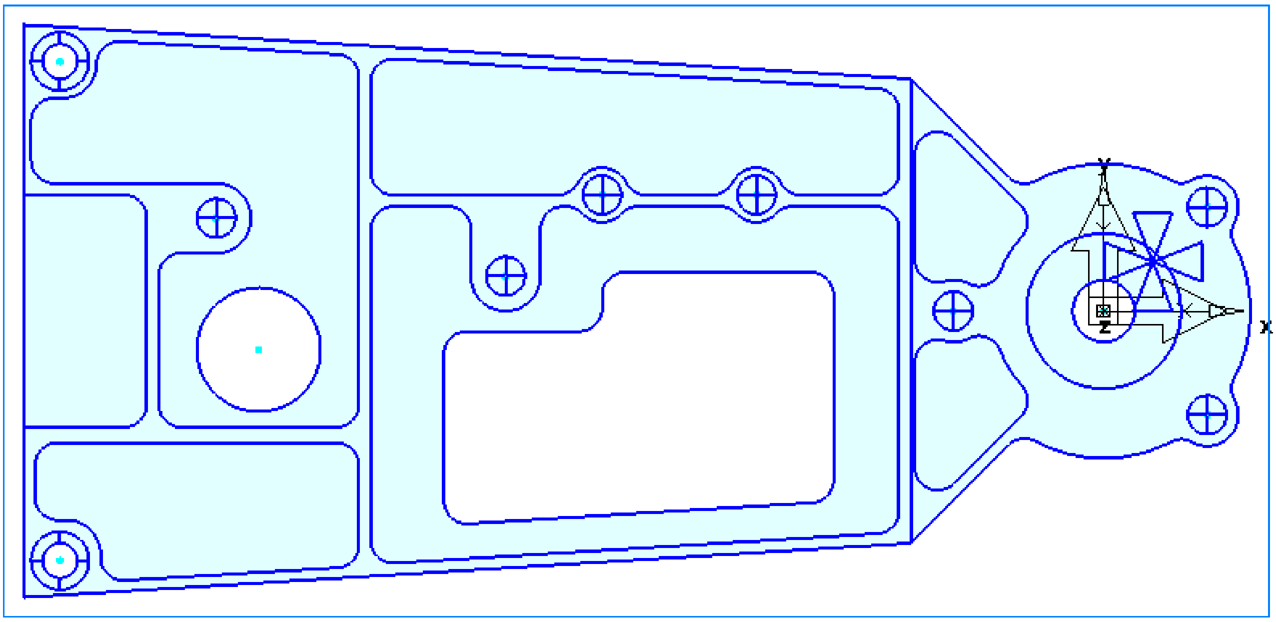
19 The **Feature Recognition Wizard** will go through and recognise features in **Setup1**.

20 The **AFR Wizard** will guide you through the next process.



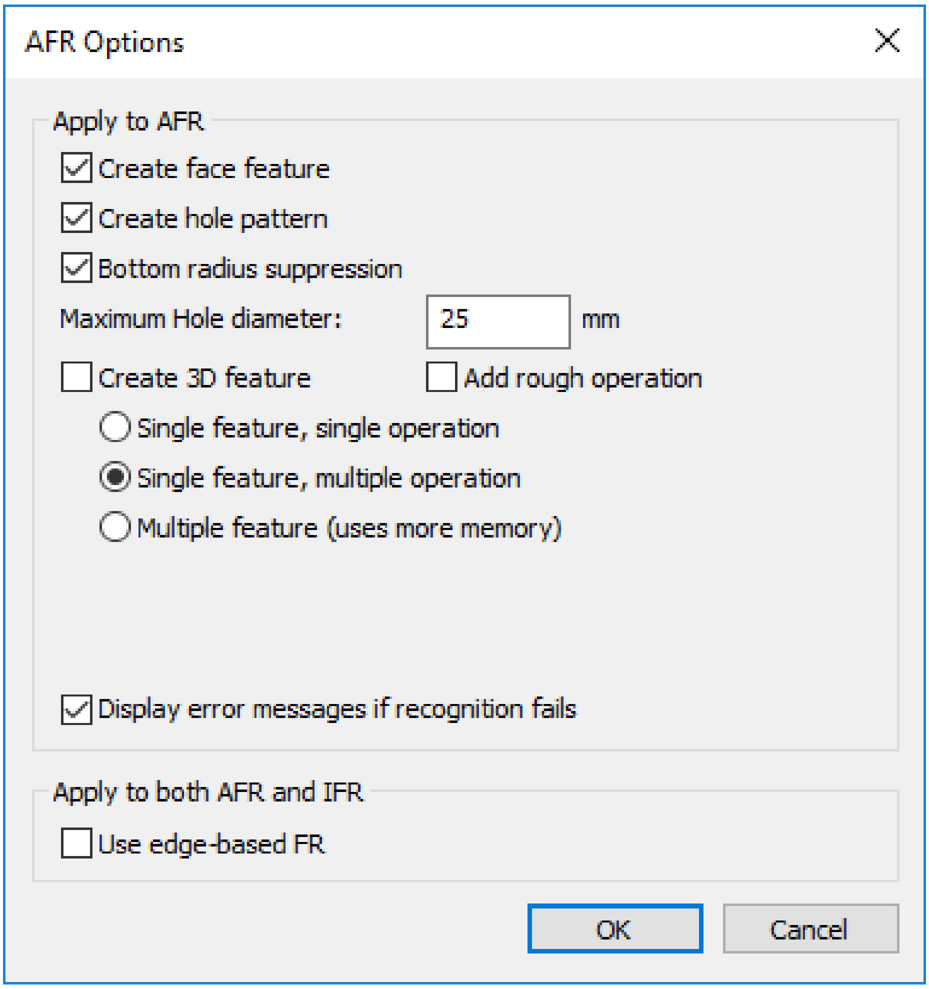
21 Select **Next**.





22 Select **Finish**.

*Select **Options>Feature Recognition**. These options set how AFR will be performed.*



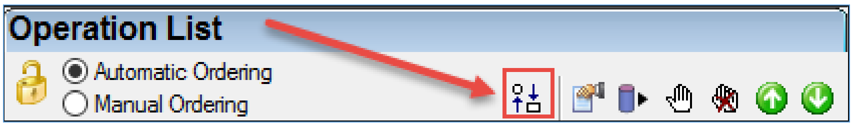
*If for some reason the **Feature Recognition** fails. Try using the **Edge-based FR**. This could give you better results.*

23 Click **OK** for options and select **Next**.

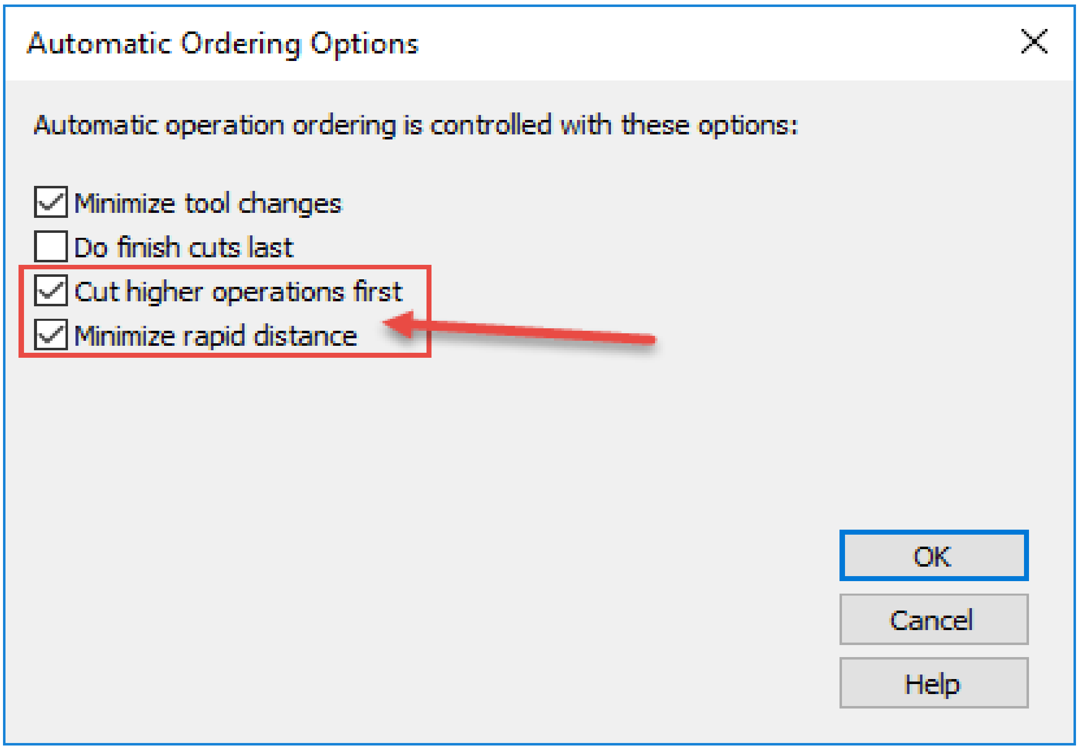
24 Setups can recognize multiple setups. Select **Finish** accepting all features.

25 Run the 3D Simulation.

26 If you see a gouge you will need to change the Automatic Ordering to Cut Higher Operations First.

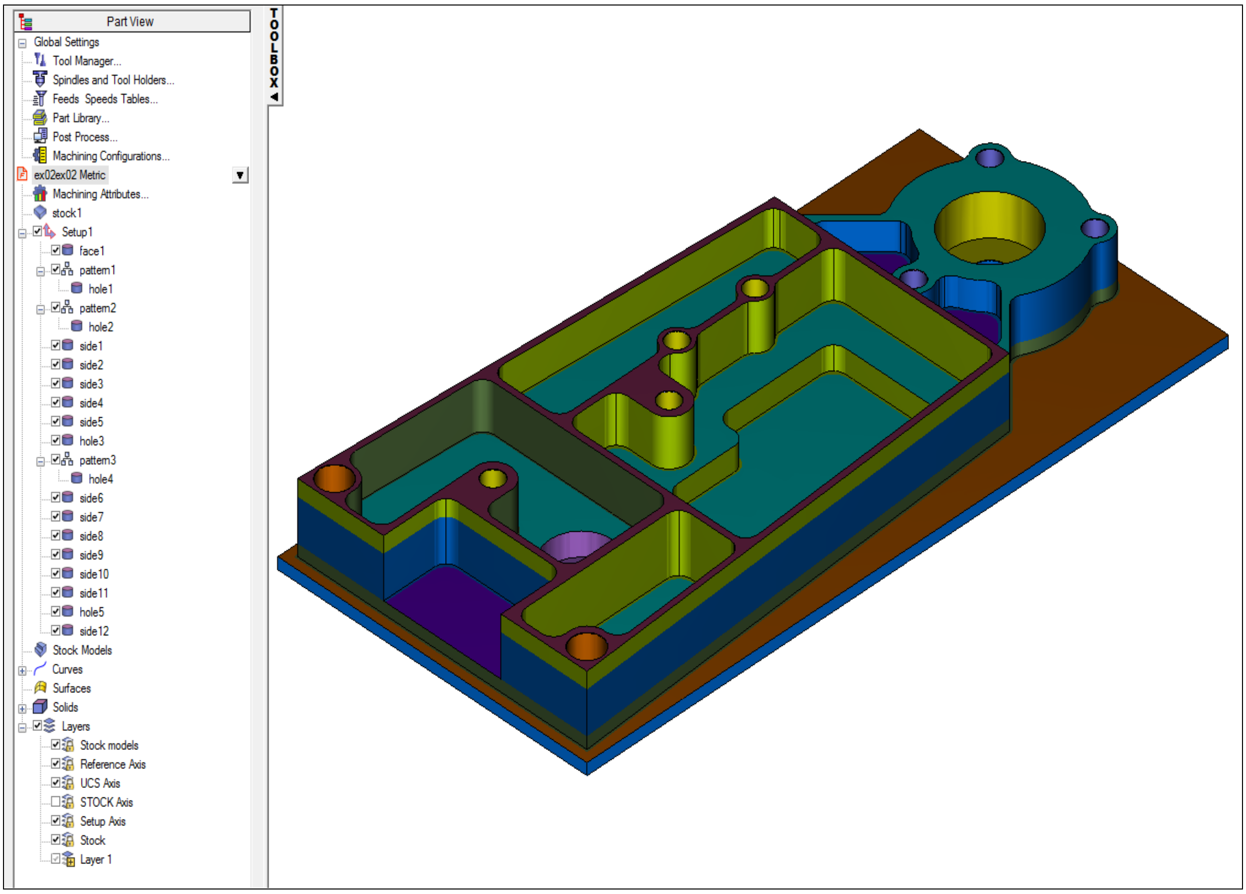


See Automatic Ordering Options on the next Page.



***All Features**, tooling, feeds and speeds, coolant, depth of cut and Step overs, etc. have been created for the entire part in this Setup.*

See finished image on the next page.

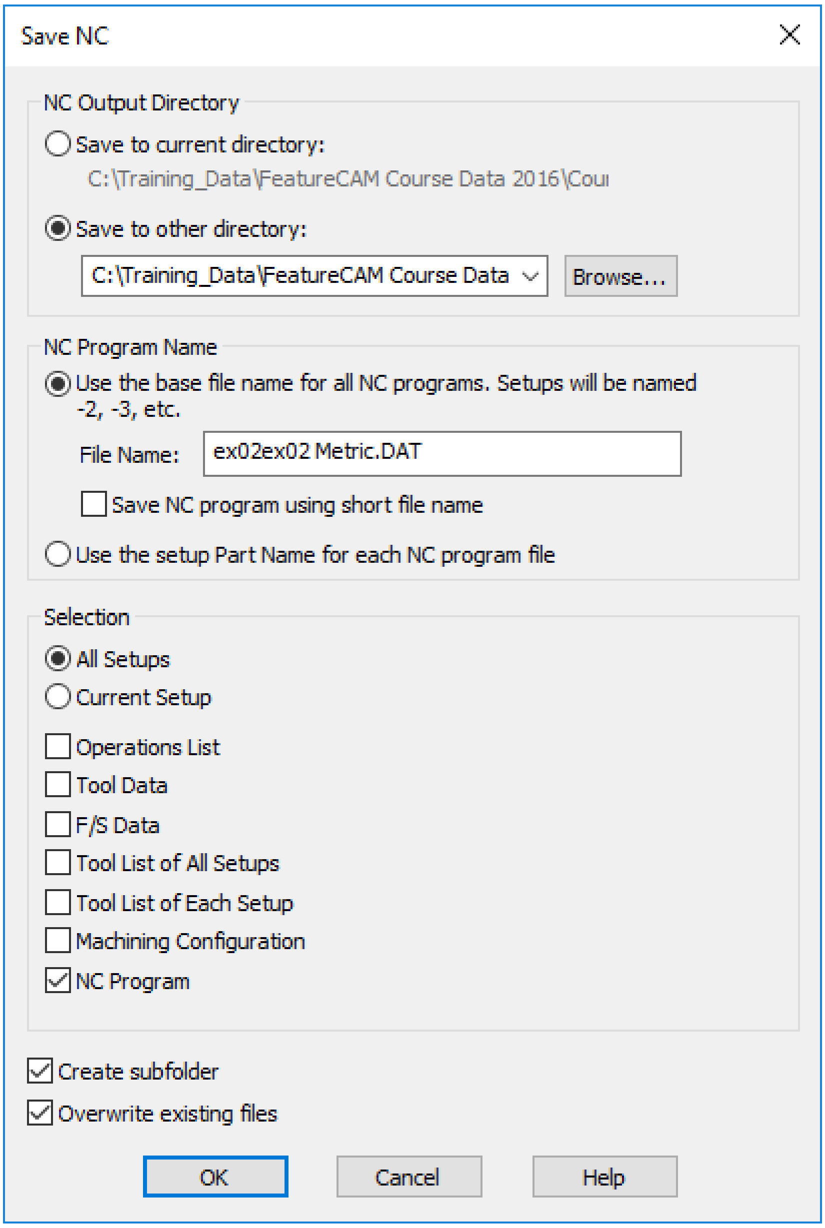


27 Click on **File** then **Save as** and use Windows Explorer to navigate to the Folder where your Instructor placed the files for your course. Name the file **ex02ex02 Metric.fm**

28 To **output** the **NC Code** make sure you have selected the correct **Post** and **Tool Crib**. For example. Fanuc_Robodrill.cnc | tools | 4 cores

29 Go to **File**>**Save NC.** Select the directory of your choice to output the NC Code.

30 Select **OK**.



IFR (Interactive Feature Recognition)

IFR Example #1

*IFR has three types of strategies available **IFR** Automatic by feature type, **IFR** using surfaces or faces and **IFR** chaining.*

IFR Automatically examines the model for a particular feature such as a side, pocket, face, slot, hole and boss. Side, pocket and boss features are not created using AFR side features but separate side, pocket and boss entities.

IFR using surfaces or faces extracts information about the highlighted entities extracting depth and feature contour.

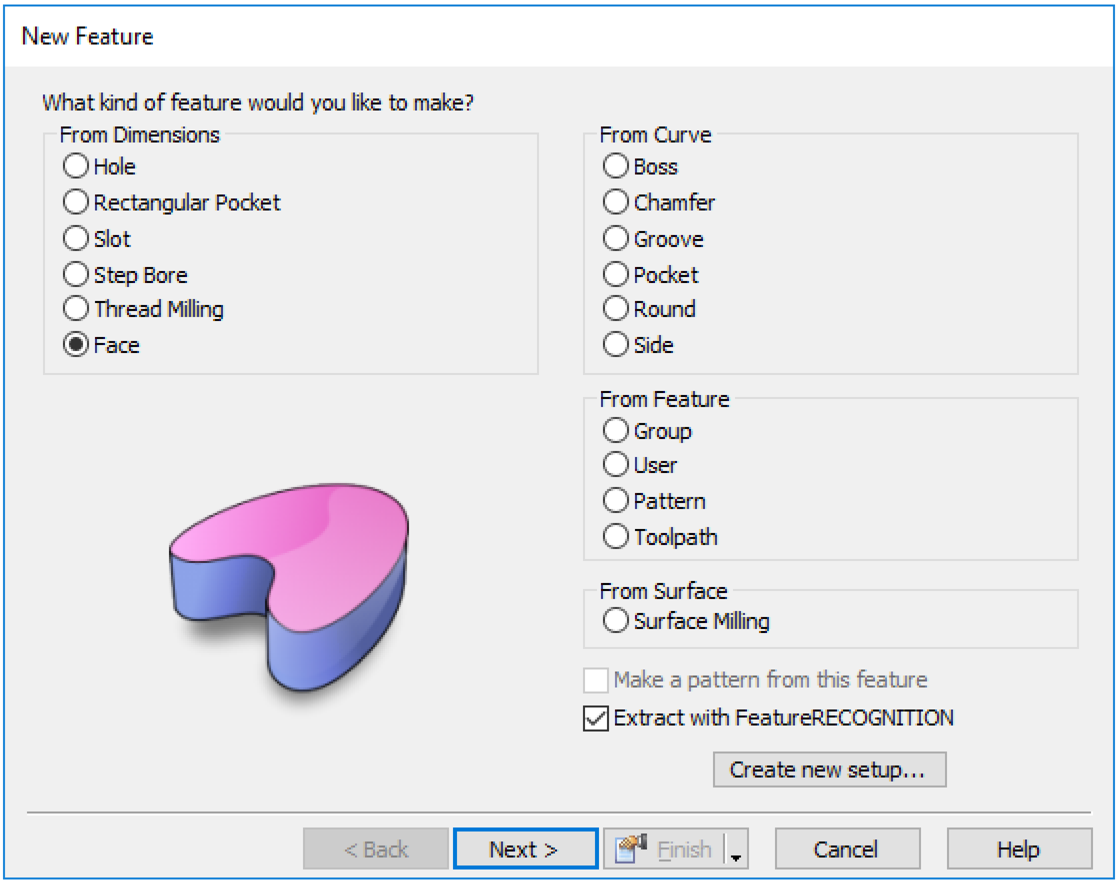
The IFR using chaining method takes a horizontal slice of the model then the slice profile may be chained into a curve.

Depending on the geometry of the feature to be extracted one technique may be more advantageous than others. Usually when programming a part using a solid model a combination of these techniques is used.

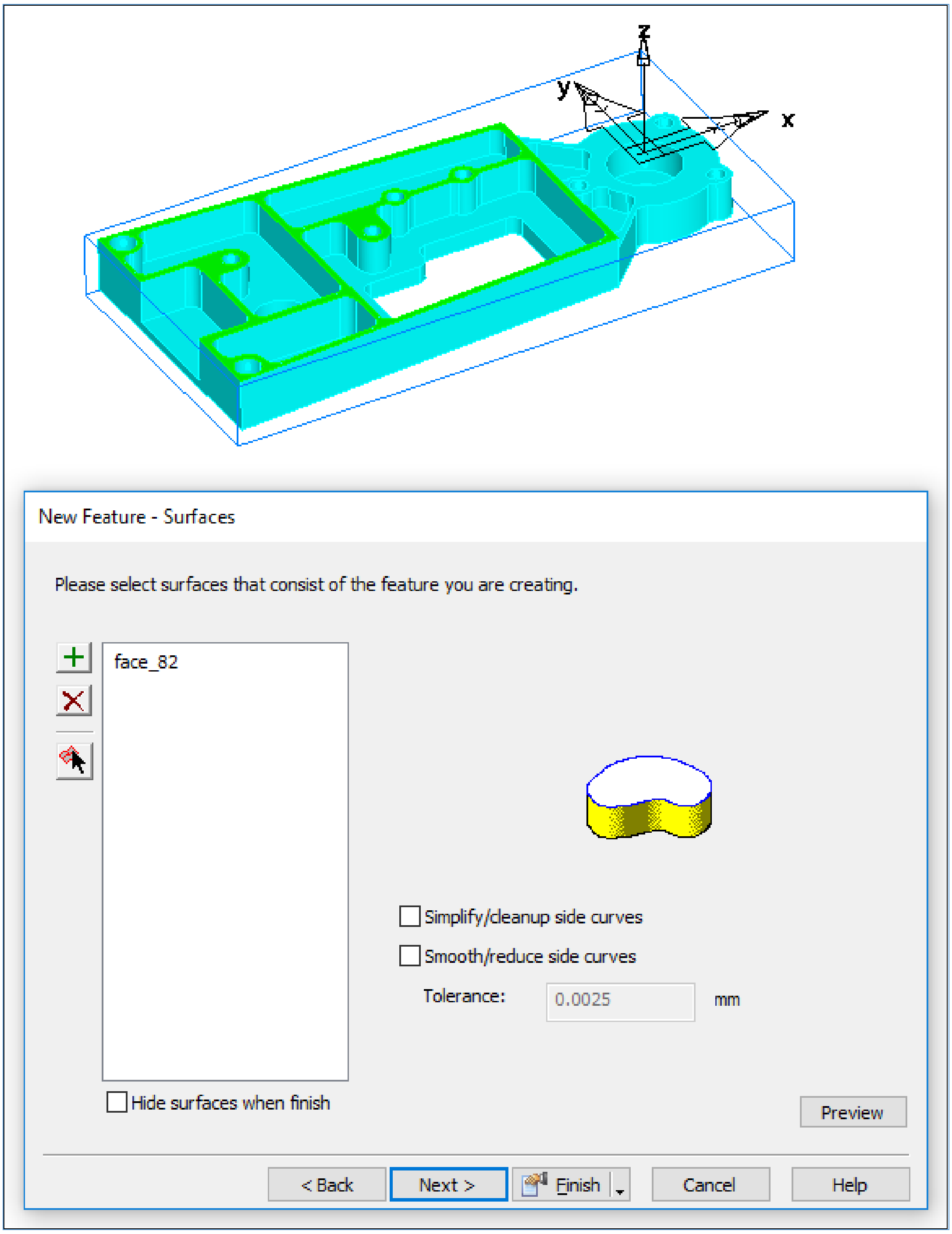
1 Using the same part which has already been aligned. **Delete** all of the **Features** in **PartView**. Delete all **Curves**. Rename part to **exo2exo2 Metric IFR.fm**

2 Create a New Feature by selecting **Ctrl+R** and select ***Face***. Check **Extract with Feature RECOGNITION.**

3 Select **Next**.



4 **Left Click** on the **Top most horizontal face** of the part and then click the **Green +** button to add it to the list. Then press the **Finish** button.

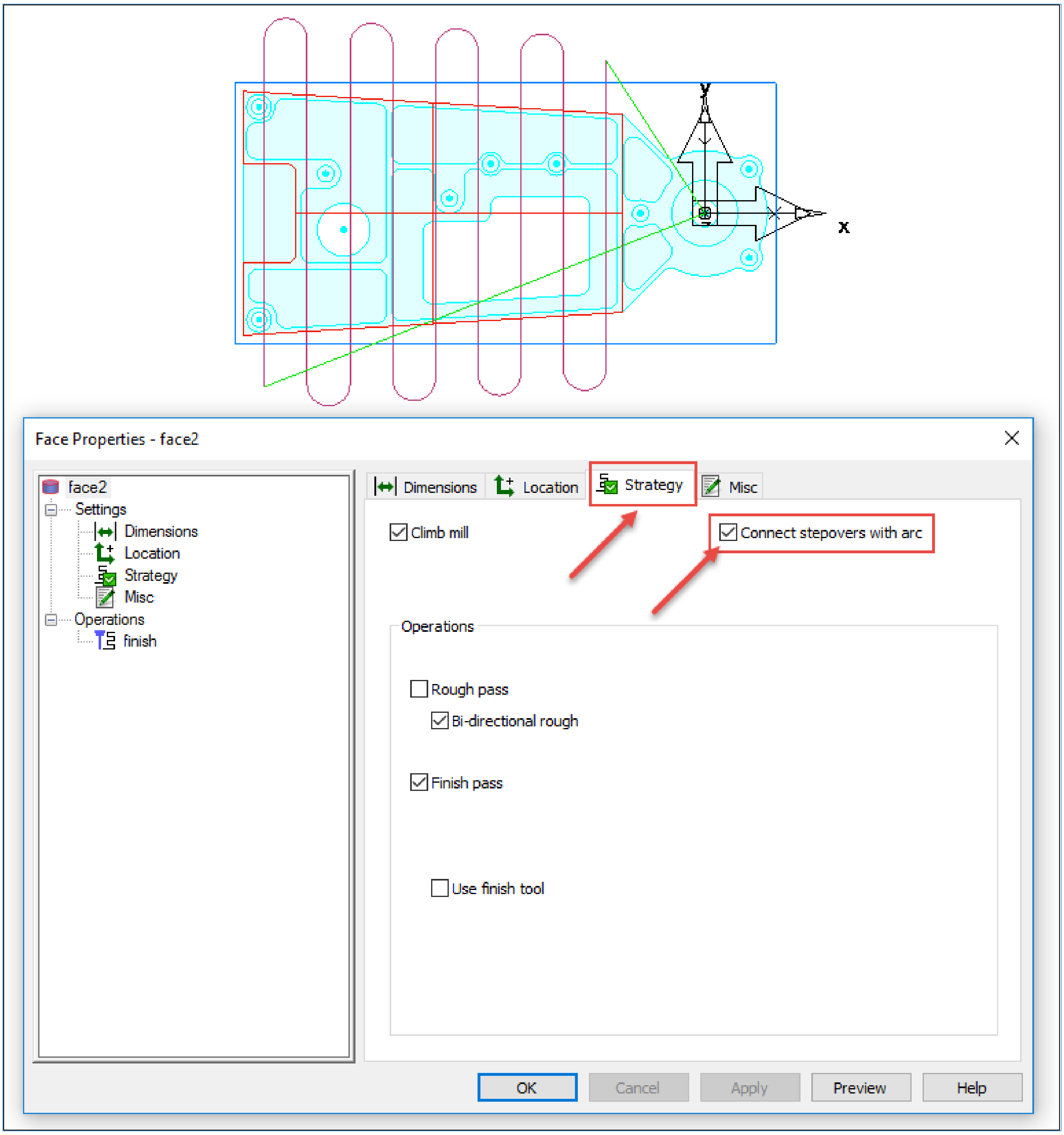


5 In the **properties of the Face Mill Feature**, Click the **Finish** Operation and the **Milling Tab** and change the **Zig-Zag Angle** to **90** and **the Lateral overcut** to %=**200.** This saves the tool feeding down onto the job.

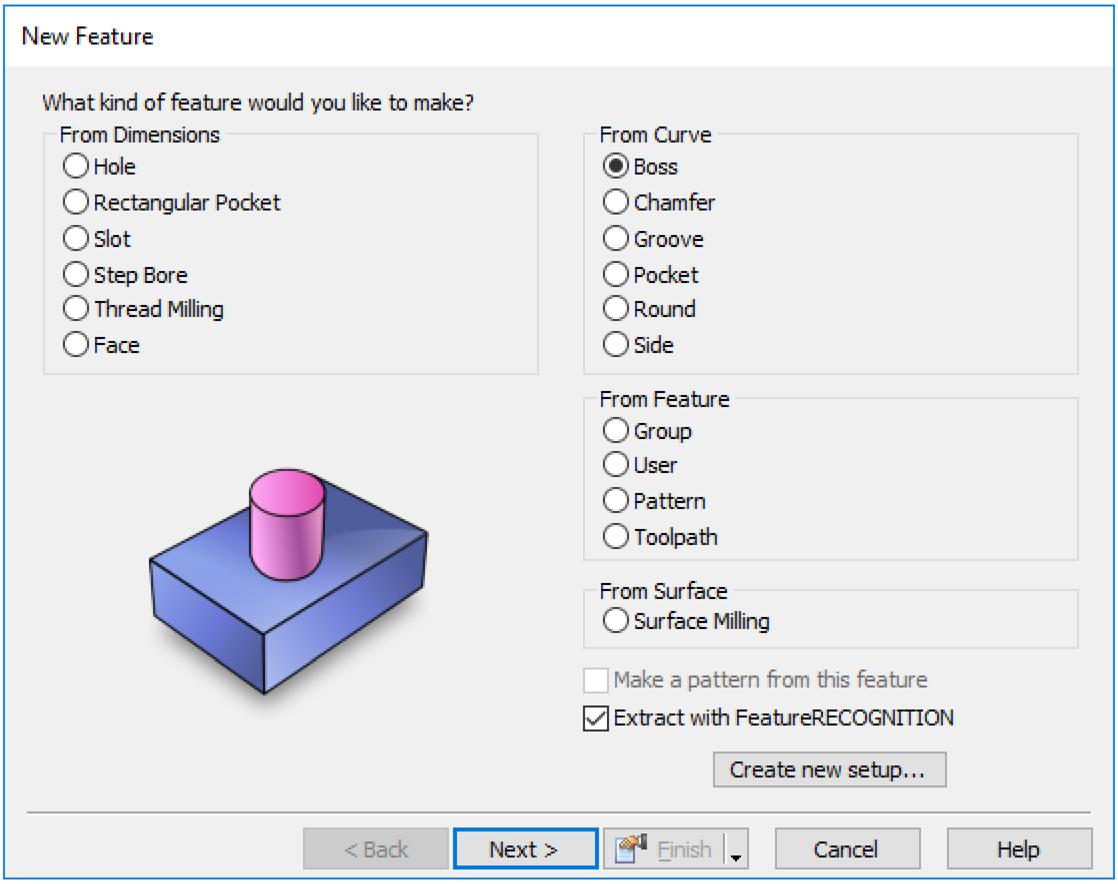
This will cause the toolpath to cut along the Y axis. You will see that it leaves an area that is not machined on the right end of the part. Think of this as saving time during production since you are about to program to cut that step away in the next feature.

6 Under the strategy tab select **Connect Stepovers with arc**.

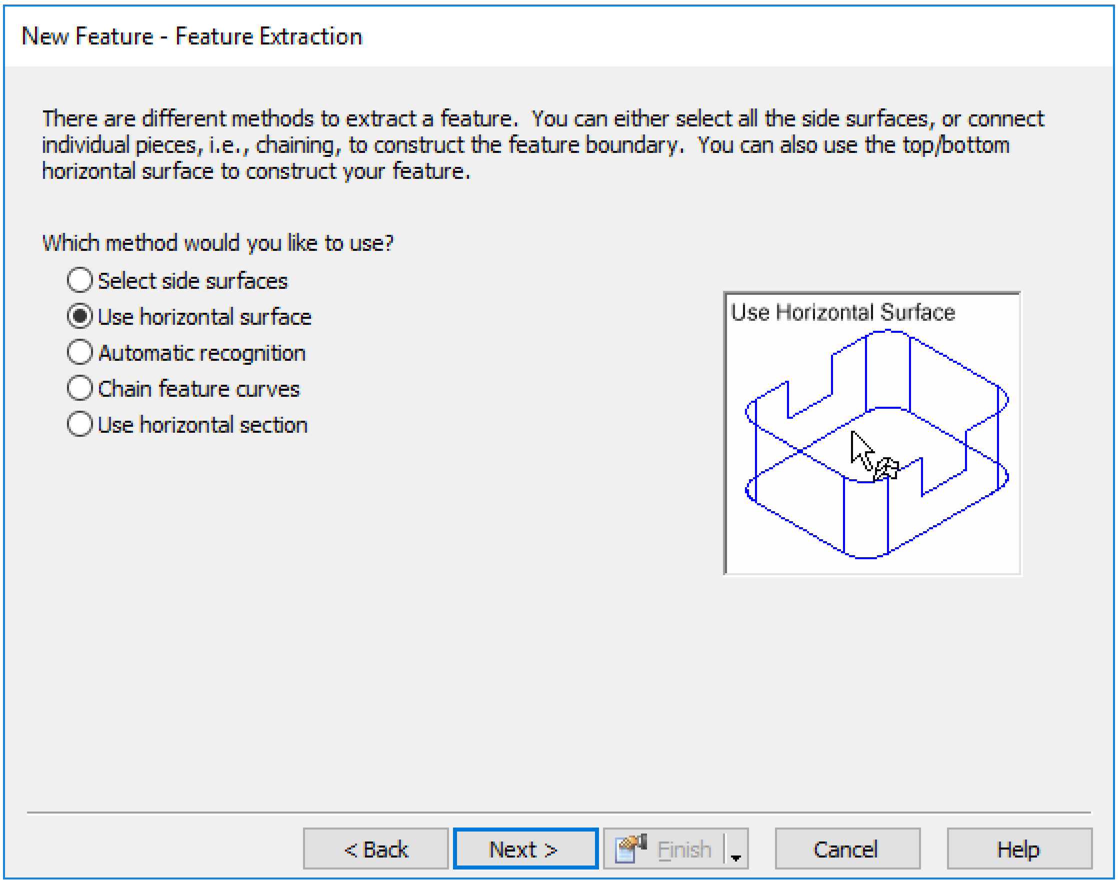
7 Select **Apply** and then **OK** to close the form.



8 Select ***Boss*** with **Extract with Feature Recognition** from the New Feature Wizard.

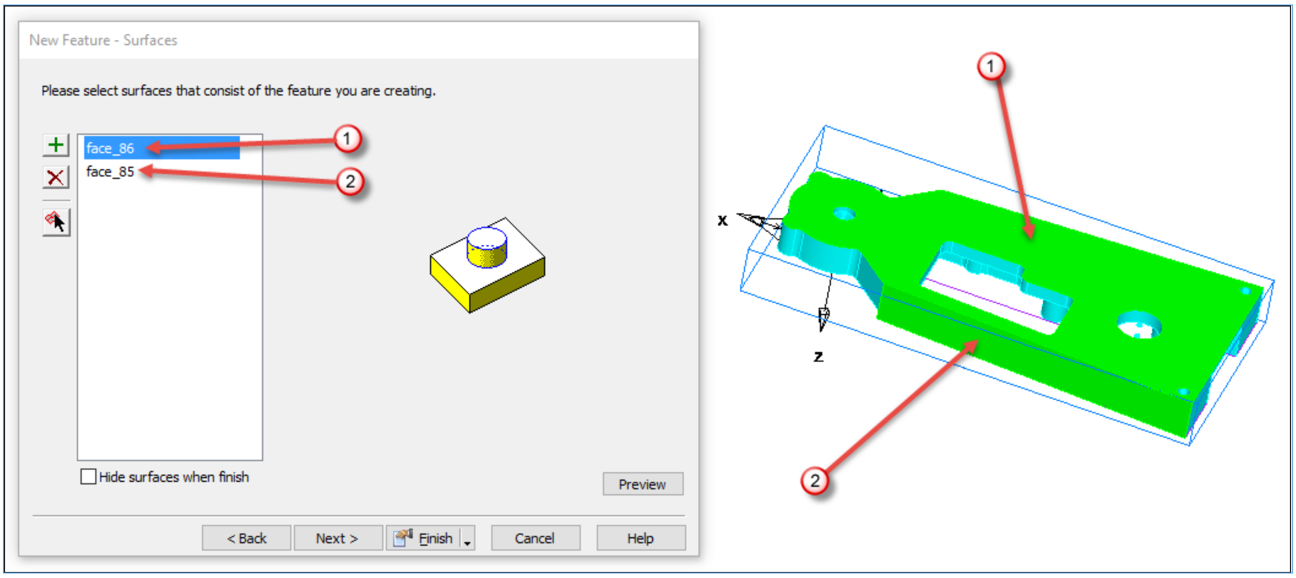


9 Select **Next**. Select "**Use Horizontal surface".**



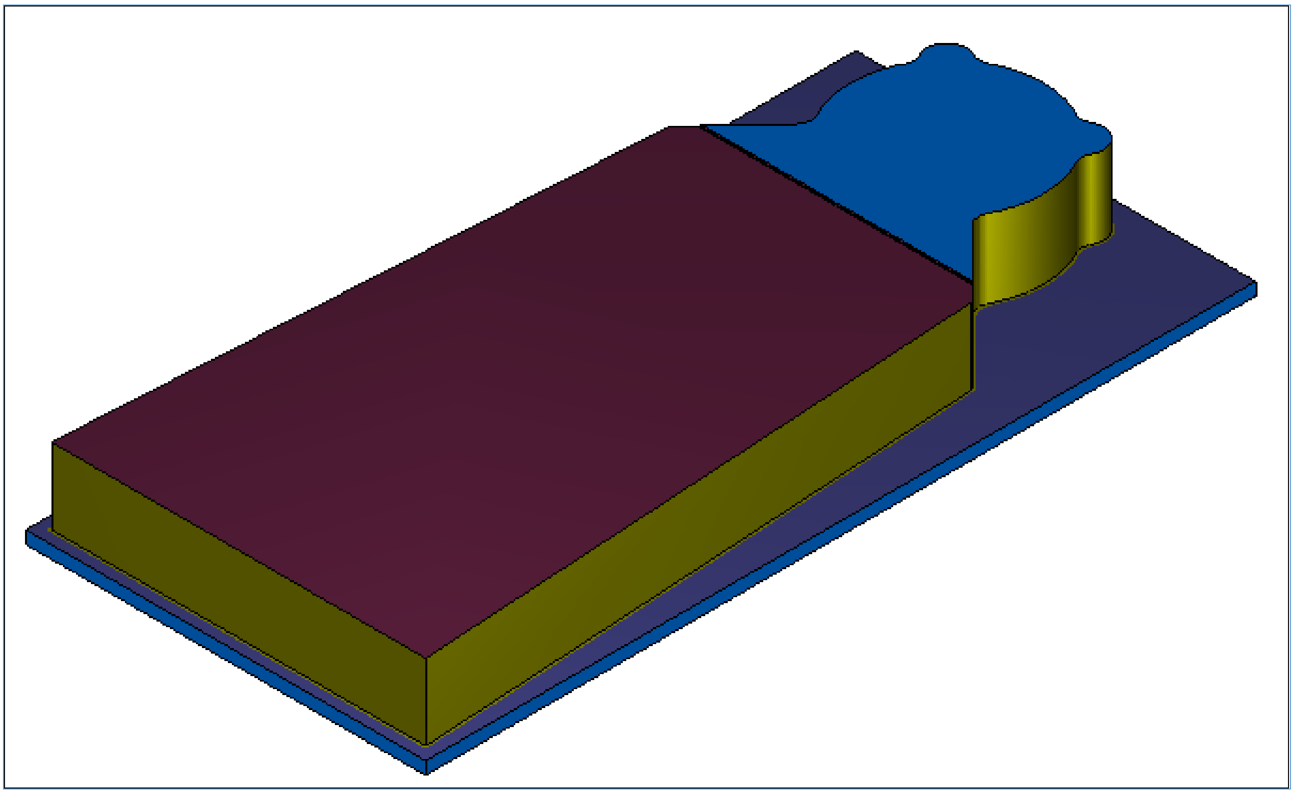
10 Then select **Next**.

11 Rotate the Part view and select the Bottom horizontal face for the shape and One Vertical face to determine the Top and Bottom Z level. **Use Horizontal Surface** for both selected faces.

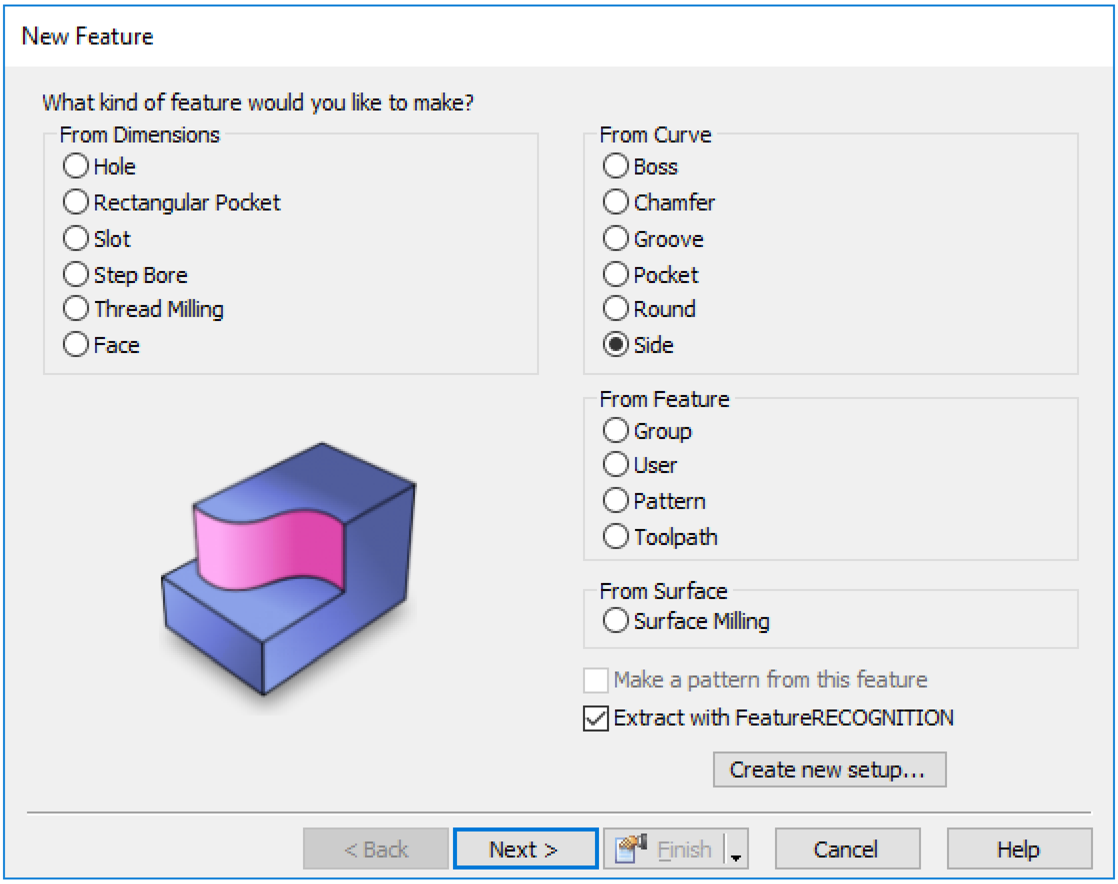


12 Select **Green "+"** to add selection to the list.

13 Select **Finish**. Then run a **3D Simulation**. FeatureCAM will automatically select the required tools from the Tool Crib. This can be changed at any time.

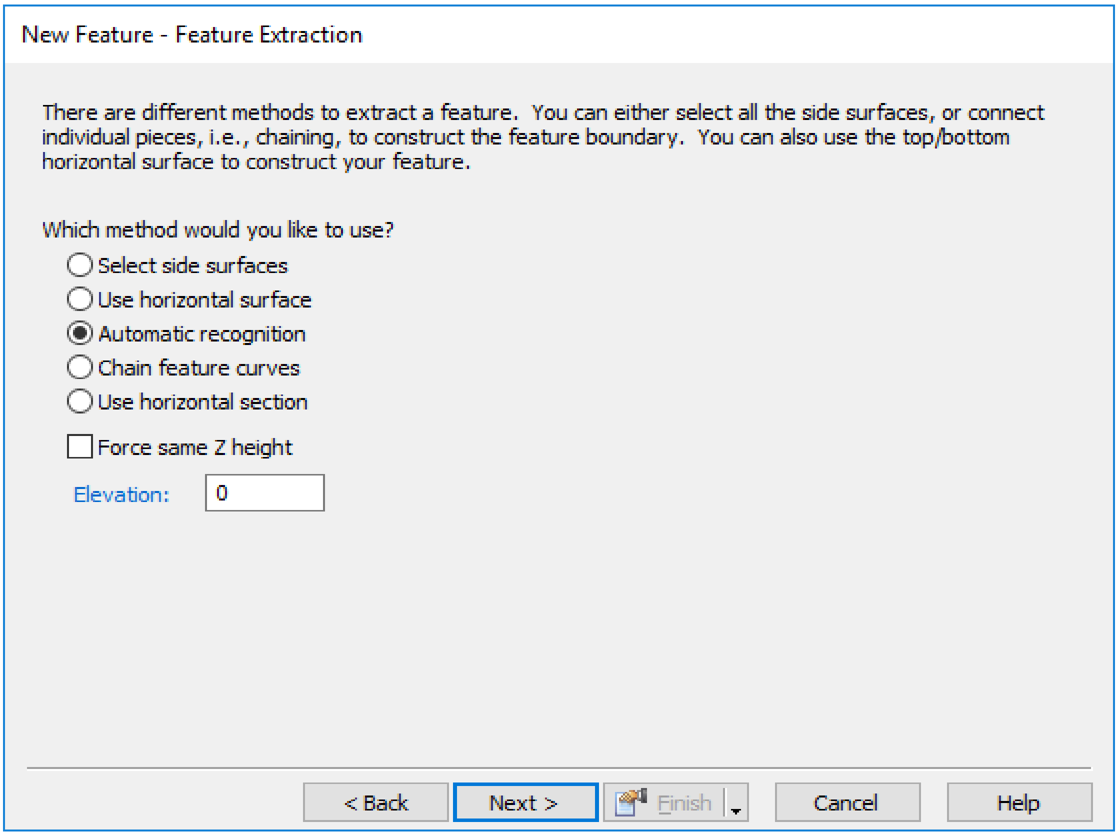


14 Select New Feature wizard and select ***Side*** with Extract with Feature Recognition.

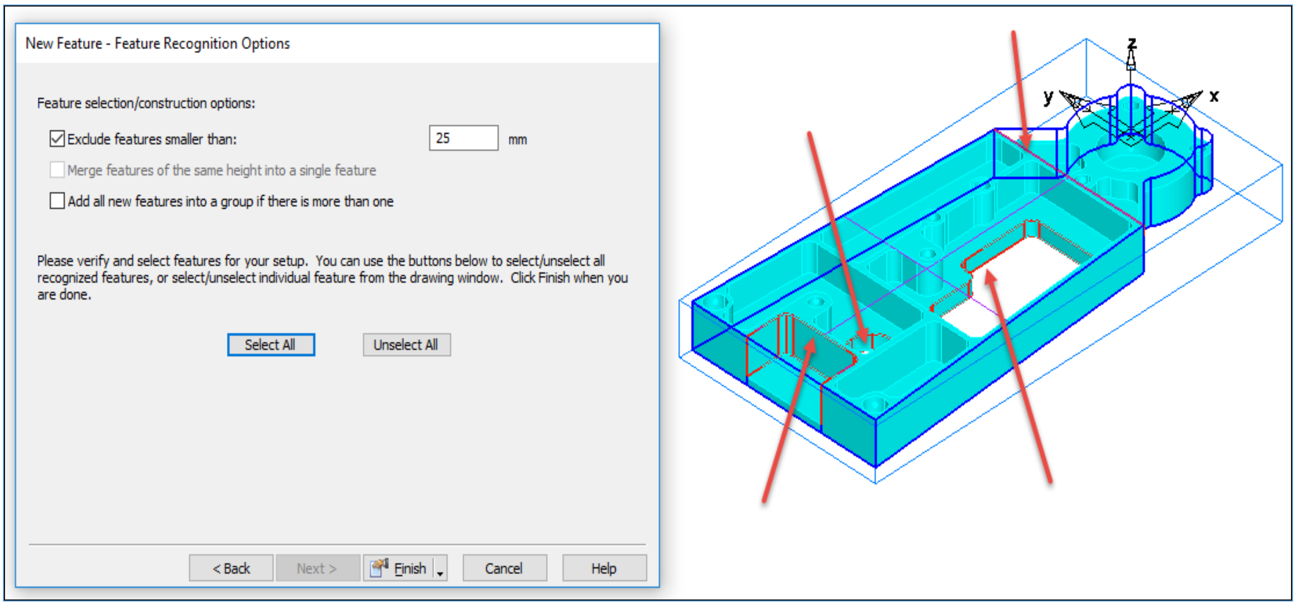


15 Select **Next**.

16 Then Select **Automatic Recognition.**



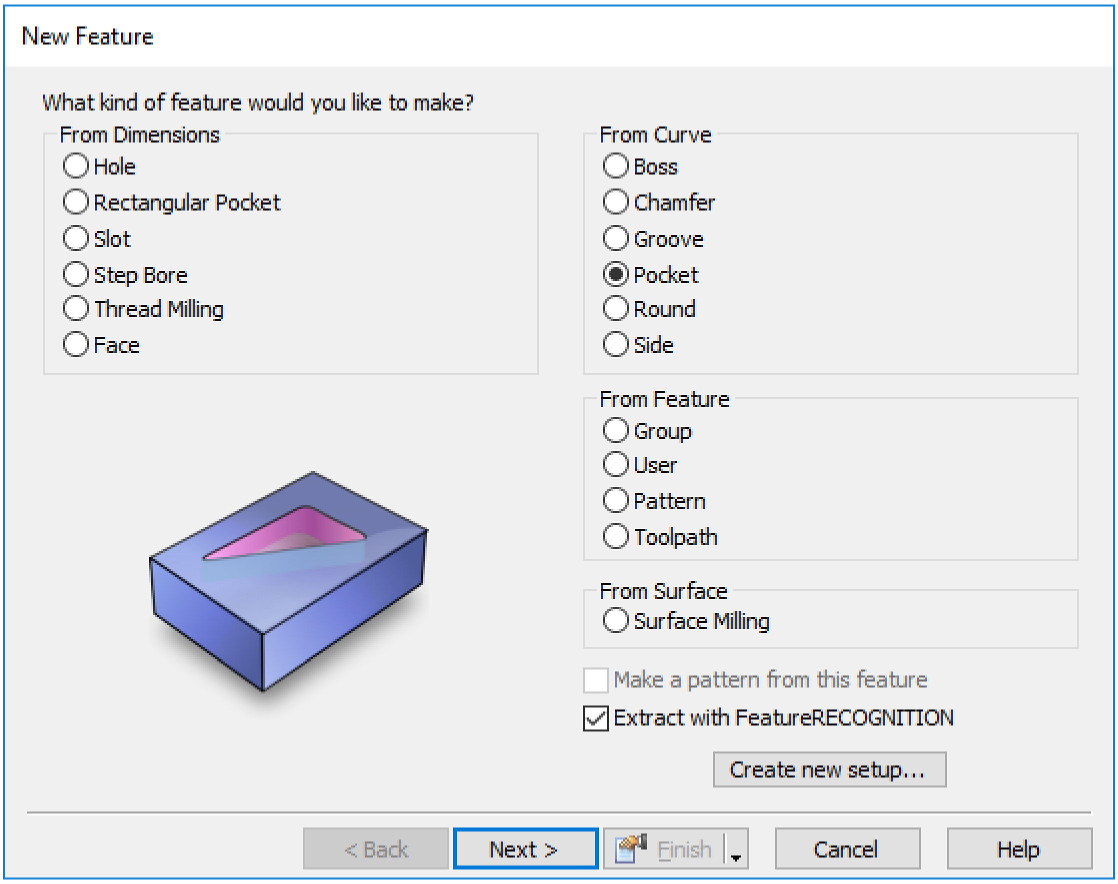
17 Only **select** the **features shown**. The best way to do this is to use **Select All** then **un-select** the outer profile**.** The will turn **Blue.**



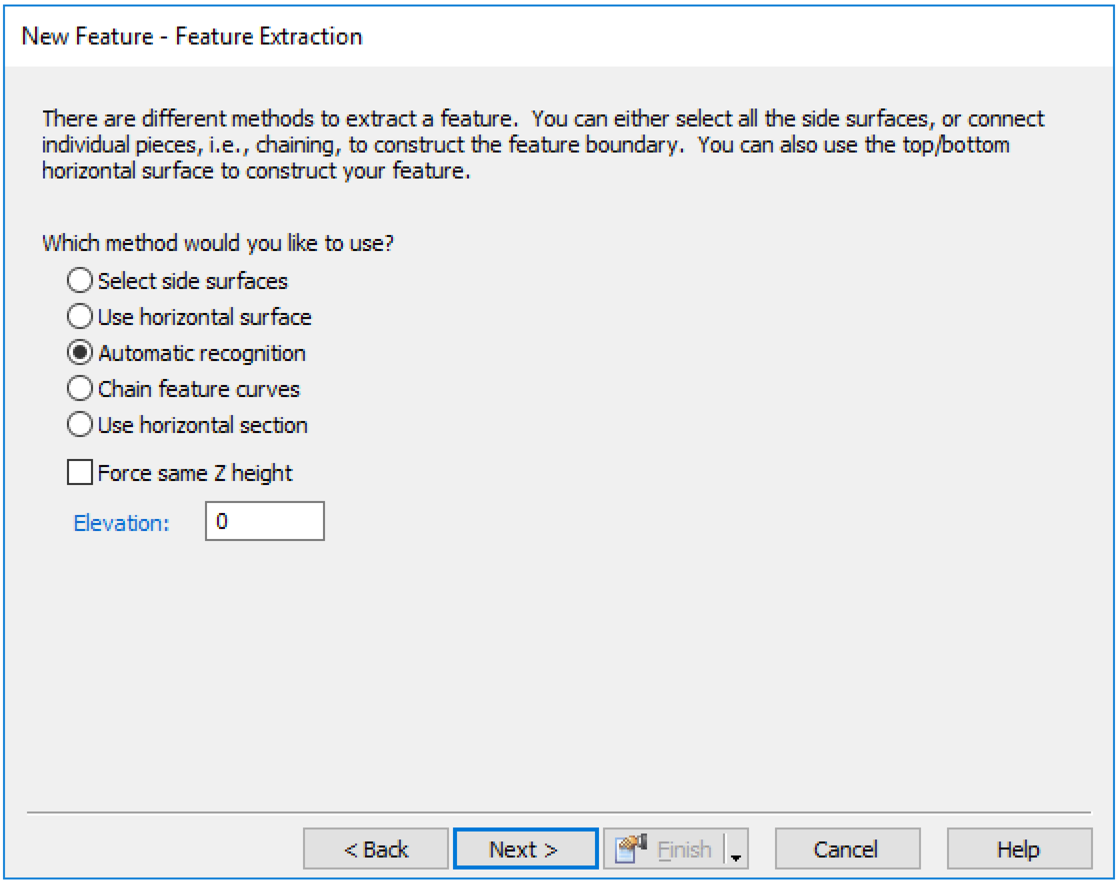
18 Select **Finish**. All open Features will be machined.

