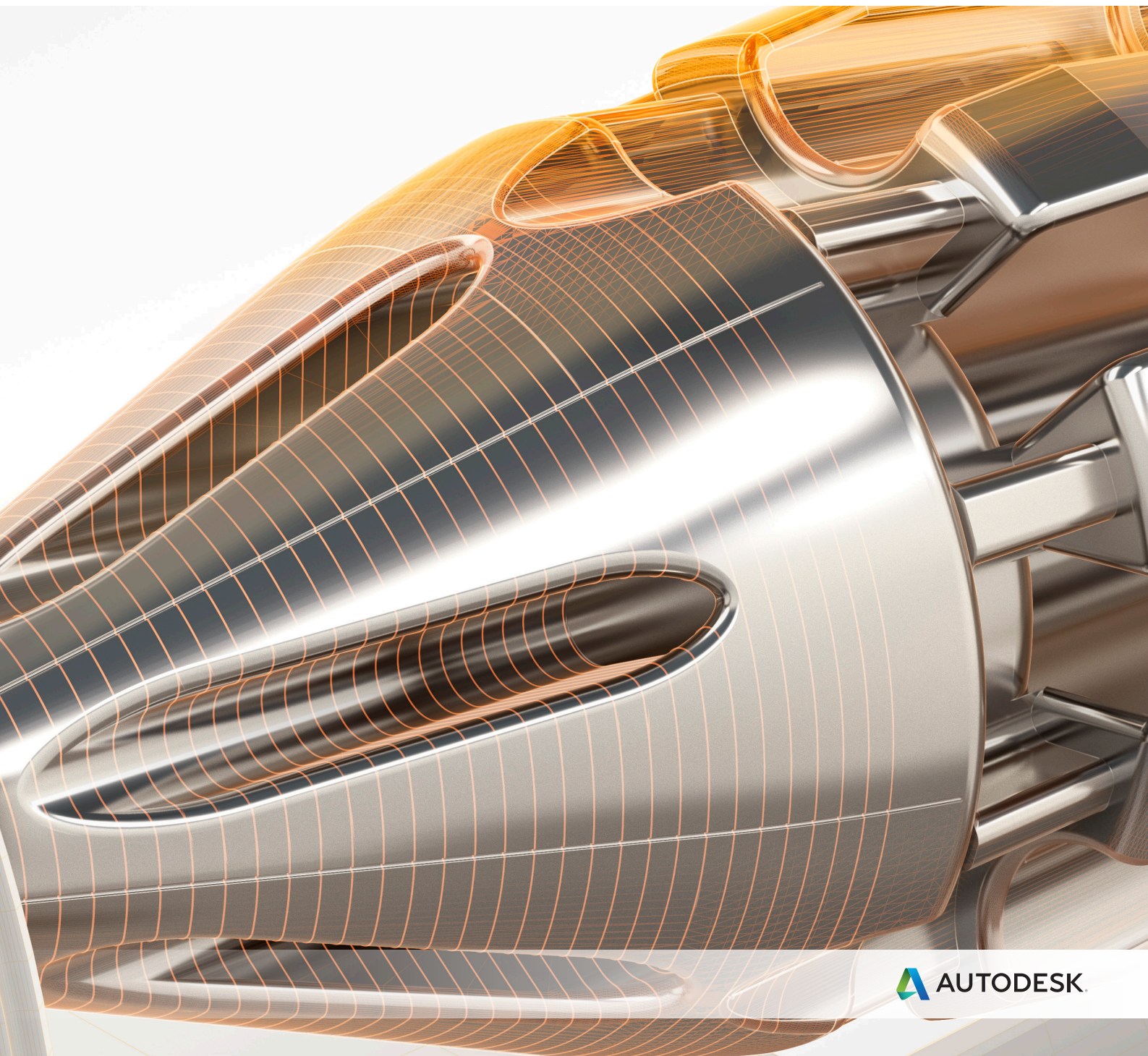




Automated to make parts faster

Training Course



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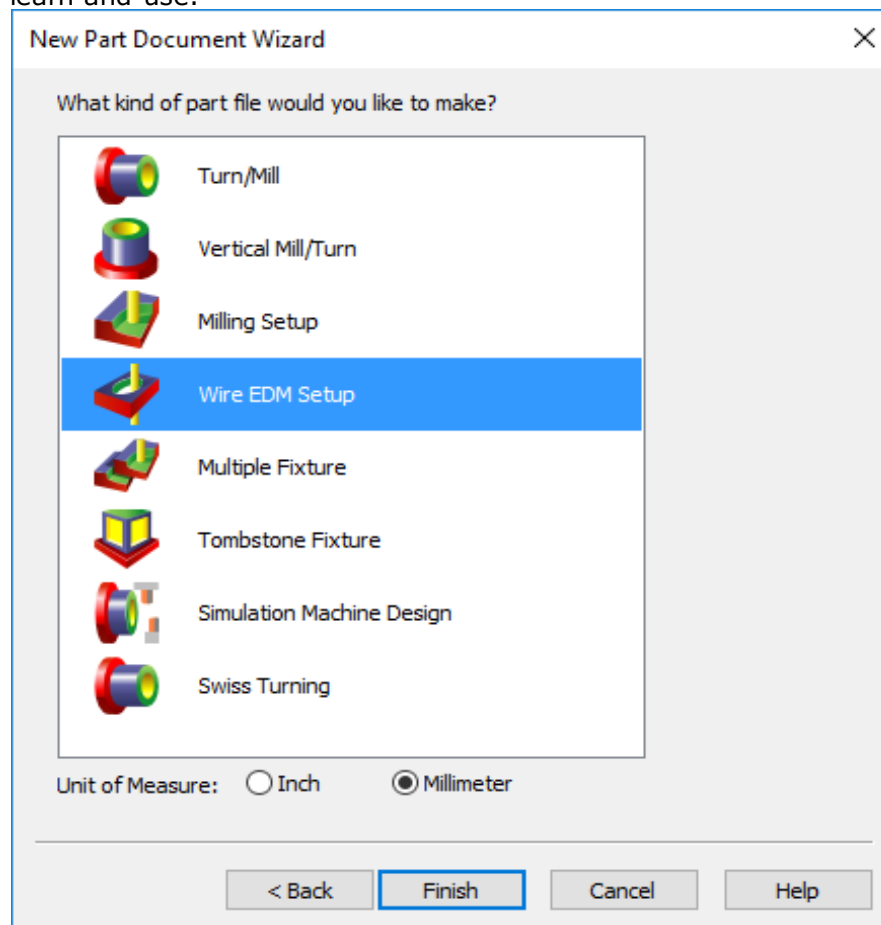
 AUTODESK® FEATURECAM® FeatureWIRE (EDM)

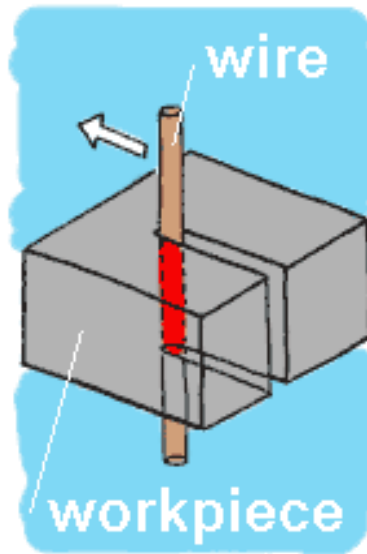
Introduction

This module will familiarize you with **Autodesk FeatureCAM's** history, describes what **FeatureCAM** is and show how it is advantageous over other **CAM** systems.

In this **FeatureWIRE** module the user will learn the basic user interface such as tool bars, menus and how to make certain functions become the default which can reduce programming time by using these predetermined settings. You will learn about drawing with geometry and using curves and the added benefit of using solid models and Feature Recognition and each of these will be used to create simple **2Axis** and more complicated **4Axis features**.

Ease of use is another guiding principal of **FeatureCAM** products. Our system was originally developed as a graphical Windows-based program. **FeatureCAM** programs are full of graphical feedback, **Step-by-Step Wizards** and tutorial-style animations that run directly in the program dialog boxes. Ease of use has always been a **FeatureCAM** advantage and we continue to study ways to make our system even easier to learn and use.





Electrical Discharge Machining



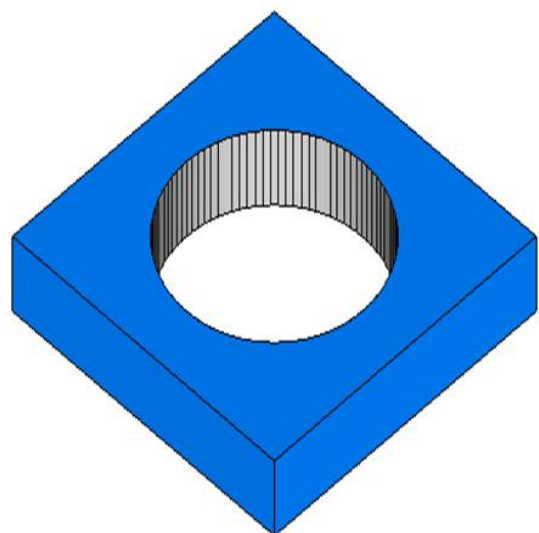
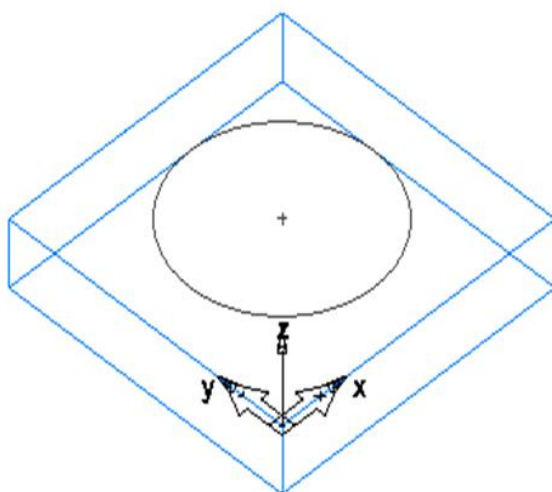
The process of wire erosion enables you to cut difficult to machine conductive materials with tremendous accuracy. Parts can be from less than 1mm up to 450mm in thickness and very small from a few grams up to 400kg in weight. Wire erosion can cut tungsten carbide, exotic steel alloys, titanium and aluminium – in fact as long as it conducts electricity it can be cut with Wire EDM.



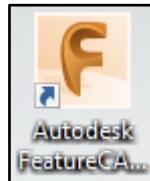
With an electrically charged wire machine (**EDM**). A spark jumps from the wire to the work piece and the material is locally vaporized, the wire never contacts the part. The wire diameter is typically **0.025mm**



Simply draw your shape and create the feature.



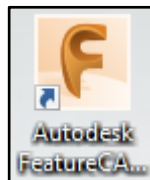
Starting FeatureCAM



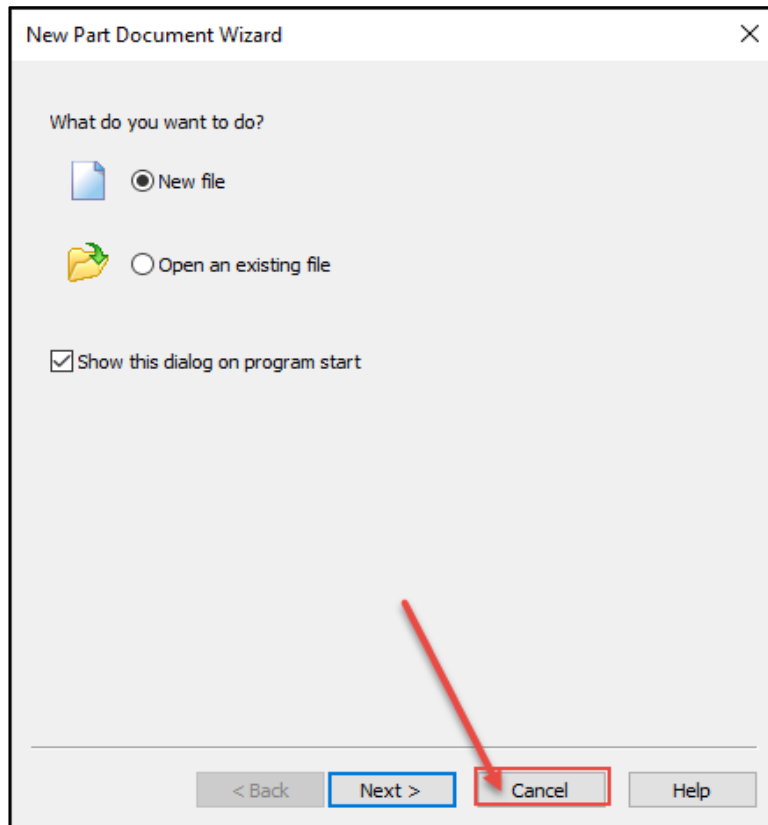
- Select the FeatureCAM icon from your desktop and select **New File** and **Wire EDM Setup** and **Millimetre**. Select **Finish** twice and then **Ok** to close the Stock menus. The stock menu will be covered later on in the training notes.



Evaluation Options



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



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

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



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



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

Evaluation Options ✕

Select the FeatureCAM product components you would like to activate. OK

Cancel

Product component

Standard
 Premium
 Ultimate

2.5D Milling

2.5D and 3D HSM
2.5D and 3D MX

2.5D and 3D Lite

Turning

Turn/Mill

FeatureRECOGNITION

Tombstone Machining
5 Axis Positioning
Catia Import
Solid Modeling

Wire EDM

Step Import
Catia V5 Import
ProE Import
NX Import
Machine Simulation
Advanced Turn/Mill (MTT)

Network Database

5 Axis Simultaneous
Probing
Vericut verification
NCSIMUL verification
CAMplete verification
Swiss Turning

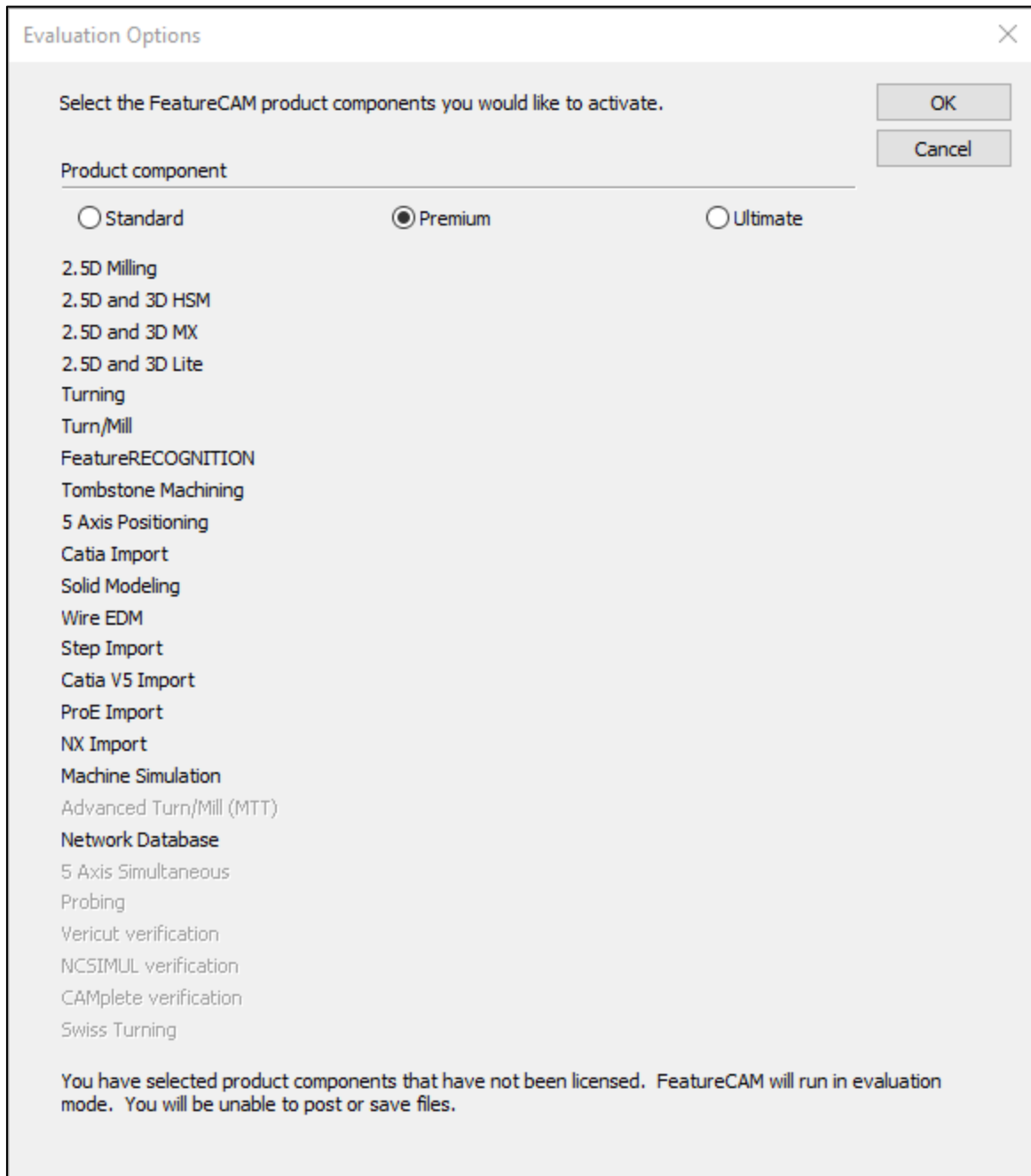
You have selected product components that have not been licensed. FeatureCAM will run in evaluation mode. You will be unable to post or save files.



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



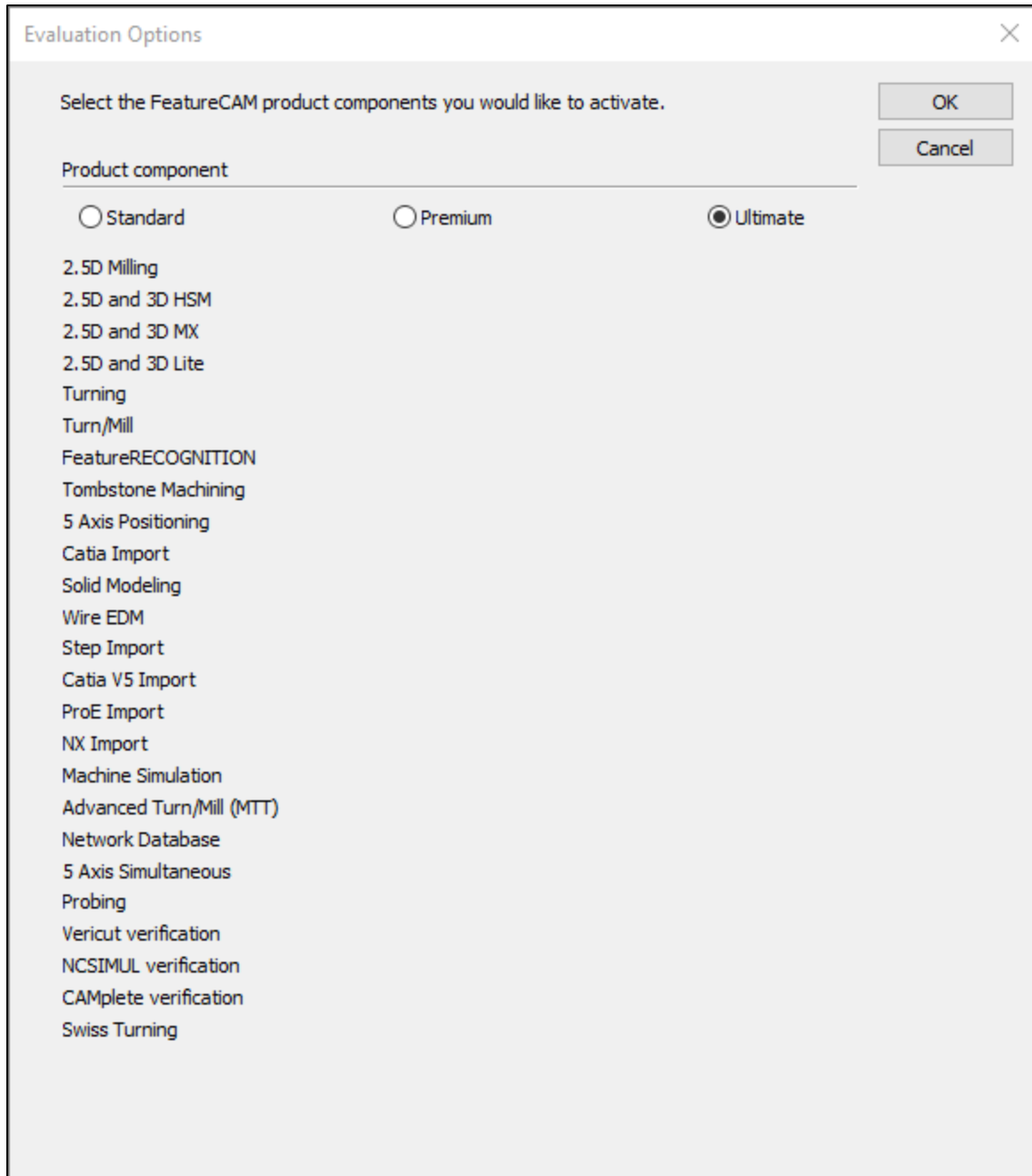
The image below show the available options in **Premium FeatureCAM**.



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



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



Evaluation Options

Select the FeatureCAM product components you would like to activate.

Product component

Standard Premium Ultimate

2.5D Milling
2.5D and 3D HSM
2.5D and 3D MX
2.5D and 3D Lite
Turning
Turn/Mill
FeatureRECOGNITION
Tombstone Machining
5 Axis Positioning
Catia Import
Solid Modeling
Wire EDM
Step Import
Catia V5 Import
ProE Import
NX Import
Machine Simulation
Advanced Turn/Mill (MTT)
Network Database
5 Axis Simultaneous
Probing
Vericut verification
NCSIMUL verification
CAMplete verification
Swiss Turning

OK
Cancel



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

Desktop Subscription



This is what you get when you subscribe:-

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

Access options



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

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

Flexible term lengths

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

Access to the latest software



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

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

More about updates and releases

- Software when you need it, where you need it.

Use your software in more places

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

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

Autodesk support

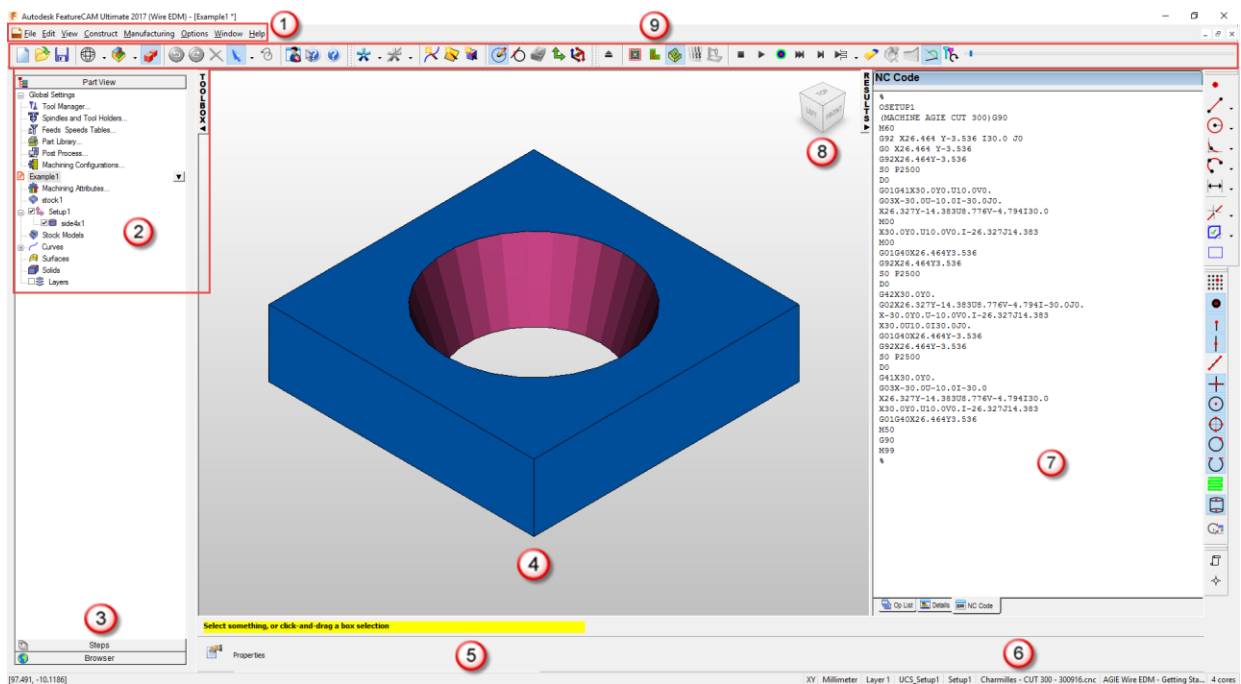


Let us help you to resolve issues quickly. You will have access to:

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

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.



1 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.

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

- 3 **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.
- 4 **Graphics Window** Located in the center. This is where most of the activity 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 wire.
- 5 **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.
- 6 **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, and how many CPU cores are being utilized.
- 7 **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.
- 8 **View Cube**

View Cube



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



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


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


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


Manipulating orthogonal views

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

Click:

 To show the view of an adjacent face.

 To rotate the view clockwise through 90 degrees.

 To rotate the view counter-clockwise through 90 degrees.

Configuring the ViewCube


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

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

Setting ViewCube options

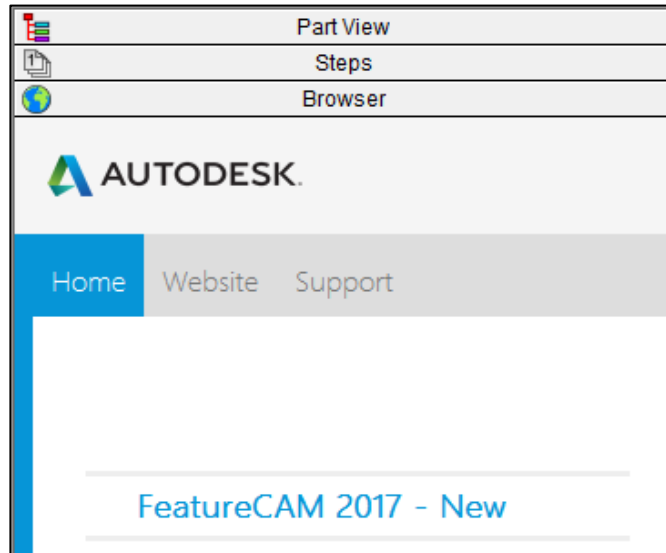
To modify the behaviour and appearance of the ViewCube, set these options on the **ViewCube** tab of the **Viewing Options** dialog:

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

 **9 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.

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.



Steps

- 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:

Specify type of stock and stock dimensions,

Draw Geometry

Create Curves from Geometry

Create Surfaces (ignored since creating a solid model)

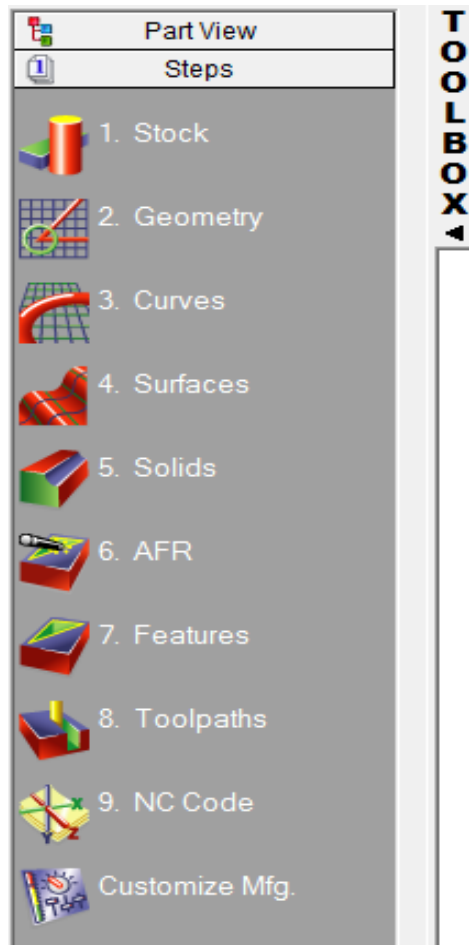
Create Solid from Curves.

No AFR for Wire Features

To view toolpaths and

To generate NC Code

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



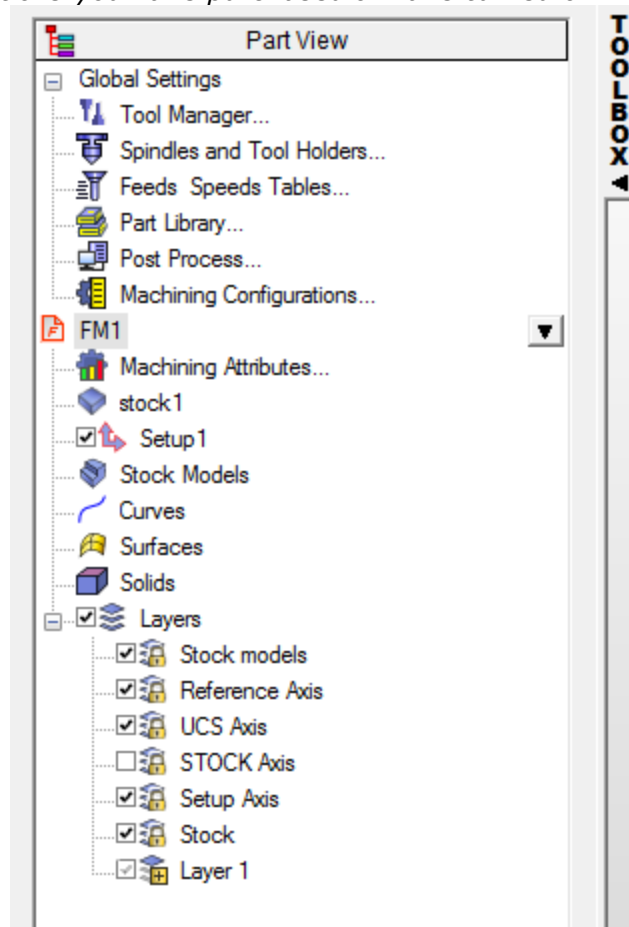
Note: We recommend that if you are just starting out with the program or demonstrating it to others that you use the steps toolbox instead of toolbar buttons in other areas of the interface to keep the interface simple.

- **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 – Not available for Wire. AFR** (Automatic Feature Recognition)
- **Step7 – Features.** Features are common shop terms like pocket, or side. 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.



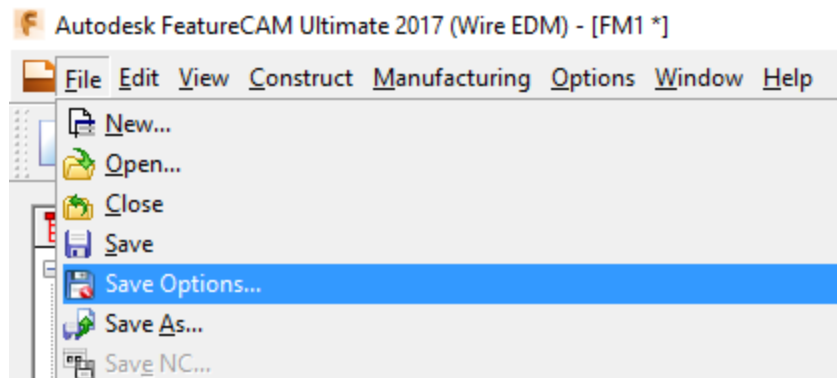
- Above the Steps Toolbar is the **Part View Tool Box**. This contains the names of the objects that are created on the screen. Objects that can be contained in this area are Stock, Setups, Curves, Surfaces, and Solids etc.
- Clicking on the objects in the part view enable different functions. Left-click 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.

Saving your work

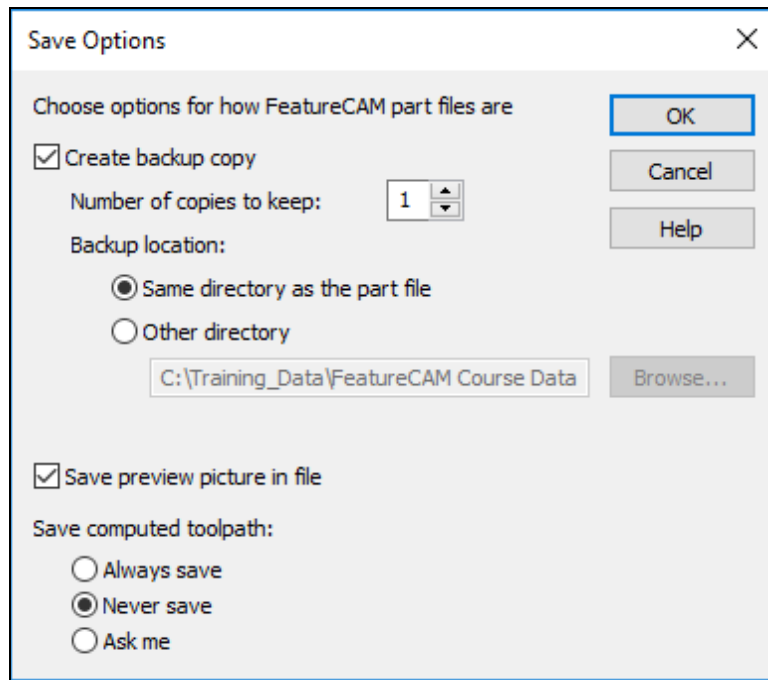
- **Question?** What is the difference between **Save** and **Save as**?
- **Answer:** When you are working on a document, it is a good idea to save your work every couple of minutes. In the **File** menu of the program, you will typically find the options "**Save**" and "**Save As**"
- Choosing "**Save**" simply saves your work by updating the last saved version of the file to match the current version you see on your screen.
- Choosing "**Save As**" brings up a prompt to save your work as a file with a different name. For example, you might choose to save a document called "**FeatureCAM_opp1**". This way, you can save your file at different stages and keep multiple versions on your hard drive.
- Choosing "**Save**" and then "**Save As**" is also a good way to make a backup of your file. Just be sure to rename the file something new when you choose "**Save As**" or you will overwrite the current saved version, just like choosing the "**Save**" command would do.

Save Options

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



- See **Save Options** on the next page. Adjust your computer to the same as the options shown.



- **Create backup copy:** - Select this option to save a number of previous versions of your part as you work. Enter the **Number of copies to keep** and set the **Backup location**. When you save a file, the previous version(s) are saved to disk using the name, but prefixed with **Backup of**. The latest version of the file is always saved using the name of your FeatureCAM part.
- **Save preview picture in file:** - Select this option to store an image of the part in the file. This image is displayed in the **Open** dialog.
- **Save computed toolpath:** - For certain parts, generating toolpaths can be time-consuming and you may want to save them for the next time you open a part. The setting of save computed toolpath controls the default behaviour for saving toolpaths. Select from **Always save**, **never save**, or **ask me** to be prompted each time you save a part. The toolpath is saved as an **.fmp** file.



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



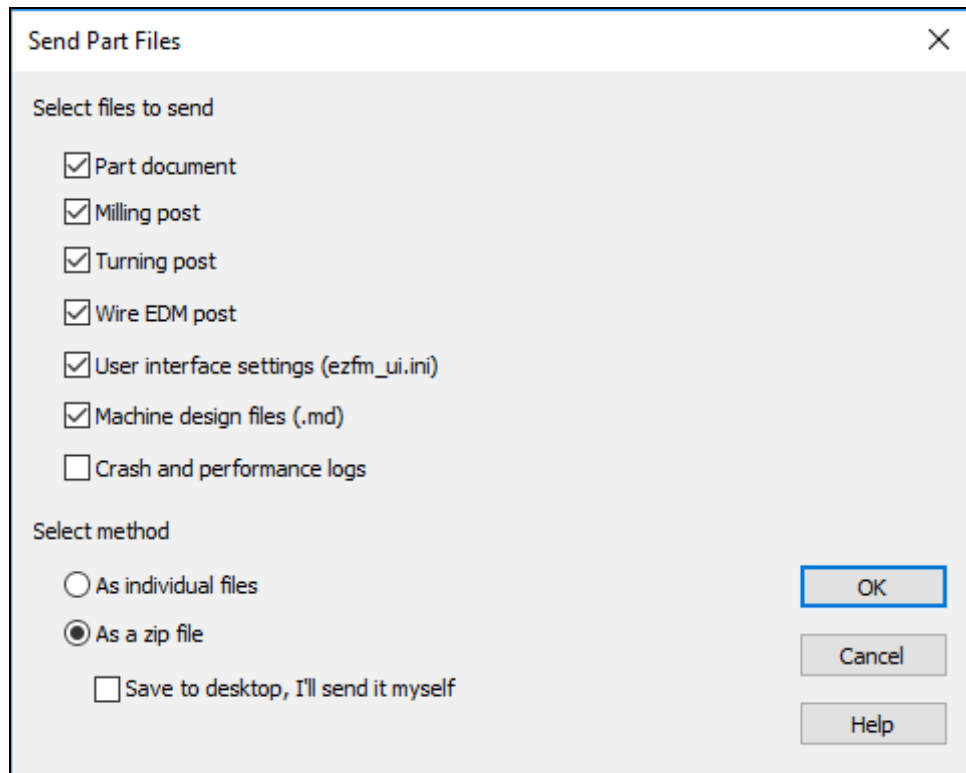
*If you upgrade to a newer version of **FeatureCAM**, the toolpath is disregarded and recomputed.*

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

Send Part Files dialog



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



Select the files you want to send from:

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

Size confirmation

- If you are sending **.zip files**, a Size Confirmation warning dialog is displayed:
- The average email server limits attachments to **10 MB**.
- The total size of the attachment(s) is: N (where N is the size of your attachment).
- Would you like to continue sending?

- Click **Send** to attach the files to an email.
- Click **Don't Send** to keep the files on your desktop.
- Click **Cancel** to discard the files.

FeatureCAM file types

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

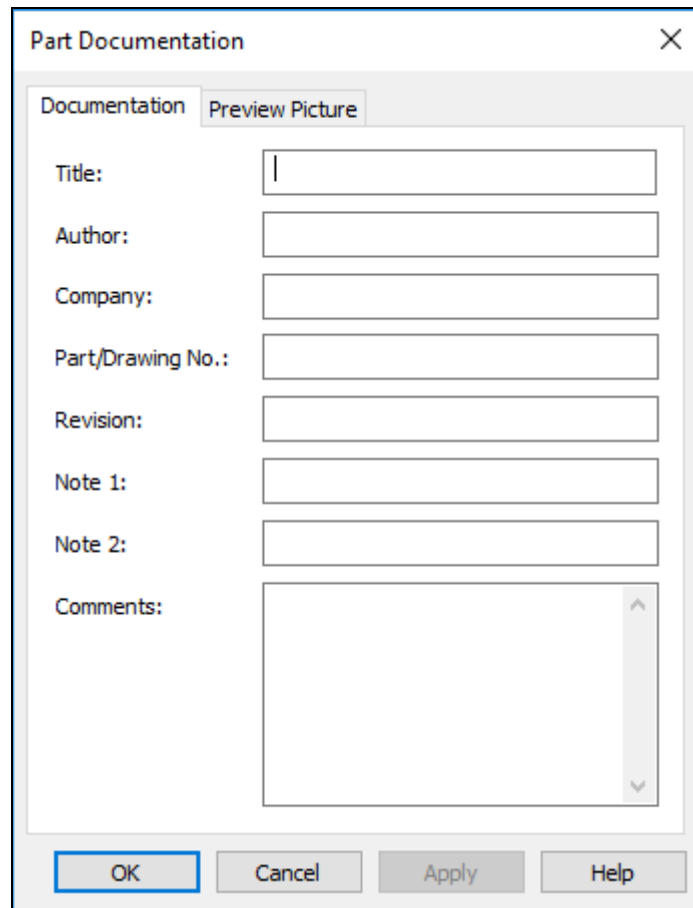
- **.op** is the Manufacturing Operation Sheet and is the same information shown on the **Details** tab when you select **Operation List**.
- **.tl** is the Manufacturing Tool Detail Sheet and is the same information shown on the **Details** tab when you select **Tool List**.
- **.txt** is a text file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

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

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

Documentation tab

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




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

Preview Picture tab

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

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

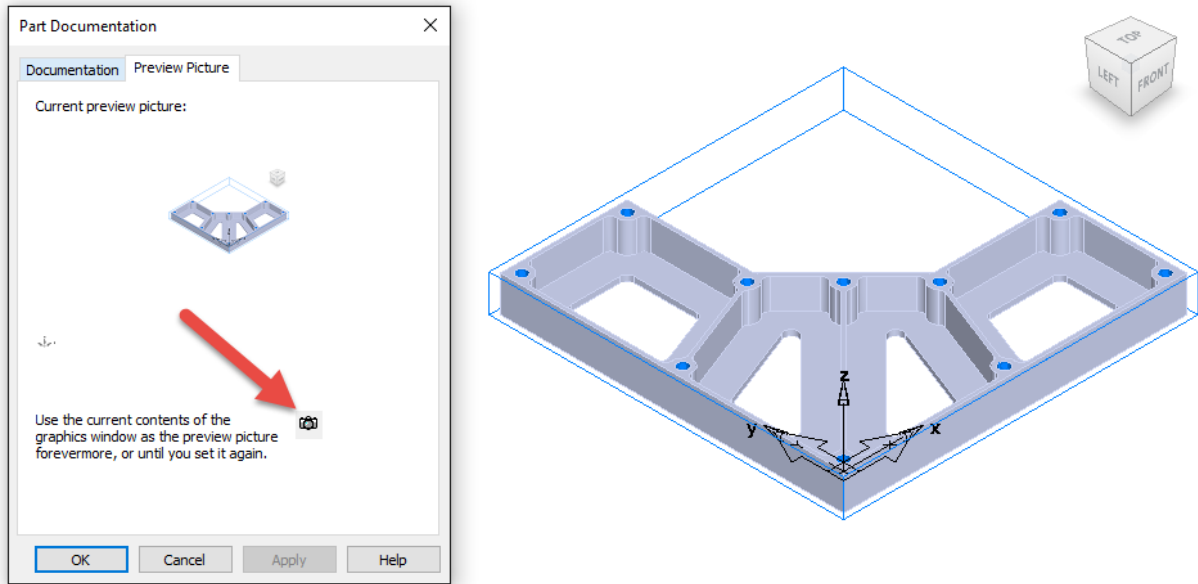
- 1 Create the view of the part you want to store.



