Delcam Designer 7
Training Course

Delcam plc,
Talbot Way, Small Heath Business Park, Birmingham, B10 0HJ.
www.delcam.com

Important Notice

This document is supplied as part of a Delcam Training Course. It is not intended to be distance learning material: rather as an aid for Tutors when presenting material to course delegates and as a subsequent aid memoir to those delegates.

Delcam does not accept responsibility for any personal belongings / valuables whilst on the premises. Delegates are advised to keep their belongings on their person at all times.

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

The software described in this document is furnished under a license agreement and may be used only in accordance with the terms of such license.

Copyright © 2008 – Delcam plc. All rights reserved

Training Centre
Tel: 0121 683 1050
Fax 0121 7665511

Customer Support
Tel: 0121 683 1010
Fax: 0121 7665542

Delcam Designer Contents

Chapters	Page Number
1. Introduction	1.1 – 1.10
2. Wireframe for Design	2.1 – 2.40
3. Workplanes	3.1 – 3.8
4. Surface Modelling	4.1 – 4.26
5. Solid Modelling	5.1 – 5.10
6. 3 Stone Ring Design	6.1 – 6.32
7. Tapered Ring Design	7.1 – 7.26
8. Chain Link	8.1 – 8.16
9.Twisted Collet	9.1 – 9.8

Appendix - Further Exercises

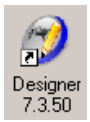
Carved Bowl	1 - 24
Decorative Column	1 - 20
Corbel Exercise	1 - 34
Guitar Body	1 -18
3D Scroll	1 - 14

1. Introduction

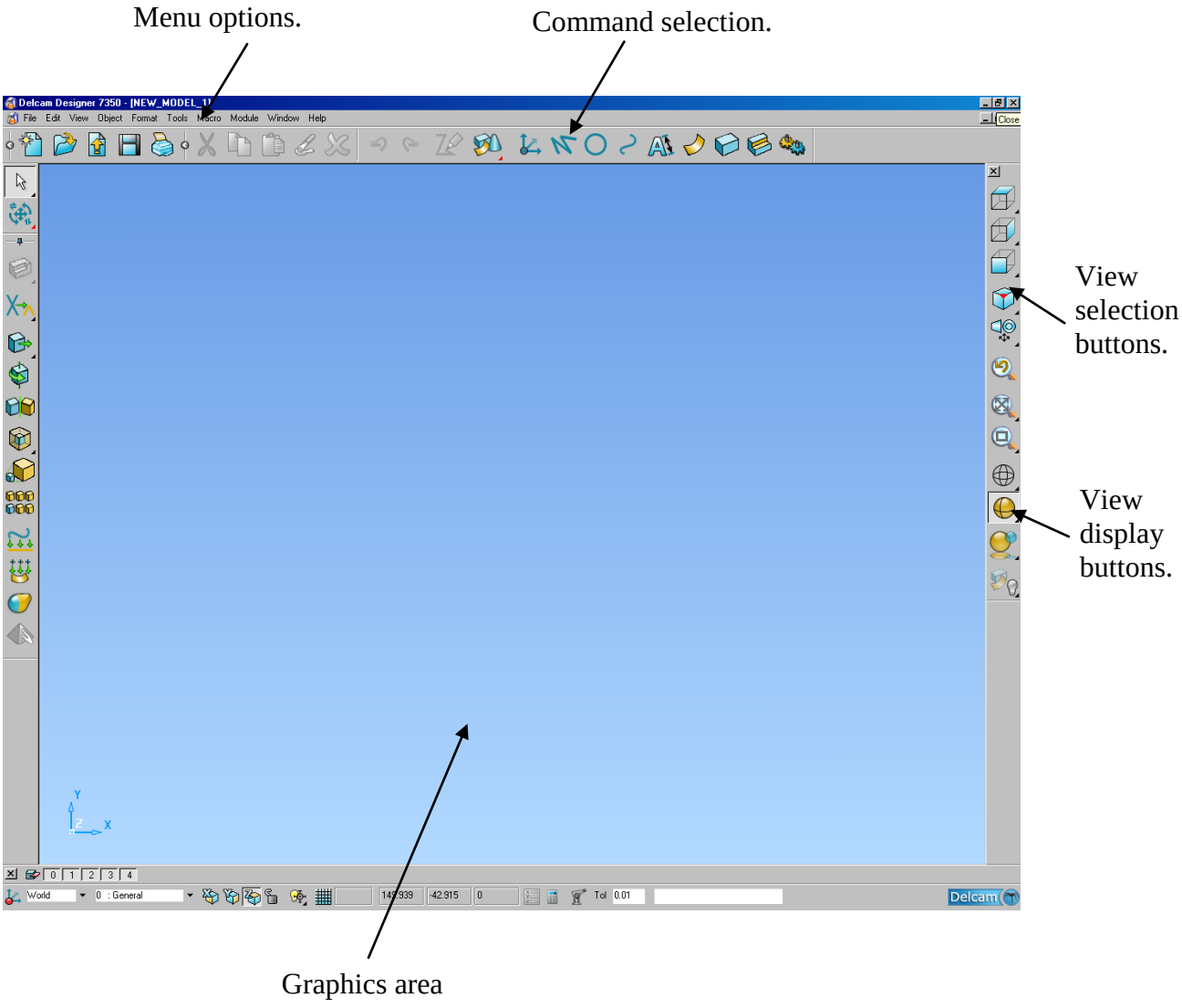
Delcam Designer Introduction.

Delcam Designer is a hybrid surface and solid Modelling package and has been created from **Delcam's** main CAD package called **PowerSHAPE**. Delcam Designer allows complete freedom in 3D design, yet at the same time is easy to learn and use. Interaction between **ArtCAM** is such that you have the ability to copy models or wireframe between the two interfaces.

To open **Delcam Designer**, **double click** on the Delcam Designer icon on the desktop.



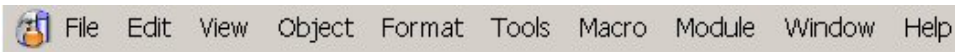
Once loaded, the screen looks as shown below.



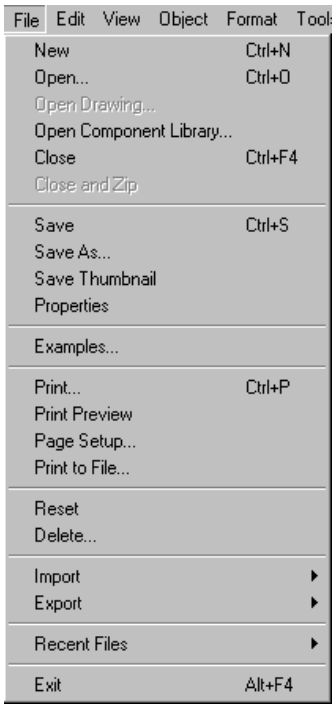
Delcam Designer automatically loads a New Model to work in. The name (New_Model_1) is given at the top left hand corner.

This model can be saved with a new name or closed so a saved model can be opened up to work on. Delcam Designer has the ability to have several models open and data can be copied from one model to another.

At the top of the screen there are a series of pull down menus.



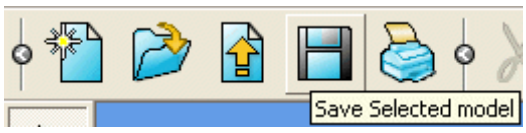
- Select the **File** menu, using the **Left** mouse button.



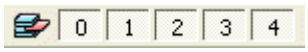
This menu contains a comprehensive list of operations available.

Beneath the pull down menus are the icons which when pressed bring up a form or do an action. The picture on the icon represents the action it will perform; for example the one with the floppy disc will save your model.

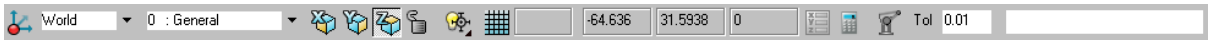
If you leave the mouse over an icon for a second, a description of the icon will appear.



To the bottom left is the levels menu.



At the bottom of the screen is the **Workplane Selector**, **Principal Plane** selector, the grid definition, the cursor position, the tolerance and the **XYZ** position input area.

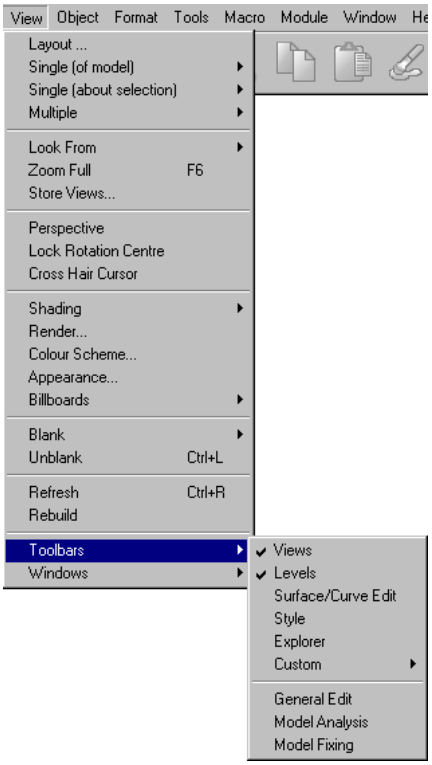


To the top right are the options Workplane, Line, Arc, Curve, Annotation, Surface, Solid, Feature, Assembly and Wizards.

When one of these icons is selected a context sensitive menu appears to the left of the screen. The context sensitive menu displays all of the options for creating the type of object selected.

- **Select** the menu for **line creation**.

The context sensitive menu on the left now shows the options for creating lines.



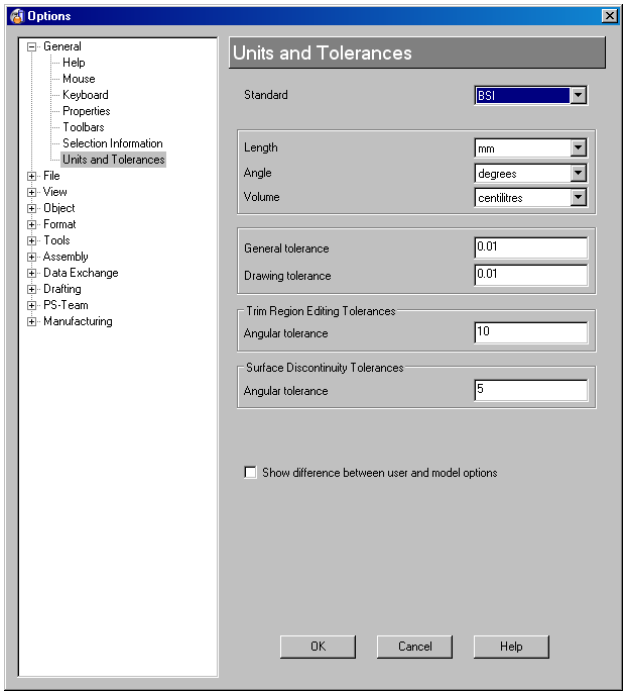
Toolbars

All of the toolbars, can also be switched on and off from the **View** pull down menu and selecting

View ➔Toolbars ➔.

Delcam Designer Options

The **Tools ➔Options** page allows you to configure Delcam Designer and change the settings.



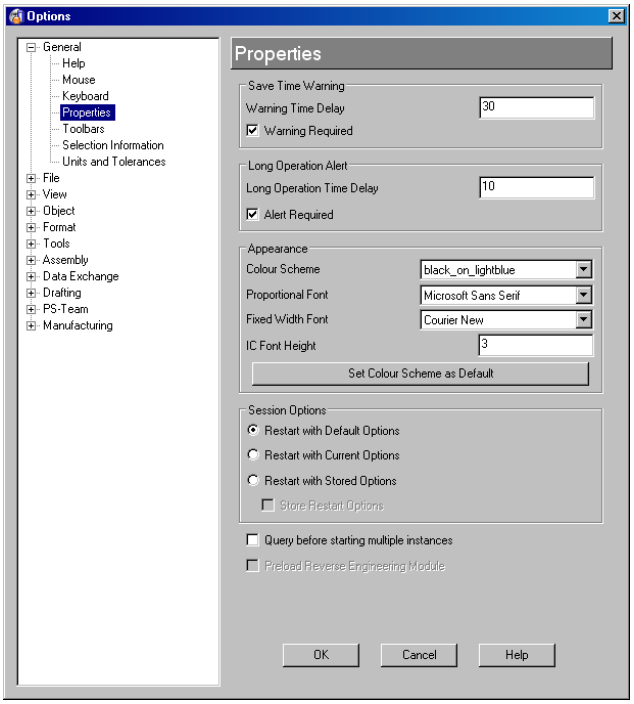
The options are stored under main categories.

By altering these options you can change how Delcam Designer works and you can save these changes so that they are used every time you open Delcam Designer. Alternatively you can make the changes just for this Delcam Designer session.

- Under **General** select **Properties**.

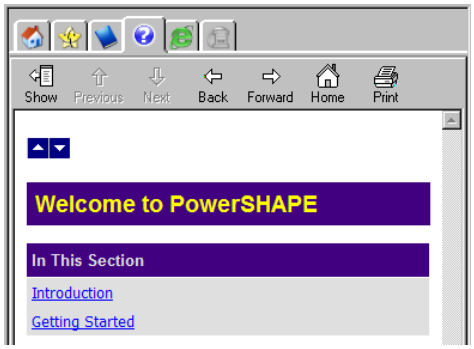
This is the main control area where you can save any changes and also set the time for Delcam Designer to remind you to save the model.

- Select **Cancel**.



Help Contents and Index

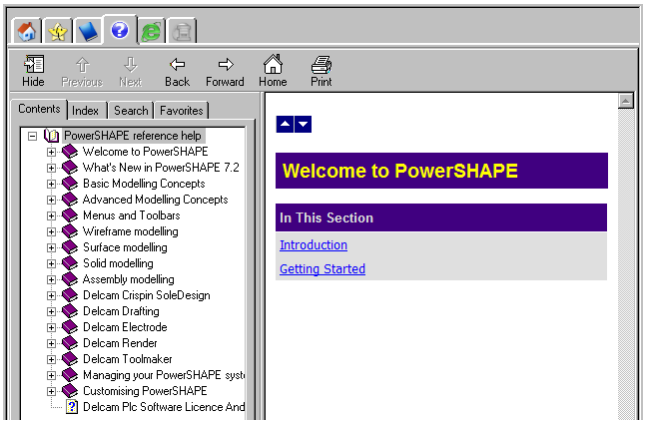
Delcam Designer has an extensive Help index that can be accessed by either selecting Help from the top menu or by simply pressing **F1** on the keyboard.



- Press **F1** on the keyboard.

The help is displayed in a new window on the left hand side of the screen.

- Press the **Show** button.



Topics are listed on the left side and the right side displays the help for that topic.

Select the Index tab for more search topics.

The Hide button can be pressed to hide the topics.

- Select the **home** icon to access the **Learning Assistant** and **Interactive Tutorials**.

- **Close** the help window using the hide button in its top right hand corner.

Mouse buttons

Each of the three mouse buttons performs a different operation in Delcam Designer. By using the ALT, Ctrl or Shift key, these operations can be extended as the following details explain.

Left Mouse button 1: Picking and selecting

This button is used for selecting items from the main pull down menus, inputting and selecting parts of the model.

To select multiple items, either –
Press the left mouse button and *hold it down* while moving it over the items you wish to select
OR-
Use Shift+Click or Ctrl+Click to change the selection. Shift will **add** items – Ctrl will **toggle** items – non-selected items will be selected, selected items will be de-selected.

Middle Mouse button or wheel: Dynamics

Zooming: - Hold down the CTRL key and button/wheel and move the mouse up and down to zoom in and out of the view. Hold down the CTRL, Shift Key and button/wheel to select an area by box to zoom into.

Panning: - Hold down the SHIFT key with the button/wheel and move the mouse in the direction you want to see.
Rotating: Hold down button/wheel and move the mouse. A tracker ball shows the rotation as does the view orientation axis in the bottom left corner of the screen. If view spinning is on the view will spin around until you click again.

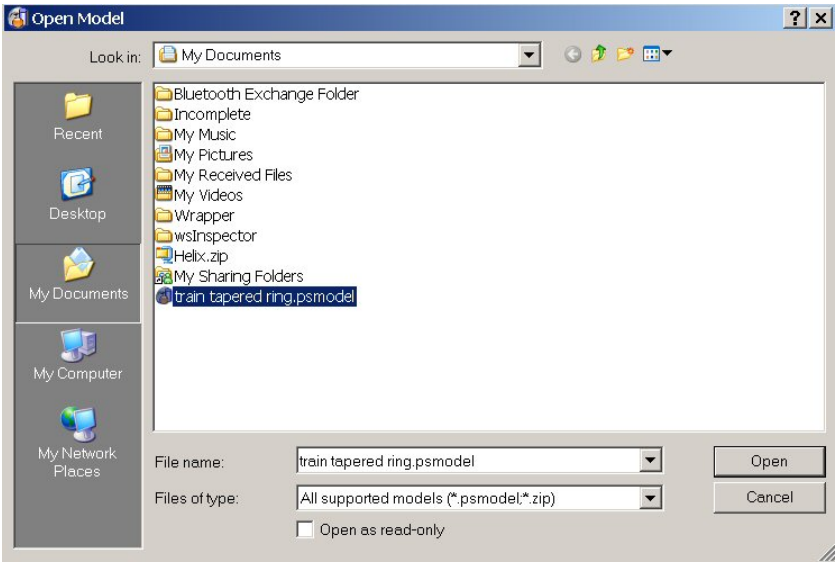
Mouse button 3: Special Menus

When this button is pressed it brings up a new menu based on whatever the mouse is over. If nothing is selected, the View menu appears. If a line is selected, for example then the Line menu appears.

Exercise 1: Dynamic mouse controls.

For this exercise you will be opening a stored model and then changing the views around and using the dynamic option.

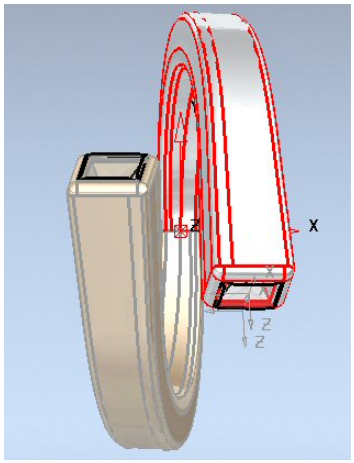
- Click on **File**, select the option **Open.**



This dialogue allows you to browse for a model.

The model format for Delcam Designer is **.psmodel**.

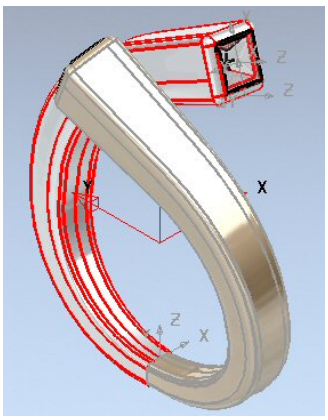
- Locate **D:\users\training\Delcam Designer Data**.
- Select the model **train tapered ring** and press **Open.**



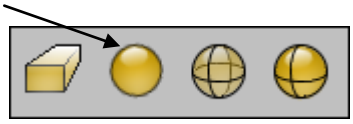
When a model is opened it is shown in a view down the Z-axis.

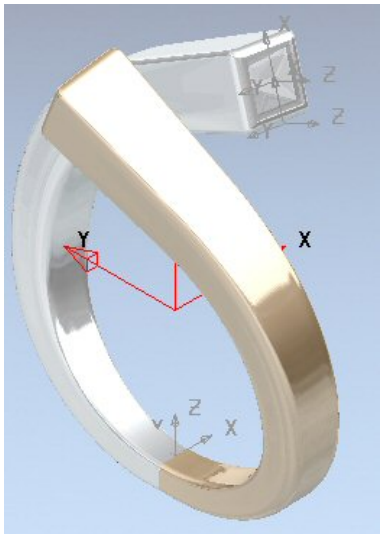
The view toolbar, on the right hand side of the Delcam Designer window, can be broken down into 3 main areas, **standard views**, **dynamics** and **shading**.

- **Select** the view **Iso 1**.



The model is now positioned in an isometric view. To provide better clarity a shaded version can be shown, instead of the wireframe.

- From the views toolbar select **Shaded view** pull out menu.
- From the available options pick **Shaded View.** 

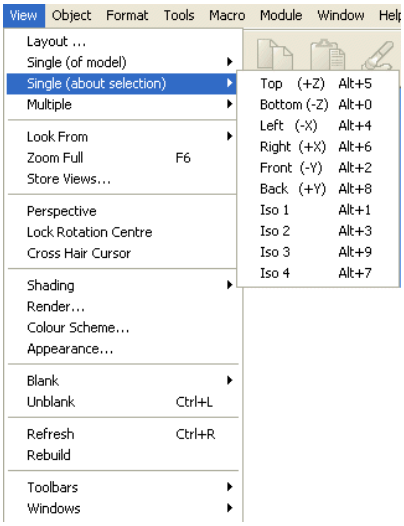


The surfaces have been shaded in to make it look like a completed model.

- **Hold down** the **middle mouse button** and **move** the **mouse** around to rotate the model.

- **Hold down Shift** and **middle mouse button** and pan the model around the graphic area.

There are many different options for viewing the model that you can select. There are also a variety of keyboard shortcuts that you can use.

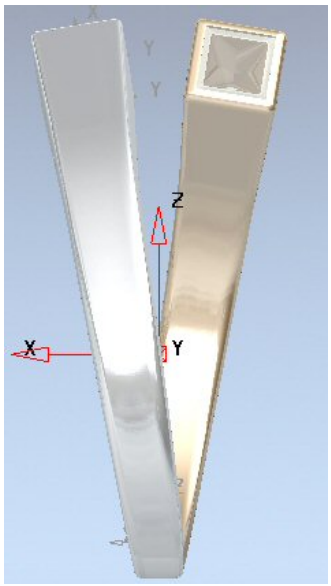


- Select **Views ➔ Single [about selection]**.

The keyboard shortcut for the views are listed, i.e. **Alt + 1 for Iso 1**. By switching **Num Lock** on your keyboard, you can use the number pad with the Alt key to change the views.

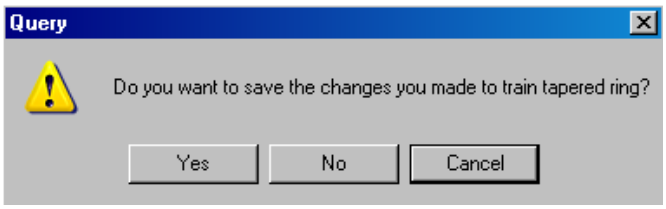
There is also a Perspective Mode option, which displays the model in any view as in real life with the parts nearer the front shown larger while the parts further away seem smaller.

- **Select Back [+Y].**



Another view is displayed. The Delcam Designer model needs to be closed. In this case it does not need to be saved, as the model has not been altered.

- Select **File ➔ Close**.

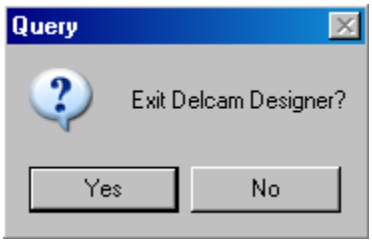


A message box appears asking if you want to save these changes. No changes have been made so the answer is NO.

- Select **No**.

The model called golf-fin is closed and this leaves the New-Model_1:1 open to work on if requires. This model needs to be closed to exit Delcam Designer. To close this model and exit Delcam Designer at once, select Exit.

- Select **File ➔ Exit**.



A message box appears asking if it is okay to exit Delcam Designer.

- Select **Yes**.

Delcam Designer has been closed.

Delcam Designer models

Delcam Designer models can consist of a variety of different entities, which fall into three main types, **wireframe, surfaces**, and **solids**.

Wireframe in Delcam Designer

Wireframe items include lines, arcs, curves, points, text, dimensions etc. These can be 2D or 3D and are essential for drafting in addition to generating most types of surfaces or solids.

Surfaces in Delcam Designer

Surfaces can be generated from wireframe, created directly as standard primitive shapes, or by conversion from a solid model. A surface is a skin stretched across the 4-sided elements of a wireframe network. A 3D model created as **surfaces** behaves as a hollow form.

Solids in Delcam Designer

Solids are generated directly from wireframe, created directly as standard primitive shapes, or by the conversion of a selection of surfaces. A 3D model created as a **solid** behaves as an enclosed mass.

2. Wireframe for Design

Introduction

Delcam Designer lets you sketch wireframe geometry using any combination of lines, arcs and curves. Delcam Designer uses a powerful **Intelligent Cursor (IC)** that continually feeds back positions, distances and tangency conditions. Once you understand what the IC is telling you, and how to use it, you will find that creating even the most complex geometry is fast and efficient.

As soon as you select any object in Delcam Designer, it will become highlighted (usually in yellow). Depending on the object selected you will also see varying degrees of 'instrumentation' (usually in blue and yellow). This instrumentation allows you to edit your geometry in a variety of ways, again using the IC.

Wireframe Line and Arc Options

The **wireframe Line** and **Arc** options are accessed from the **main** toolbar.

Single Line

Continuous lines

Rectangle

Polygon

3D box around selected items

Shortest line between 2 objects

Trimmed Chamfer

Untrimmed Chamfer

Full Circle

Arc Through Three pts.

Swept Arc About Centre

Trimmed Fillet

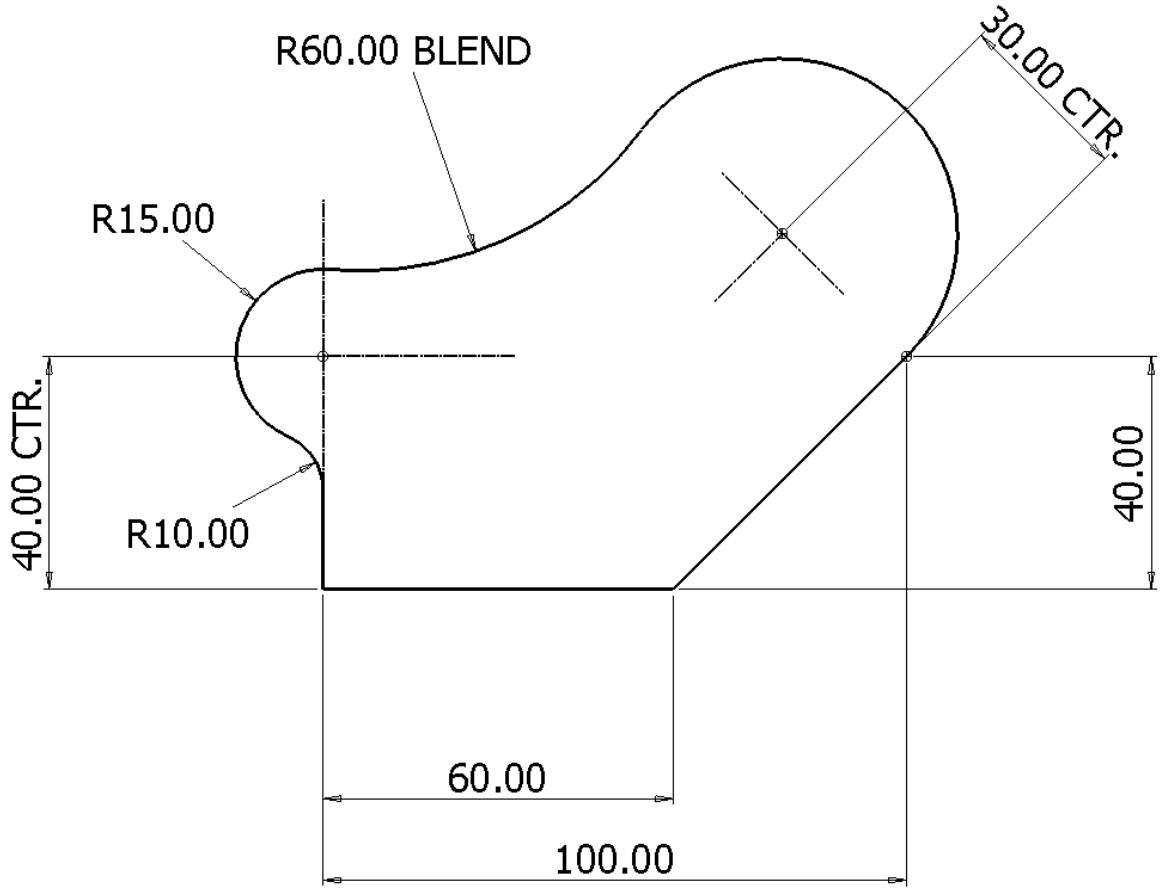
Untrimmed Fillet

- Restart Delcam Designer from the Desktop.

A ***new model*** will be opened in the **Delcam Designer** session.

Wireframe Lines and Arcs Exercise

The following tutorial will show the basic operation of the **Wireframe** modeller by recreating the ***2D profile*** shown below.



Line Creation

A **single line** will be generated from **0** to **Y40** to coincide with the **R 15 arc *centre***.

- **From** the **Line menu** select **Single Lines**.
- Enter the start co-ordinate of **0** and press **Return**.
- **Input** the values **0 40** in the **Command input box** (lower right of graphics area) and press **Return**.

The next part will be generated using **continuous lines**.

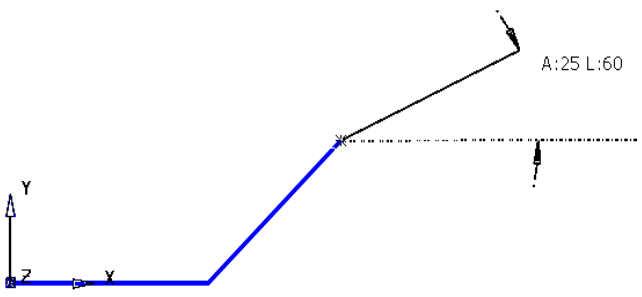
- From the **Line menu**, select **Continuous Lines** option.
- **Snap** to the **start** of the **first line** (or enter **0** in the ***command input*** box).
- **Input** the value **60** in the **Command input** box followed by **Return**.
- **Input** the value **40 40** in the **Command input** box followed by **Return**.

The Intelligent Cursor (IC)

The **Intelligent Cursor** displays preview information and locks onto ***key points*** and ***major directional*** axes during dynamic **wireframe** creation.
With the **continuous line** mode still active, a construction **line** to locate the **R30 *centre*** will be dragged ***dynamically*** while being monitored by the **Intelligent Cursor**.

Note:- If the **Intelligent Cursor** functionality does not appear, right click within the graphics area to open the local ***view menu*** and check that the option **Intelligent Cursor** is **ticked**.

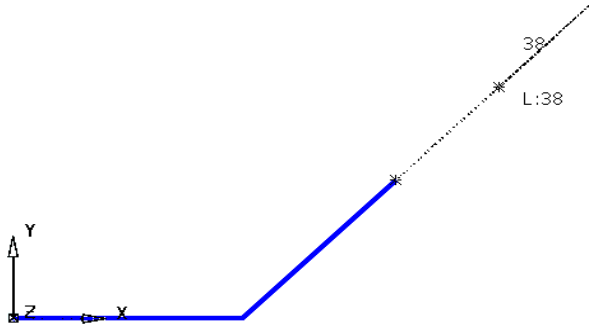
- By moving the mouse, dynamically **drag** the ***end*** of a new **line** away from the current position (as shown below) and observe the effect of the **Intelligent cursor** (<u>do not</u> click the left mouse key yet).



In the diagram the end of new line has currently been dragged to a relative, preview position at an angle of **25 degrees** with a length of **60**.

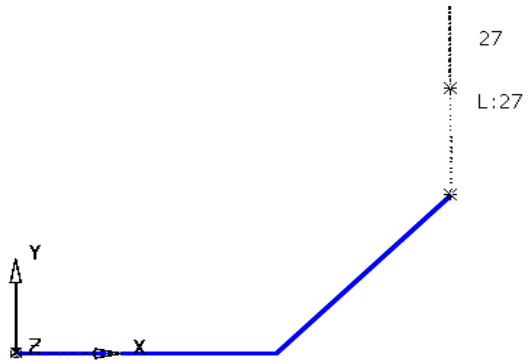
Note:- The **X** direction is **0 degrees** and the **Y** direction is **90 degrees** anticlockwise.

- By moving the mouse further, dynamically drag the end of a new **line** to a position ***tangential*** to the previous **line** and observe the effect of the **Intelligent cursor** (do not click the left mouse key yet).



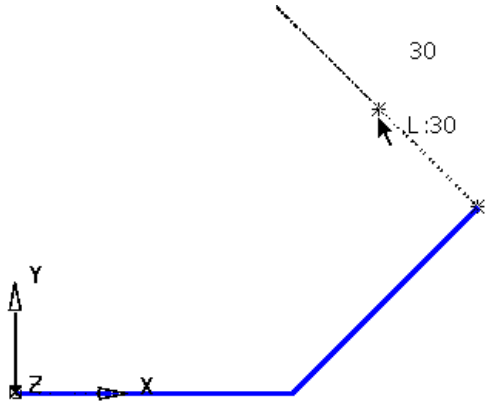
In the diagram the end of new **line** has currently, ***locked*** onto the ***tangency*** of the previous line and has been dynamically dragged away with a preview length of **38**.

- By moving the mouse further, dynamically drag the end of a new **line** position to a position along the **Y** direction and observe the effect of the **Intelligent cursor** (do not click the left mouse key yet).



In the diagram the end of new **line** has ***locked*** onto the **Y *direction*** and has been dynamically dragged away with a preview length of **27**.

By moving the mouse further, dynamically drag the end of a new **line** to a distance of **30** along the ***normal*** (**90** degrees) direction and this time ***click*** the left mouse key at accept the **line**.



The intelligent cursor has locked onto the **normal** (**90 degrees**) direction from the end of last **line**. The cursor has been dragged along this normal until a length of **30** is displayed.
The ***end position*** is accepted by applying a ***left mouse click***.

- Press **Escape** to exit **line** creation.

Arc Creation

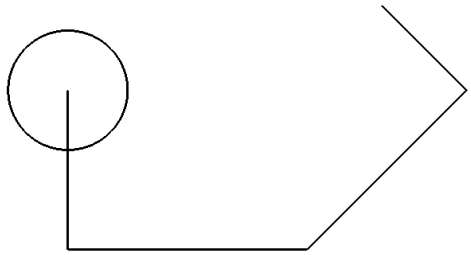
The remainder of the **wireframe** is to be created using various **arc** options.

- **From** the **Arc Menu** **select Full Arc** .
- Input the value **r 15** in the **Command input box** and press **Return**.

(This input will redefine the ***default radius***).

- **Move** the **cursor** over the open end of the vertical line until the text **End** is displayed then **click** the **left mouse key** to snap the circle centre position.

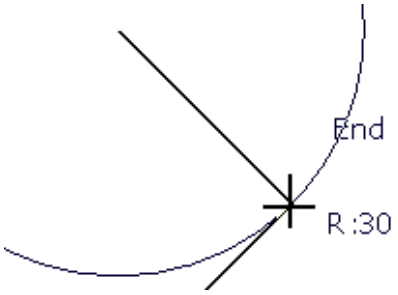
- Press **Select**.



If it is required to modify the **Circle**, quickly double click it with the left mouse key and the relevant **Arc** editor **Form** will appear.

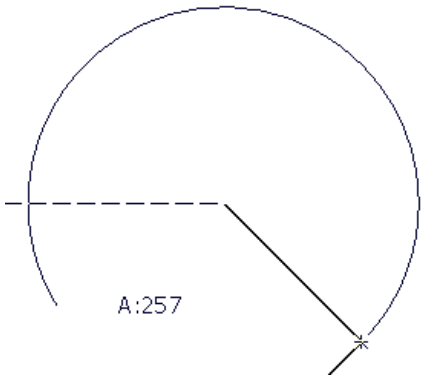
- **From** the **Arc Menu** select **Arc through centre**, **radius and span**.

- **Snap on** the **open end** of the **construction line**, for the centre point.
- **Move** the **cursor** to the **other end** of the **line** and **click** with the **left mouse button**.



The position is now set for the start of the **arc**.

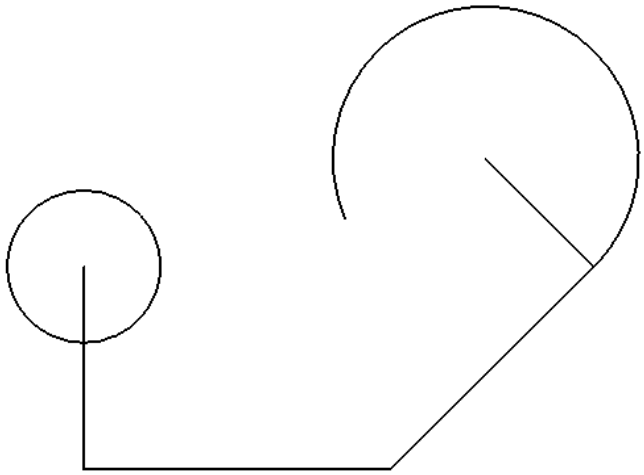
- **Release all mouse buttons** and **drag** the **arc around anti-clockwise**.



The **intelligent cursor** displays the current span angle.

When the left mouse button is clicked, the **arc** is created.

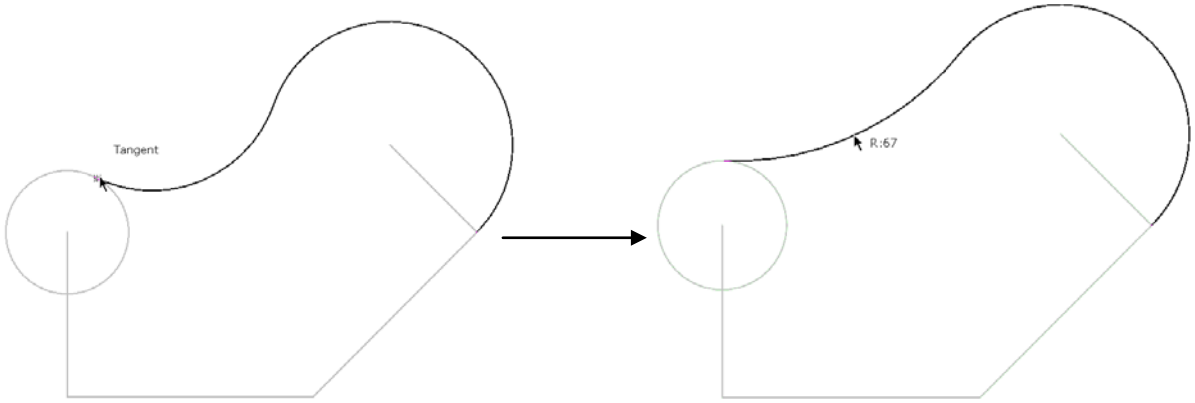
- **Click** the **left mouse button** when the arc reaches about **250 degrees**.



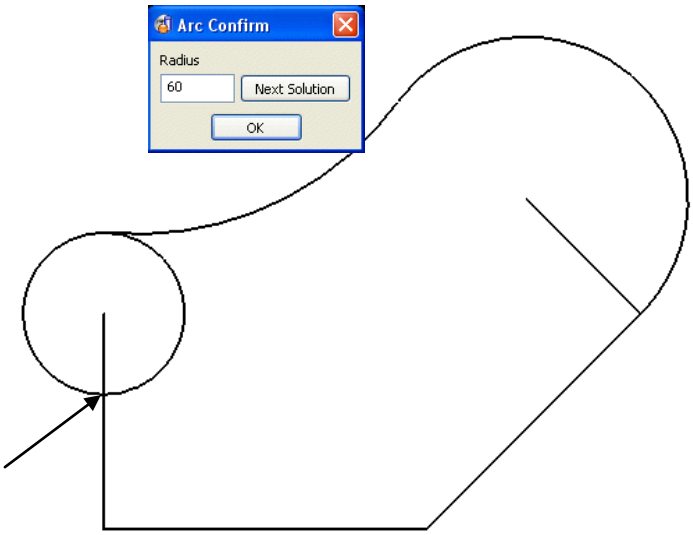
The blend radius (**R 60**) will now be generated using **Create an arc through three points**.

The 1st and 2nd points will be snapped as ***tangencies*** to the existing **full circle** and **arc** with the 3rd point being input as the specified blend radius value (**60**).

- **From** the **Arc Menu** select the **Arc through three points** option.
- Use the **intelligent cursor** to locate, and **Click** on **Tangent points** on the **arc** and **circle** (Do not locate a **Key**, **End** or **Mid point** as these are fixed positions).



- **Drag** the **radius point** to the required shape and to a value as close as possible to **60** before **left clicking** for initial **acceptance**.
- If necessary, In the **Arc Confirm** form **Modify** the **radius** value to **60** and toggle through **Next Solution** until the correct shape is achieved, before pressing **OK**.



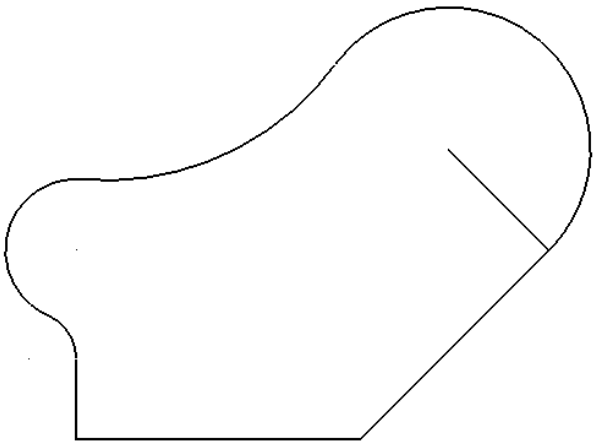
The new **arc** has been trimmed back both to the adjacent arc and circle.

The **Arc through three points** option will also be used to create the **R 10** fillet between the **line** running along Y and the remaining full **circle**.

- **Select** the **Arc through Three points** option.
- Use the **Arc through three points** option to create the **radius** of **10** between the **full circle** and the **vertical line**.

Composite Curves

In many applications it is necessary for the required **wireframe** to be a single entity. This is achieved by creating the wireframe as a **composite curve**.
Note:- A **composite curve** can only by driven along existing model entities.



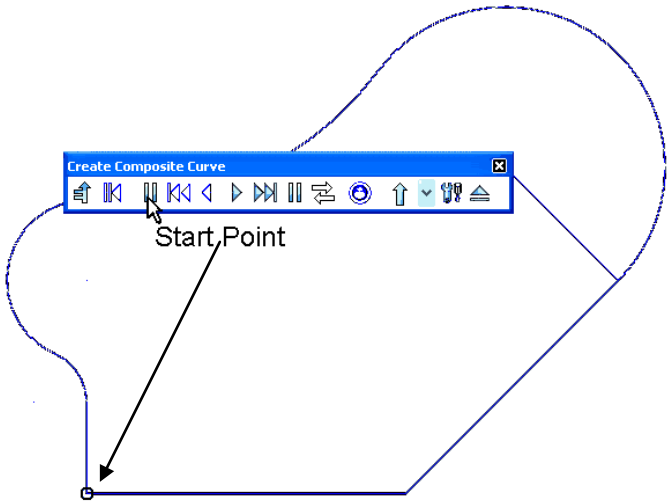
The basic **wireframe** shape now is complete but is currently made up of several separate **lines** and **arcs**.

The extent of a **composite curve** can be limited between selected **key** points along the potential route (**Define start point** and **Define end point**).

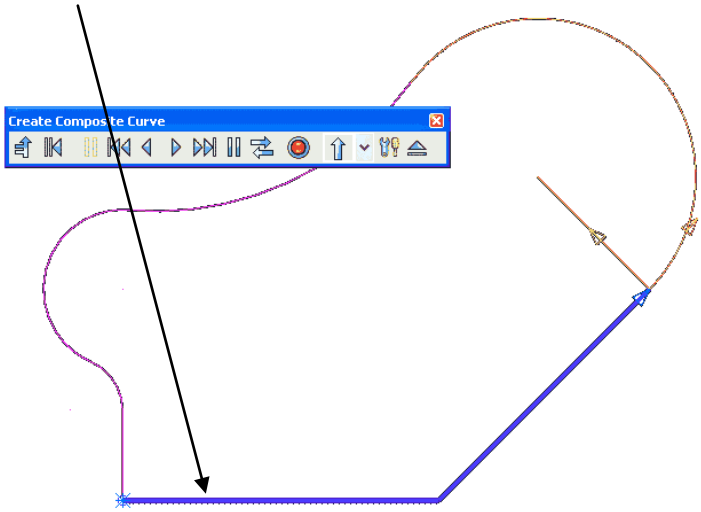
- From the **Curve Menu**, select **Composite curve**.

The **composite curve** toolbar appears.

- Select the **Start Point** option in the toolbar and ***left click*** the lower left corner on the **wireframe** model.



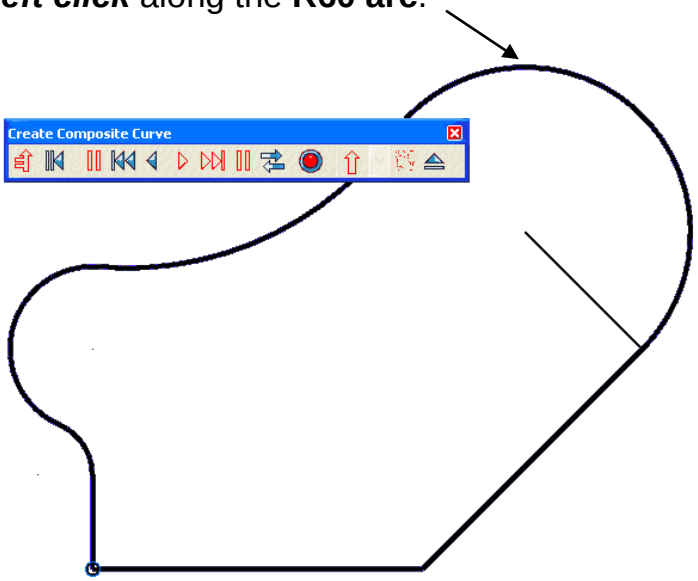
- ***Left click*** on the **bottom line**.



The **Composite Curve** is generated along the route until it hits a branch point where optional directions will be arrowed and change colour for the user to manually select the required route.

Note: The ***asterisk*** at the **start point** indicates that the **composite curve** is currently open.

- ***Left click*** along the **R60 arc**.



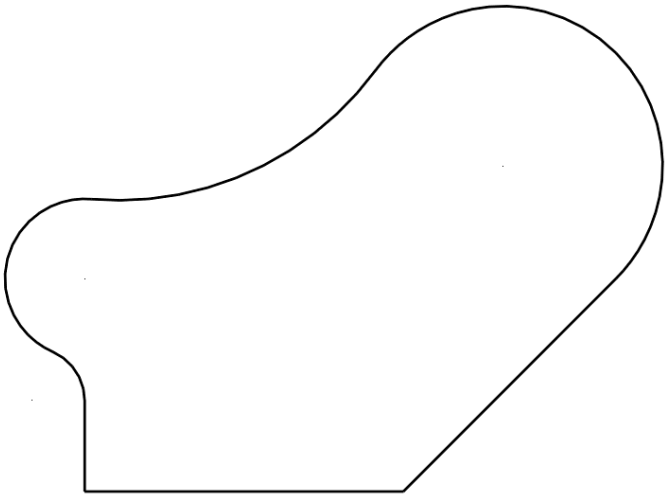
The **Composite curve** has now continued around the **wireframe** back to the defined **Start Point**.

Note: a *circle* at the **Start Point** indicates that the **composite curve** is a ***closed*** form.

- On the toolbar select **Save** to accept the **composite curve**.

- Select **Eject** to remove the toolbar.

- Select the **construction line** with the **left mouse button**.
- From the **right mouse button - menu**, select **Delete**.



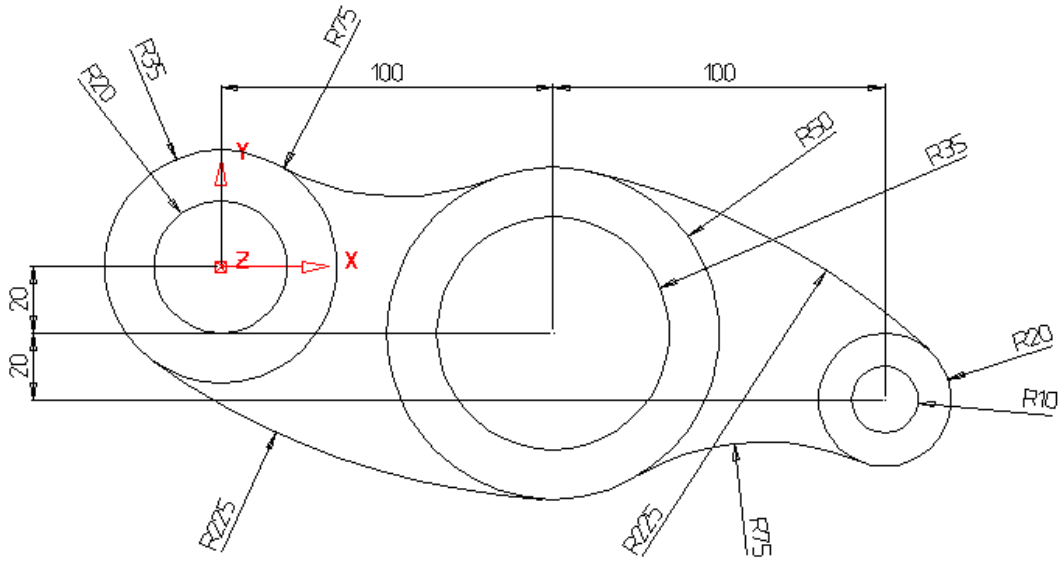
The **wireframe** is now complete and will be **Saved** as a permanent **model**.

- Select **File ➔ Save As**.
- In the form **Save the Model As**:-
  D:\users\training\COURSEWORK\Delcam Designer-Jobs\cam-example

- Select **File ➔ Close**.

Wireframe Arcs Example

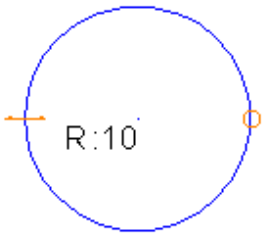
The next example demonstrates the use of **Arc**s to create the lever design shown below.



- **Open a New Model**
- **Select** the **Arc Menu**.
- **Select** the **Full Arc** option from the menu.

When this command is selected, the centre position of the circle is required first. This can be input as a value or by snapping using the **Intelligent Cursor**.

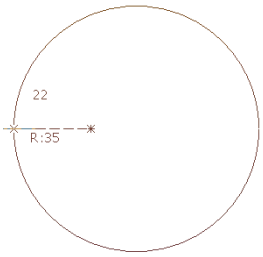
- In the **Command Input box** enter **0** for the circle centre and press return.



A preview of a circle has been generated with a **radius** input from the default value. The radius can be altered by dragging one of the displayed arrow to the required unit value.

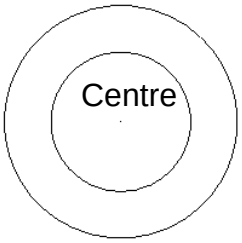
To accept the circle, press the **escape key**.

- With the **circle selected**, click and hold the handles (arrows) and dynamically move the mouse to **drag** a **new radius** of **35**.



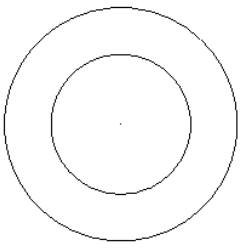
The **circle** now has the required **radius**. This method can be used to create the second circle at the same centre position, using the **Intelligent Cursor**. Alternatively the user can ***right click*** on the **arc** and select ***Modify*** from the local menu.

- **Select** the **Circle** command.
- **Position** the **cursor over** the **centre** of the **first circle** so the word ***Centre*** appears.
- **Click** the **Left mouse button** to accept the centre of the new circle.



This new **circle** is exactly the same initial radius as the first circle. To change the default, creation radius a new value is entered into the ***Command Input window*** i.e**. R 25**.

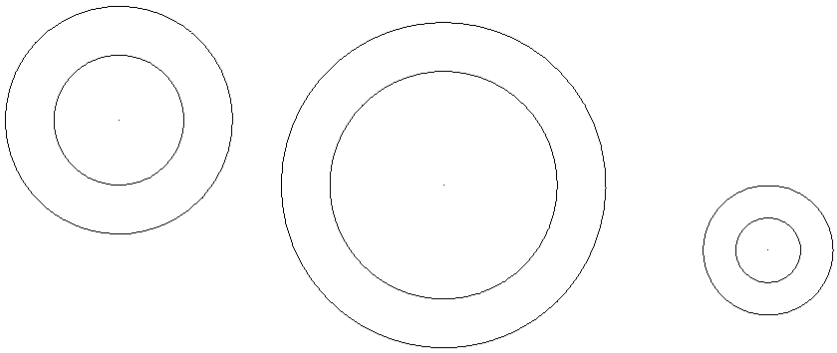
- **Drag** the second **circle** to a **radius** of **20mm**.



The two basic circles are completed. Further circles are required for this model.

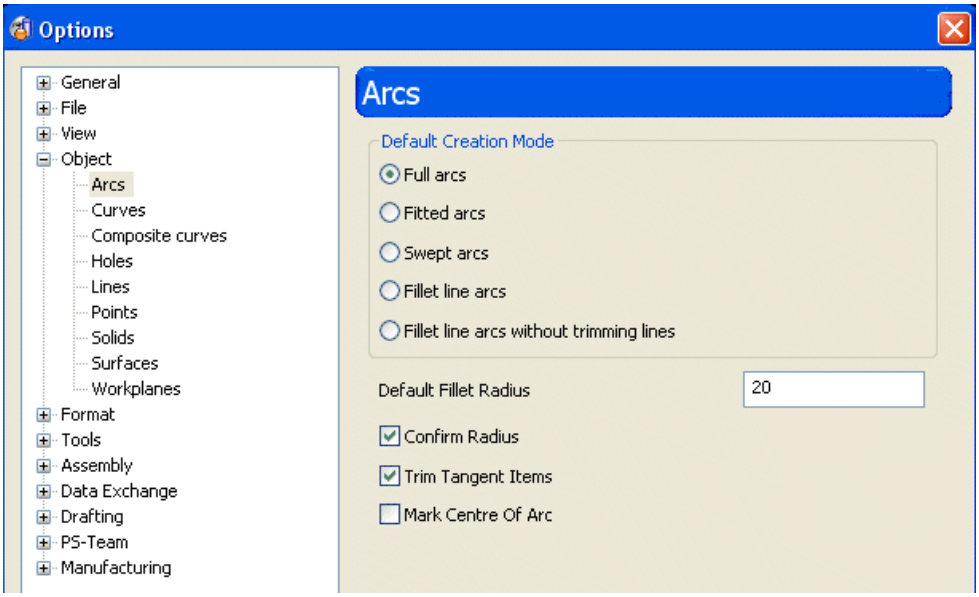
Hint: Use the command input box to enter the circle centre if it cannot be snapped.

- **Create** a **circle** at **centre 100 –20** with a **radius** of **50mm**.
- **Create** a **circle** at **centre 100 –20** with a **radius** to **35mm**.
- **Create** a **circle** at **centre 200 -40** with a **radius** of **20mm**.
- **Create** a **circle** at **centre 200 -40** with a **radius** of **10mm**.
- Press **Escape**.



The default option when creating tangent arcs is to trim back the associated geometry. In this case it is not required to do this, so the flag (**Trim Tangent Items**) is switched off. This is found in the **Options** form under **Tools** at the top of the screen.

- **Select Tools ➔ Options**, select **Object** then **Arcs**.

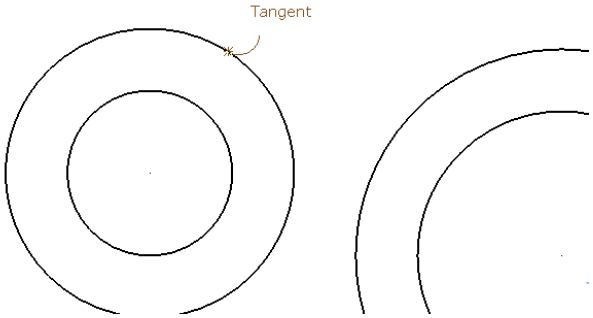


- *Untick* **Trim Tangent items**.

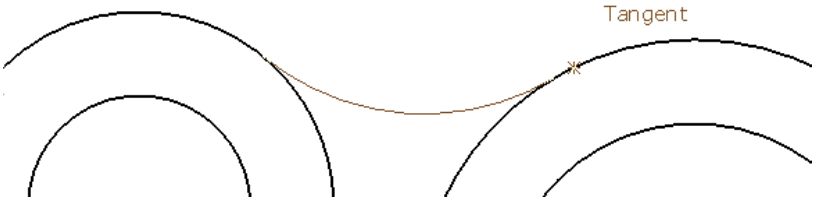
- *Press* **OK**.

For the tangent arcs, the fitted arc option is the most suitable as it provides full dynamic control through all possible combinations.

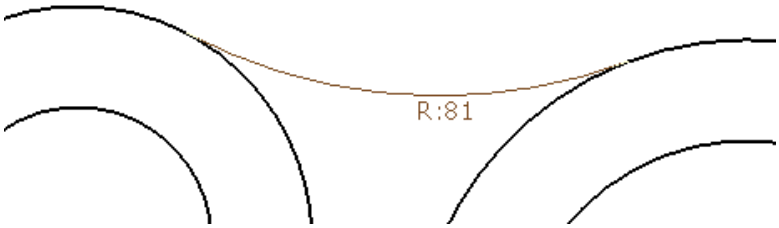
- From the **Arcs** menu select the **Three point arc** option.
- **Move** the **cursor over** the **outer left circle** until the word **Tangent appears** and **press** the **Left mouse button**.



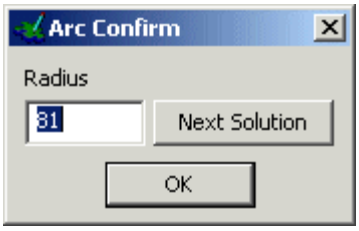
- **Move** the **cursor over** the **middle outer circle** so the word **Tangent appears** and **press the left mouse button**.



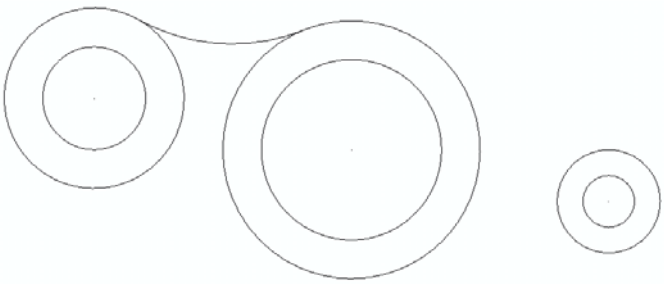
- **Move** the **cursor over up** and **down** to get the **tangent arc** in the correct place and **press** the **Left mouse button**.



The Arc Confirm form appears. This can be used to enter an exact radius.



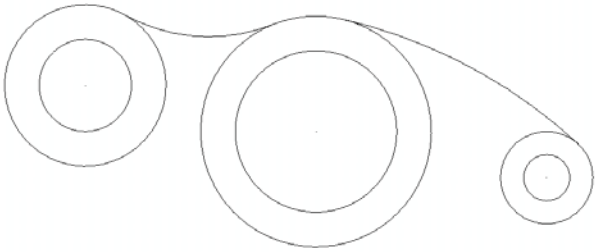
- **Enter** a **value** of **75** and press **OK**.



The tangent arc is drawn.

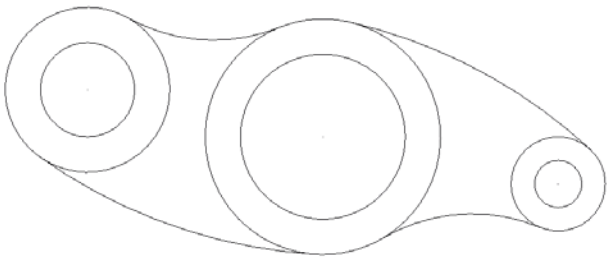
The command automatically repeats until exited by pressing escape or the selector.

- **Move** the **cursor over** the **middle outer circle** until the word **Tangent appears** and **press** the **Left mouse button**.
- **Move** the **cursor over** the **right outer circle** so the word **Tangent appears** and **press the left mouse button**.
- **Drag** a **radius** of **225** and **press OK**.



The second radius has been generated. The further two can be generated in the same manner.

- **Create** a **three point arc** between the **left outer circle** and the **middle outer circle** with a **radius** of **225mm**.
- **Create** a **three point arc** between the **right outer circle** and the **middle outer circle** with a **radius** of **75mm**.

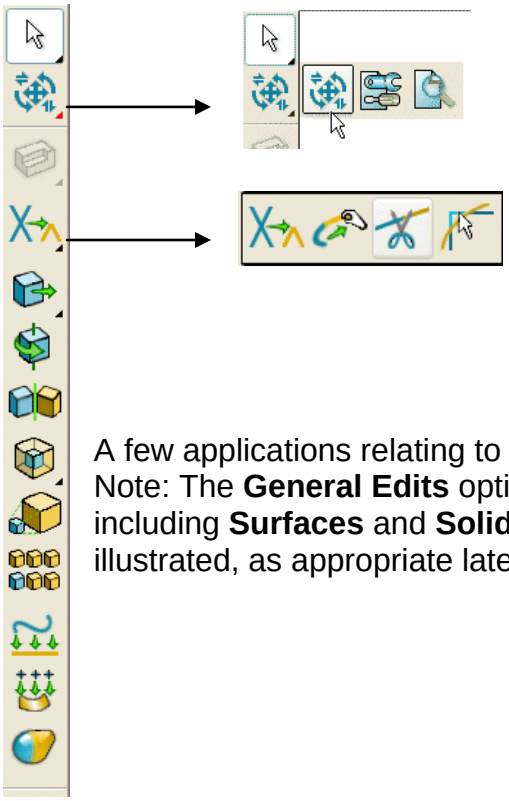


- Select **File ➔ Save As**.
- In the form **Save the Model As**:- **D:\users\training\COURSEWORK\Delcam Designer-Jobs\lever-example.**

- Select **File ➔ Close**.

General Edits

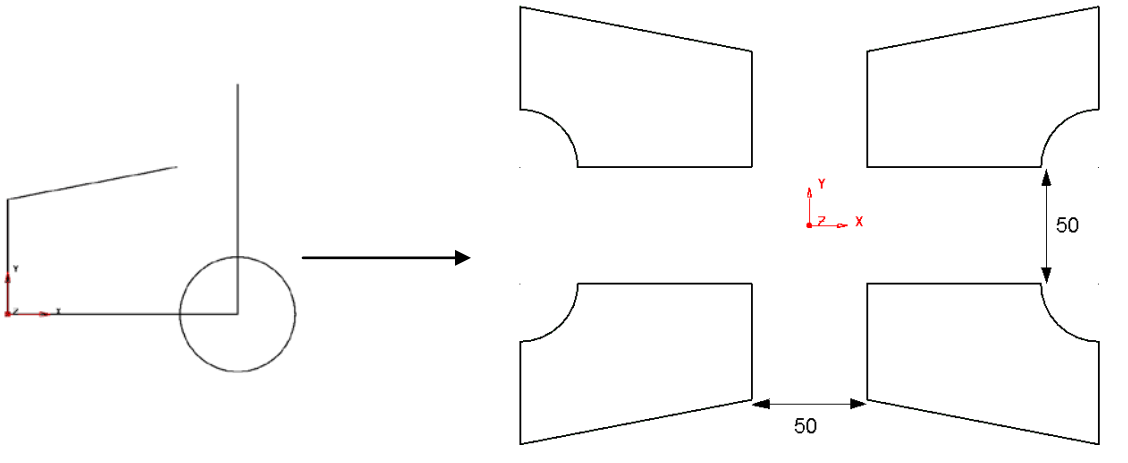
At the top of the left hand toolbar are 2 fixed icons, the second of which includes access to the **General Edits** toolbar (default).



A few applications relating to **wireframe** data will be illustrated in this section. Note: The **General Edits** options are also frequently used on other entities including **Surfaces** and **Solids**. Further **General Edit** options will be illustrated, as appropriate later on during the course.

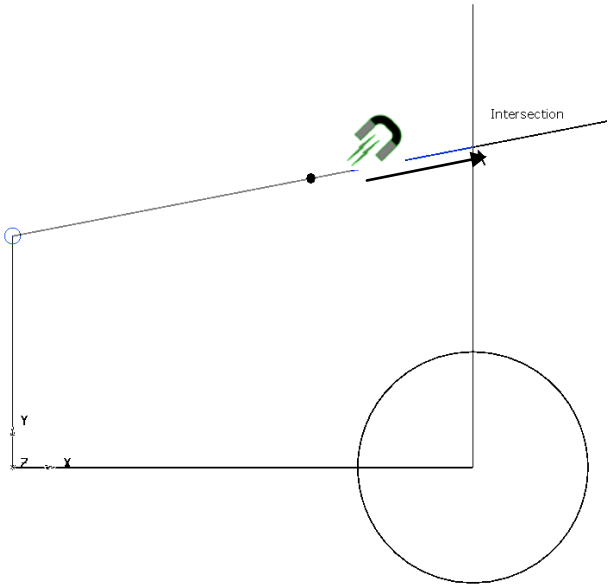
- Open a **New** model.
- Select **File - Import** and load in the **wireframe** model:

D:\users\training*Delcam Designer data*\ *GeneralEdit_Wireframe.dgk*

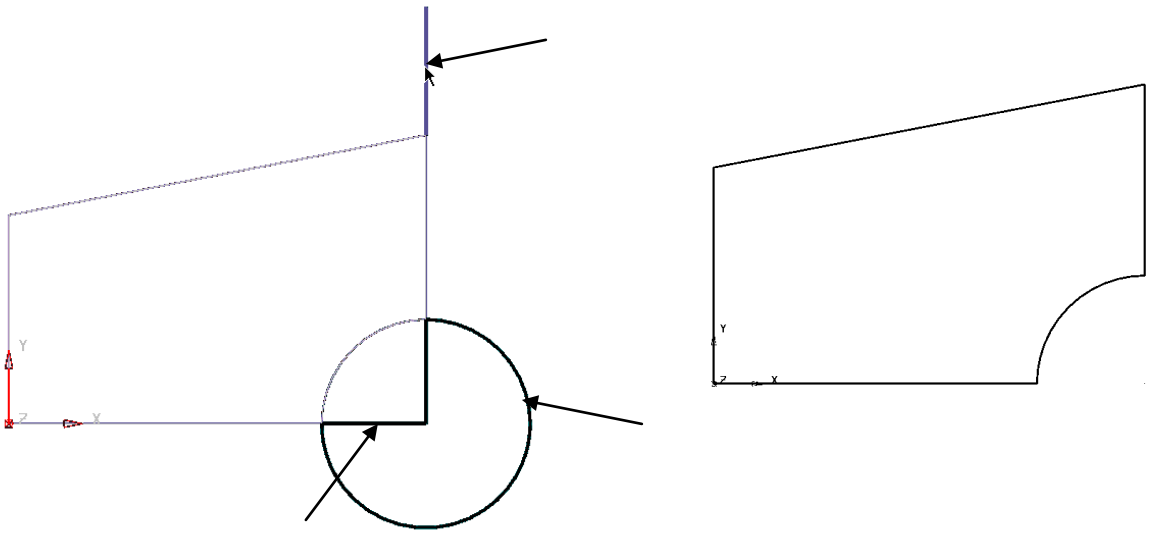


The imported wireframe (shown above left) will be modified using the **general edits options** initially to produce the 4 finished **wireframes** (shown above right).

- Select the **Point limit** option and move the cursor to the end of the line to be extended and hold down the left mouse key (The cursor will take the form of a magnet).
- Drag the end of the **line** tangentially (along the preview line) until the word **Intersection** is displayed at which point release the left mouse key,



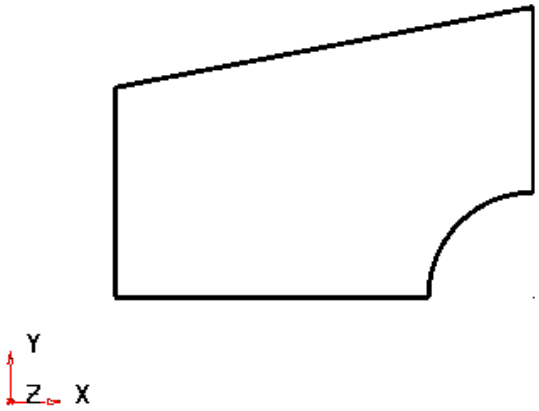
- Select the **Interactively limit wireframes** option and left click the ***mid-span*** of any sections of the **wireframe** network that are to be trimmed away (as illustrated below).



- Create a **Composite Curve** from the network of **lines** and **arc** wireframes (Hint: For a 'shortcut' method hold down the **Alt *key*** and **left click** anywhere along the **wireframe** network).

- With the **Composite Curve** selected, open the **general edit** - **Move** form.

- While the form is open, enter the coordinate values **25 25** in the **Command Input** box located at the bottom right of the graphics area.

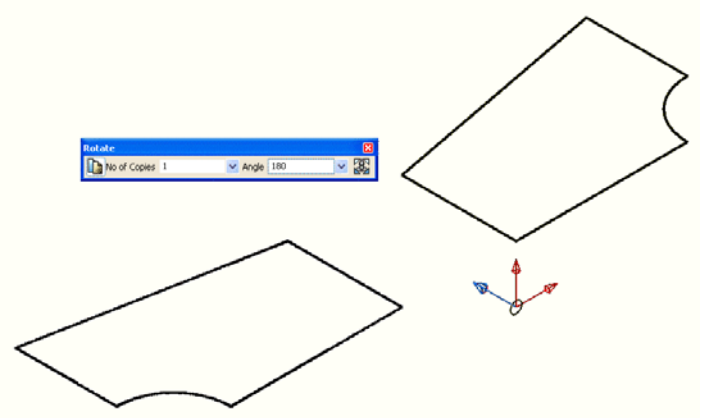


The **Composite Curve** is shifted by **25mm** along both **X** and **Y**.

- Select **Y** as the operational axis by selecting the Icon located below the graphics area towards the left hand side.

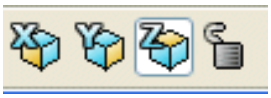
- With the **Curve** selected, open the **general edit** - **Rotate** form.

- In the **Rotate** form select the ***keep original*** option , enter ***No of Copies***:- **1**, and ***Angle***:- **180** followed by **Return**.



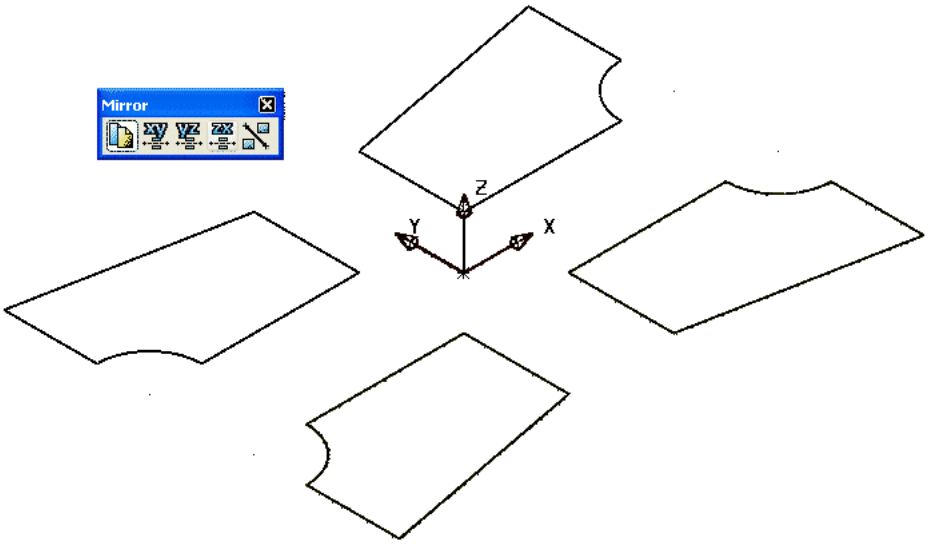
A copy of the **composite curve** has been **rotated** by **180** degrees about the active **Y** axis.
Note:- In this case the same result would have been obtained if the curve had been **mirrored** about the **YZ** plane.

- Reinstate **Z** as the operational axis by selecting the Icon located below the graphics area towards the left hand side.



- With both **Curves** selected, open the **general edit** – **Mirror** form.

- In the **Mirror** form select the ***keep original*** option and select the icon labelled **ZX** followed by **Return**.

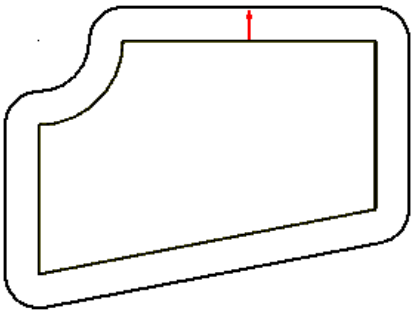


Copies of the 2 **composite curves** are **mirrored** about the active **ZX** plane.

Further General Edit options

Offset

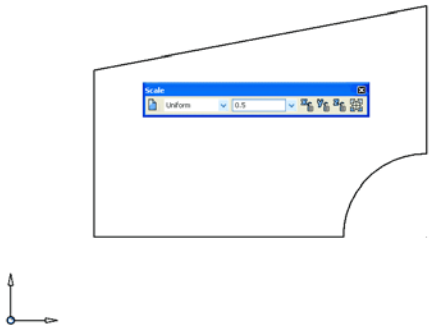
- Select the **composite curve** located in the **-X -Y** quadrant and in the **General Edits** form select the **Offset** options.



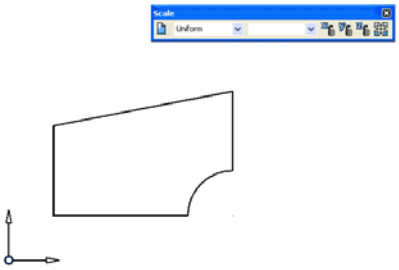
- Select the **Round discontinuities** option and input ***Distance* 10** to create an **offset copy** as shown above (The arrow denotes the offset direction for a +ve **Distance** value).

Scale

- Select the **composite curve** located in the **X Y** quadrant and in the **General Edits** form select the **Scale** option.

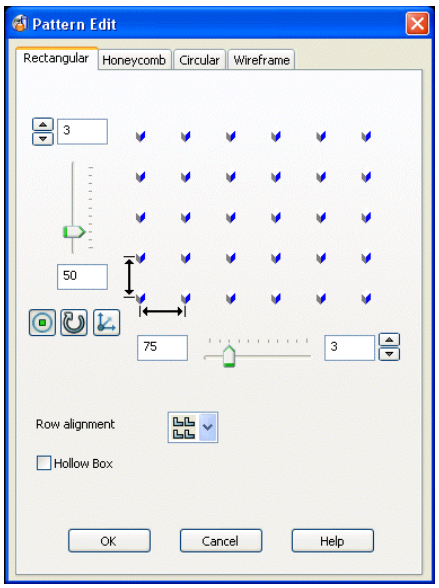


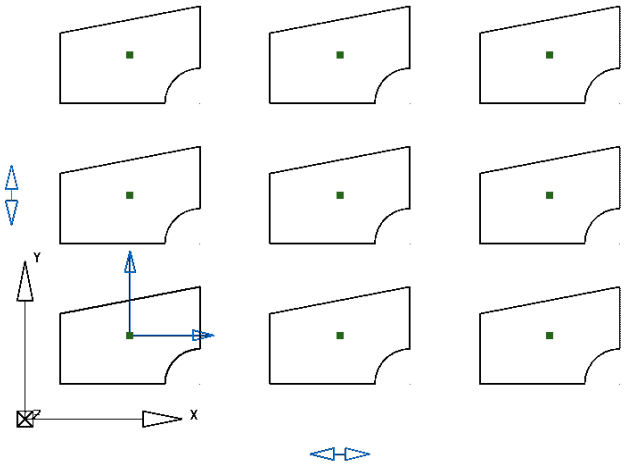
- Enter a ***Scale*** factor of **0.5** to reduce the size of the **composite curve** by **50%** of the original size (as shown below).



Create Pattern of objects

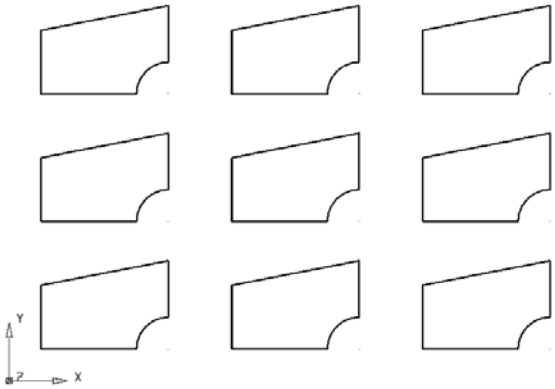
- With the scaled down **composite curve** still selected, click the **Create Pattern of objects** option.



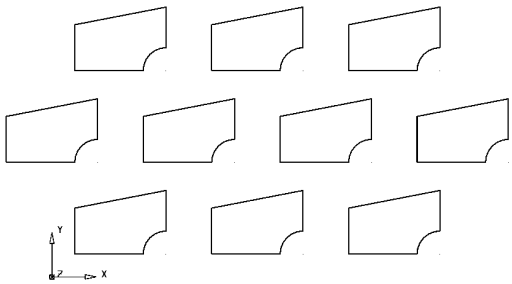


The current settings are immediately previewed.

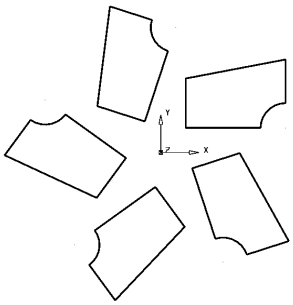
- Input **3** rows along both **Y** and **X** with a **Y pitch** of **50** and **X pitch** of **75**.
- Select **OK** to accept.



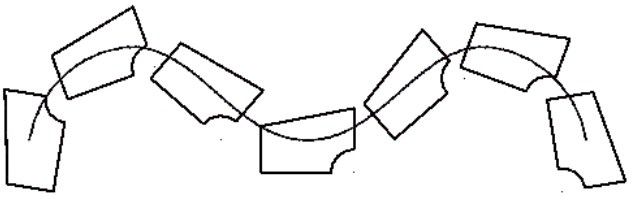
Rectangular is one of 4 main functions in the **Pattern Edits** form. The other 3 include **Honeycomb**, **Circular**, and along **Wireframe**. Note: Within each specific **Pattern Edit** form further advanced options are available.



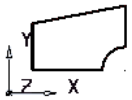
Honeycomb



Circular



Wireframe



The **General Edits** options will be applied as required, later during the course on other **Delcam Designer** entities such as **Surfaces** and **Solids**.

Gridding

Often when sketching, it is useful to draw onto a rectangular grid to give an idea of scale. To activate the grid, click the Grid icon on the status bar.

The default grid spacing is 10mm, but you can change the size to anything you like, by entering a different value is the box to the right of the icon.

- Activate the grid, and sketch a few more lines. Notice that key geometry points (end points, mid-points etc.) take precedence over grid positions.

- Turn the Grid Off, and Delete all your geometry.

Note: Selecting the icon allows you to switch the **IC** on/off.

Fillet Arcs

Fillet arcs are used when you want to make a smooth join of known radius between two items (two lines, two arcs, an arc and a line).

- Draw two lines that join (roughly) at a right angle –

- Click the ‘Fillet’ option

(You will almost never use the Untrimmed Fillet command, but it works exactly the same way).

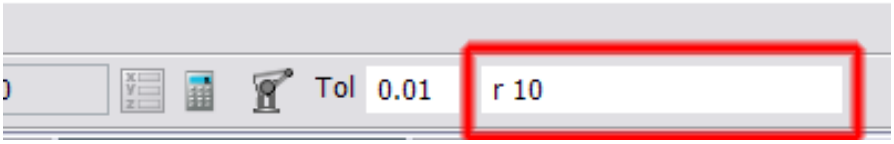
Notice that the cursor changes to an arc, and also displays a number (the value of the radius)

- Move the cursor over one of the lines and see that it changes to read “Fillet 1st Item”.
- Click the mouse to select that line.
- Move the mouse over the other line. The IC changes to “Fillet 2nd Item”.
- Click to select that line and create the fillet.

If the radius is the wrong size, you can double–click the fillet to raise the edit form, and then change its radius.

To set the value of the radius *before* you create it, by typing **R 10**

Note that you do not need to put the cursor into the command box. What you type automatically appears in the box.



TIP: To make sure the command box is empty before you type anything else, press ‘Enter’.

- Delete all the geometry you have made so far.

Designing With Curves

As well as enabling you to design with arcs and lines, Delcam Designer lets you create shapes using smooth, free-form curves.

Bezier Curves

Bezier curves are Delcam Designer’s default curve type.

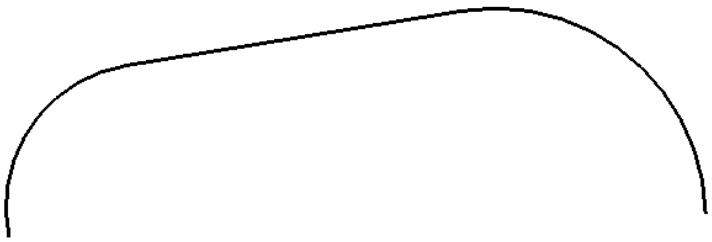
To create a curve, click on the main toolbar to raise the curve toolbar. Then click the Bezier icon

- Using the left mouse button, sketch a curve using 4 or 5 points, as below.

To exit curve creation, press . You can also finish the current curve by double-clicking the current end point. You can then create another curve.

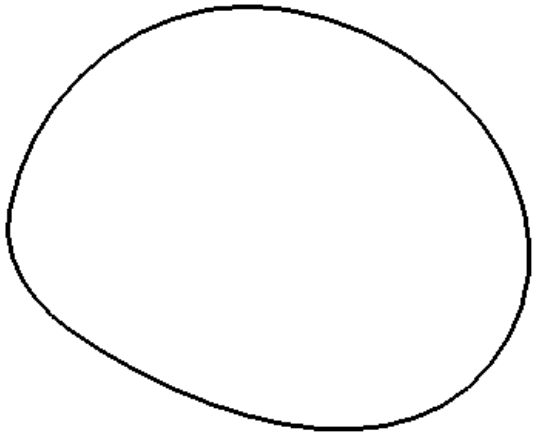
Note that the IC will operate in the usual way during this process. If there is any other geometry nearby the IC will let you ‘snap’ exactly to it.

- Delete the curve, and draw this, instead. Note that the span between the two middle points is to be a straight line.



To draw a straight line segment when creating curves, hold the Ctrl key down as you click.

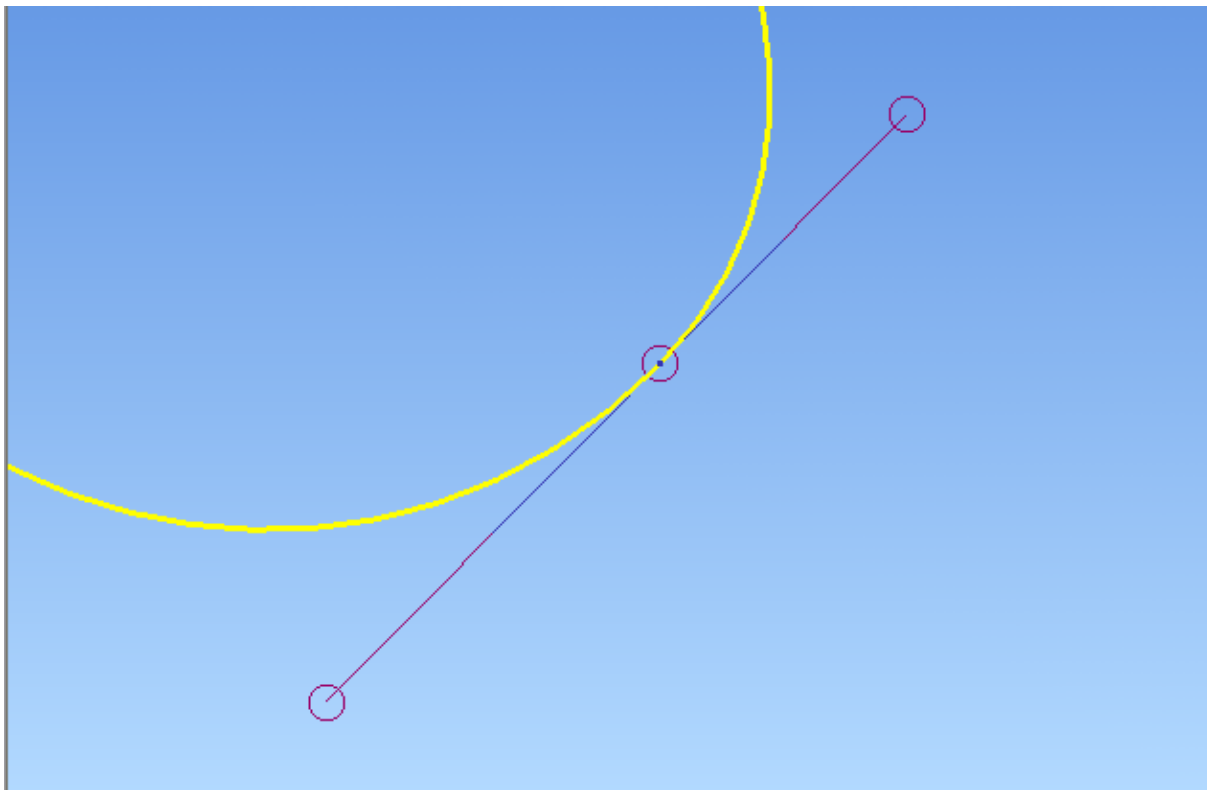
Delete the curve, and draw a closed shape. Note how the curve will automatically change so that the tangent directions of the first and last points match up, giving a smooth shape. Note also that the cursor changes as you place it over the curve’s start point, and the IC displays the word ‘Key-point’. Remember that to obtain the correct closure you must double-click. Then press to exit curve creation.



Editing Bezier Curves

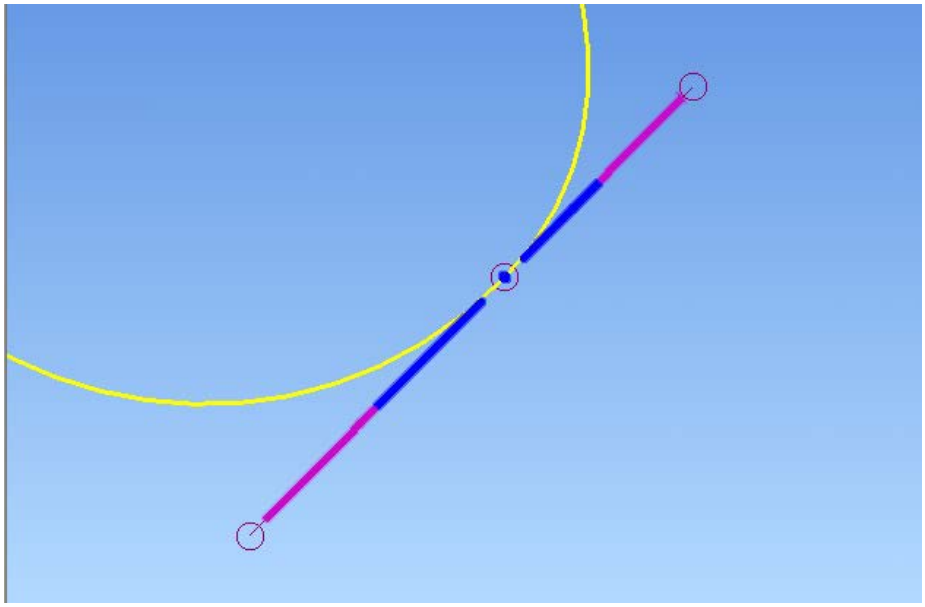
Delcam Designer allows you to change your curves in almost limitless ways. As soon as you select a curve, a special toolbar opens, that contains many different editing tools. For full details of each of the tools, and examples of its use, you can use the Learning Assistant, which is available from the Help menu.

Select the curve and you will see the individual points marked with small blue dots. Pick the curve at any of the points. The curve point will become *instrumented* and you can make edits to the curve graphically.



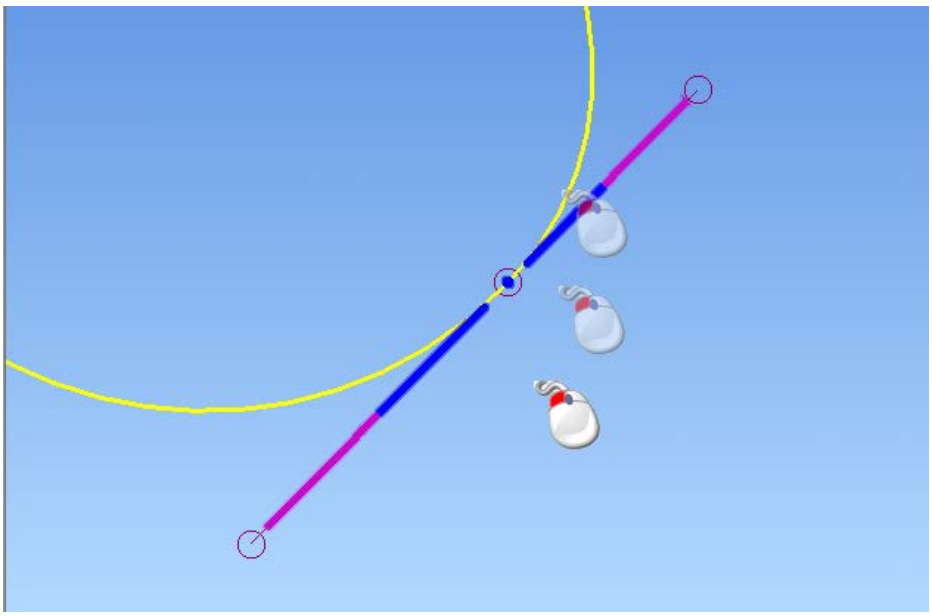
This instrumentation is very important. Curves on surfaces may be edited using exactly the same techniques. If you look closely at the lines that join the dots you will see that they are (slightly) different colours.

This is exaggerated in the following diagram.

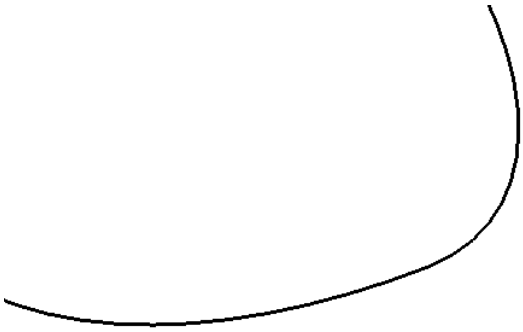


The lines, and the circles (including the point that defines the curve), may all be *dragged* (by holding down the left mouse key while moving the mouse).

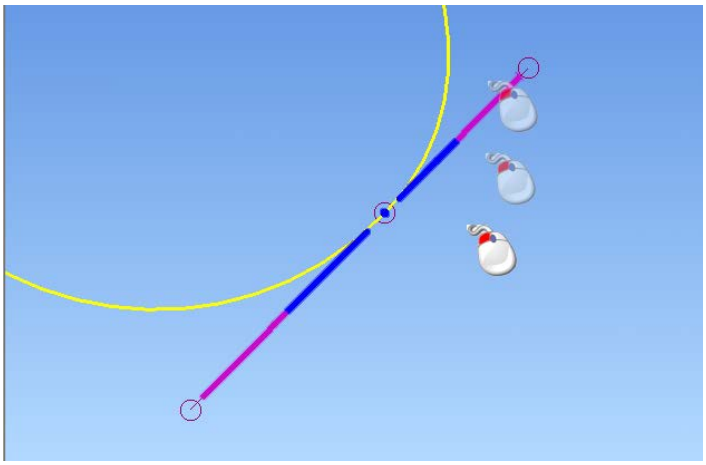
- Start by clicking on either of the blue pieces. Hold the mouse button down, and drag.



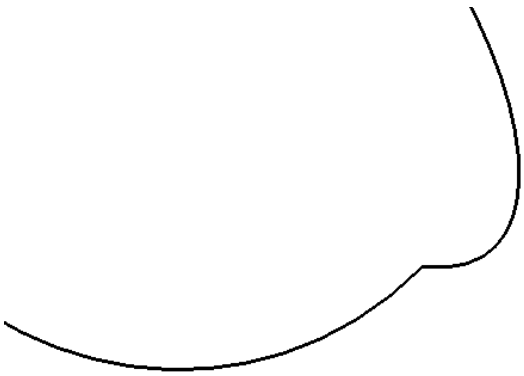
This has changed the shape of your curve, by changing the *angle* at which the curve passes through the point. This angle is called the ‘Tangent’.



Now do the same again, but this time, drag one of the purple lines.



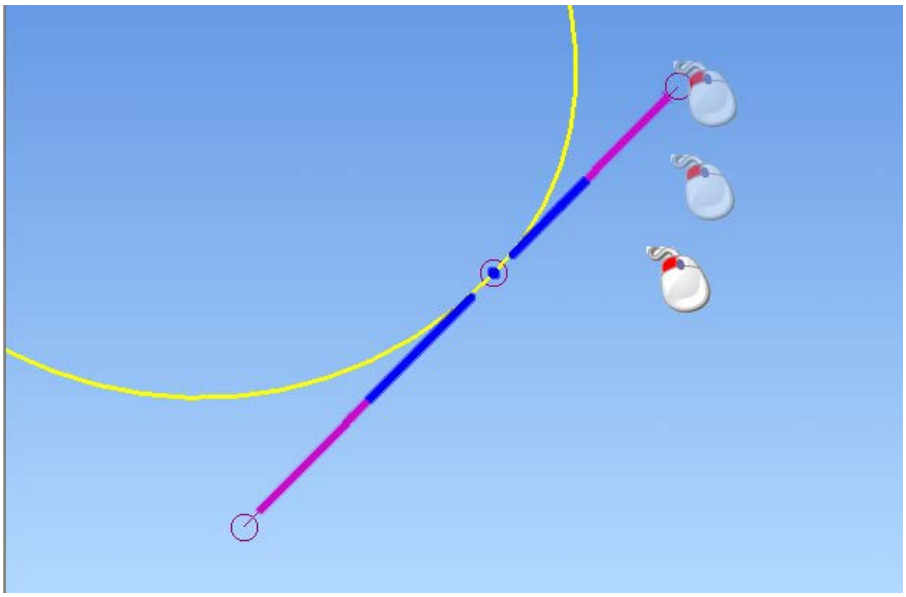
This has altered the Tangent angle again, but only on the side you picked. Therefore the curve now has a ‘kink’ or discontinuity.



As well as allowing you to dynamically change Tangents, this mechanism also allows you to change the *Magnitude* of the curve.

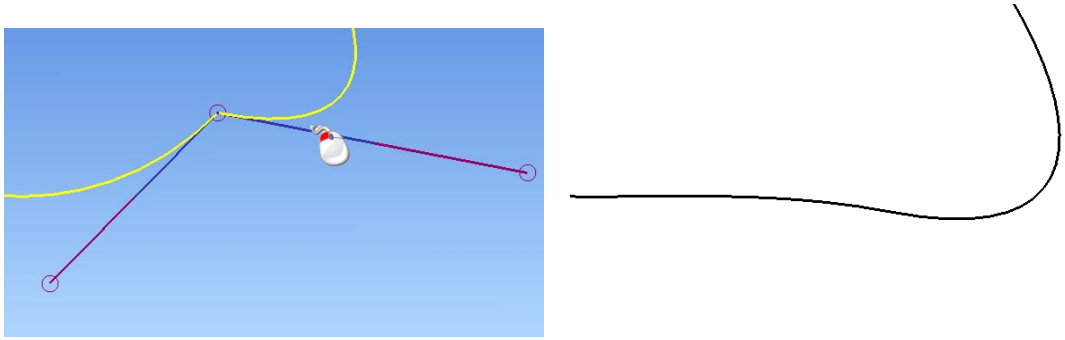
Imagine taking a piece of steel spring and bending it. Now imagine doing the same with a piece of elastic – or a glass rod. The *Magnitude* of a curve (or a point on that curve) controls how stiff or sloppy the curve is. High magnitudes give curves that resist changes of direction and are ‘stiffer’. Low magnitudes give curves that are very ‘soft’ and floppy.

- Do the drag/move again, but this time grab the circle at the end of the right hand purple line**.**

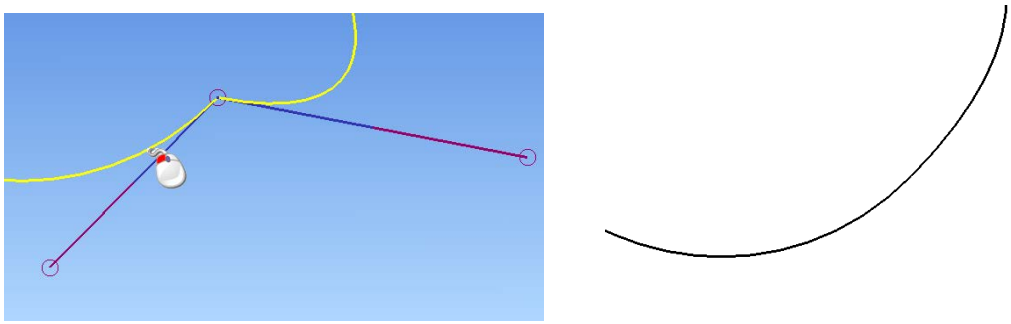


Notice that the shape of the curve becomes very different, as now you are adjusting both Tangency and Magnitude.