19 You are probably thinking to yourself. Why are we machining these pockets as they are at the bottom of the part? All will be explained in the next section.

20 Create a New ***Pocket*** Feature and **Extract with Feature RECOGNITION**.



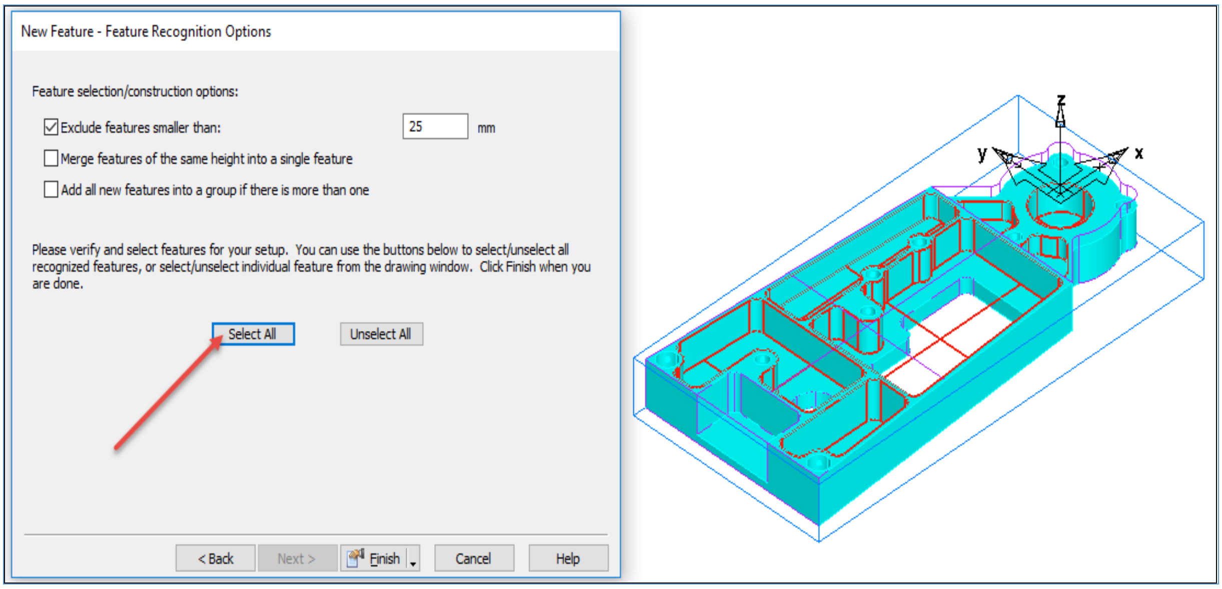
21 Select Automatic recognition.



***FeatureCAM** using **IFR** and **Automatic recognition** will identify all pocket features.*

22 Select **Next**.

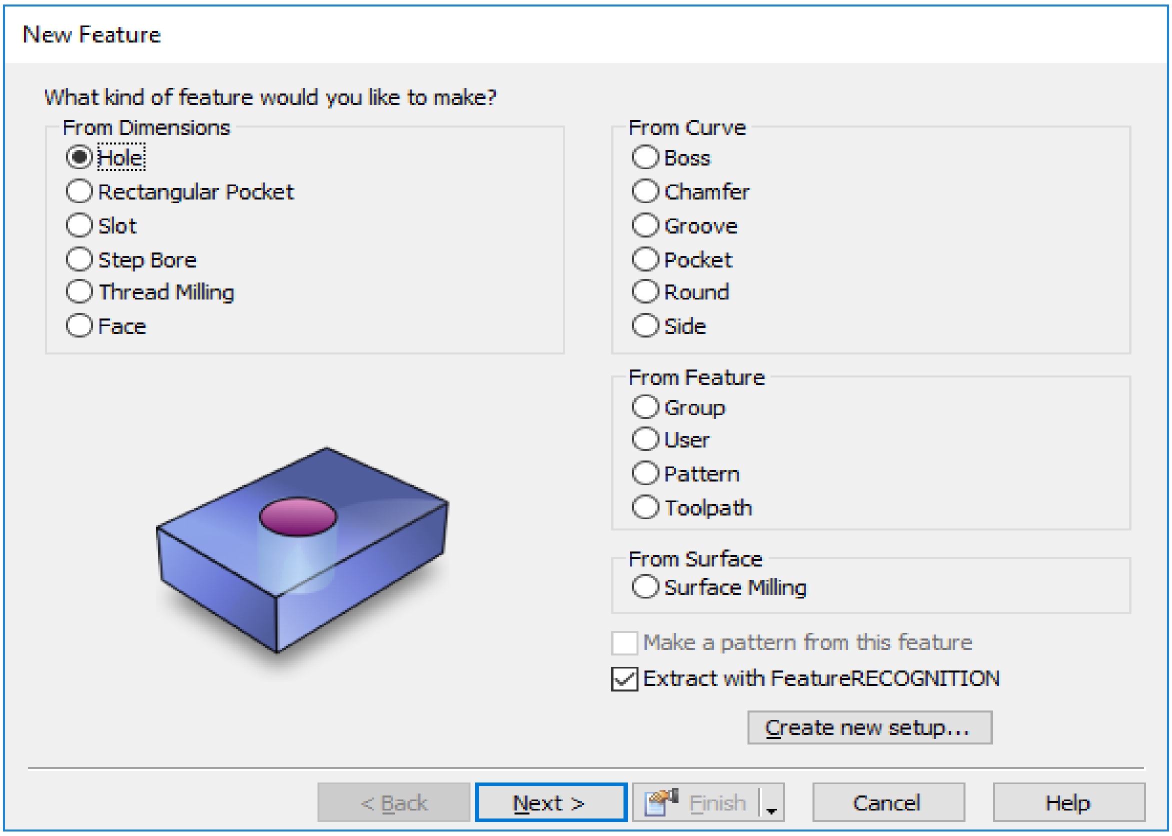
23 Press the **"Select All"** button to select all identified features.



All selected pockets turn red when selected

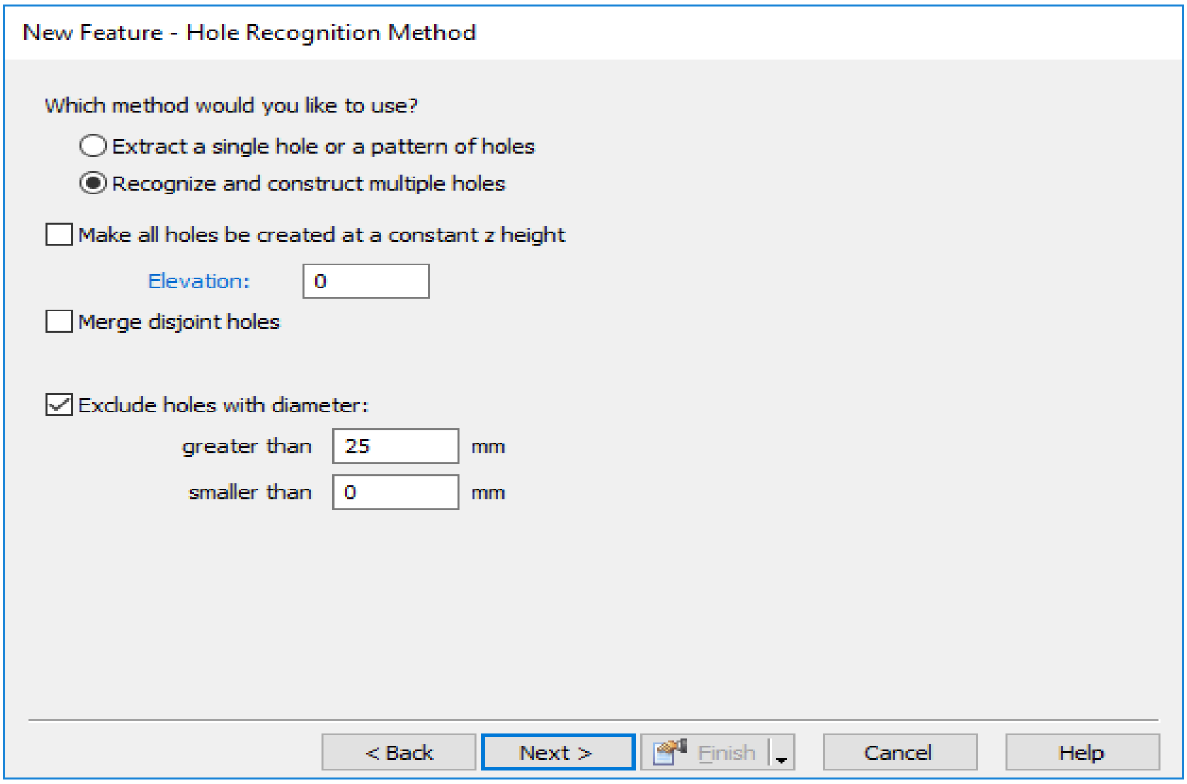
24 Select **Finish.**

25 Open the New Feature Wizard and select ***Hole*** and check Extract with Feature RECOGNITION.

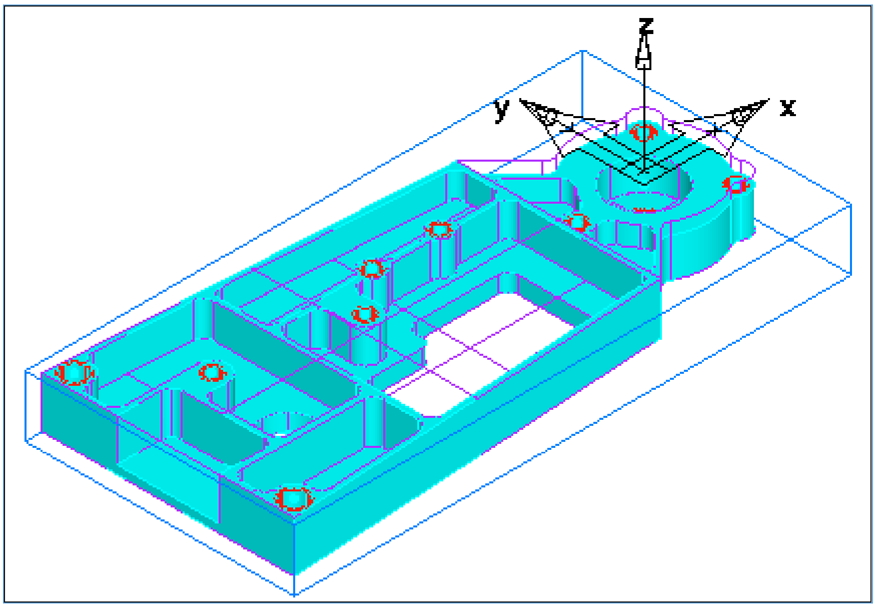


26 Select Next. Select the Recognize and construct multiple holes radio button.

27 Check the Exclude holes with diameter > (greater than) and enter a value of 25mm.



28 Select **Next**.

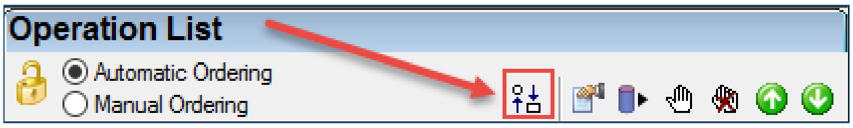


29 Select, **Select All**.

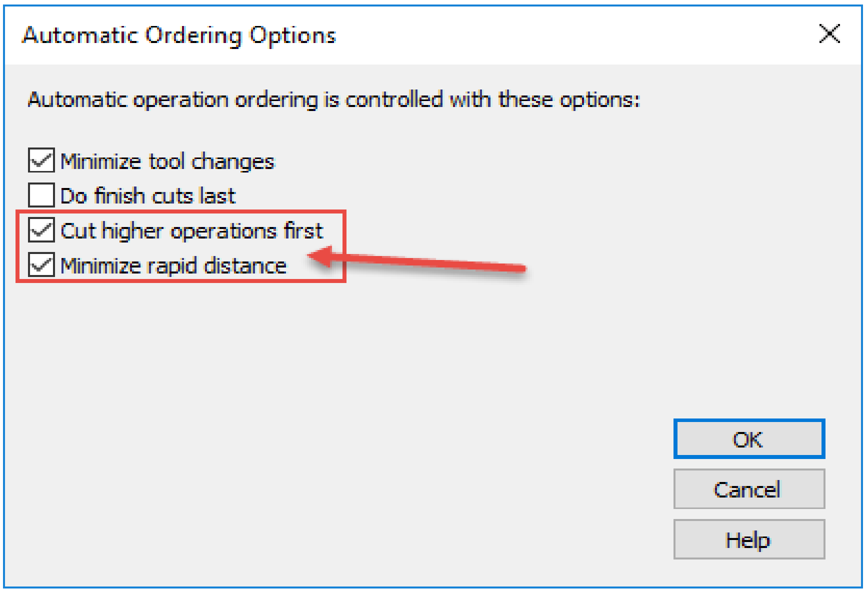
30 Select **Finish**.

31 In **Automatic ordering options.**

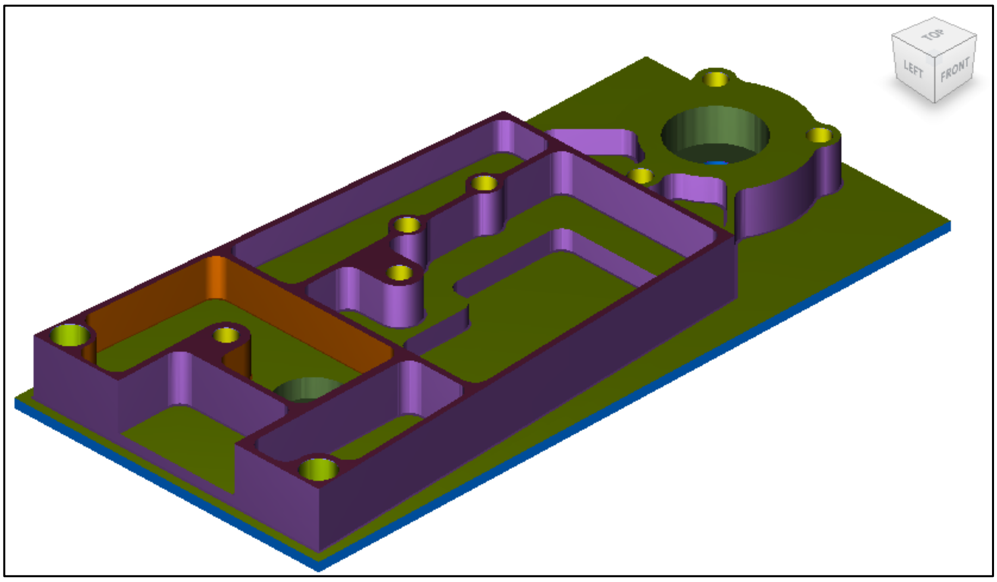
If you see a gouge you will need to change the Automatic Ordering to Cut Higher Operations First.



See Automatic Ordering Options on the next Page.

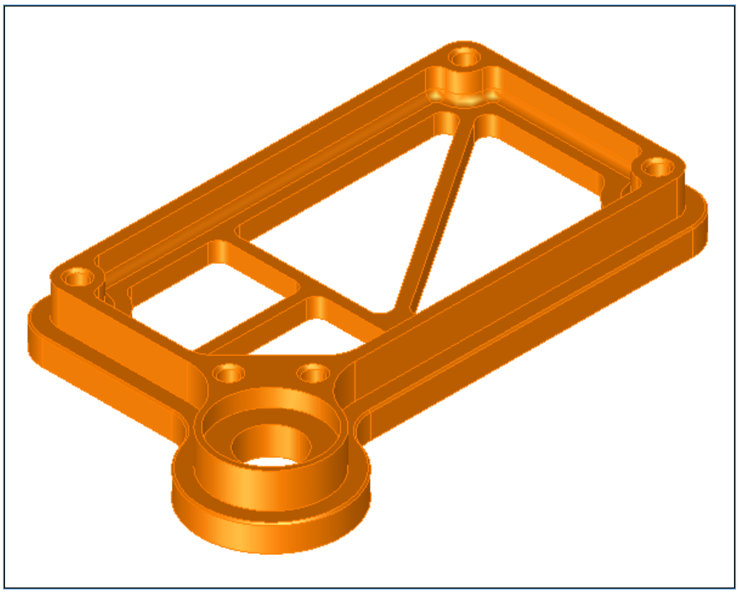


32 Run **3D Simulation**.



***Tip**: Even though we had machined the pockets at a lower level. When we selected **Cut Higher Operations First**. This has **re-ordered** the operation sequence.*

AFR Part 2 – 2 setups (Class Exercise)



*The next example will show you how to machine a component with multiple setups. **Setup1** for side one and **Setup2** for side two using **AFR**.*

1 Import File (**AFR Part2.x_t**) (Parasolid)

2 Use the **Wizard** to establish the initial **Setup** location and **Stock** size.

3 **Do not** tick **Launch AFR** after finish. Select **Next.**

4 Setup Z direction is ok. Select **Next** and **Next.**

5 Stock Type is block. Select **Next.**

6 Use **Compute stock size from the size of the part**. Change extra stock to **5mm** for **-X +X -Y +Y.**

7 Set Z- and Z+ to **2mm** for stock size. Select **Next**.

8 Set Z origin **Top View centre +** to **-2mm.** Select **Next**.

9 Are you going to use Multi-Axis positioning to machine this part? **No.** Select **Finish**.

10 Change Solid model Colour. **Select** Model in **Part View**. Go to **Options** and Select **Change Selected>More Colours**. Pick Colour and select **Apply** and then **Done**.

11 Create **Setup2** for **Side 2** by double clicking on **Setup1. New>Align to stock Face. Next.** Select **Bottom Centre +.**

12 Select **Next.** Change **Z Offset to -2mm.** Select **Finish.**

13 Go to **Construct** and Run **Automatic Feature Recognition** for **Setup1** and **Setup2**. **Delete** unwanted or duplicate operations on **Setup2**.

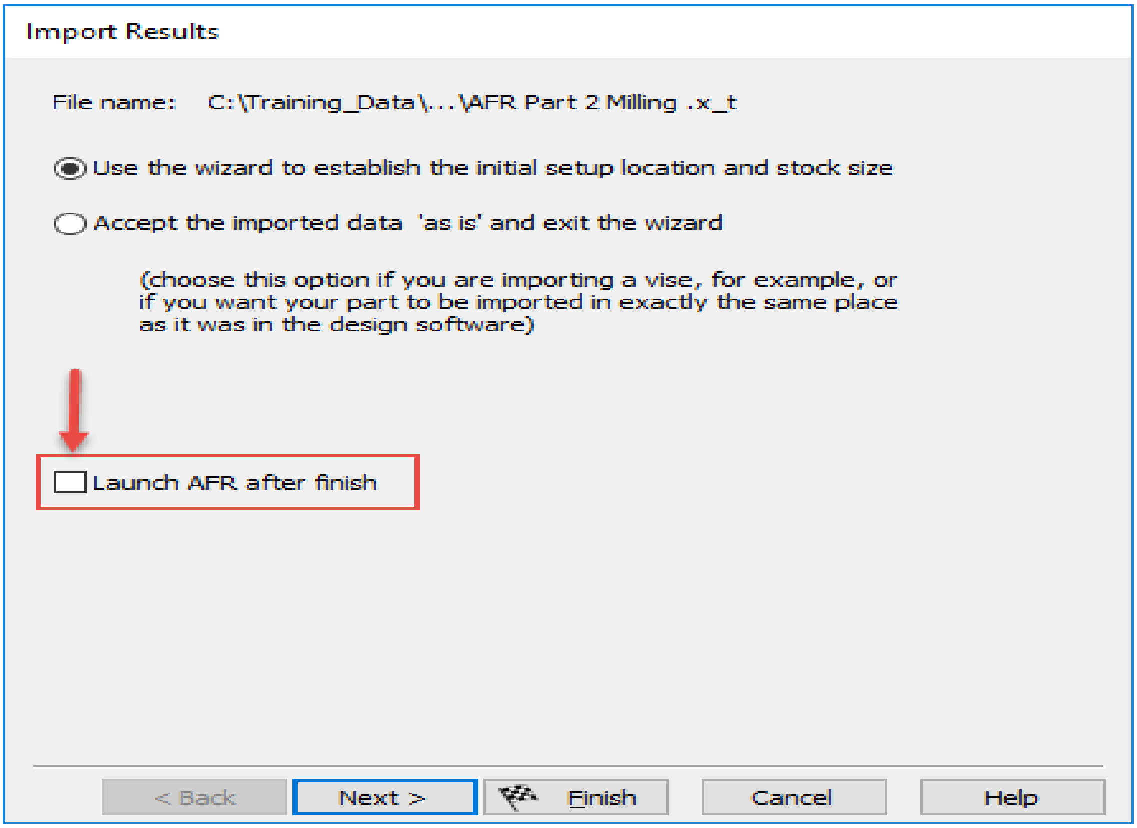
14 Run **3D Simulation.**

15 **Output Code**.

Import Wizard

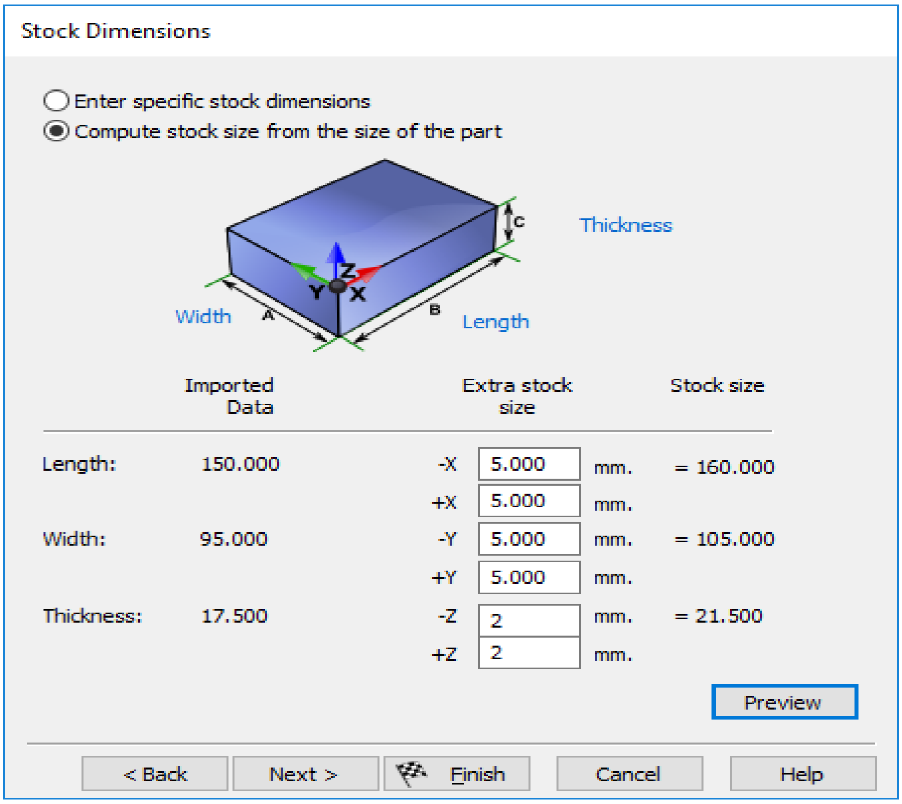
*When you import a file, **FeatureCAM** steps you through the import wizard. The import wizard pops up automatically and assists you in importing the file into **FeatureCAM**.*

1 Go to **File>Import> AFR Part2.x_t.** Please navigate to **C:\Training_Data\FeatureCAM Course Data 2017\Milling Files to import.** Then select **AFR Part2.x_t**



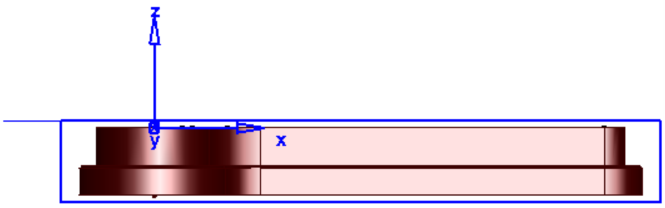
***Do NOT** use **Launch AFR after Finish**. Use the Wizard to establish the initial setup location and stock size. Select **Next**. The part is correctly orientated so Click **Next** until you get to **Stock Type** which is **Block**.*

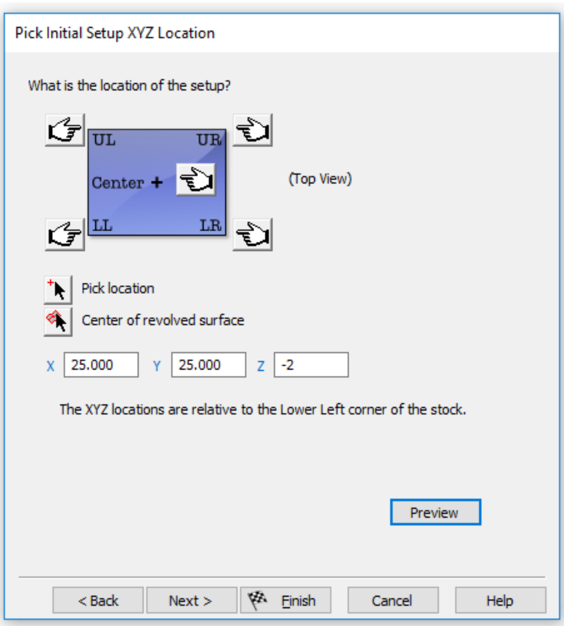
2 On the **Stock Dimensions** page check the radio button, **compute stock size from the size of the part**. Enter dimensions as shown on next page.



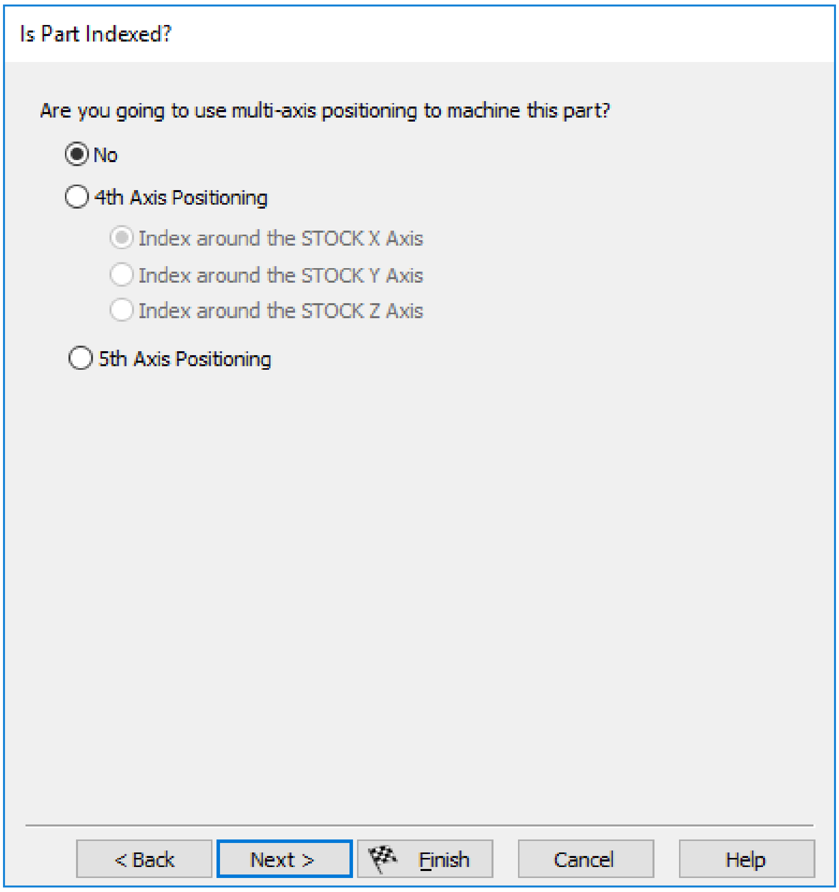
3 The Units are in Metric and we have allowed **5mm** for each side edge and **2mm –Z** and **+Z.** Select **Next**.

4 Leave **Setup1** in the same **X, Y, position**. Then change only the **Z figure to -2.** Select **Next.**





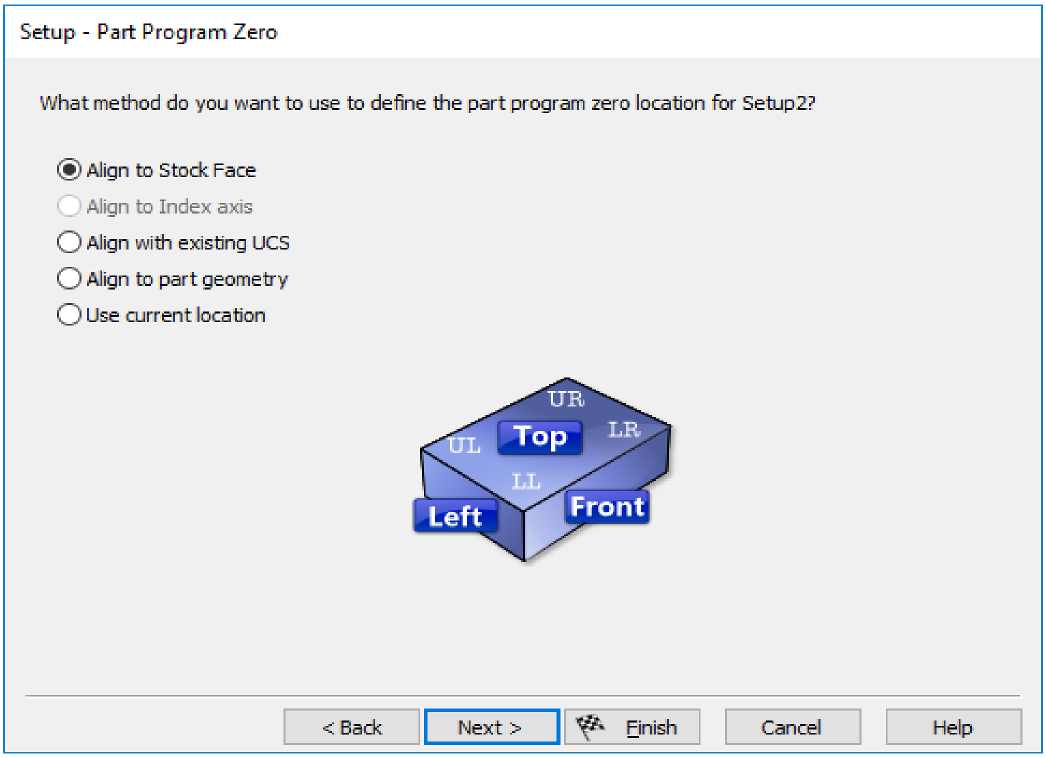
5 **Is the part indexed?** In this example the answer is **No**. Select **Finish.**



6 We have to create another Setup called **Setup2.**

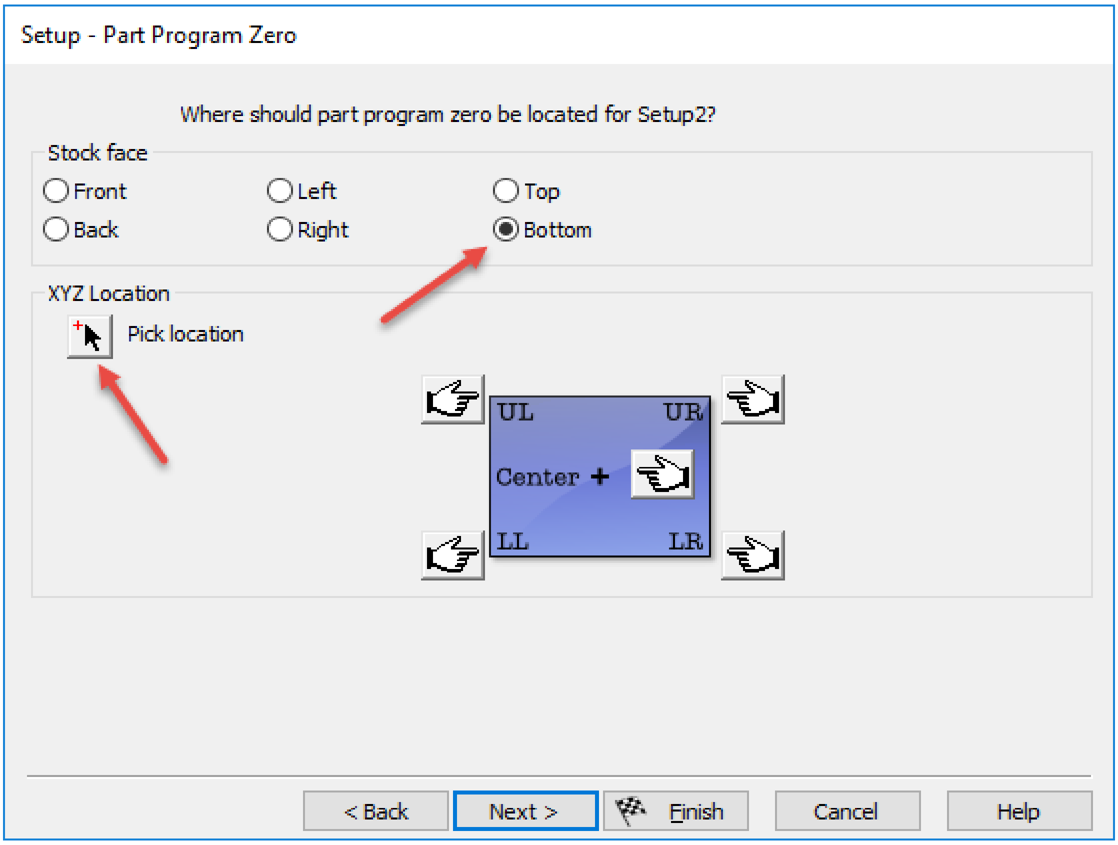
7 Double click on **Setup1** and select **New.**

8 Select **Align to Stock Face.**



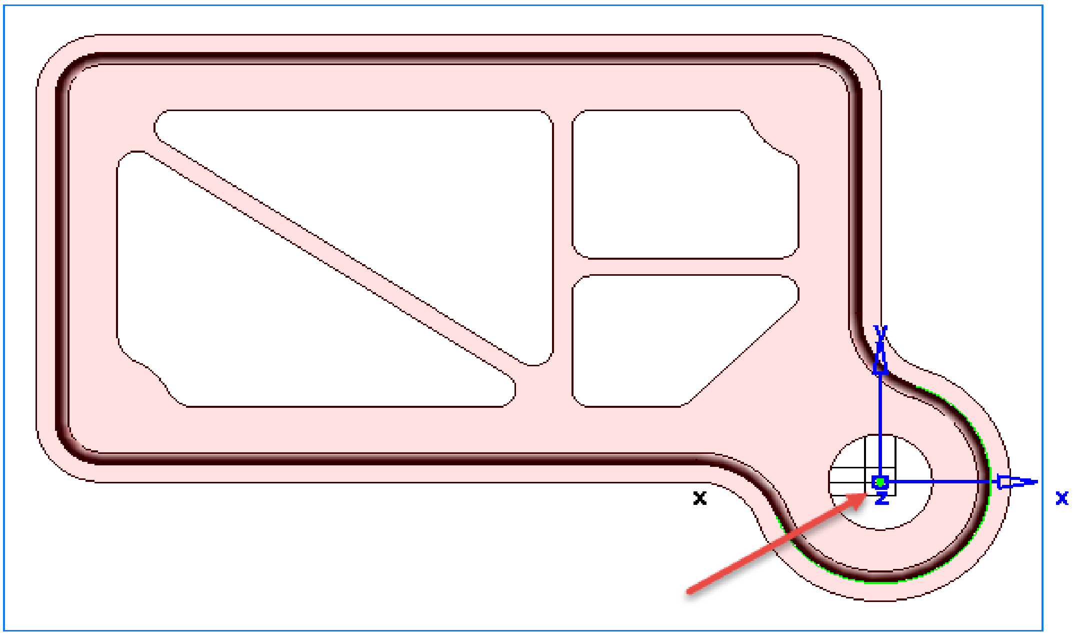
9 Select **Next.**

10 Select **Bottom** then select **Pick Location.**



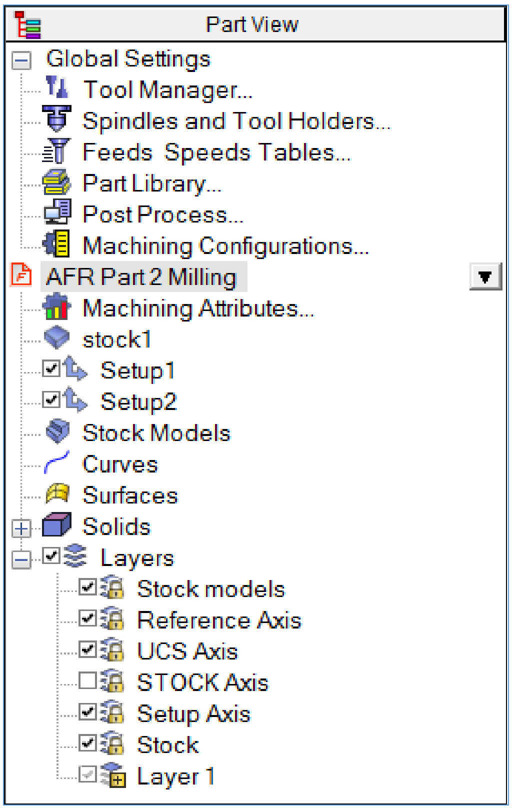
11 Make sure you have snap to cylinder active.

12 **Snap** to the **centre** of the same **bore** as shown below.



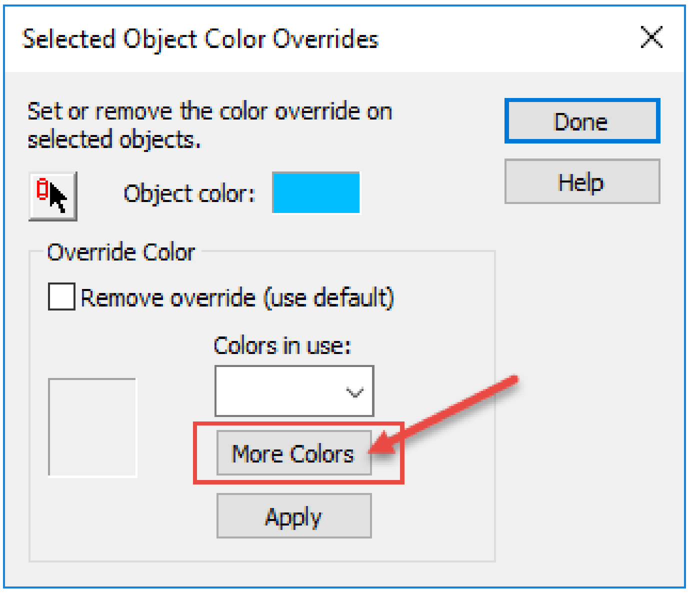
13 Select **Finish** and **Close**.

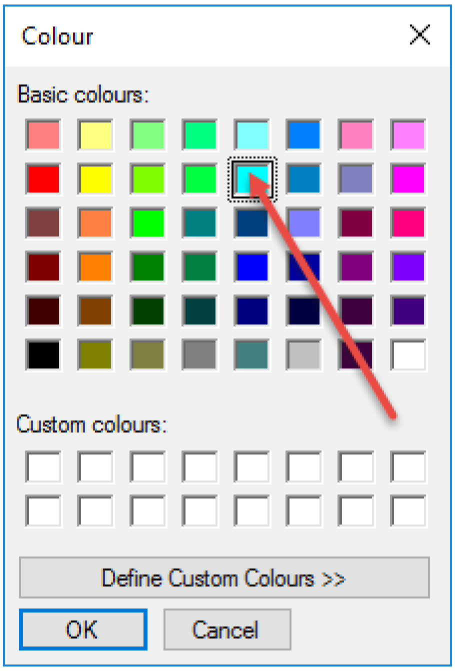
14 We now have **Setup1** & **Setup2** displayed.



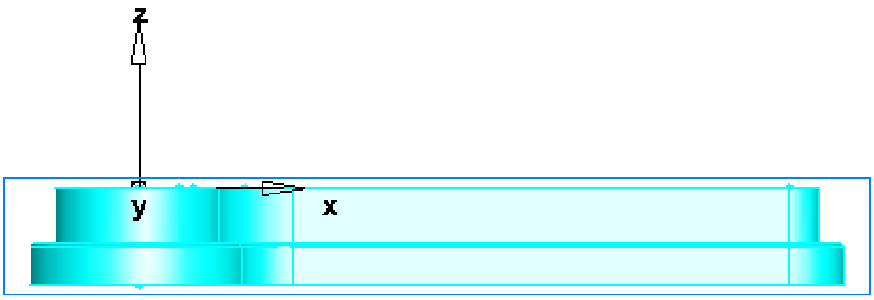
We are now going to change the colour of the model.

15 Select the model in **PartView** under **Solids.** If you cannot see the model select the **+** button. This would be a good time to rename the solid model to a name of your choice. I have **renamed** it to **AFR Part 2 Milling**. Select the **model** then go to **Options>Colouring>Change Selected>More Colours.**

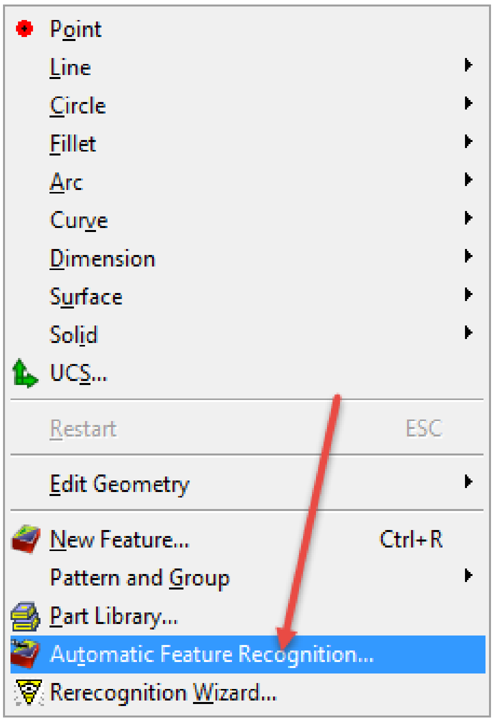


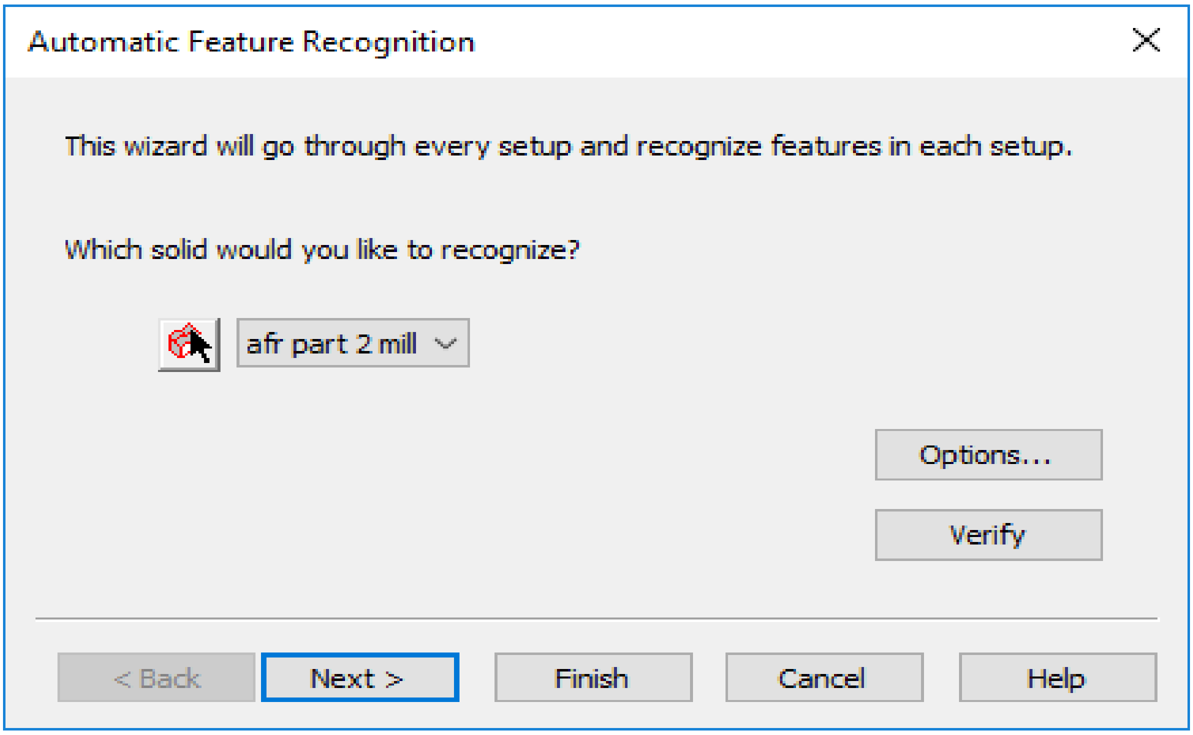


16 Select **Ok**, **Apply**, and **Done**. The model has now changed **colour**.



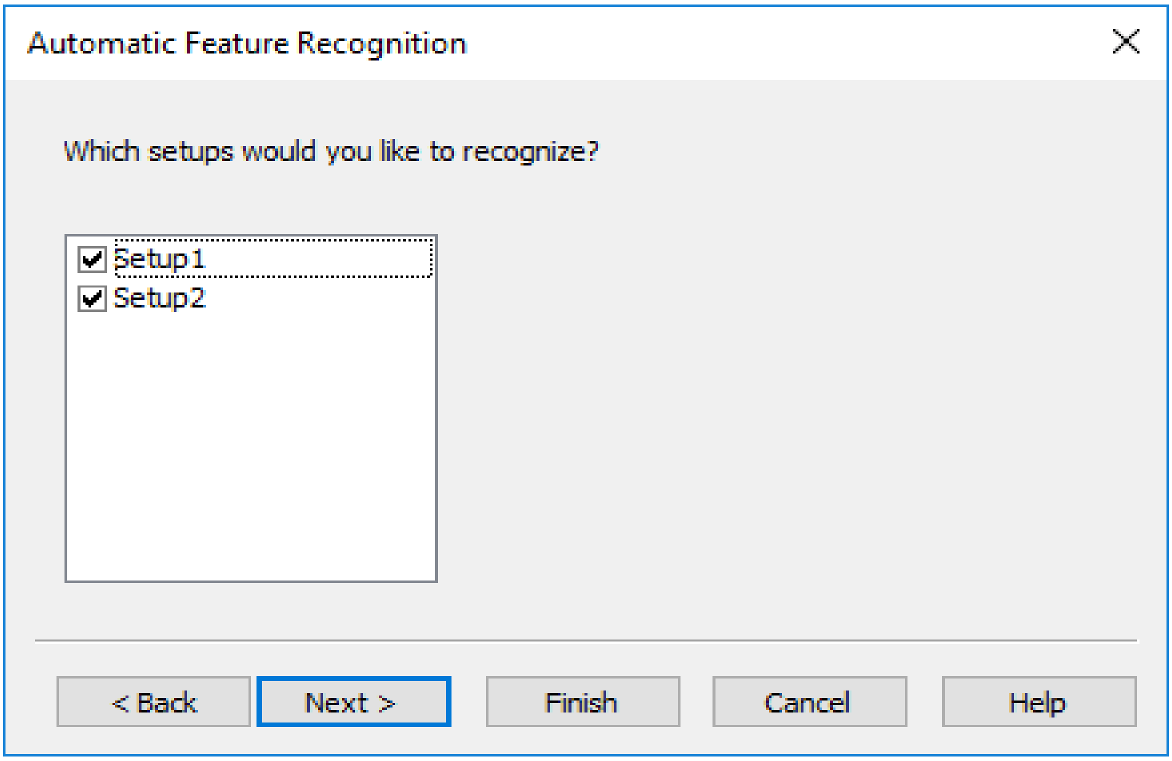
17 Go to **Construct** and select **Automatic Feature Recognition**.