- 2 Click Update preview picture



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



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

Saving your settings

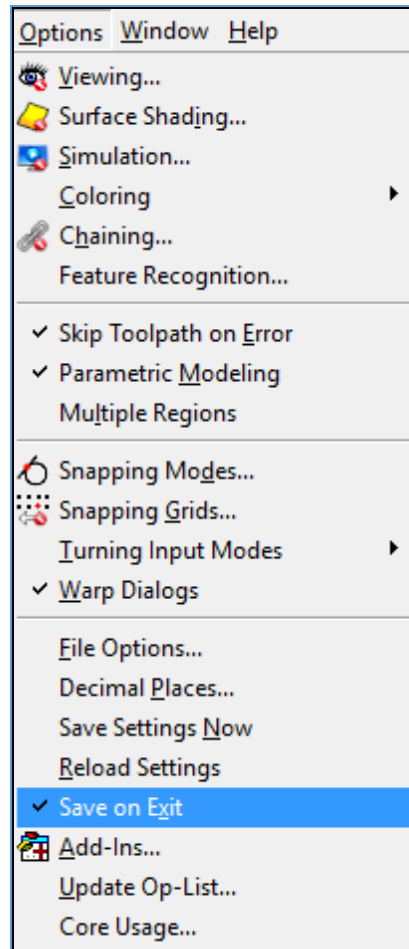
- FeatureCAM uses two **.ini** files to store your settings:
- Ezfm_ui.ini** contains toolbars, dialog locations, graphics settings, colours, and other user settings.
- Ezfm_mfg.ini** contains manufacturing defaults and **.cfg** settings.

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



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

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



Import/Export

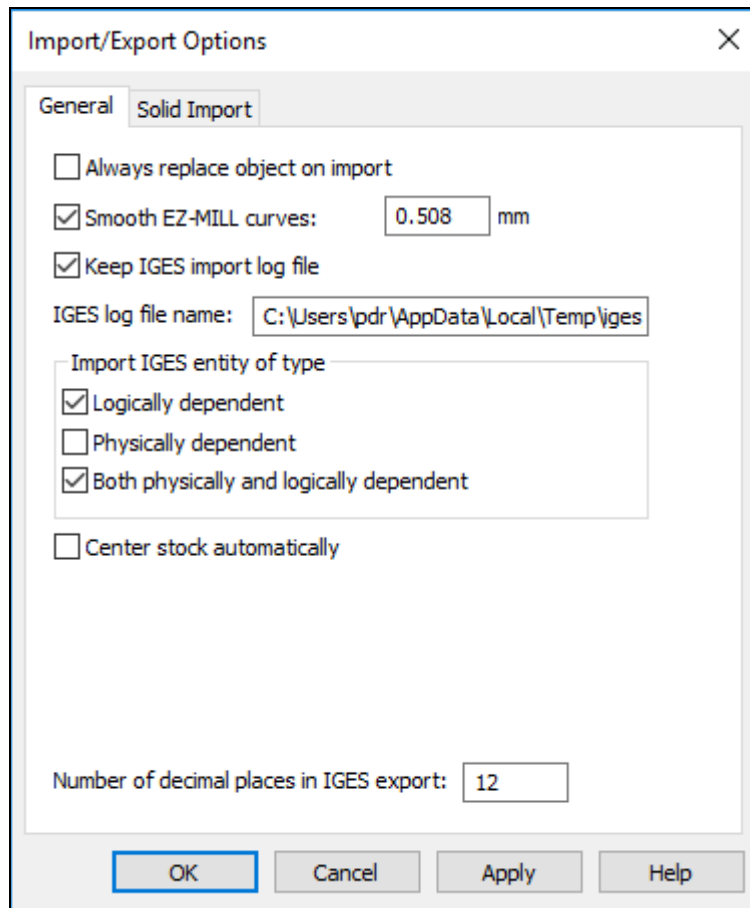
- You can import and export various file types into and from FeatureCAM.
- The settings for import and export are in the **Import/Export Options**
- To open this dialog select **File > Import/Export Options** from the menu.

Import/Export Options

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

General tab

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

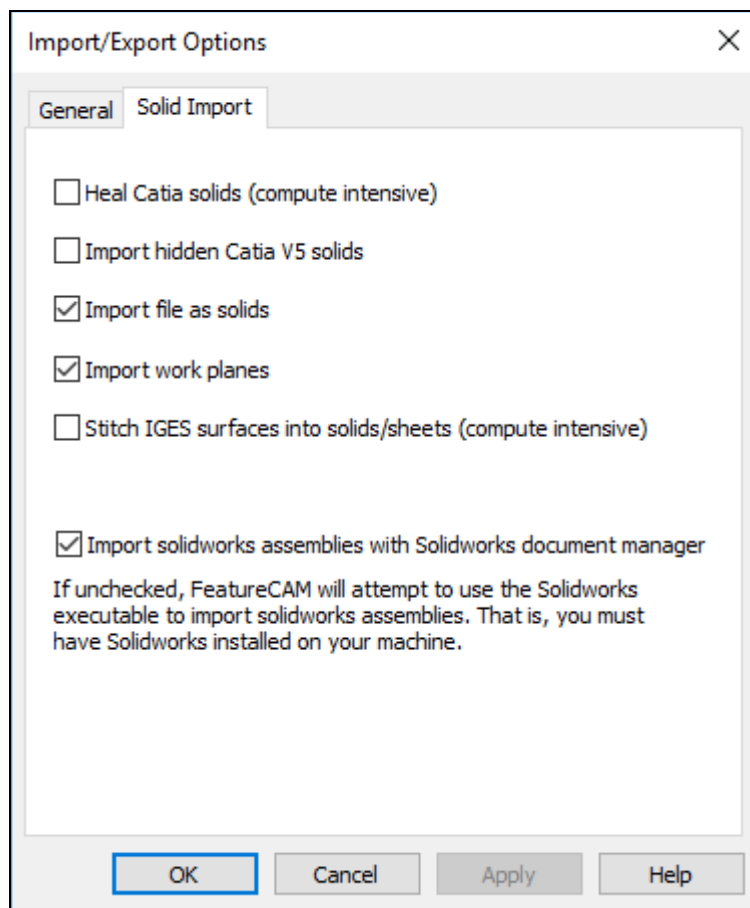


- **Always replace object on import:** - Select this option to overwrite objects of the same name when you import. Deselected prompts for every replacement. You can't have two objects with the same name.
- **Smooth EZ-MILL curves:** - Select this option to improve EZ-MILL curves on import because FeatureCAM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

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

Solid import tab

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



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

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

Importing Files

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

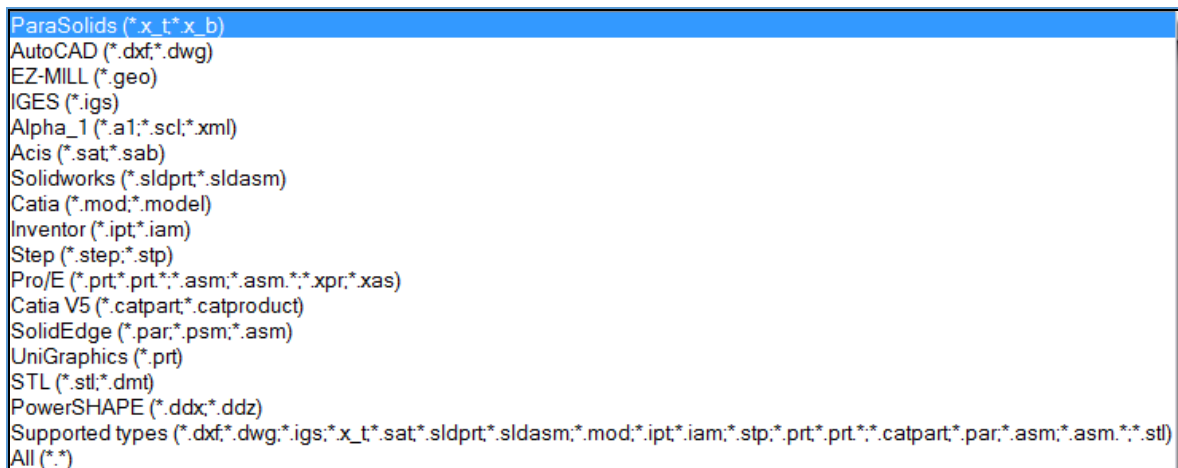
To import a CAD model from a file:

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



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

- 8 Follow the instruction in the wizard to complete the process.



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



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



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



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



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

Mouse buttons

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



Left mouse button Picking and selecting.

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



Middle mouse button Dynamics.

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

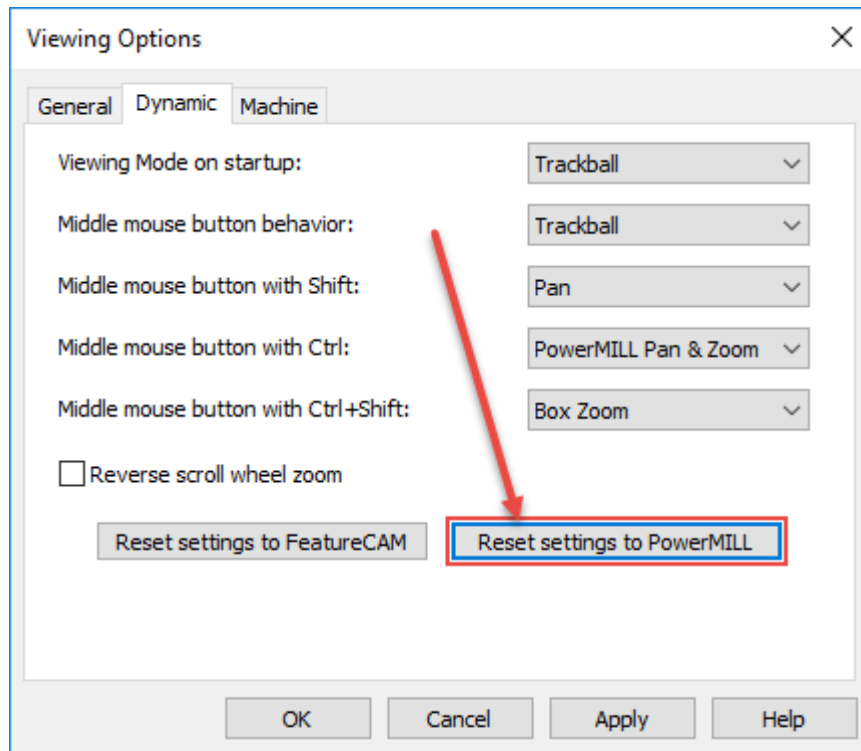
- The most commonly used mouse feature is to hold the **Centre wheel down** and moving the mouse, this will rotate your part. Hold the shift key down at the same time and you can **Pan** anywhere on the screen. Hold the **shift key** down and where ever the mouse is it will **zoom** to that point by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

Right mouse button Special Menus & FeatureCAM Options.



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

Viewing Options



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

Reset to FeatureCAM Settings

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

- **Dynamic Viewing Choice** upon FeatureCAM Start-Up - Pan and Zoom
- Middle-mouse button behaviour - **Current Viewing Mode**
- Middle-mouse button with Shift - **Pan and Zoom**
- Middle-mouse button with Ctrl - **Rotate**
- Middle-mouse button with Ctrl+Shift - **Zoom**

Reset to PowerMILL Settings

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

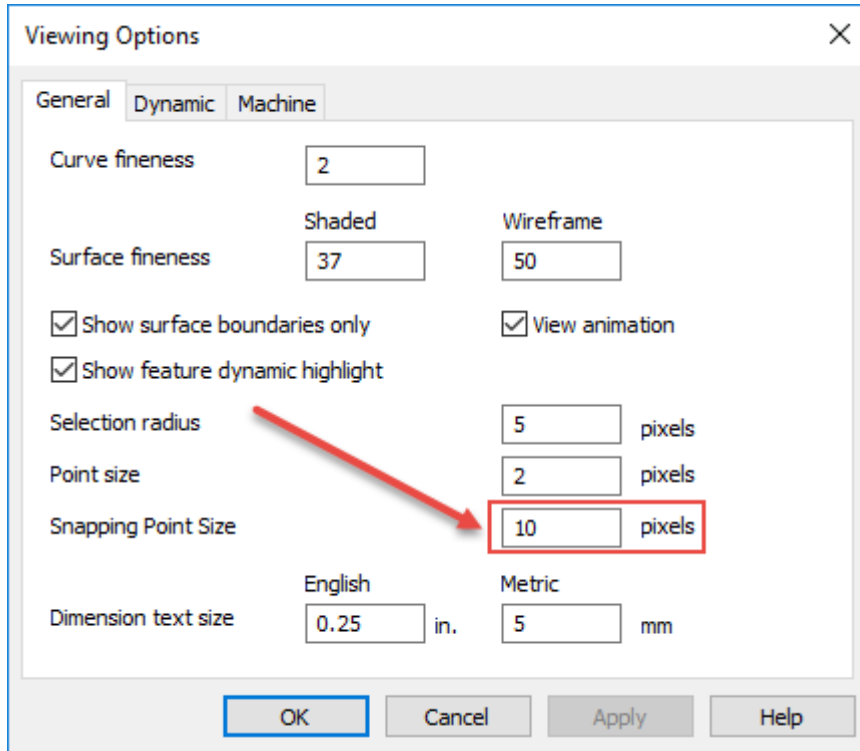
Dynamic Viewing Choice upon **FeatureCAM** Start-Up - **Trackball**

- Middle-mouse button behaviour - **Trackball**
- Middle-mouse button with Shift - **Pan**
- Middle-mouse button with Ctrl - **PowerMILL Pan & Zoom**

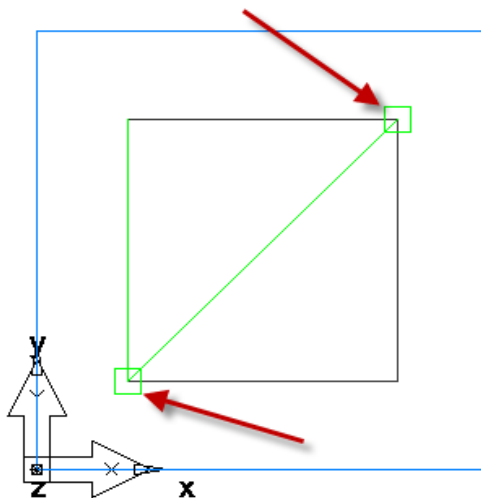
- Middle-mouse button with Ctrl+Shift - **Box Zoom**
- This menu controls the default viewing mode when **FeatureCAM** first opens. For example, if you select **Trackball**, the next time **FeatureCAM** starts up, the viewing mode is **Trackball**.

Viewing Options General

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

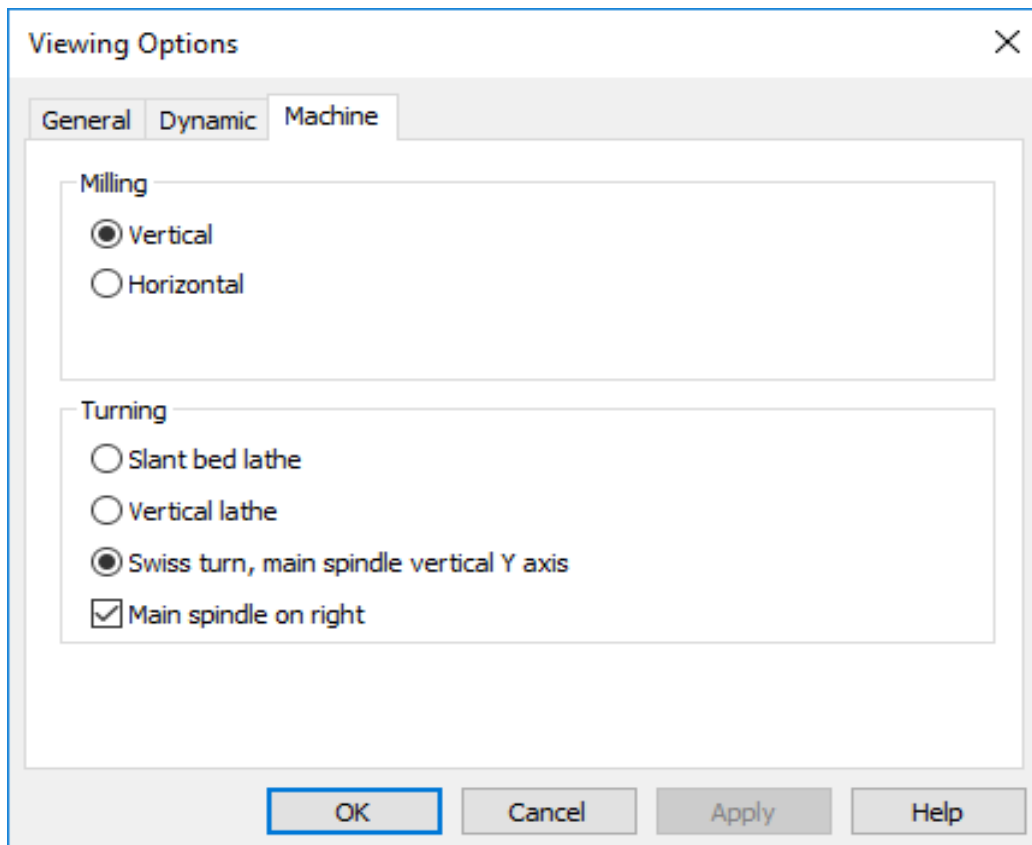


An example of this is shown below



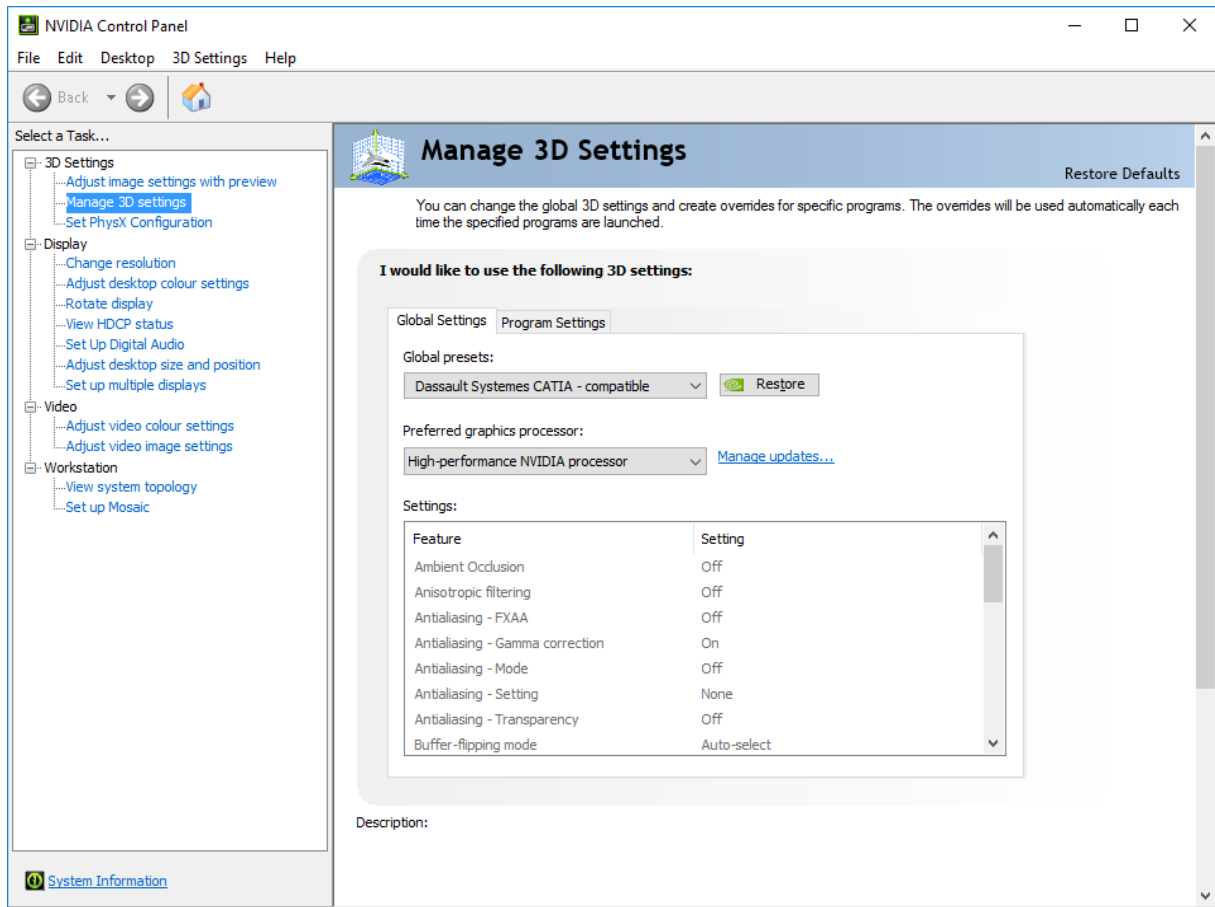
Viewing Options Machine

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




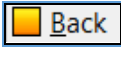
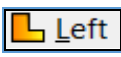

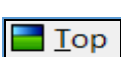
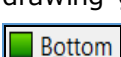
Display options

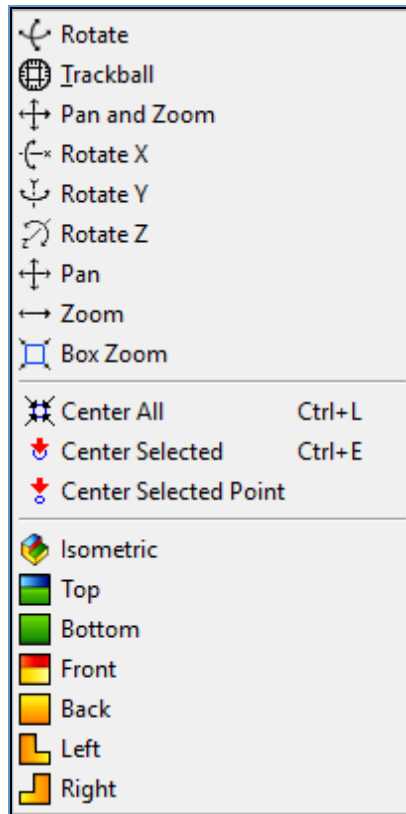
- For the best **Display Options** for your computer set the **NVidia** [®] **Control Panel** options to the following.



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

Principle views

-  **Front** Changes to a view of the part from the front with no other surfaces visible.
-  **Back** Changes to a view of the part from the back with no other surfaces visible.
-  **Left** Changes to a view of the part from the left side with no other surfaces visible.
-  **Right** Changes to a view of the part from the Right side with no other surfaces visible.
-  **Top** Changes to a view of the part from the top only. Useful for drawing geometry, but harder to see the wireframe model of the part.
-  **Bottom** Changes the view to the bottom of the part. No sides of the part are visible from this perspective.



Right Click in the graphics area to show this menu.

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

Keyboard shortcuts

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

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

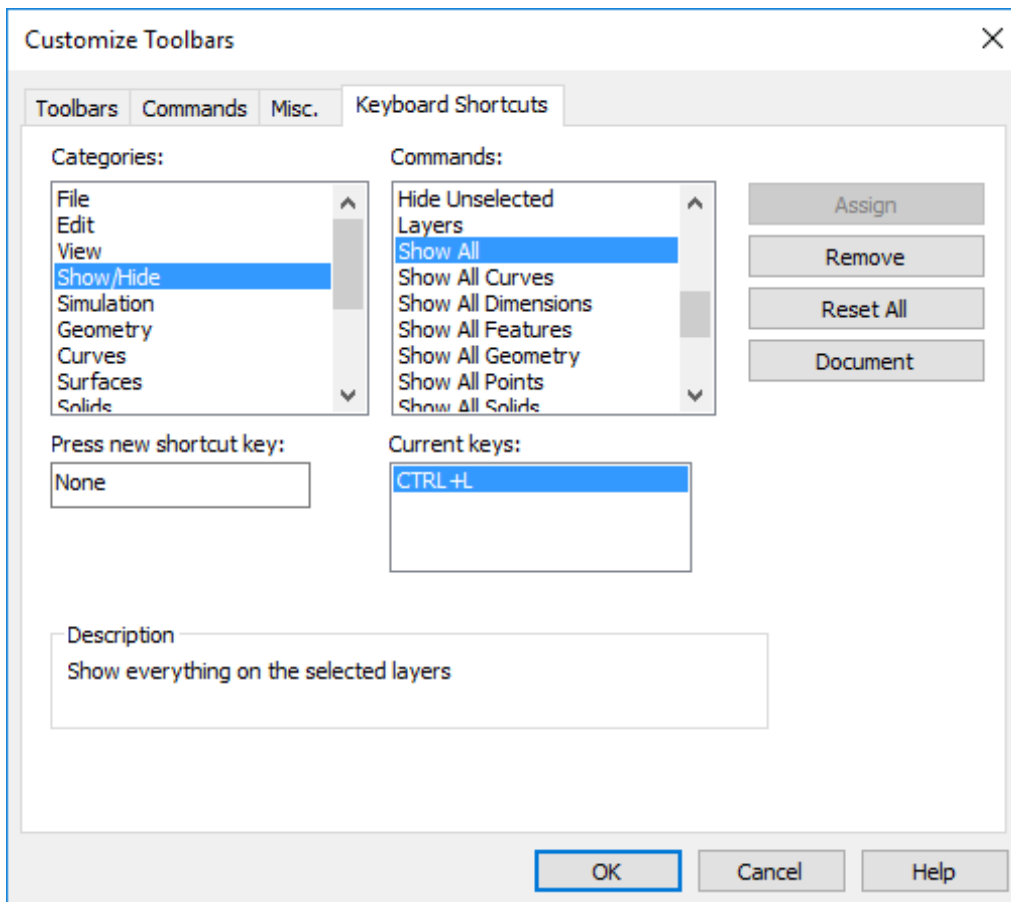
Ctrl+Alt +click-and-drag on any toolbar button	Duplicate and move button around in toolbar.
Ctrl+Shift +click the 3D Simulation button. Release keys and click the Play button.	Continuous loop simulation for 3D simulation.
Ctrl+Shift +click the Machine Simulation button. Release keys and click the Play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

Commonly used Ctrl Keys

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

User interface: Keyboard shortcuts

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



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

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

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



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

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

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

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

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


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

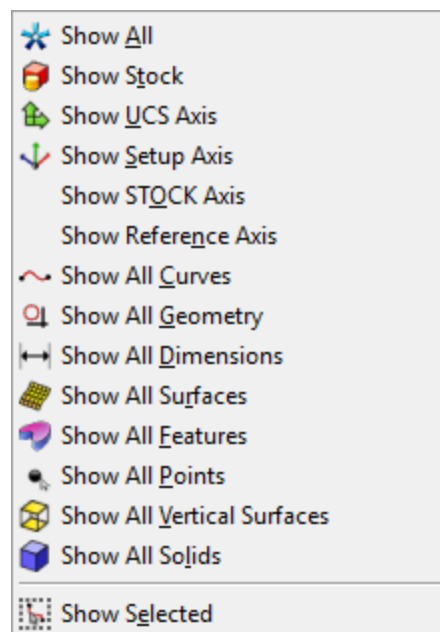
- **Remove** — Click this button to remove the shortcut assigned to the selected command.
- **Reset All** — Click this button to reset all shortcuts to the **FeatureCAM** defaults.

Blanking model entities

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

Show menu


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

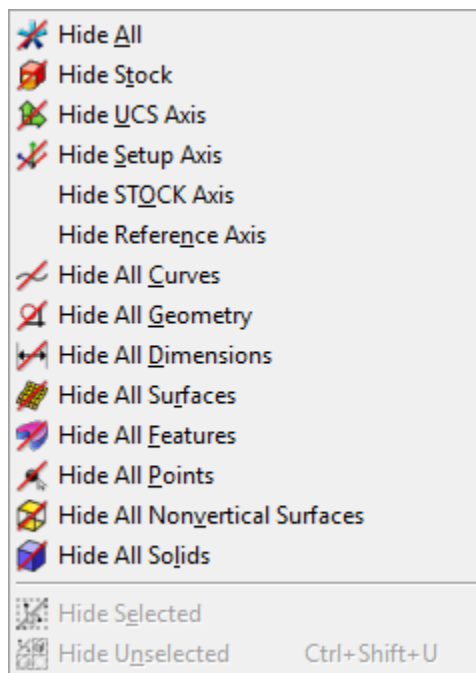


- Show **All** — Use this menu option to Show everything in the part model.
- Show **All Geometry** — Use this menu option to Show all geometry (points, lines, arcs, and circles).
- Show **All Dimensions** — Use this menu option to Show all dimension information added with the Dimensions Tool.
- Show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

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

Hide Menu

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



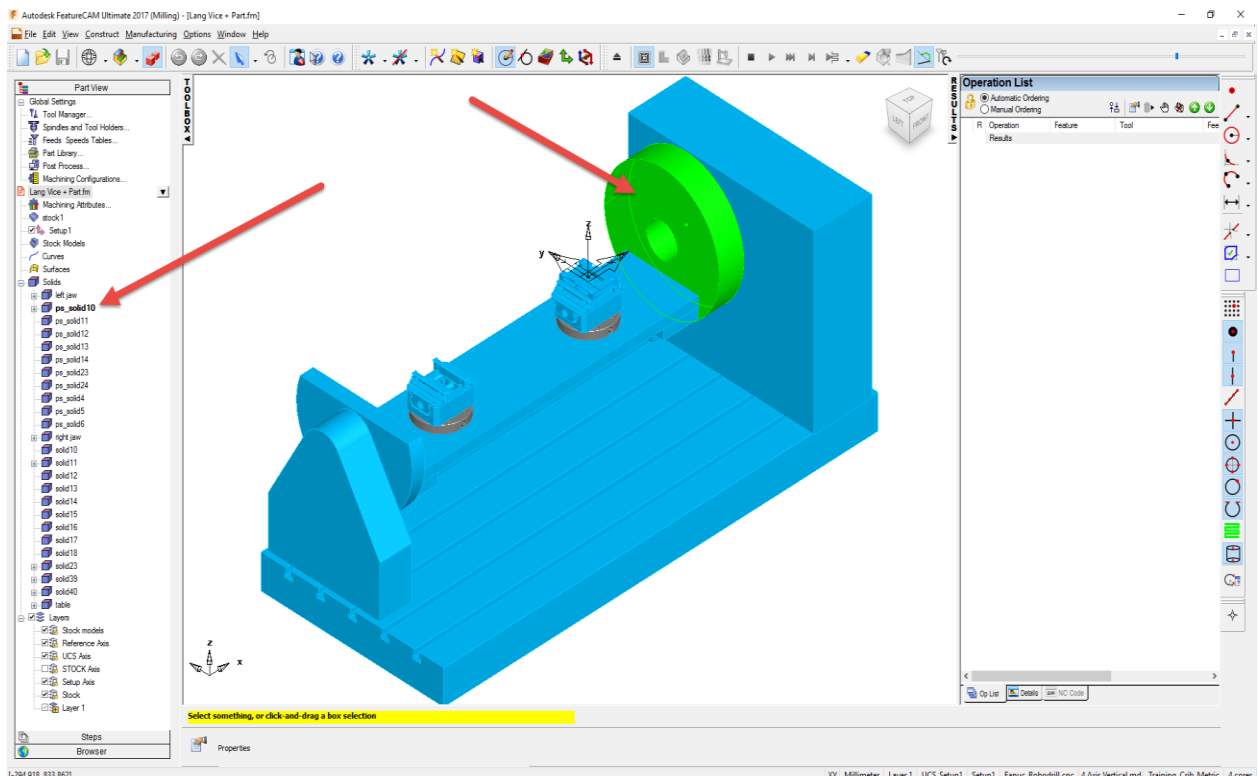
- Hide **All** — Use this menu option to hide all geometry, curves, Features. The stock and axis icon remain visible. A common procedure is to Hide **All**, then **Show** only one type of entity, for example features.
- Hide **All Geometry** — Use this menu option to hide all geometry. Other entities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

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

Highlighting objects from **Part View**

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

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



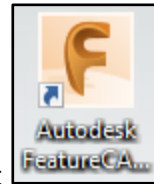
Setting file location options



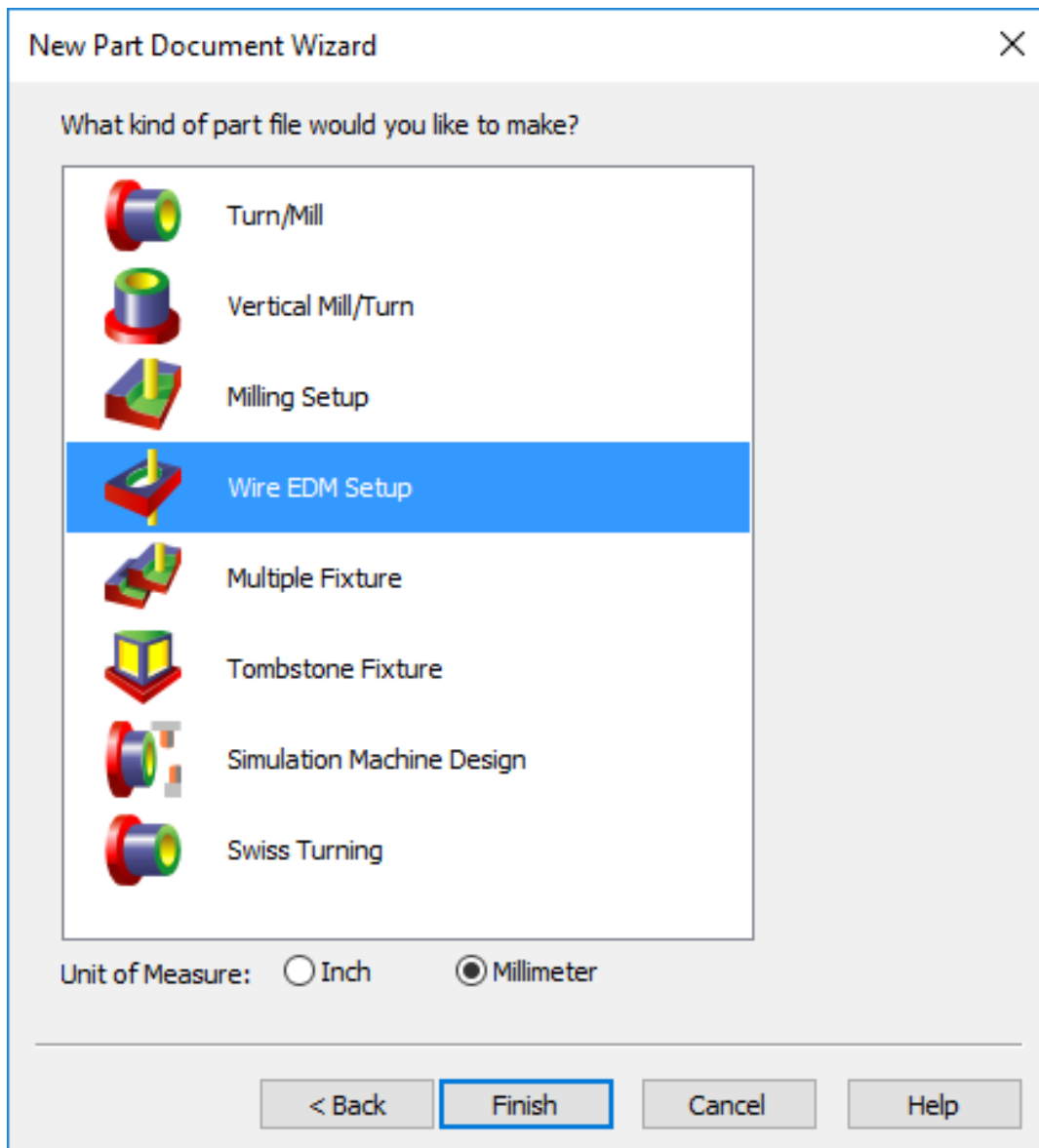
*It is a good idea to set the file location for where you will save your files. Please select **Options/File Options/Existing Files**.*

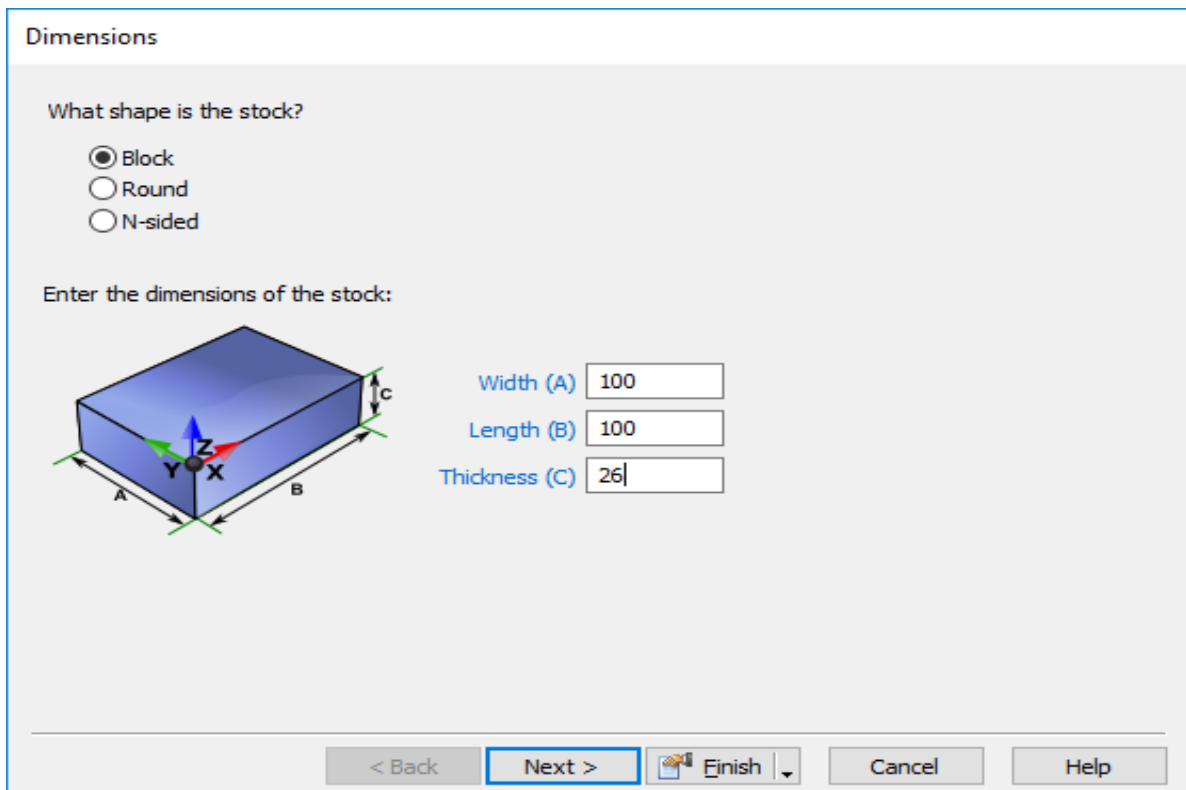
- 9 Select Browse. Then select the following location.
C:\Training_Data\FeatureCAM Course Data 2017.
- 10 Select **Apply** and **Ok** to close the form. When we save or import files **FeatureCAM** will take us to this location.
- 11 Please save all files in **Coursework**.
- 12 Tool Crib = **BasicMetric** or **Training_Crib_Metric**
- 13 Post processor used = **Fanuc_Robodrigill.cnc**

Stock dimensions

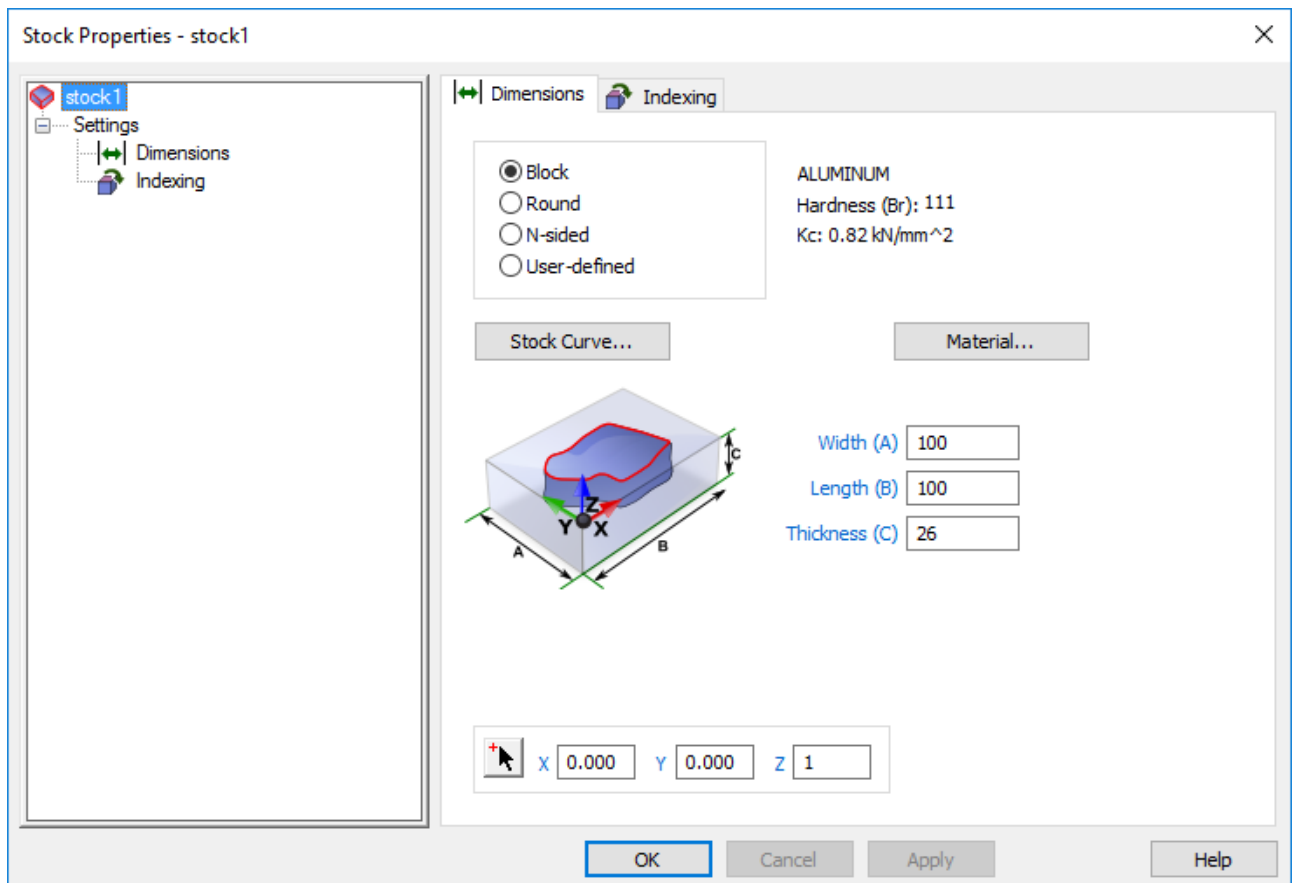


- 14 Open **FeatureCAM** by double clicking the **FeatureCAM** shortcut
- 15 Select **New File. Wire EDM Setup**. Select **Finish**.



