Delcam

Advanced Manufacturing Solutions

FeatureCAM 2012

Training Course

FeatureCAM 2012

Training Course

FeatureMILL 2.5D

FeatureCAM

Acknowledgements

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

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

Contents

FeatureCAM overview

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

FeatureCAM includes five stand-alone modules:

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

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

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

The following add-on modules are also available:

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

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

Why creating toolpaths is so fast

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

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

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

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

Introduction to FeatureCAM

Introduction

This module will familiarize you with Delcam FeatureCAM's history, describe 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, a 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.

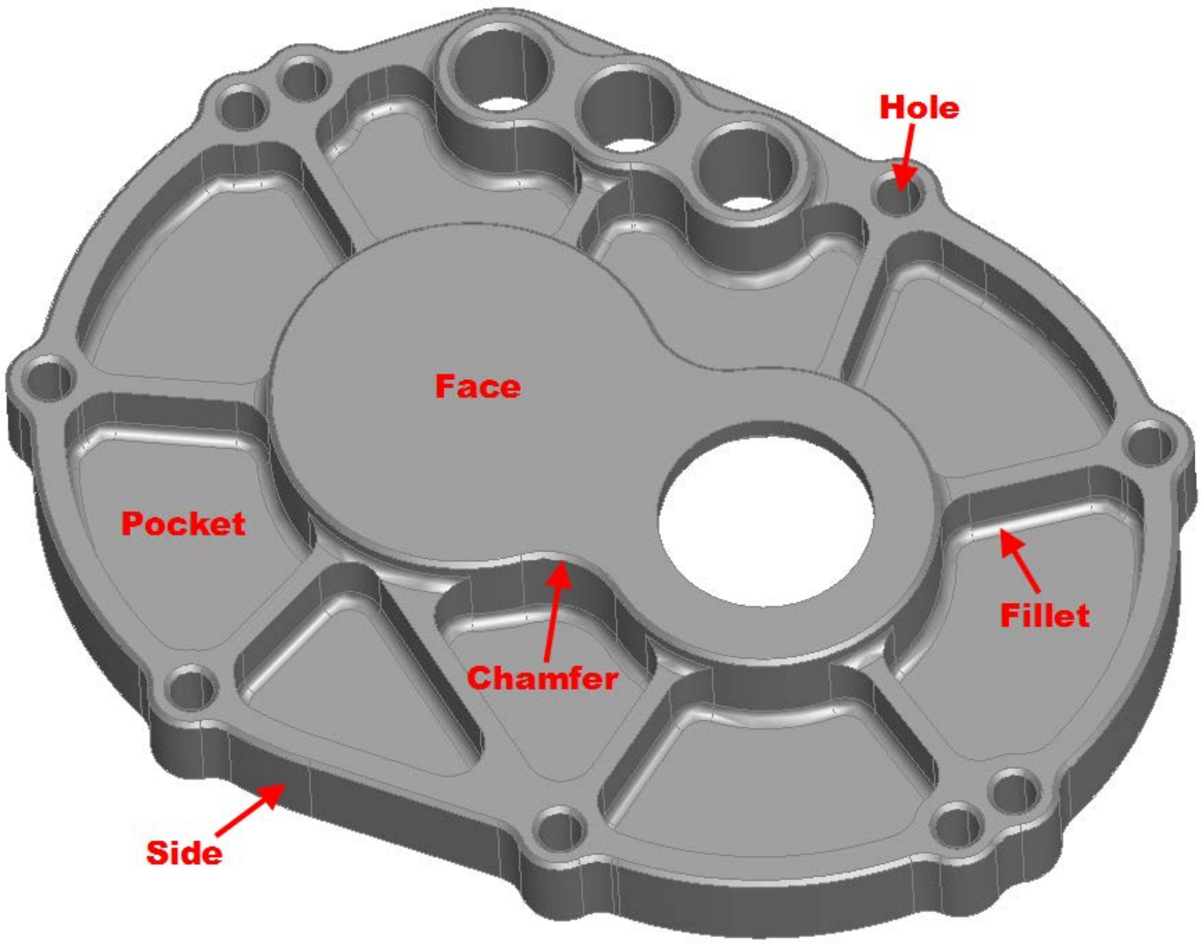
Ease of use is another guiding principal of FeatureCAM products. Our system was originally developed as a graphical Windows-based program. As a result it does not suffer from an outdated DOS look. FeatureCAM programs are full of graphical feedback, step-by-step wizards and tutorial-style animations that 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 may be quickly programmed in FeatureCAM

What is a Feature?

If we look at a part such as that shown on the previous 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 off top of the part, then machine around the two raised areas down to the next flat level. Outside area of the part would need to be roughed out and then the side of the part finished. There are then a series of pockets which would need to be rough and finish machined and finally we would drill the holes and add chamfers to the part. Each of these areas of the part, Face, Side, Pocket, Hole, Chamfer and so on is a Feature of the part.

FeatureCAM defines a part in terms of Features using common shop terms so for instance and 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.



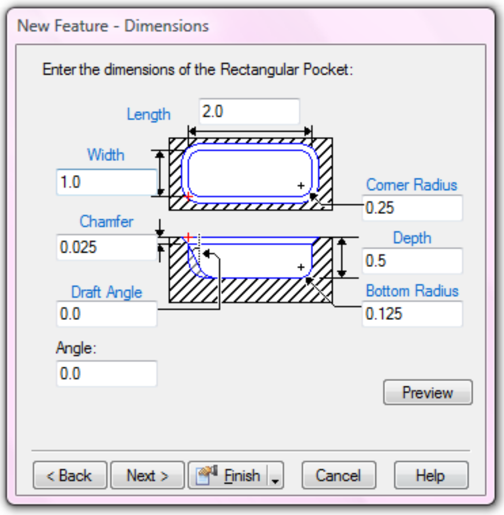
The image above shows some of the features of the part. There are many more feature types that can be created, the illustration is just to give you an idea of how the part can be broken down into its component features for manufacturing purposes.

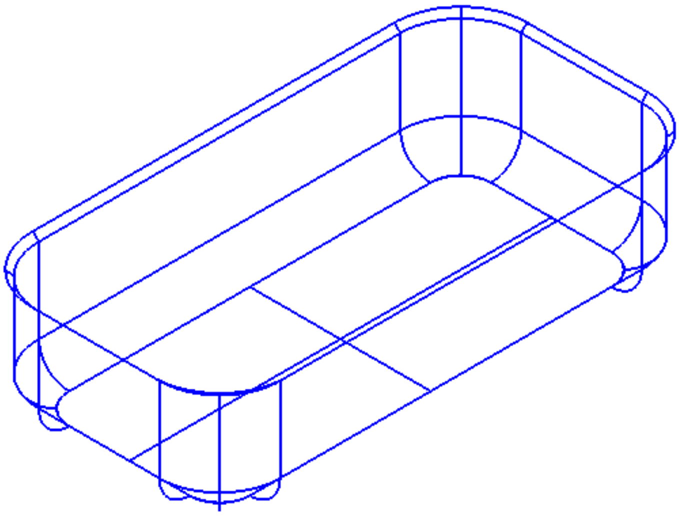
How are Features Created?

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

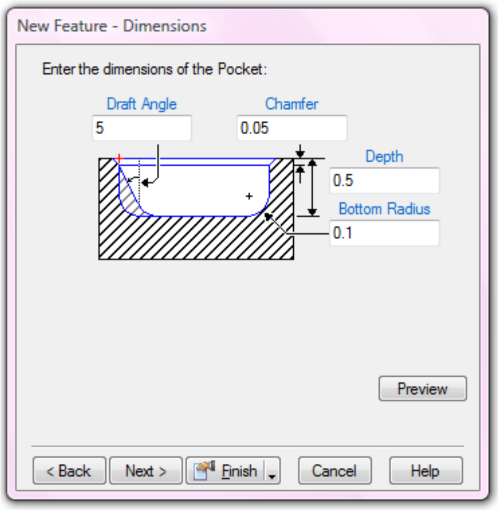
If you are working from a print then often the quickest way of creating features is to pull the dimensions from the print and enter them directly into FeatureCAM. For example you can create a rectangular pocket by entering its length, width, depth and location. Additional attributes for the feature can also be added such as a draft angle, fillet radius or chamfer.

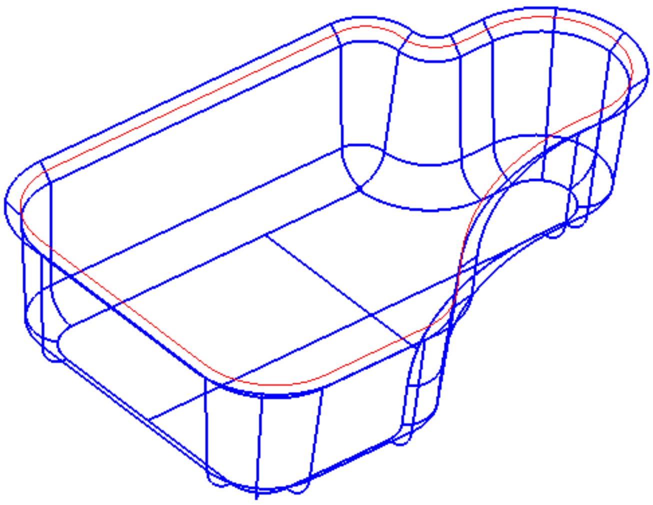




From Curves

If you have a print or a 2D drawing of a part in electronic format you can create curves for more irregularly shaped parts and then create the feature from the curve.





Automatic Feature Recognition (AFR)

If you are working from a solid model then FeatureCAM has the capability to analyze the part and identify features automatically. Basically you say, "Here is the part, go and program it". FeatureCAM then looks for Pockets, Bosses, Sides, 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.

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 for holes with a diameter of less than 0.75in". 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.

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 stepover 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 to 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, stepovers and speeds and feeds were selected because a static machining process does not automatically adapt to every part.

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 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 center 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.

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 Stepover & 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 stepover and Z steps. 8. Select a finish operation. 9. Select finish boundary. 10. Select type of toolpath. 11. Select a finishing tool. 12. Select stepover 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 analyze 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 stepover and Z increments?

Feature-based machining has many advantages such as:

- Dramatically shorter part programming time.
- 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 makes FC very easy to use.

- User Friendly Interface (Steps & Wizards)
- Automatic Tool Selection (2D & 3D)
- Automatic FeatureRECOGNITION
- Standard Databases (Tools, Materials, Feeds & Speeds)
- Customizable Post Processors
- Outstanding Technical Support

Built in intelligence

A major advantage of FeatureCAM is 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 toolcribs. 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 workpiece and the toolcrib you want to work with and FeatureCAM will then make selections of tooling for a feature automatically. If you change the toolcrib 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 toolcrib. Similarly if you wish to cut a test part in Aluminum and then cut the final part in Stainless Steel you program the part with Aluminum 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, stepovers etc. without any further intervention from the operator.

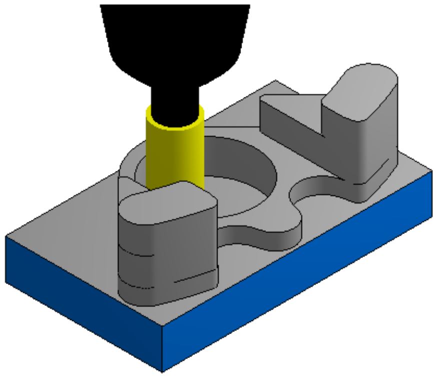
By providing a set of standard machining rules already in place, it's easier for you to customize the software to reflect the machining preferences of your shop.

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.

Beginning FeatureCAM

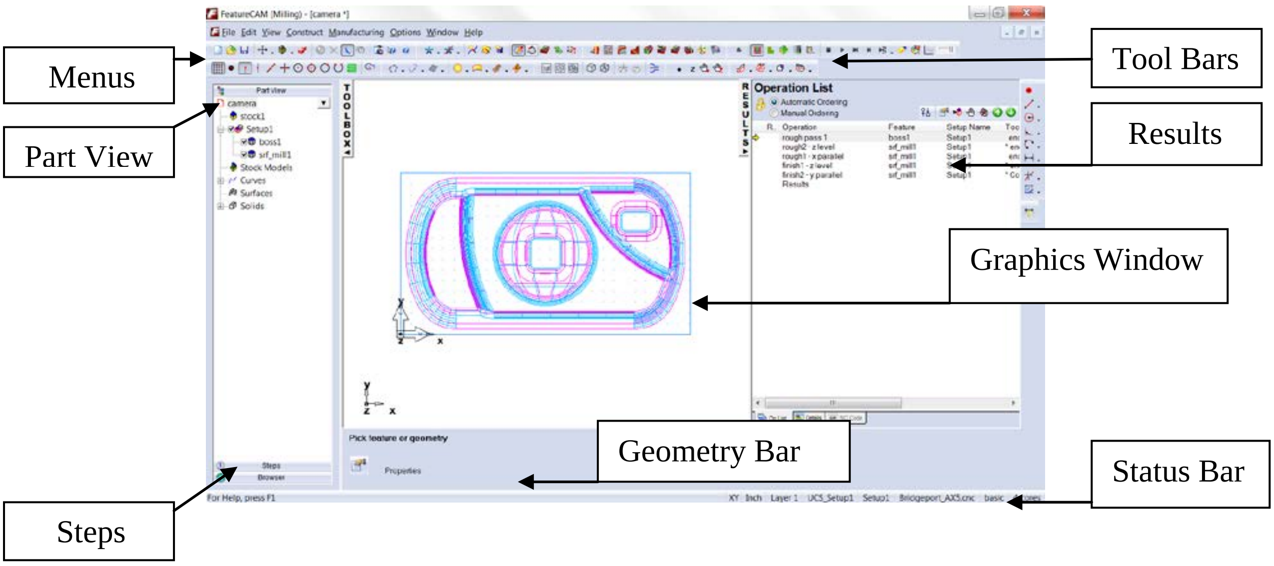
Introduction

This module will provide the user with an understanding of the basic user interface and different stock types. At the end of the module the user will be able to generate a program for a simple part.



User Interface

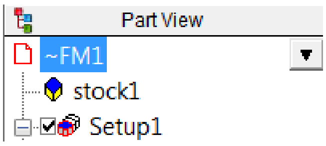
The user Interface is the entire viewing area of FeatureCAM and is designed to be intuitive, easy to use and provides the many functions to assist you in creating programs accurately and in the fastest possible time.



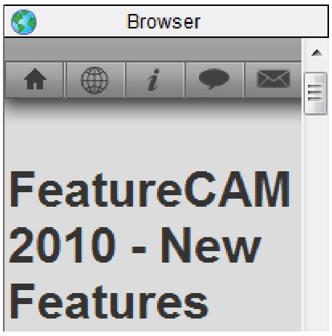
Menu Bar Located at the top. Like many other Windows based software, FeatureCAM includes a series of drop down menus. These menus are unique to FeatureCAM and provide one or more functions used to facilitate the constructing, manufacturing, viewing and editing of your part program.

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

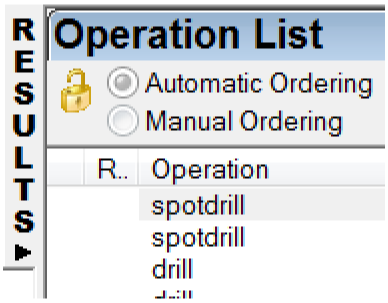
Steps Located on the left side. A systematic sequence to guide, especially the beginner, as to the order of which needs to be completed first, second, etc.



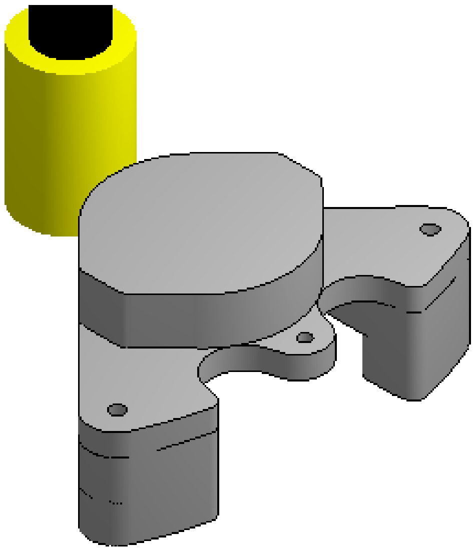
Part View Located behind the Steps. This area gives a listing and provides access to the properties of the stock, setups, features, turrets, curves, (surfaces and solids when available).



Browser Located on left side “Toolbox” below Steps. Your computer must be connected to the internet and the Browser provides a direct 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.

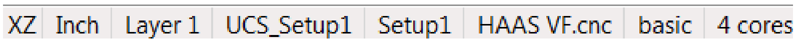


Results Located on the right side. It contains the Operation list. Whereas the Part View contains all of the features, the Operation List contains all of the hits. It also provides the details of each operation, including a tool list. The NC code is also here. All of the information residing in this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

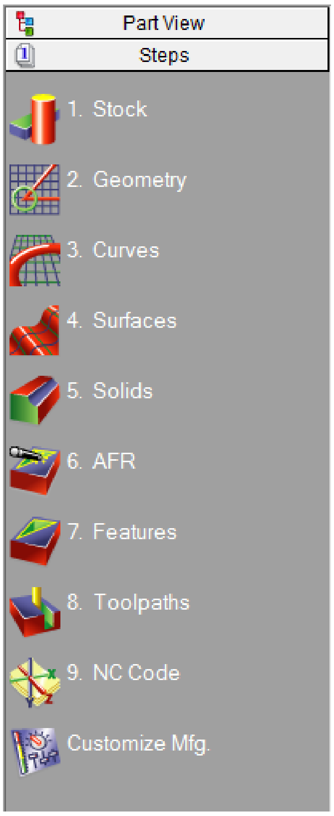


Graphics Window Located in the center. This is where most of the activity is takes place. These activities include: creating stock, importing drawings, solid and surface models, constructing geometry, chaining with curves, creating features, and simulating the tool path in 3D or center line of the tool.

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.



Status Bar Located at the very bottom of FeatureCAM. It displays the plane you are working in (XY–XZ-YZ), the unit of measure (inch-metric), the layer you are working in, the active setup, post processor, the tool crib, and how many CPU cores are being utilized.



- The basic interface has three different sections. On the left is the Steps toolbar.
- 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.
- For example suppose you have a 2D drawing on your desk and would like to create a solid model and a CAM program for it. 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 (ignored since creating a solid model)
5. Create Solid from Curves.
6. Or 7. to create Features
8. To view toolpaths and
9. To generate NC Code

- You can access a majority of FeatureCAM's functions through these buttons. A summary of each steps function follows.

Note: 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.

Step 1 - Stock. The 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.

Step 2 – Geometry. Points, arcs, lines and other shapes are used to describe the overall shape of parts. Many different geometry tools are available. Geometry can also be imported from CAD systems.

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

Step 4 – Surfaces, opens the surface wizard

Step 5 – Solids, opens the solid wizard

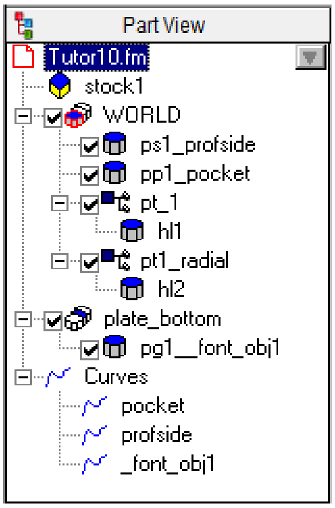
Step 6 – AFR (Automatic Feature Recognition)

Step7 – Features. Features are common shop terms like *pocket*, or *thread*. They are created from curves and dimensions. These objects describe your part in 3D and are used to generate toolpaths.

Step 8 – Toolpaths. Toolpaths are generated from collections of features. You can simulate them in FeatureCAM using toolpath center lines, 2D shaded or 3D solid shaded simulations.

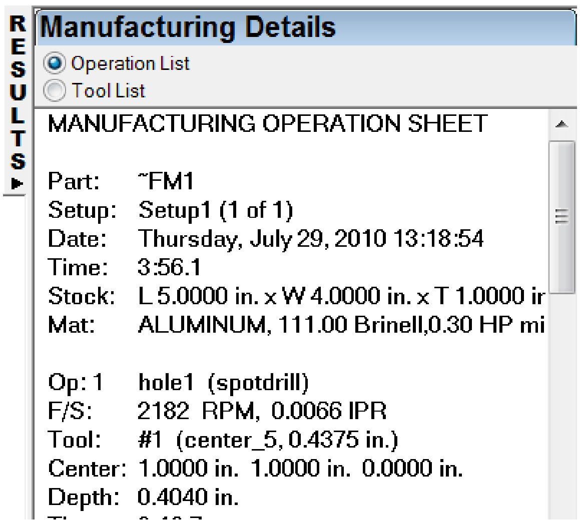
Step 9 – NC Code. Machine-specific G-codes are generated from the toolpaths. Translators are provided for many different NC controls and include a program for creating new translators.

Customize Manufacturing. FeatureCAM automates the entire part programming process; you can customize all of the system settings including feed/speed tables (Stock Properties), tooling databases (Tool Manager) or feature settings (Machining Configurations in another Module)

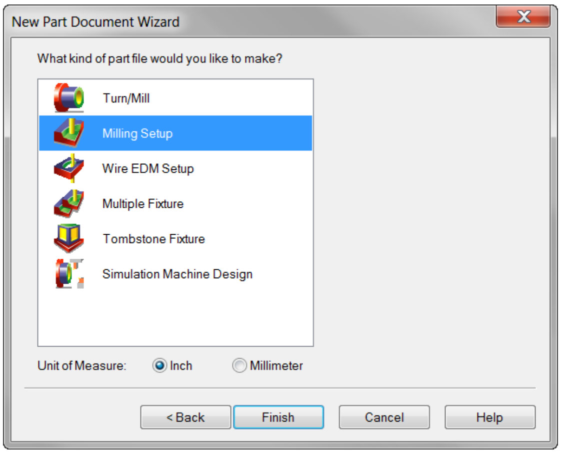


Note: In Steps, some steps may not be displayed depending on which options you have purchased or have turned on.

- Above the Steps Toolbar is 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, solids and stl.
- Clicking on the objects in the part view enable different functions. Left-click selects the object in the graphics window. Right-click reveals menu of options available for the object. Clicking on a "+" to expand the view will reveal objects that are subordinate.



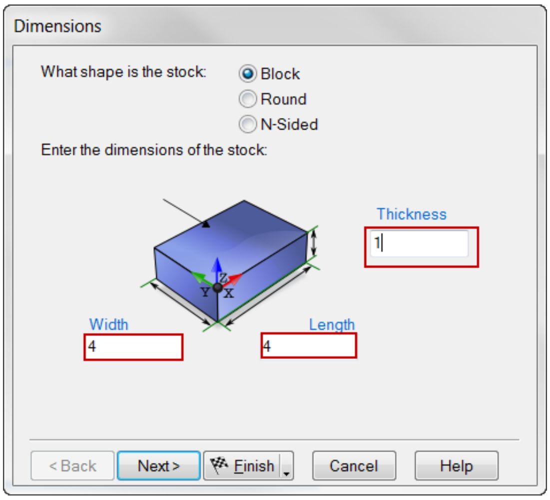
On the right is the Manufacturing Results Window. The automatically generated documentation and part programs are shown in this window.



Your First part exercise

- From the New Part Document Wizard select **Milling**, then select **Inches** as unit of measure then press the **Finish** button.

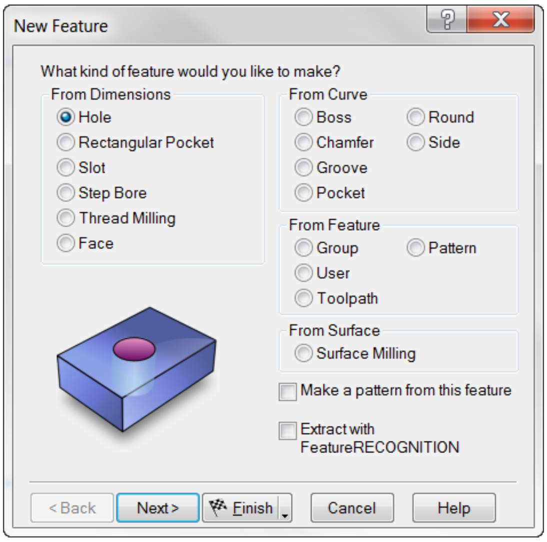
Notice the different types of documents available. The different types of documents available; depends on the modules that you own and are licensed for.

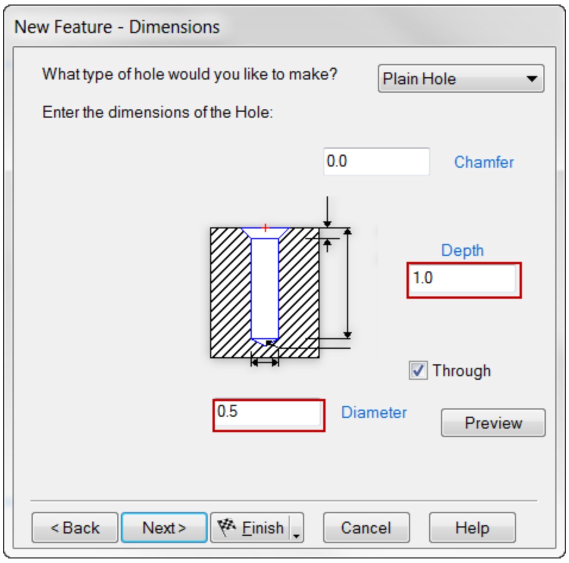


- In the stock dimensions dialogue **Enter** 4” for the Width, 4” for the Length and 1” for the Thickness then **press** the **Finish** then **OK**.

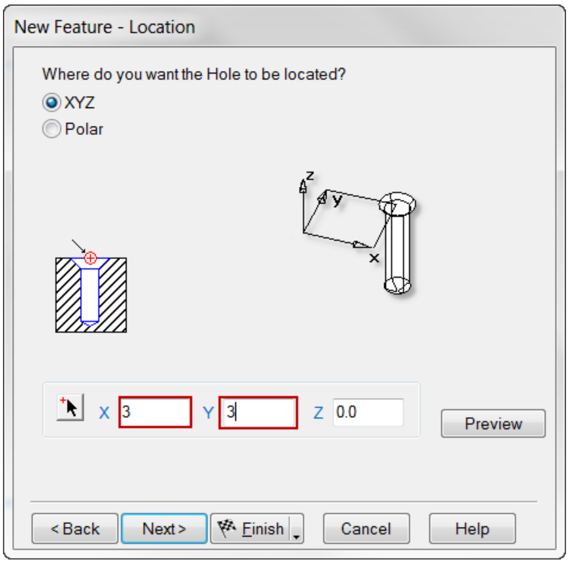
.

- From the steps view on the left hand side of the screen click the **Features** button.
- Select Hole and press **Next.**

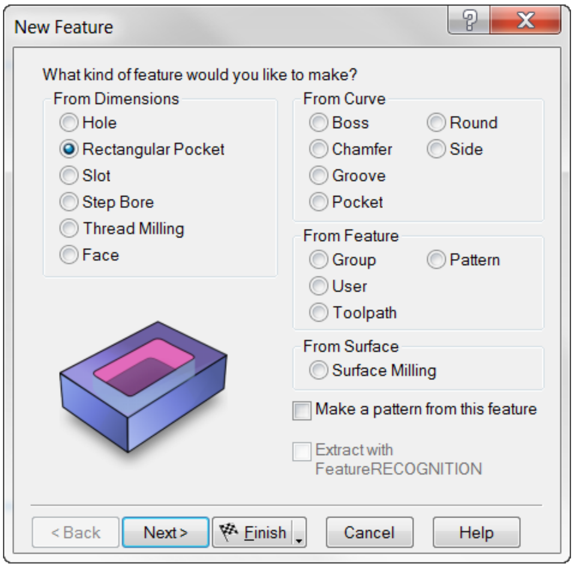




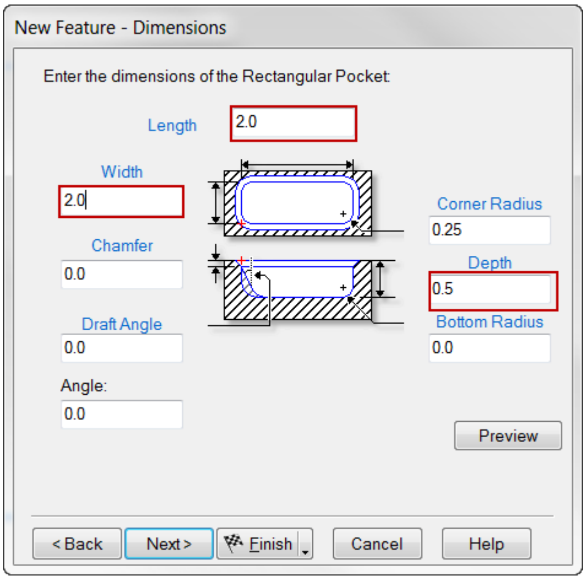
- In the New Feature Dimensions window **Enter** the hole dimension as 0.5" in Diameter, 1.0" in Depth, then check the Through checkbox and press **Next**.



- For the X location **Enter** 3", the Y location enter 3" and the Z location enter 0. Press **Finish** then **OK**.



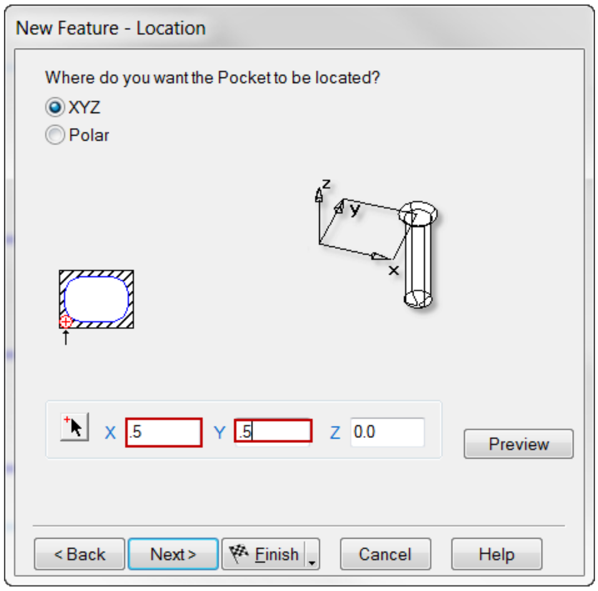
- From the steps view on the left hand side of the screen **Click** the **Features** button.
- **Select** Rectangular Pocket. **Next**.



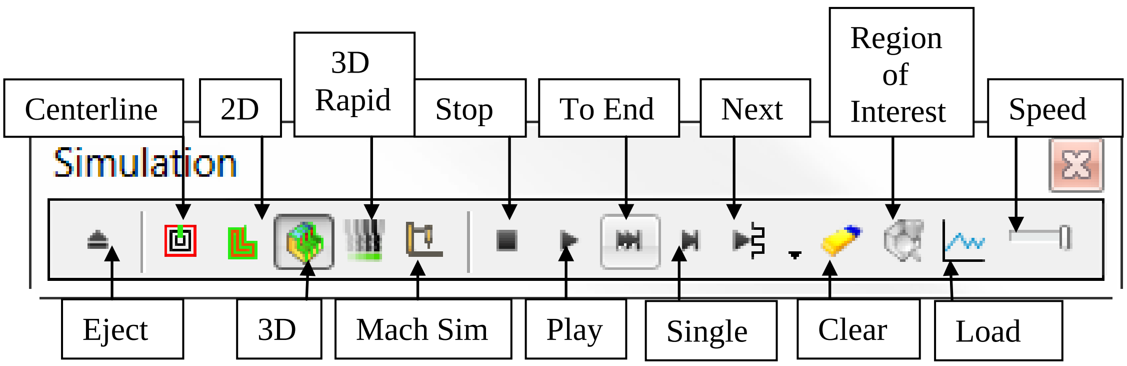
- On the Dimensions tab **Enter 2.0”** for the **Width**, **2.0”** for the **Height** and **0.5”** for the **Depth** then press **Next**

.

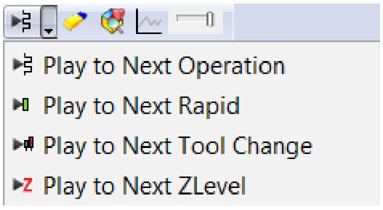
- On the Location Tab **Enter 0.5”** for the **X location**, **0.5”** for the **Y location** then press **Finish** and **OK.**



- **Right-click** in the Graphics window then press **isometric**.
- From the steps menu **Click** on the **Toolpaths**. Simulation bar flies out, unless already out.
- Using the Simulation Bar; **Read** the purpose of and **Execute** each button.



- **Show centerline**: A line drawn of the center of the tip of the tool is displayed.
- **2D simulation**: A two-dimensional simulation showing the regions cut by each operation is displayed. The view is changed to the top view automatically.
- **3D solid simulation**: A 3D solid simulation is displayed where the tool is animated through all of its moves.
- **Rapid Cut simulation**: 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. Note this type of simulation is only available in FeatureMILL3D.
- **Machine Simulation:** A 3D solid simulation is displayed where the tool is animated through all of its moves along with the machine tool
- **Eject** removes the Simulation toolbar from the screen and erases the simulation from the graphics window.
- **Stop** cancels a simulation.
- **Play** starts the selected simulation (Centerline, 2D, 3D or Rapid Cut), or restarts a paused simulation.
- **Pause** pauses the simulation. The Play button transforms into the play button after it is pressed. If the Pause button is pressed during a simulation, it pauses the graphics.
- **Single** step moves the simulation ahead one tool move. The keyboard accelerator for this button is *ALT+F3*.
- **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

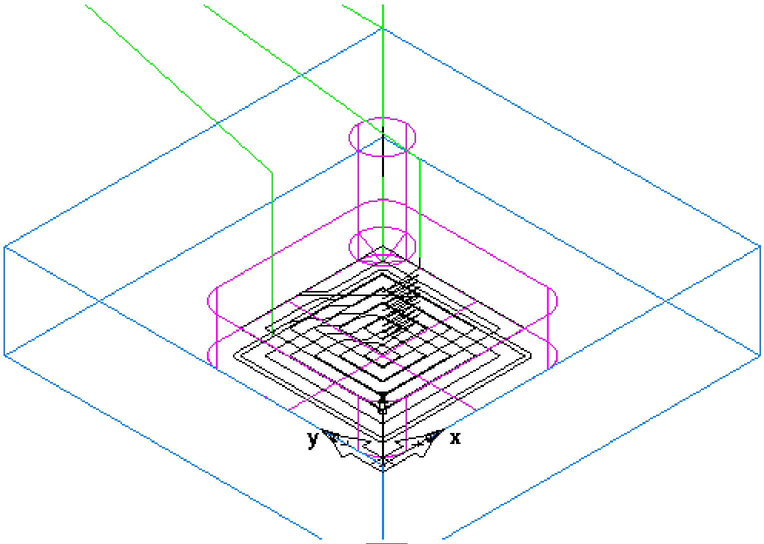


:

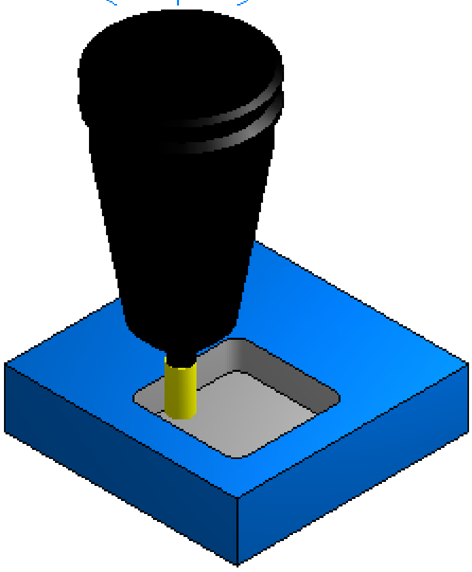
- **Next rapid** simulates until the next rapid tool move.
- **Next tool** change simulates until the next tool change.
- **Next Z level** will simulate the next Z of a Z level toolpath. For other toolpaths it will play the entire next operation
- **Clear** erases any centerline toolpaths on the screen.
- **Region of interest** limits the portion of the part that is simulated.
- **Show tool load** indicates whether or not to display a graph of the tool load when the next 3D Simulation is performed.
- To adjust the speed of a centerline, 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 the simulation down. The slider of 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.

- Click the **show centerline** from the simulation toolbar.
- Adjust the simulation slider button to the minimal setting by **Left-clicking** and **Dragging** then click the **Play**.

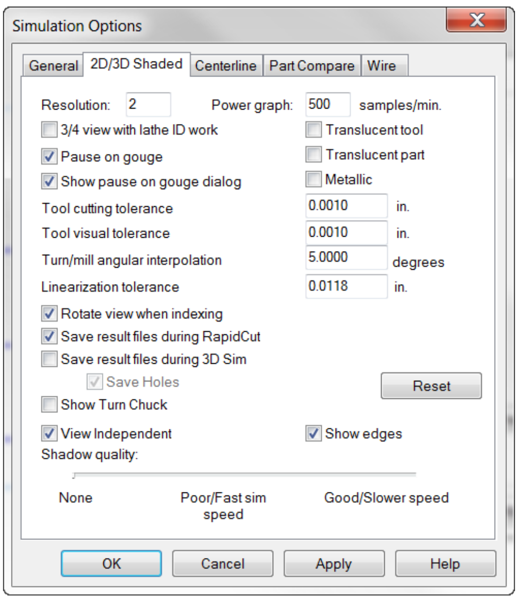
A centerline display of the toolpath is shown where the line displayed represents the center of the bottom of the tools movement. Green lines represent rapid movement where black lines show feed moves. **Press** the **Eject**.



- **Click** 3D Simulation and Play.

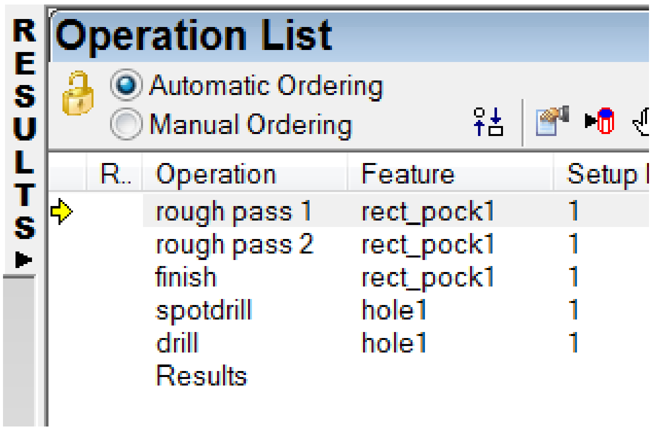


- In conjunction with 3D, Click on **Options** and **Simulation** from the menu bar. Check show holder displayed on the General Tab and on the 2D/3D Shaded Tab Check the Pause on gouge and Show pause on gouge dialog. Click **OK.**



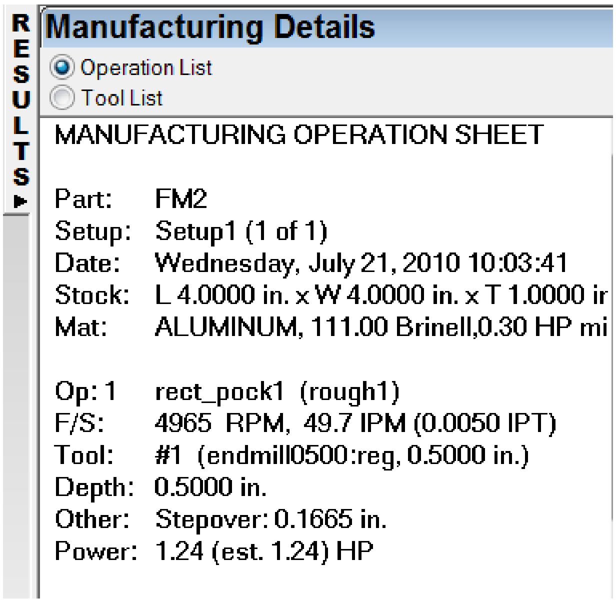
This will not only show the holder for the milling tool, but reveal any tool striking the stock in rapid traverse, or if the holder touches any part of the stock. These gouges will show as pink in color, the simulation will stop, and a pop up will appear telling of a possible gouge.

- In the Manufacturing results window on the right hand side of the screen **Click** on the Op. List tab at the bottom of Result Window.

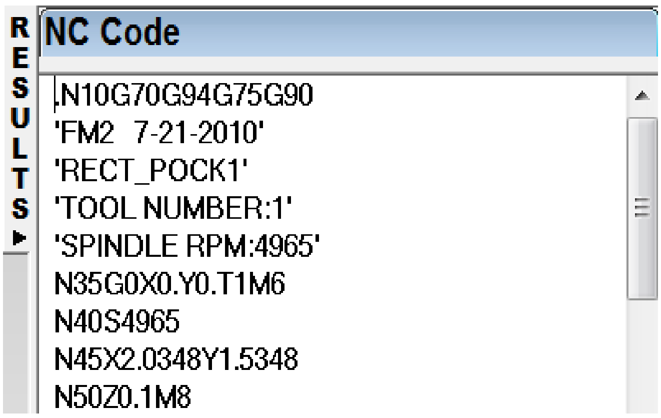


Op List shows each operation listed in the order they are to be machined, feature name, tool used, Feed and Speed data. Properties may be accessed here also (double click on each operation).

- **Click** on the Details tab.



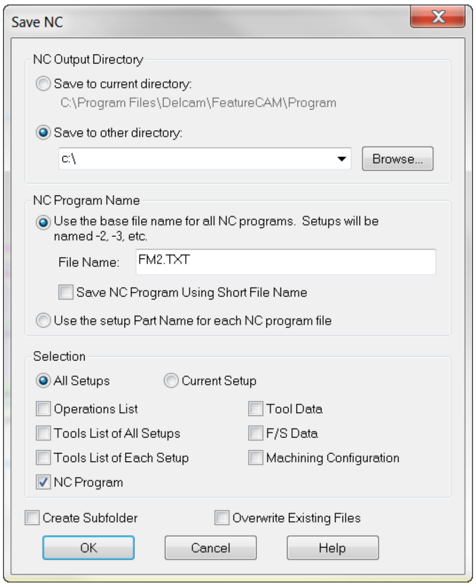
You will see an operation summary showing the Part name, Setup name, Date, Stock Dimensions and Stock Material followed by the summary of each operation. This shows the feature name, operation name, F/S, Tool used, power, stepover and depth of cut. The Tool List button at top provides a list, in detail, all tools being used and where they are placed on the tool changer.



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

NC code is generated during simulation and the output is in accordance with the post processor for the machine that is going to be used to run your part. The Part Program number on the O line and the G54, line N50 are automatically entered in the code.

- Click on **File** then **Save NC**. **Click** the radio button **save to other directory**, then **Type** in **“C:\”**. This will save the NC code to the C drive. In the selection window **Uncheck** all check boxes and only check NC Program. Then **Press OK**. The check boxes will create a text document that can be printed.

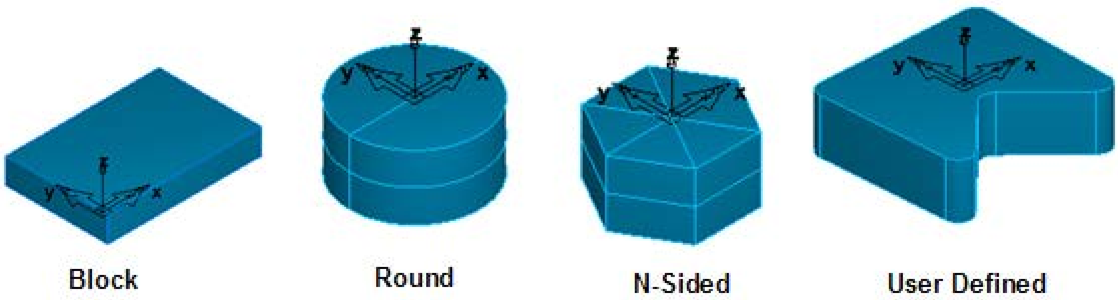


Now that the NC code is saved it can be downloaded to the machine and the part with the simple pocket and hole can be machined.

- **Close** this document and **Save** it if you want to keep it.

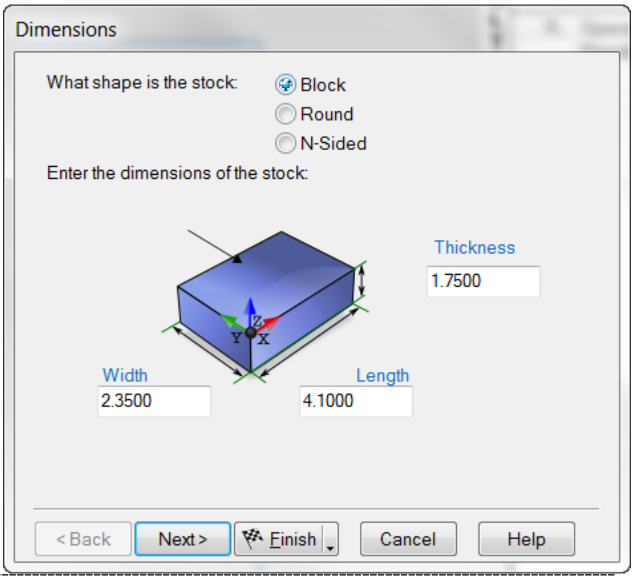
Stock Types

Inside of FeatureCAM milling document there are several different stock types. The stock can be defined as a block, round, stock curve or user defined.

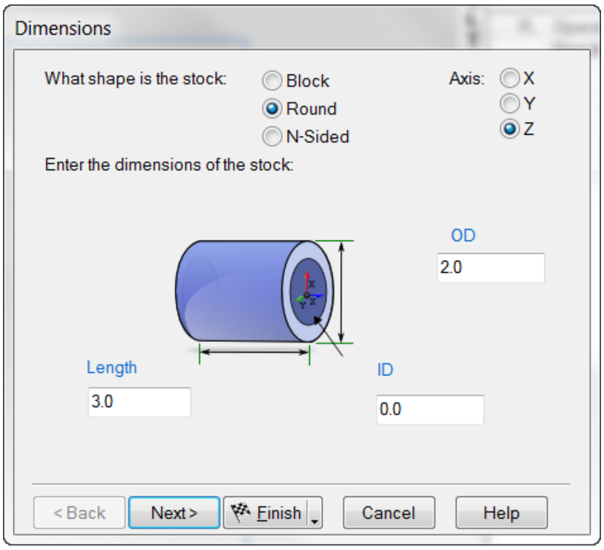


Stock Exercise

- **Open** a New Milling Document in inch. **Finish.**

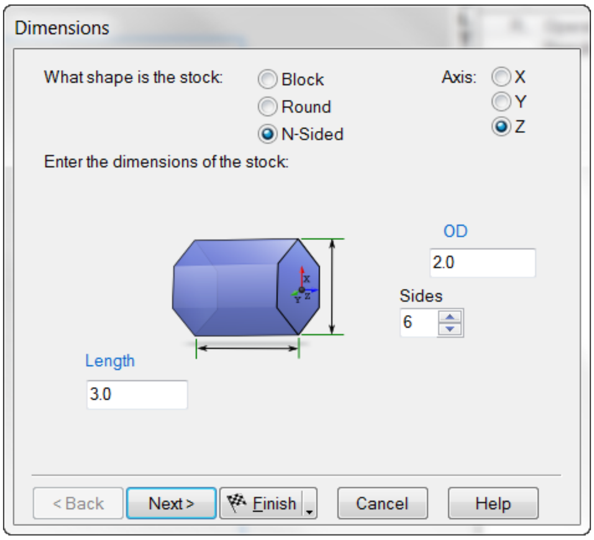


The stock wizard dialogue will appear. By default block will be selected. You can enter the rectangular dimensions of the stock here.



- Click on the **Round** radio button.

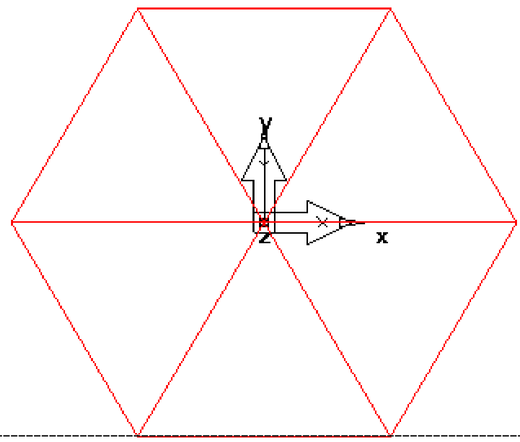
The ID, OD and length of the round stock may defined. Additionally the axis of the stock may be defined as needed for 4 axis..



- **Click** on 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 such as hex.

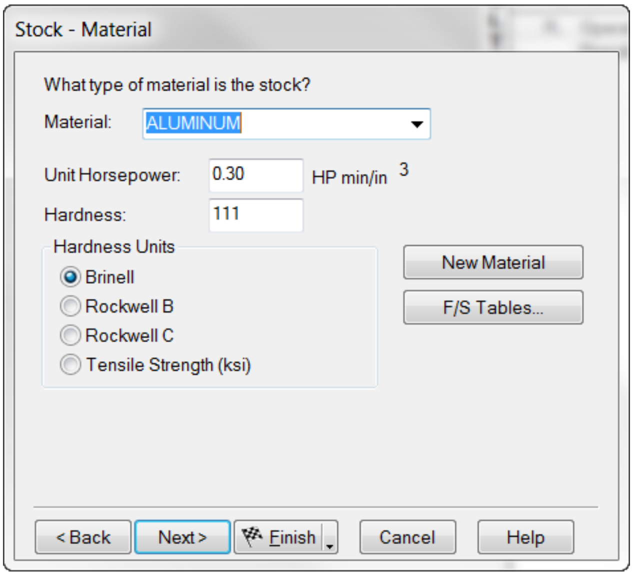
- Enter the stock dimensions as shown to the right and press **Next**.

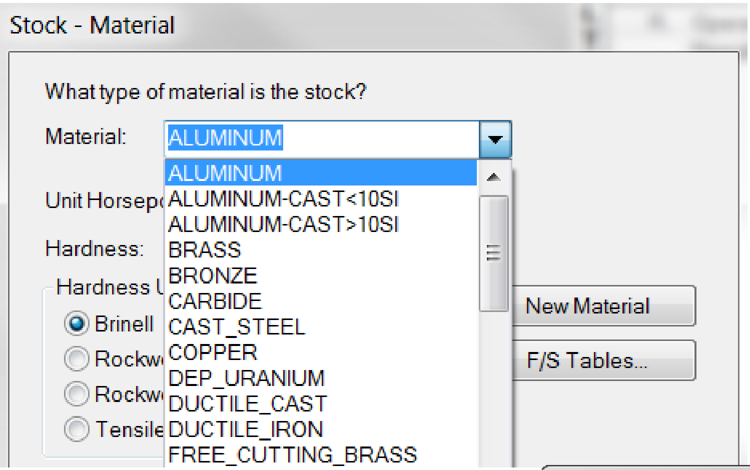


A Hexagonal outline of the stock will appear in the display window. The Axis may also be selected like the round stock.

- **Press Next.**

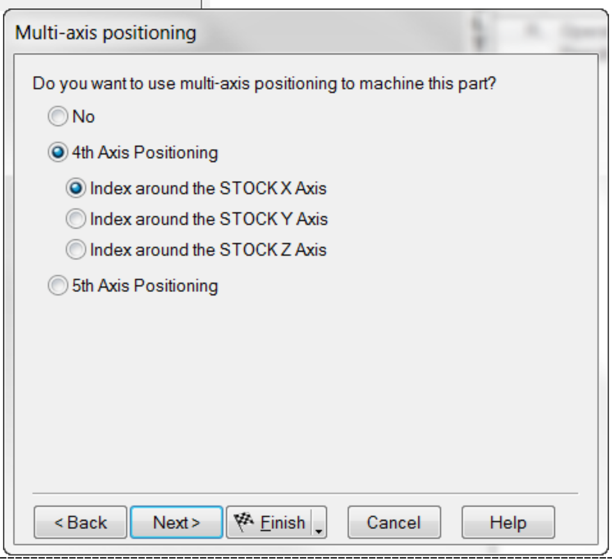
On this page of the stock wizard you can select the Material, Unit Horsepower, Hardness and Hardness Units. You can also access the existing Feed and Speed tables for the material or create a new Material

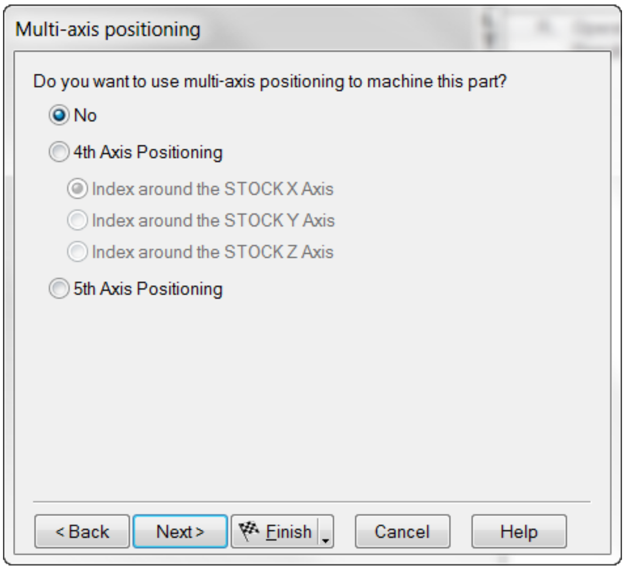




- From the pull-down menu select Aluminum then press **Next**.

The multi-axis positioning options appear.



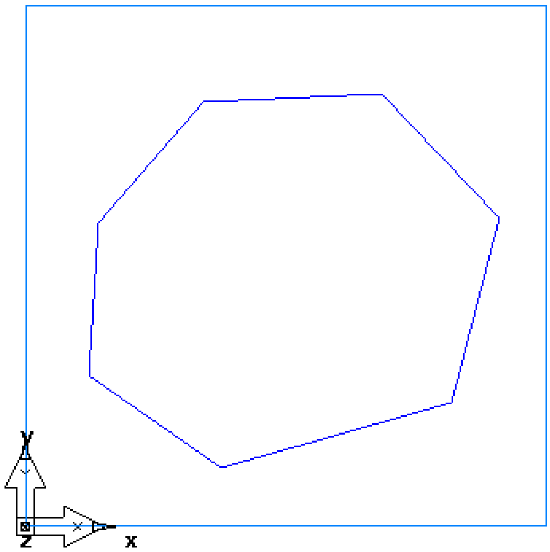


- Press **No** to Multi-axis positioning and press **Finish**. Do not press **OK** at this time.

- **Right Click** in the Graphics window, off the part, and **Select** an isometric view. The stock is hex shape and FeatureCAM simulates that shape in 3D simulation and also uses it as the stock boundary when determining where cuts start. **Right Click** in the Graphics window and select a Top view. **Click OK**.

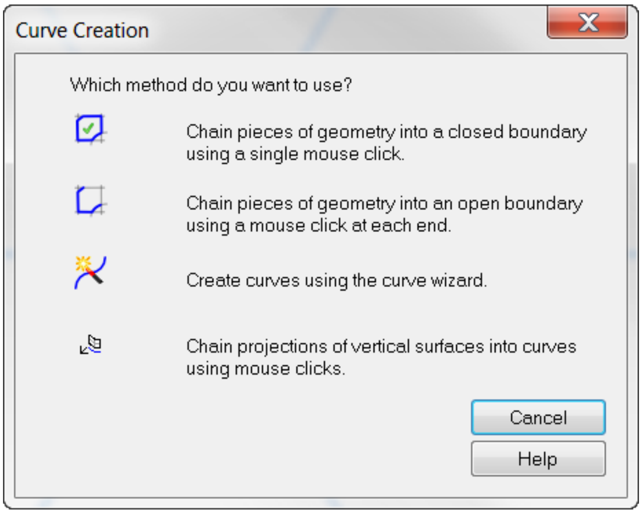
The Fourth way to create a Stock shape is by drawing the shape using geometry and then chaining a curve, it is called a Stock Curve. **(Creating shapes using geometry and curves will be covered in a later module and you may wait until you finish with them and then return to this step and create the Stock Curve although the following steps are easy).**

Note: The Stock Curve must lie in the XY Plane. In the Top View, this is the XY Plane when the UCS (Part Program 0) is in this position (Lower Left)



- **Click** on **Stock** in the Steps on the left side of FeatureCAM. **Select** Block and enter in the Width as 6.0, Length 6.0, and Thickness 6.0. Click **Finish** and **OK.** Center the stock by holding **CTRL** and **L** keys simultaneously.

- **Draw** by **Clicking** on **Geometry** from the Steps and from the Geometry Constructors use Line/Connected Lines and **Draw** freehand a shape similar to the one above. Be sure the geometry is connected.

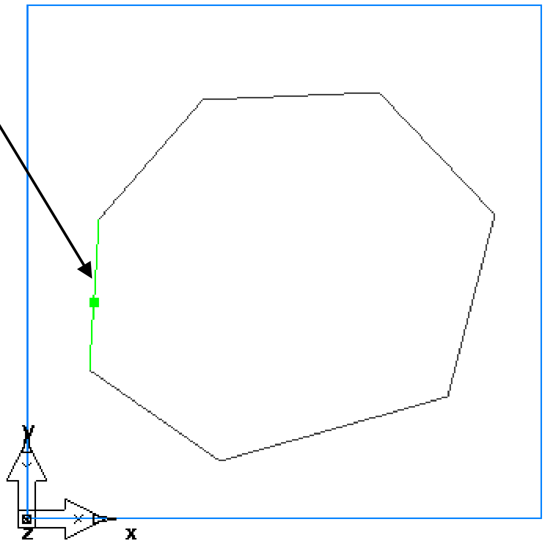


- **Click** on Curves from Steps and **Click** on Closed Curve (red check).

Chain a closed curve on the Geometry by **Clicking** once on any line then **Click** on the **Create** button at the bottom of the screen.
The curve (blue) will close all the way around the closed Geometry. The Curve must be closed and in the XY Plane.

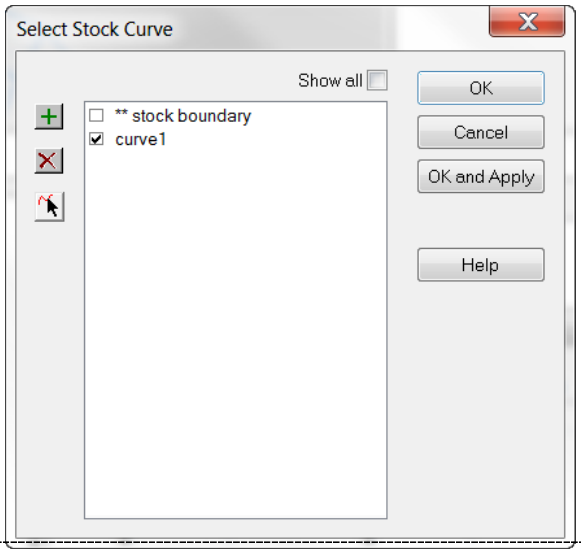
- XY Plane is Top View.
- XZ Plane is Front View.
- YZ Plane is Left View.

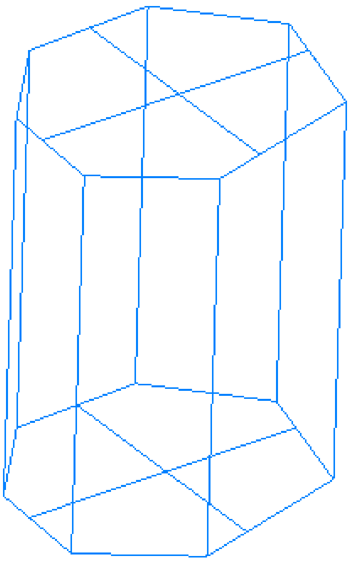
This Curve now becomes the NEW shape of the Stock and is used for Stock Boundary and 3D simulation stock solid.



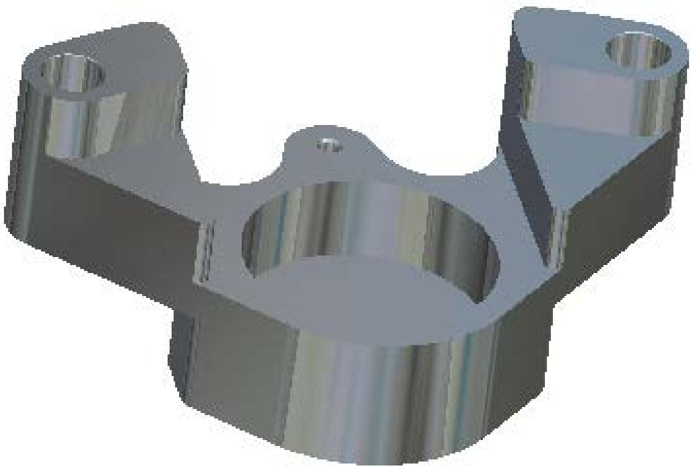
Click on Stock in the Steps and then **Finish**.

- On the Dimension Tab, of the Stock Properties dialog, **Click** on the Stock Curve button and from this list **Check** the box in front of the curve just created. **OK and Apply**, then **OK.** The Stock now changes its shape.



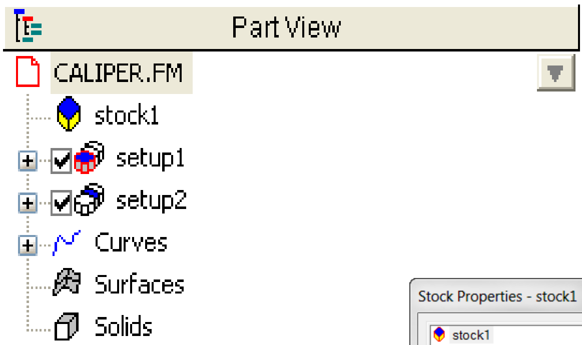


User interface exercise



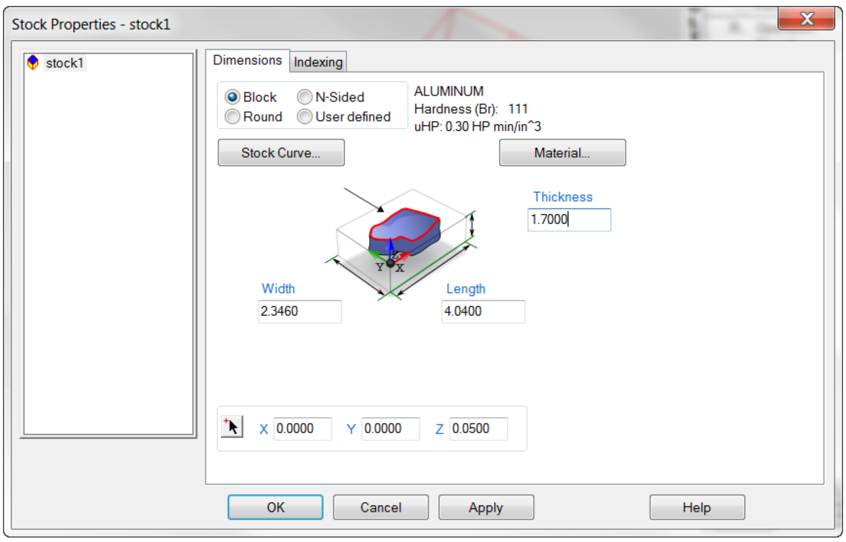
In this exercise you will become more familiar with the FeatureCAM user interface.

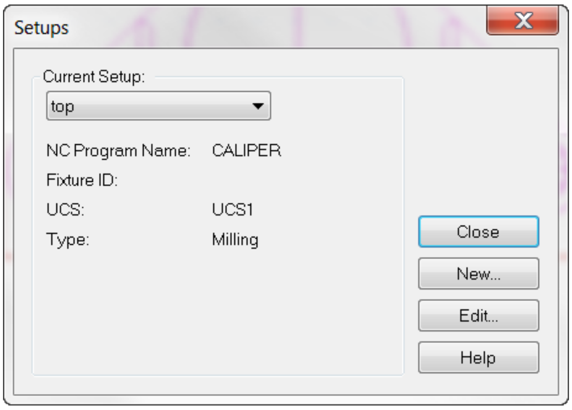
- From the Beginning FeatureCAM Data folder, or the FeatureCAM 2.5D Data folder. **Double Click** on the file **Caliper.fm.**
- **Click** on the Part View on the left hand side of the screen.



Observe all the different objects the part contains under the part view. At the top level of the part view is the part name. In the level directly below the part name you have stock, setup1, setup2, curves, surfaces, and solids.

- **Double-click** on stock in the part view for the Stock Properties showing Length, Width Thickness and an offset in Z for facing stock. **OK.**



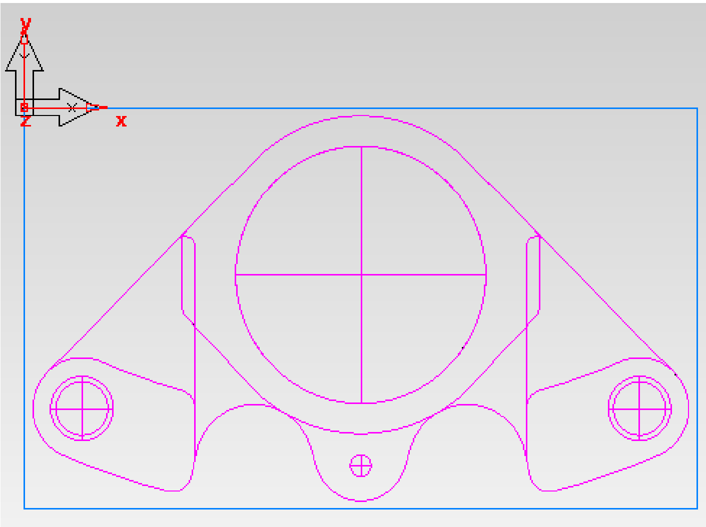


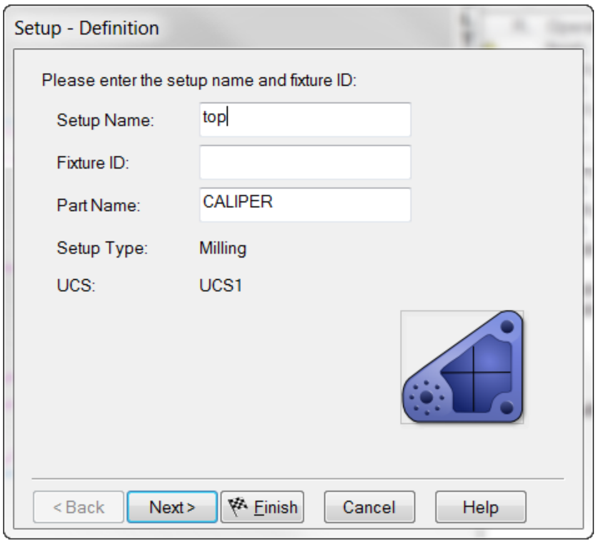
- From the Part View menu **Double-click** on top.

The setups dialogue is displayed. From here an existing setup may be modified or a new setup may be created.

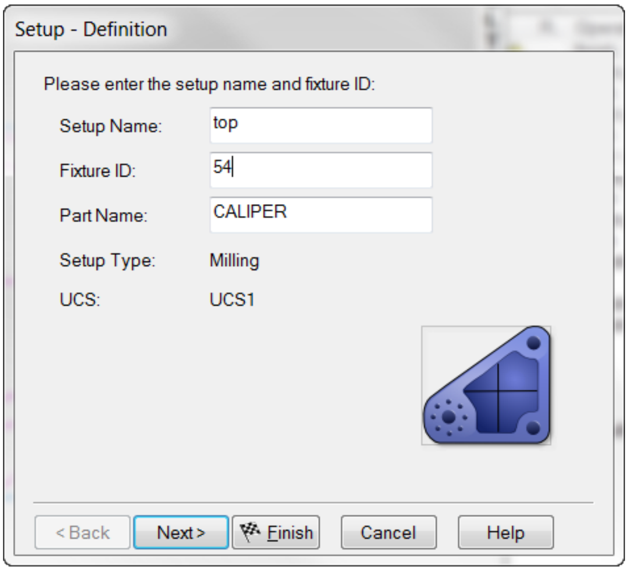
- Click on **Edit**

Observe that the current setup location is highlighted in Red in the graphics window. The setup location is on the upper left hand corner of the stock. The setup, fixture name and part name can be edited from this window.



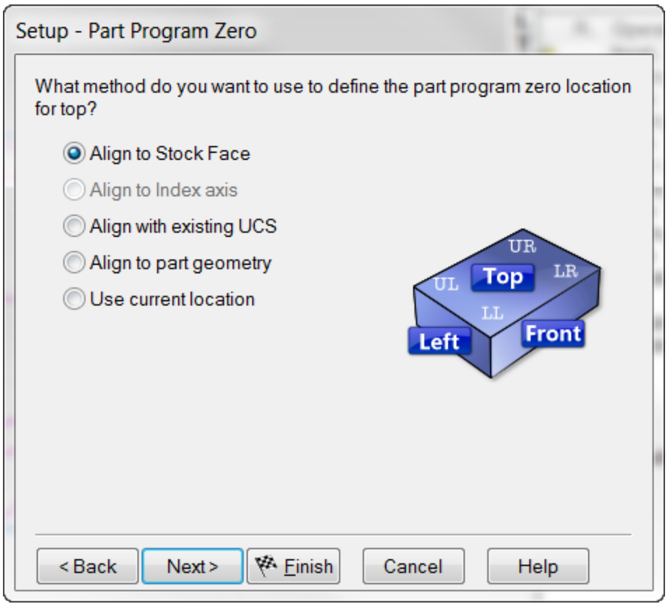


- Enter **Fixture ID** of **54** then press **Next**.

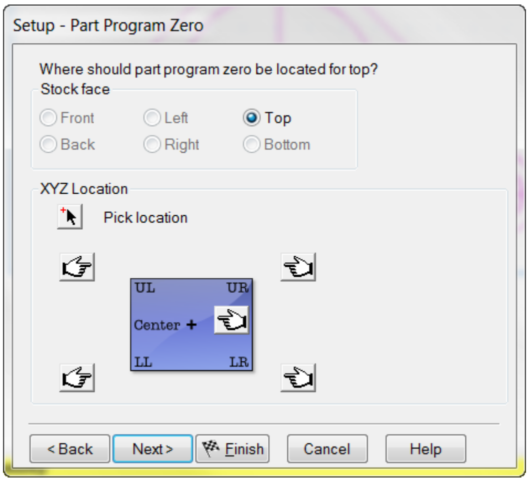


The part program zero window is displayed. In this window we have the ability to change the setup location or the part program zero. This is where fixture ID 54 is and where the machine tool fixture offset 54 zero is.

- **Select** the Align to Stock Face and press **Next**.

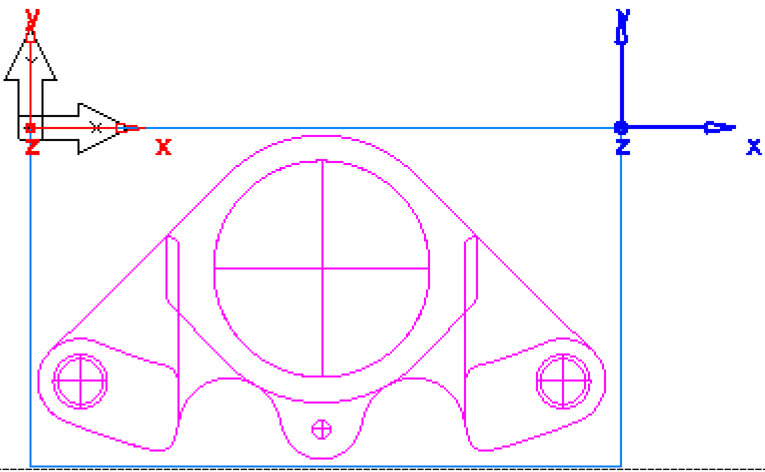


A page appears that will enable you to change the setup location. Currently the setup is located in the upper left hand corner.

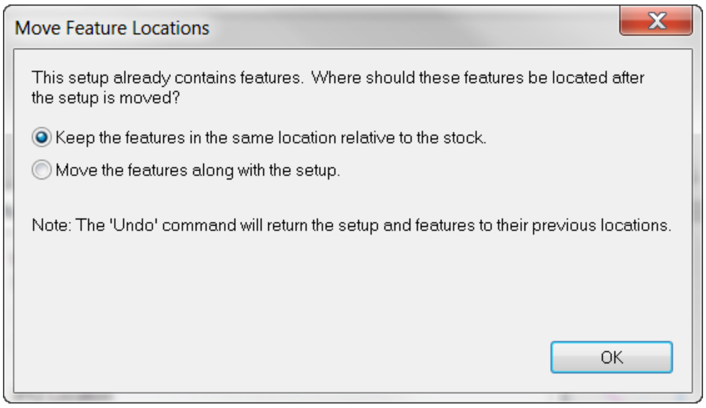


- Change the setup to the upper right; **Press** the **Hand** next to **UR.**

You should see in red in the upper left hand corner the previous location of the setup, and highlighted in blue is the new location of the setup.



- **Click Finish.**

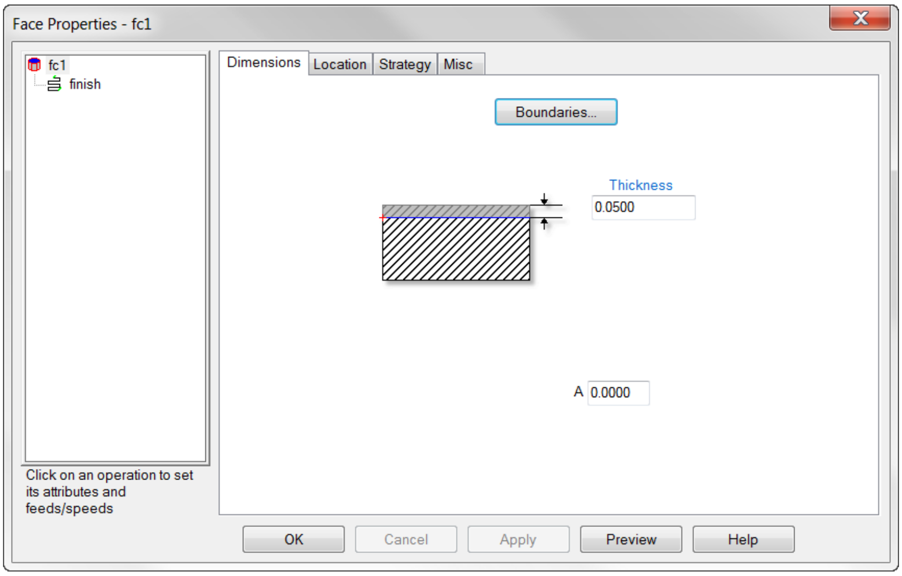


- Select to Keep Features in the same location. **OK** and **Close**.

- In Part View **Click** the **+** Plus Sign next to top to expand and reveal all of the Features for Setup1.

Notice how the features in setup1 have been displayed, in the order they were created.

- **Double-click** on fc1 from the part view or Right Click on feature for Properties.



The feature properties from setup1 are displayed. From here the face properties may be changed. All the feature properties may be accessed from the part view by simply double clicking on them.

- Click **OK**.

Cut, Copy and Paste are Windows functions that are available in FeatureCAM. There are two Setups on this Part and Setup2 needs to be faced off. You already have a face feature in Setup1 so you will copy it from Setup1 and paste it in Setup2.

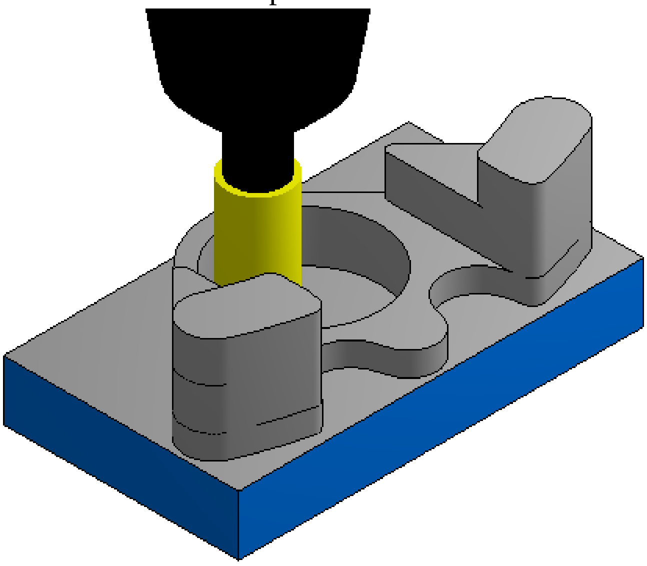
- In the Part View, **Right Click** on the Face feature in Setup1 and **Click** Copy.

- **Click** on Setup2 to select which Setup you wish to place it in. **Right Click** on the same Face feature you copied from and **Click** Paste. This adds the Face feature to Setup2.

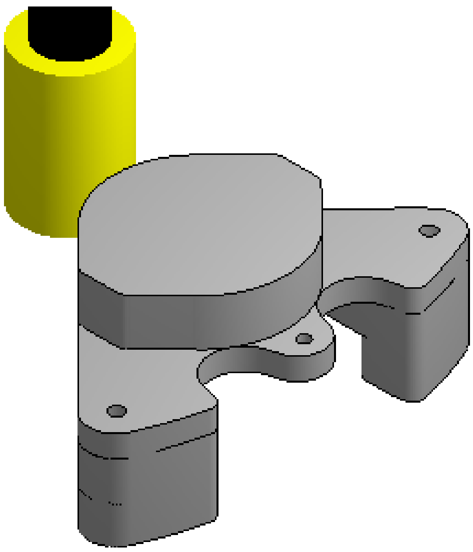
Now both sides have a Face feature by simply copying and pasting. You can even copy from one Document and Paste into another Document.

- In Part View **Click** Setup1 then **Right Click** in Graphics Window and **Click** Isometric View.
- In Steps **Select** Toolpaths and the Simulation toolbar flies out. (unless it is already visible) It may be dragged to several docking areas of your choosing.
- On the Simulation Bar, **Click** 3D simulation and **Click** Play. (Setup1)
- Do **NOT** cancel 3D Simulation. Now in Part View **Click** on Setup2 then **Right Click** in Graphics Window and **Click** Isometric View and Run the 3D Simulation again. (Setup2)

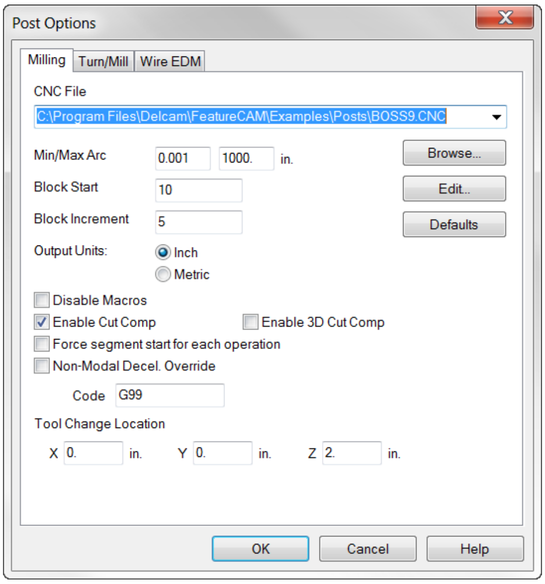
Setup1



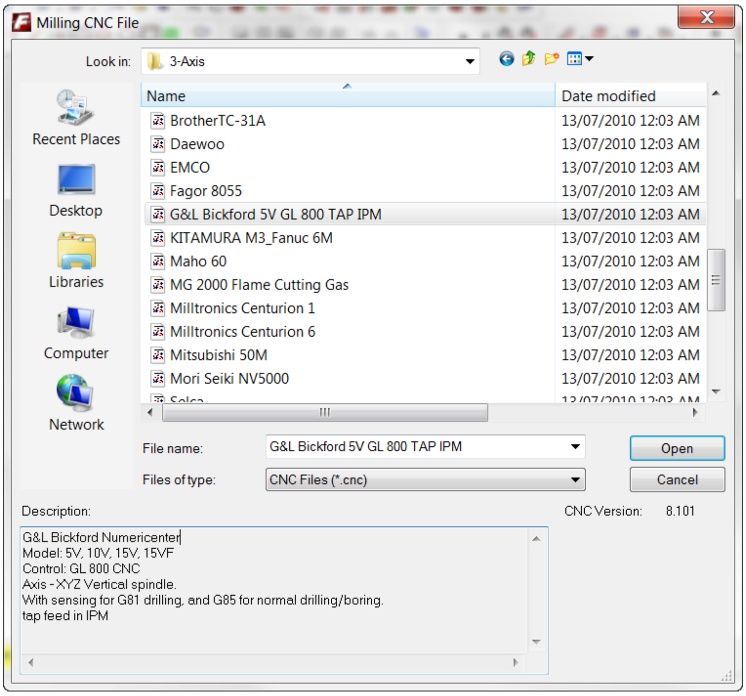
Setup2



  - When you run the Simulation, the NC G-Code that will go to the machine is generated.
  - The Post Processor is what determines what the code contains and it must correspond to the machine control that it will be loaded into.
  - There are many Post Processors that come with FeatureCAM and the correct Post must be selected before running the simulation to obtain the proper code. Post processors for Milling are located in C:\Program Files\FeatureCAM\Posts\Mill.

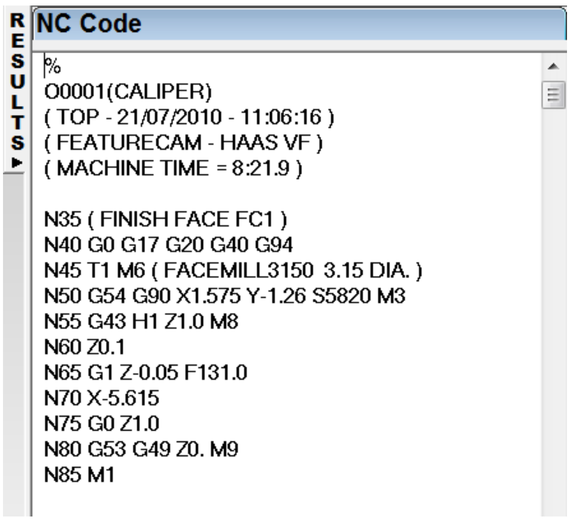


- **Click** on Manufacturing then Post Process to open the Post Options window.



- **Click** on Browse and enter the previous path given to search for and **Select** your Post. **Open** and **OK**.

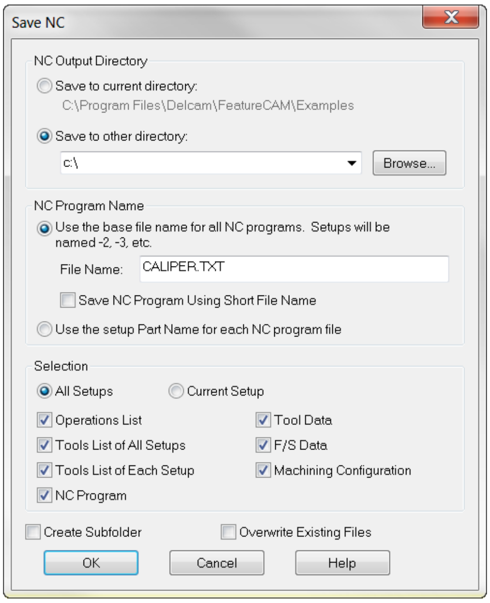
- **Simulate** either 3D or Centerline Simulation again.



- From Results **Click** the NC Code tab.

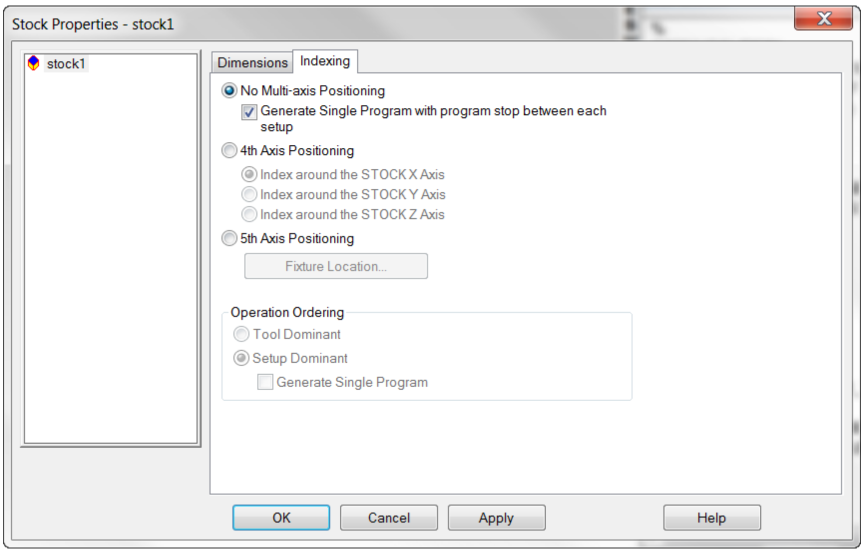
This is what will be sent to the machine control. It is a text file and may be edited but the goal is not to edit the Code but to correct the Post so as to eliminate any editing.

There might be a slight delay to receive the NC Code in the results window. This is because the NC is generated once the NC Code tab is pressed.



- From the file menu click on **Save NC**.
- Save the code to the C:\ drive by choosing "Save to other directory". Check NC Program only in the selection window then press **OK**.
    - If additional boxes are checked a text file will be generated for each that may be printed and sent to the shop to facilitate setup. Check them only as needed because a file is saved on your computer every time you save NC if you have boxes checked even if you do not need them.
    - Now you can take the Caliper.txt NC code file from the C:\ drive and transfer it to the machine to cut setup1 for the part. The Caliper.txt NC file contains G code for setup1 only.

In the Stock Properties and the Indexing tab, if you check the box "Generate Single Program with program stop between setups" then the NC Code will include both setups with a stop between setups. This is so you can run Setup1 then the machine will stop to flip the part then you push the machine start button again to run Setup2.



To open the Stock Properties window, **double click** on stock1 in the part view window on the left side of FeatureCAM.

The Part is Complete.

Snapping and Geometry

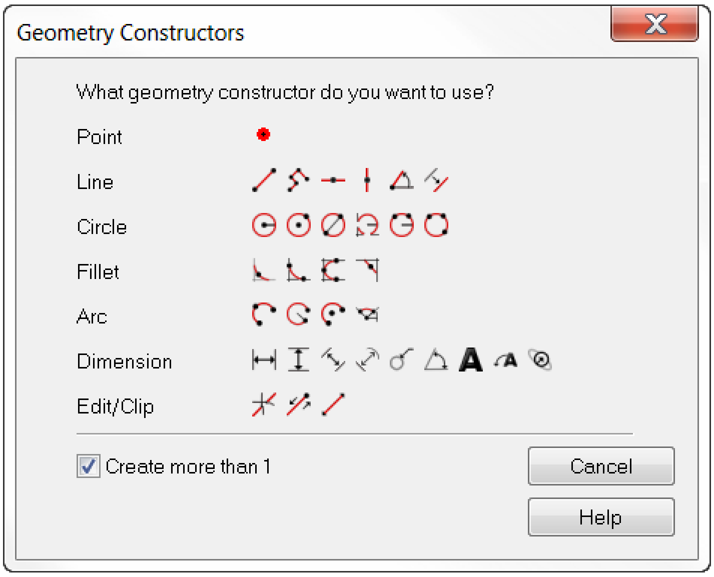
Introduction

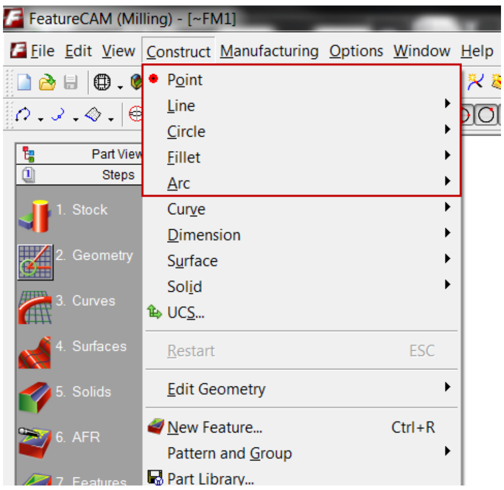
This module will provide an understanding of how different snapping modes work and how Geometry is created within FeatureCAM. By the end of the module the user will be able to construct geometry and understand how each snapping mode functions.

User Interface

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

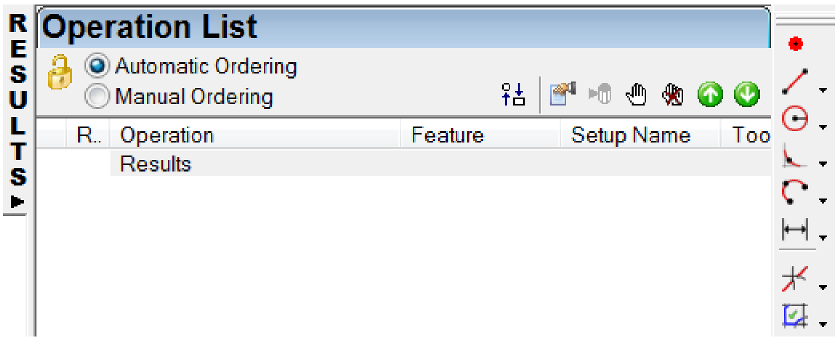
- Commands can be accessed by clicking the Geometry icon in the **Steps** view:





- Can be accessed through the File Menu by clicking on **Construct**

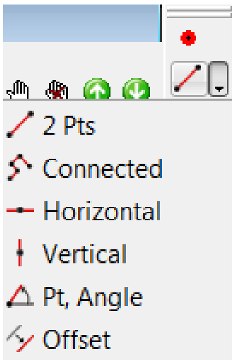
- Or may be accessed by the geometry toolbar (to show the geometry toolbar from the **File Menu** click on **View** then **Toolbars** then check the **Geometry** checkbox and press **OK**)



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

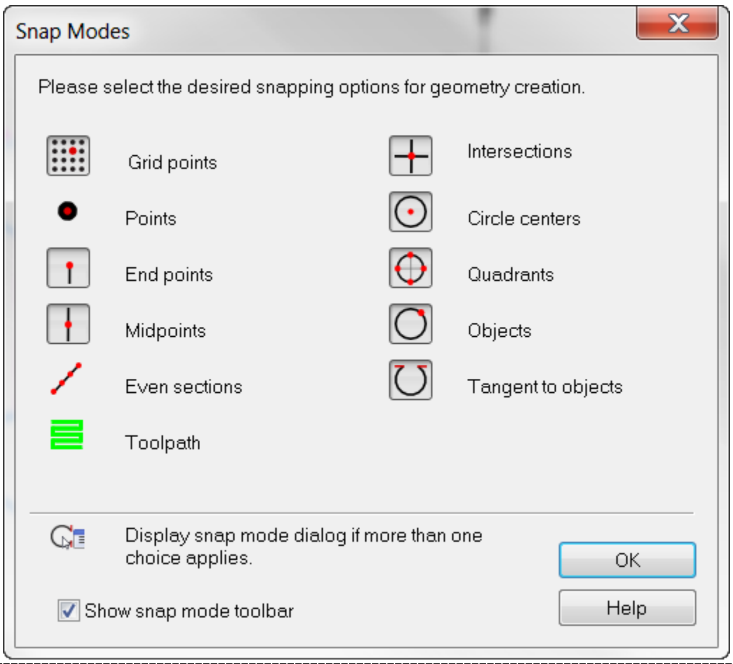
- Click on the triangle to reveal the flyout menu.

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

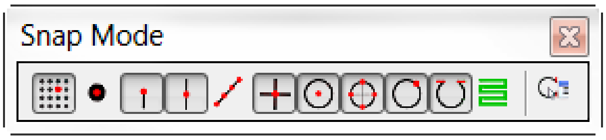


Snapping mode commands can be accessed in two different areas of the user interface.

- Snapping mode commands can be accessed through the **file menu** by clicking on **options** then **snapping modes**



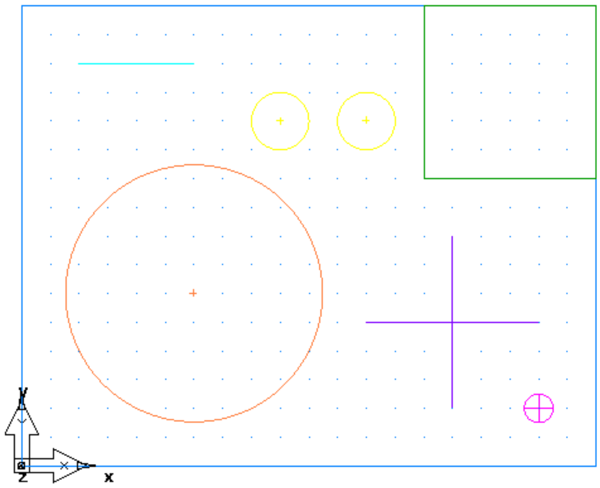
- Snapping modes may also be accessed through the snapping toolbar(to show the snapping toolbar, from the **file menu** click on **view** then **toolbars** then check the **snap mode** checkbox and press **OK**).



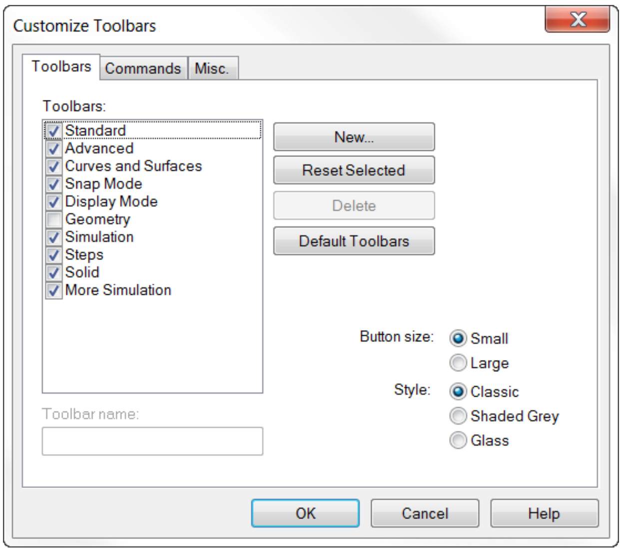
Snapping

- Open FeatureCAM by **double-clicking** the FeatureCAM **shortcut icon** on the desktop: 
- **Click** on **Open an existing file** in the New Part Document wizard window then press **Next**.
- Browse to the examples folder and click on "**Snapping.fm**" then press **open**

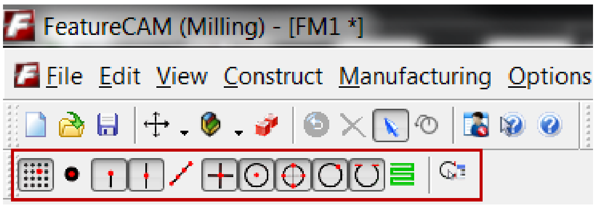
The part when open should look similar to the image below:



- Make sure the snapping toolbar is shown. To show this menu from the **File Menu** click on View then **Toolbars** then check the **Snap Mode** checkbox. Also make sure **Standard** and **Geometry** are also checked then press **OK**.



The **snapping toolbar** will appear:



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

Snap to grid causes the cursor to snap to predefined increments defined under **Options** and **Snapping Grids**

Snap to Point snaps to a point object

Snap to Endpoint snaps to the ends of finite lines and arcs. Endpoint also applies to the corners of the stock and vertices of a solid

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

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.

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

Snap to Center snaps to arc and circle centers. This setting also controls the display of circle and arc center points.

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

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.

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

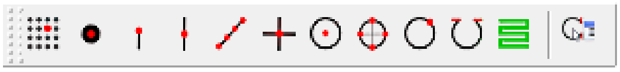
Snap to Toolpath will snap to toolpath lines

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.

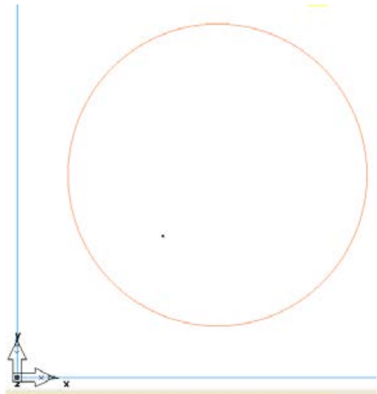
Snapping Exercise

The exercise below will review each of the snapping modes on the snapping toolbar from left to right. When complete it will provide a basic overview of each snapping modes function.

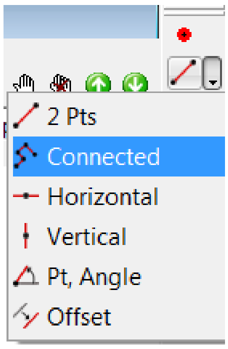
- Make sure none of the snapping buttons are depressed



- **Right-click** on the Orange circle and press **Center Selected**

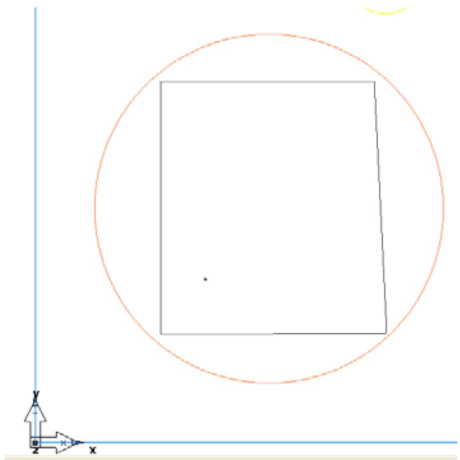


- Click on the **connected** line constructor button from the geometry toolbar



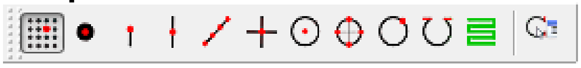
- Draw a square in the center of the circle by **clicking** once in the **upper left**, then **upper right**, then **lower right**, then **lower left** then **upper left**.

If your hand does not move perfectly it will be impossible to do this. We have a tool that can help snap to a Cartesian grid

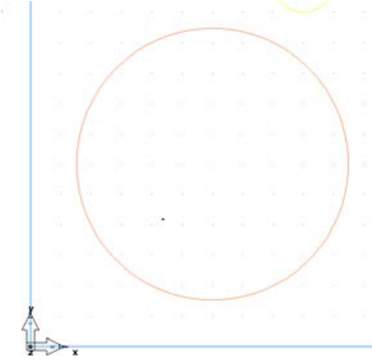


- Press **Undo**  until the line segments disappear. This button is at the top of FeatureCAM.

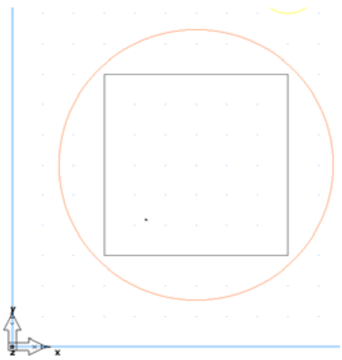
- Make sure none of the snapping buttons are depressed then press the **Snap to Grid** button



You should then notice a faint grid appears



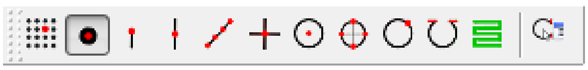
- Draw a square in the center of the circle by **clicking** once in the **upper left**, then **upper right**, then **lower right**, then **lower left** then **upper left**.



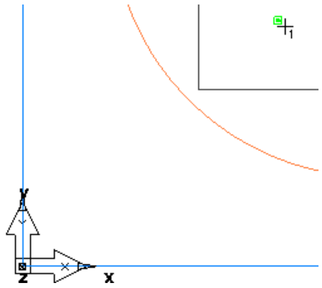
Click on the Select icon  at the top of FeatureCAM on the Standard Toolbar to cancel the line drawing. Use this button at any time to exit selection mode.

You should notice that the square created was much easier to create. This is because the cursor snaps to a predefined grid designated under the **File Menu** under **options** then **snapping options.** Snap to grid overrides all other snapping options so it is suggested to leave this option off if you need to use any of the other snapping modes.

- **Uncheck** the **snapping grid** and check snap to point on the snapping toolbar. Select **connected line** from the geometry toolbar again.

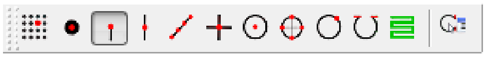


- **Move** the mouse **cursor** around the screen and try to get it to **snap** to an object on the screen

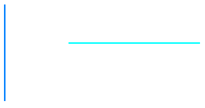


Notice that the only object we can get this to snap to is a point. The only object snap to point will snap to is a point. There is a point inside the square you just created, snap to it.

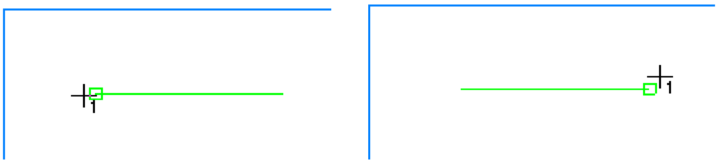
- Uncheck **Snap to Point** and check snap to endpoint.



- Locate the teal horizontal line segment located in the upper left hand of the screen. If you are still zoomed in, right click your mouse in the graphics window, off the part, and select Center All. **Select** connected line again from the Geometry toolbar.

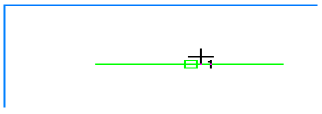


- **Move** the mouse **cursor** from the **left to the right** over the line.



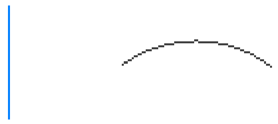
Observe that only the line segment endpoints are snapped to. Snap to Endpoint snaps to the ends of finite lines and arcs. Endpoint also applies to the corners of the stock and vertices of a solid.

- Uncheck **Snap to Endpoint** and check **Snap to Midpoint**. Again position the mouse cursor over the teal horizontal line segment. **Hover** the mouse cursor over the center of the line segment.

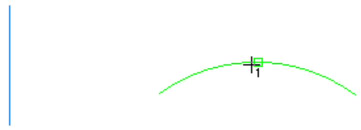


The cursor should snap to the center of the line segment.

- Locate the black arc below the teal horizontal line segment

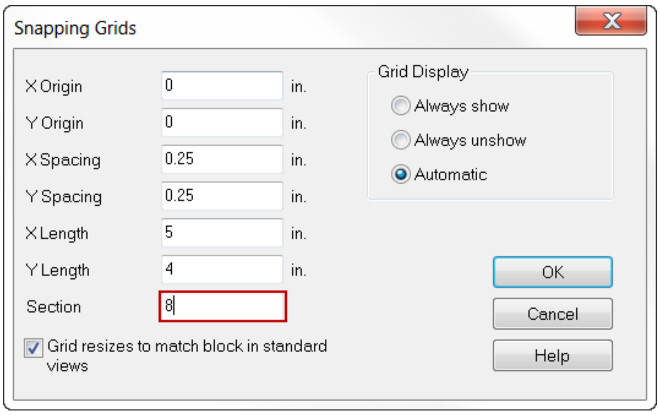


- **Snap to the center** of the line segment

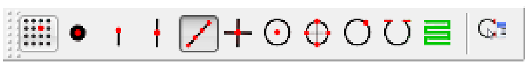


Snap to Midpoint snaps to the middle points of finite lines and arcs.

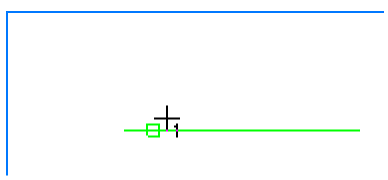
- Click on **options** then **snapping grids** then change **Section** to 8.



- **Uncheck snap to midpoint** then **check snap to segment**

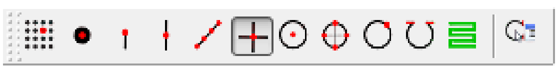


- **Hover** the mouse cursor over the teal line segment and **move** mouse **cursor** slowly from **left to right**.

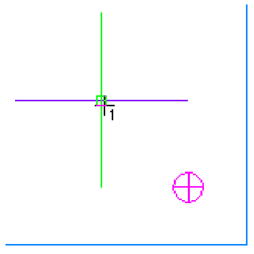


Notice how the line segment snaps to 8 different sections.

- **Uncheck snap to segment** then **check snap to intersection**

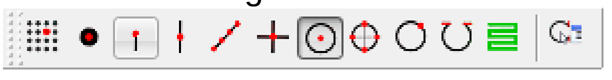


- Locate the violet horizontal and vertical line segments towards the lower right hand part of the stock **hover** the mouse **cursor** over the **intersection**.

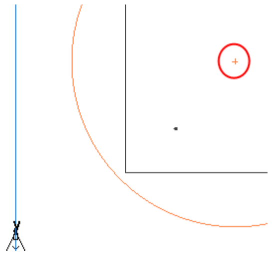


Notice it snaps to the intersection of the lines. This snap mode works for the intersections of arcs, lines and circles.

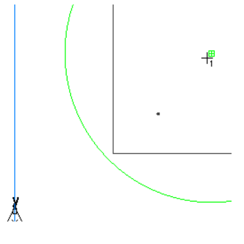
- Uncheck the **snap to intersection** button and make sure no snap modes are enabled. Observe the center of the large circle on the left hand side of the part. Then toggle the **snap to center** button then re-examine the center of the large circle.



What you should notice is that a cross-hair appears at the center of the circle. This is the snap point for the center of the circle

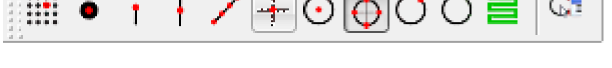


- Hover the mouse cursor over the cross-hair and notice the cursor snaps to that location

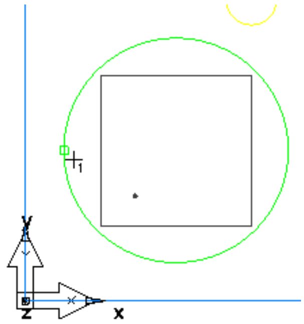


Snap to center snaps to arc and circle centers. This setting also controls the display of circle and arc center points.

- **Uncheck snap to center** then **check snap to quadrant**

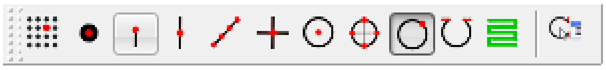


- Locate the large circle and move the mouse cursor around the perimeter of the circle.

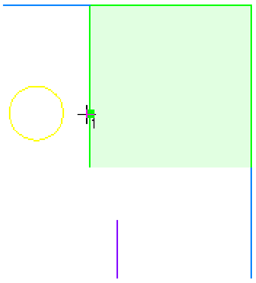


The circle will snap in four places. Snap to quadrant snaps to the four points on a circle corresponding to 0°, 90°, 180° and 270°.

- **Uncheck snap to quadrant** and **check snap to object**

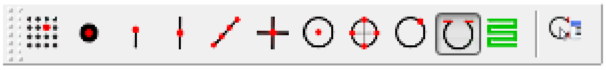


- Hover the mouse over all objects in the viewing window.

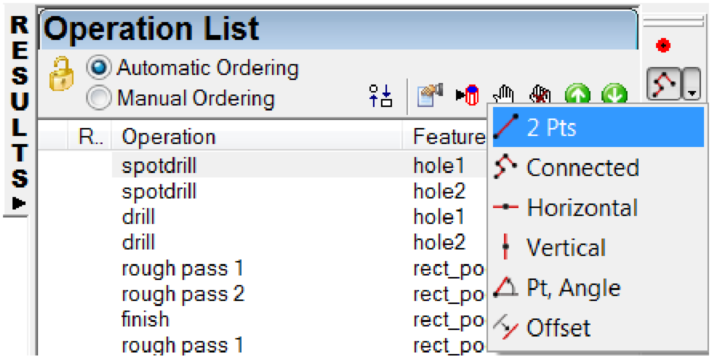


What you will see is that the cursor snaps to all objects in viewing window. This snapping mode is very useful to snap to any type of object including solids and surfaces.

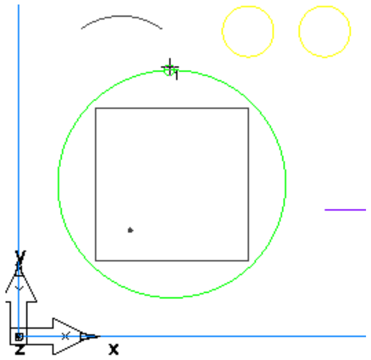
- **Uncheck snap to object** and then **check snap to tangent**



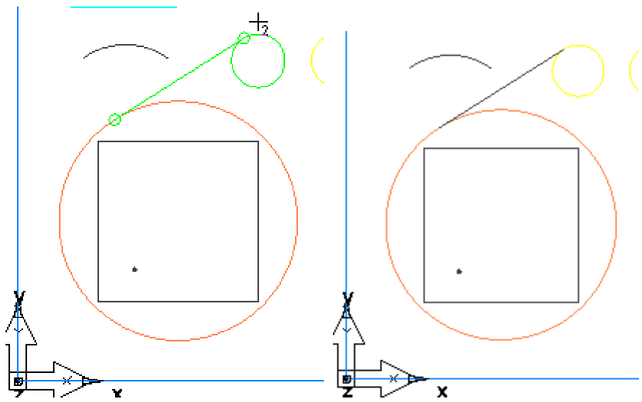
- **Click** on the **line** from **2 points** constructor.



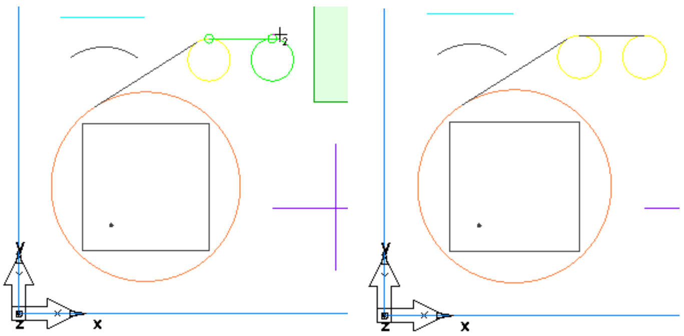
- **Click** once on the large orange circle.



- **Click** next on the small yellow circle on the left.

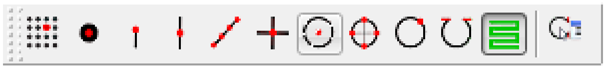


- **Click** again near the top of the small yellow circle on the left, then click near the top of the yellow circle on the right.

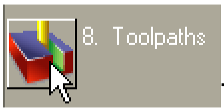


Observe that Snap to Tangent snaps the point so that the object you are creating will be tangent to the object you snapped to.

- **Uncheck Snap to Tangent** then **check Snap to Toolpath**



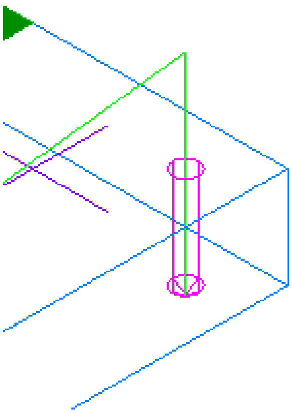
- **Click** on the **toolpaths** icon from the steps menu



The simulation toolbar should appear

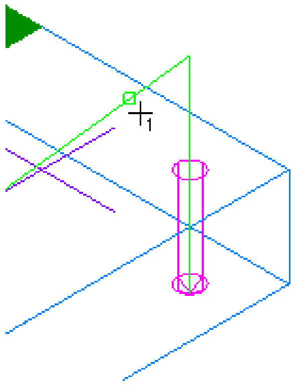
- Click on the **play** icon from the simulation toolbar

- **Right-click** in the viewing window and click **isometric**



Notice the green line representing the toolpath for the drilled hole feature.

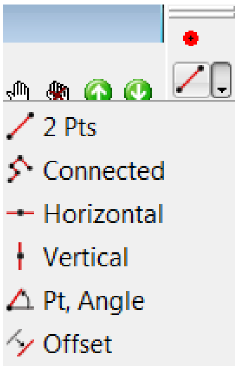
- Select a **line** from **2 points** again from the Geometry toolbar and hover the mouse over the toolpath



Observe how the cursor snaps to the toolpath at any point along that toolpath

Line Constructors

We will now give a summary of what each items function on the line constructor flyout toolbar followed by an exercise.



- Click on the line constructor flyout menu from the geometry toolbar

2 Pts: Line created with two endpoints.

Connected: Multiple lines created in succession. Endpoint of one line becomes start point of next line.

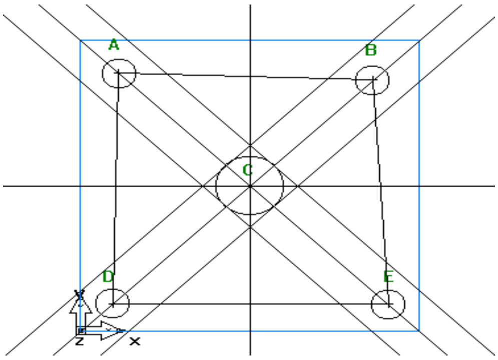
Horizontal: Infinite horizontal line created through a point.

Vertical: Infinite vertical line created through a point.

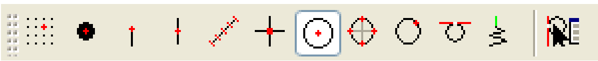
Pt. Angle: Infinite line created through a point at an angle specified in degrees.

Offset: Line, circle or arc created offset from another. New lines have the same length as original

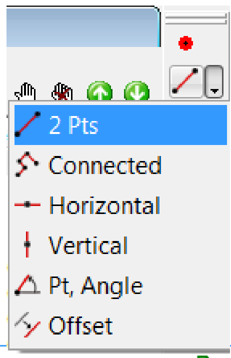
Line Constructor Exercise



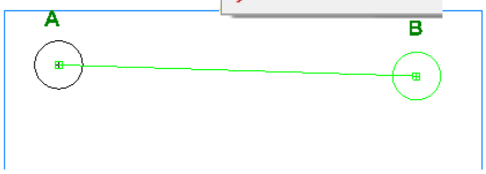
- **Open Lines_example.fm**
- From the snapping toolbar click on **Snap to Circle/center**.

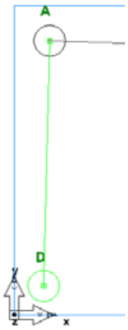


- From the line constructor flyout menu select **Line from Two Points**.

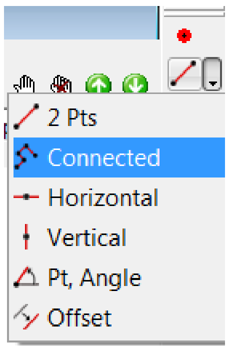


- Use a 2 point Line to create a line from A to B. To do this **click** once on the circle center at **A** then **click** again at point **B**.



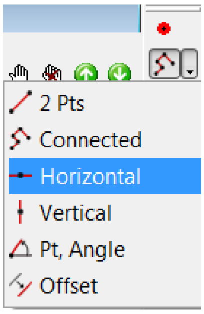


- Use a 2 point Line to create a line from A to D.

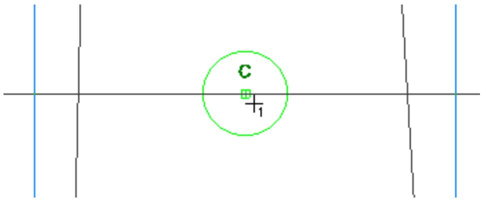


- From the line constructor flyout menu click on **connected**.
- Now use a connected line to create a line from B to E to D. **Click** once on **B** then **click** once on **E** then once on **D**.

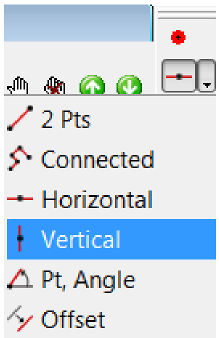
- Use the horizontal line to place a line through C. **Click** on the **horizontal** line constructor from the flyout menu

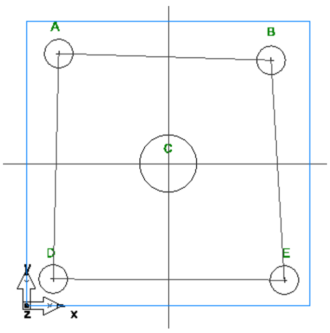


Click once at point **C** to create a horizontal line



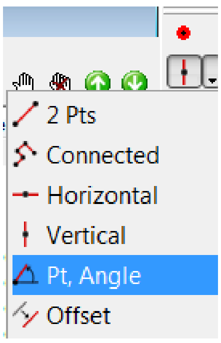
- Use the vertical line tool to place a line through C. Click on the **vertical line constructor** from the flyout menu



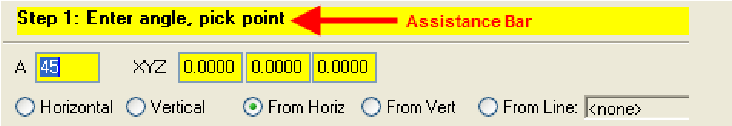


Click once at **point C** to create a vertical line

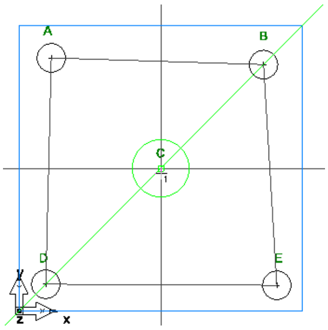
Use the Pt, Angle line to make a 45 deg line through C. Click on the **pt, angle** line constructor from the flyout menu



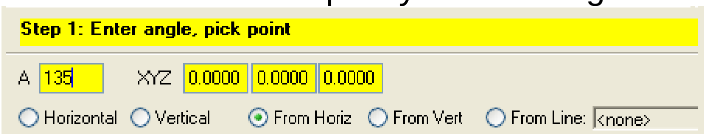
Click below the Assistance Bar and specify the line angle as **45**.



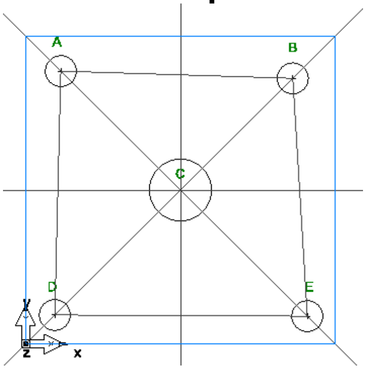
Then **click** at point **C**

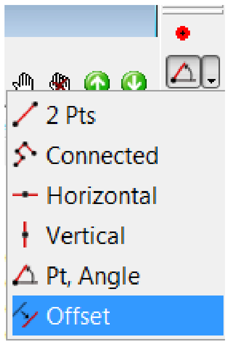


- Use the Pt, Angle line to make a 135 deg line through C. **Click** below the Assistance Bar and specify the line angle as **135**

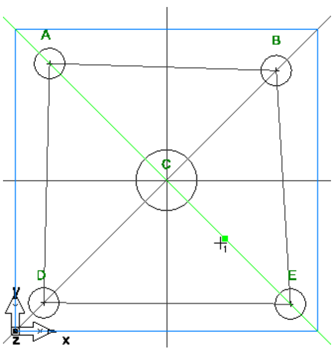


Then click at **point C**.

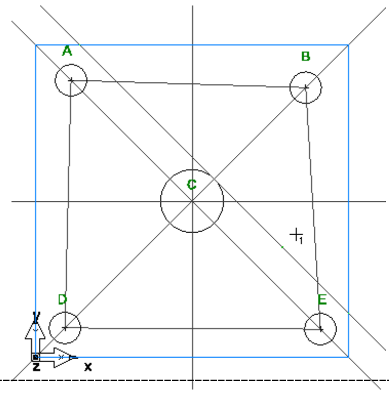




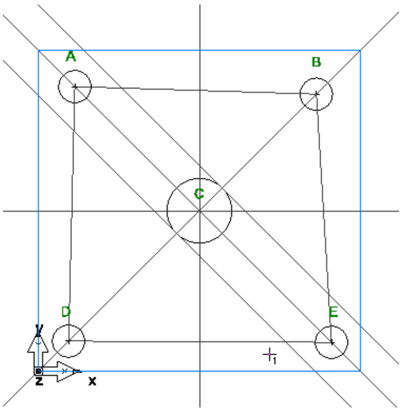
- Use the offset command to offset line A-C-E 0.5 inches on each side. Click on the **Offset** Constructor from the flyout menu



Click once on the Infinite Line through **A-C-E**

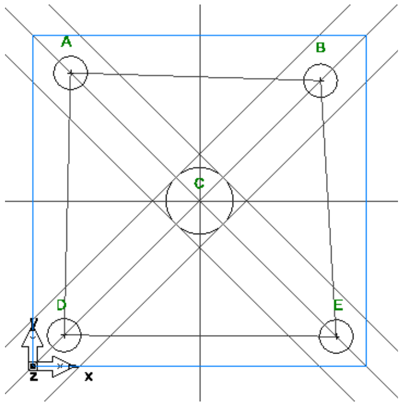


Then **click** once on either side of the line



Repeat for the other side of the line

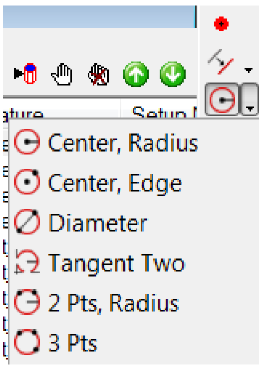
Use the offset to offset line B-C-D 0.5 inches on each side. **Do the same** as the previous step but now offset the infinite line **B-C-D** on both sides



Circle Constructors

Below is a summary of what each item's function is on the circle constructor flyout menu followed by an exercise.

