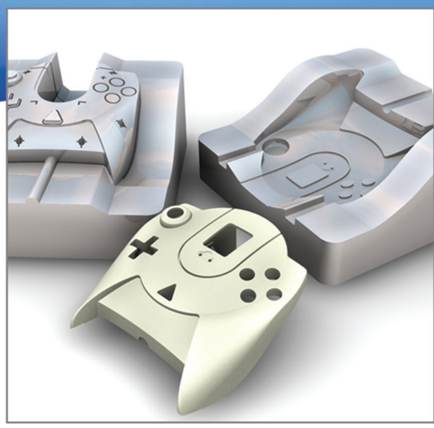
Delcam

Advanced Manufacturing Solutions

Powering your productivity



PowerSHAPE

Training Course

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 are checked by a competent person in accordance with good quality control procedures.

Contents

1. Introduction

PowerSHAPE Pro offers you "**Tribid Modelling**"**.** This means that you will be able to use the **PowerSHAPE** surface and solid modelling functionality with the **reverse engineering** and triangle modelling options**.**

With **PowerSHAPE Pro** you can:

- Move data captured and **reverse engineer** into the **design environment** easier and so incorporate additional features into any **reverse engineered design** quicker;
- Easily create **CAD surfaces** from **triangle data** through new options added in the **Automatic Surfacing** and **Mesh Segment dialog**;
- Use the **Compare Analysis** option to see the differences between **any two objects** (new to **PowerSHAPE Pro 2015**);
- Perform **Boolean operations** between the **triangle models** and either **surfaces** or **solid models**.

In this training course you will:

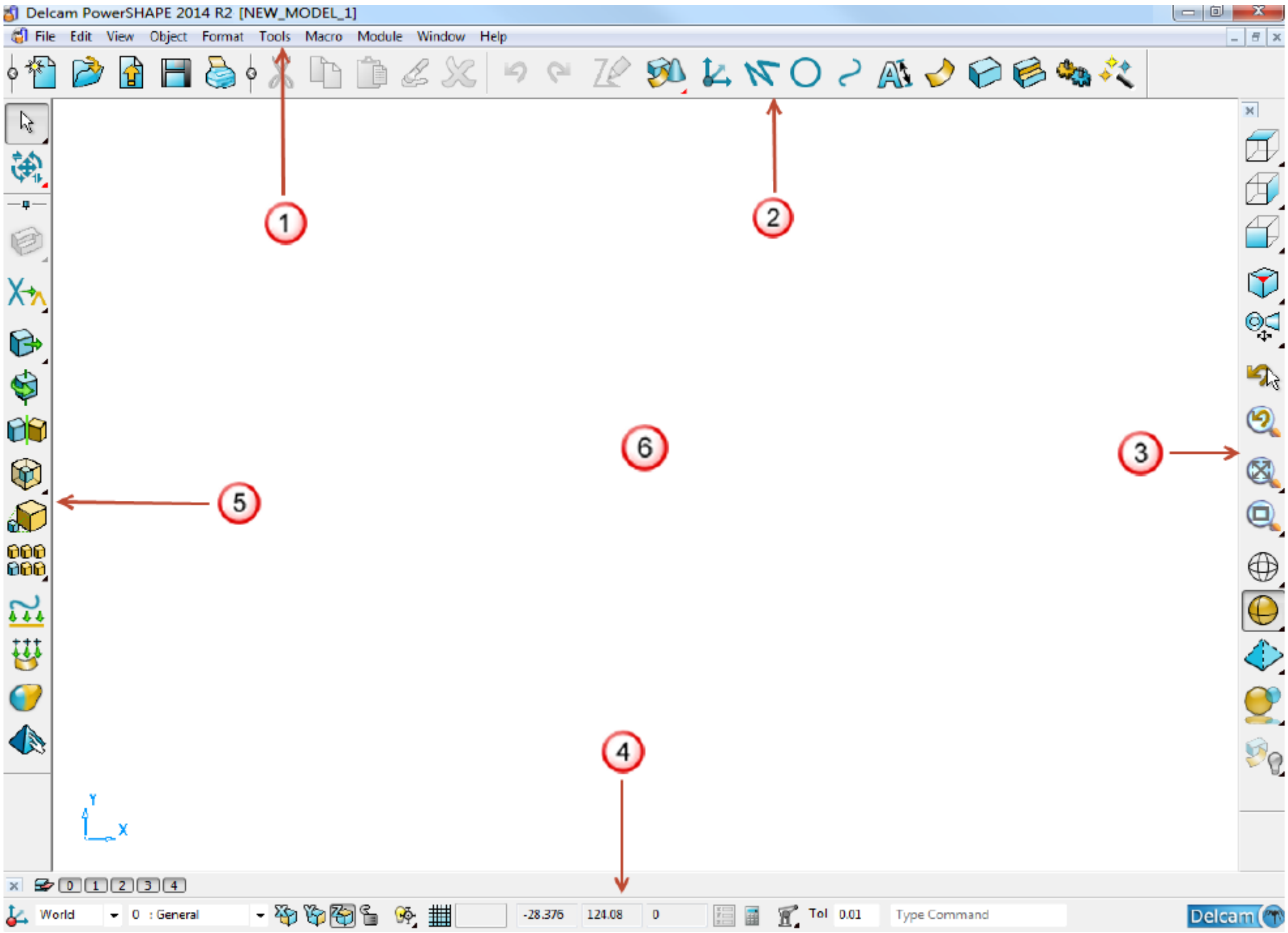
- **Generate triangles** from **scanned data**;
- Edit the point clouds to get the best triangulation result;
- **Fix**, **smooth** and **morph** triangles;
- **Sculpt triangles**;
- Perform boolean operations between meshes and a mesh and a solid;
- **Wrap triangles** onto a **surface** or **solid**;
- Use the "**tribid**" **modelling techniques** to create **surfaces** or **solids** from **mesh**.

PowerSHAPE Pro User Interface

1 Double-left-click with the on the **PowerSHAPE Pro 2015** icon displayed on your desktop.

2 The **PowerSHAPE Pro 2015** interface is displayed and a **New Model** is immediately opened.

PowerSHAPE Pro uses a Windows-style interface with pull-down menus, toolbars and flyouts. The following areas of the screen are identified as follows:



1 **- Pull-down menus**

2 **- Main toolbar**

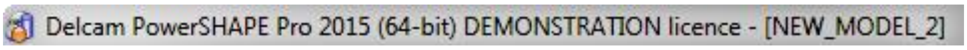
3 **- Viewing and shading options**

4 **- Status bar**

5 **- Command toolbar**

6 **- Graphics window**

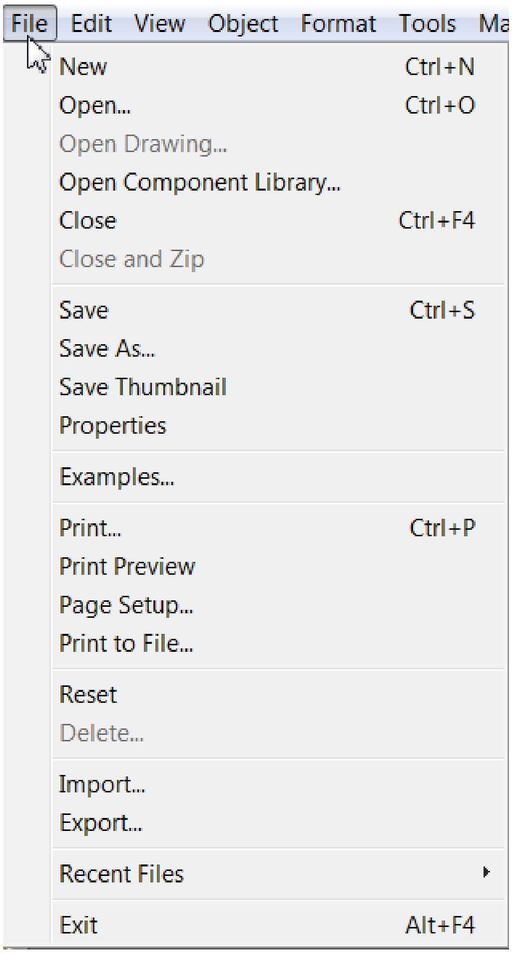
PowerSHAPE Pro automatically starts a **new empty model**. The model name is displayed at the top left corner. The new model is not stored externally.



The model should be **saved permanently** (recommended) with a more appropriate name and stored to an **external directory**. One or more previously stored models can be opened within the same **PowerSHAPE Pro** session.

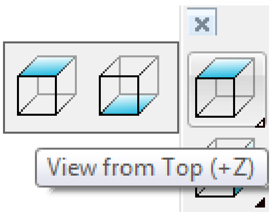
At the top of the screen there are a series of pull-down menus. 1

3 Select the **File** menu, using the left mouse button.



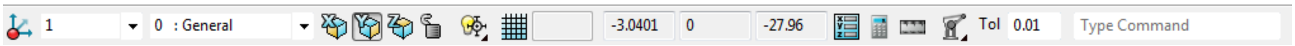
In the **File** menu models can be created, saved, imported, exported and printed etc while further **sub-menus** are accessed by **clicking** on the  arrow.

If the mouse is held over a button for a short period, a 'tool tip' box containing description of the command is displayed.

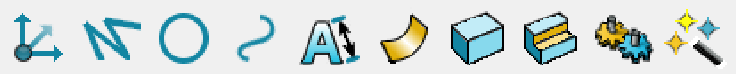


Towards the **bottom left** of the **main graphics area** is the **Levels** menu. **Entities** and **objects** can be assigned to a **level** for long term **organisation** of the **model**.

At the bottom of the screen are the **Workplane** and **Principal Plane selectors**, the **grid** definition, the **cursor** position, the **tolerance** and the **command input area**.



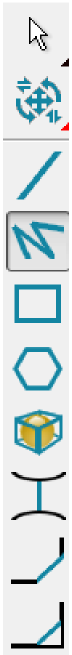
PowerSHAPE Pros Main toolbar (2) provides access to the **Workplane**, **Line**, **Arc**, **Curve**, **Annotation**, **Surface**, **Solid**, **Feature**, **Assembly**, and **Wizard** entity creation toolbars.

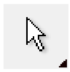


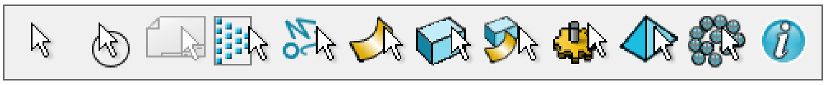
Beneath the pull-**down menus** are the Main toolbar buttons (2). When selected, these buttons display the relevant functions on the **Command** toolbar as a **sub-menu** (5) to the **left** of the **graphics area**. (6)

This **Command** toolbar contains all of the **main options** for **creating** within the chosen function.

4 From the **PowerSHAPE Pro Main** toolbar, select the **Line**  icon, which will display the following **sub-menu** to the **left** of the **graphics area**.



On this side the **top two icons** are fixed and compromise of the **selection**  icon and the **quick select flyout menu**.

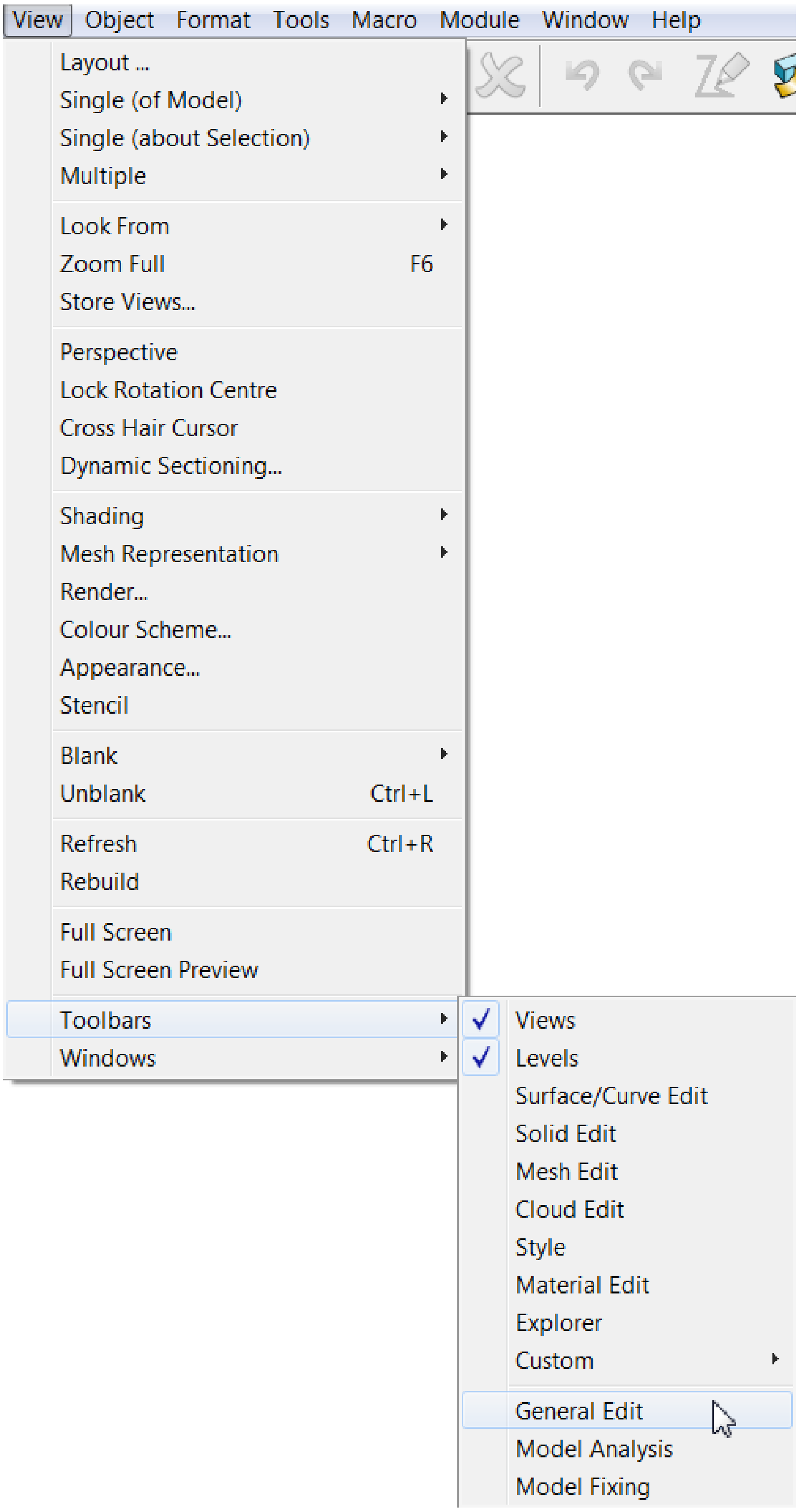


While the lower icon will either show the **General Edits** , **Model Fixing**  and **Model Analysis**  **sub-menus**.

A **small black triangle** in the **lower right hand corner** of an **icon** denotes and **flyout menu** will appear if the mouse is held over, while the equivalent **triangle** in **red** denotes the same **flyout menu** can be **cycled** by **clicking the icon** itself.

Toolbars

All the available toolbars, including any custom toolbars cab be **shown** or **hidden** from the **View drop down menu** in **View>Toolbars.** Any toolbar with a **tick** adjacent to the name is **currently visible**. A **single click** on the **name** will **toggle** between **visible** and **hidden.**

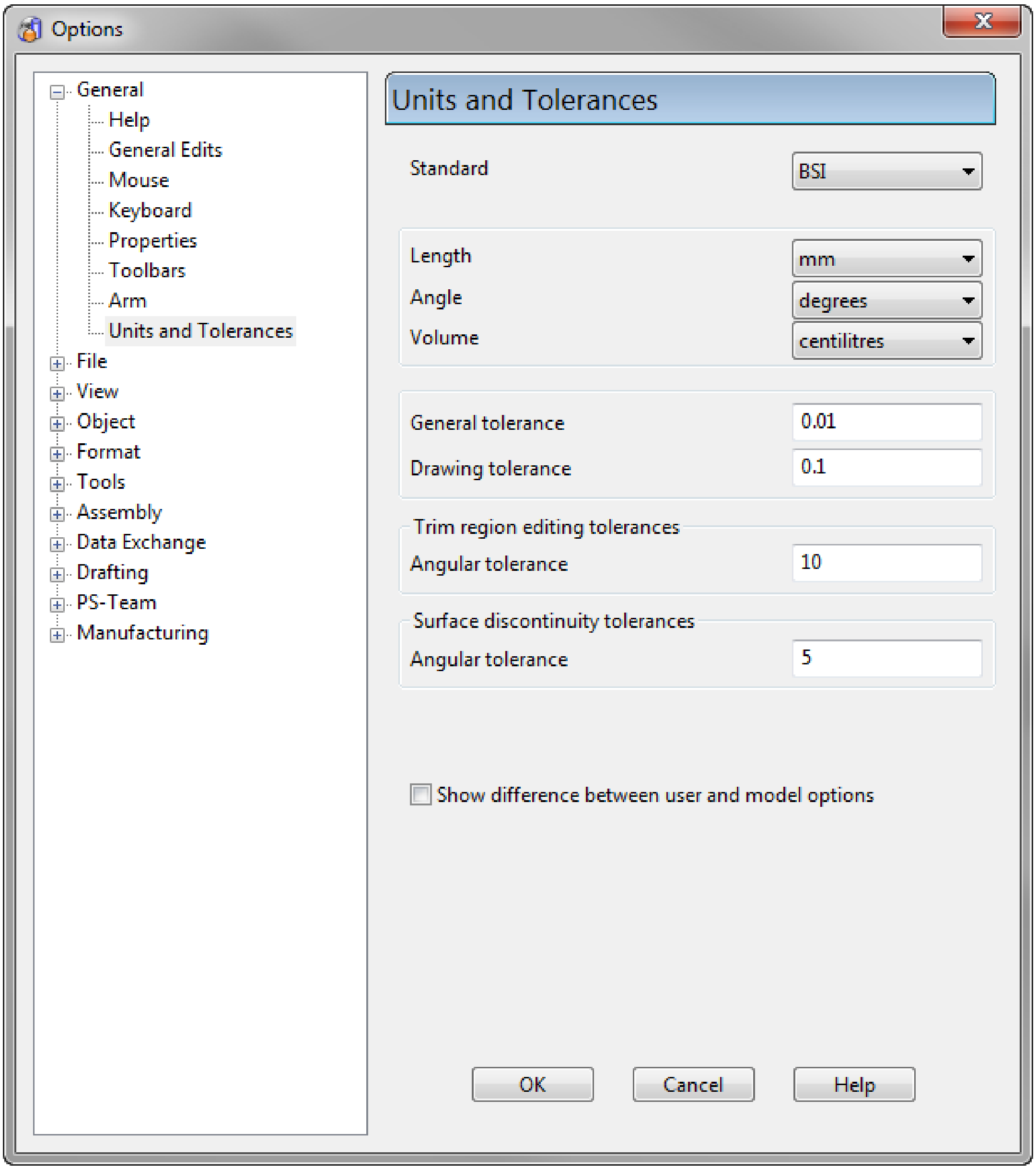


The **Surface/Curve**, **Solid** and **Mesh Edit** toolbars will only tend to **used** and **visible** while a **relevant object** is **selected** and **worked on** in the **graphics area**, so these will come and go from **above** the **graphics area** with your **selection**.

PowerSHAPE Pro Options

The **Tools > Options** page allows the user to configure **PowerSHAPE Pro** from the **factory default** settings. The options are stored within **several main categories**, accessible by clicking on the **adjacent** **symbol**.

It is possible to make changes to the **factory default** settings and store them as the **new default settings** to be active with subsequent **PowerSHAPE Pro** sessions.



Mouse buttons

Each of the three mouse buttons performs a different operation in PowerSHAPE. By using the **ALT, Ctrl** or **Shift** key, these operations can be extended.

Left Mouse button 1: Picking and selecting

This button is used for selecting items off the **Main** pull-down menus, entering data, and selecting parts of the model.

Middle Mouse button or wheel: Dynamics

Zooming: Hold down the **Ctrl** key and middle button/wheel and move the mouse up and down to zoom in and out of the view. Hold down the **Ctrl, Shift** Key and middle button/wheel to select a framed area to zoom into.

Panning: Hold down the **Shift** key with the **button/wheel** while moving the mouse, to move the view across the component.

Rotating: Hold down the **middle button/wheel** and **move** the mouse. A tracker ball appears at the centre of rotation, as does the view orientation axis in the bottom left corner of the screen. If **View Spinning** is switched on (**Tools > Options > Views**) the view will spin around until the user executes a further mouse click.

If available, the middle mouse wheel can be used to scroll text.

Right Mouse button 3: Special Menus

When this button is held down a local menu is opened. The contents of the menu depend on the entity selected. For example, when the cursor is over a line, the **Line** menu appears; when the cursor is in the graphics area, the **View** menu appears.

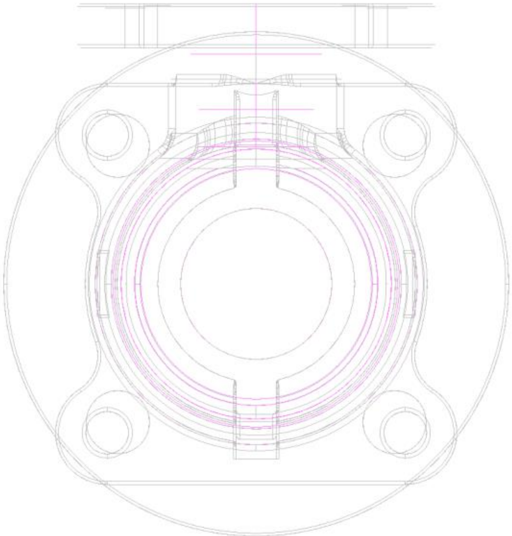
Exercise 1: Dynamic mouse controls.

In this exercise you will Import an existing, stored model and change the views using the dynamic mouse options.

1 From the main menu select **File > Open** or  to open an existing PowerSHAPE model (.psmodel).

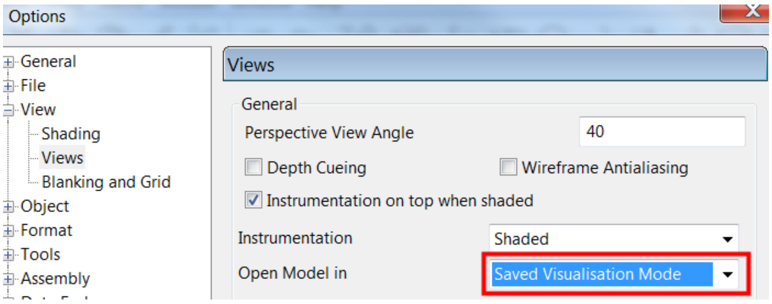
2 **Open** the model:

.....\PowerSHAPE_Data**pump housing.psmodel**



When a model is imported into a new PowerSHAPE session, the default view is down the Z-axis.

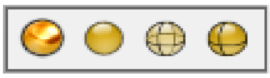
The model has opened in wireframe view as originally saved. Preferred shading options on opening models can be changed from **Tools>Options>View>Views**

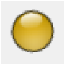


The view toolbar, on the right of the PowerSHAPE window contains three main areas

- Views.
- Dynamics.
- Shading.

3 Select the view **Iso 1** , the model is now displayed using the first of 4 isometric views. Surface or Solid entities of the model can be displayed as shaded if required.

4 From the views toolbar open the **Shaded** view pull out menu. 

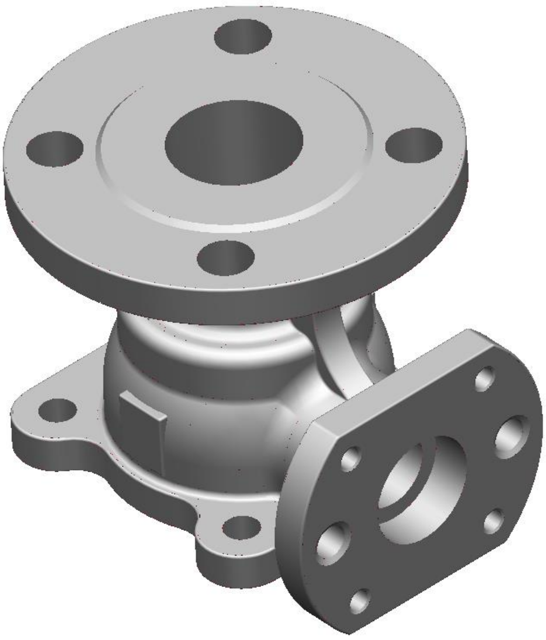
5 From the available options pick **Shaded** View. 

The surfaces have been shaded to show the full extent of the surface area. The wireframe from which it is constructed can also be shown if required. The model can be rotated dynamically so that the user can view the underside detail.

6 Hold down the middle mouse button and move the mouse to rotate the view to display the underside.

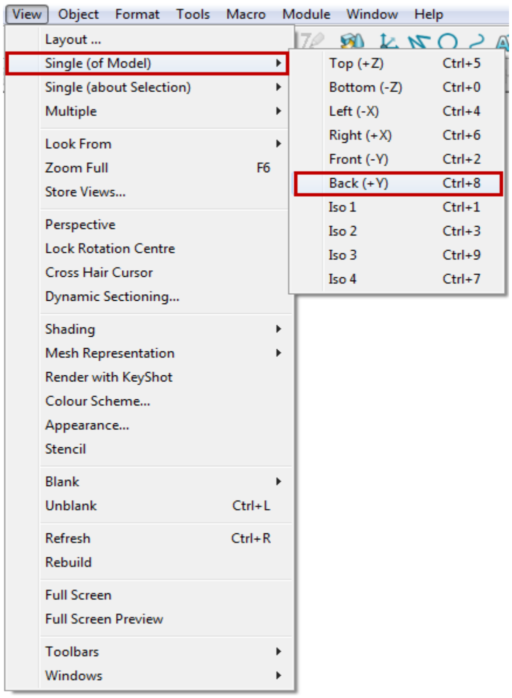
7 Hold down the **Shift** key and middle mouse button and move the mouse to position (Pan) the view suitably.

When scrolling the middle button (wheel) to zoom, the position of the cursor is the focal point.



The underside can now be seen clearly. 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.

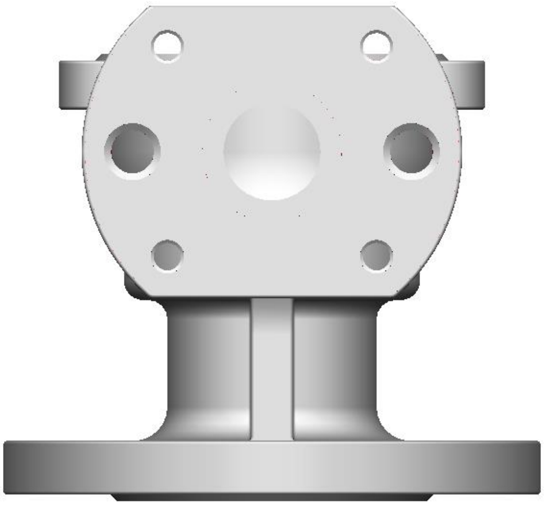
8 Select **View > Single [about selection].**



The keyboard shortcuts for the views (and other functions) are listed, such as Ctrl+1 (Press Ctrl on keyboard then number 1) for the **Iso 1** view. By pressing **Num Lock** on the keyboard, the number pad can be used with the Ctrl key to change the views.

There is also an enhanced shading option that displays the model in a Perspective view with the lines running to a vanishing point as if on the horizon.

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



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

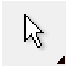
All PowerSHAPE help options and keyboard shortcuts are listed in chapter 13 of the training manual.

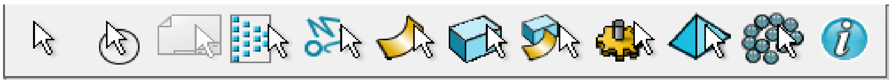
Selecting Entities

To be able to perform any edits, individual entities must first be selected.

If an entity is selected with the left mouse button it will be displayed with a yellow colour. If a box is dragged across a group of entities they will all be selected (The default setting does not require an entity to be completely boxed).

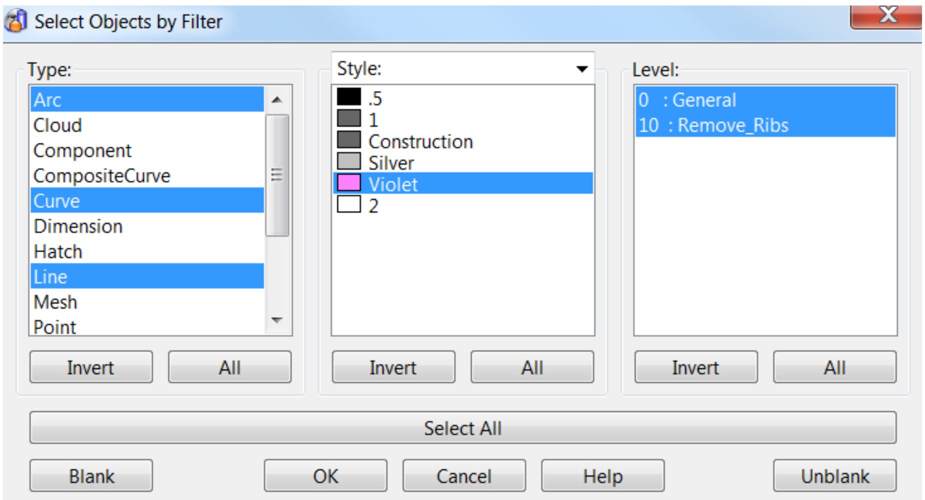
To select more than one item within a group, hold down the **Shift** key while selecting individual entities to add to the selection. To toggle a selected item on or off, the **Ctrl** key is depressed while using the left mouse key to pick the entities.

The **Select** flyout (above command toolbar) reveal the following options.



There are options within this **Selection** toolbar to enable the user to quickly select all items in the view of a particular type. They are Wireframes, Surfaces, Solids, Surfaces & Solids and components (assembly), meshes and point cloud.

For more complex models, the **selection** filter  can be used to discriminate entities using a more specific combination of search criteria, such as the type of wireframe, colour, and line style.



Blanking Entities

If one or more entities are selected, they can be temporarily removed from the graphics area by selecting **Blank**. This is available from the local menu (right mouse click) or selecting the **Blank** icon  from the **Viewing** and **Shading** toolbar.

This command is useful when you want to temporarily undraw some objects from the current view where **Levels** is not appropriate. This is because, a level may contain objects you don't want to be hidden and/or changing the levels may affect the model's logical structure.

Blanking allows you to remove selected objects from the screen. They are not deleted, but merely hidden and can be unblanked when required. This allows you to concentrate on the objects you need to work on without being hindered by other irrelevant entities.

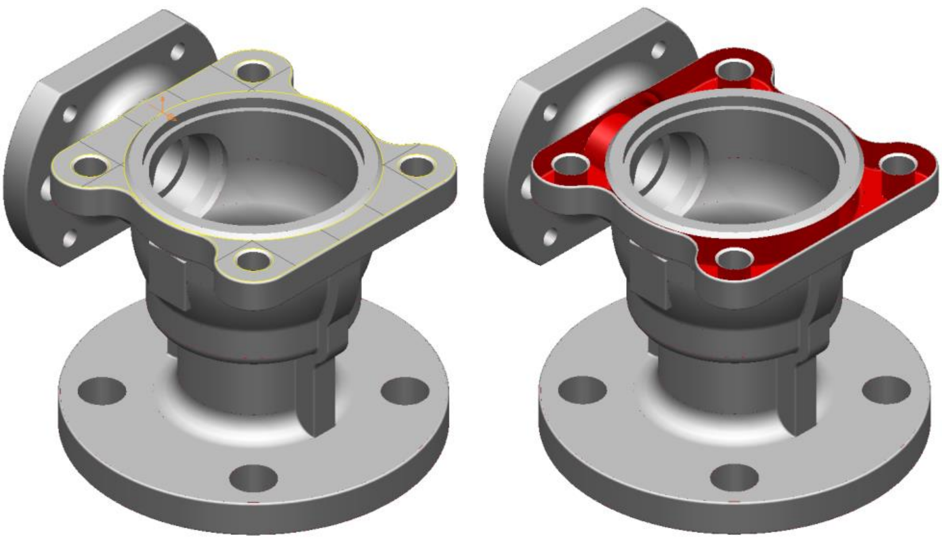
If it is required to temporarily keep the selected entities from the graphics area and hide the rest then Blank Except is applied. To return all entities back to the graphics area, Unblank is applied from the screen menu. Blank Toggle allows the user to toggle between the two views of Blanked and unblanked entities.

When using often, the shortcut keys are useful

Ctrl+J	**Blank**
Ctrl+K	**Blank Except**
Ctrl+L	**Unblank**
Ctrl+Y	**Blank Toggle**

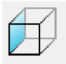
Each will be demonstrated using the current model

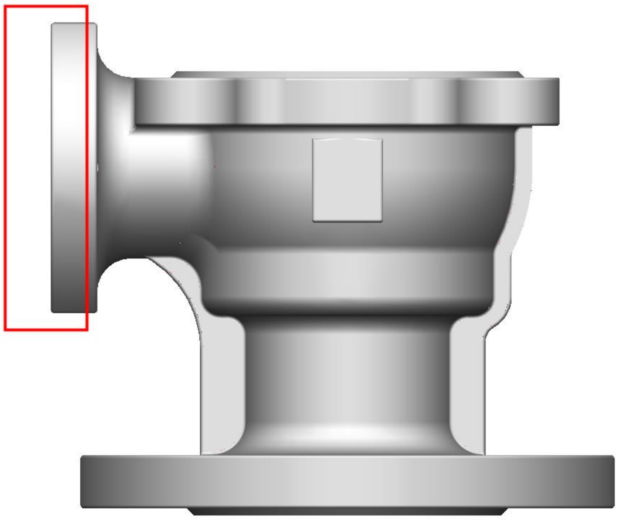
10 Select the upper surface of the pump housing and its outline will change to a yellow colour.



11 Select **Ctrl+J** to temporarily remove the upper surface from the graphics area as shown to the right (above).

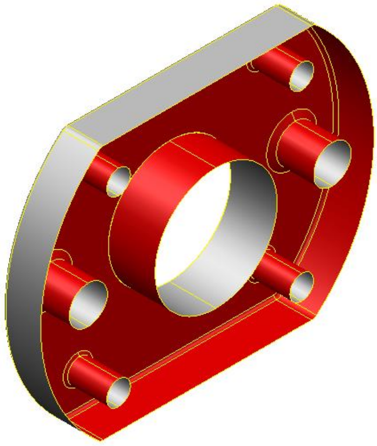
12 Select **Ctrl+L** to return the blanked surface back as the selected item.

13 **View the model from Left (–X)** , then box select the flange surfaces as shown.

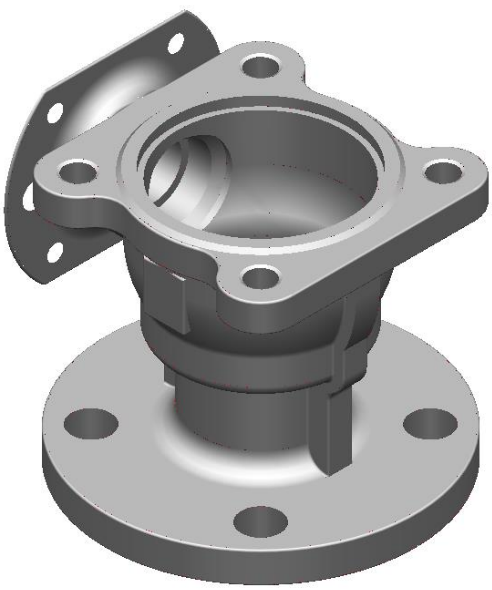


14 Select **Ctrl+K** to keep these surfaces and temporarily hide all others.

15 View in **ISO 1**. 

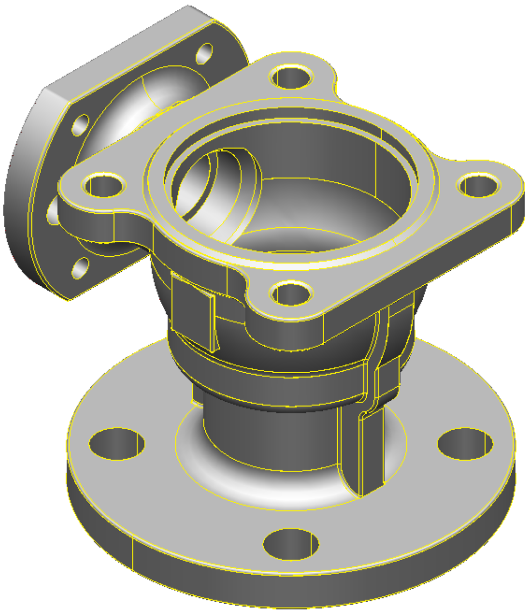


16 Select **Ctrl+Y** to blank **toggle** and view the blanked surfaces

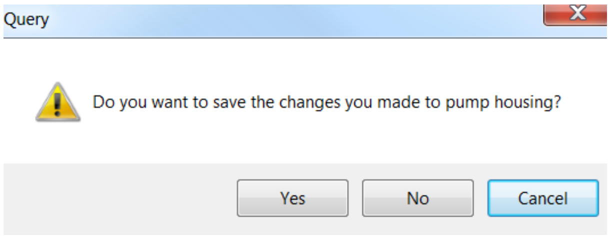


17 Select **Ctrl+Y** to toggle the view again.

18 Finally select **Ctrl+L** to return and show all surfaces.



19 Select **File > Close.**



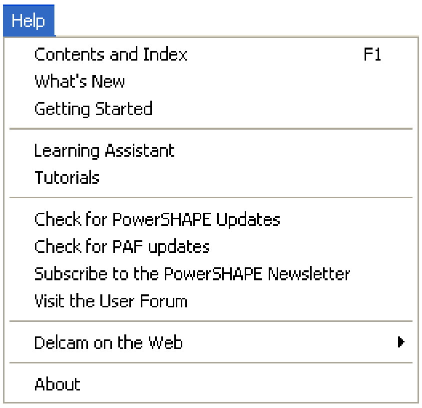
A message box appears asking if it is required to save the changes. In this case no changes have been made to the model.

20 Select **No**. The current model is now closed.

2. Help and Shortcuts

Help menu

The Help menu provides the tools related to the on-line help system.



- Contents and Index
- What's New
- Getting Started
- Learning Assistant
- Tutorials
- PowerSHAPE updates
- PAF updates
- PowerSHAPE newsletter
- User Forum
- Delcam on the Web
- About

Help > Contents and Index

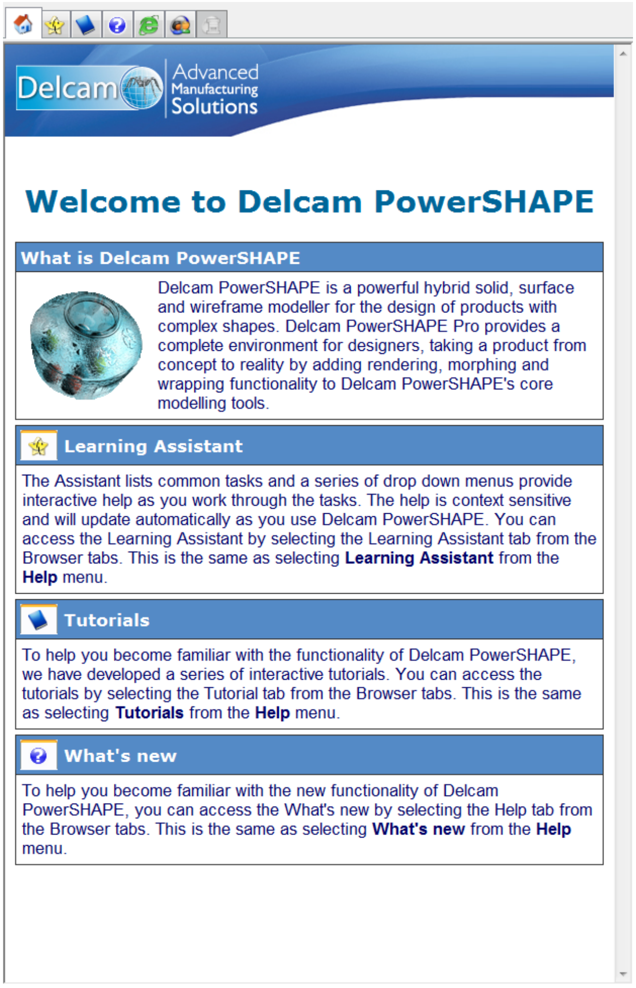
Select this option to display the contents and index of the help file.

The right hand frame displays the welcome page and provides links to further information about the program and the on-line tutorials.

PowerSHAPE Help Contents

PowerSHAPE has an extensive support that can be accessed by either selecting Help from the top menu or by pressing **F1** on the keyboard. The

1 Press **F1** on the keyboard to launch the HTML browser window.

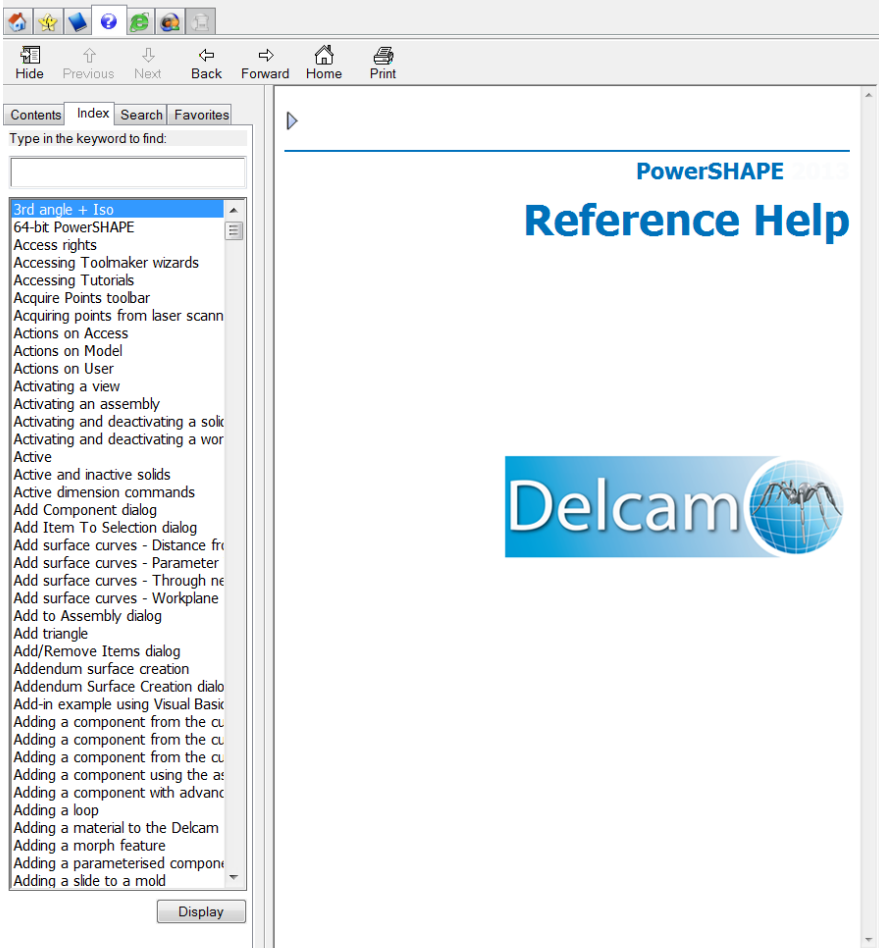


The window displayed above is the welcome page and can be accessed by selecting the small home tabbed icon at the top of the window. A number of other options are available to help navigate the information.

2 Select the Learning assistant to access an interactive list of common tasks.

3 Select Tutorials to access free tutorials which can be downloaded or installed from the installation DVD.

4 Select Help to access the complete online reference help which allows you to **search** by **index** or **keyword(s)**.



Topics are listed in the left-hand window and the specific help for a selected item is displayed in the right pane.

The **Hide** button can be pressed to hide the topic list. The button then changes to **Show** which again redisplays the topic list.

5 Close the browser window using the button at the top right corner.

Help > What's New

Use this option to view details of new features in the latest release. The What's New is displayed in one of the following ways:

- If full documentation is not installed, What's New will appear in the browser window.
- If full documentation is installed, What's New will appear in a new, floating window.

Help > Getting Started

Select this option to display the browser. Use the browser tabs to select the type of help you require.

Help > Learning Assistant

This displays and removes the Learning Assistant window.

The Learning Assistant lists common PowerSHAPE tasks. A series of dropdown menus provide interactive help as you work through the tasks. The help is context sensitive and will update automatically as you use the program.

You can also use the Learning Assistant's drop down menus to execute a command.

To create a single line using the Learning Assistant,

1 In the Assistant window, click the **Creating Objects** drop down menu.

2 Click the **Lines** button. This will display a new list of options.

3 Click **How?** to see details about creating a line.

Help > Tutorials

PS-Tutorials are a free add-on module. If you are using a non-English version of PowerSHAPE, the tutorials may be displayed in English.

Help > Check for PowerSHAPE updates

Use this option to check the Product Download site so that you can download the latest release version of our products.

Help - Check for PAF updates

Use this option to check the Product Download site for a new PAF to download. To check for a PAF updated you need to be connected to the internet and have a valid licence file.

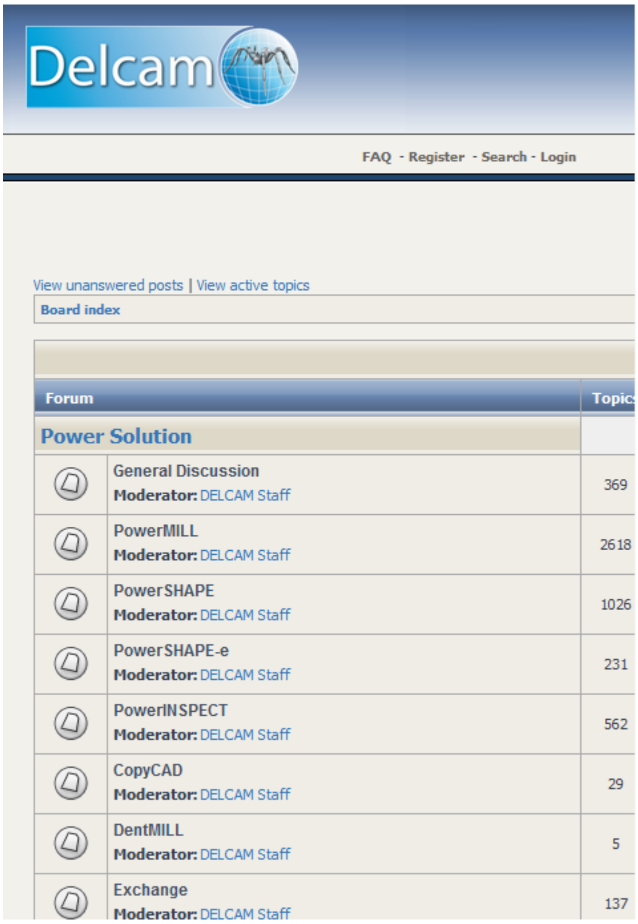
Help > Subscribe to the PowerSHAPE newsletter

This generates an email using your default mail tool. When you click Send, your email address is added to the distribution list for the Delcam email newsletter for PowerSHAPE.

Help > Visit the User Forum

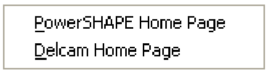
This links to the Power Solution User Forum on the www.delcam.com (http://www.delcam.com) website.

The User Forum is intended for new and established users to join and participate in user discussions to share ideas and experiences of PowerSHAPE or other Power Solution products.



Help > Delcam on the Web

This gives options to quickly access our website (providing you have internet access from your computer).



PowerSHAPE Home Page - This displays PowerSHAPE's home page www.powershape.com/powershape/powershape.htm (http://www.powershape.com/powershape/powershape.htm), which gives news and information on PowerSHAPE.

Delcam Home Page - This displays our home page www.delcam.com (http://www.delcam.com). You can find here full PowerSolution news and information.

Help > About

Select this option to display a dialog showing the PowerSHAPE version and copyright details.

The dialog also provides:

- licence and user information.
- the language set on your PC and that used by PowerSHAPE.
- patent information.

Click **OK** to close the dialog.

PowerSHAPE Shortcuts

Shortcuts to menu options

The keyboard can be used to access the menus. Every menu item has an underlined letter.

1. Look at the menu bar and find the menu you wish to access. One of its characters is underlined.

   For example, **F** is underlined for the **File** menu.

2. Hold down the **Alt** key and press the appropriate character to display the menu option.

   For example, **Alt - f** opens the **File** menu.

3. Look at the option menu and find the option you wish to access. One of its characters is underlined.

   For example, **O** is underlined for the **Open** option.

4. Press the character underlined in your required option.

   For example, **O** opens the **Open** dialog.

If you prefer to use the keyboard, here are some other useful controls:

- Once the **Alt** key has activated the menus, you can navigate using the arrow keys.⇧⇩⇦⇨
- To accept a menu, press **Enter**.
- To do nothing and close a menu, press **Esc.**

Shortcuts to frequent commands

The following table lists shortcuts to frequently used commands. These shortcuts also appear on the menus alongside their respective options.

Menu option	Shortcut*
File - New	Ctrl + n
File - Open	Ctrl + o
File - Close	Ctrl + F4
File - Save	Ctrl + s
File - Print	Ctrl + p
File - Exit	Alt + F4
Edit - Undo	Ctrl + z
Edit - Cut	Ctrl + x
Edit - Copy	Ctrl + c
Edit - Paste	Ctrl + v
Edit - Paste Special	Ctrl + e
Edit - Select - Clear selection	Ctrl + d
Edit - Select - Select All	Ctrl + a
View - Single (of model) - Top	Ctrl + keypad 5

Menu option	**Shortcut***
View - Single (of model) - Bottom	Ctrl + keypad 0
View - Single (of model) - Left	Ctrl + keypad 4
View - Single (of model) - Right	Ctrl + keypad 6
View - Single (of model) - Front	Ctrl + keypad 2
View - Single (of model) - Back	Ctrl + keypad 8
View - Single (of model) - Iso 1	Ctrl + keypad 1
View - Single (of model) - Iso 2	Ctrl + keypad 3
View - Single (of model) - Iso 3	Ctrl + keypad 9
View - Single (of model) - Iso 4	Ctrl + keypad 7
View - Single (about selection) - Top	Alt + keypad 5
View - Single (about selection) - Bottom	Alt + keypad 0
View - Single (about selection) - Left	Alt + keypad 4
View - Single (about selection) - Right	Alt + keypad 6
View - Single (about selection) - Front	Alt + keypad 2
View - Single (about selection) - Back	Alt + keypad 8
View - Single (about selection) - Iso 1	Alt + keypad 1
View - Single (about selection) - Iso 2	Alt + keypad 3
View - Single (about selection) - Iso 3	Alt + keypad 9
View - Single (about selection) - Iso 4	Alt + keypad 7
View - Shading - Wireframe	F2
View - Shading - Shaded	F3
View - Shading - Transparent Shaded	F4
View - Shading - Transparent Shaded Wire	F10
View - Shading - Shaded Wire	F11
View - Shading - Dynamic Hidden Line	F12
View - Look From - Last View	F5
View - Zoom Full	F6
Tools - Model Analysis - Smoothness Shading	F7
Tools - Model Analysis - Undercut Shading	F8
Tools - Model Analysis - Minimum Radius Shading	F9
View - Blank - Blank Selected	Ctrl + j
View - Blank - Blank Except	Ctrl + k
View - Blank - Blank Toggle	Ctrl + y
View - Unblank	Ctrl + l
View - Refresh	Ctrl + r
Object - Surface - Smart Surfacer	Ctrl + t
Object - Surface - Fillet	Ctrl + f

Menu option	Shortcut*
Object - Curve - Composite Curve	Ctrl + h
Macro - Run	Ctrl + m
Swap individual graphics windows in the Window menu	Ctrl + F6
Help - Contents	F1

* For the keyboard shortcuts to work, you must have Caps Lock Off and Num Lock On.

Other shortcuts

Operation	Shortcut*
Bold text	Ctrl + b
Italic text	Ctrl + i
Underline text	Ctrl + u
Diameter character in text	Ctrl + Alt + d
Degrees character in text	Ctrl + Alt + s
Plus/minus character in text	Ctrl + Alt + p
Interrupt current operation	Esc
Exiting a creation mode	Esc
Halt program (after accepting confirmation dialog)	Esc - 3 times within 1 second

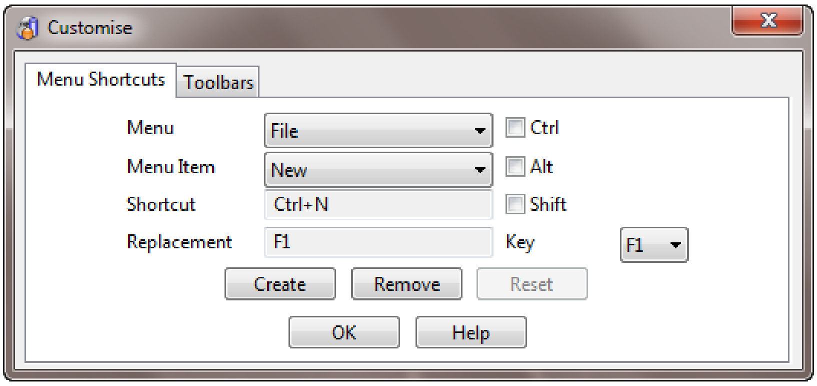
* For the keyboard shortcuts to work, you must have Caps Lock Off and Num Lock On.

Customising menu shortcuts

You can create your own shortcuts for menu options.

1 From the **Tools** menu, select **Customise**.

2 Use the **Menu Shortcuts** tab of the **Customise** dialog to set up your own shortcuts.



Menu - Select a menu from the list of the menus that appear at the top of the screen. .

Menu Item - The options for the selected menu. Select the option for which you want to create a shortcut.

Shortcut - The current shortcut for the selected option. If a shortcut does not exist, **None** is displayed in the box.

Replacement - The shortcut that you want to use for the selected menu option.

Ctrl/**Alt**/**Shift** - Turn *ON* these options if you want to use any of these keys as part of the shortcut.

Key - A list of keys you can use for the shortcut. If none of the **Ctrl**/**Alt**/**Shift** keys are selected, you can only choose one of the **F** function keys. If any of the **Ctrl**/**Alt**/**Shift** keys are selected, you can choose any alphanumeric character or an F function key.

Create - Assigns the replacement shortcut to the select menu option. The replacement shortcut now appears in the **Shortcut** box. The new shortcut will also appear next to the menu option. If a shortcut already exists, you are asked if you want change its use.

Remove - Removes the current shortcut for the selected menu option. None appears in the **Shortcut** box.

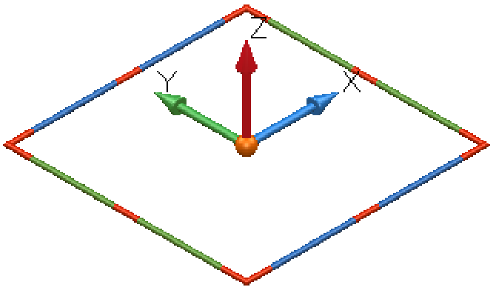
Reset - Resets all the shortcuts back to the original shortcuts.

OK - Removes the dialog from the screen.

3. Workplanes

Workplanes

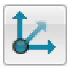
Workplanes are **user-defined datums**. They can be **positioned** and **aligned** as required to simplify model creation.

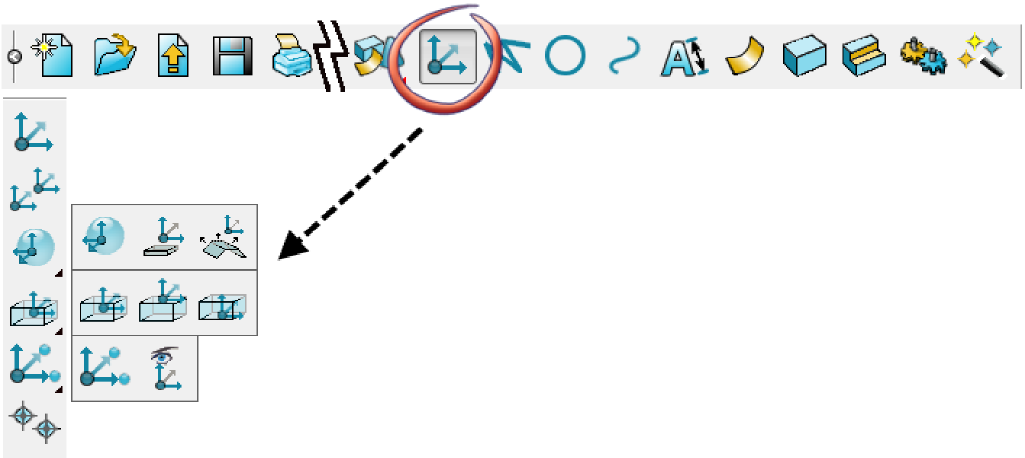


An active **workplane** represents the current **workspace** and its **origin**.

Any number of **workplanes** may exist in your model, but only **up to one can** be **active** at a time. You may **activate** and **deactivate workplanes** at any stage.

When a **workplane** is **active** it becomes the **XYZ datum**, visually larger in size, and changes colour from grey to the coloured symbol above (when selected) or red. **Model entities** can be **copied** or **cut** from the **currently active workplane** and then **pasted** back in a **different position**, **relative** to a **new active workplane**.

The main **Workplane** button  is located in the **Main** toolbar. When selected, it displays the following buttons in the workplane toolbar to the left of the graphics window.

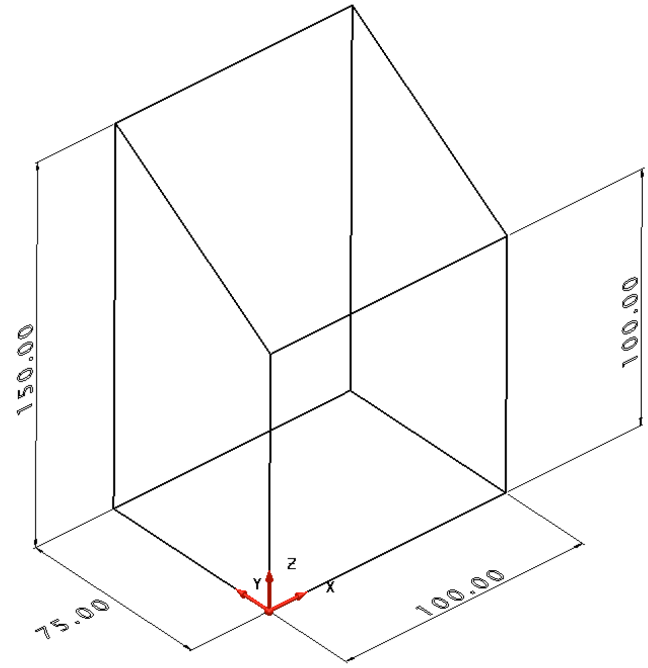


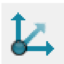
A simple wireframe box with angled top will be initially created to demonstrate the basic applications of workplanes.

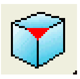
Box Example

1 Open a **new model**.

The wireframe for the dimensioned model (as shown below) will now be created.



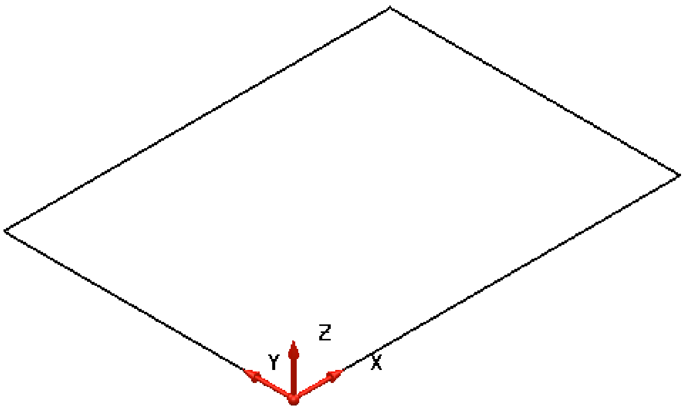
2 Select **Single Workplane**  and enter **0** into the **Command Input** window to position it on the default **World** datum.