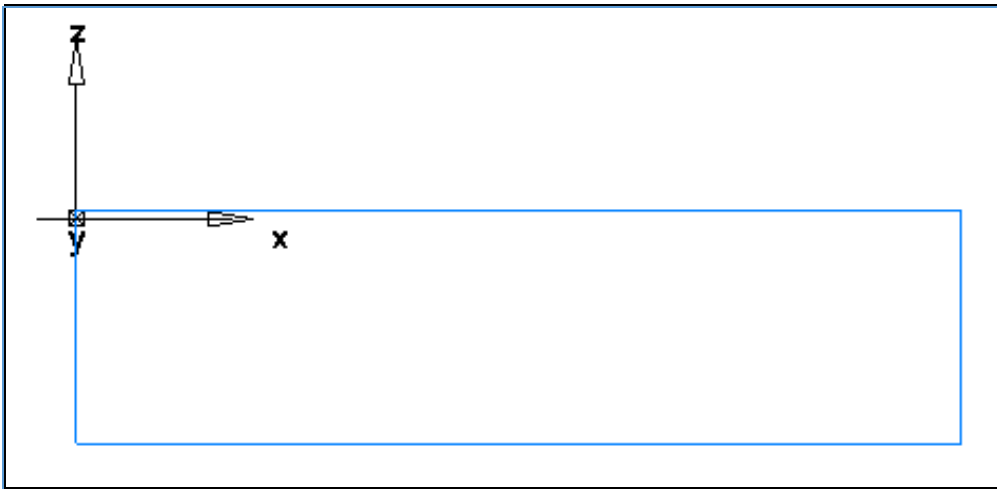


- 16 Enter the dimensions of the stock. Stock Size = **100mm x 100mm x 26mm thick** into the dimensions menu. Select **Finish** and the following menu will appear. Enter 1mm for Z to allow material above the datum. Select **Finish**.

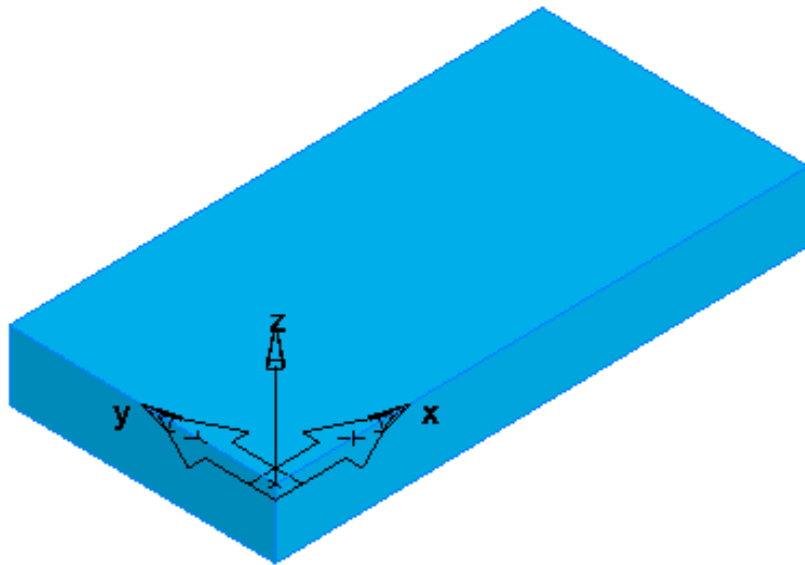




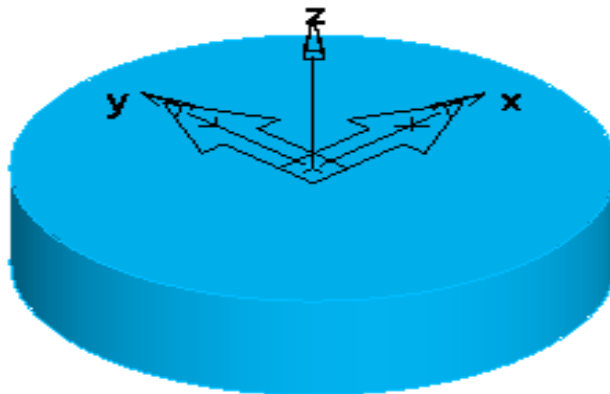
Notice the **1mm** in the **Z** Menu. This adds 1mm above the Z Datum for machining off later when creating a Face Feature.



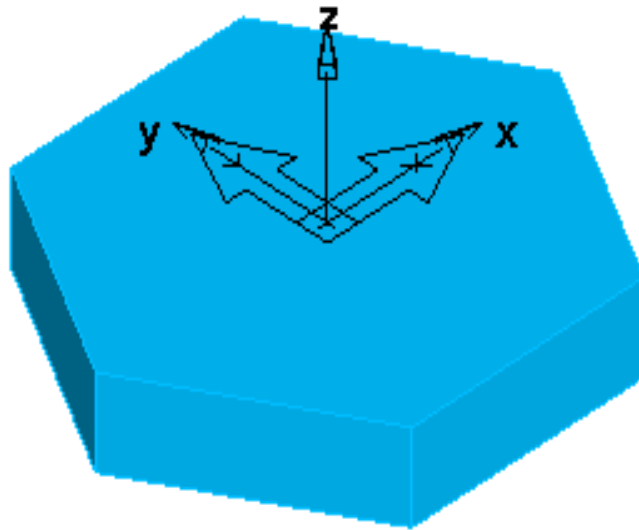
Stock Types available (Information Only)



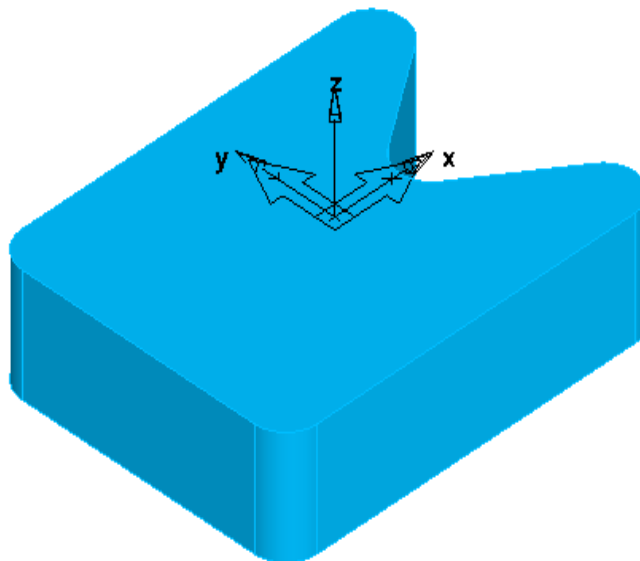
Block



Round

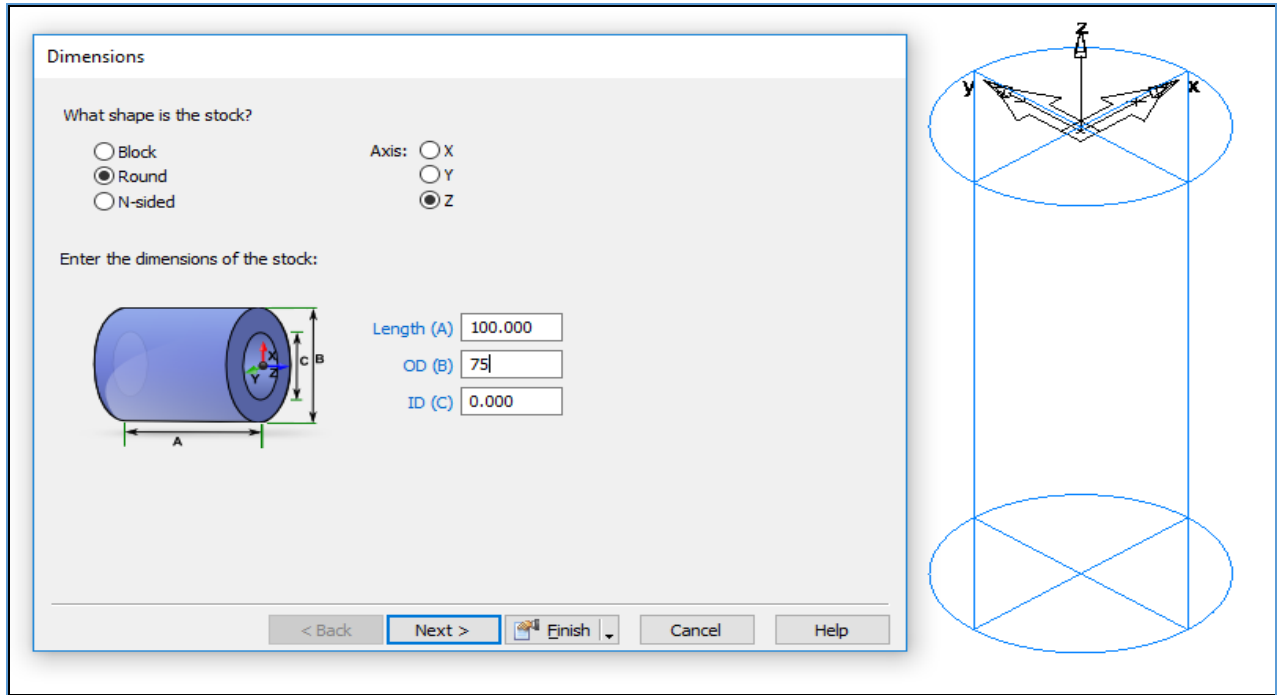


N-sided



User-defined

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

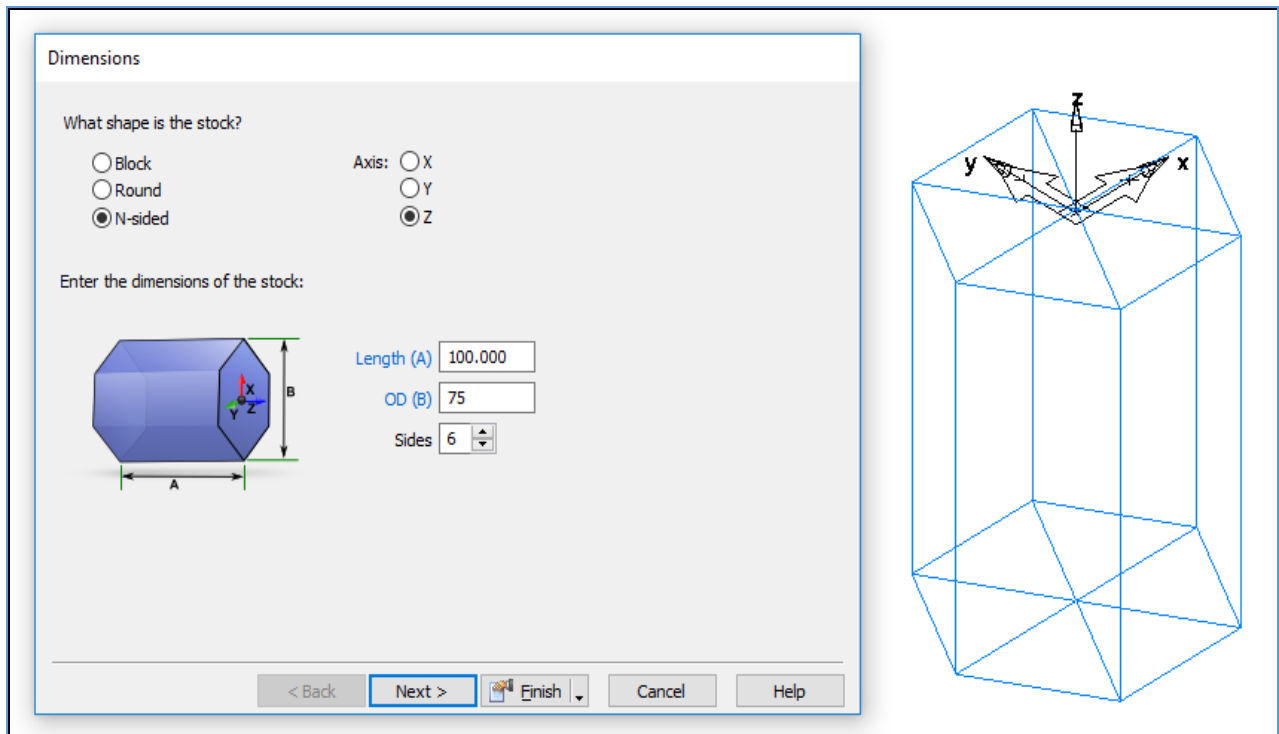


This option is ideal for work held in a **3 or 4 jaw chuck** on a **3 or 4 Axis** vertical machining Centre.

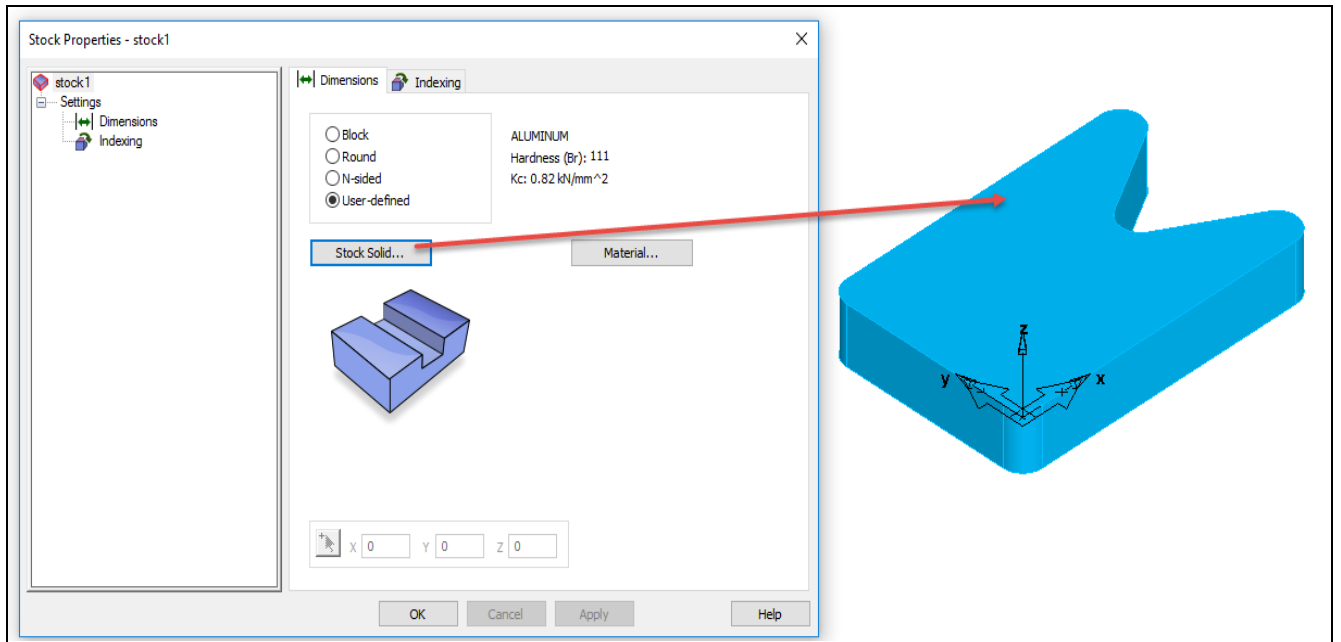
N-Sided



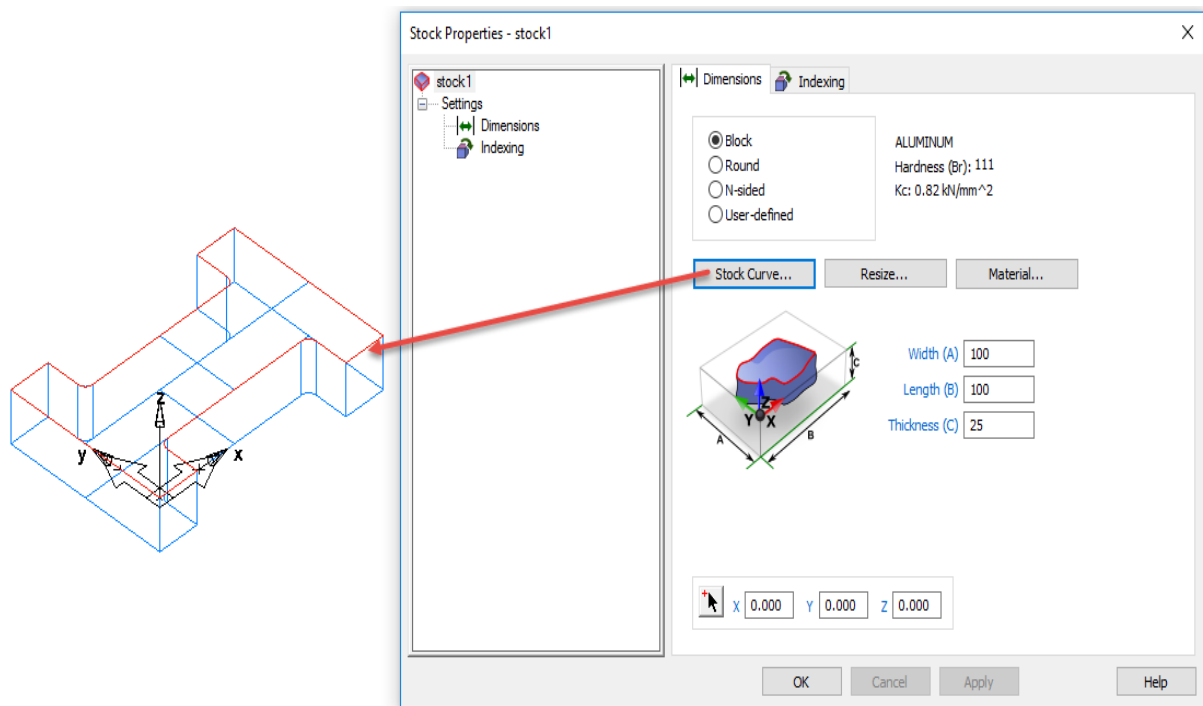
Select the **N-Sided** Radio button. When this button is pressed the stock can be defined along a specific axis with a multiple number of sides. The dimensions shown are just an example only.



User Defined – Stock Solid



Stock Curve



Stock – Material

Stock - Material

What type of material is the stock?

Material:

Specific cutting force: kN/mm²

Hardness:

Hardness Units

Brinell

Rockwell B

Rockwell C

Tensile Strength (ksi)

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

Stock - Material

What type of material is the stock?

Material:

Specific cutting force: kN/mm²

Hardness:

Hardness Units

Brinell

Rockwell B

Rockwell C

Tensile Strength (ksi)

ALUMINIUM

ALUMINIUM-CAST < 10SI

ALUMINIUM-CAST > 10SI

BRASS

BRONZE

CARBIDE

CAST_STEEL

COPPER

DEP_URANIUM

DUCTILE_CAST

DUCTILE_IRON

FREE_CUTTING_BRASS

GRAPHITE

GREY_CAST_IRON

HARD_STEEL

HEAT_RES_CO

HEAT_RES_FE

HEAT_RES_NI

LEAD_ALLOY

MAGNESIUM

MANGANESE

NODULAR_IRON

PLASTIC

STAINLESS-15-5_PH

STAINLESS-16-6_PH

STAINLESS-17-14_CU_MO

STAINLESS-17-4_PH

STAINLESS-17-7_PH

STAINLESS-201

STAINLESS-202



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



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

Multi-axis positioning

Do you want to use multi-axis positioning to machine this part?

No

4th Axis Positioning

Index around the STOCK X Axis
 Index around the STOCK Y Axis
 Index around the STOCK Z Axis

5th Axis Positioning

Finish ↓ " style="border: 1px solid #0070C0;"/>

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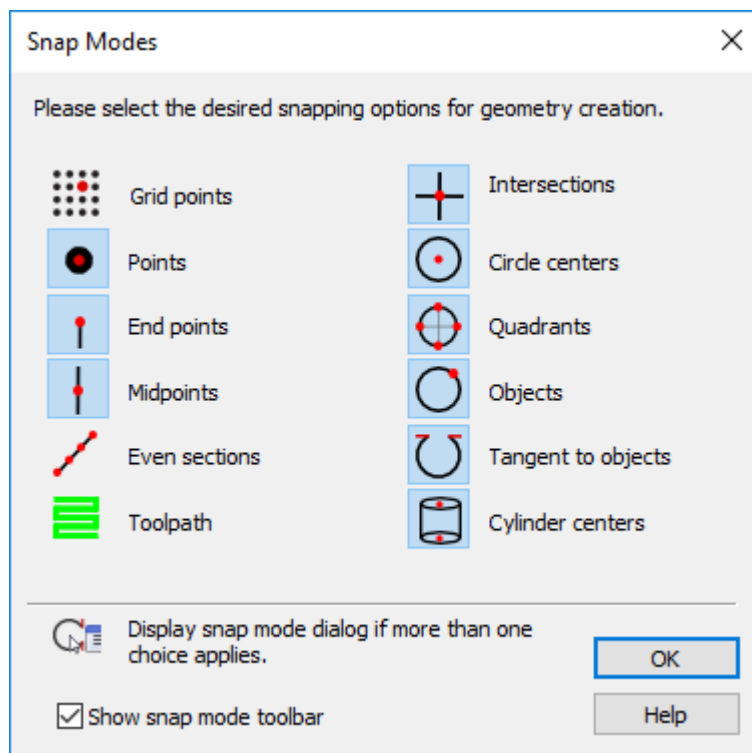
Wire Geometry Creation Techniques

Introduction

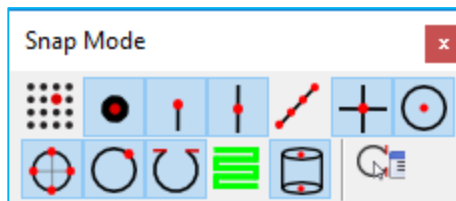
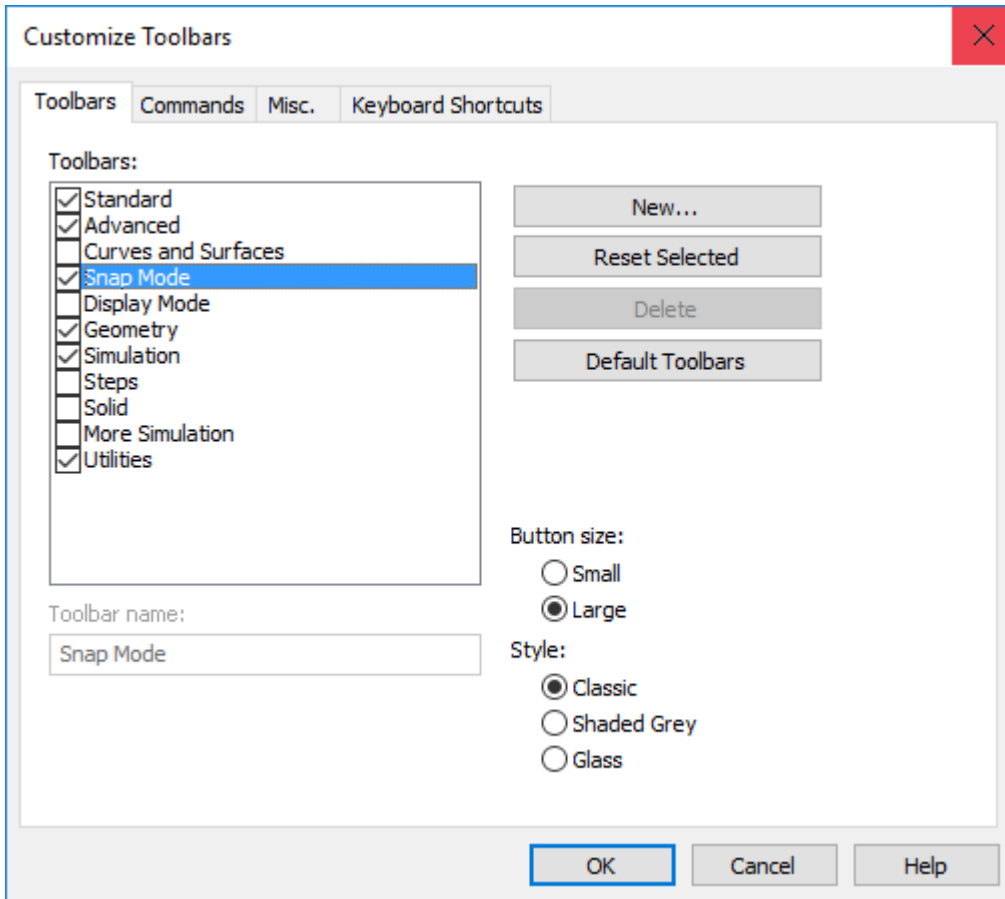
This module will provide an understanding of how different **Snapping Modes** work and how **Geometry** is created within **FeatureCAM**. By the end of the module the user will be able to construct **Geometry** and understand how each **Snapping Mode** functions.

Snapping

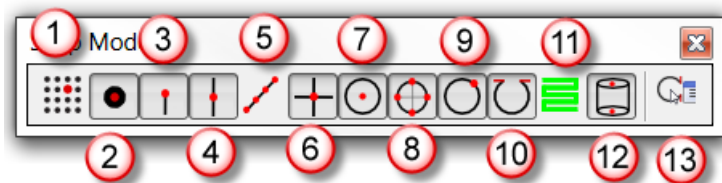
- **Snapping Mode** commands can be accessed in two different areas of the user interface. Select the **Options** menu then **Snapping Modes**.







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





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





- 1 The **Snap to Grid Points**  button displays a grid and enables snapping to the grid. Selecting snapping grids from the options menu controls the spacing of the grid. Grid snaps to a point on a coordinate system that is laid out on the **stock**. The grid size can be modified from the **Options** menu.
- 2 The **Snap to Point**  button snaps to a point object.
- 3 The **Snap to Endpoint**  Snaps to the ends of lines or arcs. **Endpoint** also applies to the corners of the stock and verticals of a Solid.


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


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

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

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

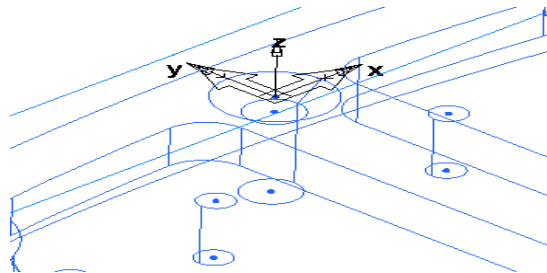
8 **Snap to Quadrant**  snaps to the four points on a circle corresponding to 0°, 90°, 180° and 270°.


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

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

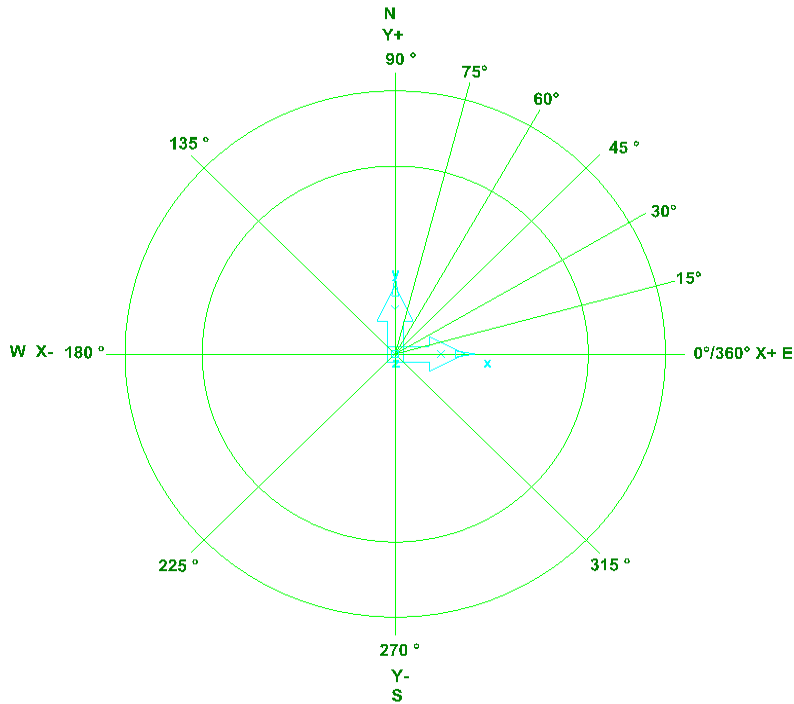
11 **Snap to Toolpath**  snaps to toolpath lines.

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



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


Drawing conventions when using Autodesk products.

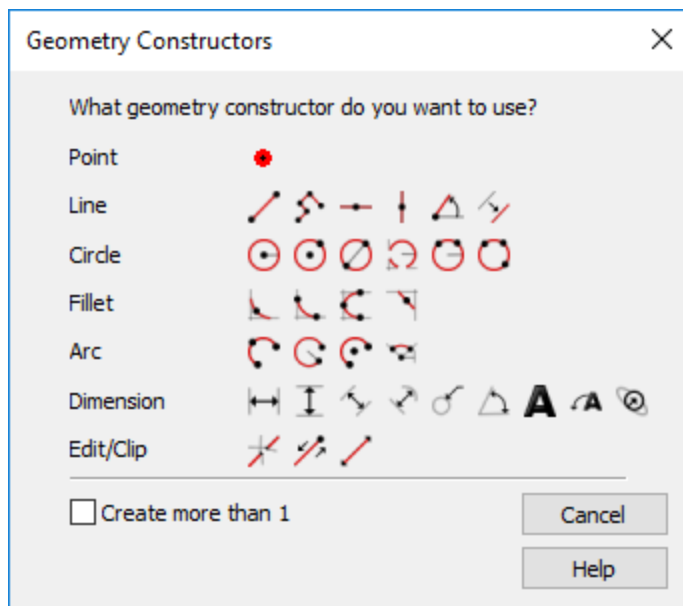


- When you start using **Autodesk** products and start to construct Geometry, the illustration above clearly shows you the directions of the cardinal drawing angles and positive and negative directions. This should help when creating the exercises in this chapter.

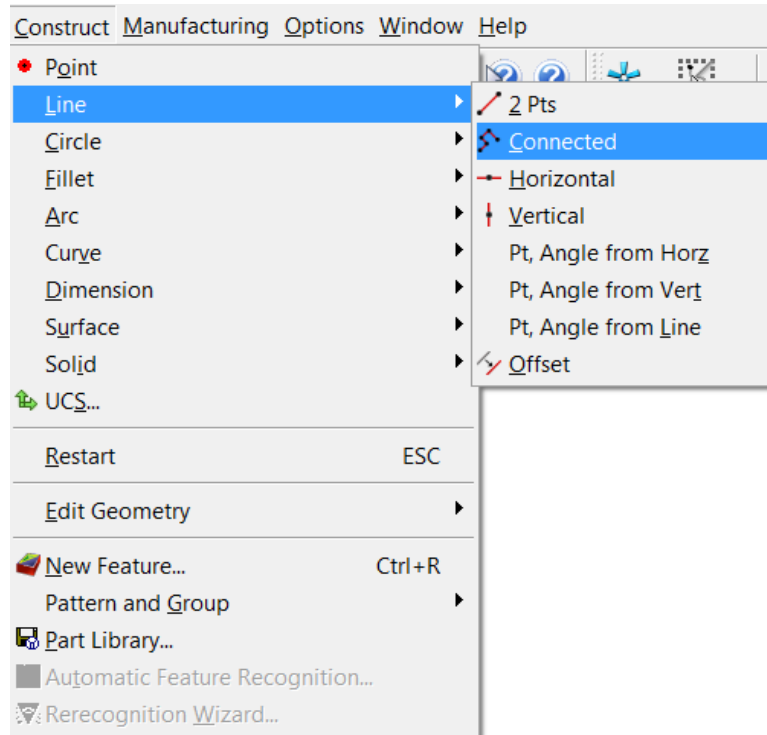
User Interface

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

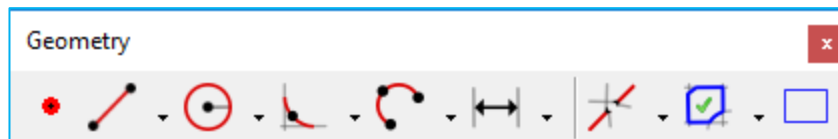
- Commands can be accessed by clicking the **Geometry** icon in **Steps**: 



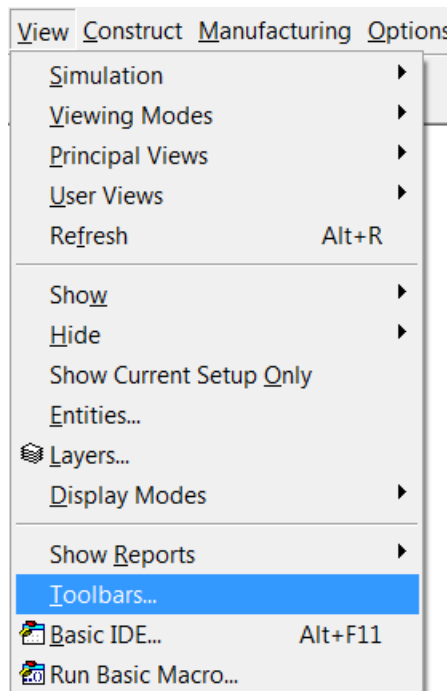
- This can also be accessed through the **Construct** Menu.

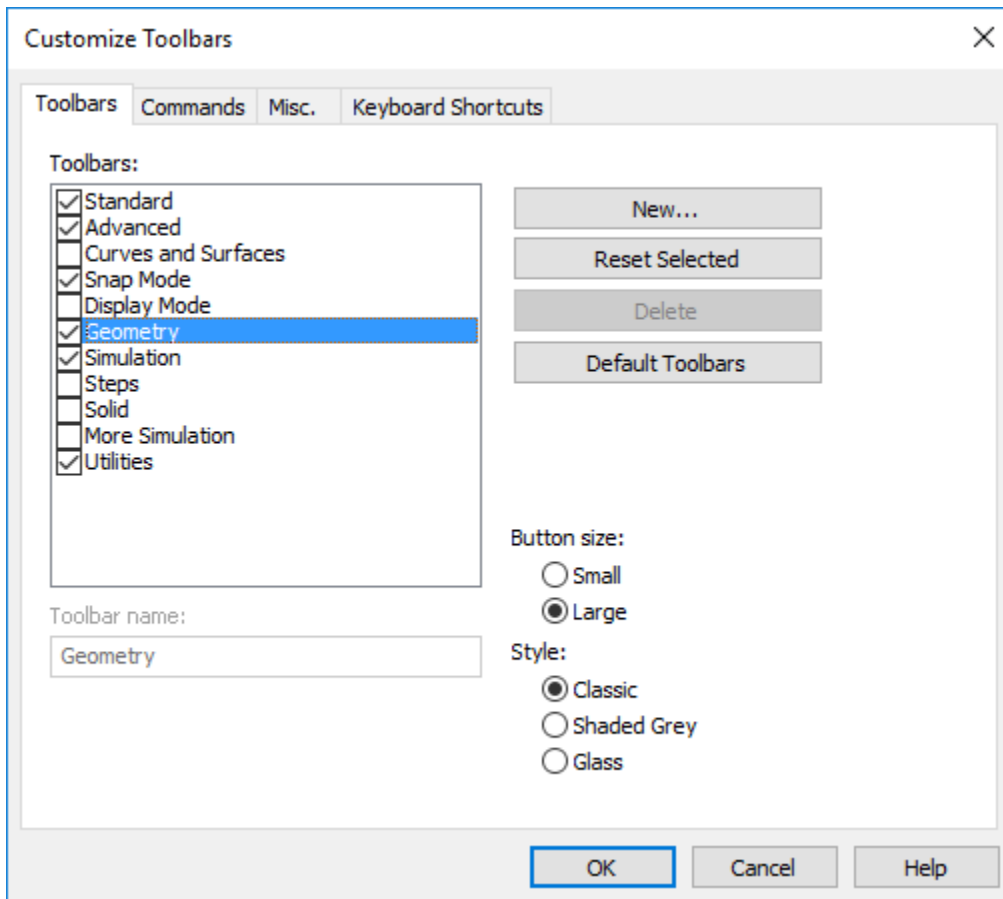


- Or may be accessed by the **Geometry Toolbar**

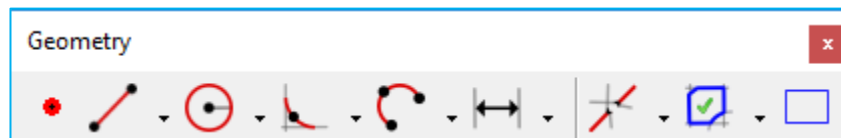


To show the geometry toolbar click on **View>Toolbars**, then check the **Geometry checkbox** and press **OK**





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

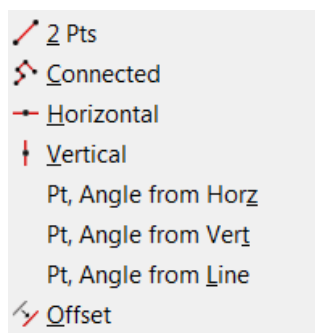


- Click on the **triangle** to reveal the flyout menu.

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

- The following Menu show the Geometry creation options available.

Line

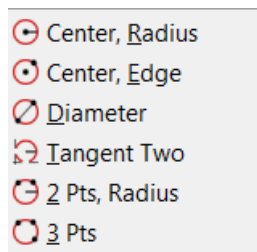


- **2 Pts:** - Line created with two endpoints.
- **Connected:** - Multiple lines created in succession. Endpoint of one line becomes start point of next line.
- **Horizontal:** - Infinite horizontal line created through a point.
- **Vertical:** - Infinite vertical line created through a point.
- **Pt.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



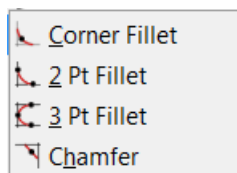
- Click on the Select icon at the top of **FeatureCAM** on the Standard Toolbar to cancel the line drawing. Use this button at any time to exit selection mode. Or press the **Esc** key at any time.

Circle



- **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

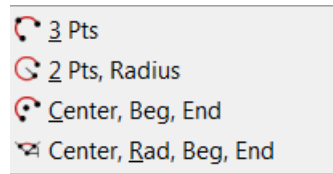
Fillet



- **Corner Fillet:** - Creates a Fillet in a corner originally defined by an intersection of lines or arcs.
- **Two point Fillet:** - Creates a Fillet in a corner originally defined by an intersection of lines or arcs.

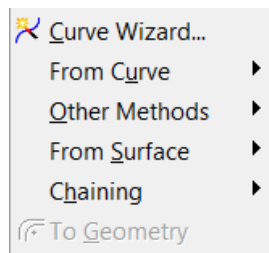
- **Three points:** - Defines a fillet by selecting three points, similar to the three point circle.
- **Chamfer:** - Creates a Chamfer in a corner originally defined by an intersection of lines or arcs.

Arc



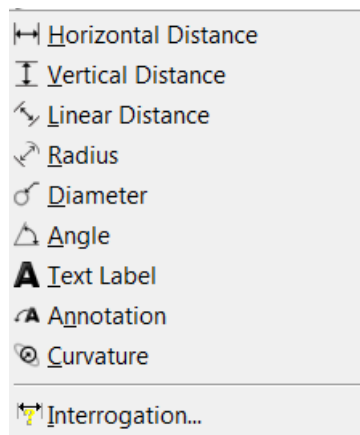
- **Three points:** - Constructs an arc through a start point, edge point, and a finish point.
- **Two points, radius:** - Constructs an arc through two points with a specific radius.
- **Center, Beg, End:** - Constructs an arc from a center point and given beginning and end points.
- **Center, Rad,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

Curve



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

Dimension



- **Horizontal:** - creates dimension information based on the horizontal axis of the part.

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

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

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

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

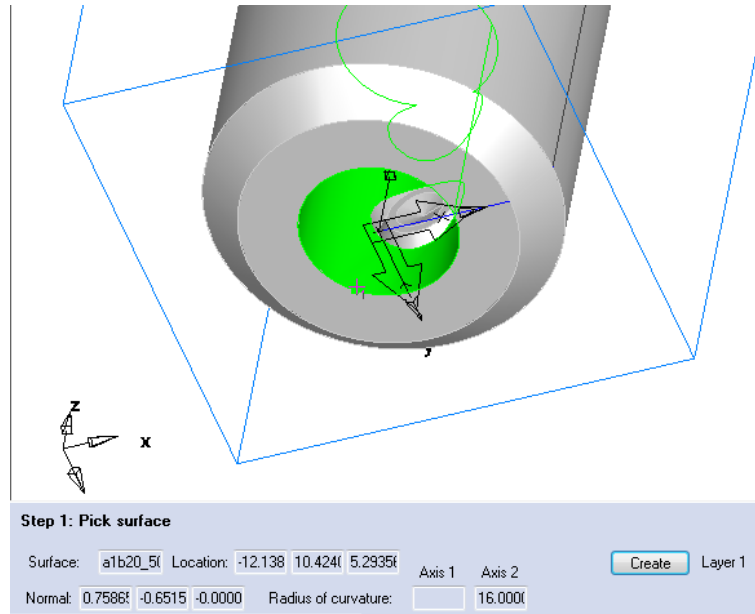
Annotation: - places explanatory text, entered in the dialogue with an arrow of what is being explained

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

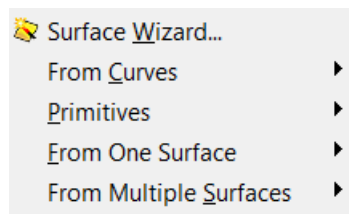
Interrogation: - This dialog helps you extract numbers from the graphics window using snap modes and pick filters. You can then cut and paste these values into other dialogs.