- **Click** on the Circle Constructor Flyout Menu from the geometry toolbar



Center, Radius: Creates a circle by clicking on the center and specifying the radius or dragging the mouse to a radius

Center, Edge: Creates a circle by clicking once on the center then again on the edge

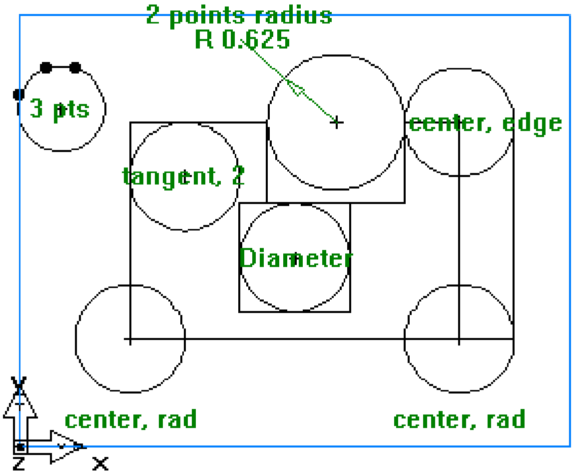
Diameter: Creates a circle with two points with the diameter being the distance between the two points

Tangent Two: Circle created by snapping the circle's edge tangent to two objects

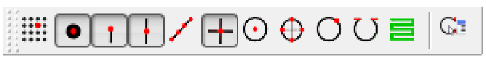
2 Pts, Radius: Creates a circle by specifying two points and a radius

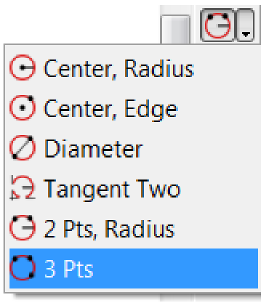
3 Pts: Creates a circle from three points

Circle Constructor Exercise

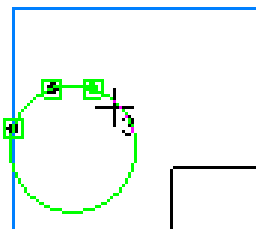


- **Open circles_example.fm**
- From the snapping toolbar **enable snap to point**, **snap to endpoint**, **snap to centerpoint** then **snap to intersection**

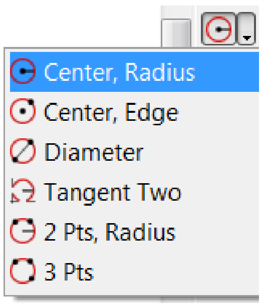




- From the circle constructor flyout menu **click** on **3 Pts**

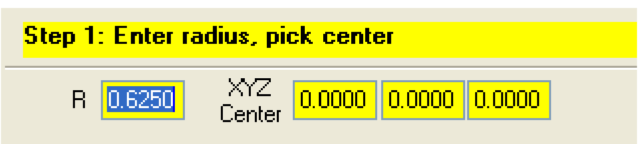


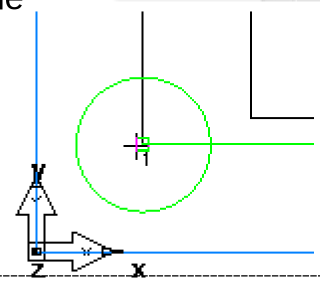
- In the upper left hand corner locate the 3 points and **click** once on **each point**



- From the circle constructor flyout menu **click** on **Center, Radius**

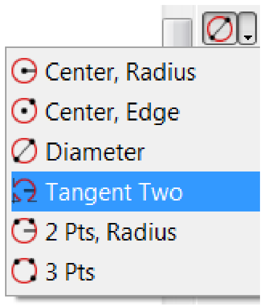
  Below the assistance bar there is a field for the circle radius. Enter **0.625”**.

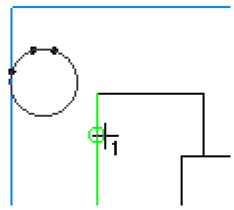




Then click on the corner of the Geometry in the lower left

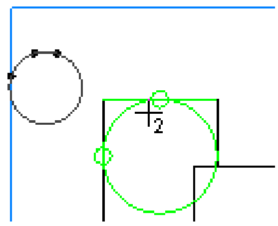
- From the circle constructor flyout menu **click** on **Tangent Two**



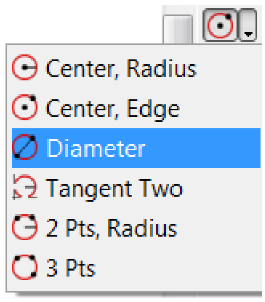


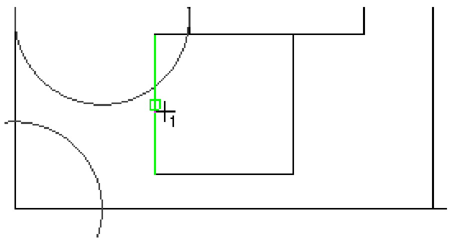
Click once on the line segment in the upper left area as shown below

Then **Click again** on the line segment perpendicular to it



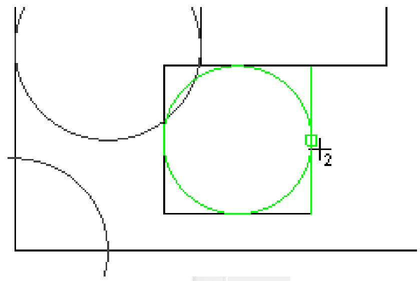
- From the circle contructor flyout menu click on **Diameter**



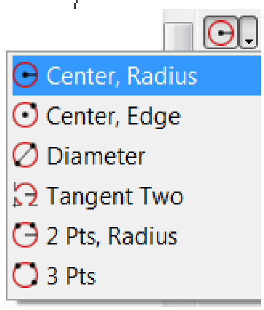


In the square located on the center of the part **Click** once on the left vertical line segment

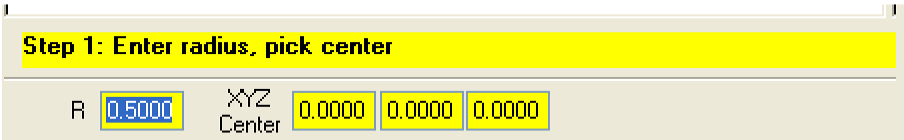
Then **Click** again on the right vertical line segment



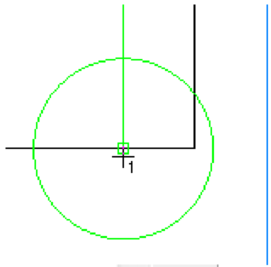
- From the circle constructor flyout menu click on **Center, Radius**



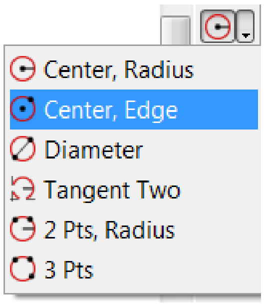
- In the assistance bar below the display window enter a **Radius** of **0.5”**

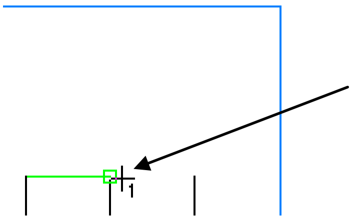


Click once where the perpendicular and vertical lines meet close to the right hand corner:



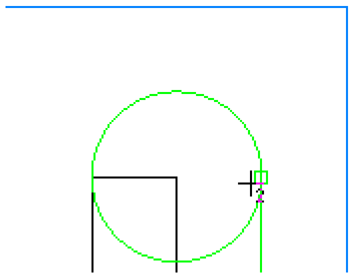
- From the circle constructor flyout menu **click** on **Center, Edge**



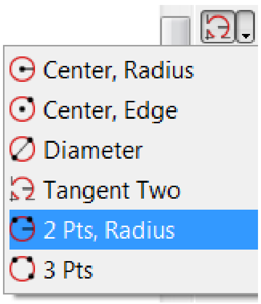


Snap to the rectangular corner located toward the upper right hand of the part and **Click** once

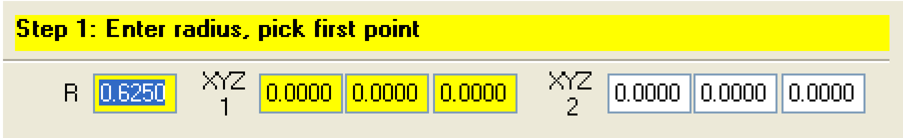
Then **Click** again at the end of the vertical line segment to the right



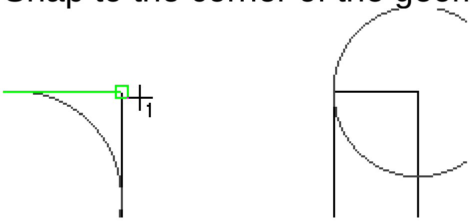
- From the circle constructor menu **click** on **2 Pts, Radius**



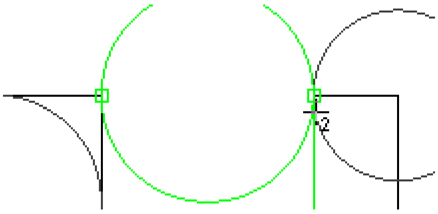
- Enter a Radius value below the assistance bar as 0.625”.



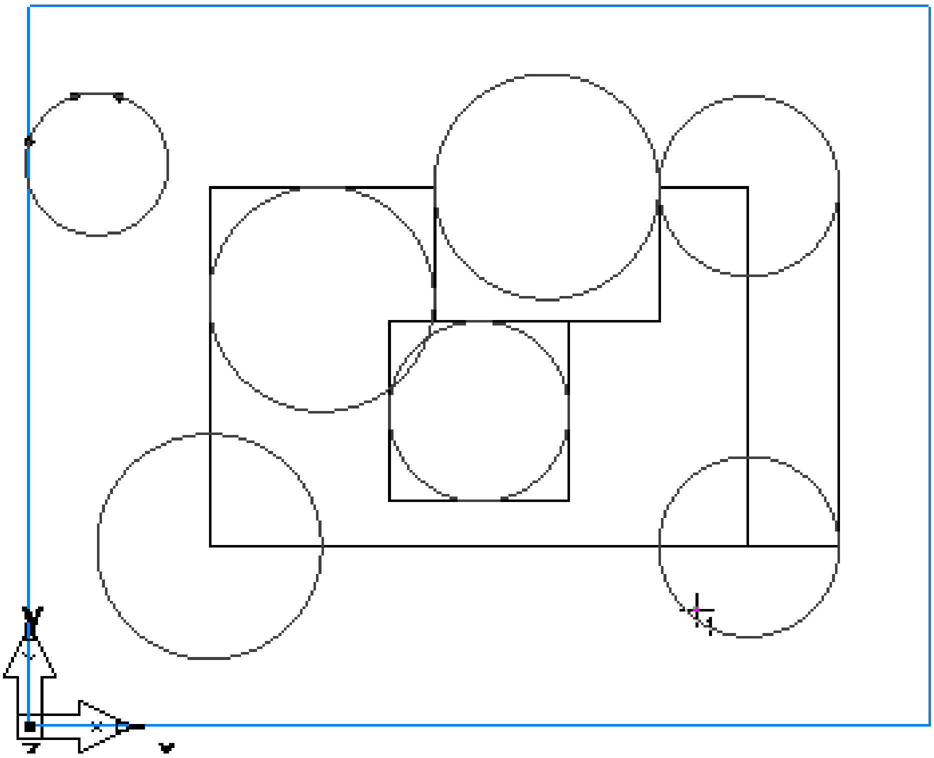
Snap to the corner of the geometry segments shown below and **click**



Then **click** again snapping to the opposite corner shown below



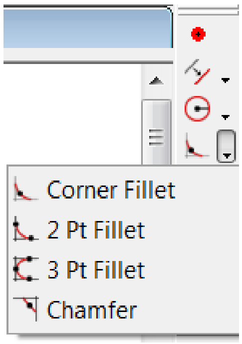
When finished the project should look like the image below



Fillet Constructors

Below is a summary of each item's function is on the fillet constructor flyout menu followed by an exercise.

- **Open fillet_example.fm**
- Click on the fillet constructor flyout menu from the geometry toolbar.



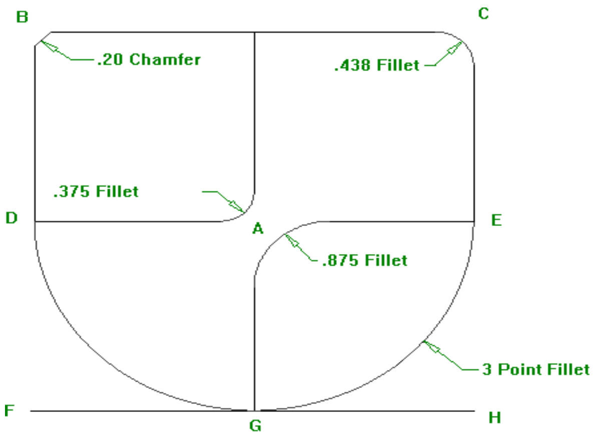
Corner fillet creates a fillet in a corner originally defined by an intersection of lines or arcs.

Two point fillet creates a fillet in a corner originally defined by an intersection of lines or arcs.

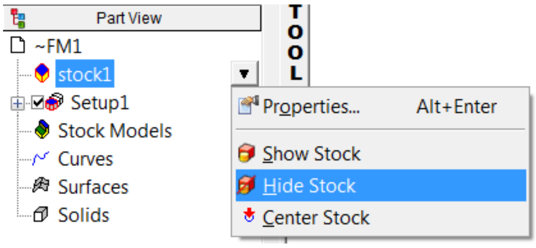
Three points defines a fillet by selecting three points, similar to the three point circle..

Chamfer creates a chamfer in a corner originally defined by an intersection of lines or arcs.

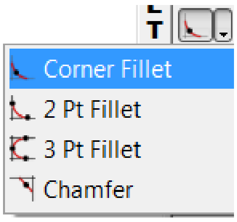
Fillet and Chamfer Constructor Exercise



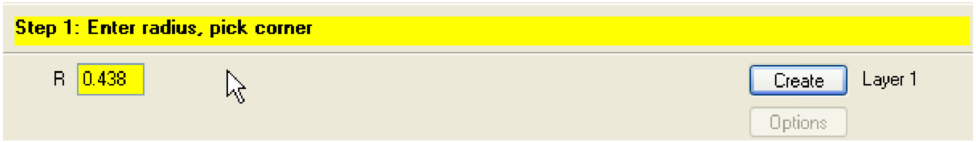
- From the part view **Right-click** on stock then **Left-click** on **Hide Stock**.



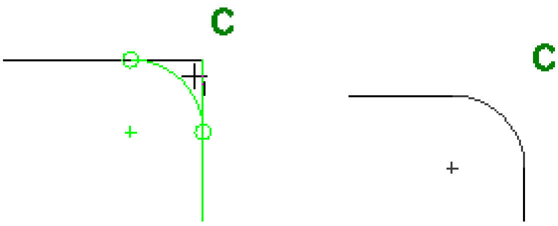
- From the **Fillet Constructor** flyout menu select **Corner Fillet**



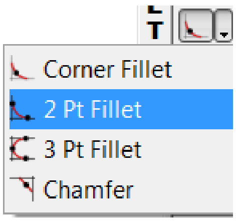
- Below the assistance bar enter **0.438"** for the **Radius**.



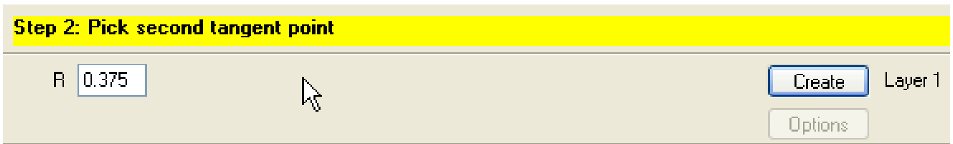
- Move the mouse close to **C** then **Left-click**.



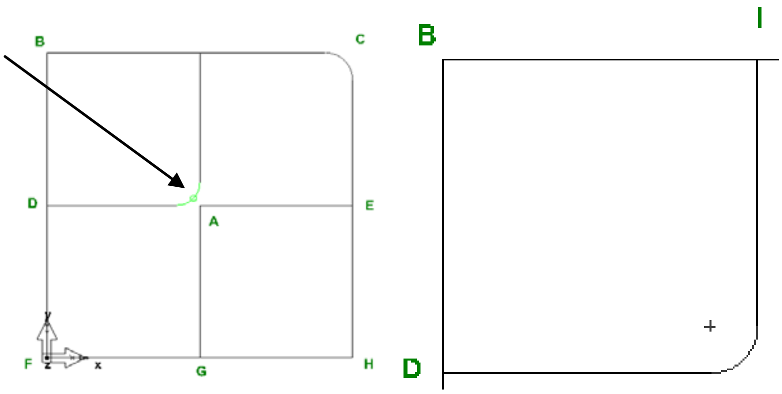
- From the **Fillet Constructor** flyout menu select **2pt fillet**.



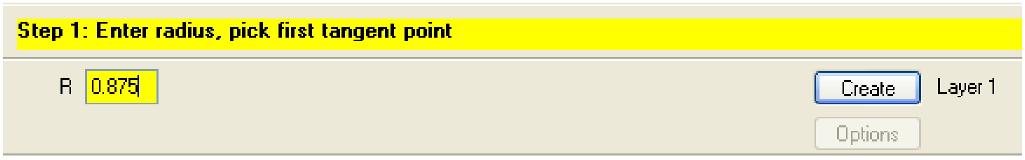
- Below the assistance bar enter **0.375"** for the radius value.



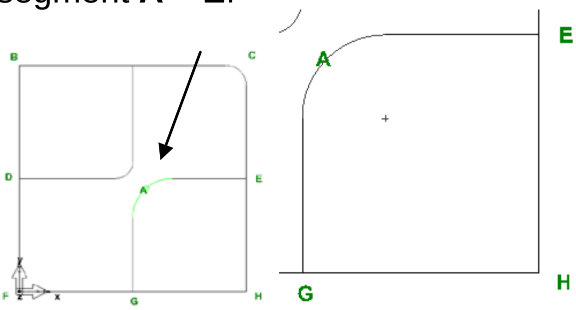
- Create a 2 Pt Fillet between A and D shown below by **clicking** once on the vertical line segment **I - A** then once on the horizontal line segment passing through **A – D**.



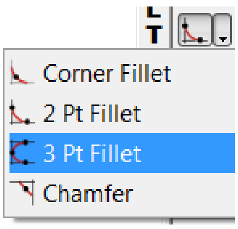
- Below the assistance bar enter a value of **0.875"**.



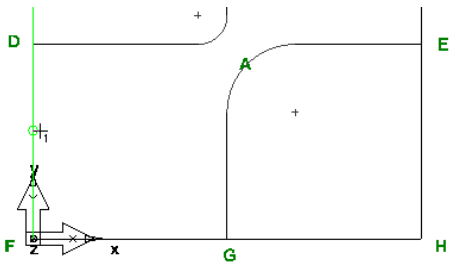
- **Click** once on the line segment between **A – G** and once on the line segment **A – E**.



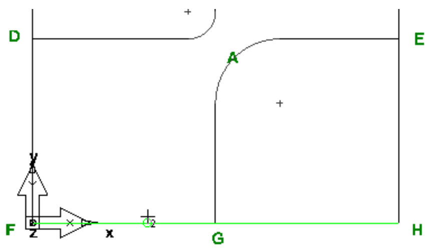
- From the fillet fly out menu **Click** on **3 Pt Fillet**



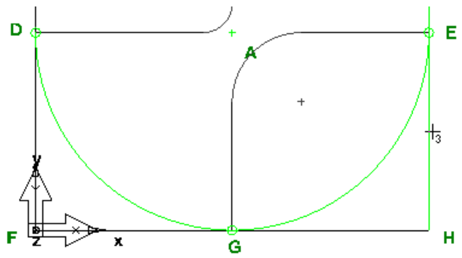
- **Click** once on the line segment **D-F**

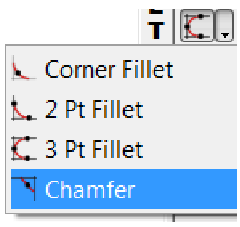


then again on the line segment **F-G**

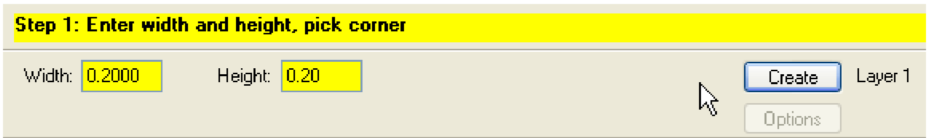


- then once more on the line segment **E-H**

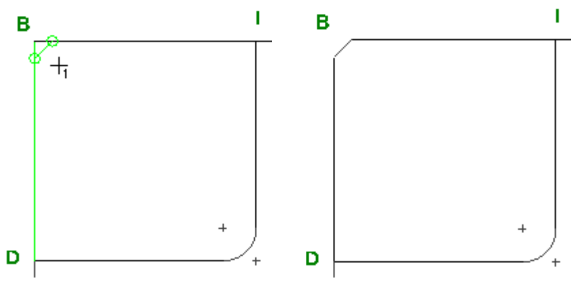


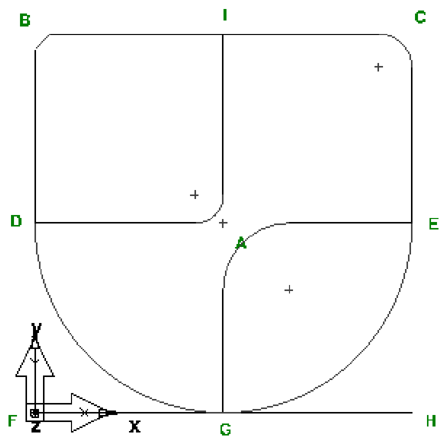


- From the **Fillet Flyout Menu** click on the **Chamfer** command
  From the assistance bar enter the width and height of **0.20"**



- Move the mouse near the line intersection at point B and click



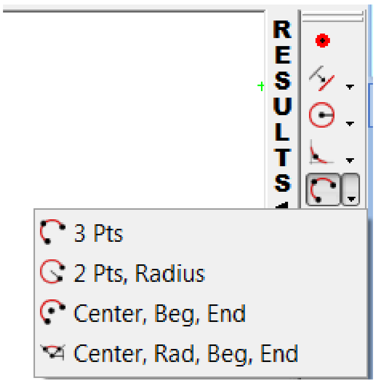


When this exercise is finished you should see results similar to that shown on the right.

Arc Constructors

Below is a summary of what each items function is on the arc constructor flyout menu followed by an exercise.

- **Click** on the arc constructor flyout menu from the geometry toolbar



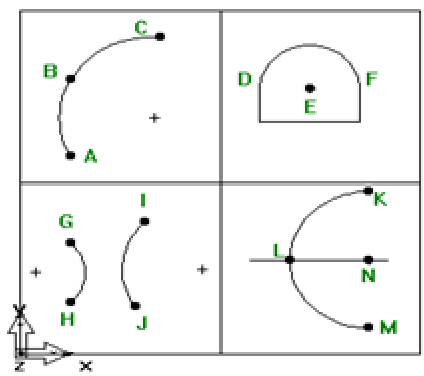
Arc from three points constructs an arc through a start point, edge point, and a finish point.

Arc from two points, radius constructs an arc through two points with a specific radius.

Arc from center, beginning, end constructs an arc from a center point and given beginning and end points.

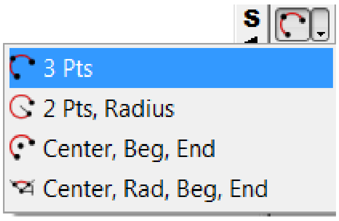
This selection constructs an arc with a specific center and radius with the starting and ending points determined by angles.

Arc Constructor Exercise

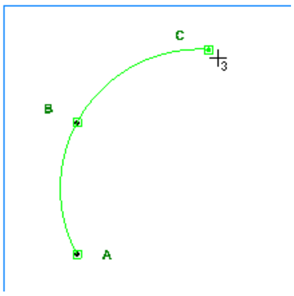


- **Open arcs_example.fm**
- From the snapping toolbar **click** on **snap to point** and **snap to endpoint** as the only snapping modes checked

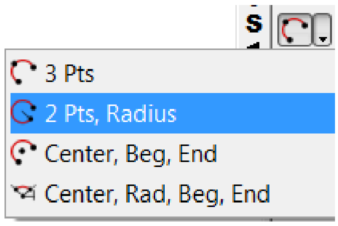
- **Click** on the **arc from three points** constructor from the arc constructor flyout menu



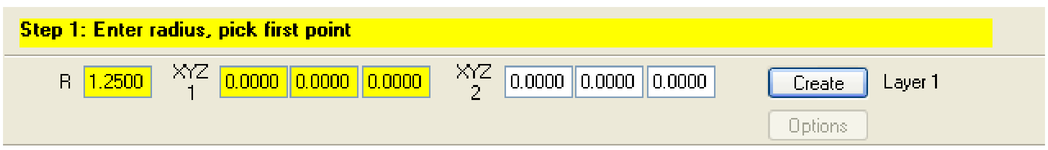
- In the upper left hand quadrant of the stock **click** once on **A**, **B** then **C**.



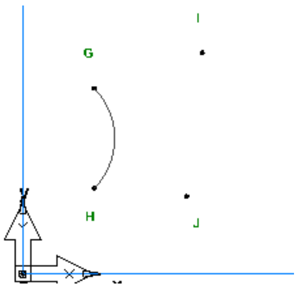
- Click on **2 Pts, Radius** from the arc constructor flyout menu



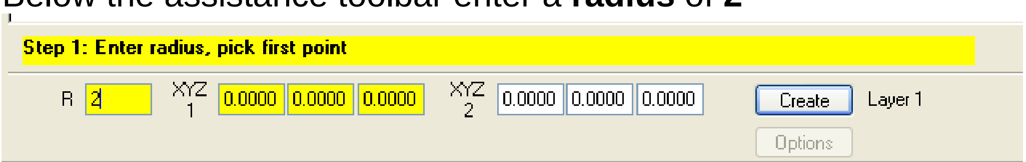
- Below the assistance toolbar enter a **radius** of **1.25”**



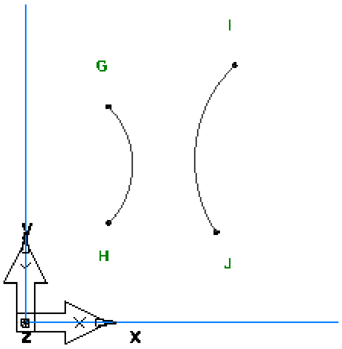
- **Click** once at point **G** then again at point **H**



- Below the assistance toolbar enter a **radius** of **2”**

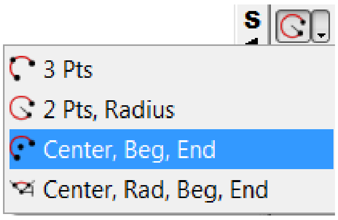


- **Click** once at point **J** then again at Point **I**

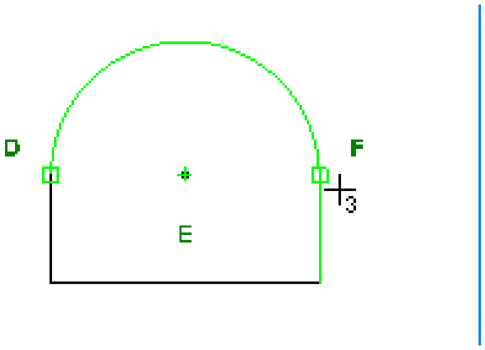


Notice that depending on which point is picked first determines the direction of the arc.

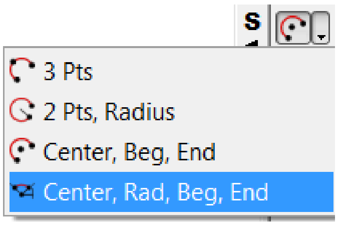
- **Click** on the **arc from Center, Beg, End** from the arc constructor flyout menu



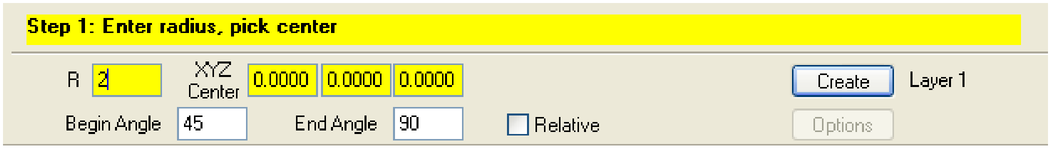
- **Click** once at **point E**, then at **point F** and then at **point D**.



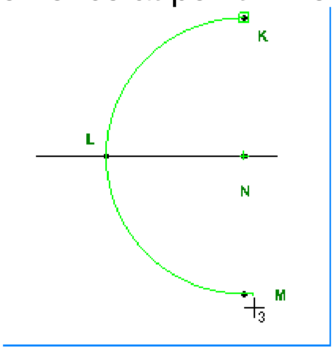
- **Click** on the **arc from Center, Rad, Beg, End** from the arc constructor flyout menu



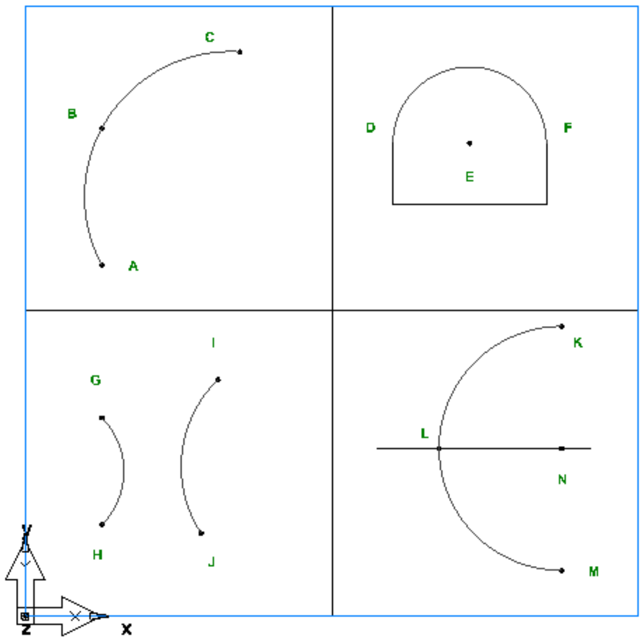
- Below the Assistance toolbar enter **2"** for the **Radius**



Click once at point **N** next at point **K** and finally at point **M**



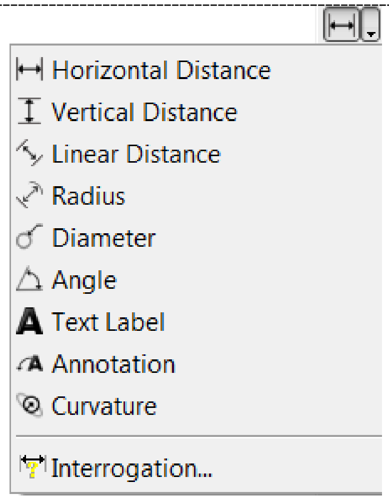
When the project is complete it should look similar to the image on the right.



Dimensioning

Below is a summary of what each item's function is on the Dimensioning flyout menu followed by an exercise.

- Click on the dimensioning flyout menu from the geometry toolbar



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.

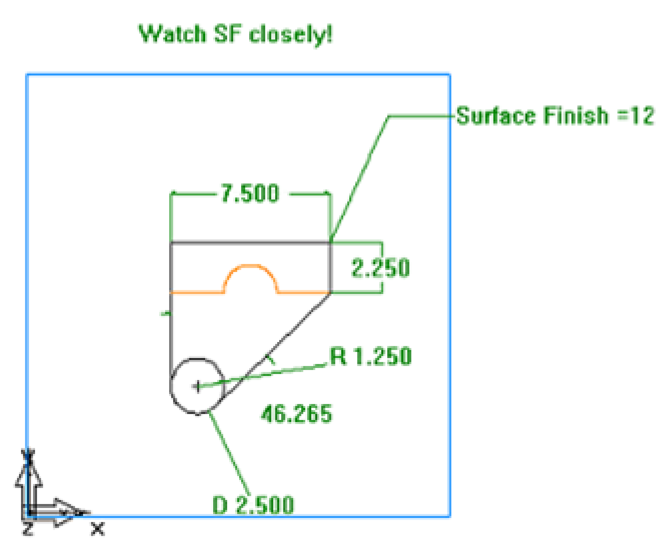
Text label creates a text label entered in the dialogue bar

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

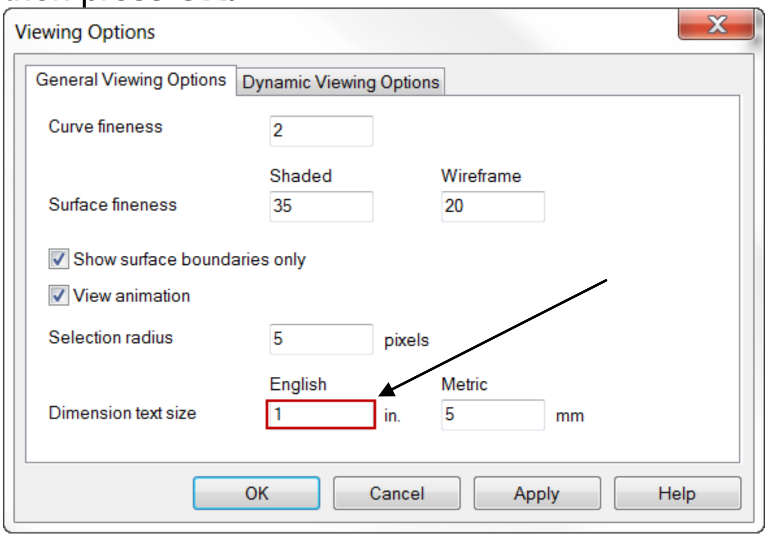
Curvature samples the surface and computes the curvature in two directions to describe how the surface behaves at the point shown in the dialog bar. Unlike the other dimensioning tools, Curvature is a real-time rubber-banding effect where you traverse the surface to find the point with the smallest curvature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

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.

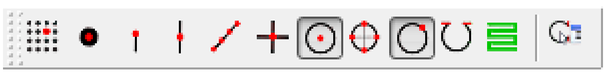
Dimensioning Constructor Exercise



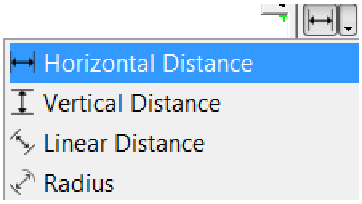
- **Open dimension_ex.fm**
- **Click** on **options** then on **viewing**. In the **Dimension text size** enter **1** then press **OK**.



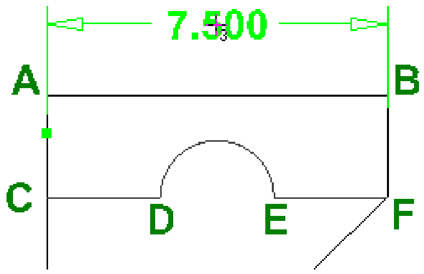
- On the snapping toolbar check **snap to center** and **snap to object** only.



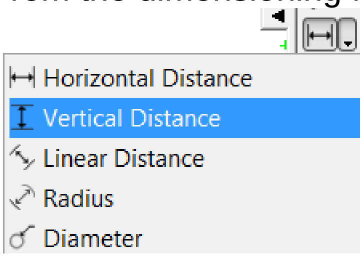
- From the Dimensioning flyout menu on the geometry toolbar select **Horizontal Distance**



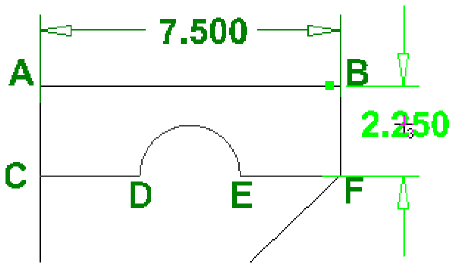
- Click once on the vertical line segment, close to A, then click again on the vertical line segment at B then click once again above the horizontal line segment between A and B



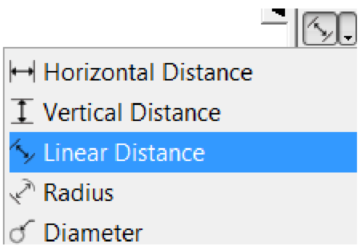
- From the dimensioning flyout menu **click** on **Vertical distance**



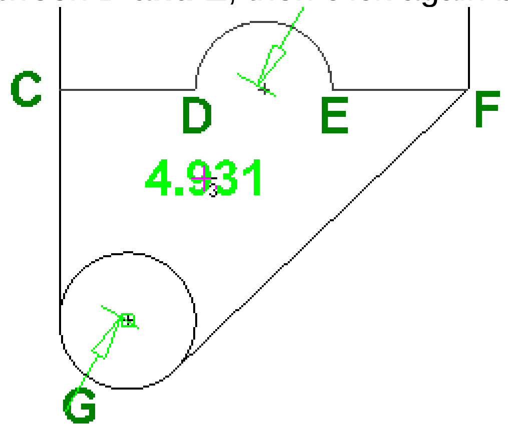
- **Click** once on the horizontal line segment between **A and B** , **click** once on the horizontal line segment **E – F** then **click** again to the right of the vertical line segment **B-F**



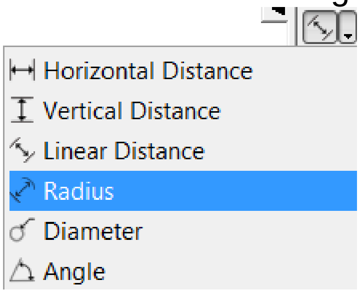
- From the Dimensioning Flyout menu **Click** on **linear distance**



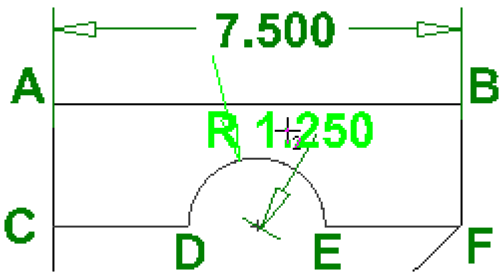
- **Click** once at the circle center of **circle G** , click again at the **arc center** between **D and E**, then click again between the two previous points



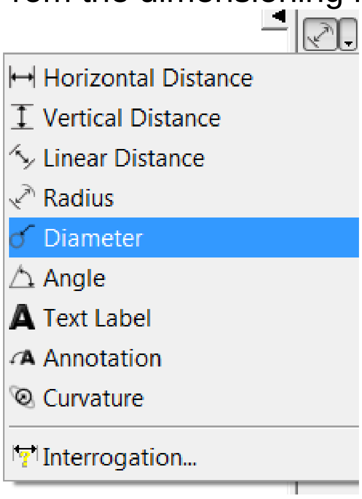
- From the dimensioning flyout menu **click Radius**



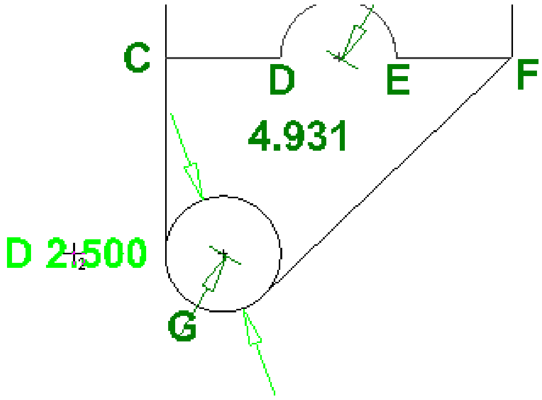
- **Click** once on the Arc at **D – E** then **Click** again slightly **above the arc**



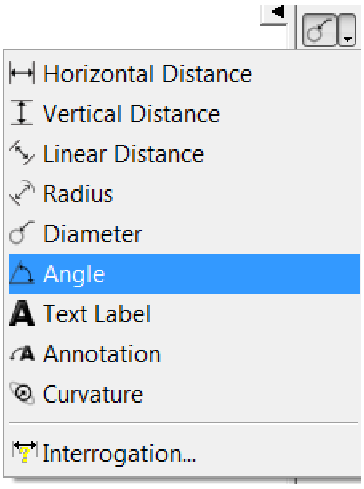
- From the dimensioning menu click on **Diameter**



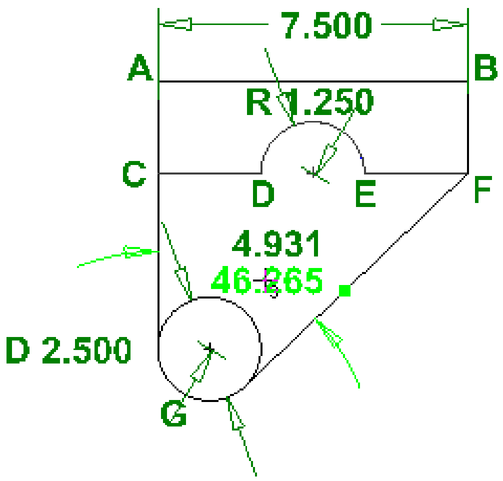
- **Click** once on the circle located at **G** then **click** again to the **left** of circle **G**



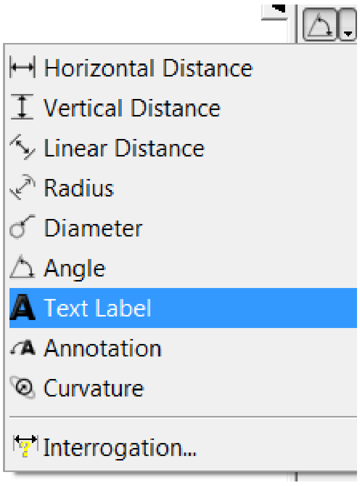
- From the Dimension flyout menu select **Angle**.



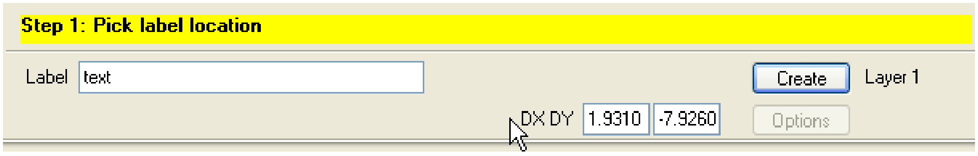
- **Click** once on line segment **F–G** then click once on line segment **C-G** then **Click again** somewhere between the two



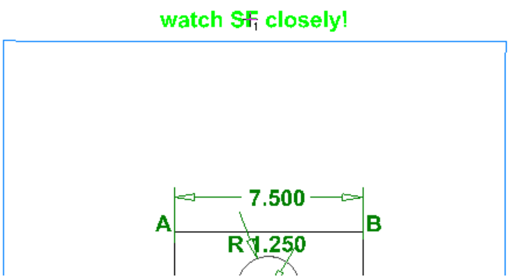
- From the dimensioning flyout menu **click** on **Text Label**.



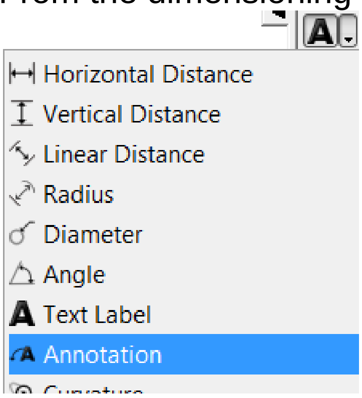
- Below the assistance bar **enter** the **text** “Watch SF closely!”



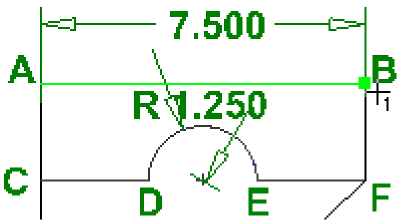
- **Click** once above the stock outline



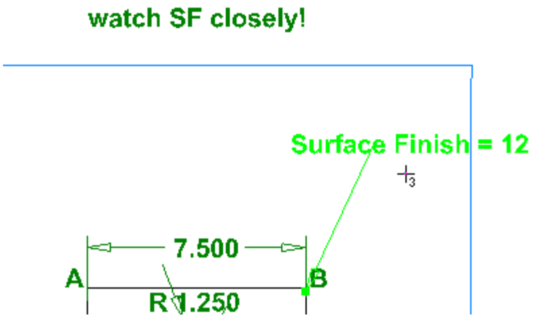
- From the dimensioning flyout menu select **Annotation**



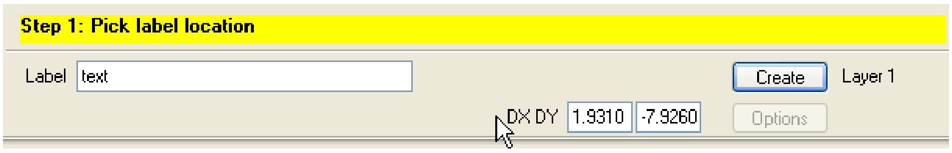
- **Click** once at point **B** then **click** again towards the upper right hand corner of the stock outline



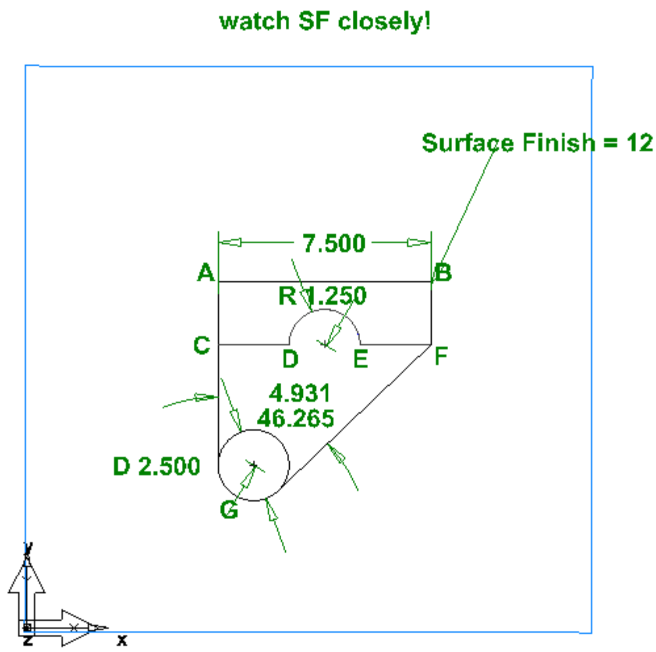
- **Click** again towards the upper right hand of the stock



- Below the assistance bar enter the text “Surface Finish = 12” then press the Create button.



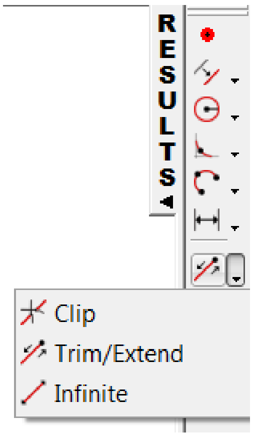
When the project is finished you should have results similar to those shown below.



Clipping

Below is a summary of what each item’s function is on the clipping flyout menu followed by an exercise.

- Click on the clipping flyout menu from the geometry toolbar

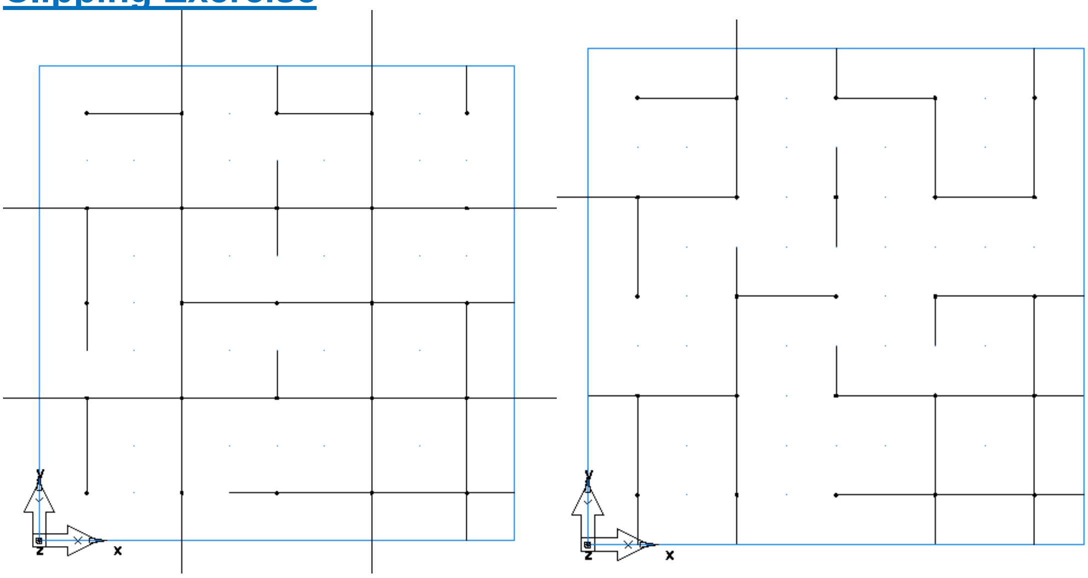


Clip: Clip removes a region of a line, arc, circle or curve. A region is defined as a portion of an object between two intersection points.

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:

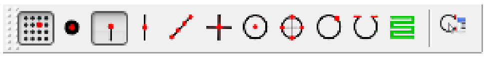
Infinite: changes the length of a line or an arc.

Clipping Exercise

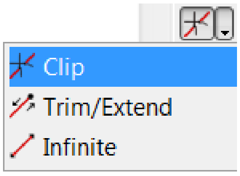


The purpose of this exercise is to become familiar with the clipping, trim/extend and infinite geometry modification functions. These functions will be shown once then try to make the file clipping example look similar to the image above

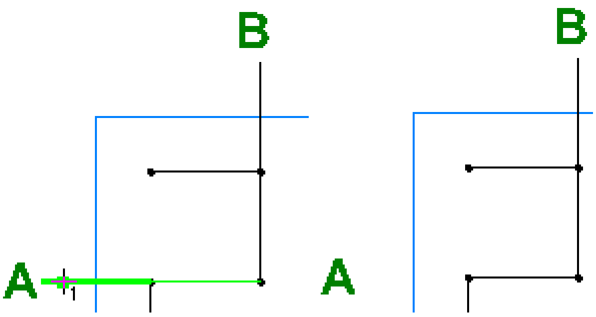
- Open **clipping_example.fm**
- From the snapping toolbar check **snap to grid** and **snap to point** only.



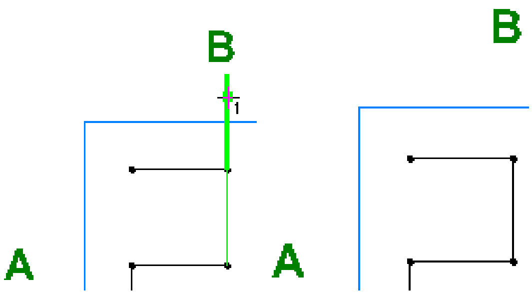
- From the geometry toolbar **Click** on **Clip.**



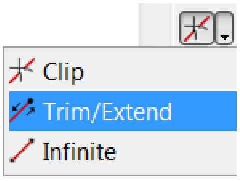
- Locate the horizontal line segment at **A** then **click** on it once



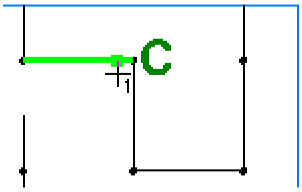
- Locate the vertical line segment at **B** then **click** once



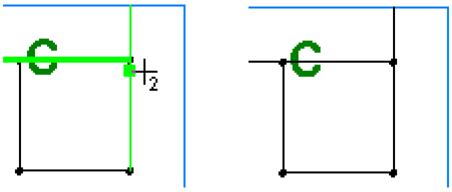
- From the geometry toolbar click on **Trim/Extend**



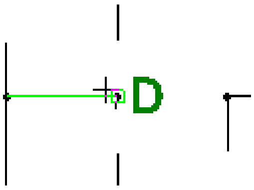
- Locate the Horizontal line segment to the left of **C** and **Click** once



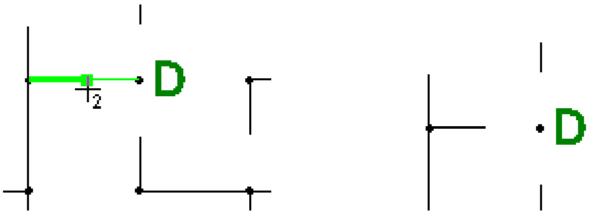
Then **Click** again toward the vertical line and point across from **C**.



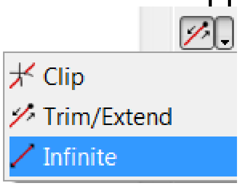
- Locate the horizontal line segment at **D** and **Click** once.



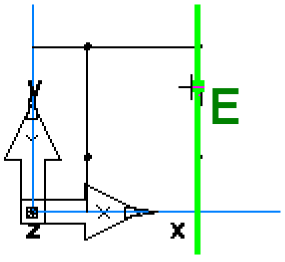
Then **Click** again between and the vertical line



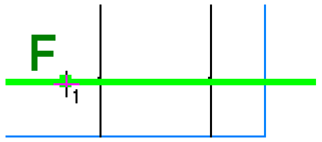
- From the snapping toolbar **Click** on **Infinite**.



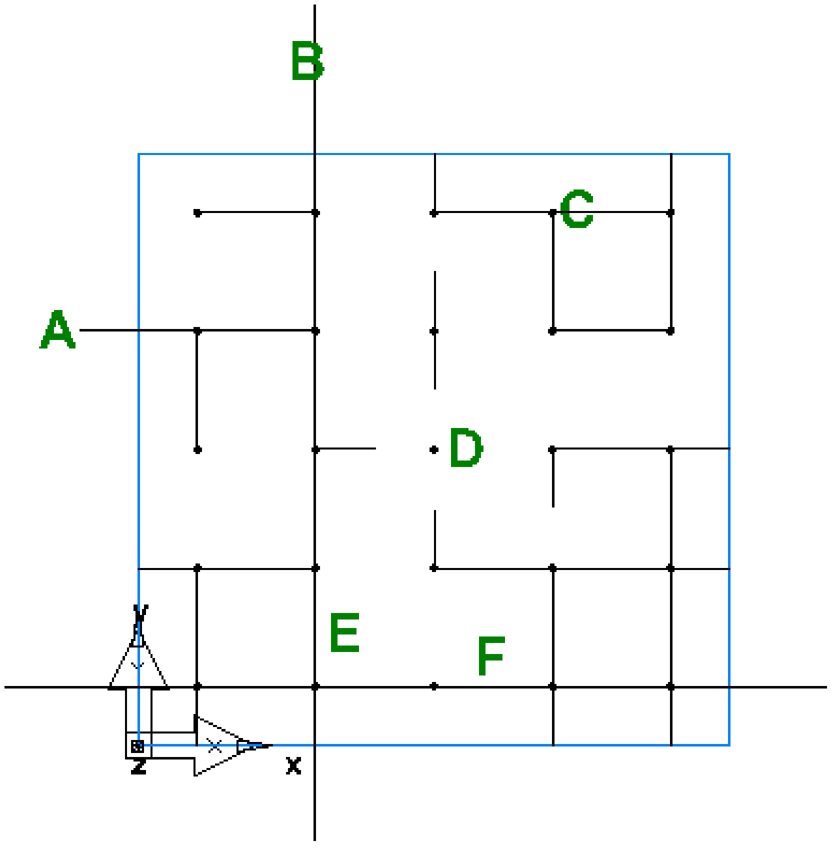
- Locate the Point near the vertical line segment at point **E** and **Click** once



- Locate the horizontal line segment at point **F** and **Click** once



When finished, you should have results similar to the image below:



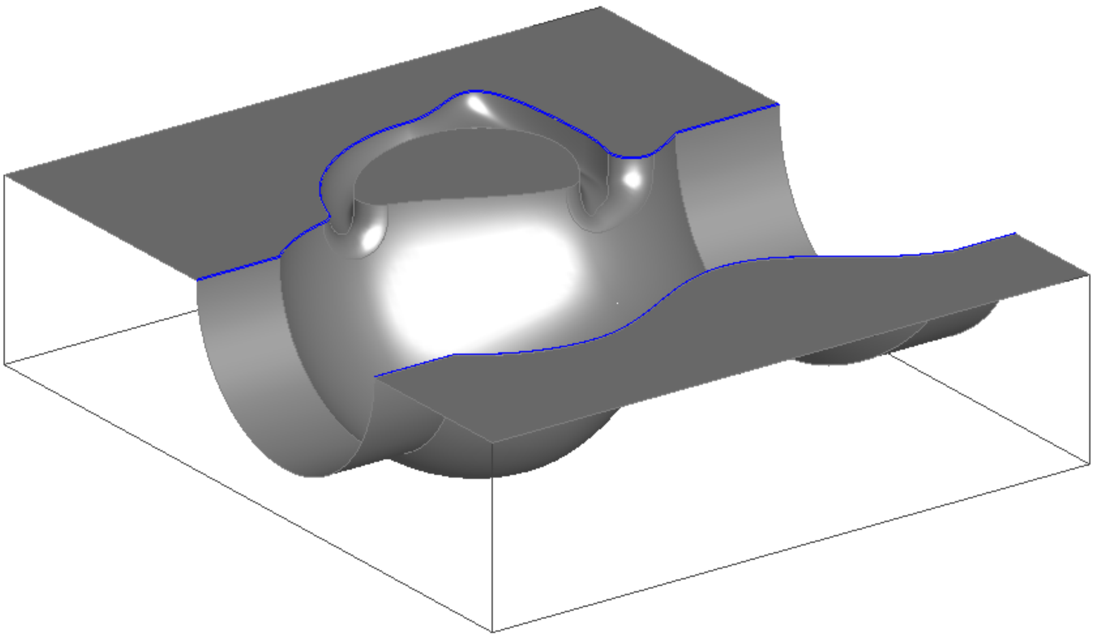
Curve Creation

Introduction

This module will provide an 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 and import curves from other objects.

What is a 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. You can create curves from lines and arcs, from other curves, from points, from CAM dimensions, from Windows fonts, or from surfaces.

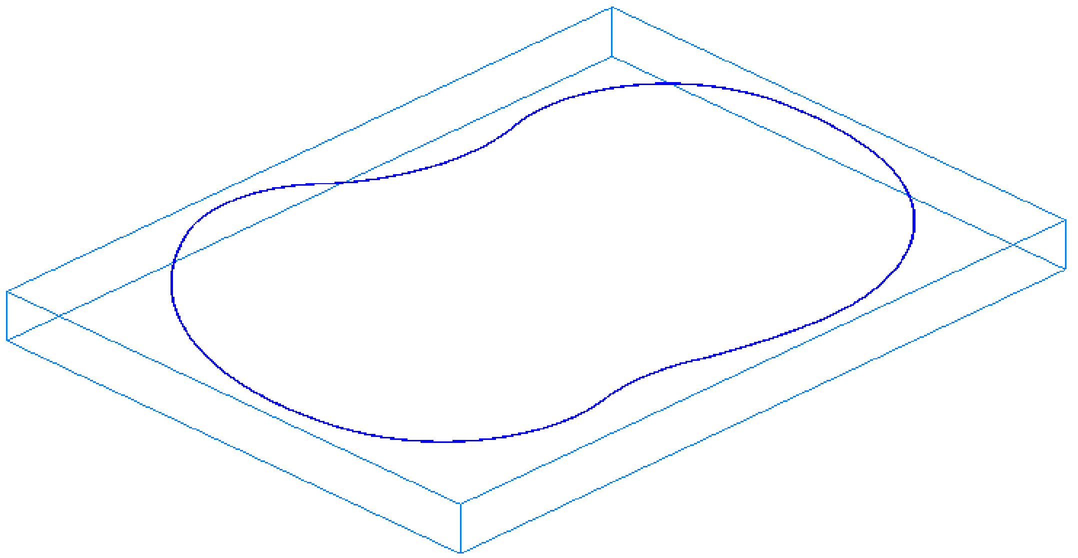


Chaining

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.

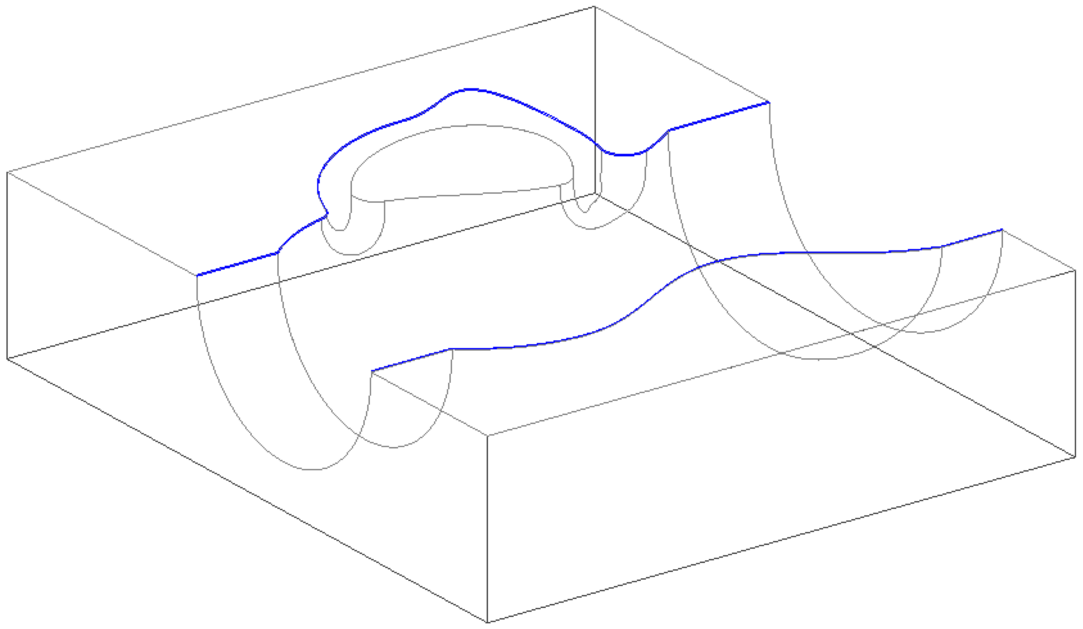
Closed Curves

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 or by simply double-clicking the closed geometry using the pick pieces command.



Open Curves

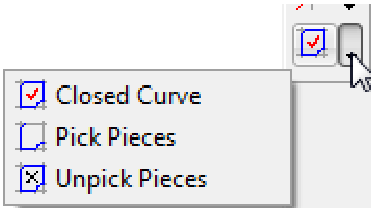
Open curves have end points that do not meet. You can only use open curves in Side and Groove features. To chain open curves we use the pick pieces command. The image below shows two open curves on a part.



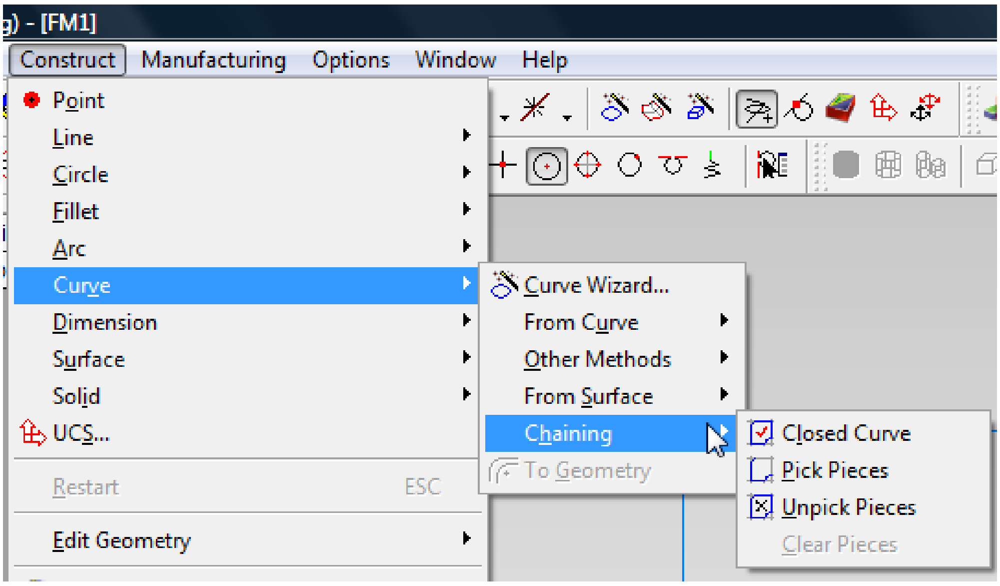
User Interface curve chaining

Curve chaining commands can be accessed from three different areas of the user interface:

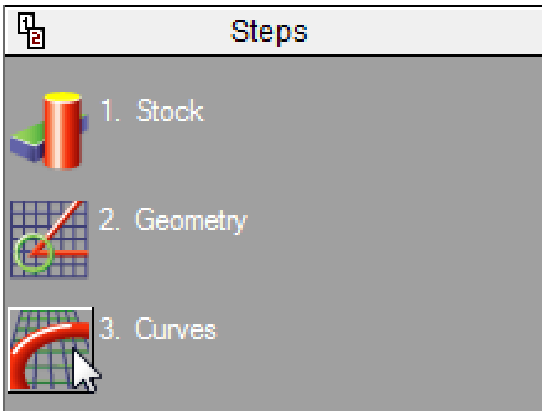
Curve chaining commands may be accessed through the **Geometry toolbar**



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



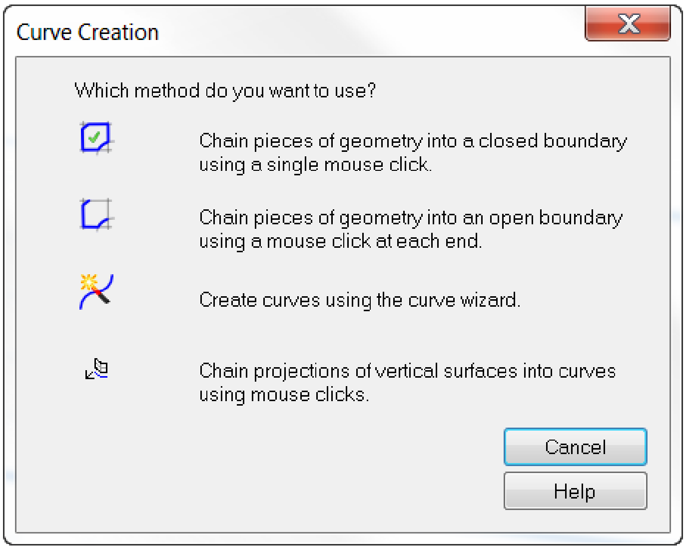
Or may be accessed by clicking on **Curves** from the **Steps Toolbox**



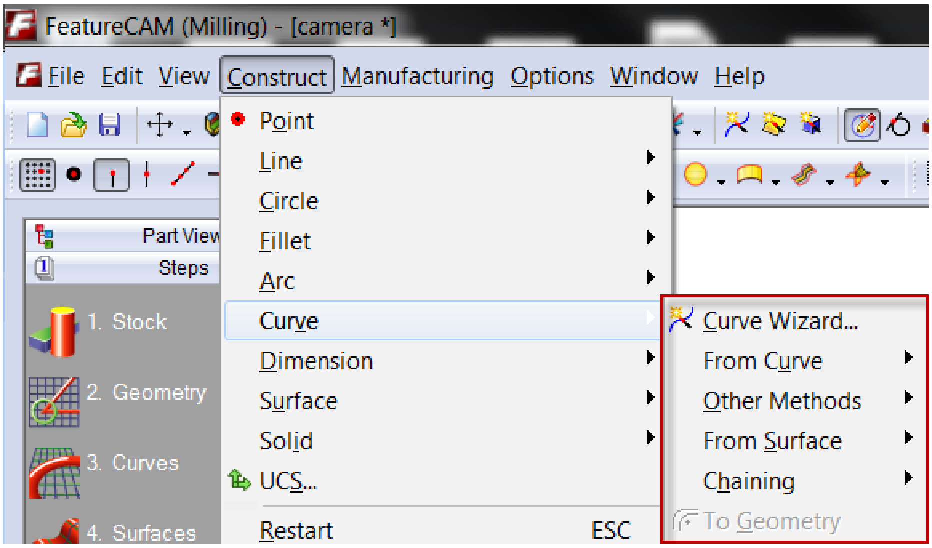
Curve Wizard

Curve constructor commands can be accessed in four different areas of the user interface:

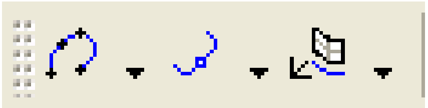
Curve constructor commands are available through the **Steps** menu:



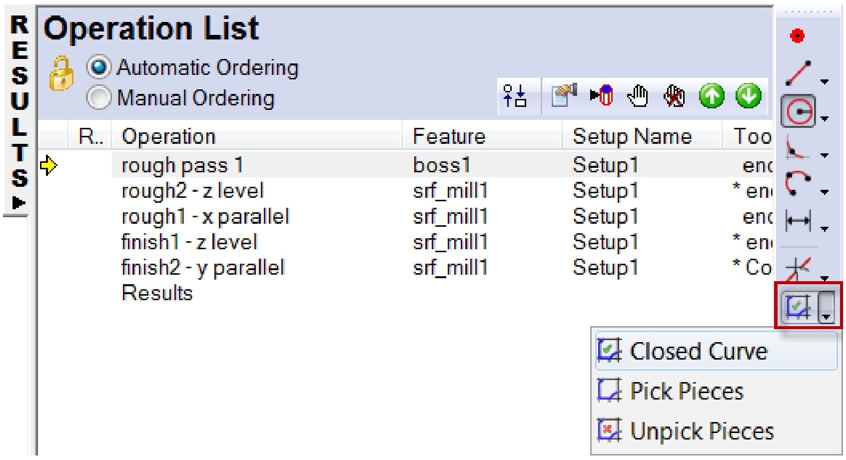
They may also be accessed through the **File menu** by clicking on **Construct** and **Curve:**



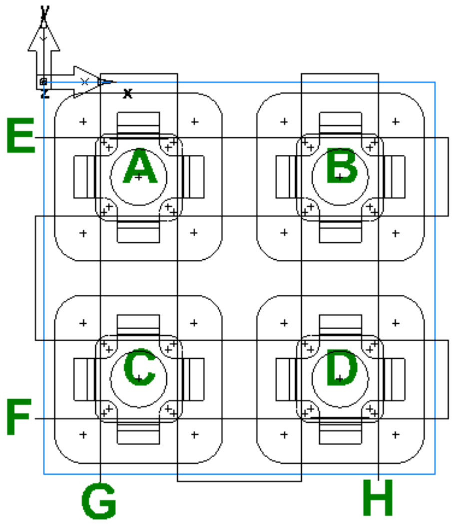
May be accessed through the curve toolbar (to show the curve toolbar, from the file menu click on **View** then **Toolbars** then check the **Curves and Surfaces** checkbox and press **OK**)



Or may be accessed at the bottom of the **Geometry toolbar**(not all functions available here)



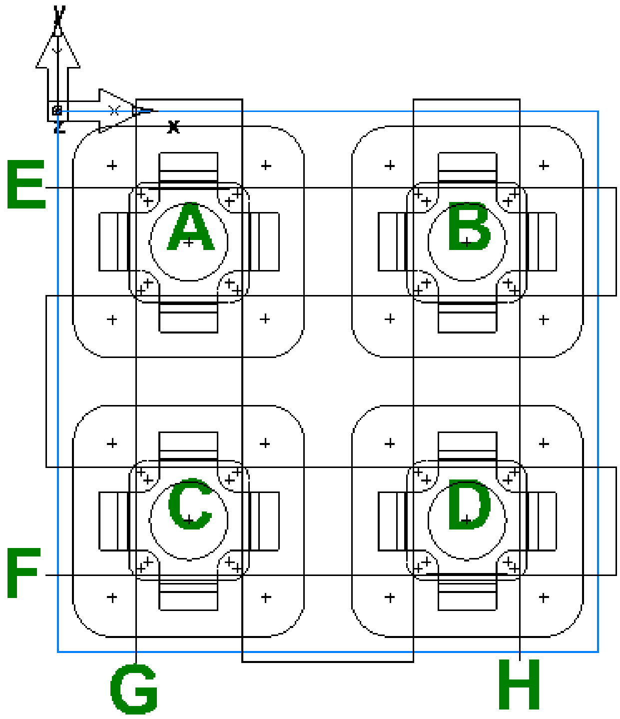
Chaining Open and Closed Curves

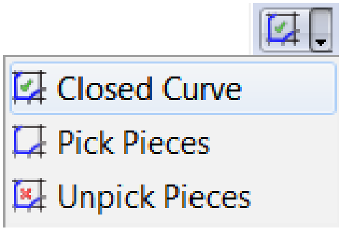


- Open FeatureCAM by **double-clicking** the FeatureCAM shortcut **icon** on the desktop: 

- Click on **Open an existing file** in the **New Part Document** wizard window then press **Next**.
- Select **Curve_Chaining.fm** then press the **Open** button.

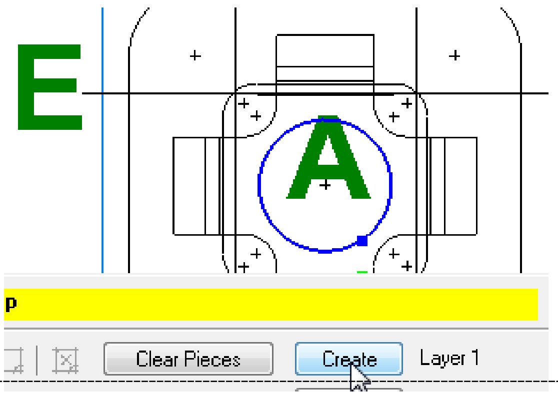
When the file is open it should look similar to the image below:





- From the **Geometry toolbar** select the **Chain Menu flyout** then click on **Closed Curve**

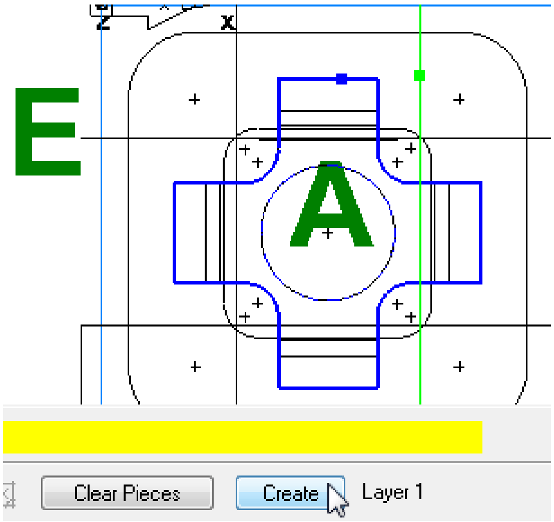
- **Click** once on the circle at "**A**" then press the **Create** button below the graphics window.



- Do the same for the circles located at **B**, **C** and **D**.

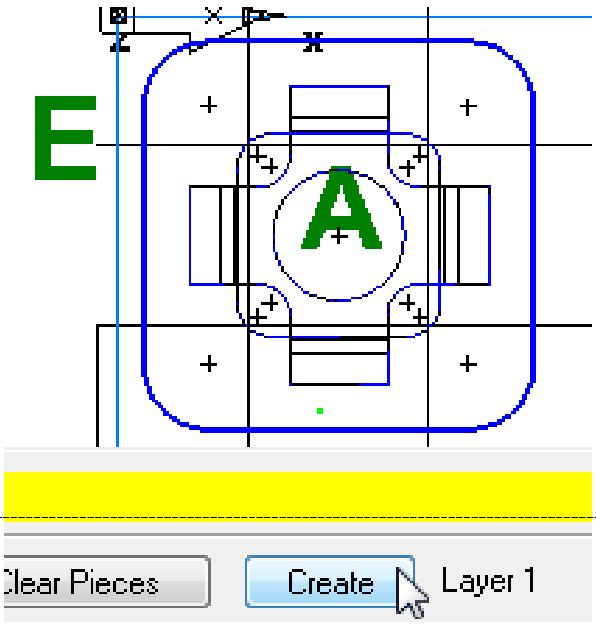
- From the **Part View** menu **click** on the **+** next to the word **Curves**. Four curves should be listed. These four curves are the curves that were created in the previous step.

- **Click** on the cross shape centered on "**A**" then press the **Create** button below the graphics window



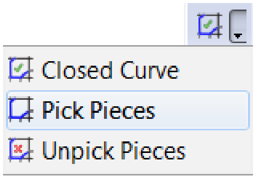
- Do the same for the cross shapes located at **B, C** and **D**

- **Click** on the large square with a corner radius centered around "**A**" then press the **Create** button below the graphics window

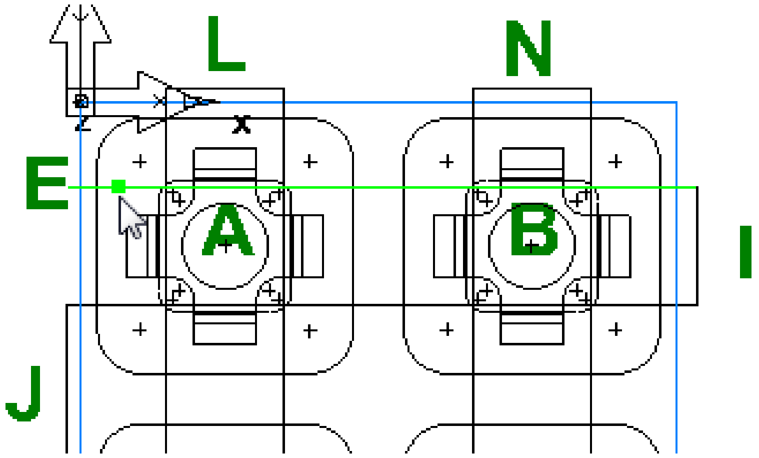


- Do the same for the large squares located at **B, C and D**.

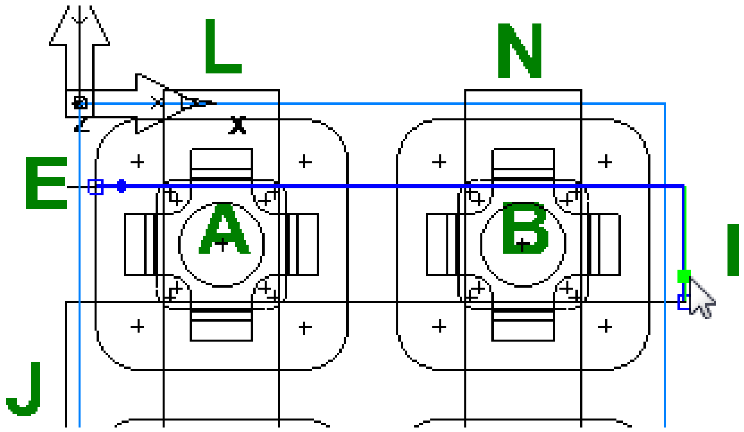
- From the **Geometry toolbar** select the **Chain Menu flyout** then click on **Pick Pieces**



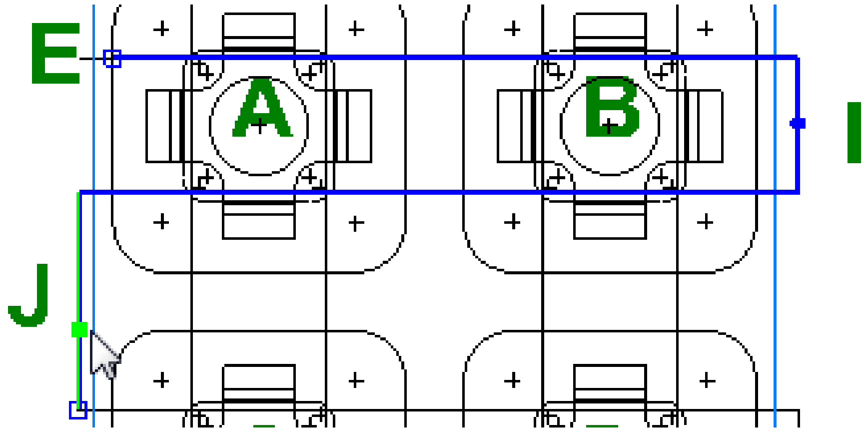
- Click once on the **horizontal line segment** located at "**E**"



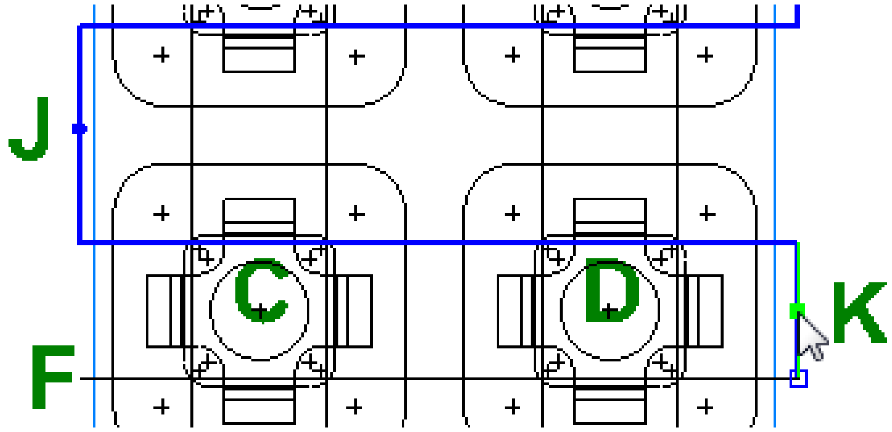
Then click once on the **vertical line segment** at "**I**"



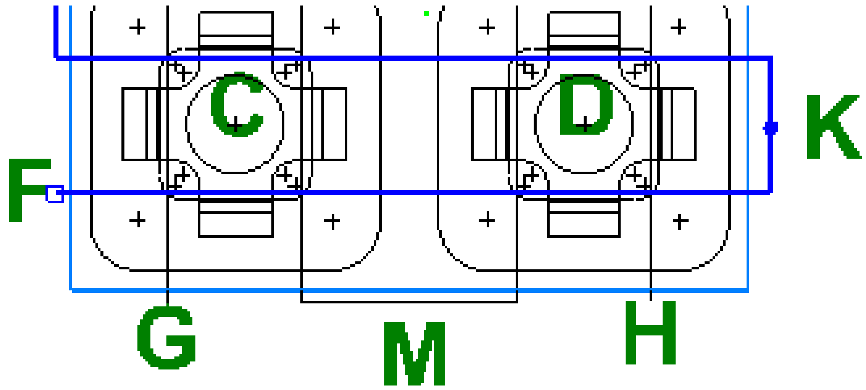
Then click once at the **vertical line segment** located next to "**J**"

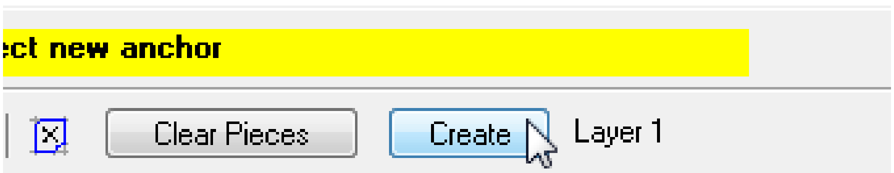


Then click on the **vertical line segment** located at “**K**”



Then click on the **horizontal line segment** located next to “**F**” then press the **Create** button below the graphics window.





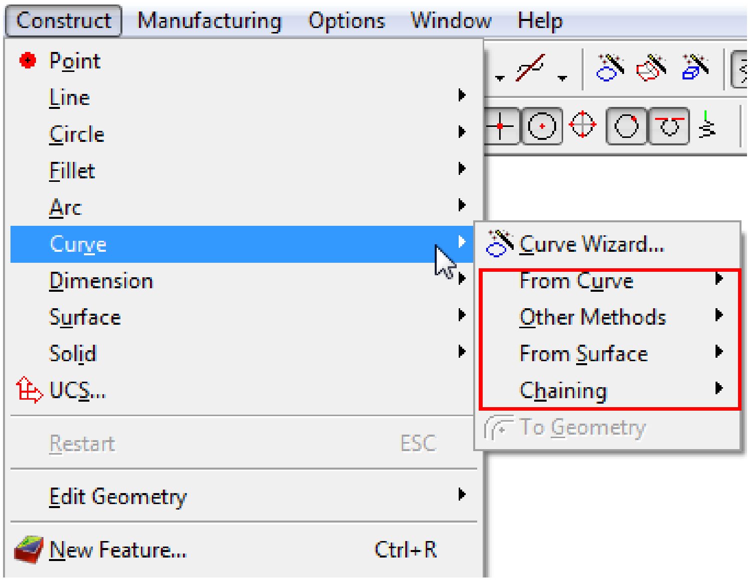
- Following steps similar to those shown above create a zigzag curve between “**G**” and “**H**”.

In this exercise curves have been made from Geometry. These curves can be useful for creating features, surfaces and solids.

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 followed by an exercise.

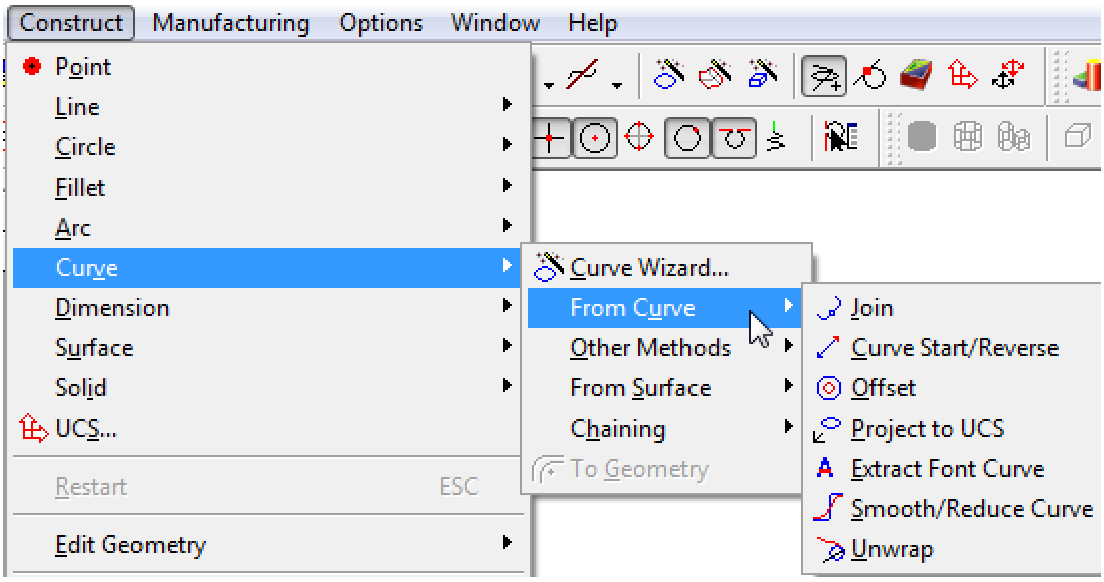
- From the file menu click on **Construct** then on **Curve**. Let's review these.



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

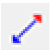
Curve from Curve

- 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.

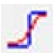


Curve Start/Reverse changes direction and starting point of the curve

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

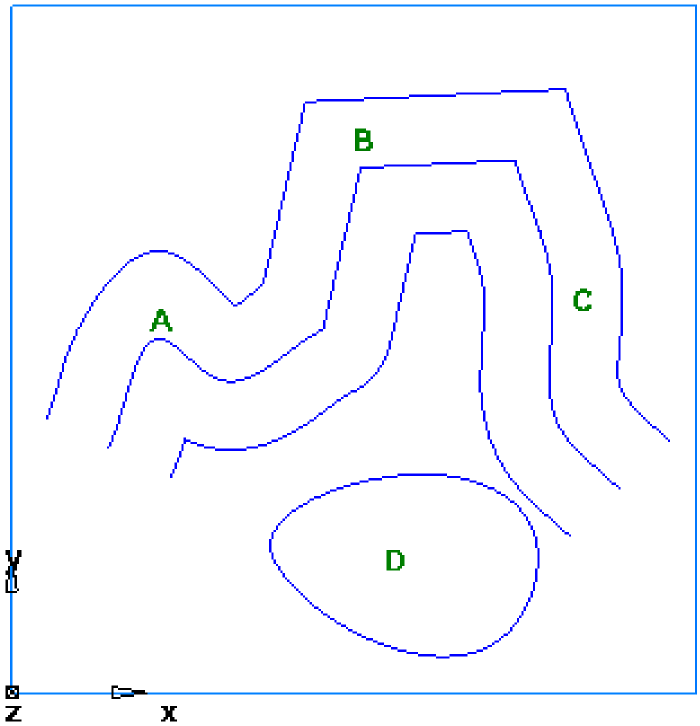
Extract Font Curve extracts curve segment from a font curve



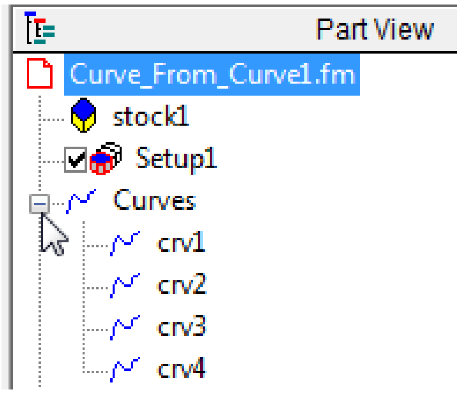
Smooth/Reduce Curve reduces the amount of data in a curve and smoothes out any bumps within a user specified tolerance.

Unwrap unwraps a curve that has been wrapped around a cylinder

Curve from Curve Exercise #1



- From the file menu select **File** then **Open**
- Select the file named **Curve_From_Curve1.fm** then press **Open**

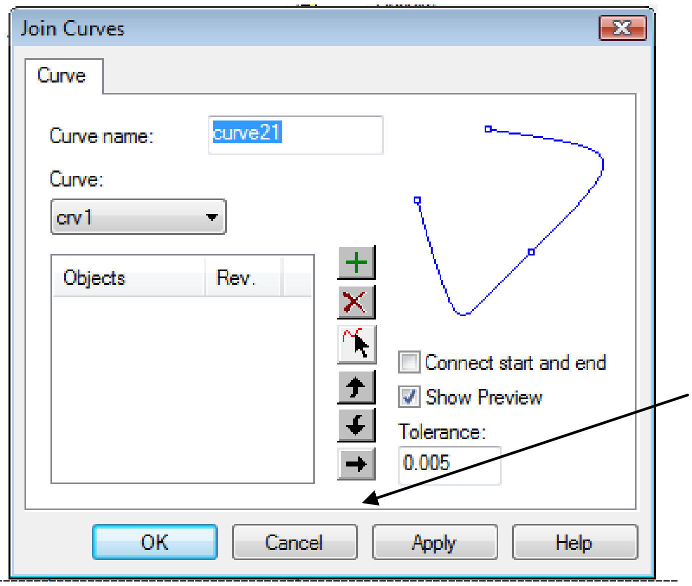


- **Click** the "**+**" in the part view next to curves

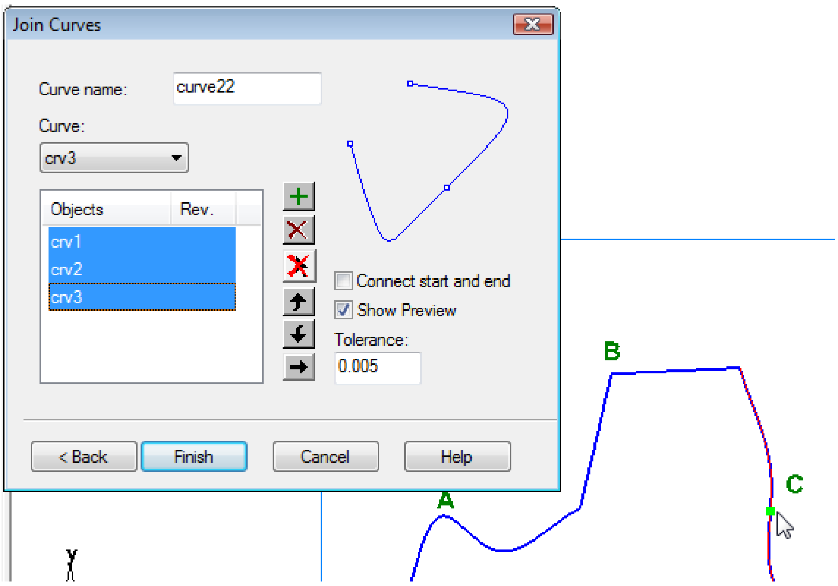
You will see that in this file there are four curves.

- From the file menu **click** on **Construct** then **Curve, From Curve** and **Join**.

A dialogue window for Join curve appears.

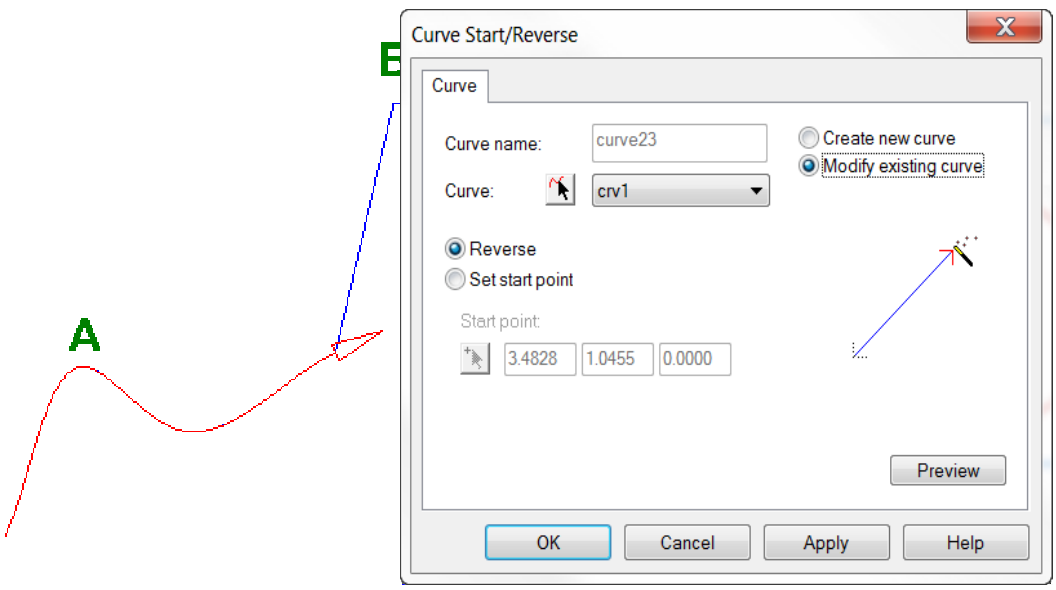


- **Click** on the **selection arrow** (black arrow with the red wavy line) **icon** . **Click** once on the **curve next to A**, **click** again on the **curve next to B** and **click** again on the **curve next to C** then press **OK/Finish**.



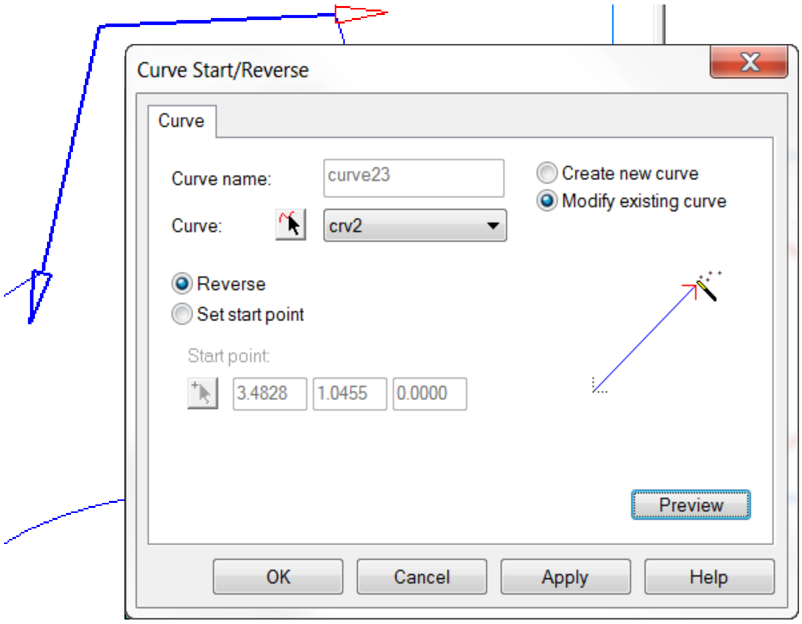
Observe the list of curves in the **Part View**. A new curve has been added from joining the curves.

- From the file menu **click** on **Construct** then **Curve, From Curve** and select **Curve Start/Reverse**.



By default the first curve should be highlighted. Notice there is an arrow at the end of the curve that shows the direction of this curve.

- From the pull down menu select **crv2.** Crv2 should highlight then press preview

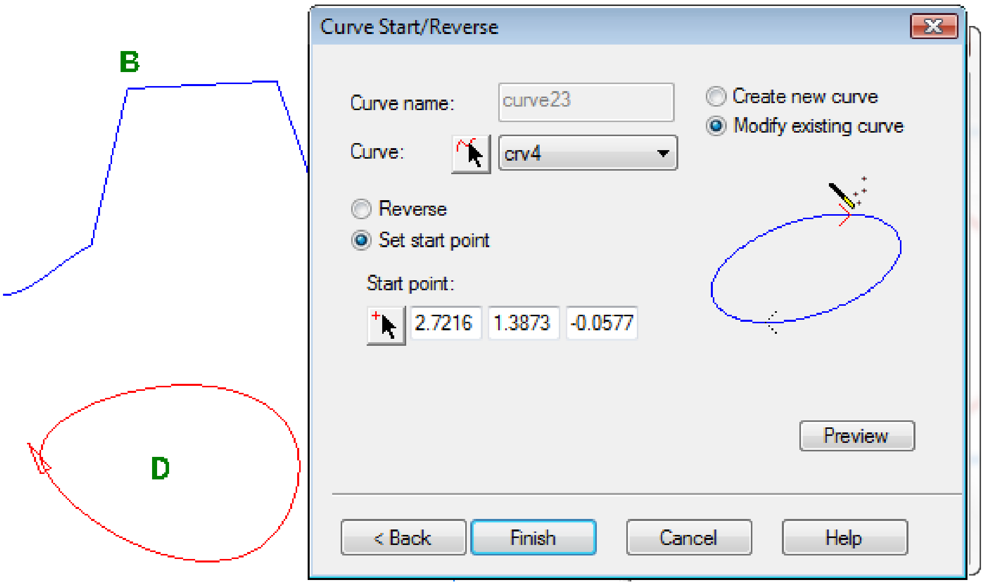


The blue arrow shows the new reversed direction of the curve.

- Press **OK/Finish**.

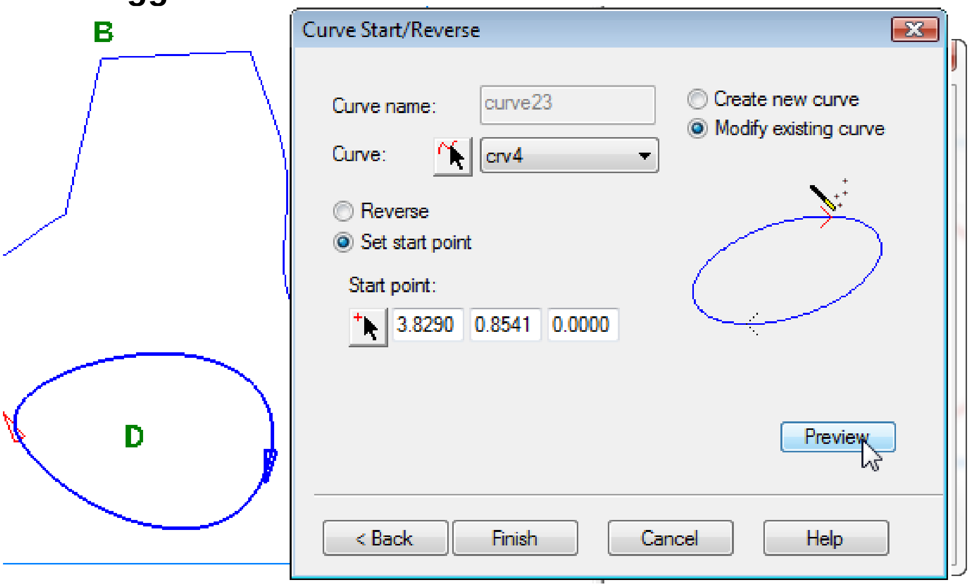
Reversing of curves is useful for cutting simple grooves for milling. Simple groove toolpaths plunge at the start of the curve and end at the end of the curve.

- From the file menu **click** on **Construct** then **Curve** then **From Curve** and select **Curve Start/Reverse**. From the pull-down menu select **crv4** then activate the radio button **Set start point**.



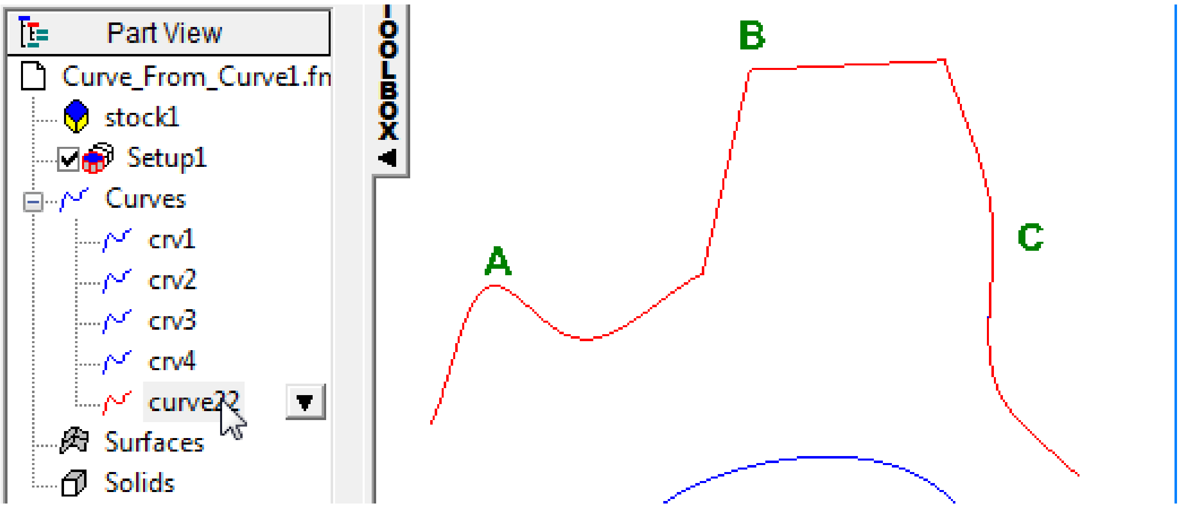
This will highlight the red egg-shape centered around point D. Notice that the direction of the curve is clockwise and the start point is on the left hand side of the egg. If we were to apply this curve to a simple groove feature the tool would plunge on the left hand side and continue clockwise around the profile.

- **Click** the **Pick new start point icon** , then **click** on the **right hand side of the egg** then **click** the **Preview button**.

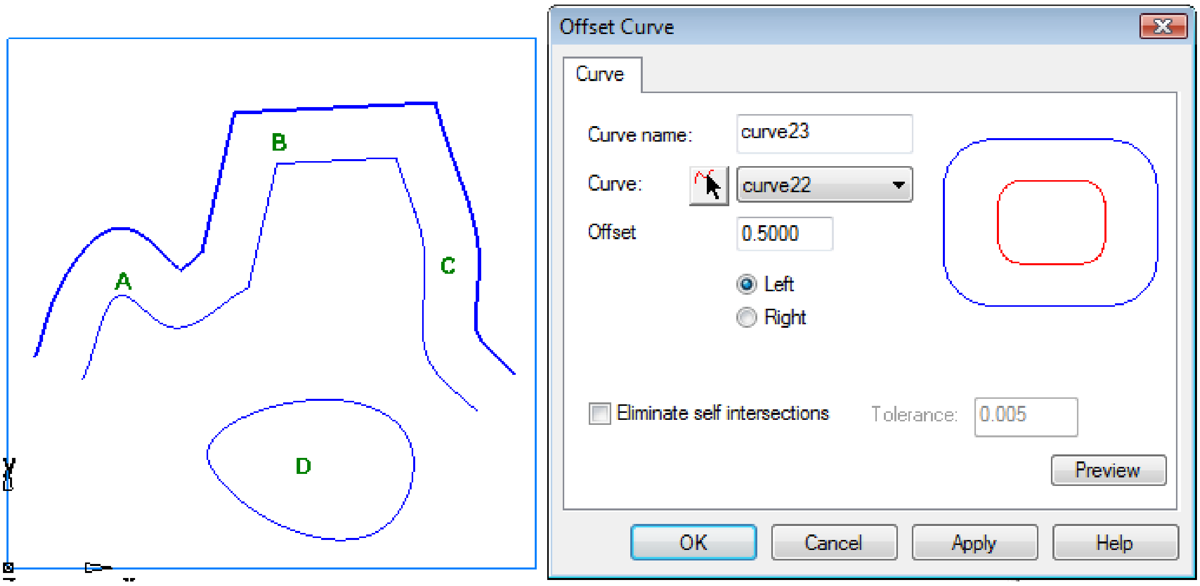


A profile of the new curve direction and start point preview is shown.

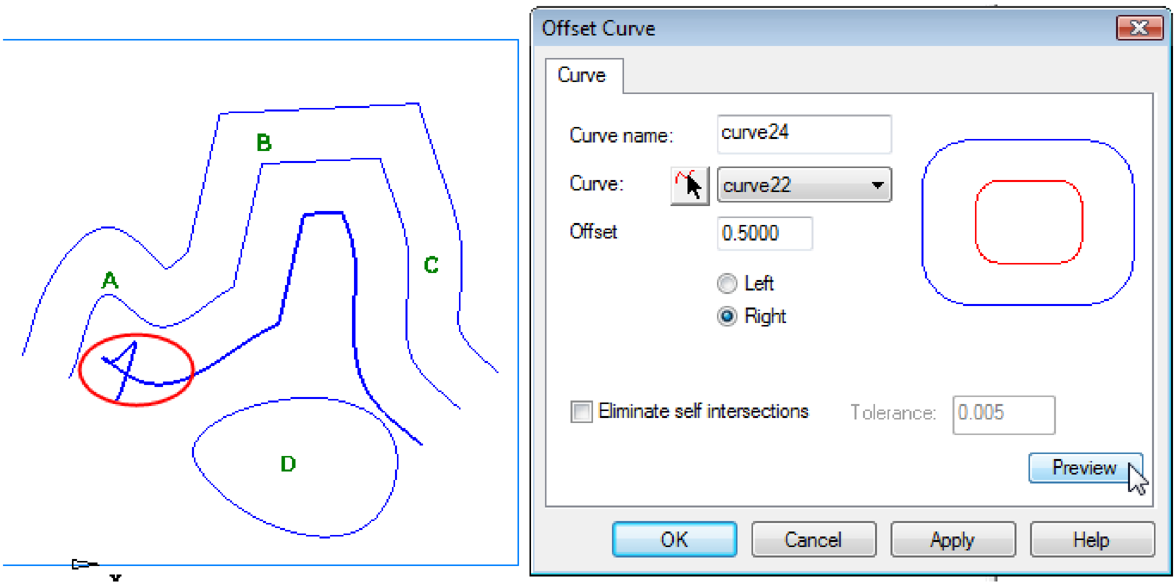
- Press **OK/Finish**.
- In the **Part View** locate and **click** on the curve that was joined from the 3 separate curves at the beginning of the exercise.



- From the file menu **click** on **Construct** then **Curve** then **From Curve** then **Offset**.
- Enter an **Offset value** of **0.5**, select **Left** then click **Preview** and **Apply**.

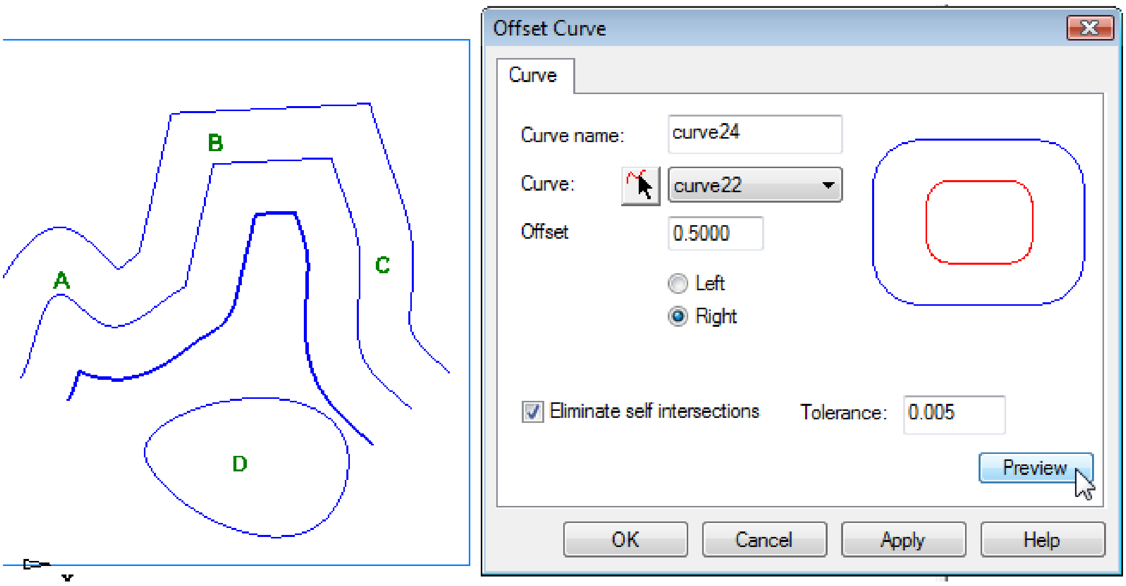


- Using the same curve select an **Offset** of **0.5**, select **Right** then **Preview**.



Observe the area circled in red. This is a self-intersection that could be bad for cutting as an offset profile.

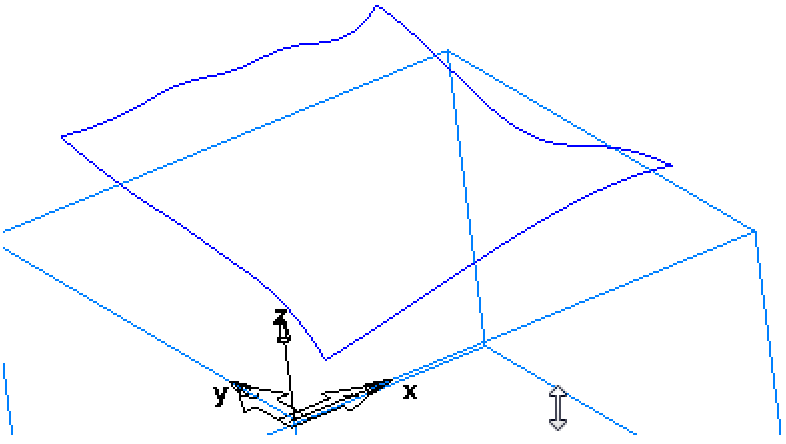
- Click on the checkbox that is labeled **Eliminate self-intersections** then press **Preview**.



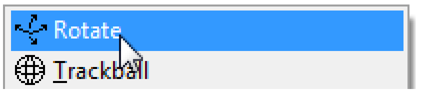
Notice that there is no longer a self intersection.

- Press **OK**

Curve from Curve Exercise #2

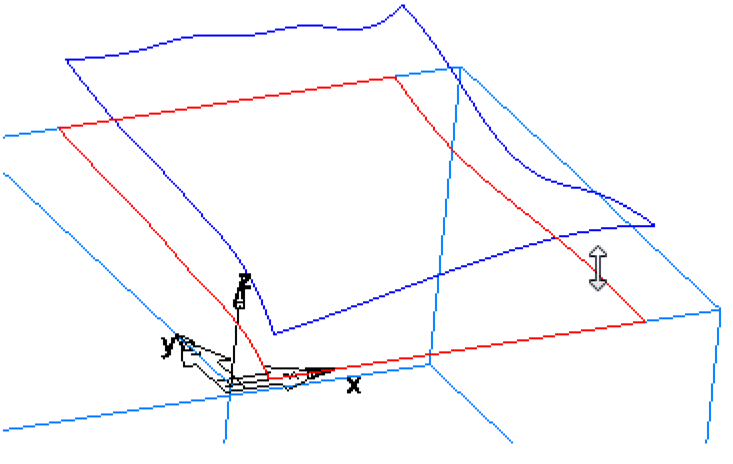


- From the file menu select **File** then **Open**
- Select the file named **Curve_From_Curve2.fm** then press **Open**
- **Right-click** in the viewing window and press rotate.
- Rotate the part to observe the curve.



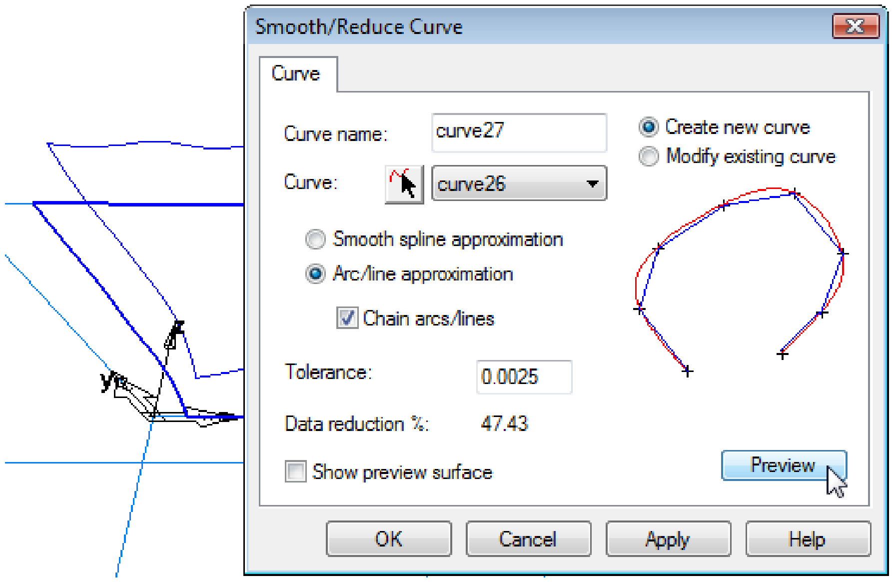
Notice that the curve is 3 dimensional. Most features from curves do not allow you to use a 3D curve for a feature. The only feature that will accept a curve that is not in the plane of the current setup UCS is a simple groove.

- From the file menu **click** on **Construct** then **Curve**, select **From Curve** then select **Project to UCS** and then press **OK**.



A curve is created by projecting the existing curve profile in Z to the current UCS. This curve can now be used to create 2.5D features from a curve such as pockets and bosses.

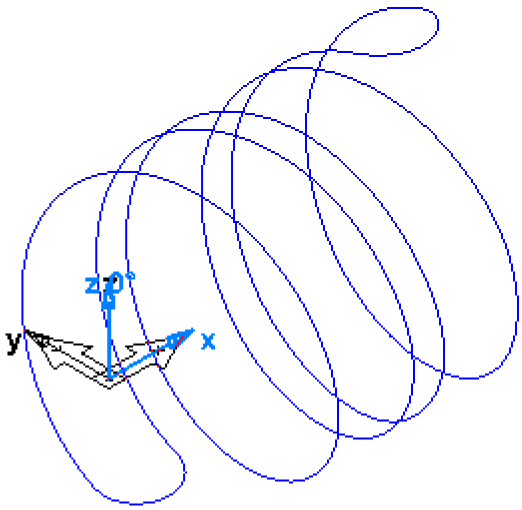
- Click on the curve that was just created then from the file menu **click** on **Construct** then **Curve** then **From Curve** then select **Smooth/Reduce Curve** then press **Preview**.



The Smooth/Reduce dialogue appears. When the preview button is pressed the Data reduction% field is populated. In this example the data reduction is about 50%. The new curve contains 50% less data but also has the ability to deviate from the original curve within tolerance.

- Press **OK** and the new curve is created

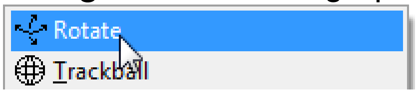
Curve from Curve Exercise #3



- From the file menu select **File** then **Open**.
- Select the file named **Curve_From_Curve3.fm** then press **Open**.
- From the file menu click on **View->Show->Show STOCK Axis**.

The part that has been opened is a 4^{th} axis part to be indexed about the stock x axis.

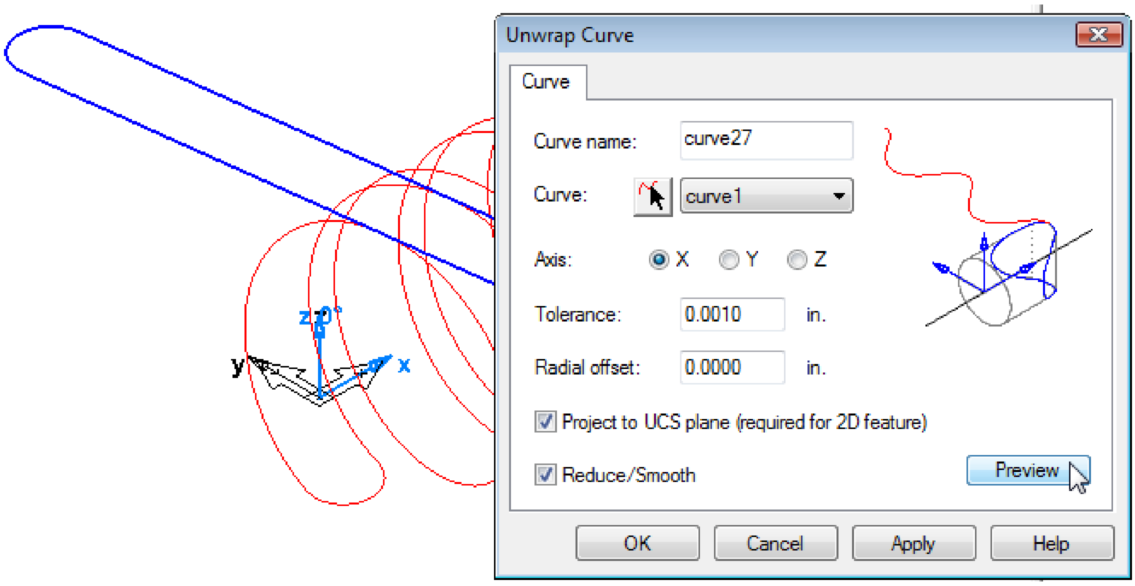
- **Right-click** in the graphics window and press **Rotate**.



- Rotate the part by moving the mouse to observe the curve

Notice that the curve is 3 dimensional as in the previous example. The only feature that would be able to use this curve would be a simple groove. So the curve needs to be unwrapped so it lies in a plane aligned with the current UCS.

- Click on the curve so that it is highlighted then **click** on **Construct** >**Curve->From curve**-> then select **Unwrap**.
- In the **Unwrap Curve** dialog press the **Preview** button.



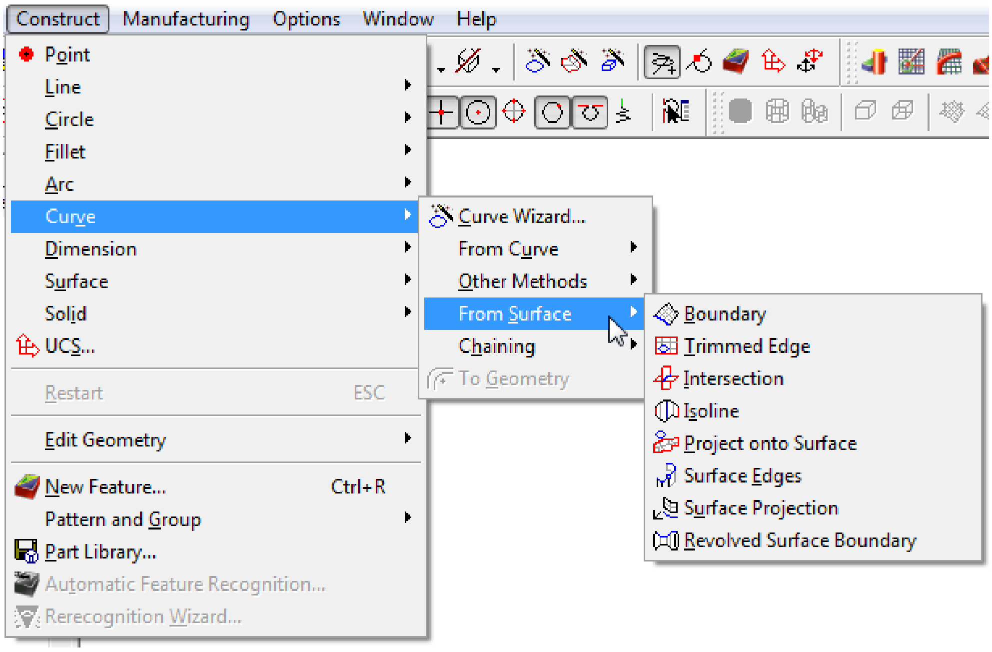
Notice that when the preview button is pressed the curve is unwrapped around the stock axis.

- Press **OK**

Now the curve can be used to create a 2.5D feature since it is aligned with the current UCS.