Another graphical edit that you can do is to *copy* the Tangent and Magnitude settings from one side of the point to the other. To do this simply click (just one click, do *not* drag the mouse) on the bit of blue line on the side whose values you wish to *keep*.

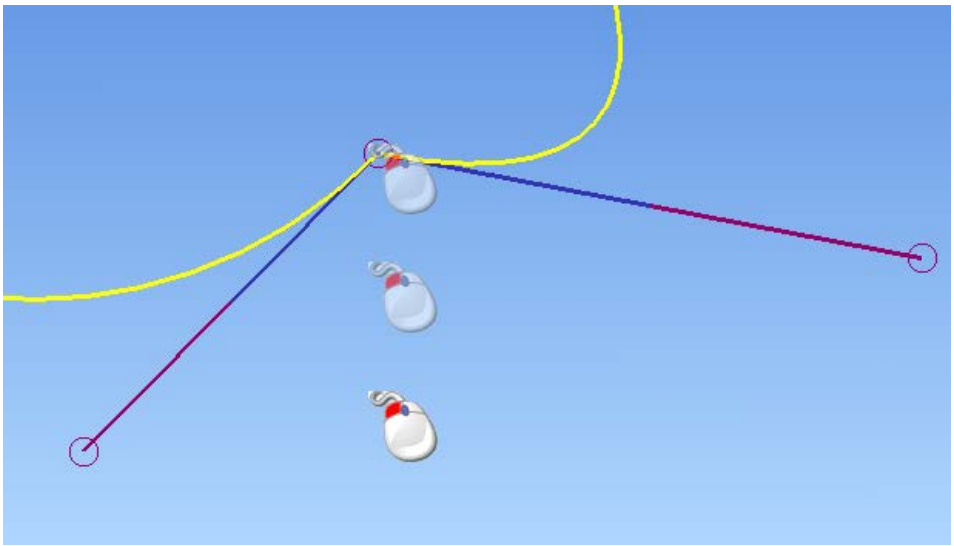


OR-



TIP: While setting Tangencies, the IC will still allow you to ‘snap’ to other geometry. This provides a very fast, yet simple and accurate method for aligning curves with other elements of your design.

The final edit that can be done graphically on a curve is to move the defining points. Repeat the drag/move exercise from earlier, but this time, pick the point as shown.



The IC will operate as usual during such a move, allowing you to ‘snap’ points to other elements.

TIP: When moving points in this way, you can also press and hold the ‘X’ or ‘Y’ key on your keyboard down to constrain the move.

An Important Edit Flag

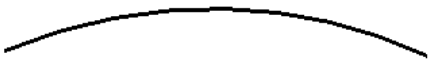
At the right hand side of the Curve Edit toolbar which appears every time you select a curve, is an icon which has a very big effect on how the edits occur.

This icon can effectively be either ON or OFF.

If the icon is turned ON it looks like this

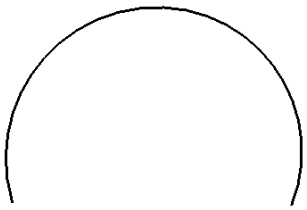
If the icon is turned OFF, it looks like this

To investigate the difference, draw simple Bezier curve using three points, as shown below.



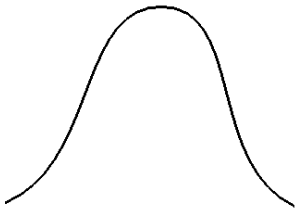
- Turn the icon ON , so it looks like this by clicking it with the left mouse.

Now *drag* the middle point of the curve upwards. Notice how the shape of the curve changes as you do this. The spans to *either side* of the point are automatically recalculated to give the smoothest possible curve flow. This is known as ‘floating’.



- Undo the change, and repeat with the icon turned OFF

Now, the tangent angles of the adjacent spans remain unaltered, giving a very different result.



Sketching Over Concept Artwork

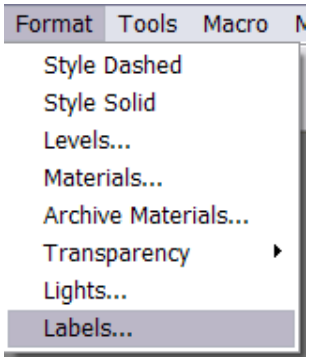
In most cases, designs begin life as concept artwork, either as ink on paper drawing, or as renderings created in paint-box software. Delcam Designer allows you read images and use them as templates for sketching geometry in two different ways.

Label Wrapping

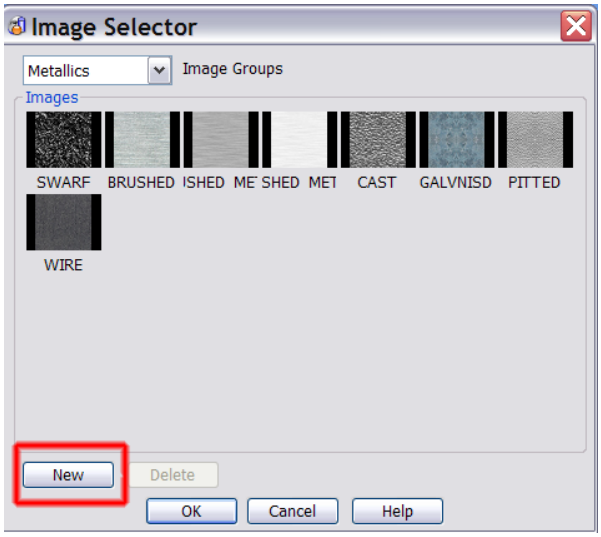
Create a simple planar surface by activating the ‘Create Surface’ toolbar.

Then pick the Plane option,

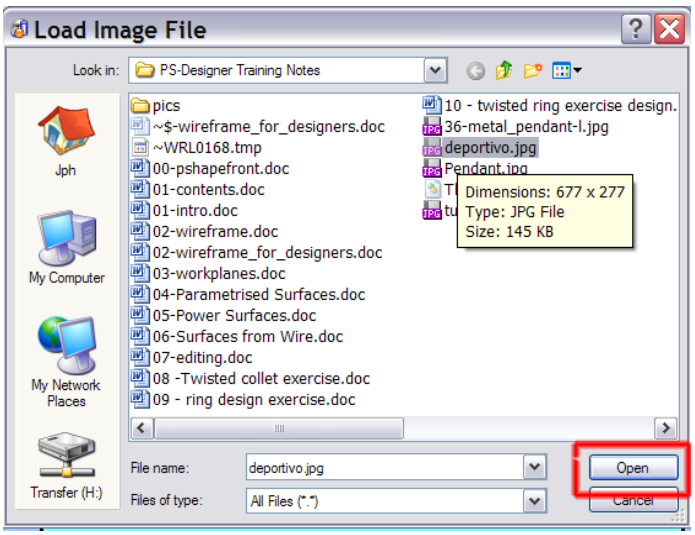
- Click the left mouse button to create the surface.
- From the top menu bar, select “**Format > Labels**”.



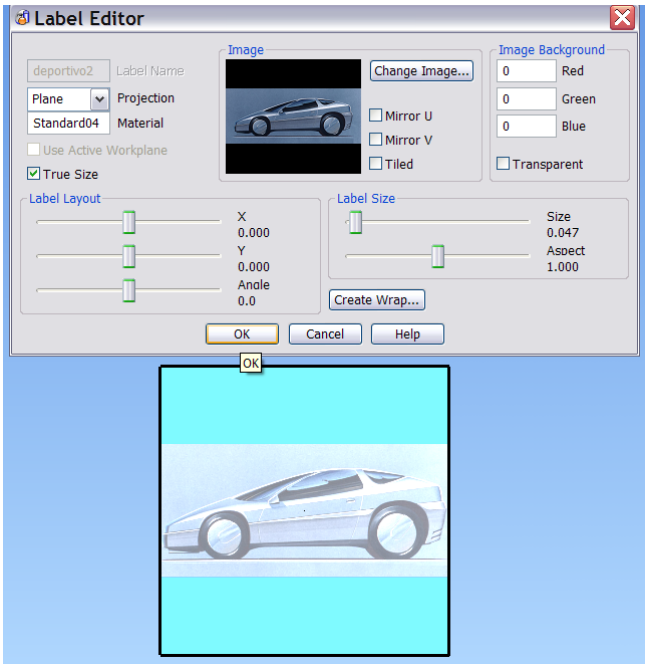
- When the label selector appears, click **'New'**.



- Use the next form to browse **D:\users\training\Delcam Designer Data**
- Select the image **deportivo.jpg** and then **Open**.



If necessary, adjust the size of your images, or its orientation, using the sliders. When you are happy with the result, click ‘OK’.



You can now sketch directly onto the surface to create your curves and other wireframe geometry, using the image as a guide.

Note that the image does not have to be wrapped onto a simple plane. You can wrap an image onto a more complex shape to allow creation of 3D curves.

Billboards

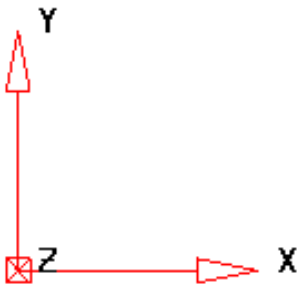
As an alternative to label wrapping, you can instead sketch onto billboards. Billboards are more flexible as you can define up to 6 separate images to represent different views of the design (front, top, left, right and so on).

Before you can create a Billboard, you need to define the plane or surfaces onto which you are going to sketch. Create a single Workplane at the origin.

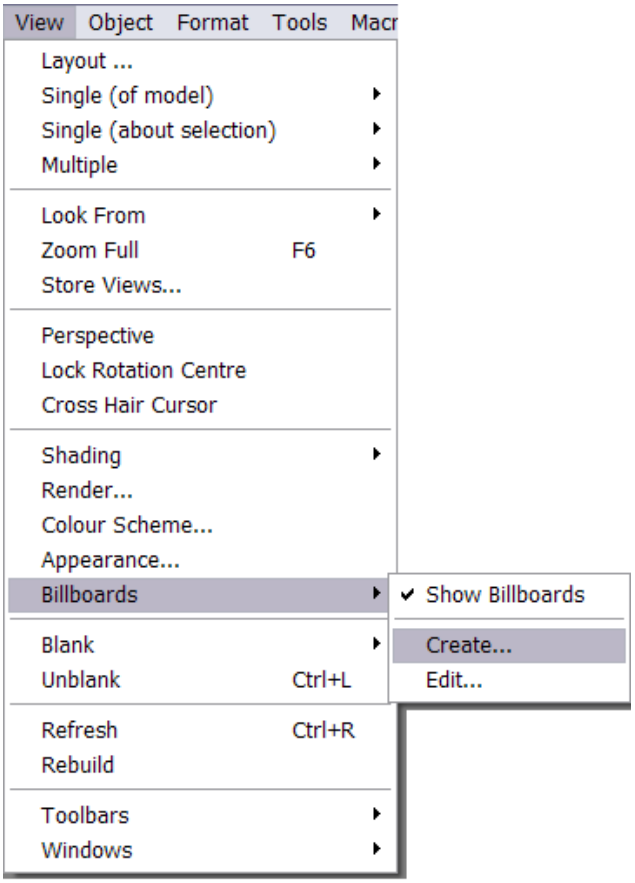
- Click a position on the screen.

TIP: Rather than simply click on screen to establish a position, you can enter exact values. Typed values automatically appear in the Command Window. Numbers are entered in the order X, Y, Z but you only need enter enough numbers to make the value unique.

Entered Value	Values Used
10	10, 0, 0
0 10	0, 10, 0
0 0 10	0, 0, 10

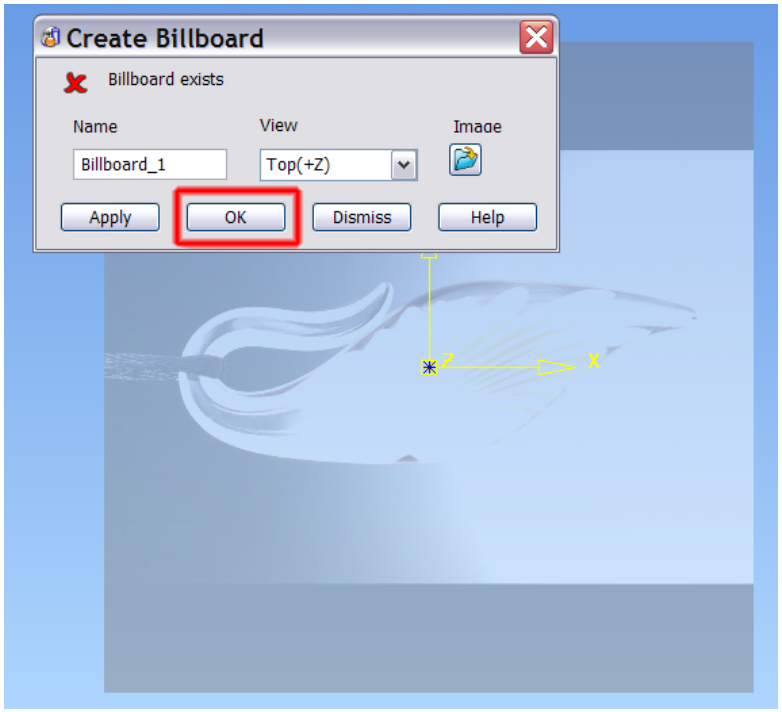


- Select the Workplane, the choose “View .. Billboards … Create” from the top menu.

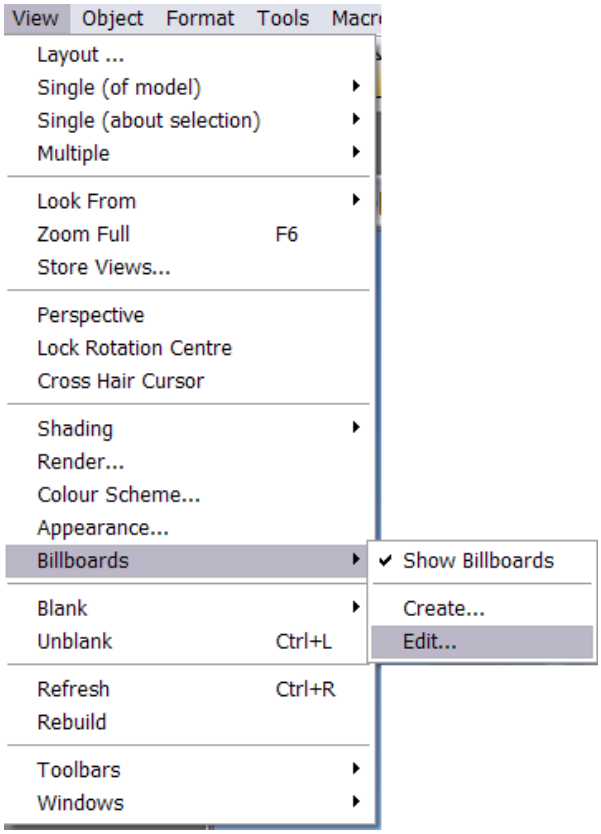


The ‘Create Billboard’ form opens, from which you can specify the direction(s) for the view, and the Image(s) to use.

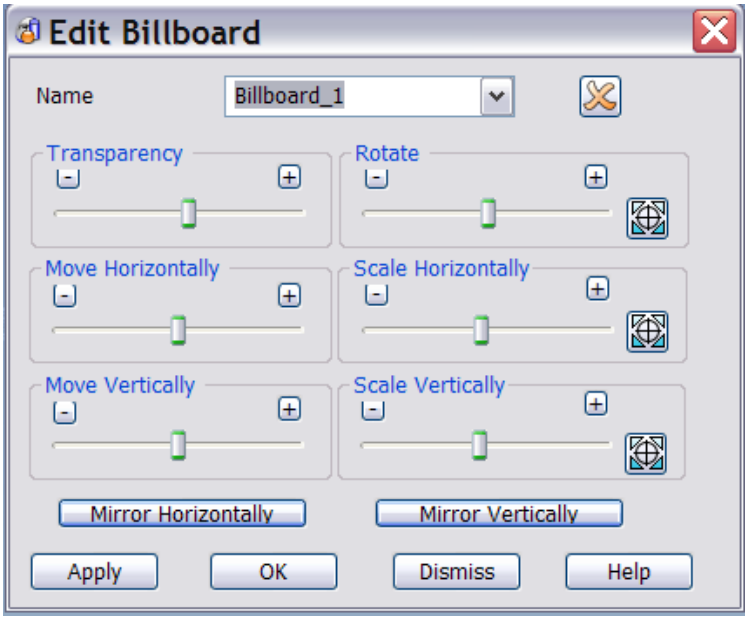
- **Click the ‘Image’ button and locate D:\users\training\Delcam Designer Data.**
- Select the image **Pendant.jpg** and then **Open**



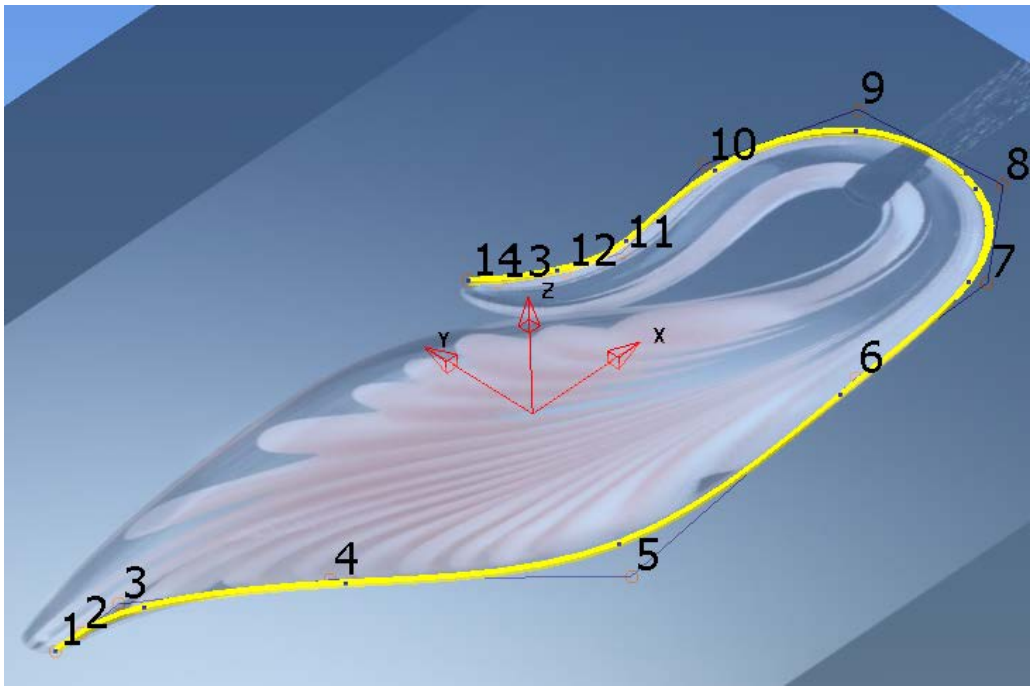
If the orientation or size of the image is not what you want, you can edit the Billboard, by selecting “View… Billboards…Edit” from the top toolbar.



From the edit form you can change the attributes of the image to achieve the desired result.



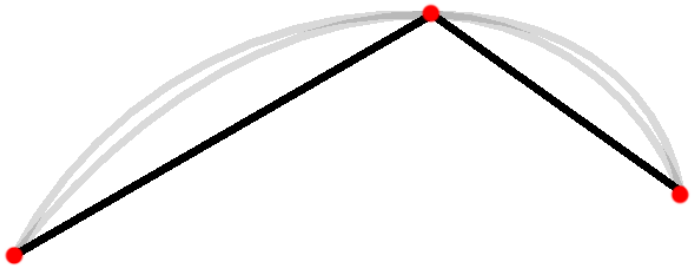
Once the Billboard is in place, use any of Delcam Designer's sketching tools to create accurate wireframe geometry.



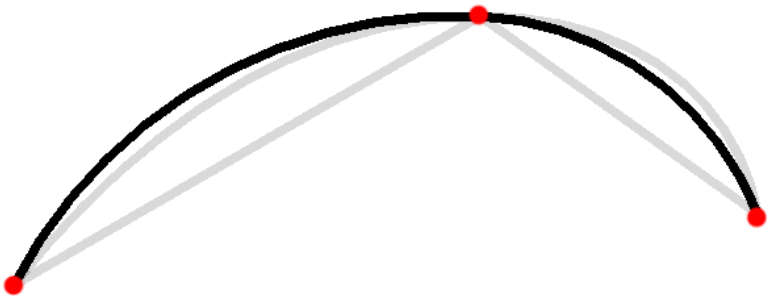
Appendix - Other Types of Curve

Delcam Designer uses several different types of curve, and (unfortunately) in order to understand the differences we need a little theory.

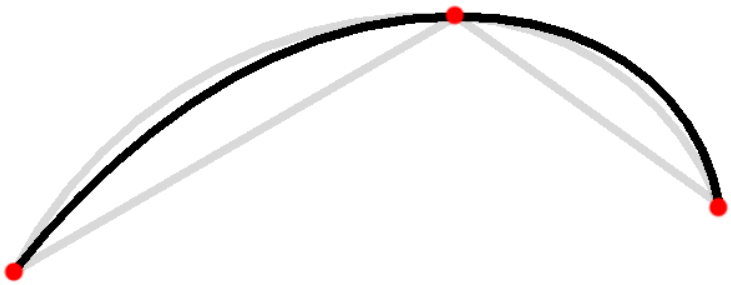
There are three classes of curve, known as **G_0, G_1 and G_2**. All curves are composed of a set of defining positions, known as 'points'. The distinction comes from the way the curve behaves as it passes through each point.



A G_0 curve has *positional continuity*. The ends of the span leading *to* and *away from* the point are in exactly the same place. However, the two spans may point in very different directions. If this type of curve were used in a surface you would see a crease. In the figure above you can see the sharp change of angle.



A G_1 curve has *tangent continuity*. The curve exits the point in exactly the same direction as it enters. If you used this type of curve to construct surfaces you would not see a sharp edge. However, under certain lighting conditions you can still see what is usually known as a 'hard edge' due to the way light is reflected. In the figure above, you can see that the curve's shape changes as it passes through the mid-point.



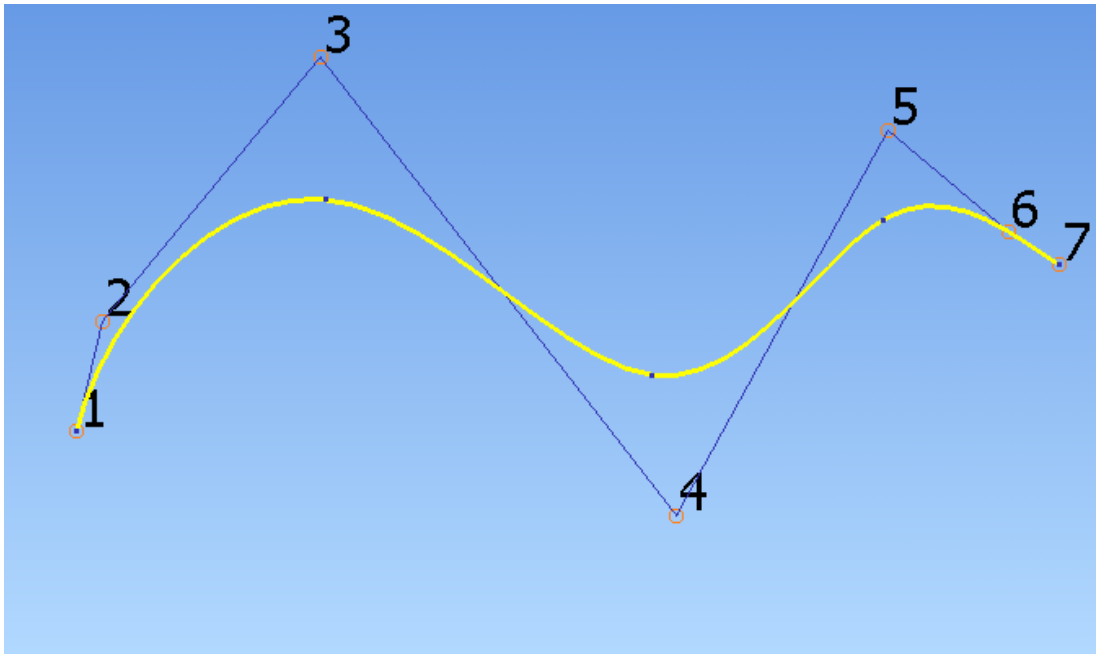
A G_2 curve has *curvature continuity*. Not only does the curve stay in the same direction as it passes through the point, but the shape of its trajectory is exactly the same either side. If you were to replace a segment of curve with arcs that were a perfect fit, this type of curve would have identical radii either side of the point. A surface made with this type curve has no visible transitions between adjacent segments, but is smooth and free-flowing. This type of surface (known as an 'A-class' surface) is typically found in automotive bodywork.

Delcam Designer offers all these types of curve, and has extensive tools for editing them too, to help you achieve the perfect shapes. For most applications G_0 and G_1 curves give perfectly acceptable results. The type of curve you use will depend on the object you are designing.

NURBS

A NURB is a special type of curve that maintains a smoother shape than a Bezier. A NURB curve cannot have a discontinuity, and you also have less control over the ways in which the points can be edited after the curve has been created.

- Click the NURB icon, from the Flyout.



G_2 Curves

Delcam Designer can also make G_2 curves in the same way. Click the G_2 icon

.

When editing a G_2 curve, the curve will *always* retain its *curvature continuity.* Therefore it is not possible to make the curve have any kinks.

3. Workplanes

Workplanes

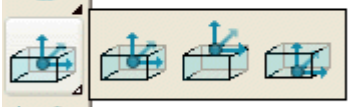
Workplanes are user defined, local datum points, which are positioned and aligned as required to simplify modelling. The model can contain several **workplanes**, but only one can be **active** at any time. When a **workplane** is **activated** it becomes the XYZ datum and changes colour from grey to red. Model entities can be **copied** or **cut** from the currently **active workplane** and then **pasted** back in a different position, relative to a newly **activated workplane**.

The main **Workplane** icon is located in the **main toolbar** which when selected will provide the following icons in the left hand toolbar:

Single Workplanes.

Multiple Workplanes.

Single Workplane aligned to geometry.



Single Workplane at Top/Centre/Bottom of selection.

Workplane from three points.

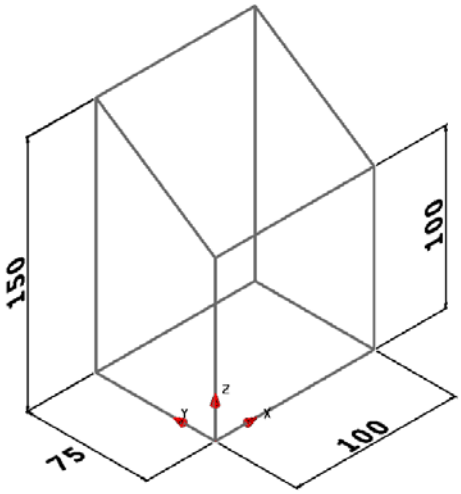
Create Points.

In the next example the user will be required to apply **wireframe Lines** to construct a box with an angled top. This in turn will be used as the model to demonstrate some basic applications of **Workplanes**.

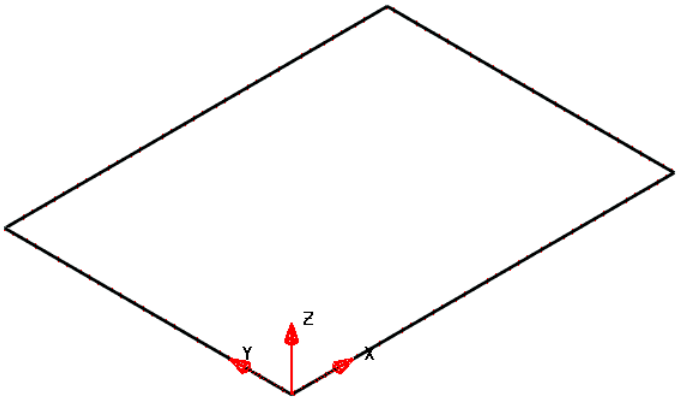
Box Example

- Open a **New Model**.

The **wireframe** for the dimensioned form (as shown right) will now be created.

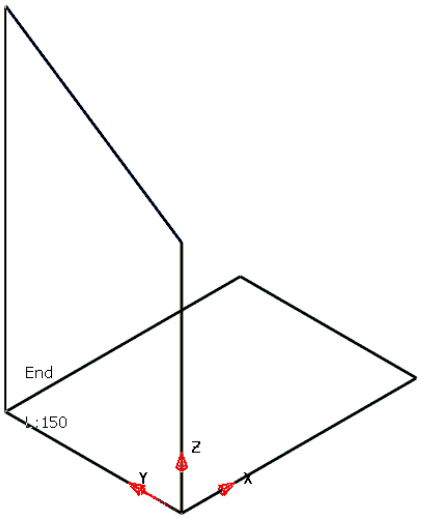


- Open the **Single Workplane** option and enter **0** into the **Command Input** window to position it at the ***World Datum***.
- Select the **view Iso1**.
- **From** the **Line** menu select **Rectangle**.
- Enter **0** in the **Command input box** and press **Return**.
- Enter **100 75** in the **Command input box** and press **Return**.



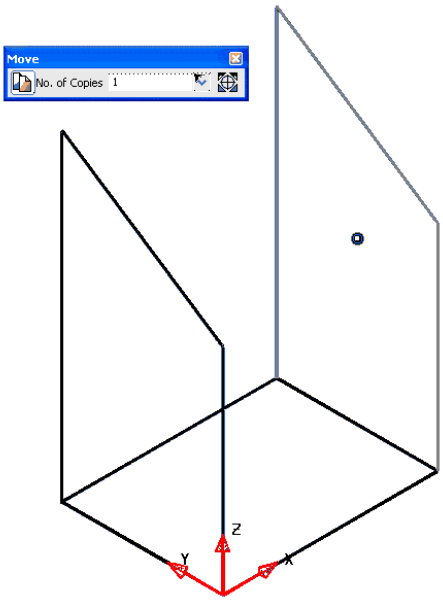
Note:- The ***rectangle*** is not a single entity but consists of 4 separate, **wireframe lines**.

- From the **Line** menu select **Continuous Lines**.
- **Snap** the start of the **line** onto the **Workplane**.
- Enter **0 0 100** in the **Command input box** and press **Return**.
- Enter **abs 0 75 150** in the **Command input box** and press **Return**.
- Snap (left mouse click) the end of the current line to the top left corner of the **rectangle** (Marked as ***End*** by the **Intelligent Cursor** below).



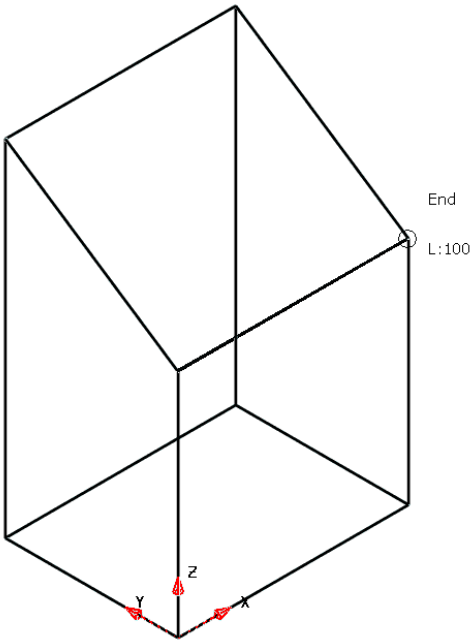
- Select the last 3 **lines** created.
- Open the **General Edits - Move** option.

- Select the option **Make a Copy** and enter **100** in the **command input box** before pressing **Return**.



The final 2 **Single lines** are input by snapping to appropriate key points on the wireframe model.

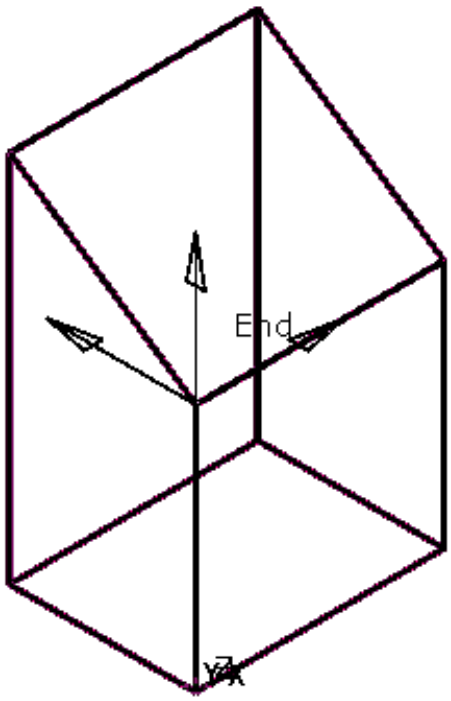
- Select **Single Lines**.
- Press **Escape**.



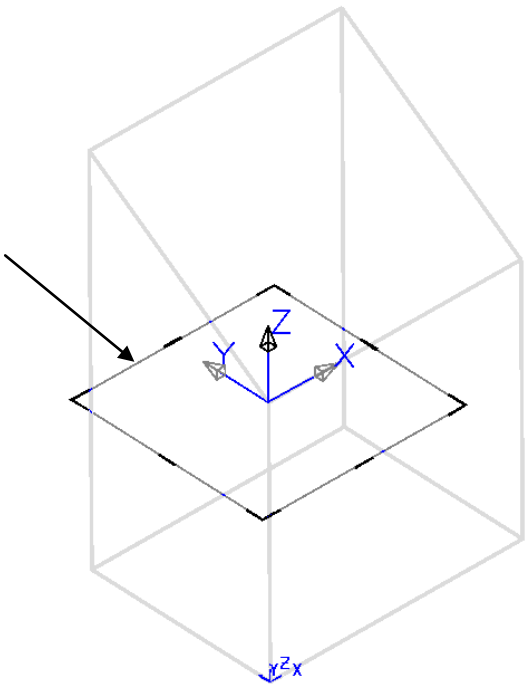
The main **wireframe *model*** is now complete.

An additional **wireframe *model*** based on a hexagonal **Polygon of lines** will now be created to a new **Workplane** coordinate system at **X -100** away from the main model. A copy of this separate model will later be placed onto the angled, top face of main model.

- Open the **Single Workplane** option and ***snap*** a new **Workplane** to the top ***End*** of the line running up **Z** from the original **Workplane**.



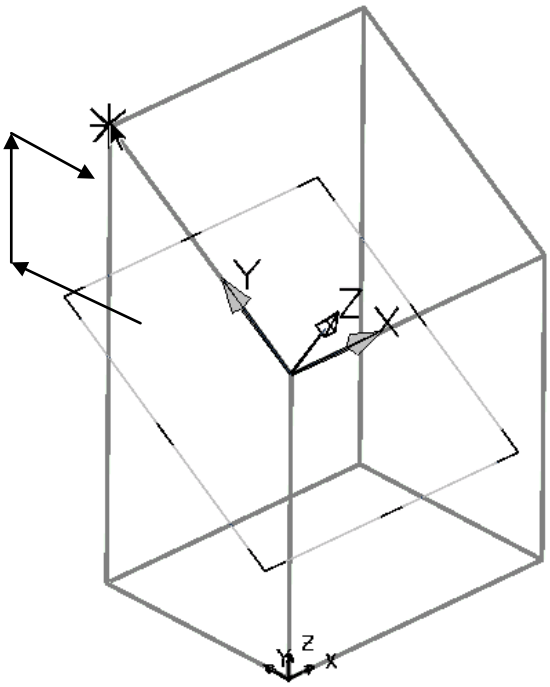
- **Zoom** into the new (**Active**) **Workplane** and then use the left mouse key to ***select*** it.

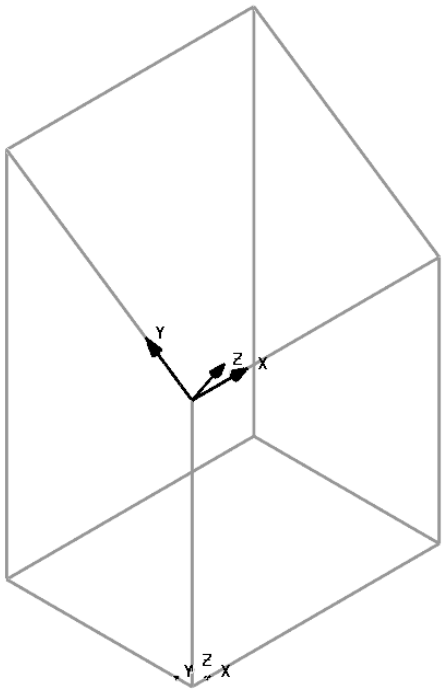


When a **Workplane** is selected a square appears on the **XY** plane. The corners and mid points of the square, along with the **Z** axis arrowhead are thicker and darker in colour.
If the left mouse key is held down on one of the dark bands then the **Workplane** can be dynamically rotated about the **Z** axis.
If the left mouse is held down one of the lighter coloured bands then the **Workplane** can be dynamically rotated about the corresponding, ***parallel*** **X** axis or **Y** axis.

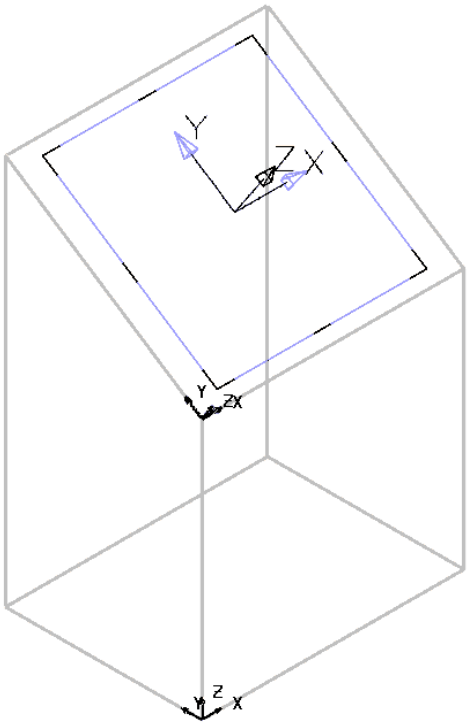
Note: If one of the **Workplane** ***arrowheads*** is selected (instead of a coloured band) during dynamic re-orientation then all **Workplane** axes will rotate simultaneously.

- Select the light band on the **+Y** side and parallel with the **X** axis and with the left mouse key depressed drag and drop on to the upper left corner of the model (arrowed above).



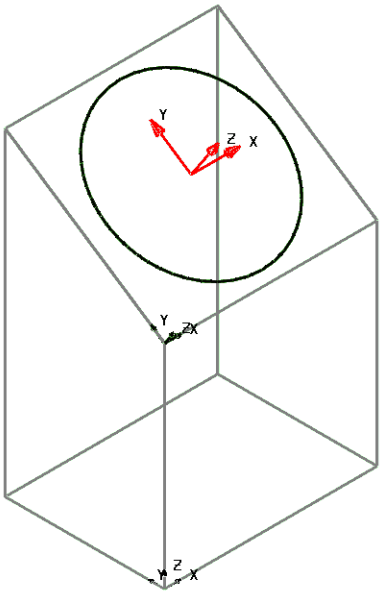


- Select the **4 lines** (Use **Shift - left mouse** for multi-selection) enclosing the angled top face of the model and select the **Workplane** option - ***single workplane at centre of selection***.



As soon as the icon is clicked a new **Workplane** is created, central to the 4 selected **lines**.
This **Workplane** will both be **Active** and **selected**.

- From the main toolbar select the **Arc** options and create a **40 radius** - **Full Arc** about the currently **Active workplane**.



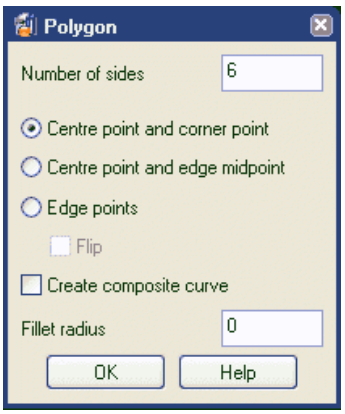
The **R40 Circle** has been created on the **XY** plane of the **Active workplane** central to the angled top face of the **wireframe** model.

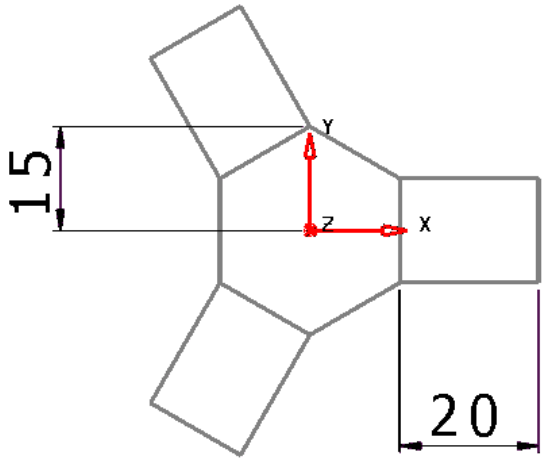
- Right click on the ***Active*** **Workplane** to open it's local menu and ***untick*** **Activate**. As no **Workplane** is now **Active** all measurements will now be controlled by the original **World** coordinates.

- Open the **Single Workplane** option and enter **-100** into the **Command Input** window to position it (along **-X**) away from the main model (The new **Workplane** will become **Active** by default).
- Create the following network of **wireframe lines** central to the new **Workplane**.

Hint:-

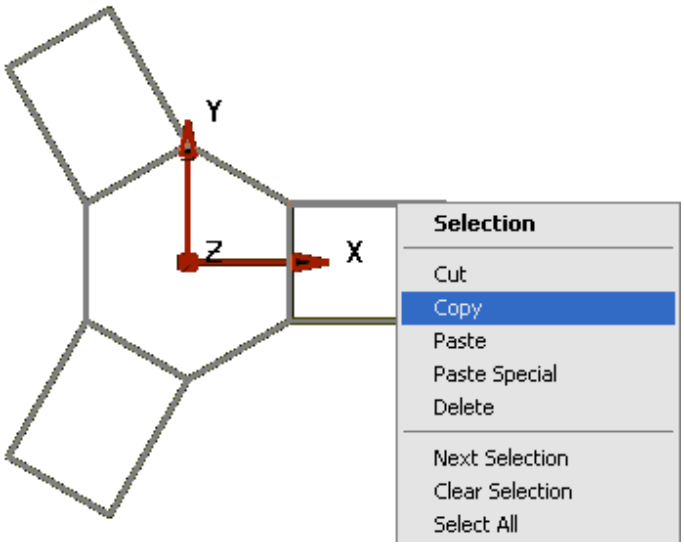
- ***Use Lines*** – ***Polygon***.
- In the **Command Input** box, **Input** (***and return)*** **0** (***Centre point***) followed by **0 15** (***corner point***).





This **wireframe** will be copied from the **Workplane** local to the geometry, to the **Workplane** central to the angled face on the main model.

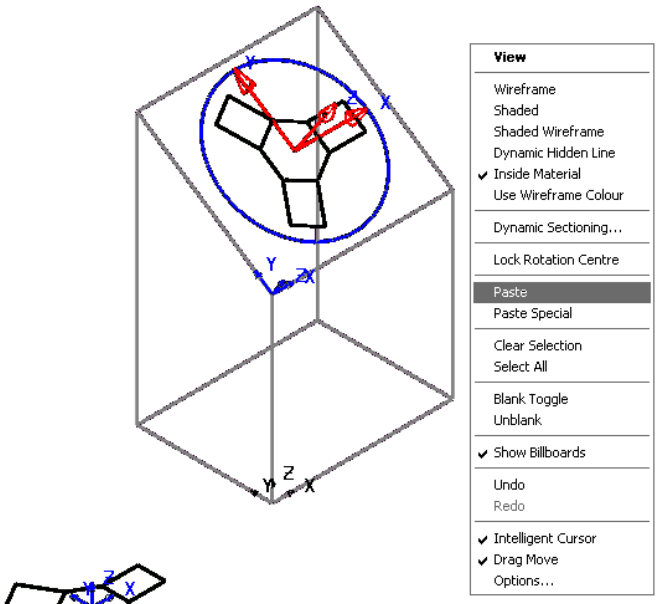
- With the above **workplane Active**, select all the new locally created **wireframe** and with the cursor on one of the **lines**, right click to open the local **Selection** menu.

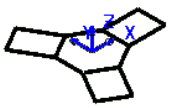


- Select **Copy** in the menu.

The selected **wireframe** is effectively, copied via the mouse!

- **Activate** the **workplane** positioned central to the angled, top face on the ***main*** **wireframe** model.
- Right mouse click in the graphics area away from the model and select **Paste** from the local menu.





The **wireframe** created away from the main model has been copied relative to the **workplane** central to the top angle face of the **main wireframe** model.

Note: Data relative to an ***Active*** **workplane** in one PowerSHAPE model can be **copied** and **pasted** relative to an ***Active*** **workplane** in a different PowerSHAPE model

- From the **main** pulldown menus select **File – Save As**, and store the ***psmodel*** as:-

D:\users\training\COURSEWORK\Delcam Designer-Jobs\Workplane-Box

4. Surface Modelling

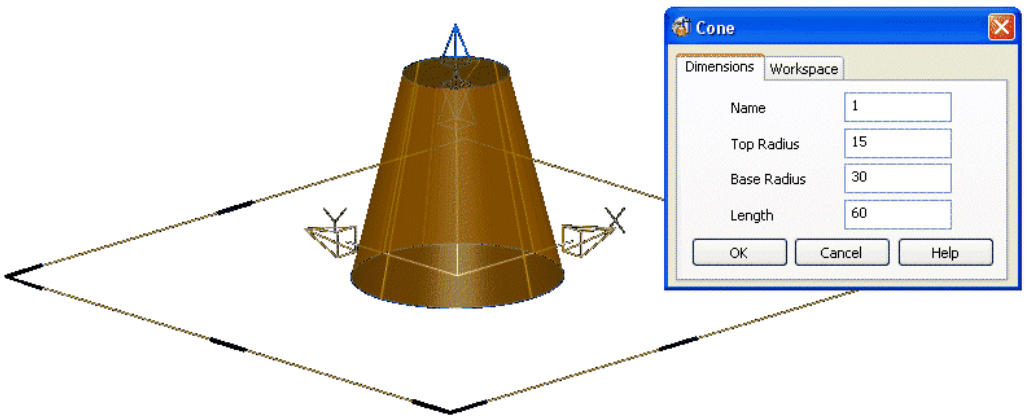
What is a Surface?

A **surface** is best described as a skin of negligible thickness stretched across a defined 2D or 3D area.
There are 3 main types of **surface** supported by **Delcam Designer** which include:- **Primitives**, **NURBS** and **Power Surfaces**. The constructional attributes and editing capability of these are very different.

Primitive surfaces

A **Primitive Surface** is based on simple, ***standard shapes***, and ***wireframe extrusions*** or ***rotated*** forms. A **Primitive Surface** is defined or edited by entering set parameters into a specialist form (The actual input criteria depends on the type of **Primitive Surface**).
A major restriction of a **Primitive surface** is that it is only possible to modify the existing defined parameters such as ***length***, ***radius***, ***orientation*** and where applicable, the ***original wireframe***. **Primitive surfaces** also have an inherent workplane like datum, which the user can manipulate to perform modifications dynamically.

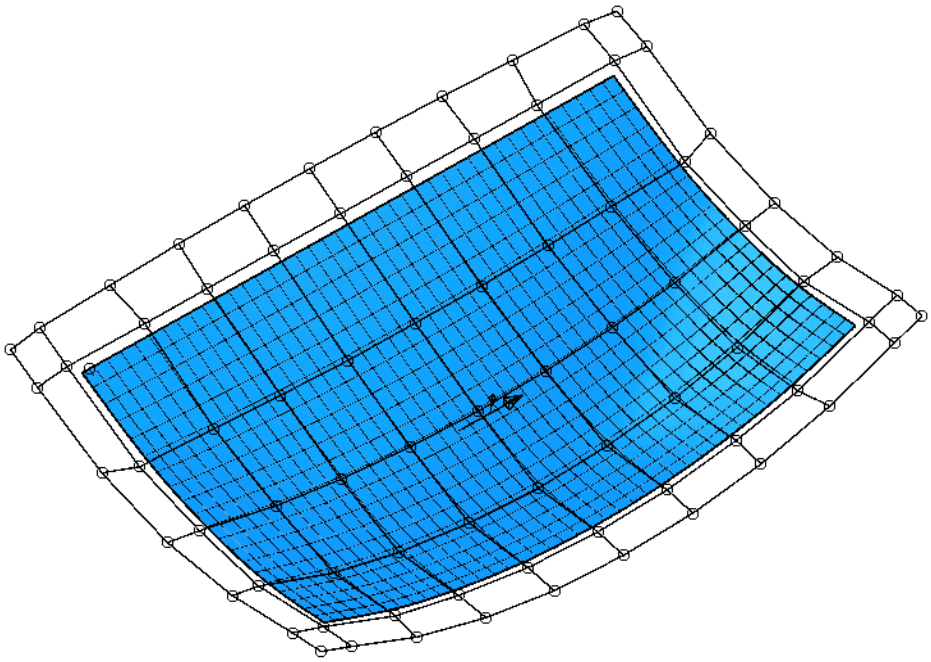
Delcam Designer - Primitive surface options include:-
Primitive - ***Plane, Block, Sphere, Cylinder, Cone, Torus. Helix***.
Extruded surfaces (From pre-defined wireframe).
Surfaces of revolution (From pre-defined wireframe).



The **Primitive Cone** is shown left **selected** with the local **Cone** editing form open.

NURBS surfaces (Non-Uniform Rational B-Spline)

It is common for a **surface model** created using a different CAD system to be imported into **Delcam Designer**. For this to be possible, **Delcam Designer** supports other types of **Surface** definitions such as **NURBS surfaces**. **PowerSHAPE** will also create **NURBS surfaces** during certain applications or if dictated by the user in the **Delcam Designer - Surface Options**.



The control points on a **NURBS** surface can be dynamically moved around in space, but not to any degree of dimensional accuracy.
Any such changes are tricky to apply and depend on the users visual acceptance of the new form.

Conversion of Primitives and NURBS to Power Surfaces

If it is required to perform more complex modifications to the shape of a **Primitive** or **NURBS surface**, it must first, be **converted** to a **Power Surface**. With a **Power Surface** the user has access to a full range of editing options via a comprehensive **surface/curve editing** toolbar or **dynamic** operations.
Note:- It is not possible to ***convert*** a **Power surface** back to a **Primitive** or **NURBS** type.

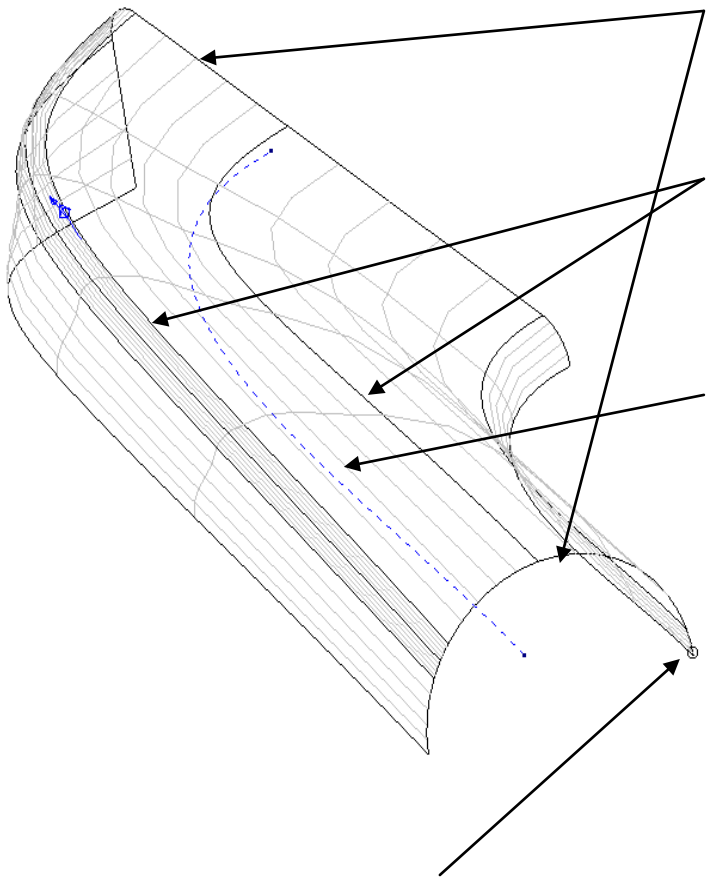
In the **Main** pulldown menu option:-
Tools – Options – Objects – Surfaces ensure that in the section labelled ***Primitives*** that **Create as NURBS** is unticked if direct conversion from **Primitives** to **Power Surfaces** is required (Otherwise when **Convert Surface** is applied to a **Primitive** it becomes a **NURBS** which in turn will have to be converted to a **Power Surface**).

Note:- Several of the **Surface** creation options directly create a **Power Surface**.

Power surfaces

A **Power surface** is based on a network of 4 sided **wireframe** elements appearing as curves along (**longitudinals**) and across (**laterals**) the surface area.
A **Power Surface** can generate complex forms while retaining ***full editing capability*** including the **direction** and **magnitude** through **surface curve** intersections.
To define holes within the **surface** area or an outside profile not compliant with a 4 sided **wireframe** structure, specialist ***trim curves*** called **Boundaries** are applied.
The area of **surface** between adjacent pairs of **laterals** and **longitudinals** is called a *patch*.

The curves on a **POWER Surface** are called longitudinals (along the surface) and laterals (across the surface). In some cases another (optional) curve exists called a spine. This generally runs along the longitudinal direction, often in free space controlling the orientation of the laterals.



Laterals
This surface contains the minimum, two laterals.

Longitudinals
This surface contains a total of 7 longitudinals flowing from corresponding points from the first to the second lateral

Spine (Drive Curve)
The spine (shown dotted) is used to control the orientation of the laterals. A spine is not mandatory and can be created or deleted as required without changing the surface shape. It occurs automatically such as during the creation of Fillet surfaces or as part of the controlled geometry of a Drive Curve surface.

The Cato mark identifies the start point for Laterals and Longitudinals on a POWER surface. It is positioned a short distance from point 1 along lateral 1 with a short line pointing from it representing the Longitudinal direction.

Basic Primitive Example

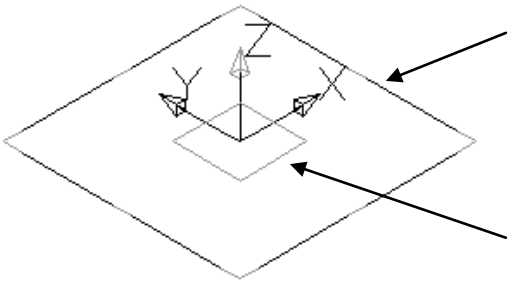
- Select **Create New Model.**
- Select the **Surface menu** icon from the **Main Toolbar.**
- Move the mouse over the **Surface Primitives** fly-out commands.

This menu allows the user to create a range of surface primitives. They are, from left to right, Plane, Block, Cylinder, Cone, Sphere and Torus.

- Select **Plane Primitive.**

A moveable **Primitive Surface** - **Plane** is attached to the cursor, with the centre of the **Plane** being it's local origin. This can be positioned by locking onto an item using the **intelligent cursor** or by typing in **X Y Z** co-ordinate value in the **Command Input** window.

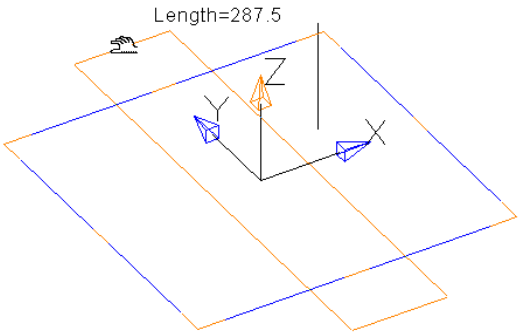
- Enter **0** in the **Command input box** and **press Return**.
- Select the **view Iso1.**



The selected **Plane** is shown is positioned with an inherent workplane attached to it. The orientation of the **Primitive Plane** can be changed dynamically by selecting and dragging on workplane attributes.

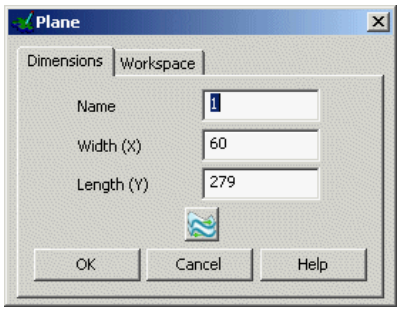
The sides of the actual **Plane** can be dynamically dragged to a different size.

- Select the **blue edge** at the **top** of the **square** and **dynamically pull** it out wards until a **value** of **280** is displayed and **release** the **mouse button.**



The width can also be modified at this point using the same method.

- Select the **blue edge** at the side of the square and **dynamically pull** it outwards to **100**.
- **Double click** on the **Plane** to open it's editing form.



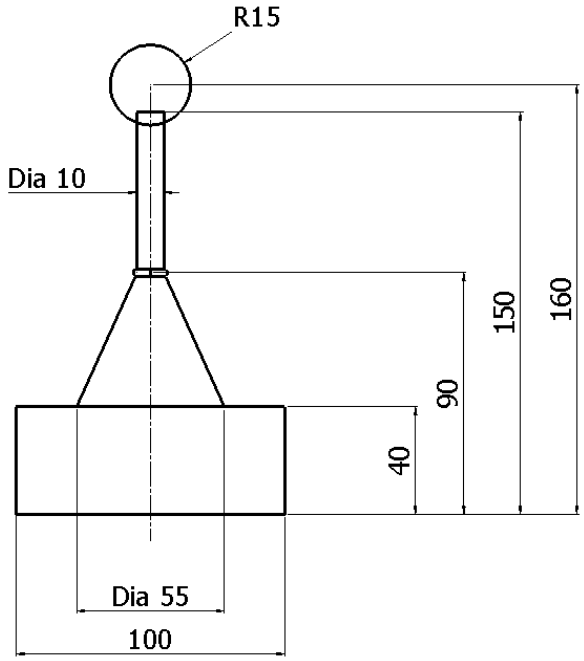
This form has two tabs and allows the user to rename or reverse the surface, input different values for width and length, re-orientate directly along a major Axis, or by applying a twist about a major Axis to change the angle.

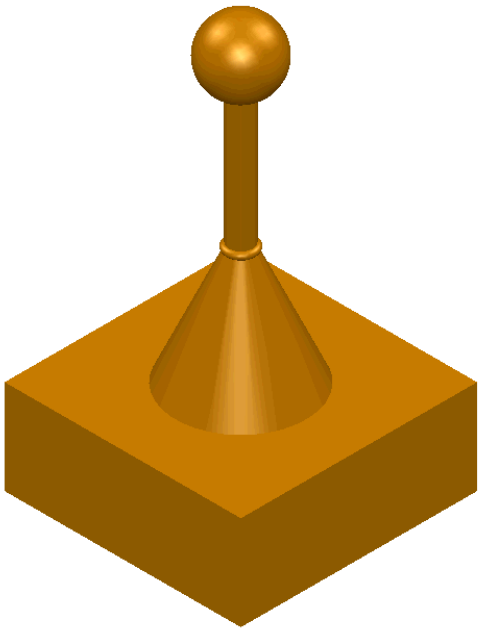
- **Change** the ***Length (Y)*** to **100.**
- Select the Workspace tab and change the ***define* Z position** to **40** before selecting **OK.**
- Press **Select** to exit **Primitive Plane** creation mode.

This **Plane** will be used as part of the next example, so do not delete it.

Simple design using Primitives

In the following example each of the six **Primitive Surface** options will be applied to the construction of a basic Gear Shift design.





- Click the **Workplane** icon and the option **Single Workplane.**
- Enter **0** in the **Command Input** box and **press Return.**

This **Workplane** will provide a visual marker for the dimensional datum.

- From the **Surface** fly-out menu , select **Block Primitive.**
- Enter **0** in the **Command Input** box and **press Return.**
- **Drag** or **Edit** the **Block** dimensions to **X 100**, **Y 100** and **Z 40.**

- From the **Surface** fly-out menu , select **Cone Primitive.**
- Enter **0 0 40** in the **Command Input** box and **press Return.**
- **Drag** or **Edit** the **Top Radius** to **5, Base Radius** to **27.5** and the **Length** to be **50.**

- From the **Surface** fly-out menu , select **Cylinder Primitive.**
- Enter **0 0 90** in the **Command Input** box and **press Return.**
- **Drag** or **Edit** the **Radius** to **5** and the **Length** to **60.**

- From the **Surface** fly-out menu , select **Sphere Primitive.**
- Enter **0 0 160** in the **Command Input** box and **press Return.**
- **Drag** or **Edit** the **Radius** to **15.**

- From the **Surface** fly-out menu , select **Torus Primitive.**
- Enter **0 0 90** in the **Command Input** box and **press Return**.
- **Drag** or **Edit** the **Minor Radius** to **1.5** and the **Major Radius** to **6.5**.

- Press **Select** to exit **Primitive Plane** creation mode.
- Select **File** - **Save**
- Search **Location D:\users\Training\Coursework\Delcam Designer Jobs**

- Enter name **GearStick.psmodel** and select **Save**.

Extrusion Surfaces

An **Extrusion Surface** uses the shape of a selected **wireframe** entity, and extrudes it as a surface, normal to the **wireframe's** planer base (default). If required, it is possible to change the settings in **Tools – Options – Objects -Surfaces** for the **extrusion surface** to generate along the active major Axis. If several items are selected for **extrusion** then a separate **surface** will be generated for each. To create as one **surface**, the **wireframes** must first be merged into a single entity, **composite curve**.

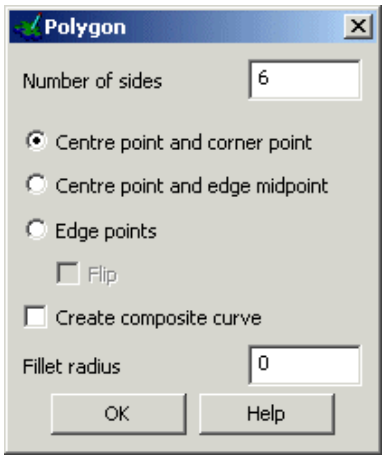
As with standard **Primitives** an **Extrusion Surface** can only have basic **parameters** modified unless it is irreversibly **converted** into a **Power surface** in which case a more comprehensive **editing toolbar** is available.

Extruded Polygon example

This example demonstrates the option to create a wireframe, **polygon** which in turn will be used to generate an octagonal **extrusion surface** with a side length of 50 and a height of 100.

- Select **Open New Model**.

- **Open** the **Line menu.**
- From the **menu** select **Polygon.**



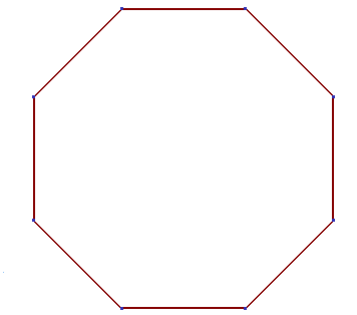
The polygon creation form is displayed. This allows you to specify the number of sides points to define a polygon that fits a circle around the outside.

The option to automatically generate a composite curve is useful as other commands, such as extrusion require a composite curve. If this option is switched off the polygon will be made up of single lines.

- Set the **Number of sides** to **8.**
- Select **Edge points** and **tick** - **Create composite curve.**

Start and end points of the first span need to be defined before dismissing the form.

- Enter **0** in the **Command Input** box and **press Return.**
- Enter **50** in the **Command Input** box and **press Return.**
- **OK** the **Polygon** creation form.

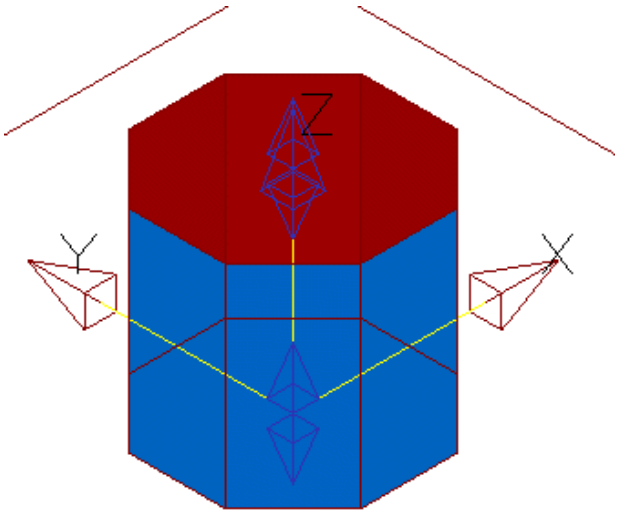


The **wireframe** is completed and is already a single closed **composite curve**.

This will be used as the shape of the **Extrusion Surface**.

- Select the **view Iso1.**

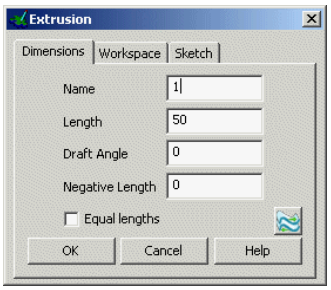
- Select the **composite curve.**
- Select the **Surface** menu and select **Extrusion.**



This produces an **extrusion surface** from the selected composite curve up the **Z-axis** by a default value.

The **extrusion** has two sets of Double Arrows which are used to dynamically change the length up or down the Z axis and if required to create a negative extrusion (lower set).

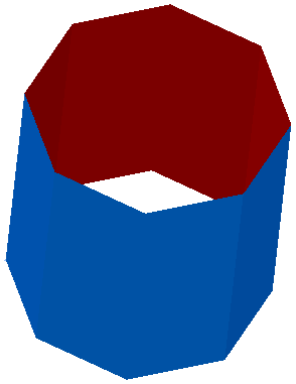
- **With** the **Left mouse button** select the **upper** set of **double arrows**.
- **Drag** the **extrusion upwards** to a **length of 50.**
- **Double-click** on the **surface edge**.



The **Primitive Extrusion** editing form will open providing the option to input settings and dimensional values. The **Extrusion** can be given a positive or negative draft angle. The **Keep** option will restore the original wireframe used to generate the extrusion.

The **extrusion** can be repositioned by entering new origin coordinates, and aligned to, or rotated about an Axis.

- **Enter** a **length** of **100** and press **OK**.
- **De-select** the **surface** by clicking away from the surface.



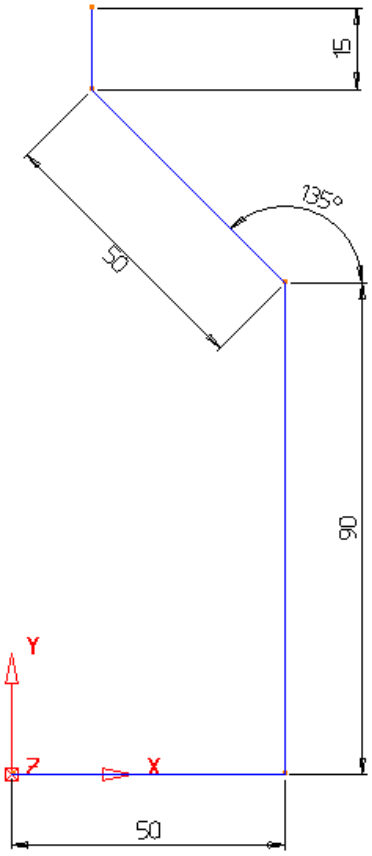
The **Extrusion Surface** is completed.

Surfaces of Revolution Example

A surface of revolution is creating by revolving single geometry or a composite curve around a specified principal plane.

- **Select Open New Model**

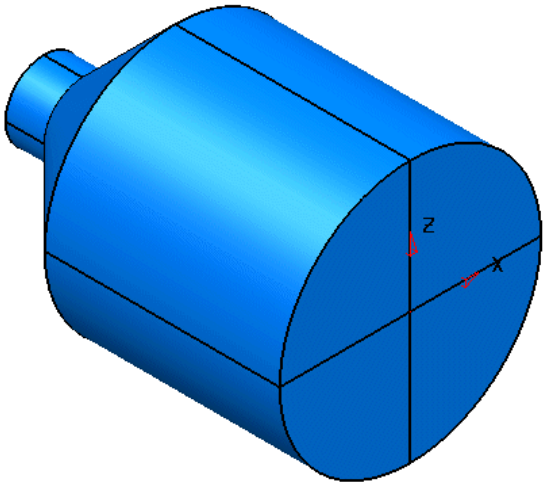
- Create a **Workplane** at **0** and create the bottle section geometry.



This shape will be connected together with a composite curve.

This curve will be revolved by 360 degrees around the workplane generating a surface. The correct axis for rotation around must be pre selected.

- Select the **Y-Axis**
- Create a **composite curve** of the section geometry.
- **Select** the **wireframe** and select the **Surface** menu.
- Select **Surface of Revolution.**

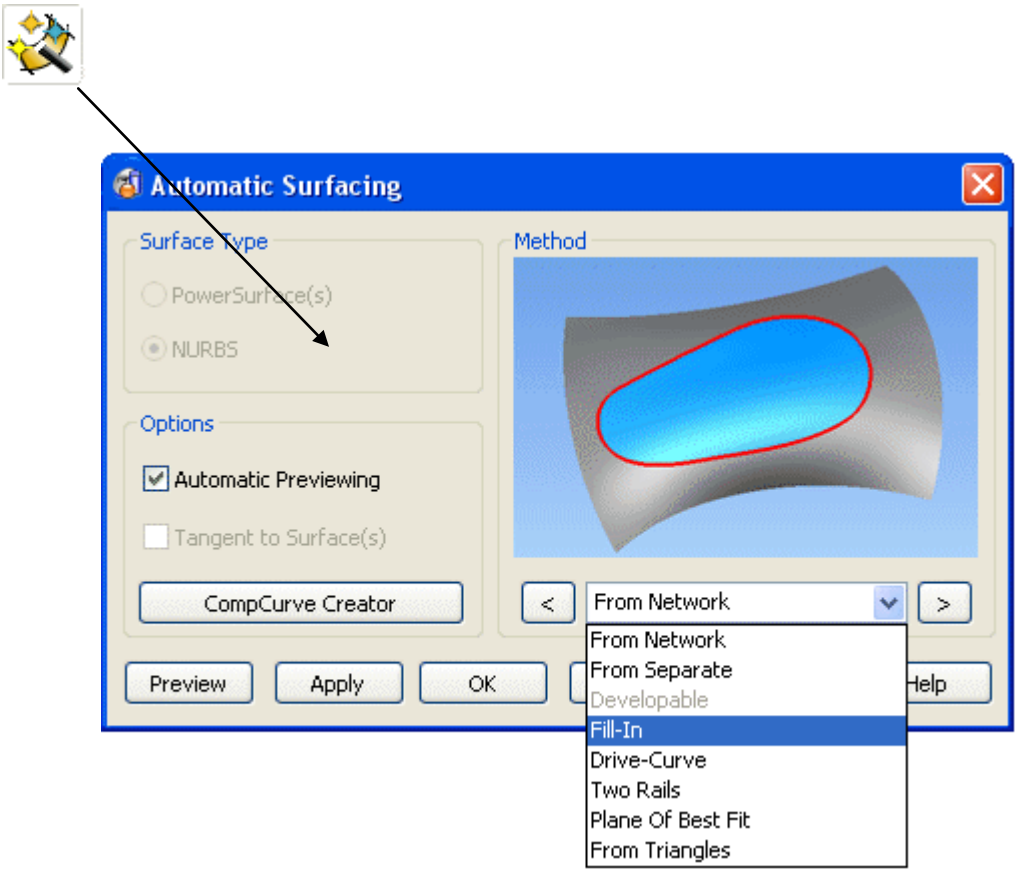


The selected composite curve is spun around the active axis (Y) producing the surface of revolution producing the bottle shape. If a lid was required on the open end of the bottle, this can be made by capturing the end curve and using the Fill in surface command.

To capture the end curve a composite curve is required.

Automatic Surfacing Wizard

Several of the **Delcam Designer surface** creation options have been included into an easy to apply, **Automatic Surfacing Wizard**. If the user selects **wireframe** before opening the wizard then the most likely surfacing method is selected ready to be previewed and/or applied. If an alternative, valid surfacing method is required then the user can toggle to it by opening the list of options and clicking on the downward pointing chevron.

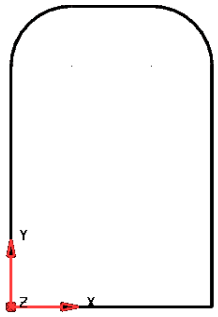


Fill-In *Surface*

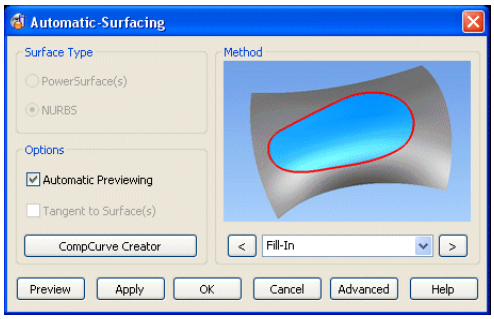
A **Fill-in surface** is usually created from one or more enclosed **wireframes**. It can also be created from open ended, **wireframes** as well as **Point** data.

Example

- Create a ***wireframe***, **Rectangle of lines** from the **0** ***datum*** with sides of **X50** and **Y75**.
- Create **R 15 fillets** at both of the top corners.

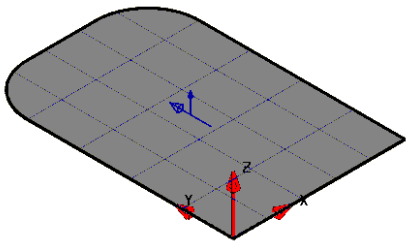


- Select all the **wireframe** entities.
- From the **main** toolbar select the **Surface** icon.
- In the ***left hand*** toolbar select the **Automatic Surfacing** option.



The most obvious option for the selected ***wireframes***, **Fill-in** will appear in the form.

- Check that **Fill-In** is the selected **surface** creation option and **Apply**.

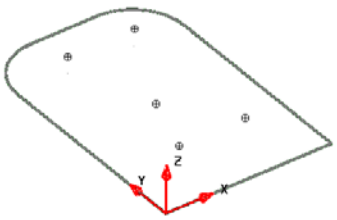


A Fill-In surface will is patched inside the enclosed **wireframe**.

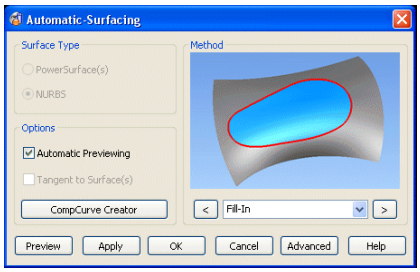
- Select and **Delete** the **Fill-in Surface**.
- From the **Main** toolbar select the **Workplane** options.

- From the **workplane** toolbar select the **Create a point** option.
- In the **command input** box enter the following 5 co-ordinate positions pressing the **Return** key after each one to accept:-

25	**37.5**	**0**
15	**15**	**5**
15	**60**	**5**
35	**60**	**5**
35	**15**	**5**

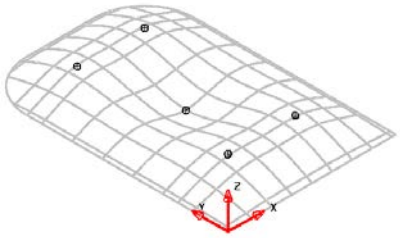


- Select all of the **wireframe** including the **5 points**.
- From the **main** toolbar select the **Surface** icon.
- In the ***left hand*** toolbar select the **Automatic Surfacing** option.



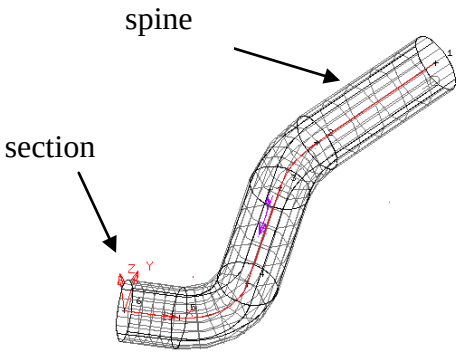
The most obvious option for the selected ***wireframes***, **Fill-in** will appear in the form.

- Check that **Fill-In** is the selected **surface** creation option and **Apply**.



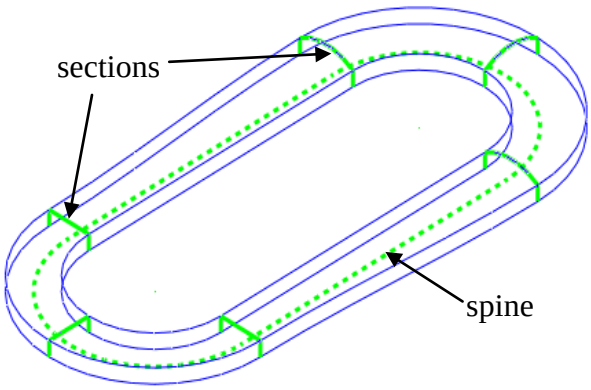
The **Fill-in surface** has blended through the selected **point** data as well as the outside wireframe.

Drive-Curve *Surface*



A **Drive Curve** surface consists of a **spine *curve*** along which **section *curves***, are aligned perpendicular relative to the ***spine points***. The illustration left shows a **Drive Curve Surface** using a single **section** curve running along the whole length of the **Drive Curve**. Minimum requirement is **one sectional *wireframe*** positioned anywhere along the single **Drive Curve**.

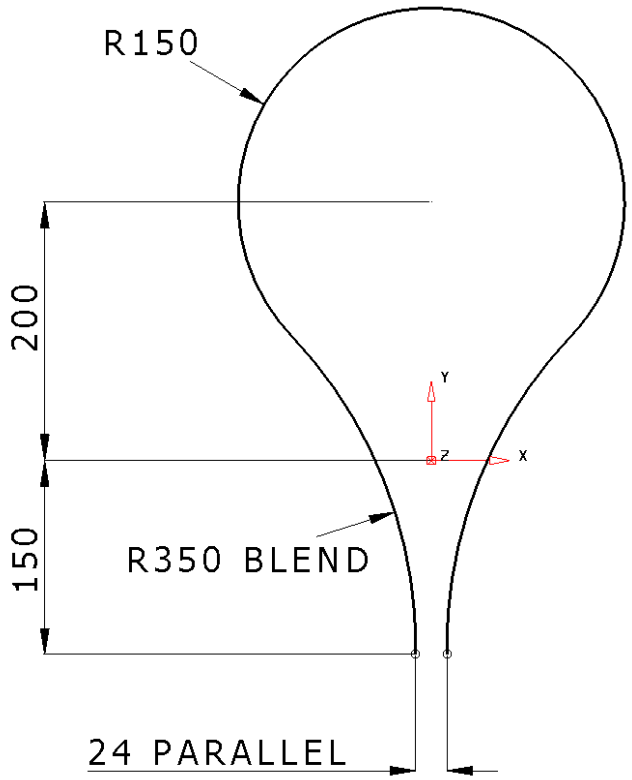
The illustration right shows a closed **Drive Curve Surface** with **2** different shaped **section** curves strategically attached to the ***spine points***.



Racket Example

The first step is to create the wireframe for the drive curve of the tennis racket.

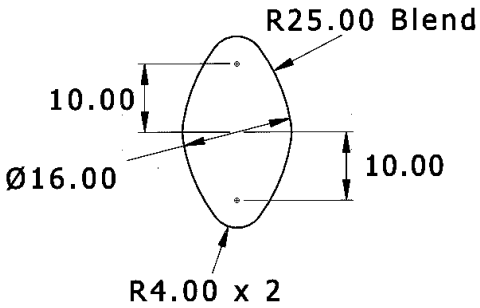
- Create a **workplane** at **0** and then construct the following **wireframe**.



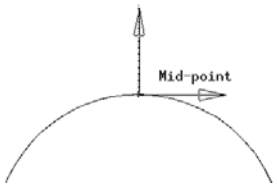
This shape will form the single **drive curve** along which the **surface** will be generated.

Wireframe sections will be defined, in this case, perpendicular to and at strategic positions along the drive curve.

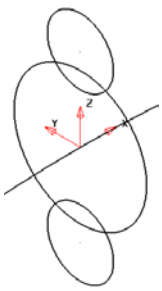
Upper Section



- Create a **workplane** and position it at the **midpoint** at the top of the racket.

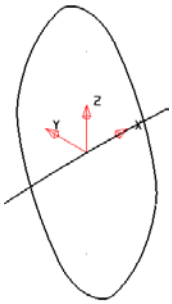


- **Zoom** into the area around the new **workplane**.
- Select the **X** principle **plane**.
- **Generate three circles**, circle 1, **radius 8** at **Z 0**, circles 2 and 3, **radius 4** at **0 0 10** and **0 0 –10**.



These will be joined by using **4 fillet arcs** of **radius 25**. During this process it will be advised to split the existing circle of radius **8** into **2** separate **arcs**, if the default ***automatic trimming*** is active.

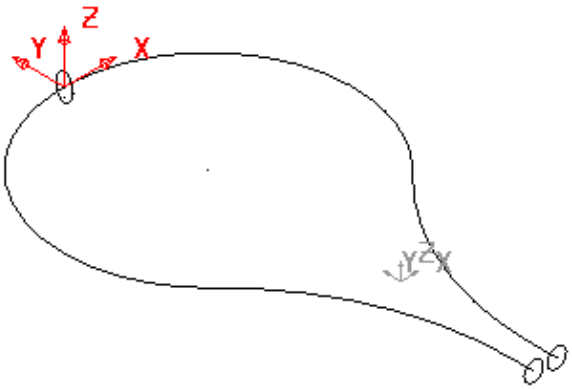
- **Fillet** the circles with an **arc** of **radius 25**.



This will be the main centre section of the racket.

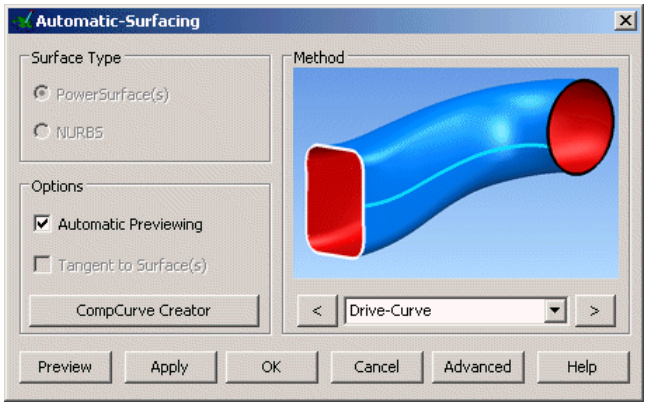
The other sections are circles and they can be placed directly at the grip end, without the need for another **workplane**.

- Select the **Y** principal **plane**.
- Generate **two circles of radius 10** with their **centres** snapped to the **end** of the lines.



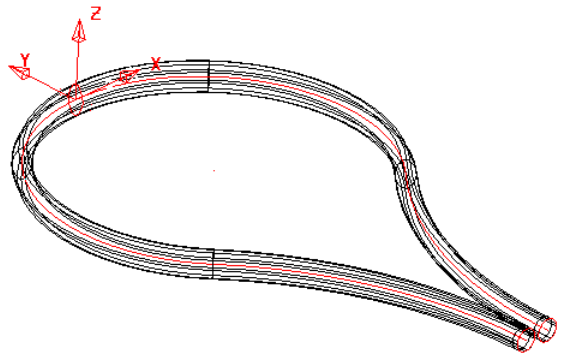
The **Drive Curve** and each **Cross Section** must be single entities before the required ***Drive Curve*** **surface** can be produced. The sections are already single **Full Arc** entities but the **Drive Curve** is a series of **arcs/lines** which must first be made into a single **composite curve**.

- **Create** a single **composite curve** out of the **Drive Curve** entities and **mid section** entities.
- **Select all three sections** and the **drive curve**.
- From the **surface menu** select the **surface wizard**.



The **surface wizard** recognises that the **wireframe** selection is suitable for creating a **drive curve surface** and changes the pull down menu accordingly. A preview of the surface is also displayed.

- Click **OK**.



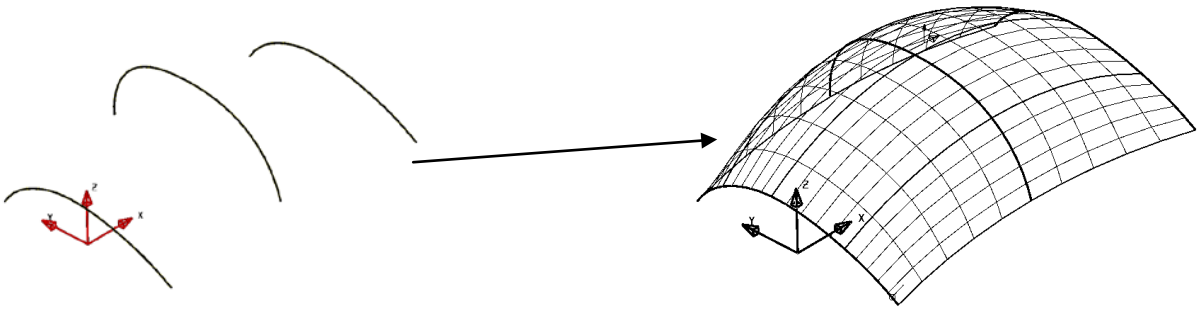
This generates a surface with a changing section from the first circle to the defined section and then back to the last circle.

- **Select File➔ Save As**:-
D:\users\training\COURSEWORK\Delcam Designer-Jobs\my-racket

- **Select File ➔ Close**.

Surface From Separate *Curves*

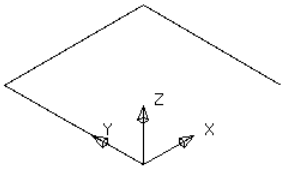
If separate curves are selected the **Automatic surfacing** wizard will use create a **surface *From Separate* curves** option.



The separate **curves** are defined across the surface (**laterals**), which are then linked with curves of best fit along the surface (**longitudinals**).

Lateral Curve Example

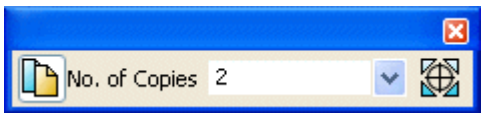
- Create a **workplane** at **0.**
- Create a **continuous line** from the **workplane**, **10mm** in **Y**, **10mm** in **X**, and **10mm** in **-Y** to produce the 'n-shape'.



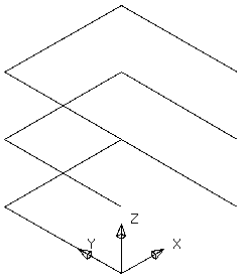
This section will be the basic for the other sections so it will be copied up the Z-axis as individual lines.

- **Select** the **three lines.**

- From the **Edit toolbar** select **Move/Copy Object.**

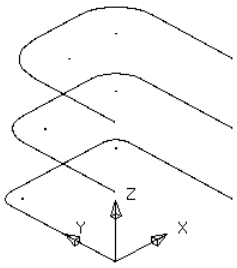
This opens the **Move** form. 

- Enter **2** for the number of copies and the value of **0 0 5** in the **position window**, giving you three sets of lines.
- **Dismiss** the **form**.



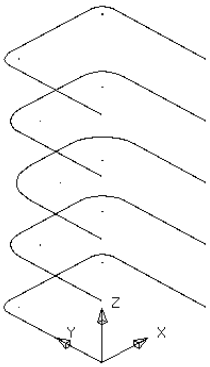
Each **composite curve** will be filleted by a different radius.

- Make **composite curves** from each set of lines (Alt key and left click).
- Create a **fillet radius** of **1mm** on the **bottom** curve.
- Create a **fillet radius** of **2mm** on the **middle** curve.
- Create a **fillet radius** of **3mm** on the **top** curve.



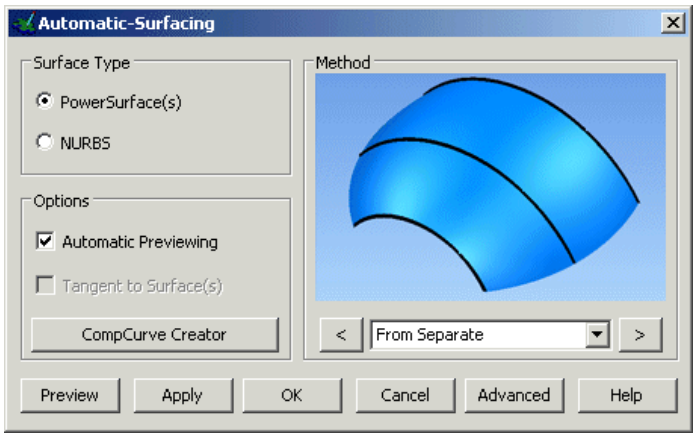
An additional two curves will be copied to produce a total of 5 curves.

- **Select** the **lower composite curve** and **Move/Copy** it up the **Z-axis** by **20mm.**
- **Select** the **second from bottom composite curve** and **copy** it up the **Z-axis** by **10mm.**



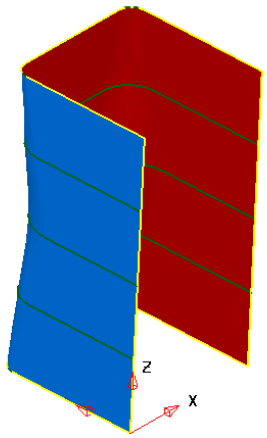
All of the shapes have been turned into composite curves, ready for surface generation. Each of the composite curves will be turned into a lateral on the surface.

- Select **all** of the **composite curves**..
- From the **surface menu** select the **surface wizard.**



The Wizard analyses the selected wireframe and selects the surface type to be created as **From Separate**

- **OK** the form.

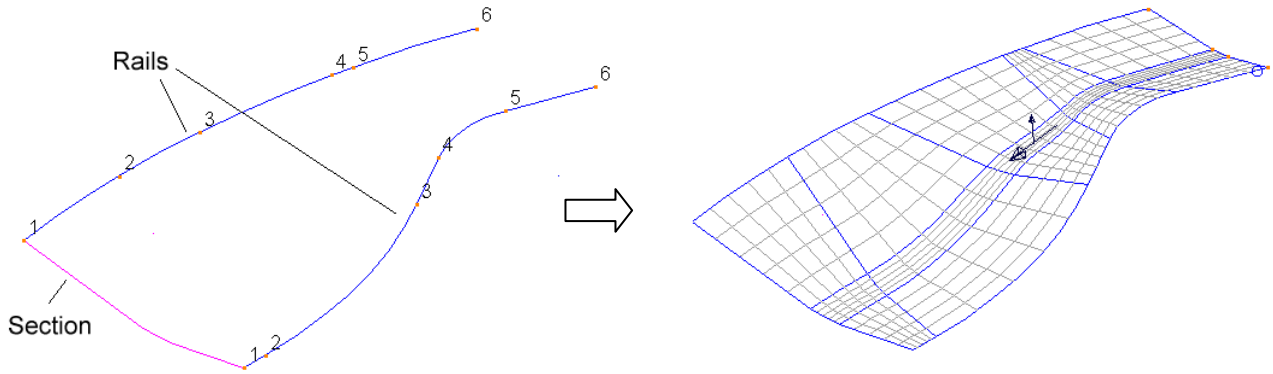


The **surface *From Separate*** curves is as shown left.

- **Select File ➔ Close** and select **No**. (to Save)

Surface from Two Rails

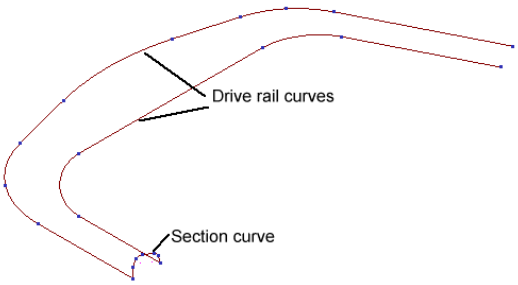
A surface can be generated from a **section**, which is scaled along two **drive rails**, defining the edge of the **surface**.



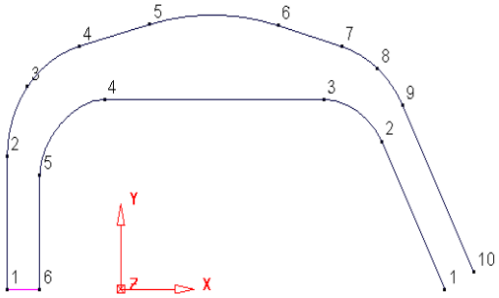
The **section** is scaled across corresponding points along the **drive rails** to generate the **surface**. **Drive rail** pairs must contain the same number of **points** and can both if required form a closed loop. The end points on an open **section** must be positioned exactly on the start point of each **drive rail**. Both the **section** and the **drive rails** must be defined as **composite curves**.

Surface from Two Rails Example

- Select **File > Import** and then select the model:- **D:\users\training\Delcam Designer data\ two_rail_data.dgk**.

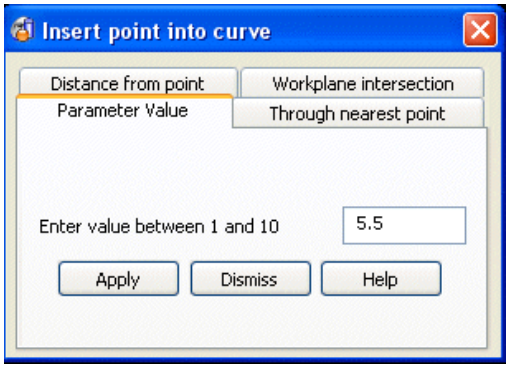


- **Double click** on one of the **drive rail** curves to bring up the **curve toolbar**.
- Shift-select the other composite curve.
- **Select show point labels**.



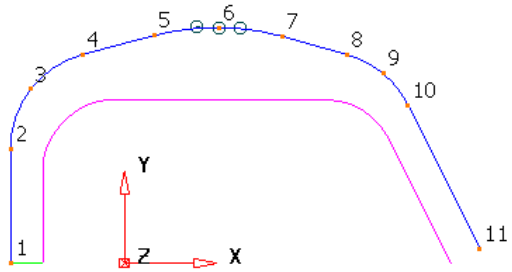
Each ***drive rail*** must have the ***same direction*** and ***number of points***. The ***section curve*** will be scaled in relation to the distance between the start and end points and corresponding points on the two **drive rails**.

- **Select** the **top curve** (10 points).
- From the **curve toolbar**, select **Create a point**.



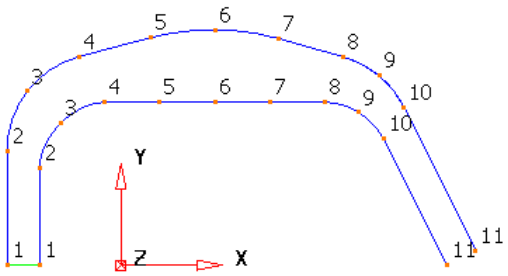
The **Insert point into curve** form appears. For this example the **Parameter Value** tab is used. This enables the user to ***insert a point*** positioned at a proportional distance between 2 existing ones. For instance **5.5** is half way between points **5** & **6**.

- Select the **parameter tab** and enter **5.5** and press **Apply** and then **Dismiss**.



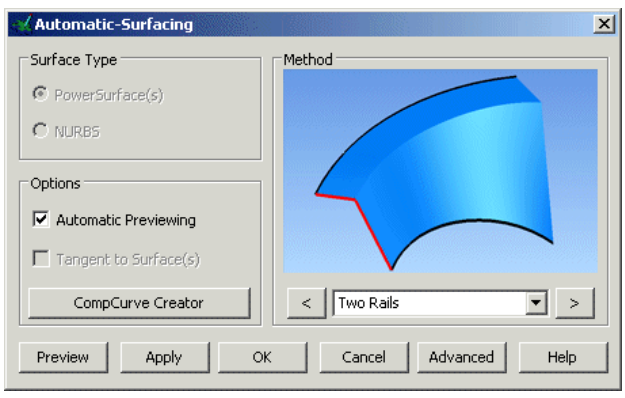
When adding each point, the curve is renumbered and it is therefore easier to add points from the last number backwards.

- **Select** the **bottom curve** (6 points).
- Select **Reverse curve** then **Create a point**
- Select the **parameter tab** and enter **4.5** and press **Apply**.
- Enter **3.5** and press **Apply**.
- Enter **4.5** and press **Apply**.
- Enter **3.5** and press **Apply**.
- Enter **2.5** and press **Apply** and then **Dismiss. Select both curves**.