Curvature

- Curvature samples the surface/Solid and computes the Curvature in two directions to describe how the surface behaves at the point shown in the dialog bar. Unlike the other dimensioning tools, Curvature is a real-time rubber-banding effect where you traverse the surface to find the point with the smallest Curvature radius. Knowing the smallest radius tells you the smallest tool to use to manufacture the surface.
- Select **Construct > Dimension > Curvature** from the menu.
- Move the cursor over the surface, particularly in the tight constrained areas of the surface or joints.
- Note the smallest value shown for **Curvature**. That is the smallest tool end radius you need to accurately machine the surface.
- Set up rough and finish passes for the surface feature based on this knowledge, and make sure the tool is available for production.

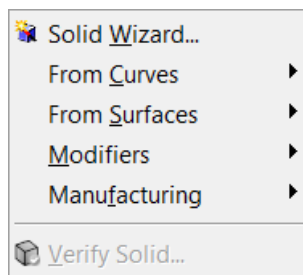


Surface



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

Solid

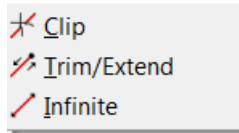


- The **FeatureCAM** support of solids lets you work with imported CAD designs from solid modelling systems and create 3D solid models using **FeatureCAM's** Solid Modelling Tools. Solids are a convenient representation for 3D parts because they group collections of surfaces into 3D volumes. By working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



Please Note that the Creation of Solid Models in **FeatureCAM** is an **extra cost option** and will be **greyed out** if you have not **paid** for this **option**. Please speak to your local sales representative for more details.

Edit Geometry



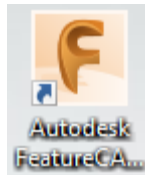
- **Clip:** - Clip removes a region of a line, arc, circle or curve. A region is defined as a portion of an object between two intersection points.
- **Trim/Extend:** - Changes the length of a line or an arc. Trim/extend can be used to lengthen or shorten lines and arcs as follows:
- **Infinite:** -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

Geometry Creation Exercise 1 (Class Exercise)

Introduction

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

- 1 Please **start** a **New** File document in **FeatureCAM**.



- 2 Select the **FeatureCAM** Icon on your **desktop** or **Taskbar**.
- 3 Select **New File** then **Milling Setup** and select **Millimeters**.

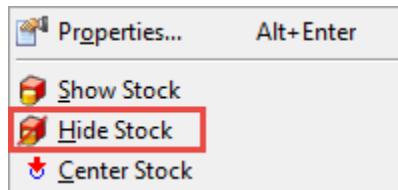


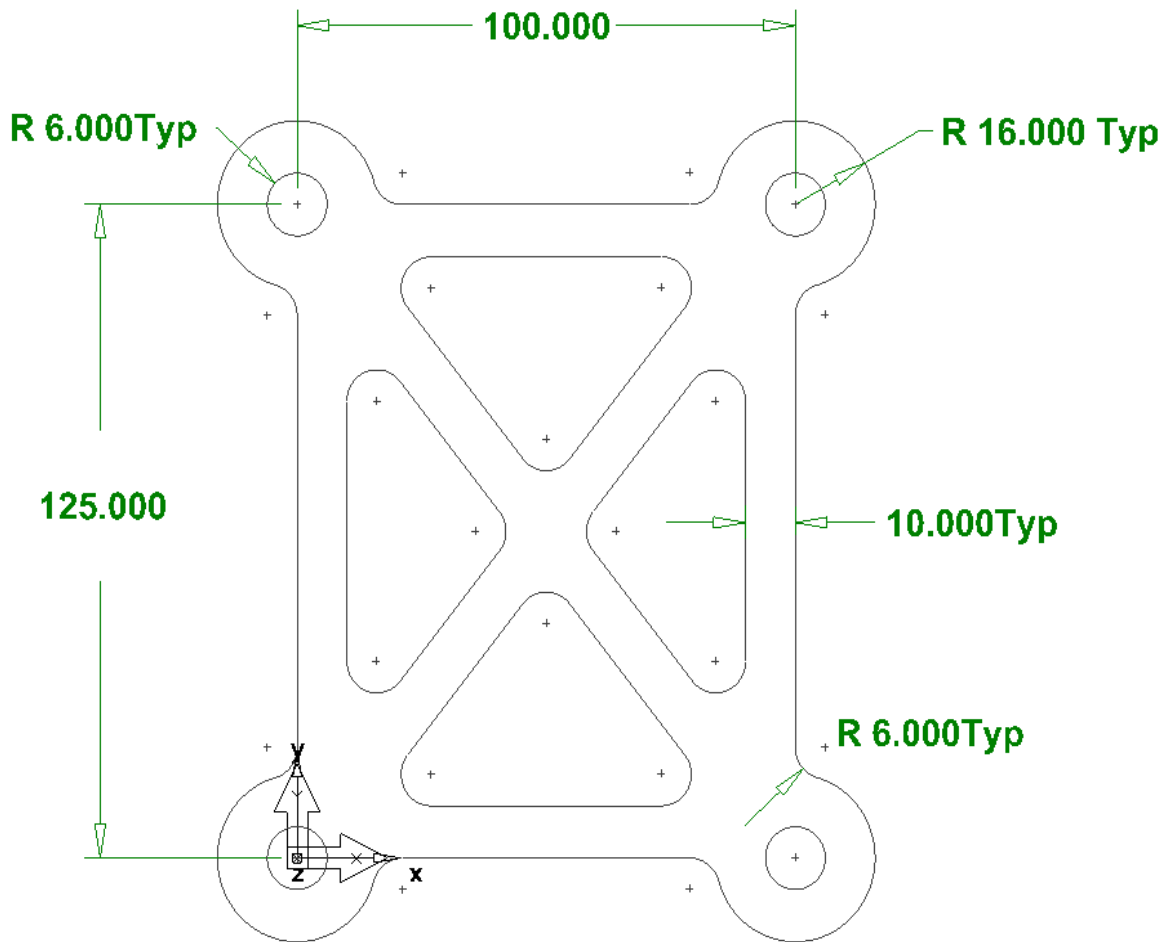
Typ means *Typical for all dimensions that are the same unless otherwise stated. **If in doubt ask***



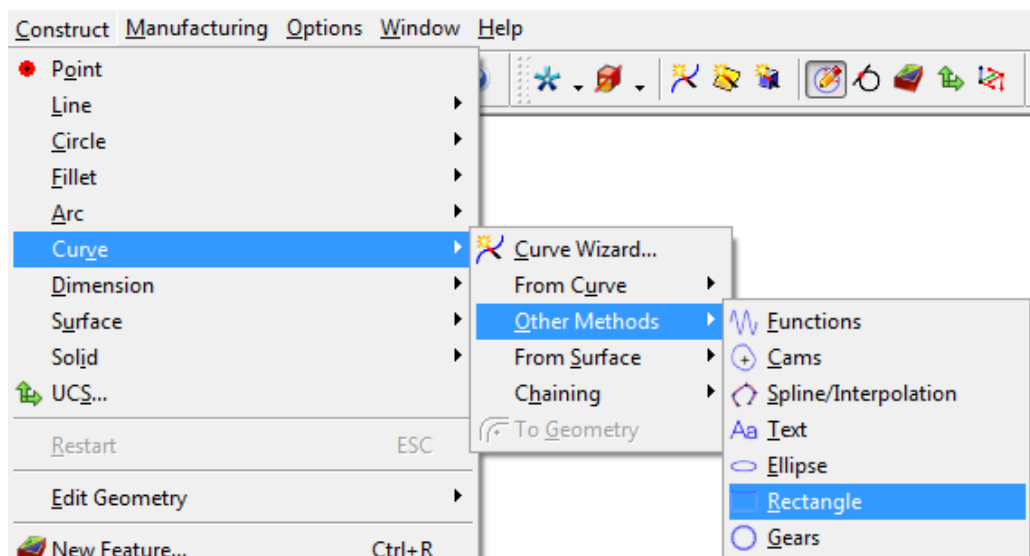
Do not worry about the stock size at this stage as we will resize this later. We are concentrating on Geometry creation techniques only.

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





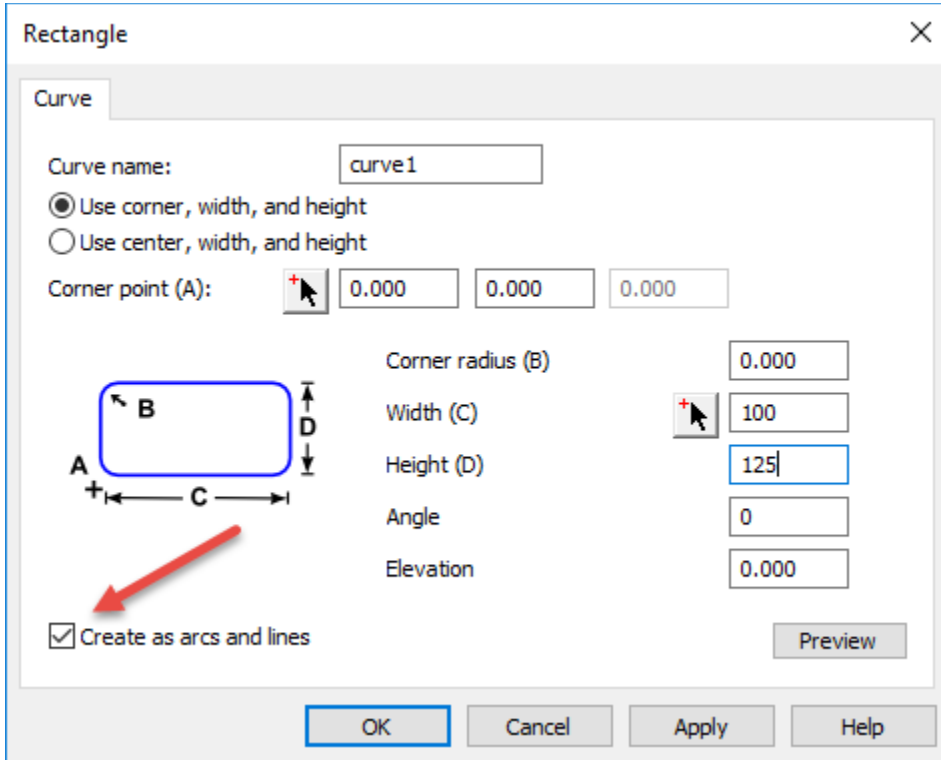
- 5 We will create a **Rectangle** first, go to **Construct>Curve>Other Methods>Rectangle**.




- 6 Enter the following values **Width 100mm Height 125mm**.

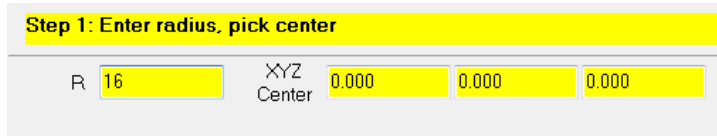


*Remember to select **Create as arcs and Lines**. See image below.*



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

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

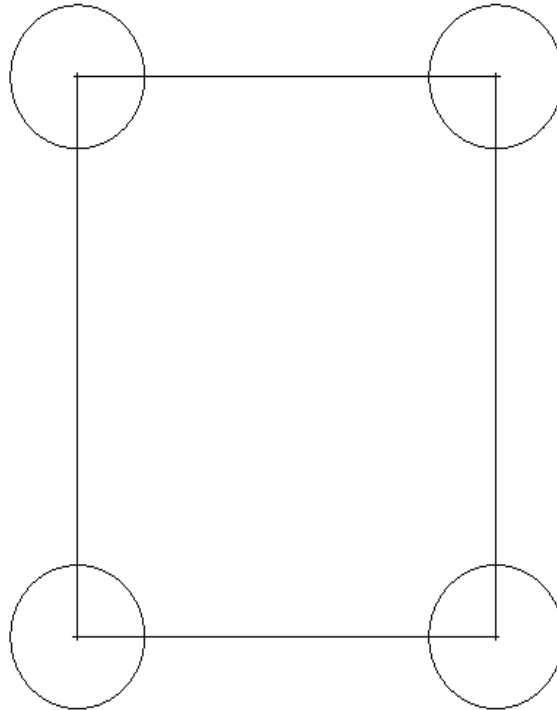




Please note that all Snapping mode functions can be switched on and off at any time. All of the snapping modes can be switched on together or used individually.

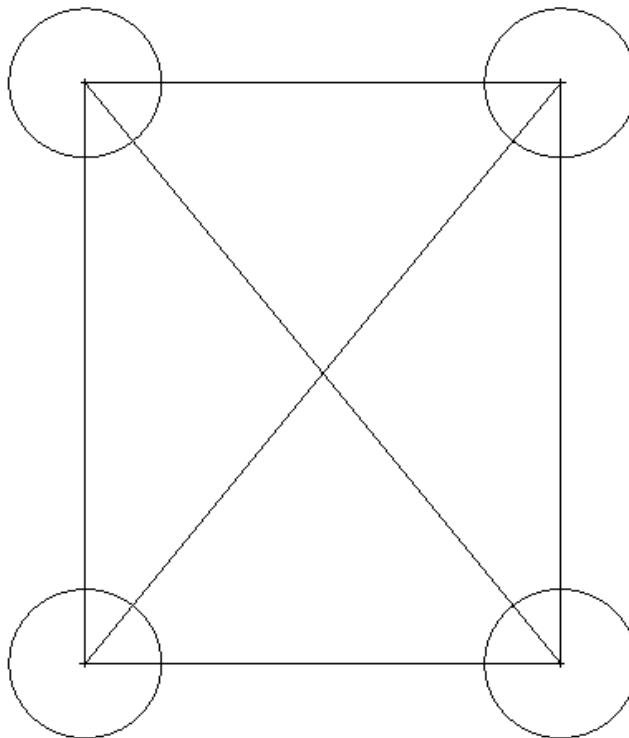
- 8 We will use **Snap to End Point**  as the most efficient snap mode in this instance. **Remember to snap to the four corners of the rectangle.**




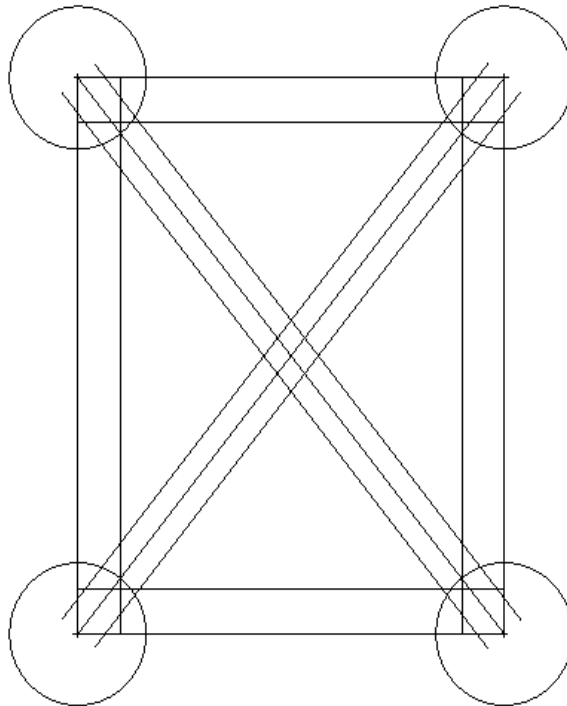
Your image should look like the one on the next page.




- 9 We will now draw two lines across the diagonally opposite corners.
- 10 Select **Line from 2 Points**  from the **Geometry** Toolbar. Use **Snap to End Point**  as the most efficient snap mode in this instance.

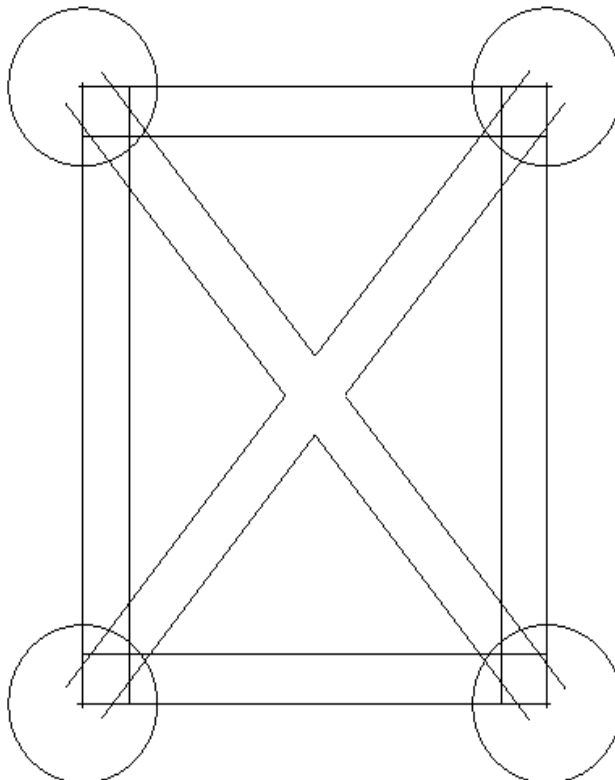


- 11 Using  Offset we will offset the outer lines only **10mm** towards the centre. Then whilst the Offset command is still active. Offset the diagonal lines **5mm** either side of the lines as shown on the next page.




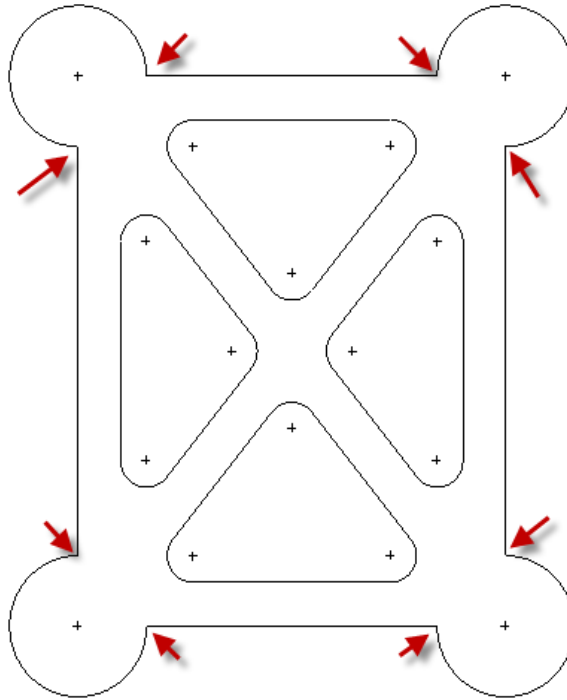
12 Delete the 2 centre diagonal lines. To do this just select the lines and then

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

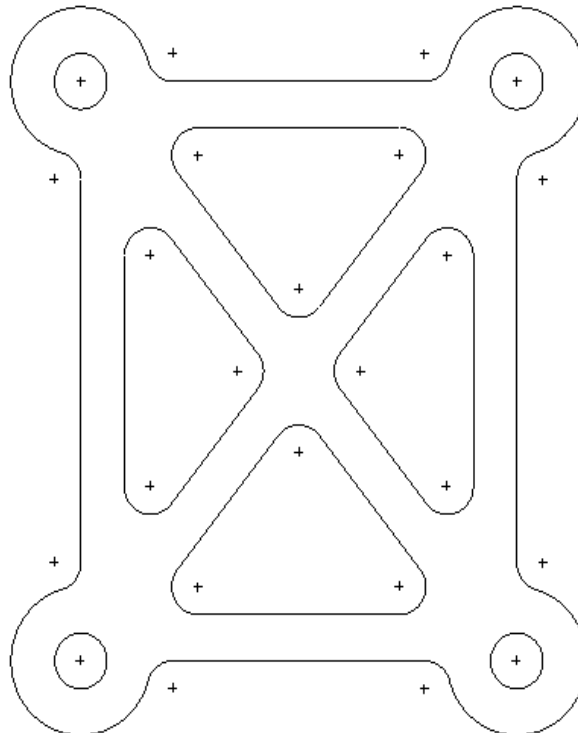


We will now **Fillet** all pocket corners **6mm**

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



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



If in doubt ask.

Dimensioning the part (Class Exercise)



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

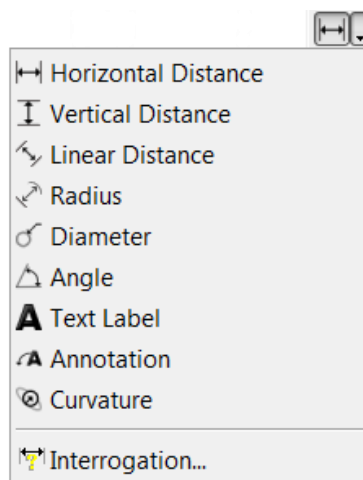
- 1 Next Select **Dimension** from the **Construct** menu, or the dimensioning flyout menu from the geometry toolbar. We will have a go at the different commands for example, **Horizontal Distance**, **Vertical Distance** etc.



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



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



Horizontal

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

Vertical

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

Linear

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

Radius

Creates dimension information for the radius of the selected object.

Diameter

Creates dimension information for the diameter of the selected object.

Angle

Creates dimension information for the angle between two selected lines.

Text label

creates a text label entered in the dialogue bar

Annotation

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


Curvature

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

Interrogation

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

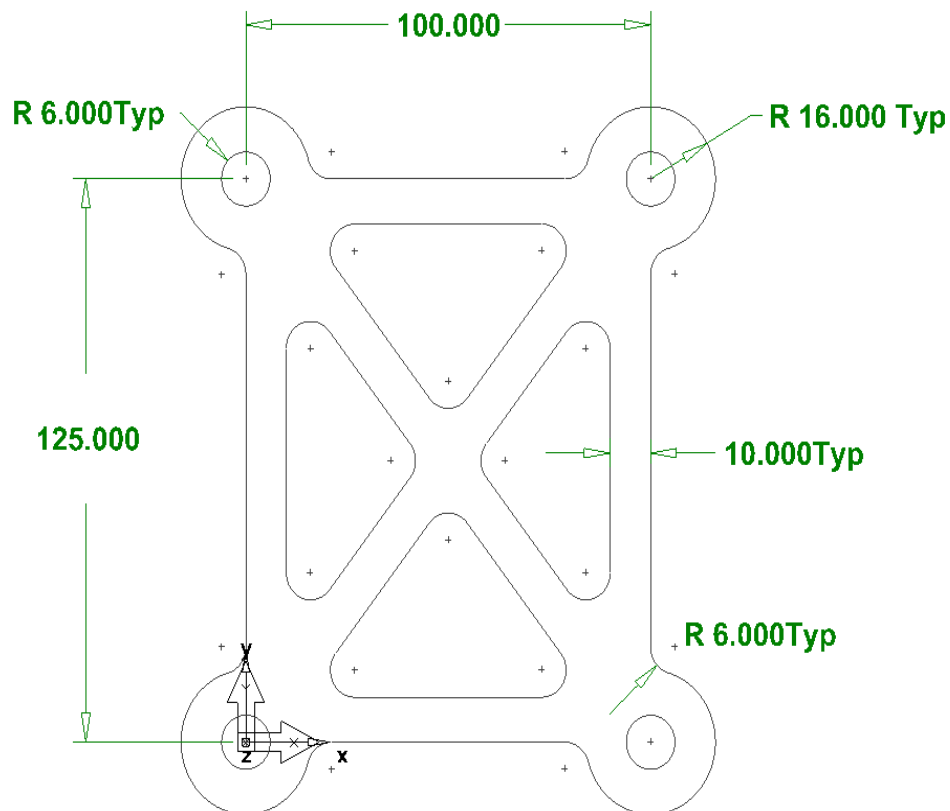


*When creating dimensions. Do not worry if you make a mistake. If you do, just select  undo or select **Ctrl + Z** then have another try.*

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

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

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



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

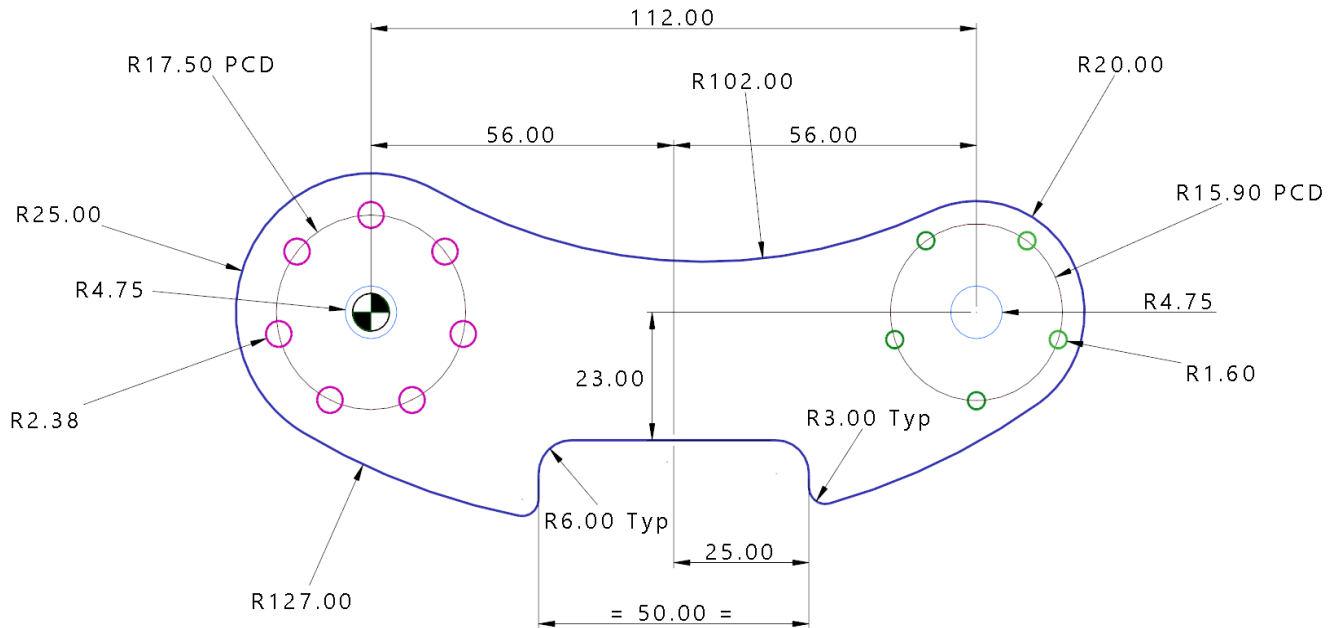
Construction Exercise 2



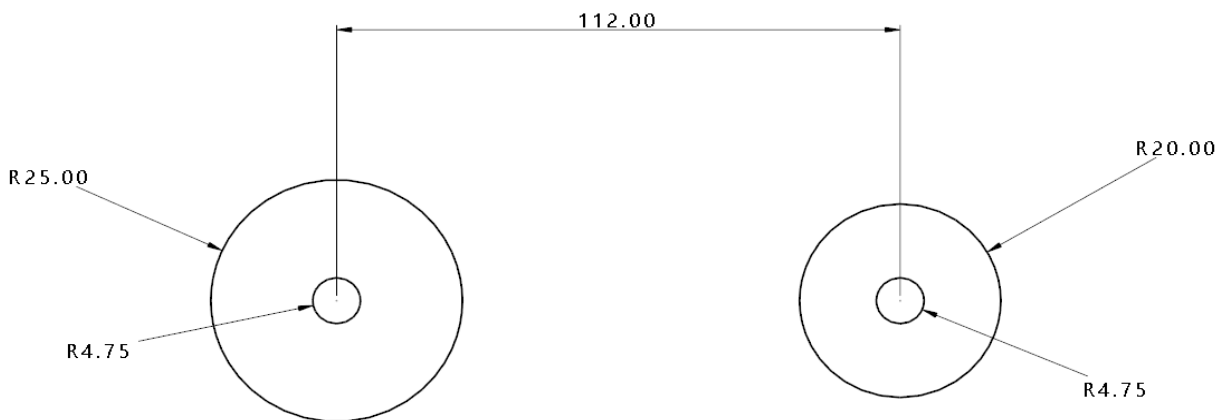
There are additional optional exercises that follow. This will give you more practice drawing and using tools you have learned up to this point.



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



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



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

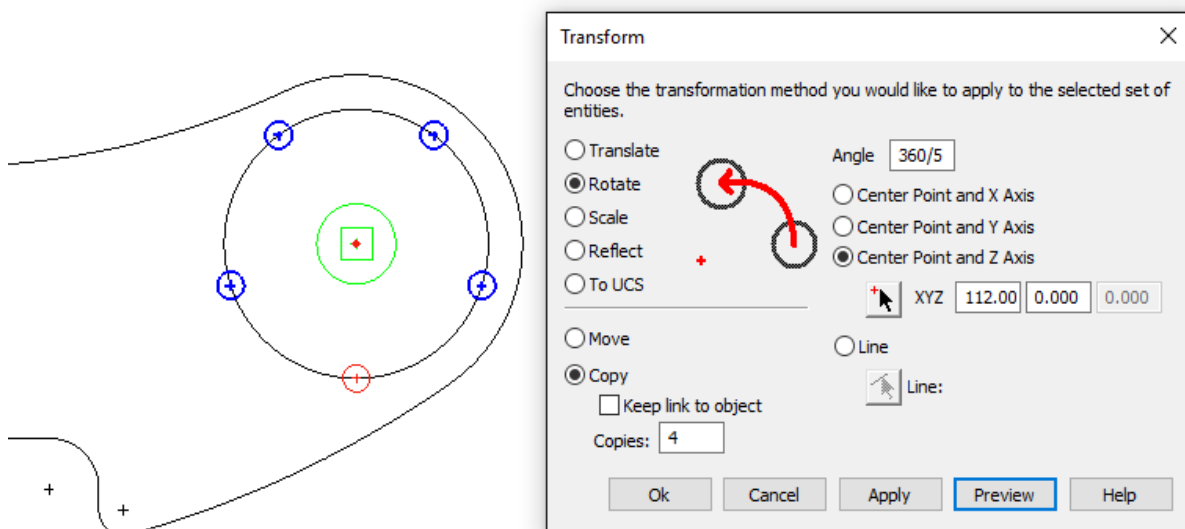
Additional machining information


- Product thickness is **25mm**
- **9.5mm** Diameter holes are through
- **4.75mm** Diameter and **3.2mm** Diameter holes are **through**.

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

Using Transform

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



- 2 Open the **Transform** dialog in one of these ways:
- 3 Select at least one object and click the **Transform** button in the **Standard**
- 4 Toolbar.  Or select **Edit > Transform** from the menu.
- 5 Or right-click on an object and select **Transform** from the context menu.
- 6 You can transform **Geometry, Curves, Features, Surfaces,** or **Solids**.



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



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



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



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

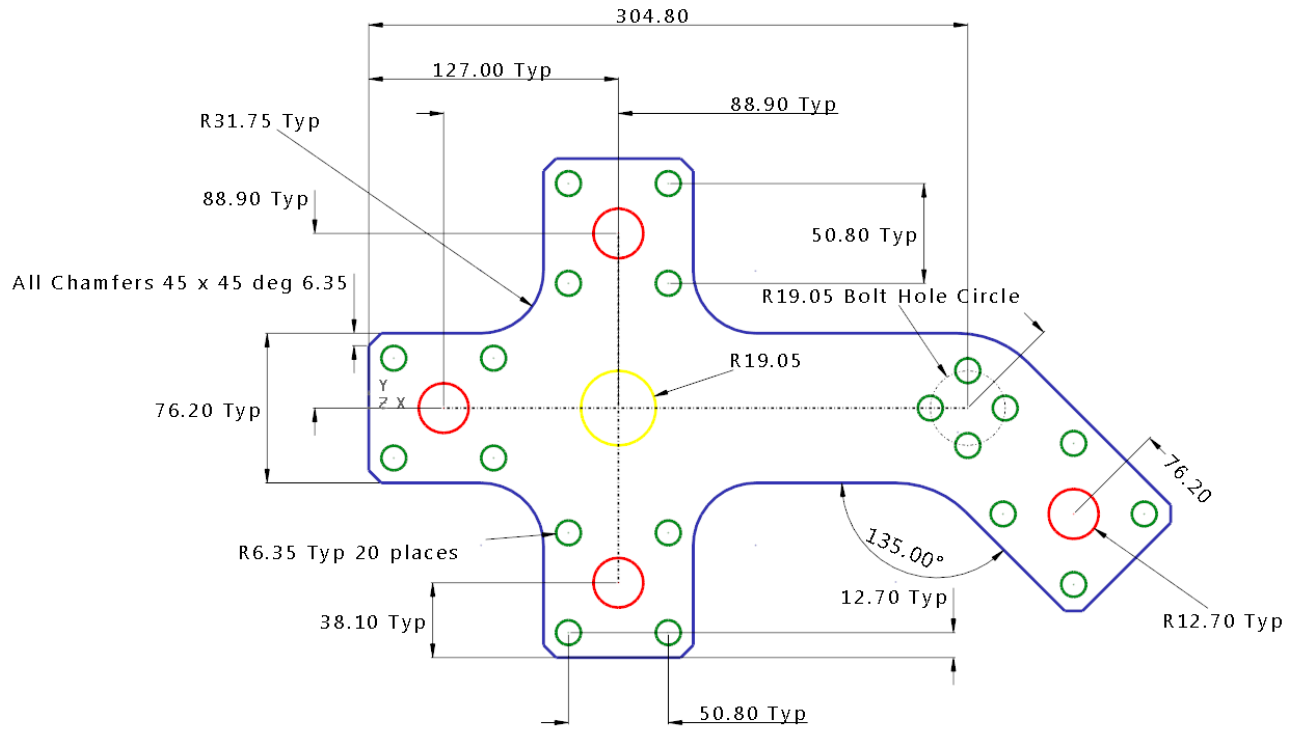


Reflect mirrors the element about a line. The line can be an existing axis, or any other line including one created just for reflecting around. The object can be flipped top to bottom, left to right or even both depending on the line you choose to reflect around. Set whether you want to **Move** or **Copy**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



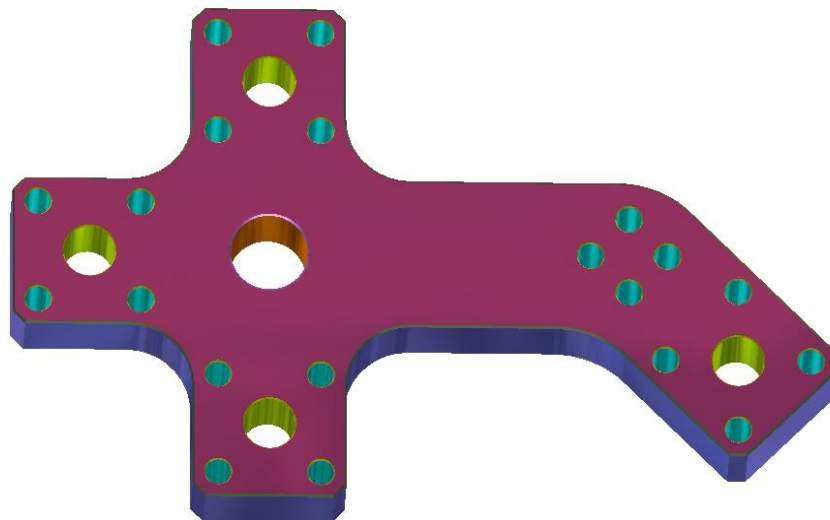
There are additional optional exercises that follow. This will give you more practice drawing and using tools you have learned up to this point.

Construction Exercise 3

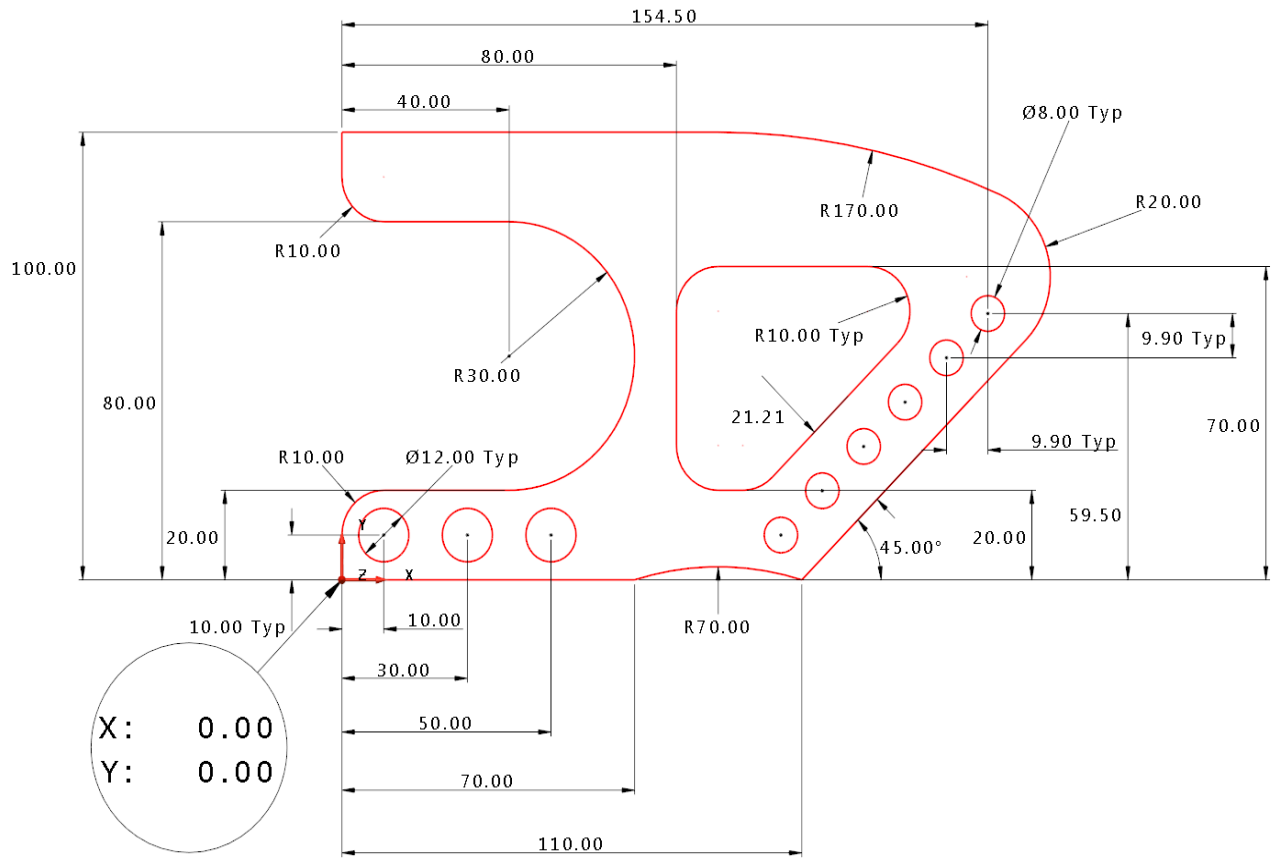


Additional machining information

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



Construction Exercise 4



Additional machining information

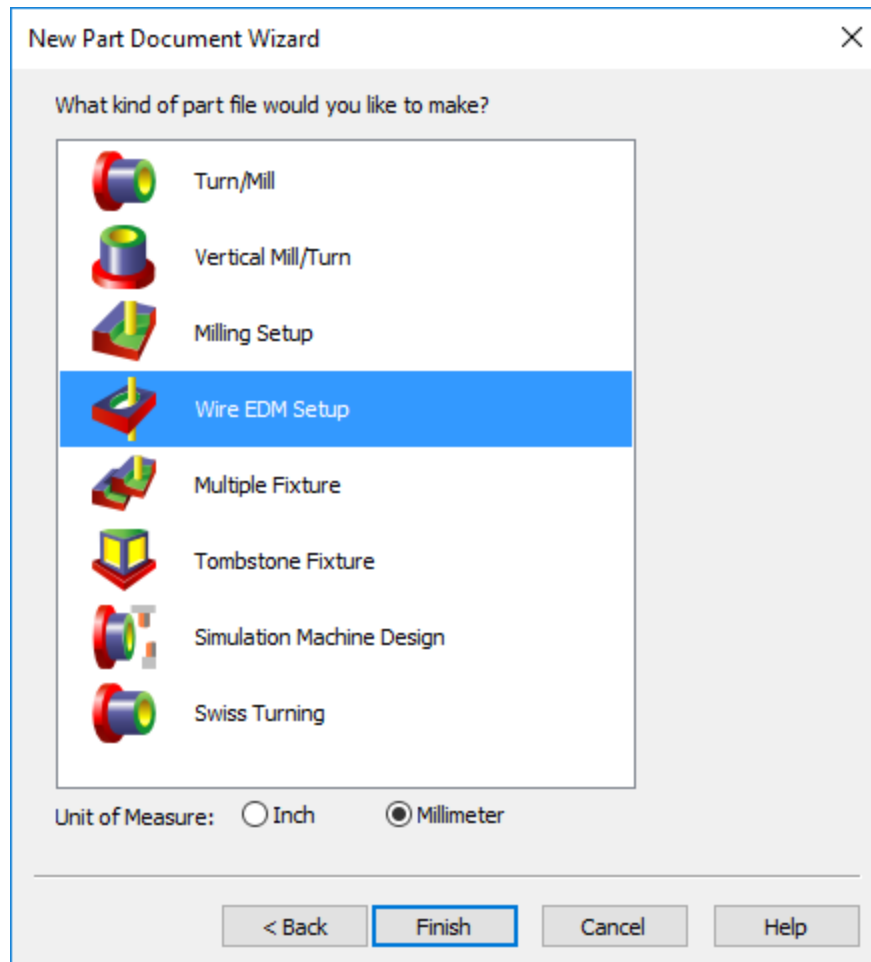
- Product thickness is **45mm**
- All Holes are through.

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First part exercise – Simple 2-Axis

- 1 From the **New Part Document** Wizard Select **New file** then select **Next** then select **Wire EDM Setup**, Select **Millimeter** as **unit of measure** then select 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.

Stock Dimensions Dialog

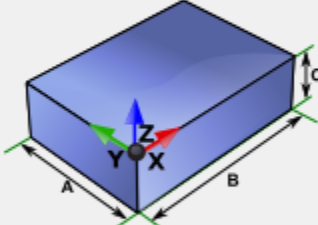
- 2 In the **stock dimensions** dialogue, Select **Block**, Enter **150mm** for the **Width**, **150mm** for the **Length** and **25mm** for the **Thickness**.
- 3 Select **Next**.

Dimensions

What shape is the stock?