Curve 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 isolines

 **Project onto Surface** project existing curve onto a surface

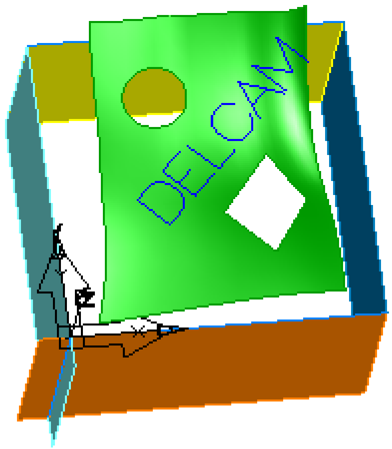
 **Surface Edges** extracts selected surface edges

 **Surface Projection** extracts curves from all vertical walls

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

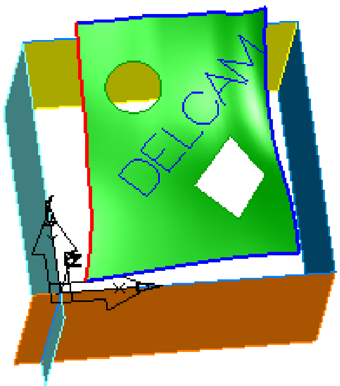
Curve From Surface Example #1

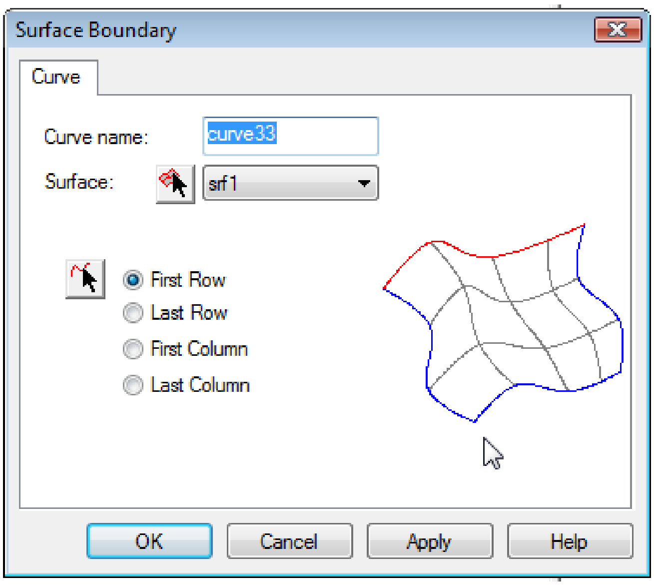
- From the file menu select **File** then **Open**



- Select the file named **Curve_From_Surface1.fm** then press **Open**
- From the file menu click on:

Construct->Curve->From Surface->Boundary





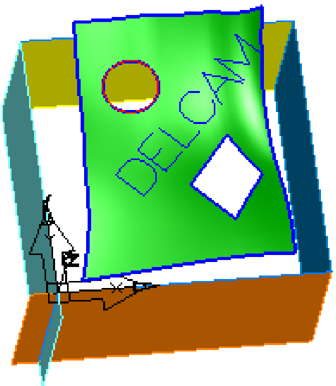
- Select **Srf1** from the pulldown menu select **First Row** then press **Apply**.

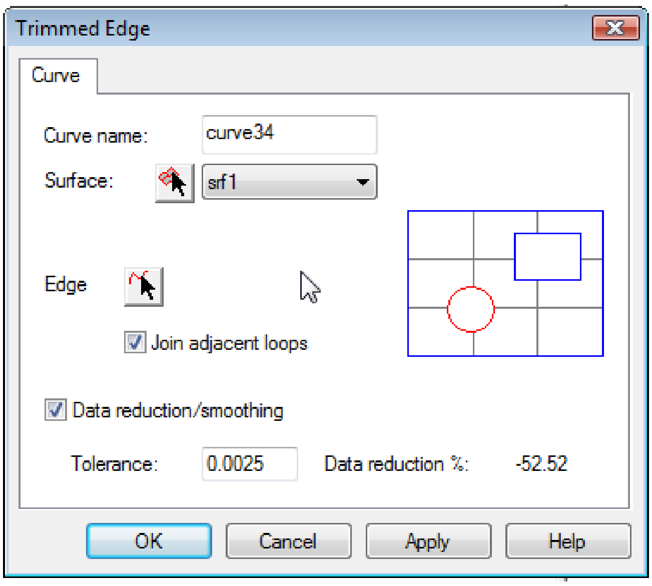
In the **Part View** window under Curves a new curve appears. This curve is created from the first row of srf1 boundary

- Press **OK**

- From the file menu click on:

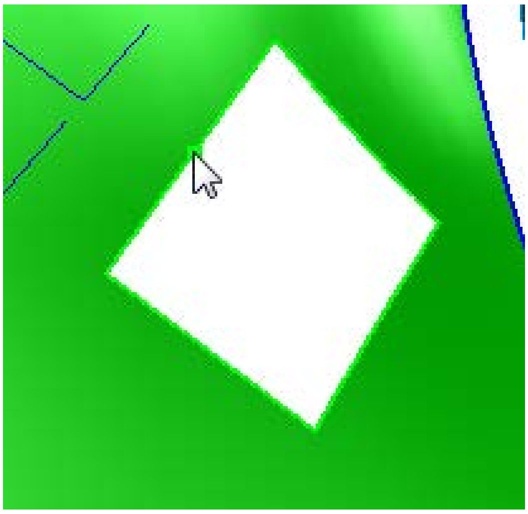
Construct->**Curve**->**From Surface**->**Trimmed Edge**.





The trimmed edge dialog appears that gives the ability to extract trimming curves used to trim the surface.

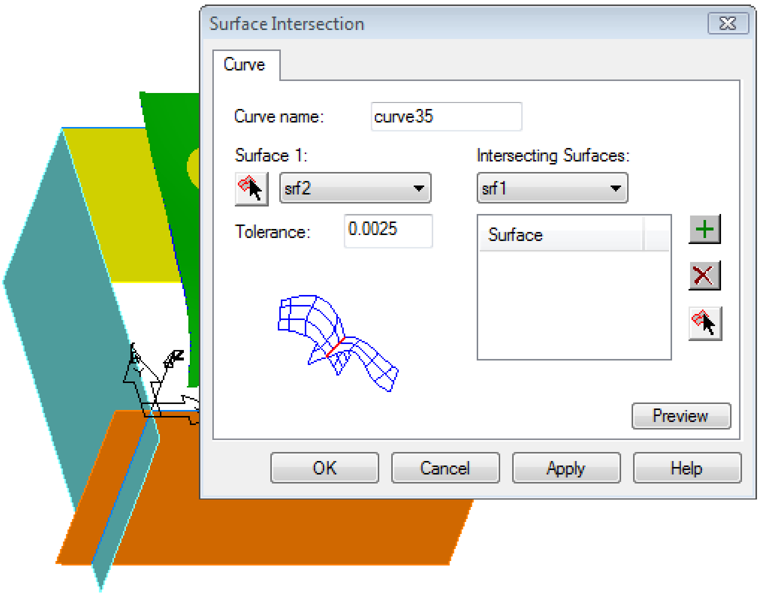
- Click the **Pick curve on surface icon** with the red wavy line 
- Click one of the edges of the green diamond on **srf1** then press **OK**.



A curve is extracted from the trimmed surface

- From the file menu click on:

Construct->**Curve**->**From Surface**->**Intersection**

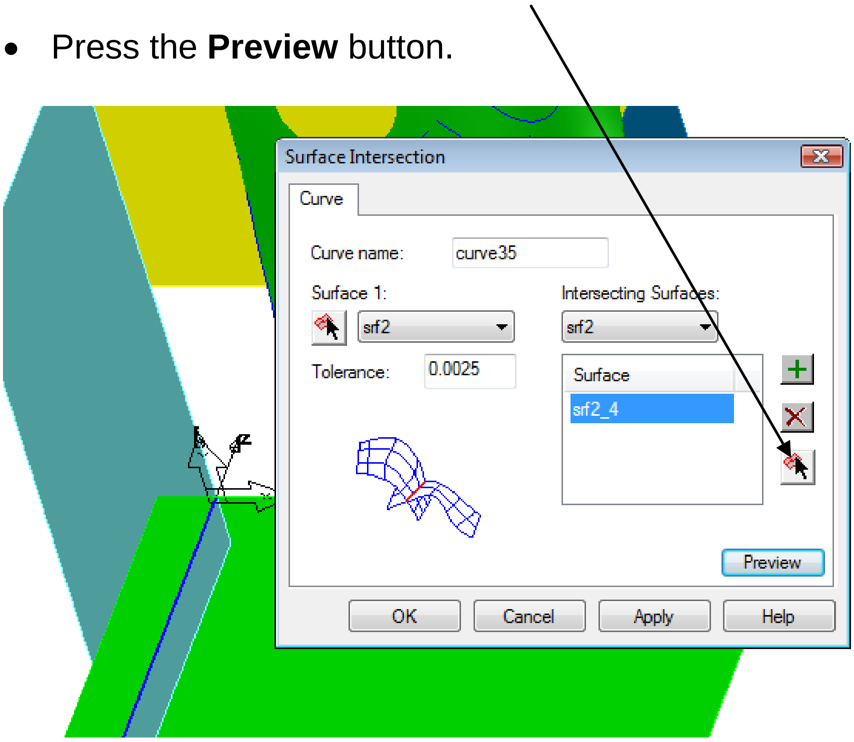


The surface intersection dialog appears that enables a curve to be constructed where the surfaces intersect.

- Locate the aqua blue and orange surfaces in the lower left hand corner.

For **Surface 1** select **srf2** (aqua blue surface) from the pull-down menu.
Click on the **Pick surface icon**  and click once on the **orange surface**.

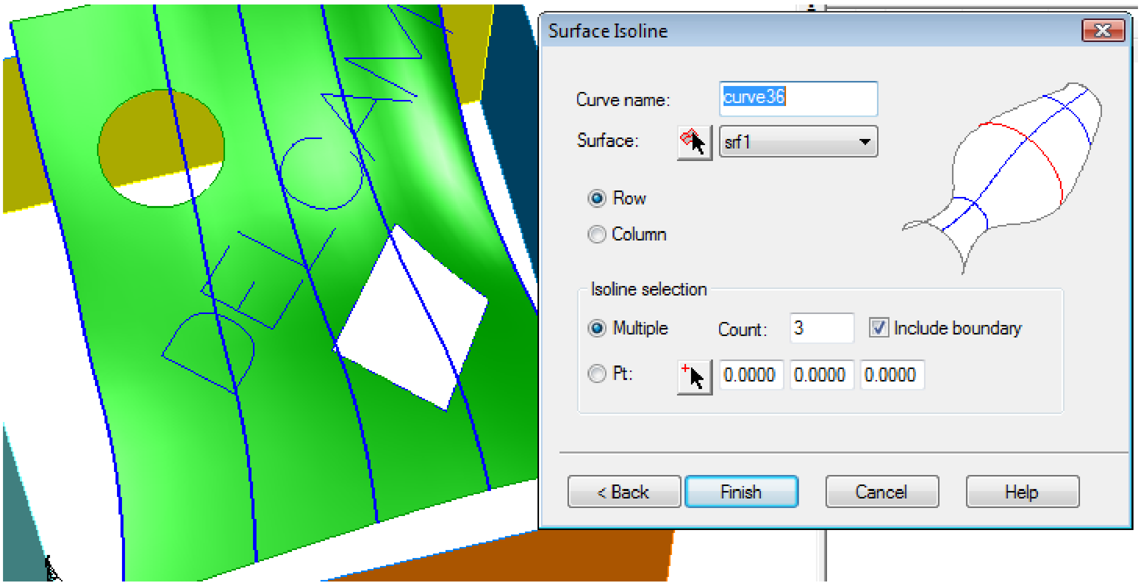
- Press the **Preview** button.



Observe that a curve is highlighted in blue in the lower left hand corner where the two surfaces intersect. This is the curve that will be created when OK or Apply are pressed.

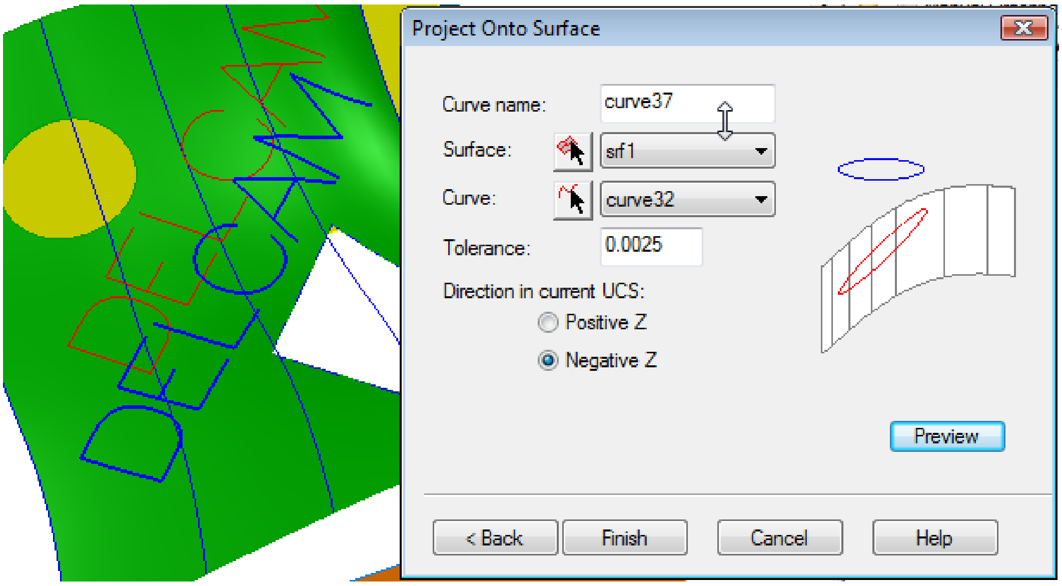
- Press **OK**

- From the file menu click on:
  Construct->**Curve**->**From Surface**->**Isoline**
- **Select srf1** as the surface and in the **count** field enter **3**



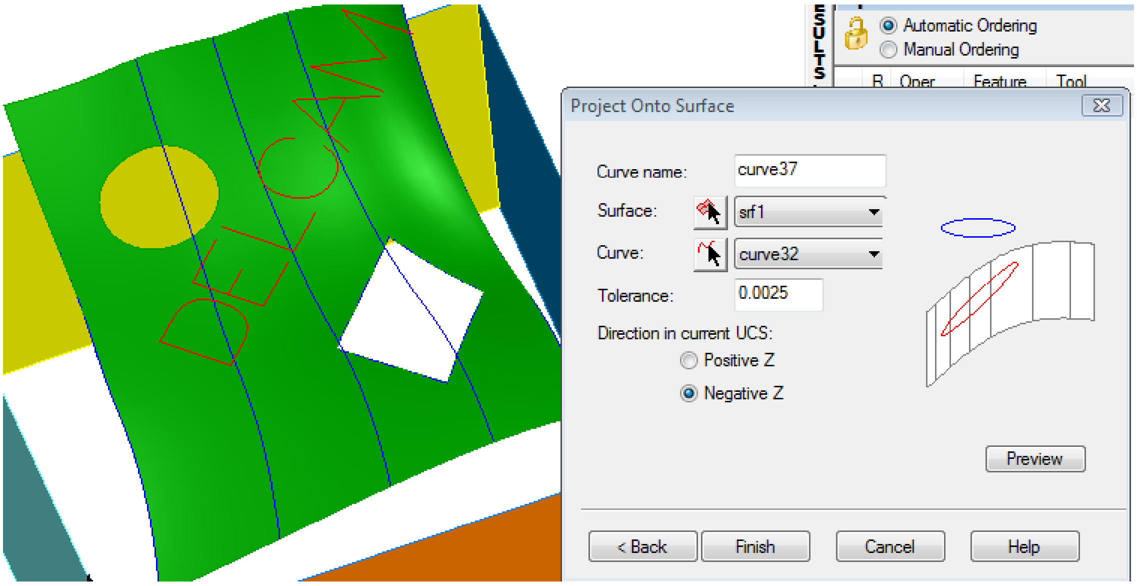
Notice the preview of the surface isolines running along surface rows in the current view.

- Then press **OK/Finish**.
- From the file menu click on:
  Construct->**Curve**->**From Surface**->**Project onto Surface**.
- In the **Project Onto Surface** dialog **select srf1** for the **Surface** and **select curve32** for the **Curve**.
- **Press** the **Preview** button.



Curve32 (DELCAM) is projected in the Negative Z direction. The resulting curve could be useful for engraving text on a surface using a simple groove.

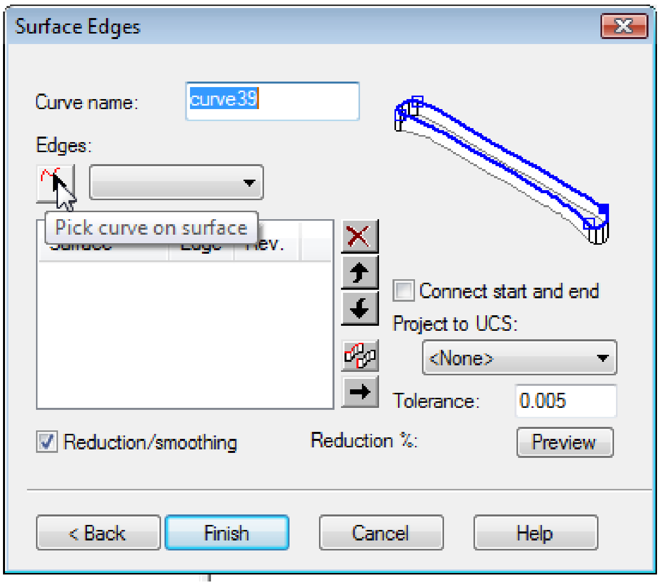
- Press **OK/Finish**.



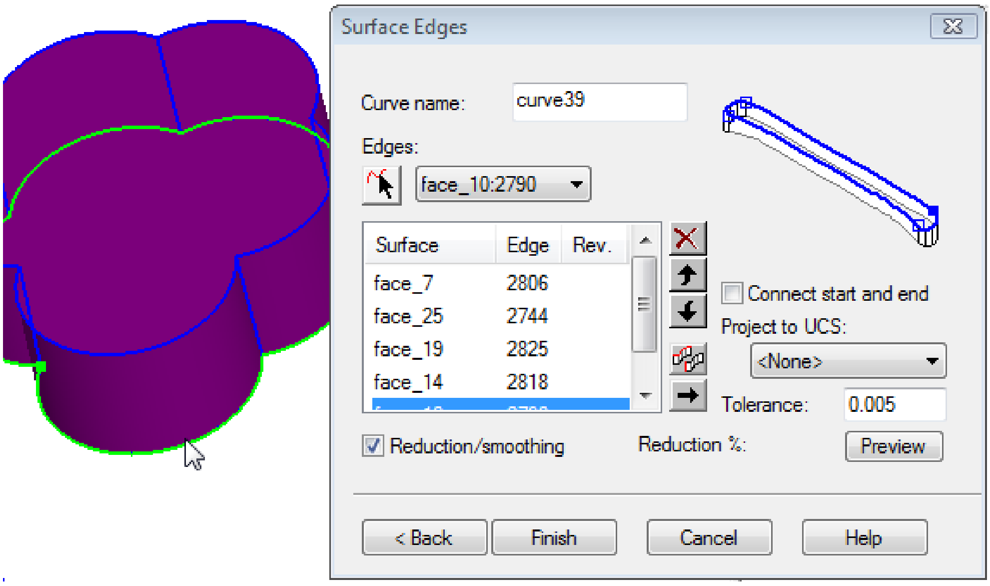
- From the file menu click on:

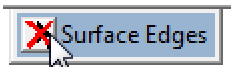
Construct->**Curve**->**From Surface**->**Surface Edges**.

- **Click** on the **Pick curve on surface icon**.



- **Pick** the **lower surface edges** on the purple shape in the upper right hand corner until all edges are selected. Be sure to pick the surface edges consecutively.



- When all edges are pick press the **red X** next to the minimized surface edges window 

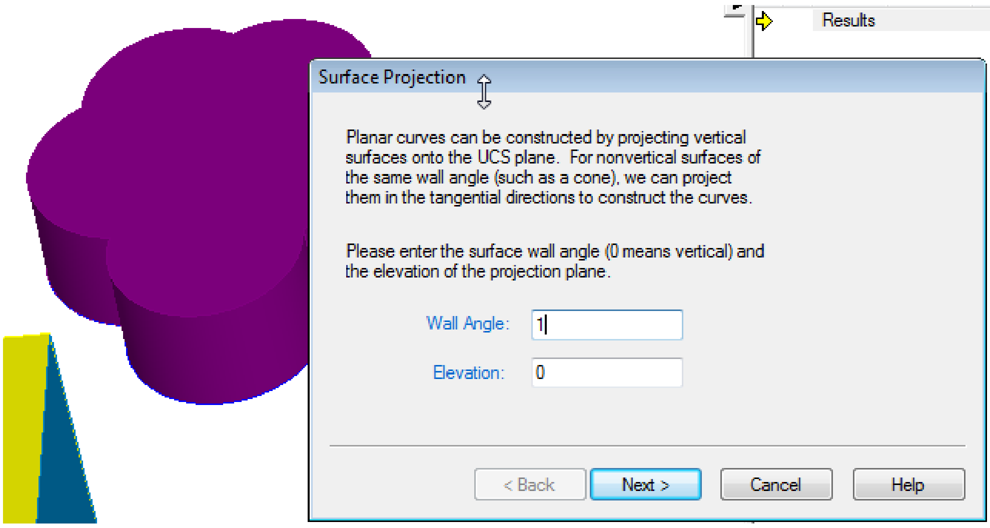
- Press **OK/Finish**.

The curve constructor from surfaces surface edges is a very useful tool for extracting curves from surfaces or from the faces of a solid.

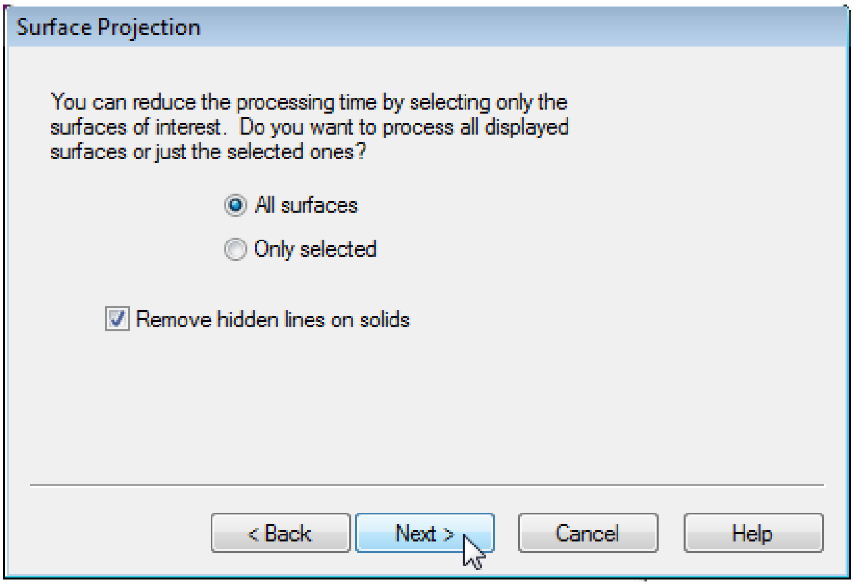
From the file menu click on **Construct->Curve->From Surface->** then **Surface Projection**.

The first dialog that appears gives you the ability to specify the draft angles and elevation relative to the UCS where the curve is to be created.

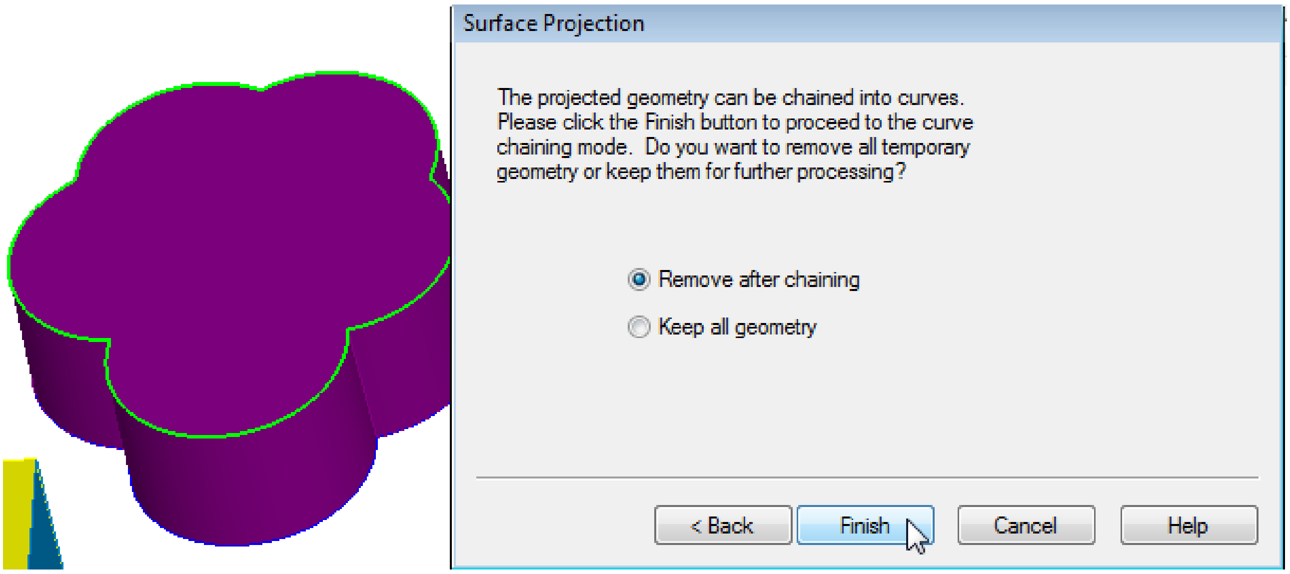
- In the **Surface Projection dialog** enter **1** for the **Wall Angle** and **0** for the **Elevation** then press **Next**.



- In the following window select **All surfaces** then press **Next**

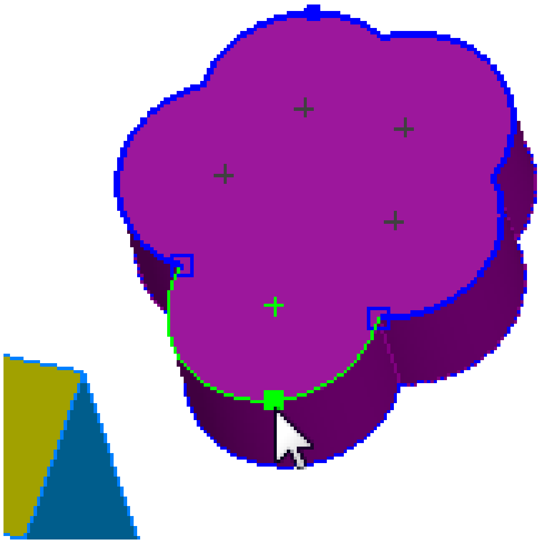


A preview of the geometry to chain in the viewing window appears highlighted in green.

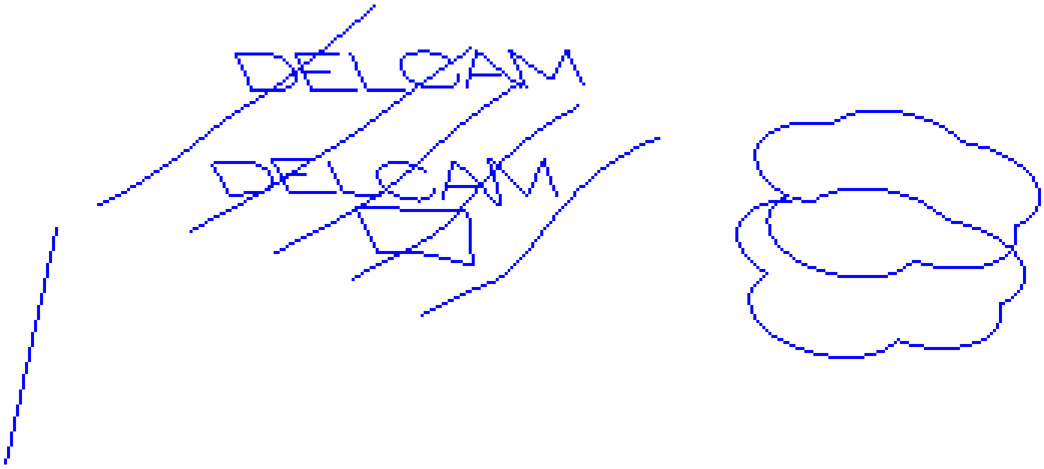


- Check the radio button **Remove after chaining** then press **Finish**.

- **Chain** the geometry into a **Closed Curve** then press **Create**.

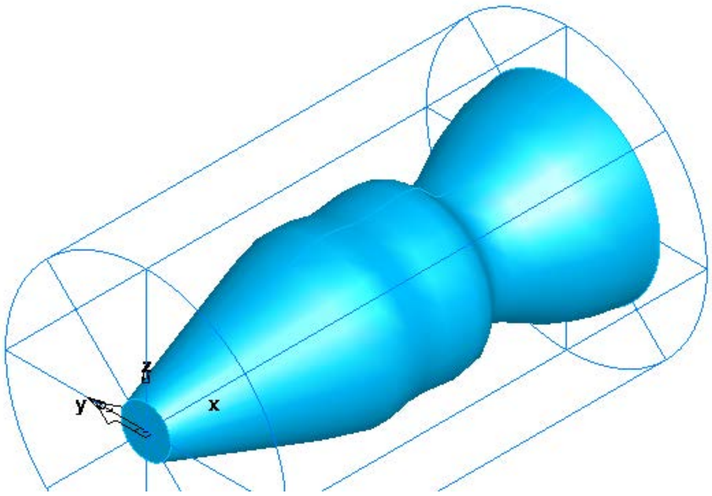


The curves created from this exercise should look similar to the results shown below:

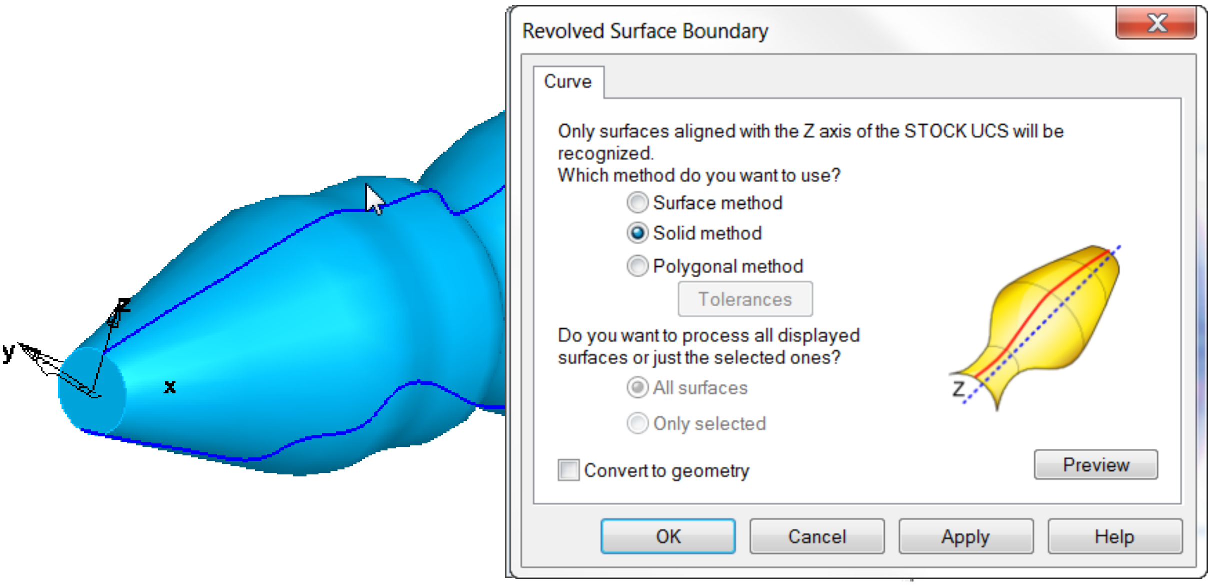


- To see the curves click on **View**->**Hide**->**Hide All**. Then click on **View**->**Show**->**Show All Curves**.

Curve From Surface Example #2



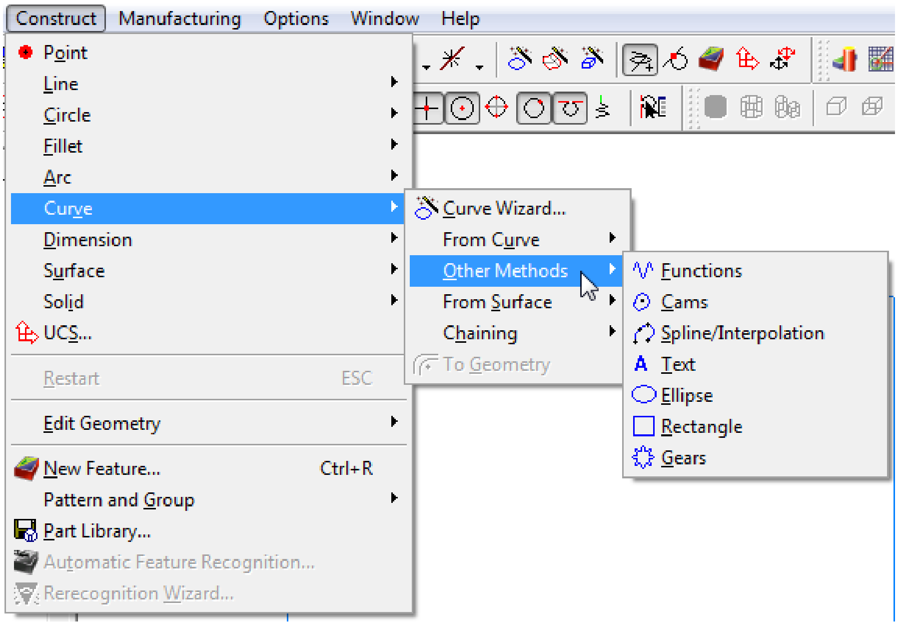
- From the file menu select **File** then **Open**
- Select the file named **Curve_From_Surface2.fm** then press **Open**
- From the file menu click on:
  Construct->**Curve**->**From Surface**->**Revolved Surface Boundary**.
- **Select** the radio button that is labeled **Solid method** then **Preview**.



- Click **OK/Finish**.
- **Hide all solids** to see the curve that has been created.
- **Close** the file **without saving**.

Curve From 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.

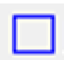
 **Function** creates user-defined mathematical relationships to generate a graphical figure.

 **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.

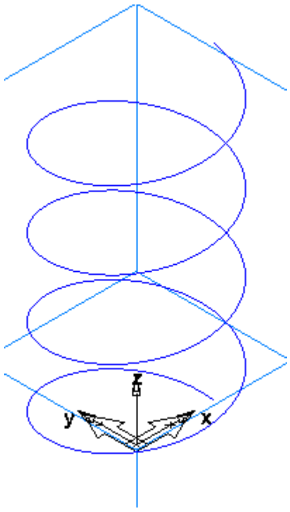
 **Ellipse** curve tool creates an elliptically-shaped curve in the plane of the current UCS.

 The **Rectangle** curve tool creates a rectangular shaped curve in the plane of current UCS.

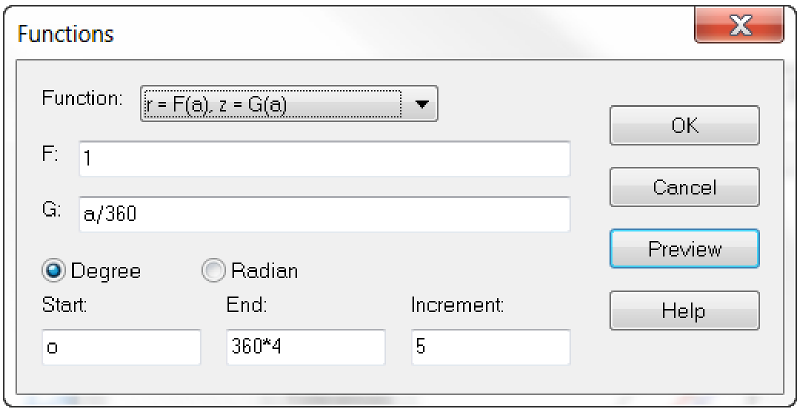
 The **Gear** curve tool creates a 2D gear profile.

Other methods Example #1 functions

The Functions method of creating a curve allows the user to define a 2D or 3D curve in terms of a mathematical function. In this example we will create a helix.



- From the file menu click on **New.**
- Select a **milling** setup with **inches** then press **OK.**
- From the file menu click **Construct**>**Curve**>**Other Methods**>**Functions.**
- From the **Function pulldown** select **r=F(a), z = G(a)**



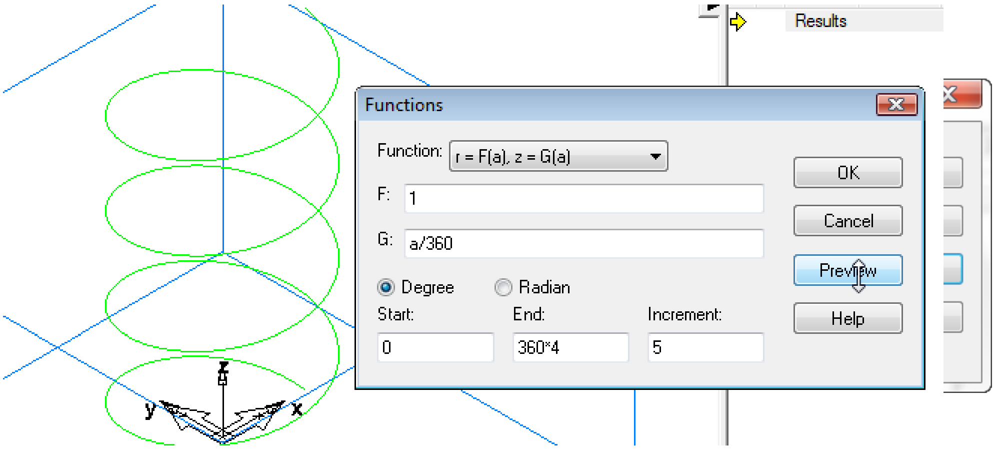
In this example a Helix will be constructed. The helix will have 4 revolutions with a constant radius of 1 and a pitch of 1. F is the function for the radius of the curve and G is the Function for the pitch. The letter “a” represents angle.

- Change the parameter **F** to **1** since there is a constant radius of 1”
- In the **G** field enter **a/360**. This means that every 360 degrees the helix will rise 1 inch. If you would like to rise a half inch every 360 degrees change the parameter to (a/360)*.5
- Select **Degree.**
- Enter **0** for **Start.**

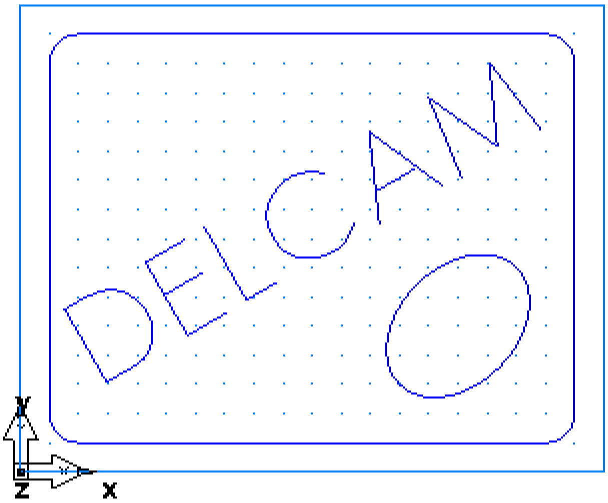
- In the **End** field enter **360*4**. This means the helix will go around 4 times.
- Change the **Increment** to **5**. This is how course or rough the helix will be.

Changing the value to a large value will make the helix faceted.

- Press **Preview** and press **OK**.



Other methods example #2

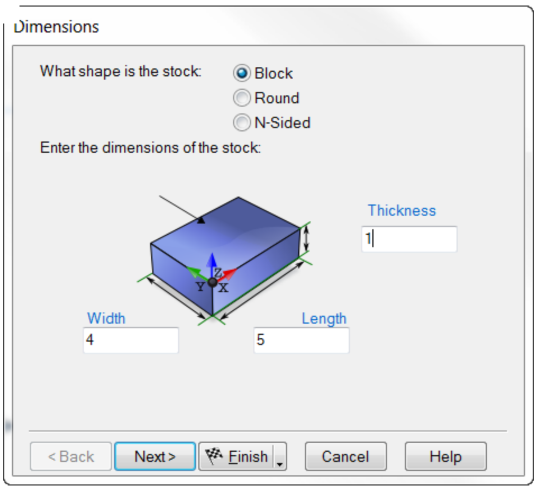


- From the file menu click on **new**
- Select a **milling** setup with **inches**

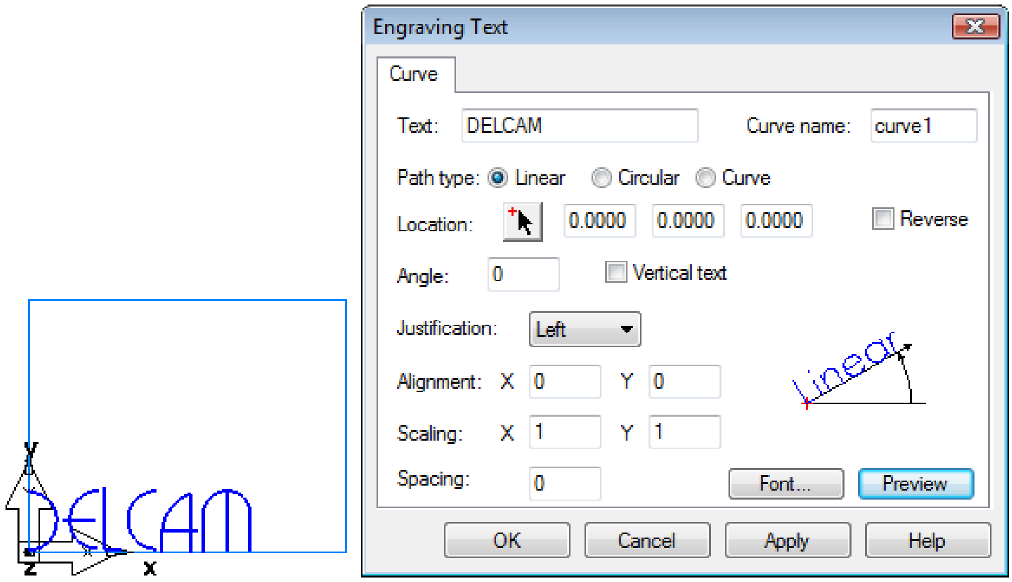
then press **Finish**.

- Enter the stock dimensions as **4" x 5"**

x 1" then press **Finish**.

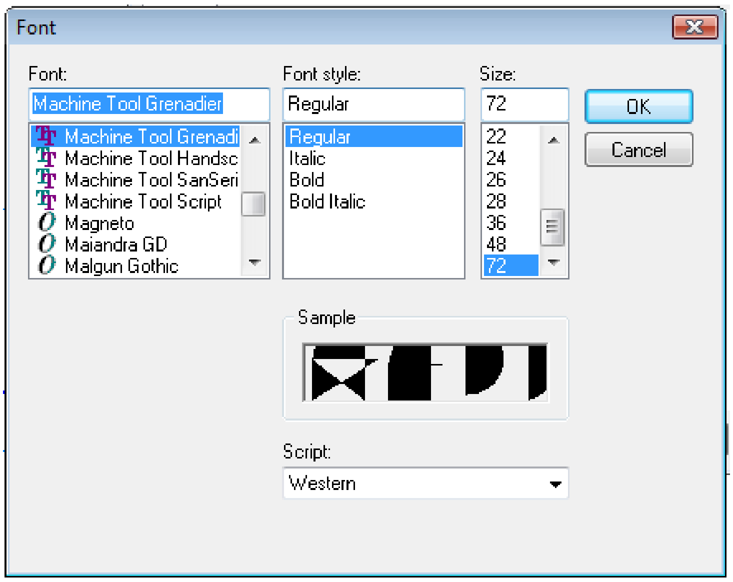


- From the file menu click on **Construct->Curve**->**Other Methods**>**Text**
- In the **Text** field type in **DELCAM** then press the **Preview** button.



A preview of the text Delcam is shown with the default machining font.

- Click on the **Font** button

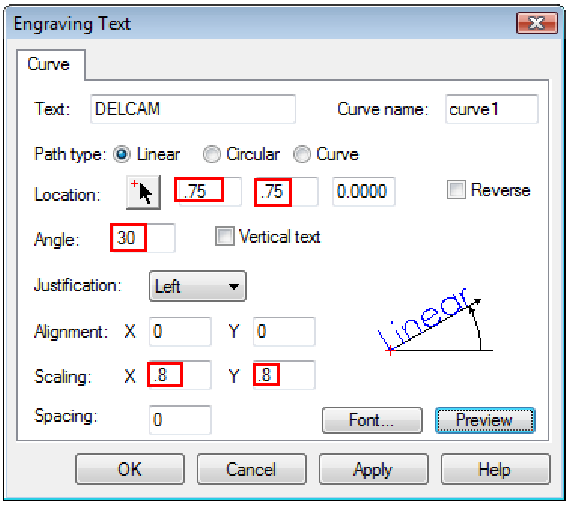


The available fonts are shown in the left hand window. All windows fonts are listed in this window as well as 5 machining fonts installed with FeatureCAM. All of these fonts start with “Machine Tool...”. These fonts are unique because they are single line engraving fonts that are useful for engraving.

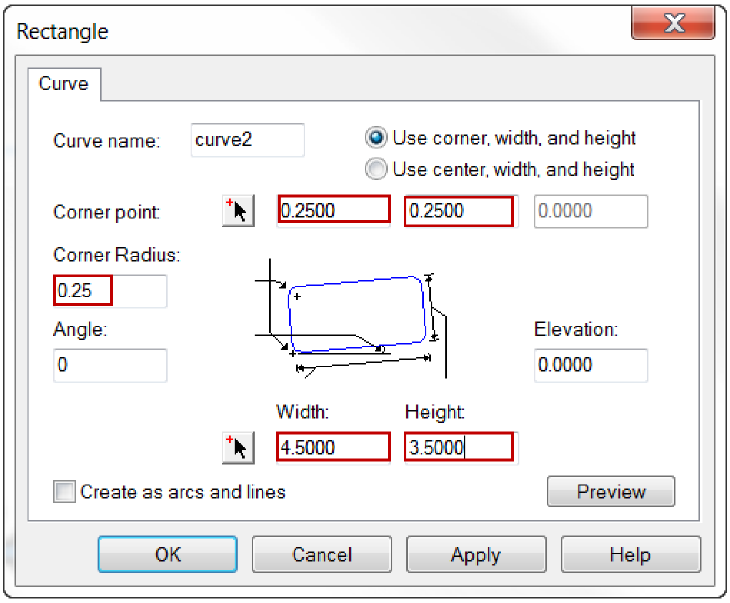
- From the available Font list select **Machine Tool Gothic** and press **OK**

***Note: it is suggested to not change the size of the font in the Font window as undesirable results could occur. The value for all the machining fonts should be set to 72. If changes are needed to the size of the text, use the scaling option.**

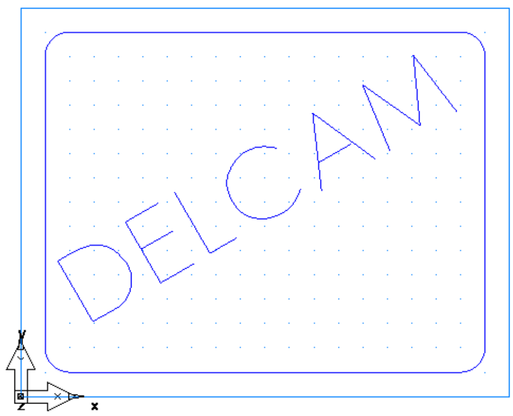
- In the Engraving text dialog change the **Location** to **(0.75, 0.75, 0)**. Change the **Angle** to **30** and change the **Scaling** to **0.8** then press **OK**.



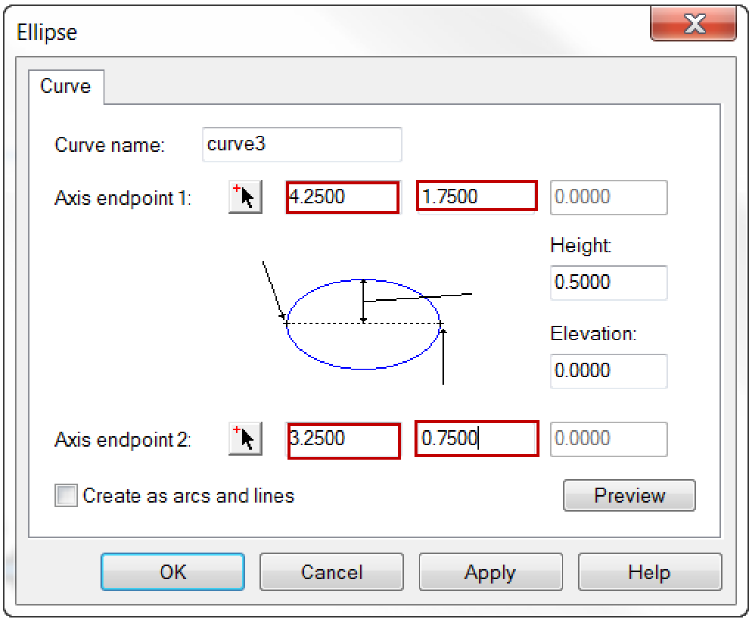
- From the file menu click on: **Construct**->**Curve**->**Other Methods**>**Rectangle.**
- In the **Rectangle** dimension dialog enter the values highlighted and press **OK/Finish**.

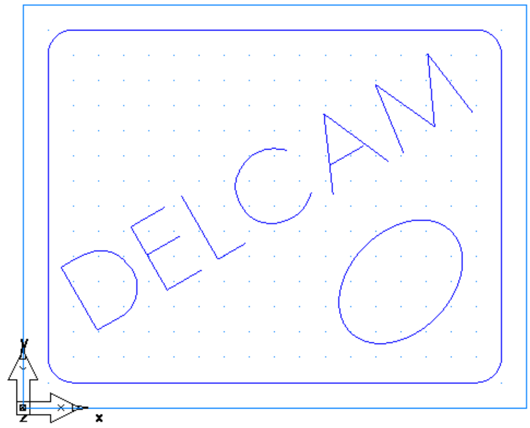


Results now should look similar to the image on the right:



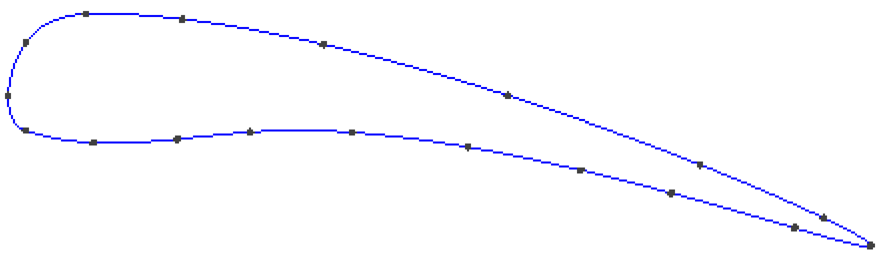
- From the file menu click on **Construct**->**Curve**->**Other Methods**>**Ellipse**
- In the **Ellipse** dialog **change the values of the fields highlighted** below then press **OK/Finish**.



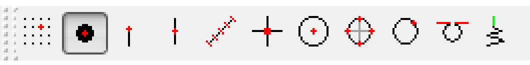


The resulting curves can be used for engraving by creating a simple groove.

Other methods example #3

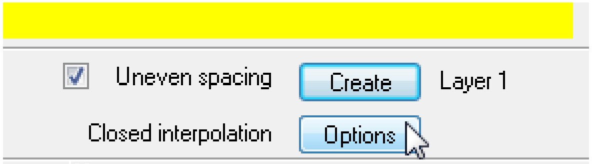


- From the file menu select **File** then **Open**
- Select the file named **Curve_From_Other_Methods3.fm** then press **Open**
- Enable **Snap to Point** from **Snap Mode** toolbar

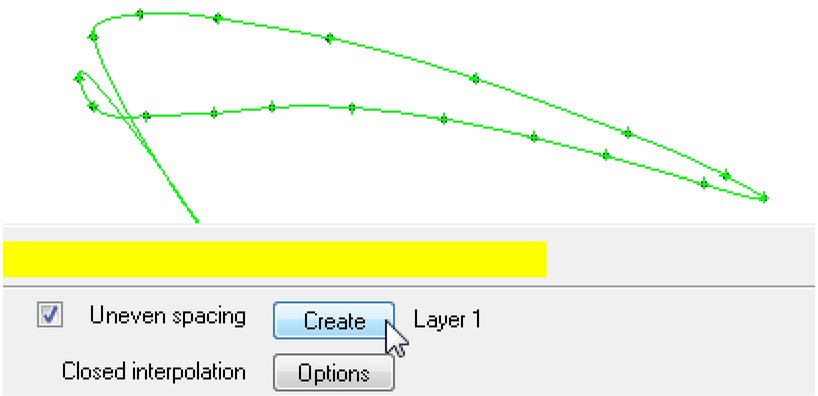


- From the file menu click on:
  Construct->**Curve**->**Other Methods**>**Spline/Interpolation**

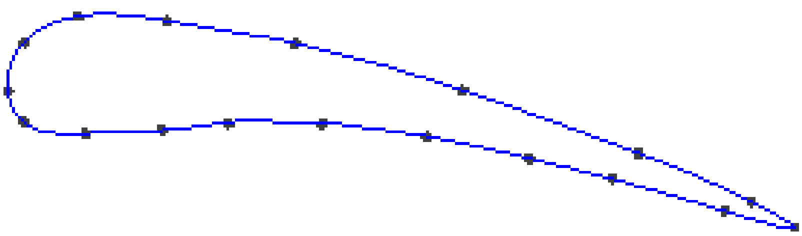
- Below the assistance bar press the **Options** button so that **Closed interpolation** is activated.



- Click on each point sequentially then press the **create** button below the assistance bar.



When finished the resulting curve should look similar to the curve below.



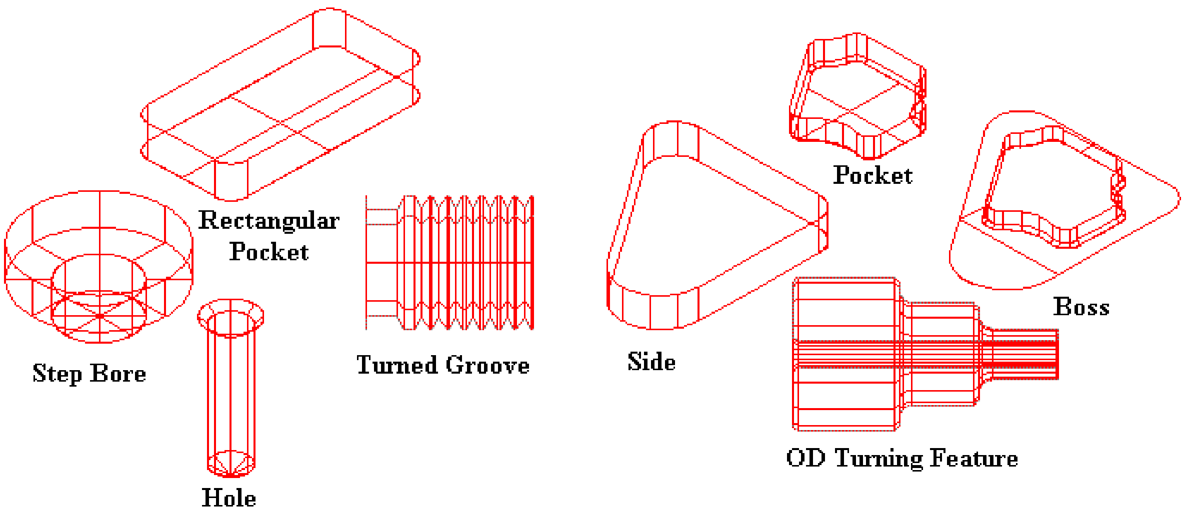
Basic 2.5D Features

Introduction

This Module will provide an understanding of what a feature is, the feature types available and how to create various types of 2.5D features.

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 make these shapes. Features may consist of several operations to create the final result or shape.



Features may consist several operations including: spot drill, center drill, drill, pre-drill, tap, roughing, semi-finish and finish passes.

A 2.5D Feature is a Feature that in a single plane, primarily the XY plane. A more complex Feature that will be introduced in later modules is a 3D Feature that cuts in X, Y and Z axis simultaneously.

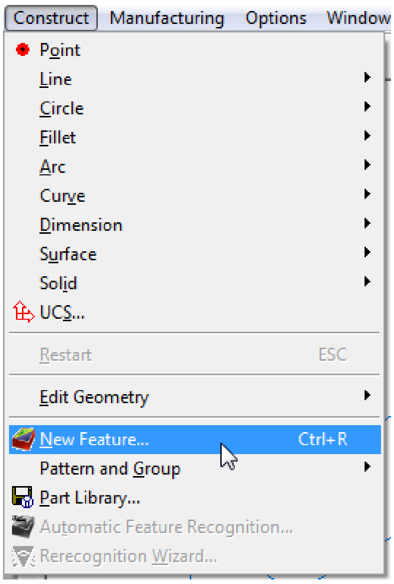
User Interface

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.

- Accessed through the **Advanced Toolbar**.

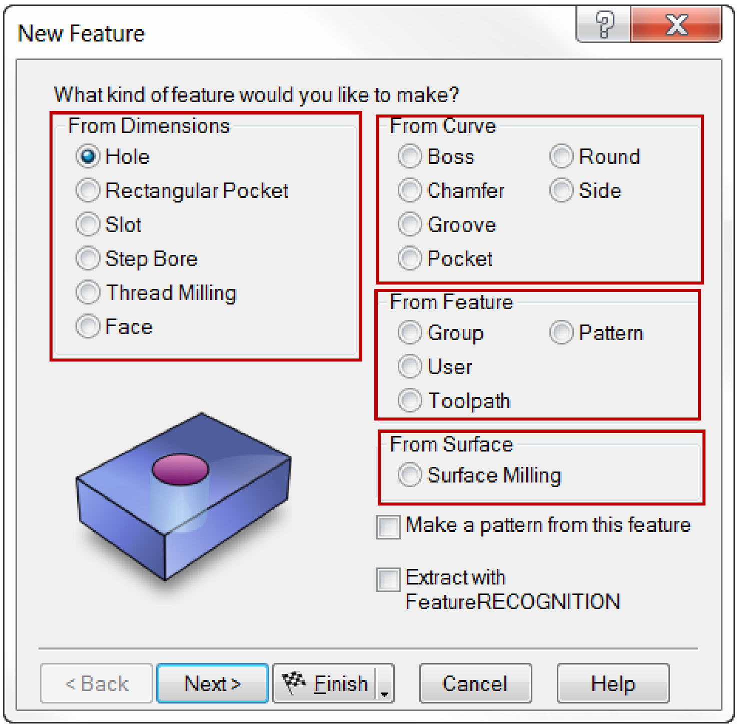
- Accessed through the **Steps Toolbar**.

- Accessed through the **File Menu**.



- Or the New Feature Wizard may be accessed through the **Steps Menu**.

The new Feature Wizard is broken into four sections:



From Dimensions: Feature From Dimensions specific dimensions for the hole, rectangular pocket, face, etc... may be entered. No curve is required to create the feature boundary.

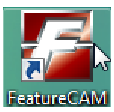
From Curve: Feature From Curve defines the feature boundaries by using curves.

From Feature: Feature from Feature allows the user to create a group of features, a pattern of features, a feature from existing toolpath or a user defined feature.

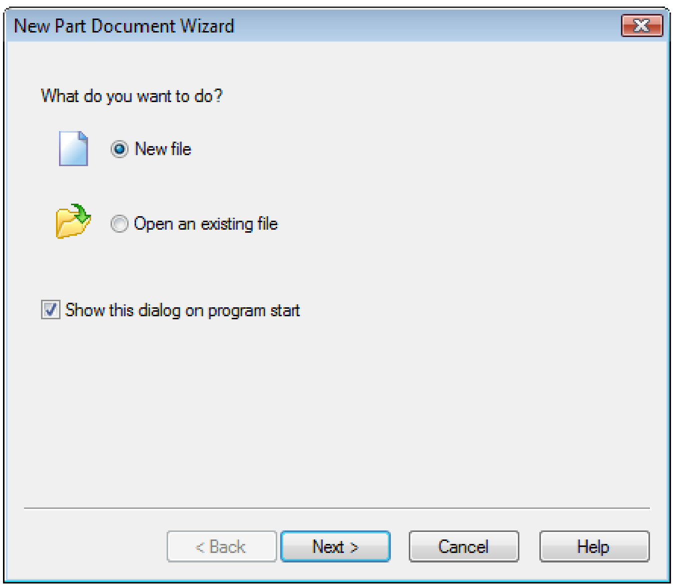
From Surface: Feature from Surface is used to create surface milling features by either selecting part surfaces or faces.

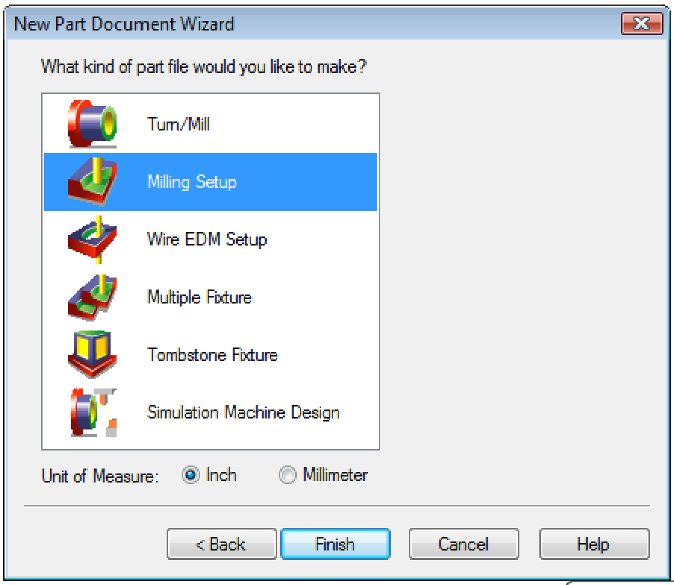
Feature Representation

- From the desktop double-click on the FeatureCAM shortcut icon.

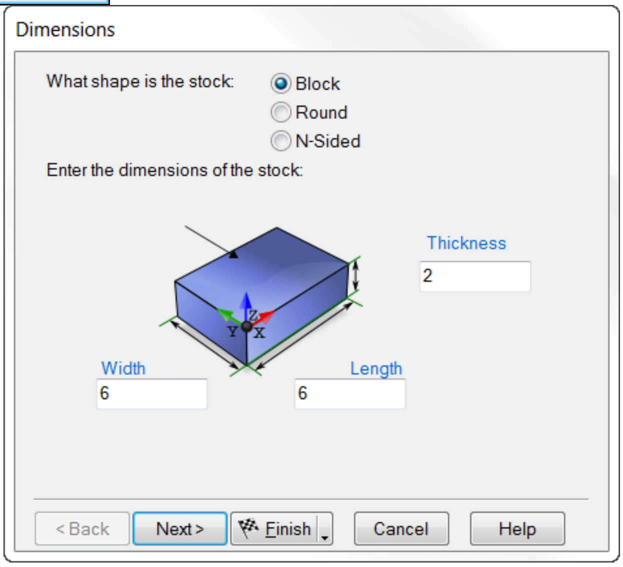


- From the **New Part Document Wizard** select New File then press **Next**.

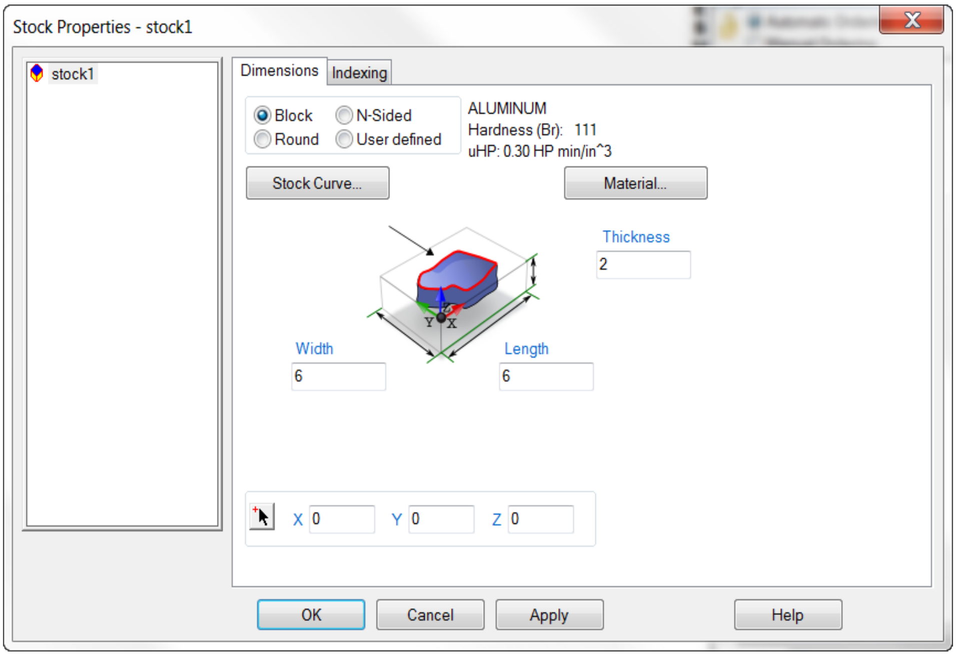




- Select **Milling** then **inches** and press **Finish**.

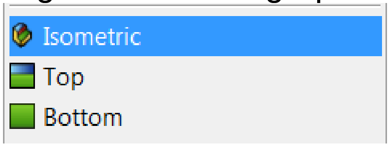


- On the **Dimensions** page press **Finish**.

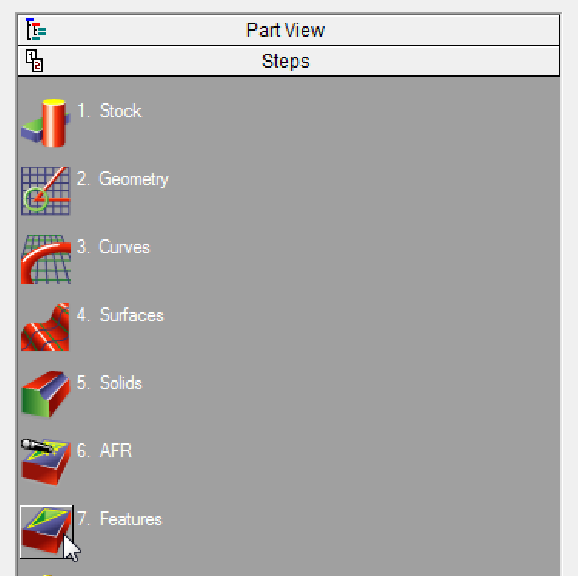


- On the **Stock Properties** page press **OK**.

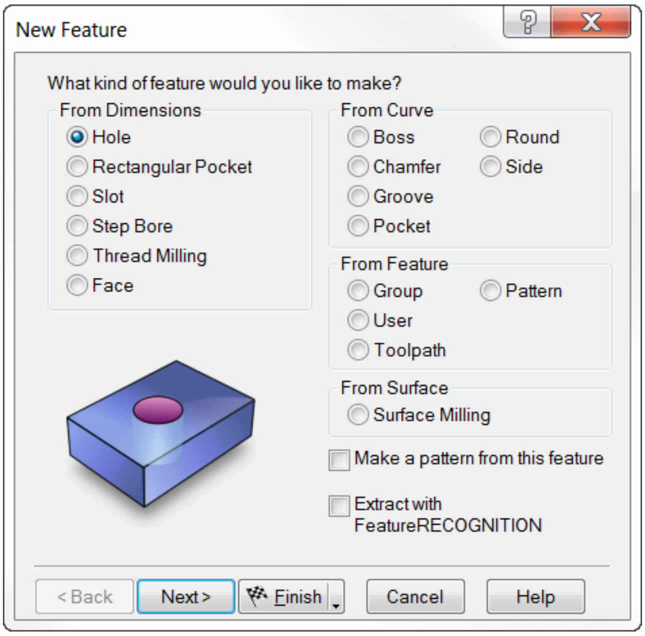
- Right click in the graphics window and click on **Isometric**.



- From the **Steps Toolbox** click on **Features**.

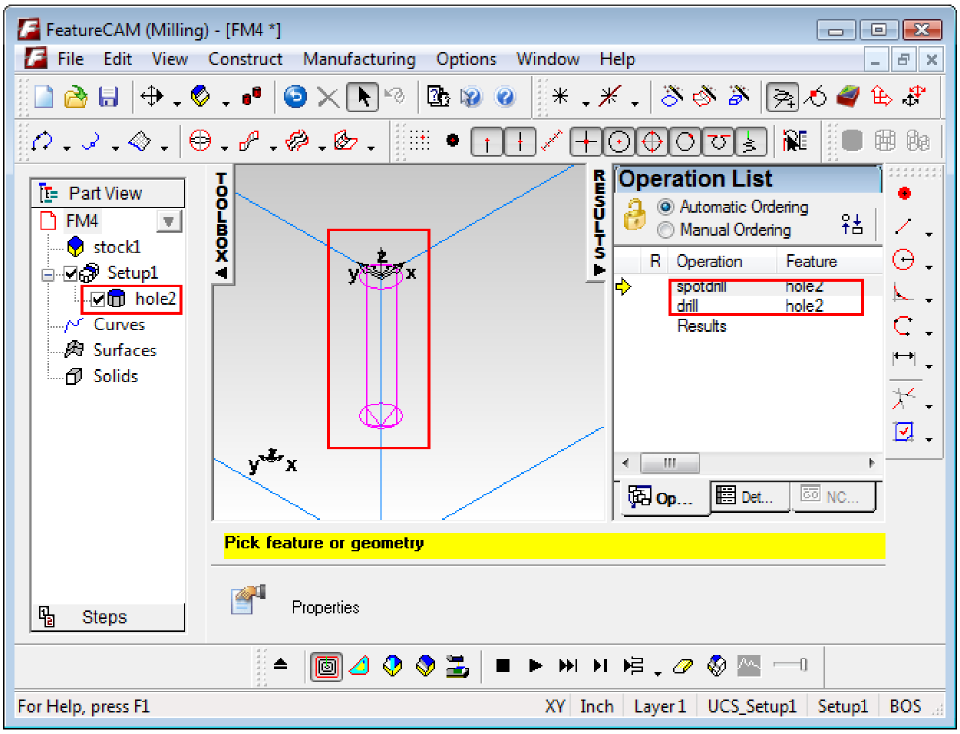


- Select **Hole** and press **Finish**.



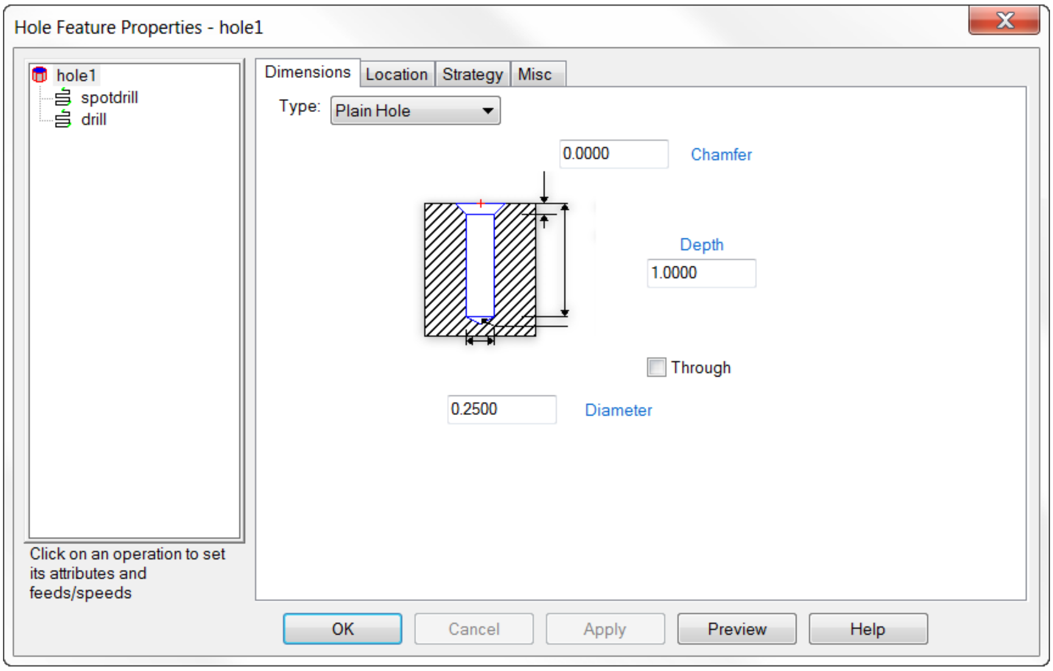
- In the **Hole Feature Properties** window press **OK**.

- In the **Toolbox Window** click on **Part View**.



Notice that the hole feature created is represented in three different sections of the user interface: the feature shown in a tree structure under the part view on the left, a graphical representation of the feature in the viewing outlined in pink and the feature listed with its associated operations on the operation list.

- Double-click on the **hole** in the **Part View**. The **Hole Feature Properties** window should appear.

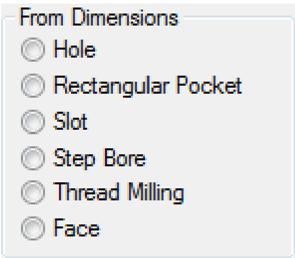


It should

be noted that the feature properties can be accessed by double clicking on each representation of the feature: in the Part View in the Graphics Window or the Operations List.

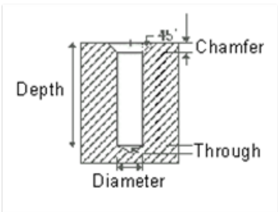
Features from Dimensions

Features from Dimensions consist of Hole, Rectangular Pocket, Slot, Step Bore, Thread Milling and Face.

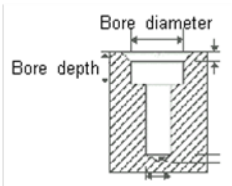


Holes

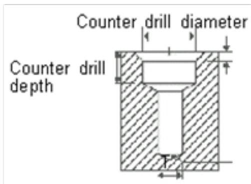
There are six different types of hole features in FeatureCAM. Plain, Counter bore, Counter drill, Counter sink, Tapped, Counter drilled tapped. Each hole type consists of different operations to create the resulting feature. Below is a graphical representation of the available holes in FeatureCAM



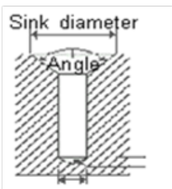
Plain
- Optional chamfer



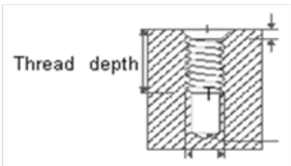
Counter bore
- Top hole bored
- Bottom drilled



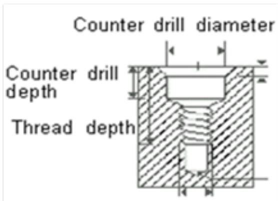
Counter drill
- 2 drill diameters



Counter sink
- Top countersunk
- Angle can vary



Tapped
- Dimensions manual or from a list

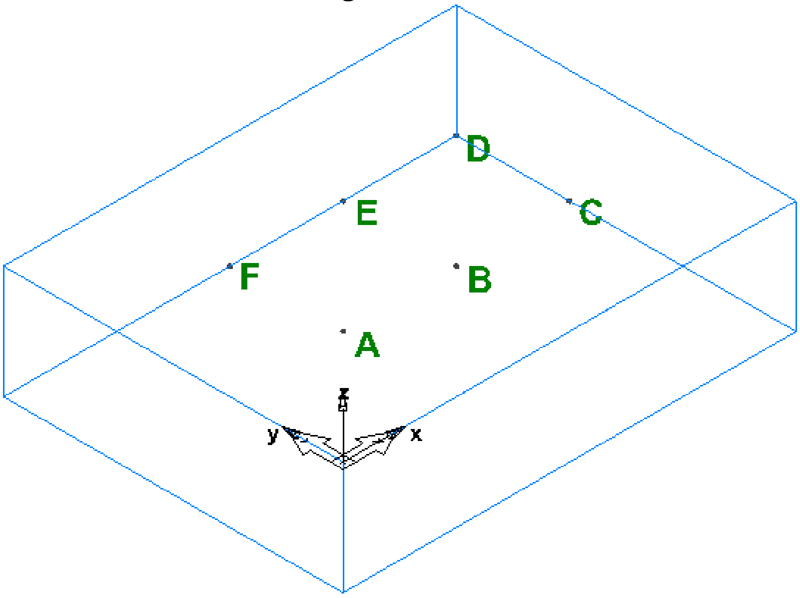


Counter drilled tapped
- Combines both

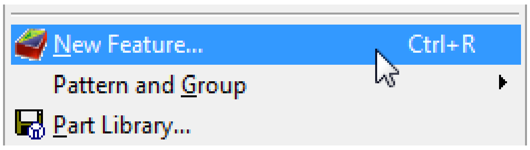
Feature From Dimensions Exercise #1

- From the file menu click on **Open**.
- Select **holes.fm** then press **Open**.

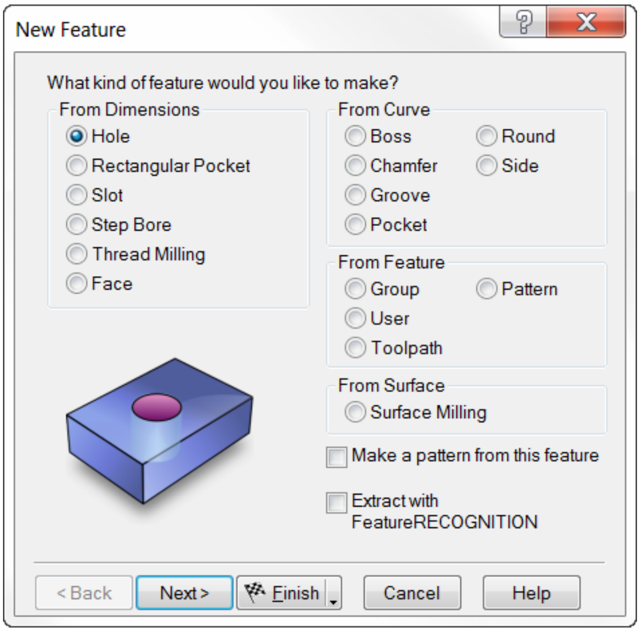
The opened file should look similar to the image below:



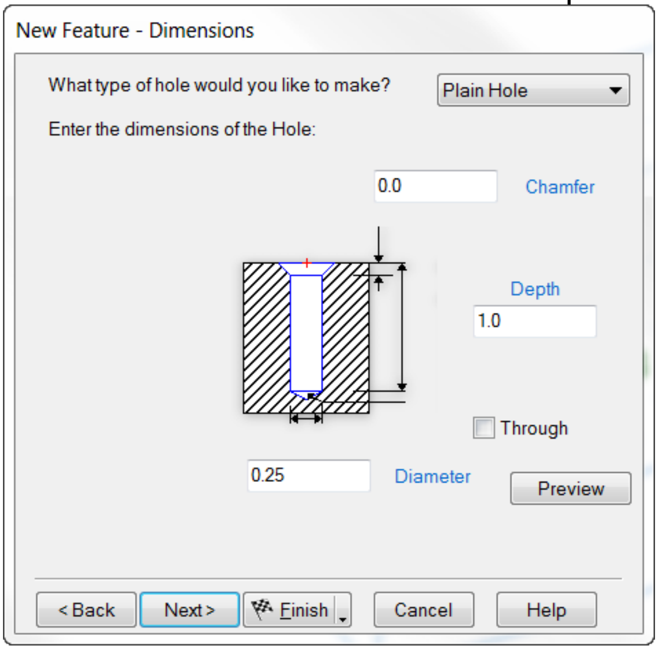
- Click on **Construct** then **New Feature**.



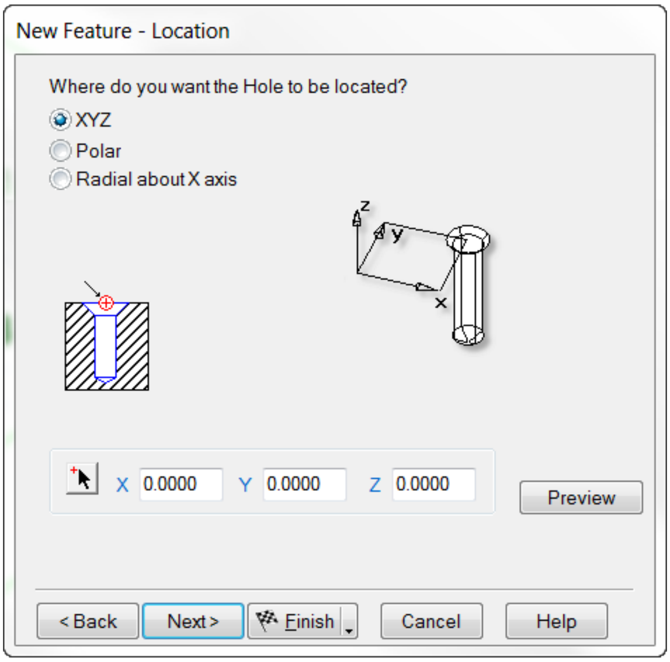
- In the **New Feature** Wizard select **Hole** then press **Next**.



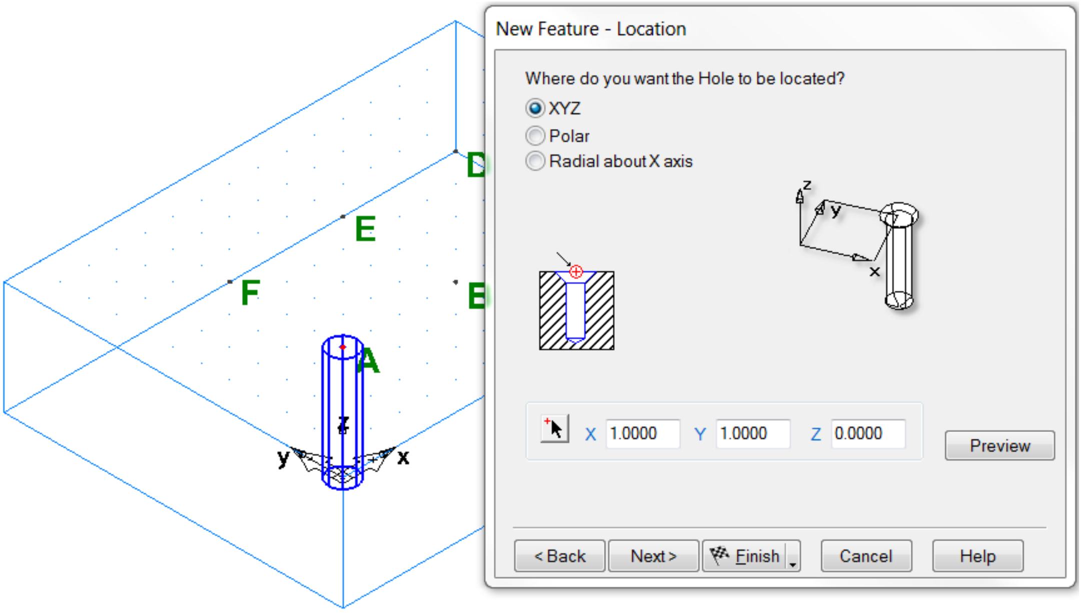
- On the new feature dimensions page select **Plain Hole**, enter a **Depth** of **1.0** and a **Diameter** of **0.25** then press **Next**.



- Click on the **Pick location** button.

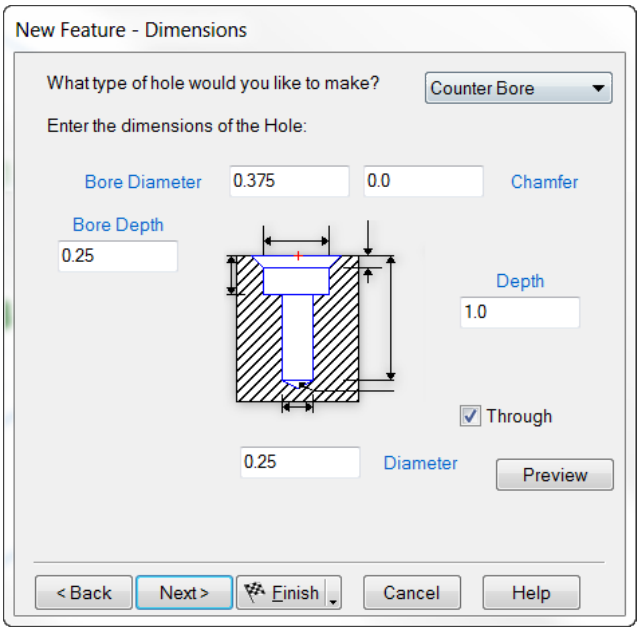


- Click once on **point A** (values should be identical to those listed below once point A is clicked) then press **Finish** and **OK.**

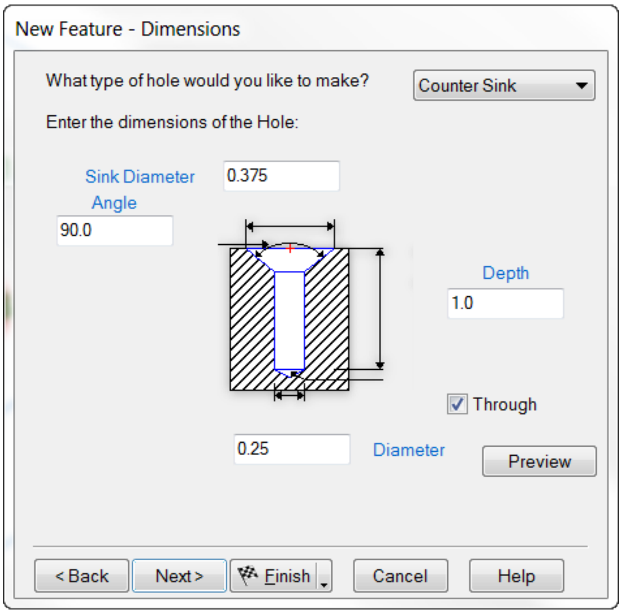


A plain hole is created at point A. The next step is to create the 5 different types of remaining holes at B, C, D, E and F.

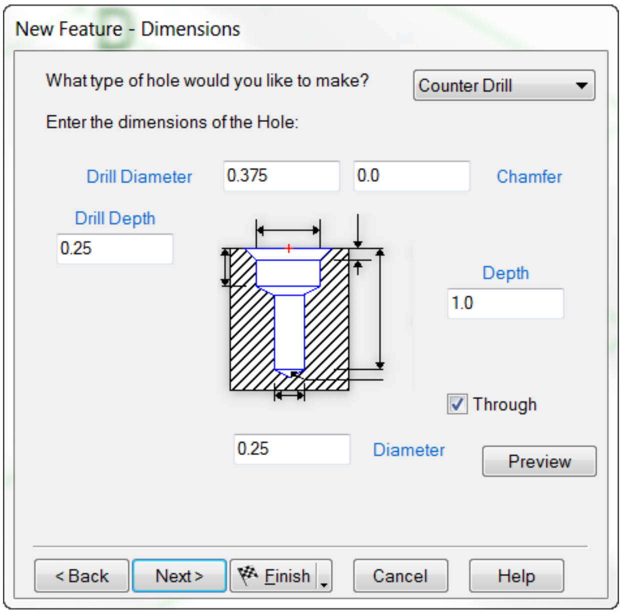
- At point **B** create a **Counter Bore Hole** with the dimensions shown.



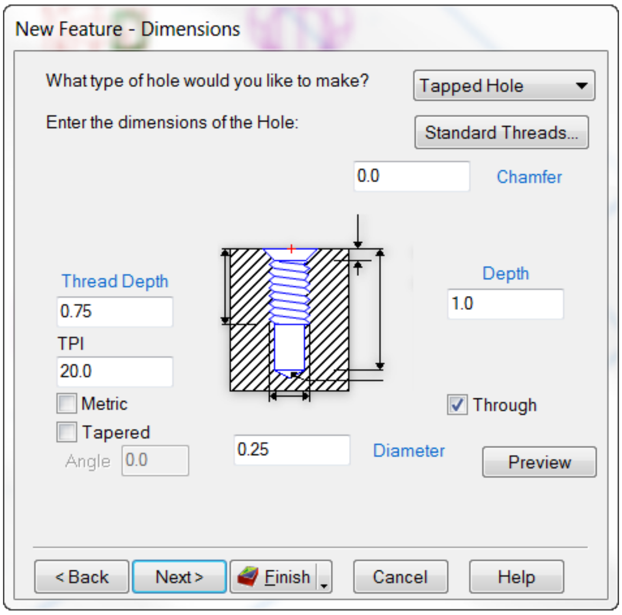
- At point **C** create a **Counter Sink Hole**.



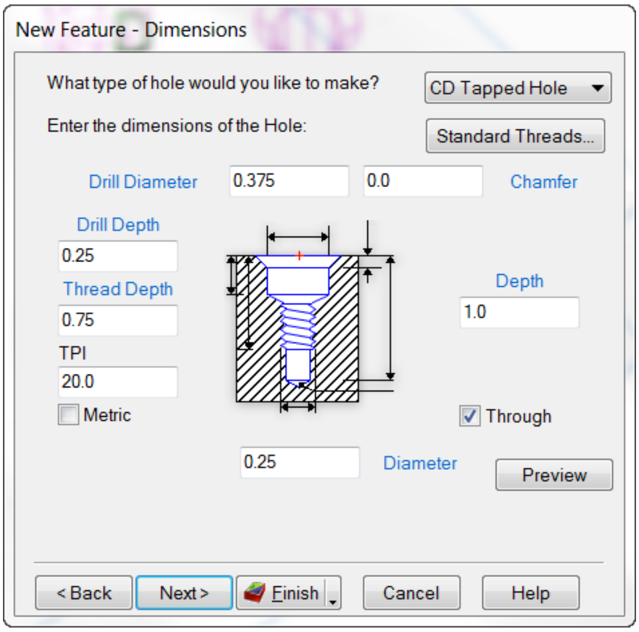
- At point **D Counter Drilled Hole**.



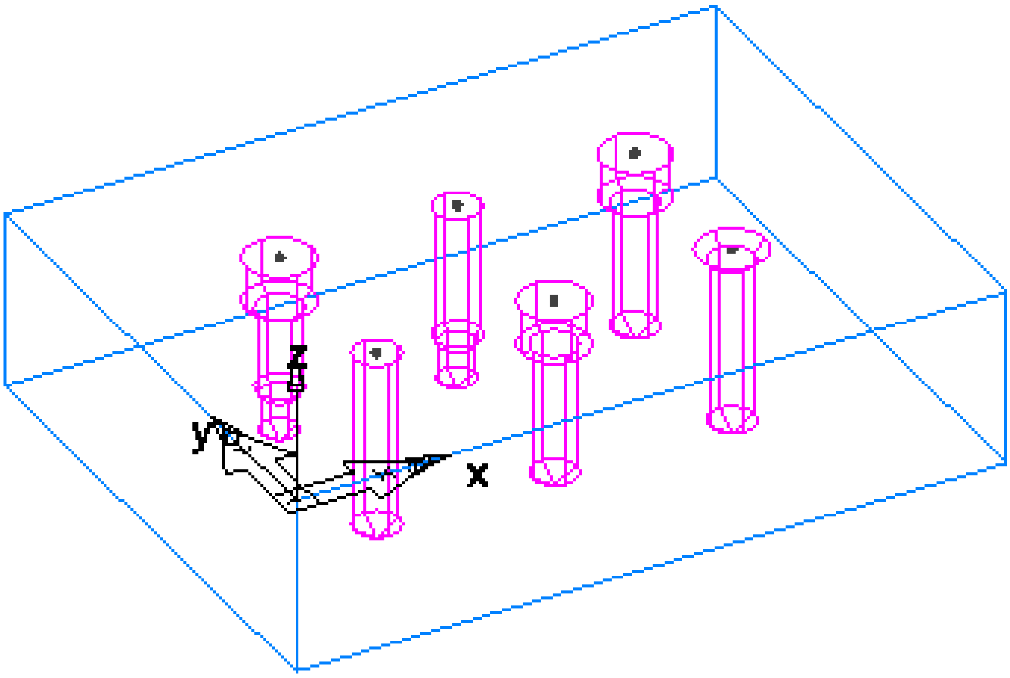
- At point **E** create a **Tapped Hole**



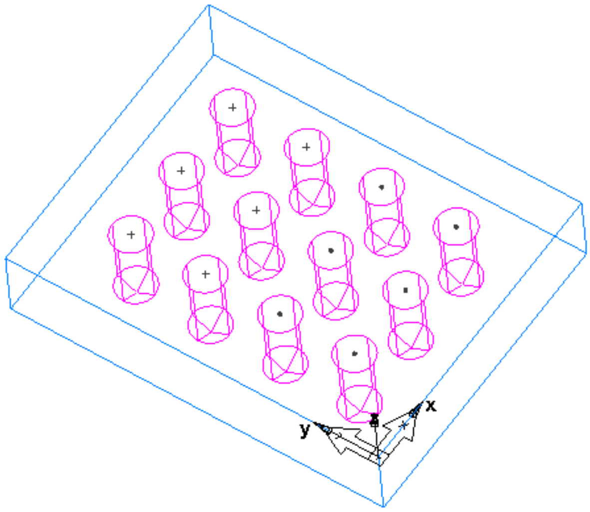
- At point **F** create a **Counter Drilled and Tapped Hole**.



When this exercise is finished you should have results similar to the image below:

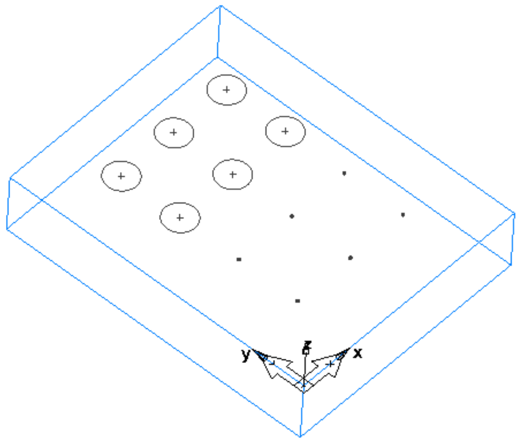


Feature From Dimensions Exercise #2

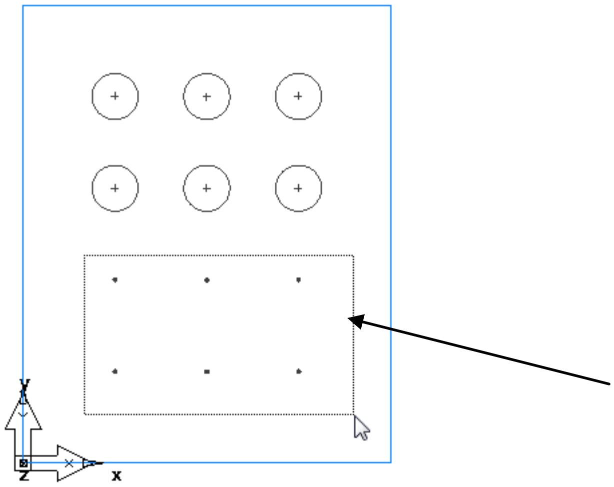


- From the file menu click on **Open**.
- Select **hole patterns.fm** then press **Open**.

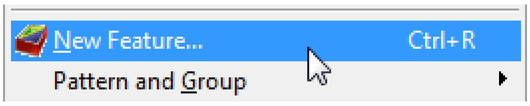
The opened file should look similar to the image below:



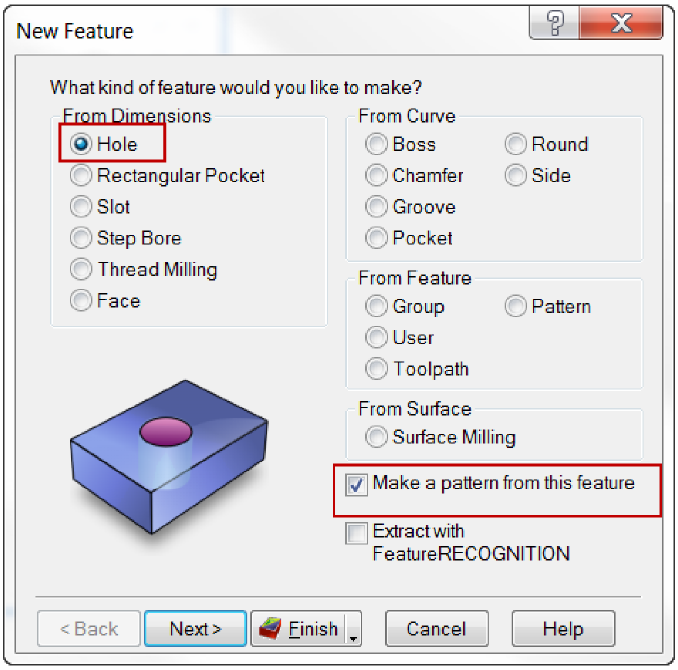
- Left click and **drag a bounding box** to enclose all six points so they are highlighted.



- With the six points still highlighted click on **Construct** then **New Feature**.

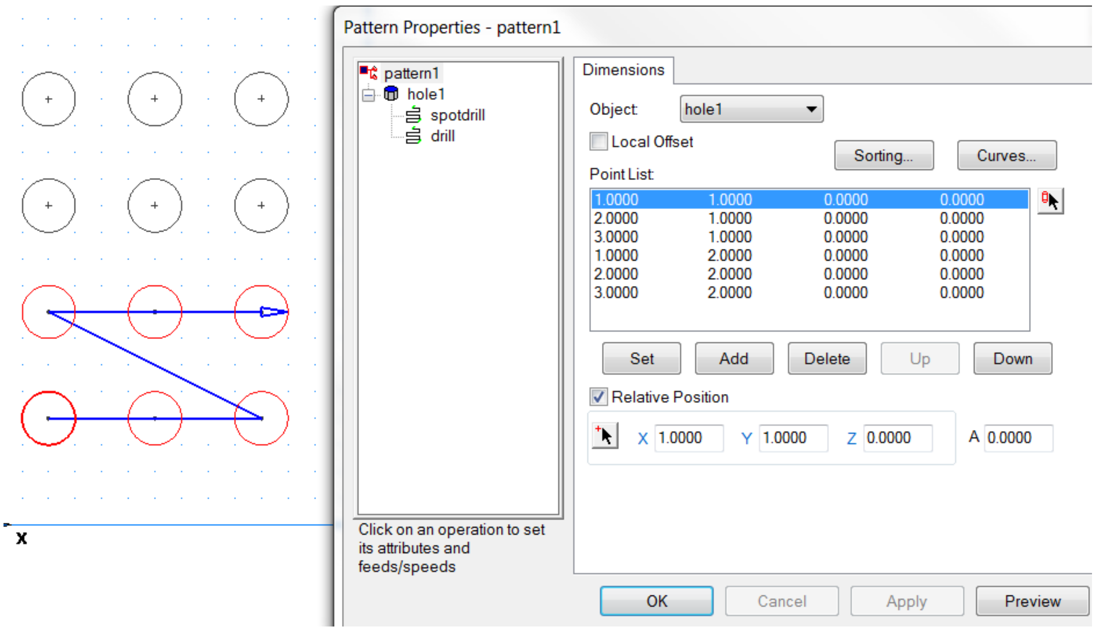


The New Feature wizard should appear notice that hole is automatically selected and that **Make a pattern from this feature** is automatically checked.

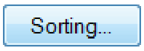


By selecting these points and then entering the New Feature Wizard FeatureCAM knows to create a pattern of holes.

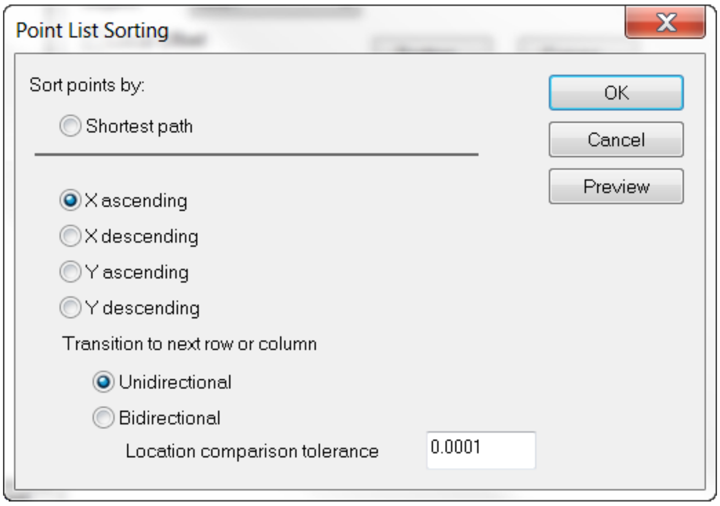
- Press **Next.** Select a **Plain Hole**.
- Press **Finish**



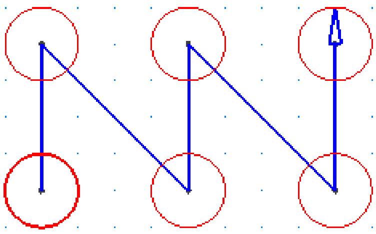
The **Pattern Properties** appear and a **Point List** for the pattern appears. The order in which the holes are drilled depends on the order the points listed in the table.

- Press the **Sorting** button 

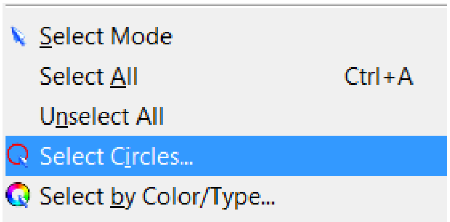
- Press **X-ascending** then press **OK** then **Apply**.



Observe the graphics window. Notice that the order in which the holes are drilled has been changed.

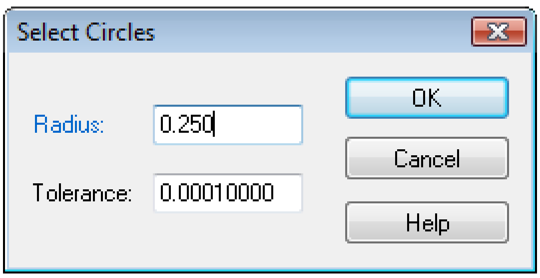


- Press **OK**
- From the file menu click on **Edit** then **Select circles**

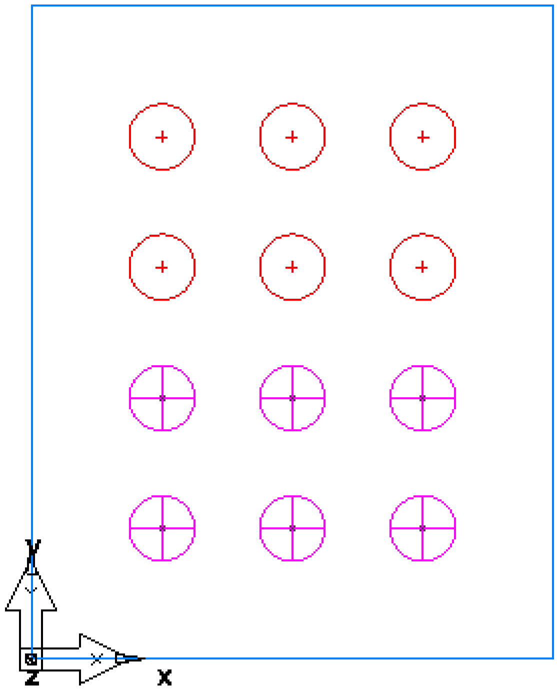


A filter dialog will appear for selecting circles of a specific diameter and tolerance.

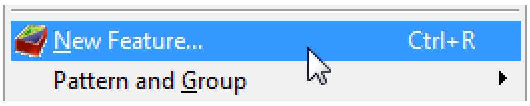
For the **Radius** value enter **0.250** and enter **0.0001** for the **Tolerance** then press **OK**



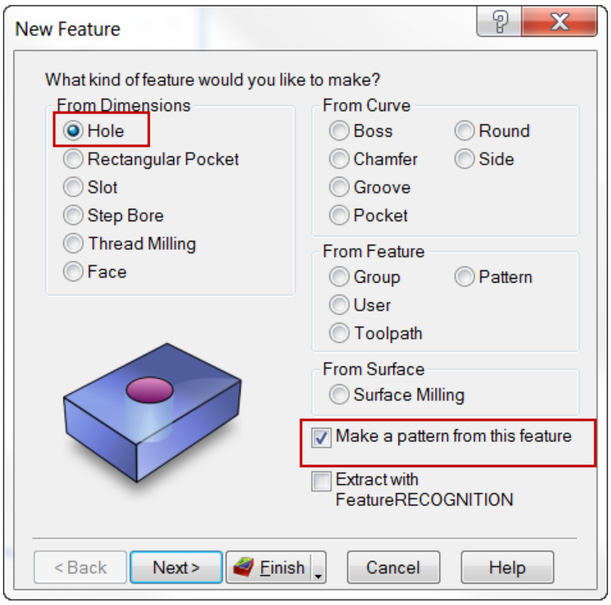
The circles on the upper half of the part drawing should highlight red.



- With the six circles still highlighted click on **Construct** then **New Feature**.

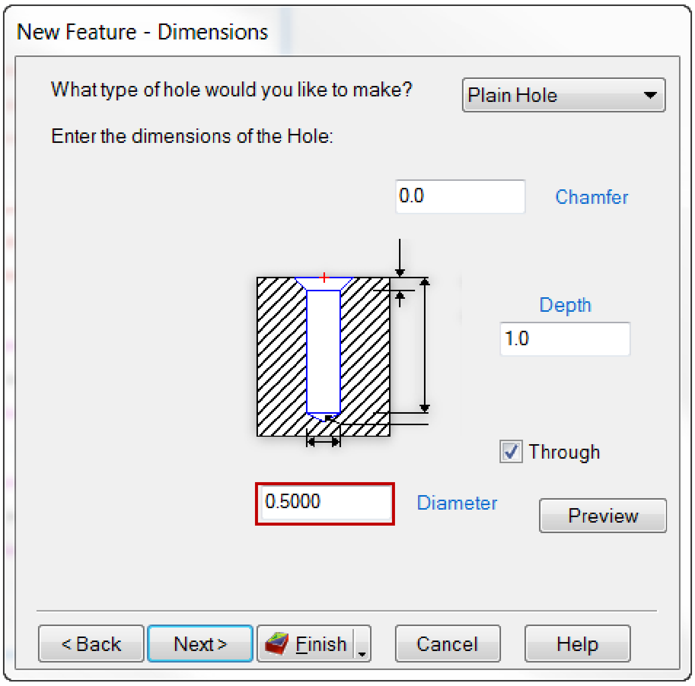


- Select **Hole** and check **Make a pattern from this feature**.

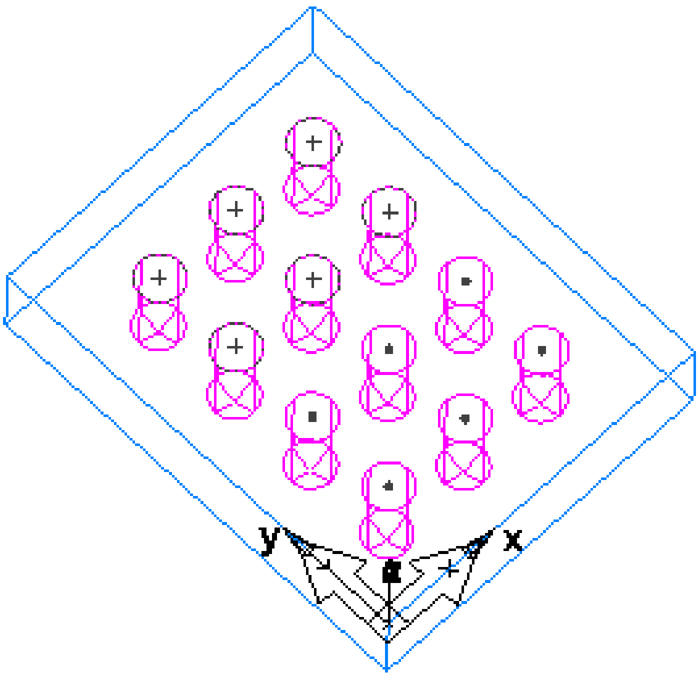


- Press **Next**.

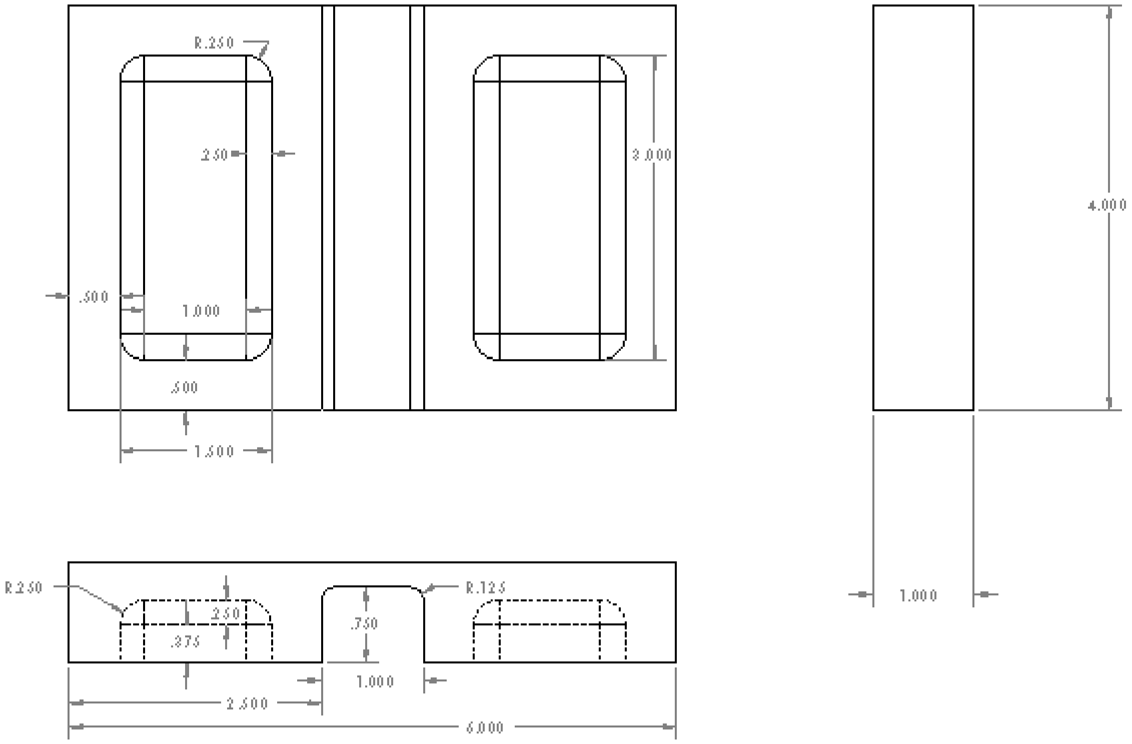
The hole **Diameter** field is automatically populated because the circle diameter was previously selected.



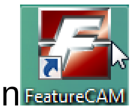
- Press **Finish** then **OK** and results should look similar to those shown below.



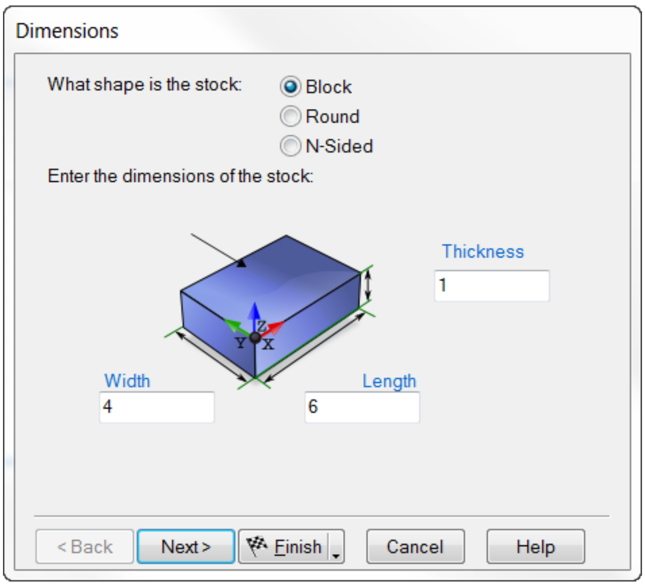
Feature From Dimensions Exercise #3



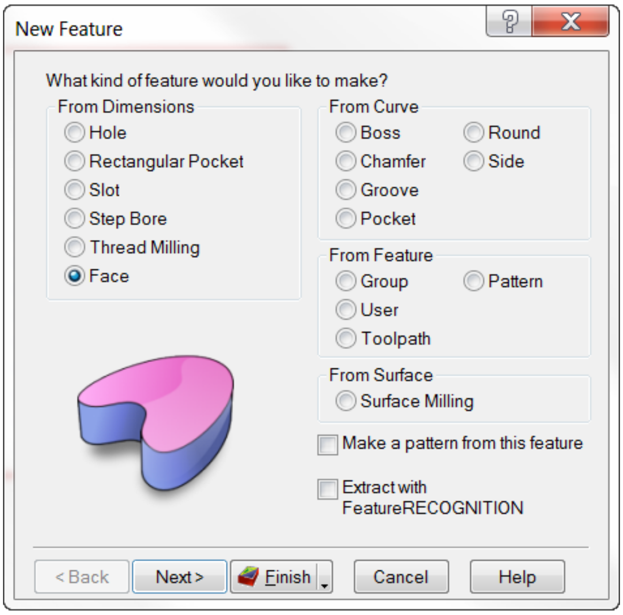
This exercise will familiarize the user with creating pockets and slots from dimensions.

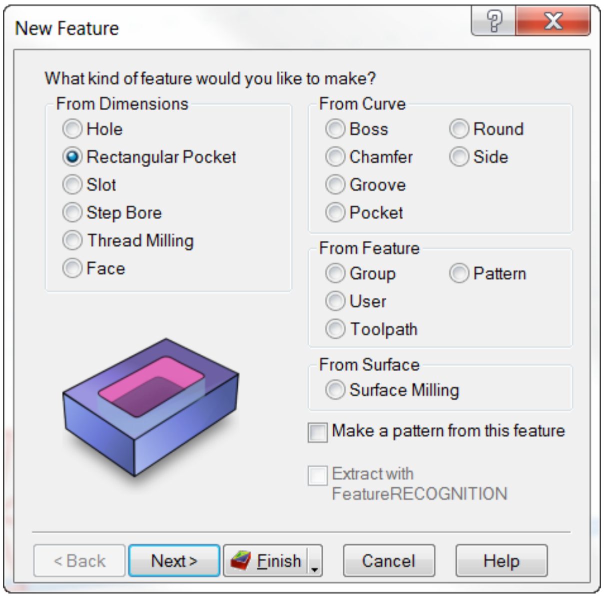
- From the desktop double click on the FeatureCAM shortcut icon 

- From the **New Part Document Wizard** select **New File**, press **Next**.
- Select **Milling** then **inches** and press **Finish**.
- On the dimension page enter **4** for the **Width**, **6** for the **Length** and **1** for the **Thickness** then press **Finish** and **OK**.



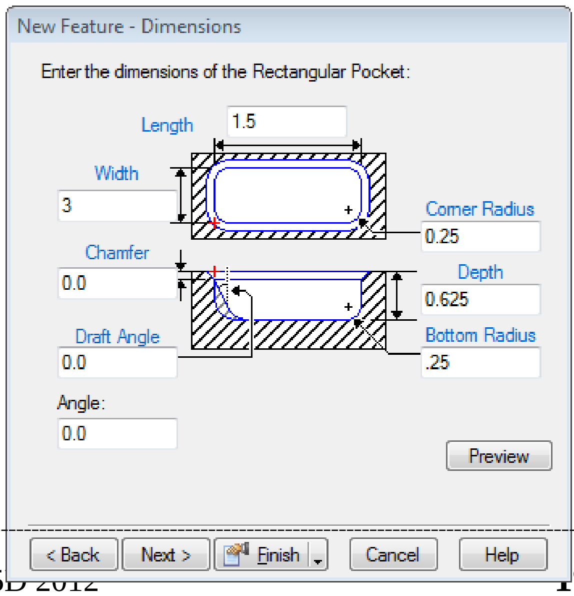
- Open the **New Feature** wizard from the **Steps Toolbox**. Select **Face** then **Finish**.



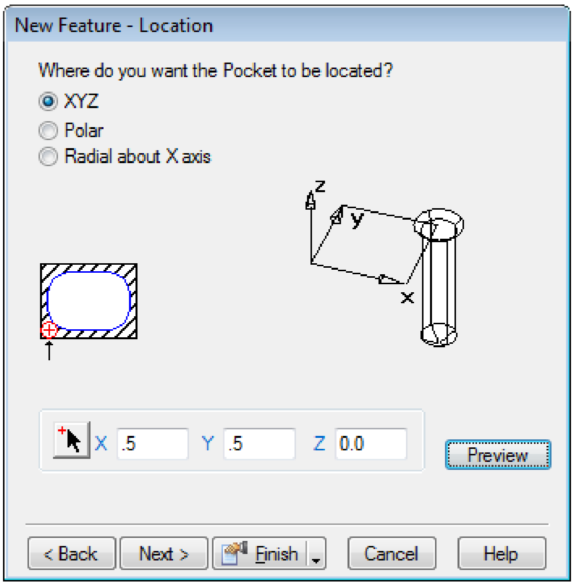


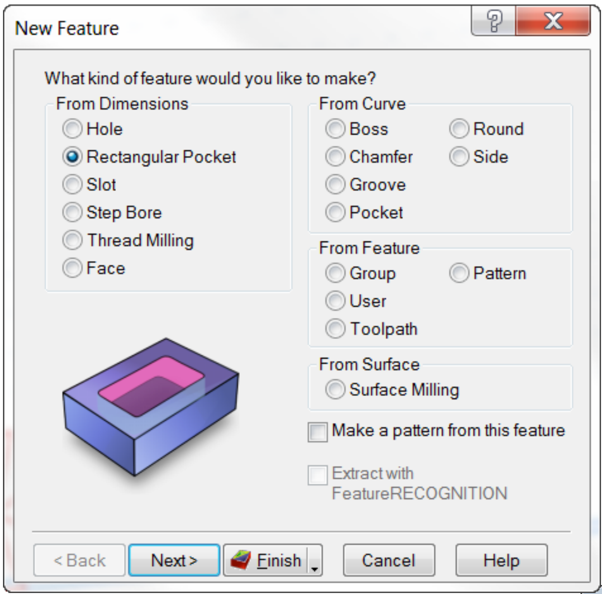
- Press on the **Features** icon to open a **New Feature** from the **Steps Toolbox**. Select **Rectangular Pocket** then press **Next**.

- Enter the dimensions of the pocket as shown. **3** for the **Width**, **1.5** for the **Length** and **0.625** for the **Depth**. The **Corner Radius** should be set to **0.25** with a **Bottom Radius** of **0.25**. Once the dimensions have been entered press **Next**.

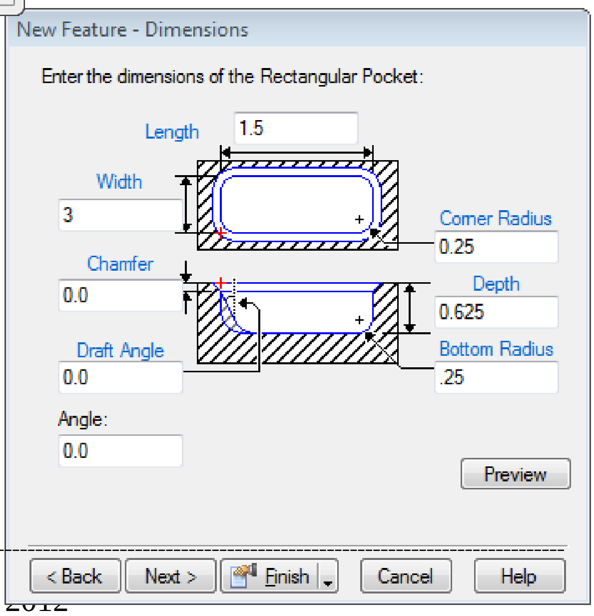


- On the location tab select the **XYZ** radio button and enter a location of **(.5, .5, 0)** then press **Finish**.

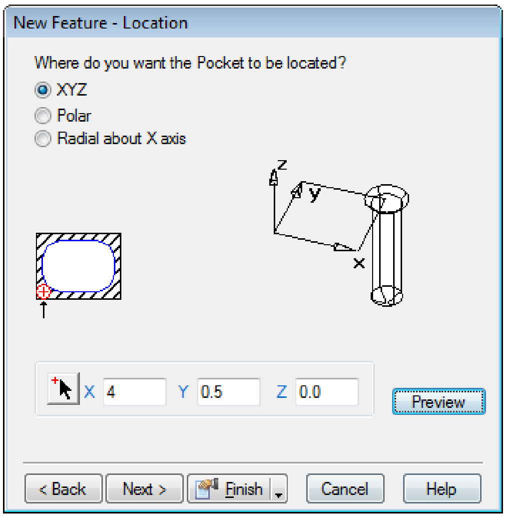




- Open the **New Feature** wizard from **Steps**.
- Select **Rectangular Pocket** then press **Next**.