3 Select the view **Iso1** .

4 From the **Line** toolbar , select **Rectangle** .

5 Enter **0** in the **Command input** box and press **Enter.**

6 Enter **100 75** in the **Command input** box and press **Enter**.



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

7 From the **Line** toolbar, select **Continuous Lines.** 

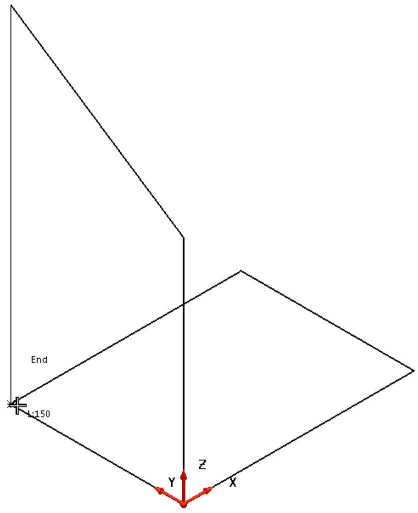
8 Snap the start of the line onto the workplane (or enter 0)

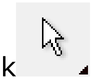
9 Enter **0 0 100** in the **Command input** box and press **Enter**.

10 Enter **abs 0 75 150** in the **Command input** box and press **Enter**.

Adding the prefix abs changes the input from the default relative to absolute coordinates.

11 Snap (left-click) the end of the current line to the top-left corner of the rectangle (marked as *End* by the **Intelligent Cursor** below).



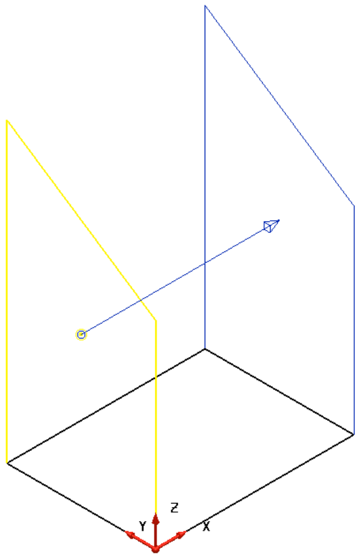
12 Press the **Esc** key or click  to end creation mode.

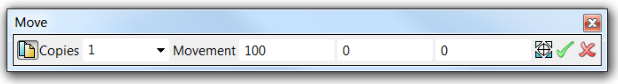
13 Select the last three lines generated.

A copy will be moved to the other side.

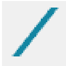
14 Select **Move**  to display the **Move** toolbar.

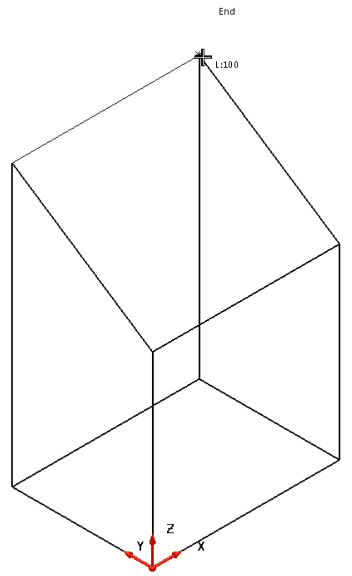
15 Select **Copy the selected items** , enter **100** in the **X input** box, and press **Enter** to display the preview





16 Select the **green tick**  to **accept**.

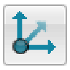
17 Use **Single Lines**  to complete the model by snapping to End points as shown.

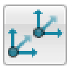


The main wireframe model is now complete.

Creating and Editing Workplanes

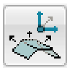
Workplanes within **PowerSHAPE** can be created in the following ways

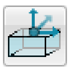
 Create a Single Workplane

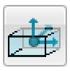
 Create Multiple Workplanes

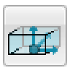
 Create a Workplane aligned with geometry

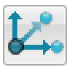
 Create a Workplane Aligned with minimal depth of selection

 Create a Workplane Aligned with the average normal of selection

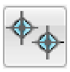
 Create a Single workplane at top of selection

 Create a Single workplane at centre of selection

 Create a Single workplane at bottom of selection

 Create a Workplane from three points

 Create a Workplane Aligned with the current view

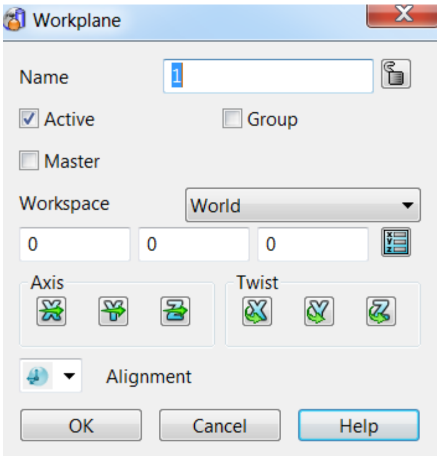
 Create a point

Depending on the option selected, **Workplanes** can be **created by**:

- Snapping onto the screen or geometry using the left mouse button.

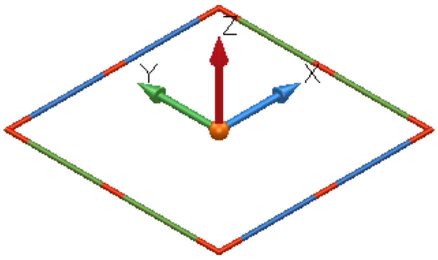
- Entering the coordinates using the command input box
- Generated from the geometry and option type selected.

Workplanes can be edited using the Workplane dialog. This is accessed by double clicking on the workplane or selecting modify from the right click menu.

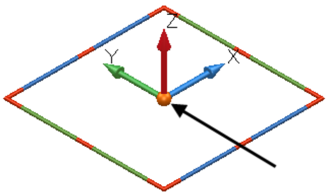


Editing using Graphical handles

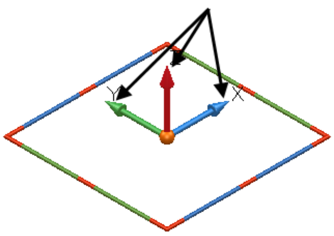
Single selecting a Workplane will display its graphical handles.



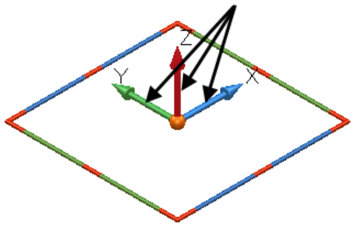
For each graphical edit, the handle to select is shown and summarised below.



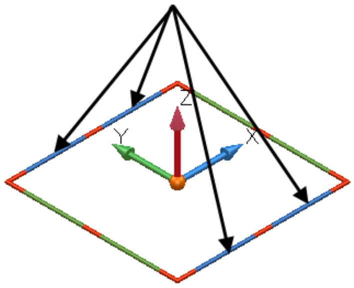
To move the Workplane origin, select the centre point (sphere) and drag. The Confirm Drag dialog will appear for confirmation and/or changes.



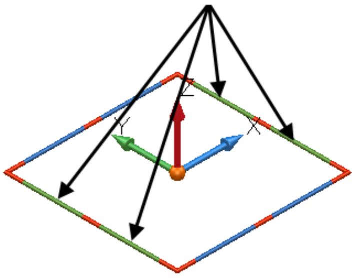
To change the direction of the X, Y or Z axes, select the end arrow head and release over an item. The selected axis will point in that direction.



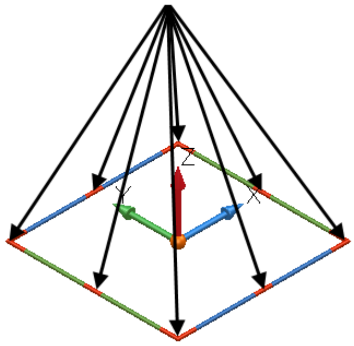
To move the workplane along the X, Y or Z axes, select the handle of the axis itself and drag.



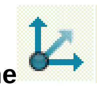
To twist around the X axis, select any blue handle.

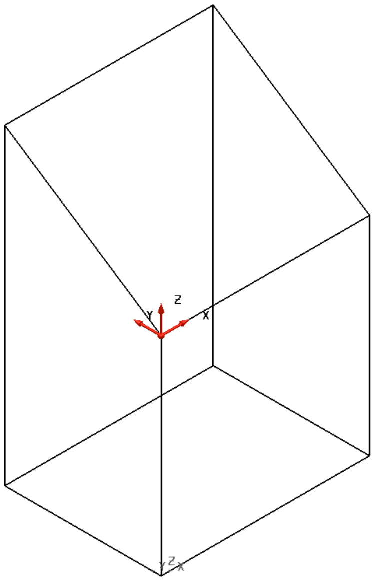


To twist around the Y axis, select any green handle.



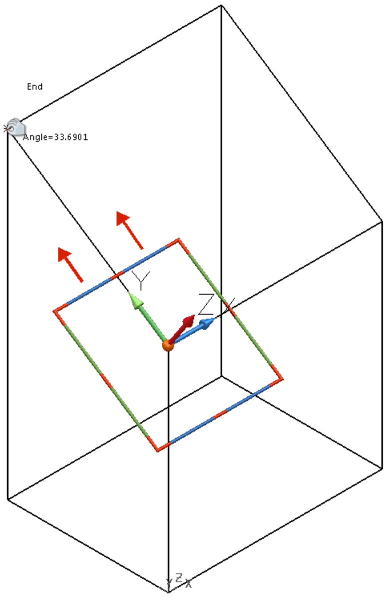
To twist around the Z axis, select any red handle

18 Select **Single Workplane** , and snap a new workplane to the top end of the line running up **Z** from the original **Workplane**.



The **new workplane** is now **active** displayed in **red**, with the **previous workplane** now **deactivated** (shown in **grey**).

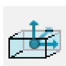
19 Ensure the graphical handles are displayed, then drag any of the blue handles on the +Y and release at the upper left corner as shown.

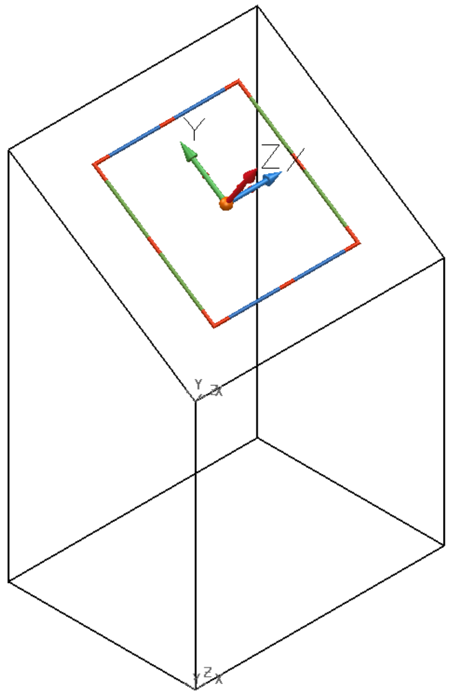


By rotating about the **+X axis**, the **+Y axis** is in line with the angled face.

With the **workplane** now in the **correct orientation**, it will now be centred within the **top wireframe boundary**.

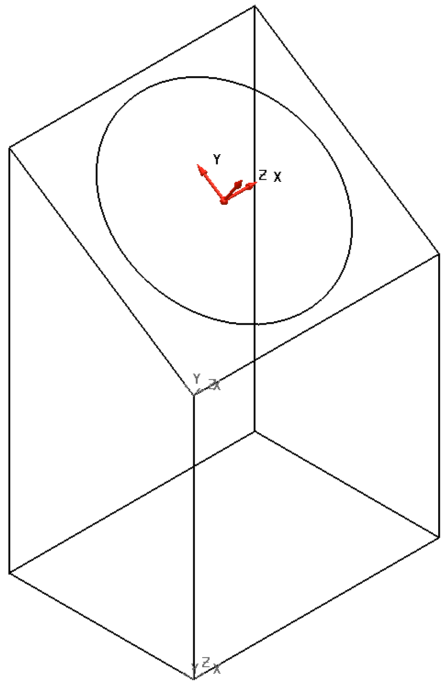
20 Use **Ctrl** and **left-click** to select the four lines that enclose the angled top-face of the model

21 From the **Workplane** toolbar, select **Single workplane at centre of selection.** 

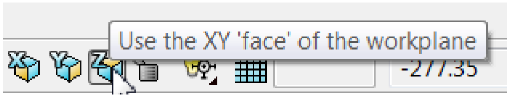


A **new workplane** is **created**, **central** to the **four selected lines**. This workplane is both **active** and **selected**.

22 Create a **Full Arc (Circle)** of **radius 40,** about the currently active workplane.

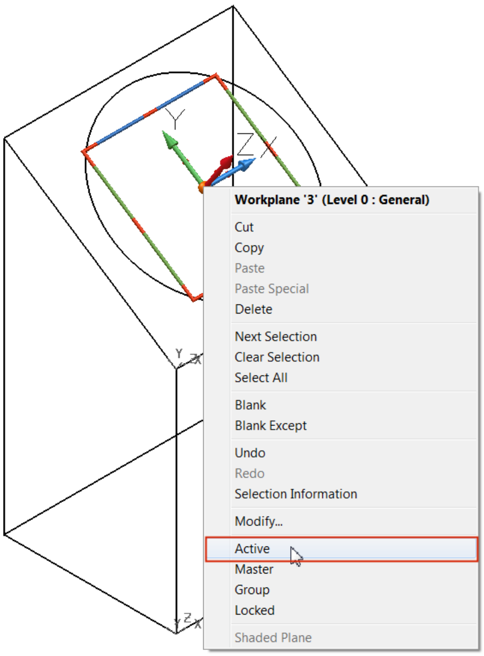


The **Radius 40 circle** has been created on the **XY** face of the **active workplane** and is **central** to the angled-top face of the wireframe model.



Master Workplane

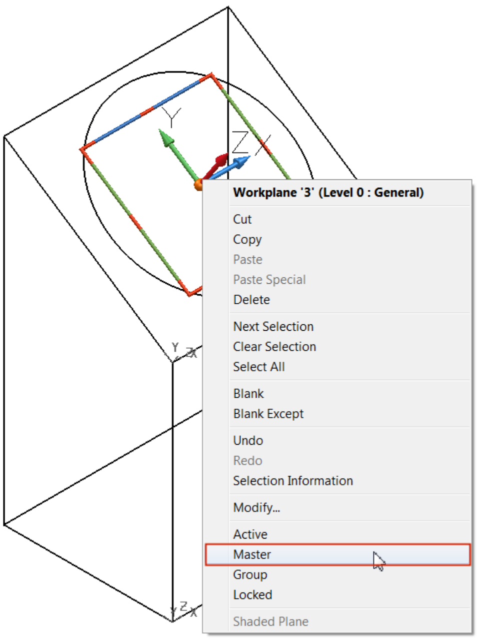
1 **Right-click** on the **active workplane** to open the popup menu and then deselect **Activate.**



There is **no active workplane**; all **coordinates** are now defaulted to the **World** coordinate system.

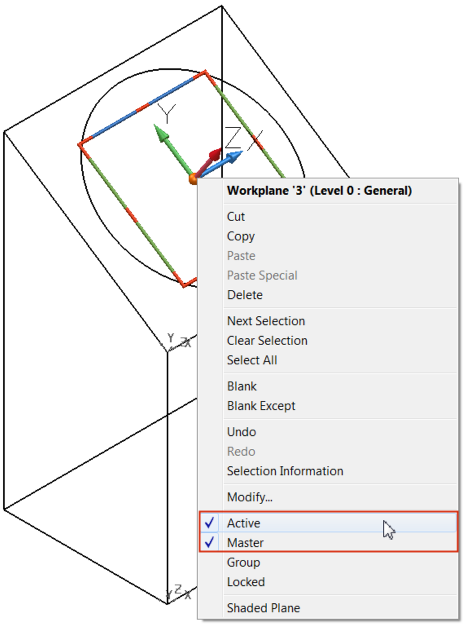
If required, any **Workplane** can be assigned to be the **Master Workplane**.

2 Right-click on the previous active workplane to open the popup menu and then select **Master.**



*The **Workplane** is not **currently active** but has now been assigned as the **Master Workplane**.*

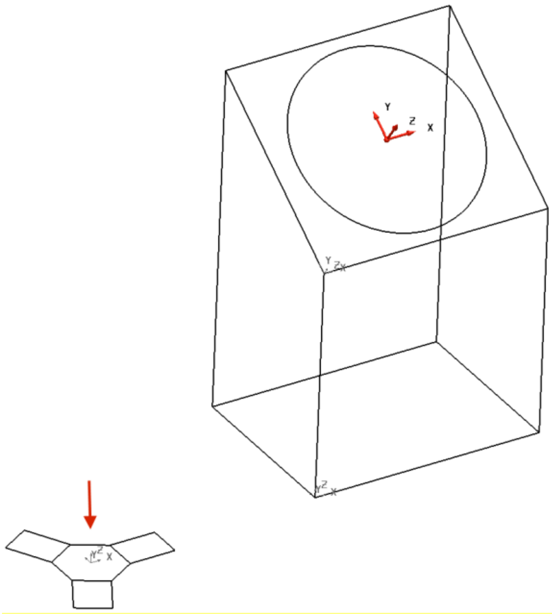
3 **Re-activate** this **Workplane**.



A previously created **2D model** will now be **copied** and **pasted** onto the **active workplane**.

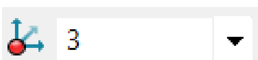
4 **Import** the model:

C:\\Training Data\PowerSHAPE Data**workplane feature.dgk**

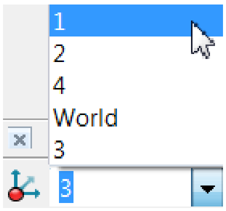


Workplane toolbar

In the **bottom left corner** of the **graphics area**, is a drop-down menu that provides an alternative means to control the **naming** and **activation** of **workplanes**.



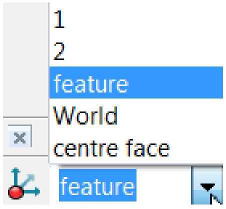
1 Select the **small black arrow** to reveal the **workplanes** currently **available** in this session.



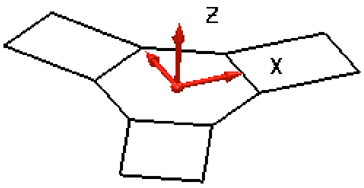
Simply selecting the workplane in this menu makes it **active**. In addition, **Workplanes** can be renamed from the default number provided.

2 Select **Workplane 3** and rename it to **centre face** by typing over and pressing Enter.

3 Rename **Workplane 4** to **feature**.



4 **Activate** the **workplane** renamed feature.

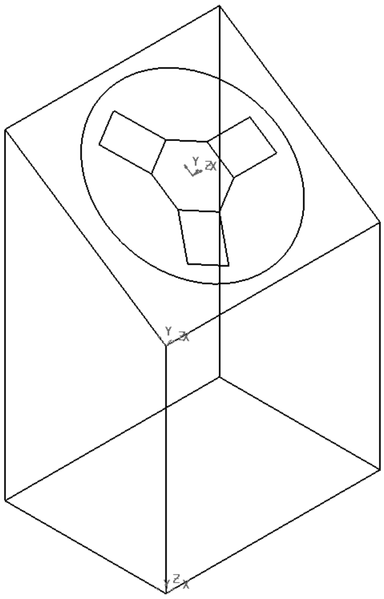


5 Select all the **wireframe** above of the **imported model** then **copy**  from the **main menu**.

The **selected wireframe** is **copied** to the **clipboard**.

6 **Activate** the **workplane** renamed **centre face** using the **Workplane** toolbar.

7 Select **Paste**  from the local menu.



The wireframe imported away from the main model is copied relative to the workplane central to the top-angle face of the main wireframe model.

Data relative to an active workplane in one model can be copied and pasted relative to an active workplane in a different model.

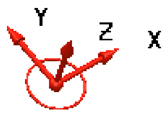
8 Save the model as:

...... Workplane box.psmodel

9 is an section of toolbar that includes a **Workplane** button  for creating temporary workplanes

Temporary Workplanes

If a permanent Workplane is not required, then PowerSHAPE provides the ability to create a temporary Workplane. The icon is visible next to the workplane toolbar.



A temporary workplane cannot be aligned dynamically, nor does it have access to a local editing dialog, but it can be modified using the **General Edits** toolbar. It is automatically named **Temporary** and can be deleted using the local (right click) menu or by reselecting the **Temporary Workplane** button.

Workplane alignment of molding component

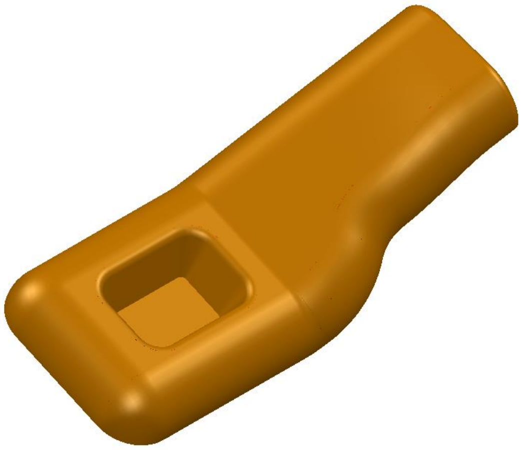
When a component model is imported, it may not be in a suitable orientation for such tasks as creating a tool around it, or for maximum accessibility to a 3-axis machining operation. In these cases, workplanes are applied to achieve a more suitable location and orientation for the model. To maintain dimensional accountability to the original component, it is essential to move workplanes around the model (as opposed to physically moving the model relative to the world datum).

1 **Close** any previous PowerSHAPE models.

2 **Import** the surface model:

.....\PowerSHAPE_Data**InteriorTrim.dgk.**

3 If the model is not visible, select **Resize to Fit**  to centre in page.

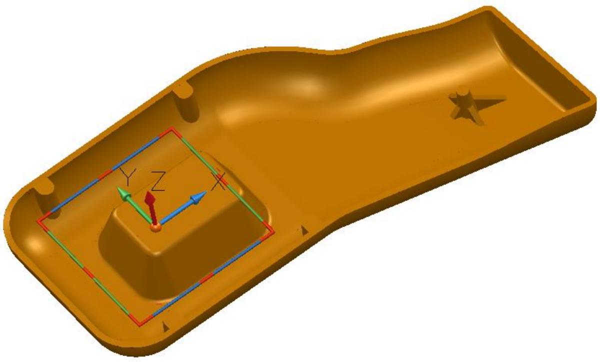


The **imported model** of a **plastic molding** is **not** in a **suitable orientation** from which to create a **mold tool**. A **suitably positioned workplane** will be **created** to provide an **appropriate tooling datum**.

The most obvious choice for a **tooling alignmen**t is **normal** to the **base** of the pocket.

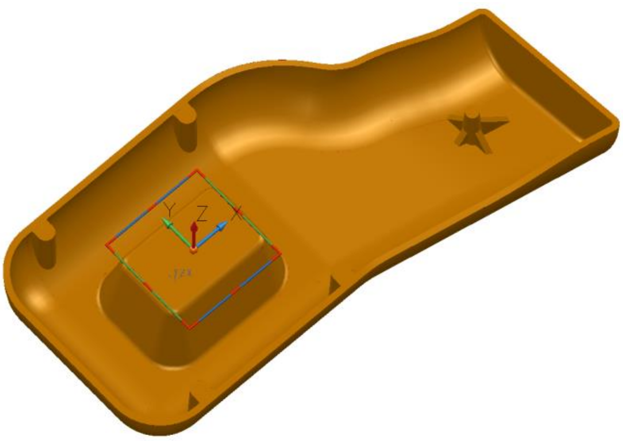
4 Use the middle mouse key to rotate the view to display the underside of the component.

5 Select **Single workplane aligned to geometry**  and left-click anywhere on the flat surface defining the underside face of the pocket.



The Z axis is normal to the surface. With this orientation, the centre of the pocket will be calculated.

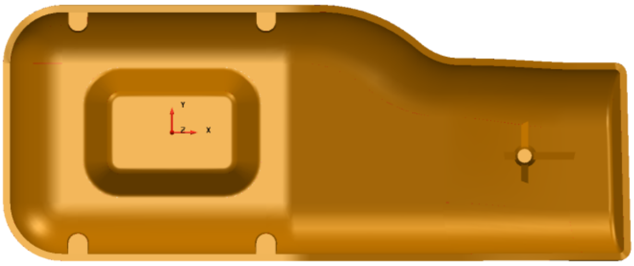
6 Select the surface only defining the underside face, the select **Single workplane at top of selection.** 

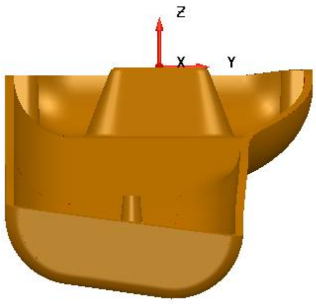


The **workplane** is **positioned central** to the **selected surface**.

7 Select and **delete** the original workplane used to create the alignment with the base of the pocket.

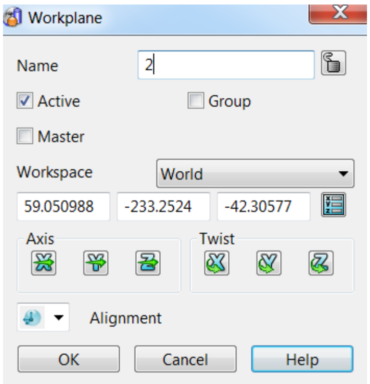
8 View the model in all directions to check that a suitable tooling alignment has been achieved.

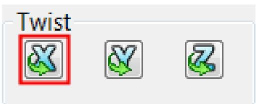




It is possible that the model may need to be rotated 180 degrees about the **X** axis at some stage. This can be performed using the **Workplane** editing dialog.

9 Right-click on the workplane, and select **Modify** in the popup menu.

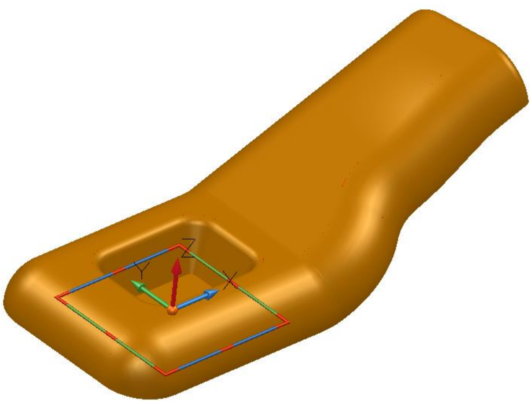




10 Select **Twist X**.

11 Enter **180** in the **Calculator** dialog, and select **OK**.

12 Select **OK** in the **Workplane** dialog.



The model is now viewed the other way up, relative to the new orientation of the workplane.

13 Save the model as:

......\PowerSHAPE-Models\Interior Trim ex1

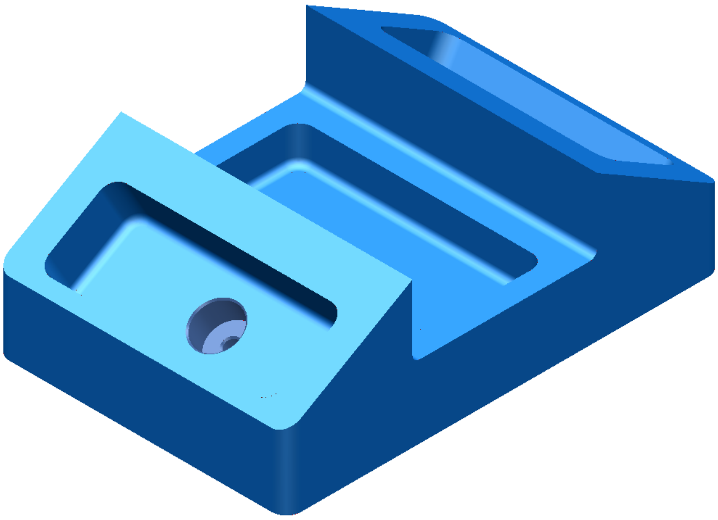
14 Close the model.

Exercise 3: Workplane creation

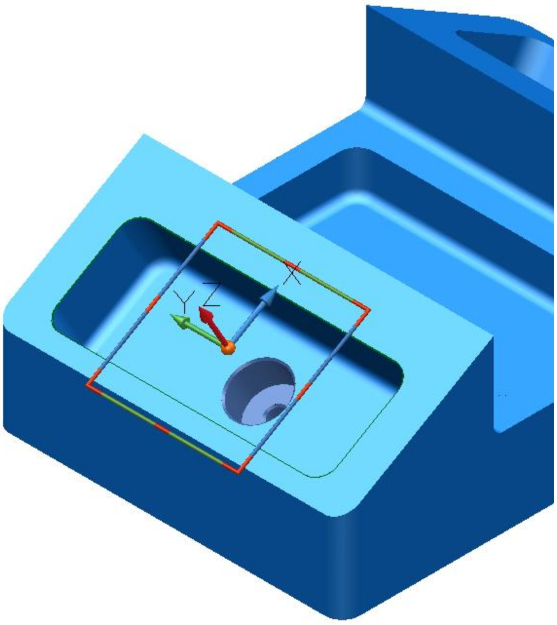
1 **Open** the surface model:

.....\PowerSHAPE_Data**Exercise 3 model.psmodel**

2 View in ISO 1. 

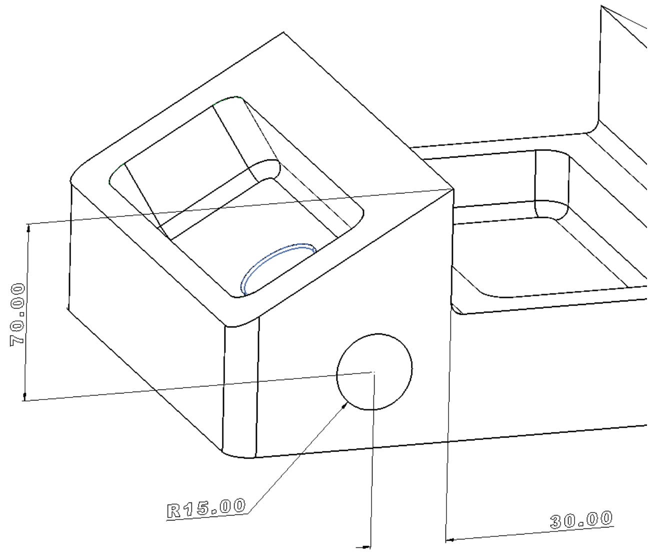


3 Create a workplane at the **centre of the pocket** aligned to the top sloped face as shown.

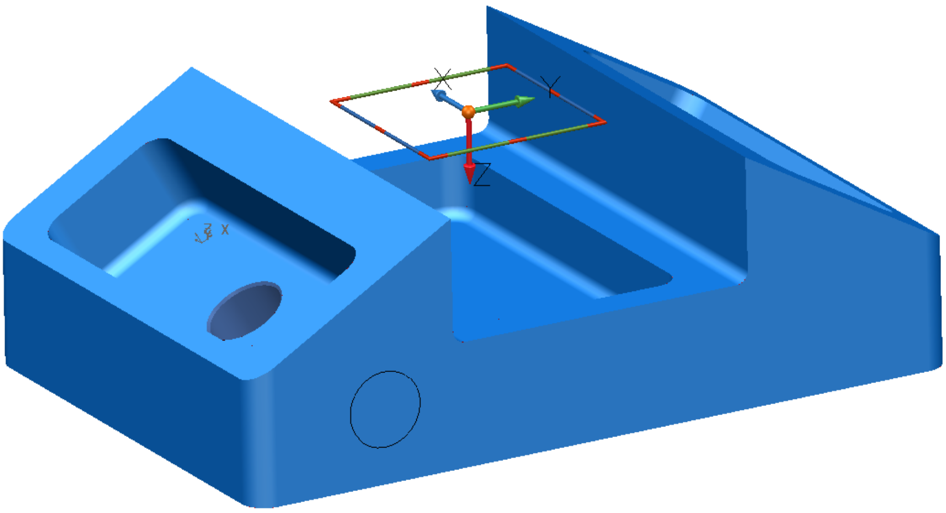


4 **Rename** the Workplane **Angled left pocket**.

5 Create a **radius 15** circle in the position shown.



6 Create a Workplane at the **top centre** (**highest point**) ensuring the **+Z** is **directed towards** the **component** as shown.



7 **Save** the **model** as:

....../PowerSHAPE-Models/**Ex 3 Workplane creation.psmodel**

4. Levels

Levels enable you **to group model entities** and help **simplify the display** and selection of items. Levels can be **distinct groups of component surfaces**, or **different types of entity**, and they can be **renamed** to identify the items they contain.

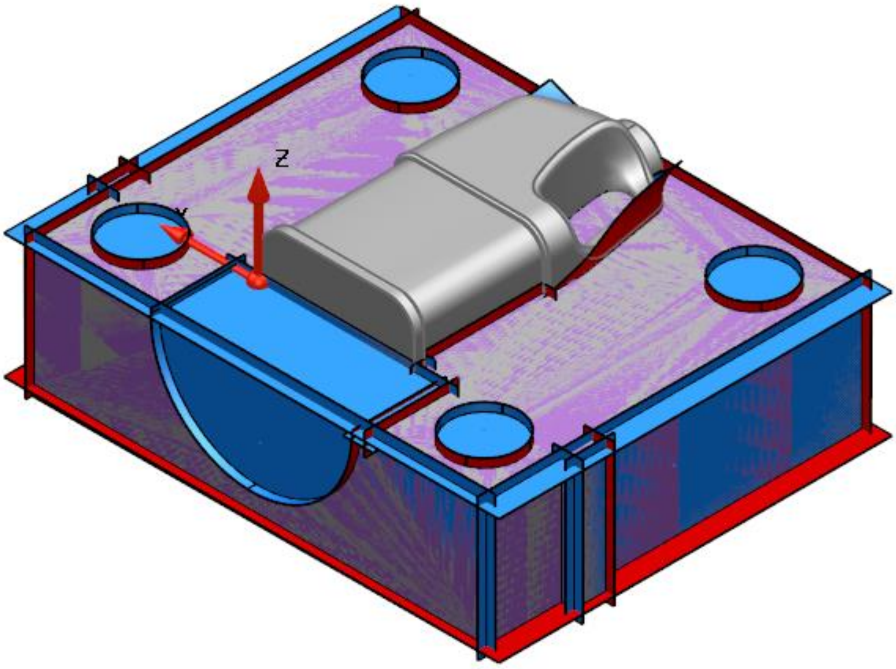
Objects can be assigned to **different levels**, which can then be **displayed** or **hidden** to help **manage the selection** and **visualisation of entities** within a **model**. A typical use is to assign the inner wall surfaces of a component to a one level, and the outer wall surfaces to another.

PowerSHAPE Pro supports up to **1000 levels** (0 – 999), but levels 998 and 999 are reserved for internal use.

Example

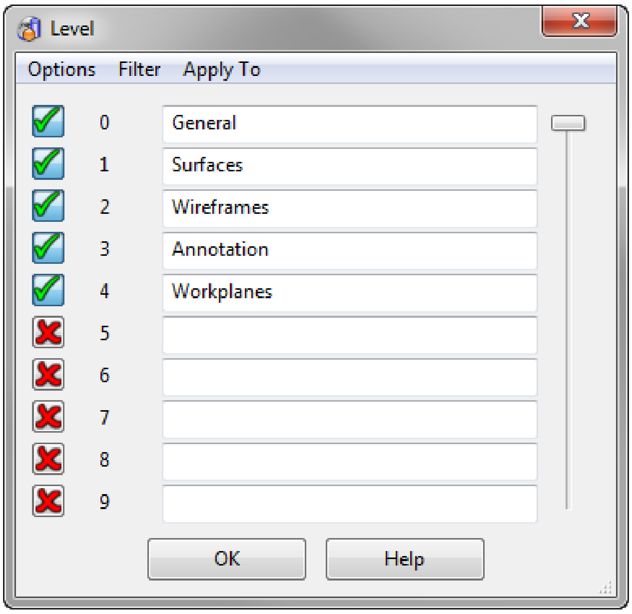
1 **Open** the model......PowerSHAPE Pro data/**re-eng levels.psmodel**

As you can see this **model** contains a mixture of **Solids**, **Surfaces** and **Mesh** used to aid the **reverse engineering** process for this model. Currently all the different entities are visible and overlapping each other due to the nature of the process. Blanking (**Ctrl+J**) or Blank Except (**Ctrl+K**) certain **entities** is good for more **short term requirements**, however a more **semi-permanent solution** is **best** in this case.



2 Ensure **everything** in the model is **visible** by **unblanking** all entities (**Ctrl+L**).

3 From the lower-left side of the window open the **Levels** form. 

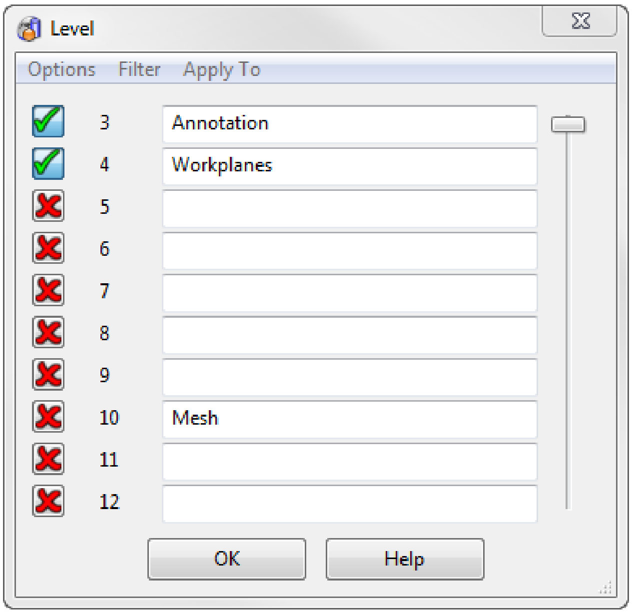


Each **level** has a **number**. **Visible** levels are indicated by . **Hidden** levels are indicated by . You can turn ***ON*** a level by clicking and ***OFF*** by clicking .

PowerSHAPE Pro has a number of **default levels** (0 – 4), while a **new level** is created by simply typing a **name** into the **name box** adjacent to the **level number**.

Level names can be edited by just modifying the existing name and are **visible** by **hovering** the **mouse** over the **level number**.

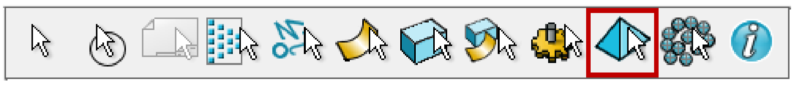
4 In level **10**, enter the name **Mesh** which will create a level called 'Mesh' on number 10 which is turned **OFF** by default.



5 Select **OK** to close the **dialog**.

6 From the **Quick Select** toolbar, select **Quick Select All Mesh**.

This will select ALL visible mesh in the model.



7 **Click** the **middle mouse button** over the level **10** icon on the **levels toolbar** at the **bottom left** of the **window**, to move the selected items to that level.

*The **Mesh**, which is purple in **PowerSHAPE Pro**, should now **disappear** from the **graphics window** because the level is turned **OFF**.*

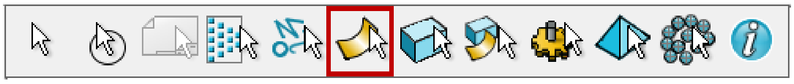
8 Toggle **Level 10 ON** and **OFF** to highlight that the **Mesh** has been successfully placed on the correct **level**.

9 Open the **Levels** form and create a level called '**Untrimmed Surfaces**' on **number 20**.

*This has created a level to place the **untrimmed surfaces** creaed using the **Automatic and Manual Mesh Segmenter** tools that we will be using later in the notes.*

10 Select **OK** on the form.

11 **Quick Select All Surfaces** from the **Quick Select** flyout menu.



12 Place all **the selected** surfaces on **level 20** using the **middle mouse button**.

*Again, as the level is turned **OFF** the **surfaces** should **disappear** from view.*

13 **Right-click** over **level 20** on the **levels** toolbar and select **Solo**.

*This will turn **ALL** other levels **OFF except level 20**. Again this will highlight that the **surfaces** have been **correctly located** on the **level**.*

This **model** also contains a number of **solids** and some **contruction wireframe** making up different components of the same group of parts. We have the **lower half** of the **mold** and the **complete component solid models** in this session. When we would like to create a number of different levels to organise a family of parts and construction information we can Group Levels by defining an overall group name and then the individual levels names, separated by a colon.

Grouping levels

This **model** also contains a number of **solids** and some **contruction wireframe** making up different components of the same group of parts. We have the **lower half** of the **mold** and the **complete component solid models** in this session. When we would like to create a number of different levels to organise a family of parts and construction information we can Group Levels by defining an overall group name and then the individual levels names, separated by a colon. By grouping levels together, you can manage parts of the model more efficiently and effectively.

1 From the lower-left side of the window, open the **Levels** form.

2 In **level 30**, enter the name ***Component : Construction Lines***.

3 In **level 31**, enter the name ***Component : Solid Mold***.

4 In **level 32**, enter the name ***Component : Bottle.***

*You may have become aware that we are leaving **defined gaps** between each of the separate '**groups**' of **levels** we create, e.g **10, 20, 30 etc**. This is because looking forward we may require levels in between such as **11, 12, 13 for further objects** and **entities** relevant to each '**group**'.*

The word ***Component*** is the group name and ***Wireframe* and *Solid*** are the level names.

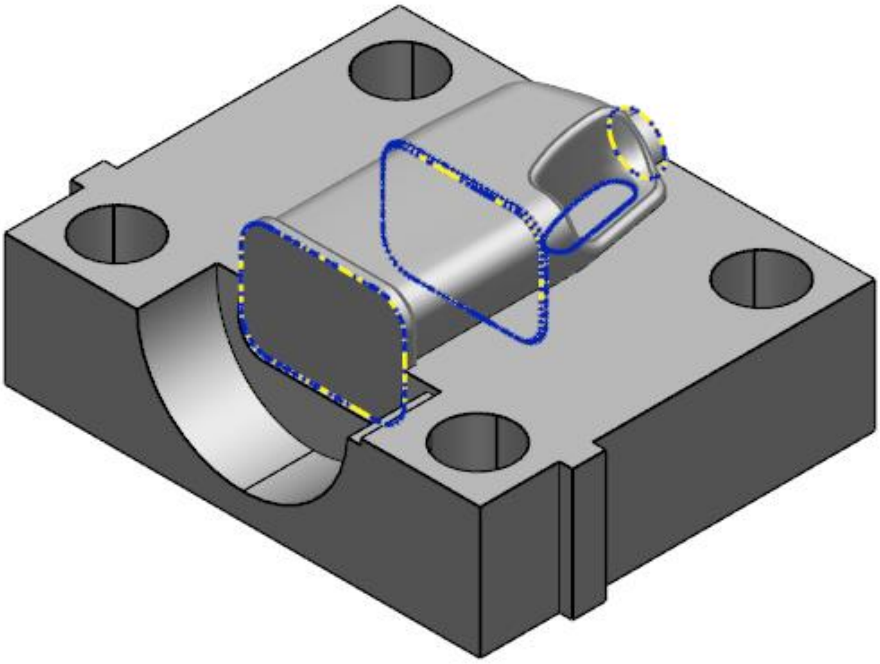
5 Select **OK**.

The **three new levels** now belong to the group called ***Components***.

On the **Levels** toolbar, the buttons of levels **30, 31** and **32** have changed to purple. In addition, a new **Levels Group** button is displayed, also in blue. You can use this **Levels Group** button to turn all levels in the group ON and OFF simultaneously.

*Each new **group** is assigned a **different colour** so you can **distinguish** between them.*

6 Select **Quick select all wireframes** from the **Select** toolbar.



7 With all the **wireframes** highlighted, middle mouse button over level **30.**

8 Select the **component mold solid** and place it onto level **31.**

9 Repeat for the **Bottle** solid onto **Level 32**.

10 Select to switch off **all** the associated **components levels**.

Using Copy and Paste to make copies of entities

In some instances, it is advisable to make safe/backup copies of **solid**, **surface** or **wireframe** entities. For this exercise, we will copy the **mold** solid and paste it onto the general level.

1 Turn on level **31** (**Component : Solid Mold**) using the **right-click solo function** and select the **solid**.

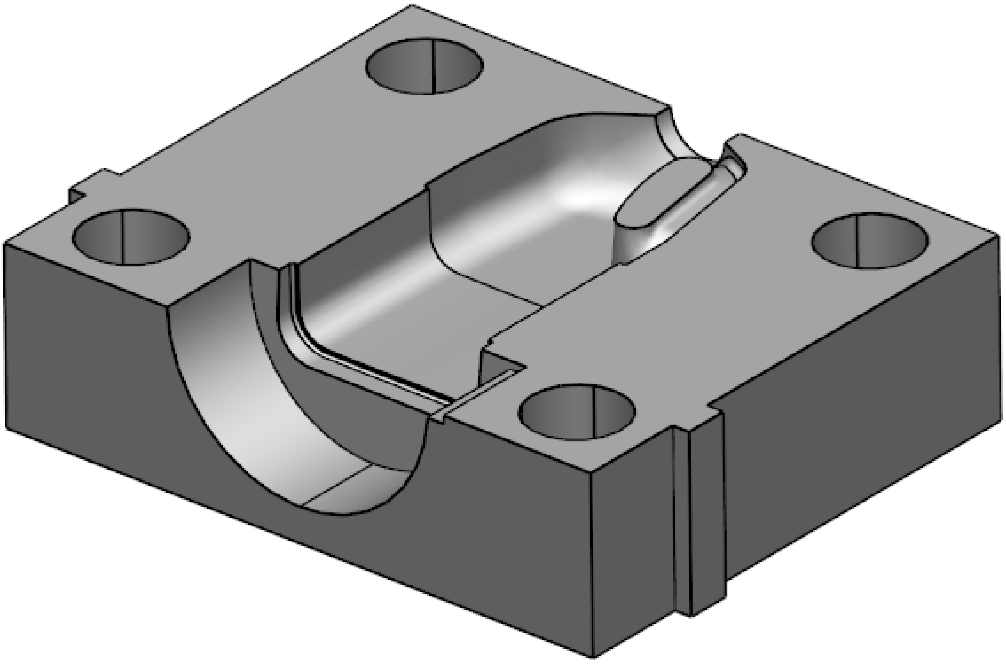
2 From the top toolbar, select **Copy**  and then **Paste.** 

Two identical solids are on **level 31**.

3 Select **one** of the **solids** and move it to **level 0 : General.**

4 Switch off level **31.** Toggle level **0** on and off to see the new copy.

5 **File >Save as**
\PowerSHAPE Pro Models\Mold Levels Example.psmodel



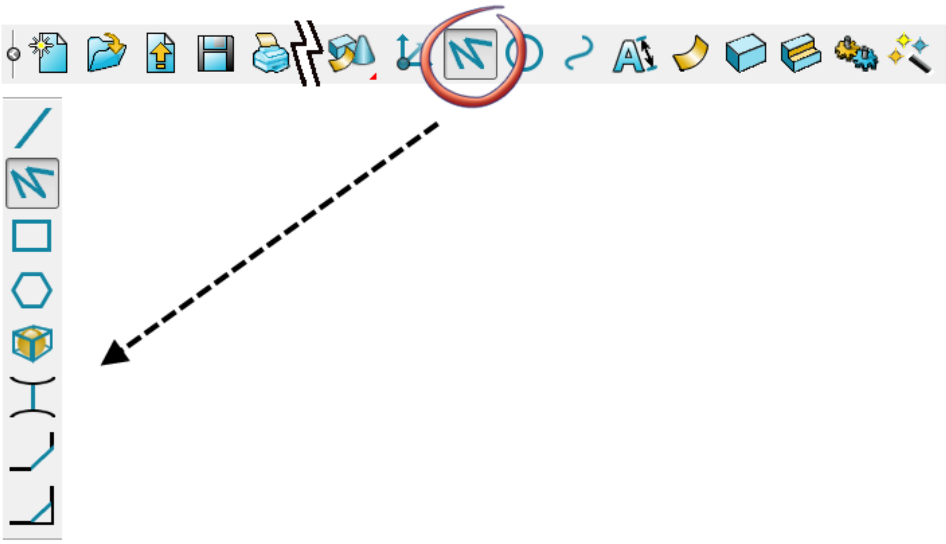
5. Wireframe Modelling

Introduction

PowerSHAPE can generate **points**, **lines**, **arcs**, and **curves** in **2D** and **3D** space, which are collectively known as **Wireframes**. These **wireframes** have several functions, which include being the **basic framework** for several types of **surfaces** and **solids**, and as the drawing entities in **Delcam Draft**. **Wireframes** can be exported in a variety of **file formats** for use in other software products.

The buttons for generating **Wireframe objects** are located within the **main** toolbar.

1 Select **Line** from the **Object** menu.

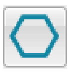


The following **Line** creation options are available.


Single Lines


Continuous Lines


Rectangles


Polygons

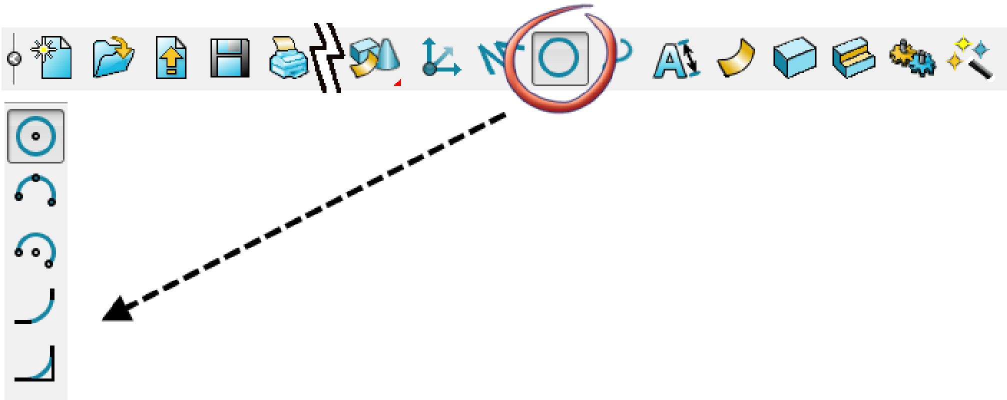
 Selection Box

 Create the shortest line between two selected objects

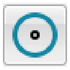
 Chamfers

 Untrimmed chamfers

2 Select **Arc** from the **Object** menu.

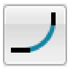


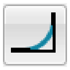
The following **Arc creation** options are available.

 Full arc (circle)

 Create an arc through three points/items

 Create an arc through centre, radius and span

 Fillet arc

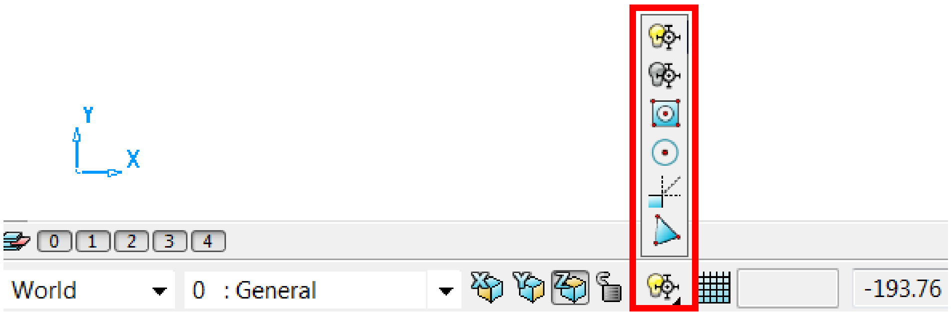
 Untrimmed fillet arc

The Intelligent Cursor

The **Intelligent Cursor** provides dynamic assistance for its two primary functions:

1. **Point input** - it activates construction lines and labels for dragging, snapping and for point creation.
2. **Selection** - it highlights the object under the cursor ready for selection

The Intelligent Cursor is *ON* by default. You can turn the Intelligent Cursor *OFF* (or back on) by using the options on the Intelligent Cursor flyout.



1. Click  to turn the Intelligent Cursor off.
2. Click  to turn the Intelligent Cursor on (default).

When the **Intelligent Cursor** is turned on, objects are highlighted as you move the cursor over them. This shows which objects can be selected if you click at that point

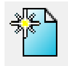
Key points and **features** available are:

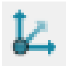
- **End-point**
- **Mid-point**
- **Centre-point**
- **Centre Key-point**
- **On**
- **In**
- **Intersection**
- **Tangent**
- **End Closing a curve**

Construction lines

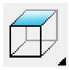
When you move the **intelligent cursor** over an important **key point**, construction lines are created from that **key point**. You can now drag the cursor along a construction line and see the appropriate labels display. Clicking causes the position to snap to that point.

A simple box will be drawn to illustrate this:

1 From the top Menu bar, select **File > New**, or click on **Open New Model**  on the Main toolbar. A **New Model** is opened.

2 Select the **Workplane Icon**  and type **0** into the **Command Window** and then press **Enter**.

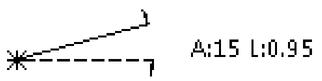
*This will create a **Workplane** at **0, 0, 0** to allow us to visualise the position of the **World UCS**. We will cover workplanes in full later in the course*

3 Select a **View in (+Z)**. 

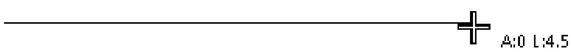
4 From the **Line** toolbar , select create a **Continuous line**. 

5 Click anywhere in the graphics area to start a line.

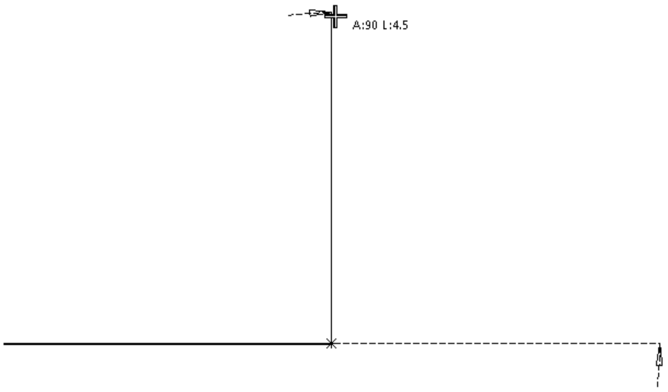
Dynamic feedback on the cursor position indicates **A for angle** (from dashed line) and **L for length**.



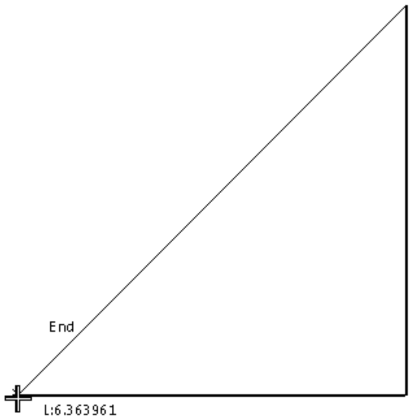
6 Drag along a **horizontal** construction line (A:0) and then click to create it.



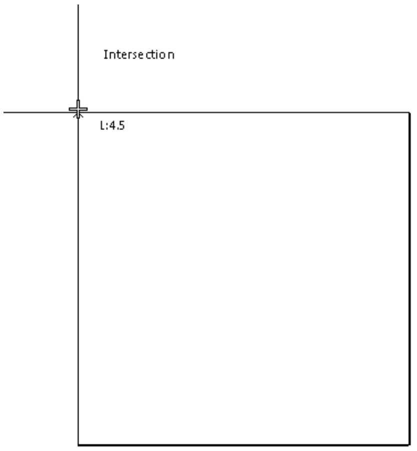
7 Drag along a **90°** construction line (A:90) and click. The line snaps to give a square intersection.



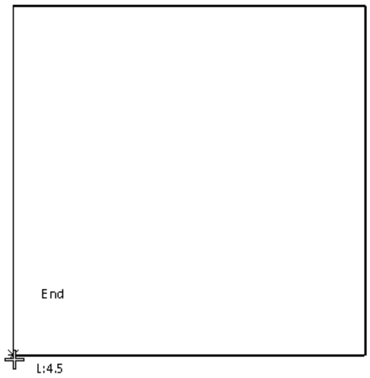
8 Drag the cursor down to the original start point and hover. The cursor displays **End** (do not click).



9 Drag along the vertical construction line until **Intersection** is shown.



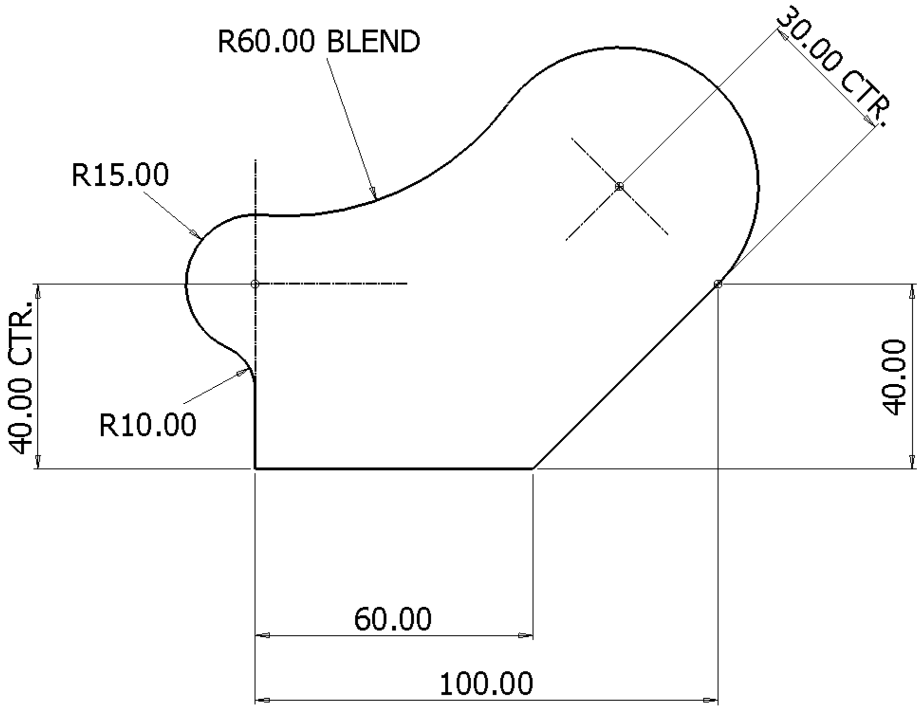
10 Click to snap at the intersection point then finally back down to the start point when **End** is displayed.



11 **Select** and **delete** all wireframes.

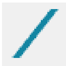
Wireframe Lines and Arcs Exercise

The following exercise demonstrates basic **Wireframe** modelling by recreating the 2D profile shown below. Lines and Arcs will be used to complete the design.

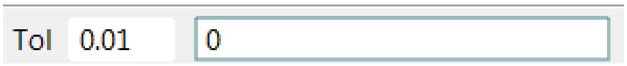


Line Creation

To begin, a single line will be created from **0** to **Y40** to coincide with the **R 15 arc centre** shown above.

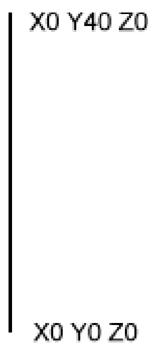
1 From the **Line** toolbar , select **create a single line.** 

2 In the **command input** box, enter the value **0** followed by **Enter**.



This defines the start coordinate for the single line (bottom left corner).

3 Input the values **0 40** followed by **Enter.**



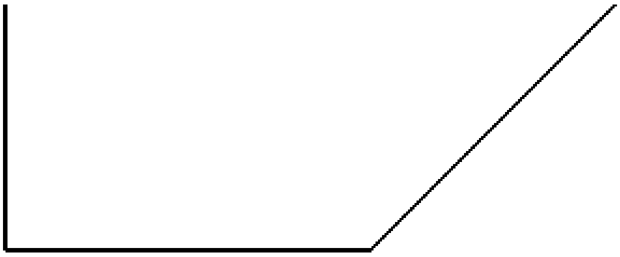
The X Y Z coordinates are entered in this order by default. Each input must be separated by a space and if the second or third values are not given, they are treated as zeros.