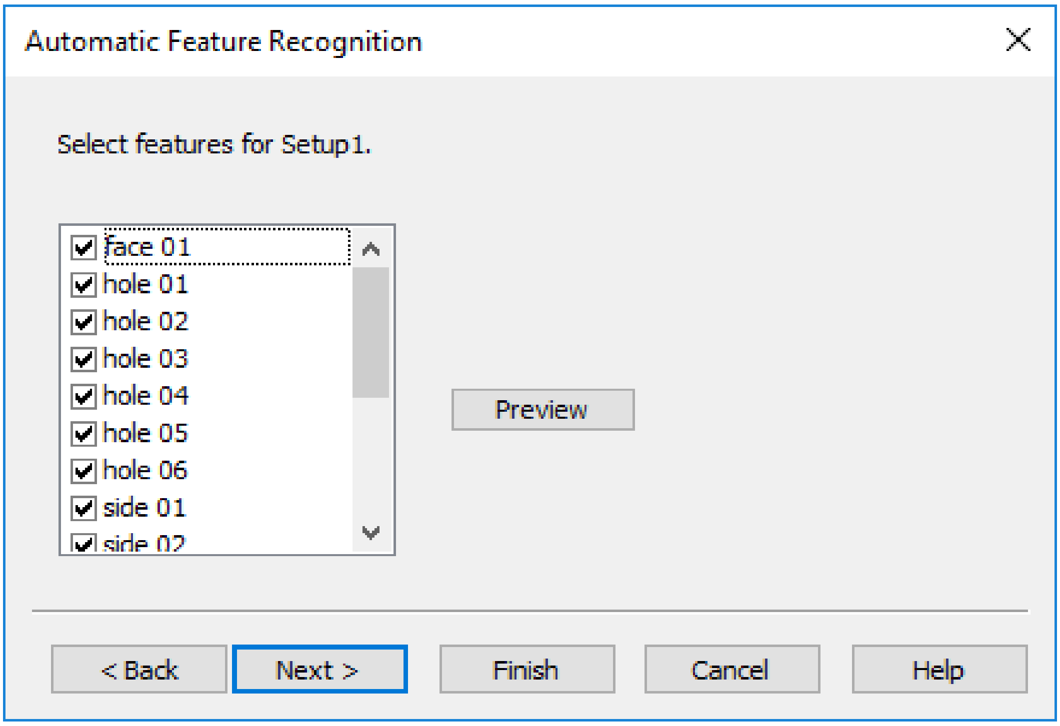


18 Set the **AFR Options, Verify** the Solid, Select **Next**.

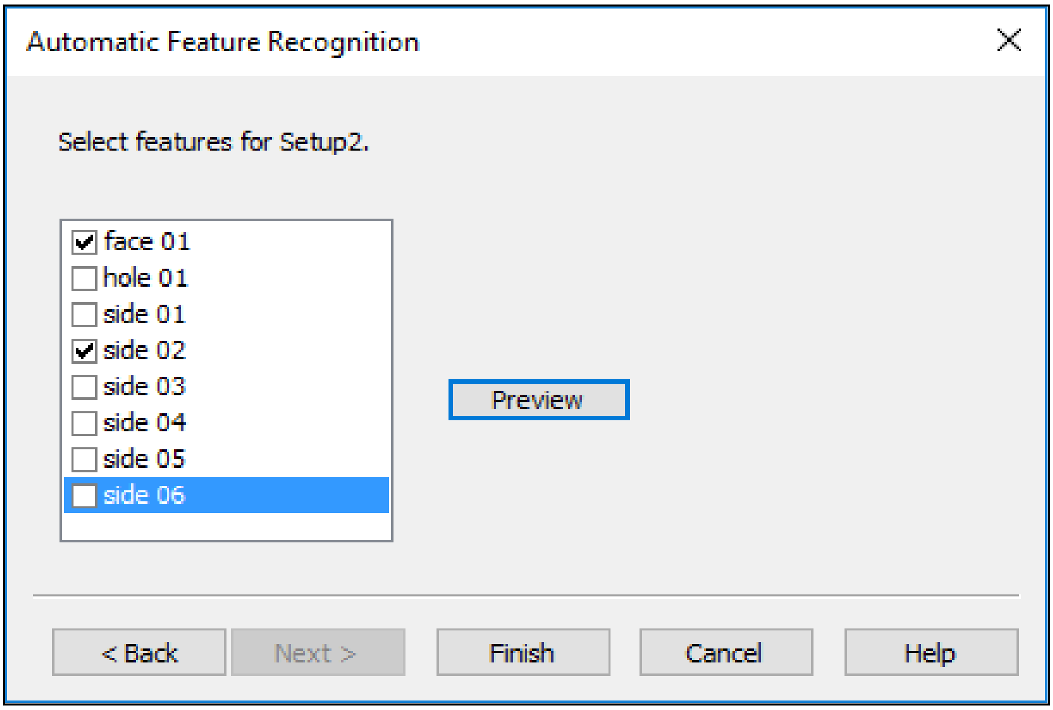
19 This will look at both **Setups** and machine both sides of the component.



20 Select **Next**.



21 Select **Next. Unselect** the following Features.



We now have duplicate machining. In other words we have operations that do the same machining from both sides. We can select each Feature and establish which machining operations we need to delete.

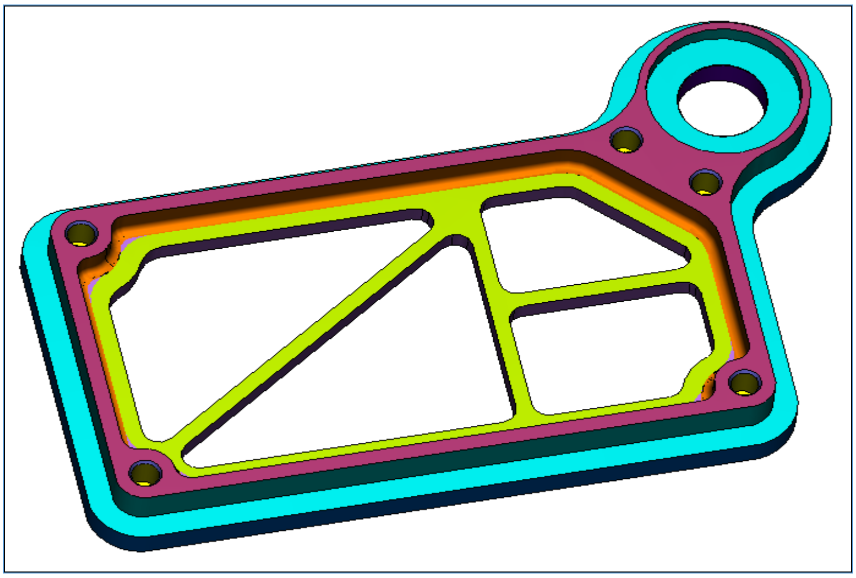
22 Unselect **hole1, side1, side3, side4, side5, side6**. Your numbering sequence may be different.

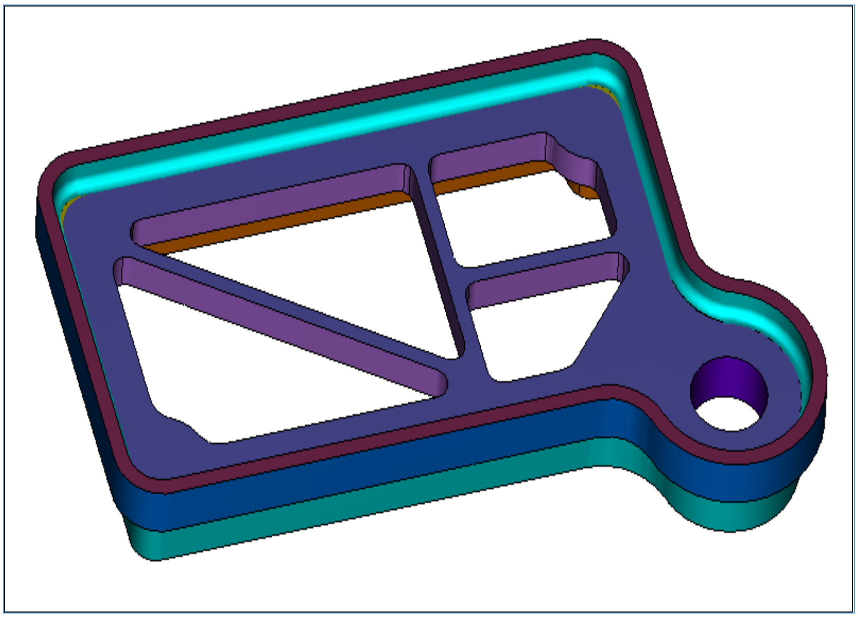
23 Select **Finish.**

24 Run 3D Simulation.

25 Press Eject or Stop on the Simulation toolbar. Make any changes then Re-run the **3D Simulation** to confirm that the part is finished.

Use existing curves to machine chamfers around each pocket.

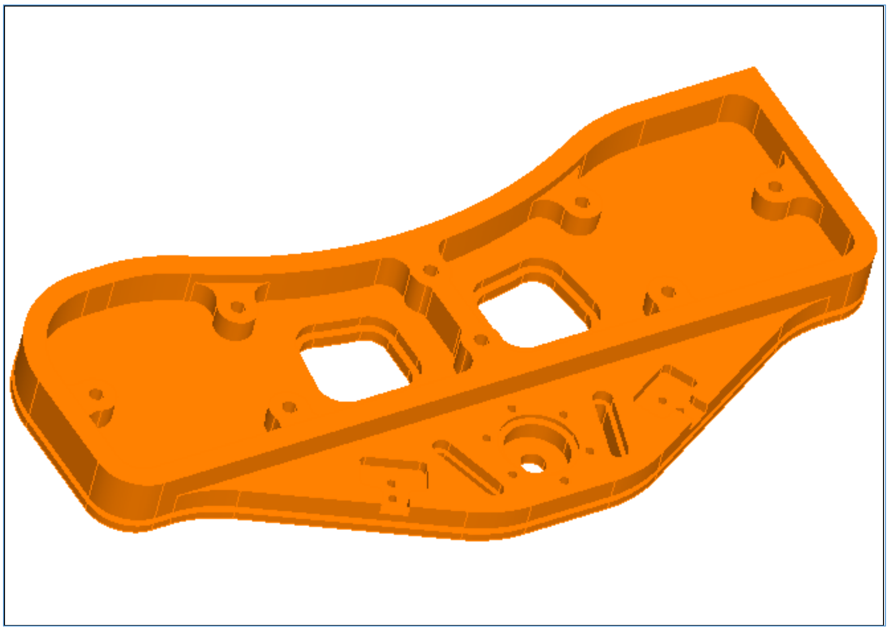




AFR Test. (Work through this on your own)

*The following example is a test to see if you have learned how to machine a complex 2.5D part complete using **AFR** and **IFR.***

*Import file **New 2.5D Part for AFR & IFR Rotated X & Z.x_t** from **C:\Training_Data\FeatureCAM Course Data 2017\Milling Files to import***



This file has been rotated in X & Z

*Your challenge will be to rotate the part so the longest edge will be parallel in X and the bottom and Top faces are Horizontal to the XY Plane or **Top View**. Use the Tools in the **Feature Recognition Wizard** to align the part as described earlier on in the chapter.*

Make sure you have created your Stock.

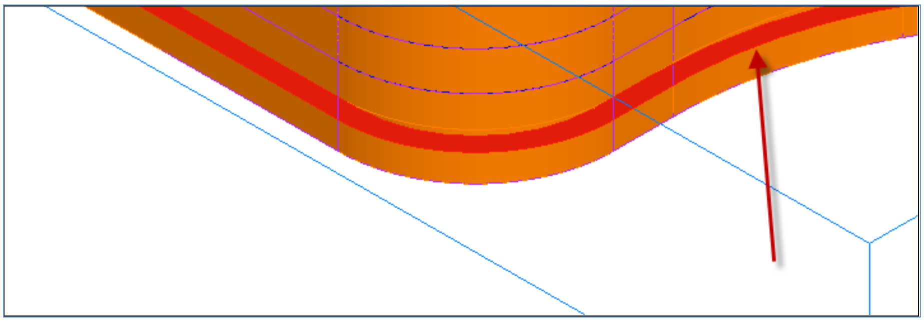
*There is machining on both sides of this part. Create Setups for **Op1 Setup1 & Op2 Setup2**.*

***AFR** will not recognise the groove around the outside. Machine this using **IFR**.*

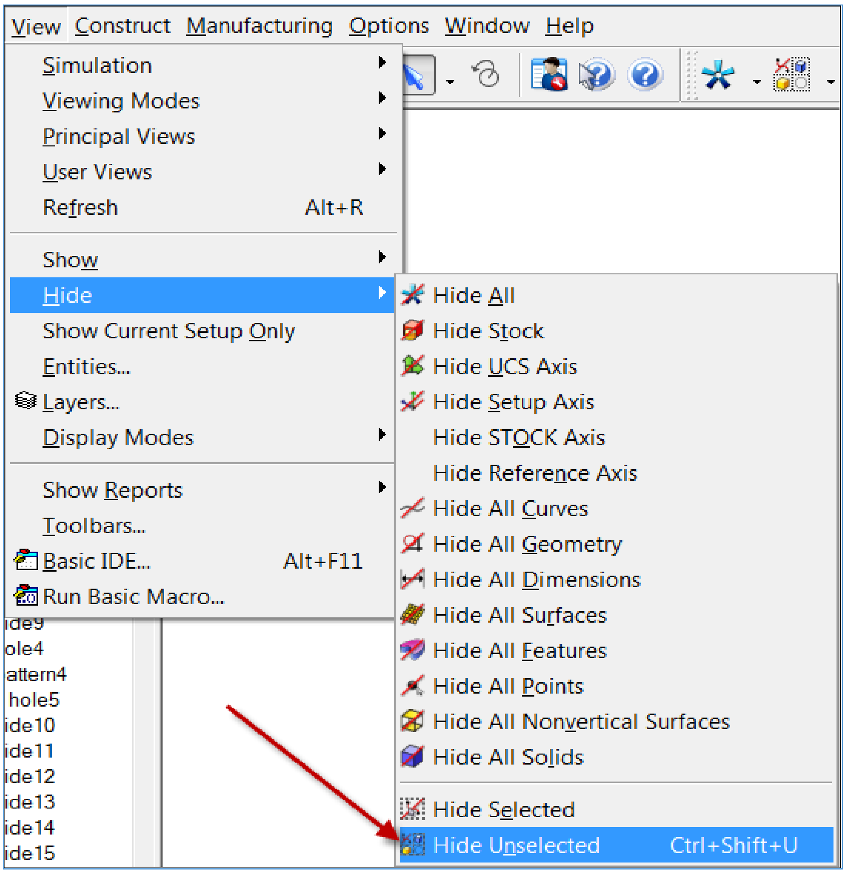
See the example below on how to machine the groove.

The following guide will help machine the groove.

1 After machining the part, **select** the lower bottom face of the groove as shown.



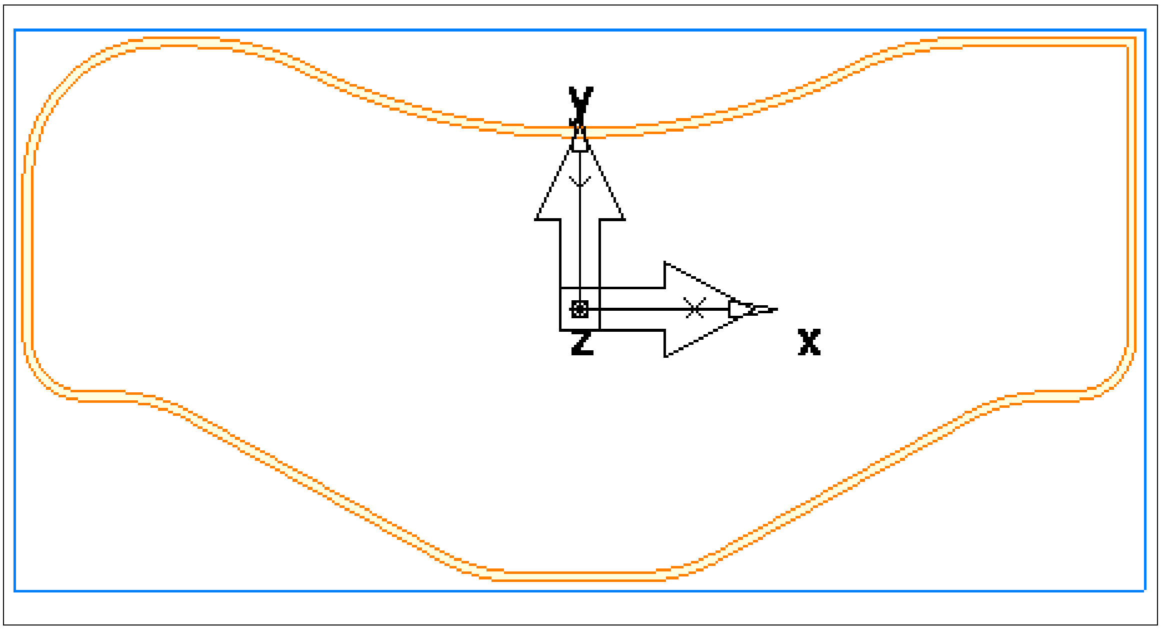
2 Go to **View>Hide>Hide Unselected**.



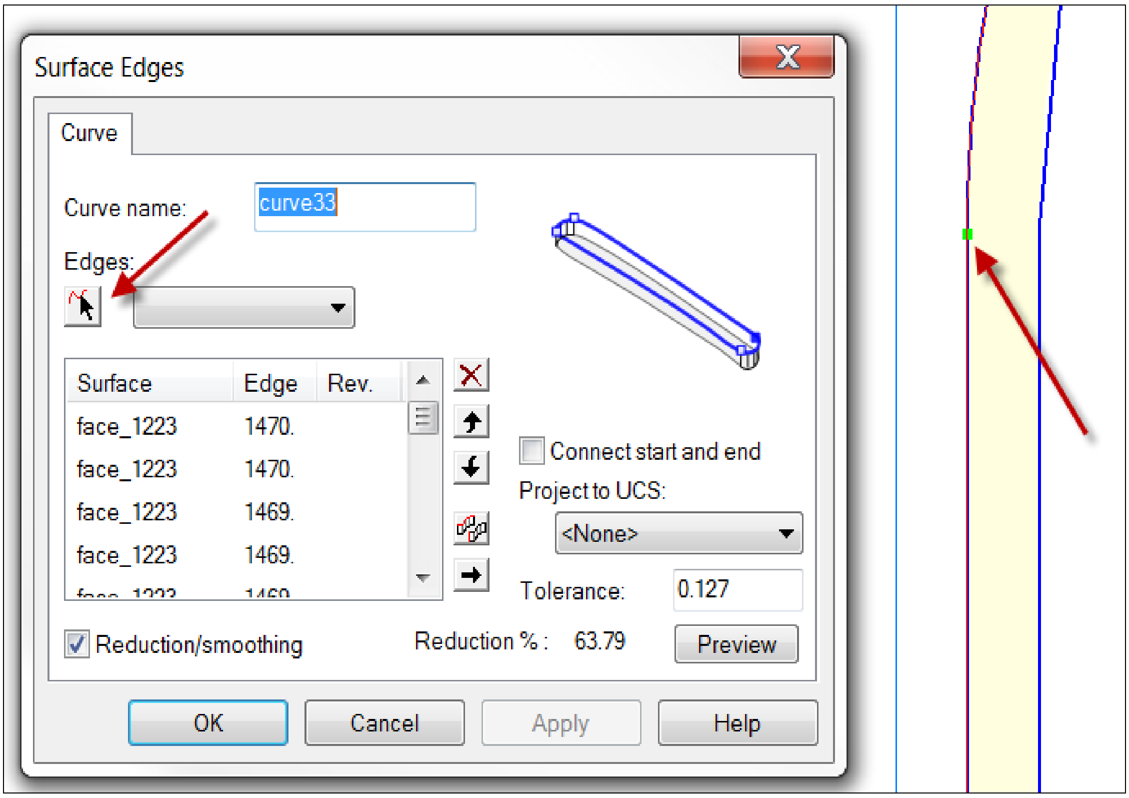
All that will be shown on the screen will be the face we selected earlier.

3 Click on the screen anywhere to deselect the surface.

4 Select **Ctrl+5** or Right click on the screen and select **Top.**



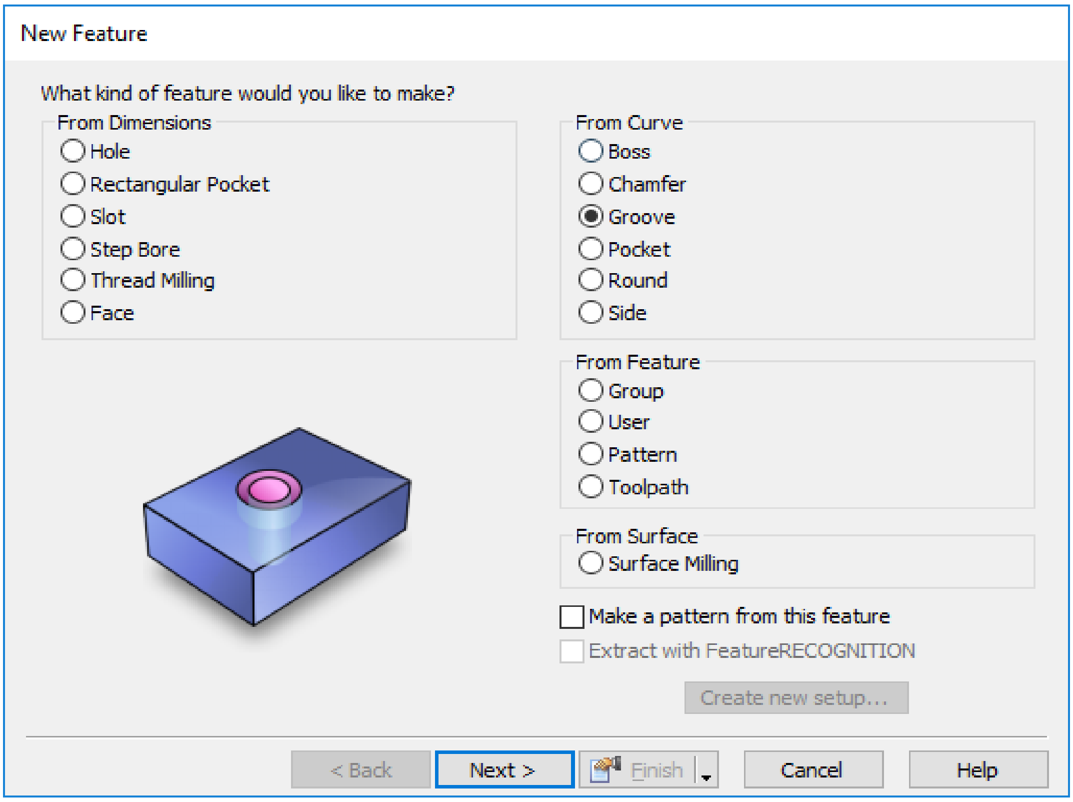
5 Go to **Construct>Curve>From Surface>Surface Edges.** Then select the **Edges** Icon and then double click on the outer edge as shown.



6 Select **Apply** and **OK**. We have now created our Curve all the way around our profile.

7 Select the visible surface and **Right click**. Select **Hide Selected**.

8 Select New Feature Wizard. Then select Groove from Curve.



9 Select **Next.**

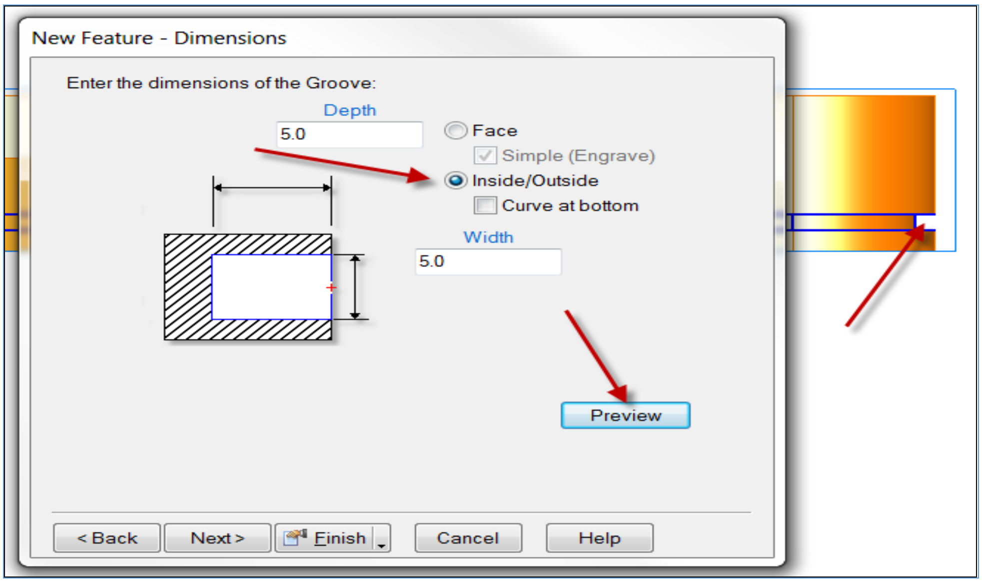
10 **Show** the Solid by selecting the **Ps-Solid** in **Part View** and **Right Click** and select **Show Selected**.

11 Select **Curve.** Select **Next.**

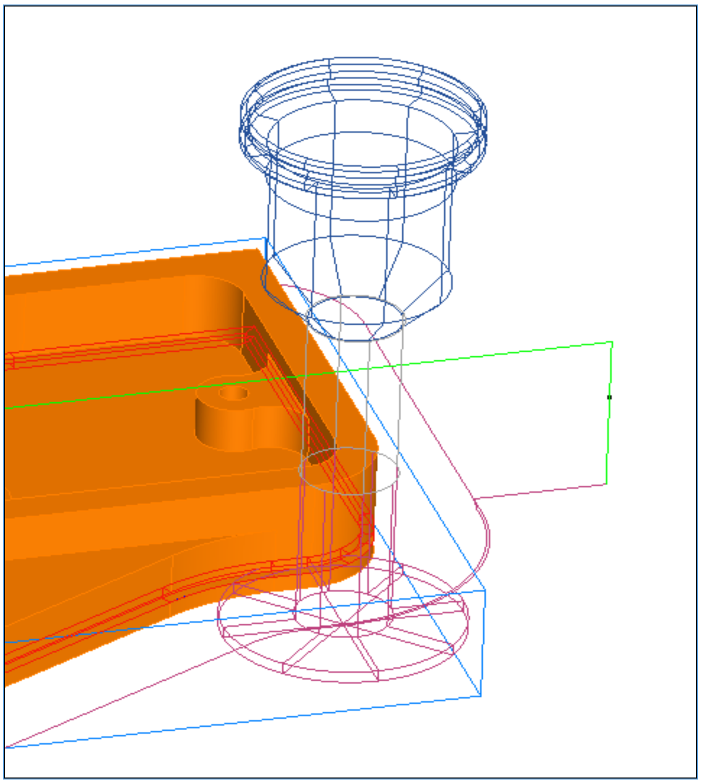
12 In the **New Feature - Location** menu type in **2.5mm**

13 Select **Next**.

14 Select the **Inside/Outside** radio button. Select **Preview**. Select **Finish.**



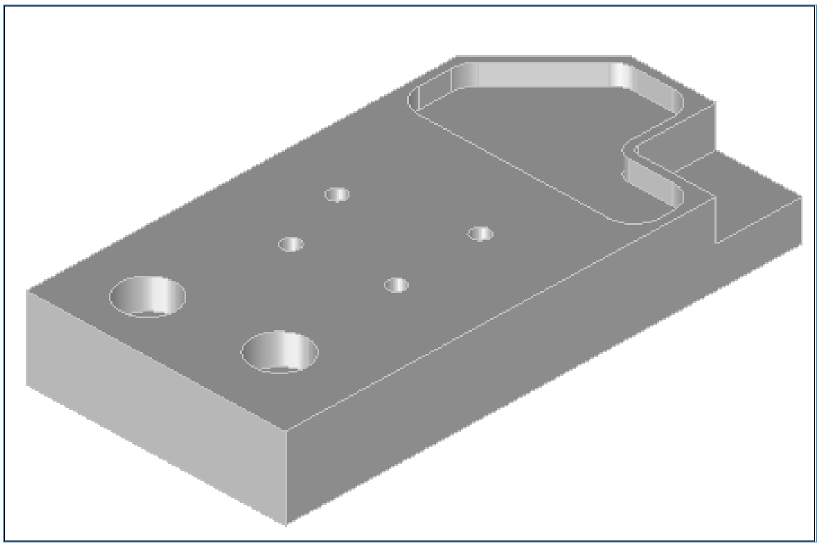
15 You can see from the image below that the groove is now being machined.

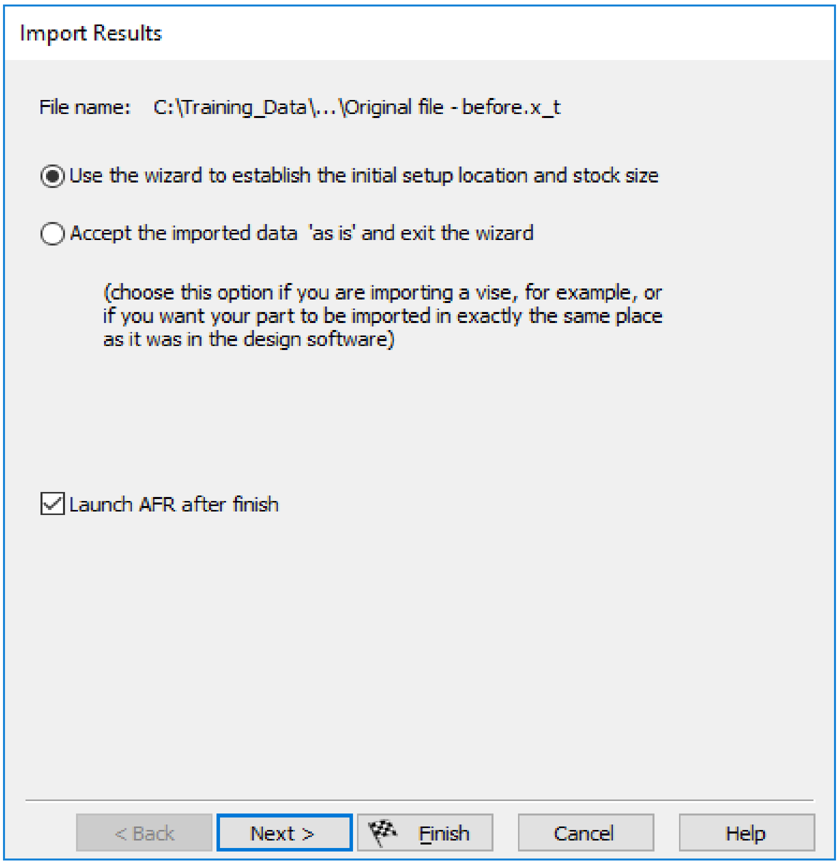


AFR Example 3 (Feature Re-recognition)

(Work through this on your own)

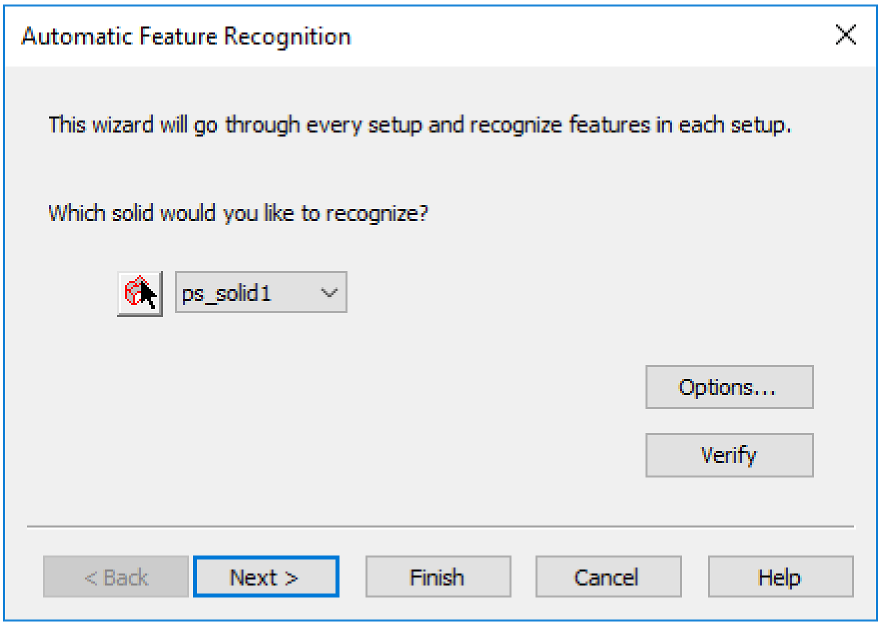
1. In the New Part Document window select **Milling Setup**, **Millimetres** then press **OK**.
2. Find the file in **C:\Training_Data\FeatureCAM Course Data 2017\Milling Files to import** from the files of type pull down menu select **Parasolid (*.x_t,*.x_b).** Select the file named **Original file - before. X_t** then press the **Open** button.
3. Select the **Launch AFR after finish** checkbox then press **Next** four times.



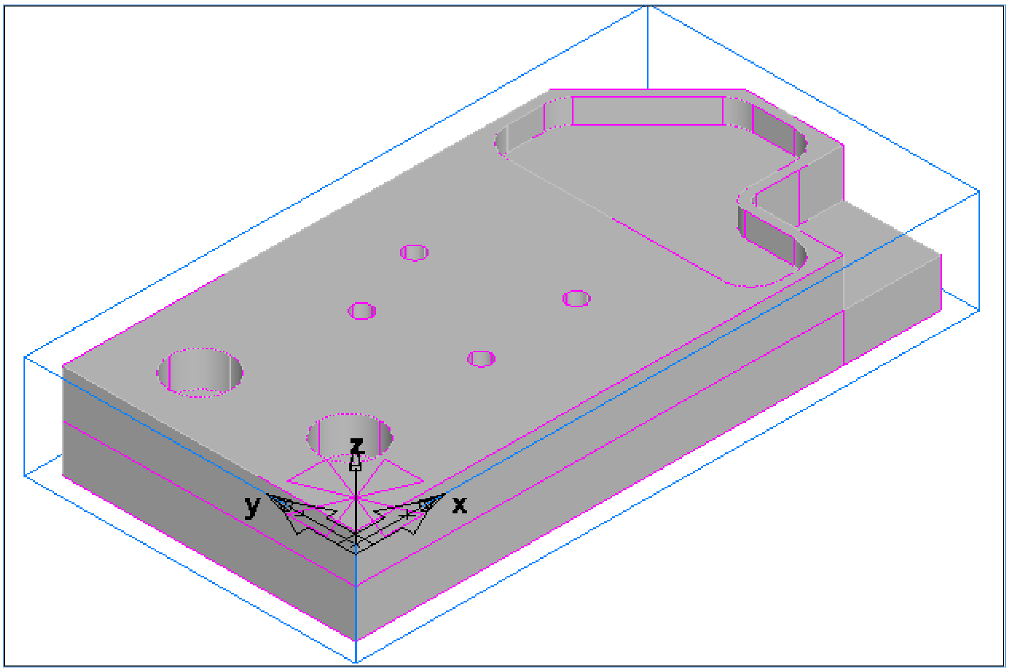


4 On the **Stock Dimensions** page check the **Compute stock size from the size of the part** and Zero the extra material fields. Then select **Finish**.

5 From the **(AFR) Automatic Feature Recognition** press **Next** then **Finish**.

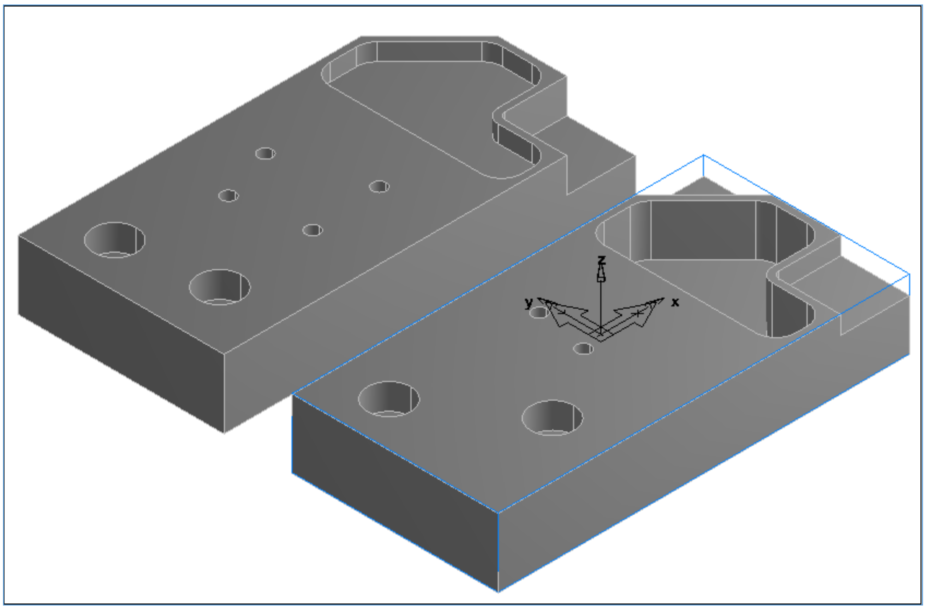


6 **Feature recognition** completes the part by identifying all the features in the setup. Select **Next** then Select **Finish**.



7 The next step is to import the similar solid model that has been revised into the same document as the first and perform **Feature Re-Recognition**.

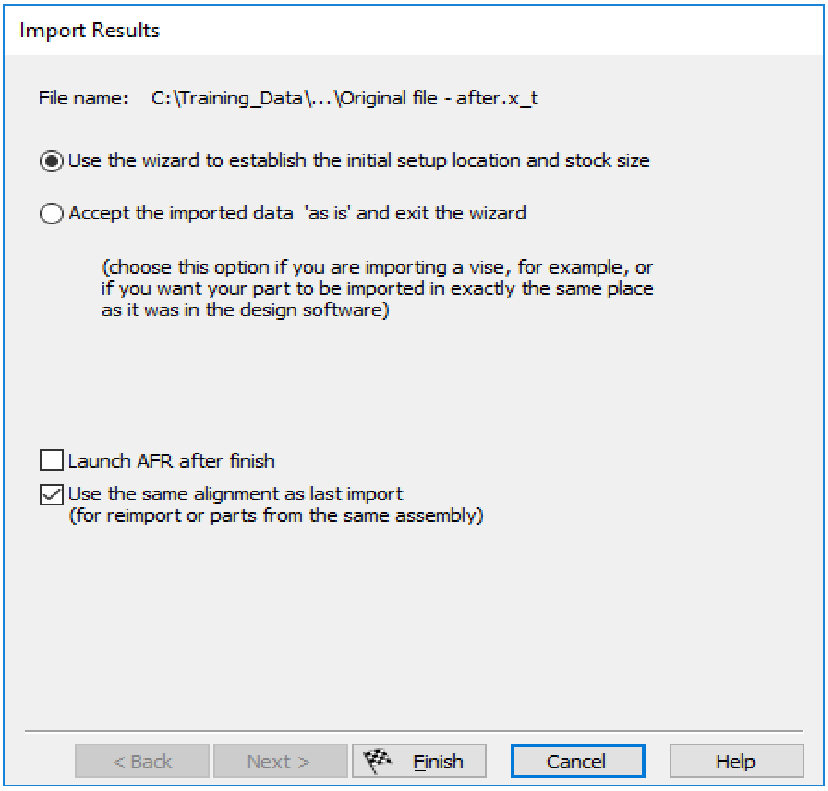
8 From the file menu click on File then **Import.** From file location **C:\Training_Data\FeatureCAM Course Data 2017\Milling Files to import.** Select the file named **Original file - after. X_t**. Then select Open.



It is only shown like this for clarification to show you the difference between the two models.

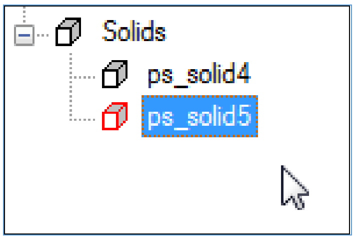
*Upon closer examination there should be a noticeable difference between the two solids. Some holes are removed and in a different location, as well as the depth and dimensions of the side and pocket features recognized using **AFR**.*

9 On the **Import Results** window select the top radio button, **Uncheck** Launch **AFR** after finish and Check **Use the same alignment as last import**. Select **Finish**.

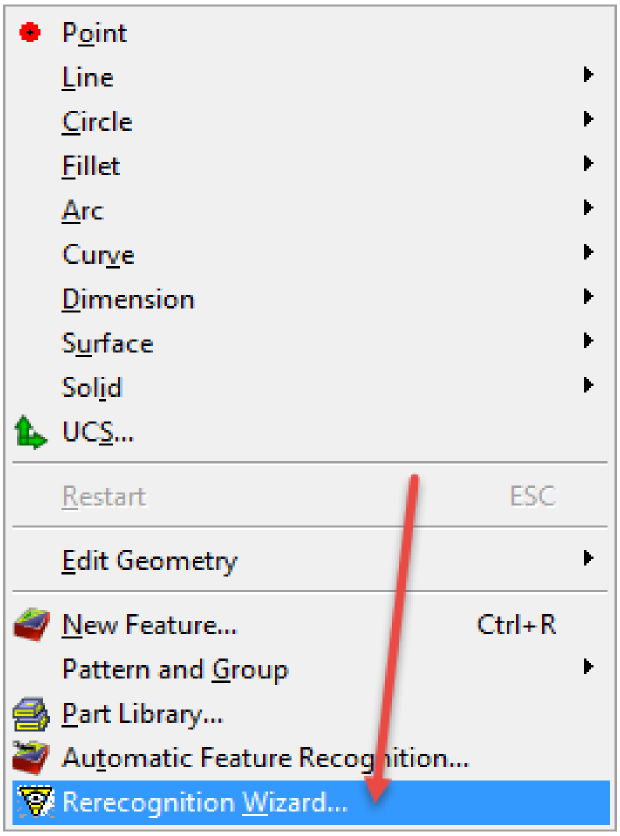


They will merge together.

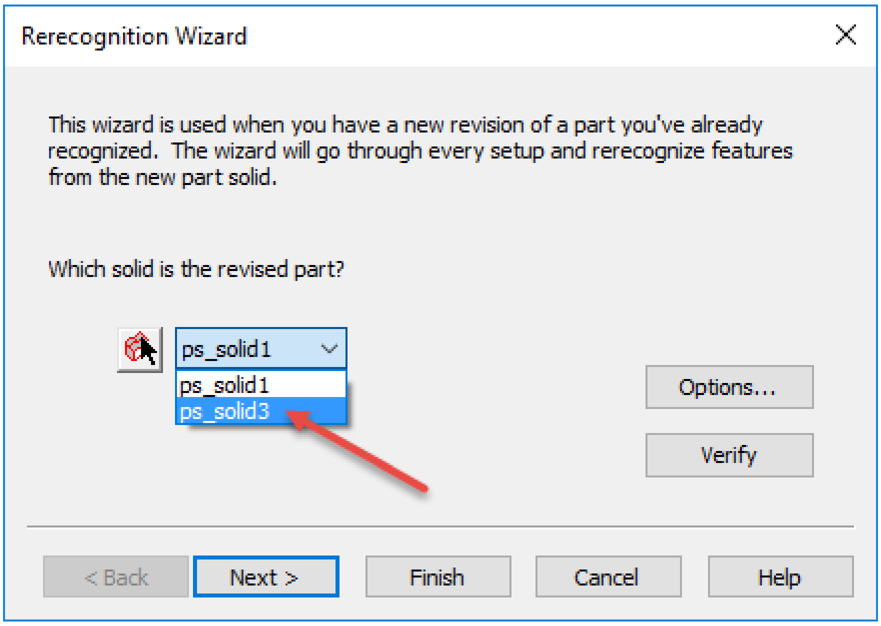
10 In the **Part View** on the left hand side under Solids click the plus sign to expand if necessary then toggle between each solid that is highlighted. When the solid name is clicked in the part view.



11 Click on **Construct** from the menu Bar above then select **Re-Recognition Wizard** at the bottom of the menu. Select **Finish.**



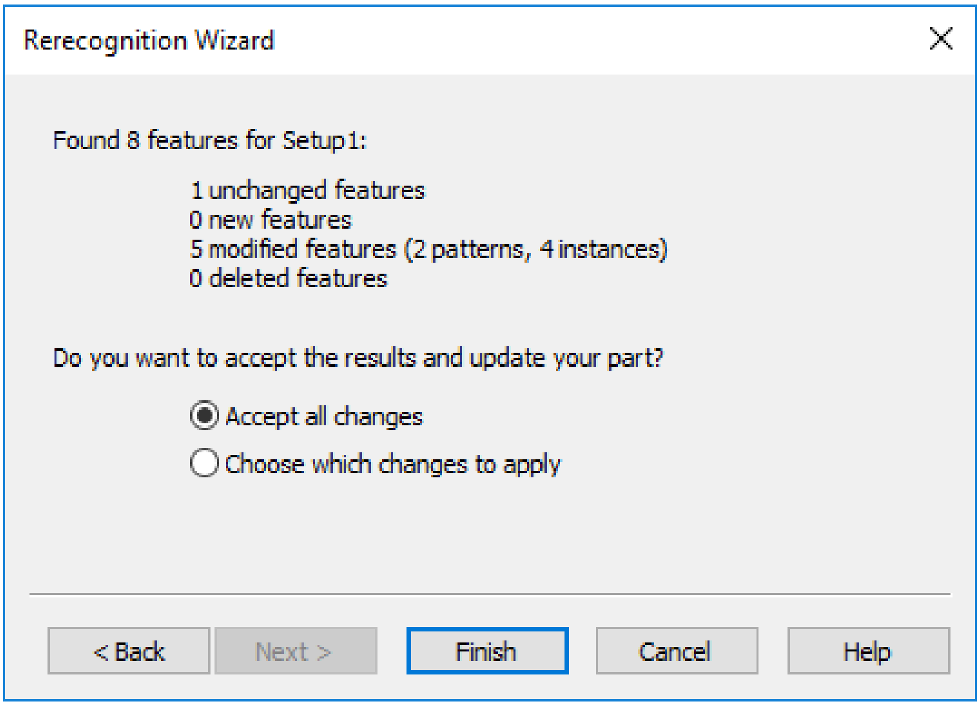
12 The new model overrides the old one but is still visible.



13 From the solid pull down menu select the **second**, last imported, solid in the list then press **Next** twice.

14 The **Re-Recognition wizard** will show you what has changed. This is very useful so the programmer is not required to completely reprogram the part upon design changes.

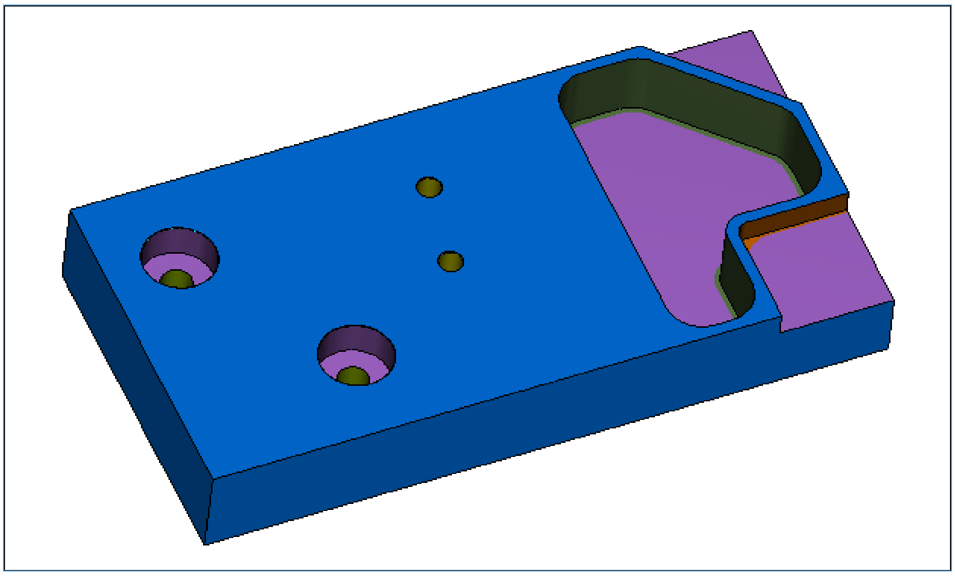
*This method is only effective for features that have been programmed using **AFR** or **IFR**. Features created using curves will not be Re-Recognized upon changes to the model.*



15 Depending on how your **AFR Options** are set you can get Varying results.

16 It is suggested you use the same **Options** for both parts.

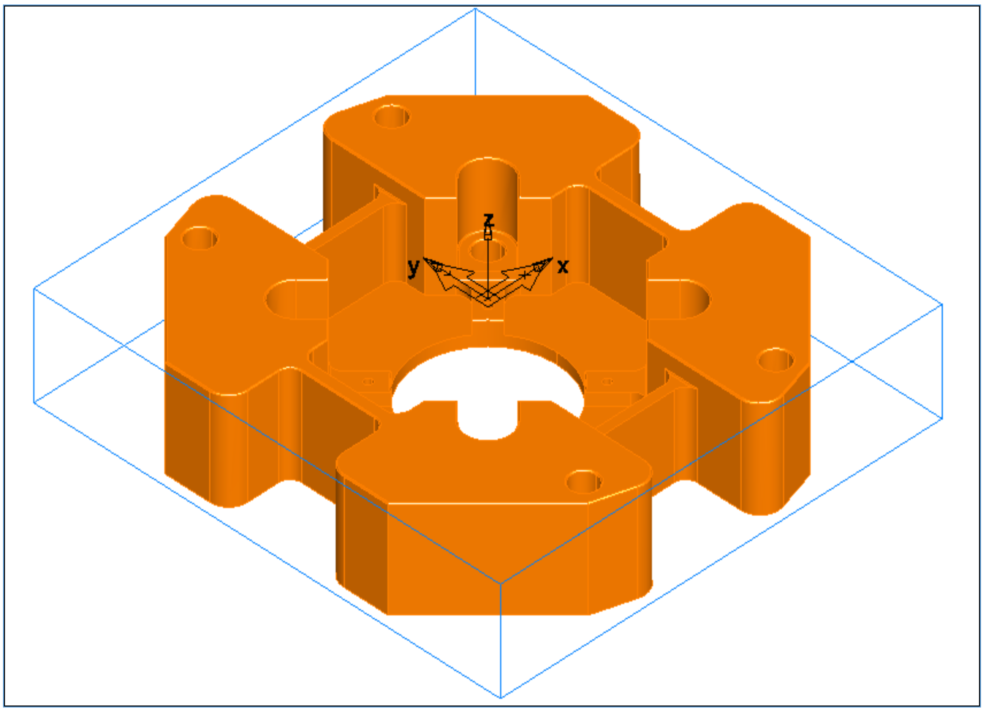
17 Run **3D Simulation**.