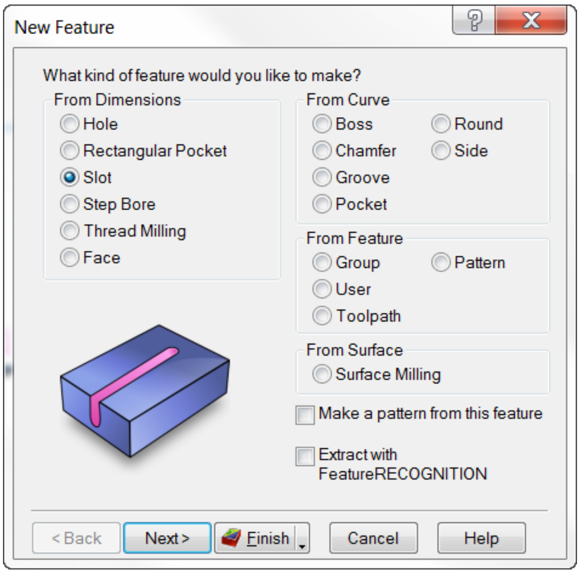


- Enter the dimensions of the pocket as shown below. **3** for the **Width**, **1.5** for the **Length** and **0.625** for the **Depth**. The **Corner Radius** should be set to **0.25** with a **Bottom Radius** of **0.25**. Once the dimensions have been entered press **Next**.

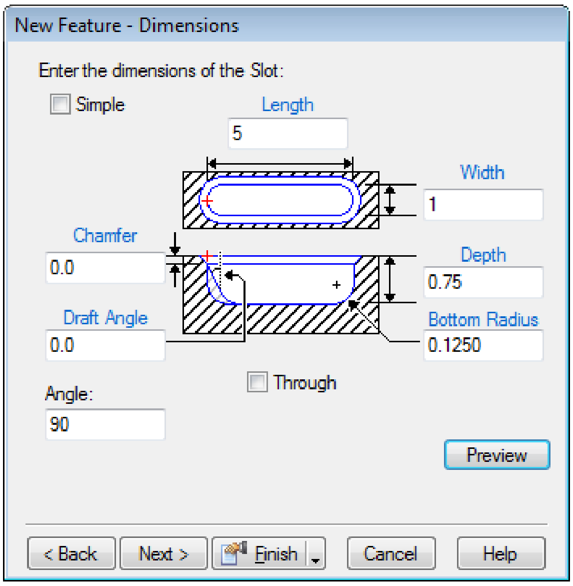


- On the location page select the **XYZ** radio button and enter a location of **(4, .5, 0)** then press **Finish**.

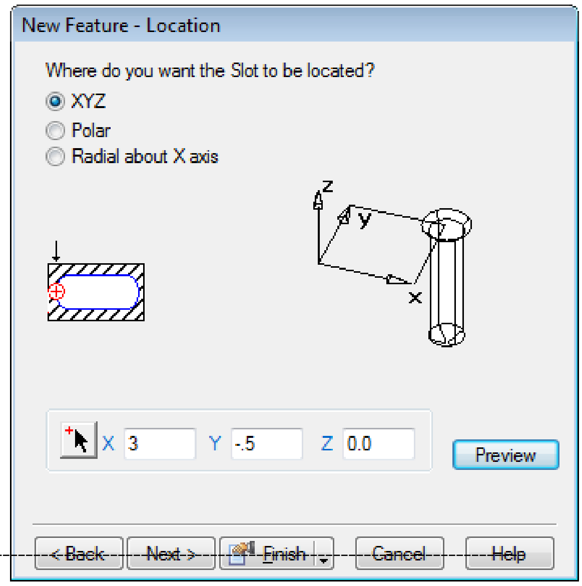
- Press on the **Features** icon to open the **New Feature** wizard from the **Steps Toolbox**. Select **Slot** then press **Next**.



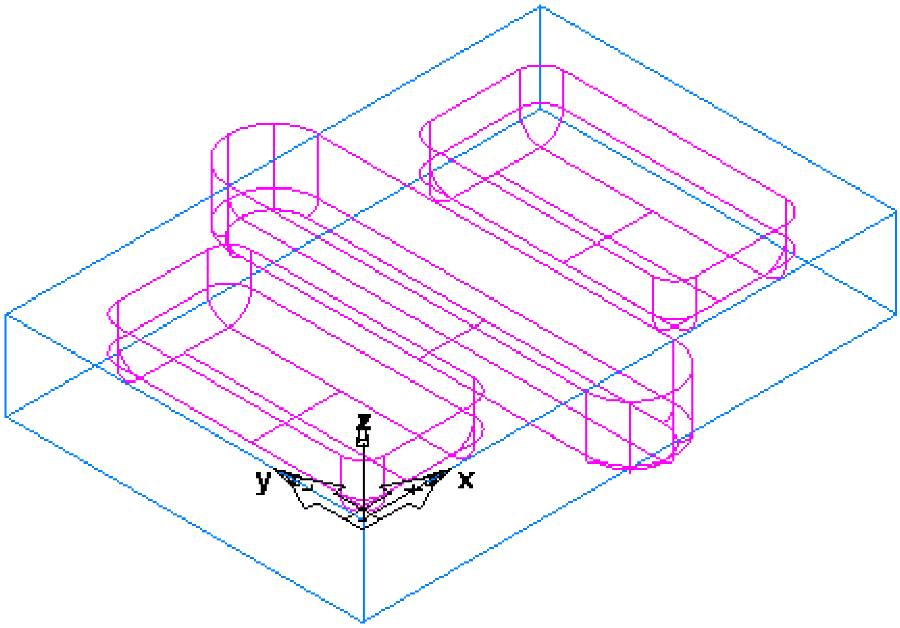
- Enter the slot dimensions as shown below. Enter a **Length** of **5.0**, a **Width** of **1.0**, **Depth** of **0.75** ,a **Bottom Radius** of **0.125** and an angle of **90** degrees then press **Next**.



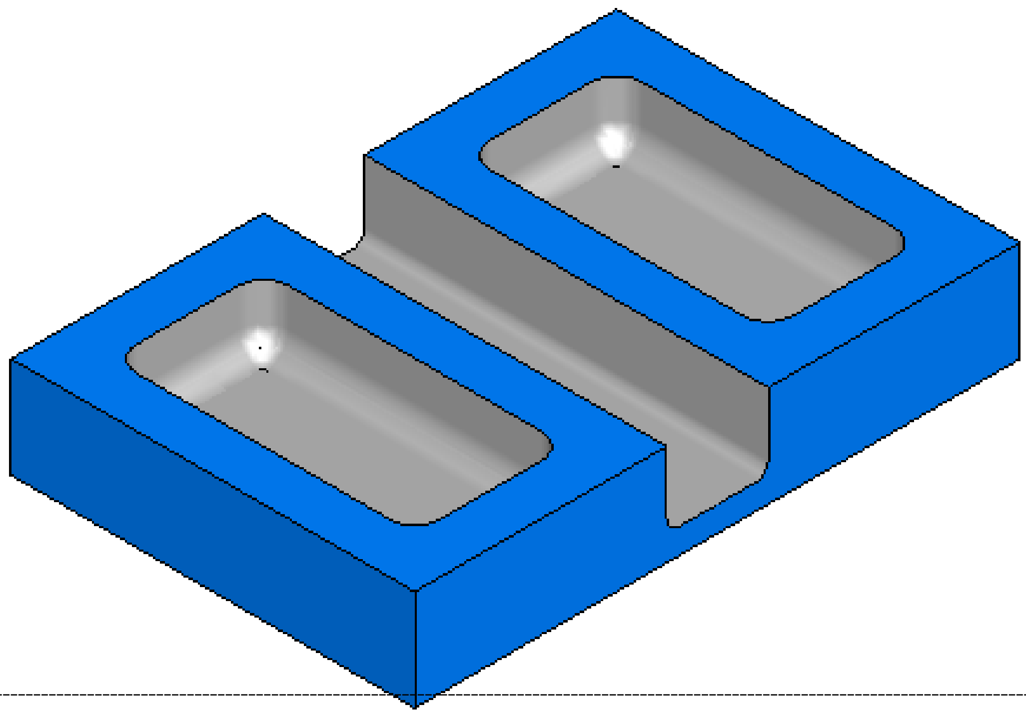
- On the location page select the **XYZ** radio button and enter a location of **(3, -0.5, 0)** then press **Finish**.



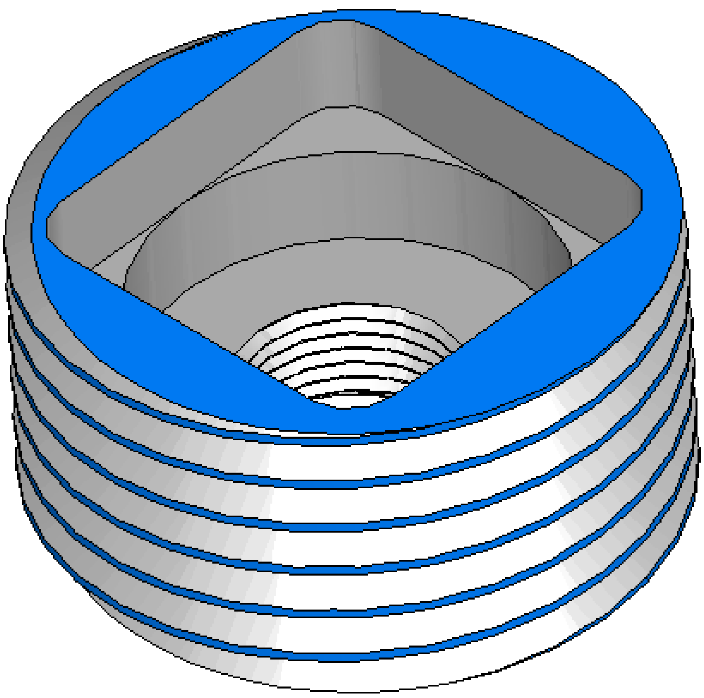
When this exercise is complete you should get feature results similar to the image shown below:



- Right click in the graphics window and select **Isometric**.
- From the **Steps** click the **Toolpaths** icon, then run a **3D simulation**.



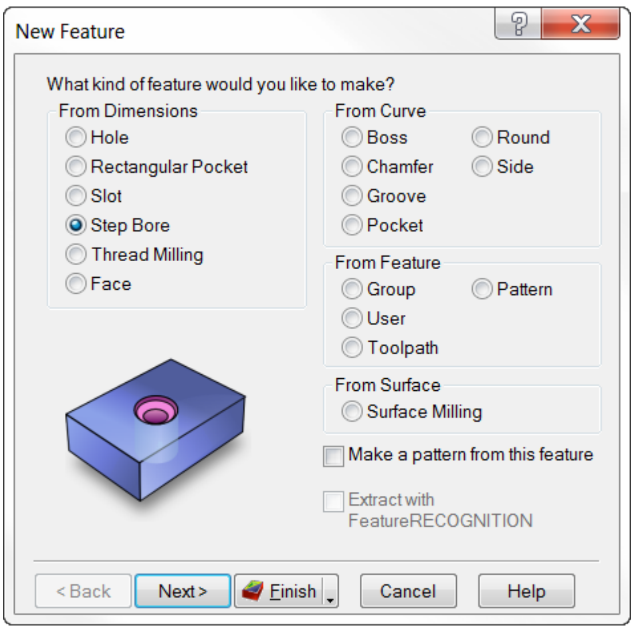
Feature From Dimensions Exercise #4



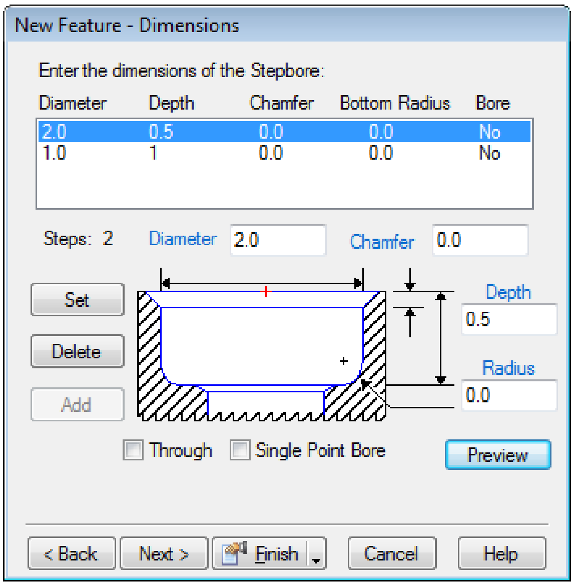
- From the FeatureCAM file menu click on **File** then **Open**.
- Select the file named "**ThreadmillStepbore.fm**" then press **Open**.

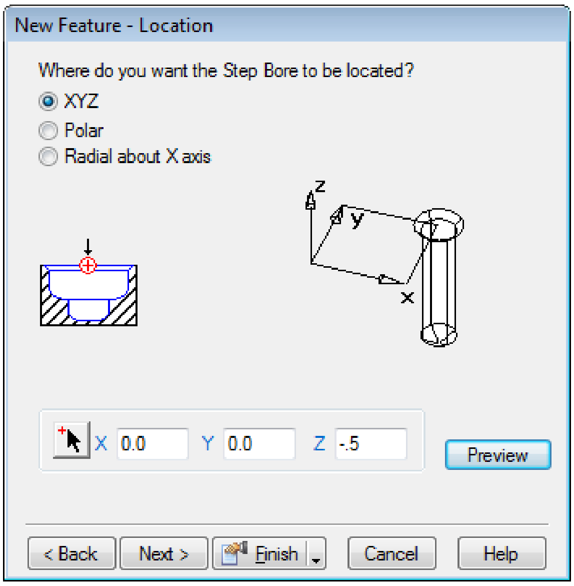
Examine the features in setup 1 (**Part View**). Notice that a face feature and rectangular pocket have already been created.

- Press on the **New Feature** Wizard icon from the **Steps Toolbar**.
- Select **Step Bore** then press **Next**.



- Enter the values as shown for each of the steps, press **Set** then press **Next**.

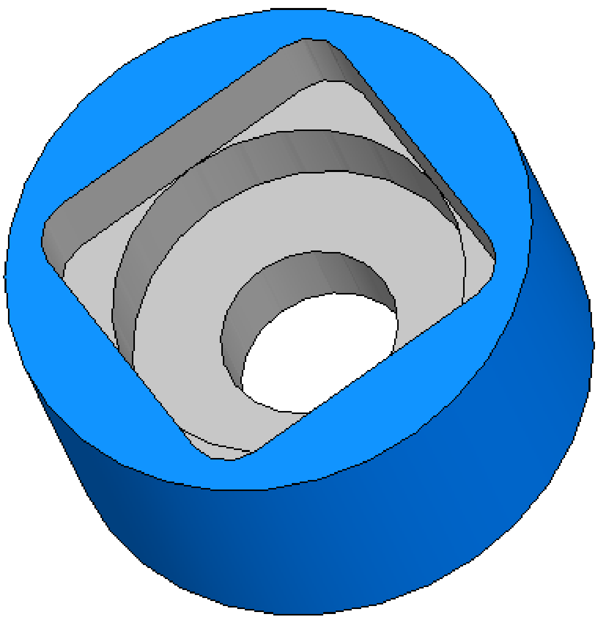




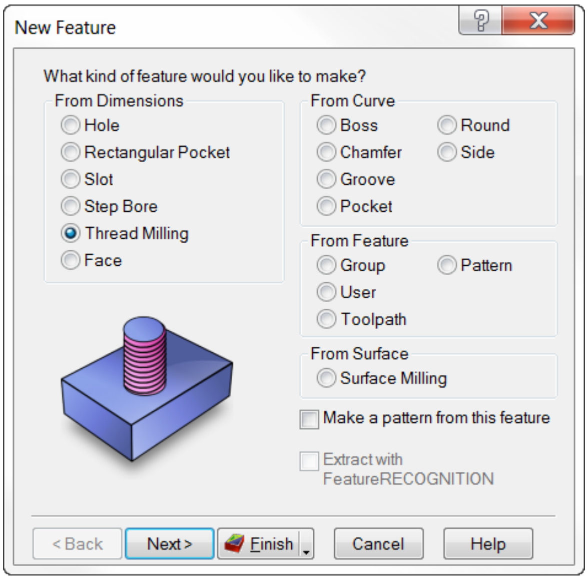
- Press the **XYZ** radio button then enter the location dimensions as **(0,0,-0.5)** then press **Finish** and **OK**.

- From the **Steps Toolbox** press the **Toolpaths** icon then run a **3D simulation**.

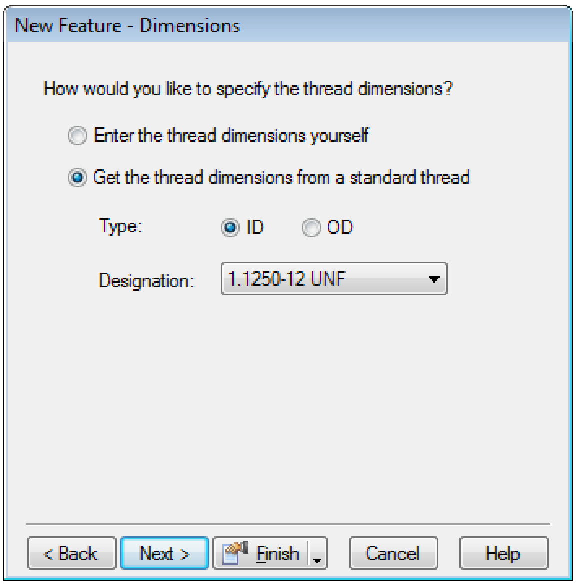
Results should look similar to the image to the right:



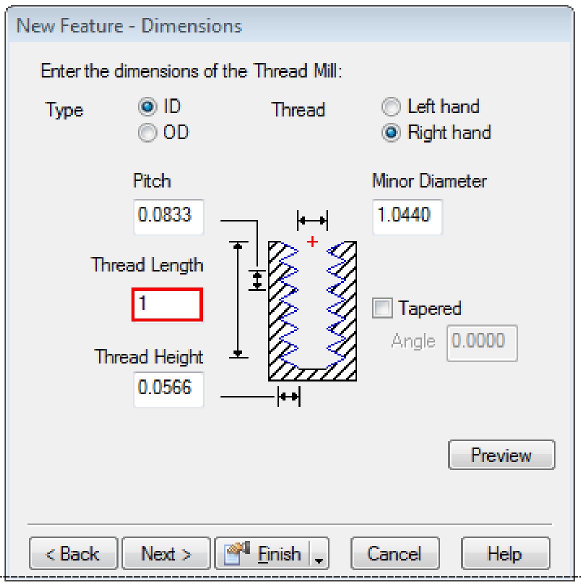
- **Eject** the simulation.
- Press the **New Feature** Wizard icon from the steps toolbar.
- Select **Thread Milling** then press **Next**.



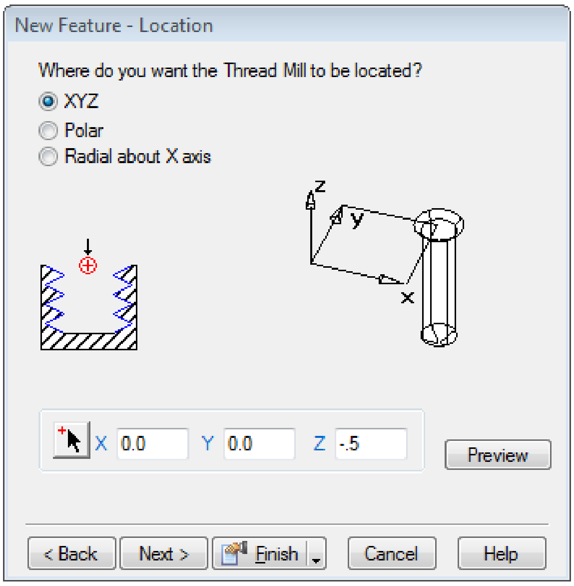
- Select **Get the thread dimensions from a standard thread**.



- Select **ID** then from the **Designation** pull-down menu select **1.1250-12 UNF** then press **Next**.

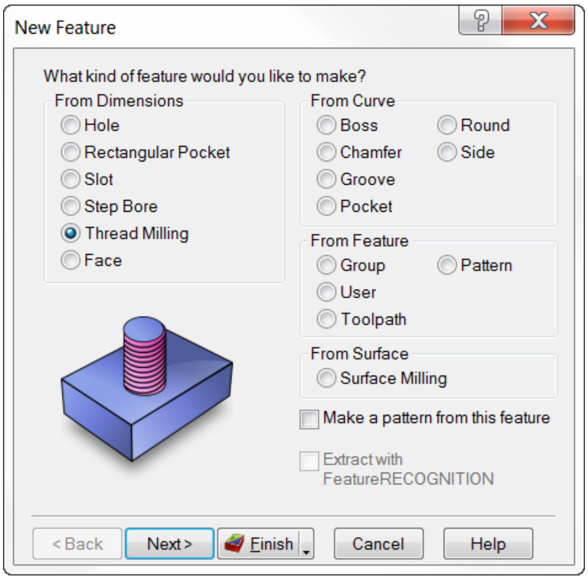


- In the **Thread Length** field enter **1** then press **Next**.

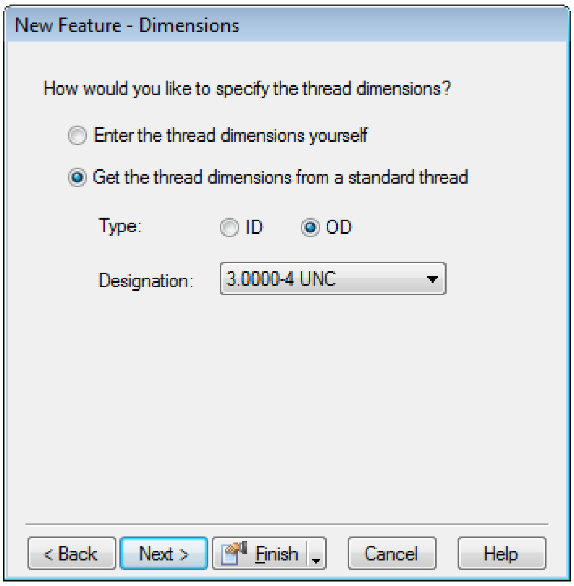


- Pick the **XYZ** radio button then enter a **Z location** of **-0.5** then press **Finish** and **OK**.

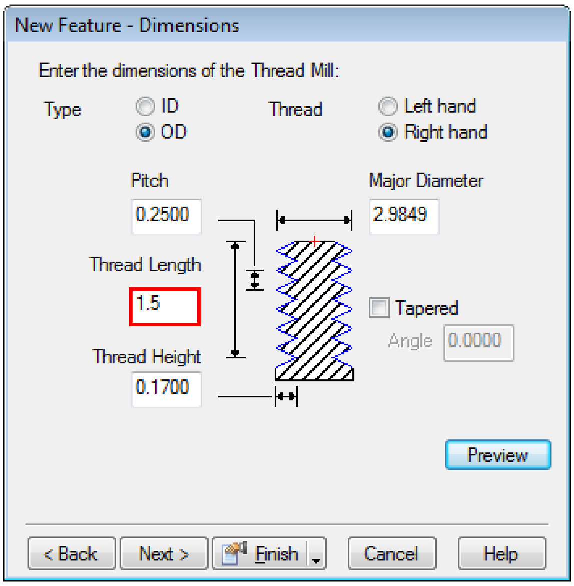
- Press the **New Feature** Wizard icon from the steps menu.
- Select **Thread Milling** then press **Next**.



- Select **Get the thread dimensions from a standard thread**.



- Select **OD** then from the Designation Pull-down menu select **3.0000-4 UNC** then press **Next**.

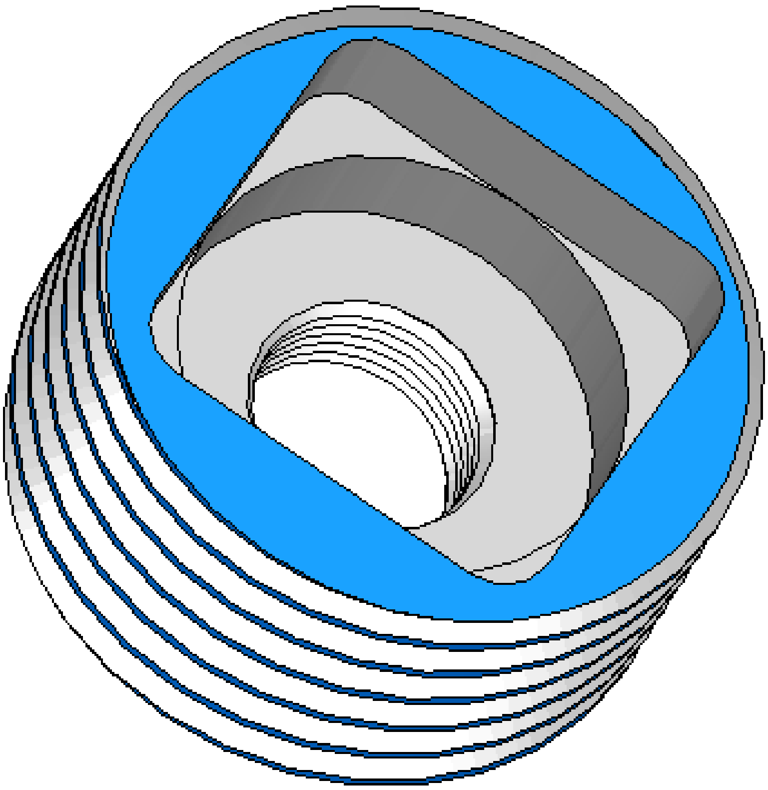


- In the **Thread Length** field enter 1.5 then press **Next.** Select the **XYZ** radio button enter all 0's in the X, Y, and Z fields.

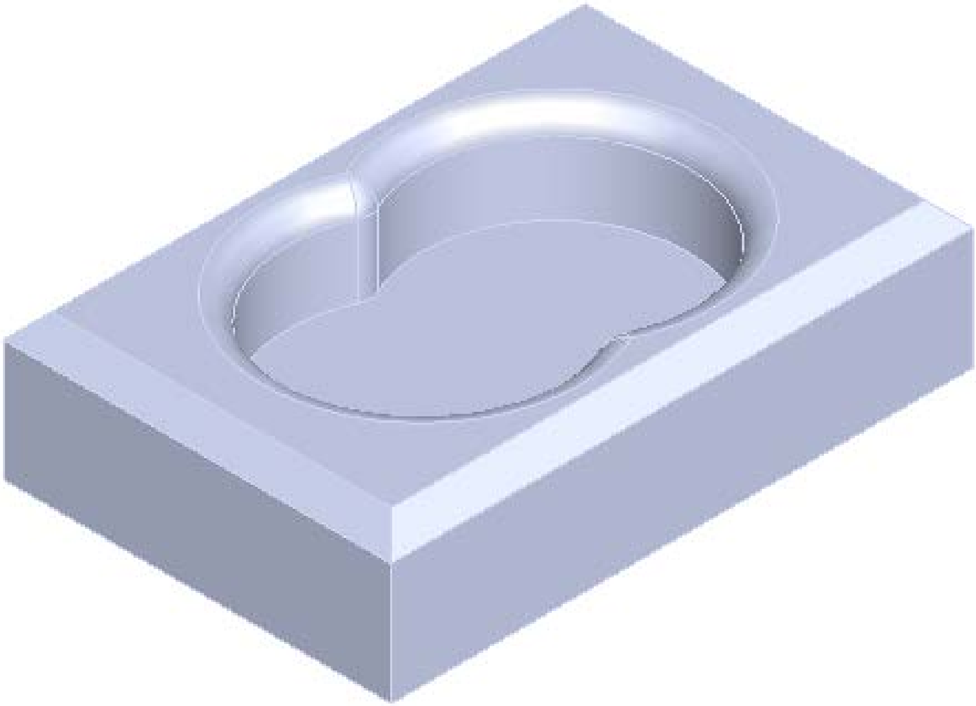
- **Finish** and **OK**.

- Run a **3D simulation.**

You should have results similar to the image below:



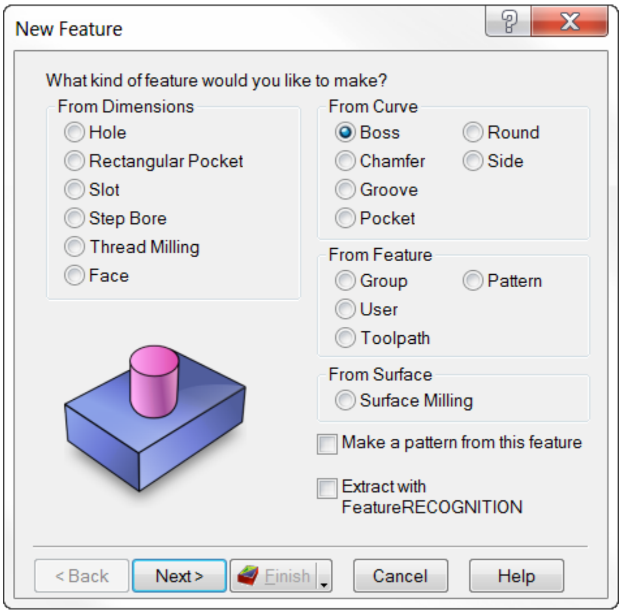
Features from Curves Exercise #1



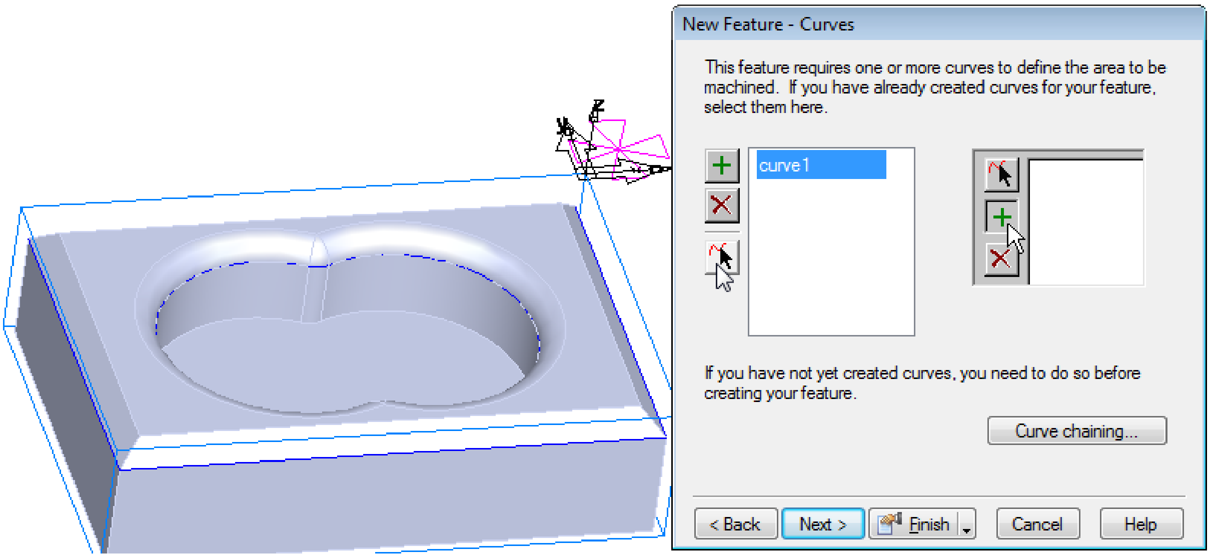
- **Open** the file named **ChamferFillet.fm**.

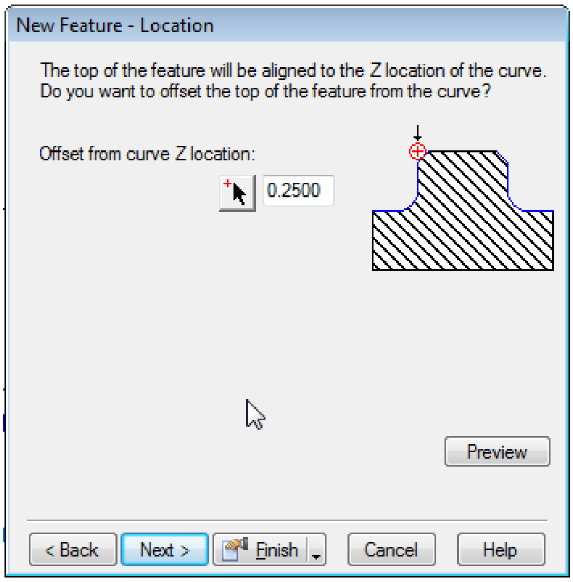
Examine the features in setup 1. Notice that a face feature has already been created.

- Press on the **New Feature** Wizard icon from the steps menu.
- Select **Boss** and press **Next**.

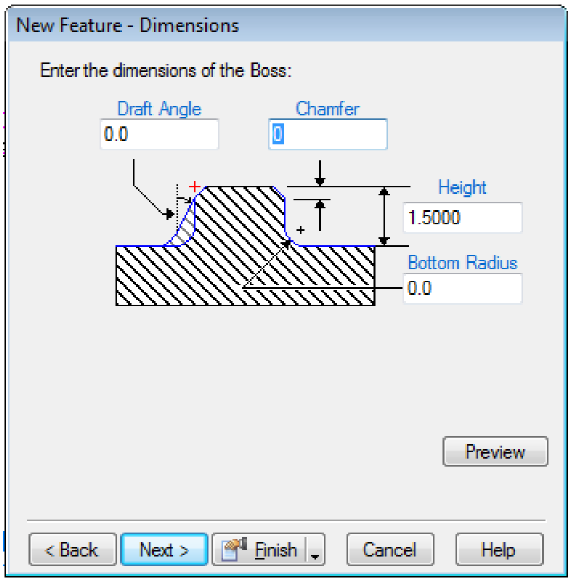


- Use the **Pick curve or geometry arrow** to select the curve in blue that goes around the outer profile of the part. The curve name should be **curve1** then press **Next**



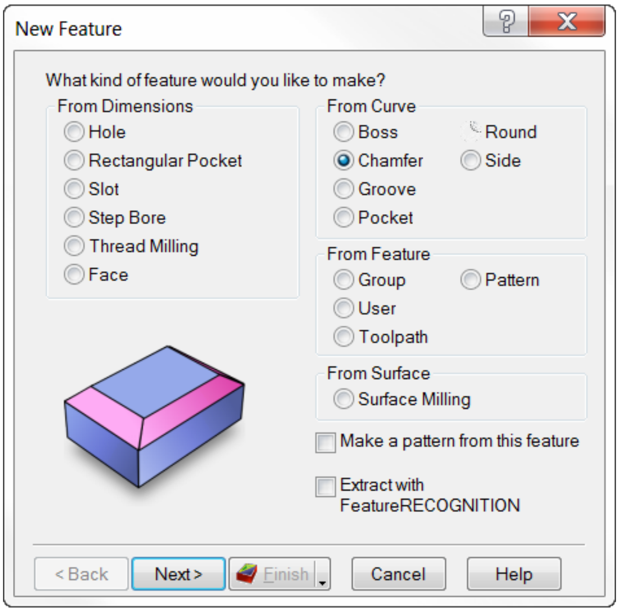


- For the **Offset from curve Z location** value enter **0.25**, then press **Next**.

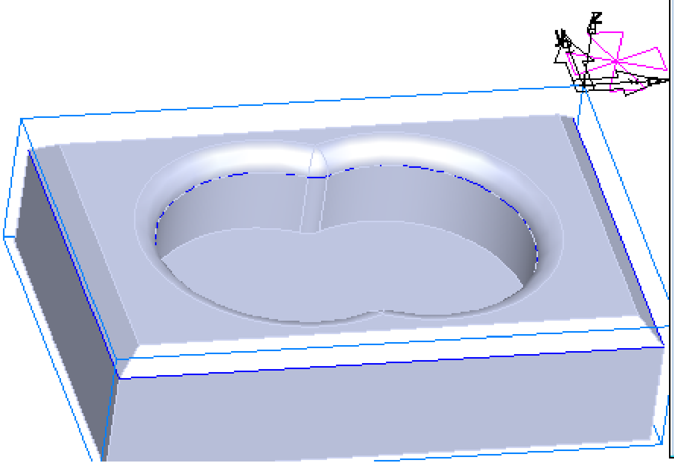


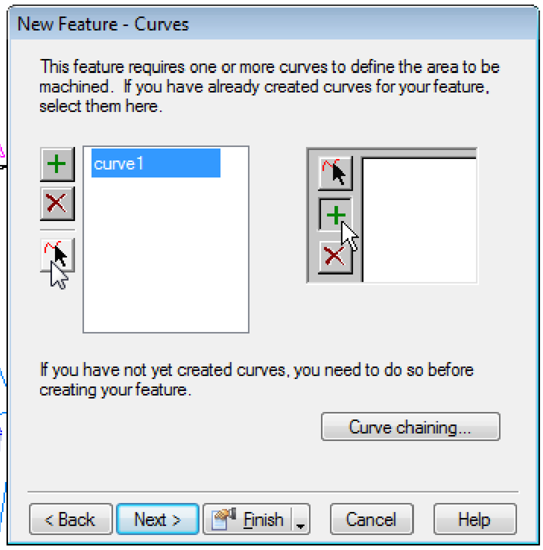
- On the **New Feature - Dimensions** page enter a **Height** of **1.5** then press **Finish** and **OK**.

- Open the **New Feature** Wizard select **Chamfer** then press **Next**.

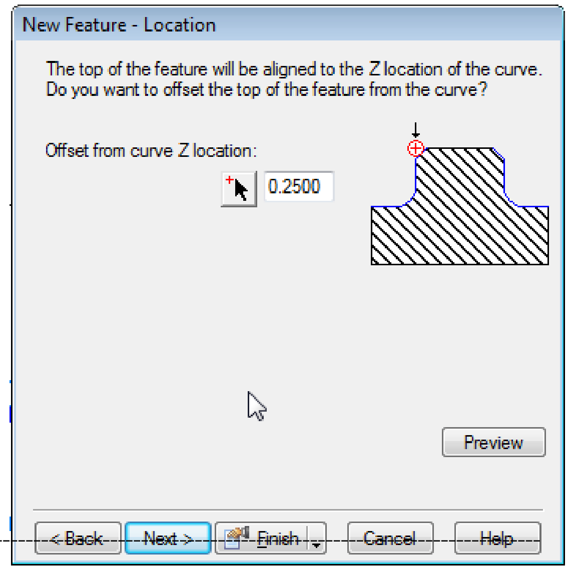


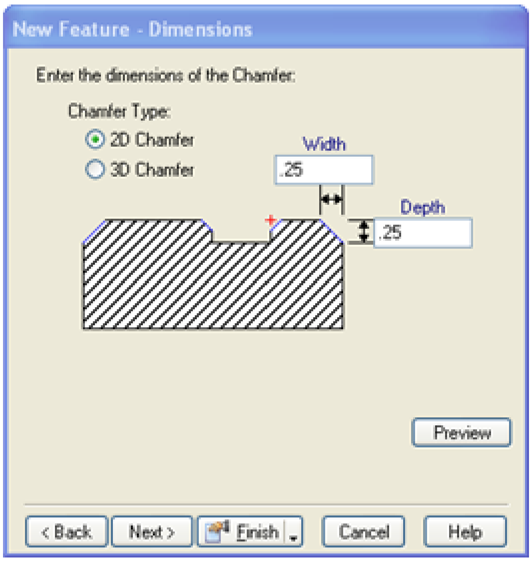
- Use the selection arrow to select the curve in blue that goes around the outer profile of the part. The curve name should be **curve1** then press **Next** twice.





- For the **Offset from curve Z location** value enter **0.25**, then press **Next**.



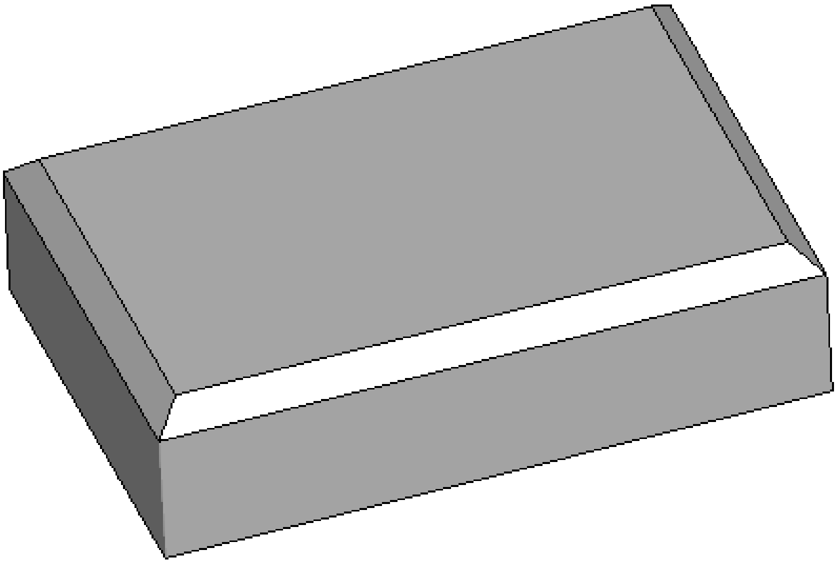


- Select **2D Chamfer**.

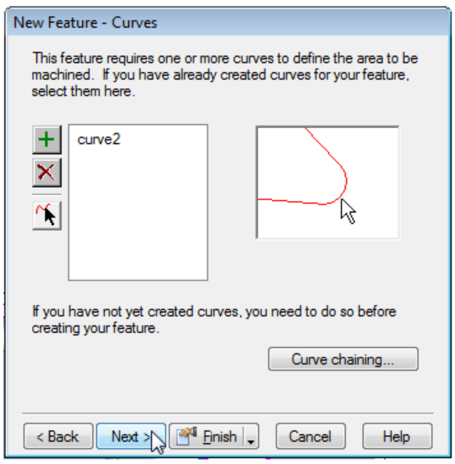
- For the **Depth** and **Width** dimensions for the chamfer enter **0.25** then press **Finish** and **OK**.

- Run a **3D simulation**.

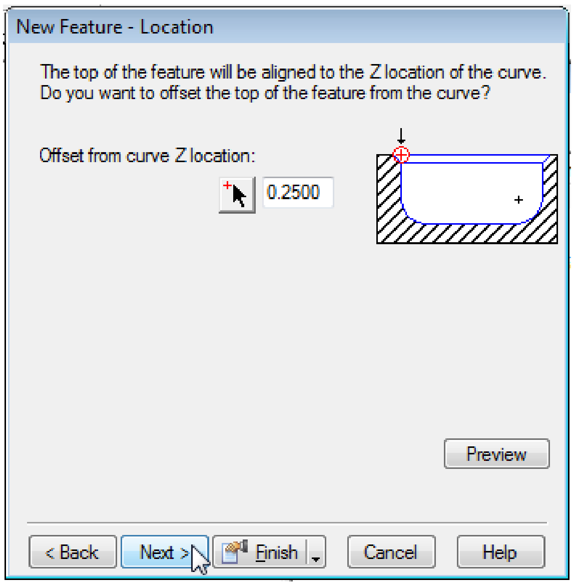
You should see results similar to the image below



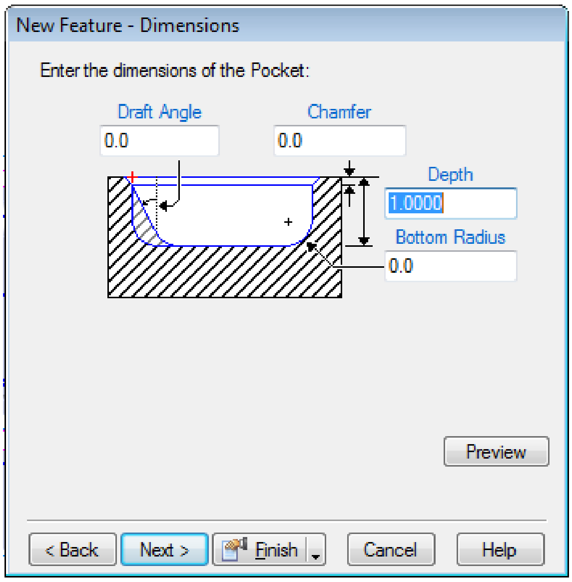
- Press the **Eject** button on the simulation toolbar to clear simulation results
- Enter the New Feature and select a **Pocket** then Press **Next**.



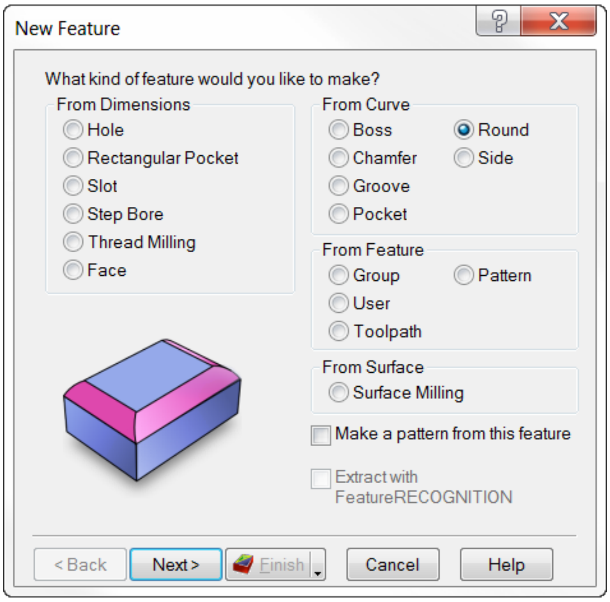
- Use the **Pick curve or geometry arrow** to select the curve in blue that goes around the outer profile of the pocket in the solid. The curve name should be **curve2** then press **Next**.



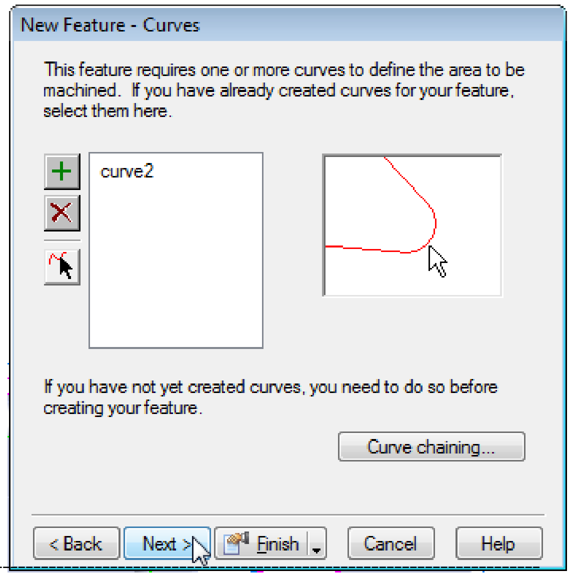
- For the **Offset from curve Z location** value enter **0.25** then press **Next**.



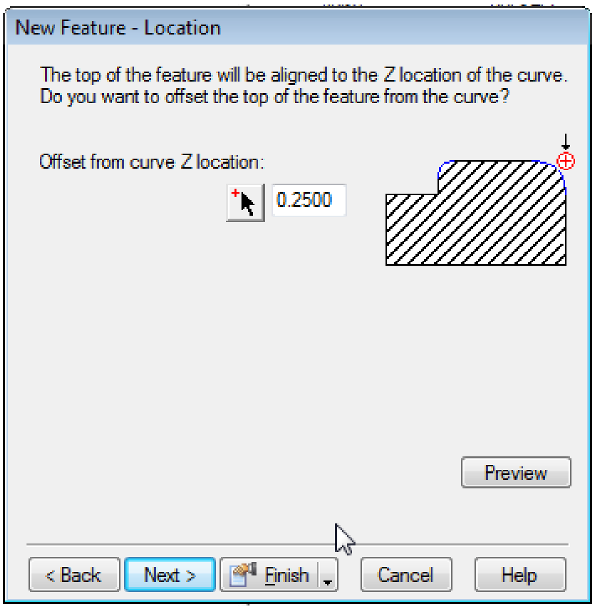
- On the Dimensions page enter the **Depth** of **1.0** then press **Finish** and **OK**.



- Click on the **New Feature** Wizard icon select **Round** then press **Next**.

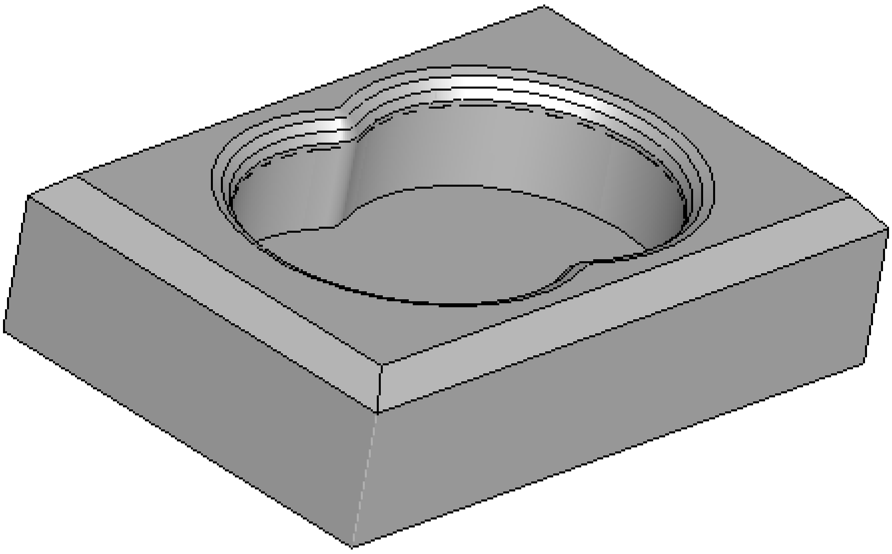


- Use the selection arrow to select the curve in blue that goes around the outer profile of the pocket. The curve name should be **curve2** then press **Next** twice.

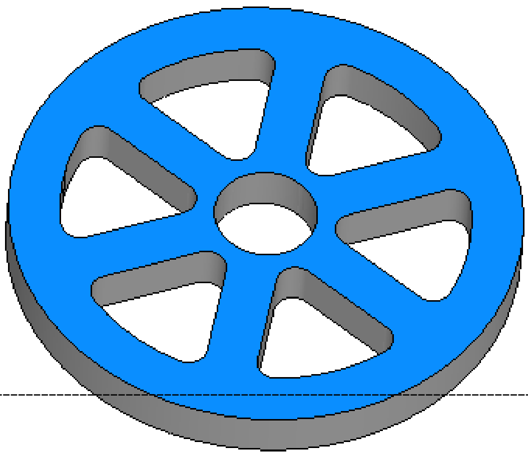


- For the **Offset from curve Z location** enter **0.25** then press **Finish** and **OK**.
- Run a **3D simulation** of the part.

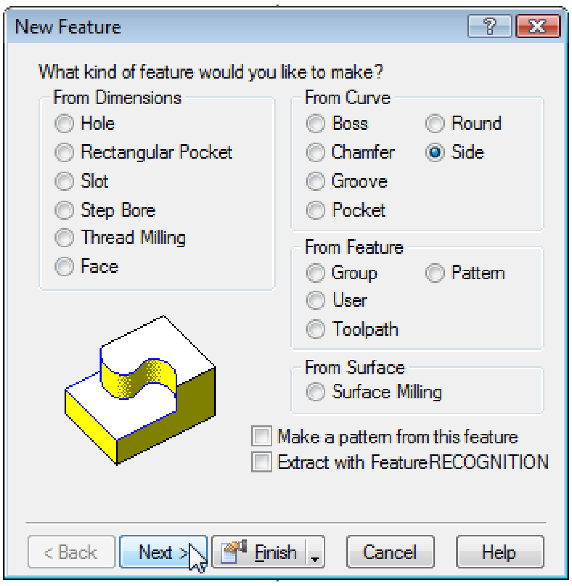
Results for the 3D simulation should be similar to the image shown below.



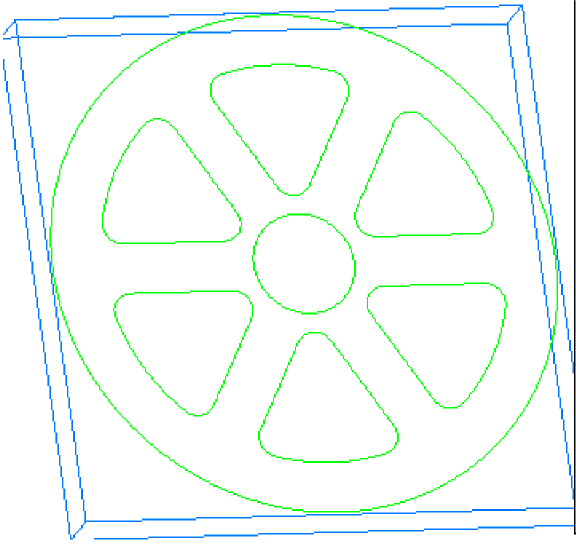
Features From Curves Exercise #2

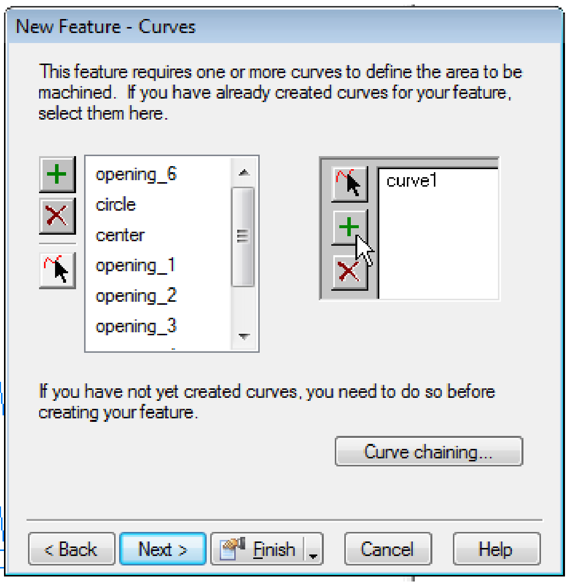


- Open **SideWheel.fm**.

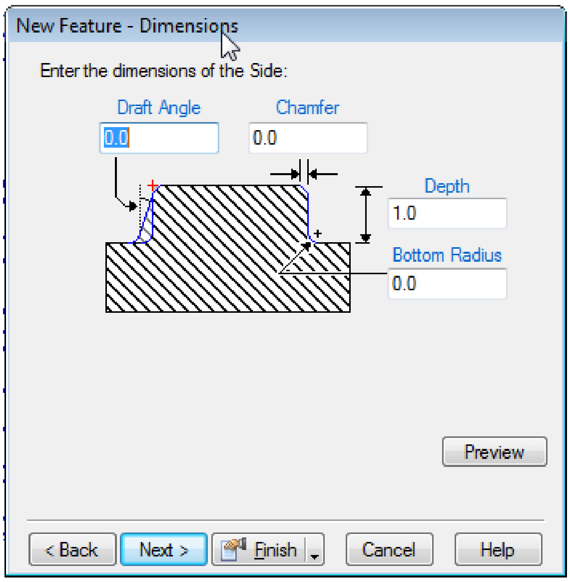


- Open the **New Feature** Wizard.
- Select **Side** then press **Next**.
- Left click and **drag a bounding box** so all curves are selected in red then press the **green plus** sign to add all selected curves to the list then press **Next** three times.



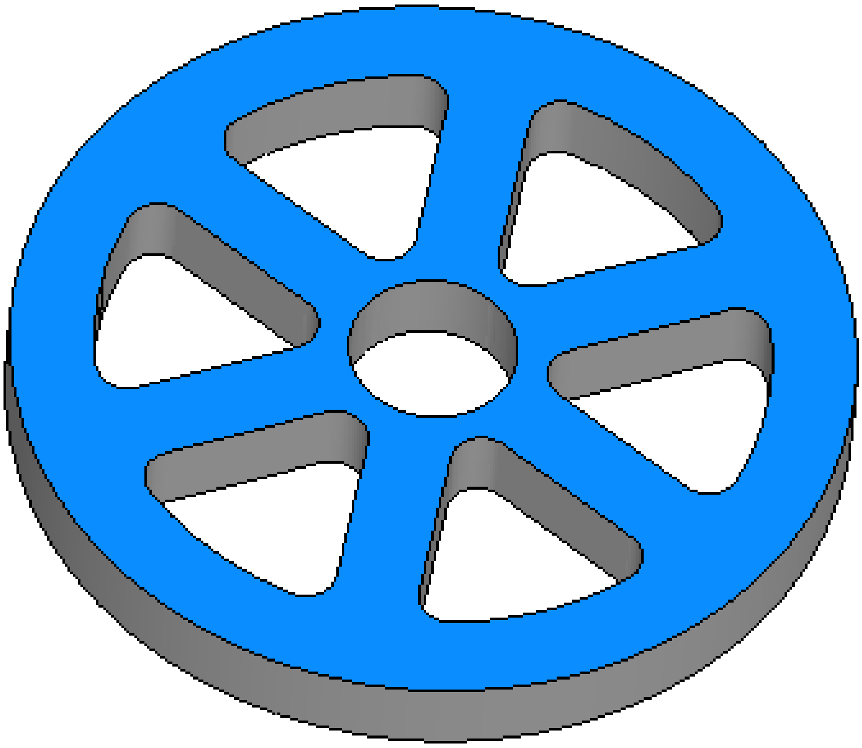


- On the **Dimensions** page enter the **Depth** of **1.0** then press **Finish**.

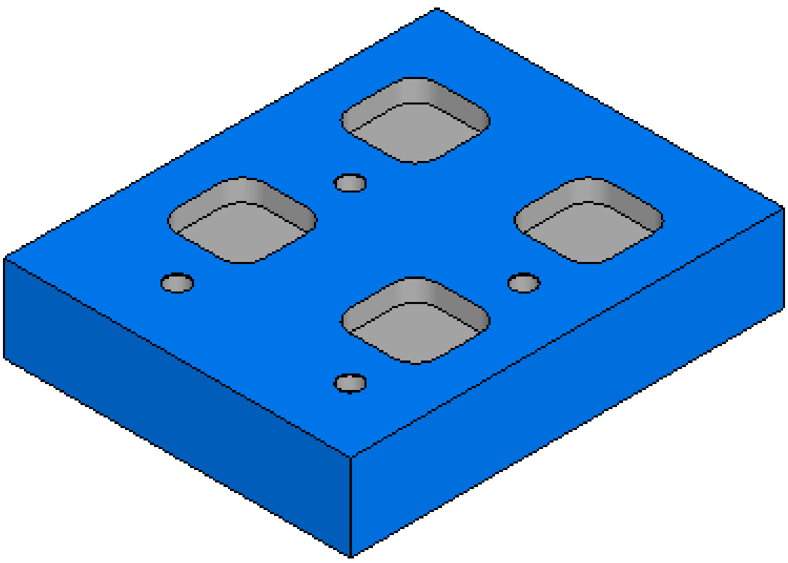


- Run a **3D simulation**.

The simulation results should look similar to those shown below



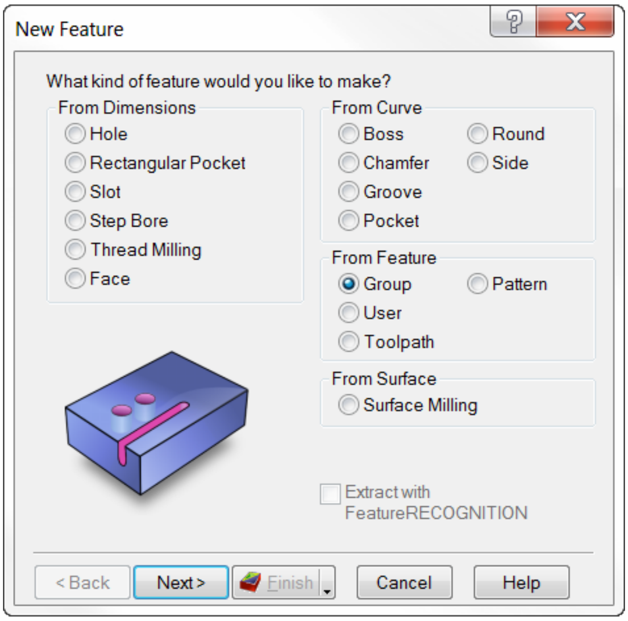
Feature from Feature Exercise #1

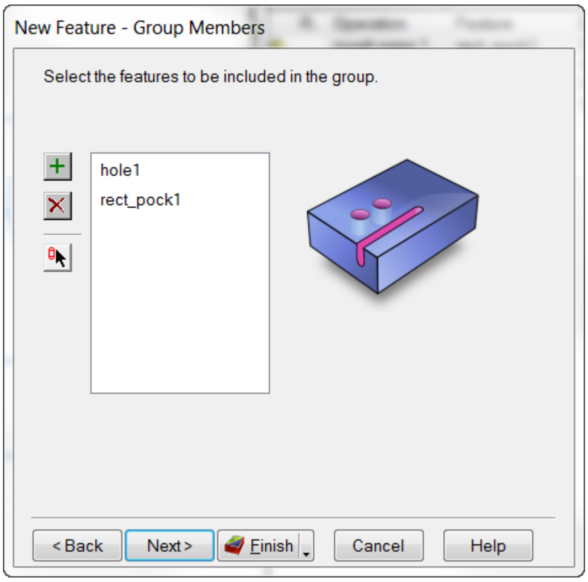


- From the file menu click on **File** then **Open** and select the file named **PatternGroup.fm** then press **Open**

Examine the features in setup 1. Currently there is a single hole and a rectangular pocket. Say for example you would like to create a pattern of the hole/ rectangular pocket combination. 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 that would be more effective.

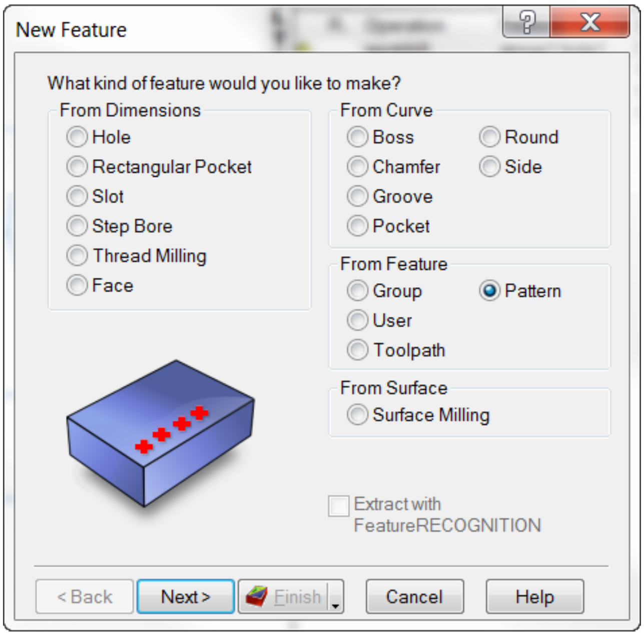
- Open the **New Feature** Wizard in the **From Feature** group select **Group** then press **Next**.

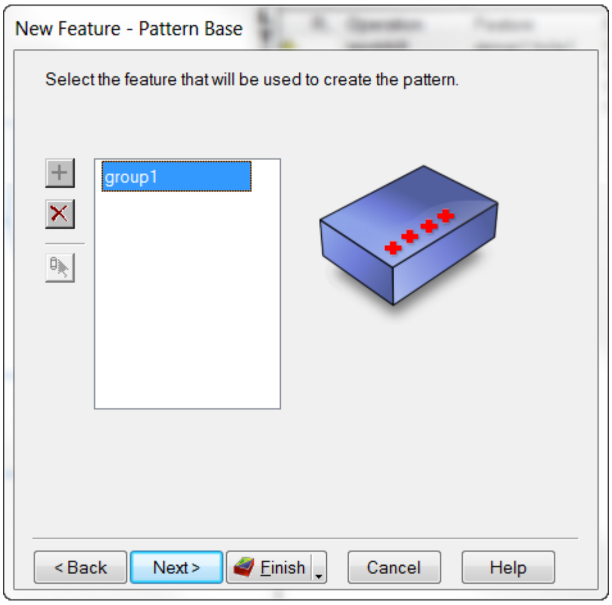




- Add the **hole** and **rectangular pocket** to **Group Members** then press **Finish**.

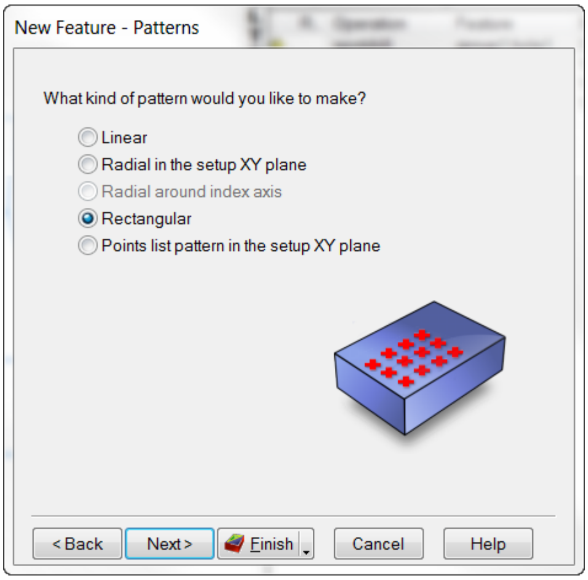
- Open the **New Feature** Wizard select pattern and press **Next**.



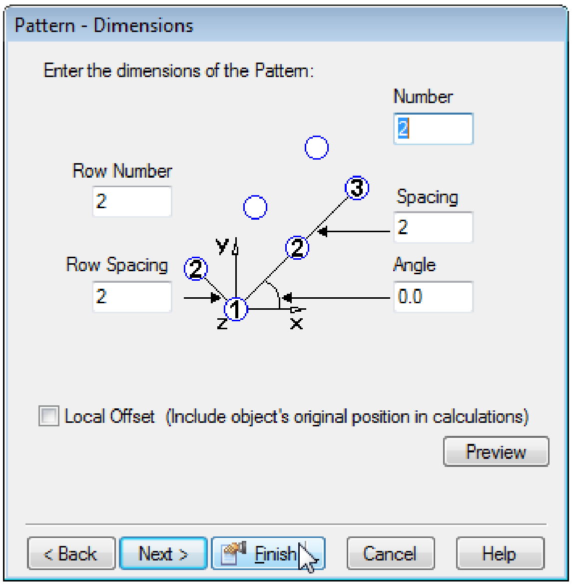


- Select the newly created group then press **Next**.

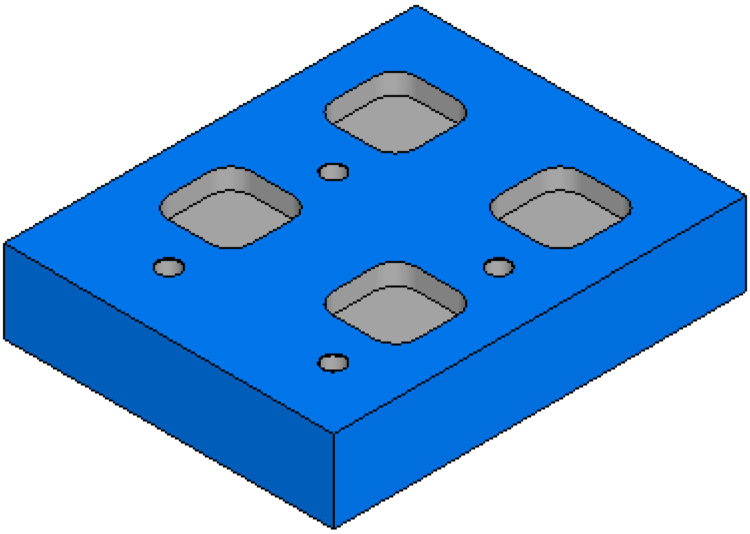
- Select **Rectangular** then press **Next**.



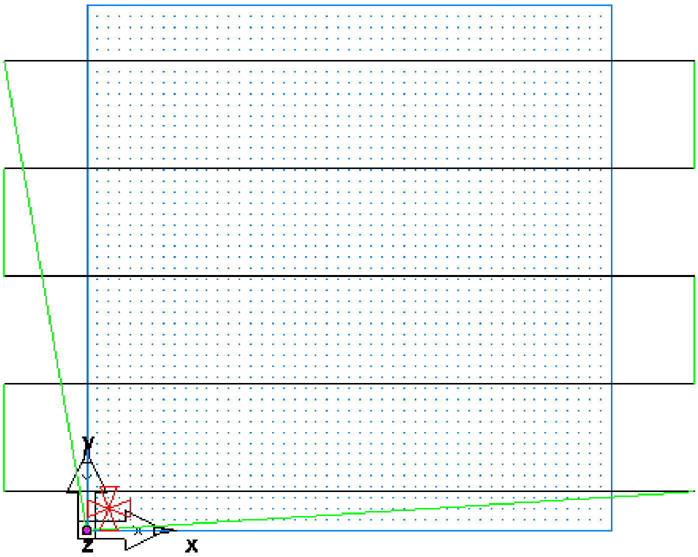
- Enter **2** for both the number of **Rows** and **Columns** and enter **2** for the **Row Spacing** and **Column Spacing**, then press **Finish**.



- From the simulation toolbar select **3D simulation** then press **Play**.



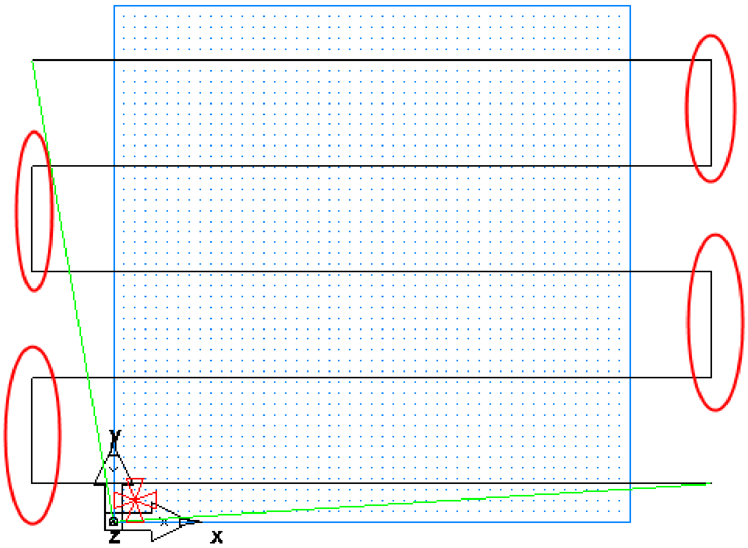
Feature from Feature Exercise #2



- 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 Z=0

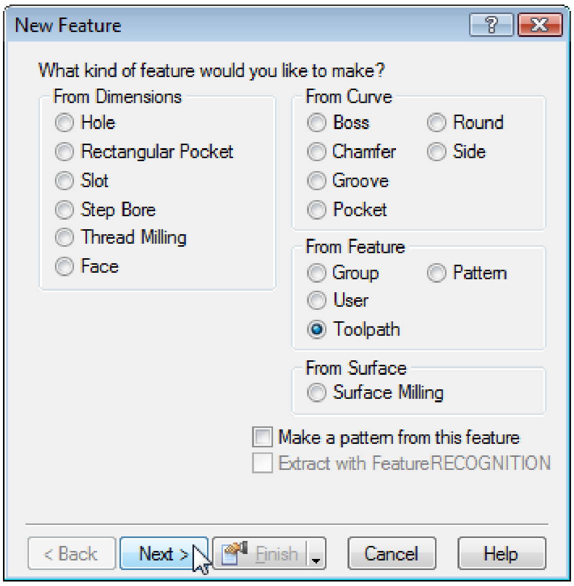
- Run a **Centerline simulation** of the current toolpath. To do this press the centerline simulation button on the toolbar then press the play button

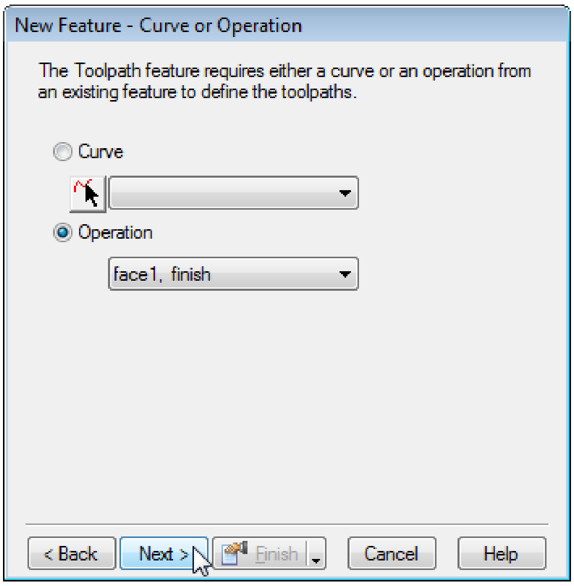


On this facing toolpath there are four undesirable areas where we would like to change the toolpath. For the toolpath representation there are both feed moves and rapid moves. Black toolpath segments represent the feed moves while the green segments represent rapid moves. Currently circled in red are 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.

- Press **Eject** on the simulation toolbar

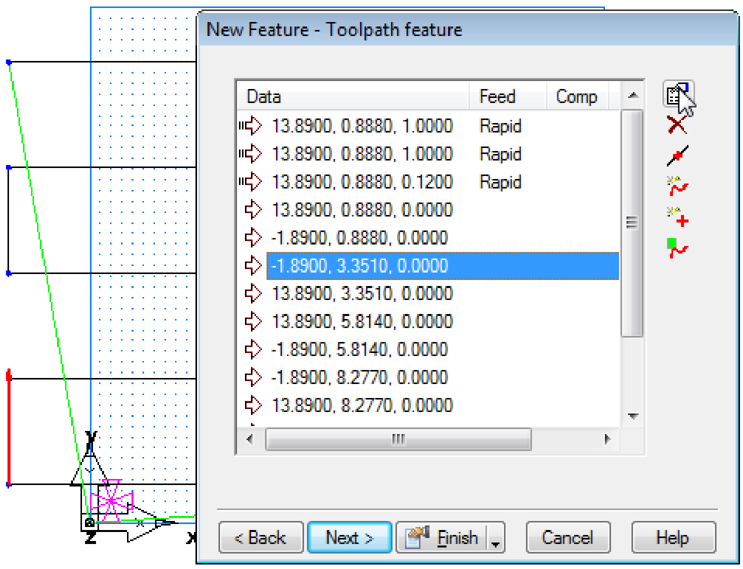
- Open the **New Feature** Wizard and select **Toolpath** in the **From Feature** section then press **Next.**



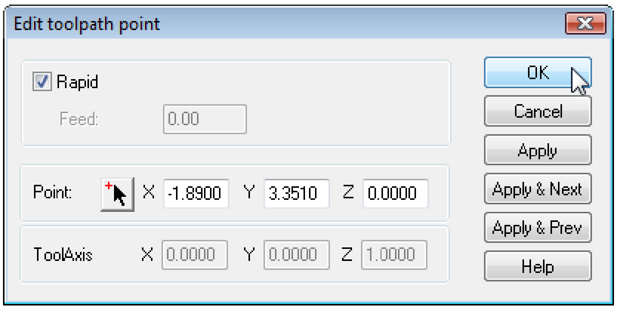


- Press the **Operation** radio button select **face1, finish** then press **Next**.

- Select the first **Y** feed move from the list, then press the **Edit segment** button.



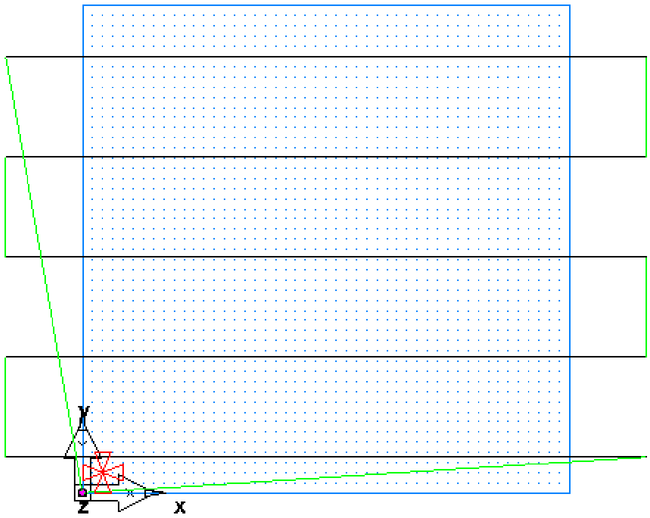
- Check the **Rapid** checkbox then press **OK**.



- Locate the remaining y stepover feed moves and change them to Rapid also then press **Finish**.

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.

- Uncheck **face 1** in the part view (so there are not two facing operations)
- Press **Play** on the **Centerline simulation** and observe the toolpath.



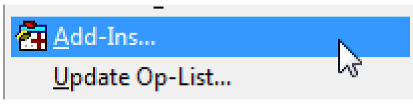
Now notice that the stepover moves have been converted to rapid moves that will reduce the cutting time of the part.

Feature from Feature Exercise #3

Often when creating NC programs it may be desirable to insert G code between operations or to add comments. In this example we will explore the “User Defined Feature” that will give us the ability to add code or comments to their program.

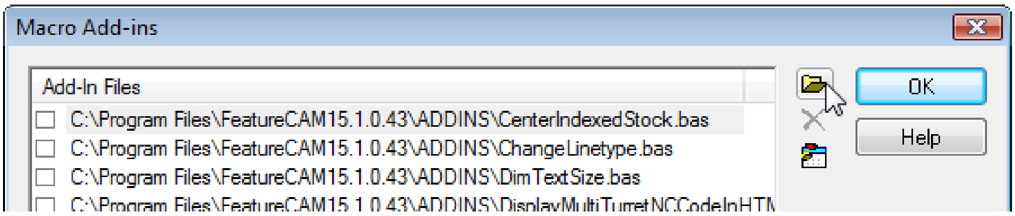
- Click on **File** then **Open** and select **CodeComment.fm** then press **Open**.

In the file the should be a hole and rectangular pocket feature. Observe the operation list. There is a Spotdrill->Drill->Rough Pass 1-> and Finish. Between the rough pass 1 the finish pass we want to put in an optional stop (M01) so the roughing chips can be cleared out for the finish pass.

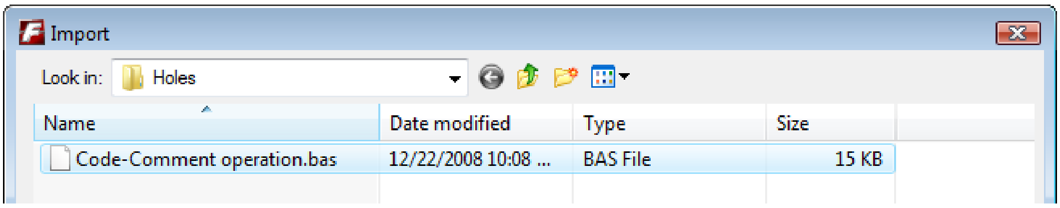


- Click on **Options** then **Add-Ins**.

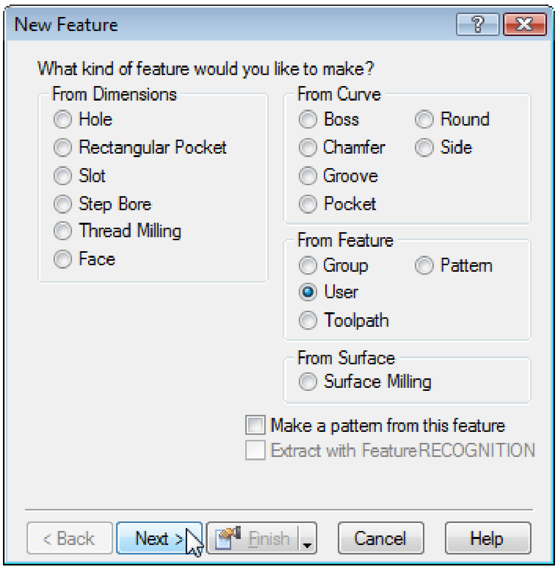
- Click on the **Browse** button.



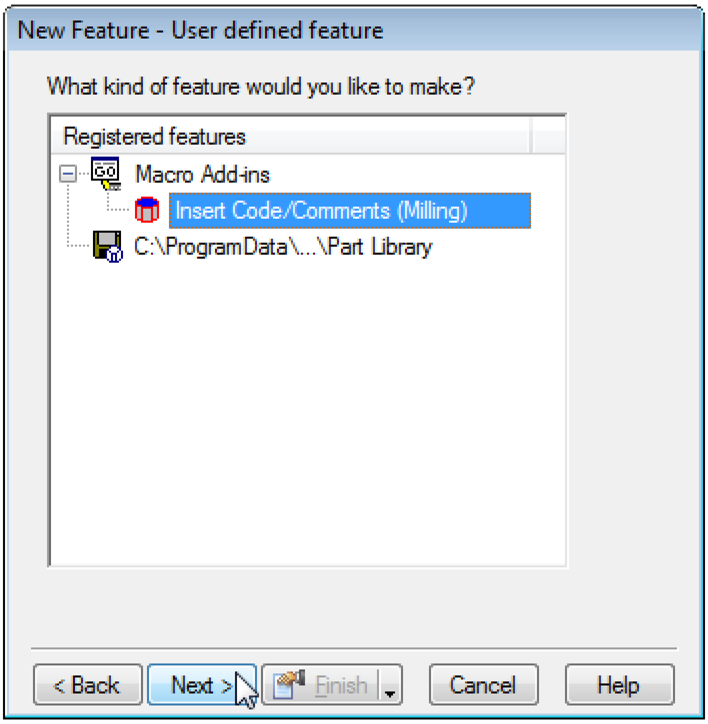
- Browse to the **Code-Comment operation.bas** in the examples folder then press **OK** and **OK** again.



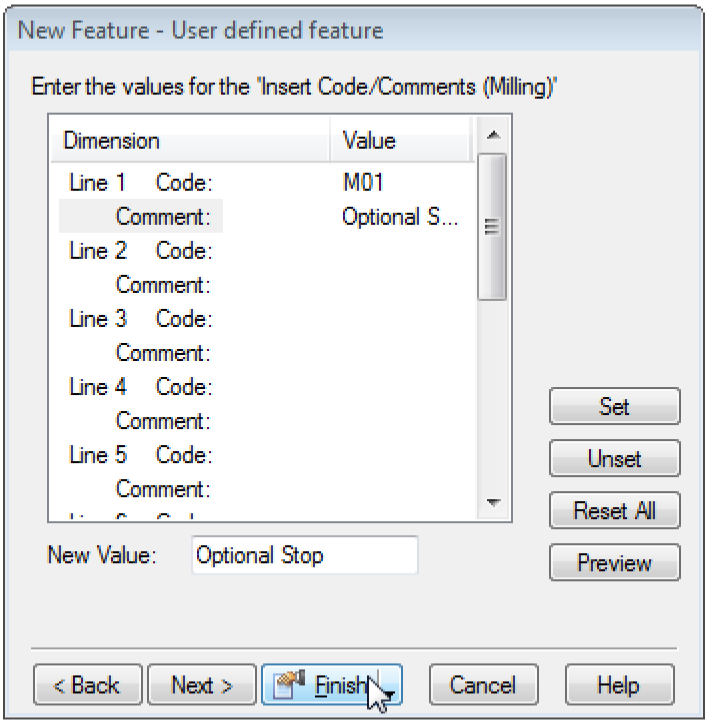
- Open the **New Feature** Wizard and select **User** then press **Next**.



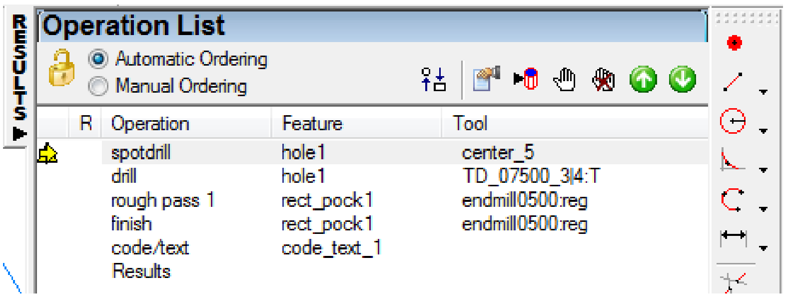
- Select **Insert Code/Comments (Milling)** then press **Next**.



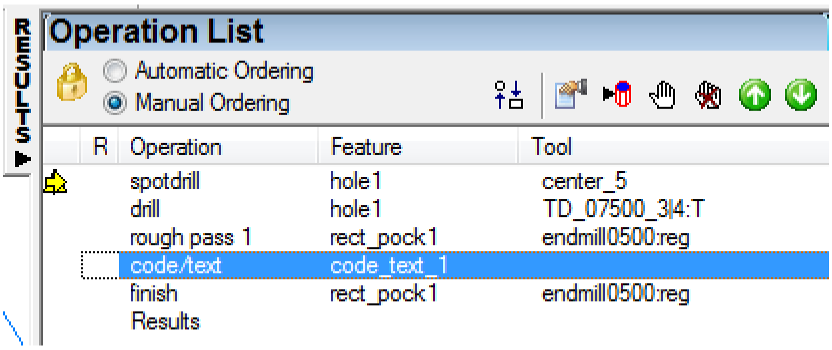
- Click on Line 1 Code and in the New Value field type in "M01" then press the Set button. Click on the word comment below Line 1 and in the New Value field type in optional stop then press set then press Finish.



In the operations list in the results window notice there is a new operation called code/text. This is the code comment operation that was just created



- Left click and drag the code/text feature so that it occurs right after rough pass 1 (a warning message may appear depending on your configuration press OK if it appears)



Now an M01 should appear in the G-Code.

Note: Some posts may not be configured for this functionality to ensure G-Code is properly output check to see that there is {N<SEQ> }{<UDF-TEXT> }{(<UDF-COMMENT>)}<EOB> in Formats/Move/UDF Text.

UDF’s (User Defined Features) are not only available for inserting text into the G-code but can also be configured to create your own customized features. To discover the capabilities of FeatureCAM’s UDFs contact your local salesperson or support representative.

2.5D Advanced Milling

Introduction

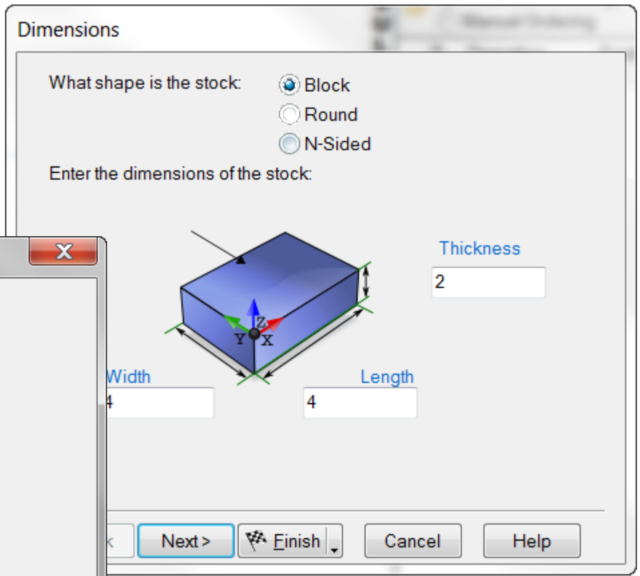
In 2.5D Milling there are more advanced features than milling and drilling, such as 4axis, multiple fixtures, cross section, multiple islands, bosses and side features, etc. The user will learn how tapered walls and other shapes can be machined to emulate 3D milling surfaces using a ball end mill, but are still 2.5D features.

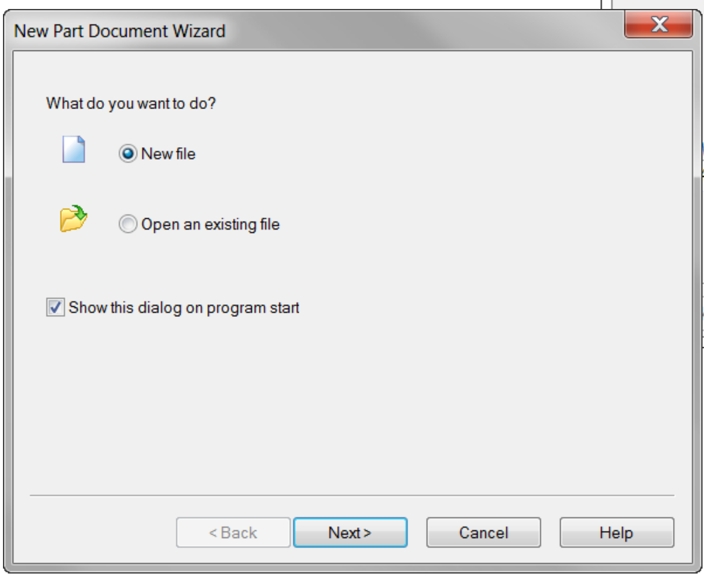
Multiple Curves in One Feature

Boss and Side features begin cutting the feature at the stock boundary. If your part contains more than one boss you must select all of the feature curves for the bosses in one feature. If they are separate features, each will start cutting at the stock boundary thereby removing all of the material instead of leaving the individual shapes of each boss. Side features are the same. A pocket with one or more islands (a shape that the tool cuts around and leaves that shape inside the boundary of the pocket) has an island button to select the curves for the island, or islands, and then the tool will cut both the pocket shape and the island.

Multiple Bosses

- Open a new **Milling** document, select **Inches**.



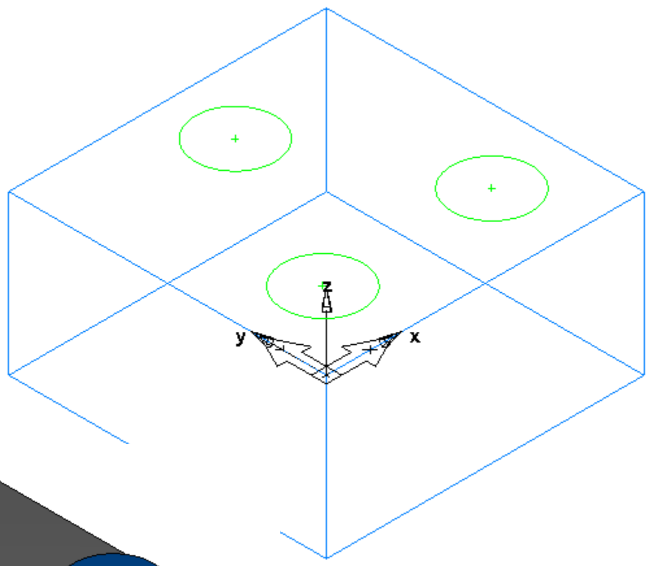


- Establish the stock size and Block shape (4 x 4 x 2) or use Sample:
  Boss-Side-Pocket.fm.

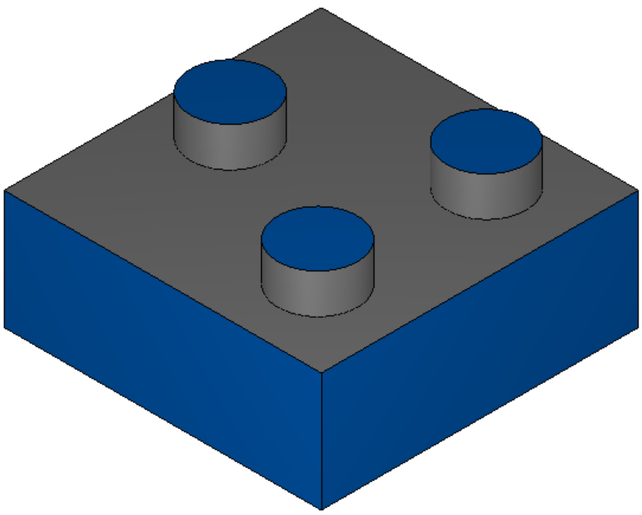
- Draw the **circles** for the multiple Bosses (.500 radius Circles as a triangle as illustrated). These are the multiple shapes you intend to leave on the top of the stock after machining.

Shape curves do not have to be round but they cannot touch, be inside each other or be higher than the stock, they will have a common bottom depth although they can be at different heights. If they are different heights draft angles may not be selected, chamfers will have to be separate chamfer features but bottom radiuses will be common to all.

- Select **New Feature>**Boss from Curve.
- ALL curves must be included in this one feature
- Enter the **height** to be **0.500.**



- Click **Finish** and run a **3D simulation**.

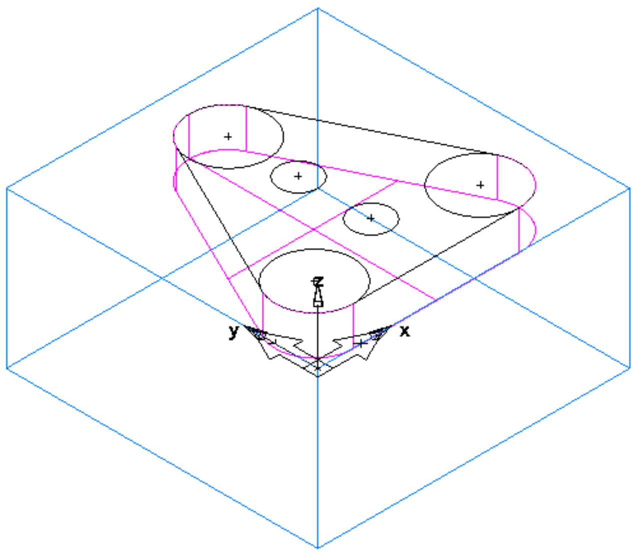


Pocket with Islands

- Open a new **Milling** document and select **Inches**. Establish the block stock size (**4** x **4** x **2**) alternatively you may open the Sample: **Boss-Side-Pocket.fm**.

- Draw **geometry** to create a triangular pocket shape with .500 radius corners, roughly as shown.

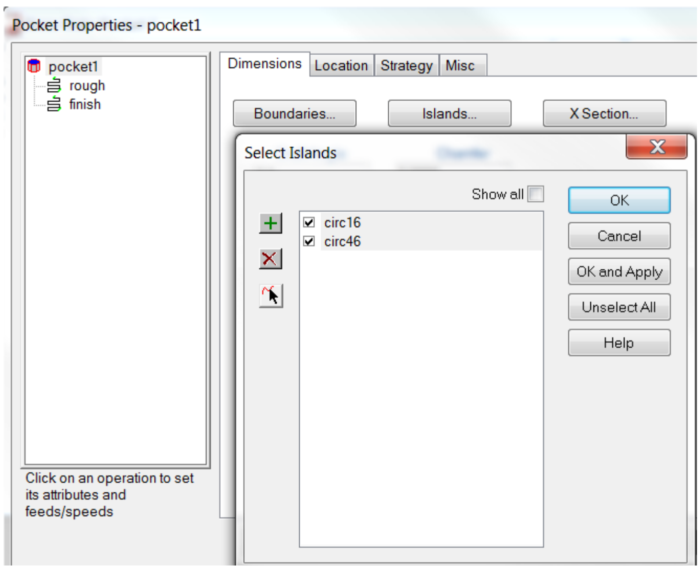
- **Chain** a **closed curve** from the triangular shape. Add two .250 radius circles, roughly as shown, inside the pocket.



The islands can be any shape and have a common depth. They can have tops at different heights, but may not be above the top of the pocket. If they are different heights, draft angles may not be selected. Also, in the case of different heights, chamfers need to be applied to the pocket, and each island, as separate chamfer operations. If you add bottom radiuses they will be common to all.

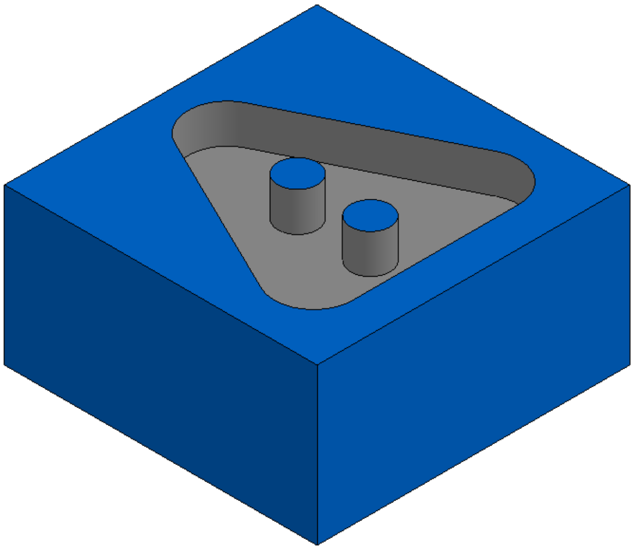
- Click on **New Feature** > **Pocket From Curve** >select the triangular curve (.500 depth) **Finish**, then on the **Pocket Properties** page, **Dimensions** tab, select **Islands** and include the two circles whose shape will be left in the pocket after machining.

- Click **OK and Apply.**

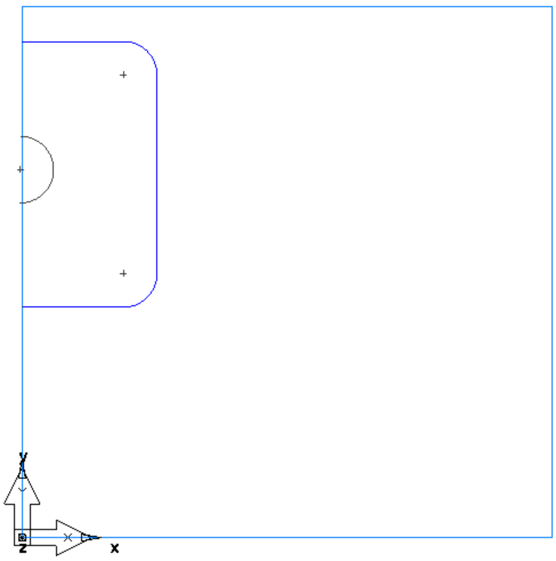


All island curves or circles must be included in the feature.

- Click **OK** and **3D simulation**.



Multiple side features

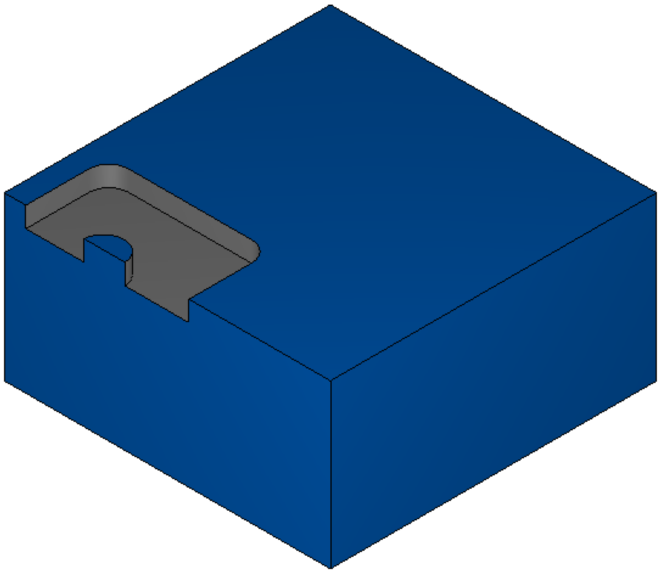


- Open a new **Milling** document and select **Inches**. Establish the stock size and block shape (**4** x **4** x **2**) or you may open the Sample: **Boss-Side-Pocket.fm**.
- Draw a "U" shape with a 2.000 opening and .250 corner radiuses, roughly as shown, and chain it as an open curve. Add a .250 radius half circle, centered in the opening, and clip the left side. Click **New feature** > **Side From Curve** and select **both** the "U" shaped curve and the half circle.

Side features can cut on either side of the curve so it is important that the **machining side** tab in the properties is checked to make sure the arrow (which indicates the side to be cut for each shape) is pointing toward the **other** side curve. The arrows do not have to point at each other but they must point at the other side feature. Although there is no island, so to speak, in a side feature, multiple sides will have the same effect.

- Click **Finish**.

- Set the **depth** at .250 and run a **3D Simulation**.



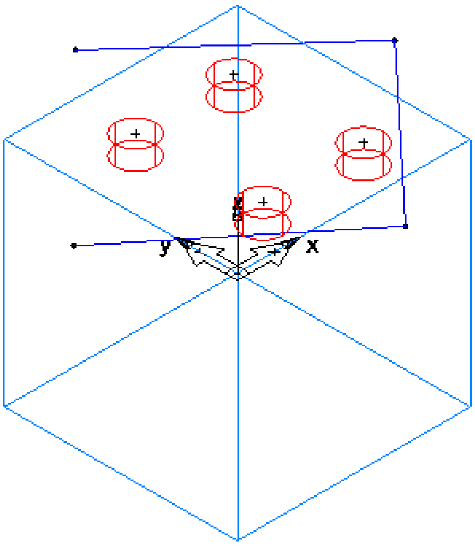
Multiple Plunge Point Curve

When machining a part, it is usually more desirable to plunge the tool to depth off of the material, wherever possible. For example, a part with four bosses that needs to have the stock removed all the way to the center. A Boss feature is designed to plunge off the part, but somewhere in the tool path it may also plunge into the material to reduce the travel distance. However, center cutting end mills, though designed to cut to the center, may find this difficult. Because of this, Featurecam defaults to using 50% of feed rate when plunging.

In the **Boss** feature, under the **Milling** tab, there is a selection called **Plunge Points**.

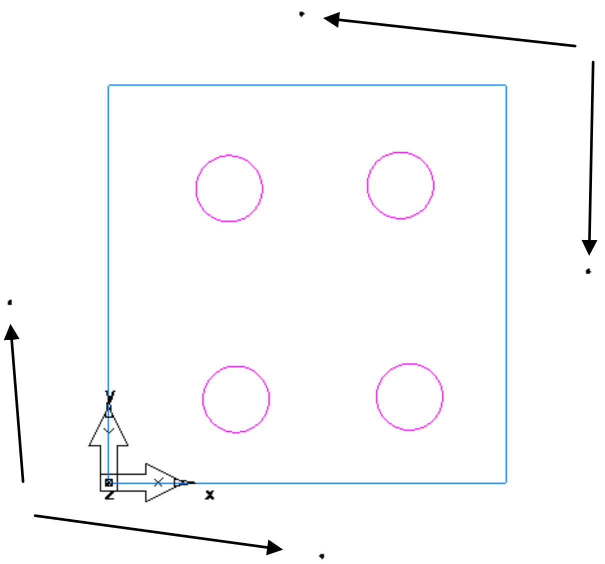
- Open the **Plunge Point Curve.fm** file. **Hide all Curves** and **Geometry**. Double click on one of the Bosses in the graphics window to open the **Boss Properties**.

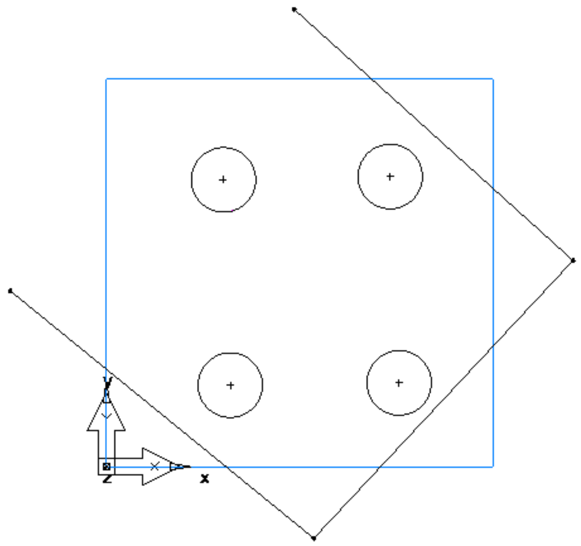
- Click on the **Rough** operation and the **Milling** tab. Select **Plunge Point** and **Unset**. Now run a **3D simulation**.



Using a 1.5 diameter end mill, notice how the tool plunges once in the middle of the Stock. We can use a curve to force the tool to plunge in certain locations instead of the stock.

- To create this type of curve, pick **Point** from the geometry toolbar. Place **multiple point locations** in the graphics window at locations where you desire the tool to plunge, as shown to the right. These multiple points force the tool to plunge only at these four locations, not in the stock.

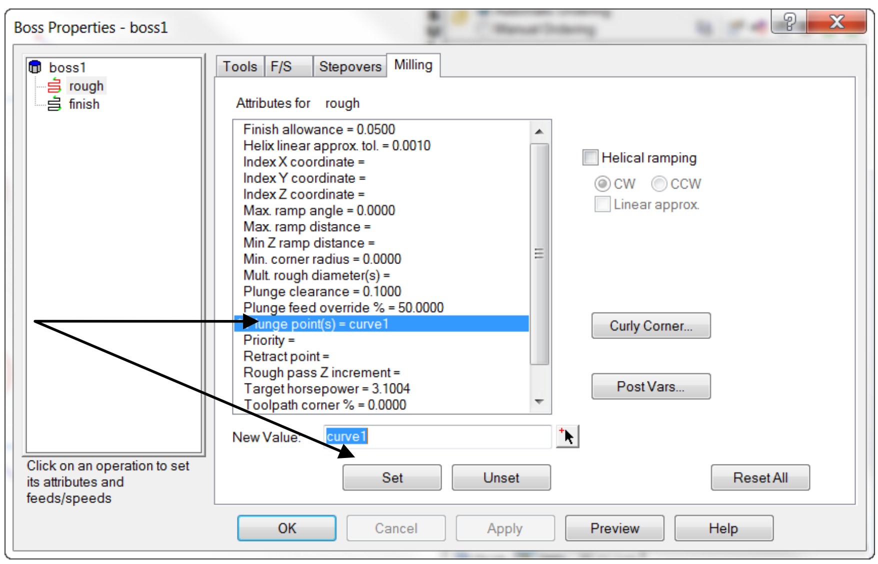




- Connect the dots with **geometry** using **Connected Lines** and then **chain** as an **open curve**.

Note: When you create the curve, be sure to notice exactly how it is **named**. For example: curve1.

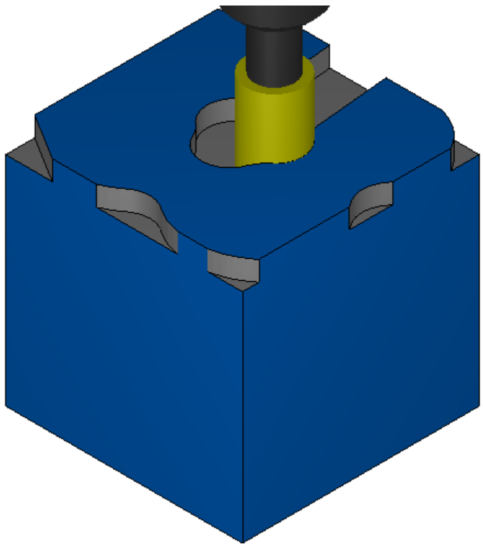
- Open the **Boss Properties**> **rough**>**Milling** tab, and select **Plunge Points** then in the **New Value** field type in the **exact name of your curve**. Example: “curve1”.



- Click **Set**, **Apply**, and **OK**.

- Run a **3D simulation** and notice that the rough tool feeds to the center, but does not plunge in the center.

Note: The curve you create for use in **Plunge Points** must have sharp corners at the desired plunge point locations. As seen in the exercise, the easiest way to create such a curve is to use **Connected Lines**.



Cross Section (X section); Side, Pocket and Boss Features.

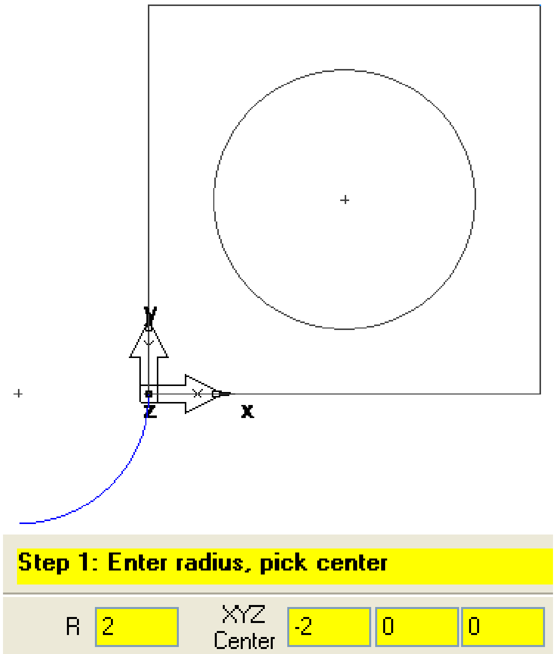
When creating Boss, Pocket and Side features there is an option of selecting **X section**. This allows you to apply a shape, other than an angle, to the walls. FeatureCAM will even select a ball end mill where necessary. Surface or solid models are NOT required, although the end result is similar to a 3D finished surface.

Cross Section (X section)

- Create a new document, **Milling**, and **Inches**. Select **Block** and use **Stock Dimensions** (**6** x **6** x **6**). Alternately, you may open the file: **Cross X Section.fm**

- Create a Pocket by drawing a 2.0 radius circle, centered on top, XY plane of the stock. Click **New Feature**>**Pocket From Curve**, a 1.0 Depth and **Finish**.

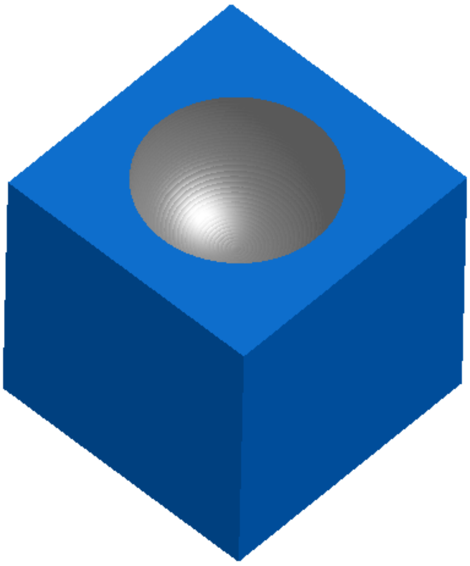
- For use in Cross (X Section), draw a **2.0 radius circle**, using **Center, Radius** from the **Geometry** toolbar. Enter X=-2.0 and Y=0, Z=0. Using geometry, **draw a horizontal and**



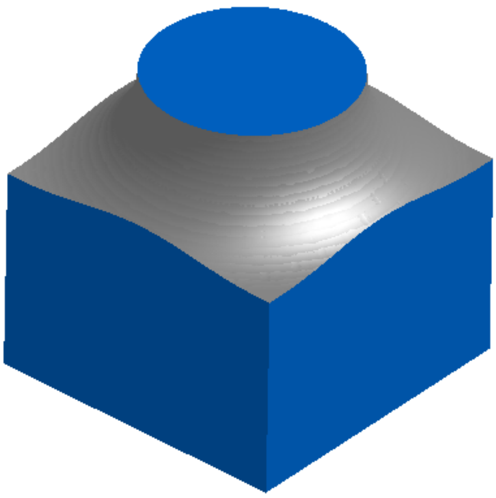
vertical line through the center of this new circle. These lines will be used for trimming. **Trim the geometry** until you have the lower right quadrant of the circle remaining as shown to the right.

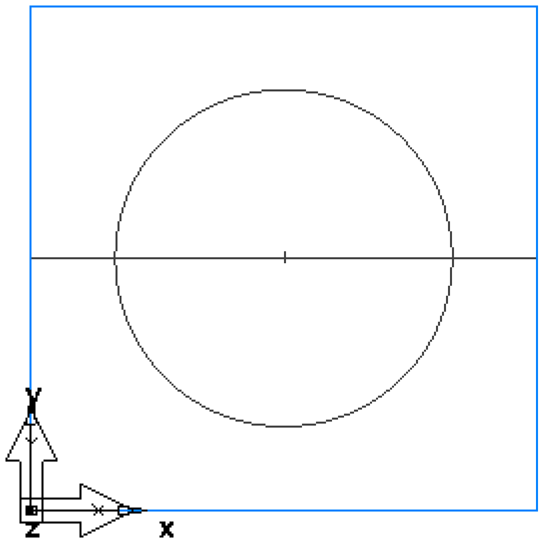
This arc could be used as the wall shape of the pocket.

Rules for an X-Section Curve

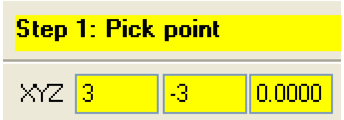


- When using geometry for your cross section shape it does not have to be round. It can be other shapes as well.
- Once you start drawing away from XYZ=0 you may **not** draw back toward the start point in the X or Y direction. This would create an undercut.
- A straight line in the X or Y is ok as long 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 can **NOT** be used.



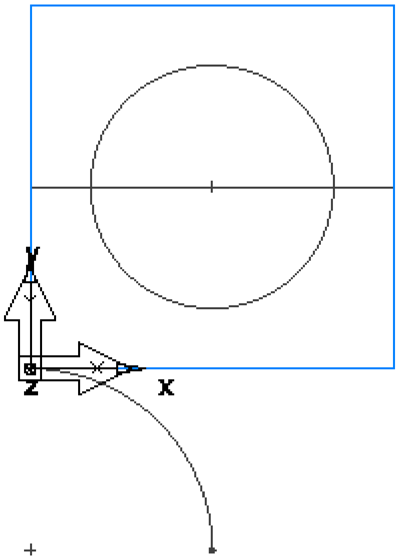


- **Open** the file **Cross (X) Section.fm**. You should only see this line and circle, as shown to the right.



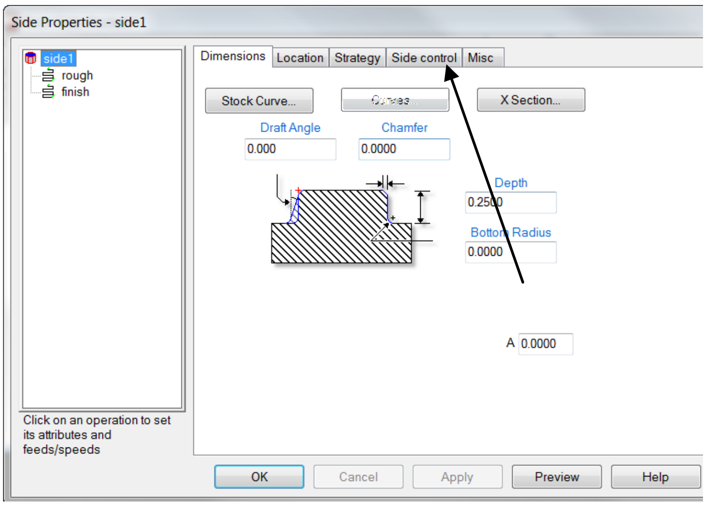
- Create a **point** located at X=3, Y=-3, Z=0.

Using the **Geometry** toolbar and the **Arc** menu, create a **3 inch radius arc** from **2 points, Radius**. Click once at XYZ=0 and click a second time at the **point** location (point that was just created). This will create a 3 inch arc as illustrated below.



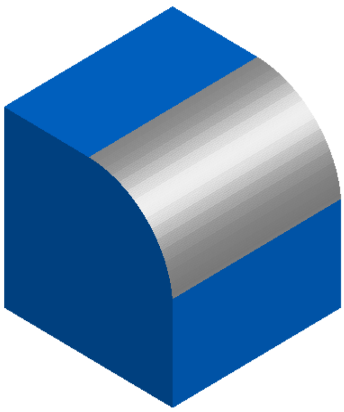
- Click **New Feature** > **Side From Curve.**

- Click the **line across the middle** of the stock and add it to the **Curves** list (depth is not important). **Finish**.



- Click the **X-Section** button and **add the arc**. Change the machining side if necessary from the **Side Control** tab.

- Run a **3D simulation**.



Note: It may be necessary to add a short straight extension on the bottom end of the X section curve so as to carry the ball end mill flush with the stock face.

2.5D Plunge Milling

Open a new **Milling** document. Set the stock as **Block**, Enter 10 inches for **Length** and **Width**, and 4” for **Thickness**. **Finish** and **OK.**

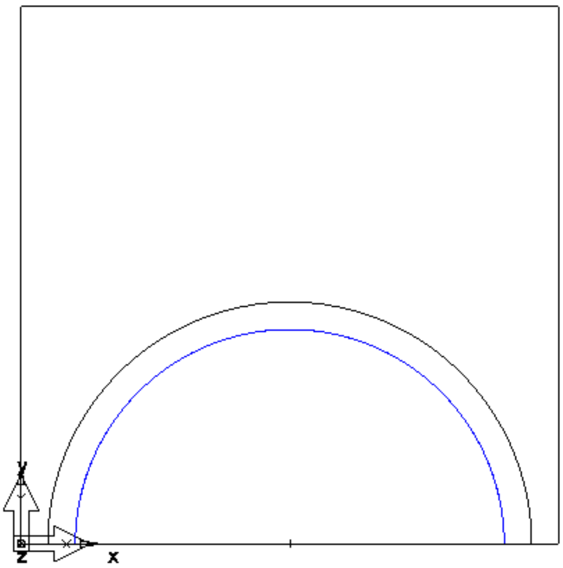
Now we will need to create two curves in our new milling document.

With the UCS and Setup on the Top and Lower Left of the part create a point at (X.5, Y0, Z0), and a second point at (X9.5, Y0, Z0). Draw a 4.5 inch radius, from **Geometry** by selecting **arc** from **2 points, Radius** and attach to the two points.

Important: select the point we created first, then the second point to create the arc properly. Chain the arc as an **open curve**.

This curve is the finish curve, the finished shape, or profile that you desire.

- Open the **Curve Wizard**. Select **Curve from curve**, **Offset**. **Next.**
- Set the **Offset** value to .500, and to the **Left**. This represents the radius of the tool **(1.000 dia.)** to be used for plunging. **Finish.**



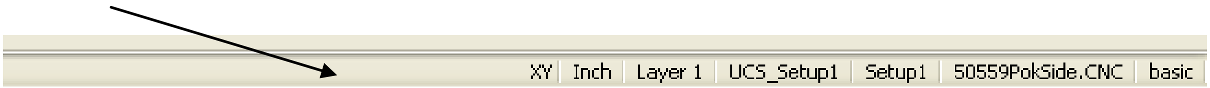
You can **add to the offset** value, any **finish allowance** if you desire, for an additional finish pass. You can do this to remove the scallops left by the plunge mill tool stepover.

The new curve (the offset curve) is going to be the **center of the plunge milling cutter** and the milling cutter will follow this curve.

Note: The Curves created can have any number in their name.

In order to get the tool to plunge on the curve in an equal stepover amount we have to use SCL Code to achieve this. The following instructions will introduce you to this concept.

- Hold the **Control** key and **Right Click** your mouse on the **Status Bar** at the bottom of the FeatureCAM user interface and check the option to **Enable SCL dialog.**

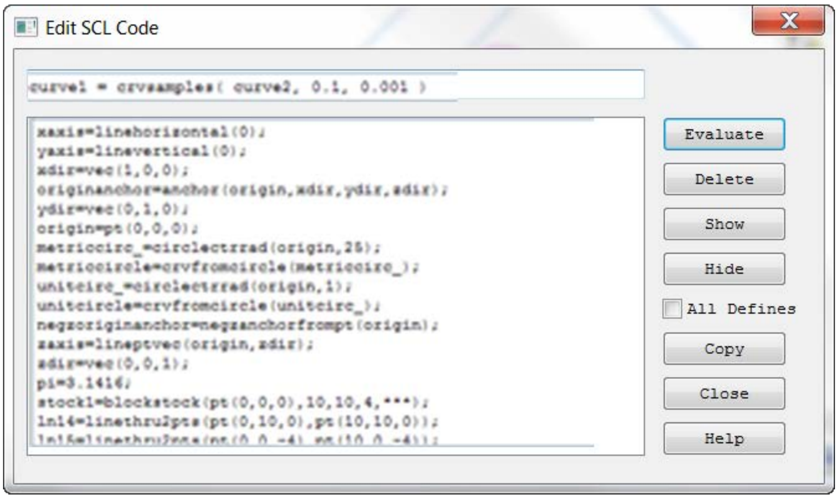


- Click on **Edit** > **SCL Code** > Check the **All Defines** checkbox.

- **Copy** the bolded code below (minus quotations):

  "**curve1 = crvsamples(curve2, 0.1, 0.001)**" and **paste** into the **Edit SCL Code** top field.

  Note: If you fail to check All Defines **first** you will have to close and reopen SCL. Also **know YOUR curve** numbers before doing SCL.



What does this Code Mean?

- Curve 1 in the code is finish curve number, change to whatever **YOUR** finish curve name is.

- Curve 2 in the code is offset curve number (toolpath), change to whatever **YOUR** curve (offset curve) name is.

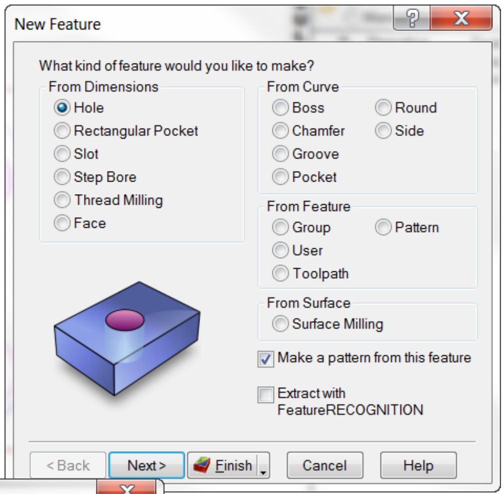
- 0.1 is stepover for plunges (You may change).