Block
 Round
 N-sided

Enter the dimensions of the stock:



Width (A)
 Length (B)
 Thickness (C)

4 Select **Next**.

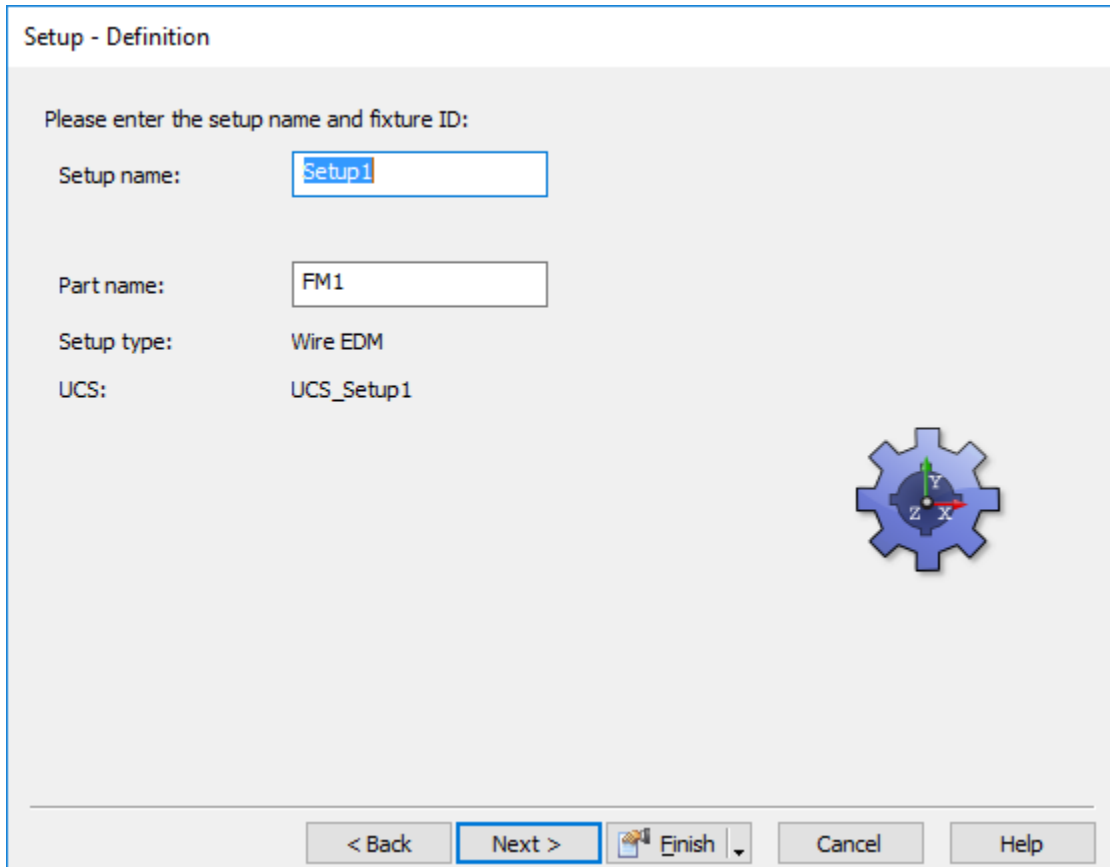
Stock - Condition

What is the cutting condition for the part?

Material:	<input type="text" value="ALUMINUM"/>		Machine:	<input type="text" value="Fanuc"/>	
Wire:	<input type="text" value="Generic"/>		Nozzle position:	<input type="text"/>	
Wire diameter:	<input type="text" value="0.250 mm"/>		Fluid:	<input type="text"/>	

Stock - Condition dialog

- The material, wire type, wire diameter and machine type for an EDM part is specified using the Stock Properties dialog.
- To enter these values, select each field using the drop down list and make your selection, if you wish to change any category Click the Cutting Data button and enter the desired values. The Condition button also may be accessed from the Stock Properties. The actual cutting conditions are on the machine and the cutting data in the feature is merely a call for that condition.



Setup - Definition

Please enter the setup name and fixture ID:

Setup name:

Part name:

Setup type: Wire EDM

UCS: UCS_Setup1

< Back Next > Finish Cancel Help

- 5 Select **Next**.

Setup Dialog

- 6 Enter Setup Name, you may call it whatever you wish, FeatureCAM defaults to Setup 1 and in sequence. Additional Setups may be designated for other sides on the part. Use whatever Part Name you desire.
- 7 Click **Next**

Part Program Zero Dialog

- 8 Align your Part Program Zero (UCS User Coordinate System) by **Clicking** on the desired selection available. (See image on the next page)

Setup - Part Program Zero

What method do you want to use to define the part program zero location for Setup1?

Align to Stock Face

Align to Index axis

Align with existing UCS

Align to part geometry

Use current location

< Back **Next >** Finish ▾ Cancel Help

- 9 Select **Next**.

Setup - Posting Dialog

- 10 Enter the desired values for your machine and part location. This is shown on the next page.

Setup - Posting

Please enter the following information for the posting of the current setup:

Machine Z-zero is offset from the setup Z-zero by:

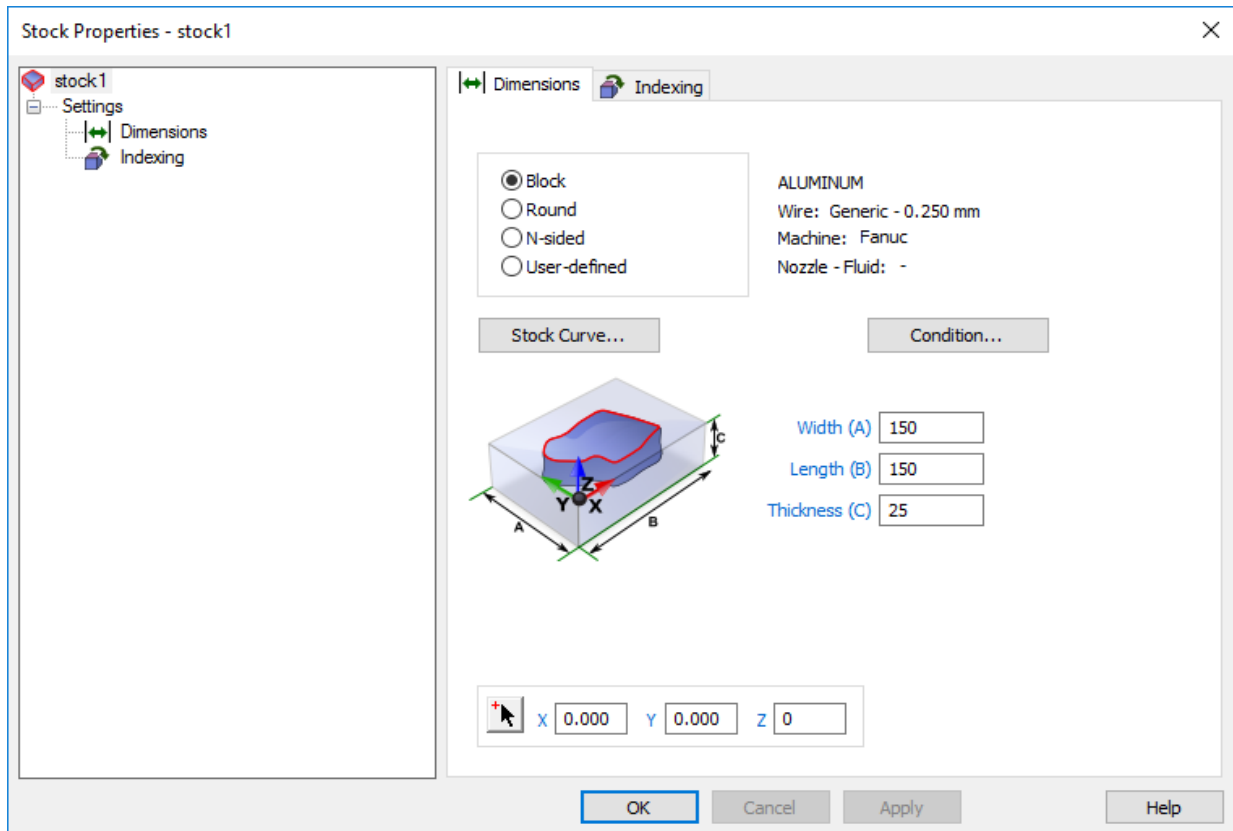
Generate 4 axis toolpath at the wire guide planes

Upper guide on machine (relative to machine Z-zero):

Lower guide on machine (relative to machine Z-zero):

< **Back** Next > Finish ▾ Cancel Help

11 Click **Finish**.




12 Select **OK** to close the form.

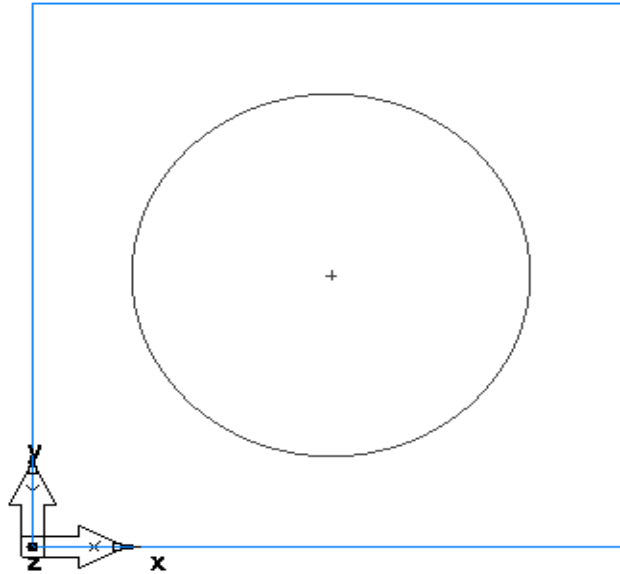


You now have a Block of Stock of D2 Material, using Hard Brass Wire with a 0.250mm Diameter and the machine is a Fanuc which contains the Condition Values.

13 Click on the Geometry Button from Steps and using the Geometry constructors

14 **Select**  Circle from radius, center.

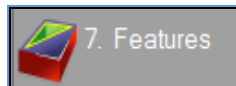
15 **Enter** Radius of **50mm** and X= **75mm**, Y=**75mm**, Z=**0**.



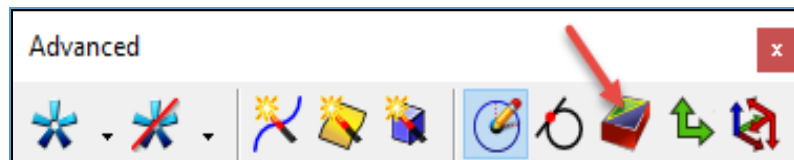
To activate a **new Feature menu** select from the following options.



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

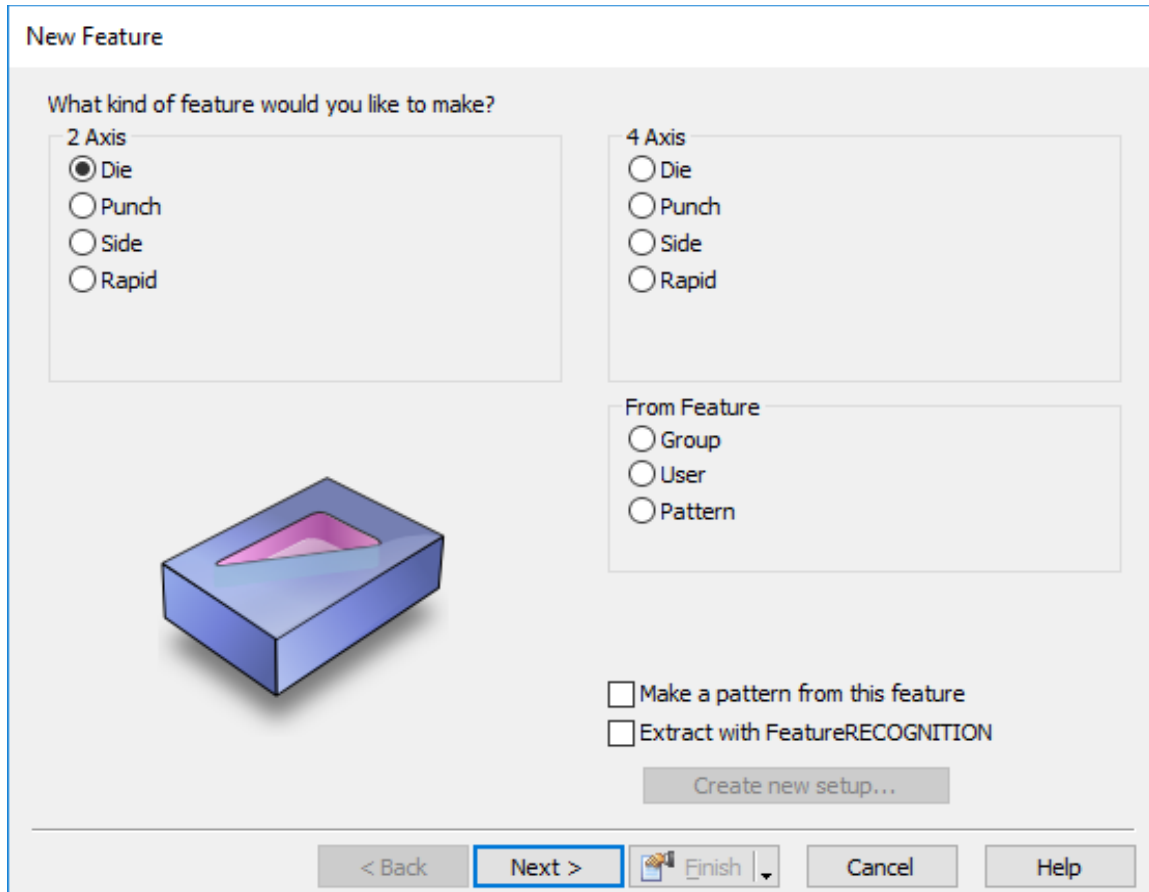


- Select **Features** from the **Steps** menu.
- Select **Ctrl + R New Feature**.
- Select **Construct>New Feature**.
- Select **New Feature** from the **Advanced Toolbar**.

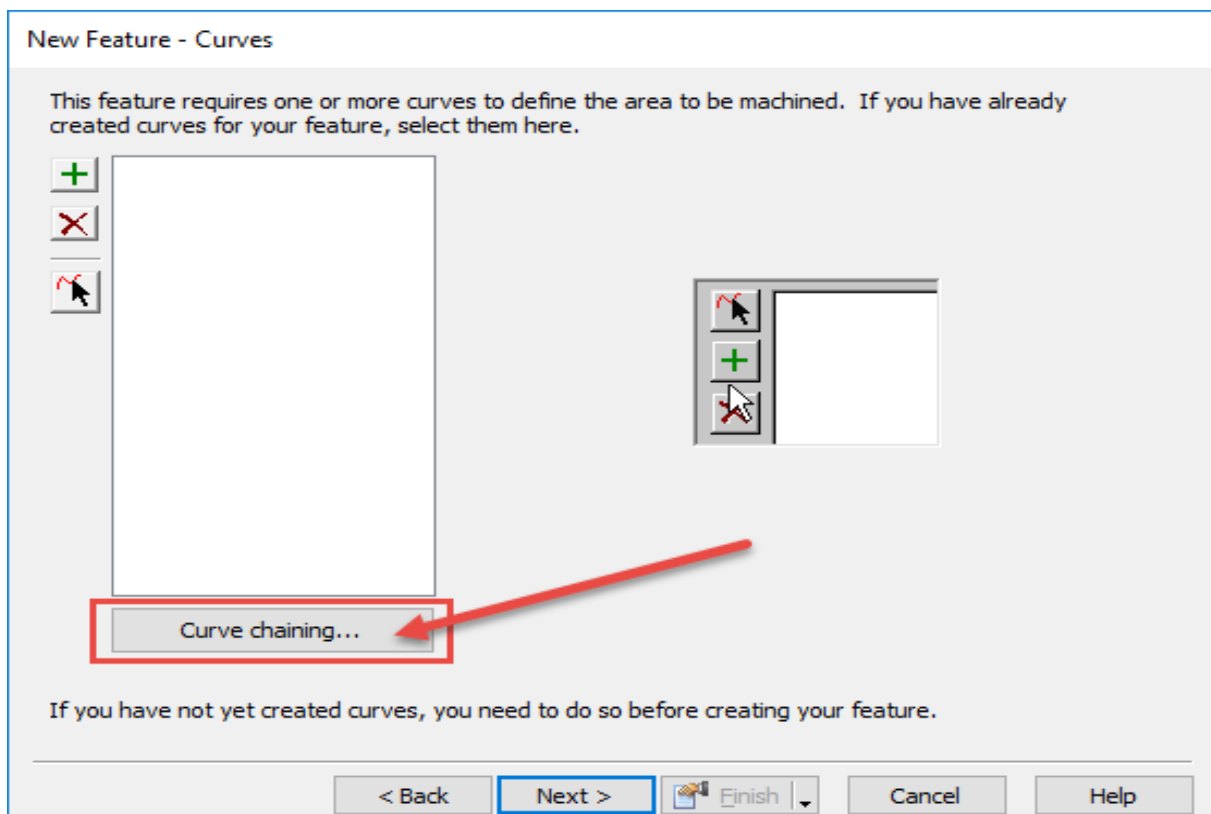


16 Choose the option of choice to create a new Feature. The following menu will appear.

17 Select **Die** under **2 Axis**. Click **Next**.



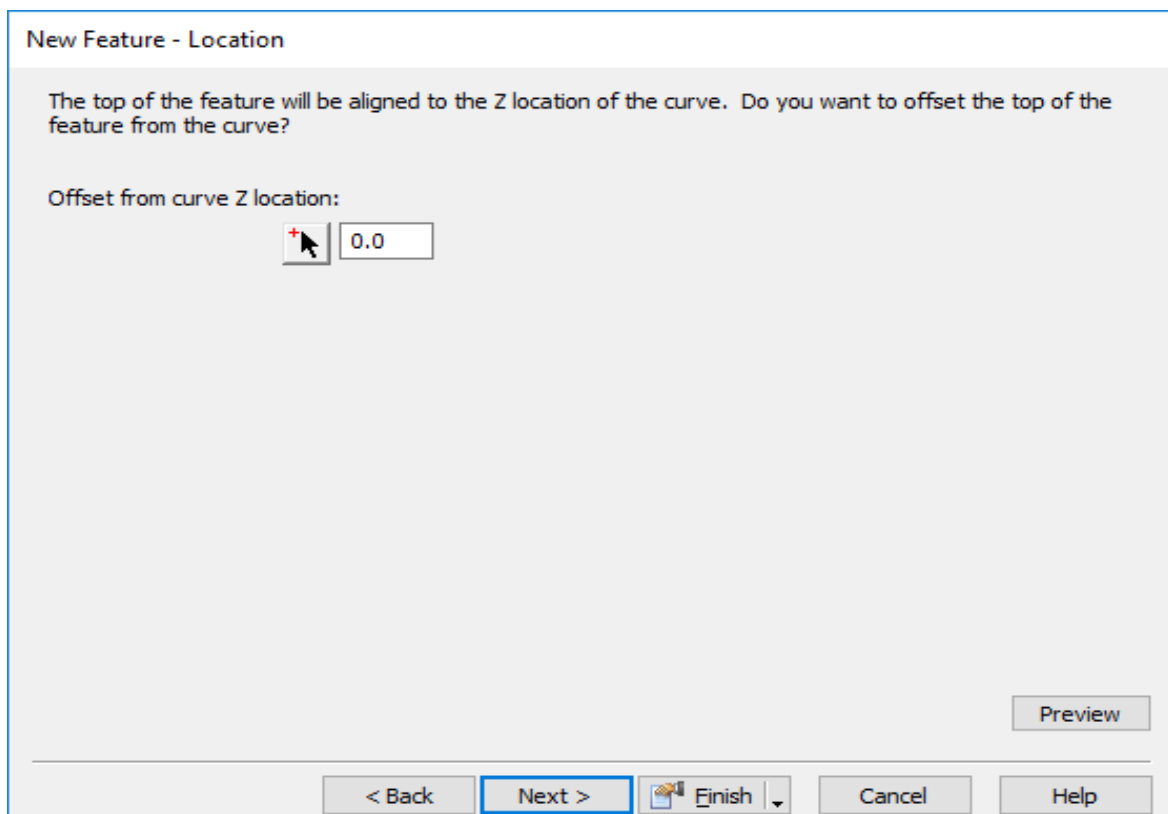
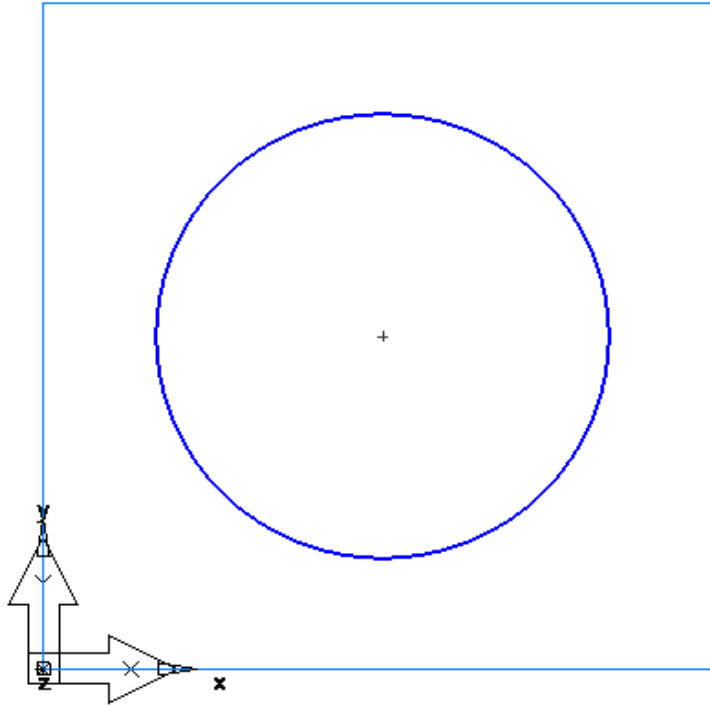
18 Select **Next**.

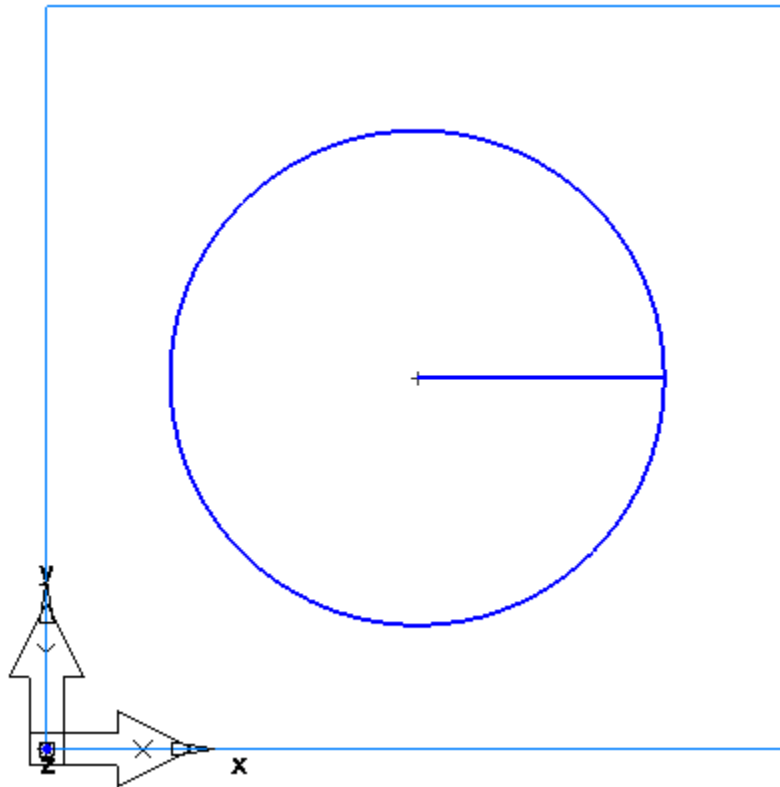




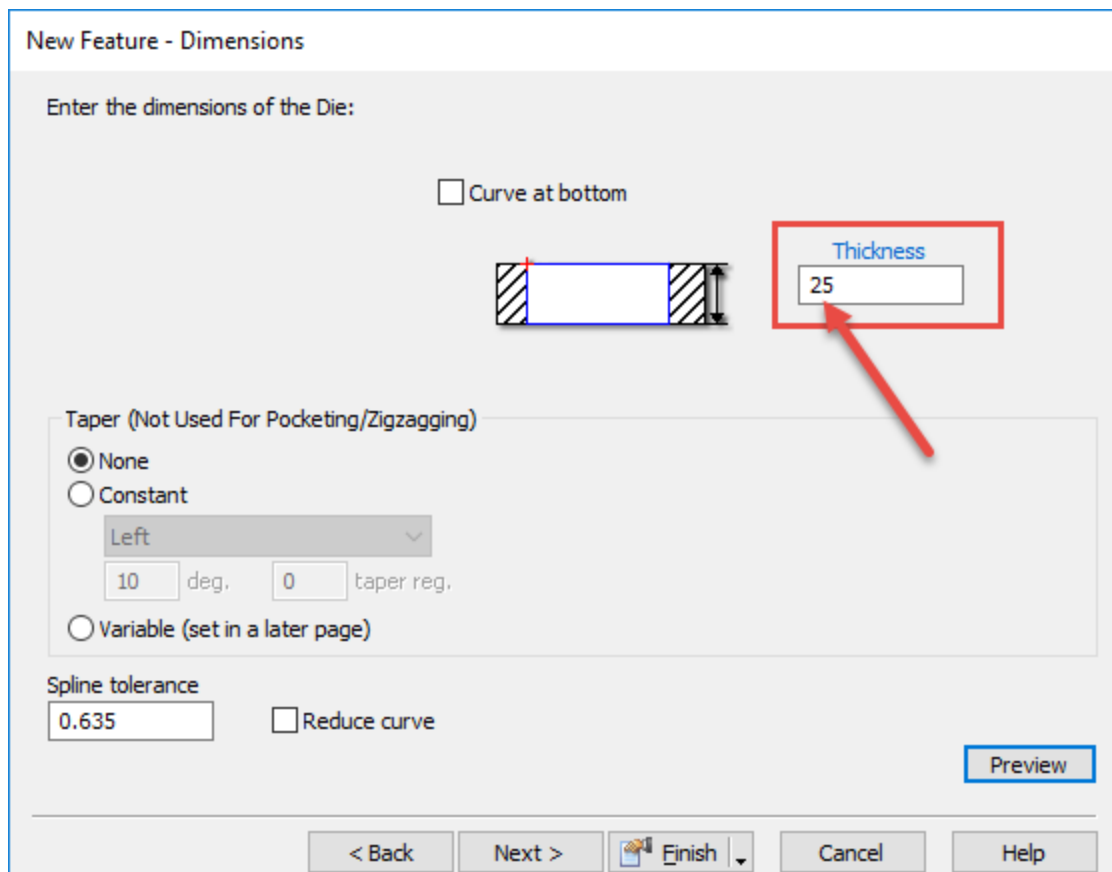
We need to create a Curve.

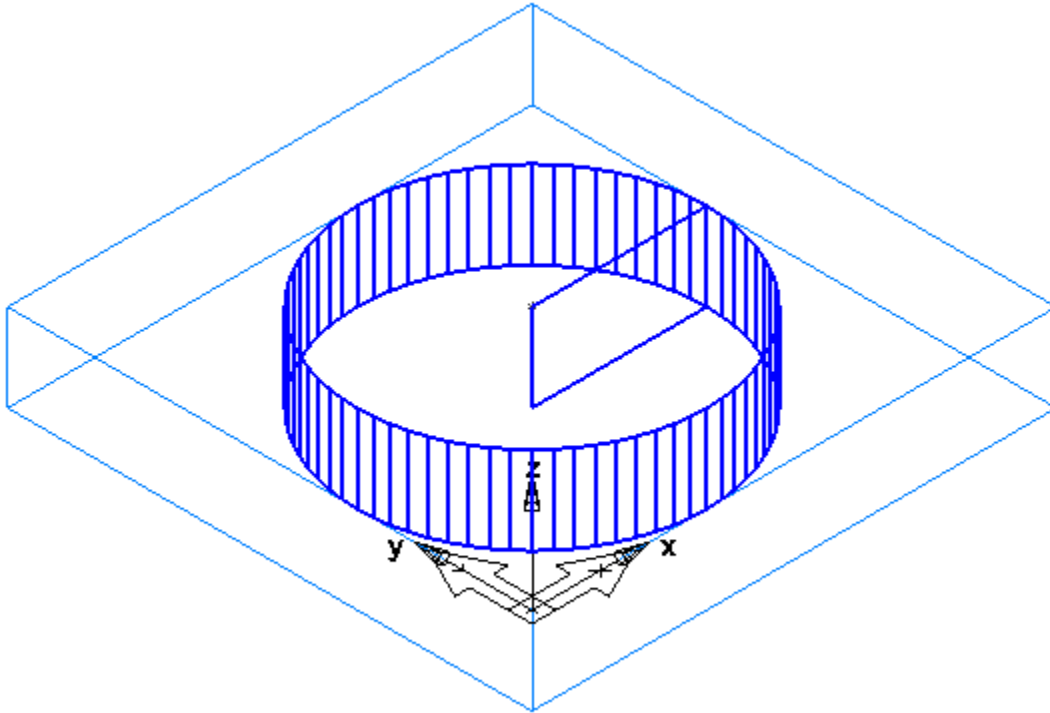
- 19 **Select Curve chaining** from the menu.
- 20 **Select the Geometry** as shown. **Select Next**. The following menu will appear.



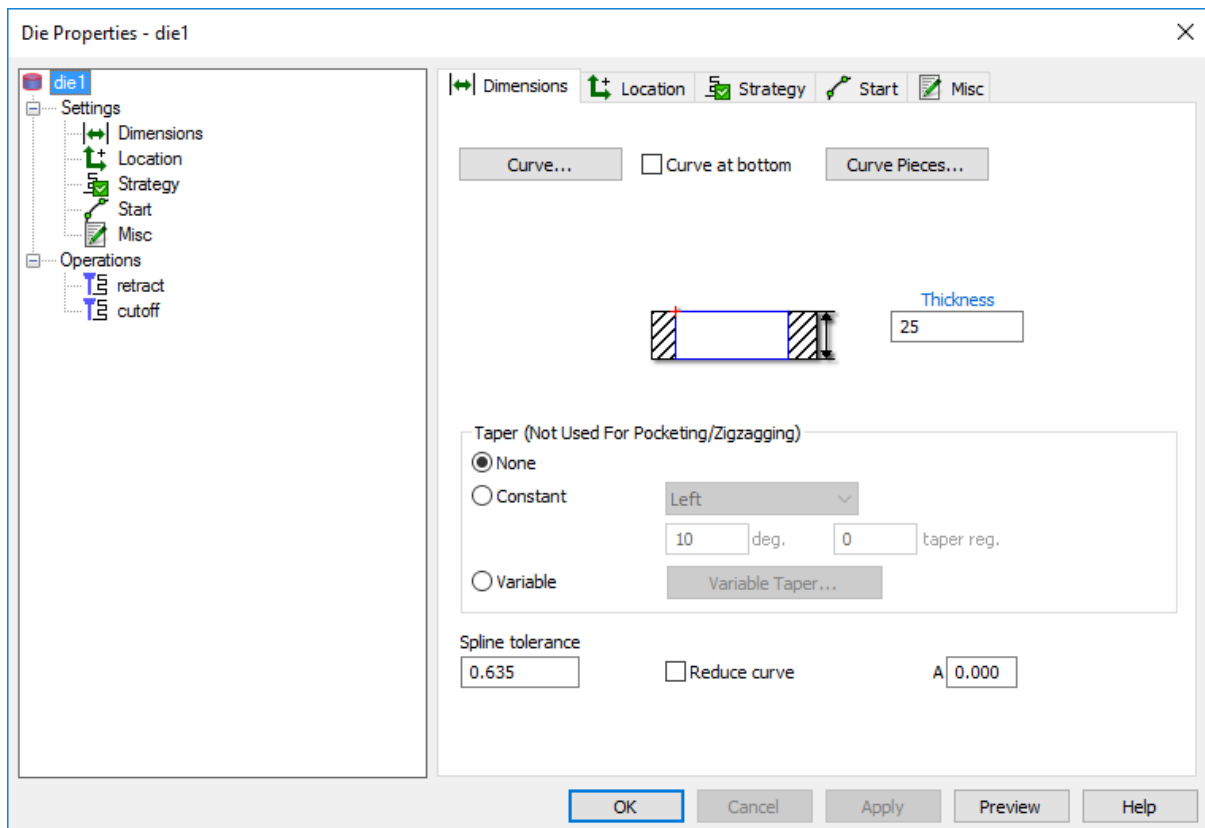


21 Select **Next**. Change the thickness to **25mm**



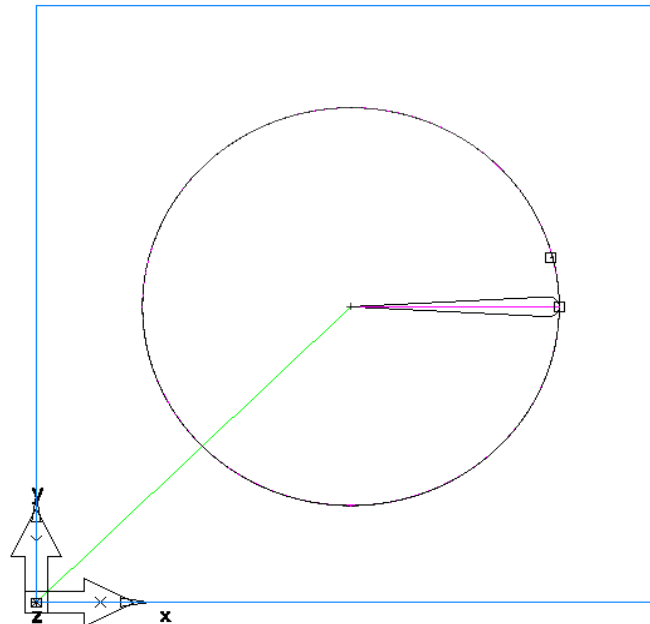
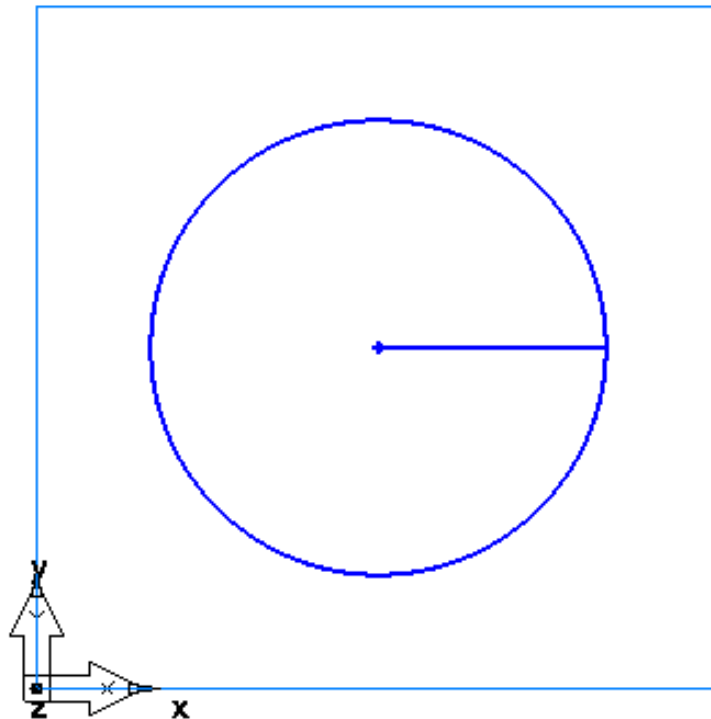


22 Select **Finish**. The following menu will appear.



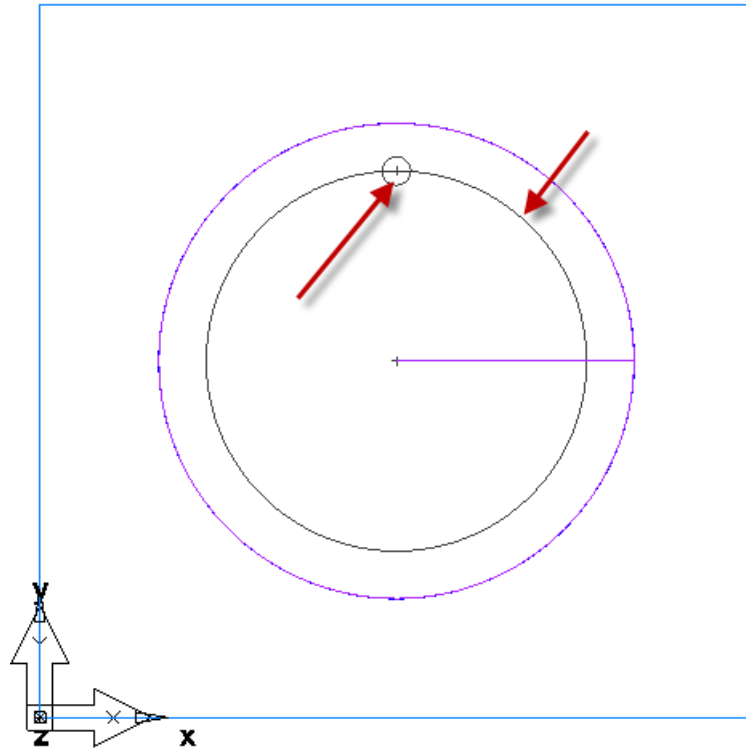
23 Select **Ok** to close the form.

24 Run a Centreline simulation.



There are a couple of ways to change the Start point in the feature.

- 25 First of all **hide** the **die Feature** in **Part View**.
- 26 Select **Offset** from the geometry menu and offset the Circle inwards by **10mm**
- 27 Create a **3mm** Radius Circle. Snap to a quadrant.

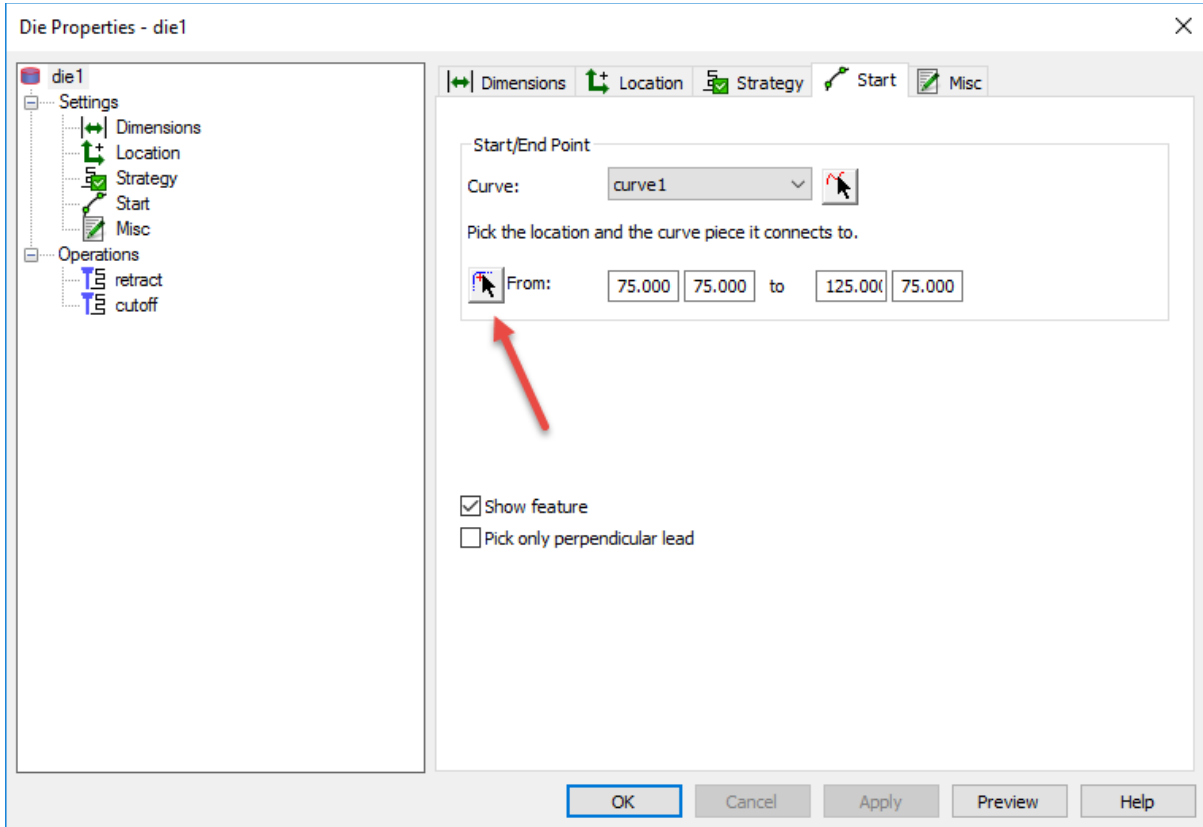


- 28 To change the location of the start point
- 29 **Double-click** the die feature in the **Part View** area of the **Toolbox** found at the left side of FeatureCAM's User Interface.

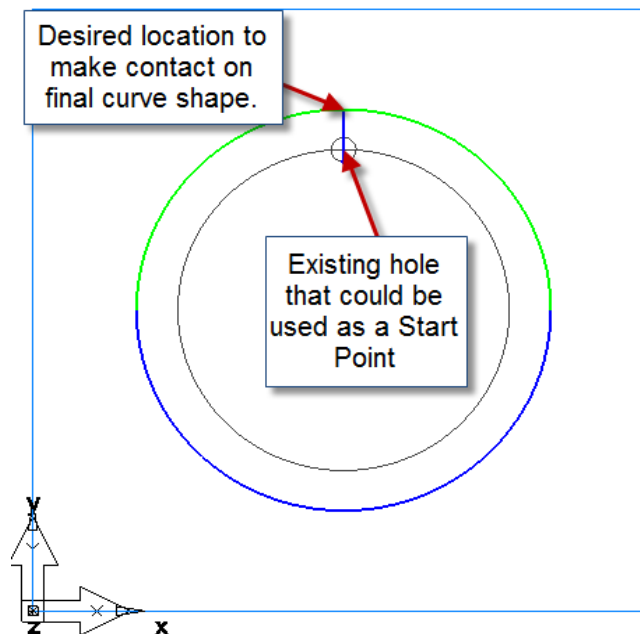


Double clicking opens the feature's properties.

- 30 Select the **Start** tab



- 31 Use the **Pick Arrow** to pick the start point, in this case the **Centre of the circle** we used to create the feature, then select along the circle where you want the wire to start cutting the curve shape.