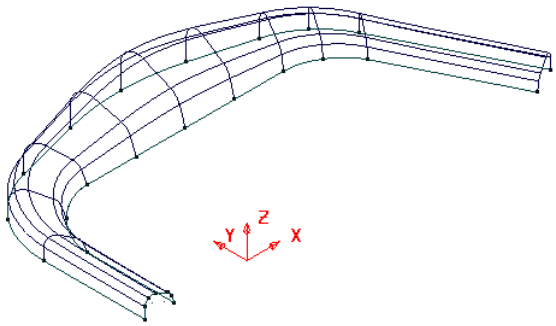


Each curve has the same number of points so when the surface is generated it will align smoothly.

- **Select** all of the **composite curves**.
- From the **surface menu** select the **surface wizard**.



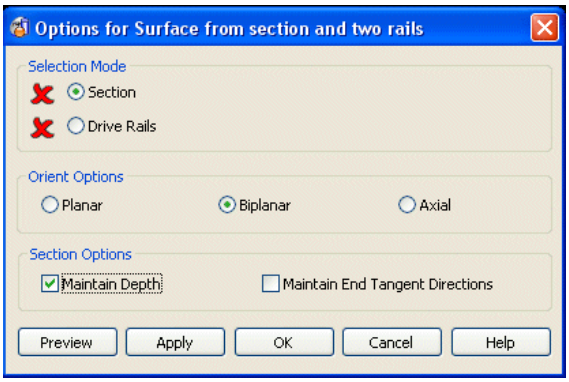
This time the wizard recognises that a Two Rails surface is the best choice for surface creation.



The preview of the surface should appear as in the picture to the left.

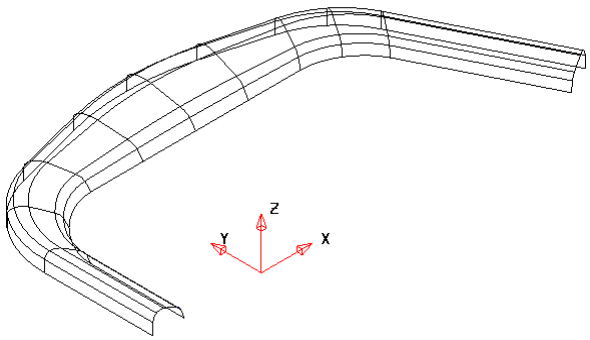
Note: The section height has scaled in proportion to the distance between the drive curves at each section position.

- Select the **Advanced** button from the bottom right of the form.



This form provides the user with more control over the way in which the surface is defined from the selected wireframes.

- ***Tick*** the option **Maintain Depth** and **Preview** again.



This time the height has been maintained to that of the original section.

- **Select OK**.
- Select and **Delete** the **surface**.

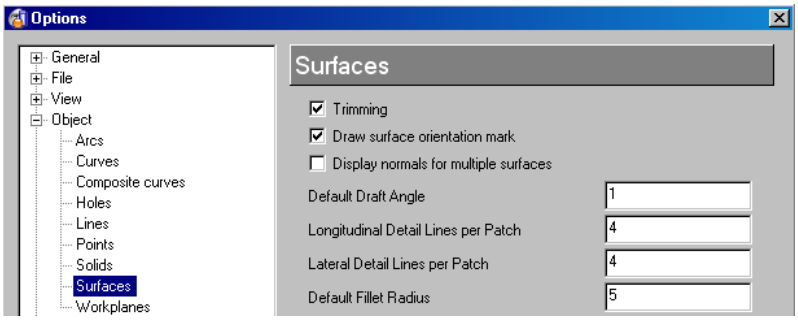
Editing Surfaces

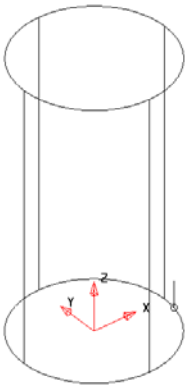
Surfaces can be edited in many ways. A whole surface or any part of it may be modified using the edit toolbar. The following example will illustrate some of the main techniques.

Adding Laterals and Longitudinals

The points on laterals or longitudinals control the surface shape. It is often necessary to add extra curves into a surface to give control at a particular place.

- Create a **workplane** at **0**
- Create a **cylinder** at **0,** along the **Z** axis, **radius 30, length 100**
- **Convert** the surface
- Switch on the **Surface Orientation Marker** in the Options form by selecting **Tools➔ Options** and under **Objects** select **Surfaces**



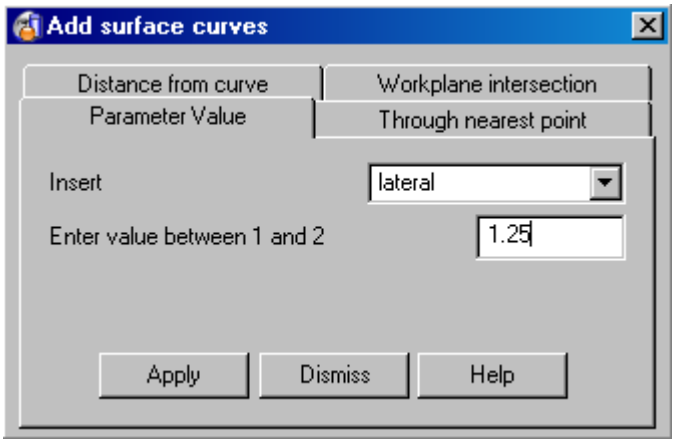


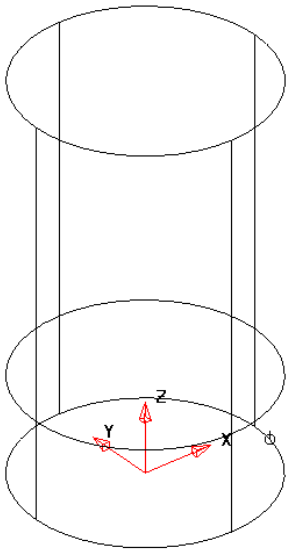
The surface orientation marker always lies on lateral 1 near to point 1. This line shows the direction the longitudinals flow along. In this case lateral 1 is the circle at the base of the cylinder, lateral 2 is at the top.

Extra laterals will now be added into the cylinder.

- Select the surface to raise the **Surface Edits Toolbar**

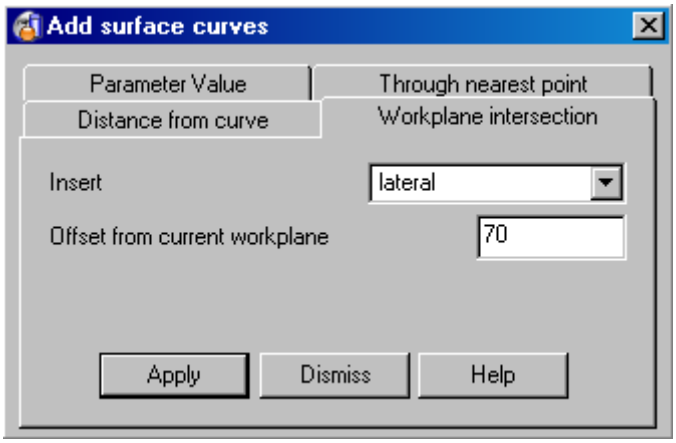
- Select the **Add Curve** icon
- In the form, select **Lateral** and enter the **Parameter Value** of **1.25**
- **Apply** the form

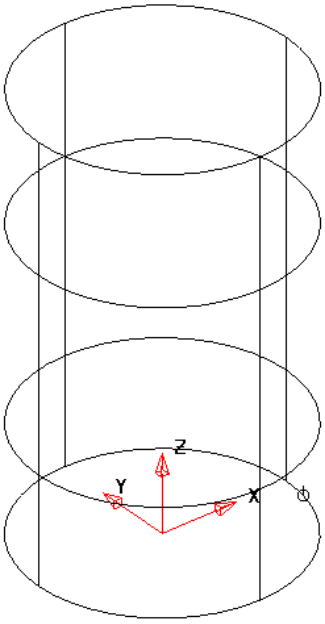




A new lateral has been created one quarter of the way between lateral one and lateral 2

- Select the **Workplane Intersection** tab on the form
- Select **lateral** and enter an **offset** of **70**
- **Apply the form**



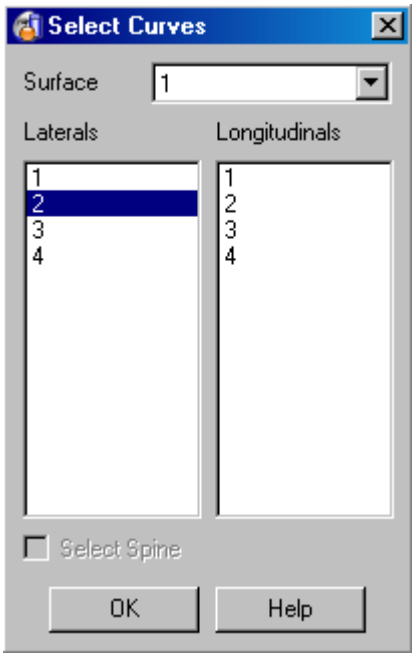


A new lateral has been created at 70mm along the active axis (Z) of the workplane.

Longitudinals are entered in the same way, the Insert option on the form is just changed from Lateral to Longitudinal.

The laterals will now be changed to give the cylinder a new shape

- Dismiss the **Add curve** form
- Select the **select curve** icon



This brings up the **Select Curve** Form with all of the laterals and longitudinals listed for the selected surface.

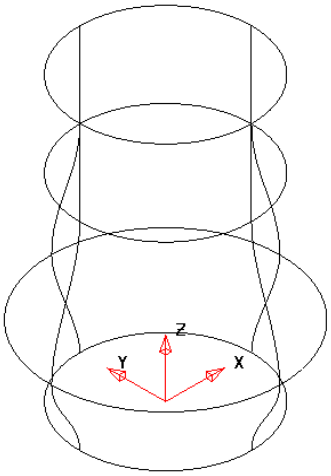
- Select **Lateral 2** as shown and dismiss the form

Curves can also be selected by picking with the cursor

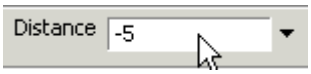
- From the **General Edits** menu , select **Edit selected sub Items** as shown.

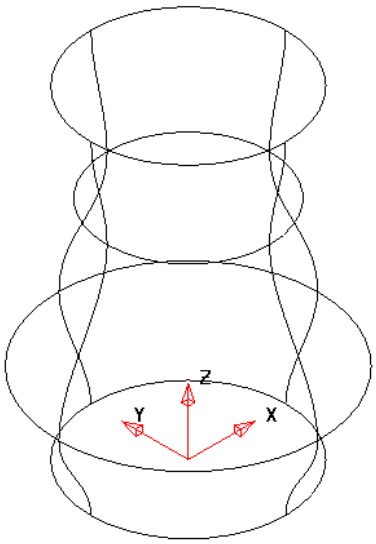
When this icon is selected, only the part of the surface that is selected will be modified, i.e. lateral, longitudinal or point. If it is not switched on, the whole surface will be modified.

- Select the **Scale** icon and then **non-uniform.**
- Expand the section by **1.33** in **X, 1.33** in **Y** and **lock** the **Z.**



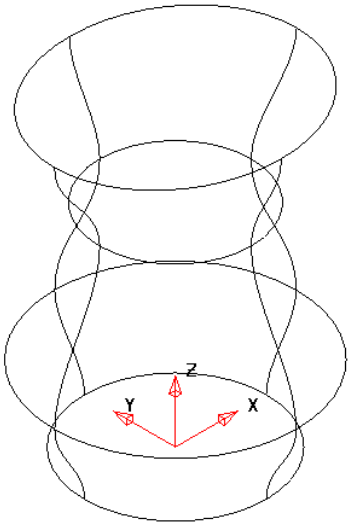
The Lateral has been scaled in X and Y but the Z position has not changed

- Select the **Lateral 3** by picking it (left mouse).
- Apply the **Edit selected sub items** as before
- Select the **Offset Object** icon
- Enter an Offset **Distance** of **–5.** 



The Lateral is offset by 5mm inwards giving this shape. The direction of the positive offset is indicated by an arrow on the lateral when you first pick the Offset icon

- Select the **Lateral 4 (Top)**
- Apply the **Edit selected sub items** as before
- Select the **Scale** icon and expand the section by **80/60** in **X, 70/60** in **Y** and **lock** the **Z.**



Note how the shape of the lateral has been changed

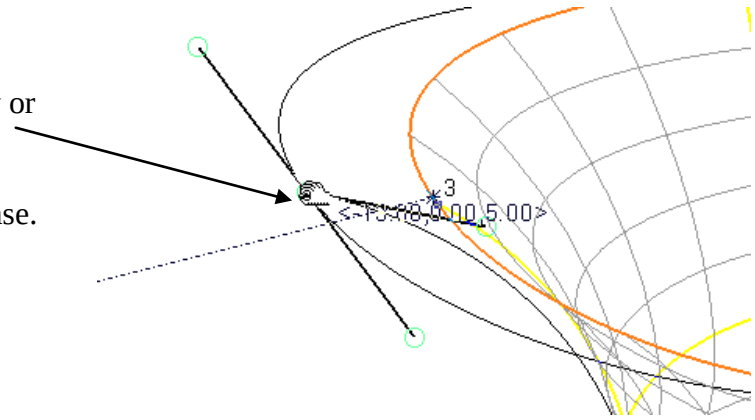
Selecting **Turn points Label ON/OFF** will show points number on the selected lateral/longitudinal.

- Select **Lateral 4**
- Pick point **1**
- Apply the **Edit selected sub items** as before

Any edits applied now will only be applied to this single point, it will now be moved by X –5

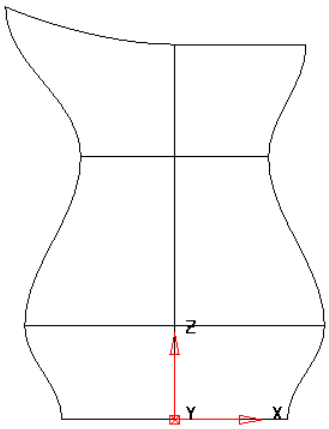
Points can be dragged manually or from inputted values.

Values will be entered in this case.

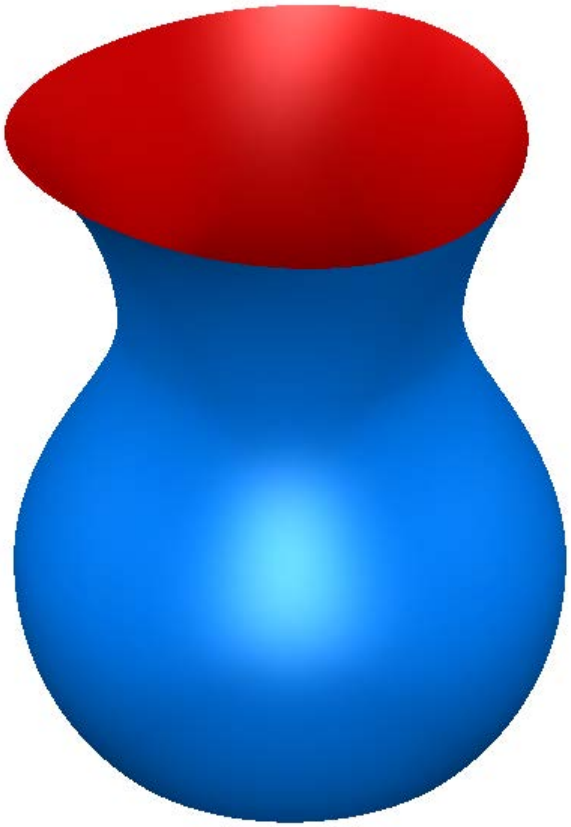


- Enter **–5** in the command window
- Pick point **3**
- Enter **–5 0 10** in the command window

- View along the **Y** axis to see the changes



Iso 1 and **Shaded** view



5. Solid Modelling

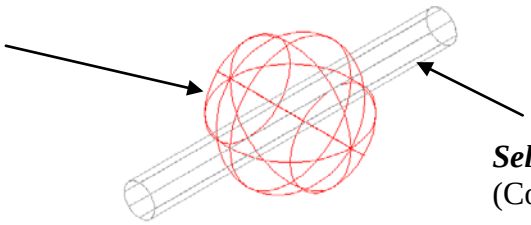
Introduction

Solid modelling applies a completely different approach in the creation of a **CAD** model. The main physical difference is that a **Surface** model is formed by a hollow, ***zero thickness skin***, whereas a **Solid Model** is a ***mass of material***. There main advantages of using **Solid Modelling** are the ***model creation speed*** and the ***history tree***, in which the user can **reposition** or **edit** operations performed earlier resulting in the ***automatic update*** of other affected items. **Delcam Designer** is unique in the ability to convert **Solids** to **Surfaces** and vice versa depending on which approach is most suitable at the time. **Open Surfaces** can be **converted** to a **Solid,** an extremely useful capability which would not be available with a dedicated **Solid Modelling Package**.

Boolean Operations

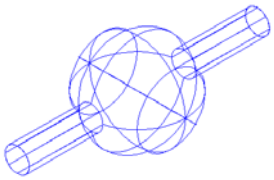
As an ***Active*** **solid** is built up other **solids/surfaces** can be absorbed into it, which is best illustrated via the basic **Boolean** operations **ADD**, **REMOVE** and **INTERSECT**.

Active **solid sphere**
(Coloured Red)



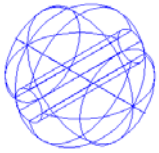
Selected **solid cylinder**
(Coloured Yellow)

Adds the **selected solid** to the ***active*** **solid**.



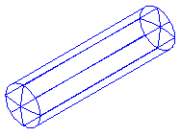
The **solid cylinder** is joined onto, and becomes part of the ***Active*** **solid sphere**.

Removes the **selected solid** from the ***active*** **solid**.



A hole appears through the ***active*** **solid sphere** caused by the removal of the **solid cylinder**, which again is now part of the ***Active*** **solid sphere**.

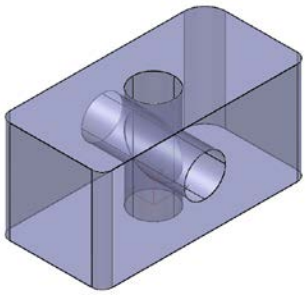
Creates the **intersection** of the **selected solid** and the **Active solid**.



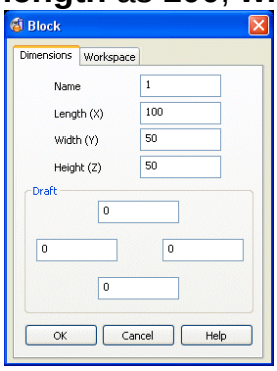
The ***active*** **solid** becomes the common volume of both **solids**, in this case producing a solid tube with spherical ends. As before the cylinder is now part of the original ***Active*** **solid sphere**.

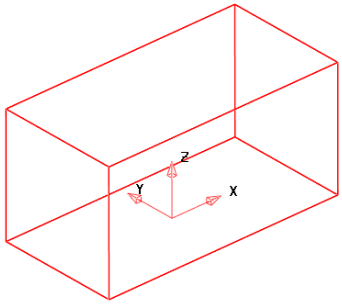
Basic Solid Model Example

The following example illustrates the creation of a simple **solid model**. It emphasises the benefits with the ease of creation, and the ability to retrospectively make changes to existing features, causing the remainder of the **solid** to be automatically updated to comply.



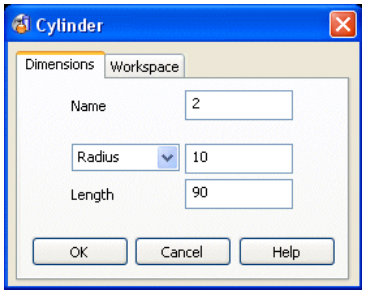
- **Open** a **new model**.
- **Create** a **workplane** at **0** and set the **principal plane** to **Z**.
- Select the **Solid** creation options icon from the **main** toolbar.
- From the **Solids toolbar** select **Create Solid Block**.
- Centre the base of the **solid block** about the **workplane** by typing **0**.
- Press **ESC** on the **keyboard** to break out of the command.
- **Double click** on the solid block in the graphics area opening the form shown below.
- Input the **length** as **100**, **width** as **50** and **height** as **50** and press **OK**.

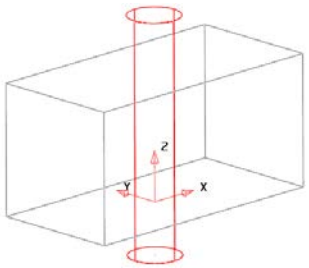




The newly created **solid block** will automatically become **active** (red).

- Select the **Solid** icon from the **main** toolbar.
- From the **Solids toolbar** select **Create Solid Clylinder**.
- In the coordinate input window to the bottom right of the graphics area input the values **0 0 –20** to position the base of the **solid cylinder**.
- Press **ESC** on the **keyboard** to break out of the command.
- **Double click** on the **solid cylinder** in the graphics area opening the form shown below.



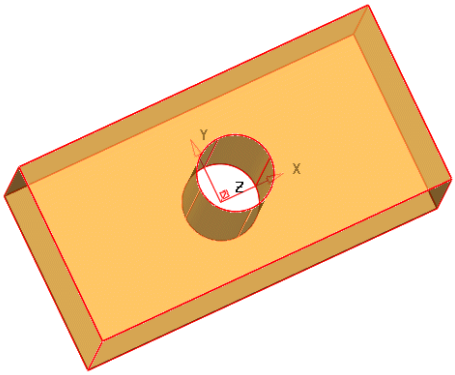


- Right click on the **solid block** and from the local menu select **Active**.

- With the left mouse, select the **solid cylinder** and from the main toolbar select the **Feature** option.

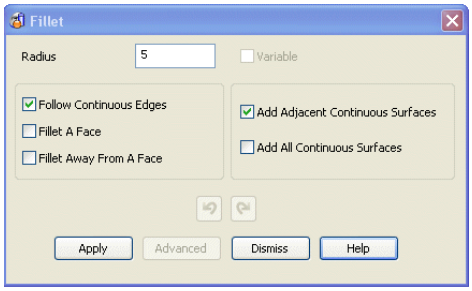
The **Feature** icon is next along from the **solid** creation icon in the main toolbar. It contains options for performing constructional operations on the initially created **active solid**.

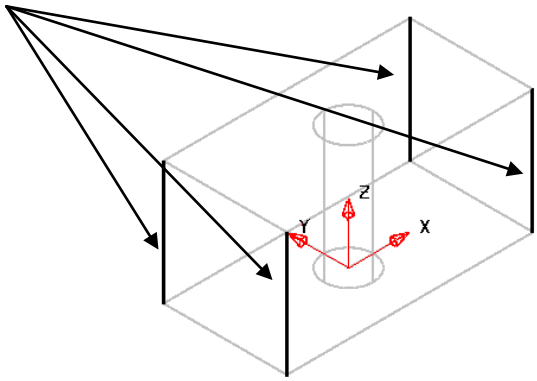
- In the **Feature** toolbar select the **Boolean** option, **Remove the selected solid …** to create the **solid cylinder** as a hole through the ***active*** **solid block**.

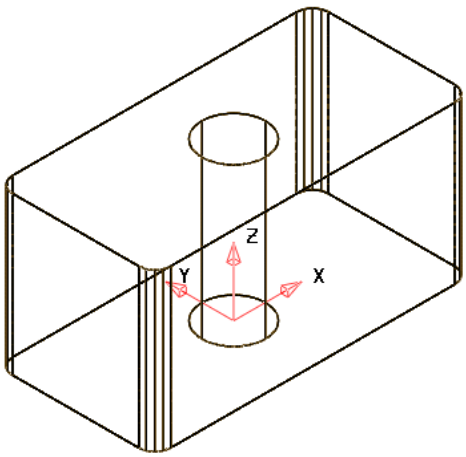


- In the **Feature** toolbar select the **Create solid fillet** option.

- In the form input ***Radius* 5** and with the **shift key** held down multi-select (***left mouse key***) the ***vertical edges*** of the **solid block** before selecting **Apply**.







All ***4 vertical edges*** will now have a **Radius 5** ***fillet***.

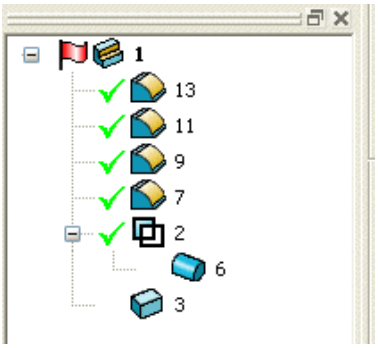
- Select **File** - **Save As** and store the model to the following location:- **D:\users\training\COURSEWORK\Delcam Designer-Jobs**

- Name file as **SolidBlockExample**, Press **Save.**

Solid History Tree

As an **active solid** evolves it absorbs other **solids/surfaces/symbols**. All these items are added to a ***history tree*** located on a dedicated **explorer** window that appears to the left of the graphics area (if it is not there double left click on the **active solid**). The last operation to be performed appears at the top of the history tree. These ***items/operations*** stored in the history tree can be ***modified retrospectively*** resulting in an automated update of the **active solid** (provided the changes are physically possible). It is also possible to ***defer the update*** if required to allow a series of changes to be implemented at the same time.

- If the ***history tree*** is not already displayed open it by double left clicking on the ***actiive* solid**.

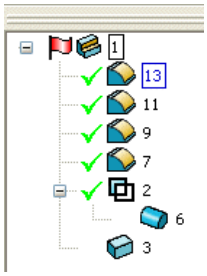


A **solid** can be made **Active** or ***deactived*** by left clicking on the **flag** (***Red flag*** = ***Active***).

If sub-items are not displayed click the open the sub-directory. To switch off the display of sub-items click on the

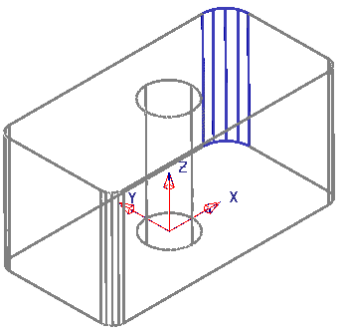
As is clear from the illustration the first action; create **solid block** operation is at the bottom of the history tree whereas the final **solid fillet** operation is at the top.

- Double click on the **solid fillet** at the top of the history tree.



A box appears around the selected **sub-item** in the history tree and the **Edit Solid Fillet** form opens.

- Input a new ***Radius* 10** and click the **OK** tab.

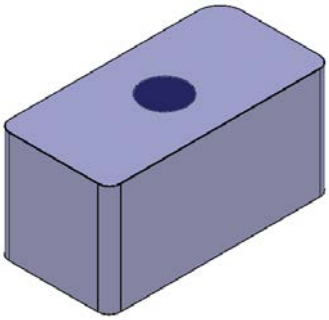


The selected **Radius** in the on the **solid** is changed to **10** and the affected area of the **solid model** automatically re-trimmed to suit.

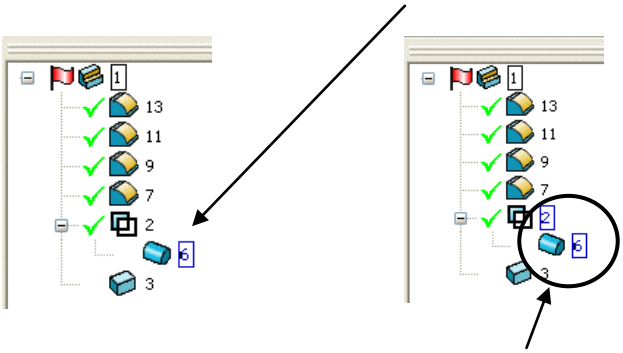
Note: It is also possible to select **solid**, ***sub-items*** using a ***double left mouse click*** directly on the **solid** in the graphics area.

As well as the ablility to modify the original parameters used in the creation of a **sub-solid**, it is also possible to apply the **General Edits** options to preceding **Boolean** operations.

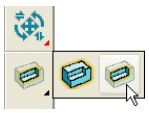
- Select the **cylinder**, ***sub-solid*** with a single mouse click and identify where it is located (boxed) in the ***history tree***.



If a **sub-solid** is selected it changes to a different colour from the **main solid** (this also applies if a **wireframe view** is displayed). The sub solid is also identified by being enclosed by a ***box*** in the **history tree**.

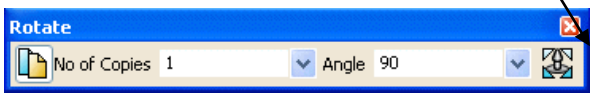


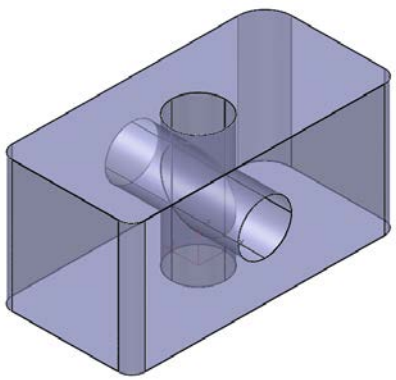
- ***Shift - Select*** the boolean **Remove** operation in the ***history tree*** such that both the **solid cylinder** and the **Remove** operation are selected (boxed).
- Select the **General Edits** options followed by **Edit Sub Item** (if not already active).



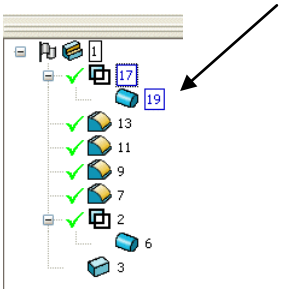
- Set **X** as the operational axis.

- Select the **Rotate object** option, entering values in the form to **Keep original**, with ***No of Copies*** **1**, ***Reposition rotation axis*** as **0 0 25**, and ***Angle*** **90** before pressing **Return**.





The original ***sub-solid*** **cylinder** has been **copied** and **rotated** within the ***active*** **solid**. The new ***sub-solid*** **cylinder** appears at the top of the ***history tree***.



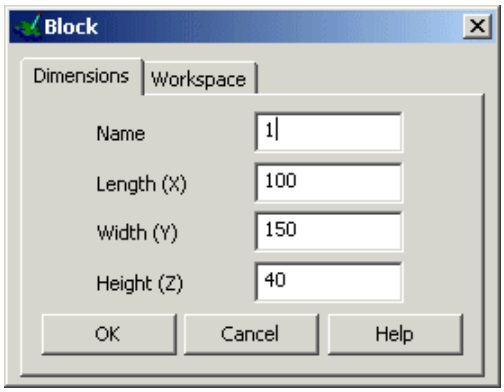
- Select **File** – **Save** to update the content of the stored model file.

- **Close** the model.

Die Example

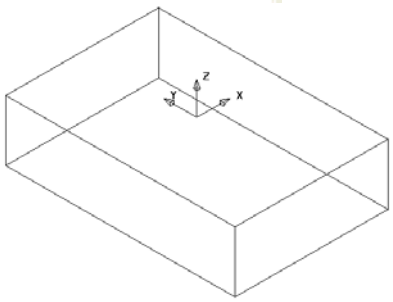
The following Die example shows the application of solids to produce complex parts quickly and easily.

- **Open** a **new model.**
- **Create** a **workplane** at **0** and set the **principal plane** to **Z.**
- Select the **Solid** icon from the top toolbar.
- From the **Solids toolbar** select **Create Solid Block.**
- Position the block by typing **0 0 –40.**
- Select to break out of the command.
- **Double click** on the solid block in the graphics area opening the form shown below.



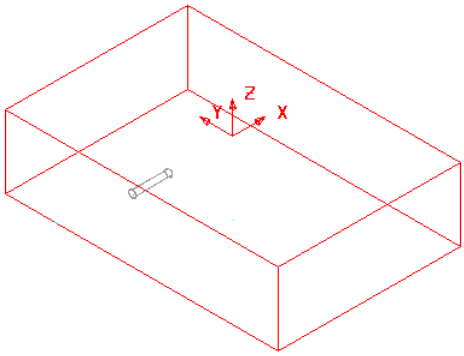
- Set the **length** as **100**, **width** as **150** and **height** as **40** and press **OK**.

PowerSHAPE also displays a **History tree** of all the solid operations. The red flag indicates that the solid called **1** is the Active solid.



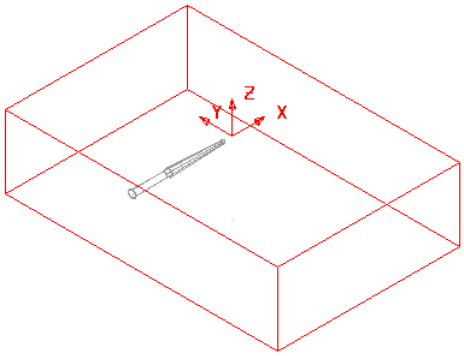
The first solid automatically becomes **Active** and is shown in **Red or Black**.

- Set the **principal plane** to **X**.
- From the **pop up** under Solid Block select **create solid cylinder.**
- Position the cylinder with the co-ordinates: **-55 0 0.**
- Change the **radius** to **2** and the **length** to **20**.



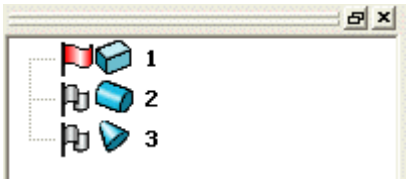
Further solids are shown in Grey, as only one solid can be Active at any time.

- Create a **solid cone** with the co-ordinates: **-35 0 0.**
- Change the **top radius** to **1, base radius** to **2** and **length** to **30.**



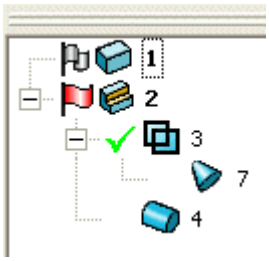
This model contains one active and 2 non-active solids. These non-active solids need to be joined together to generate a new solid.

The feature tree now contains 3 solids. The active solid has a flying red flag to indicate that it is active.



- Select the **solid cylinder** and **Right click** over it to bring up the sub-menu and select **Active.**
- Select the **solid cone.**
- From the **Features menu,** select **Add the Selected Solid to the Active Solid**.

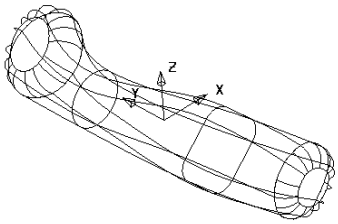
The cylinder and cone will be automatically joined together as a new active solid.



In the history tree, the second solid has become the active one and it shows that solid cone has been added to the solid cylinder. This solid can be removed from the active solid by clicking on the tick. The solid can be added in again by selecting the cross.

The component to go into the die will be imported as surfaces and then turned into a solid.

- **Blank** all of the **solids** and **import** the model **handle.dgk** from **D:\users\training\Delcam Designer data**



The model is made up of various surfaces, which will be turned into a single solid.

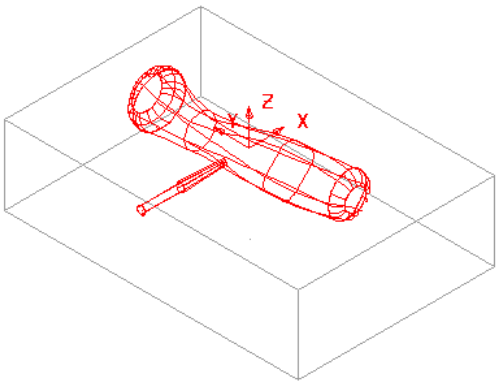
- **Select all surfaces.**
- From the **Solids toolbar** select **Solid from Surfaces.**
- **Select Unblank**.(Ctrl + L).

The surfaces have become one single solid. If the surfaces do not match a watertight wizard will appear to fix any gaps.

A copy of this solid should be made for later use.

This new solid can now be added to the original solids.

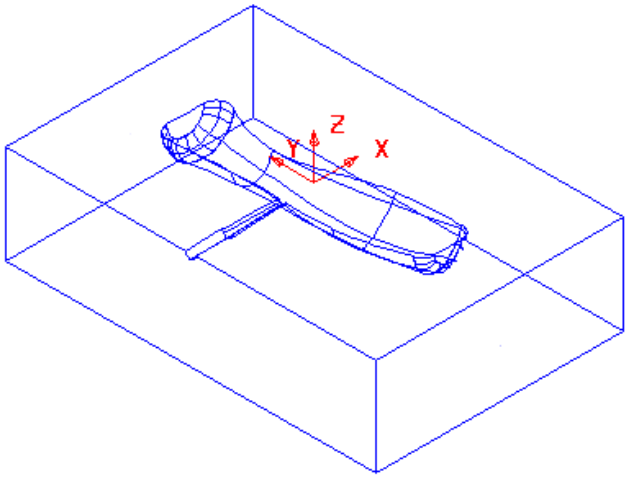
- With the 'gate' **solid Active**, **select** the handle **solid.**
- From the **Features toolbar** select **Add the Selected Solid to the Active Solid.**



To produce the cavity in the die block, this final solid can be subtracted from the die block. .

- Make the **block solid active** and **select** the **Handle & Gate Solid.**

- From the **Features toolbar** select **Remove the selected Solid from the Active Solid.**



- **Close** the Model without Saving.

6. Three Stone Ring

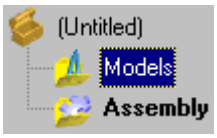
Ring Design

The object of this exercise is to design the Ring seen here.

We will be creating the design using a mixture of ArtCAM JewelSmith and Delcam Designer functionality.

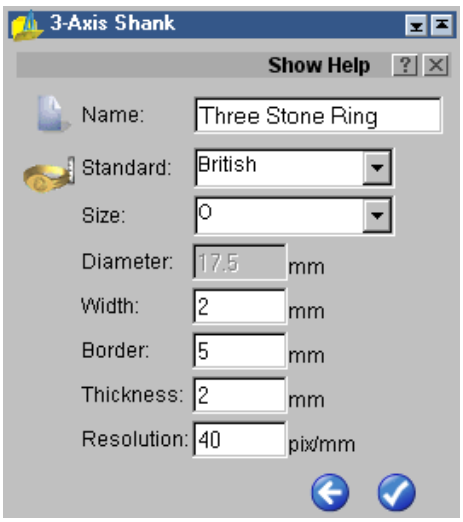
- Open ArtCAM JewelSmith

- Select the Create New Project button.

- Select Models from the Project Tree. 

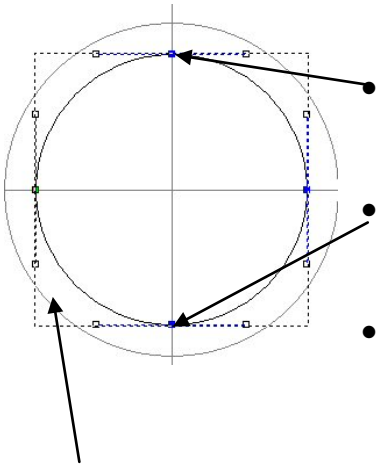
- Select the Blank - 3-axis

- In the File Name area type in PS-Designer Ring with British Size O, Width 2, Border 5, Thickness 2 and Resolution 40.

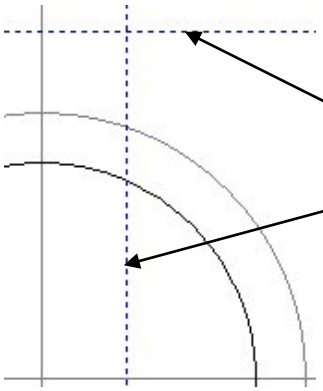


- Select the **Accept** button.

- Make a copy of the inner ring vector. **(Ctrl + C, Ctrl + V)**
- Make sure the inner ring vector is selected and press **N** on the keyboard to enter **Node Editing**.

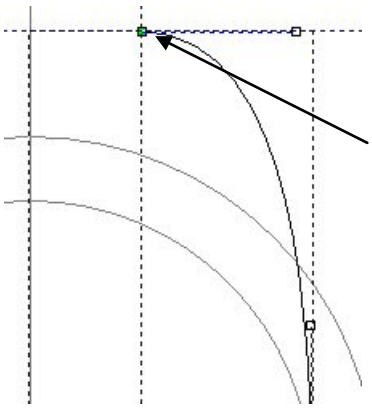


- Position the cursor over this node and press the **C** button on the keyboard to **Cut** the vector.
- Repeat the procedure for the bottom opposite node.
- This will have cut the ring vector into two equal halves.
- Delete the left of the two vectors.
- Press the **Esc** key on the keyboard to exit **Node Editing**.

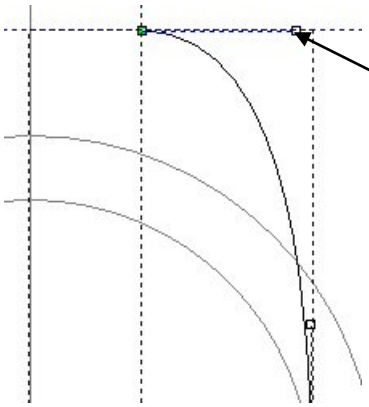


- Drag down a **horizontal guide line** and position it at **Y 14**.
- Drag across a **vertical guide line** and position it at **X 3.5**.
- We will use the intersection of these two guide lines to help create the ring profile.

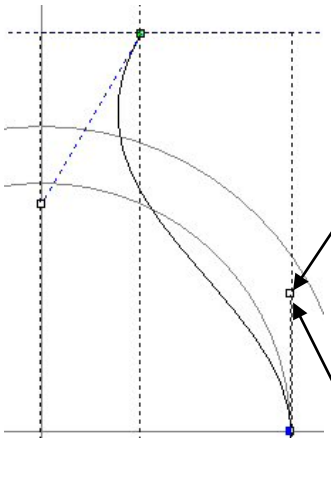
- Select the half vector created earlier and press **N** to enter **Node Editing**.



- Drag the top node to the intersection of the two guide lines as seen here.

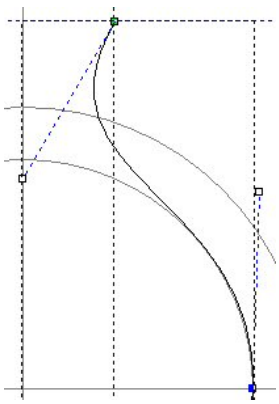


- Right mouse click over this node and select **Properties**.
- Change the **X Coordinate** to **0** and the **Y Coordinate** to **8**.
- Select **OK** on the form.



- The vector should look something like this.
- We now need to move this node to complete the shape required.

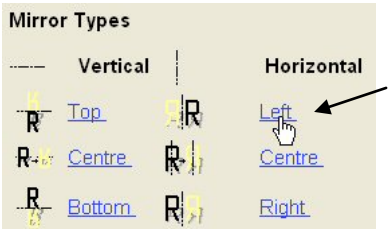
- Right mouse click over this node and select **Properties**.
- Change the **X Coordinate** to **9** and the **Y Coordinate** to **7.5**.
- Select **OK** on the form.



- The vector should look something like this.
- Make any minor adjustments of your own choice to the nodes if you so desire.
- We now need to mirror this vector to the left to create the other half.

- Press the Select button to exit **Node Editing**.

- The new vector should still be selected so click on the **Mirror Vectors** button in the **Vector Tools** area of the **Assistant page**.



- Select **Left** to create a new copy of the original vector.
- **Close** the form.

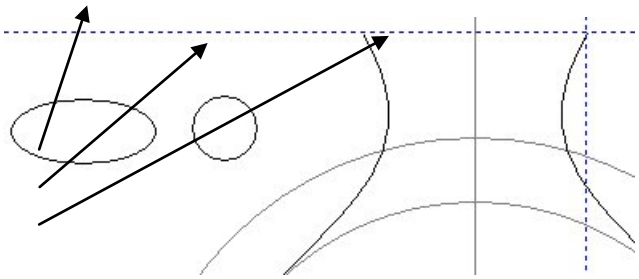
Both of the two new vectors need to be joined together as one.

- Hold the **Shift** key down and select both new vectors.
- Click on the Join Vectors with a line button.

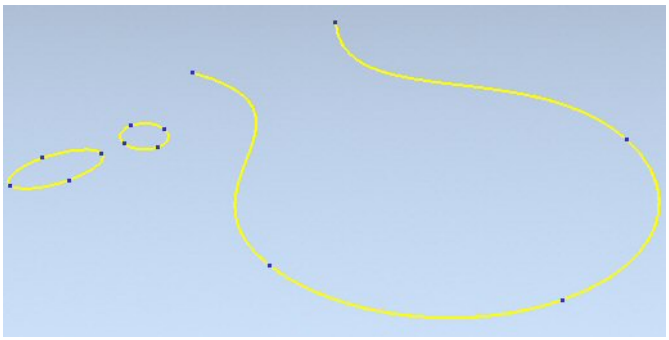
This has now completed the design of the ring profile that will be used in Delcam Designer.

We now have to create the cross section vectors that will be used in conjunction with the ring profile vector to create the main ring shape.

- Create a Circle of diameter 2mm.
- Create an Elipse with Width 4.5mm and Height 2mm.
- Holding down the **Shift** key select the three vectors created.
- Press **(Ctrl + C)** to copy these vectors to the clipboard.



- Open **Delcam Designer**, right click in model space and select **Paste** from the menu.



The vectors from ArtCAM are pasted in flat down in Z and the X 0 Y 0 position remains at the centre of the ring profile here.

The vectors are yellow in colour, which means that they are selected.

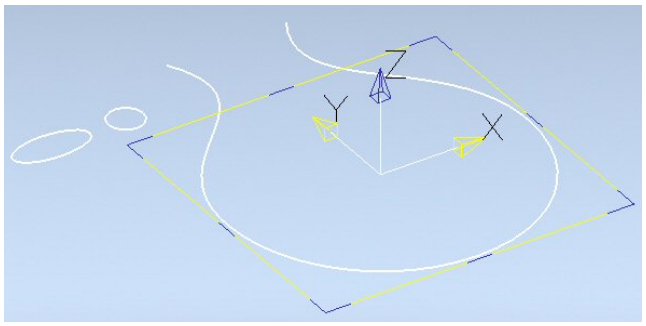
- Click away from the vectors to deselect them and they are now white in colour.

N.B. Most items when selected in Delcam Designer are yellow.

The next part of the exercise is to position, rotate and move the vectors into the correct orientation to allow the main shank surface to be created.

- Select the **Workplane** icon on the top toolbar.
- Type **0** in the command window at the bottom of the screen.

- Press **return** on the keyboard.



This new **Workplane** will become the main datum for the ring design.

- Click away from the Workplane and you will see it has turned red.

This indicates that it is the **Active** Workplane and hence **X 0, Y0** and **Z 0**.

More Workplanes can be created in different areas and orientations on a model to make it easier to create objects such as wireframe around it.
Each time a new Workplane is created it becomes Active and also the new X 0, Y0 and Z0 position.
Workplanes can be positioned by using the mouse onto keypoints, intersections etc. or by typed position input in the command window.

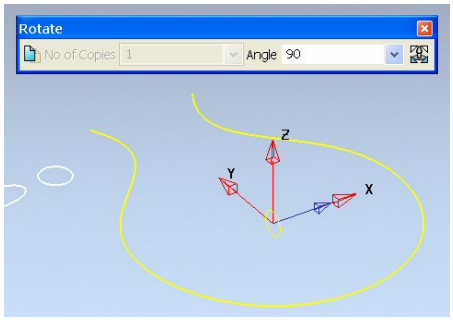
Now that the Workplane has been created we can rotate the ring profile vector into position.

- Select the ring profile vector.
- Select the **Show general edits options** button on the right toolbar.

- We need to Rotate the curve by 90 degrees firstly around the X axis then around the Z.
- Click on the **use the YZ 'face' of the Workplane** button on the bottom toolbar.

This sets the rotation axis as X.

- Click on the **Rotate Object** button on the left toolbar.

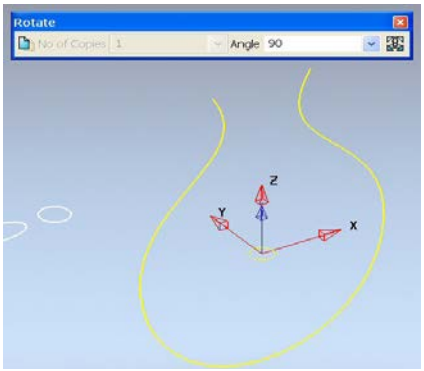


The blue arrow here indicates the current rotation axis. In this case the X.

- Type in **90** in the **Angle** box and press return on the keyboard.
- Do not close the form yet.

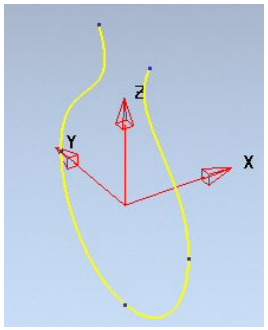
We now need to rotate the curve around the **Z** axis.

- Click on the **use the XY 'face' of the Workplane** button on the bottom toolbar.



The blue arrow points upwards indicating the current rotation axis is now Z.

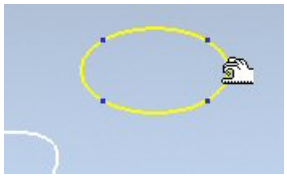
- Type in **90** in the **Angle** box and press return on the keyboard.
- **Close** the Rotate form.



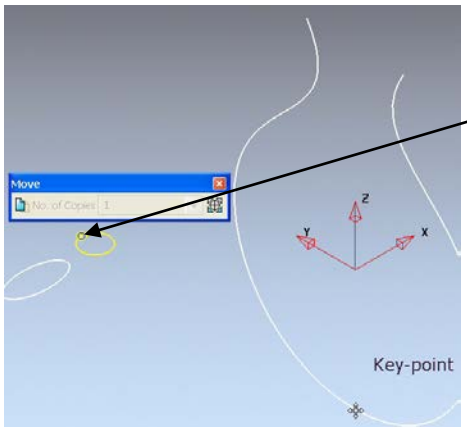
The ring profile curve is now in the correct orientation as it would be in ArtCAM JewelSmith.

We will now start to position the circle and elliptical cross sections.

- Click away from the curve to deselect it.

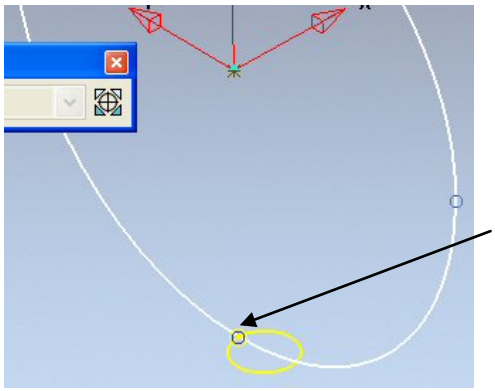


- Select the **2mm diameter circle** that we pasted in earlier.
- Select the **Move/Copy Object** button from the left hand toolbar.



The 'move from' indicator appears on the circle here.

- Position the mouse cursor at the **Key-point** at the bottom of the ring profile curve.
- Click the left mouse button to position the 2mm circle.

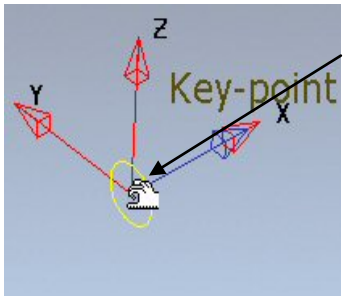


The circle moves to the key-point but remains in the wrong orientation.

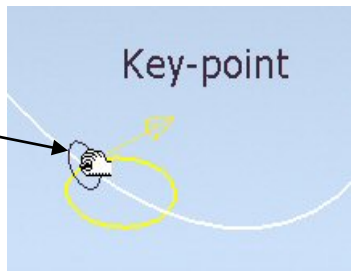
We need to rotate it 90 degrees around this point and in the X axis.

- Click on the **use the YZ 'face' of the Workplane** button on the bottom toolbar.

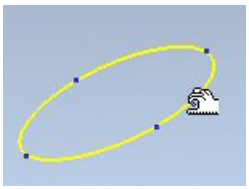
- Click on the **Rotate Object** button on the left toolbar.



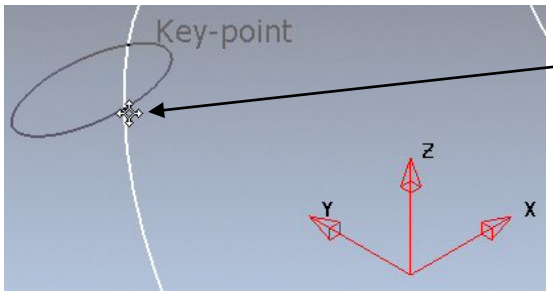
- Grab the rotation axis arrow with the left mouse button and drag it to the key-point at the bottom of the ring profile curve.
- Click the left mouse button to position it.



- Type in **90** in the **Angle** box and press return on the keyboard.
- **Close** the Rotate form.

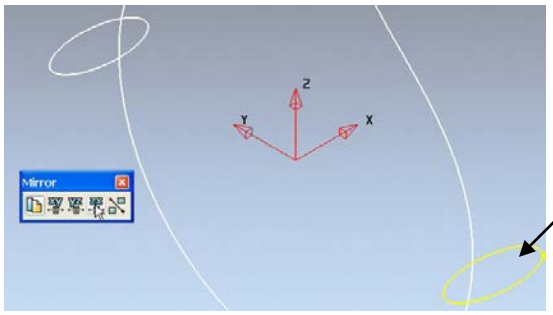


- Select the elipse curve with the left mouse button being careful not to select one of the blue node points.



- Drag the elipse to the key-point shown here.
- Click the left mouse button to position it.

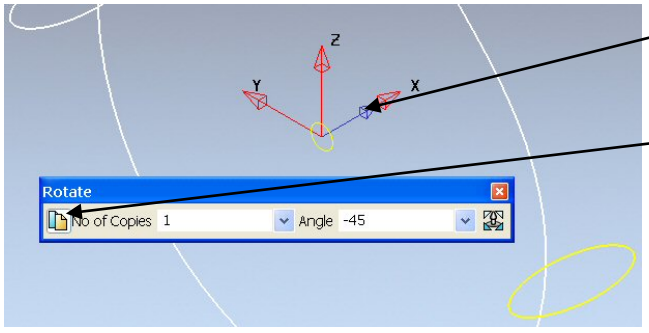
- Select **Yes** to the **Confirm Drag by** box.
- With the elipse still selected, click on the **Mirror/symmetrise object** button on the left toolbar.



- Click the **ZX** button to create a new elipse on the opposite side.

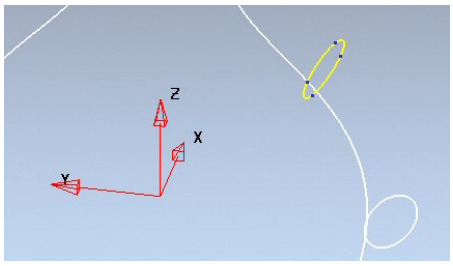
We will now create two further elipses up and around the ring profile curve.

- With the second elipse still selected click on the **Rotate object** button on the left toolbar.



The blue arrow should still indicate that we are still working in the YZ.

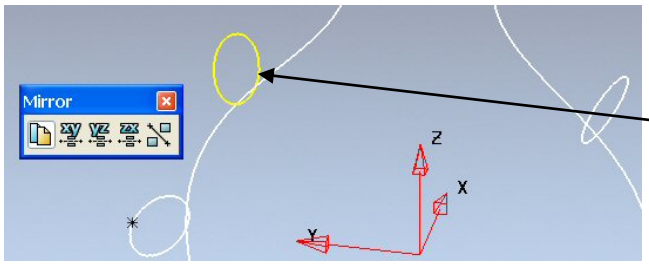
- Click on the **Keep original** button on the left of the form and type **45** in the Angle box.
- Press return on the keyboard.



A new copy of the elipse is positioned around the ring profile curve.

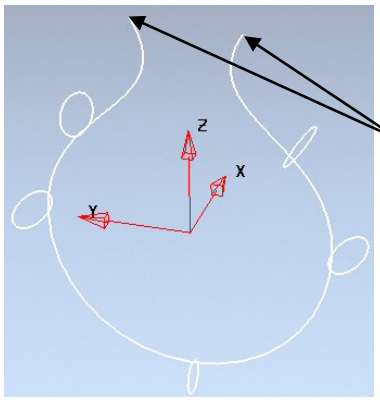
We will now mirror this to create one on the opposite side.

- With the elipse still selected, click on the **Mirror/symmetrise object** button on the left toolbar.



- Click the **ZX** button to create a new elipse on the opposite side.

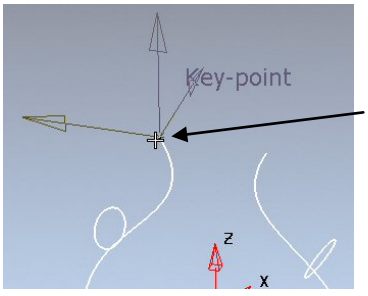
- **Close** the **Mirror** form.



To finish off the cross section construction we need to place two 5mm diameter circles on the ends of the profile curve and at the same time be aligned to the geometry.

We can do this by creating a Workplane Aligned to Geometry.

- Click on the **use the XY 'face' of the Workplane** button on the bottom toolbar.
- Select the **Workplane** icon on the top toolbar.
- Select the **Single Workplane aligned to geometry** button on the left toolbar.



- Position the Workplane on the **key-point** seen here.
- Click the left mouse button to position it.

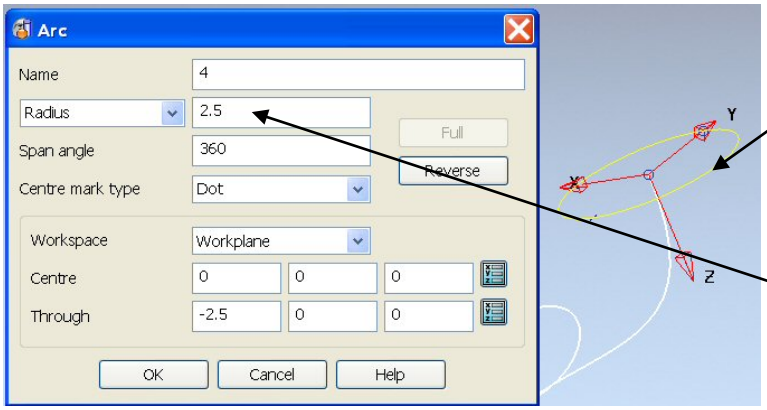
- Select the **Arc** button on the top toolbar.

- Select the **Create a full arc (circle)** button on the left toolbar.

- Type **0** in the command window at the bottom of the screen.

- Press **return** on the keyboard.

- Press to exit circle creation mode.



- Double click the new circle with the left mouse button to open the **Arc** form.

- Enter a **Radius** of **2.5** then click **OK**.

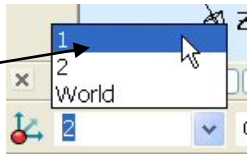
- Select the **Show general edits options** button on the right toolbar.

- With the circle still selected click on the **Move/Copy Object** button from the left hand toolbar.

- In the bottom command window type in **2.5** then hit return on the keyboard.

- **Close** the Move/Copy Object form.

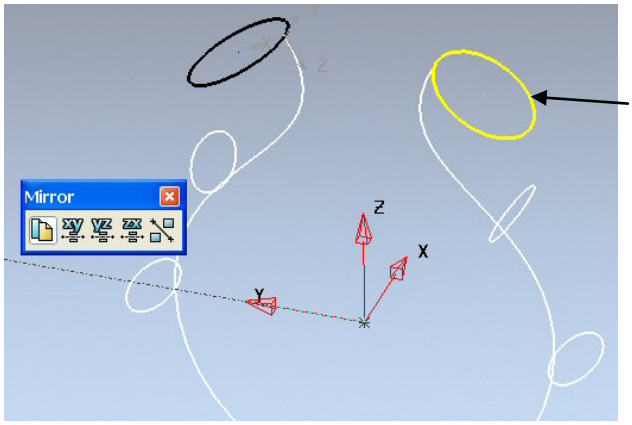
- At the bottom left corner of the screen select **1** from the Workplane list.



This will deactivate the second Workplane and activate our first Workplane in the centre on the ring profile.

- Select the **Show general edits options** button on the right toolbar.

- With the **5mm circle** still selected, click on the **Mirror/symmetrise object** button on the left toolbar.



- Click the **ZX** button to create a new 5mm circle on the opposite side.

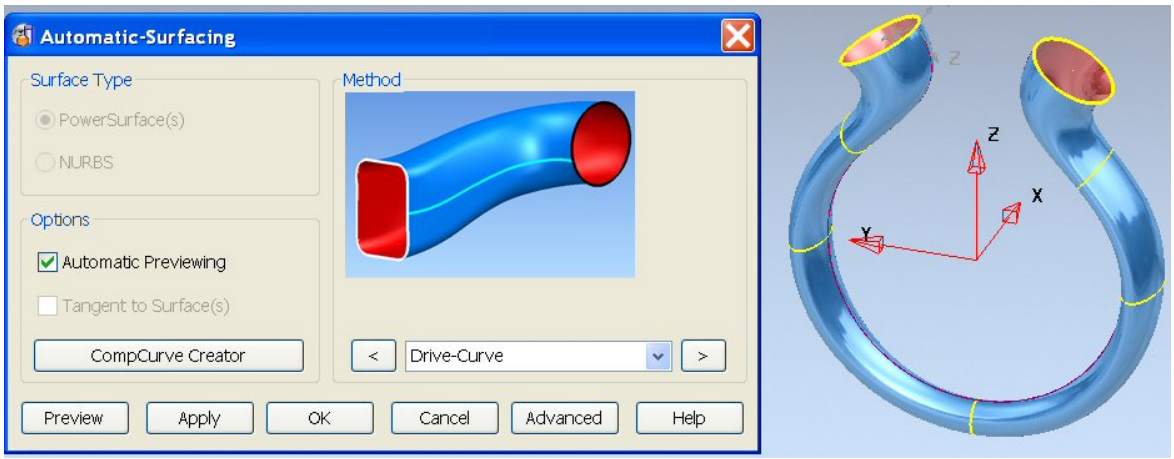
- **Close** the **Mirror** form.

- Our wireframe for the ring shank has now been created.

- Drag a window around all of the curves in the model. They will all turn yellow.

- Select the **Surface** button on the top toolbar.

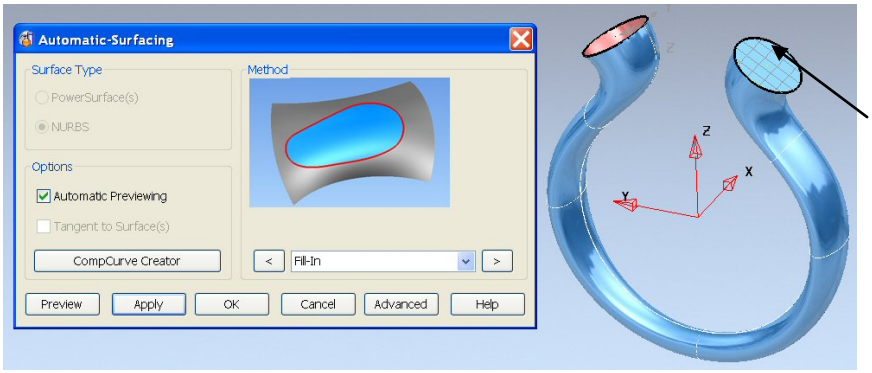
- Click the **Automatic-Surfacing** button on the left hand toolbar.



A new surface is automatically created based on the curves we have made earlier.

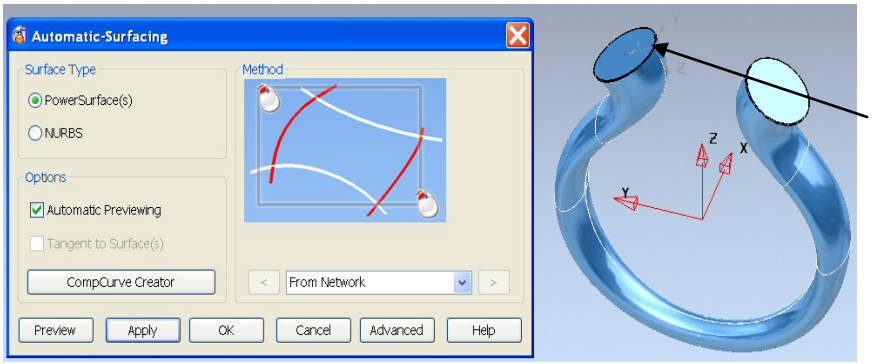
- Click on the **Apply** button.

- The Automatic – Surfacing form can stay open at this stage so we can continue to create the surfaces to fill in the ends of the design.



- Click on this 5mm diameter circle to see it automatically create a fill in surface.

- Click on the **Apply** button.

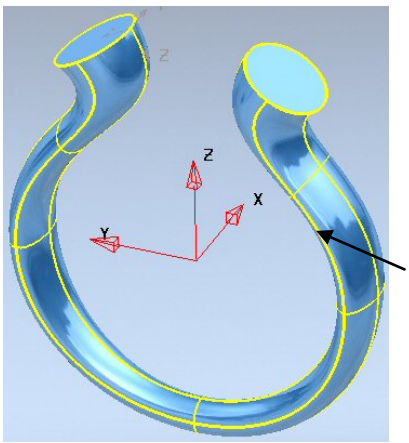


- Click on the remaining 5mm diameter circle to create the second fill in surface.

- Click on **Apply** then **OK** to close the form.

- The main design of the shank is almost complete. Next we will create the setting area for the stones.

- Because **Delcam Designer** is a hybrid surface and solid modeller we will convert these three surfaces into one solid, hence speeding up the process to create cavities etc using Boolean functionality within the solid options.

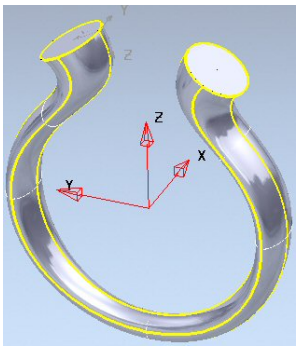


- Drag a window around the whole model to select all items.

When selected they should look like this.

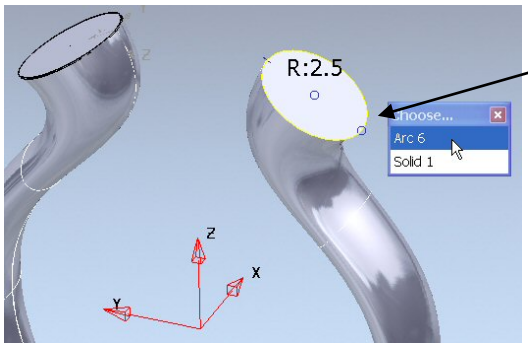
- Click on the **Solid** button on the top toolbar.

- Click on the **Create solid from selected surfaces** button on the left toolbar.



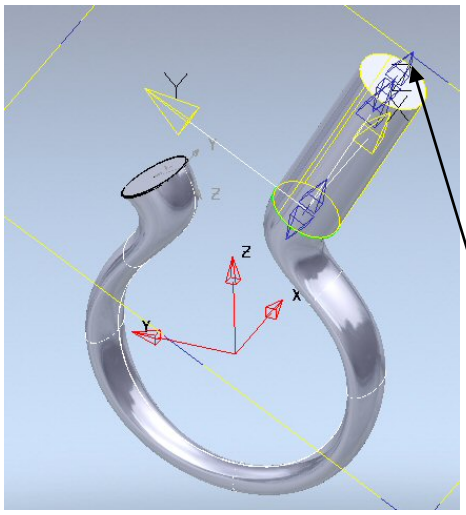
The three surfaces have now become one solid, hence the indication by change in colour. Grey represents a solid.

- Deselect all items by left clicking anywhere in model space away from the design.



- Select the right hand **5mm diameter circle** by clicking on it then accepting the choice from the Choose form.

- Click on the **Create one or more solid extrusions** button, on the left toolbar.



The extrusion is created but needs to be adjusted to the correct length.

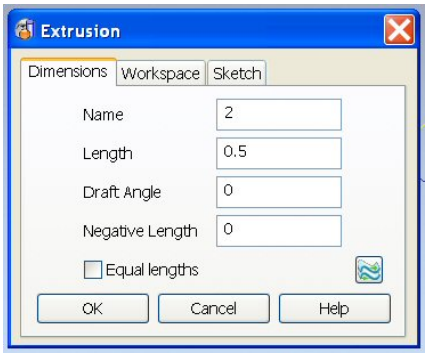
There are two ways you can do this.

One is to drag the blue arrow down until you can see the correct value appear.

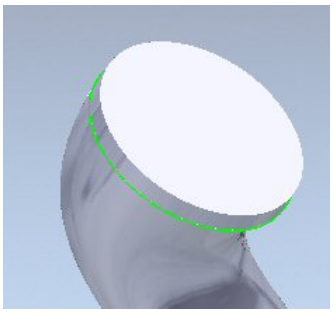
Two is to double click the extrusion with the left mouse button to open the Extrusion form.

We will use option 2 in this case.

- Double click the extrusion with the left mouse button to open the Extrusion form.

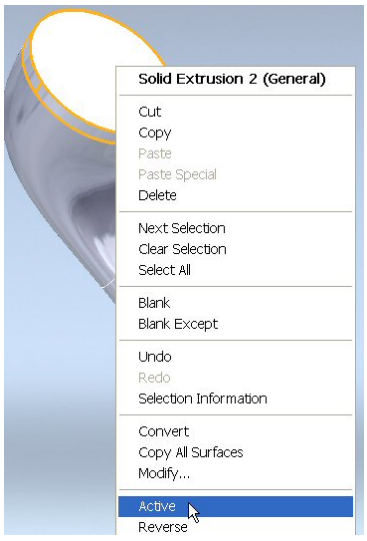


- Change the **Length** to **0.5**
- Select **OK**.



The extrusion is now the correct length but requires a fillet to be added to the top edge.

For the Solid Fillet functionality to work we need to make the extrusion the **Active** solid.



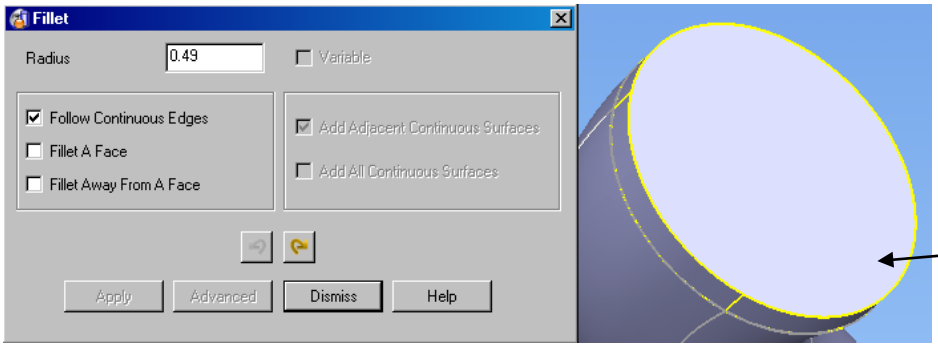
- To do this, simply right click over the extrusion and select **Active** from the menu.

A tick will appear next to the word Active to indicate whether Active or not.

N.B. For other Solid Boolean functionality to succeed, such as Solid Addition and Subtract you need to have one active solid (not selected) and at least one inactive solid (selected).

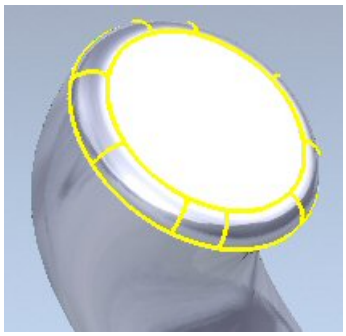
- Select the **Feature** button on the top toolbar.

- Select the **Create solid fillet** button on the left toolbar.



- Type in **0.49** in the Radius box then select the top edge of the extrusion as seen here.

- **Apply** and **Dismiss** the form.



The fillet is successfully created.

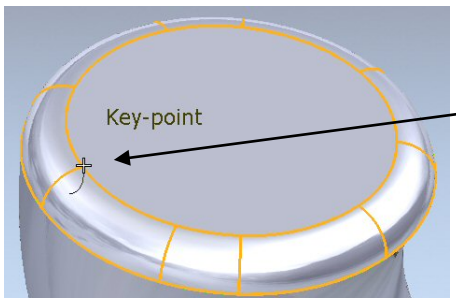
We now need to create the cavity for the setting.

To do this, first we need to create a Workplane that sits on the top of the extrusion and perpendicular to the top face.

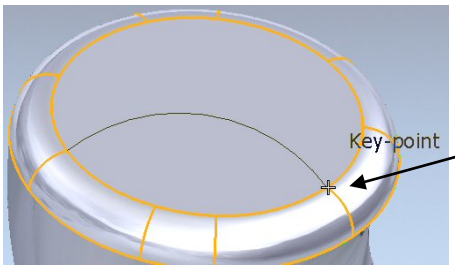
The easiest way to do this is to create a piece of wireframe that the Workplane can be aligned to.

- Select the **Arc** button on the top toolbar.

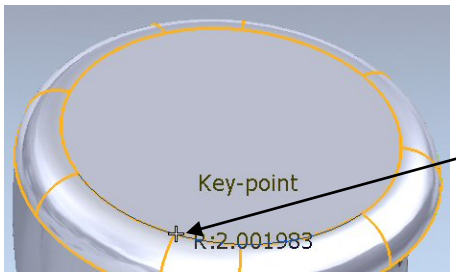
- Select the **Create an arc through three points** button on the left toolbar.



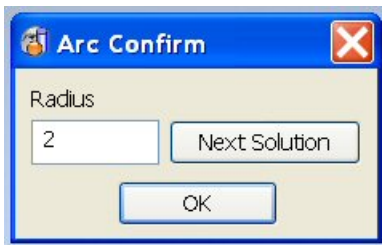
- For the first point click this Key-point here.



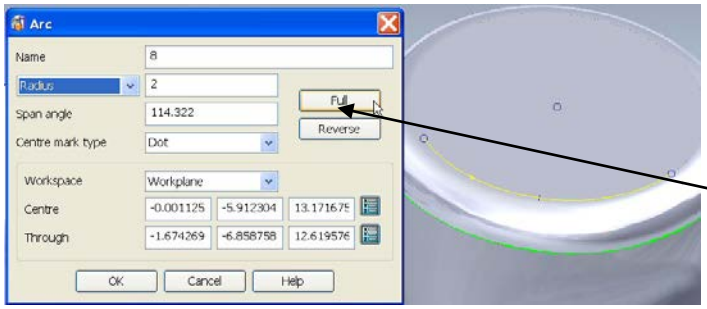
- Click on the second Key-Point here.



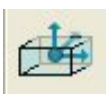
- Finally click on the third Key-Point here.

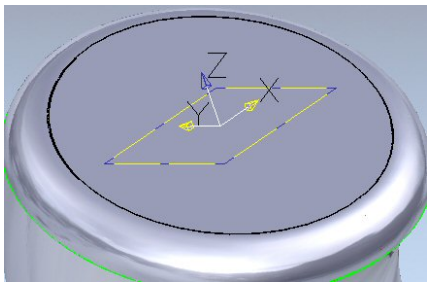


- Round up the Radius value to **2mm** then click on **OK**.



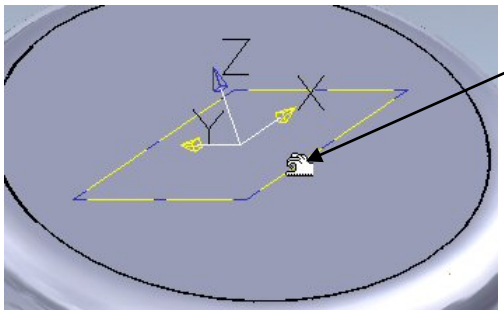
- Double click the 4mm diameter circle to open the Arc form.
- Click on **Full** to make the circle complete.

- Click on **OK** to close the form.
- Select the **Workplane** icon on the top toolbar.
- Click on the **Single Workplane at centre of selection** button on the left toolbar. 



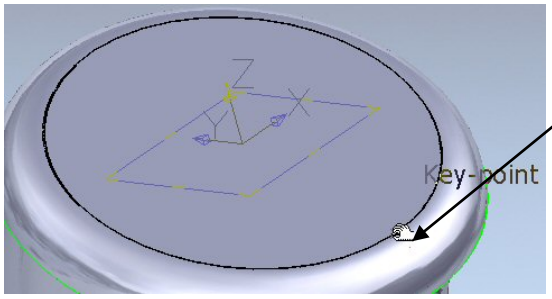
The Workplane still appears at an angle to the top face. This is because it is still relative to the original Workplane 1 in the centre of the shank.

We will use the yellow handle bars to drag around the X and snap to the top face.

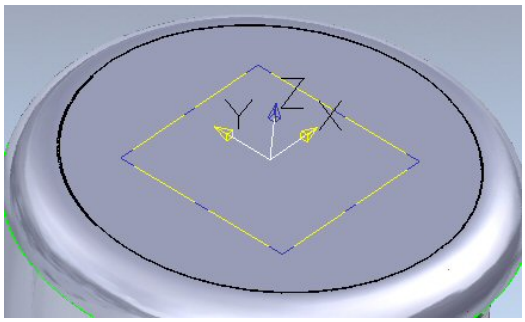


- Position the mouse cursor over the yellow handle bars here.

The intelligent cursor hand will appear.

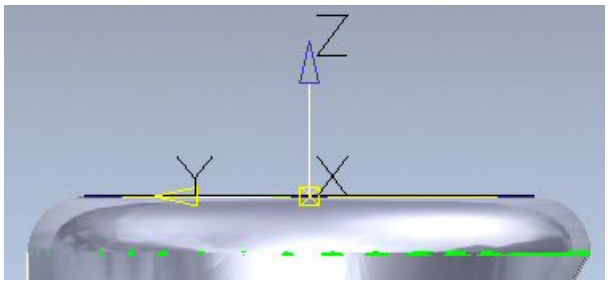


- Holding down the left mouse button move the grab hand to the Key-Point shown here then let the mouse button go.



The Workplane should now be aligned and perpendicular to the top face.

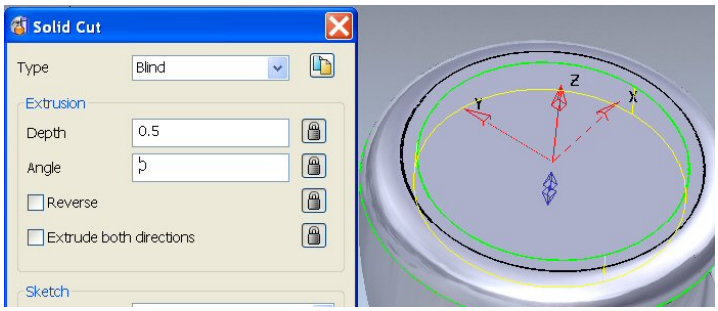
To check this you can view the model in X and Y individually to see.



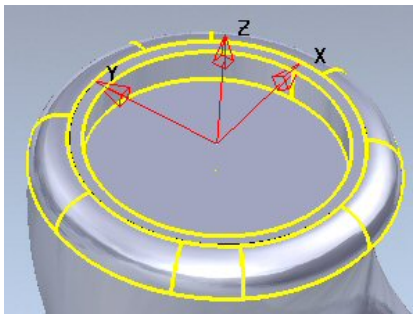
- View from left (X-).

- Select the Diameter 4mm circle on the top face created earlier.
- Change the Diameter to **3.8mm** (Radius 1.9mm).
- Select the **Feature** button on the top toolbar.

- Select the **Solid Cut** button on the left toolbar.



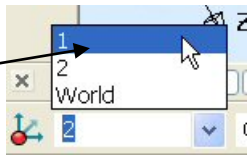
- Type in **0.5** in the Depth box.
- Select **OK**.



The cavity should look like this.

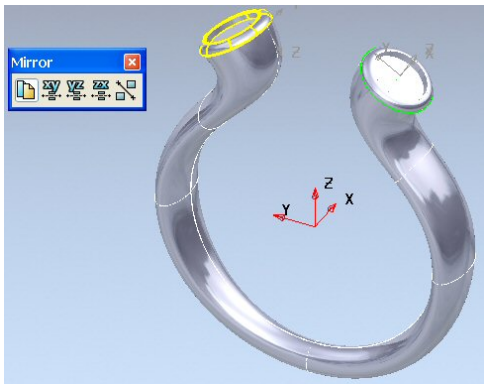
We will copy this design to the opposite side.

- At the bottom left corner of the screen select **1** from the Workplane list.



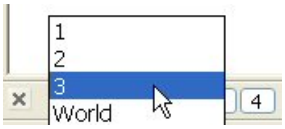
- Select the **Show general edits options** button on the right toolbar.

- With the top design still selected, click on the **Mirror/symmetrise object** button on the left toolbar.



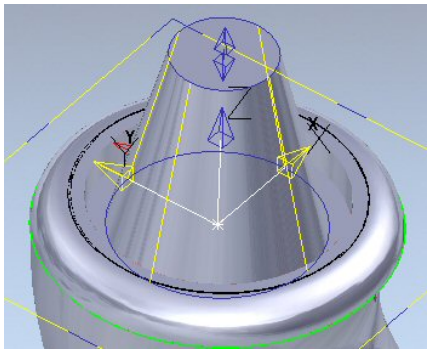
- Click the **ZX** button to create a new design on the opposite side.

- At the bottom left corner of the screen select **3** from the Workplane list.

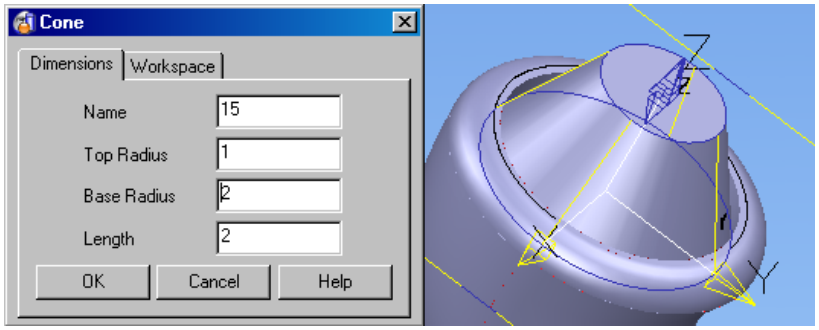


- Click on the **Solid** button on the top toolbar.

- Select the **Create solid cone** from the pull out menu on the left toolbar.



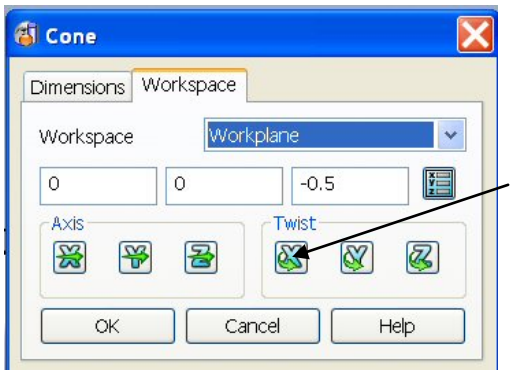
- To position the Solid Cone, type **0 0 -0.5** in the command window then press return.



- Double click the cone to open the Cone form.

- Make the Top Radius 1, Base Radius 2 and Length 2.

- Click the **Workspace** tab.

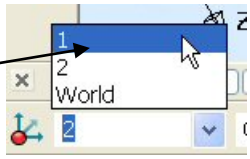


This form allows you to position and rotate the cone when needed.

- Click on the **Twist X** button here.

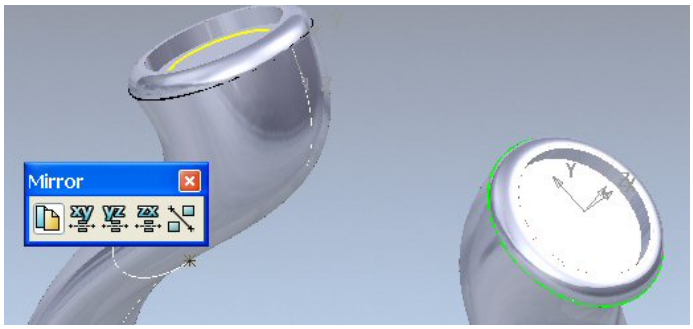
The calculator will appear allowing you to type in the required angle.

- In this instance we need to **type in 180** then click **OK**.
- Select **OK** on the Cone form to close it.
- At the bottom left corner of the screen select **1** from the Workplane list.



- Select the **Show general edits options** button on the right toolbar.

- With the cone still selected, click on the **Mirror/symmetrise object** button on the left toolbar.



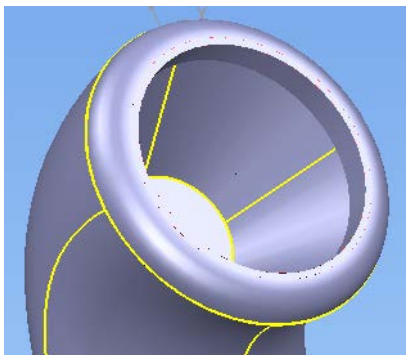
- Click on the **ZX** button to create a new copy of the cone on the opposite side.

To create the cavities we need to remove the two Solid Cones from the main Shank solid.

First we need to make the main Shank solid Active.

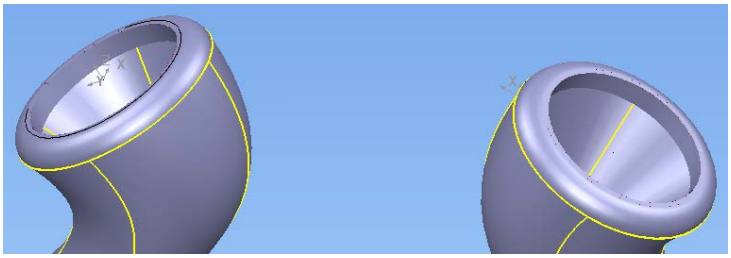
- Right click over the main shank solid and select **Active** from the list.
- Select either one of the two cones.
- Select the **Feature** button on the top toolbar.

- Select the Remove the selected solid, surface or symbol from the active solid button on the left toolbar.

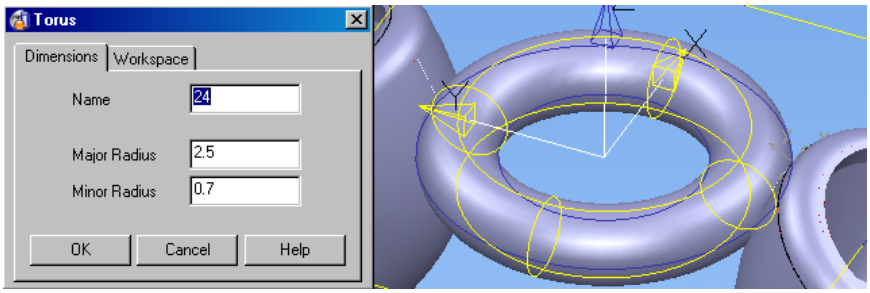


The cone is removed to leave the cavity.

- Repeat the procedure to complete the opposite side.

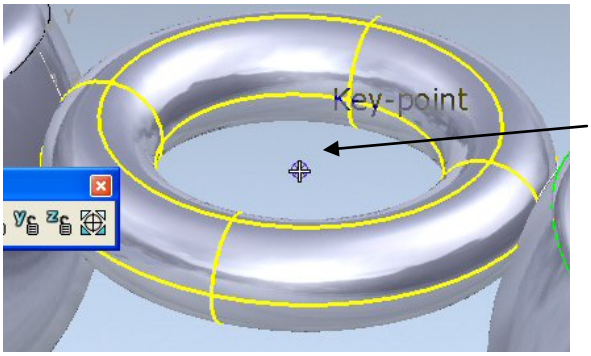


- Click on the **Solid** button on the top toolbar.
- Select the **Create solid torus** button from the left toolbar.
- In the command window at the bottom of the screen, type **0 0 13** to position the torus.
- Double click the torus to open the torus form.



- Fill in the values shown then click on **OK**.

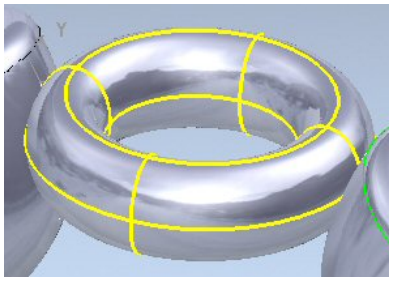
- Select the **Show general edits options** button on the right toolbar.
- Select the **Scale the object(s)** button on the left toolbar.
- Click on the **reposition scale origin** button on the Scale form.



- Click on the **Key-Point** at the centre of the Torus.

- Click on the **Lock X** and **Lock Y** buttons.

- Type **1.4** in the scale box then press return on the keyboard.



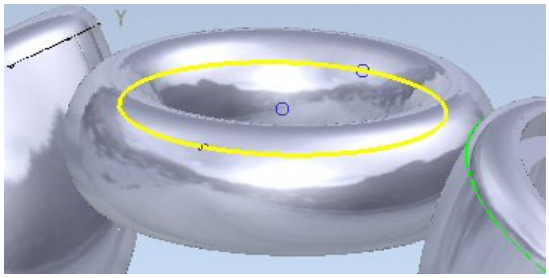
By locking the X and Y we have only scaled the Torus in Z leaving the major diameter as it was.

- Right click over the solid torus and make it **Active.**

- Select the **Arc** button on the top toolbar.

- Select the **Create a full arc (circle)** button on the left toolbar.

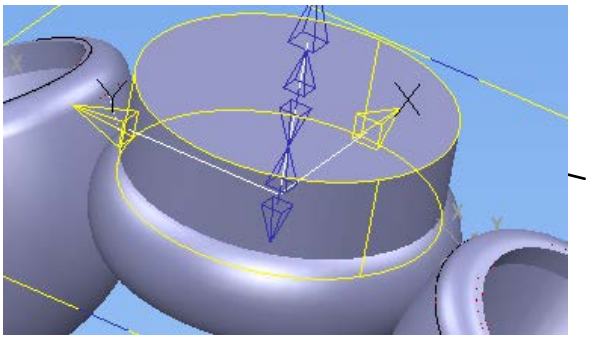
- In the command window at the bottom of the screen, type **r 2.4** and then press return to size the circle.



- In the command window at the bottom of the screen, type **0 0 13.5** then press return to position the circle
-

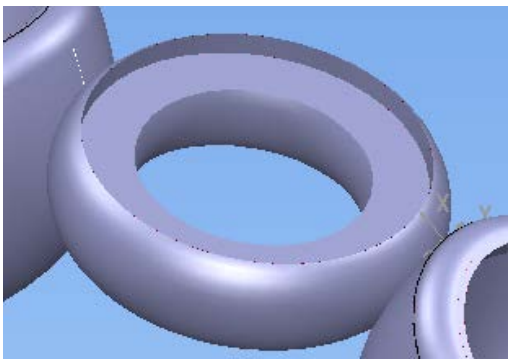
- Click on the **Solid** button on the top toolbar.

- Click on the **Create one or more solid extrusions** button on the left toolbar.



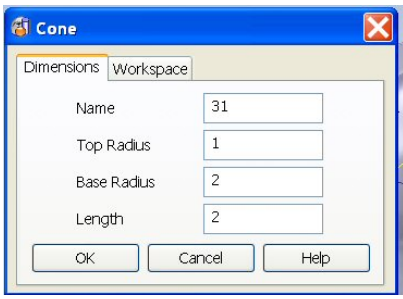
- Adjust the extrusion length so that it clears the top of the torus similar to that seen here.

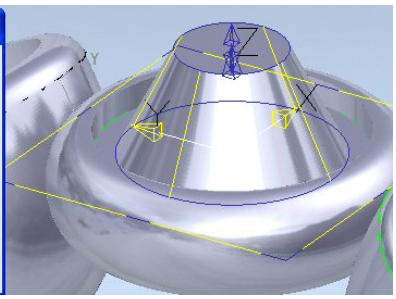
- Select the **Feature** button on the top toolbar.
- Select the Remove the selected solid, surface or symbol from the active solid button on the left toolbar.



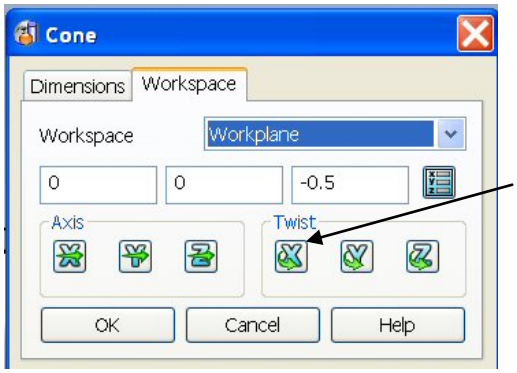
The extrusion is removed to give the seat for the setting.

- Click on the **Solid** button on the top toolbar.
- Select the **Create solid cone** from the pull out menu on the left toolbar.
- In the command window at the bottom of the screen, type **0 0 13.5** to position the cone.





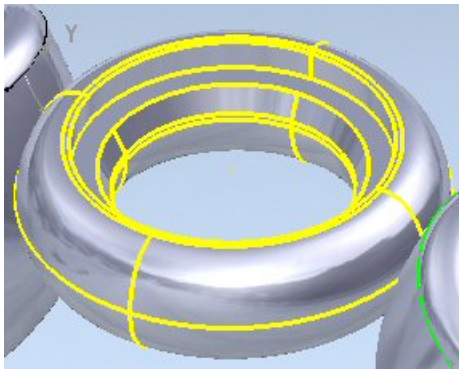
- Make the Top Radius 1, the Base Radius 2 and Length 2.
- Click the **Workspace** tab.



This form allows you to position and rotate the cone when needed.

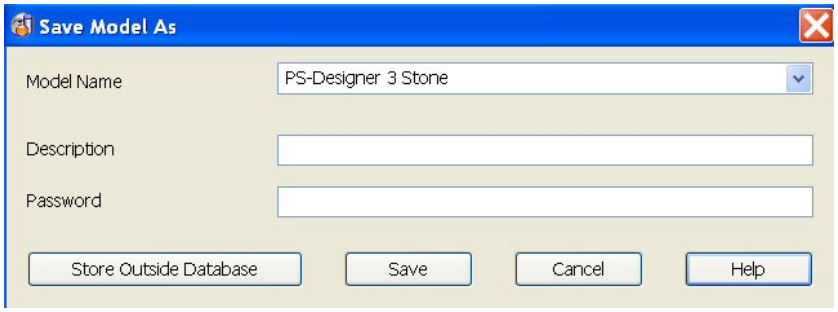
- Click on the **Twist X** button here.

- The calculator will appear allowing you to type in the required angle.
- In this instance we need to **type in 180** then click **OK**.
- Select the **Feature** button on the top toolbar.
- Select the Remove the selected solid, surface or symbol from the active solid button on the left toolbar.



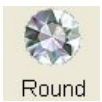
The cone is removed to give more clearance for the gem.

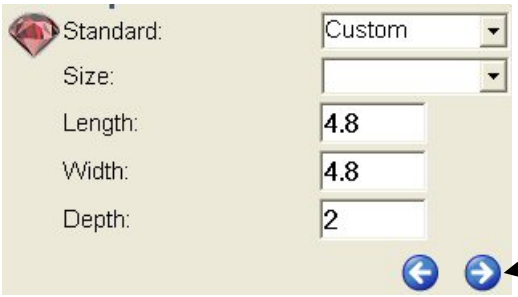
- Click on the **Save selected model** button on the top toolbar.



- Search Location **D:\users\Training\Coursework\Delcam Designer Jobs**
- Type in a name as **3 stone ring.psmodel** then **Save**

To finish the design we need to add some diamonds. One for the centre setting, and two for the outer settings.

- In an ArtCAM JewelSmith Project select the Add Gem button in the Assembly Manager area.
- Select a **Round** gem. 



- Fill in the form with the values seen here.
- Click the Forward arrow.

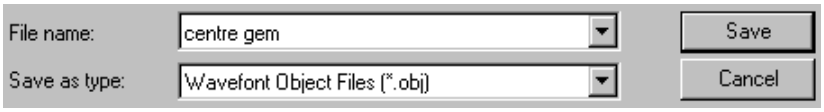
- Choose a **Diamond** colour.

- Type in the name **Centre Gem**, then click the forward button.

- The centre gem is created and ready to be exported for use in Delcam Designer.
- Repeat the above procedure to create one more **diamond** with **Custom** sizes of **Length 4**, **Width 4** and **Depth 1.7**.
- **Name** this diamond **Side Gem**.

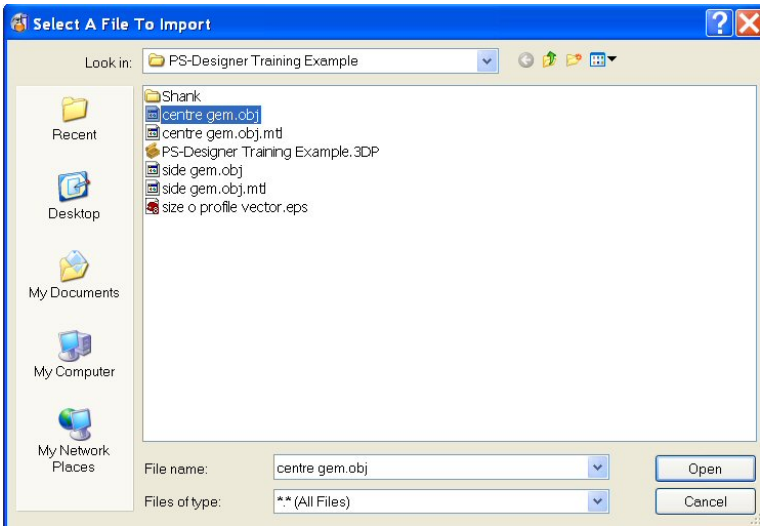
In the JewelSmith Assembly tree you should see the two created gems.