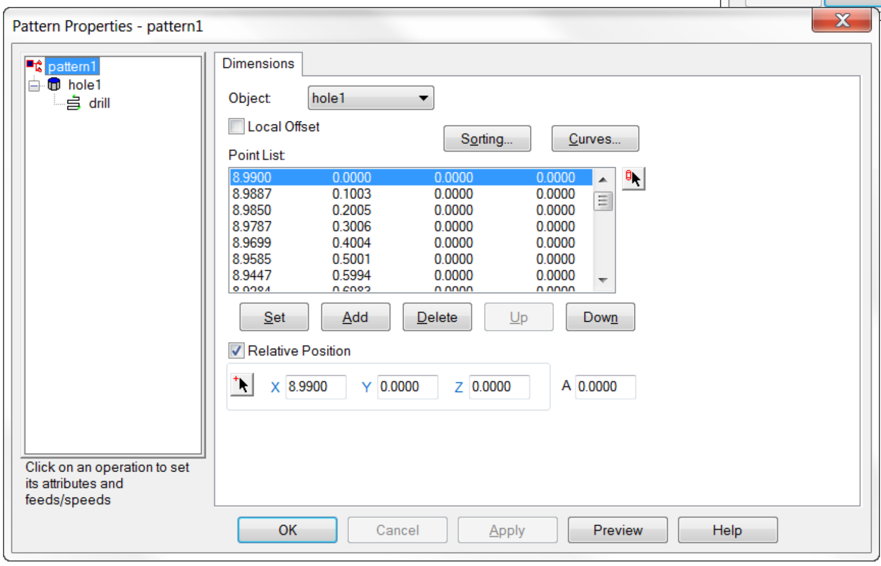
- .001 is tolerance for following Curve 2 (You may change).

Click **Evaluate** and then **Close**.

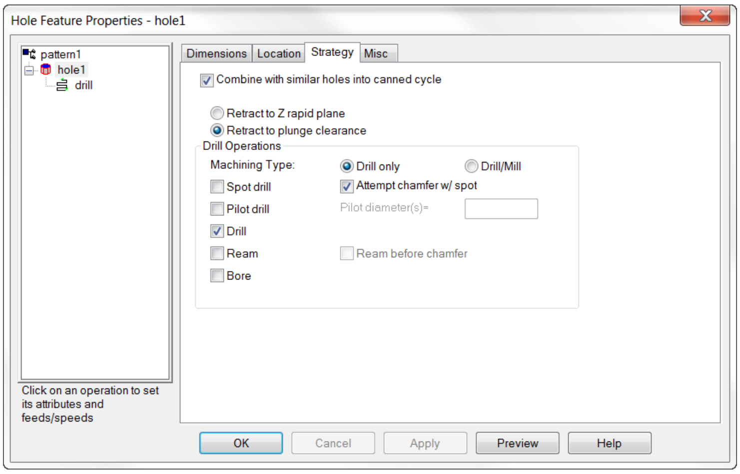
Important: You may have noticed when you clicked evaluation that the finish curve moved to the toolpath curve and became the same size, both curves are now in the second or offset position, it kept the original curve number from the first position and that is the curve number that must be selected, it may already be in the curve selection window when you click the curve button.

- Create **New Feature** > **Hole** and **Make a pattern from this feature** (Points Pattern). **Next**.
- Set **Depth** 2.0, check **Through**, and **Diameter** 1.0. **Next**.
- Select **Points list pattern in the setup XY plane**. Click **Finish**



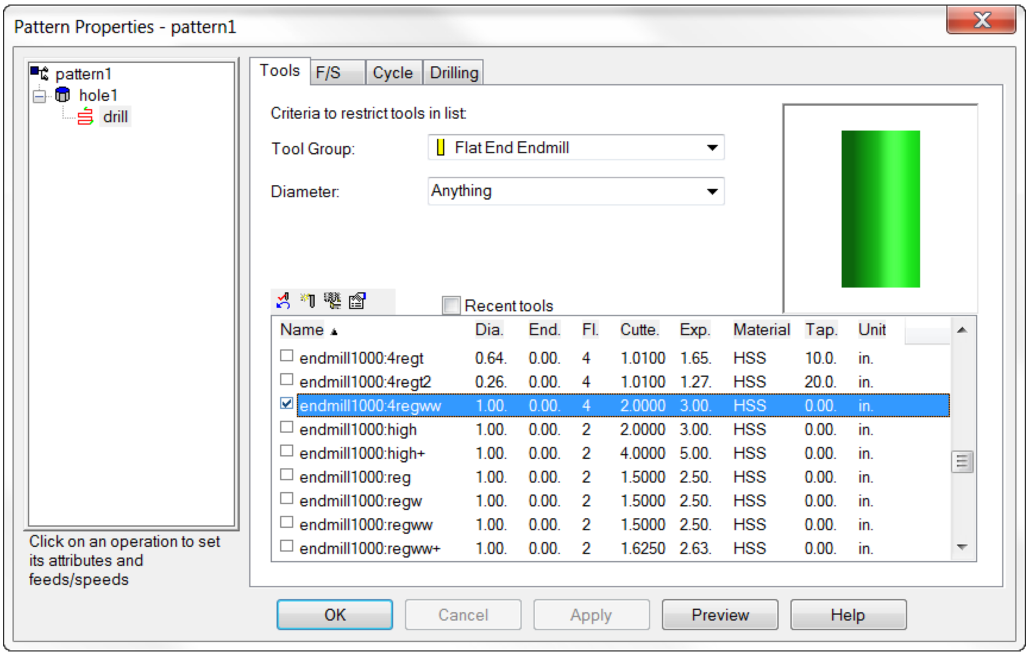


- Click on the **Curves…** button in the **Pattern Properties** dialog and **select the curve you offset earlier**.
- **OK and Apply.**



- **Uncheck Spot Drill** in the **Strategy** tab.

- Change the tool (matching the offset curve radius) to the appropriate milling cutter: **Tool Group**, pick **End Mill**, then for **Diameter** select **1.00** inch.

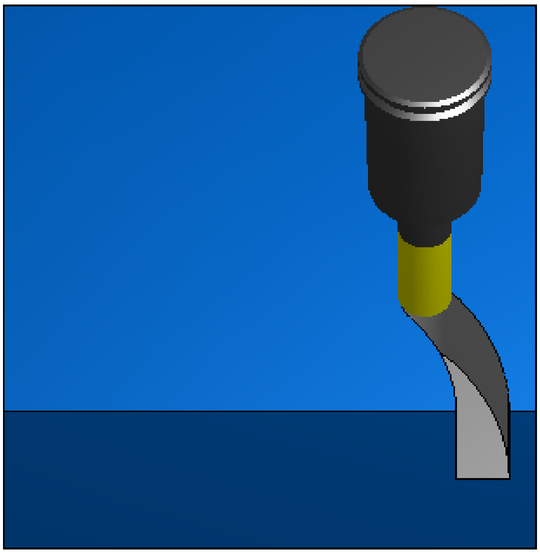


- In the **Strategy** tab of the **Hole Feature Properties**, check “**Combine with similar holes into a canned cycle**”.

- **Apply** and **OK.**
  Note: The canned cycle setting helps reduces the retract distance. It will now retract to the **plunge clearance** value on each plunge cut.

Grooving

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.



- Create a **New File**. Stock dimensions (6” X 6”, Length and Width X 2” Thick). **Finish** and **OK**.

- Create a **Point** (X 1.0, Y 0.0, and Z 0.0) and another **Point** at (5.0, 0, 0).

- Create an **Arc from Radius, 2 Pts**.
  Enter a **2 inch radius** and select the first point you created, then the second.

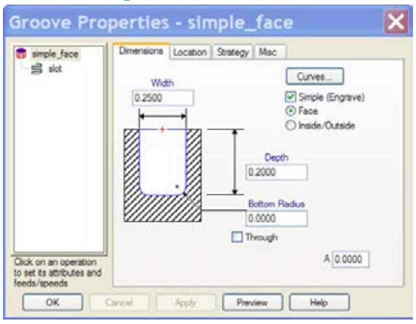
- Create a **New Feature** > **Groove > From Curve** and select the circle. **Next** 3 times.

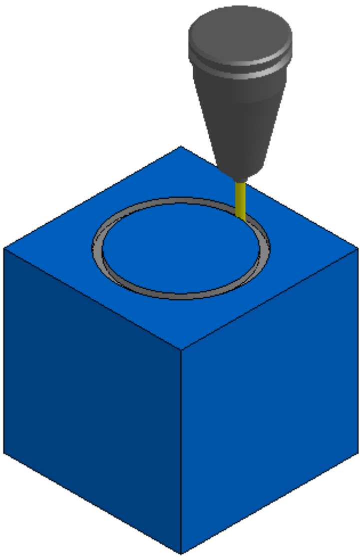
- In the **Dimensions** dialog, enter the **Width** .25, **Depth** 0.2, and check **Face** and **Simple (Engrave)**. **Finish** and **OK**. FeatureCAM picks a tool the same diameter as the groove width and plunges to the bottom, cuts around the circle once, then retracts.

  Note: There may be more passes, depending on depth, but there is no radial movement of the tool. The walls are cut to size.

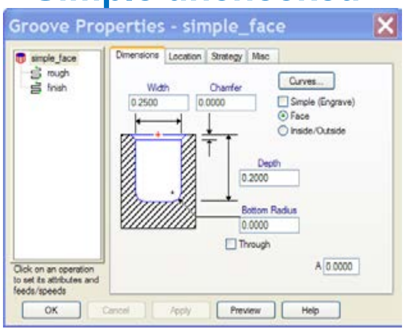
In a **Face Simple (Engrave)** groove the **cutting tool is the same width as the groove** and makes one pass. **When Simple (Engrave) is unchecked** a tool smaller than the width of the groove is used, and makes a rough and finish pass. This method is usually used when tolerance and finish are more critical.

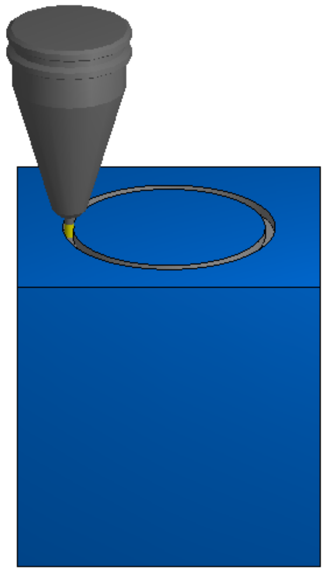
Simple checked





Simple unchecked

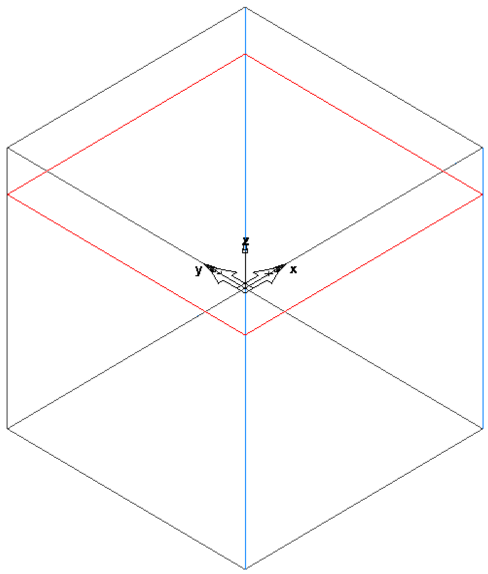


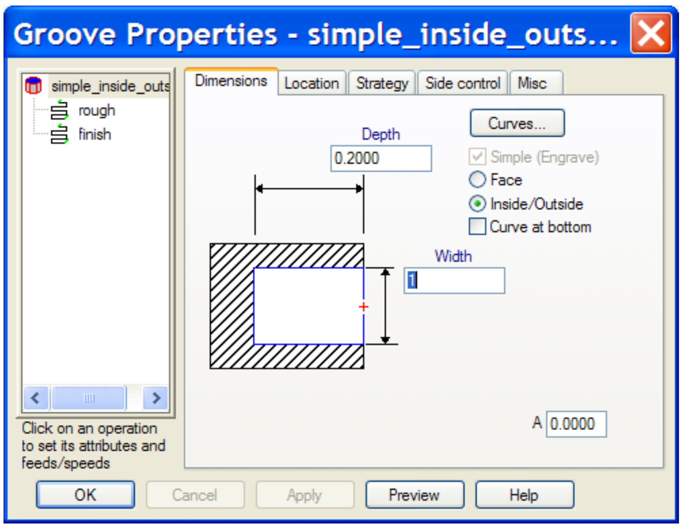


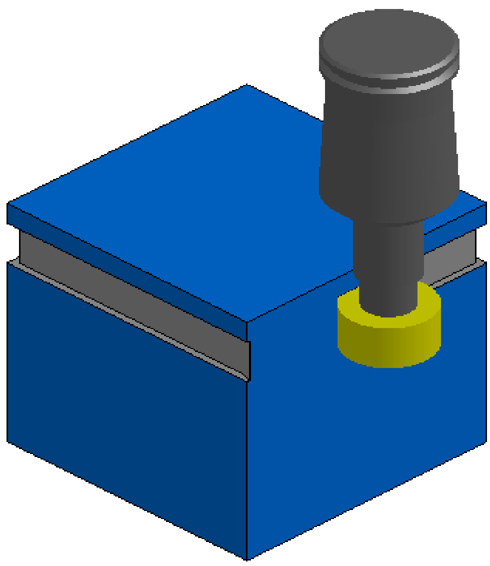
An **Inside/Outside groove** may be created on the **outside** of the part, or the **inside of vertical walls** using a side-cutting tool. **When the tool is the same width as the groove**, one rough and one finish pass is made. A single finish pass may also be used.

When the tool is smaller than the width of the groove, the first pass places the center of the tool width on the curve, next makes a pass using the top of the cutter to cut the top of the groove and next pass using the bottom of the cutter to cut the bottom of the groove, this method is usually used when width tolerance and finish is critical. The geometry and curve for the groove must be on vertical walls, outside or inside and the curve may be at the top or bottom (radially), remember the opening of the groove is facing out (or in) so when it says curve at bottom it means radially of the tool (check box) because that is the depth of the groove now and the depth is not measured in the Z direction.

Inside/Outside groove







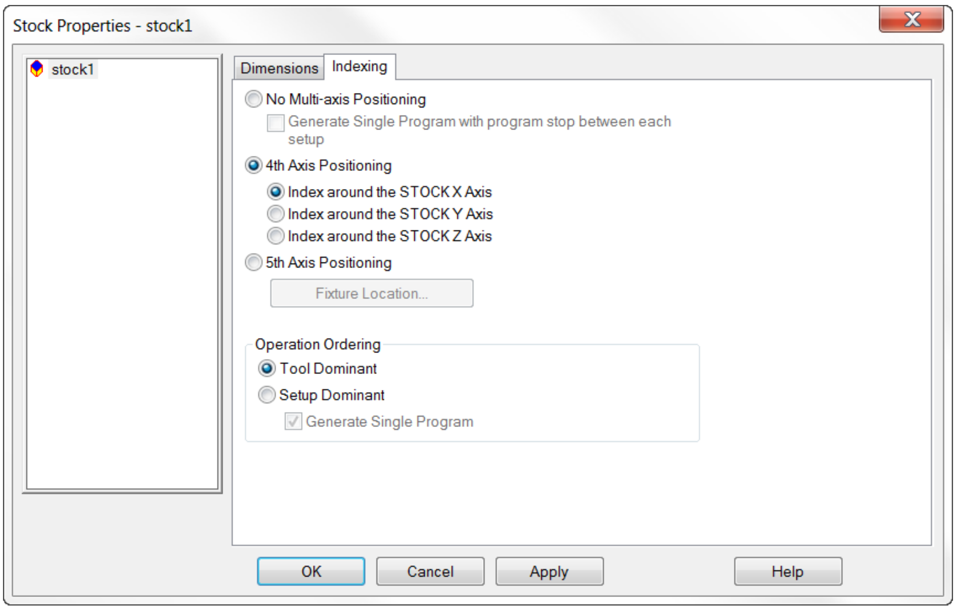
4th Axis Indexing and Wrapping:

Introduction

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.

- The **Stock Axis** is different from UCS or Setup Axes. It corresponds to machine zero, along with the axis of rotation for an indexer or rotary table.

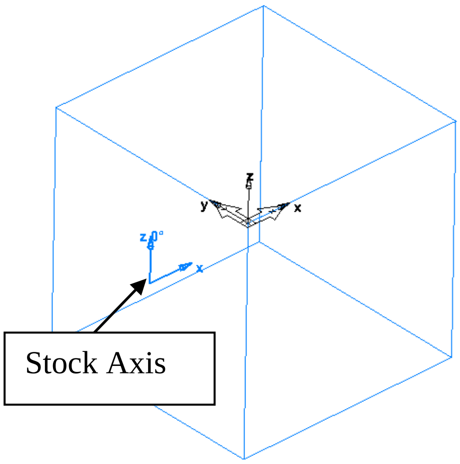
- **The Stock Axis is not normally displayed** but to view the stock axis click **View** and select **Show/stock axis** from the **Show** menu. It is displayed as two vectors (blue). One shows the axis of rotation (X or Y) and the other indicates the orientation of a 0 deg rotation (pointing at the spindle). **You must align your part center of rotation to the rotation axis of the *STOCK AXIS.***



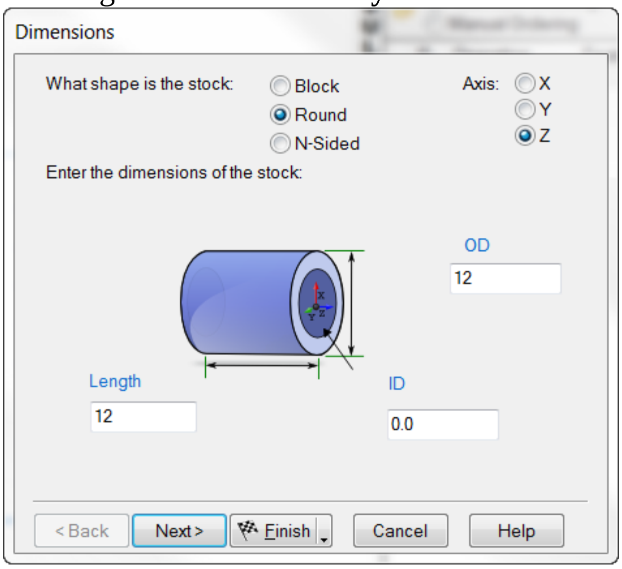
If the Stock Axis is not centered, any features that you will be adding to the part will be out of place. When a block stock is being used the UCS and Stock Axis are many times together but not always centered.

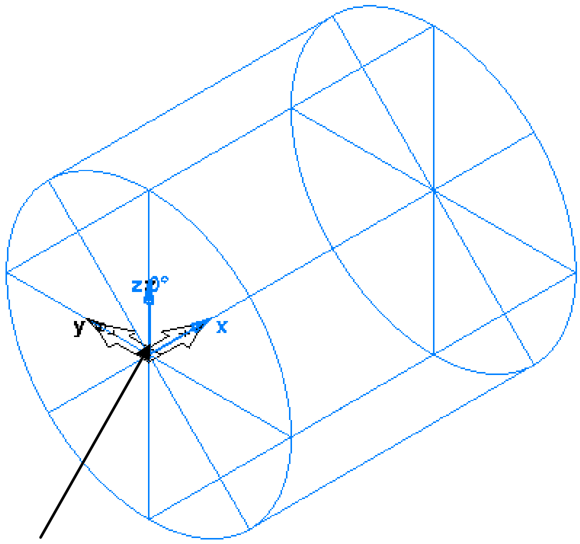
The **Stock axis CANNOT** be moved, the stock must be positioned around it.

- Click on **Options** then **Addins**.
- Check the box in front of the Macro called "**Center Indexed Stock.bas**" and a little tool bar appears, usually by the Part View.
- **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 center 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 centered on the stock. Now the features can be placed around the indexing axis. The **UCS may be moved** for Feature creation, if desired.





Stock Axis is Light Blue in color.

Caution: 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 worse.

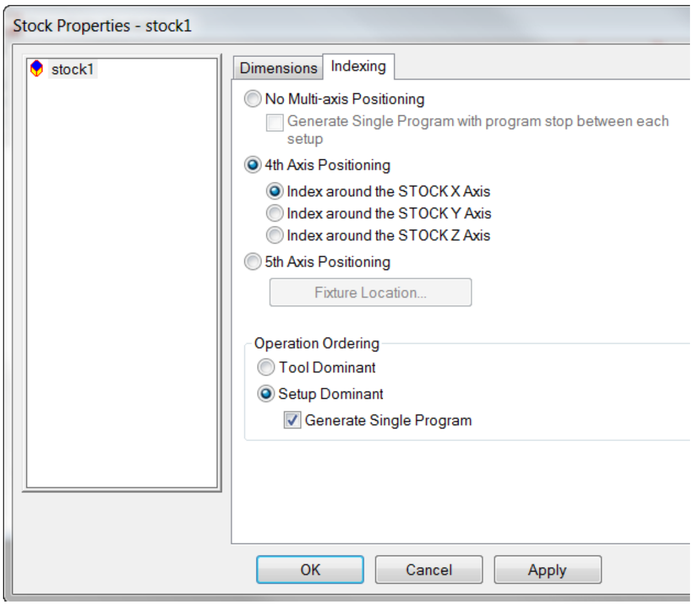
Once you have completed the prior steps, you may begin creating and placing features on your part. All 2.5D, and 3D, features may be programmed on an indexed part. FeatureCAM can index from face to face and cut features, or machine a continuously wrapping feature. Geometry, curves and features are applied 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 center of the part, the feature may be created at Z 0, 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.

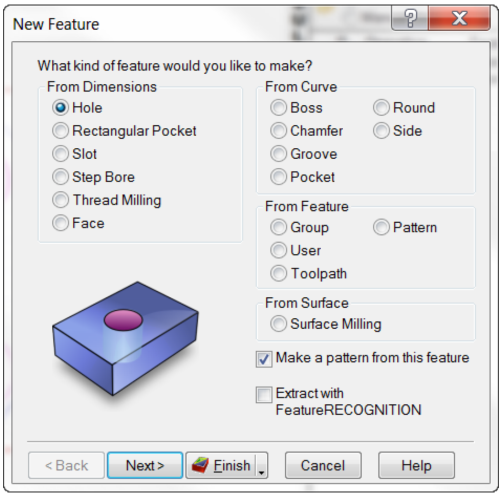
On the indexing tab you may choose Tool Dominant or Setup Dominant. **Tool Dominant** does all of the machining on every feature that uses that particular tool before it changes the tool, more indexing, less tool changes. **Tool dominant can have just one setup** for all of the part features.
Setup Dominant 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.

- Open a **New Document** and, using **one Setup**, create a **Stock (Block)** with the dimensions, **6 inch Length,Width** and **Thickness**. Place the **UCS** in the **Lower Left** corner, and material as **Aluminum**.

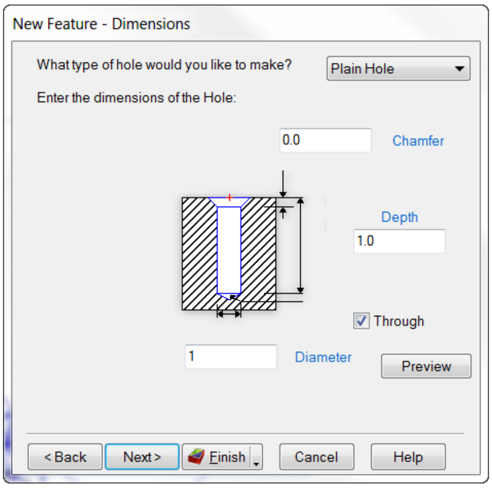


- Click on the **Indexing tab** and select **4th axis positioning, Index Around the Stock X axis** and **Tool Dominant. OK**.

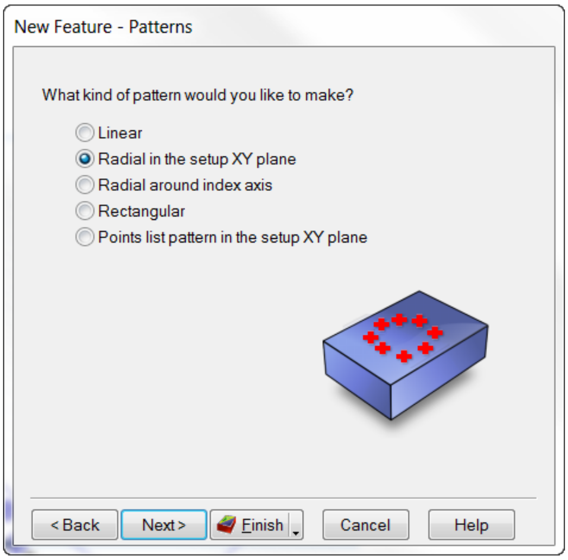
- Create a **New Feature**, select **hole** from dimension and **Make a pattern from this feature. Next.**



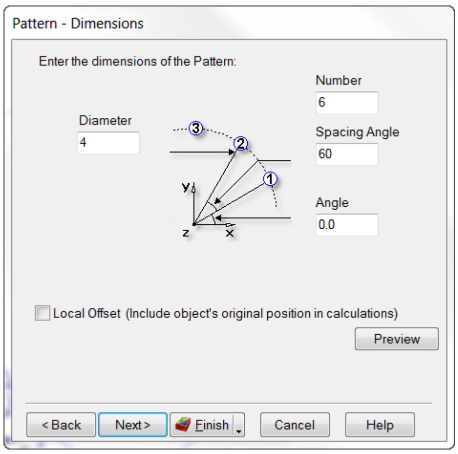
- Enter **Plain Hole**, **Diameter** and **Depth** of 1.0 and check **Through. Click Next**



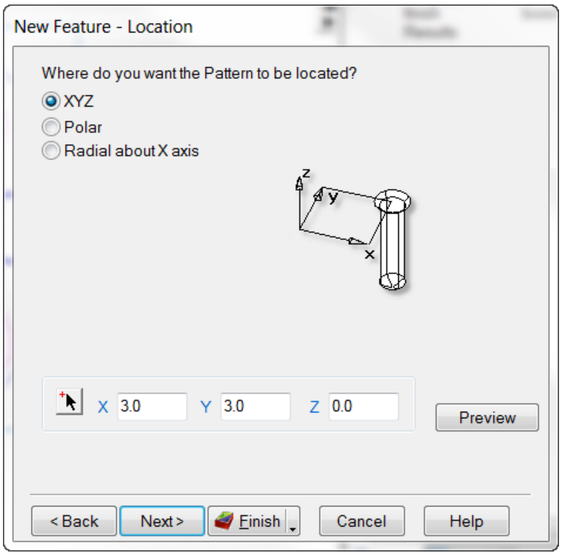
- Pick Radial in the Setup XY plane.

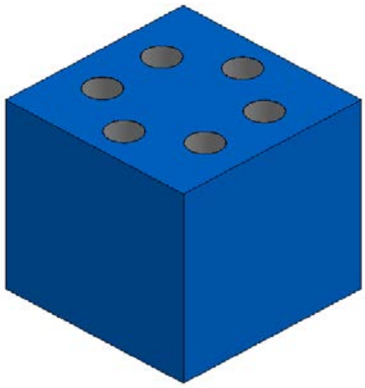


- **Diameter** 4, **Number** 6, **Spacing Angle** 60, **Angle** 0. This creates a pattern on one face and will be used to create the same pattern on the other 3 faces, a pattern within a pattern. **Next**

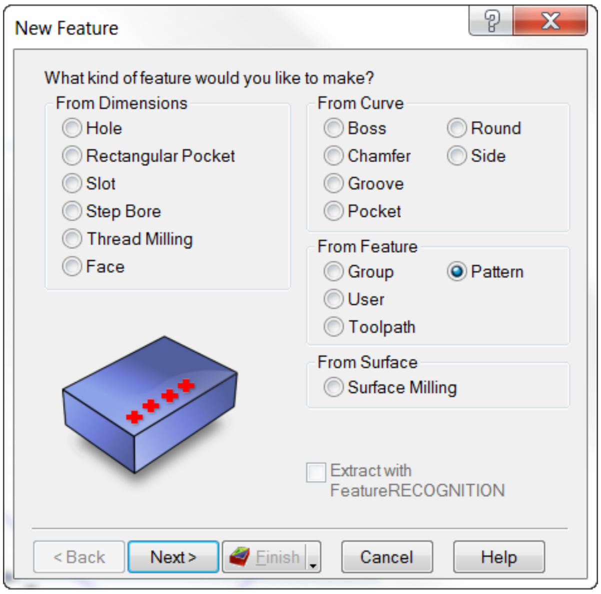


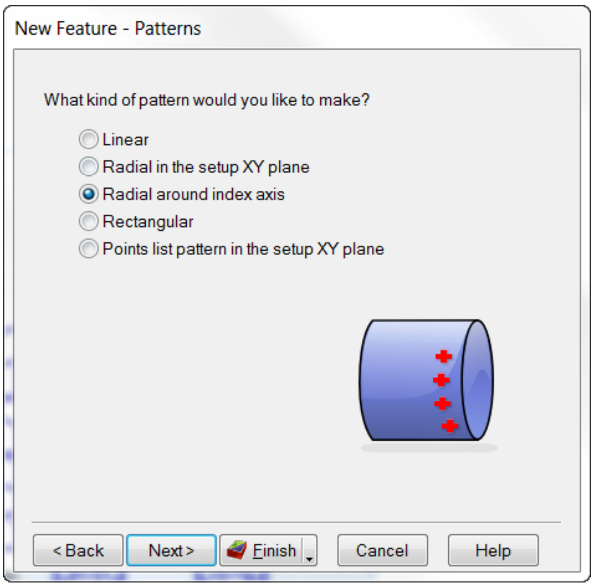
- **Location** X 3.0, Y3.0, Z 0. **Finish**



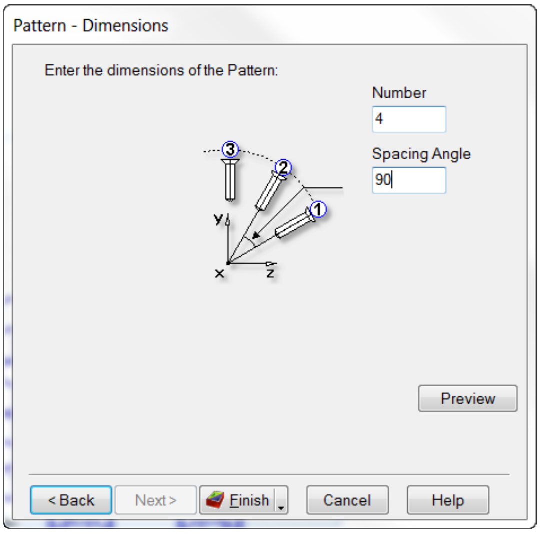


- Create a **New Feature**. **Pattern**. **Next.**
- Use the **pattern just created**. **Next**



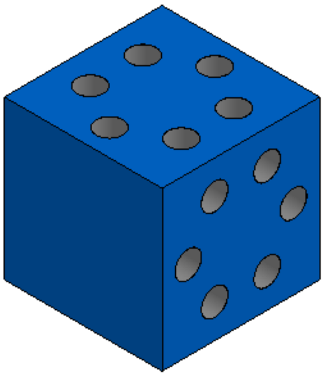


- Select **Radial Around the index axis**.



- **Number** 4, **Spacing Angle** 90.
- **Finish.**

This places the (6) hole pattern on four faces positioned the same as the original pattern.



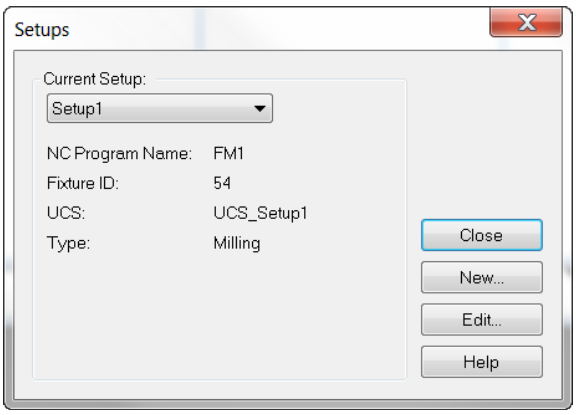
When you choose to put a different feature on each face this is **not** a **pattern** around the indexing axis. There are several ways to accomplish this.

Note: Remember you **CAN'T transform/rotate/copy** features **around** the **indexing axis**.

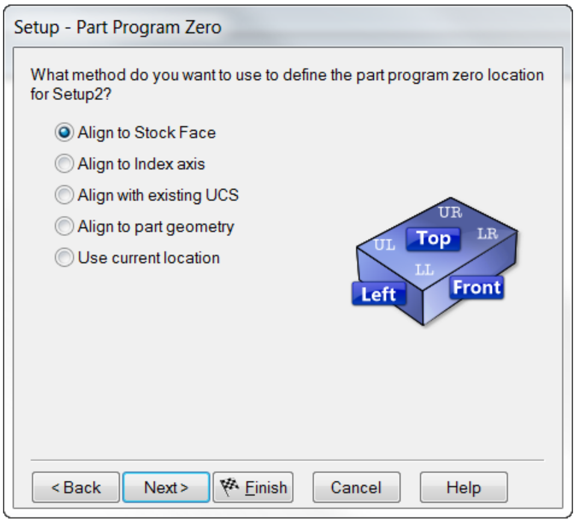
An **alternate way is to create a Setup on each face**. For example, on the 6 inch cube stock place a Setup on lower left corner of the stock on each face to match Setup1. When creating a new Setup you want the X axis to point in the same direction on ALL setups. If you need to, rotate the Setup around the Z axis so the X is pointing the same direction as the **Stock Axis** X.

Let's create 4 setups, top, front, bottom and back of the part.

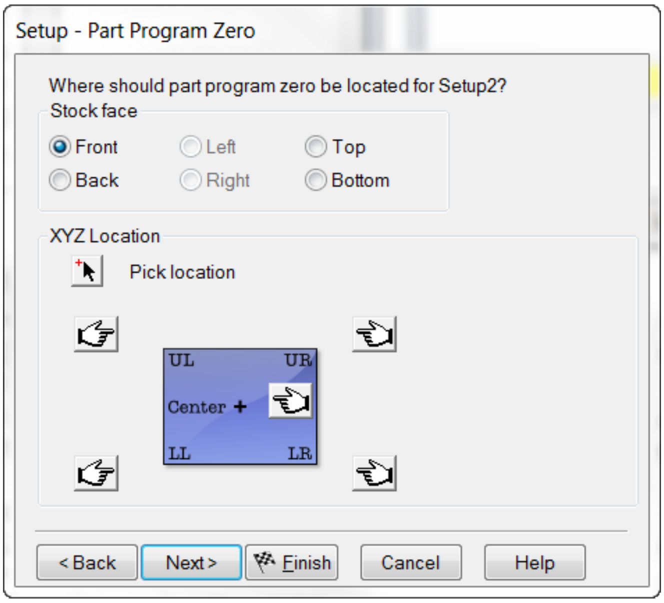
- Open a **New File** with the **Stock** dimensions 6" X 6" x 6", and align the **UCS and Setup** on the **Top** and **Lower Left** of the part.



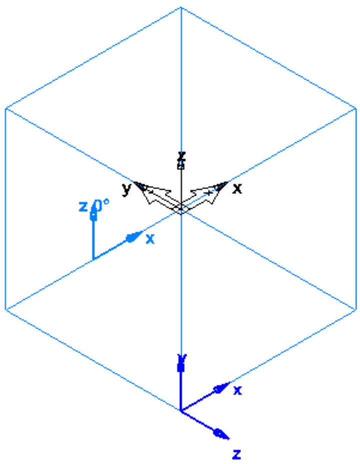
- Click the **Setups** icon on the **Advanced Toolbar**. 



- Click **New** and **Next.**
- Select **Align to Stock Face**.
- **Next.**



- Select **Front** for the Stock Face and the little hand at **LL** (lower left).
- **Next**

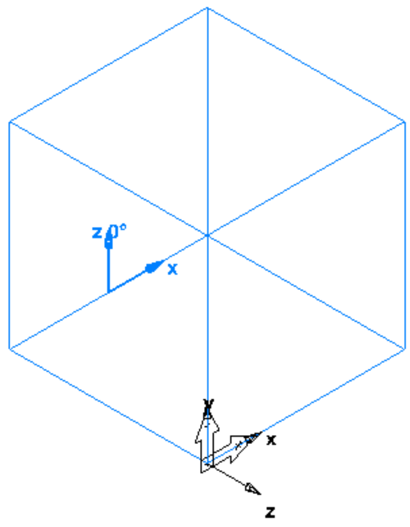


Note: If necessary use Z Rotate to **align X with Stock Axis X**.

- **Finish.**
- **Repeat the steps** for two other faces on the **Back** and **Bottom** of the part.

You can now create the feature, or features, as you would normally, (UCS – XY plane) on each face.

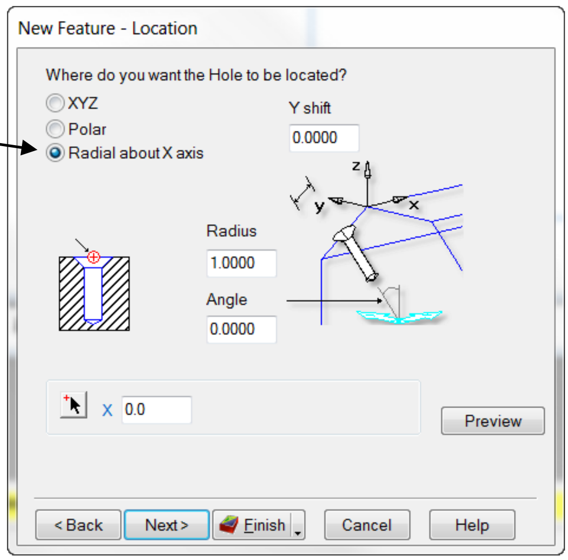
Hint: Keep Setup1 in the isometric view as you create each Setup. All faces relate to the first Setup as to which is Top, Front, Bottom and Back. This helps to keep the orientation straight in your mind. Also, **remember that you must have a Setup on each face to use Setup Dominant**.



- **Before** creating the feature, **select the Setup first** (Setup2 – Front Face) in **Part View** window that belongs to the face that you want to apply the feature to.
- Select the **Top view**, then create the features for that setup.
- Select next setup, top view, until all features are on all faces.

Another way is using **one setup** and creating new features, **from dimensions**, on multiple the faces. There is the ability to specify the **Location around the part** when **Radial about the X axis** is chosen on the **Location** dialog.

- Create a **New Feature**, select **Hole from dimensions**, **Plain Hole, Diameter** and **Depth** 1.0. **Next.**
- On the location page check **Radial about X axis.**



The **Radius** value is the **top of the feature.** It is measured from the center of the **Stock Axis** in the **Z direction** and will place the top of the feature at the specified radius which would be the stock face.

- Enter 3 for the **Radius, Angle** 270.

This places the top of the feature to the desired face around the Indexing Axis.

- Enter 3 in the **X Location**.

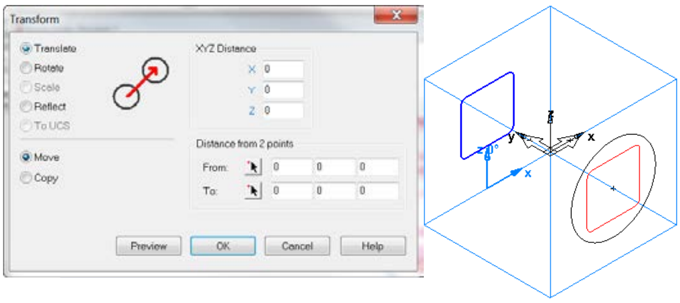
This will position the feature on the face measured from the UCS in the X direction.

- Enter 0 for the **Y shift**.

This will position the feature on the face in the Y direction, in this case no shift.

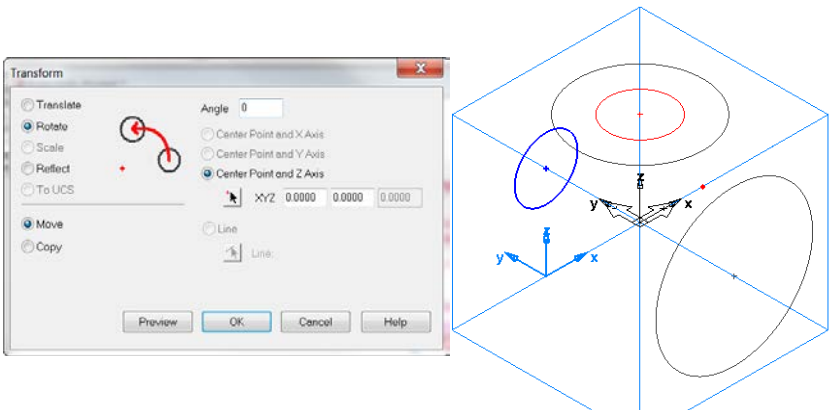
- **Finish**.

Another way is when creating new features **from a curve** using one setup. You need to view the XZ plane (Front), so **geometry** and **curves** created on the front face can be transformed to the back side using the Y direction. This also applies to curves created in the XY plane, top of the part that can then be transformed to the bottom using the Z direction. Once the geometry and curves are in place the New Feature can be created and you need only to select the curve on any face and FeatureCAM knowing the position around the indexing axis to machine the part.



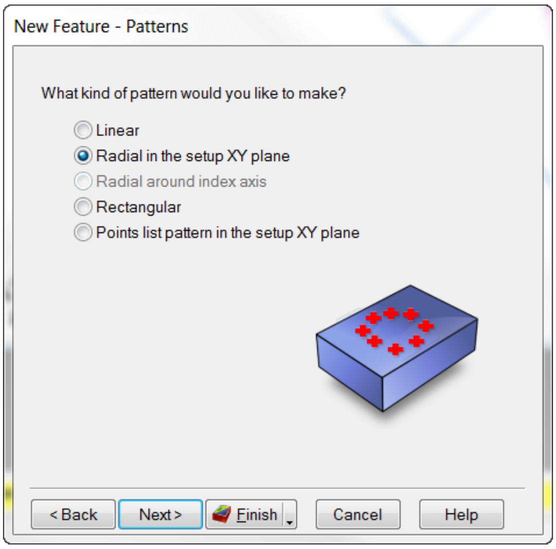
Another way, using one setup, is to create **ALL geometry and curves in the XY plane** and **Transform/rotate** them around the **center point and X axis**. Each feature needs to be rotated to the correct face, individually, then create the new features from curves.

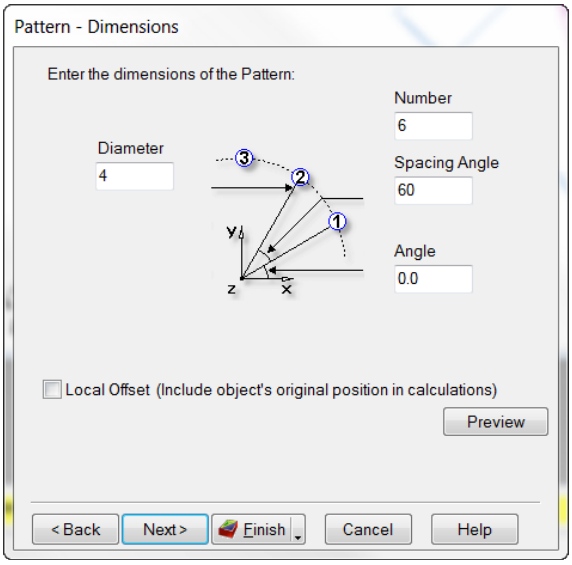
Remember you CAN'T rotate FEATURES around an index axis but you can rotate geometry and curves).



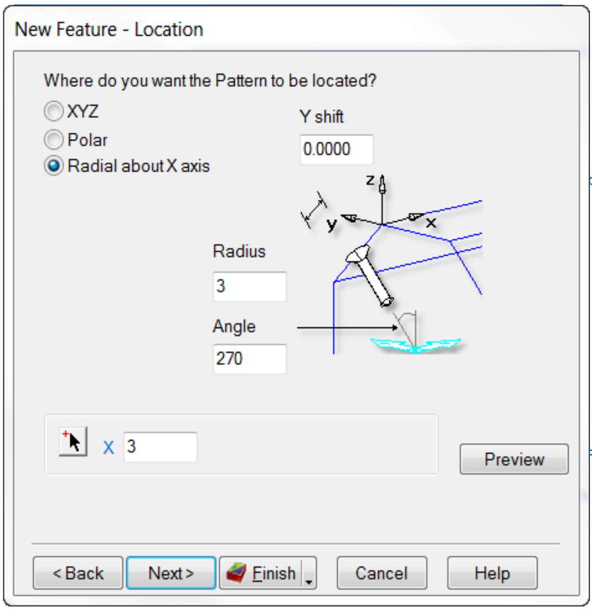
The **last way** using one setup is when you create a hole pattern in the XY plane (for example a bolt circle) and then you need to place it on one other face. In this case, you don't use a pattern from a pattern, because you only need it on one other face, and you can't Transform/Rotate features around the indexing axis.

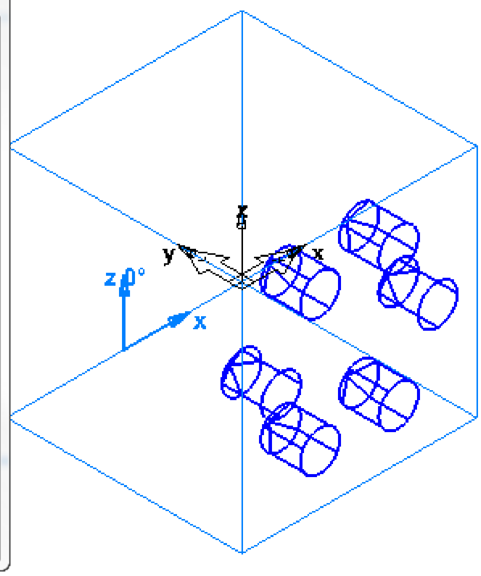
- Open a **New File** and create 6” X 6” X 6” **Block** of **Stock**.
- **New Feature. Hole** from dimension and **Make a Pattern from this Feature**. **Next.**
- Create a **Plain hole**, **Diameter** and **Depth** 1.0. **Next**.
- **Radial in the XY plane**. **Next.**





- **Diameter** 4, **Number** 6, **Spacing Angle** 60. **Next**.
- In the **Location** window select **Radial about the X axis**, **Y Shift** 0, **Radius** 3.0, **Angle** 270, **X Location** 3.0. **Finish**.



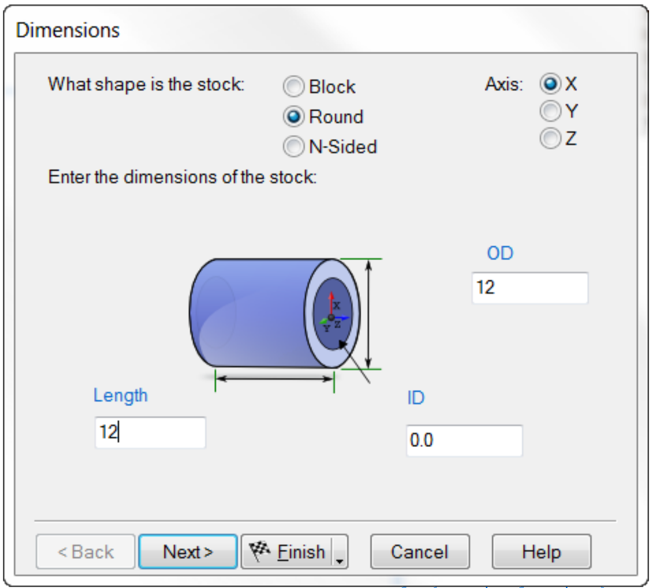


Round Stock

The **Stock Axis** is in the center 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 **CAN'T** 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 pocket and groove features, including engraving, around the Stock Axis.

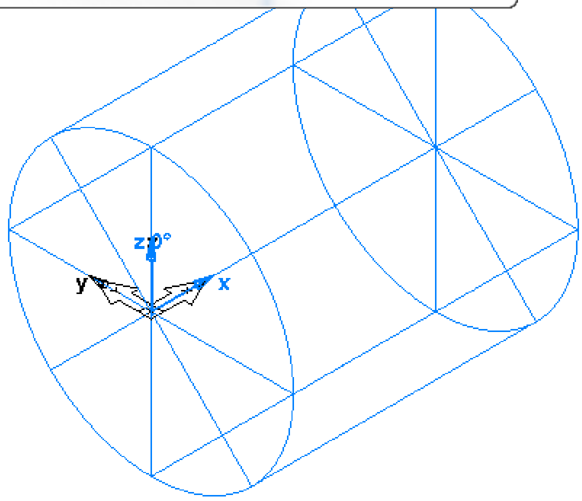
To wrap a pocket you must first create the pocket, and if the UCS is at the center of the part, you must translate the feature in the Z direction to put the top of the feature at the outer radius of the stock.

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 center of the part and you failed to move it up to the desired radius on the part, it cannot wrap around the center.



Wrapping a Groove

- Open a **New Milling Document**, **Inch**, **Round Stock**, **X axis**, **OD** 12.0, **Length** 12.0 and **ID** 0.
- On the **Indexing Tab** select **4th**
- **Axis Positioning**, and **Index around the STOCK X Axis.**
- Click on **View** from the top **Menu bar**. **Show > Show Stock Axis**.



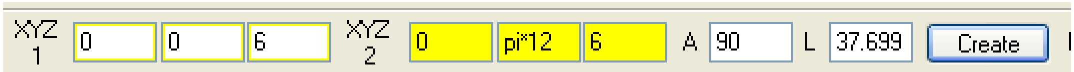
The Stock Axis and the Setup/UCS are together in the center of the part.

The stock and length of the stock are aligned with the X axis. Alternatively, open the **4axis wrapping.fm** example.

Note: 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 soup can. The values entered as indicated below will start the groove with the center of the tool being on X 0.0 and end on X 0.0 with no movement in the X direction. The Y is Pi*12 determines the length of the geometry and the part will rotate 360 degrees. The Z 6.0 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**.

- From **Geometry** draw a **line from 2 points** to the dimensions below. **Disregard length and angle**.

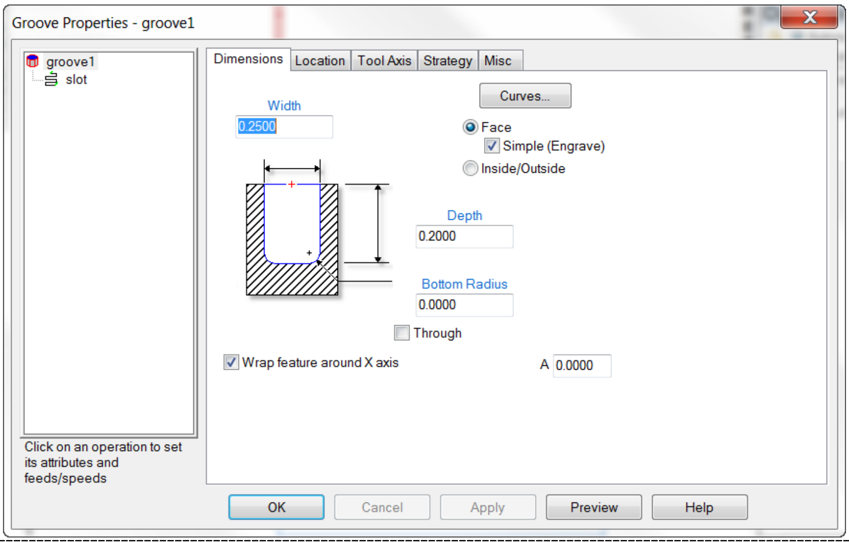


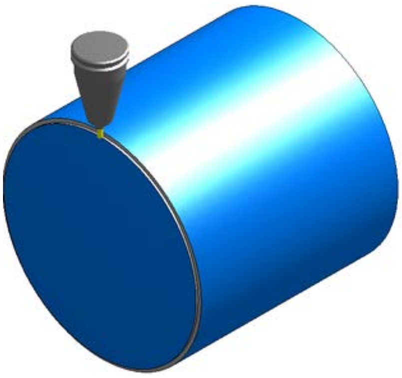
XYZ 1 sets the **start** point of the groove tool at the top or radius of the stock centered on the end where the Setup **X 0.0** is located. **Y 0.0** means it starts on 0 in the Y direction. **Z 6.0** means it starts at a 6.0 elevation in the Z direction above the center on the 6 inch radius of the 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** - 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*12** - Means the part will rotate once around the part. Notice that the value is pi*12, this means pi times the diameter which is the circumference of the 12 inch diameter stock and FeatureCAM understands what pi means and its value and the *(asterisk) is multiplication. **Z 6.0** - Means the tool ends at the same Z elevation as when it started.

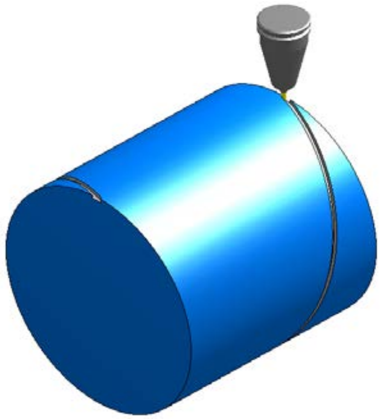
- Select **New Feature**, **Groove From Curve** and select the line.
- Groove **Width** .250, **Depth** .200, **Simple (Engrave)** and **Face. Finish.**
- Double click the groove feature in the **Part View** window. Check the **"Wrap feature around the X axis"** box at the bottom of the dimensions tab. Click **OK** and run a **3D simulation**.

The tool plunges to depth, rotates 360 degrees, retracts, and finishes.





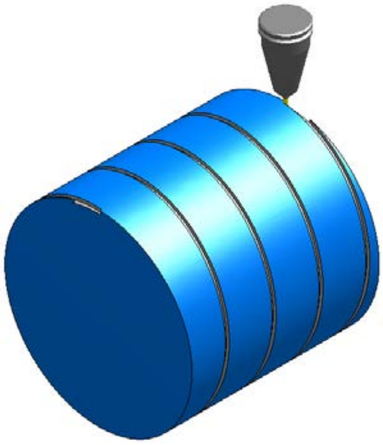
- Right click on the groove feature in the **Part View** and pick **Hide Selected.**



- Select the **Line** just created to change the dimensions.

- Change only the end point (XYZ2) in the **X** to **12.0**, click **Modify**, run a **3D simulation.**

Now the tool plunges to depth, part rotates once around but travels the full length of the part, retract and finished.



- Select the **Line** again.

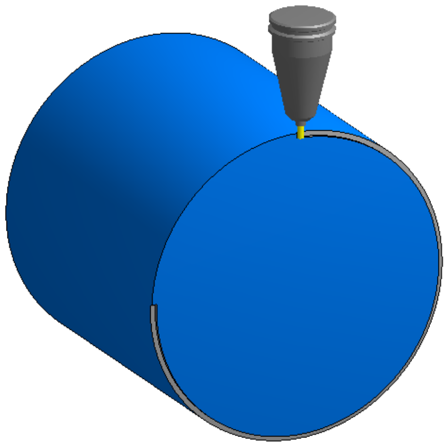
- Change the end point (XYZ2) **Y** to **4*pi*12**, **Modify**, and run a **3D simulation.**

Now the tool plunges to depth, part rotates 4 (4 x 360 degrees) times around the full length, retracts and finished.

- Select the **Line** once more.

- Change only the end point (XYZ2) **X** to **0** and **Y** to **270/360*pi*12**, run a **3D simulation.**

Now the tool plunges to depth, part rotates 270 degrees or ¾ of the way around, retracts and is finished.



Wrapping a Pocket

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 center 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.

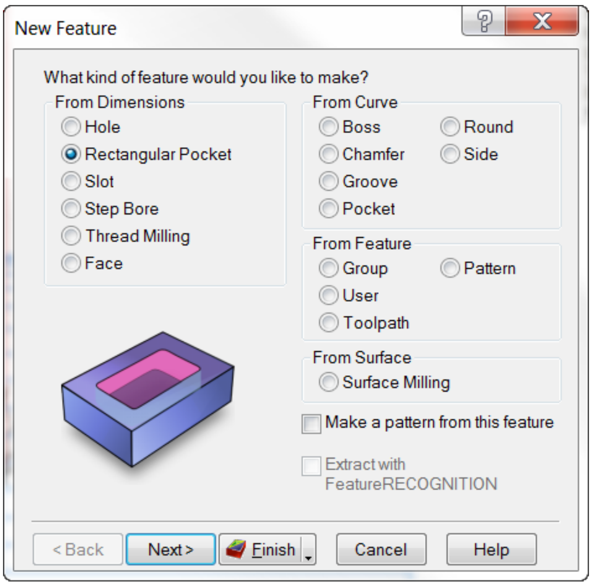
The feature, when completed, may be Transformed/Translated in the Z direction to locate the top at the radius of the stock.

The UCS can be translated in the Z direction, to the radius of the stock, and then the feature can be created. The feature remains at 0. in the Z Location.

Remember, if the 4th axis is not checked, on the **Stock** properties **Indexing tab,** or the feature is located down on the center of the part, the **"Wrap feature around the indexing axis"** on the dimension tab **will not be visible**.

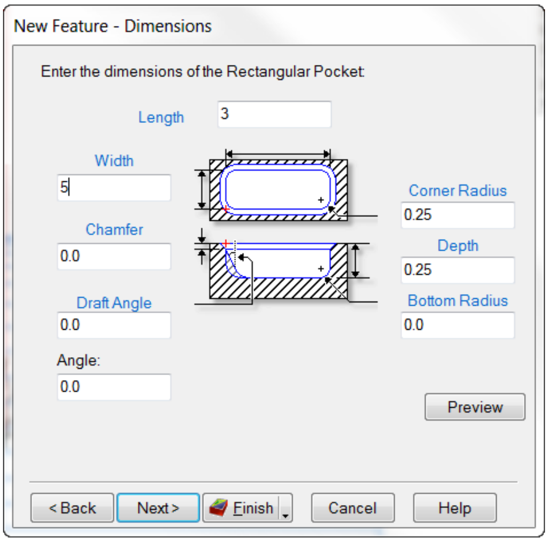
- Open a **New Milling Document**, **Inch, Stock** as **Round,** and **X axis.**
- Set the **Diameter** 12.0, **Length** 12.0, **ID** 0.
- **Next** twice, **4th Axis Positioning**, and **Index around the Stock X axis**.
- **Next** twice. **Rename** the Setup, if you choose. **Next.**
    - **Use the current location** for alignment. **Finish** and **OK**.

- Create a **New Feature**, **Rectangular Pocket** from Dimensions.
- **Next.**

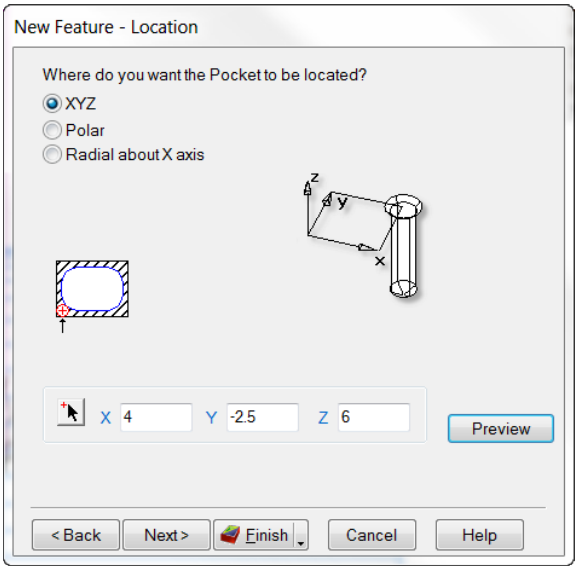


- Enter **Length** 3.0, **Width** 5.0, **Corner Radius** .250, **Depth** .250.

- **Next.**



- In the **Location** window enter X4, Y-2.5, Z6. **Finish.**

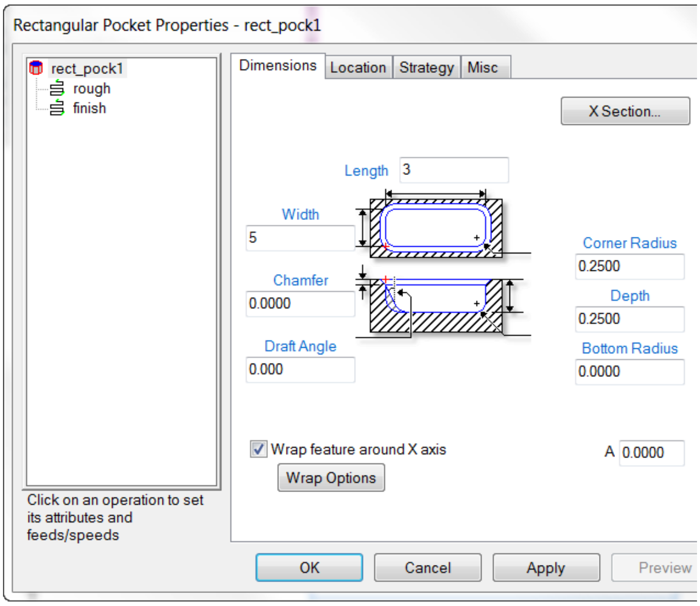


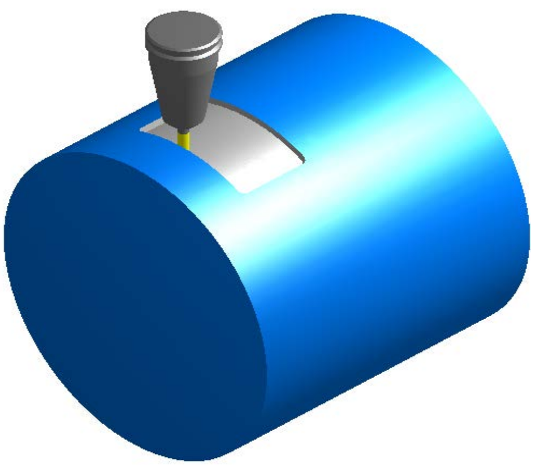
This centers the pocket and places the top of the feature at the outside radius of the stock.

- Double click the rectangular pocket feature in the **Part View** window. On the **Dimension** tab **check** the box **"Wrap Feature around X axis"**.

Note: If it's not visible, either the 4th axis is off, or the feature is at the center of the stock.

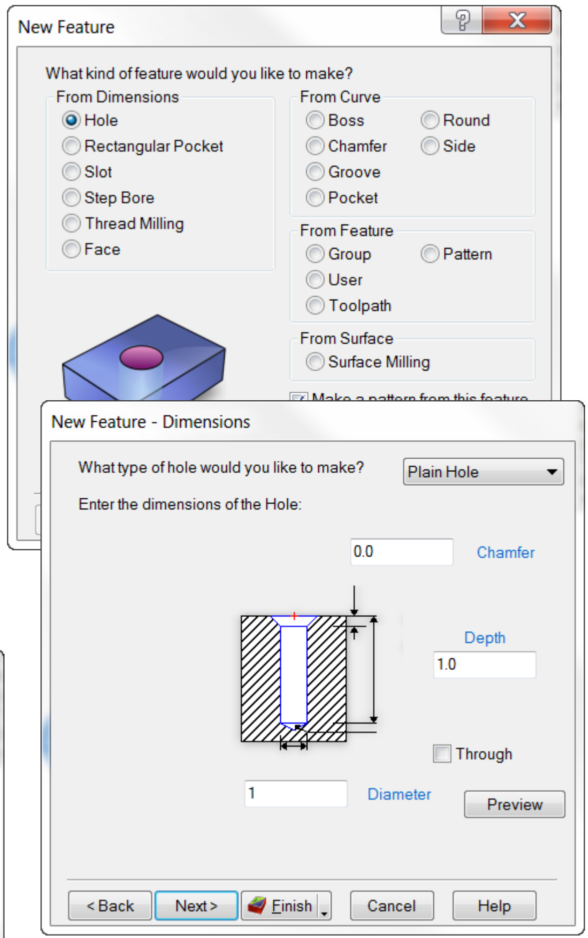
- Click **Apply** and **OK.** Run a **3D simulation**.



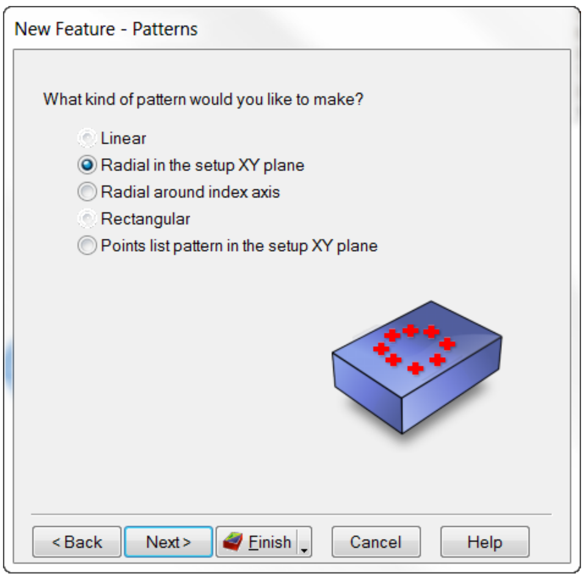


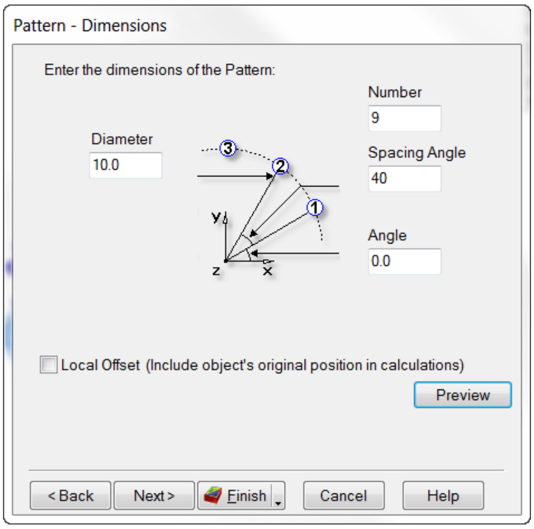
Wrapping a drilled hole pattern

- Open a **New Milling Document**, **Inch, Stock** as **Round, X axis**.
- Enter **Diameter** 12.0, **Length** 12.0, **ID** 0. **Next** twice.
- **4th axis Positioning**, and **Around STOCK X axis**. **Next.** Set it to **Tool Dominant**. **Next** twice, **Use current location**. **Finish.**

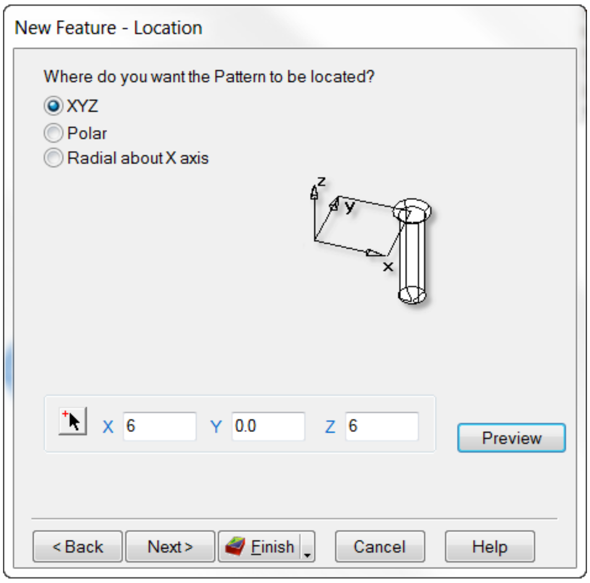


- Create a **New Feature**, **Hole** From Dimensions and **Make a pattern from this feature.** **Next.**
- **Plain Hole**, **Diameter** 1.0, **Depth** 1.0.
- **Next.**

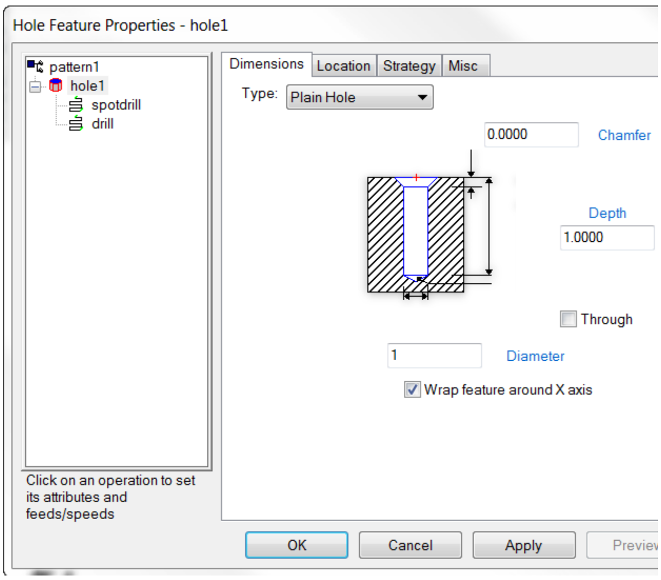




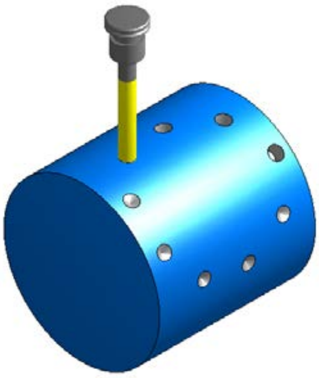
- **Radial in the setup XY plane.**
- **Next.**
- **Diameter** 10.0, **Number** 9, **Spacing Angle** 40, **Angle** 0. **Next.**



- Set the **Locations** at X 6.0, Y 0, Z 6.0.
- **Finish.**



- Double click the **hole** feature under the **Pattern** from the **Part View** window and check "**Wrap Feature around X axis**" **OK**.
- Run a **3D simulation**.

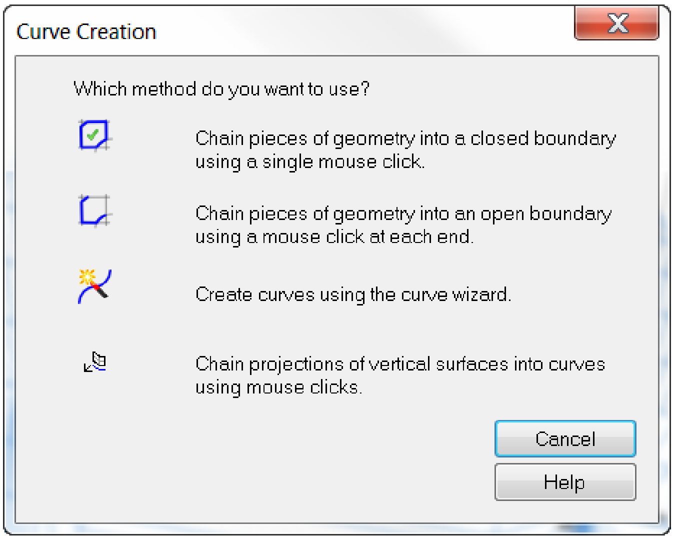


The wrapping keeps the point of the drill lined up with the centerline of the part.

Wrapped Engraving

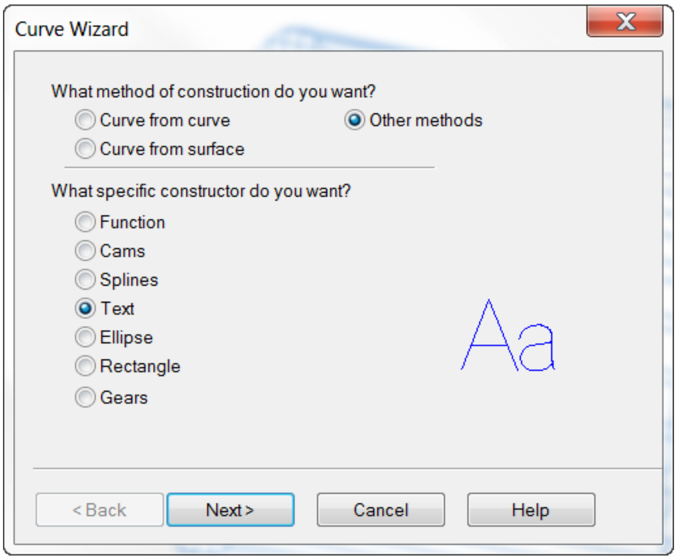
Engraving around a cylinder can be accomplished, by first creating the text, and then applying a grooving feature to the text curve.

- Open a **New Milling Document**. **Inch, Round Stock, X axis**.
- **Diameter** 12.0, **Length** 12.0, **ID** 0.
- **Next** twice.
- 4^{th} **axis Positioning**, and **Around STOCK X axis**.



- Click on **View** from the **Menu bar** at the top. **Show, Show STOCK Axis** to make sure the **Stock Axis** and the **Setup** are aligned and centered on the part.

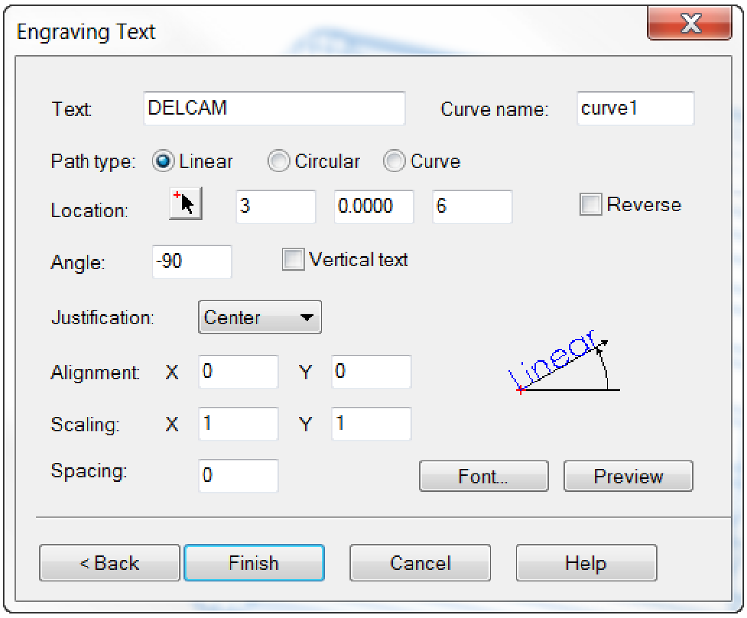
Change to a **Top view**, Select **Construct**, **Curve**, **Curve Wizard**.



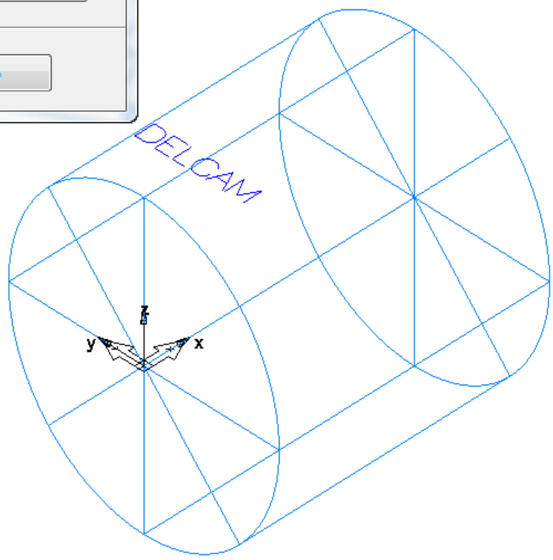
- Select **Other Methods** and **Text**. **Next.**

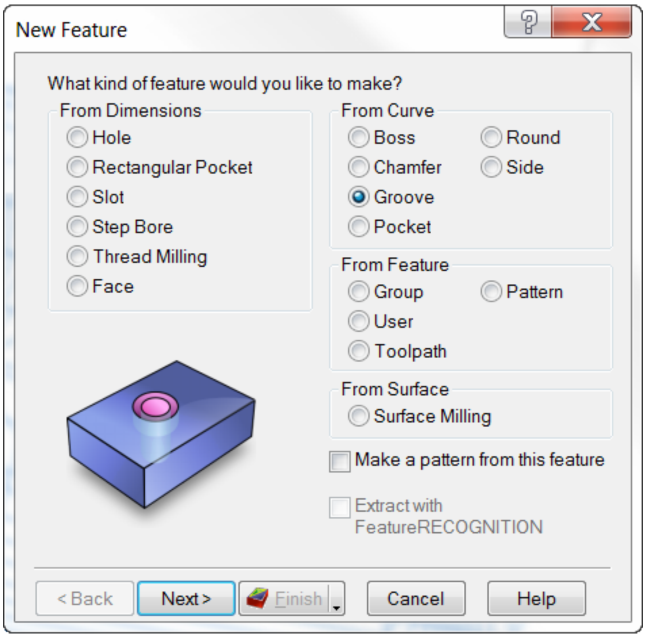
- In the **Text** field type **DELCAM**, click the **Font** button, select **Machine Tool San Serif** from the list, and **Size** 72. **OK.**

- Select **Linear**, **Location** of X 3, Y0, Z 6, **Angle** -90 and **Justification** Center. **Finish**.



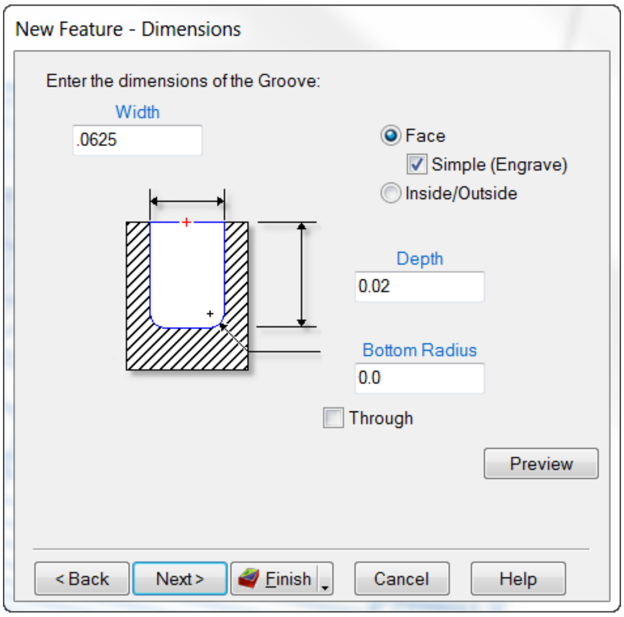
This will move the text to the outside radius of the Stock. It will also rotate and center it to the proper alignment position.

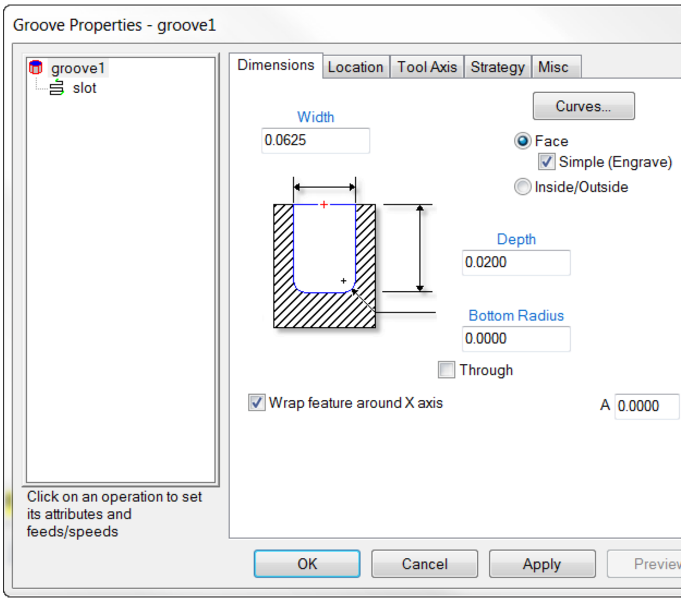




- Create a **New Feature, Groove From Curve.**
- **Next.**
- Select the **text curve** (all letters are one curve).
- **Next** twice.

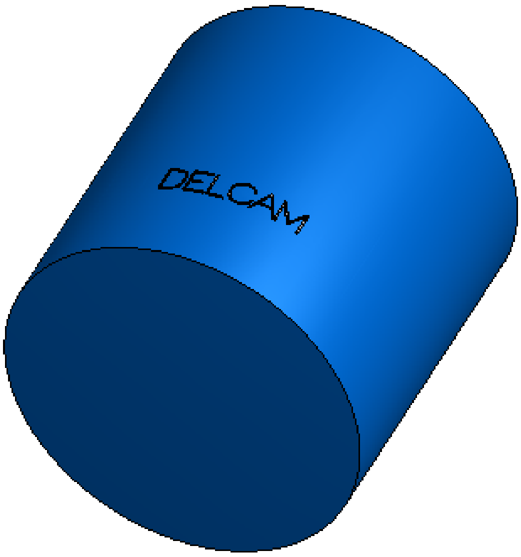
- Enter **Width** .0625, **Depth** .02, **Simple (Engrave)**, and **Face**.
- **Finish.**





- Double click the groove from the **Part View** window.
- On the Dimension Tab, check "**Wrap feature around X axis**". **Apply** and **OK.**

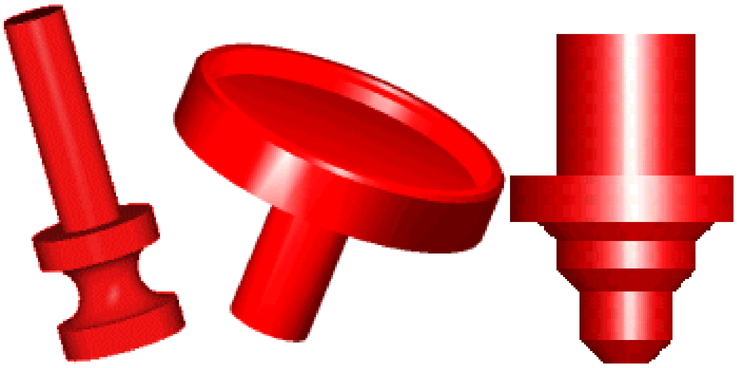
- Run **3D simulation**.



Form Tools

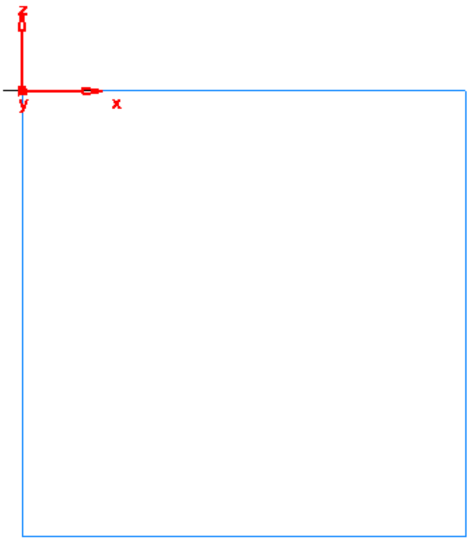
Introduction

In this module the user will learn how to create Form Tools for milling with end mills and side cutting milling cutters and also for drilling. 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 anytime.



Create a Dovetail Milling Form Tool

- **Open** a New Milling Document, inch or **Dovetail Form Tool.fm** for reference.
- **Create** Stock size 6x6x6, any material, no 4th axis. Viewing from the top, **Locate** the UCS in Lower Left Hand corner of stock.
- **Right Click** in graphics window and View from the front. The **XZ** plane.



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.

- To view the STOCK AXIS **Click** View in top toolbar>Show>Show STOCK AXIS.

Another set of arrows (blue) will show up in the graphics window merged with the UCS.

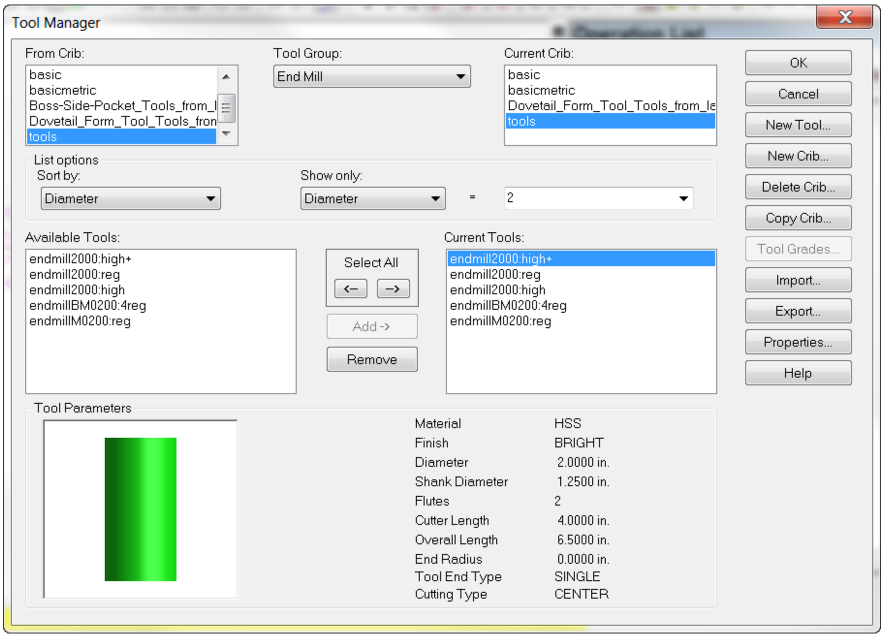
The Z0 will be the bottom of the Dovetail tool. The X0 is the center of the tool and only half (radius) of the tool will be created (+ side of X). The finish curve will be an open curve.

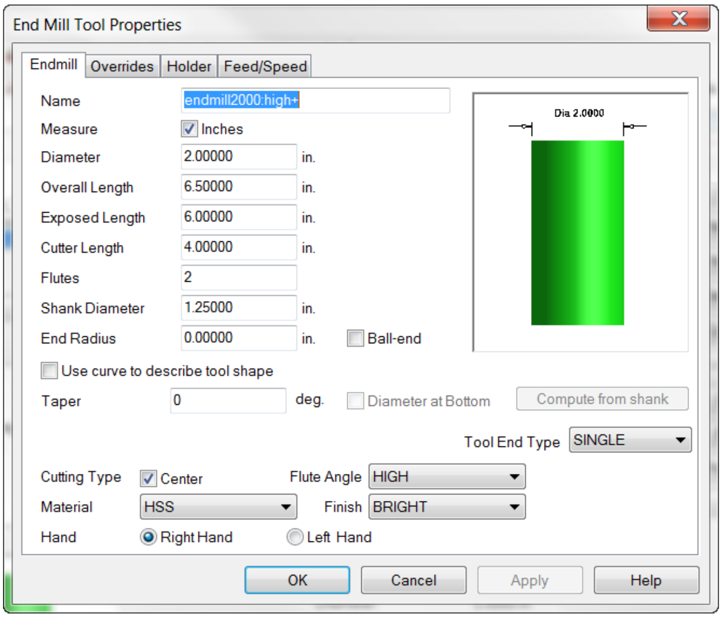
- Using the Geometry constructors **Draw** this shape and chain an open curve. (no centerline)

UCS = XYZ 0 and STOCK AXIS 0

- **Click** on Manufacturing on top toolbar and **Select** Tool Manager.

- Select "From Crib" Tools.
- "Tool Group" End Mill.
- "Current Crib" Tools.
- "Sort by" Diameter.
- "Show only" Diameter.
- "= 2" Press Enter. This takes you to the 2 inch diameter end mills. **OK**.
- **Double Click** on a 2 inch tool in the current list.

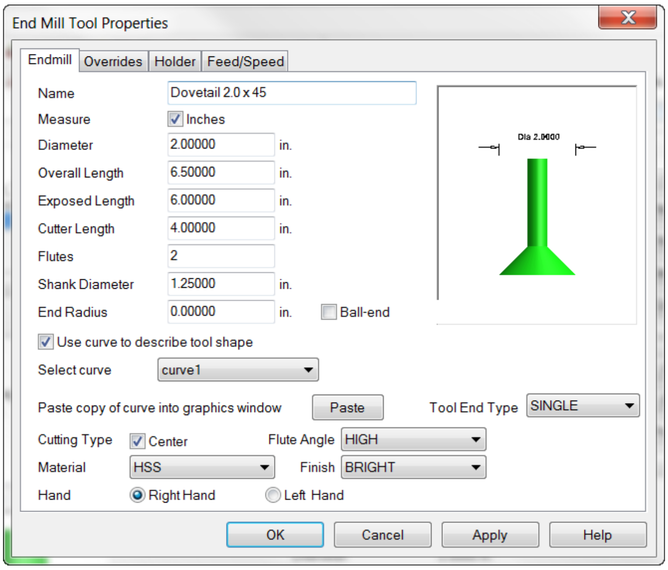




The 2" inch diameter end mill was chosen because our Dovetail tool is 2 inches.

When you specify the shape of a tool with a curve, many of the dimensions normally entered for a tool are no longer used, but the **diameter** dimension is **still critical** for **milling** operations. The **diameter** is still used for **calculating** stepovers and generating the paths.

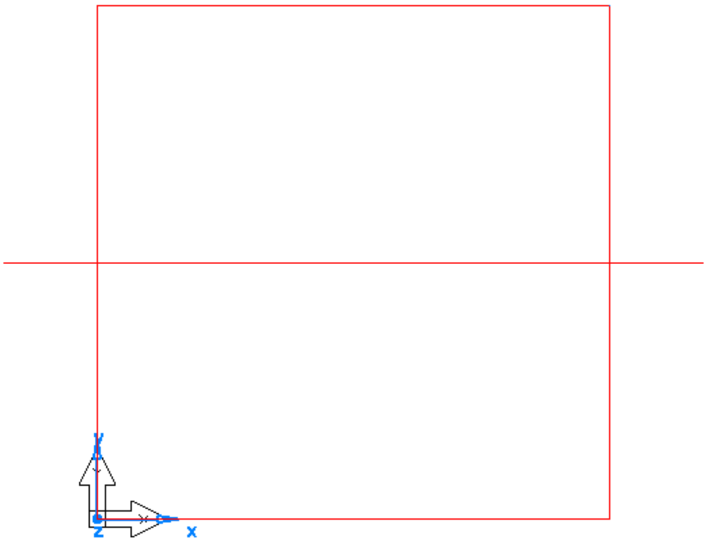
- **Change** the Name to “Dovetail 2.0 x 45”
- **Check** “Use curve to describe tool shape” and **Click** on the curve in the drop down list. **OK.**



Remember if the curve is not in the drop down then you either do not have the **STOCK AXIS** at the XYZ0 or the curve does not **start** at XYZ0.

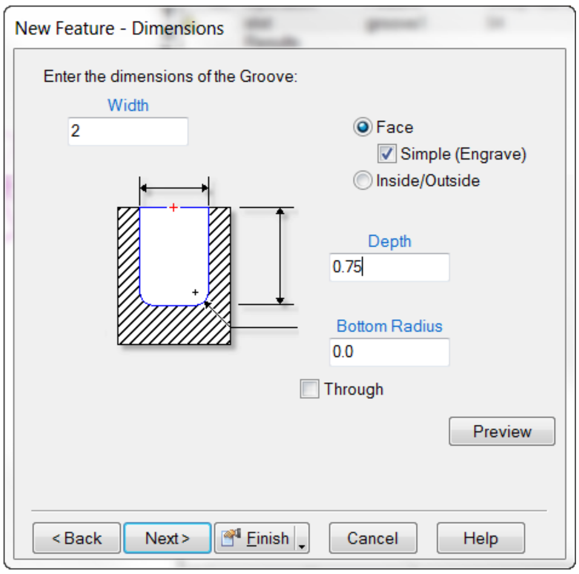
The New Tool shape appears in the window. Be sure to make any necessary changes such as “Cutting type”, “Material”, etc.

This only creates the tool; you must now create geometry and a groove feature.

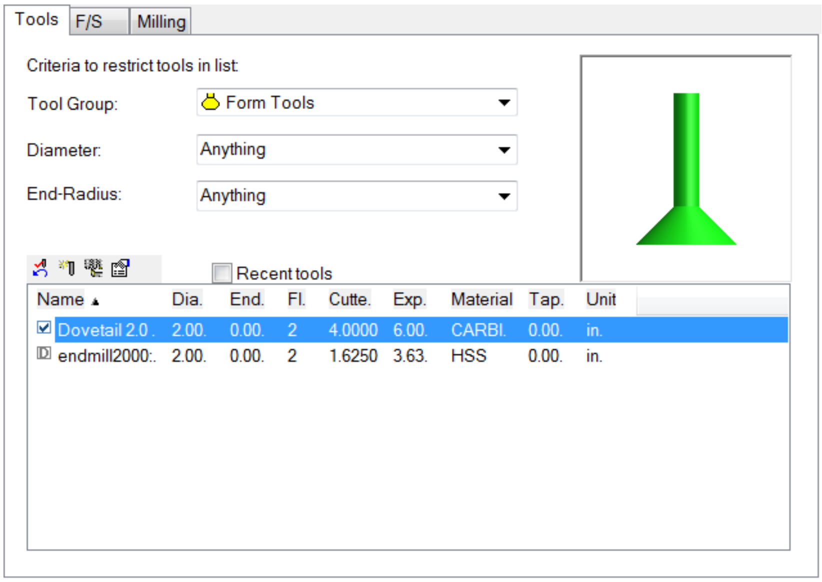


- Viewing from the top, **draw** a line across the center of stock. Extend the line 1.100 inches beyond each side of the stock so the tool will plunge off the material and retract when it’s clear at the end of the cut.

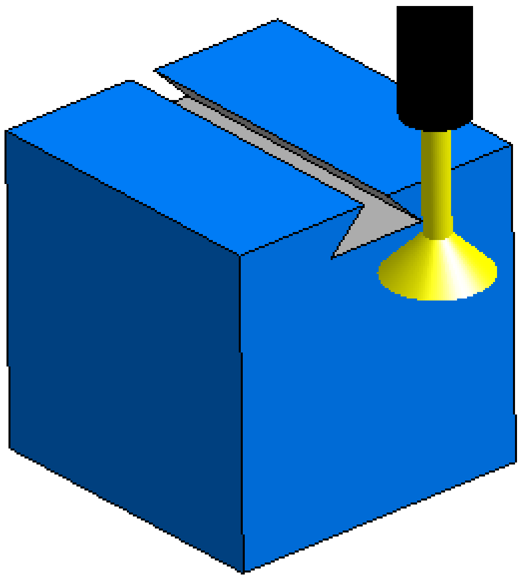
- **Click** on New Feature and **Select** “Groove from curve”, **Select** geometry line and **Enter** Width 2 inches. Simple (engrave). Depth of .75. **Finish and edit properties.**



- **Click** on slot operation and Tools tab and from Tool Group **Select** Form Tools. **Check** the box in front of Dovetail. **OK.**

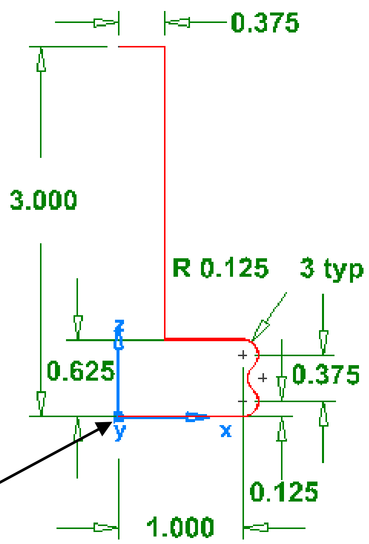


- Run **3D Simulation**



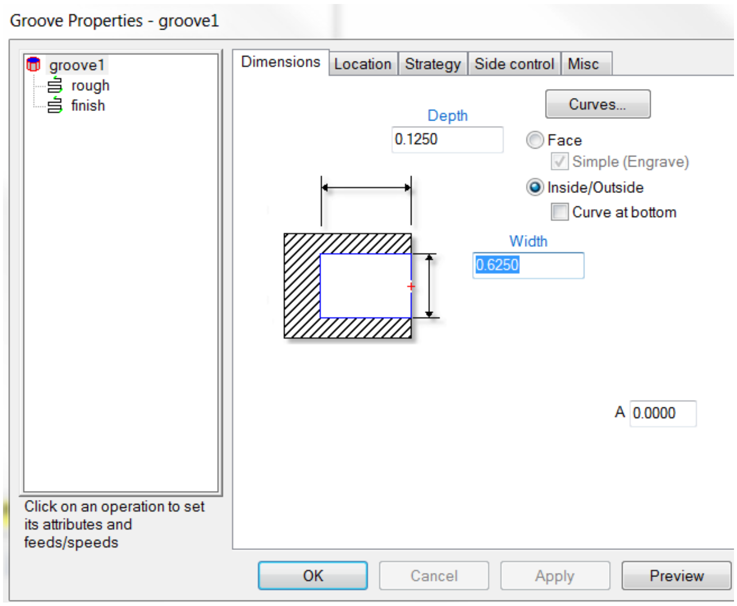
Create a Side Milling Form Tool

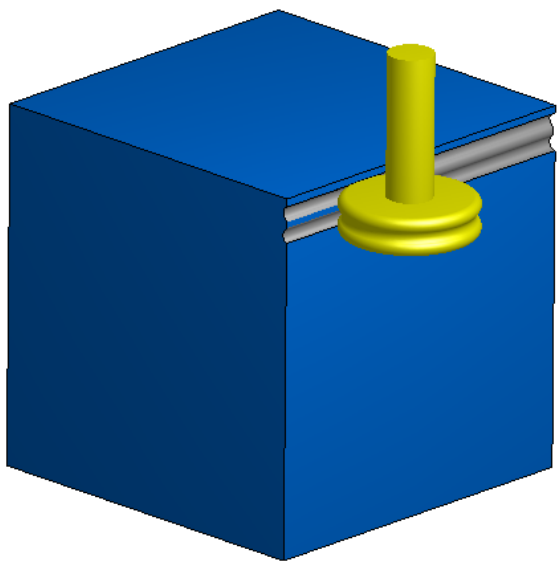
- Open a New Milling Document, inch or Side Milling Form Tool.fm for reference.
- **Create** Stock size 6x6x6, any material, no 4th axis. Viewing from the top, **Locate** the UCS in Lower Left Hand corner of stock.
- **Right Click** in graphics window and View from the front. The **XZ** plane.
- Using the Geometry constructors **Draw** this shape and **chain** an open curve. (no centerline)



UCS = XYZ 0 and STOCK AXIS 0

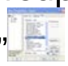
- Viewing from the front **draw** a line from 2 points at X0, Y0, Z -.4375 across the front face for the groove.
- Repeat the Groove from curve feature but instead select Inside/Outside with a Depth of .125 and a Width of .625. **Finish**.
- **Click** on Rough operation and Tool tab. From the list of Side Mills Select a KEYSEAT cutter then **Click** on the new tool button.
- **Change** the name to **Side Milling Form Tool 2.25D x (2).125r**.
- **Change** Diameter to 2.250. Change the Cutter width to .625. **Check** "Use curve to describe tool shape" and **Click** on this new curve in the drop down list as before. **OK.**
- **Click** on Finish and Tool tab and Tool Group Form Tools. Check the same tool as for the rougher. **OK** and **Run 3D Simulation.**

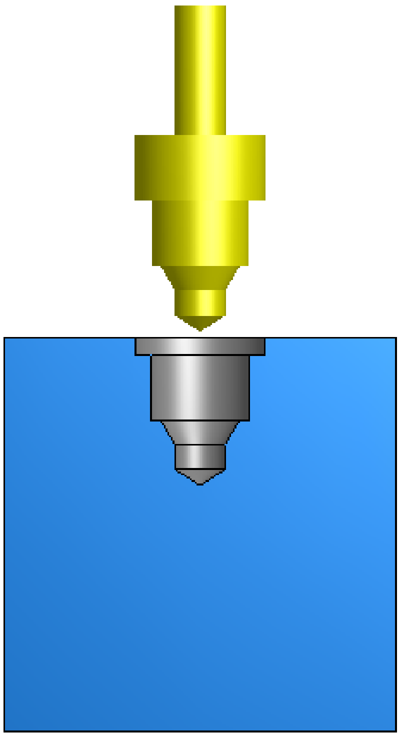


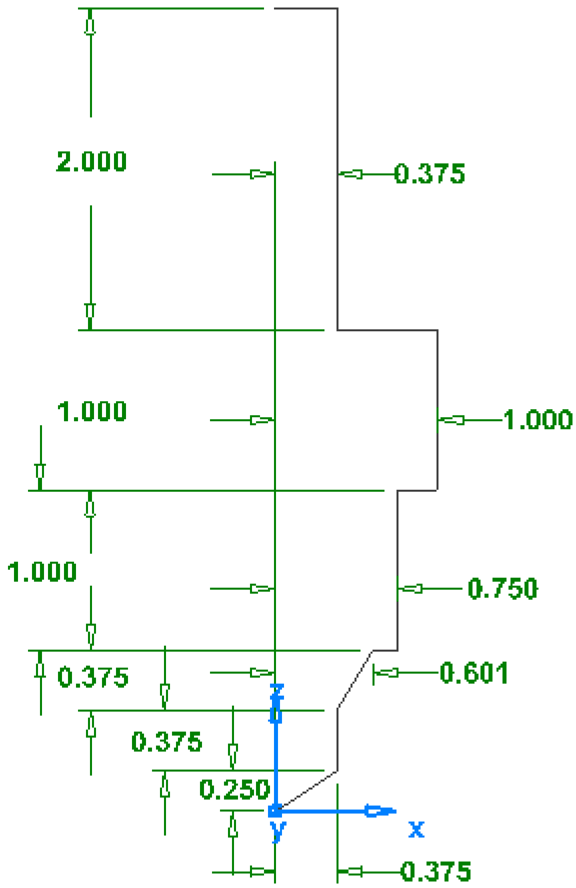


The Tool makes a Rough and Finish pass.

Create a Drilling Form Tool

- **Open** a New Milling Document, inch or **Drilling Form Tool.fm** for reference.
- **Create** Stock size 6x6x6, any material, no 4th axis. Viewing from the top, **Locate** the UCS in Lower Left Hand corner of stock.
- **Right Click** in graphics window and View from the front. The **XZ** plane.
- Using the Geometry constructors **Draw** shape from the dimensions shown on the next page and **chain** an open curve. (no centerline)
- **Click** on New Feature Wizard and **Select** Hole from Dimensions. **Next.**
- **Select** Plain Hole. 2.0 Diameter. 2.25 Depth. No Chamfer. Uncheck Through. **Next.**
- **Enter** X3, Y0, Z0 for Location. **Next.**
- **Uncheck** Spot Drill. **Next** and **Next**.
- Default Tool **Select** "I want to Search for another tool or make new one". **Next**.
- Tool Group **Select** Twist Drill and Diameter 2.0. Click "Create New Tool button" and **Change** Cutter Length to 3.0, Overall Length to 5.0 and Exposed Length to 4.0. **Check** "Use curve to describe tool shape" then **Select** the curve. Re-Name "Drill Form Tool 2.0" and **Apply**.
- **Check** Create New Tool when asked in Rename Tool popup. **OK** and accept as override. **Finish**.
- **Run 3D Simulation**. You can see the cross section.





Multiple Fixture Documents

Introduction

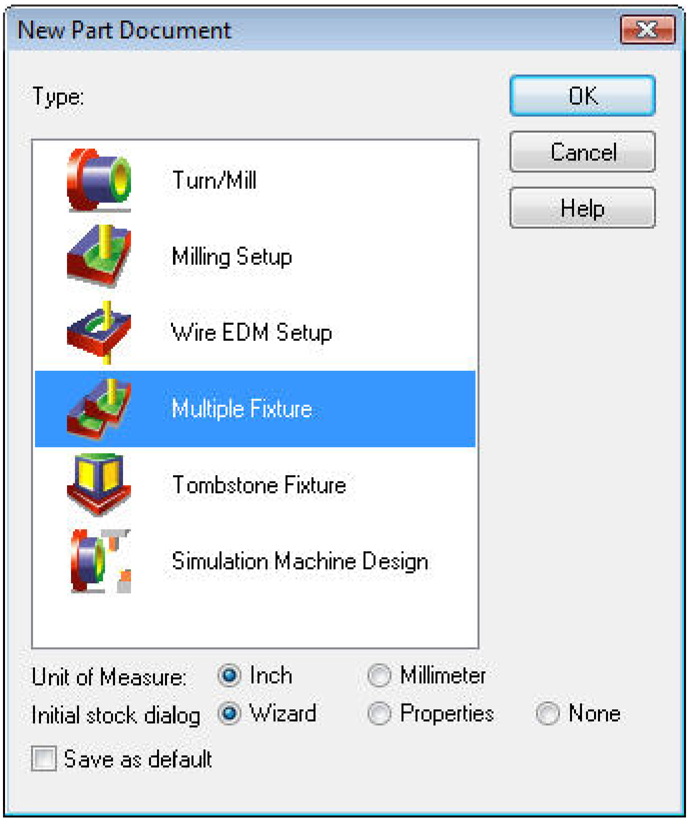
FeatureCAM allows the user to program multiple parts or multiple instances of the same part within a single part called a **Multiple Fixture (mf)** document. These documents are for milling parts that have already been created and saved in individual **fm** files. The parts can be arranged relative to a single Setup, or they can each have their own individual fixture positions. They can also be machined from a single stock model or from individual stock pieces.

Since complete parts are loaded into Multiple Fixture documents, they cannot be edited within the Multiple Fixture environment. They must be changed in the fm files and then manually reloaded into the mf file.

This chapter will describe the process for loading parts into Multiple Fixture documents.

Getting Started

In order to begin programming a part, the Multiple Fixture option must be selected from the New Part Document wizard.



Multiple Fixture Interface

The user interface for Multiple Fixtures is slightly different from the typical fm file. You will notice that many functions are dimmed or completely unavailable. Again, this is because the purpose of the Multiple Fixture document is to collect and arrange multiple parts into a single file – it is not for creating features of any type or making changes to existing parts. So, any functions related to creating toolpath will not be available.

Steps

The Steps mode of the Toolbox is open by default (Part View is not available at any time), and it is quite simple. The only entries that are unique to the mf file are Fixture ID and Parts. The rest of the options operate exactly as they do in any other FeatureCAM file.

Fixture ID

The Fixture ID step allows the user to define and locate established fixture locations. Those locations can later be used to anchor single or multiple parts.

Parts

The Parts step is the Wizard that is used to build and edit the part layout. It allows the user to add, delete, and edit all of the part configurations in the file. Remember, the parts themselves cannot be edited in the Multiple Fixture environment. We will discuss making changes later on in this chapter.

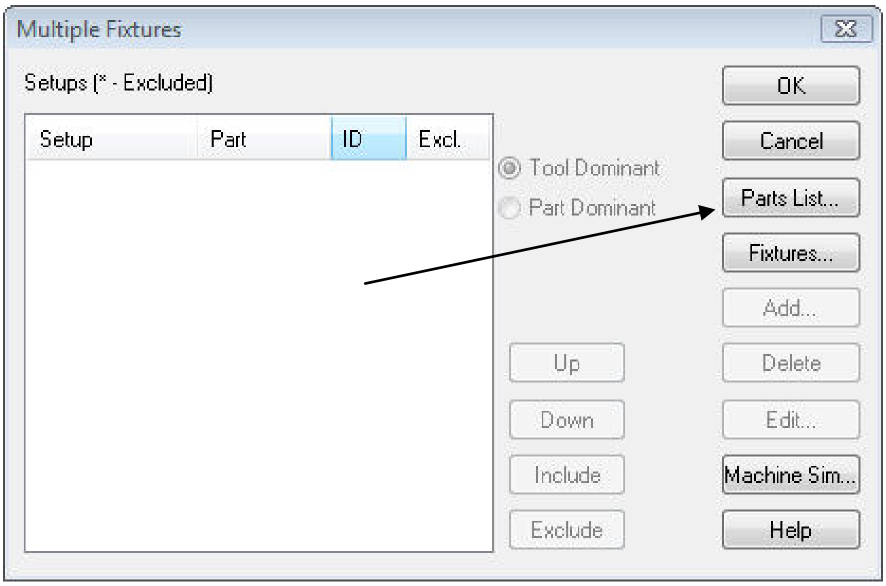
The following examples will examine how Multiple Fixture files can be programmed:

Example 1 - Multiple Parts as Separate Blocks

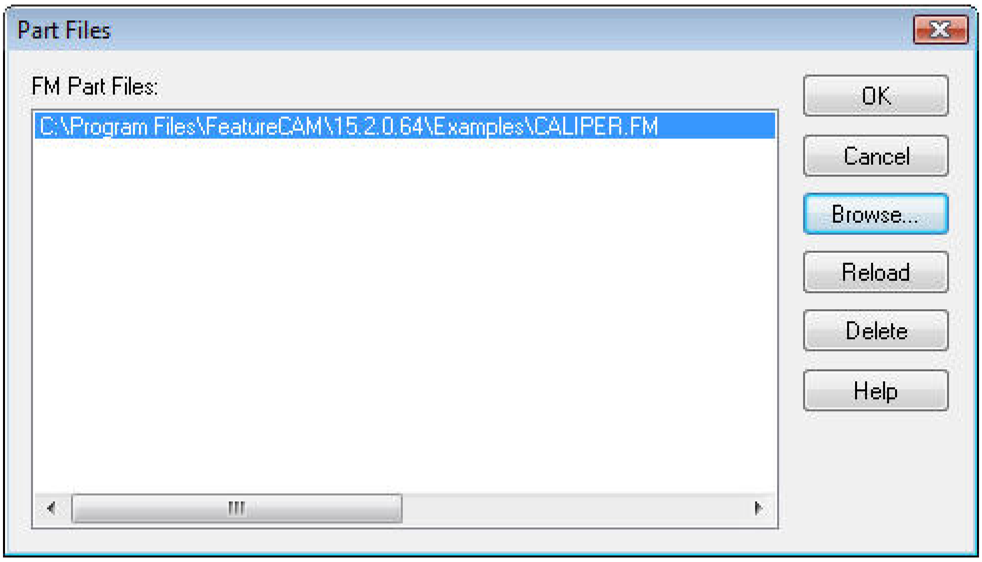
For this example, we will use the Caliper part that is found in the FeatureCAM Examples folder. This part has two setups – a top setup and a bottom setup. We will use this to create a row of tops and a row of bottoms. This will depict a scenario where you have parts that you are rotating through fixtures on the machine. For each cycle, you will be adding a set of material blanks and removing a set of complete parts.

The first step for creating a Multiple Fixture file is to add parts that you want to machine to the Parts List.

- Make sure the Parts step is active, and select **Parts List** from the buttons along the right edge of the dialog box



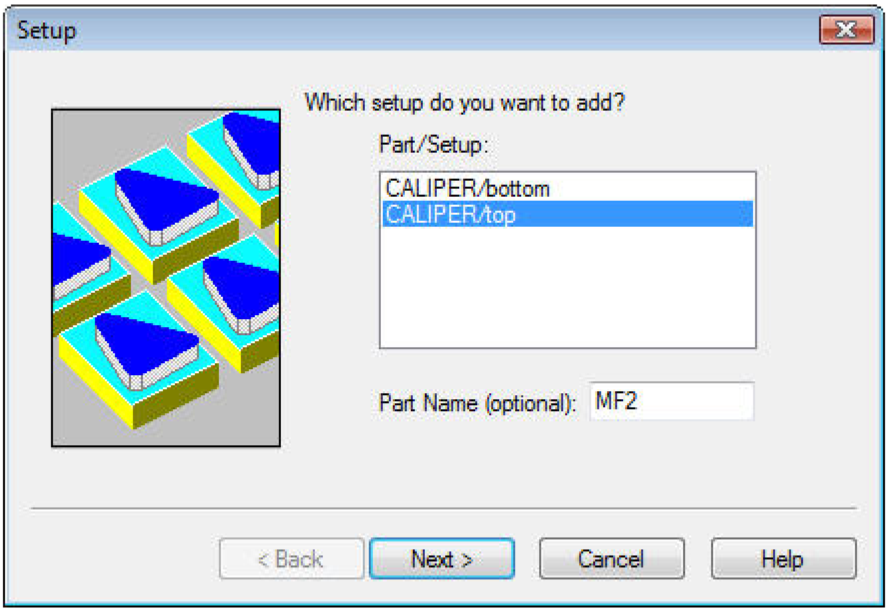
- When the **Part Files** window opens, click **Browse** and navigate to the **Data** folder
- Open **Caliper.fm.**
- **Click OK** and **Add**



As stated earlier, the Caliper part has two setups, and we will be machining both in this example. First, we will start with a row of Top setup parts.

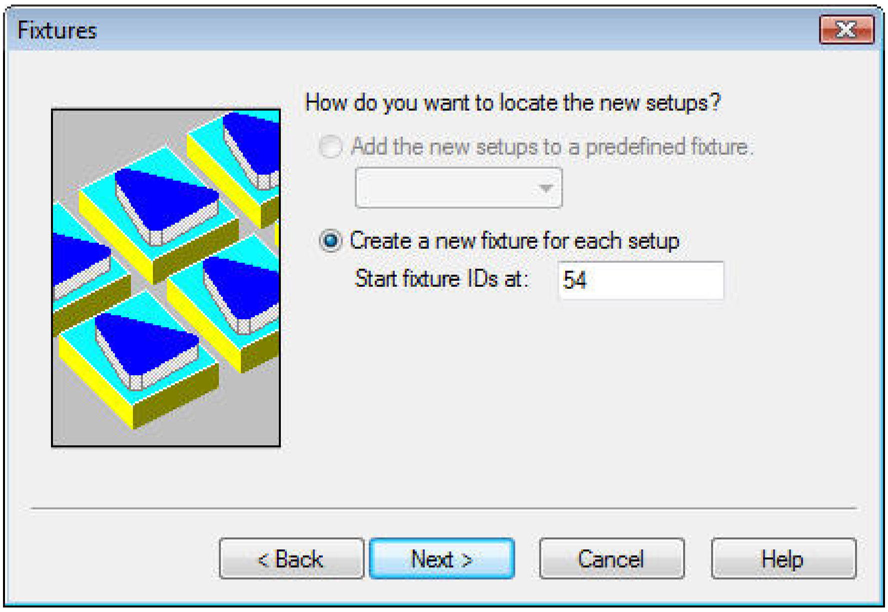
- Click the **top** setup

Notice that there is also a field where you can name the Multiple Fixture file. This is the name that will appear in the NC code, if desired.



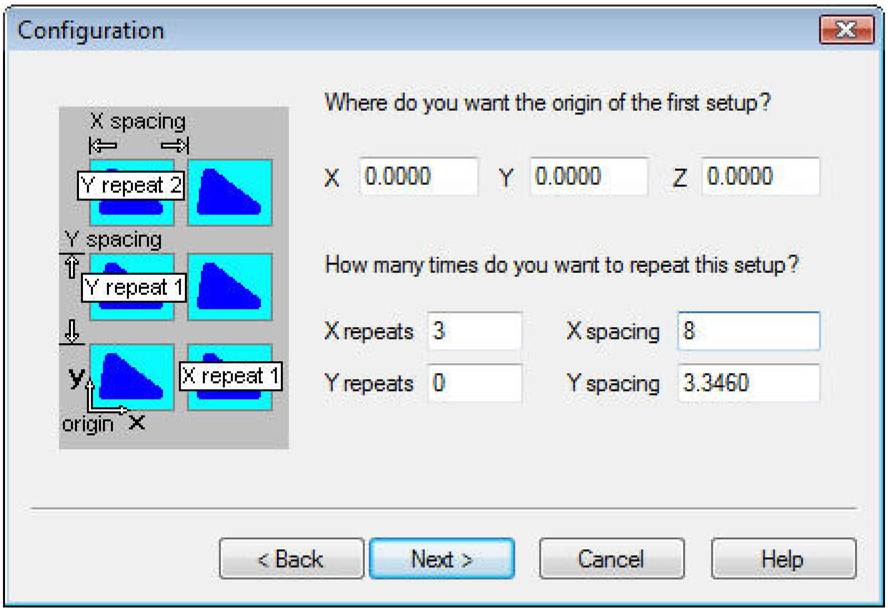
The next screen allows the user to select a starting offset for the parts. Since we will be cutting our parts from separate fixture offsets in this example, this dialog will determine the beginning offset, and other offsets will be added incrementally as we add parts.

- Accept **54** as the initial offset by clicking **Next**

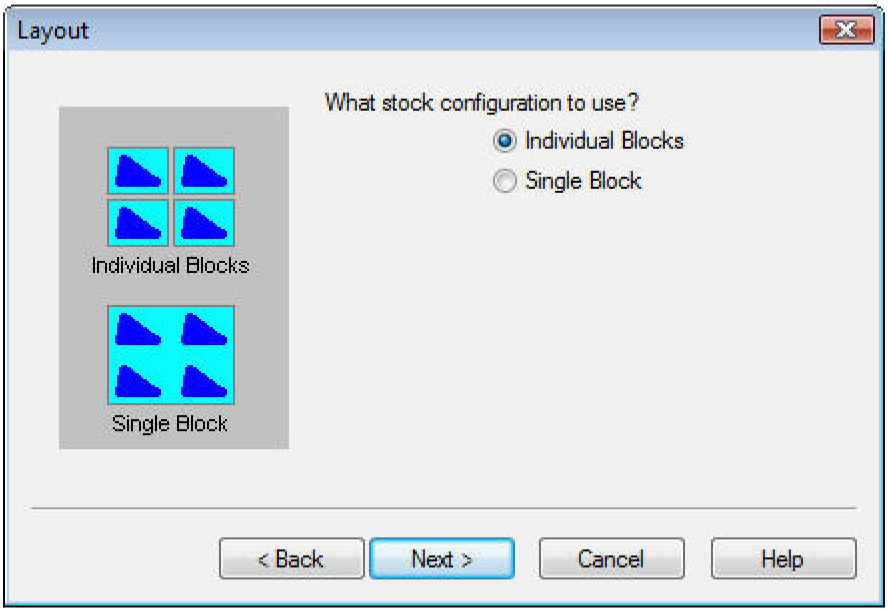


The next screen in the wizard defines how this part will be arranged. The XYZ location is the position of the first setup, and the other boxes determine the number and spacing of the repeats. Since these are repeats, a value of 3 will give us 4 instances. FeatureCAM fills in the spacing automatically by adding a set distance to the part's stock size, but this can be overridden to any value the user wants. Note: When using separate blocks of stock, the spacing is for simulation purposes only. The actual offset positions will have to be determined at the machine and entered into the control registers.

- Fill out the Configuration dialog box as shown
- Click Next

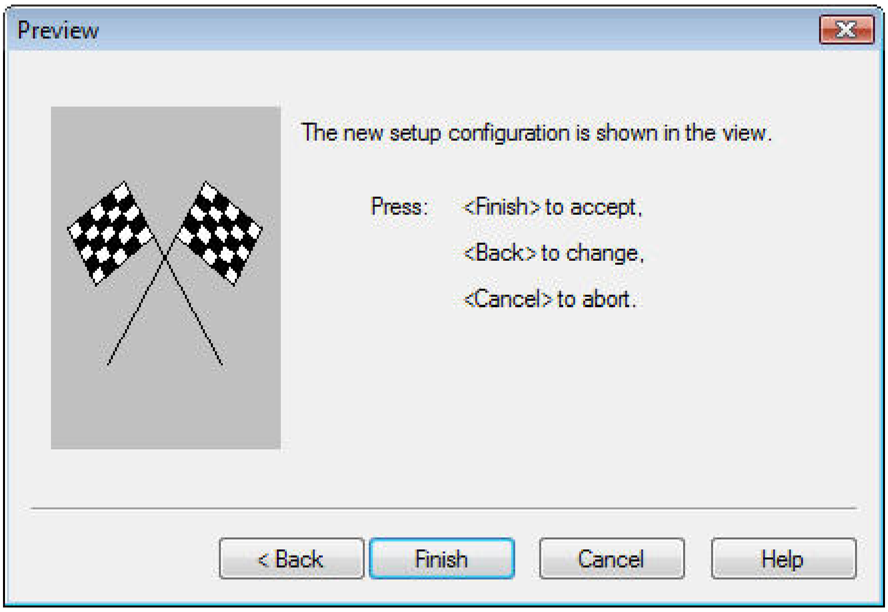


- Select Individual Blocks and click Next



At this point, we have completed the layout of the first set of parts. The Preview dialog appears. You can drag this window aside to inspect the part placement. If you want to make any changes, click Back to revisit the wizard screens.

- Click Finish

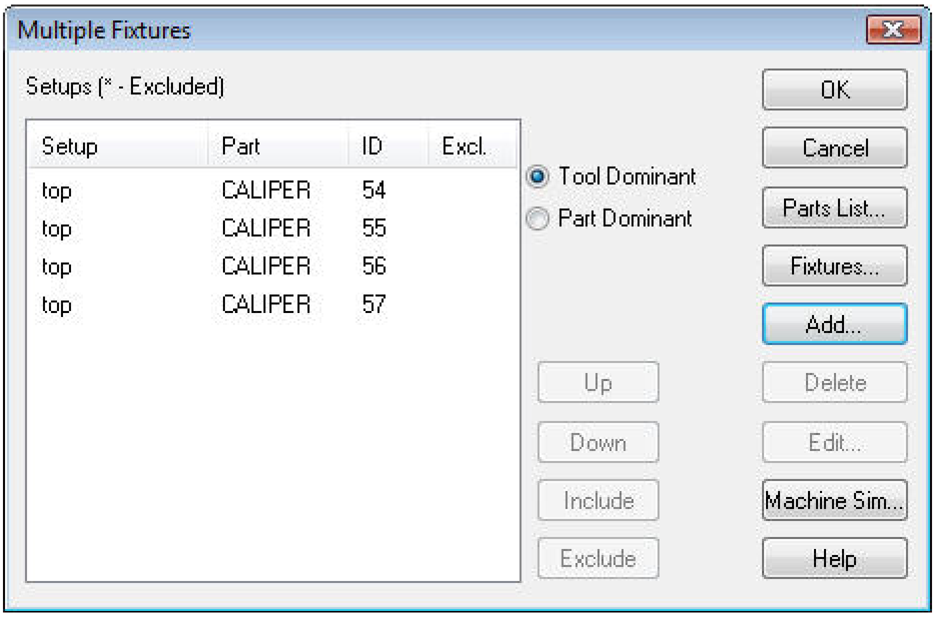


Once you have clicked Finish, the Multiple Fixture screen reappears showing you all of the parts and offsets in the file.