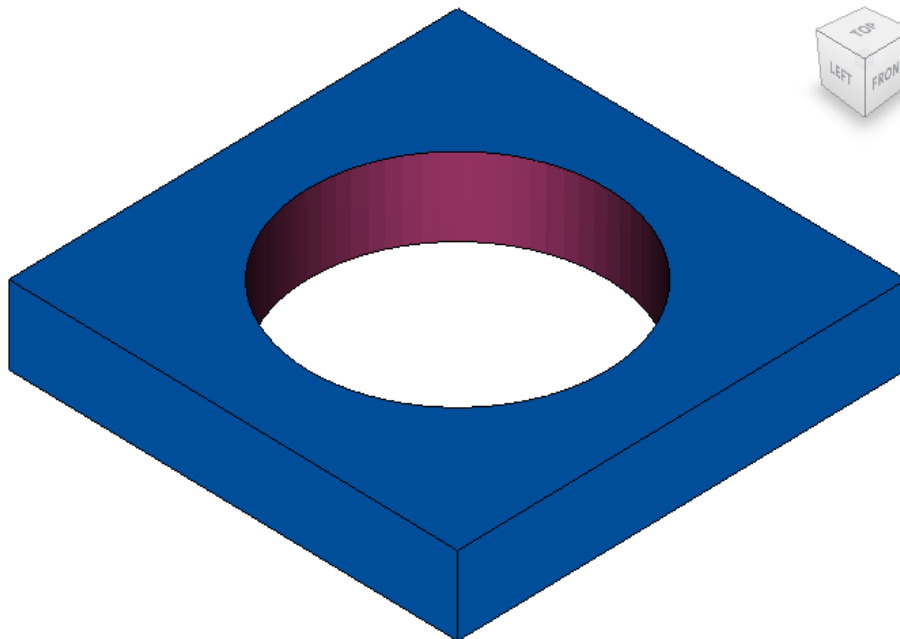
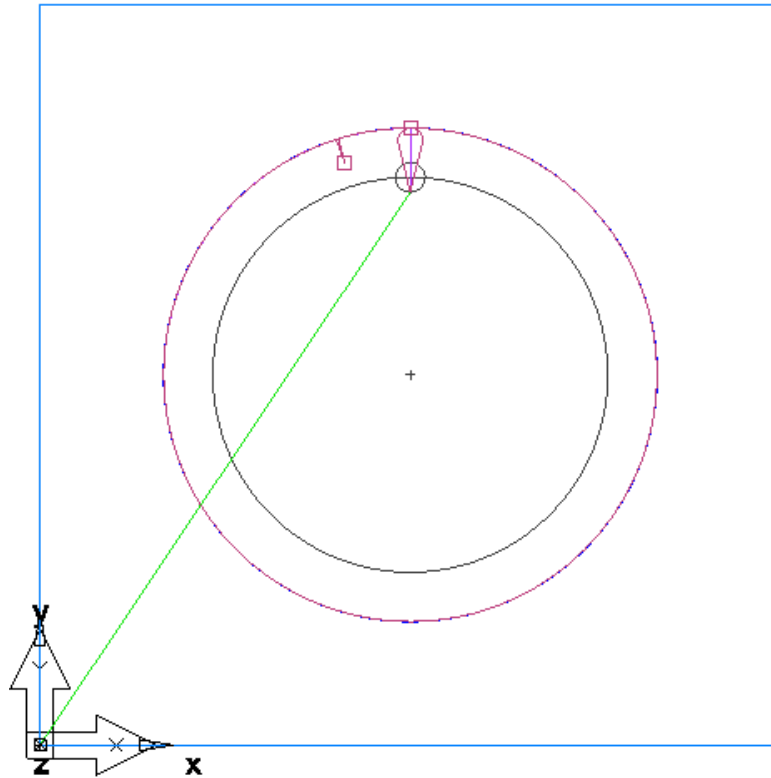


Or simply **enter** the dimensions for the start of the Wire path a known location along the curve shape. (Not Shown)



Since Wire EDM processes can take a lot of time to cut the part it is wise to shorten the distance travelled by the wire, whenever possible.

An example of this is if you want to use an existing hole in the stock where the start point is closer in location to the curve shape.



More 2 Axis Features



There are four types of 2 Axis features to select from in a Wire part program. We have seen a simple 2 Axis Die feature so a quick review of the other types is appropriate.

New Feature

What kind of feature would you like to make?

2 Axis

Die

Punch

Side

Rapid

4 Axis

Die

Punch

Side

Rapid

From Feature

Group

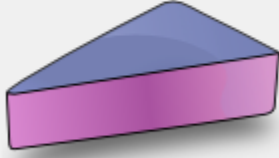
User

Pattern

Make a pattern from this feature

Extract with FeatureRECOGNITION

Create new setup...

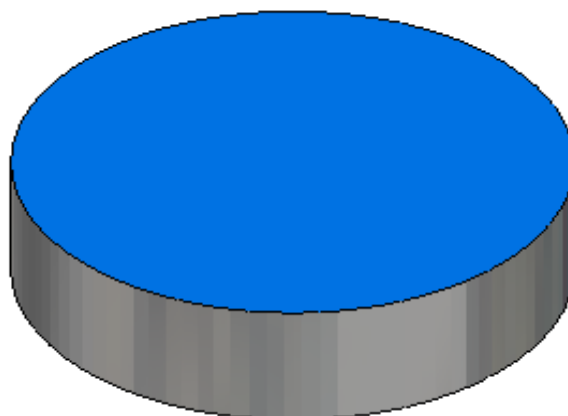


< Back **Next >** Finish Cancel Help

2 Axis Punch feature



The **Punch** feature requires one or more closed curves. The wire travels on the outside of the curve(s).





A 2-axis punch feature can create a number of cutting operations. These are specified on the **Strategy** tab. A 2-axis punch feature can have a taper.



Creating a punch feature is done following the same steps as we have done for the die feature.

Open the file in the **FeatureWIRE Metric data** folder named **2 Axis Punch.fm** C:\Training_Data\FeatureCAM Course Data 2017\FeatureWIRE Metric Data

- 1 In the **Steps** window of the **Toolbox** choose **Features** to create a new feature. Choose **Punch** as the feature type.

New Feature

What kind of feature would you like to make?

2 Axis

Die

Punch

Side

Rapid

4 Axis

Die

Punch

Side

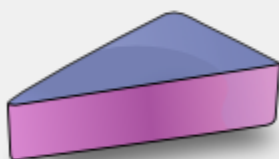
Rapid

From Feature

Group

User

Pattern



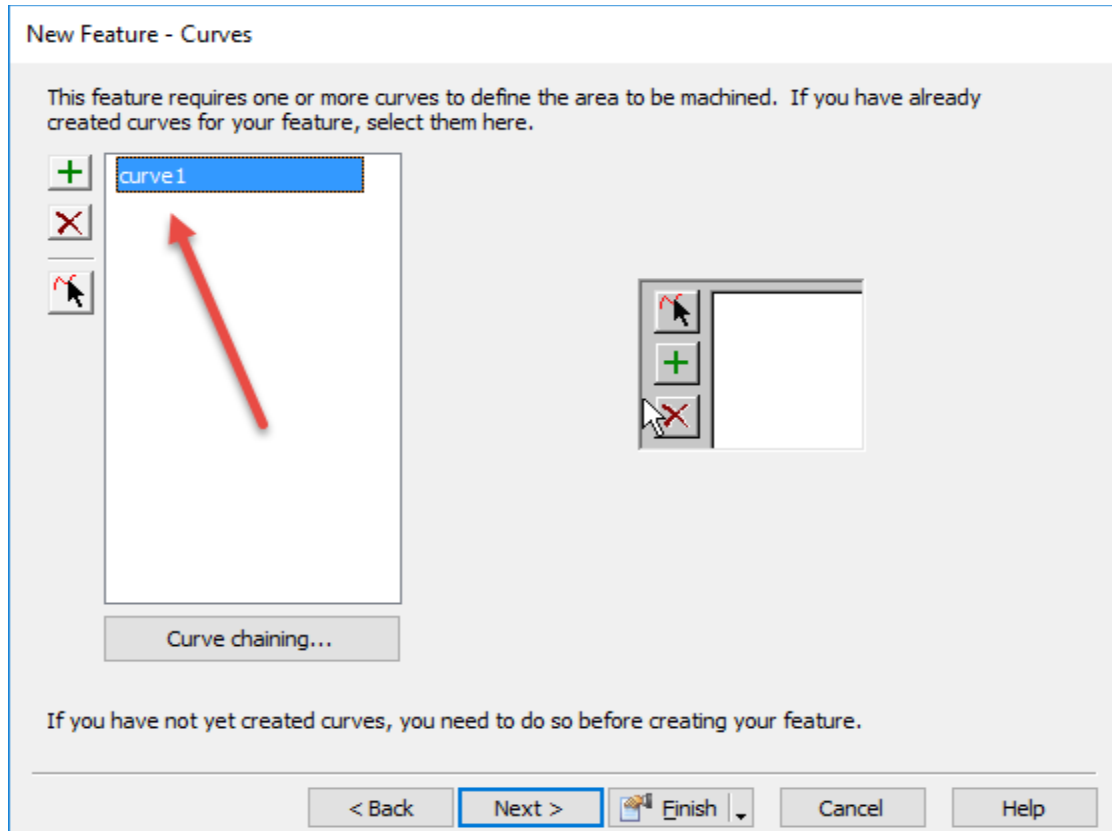
Make a pattern from this feature

Extract with FeatureRECOGNITION

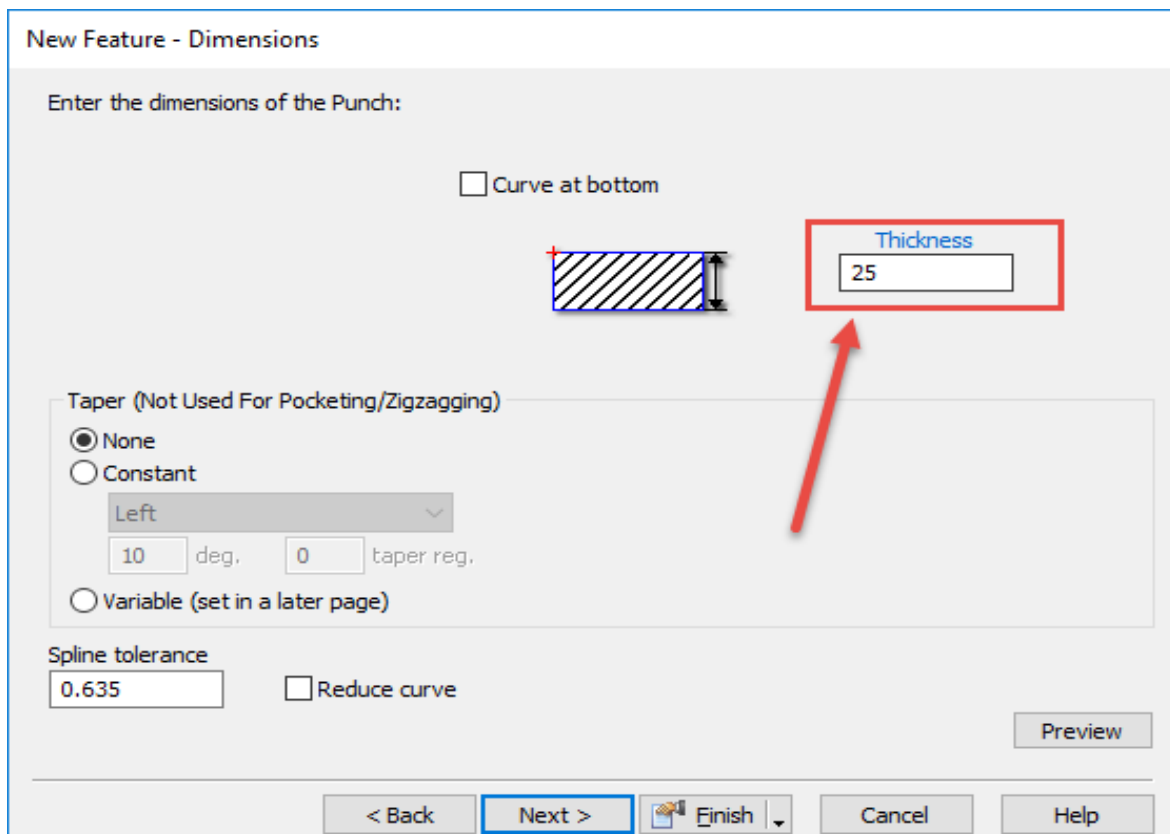
Create new setup...

< Back
Next >
Finish ▾
Cancel
Help

- 2 Select **Next**.
- 3 You may wish to hide **Geometry**. To do this go to **Hide** and **select Hide Geometry**.
- 4 Use the **Pick Arrow** and select **Curve1** from the graphics window.



- 5 Select **Next** Twice.
- 6 Set the **Thickness** to **25mm** and leave the other parameters as shown.



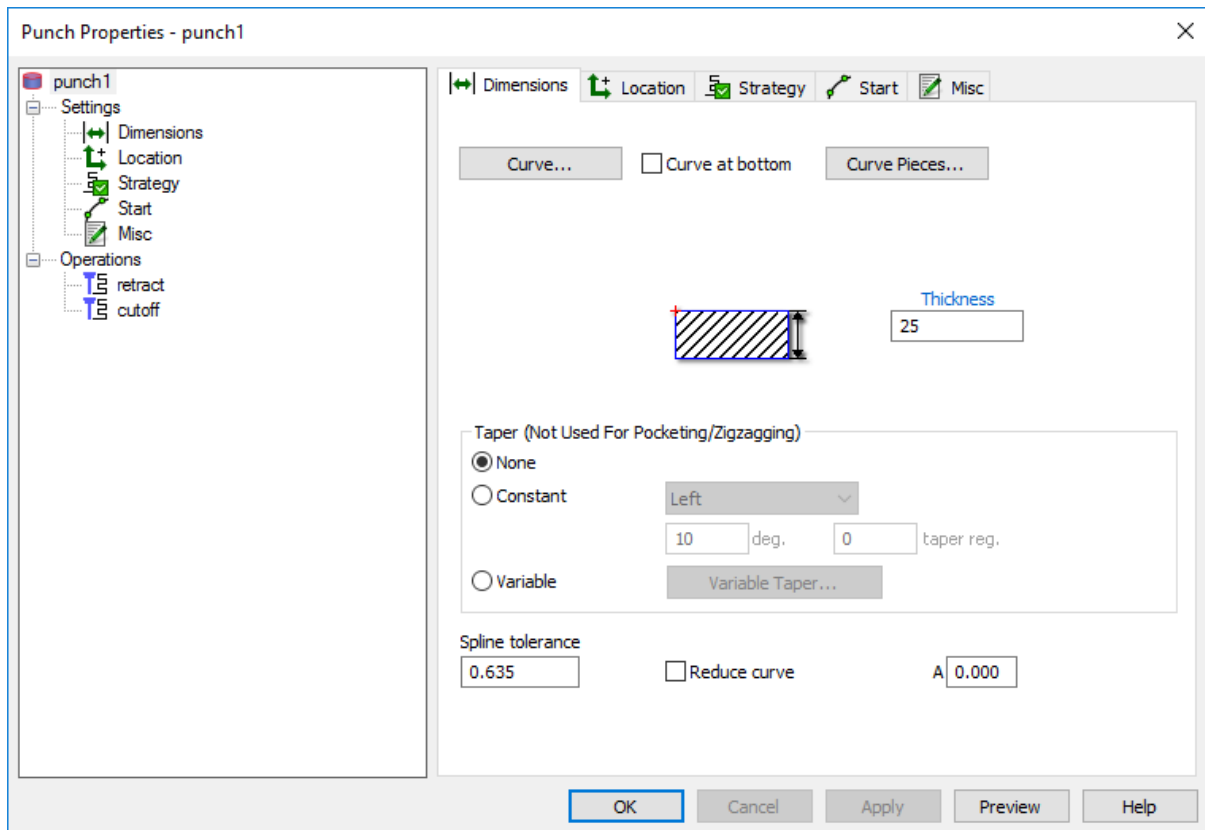
7 Select **Finish**.



Please note: The thickness parameter is used to access the appropriate cutting data table.

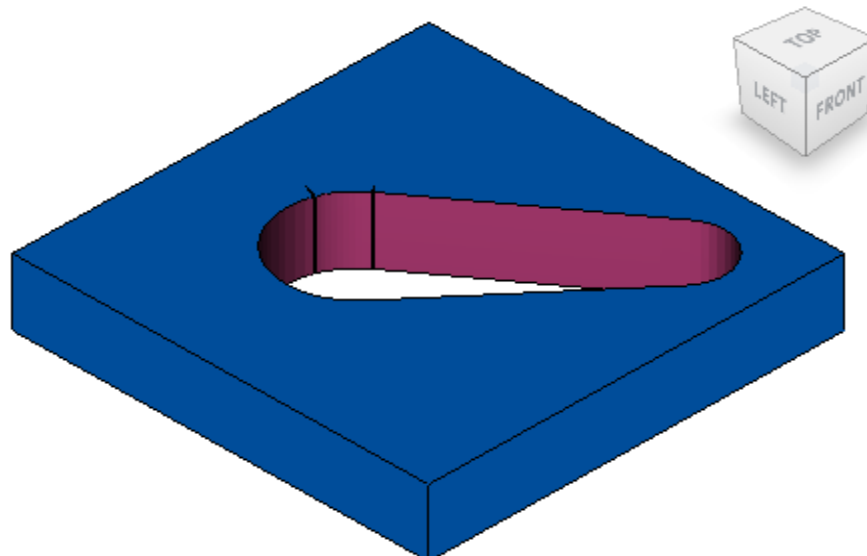


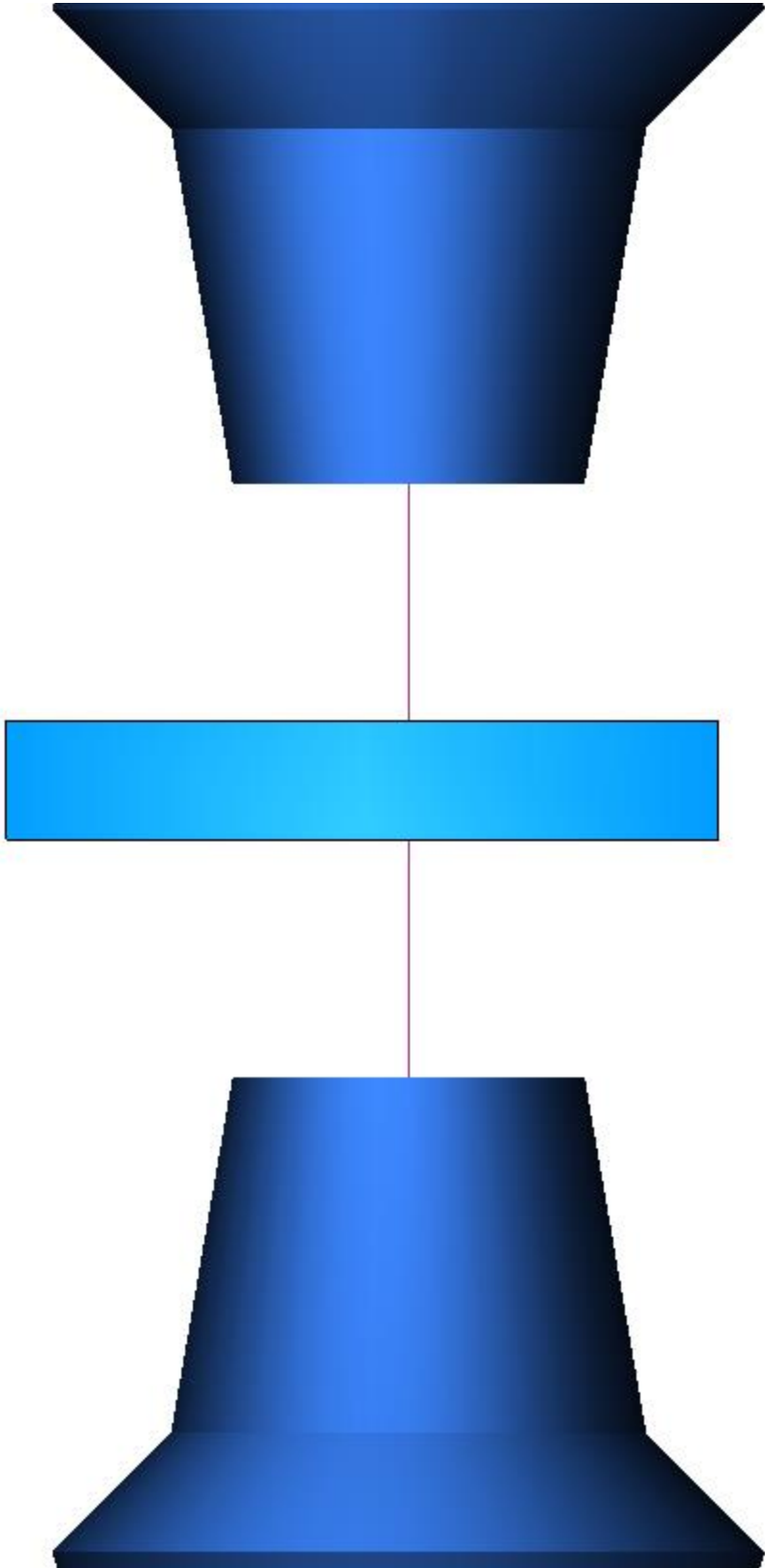
*The **Punch Properties** dialog opens to give you a chance to review the feature and make any changes you want to. This review of the properties happens each time you create a new feature.*



8 Press **OK** to **accept** the **feature dimensions**.

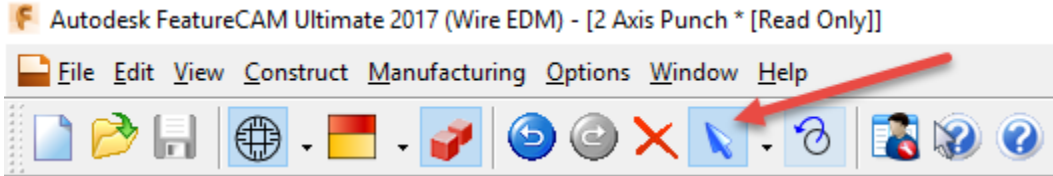
9 Run a **3D Simulation** to see the wire path in the graphics window.



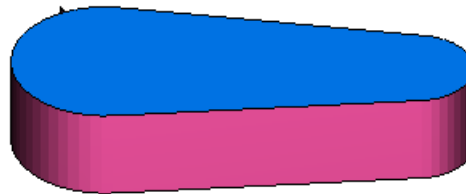




Notice you see the whole part still with the wire path through the 3D display of the stock and the feature. You can remove either the stock or the part from view by using the **Select Arrow** from the **Advanced Toolbar**. The select arrow is found under the word Options on the **Menu Bar**.



Click on the **Select Arrow** shown above and move your mouse cursor over the stock display in the graphics window.



10 Click on the Stock and it disappears from view.



You can alternately click on the part itself to remove the part from view, but one item at least needs to remain in the graphics window when viewing 3D simulation results.

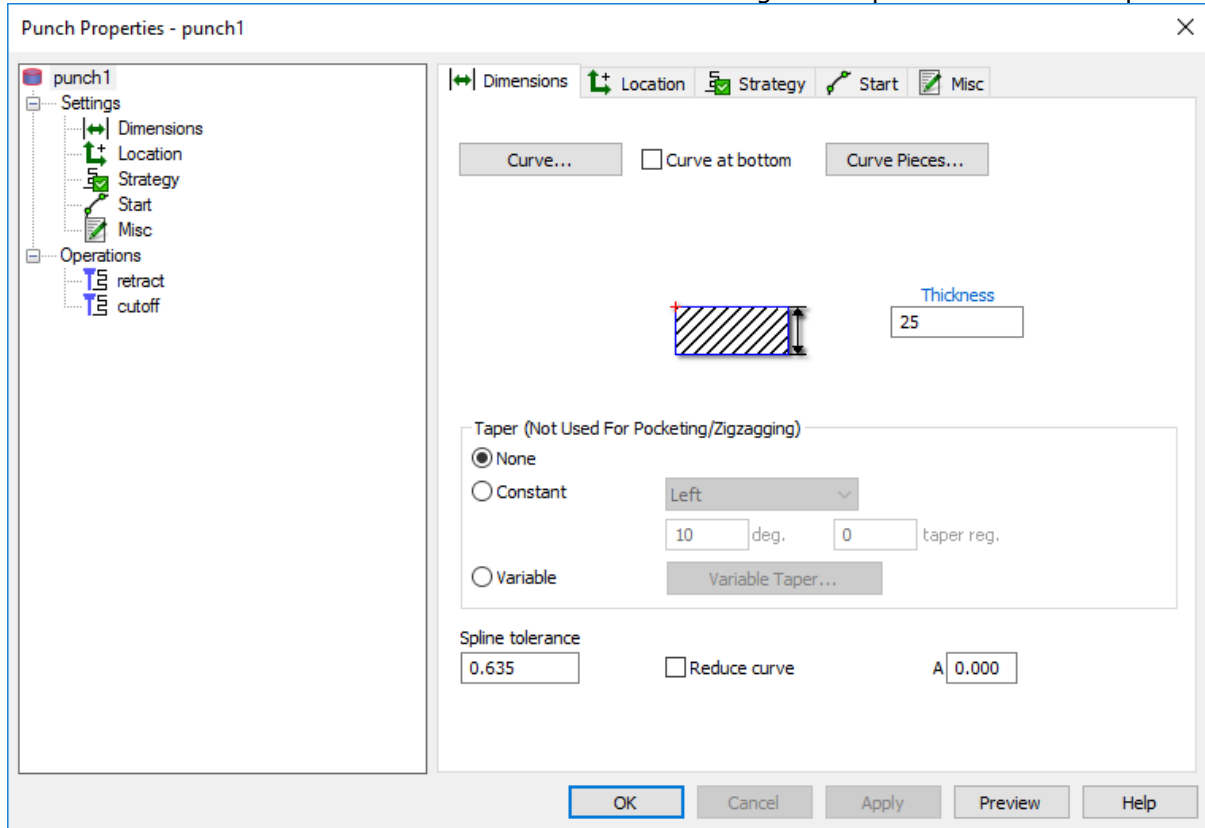
11 Eject the Simulation by pressing the **Eject Button** on the **Simulation Toolbar**.



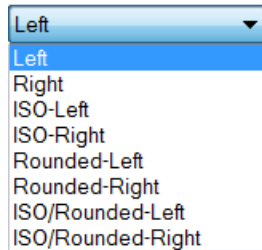
Open the **feature properties** by double-clicking **Punch1** from **Part View** in the **Toolbox**.

On the **Dimensions** tab of the feature change the **Taper** type to **Constant** with a **10 degree** value entered in the field as shown.

Press the **Preview** button to see the change and updated feature shape.



As a review, toggle through each of the **taper types**



(Left, Right, ISO-Left, etc.) by using the pull-down menu to **Preview** the feature shape changes in the graphics window.

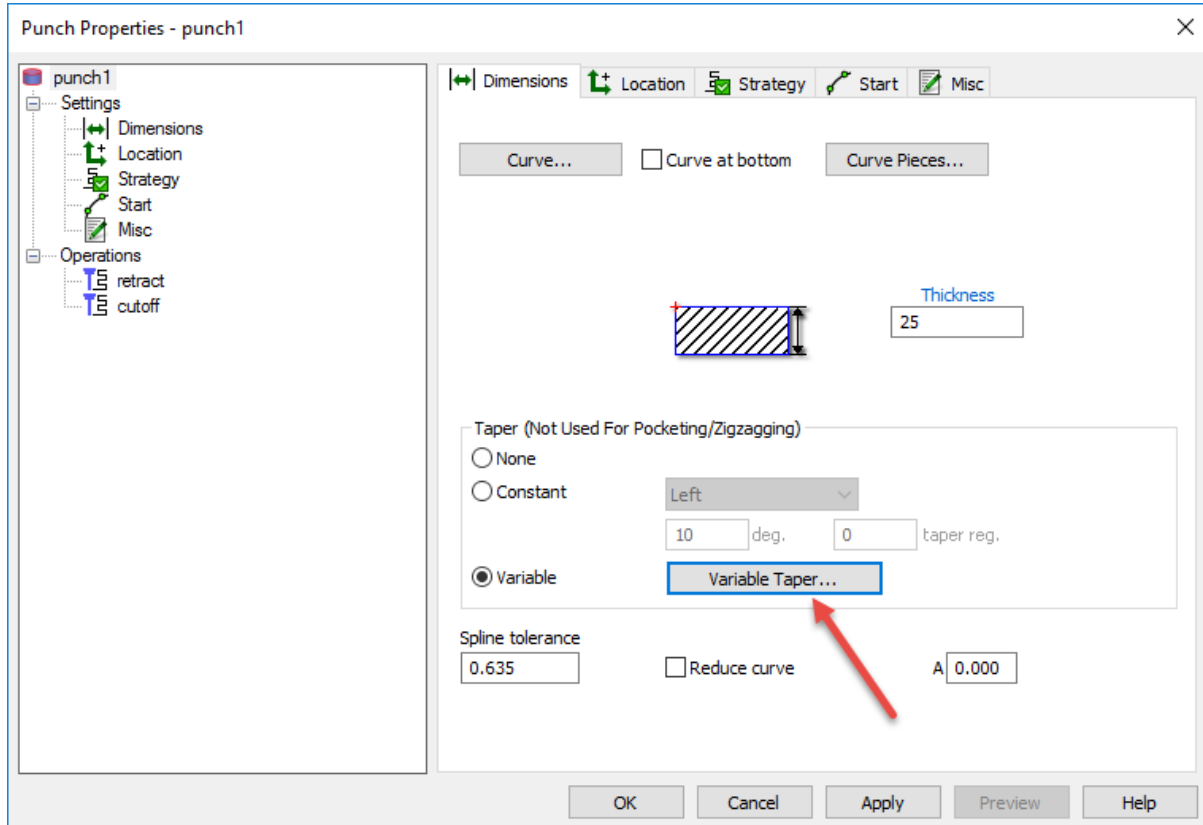
Types of Wire EDM Constant tapers

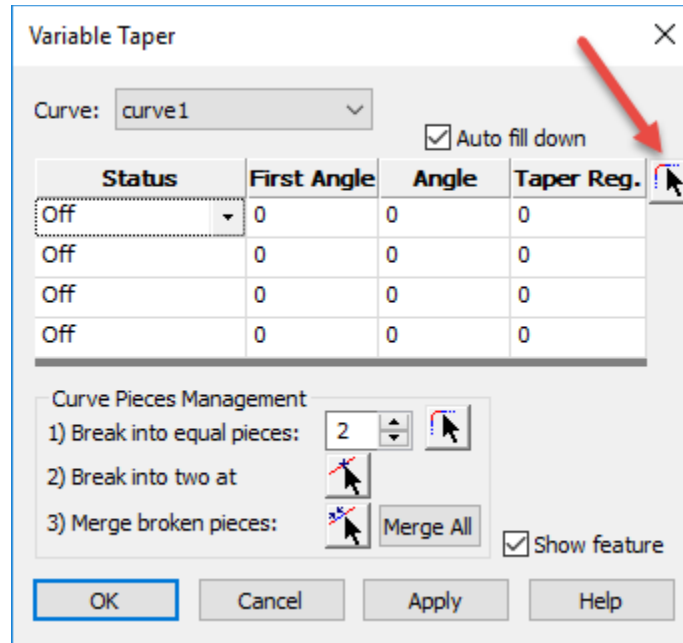
- All left tapers are performed to the **left** of the curve (at the bottom of the feature) relative to the **Primary Cut Direction**, and all right tapers are performed to the **right** of the curve (at the bottom of the feature) relative to the **Primary Cut Direction**.
- **Left:** Sharp corners tapered into sharp corners (G29 on a Charmilles machine) and arcs tapered into conical corners (G28).

- **Right:** Sharp corners tapered into sharp corners and arcs tapered into conical corners.
- **Rounded-Left:** Sharp corners tapered into arcs (G28) and arcs tapered into conical corners (G28).
- **Rounded-Right:** Sharp corners tapered into arcs and arcs tapered into conical corners.
- **ISO-Left:** Arcs tapered into cylindrical corners (G29) and sharp corners tapered into sharp corners (G29).
- **ISO-Right:** Arcs tapered into cylindrical corners and sharp corners tapered into sharp corners.
- **ISO/Rounded-Left:** Arcs tapered into cylindrical corners (G29) and sharp corners tapered into arcs (G28).
- **ISO/Rounded-Right:** Arcs tapered into cylindrical corners and sharp corners tapered into arcs.

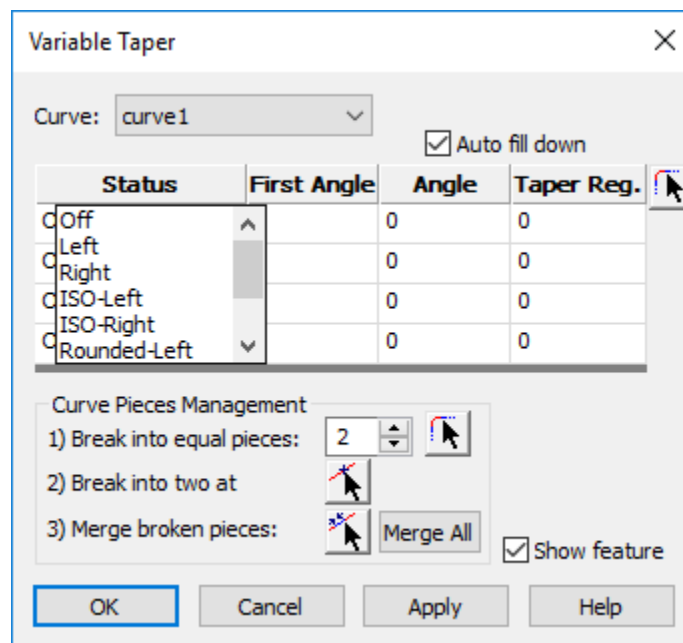
Variable taper table

- If the feature has multiple curves, select the appropriate curve from the **Curve** list. The curve is broken down into arc and line segments and each segment is represented by a row in the table. As you click a row of the table, the segment of the curve is highlighted in the Graphics window.
- If you wish to find the row of the table that corresponds to a segment, click the **Pick curve piece** button and select the segment. The row of the table is then selected.





The columns of each table are:



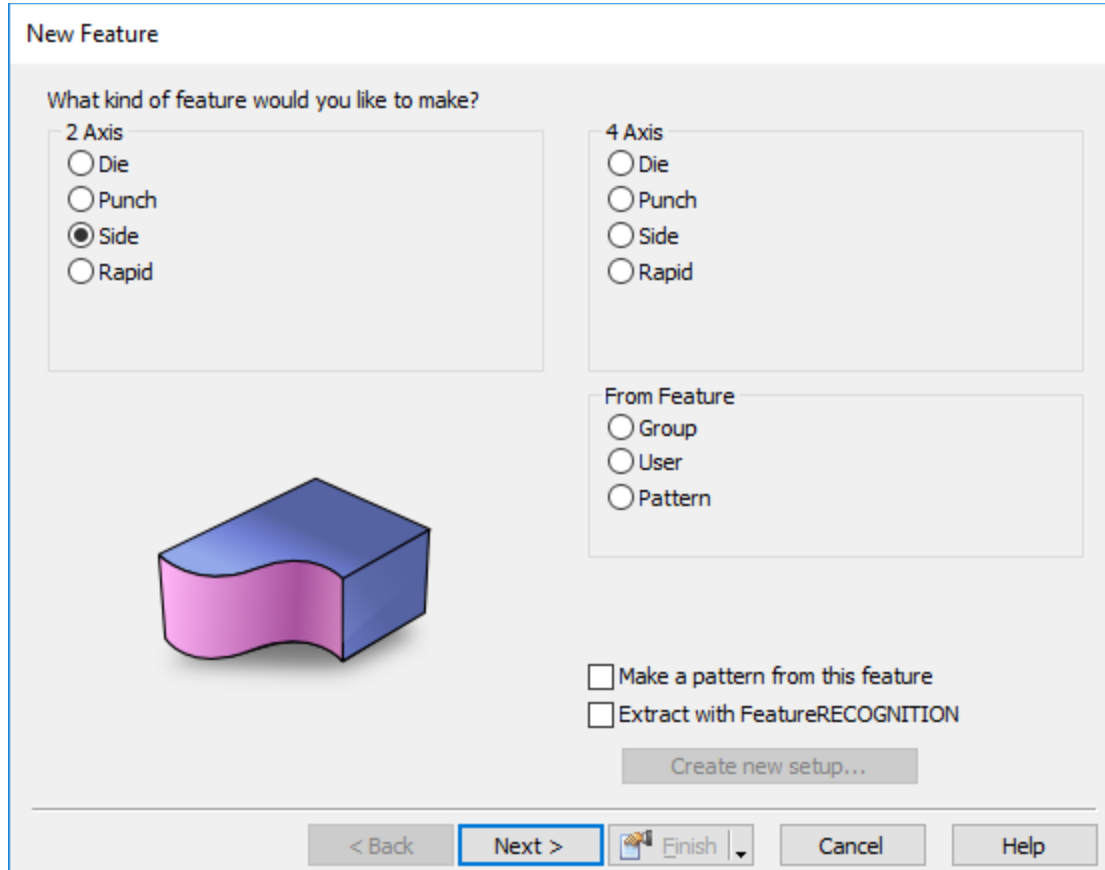
- **Status** - Double-click box in the status column to display a list of Types of wire EDM tapers.
- **First Angle** - This is the taper angle, specified in degrees, for the first end point of the segment.
- **Second Angle** - This is the taper angle, specified in degrees, for the second end point of the segment.
- **Taper reg** - This is the register on the machine containing taper information for this segment.
- If you want smooth walls on your part, the angle of one segment must match the first angle of the next segment.



This is covered in an exercise later in this module

Close the file **without** saving.

2-axis Side

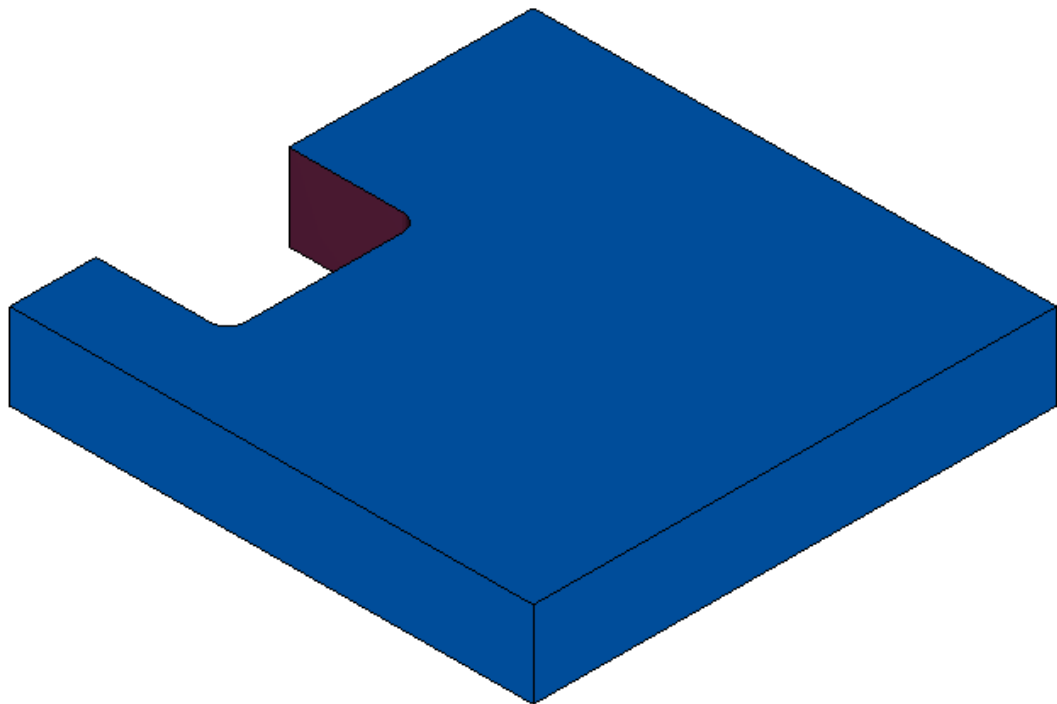


- The **Side** feature can use one or more **closed or open curves**. The **Punch** or **Die** features provide more cutting options for closed curves. You can optionally place the wire on either side of the curves. A side feature must have at least three arcs or lines in it including the lead moves.
- The thickness parameter is used to access the appropriate cutting data table.
- A **2-axis side feature** can create a number of cutting operations. These are specified on the **Strategy** tab.
- A **2-axis side feature** can have a **taper**.
- A side feature must have at least three moves in it including the lead moves. If it does not, then no feature is displayed in the start page that is displayed in the wizard. For example, if you are creating a feature from a single line, you must change the start point and end point so that the feature has three moves. If you add these moves and click the **Next** button in the wizard, the feature displays.

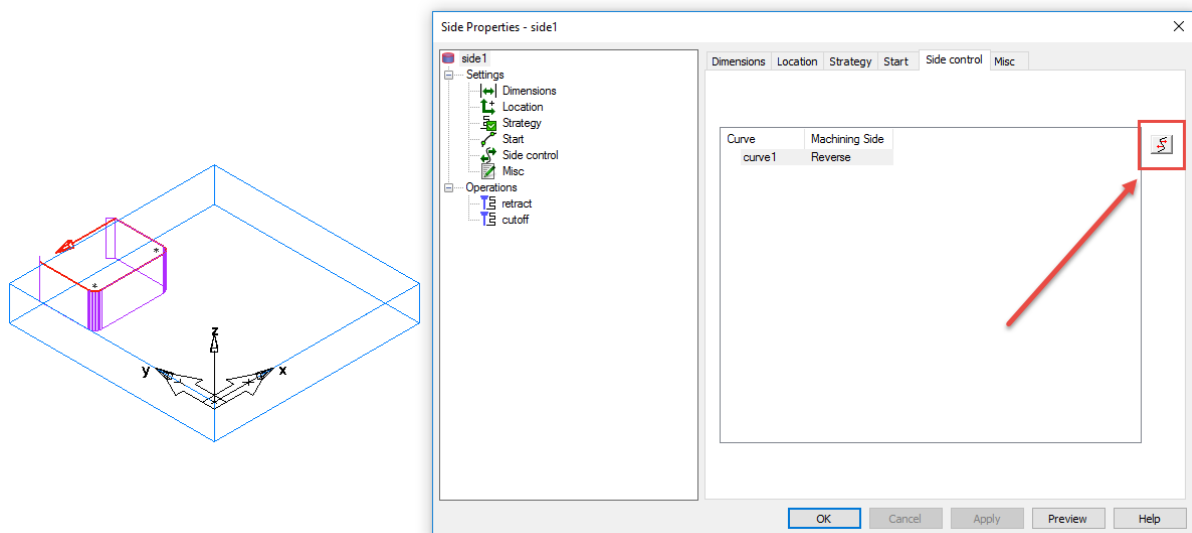
- 1 Open the file named **2 Axis Side.fm** from the Data folder.
C:\Training_Data\FeatureCAM Course Data 2017\FeatureWIRE Metric Data

- 2 Run a **3D Simulation** and use the **Select Arrow** to remove the piece to be removed from the part.