4 From the **Line** toolbar, select **create a continuous line**. 

5 Snap to the start point of the first line, or enter **0** in the **command input** box.

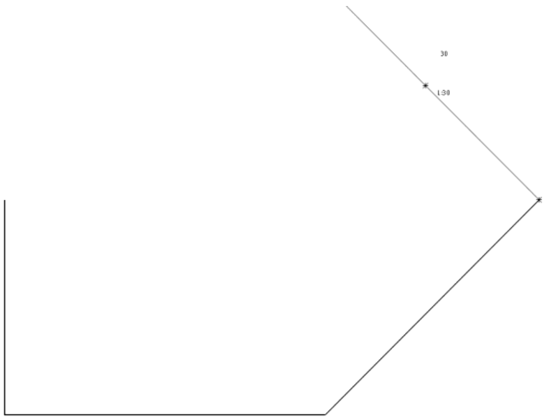
6 Input the value **60** in the **Command input** box, then **Enter**.

7 Input the value **40 40** in the **Command input** box, then **Enter**.



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.

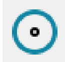
8 By moving the mouse further, drag the end of a new line to a distance of **30** along the normal (**90** degrees) direction and left-click to accept the line.



9 Press the **Esc** key to exit line creation.

Arc Creation

The remainder of the 2D model is to be created using various arc options.

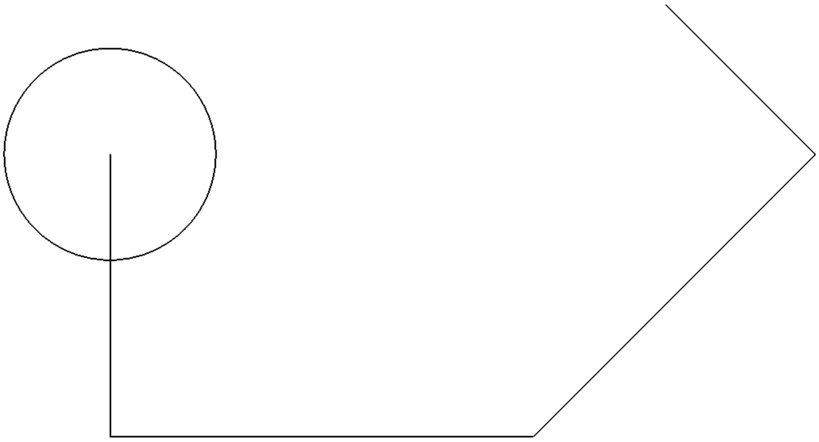
1 From the **Arc** toolbar , select **Full Arc**. 

2 Input the value **r 15** in the **Command** box and press **Enter**.

This input redefines the default radius

A space must be included between the command r and value.

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

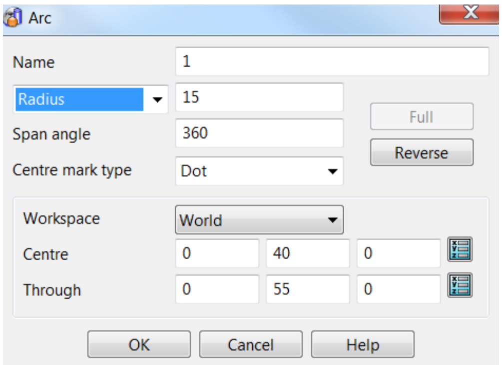


4 Press **Esc** to exit.

Wireframe objects can be edited by:

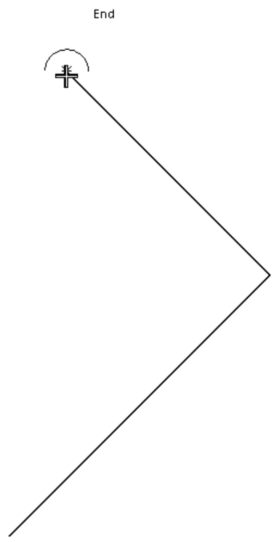
- Double left mouse clicking on the object.
- Right mouse select on the object then Modify from the local menu.

In both methods the editing form appears

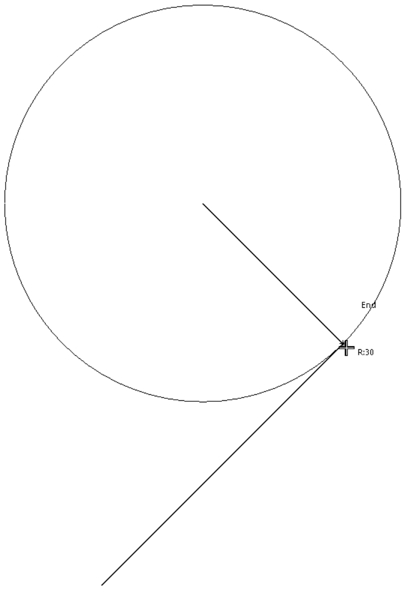


5 From the **Arc** toolbar , select **Arc through centre, radius and span**. 

6 Snap on the open end of the construction line for the centre point as shown.



7 Move the cursor to the other end of the line and left-click.

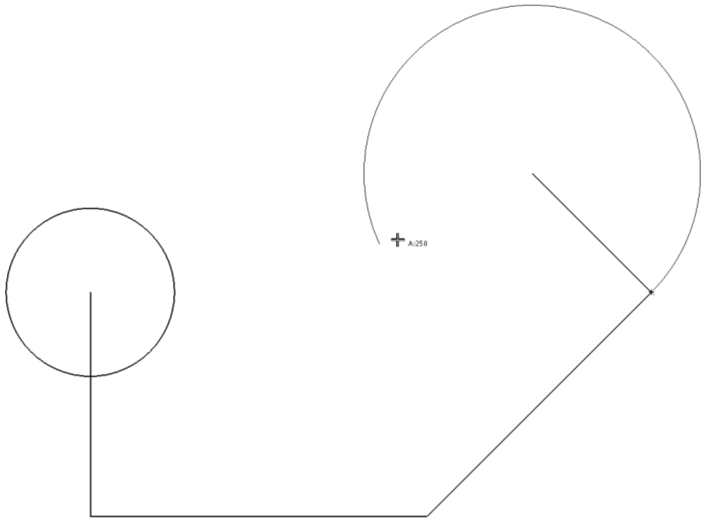


The position is now set for the start of the arc.

8 Release all mouse buttons and drag the arc around counter-clockwise.

The **intelligent cursor** displays the current span angle. When you left-click, the arc is created.

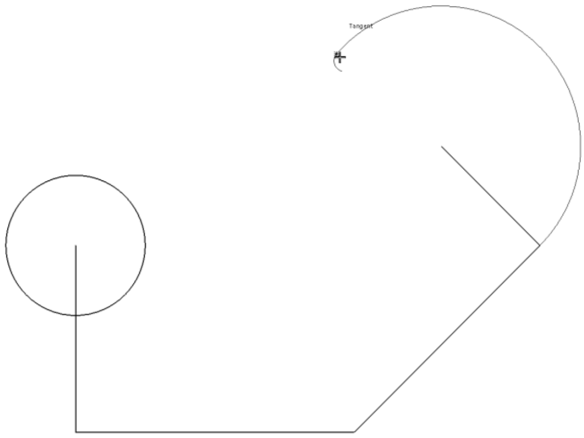
9 Left-click when the arc reaches about **250** degrees.



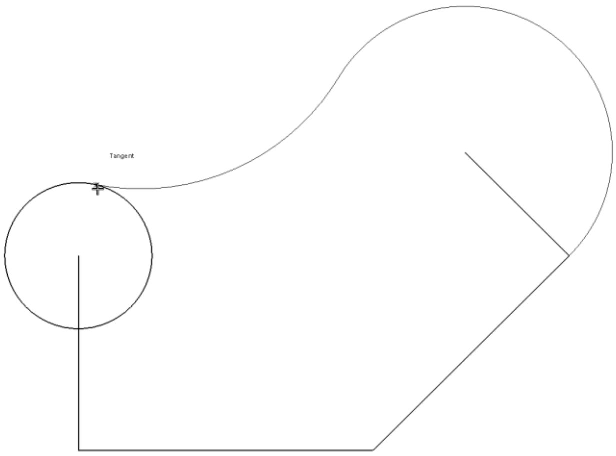
Next, a blend radius (**R 60**) will be generated between the arc and circle using **Create an arc through three points.** The first and second points will be snapped as tangencies to the existing full circle and arc with the third point being entered as the specified blend radius of **60**.

10 From the **Arc** toolbar , select **Arc through three points**. 

11 Use the **Intelligent cursor** to locate and click on the tangent point of the arc.

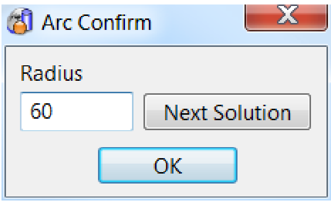


12 Locate and click on the tangent point of the circle.

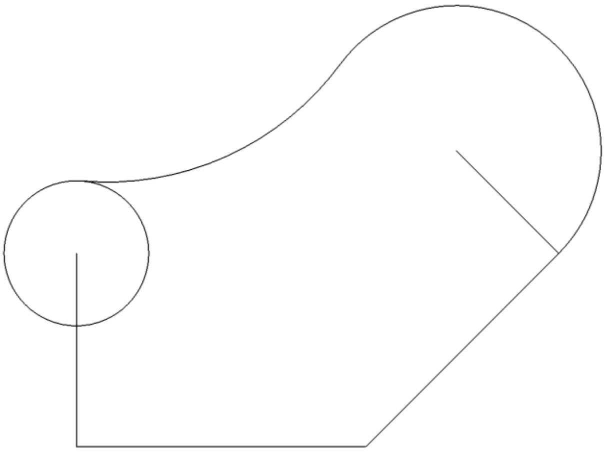


13 Drag the third radius point to the required shape and to a value as close as possible to **60,** then left-click.

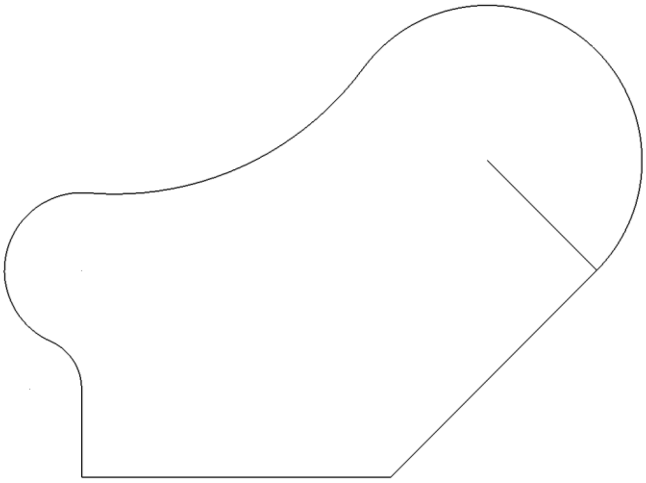
14 If necessary, in the **Arc Confirm** dialog, modify the **Radius** value to **60** and toggle through **Next Solution** until the correct shape is achieved. Select **OK**.



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



15 Use **Arc through three points/items**  to create the radius of **10** between the full circle and the vertical line.



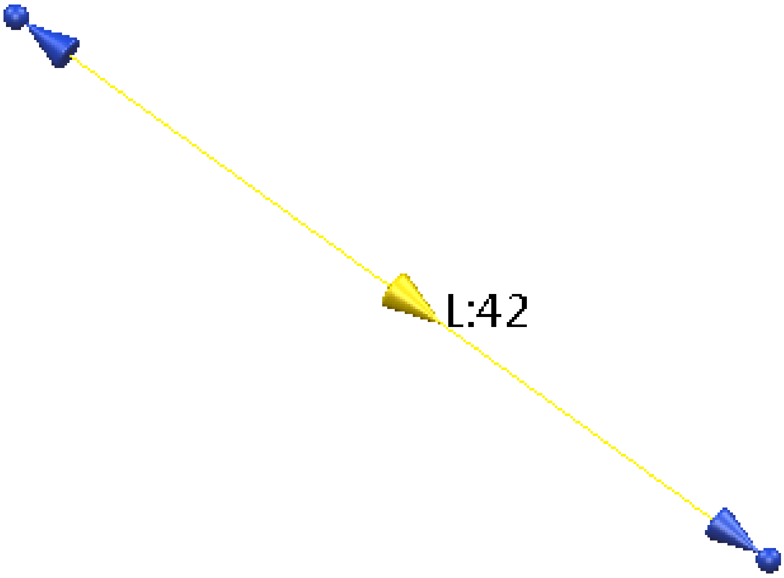
Wireframe Lines and Arcs Editing

Once **Lines** or **Arcs** have been created there are two main ways which they can be then edited. This can be done **Graphically** from within the **graphics window** or using the **Line/Arc Editor** dialog.

Line Edits

As with all **Objects*** in **PowerSHAPE** a single left mouse click on the Object, in this case a **Line** enters the **Graphical Edit** mode with the **3D instrumentation** to allow quick edits.

*An **Object** within **PowerSHAPE** can be decribed as anything you create within the **graphics window** such as: Workplane, Line Arc, Curve, Sold, Surface, Mesh etc.*

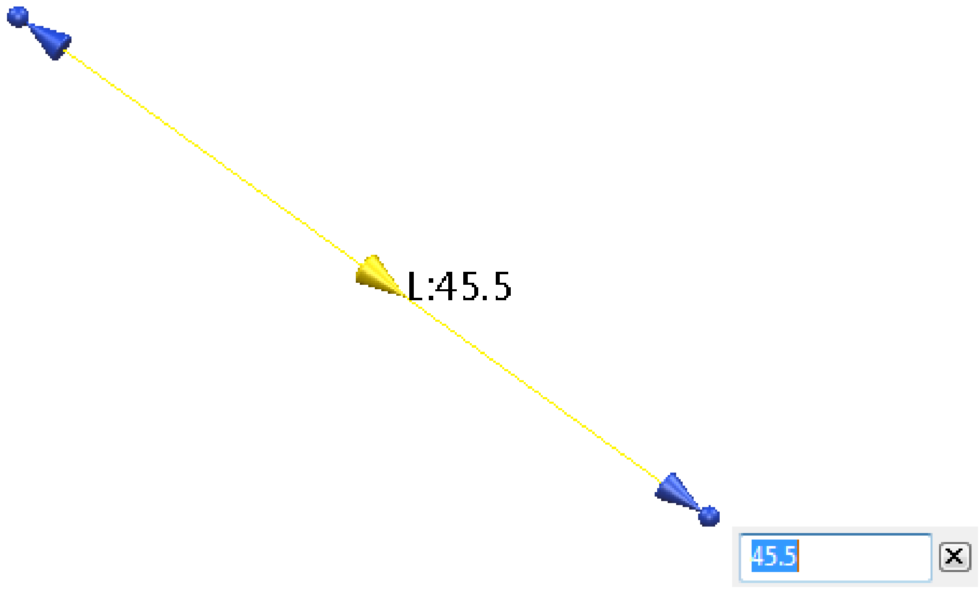


Within this mode the **Line** is highlighted yellow with an arrow indicating the **Direction** of the line and text showing its current **Length**.

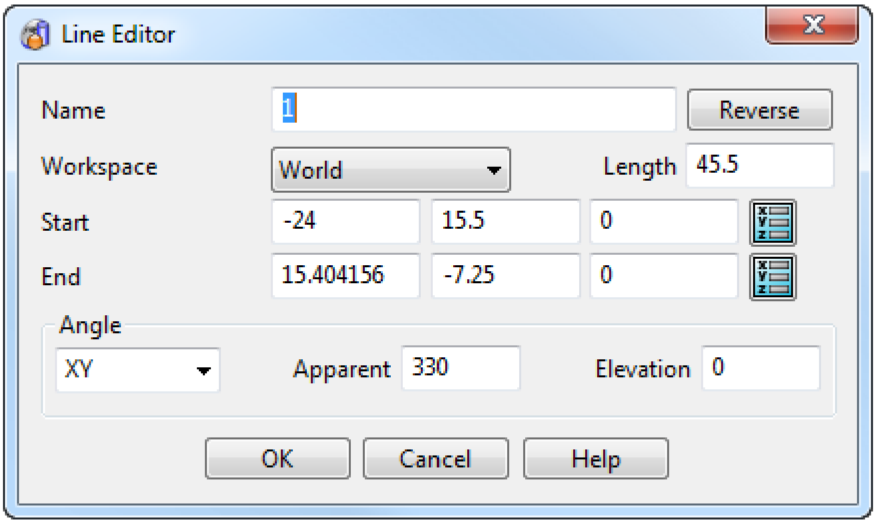
The blue **Arrow** and **Sphere** at either end of the **line** allows you to **edit** the characteristics quickly and easily while the **intelligent cursor** shows the **original position** and allows you to **snap** to other objects.

Clicking and **dragging** the **spheres** will allow either end of the line to move **freely** in **3D space** to change both the **length** and **direction**, while using the blue **arrows** will only affect the **length** and keep the same **axial direction**.

When using the end arrows to change the **length** of a **line**, once the mouse button is **released**, **PowerSHAPE** will give the opportunity to **fine tune** the length as seen in the next image.



To fully fine tune the **Line**, the most accurate option will be to enter the **Line Editor** dialog. Again, as with all **Objects** in **PowerSHAPE** a double left mouse click on the Object, in this case a **Line** enters the **Editor** dialog. This dialog is particularly useful for naming and editing the **relative Azimuth (Apparent)** and **Elevation angles** of the line **accurately**.



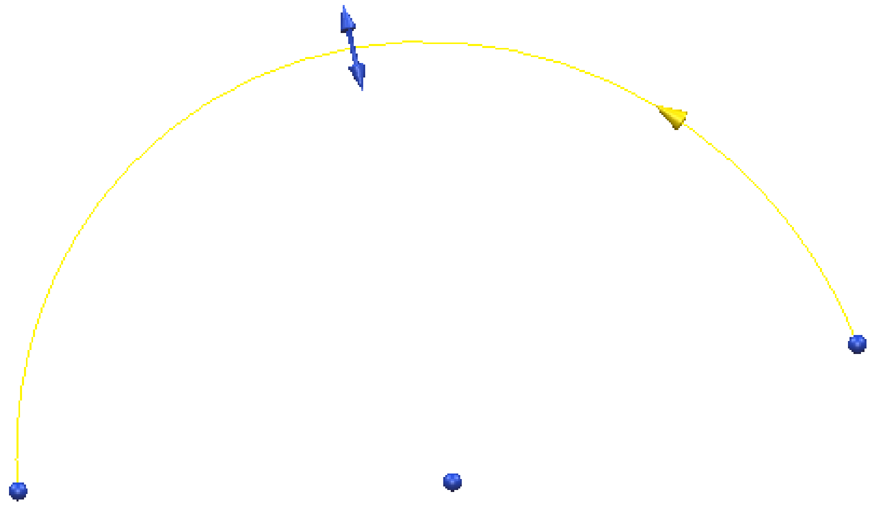
Within this dialog we can edit the **Name**, **Direction**, **Length**, **Start** and **End Point** and **Angles** highly accurately.

Arc Edit

Similarly to **Lines** a single left mouse click on the **Arc** enters the **Graphical Edit** mode with the **3D instrumentation** to allow quick edits.

Within this mode the **Arc** is highlighted yellow with an arrow indicating its **Direction** and text showing its current **Radius**.

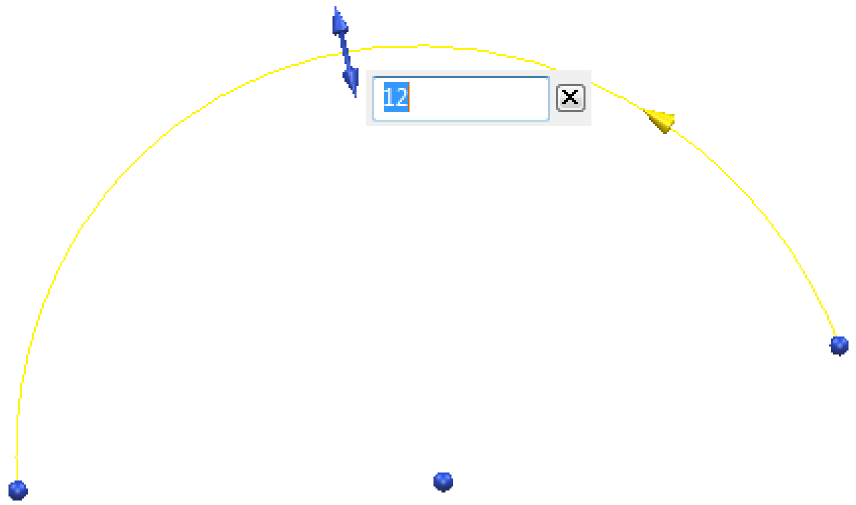
The blue **Sphere** at either end of the **span** allows you to **edit** the characteristics quickly and easily while the **intelligent cursor** allows you to **snap** the **span** to other objects.



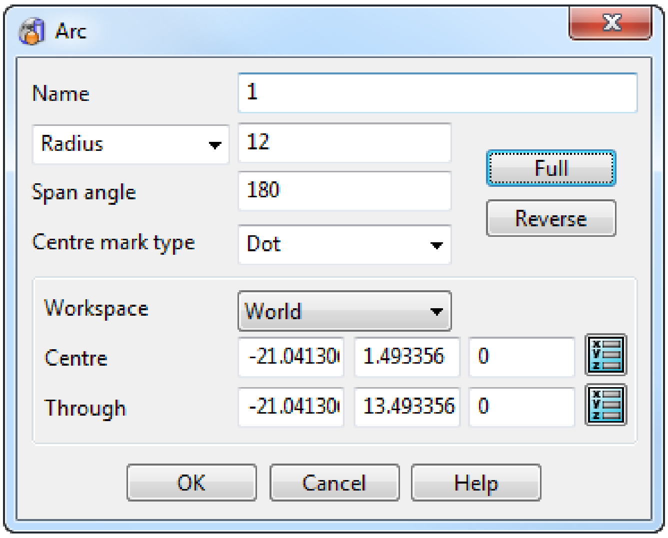
Clicking and **dragging** the **spheres** will allow either end of the **arc** to move in a **fixed circular motion** dictated by the **radius** in order to **graphically edit** the **span angle** of the **arc**.

The bidirectional blue arrows at the **centre** of the span allow the user to **edit** the **radius** of the line from the **fixed arc centre**.

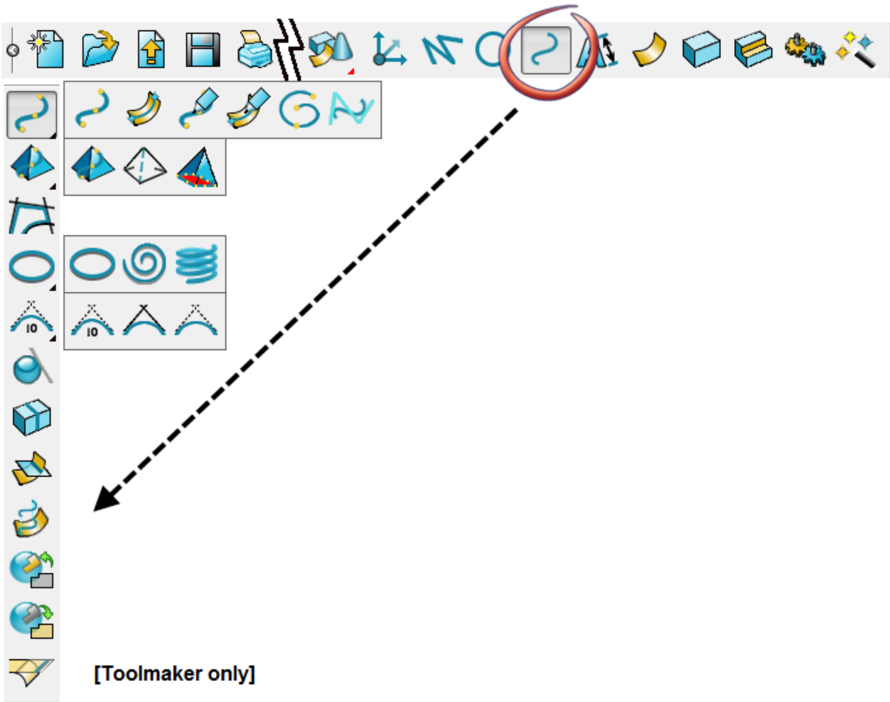
When using the **arrows** or **spheres edit** the **arcs characteristics**, when the mouse button is **released**, **PowerSHAPE** will give the opportunity to **fine tune** these values such as **span angle** and **radius**.



To fully fine tune the **Arc**, the most accurate option will be to enter the **Arc Editor** dialog. Again, as with all **Objects** in **PowerSHAPE** a double left mouse click on the **Object**, in this case an **Arc** enters the **Editor** dialog.



Curve toolbar



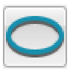
The following **Curve creation** options are available in <u>standard</u> PowerSHAPE.

 Bezier curve

 Sketch a Bezier curve

 G2-curve

 B-spline curve

 Ellipse

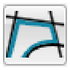
 Spiral

 Helix

 Fillet

 Untrimmed blend

 Trimmed Blend

 Composite Curve

 Draft curve

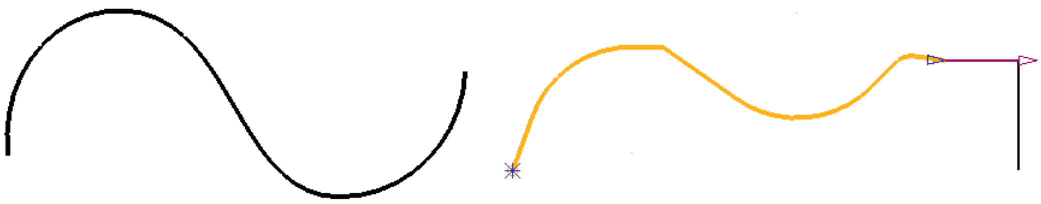
 Curve by oblique sections

 Surface or solid intersection

 Curve projection

 Curve wrapping

 Curve unwrapping



After creating geometry consisting of lines and arcs, it is often necessary (and good practice) to combine these into single entities, as wireframe curves. You will often require wireframe that is defined directly as complex curves.

The most commonly used curve definitions are:

Bezier curve - free form curve

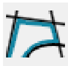
Composite curve – curve defined along existing wireframe and/or model edges.

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.

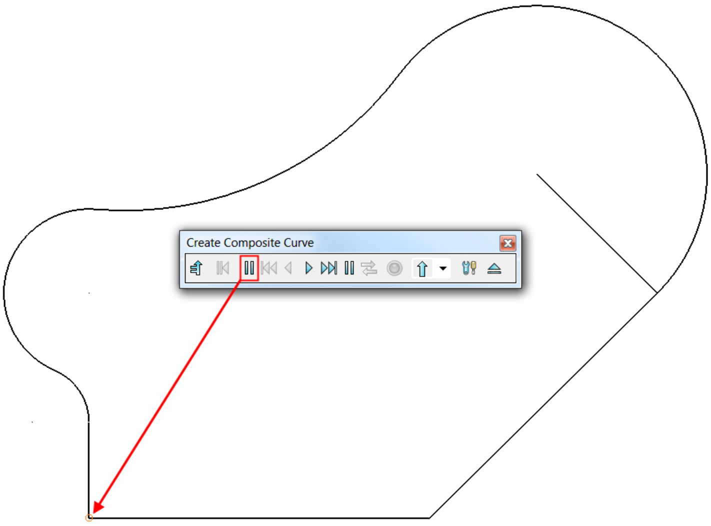
A composite curve can only be created along existing model entities.

The basic wireframe shape is complete, but is 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).

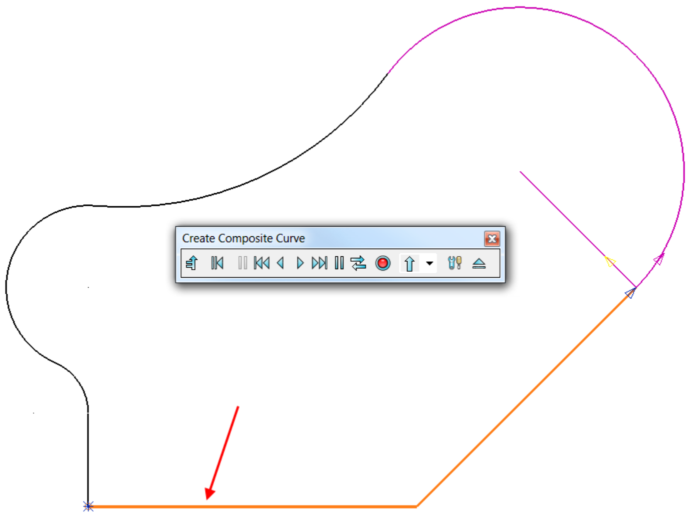
1 From the **Curve** toolbar,  select **Composite curve**. 

The **Create Composite Curve** toolbar appears.

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



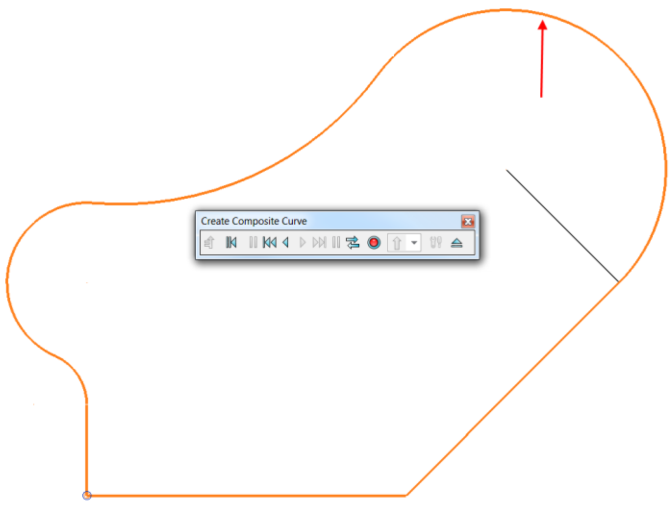
3 Left-click on the bottom horizontal line.



The composite curve is generated along the route until it hits a branch (intersection) point where optional directions will be arrowed for you to manually select the required route.

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

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

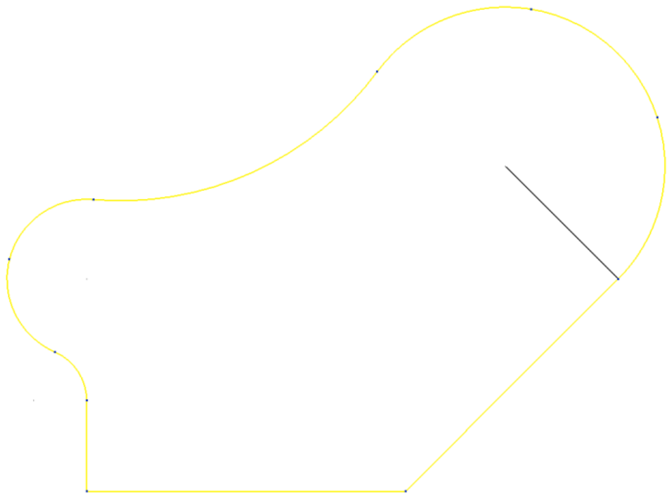


The composite curve has continued around the wireframe back to the defined start point.

A circle at the start point indicates the composite curve is a closed form.

5 On the toolbar, select **Save** to accept the composite curve.

The single composite curve is shown yellow in colour when highlighted.



6 Select **Eject**  to close the toolbar.

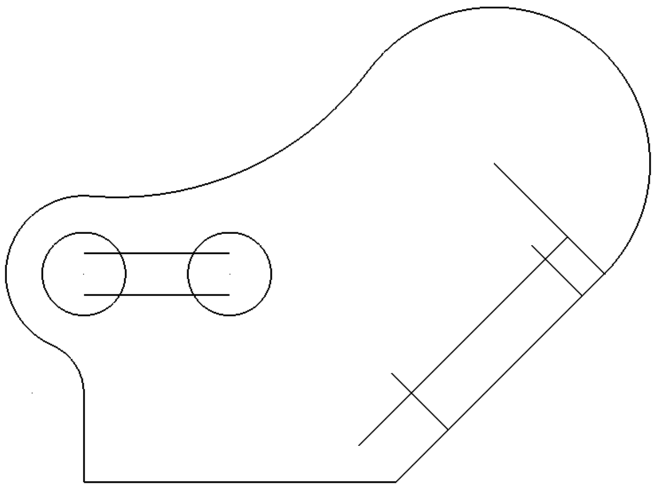
7 Select **Undo**  to revert the last command (composite curve creation).

Exercise 1: Composite Curves

1 From the main menu select **File > Import** to import existing geometry into the current PowerSHAPE model.

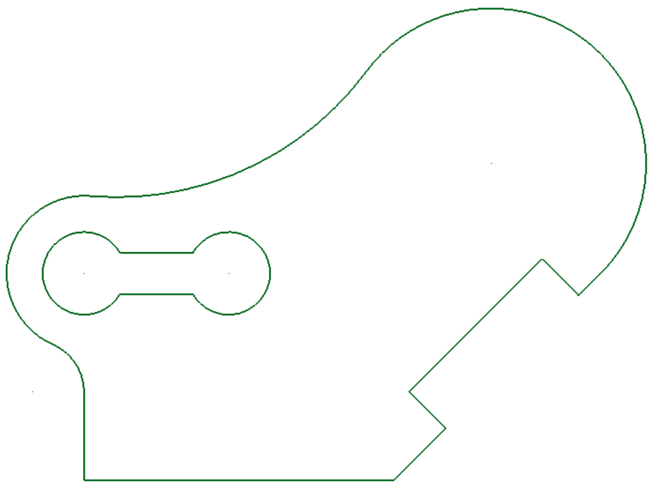
2 Import the model:

.....\PowerSHAPE_Data**composite curve Ex1.dgk**



3 Use Composite curve to generate the result shown below.

Use blanking to hide items not required.



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

Composite curves can be converted back to wireframe by selecting Edit>Convert>To Wireframe from the main pull down menu.

4 Select **File > Save As**.

5 In the form save the model as:

......\PowerSHAPE-Models\Ex 1 composite curves.psmodel

6 Select **File > Close**.

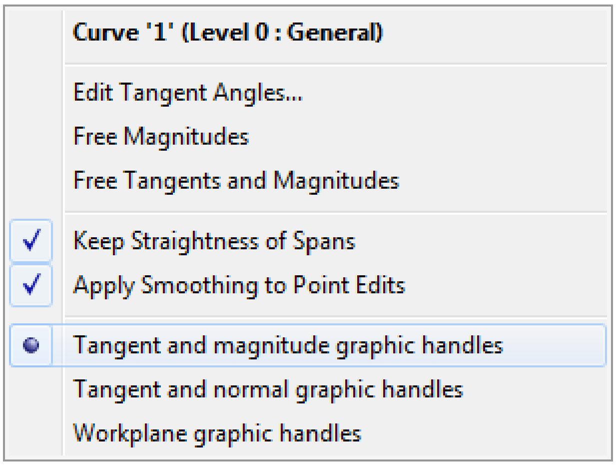
Single Curve Point Edits

Single points on curves can be edited directly in the graphics window using a variety of different graphical edit handles, as well as using the Command Box on the Status Bar. Use the handles on curve points to graphically modify single curve points. The handles will display only when a single point is selected.

It is possible to change they different curve point handles within **Edit>Surface and Curve Edits>Common Edits> Selected Point Graphical Handles** or by right-clicking with a single point selected.

The following types of handles are currently available to use:

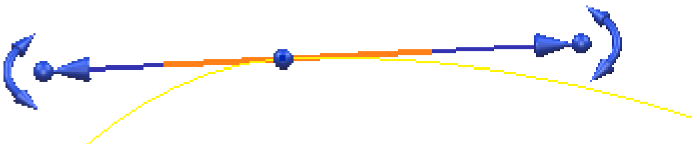
- **Tangent and Magnitude graphic handles**.
- **Tangent and Normal graphic handles**.
- **Workplane graphic handles**.



Tangent and Magnitude graphic handles

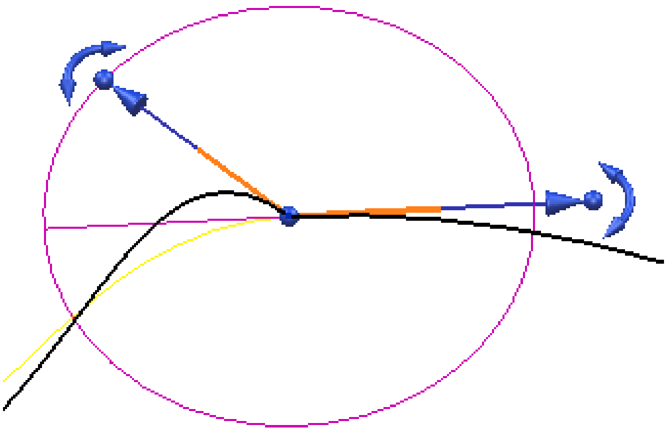
1 From the **Curve Menu**  select **Bezier Curve**  and create a freeform curve within the **Graphics Window**.

2 Select a **point** on the curve and right-click and ensure that **Tangent and Magnitude graphic handles** is selected in the menu.

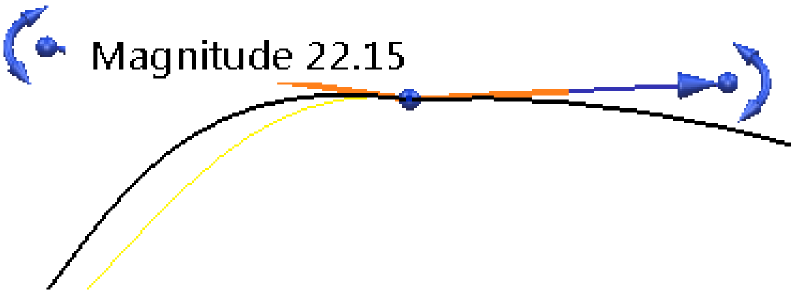


These handles allow the user to graphically edit the **Tangency** and **Magnitude** of the curve point.

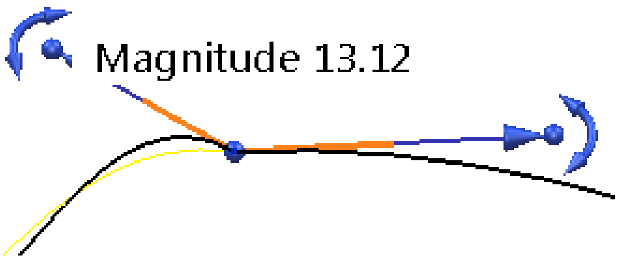
The **arced handles** offset from the end will modify the **tangent angle** of the **curve** in a **fixed circular motion**, without affecting the magnitude as can be seen below.



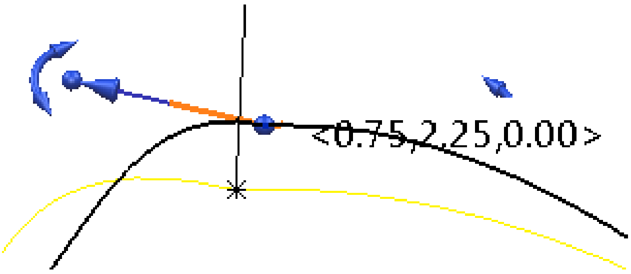
The **arrows** at the end of each handle will **graphically modify** the **magnitude** of the curve along a **fixed tangent angle**, in this case to help the user, PowerSHAPE will give interactive feedback of the magnitude.



The **Spherical** handles will **modify** both the **Tangent** and **Magnitude** freely in **2D** or **3D** space. Again, **PowerSHAPE** will give interactive feedback of the magnitude to the user.

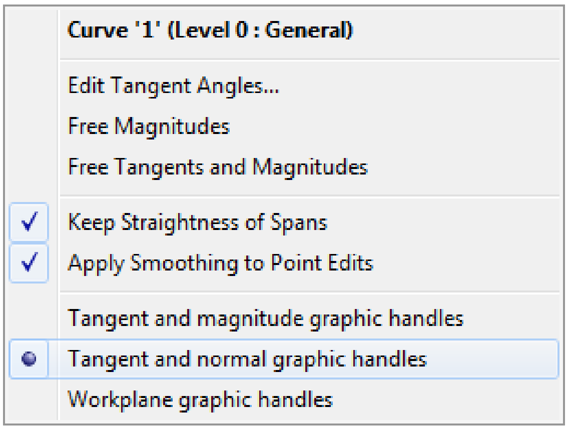


Finally, the **spherical handle** at the selected curve point position at the **centre** will **edit** the **position** of the **point**, without affecting the **direction** or **magnitude** of the point. For increased usability the intelligent cursor will remember the **original position**, and give coordinates (**X, Y, Z**) of the movement.

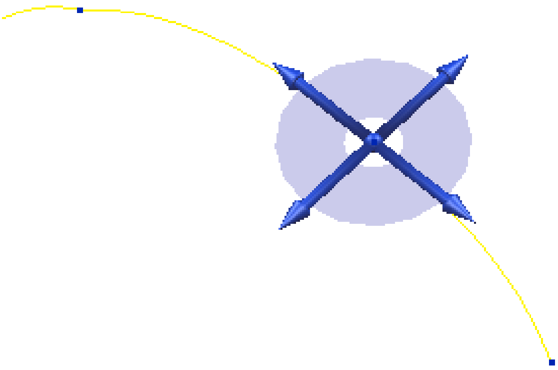


Tangent and Normal graphic handles

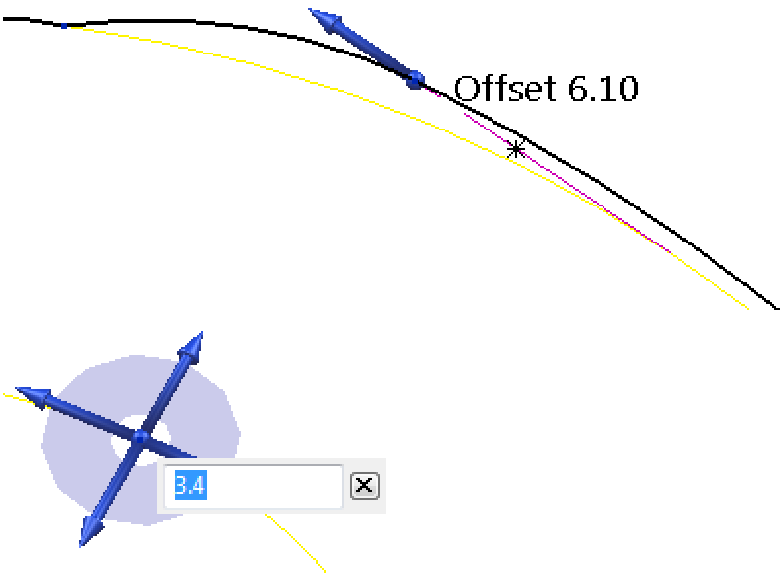
1 Select a **point** on the curve and right-click and ensure that **Tangent and Normal graphic handles** is selected in the menu.



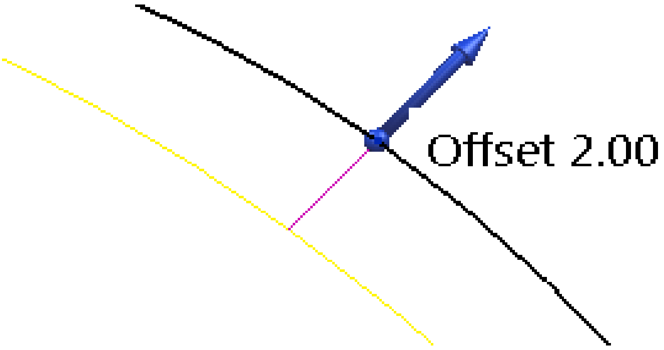
These handles allow the user to graphically edit the position of the selected point in fixed **Normal** or **Tangent** directions, and alternatively **freely in a 2D plane**.



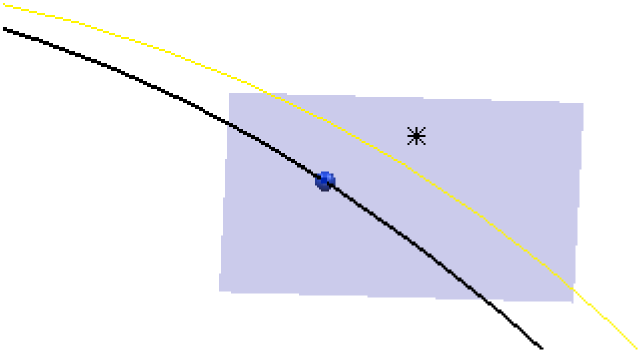
The **arrow** handles **tangent** to the curve will modify the **position** of the **point** in a **fixed direction** along the **tangent direction**. For accuracy the offset distance from the **orginal location** will be visible, as well as **editable** once the mouse button has been released.



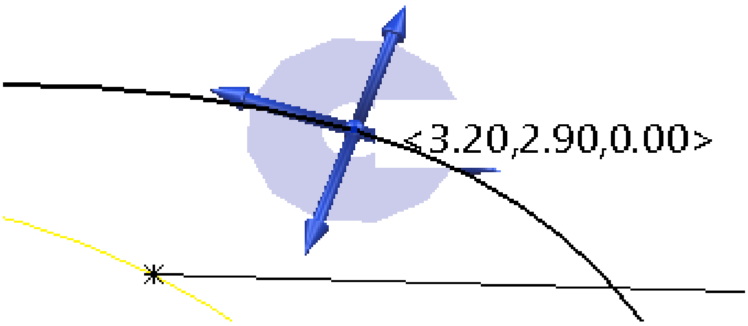
Similarly, the **arrow** handles **normal** to the curve will modify the **position** of the **point** in a **fixed direction** along the **normal direction**. For accuracy the offset distance from the **orginal location** will be visible, as well as **editable** once the mouse button has been released.



The **4 Planar Handles** surrounding the point will allow you to move the **point** freely in a **2D plane** while again the original location is kept visible throughout**.**

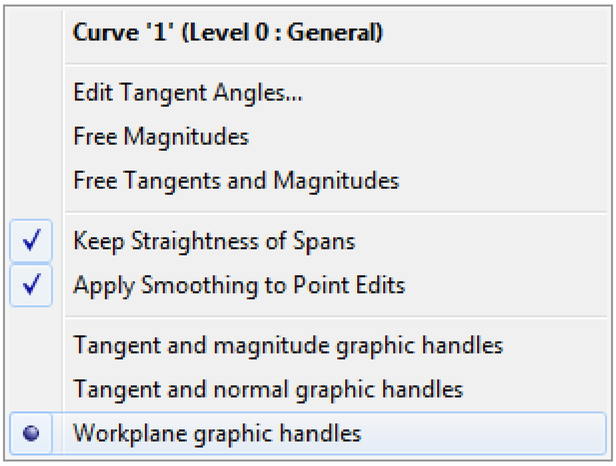


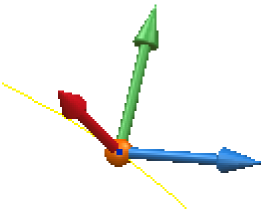
Again, the central **spherical handle** at the centre will move the point **freely** in **3D space** while showing **X**, **Y**, **Z** coordinates offset from its original location.



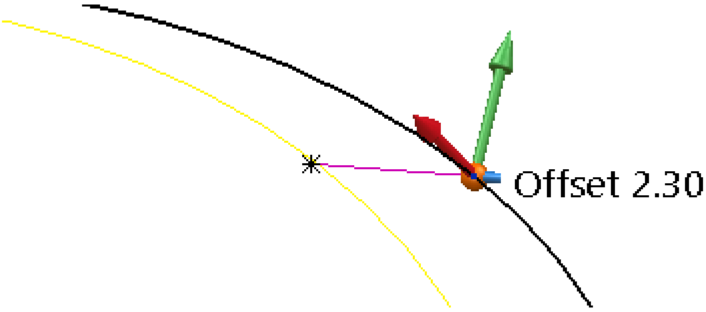
Workplane graphic handles

1 Select a **point** on the curve and right-click and ensure that **Tangent and Normal graphic handles** is selected in the menu.



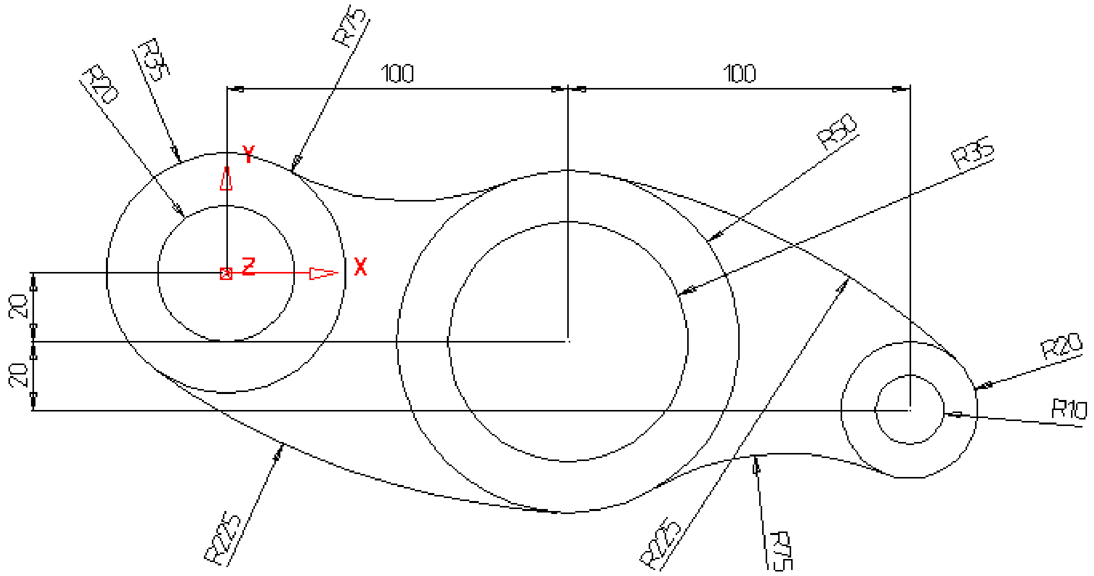


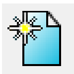
Clicking and **dragging** the **workplane axis** arrow handles will move the point in a **fixed axial** (**X**, **Y** or **Z**) direction with a **visible offset distance** from the original position, with the option to **further edit** or fine tune the offset once the mouse button has been released. The **workplane origin** in this case allows the point to be moved freely in **3D space**.

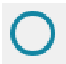


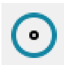
Wireframe Arcs Example

This example demonstrates the use of arcs to create the lever design shown below.



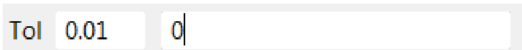
1 Start a **New Model** 

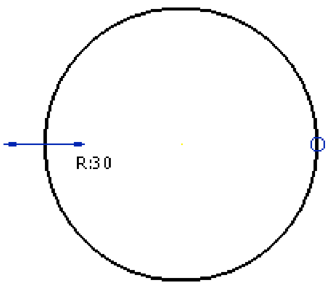
2 Select the **Arc** toolbar. 

3 Select **create a full arc.** 

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

4 In the **Command Input** box, type **0** for the circle centre and then press **Enter**.





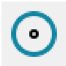
A preview of a circle is generated with a **radius** of the default value. You can change the radius by dragging one of the displayed arrows to the required value.

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

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

Alternatively open the arc editing form and modify the radius value.

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**.

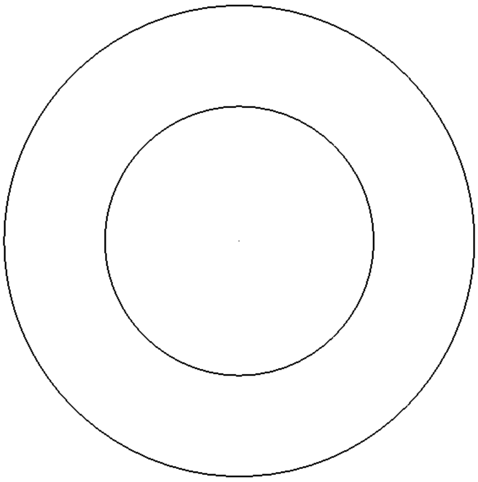
6 Select **create a full arc.** 

7 In the **command input** box enter **r 20** then **Enter**.

8 Position the cursor over the centre of the first circle so the word **Centre** appears.

*By default, the new circle will be exactly the same initial radius as the first circle. To change the default radius, type the new radius into the **Command Input** window.*

9 **Left-click** to accept the centre of the new circle.



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

Creating additional circles

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

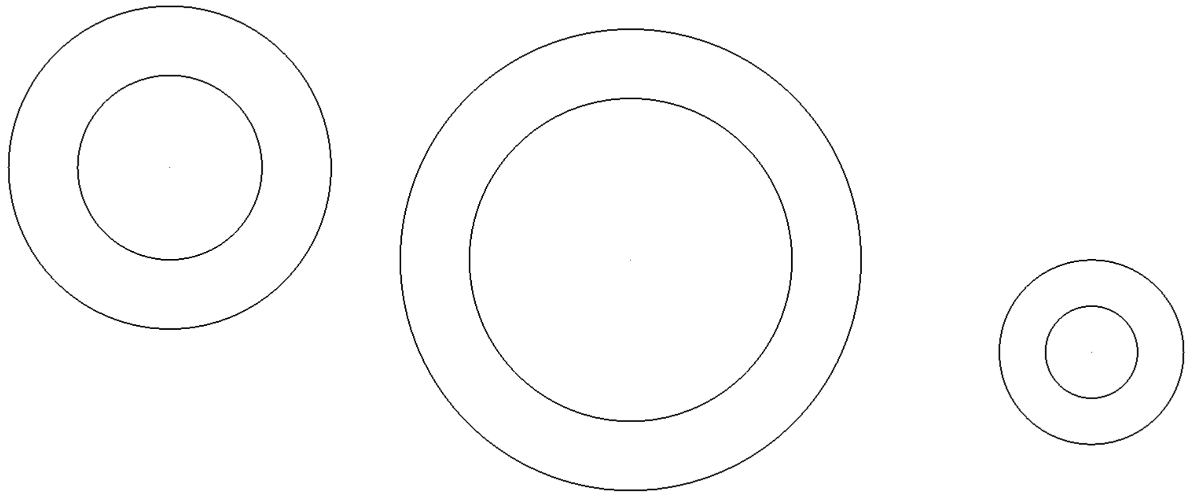
10 Create a circle at **centre 100 –20** with a **radius** of **50**.

11 Create a circle at **centre 100 –20** with a **radius** to **35**.

12 Create a circle at **centre 200 -40** with a **radius** of **20**.

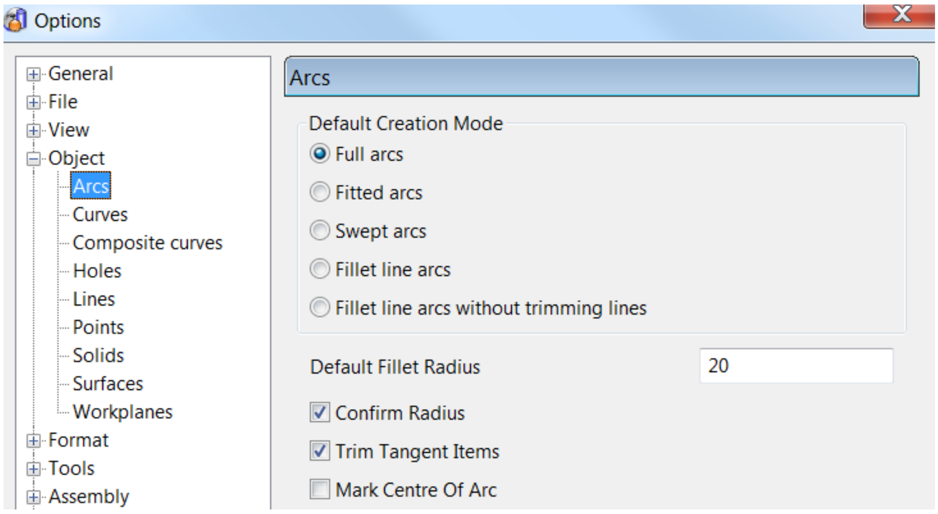
13 Create a circle at **centre 200 -40** with a **radius** of **10**.

14 Press **Esc**.

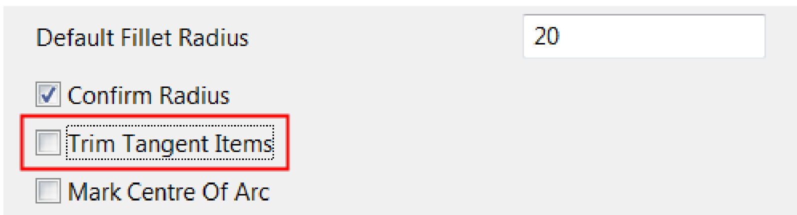


The default option when creating tangent arcs is to trim back the associated geometry. In this case, it is not required, so the option (**Trim Tangent Items**) needs to be deselected.

15 Select **Tools > Options**, expand **Object** then select **Arcs**.



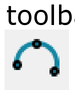
16 Deselect **Trim Tangent items**.



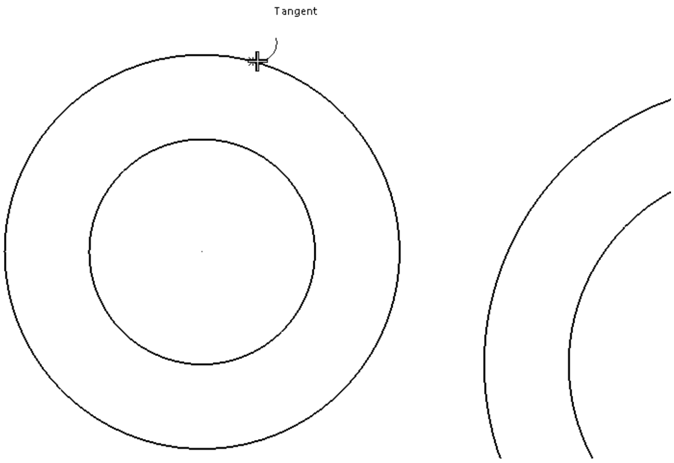
17 Select **OK**.

Creating the tangent arcs

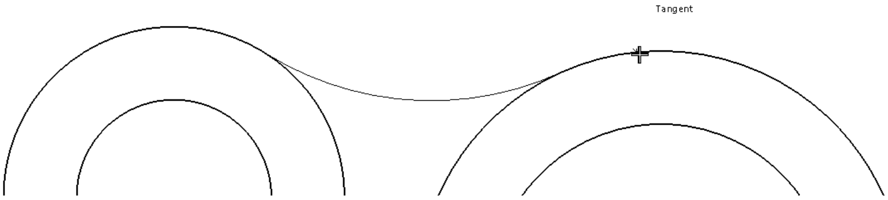
For the tangent arcs, the fitted arc option is the most suitable as it provides full dynamic control through all possible combinations.

1 From the **Arcs** toolbar, select **Create an arc through three points/items.** 

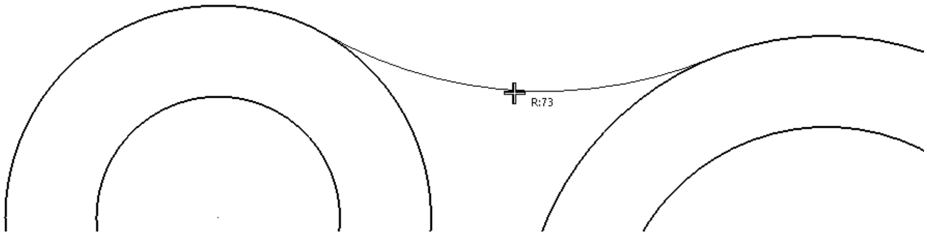
2 Move the cursor over the outer left circle until the word **Tangent** appears and left-click.



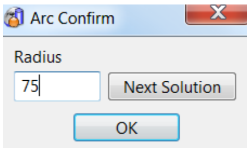
3 Move the cursor over the middle outer circle so the word *Tangent* appears and left-click.