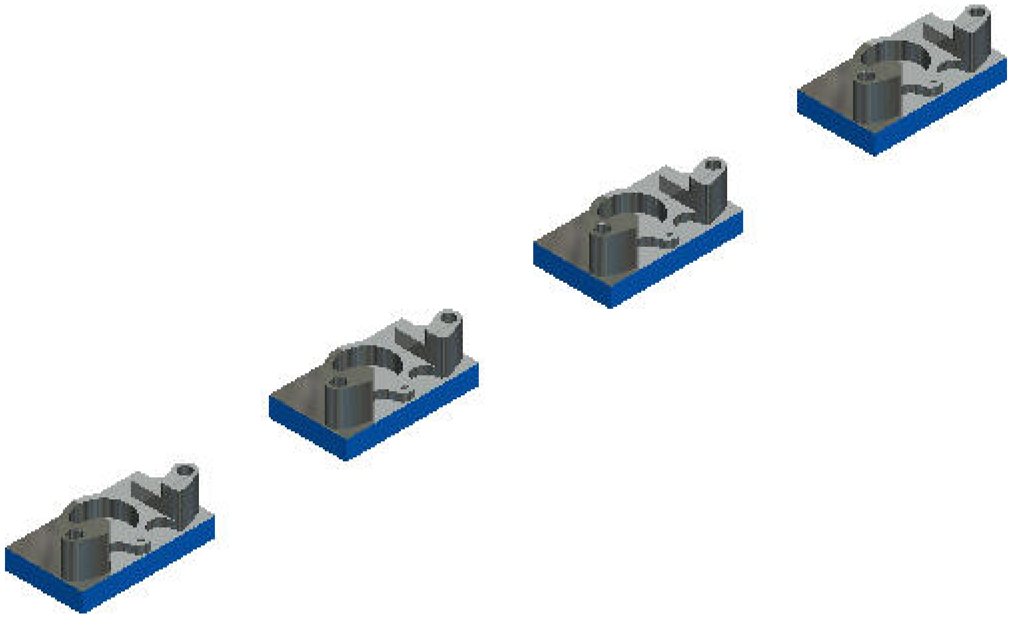
The parts in the Multiple Fixture file can be machined two different ways:

Tool Dominant – Each tool cuts all of the part instances before changing to the next tool.

Part Dominant – Each part is cut completely before proceeding to the next part.



- Click **OK** to exit the Multiple Fixtures window
- Run a simulation

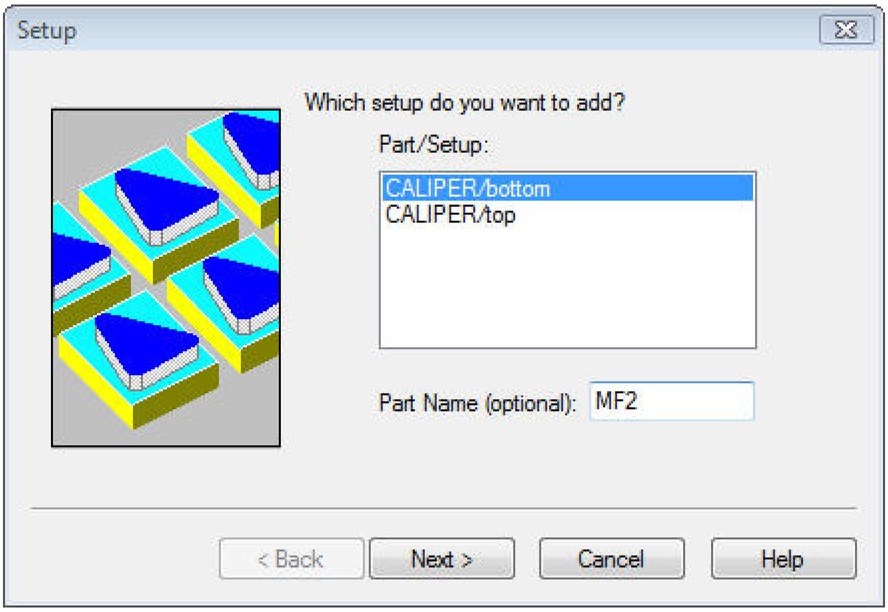


Next, we will add another set of parts to the file.

- Clear the Simulation Results
- Click on the Parts step to reopen the Multiple Fixtures window
- Click Add

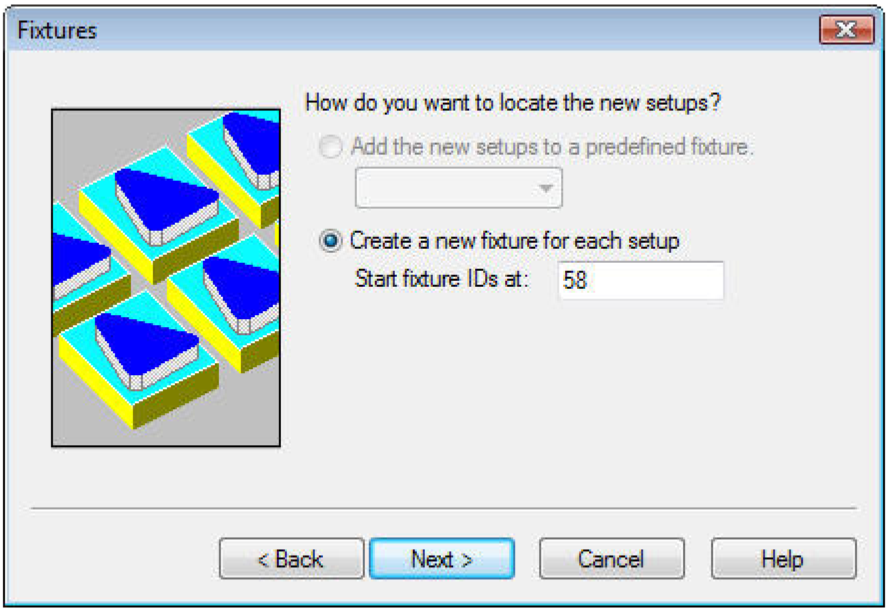
This time, we will add some Bottom setups to our layout.

- Select the **CALIPER/bottom** setup
- Click **Next**



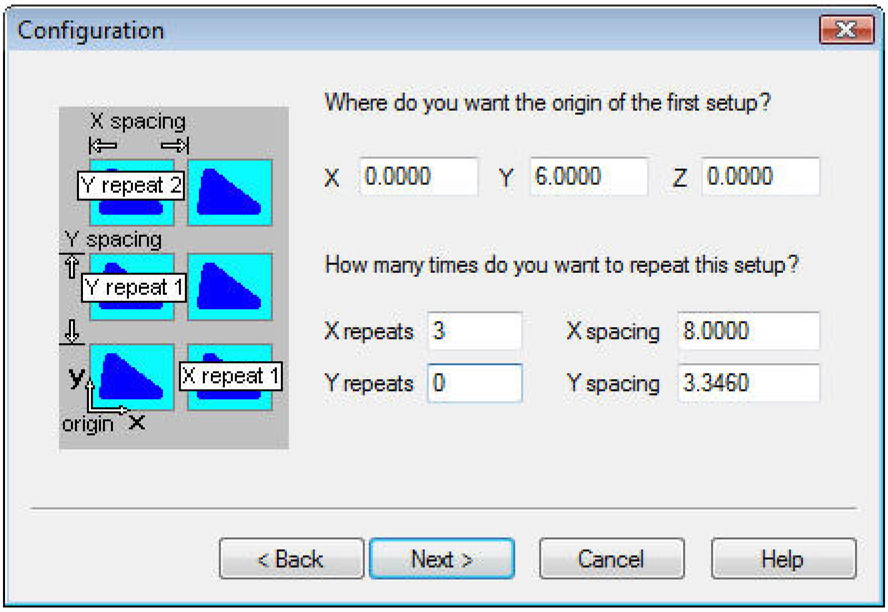
Notice how the Fixture IDs have picked up where we left off at G58. Of course, this can be overridden, but we will keep the default value

- Click Next



On the configuration screen we have an opportunity to locate our next setup and determine the repeats and the spacing of the new part.

- Enter **X0, Y6, Z0** as the position of the next setup
- Enter **3 X Repeats** with a spacing of **8** inches
- Click **Next**

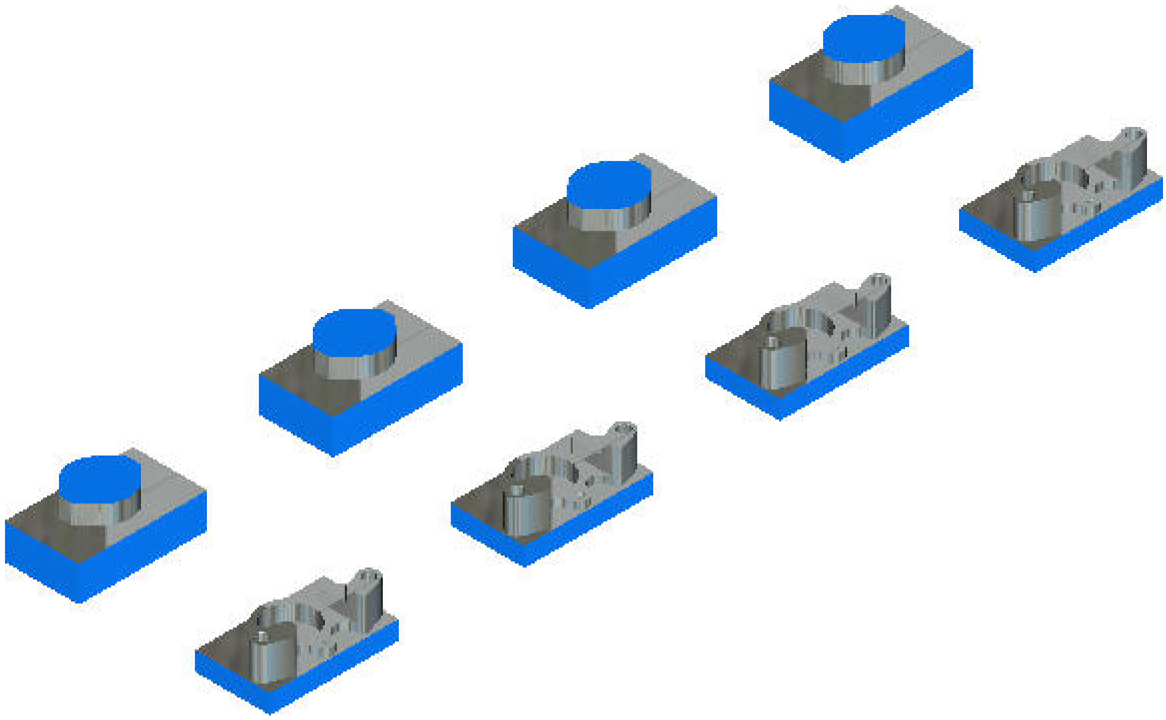


- On the **Layout** screen, select **Individual Blocks**
- Click **Next**
- Examine the Preview and click **Finish**

Again, we see the summary of our efforts detailed on the Multiple Fixture screen. The setup IDs are based on the available registers in the post processor.

- Click **OK** to exit

- Run a Simulation



- **Close** the part

Example 2 - Multiple Parts out of a Single Block of Material

In this exercise, we will create an array of parts that are cut out of a single block of material.

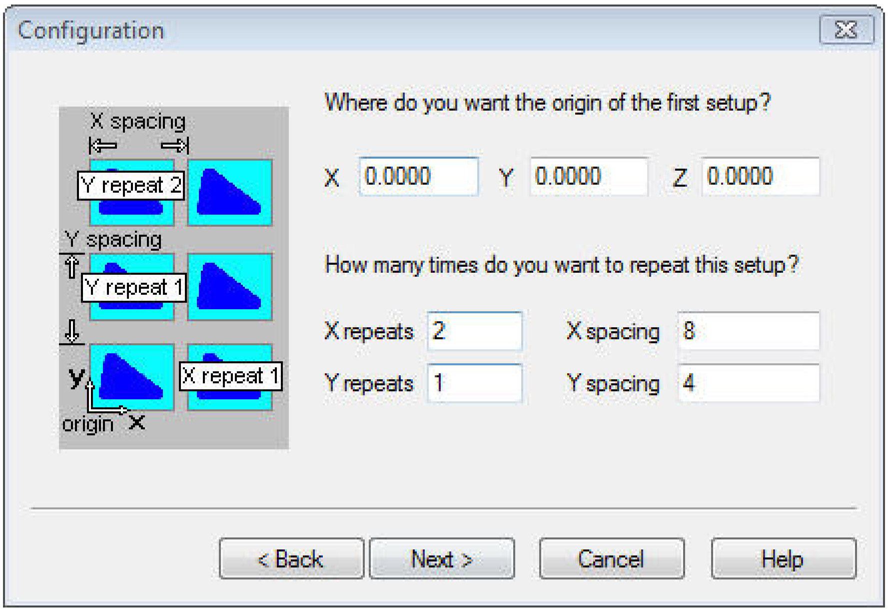
- **Open** a **new** Multiple Fixture file
- In the Multiple Fixture window (Parts Step) click on the **Parts List** button
- **Browse** to the **Caliper.fm** part in the **Data** folder
- Click **OK** to accept the Caliper and return to the Multiple Fixtures window
- Click **Add**
- Choose **CALIPER/top** from the available setups and click **Next**

Since we are going to machine all of these parts from a single block of material, there will only be one fixture offset used.

- Accept the default Fixture ID by clicking **Next**

We are going to lay out two rows of parts in this example. To do this, we will enter values for both the X and Y repeats. In this situation, spacing will also be critical since we must allow room for the tooling to move between parts without gouging.

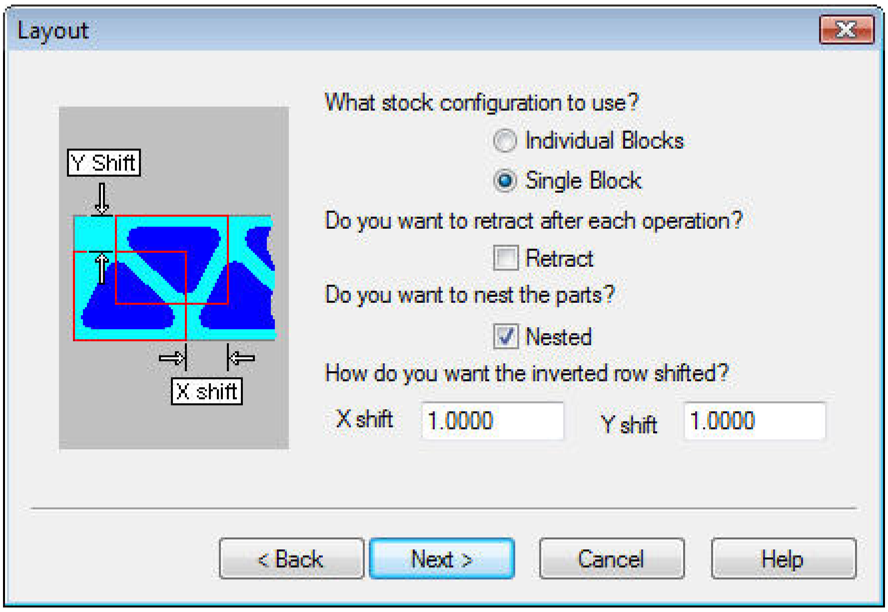
- Fill out the **Configuration** screen exactly as shown
- Click **Next**



- Select **Single Block** on the Layout page

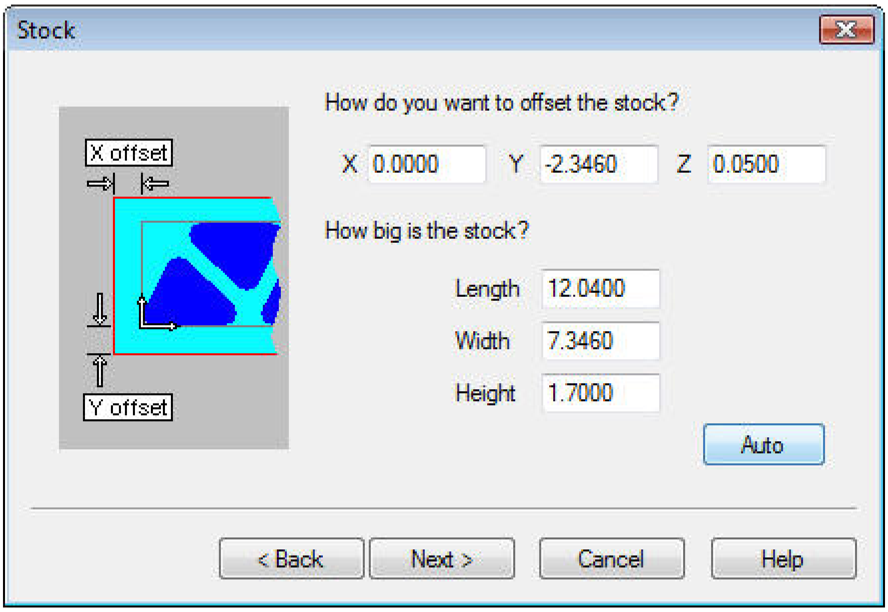
When you chose a single block, FeatureCAM gives you an opportunity to nest the parts by inverting alternating instances of the part.

- Check the **Nested** box and click **Next**



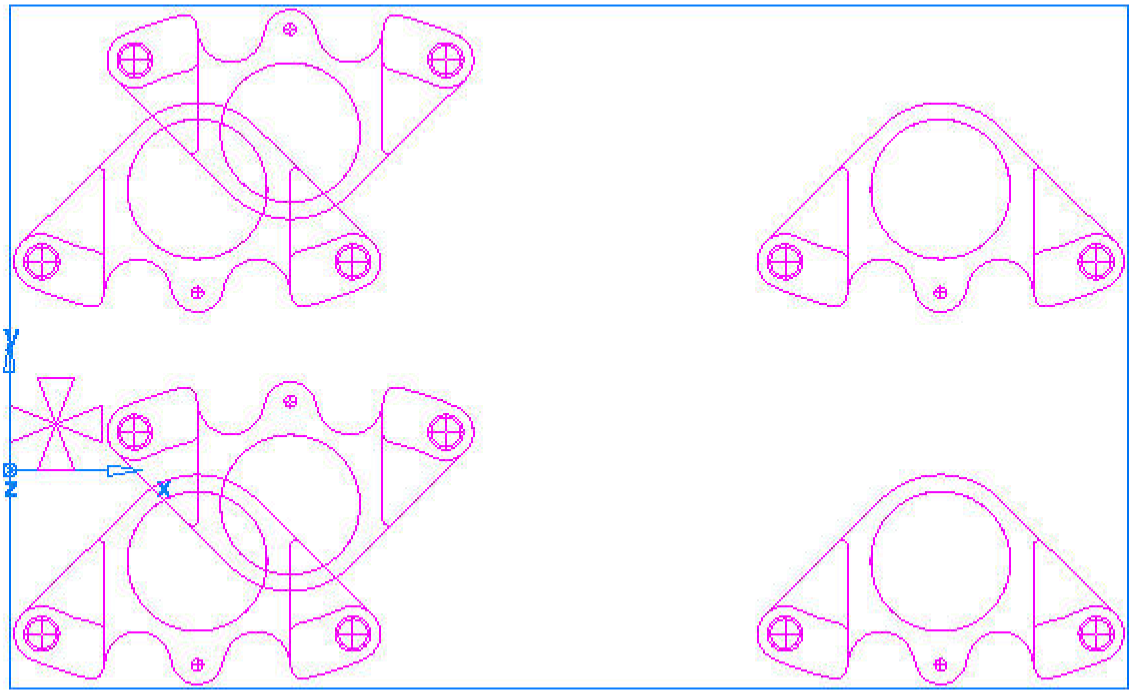
Now, the stock size must be determined. The user has a choice to enter the stock size manually or let FeatureCAM automatically calculate the required stock size.

- Click **Auto** to let FeatureCAM calculate the stock size



- Click Next to advance to the Preview page
- Click Finish to exit the wizard
- Click OK to close the Multiple Fixtures window and view the results

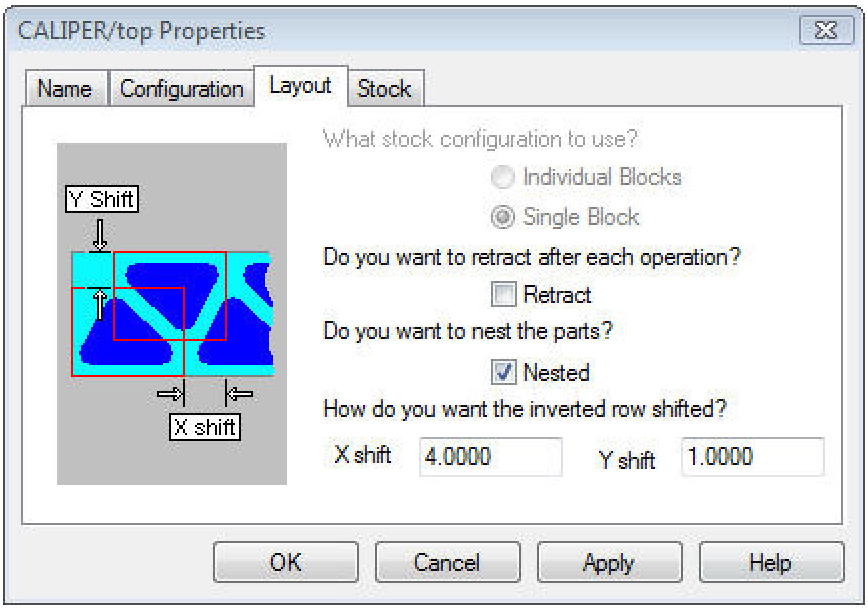
As you can see from the results, the spacing of our nesting is incorrect. This was visible from the preview page, but this will demonstrate how Multiple Fixture layouts can be edited after the fact.

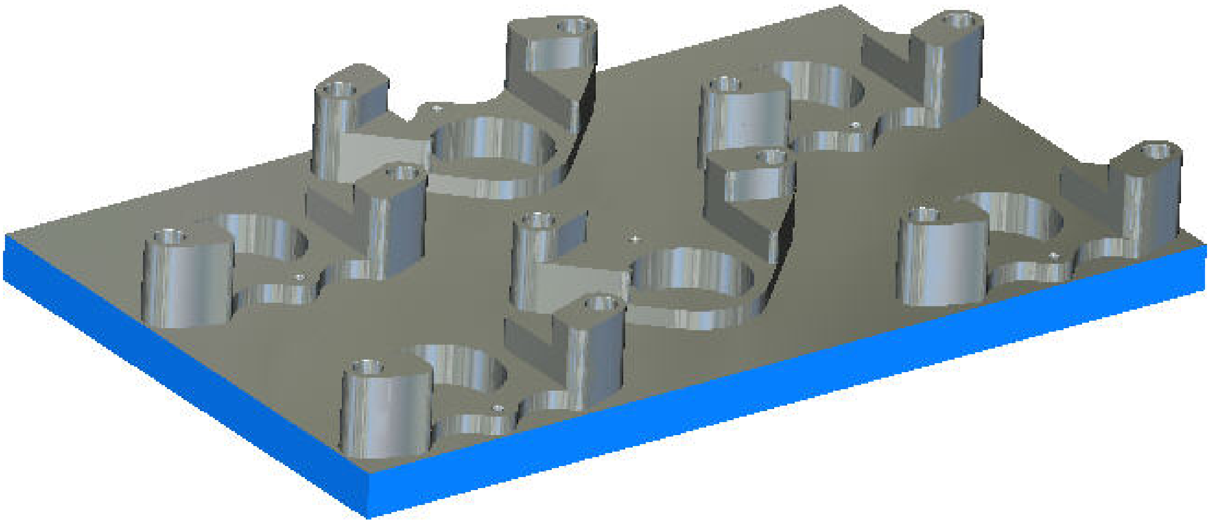

.

- Click on Parts in the Steps window to reopen the Multiple Fixtures window
- Highlight the setup and click Edit

The Edit function will open the properties of our layout and give us access to all of its parameters.

- Click on the **Layout** tab
- Change the spacing of the nest as shown
- Click **OK** to exit
- Run a simulation to see the results





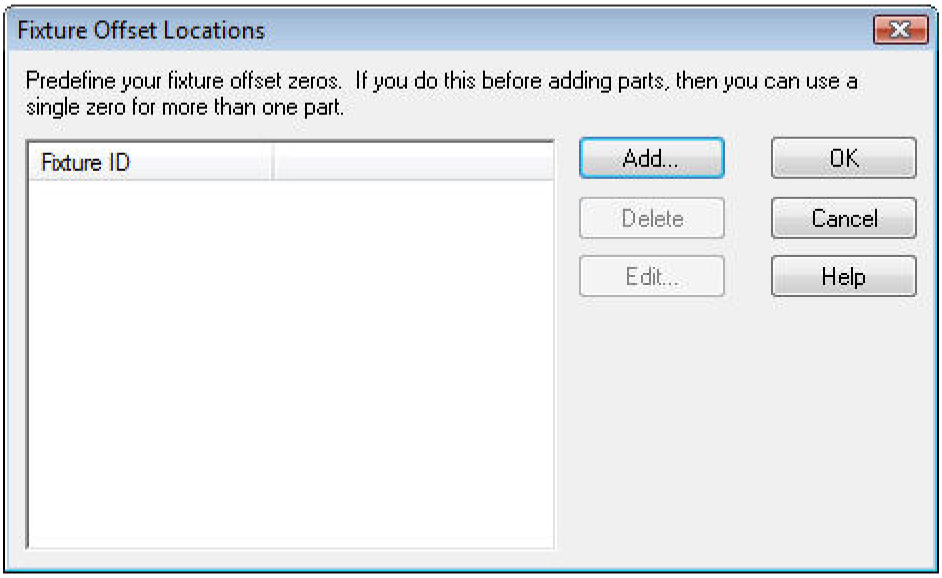
- **Close** the part

Example 3 - Multiple Parts Located on Predefined Fixture IDs

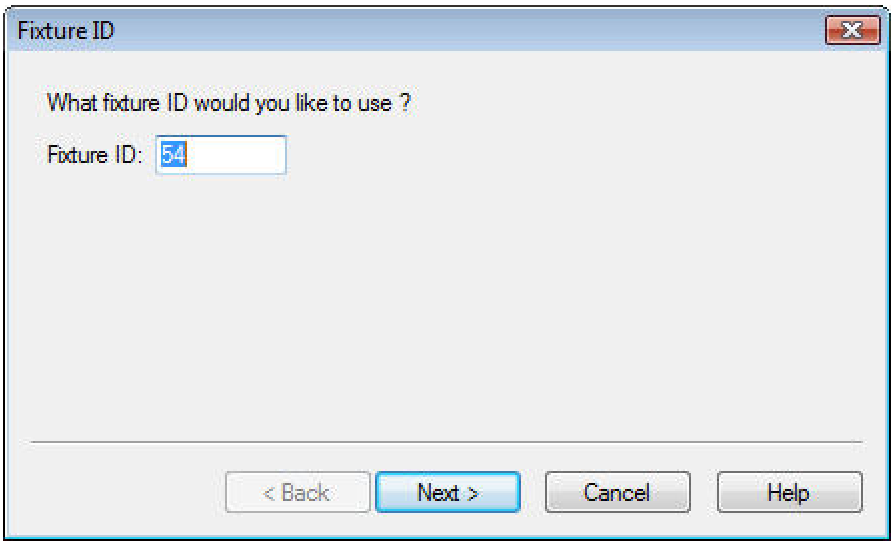
The next example will demonstrate how to create predefined Fixture IDs and locate parts on them. Keep in mind that the locations for the setups do not automatically translate to the machine tool. Each fixture ID location must be determined at the machine and entered into the control.

- **Open** a **new** Multiple Fixture file
- **Click** the **Fixtures** button in the Multiple Fixtures box (Note: This is the same as clicking Fixtures in the Steps view)

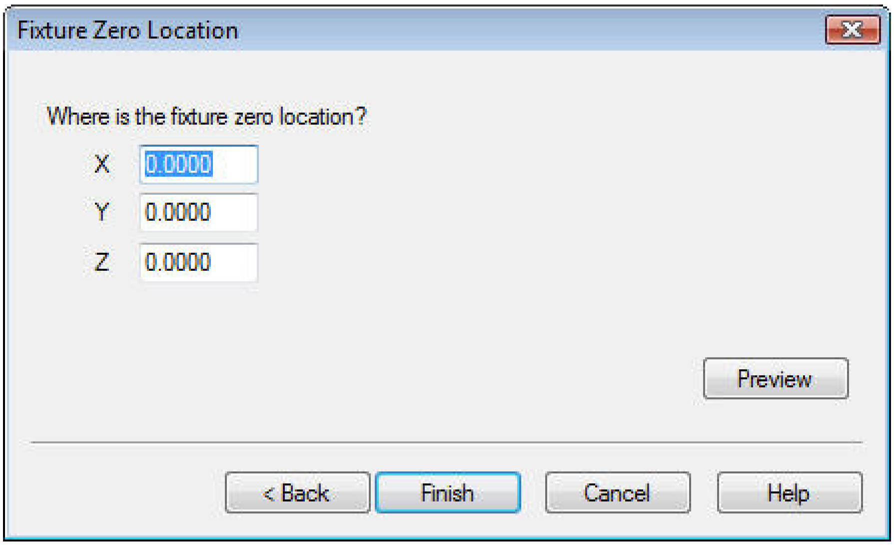
- Click the **Add** button to begin defining a new fixture location



- Accept 54 as the first fixture location by clicking **Next**

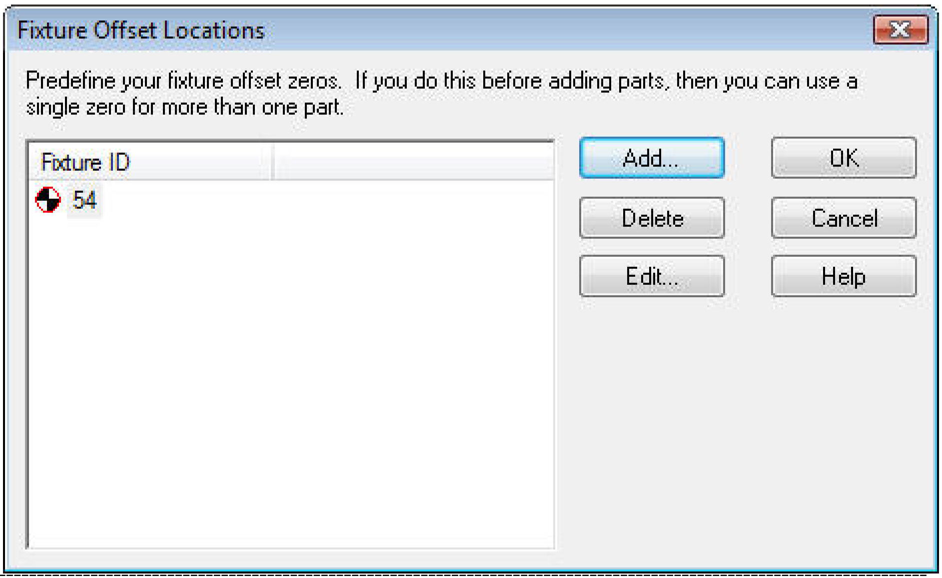


- Accept 0,0,0 as the location for offset 54 by clicking **Finish**

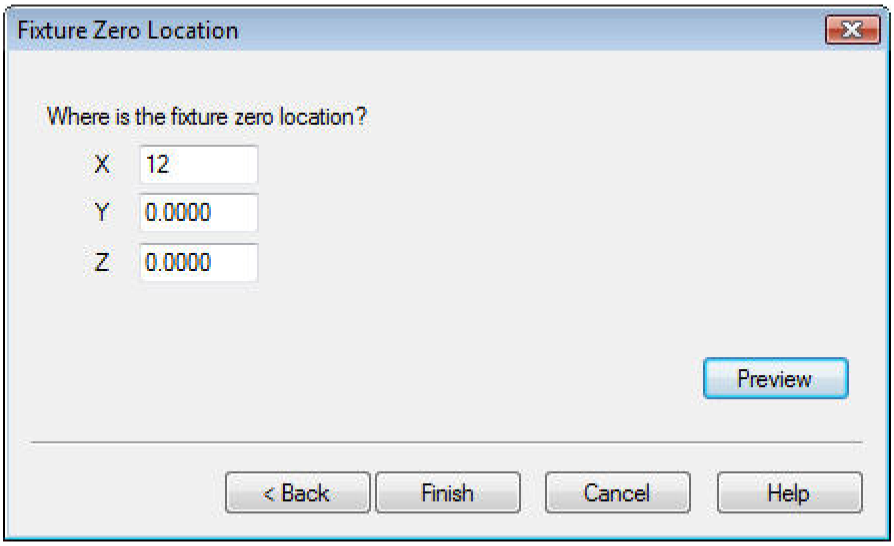


Below, you see that Fixture ID 54 has now been defined and is ready to be used. Before we start adding parts to this file, we will create a second setup location.

- Click **Add** to create another Fixture ID



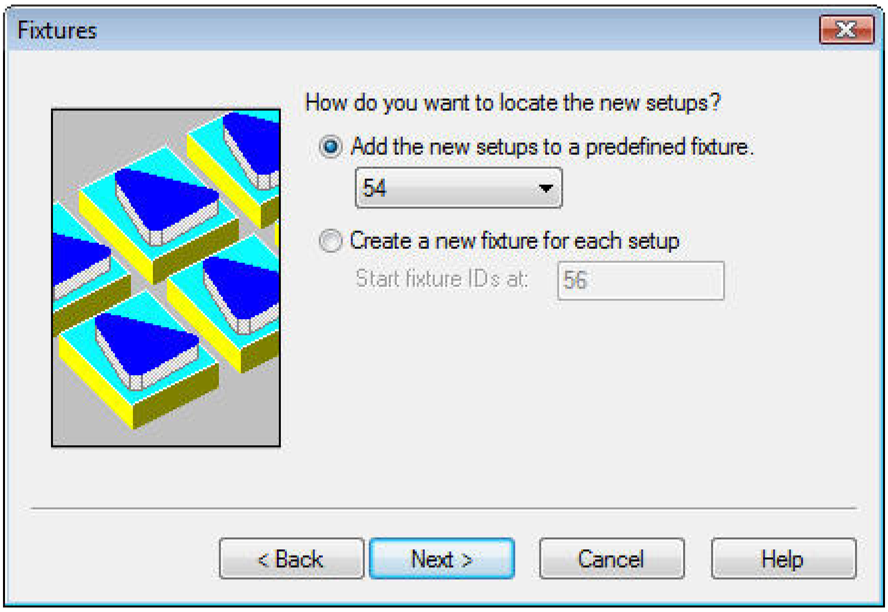
- Accept 55 as the next Fixture ID by clicking **Next**
- Enter the location of this Fixture ID as shown and click **Finish**



- Click **OK** to exit the Fixture Offset Locations screen and return to the Multiple Fixtures window
- Click on the **Parts List** button an Browse to the **Caliper.fm** file in the **Data** folder
- Click **OK**
- Click **Add**
- On the **Setup** screen, select the **CALIPER/top** setup and click **Next**

Now that we have predefined fixture locations, we are given the opportunity to select them as locations for out parts.

- On the **Fixtures** screen, select **Add the new setups to a predefined fixture** and verify that **54** is selected
- Click **Next**



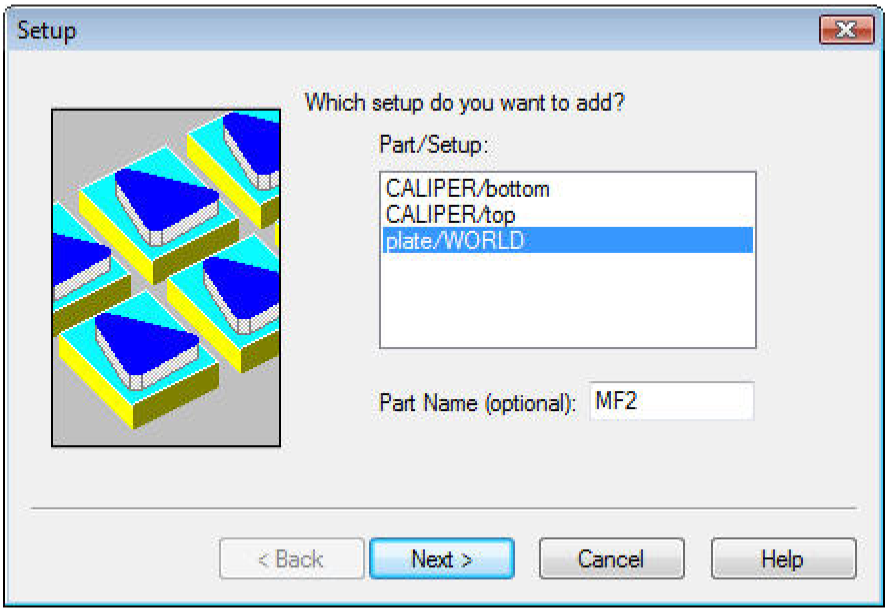
Notice on the Configurations screen that you still have the opportunity to offset the part origin from the Fixture ID, and you are able to create both X and Y repeats from that location.

- Accept the default offset (0,0,0) and make sure the there are **0 X repeats** and **0 Y repeats**
- Click **Next**
- Click **Finish** to accept the single part located at Fixture ID 54 and return to the **Multiple Fixtures** window

For the next step, we will bring an entirely different part to locate at Fixture ID 55.

- Click **Parts List**
- Click **Browse**
- Select the **plate.fm** file located in the **Data** folder and click **Open**
- Click **OK** to return to the **Multiple Fixtures** window
- Click **Add**

- Select **plate/WORLD** from the Part/Setup list
- Click **Next**

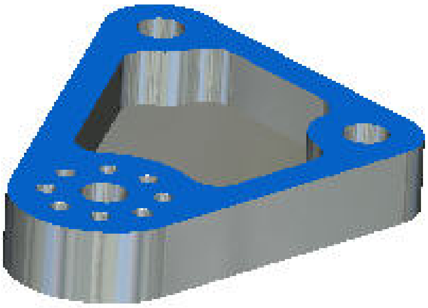


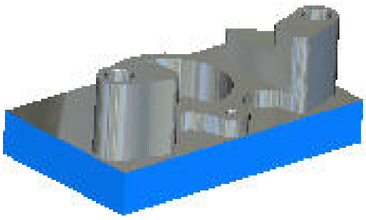
- On the **Fixtures** screen, select **Add the new setups to a predefined fixture** and select 55 from the drop-down list
- Click **Next**

- Accept the default offset (0,0,0) and make sure the there are **0 X repeats** and **0 Y repeats**
- Click **Next**

- Click **Finish** to accept the single part located at Fixture ID 54 and return to the **Multiple Fixtures** window

- Click **OK** and run a simulation





- **Keep this part open** for the next section!!!

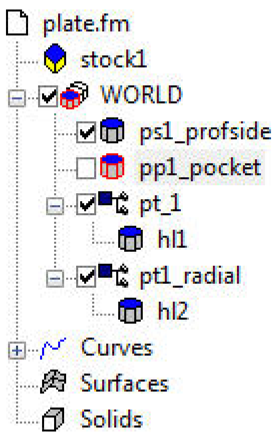
Example 4 – Making Changes

As mentioned earlier, there are very few changes that can be made to files programmed in a Multiple Fixture document. As we have seen, layouts can be modified by accessing the Parts step, but many of the other functions normally available in an fm file are dimmed. This final example will show how modified parts are handled in a Multiple Fixture file.

For this example, we will modify the plate.fm file.

- Making sure not to close the mf file, click File > Open, and open the plate.fm file located in the Data folder

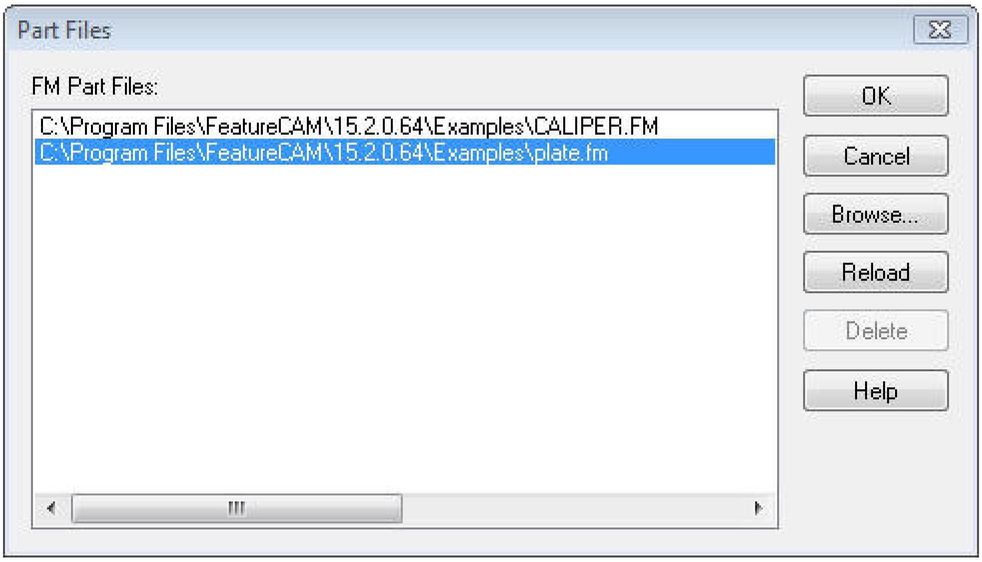
We will make a very simple change and see how the mf document is affected.



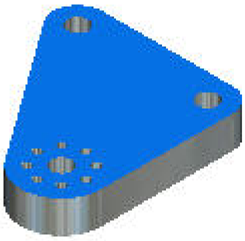
- Turn off the pocket feature by unchecking the **pp1_pocket** feature in the Part View
- **Save** and **Close** the file
- Once back in the mf document, run a simulation

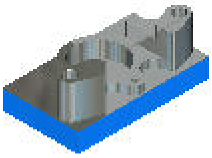
Nothing has appeared to have changed, has it? There is one more step that must be performed in order for the part to be updated in the Multiple Fixture document.

- Clear the simulation
- Click on the Parts step in the Steps view to open the Multiple Fixtures window
- Click the **Parts List** button
- Highlight the **plate.fm** file and click the **Reload** button



- Click **OK** to return to the Multiple Fixtures window
- Click OK to exit
- Run a simulation



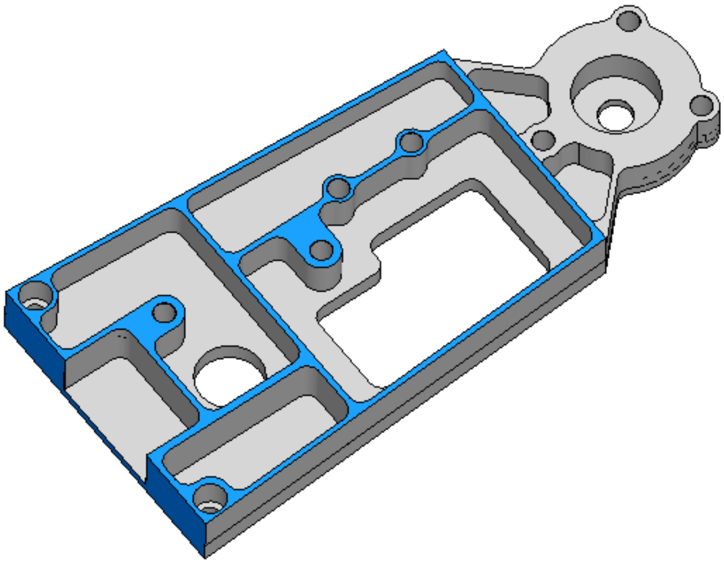


This is the procedure that will be required every time a change is made to how the individual part is machined. None of the parameters controlling Attributes, Tooling, etc. can be edited from the Multiple Fixture document, so the original part file must be opened, edited, and reloaded into the mf file.

FeatureRECOGNITION 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, Interactive Feature Recognition and Feature Re-recognition.



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, center drill, drill, pre-drill, tap, roughing, semi-finish and finish passes. FeatureCAM takes, for example, a tapped hole and combines the operations like the center drill, drill and tap and automatically selects the necessary tooling, feeds, speeds, etc.

What is Feature Recognition. Why is it necessary?

Feature Recognition is 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 necessary 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 necessary is that in modern manufacturing a majority of the design software used for part design is solid or surface based. The 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 F/S, tools, toolpaths and NC code

The third reason FeatureRECOGNITION is necessary 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 FeatureRECOGNITION Work?

There are two different types of FR: Automatic Feature Recognition (AFR) and Interactive Feature Recognition (IFR). Both tools search the CAD data (cylinders, surfaces, and faces) and matches 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

In order to use Feature Recognition a surface or Solid Model is required. There is a wide variety of Import formats supported:

- Iges, Acis, Parasolid, SolidWorks, Inventor, SolidEdge
- **Optional import plugins:** Catia V4 & V5, Step, Unigraphics and ProE

There are several ways that the model can be imported directly into FeatureCAM:
- Click on **File** then **Open**.
- Click on **File** then **Import**.
- Locate the file, **left click** on the file to import, **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).

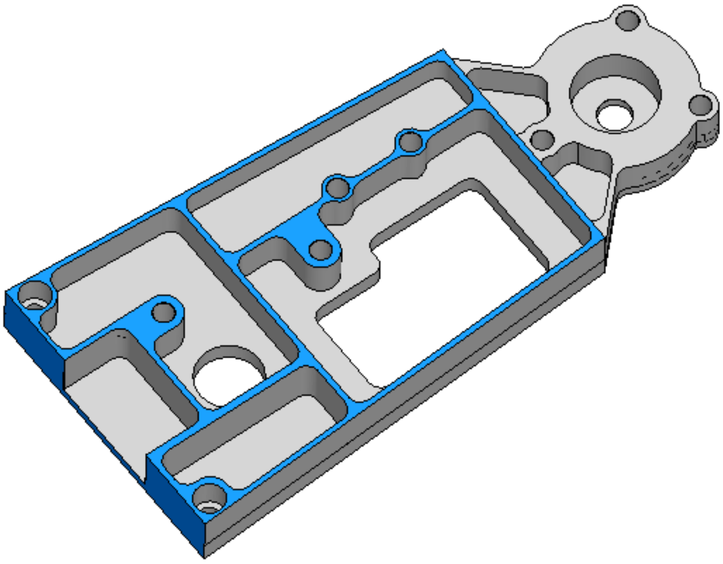
Import Wizard

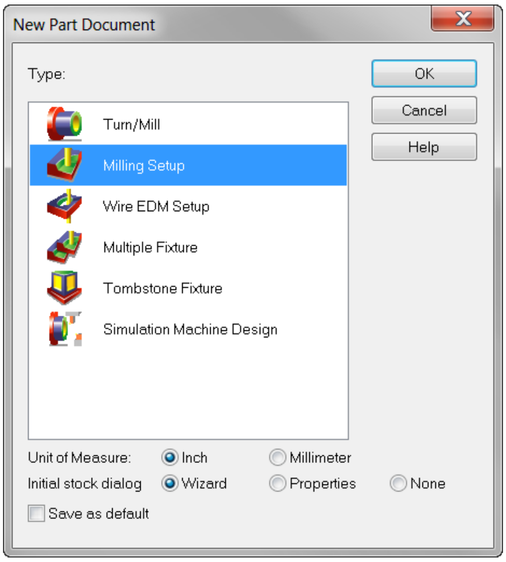
When you import a file, FeatureCAM steps you through its import wizard. The import wizard pops up automatically and assists you in importing the file into FeatureCAM. This wizard helps to:

1. Import the file into FeatureCAM
2. Size the stock
3. Orient the stock
4. Position the part program zero
5. Set up a milling part for indexing
6. For some solid file formats, it even helps you recognize and suppress some part features

AFR Example #1

- **Click** on **File** then **Open**. Under **Files of type** at the bottom of the screen select ParaSolid (*.x_t, *.x_b). Find the file in the FeatureRECOGNITION Data folder named **ex02ex02.x_t** then press the **open** button.

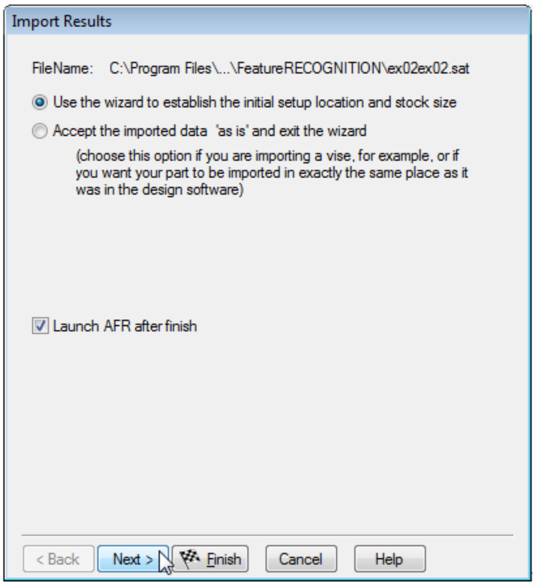




- If asked, select **Scale the imported geometry into the document's units**.

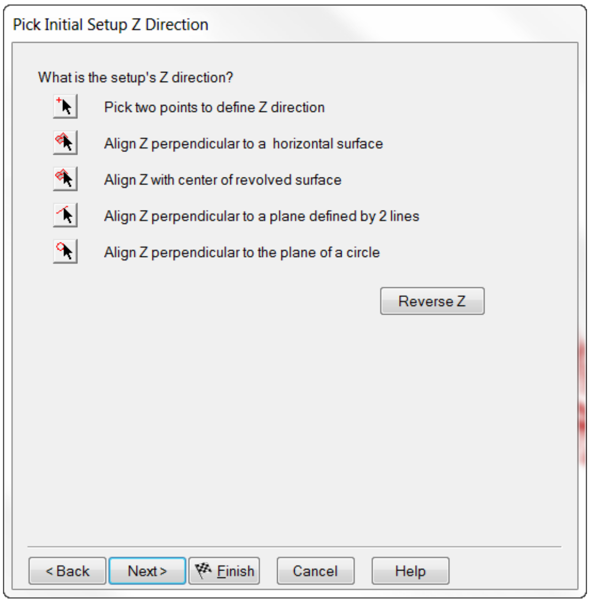
- Select **Milling Setup** and select the Unit of Measure to be **Inch** then press **OK**.

- Select the button "**Use the wizard to establish the initial setup location and stock size**" then check the "**Launch AFR after finish**" checkbox. **Next**

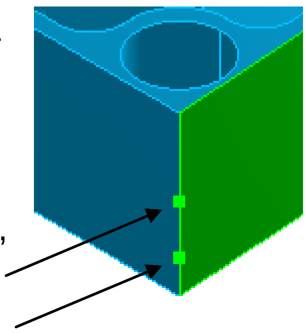


To aid in aligning the model, when the model is first brought into the Graphics Window, right click and view Top or Isometric and Shade the model so you can see how it is positioned in the window. (**Shade**-upper tool bar-button with two little red barrels) 

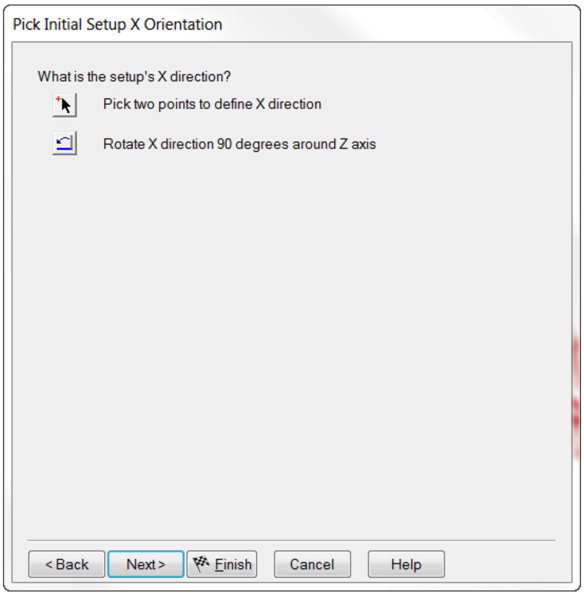
Because the CAD model may have been saved in a position that is not aligned with your Z and X axis, FeatureCAM in the "**Pick Initial setup Z Direction**" form provides five tools to assist in aligning the model to your document in the Z direction, which in turn aligns to the machine spindle, table and axis.



- **1)** Pick two points to define Z direction. Click on arrow then click twice on a vertical surface edge starting low, then the second click **above** will point to the spindle.



- **2)** 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.
- **3)** Align with center of a revolved surface such as a hole by selecting the arrow and clicking on the round surface.
- **4&5)** can be used even when geometry alone is imported by clicking on the defined geometry. **Next.**

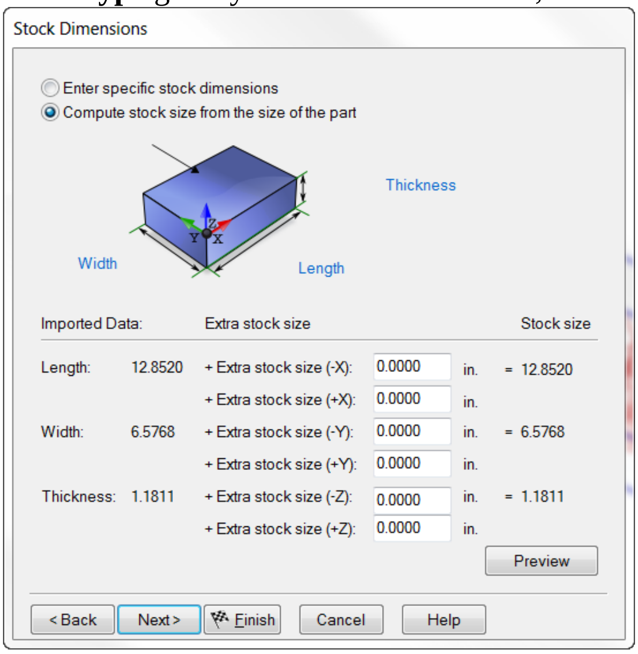


"**Pick Initial Setup X Orientation**" offers assistance to align the setups X direction. The instructions below does not rotate the part, you must do a top or isometric view for the model to move.

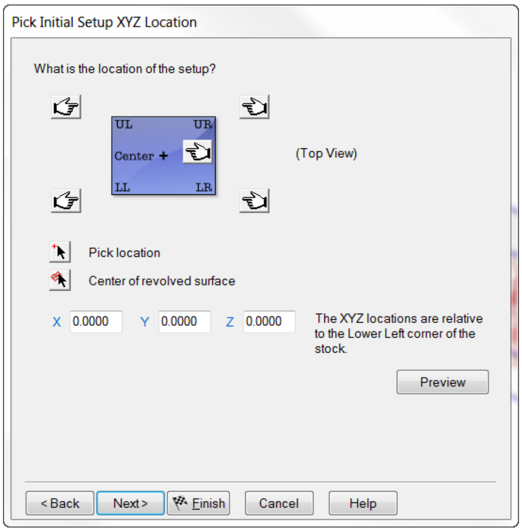
1) Pick two points to define X direction works the same as the Z, except the edge you click on will be parallel to the table.

2) Rotate X direction 90 degrees around Z axis will rotate the model 90 degrees for each click, three times will rotate 270 degrees. **Next.**

Stock type gives you the choice of Block, Round or N-Sided such as hex stock.



- On the **Stock Dimensions** page check the radio button, "**Compute stock size from the dimensions of the part**". Enter 0 in all dimension boxes. Additional stock may be added on all sides. **Next.**

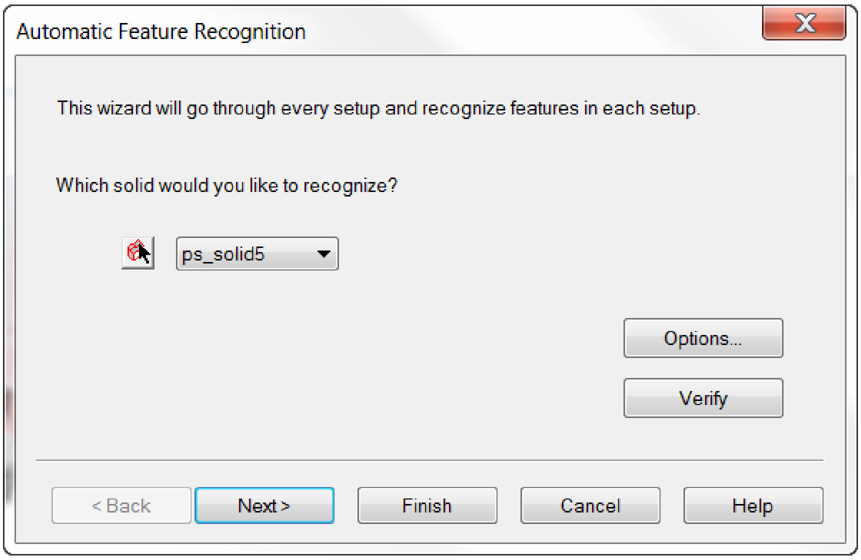


- On the "**Pick Initial Setup XYZ Location**" form (Part program 0) select **LL** (lower left from top view). You can also use the pick arrow and click on a location, use the center of a revolved surface or enter the XYZ location measured from the UCS. **Next.**

- **No** 4 axis Indexing. **Finish.**

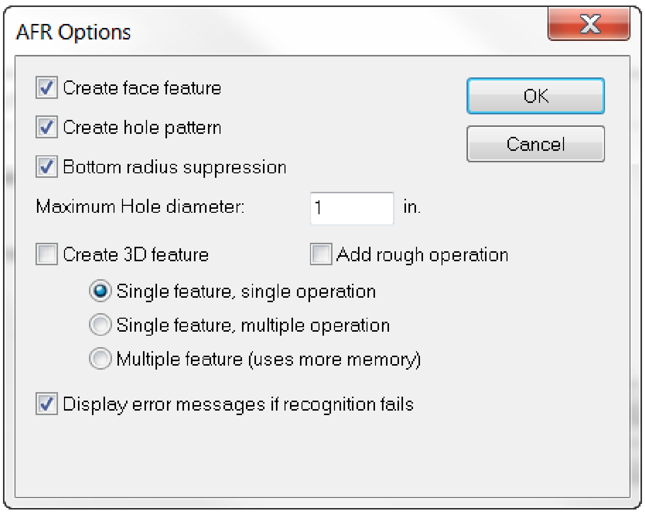
Because the “Launch AFR after finish” box was checked on the initial Import form, the **Automatic Feature Recognition** pops up.

It identifies the solid just imported.



- **Click** the **Verify** button to assure you have a good Solid Model.
  (AFR does not work with Surface Models)

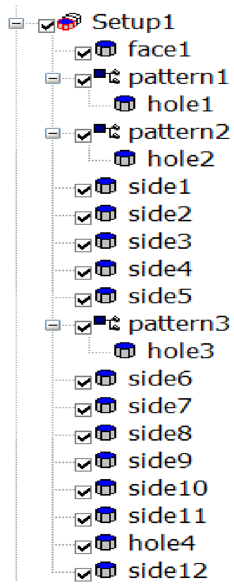
- **Click** the **Options** button.



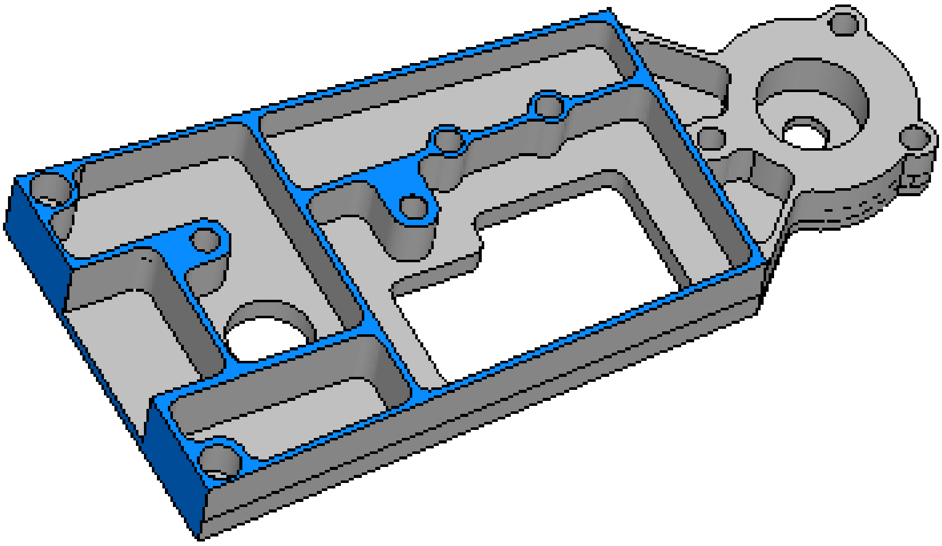
- o Create a facing feature.
- o Combine like holes into a pattern.
- o When checked, Bottom radius suppression will recognize bottom radiuses and when unchecked it skips them and leaves material there instead.
- o Maximum Hole diameter value is the largest hole AFR will create with a drill, anything larger uses an endmill.

(Disregard 3D for now)

- **Click** OK for Options. **Next.**
- Setups can recognize multiple setups. **Next.**
- Select features will allow you to accept all or pick and choose. **Finish.**
- Run the **3D simulation**.



ALL features, tooling, feeds, speeds, coolant, depth of cut and stepovers, etc, have been created for the entire part in this setup and looks like the finished part illustrated.



- **Click** on **File** then **Save as** and name the file **ex02ex02.fm**

Types of FR Available

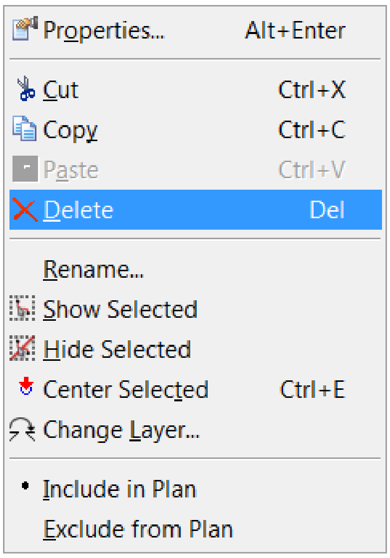
There are four types of FR available:

1. Fully automatic – "AFR" as in the previous example,
2. IFR Automatic by feature type,
3. IFR using surfaces or faces
4. IFR chaining.

Each of these is a different tool used increase the programming options available to you. Each technique has its own benefits and disadvantages.

AFR (Automatic Feature Recognition)

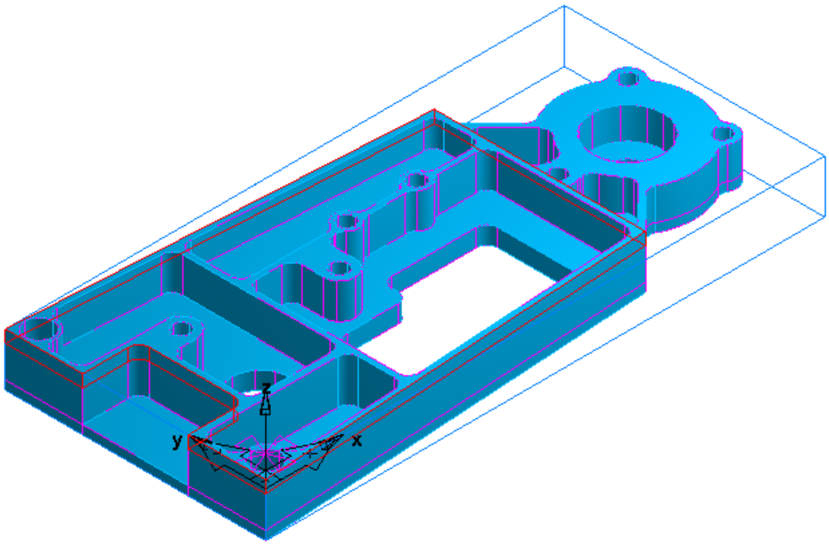
- **Open** the file saved previously **ex02ex02.fm**.
- In the **Part View** right-click on and **delete** the single **face** feature and **all of the holes** in setup1.



The reason all the face features and holes are deleted is so we can easily see all the side features that AFR created to cut the part. In order to completely cut the part (excluding face feature and holes) AFR uses side features. For Bosses, pockets and sides, **AFR always uses side features**.

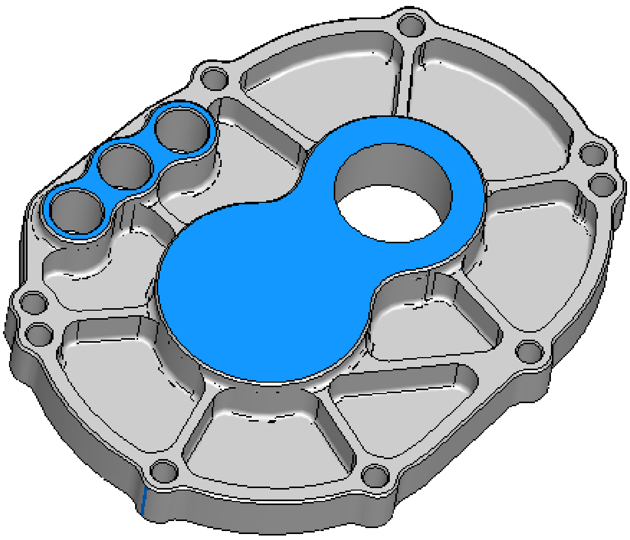
- On the top tool bar, **if the model is not already shaded**, shade the part by clicking on the **shading icon** (two little red barrels). 
- **Right-click** in the Graphics Window and select **isometric**.

- **Click** on each of the side features in part view one by one

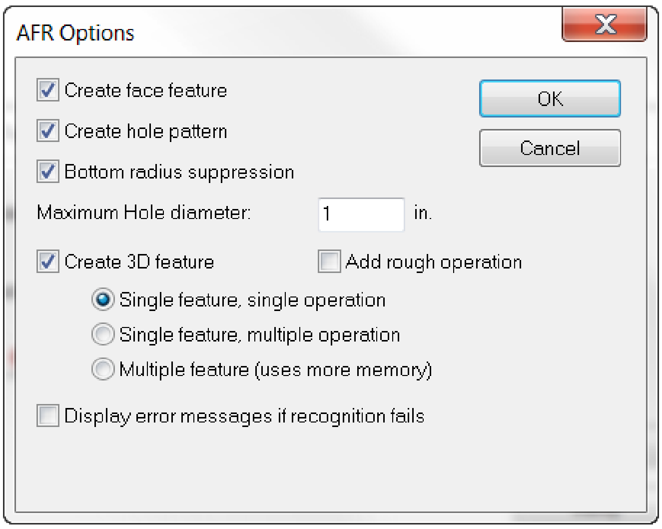


  - Notice that it creates a set of features that will completely cut the solid, but it may create more features than you might create if you have modeled the features yourself.
  - The way AFR works is that it examines the model and looks for any flats on the part. It creates features by dividing the model into horizontal slices at these flats and automatically determines side control. Any features (excluding holes) remaining will be cut using surface milling.
  - 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 disadvantage of this method is 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 note 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.

AFR Example #2

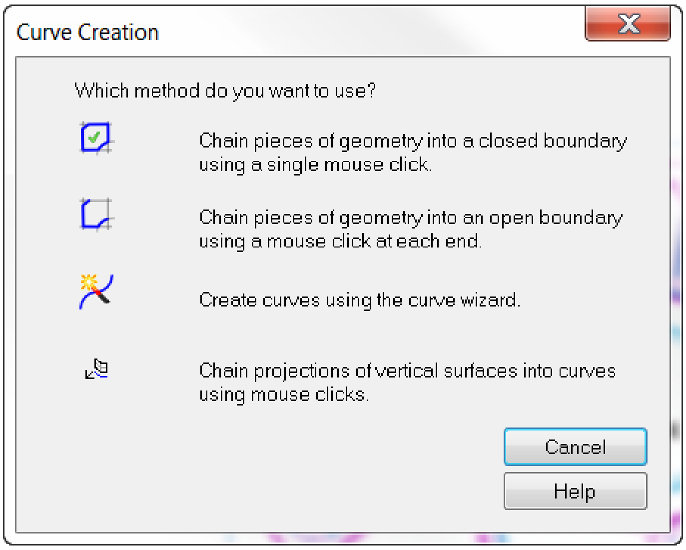


- Browse to the FeatureRECOGNITION Data folder, locate the file named **EndPlate.x_t** then **left-click** and **drag** into FeatureCAM.
- Select **Milling** Setup and **Inch** Units then press **OK**.
- Select the **top radio button** then check "**Launch AFR after finish"** then follow the same import procedure as the previous part.



- On the **AFR Options** page the **Maximum** hole diameter is set at **1 inch** so the through hole in the part will be machined with an endmill. If you also check the **Create 3D feature** check box FeatureCAM will apply a ball endmill to the chamfers. (FR does not do chamfers and you must be licensed for 3D surface milling). **Do NOT select 3D now.** Click **OK**.
- Run the **3D simulation**.

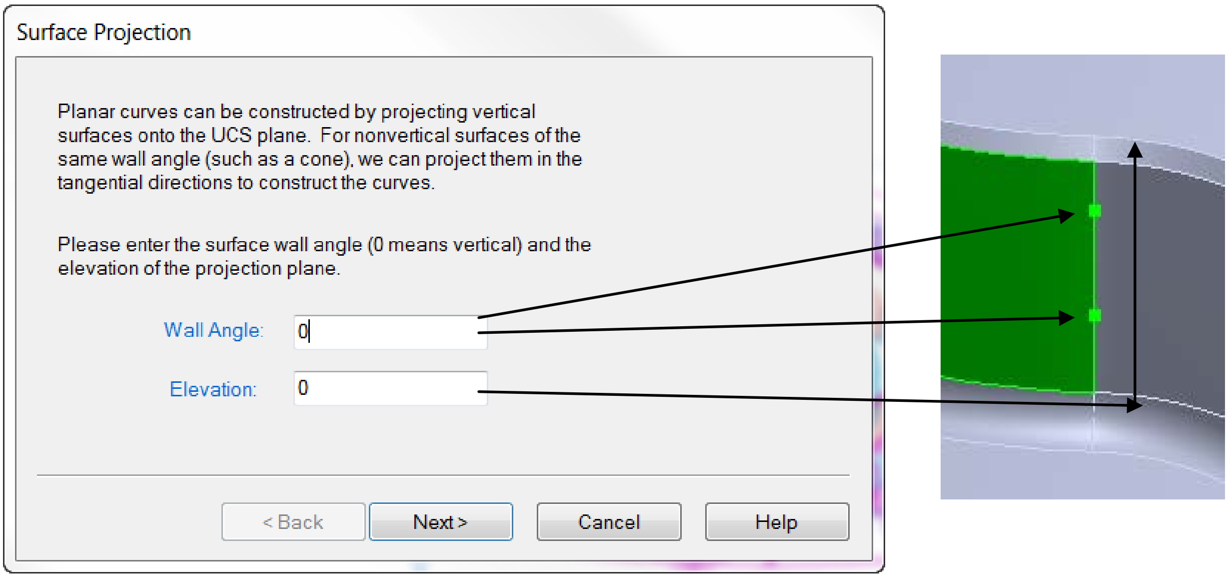
The chamfers on this model are standard 2.5D features and first require a curve, three in all. A curve can easily be created by projecting (**ALL** surfaces) geometry in the Z direction, this will include the shape required and will be located at the proper Z elevation you choose. During this process a curve can quickly be chained on only the geometry you need then all geometry is automatically deleted when curve chaining is complete leaving only the curve required for the feature. Multiple curves may be created by clicking the create button after each curve. If all curves are created at the same elevation in Z, any curve when selected may be transformed to the correct elevation.



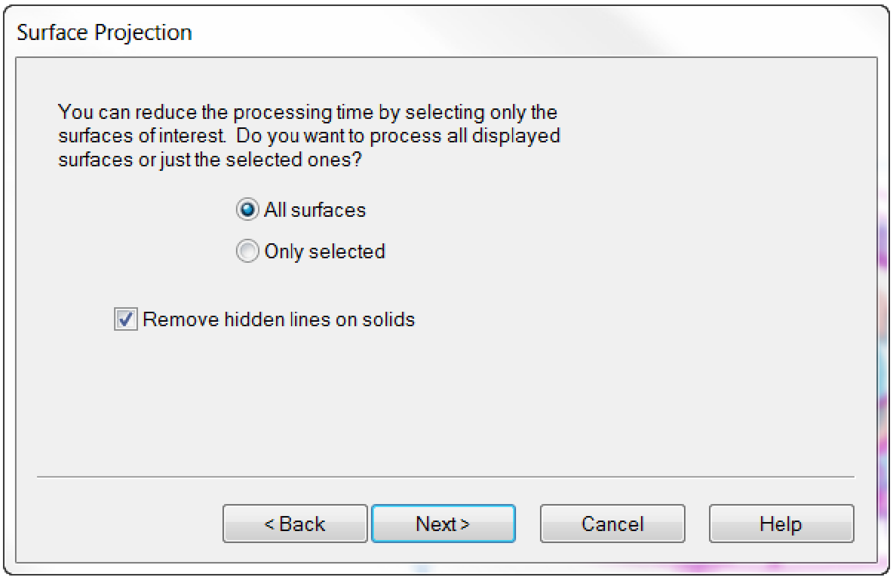
- Click on **Curves** from the **Steps Toolbox**, Click "**Chain projections of vertical surfaces into curves using mouse clicks**".

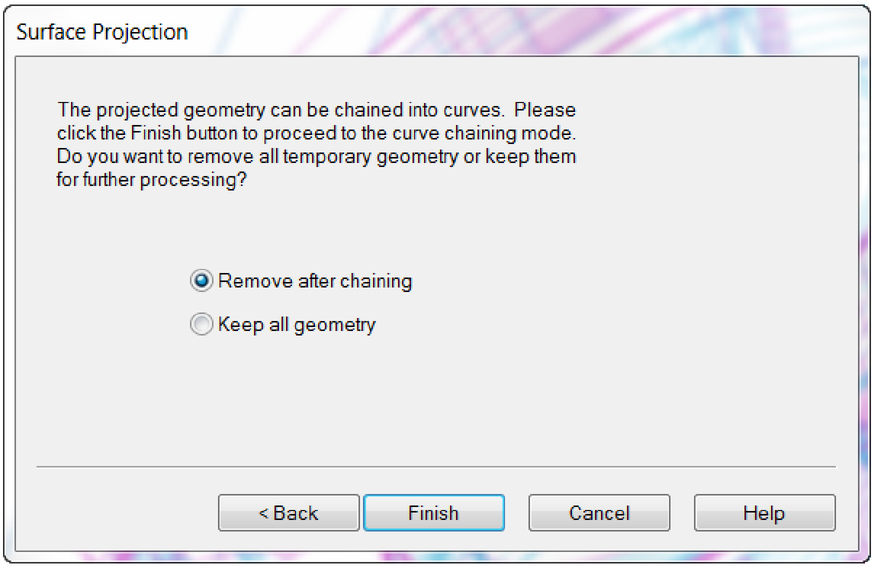
- The wall angle can be acquired by clicking on the Blue Words "**Wall Angle**" and then **click twice** on an **isoline** along the vertical edge of a surface as illustrated.

- Click on the "Blue Word" **Elevation** and then click once on any surface or edge that is the Z Elevation or height you desire. Chamfer curves must be at the top of the chamfer. **Next.**



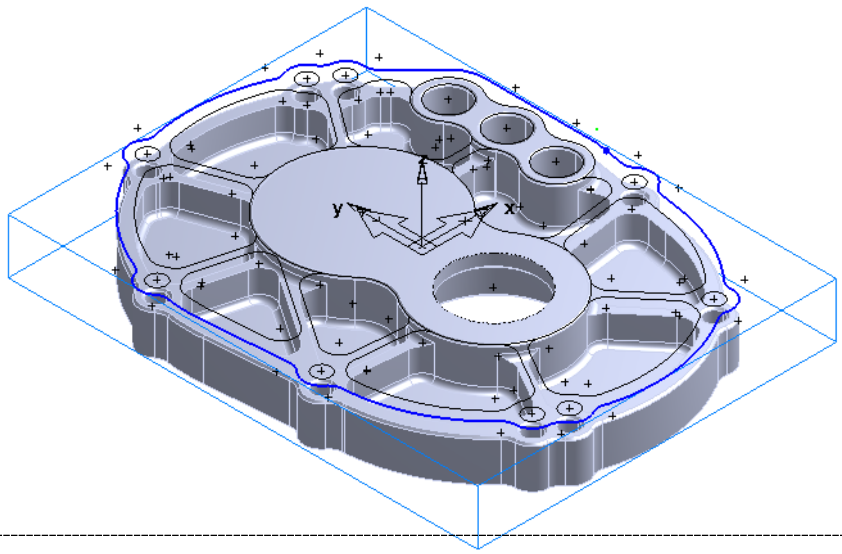
- Select “**All Surfaces**” and check the box “**Remove hidden lines on solids**”. **Next.**





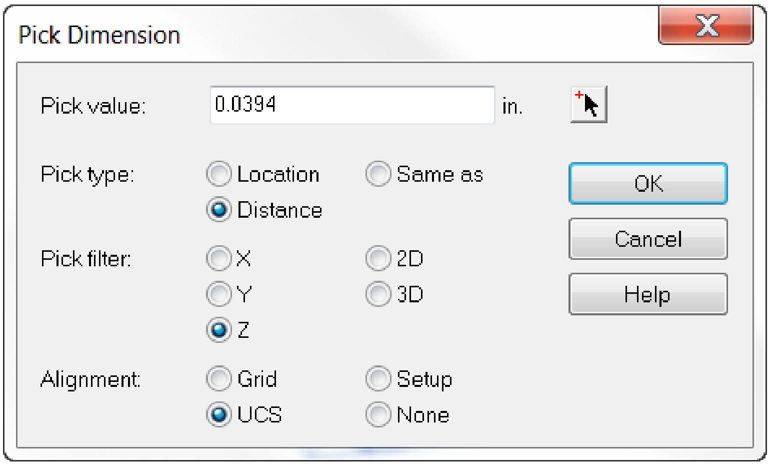
- Select “**Remove after Chaining**”. **Finish.**

The cursor now changes to the mode for chaining curves. Chain the desired geometry for the shape and elevation required for the chamfer curve. Geometry is deleted when the select arrow is clicked canceling curve chaining. A Chamfer Feature can now be created for each curve.

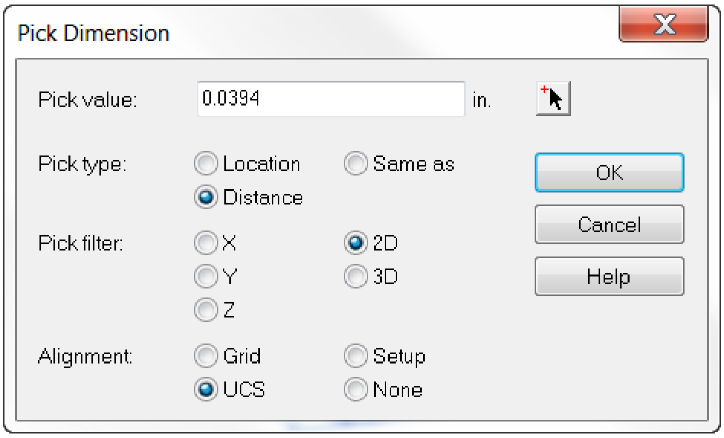


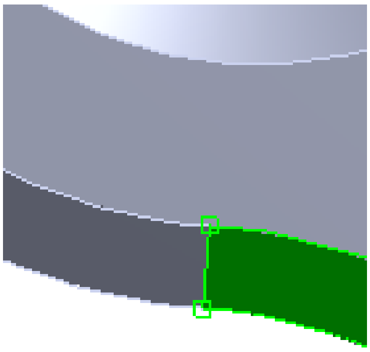
- To determine the size of the chamfer from the model, view from the **front** then use **Interrogation**. The **Interrogation** button may be obtained by clicking on **View>Toolbars>Command tab>Geometry**. Left click on the little icon with the yellow question mark then drag it and place in the tool bar docking area.

- With **Distance, Z,** and **UCS** selected use the arrow button and **click** on the **top edge** of the surface and the **bottom edge** to measure the distance in Z, this dimension is used for the **height** in the chamfer feature.

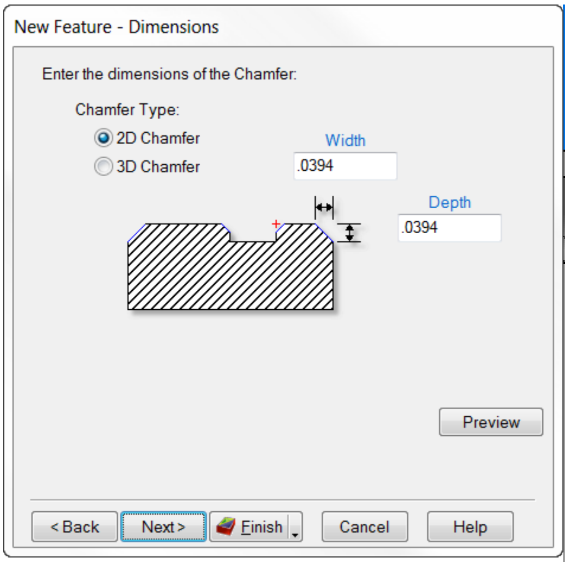


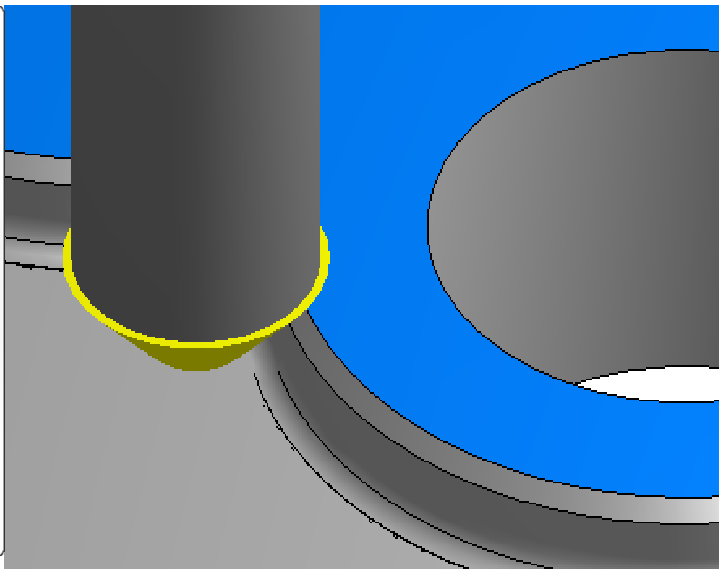
- To determine the width, view from the top and using **Distance, 2D,** and **UCS** click on each end of any isoline on the chamfer faces as shown for the width of the chamfer. If they are equal you of course have a 45 degree angle but if they are not remember this is a formed chamfer with a tapered tool and your tool crib may not have that angle tool, it will have to be created and purchased or ground to the taper.





- **Create a 2.5D chamfer**, no Recognition and use the **.0394** dimensions for both the **height** and **width** for each of the **three chamfers**.



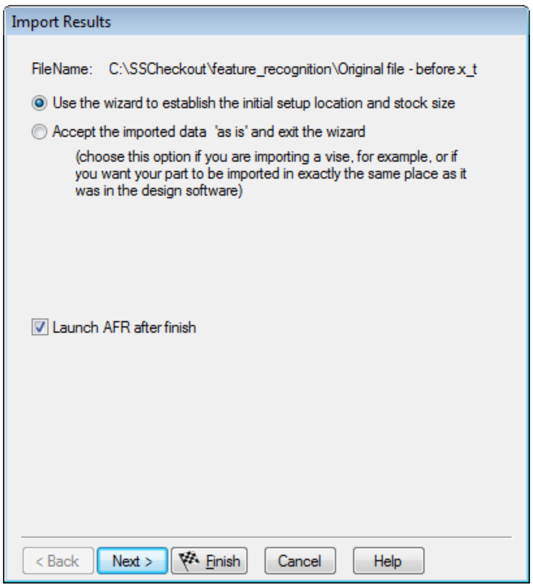


AFR Example #3 (Feature Re-recognition)

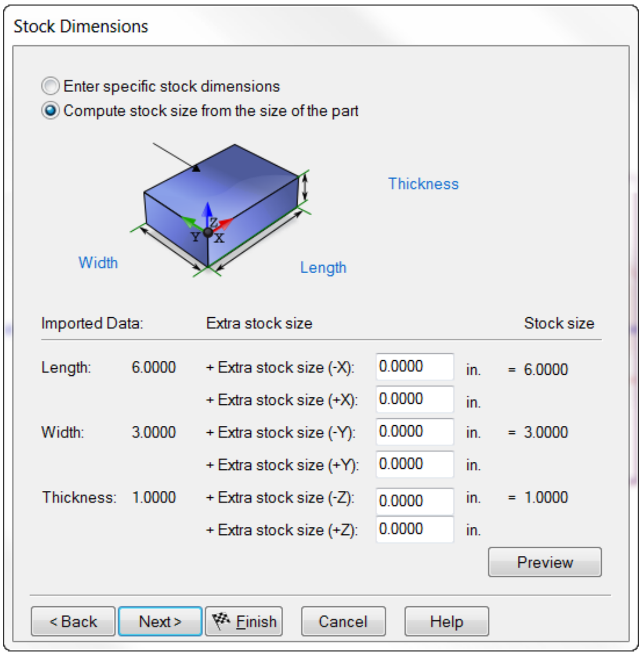
- From the file menu click on **File** then **Open**. From the **files of type** pull down menu select **ParaSolids (*.x_t,*.x_b)**. Select the file named "**Original file - before.x_t**" then press the **Open** button.

- In the **New Part Document** window select **milling** setup, **inches** then press **OK**.

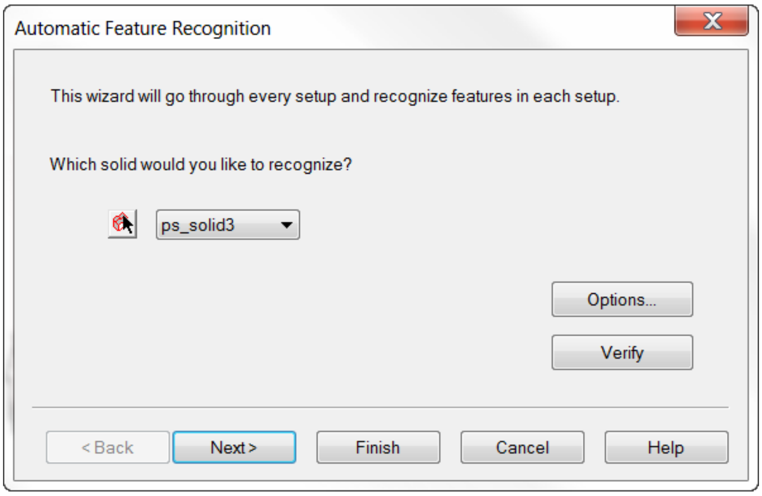
- On the **Import Results** screen select the **top radio button** and check the **Launch AFR after finish** checkbox then press **Next** four times.

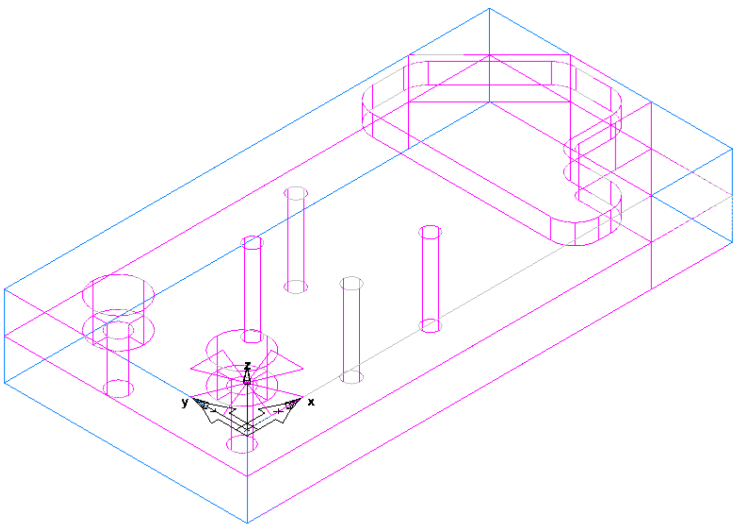


- On the **Stock Dimensions** page check **Compute stock size from the size of the part** then press **Finish**.



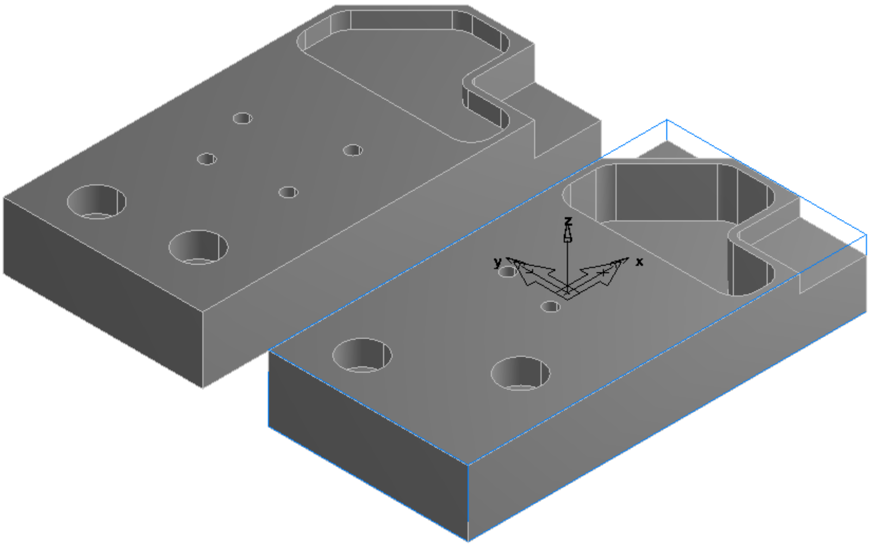
- On the **Automatic Feature Recognition** press **Next** then **Finish**.





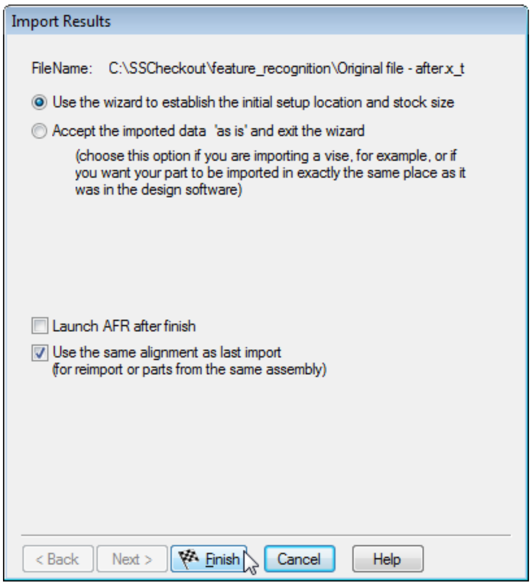
Feature recognition completes the part by identifying all the features in the setup.

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**.



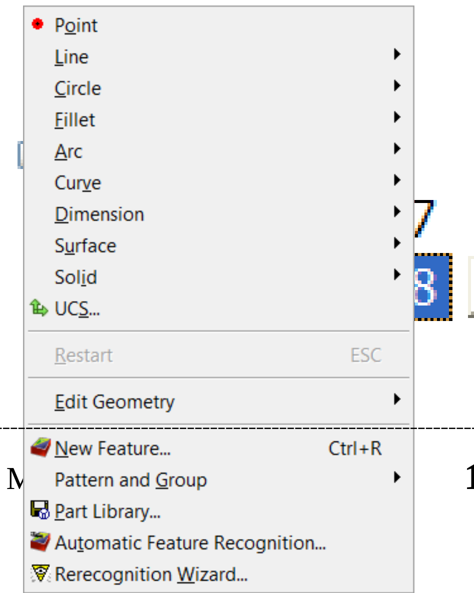
- From the file menu click on **File** then **Import**. Select the file named "**Original file - after.x_t**" then press **Open**.

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.

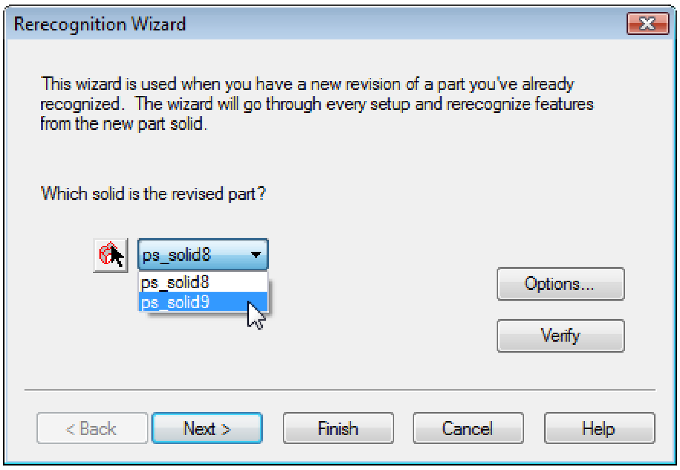


- On the **Import Results** window select the top radio button, **Uncheck Launch AFR after finish** and **Check** "**Use the same alignment as last import**".
  Finish.
  They merge together.

- 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 the solid will be highlighted in red.

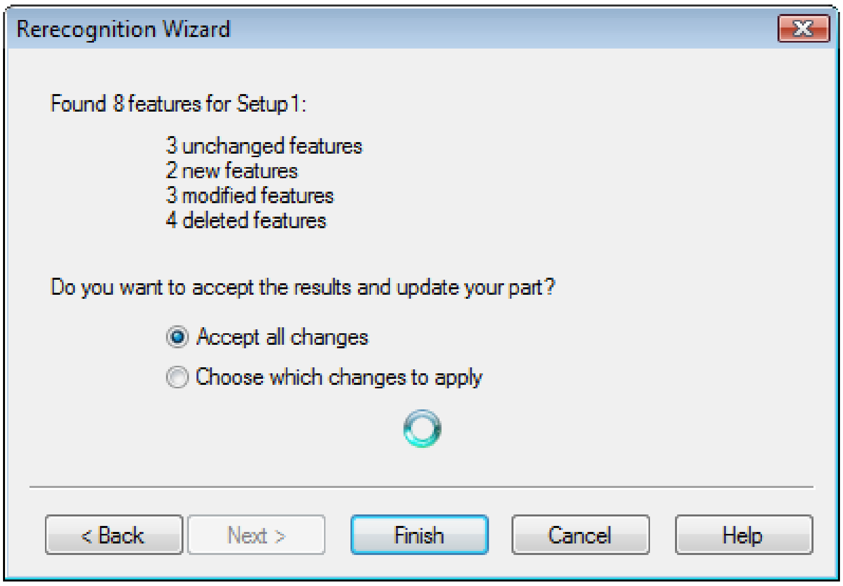


- Click on **Construct** from above then select **Re-Recognition Wizard** at the bottom of the menu.



- From the solid pull down menu select the second, last imported, solid in the list then press **Next** twice.

The Re-Recognition wizard will show what has changed. This is very useful so the programmer is not required to completely reprogram the part upon design changes.

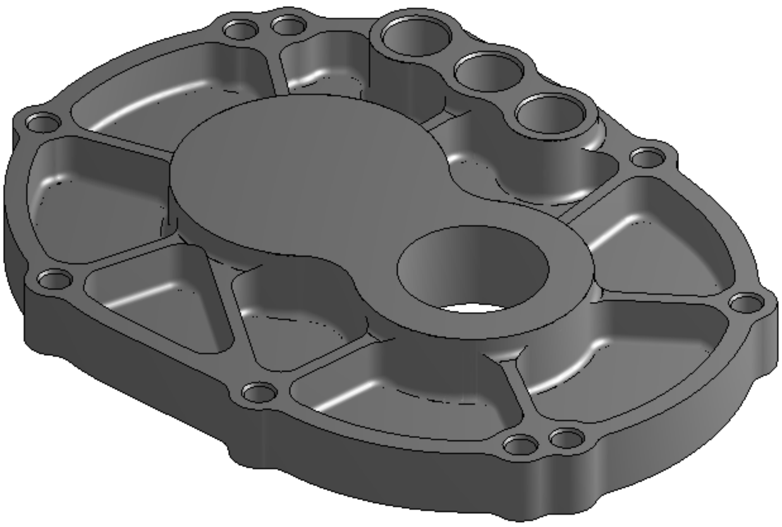


Note: This method is only effective for features that have been programmed using **AFR** or **IFR**. **Features created using curves will not be Re-Recognized upon changes to the model**.

- Press **Finish** then run a **3D simulation**.

- **Open** or drag and drop **EndPlate.x_t** into a **new milling document**, **inch**. Repeat the **Import Wizard** process including "**Launch AFR after finish**". Click **Next** four times**.**

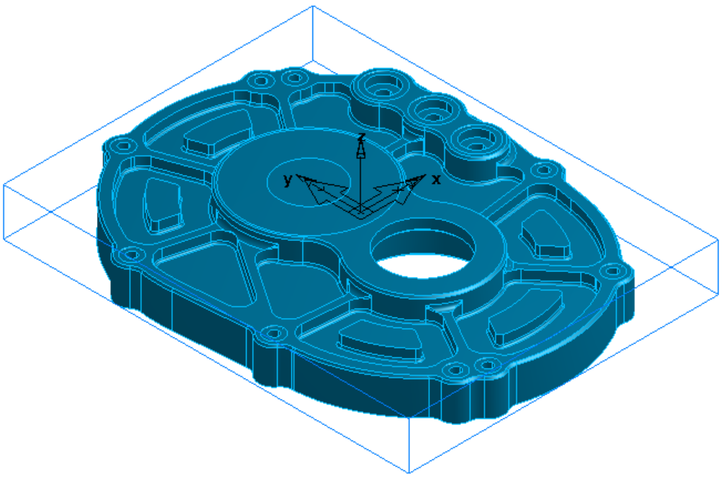
- **Compute stock size from the size of the part** then **Finish**. This will establish the Stock and Setup the same as before and then execute AFR but first in **AFR Options**, **Uncheck 3D**. This completes the initial Setup.



- **Import** or drag the model **EndPlate Changes.x_t** located in the FeatureRECOGNITION folder and click **YES** to Import into the same document.

- **Uncheck** "Launch AFR after finish". **Check** "Use same alignment as last import". **Next.**

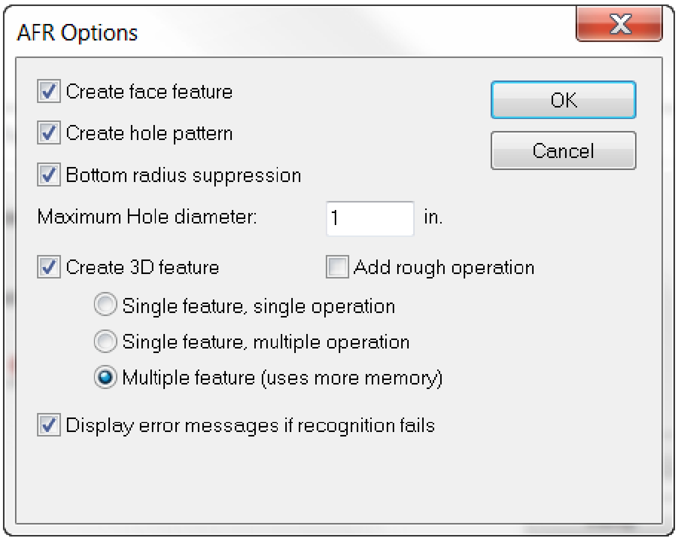
- Select **"Accept the imported data as is and exit the Wizard**". **Finish.** The revised model, **EndPlate Changes.x_t** is merged with the first.

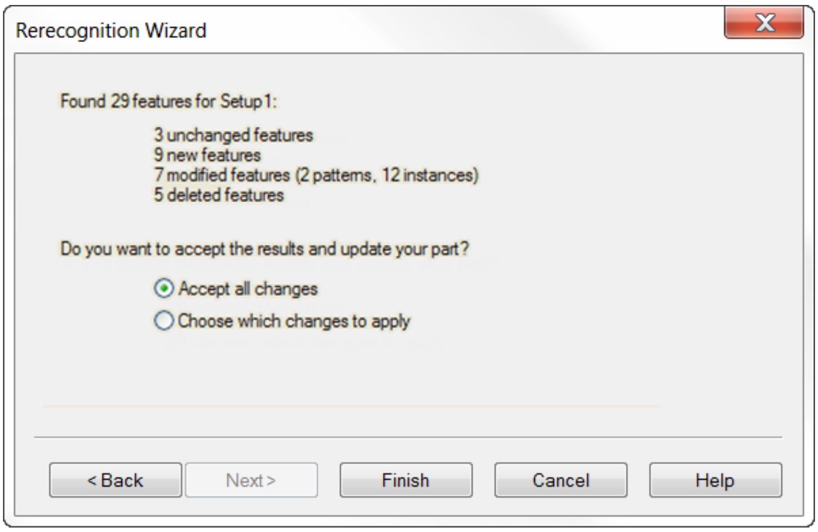


Note: The "Use same alignment as last import" button will disappear. This is expected.

- Click **Construct** and Click on the **Re-Recognition** Wizard and drop the menu down and select the **second** model.

- Click **Options**. Check "**Create 3D feature**" and Select "**Multiple feature**". This will apply the 3D tool path to the three chamfers and a new 3D (**only if licensed)** addition in the revised model. The three chamfers can be Unchecked in the Part View under Setup 1 to execute only the new 3D surface features.





Notice the numerous changes in the holes, pocket depths, islands and even 3D.

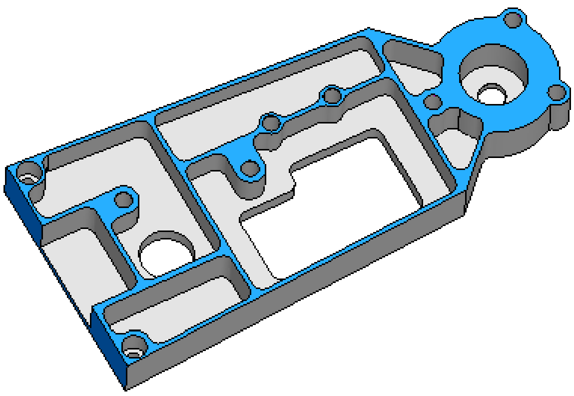
- Run a **3D simulation**.

IFR (Interactive Feature Recognition)

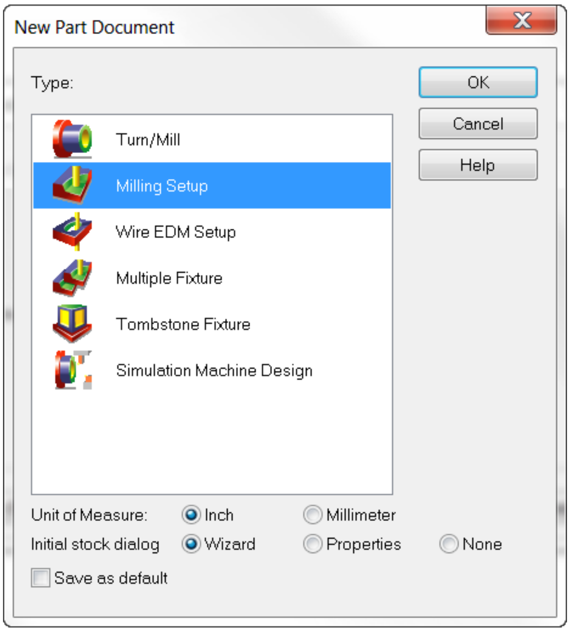
- IFR has three types of strategies available IFR Automatic by feature type, IFR using surfaces or faces and IFR chaining.
- IFR Automatic examines the model for a particular feature such as a side, pocket, face, slot, hole and boss. Side, pocket and boss features are not created using AFR side features 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.

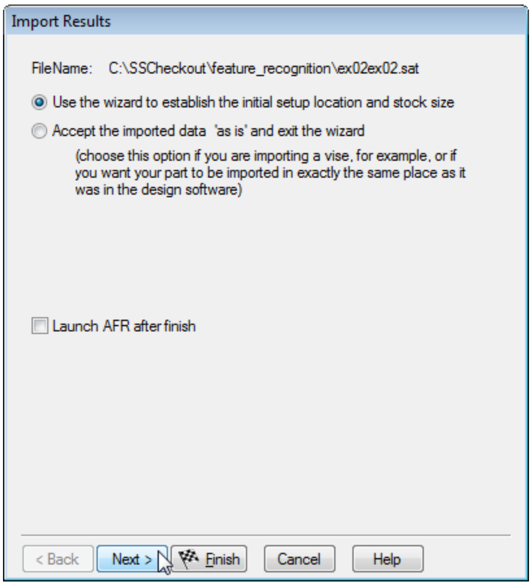
IFR Example #1

- Click on **File** then **Open**. Under Files of type at the bottom of the screen select **ParaSolids (*.x_t, *.x_b)**. Find the file named **ex02ex02.x_t** then press the **Open** button.



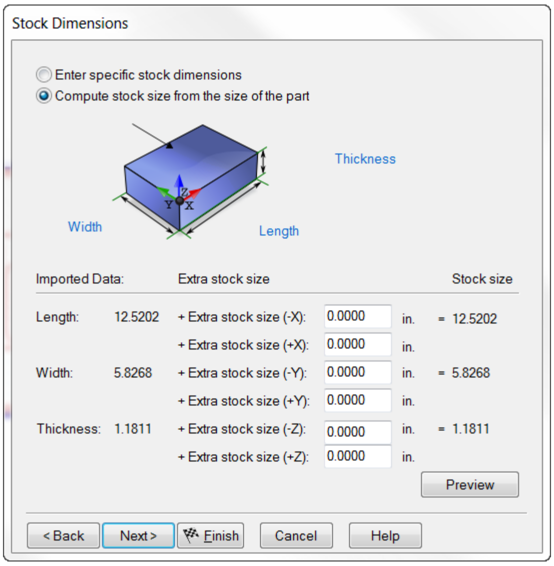
- Select **Milling** Setup and select the Unit of Measure to be **Inch** then press **OK**.

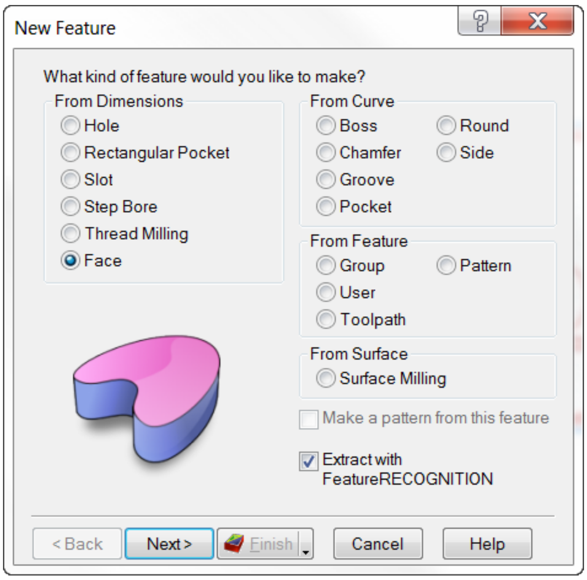




On the **Import Results** page select the **top radio button** and **Uncheck** “Launch AFR after finish” then press **Next** four times.

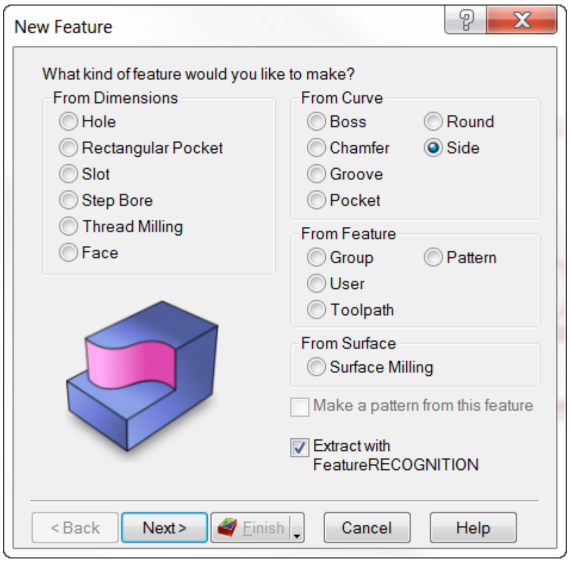
- On the **Stock Dimensions** page check the lower radio button **Compute stock size from the size of the part. Finish**.



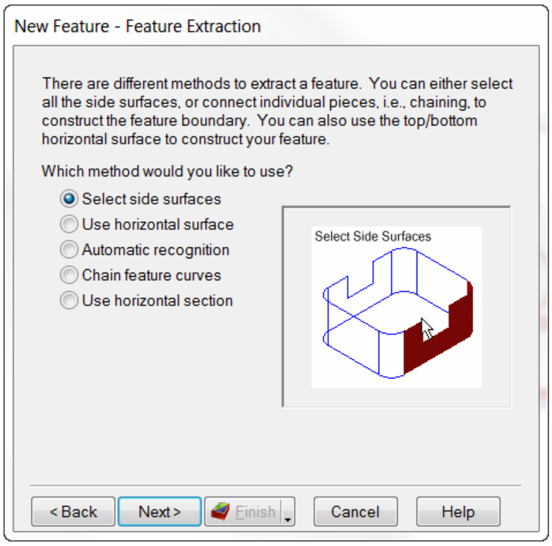


- Start the **New Feature Wizard** select **Face** then check “**Extract with FeatureRECOGNITION**”.

- **Left click** on the top most horizontal face of the part then click the green "**+**" to add it to the list. Then press the **Finish** button.
- In the Properties, Click the **finish** operation and **Milling Tab** and change the **Zig-zag Angle** to **90.**

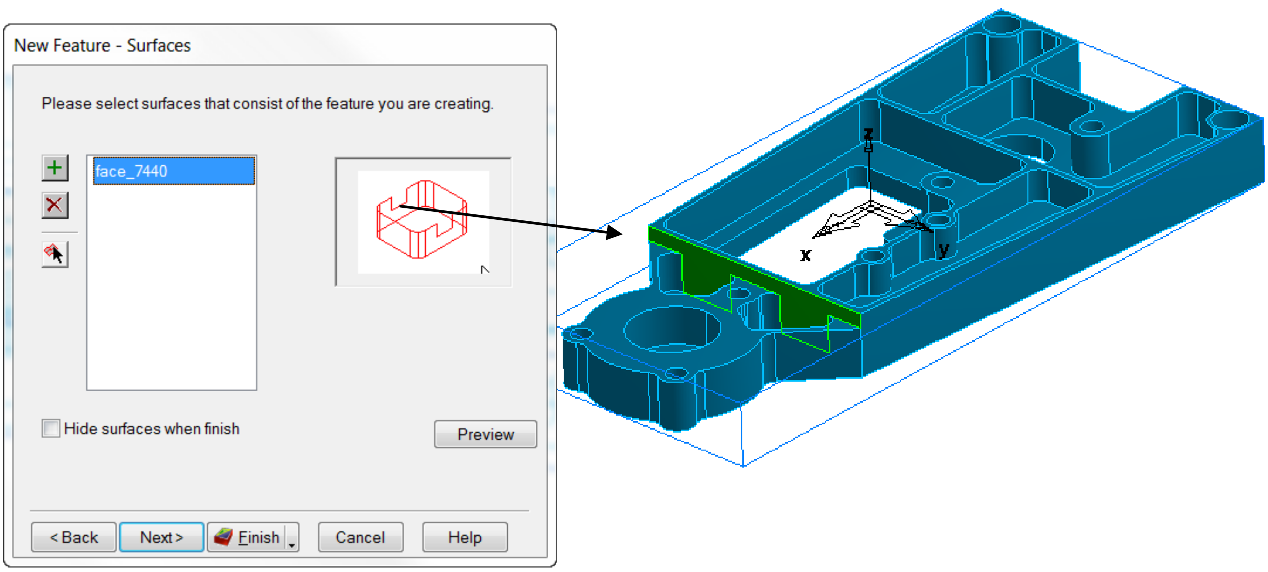


Open the **New Feature** wizard and select **Side** and check **Extract with FeatureRECOGNITION. Next.**



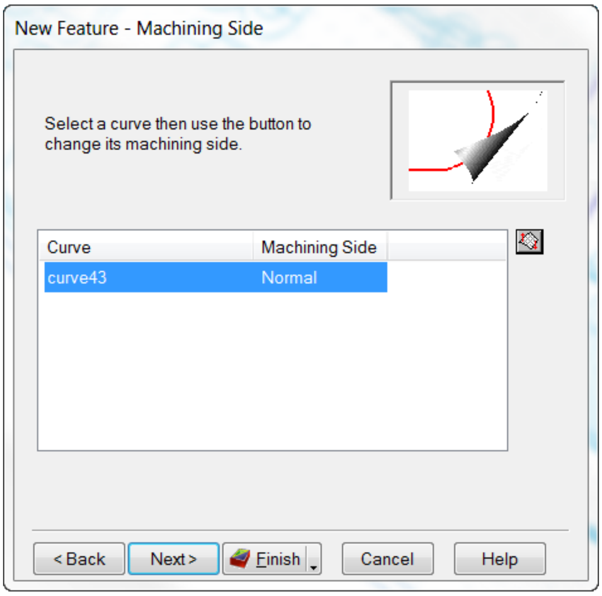
- Select side surfaces. **Next**.

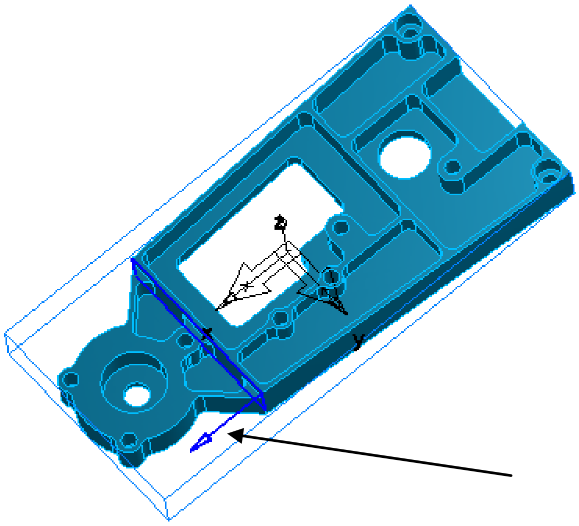
- Using the arrow, select this **vertical face** and press the green + to enter. **Next.**



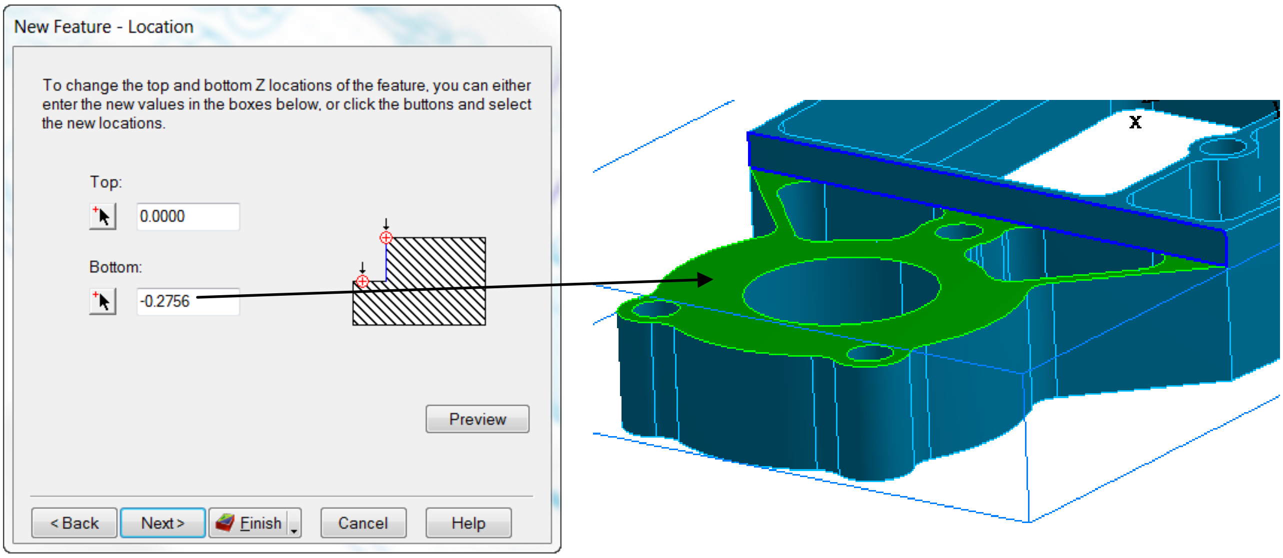
Observe the arrow, it points to the side that is cut. **Reverse if pointing to the wrong side** of the curve by using the button on the right of the machining side window.

- **Next.**



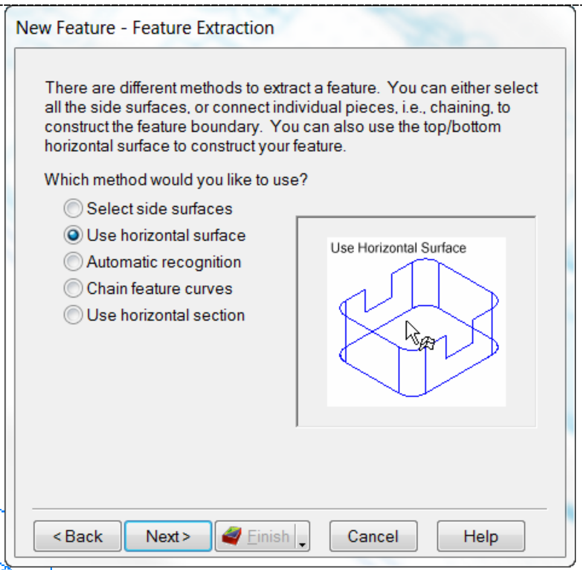


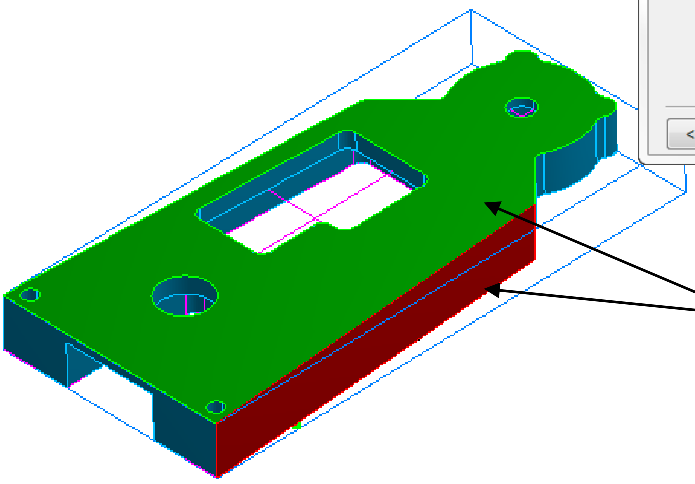
- Because the side selected for the feature descends into the pockets you must Click on the **Bottom** arrow and Click on the **horizontal** face to control the depth. Both the Top and Bottom can be controlled here. **Next.**



- Select **Boss** with Recognition from **New Feature**. **Next.**

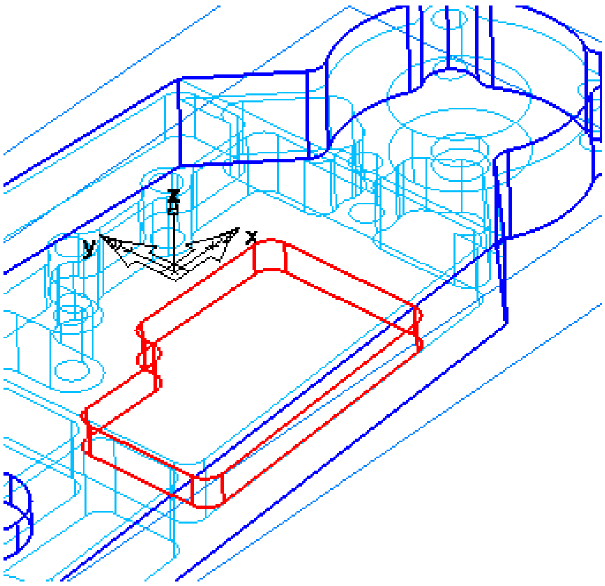
- Select “**Use horizontal surface**”. **Next.**



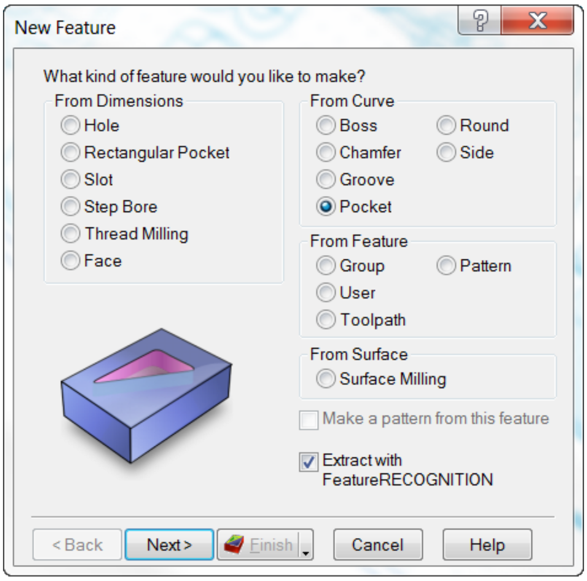


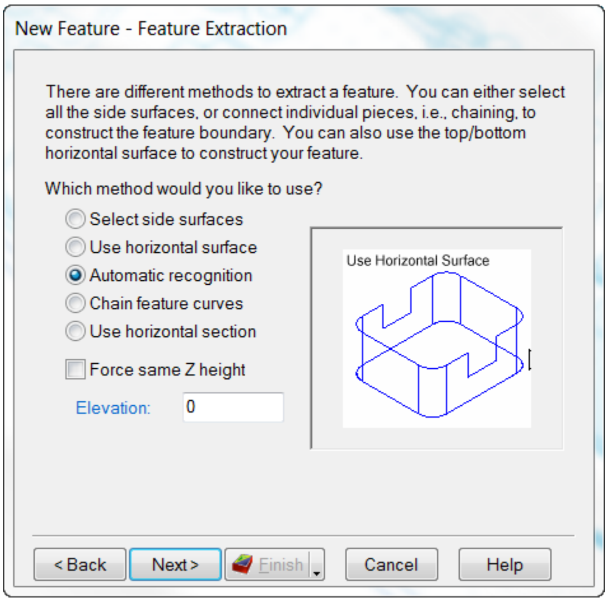
- Rotate the part and select the bottom horizontal face for the shape and one vertical face to determine top and bottom and Click the green + to enter. **Finish.**

- Select **New Feature** Wizard and Select **Side** with Recognition. **Next.**
- Automatic, then select only the side feature that goes through. **Finish.**



- In **New Feature** select **Pocket** and Extract with FeatureRECOGNITION. **Next.**

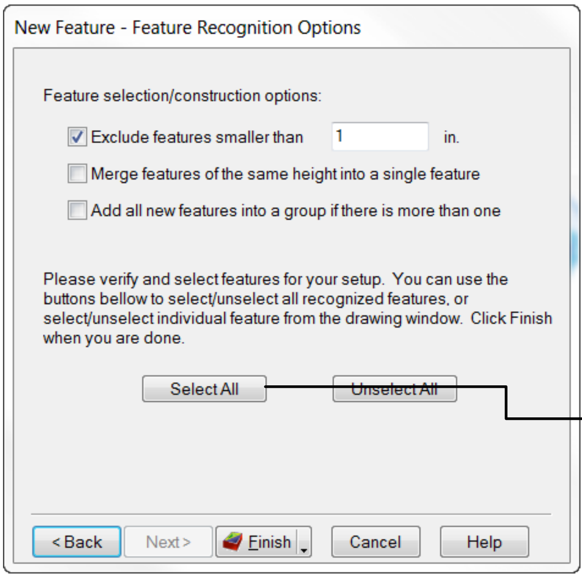




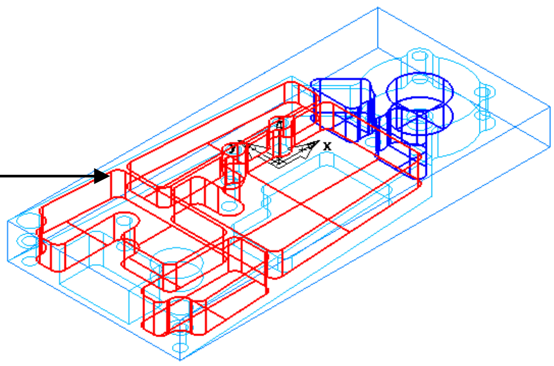
- Select Automatic recognition. **Next.**

FeatureCAM using IFR Automatic recognition will identify all pocket features.