- Click on the **Centre Gem** name.
- Right mouse click and select **Export Gem.**
- In File name type **centre gem** and Save as type select **Wavefront Object Files** (*.obj).

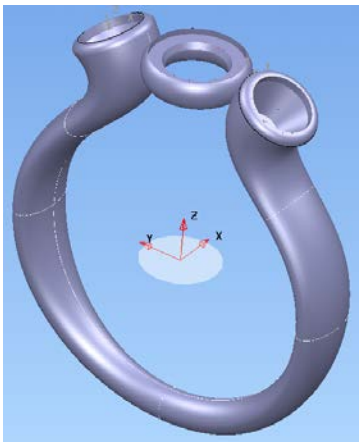


- Save in Location D:\users\Training\Coursework\Delcam Designer Jobs
- Click on **Save**.

- Repeat this procedure for the **Side Gem** giving the name **side gem** in the **File name** area.
- Return to the **Delcam Designer** window
- From the top left corner select **File > Import**.



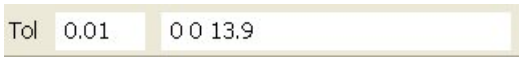
- Browse to the location of the gems.
- Select **centre gem.obj** from the list
- Select **Open**.

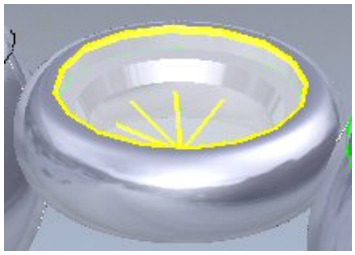


- The gem gets imported and angled to the current active Workplane.
- Select the gem with the left mouse button.
- Select the **Show general edits options** button on the right toolbar.

- Select the **Move/Copy Object** button from the left hand toolbar.

- In the command window at the bottom of the screen type in **0 0 13.9** then press the return key.

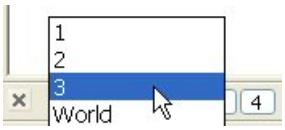




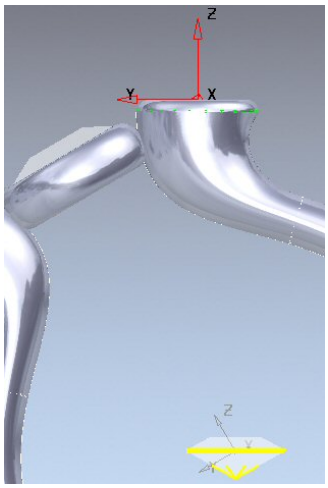
The gem is positioned correctly.

You can change its position to suit yourself by adjusting the 13.9 Z value if required.

- At the bottom left corner of the screen select **3** from the Workplane list.



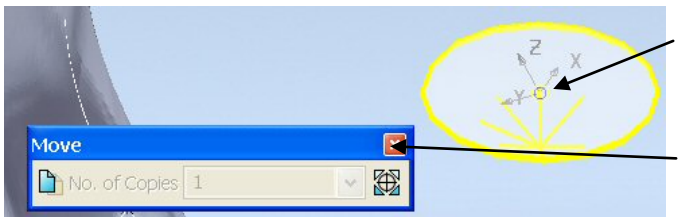
- From the top left corner select **File > Import**.
- Select **side gem.obj** then **Open**.



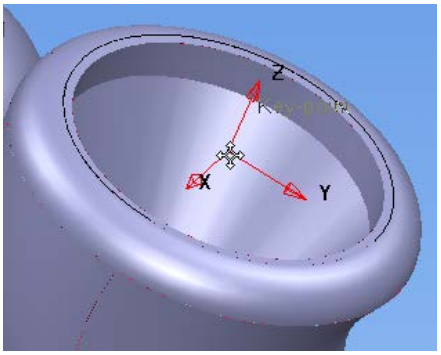
The side gem gets imported and angled relative to the new active Workplane.

- Select the gem with the left mouse button.
- Select the **Show general edits options** button on the right toolbar.

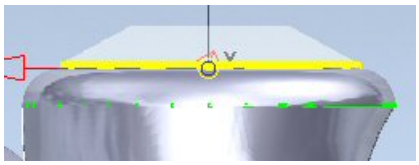
- Select the **Move/Copy Object** button from the left hand toolbar.



- The move origin of the gem should be here at the gem centre.
- If not, use the reposition move origin button and click at this point.

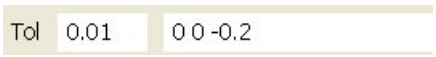


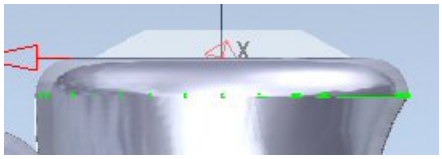
- Position the cursor at the active Workplane Key-Point and click the left mouse button.



The gem appears to be slightly too high.

- In the command window type **0 0 -0.2** then hit return.



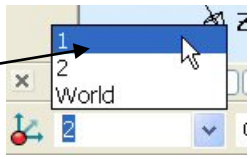


- The side gem is adjusted correctly.

- **Close** the Move form.

- We will now make a copy of the side gem to the opposite side.

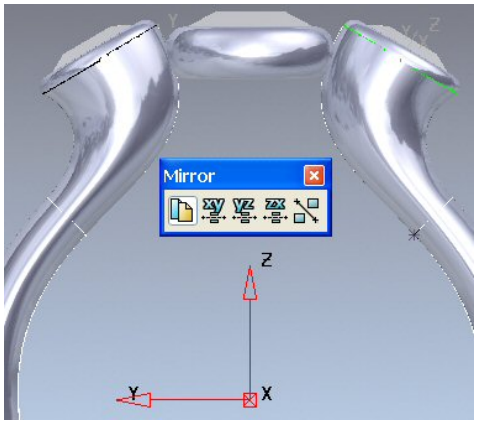
- At the bottom left corner of the screen select **1** from the Workplane list.



- Select the side gem.

- Select the **Show general edits options** button on the right toolbar.

- With the cone still selected, click on the **Mirror/symmetrise object** button on the left toolbar.



- Click on the **ZX** button to create a new copy on the opposite side as seen here.

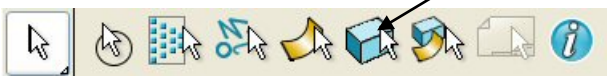
The design is almost complete.

- Select the **Save** button on the top toolbar.

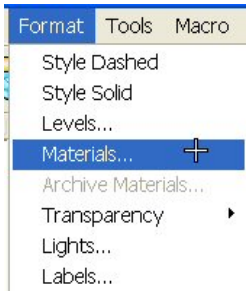
We will now create a rendered image that can be used to email to prospective customers.

First we need to change the material properties of the shank and settings to **18ct Gold**.

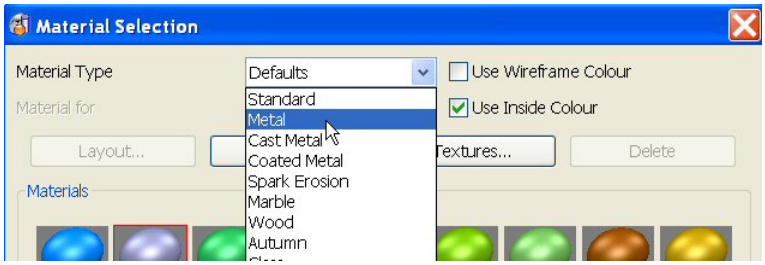
- From the left toolbar click on the **Quick select all solids** button.



- This should highlight all the solids in the current model.



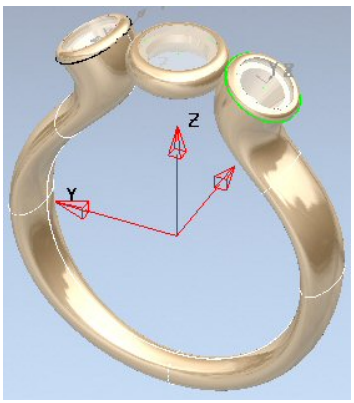
- At the top of the screen select **Format** then **Materials**.



- Select **Metal** from the pull down list.

- Select **Gold 18ct** button.

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



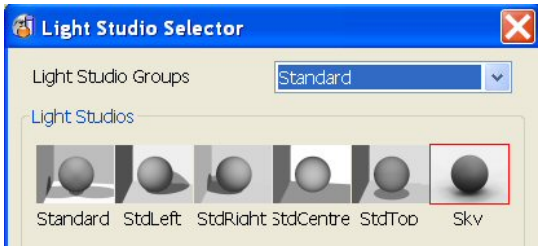
The solids have changed colour and now we are ready to create a rendered image.

- Select the **Restore down** button on the top right hand corner of the screen.

- Adjust the model window smaller to allow faster rendering.
- Also rotate the design into the desired position, similar to that seen here.

- Select the **Raise the render form** button from the right hand toolbar.

- In the **Lights** area of the form select the **Change** button.



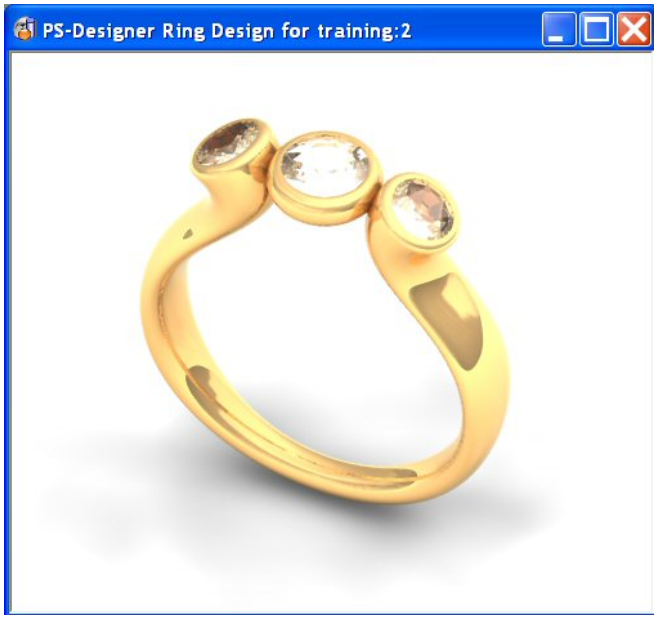
- From the pull down list select **Standard.**
- From the **Light Studios** area click on the **Sky** button.

- Click on OK to accept the form.
- Click the **Render** button on the Render

form.

There may be a slight delay before rendering starts.

Eventually you will be left with an image such as the one seen on the next page.



- If you are happy with the result select **OK** on the Render form.
- To save the image select **File** at the top left of the screen then **Print to File**.
- Search Location **D:\users\Training\Coursework\Delcam Designer Jobs**
- Type in a name as **Rendered 3 stone ring.jpg** then Save.

- Select the **Save** button on the top toolbar and name model as **3 stone ring.psmodel** in the same location

7. Tapered Ring

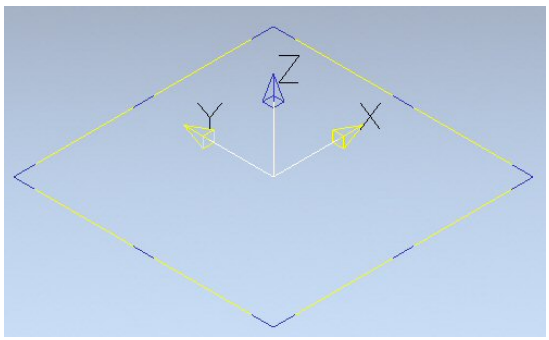
Tapered Ring Design

The object of this exercise is to design the Ring seen here.

We will be creating the design using the **Delcam Designer** functionality.

- Create a New Model
- Select the **Workplane** icon on the top toolbar.
- Type **0** in the command window at the bottom of the screen.

- Press **return** on the keyboard.



This new **Workplane** will become the main datum for the ring design.

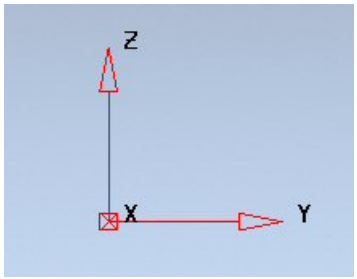
- Click away from the Workplane and you will see it has turned red.

This indicates that it is the **Active** Workplane and hence **X 0, Y0** and **Z 0**.

More Workplanes can be created in different areas and orientations on a model to make it easier to create objects such as wireframe around it.
Each time a new Workplane is created it becomes Active and also the new X 0, Y0 and Z0 position.

Workplanes can be positioned by using the mouse onto keypoints, intersections etc. or by typed position input in the command window.

- Select the **View from right (+ x)** button from the right hand toolbar.

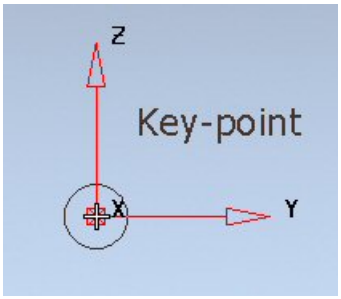


The Workplane will reflect that we are now working in the ZY plane.

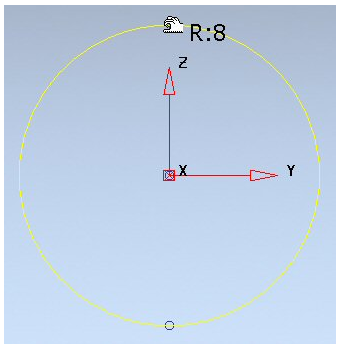
The reason for this is so that the ring is designed in the correct orientation ready for copy/pasting directly into ArtCAM JewelSmith for machining later.

- Select the **Arc** button on the top toolbar.

- Select the **Create a full arc (circle)** button from the left toolbar.



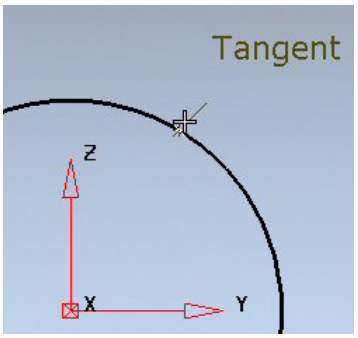
- Position the circle at the **Workplane key-point**.
- Click the left mouse button to place the circle.



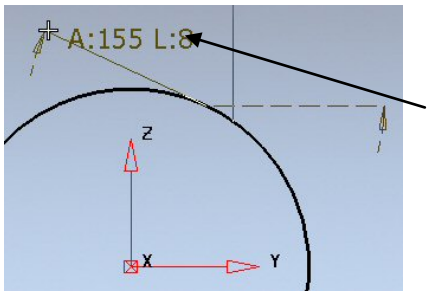
- Modify the circle radius until it has a Radius of 8mm.

- From the top toolbar select the **Line** button.

- Select the **Single lines** button from the left toolbar.



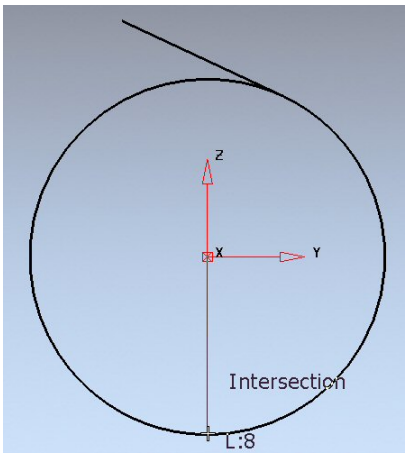
- Position the line on the cursor at approximately this point until the word **"Tangent"** appears.
- Click the left mouse button to place the start of the line.



- Move the mouse to the left until the intelligent cursor indicates an **Angle (A)** of **155** and **Length (L) 8** as seen here.
- Click the left mouse button to finish the line.

- From the top toolbar select the **Line** button.

- Select the **Single lines** button from the left toolbar.



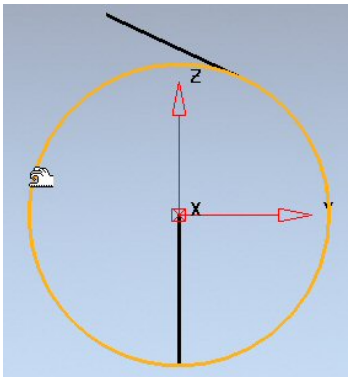
- Click firstly on the Workplane key-point then secondly on the intersection at the bottom of the circle.

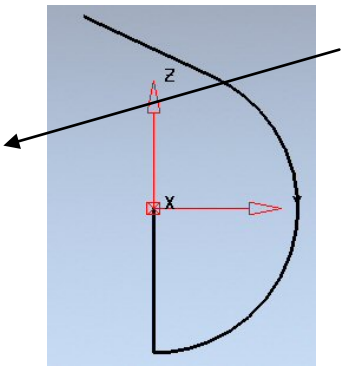
We will use this line to enable us to trim the circle on the left side of the Workplane to the tangential line at the top to the intersection on the vertical line with the circle at the bottom.

- Click on the **Show general edits options** button o the left toolbar.

- Select the Interactively limit wireframes button.

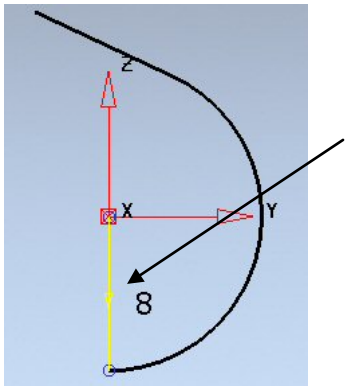
- We now need to click on the part of the circle we do not want.



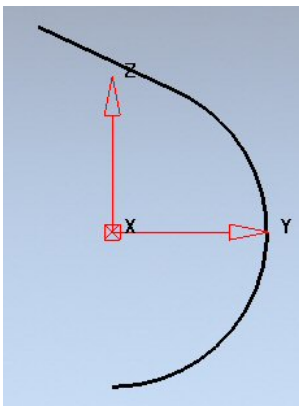


- Click on this part of the circle to remove it.

Press to exit the Interactive limit of wireframe.

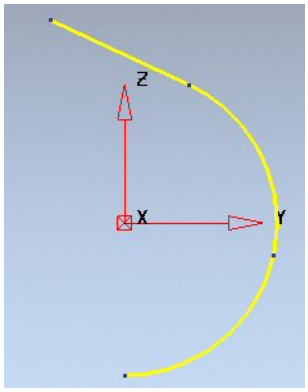


- Select the vertical line we created earlier.
- Press **Delete** on the keyboard to delete it. It is no longer required.



We are now left with the circle and line which form the first part of the ring design.

Because they are two separate items we need to connect them together as one. Once done this will be a composite curve.

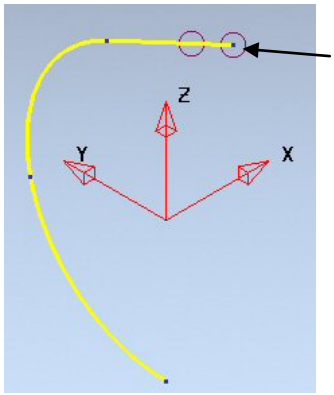


- Holding down the Alt key on the keyboard, click on either the line or circle.

The two curves are now joined together to create one composite curve.

The curve is made up of node points. These points will be moved to create a twisted effect for the ring design.

- Select the **View ISO 1** button on the right toolbar.

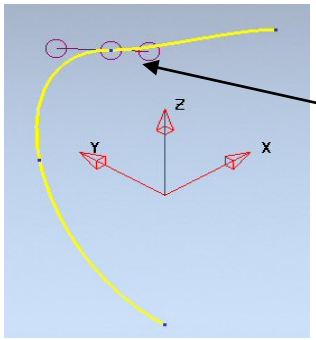


- Select the top point with the left mouse button.

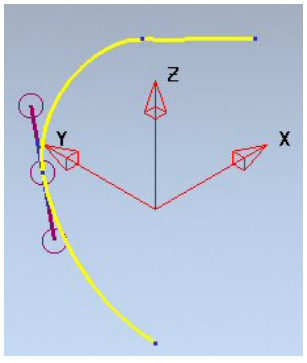
To move the point by a given amount, just type in the distance in the command window at the bottom of the screen.

In this case we want to move the point by 3mm in X+.

- Type **3** in the command window then press return on the keyboard.

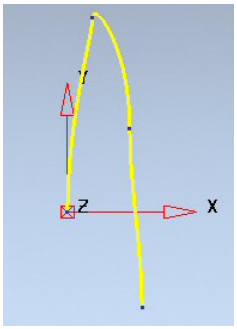


- Select the second point and this time type **2.5** in the command window then press return on the keyboard to move it 2.5mm in X+.



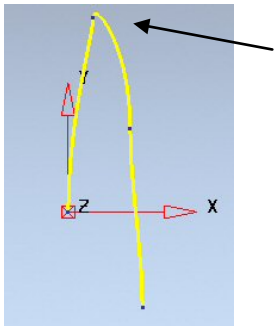
- Select the third point and move it 1mm in X+.

- Select the **View from Top (Z+)** to see the effects.



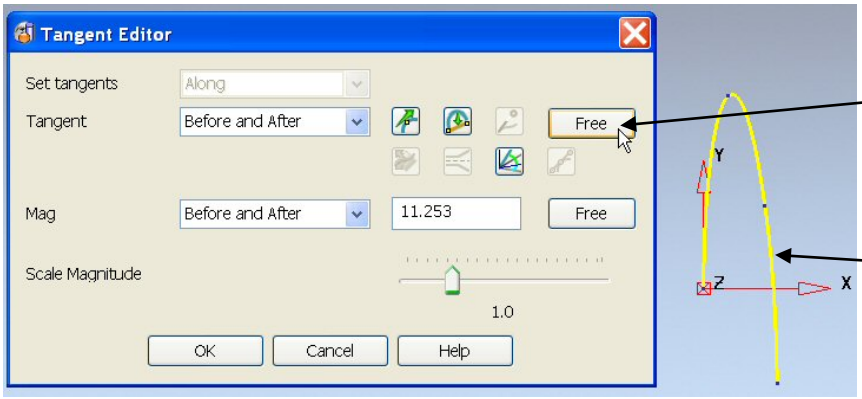
You will notice that the curve is not smooth running through the points.

We can improve this by freeing the tangencies through all points along the curve.



- Make sure that the curve is selected. Click between the curve points so that only the curve and not an individual point is picked.

- Select the **Edit Tangent Angles** button on the curves toolbar at the top of the screen.



- Click on the **Free** button.

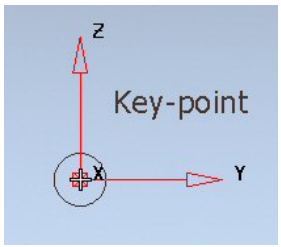
See the curve smoothed.

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

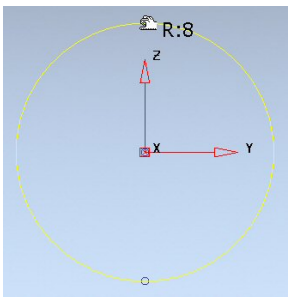
- Select the **View from right (+ x)** button from the right hand toolbar.

- Select the **Arc** button on the top toolbar.

- Select the **Create a full arc (circle)** button from the left toolbar.

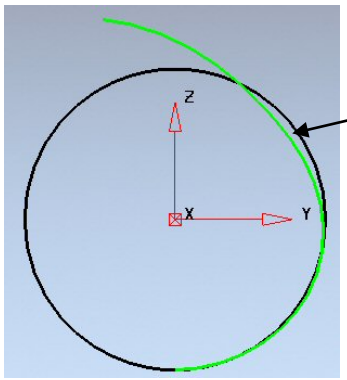


- Position the circle at the **Workplane key-point**.
- Click the left mouse button to place the circle.

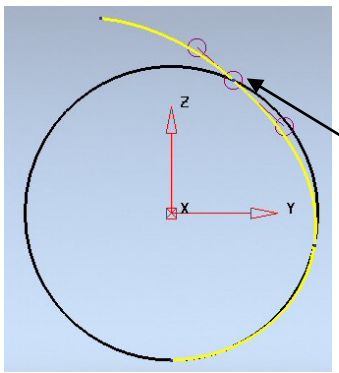


- Modify the circle radius until it has a Radius of **8mm**.

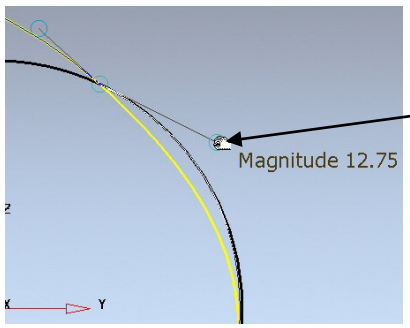
This will be used as a guide to see if the curve is correct “down the finger”.



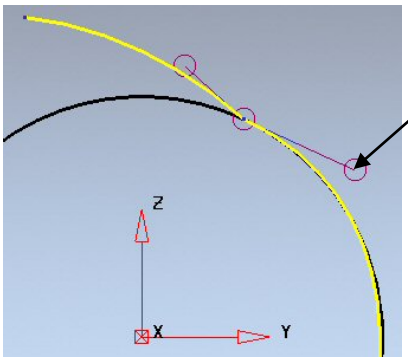
- Viewing the ring design down the x (finger) we can see that the curve needs re-adjusting in places to give a more circular shape.



- Select this point on the composite curve.

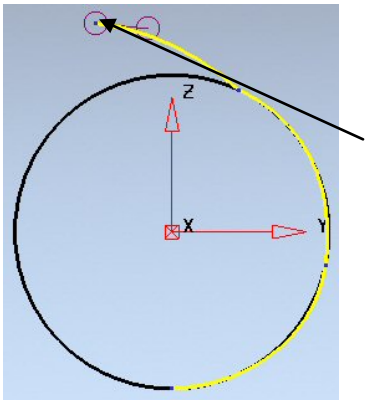


- Grab this handle with the left mouse button then drag and modify the curve until it matches the 8mm radius circle then let go.



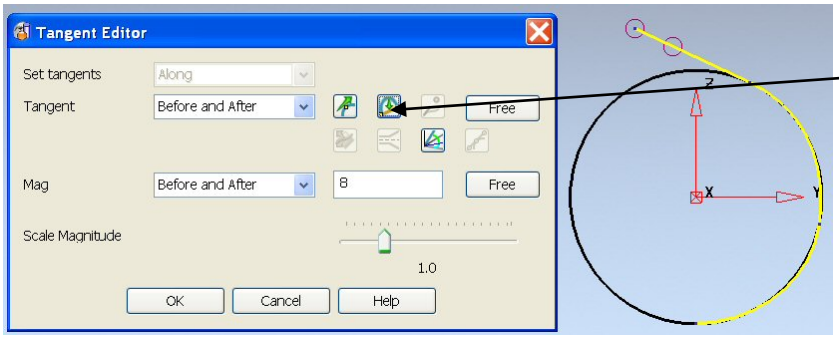
- Adjust the curve until you have a similar result to this.

We will now straighten the curve between the top two points using the Edit Tangent Angles form.



- Select the top point on the curve.

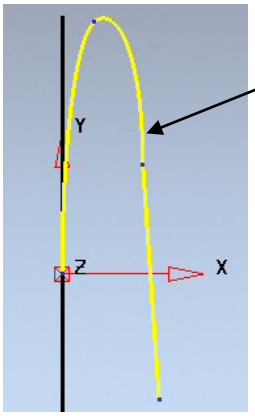
- Select the **Edit Tangent Angles** button on the curves toolbar at the top of the screen.



- Select the **Straighten** button on the form to give the result shown.

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

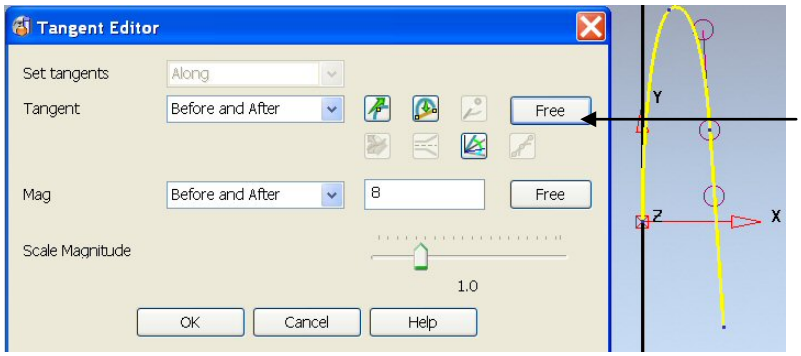
- Select the View from Top (Z+).



You can see that the curve is not smooth going into this point.

We can free the tangencies once again through this point but first we have to select it.

- Click on the point to select it then on the **Edit Tangent Angles** button.



- Click on the **Free** button and the curve is smoothed.

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

N.B. It is advisable when modifying curves in this way to continually change the view to see how the curve looks and if required free the tangencies where necessary.

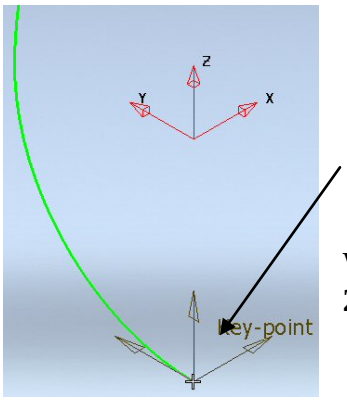
The main “drive curve” for the shank is now complete. We will now create the cross sections needed to produce the square tapered design.

- **Delete** the **8mm radius circle** used as a guide as we no longer need it.

- Select the **View ISO 1** button on the right toolbar.

- Select the **Use the XY face of the Workplane** button at the bottom left of the screen.

- Select the **Workplane** icon on the top toolbar.

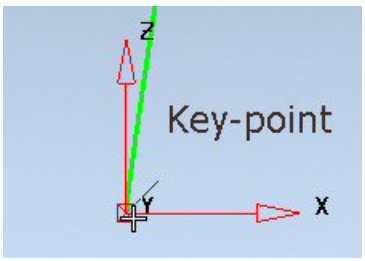


- Position the new Workplane at the **key-point** at the bottom of the curve.

We are going to create a rectangle around this Workplane **2mm x 1.5mm**.

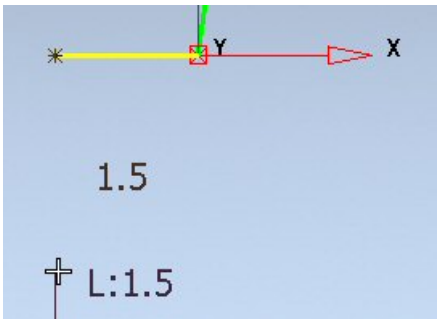
- Select the **View from front (Y-)** button.

- From the top toolbar select the **Line** button.
- Select the **Continuous lines** button from the left toolbar.



- Click on the Workplane key-point to position the start of the continuous lines.

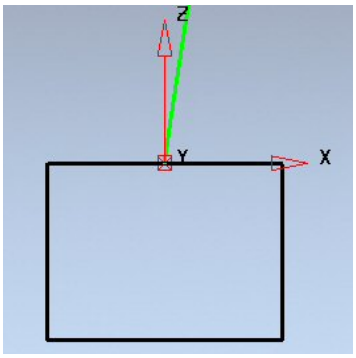
- Move the cursor to the left of the Workplane until the value of **L:1** appears then click the left mouse button.



- Move the cursor down until you achieve a length of **1.5mm** then click the left mouse button.

- Move the cursor to the right X+ to a length of 2mm then up to a length of 1.5mm and finally back to the Workplane origin.

- Press to finish the line creation.



The final rectangle should look like this.

To remove the sharp edges we will fillet the corners with a 0.3mm Radius.

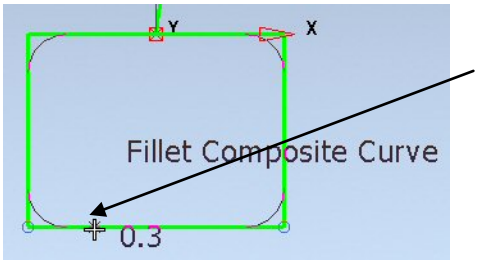
- Holding down the **Alt** key, click on any of the rectangle lines. This should create a composite curve.

- Select the **Arc** button on the top toolbar.

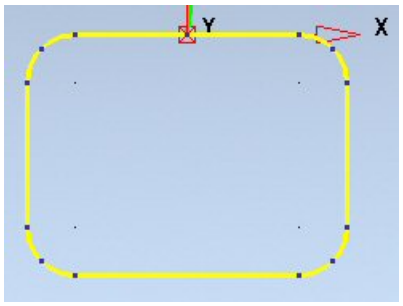
- Click on the **Create a fillet arc** button on the left toolbar.

- Type **r 0.3** in the command window then press return on the keyboard.

This sets the arc radius before applying.



- Hold the mouse cursor over the composite curve then click the left mouse button to apply the fillet on all corners.



All corners are filleted 0.3mm.

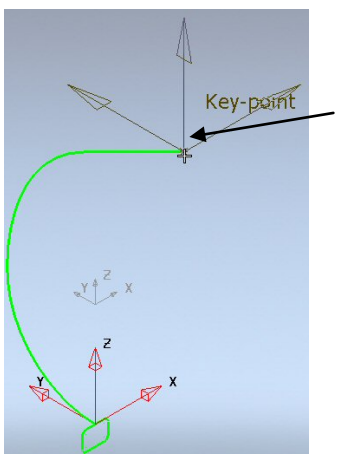
This cross section is complete.

- Press to end the filleting function.

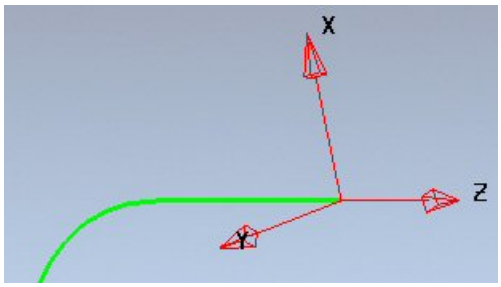
We will now create a 4mm square cross section at the other end of the shank profile curve.

As this end is angled we will create a Workplane that is automatically aligned to the curve when positioned there.

- Select the **View ISO 1** button on the right toolbar.
- Select the **Use the XY face of the Workplane** button at the bottom left of the screen.
- Select the **Workplane** icon on the top toolbar.
- Select the **Single Workplane aligned to geometry** button on the left toolbar.

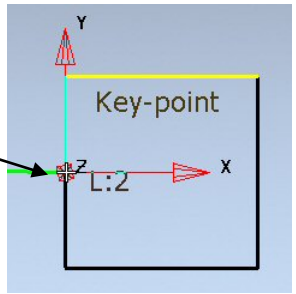


- Position the Workplane at the remaining end of the shank profile curve.

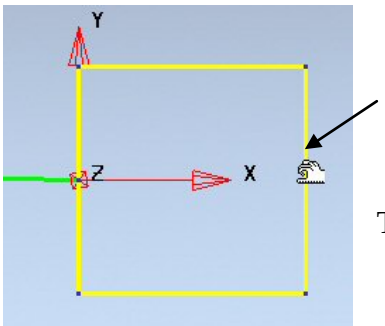


The Z of the Workplane is automatically aligned with the geometry making it easier to design around it and angled correctly to the curve.

- Select the View from Top (Z+).
- From the top toolbar select the **Line** button.
- Select the **Continuous lines** button from the left toolbar.
- Type **0** in the command window, then press return.
- Type **0 -2** then return.
- Type **4** then return.
- Type **0 4** then return.
- Type **-4** then return.
- Click on the Workplane key-point to finish the 4mm square.



- Press to end the Line creation mode.

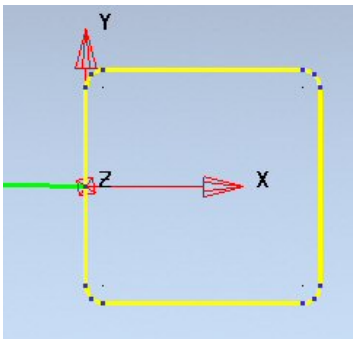


- Holding down the **Alt** key, click on any of the lines.

This should create a composite curve.

- Select the **Arc** button on the top toolbar.
- Click on the **Create a fillet arc** button on the left toolbar.

- Type **r 0.3** in the command window then press return on the keyboard.
- This sets the arc radius before applying.

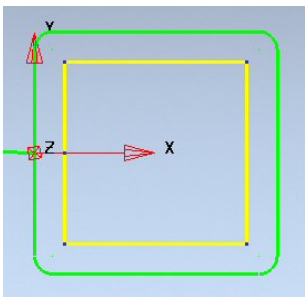


- Click on the composite curve to fillet all corners with a 0.3mm radius.
- Press to end the filleting mode.

We will now create further vectors inside this one for creating the setting area of the stone.

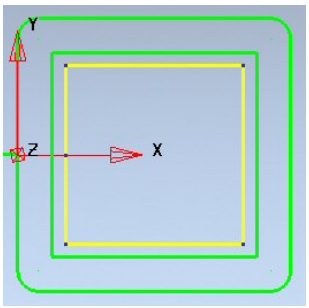
- With the 4mm square curve selected click on the **Show general edits options** button o the left toolbar.
- Click on the **Offset object** button on the left toolbar.

- Enter **0.5** in the Distance box and make sure the **brown arrow** points inwards as is here. Press **Return**.
- **If not** you can enter a value of – 0.5 in the Distance box then press return on the keyboard.



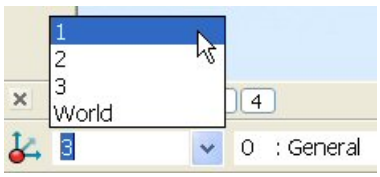
We now have a 3mm square curve. This will represent the Princess Cut stone size and will be used to cut the seat for the setting area.

- **Close** the Offset form.
- With the **3mm** square curve selected click on the **Offset object** button on the left toolbar.
- Enter a value of **0.2** in the Distance box then press return on the keyboard.



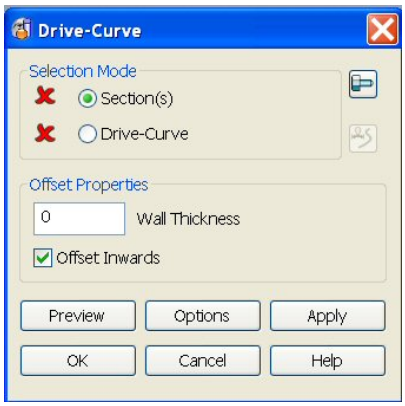
This curve will be used to create the clearance for the depth of the Princess cut stone.

All curves for the design are now complete.



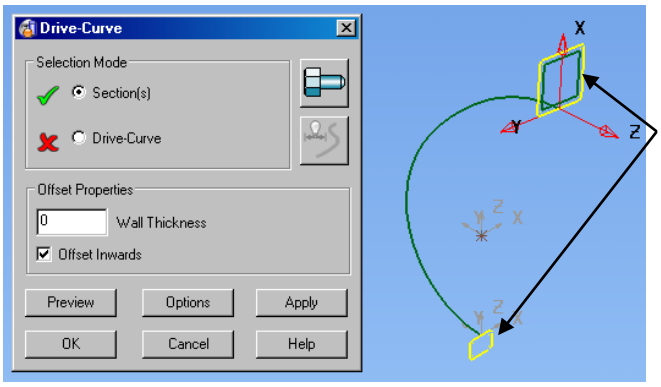
- From the list of Workplanes in the bottom left corner select **1** to activate it.

- Select the **View ISO 1** button on the right toolbar.
- Select the **Solid** button on the top toolbar.
- Select the Create a solid from drive curve and sections button on the left toolbar.



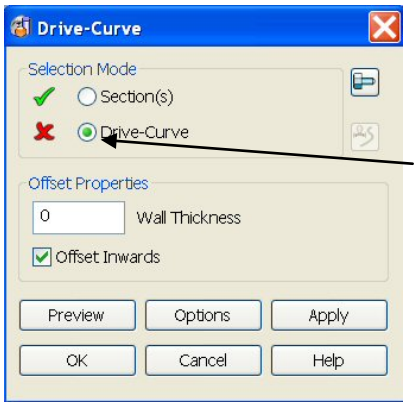
The red crosses next to Section(s) and Drive-Curve indicate that nothing is selected.
When items are selected a green cross appears instead of the Red Cross.

We need first to select the two cross sections created earlier at each end of the shank drive curve.



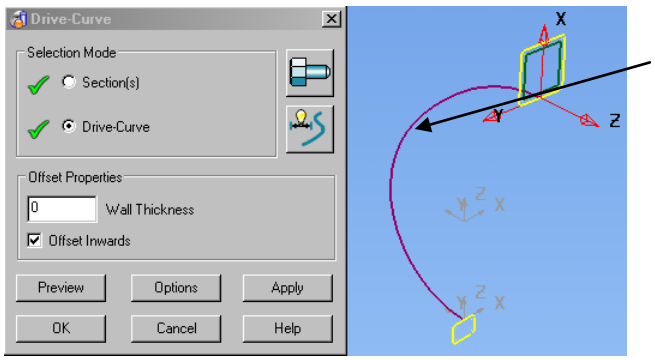
- Holding down the Shift key, select the two cross sections seen here.

Once selected you will notice the green tick appear next to Section(s).

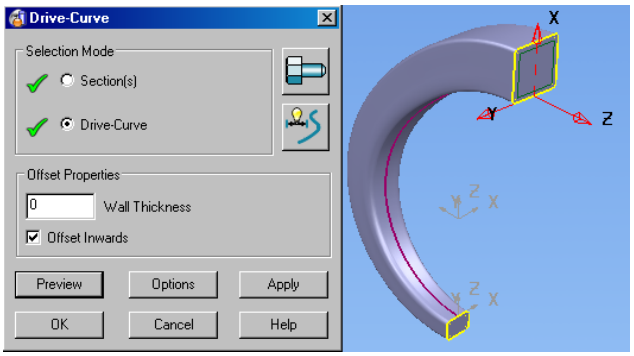


- With the left mouse button, click this button here to select it.

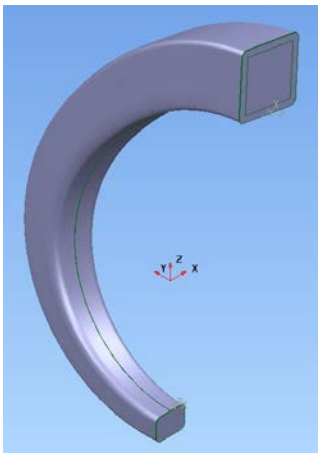
We will now select the ring shank drive curve.



- Click on the shank drive curve here and a green tick will appear indicating the selection.



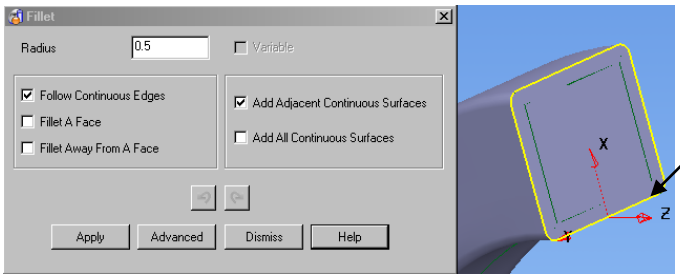
- Select the **Preview** button to see the results.
- Select **OK** to accept and close the form.



This end of the shank will be modified further to include a fillet radius and solid cut detail.

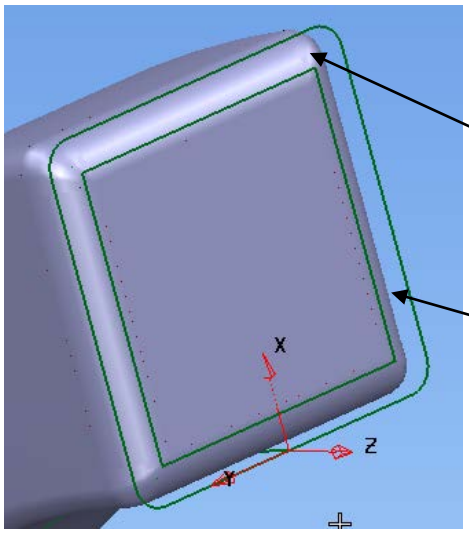
- Select the **Feature** button from the top toolbar.

- Click on the **Create solid fillet** button on the left toolbar.



- Enter **0.5** in the Radius box then select this edge.

- Select **Apply** and **Dismiss** the form.

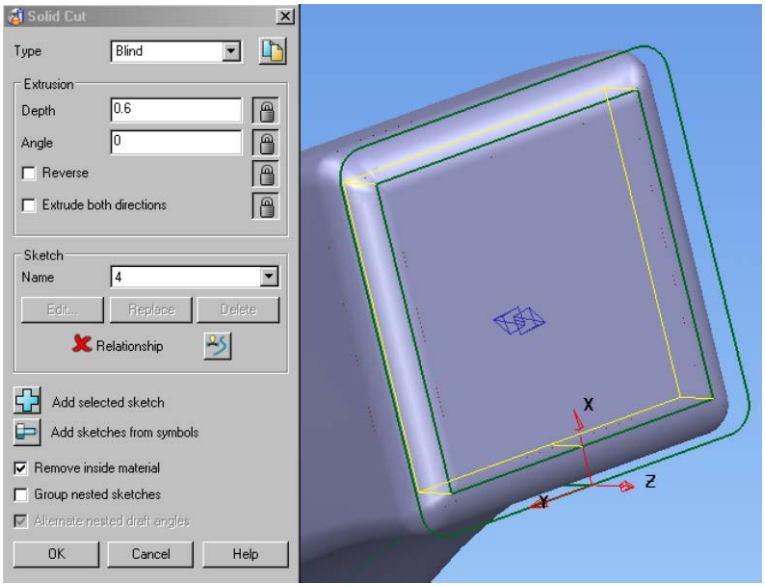


The fillet is applied around the top edge of the design.

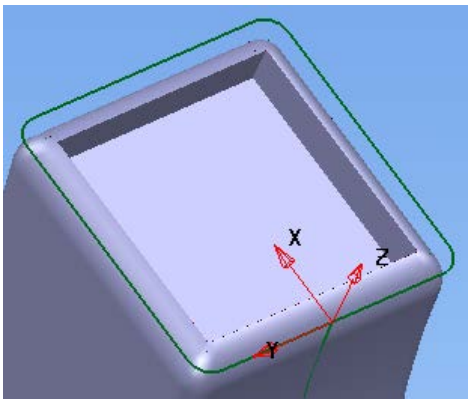
We will now create a cut at **0.6mm** deep using the 3mm square curve.

- Select this curve.

- Select the **Create solid cut feature** button on the left toolbar.

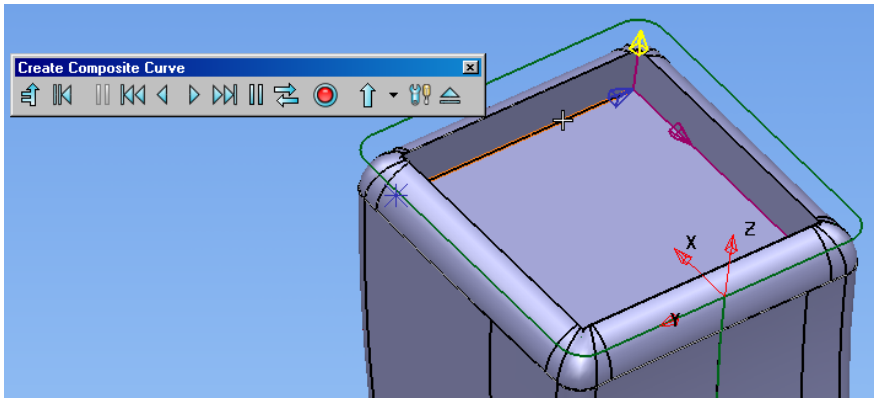


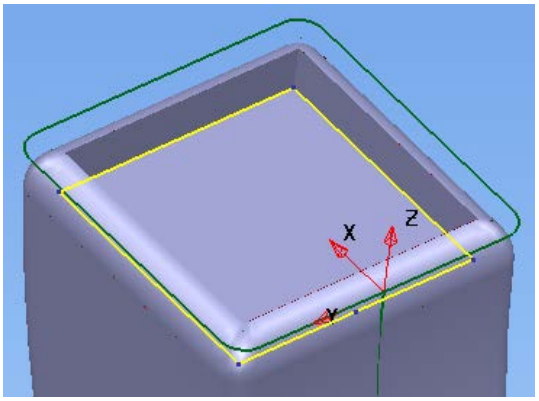
- Enter a value of **0.6** in the **Depth** box.
- Select **OK** to create the cut feature and close the form.



The pocket is cut to a blind depth of 0.6.

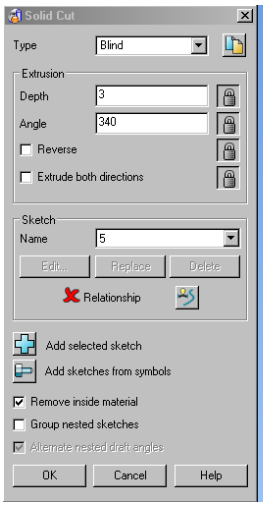
- Trace a new composite curve at the bottom of the pocket edge as shown

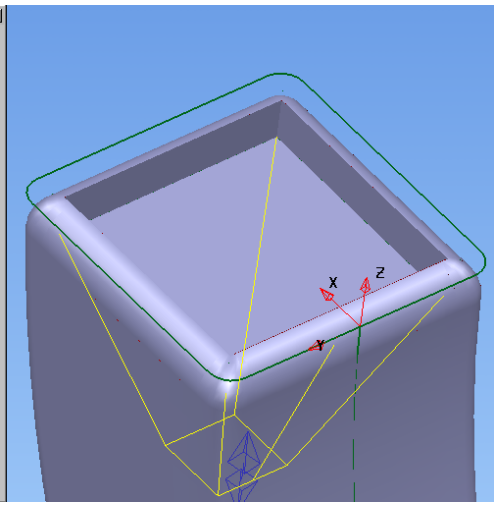




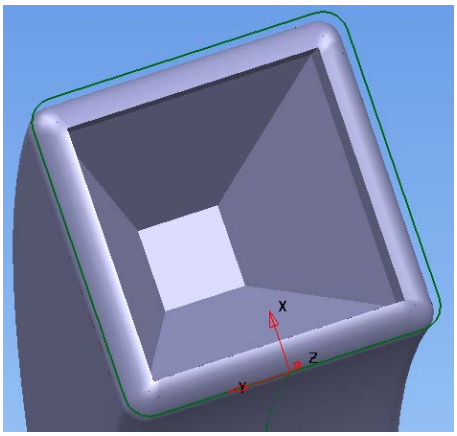
- Select this new composite curve and then

- **Create solid cut feature** button from the Solid feature toolbar.





- Enter a **Depth** of **3** and an **Angle** 0f **340** (20 deg per side).
- Select **OK** to accept and close the form.

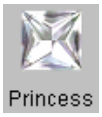


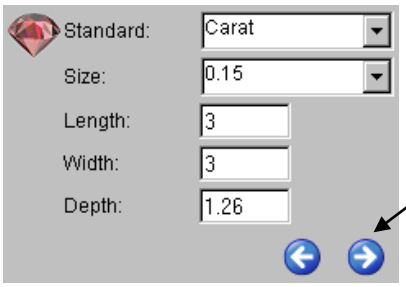
The end design is now complete.

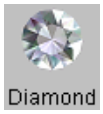
If you wish you can **undo** any changes and apply your own values.

We are now ready to bring in the Princess Cut gem.

- Open **ArtCAM JewelSmith**.
- Create a **New Project**.

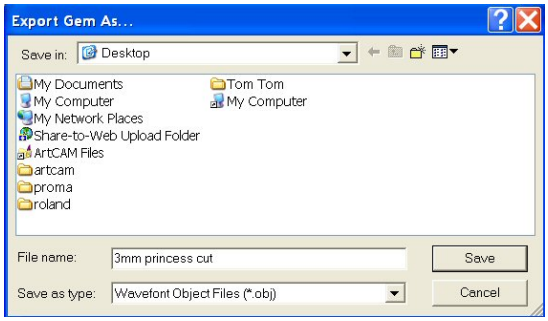
- Select the **Add Gem** button from the **Assembly Manager** area.
- Select the **Princess Cut** button. 



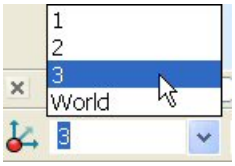
- Select a **Carat** size of **0.15**.
- Click the forward arrow to continue.
- Select the **Diamond** colour button. 

- Select the **forward** arrow to create the gem.

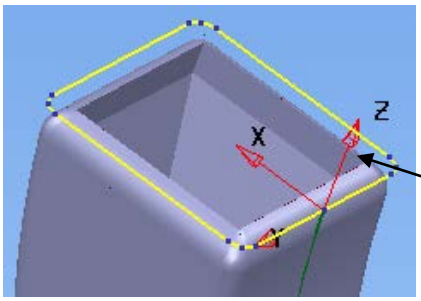
- Click on the gem name in the project tree.
- Right mouse click and select **Export Gem.**
- In File name type centre gem and Save as type select **Wavefront Object Files** (*.obj).



- Save in Location **D:\users\Training\Coursework\Delcam Designer Jobs**
- Enter a name **3mm princess cut**.
- Change the Save as type to **Wavefront Object Files (*.obj).**
- Select **Save**.
- Return to the **Delcam Designer** window.

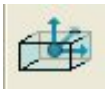


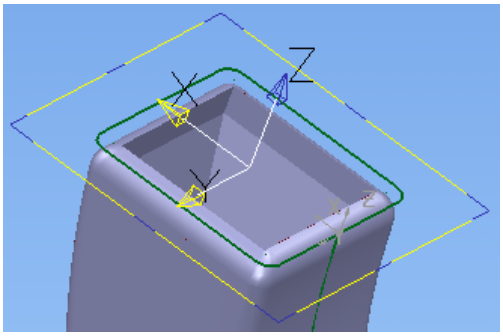
- From the bottom left corner select **Workplane 3** from the list.



To make positioning of the Princess cut gem easier we will create a new Workplane in the centre of this curve.

- Select the curve.

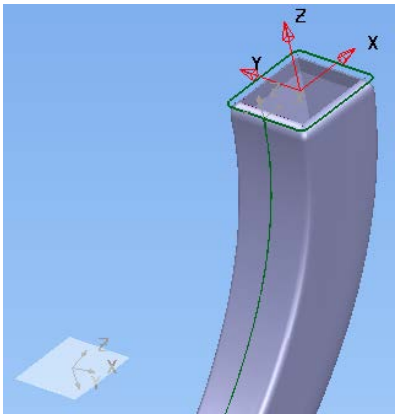
- Select the **Workplane** icon on the top toolbar.
- Select the **Single Workplane at centre of selection** button on the left toolbar. 



The Workplane is positioned at the centre of the selected curve.

- From the top left of the screen select **File > Import**.

- Locate and Open the **3mm princess cut.obj** file.



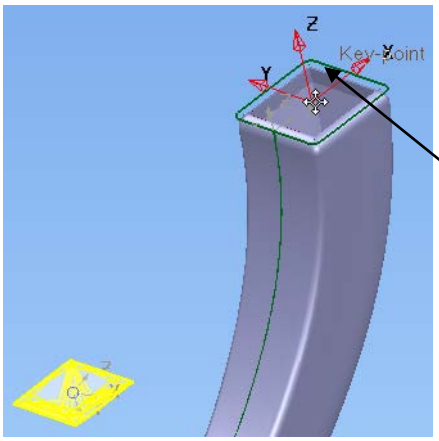
The gem is imported aligned to the current active Workplane but at the world origin.

We need to select the gem and move it to the correct position at the end of the shank design.

- Select the gem.

- Click on the **Show general edits options** button o the left toolbar.

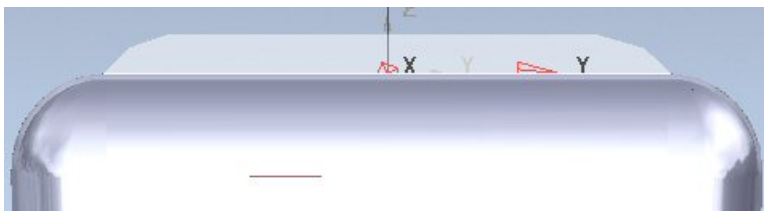
- Select the **Move/copy object** button.



- Make sure that the move origin is at the world origin (centre of the gem) before moving it.
- Click on the key-point at the active Workplane to move the gem.

- Press to finish the move/copy function.

- View the model **from right X +** to see if the gem is at the correct height.

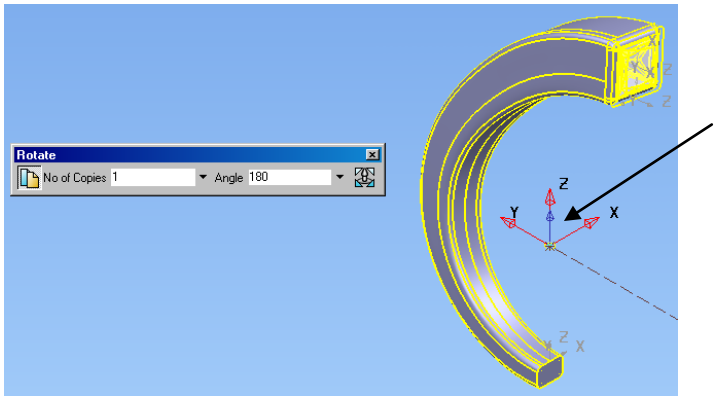


In this case the princess cut diamond is perfectly positioned.

- From the bottom left of the screen select **Workplane 1** from the list.

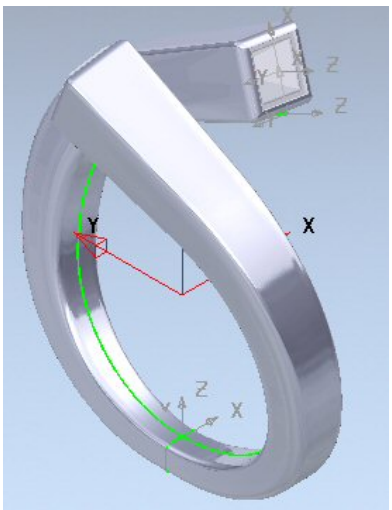
- Select the **View ISO 1** button on the right toolbar.

- Drag a window around the whole model including the gem and select the **Rotate object** button on the left toolbar.



- Make sure that the blue rotation arrow points up the **Z axis**.
- Enter No of **Copies 1** and an
- **Angle 180**.
- Press return on the keyboard.

- **Close** the Rotate form.
- Select the **View ISO 1** button on the right toolbar.

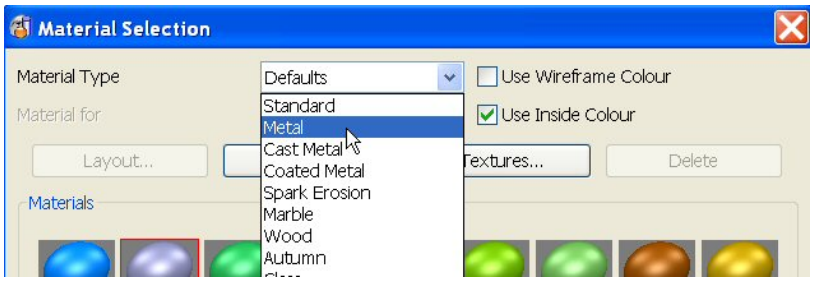


The design is complete.

All that remains is to change the materials to 18ct gold for one side and Silver for the other.

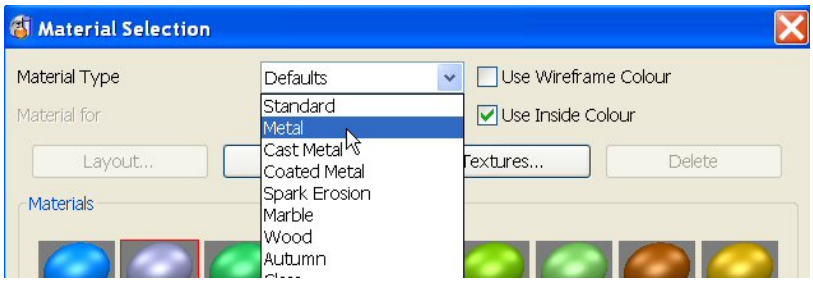
We will then create a rendered image to finish.

- Select one of the two shank solids.
- Select **Format > Materials** from the top of the screen.

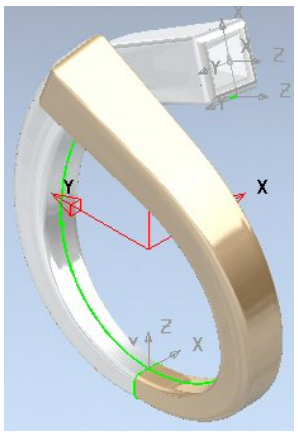


- Select **Metal** from the pull down list.
- Select **gold 18ct** then **OK** to accept and close the form.

- Select the remaining solid shank.
- Select **Format > Materials** from the top of the screen.



- Select **Metal** from the pull down list.
- Select **Silver** then **OK** to accept and close the form.

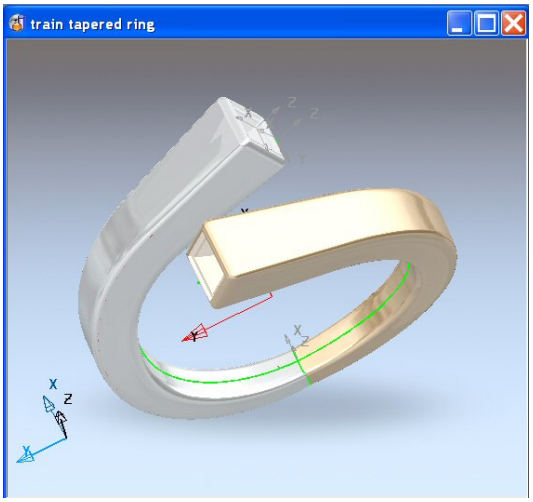


- The model is now ready for rendering.

  First you need to rotate and position the model to suit your preference.

- Rotate your model into the desired angle and size.

- Select the **Restore down** button on the top right hand corner of the screen.

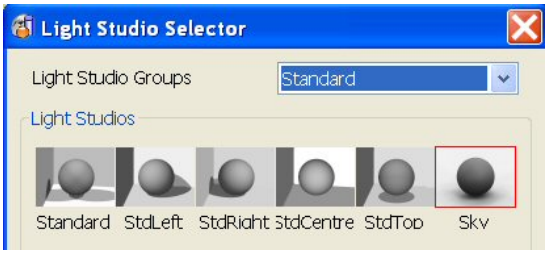


- Adjust the model window smaller to allow faster rendering.

- Also rotate the design into the desired position, similar to that seen here.

- Select the **Raise the render form** button from the right hand toolbar.

- In the **Lights** area of the form select the **Change** button.



- From the pull down list select **Standard.**

- From the **Light Studios** area click on the **Sky** button.

- Click on OK to accept the form.

- Click the **Render** button on the Render form.

- There may be a slight delay before rendering starts.

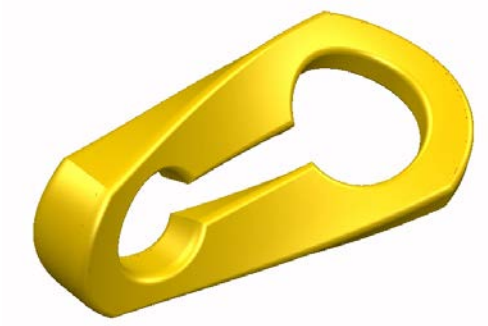
- Eventually you will be left with an image such as the one seen on the next page.

- If you are happy with the result select **OK** on the Render form.
- To save the image select **File** at the top left of the screen then **Print to File**.
- Search Location **D:\users\Training\Coursework\Delcam Designer Jobs**
- Type in a name as **Finished Ring.jpg** then **Save**.

- Select the **Save** button on the top toolbar to save the model itself.
- Type in name **Tapered twist ring.psmodel** (in same location) then **Save**.

8. Chain Link

Introduction.



This tutorial will demonstrate the interoperability between Delcam Designer and JewelSmith. The demonstration will move you between the two programs for the creation of a chain link from complete design to machining.

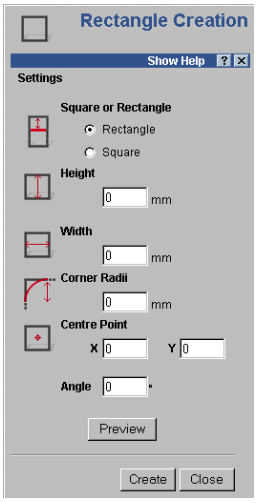
- Select **Create New Project**

- Select **Blanks – 3 Axis Flat** from the Models Tree.

- Leave name as **ThreeAxisFlat**, **Width** and **Height** of **25** with a resolution of **30 pix/mm** and press **Accept**.

- Select the **Assistant** tab.

- Select **Create Rectangle**.



The rectangle creation page appears, showing the options available.

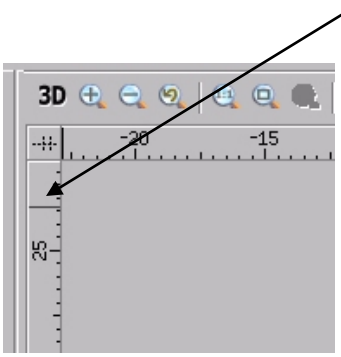
Clicking in the screen and dragging to a size, or by entering in the values can create the rectangle.

Several rectangles can be created without closing down the page.

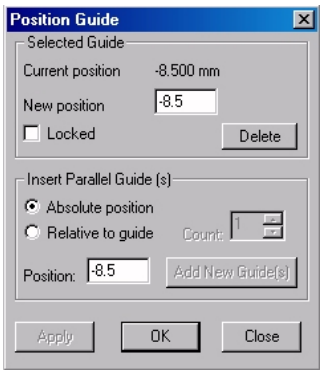
- Select the option **Rectangle**, a **Height** of **9** and **Width** of **14**

- Leave **Invert corners unticked** and **Centre Point** is at **X 0 Y 0** and **Angle** at 0**.**
- Select **Preview** and then **Create** and then **Close**.

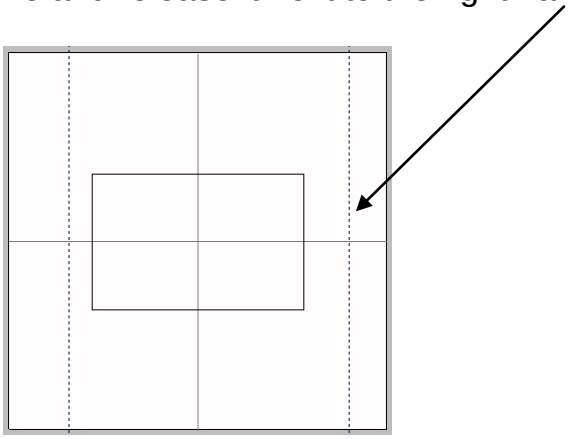
The remaining shape of the link will be completed with the aid of guidelines. Guidelines can be accessed from the left hand and top rulers on the model border.



- Hold down the **left mouse button** within the **left ruler** and drag inwards a **vertical guideline**, releasing the mouse at the left hand side of the rectangle.
- Right mouse click on the guideline and enter **–8.5** and **Apply** then **OK.**



- Drag a second **vertical guideline** and release it next to the right hand side of the rectangle.
- Change the position to **10**.
- **Apply** and then **OK**.



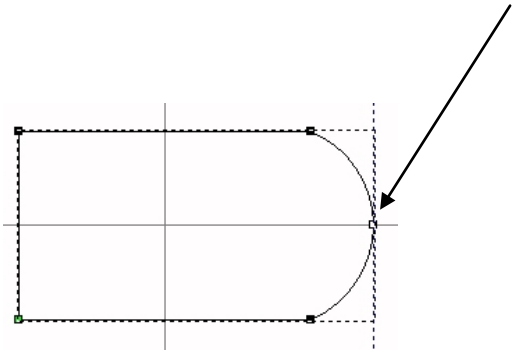
The rectangle will be modified to create the basic outline of the link. This can be done more effectively in **Node Editing**.

- Select the **Rectangle** and then select **Node Editing**. (Or press **N** for shortcut on keyboard)

When a vector is selected, the option Node editing will display the points (nodes) that are used in this vector. In this mode the individual points can be moved manually by dragging them to a new position or the curve shape altered by moving one of the control points (shown as empty squares on a dotted line).

- Right click on the right hand side vertical vector of the rectangle Select **Convert span to Arc** (or press **A** for shortcut)

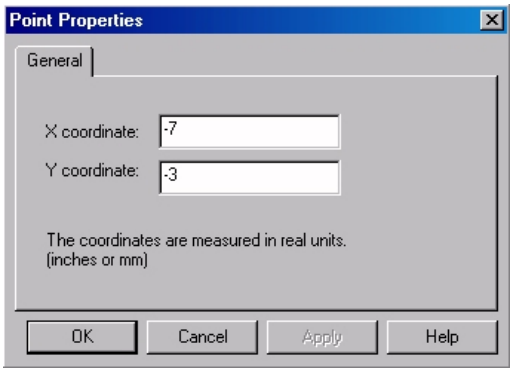
- Drag the **Arc** until it intersects with the right hand **guideline**.



- Convert the left hand vertical **Span** to and **Arc.**

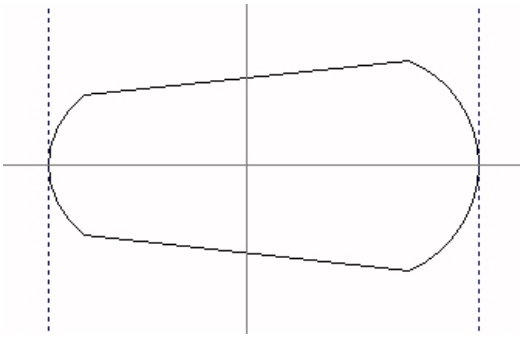
The position of each end **Node** (of the new arc) will be changed to alter the shape further.

- Move the cursor over the **bottom Node, right mouse click** and then select **Properties**.
- Enter **–3** for **Y coordinate** and select **OK**.

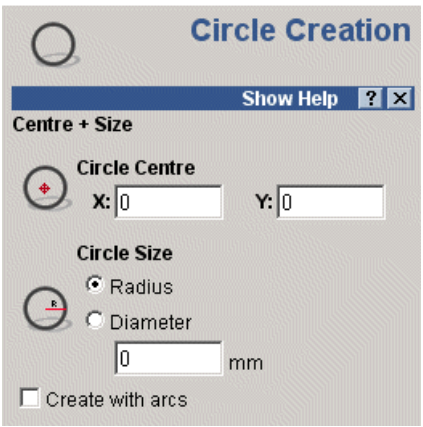


- Repeat the above for the top node but enter **3** in the **Y coordinate** box.
- Drag the left hand **arc** until it intersects the **guideline.**

The shape should now look as below.



- Select **Create Circles**.



Circles are generated from a circle centre and a radius or diameter.

The option, create with arcs is mainly used if you are going to offset the circle, or for machining purposes.

- Set the **Circle Centre** as **X 5.5 Y 0**. Select **Diameter** and enter **5.5mm**.
- Press **Create** and then **Close.**

- Create a **second circle** with coordinates of **X –4.5 Y 0** with a **Diameter of 4mm**.

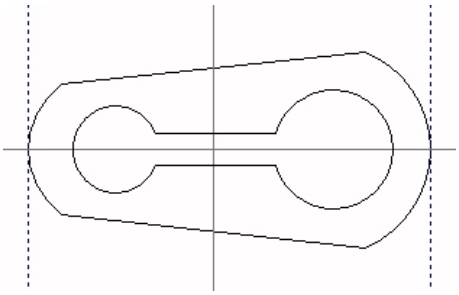
- Select Create Rectangle.

- Enter a **width** of **8mm** and **height** of **1.5mm.** Leave the other settings as **default** and then select **Create** and **OK.**

The two circles and rectangle will be welded together to create a single vector. This will complete the inner 2D design of the link.

- Select the **two circles** and **rectangle** (Shift button can be used to multi select)

- Select Weld vectors from the assist page to create a single vector.



The vector design of the link is now complete in ArtCAM. These vectors will now be used in **Delcam Designer** to create the 3D model.

Creating the 3D model

- Select all the vectors in **Jewelsmith** and select **Edit – Copy**
- Open **Delcam Designer**

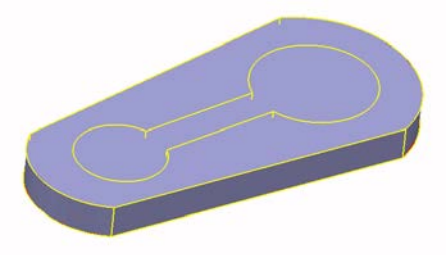
and select **Edit – Paste**.

This will paste the curves from Jewelsmith into Delcam Designer. We will now convert the curves into solids.

- Adjust the view / position the vectors to obtain a good working view
- Ensure all the **vectors** are **selected** (highlighted yellow)

- From the **Solids toolbar** select **Solid Extrusion**
- Enter a **length of 1.5mm** and select **OK.**

A 1.5mm thick solid(s) has been created from the selected vectors. Note: As two separate vectors were selected, two separate solids have been created.

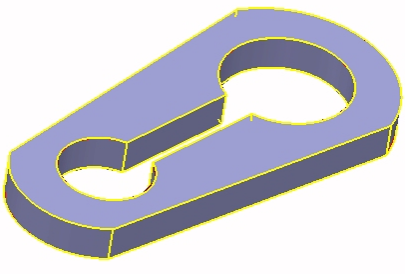


The **inner solid** will be removed from the **outer solid** to complete the basic flat shape of the link.

- Select the **outer solid** and **Right click** over it to bring up the sub-menu and select **Active.**
- Select the **inner solid** with the left mouse button.

The solids history tree now shows the active outer solid (denoted by the red flag) and the inner solid selected (boxed).

- From the **Features toolbar** select **Remove the selected Solid from the Active Solid.**

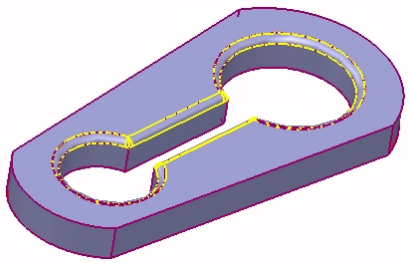


The edges and corners will now be filleted

- **Select** the **top face** of the model
- From the **Features menu** select **create solid fillet**.

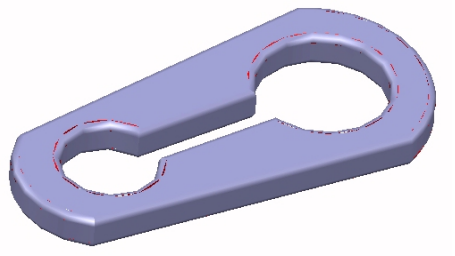
- Enter a **radius of 0.3mm** and select **Fillet a face.**

- Select **Apply** but do **not** close the form.
- **Rotate** the model to apply the same fillet on the other side.
- **Select** the surface. Select **Apply** then **Dismiss.**



This completes the fillets on the inner edges of the model. The outer edges also require filleting.

- Select **Create solid fillet** again from the **Features menu**
- On the **Fillet form**, select the **Follow continuous edges** option.
- Multi select (shift button) the complete outer edge of the model.
- Ensure the **radius is still 0.3mm** and select **Apply.**
- **Repeat** for the other side and **dismiss** the Fillet form.



The model is now completely filleted. The shape will now be further altered using the powerful **Morphing** function of Delcam Designer.

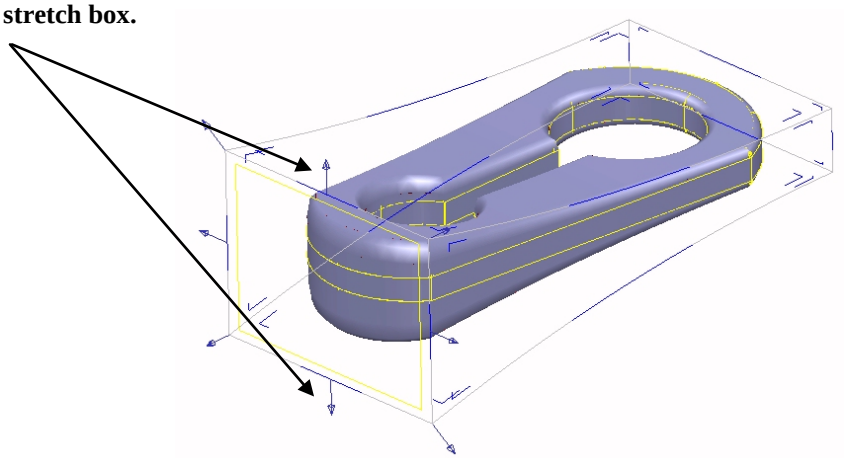
Morphing the model

- From the **Feature** tab select **Create a morph feature**

- In the **Morphing box** select **Flexible box** and then **Stretch box**

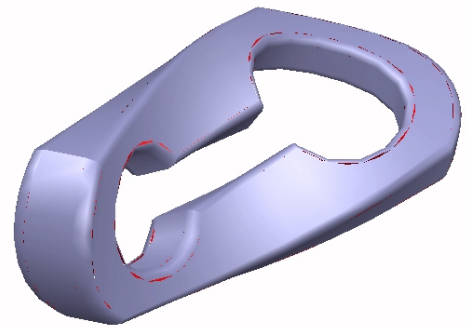
- **Select** the **vertical face** on the narrower end of the model. Use both the **up and down arrows** to stretch the model until it looks like the image below.

Pull arrows to stretch box.



- With the face still **selected** and the **Morphing form open**, Select **Twist the box** .

- **Twist** the **circular arrow** approximately **90 degrees**, and then **OK** the morphing form.



Rendering the model

If required at this stage, the model is ready for exporting back to JewelSmith to calculate the toolpaths.

But before that, the Rendering functionality of Delcam Designer will be demonstrated. Rendering allows realistic shading of the model to represent the intended material.

- Ensure the model is shaded

- Select the model and from the top **menu bar** Select **Format – Materials**
- Select **Custom** and from the **pull down menu** select **gold new.**
- **Apply** and **OK** the **material selector** form and then **OK** the **materials selection form.**

The link is now shaded in the gold, but can be enhanced further.

- From the shading toolbar, select Enhanced shading

The model is now enhanced to a more realistic (metallic) shade.

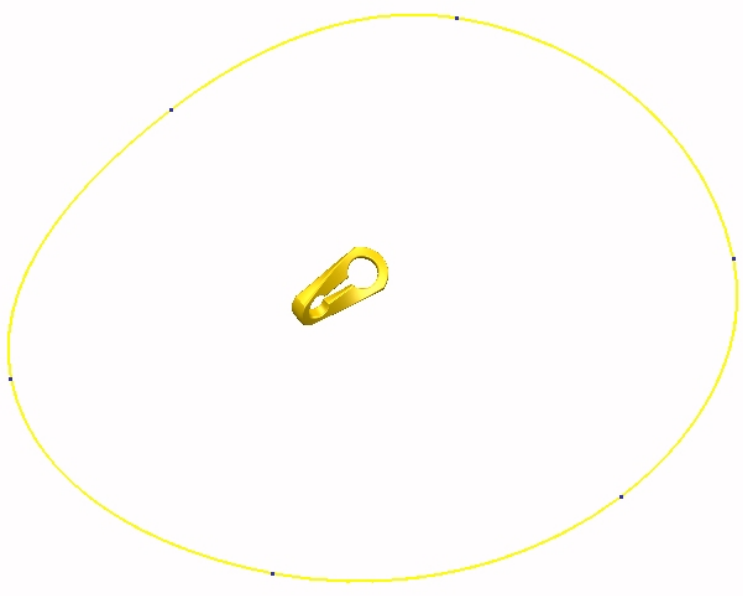
Chain assembly

Multiple copies of the model link can be easily created then joined together to create a chain (e.g. bracelet), and presented as a finished piece.

- Revert the model back to standard **shading**

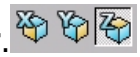
- From the top **curve menu** select **create a bezier curve**
- **Zoom** and position the model accordingly, and then draw a complete **bezier curve** around the model as shown below.

Note: snap to the **keypoint** to complete a single curve



This **bezier curve** will be used as the **drive curve**. Multiple **copies** of the link model will be created and positioned onto this **drive curve** to create our chain.

The orientation must be firstly checked to obtain the correct result

- Ensure the **Z workplane** is selected from the bottom **views toolbar**. 

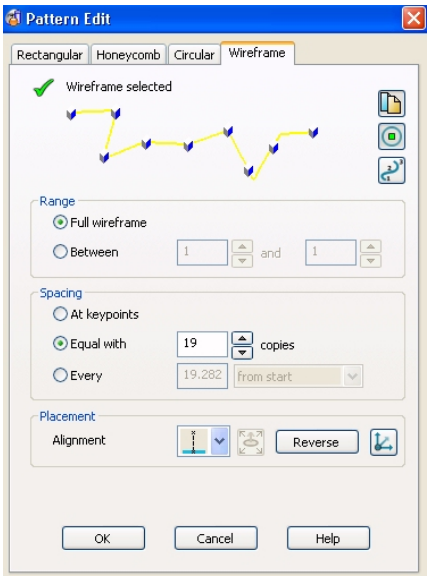
- Select **view from top (+Z)**

The model and drive curve are positioned correctly and now ready to create the chain link.

- Select the model and from the **General edits toolbar** select **Create pattern of objects**

- Select the **Wireframe** tab and then select the **drive curve** from the workspace.

- Select **Rotate copies to axis** from the orientation pull down menu, and then select the option **equal with** from the **spacing menu.**



- Adjust the number of copies using the **arrows** until the links combine correctly.

- Select **OK** to generate and complete the chain assembly.

Note: The single link can be hidden from view by selecting it first then **CTRL-J** (Blank)

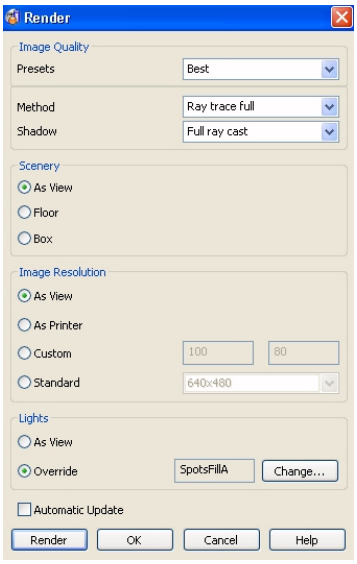
Advanced Rendering

Delcam Designer includes further Rendering options to enhance this completed chain design.

- Raise the **Render form** by selecting the option from the **Rendering toolbar.**

- In the **Render form** select the option **As View** in the **Scenery** and **Image resolution** menus.
- In the **Lights** menu, select **Override** and then change the option to **SpotsfillA**

The form should like as shown below.



- Select **Render**

The model is rendered in a new window. Note: The time taken to render can be reduced if the window size is decreased.

- Experiment with the different options to see the different effects before selecting **OK**.

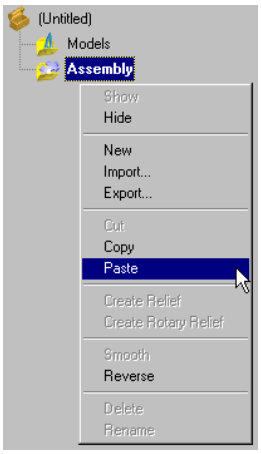
The toolpaths for the link will now be calculated using JewelSmith.

- Select the **original** model using the **window menu**.
- Select the single separated **link** and then **CTRL-K.**

This keeps the link but hides (blanks) the remaining chain from view

- Select the **model** and then **Edit-Copy** from the **menu bar**.

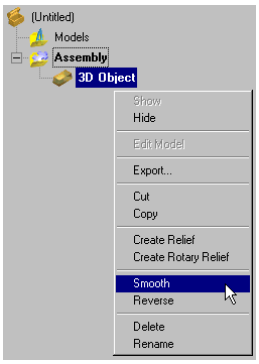
Machining the model



- Return to the original **Chain Link Project** in ArtCAM **JewelSmith.**

If the **ArtCAM** model (vectors) is still open then close the current model.

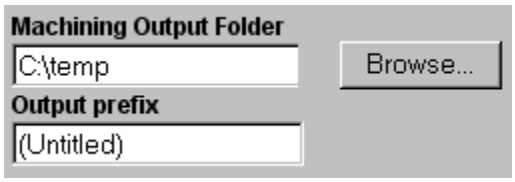
- Right click on **Assembly** in the **Project tree** and select **paste**.



The Delcam Designer model is now pasted into JewelSmith. The **'triangulation'** surface finish can be improved by right mouse clicking the **New Assembly,** and then selecting **Smooth.**

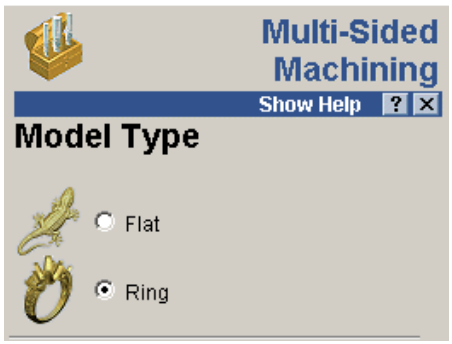
- Select the **Assembly** root folder and **Multi-Sided Machining**

- Change the output folder to **D:\users\Training\Coursework\Delcam Designer Jobs**



- Change the Output Prefix to **Chain Link**

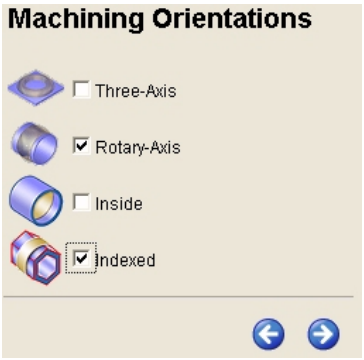
- Select **Edit Settings**.



- Select **Ring** and then **Forward**.

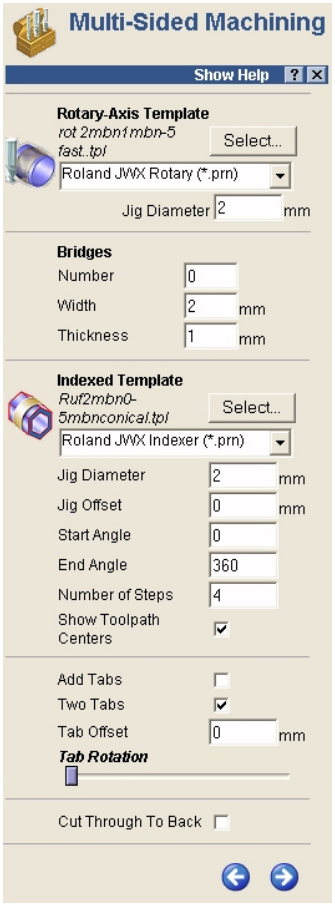
- Select **Forward**. On **Model Size.**

- Select the **Rotary** and **Indexed** options. This will combine rotary and indexed (predetermined angles) to machine the complete link.



- Select **Forward**.

- Complete the form as shown below.

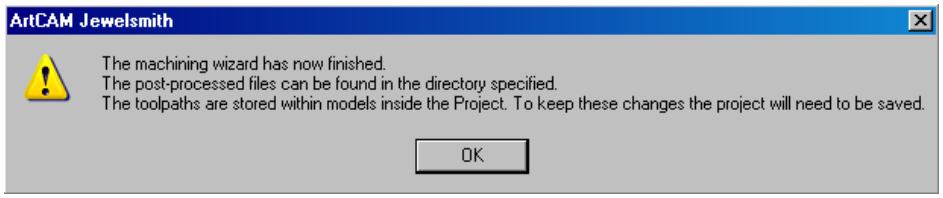


Select the templates from **ArtCAM Files\Toolpath Templates\Alcami\Roland JWX**

Ensure the correct JWX Roland machine is selected for Rotary-Axis and Indexed respectively.

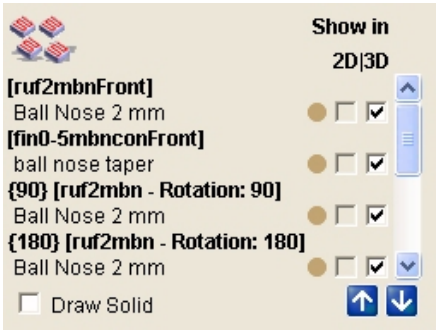
For the Indexed Template, 4 steps within a total angle of 360 degrees will ensure rotary increments of 90 degrees.

- **Select Forward** and press **Accept** to commence the toolpath calculation.



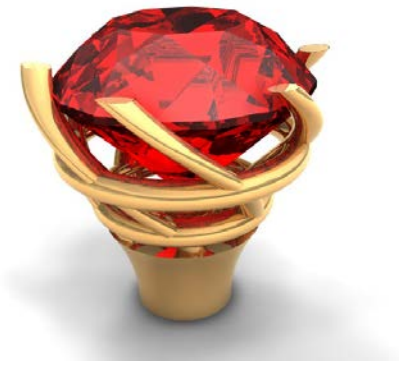
- Select **OK** to accept the Machining

A total of eight toolpaths have been created. They can be simulated to show the final result. Note: Simulation is displayed un-wrapped for clarity.



9.Twisted Collet.

Introduction.

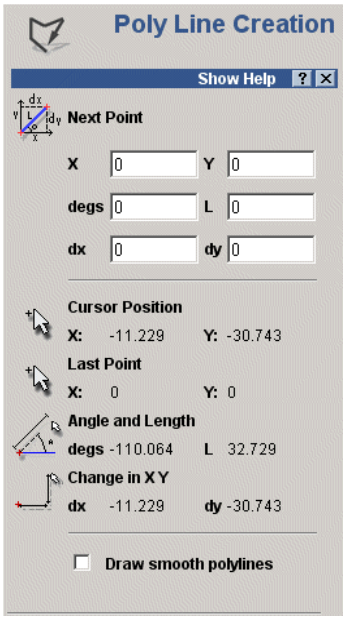


This tutorial will demonstrate the interoperability between Delcam Designer and JewelSmith. The demonstration will move you between the two programs for creation of the custom collet.

- Select **Create New Project**

- Select **Blanks – 3 Axis Flat** from the **Models Tree**.

- Change name as **Untwisted Collet**, **Width** and **Height** of **10** with a resolution of **30 pix/mm** and press **Accept**.



- Select the **Assistant** tab.

- Select **Create Polyline**.

A poly-line is a continuous line made up of two or more points.

These points can be entered by clicking on the screen, clicking on the end of lines, or by entering the actual or relative co-ordinates.

By holding down the mouse button and moving the mouse, freehand sketching is generated.

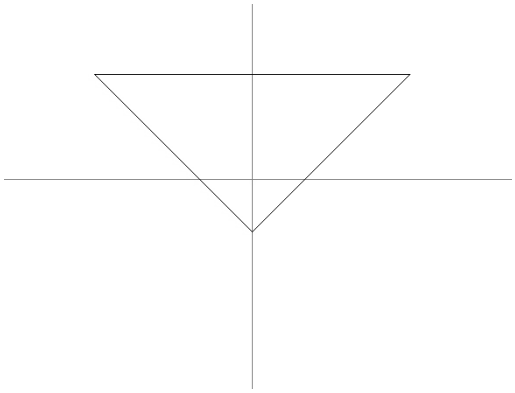
The ability to smooth poly-lines when creating turns the lines into curves as you enter the points.

- **Untick Draw Smooth polylines**. Enter **X –1.5** and **Y 1** and press **Add.**

This is the starting point of the poly-line and nothing is seen in the 2D view.

- Enter **L 3** and press **Add.**
- Enter **dx –1.5** and **dy –1.5** and press **Add.**
- Enter **dx –1.5** and **dy 1.5** and press **Add.**

- Select **Close.**



The diamond is now complete the next thing that we will create will be the shapes for the collet.

- Select **Create Polyline**.
- Enter **X –1.75** and **Y 1** and press **Add.**
- Enter **X –0.6** and **Y -2.5** and press **Add.**
- Select **Close**

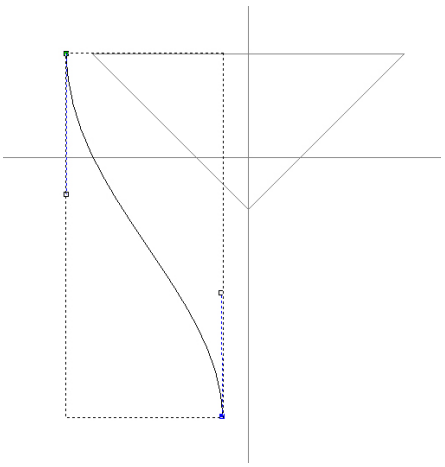
The **straight** line will now be converted to a **curved** line

- Select the **line** and then Select **Node Editing**. (or press **N** for shortcut on keyboard)

When a single vector is selected, the option Node editing will display the points (nodes) that are used in this vector. In this mode the individual points can be moved manually by dragging them to a new position or the curve shape altered by moving one of the control points (shown as empty squares on a dotted line).

- Right click on the **line** and Select **Convert span to bezier** (or press **B** for shortcut)

- Drag the handles (empty squares) so they look like the image below.

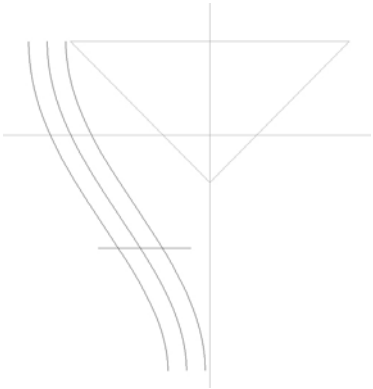


- Press **select vectors**.
- With the **line** selected. Select **Offset vectors**.

- Select an **Offset Distance** of **0.2**. Select **Outwards**.

- Select the original **line** again, Select **Inwards.**
- Select **Close**.

- Select **Create Polyline**.
- Enter **X –1.2** and **Y –1.2** and press **Add.**
- Enter **L 1** and press **Add.**
- Select **Close.**

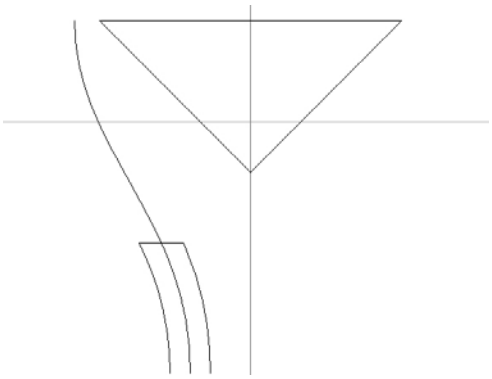


This will give you a horizontal line across the 3 vectors. This new line will be used to create the base of the collet.

- Select **Trim vectors to intersections.**

The mouse arrow is replaced by a scissors icon. This dynamic icon changes to open scissors when it has found some vectors it can trim back.

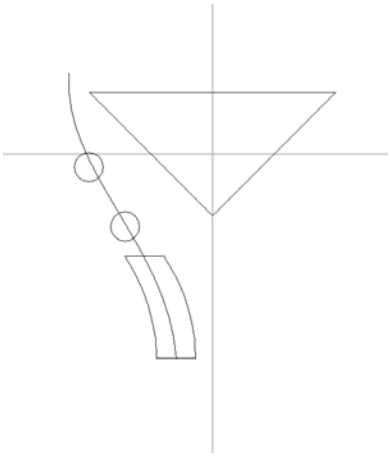
- Remove the top left vector the top right vector and either sides of the horizontal line as shown below.



- Select the 3 vectors and **Join** them together then close the line with **Close Vector With A Line**

- Select Create Circles.
- Enter **X –1.25** and **Y –0.5** and **diameter 0.4**.
- Select **Create.**
- Enter **X –1.6** and **Y 0.3** and **diameter 0.4** to create a second circle.
- Select **Create** and **Close.**

We now have the completed vectors that we will use to create the surfaces in Delcam Designer.



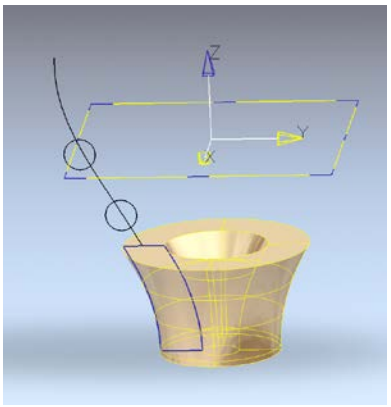
Creating the surfaces

- Select all the vectors in Jewelsmith except for the diamond and select **Edit – Copy**
- Open Delcam Designer

and select **Edit – Paste**.

This will paste the curves from Jewelsmith into Delcam Designer so that we can convert them into surfaces.