Chain Feature Curves

Lines, circles, and arcs typically represent the shape of a part. To use a sequence of lines and arcs to create a feature, you must chain them into a curve. Chaining is the primary way of creating curves by connecting pieces of geometry. In many cases you do not need to trim the geometry before creating a curve; chaining works better with smooth, tangent-continuous paths because these paths are more conducive to manufacturing.

1 Import solid model Chain Feature Curves from directory **C:\Training_Data\FeatureCAM Course Data 2017\Milling Files to Import\Chain Feature Curves. X_t**



2 Unselect Launch AFR after Finish.

3 Use the wizard to establish the initial setup location and stock size.

4 Allow 5mm all-round 1mm on top face and zero on the bottom face.

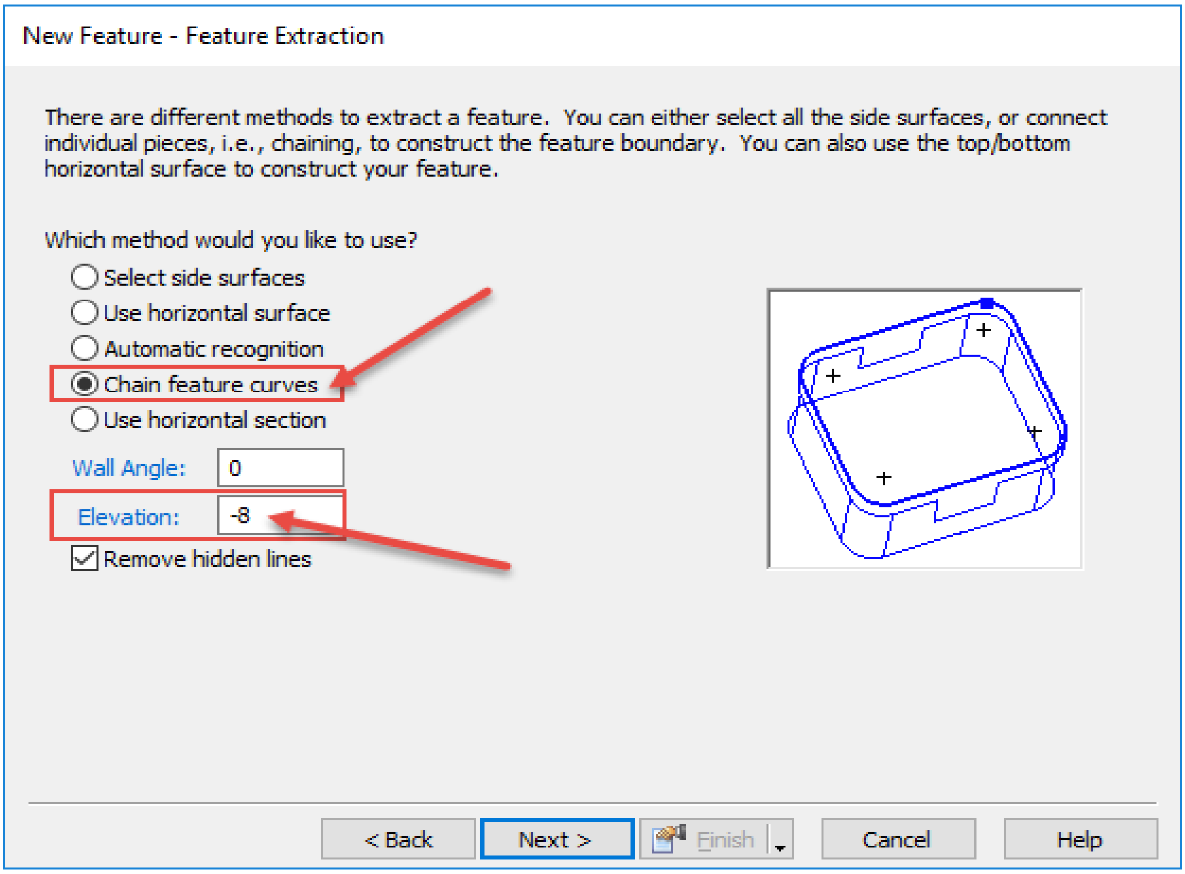
5 Setup is centre **+ and Z Offset is Z-1**

6 Create a Face Feature.

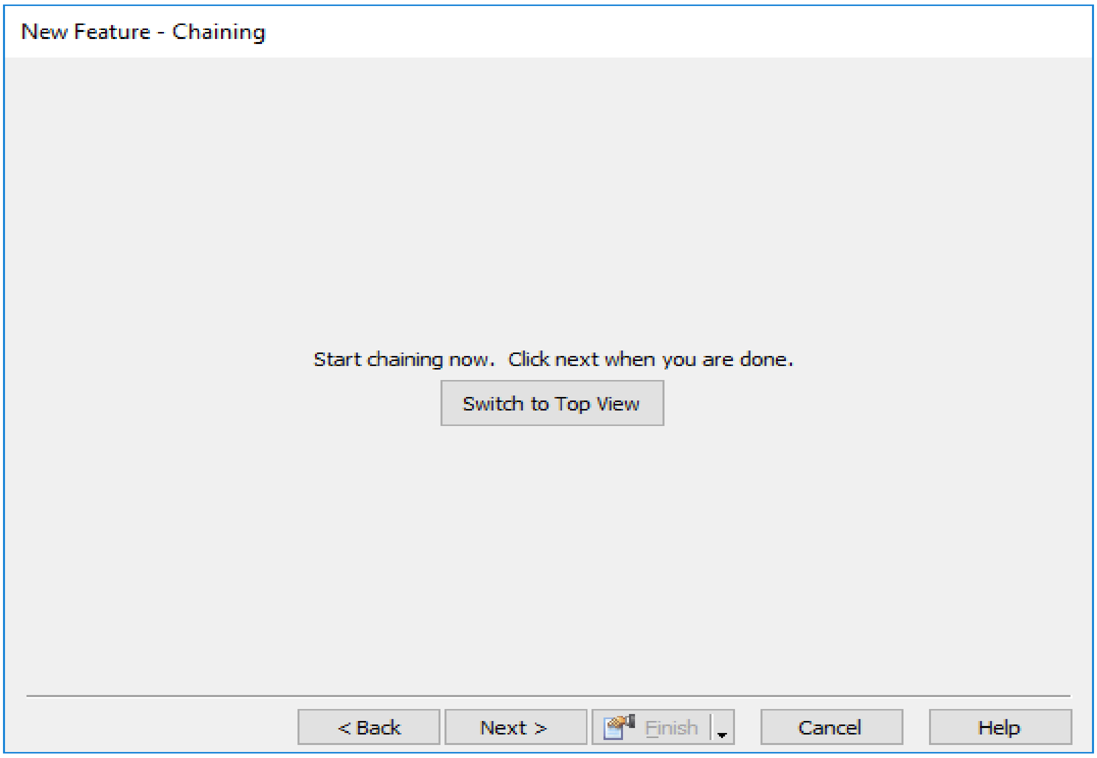
7 Create a Side Feature using Extract with Feature Recognition.

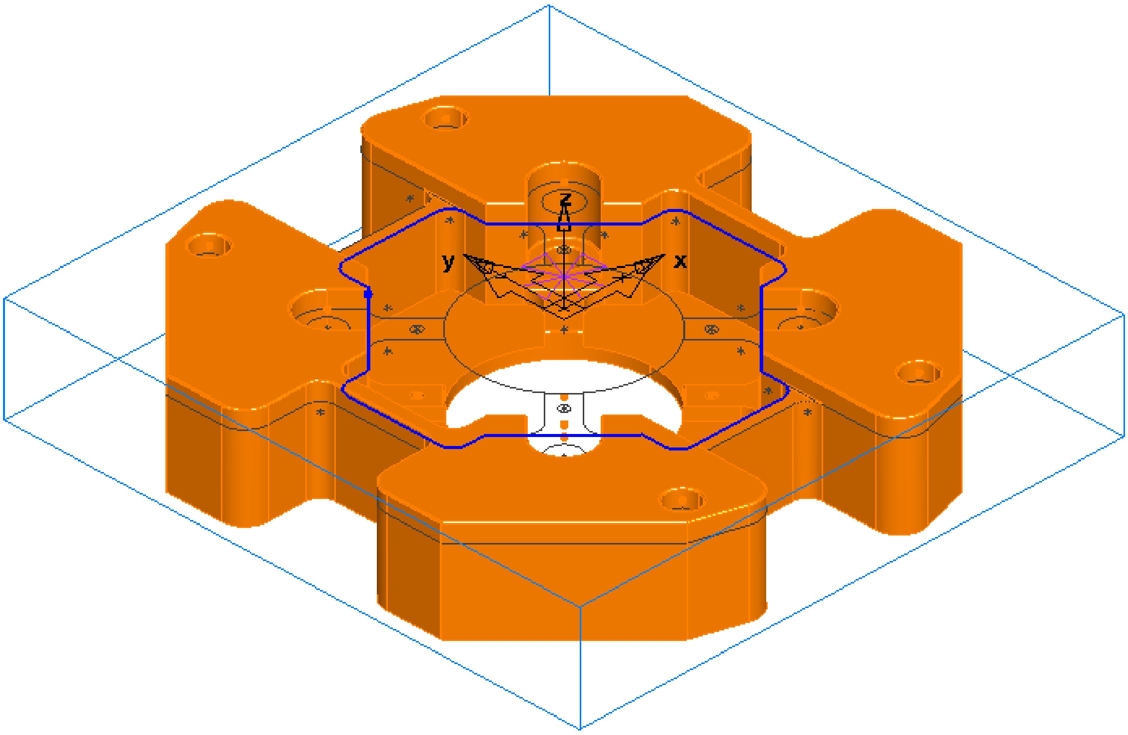
8 Orientate the view so you are looking from the Top. (Ctrl+5)

9 Select Chain feature curves. Select Elevation -8mm. Select Next.



You will see a form that asks you to start chaining now. Work your way around until you have completed the complete profile.

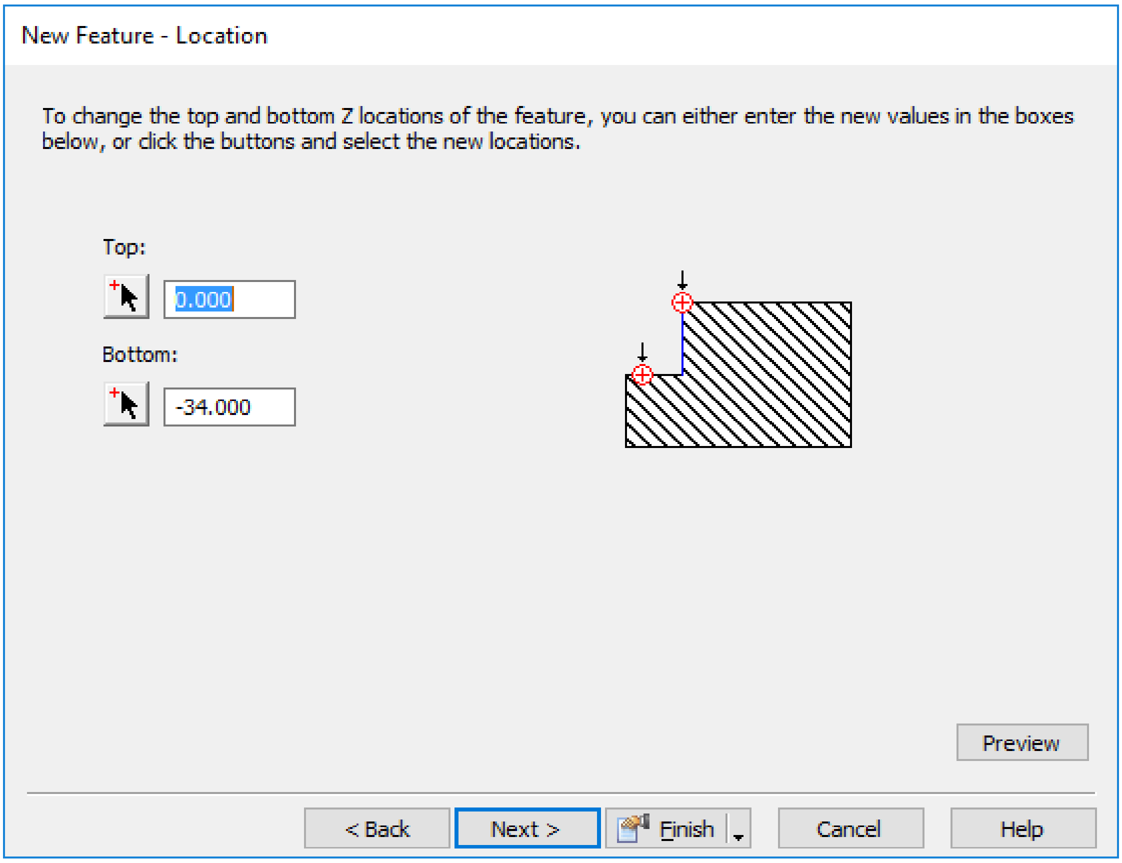


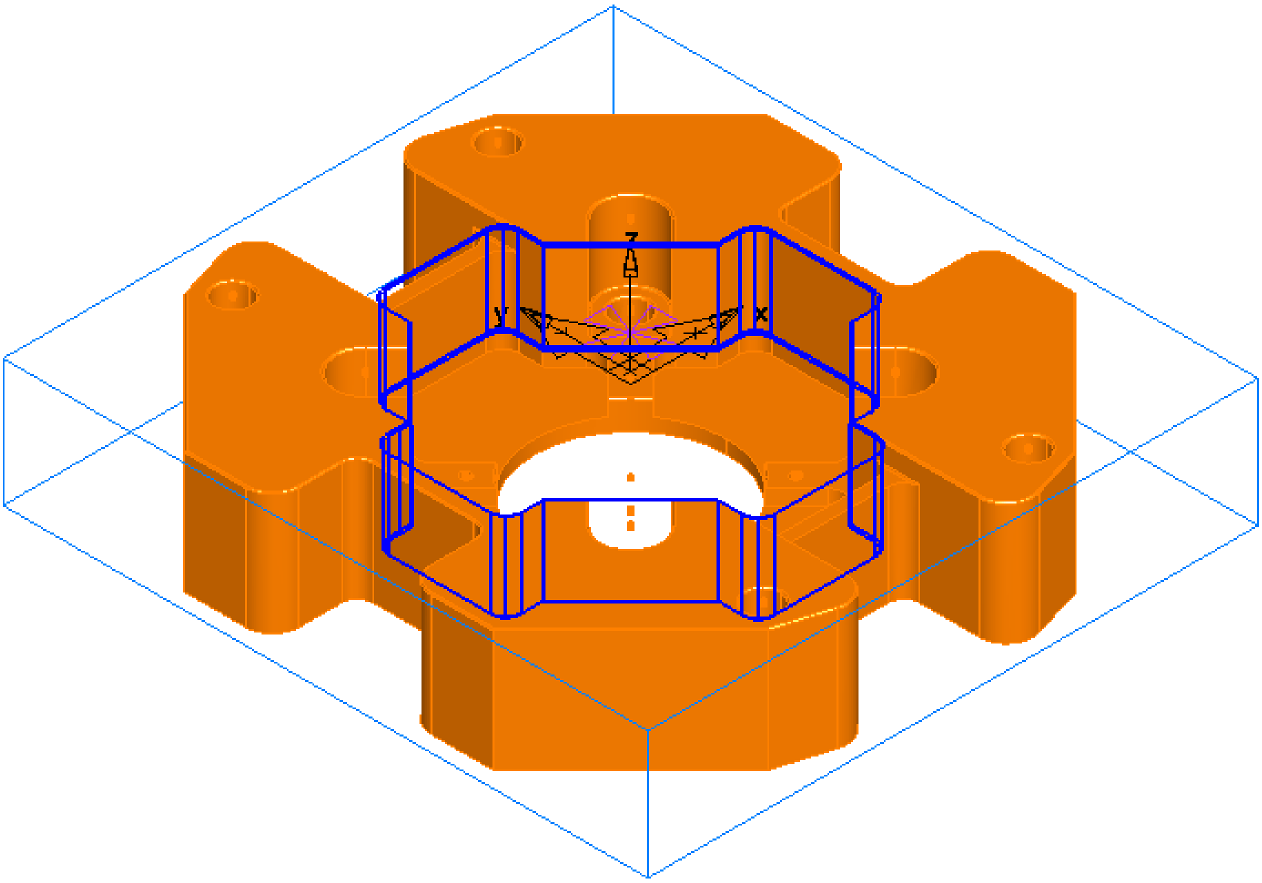


10 Select **Next** which will give you the option to select the **machining side**.

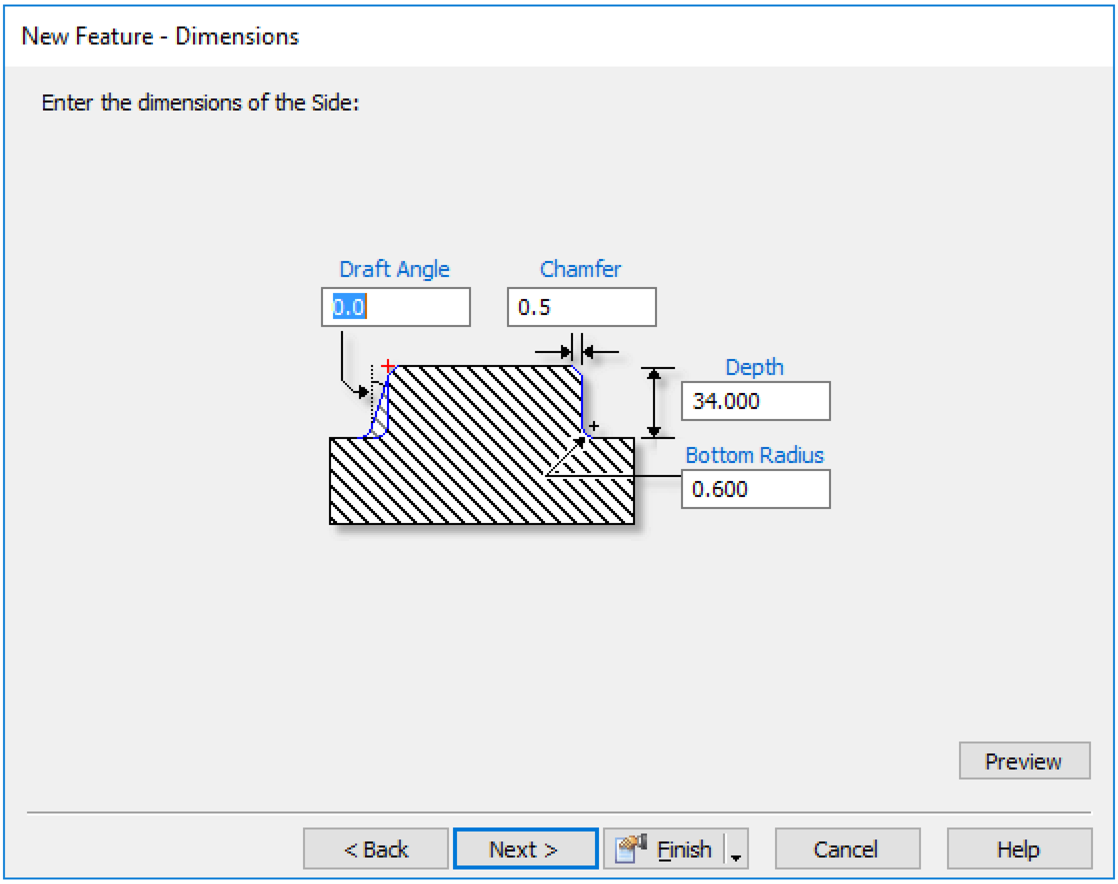
11 Select **Next** to set **Top** and **Bottom** Z locations.

12 Top should be zero and Bottom should be **-34mm** enter this information or select the model to extract the information.

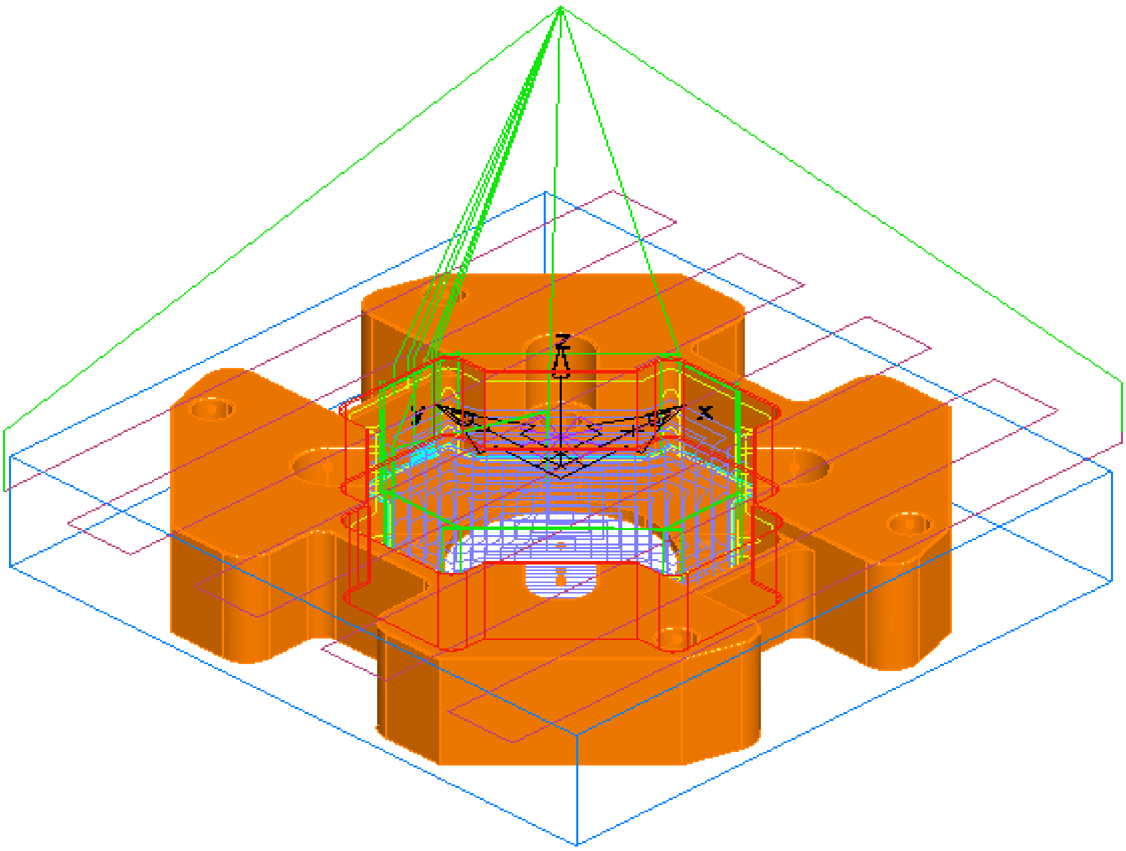




13 Select **Next**.



14 Change to the following settings and then select **Finish**.

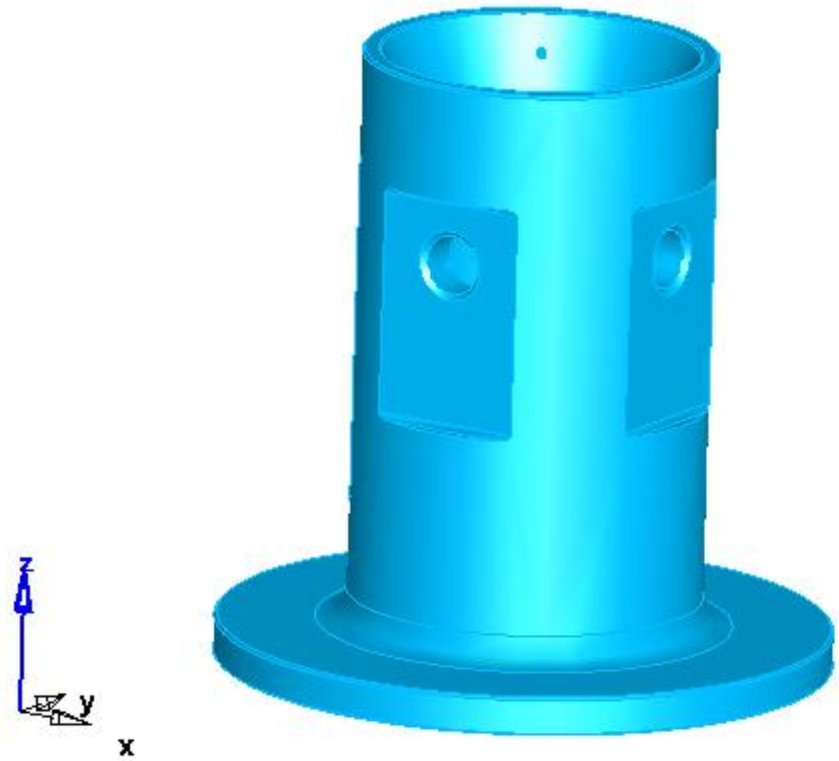


15 Using the same techniques work through this example on your own until all the features are machined. Create a **Hole Feature** to machine the holes using **Feature Recognition**.

Milling Flats and Holes on a finished turned part

Although we are moving towards a complete solution on new Turn-Mill centres. It is still a common method of first turning a part and then setting this up on a 4 axis milling machine to machine flats and holes as a second setup. The following example will take you through this process in FeatureCAM.

1 Please import the following model **FC XZCY Axis Part. X_t** into a Milling Setup. Import **from C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import.**



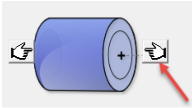
You will notice that the part is aligned in the Z Axis. This is correct as we need to extract the turned geometry from the solid model and this only works whilst the part is in the Z Axis. We will rotate the part and Geometry after we have extracted the turned geometry.

2 Make sure you unselect Launch AFR after Finish.

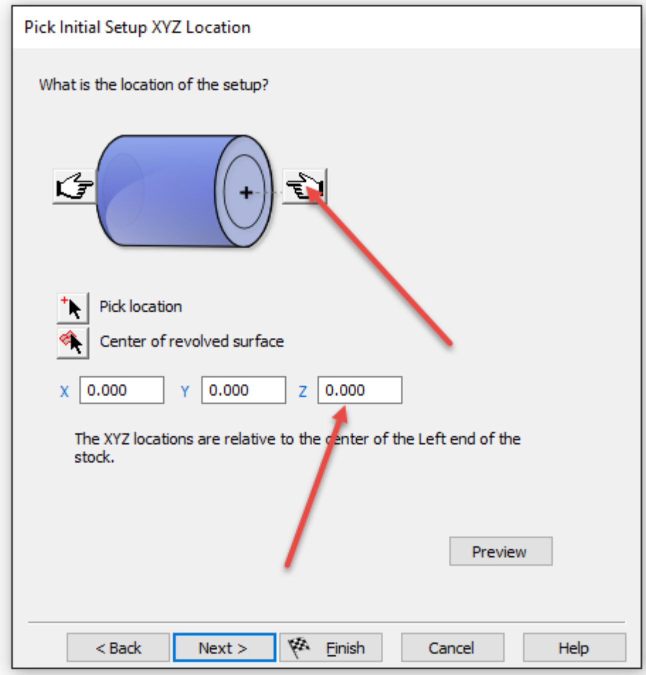
3 Please select **Next** until you get to the **Stock Type** menu. Please select **Round** and **Z Axis**. Select Next.

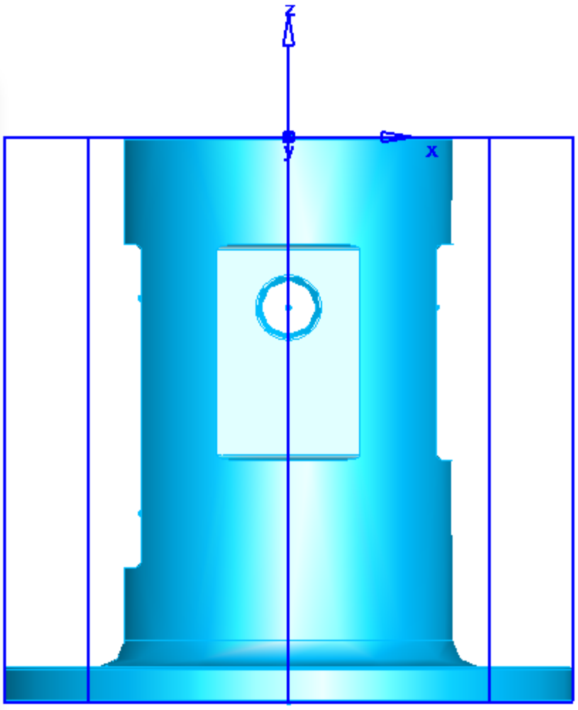
If the part is not on centre please select ***Centre of revolved surface*** *and select a cylindrical surface on the solid model. This will align the part to X0, Y0.*

4 Select **Next** and go to **Stock Dimensions**. In this instance set all of the extra stock sizes to Zero. Select Next.

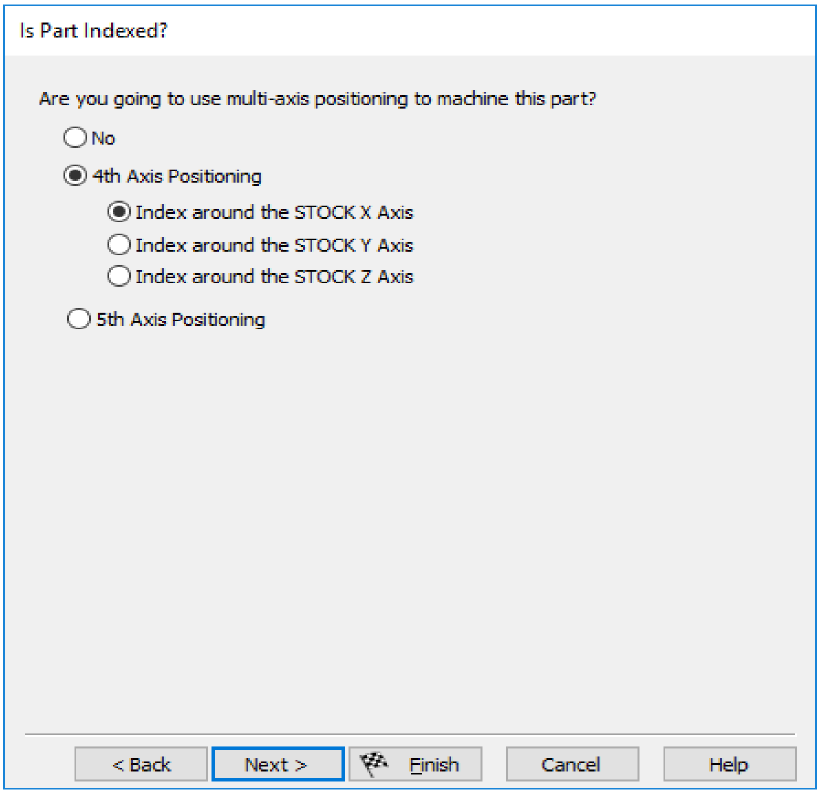
5 Select the **Right Hand**  this will reset the setup location to the end of the part.

6 The image below shows the part with the setup at the end of the part.





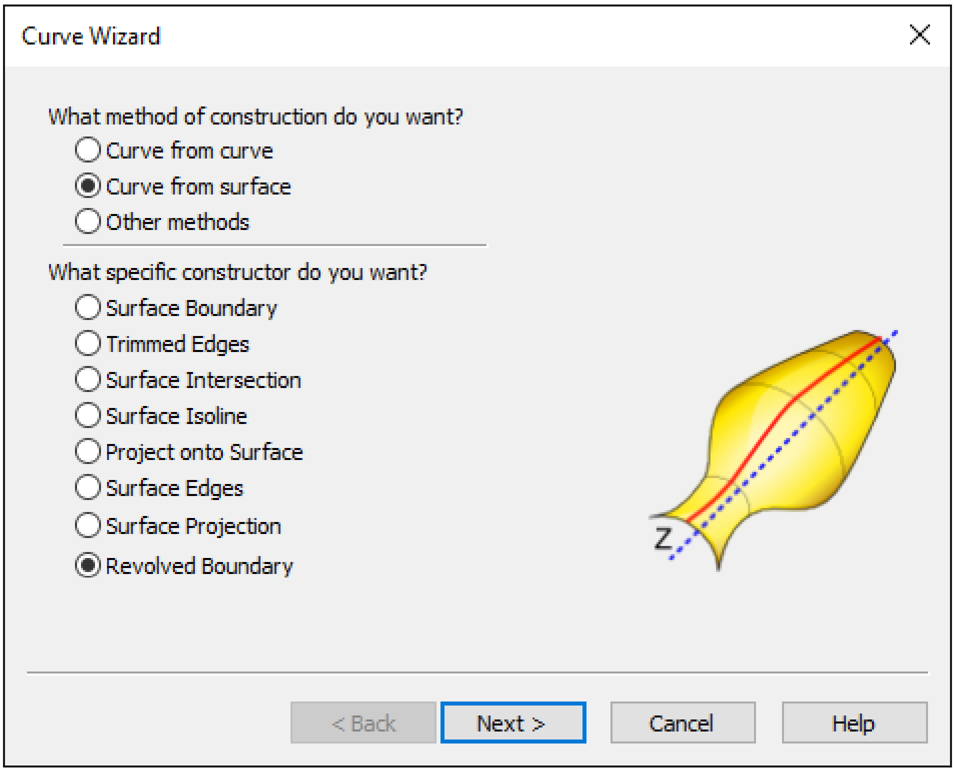
7 Please select **Next**. This will take you to the next menu. **Is Part Indexed?** Select the following.



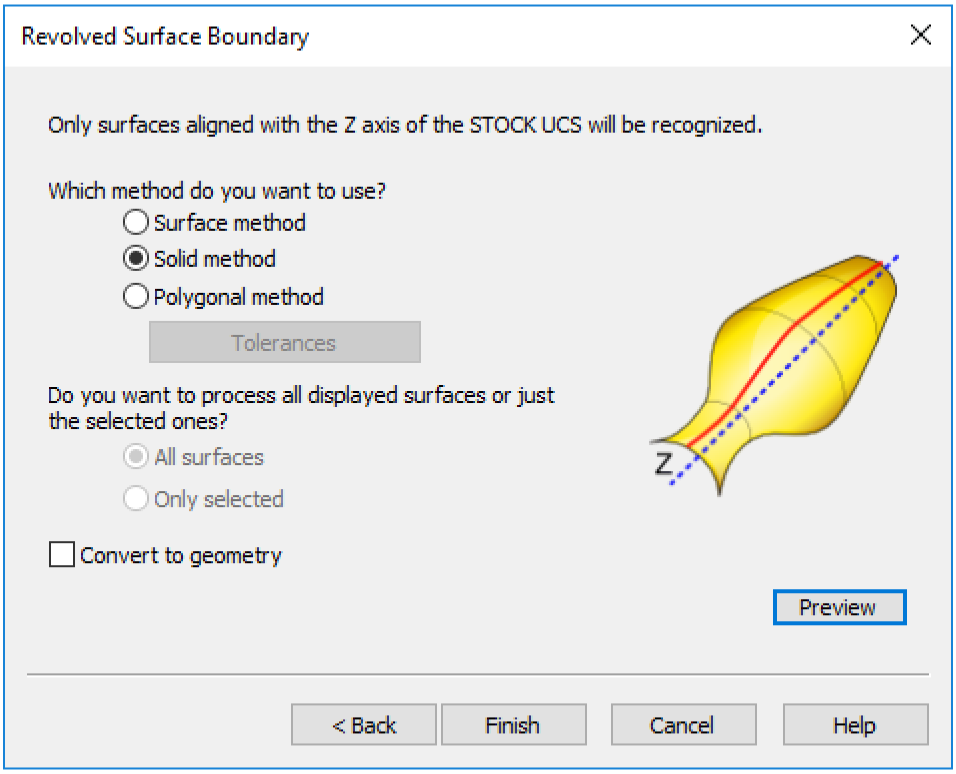
We are now going to extract a Revolved surface boundary.

8 Select the Curve Wizard icon 

9 Then select Curve from surface and Revolved Boundary.



10 Select **Next**. Select the following.



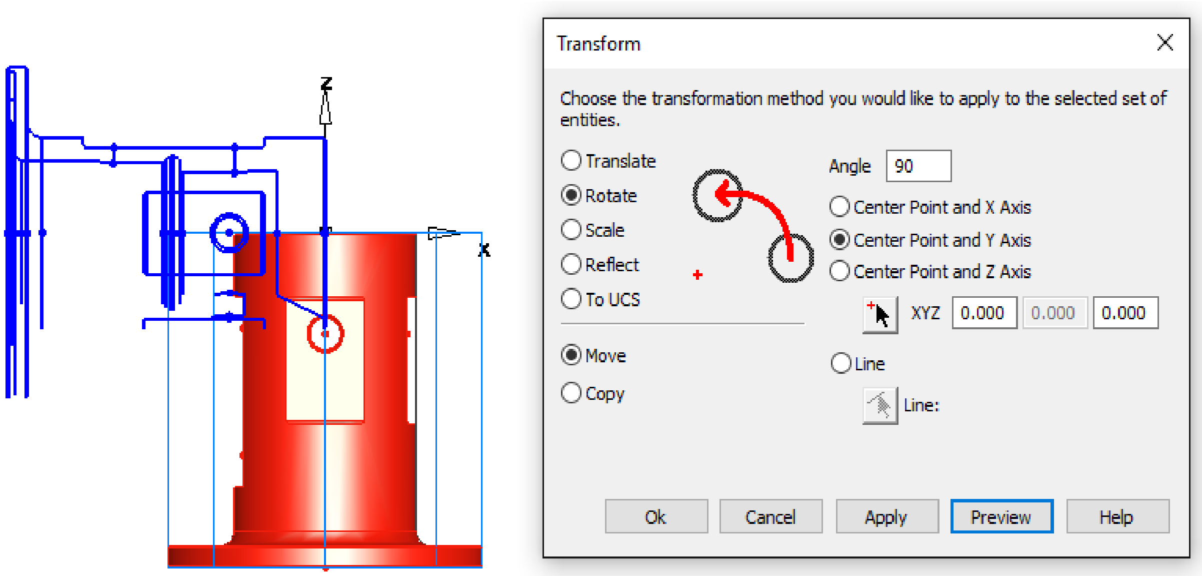
*If the part is complicated then you may have to reduce the tolerance to **0.05** within the **Tolerances** menu.*

11 Select **Preview**. Select **Finish**. This has now produced a Curve profile or section of the part.

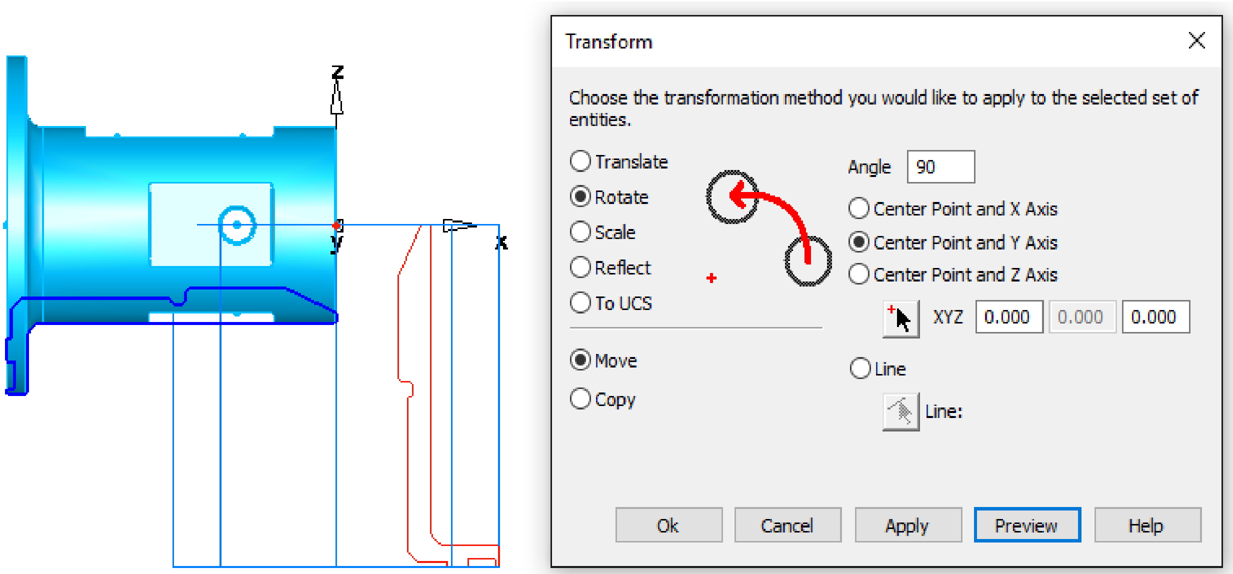
12 We now need to rotate our **solid model** and **Curve** so it is in alignment with the **X Axis**.

13 Select the **solid model** in **PartView**.

14 Select **Edit/Transform**. Then select the following. Select **Apply** to action the command.



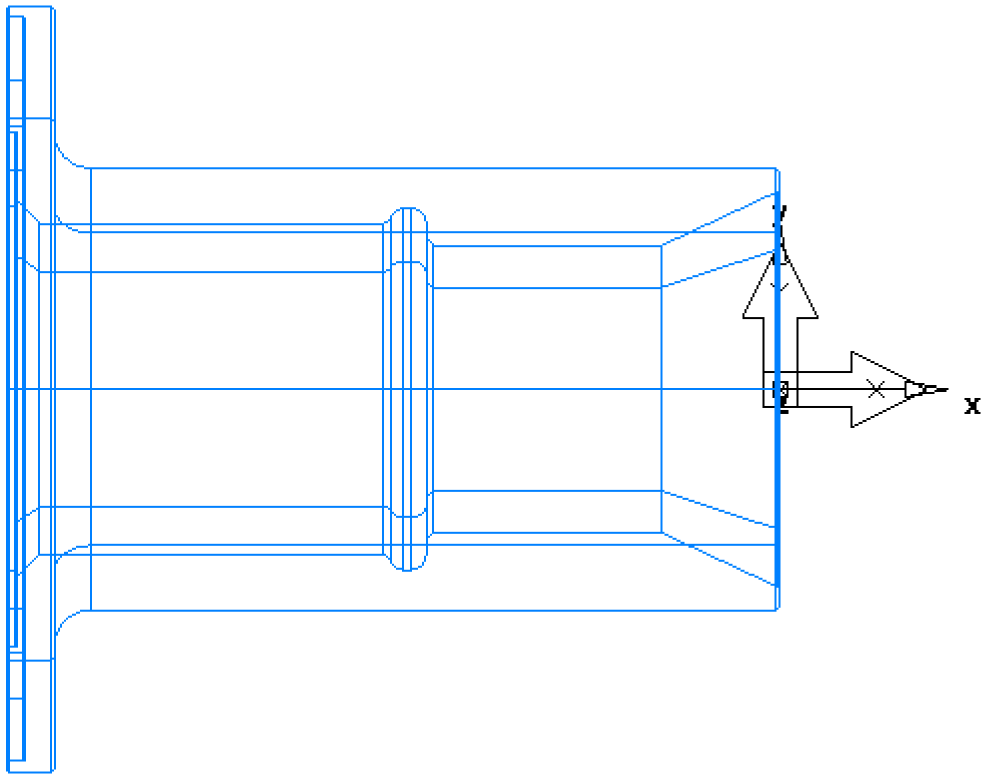
15 Then rotate the geometry so it is in alignment with the solid model.



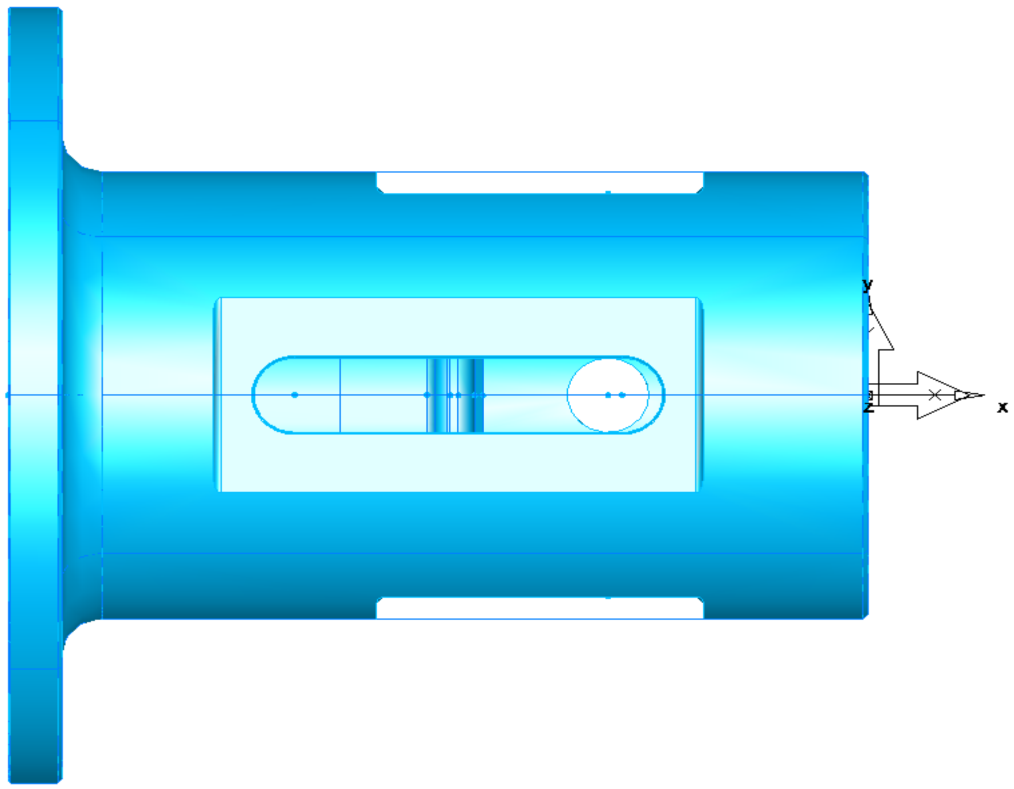
16 Select the **Curve**. Then select **Apply** to action the command.

17 Then select **Cancel**. **Hide** the solid model.

18 Double click on the **Stock1** properties. Select **X axis** and **Stock Curve**. Select the Curve and this will then create a revolved stock. As shown on the next page.

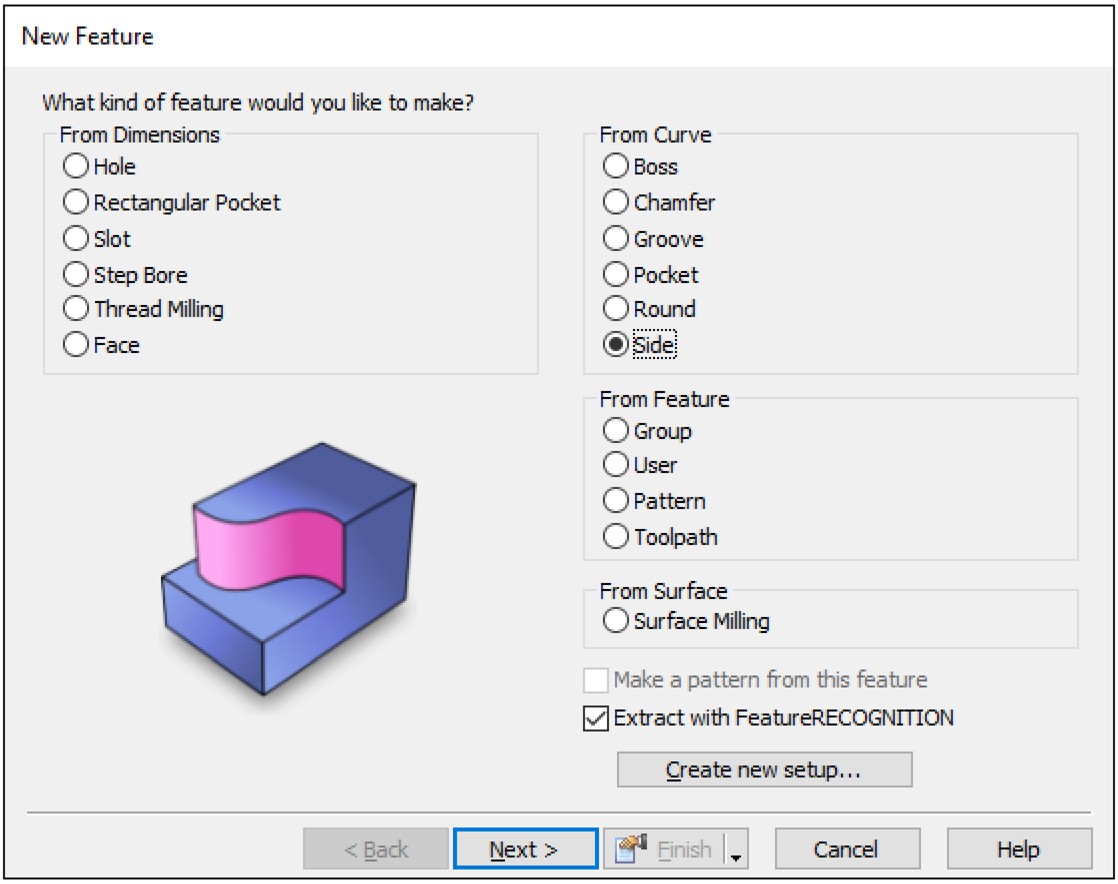


19 **Unhide** the solid model.