4 Move the cursor over up and down to get the tangent arc in the correct place and left-click.

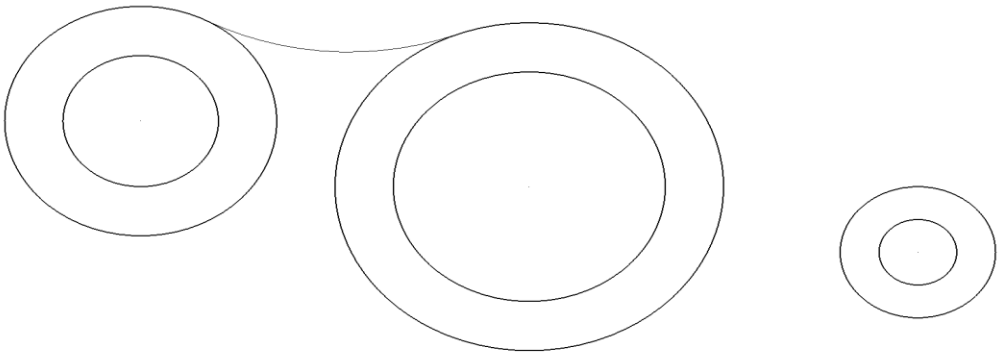


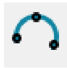
The **Arc Confirm** dialog is displayed. This can be used to enter an exact radius.

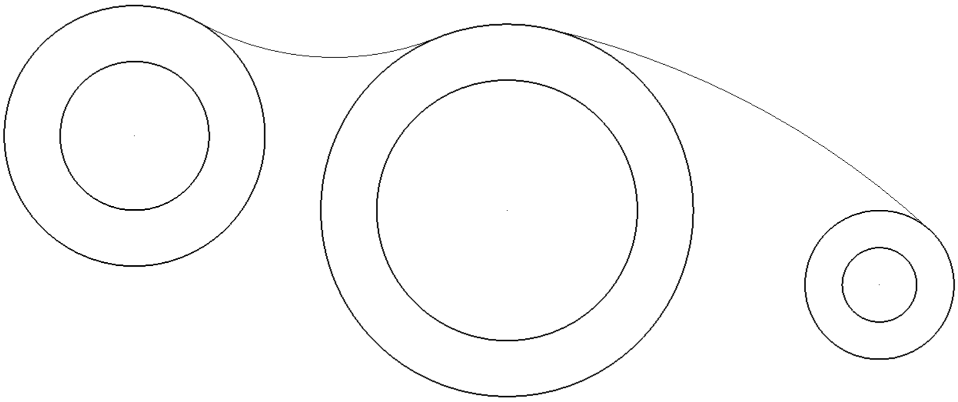


5 Enter a **Radius** of **75** and select **OK**.

The tangent arc is drawn.

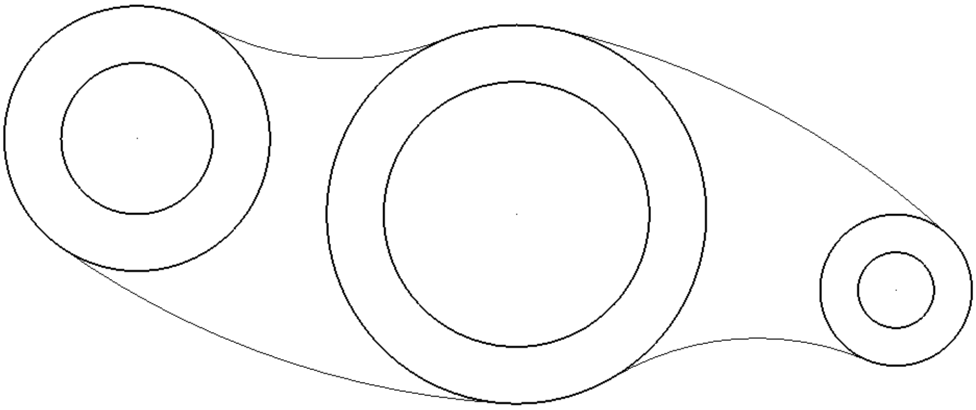


6 Create an **arc through three points/items**  of radius 225 between the middle and outer circles shown below.

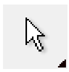


7 Create an arc between the left outer circle and the middle outer circle with a **radius** of **225**.

8 Create a three point arc between the right outer circle and the middle outer circle with a **radius** of **75**.



The model is complete

9 Select **Esc** or click **Select**  to exit creation mode.

10 Select **File > Save As**.

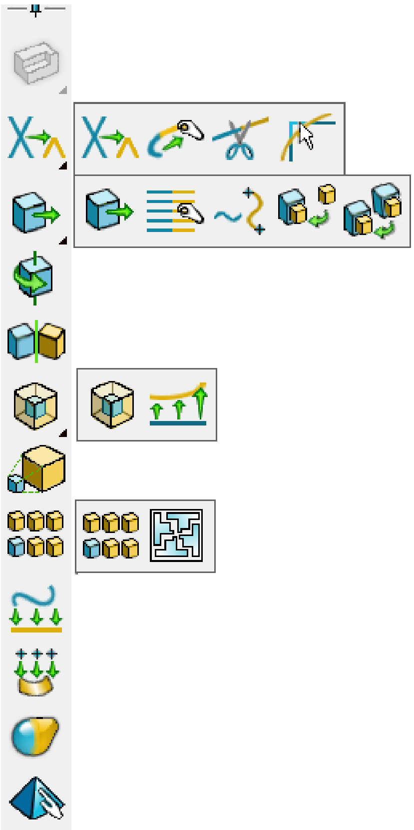
11 In the form save the model as:

.....\PowerSHAPE_Models\lever example

12 Select **File > Close**.

General Edits example

At the top of the left toolbar are two fixed buttons; the second button  includes access to the **General Edits** toolbar. The buttons on this toolbar allow for standard editing functions that apply to all object types.

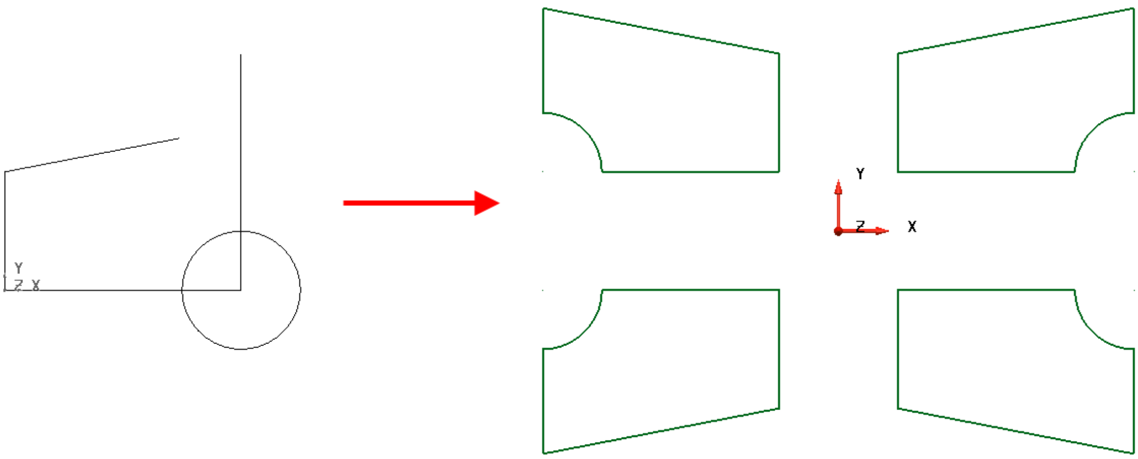


A number of key editing options will be demonstrated in the following example.

1 Start a **new model**.

2 Select **File > Import** and load in the wireframe model:

.....\PowerSHAPE_Data\GeneralEdit_Wireframe.dgk

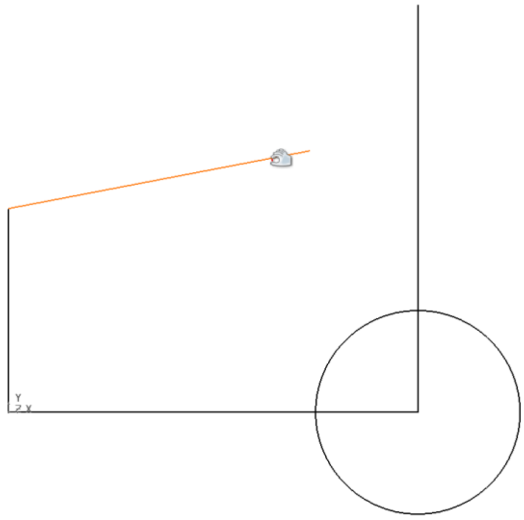


The imported wireframe (shown above, left) will be modified using the **General Edits** options, to produce the 4 finished wireframes (shown above, right).

3 Click in the graphics area to deselect the wireframe.

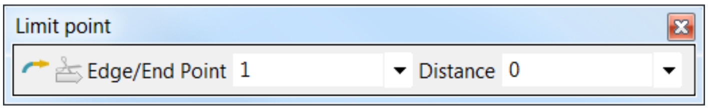
4 Select  to display the **General Edits** toolbar.

5 Select the line that needs extending as shown



6 Select the **Point limit** option. 

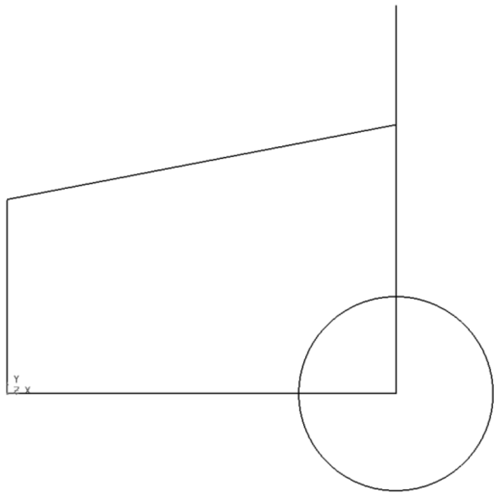
The **Limit point** toolbar is displayed, floating in the graphics window.



The cursor also changes to a magnet symbol. 

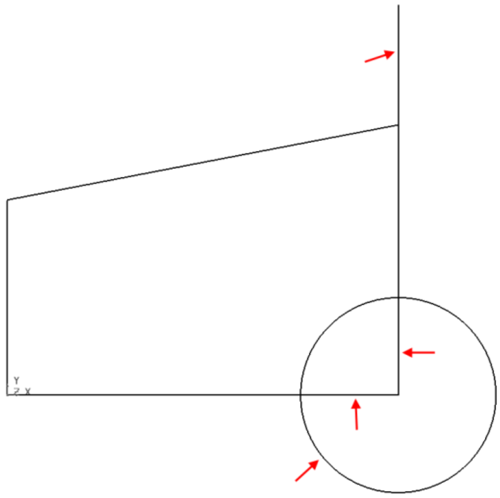
7 Move the cursor to the **End** of the line to be extended.

8 Hold down the left mouse button and drag the end of the line tangentially (along the preview line) until the word **Intersection** is displayed, then release the left mouse button.



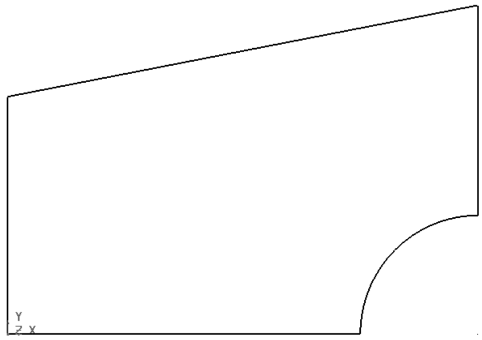
9 Select the **Interactively limit wireframes** option  from the **General Edits** toolbar. The **Limit point** toolbar will be closed automatically.

10 Left-click the mid-span of any sections of the wireframe network that are to be trimmed away (as illustrated below).



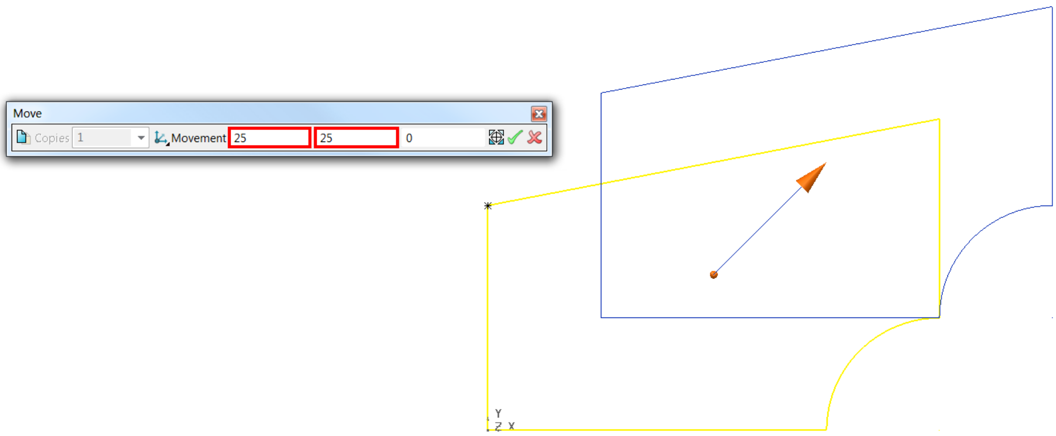
11 Create a composite curve from the network of lines and arc wireframes.

A quick way to create a composite curve is to hold down the ***Alt*** *key and left-click anywhere along the wireframe network.*

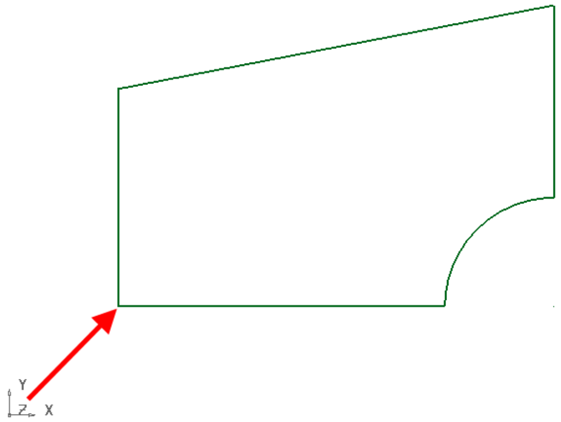


12 With the composite curve selected, select **Move**  from the **General Edits** toolbar.

13 With the toolbar displayed, enter the coordinate values **X 25 Y 25** in the position boxes as shown. A preview of the transformed item is displayed and if required, can be changed.



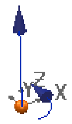
14 Select the green tick  to accept the preview.



15 The composite curve is shifted by **25** mm along both **X** and **Y** relative to the World datum

Workplanes and datums will be discussed in the next chapter.

16 With the composite curve still selected, select **Rotate** from the **General edits** toolbar.

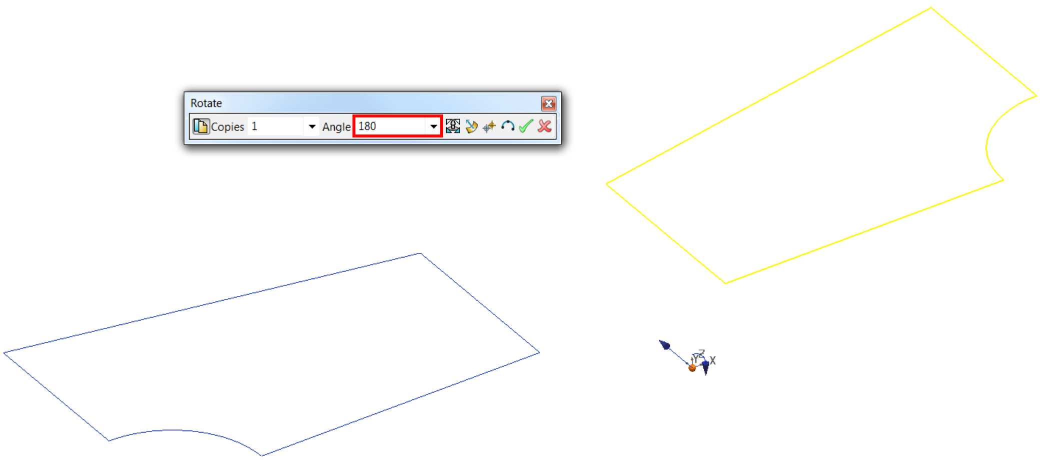


The Rotate toolbar is displayed. In addition, the blue arrow on the datum shows the current rotational axis (Z). In this example the wireframe will require rotation about the Y axis, so this must be changed.

17 Select the **Y** button to make **Y** the operational axis. The axis buttons are located at the bottom-left of graphics area.

The rotational axis in the graphics area reflects this change.

18 In the **Rotate** toolbar, select copy the selected item (to keep original copy), Leave Copies as 1, and enter Angle of 180. Press Enter to display a preview.



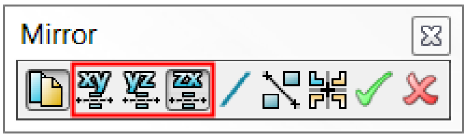
19 Select the green tick to accept the preview.

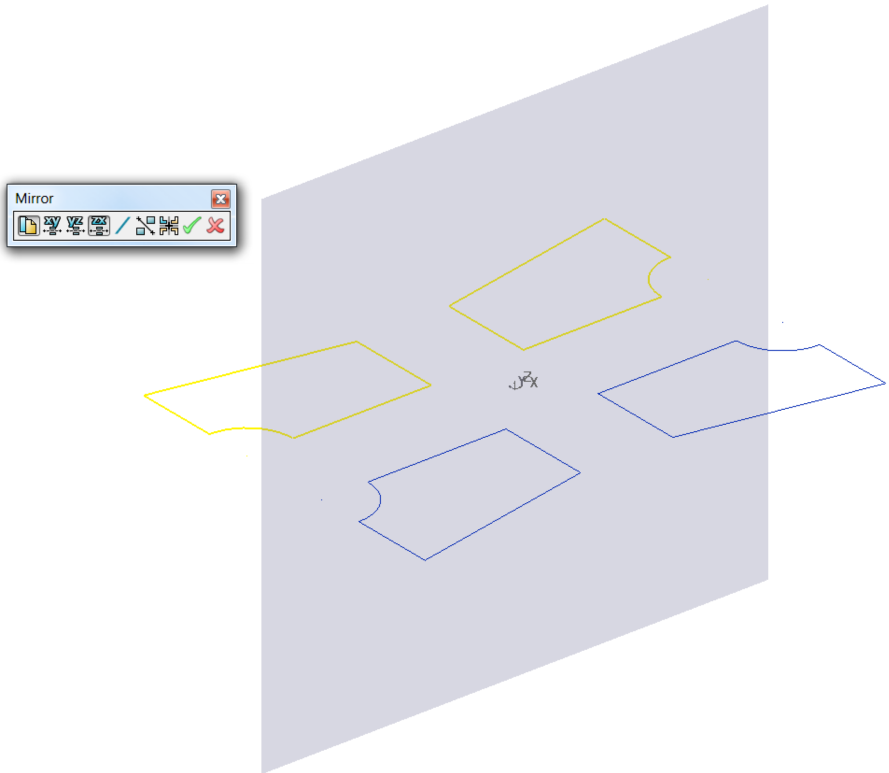
A copy of the composite curve is rotated by **180** degrees about the active **Y** axis.

*In this case, the same result would have been obtained if the curve had been **mirrored** about the **YZ** plane.*

20 With both curves selected, select **Mirror**  from the **General edits** toolbar.

The preview shows the desired result. The selected wireframes are mirrored across the ZX plane (i.e. Y as operational axis).





21 Select the green tick  to accept the preview.

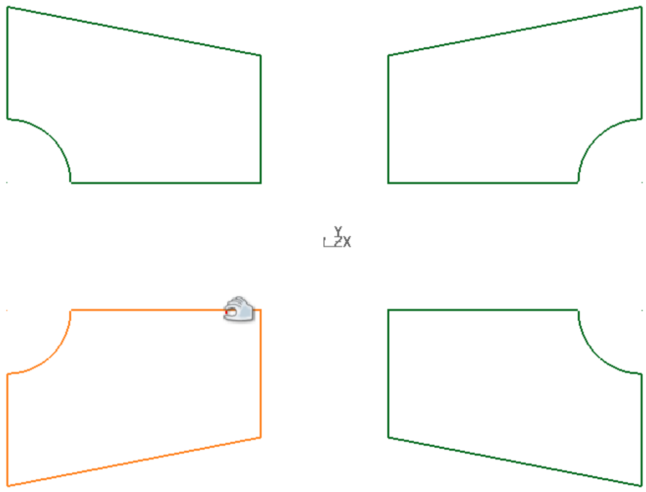
22 Close the **Mirror** toolbar. 

Further General Edits options

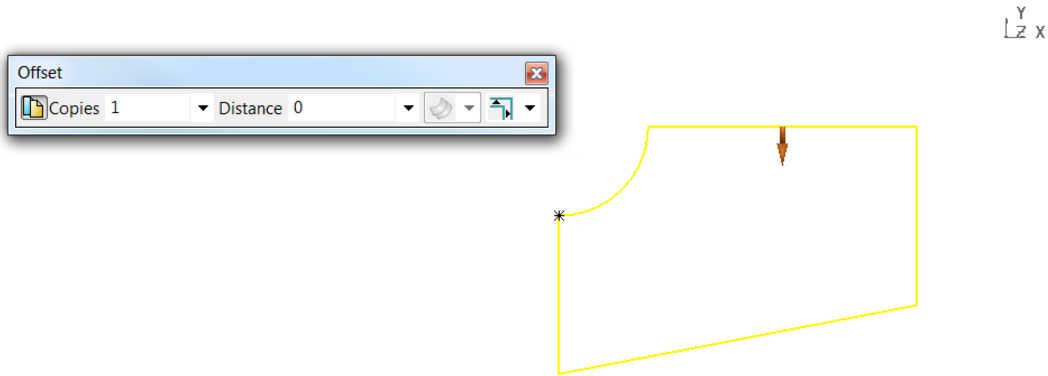
In addition to the General Edits options used, other options are used later in the course with other PowerSHAPE entities, such as surfaces and solids.

Offset

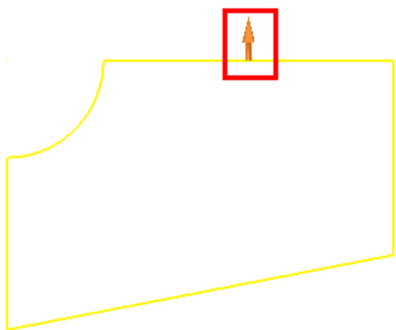
1 Select the composite curve located in the **-X -Y** quadrant.



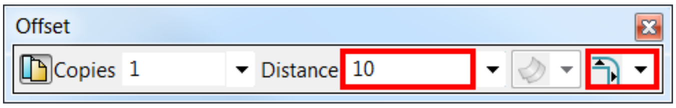
2 Select **Offset**  from the **General edits** toolbar.



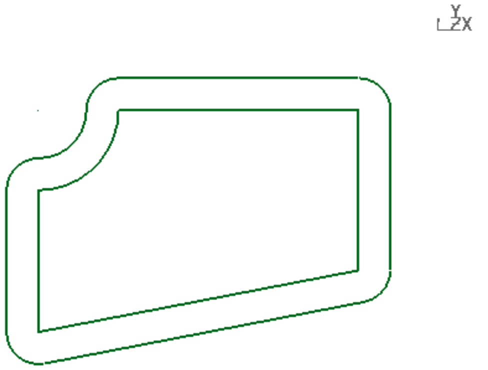
3 Select the small red arrow on the curve to switch the positive direction outwards.



4 Select **Round discontinuities**  and enter a **Distance** of **10.**



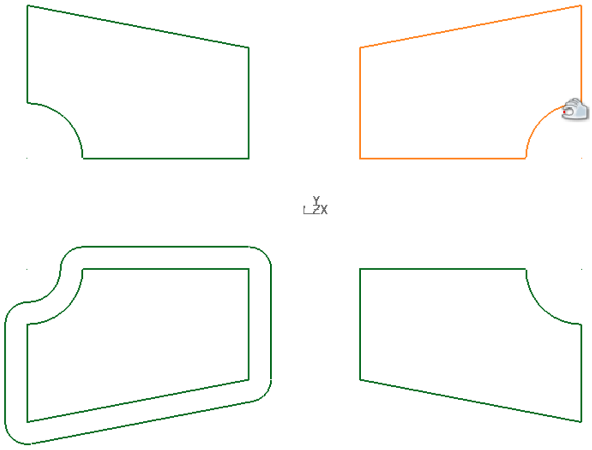
5 Close the **offset** form by selecting the small cross in the corner. 



A 10mm offset with round corners has been applied.

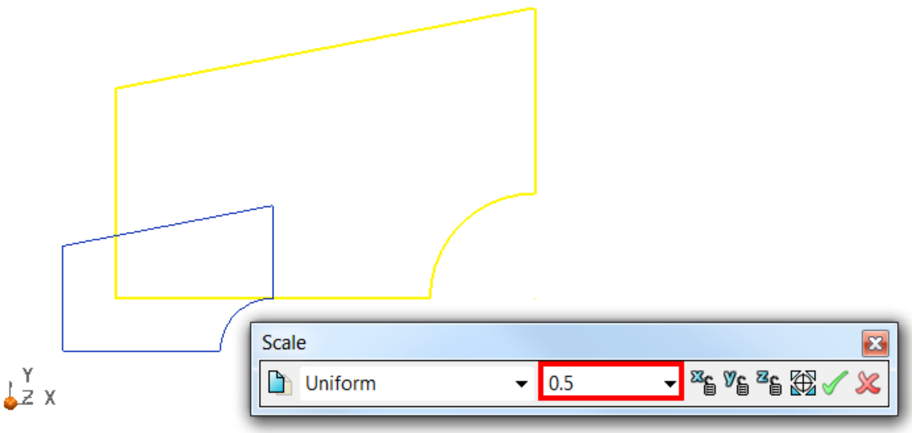
Scale

1 Select the composite curve located in the **X Y** quadrant.



 *If not already, ensure ts to/from he operational axis is set to Z.*

2 Select **Scale**  from the **General edits** toolbar



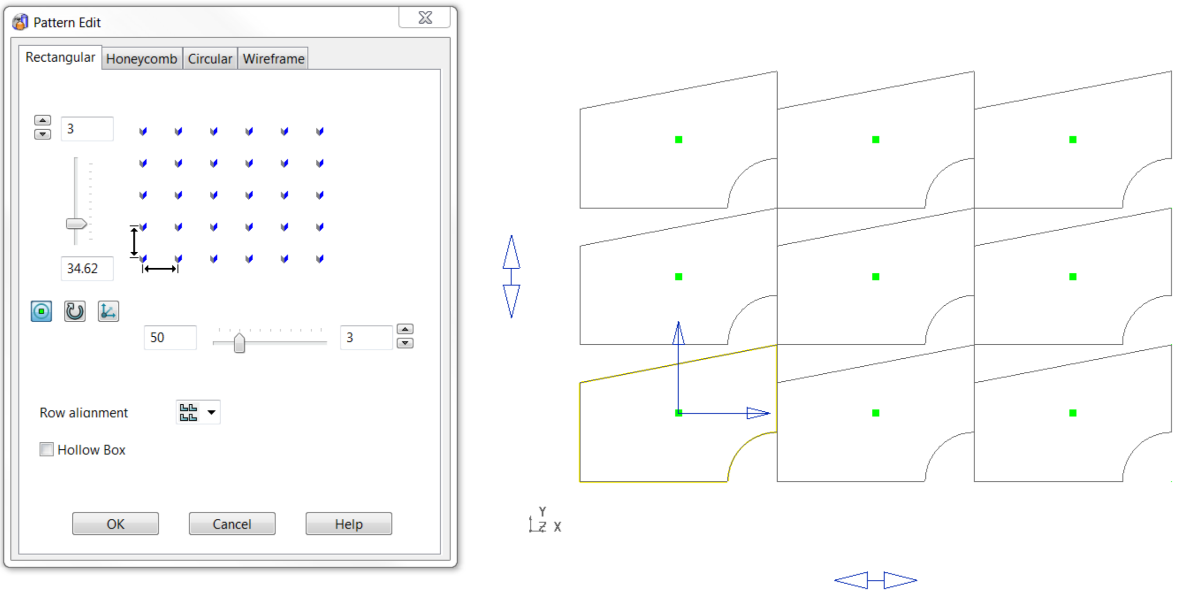
3 Enter a **Scale** factor of **0.5** to reduce the size of the composite curve to 50% of the original size.

4 Select the green tick to accept the preview.

Scaling is from active workplane, therefore vectors to the active workplane origin also scale. This can be repositioned as required within the scale dialog

Create Pattern of objects

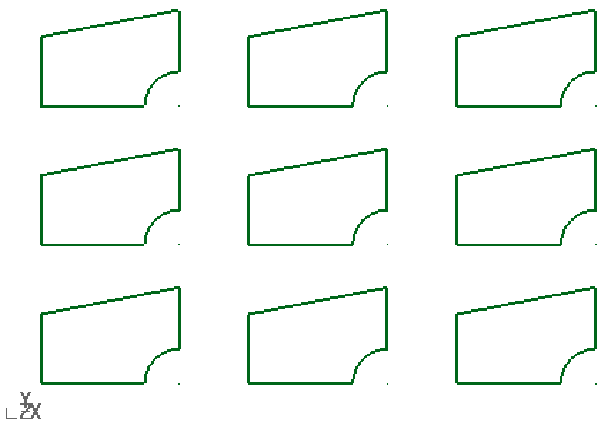
1 With the scaled down composite curve selected, select **Create Pattern of objects.**



The current settings are immediately previewed.

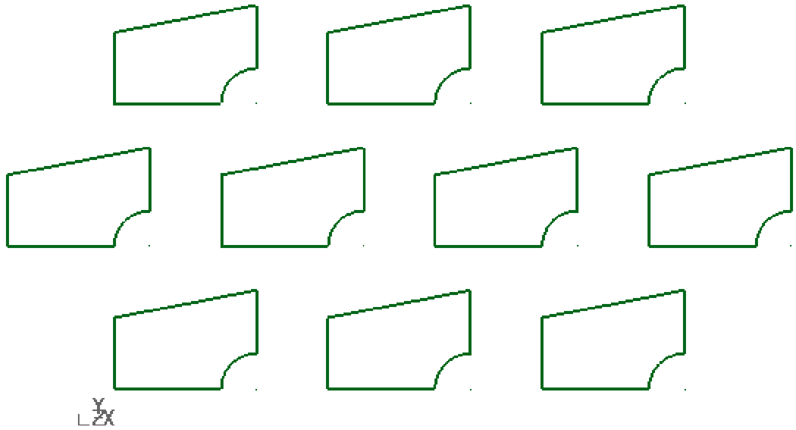
2 Enter **3** rows along both **Y** and **X** with a **Y pitch** of **50** and **X pitch** of **75**.

3 Select **OK**.

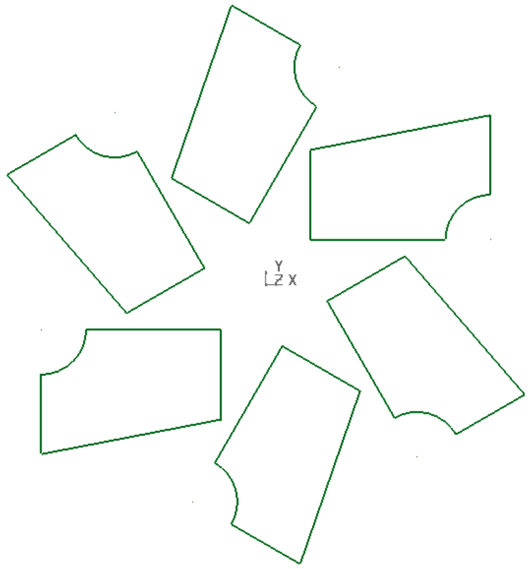


Rectangular is one of 4 main functions in the Pattern Edits dialog. The other three include Honeycomb**,** Circular, and along Wireframe.

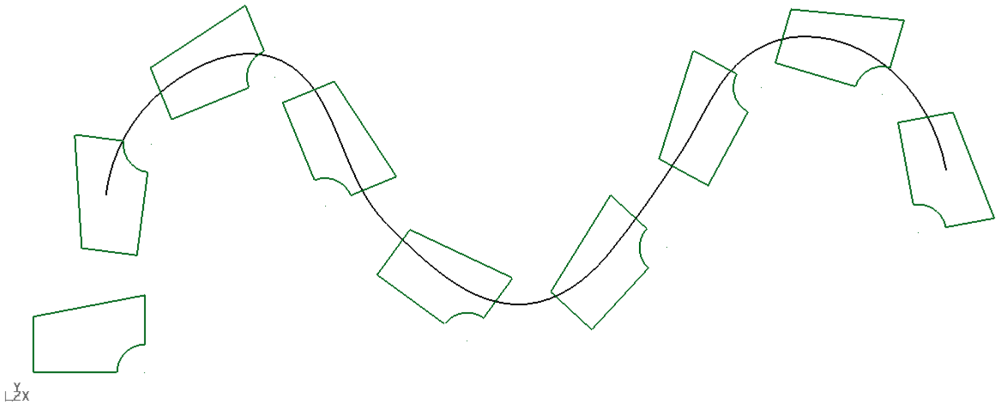
Honeycomb



Circular



Wireframe



Further layout options are available within each ***Pattern Edit*** *tab.*

4 Select **File > Save As**.

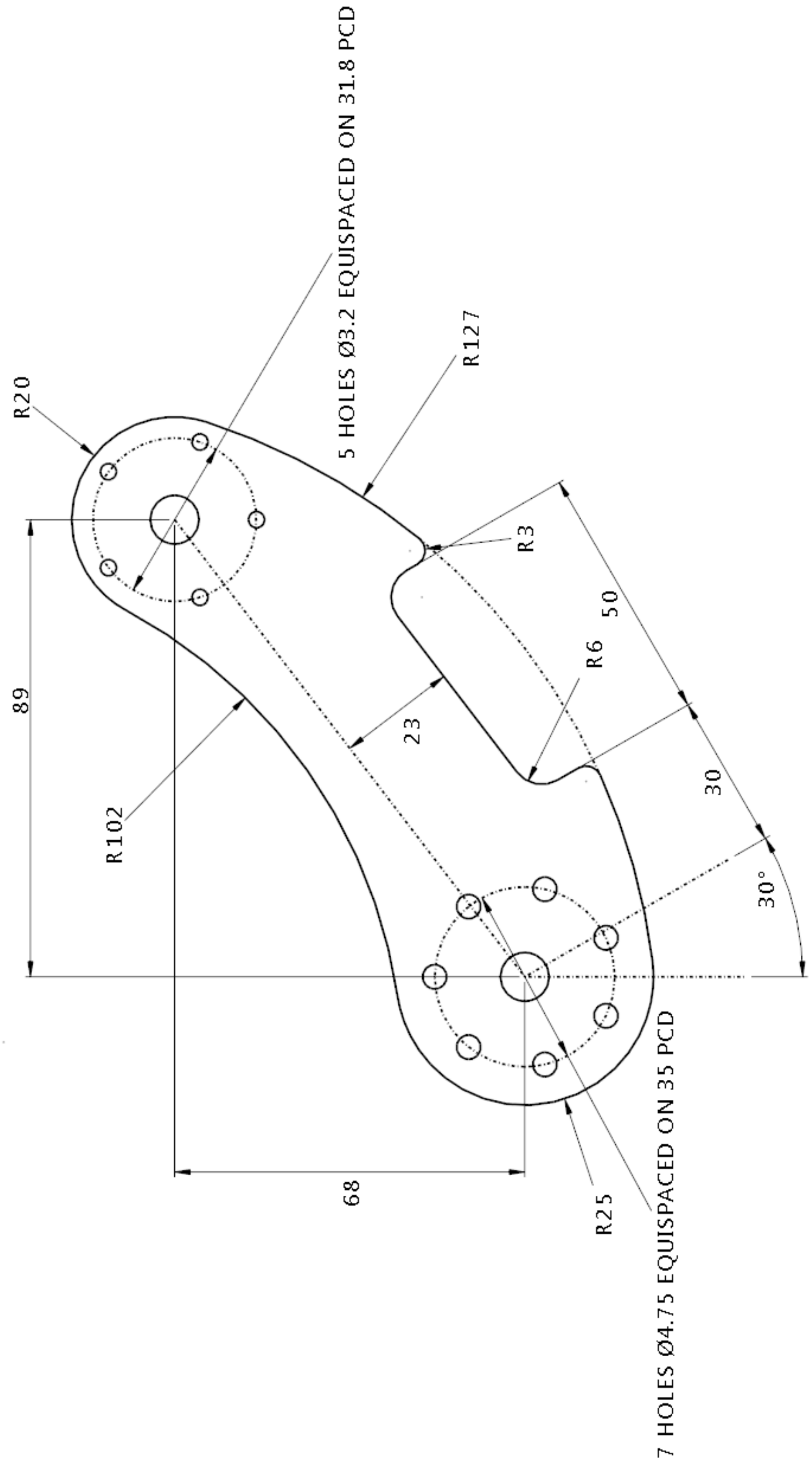
5 In the form save the model as:

.....\PowerSHAPE_Models\General Edits

6 Select **File > Close**.

Exercise 2: 2D Drawing

1 Use the dimensions provided to generate a Wireframe drawing.



2 Upon completion save the model as …/PowerSHAPE_Models/**Exercise 2 2D Drawing.psmodel**

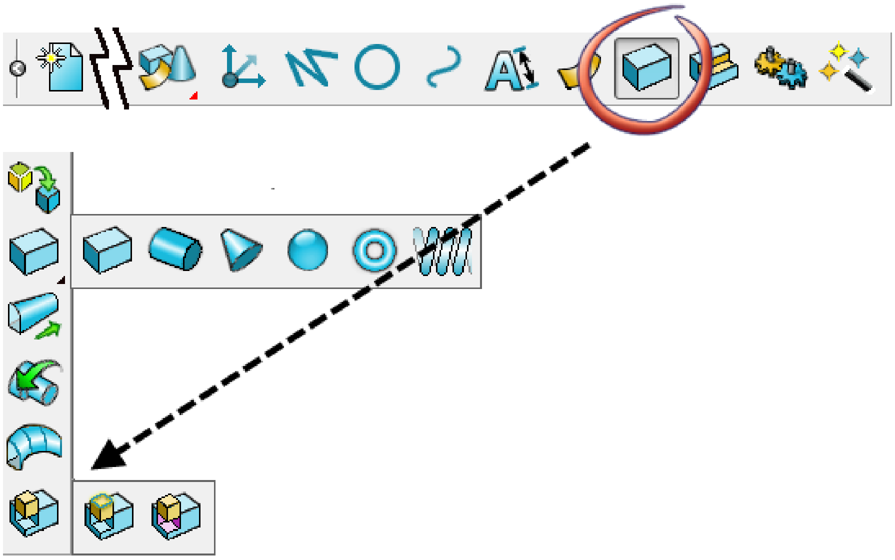
6. Solid Modelling

Introduction

Solid modelling applies a different approach to the creation of a **CAD** model compared to surface modelling studied later in the course. The main difference is that a surface model is formed by a hollow, zero thickness skin, whereas a **solid model** is a **mass of material**. The main **advantages** of using **solid modelling** are the model **creation speed, parametric qualities** and the **history tree**. The user can use the **history tree** to **reposition** or **edit operations performed** earlier, resulting in the **automatic update** of other **affected items**. **PowerSHAPE** is unique in the ability to **convert solids to surfaces** and **surfaces to solids**, depending on which approach is most suitable. Open surfaces can be converted to a solid, an extremely useful capability which is not available with dedicated solid-modelling packages.

Within the **Main** toolbar there are two buttons for creating and Editing Solids.

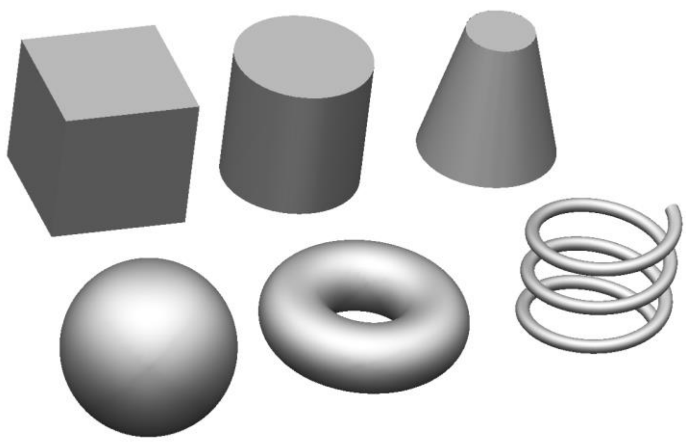
When **Solid options**  is selected, it displays the following buttons in the toolbar to the left of the graphics window.



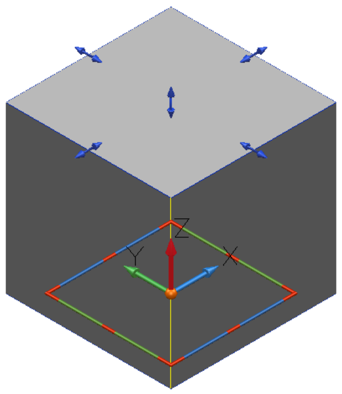
- Creating a solid from surfaces
- Create solid block (primitive)
- Create solid cylinder (primitive)
- Create solid cone (primitive)
- Create solid sphere (primitive)
- Create solid torus (primitive)
- Create a solid spring (primitive)
- Create solid extrusion
- Create a solid of revolution
- Create solid from drive curve and sections
- Create solid core
- Create a solid core from selection

Solid Primitives

In addition to the **primitive solids** which are generated from wireframe items (e.g. Extrude, Rotate), the toolbar contains six pre-defined shapes.

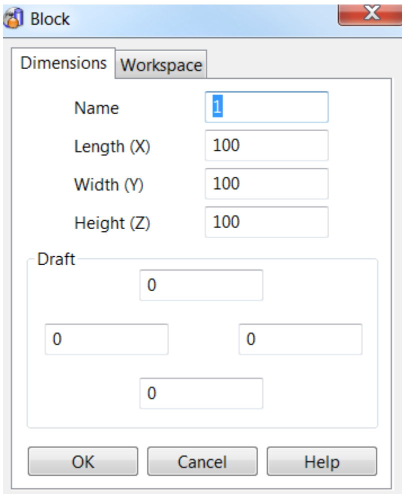


Primitive shapes can be edited dynamically with a single mouse click (select)

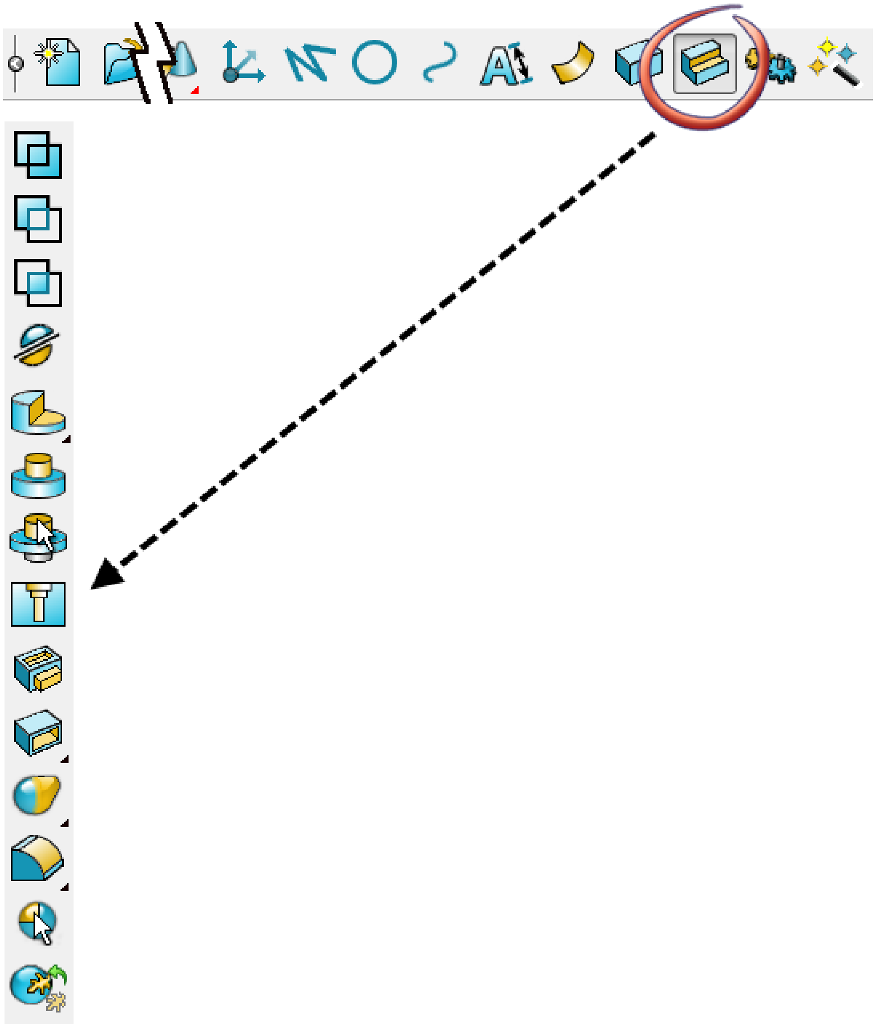


The **workplane 'type' handles** can be used to **move**, **change orientation** and **twist** about its **axis**. The **blue handles** can be **dragged** to **edit the dimensions** of the **primitive solid**.

Alternatively, **double clicking** on the **primitive solid** will display the **dialog** in which edits can be applied.



When **Solid Features**  is selected, it displays the following buttons in the toolbar to the left of the graphics window.



Solid from addition

Solid from subtraction

Solid from intersection

Solid split

Solid cut

Radial Cut

Solid boss

Solid hole

Pocket or protrusion

Hollow solid

Thicken solid

Bulge

Morph feature

User defined feature

Rib fillet

Solid fillet

Solid chamfer

Solid wrap feature

Active and inactive solids

In older versions of **PowerSHAPE** (Before **2015**) many **solid operations** were performed on the **currently active solid**.

- The active solid is usually the main component of your model
- Only one solid can be active at any time
- By default, the first solid you create automatically becomes the active solid and other solids are created as inactive
- If you activate an inactive solid, the currently active solid (if any) becomes inactive

The **active solid** is represented by the **Red Flag** in the **solid history tree** and by **darker wireframe edges** in the **graphics area**.

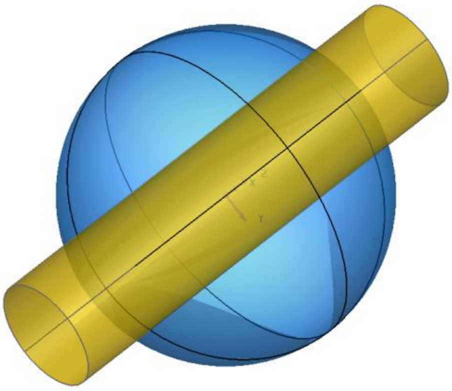
Since **PowerSHAPE 2015** the **selected solid** or the **active solid** can be used to perform **solid feature operations**. However it is still good practice to keep control over which **solid** is used as the **active**.

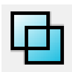
Boolean Operations

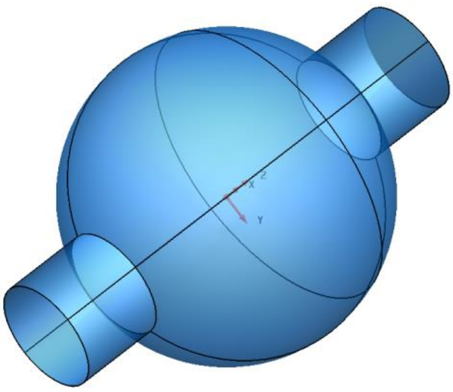
Boolean operations define the relationship between entities. As a **solid** is built up, **additional solids** and **surfaces** can be **absorbed** into it. This behaviour is illustrated below using the **three Boolean operations** available from the **solid features** toolbar.

Observe the two primitive shapes below. With transparent shading applied, the yellow cylinder can be seen fully intersecting the blue sphere.

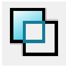
In this case the Sphere is the **primary solid** and the **cylinder** is **selected** only.

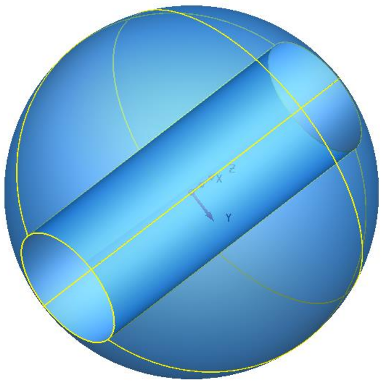


Selecting **Add**  will **combine** the **selected solids**.

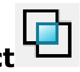


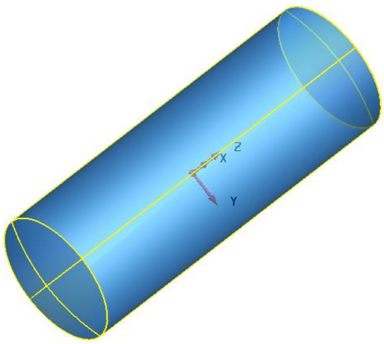
The **two separate solids** have merged into **one solid object**. The cylinder is joined onto, and becomes part of the solid sphere.

Selecting **Remove**  will **subtract** the **secondary solid** from the **primary solid**. If the **active solid** is **visible** in the **graphics area** that will **automatically selected** as the **primary**, although this can then be **overridden** in the dialog.



A **hole** appears through the **solid sphere** caused by the **removal** of the **solid cylinder**. The hole is now part of the **active solid sphere**.

Selecting **Intesect**  keeps the **intersection** of the **selected solids**.



The **new solid** becomes the **common volume** of **both solids**, producing a solid tube with spherical ends.

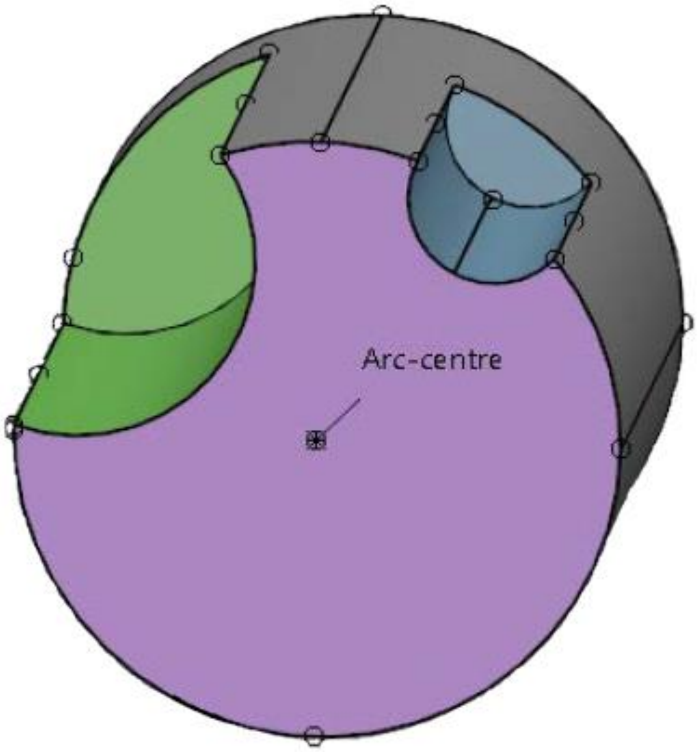
Snapping to Parasolids

Following a recent update released in **PowerSHAPE 2014 R2**, solids now support much improved snapping options using the intelligent cursor to parasolids within the graphics window.

The intelligent cursor now snaps to the following, and are only visible once in a relevant creation mode:

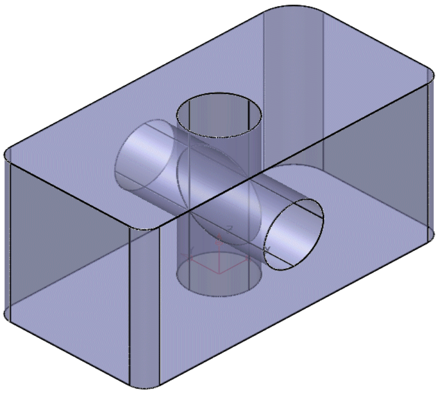
- **Corners**
- **Centres of Planar Faces**
- **Centres of Arc Edges**
- **Midpoints of Straight Line Edges**
- **Tangent to Arc Edges**

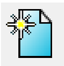
Examples of these can be seen on the image on the next page.



Solid Modelling Example 1

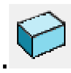
The following example illustrates the **creation** of a **simple solid model**. It emphasises the **benefits**; the **ease of creation**, and the **ability** to **retrospectively** make changes to **existing features** causing the **remainder** of the **solid** to be **automatically updated**.



1 Create a **new model**. 

2 Create a **workplane** at **0** and set the **Principal Axis** to **Z**.

3 Select **Solid**  from the **main toolbar**.

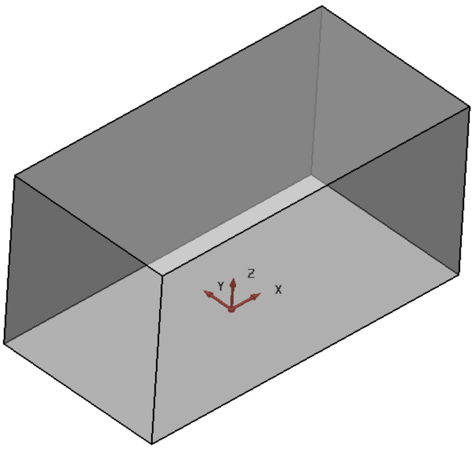
4 From the **Solids** menu, select **Create Solid Block**. 

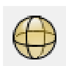
5 **Centre** the **base** of the **solid block** about the **workplane** by typing **0** in the **command box** (or **snapping** to **workplane origin**) then pressing **Enter**.

6 Press the **Esc** key to **break out** of the **command**.

7 **Double-click** the **solid block** to open the **Block** dialog.

8 Enter **Length 100**, **Width 50**, and **Height 50**.



 ***Screen shots have transparent shading applied.*** 

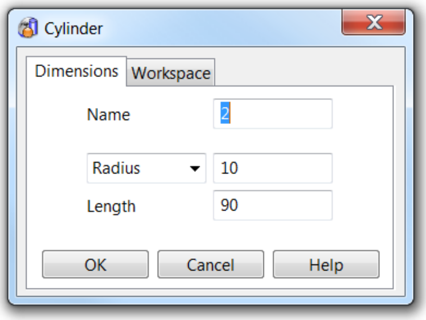
9 Select **OK** to apply the **changes** and **close** the **dialog**.

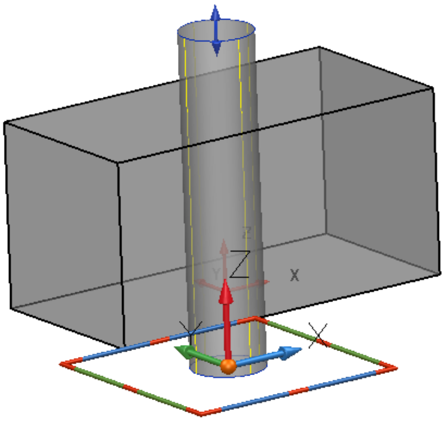
10 From the **Solids** menu select **Create Solid Cylinder**. 

11 In the **Command Input box** at the **bottom-right** of the **graphics area**, type the values **0 0 –20** to position the **base** of the **solid cylinder**.

12 Press the **Esc** key to **break out** of the **command**.

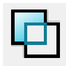
13 **Double-click** the **solid cylinder** in the **graphics area** to open the **Cylinder** dialog and enter **Radius 10mm** and **Length 90mm**.

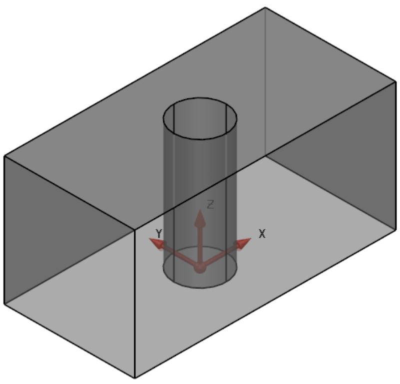




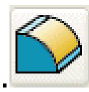
14 Click **OK** to apply and close the dialog.

15 Select **Solid Features**  from the **main** toolbar.

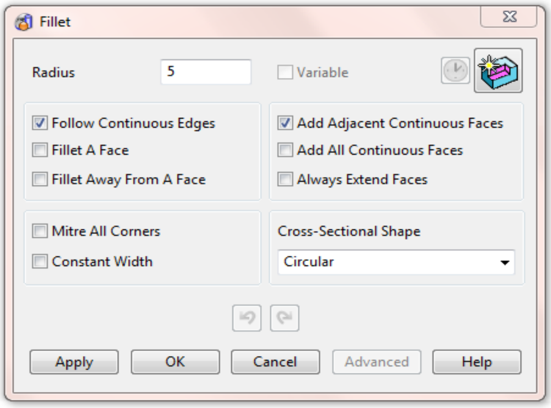
16 Ensure the **cylinder** is **selected** then **Boolean Subraction.** 

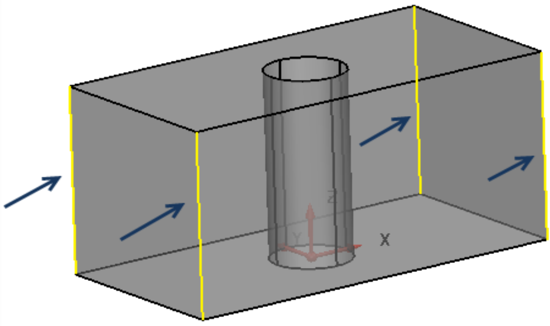


 *The **selected cylinder** is **removed** from the **active block** (created first) to leave the hole. **Fillets** will now be applied to the **vertical edges**.*

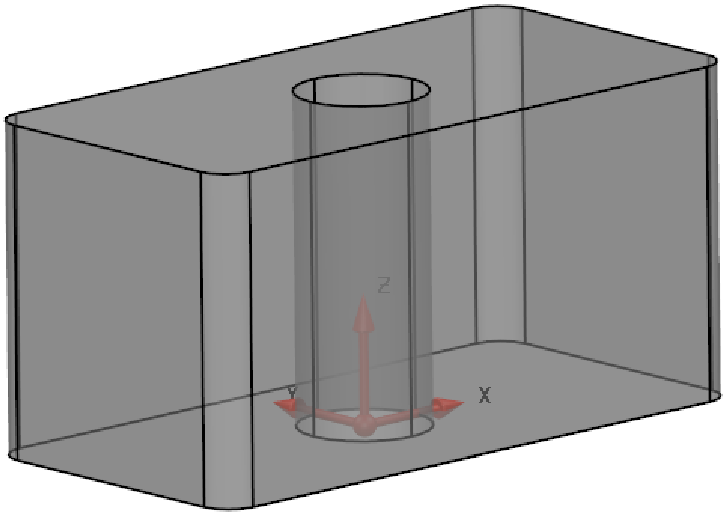
17 From the **Solid Features** toolbar , select **Create solid fillet.** 

18 In the **Fillet** dialog, enter a **Radius** of **5mm**. Hold down the **Ctrl** key and click all the **vertical edges** of the **solid block**.





19 When **all** the **edges** are **selected** (**yellow**), select **OK** to **apply** and **close** the **dialog**.



 *All **four vertical edges** now have a **fillet** of **radius 5mm**.*

20 Select **File > Save As**:

.....***Training Data******Coursework******Solid Modelling Example 1***

Solid Feature Tree

The **Solid Feature Tree** contains the **history** of **solid operations performed** on the **visible solids**.

Each **solid operation** is defined as a **feature** on the **solid**.

When you create a **solid**, it is **automatically added** to the **solid feature tree** (also known as the **solid history tree**).