- Select the closed rectangular **polyline** and the y-axis (use the zx 'face' of the workplane).
- Go to the **Solid** tab and select the **Create a solid of revolution button**



- Select the first **circle** and select the **Create a solid of revolution button**
- Repeat this with the remaining **circle**

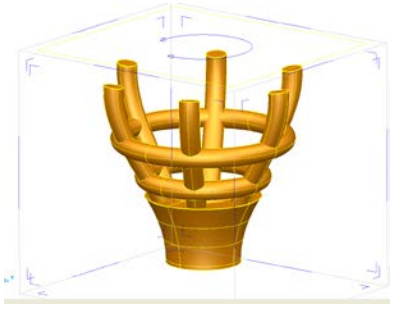
The next thing that we are going to create will be the **prongs**. To do this, we can create one prong, and then rotate copy the remaining.

- Select **the** Arc Menu
- Select **the** Full Arc **option from the menu.**
- With the **y** – axis still selected, place the centre of the **circle** at the top of the single polyline and scale the circle to a **radius** of **0.15mm.**
- Select the single **polyline** and the newly created circle and using the **Automatic – Surfacing** wizard **apply** a surface, which will create a tube.
- With the **Automatic – Surfacing** wizard still open, select the **circle** and **apply** a **fill-in** surface to the top of the tube.

- Select the two new surfaces the tube and the cap (make sure you are in the **y-axis**) and from the **General Edits** tab, select the **Create pattern of objects** button.
- In the **Pattern Edit** box select **Circular.**
- Enter **6** into the **Element box,** and press **OK.**

Creating the Twist

- Select the 6 prongs and from the pull **Solids toolbar** select the **Make solid from surfaces.** This will turn the prongs into solid objects
- Select the base and make it the active solid this can be done by selecting the base then using the right click mouse button and selecting **Active**.
- Select all the other surfaces and from the **Feature** tab select the **Add the selected solid, surface or symbol** button, this will turn the individual solids into one solid
- From the **Feature** tab select **Create a morph feature**
- In the **Morphing box** select **Flexible box** and the **Twist box** select the top of the box and rotate the arrows till the twist looks like the model below.



The collet has now been completed

Save the model in **D:\users\Training\Coursework\Delcam Designer Jobs** as **Twisted collet.psmodel**

Exercise – A Carved Bowl

Introduction

This tutorial will guide you through the process of creating the bowl shown above, and decorating it with a 3D relief created in ArtCAM.

Key features covered in this tutorial are –

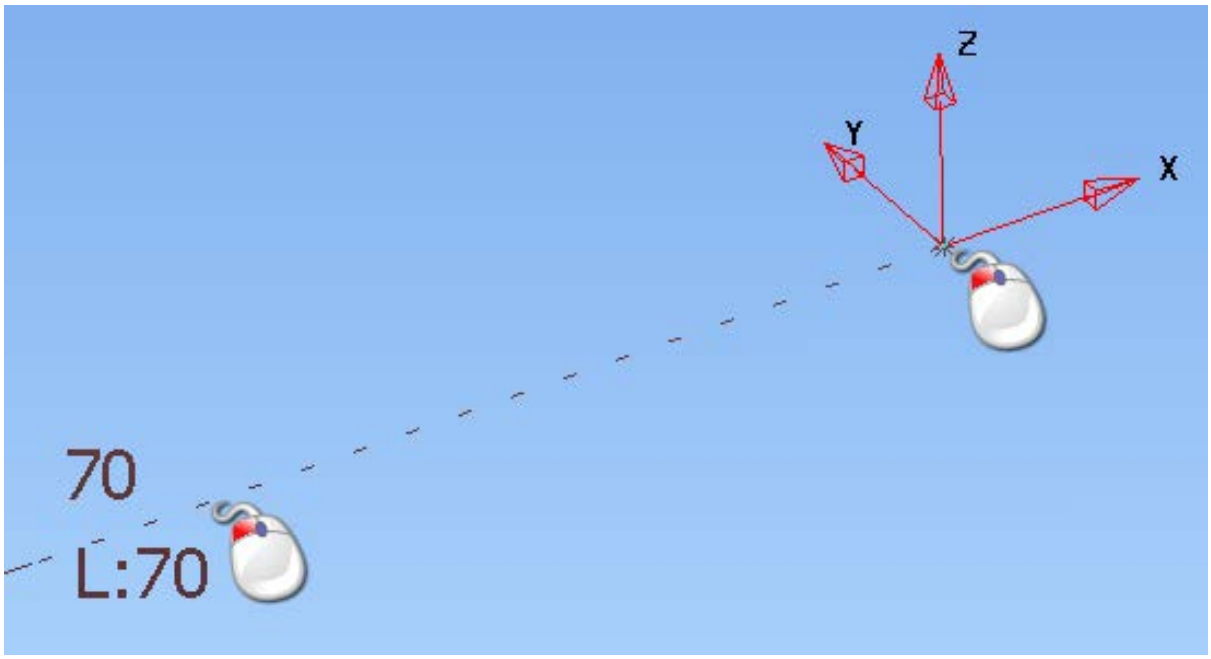
- Sketching complex wireframe
- Creating a surface
- Wrapping an ArtCAM relief

Sketching the Wireframe

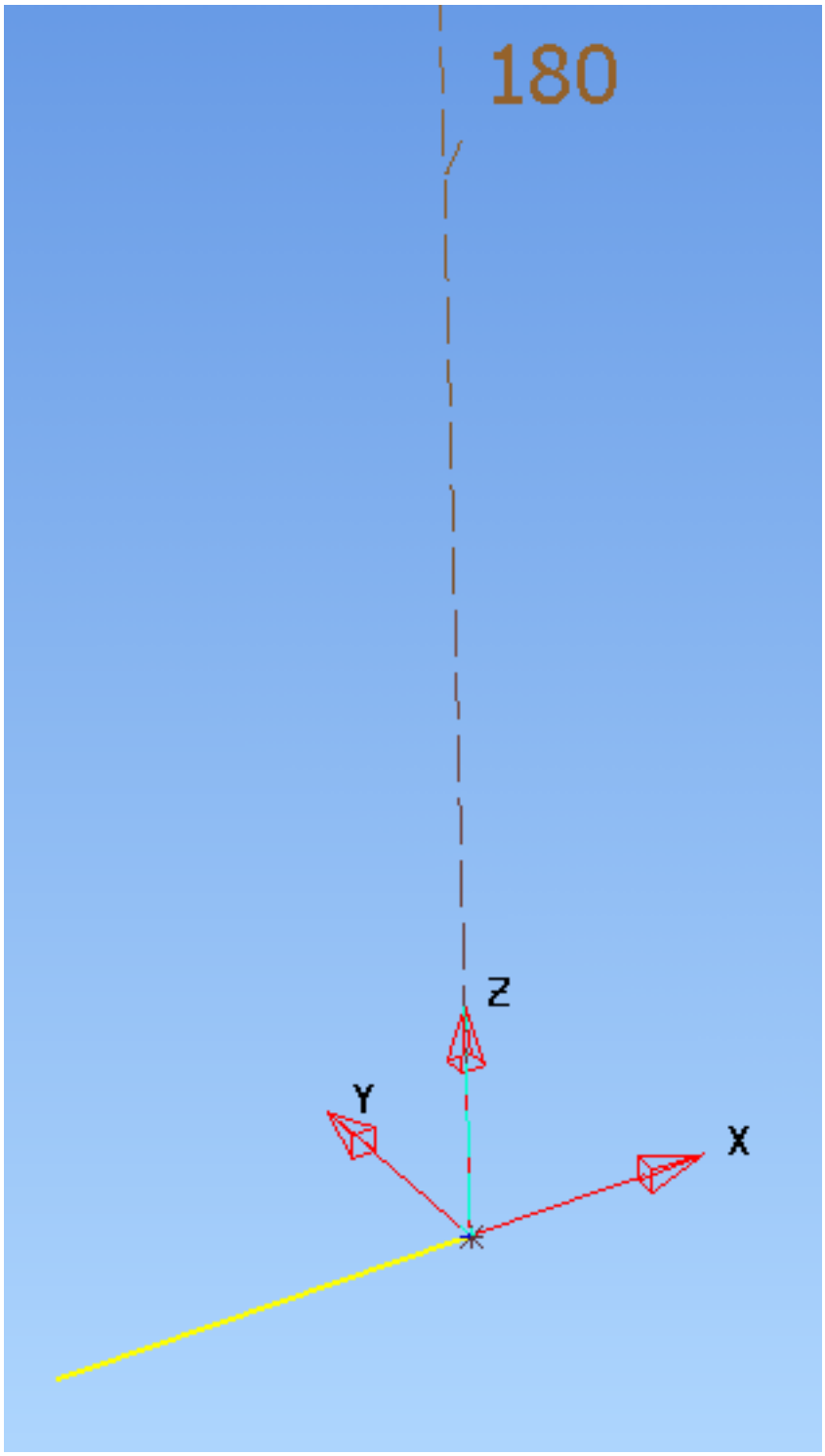
Begin by creating a Workplane at the world origin (**0,0,0**) and rotate the view to (approximately) an isometric orientation.

Click on

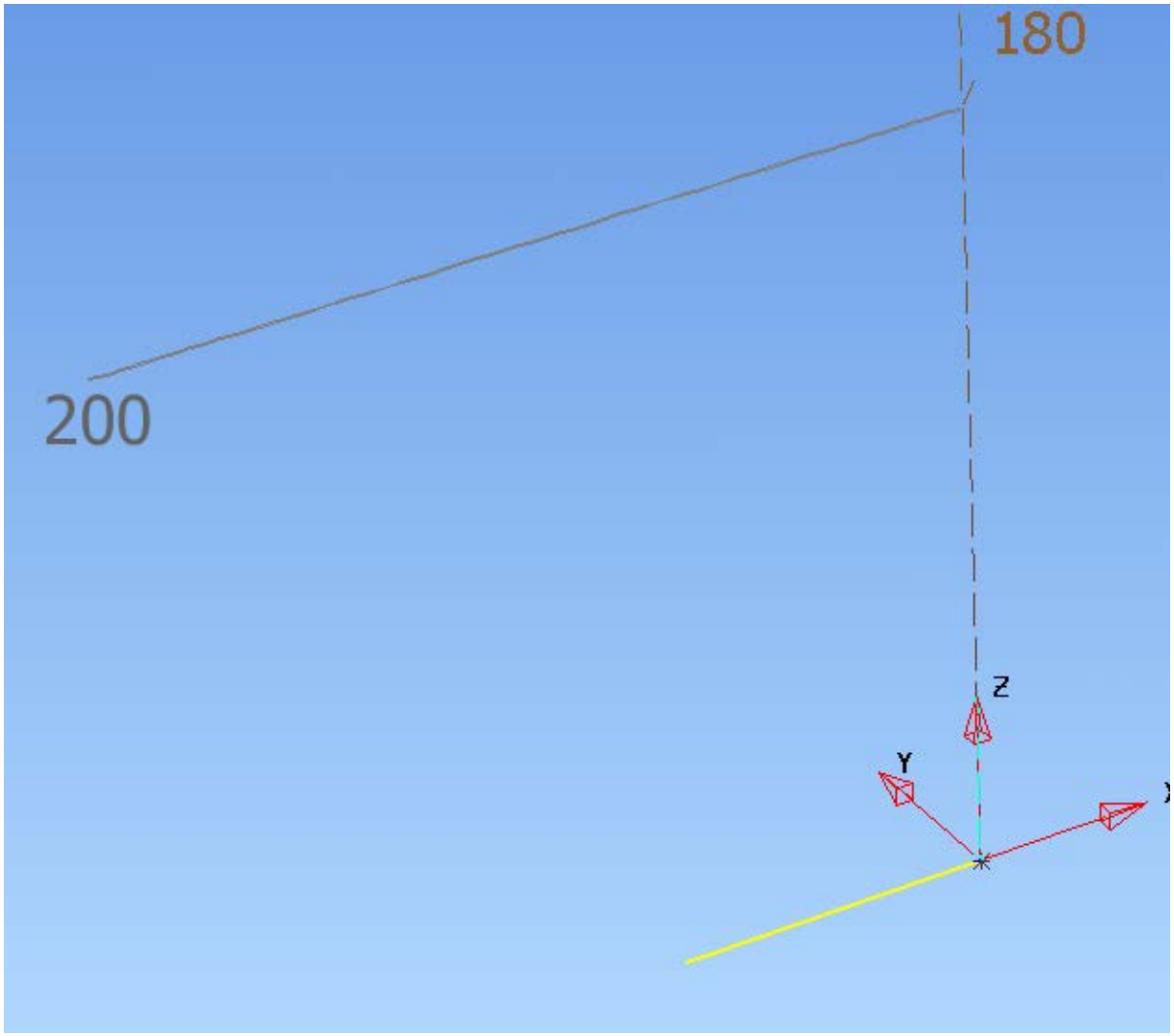
Create a line (in single line mode) from the origin, 70mm along -X

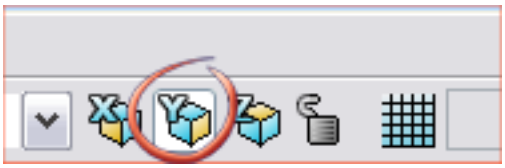


Create a second line that begins at Z 180.

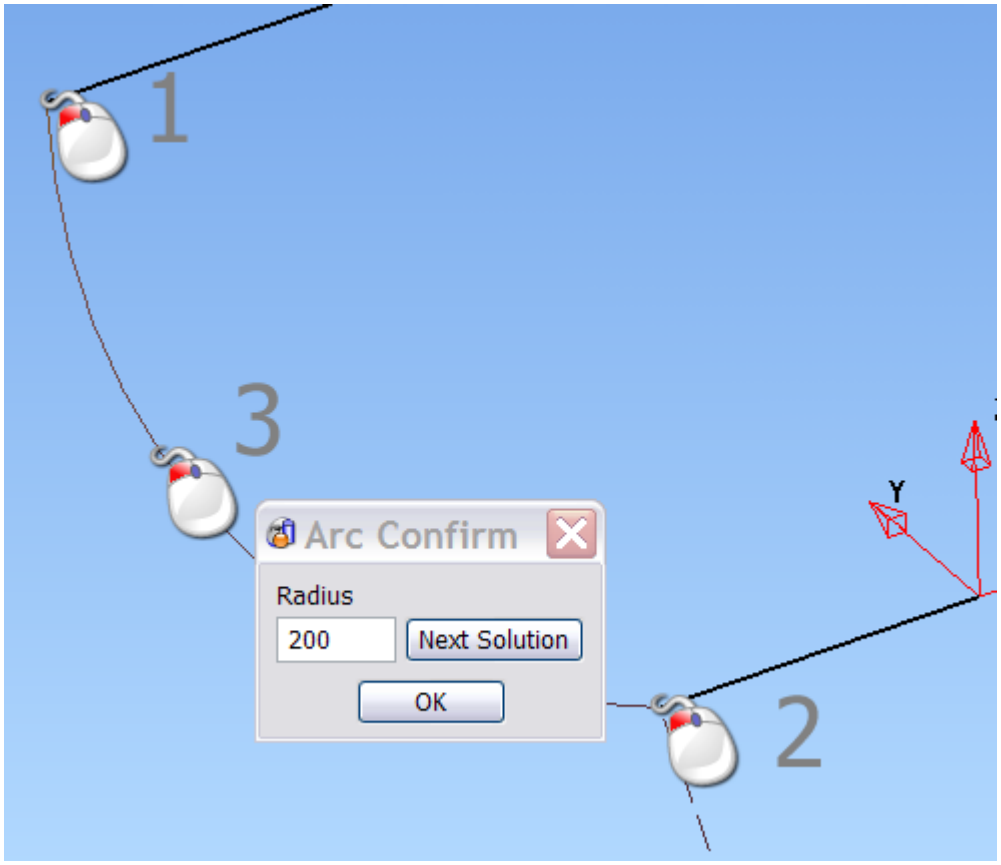


Set the length to be 200mm along –X.

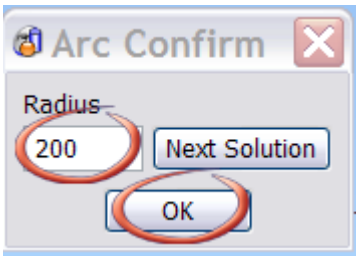


Set the working axis to Y 

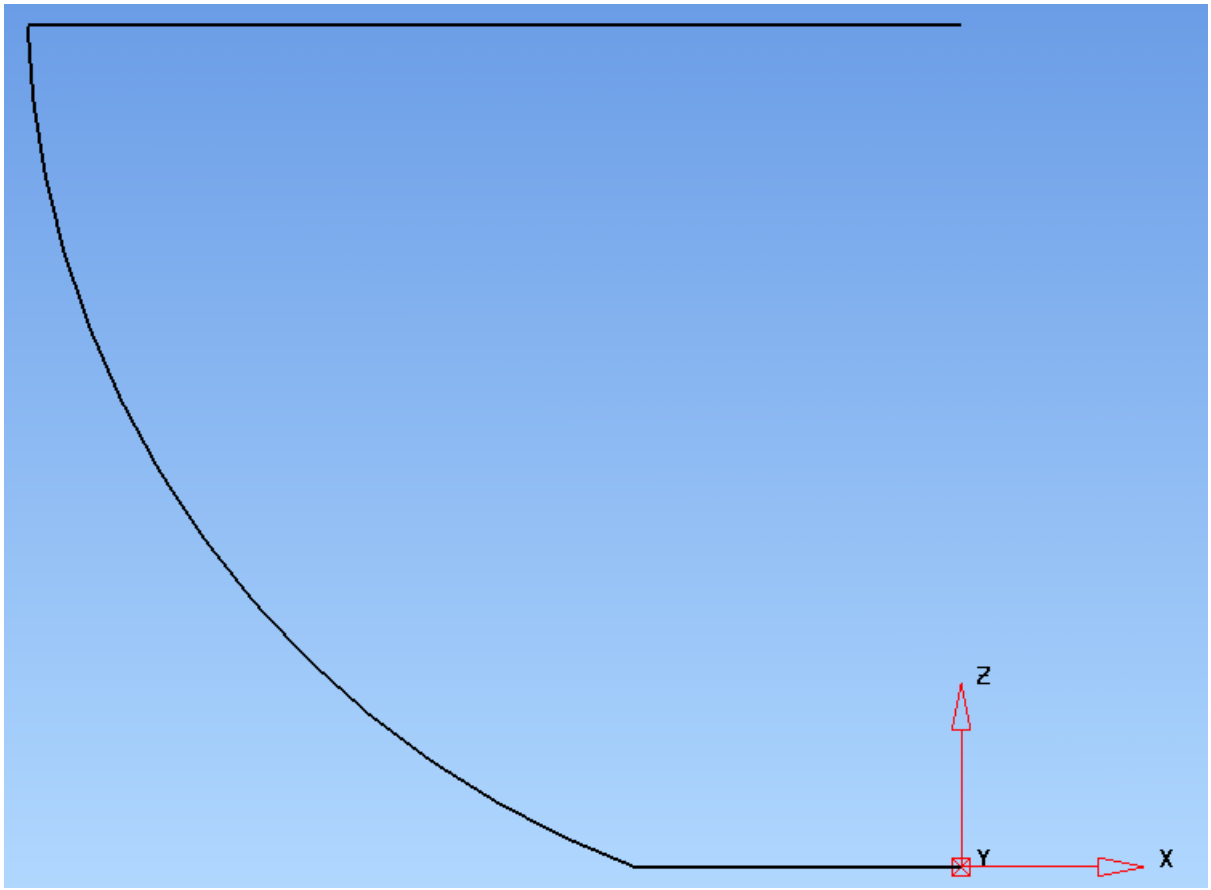
Create a swept arc between the end points of the two lines (click the points in the order indicated).



Set the Radius to 200, and click OK.

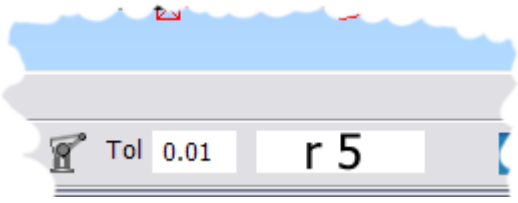


Set the view to be along the Y-axis (Ctrl + 2).



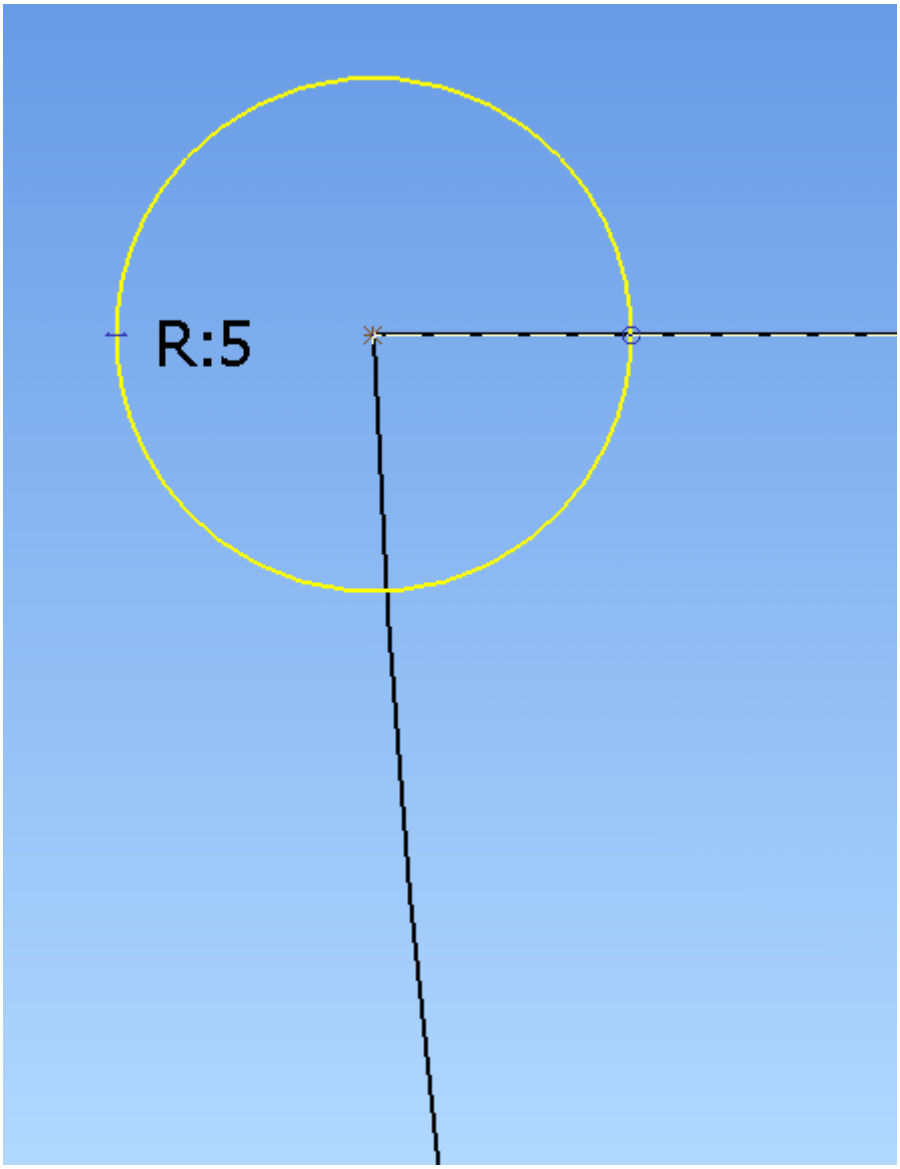
Click the ‘Create Circle’ icon

Set the radius of the arc to 5mm by typing ‘r 5’ in the Command Input box.

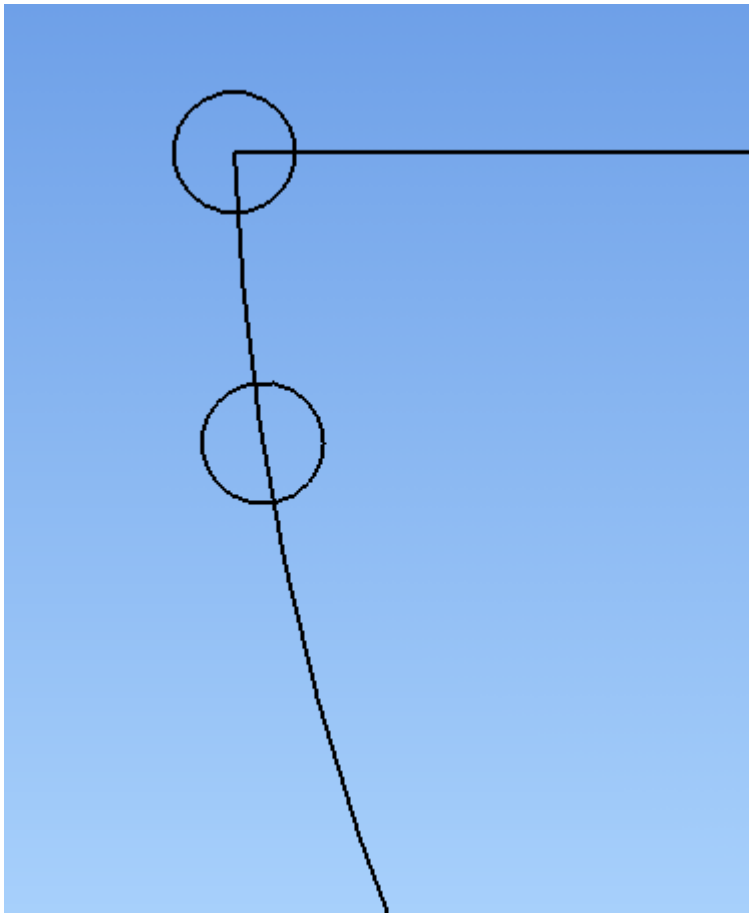


(Press <Enter> to accept)

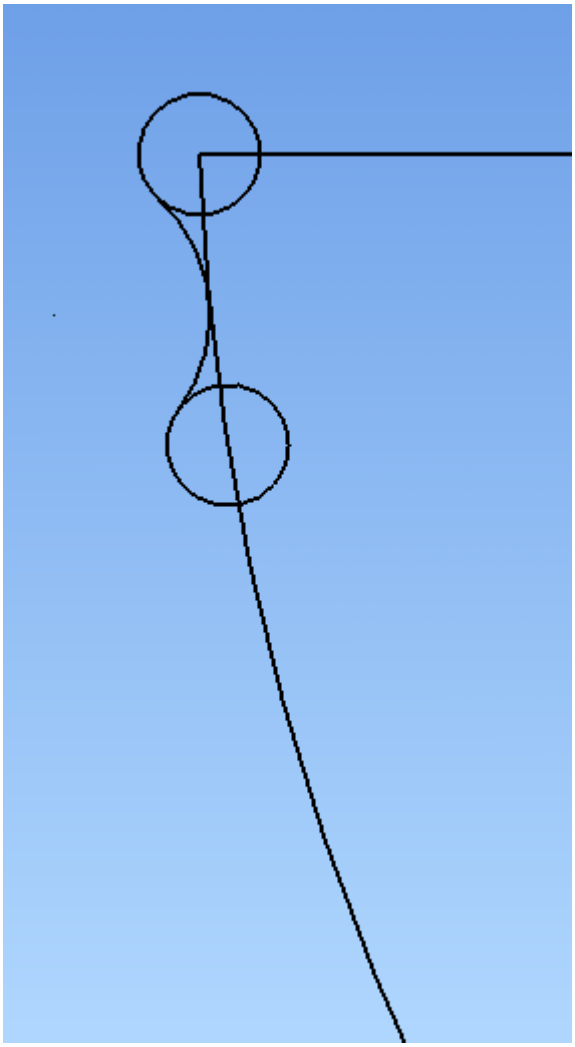
Create the first circle at the top of the original arc as below.



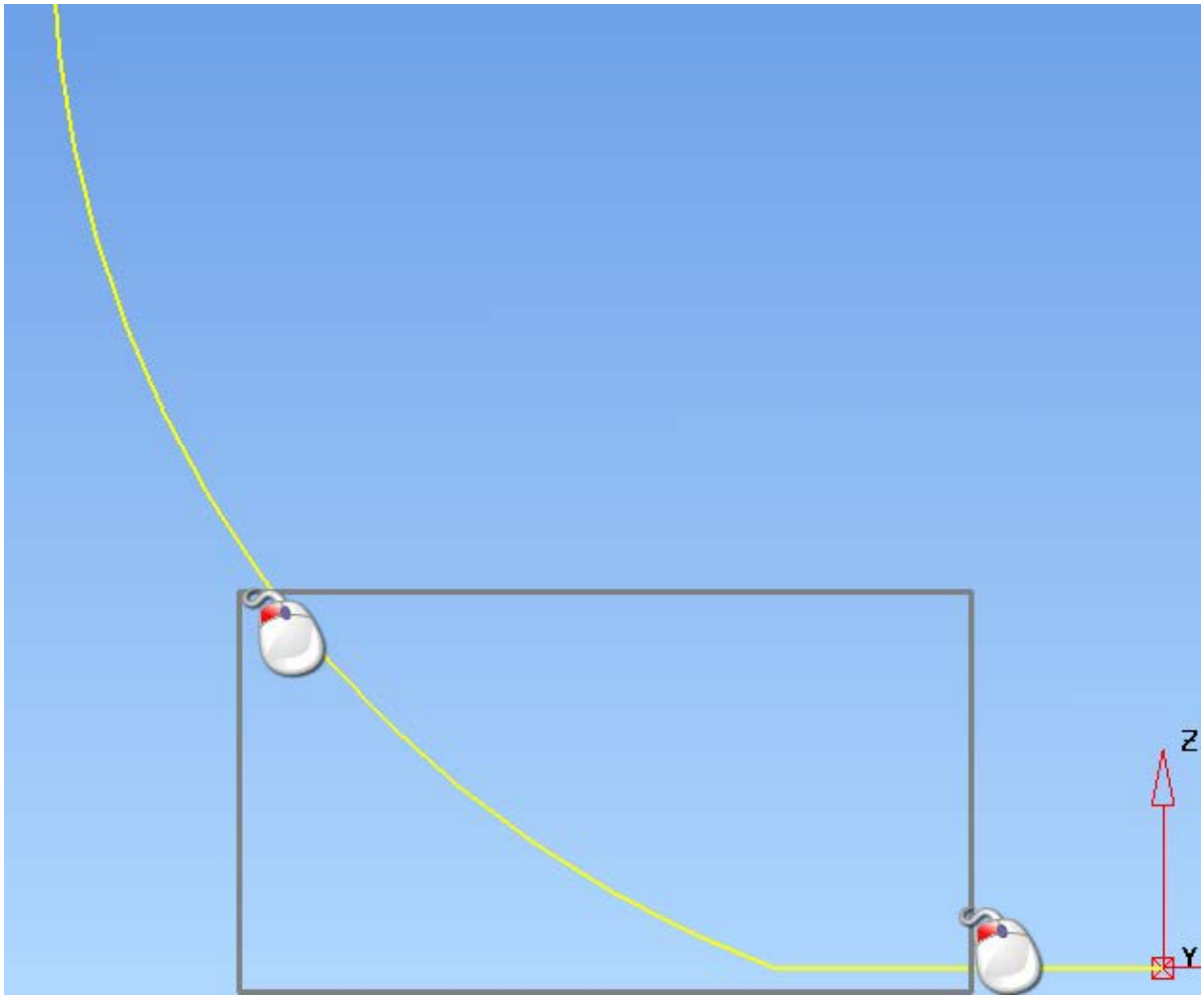
Create a second circle a little lower, but make sure its centre is still snapped to the first arc.



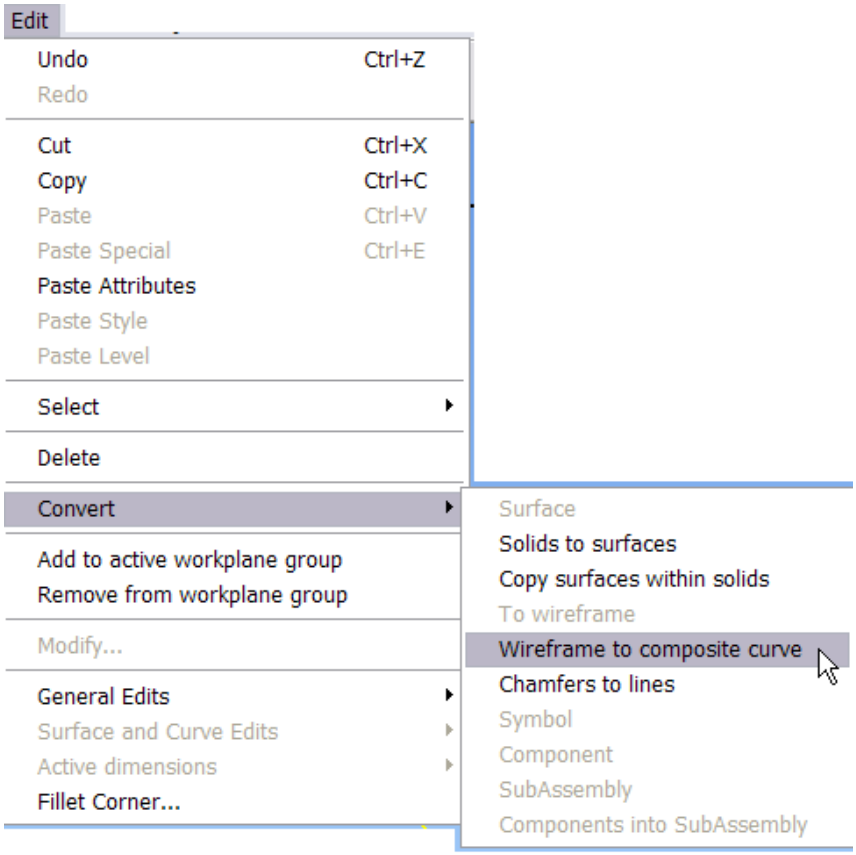
Create a swept arc tangent to both circles and touching the first arc.



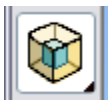
Select the large arc and the line at the base of the bowl.



Convert the arc and line to a composite curve.

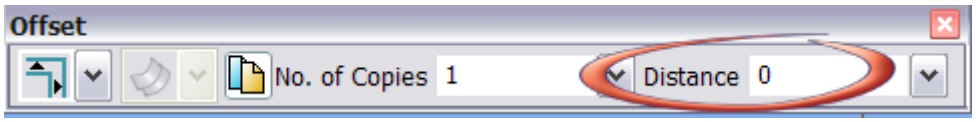


Open the General Edits toolbar

Click on Offset 

Offset the curve by 5mm – making sure it goes in the correct direction!

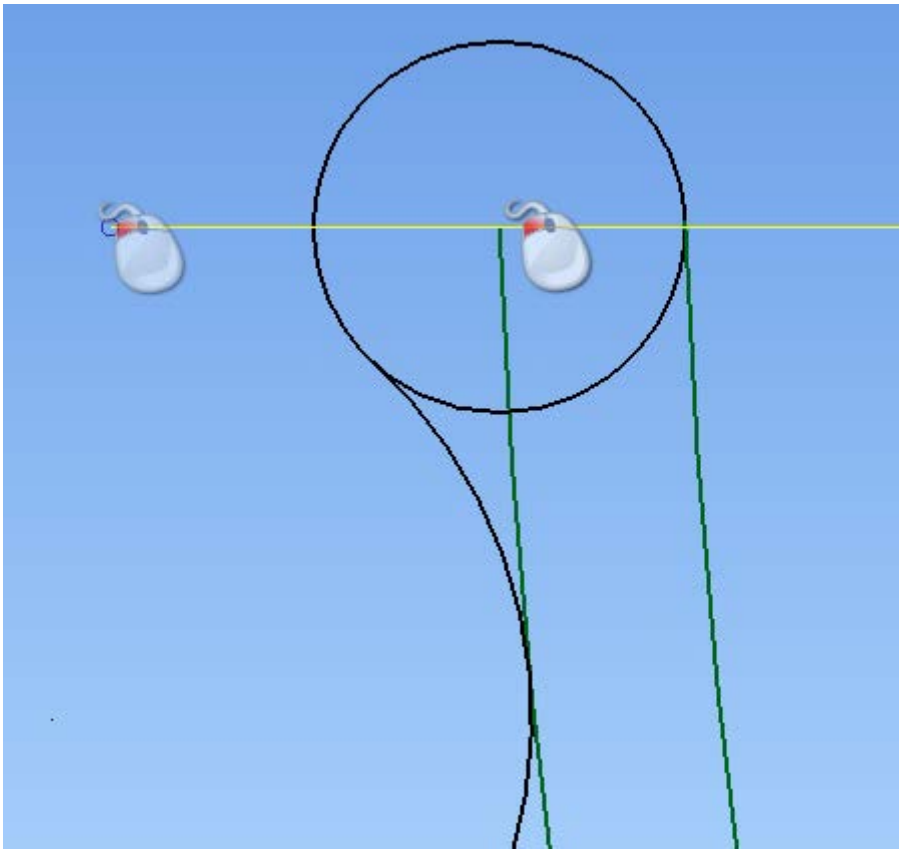
Notice that a small arrow is drawn on the curve. This is the direction the curve will move in if you enter a *positive* distance. If the arrow points down (or out) as in this case, you must enter -5 in the Offset Distance box.



If the arrow points up (or in) enter a positive value.

- Select the line at the top and change its length

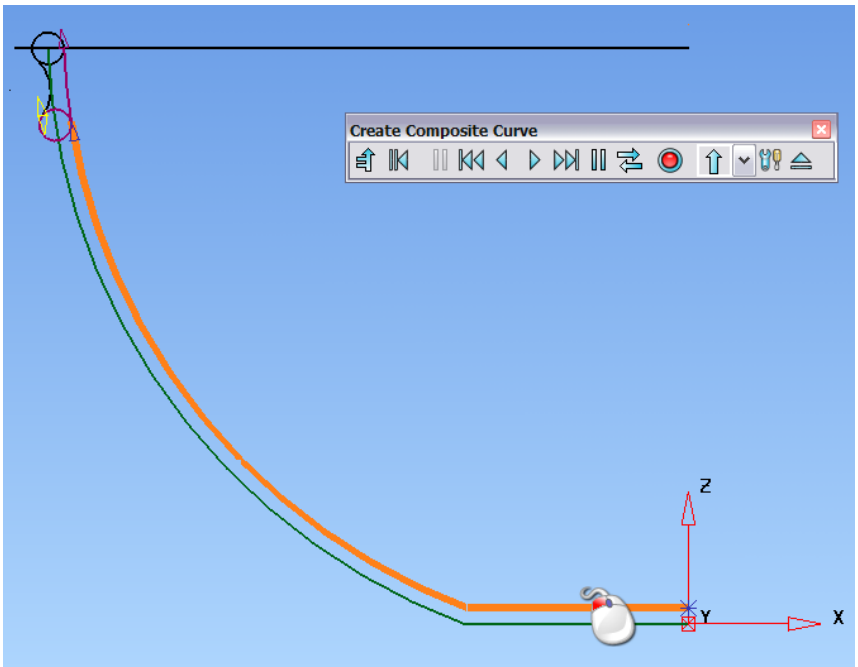
(Zoom in on the line at the top of the bowl. Grab the end of the line, and *drag* a little to the left to lengthen the line so it passes through the circle)



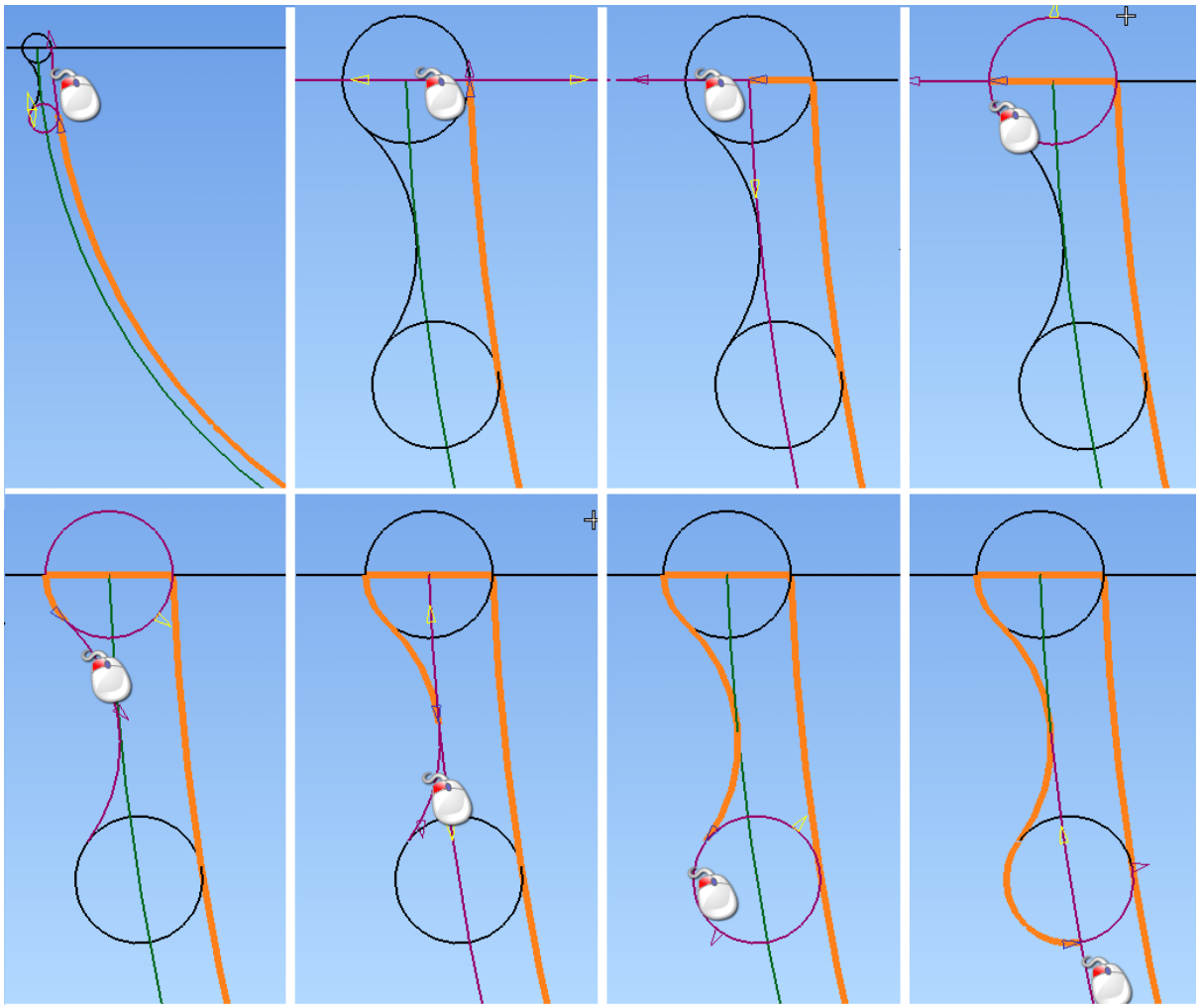
Take care not to change the *angle* of the line, but only its length.

Create a composite curve

Start the curve at the bottom the inner curve.



Trace the curve step by step, as in the following images.

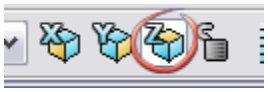


Save the composite

Press to exit the composite curve creator.

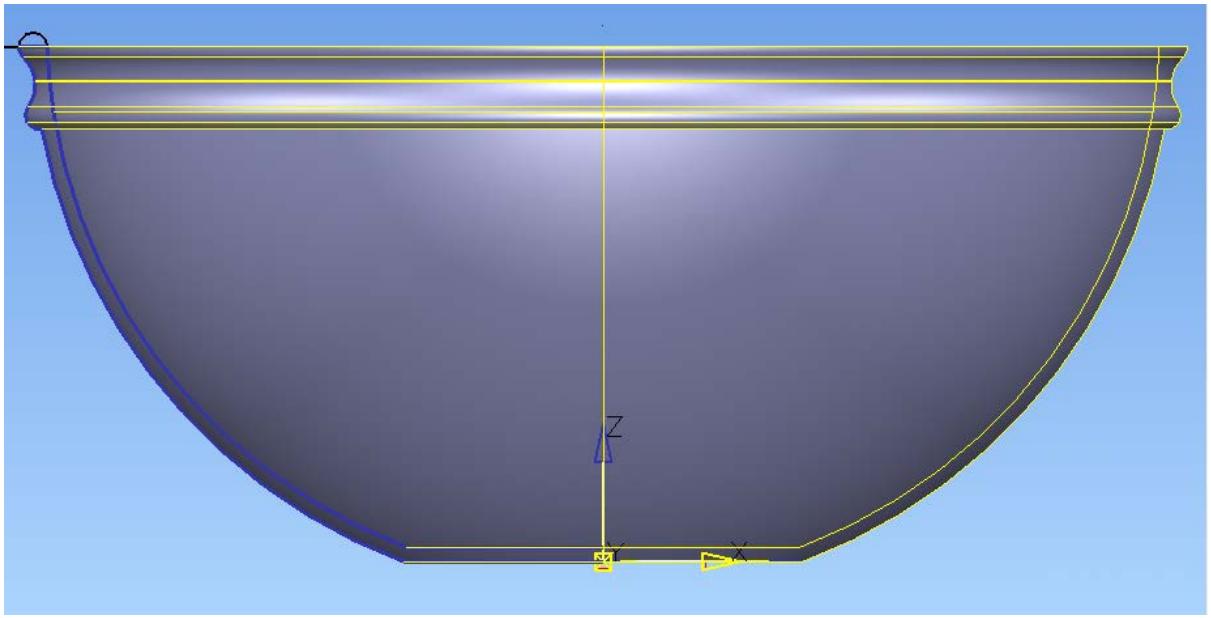
Making the bowl.

The bowl is created as a simple solid of revolution.

Set the working axis to Z 

Open the solid toolbar

Create the solid of revolution



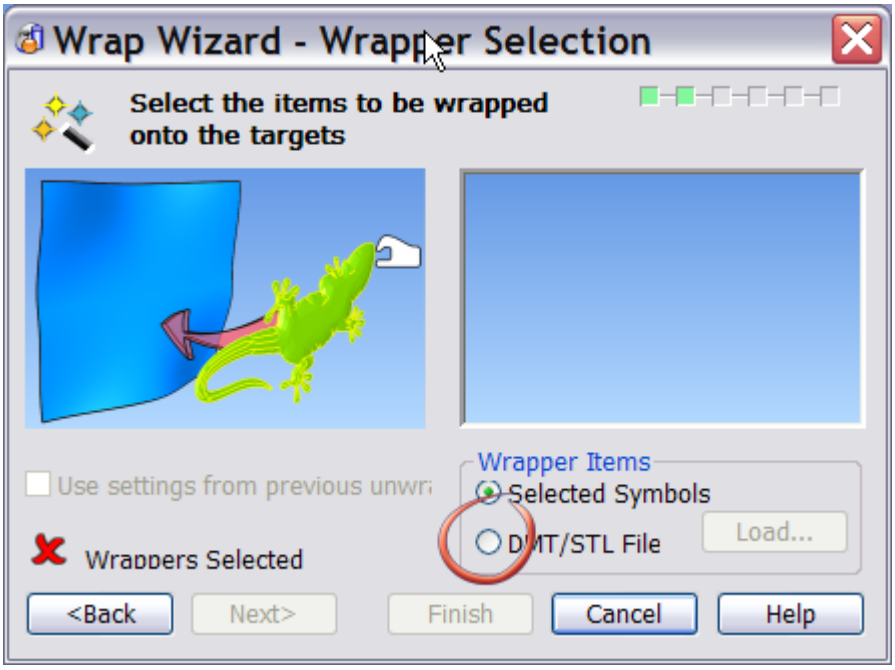
Decorating the Bowl

The decoration for the bowl is an ArtCAM relief model that has already been triangulated and exported in dmt (Delcam Machining Triangle) format.

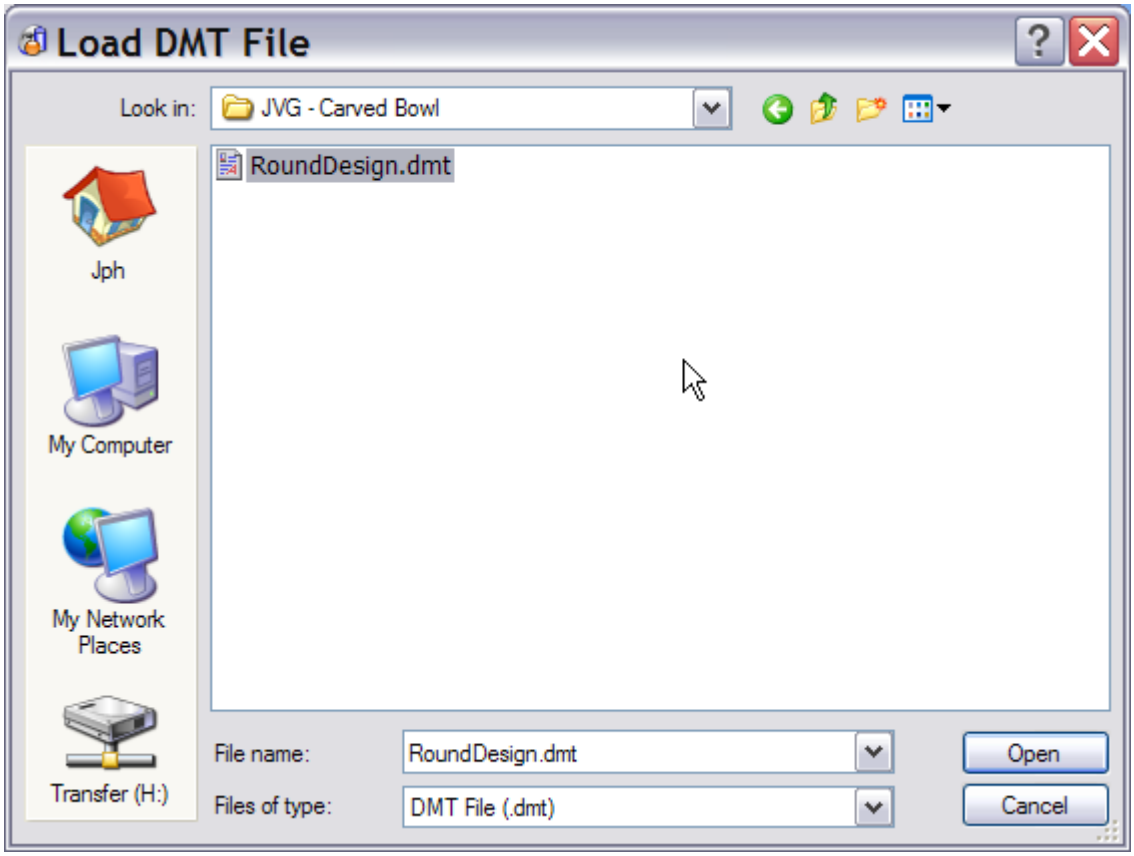
View the model from underneath (Crtl + 0)

Open the Solid Feature toolbar

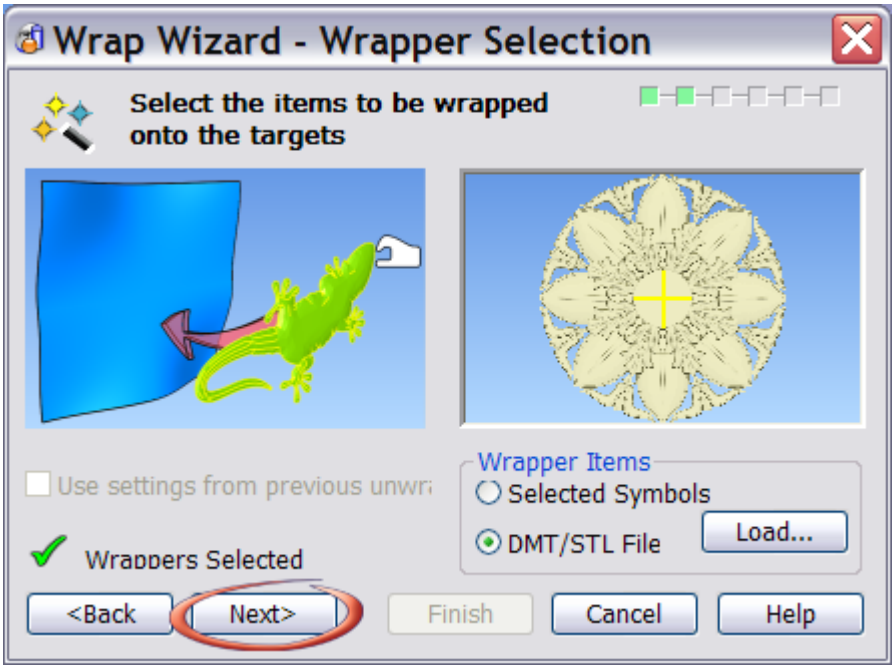
Open the Wrapping form



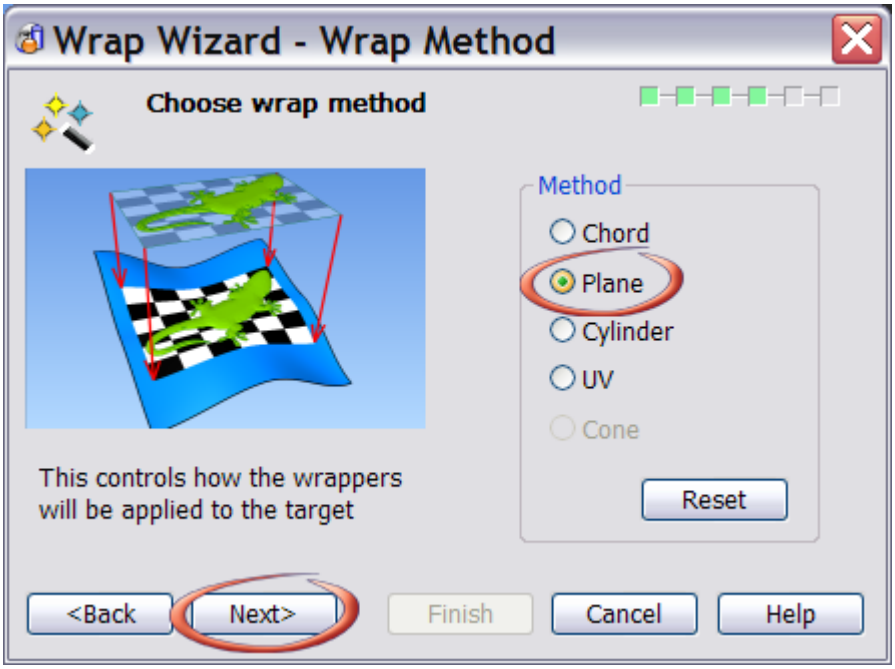
Click the DMT/STL File button to open the file selector and browse to the file- ***RoundDesign.dmt***



Click ‘Open’.



Choose the ‘Plane’ wrapping method, and then click ‘Next’.



IMPORTANT NOTE: The correct choice of wrapping method is absolutely essential to getting a good quality result. For a full understanding of the different methods available you are advised to read the relevant manual section. The following short descriptions will hopefully give you an idea of what is going on.

Chord. This method does a true ‘wrap’ with the minimum possible distortion. Imagine taking a flat sticky label and attaching it to a curved surface. Because the label is not very ‘stretchy’ there is a limit to how highly curved the shape can be

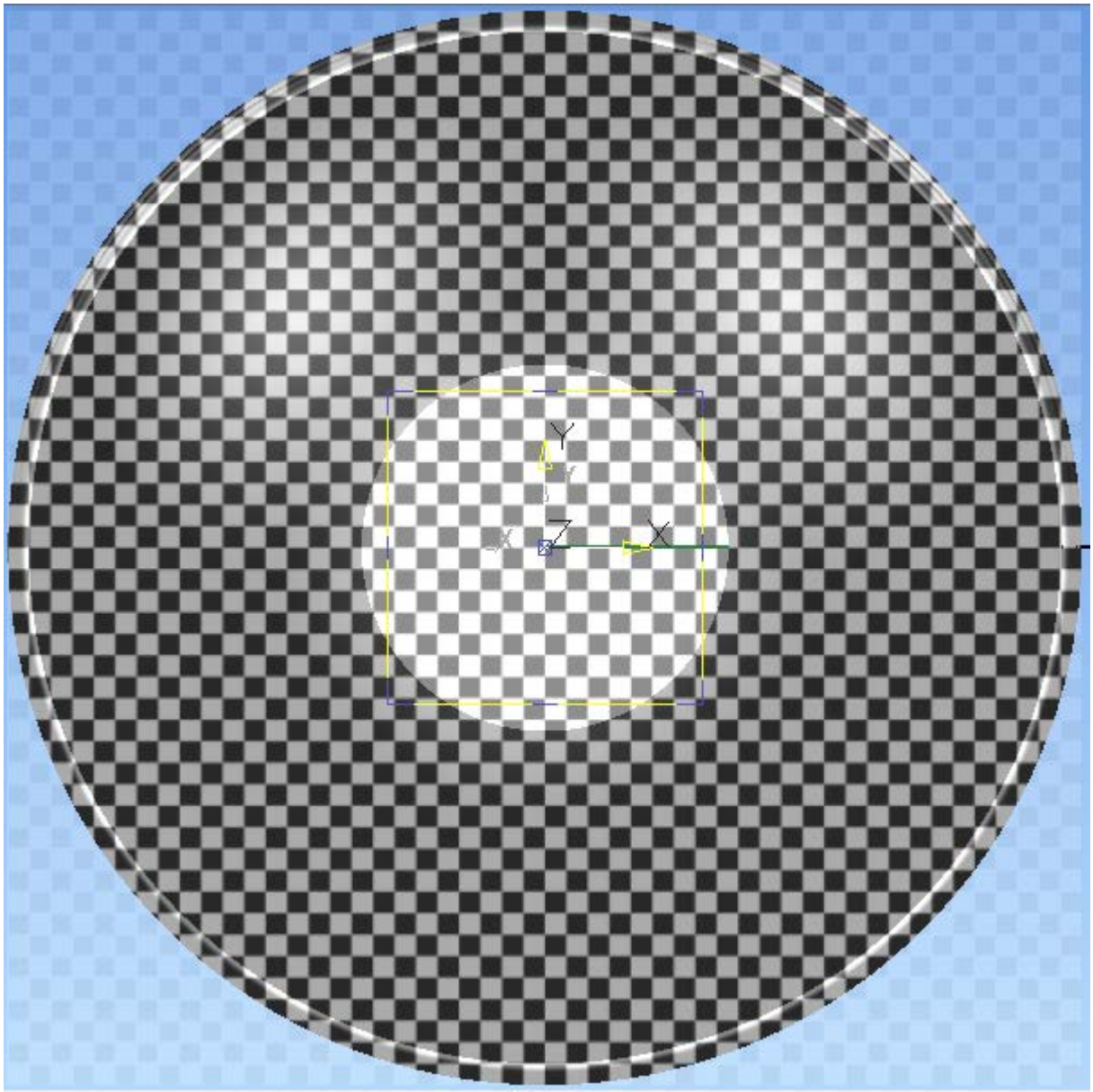
before the edges of the label begin to ripple. Also note that method *will not* work on closed objects, as in order to calculate the 'wrap map' Delcam Designer must first *unwrap* the target to a flat condition.

Plane. This method works by projecting the label onto the target along the current view direction. This method works best for shapes where curvature is not too extreme.

Cylinder. This method constructs a cylinder that contains the target surfaces/solid, and projects the label *inwards* from the cylinder onto the target(s). This method works very well for parts such as bottles and rings.

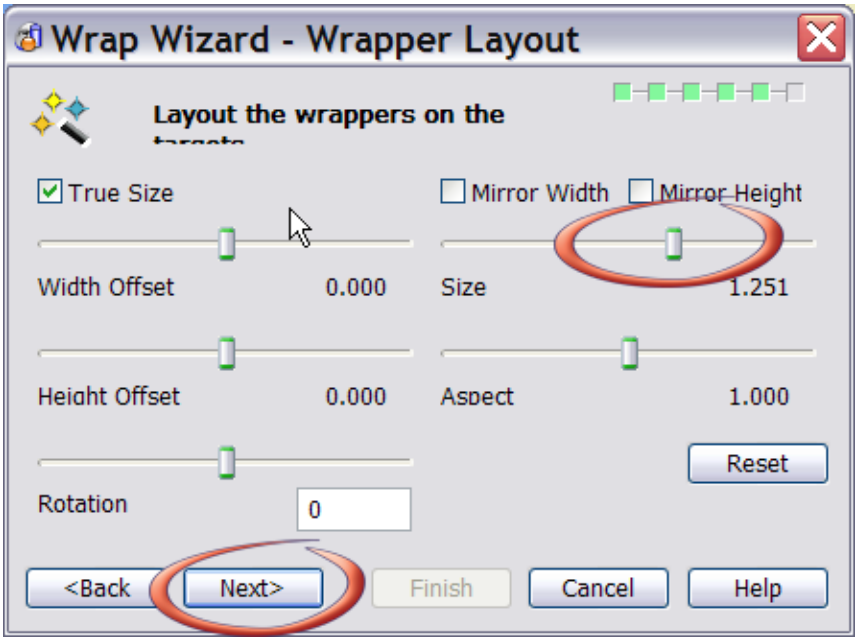
UV. The UV method constructs its wrap map according to the layout of the surface's Lateral and Longitudinal curves. Results will therefore be best when the spacing of these curves (and their relative lengths) is fairly uniform.

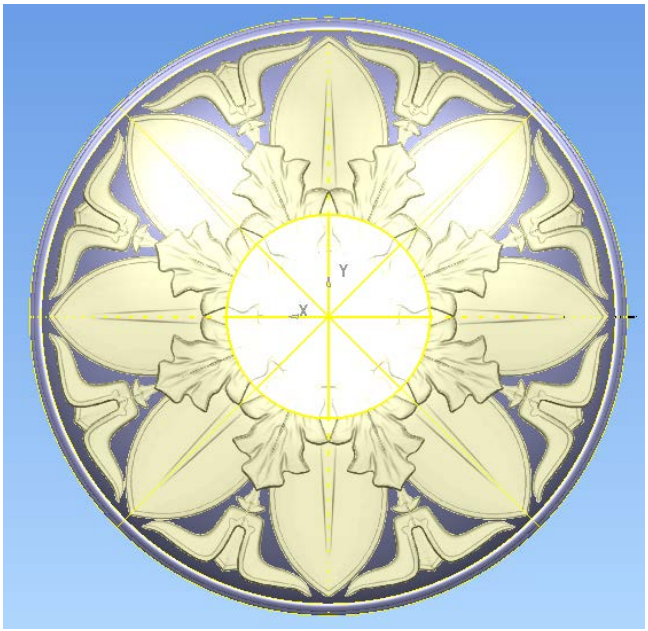
Cone. A perfect cone can be unwrapped into a segment of a circle with no distortion. Conical unwrappping projects the geometry to be unwrapped onto a cone and then unwraps the cone, and the geometry with it. Create a cone primitive that approximates the geometry to be unwrapped, select the geometry and the cone and enter the wrap wizard. The amount of distortion in the result depends on how well the geometry approximates a cone.

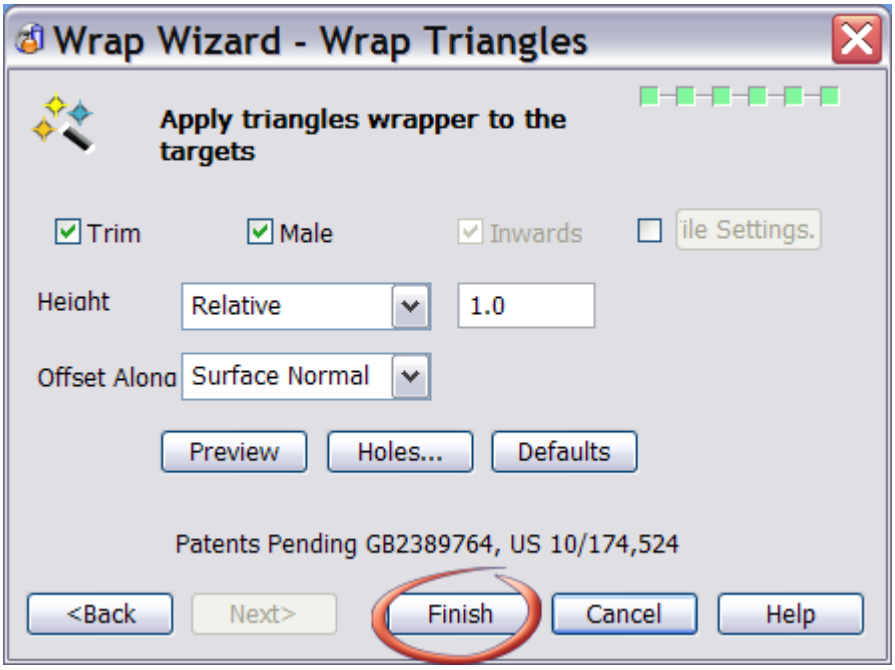


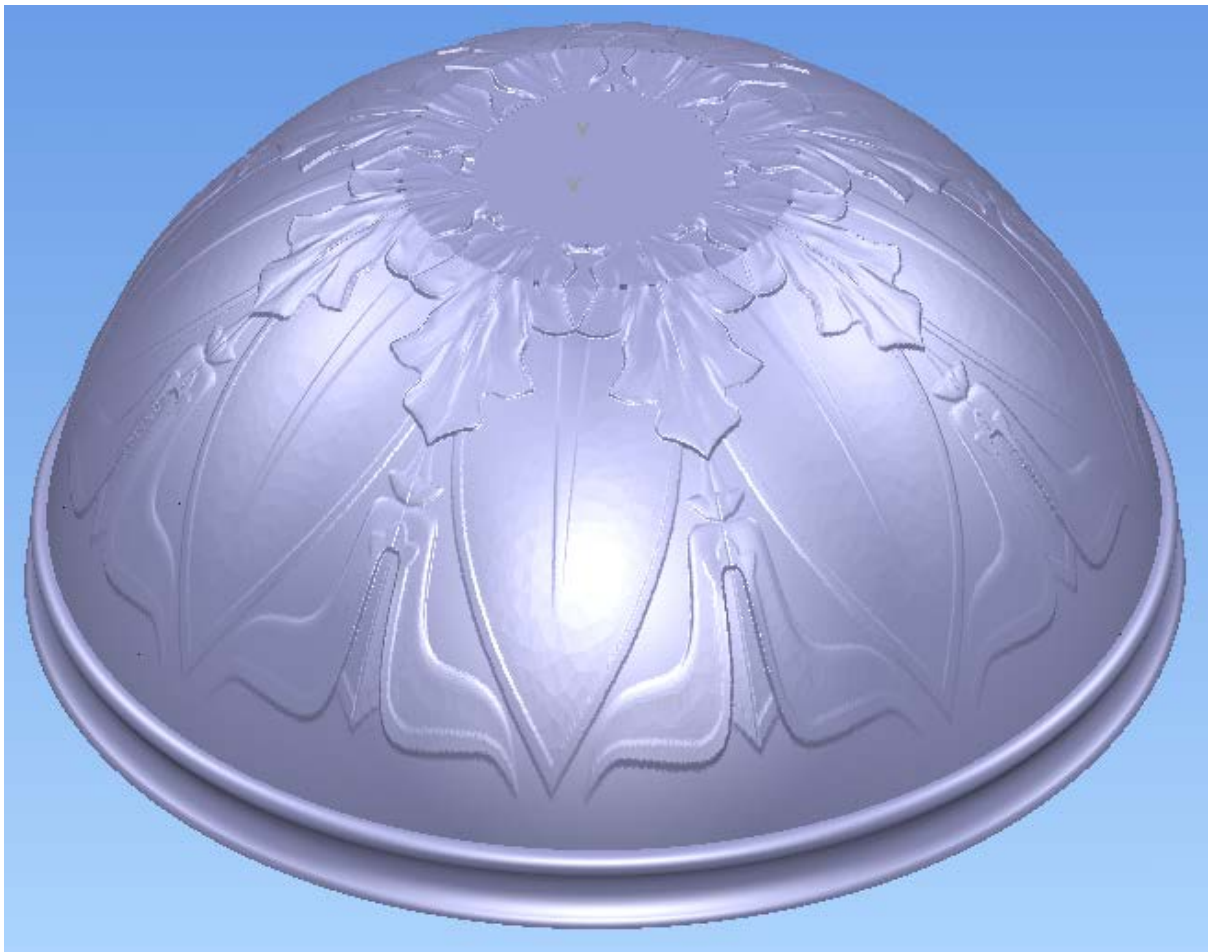
The wrap preview image shows how successful the wrap will be. If the preview is nicely uniform (as in this case) the wrap should also be even. If the preview is badly distorted, that distortion will be present in the wrapped relief as well, and you should go Back and choose different options.

Use the sliders to adjust the size of the relief until it looks as you want.

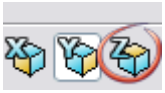

Wrap Wizard - Wrapper Layout
Layout the wrappers on the
True Size
Mirror Width
Mirror Height
Width Offset
0.000
Size
1.251
Height Offset
0.000
Aspect
1.000
Reset
Rotation
0
<Back
Next>
Finish
Cancel
Help



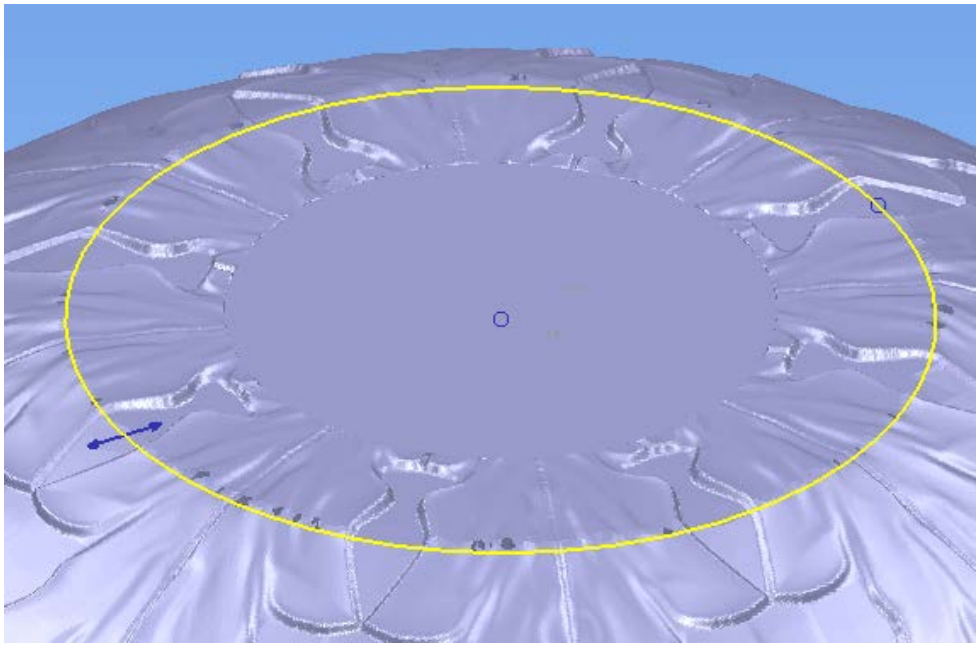

Wrap Wizard - Wrap Triangles
Apply triangles wrapper to the targets
Trim
Male
Inwards
ile Settings.
Height
Relative
1.0
Offset Along
Surface Normal
Preview
Holes...
Defaults
Patents Pending GB2389764, US 10/174,524
<Back
Next>
Finish
Cancel
Help



Adding the Base

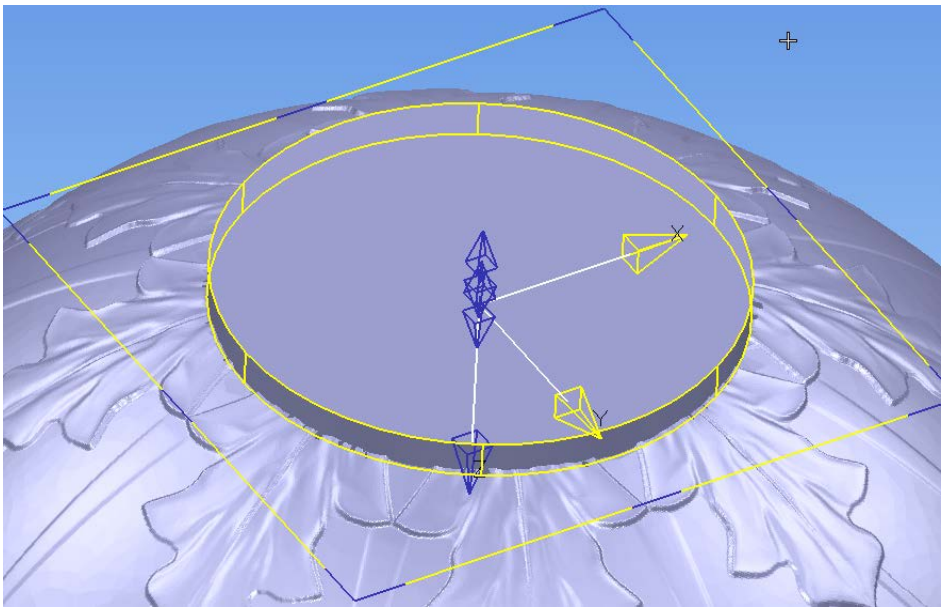
Set the working axis to Z. 

Create a circle of radius 70mm with its centre at *0,0,0*
Create a second circle on the same centre with a radius of 60mm

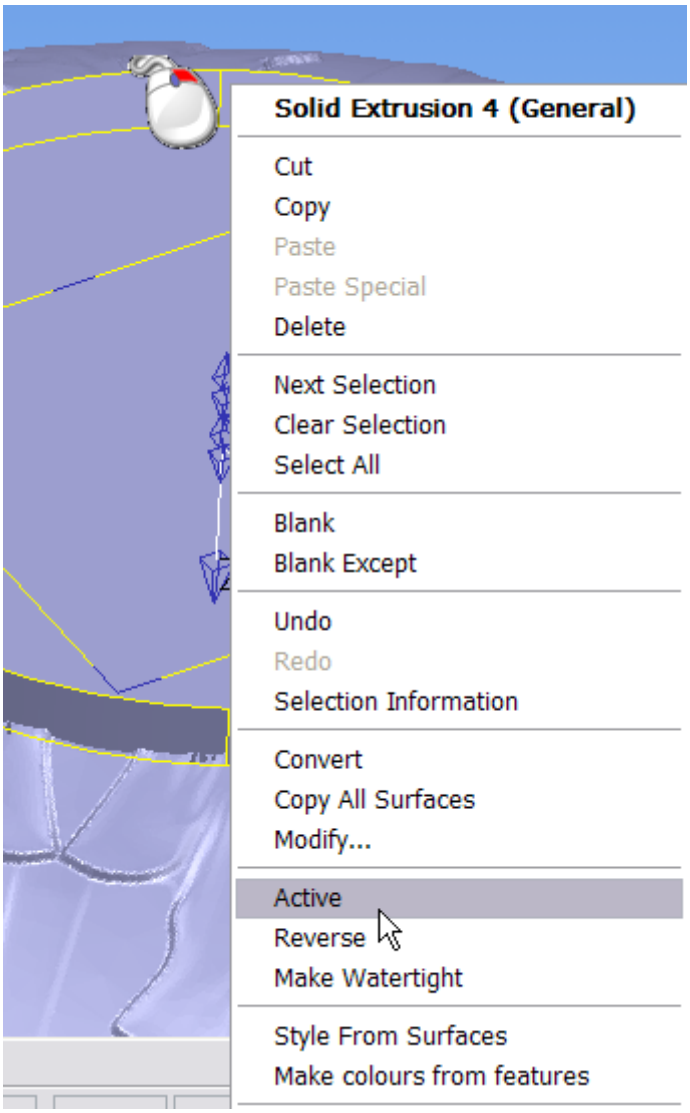


Select the 70mm circle, and create a solid extrusion.

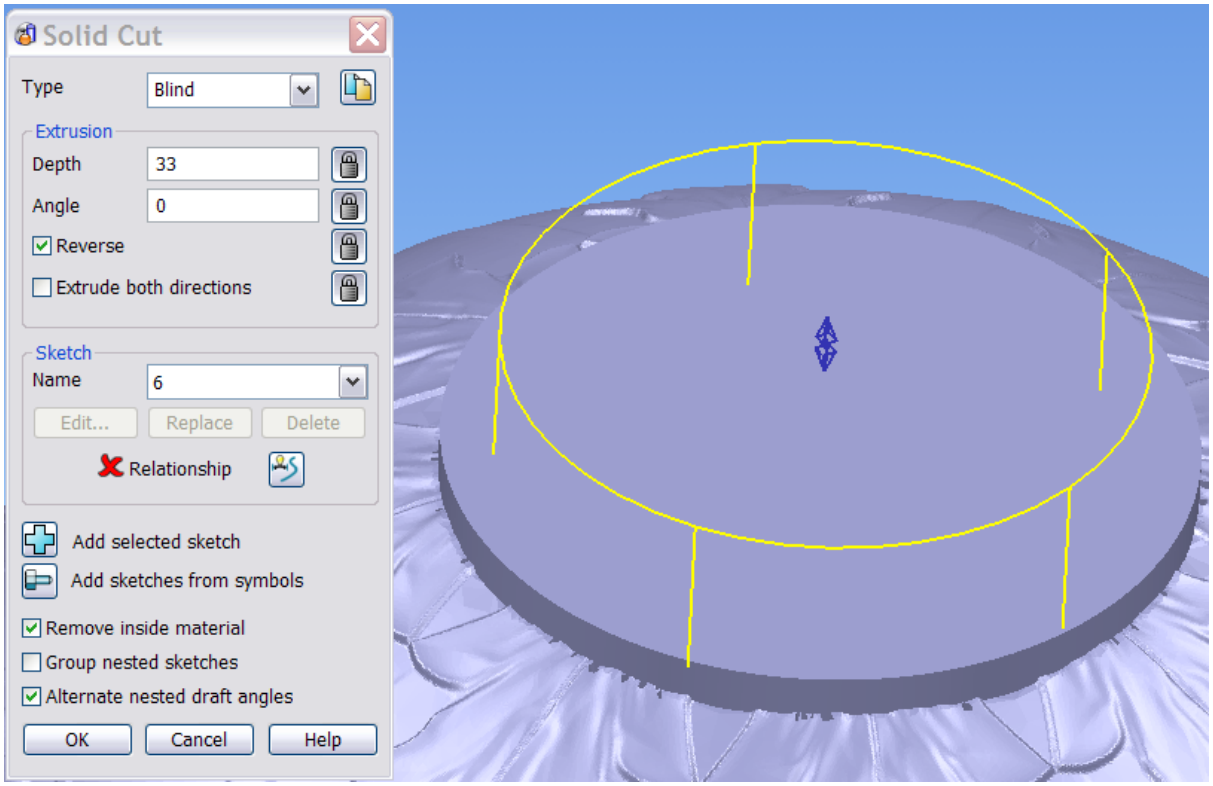
Set the extrusion so it looks as below.



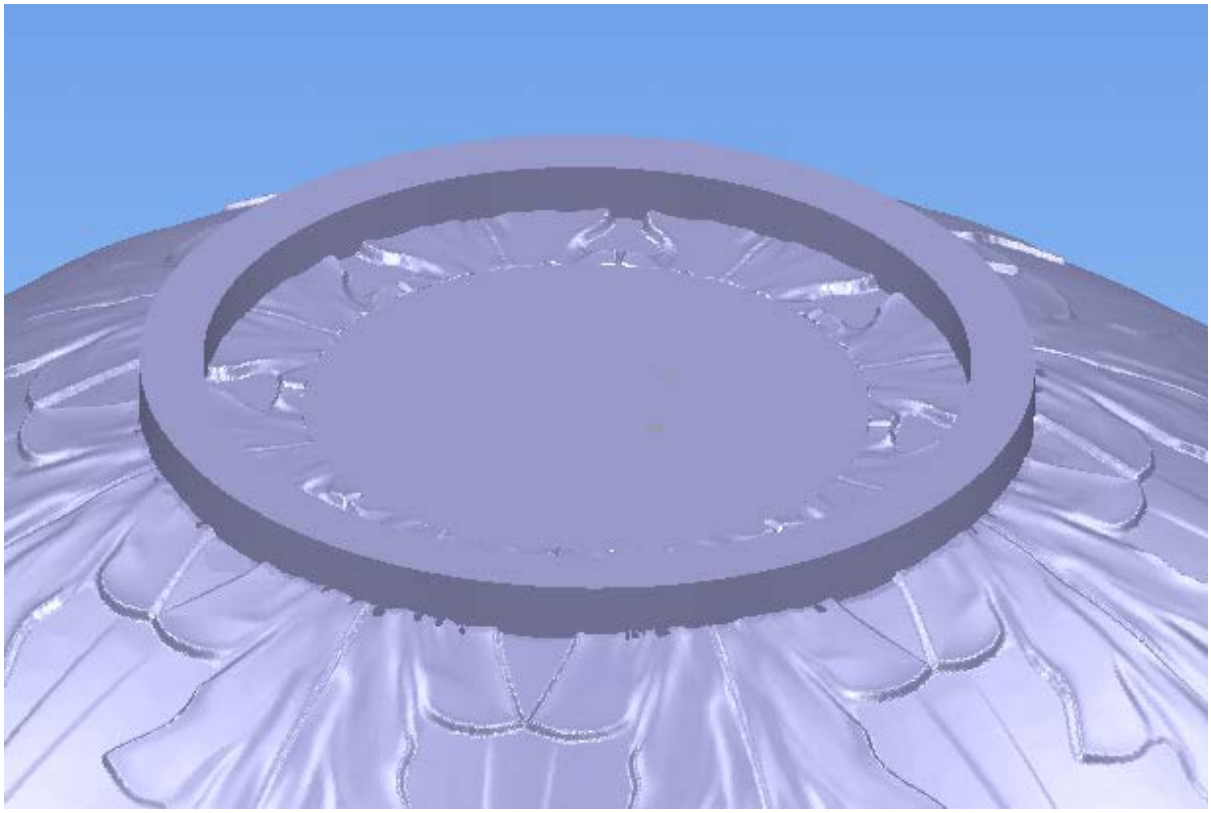
Make sure the new solid is Active.

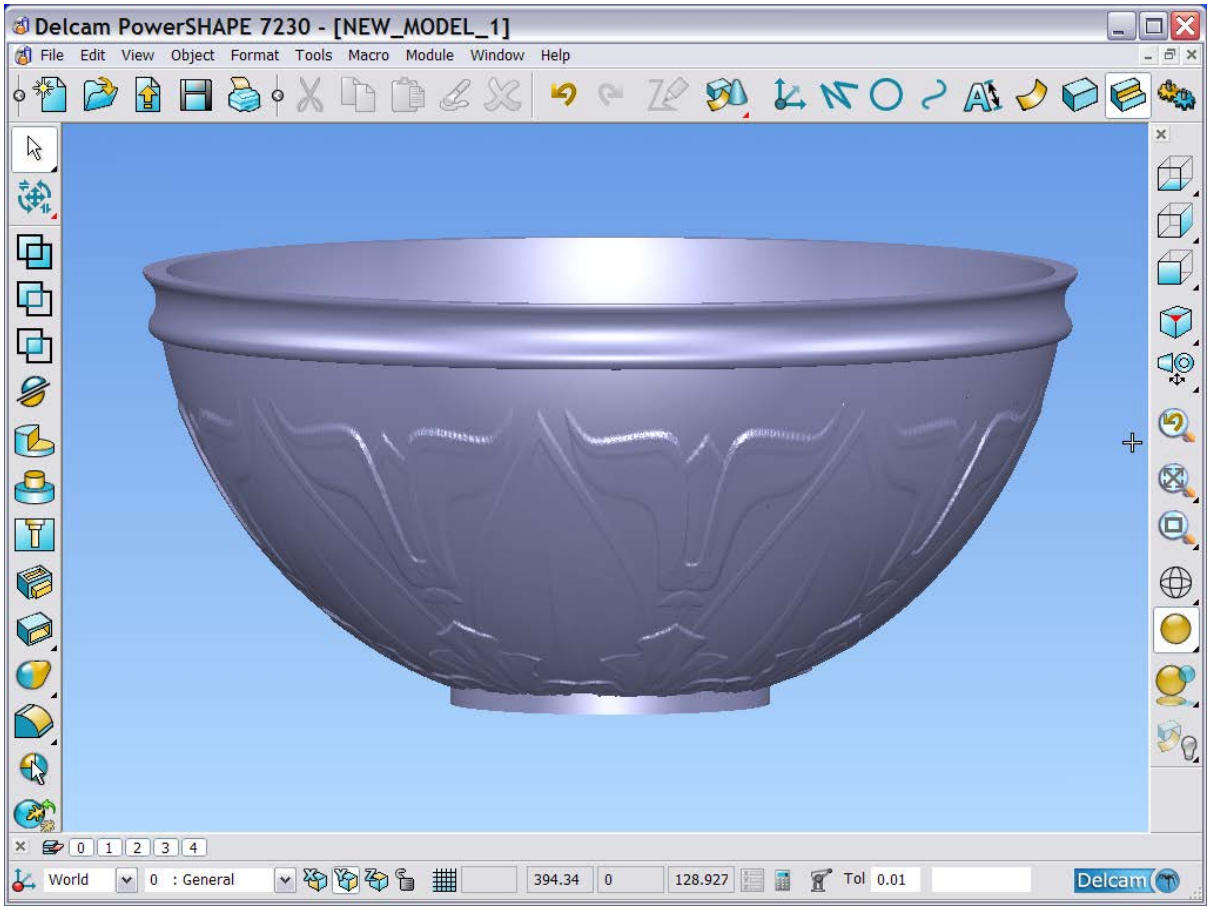


Select the 60mm circle, and use the Solid Cut feature to remove it from the active solid.



Make sure the cut feature will pass completely through the first solid, then click OK.





Save the model in **D:\users\Training\Coursework\Delcam Designer Jobs** as **Carved Bowl.psmodel**

Exercise – Modelling a Corbel

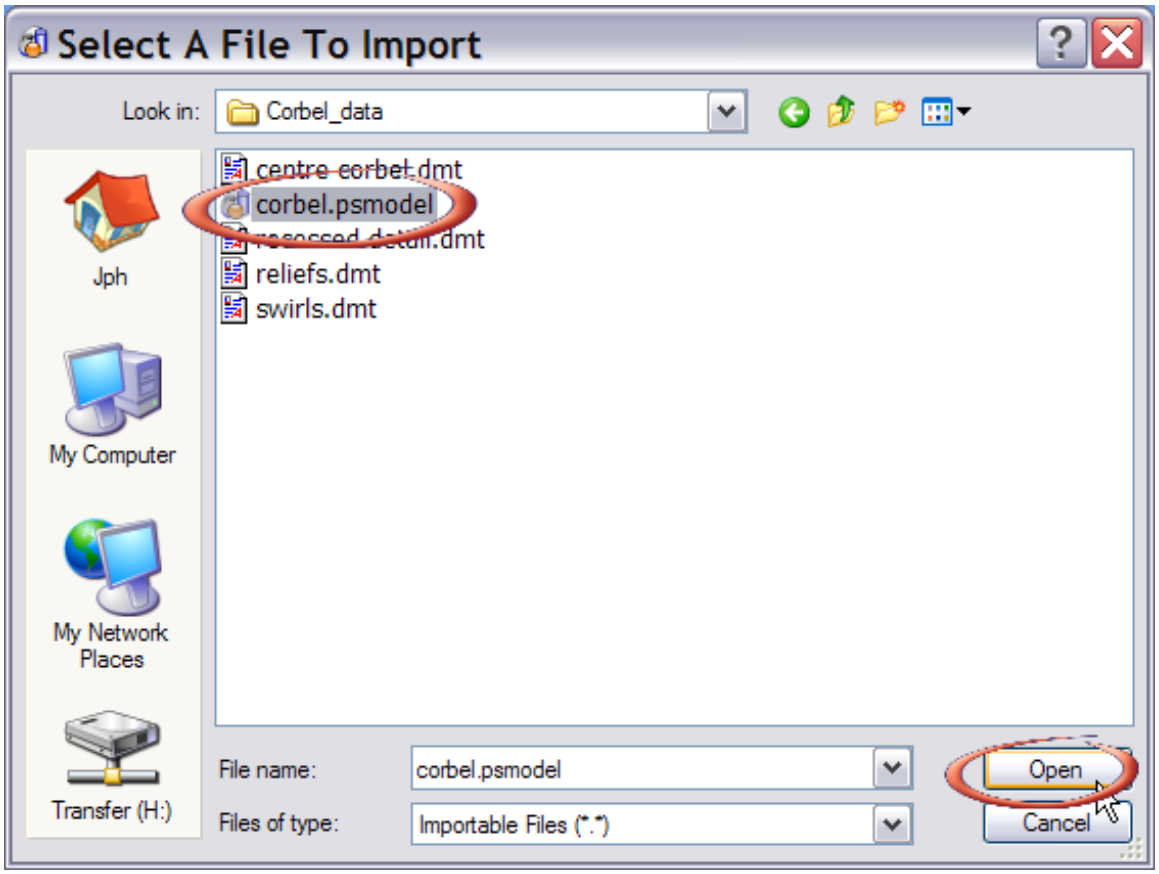
Introduction

The aim of this exercise is to design the corbel [*a support for an arch or similar heavy structure, which sticks out of a wall and is usually made of stone or brick*] shown in the picture above.

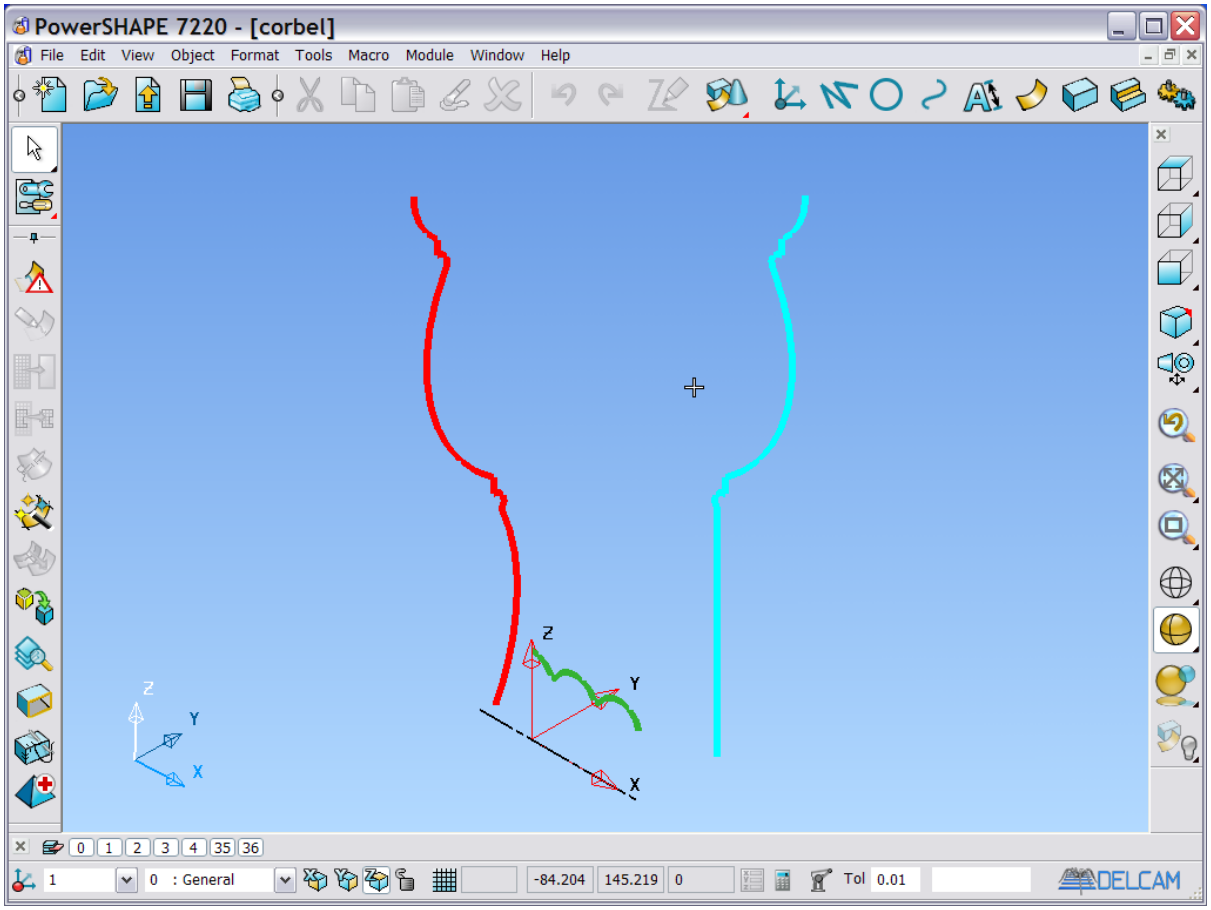
Beginning with a Delcam Designer model that contains the key wireframe curves for the design, simple surfaces will be created. These will be trimmed together (to remove unwanted material) to complete the shape. Finally, decorative reliefs will be wrapped onto the surfaces to finish the design.

Stage 1 - Creating the Main Surfaces

Click to import the Model data.



Browse to the model file and click Open.



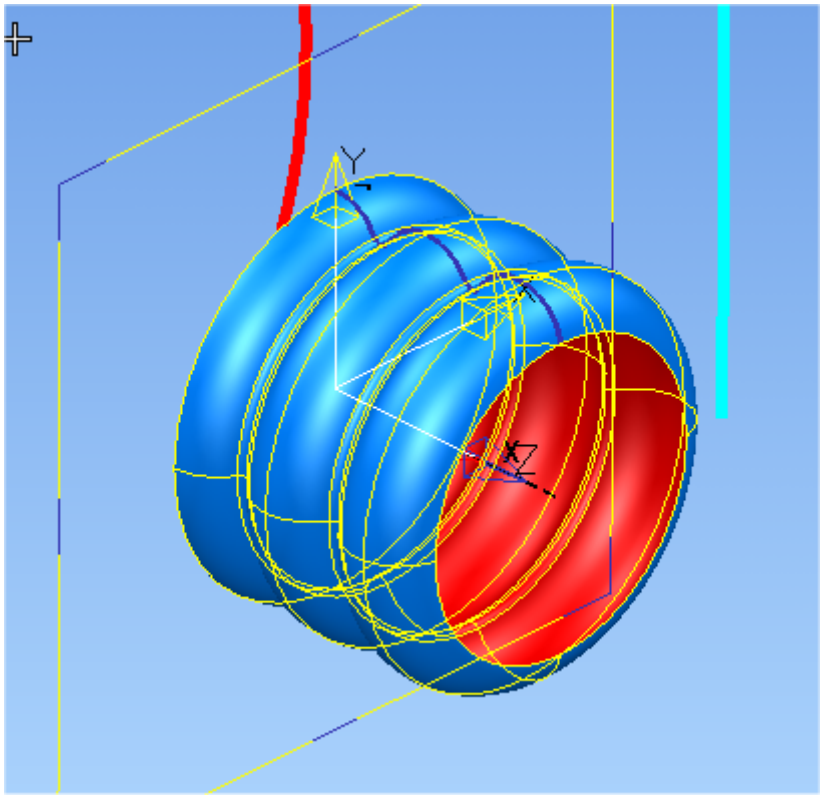
Rotate the view so as looks like the image above.

Click the X Axis icon (on the bottom toolbar) to set the Working direction.