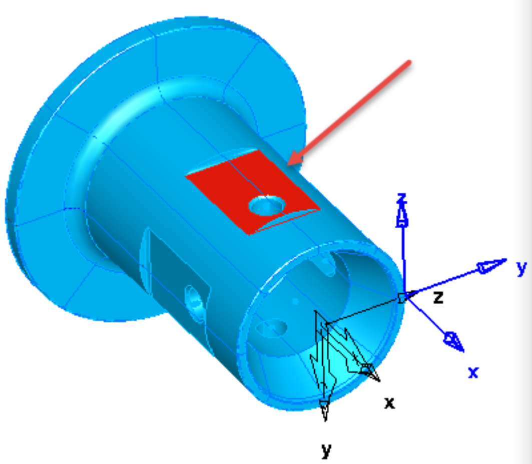


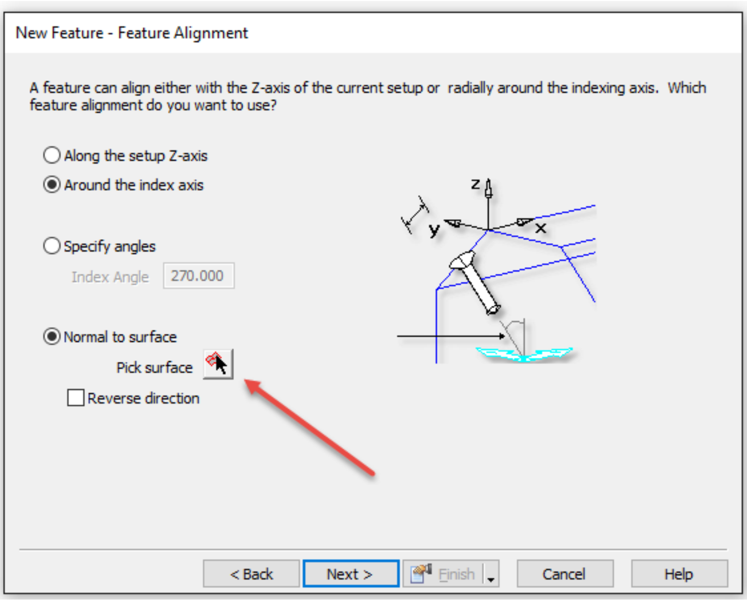
20 To extract Features from the solid model we are going to use Interactive Feature Recognition.

21 Create a new **Side** Feature and tick the box **Extract with Feature Recognition**.



22 Select **Next** and then select the surfaces shown. Select **Around the index axis** and **Normal to surface**. Use the **Pick surface** icon to select face.



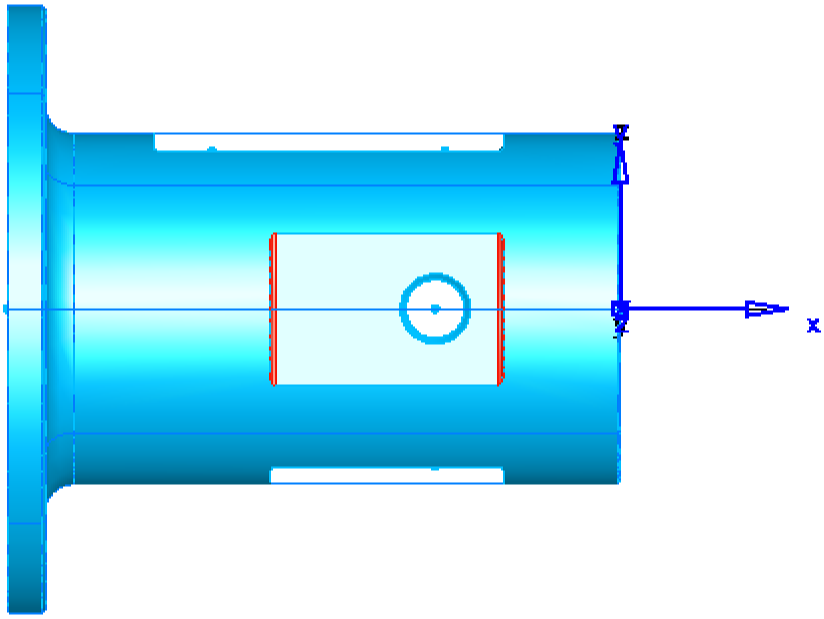


23 Select **Next**.

24 Use the **Select Surface Icon** as shown to select side surface.

25 Select **Next**.

26 Select the four surfaces as shown.



27 Select **Next**.

28 You will be presented with a form that shown machining side accept as is.

29 Select **Next**.

30 Accept Top and Bottom **location.**

31 Select **Next**.

32 Choose the Bottom Radius **Hyperlink** and select the **radius** on the model.

33 Then select **Finish**.

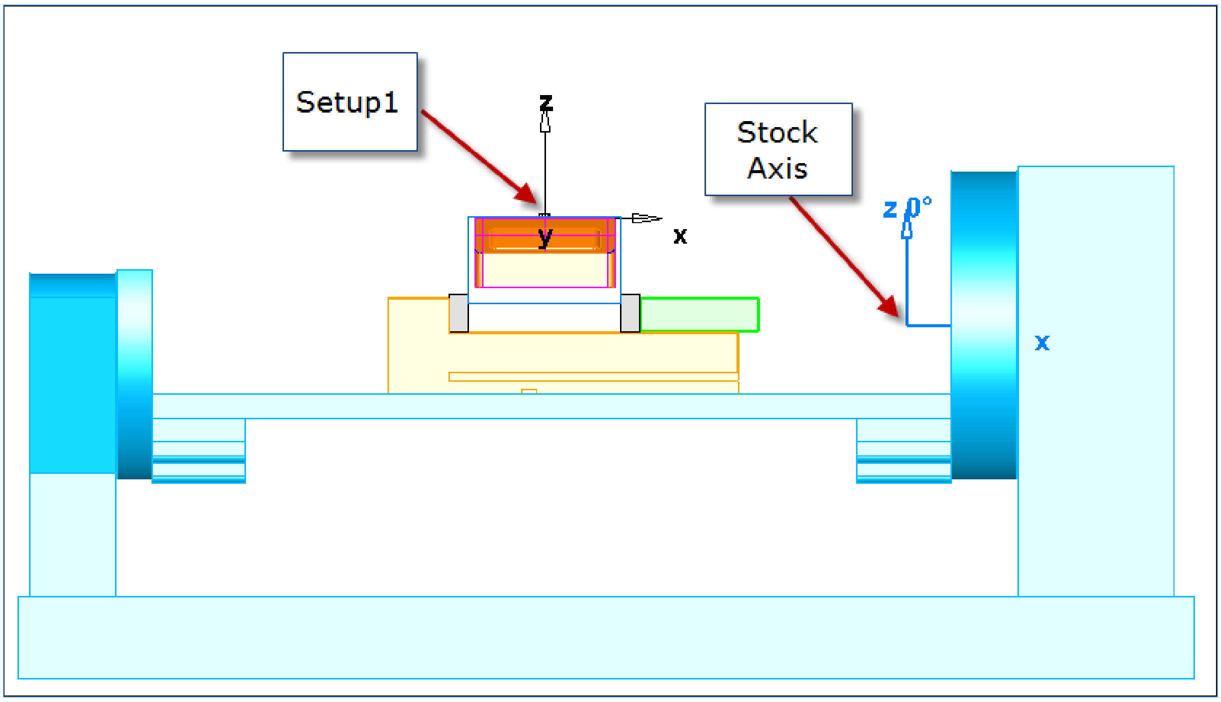
34 Then create a new **Pattern** Feature **Radial around the index axis Number 3** and **Spacing angle 270.**

35 Select **Finish**.

Apply the same technique to the large face and slot.

Create a Hole Feature and use ***Around the index axis*** *and* ***Automatic.***

Stock Axis Example - (4 Axis). (Class Exercise)



*The following example will show you the importance of positioning the **Stock Axis** (World Coordinates) With regard to your setup as shown in the example above.*

*We will **Open** a FeatureCAM file of a machine template for this exercise then **Import** – **IFR Work Planes Milling Metric. X_t** and position this into a machine vice by using the **Translate** and **Rotate** commands. Then create 3 Setups **(Setup1)** Top Face **(Setup2)** and **(Setup3)** for the angled faces. We will machine this when we start using **Interactive Feature Recognition (IFR)** on day three.*

1 **Go to File >Open** and navigate **to C:\Training_Data\FeatureCAM Course Data 2017\Milling Data .fm Files** and then **select Machine & Points to snap too Metric.fm** select **Open.**

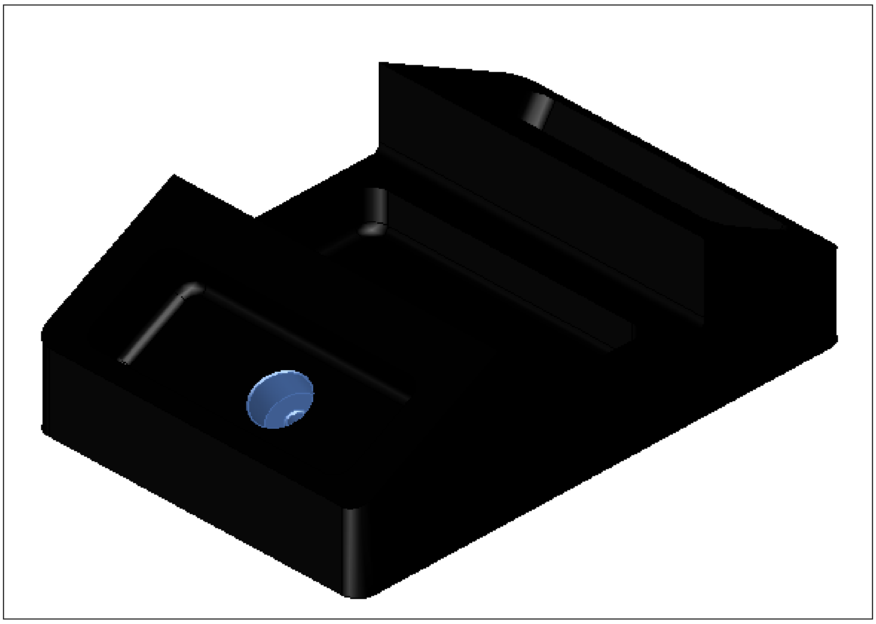
2 Select View>Hide All.

3 Go to **File >Import** and navigate to C**:\Training_Data\FeatureCAM Course Data 2017\Milling Files to import.** Then select **IFR Work Planes Milling Metric .x_t**

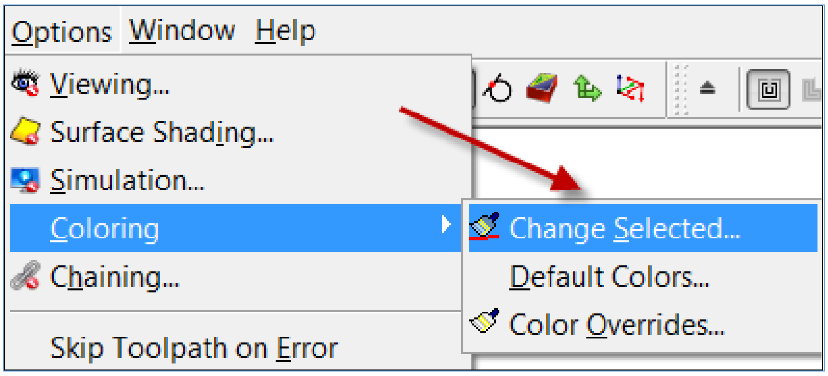
4 Select Cancel to Cancel Import Results.

5 In **PartView** click on the **+** next to **Solids**, this will show you all the available solids. Select each solid in turn until you select the solid we are currently working on. It will turn red for selection.

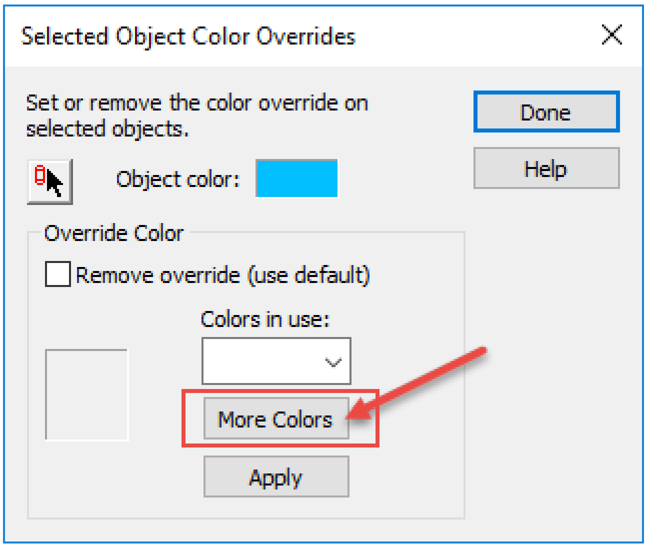
6 Right click on this solid and select **Rename**, Call the solid **Part**.



7 You will notice that the **default colour** on import is **Black**; this is because we have created the part in **PowerShape**. We need to change this. Go to **Part View** and select **Part**, it will turn Red for selection, and then go to **Options** and **Colouring.** Then select **Change Selected**.

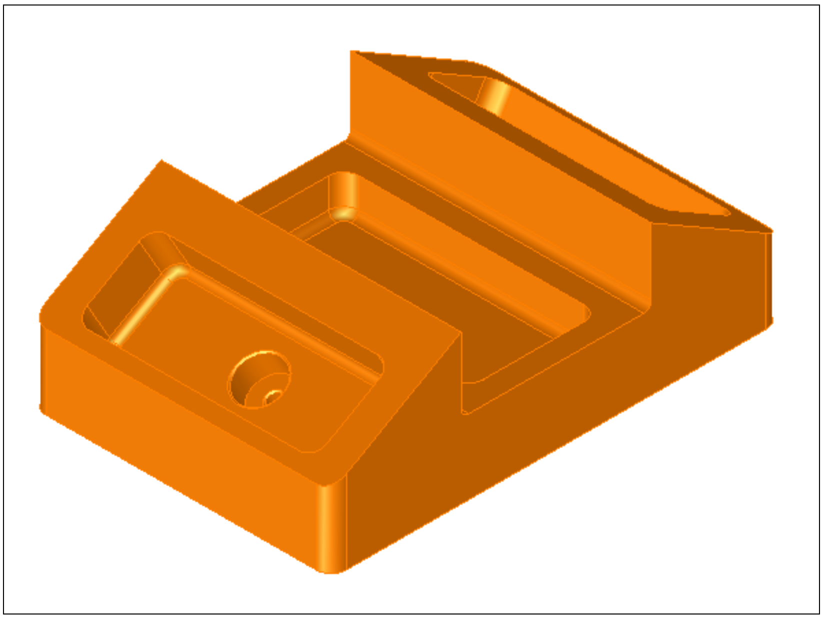


8 Select the model then go to Options>Coloring>Change Selected>More Colours.



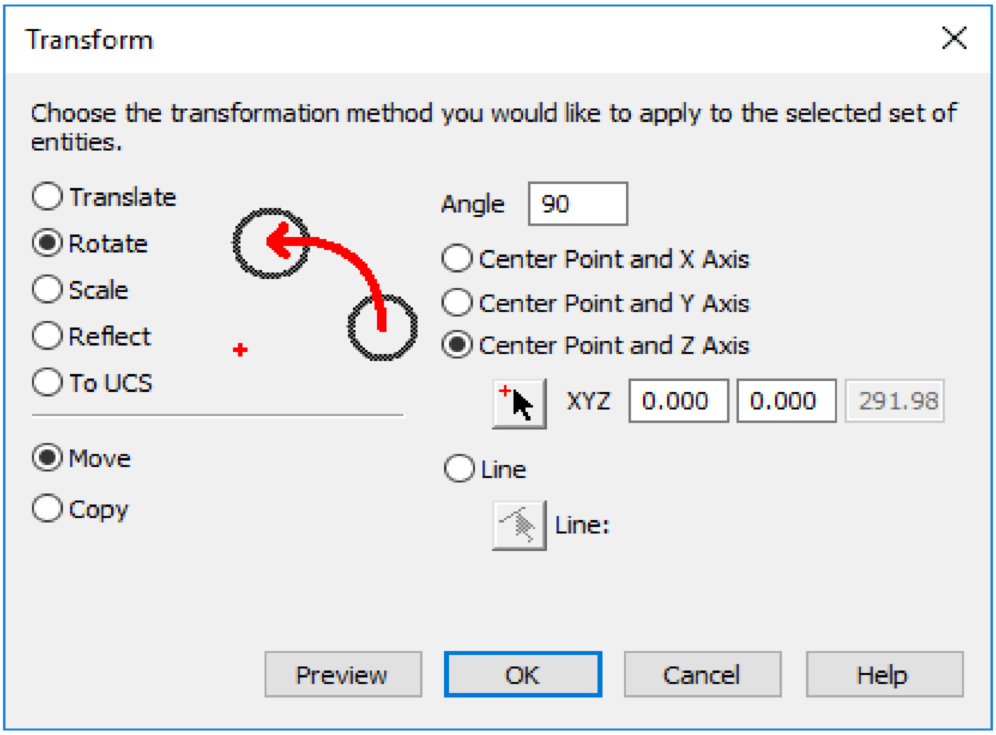
9 Select **More Colours** and select the colour of your choice. Then select **Apply** and then **Done**

10 The solid Model Colour will change as shown on the next page.

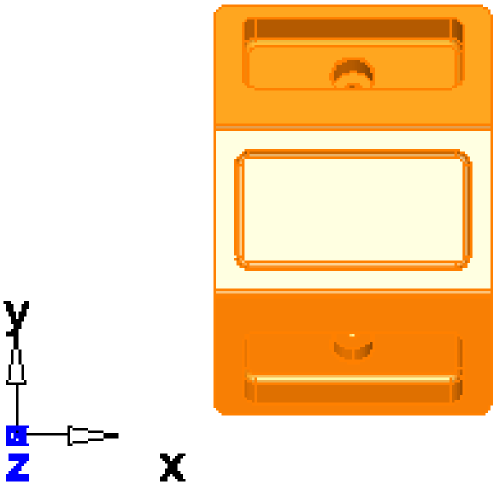


11 Select **Top View** or select **Ctrl** + **5**. This will show the correct orientation of the Solid model. You will notice that the model is **90 Degrees** out of position.

12 We need to make the longest edge in the **Y Direction**. Select the Solid model in **PartView**. Select **Edit>Transform**. Set the following in the form shown below. Select **OK**.



You will notice that the Solid model is now orientated correctly.

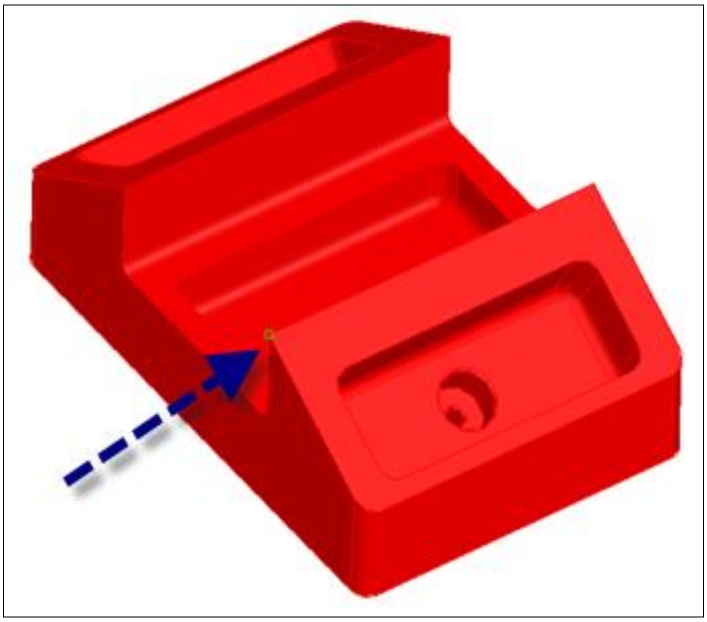


13 Select **View>Show>Select All**.

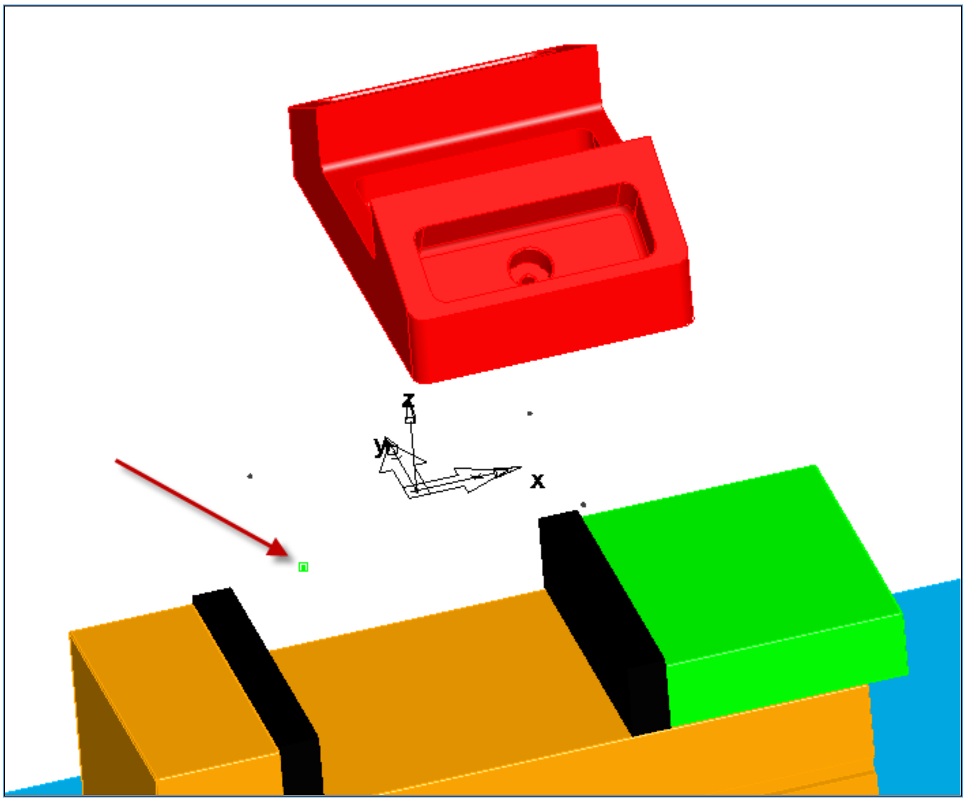
We need to move the Solid model to the correct place relative to the vice.

There are a number of ways to locate this model. But for quickness we have created some points for you to snap too. You will see 5 Points by the vice.

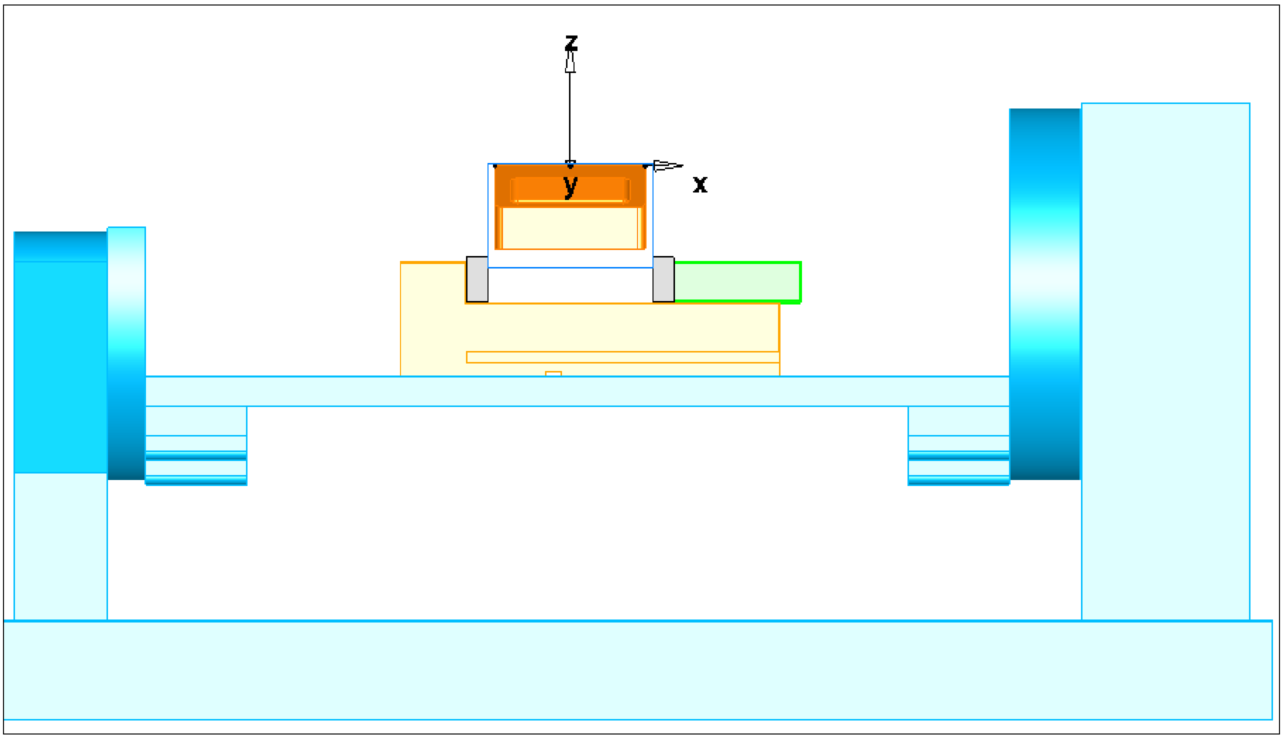
14 Select the **Part** in **PartView**. Select **Edit>Transform>Translate**. Select From and pick point as shown. Make sure you have **Snap to Object** and **Snap to Point** active.



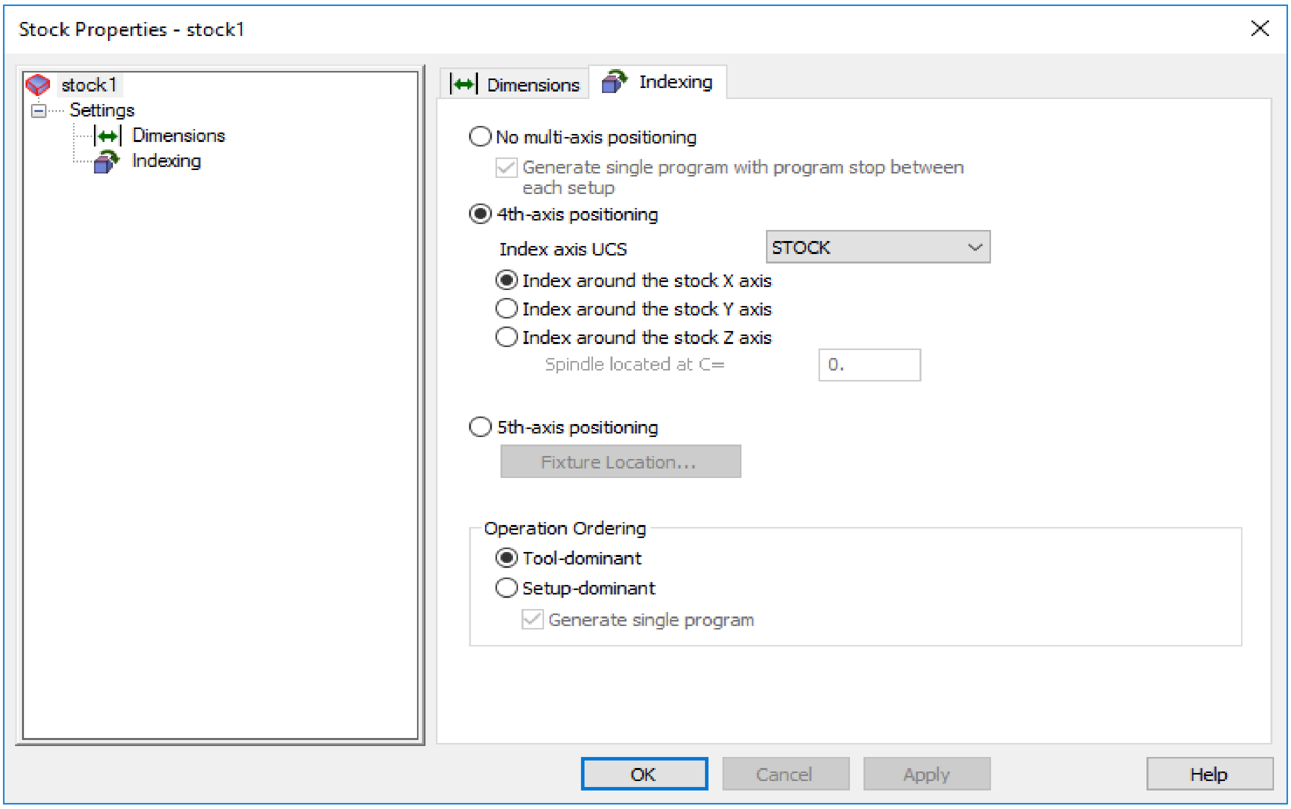
15 We now need to position too my point as shown.



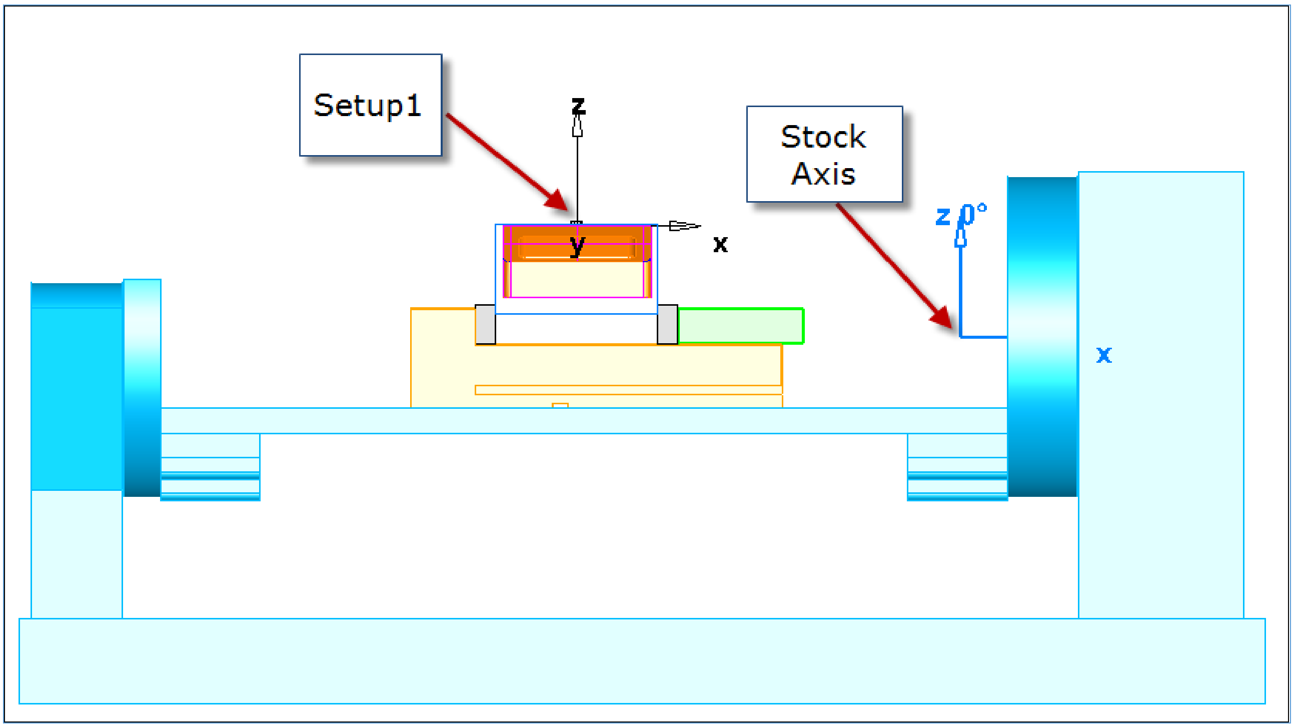
The image below shows the part correctly translated into the correct position.



*We need to set up for 4th Axis indexing. So we can Index the Part. Double click on **Stock1** and from the **Stock Properties** select **Indexing** and then select **4th Axis Positioning>Indexing around the STOCK X Axis.** This is already set for our example.*



*The **Stock** and **Setup1** is already configured in our example.*



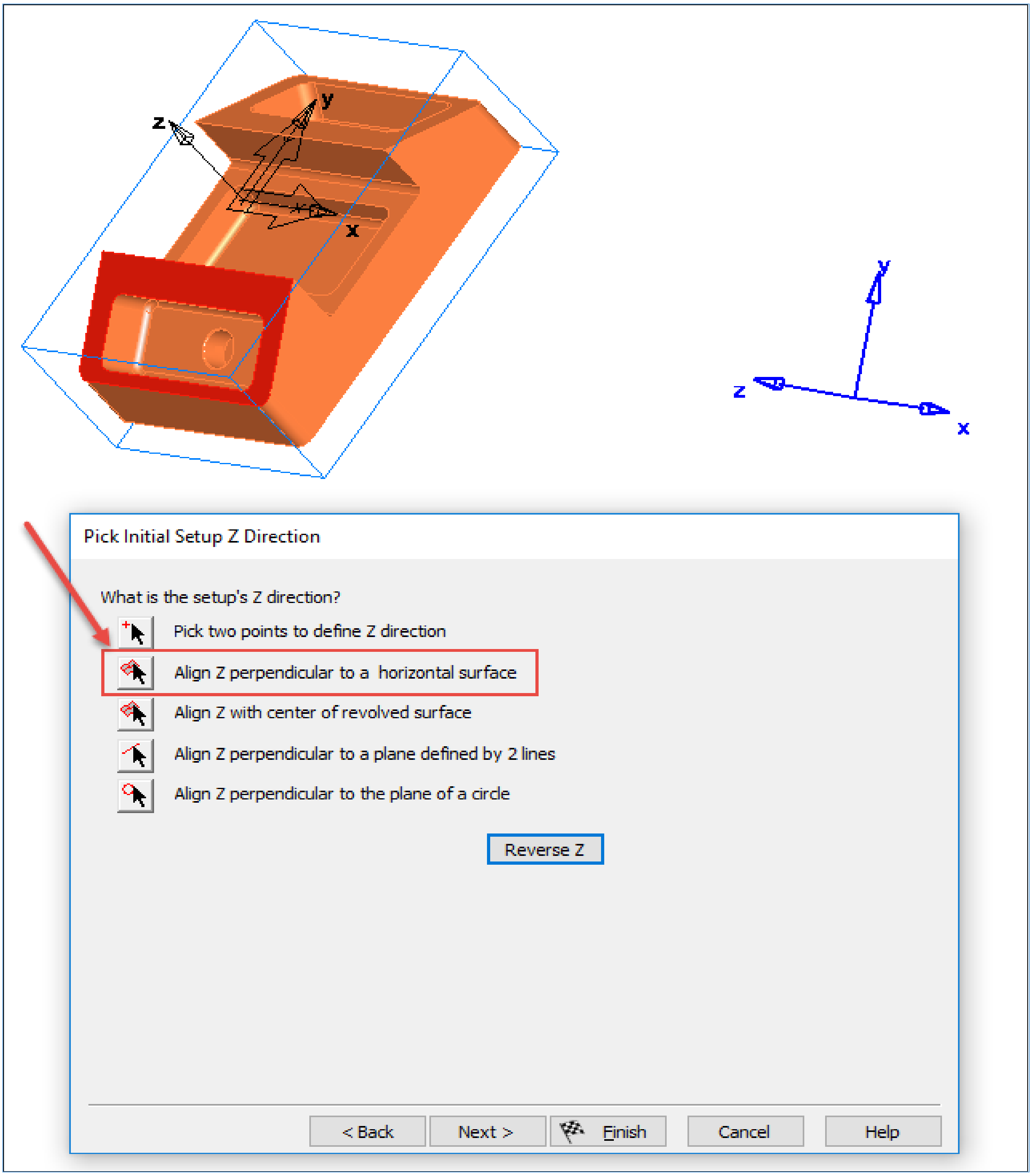
16 We now need to create two more Machine Offsets **(Setup2) G55** and **(Setup3) G56** for the angled faces.

17 Select **Setup1** in **PartView** and then **Right click** and then select **Properties**. Select **New** and then **Next.**

18 Select **Align to Part Geometry.** Select **Next.**

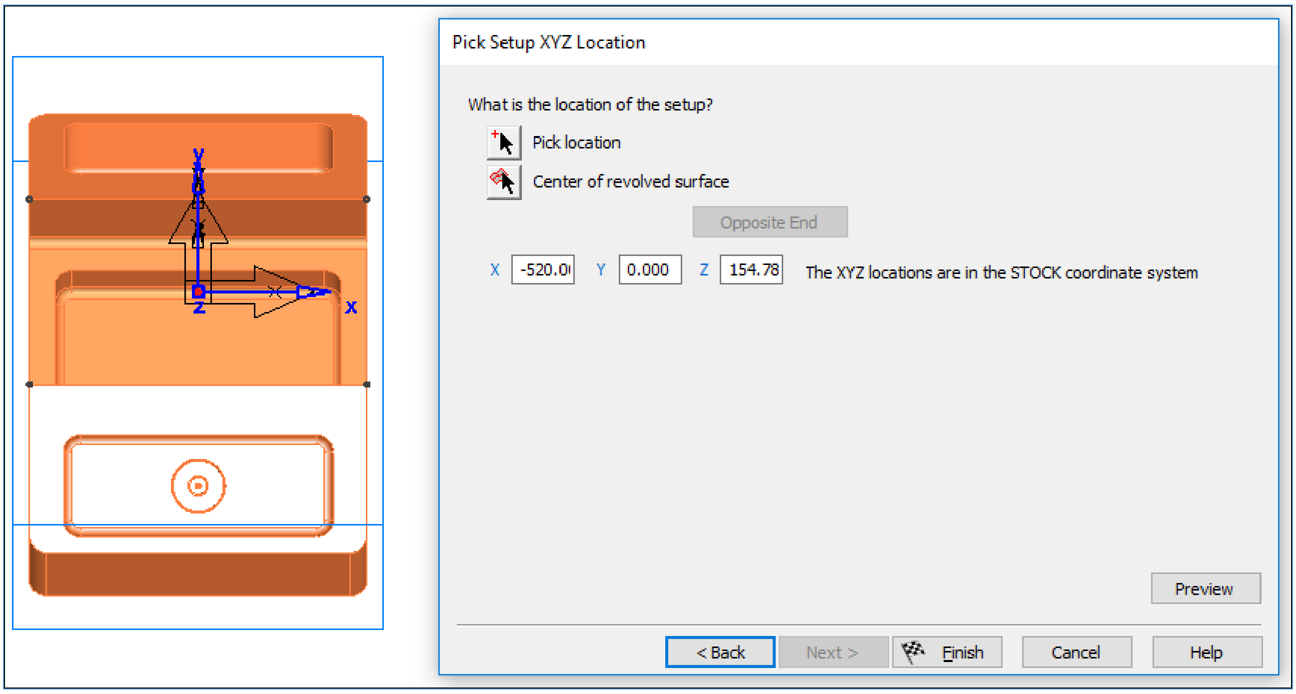
19 Then select **Align Z Perpendicular to a Horizontal Surface.**

20 At this stage you may have to reverse the **Z direction**. Select **Reverse Z**.



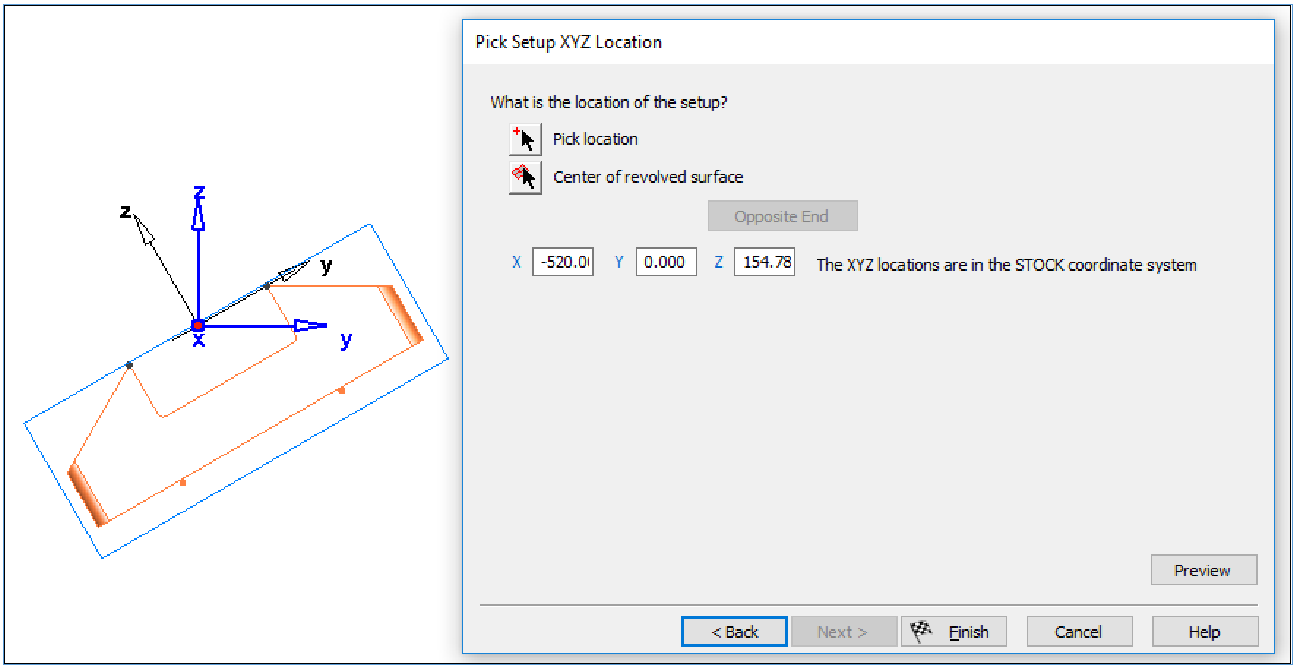
21 Select **Next** twice and then click **Pick Location**. Make sure you have Snap to Point highlighted. Then **Snap** to the point at **Setup1**. Select **Finish**. That is **Setup 2** completed.

Setup2

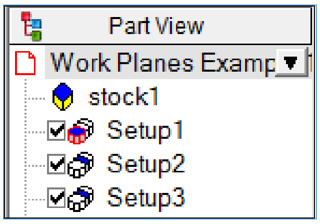


*Repeat the process for **Setup3***

Setup3



*We have now finished all of our **setups**.*



22 Save the file for IFR (Interactive Feature Recognition). We will now Machine the Part using Interactive Feature Recognition.

23 Activate **Setup1** in **Part View**.

24 Press **Ctrl** + **R** New Feature. Select **Face**. Select **Finish.**

25 Change the cutter to an **80mm Facemill** with **Zigzag** direction at 90 degrees.

26 We will now machine the outer edges of the part just slightly deeper than the actual bottom of the part but above the Vice jaws. To a depth of **105mm**

27 Press **Ctrl** + **R** or **New Feature** from the **Steps** menu and select **Boss** with **Extract with Feature Recognition ticked.** Select **Next.**

28 Select **Along the Setup Z Axis**. Select **Next.**

29 Select **Use horizontal surface** then **Next** and then select the base face and add this to the selection. Select **Next**.

30 **FeatureCAM** will ask for a **Top** and **Bottom** dimension. You can select this from the model or enter the following dimensions. **Top=0 Bottom=-105** select **Finish.** Make sure that the tool selected will achieve the depth required.

31 By default the machining strategy will Ramp down to depth and then machine all around the part and repeat this at each level. This is very time consuming. Double click in **Part View** on the **Boss** Feature. Select Strategy unselect **Rough Pass** and select Semi-Finish pass and **Helical side finish** with a pitch of **6mm** select **Apply** and then OK.

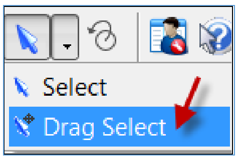
32 We will now machine the large centre pocket using **Side** and **Feature Recognition**. Select the 4 side surfaces this includes the Radius surfaces.

33 Press **Ctrl** + **R** or **New Feature.** Select **Side** with **Extract with Feature Recognition ticked**

34 Select **Along the Setup Z Axis**. Select **Next.**

35 Use Select **Side Surfaces** for the **Feature Extraction**. Select **Next**.

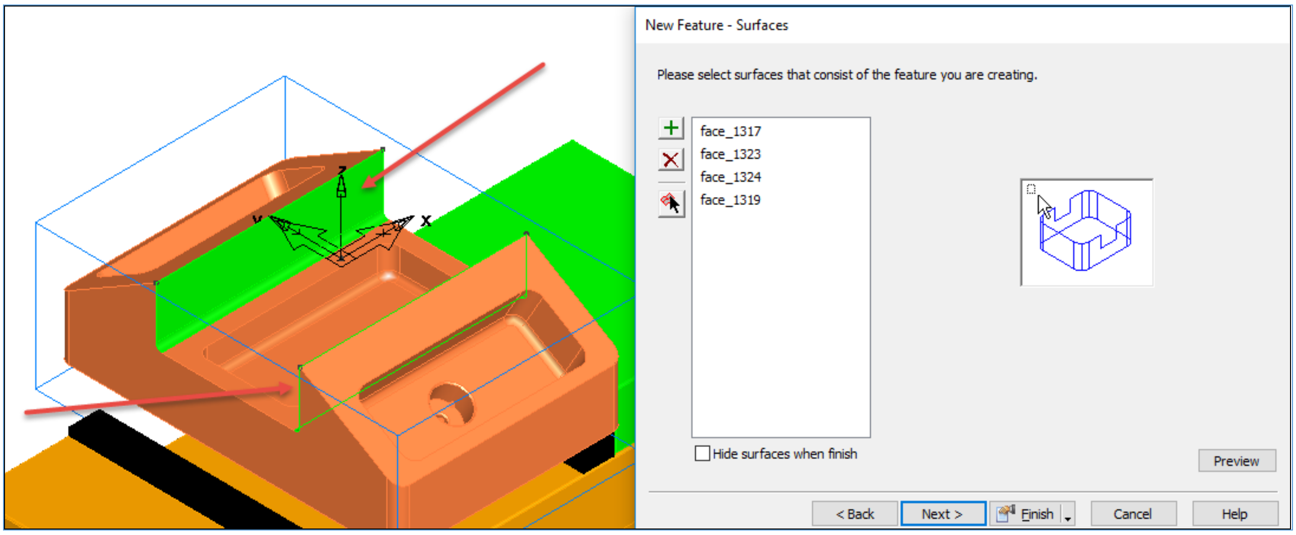
36 In **FeatureCAM 2017** we now have a new selection tool called **Drag Select**.

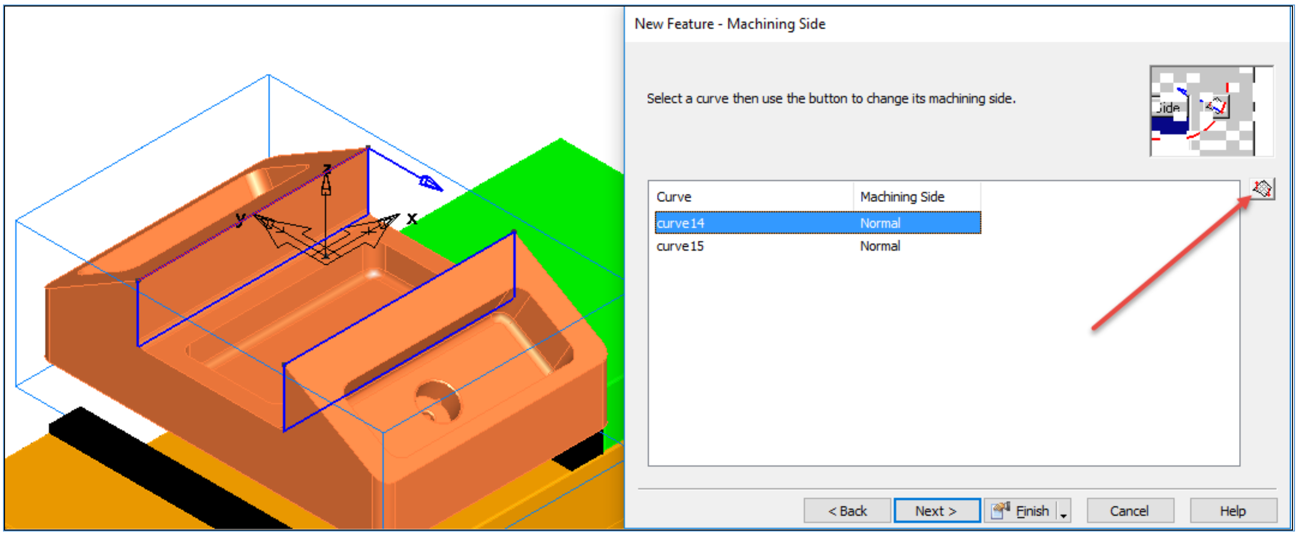


37 Select the **Side** edges of the large centre pocket as shown. **(Not the Vice)**

Select **Next**.

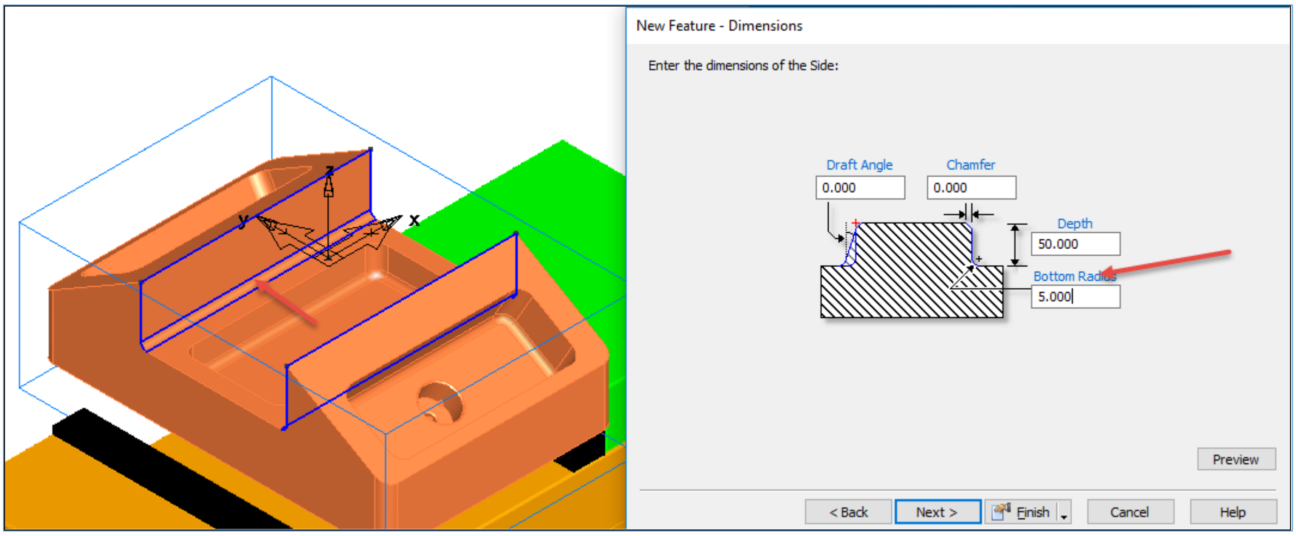
Make sure the **Side** machining arrow is pointing in the correct direction. If not select the change direction Icon. Select **Next.**





38 Make sure you have the correct depth **-50mm** if not select the Bottom Icon and click on the bottom of the surface. Select **Next**.