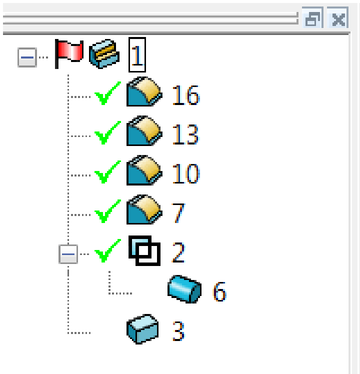
When an **additional feature** is applied a branch representing that feature is added to the tree. The **features** are shown in **reverise chronological order** with the **most recent one** applied at the **top** and the **earliest** at the **bottom**.

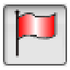
The **solid feature tree** is displayed in a **dedicated window** to the left of the **graphics area**. This can be **viewed** by selecting **Show History Tree** window  from the **Solid Edits Toolbar**, which automatically launches when any **solid** is **selected**.

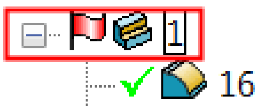
Modifying items and **operations stored** in the **Feature Tree** automatically updates the solid in the **graphics area** (provided the changes are possible). It is also possible to **defer updates** to allow a series of changes to be implemented at the same time.

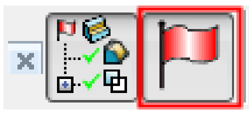
1 Select **Show tree**  window from the **Solid Edits** toolbar.

The current features are displayed.



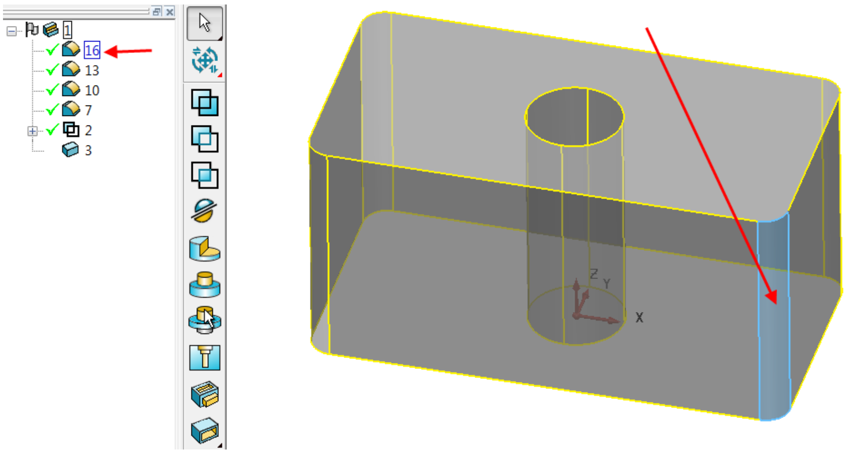
A solid can be **activated** or **deactivated** by simply **toggling** the **flag symbol**. A **Red Flag** indicates an **active solid**  and **grey flag**  a **inactivate** solid. Flags can be toggled from within the **feature tree** or from the **Solid Edits toolbar**.



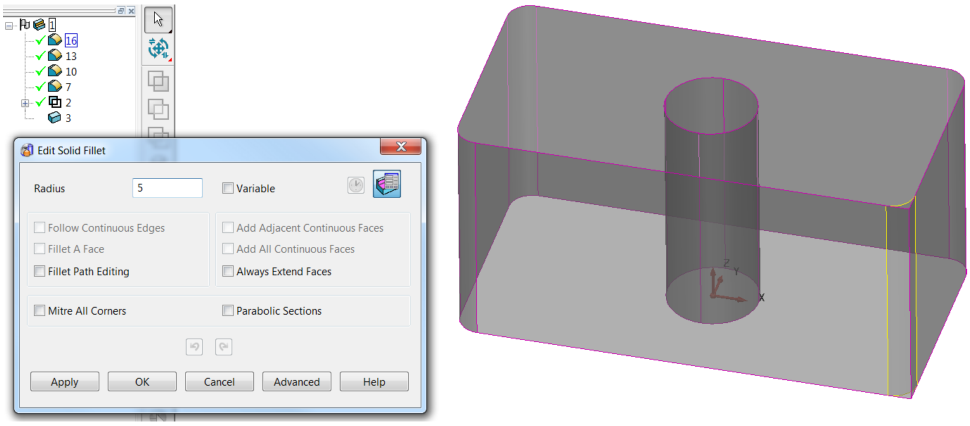


If **sub-items** are **not shown**, click  to display them. To hide the sub-items, click. 

A solid and/or feature can be selected directly on the model or from the feature tree. When a feature is selected, the name is outlined in a blue box and blue on the model itself.

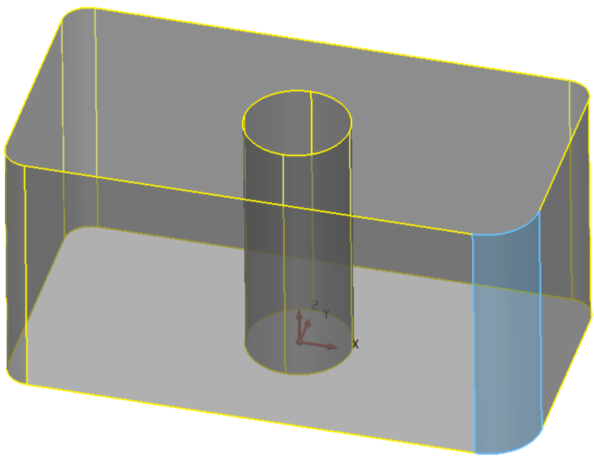


2 **Double-click** on the solid fillet at the top of the tree.



*The **editing form** for the **fillet feature** is displayed.*

3 Input a new **Radius** of **10mm** and select **OK.** The selected fillet radius on the solid changes to **10mm** and the solid **model** is **re-trimmed** to suit.

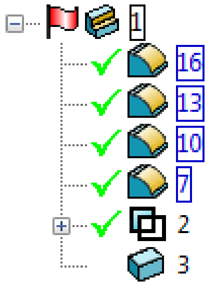


*You can also select **solid sub-items** by **double-clicking** the solid in the **graphics area**.*

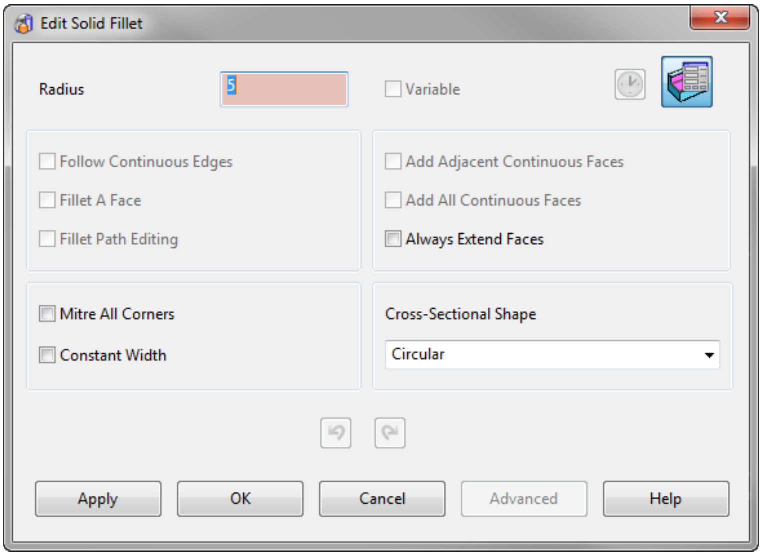
Multiple Feature Edits

In PowerSHAPE 2015 we can now make **simultaneuous edits** to **multiple versions** of different sizes of the **same feature** (ie **Solid Fillets**).

1 In the **Solid History Tree** select all **four fillet features**.



2 **Right-click** on the **selected features** and select **Modify**.



This will raise the normal Feature Edit dialog appropriate to the feature type selected. The main difference when multiple different features are selected are:

- The properties that **differ** between the **selected features** are **highlighted** in **pink**.
- The properties that **cannot** be **edited** for **ALL** the **selected features** are **unavailable**.
- The **range of values** for each of the **different properties** are **displayed** when your **mouse** is **hovered** over the **input box**.

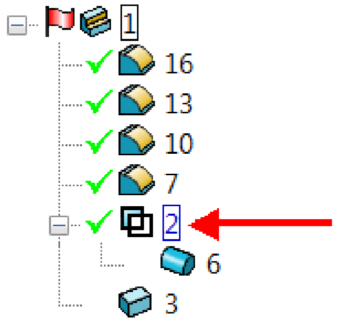
3 In the **Radius box** type **8mm** then click **OK**.

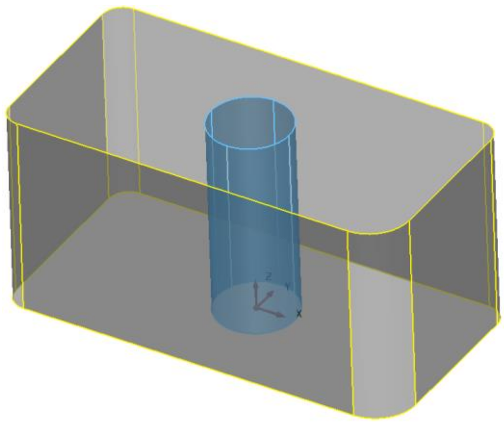
*This have **converted** both the **5mm** and **10mm fillets** to **8mm fillets** in **one operation**.*

Editing preceding Boolean operations

As well as the ability to **modify** the **original parameters** used in the **creation** of a **sub-solid item**, you can apply the **General Edit** options to preceding **Boolean** operations.

1 Select only the **Boolean Subtraction** operation in the **History Tree** as indicated.

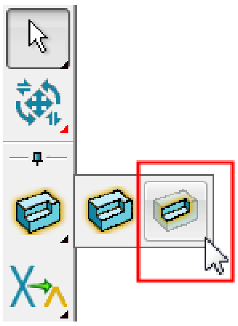




The cylinder (**Boolean operation**) is highlighted in the **graphics area**.

2 Select  to display the General Edits toolbar.

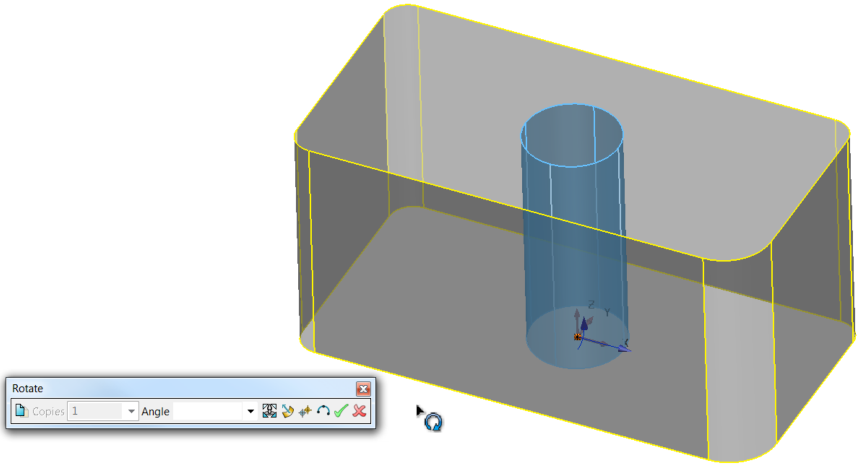
3 Select Edit Sub Item  from the flyout.



This ensures the next general edit will now apply to the selected sub-items only. In this case the **cylinder** and the **Boolean Remove** which created the hole through the block.

4 Set **X** as the principle axis. 

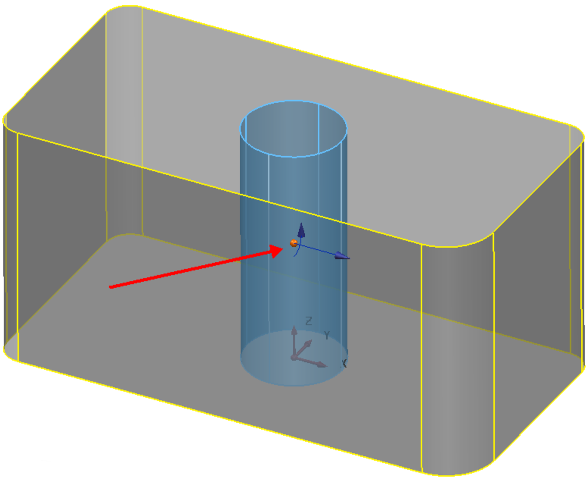
5 Select **Rotate**  from the **General Edit** toolbar.



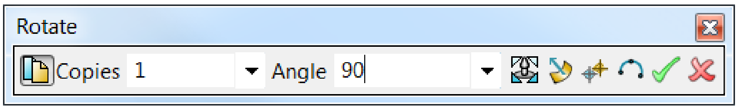
The **blue arrow** indicates the **X axis** as the **rotational axis**. However the **rotation origin** itself needs to be **transformed** to the **centre** of the **cylinder** in **Z**.

6 On the **rotate dialog** select **reposition rotation axis**. 

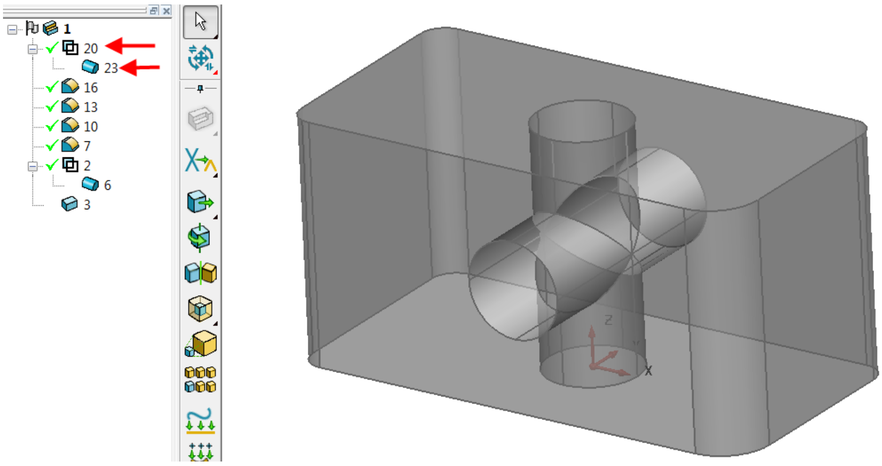
7 In the **command box** enter **0 0 25** to position the rotation axis half way up the cylinder.



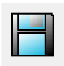
8 In the dialog, select **Copy the selected items** (to keep original) and enter an **Angle** of **90°**, press the **Enter** key to show a **Preview**.



9 Press the green tick to accept.

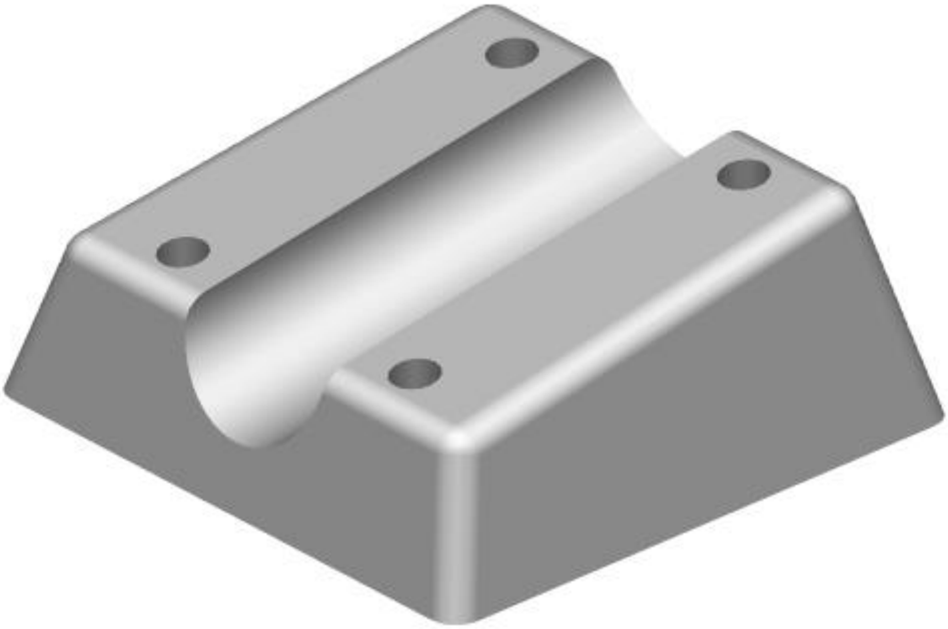


The original **sub-solid cylinder** is **copied** and **rotated** within the **solid**. The new **sub-solid cylinder** appears at the **top** of the **History Tree**.

10 **Save**  the **model** then **close** the session.

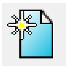
Solid modelling example 2

In this next **example** the following component will be created.



The **Mounting Block** base dimensions are **Length (X)** = **100mm**, **Width (Y) = 75mm** and **Height (Z) = 50mm** centrally positioned relative to the **workplane** at The **Draft Angle** on all **4** walls is **5°**.

A **cylindrical section** of **Radius 15mm** runs with its **axis along** and **central** to the **sloping upper face**. All **fillets** are **Radius 5mm**.

1 Create a **new model**. 

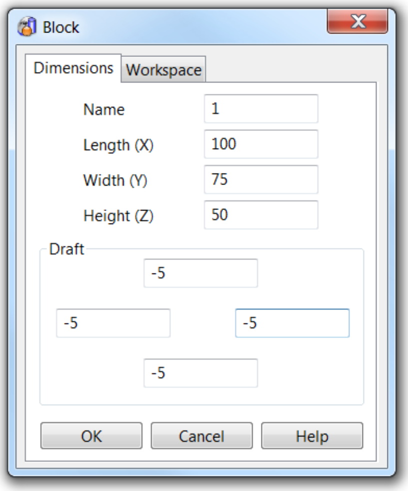
2 Set the **principal plane** to **Z** and create a **workplane** at **0**.

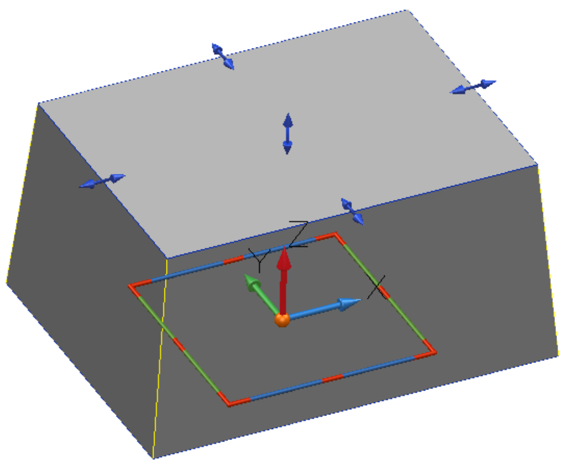
3 Select **Solid options**  from the **main toolbar**.

4 From the **Solids** menu select **Create Solid Block**. 

5 **Centre** the base of the **solid block** about the **workplane** by typing **0** in the **command box** (or snapping to the **workplane**).

6 **Double-click** on the **solid** in the **graphics area** to raise the **Block Edit** dialog.

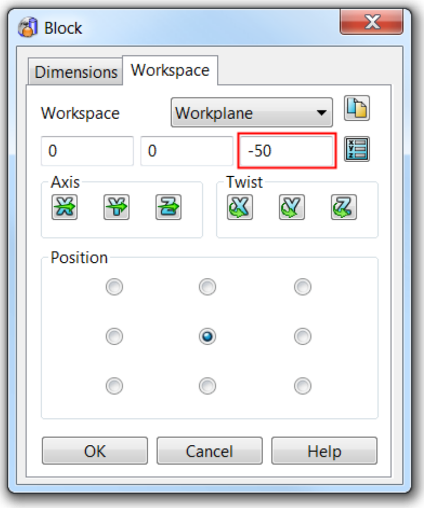


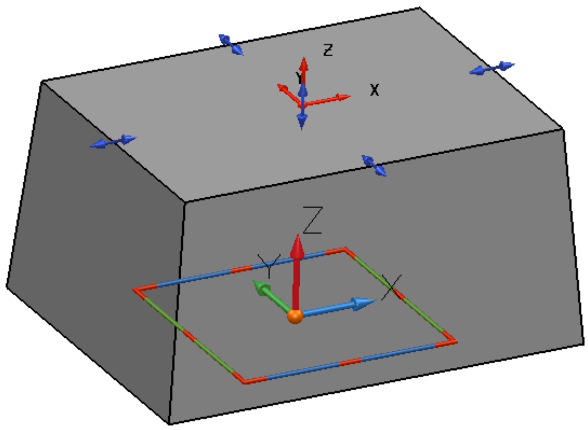


*A solid **primitive** block allows the **draft angles** to be applied immediately.*

7 Select the **Workspace** tab in the editing dialog.

8 Enter **-50mm** in the third coordinate box (**Z Axis**) and select **OK**.





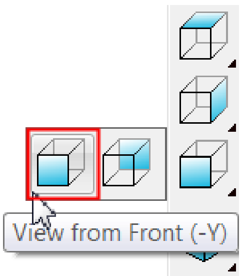
*The block has been **transformed 50mm** in the **–Z** axis. This has resulted in the **original workplane** to be **positioned** on the **top face** of the **solid block**.*

Applying fillets to the block

Prior to the fillets the **upper face** will be **modified** to create the **slope**. An **extruded surface** will be used to demonstrate the ability of **PowerSHAPE** to apply **Boolean** operations to **solids** using **surfaces**.

***Primitive Surfaces** are discussed in detail in the **next chapter**.*

1 From the **Views** toolbar, select a **View from Front** (**-Y**).

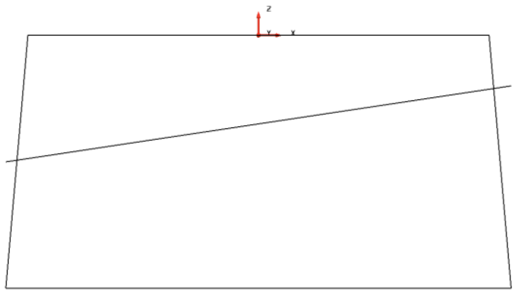


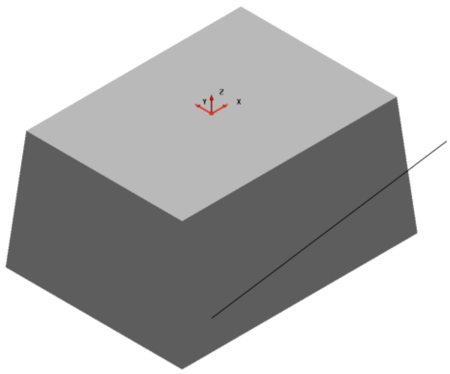
*By default the **Principal Axis automatically** switches to the **selected view**.* 

2 Select **create a single line**  from the **Line** menu.

3 Enter the **start point** coordinates **-50 -50 -25**.

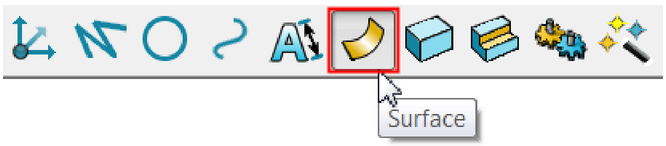
4 Next, enter the absolute coordinates **abs 50 -50 -10** to complete the line.





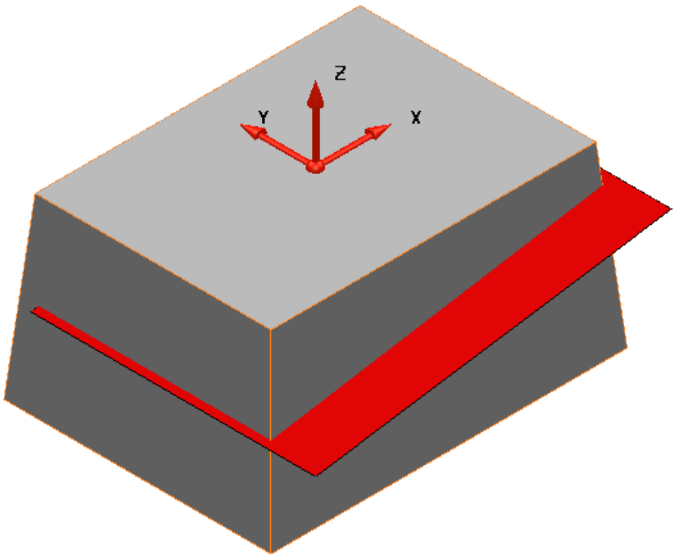
5 Select the **angled line**.

6 From the **Main** toolbar, select **Surface** to display the **Surface** menu.



7 From **Surface** toolbar, select **Create a Surface** Extrusion.

*This **training manual** will cover **surface modelling** in the next chapter.*

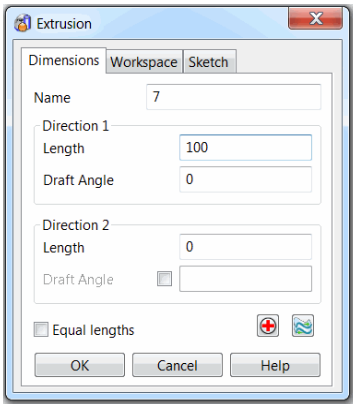


With the **line pre-selected**, an **extrusion surface** is created **immediately**.

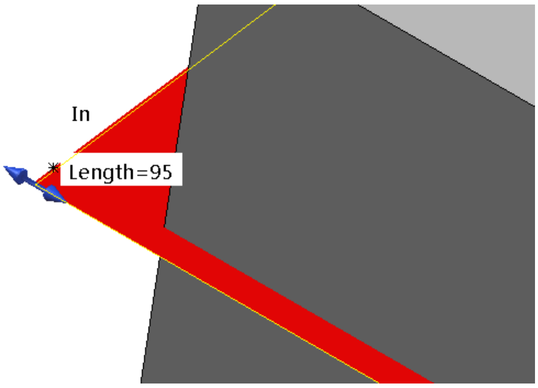
*The **upper side** of the **surface** is **shaded red** (**inside**) while the **underside** is the **current default colour** for **surfaces** (**outside**).*

8 **If** the **extrusion surface** does not **fully pass** through the **solid**, **double-click** the **surface** to display the **Extrusion** dialog. Insert a **Length** of **95mm.**

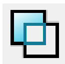
9 Select **OK.**



*You can also **change** the **length** by **dragging** the **arrow** at the end of the **selected extrusion surface**.*

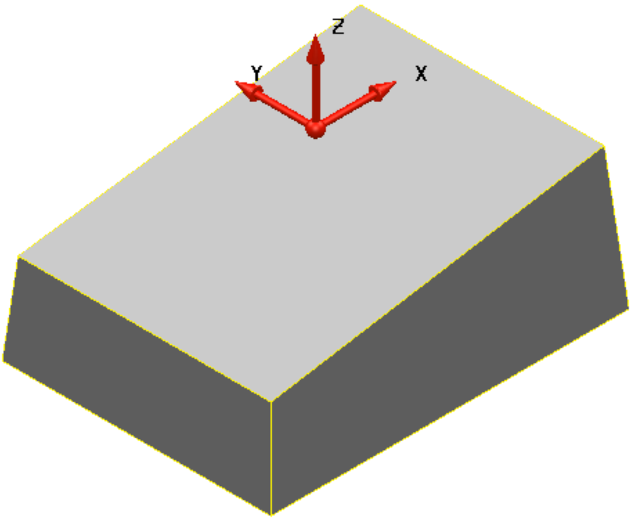


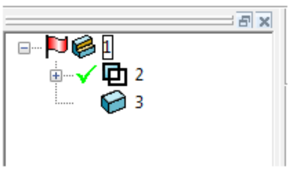
10 With the **surface** selected and the solid **active**, select **Solid Features**  from the **Main** toolbar.

11 Select **Boolean Remove.** 

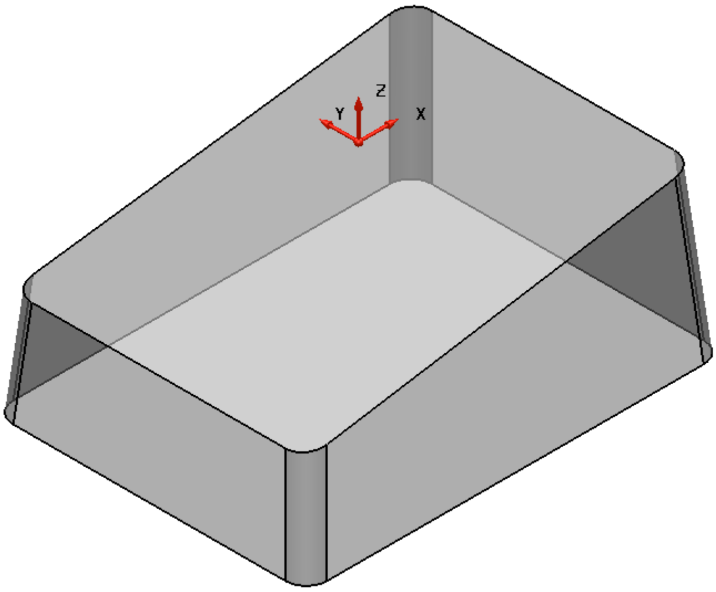
 *The **Split Solid***  *button can also be used to same affect.*

The **area of the solid** on the **red side** of the **surface** is **removed**, with the surface removal registered in the **Feature Tree**.

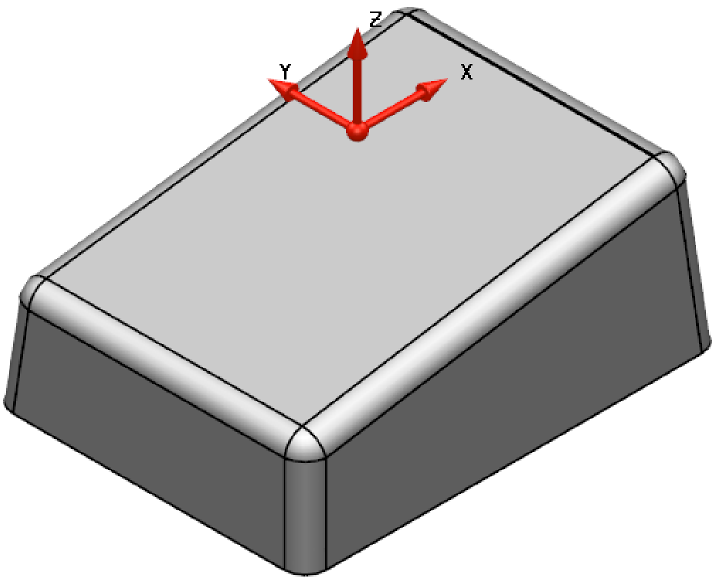




12 **Apply** a **5mm solid fillet**  on each **draft edge** of the **solid block**.



13 **Repeat** the **fillet** of **Radius 5mm** on the **sloped rectangular face**.



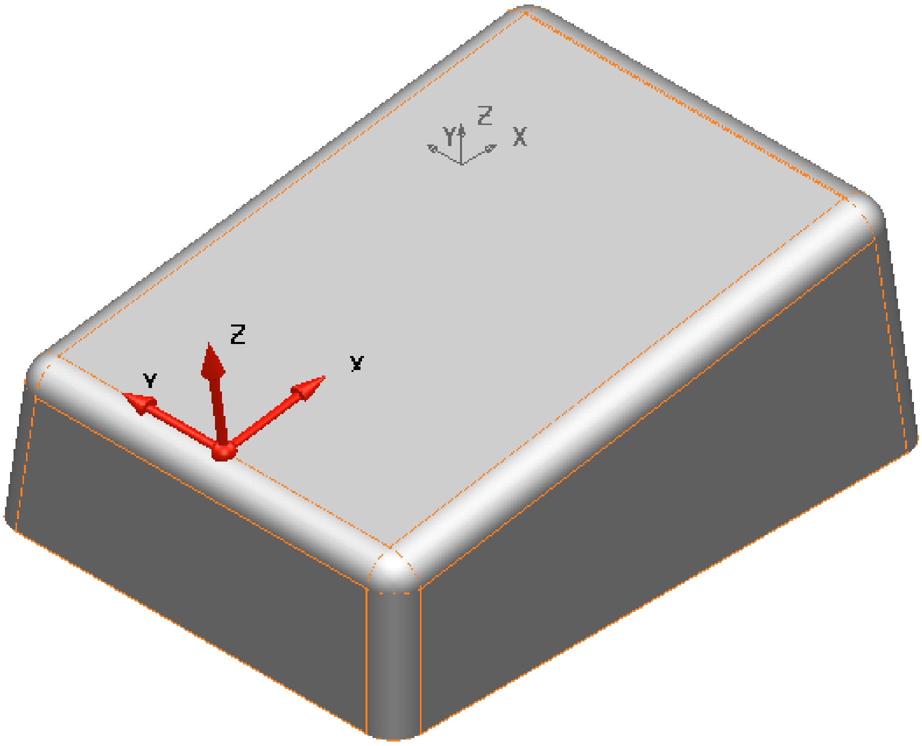
*By creating the **fillets** in **two separate operations**, the **sidewall** and **top surface fillets** are **independent** which makes it easier to make changes to the individual fillet groups at a later stage by control the grouping in the **history tree**.*

Creating a cylinder

A **new solid cylinder** will be **created** then **removed** from the **existing solid**.

Before creating the cylinder a **new workplane** is **required aligned** to the **sloping face** and **positioned** at a **known keypoint**.

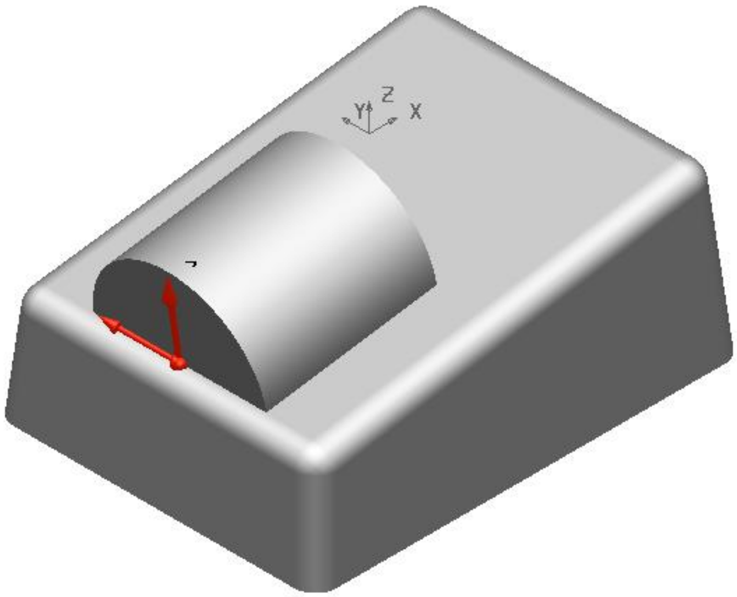
14 **Create a single Workplane aligned to geometry**  and place a **workplane** at the **centre** of the **lower edge** of the **top sloping face** to create a **workplane** with its **Z axis normal** to the **sloping face** of the **solid**. This can be done by **snapping** to the **keypoint** using the **intelligent cursor**.



15 From the **bottom-left** of the **graphics area**, select X as the **Principal Axis**.

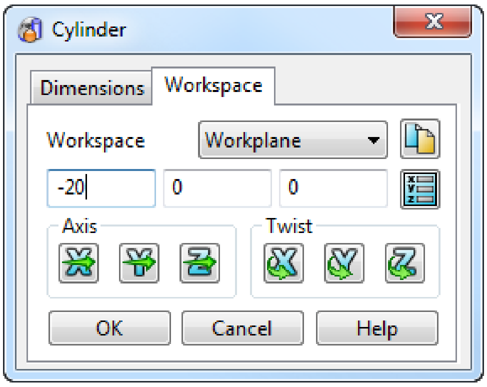
16 From the **Solid**  toolbar, select **Create Solid Cylinder**. 

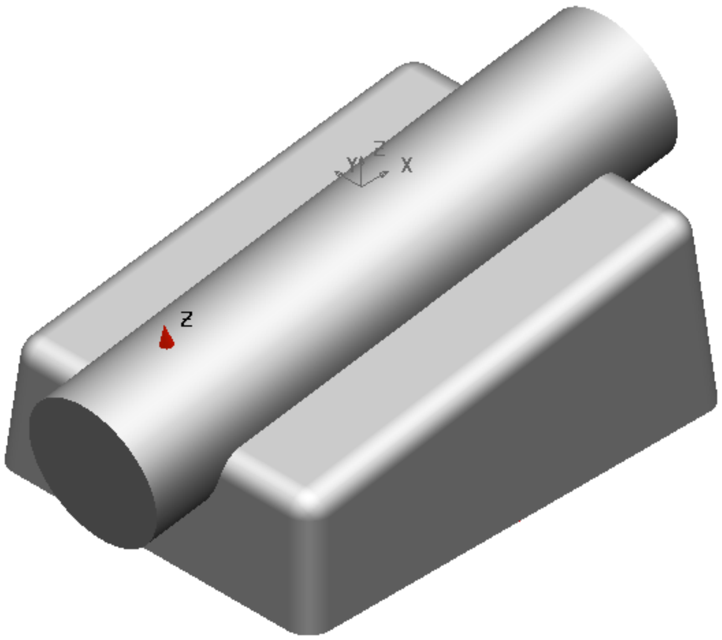
17 **Left-click** on the new **workplane** to locate the new **solid cylinder** on to it, or type **0** into the type **command box** and press the **Enter** key.



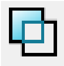
18 Modify the cylinder parameter to a **Radius of 15mm** and **length of 125mm.**

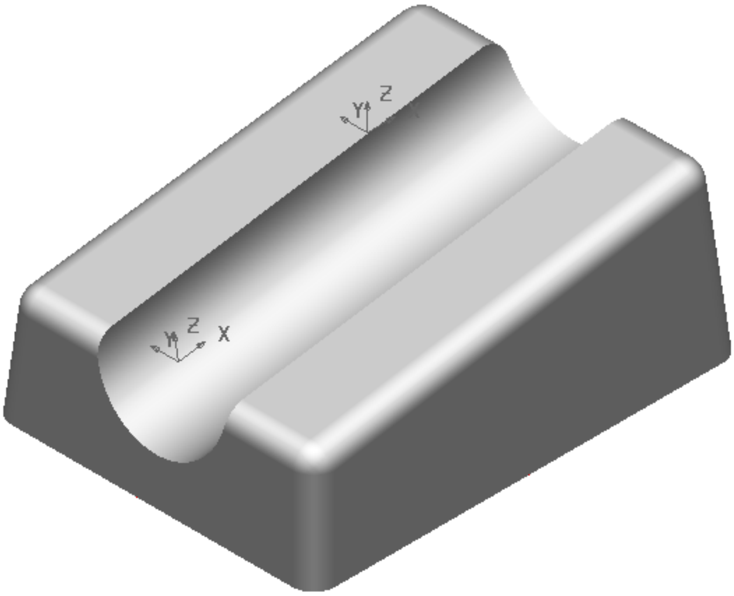
19 In the **Workspace** tab of the cylinder dialog, modify the **X coordinate to -20.**





Using a Boolean operation to remove the cylinder

1 Boolean **Remove**  the **solid cylinder** from the **main block solid**.

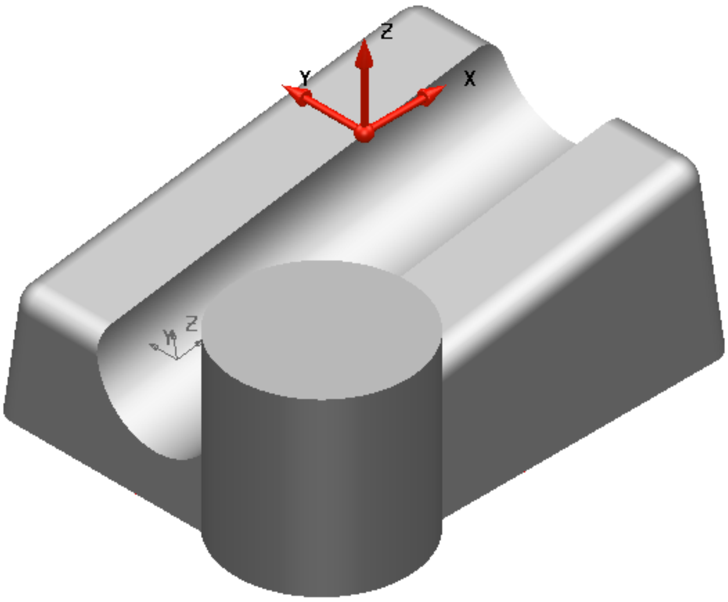


2 **Activate** the original workplane (**Workplane 1**) at the **top centre** of the **component**.

3 From the **bottom-left** of the **graphics area**, select the **Z Axis** as the **principal axis**. 

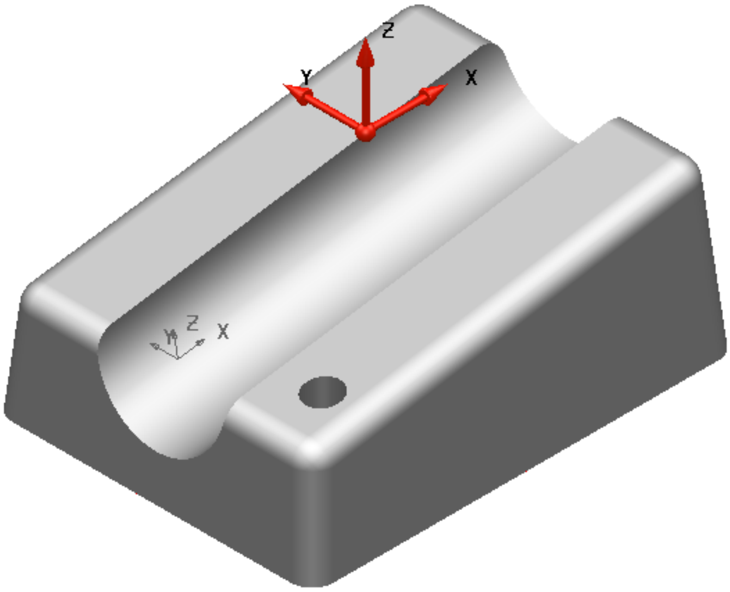
4 Create a **Solid Cylinder**  positioned at **-35 -25 -50**.

5 Modify the **Radius** to **4mm** and the **Length** to **40mm**.



 *You now need to **remove** the new **solid cylinder** from the **main solid** using a **Boolean Subtraction** operation.*

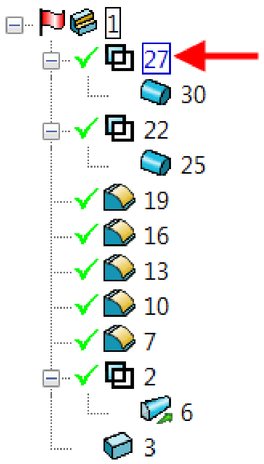
6 **Boolean Subtraction**  the **cylinder** from the **main solid**.



Mirroring the hole feature

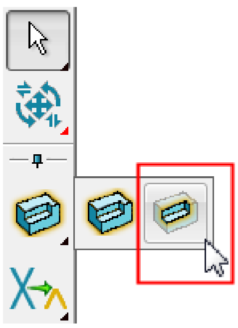
The remaining **hole features** we require will be **mirrored** across for a **more efficient process**.

1 Select the latest **Boolean Subtraction** feature displayed in the **Feature Tree**.

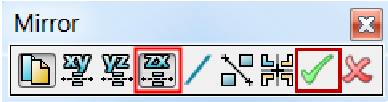


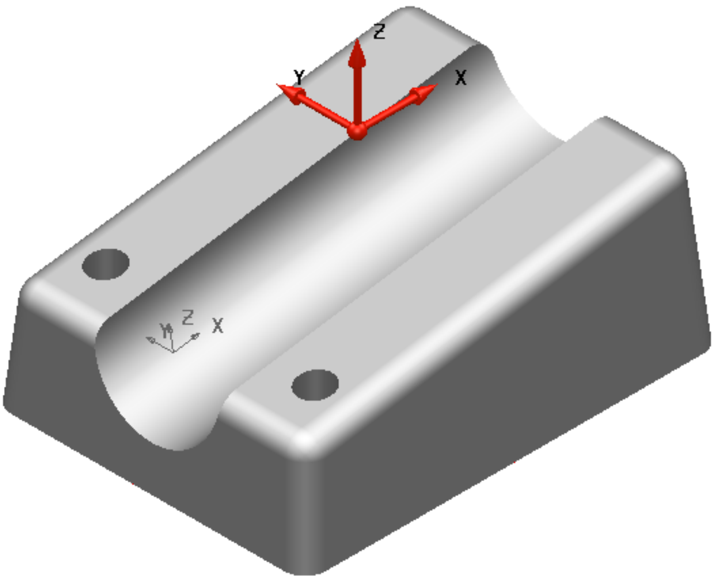
2 Select  to display the **General Edits** toolbar.

3 Select **Edit Sub Item**  from the flyout.

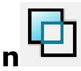


4 **Mirror**  the hole across the **ZX plane.**

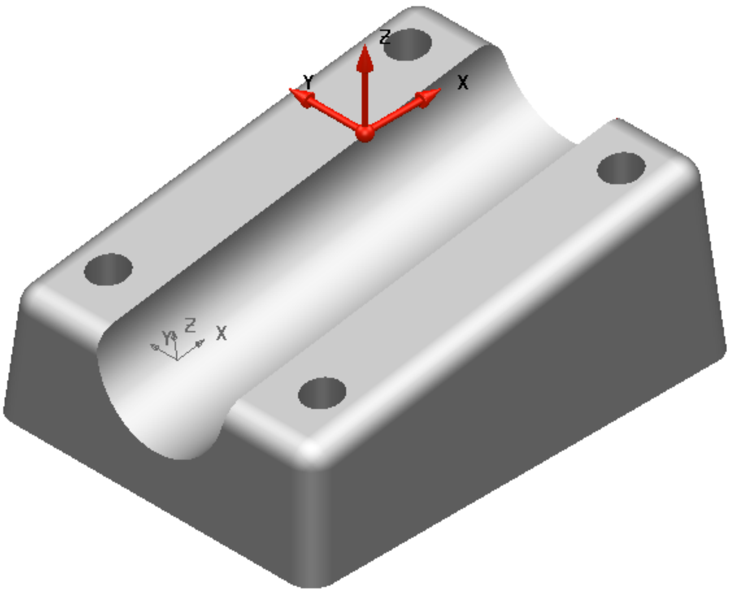




The new hole is displayed at the top of the **Feature Tree**.

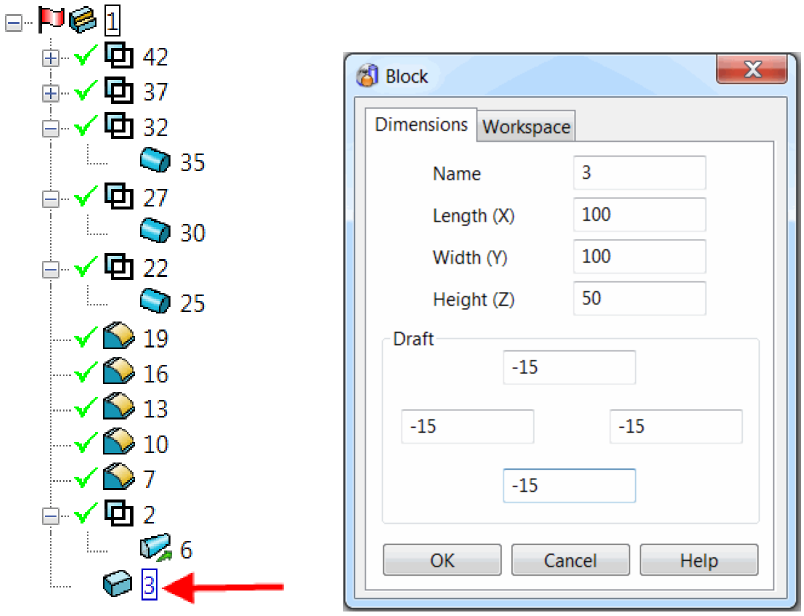
5 **Ctrl + select** (**multi-select**) the **Boolean Subtraction**  feature for **both holes** in the **Feature Tree**.

6 As before **mirror** the features but this time across the **YZ plane.**



Basic Solid Editing

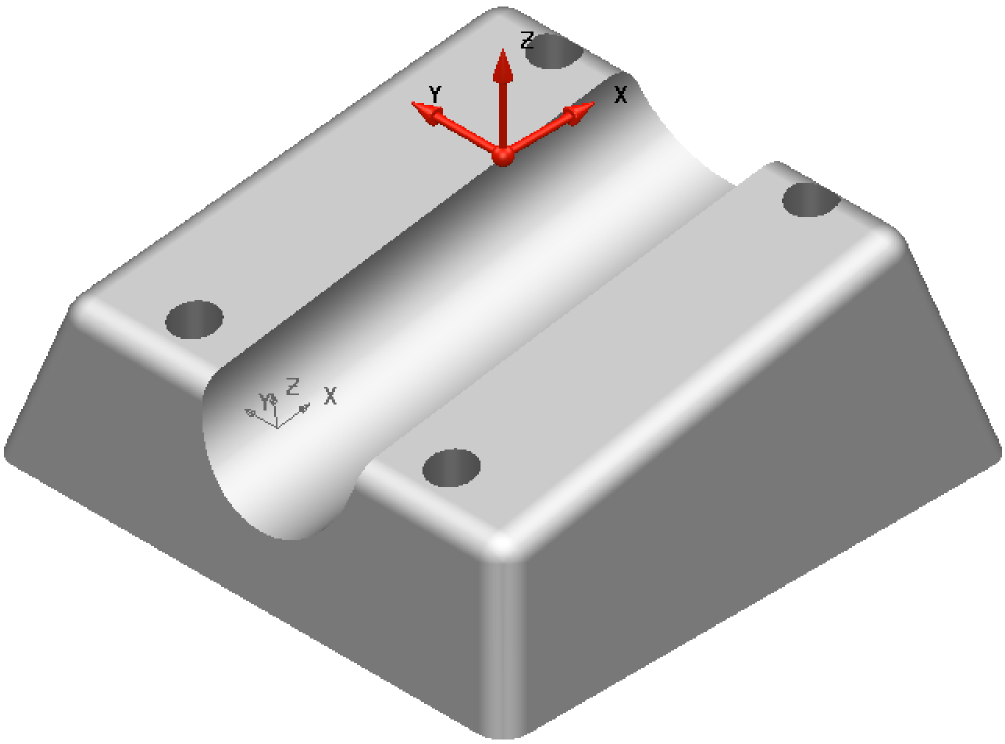
1 Double left-click on the original solid block symbol at the bottom of the **Feature Tree.** The **Block** dialog is displayed.



2 **Modify** the value for **Width** (**Y**) from **75mm** to **100mm**, and all four **Draft angles** to **-15°**

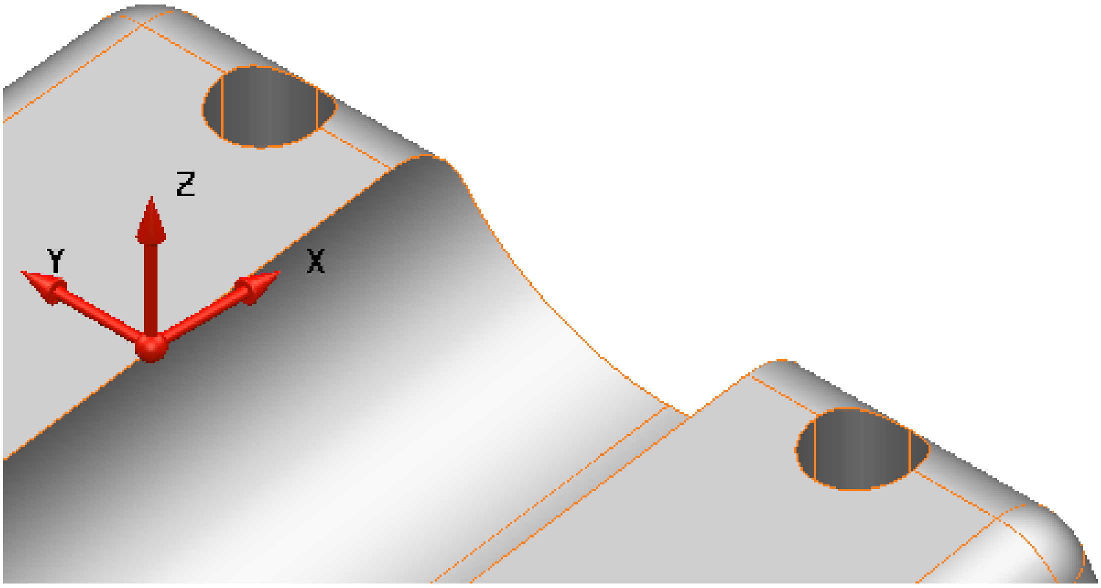
3 Select **OK**.

The **solid component** is updated with the changes.

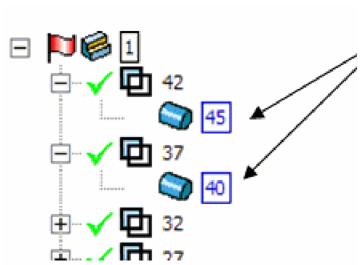


The **holes** must now be moved **clear** of the **fillets** running around the **upper sloping** face.

4 **Zoom** into the **area containing** the **two latest holes** in the **+X** direction.



5 Hold down **Ctrl** and **click** the two **Boolean operations** (**holes**) to **select** them **both**. The corresponding **cylinders** are **highlighted** in the **CAD model**.

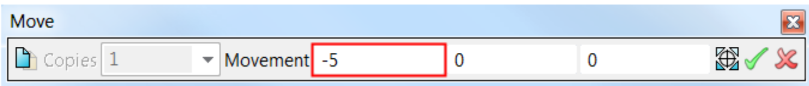


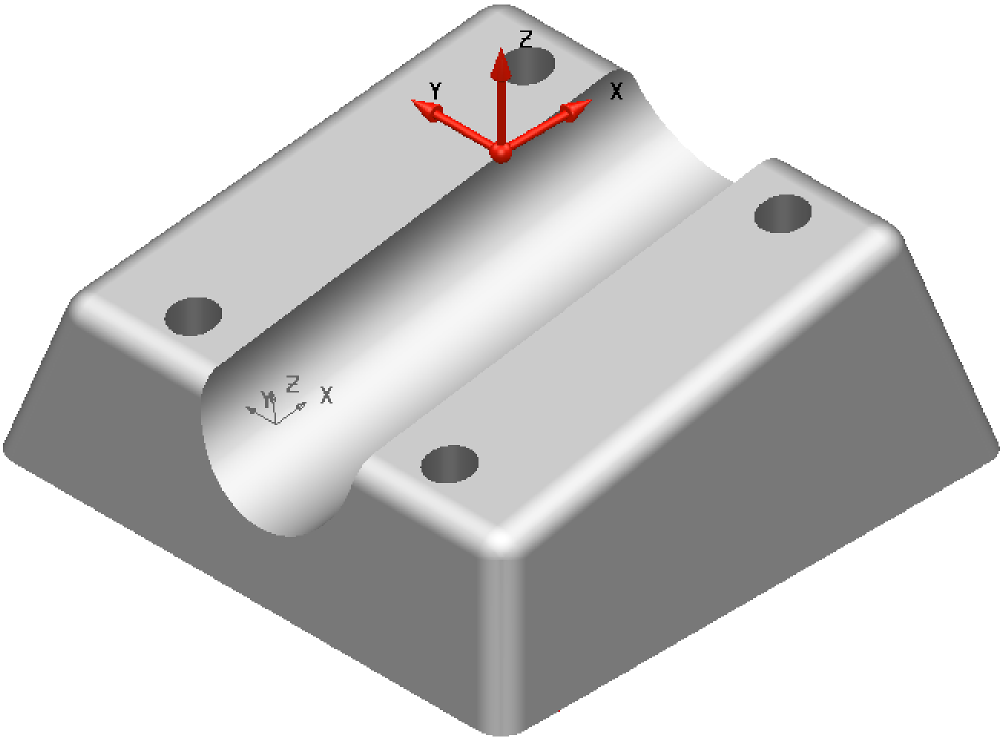
6 Select  to display the **General Edits** toolbar.

7 Select **Edit Sub Item**  from the flyout.

8 Select **Move**. 

9 In the **X Input** box, type **-5** and press **Enter**, then **green tick** to **confirm**.



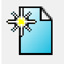


10 Select **File > Save As**:

.....***Training Data******Coursework******Solid Modelling Example2***

11 **Close** the model.

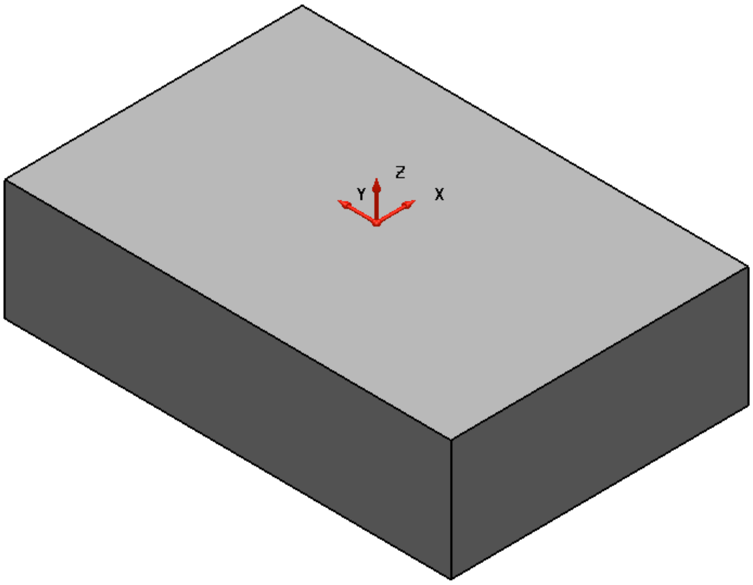
Solid Modelling Example 3 – Die block

1 Open a **new model**. 

2 Create a **workplane** at **0** and ensure the **Principal Axis** is set to **Z**.

3 **Create a solid block**  at **0 0 -40**.

4 Modify the **Length (X) 100mm**, **Width (Y) 150mm** and **Height** (Z) **40mm**.



5 Set the **Principal Axis** to X.

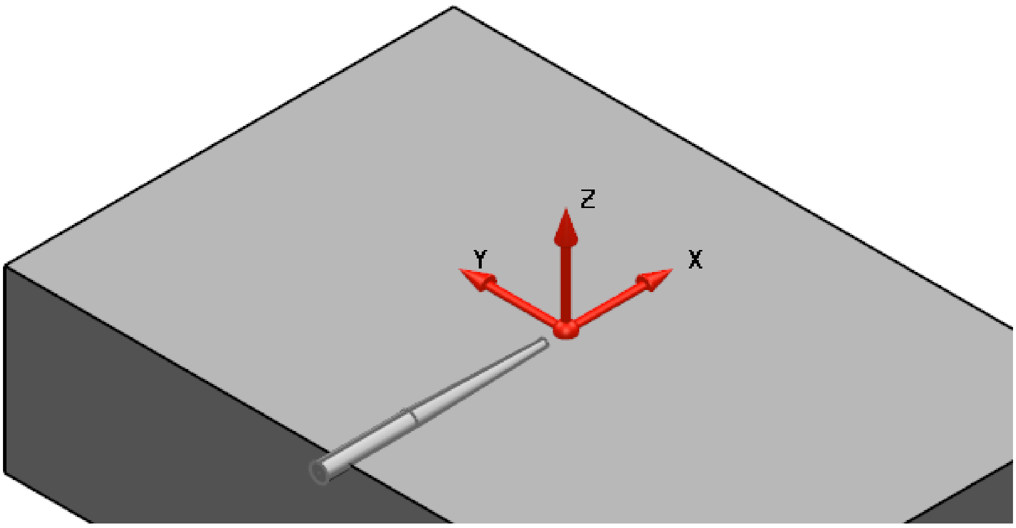
6 From the **Solids** menu select **Create a solid cylinder**  positioned at **-55 0 0**.