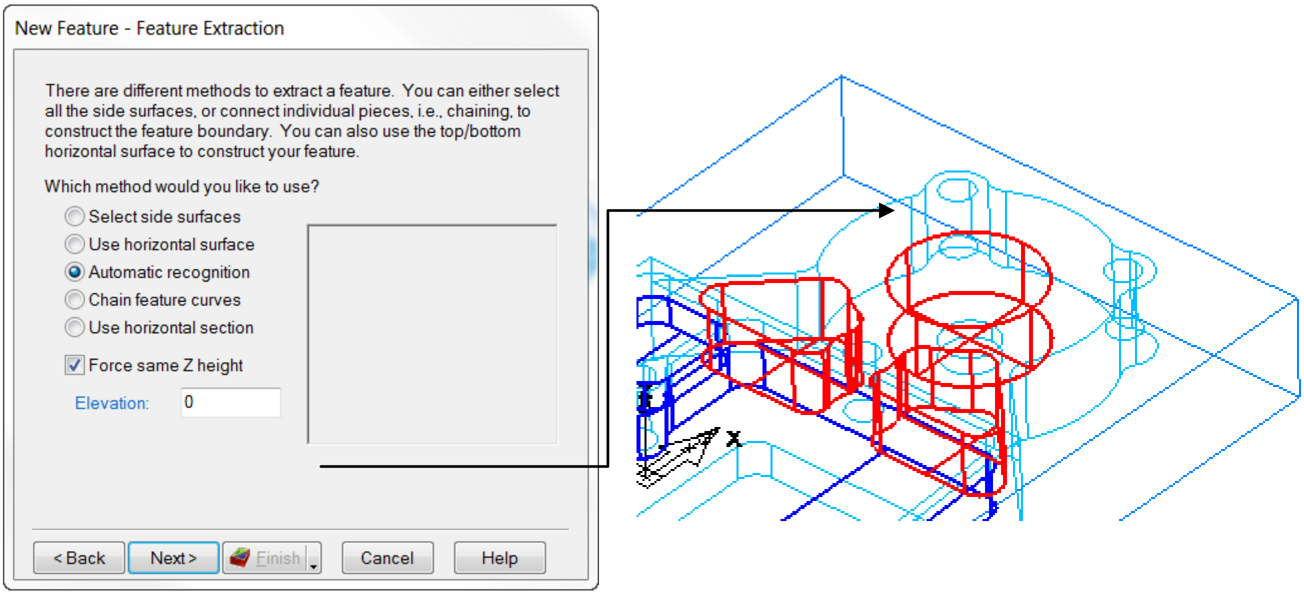
- Press the "**Select All**" button to select all identified features and then unselect the two similar small pockets which the top of these features are too high, these two should be at a lower Z elevation. This happened because a face in the pockets goes higher than the tops of the pockets. Unselect the round also.



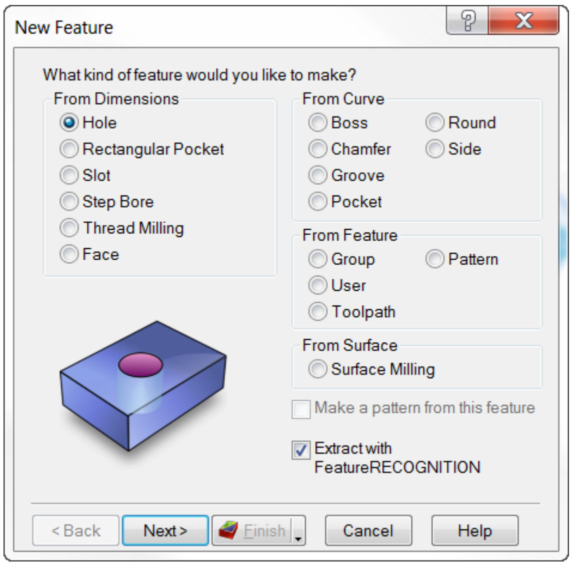
- All selected pockets turn red when selected and all you need to do to unselect them is to click on each and they will become blue again and are not included.

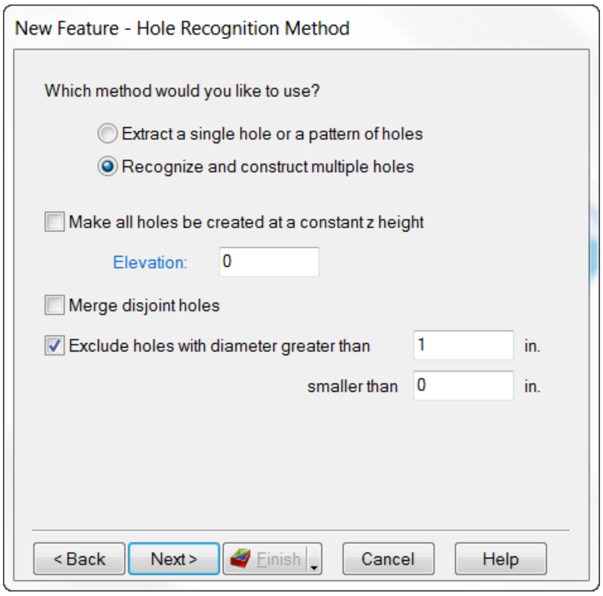


- Repeat the Automatic **pocket** process again with **New Feature** except **Check** “Force same Z height”, Click on the Blue word **Elevation** then Click on the horizontal surface to set the height (Top) of all three remaining pockets. Click on the three remaining pockets turning only those red.

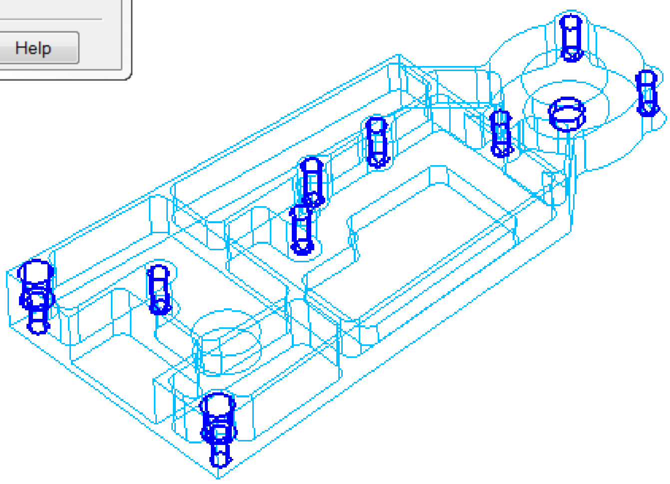


- Open the **New Feature** wizard and select **Hole** and check **Extract with FeatureRECOGNITION. Next**.

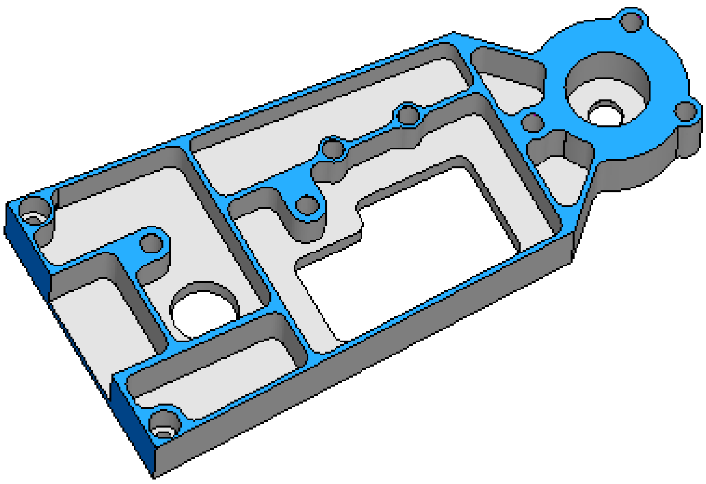




- Select the **Recognize and construct multiple holes** radio button and check the **Exclude holes with a diameter greater than** checkbox and enter a value of 1 inch. **Next.**



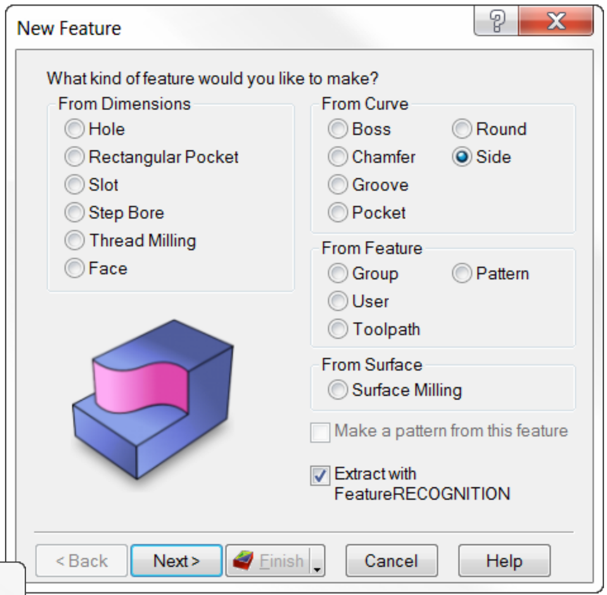
- Select All. **Finish.**
- Run **3D simulation**

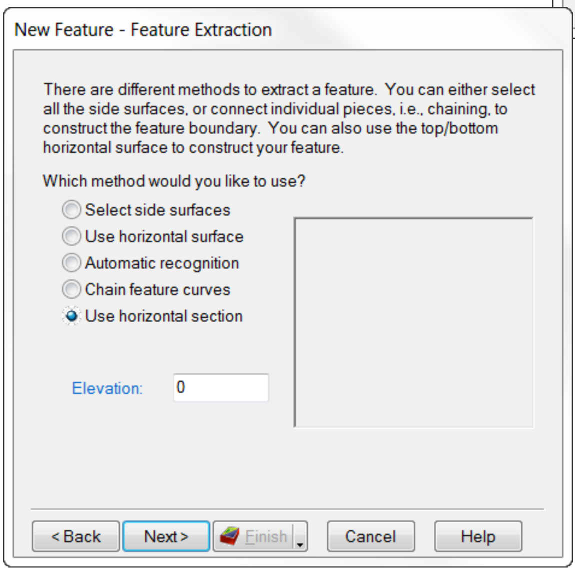


IFR Example #2

- From the file menu open **2.5D pocket with open groove.fm** from the examples folder and shade it.

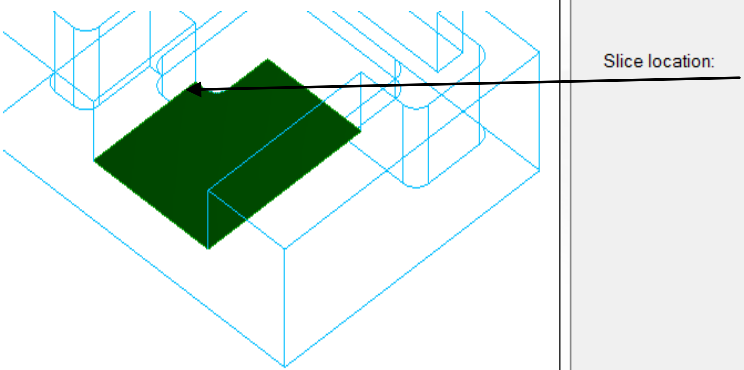
- **New Feature** Wizard and select **Side** from Curve and check Extract with FeatureRECOGNITION. **Next.**

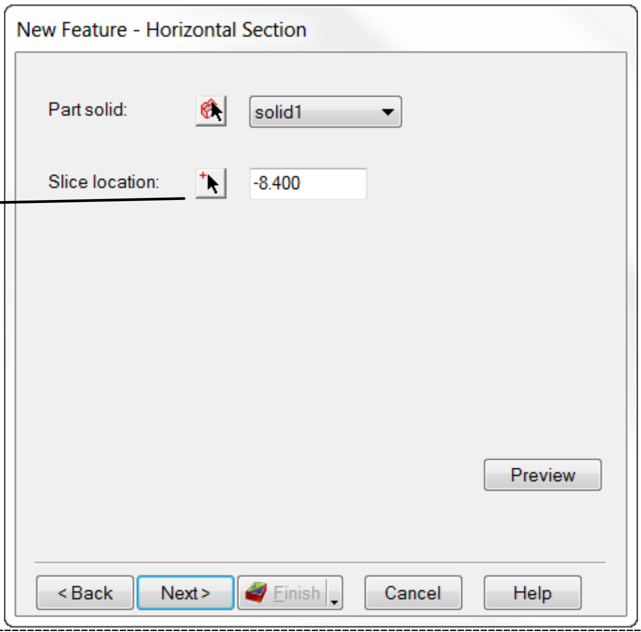




- Select "**Use horizontal section"** and "**Elevation**" of 0 (top). **Next.**

- Use "**Slice location**" arrow and **Click** on the edge of the first horizontal surface to establish the bottom of the feature. **Next.**

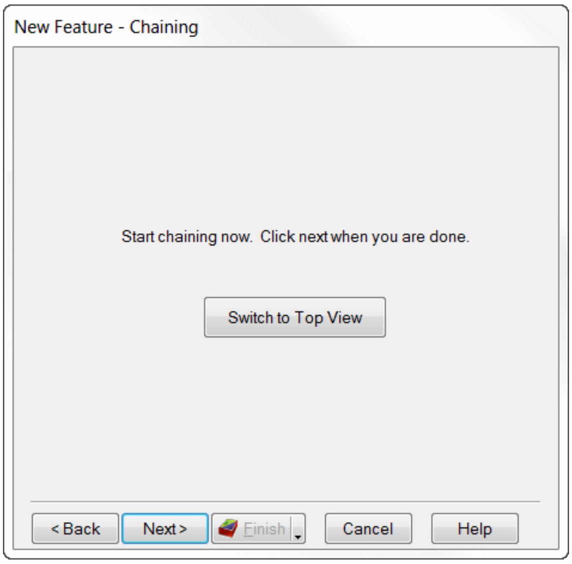


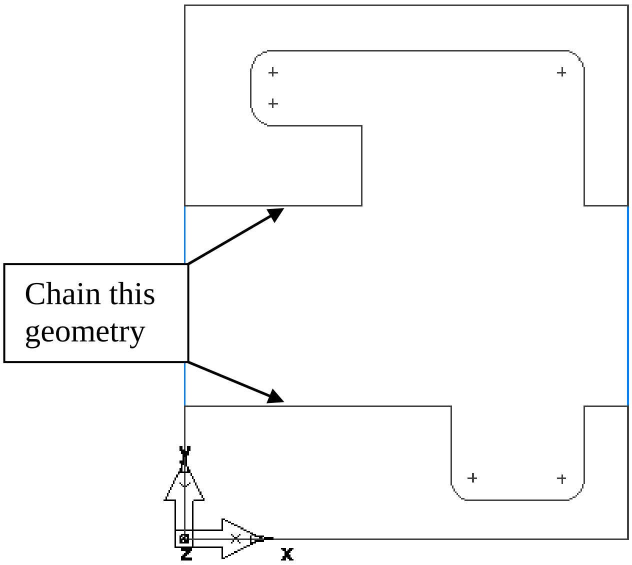


The cursor is now in the Mode to create curves and you can readily see the outline of the two side features that will require open curves.

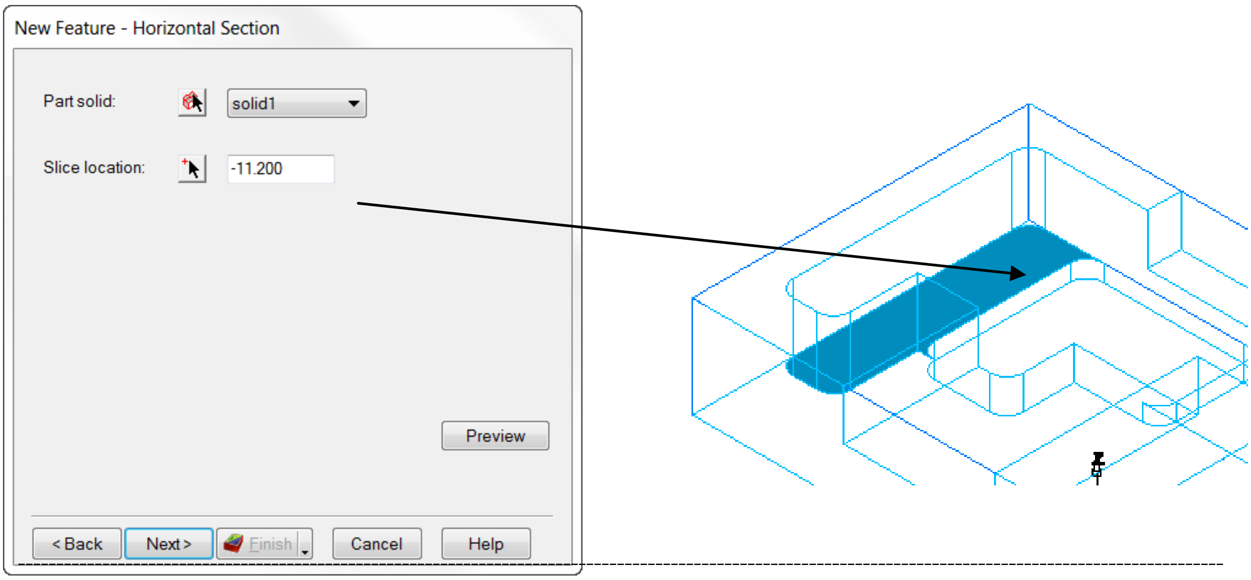
- **Chain** both **Open** geometry shapes separately. **Click** on **Create** at the bottom in the geometry dialog area after **each** curve is complete. Be certain **not** to go around the **outside** of the part with the curve, both should be open curves and end at the openings on each side. This geometry will be deleted and only the curves will remain. **Next.**

Remember these are **TWO** open curves in **One** side feature and both Machining Side arrows must point inside and each must point at the opposite curve. This will machine everything in between.



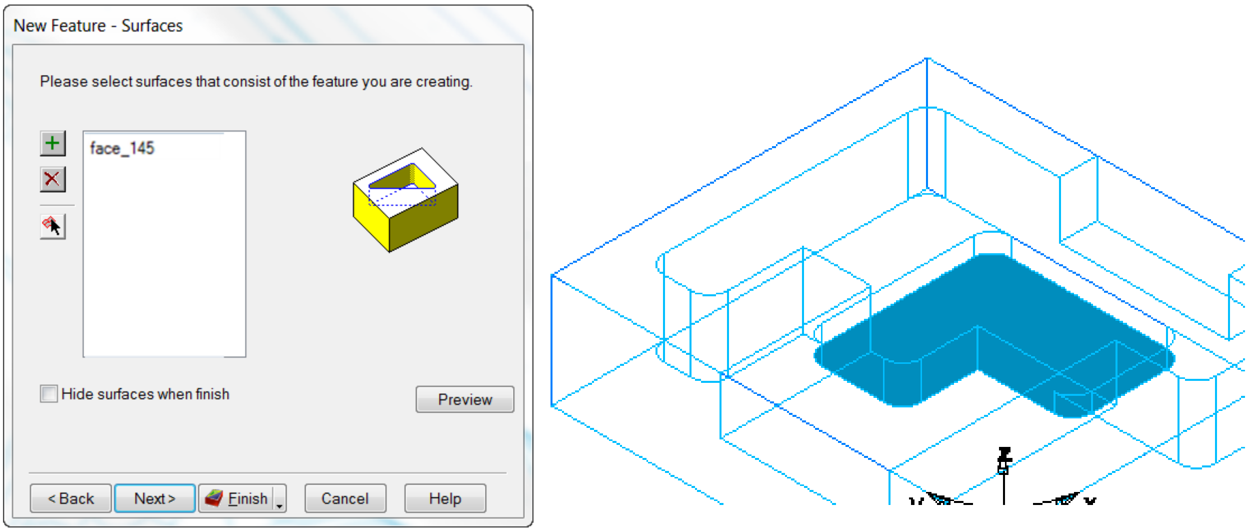


- Repeat the same steps except use the bottom of the previous feature as the “Elevation” (top) of the next Horizontal Section and the first face stepping down as the second “**Slice location**” (bottom).

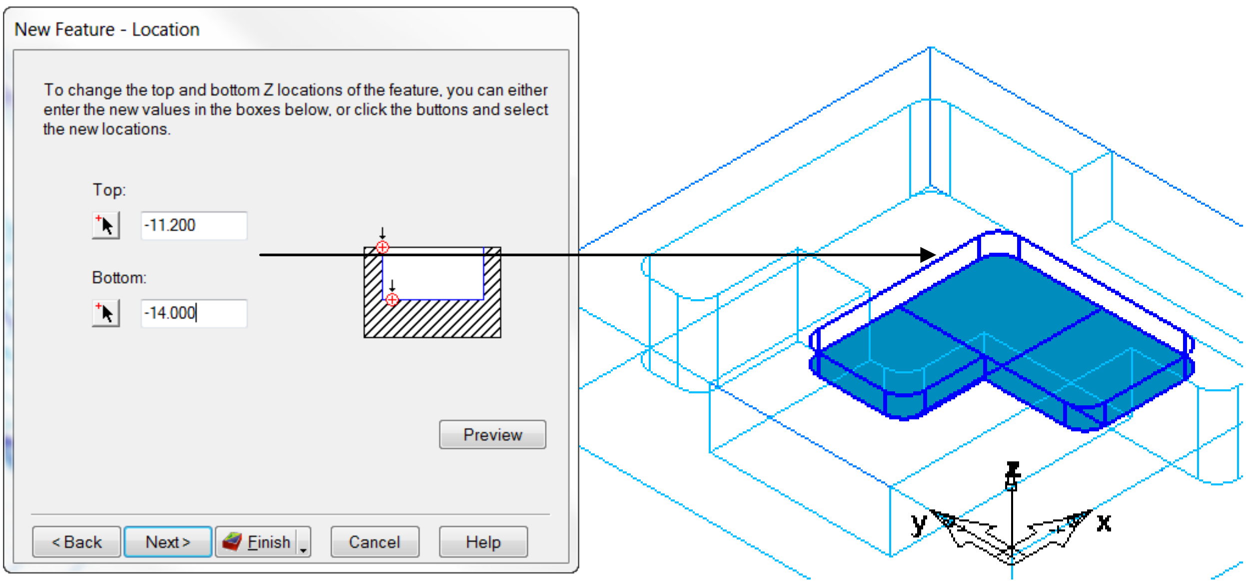


The third and final feature is a pocket.

- Open **New Feature** wizard, select Pocket and Extract with FeatureRECOGNITION. **Next.**
- Select Use horizontal surfaces. **Next.**
- Select the bottom horizontal face of the pocket. **Next.**

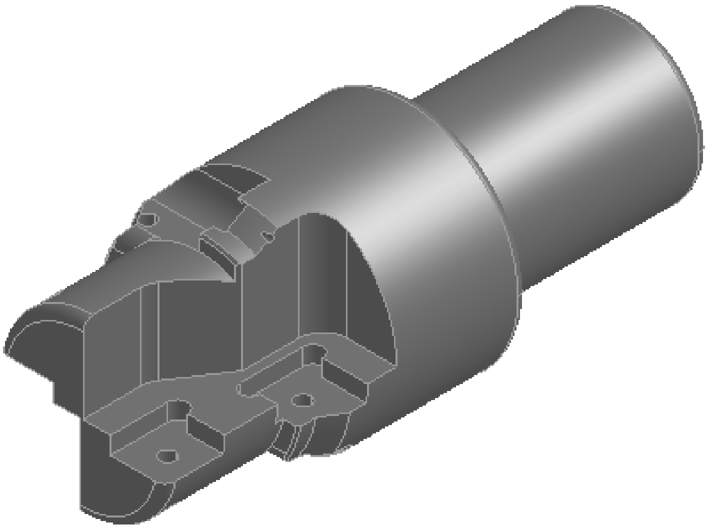


- Using the Top arrow select the top edge of the pocket. **Finish** and **3D simulation**.

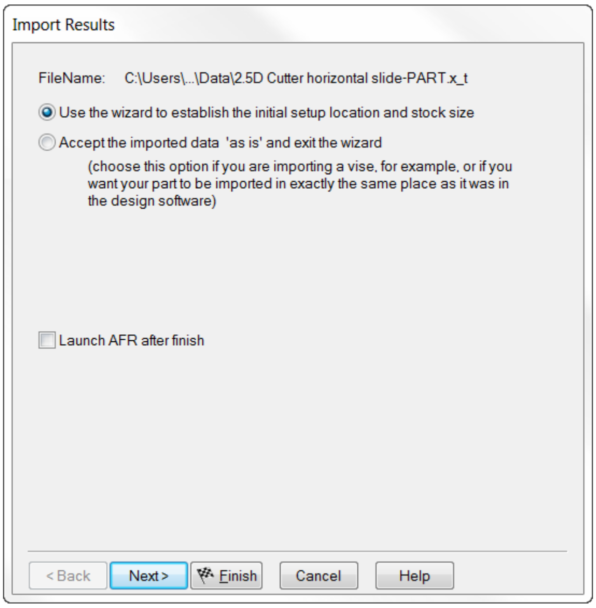


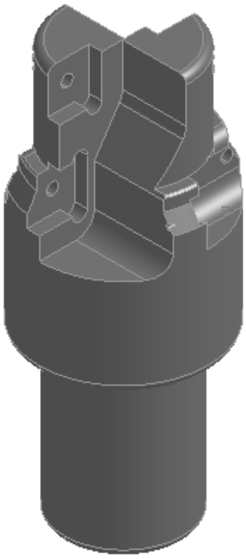
IFR Example #3 (4 Axis)

All previous FeatureRECOGNITION examples have been stationary 3 Axis parts but 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 the mill for these secondary features.



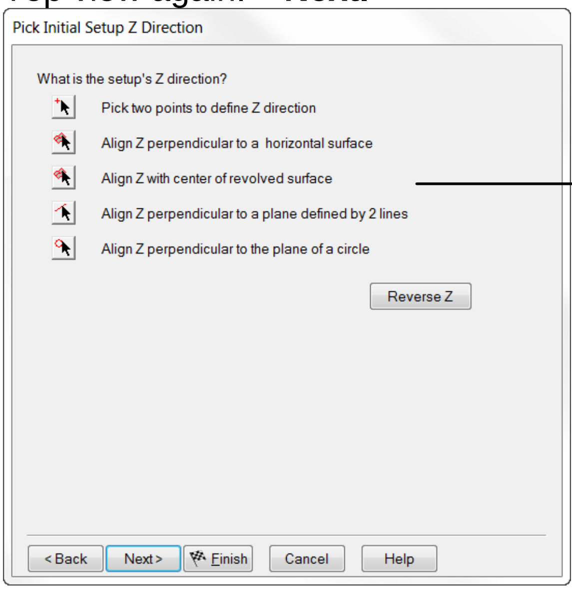
- **Open** a new **milling** document in **inch**. Click on **File** then **Import** and select from the Data folder model **2.5D Cutter horizontal slide-PART.x_t**.
- In the Import Results **Select** "Use the wizard to establish the initial setup location and stock size" and select an **Isometric View** by right clicking in the graphics window. **Uncheck** "Launch AFR after finish. **Next.**

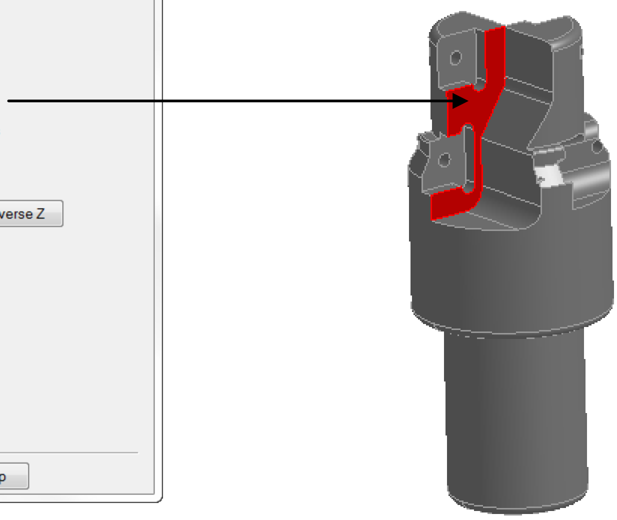




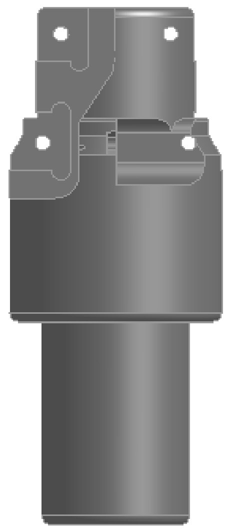
The part centerline must be aligned with the rotary table in the X Axis and perpendicular to the spindle.

- **Shade the part.** Select the Arrow to "**Align Z perpendicular to a horizontal surface**" and **Click** on the horizontal face then Top view. If your selected surface is upside down Click on the "Reverse Z" button and Top view again. **Next.**

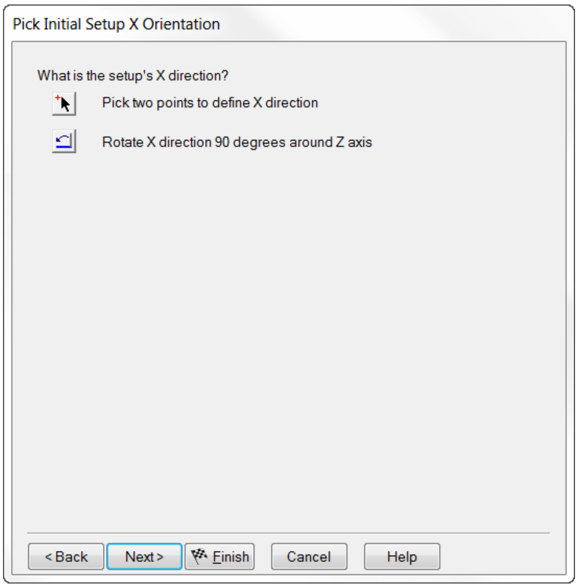


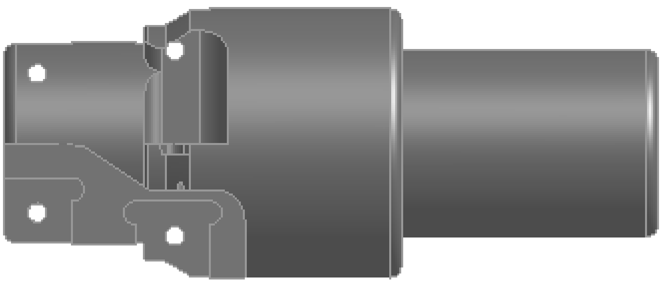


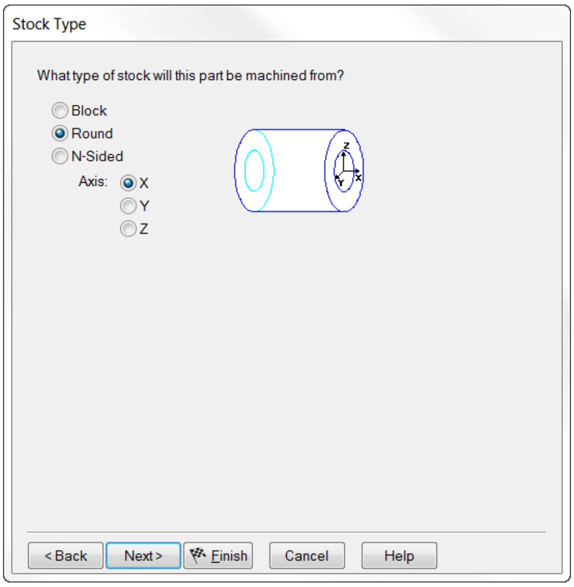
The center line is now perpendicular to the spindle but the **part must be rotated around the Z Axis** to align with the X Axis.



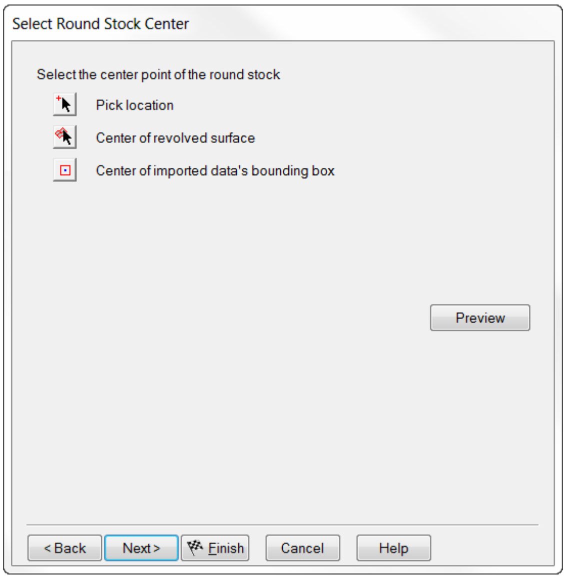
- **Click** three times on the “**Rotate X direction 90 degrees around Z Axis** and **Top View** again. Pockets to the left as shown below. **Next.**



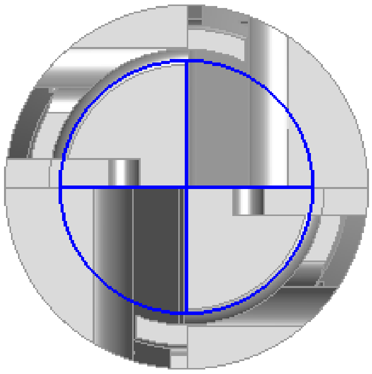




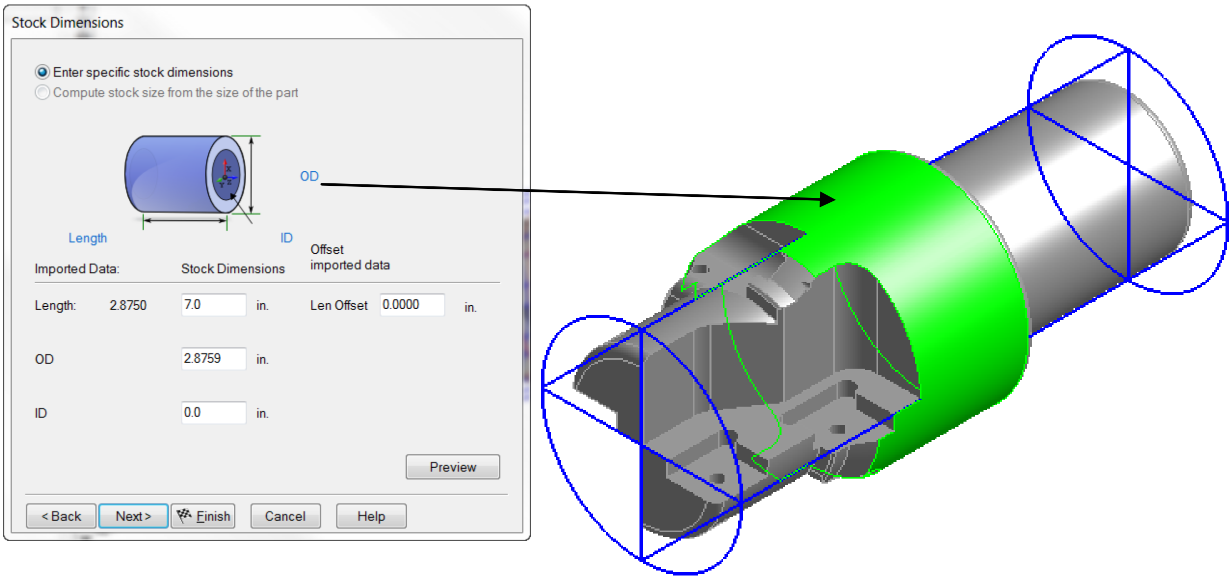
- In **Stock Type**, Click **Round**, **X Axis**.
  Next.



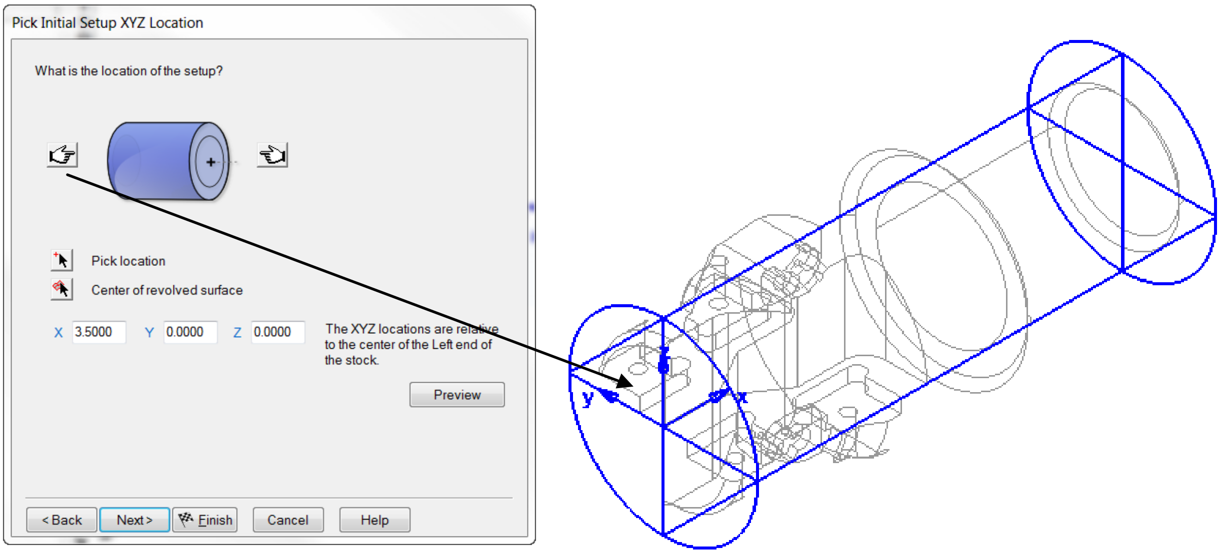
- View from **Left View** to check if stock is on center. **Next.**



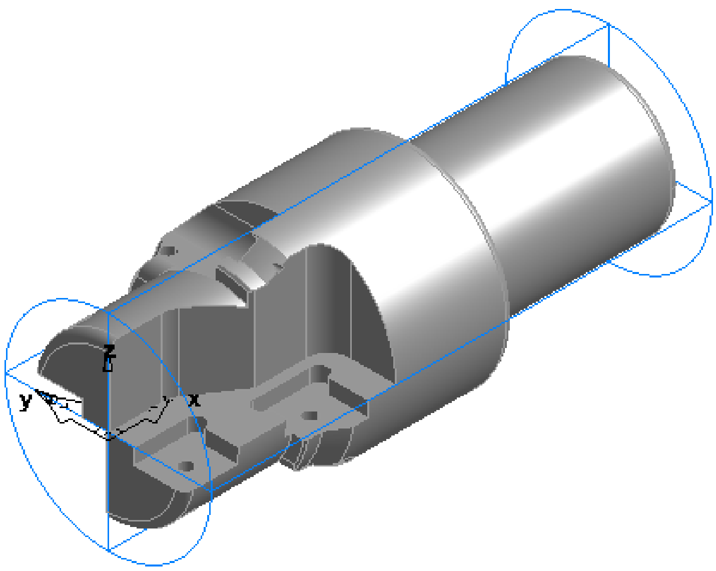
- From **Stock Dimensions** Click on the "Blue" letters "**OD**" and then click on the largest OD of the model then type the **Length 7.000** dimension in the box and Preview. **Next.**



- From the "**Pick Initial Setup XYZ Location**" Click on the **Left Hand** to set the XYZ Zero Location. **Next.**

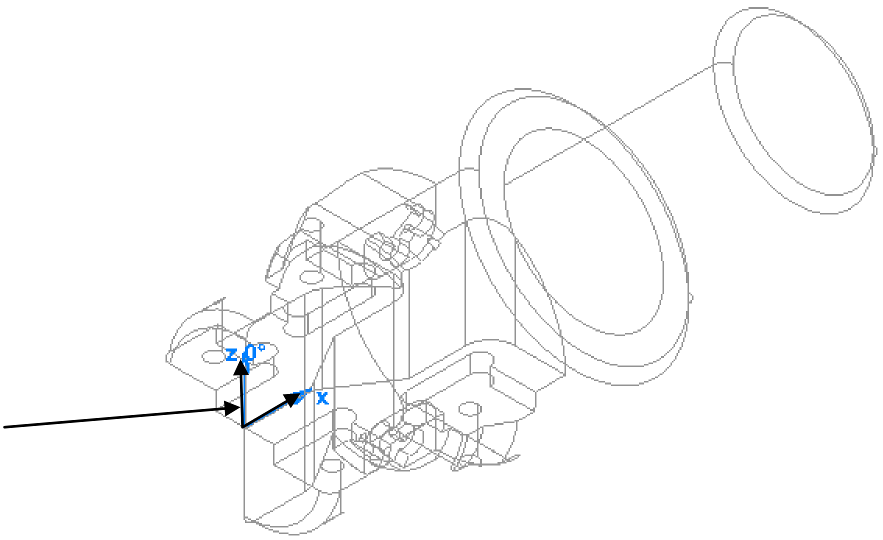


- In "**Is Part Indexed?**" Select **4th Axis Positioning** and **Index around the STOCK X Axis**. **Finish.**

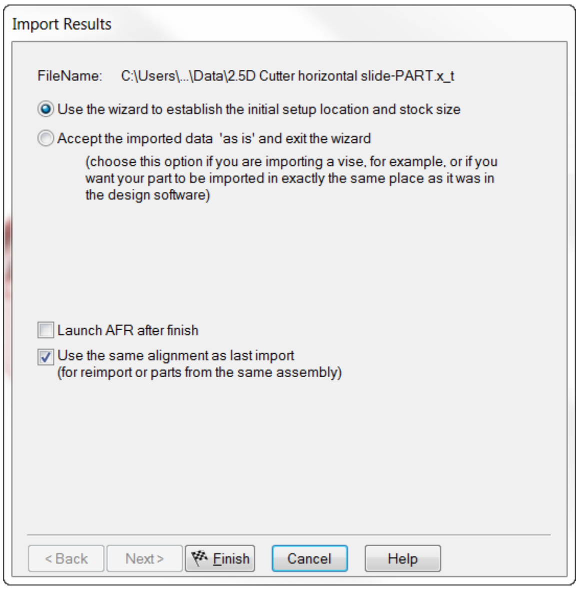


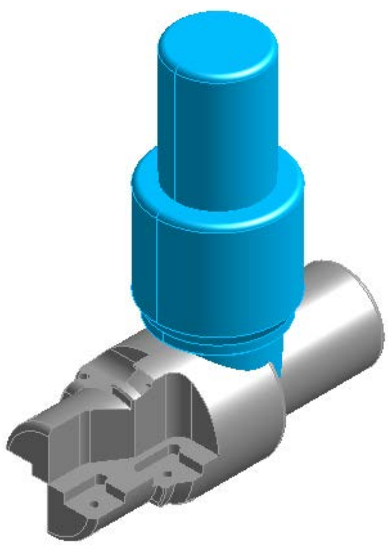
When using 4th Axis the **Part must be aligned with the "STOCK AXIS"**. This is the **Center of the Indexing** and **equally** important is the Reference for ALL features created or recognized to be placed around.

- To view the "**STOCK AXIS"** Click on **View**, upper menu bar and Select **Show** then **Show Stock Axis**. You will see another set of arrows merge with the UCS (**light blue in color**). The Z points up in the Z direction toward the spindle and is "0" degrees, the starting point of indexing angles. The X is parallel to the X Axis and will index around this axis



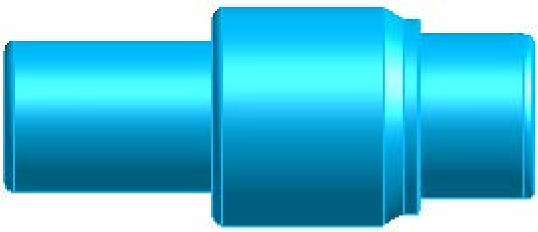
- Repeat the **Import** process for the STOCK model **2.5D Cutter horizontal slide-STOCK.x_t**. Import the STOCK **into** the same document that the PART is in, the STOCK model initially appears as below. **Click Yes**.
- In the **Import Results** select the **top radio button** to Use the Wizard and check "**Use the same alignment as last import**".

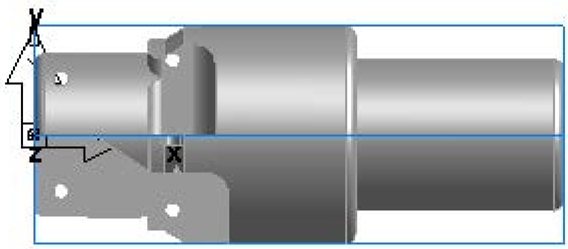




When **Finish** is clicked the STOCK model will align to the PART model in the same relationship as when they were **exported from Cad** but will not be merged, see below. For the two models to **merge** together at this point they must be created in the Cad system aligned and merged together although they can be Imported into FeatureCAM together or separately. You may be able to control how models are created in house but models from customers can be created in various positions.

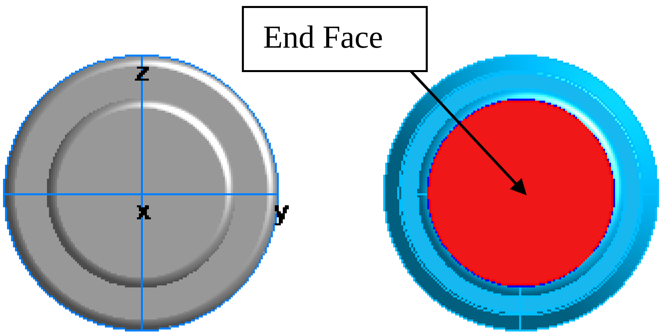
The two models now need to be merged and that means you will have to **Transform** the STOCK model by **Rotating** and **Translating** it to the PART model





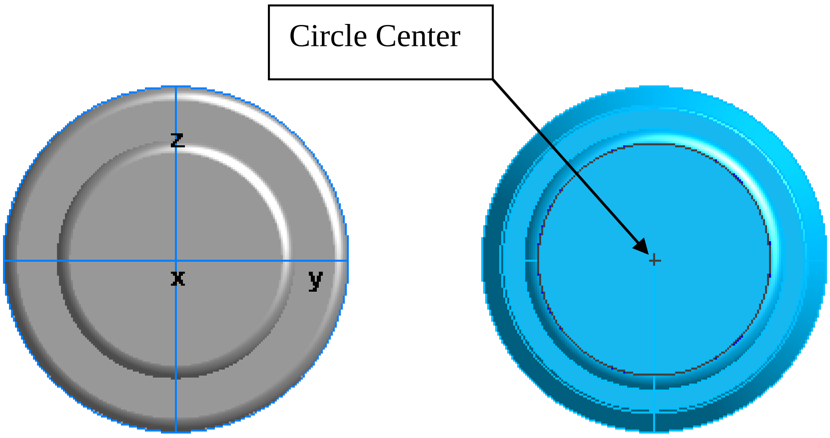
Having a STOCK model in addition to the PART model is very helpful in as much as the STOCK model actually represents the state of the stock material presented to the 4 Axis operations plus FeatureCAM now uses the STOCK model as the STOCK boundary instead of bar stock.

- To finish the alignment or the merging of these two parts, change to a **Right View** and **shade** the part.
- Using **Curves** from the **Steps Toolbox** then **curve wizard** select **Curve from Surface** and **Trimmed Edges**. **Next.**
- Using the Surface arrow select the **End Face** on the STOCK model. The face will show select color. Around the outside edge a curve is created and that is what the purpose for doing this is, you will need this curve. **Finish.**

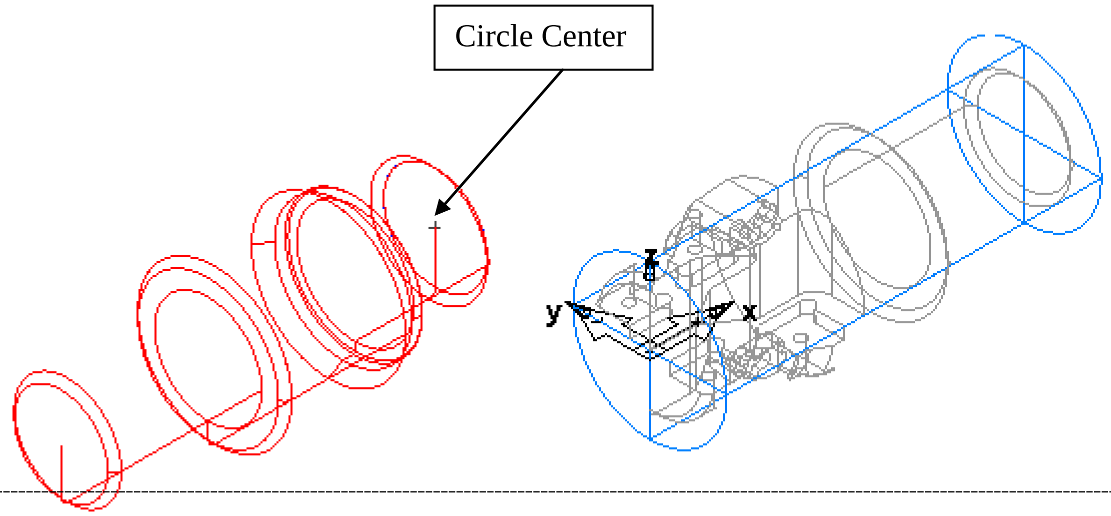


- **Click** on the curve you just created and then click on **Construct** from top menu bar, Select **Curve** and from the sub menu select "**To Geometry**". This changes the curve into geometry but keeps the curve.

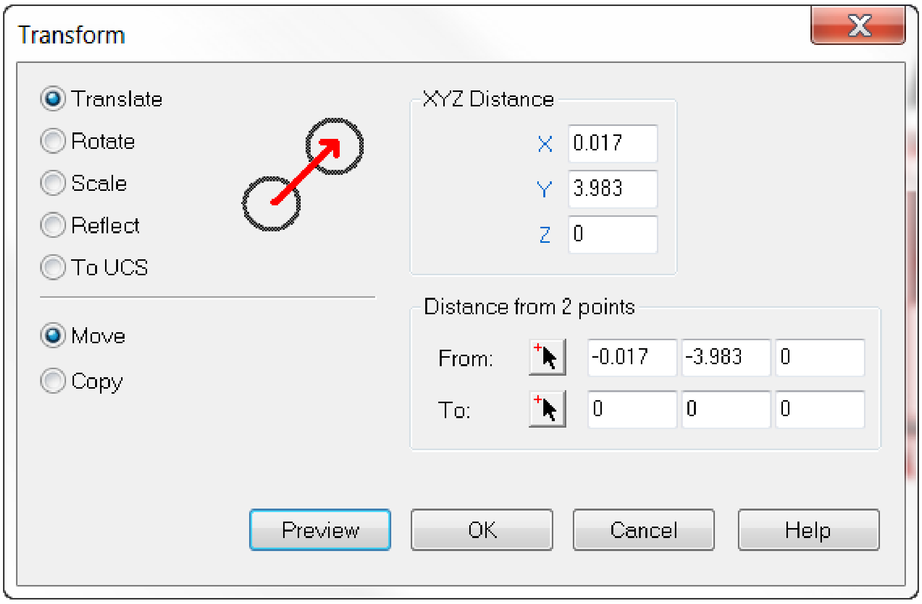
The Geometry circle gives you an exact center point of the model and can be used as an anchor to click on whereas the curve didn't provide this. These two tools are very helpful throughout FeatureCAM when working with curves and models. You are simply extracting engineering information from the model.

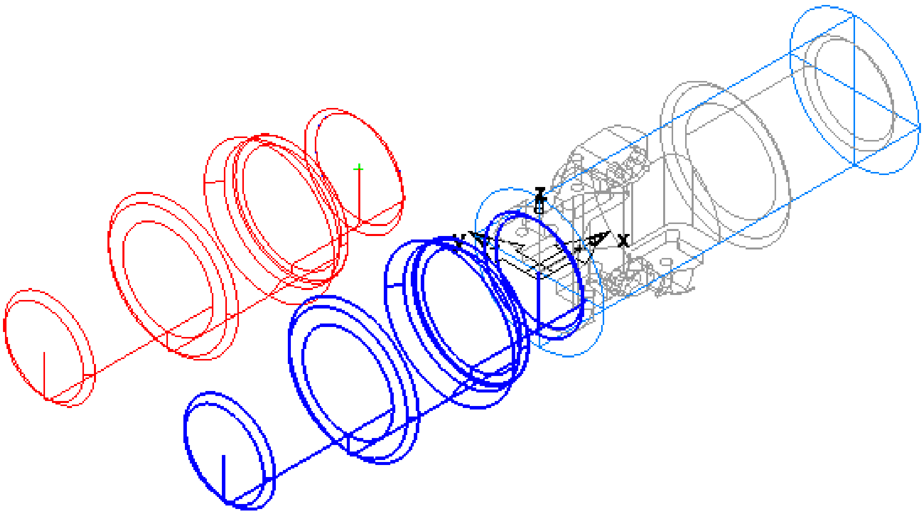


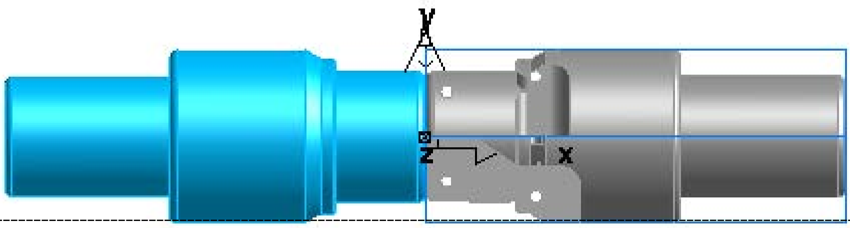
- Now change to **Isometric View** then place your cursor on the STOCK model, **Right Click** and **Select Solid**. You will need to see the circle center and the red select color covers it. Click the shade button if you need to unshade (2 little red barrels on top tool bar) and this will reveal the circle but the select color must remain on the model in order to open Transform, so be sure not to click on the graphics window or you will un-select and have to do it over again. You must always **select first** or the Transform button will not be accessible (grayed out).



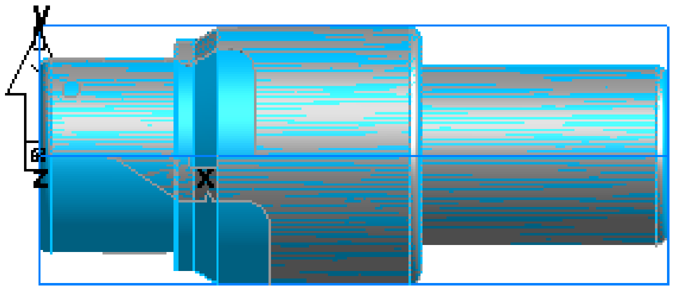
- Click on the **Transform** button (little arrow with pigtail next to the Select arrow) on upper tool bar. Select **Translate** and use the **From:** arrow click on the circle center. Enter 0 in all three boxes next to the **To:** and Click preview and you will see the STOCK model shift to the UCS at the end of the PART model. **OK.**







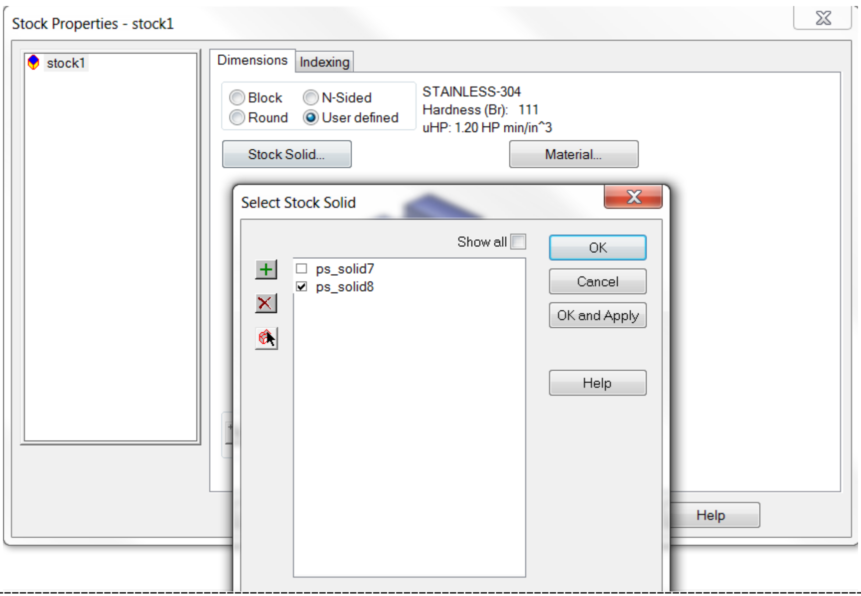
- Select the STOCK model as before and return to **Transform** and Select **Rotate. Angle** of 180 and Select **Center Point and Z Axis** and Enter **0** in both boxes. The STOCK model will rotate and merge with the PART model. **Preview** and **OK**. If unshaded, **shade** the part.



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.

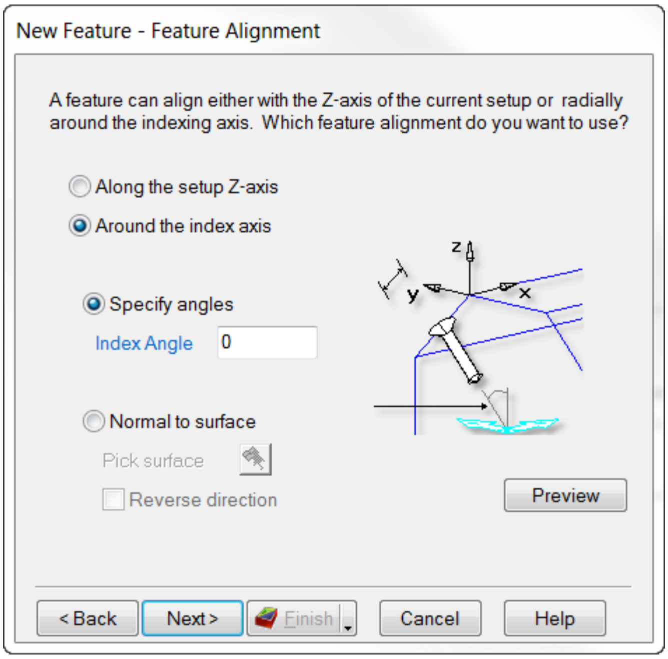
The PART model is now ready to apply the Features using FeatureRECOGNITION.

- You first must **change the Stock**. **Double-Click** on the wire frame of the stock in the graphics window. Click on "**User defined**" and check the box for the solid of the STOCK model. Click **OK** and **Apply**.



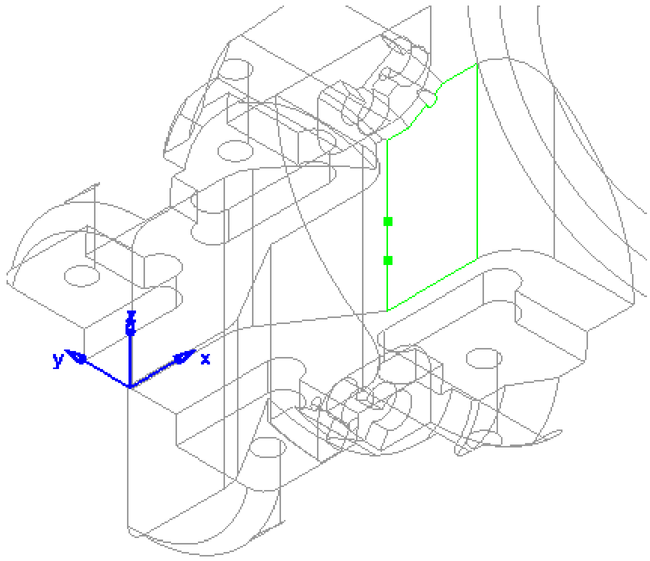
- From the Part View **Click Solid** in the list and **Right Click** on the STOCK model and **Hide Selected**. This is necessary to see the PART model and even with the STOCK model hidden it will still be visible when running 3D Simulation.

- Switch to **Isometric** view. Click **New Feature** Wizard and Select **Side** From Curve and check Extract with FeatureRECOGNITION. **Next.**



- **Select** Around the index axis and Specify angles.

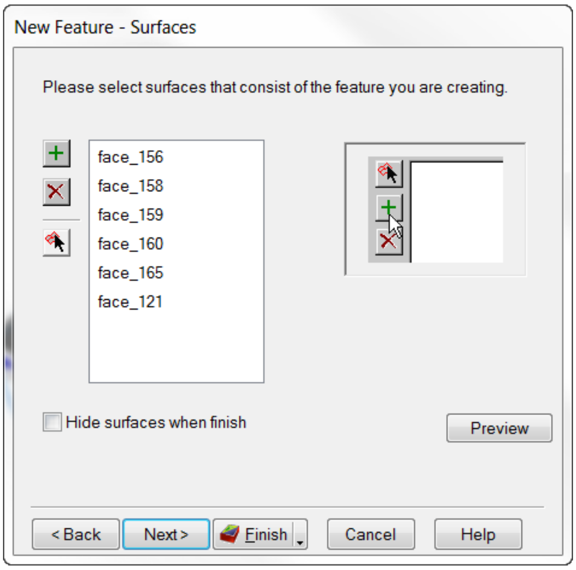
- **Click** on the “Blue” words “Index Angle”.

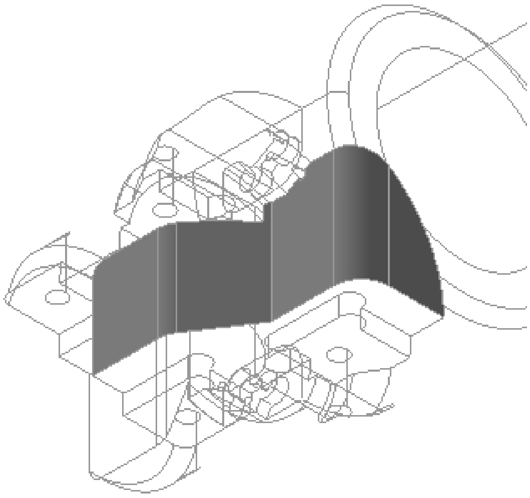


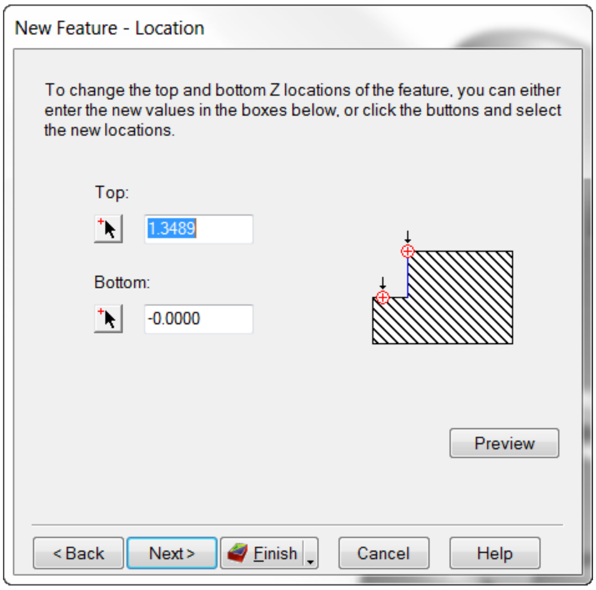
- **Click** on any vertical surface edge, starting low on the line with the first click and above the first with the second Click. **Next.**

The second click is pointing in the direction that you want the spindle to come from and at the same time you are establish the index angle for setting the features.

- **Select** side surfaces. **Next**.
- **Check** the arrow direction for Machining side. **Next.**

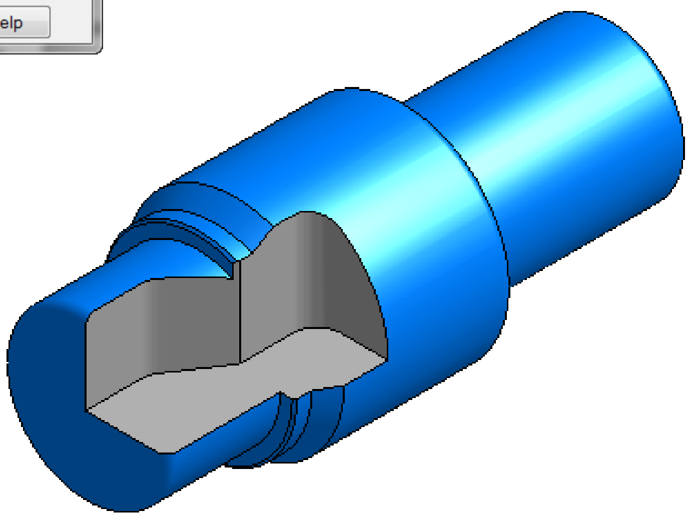






When selecting vertical surfaces, the Top and Bottom are already established but if you desire to change either the arrows can be used by clicking on a surface edge or you may enter the dimensions

- Click **Finish** and **OK**. Run **3D Simulation**.

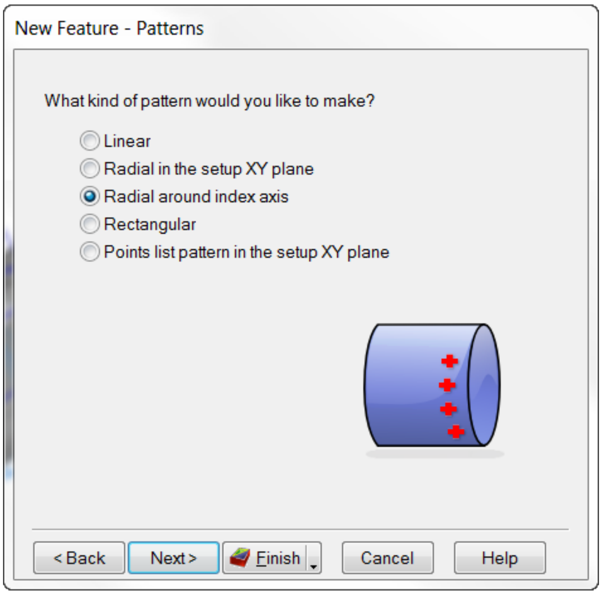


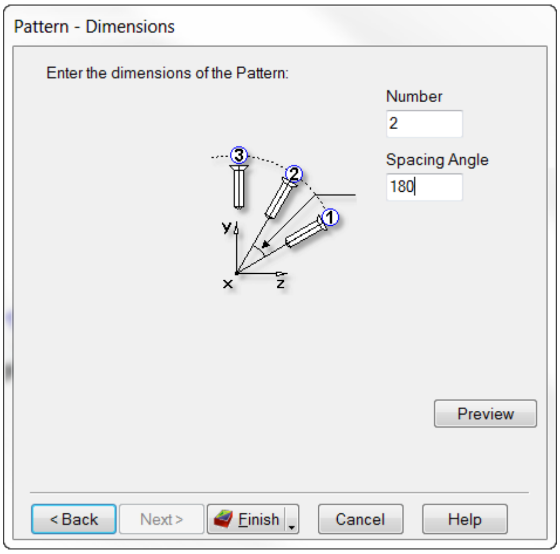
There are the same side features opposite the side features just completed. You can repeat the same steps for them. Remember when selecting the index angle the second click is always pointing to the spindle so if you select the surfaces when upside down the second click has to be lower because when the part indexes the vertical surfaces rotate and must be aligned with the spindle.
The other choice is making a **pattern from the previous feature** but should only be used when you are absolutely certain they are identical and symmetrical.

- To make a pattern first **Select** the existing feature and then the **New Feature** Wizard and select **Pattern From Feature**. **Next.**
- The previous Side feature must be selected. **Next.**

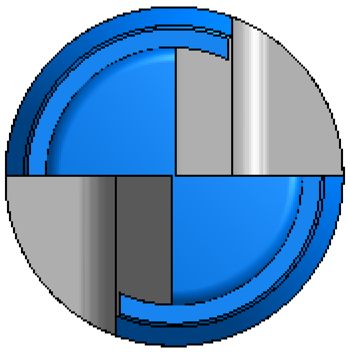
- Select **Radial around index axis**. **Next.**

.





- Select **Number 2** and **Spacing Angle 180**. **Finish** and **OK.**

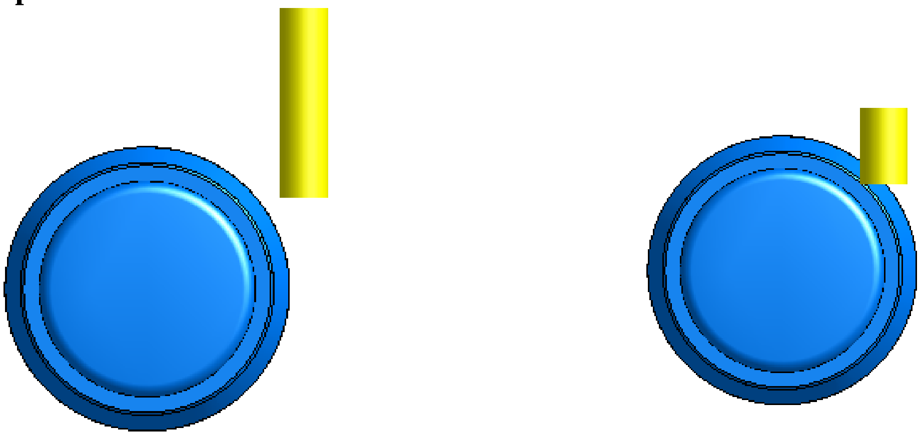


When milling cylindrical parts the side feature has to extend outside of the Stock far enough to satisfy the bottom of the feature but at the top, because of the shape of the material the tool is cutting air on some passes.

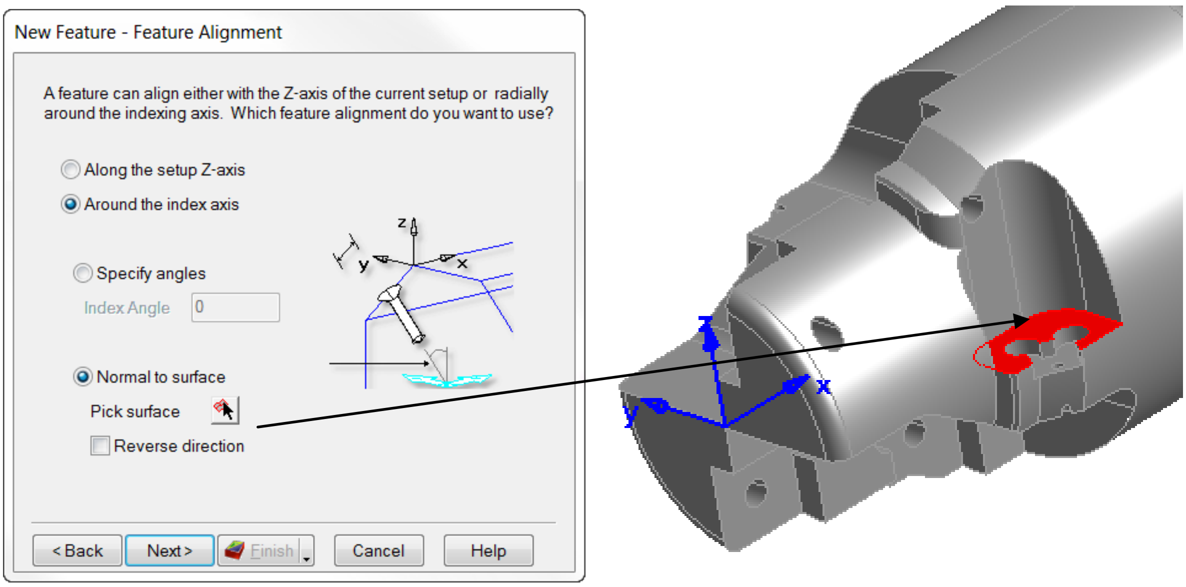
- To eliminate this, under the **strategy tab** in the milling feature select **Individual levels, Apply** and **OK**.

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.

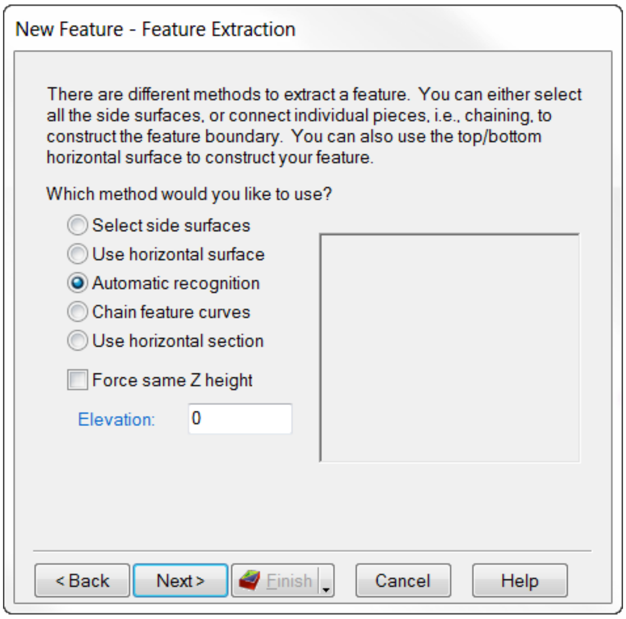
This will also make the part index on each level but can be eliminated by selecting "Cut higher operations first" from the results window (right side of graphics window), click on **Automatic ordering options** button at top. **This works only if the features are individual not a pattern**.



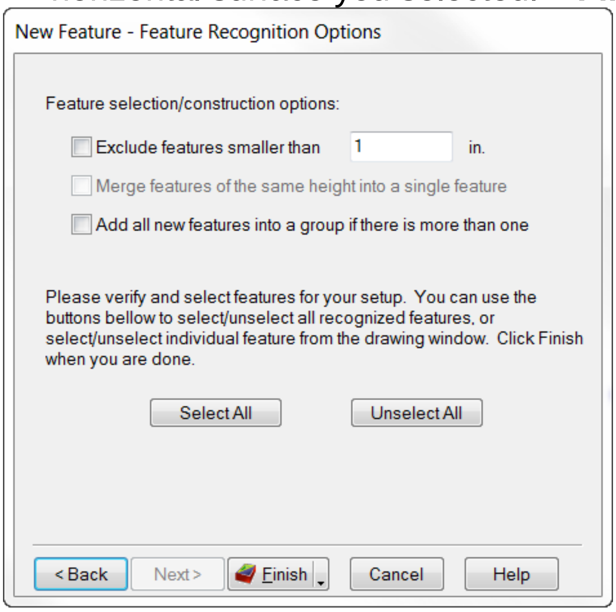
- **Rotate** the View so you can see the horizontal surface of the feature between the previous features just completed. Continue with **Side Feature** using **Recognition** and select the arrow **"Normal to surface**". **Click** on the horizontal surface that you rotated to. The vertical surfaces that FeatureCAM is looking for are Normal to this horizontal surface. FeatureCAM uses the horizontal to index to and looks for any vertical surfaces with Recognition at this angle.

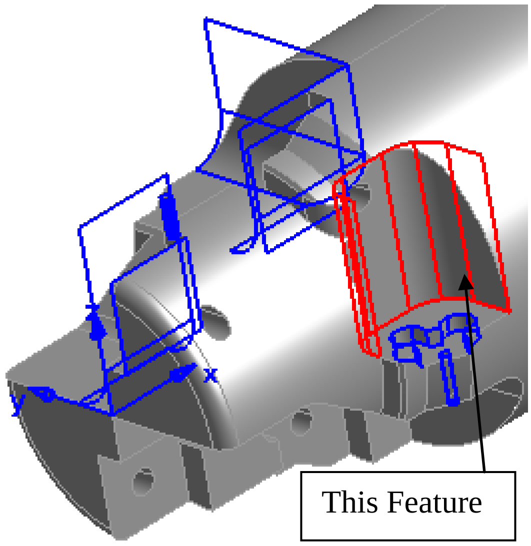


- Select **Automatic recognition**. **Next.**



- **Uncheck** “Exclude features smaller than’ and **Click only** on the vertical surface (red preview in graphics window as shown below) connected to the horizontal surface you selected. **Finish and OK**.



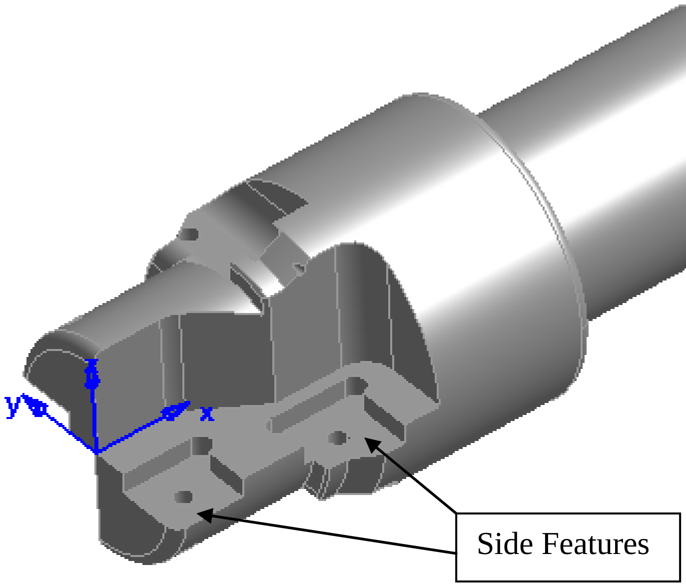


You can extend the lead on the feature if it fails to remove all the material on the short end of the feature.

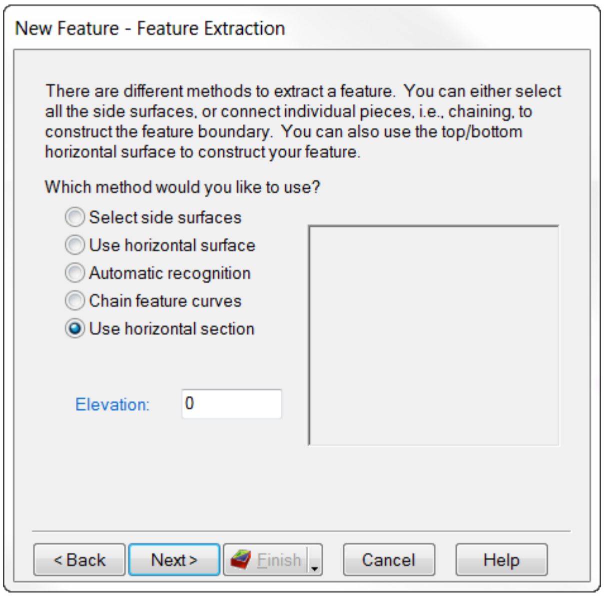
- In the feature **Click** on the **finish** operation and the **Stepovers Tab** and in the “**Lead moves for cuts with open ends**” **Enter** .250 in the **Extension dist**: box. **Apply** and **OK.**

You will do the Pockets for the inserts next.

- **Isometric** view back to 0 or the first feature.
- Open **New Feature** Wizard and Select **Side** and **Recognition**. **Next.**

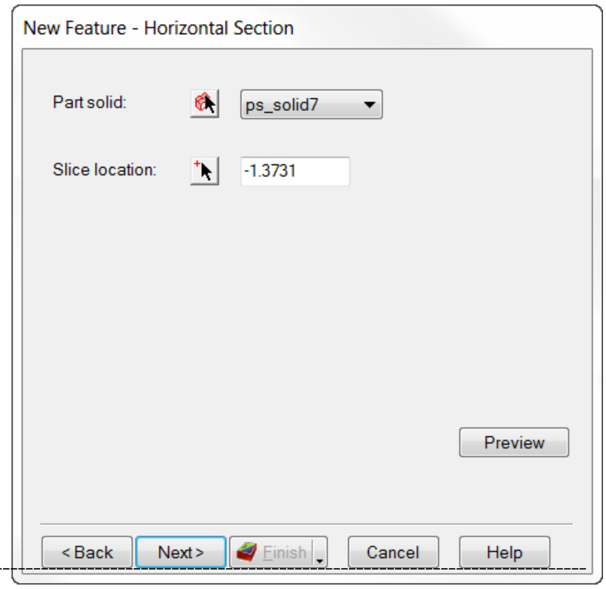


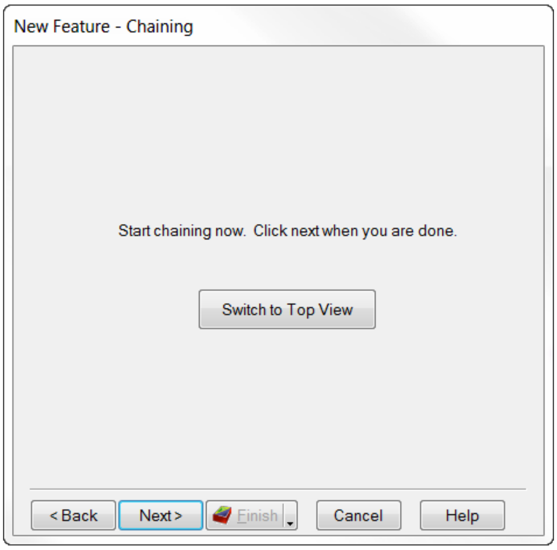
Follow the same procedure for finding the **angle of index** using the edge of a vertical face. It will be the area where the inserts will sit. These will be side features.



- **Select** Use horizontal section. **Click** on the "Blue Word" **Elevation** and **Click** on the Top edge of the vertical face for the feature, this sets the top of the feature. **Next**.

- **Click** on **Slice location** arrow and **Click** on the bottom of the same vertical face and this sets the Bottom of the feature. **Next.**



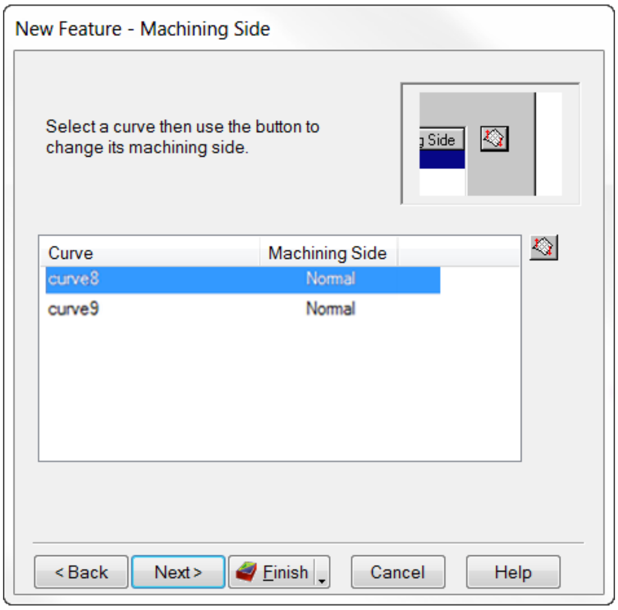


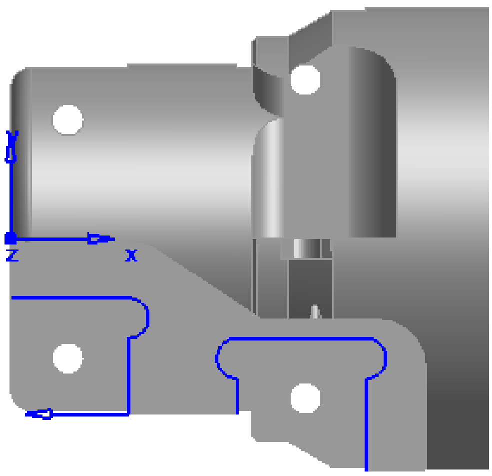
go

- **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. **Next.**

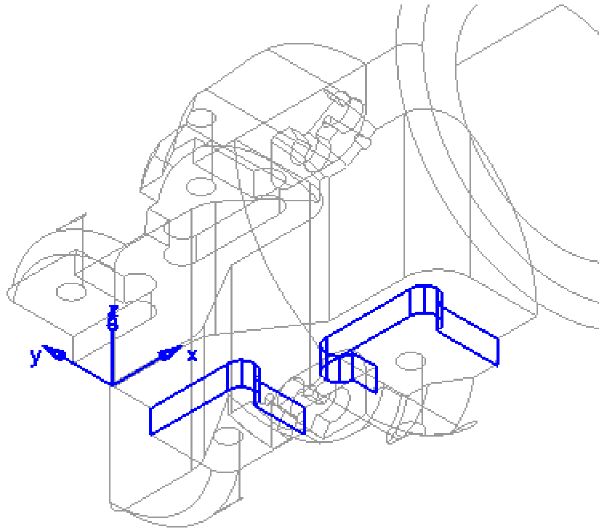
Remember this geometry goes all the way around the part; be sure the curve doesn't there.

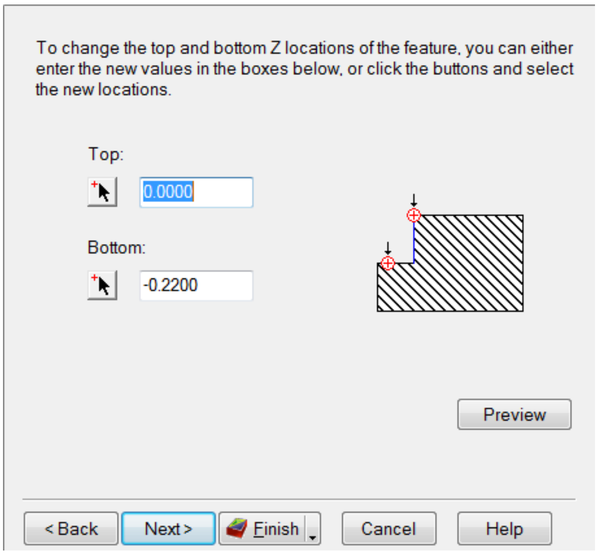
- Check the **Machining Side** arrows, they must point in. Use the reverse button to the right if necessary. **Next.**





- **Confirm the Top and Bottom** and change using the Top and Bottom arrows if necessary and extend the leads as before if necessary. **Click Finish**.

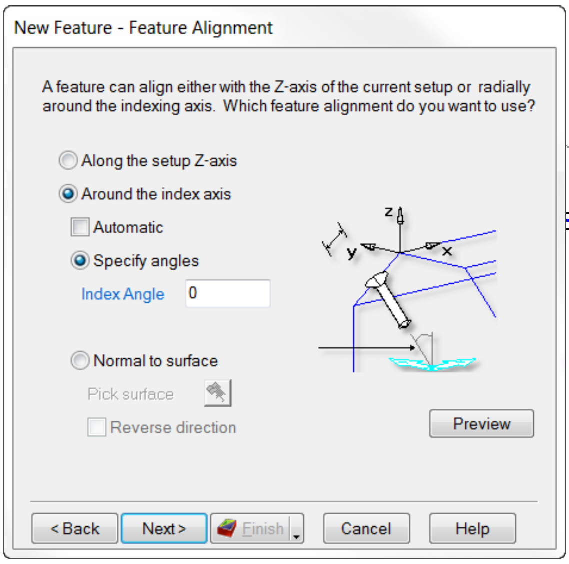


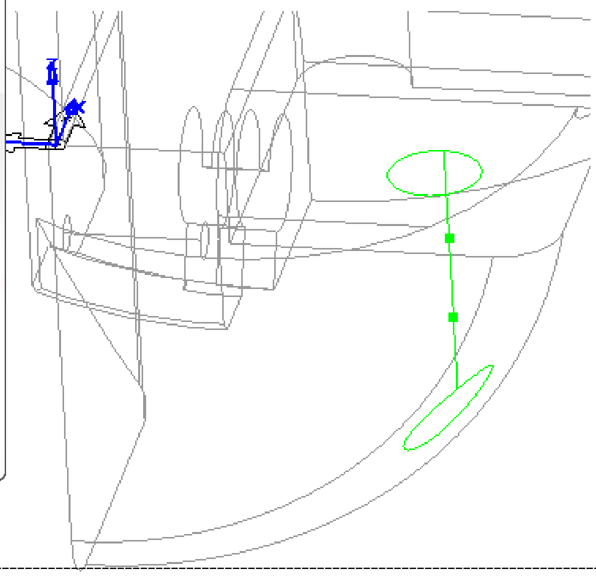


Complete all of the remaining side features.

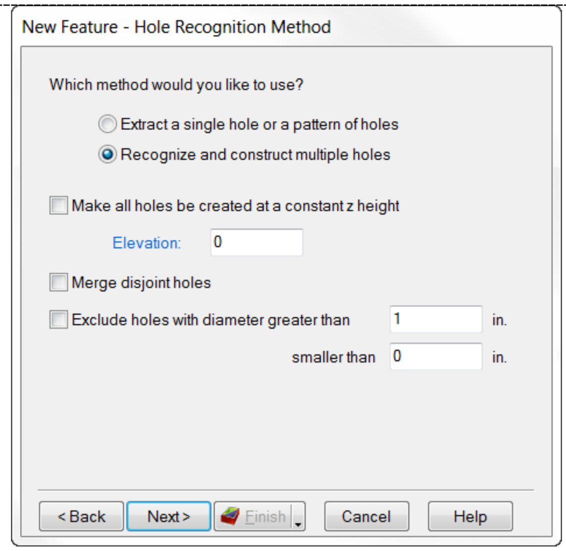
- For hole features, if Automatic is used, FeatureCAM recognizes **ALL** of the holes at **All** of the angles. When the holes have a bottom (blind) FeatureCAM knows from what direction to come with the spindle.
- When the holes are around the indexing axis and are on the centerline, whether they are blind or through like in tubing it also knows from what direction to drill the holes.
- When the holes are not on the centerline and are through like this model, Feature CAM basically has to guess from what direction. To avoid this you will use the isoline in one hole to give the indexing angle and remembering the second click on the line tells FeatureCAM the direction you want the drill to come from. It does ALL holes in Recognition but only at that angle and will not drill any of the others from the wrong direction. You will have to do the various indexing angles for the holes for Recognition until all of the holes are complete.

- Open **New Feature** Wizard and Select **Hole** and **Extract with Recognition**. **Next.**
- 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. **Next.**



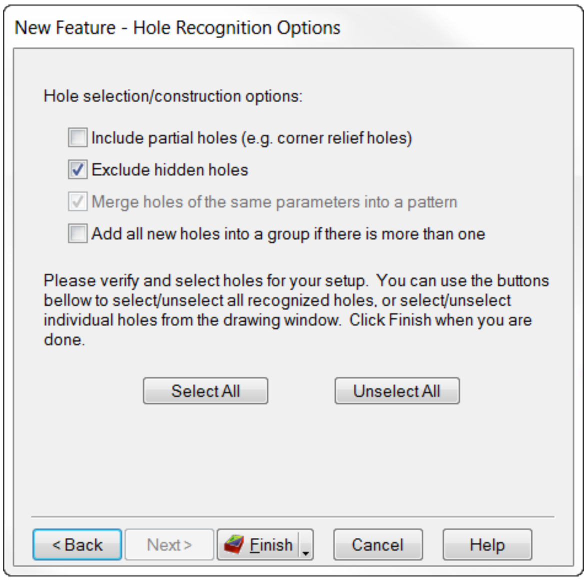


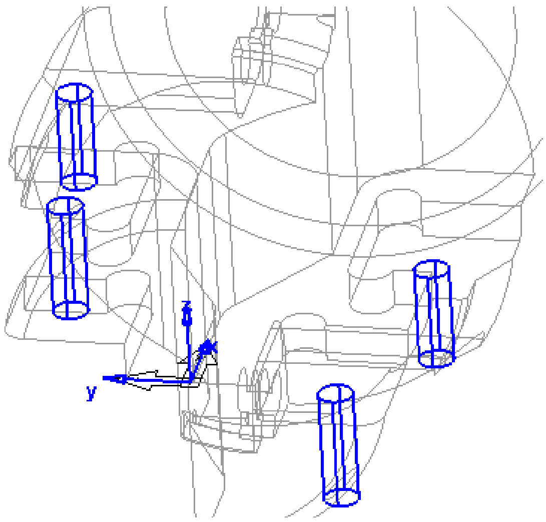
- **Select** Recognize and construct multiple holes. **Next.**



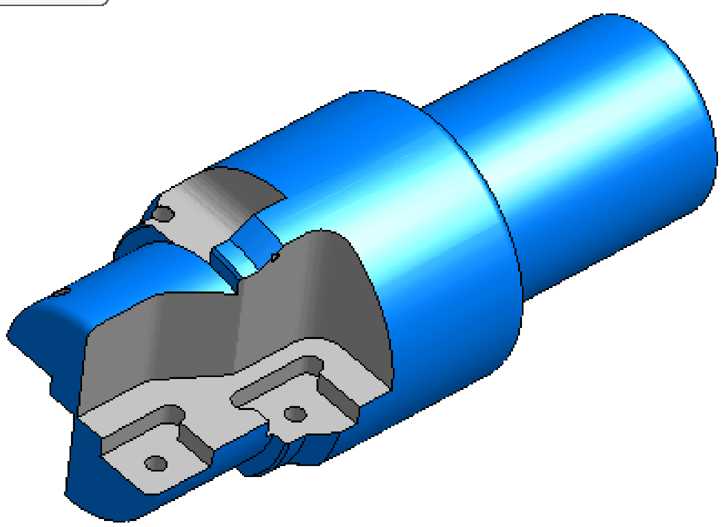
All holes have been selected at your chosen angle, even those on the other side that you don't want and that is because they are an open cylinder and could be drilled from this Z direction.

- You simple choose only the ones you want. **Click** on them and they then turn red as selected. The others remain blue and are not selected.





Continue repeating these steps until all holes are completed then your PART is finished.



Part Library

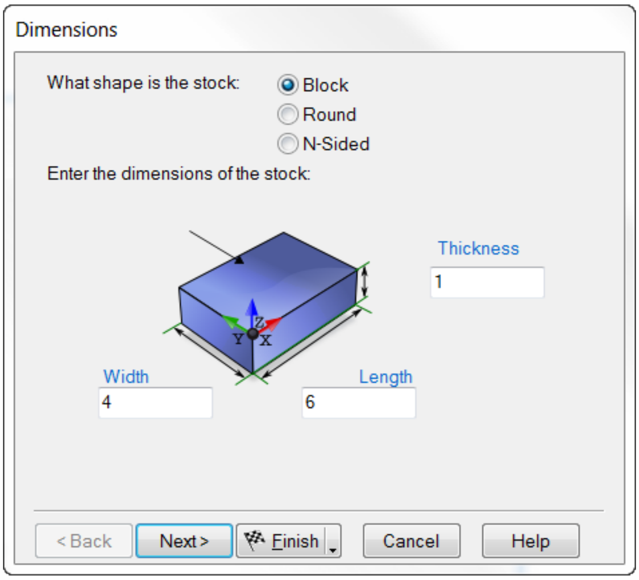
Introduction

The Part Library is a function of FeatureCAM that enables users to save items that they use repeatedly. Rather than having to recreate the same entities, the user can just recall them from the Part Library. In addition to saving specific features, the Part Library can store processes. For example, if a user machines pockets the same way all the time, one pocket can be stored, and then it's parameters and attributes can be applied to future pockets, saving the user from having to change the same settings over and over.

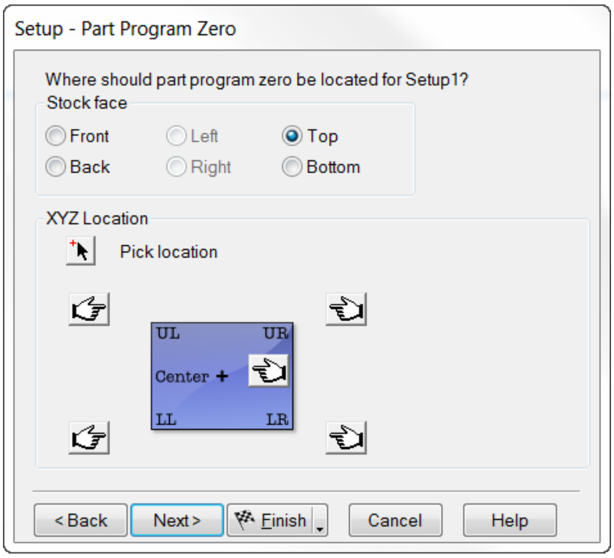
Creating a Part Library Feature

Any feature can be stored in the Part Library to be recalled later. This exercise will demonstrate how to save a feature to the Part Library, and it will offer some tips on how to do this most effectively.

- Open a New milling file
- Fill out the **Dimensions** page of the Stock Wizard as shown
- Click **Next 4 times** to get to the first **Setup – Part Program Zero** page
- Select **Align to Stock** face and click **Next**

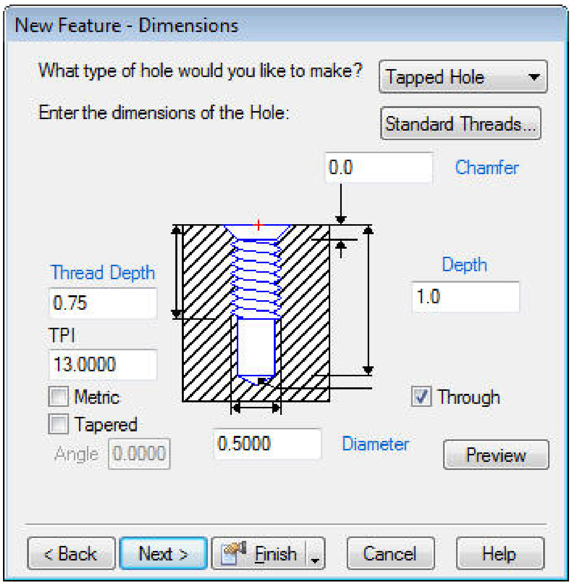


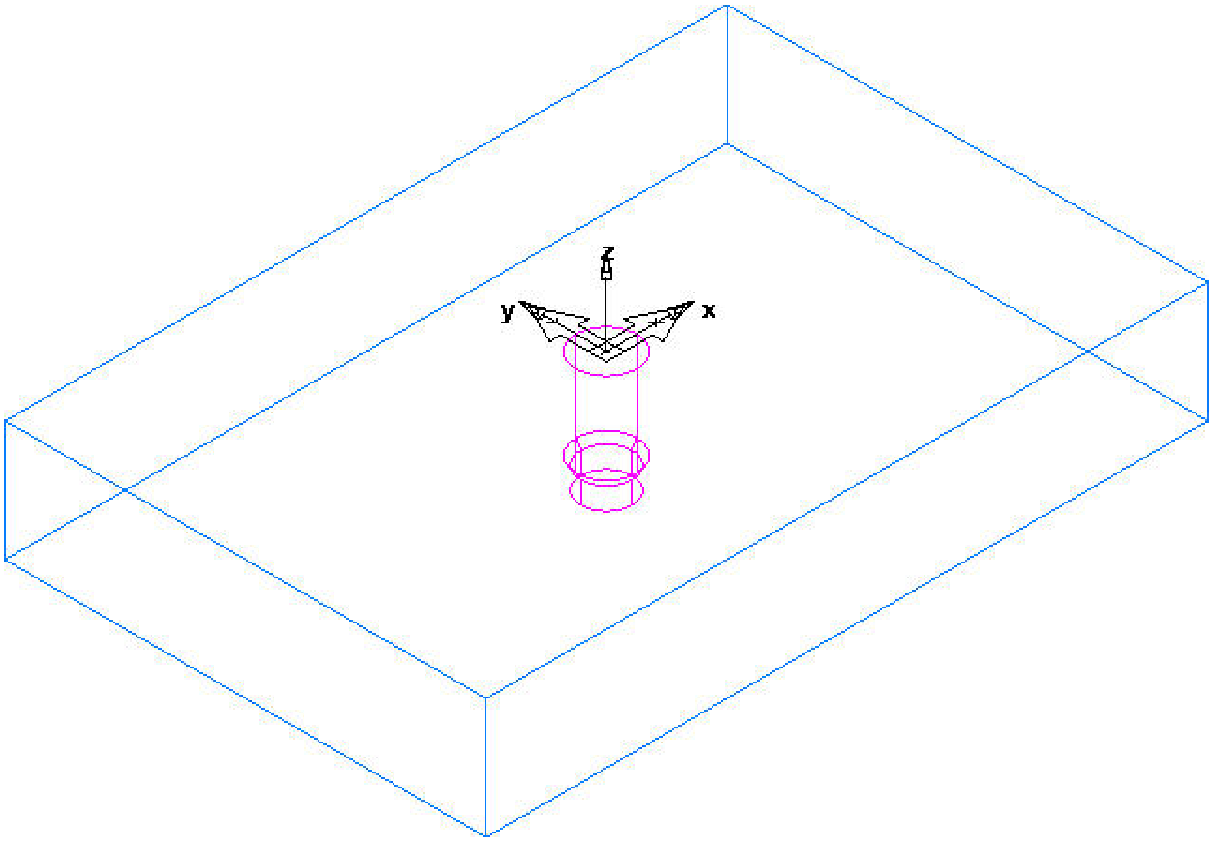
- Designate the setup location as the **center** of the part
- Click **Finish**
- Click **OK** to exit the Stock Properties



In this next step, we are going to create a ½-13 tapped hole to be stored in the Part Library.

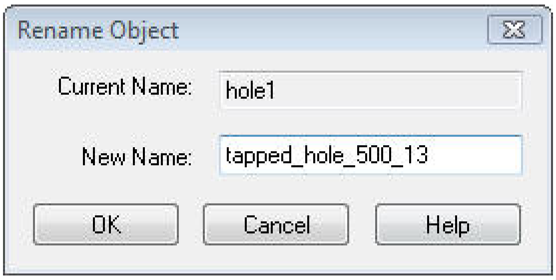
- Launch the **New Feature** wizard, and select **Hole**
- Click **Next**
- Fill out the **Dimensions** page as shown
- Click **Next**, and verify that the hole will be located at **0,0,0**
- Click **Finish**
- Click **OK** to exit the **Hole Properties**





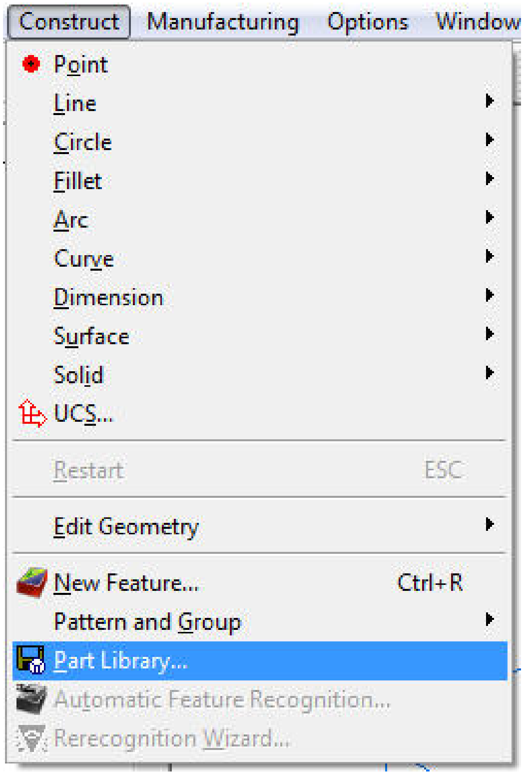
Hint: If possible, it is best to locate Part Library features at the origin. This makes it easier to locate the features accurately when they are loaded into another file. If a Part Library feature is not located at the origin, its original location has to be known and compensated for in the new document.

Next, we are going to rename the feature so that it has a more descriptive name.



- **Right-click** the hole feature in the Part View
- Select **Rename**
- Rename tapped hole as shown
- Click **OK** to accept the new name
- Select the **tapped_hole_500_13** from the Part View

- Open Part Library by selecting **Part Library** under the **Construct** pull-down menu

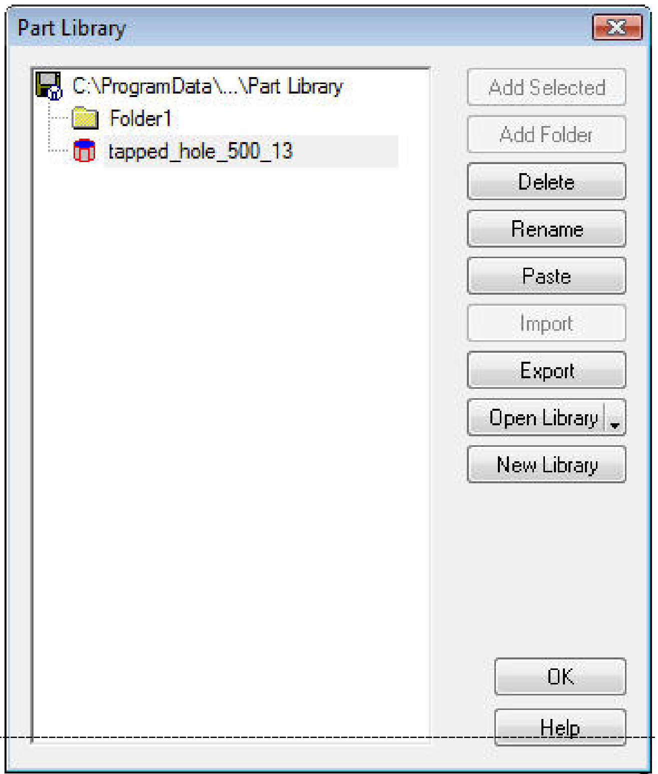


- Save the tapped hole by clicking Add selected

This will save the tapped hole to the "root" of the Part Library. In future examples, we will create sub folders so that the Part Library stays neat and organized.

- Click **OK** to close the Part Library
- **Close** the file without saving

Now that the hole is in the Part Library, we do not need to save the file that defined it. Its definition is contained entirely in the Part Library directory.

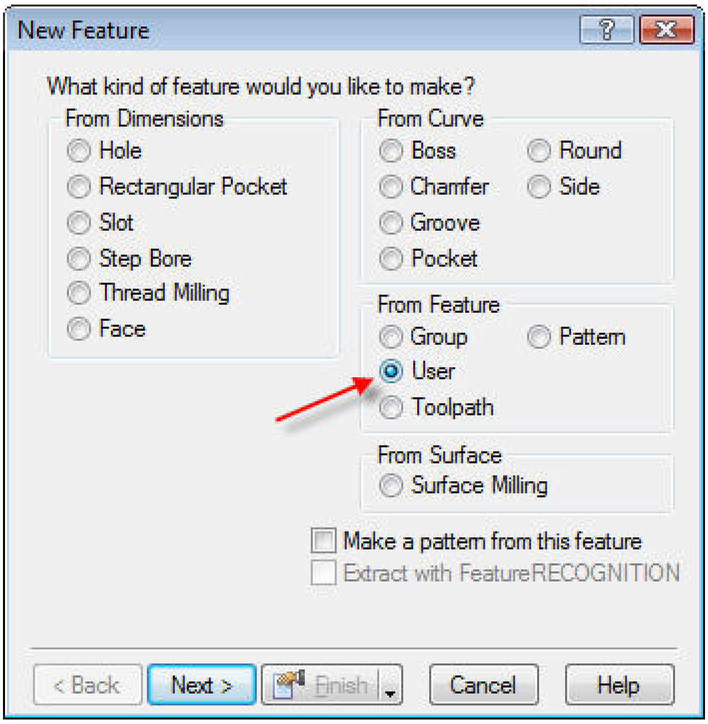


Using Part Library Features

Now that a feature has been saved away, it can easily be installed in another part.

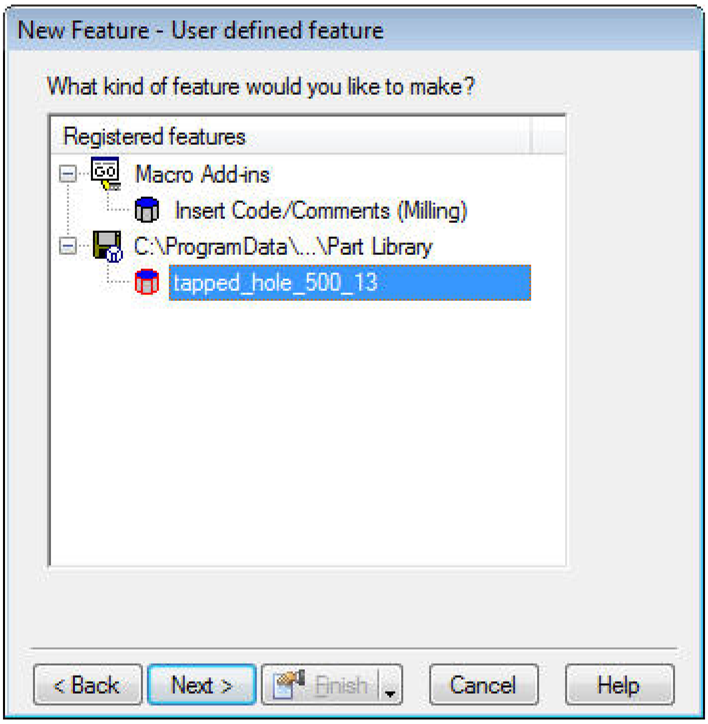
- Start a new milling file
- Use the Stock wizard to create a 4 x 6 x 1 block
- Locate setup on **lower left-hand** corner
- **Finish** the Stock Wizard and exit the Stock Properties by clicking **OK**

- Start a new feature by clicking the New Feature wizard icon
- Select a **User** defined feature



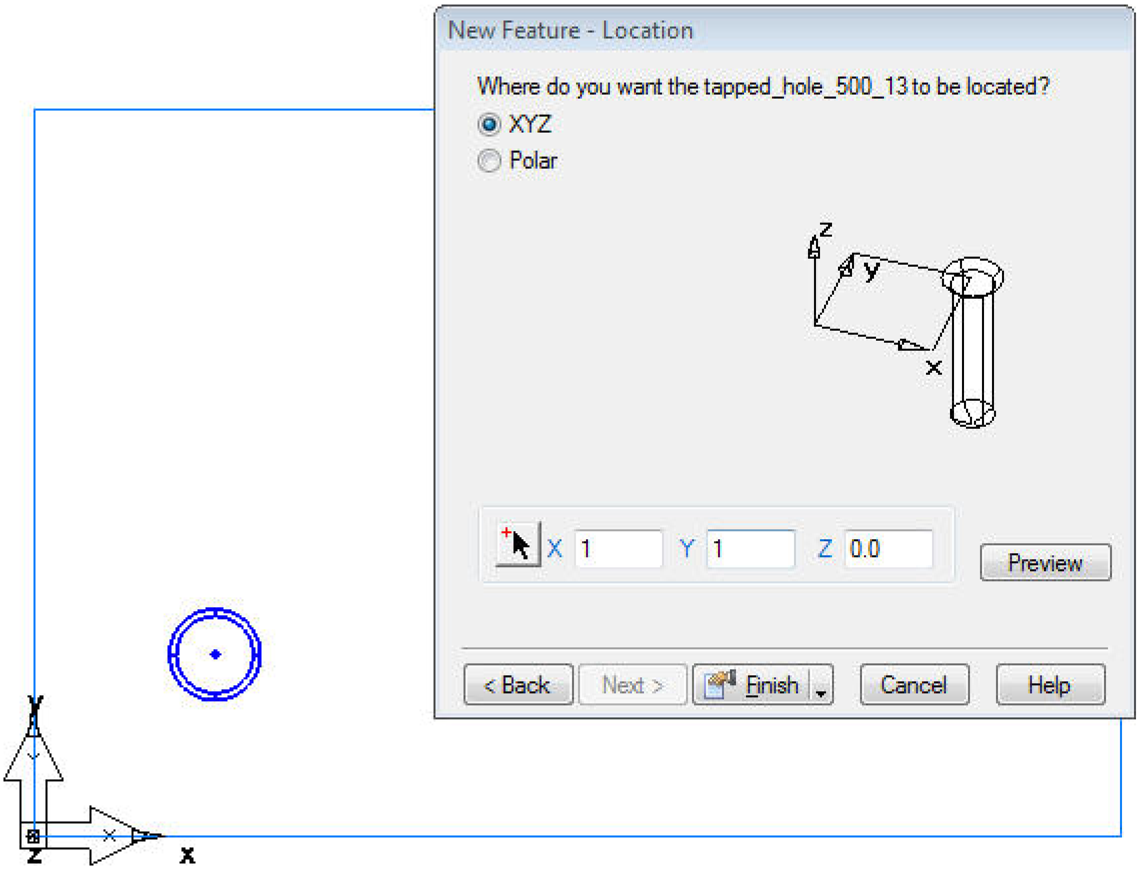
This is one way to access existing features in the Part Library. We will explore other methods later in this chapter. User-defined features include features that can be created by macros and features that are stored in the Part Library.

- Select the **tapped_hole_500_13** from the dialog box and click **Next**



To complete this process, we have to determine the location of the Part Library feature. Since we created our original feature at 0,0,0, we will be able to enter the coordinates without having to adjust for any offset.

- Enter the value **1** for both X and Y



You will notice that our tapped hole is now located in the proper position and is now ready to complete.

- Click **Finish** to complete the feature wizard.

If you open the new feature, you will see that it came in exactly as it was originally defined. This feature can now be edited, if desired, without affecting the Part Library feature - the installed feature is no longer associated to the original.

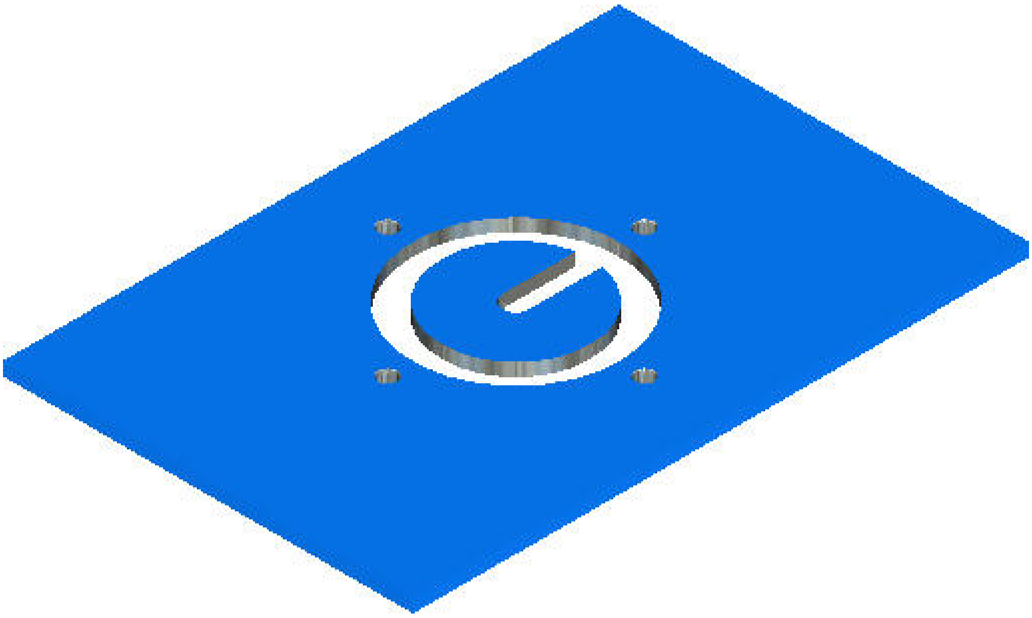
- Close the file without saving

Saving Patterns and Groups to the Part Library

Oftentimes it is not a single feature, but a combination of features that needs to be saved for future use. This exercise will show you how to combine multiple features into groups and then save the group into the Part Library.

- Open **Gauge Mount.fm** from the Data folder

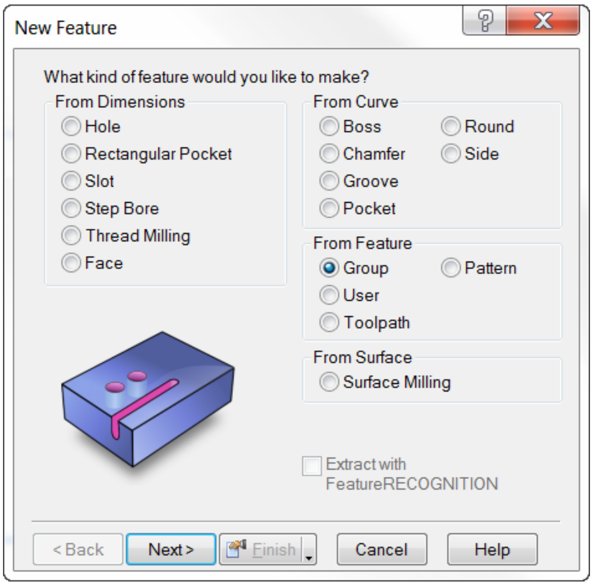
- Run a Simulation



- **Clear** Simulation results

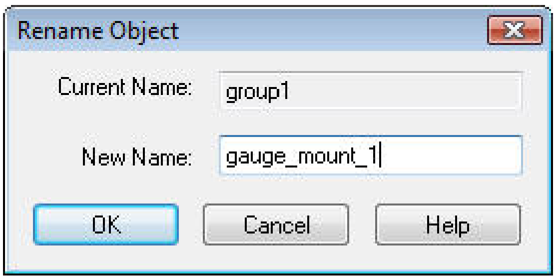
As you can see from the simulation and from the Part View, we have a pocket feature and a hole pattern that we want to always machine as a set. We would like to be able to store them together so that they could be brought in as a single Part Library item. In order for these two features to be stored together as a set in the Part Library, they have to be combined into a Group first.