39 The next menu gives you the dimensions for this feature.



40 To extract the correct Radius dimension, select the blue Bottom Radius **hyperlink** in the form as shown above and then select the radius on the solid model. This will extract the correct radius (5mm). The centre large pocket is now machined to Z-50mm.

41 The order of machining may change. If **FeatureCAM** machines this part in a different way to way you expected then we can change the **Base priority** as Follows **Face** Base priority=1 **Boss** Base priority=2 **Side** Base priority=3. To change this go to **Part View** and double click on **Face** go to **Misc** and change the **Base priority.**

42 The next thing we need to do is machine the smaller lower pocket.

43 With this Feature all you have to do is press **Ctrl + R** and select pocket with **Extract with Feature Recognition**. **Next.**

44 Along the Z Axis. **Next.**

45 **Automatic recognition**. Select all then **Finish**. The Feature is complete.

46 **Setup2**

47 We will now move onto **Setup2.** Select this in **Part View**. Once this has been selected all machining will be related to this Setup.

48 The first machining operation will be to face the angled face. Press **Ctrl + R** to create a **New Face Feature** using **Extract with Feature Recognition**. Select **Next**.

49 We will be using **along the setup Z Axis**. Select **Next.** Select the Pick Surface  Icon and move the cursor over the angled face and this will highlight green, use the left mouse button for selection; this will then turn Red to indicate your choice. By selecting this boundary this will reduce the amount of fresh air moves. Select **Next** twice until you get to the **Strategies** menu. Select **Connect stepover with arc** (ideal for high speed machining) although this will add time to the operation it will put less strain on the Facemill and machine. We also need to select **Rough pass** to machine at equal depths until roughed out. The finish pass will then take a last cut over the face. Select **Finish**.

50 Now whilst the cutter is in the spindle we will machine the angled face in **Setup3.**

51 You know what to do now. Machine the angled face in **Setup3.**

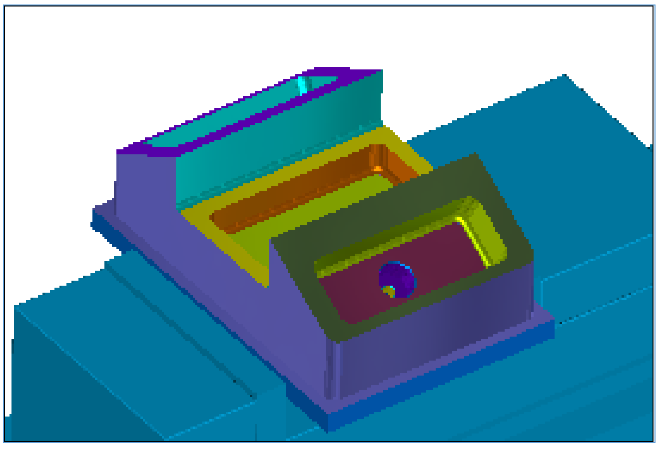
52 While we are machining **setup3** we can automatically machine the pocket on that face. Do not select all but highlight the Pocket by selecting with the left mouse button. Machine the pocket complete, you should know what to do by now if you have worked through this exercise. Complete the pocket in **Setup2**.

53 The next thing to do in **Setup2** and **Setup3** is to machine the Hole features.

54 Select **Ctrl** + **R** and select **Hole** with **Extract with Feature Recognition** ticked**.** Select **Next. Along the setup Z-axis. Next. Next** then Select, **Select all.** Select **Finish**.

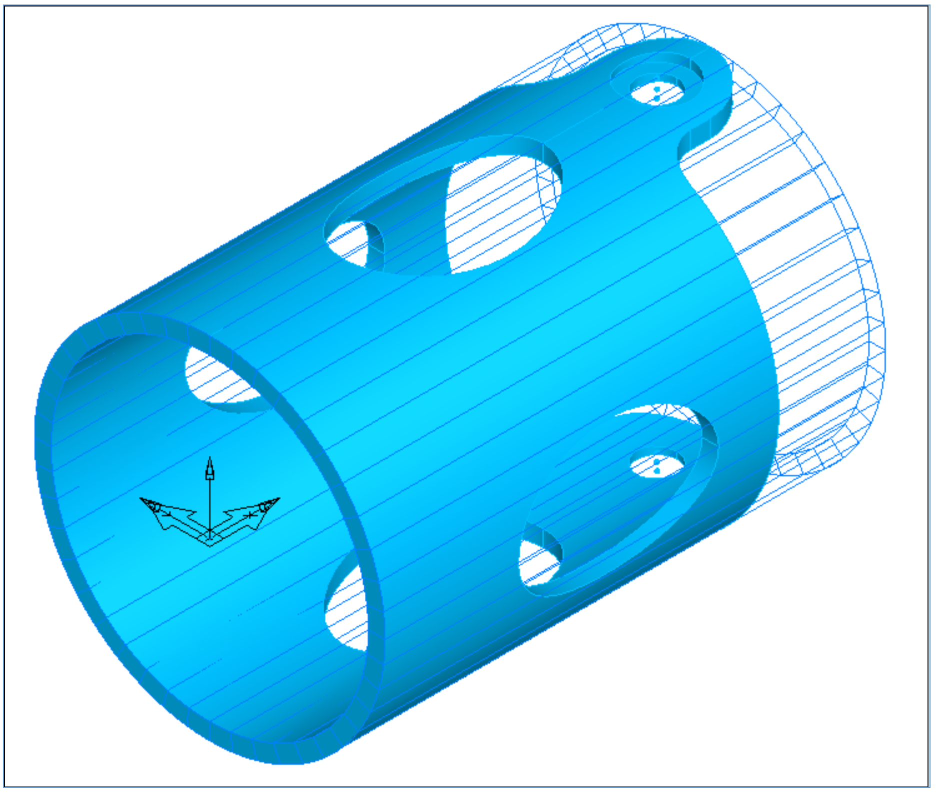
55 Select **Setup2** and repeat the above procedure for machining the **Hole** features.

56 Although there is machining still left on the underside. All you would have to do is turn the job over in the Vice and run the facing opp to machine the surplus material. The job is now complete.



4 Axis Unwrap curve options + 3D Chamfer

1 Open the file **4 Axis Unwrapped Side Feature plus 3D Chamfer – Start.fm from C:\Training_Data\FeatureCAM Course Data 2017\Milling Data .fm Files.**

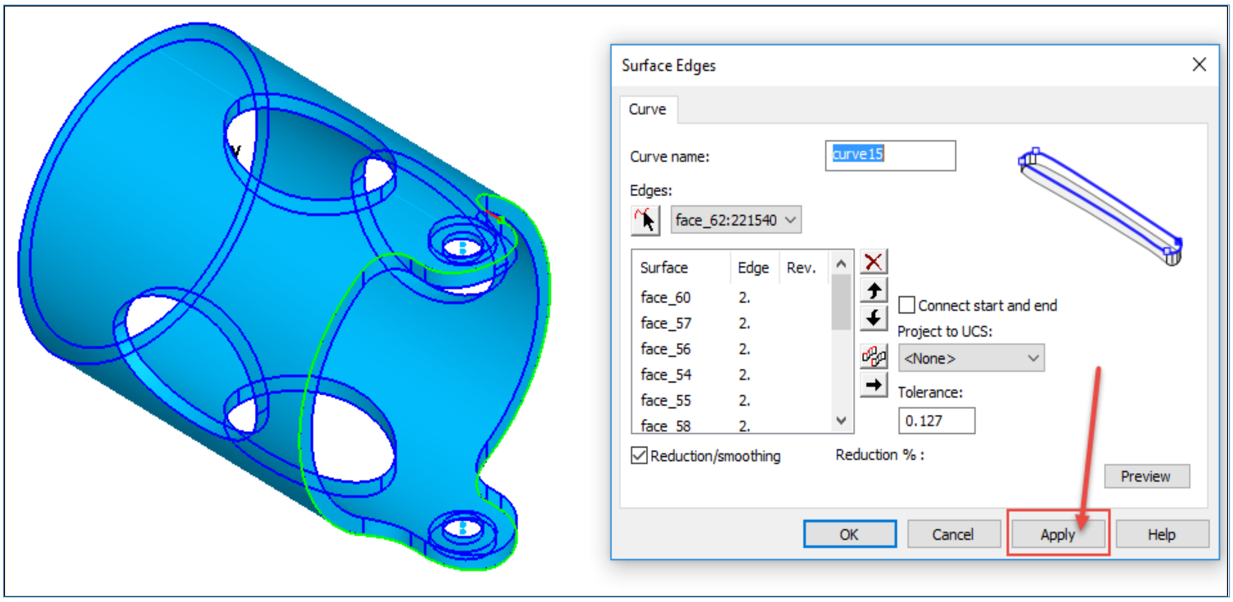


First of all we will extract a curve from the solid model and then change the start point on the curve so the machining starts in a place that we would like the Leadin/out to start from.

2 Hide Stock. Please select **Construct** from the main menu.

3 Select Curve>From Surface>Surface Edges

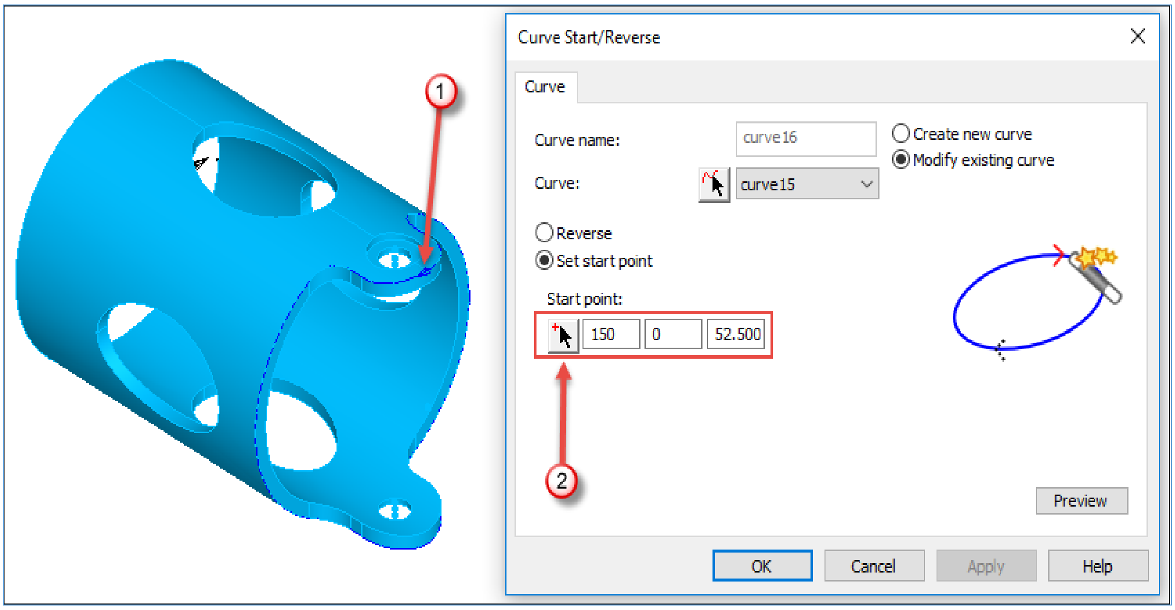
4 Work your way around the edge of the solid model as shown on the image below. Once completed select **Apply** and **Ok** to close the form.



We need to change the start point of the curve so we can decide where to lead in and out.

5 Please select Construct>Curve>From Curve>Curve Start/Reverse

6 Please select **Modify existing curve** and **Set start point**. Choose a point on the model where you want the start point to be. Or type in the values as shown below. **Point 1** is the original start point. **Point 2** is where it will be moved too.



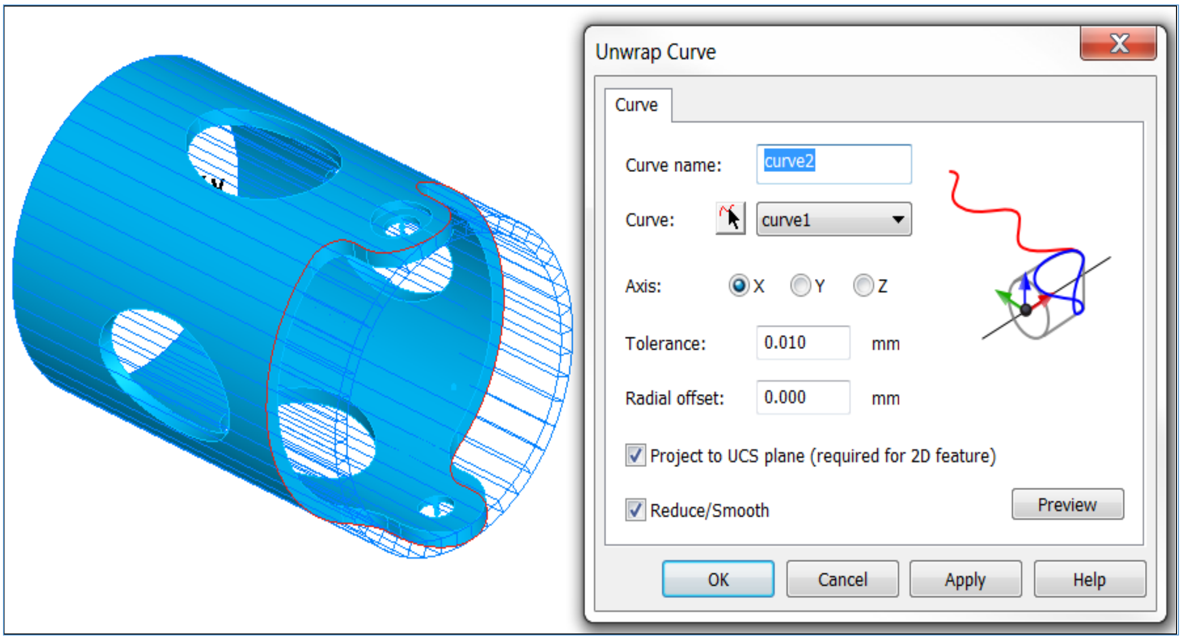
7 Select **Apply** and **Ok** to close the form.

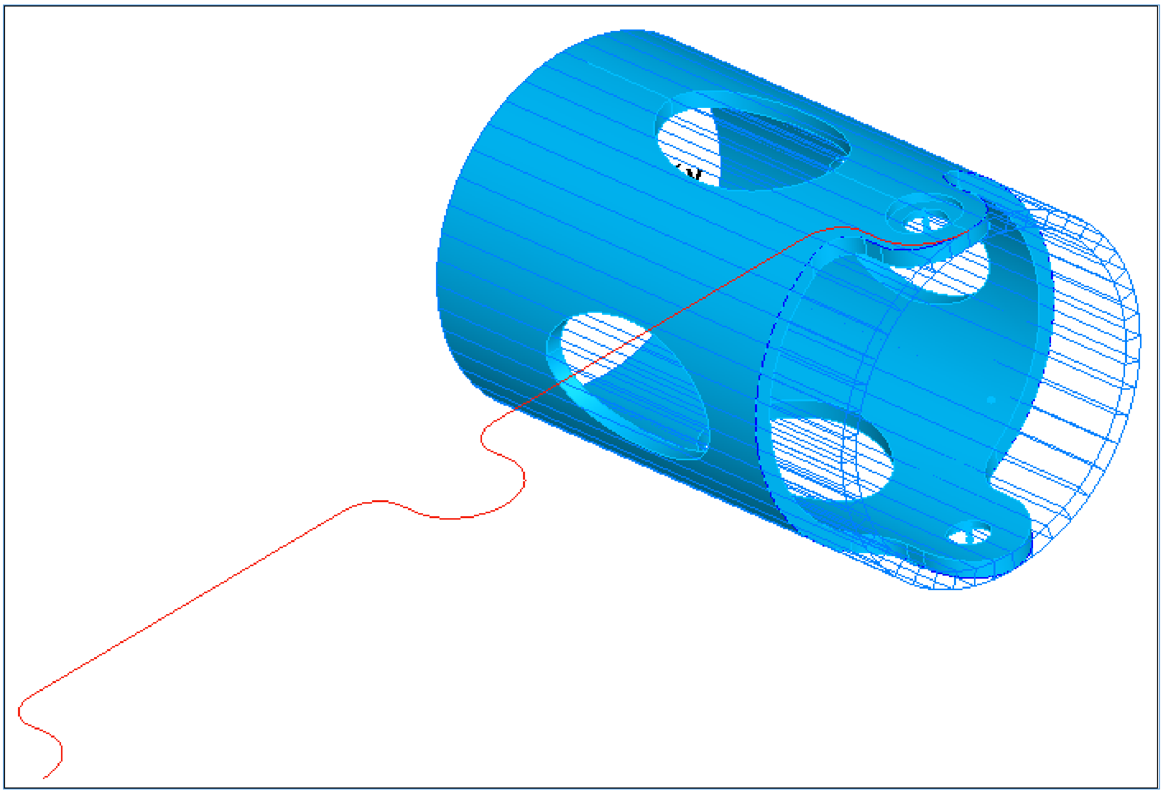
We now need to unwrap the curve so we can create a Side Feature and machine around the profile.

8 Please select the **Curve1** in **PartView**.

9 Please select **Construct>Curve>From Curve>Unwrap**

*This will **unwrap** the curve and create a new curve called **Curve2**. Select **Apply**. See lower image for the result.*





We will now use this curve to machine around the profile.

10 Please select **Ctrl+R** to create a new **Side Feature**.

11 Select **Next**. Select **Curve2**. Select **Next** twice.

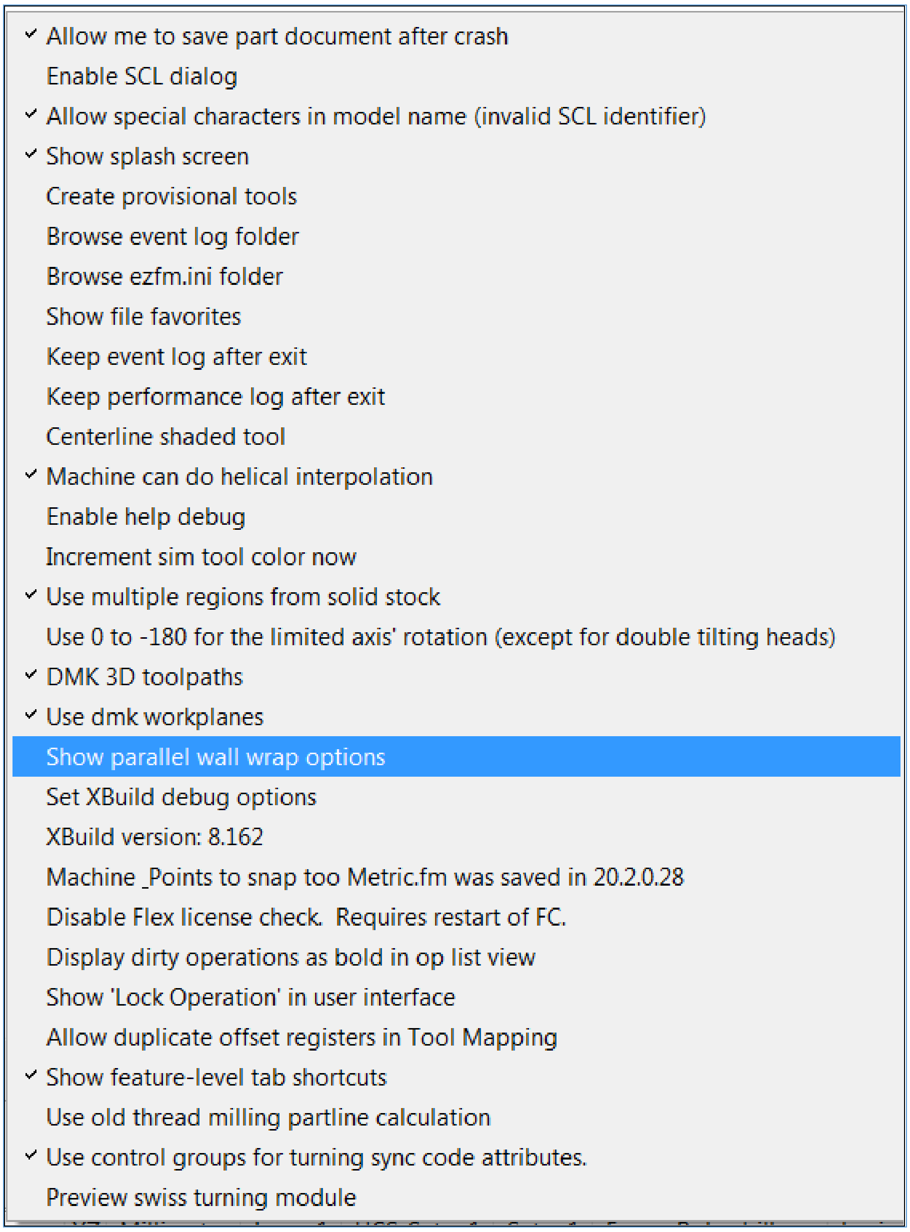
12 Change the **depth** to **10mm** and select **Finish**.

13 Go to the **Strategy** page and unselect **Rough Pass** and **Ramp from Top**.

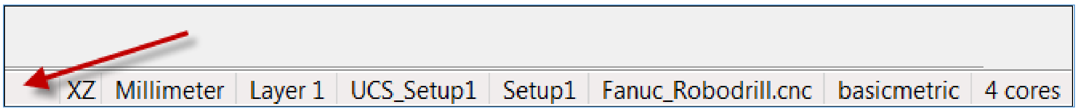
14 Please select **Semi-finish pass**. Select **Apply** and **Ok** to close the form.

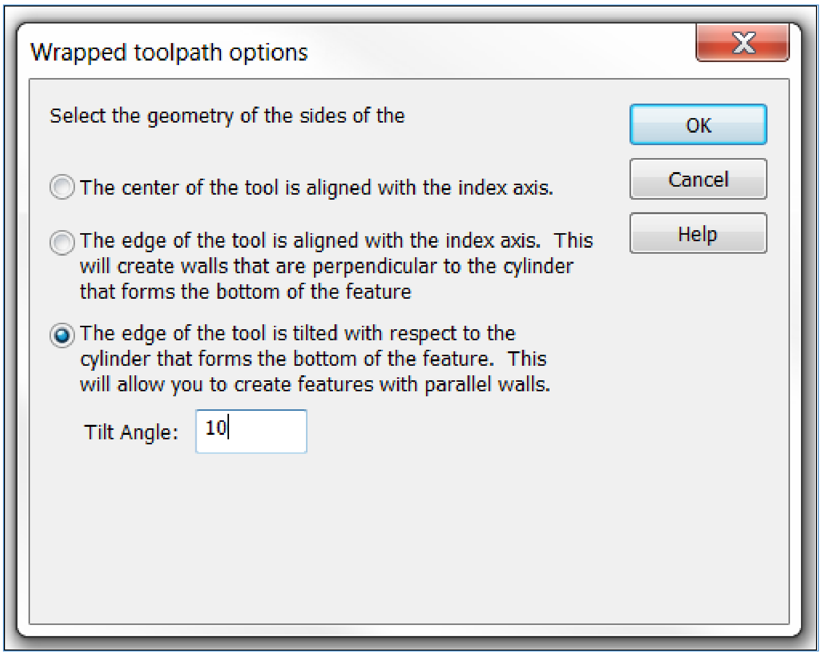
15 From the dimensions tab please select **Wrap feature around x axis**.

16 Please select **Wrap Options**. Select the following options. If this option is not available select this from the special menu. **Show Parallel wall wrap options**.

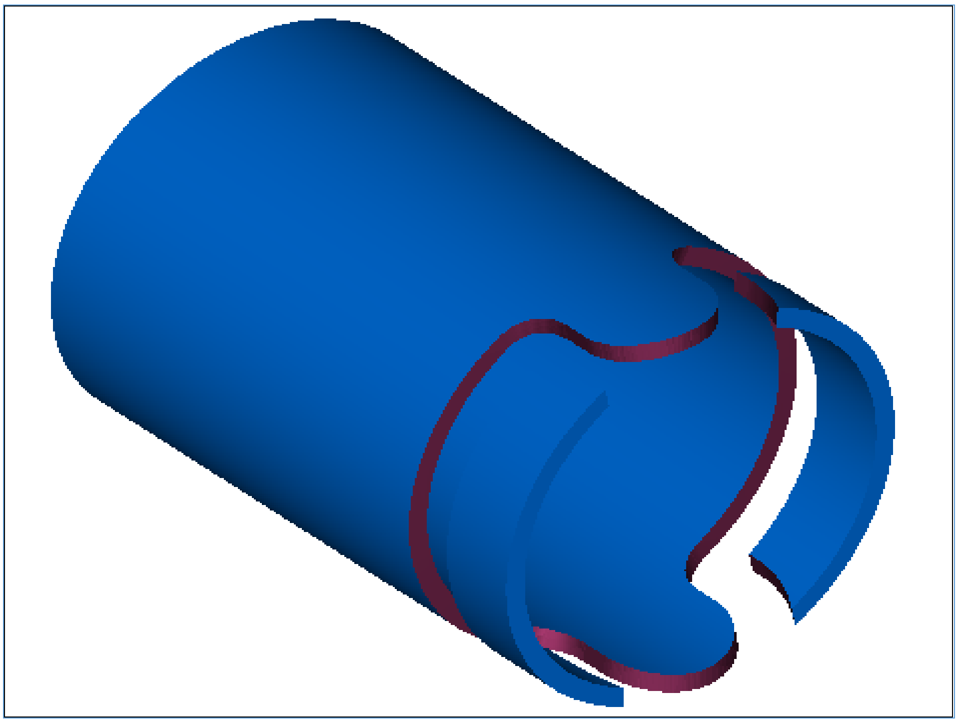


17 To activate this menu select the **CTRL** key and Right mouse click in the position as shown. Once selected a tick will appear next to this option.





18 Select **Ok** to close the form. Select **Apply** and **Ok** to close the main Side feature menu. If you want to change the tool to a **12mm diameter Endmill**.

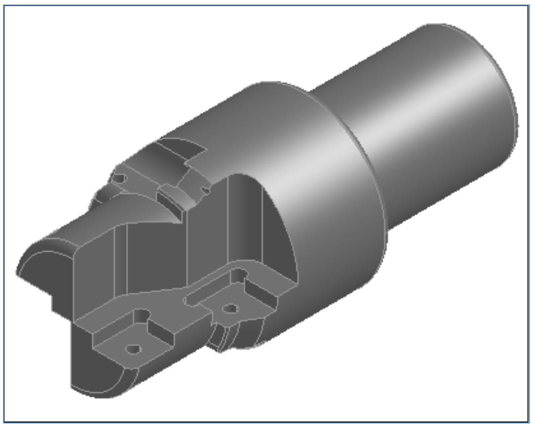


*The **Side** profile is now machined parallel to the surface feature.*

Have a go at creating the 3D Chamfer by extracting a curve from one of the Oval pockets. Create a chamfer feature from Curve and select the 3D Chamfer option. Make sure you machine the Pocket first.

Boring Tool 4 Axis (Work through this on your own)

FeatureCAM gives extraordinary results on solid models mounted on a 4 Axis indexer or rotary table. The part you will create is a special boring tool with pockets that will use carbide indexable inserts when finished. You will begin by importing TWO solid models. One is the Part and the other represents the **Stock**. The **Stock** material would logically be turned on a lathe to the size and shape of the finished holder and that is the state of the Stock when it comes to milling these secondary features. **The Index axis will be around the Stock X Axis**.



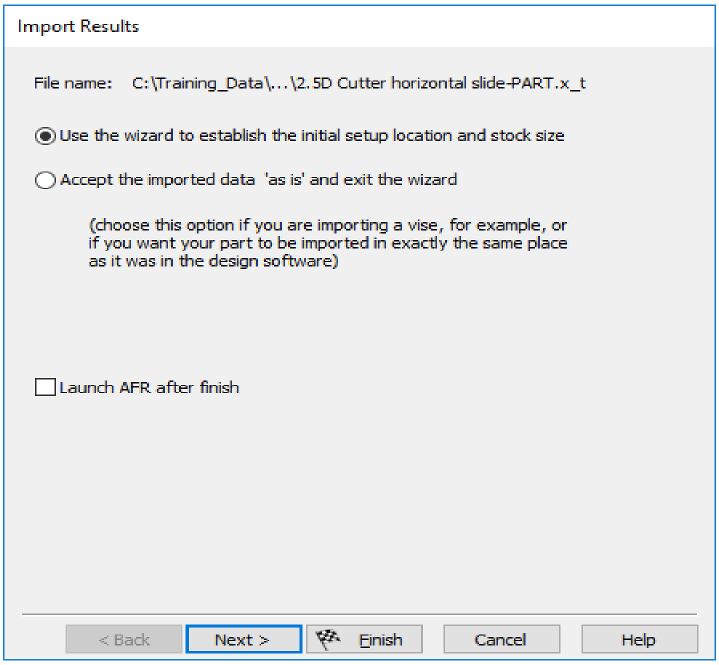
1 **Open** a new **Milling** document in **Millimeters.** Click on **File** then **Import** and Navigate to the training files your instructor placed in the appropriate folders.

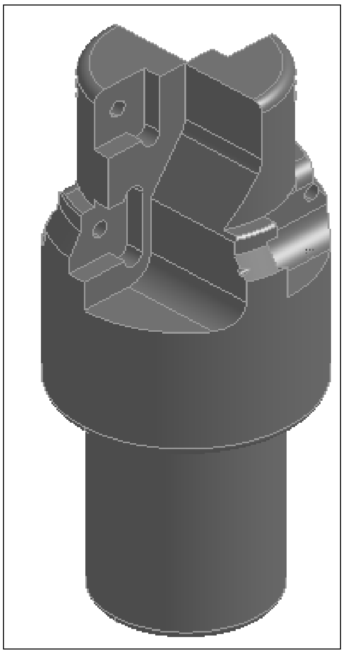
2 **C:\Training_Data\FeatureCAM Course Data 2017\Milling Files to import**. Import the model named **2.5D Cutter horizontal slide-PART.x_t**

3 In the Import Results **Select** "Use the wizard to establish the initial setup location and stock size" and select an **Isometric View** by right clicking in the graphics window.

4 **Uncheck** "Launch AFR after finish". Select **Next**.

5 Click somewhere in the Graphics Window to "de-select" the Solid model. The Solid model will turn from Red (Selection) to grey.

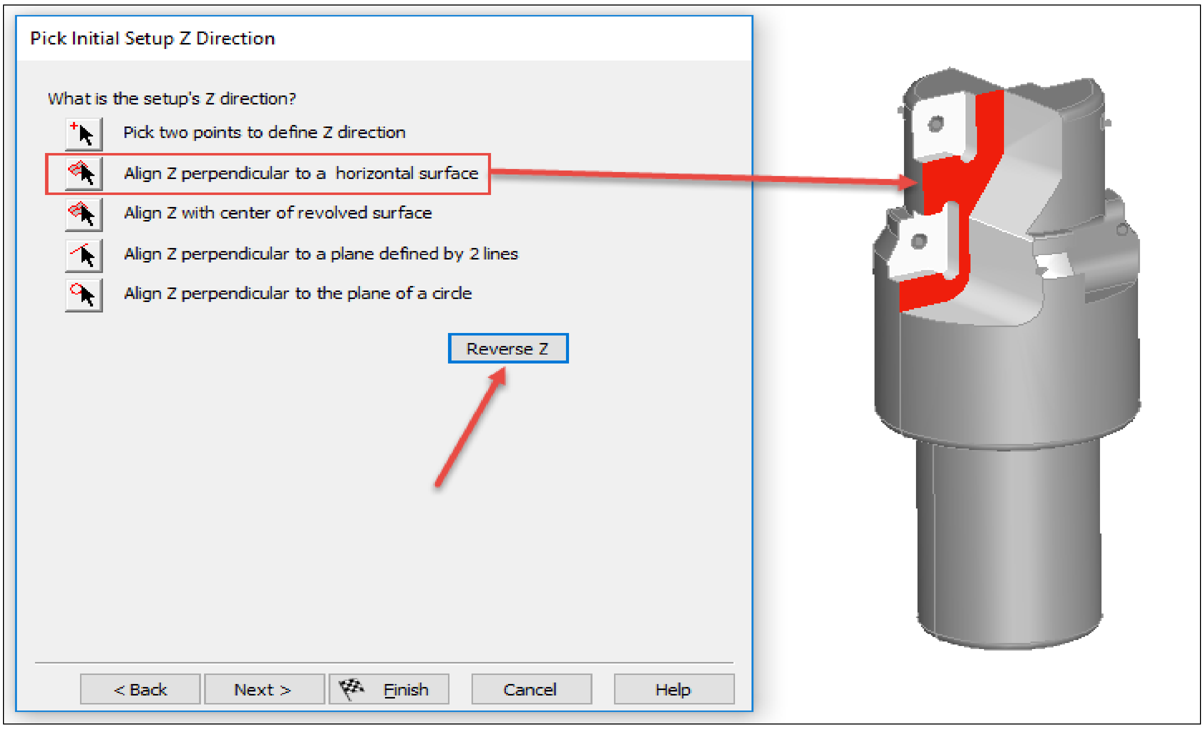




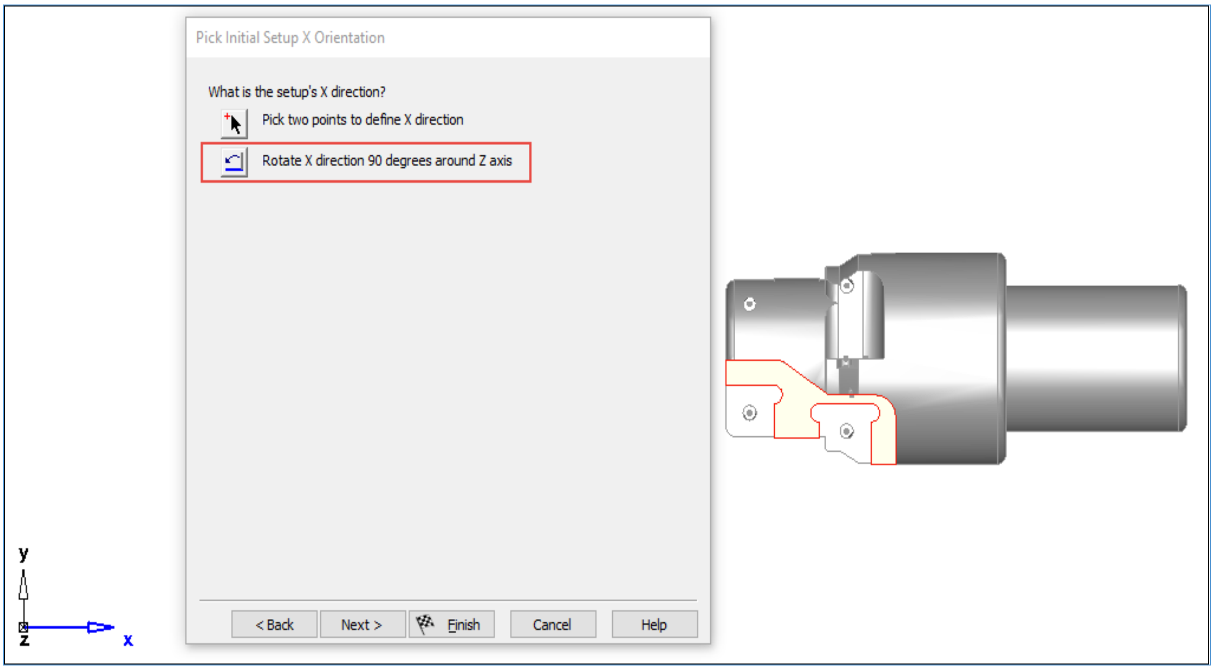
6 The part centreline must be aligned with the rotary table in the X Axis and perpendicular to the spindle.

7 Select the pick Arrow for **Align Z perpendicular to a horizontal surface** and Click on the horizontal face shown.

8 Select **Ctrl + 5** or **Top view**. If your selected surface is upside down Click on the **Reverse Z** button and Top view again. Select **Next**.

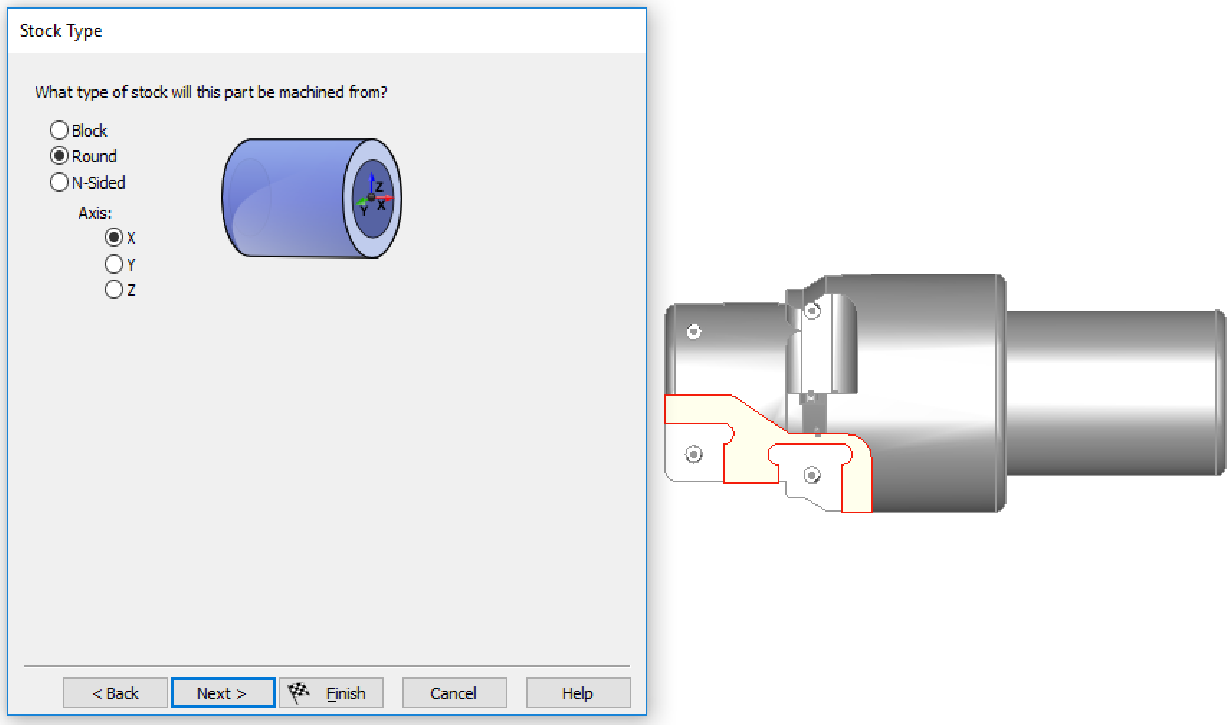


9 The Z orientation is correct but we need to align the X Axis by selecting two points or by selecting **Rotate X direction 90 degrees around Z Axis**. Select this option. Keep selecting the button until you have the part in the correct orientation in X. **See second image below**. **(Top View or Ctrl+5)** This shows the part in the correct orientation.

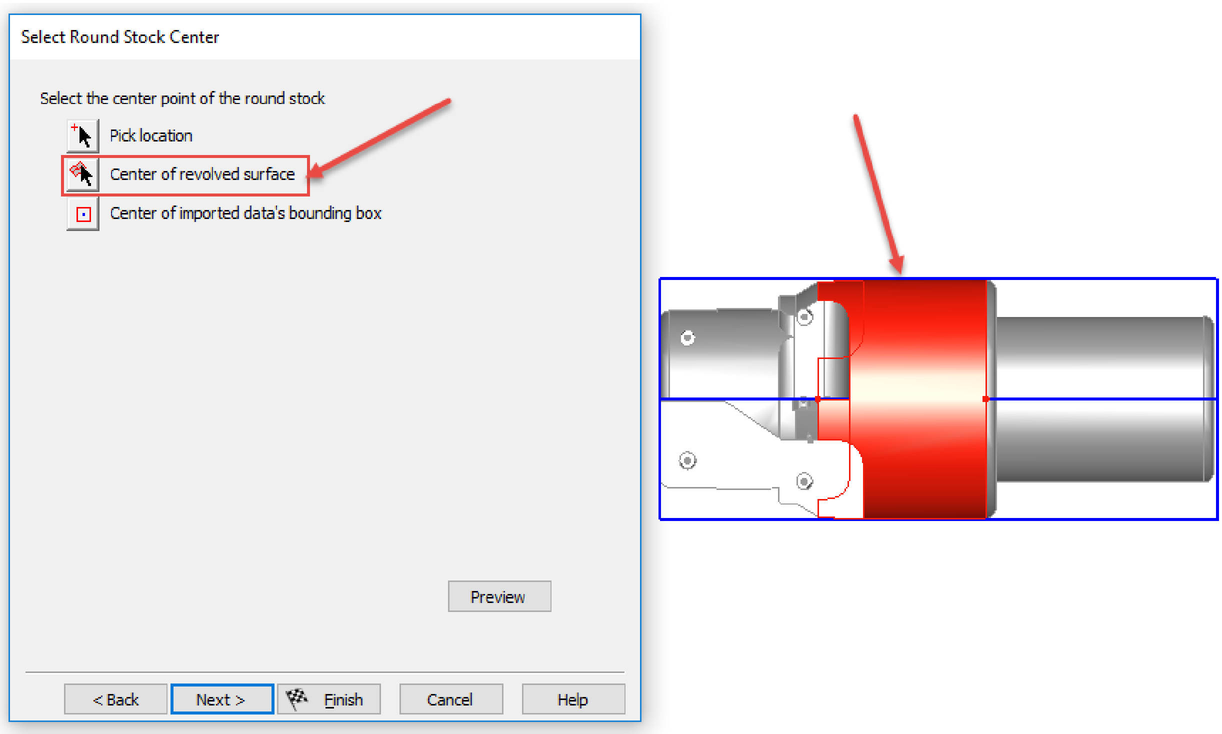


10 Select **Top View**. Pockets are to the left as shown above**.** Select **Next.**

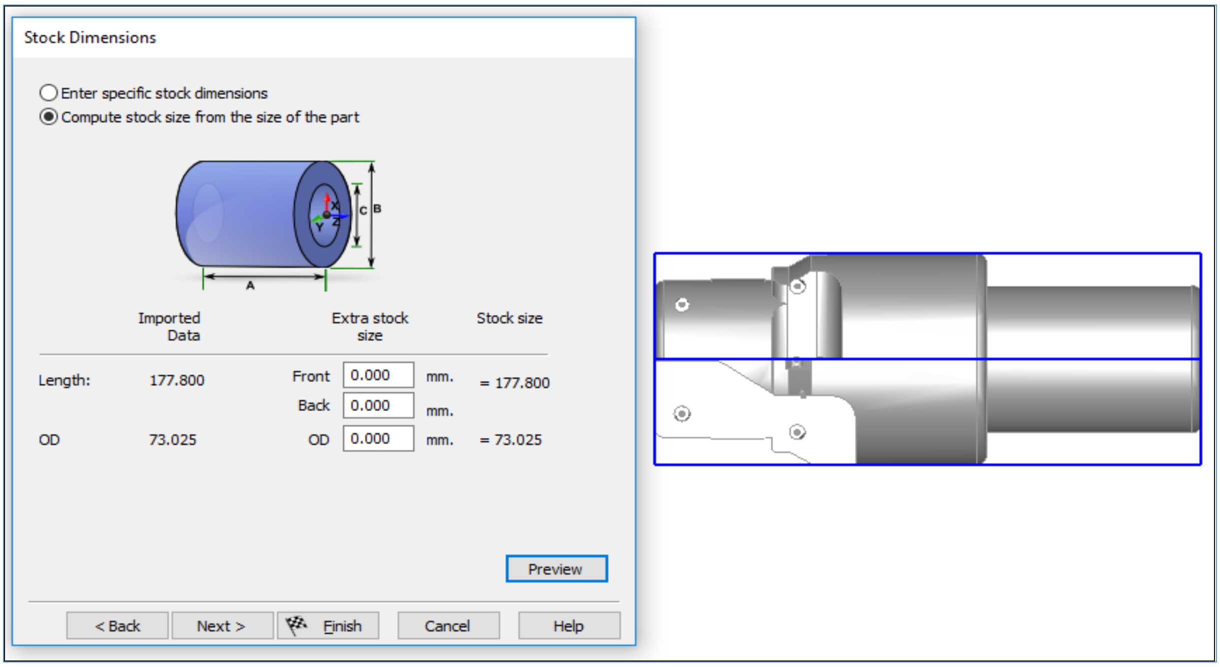
11 You will be presented with the **Stock Type** menu. Select **Round** and the **X direction** as shown below.



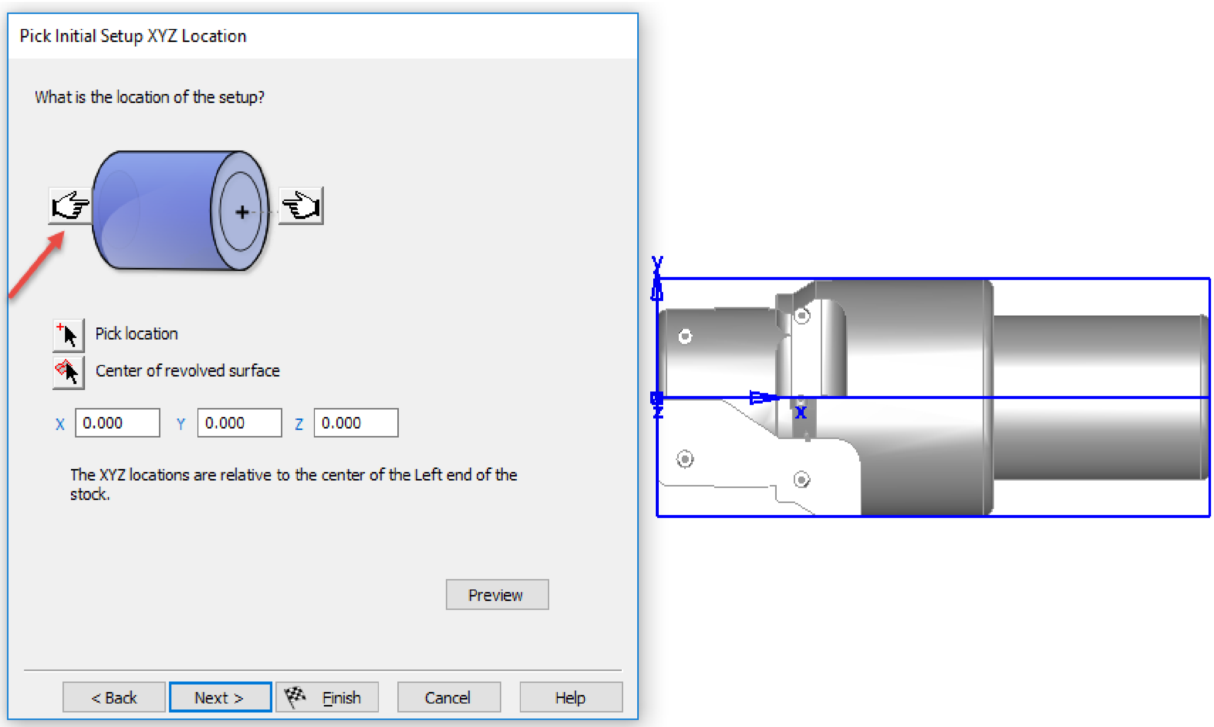
12 Select **Next**. You have the option to select the centre of the round stock. Select this option and the surface shown below.



13 Select **Next.** You will be presented with the option to set the **Stock dimensions**. Although we will be using a solid model for stock later on in this exercise it is useful to create our stock so we can create our **Setup1** in the correct position.



*Please select **Compute Stock from the size of the Part.** Make sure the Extra Stock Size for all options is **set to zero.***



14 Select **Next**.

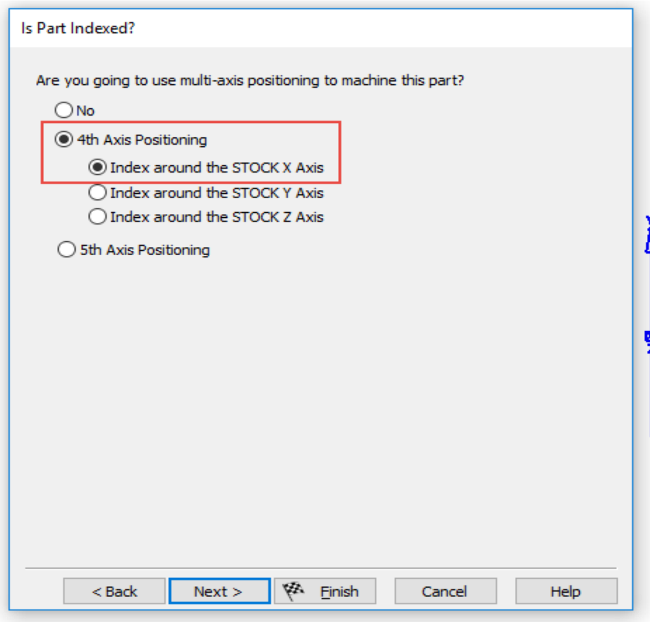
15 We need to locate our **Setup** datum point to the left hand side of our **Stock**.

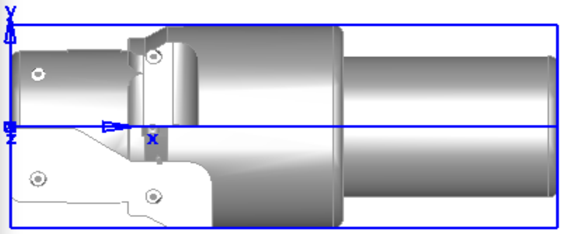
16 Select the left hand pointer as shown on the above image.