7 Modify the **Radius** to **2mm** and the **Length** to **20mm**.

8 Create a **solid cone**  at co-ordinates **-35 0 0**.

9 Modify the **top radius** to **1mm**, **Base radius** to **2mm**, and **Length** to **30mm**.

The model now contains **three solids**; the **large block** is currently the **active solid**, you should be able to see this in the **Solid Feature Tree**.

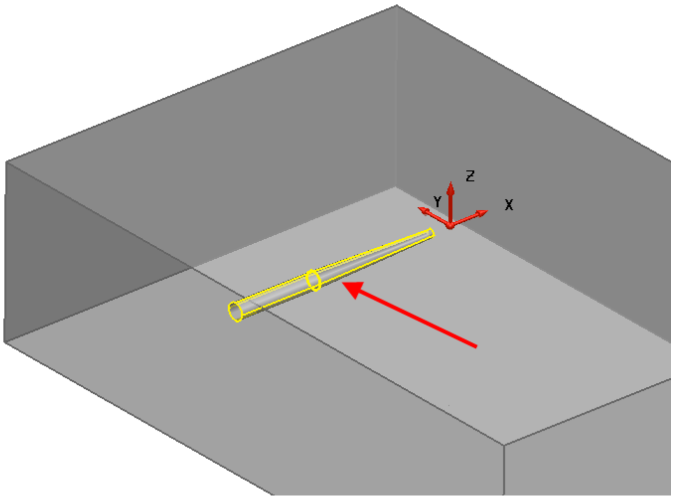


10 **Select** and **Blank (Ctrl+J)** the **solid block**.

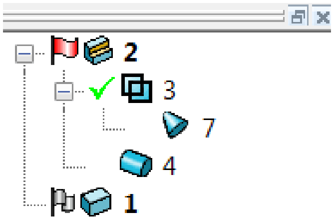
11 **Boolean Add**  the **Solid cone** to the **solid cylinder.**

The combined solids create the '**sprue**' for the **die block**. (The hole through which **molten material** is **channelled** into a **mould**).

12 **Unblank (Ctrl+K)** the **solid block**.

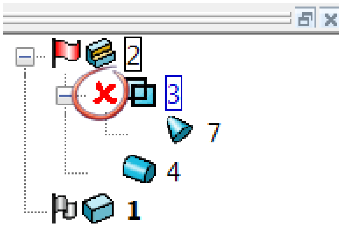


*The **Feature Tree** shows that the '**sprue**' **solid** comprises the **solid cone** and the **solid cylinder**.*



Suppressing a feature

Solid features can be **temporarily suppressed** (**removed** from the **solid**) left **clicking** on the **green tick** in the **History Tree**. The **green tick** changes to a **red cross**.



PowerSHAPE remembers the **details** and **history** of the **suppressed feature**.

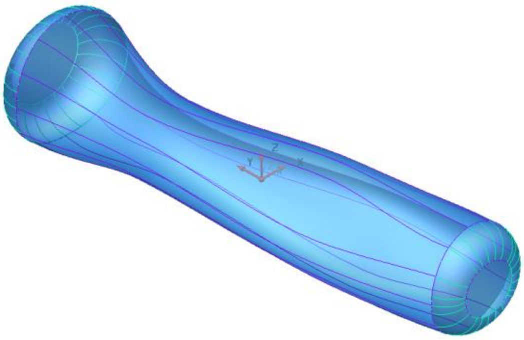
The **suppressed solid feature** can be **unsuppressed** by clicking on the **red cross** which will a **green tick** and the **feature** is **visible**.

Importing the molded component

The **molded component** to be **sunk** into the **die** will be **imported** as **surfaces** and then **converted into a solid**.

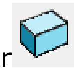
1 **Select** and **Blank** all of the solids and **Import** the model:

C:**Training Data\PowerSHAPE Data\handle.dgk**

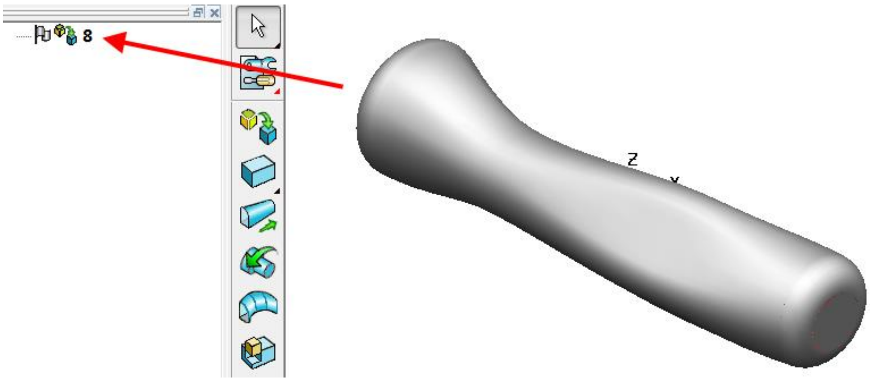


The **imported model** contains various **surfaces**, which will be used to create a **single solid**.

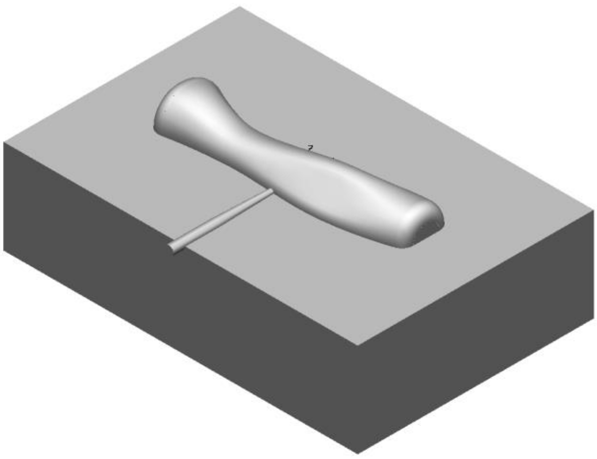
2 **Quick Select All Surfaces** from the **Selection** menu.

3 From the **Solids** toolbar  select **Create a Solid from selected Surfaces**. 

A **single solid** has been **created** and also **listed** in the **Feature tree**.



4 Select **Unblank** (**Ctrl+L**) to redisplay the **block** and **sprue** solids.

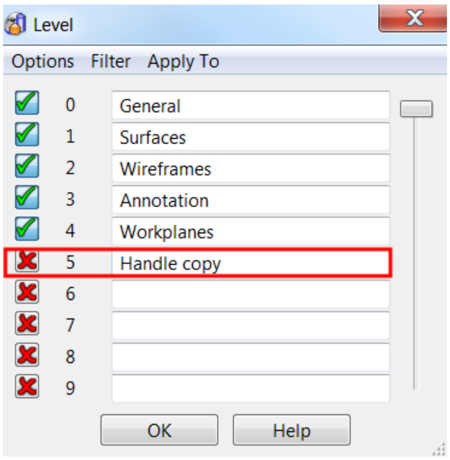


During a **Boolean Remove**, the **secondary solid** is lost as it **absorbs** into the **primary solid.** In this case, a **copy** of the **handle** will be required for later use as the basis for an **electrode.**

To achieve this a **copy** of the **handle** will be **created** and **placed** onto a **separate level**.

5 From the **lower-left corner** of the **window**, open the **levels** form. 

6 Name **level 5** as **Handle copy** and leave the **level switched off.**  Select **OK.**



7 Select the **handle solid.**

8 Select **Copy**  from the main toolbar followed by **Paste.** 

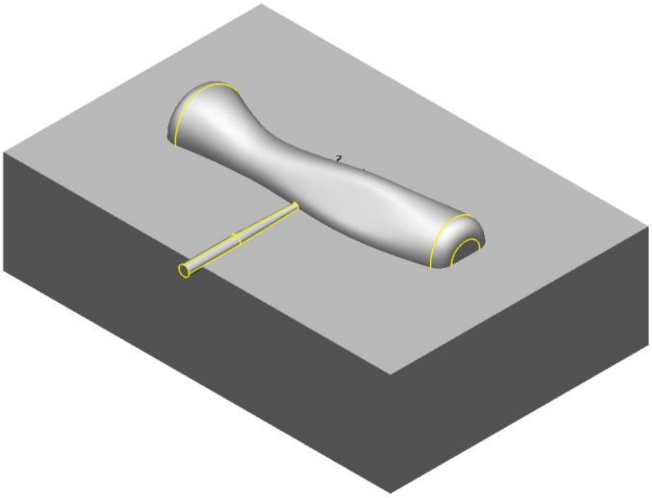
A **copy** of the **handle** has been created with **two identical handle solids** in the **feature tree**.

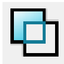
9 With **one** of the **handles selected**, **middle mouse click** over the **level 5** icon in the **levels toolbar.**

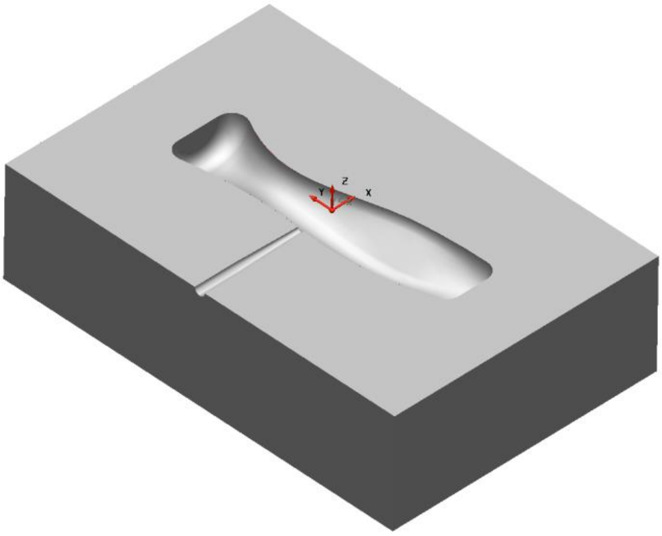
A **solid handle** has been transferred to the new level. With the **level 5 switched off** the copy of the **handle solid** is now **not visible**.

Creating cavity

1 Boolean **Add**  the '**sprue**' solid to the **handle solid**.



2 To finally create the required cavity, use the **Boolean Subtraction**  operation to remove the **handle and sprue** solid away from the **block**.



 *A **single solid** remains which has the **handle** and '**sprue**' **removed** from the **die block**.*

Creating the electrode from the solid

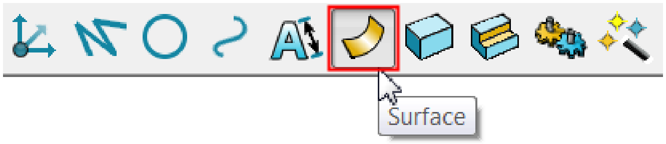
The **copy** of the **handle solid** you created earlier will now be used to create an **electrode**.

1 **Blank** the **main cavity solid** and then switch **ON level 5** to display the **handle copy**.

2 Ensure **Z** is selected as the **principal axis** is selected. .

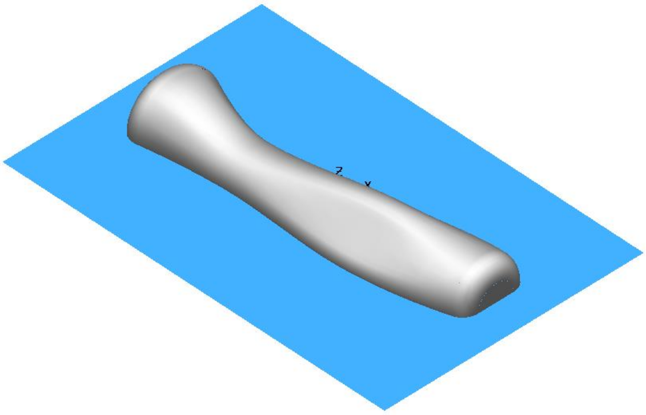
The ability of a **surface** to **split** a **solid** will now be shown.

3 From the **Main** toolbar, select **Surface** to display the **Surface** menu.



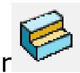
4 From Surface menu select **Create a Plane primitive**  and position on the **Workplane** at **0 0 0**.

5 Modify the **parameters** to **Width (X) 80mm** and **Length (Y) 120mm**.

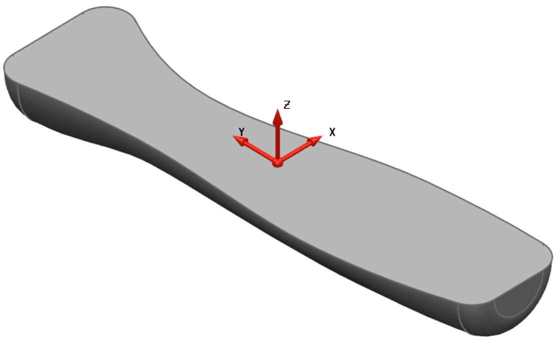


6 **Blank ALL Solids EXCEPT** the **handle**.

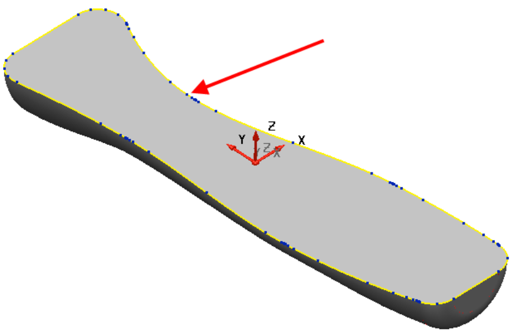
 *Select the **handle** and press **Ctrl+K**.*

7 With the surface selected, select **Solid split**  from the **Solid Features** toolbar .

8 **Select** and **delete** the **upper half** of the **handle**.

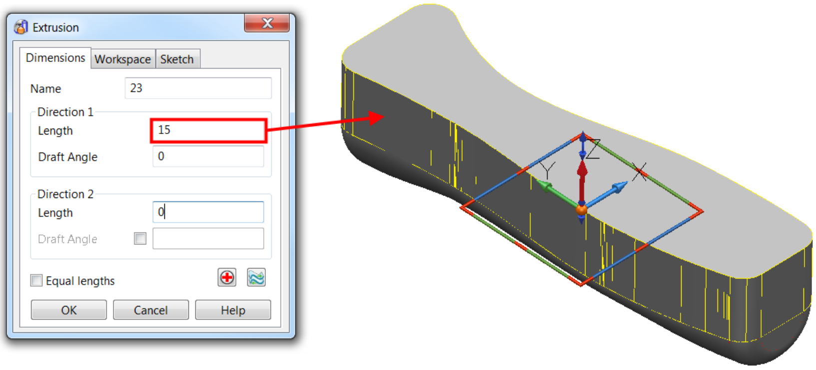


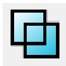
9 Create a **composite curve** around the **upper edge** of the **remaining handle**.



10 From the **Solids**  menu, **Create a Solid Extrusion**. 

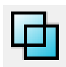
11 **Modify** the **extrusion** and change its **length** to **15mm**.

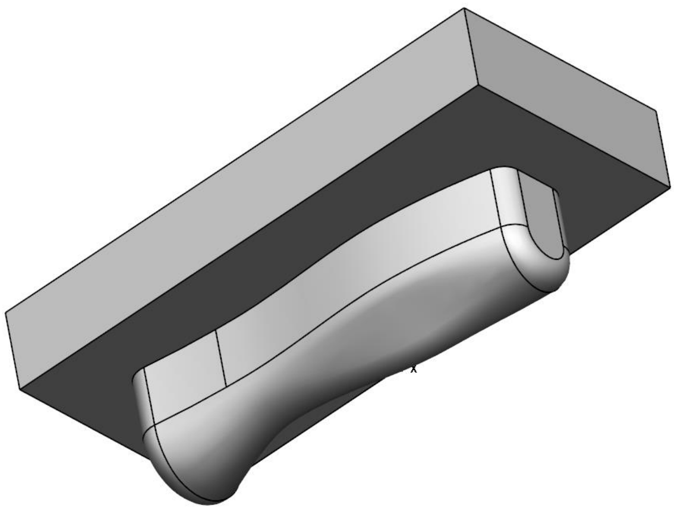


12 Using the **Boolean Addition**  operation **add** the extrusion to the handle solid.

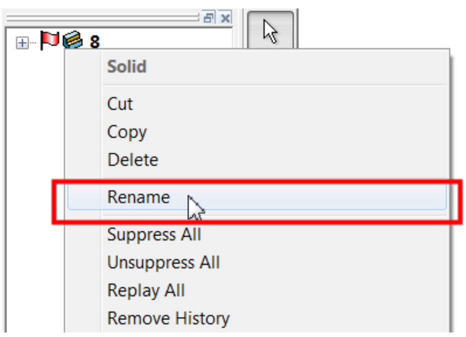
*The **solid extrusion** forms a **run off** to provide **clearance** between the **electrode** and the **back plate**.*

13 Create a **solid block**  at position **0 0 15** with **Length (X) 60mm**, **Width (Y) 130mm** and **Height (Z) 20mm**.

14 **Boolean Addition**  the **two solids** together.



15 **Rename** the **solid** as **Cavity Electrode** by right clicking the **solid** in the **feature tree** and selecting **Rename**.



16 Switch **OFF level 5**.

Creating a slide

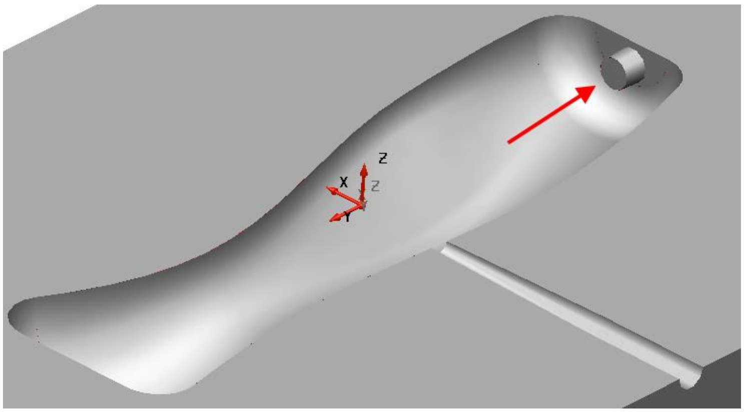
A **slide** will be created to provide an **undercut hole** in the **molding**.

1 Select **Unblank** to **redisplay** the **main cavity solid**.

2 Ensure it is **active** and set **principal axis** as **Y**.

3 Create a **solid cone**  positioned at **4 –52 -3.**

4 Modify to **Top Radius 2mm**, **Bottom Radius 2.25mm** and **Length 5mm**.



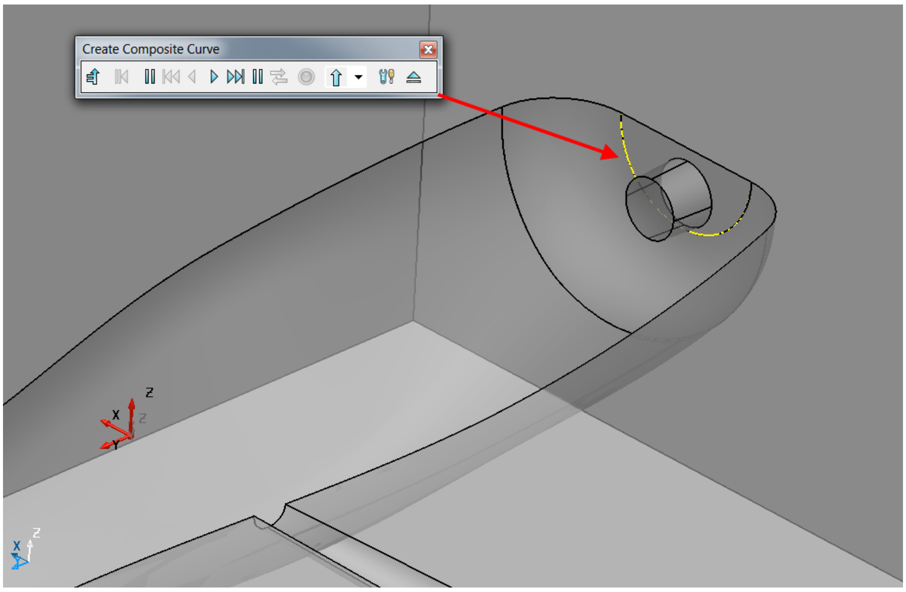
5 **Boolean Addition**  the **small cone** to the main **cavity block**.

The **solid** now has a **localised undercut core** that is to be **separated** onto a **retractable slide**. A **workplane** is required at the **end** of the **core** to **assist** with the **separation of the slide**.

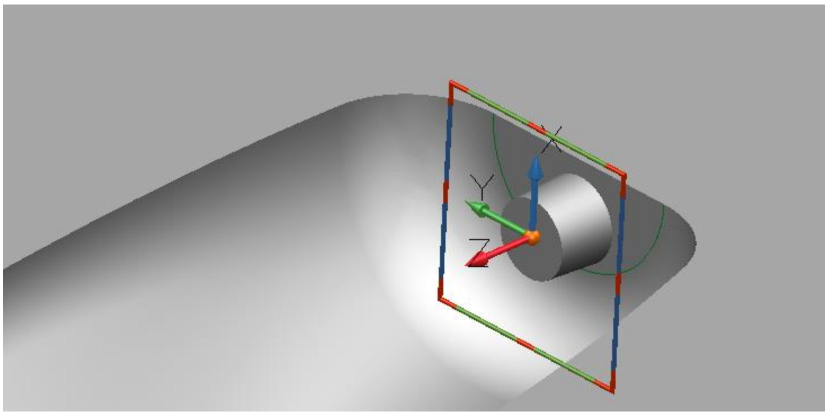
An **extruded surface** will be **created** and **utilised** again to **split** the **solid** and provide the **required component**.

6 Zoom into the small **cone solid** and create a **composite curve** on the edge of the bottom flat as shown.

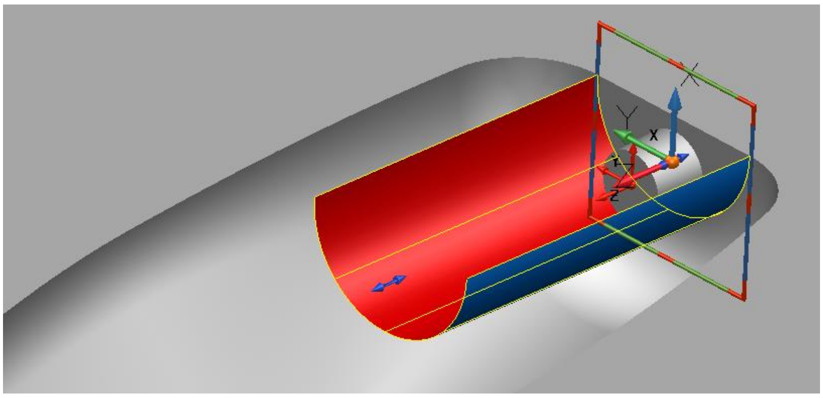
*Switching to **wireframe shaded** or **transparent** view will **display** the **edge clearer** making it easier to **select** and **trace**.*



7 **Create a single Workplane aligned to geometry**  and click and snap at the centre of the **outer planar face** of the **small cone**.



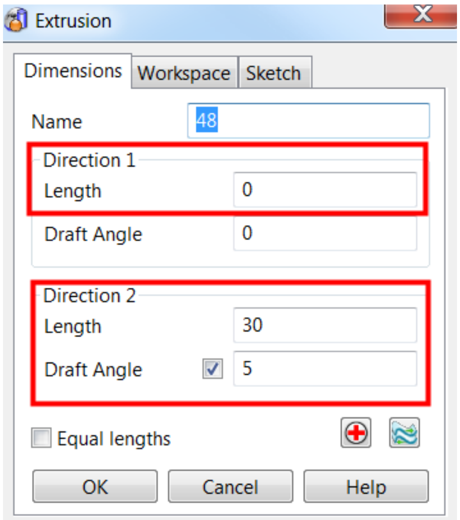
8 Select the **composite curve** and create a **Surface extrusion**. 



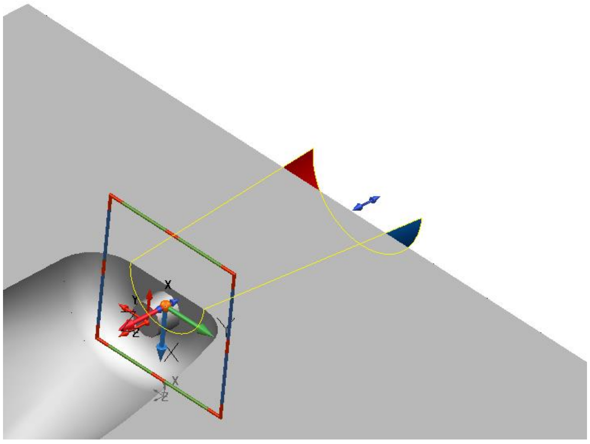
The **surface** needs to be **extruded through** the **model** including a **taper angle**. Both can be achieved simply from the **extrusion form**.

9 **Double-click** on the **extrusion surface** to open the **Extrusion** dialog so you can edit the **extrusion**.

10 Modify **Direction Length 2 to 30mm** then **Direction 1** Length **to 0mm**. Tick the **Draft angle** box in **Direction 2** and enter an **angle of 5°**.



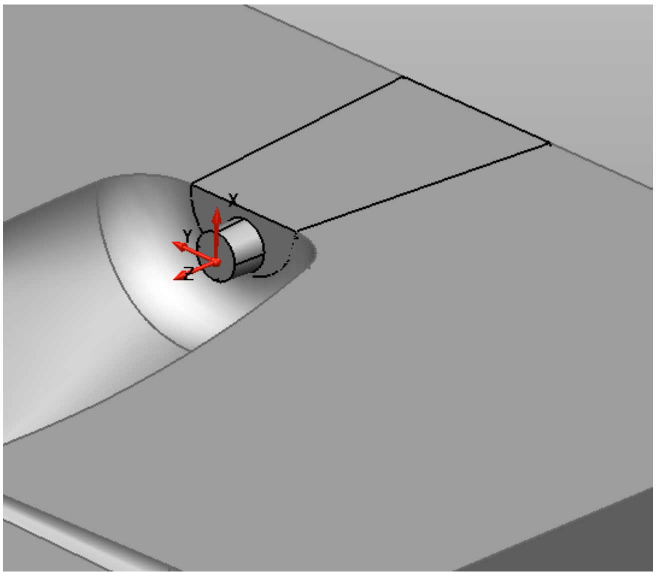
11 Select **OK.**



The **extrusion surface** will be used to **split** the **main solid extracting** the **slide** as a **separate solid**.

12 Ensure the **main solid is active** and the **extruded surface** is selected.

13 Select **Solid Split**  from the **Solid Features** toolbar .



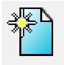
Both resultant solids retain their separate associations within the **History Tree**.

14 Select **File > Save As**:

.....***Training Data******Coursework******Solid Die.psmodel***

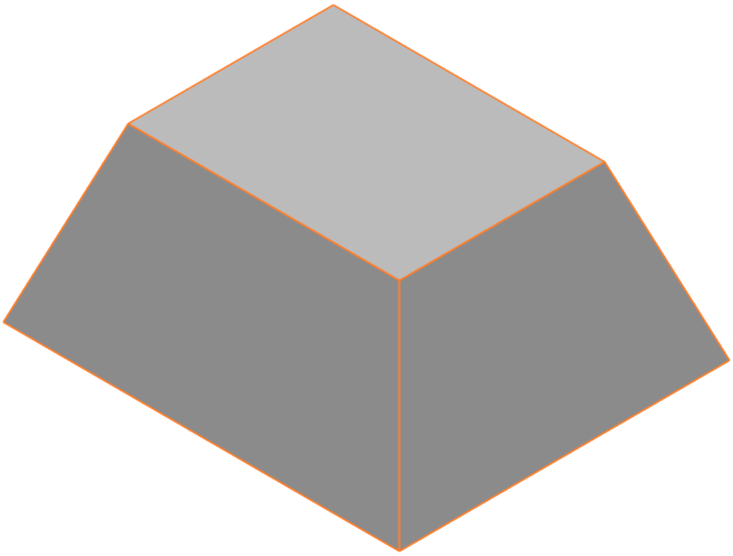
15 **Close** the **model**.

Burglar Alarm Box Example

1 Open a **new model**. 

2 Create a **workplane at 0** renamed **Datum**, and ensure the **Principal Axis** is set to **Z**.

3 Create a **solid block**  at **0 0 0** of **Length (X) 250mm, Width (Y) 300mm, Height (Z) 130mm** and **Draft 340°** on all **four** sides.



This has formed the **basic shape** for the **alarm box**.

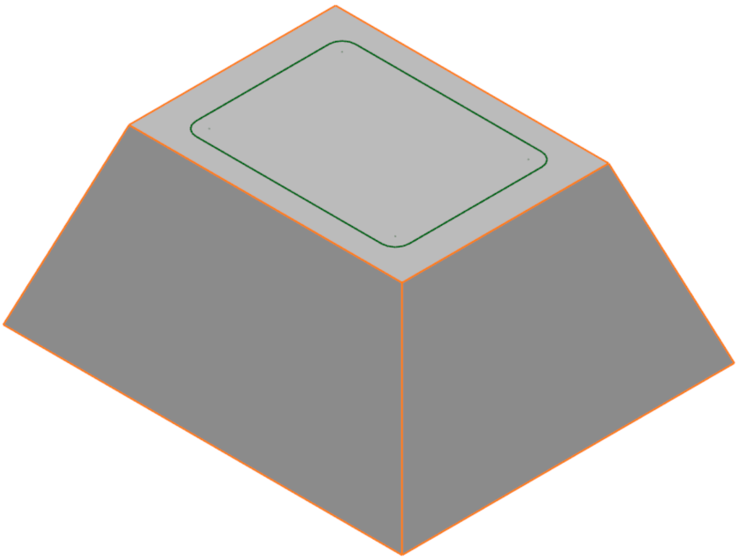
Adding solid features

Further **solid features**  will now be added.

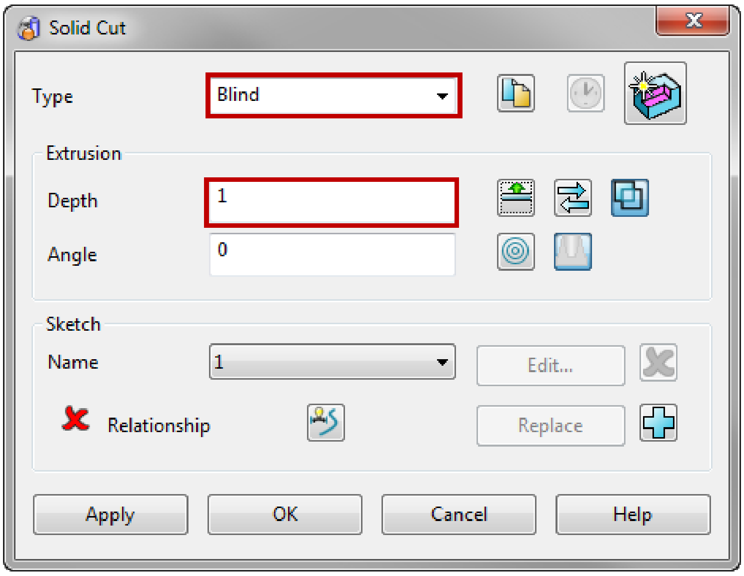
1 Create a **rectangle**  starting from **-60 -80 130** to **120 160**.

2 Create a **composite curve** from the **rectangle**.

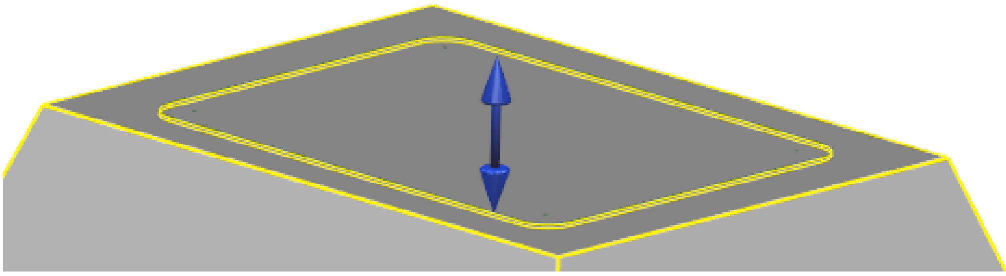
3 Create a **Radius 10mm** fillet on all corners of the **composite curve**.



4 With the **composite curve selected**, select **Create Solid cut**  from the **Solid Features** toolbar. 



5 In the **Type** drop-down list, select **Blind** and enter **Depth** of **1mm** and select **OK**.

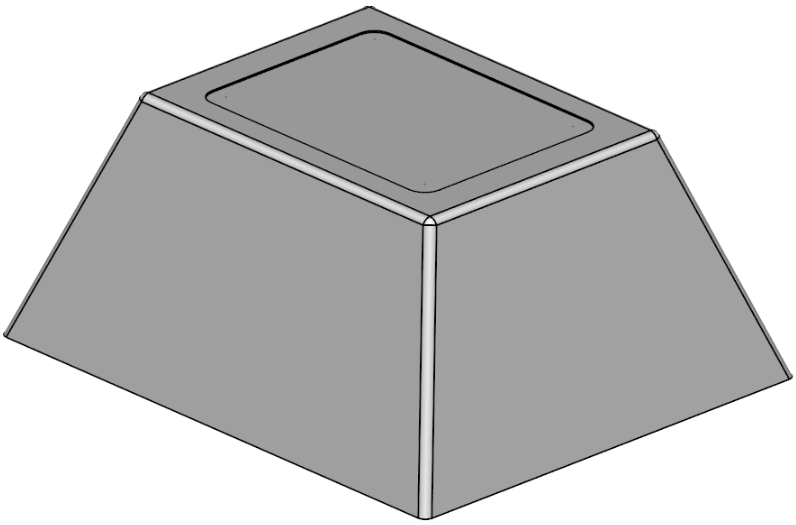


A recess of **depth 1mm** is created in the **solid** downwards from the composite curve into the **solid**.

The **next stage** is to **remove** the **sharp edges** from the **main body** by **adding solid fillets**.

6 Select **Create Solid Fillet**  from the **Features**  toolbar.

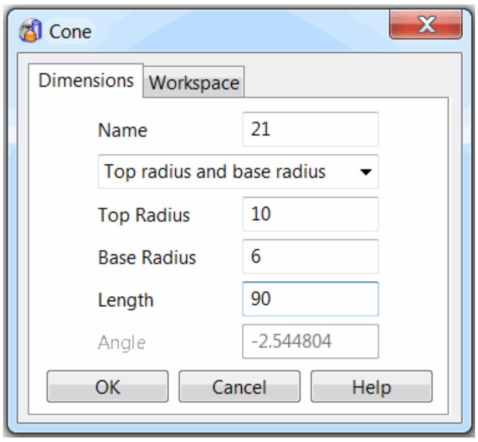
7 Apply a **fillet** of **radius 10mm** on the **four vertical** and **top edges**.

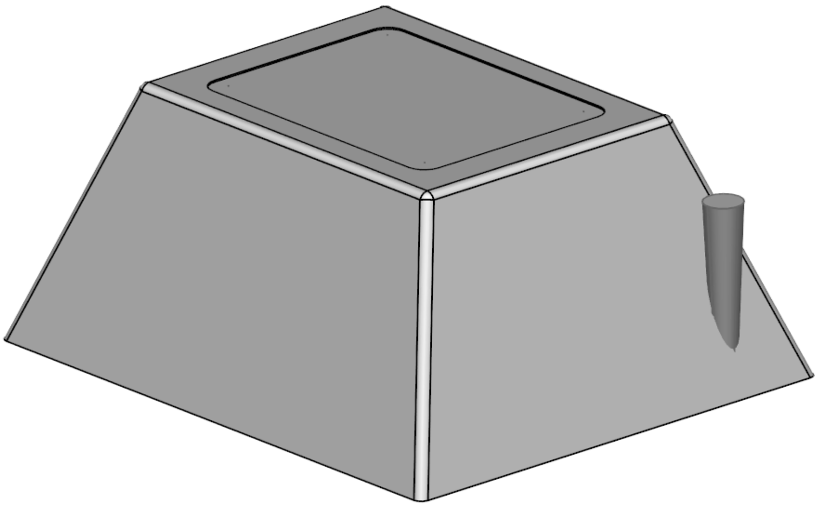


Create four screw apertures

1 Create a **solid cone**  at **80 -135 5**.

2 **Double-click** on the **cone** to display the **Cone Edit** dialog and enter the **values** as shown in the form **below**. Select **OK.**

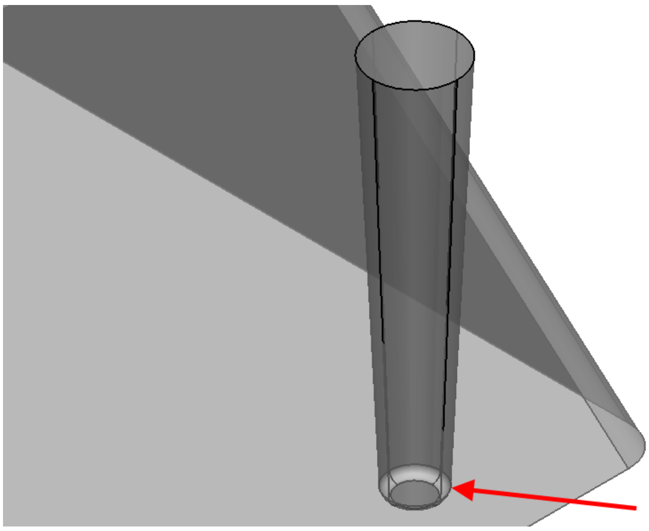




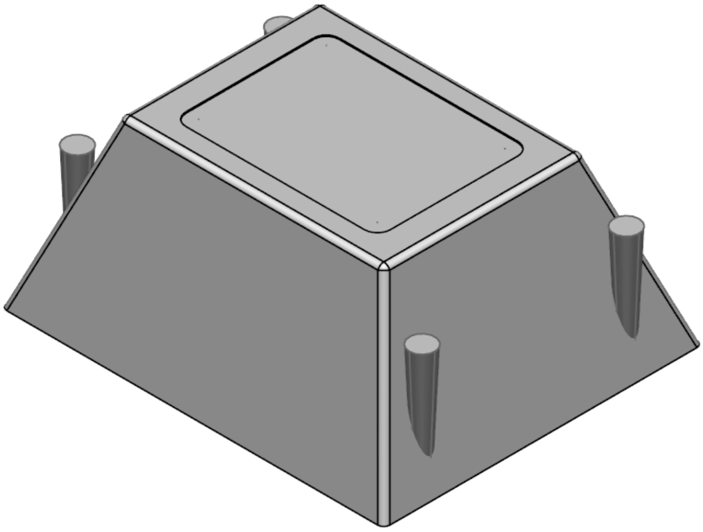
3 Make the new solid cone **active**.

4 Apply a **solid fillet of radius 2** around the **base** of the **cone**.

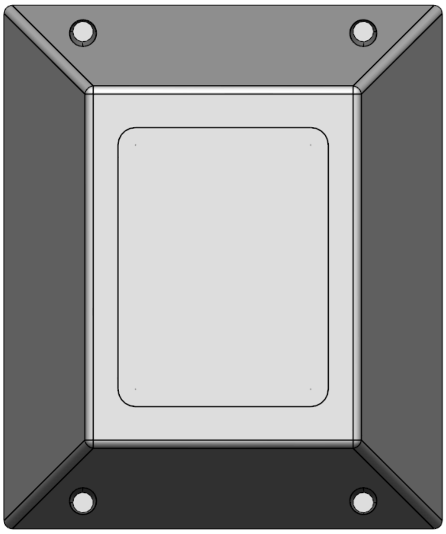
*You may need to use a **transparent shaded view**, or alternatively **Blank** (**Ctrl+J**) the **main block**.*



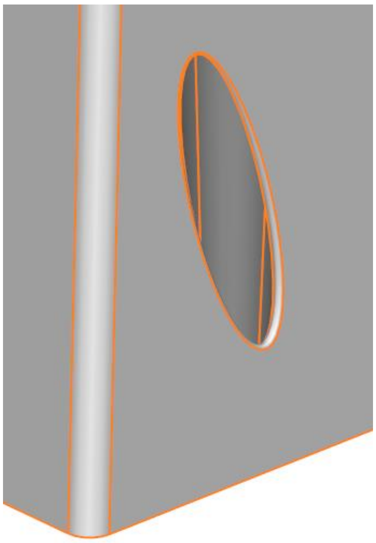
5 **Mirror** the **cone** across the **XZ** and **YZ planes** of the worklplane '**Datum'** to produce **4 cones**.



6 Use **Boolean Subtraction** to **subtract** the **4 cones** away from the **block**.



7 Create a **solid fillet** of **radius 1mm** around the **outer edge** of each **conical recess**.



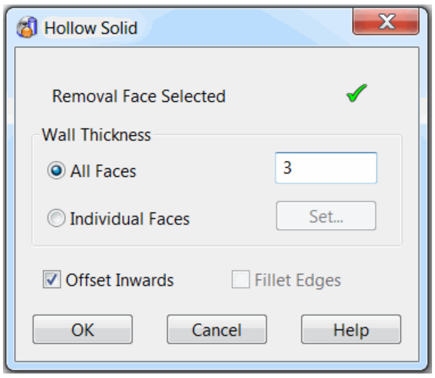
Hollowing out the base

We now need to **select** the **face** from which the **hollow solid** process is to occur. To do this, the **solid** must be **shaded** so that is possible to **select** the **base** as the **specified face**.

1 Select **Hollow Solid**  from the **Solid Features** toolbar. 

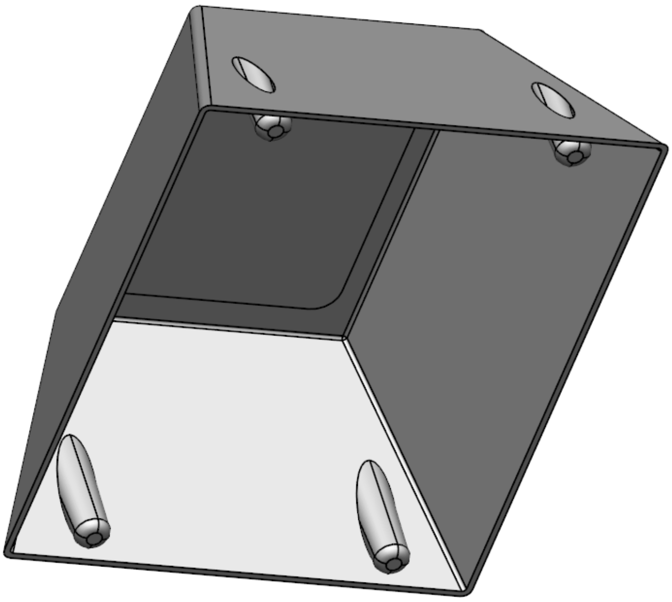
2 Select the **bottom face** of the **model** which will then be highlighted **yellow**.

The **red cross** adjacent to **Removal Face Selected** is replaced by a **green tick**.



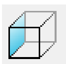
3 In the **All Faces** box, type **3mm**, and then select **OK**.

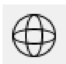
 *The **solid** now has a **uniform wall thickness** hollowed out from the **original base***



Creating the air vents

The **next stage** is to create **air vent features** on the **side walls**.

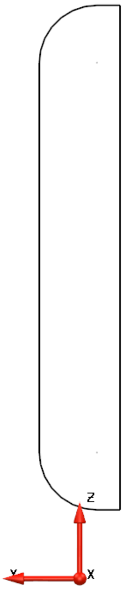
1 Select **View from left (+X)**  which **automatically** sets the **principal axis** to **X**.

2 Switch to **Wireframe** view. 

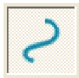
3 Create rectangle from **0 -7 12** to **0 14 88**.

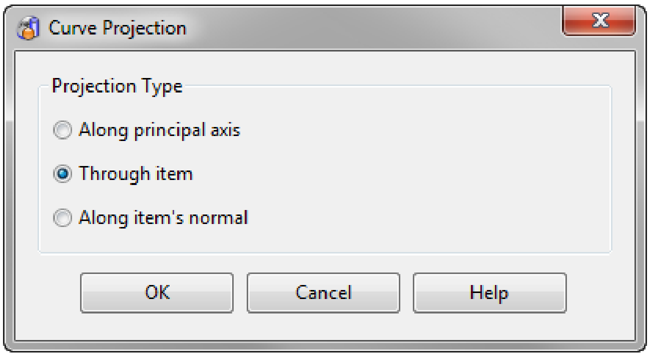
4 **Fillet**  both left corners with **radius 10mm**.

5 Create a **composite curve** of the profile.

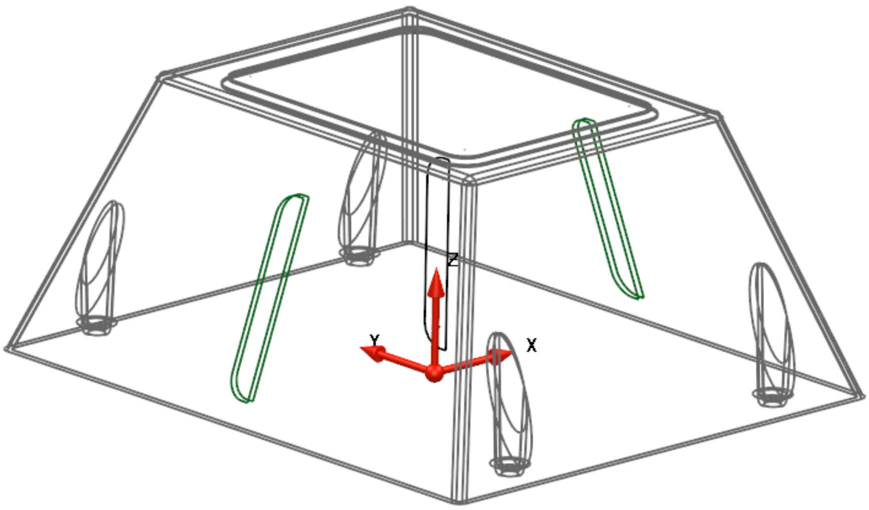


6 **Box-Select both** the **solid** and the **composite curve**.

7 Select **Curve Projection**  from the **Curve** menu. 



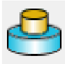
8 Select **Through item** then **OK**. The **composite curve** is projected through the **entire solid** in the **direction** of the **Principal Axis (X)**.

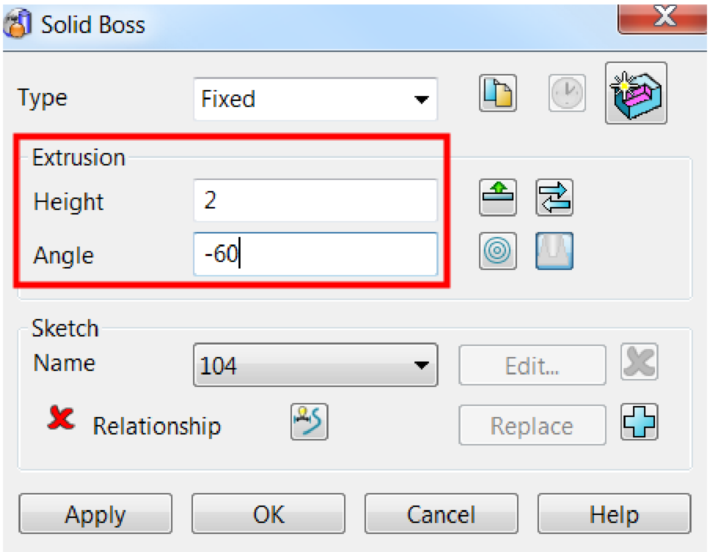


 ***Only** the **curves** on the **outside face** are required.*

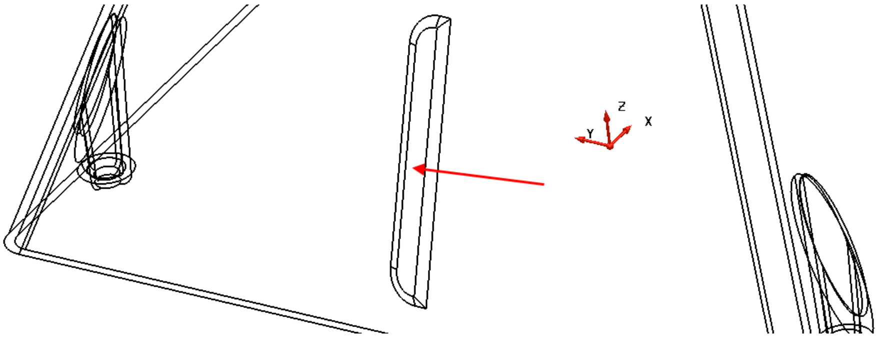
1 **Select** and **Delete** the **two curves** which were **projected** onto the **inner face** and the **original central curve**.

2 Select the **remaining outside curves**.

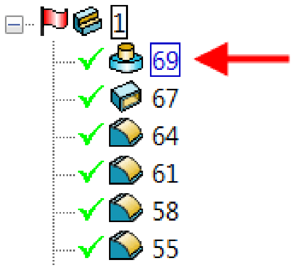
3 Select **Create solid boss**  from the **Solid Features** toolbar .



4 Change the **Extrusion height to 2mm** and **enter an angle** (**applies draft**) of - **60°**.

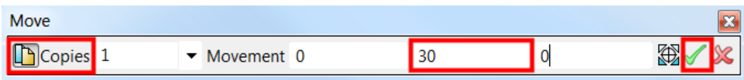


Select the **new boss feature** (created as one) from the **feature tree** as shown.

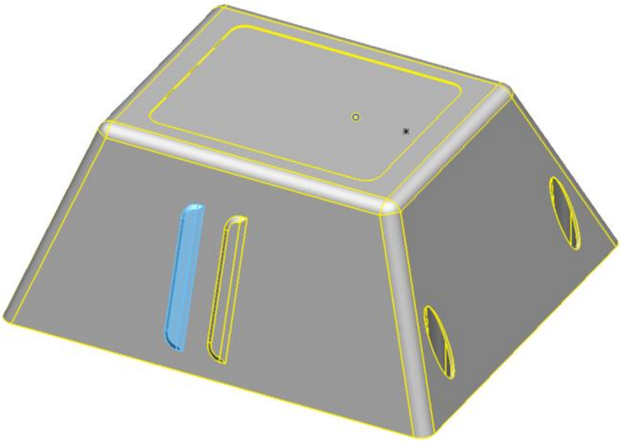


5 Select **Move**  from the **General Edits** toolbar. 

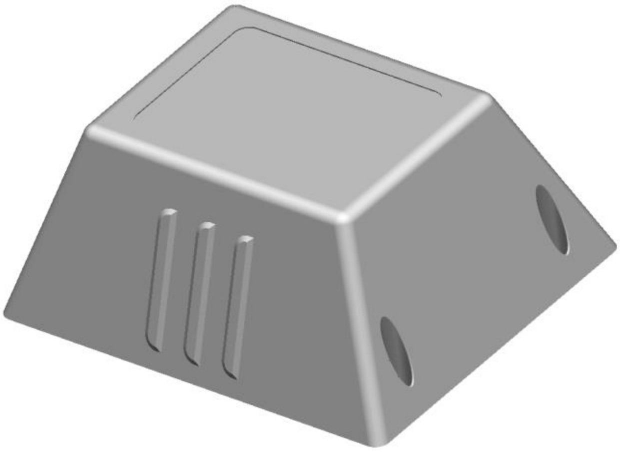
6 Select **Copy selected item** and enter **30mm** in the **Y input box**, then teh **green tick** to **confirm**.

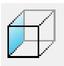


A **copy** of the **boss features** have been created to the side of the original.



7 With the **Move** dialog still **open** and the new **boss features highlighted** enter **-60** in the **Y input box** to create a **copy** on the **opposite side**.

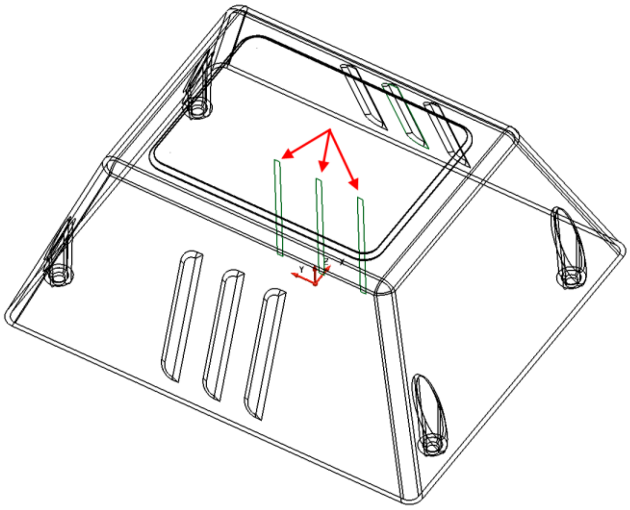


8 Select **View down from left (-X).** 

9 Create a **rectangle** of **lines**  at **0 -7 12** with sides **0 4 88**.

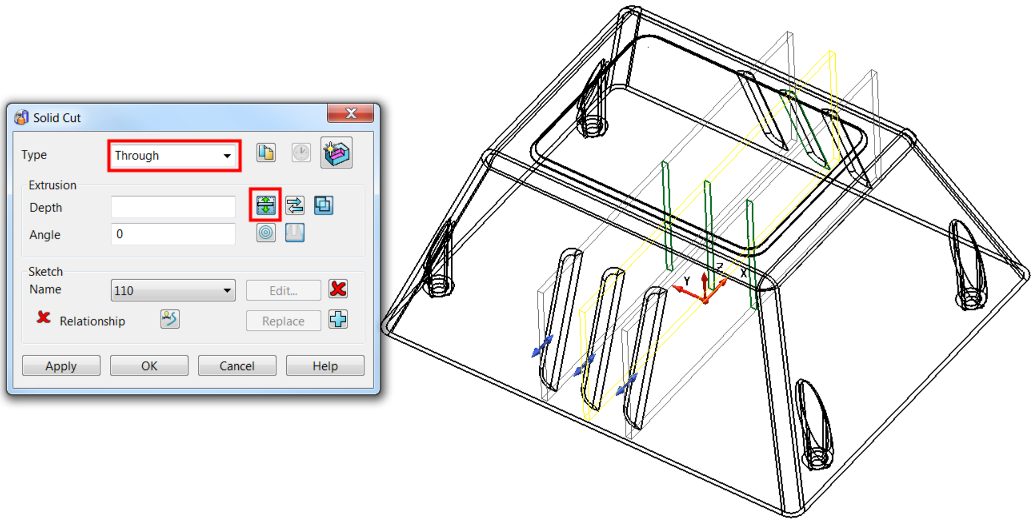
10 Create a **composite curve** from the **rectangle**.

11 Copy the **composite curve** by **30mm** along the **Y axis** on **both sides** of the **original**.

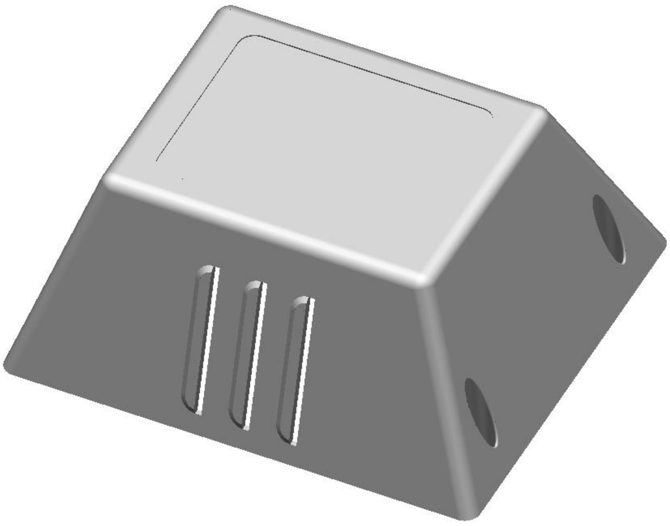


12 **Select all** 3 rectangular **composite curves**.

13 Select **Create Solid Cut**  from the **Solid Features** menu. 



14 In the **Type** drop-down list, select **Through** and select the icon highlighted to **Extrude both directions.** Select **OK**.



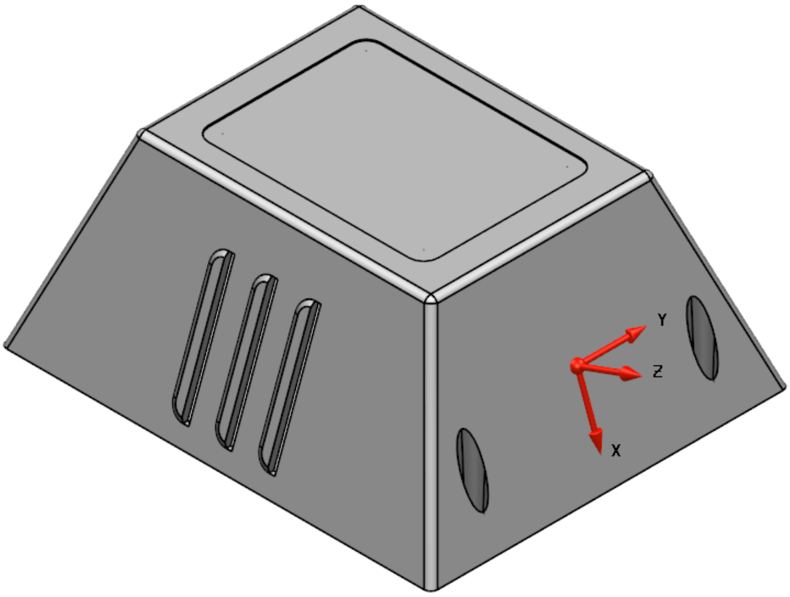
 *The **ventilation slots** are **cut** out of the **main solid**.*

Creating a workplane for the mounting

The **next stage** is to create a **mounting** for the **strobe light** on the **end face** of the alarm box. To **assist alignment** of the **mounting**, a **workplane** will be **created aligned normal** to the **face**.

1 Select the **Iso1** view. 

2 Select **Workplane Aligned to Geometry**  and select the **face centre keypoint** shown.



3 **If necessary**, **Twist** the workplane about **Z** so that its **X** axis is aligned towards the top of the alarm box.

The **Z axis** of the **new workplane** is **centrally aligned normal** to the selected **face**.

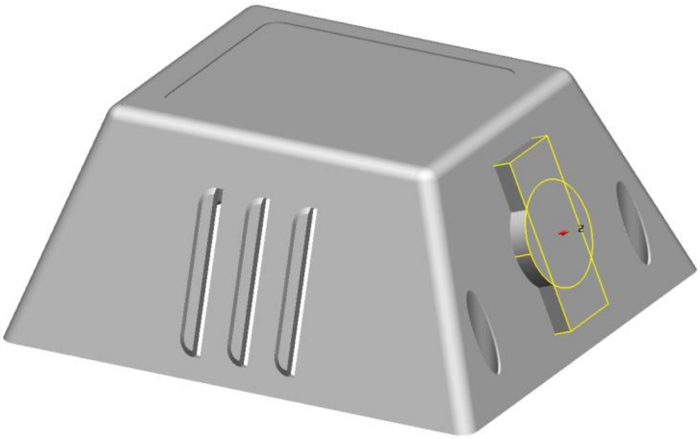
Creating a boss to provide access to the mounting

A **boss** complete with **through hole**, will be **created** and **aligned** to the **workplane** Z Axis to provide access for a **strobe light mounting**.