Open the Create Surface Toolbar

Select the ***GREEN*** curve and click the Revolve icon

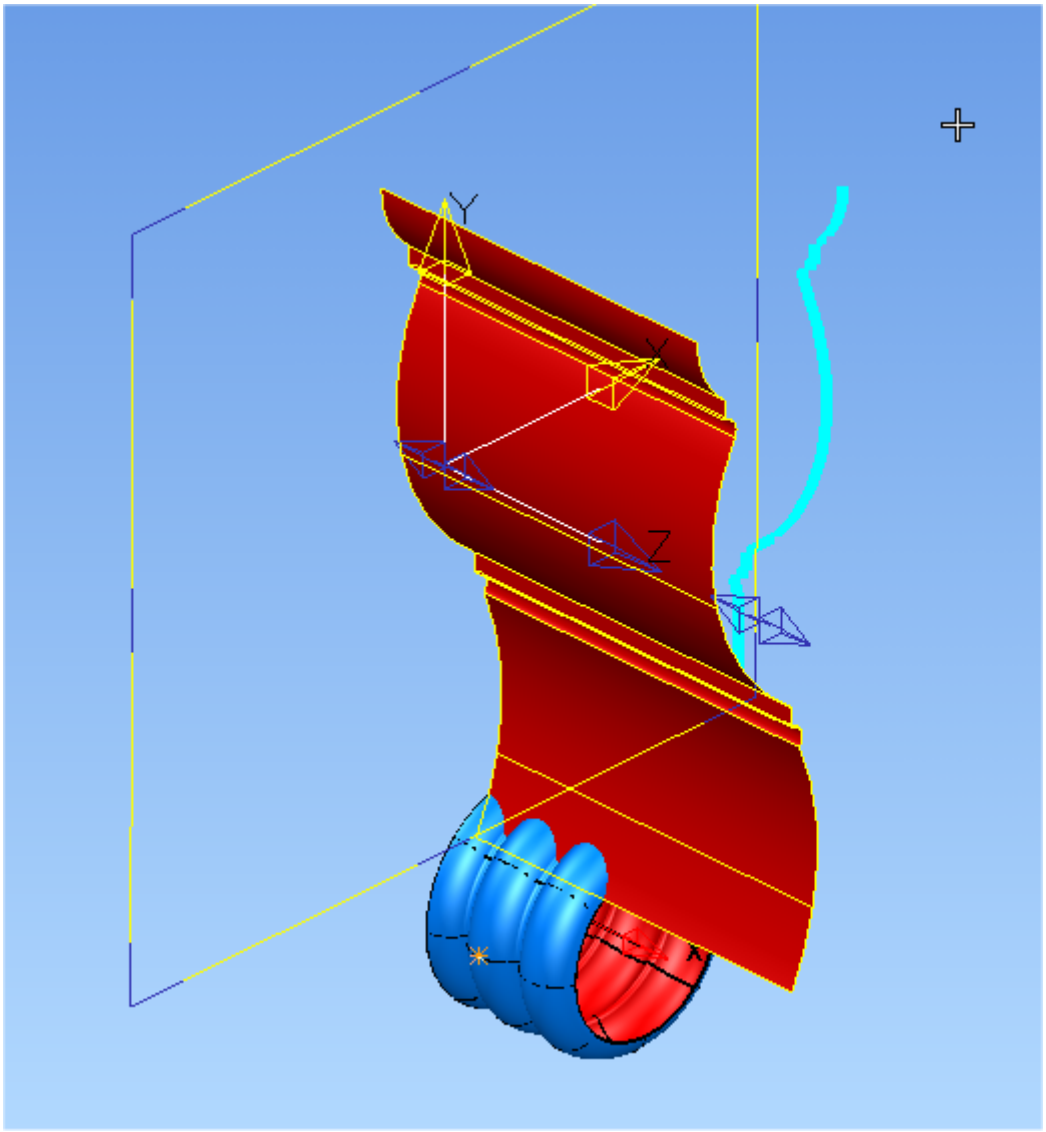
The surface will look like this –



Select the **RED** curve

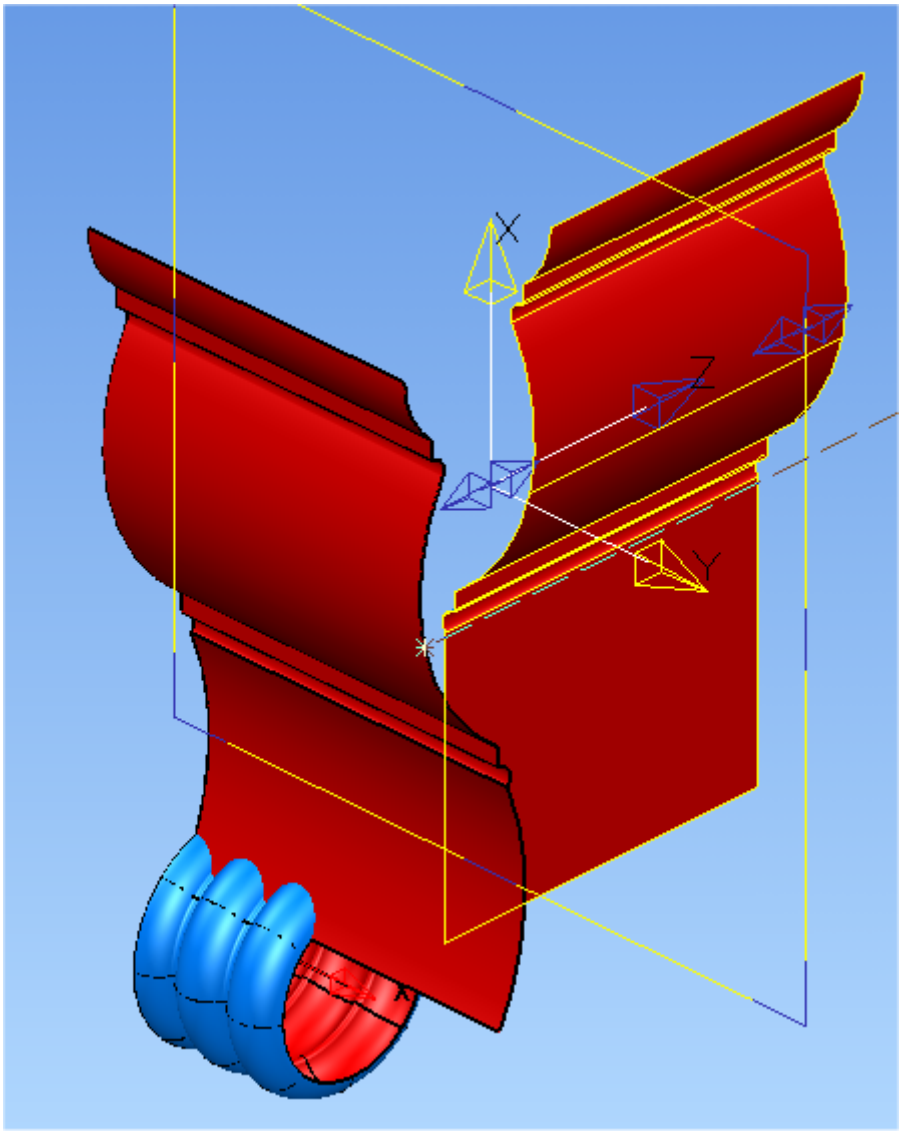
Click the Extrude icon

The extruded surface will look as below.

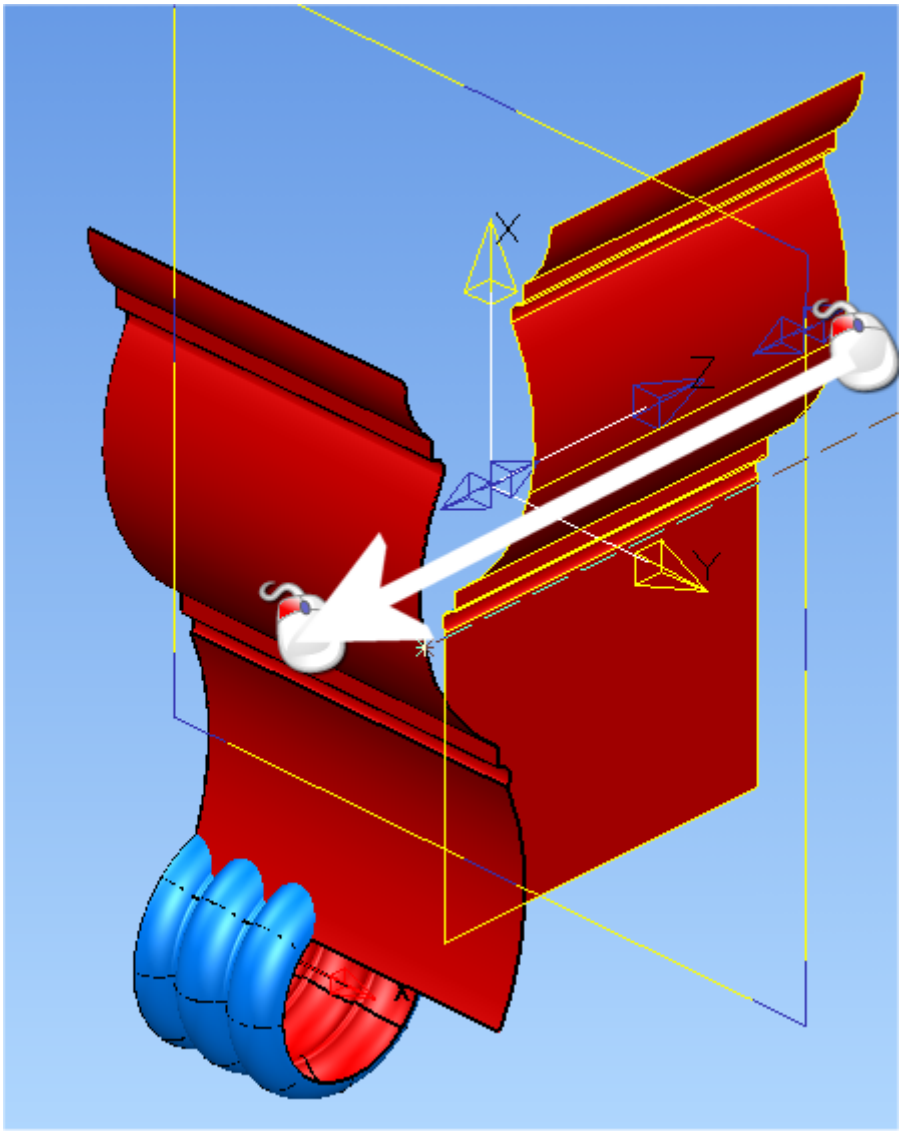


Set the working axis to Y

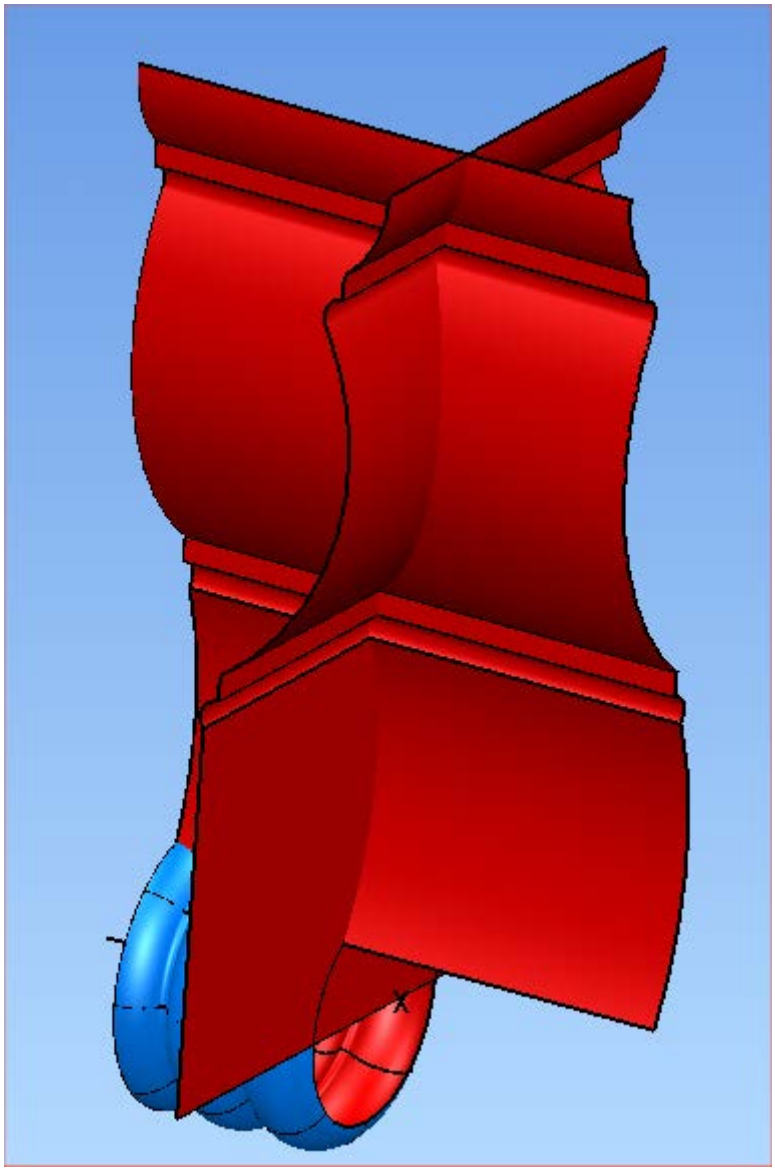
Select the **CYAN** curve and create a second Extrusion as below.



This extrusion is going the wrong way. Pick the **BLUE** arrow at the position shown, and *drag* the extrusion so it is correct.

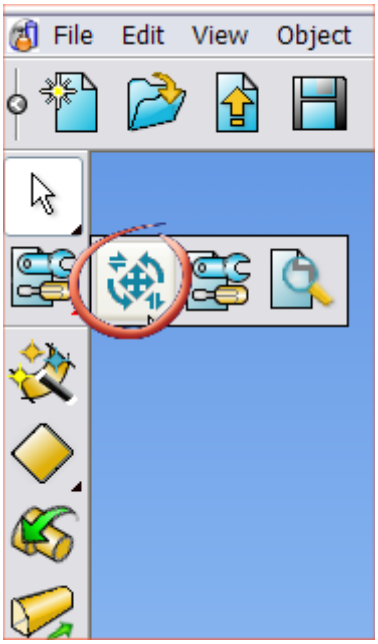


Make sure that the two extruded surface *fully cross each other* as shown below – adjusting the lengths of both as needed. The two surface *must* look as below before you go any further.

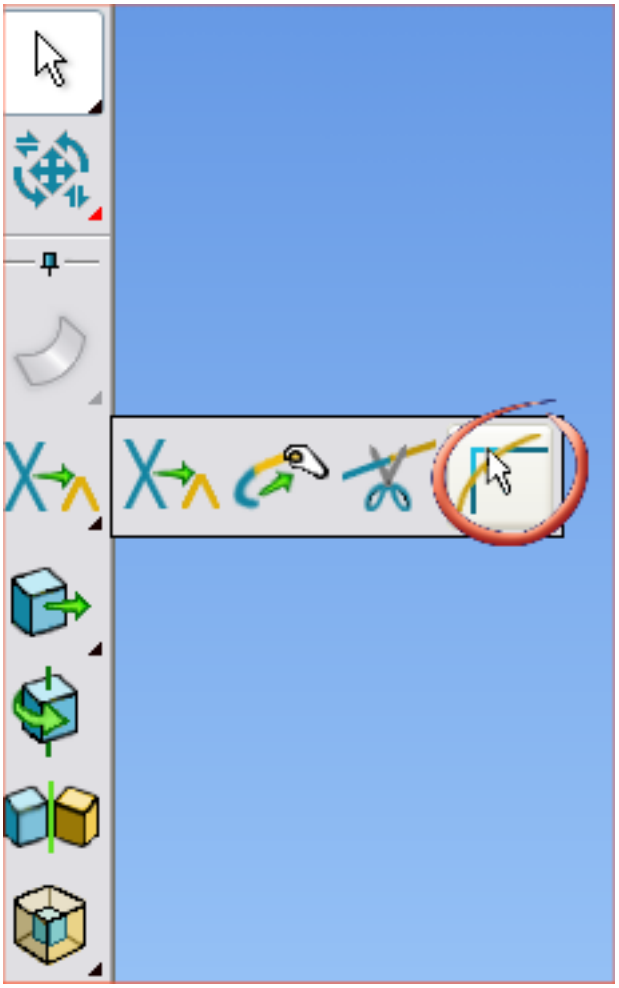


Stage 2 – Trimming the Sides

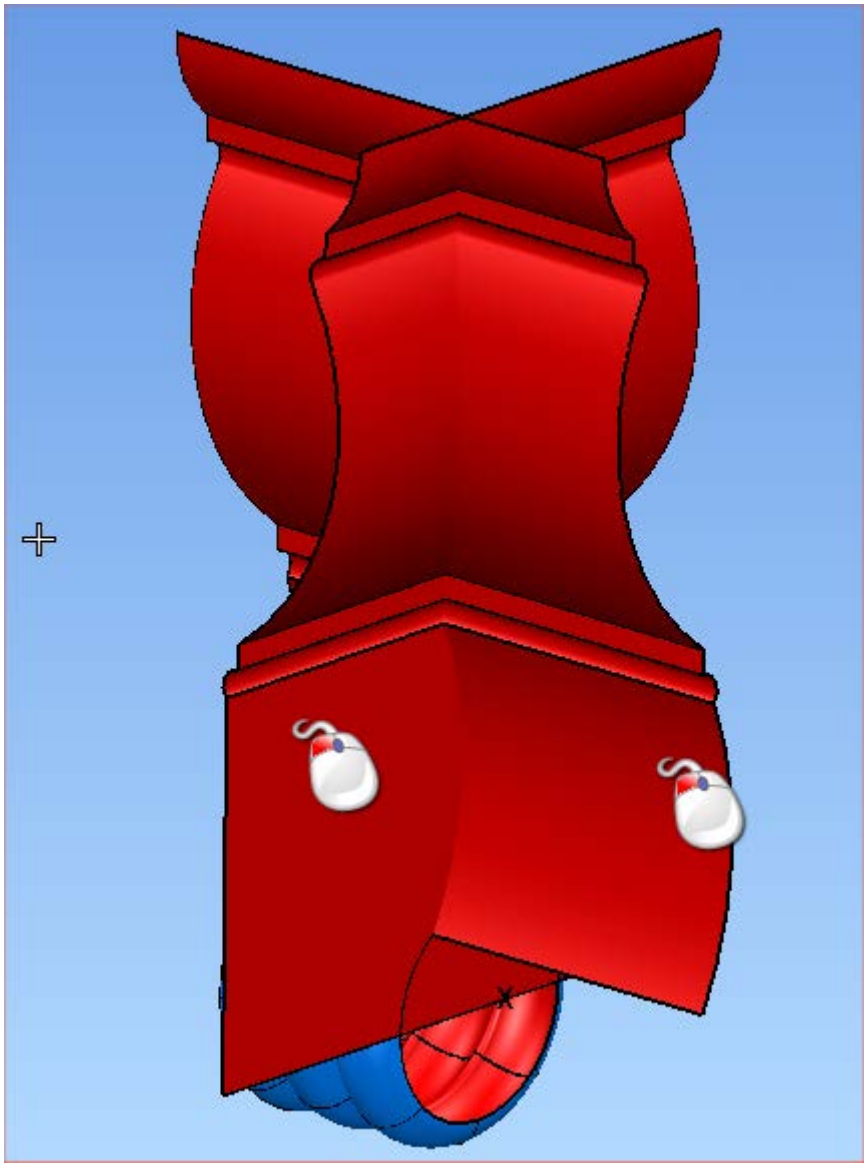
Open the General Edits toolbar –



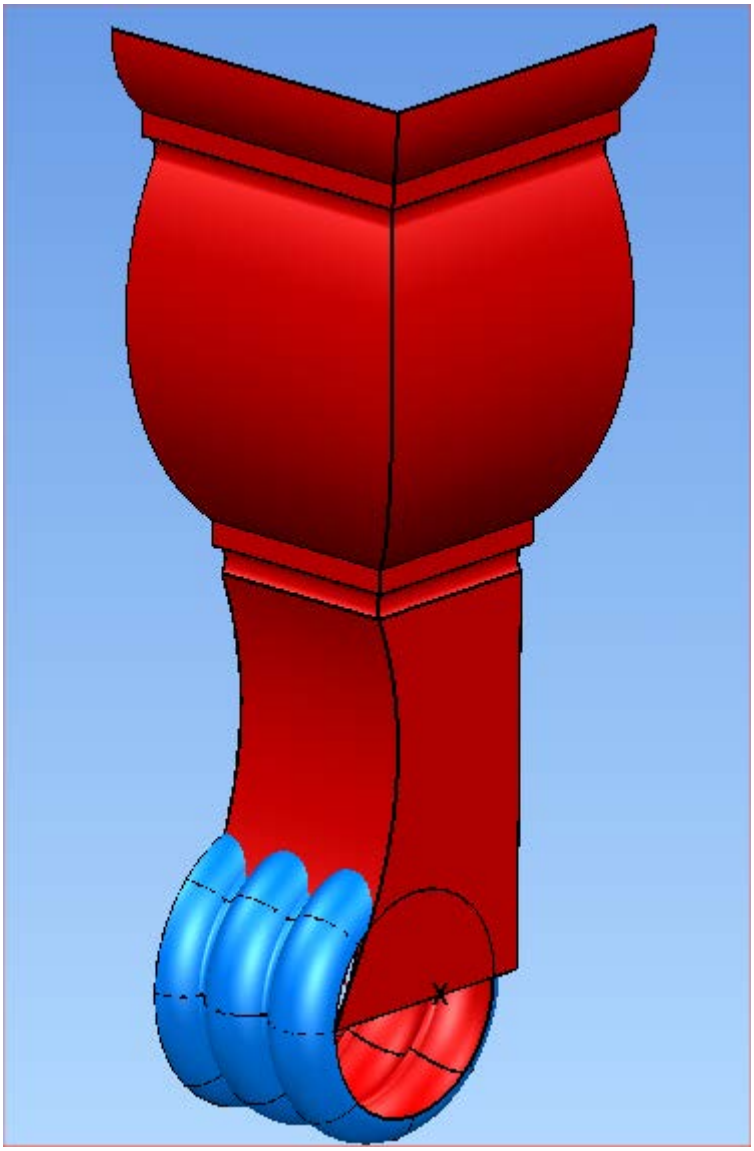
Choose the ‘Limit Selection’ icon



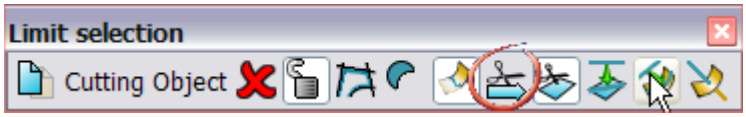
Select the two surface at the positions shown below (the order in which you make the selections does not matter).



The result should look as below –

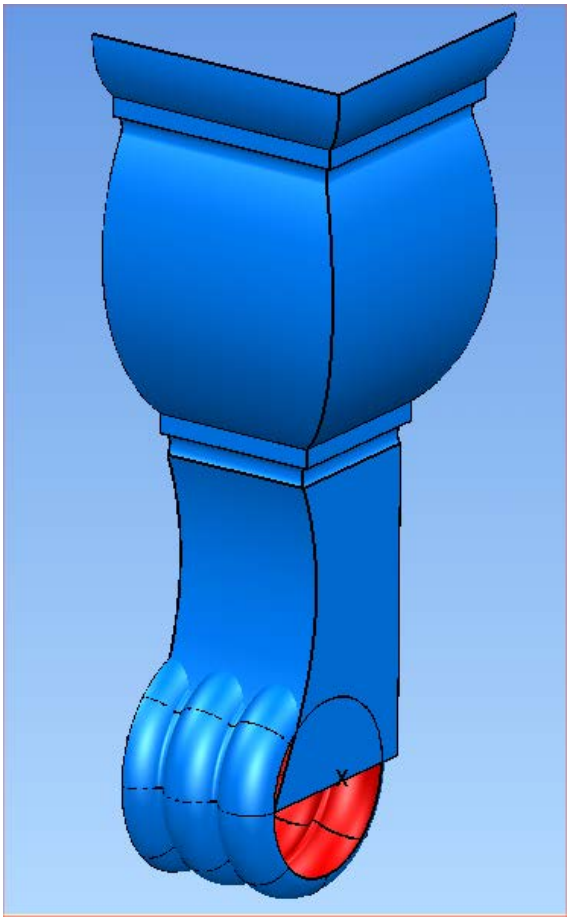


If not, simply press the ‘Next Solution’ button on the ‘Limit Selection’ form, until the desired result is achieved.

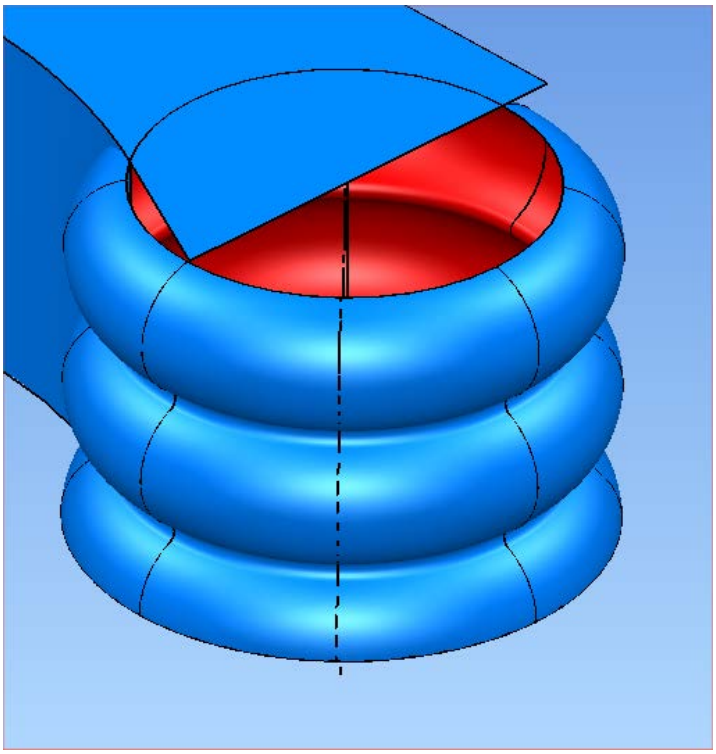


When the surfaces are correctly trimmed press the button to exit from Trimming.

Notice that these two surfaces are displayed in red. That is because they are currently incorrectly oriented; they are inside out. Select the two surfaces, click the Right Hand Mouse button and choose ‘Reverse’ from the pop-up menu.



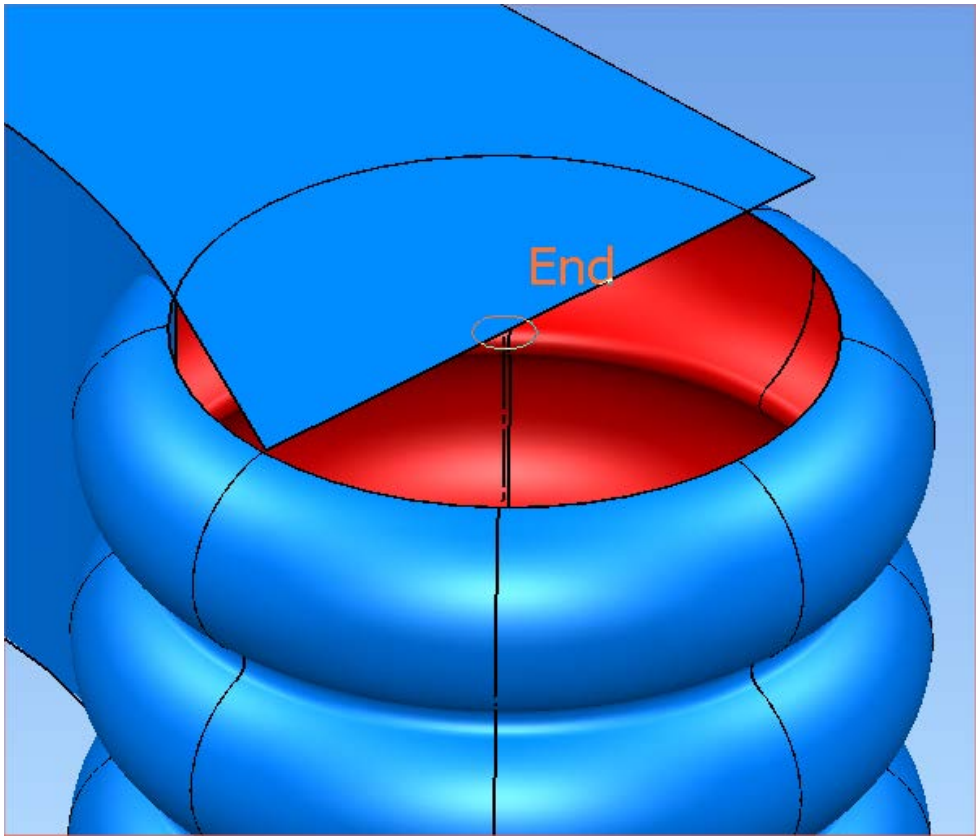
Rotate & zoom the view so it looks as below, and set the working axis to X



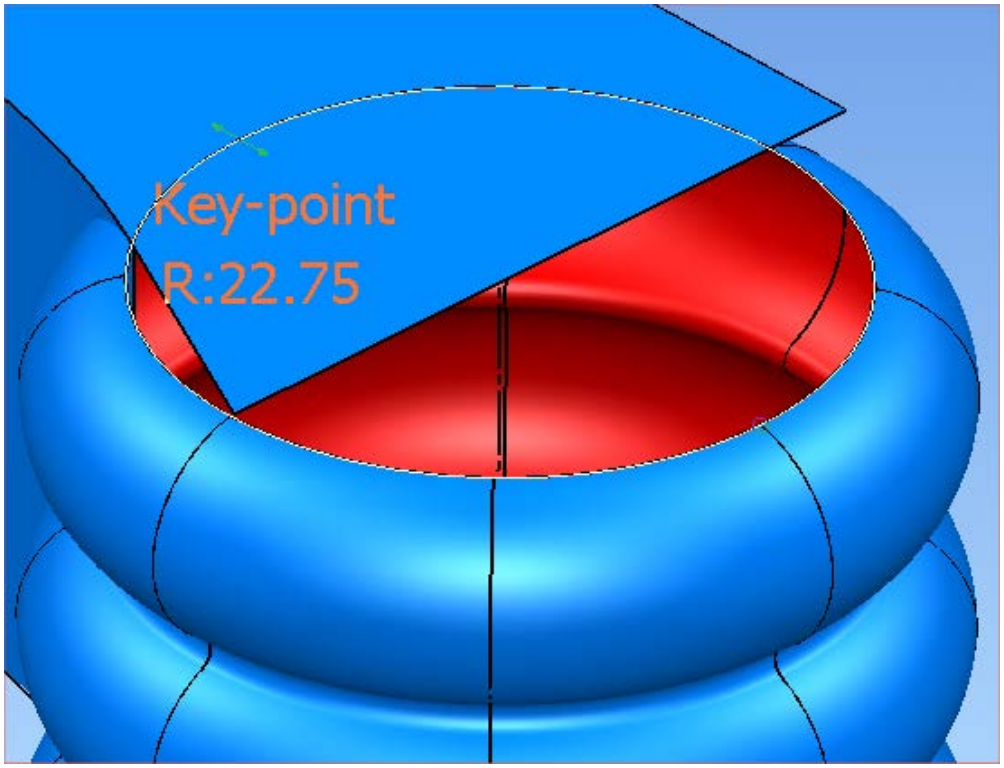
Open the Arc Creation Toolbar

Click on the ‘Full Arc (circle)’ icon

Snap the centre of the arc to the end of the centreline (DASHED).

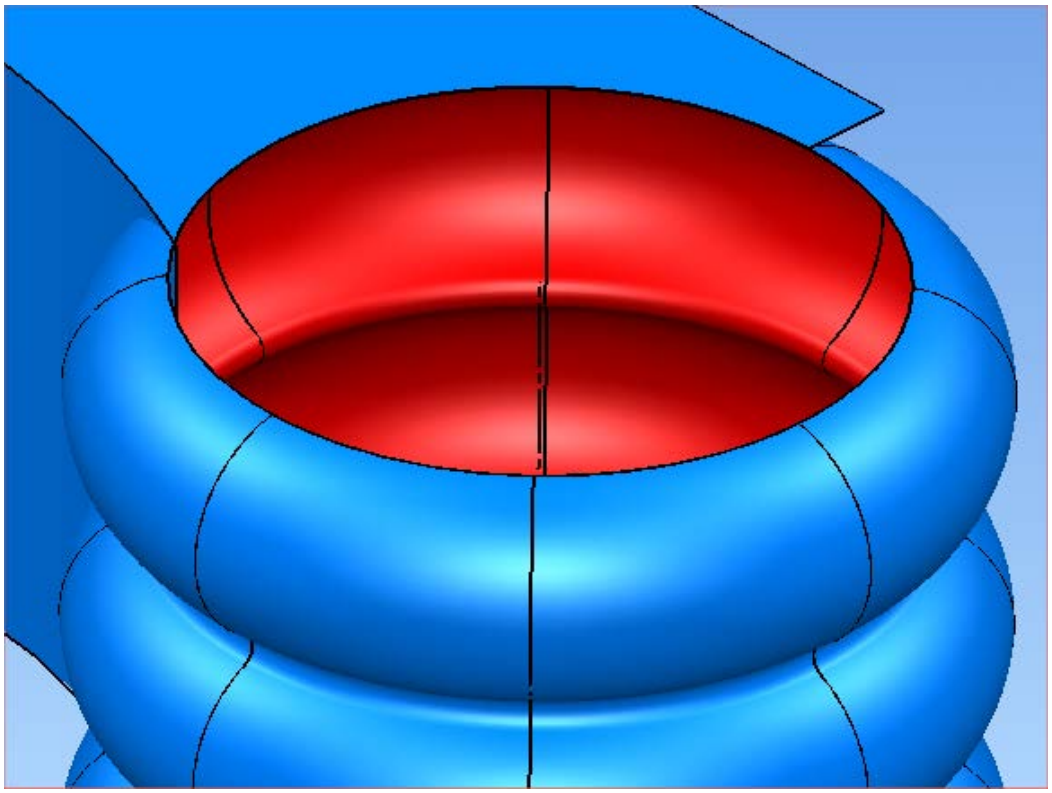


Drag the new arc’s radius so that it *exactly* matches the open end of the surface of revolution.



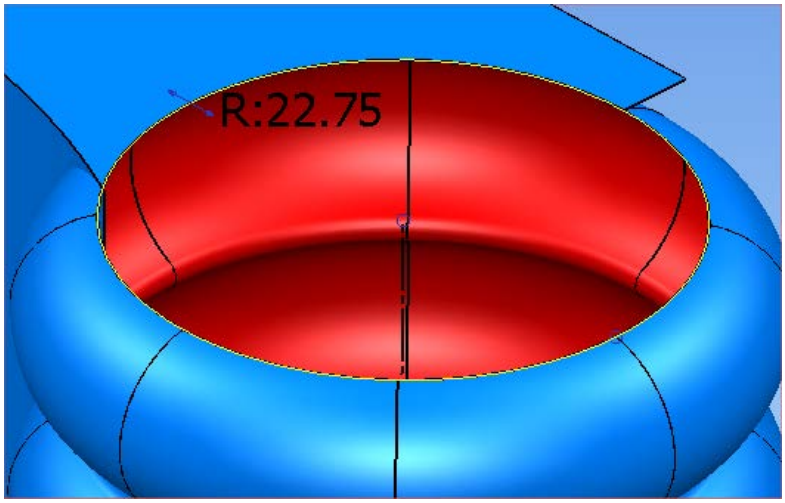
Use the ‘Limit Selection’ command

to trim away the unwanted piece of the side surface, as shown below.



Stage 3 – Creating the end cap

Select the arc created at the end of Stage 2.



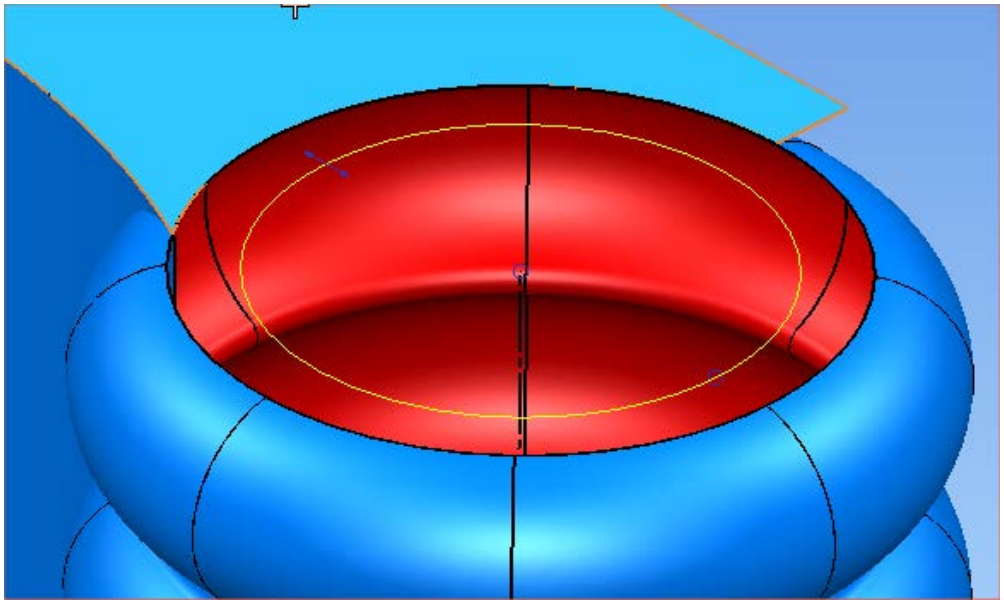
Make an identical Copy of this arc –

Either press ‘Ctrl C’ followed by ‘Ctrl V’

OR

Click followed by

Change the radius of the new arc to **18** (or simply *drag* the radius until you think it looks right).

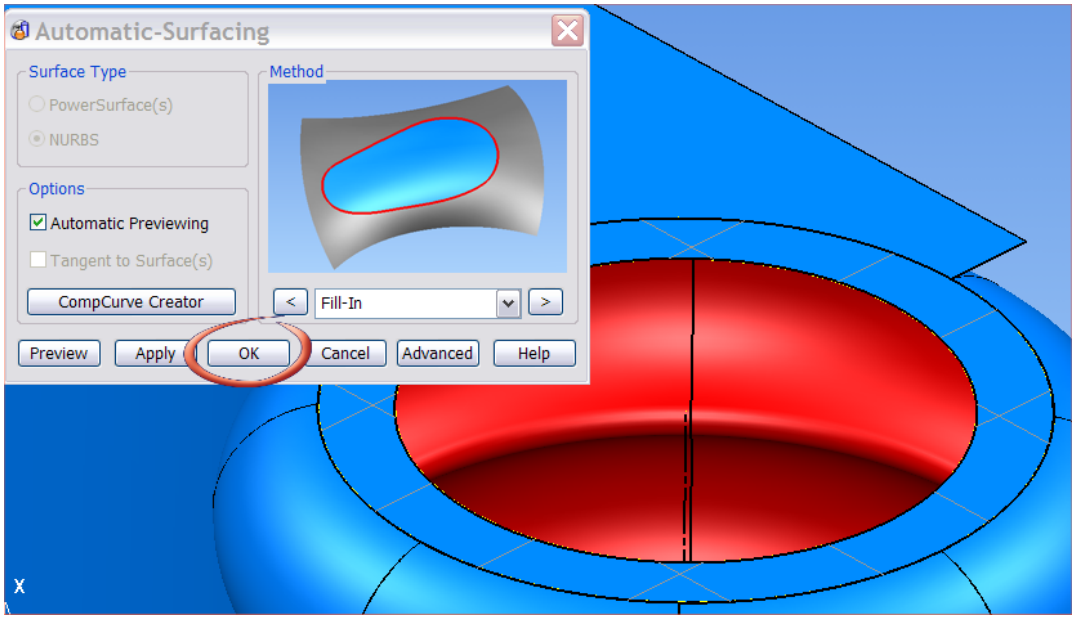


Select both arcs and user the Automatic Surface wizard to create a trimmed ‘cap’ surface.

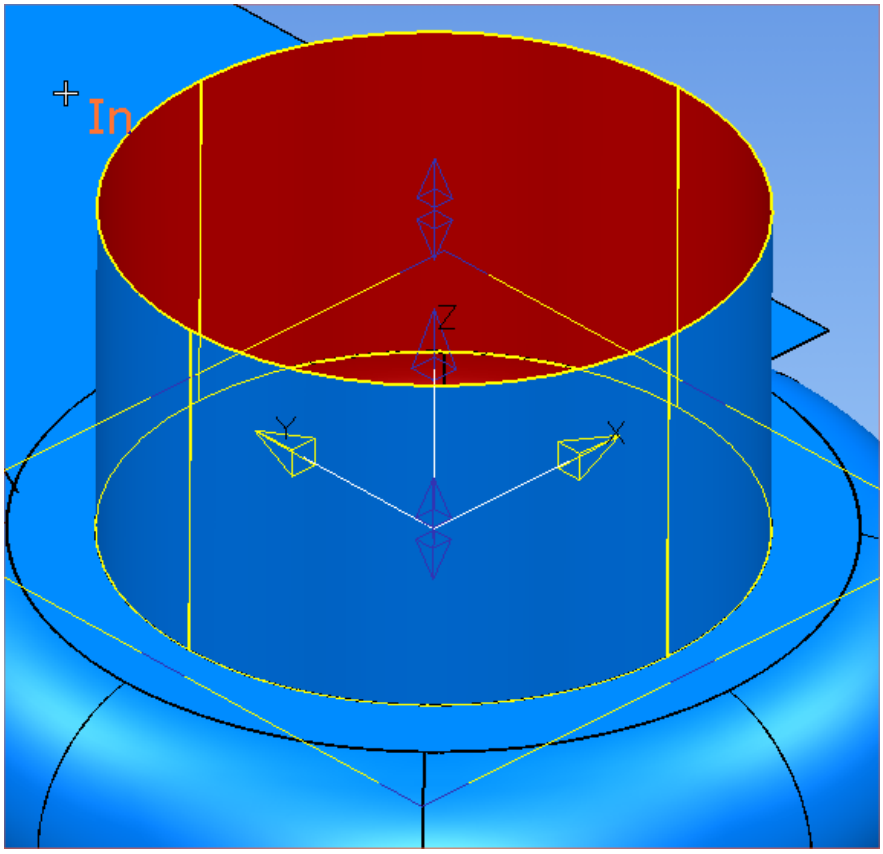
Open the Surface Creation Toolbar

Click the Automatic Surface icon

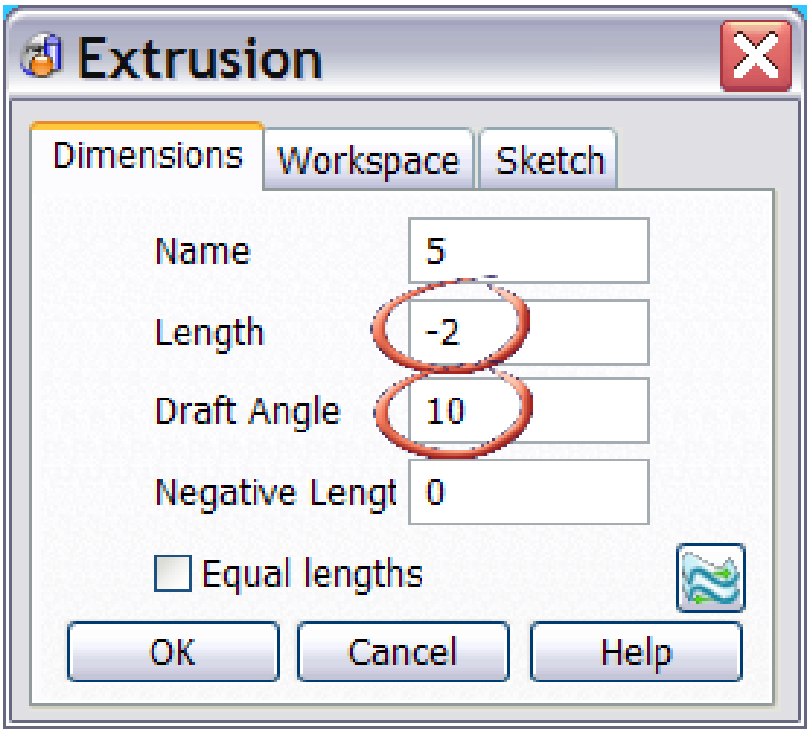
Click OK to accept the new surface.



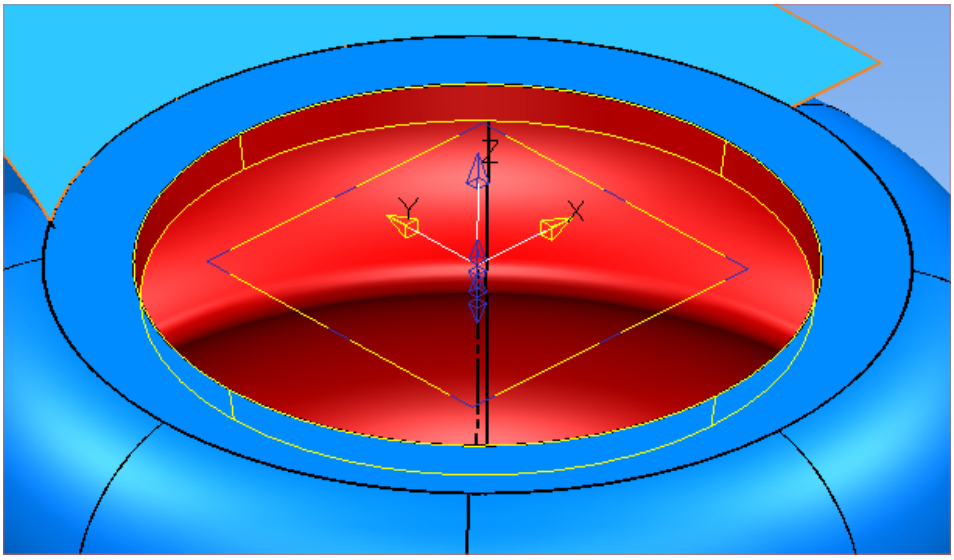
Select the smaller (inner) arc, and create an Extruded surface.



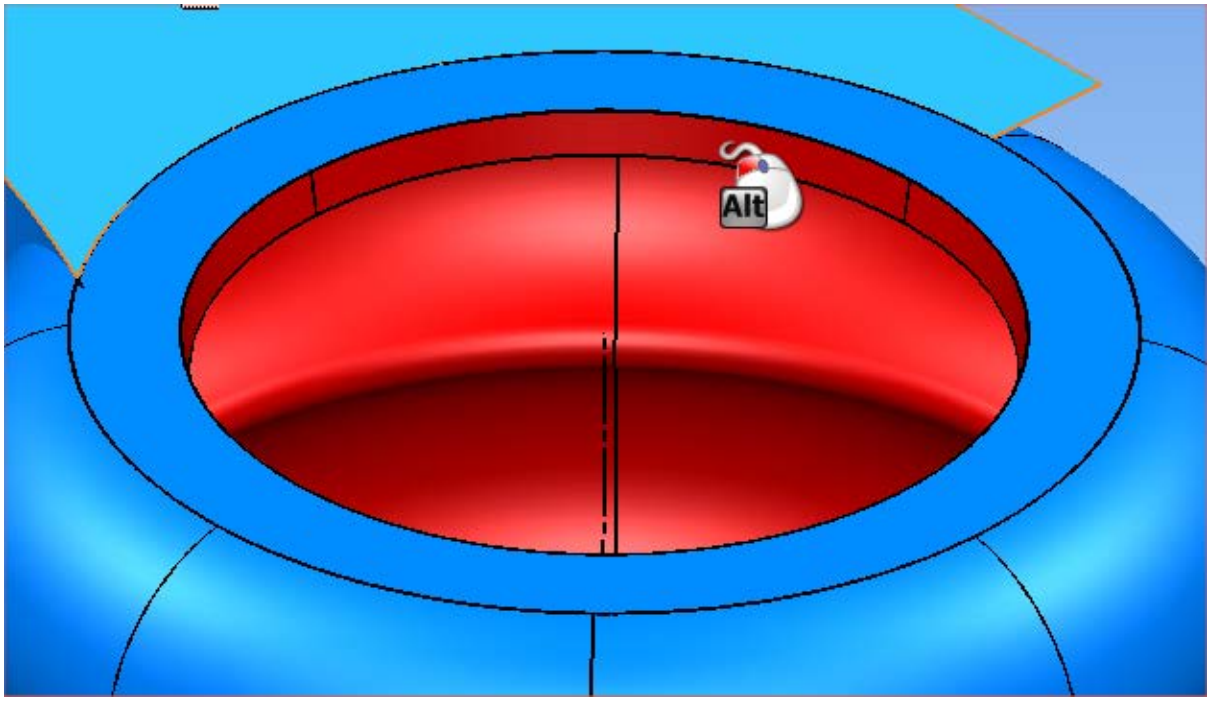
Double-click the new extrusion surface to open the Extrusion edit box. Set the parameters (roughly) as follows –



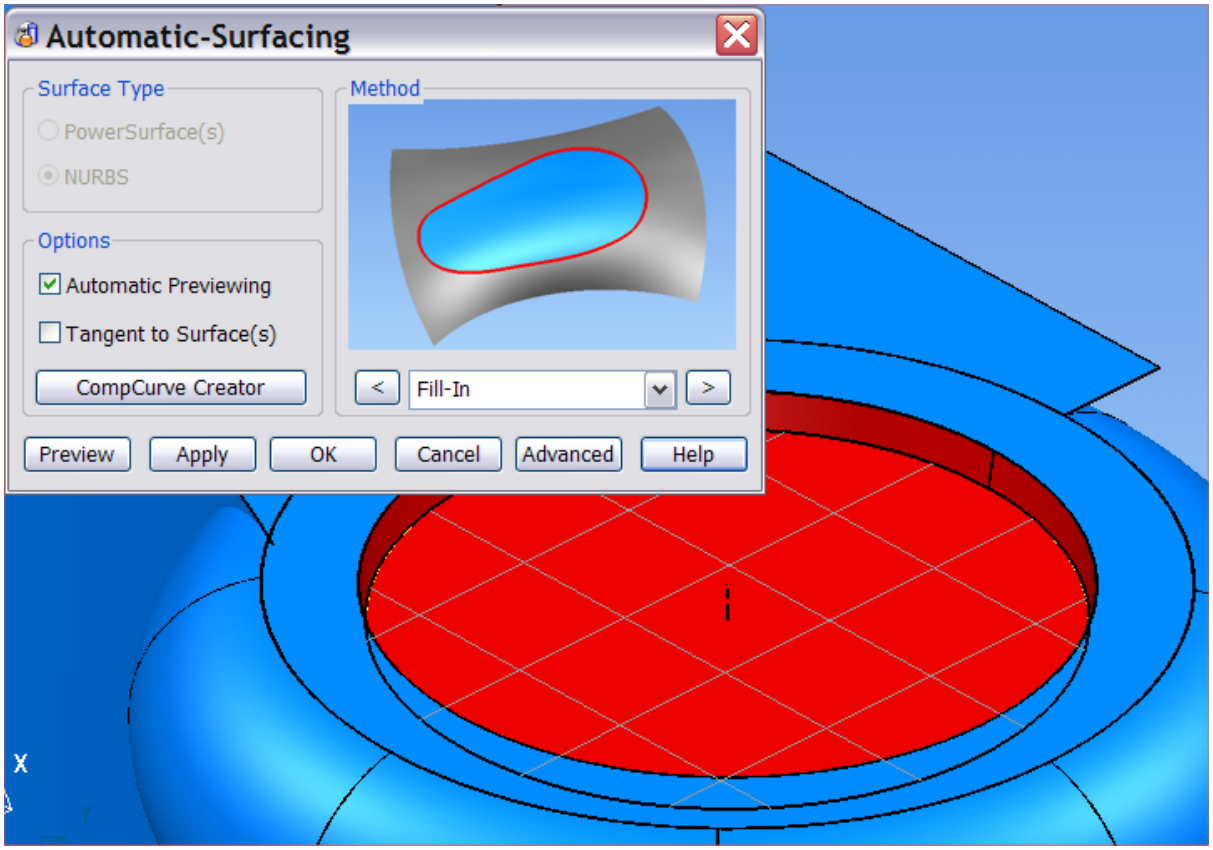
Click OK when you are happy.



Create a composite curve around the *open* edge of the new extrusion. Put the cursor over the edge and press Alt (on the keyboard) and the Left mouse button at the same time.

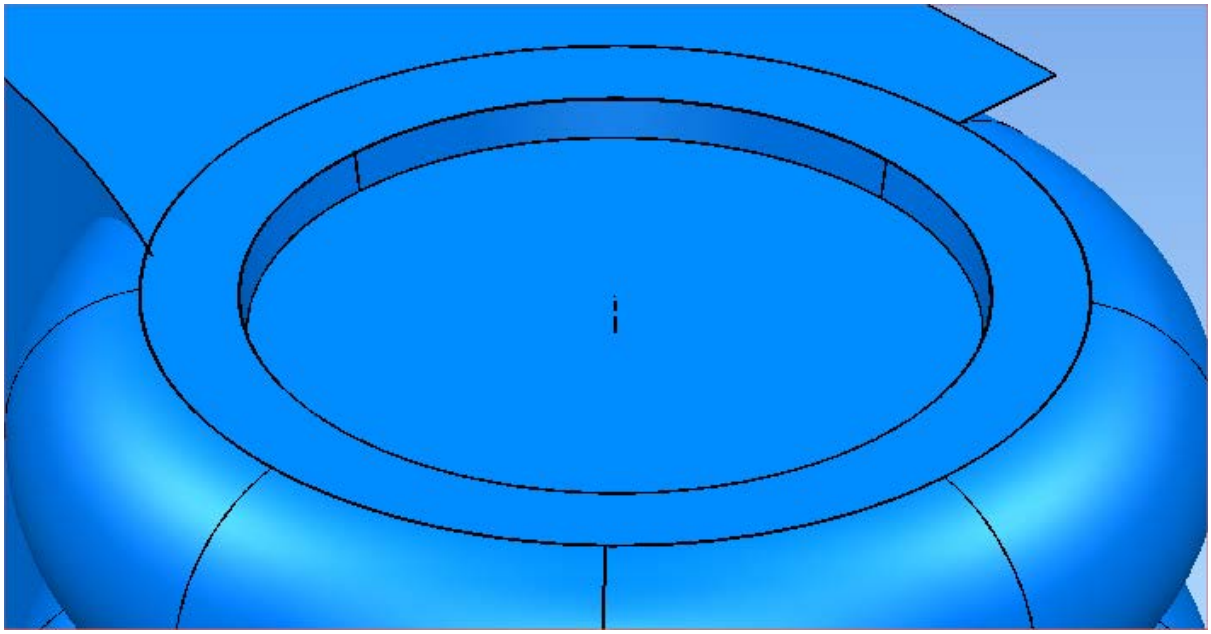


Use the Automatic Surface wizard to make another fillin (cap) surface.



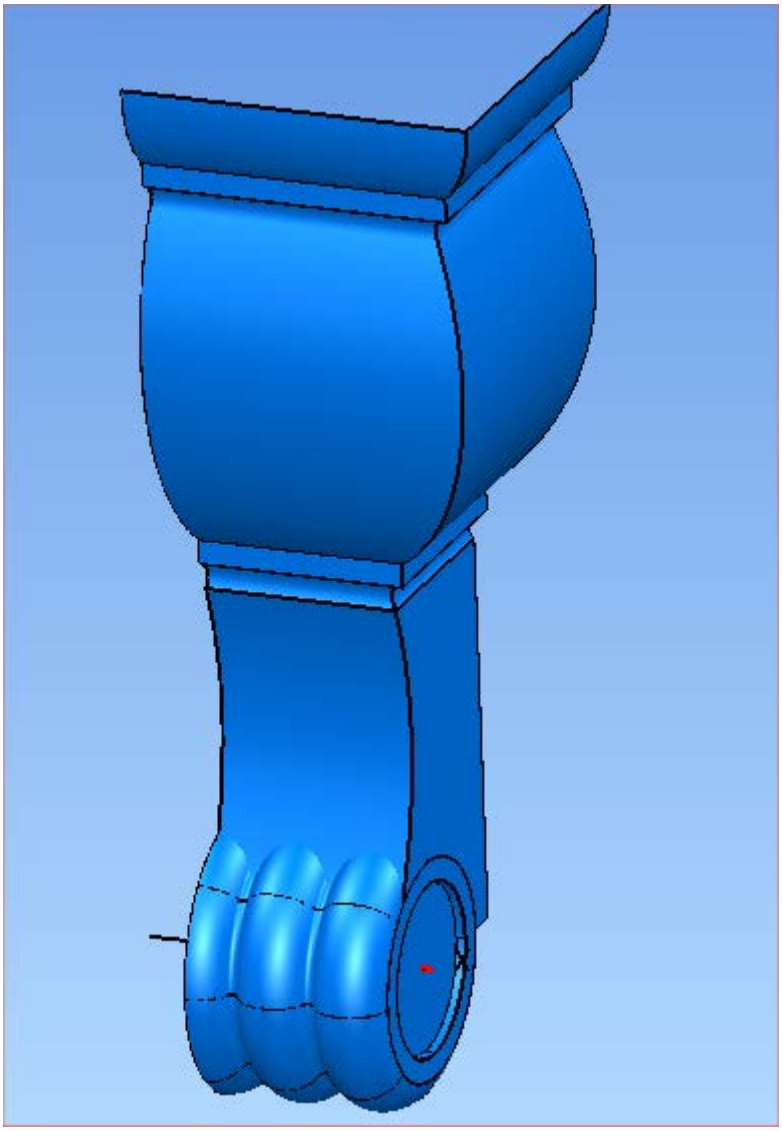
Click OK to accept the surface.

The tapered extrusion and the fillin surface are both inside out, and must be reversed.



Stage 4 – Finishing the Modelling

Rotate the view (approximately) as below.

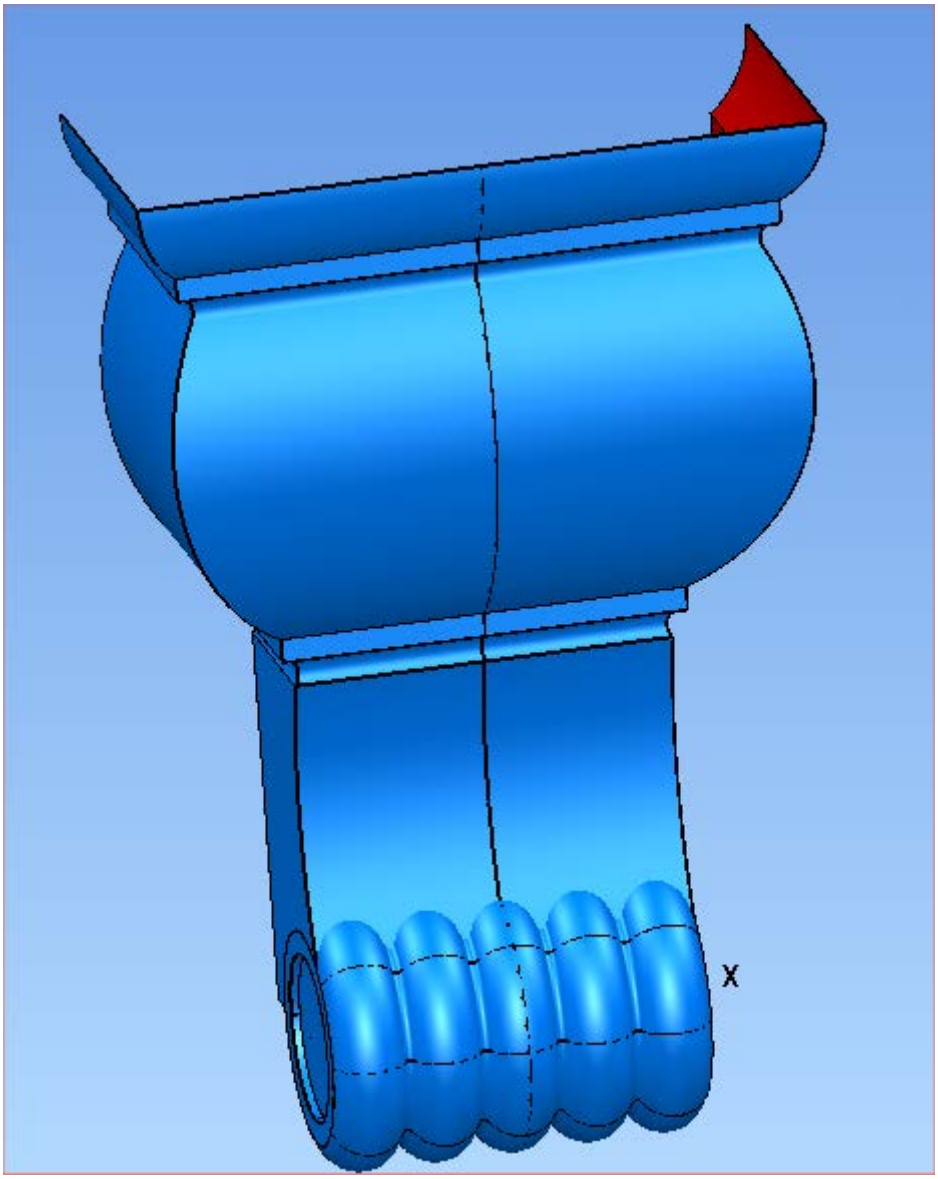


Select everything, and open the General Edits toolbar.

Click Mirror

Mirror the surface (making sure ‘Copy’ mode is ON) through the YZ plane.

The result will be as below.

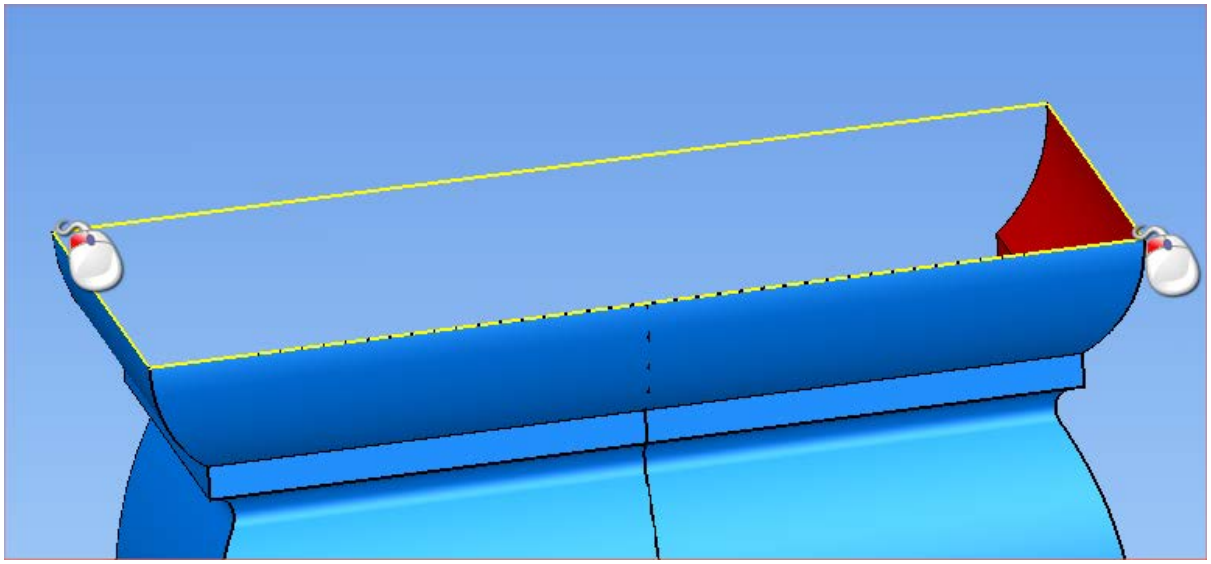


Press to quit Mirror mode.

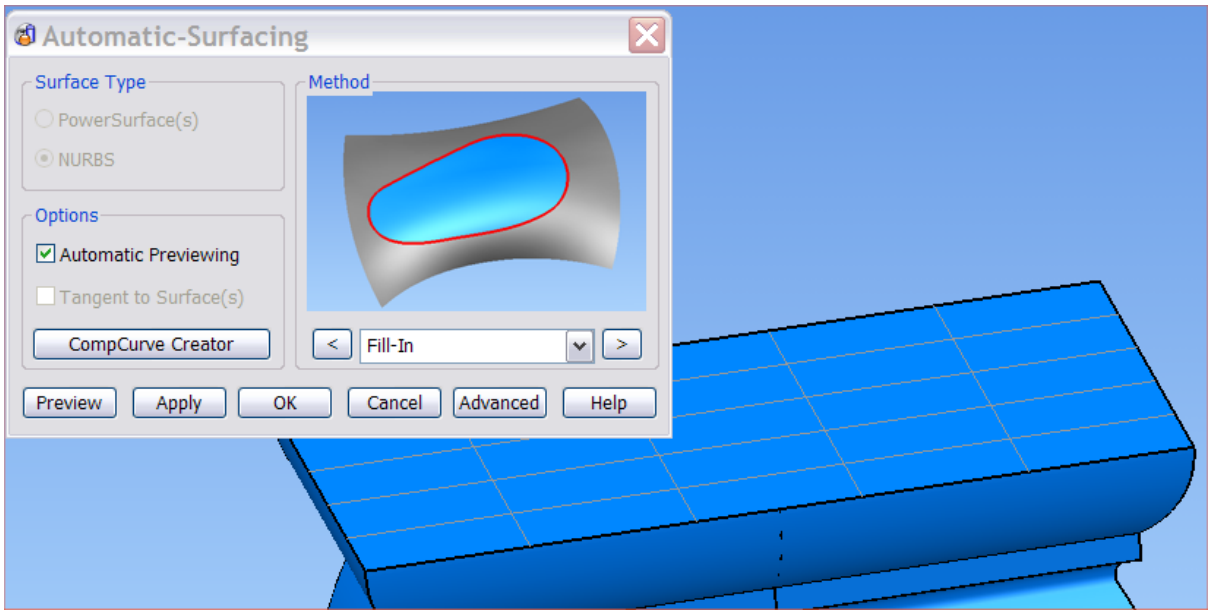
Click the Create Line icon

Select the Rectangle option

Sketch a rectangle across the diagonal corners of the open upper edge of the corbel.



While the lines are *still selected* use the Automatic Surface wizard to create a planar fillin surface,



Click OK to accept the new surface.

At this stage there is some wireframe geometry that is not longer needed. Select all the wireframe using the fast selection toolbar,

and either click from the top toolbar, or press the Delete key.

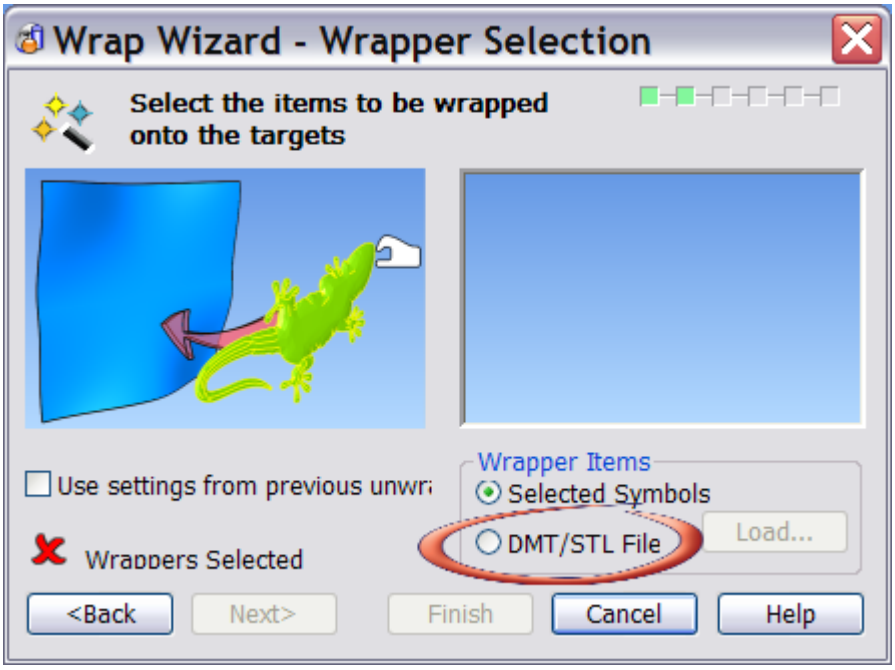
Stage 4 – Adding the decoration

Press ‘Ctrl 2’ or click the ‘Front’ icon to view the model from the front.

Select the two surfaces that form the main front face of the corbel.

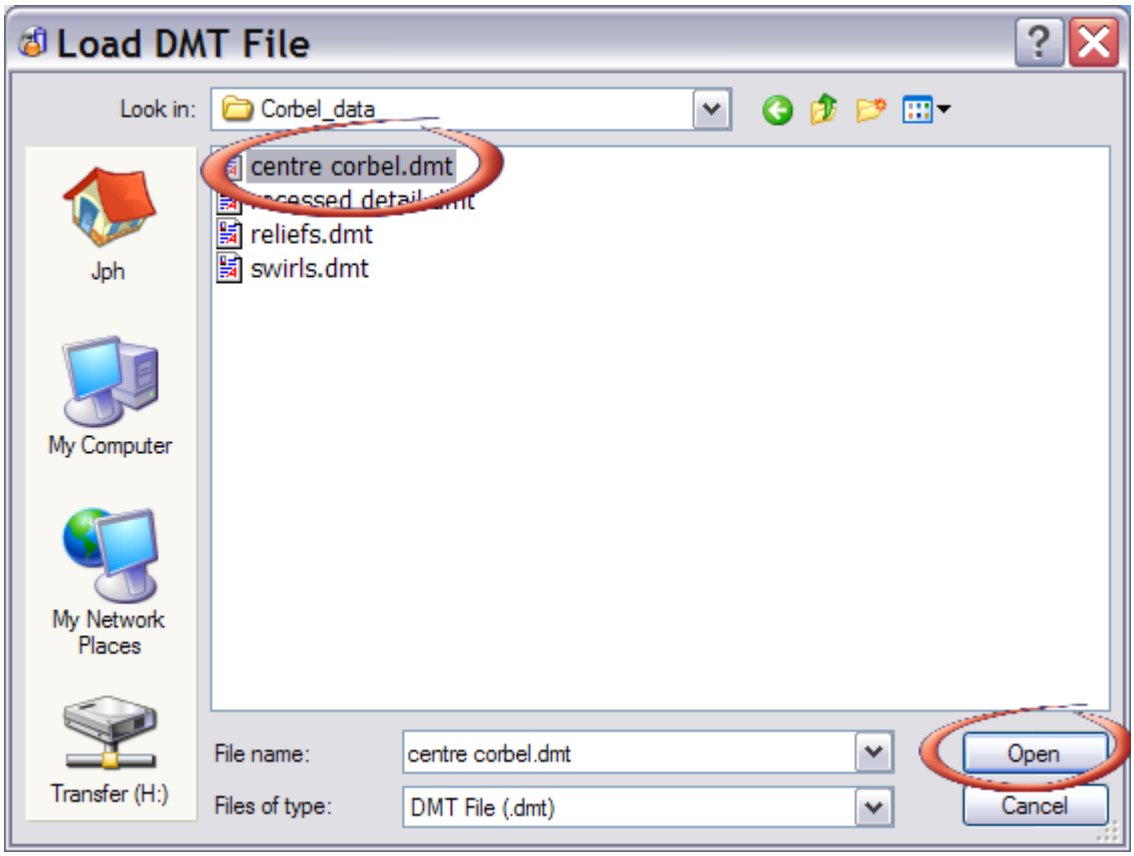
Pick the Wrap icon

The surfaces to wrap onto are already selected, so click ‘Next’.

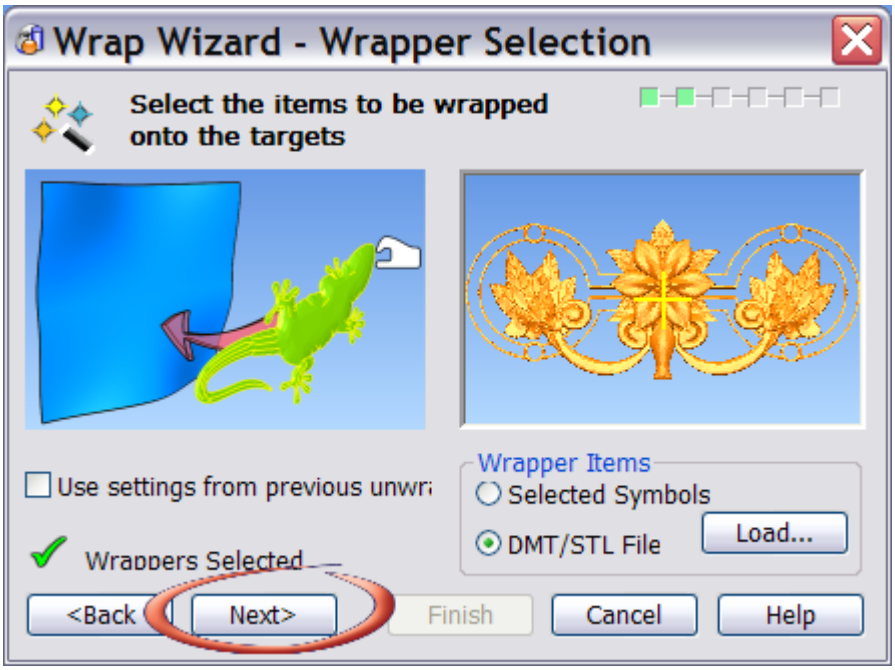


The ArtCAM relief files to wrap have already been converted to triangles and saved on disk as dmt (Delcam Machining Triangle) files.

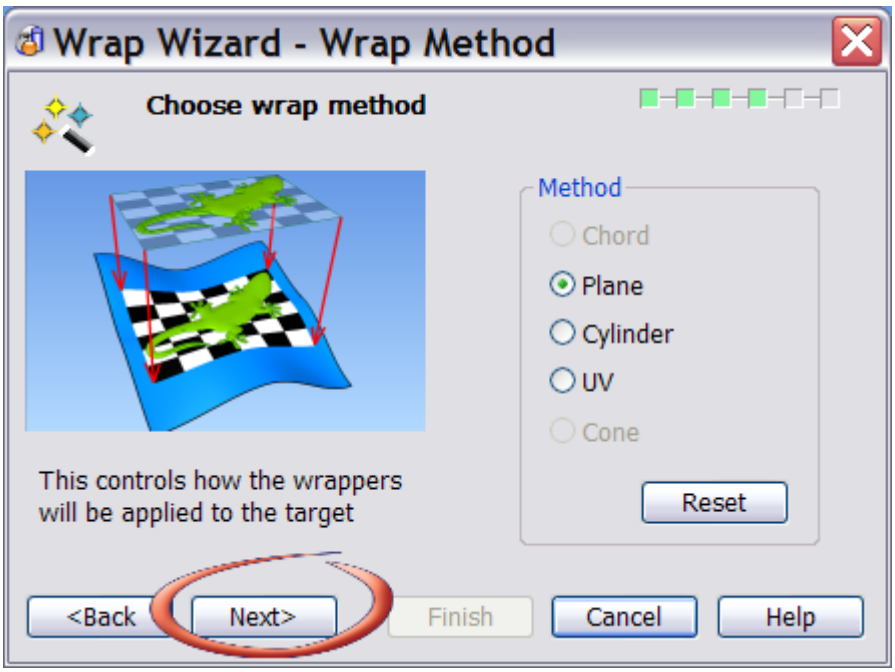
When you pick the dmt option the file browser will open allowing you to select the file to use.



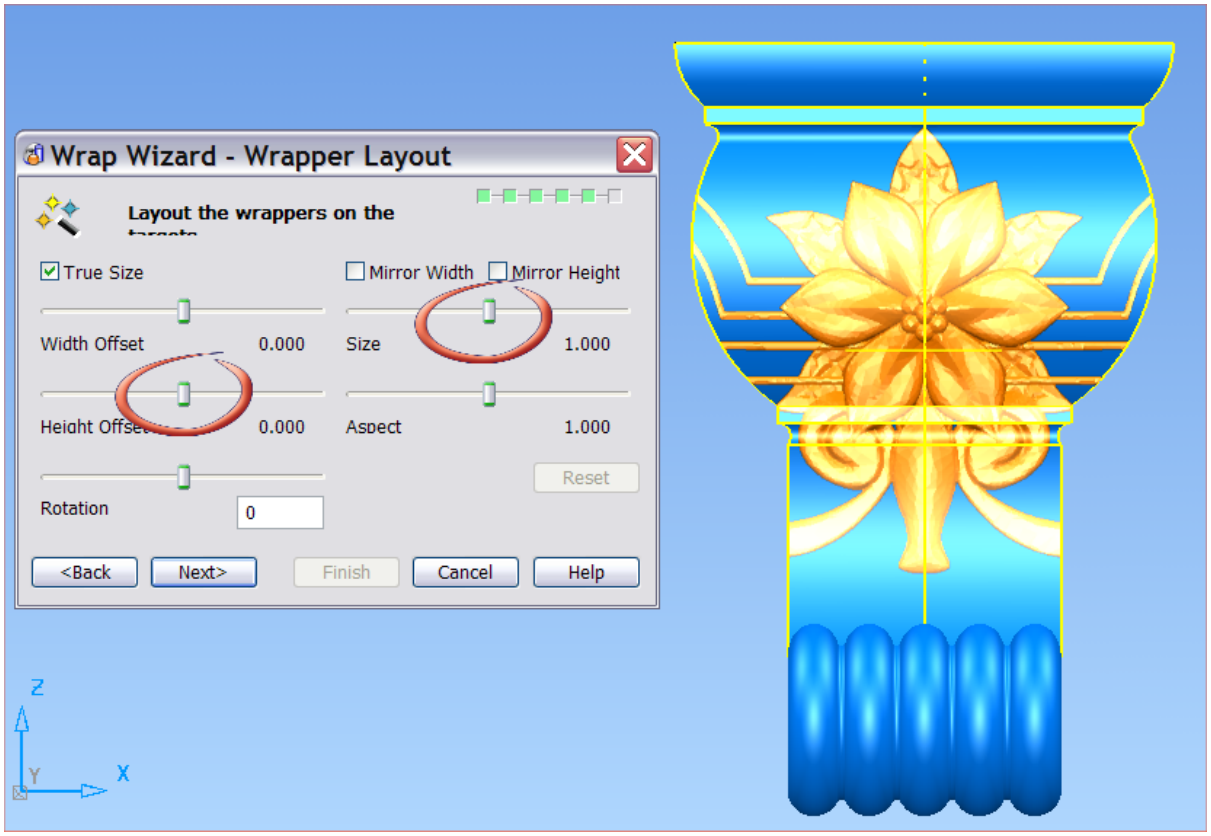
Locate the file ‘centre_corbel.dmt’ and click ‘Open’.



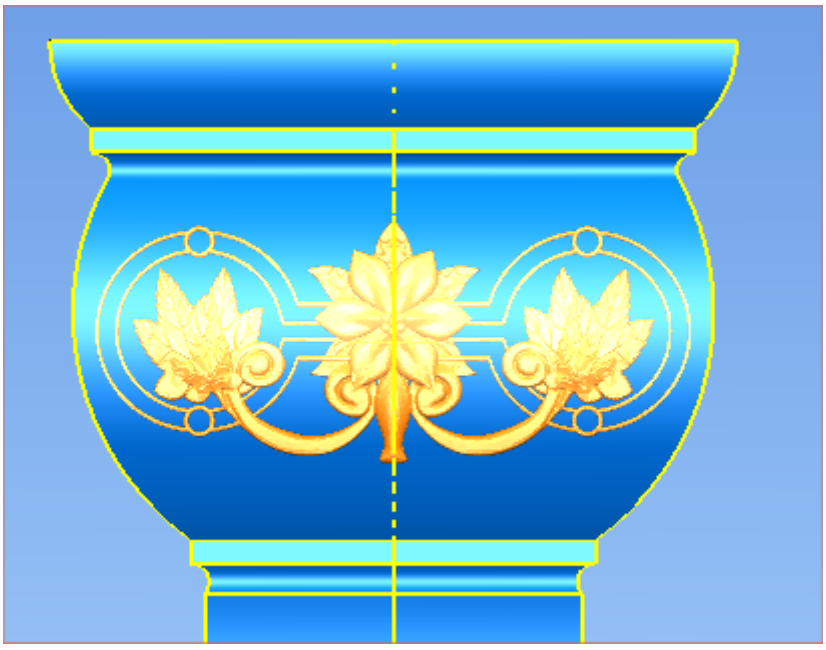
Notice that the Wrapper Selection form shows you an image of the relief file. If you select the wrong file by mistake you can simply click ‘Back’ and choose a different relief.



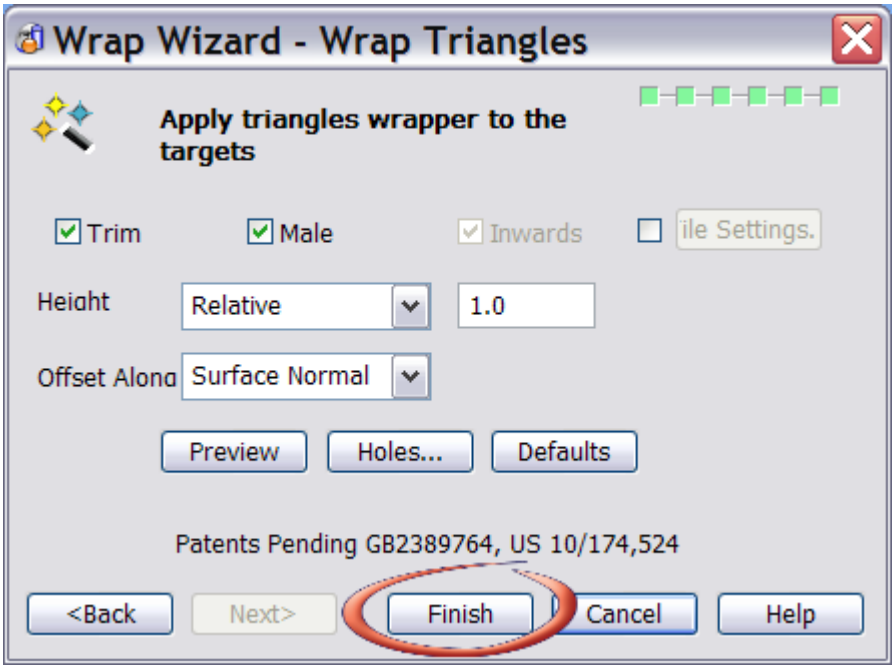
For some shapes you will find that you can achieve a better result by using a different wrapping Method. The simplest method (plane) is selected by default. See the manual and on-line help for complete descriptions of the other methods and where they should be used.

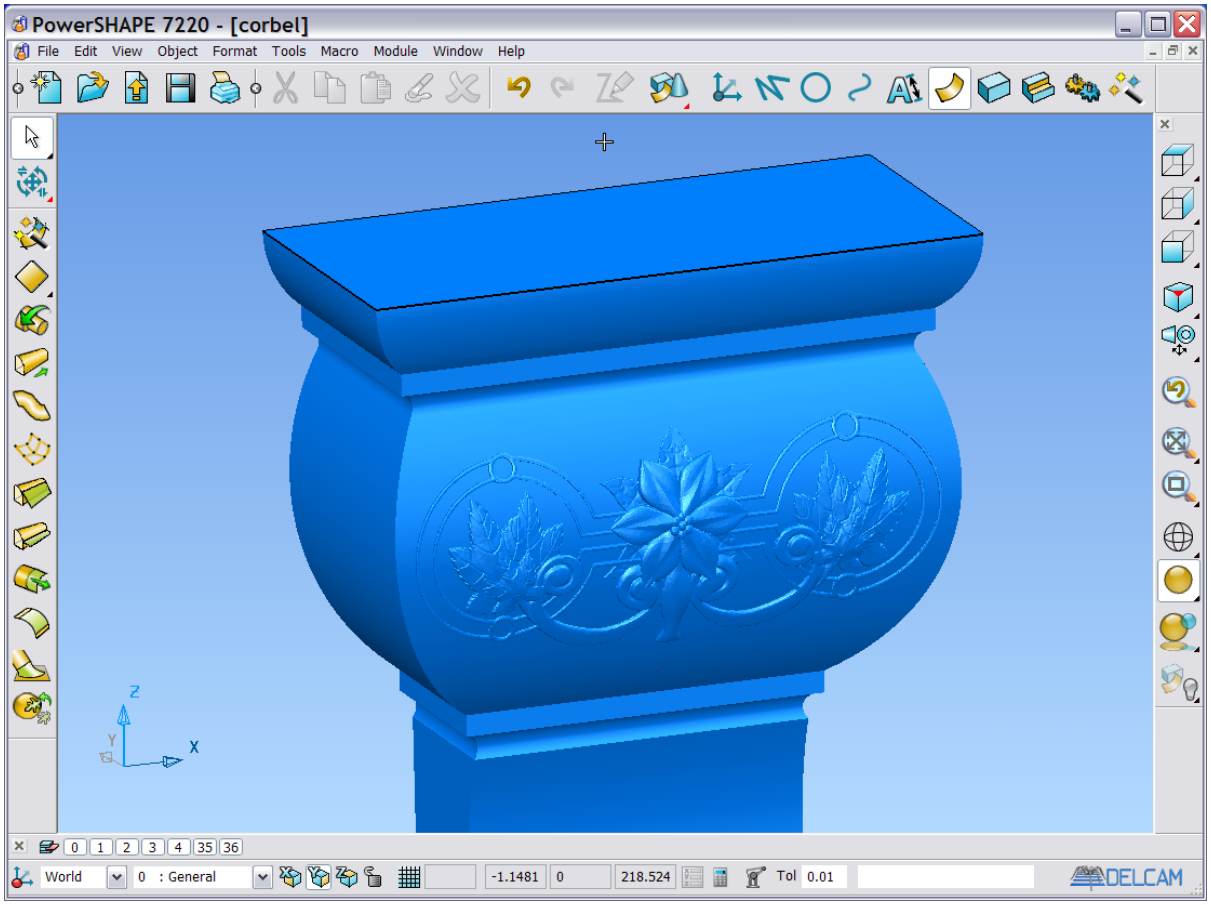


The default wrap is shown as a preview. At this stage the result is simply graphical (as if a 2D label had been applied to the surface). Using the sliders on the form you can adjust the size, orientation, aspect ratio and position of the final 3D relief. Adjust the Size and Height Offset sliders until the preview looks as below.



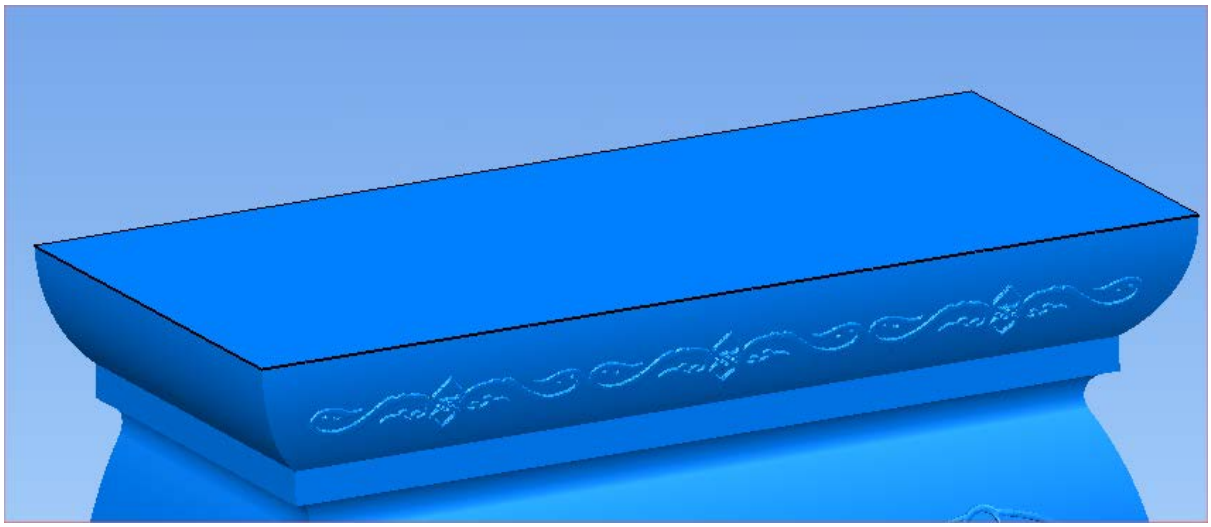
When you happy with the preview, click ‘Next’.



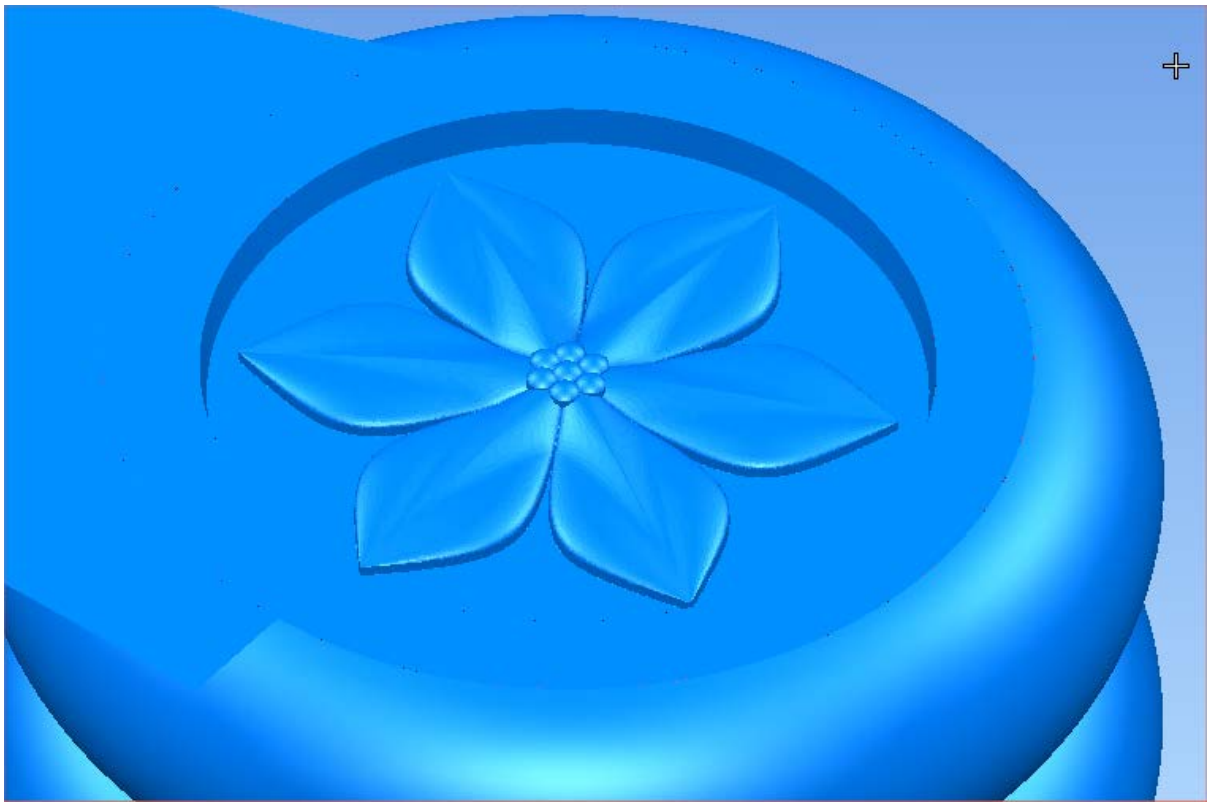


The partially decorated corbel.

Complete the decoration by adding two more reliefs.



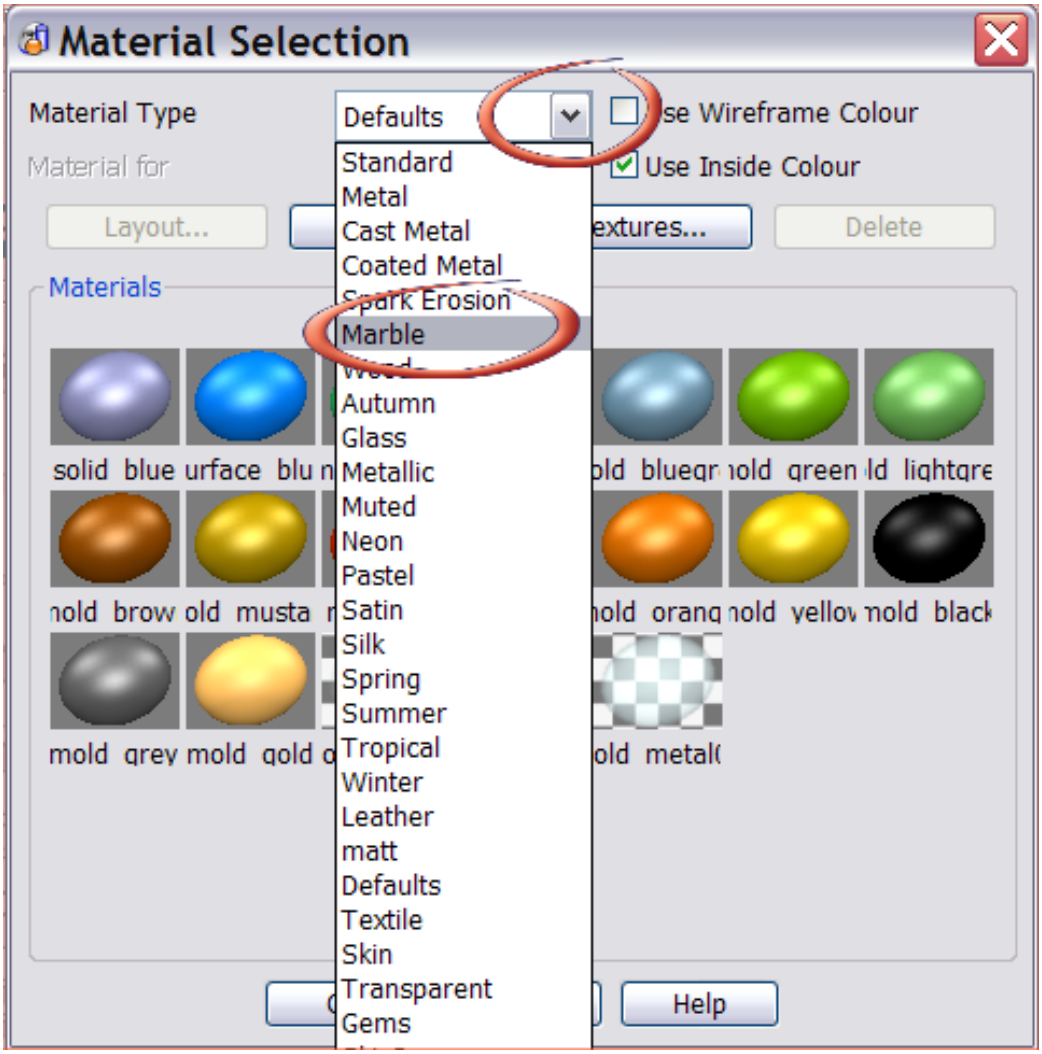
Corbel after adding ‘Swirls.dmt’.



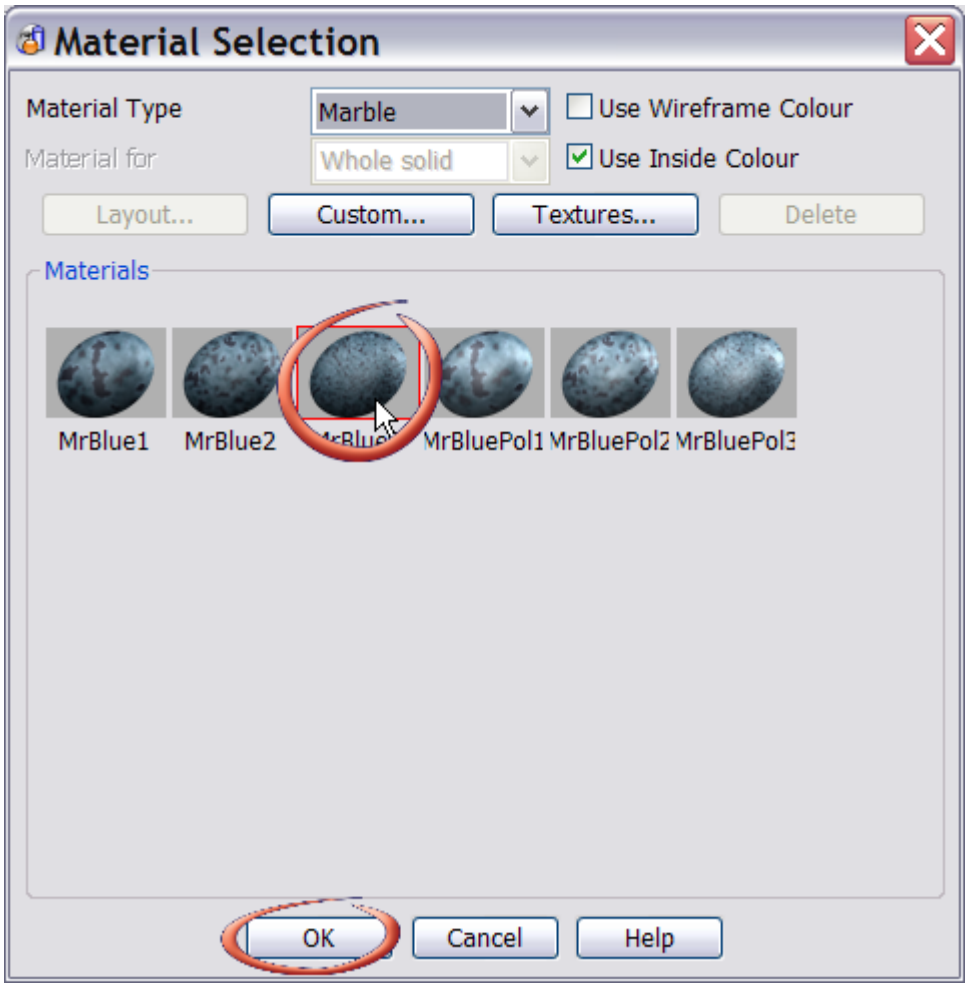
Corbel after adding ‘Recessed_detail.dmt’.