Isometric View



- 3 Open the side1 **feature properties** by double clicking the feature in the Part View, or the **Graphics Window**.
- 4 Select the **Side control** tab.



Notice the wire path will cut along the inside of the curve as shown above right.

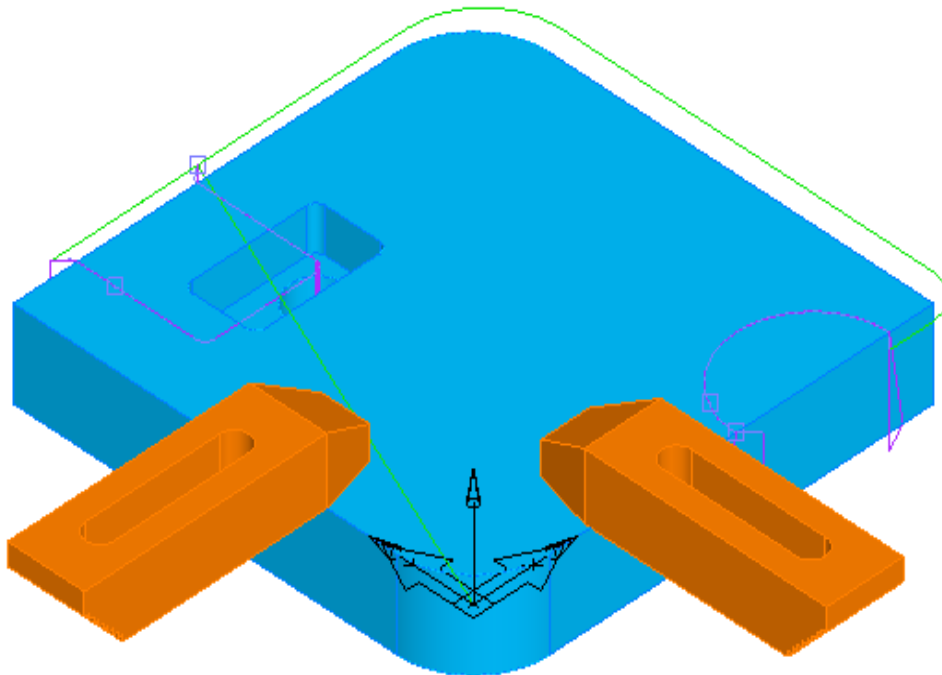


Press the **Switch machining side** button to change the side of the curve to be machined as shown.

- The lead moves have changed as a result of the change and the wire will follow the curve shape from the other side of the selected curve giving different results.
- This exercise is merely to show the user how to use the side control functionality that is available in all curve driven features throughout FeatureCAM.

2-Axis Rapid

- A **Rapid** feature is used to rapid along the length of a curve to the start point of the next feature. You can use **Rapid** features to quickly move around a part using curves to avoid possible collision with fixtures.
 - FeatureCAM programs a **Rapid** in a very similar way to a **Side** feature. The wizard pages are similar, but fewer options are needed.
- 1 **Open** the file named **2 Axis Rapid.fm** in the **Data** folder.
 - 2 **C:\Training_Data\FeatureCAM Course Data 2017\FeatureWIRE Metric Data**



If you don't see the toe clamps when the file opens:

- 3 Choose **View** from the **File Menu** - **Show** - **Show All Solids**



*Notice the **toe clamp solids** in the file. The toe clamps are there to represent the fixture that is holding the part. There are two features in the file that need to be machined. This exercise shows us how to use the **2 Axis Rapid** feature to control the rapid move of the wire to cut both features and avoid our fixtures.*

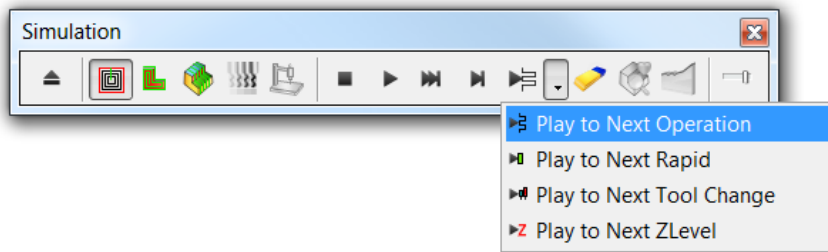


FeatureCAM has the capability to select solids that exist in the file and show them during a 3D simulation in order to help the user avoid colliding with the fixture during the machining operations. They can be toe clamps, vises, or custom fixtures. As long as you have a solid model to import into your FeatureWIRE project you can use them in the same way as this exercise.

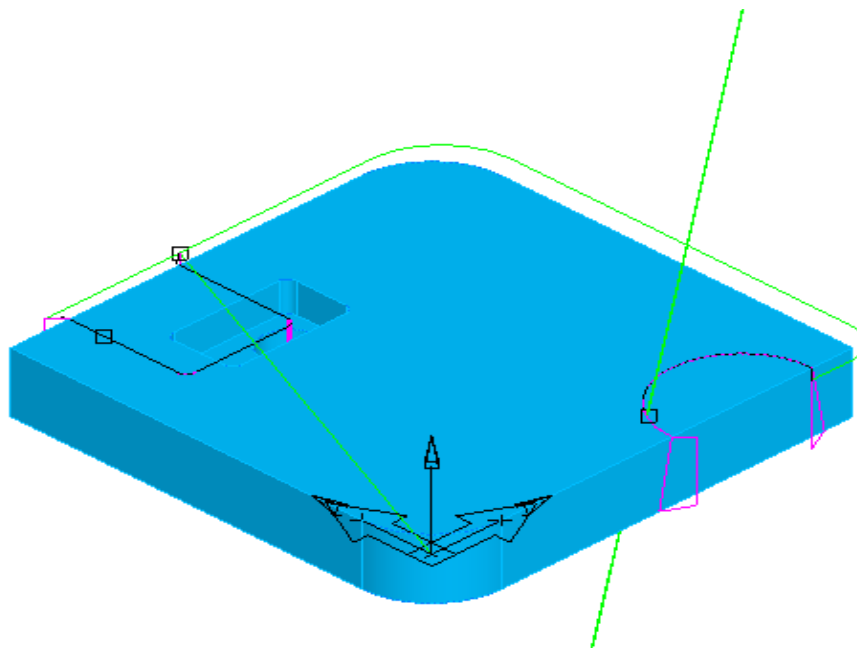
- 4 **Right click** on any face of the imported toe clamp solid and choose **Use Solid as Clamp** from the menu that pops up. If it is active to be visible in the 3D simulation, you will see a check mark next to it, as shown in the image below.
- 5 **Do this for BOTH solid toe clamps!**
- 6 Run a **Centreline Simulation** and choose **Play to Next Operation**. Do this until the simulation has completely cut the two side features.



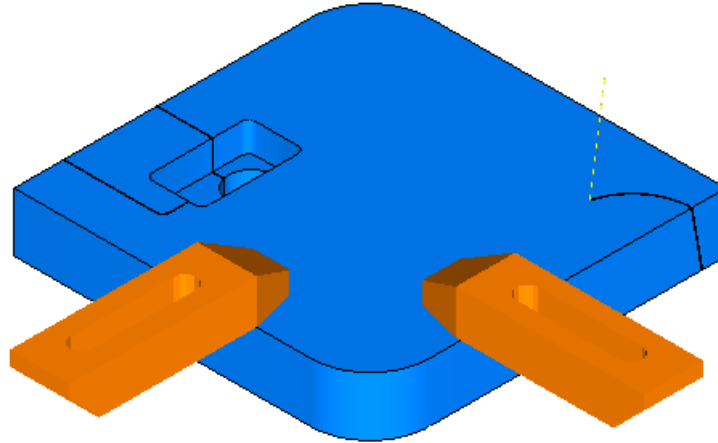
We are going to run the side1 and side2 features ONLY. Do not activate the rapid1 feature yet.



- 7 In the **Part View** window in the **Toolbox** press the + sign next to **Curves** to expand the list of curves in the file. **Click** the curve named **rapid** to see the curve in the **Graphics Window**.
- 8 This curve begins where side1 feature finishes machining and ends where side2 starts to approach, and machine the part. It is the curve that is used for the feature named **rapid**.



- 9 Hide the **toe clamps**. From the **File Menu** choose **View – Hide - Hide All Solids** to hide the **toe clamps** from view in the graphics window
- 10 In the **Part View** activate the feature named **Rapid** and run a **Centreline simulation**. Notice how the rapid feature controls the wire path rapid around to machine the second side feature.



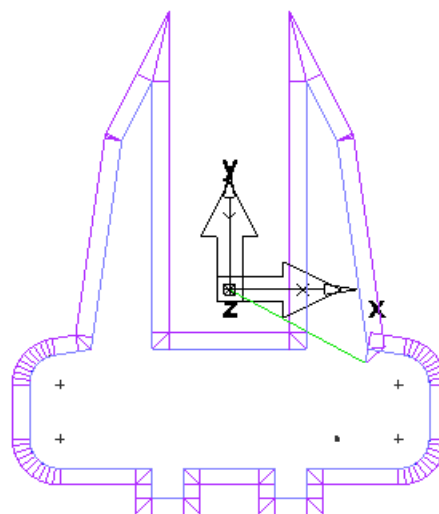
- 11 **Run a 3D Simulation** to see the clamps in the graphics window and the machining operations.



Displaying the Solids used as clamps allows the user to program the part accordingly and avoid any collisions.

More on Tapers and Start Points

- 1 Open the file named **snaggletooth.fm** from the **FeatureWIRE Metric Data** folder. **C:\Training_Data\FeatureCAM Course Data 2017\FeatureWIRE Metric Data**
- 2 Go to the **top view**.
- 3 **Run a 2D Centreline Simulation** just to familiarize yourself with the component.



- 4 Open the **die1** feature properties dialog.
- 5 Go to the **Start** tab.
- 6 Using the **Pick Arrow** select the **Point** located at **(X22.225, Y-31.75)** and then the bottom right horizontal line somewhere below the point.
- 7 **Apply** and **OK** then **Run a centreline Simulation** to see the updated wire path.
- 8 Open the **die1** feature properties again.
- 9 Go to the **Start tab** again and check the box **Pick only perpendicular lead**.
- 10 Using the **Pick arrow** select the point again and anywhere on the line below.

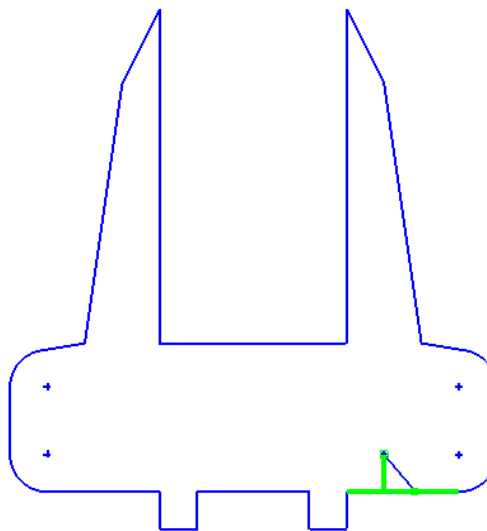


Notice that it does not matter where you select along the line, only the perpendicular tangency is allowed to be created for the wire path approach move.

- 11 **Apply** and **OK**. **Run a centreline simulation** to view the results.



Leave the file open for the rest of the exercise below.



Constant Taper Exercise

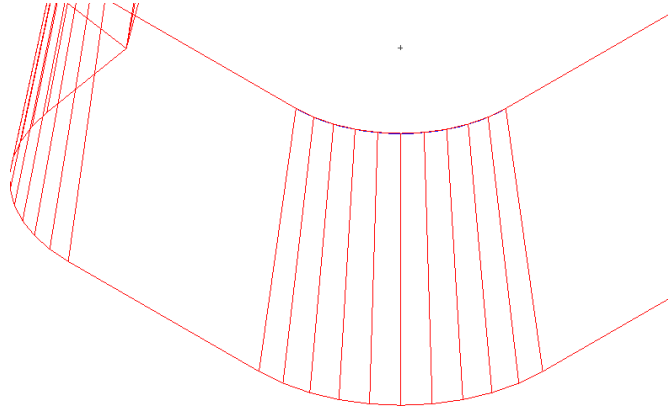
- 1 Open **die1** feature properties
- 2 Go to the **Dimensions tab**.
- 3 Set taper to **constant** and **Left. 10 degrees** and **Apply**.
- 4 Go to the **isometric view**.
- 5 Set taper to **constant** and **right**. Press **Apply**.



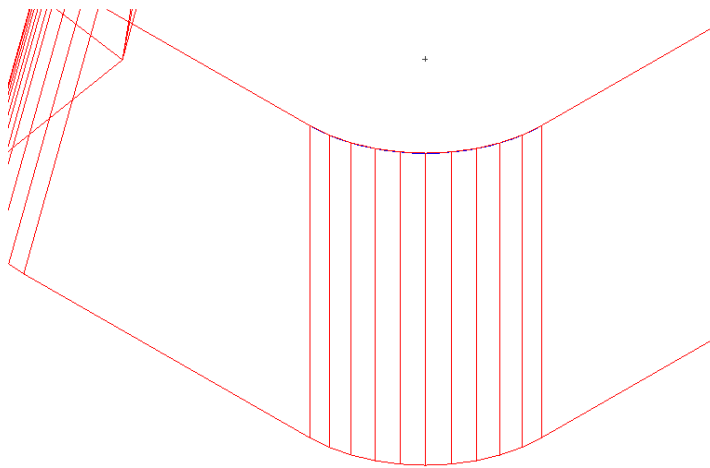
FeatureWIRE supports ISO or cylindrical taper vs. the regular or conical taper.

- 6 Go to **isometric view**.
- 7 Set taper (**10 deg.**) to **constant** and **ISO right**.

- 8 Zoom in to the lower left corner of the feature to see the bottom arc. The taper is cylindrical (same radius).
- 9 Switch back to **constant** and **right** to see the difference.



Before

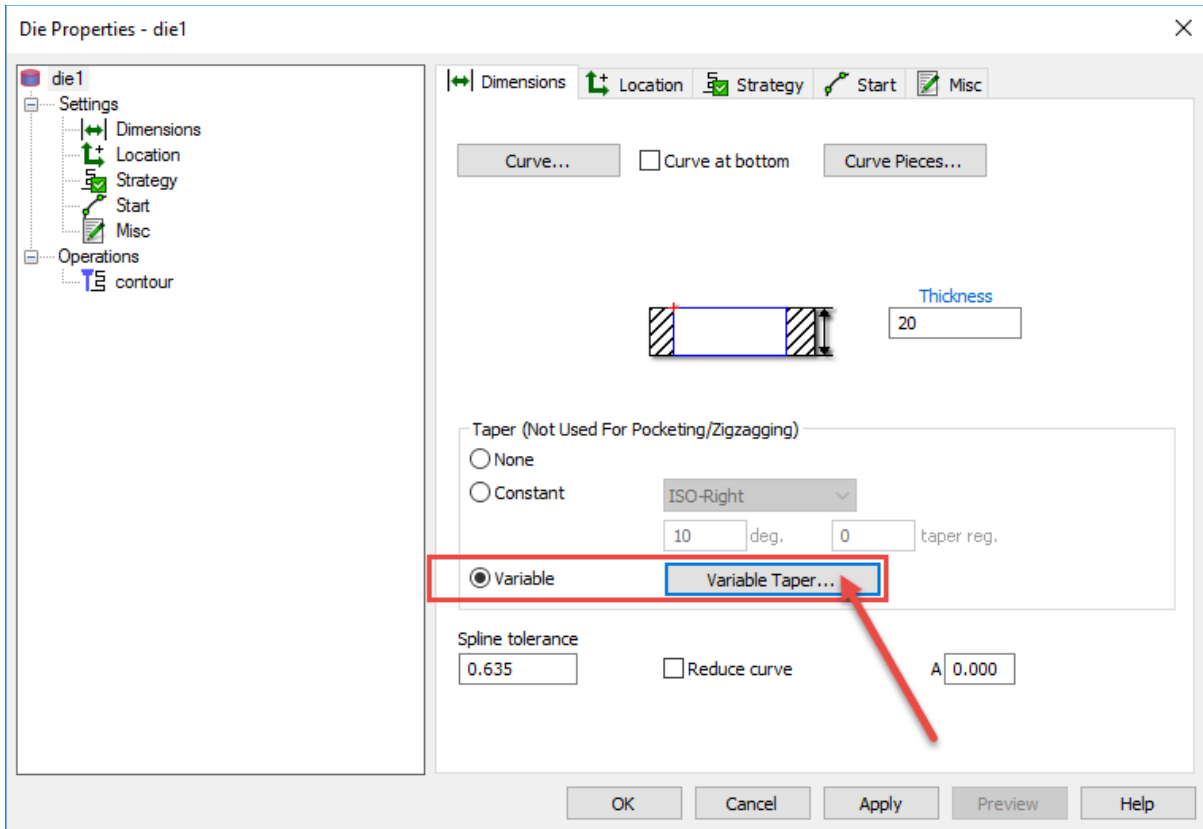


After

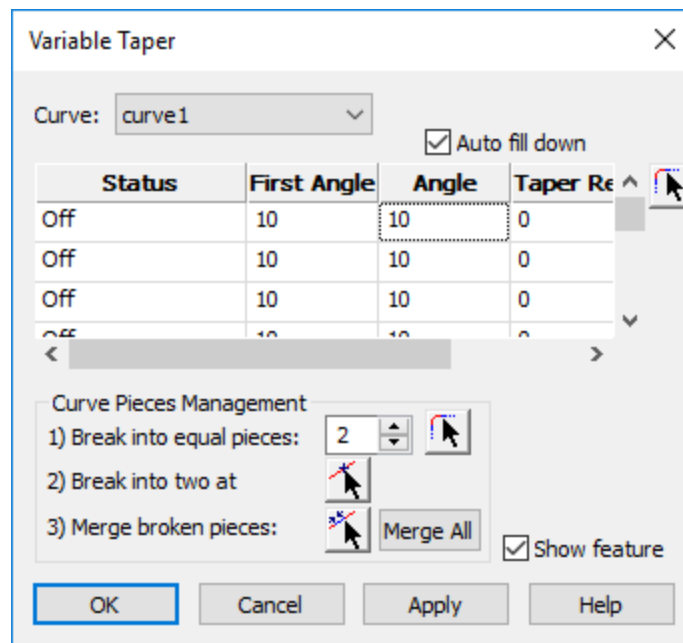


Leave the file open for the rest of the exercise below.

- 10 Hold the **Control** key and press “**Q**” on the keyboard to “**Centre all**” in the graphics window.
- 11 In **Die Properties** select **Variable**.



12 Set **taper** to **Variable** and press the **Variable Taper** button.



13 In the Variable Taper dialog, set the fields to **(Right, 10, 10)**.



*Make sure **Auto fill down** is checked*



There is no need to set anything in Taper Reg. field for this exercise.

14 **Apply** and **OK** in the **Variable taper** dialog.

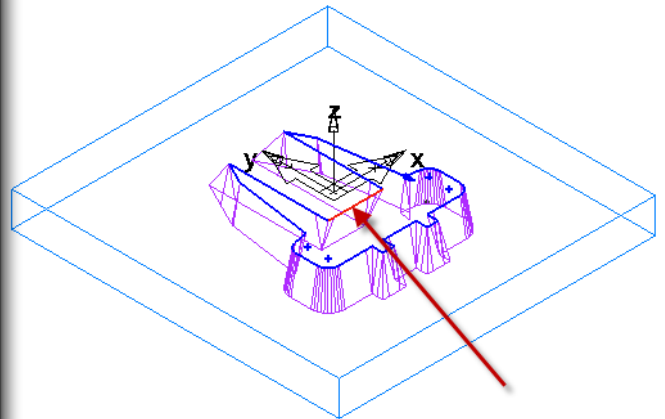
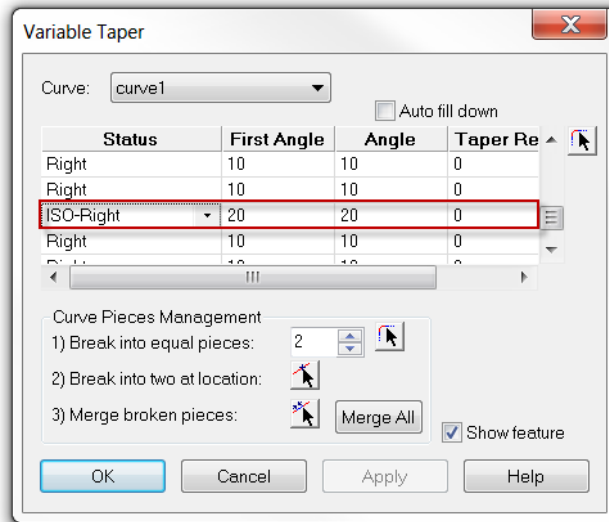
- 15 **Apply** and **OK** in the **Die Properties** dialog.
- 16 This is the same as setting a constant taper right 10 degrees.
- 17 Open the Variable Taper dialog again and **uncheck Auto fill down**.



It is possible to change the angle of different segments of the curve profile.




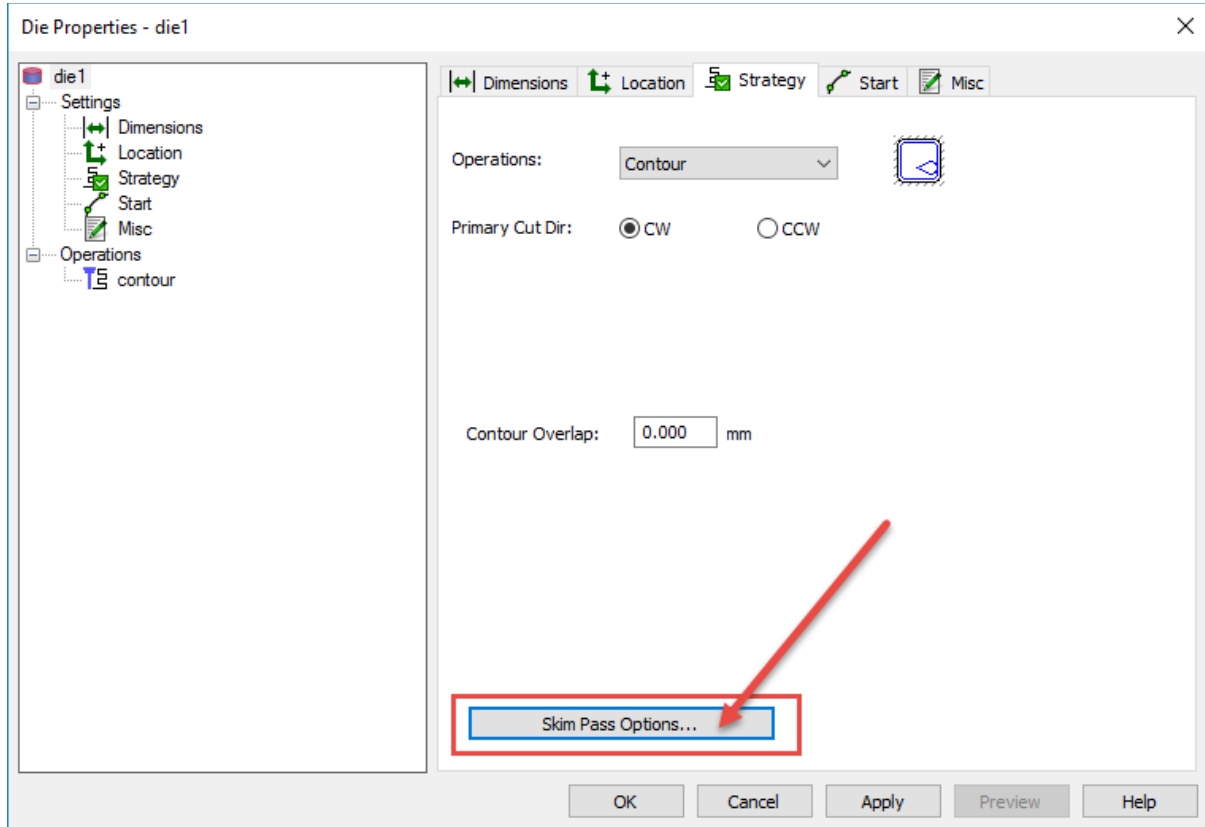
*For example one segment could be 20 degrees and all other segments could remain at **10 degrees**.*



Have a go at changing different segments to different angles.

Skim Cuts

- A common practice in cutting wire EDM parts is to make skim cuts for a better finish. This is to make multiple passes with a stepover to cut the part down to size.
- 1 Open the file named **Skim Cuts.fm** from the **FeatureWIRE Metric Data** folder. **C:\Training_Data\FeatureCAM Course Data 2017\FeatureWIRE Metric Data**
 -  *In this die feature the taper is set to none to get a better visual of how the skim cuts work.*
 - 2 Open the properties of the **die1** feature.



3 Go to the **Strategy** tab.



*There is only one operation in this feature, a **Contour Operation**.*

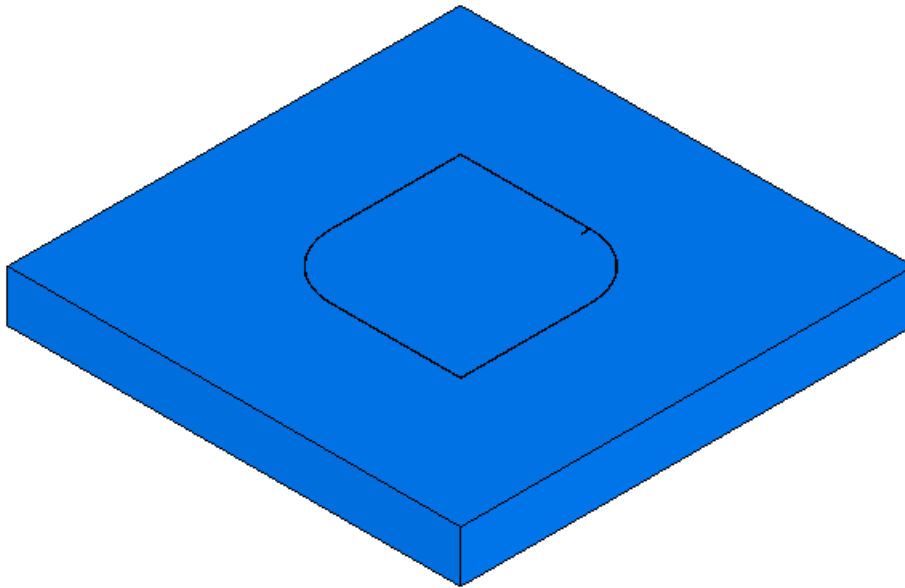


*When cutting with a contour operation the curve can be open or closed. A contour operation traces the entire length of the curve. The number of passes is determined by **Contour passes**.*

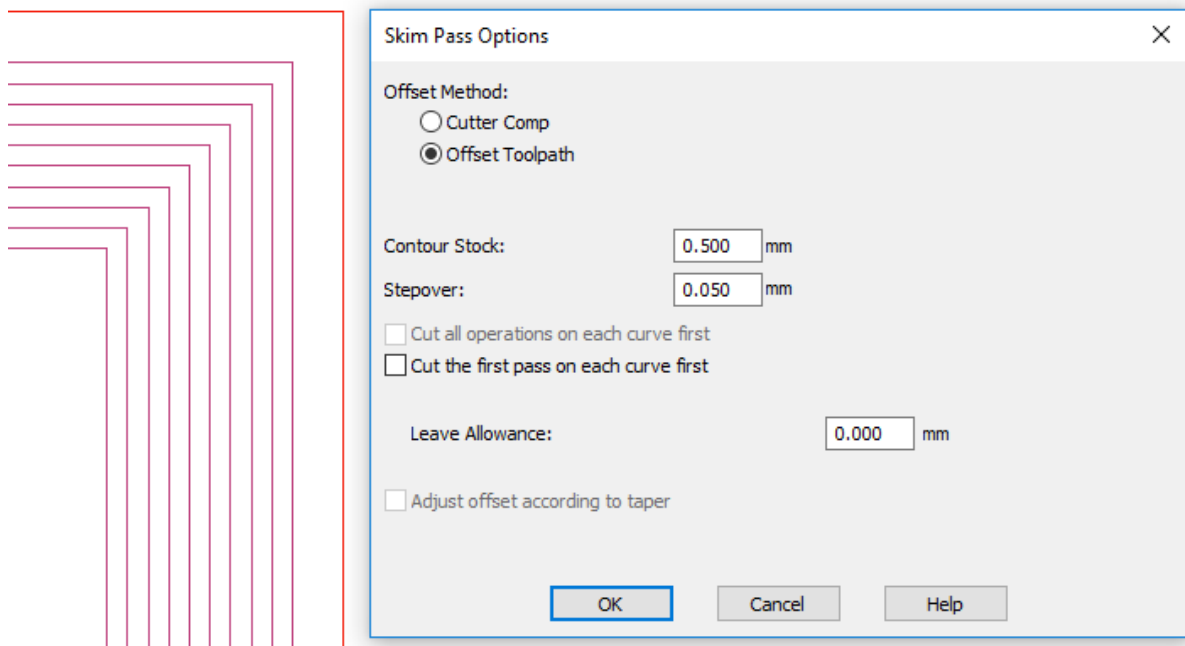
4 Select the **Skim Pass Options** button.



The number of passes is currently set to 1 so you will see only 1 pass in the simulation.



- 5 Press **OK twice** to close the dialog and the feature.
- 6 Go to an **Isometric view**.
- 7 Run a **3D Simulation**.
- 8 Slide the simulation speed slider to the half-way point on the Simulation Toolbar speed so you can see this more clearly or select **Alt+F3** Single step.
- 9 Open the **die1** feature properties again and on the **Strategy tab** press the **Skim Pass Options** button.
- 10 Change **Contour Passes to 3**.
- 11 Run a **3D simulation** again.
- 12 Right now we are using cutter comp as the offset method so the toolpath for the 3 passes are not offset. The NC code will have different cutter comps for each pass for the wire EDM machine to do the offsetting.
- 13 Another way of making skim cuts is to actually generate offset toolpath.
- 14 Open the **die1** feature properties again and on the Strategy tab press the Skim Pass Options button.
- 15 Change to **Offset Toolpath**.
- 16 Set **Contour Stock** as shown. This is the amount of material left to cut away to the curve.
- 17 Set the **Stepover** as shown. The wire will step over the specified amount after each pass.



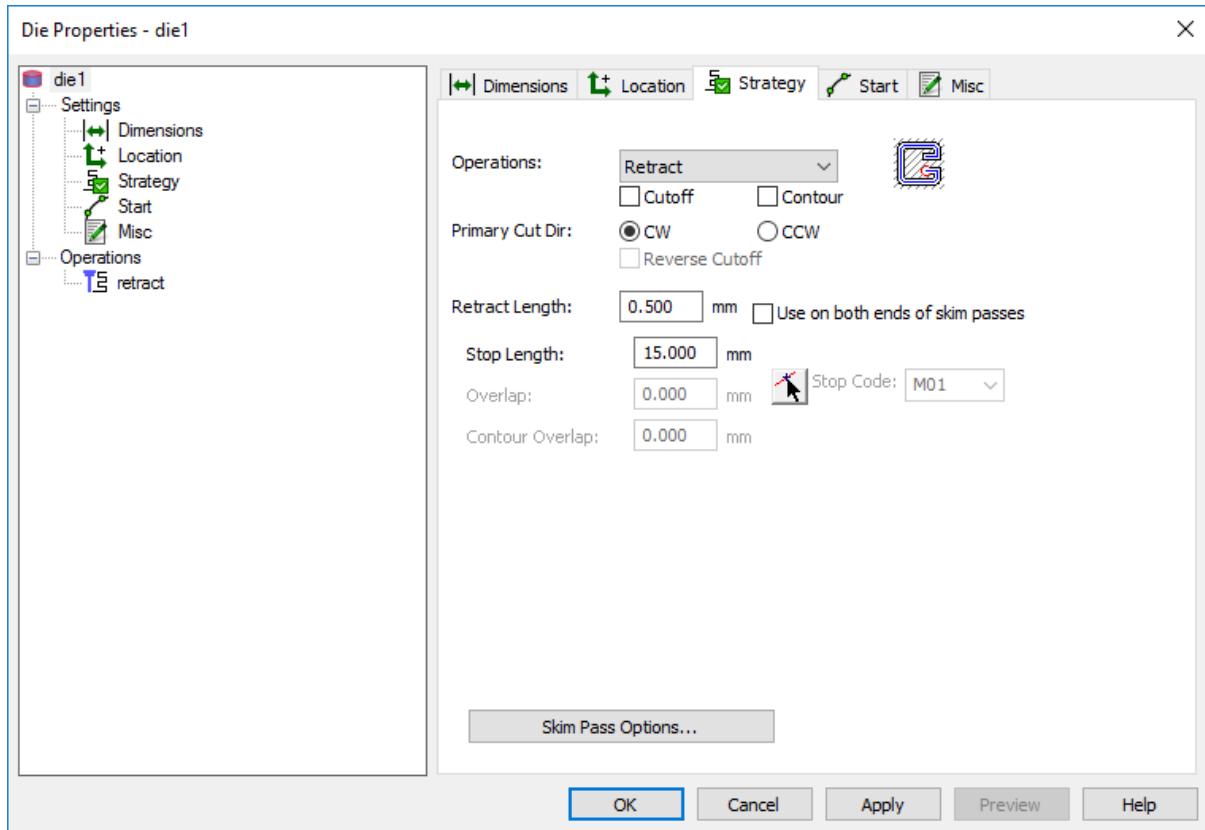
- 18 Press **OK** and **Apply** and **OK** again to apply the changes to the feature.
- 19 Go to the **Top view**.
- 20 Run a **Centreline Simulation**.
- 21 Zoom to a corner of the feature to see offset toolpath.
- 22 Select **Control** and **Q** to Centre all.
- 23 Leave the file open for the rest of the exercise below.

Retract and Cutoff Operations

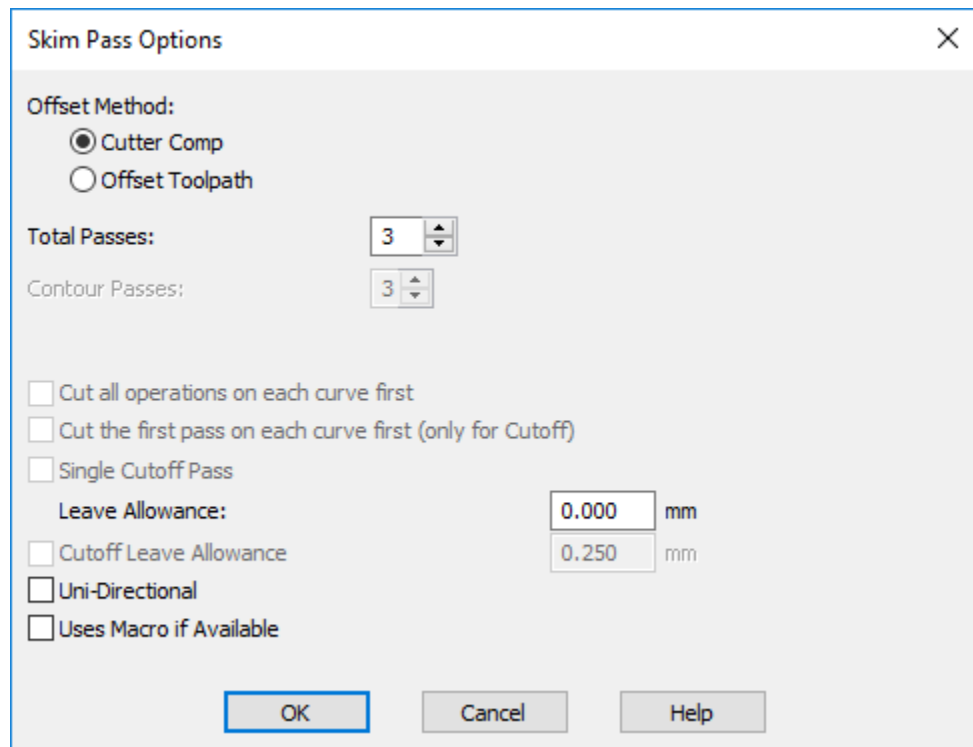


Another common practice in cutting wire EDM parts is to run the machine unattended so that most of the cutting can be done before human intervention is needed. We support this with the Retract and Cutoff operations.

- 1 On the **Strategy tab** set the operations to **Retract** in the **Operations** field using the Pulldown menu. The Die Properties dialog box updates to allow other settings.



- 2 Check the box next to **Cutoff**.
- 3 Press the **Skim Pass Options** button and change the setting back to **Cutter Comp** and the **Total Passes to 3**.



- 4 Select **OK, Apply OK**.
- 5 Go to an **Isometric view**.

- 6 Run a **3D simulation** using **Play to Next Operation**.



Use the triangle pull-down menu to find this simulation button.

- 7 Click the **Play to Next Operation** button again to finish the **3D Simulation**.



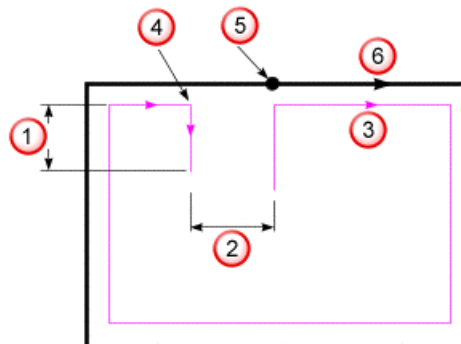
Programming the feature in this manner creates a situation where the retract operation could be run overnight. When it is done, the machine would stop. The uncut region keeps the core in place to prevent the core from interfering with the wire movement. So in the morning, the operator reconfigures the fixture to hold the core, and finishes the cutting with the Cutoff operation. The retract move makes sure the burn mark from the wire changing cut direction is away from the part. A cutoff operation is normally used after this operation to cut the part of the contour that is not cut by the retract operation. The cutoff operation machines the left over part in the opposite direction to the machining curve.

More about Retract



The retract operation is available for both 2-axis and 4-axis features.

- The contour is not cut completely but is stopped shortly before the contour end point. The distance of this position from the contour end point is set in the **Stop Length** field. This option is generally used when cutting multiple contours whereby the cut part should not be separated from the work piece, such as when the program should run automatically overnight.



- **Retract length**
- **Stop length**
- **Wire path**
- **Inserted end position**
- **Normal contour start/end position**
- **Contour**

Pocketing and Zigzag



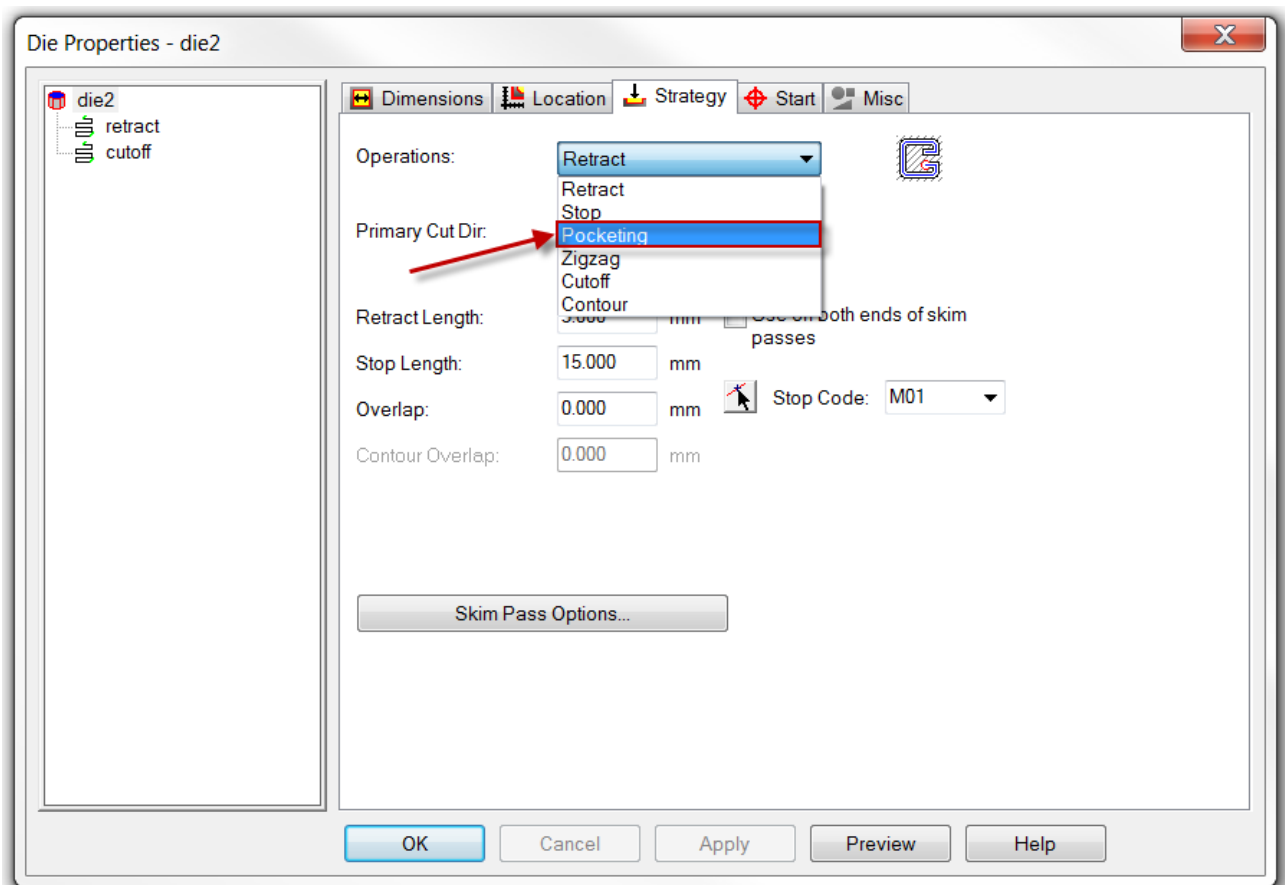
The following are two very useful operations. There are no-core operations meaning that the entire core is burned away."