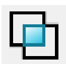
1 Generate a full arc (circle) of **Radius 30** at the workplane **0** and create a solid extrusion of 10mm in both directions (Direction 1 and 2)

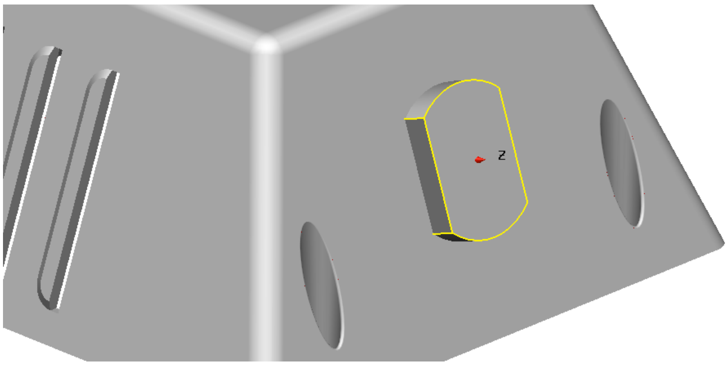
2 Generate a solid block primitive at **0 0 -10**

3 Modify the block dimensions to **Length 100**, **Width 40** and **Height 20**.



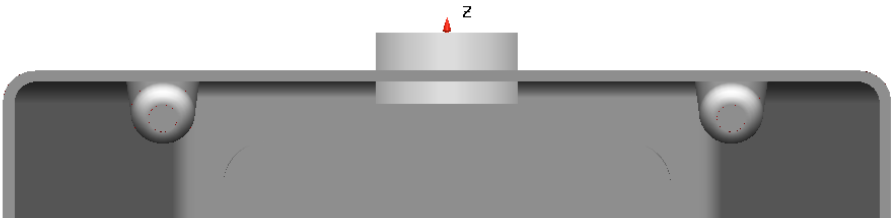
4 Make the circular extrusion **active** and **select** the rectangle block.

5 Boolean **Intersect**  the leave the combined solid.



Trimming the solid

As seen below, the new **solid** is jutting **beyond** the **inner wall** of the **main solid**.

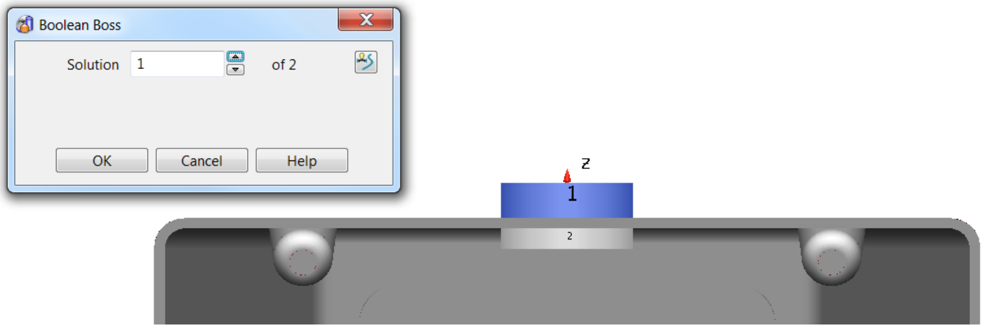


The **Solid Boolean Boss** will be used to **trim** it back to the **inside face** and at the same time add the outer part to the **main body solid** as a **Boss feature**.

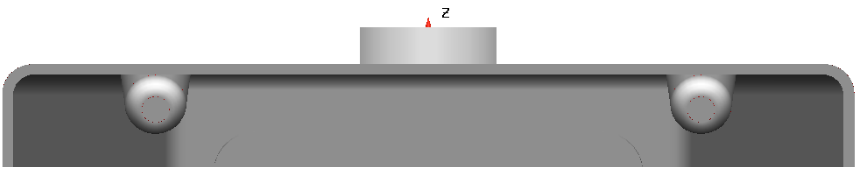
1 Select the **main casing** solid and make it **active.**

2 Select the intersected solid, and then select **Solid Boolean Boss**  from the **Solid Features** menu. 

3 The **Boolean Boss** dialog offers **2** possible solutions. It allows the **intersected feature** to be **split** to exist either on the **outside face** or on the **inside** of the **s.**



4 Toggle to the solution (in this case 1) with the **highlighted boss** (**blue**) on the outside and select **OK.**



The **solid boss** has been **trimmed** back.

Creating a hole for the wiring

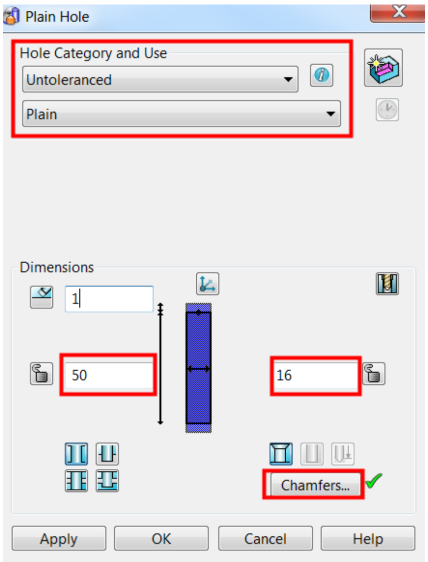
A hole for the wiring will now be created. With a single active solid, the hole will penetrate both the boss and casing.

1 Create a **Workplane Aligned to Geometry** and **snap** it to the **centre** of the **Boss face**.

2 Then select select the outer face of the boss to create a workplane at the **centre** of the **outside face**

3 Select **Create a Hole**  from the **Solid Features** toolbar. 

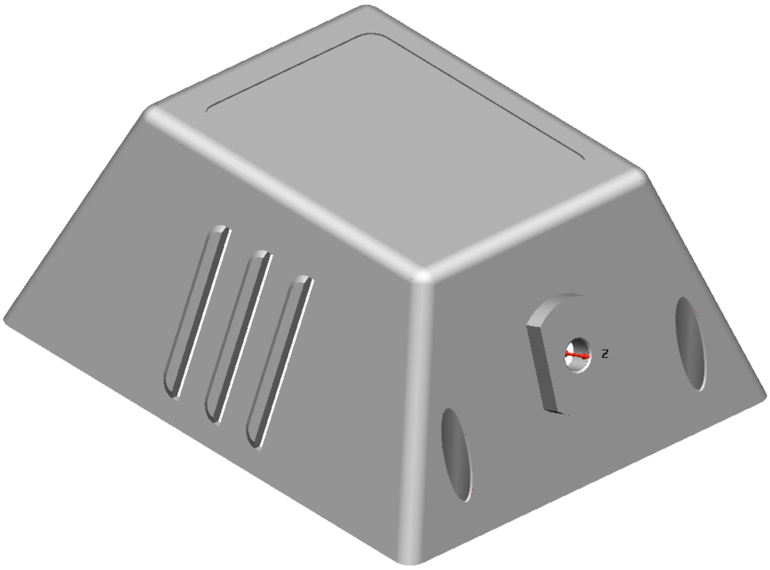
4 Position the hole by snapping to the **local** workplane, or by typing **0** in the **Command input** box.



5 For **Hole Category**, select **Untoleranced.**

6 For **Use** select **Plain.**

7 Enter a length of **50** and diameter of **16.** Select **OK.**



8 Select **File > Save As**:

.....*Training Data**Coursework**Alarm Box.psmodel*

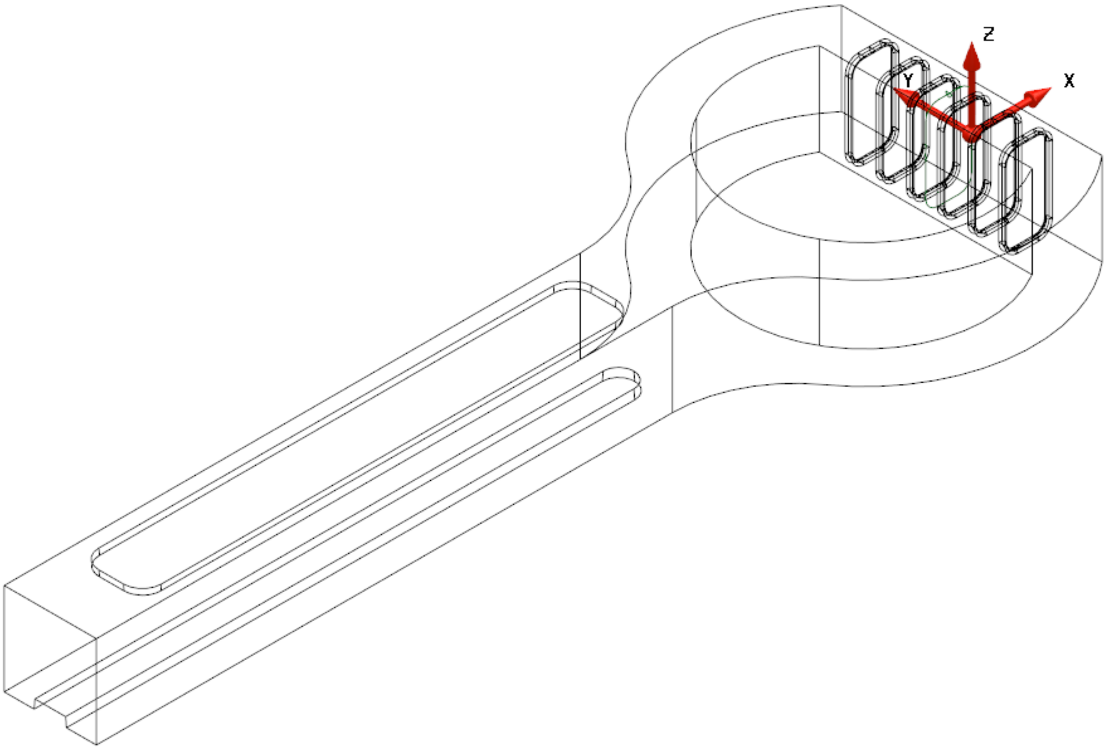
9 **Close** the model.

Plastic Handle Example

1 **Open** the model:

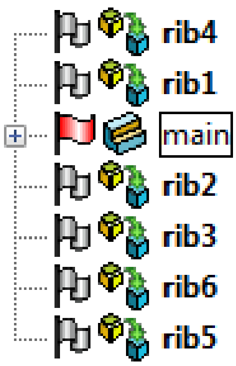
...**PowerSHAPE Data/toy_handle.psmodel**

The model contains a large **handle solid** along with **six small rib feature solids**, all of which are displayed in the **History Tree**. 



2 Select **File > Save As** and save the model to:

...**Training Data\Coursework\PlasticHandle.psmodel**



3 Make the **main handle** solid **active** in the **Solid History Tree**.

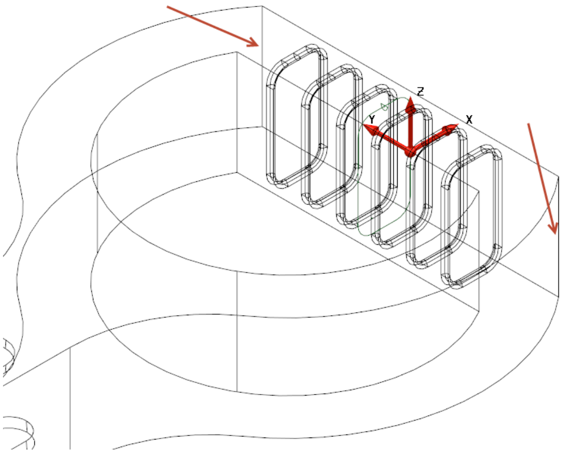
4 From the **Solid Features** menu , select **Solid Fillet** .

*The **Solid Fillet** dialog is displayed. Unlike surface filleting, **solid filleting** uses the **sharp edges** on the **solid** as the **filleting track**. The **order** and **extent** to which **solid fillets** are **created** is **important** if the correct design specification is to be achieved.*

In this case, to provide the correct shape around the **upper** and **lower edges** of the **D-shape**, the **vertical corner fillets** must be created **first**.

5 Set a **Radius** of **6mm** and select the **two outer vertical edges** shown.

*Hold down the **Ctrl** key to **multi select** edges.*

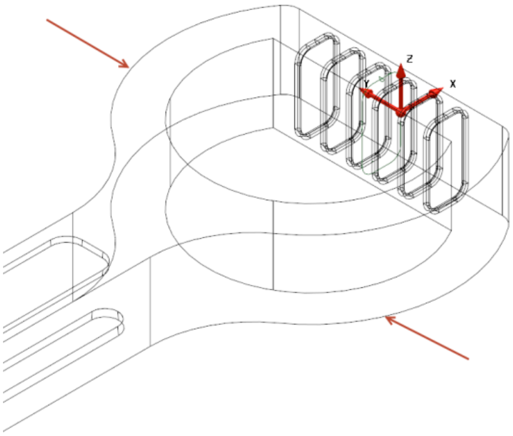


The **fillet path** runs until it finds a **sharp edge** (**Discontinuity**).

6 Select **Apply.**

*By selecting **Apply** the **fillet dialog** will stay open after the **fillet** has been created for **further operations**.*

7 Generate a radius **4mm** fillet along the **top** and **bottom faces** of the solid.



The **outer edges** are now **complete**.

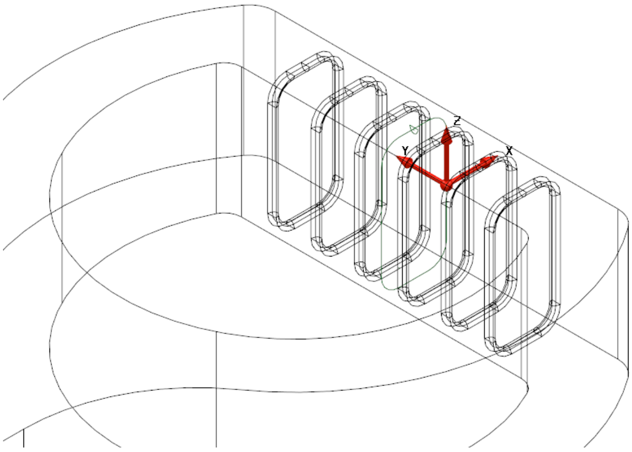
Modifying the D-shape

The **inner D-shape** of the handle will now be **modified** to include a **variable radius fillet** of **radius 4mm increasing** to **6mm** at the **mid-point** of the **curved section**.

8 **Zoom** into the **D-shape**.

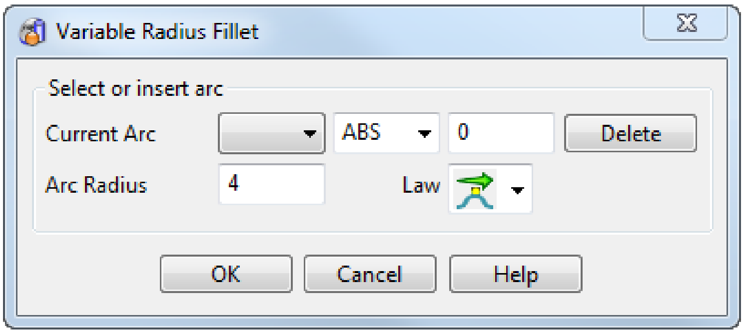
9 **Fillet** the **vertical internal corners** with a **radius** of **4mm**.

*Filletting these **vertical edges first** will provide one **continuous edge** around the entire **D-shape**.*



10 Select the **top inner edge**.

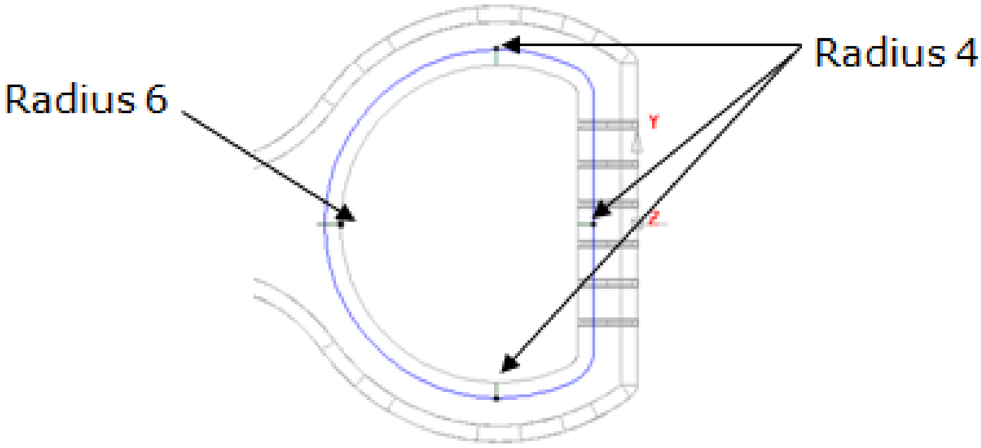
11 Select the **Advanced** option in the **Fillet** dialog to display the **Variable Radius Fillet** dialog.



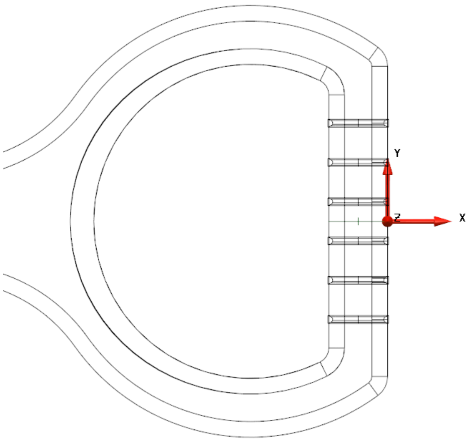
This **dialog** allows you to use the **mouse** to **select** an **arc** that represents the **radius of the fillet** at a location around the **curve** representing the shape. This works in a similar way to **variable radius filleting** of **surfaces** covered later. When you move the mouse along the track, the word ***Key*** appears. Click at these **points** to easily generate a **numbered arc**. The size of this **arc** can then be **stretched** or **modified** in the dialog.

12 Select a **View from Top** (**Ctrl+5**).

13 Click on the **track** to define locations for **radius 4mm** and radius **6mm**.

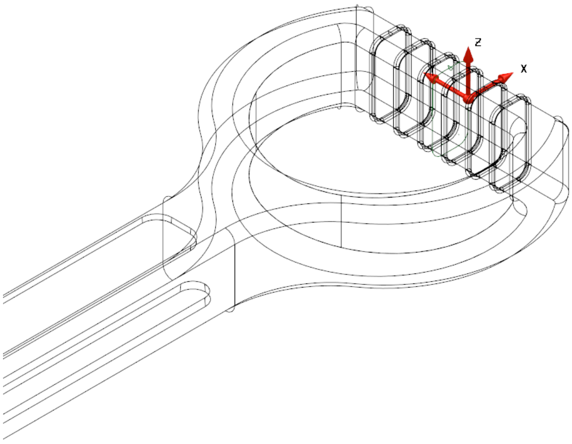


14 **Modify** the **arc radii** as indicated on the **image above** and select **OK** to produce the **variable radius fillet**.



15 Generate the **equivalent variable radius fillet** on the **underside** of the **solid**.

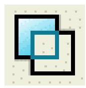
16 **Close** the **Fillet** dialog.

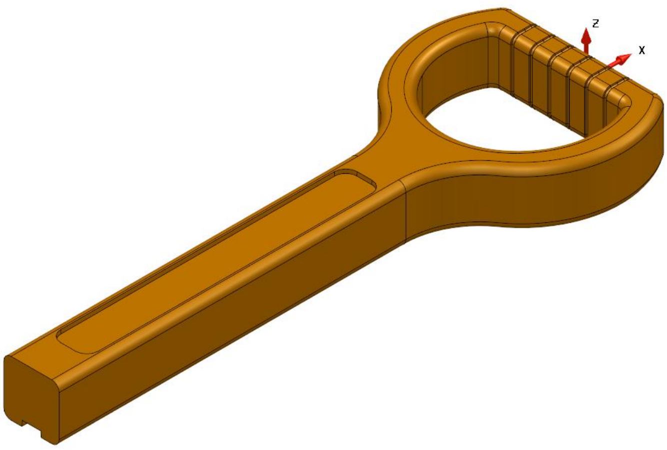


The **main fillets** have been **generated**.

Removing the solid ribs

The **solid ribs** can now be **removed** from the **main solid** to produce a grip effect on the **handle**.

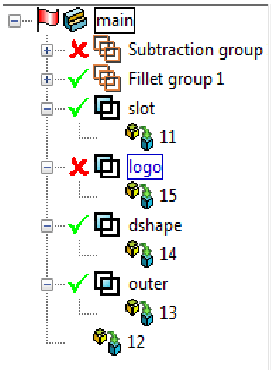
17 Select **ALL** the **solid ribs**, then navigate to **Boolean Subtraction**  from the  menu. The **completed outer model** is displayed.



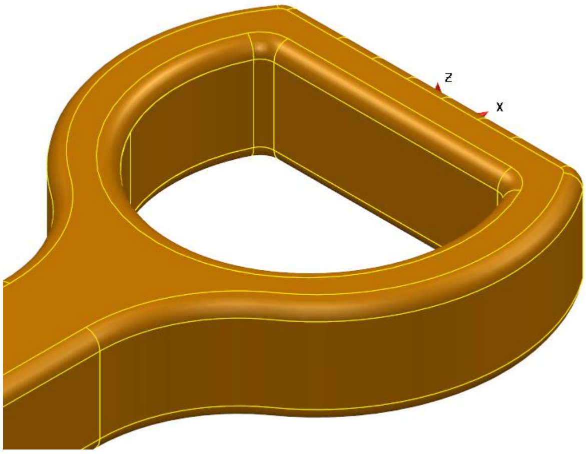
18 Select the **Solid** in the **graphics area**.

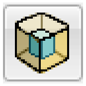
The outer must be cored with a **3 mm minimum wall thickness**. Due to **dimensional restrictions**, some of the **features** must be **suppressed** before attempting to create a **new solid** to represent the **inner**. It is also necessary to perform **additional material removal** on the **inner** along the **shaft** to **maintain** the **nominal wall thickness**.

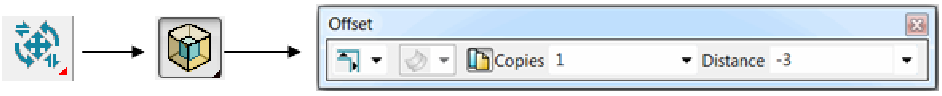
In the **History Tree** click the **green ticks** next to all **six ribs** and the **logo feature**.



*The **ribs** and **logo recess** are **temporarily removed** from the **main handle solid**.*



19 From the **General Edits** menu, **Offset**  a copy of the **selected solid** by **-3mm** to provide the **inner wall** *(excluding the ribs and logo detail).*



Now that the **basic model** for the **inner solid** has been created, the original **outer solid** must be **updated** to include the **rib** and **logo features** again.

20 In the **History Tree** click on all **red crosses** adjacent to the **suppressed features** to **reinstate** them onto the **outer solid**.

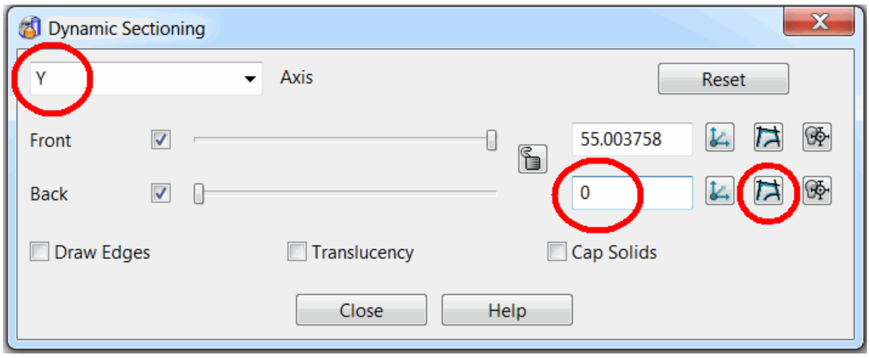
Changes to the inner core solid

Some **design modifications** are now **required** on the **inner core solid**. These include **extending** the **shaft** through the **end** of the **outer solid** and **creating** a **minimum wall thickness** of **3mm** below the **logo recess**. To show these requirements more clearly the **Dynamic Sectioning** option of **Model Analysis** will be used.

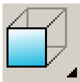
21 Select both the **inner** and **outer solid**.

22 Select **Model Analysis**  then select **Dynamic Sectioning** .

23 In the **Dynamic Sectioning** dialog, select **Y** in the **Axis** list.



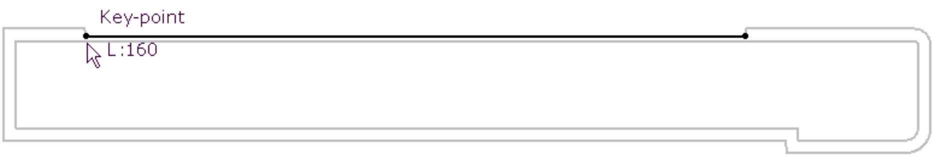
24 Enter **0** to position the **Back face** slider halfway across model, and then select the **Create Wireframe** button as indicated **above**. Select **Close**.

25 Select **View from Front** (**Ctrl+2**)  and set the **Prinicipal Axis** as **Y**.

26 **Quick select ALL wireframe**  and select **Blank Except (Ctrl+K)**.

27 Check the wireframe for **acceptable wall thickness** and then **select** the **two composite curves** defining the section along the shaft centre.

*As you should hopefully see at the **logo recess** the **minimum wall thickness** his **reduced** to **1mm** due to the **depth** of the **recess**.*

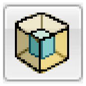


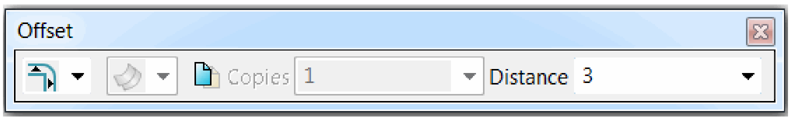
28 **Snap** a **wireframe line** to the outer **composite curve** as shown above, then **edit** the **length** to pass through the **end wall** (for example, increase the **length** to **200mm**).

29 From the **right hand end** of the existing line create another line of suitable length (for example **15mm**), at angle of **30°** as shown above to **create** a **taper internally** and **avoid sharp edges**.

30 **Select** and **Blank** (**Ctrl+J**) the **two composite curves**.

31 Create a **composite curve** along both **lines**.

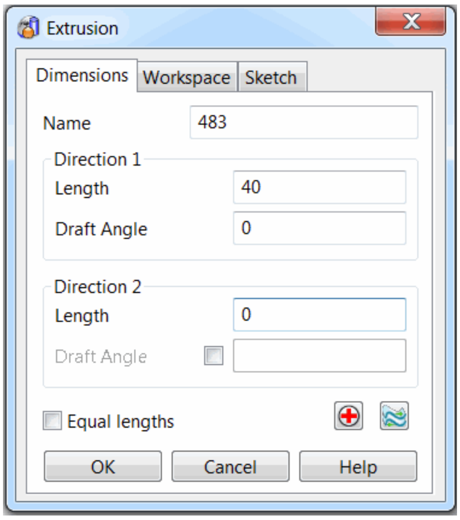
32 From the **General Edits**  toolbar, select **Offset** . Specify **Round discontinuities**, **No. of Copies 1**, and **Distance 3mm**, then press the **Enter** key.

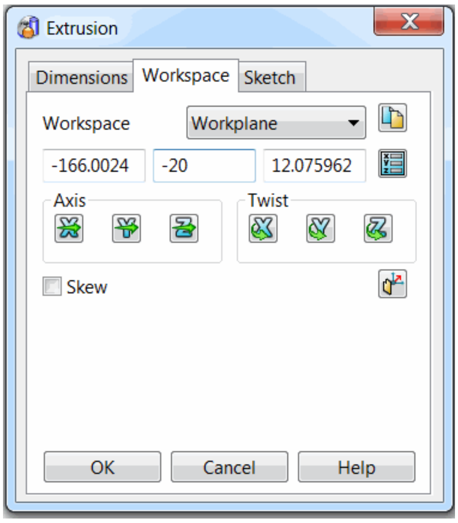


33 Select the new **offset curve**, create an **extrusion surface**  from the Surface  menu of **Length 40mm** (**along Y**).

34 Double click on the **Surface Extrusion** to open the **extrusion edit** dialog.

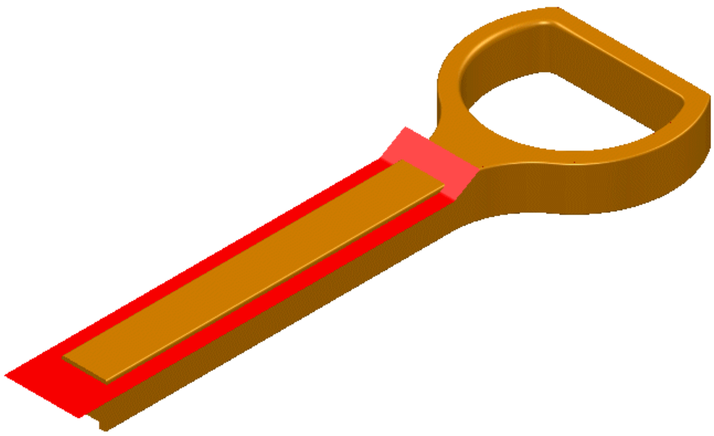
35 Select the **Workspace** tab and enter a **Y coordinate** value of **-20** to **centralise** the **surface** across the shaft.





36 **Unblank** (**Ctrl+L**) the **solids**.

37 **Blank** (**Ctrl+J**) the **Main solid**.

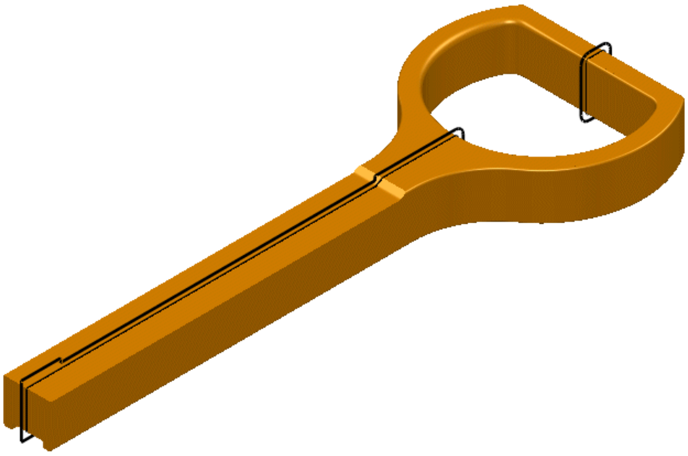


38 Make the **inner-core solid active**, and **select** the new **extrusion surface**.

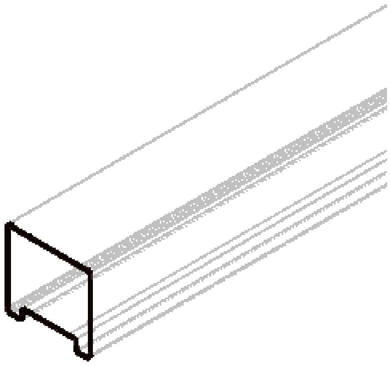
*This may be easiest in the **Solid History Tree**.*

39 From the **Feature** toolbar, select **Boolean Subtracction** to remove everything to the **red side** of the **surface** from the **solid**.

*The **core solid** illustrated **below** also includes the **revised wireframe** obtained from a further **dynamic sectioning operation** to show the new wall thickness.*

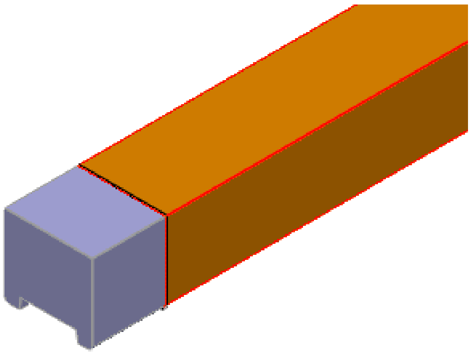


40 Create a **composite curve** around the **end** of the **core solid shaft**.

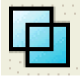


41 Set the **Principal Axis** to **X** and **select** the new **composite curve**.

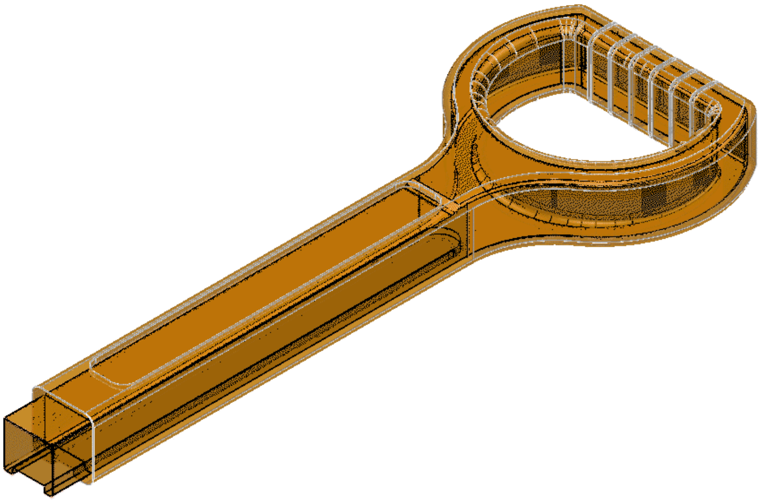
42 Create a **solid extrusion** and drag it along the **-X** direction until it has passed through the **end** of the **outer solid** (for example. **Length 20mm**).



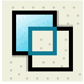
43 Make the main **inner core solid active**.

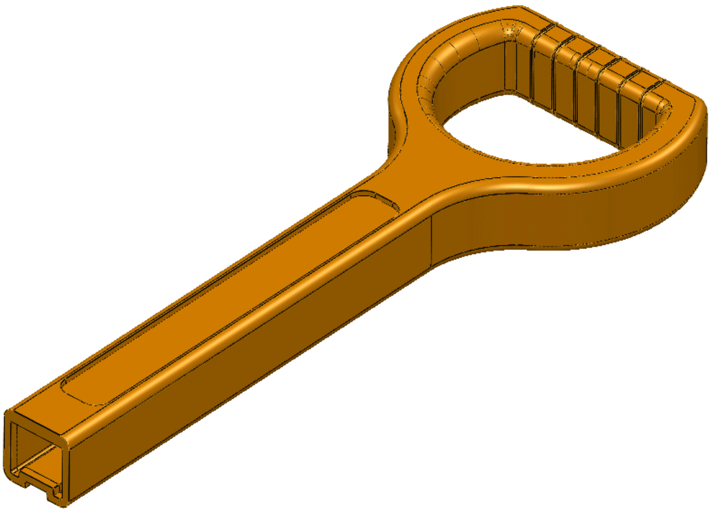
44 **Select** the new **solid extrusion** and, from the **Feature**  toolbar, select **Boolean Addition** .

*You may find that **PowerSHAPE** will not initially add the **extrusion** to the **inner core solid**. This may be because the extrusion is of **Version 8** format (mainly found in older **PowerSHAPE** sessions). To fix this select the extrusion a from the **Solid Edit** toolbar, click on the **Version 8 Converter** icon.* 



45 Make the **outer solid active**.

46 Select the **inner core solid** and, from the **Feature**  toolbar, select **Boolean Remove** .



The completed solid spade handle component is a shown above.

47 Select **File > Save** to update the content of the stored model file.

48 **Close** the **model**.

Radial Solid Cut

The **Radial Solid Cut** feature was new in **PowerSHAPE 2014 R2** (available in **PowerSHAPE Pro**, **Toolmaker Pro**, **PartMaker Modelling** and **PowerSHAPE Companion** for FeatureCAM). It allows users to quickly **wrap wireframe** around **cylindrical solid objects** and **cutout** a **pocket feature**. Use this option to create a **pocket** that can be milled using a **turn-mill** machine.

This is complete without any complex calculations by effectively **unwrapping** the selected face of the solid into a **sketch plane**, allowing **wireframe** to be created and then **wrapped** onto the **solid face**. This example will highlight the method used to create a **Radial Solid Cut** feature.

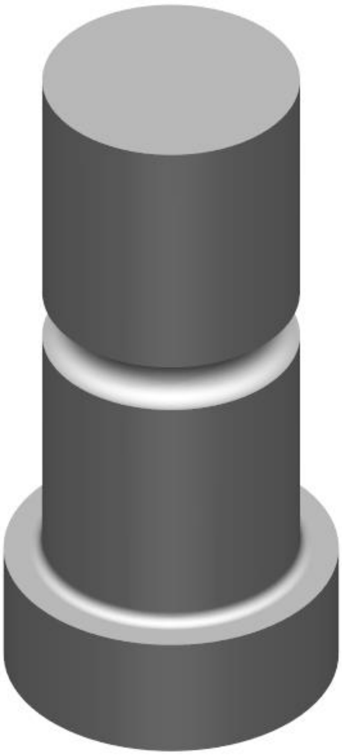
Radial Pocket Example

49 Open a **New Model**.

50 Create a **Workplane** at **0**.

51 **Import** the file **Radial Solid Cut.x_t** from the data folder.

52 Select an **Isometric View** (**Ctrl + 1**).



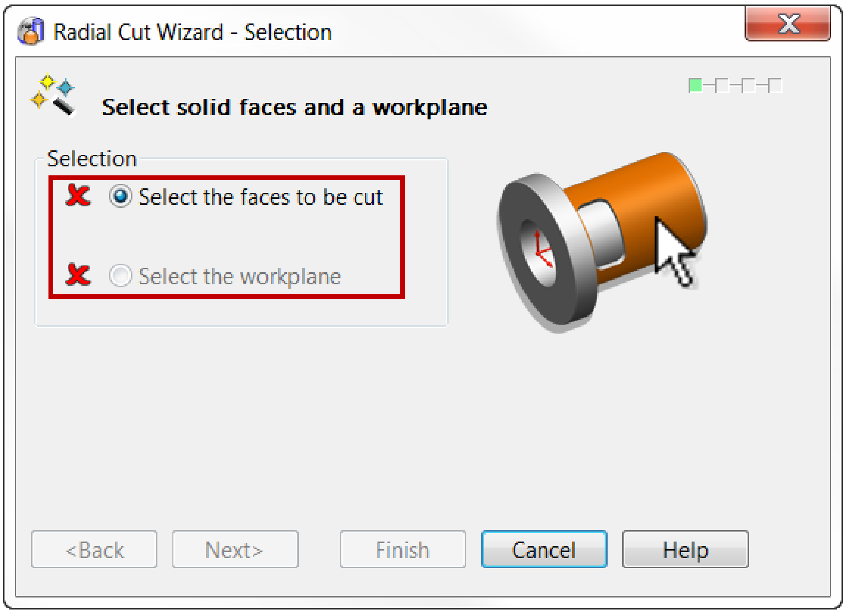
53 Ensure the **solid** is **selected** by clicking on it with the **graphics window** or in the **solid history tree**.

*The **Radial Solid Cut** feature works on a **selected** solid, however it is good practice to have this as the **Active** solid when you are editing it. The selected solid is represented by the **yellow** outlining of the solid.*

54 Click the **Solid Feature**  button to activate the associated **Sub-Menu** for solid editing.

55 Select the **Radial Solid Cut**  button from the **Solid Feature** menu. This is located in the **solid cut flyout**.

This will open the **Radial Solid Cut** wizard which will guide you through a series of step by step windows enabling you to quickly and easily create a **solid cut**.



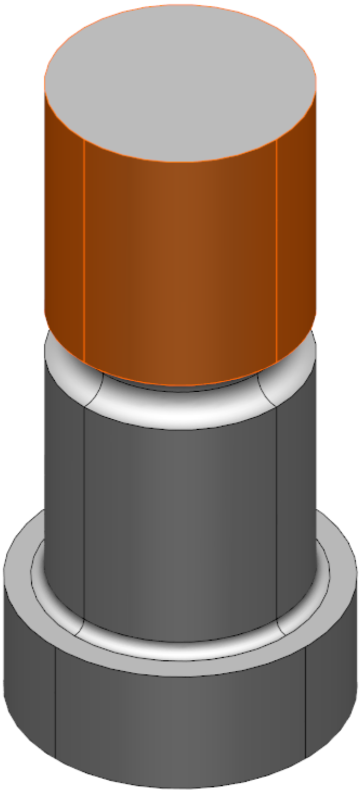
The first window of the wizard requires a selection of the face(s) to be cut. Currently the wizard is best suited to cut separate faces at a time but can handle multiple cuts.

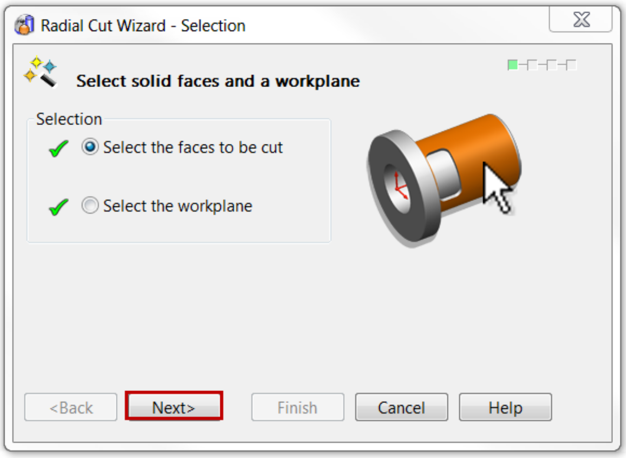
To make a **selection** simply **click** your chosen **face(s)** in the **graphics window**, these will be highlighted in orange. In this first example we will create a **rectangular pocket** on the **uppermost cylindrical face** of the solid.

56 Select the **Upper Cylindrical Face** on the solid as shown in the image on the next page.

57 The **two** green ticks in the dialog box denotes when the selection is sufficient to move on to the next window. Note: The workplane will automatically be selected as the **Active Workplane**.

*The **Z Axis** of the **Active/Selected Workplane** needs to be **parallel** to the axis of the cylindrical face(s) to be cut.*

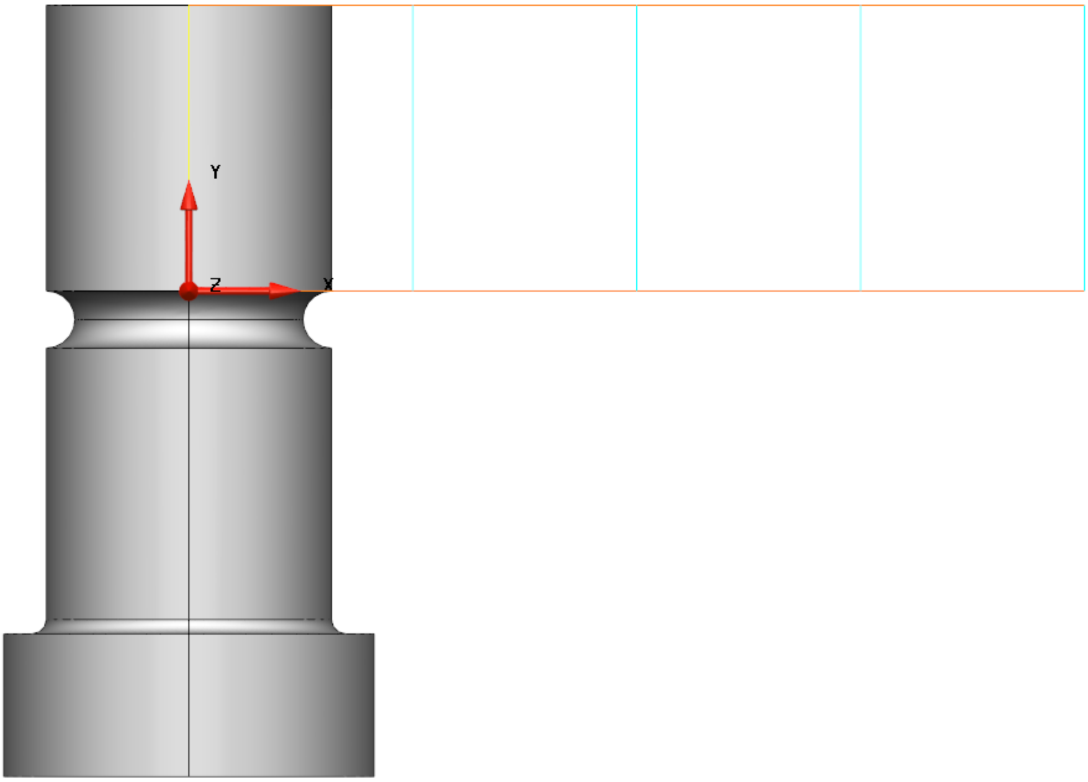




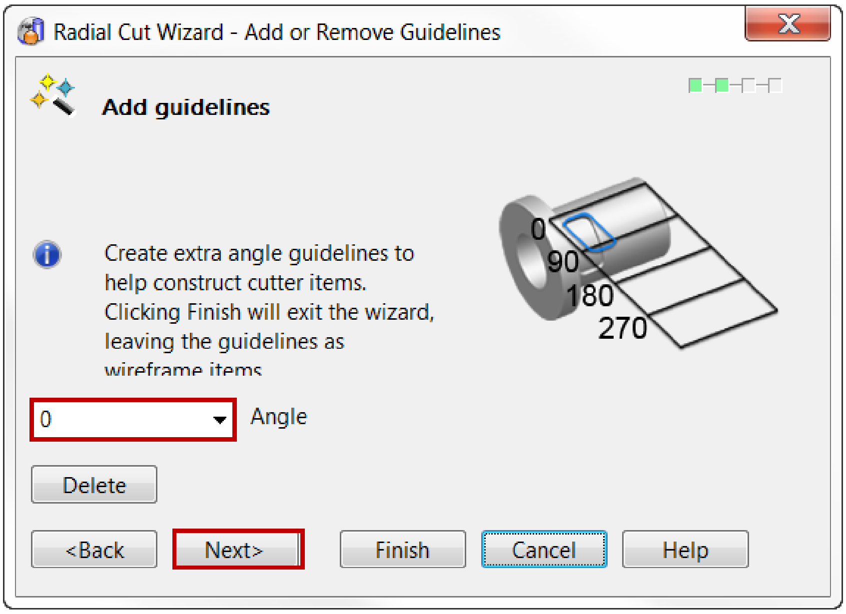
PowerSHAPE now unwraps the selected face of the solid into a plane. Its upper and lower edges outlined by **horizontal lines**, while also creating guidelines at 0, **90°**, **180°**, **270°** and **360°** (same as 0° start and end) to help visualise the distance **around** the face.

58 Select **Next**.

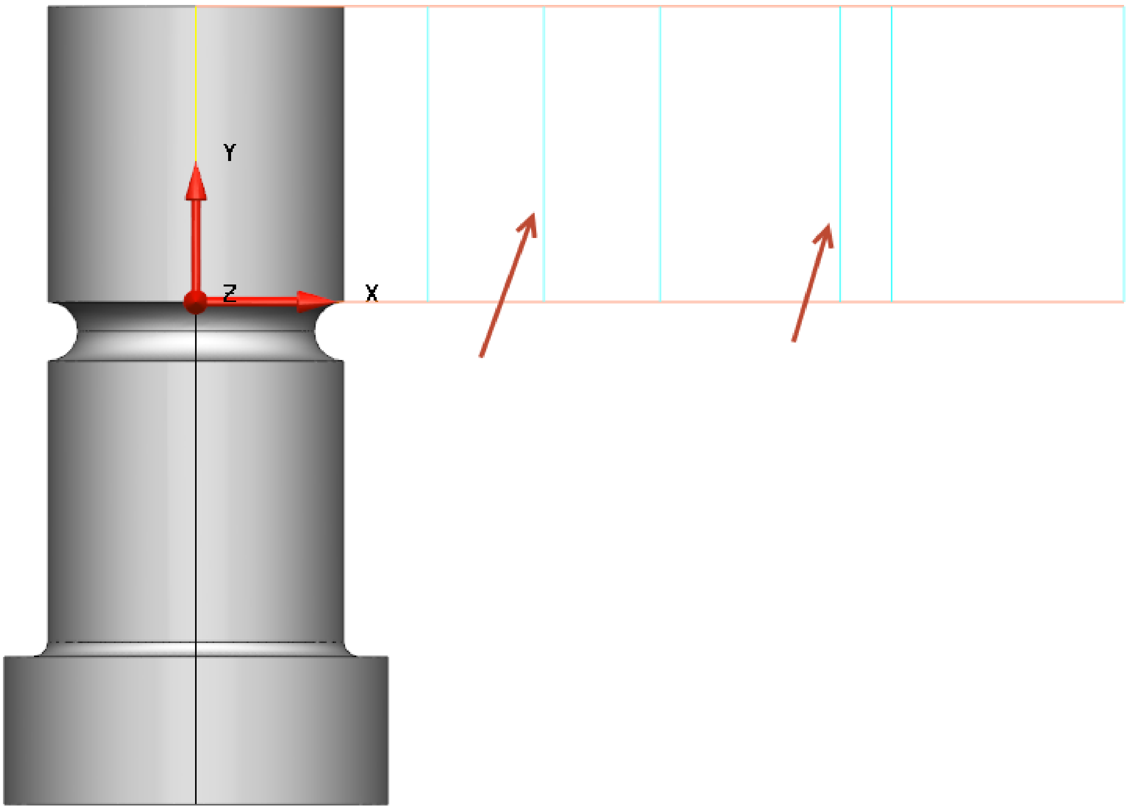
59 Select a **View from Top** (**Ctrl + 5**). While in the wizard this will view **down** on the **unwrapped face** usng a temporary workplane **normal** the the **sketch plane**.



To further help position **wireframe** around the face of the solid, it is also possible to add further **guidelines** at **any angle**. Doing this will simply add extra vertical lines onto the **sketch plane** which can be used to snap to with the intelligent cursor.

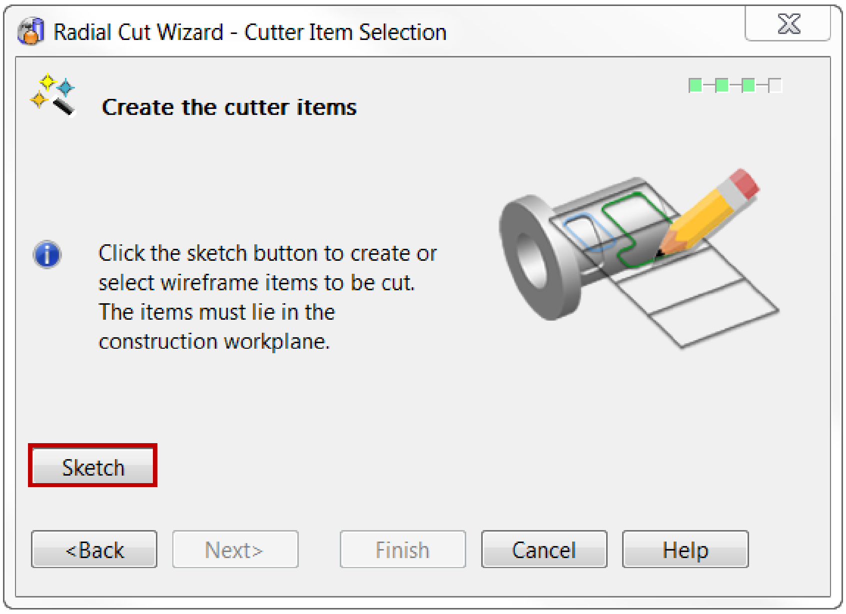


60 Within the wizard **add** guidelines at **135°** and **250°** by typing into the provided entry box and pressing the **Enter** key to **accept** each angle. **Two** new guidelines should appear in the view.



Note: *Selecting* ***Finish*** *in the wizard at this point will simply create the* ***guidelines*** *as wireframe allowing the user to create the wireframe at a later stage and re-enter the wizard to create the cut. However* ***in this example*** *we will use the wizard throughout to create a* ***Radial Cut****.*

61 Click **Next**.



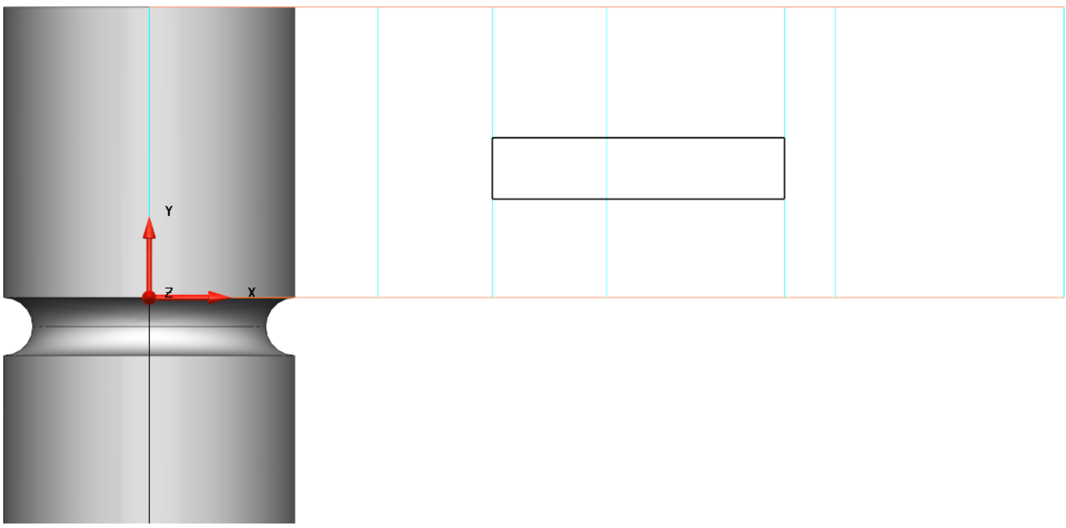
62 Select **Sketch** to **hide** the main wizard **dialog** and enter the **sketch mode**.

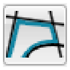
*As noted above this will **hide** the main dialog to enable **unobstructed sketching** in the **graphics window**. Reducing down to a **Save/Dismiss Changes** sketch dialog seen below.*

63 By snapping to the **135°** and **250° guidelines** created, draw a rectangle of **approximate height 4mm** near to the centre of the solid face. The **rectangle**  button from the **line menu**  is the most effective way of doing this.

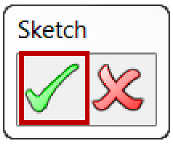
*The **angle** is measured in the **XY plane** from the **positive X axis** in a **counter clockwise** direction.*

*At this stage **don't** focus on accuracy too much, as the **process** to create a **radial cut** using the wizard is the **main aim** of this **first example**. Although snapping the the correct guideline should give a length of **20.07mm**.*

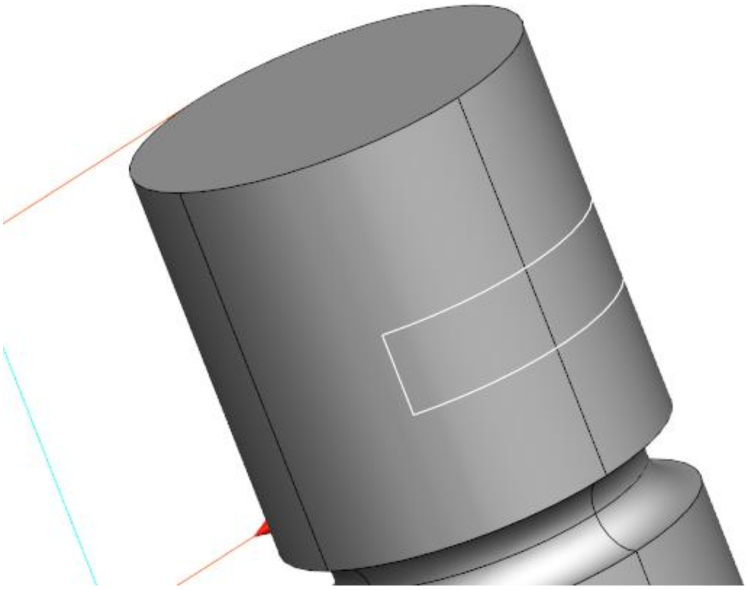


64 Create a **composite curve** from the **rectangle**. As with other solid feature function the use of composite curves is required to perform the function. The **Composite curve creator**  from the **Curves** menu  is required. Trying to use the **ALT + Click** shortcut in this case will likely fail in this case.

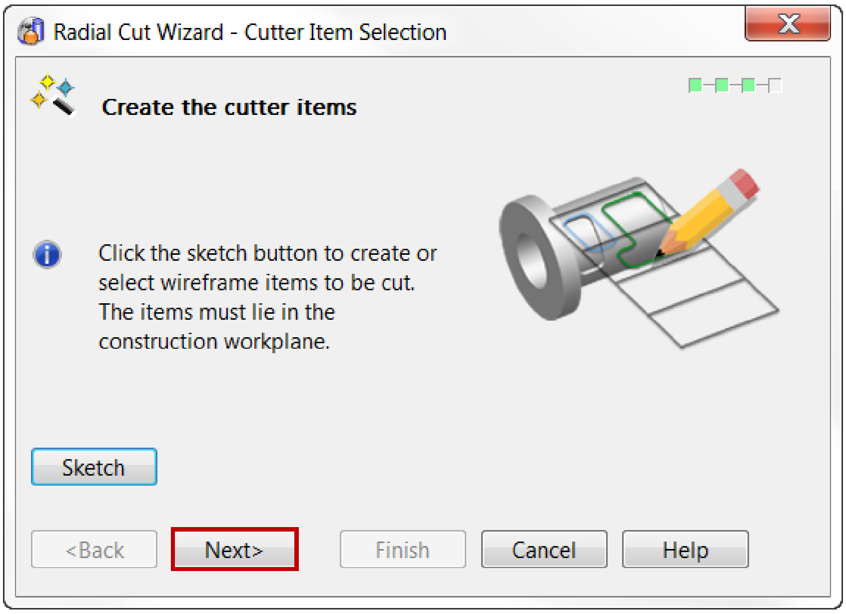
65 Once you are happy with the **shape** and **position** of you **composite curve**, **SELECT all required curve(s) that you wish to wrap/cut** and click the **green tick** to accept the changes and **re-enter** the **wizard**.

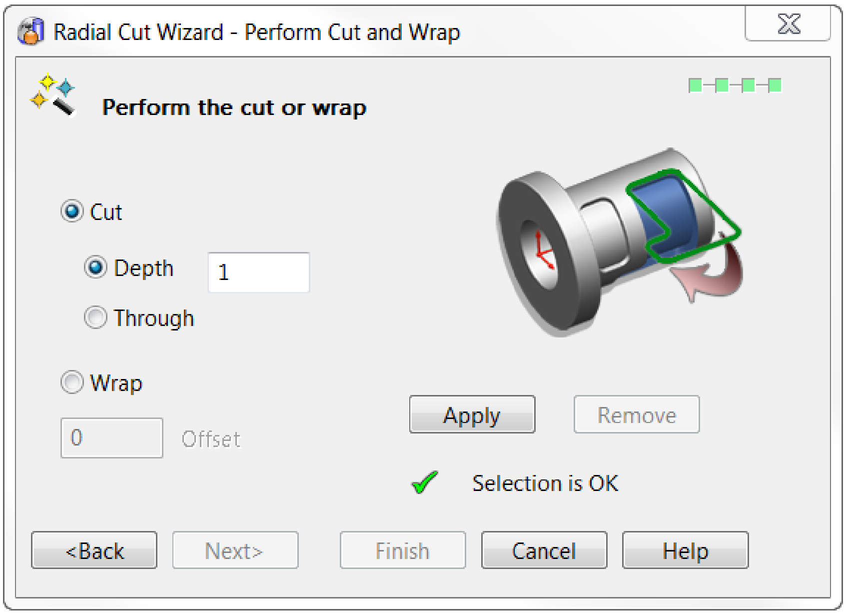


Accepting this form will also create a view of the **wrapped curve(s)** onto the **model** to show there **position** and **size** in direct relation to the solid model.



66 Select **Next** within the dialog.





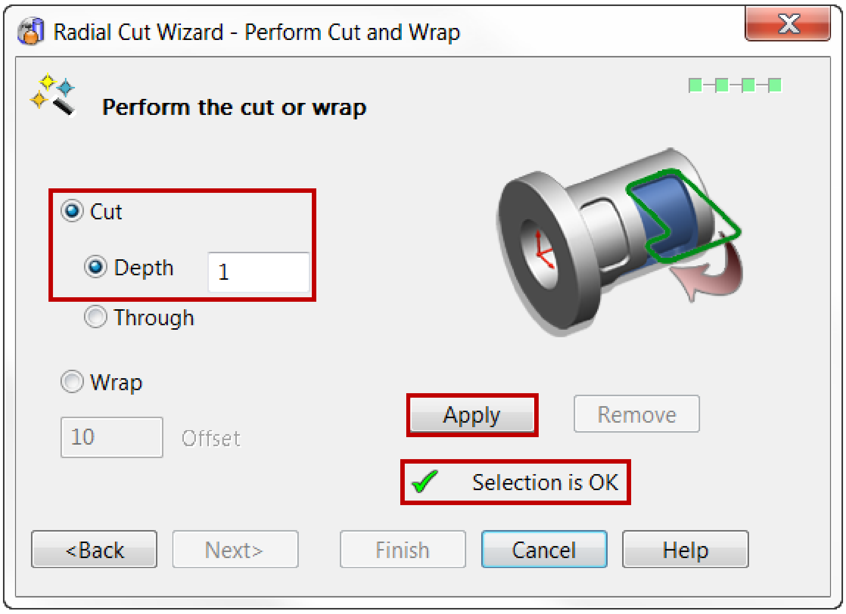
67 Within the **final window** of the **wizard** there are **two** main options; **Cut** or **Wrap**.