Stage 4 – Rendering

Rotate the View as desired.
Select the entire model.
Click on Format from the top toolbar.
Click on Materials from the Format pull-down.



Choose a marble material from the form, and click OK.



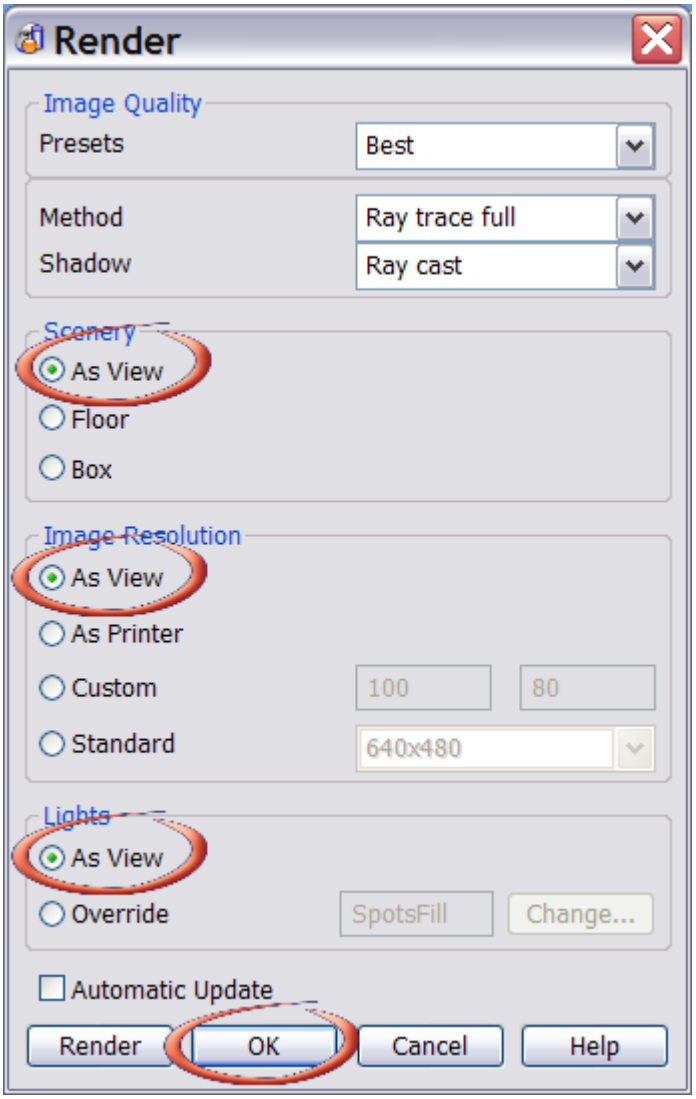
From the views menu, select ‘Enhanced Shading’.

This set the view to be perspective, so more natural, and also applies a reflection map to your scene. Shiny materials (particularly metals) will look much more realistic.

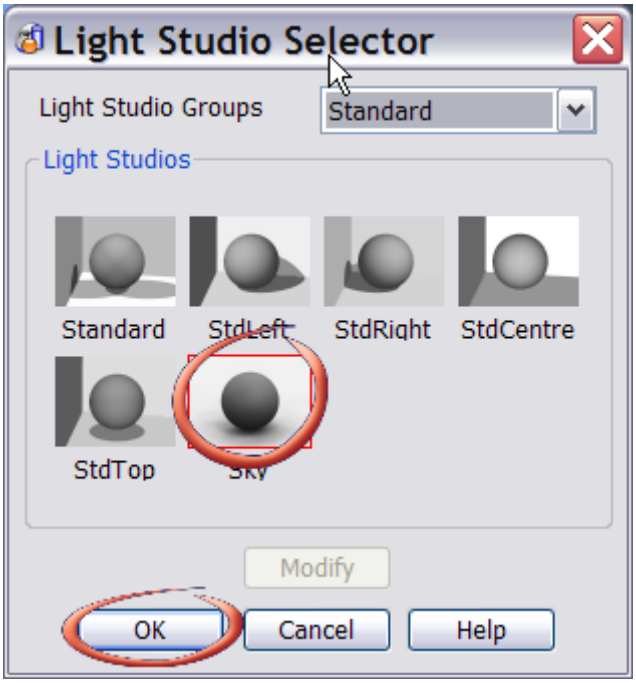
Open the Render options

Alternatively you can click on ‘View’ from the top toolbar, and then select ‘Render’ from the drop-down list.

Set the values as shown below.



Open the Light Studio selector, by clicking ‘Format’ on the top toolbar and choosing ‘Lights’ from the drop-down list.

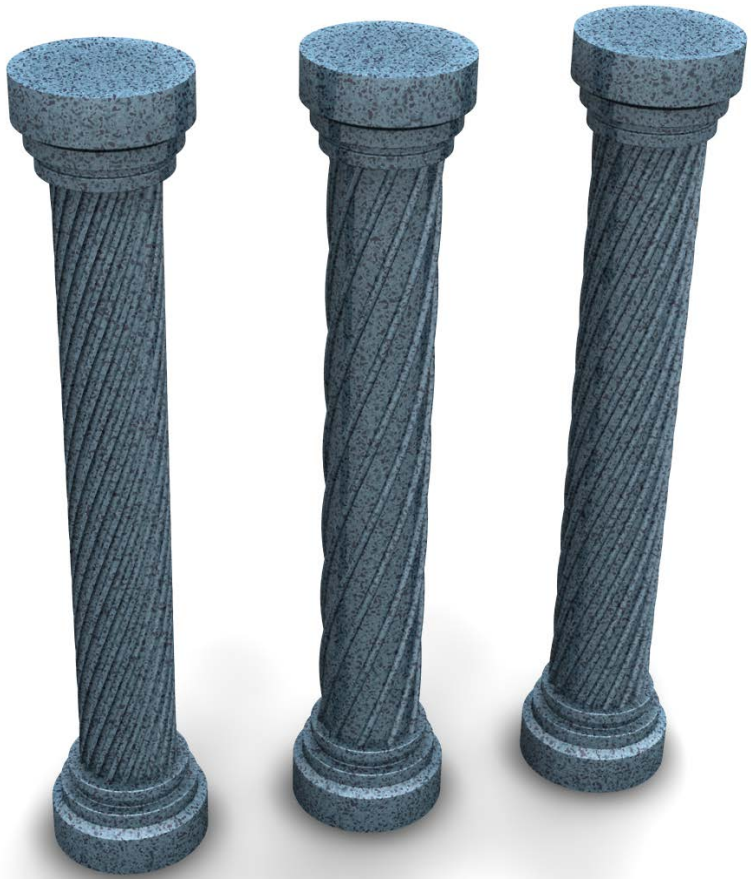


Click the Render icon.

NOTE: The ‘Sky’ light studio provides very realistic soft shadows, and provides a much more natural looking finished image. However, processing can take some time. With render set to ‘As View’ from the option form, the rendered image will be the same size as your screen view. To speed up rendering make your screen size smaller, by shrinking the view using the normal Windows controls.

Save the model in **D:\users\Training\Coursework\Delcam Designer Jobs** as **Corbel.psmodel**

Exercise – Decorative Column



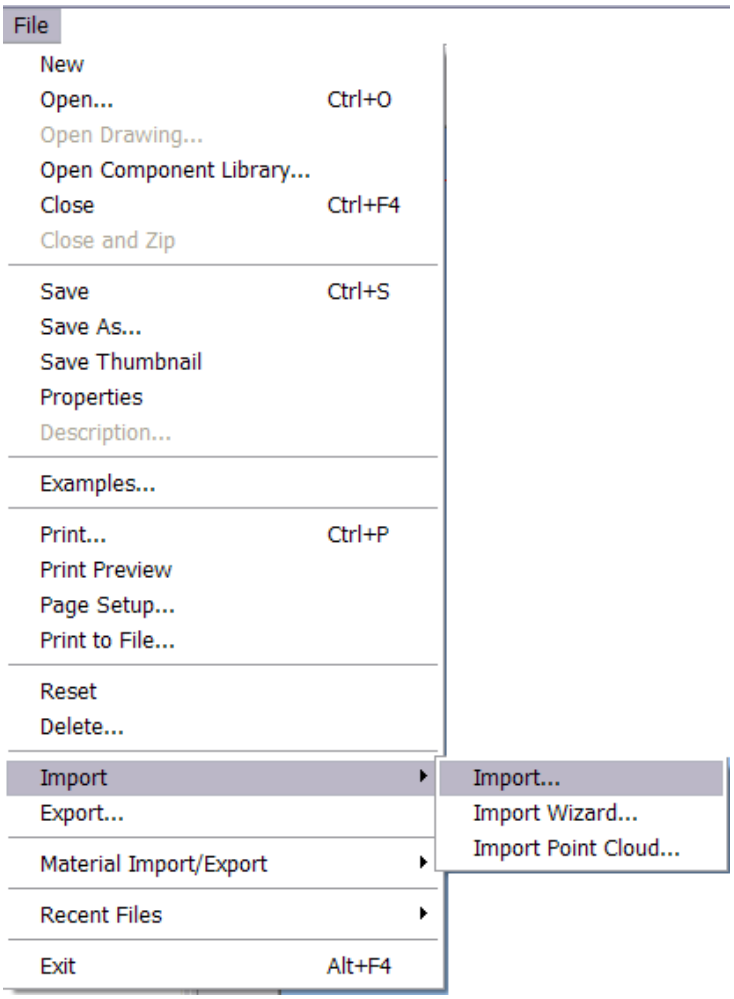
Introduction

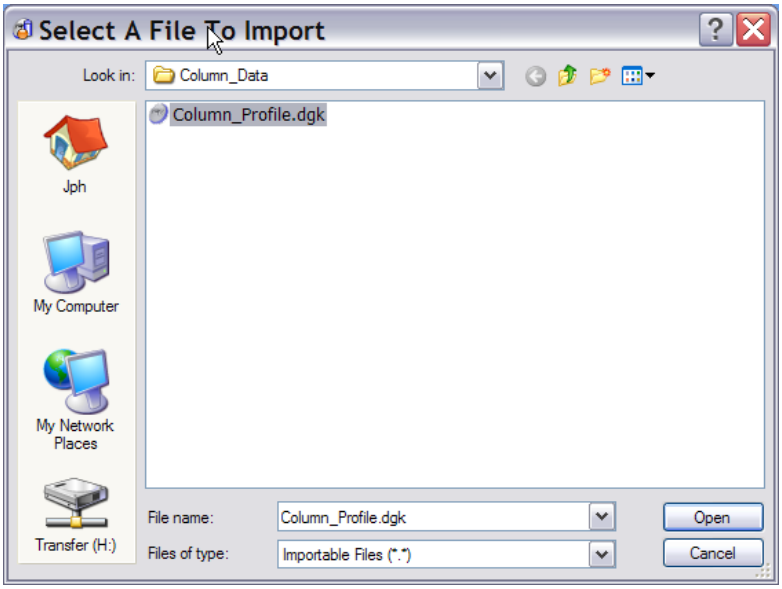
Beginning with some very simple sketches, this tutorial shows how to design the decorative columns shown above. Because the columns are designed using *features* and *patterns* elements of the design, such as the number of flutes can be changed very quickly and simply.

Importing the Data

- Select **Create new Model**
- Create a **Workplane** at **0,0,0**

Start by importing the Delcam Geometry Kernel (DGK) file that contains three simple wireframe curves.

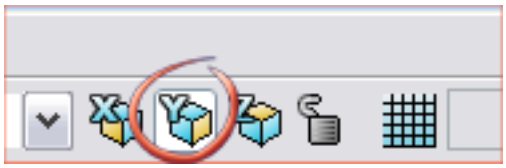




The three curves (which are composed entirely of straight lines) are shown below. If you prefer, you may sketch your own lines and curves. Do not forget that they must be converted to composite curves before you continue.

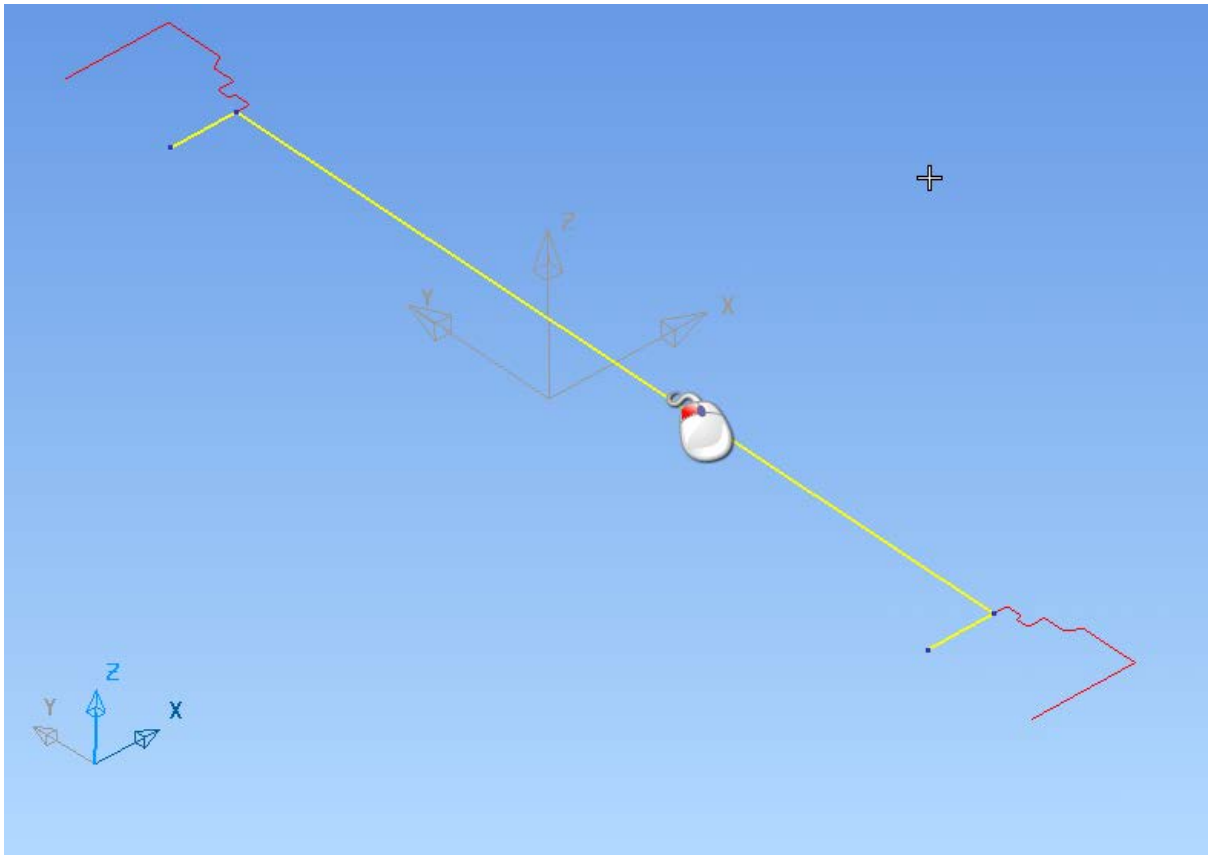
NOTE: In this example the central column will be a closed, cylindrical shape. Make sure, if you use your own curves, that they will correctly join onto the centre-line as in the imported example.

Making the Main Solid



Set the working axis to be Y.

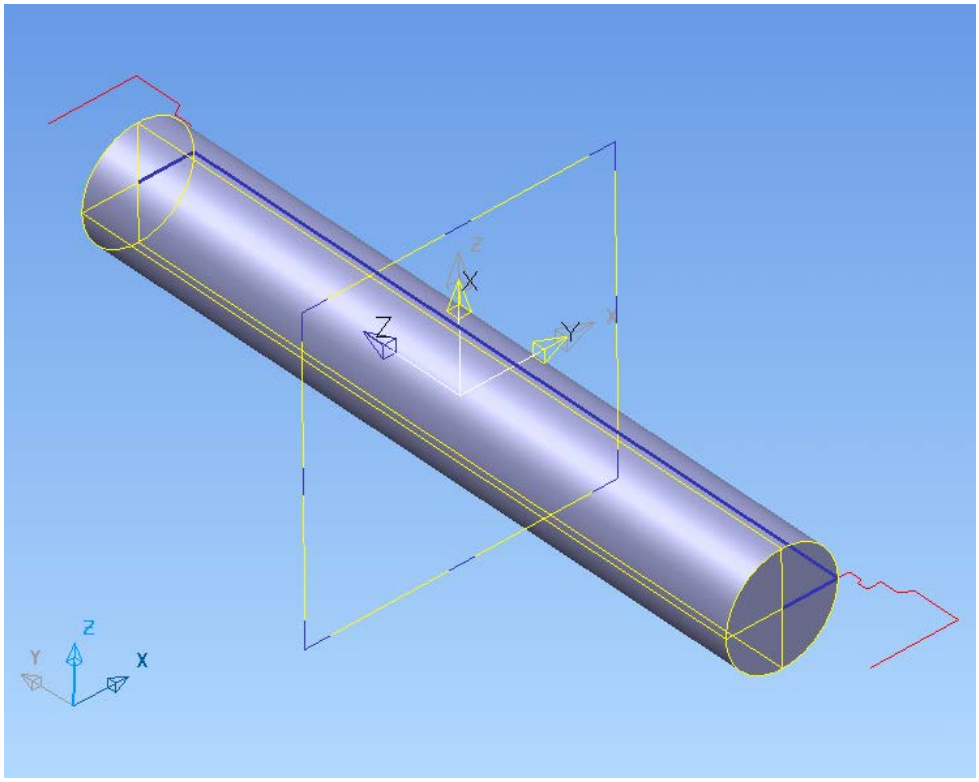
Select the black curve. Notice that its ends line up with the Y axis centre-line, so that it will close correctly at the ‘caps’ when rotated into a solid.



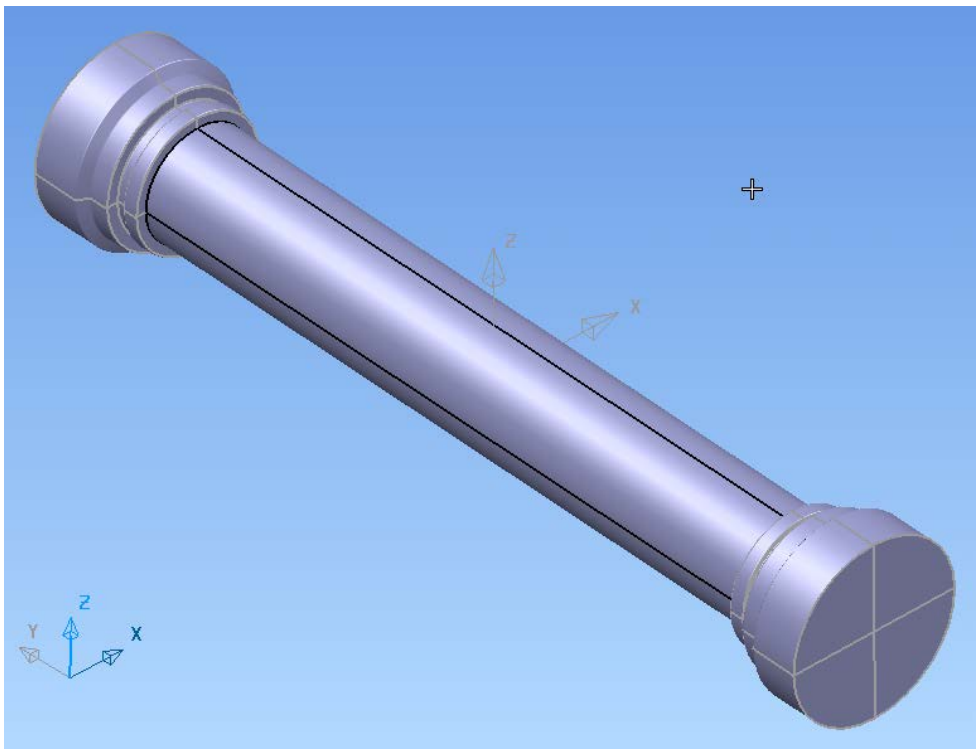
Open the solid toolbar

Click the ‘Solid or Revolution’ icon

This will create the shape shown below.



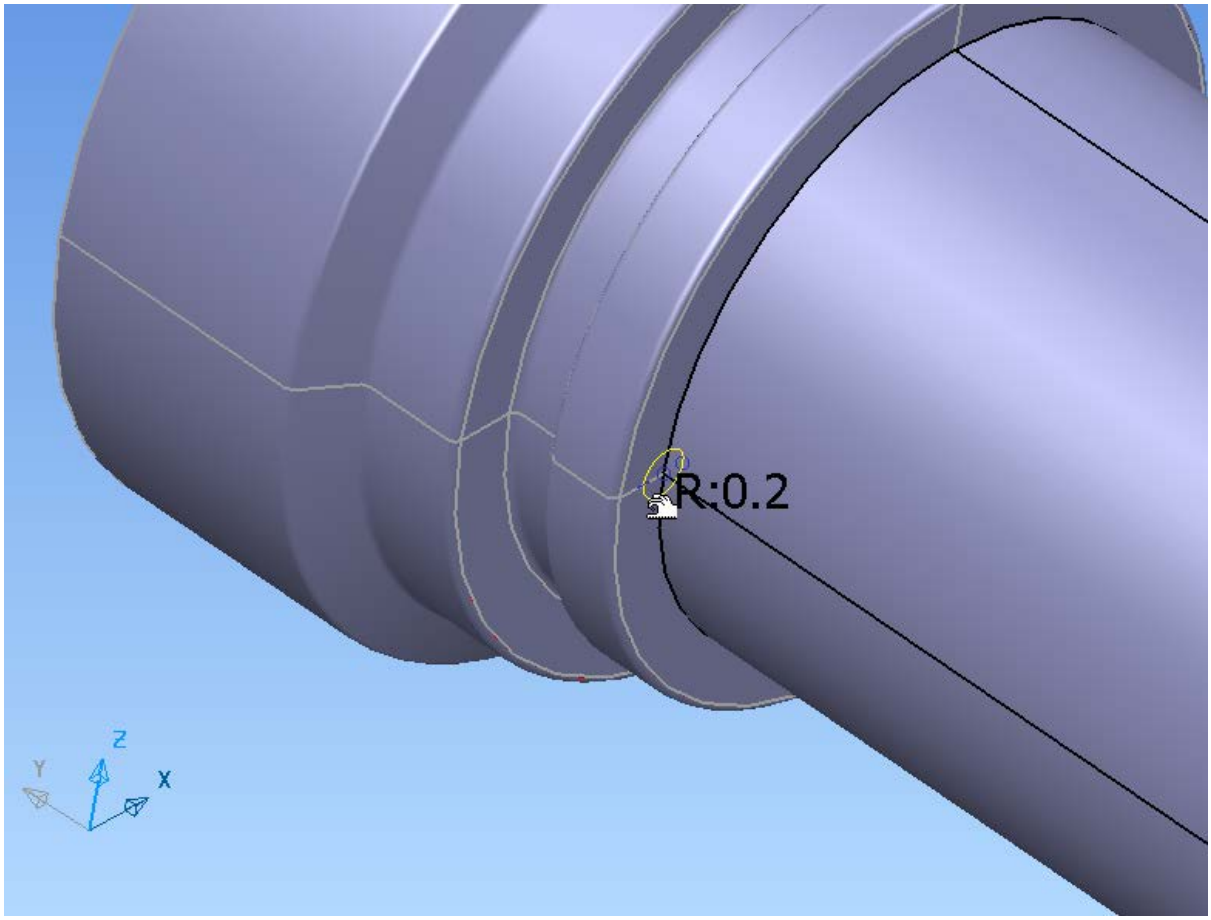
Create two more Solids of Revolution using the two remaining (red) curves.



Adding a ‘Flute’

- Click the ‘Crate Arc’ icon from the main toolbar.
- Create a ‘Full Arc’ (circle)
- Create a **circle** at centre **2.5 -16 0** with a radius of **0.2 mm**

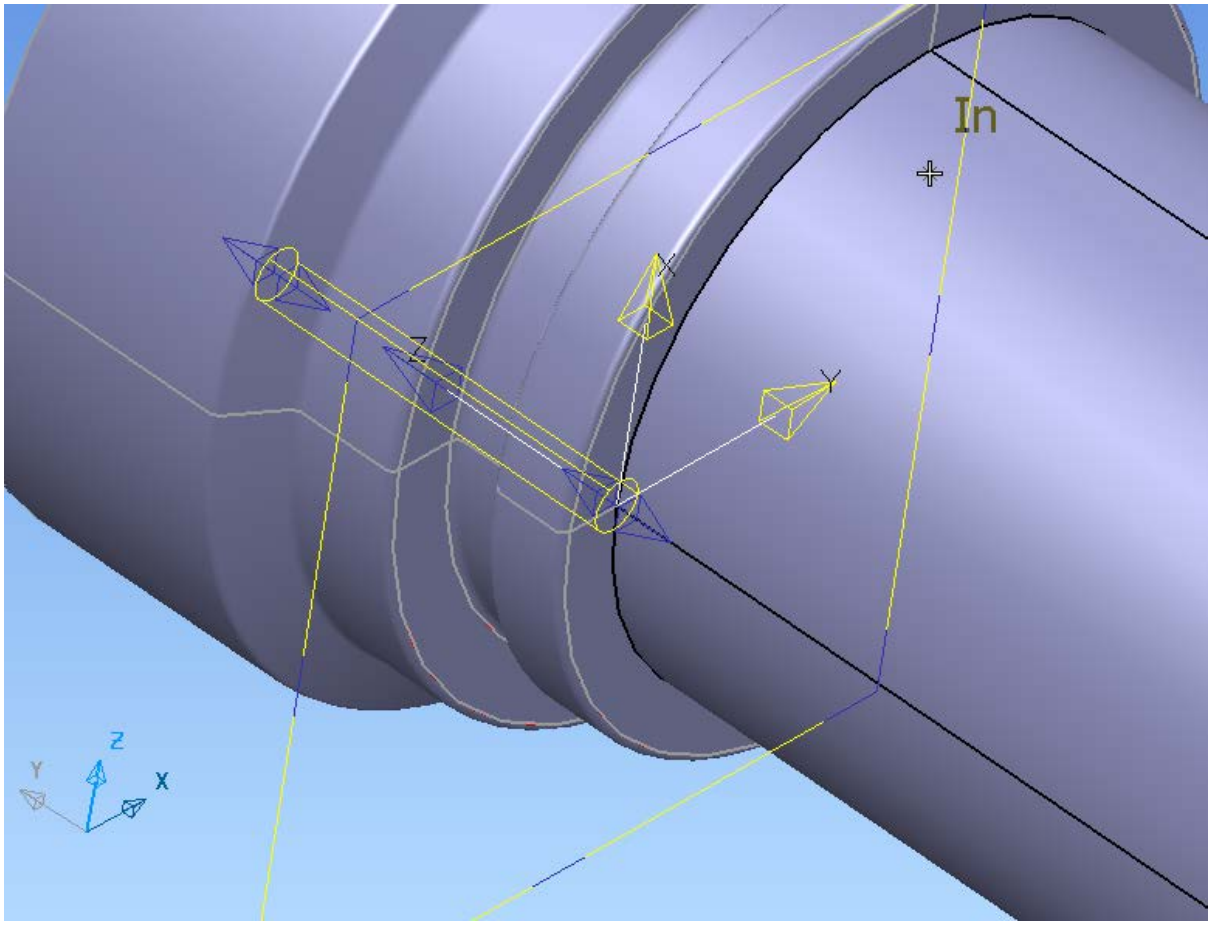
Snap the centre of the circle to the position shown below, and set the arc’s radius to 0.2 (or use your own judgement for the value if you prefer).



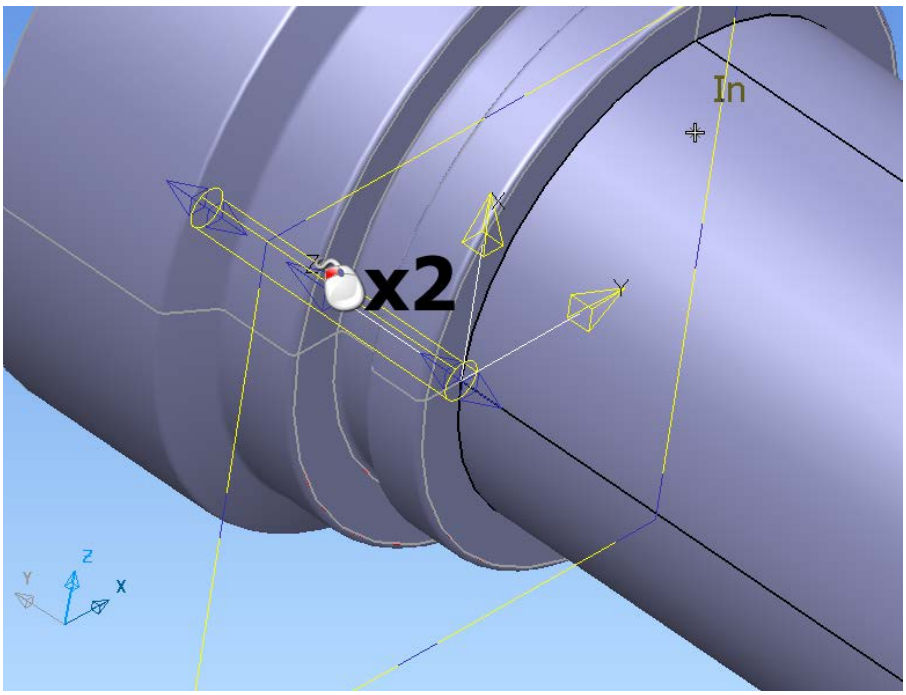
With the arc still selected, create an Extruded surface.

Click from the top toolbar, then

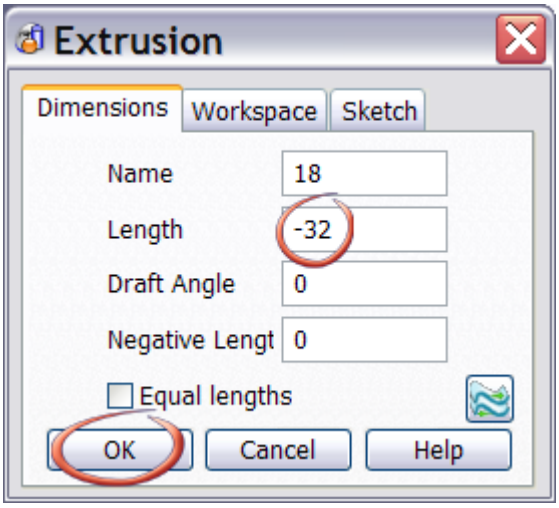
By default, the new extrusion will look as below.



The direction of the extrusion needs to be reversed, and also the length must be set correctly. Double-click then extrusion.



This opens the Extruded surface edits form. Set the length to -32 (If you have sketched your own curves, set the length accordingly).

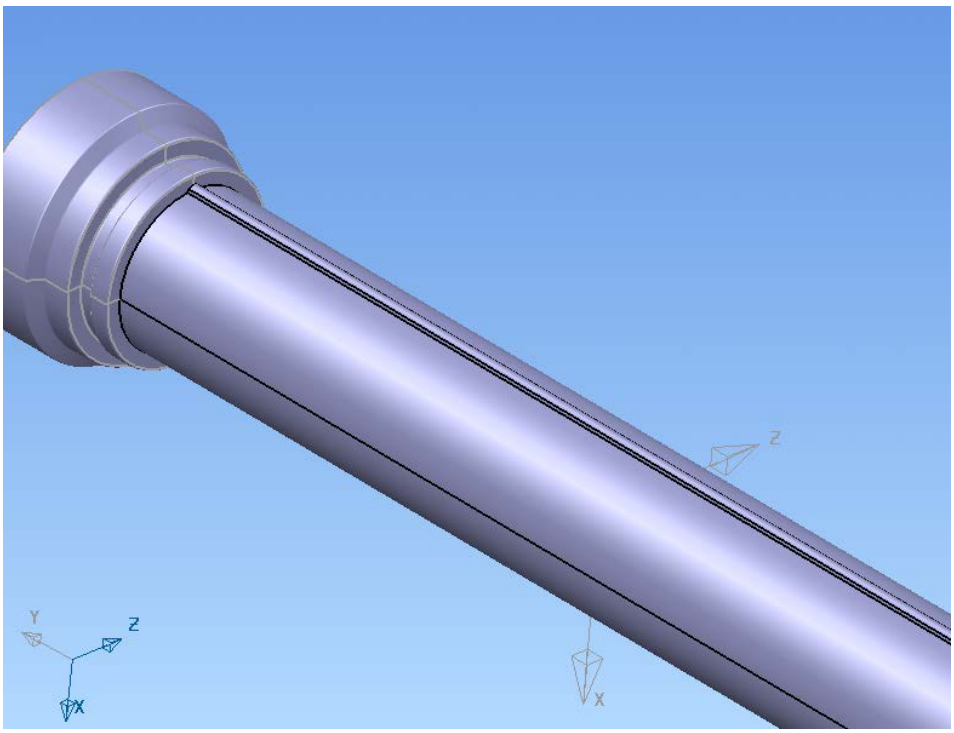


This new surface will be joined to the initial solid column using a Boolean.

Activate the Solid Features toolbar

Click on ‘Solid Add’

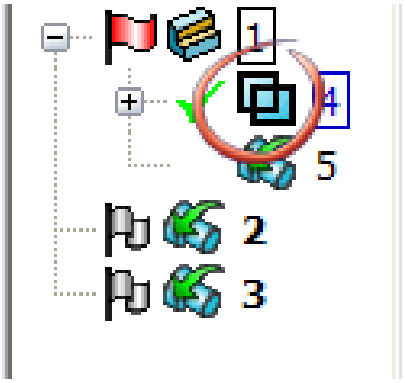
The result will be as below.



Copy the Flute as a Pattern

If the solid's feature tree is not already open, open it now by double-clicking on the solid. The history tree shows all the operations you have used so far in constructing the column.

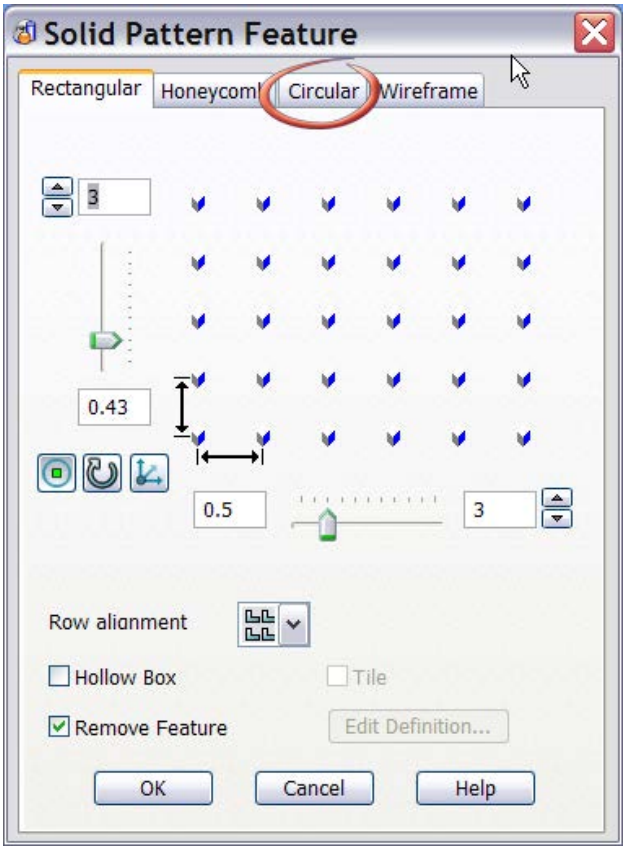
From the tree, select the 'Solid Add' feature, by clicking it with the left mouse button.



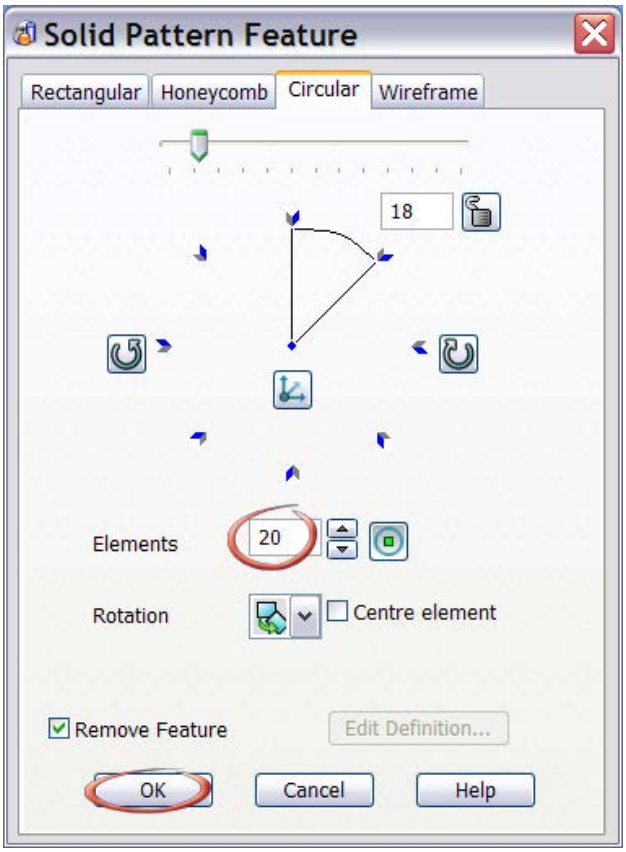
Activate 'General Edits'

Click on 'Pattern'

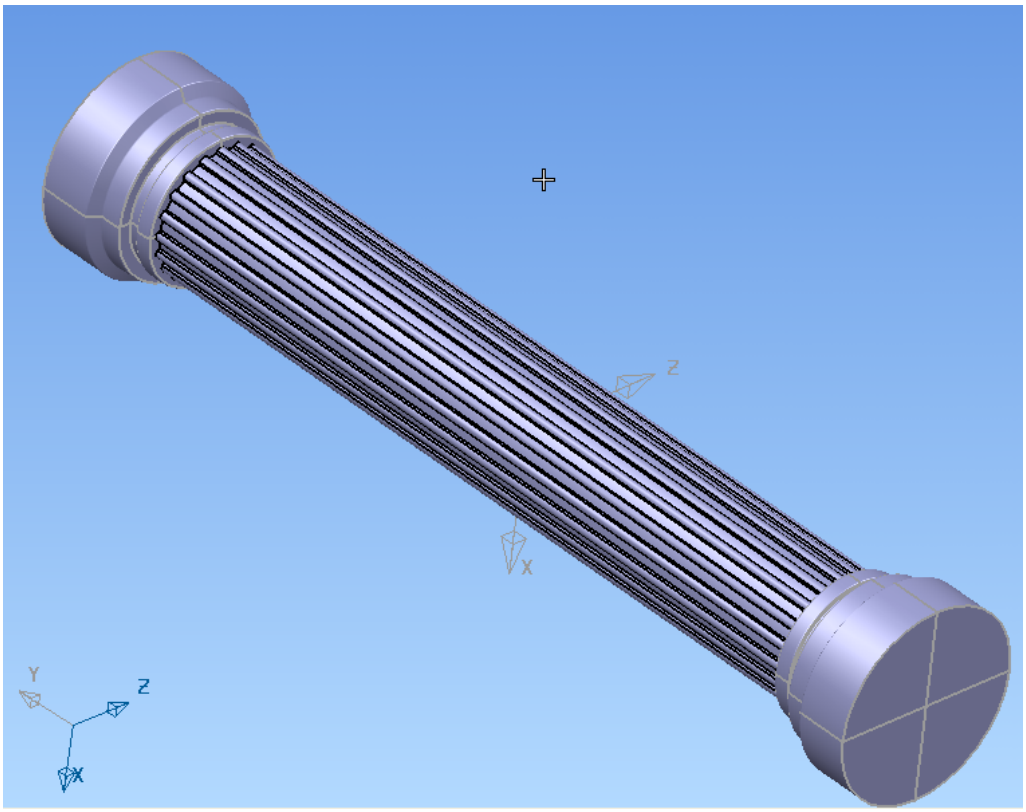
The default pattern type is a simple rectangular array. For this column we want a circular layout.



Change the number of flutes to the value you want.



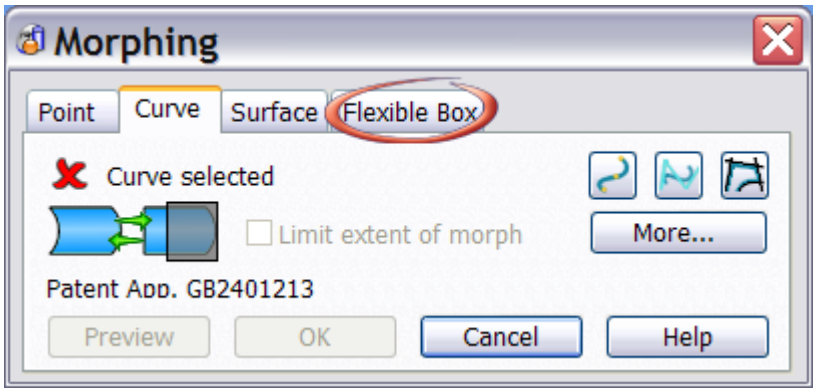
Click ‘OK’ when you are ready.



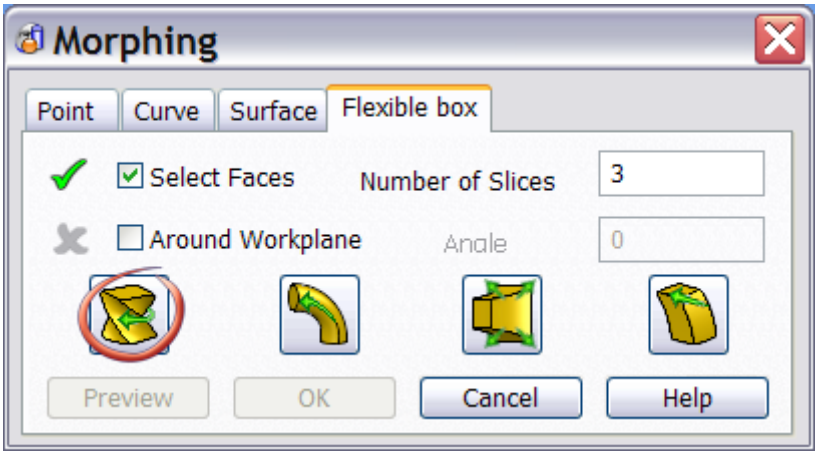
Twisting the Flutes

To make the flutes more interesting, we will twist the whole solid using Morphing.

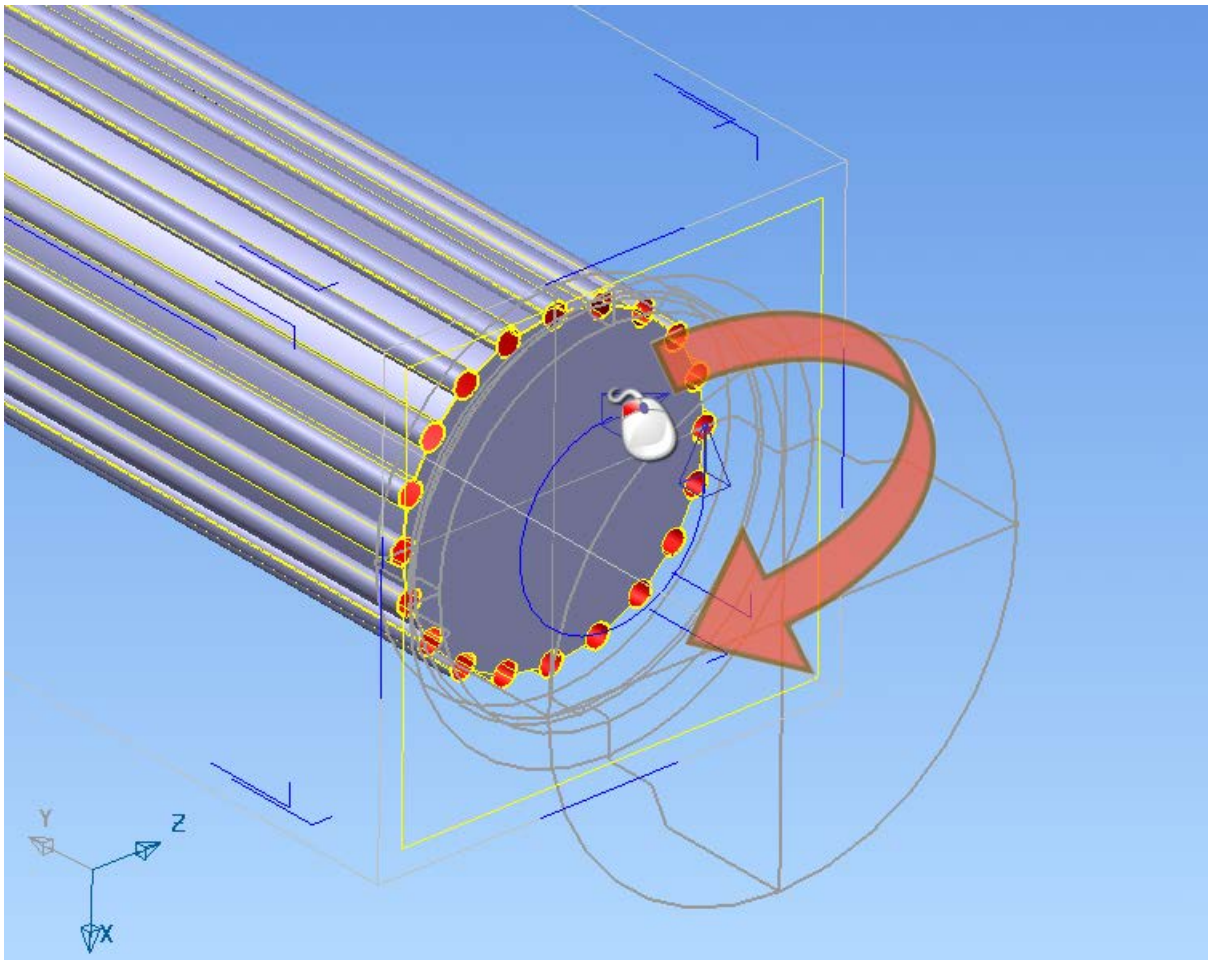
Click the Solid Morph icon



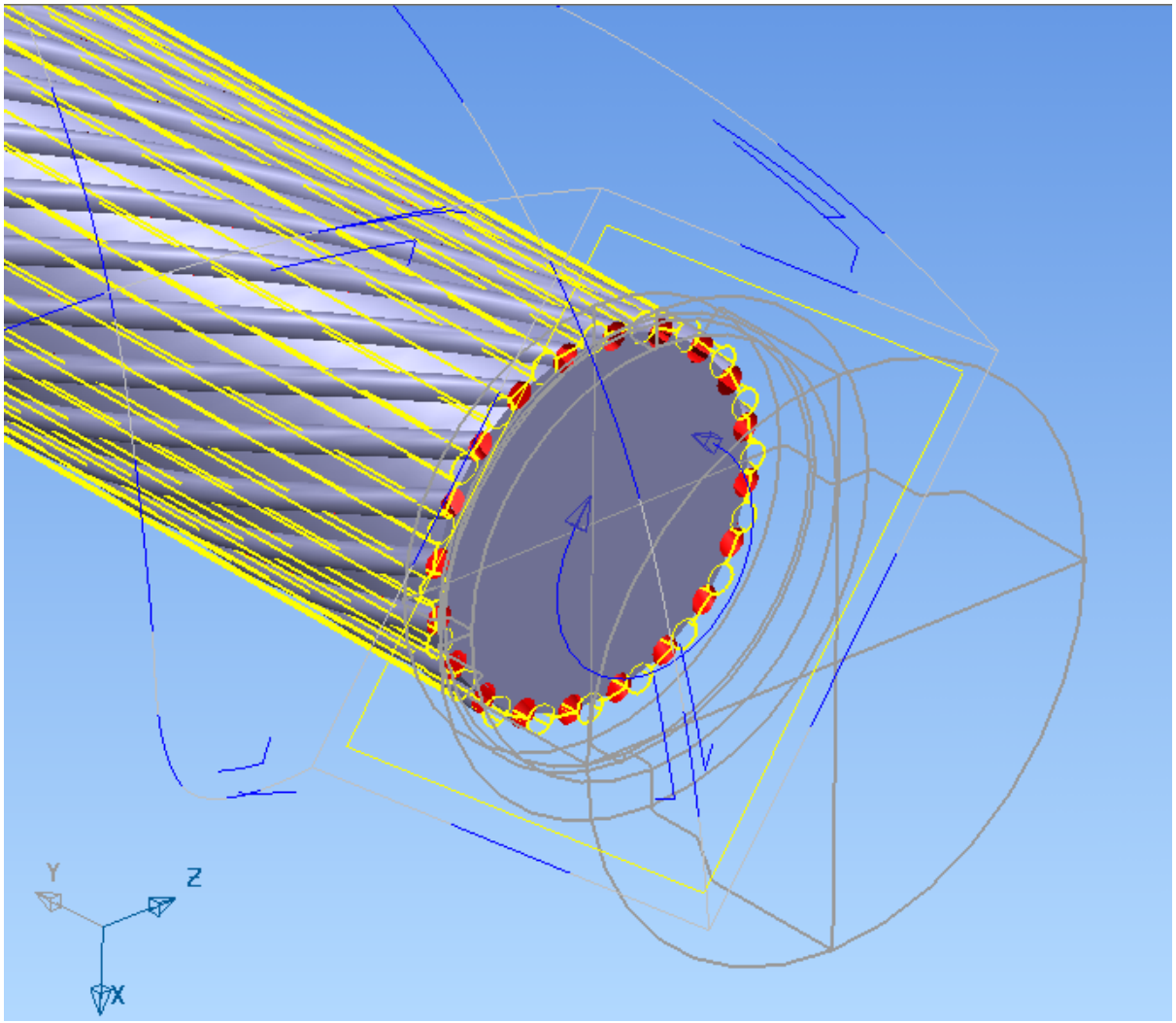
Change to the ‘Flexible Box’ pages, then click ‘Twist’.



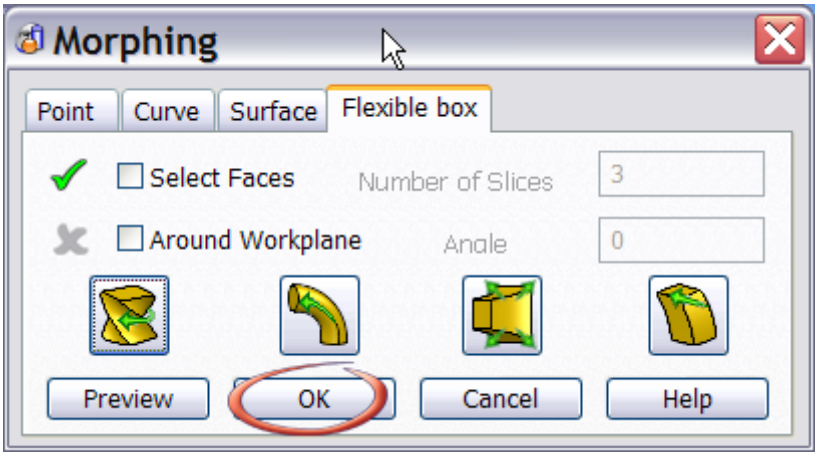
'Drag Handles' will be drawn onto the solid, allowing you to control how the shape is to be deformed. Grab the rotation handle at the bottom of the solid (it will already be highlighted). Holding the mouse down, rotate the end as many times as you want. Notice that this action is very dynamic, and that you can twist the shape through as many revolutions as you wish. Note also that you can rotate in either direction, and can also 'rewind'.

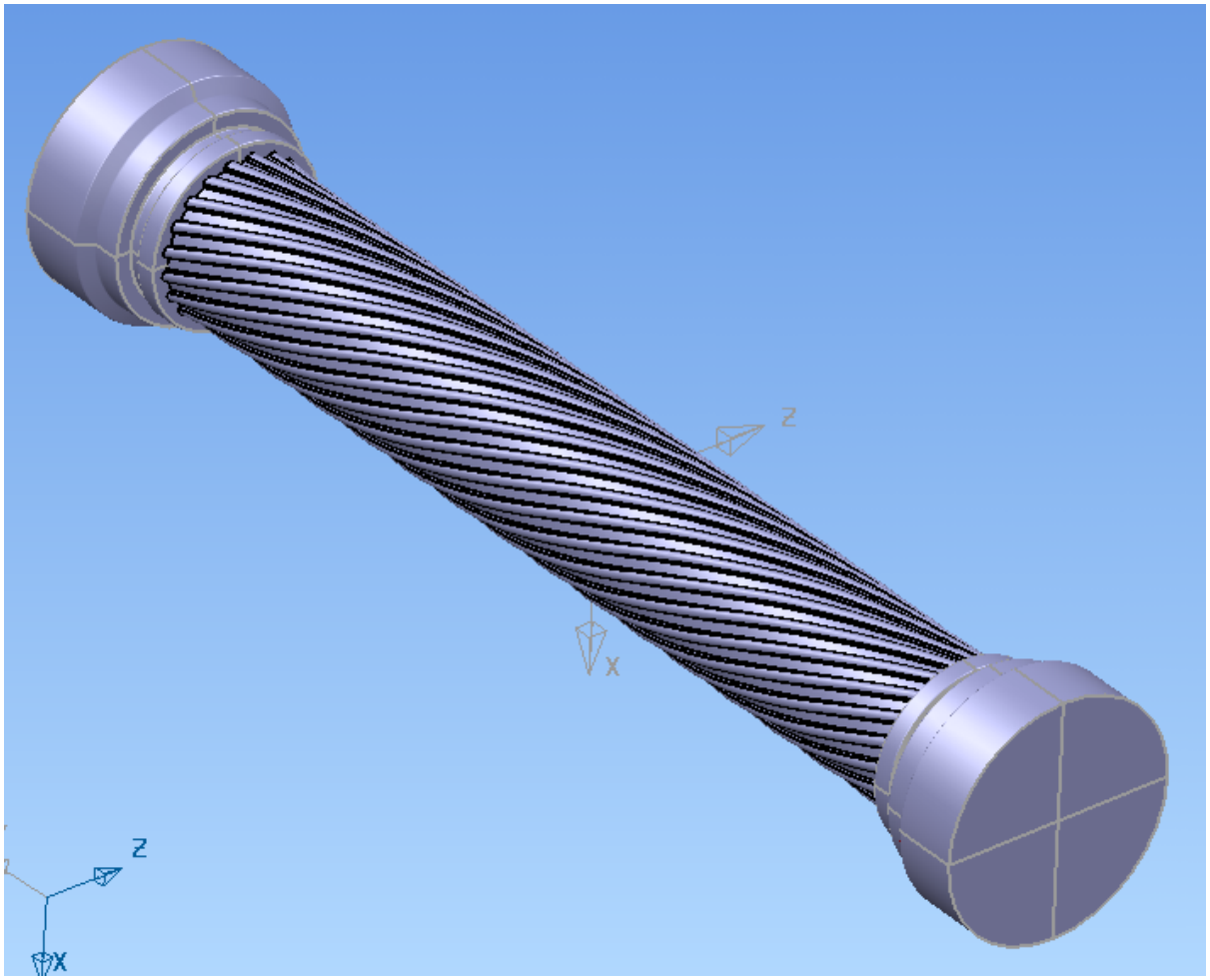


When you are happy with the shape, simply let go of the mouse button.



To finish, click OK.



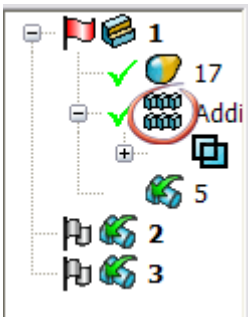


Changing the Flutes

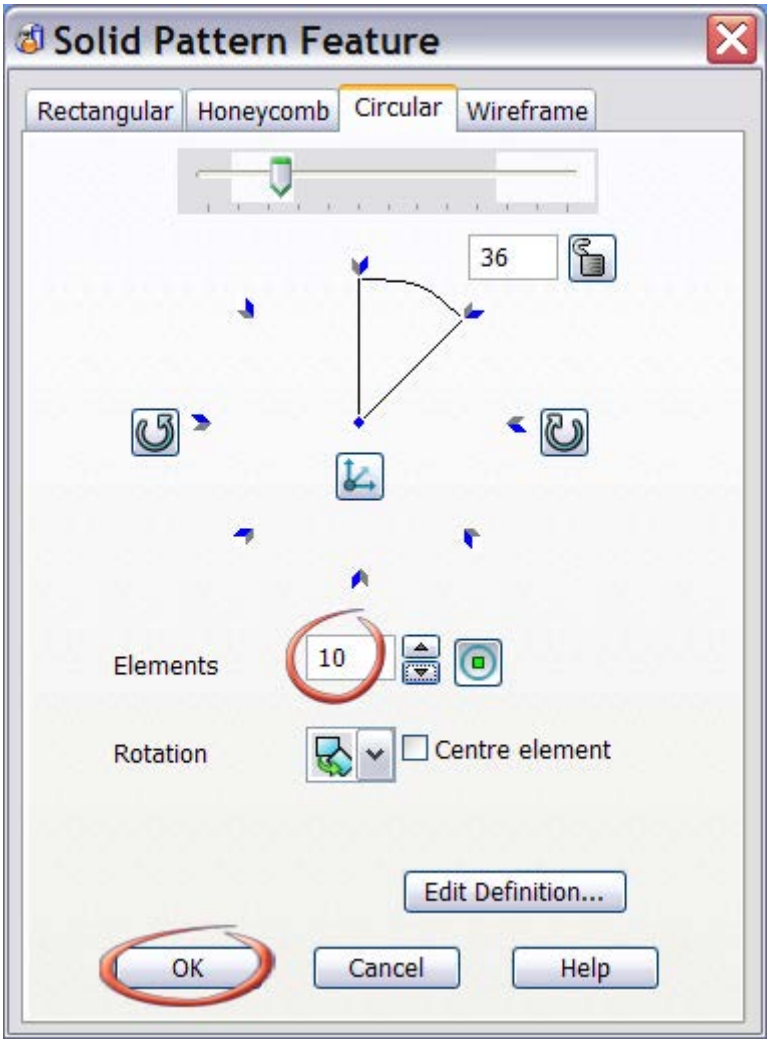
There are two feature-based changes that can be made at this stage.

Changing the Number of Flutes

Double-click the ‘Pattern’ feature in the history tree.



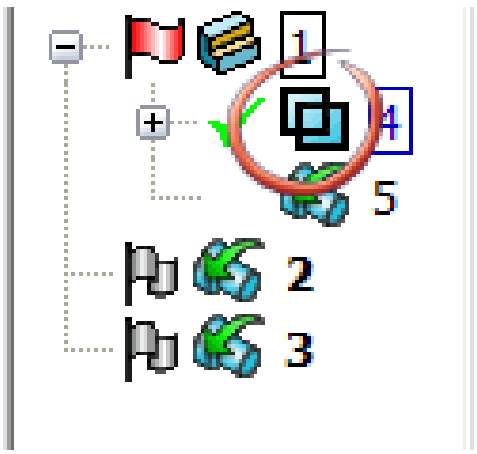
This will re-open the pattern form.



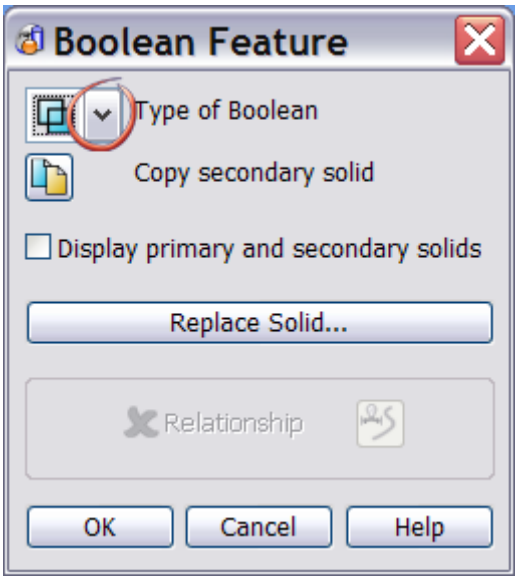
Change the number of copies, and click OK. The solid will automatically update, including the morph.

Changing the Flute Type

Next you will change the flutes from protrusions to cut-outs. Again, because the part has been constructed from solid features, this is a very simple edit.

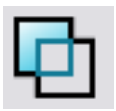


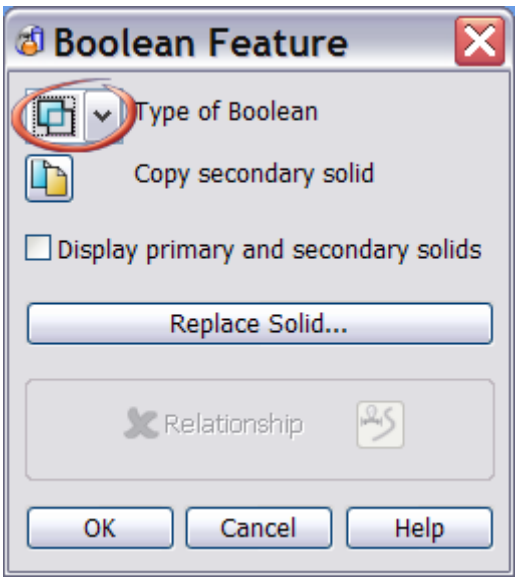
Double-click the initial solid Addition feature in the tree to open the edits form.



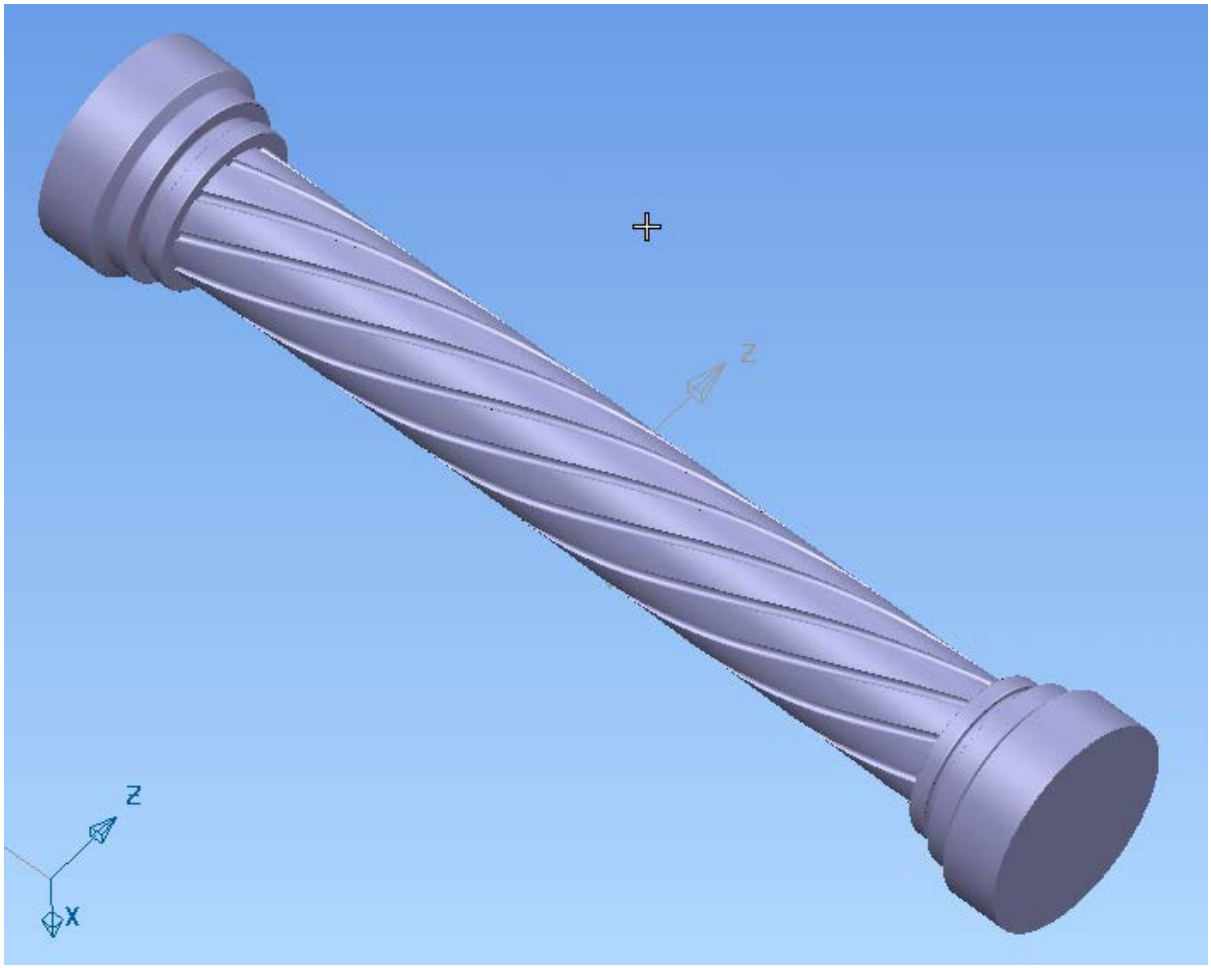
Open the drop-down list marked ‘Type of Boolean’.

From the available list, select a Subtract feature.





Click OK, and the solid will update. Now, instead of the flutes being raised from the surface of the column are cut out.

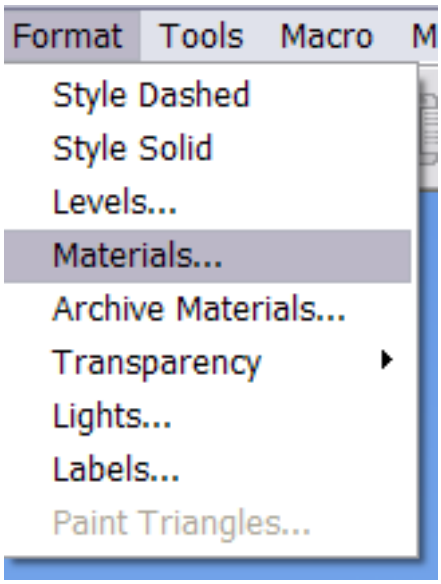


End of Tutorial

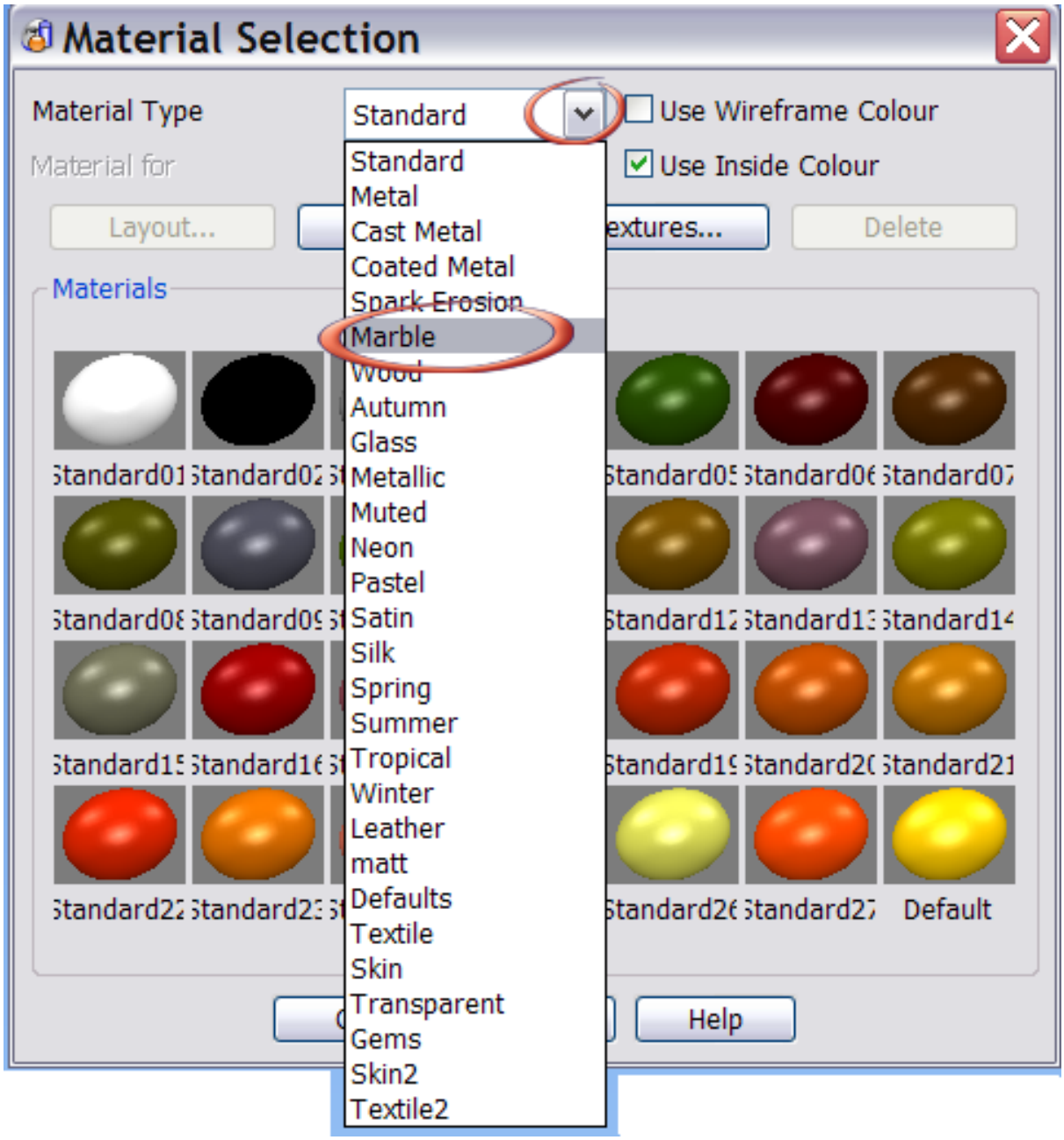
Optionally you may wish to create a rendered image.

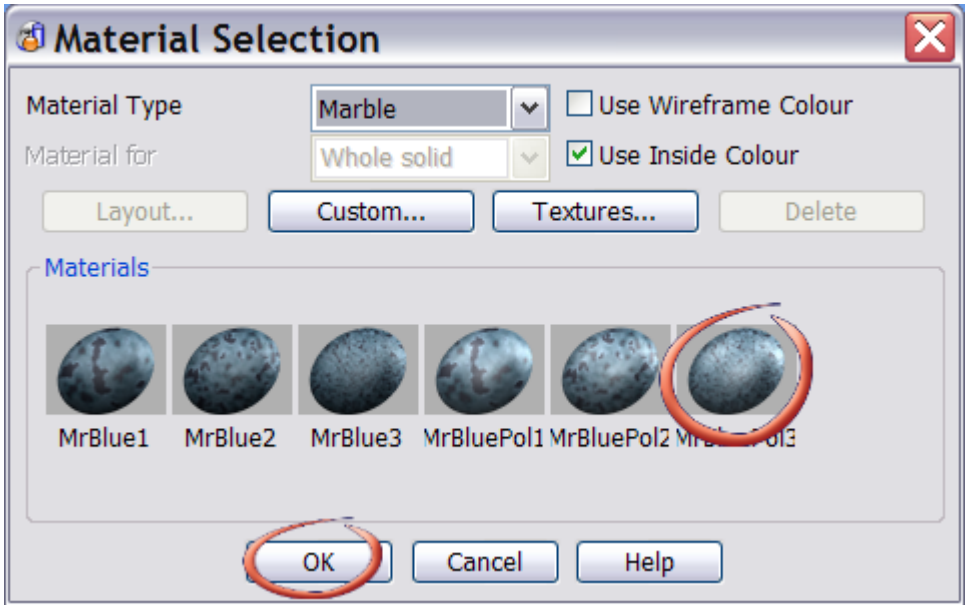
Select everything (Ctrl + A)

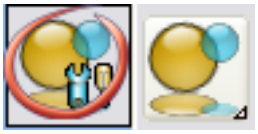
Open the materials form



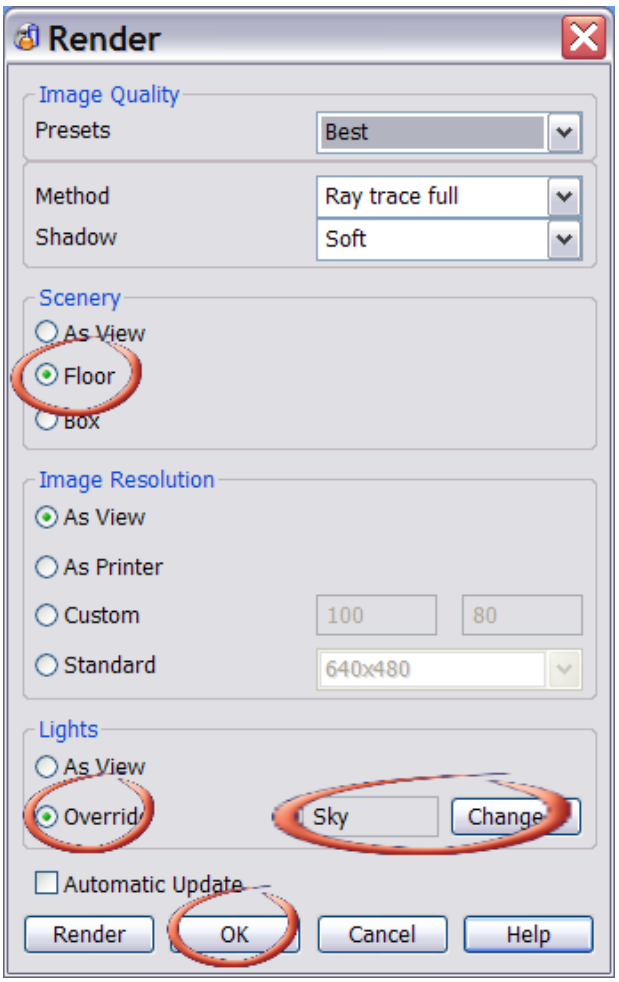
Set the Material Type to Marble



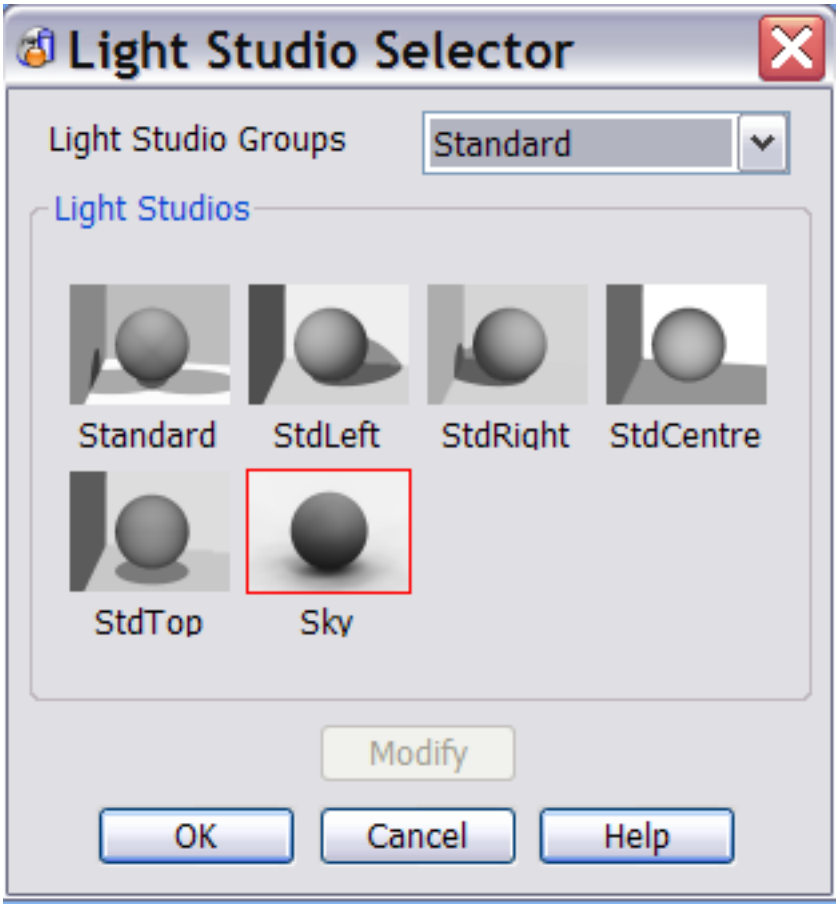


Open the Render Options page 

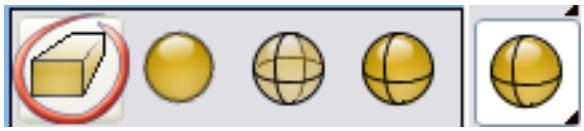
Set the options as shown below.



Note that when you click the ‘Change’ button on the Lights panel a new form will open, allowing you to select the light studio you want to use.



Set the view type to ‘Enhanced’



Finally, click the ‘Render’ icon.

NOTE: Creating a rendered image using the ‘Sky’ lighting studio can take some time, particularly in a full size window. When making rendered images of large, complex scenes, it is worth making a test render in a smaller window first. This helps ensure the final full-size render is correct, without wasting lots of time. A quick way to do this is from the Render Options form. In the ‘Image Resolution’ panel, click the ‘Standard’ button, and choose an appropriate size. The rendered view will appear in the lower left corner of the main view.

Save the model in **D:\users\Training\Coursework\Delcam Designer Jobs** as **Decorative Column.psmodel**

Exercise – A Guitar Body

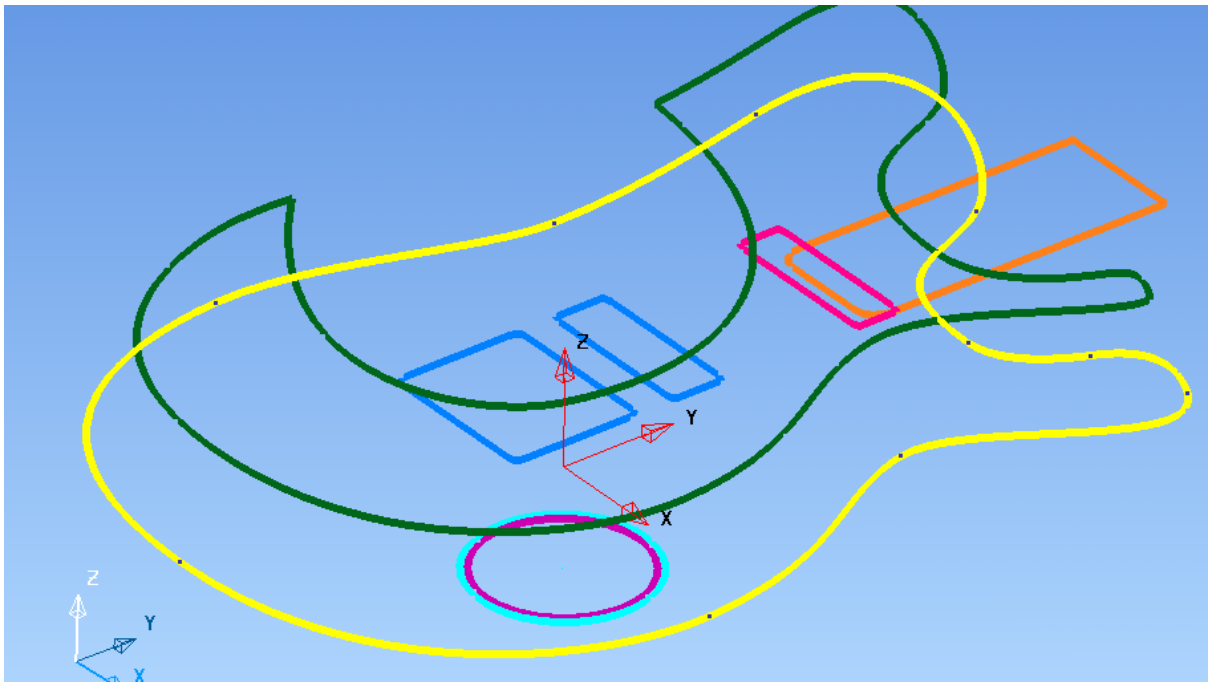
Introduction

This tutorial will show how to construct the complex guitar body shown from existing wireframe.

- Key feature of this example –
- Creating simple solids
- Solid fillets
- Flexibox morphing
- Solid Boolean operations
- Rendering

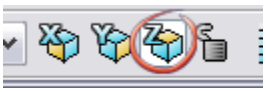
Creating the Body

Open the model ‘Guitar’. (If it is not already on your computer click ‘Import’ and browse to the data folder that contains the model **(Guitar.psmodel)**

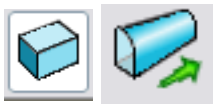


The model contains 8 curves.

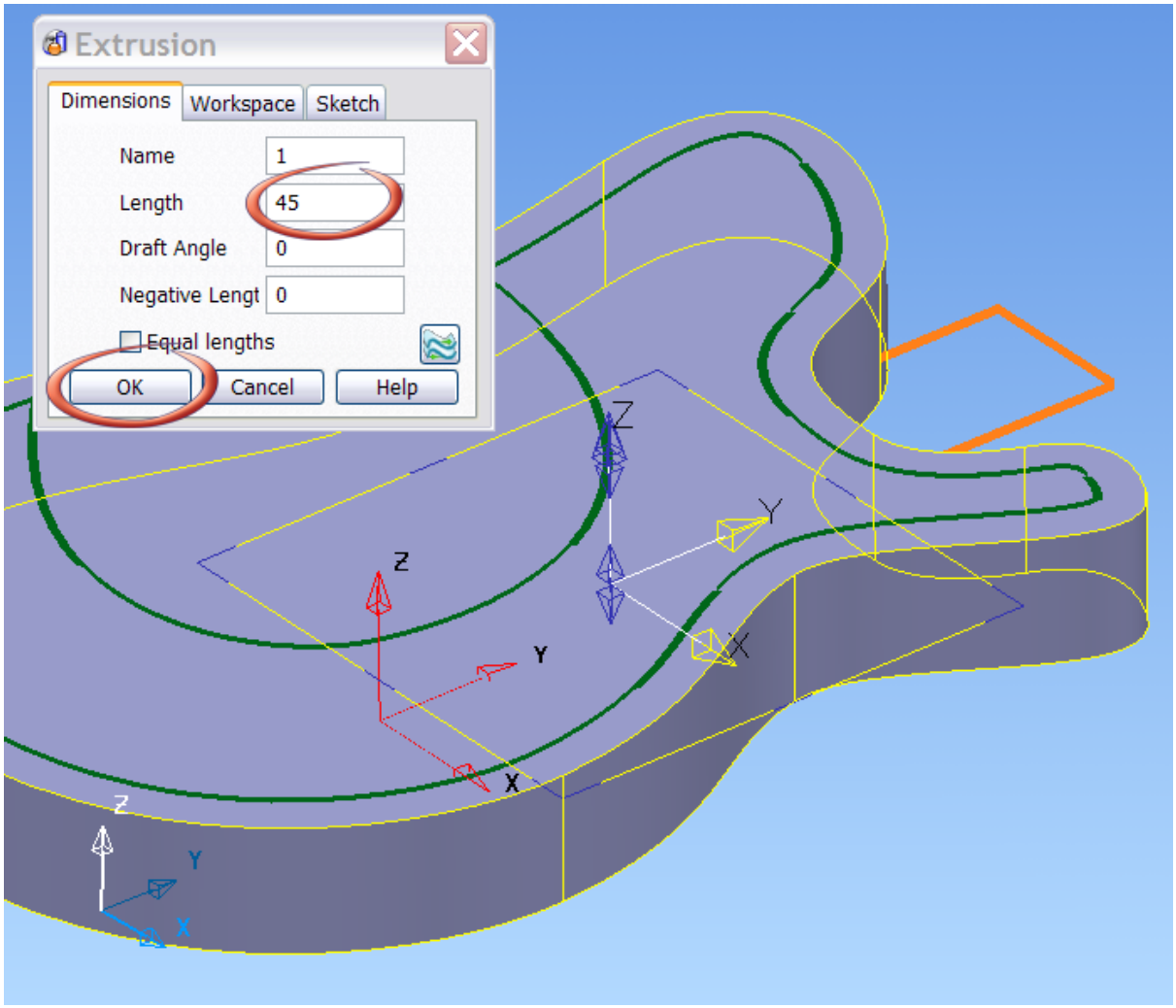
Make sure the working axis is Z.



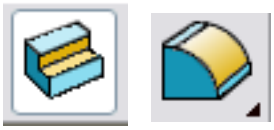
Select the main outer profile, as shown above, and make a solid extrusion.



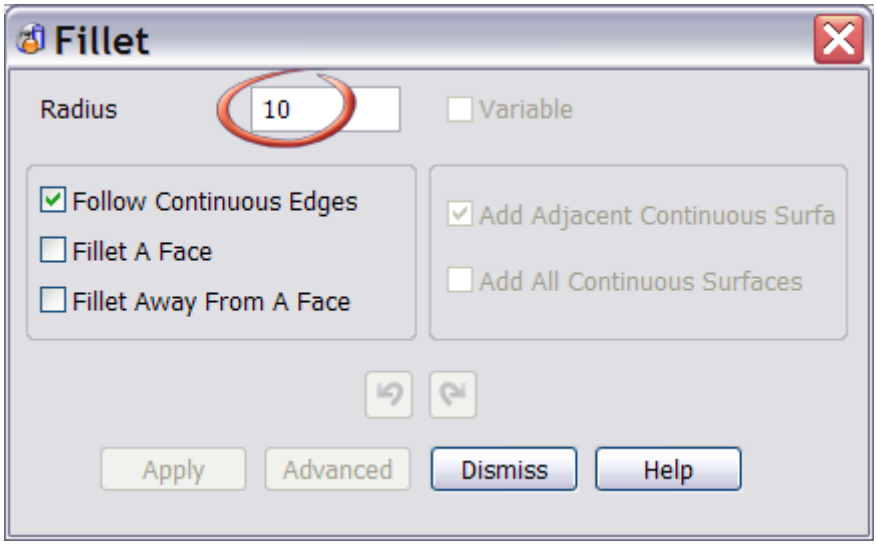
Set the length of the extrusion to 45mm



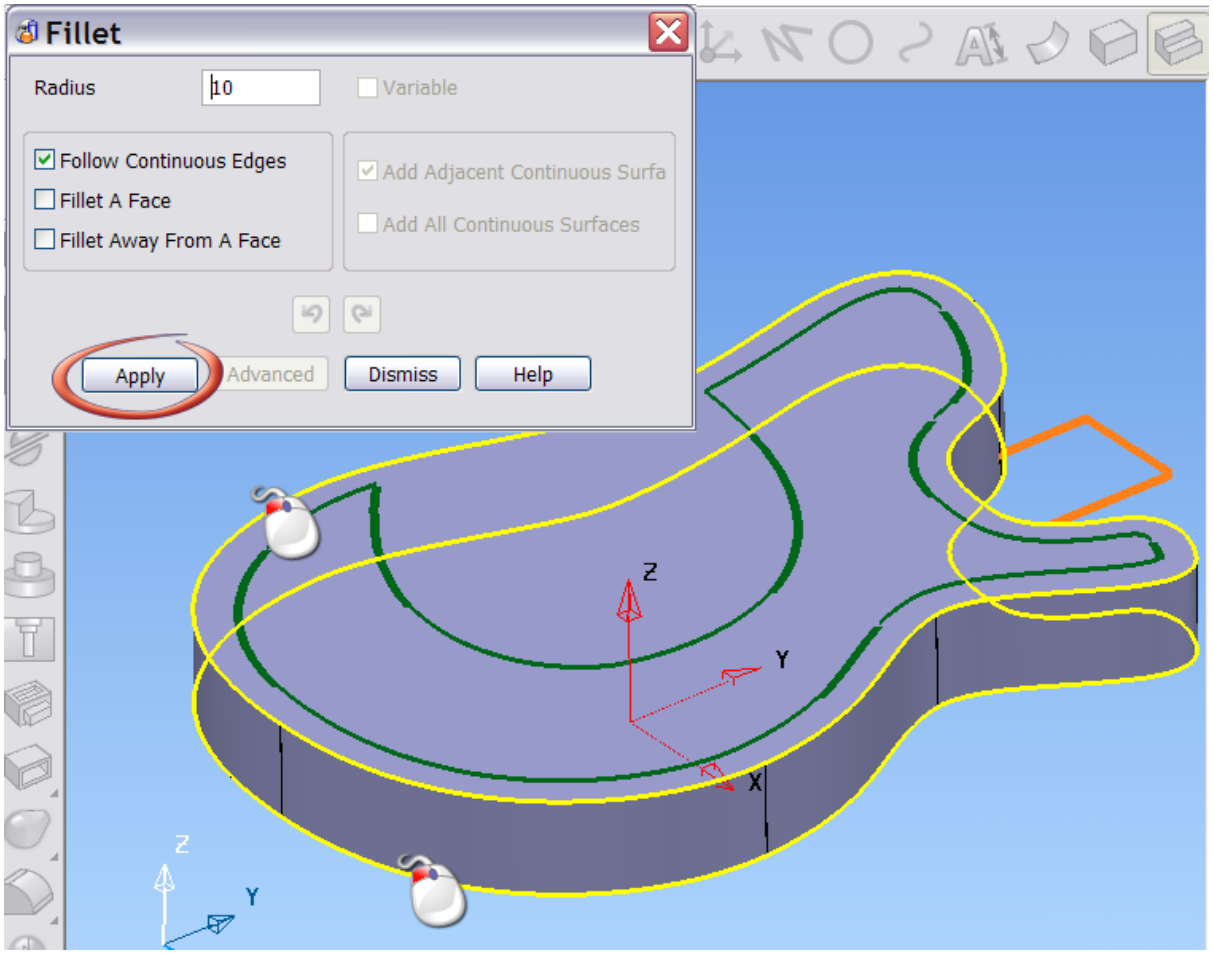
Activate solid filleting.



Set the fillet radius to 10

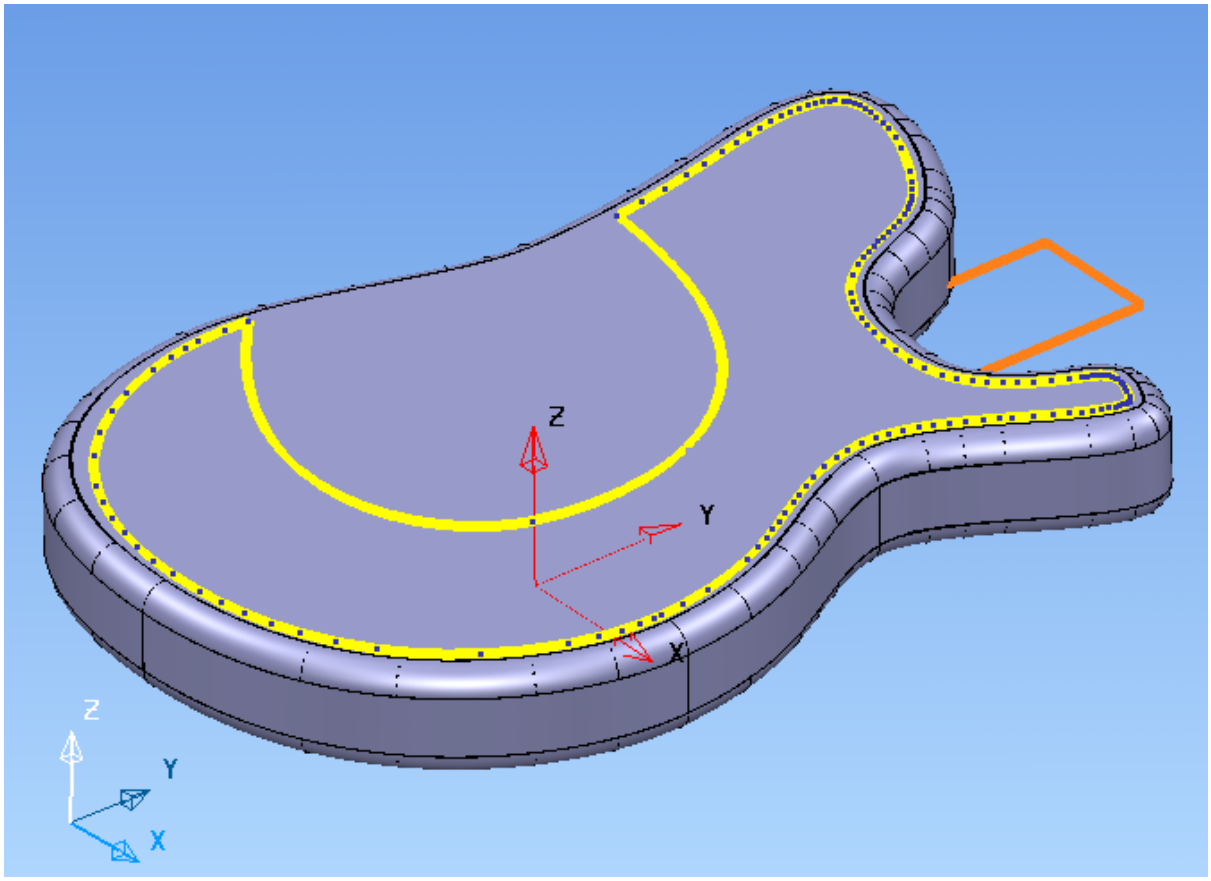


Select the two edges highlighted below. Click either edge with the left mouse button, then SHIFT-Click the other to *add* it to the selection.



When both edges are selected click the ‘Apply button’.

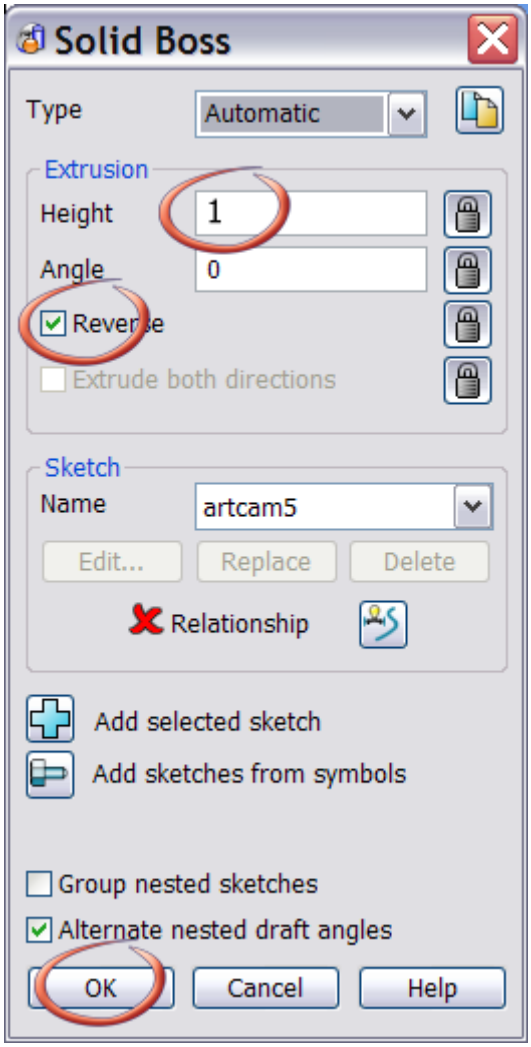
‘Dismiss’ the fillet form.

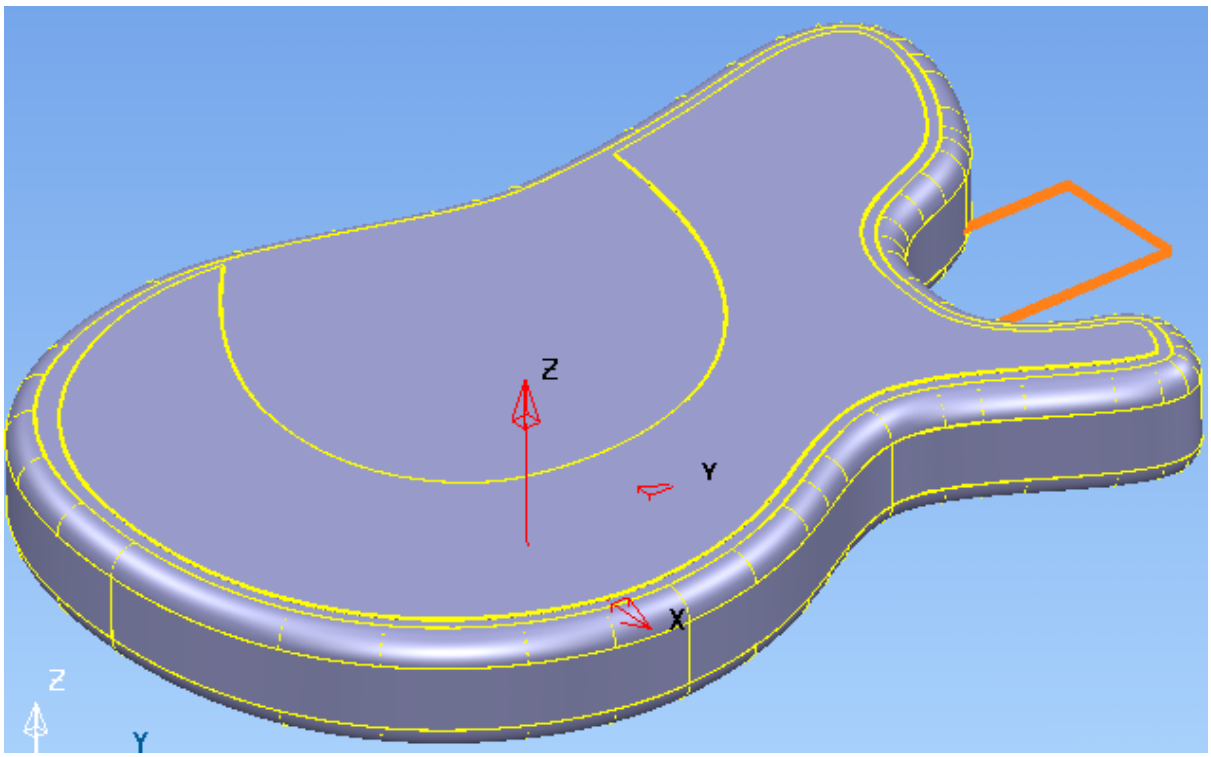


Adding the Scratch Plate

Select the curve highlighted above, and create a solid boss feature

Set the Height of the Boss to be 1mm, making sure that it points *away* from the guitar body (i.e. it is protruding, not recessed). If necessary, click the ‘Reverse’ button to change the direction. When you are ready, click ‘OK’.