- 1 Start a New File from the File Menu – Select **File > New file > Wire EDM Setup**. Choose **Millimeters** as the Unit of Measure and press **OK**.
- 2 Set the **Stock** as **Block** with the **dimensions 150mm X 150mm X 25mm thick**.
- 3 Double click on **Setup1** and Centre the Setup Axis.
- 4 Select **Finish** and **OK**
- 5 Switch to the **Top view. Ctrl+5**
- 6 Create a **12mm Radius Circle from Rad, Centre** at X0, Y0, Z0.



The Circle Constructors are available as discussed in the previous Snapping and Geometry chapter.

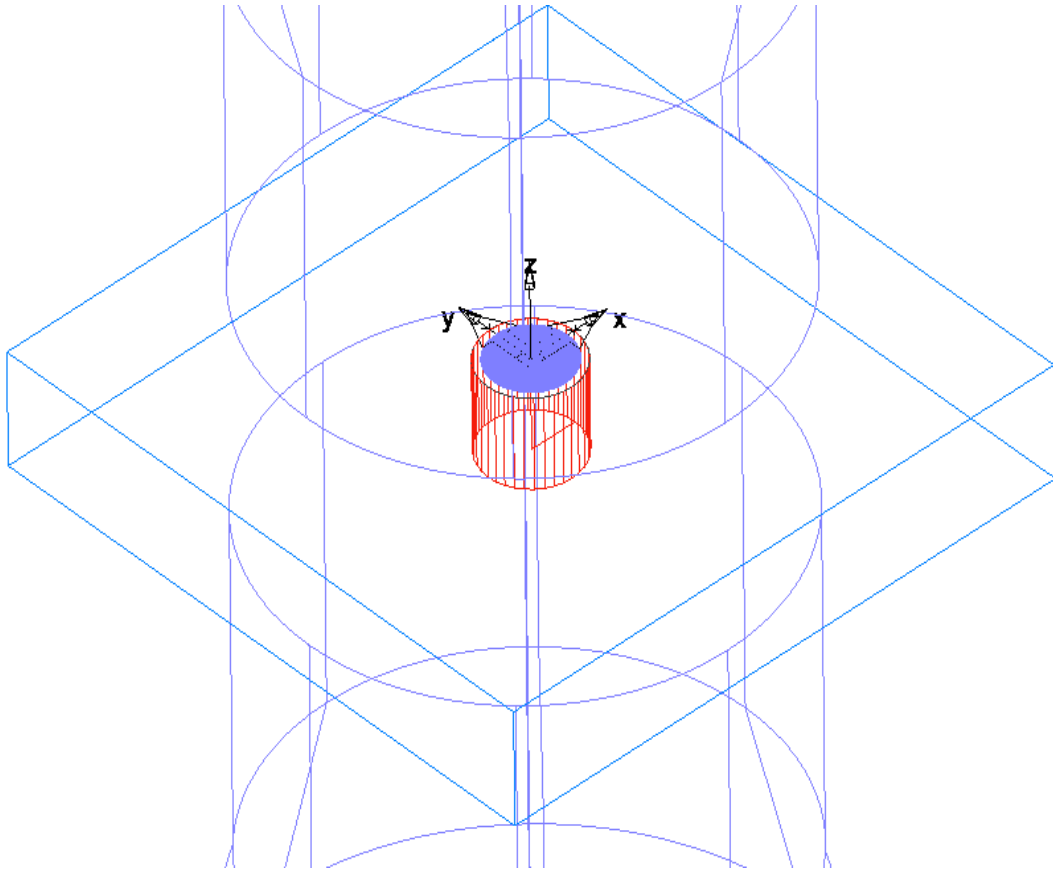
- 7 Create a **2 Axis Die feature** using the **circle** you just made. **No taper** on this feature.
- 8 Create a **Curve from the circle**. Select **Finish**. On the **Strategies** page. Set operations to **Pocketing**.





*In the existing feature this can be changed in the **Strategy tab**.*

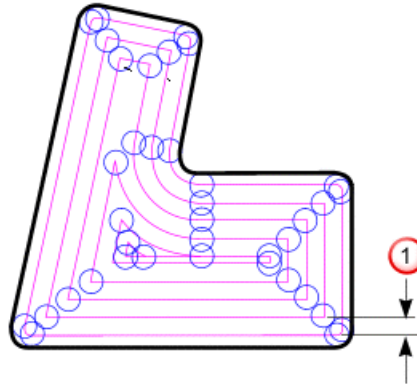
- 9 Select **Finish/Apply** and **OK**.
- 10 Select **Isometric View**.



- 11 Run a **Centreline Simulation**.

Pocketing

- The pocketing operation enables contour parallel area clearance of a closed curve. This is only available for 2-axis features.



1 - Step over

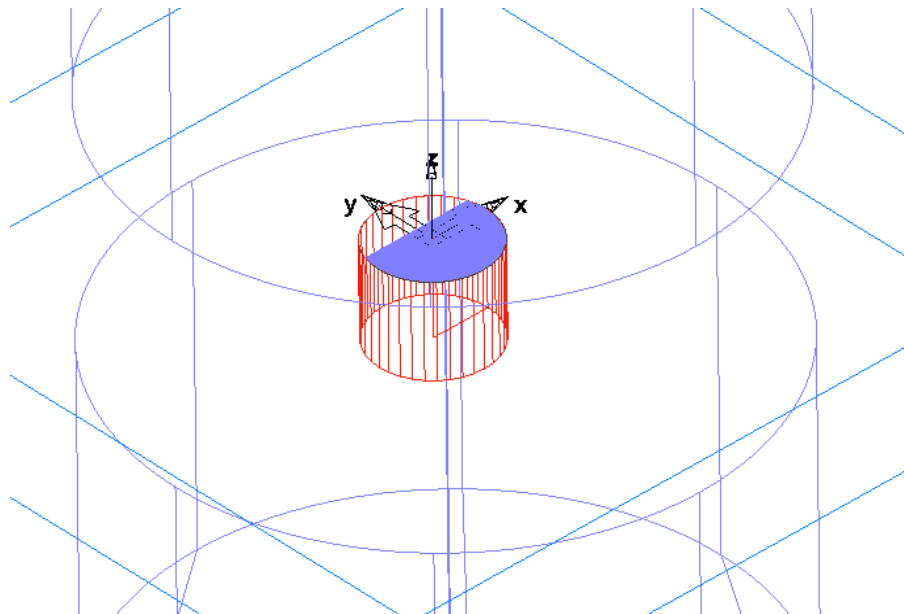
- The start position of the cut is set automatically to a point on the boundary. The wire then feeds to the Centre of the pocket and then cuts from the inside toward the boundary.
- If you wish to ensure that the wire starts in a particular position (to thread the wire for example) you should define a start point. The toolpath then starts at the start point and then feed to the Centre of the pocket.
- Cutter compensation does not apply to a pocketing operation because it outputs a wire path that is already corrected (that is, no compensation is entered on the machine).



No attempt is made to check if the stepover value is larger or smaller than the wire diameter.

12 Open the **feature properties** and change operations to **Zigzag**.

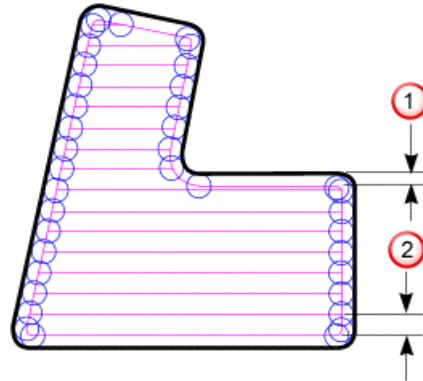
13 Run a **Centreline Simulation**.





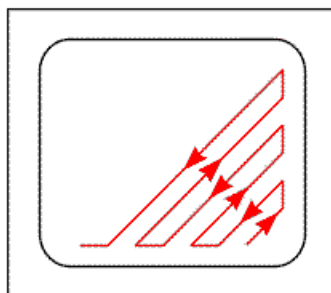
Leave the file open for the next operation setting.

Zigzag



- ① - Offset
- ② - Stepper

- The zigzag operation defines a zigzag area clearance cycle for a closed curve. It is available only for 2-axis features.
- The start position of the cut is set automatically to a point on the boundary. The wire then feeds to the beginning of the zigzag pattern.
- If you wish to ensure that the wire starts in a particular position (to thread the wire for example) you should define a start point. The toolpath then starts at the start point and then feeds to the beginning of the zigzag pattern.



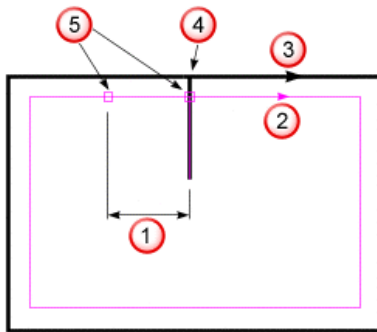
- By the definition of the **Cut Angle** you can control the direction of the wire path and also the start point of the cycle.
- Cutter compensation does not apply to a pocketing operation because it outputs a wire path that is already corrected (that is, no compensation is entered on the machine).



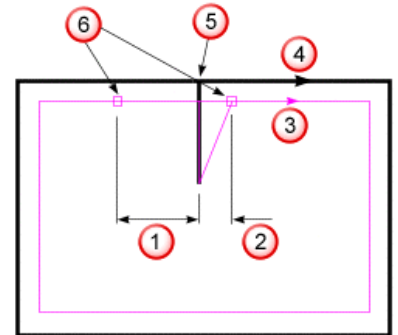
No attempt is made to check if the stepover value is larger or smaller than the wire diameter.

Stop

- The **stop operation** is available for both 2-axis and 4-axis features.
- A **stop command** is inserted into the wire path together with a further stop at the end of the contour. The distance of the new stop position from the contour end point is defined in the **Stop Length** field. If you wish to overcut the contour end position, you can also define an **Overlap**. In Centreline simulation, the locations where the wire stops are shown as small squares.
- A stop operation has two different locations where the wire stops:



- ① - Stop length
- ② - Wire path
- ③ - Contour
- ④ - Contour start/end position
- ⑤ - Inserted stop positions



- ① - Stop length
- ② - Overlap
- ③ - Wire path
- ④ - Contour
- ⑤ - Contour start/end position
- ⑥ - Inserted stop positions

Lead types

- 1 Open the file named Lead-in-out Metric Styles.fm from the FeatureWIRE Metric Data folder.
- 2 Run a Centreline Simulation using only the die1_Teardrop feature as active.

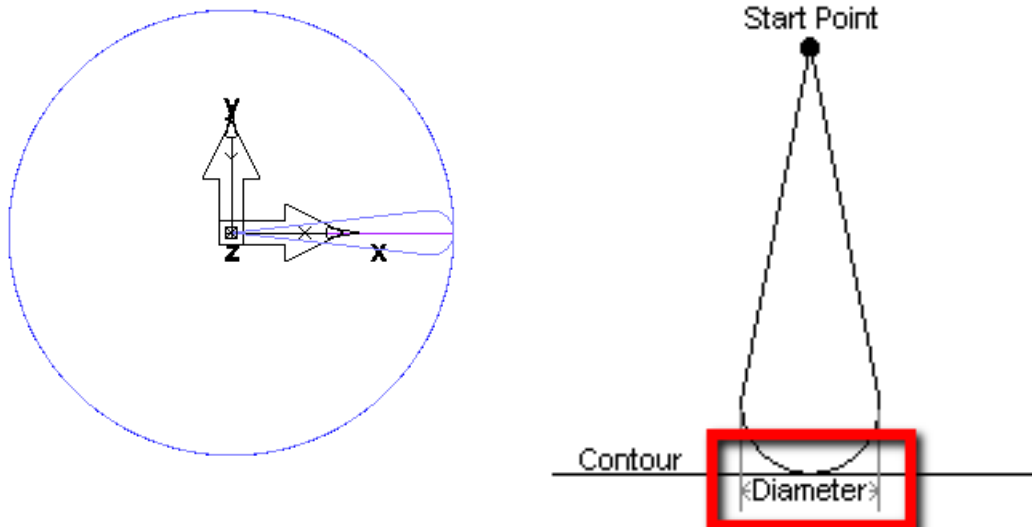


The wire path starts at the Centre of the feature where the Start Point was generated and then move in an angular direction and starts onto the curve shape with an arc.

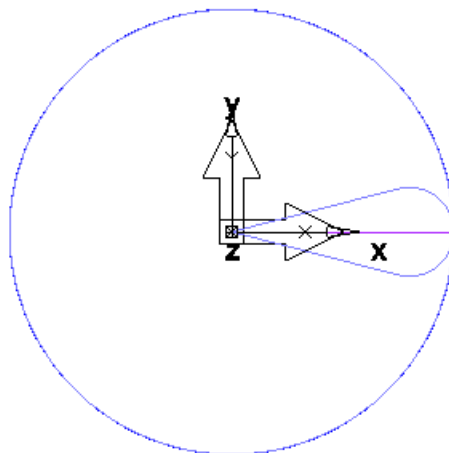


The wire path follows the curve shape around to where the approaching arc ended and started to cut the shape, then arcs off and make another linear move to the Start Point to finish the cut. The result is a teardrop shape as shown.

- 3 Use the **single step** button on the **Simulation Toolbar** or use Alt+F3 to see this more clearly.



- 4 Open the die1_teardrop feature properties and select the contour operation.
- 5 Go to the **Leads Style** tab.
- 6 Notice the Diameter for lead moves is set to 5.00mm
- 7 Change the **Diameter to 10mm** and **apply** the change.
- 8 Run a **Centreline Simulation** again to observe the change in the wire path.

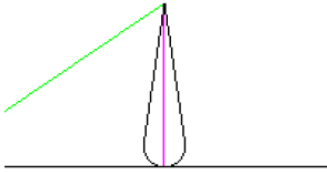
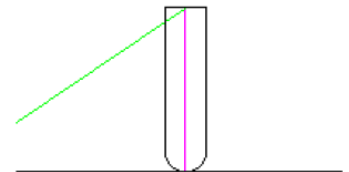
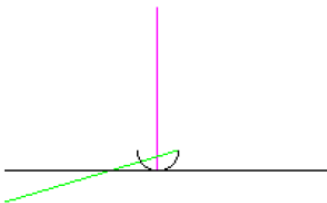
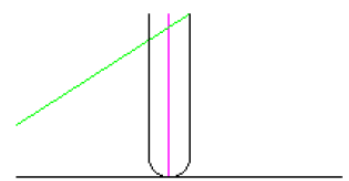


- 9 Go to the **Part View** in the **Toolbox**
- 10 Choose the feature named **die1_direct** and run **Centreline Simulation** again.



Direct is the only lead style that does not use the Diameter setting.

- 11 Select **each feature** (run them individually) and observe the **different lead moves** for each.

Teardrop:**Bullet:****Arc:****U-Shape:**

Misc. Tab Setting - Wire cutting/threading



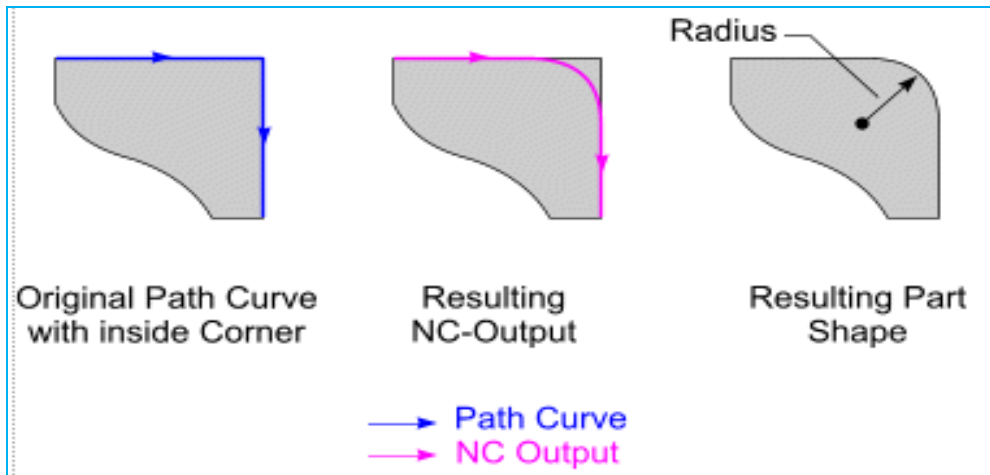
FeatureWIRE offers four settings to control the output of wire threading or wire cutting commands:

- **Off** - No wire threading or wire cutting commands are output in the NC-program.
- **Both** - The commands to thread the wire and cut the wire are output automatically at the start and end of each operation within the feature.
- **Cut** - The wire cutting command is output at the end of each operation within the feature (but no wire thread command at the start). When viewing the toolpath in center line simulation, the Cut location is denoted with a small circle.
- **Thread** - The wire threading command is output at the start of each operation within the feature (but no cut wire command at the end). When viewing the toolpath in center line simulation, the Thread location is shown as a small plus sign.

Modify outside corners



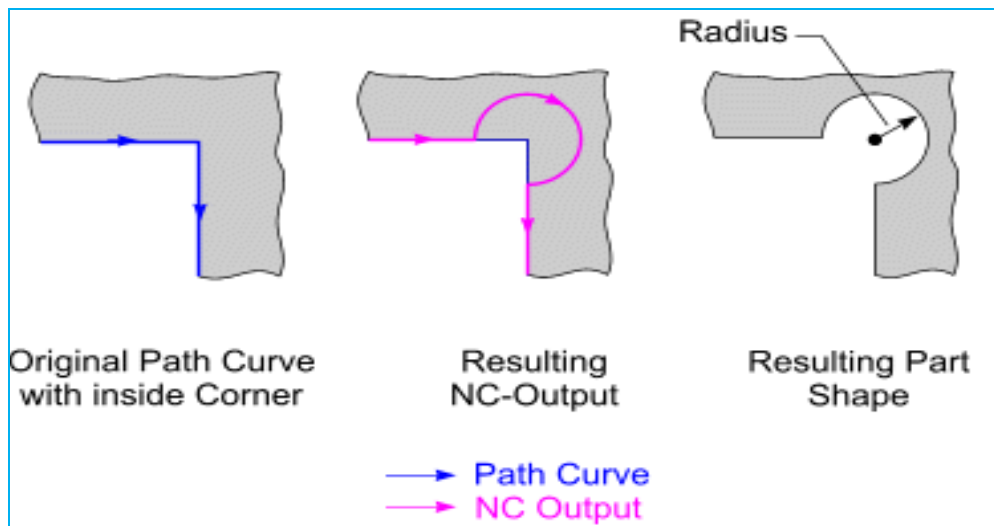
A radius will be inserted into the wire path at each outside sharp corner. This can be useful for reducing unnecessary movements and for producing cleaner corners.



Modify inside corners

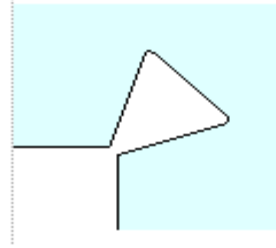


Circular - A circle is inserted in the corners. The center of the circle lies on the corner point.



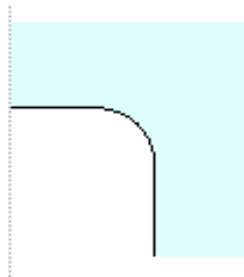
Triangular

- The path follows a triangular shape at an inside corner. Specify the length and width of the triangle.



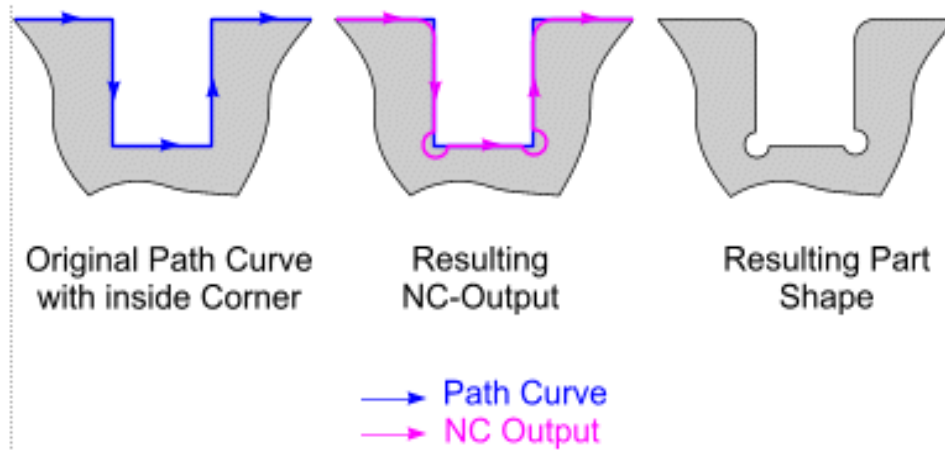
Rounded

- The path rounds inside corners. Specify the radius.



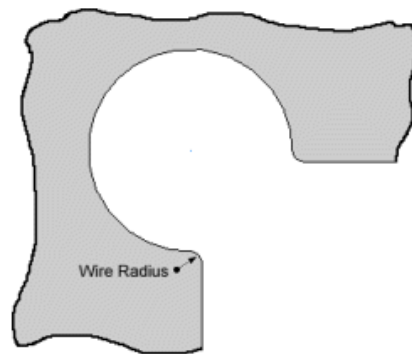
Modifying both inside and outside corners

- Both inside and outside corners are modified as shown below.



Auto round

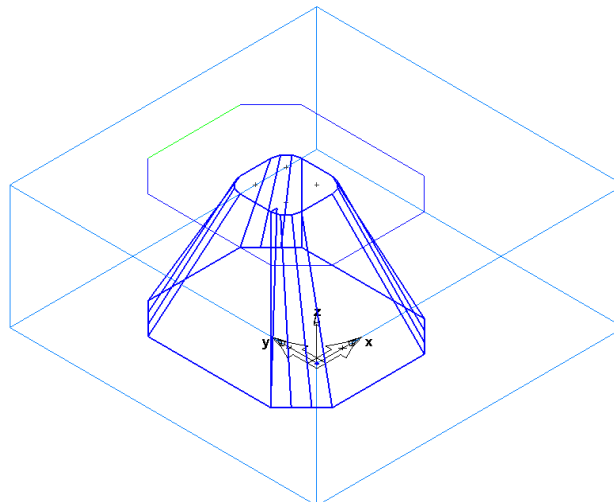
- If **Auto round** is selected, arcs are inserted at all sharp corners. This is applicable only if **Cutter comp** is set and you have a leave allowance or if **Cutter comp** is not set. If you have selected **Modify outside corners**, it will not perform any further rounding on these corners.
- It will round inside corners (even if **Modify inside corners** is selected) by inserting an arc before and after the circular corner as shown below: The radius of the inserted arcs is equal to the radius of the wire.



Four Axis Wire paths




4-axis Wire EDM requires two **closed curves** for **Die** and **Punch** features. The parts shape is controlled by a **upper curve** and a **lower curve**.



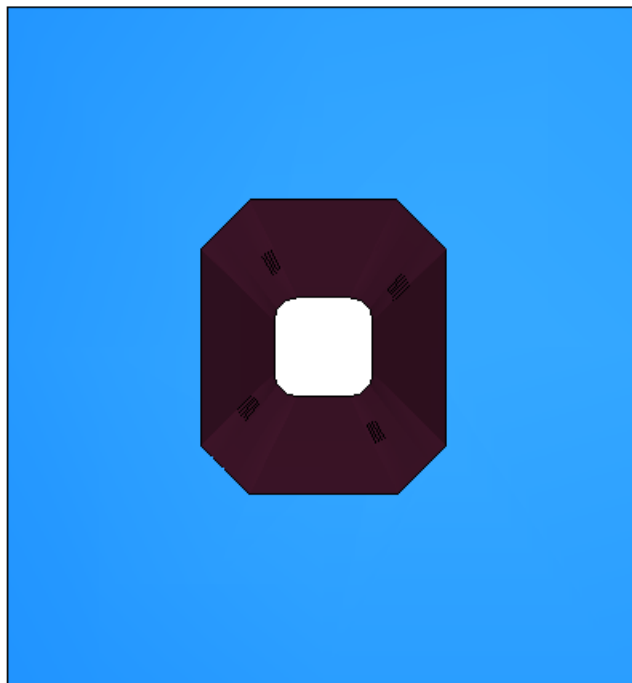
- 1 Open the file **100 x 25.fm** from the **Data** folder.
C:\Training_Data\FeatureCAM Course Data 2017\FeatureWIRE Metric Data
- 2 Create a new **4 Axis die** feature. Select **Next**.
- 3 Use the **inner geometry** for the **upper curve** and the **Outer Curve** for the **Lower Curve**.
- 4 Because there are no curves in the file use the **Curve chaining** button to **chain** the **curves**.

If you have not yet created a curve, you need to do so before creating your feature.

1. Use the geometry constructors to create the outline of the feature.
2. Then use chaining mode to connect the geometry segments together to make a curve.



- 5 Press Next after chaining each of the curves.
- 6 Set Thickness to **50mm**.
- 7 The feature may appear to be twisted because we have not set the start points.
- 8 Go to Top view.
- 9 On the New Feature – Start page check the Pick perpendicular lead check box.
- 10 Use the Pick arrow on the dialog and Set both upper and lower curve start points to the centre of the stock and the left edge of the curve.
- 11 Press **Preview** to see the updated feature
- 12 Select Finish and select Isometric View.
- 13 Since the same number of geometry pieces is used on the upper and lower curves, the match points came out nicely.
- 14 Run a **3D Simulation**.
- 15 Use the **Select arrow** to remove **Centre core piece** from view.
- 16 **Rotate** the view to see the part from the bottom.



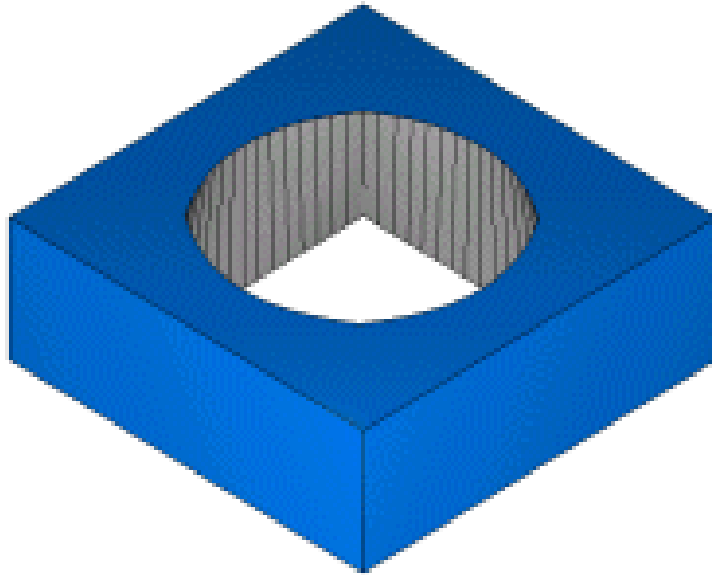
Leave the file open for the next exercise.

4-Axis Die



The 4-axis die feature needs two closed curves. It is assumed that the region outside of the curve is the part that you want to keep.

- The wire travels on the inside of the curves.



- The **Thickness** parameter is used to access the appropriate cutting data table.
- The **A** parameter is used to rotate the feature around the Z-axis of the current setup.

4-Axis Punch

- The Punch feature requires two closed curves. The wire travels on the outside of the curves.
 - The Thickness parameter is used to access the appropriate [cutting data table](#).
- 1 The A parameter is used to rotate the feature around the Z-axis of the current setup.
 - 2 Using the part **100 x 25.fm** *again* from the **Data folder**.
 - 3 **Delete** the **die1 feature** from the **Part View** (if you left the file open as instructed).
 - 4 Create a New **4 Axis Punch** feature.
 - 5 Use the inner geometry for the upper curve.
 - 6 Use the outer geometry for the lower curve.
 - 7 Set **Thickness** to **50mm**.

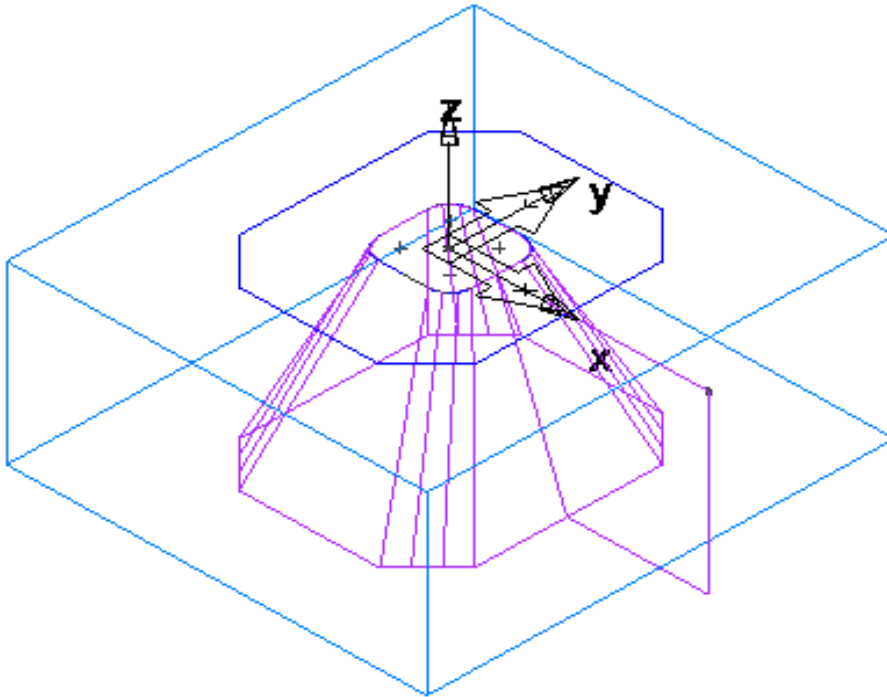


Note: There is no need to use Curve chaining this time because the curves are already in the file from the previous exercise. If you closed the file you will need to use the Curve chaining method as shown in the previous exercise.

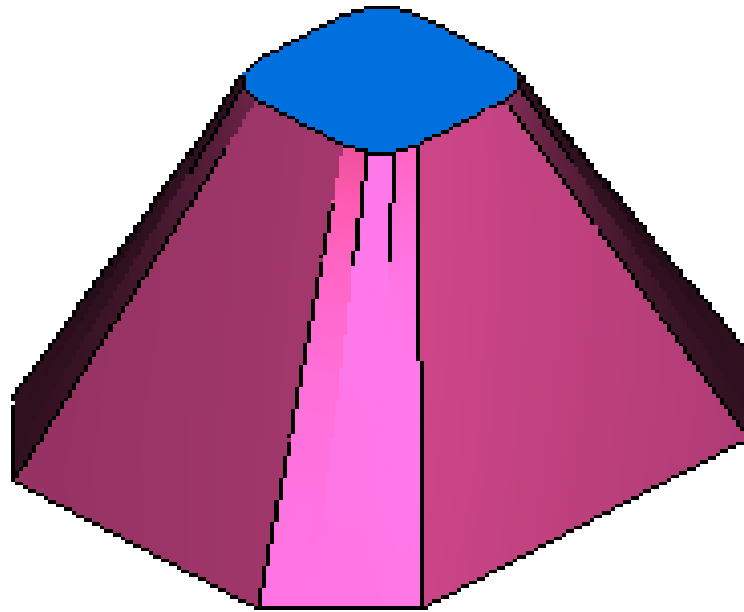


*There is a Geometry Point at **X70 Y0 Z0***

- 8 Set both upper and lower curve start points to the outside of the stock and the left edge of the curve using the point we have just created.
- 9 Press the **Apply button** to see the updated feature.
- 10 Run a 3D Simulation.



- 11 Use the Select arrow to remove the outer portion, or the stock from view.

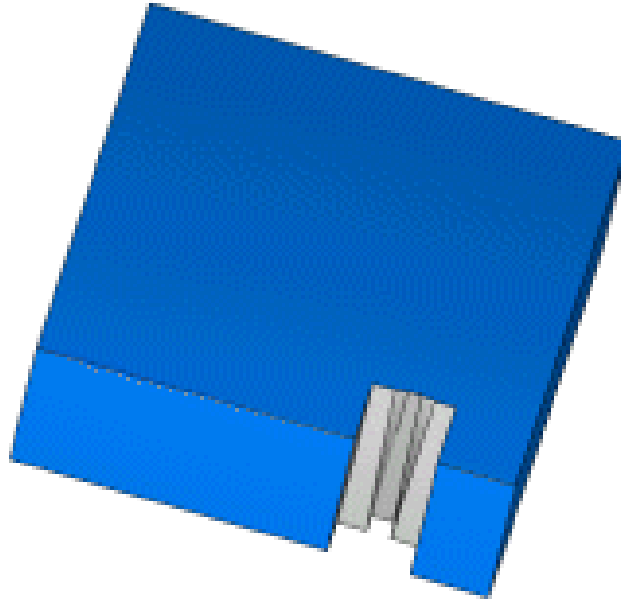


12 Close the file without saving!

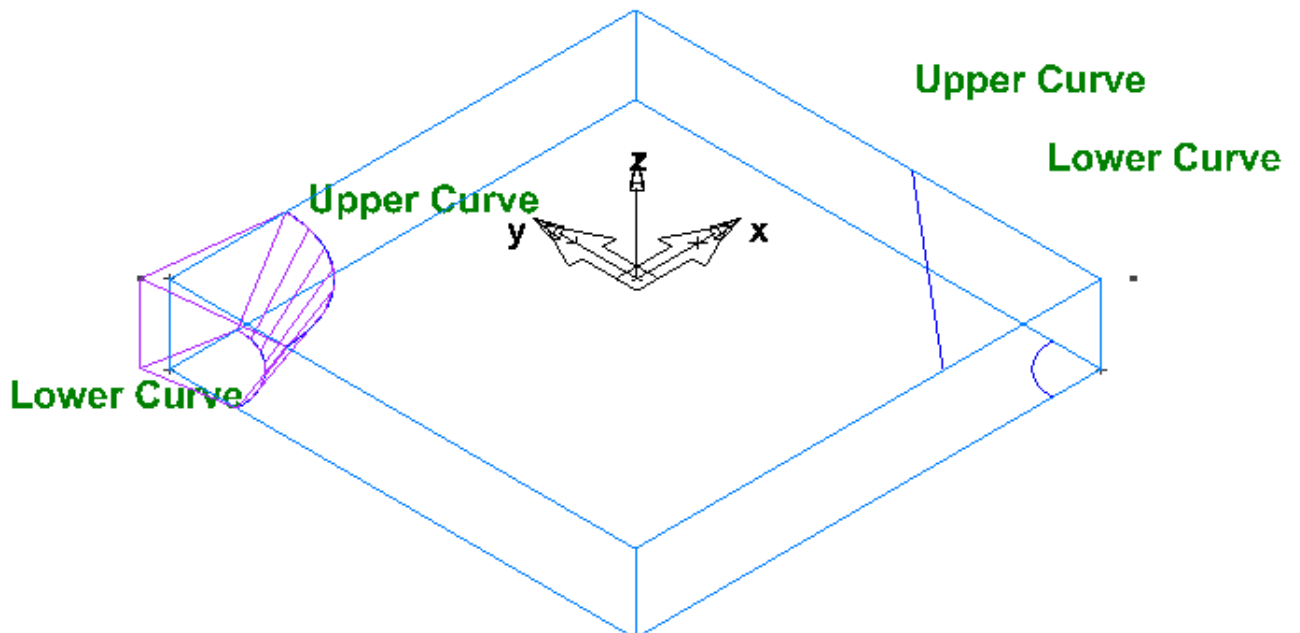
4-Axis Side



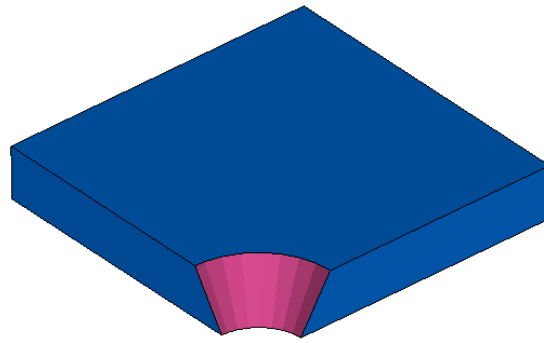
The side feature can use a closed or open curve. Punch or die features provide more cutting options for closed curves. You can optionally place the wire on either side of the curve. A side feature **must have at least three arcs or lines in it including the lead moves.**



- 1 Open **4 Axis Side Metric.fm** from the **Data** folder.



- 2 Select **Top view** then **Isometric view** to see the feature.
- 3 Run a **3D simulation** on the existing feature saved with the file.



- 4 Use the **Select arrow** to remove the piece that is to fall away.



Notice the wire **started vertically at the specified Start point**, cuts the side feature, and finally **ended vertically at the specified End point**. The end and start points are in the exact location for this feature.

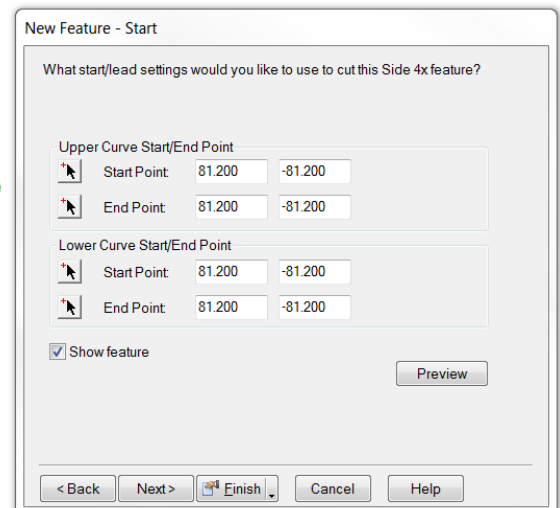
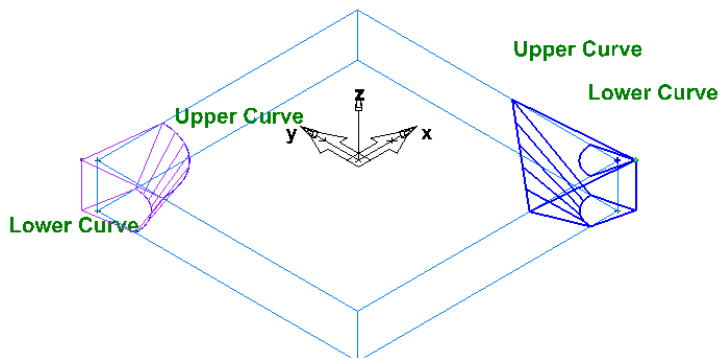


Hint: The exercise that follows will use an existing Geometry Point to snap to when setting Start and End points in the new feature.

- 5 Create a **new 4 Axis Side** feature.
- 6 Use the curves on the opposite corner of the part. Select **Next** twice.
- 7 Thickness is 25.4. Select **Next**
- 8 On the **New Feature – Start** page use the pick arrows **to snap to the point near the corner of the part** to set the Upper and Lower curves to set the Start and End Point for each curve.



You should see something similar to the image below.



- 9 **Uncheck the side4x2 feature** that was already saved in the file (**Part View**)
- 10 Run a **3D Simulation**

- 11 Use the **Select arrow** to remove the excess piece.



Close the file without saving!

4-axis rapid

- A **Rapid** feature is used to rapid along the length of a curve to the start point of the next feature. You can use **Rapid** features to quickly move around a part using curves to avoid possible collision with fixtures.
- FeatureCAM programs a **Rapid** in a very similar way to a **Side** feature. The wizard pages are similar, but fewer options are needed.



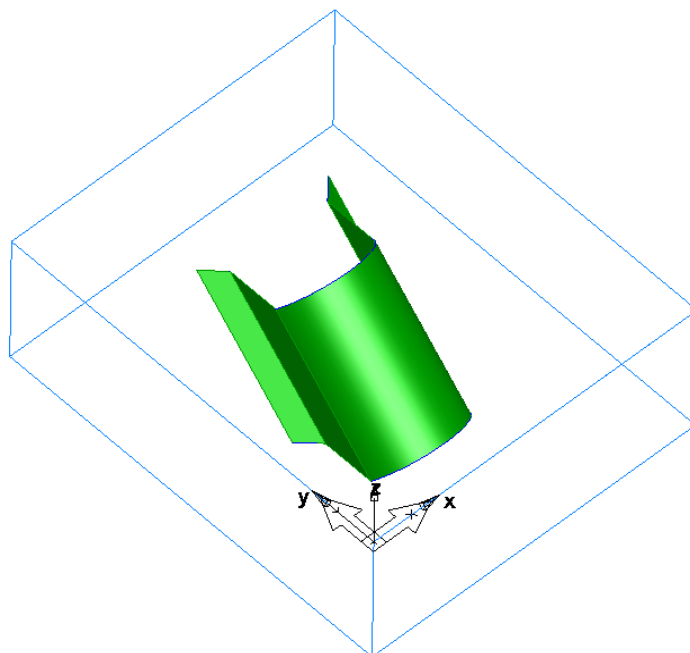
As an exercise, try to come up with a scenario where you would need to use a 4 Axis Rapid feature to maneuver the wire around and avoid collisions. This can be done at a later time, if you wish.

3D Curve in 4-axis



This is useful when you get a model from someone and the surfaces that you want to cut do not give you planar curves.

- 1 **Open** the file named **3d.fm** from the **Data folder**.
- 2 Shade the surface and rotate the view.
- 3 **Unshade** the **surface**.
- 4 Make a **4-axis side** with **Top-curve as upper** and **Btm - curve as lower**.
- 5 Using **Resize** set **Top and Bottom to Zero**.
- 6 Select **Finish**.
- 7 **Shade** the **surface**.



- **Rotate** the part around to show that it matches with the surface.



Note: If you were to actually want to wire this shape in the part you would need the surfaces to extend to the top and bottom of the stock.



Have a go at machining this part yourself.

Examples of FeatureWIRE Features

- 1 Open the file named Tutorial Features from the Data folder.



*Please note that this document is in **Inch** units.*



Click on each Setup and run a 3D simulation, one at a time, to see the feature examples included in the file for a better idea of what you could do with FeatureWIRE.