*The Setup is now set to the **Stock Axis** position.*

17 Select **Next**.

18 This next menu will give you the ability to set the index axis rotation point. Select the option shown below.



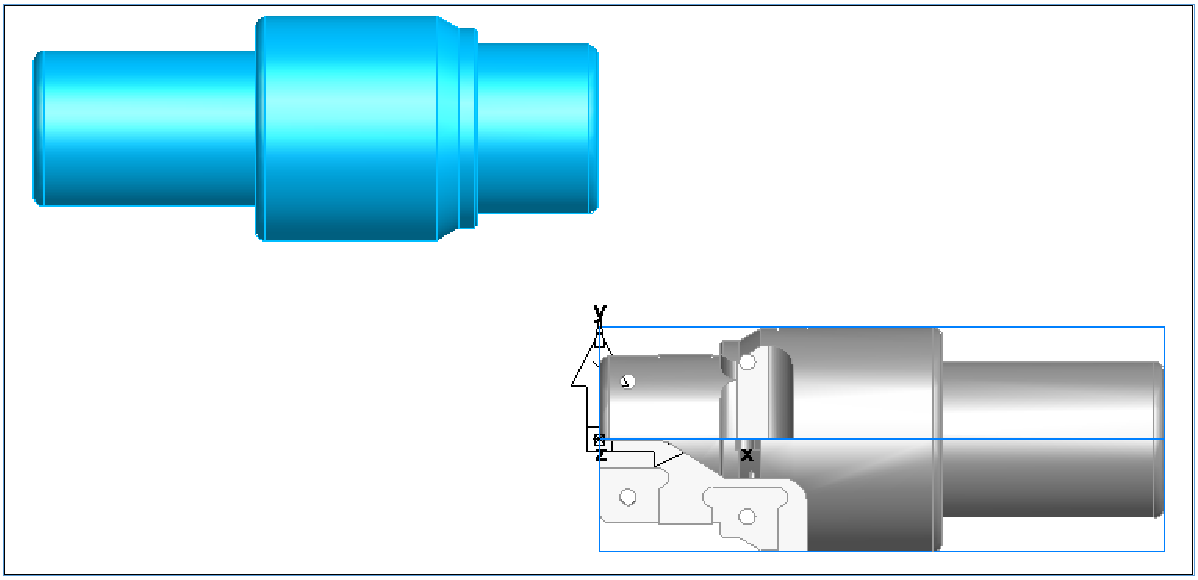


19 Select **Finish**. Then go to View from the main menu and select **Show>Show Stock Axis.**

20 Repeat the Import process for the STOCK model for **2.5D Cutter horizontal slide-STOCK.x_t**. from location **C:\Training_Data\FeatureCAM Course Data 2017\Milling Files to import.**

21 In the Import Results select the top radio button to Use the Wizard and check **Use the same alignment as last import.**

22 Select **Finish** and the part will align to the original model as shown below.

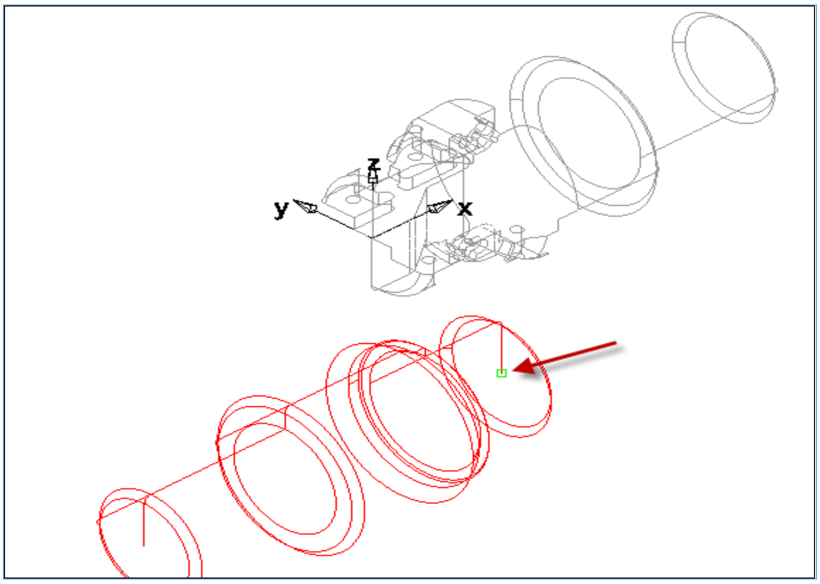


23 The two models now need to be merged and that means you will have to transform the **STOCK model** by **Rotating** and **translating** it to the PART model.

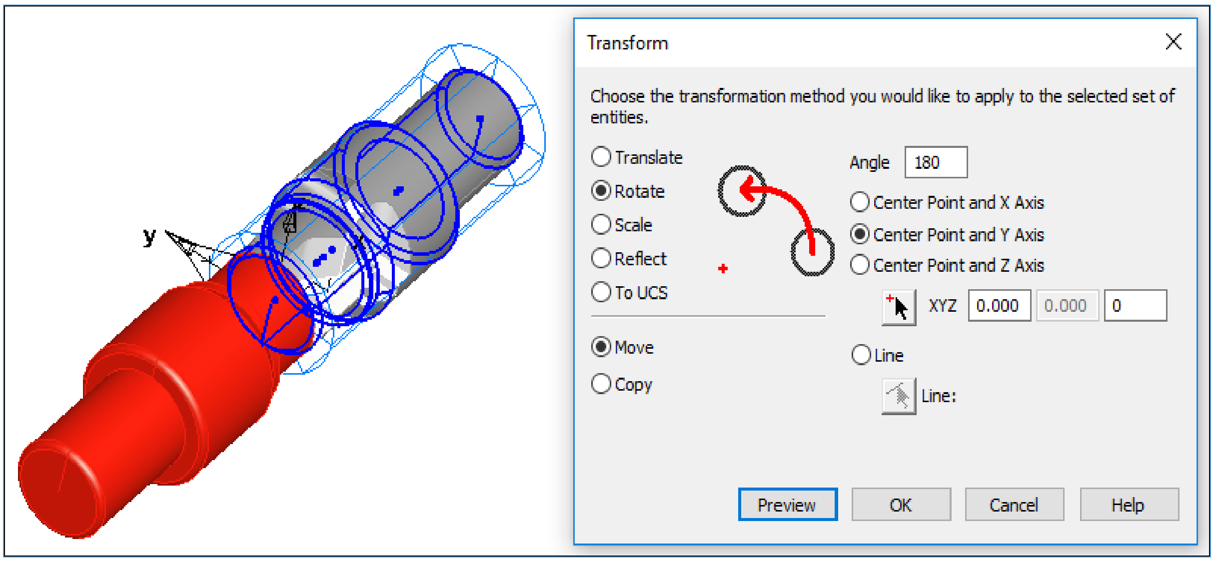
24 Start by selecting Shade Surfaces. 

25 Make sure you have **Snap to end point** and **Snap to object** active.

26 We need to highlight the model from **Part View** and then activate **Transform** from the **Edit** menu. Select **Translate** and **Move**. Select the From Arrow, Snap to the end nearest the datum as shown. To **X0, Y0, Z0**.

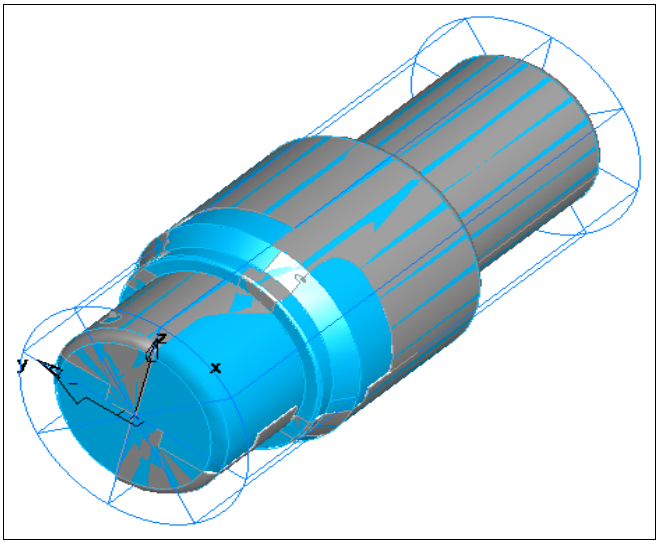


We now have to rotate the stock 180 degrees about the Y Axis.



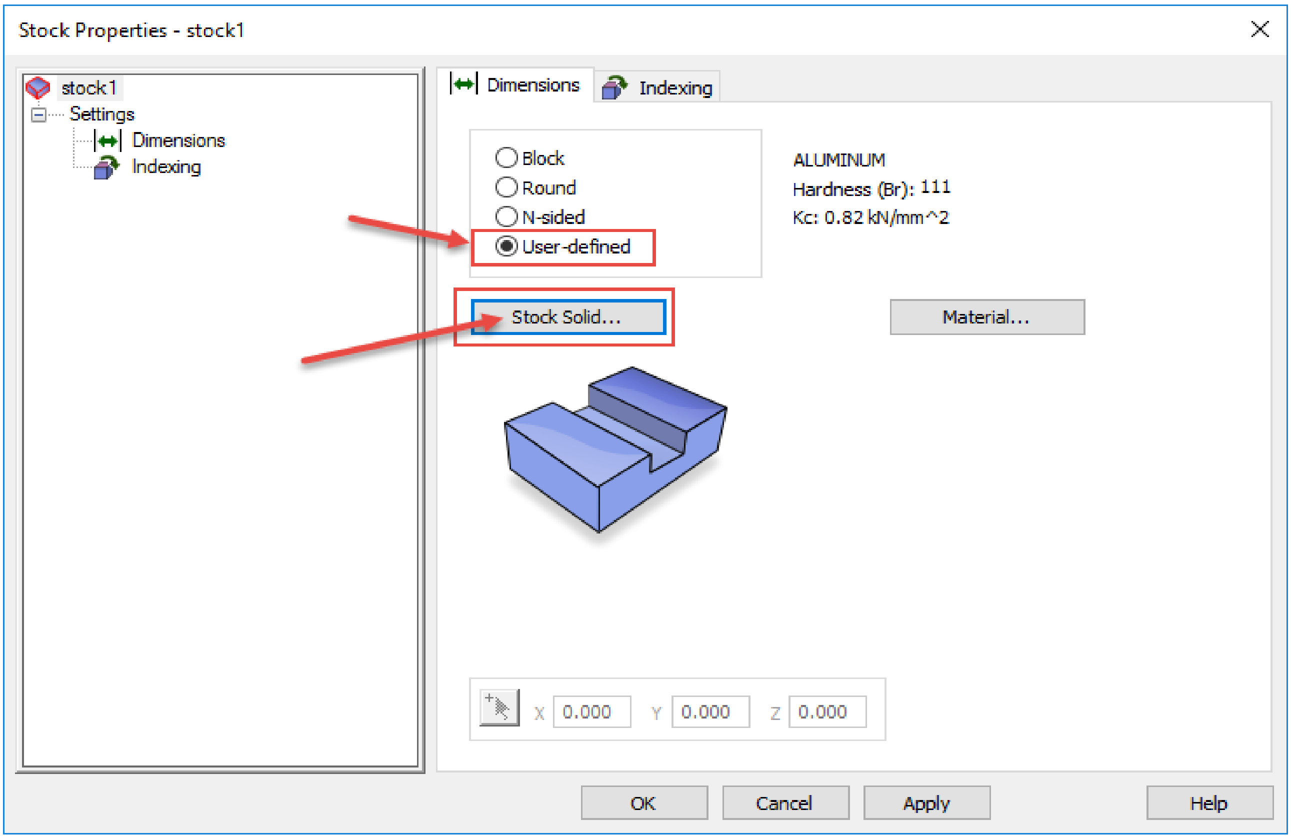
27 Select **Edit>Transform**. Select **Rotate** and **Move**. About the **Y axis 180** degrees.

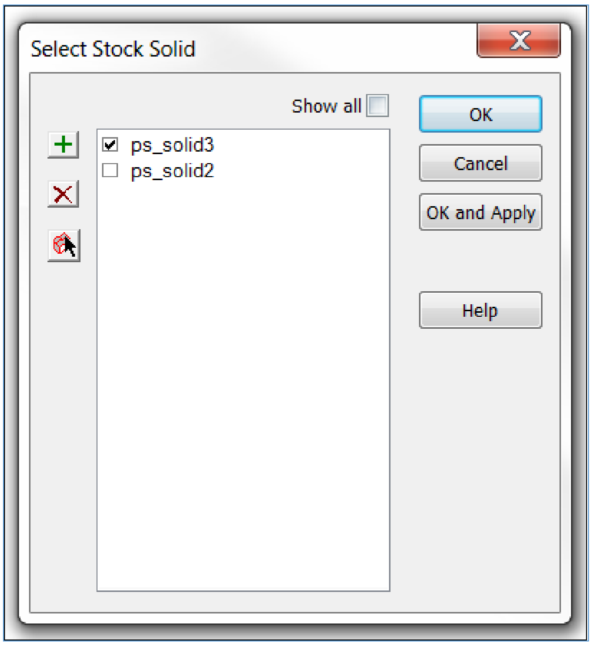
28 Select **Preview**. Then **Ok**. Switch on both Models and you will see them merged together.



***Summary**: The purpose of **Use same alignment as last import** is to take advantage of assemblies and multiple models that are created and aligned or merged together when created in CAD. If the parts are not aligned when created, it does not matter because **FeatureCAM** provides all of the necessary tools to align the parts. So you use the **Import wizard** to **Align** completely or as close as possible and finish off with these tools.*

29 We must now change the **Stock** solid model to Stock. Double-Click on the wire frame of the stock in the graphics window. Double click on Stock, select **User defined** and select the box for the **Stock Solid**.



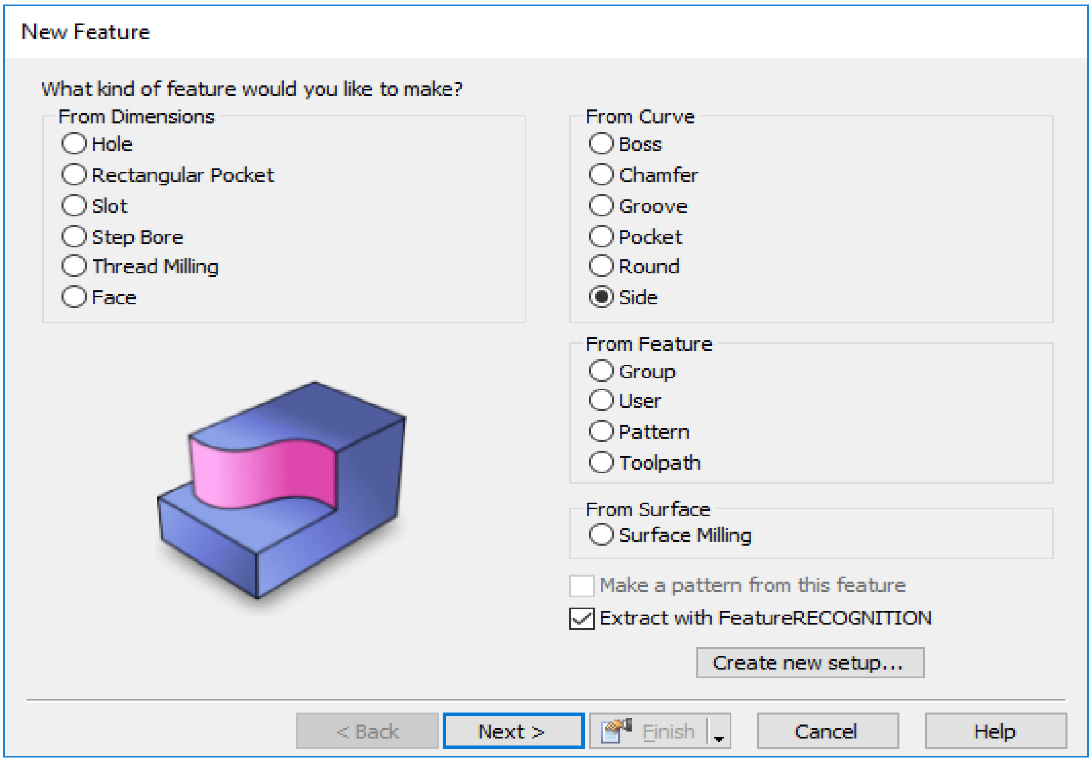


30 Select the solid model then select **OK** and **Apply.**

31 The main cutter solid model is now ready to be machined by applying **Interactive Feature Recognition.**

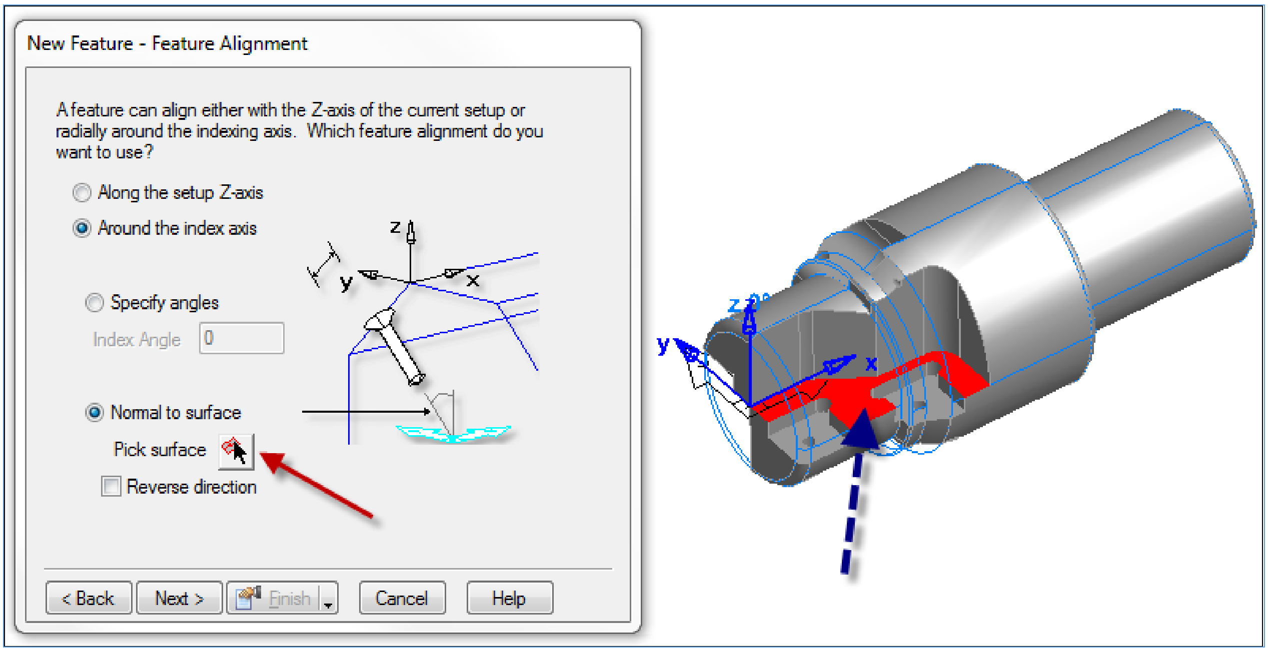
32 From **Part View** select the **Stock Solid** in the list and Right Click on the model and select **Hide** Selected. This is necessary so we just see the PART model we are going to machine. The **STOCK** model will be visible when running the **3D Simulation.**

33 Switch to Isometric view or select **Ctrl** + **1**. Select **Ctrl** + **R** or select **Steps** and **Feature**. Select **Side** from Curve and check **Extract with Feature Recognition**. Select **Next.**

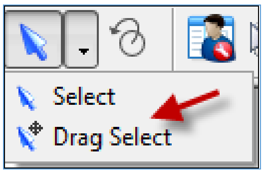


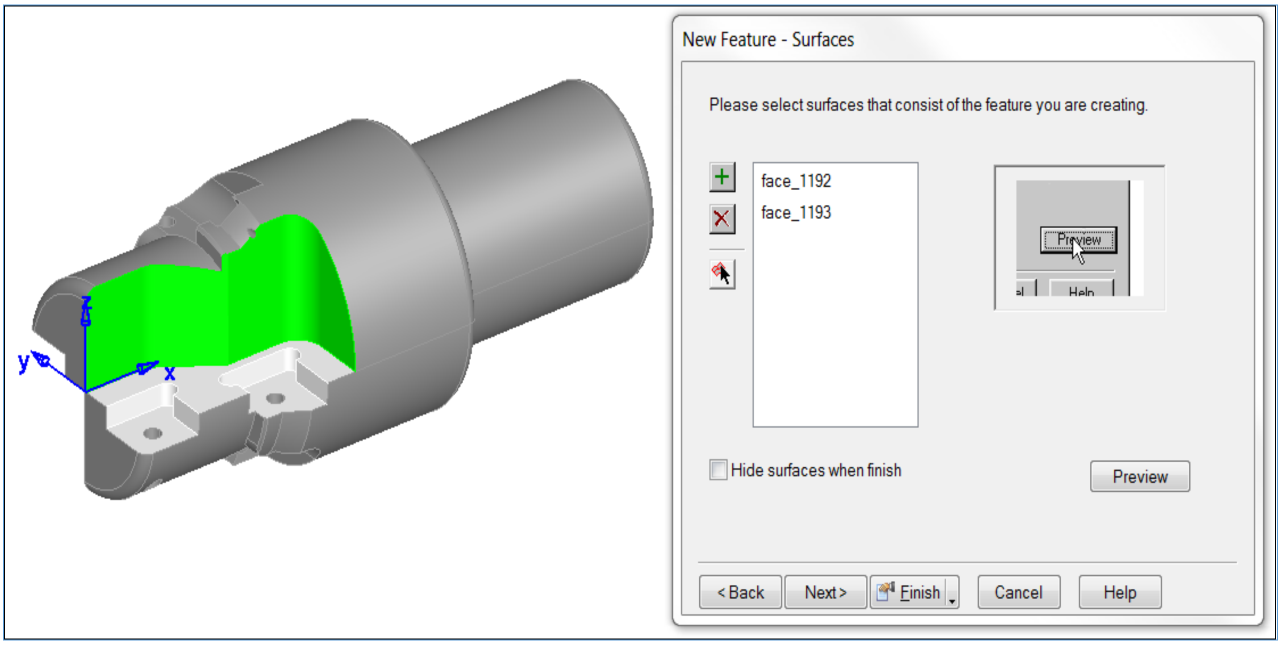
34 Select **Side** and **Extract with Feature Recognition.** Select **Next.**

35 Select **Around the index axis** then Select the **Normal to Surface** radio button. Select the Pick Surface Icon and Pick surface as shown below.

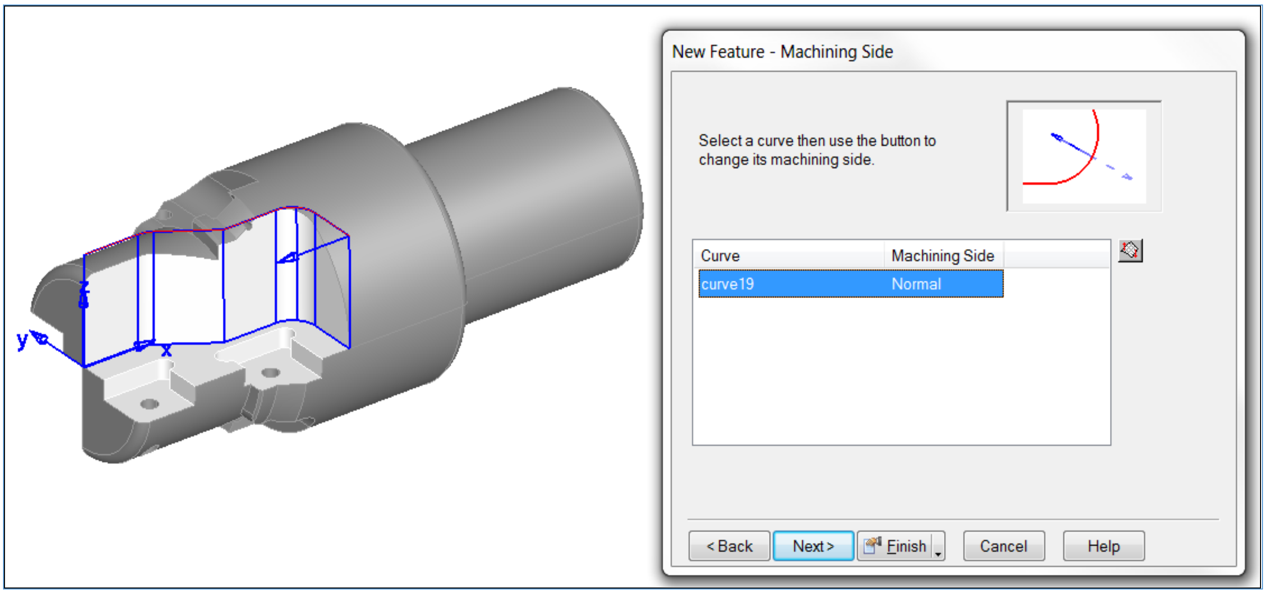


36 Select **Next**.

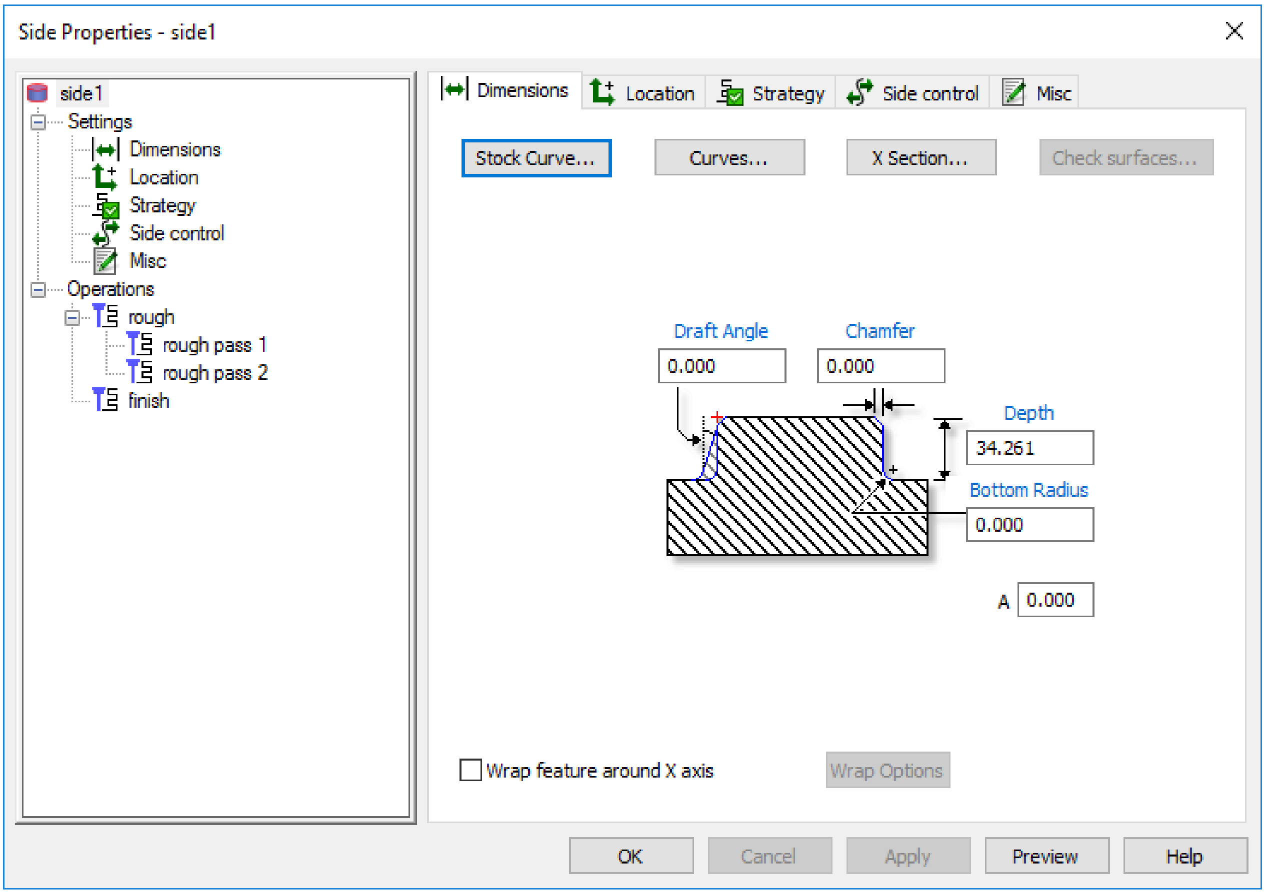
37 Use the **Drag select Icon**  Select **Surfaces**. Select **Next**.

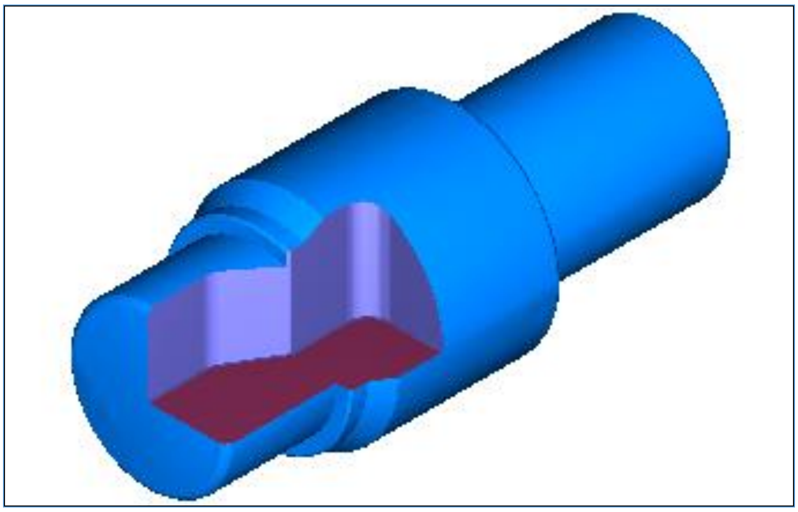


38 Select **Next**.



39 Make sure the direction arrow is pointing in the correct direction. If not select the **switch machining side** icon. Select **Finish** and then **Ok** to close the form.

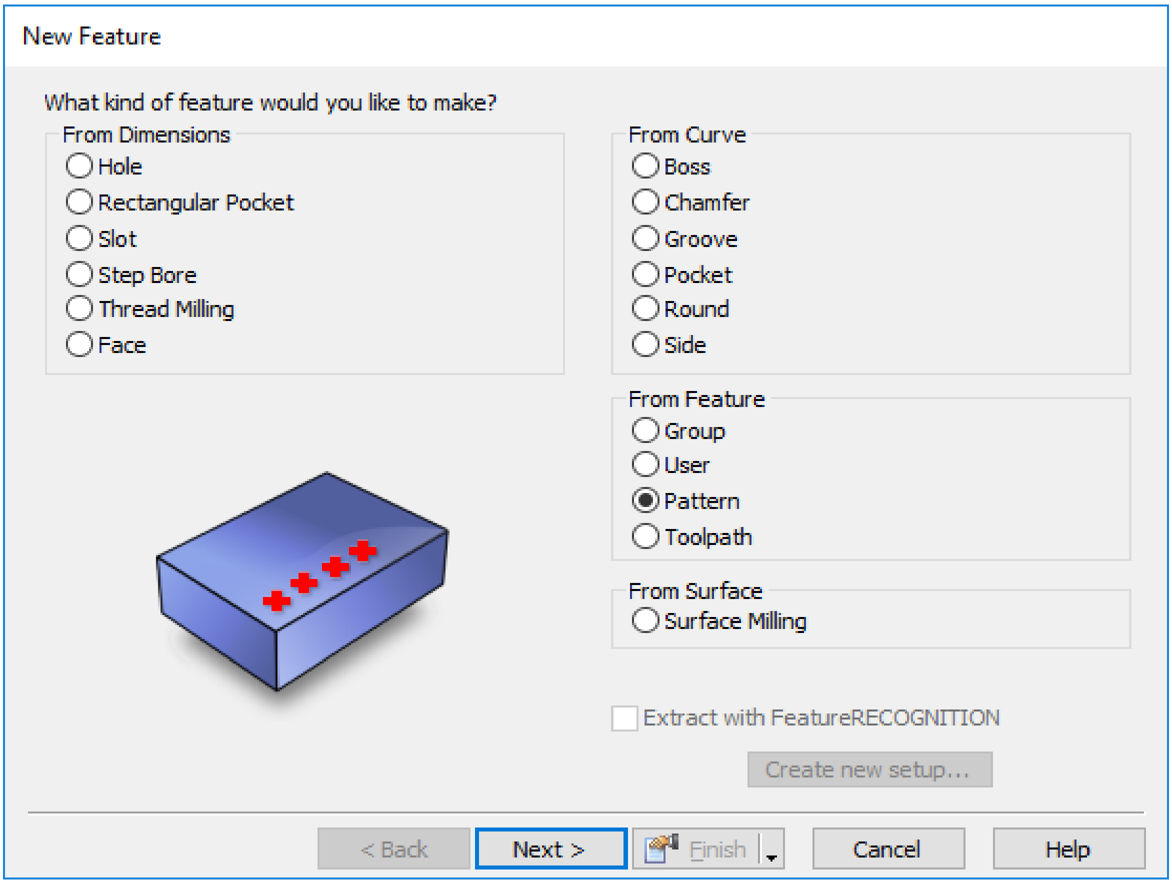




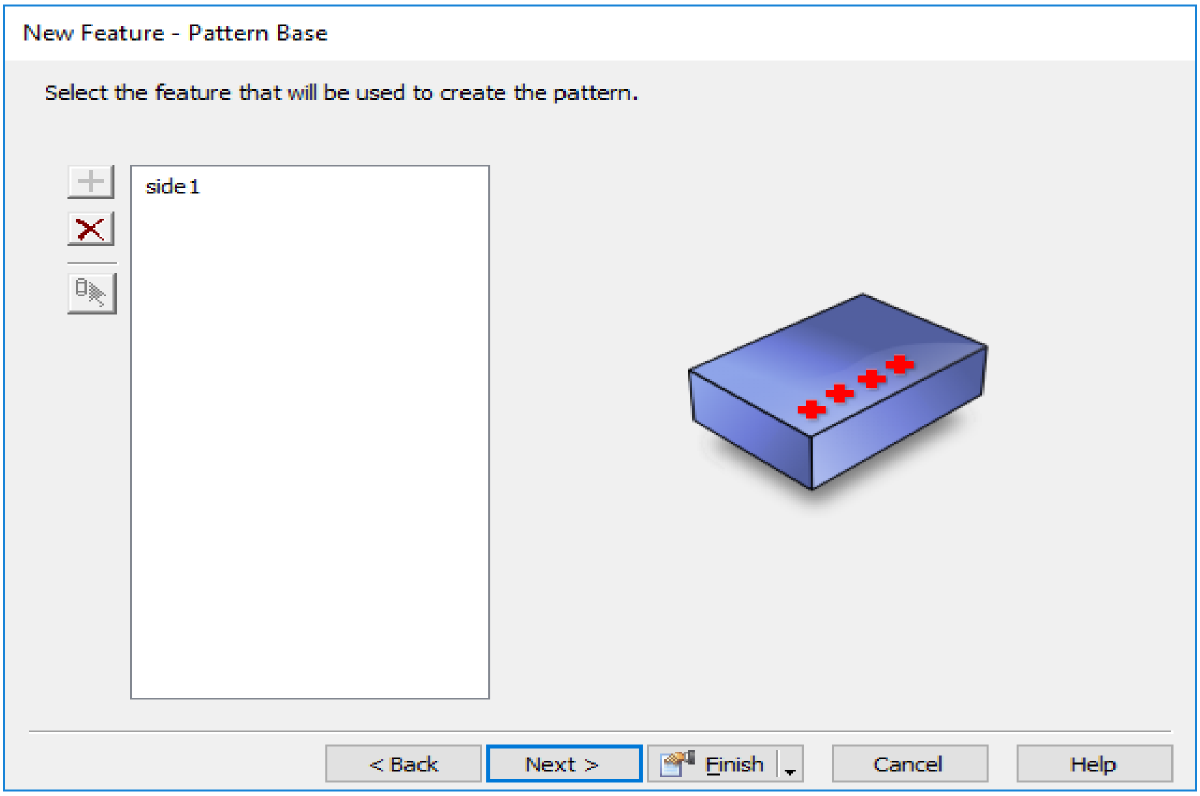
40 We can now make a pattern from the previous feature but should only be used when you are absolutely certain that the Features are identical and symmetrical.

41 Press **Ctrl** + **R** this will present you with the **New Feature** Form. **Select Pattern**.

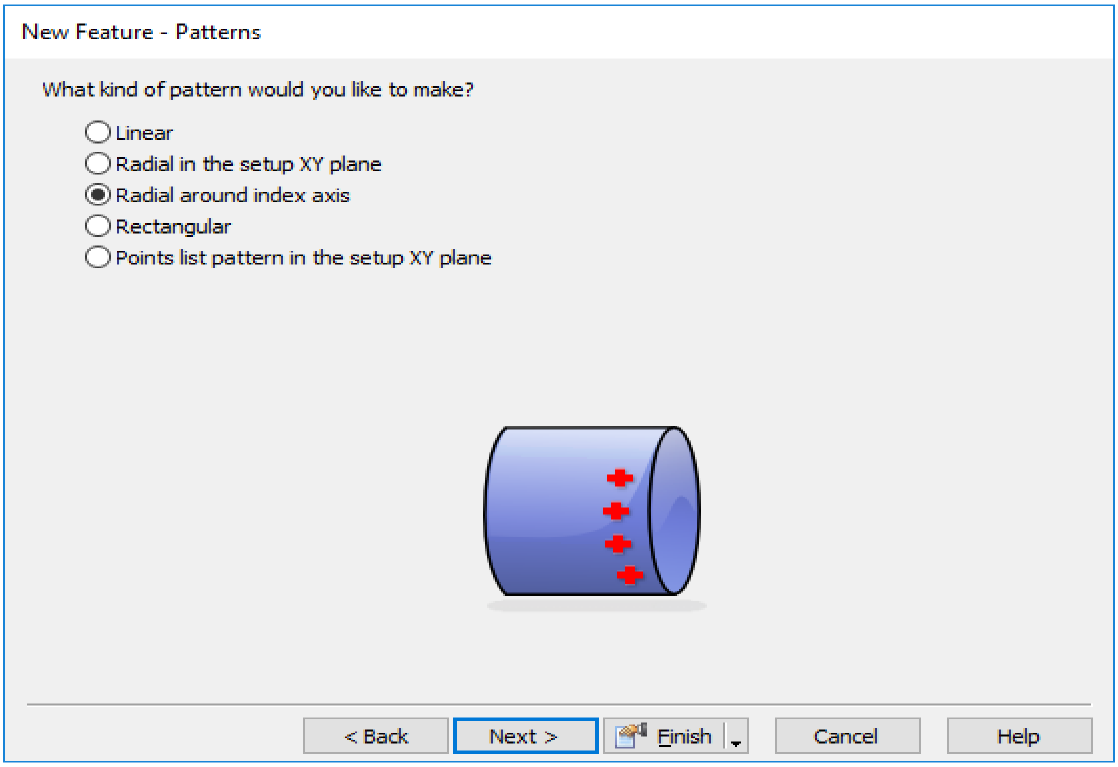
42 Select **Next**.



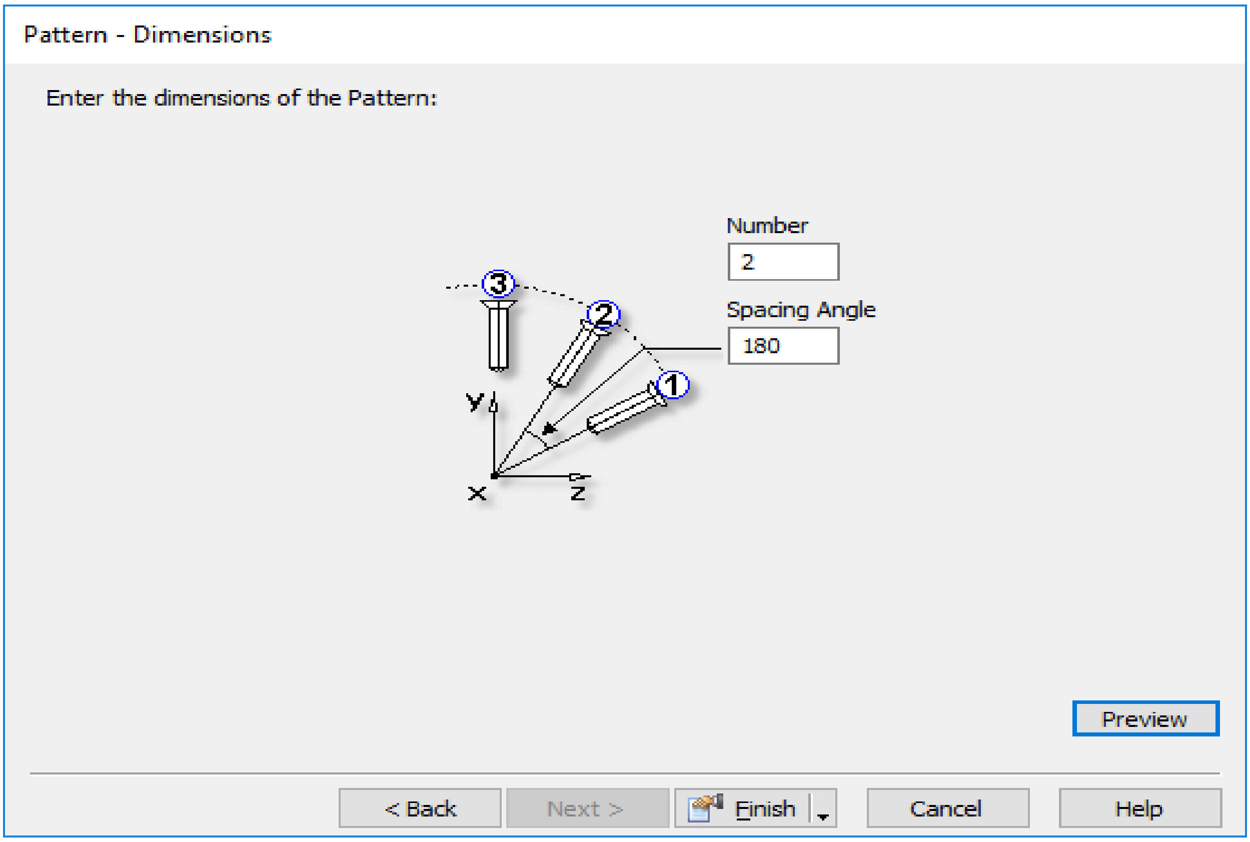
43 Select **Side1.**



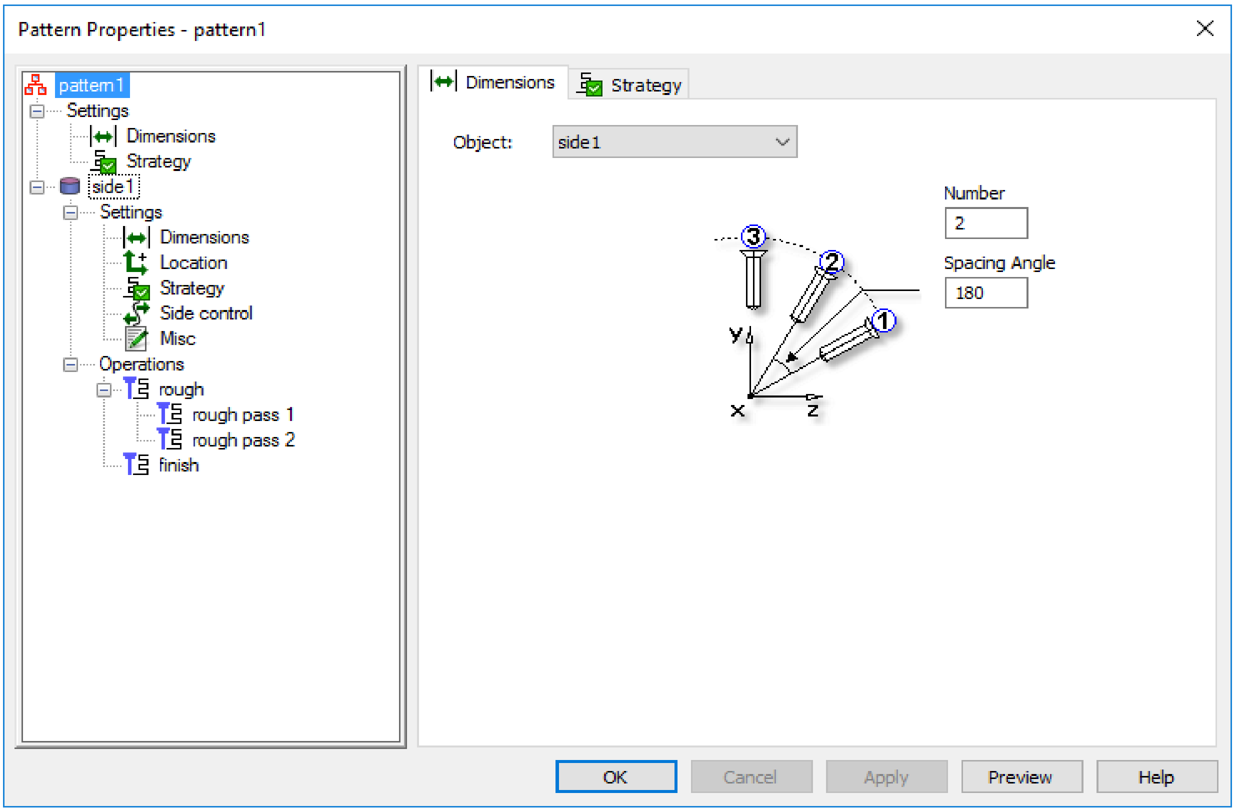
44 Select **Next**. Then select Radial around the index axis.

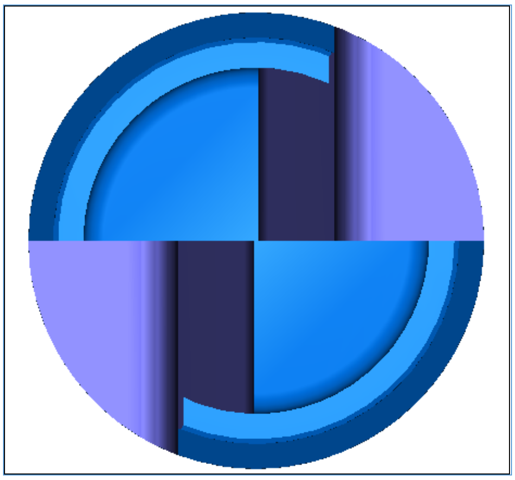


We need two copies, which include the original, rotated at 180 degrees

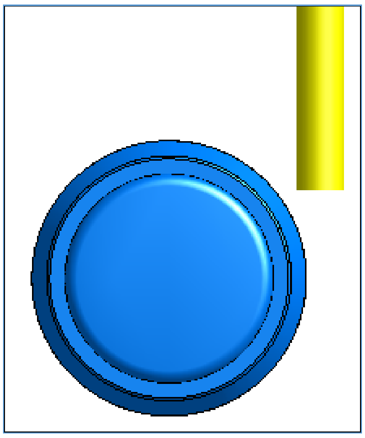


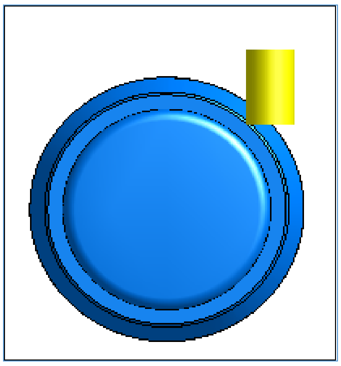
45 Select **Finish.**





- When milling cylindrical parts the side feature has to extend outside of the Stock far enough to satisfy the bottom of the feature but at the top, because of the shape of the material the tool is cutting fresh air on some passes.
- To eliminate this, under the strategy tab in the milling feature select Individual Rough levels, apply and OK.
- This creates control at each Z increment depth of cut and follows more closely to the shape of the material on each level. Higher passes start their passes closer to the material and move out as the tool descends. FeatureCAM has also selected three tools now, the first pass is the shortest tool and each pass changes to a longer tool. You may override this by selecting the same tool for all three cuts.
- This will also make the part index on each level but can be eliminated by selecting "Cut higher operations first" from the results window (right side of graphics window), click on Automatic ordering options button at the top. This works only if the features are individual not a pattern.