- Start the New Feature wizard
- Select **Group** from the From Features section
- Add both features them to the Group Members list
- Click **Finish**
- Exit the group properties by clicking **OK**



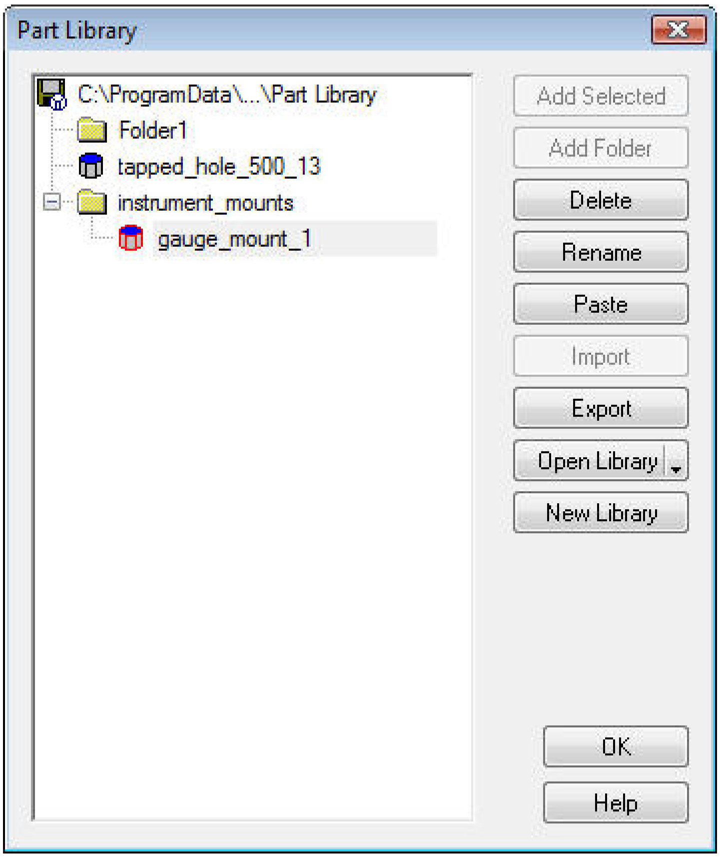
As with the original tapped hole we created in this chapter, we are going to rename our group to a more descriptive name. This is especially useful if we are going to have several similar features stored in the Part Library.

- Right-click on the group in the Part View
- Select Rename
- Rename the group as shown



From here, the process is exactly the same. For this part, though, we will create a new folder in the Part Library. You can create folders to organize your features and keep the Part Library from getting too cluttered.

- **Highlight** the Group in the **Part View**
- Open Part Library
- Click **Add Folder**
- Select the folder and click **Rename**
- Rename folder to **instrument-mounts**
- Click **Add Selected** to save group to new folder



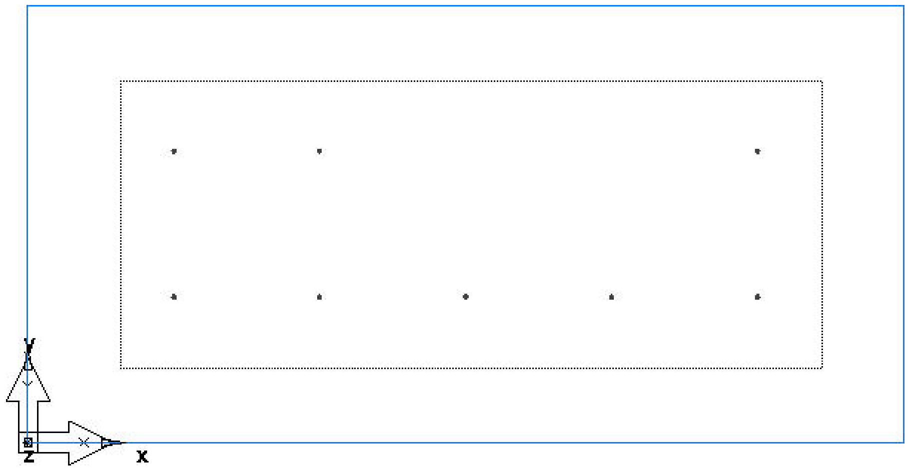
You will see that gauge_mount_1 is added to the Part Library as a child of the instrument_mounts folder.

- Exit the Part Library by clicking **OK**
- Close the file without saving

Patterning Part Library Features

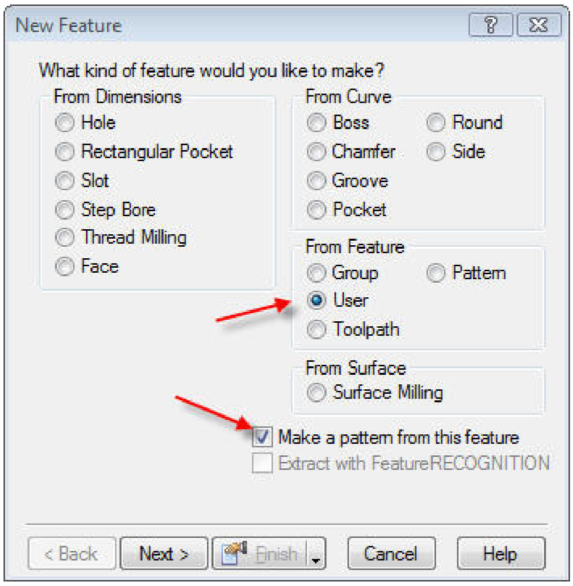
If a user wants to install multiple instances of a Part Library item into a file, this can be done by utilizing Patterns. All of the pattern types such as Linear, Radial, and Rectangular are available. If the positions of the Library features are laid out in advance, the process can be made very quick and easy by creating a Points Pattern.

- Open **Instrument Panel.fm** from the **Data** folder

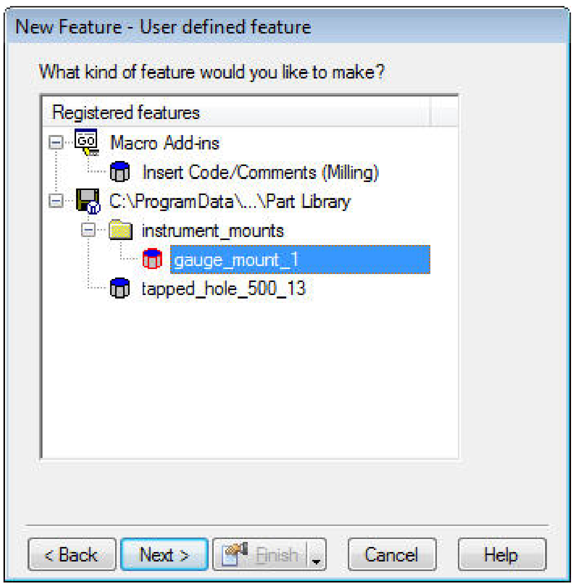


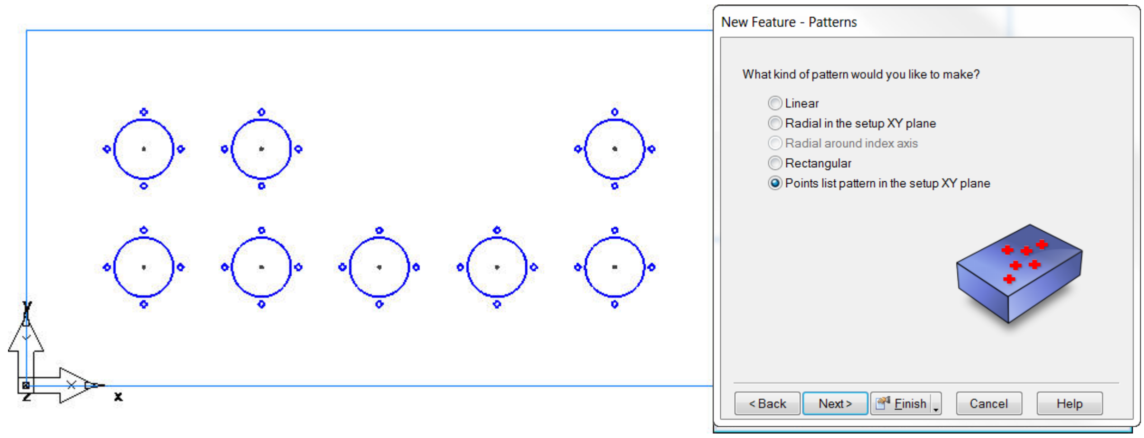
This file simply contains the locations at which we will locate our Library features. If the features are created correctly, they will come in perfectly positioned at each one of these points.

- Start by selecting all of the points by dragging a selection box around the entire part.
- Start a **New Feature**
- Select **User**
- **Make a pattern from this feature** should already be checked since we preselected more than one point
- Click **Next**



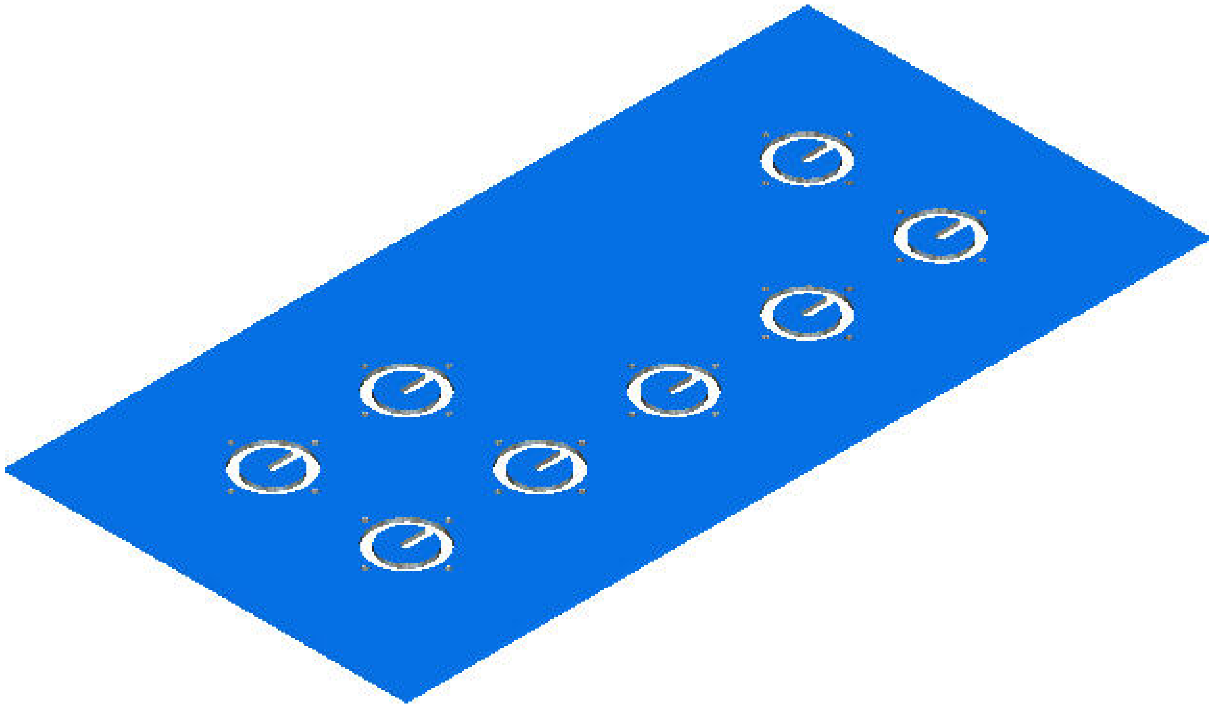
- Select the **gauge_mount_1** group from Part Library
- Click **Next**





The Points Pattern is the style of pattern already selected because of our collection of points.

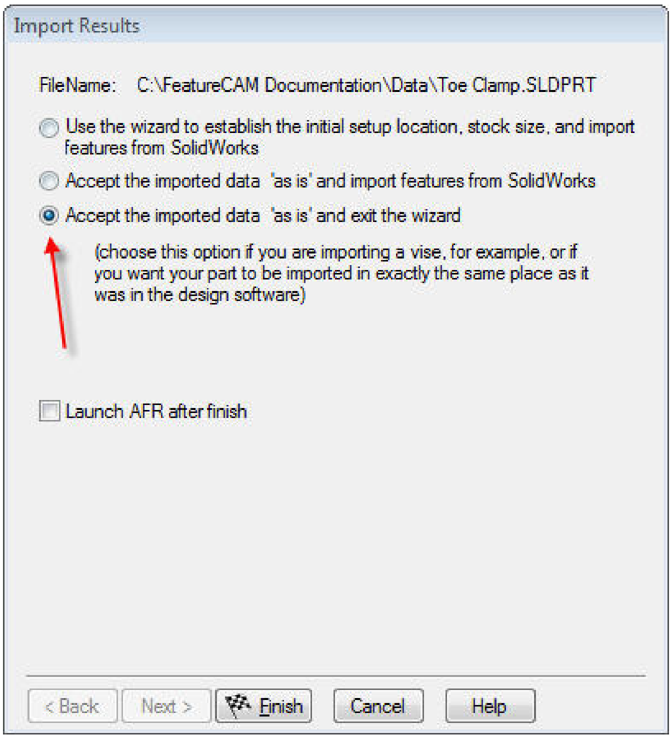
- Click **Next**
- On the Dimensions tab, you can alter the sorting order, if desired
- Click **Finish** to complete the wizard
- Click **OK** to close the Feature Properties
- Run a simulation to see the results



- Close the file

Saving Solids to the Part Library

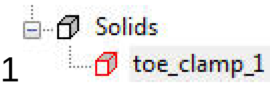
Solid models that are brought into FeatureCAM can also be stored in the Part Library for later use. This capability is especially handy for using solids as fixtures and clamps. The following example will demonstrate saving a solid to the Part Library and then loading it into a FeatureCAM document.



- Open a new Milling file
- Cancel the Stock Wizard
- Import **Toe Clamp.sldprt** from the **Data** folder
- When the Import Results screen appears, choose to **Accept the imported data "as is" and exit the wizard**
- Click **Finish**

The clamp comes in located at 0,0,0. Now we will save it to the Part Library so that it can be easily loaded – and even patterned – in future files.

- Expand the **Solids** folder in the Part View and **right-click** the solid
- **Rename** the solid toe_clamp_1



Again, it is good practice to rename entities that are going to be used at a later date with descriptive names.

- Make sure the solid is still selected and open the **Part Library**
- Click **Add Folder**
- Rename the new folder **clamps_and_fixtures**
- Highlight the **clamps_and_fixtures** folder and click **Add Selected** to add the toe clamp to the Part Library
- Click **OK**
- Exit the file without saving

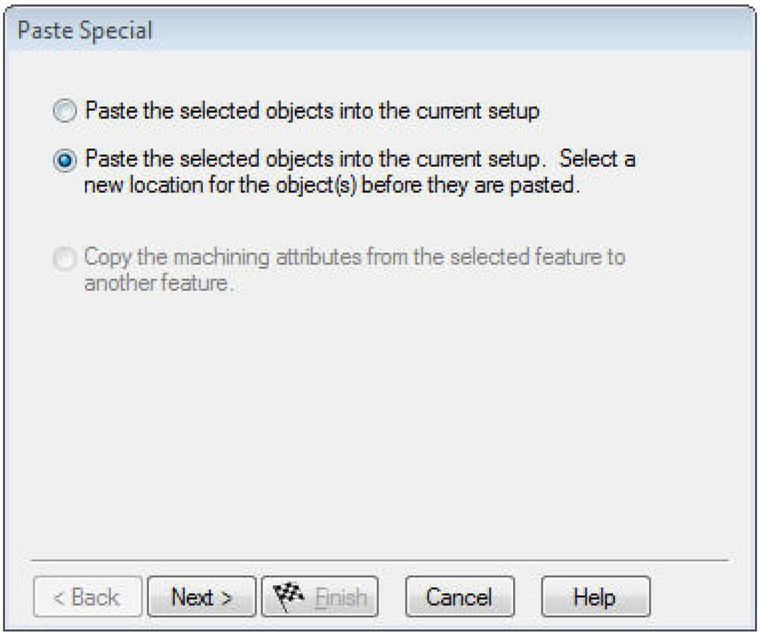
Now that the toe clamp is stored and ready to use, we will bring it into a finished version of the Instrument Panel that we worked on earlier. Since the clamp is not a feature, we will not do this through the New Feature wizard. Instead, we will copy the toe clamp directly from the Part Library.

- Open **Instrument Panel – Finished.fm** from the **Data** folder
- Click **Construct > Part Library**
- Select **toe_clamp_1** from the clamps_and_fixtures folder
- Click **Paste**

This exercise will introduce us briefly to the Paste Special function. We will study Paste Special in further detail later in the chapter.

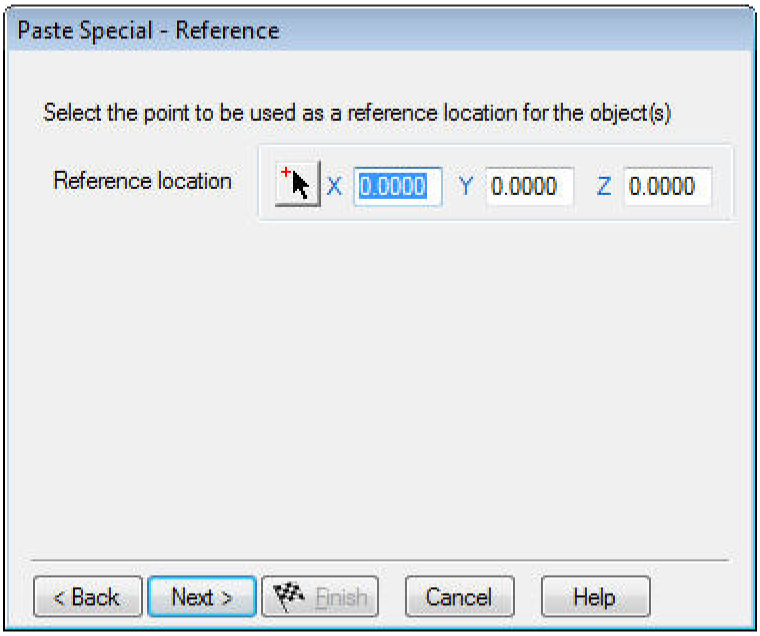
We want to paste the toe clamp into our part, but we want to be able to designate where it is located. For this, we need to use the second choice of the Paste Special options.

- Select the second option as shown



Before we give our feature location dimensions, we need to define a reference point. This is a nice tool because you do not have to locate the entity relative to an origin. If you would rather locate the clamp 3 inches from a specific point, you could select that point as your reference point. In this example, however, we will use the origin as our reference.

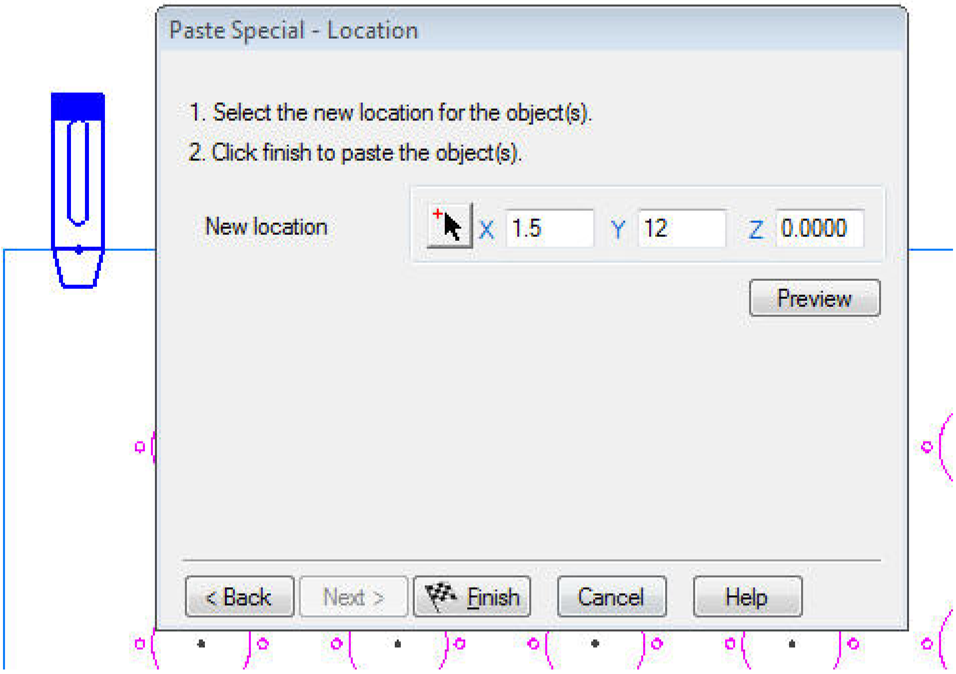
- Make sure the reference point is 0,0,0 and click **Next**



Now that the reference point is defined, enter the position of the clamp.

- Enter position for clamp as shown below

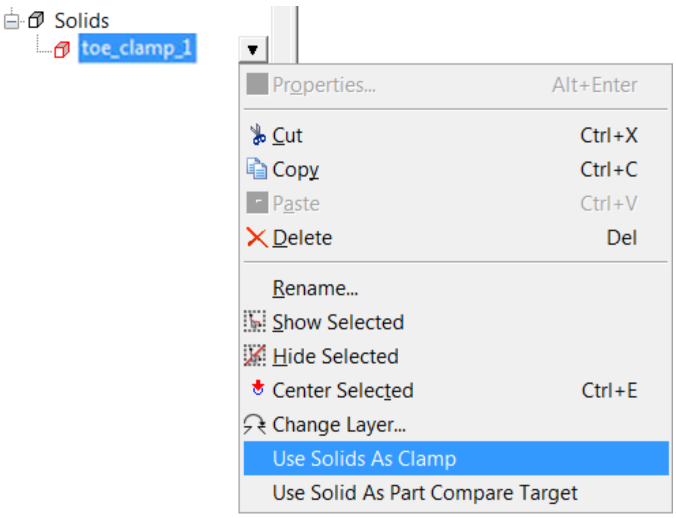
- Click **Preview** to confirm the location



- Click **Finish** to place the solid
- Click **OK** to close the Part Library

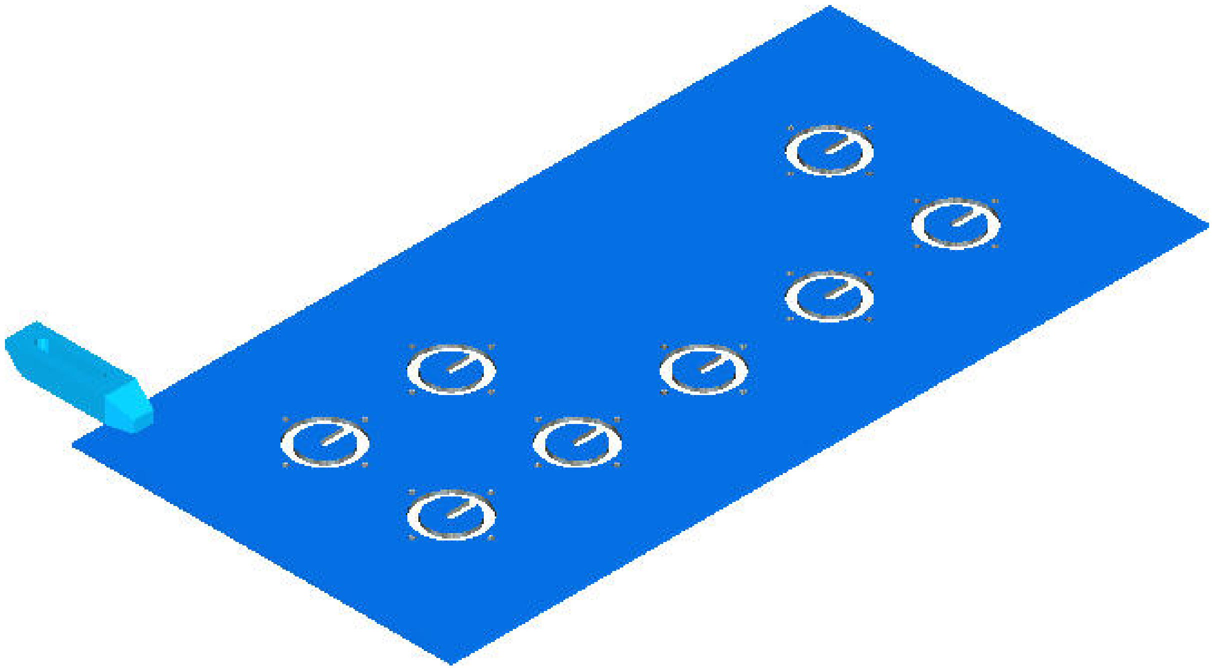
Solids that are brought into FeatureCAM files can be used to represent fixturing during simulation. However, this requires a few more steps. If you were to run a 3D simulation now, the clamp would not appear.

- Right-click on the toe_clamp_1 solid in the part view
- Select Use Solid As Clamp



Now that the solid has been designated as a clamp, it will show up in our simulation. It will even detect gouges if a tool or tool holder contacts it.

- Run a 3D simulation



- Close the file without saving.

Paste Special

We have already seen how Paste Special can be used to position a Part Library item in a FeatureCAM file. Paste Special has a couple other functions that help to load a feature or its parameters into a file.

- Open the file **Paste Special.fm** from the Data folder
- Make sure the **basic** tool crib is active

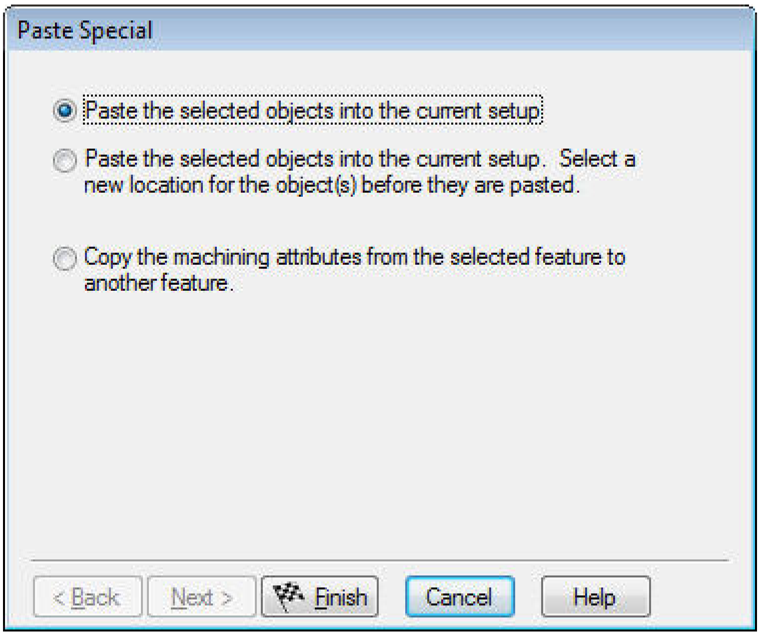
This file contains two pocket features that have different settings, depths, and parameters. We will use Paste Special to create and modify features based on the settings of a Part Library feature. First, we will save Pocket1 to the Part Library.

- Click on **pocket1** to highlight it
- Open the Part Library
- Add a **New Folder**
- Rename the folder **pockets**
- Select the **pockets** folder and click **Add Selected** to save the pocket to the Part Library

We will use this pocket as the templates for other pockets in the part file. First, we will create an entirely new pocket with all the parameters of pocket1. This strategy is very useful if you want to apply dimensions and attributes of one feature to different curves.

- Click on **pocket1** in the Part Library to highlight it
- Click **Paste**

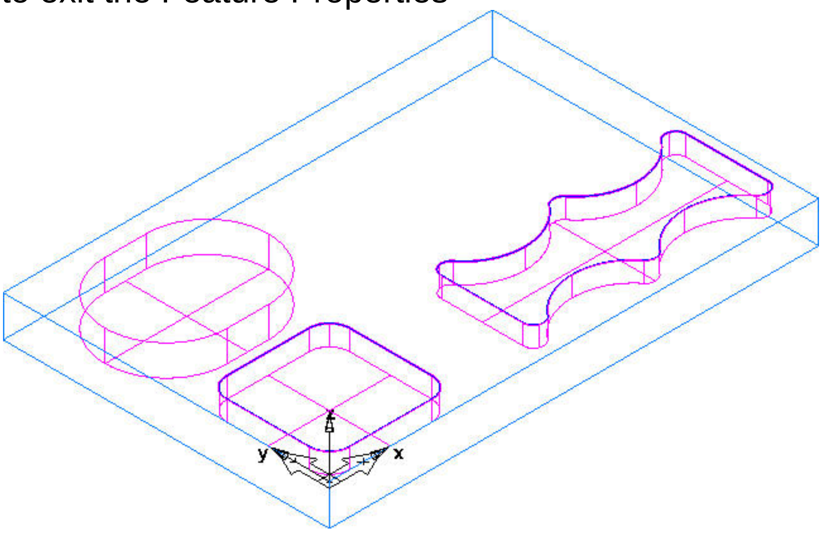
The Paste Special dialog box appears. We will paste the feature "as-is" into our part file.



- Select "**Paste the selected objects into the current setup**"
- Click **Finish**
- Click **OK** to exit the Part Library

You will notice that there is a new feature in the Part View, but it is not visible in the graphics. This is because we have two identical features in the part file. Next, we will modify the new feature be selecting a different boundary curve.

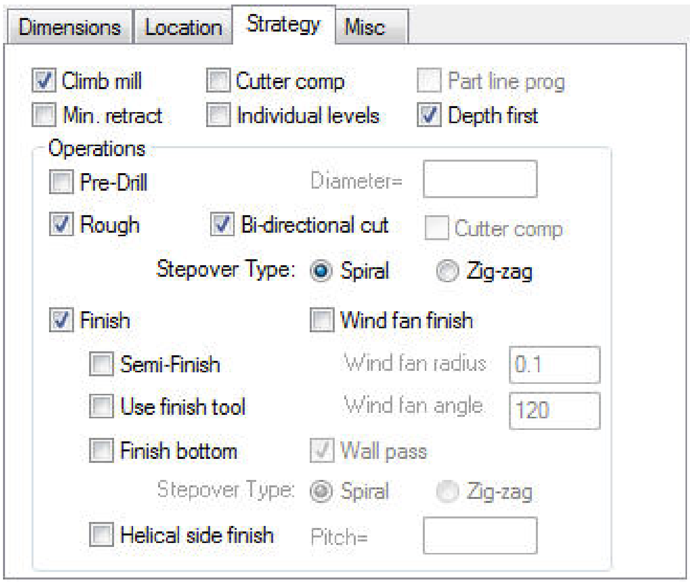
- Double-click **pocket1_1** to access its properties
- Click on the **Boundaries** button
- **Clear** the current curve selection and **select Curve2**
- Click **OK and Apply**
- Click **OK** to exit the Feature Properties



We now have a new feature attached to Curve2. This feature has the exact same *parameters and dimensions* as the original feature.

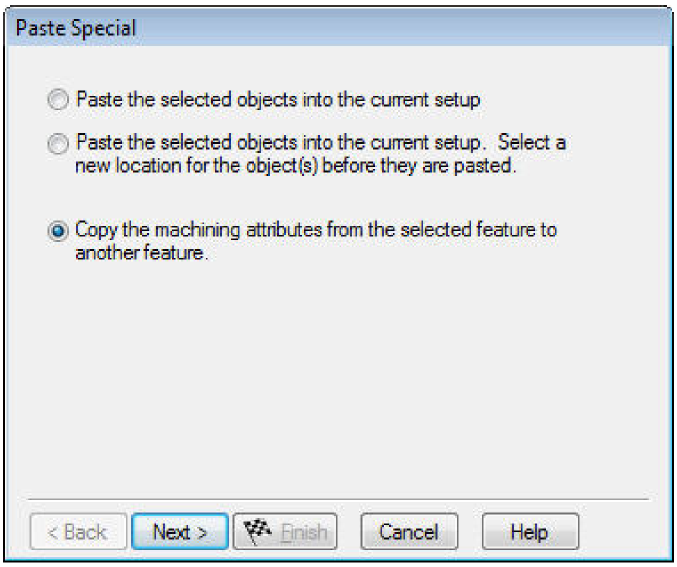
Hint: When you create a feature to be stored in the Part Library, avoid overriding any tools unless you want to designate specific tools to the feature. By keeping default tools in place, you allow FeatureCAM to change the tool selection based on the geometric conditions, and you create a more flexible Part Library feature. If you compare pocket1 and pocket1_1, you see different tools are selected because of the different radius values of the corners.

Another way to use Paste Special is to apply the Machining Attributes of the library feature to another feature. In this example, we will apply the attributes of pocket1 to pocket2. First, open the properties of pocket2 and examine the Strategy tab.



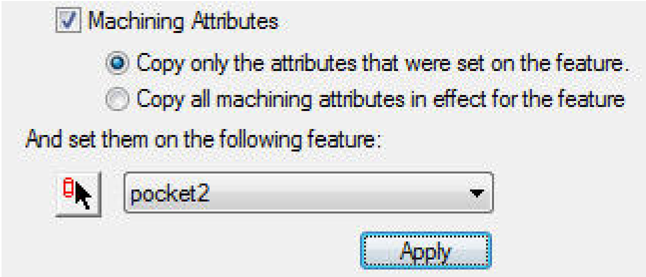
Pay attention to which attributes are checked, and exit the properties.

- Open the Part Library
- Click on **pocket1** in the Part Library to highlight it
- Click **Paste**
- Select “Copy the machining attributes from the selected feature to another feature”



- Click **Next**
- Check the **Machining Attributes**

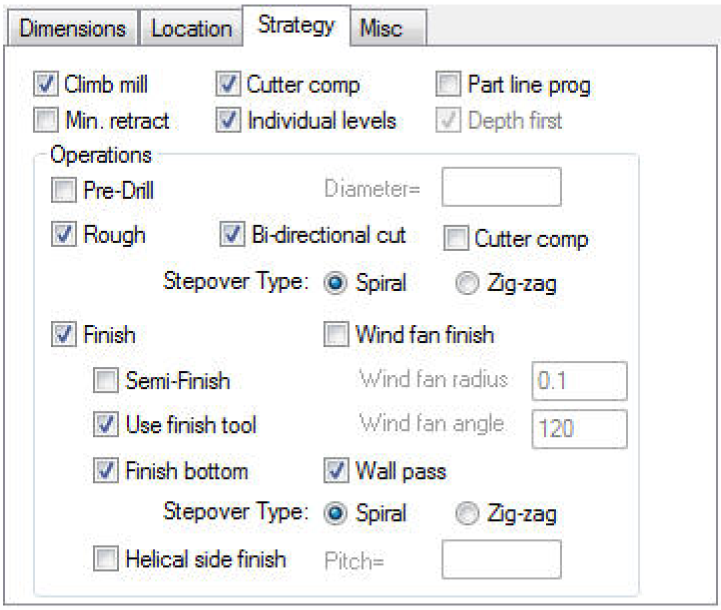
You have a choice of applying all the machining attributes of the library feature, or only the attributes that were overridden in the library feature.



of

- Choose to copy all of the attributes as shown
- Click **Apply** and **Finish**
- Close the Part Library

Now take a look at the Strategy tab of pocket2.



Options such as Individual Levels and Cutter Comp, which were characteristics of pocket1 are now installed in pocket2. This strategy is useful when you want to machine features of different dimensions in a very consistent manner. All of the attributes get applied without altering the geometry of the feature.

Another note about Paste Special: Paste Special is not limited to being used in conjunction with the part library. You can use Paste Special more “locally” by copying a feature from the part view and then using Paste Special. The Paste Special can occur within the same part, or in a different part.

Streams

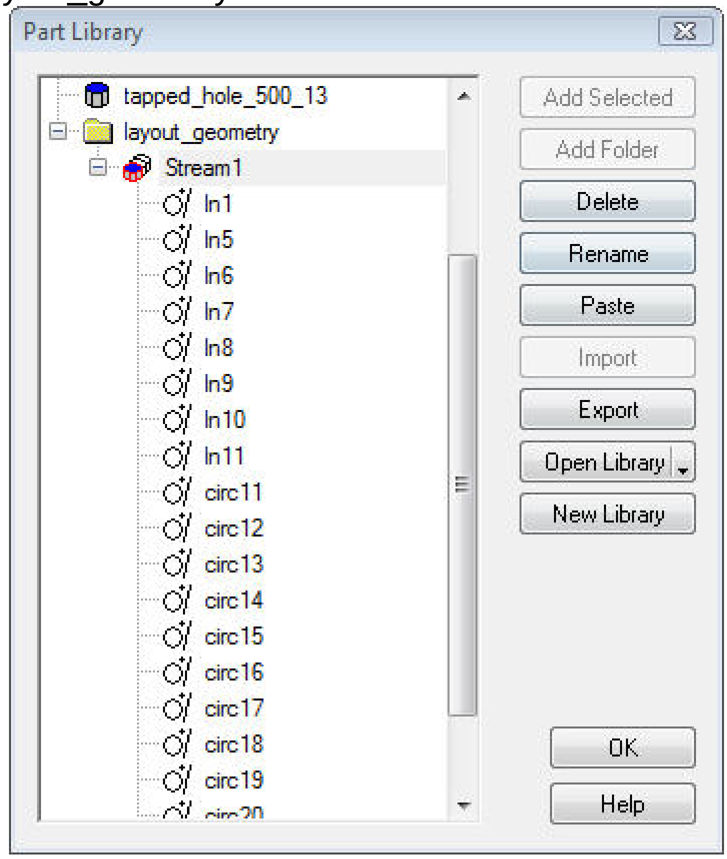
Entities other than features and solids can also be stored in the Part Library. Curves, geometry, and machine functions can also be saved and recalled. These items show up as Streams in the Part Library.

Oftentimes, geometry is reused from one part to another. For example, there might be a fixture that has specific layout geometry. In that case, the layout geometry can be stored away in the Part Library and used repeatedly as a template.

- Open the file **Layout.fm** from the Data folder

The lines and circles in this file represent a bolt hole pattern in a fixture. We would like to save this pattern to the Part Library so that it can be transferred to a new document.

- Select the geometry by dragging a selection box around the entire part
- Open the Part Library
- Add a new folder
- Rename the folder **layout_geometry**
- Select the layout_geometry folder and click **Add Selected**



- Exit the Part Library
- Close the part

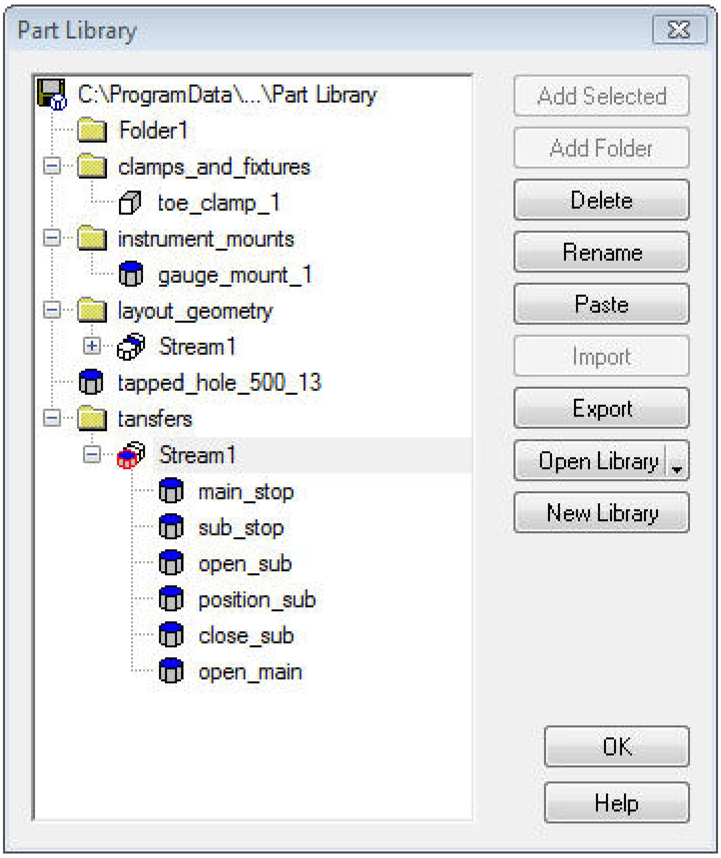
Each geometry entity is stored under the **Stream1** heading. The Stream1 heading can now be renamed to something more descriptive. This new geometry stream can now be added to new files using the Paste Special function.

Streams can also be used to store other types of features. This sample will construct a stream of lathe functions that will be very similar to the Grouping method that we have already seen.

- Open the file **Turn Part Transfer.fm** from the Data folder

This file contains Subspindle features that carry out a part transfer on a lathe. It would be quite time consuming to create each one of these steps manually for every turning file, so the entire set of steps can be stored in the Part Library as a Stream.

- Select all of the features by dragging a selection box around the entire part
- Open the Part Library
- Add a new folder
- Rename the folder **transfers**
- Select the **transfers** folder and click **Add Selected**



Each subspindle feature is stored under the Stream1 heading. The Stream1 heading can now be renamed to something more descriptive. This new feature stream can now be added to new files using the Paste Special function.

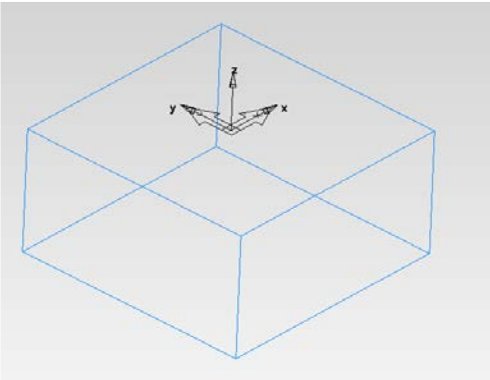
Creating Machining Set-Ups

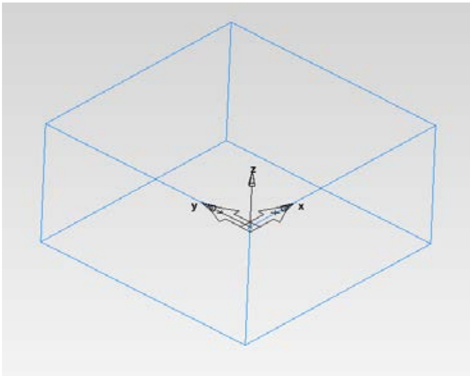
Introduction

Before starting any machining operation, it is necessary to set a **Datum** position from which to work from. In **FeatureCAM** these positions are called **Set ups** and these can be defined using a number of techniques.

Creating SetUps from the Stock

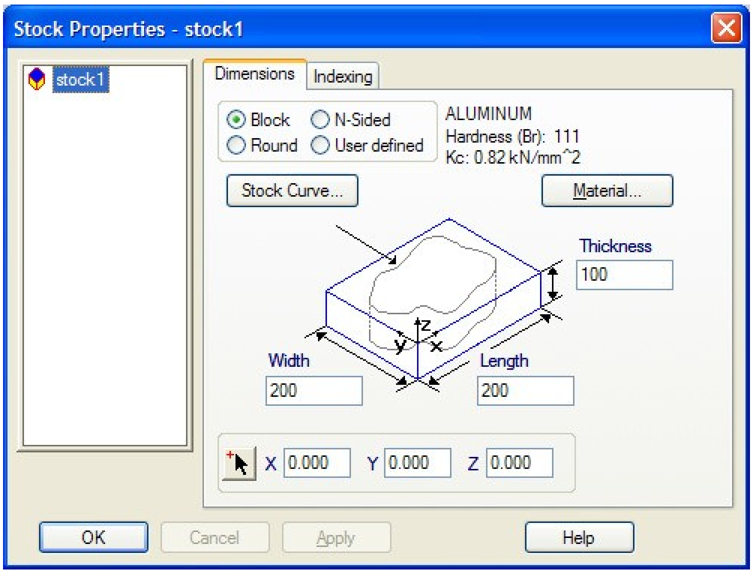
The Stock represents the material which is to be machined. The most common places that are used when defining the **Set Up** on a standard Block are the **Centre of a Face** or **one of the Corners** as shown below.



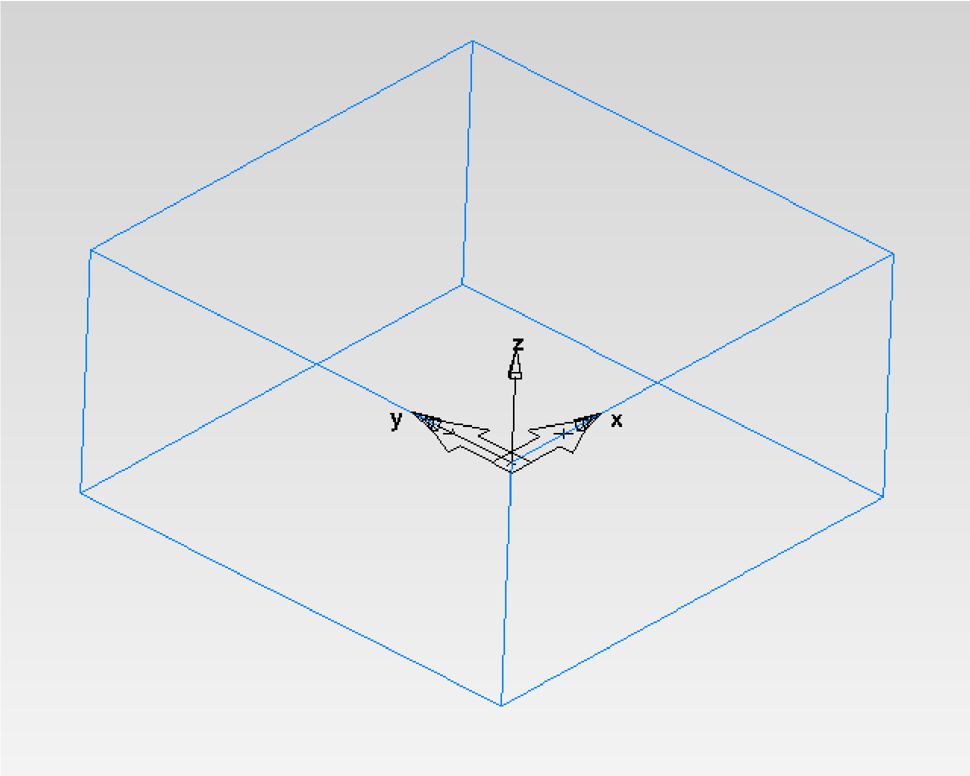


When FeatureCAM opens a new part a **Stock** is automatically displayed on the screen.

- Double click on the stock either in the **Part View Toolbox** or in the main **Graphics** area.
- Fill in the Form as shown below, **Length=200, Width= 200** and **Thickness=100**



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 Co Ordinate System** as shown in the image below

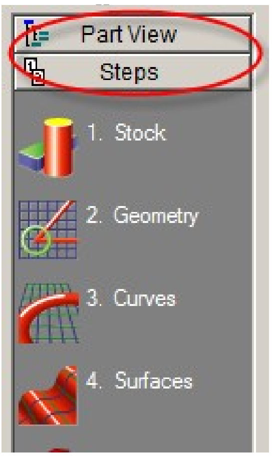


Change the figures in the input fields and watch how the **Stock** moves around.

- Reset the figures to **X0 Y0 Z0.**

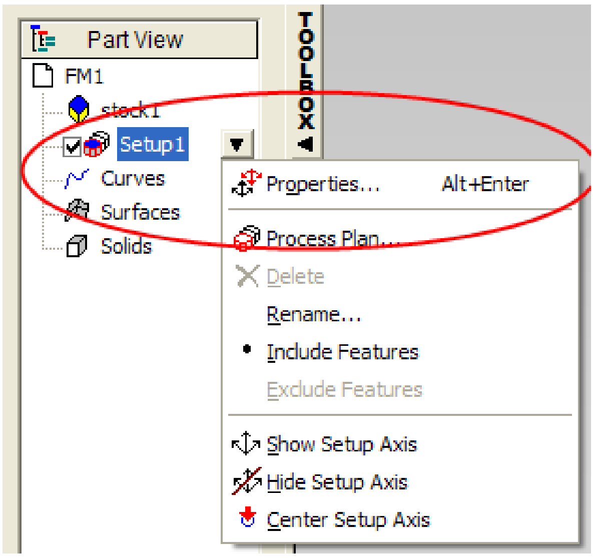
To place the **Set Up** in the centre of the **Block**, it is simply a case of moving the Set UP to the centre of a Stock face and this is achieved by changing the position, using the **Part View Toolbox**

- 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**



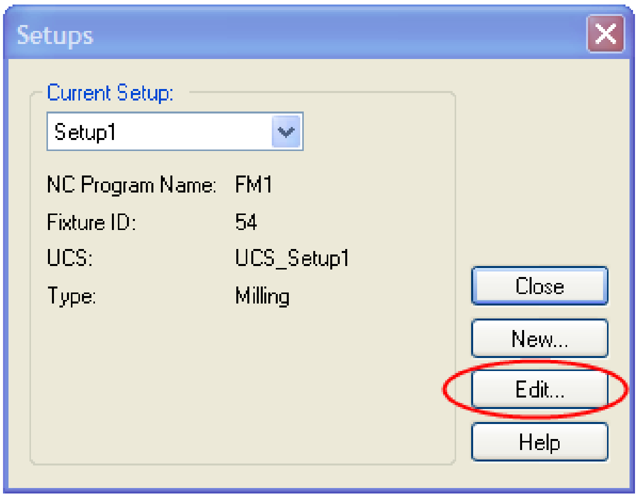
If you can only see the **Part View Toolbox** and not the Steps options as shown....Try looking at the bottom of the screen. (When it is not in use, the **Steps Toolbox** drops down to the bottom, just click on the tab to reactivate the toolbox.)

- Right click on the **Setup** in the **Part View** and then select **Properties** from the menu that appears as shown.

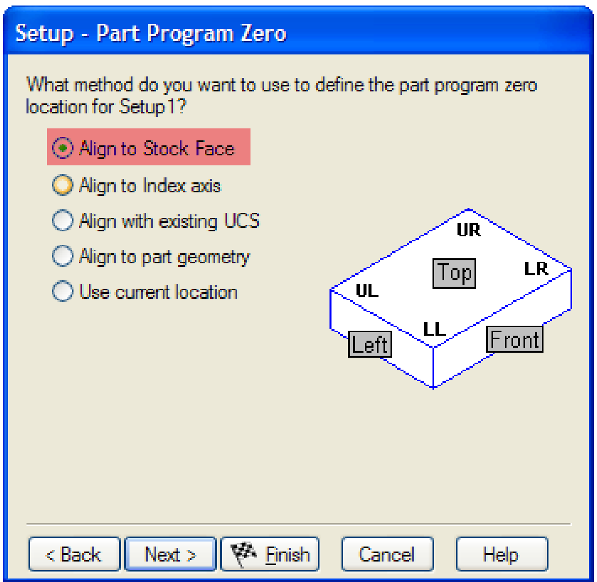


- This will open the Set- Ups form,

On this form, the user can choose whether to create a **New Setup** or to **Edit / Change** an existing one.



- Click on **Edit**, followed by **Next**

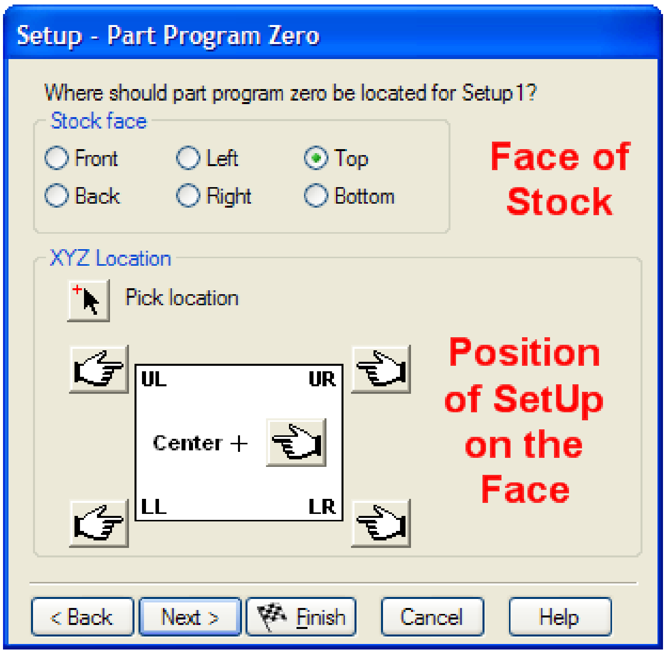


- Select **Align to Stock Face**

The form is divided into two areas and these are **Stock Face and XYZ location**

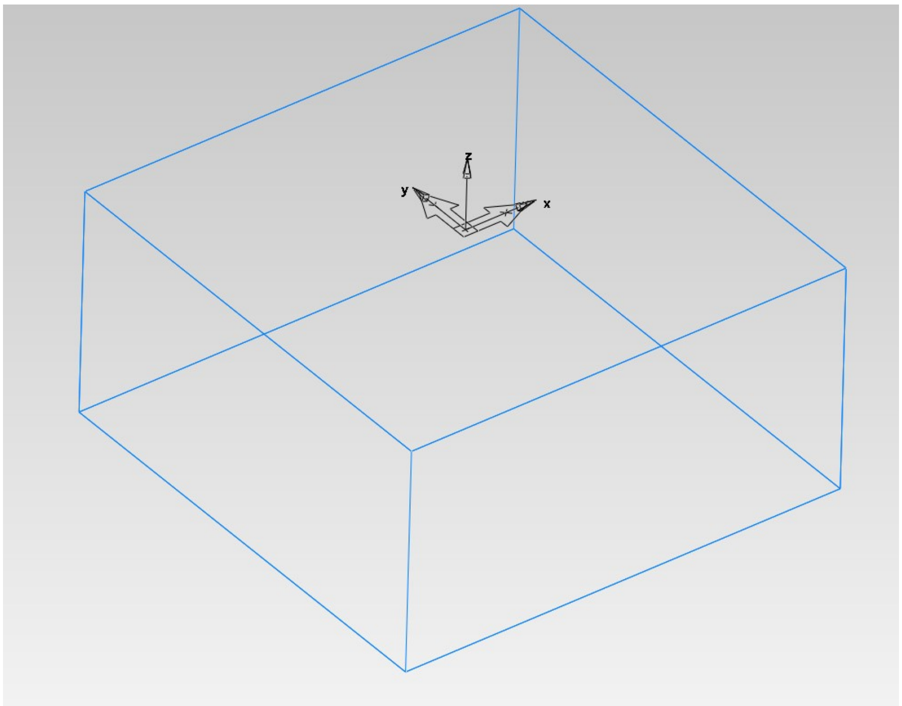
Stock Face is used to tell **FeatureCAM** on which **Face** to place the **Setup**.

XYZ Location is used to position the **Setup** on that Face



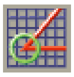
- Select **Top** followed by **Centre.** Click **Finish** and then **Close.**

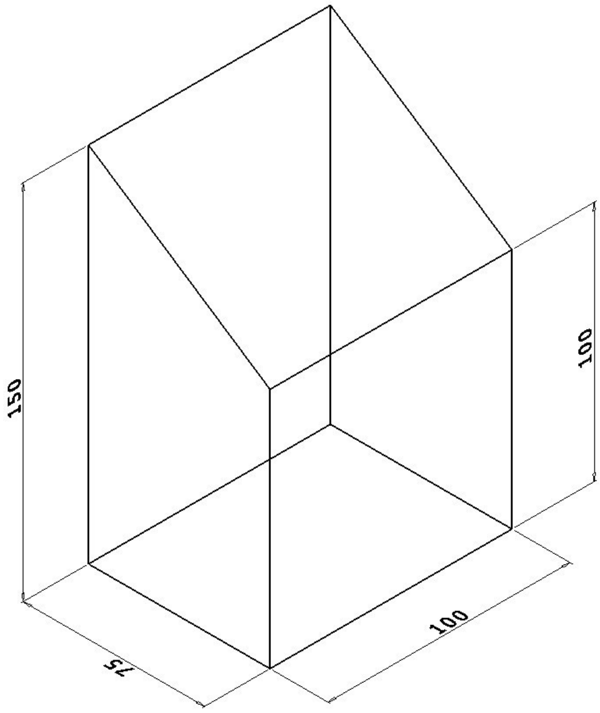
The screen should now look as shown below**.**



Using the same method, create a new set up on the **Front Face** at the **Upper Right Corner.**

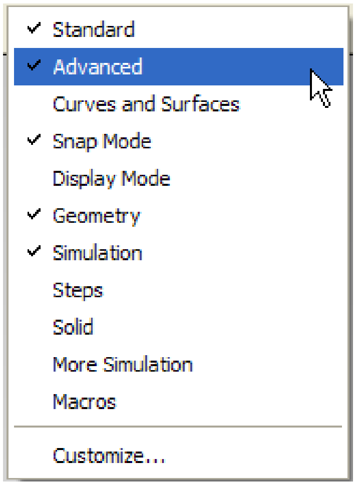
Creating Setups Using Geometry

It is also possible to create **Setups** which are aligned to existing **Geometry.** Draw the wireframe shown below, using the Geometry Step. 

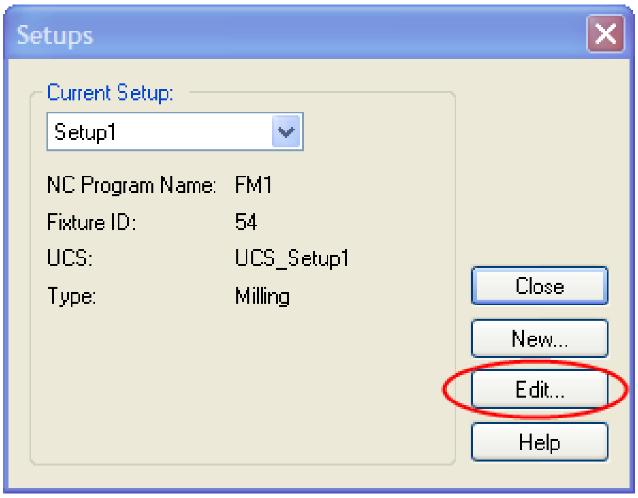


- Go to the **Setups** icon on the **Advanced Toolbar** at the top of the screen.

If it's not there, then the advanced toolbar needs to be switched on. To do this, right click on the grey area around the edge of the graphics area, and choose Advanced from the drop down list.



After selecting the **Setups** Icon, Select Setup1 from the drop down menu and click on **Edit**



This will open the **Setup Definitions** form

Here it is possible to change / set key details of the

Setup. To rename the setups so that they are easier to

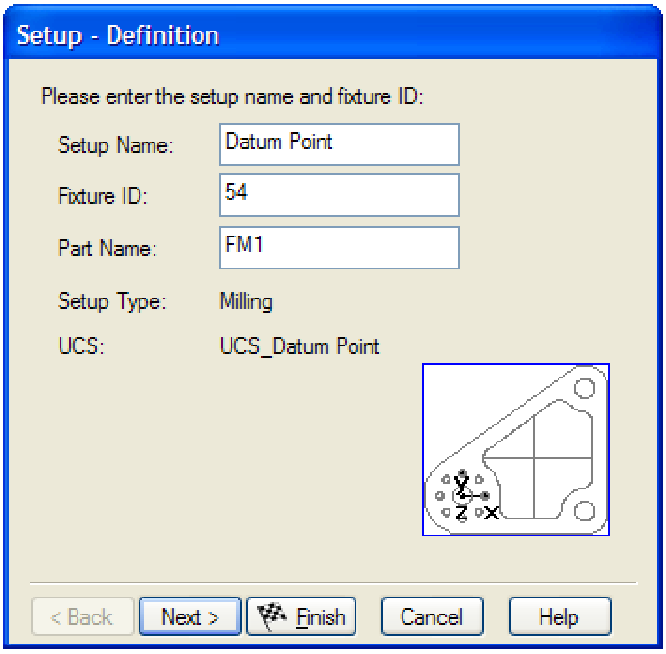
manage,

- Enter the name **Datum Point** into the **Setup Name** field. This name will appear in the **Part View Toolbox.**

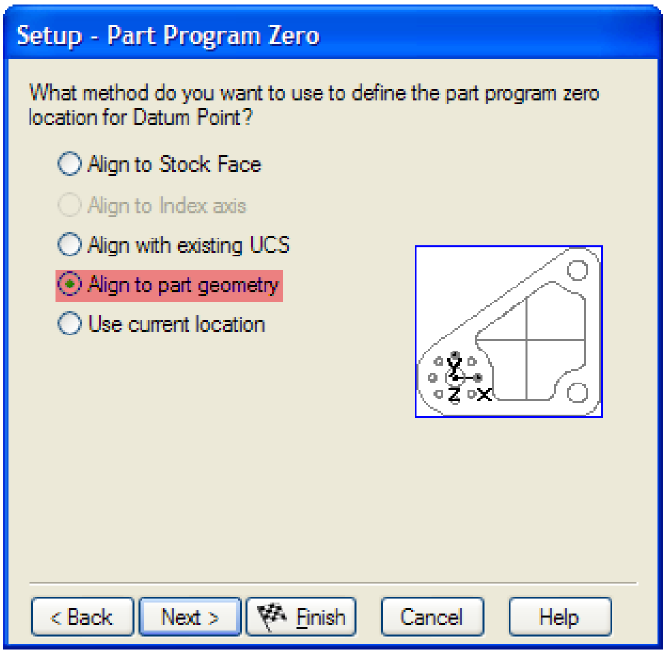
The **Fixture ID** field is where the user tells **FeatureCAM** which machine offset to use.

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.

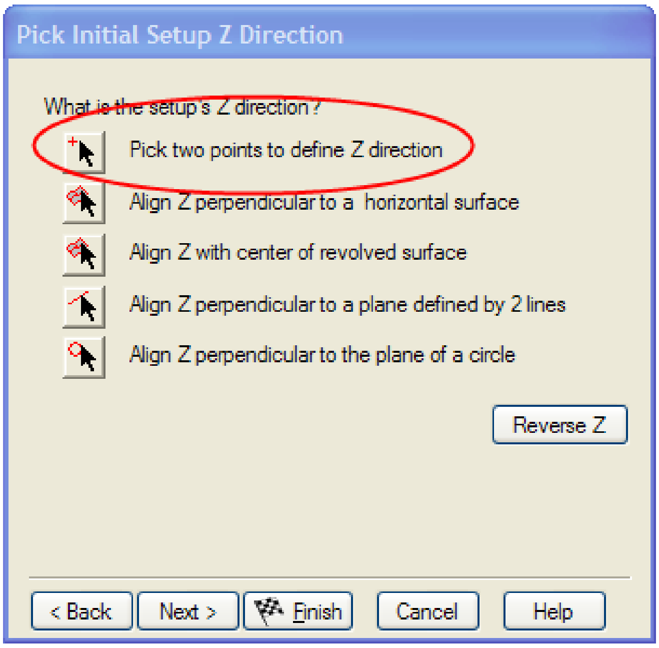


- Click **Next** and on the following page, select the **Align to Part Geometry** option.



By selecting the **Align to Part Geometry** option, FeatureCAM now knows that it has to provide the user with some tools to help them set up or align the Setup.

There are five options on the following page and depending on the geometry available; the user can choose one of these to align the Z axis .

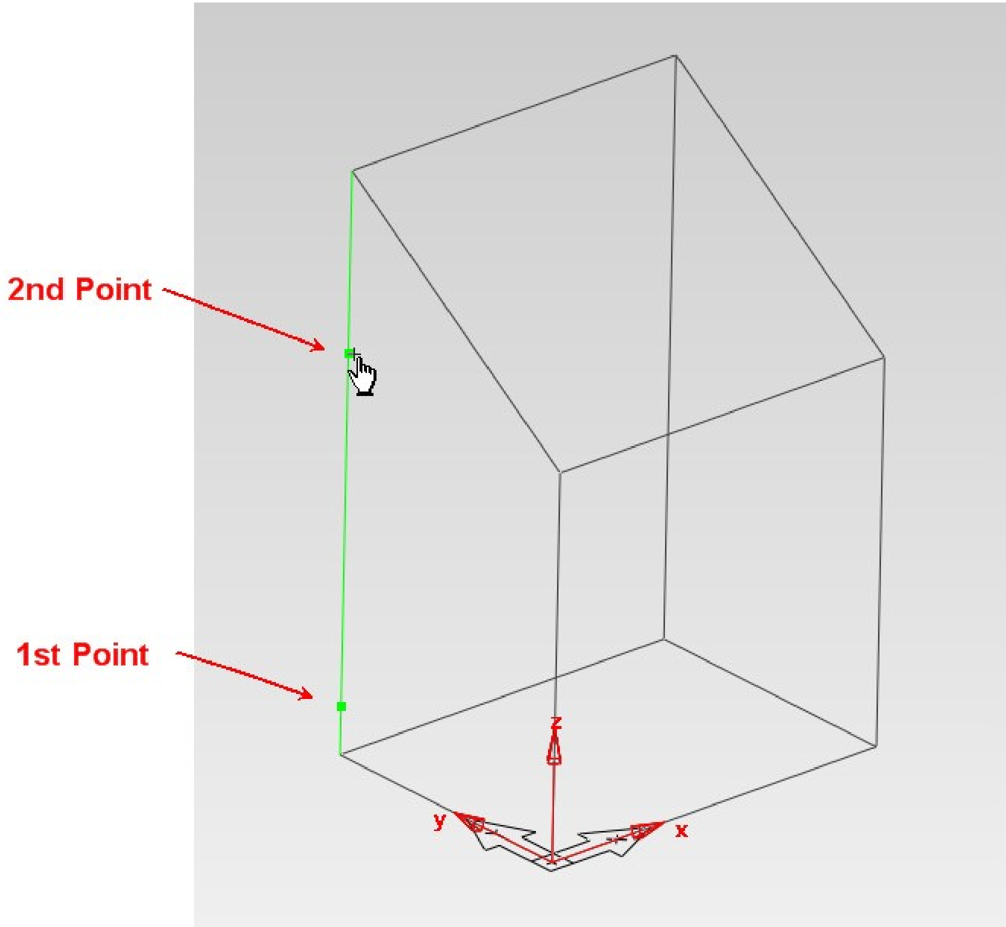


- Select **Pick two points to define initial Z direction**.

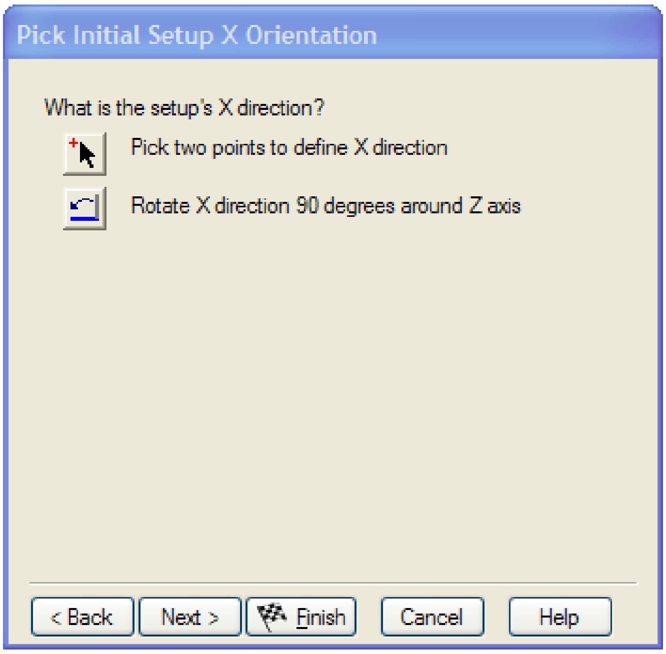
The form will shrink to one side of the screen to allow access to the geometry.

Two mouse clicks are used to define the Z Axis direction. To define the Z Axis,

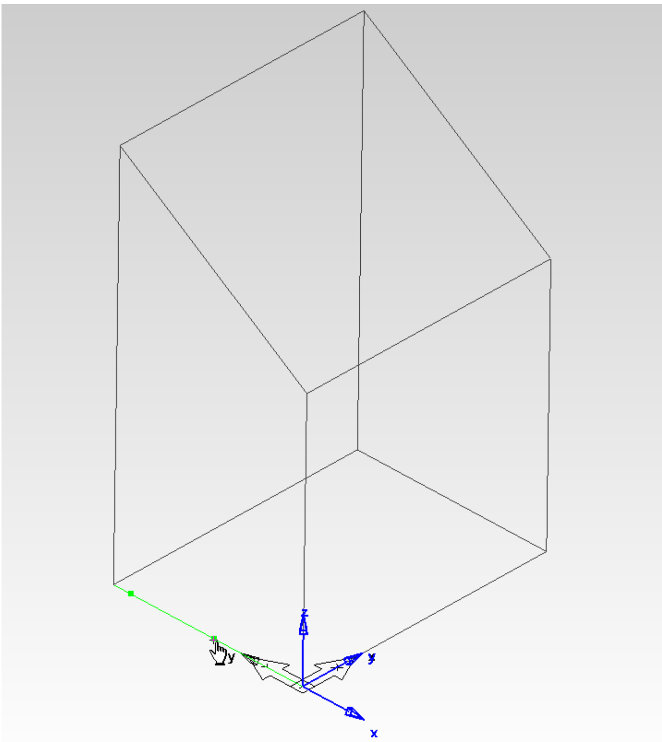
- Select the 1st point as shown, followed by the 2nd point. The position of the second click in relation to the first is what controls the direction of the axis.



- When the form returns, click **Next**



- Set the **X Axis** in exactly the same way as the B. Choose the left baseline as shown.



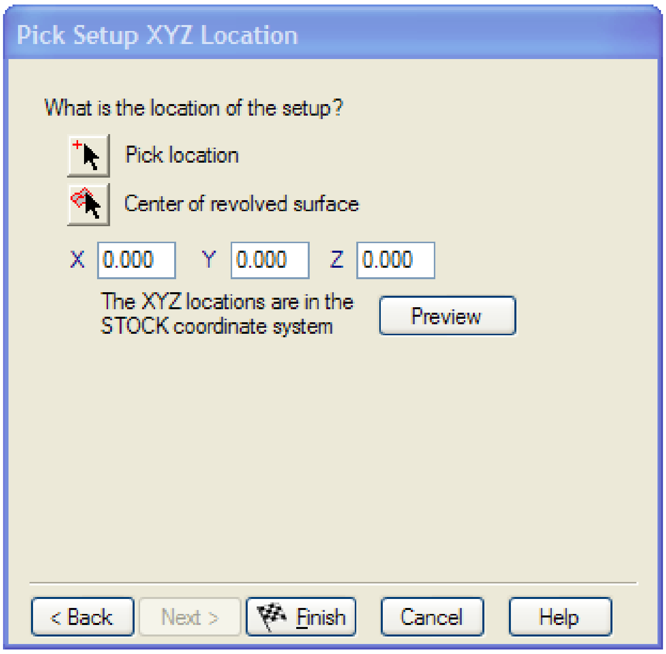
- This has aligned the **X** Axis along the short edge. Click **Next**.

- To reposition the **Setup.** Use one of the two pick buttons or enter the exact co- ordinates into the **X,Y,Z** fields on the form.

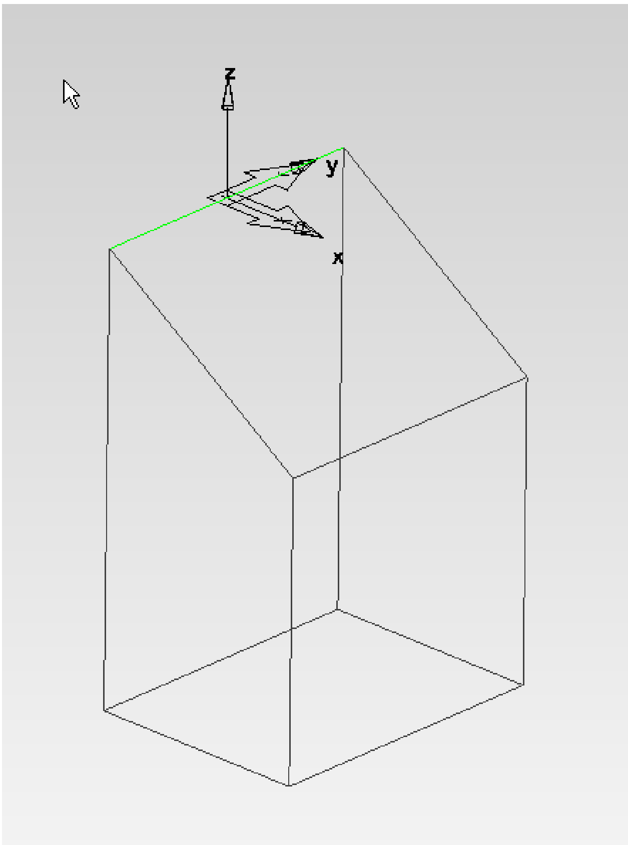
The two pick buttons are suited to different applications.

Pick Location allows the user to define positions, by picking points, directly from the screen.

Centre of revolved surface is used when there are surfaces that might represent a dowel hole for example



- Using any one of the methods, move the **Setup** until it is in the middle of the top line.

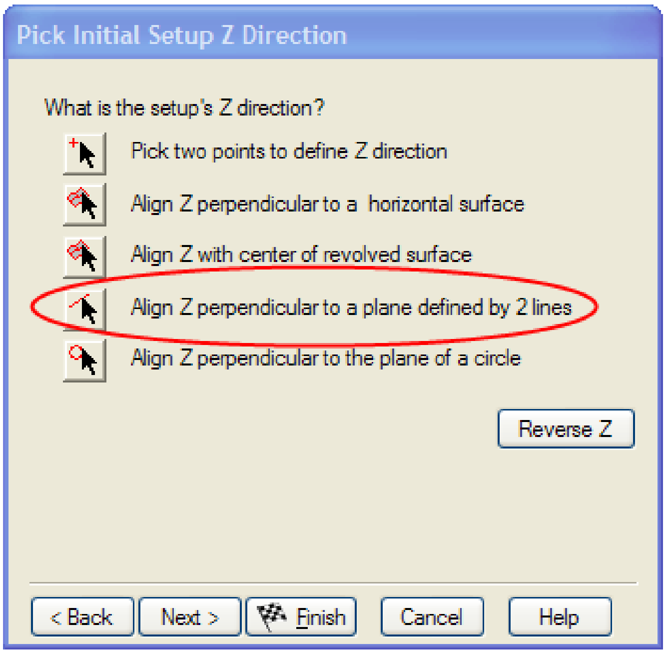


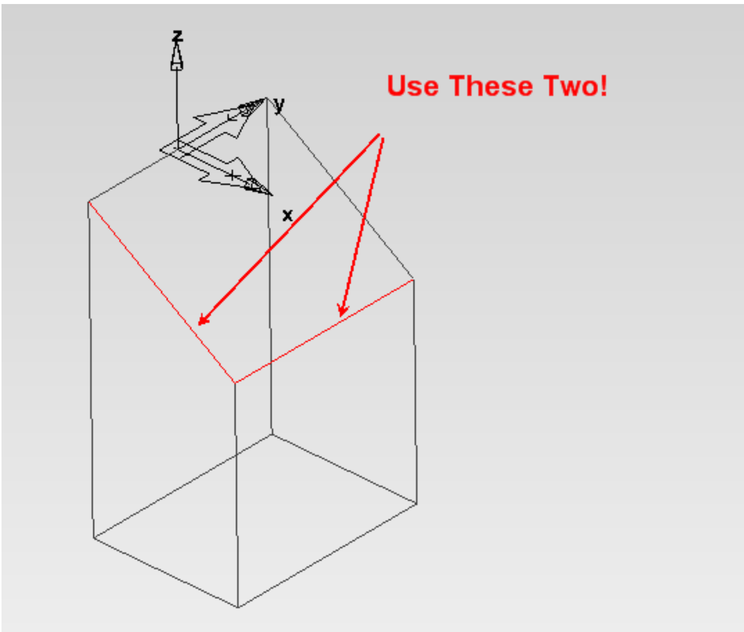
To create a new setup aligned to the angled face

- Create a new **Setup** by going to the **Advanced Toolbar** again

- Select **New** followed by **Next** and then **Align to part geometry.**

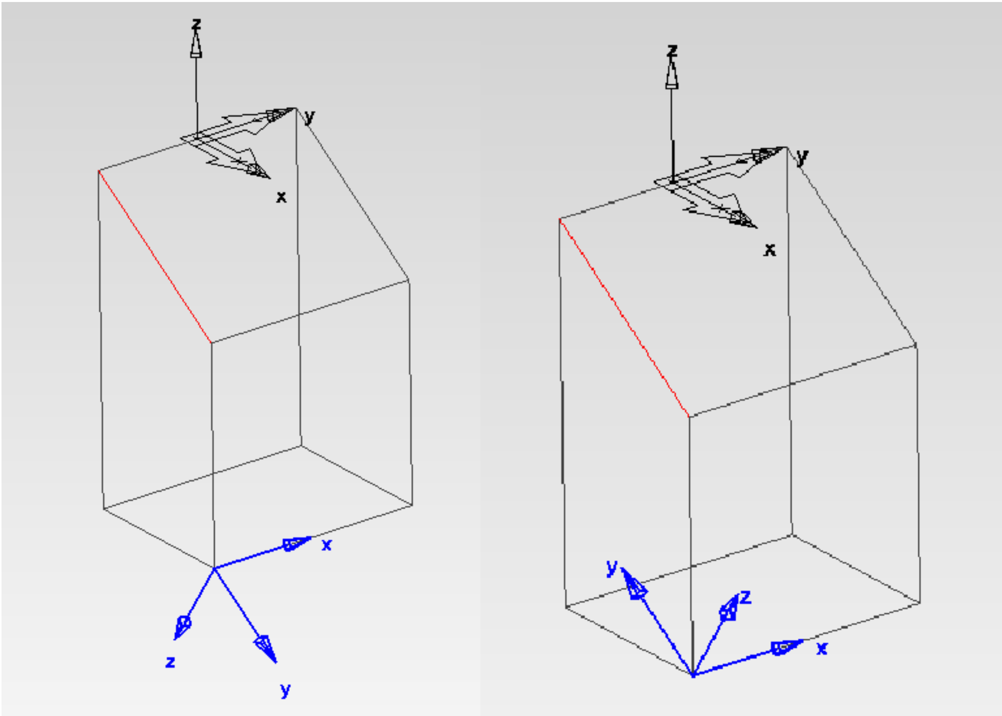
- This time on the **Initial Setup Z direction** form, select the option to **Align Z perpendicular to a plane defined by two lines** as shown.



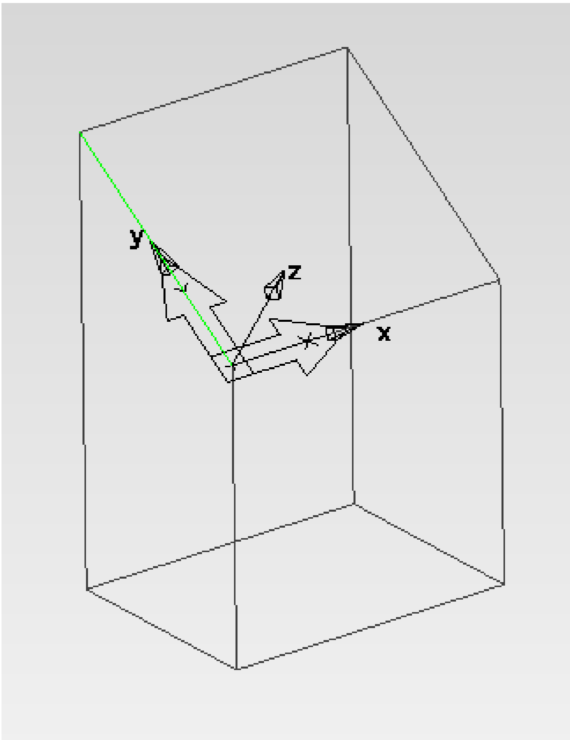


- Select two lines that are perpendicular to one another.

- If the **Z** axis is pointing the wrong way, then use the **Reverse Z** button to correct.



- Click **Next** twice and reposition the **setup** as shown below by using the **Pick Location** button.



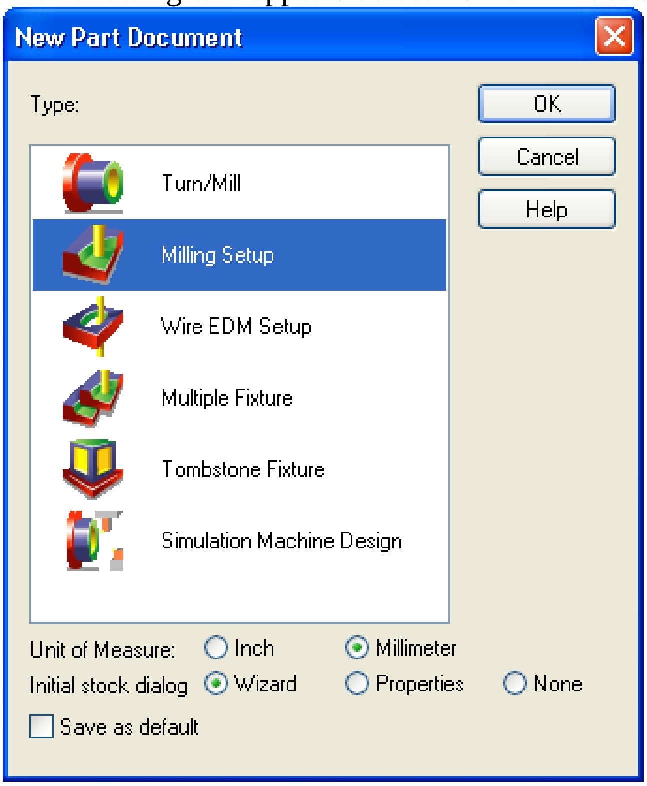
Creating Tool Libraries (Cribs)

Introduction

It is possible to create **Tool libraries or Cribs in FeatureCAM**. 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.

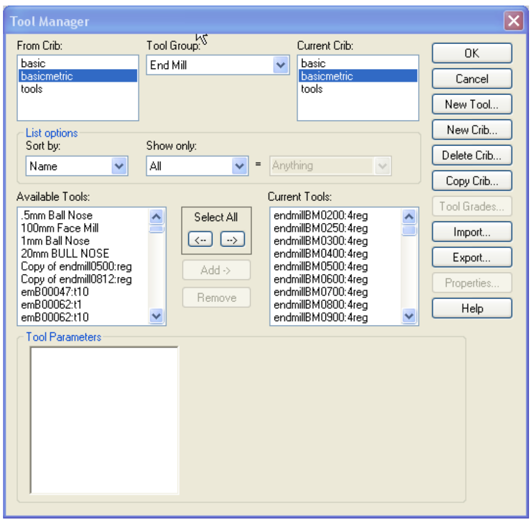
- Open a new **FeatureCAM** part by selecting **File, New**.

The following will appear. Select **Turn/Mill Set-up** and click **OK**

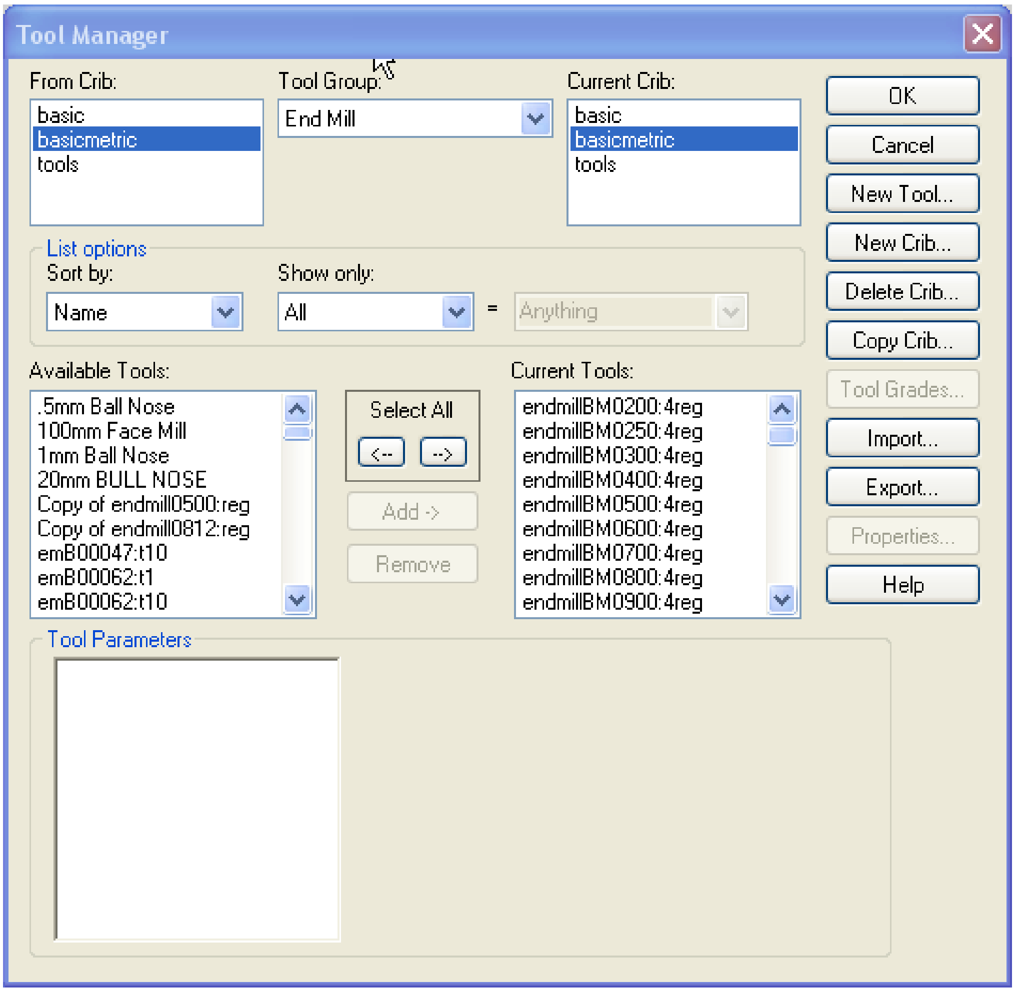


- Go up to the **Manufacturing Menu** and down to **Tool Manager.** This will open the following form.

- 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**

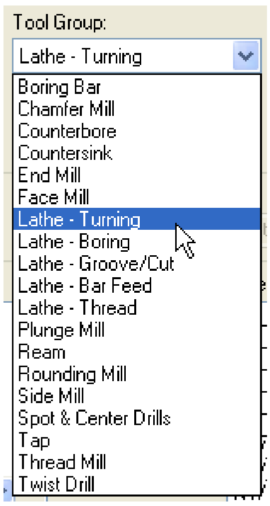


- The window is divided into separate areas and at first glance can look a little intimidating, but by applying some simple rules it becomes much easier to work with.

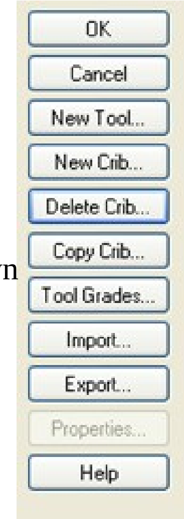


If the user divides the form in two down the middle of the form. The left side is used to select which Crib to copy from whereas the right hand side of the Form relates to the Crib that is current or the working Crib

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 on the tin.

New tool is used to create a new tool type**.**

New Crib will create a new crib, ready for the user to insert their own selection of tools**.**

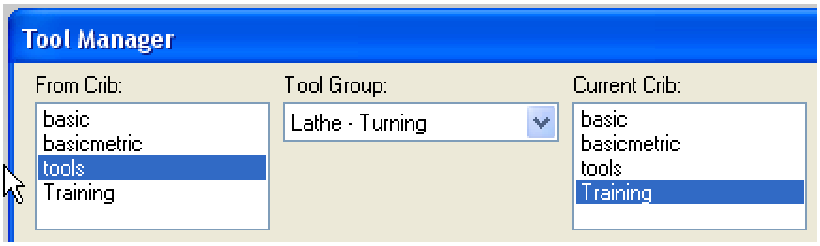
Delete Crib will delete an existing crib and **Copy Crib** will produce a copy of an existing Crib

Tool Grades is used to apply scaling to feeds/speeds to similar tool types that use different speeds/feeds

Import / Export allows the user, to either import or export tool cribs from one computer to another computer containing a copy of **FeatureCAM.**

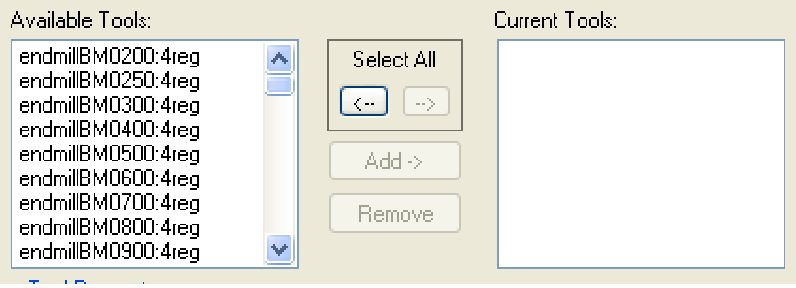
- By selecting the **New Tool Crib** option, the following form will appear. Enter the name as **Training,** and then click **OK**.

- The **New Tool Crib** will now be listed in the **Tool Manager** on the left hand side



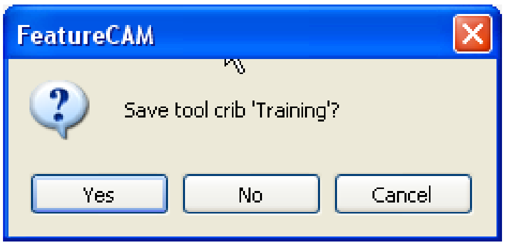
- The new **Crib** is empty so it is possible to copy tools from existing **Cribs**,

- Select the **Basicmetric Crib** in the **From Crib Menu** and the **Training Crib** from the **Current Crib Menu.** In the **Tool Group** select **End Mill**



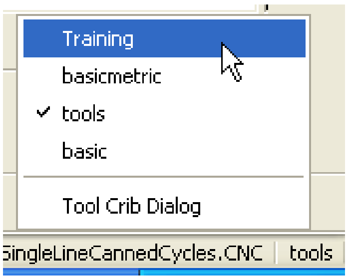
- Click on the **Select All** Button and then on the **Add** button to transfer the selected tools over to **Training Crib.**

- Repeat this procedure for the different tool groups and add some **4, 5, 8, 10 & 12mm Twist Drills** and some **Spotting** and **Centre Drills**.
- Click on the OK button and the following will appear.



- Select **Yes** and the **Crib** is complete for now. 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 left of the screen and click on the area indicated

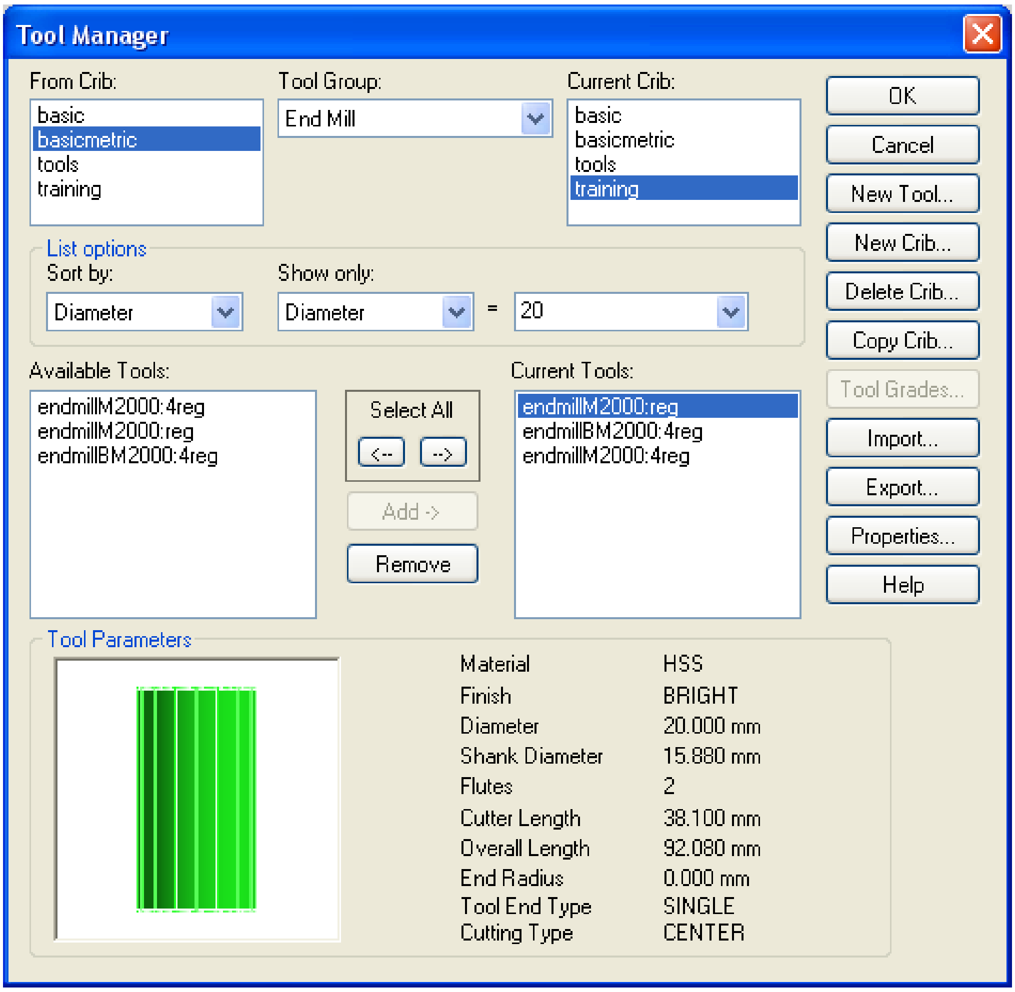
- Select the **Crib** required as shown.



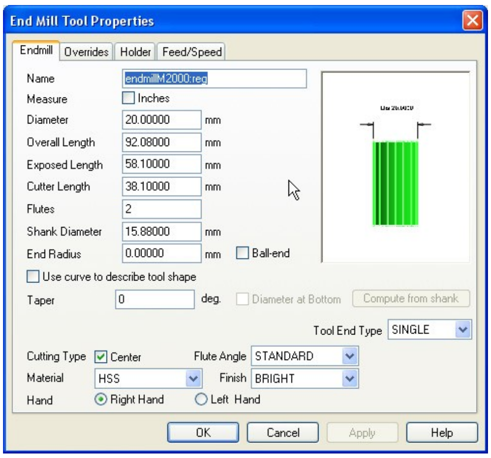
FeatureCam will now use the selected crib for any future operations
If the wrong tool is being selected, for example using a finishing tool to rough out, it could be that the tool needs to be set for a particular operation type.

Setting a Tools Operation type

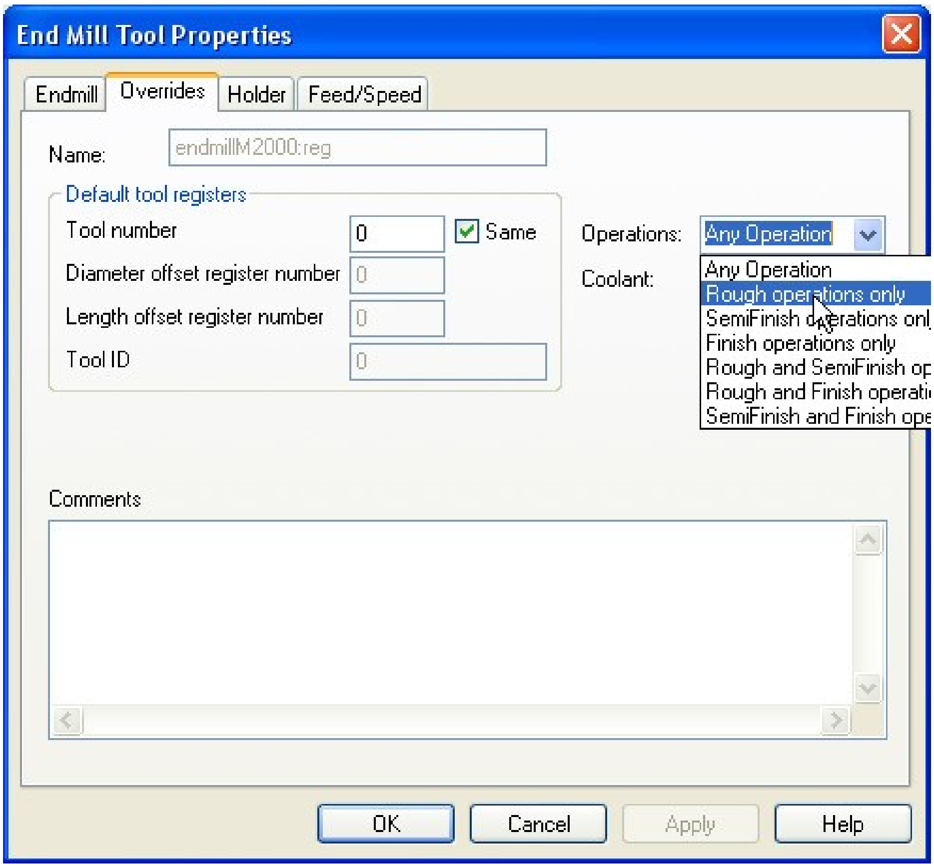
1. Open the **Tool Manager** as described above.
2. Double click the **Tool** that the user needs to edit as shown



3. This will open the



4. Click on the **Overrides** tab and then go to the **Operations** drop down menu and select the operation type as shown.



5. Check that the **Finishing tools** are set to **Finishing** by following the same procedure.
6. Click **OK** and then re-simulate the toolpath, the correct tool should now be used.

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

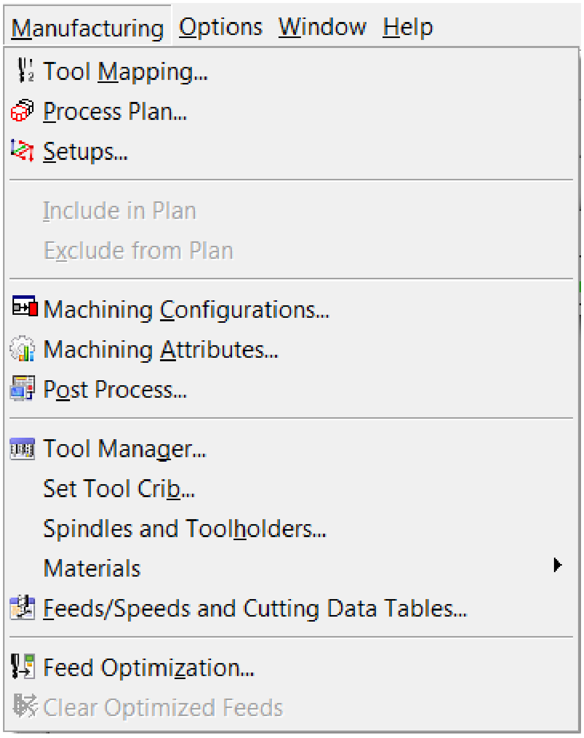
- The Machining Configurations can be accessed from the **Manufacturing** drop down menu.

- Machining Configurations are predetermined default functions that are selected, and used in all features that are created in the document.

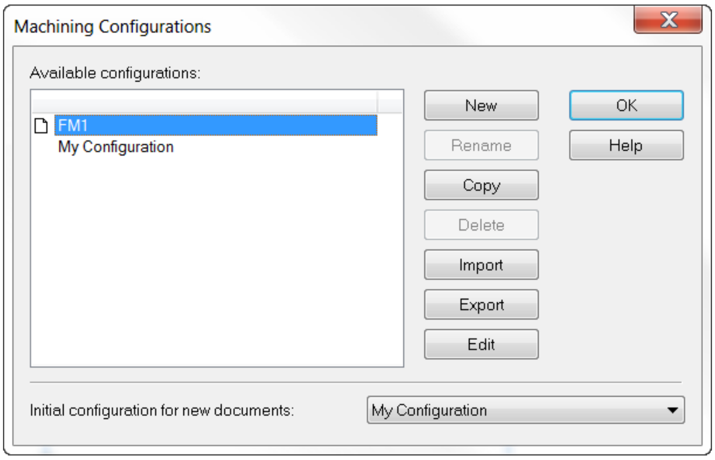
- A Machining Configuration contains approximately 150 settings.

- 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.



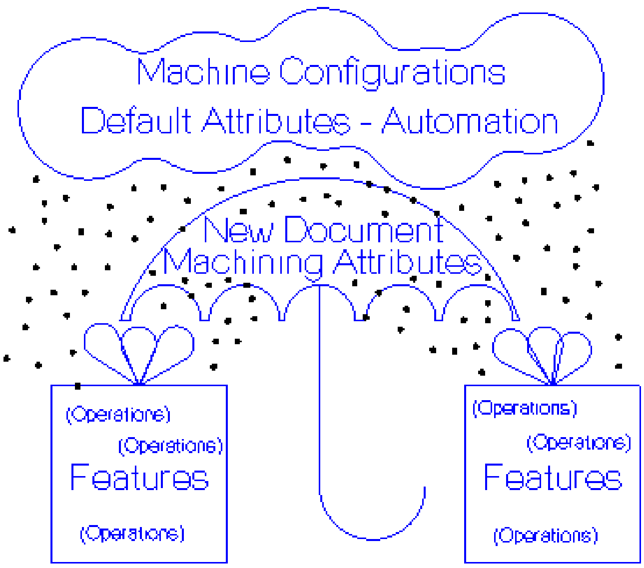
- **“Initial Configuration for new documents”**, is in the drop down menu at bottom.



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.

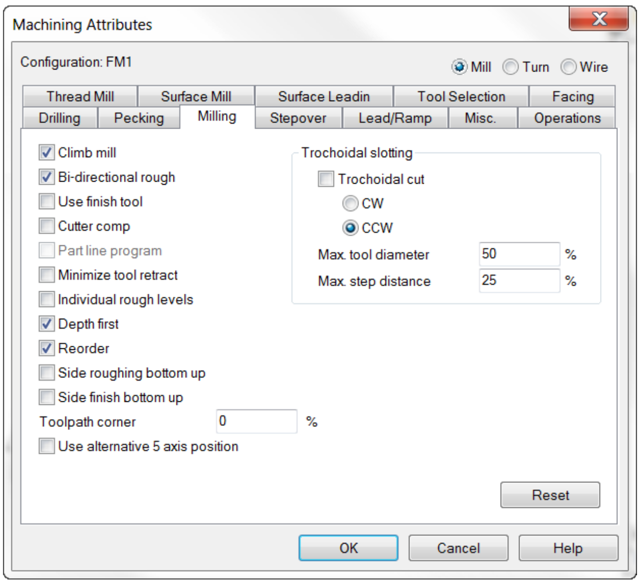
In other words, these changes do not become the defaults and when a new document is opened the old defaults are active. These settings stay with the document and are active anytime this same .fm document is opened.



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.

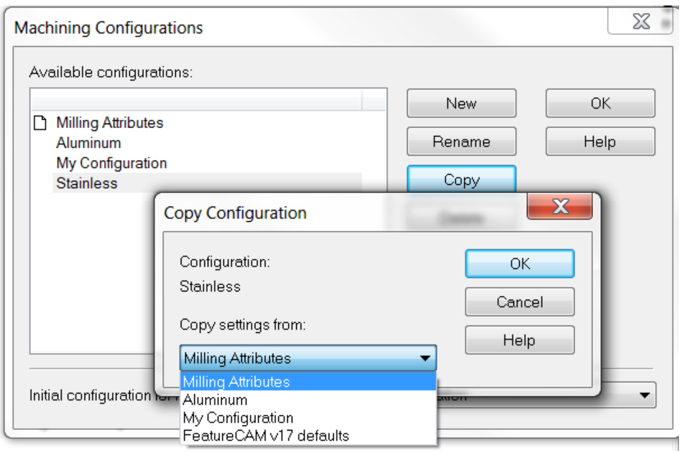
To apply the changes you have made, in the Attributes for the document, to become the defaults for any new documents they must also be changed in the Machining Configuration that opened with the file. Once set, and the next time a new document is opened, those changes will then take effect as the default settings.

- Open a **New Document** and click **Manufacturing** on the top menu and select **Machining Configurations.**

- 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.

- 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:



- Select the **desired Configuration** in the window. This is the configuration you want to **copy the attributes to.**

- Click **Copy** and **select the document** from drop down menu in the **Copy Configuration** dialog.

- **OK**.

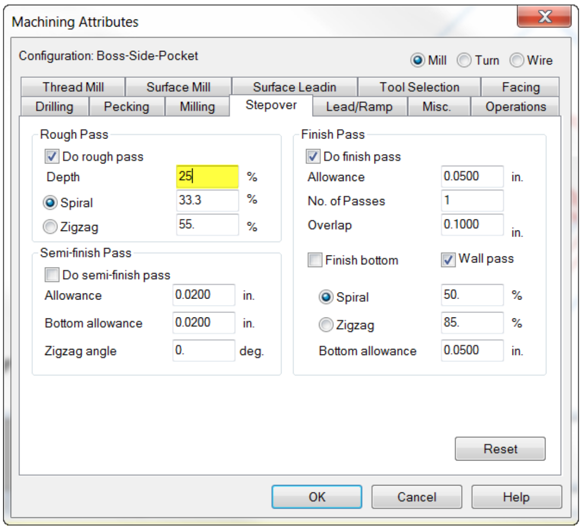
This reverses the copy and places the changes **from the document** Attributes **into the configuration selected** in the window of the **Machining Configurations** dialog.

Creating Machining Configurations

- Open a **New Milling Document**. Click **Manufacturing** on **Menu bar** then **Machining Configurations**.

- Click on **New** and type the word **Aluminum** in the field. **OK**

- **Repeat** these steps **to create a Stainless** configuration. **OK**.

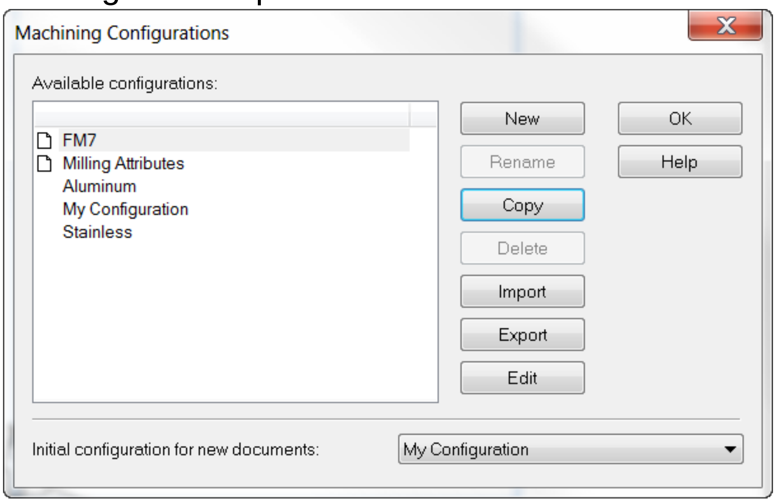


- Double click on **Stainless** in the window. Select the **Stepover Tab** in the **Machining Attributes** dialog. Change the **Rough Pass depth** to 25%.

The Rough pass depth for milling was 100% of the .500 diameter of the cutter. It is now 25%, so for a 1 inch endmill the rough pass depth is set to take cuts at a depth of .250.

Note: You now have three separate Machining Configurations. Each has approximately 150 settings.

- Select the **Document** you have open in the **Machining Configurations** dialog at the top of the window.



- **Copy** and select **Aluminum** from the **Copy Configuration** dialog.

- **OK** twice. These settings will be used in the new document.

Open an existing file named **Milling Attributes.fm**, from the **2.5D Advanced Data** folder, and run a **3D simulation**.

The .500 diameter milling cutter cuts to the bottom of the .500 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.

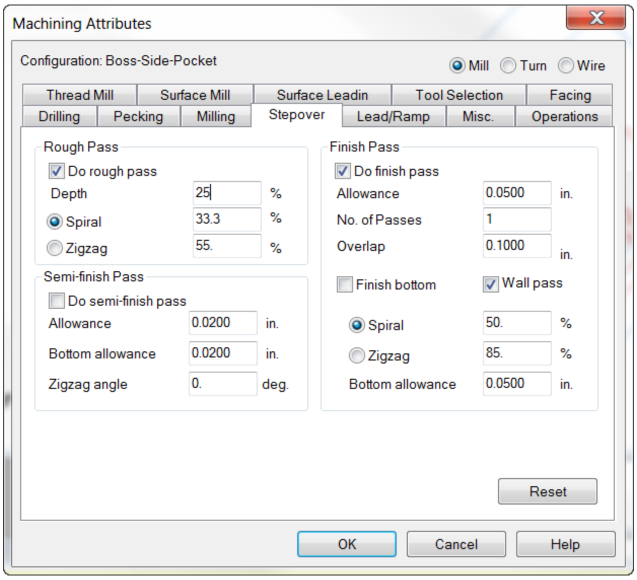
- Click **Manufacturing** then **Machining Configurations**. Select the document in the window on the left named **Milling Attributes.fm**.

- Press the **Copy** button. Select **Stainless** from the drop down list in the **Copy Configuration** dialog. Press **OK** twice.

Using the same feature and by only changing the Machining Configuration the Attributes automatically change to the Stainless settings.

- Run a **3D simulation**.

The setting for Stainless is 25% of the diameter of the cutter so now the tool makes **four passes** at .125 depth of cut **instead of one** at .500 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 counterdrill 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 spotdrill 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.

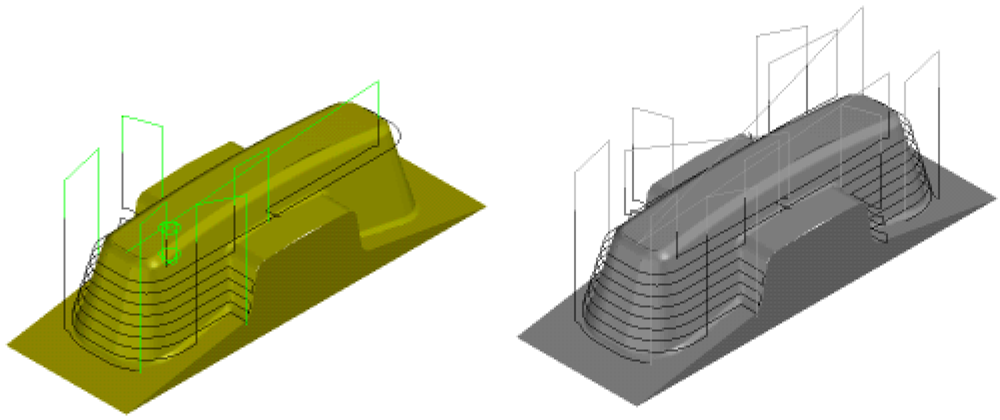
Minimize tool retract: For use when it is more desirable to feed the tool to another region of a feature instead of retracting and plunging. For example: in a pocket with islands, minimize tool retract will keep the tool from retracting and plunging.

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.

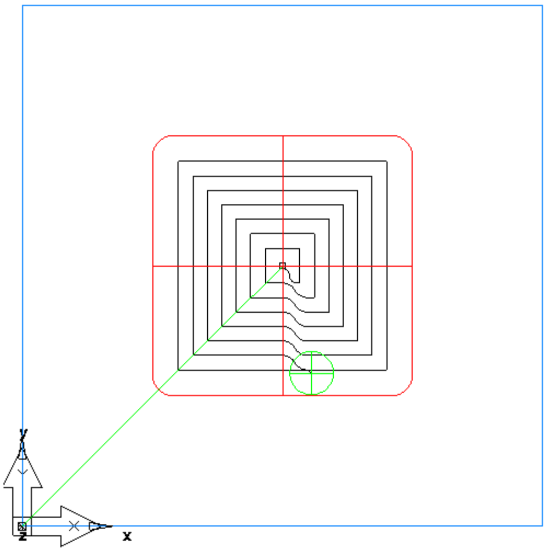


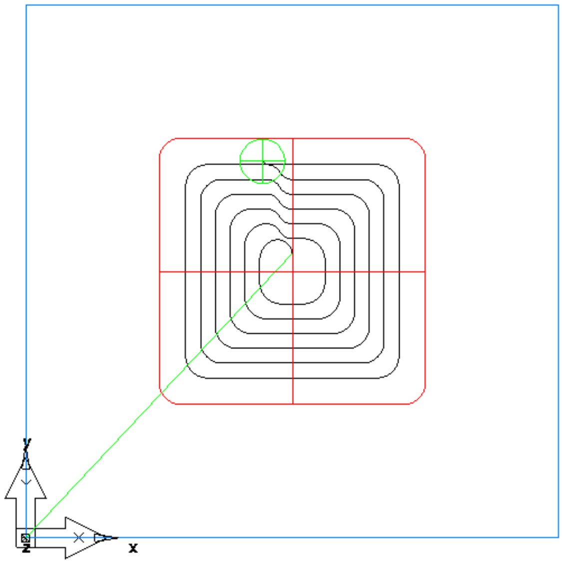
Side roughing bottom up: A 2.5D feature with a Draft Angle may be roughed from the bottom up instead of the normal top down.

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.

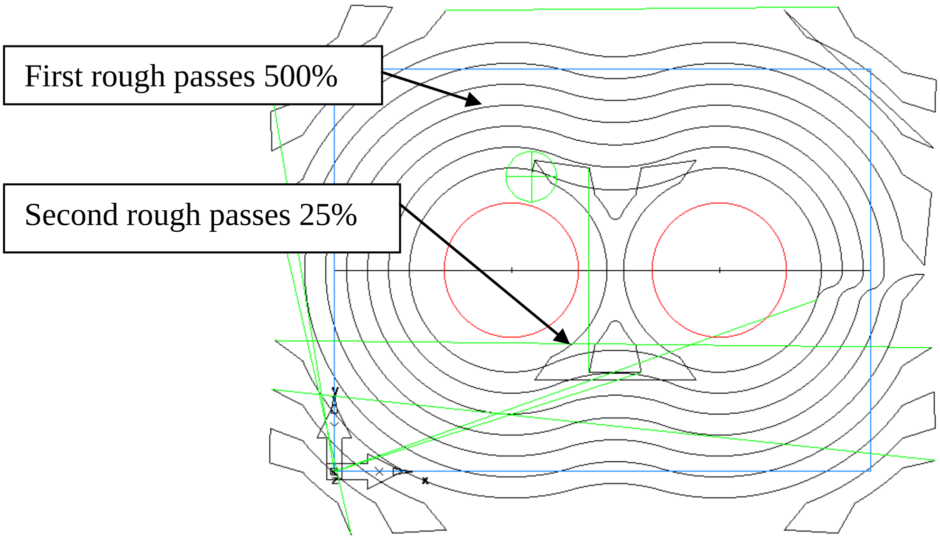
Toolpath corner (%): This attribute rounds the corners of milled roughing passes. It is specified as a percentage of the tool diameter. Rounding the sharp corners of the toolpaths provides a more constant tool velocity and reduces the tool load.

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%:**



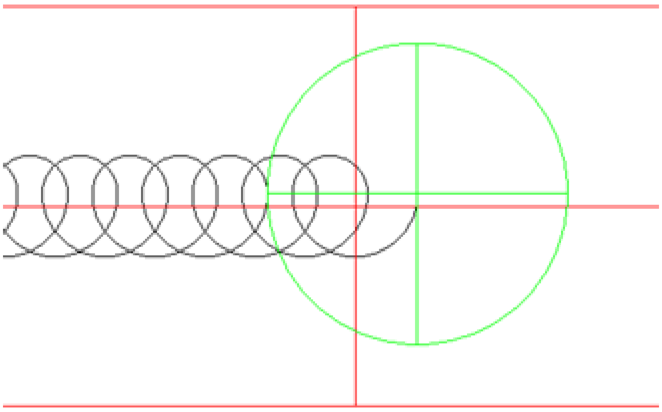


Hi speed machining technique. The first rough Tool path corner at 500% flattens the tool path, usually faster feeds and speeds. The second rough Tool path corner at 25%, cleans up only what the first rougher left. The 25% value helps smooth out some of the sharp corners, but must not be too large, or there might be an excessive amount for the finishing tool.



(Trochoidal slotting)

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.



Stepover 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 centerline 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 thd milling)

Ramp diameter is the radial size of the helical arc that is used. (thd. 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 4^{th} 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 4^{th} 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 machined. The smaller the setting the more retracting, the greater the number the 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 provides 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 4^{th} 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 stepovers 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)

Use counter bore will select a specific diameter counter bore tool. An inventory of various sizes of counter bores is usually maintained.

Use endmill will select a milling cutter to cut the counter bore. Eliminates an inventory of counter bore tools and can cut any size.

Automatic selects and 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 drill 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.

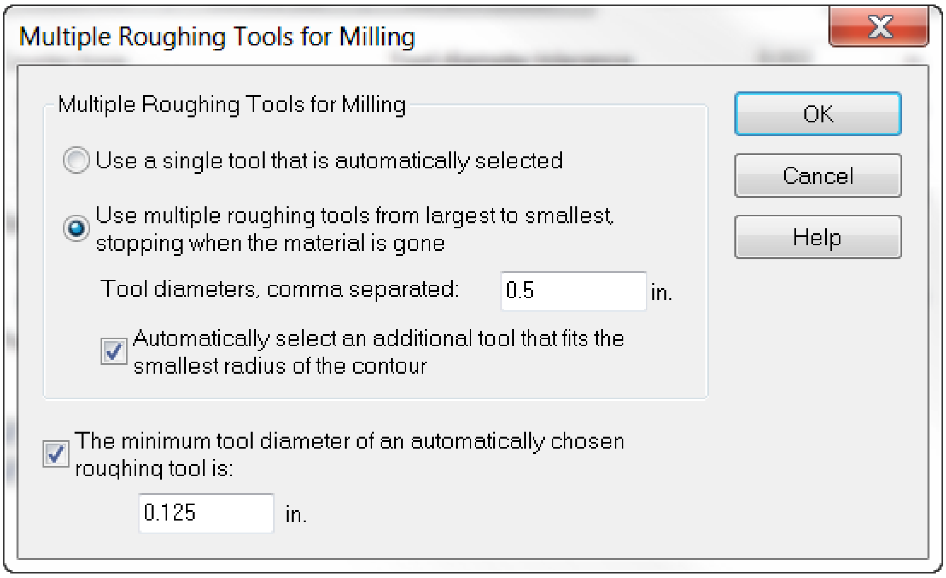
Thread % for tap drill (cut) determines the size of the drill to be used for the tapping operation to obtain a specific % of thread (example 77%)

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.

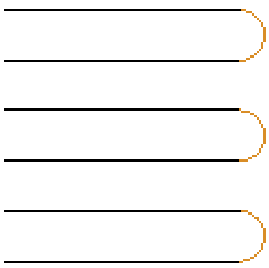


Surface Leadin

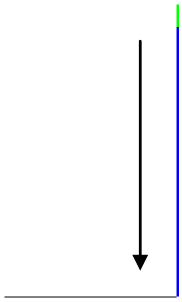
Stepover type is the transition move the tool makes radially to make the next pass (usually bi-directional) this is perpendicular to the tool path direction.

1) **Direct** is two 90 degree turns, stopping twice to change direction of 180 degrees.

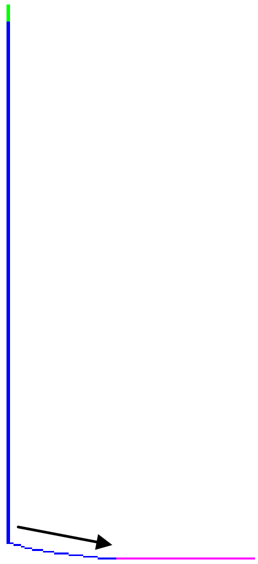
2) **Loop** is a smooth rounding transition which does not stop to change direction. Good hi-speed technique.



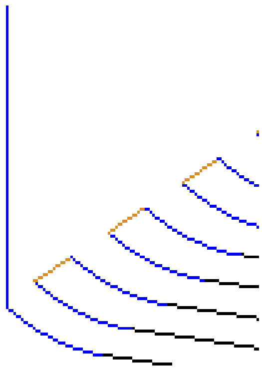
(Lead-in/out parameters)
Use lead-in/out:
Never provides no leads, it uses plunge clearance and plunges to depth.



On all plunges/retracts or **On first plunge/last retract** applies the lead type, line or arc, to **every** plunge and retract **or** the **first** and **last** only.



On all stepovers, 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.

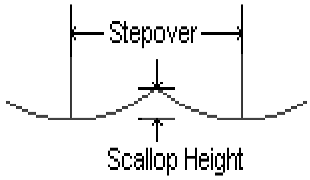
Use arc ramp-in/out allows you to choose the ramp diameter and the ramp in/out angle.

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 stepovers 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 neighboring 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 contains 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.

1) **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.

2) **Don't roll over the edge at all** cuts 100 percent cleanly to the edge but does not roll over the edge.

3) **Cut top edge: Just roll over the top edge** allows the ball only to roll over the edge.

4) **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 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.

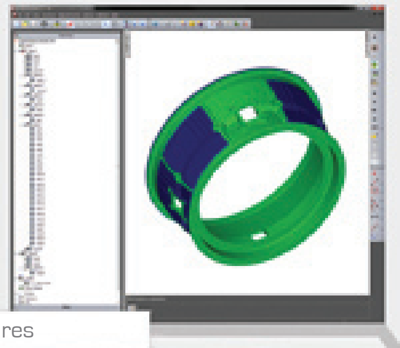
Use Finish Tool allows a separate tool to be used as a finisher.

Plunge to center plunges the tool on the center of the hole to assure clearance.

This concludes Machining Configuration/Attributes.

FeatureCAM 2012

FeatureCAM 2012



new features

Delcam TV

www.delcam.tv

FeatureCAM Learning Zone

www.delcam.tv/lz

FeatureCAM Website

www.featurecam.com

THE QUEEN'S AWARDS FOR ENTERPRISE: INNOVATION 2010

THE QUEEN'S AWARDS FOR ENTERPRISE: INTERNATIONAL TRADE 2011

Powering your productivity

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

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