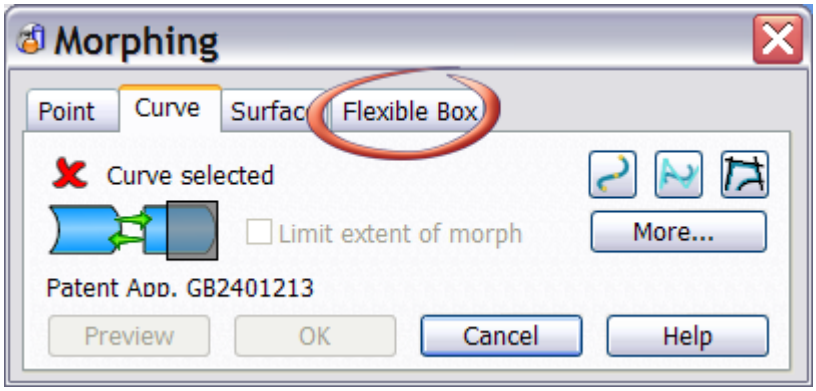

Solid Boss
Type
Automatic
Extrusion
Height
1
Angle
0
Reverse
Extrude both directions
Sketch
Name
artcam5
Edit...
Replace
Delete
Relationship
Add selected sketch
Add sketches from symbols
Group nested sketches
Alternate nested draft angles
OK
Cancel
Help


Z
Y
X
Z
Y

Sculpting the Body using Morphing

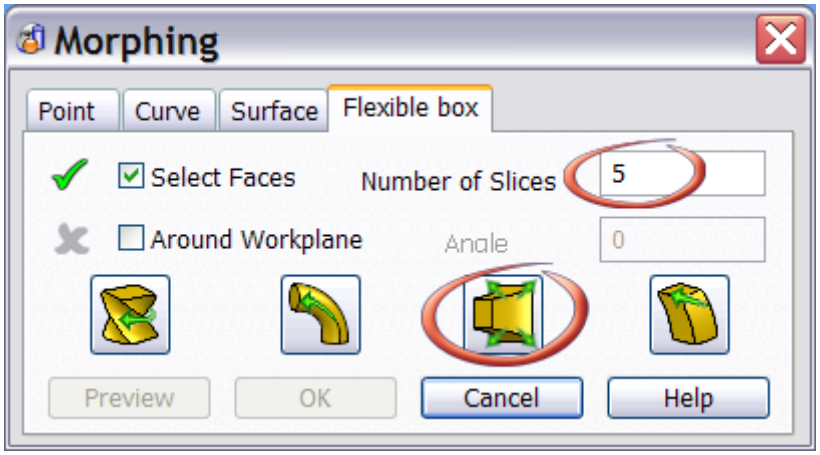
The body is currently flat, and fairly uninteresting. Use the Morphing tools to make the body much more exciting.

Start the Morphing tool by clicking

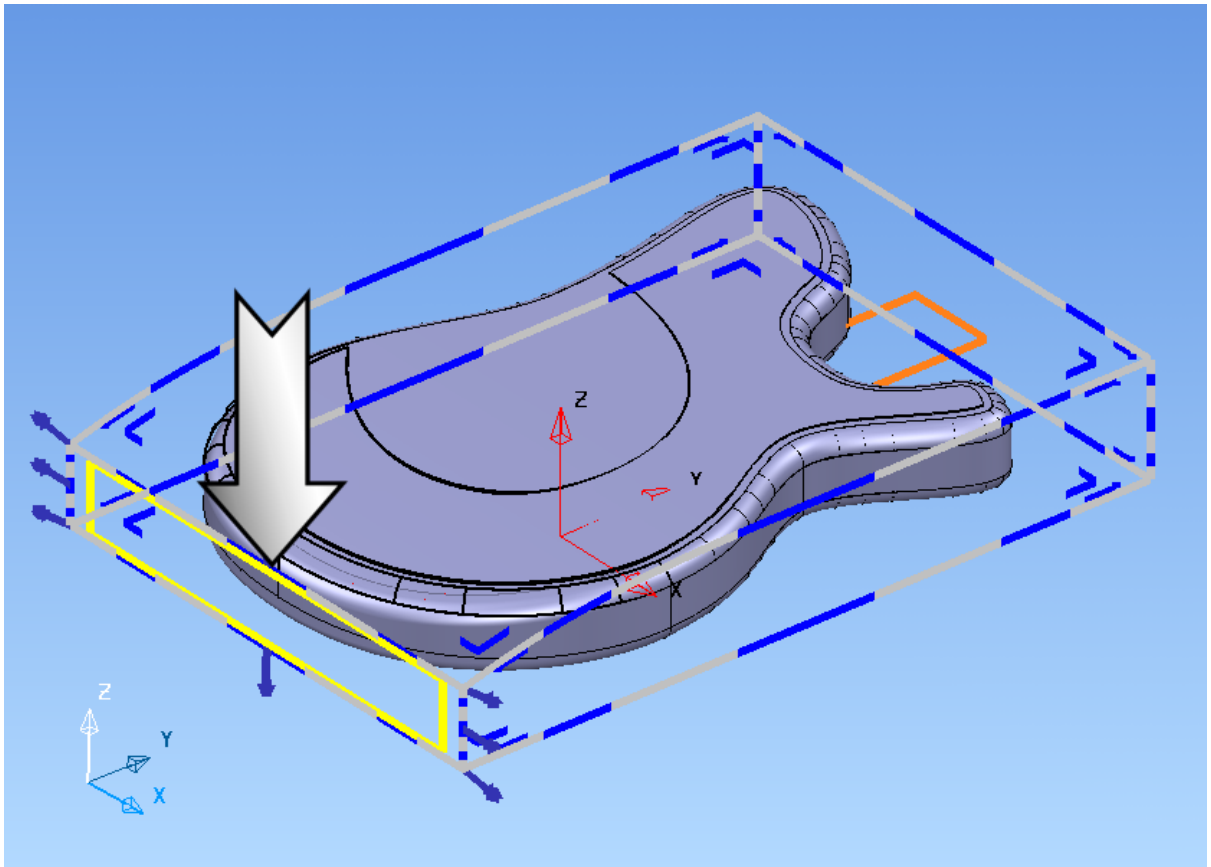


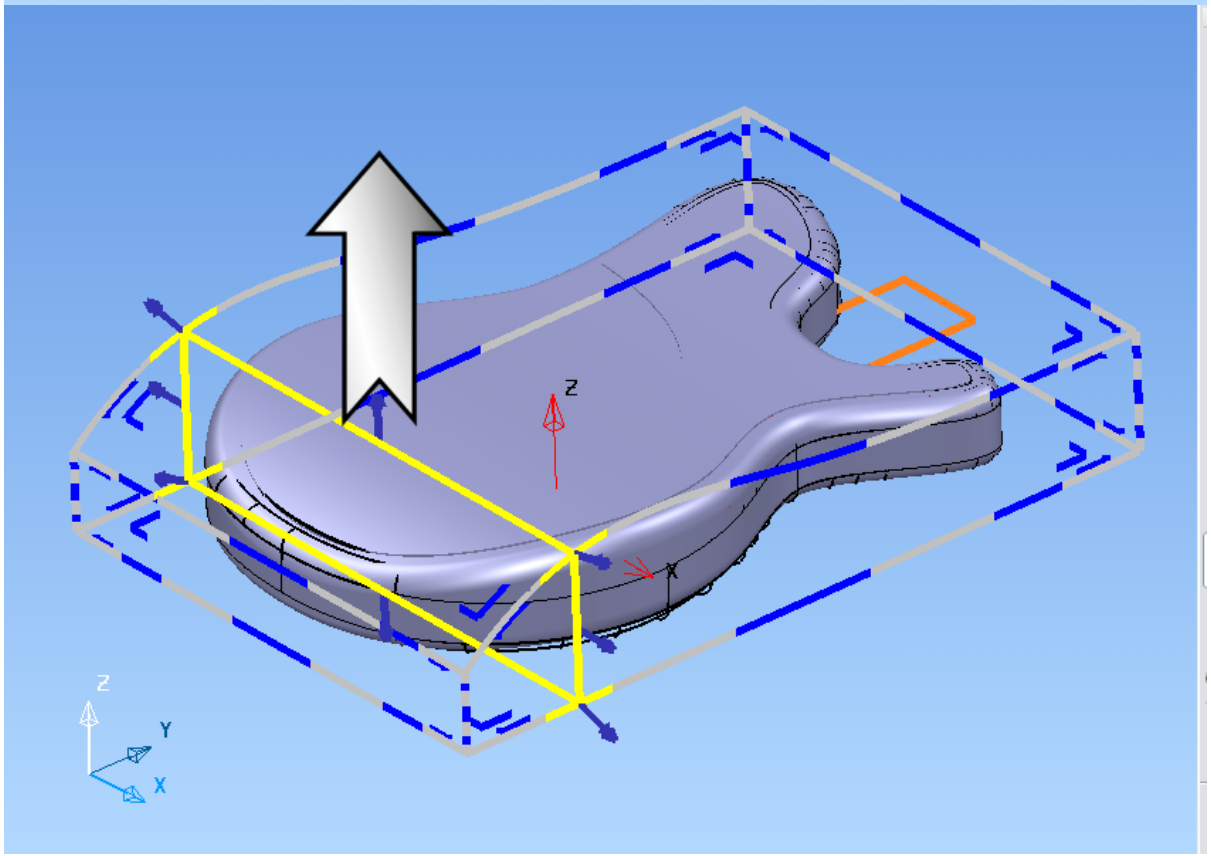
Select the Flexible Box (FlexiBox) tab.

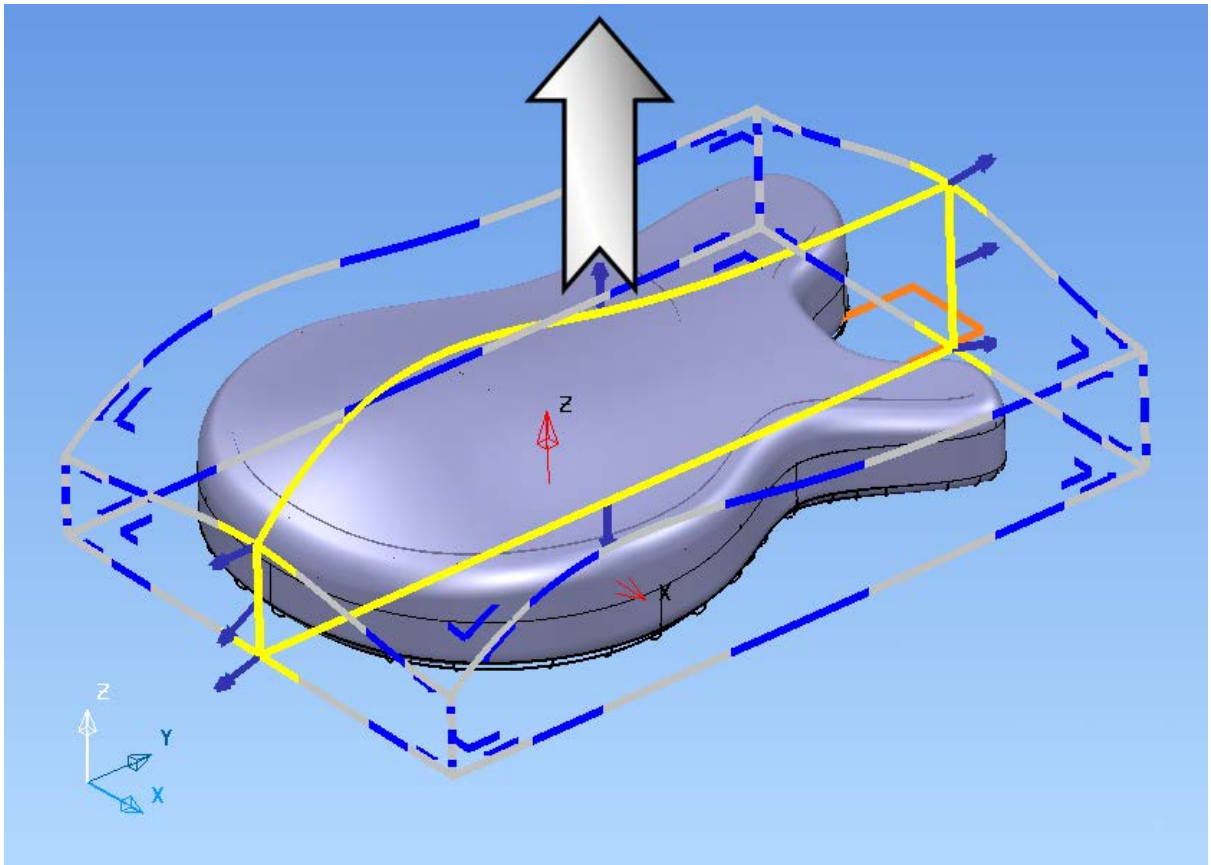
Set the number of slices to 5 (to achieve finer control) and click the ‘scale’ icon.

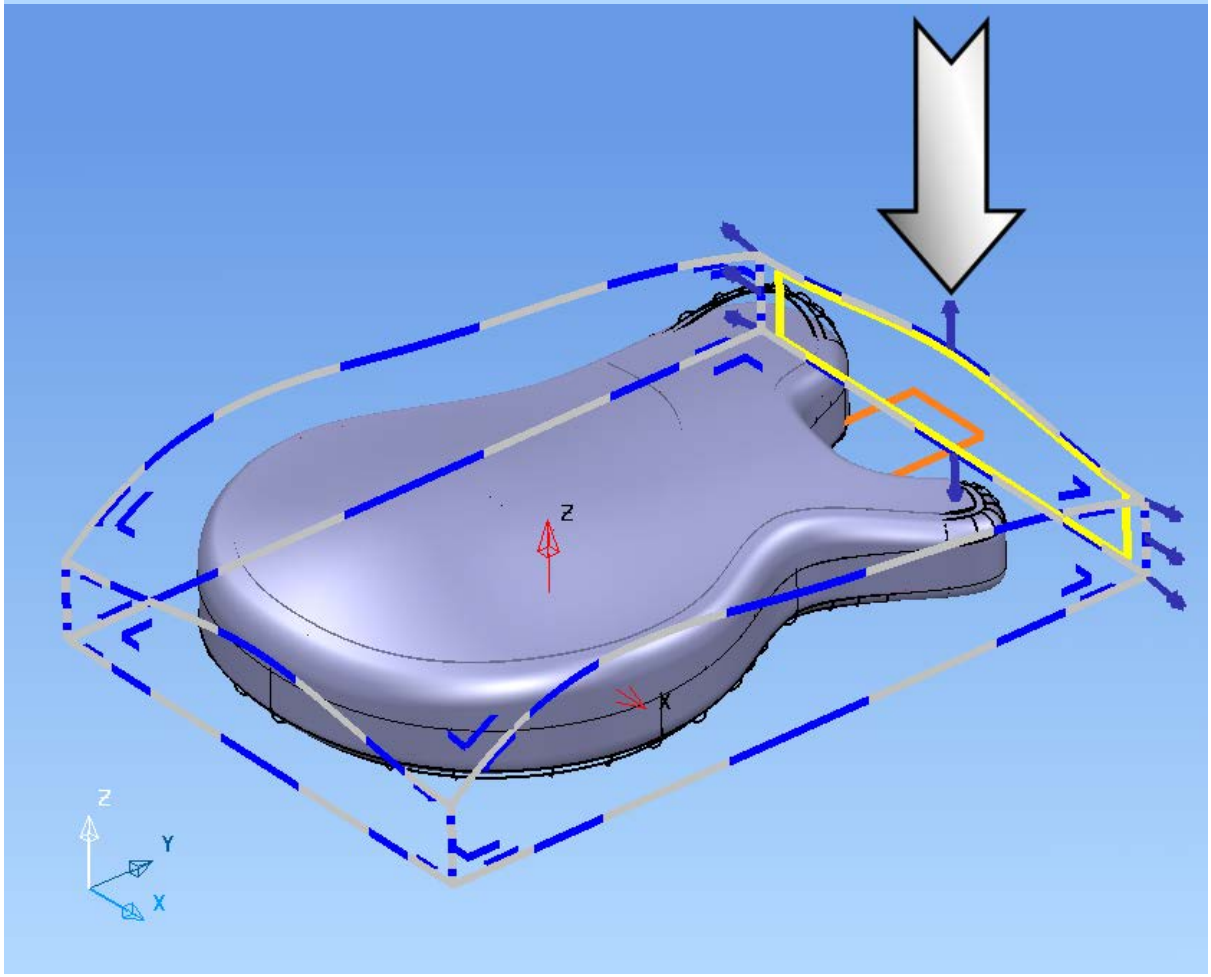


For the purposes of this tutorial example, the body was morphed in four stage as shown below. To select a different ‘plane’ to work on, select one of the **BLUE** grab handles on the instrumentation box.


Z
Y
Z
Y
X

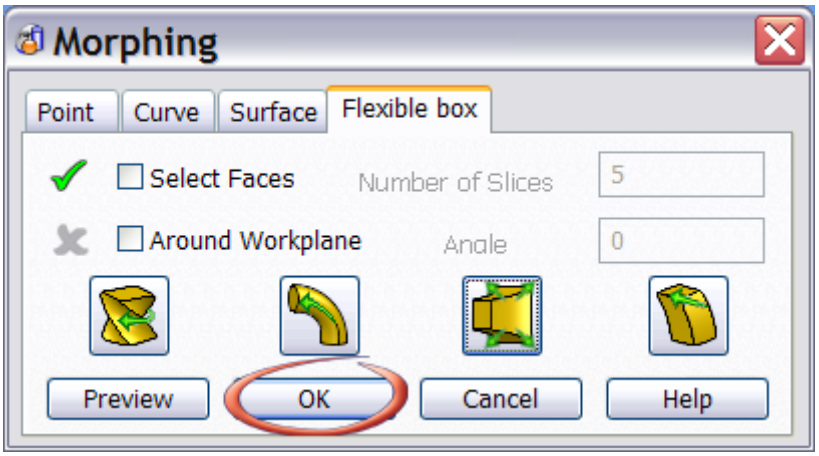

Z
Z
Y
X

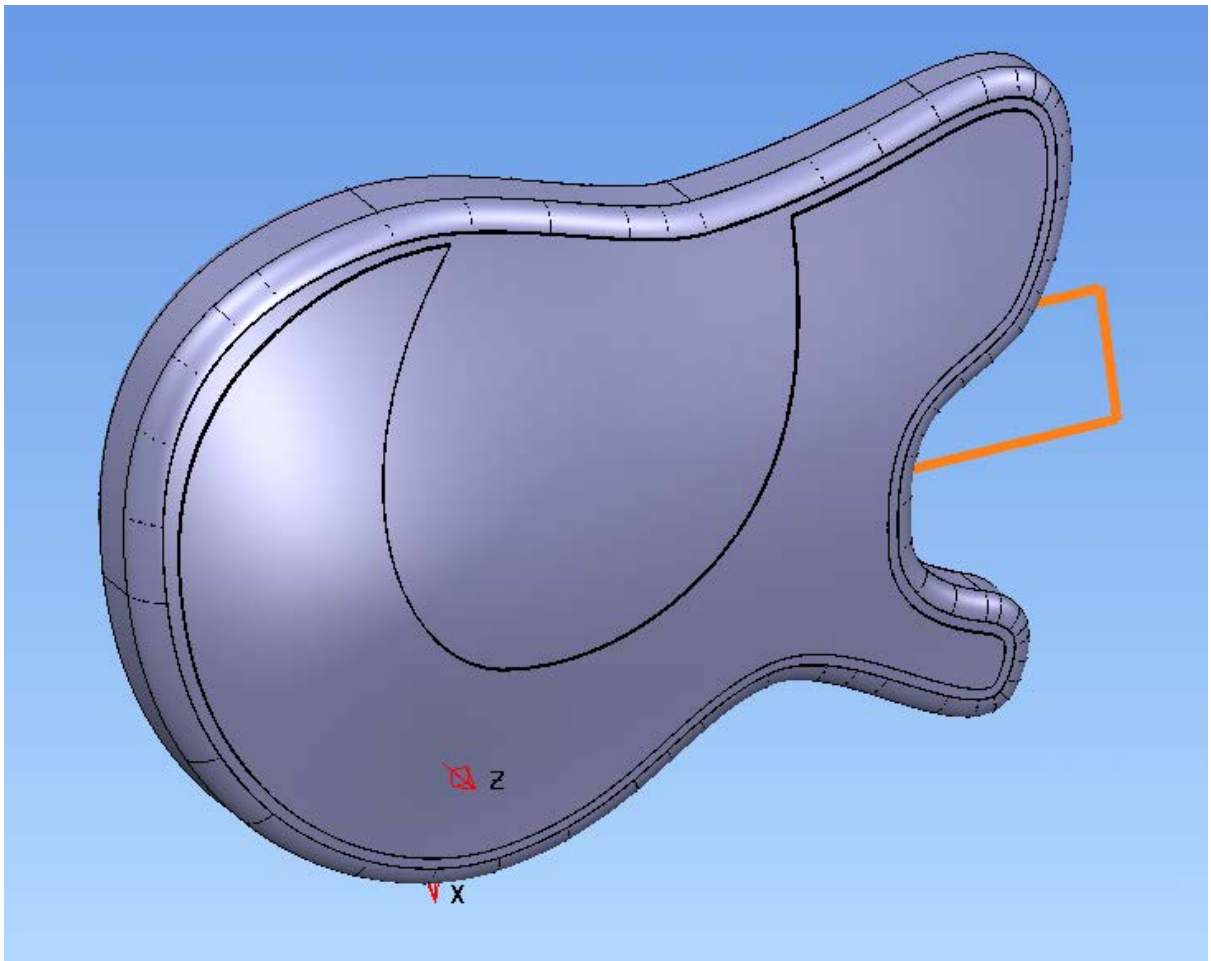

Z
Z
Y
X


Z
Z
Y
X

Feel free to experiment with your own shape.

When you are happy with the result, click OK.

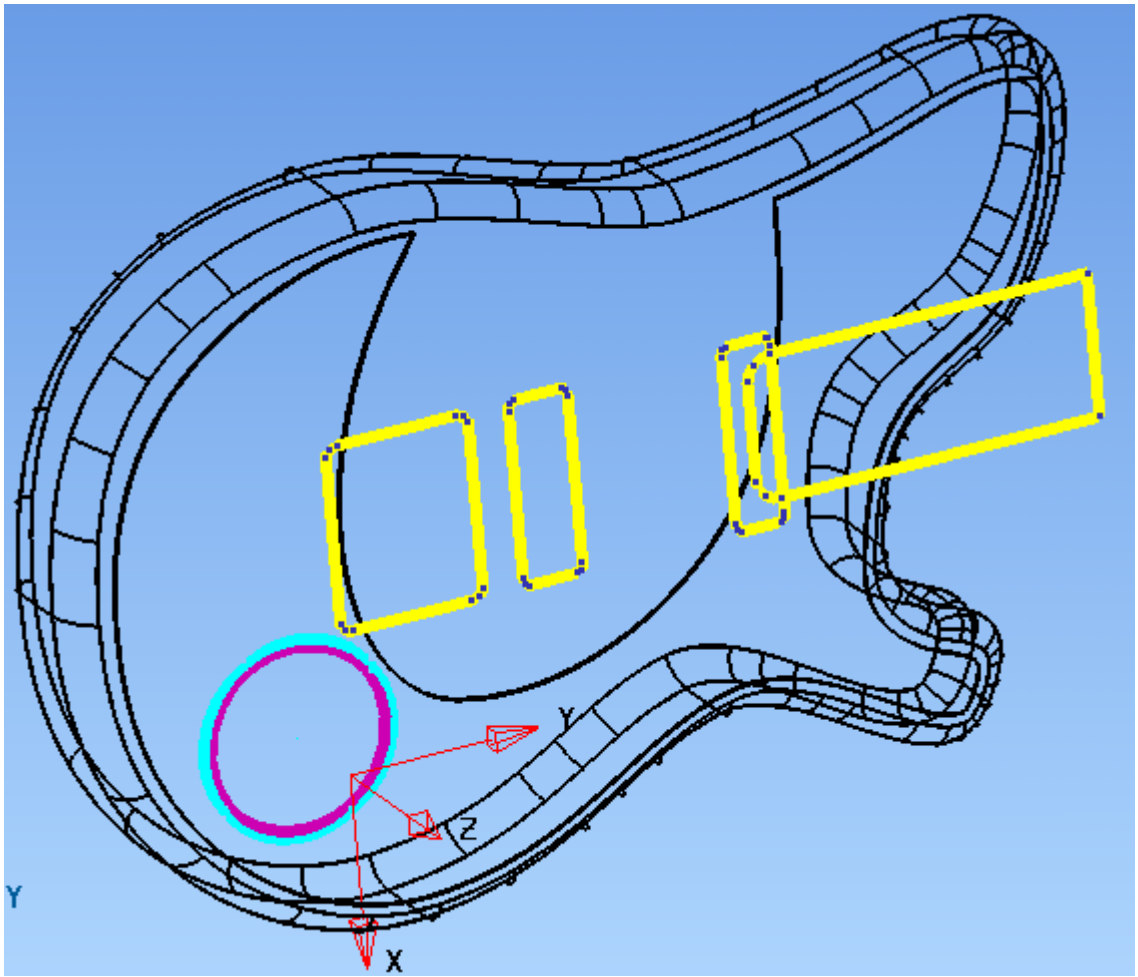




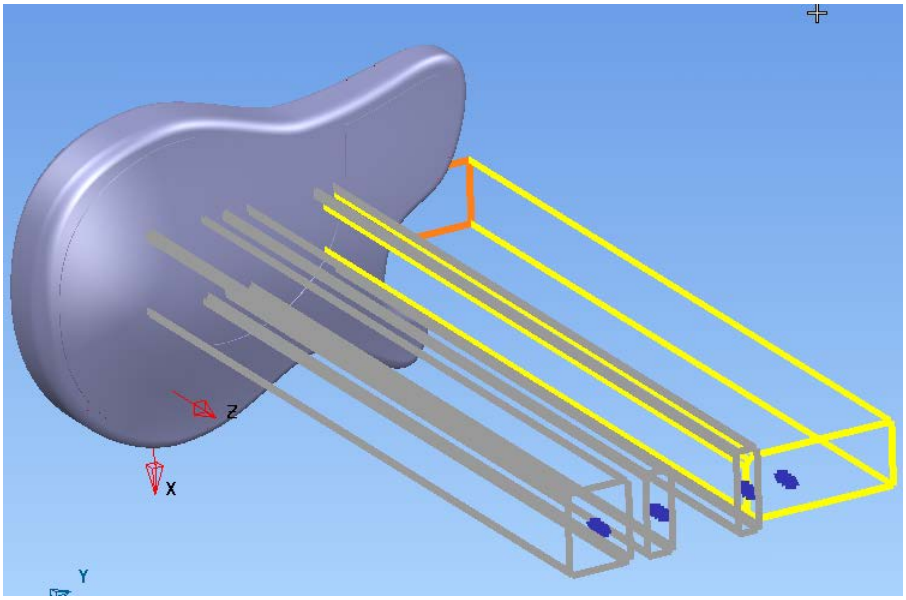
Creating the Cutouts

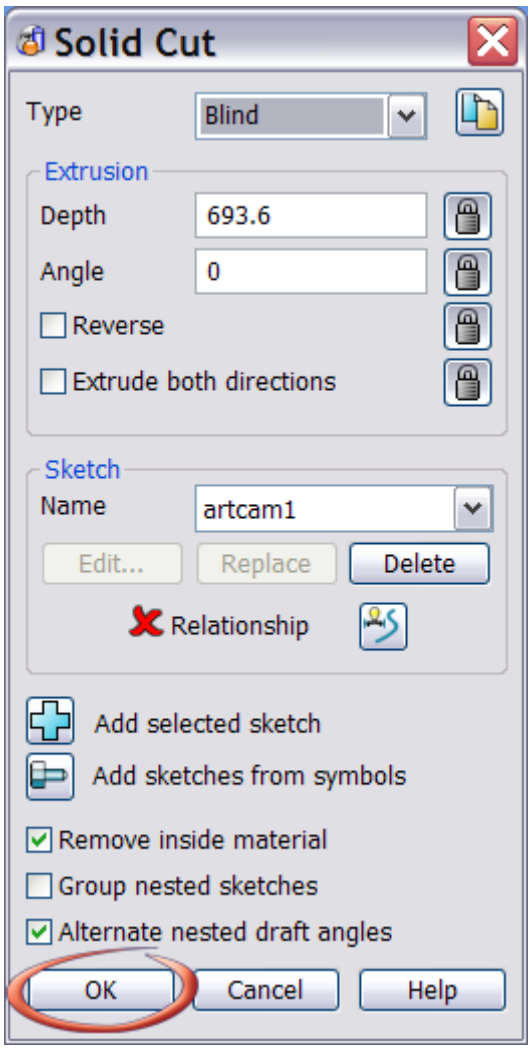
To finish the guitar body several cutouts are needed, to accommodate fitting the neck and the electronics.

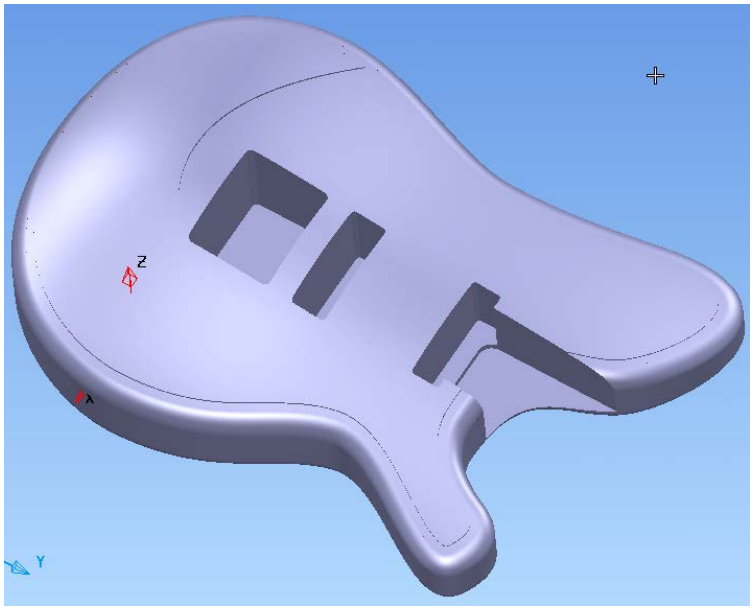
Select the four rectangular curves shown below. (Use the **shift** button to multi-select)



Use solid ‘Cut’ to remove unwanted material.

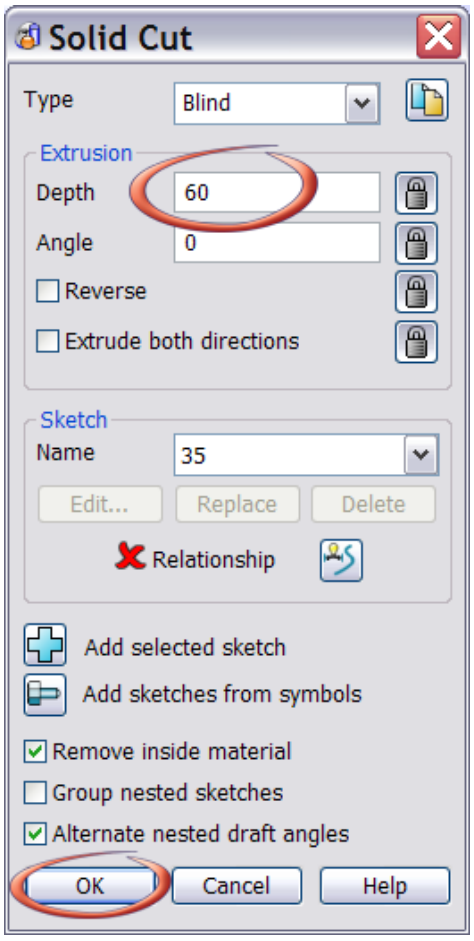

Z
X
Y


Solid Cut
Type
Blind
Extrusion
Depth
693.6
Angle
0
Reverse
Extrude both directions
Sketch
Name
artcam1
Edit...
Replace
Delete
Relationship
Add selected sketch
Add sketches from symbols
Remove inside material
Group nested sketches
Alternate nested draft angles
OK
Cancel
Help

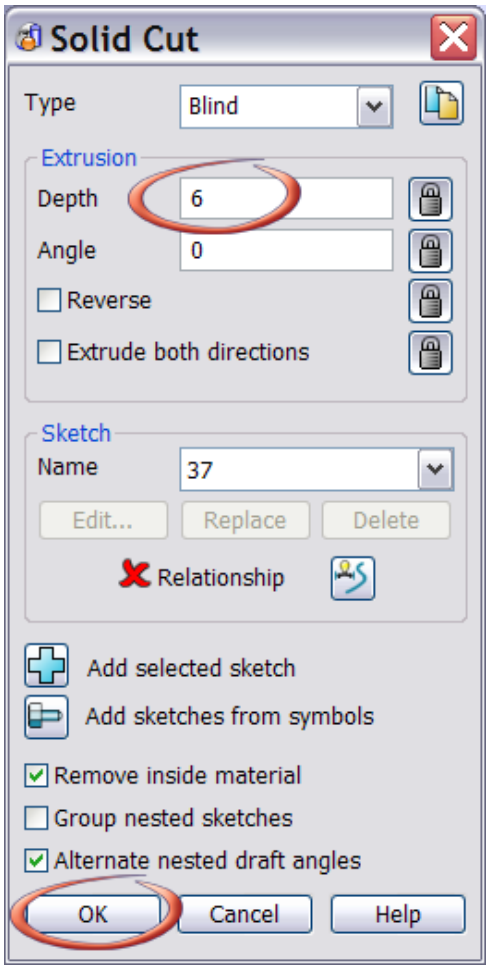


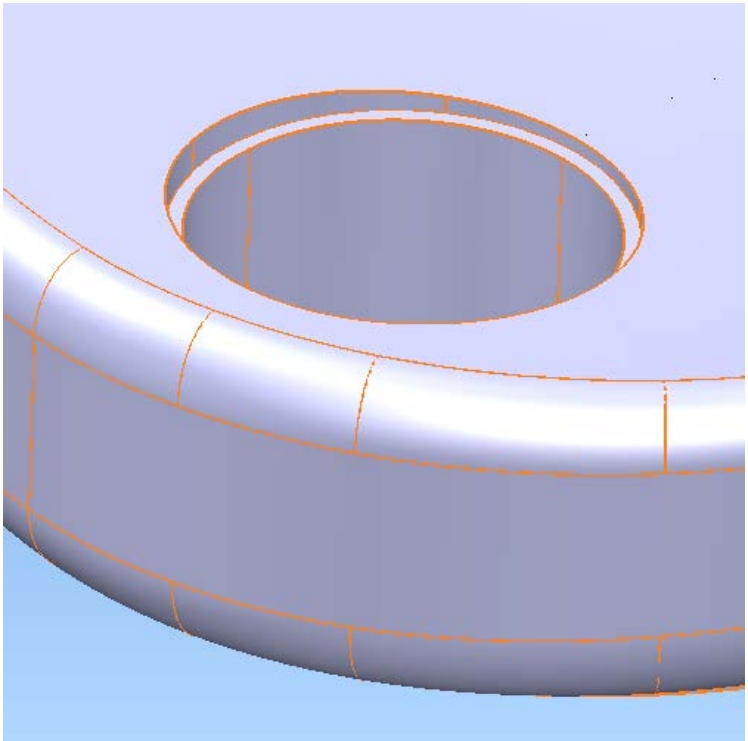
The solid cut features in the back of the guitar must be made separately as they are different depths, although they can be done in any order. Also note that the exact depths you use will probably be different from those shown here, due to differences in the morphing.

For the inner circle (smaller) enter a **depth of 45**



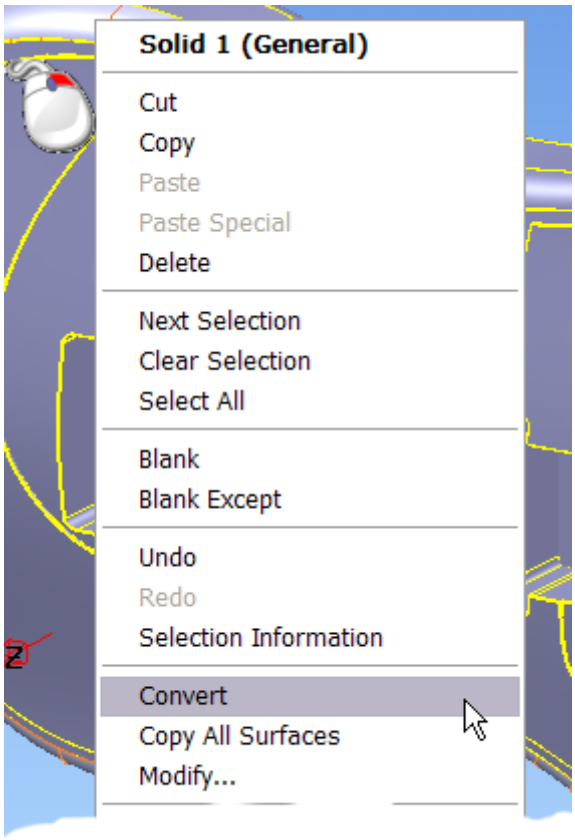
For the outer circle, enter a **depth of 6**





Rendering the Finished Body

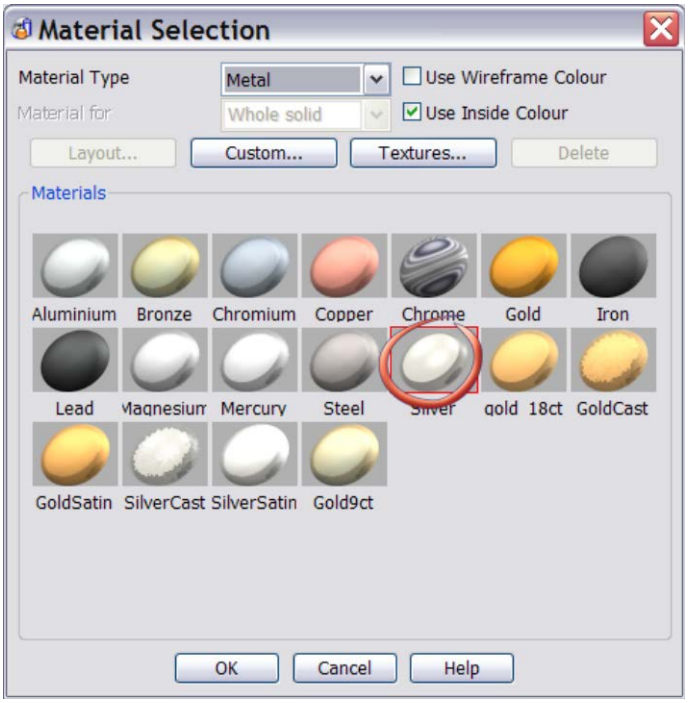
Render the part using two different materials, one for the body, and a separate one for the scratchplate. To do this, you must first convert the solid back to its constituent surfaces.



Select the entire model, and pick a nice material, for example –

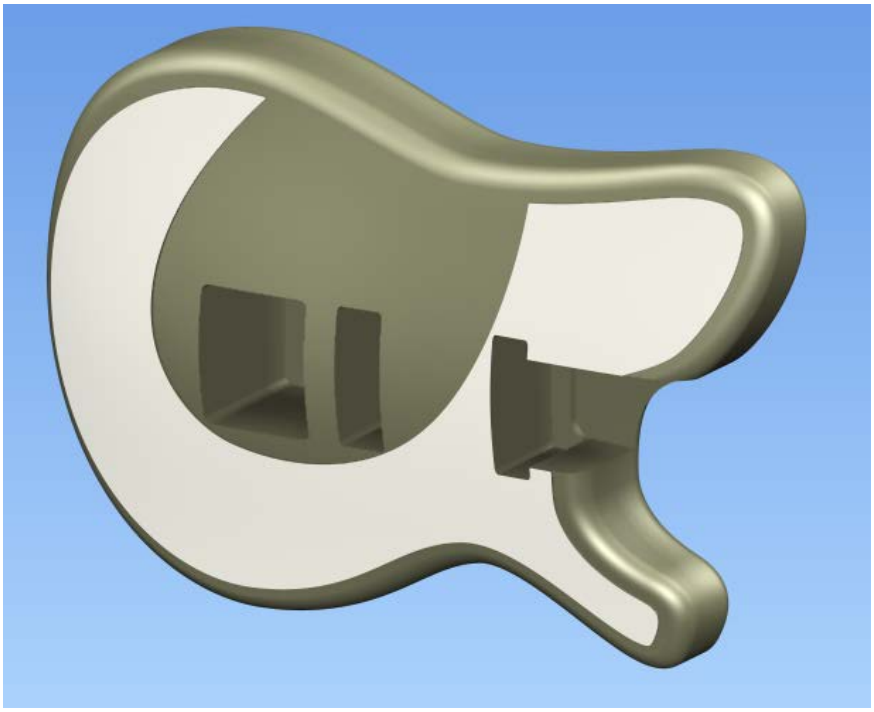
Now deselect everything (either left-click somewhere away from the model, or press Ctrl + D).

Select the scratchplate surface, and choose another material.



Click OK.

A rendered view (with default lighting) will look like this –



Save the model in **D:\users\Training\Coursework\Delcam Designer Jobs** as **Guitar exercise.psmodel**

Exercise – A 3D Scroll

Introduction

This tutorial teaches you how to make the complex 3D scroll shown above.

Key features covered in this tutorial are –

- Surface manipulation
- Curve editing
- Surface creation
- Pasting surfaces into ArtCAM Jewelsmith

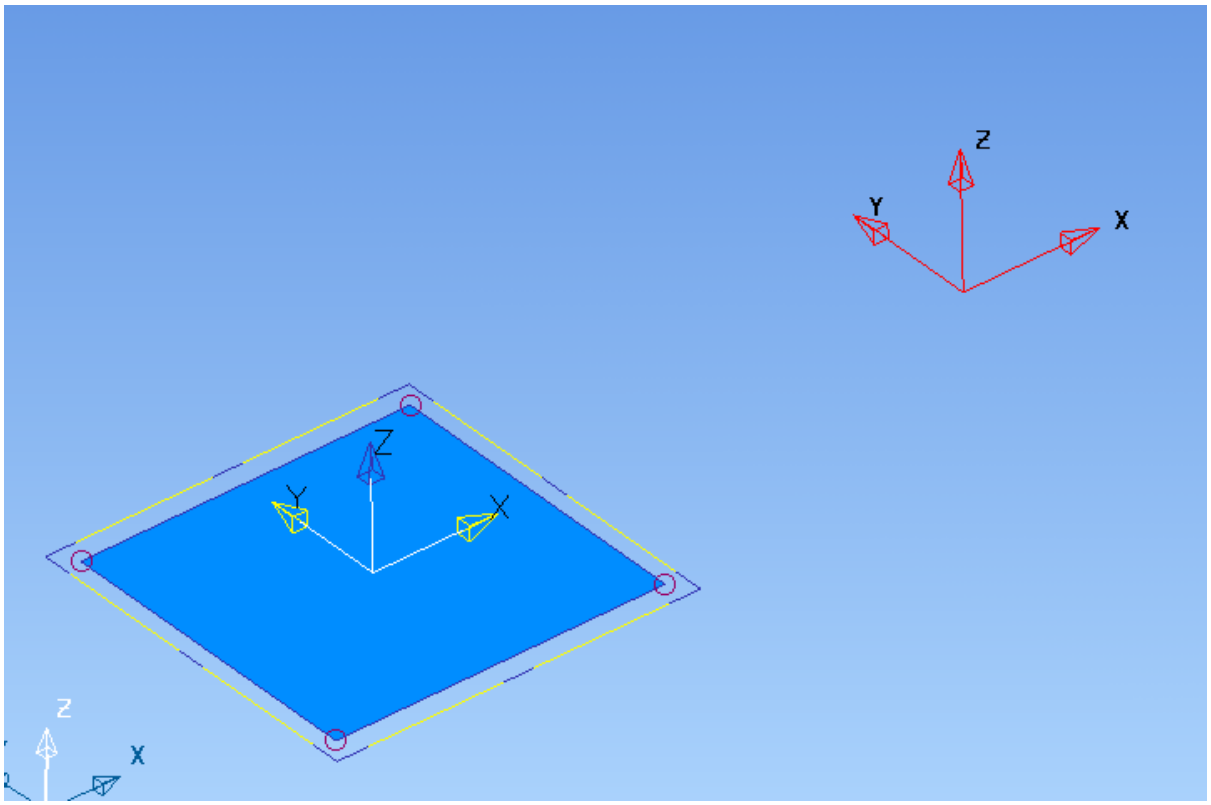
Creating a Surface to Sketch On

The main profile of the scroll is quite complex, and unlike most of the sketches you have created so far is non-planar. To create the sketch you will first make a simple surface. The exact dimensions are not critical for any part of this tutorial, and you should feel free to use your own artistic judgement.

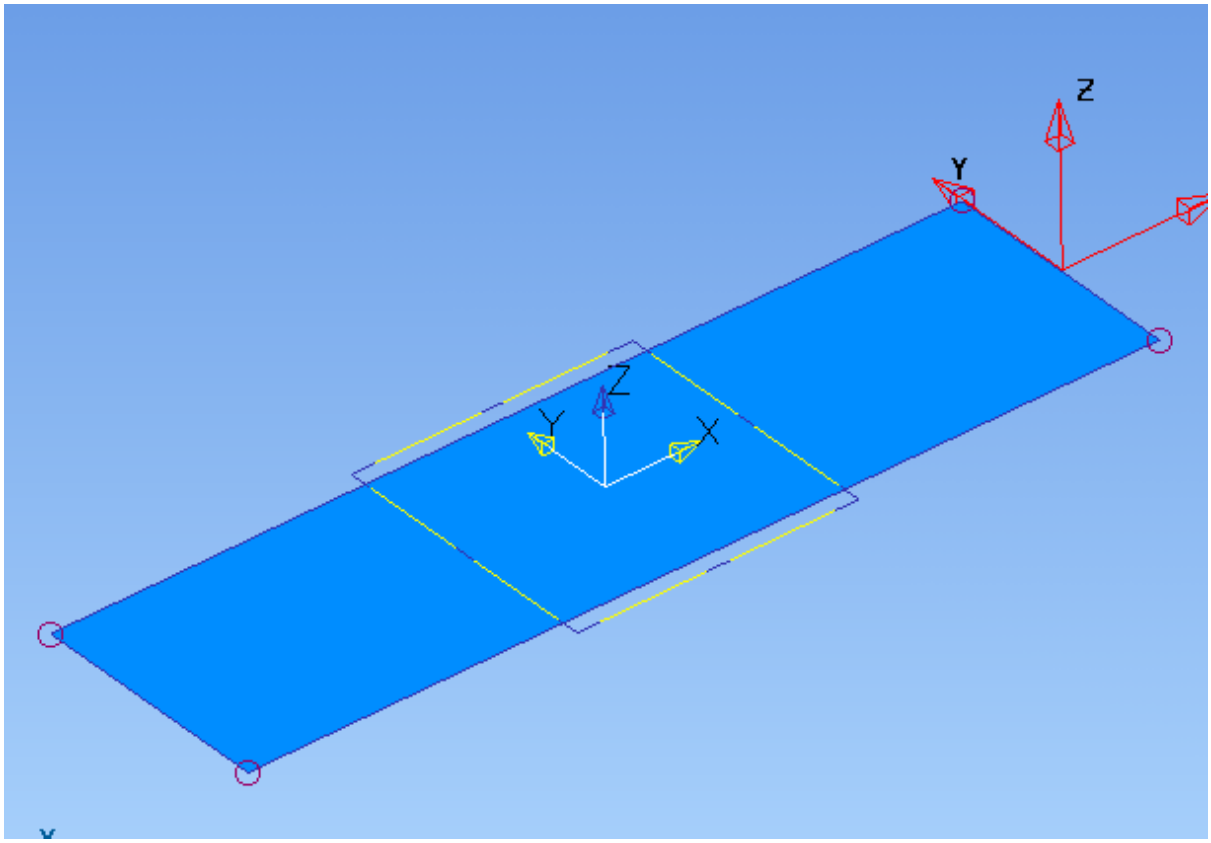
Create a workplane at ***0,0,0***

Click

Create a planar surface **(plane primitive)** at about -65 in the X axis as below.

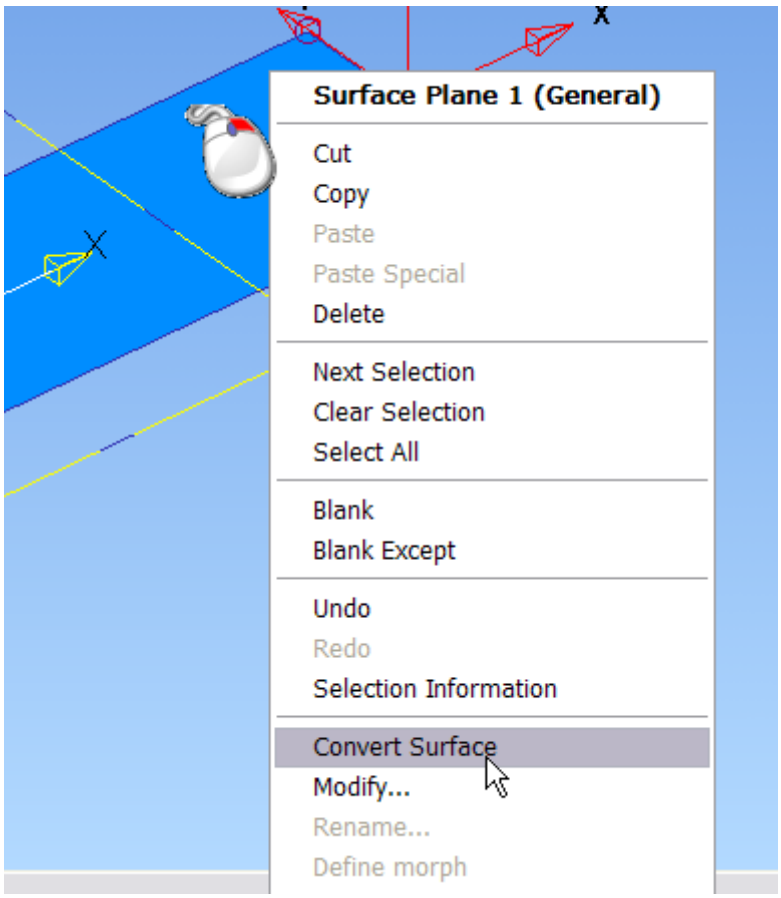


Change the plane so that it *exactly meets* the centre-line. I.e. If your plane is centred at -65 it must be 130 wide.

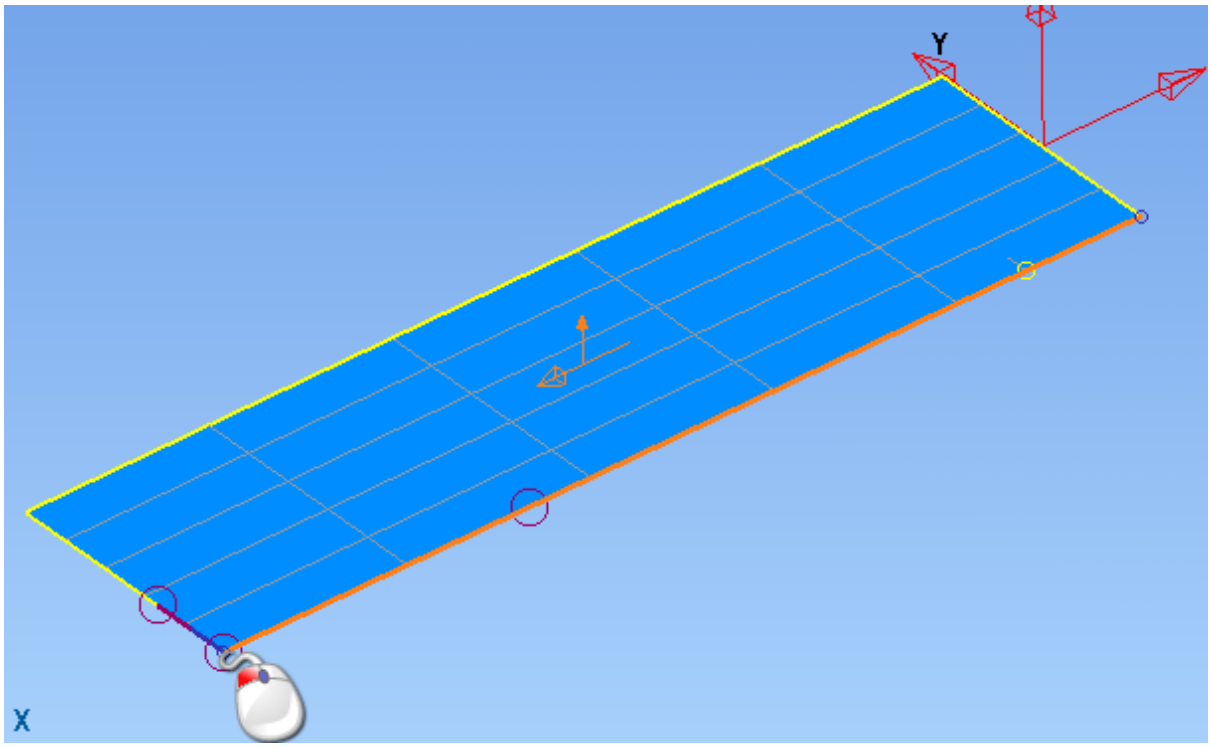


If necessary, adjust the ‘Y’ dimension of the plane as well.

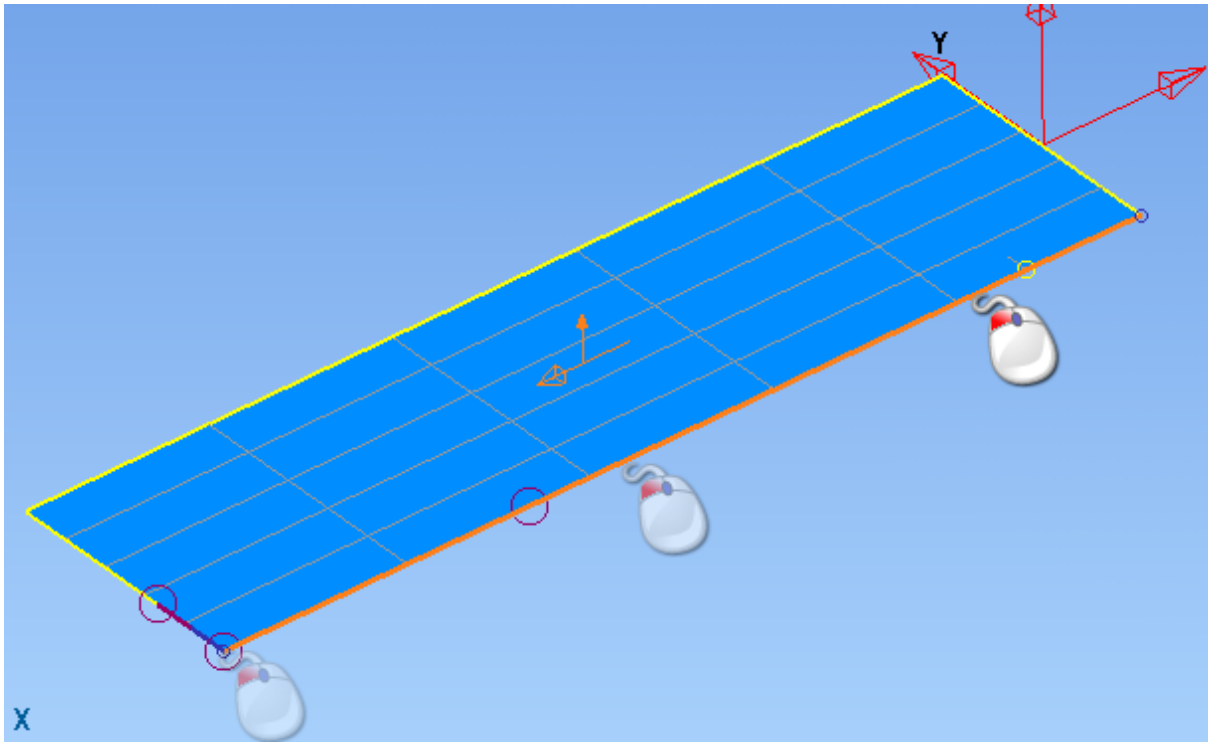
Click on the surface Using the right hand mouse button to open the surface edit pop-up and select ‘Convert Surface’. This will change the surface to a PowerSURFACE, and let you do more complex editing.



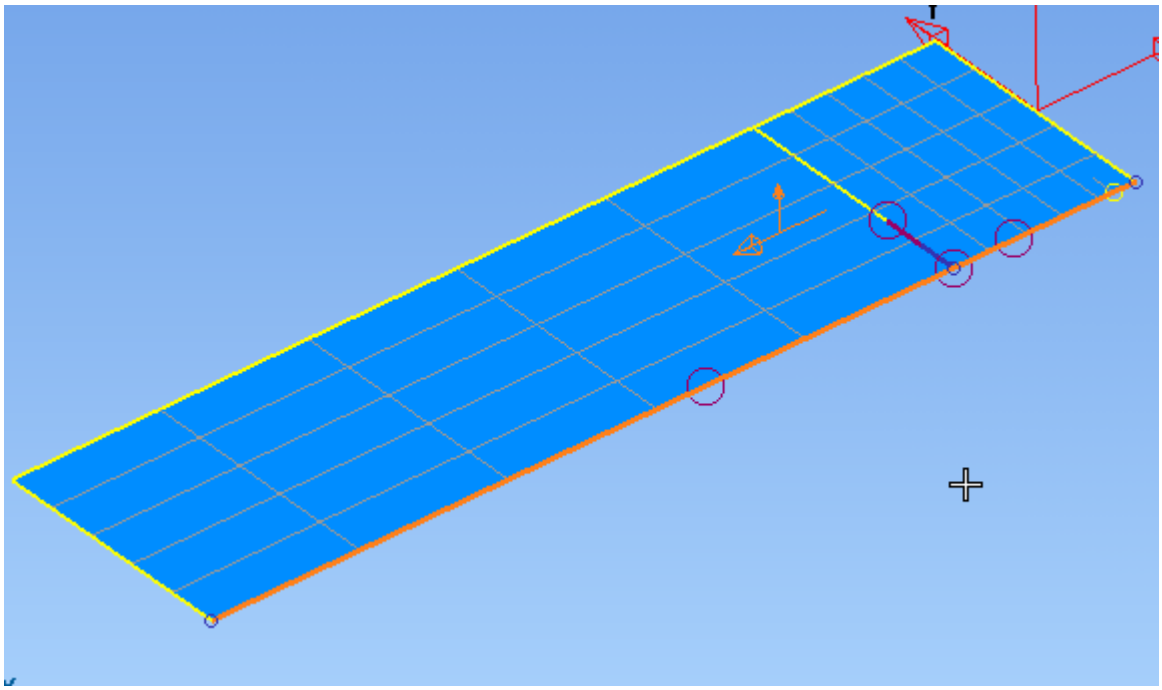
Select the surface at the point shown, making sure the hightlighting is drawn exactly as in the image below. This shows that you are ready to Edit a single Point on the surface.



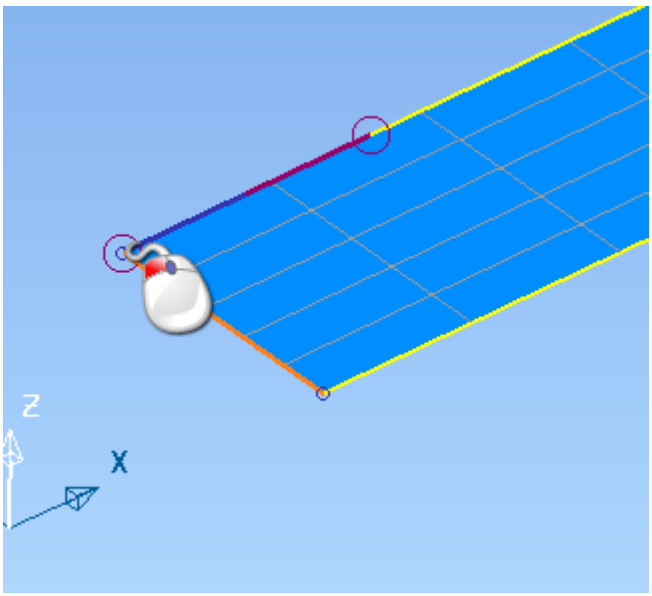
Press the left hand mouse key, and the Ctrl key *at the same time* and *hold them down.* Draw the selected point as shown to add a new curve into the surface.



NOTE: Take care to only add a single curve, in the direction shown.

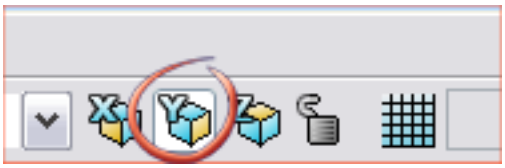


Select the point shown below.



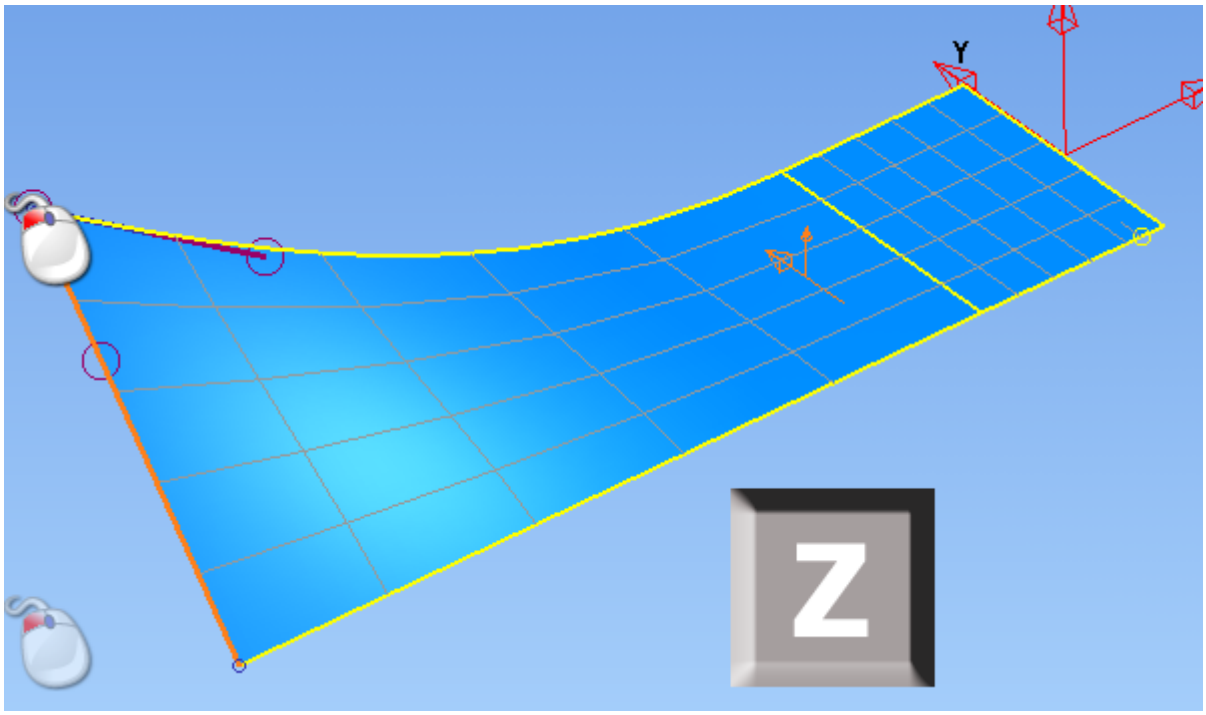
The surface will next be deformed by moving this point along the Z axis. How far you move it is entirely up to your artistic judgement.

Set the working axis to Y

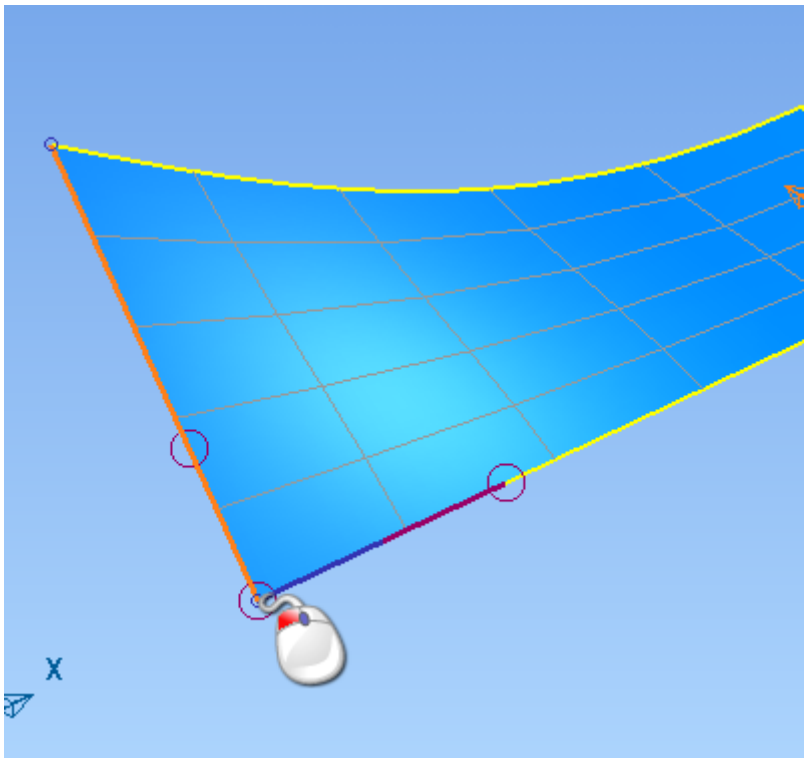


Make sure ‘Smooth Adjacent Spans’ is ON

While holing the ‘Z’ key down, *drag* the highlighted point upwards. Note that holding down ‘Z’ forces the point to move along the Z axis *only*.



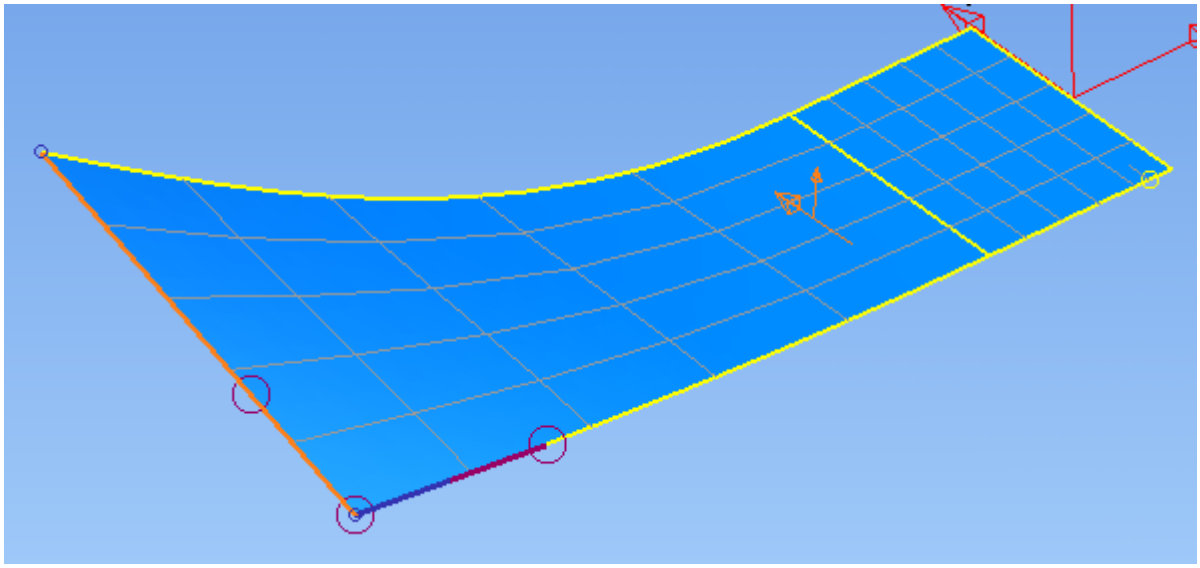
Select the point on the opposite corner, as shown below.



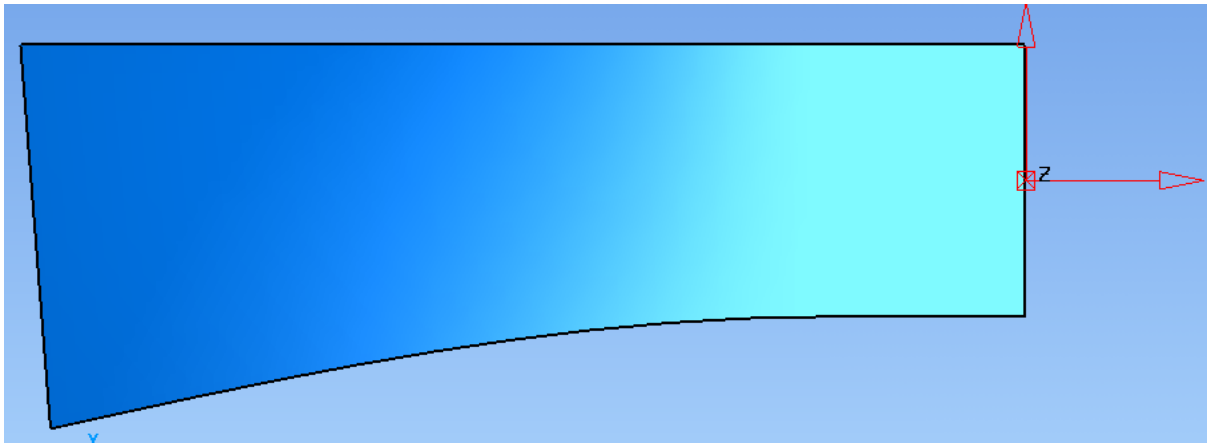
This point will be *Offset* along the *normal.* (The vector that is perpendicular to a surface).

Press the Crtl key on your keyboard and *hold it down.* (You can release the mouse). Now press the Up Arrow key. The surface will continue to change shape as long as you hold the arrow key down. When you are happy with the shape, release the arrow key. Note that if you wish, you can press the Down Arrow key to reverse the direction.

The end result will look like this –

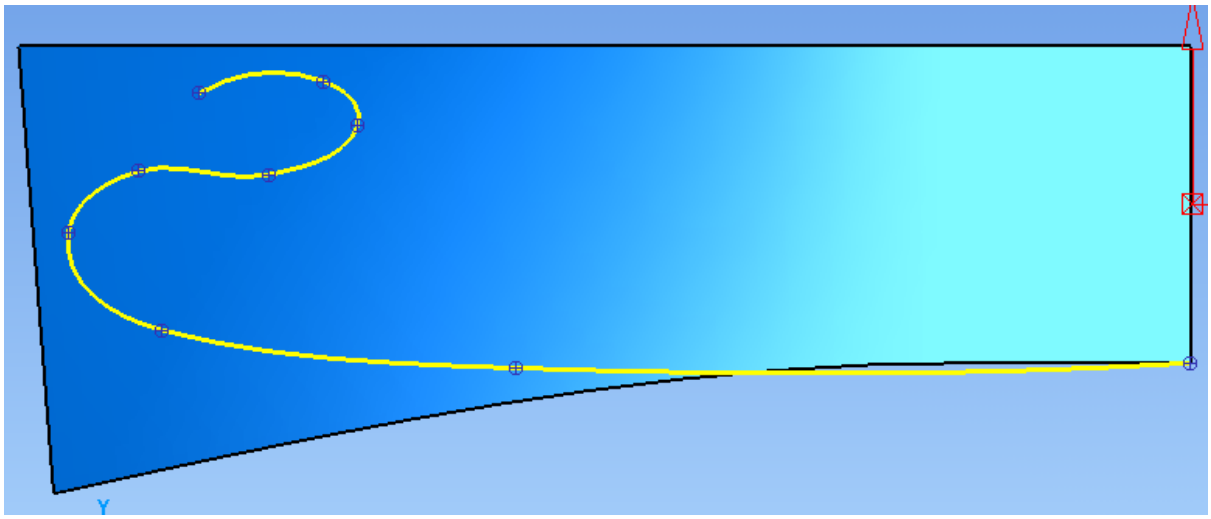


View the surface from above by pressing Ctrl + 5.

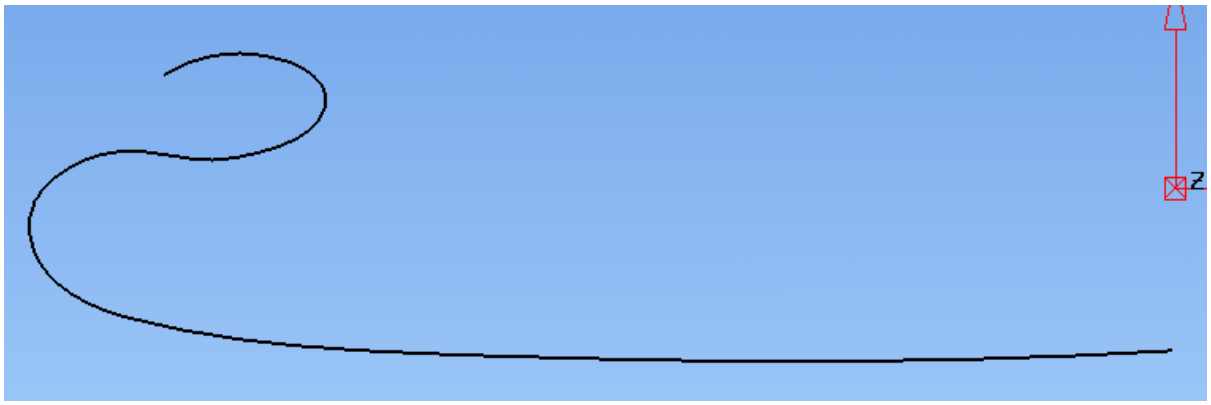


Creating the Main Curve

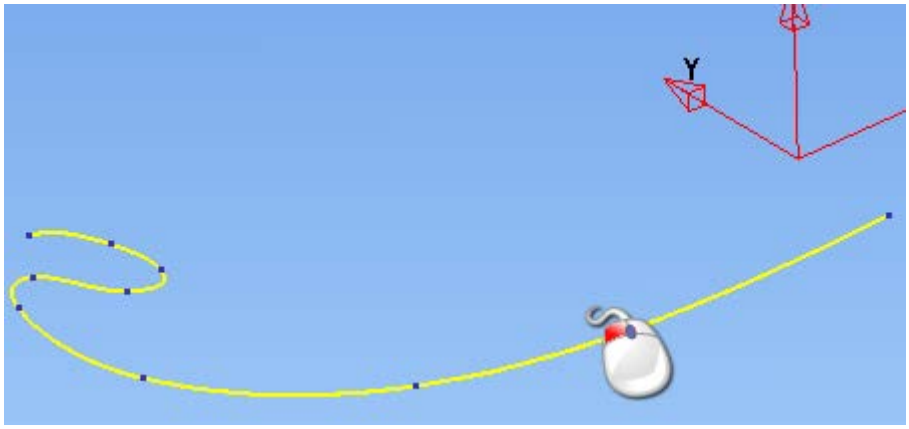
Sketch a Bezier curve similar to the one shown below. Note that as you sketch the cursor is displaying the key-word ‘IN’. This means that each point that forms the curve lies exactly on the surface. Note, though, that the spans *between* the points may not lie exactly on the surface for all their length.



Delete the surface, leaving just the curve.

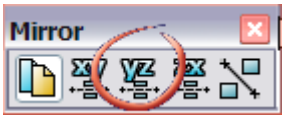


Rotate the view a little, and select the Curve.



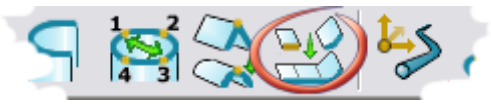
Open the General Edits page

Click on Mirror/Symmetrise

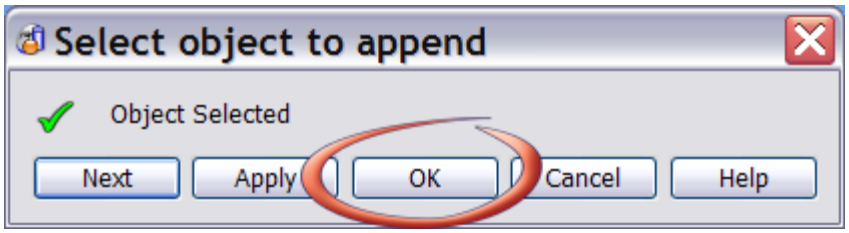
Symmetrise the curve through YZ, 

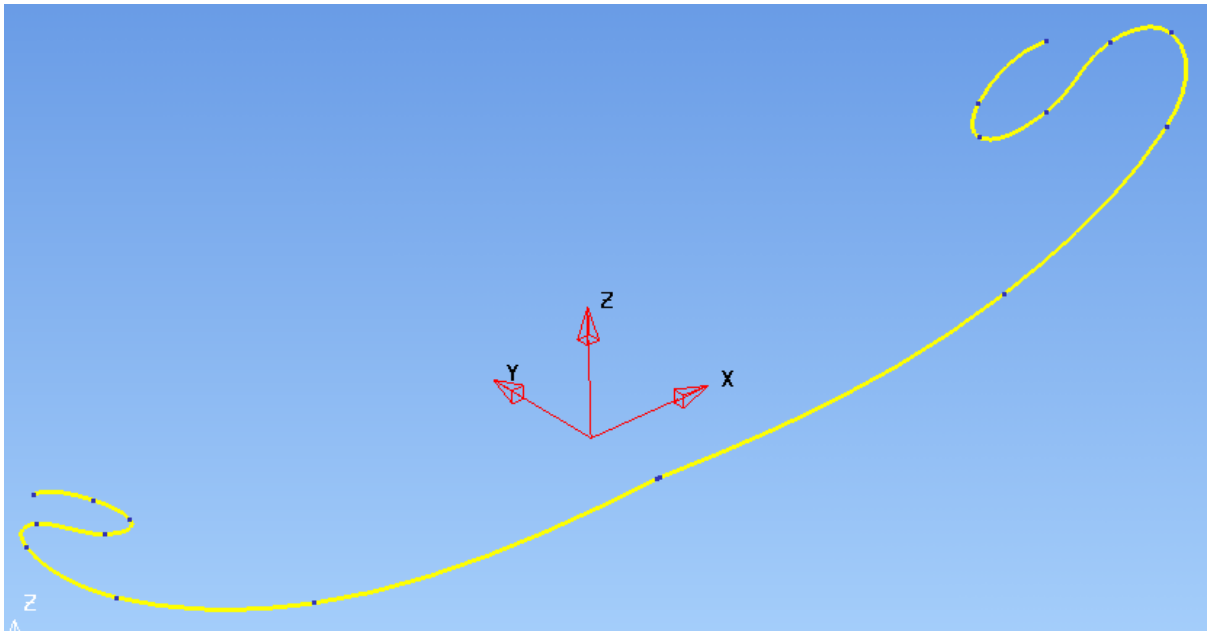
then press .

With the new curve still selected, click the Append Curve option from the curve edit toolbar.



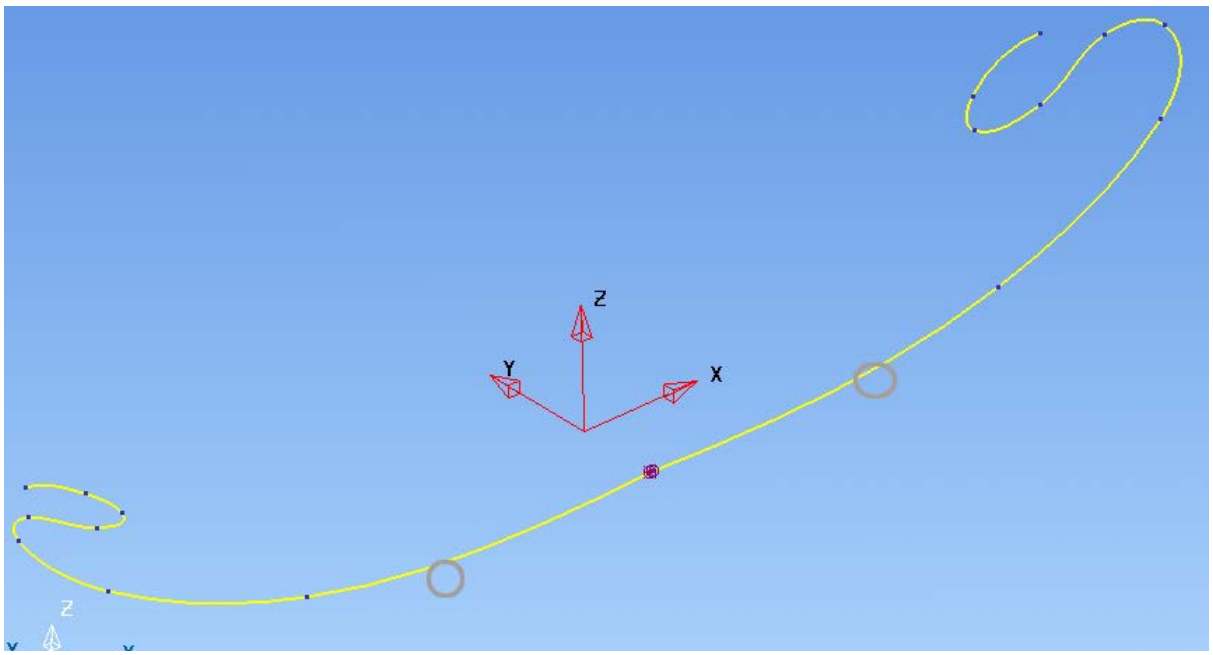
Select the original curve, and then click OK



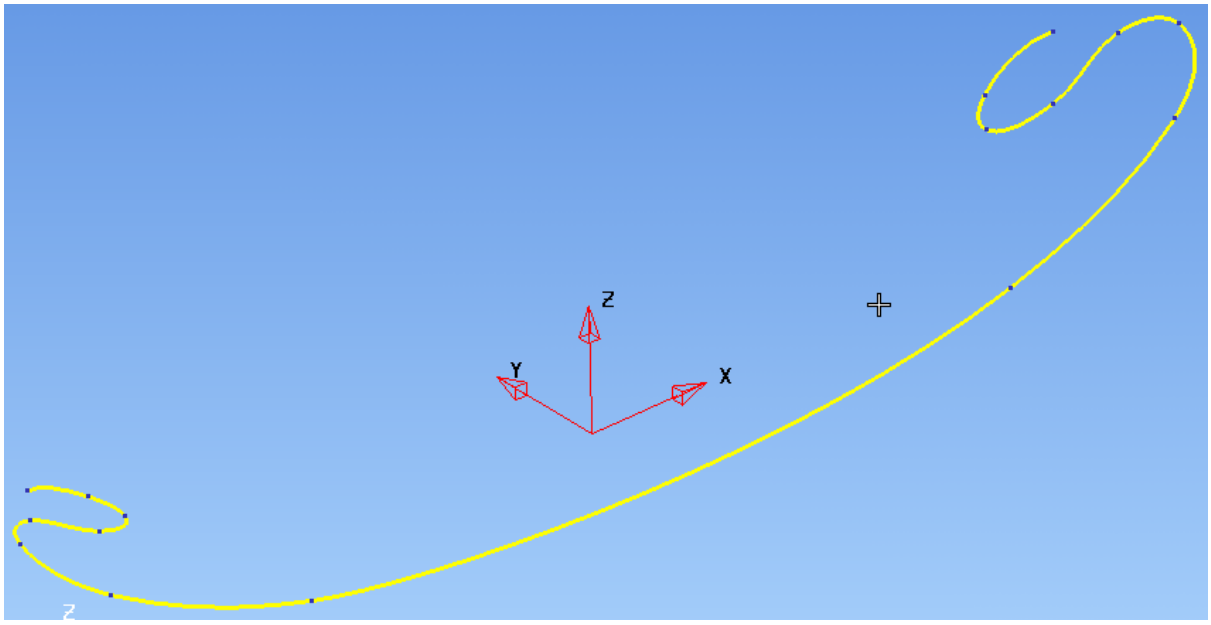


Note that there is a slight discontinuity at the mid-point of the curve, which must be corrected before proceeding.

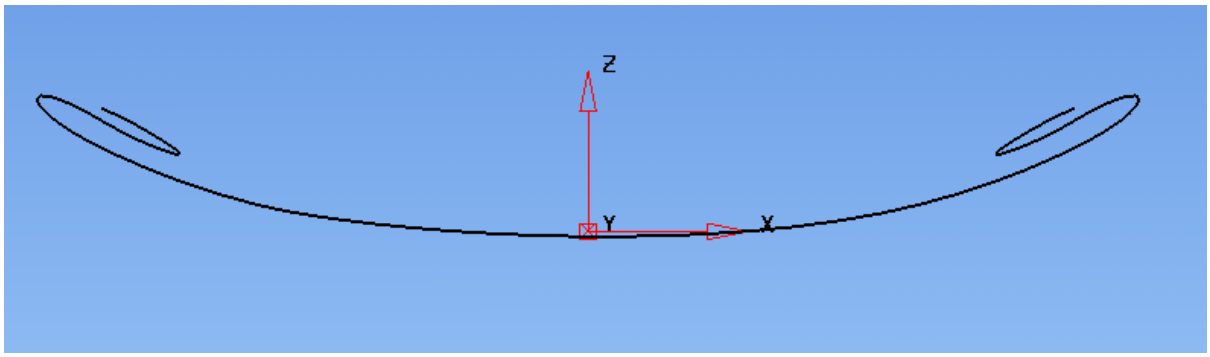
Select the mid-point so it highlights.



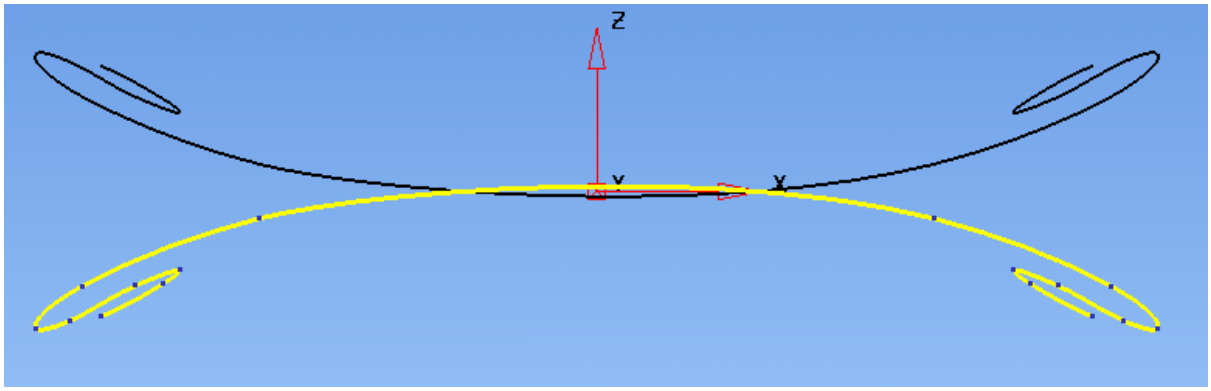
Remove the point by clicking (on the surface/curve toolbar)



Press Ctrl + 2 to view the model along the Y axis.

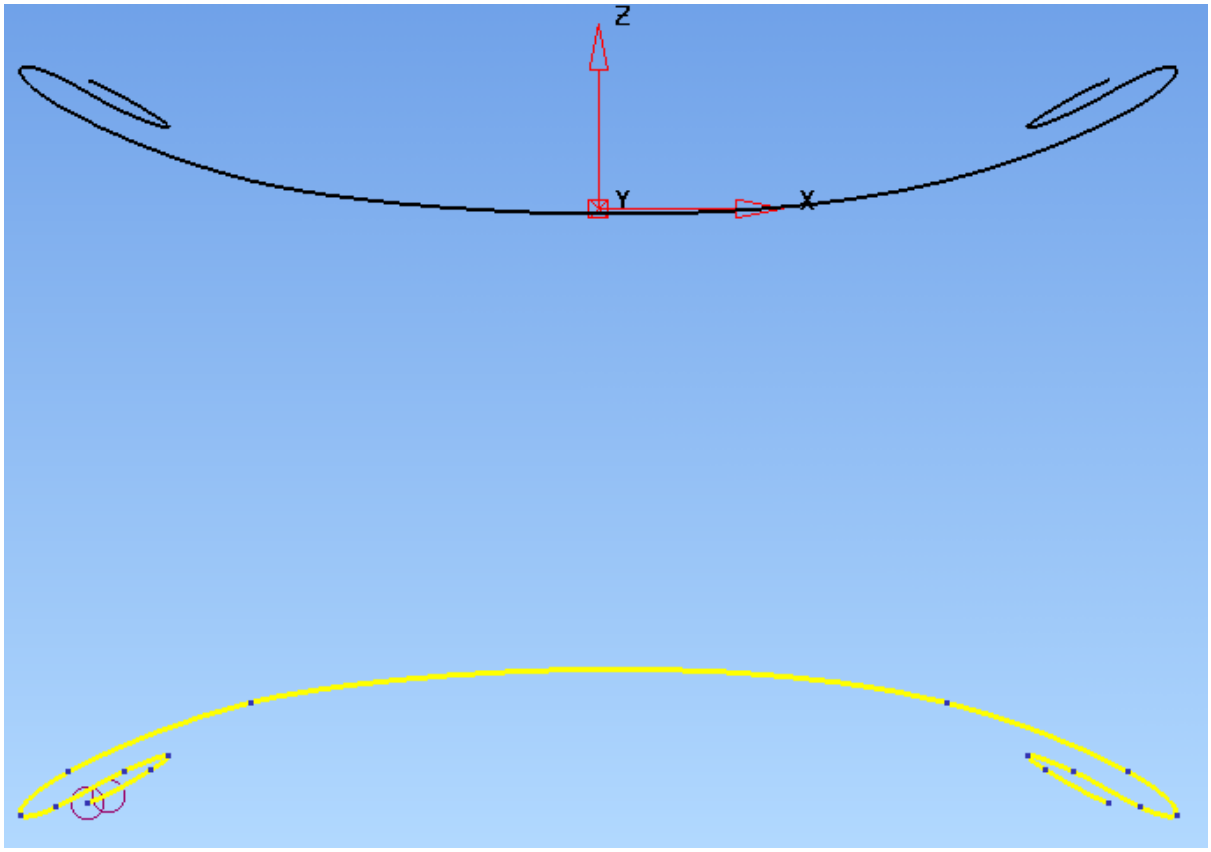


Select the curve, and make a mirror copy through XY.



Drag (move) the highlighted curve along the Z axis so it looks as below. (The exact value is unimportant).

Creating the Surface

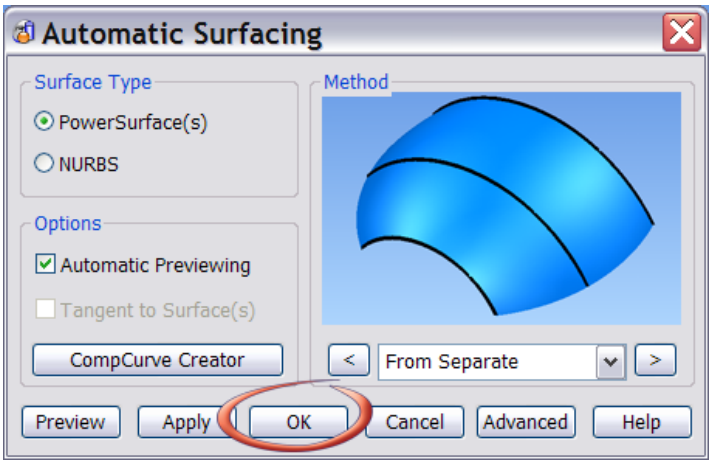


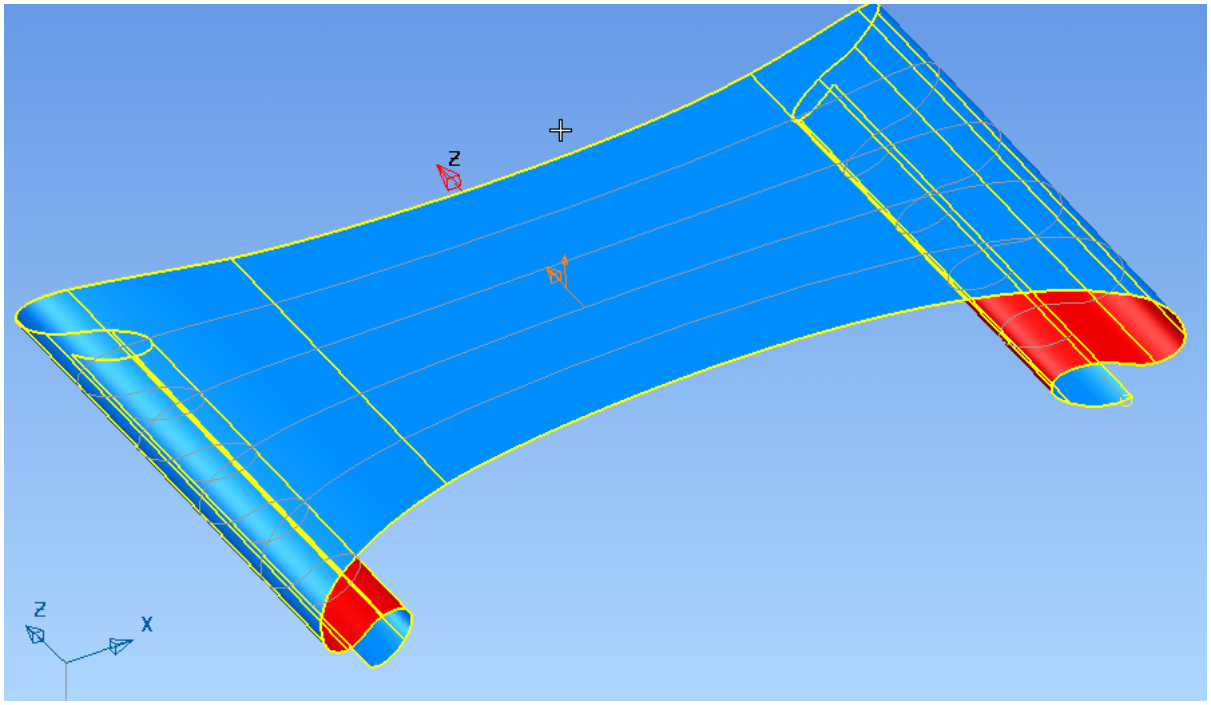
Select both curves.

Open the Surface Creation toolbar

Click ‘Automatic Surfacing’

The new surface is produced automatically, so all you need do is click OK.





Save the model in **D:\users\Training\Coursework\Delcam Designer Jobs** as **Scroll exercise.psmodel**

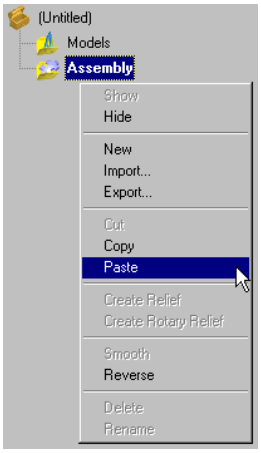
Passing the Scroll to ArtCAM

Make sure the surface is selected, and then copy the surface to the clipboard.

Open ArtCAM JewelSmith.

Create a New Project

Right click on **Assembly** in the **Project tree** and select **Paste.**



Paste the surface into the Project.