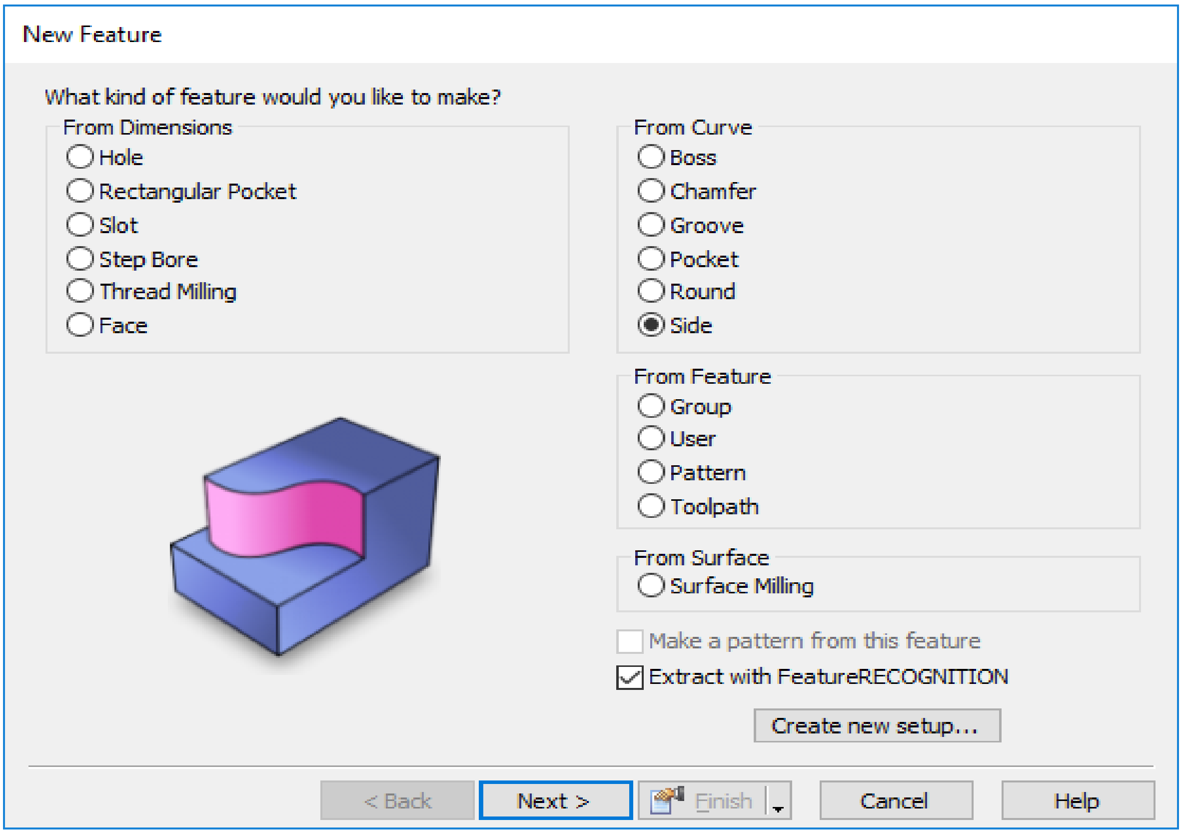




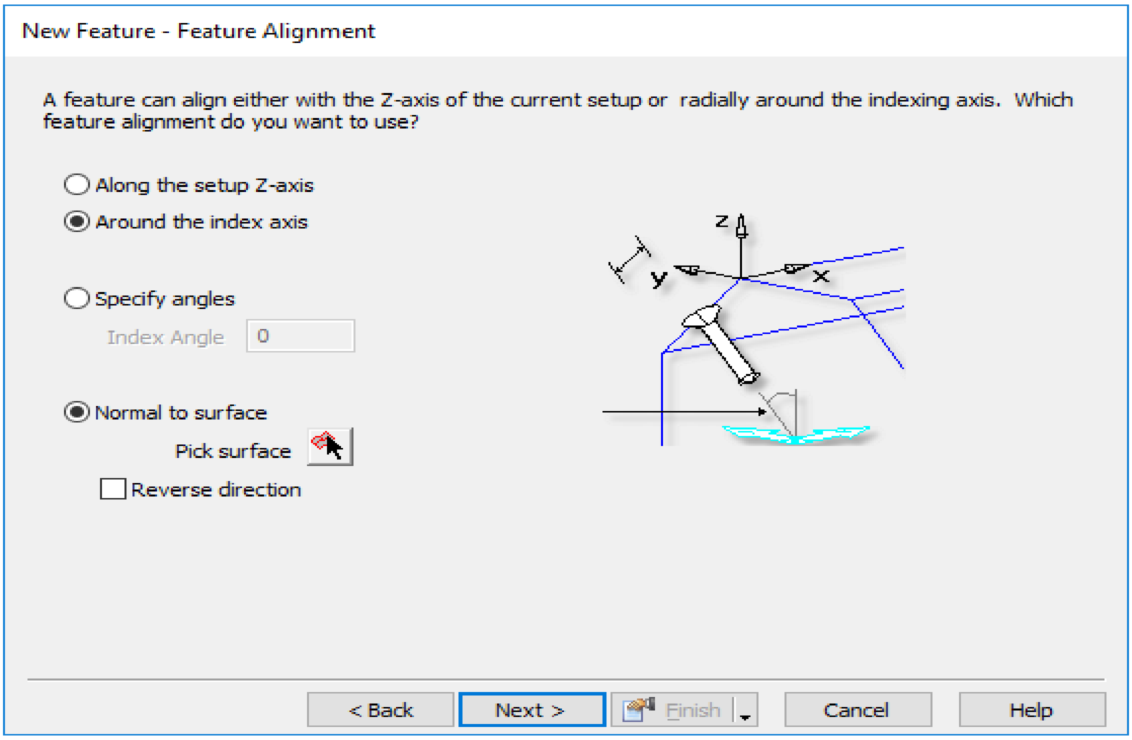
Rotate the View so you can see the horizontal surface of the feature between the previous features just completed. Continue with Side Feature using Recognition and select the arrow "Normal to surface". Click on the horizontal surface that you rotated to. The vertical surfaces that ***FeatureCAM*** *is looking for are Normal to this horizontal surface.* ***FeatureCAM*** *uses the horizontal surface to index to and looks for any vertical surfaces with Recognition at this angle.*

*Next we are going to machine the two smaller pockets that are 90 degrees to the machining we have just completed. Select **Ctrl** + **R** to select a new feature.*

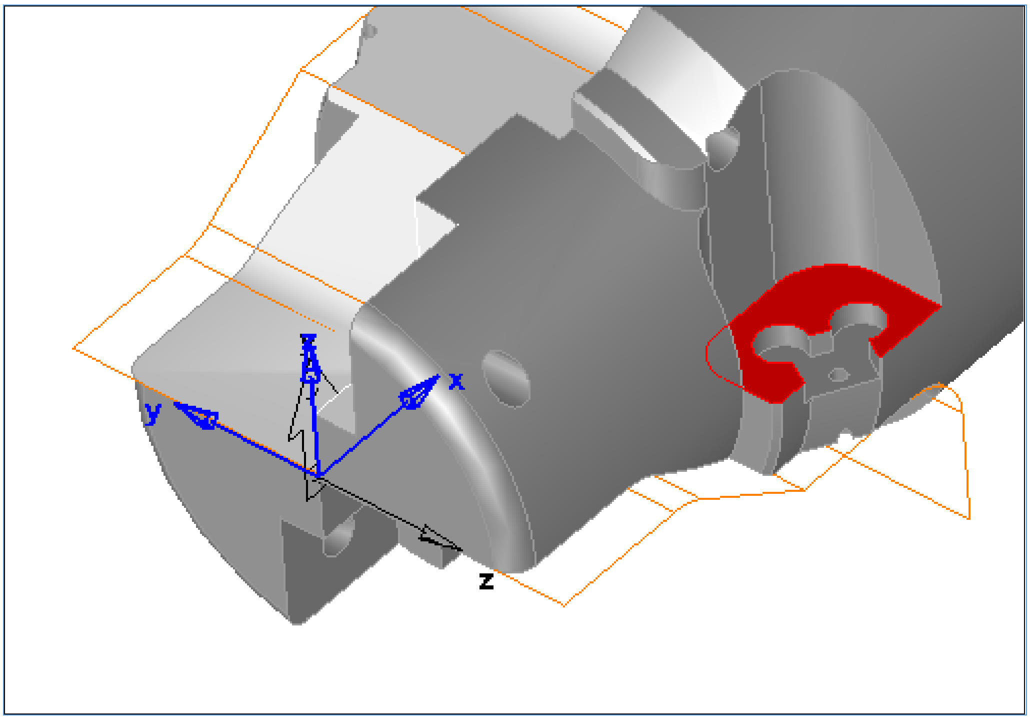
46 Select **Side** and select **Extract with Feature Recognition**. Select **Next**.



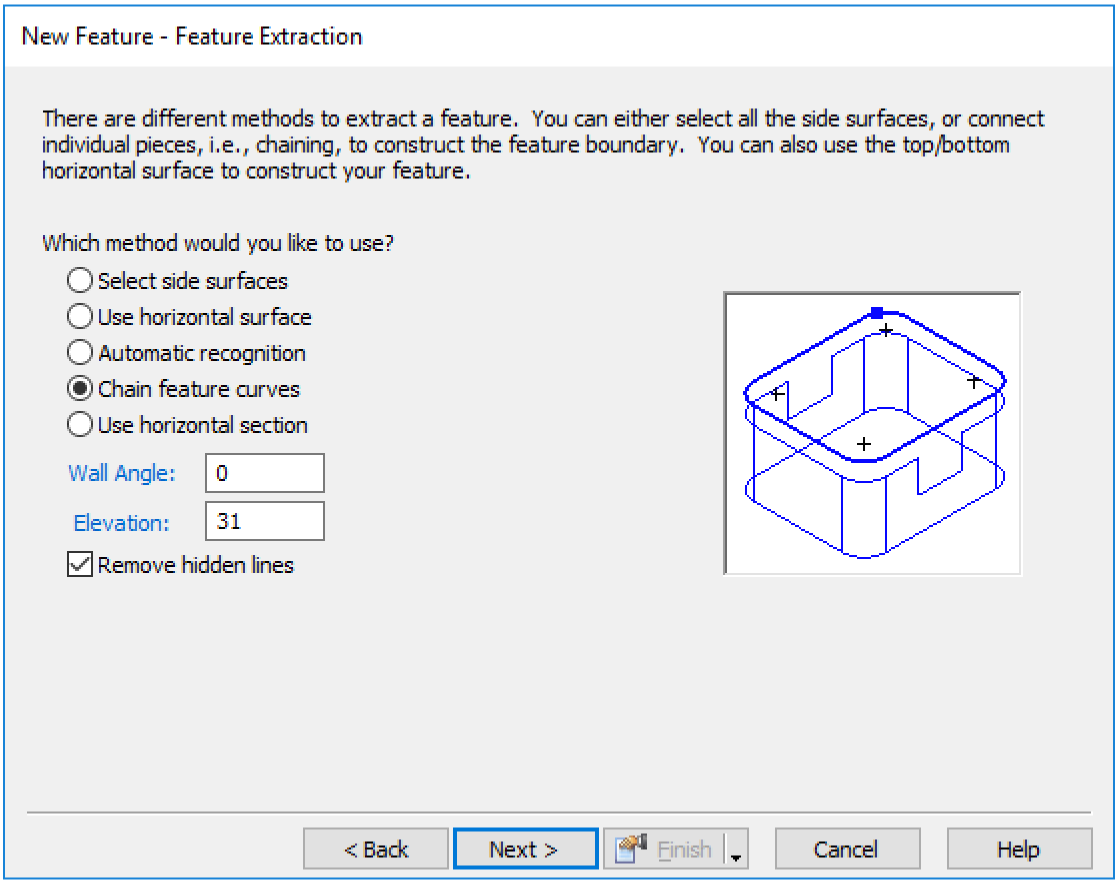
47 Select **Normal to surface** & select the Pick Arrow then Pick surfaces. Select **Next.**



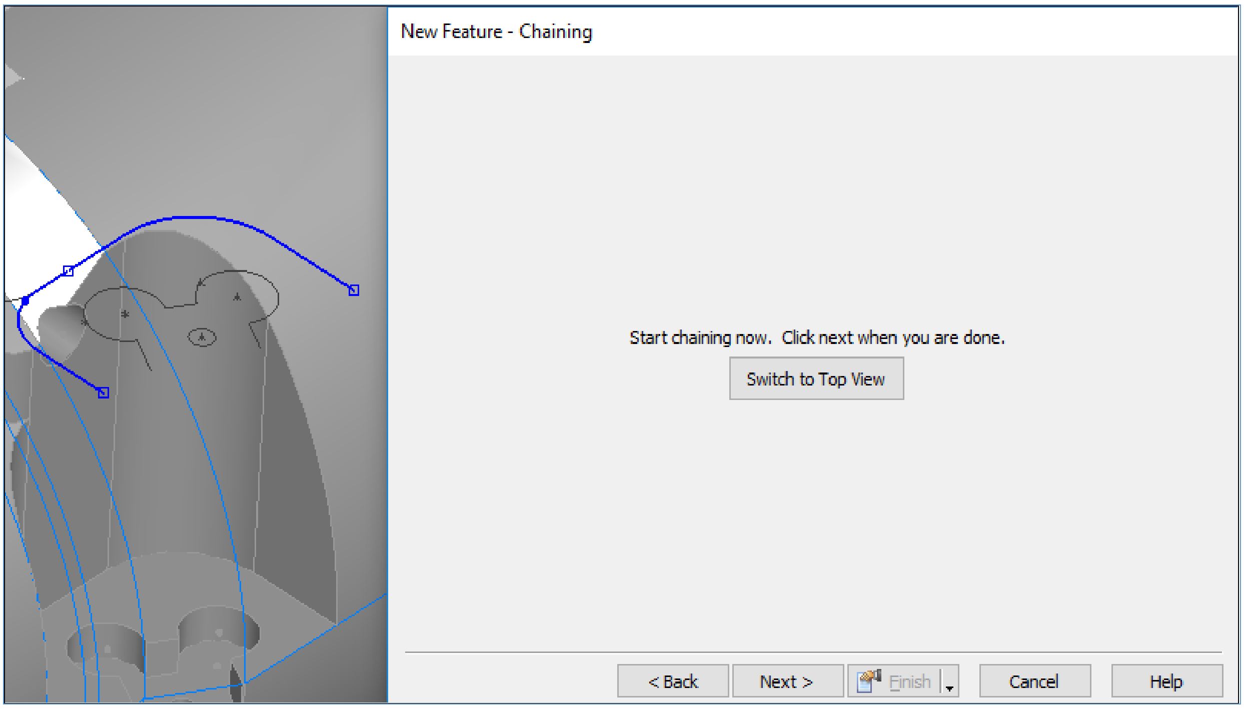
48 Pick the surface as shown.



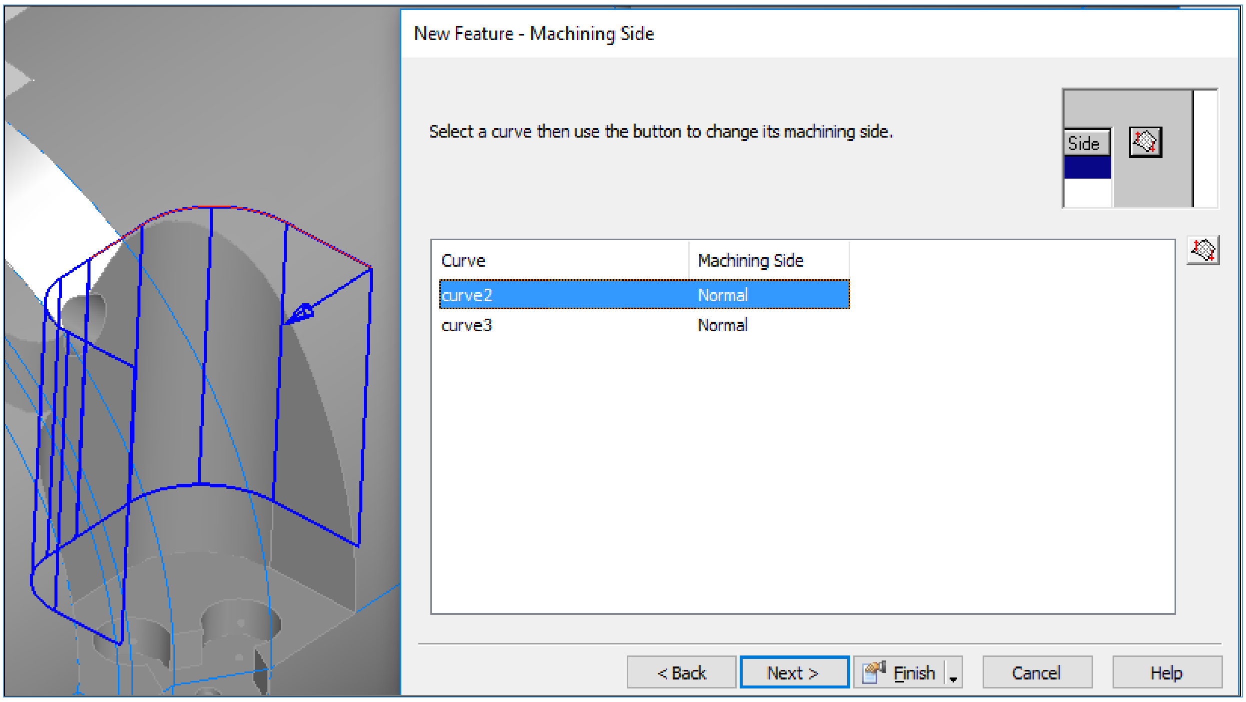
49 Select **Chain Feature Curves**. Set elevation to **31mm**.



50 Select Next. Chain around the open profile as shown on the next page.

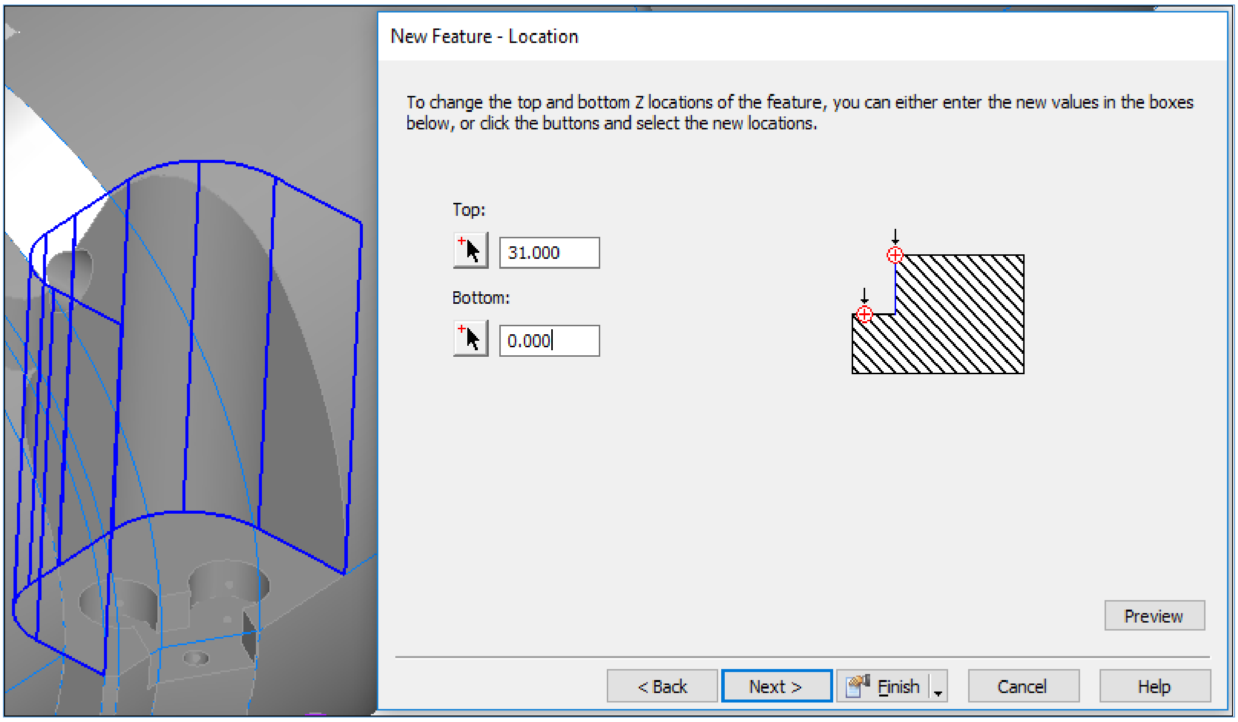


*If you have difficulty with a section of the profile select smaller sections until you have completed the profile. Select **Next** and select the machining side.*

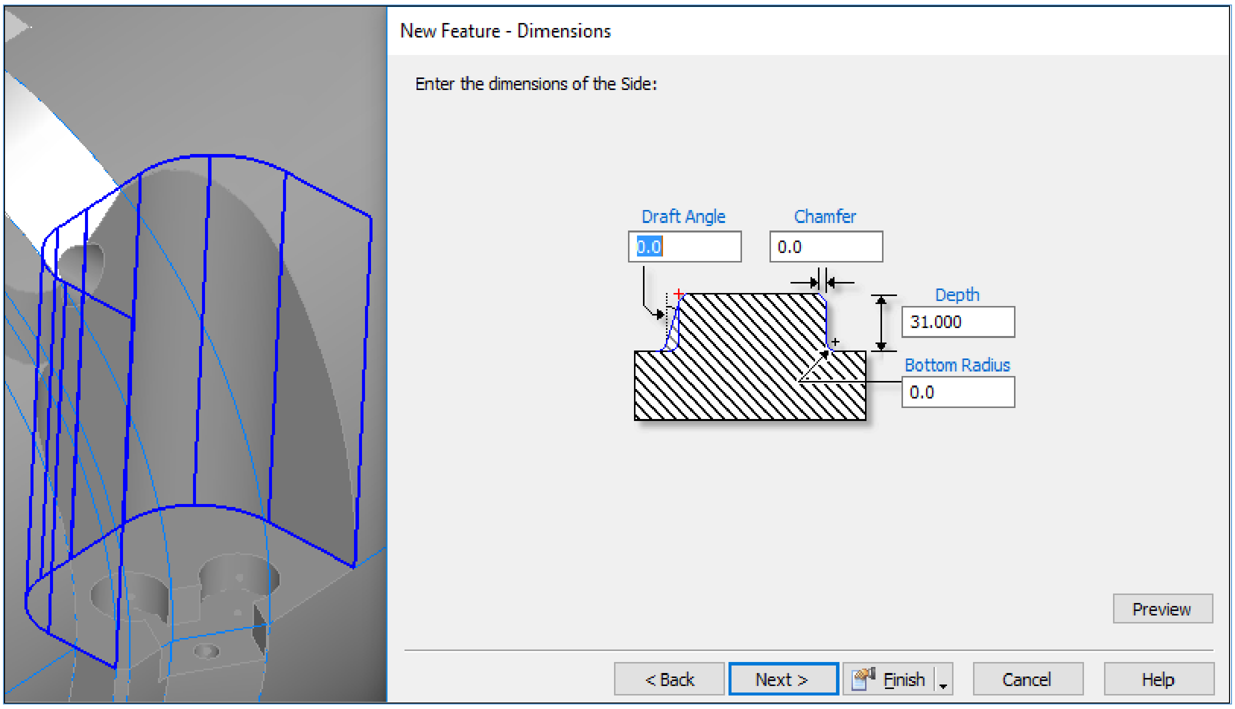


51 Once you selected the machining side select **Next** to see the location menu.

52 Set the bottom dimension to **zero**.

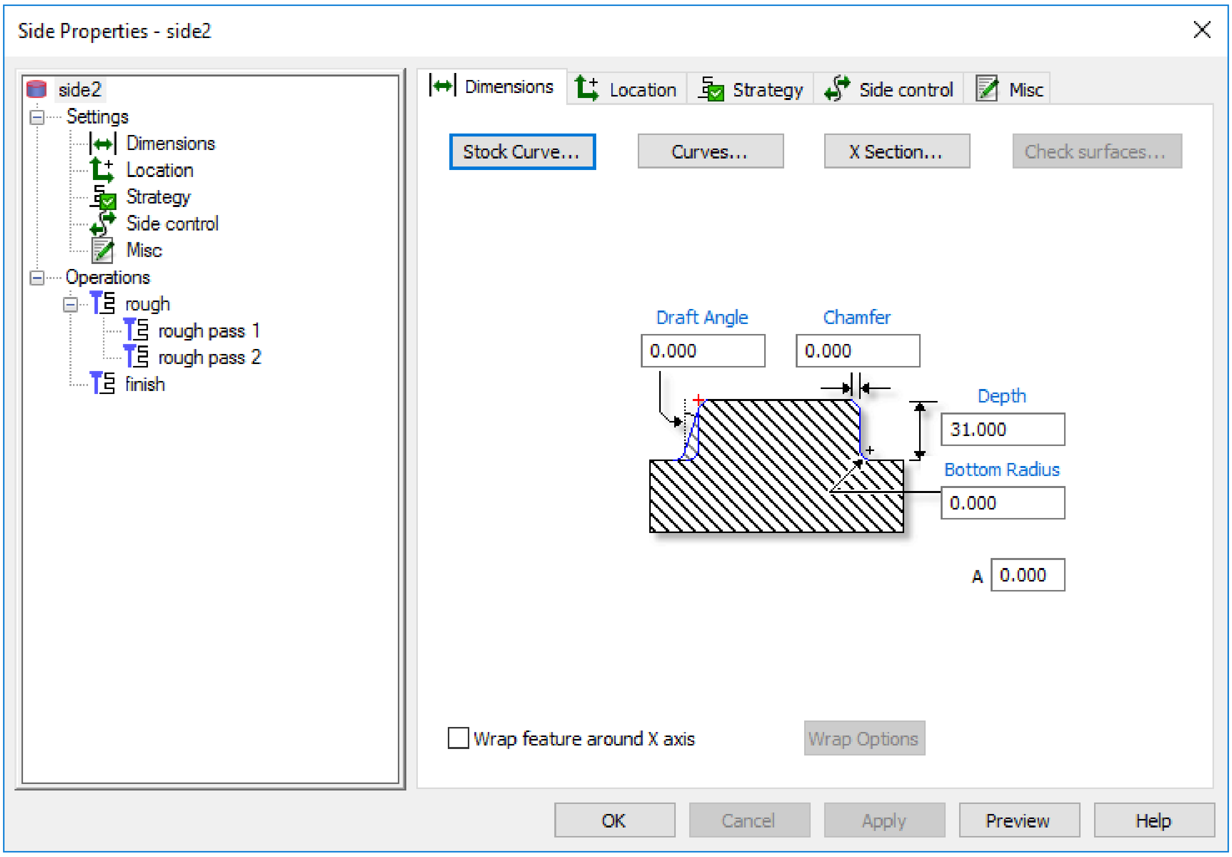


Select **Next.**

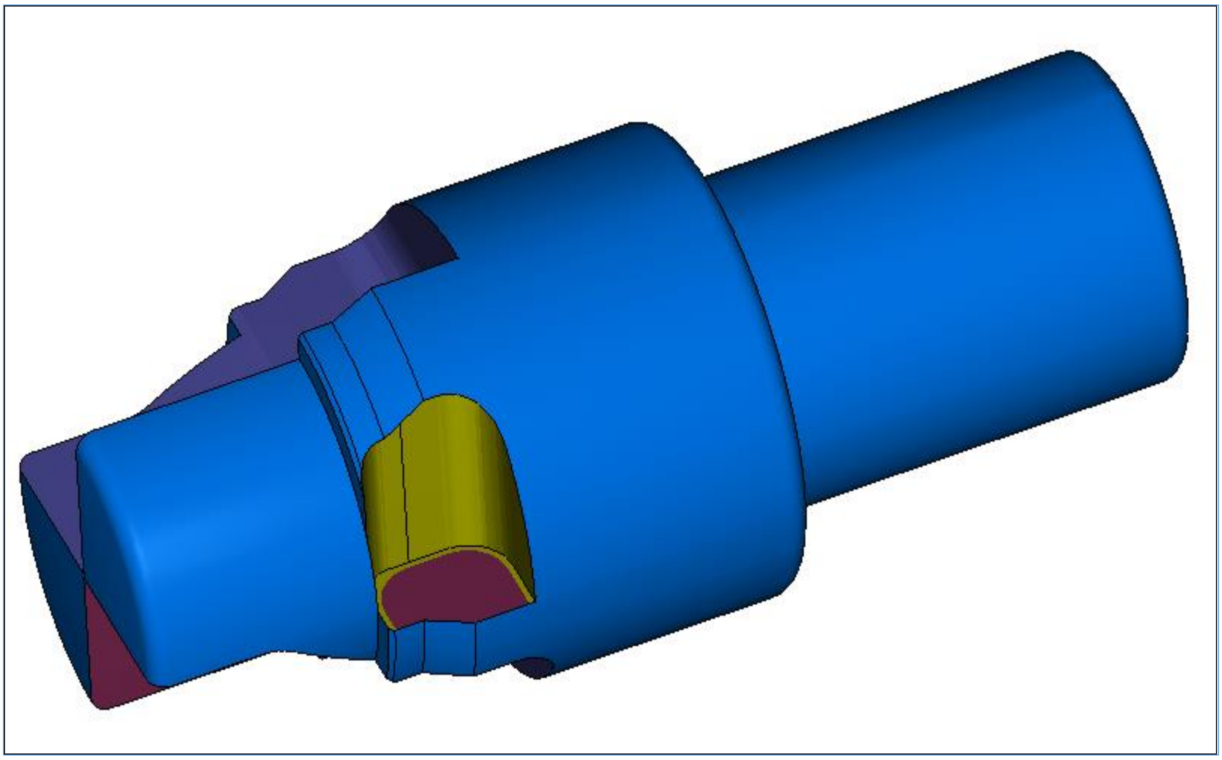


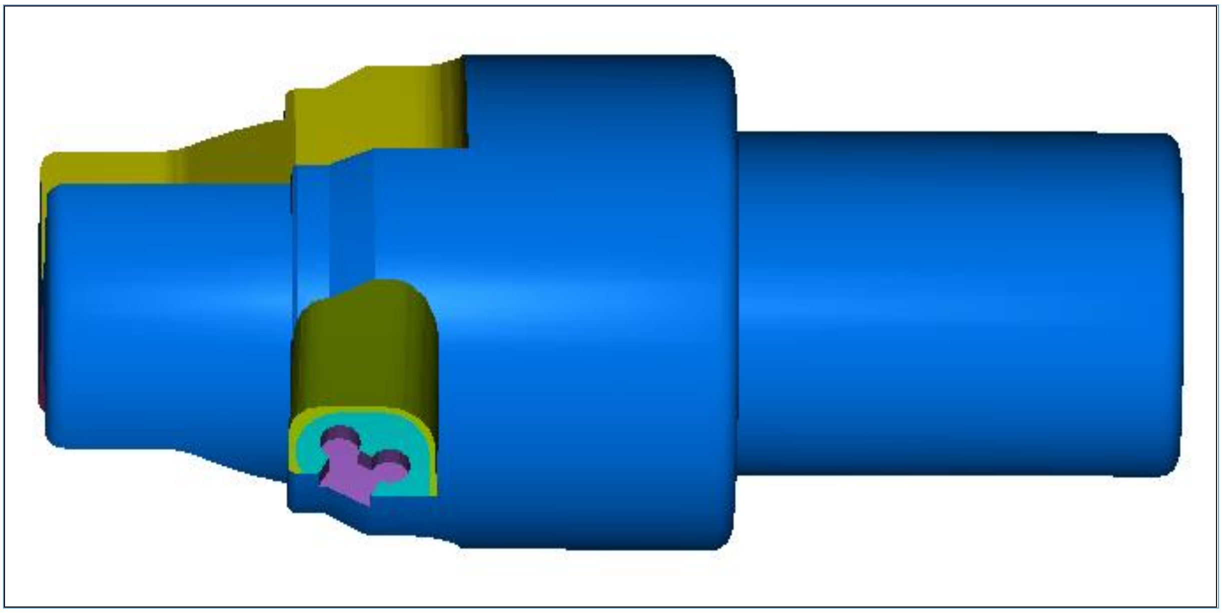
*The main reason for using **Side** with the **chain** option is that the profile will be completely machined compared to using side surfaces or Horizontal face.*

If you do use one of the other options you could always extend the curve afterwards which will increase the feature machining area.

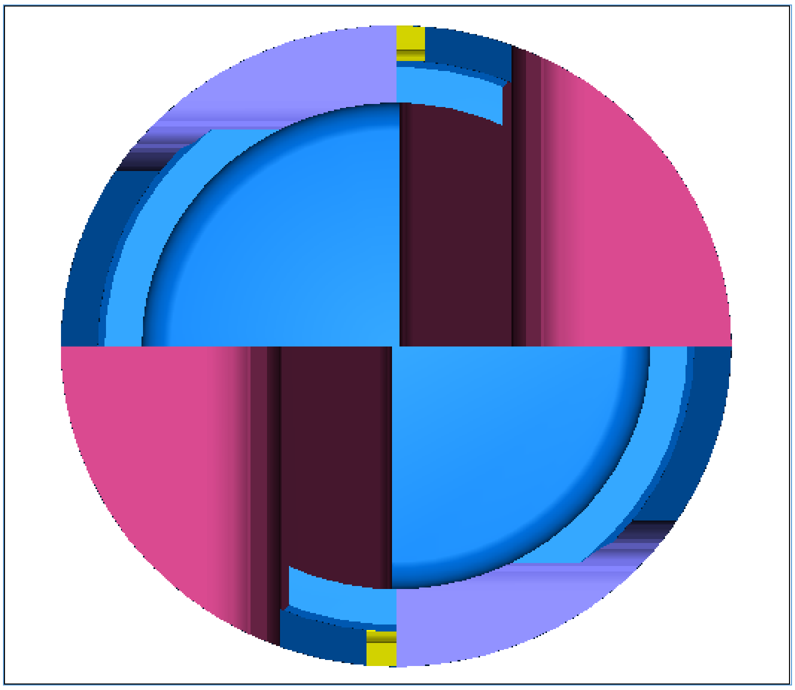


Apply the same technique to the smaller pocket or use Horizontal section.





53 Create a **Pattern** of the **Group**. Around the **Index Axis**.

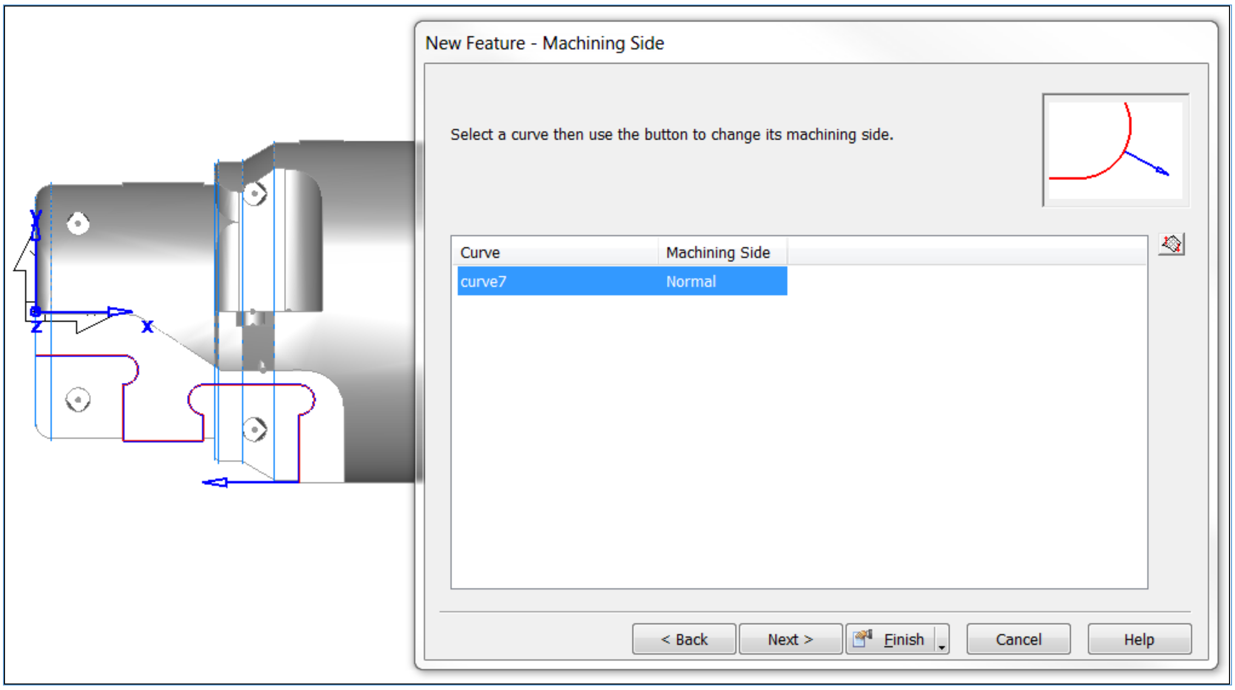


54 Create a New **Side** feature.

55 Select **Next**

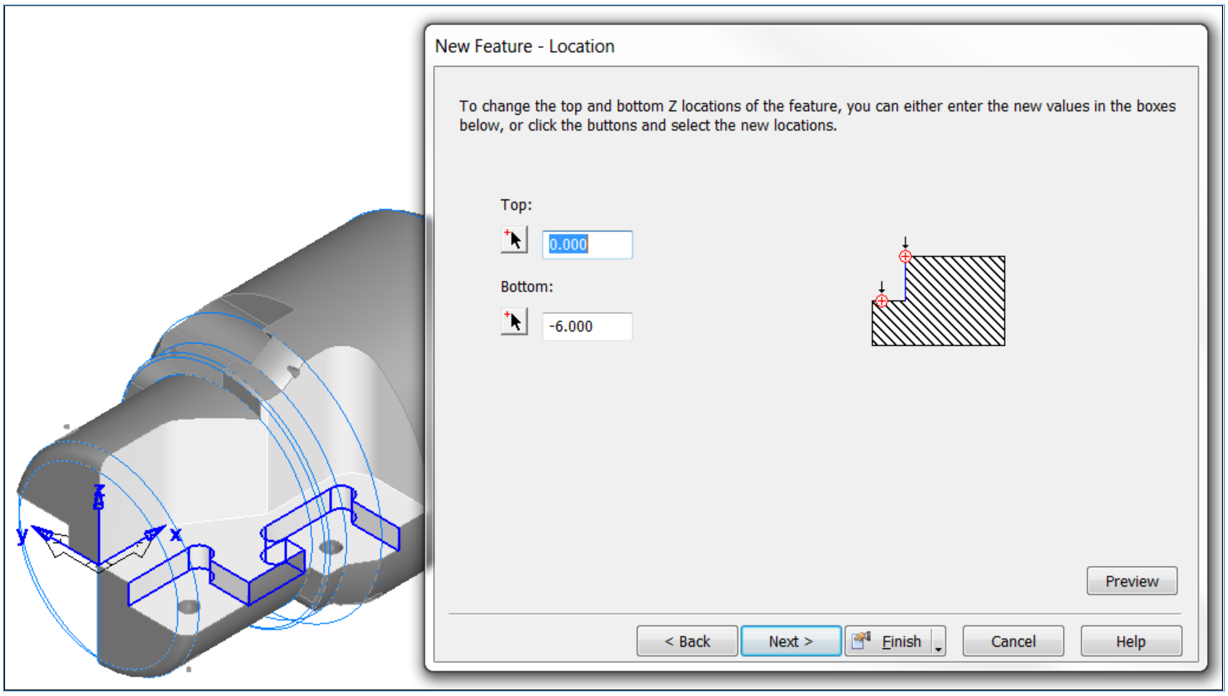
56 Click on **Slice location arrow** and Click on the bottom of the same vertical face and this sets the Bottom of the feature. Select **Next.**

57 Click on Switch to Top View and the part moves to a top view. The geometry is readily visible and must be chained with an open curve. Chain them with two open curves, not connected. Click on Create at the bottom in the geometry dialog area after each open curve. Select **Next.**

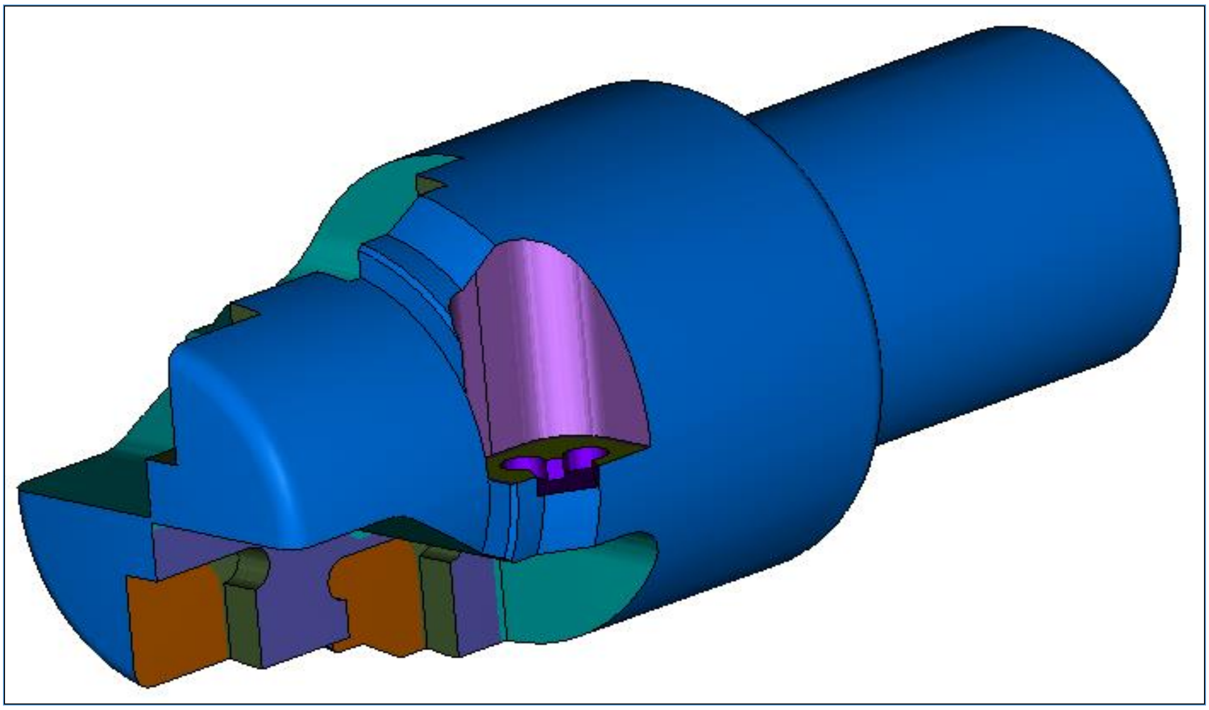


58 Check the Machining Side arrows, they must point in. Use the Icon to reverse the direction if necessary. Select **Next**.

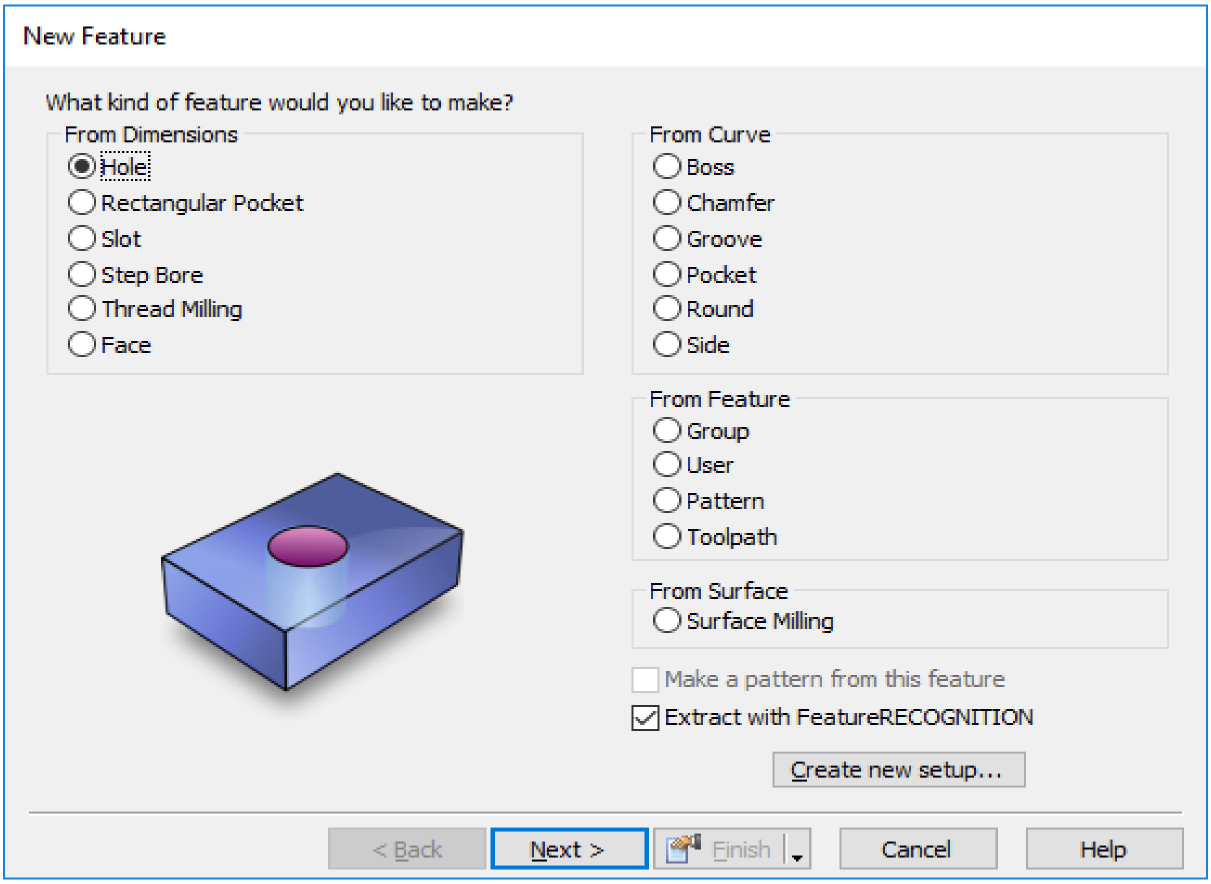
59 Confirm the Top and Bottom and change using the Top and Bottom arrows if necessary and extend the leads as before if necessary. Select **Finish.**



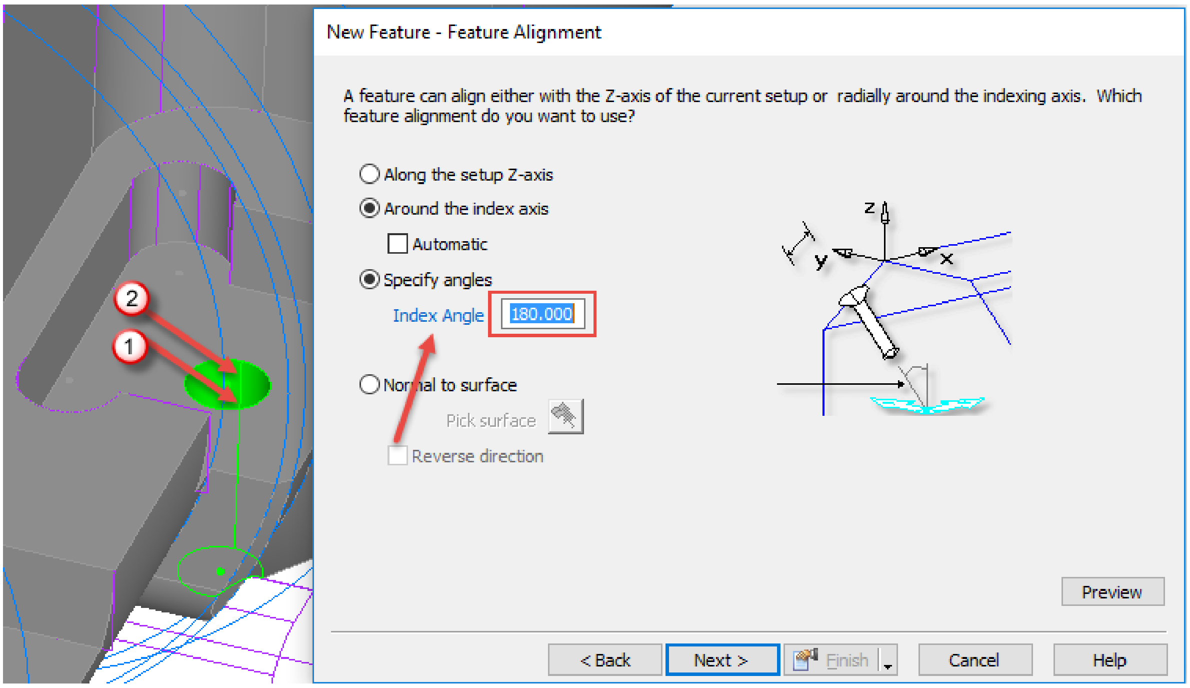
Complete all of the remaining side features.



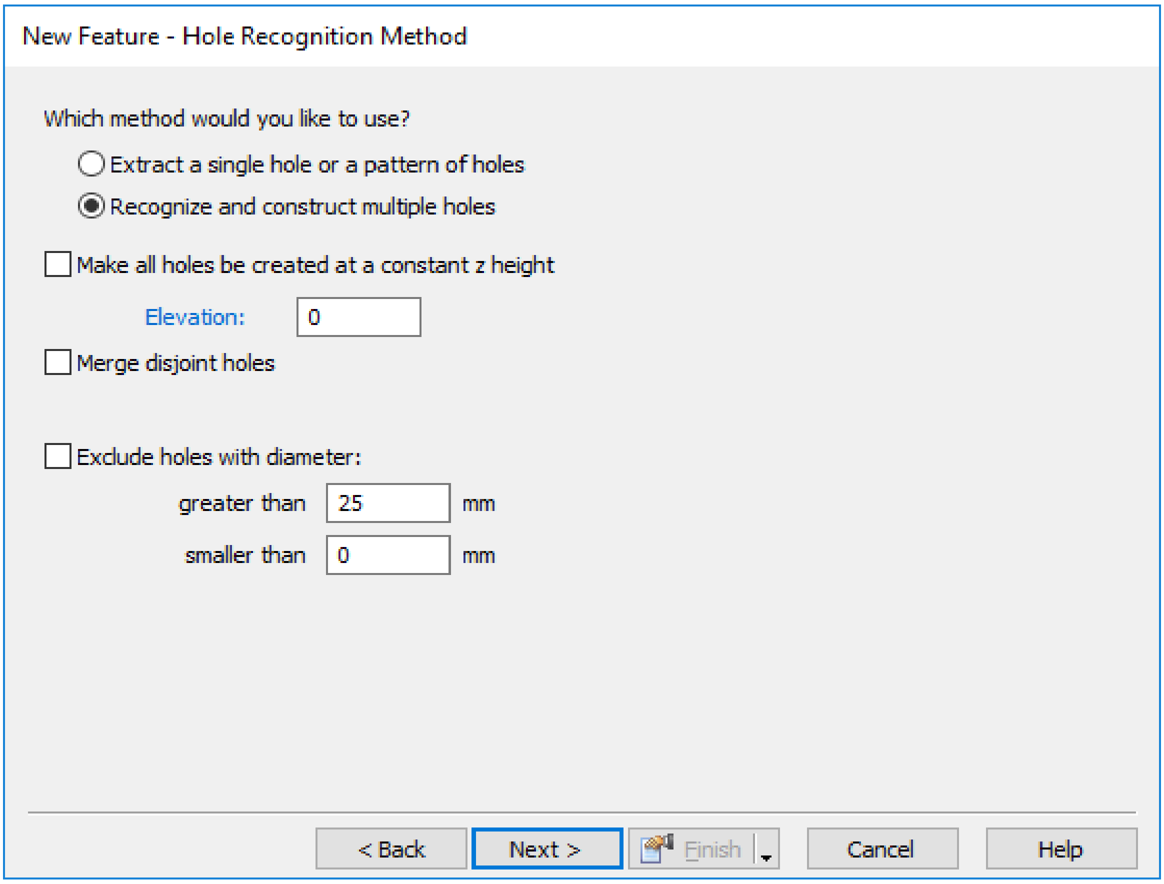
60 Open a New Feature and Select **Hole** and **Extract with Recognition**. Select **Next.**



61 Select Specify angles and click on the **"Blue Words"** Index Angle and Zoom in on the hole you choose. If you un-shade it temporarily it exposes the line very clearly for easy picking.

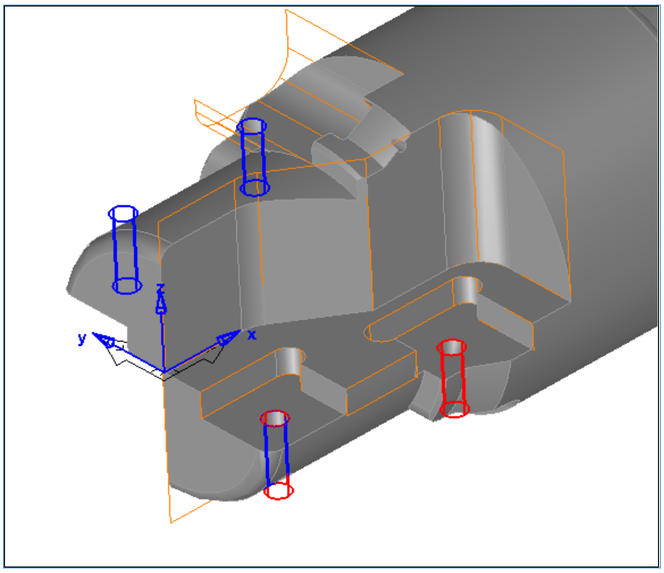


62 To achieve the correct angle remember to select on the Line from bottom to Top. This will give the correct **Index Angle**. Select **Next.**



63 Select **Next**. All holes have been selected at your chosen angle, even those on the other side that you don't want and that is because they are an open cylinder and could be drilled from this Z direction. But would drill them from the underside. Which is not desirable as it is on the diameter.

64 You simply choose only the holes you want. The selected holes will turn red for selection. The others remain blue and are not selected.



65 Create a Group of the two holes and then create a pattern for each group. Rotate the **Hole** feature 180 degrees until all of the machining is complete as shown on the next page.