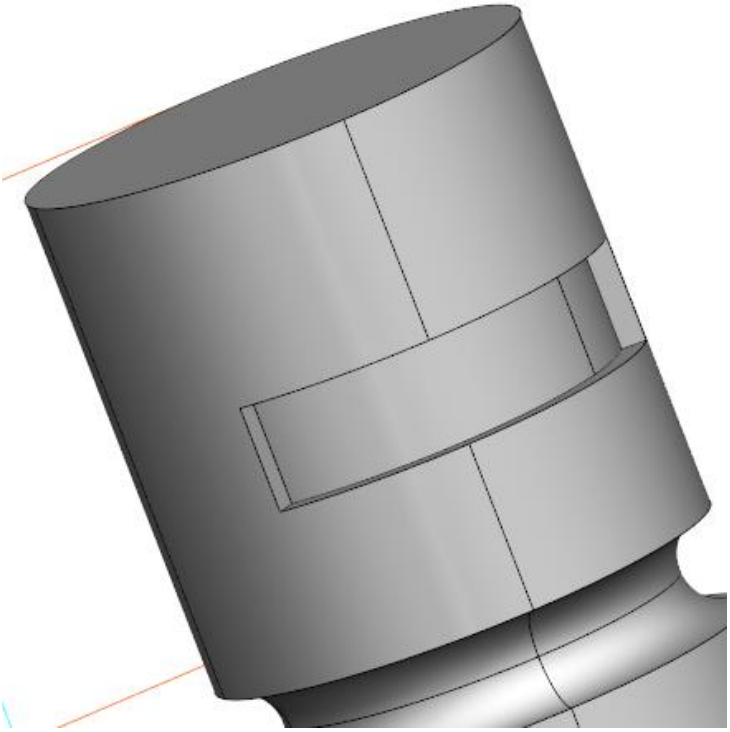
- **Wrap** will simply create a **wrapped curve** onto the **face** of the **solid**, while the **Offset** option allows the curve to be projected at a distance away from the **face** of the **solid**.
- **Cut** will apply the **wrap** and make a **cut** in to the **solid** at **any** specified **depth**, or **through**. It is important to note that this cut is normal to the face of the solid.

68 Select **Cut** and a **Depth** of **1mm**, then select **Apply**.

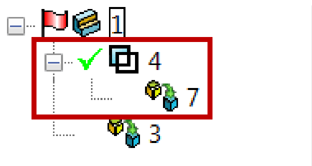
*If after you select **Apply**, changes are required and you are still in the wizard the **Remove** button will delete the current cut, allowing edits to be made.*

*A **Green Tick** will notify you that the current selection is **OK** and ready to be cut.*





69 As you can see the **Cut** has been performed successfully, Select **Finish**.



Looking to the **left** at the **Solid History Tree** we can see that this feature has been created by effectively creating a **solid** from the **wrapped curve** and the using a **Boolean Remove** function to create the cut.

The **curve** that was created on the unwrapped solid face in the wizard is also kept after the operation. This can be **deleted** altogether, or you could put it on a separate **construction level**.

*As this is a solid the cut faces are easily edited using further **Solid Features** like **solid fillet** and **wall draft**.*

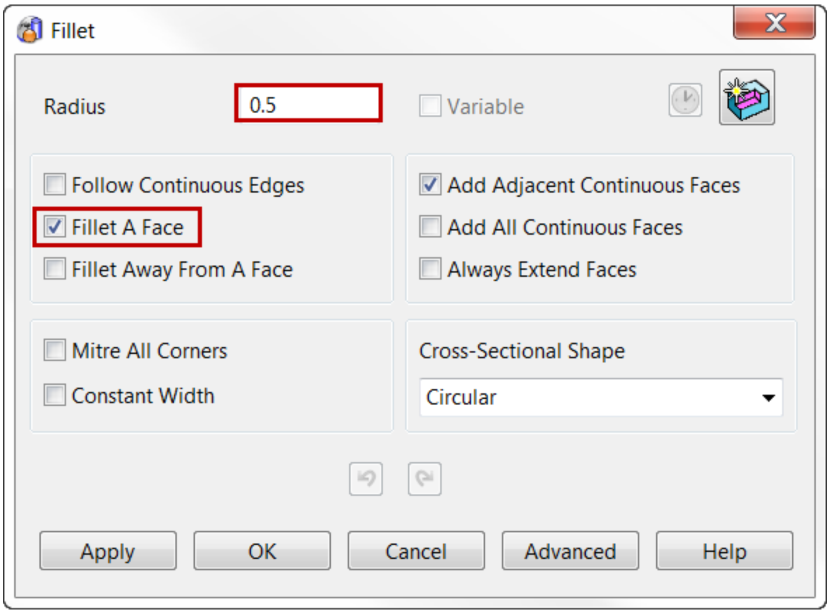
70 Ensure the **solid** is **active** which is denoted by the **red flag** next to the solid in the **solid history tree**.

71 From within the **Solid Feature**  sub-menu select the **Solid Fillet** .

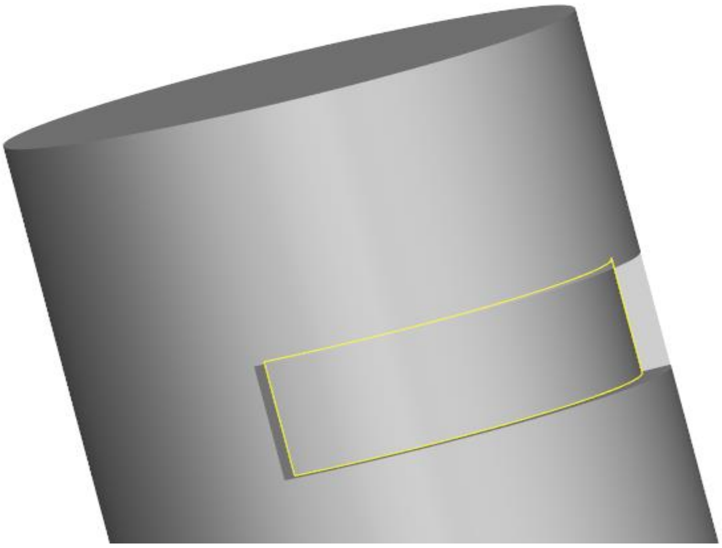
72 Enter a **Radius** of **0.5mm** and select the option to **Fillet a Face**.

*The **Fillet a Face** allows you to select faces of the solid and automatically select and fillet **all** the face edges to save time multi-selecting faces.*

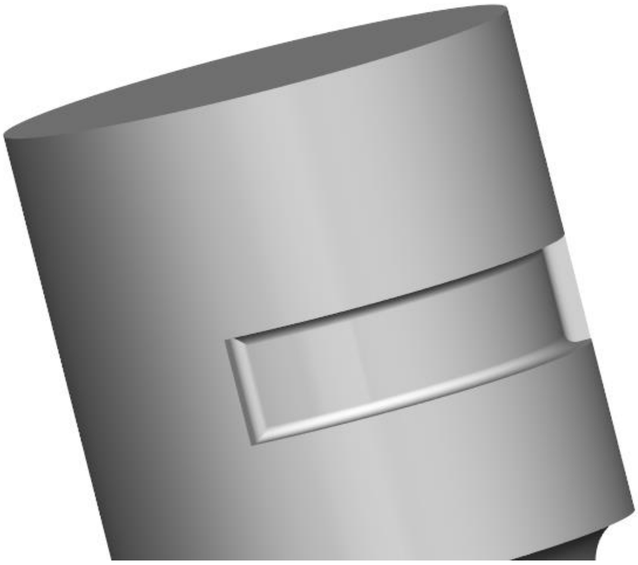
*Currently **solid edits** to the **radial cut feature** are only supported **after** the feature has been **created** and once the **wizard** has been **finished.***



73 Select the **face** at the bottom of the **radial cut pocket**.



74 Click **Apply** then **OK** to accept and save the changes and **create** the **fillet**.



Solid Base Groove

Next we will create a complex **radial groove** round the **lower cylindrical face** of the **solid**. We can easily create a simple groove using a variety a solid feature methods, however for the groove to take a **more complex path** the **Radial Solid Cut** method **simplifies** this process.

75 Ensure the **Active Workplane** has its **Z Axis parallel** to the **axis** of the **cylindrical face** you plan to cut using the **Radial Solid Cut** feature.

*This should already be the in place from the **previous example**, however it is **good practice** to develop a process of checking everything is in place **before** you start the feature wizard.*

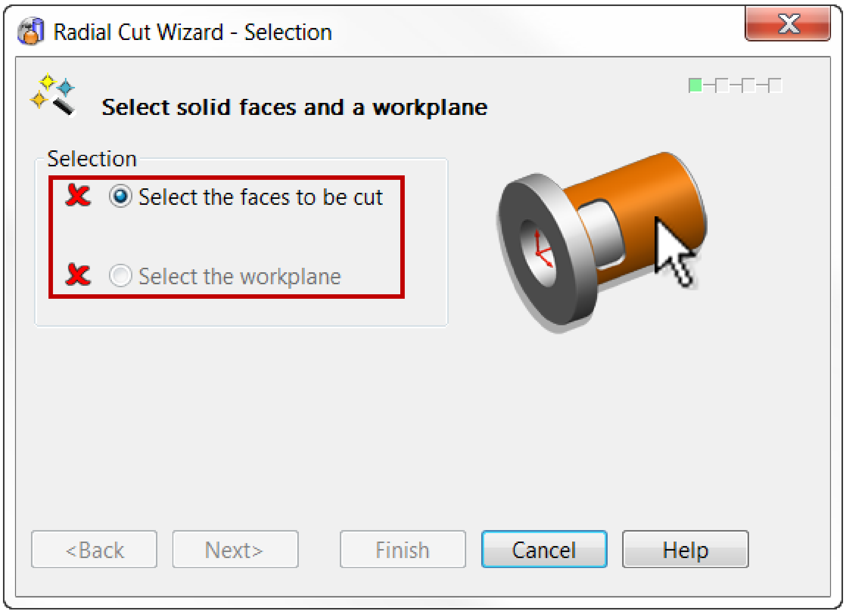
76 Select an **Isometric View** (**Ctrl + 1**).

77 Ensure the **solid** is **selected** by clicking on it with the **graphics window** or in the **solid history tree**.

78 Click the **Solid Feature**  button to activate the associated **Sub-Menu** for solid editing.

79 Select the **Radial Solid Cut**  button from the **Solid Feature** menu. This is located in the **solid cut flyout**.

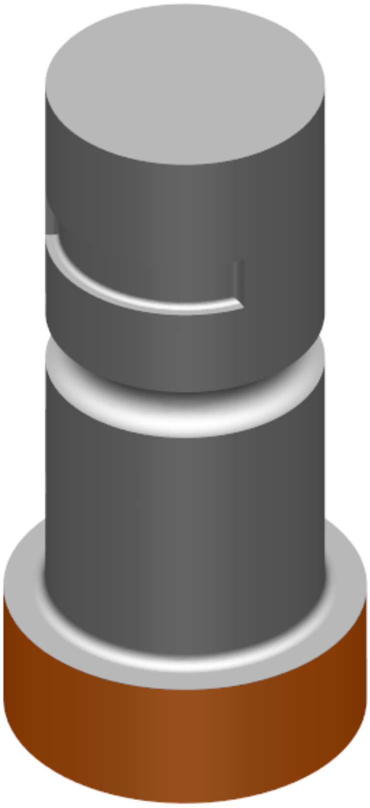
Again, this will open the **Radial Solid Cut** wizard which will guide you through a series of step by step windows enabling you to quickly and easily create a **solid cut**.

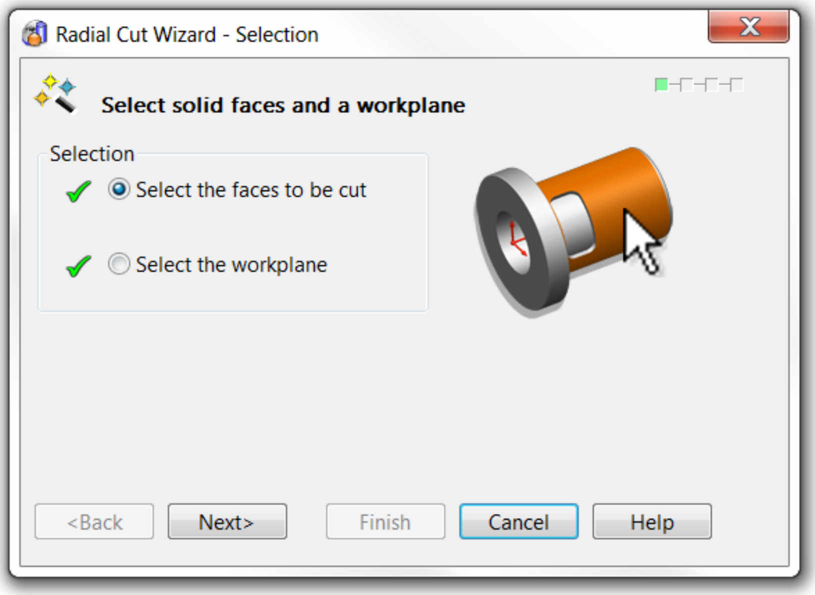


80 Select the **Lowest "Base" Cylindrical Face** on the solid as shown in the image on the next page.

81 The **two** green ticks in the dialog box denotes when the selection is sufficient to move on to the next window. Note: The workplane will automatically be selected as the **Active Workplane**.

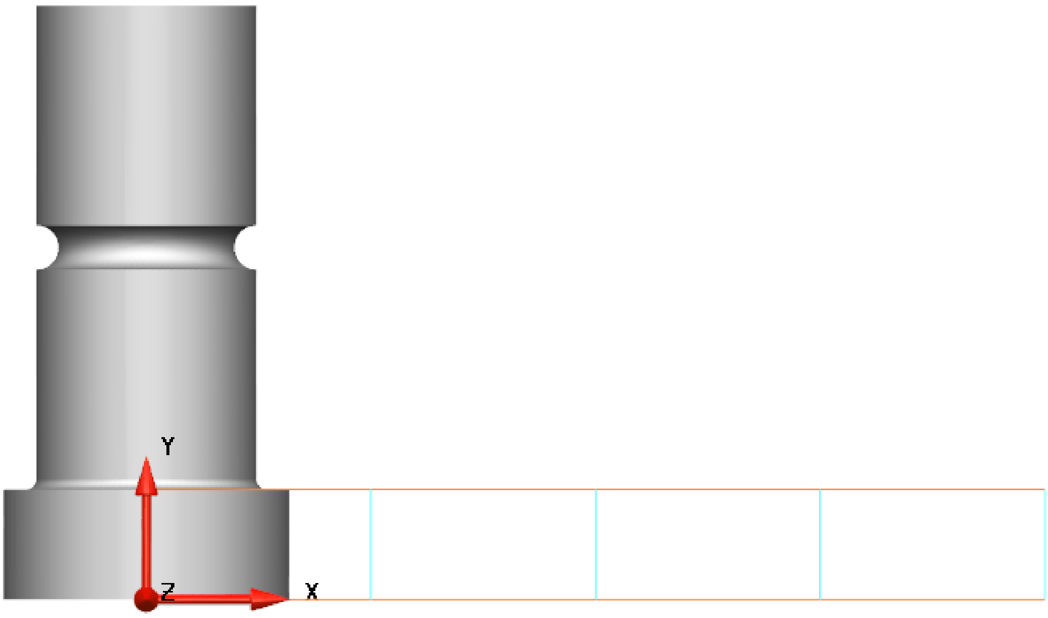
*The **Z Axis** of the **Active/Selected Workplane** needs to be **parallel** to the axis of the cylindrical face(s) to be cut.*





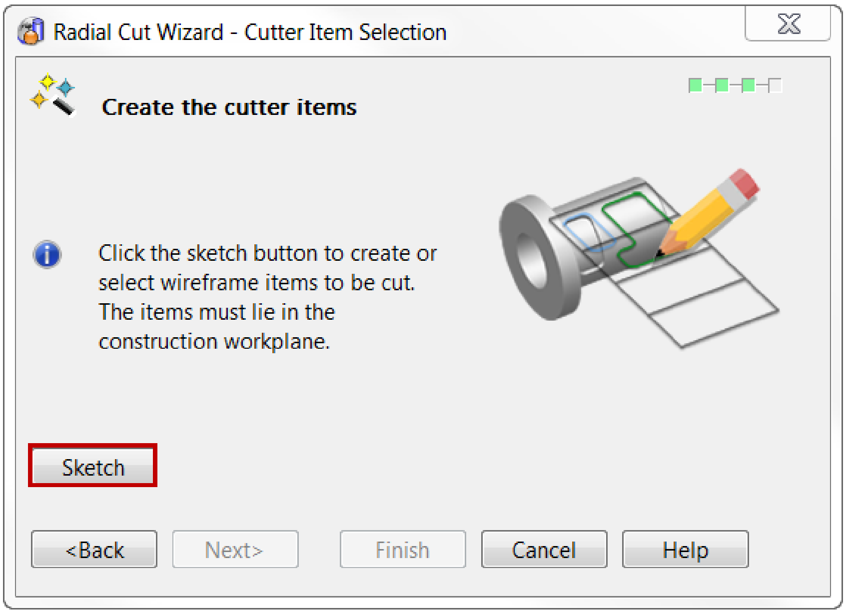
82 Click **Next**.

83 Select a **View from Top** (**Ctrl + 5**). While in the wizard this will view **down** on the **unwrapped face** usng a temporary workplane **normal** the the **sketch plane**.



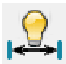
This page of the **wizard** allows extra vertical **guidelines** to be created at **any angle** around the **solid face** beyond the **default lines** to further help position wireframe around the face of the solid. In this example we will use the default angular guidelines.

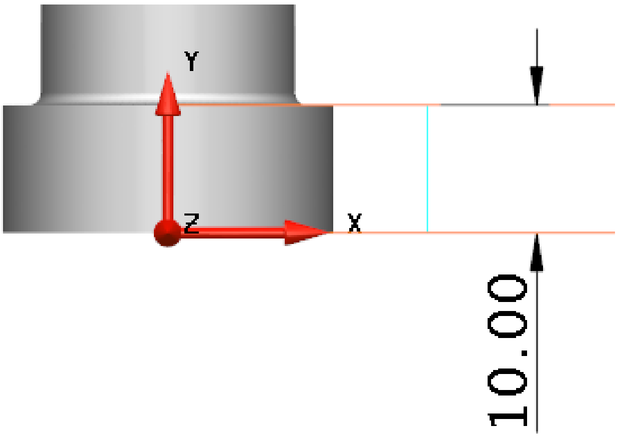
84 Click **Next**.



85 Select **Sketch** to **hide** the main wizard **dialog** and enter the **sketch mode**.

*This will **hide** the main dialog to enable unobstructed sketching in the **graphics window**. Reducing down to a **Save/Dismiss Changes** sketch dialog seen below.*

86 Using the **Automatic Dimensioning**  tool in **Annotation Menu**  select a **vertical guideline** to **measure** the overall **height** of the **base**.

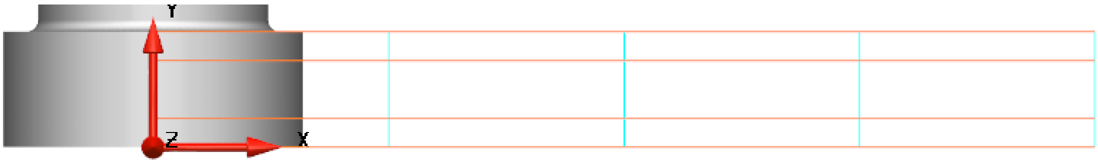


87 Select and **Delete** the **dimension**.

88 Select the **lower horizontal line** that represents the **bottom edge** of the solid face.

89 From the **General Edits**  menu, choose **Offset** . Ensuring the orange arrow is pointing **upwards** from the line, **Offset copies** of the line **2.5mm** & **7.5mm**, and then **close** the **offset** form.

*Note how in the **sketch** mode within the wizard all of the required **PowerSHAPE functionality** is **available**.*

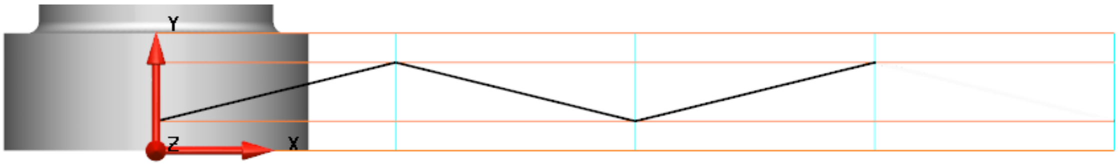


*This is currently the most efficient option to create horizontal guidelines to aid wireframe creation. As you can see by **offsetting** the lower line upwards we have created guidelines at known values in **Z** that can be **snapped** to using the **intelligent cursor**.*

Next we will create a zig-zag line with a **height** varying between **2.5mm** & **7.5mm** from the **lower edge** of the face changing direction every **90°**.

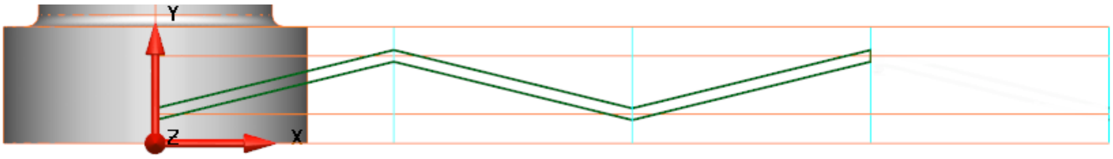
90 From the **Lines** menu on the **Main Toolbar** select **Continuous Line**.

91 Starting from **Y = 2.5mm** (current workplane in view) create the line as shown.



92 Convert the **wireframe** into a **composite curve**.

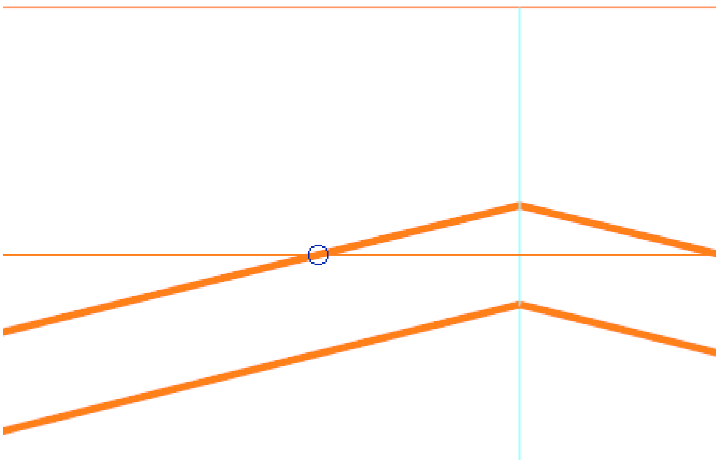
93 With the **curve selected**, **move** the **curve** and create **copies 0.5mm vertically** (in **Y** in this case) in **both** directions and delete the orginal.

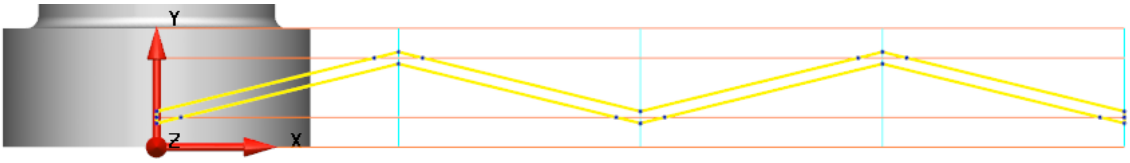


94 Using the vertical guidelines that are already in place and the two **composite curves**, **track** a **single closed composite curve** around both zig-zag profiles.

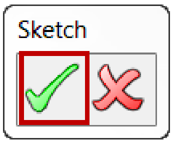
*Using the guidelines that are already in the sketch saves using having to create lines to close the curve off. **Remember** to **Zoom In** and **Pan** around the model while tracking the curve to ensure the correct curve is produced.*

95 Once the curve is **closed** as shown below, **save** the curve in the **composite curve creator** and **eject** to close the form.

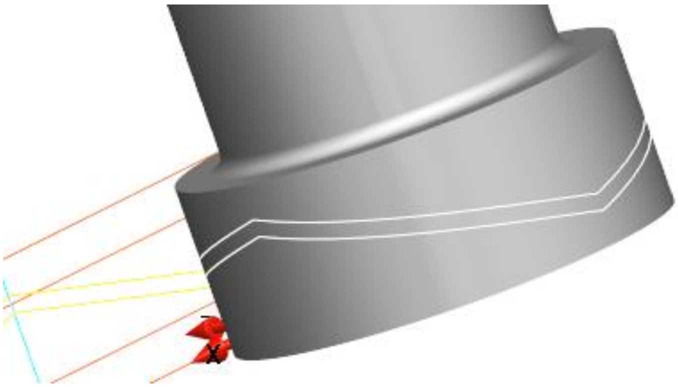




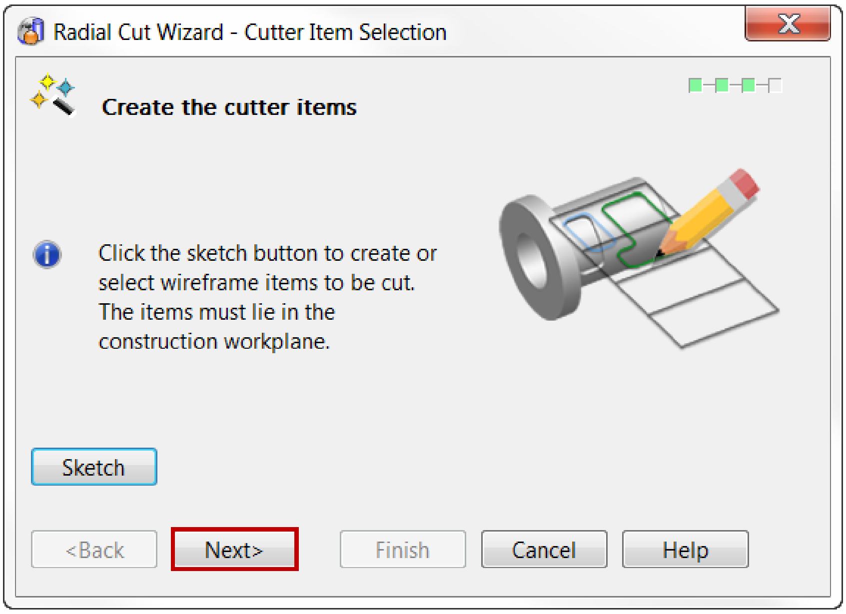
96 Once you are happy with the **shape** and **position** of you **composite curve**, **SELECT all required curve(s) that you wish to wrap/cut** and click the **green tick** on the **sketch dialog** to accept the changes and re-enter the **wizard**.

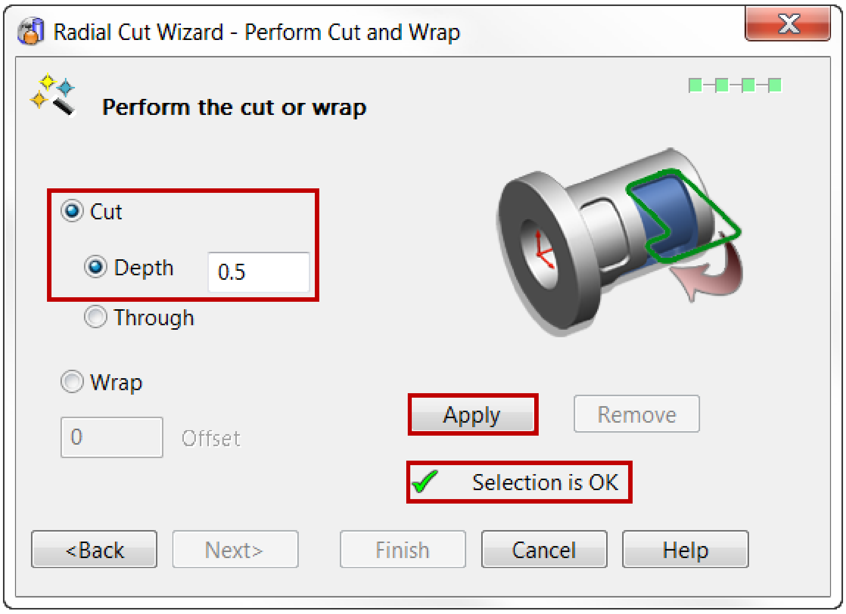


Accepting this form will also create a view of the **wrapped curve(s)** onto the **model** to show there **position** and **size** in direct relation to the solid model.



97 Select **Next** within the dialog.

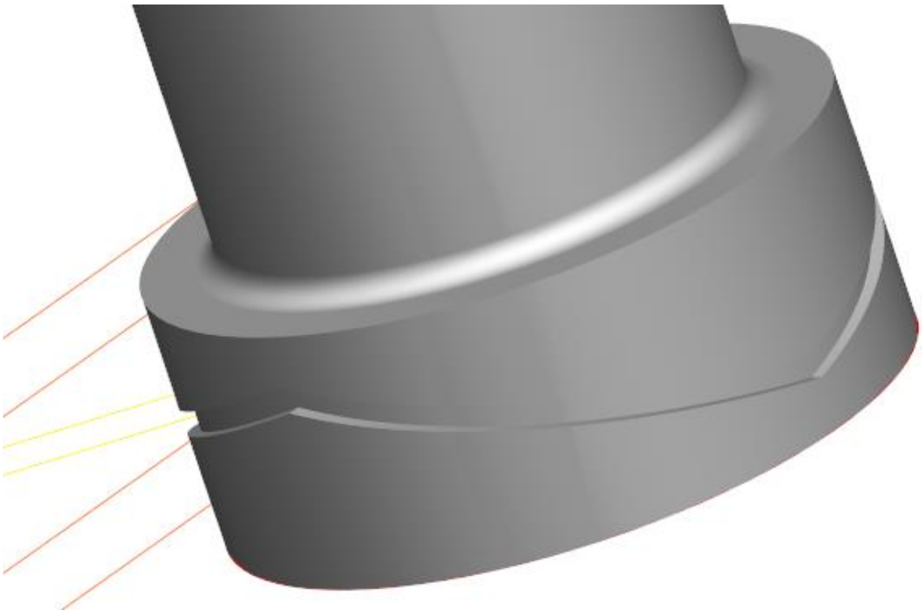




98 Select **Cut** and a **Depth** of **0.5mm**, then select **Apply**.

*If after you select **Apply**, changes are required and you are still in the wizard the **Remove** button will delete the current cut, allowing edits to be made.*

*A **Green Tick** will notify you that the current selection is **OK** and ready to be cut.*



99 As you can see the **Cut** has been performed successfully, Select **Finish**.

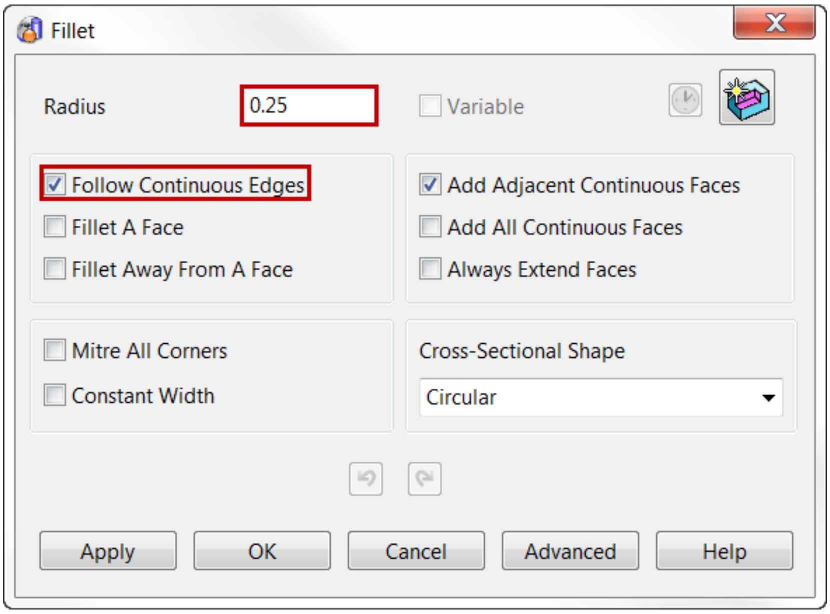
100 The **curve** that was created on the unwrapped solid face in the wizard is also kept after the operation. This can be **deleted** altogether, or you could put it on a separate **construction level**.

*As this is a solid the cut faces are easily edited using further **Solid Features** like **solid fillet** and **wall draft**.*

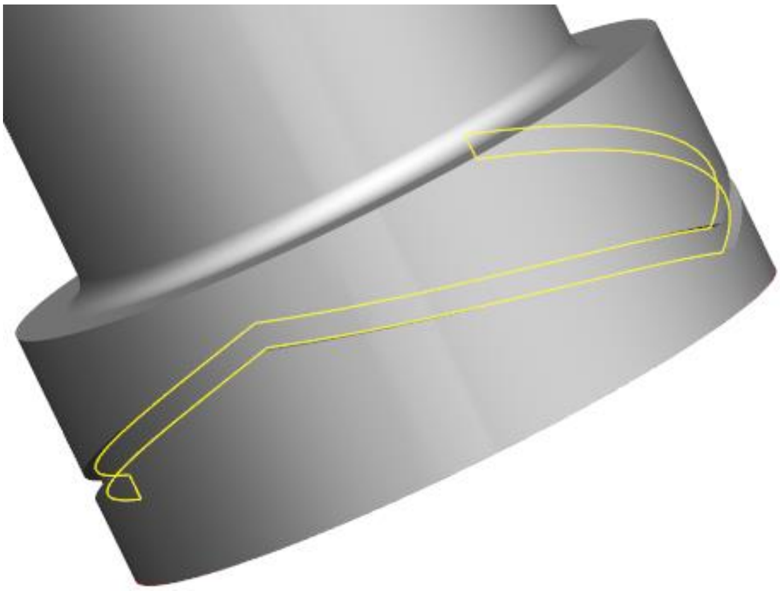
101 Ensure the **solid** is **active** which is denoted by the **red flag** next to the solid in the **solid history tree**.

102 From within the **Solid Feature**  sub-menu select the **Solid Fillet** .

103 Enter a **Radius** of **0.25mm**.



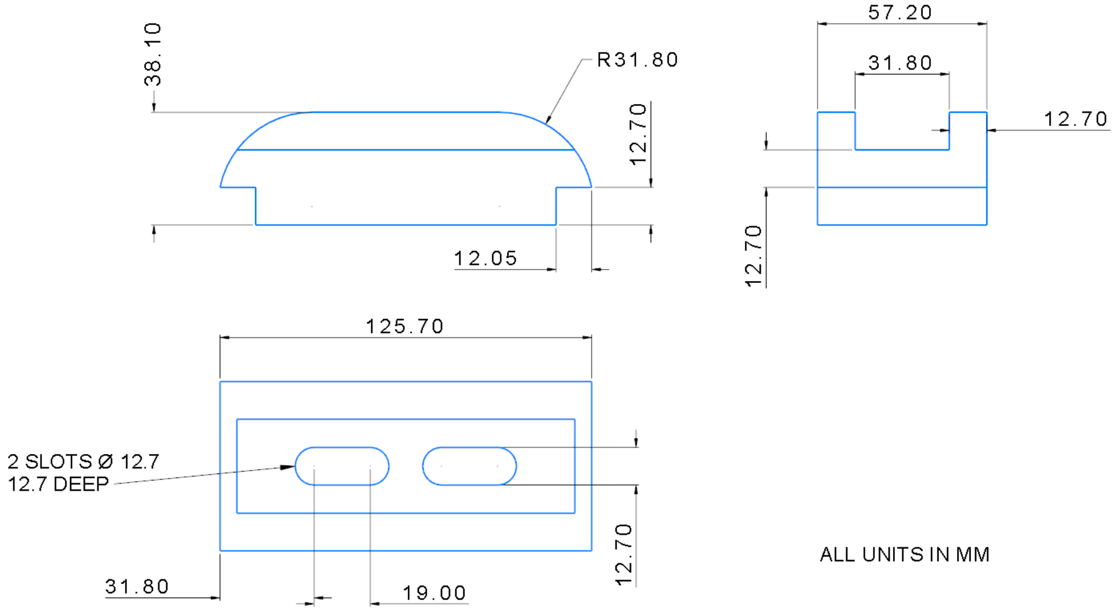
104 Select the **ALL** the edges surrounding the face at the bottom of the **radial cut zig-zag groove**.

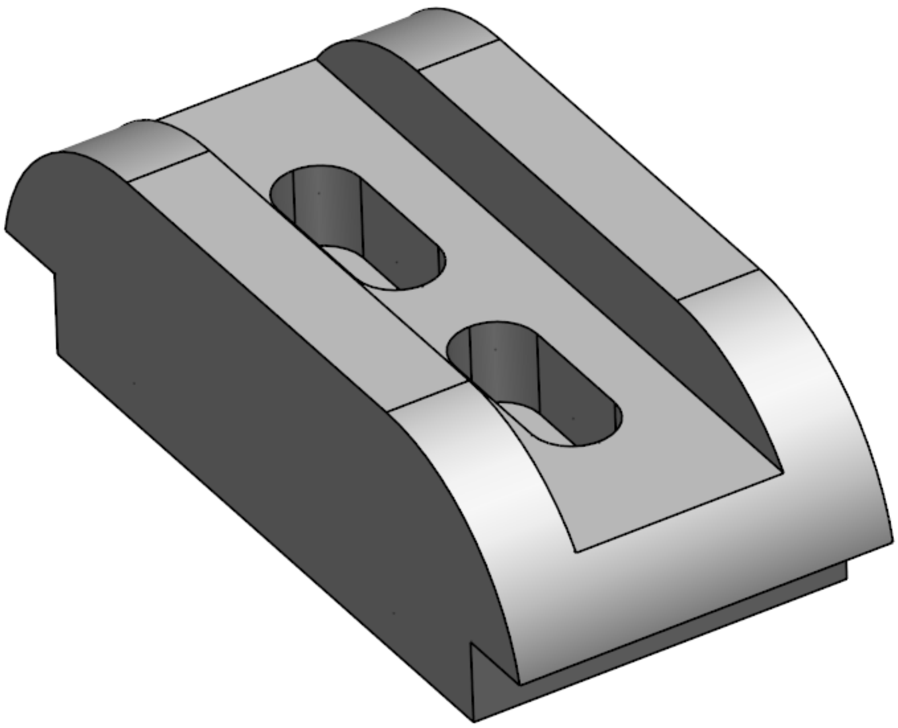


105 Click **Apply** then **OK** to accept and save the changes and **create** the **fillet**.

Exercise 4: Solid model

1 Using the basic dimensions provided generate a solid model.





1 Select **File > Save As**:

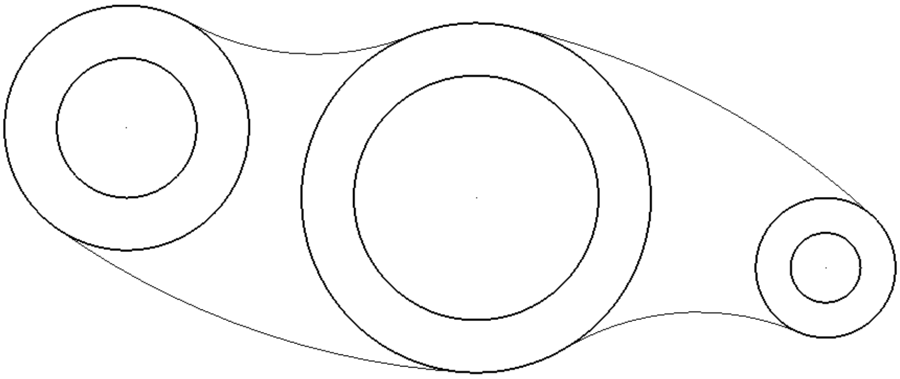
*.....**Training Data****Coursework****Exercise 4.psmodel***

Close the model.

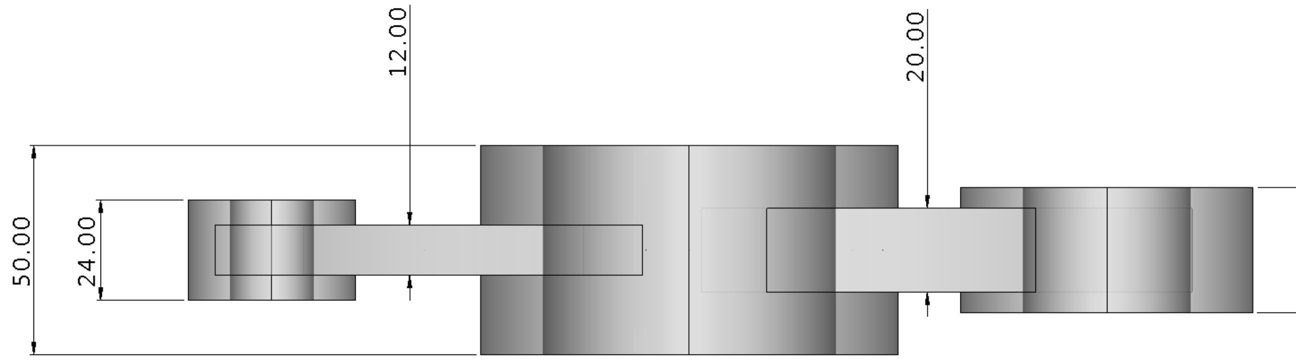
Exercise 5: Solid lever

1 Open the previously completed wireframe model from Chapter 2

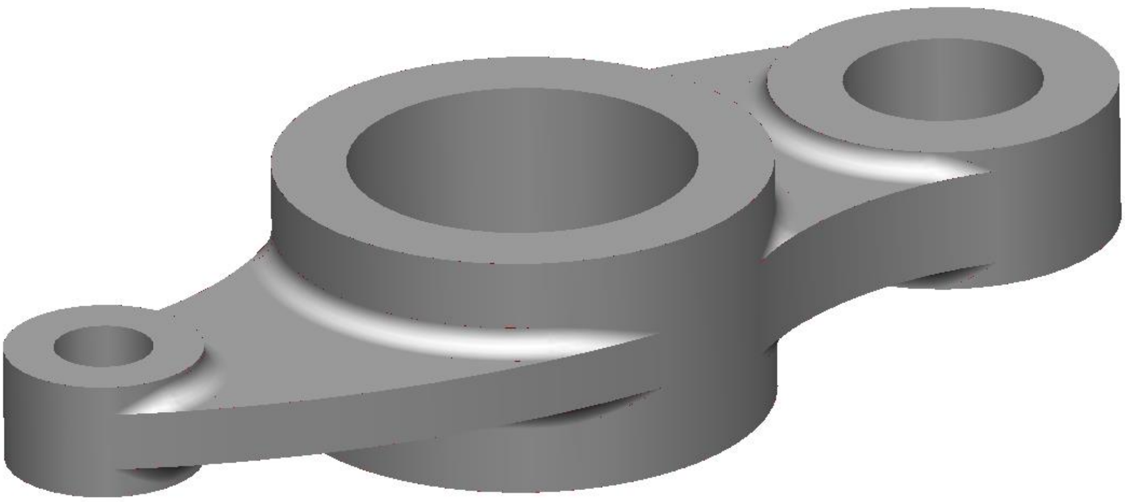
.....\PowerSHAPE_Models**lever-example.psmodel**



2 Generate a solid model using the following dimensions. Use the inner three circles to create the through holes.



3 Apply a 5mm fillet to both upper and lower intersecting features as shown



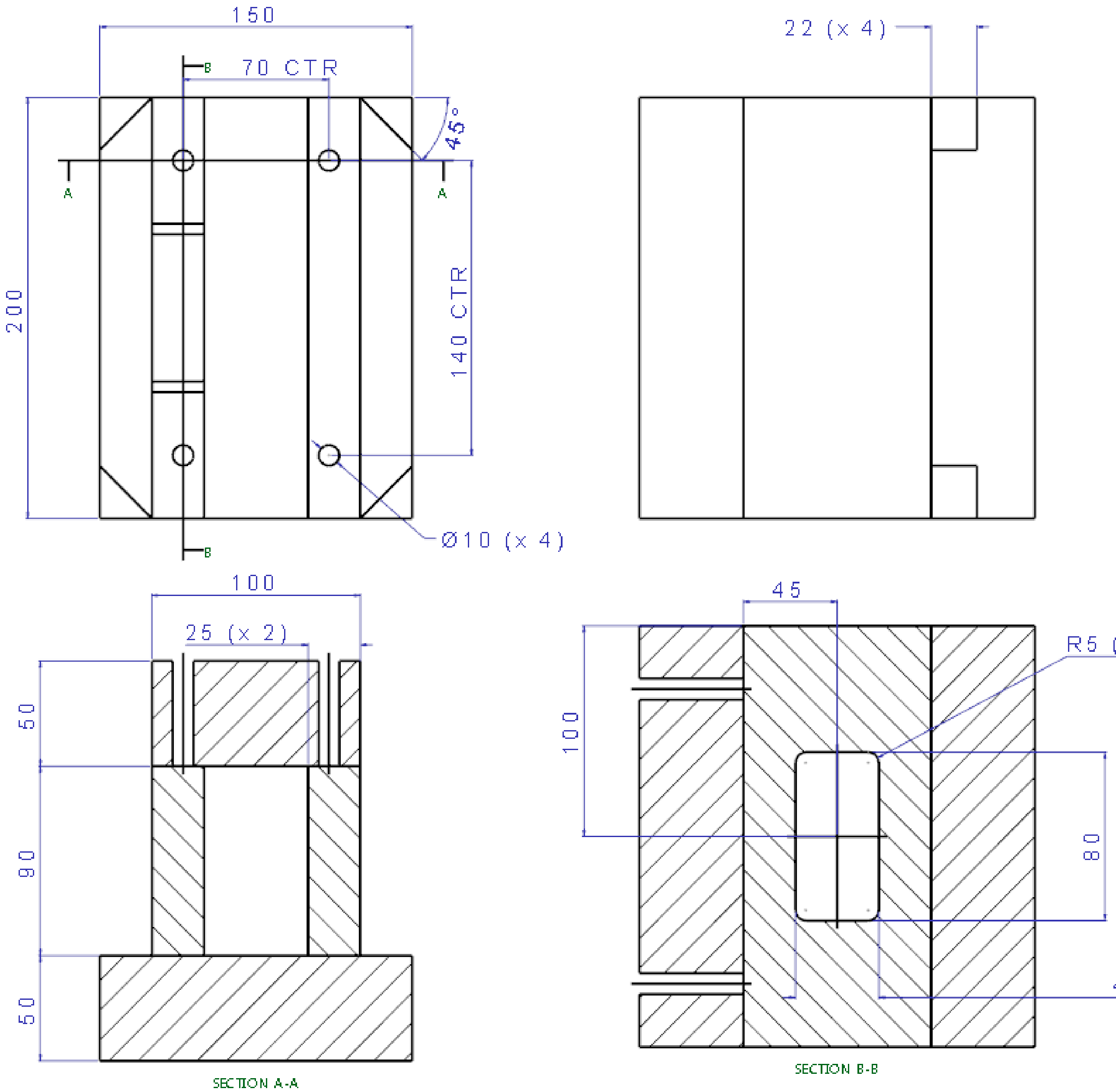
4 Select **File > Save As**:

.....***Training Data******Coursework******Exercise 5.psmodel***

5 **Close** the **model**.

Exercise 6: Assembly plate

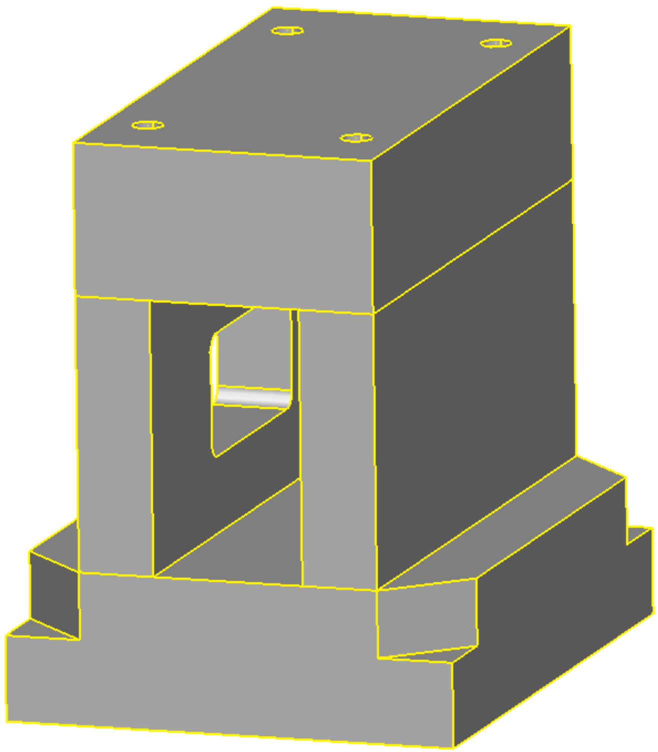
1 Generate a solid model using the dimensions provided.



1 Select **File > Save As**:

*.....**Training Data****Coursework****Exercise 5.psmodel***

2 **Close** the **model**.



7. Direct Solid Modelling

Introduction

Direct Solid Modelling provides tools to:

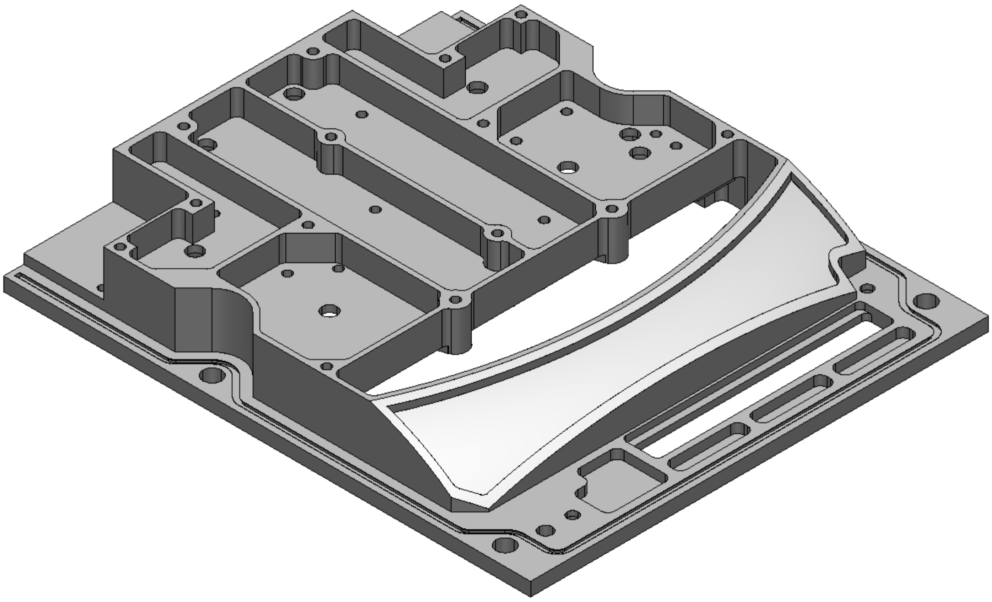
- Make **rapid edits** to **history-free solids**.
- **Recognise key solid features** for simple **editing** (for example, fillets, pockets)
- **Add draft** to faces and **automatically extend** and **intersect** all **surrounding geometry** to keep a **closed solid**.
- **Remove** and **heal** to simplify **geometry** for **downstream manufacturing** operations or to **accommodate design changes**.
- **Edit faces**, **automatically extending** and **intersecting surrounding faces** to maintain a **closed solid**. This lets you do operations like copying a strengthening rib or offsetting the side faces of a pocket.

Direct Modelling is particularly efficient for operations that are typically performed using surface modelling and would require you to extend and intersect each surrounding faces manually to create a closed solid.

These powerful tools will be demonstrated in the following example.

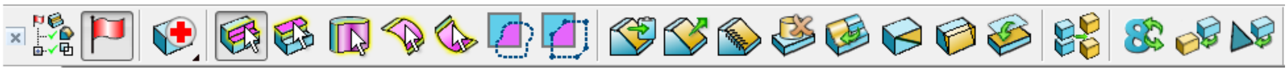
1 **Import**  the solid model:

.....**PowerSHAPE Data****Direct modelling plate.x_t**

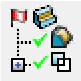


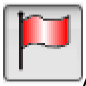
2 Select the Solid to display the **Solid Editing** Toolbar.

Solid Edit Toolbar



The following **Solid Editing** and a number of **Direct Solid Modelling** tools are available:

 - Toggle display of the **Solid Feature Tree**.

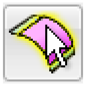
/ - Make the **Solid Active/Inactive**.

 - **Find** and **Fix faults** in the **solid**. (**Solid Doctor**)

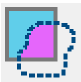
 - Select a **Feature** on the **solid**.

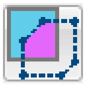
 - Select **Individual Faces** on the **solid**.

 - Select **Faces** of **Continuous Regions**.

 - Select **Faces** of **Convex Regions**.

 - Select **Faces** of **Concave Regions**.

 - Select **Faces** using a **Continuous Lasso**.

 - Select **Faces** using a **Discrete Lasso**.

 - **Copy** the **Selected Faces**.

 - **Extract** the **Selected Faces**.

 - **Select** and **Sew** items into the **solid**.

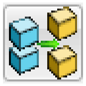
 - **Remove** and **Heal** the **Selected Faces**.

 - **Merge Selected Faces**.

 - **Divide** the **Selected Faces**.

 - **Draft Faces** of the **solid**.

 - **Replace Faces** of the **solid**.

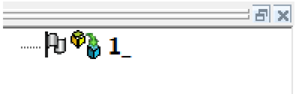
 - **Divide Selected solid** into **Multiple Solids**.

 - **Convert** between **Version 8 solid** and **Parasolid**.

 - **Convert** between **Surfaces** and **Parasolid**.

 - **Convert** between **Mesh** and **Parasolid**.

3 Select  to display the **feature (history) tree** of the imported solid.



As you can see the solid has **no features or history** associated with it.

Direct Solid Modelling demonstration

Direct modelling functionality can fall into two categories.

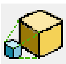
1 **Direct editing** - Tools which directly change or edit features without creating history (as a new feature).

2 **Feature recognition** - Tools which create new features and history. These can then be edited as part of the active solid.

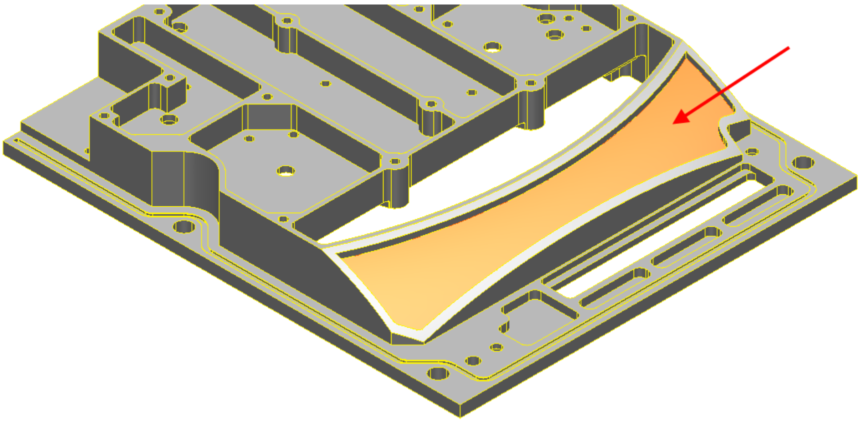
Direct Editing

Solid faces which require editing, must be selected first using one of the selection tools available in the Solid Edit toolbar.

Once selected, the following **General Edits**  can be applied.

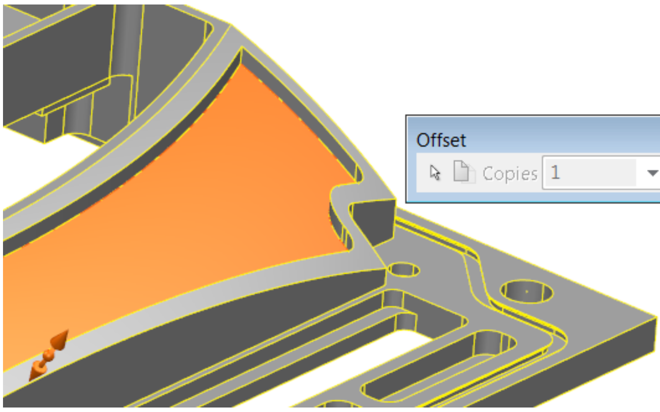
- **Move** - 
- **Rotate** - 
- **Mirror** - 
- **Offset** - 
- **Scale** - 

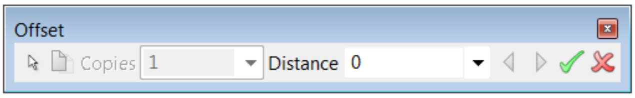
4 **Select** and **highlight** the **large curved face** shown in the **next image**.



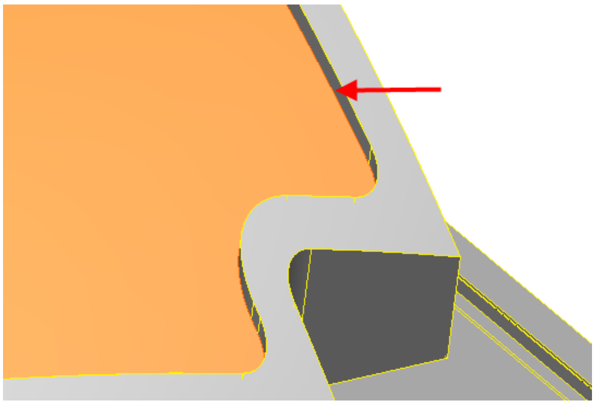
 *By default, the surface select mode is **Select individual faces**.* 

5 Select **Offset**  from the **General Edits** toolbar. 



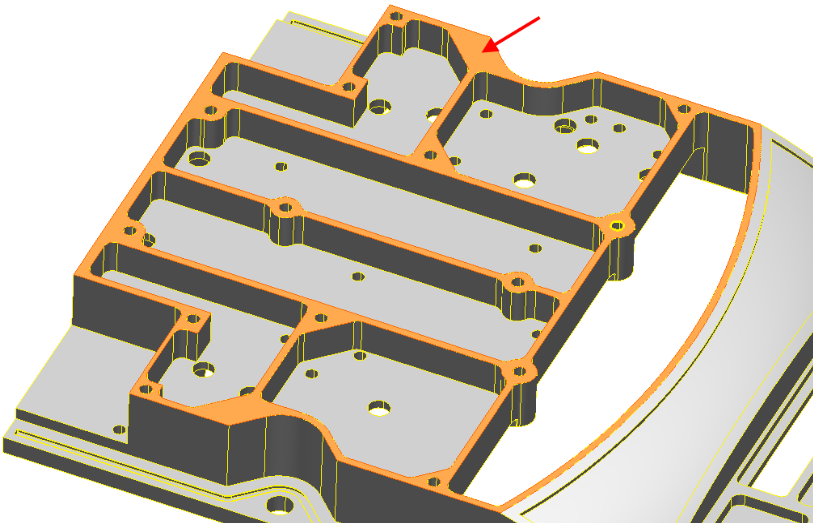


6 Enter a value of **2mm** on the **Offset** form.



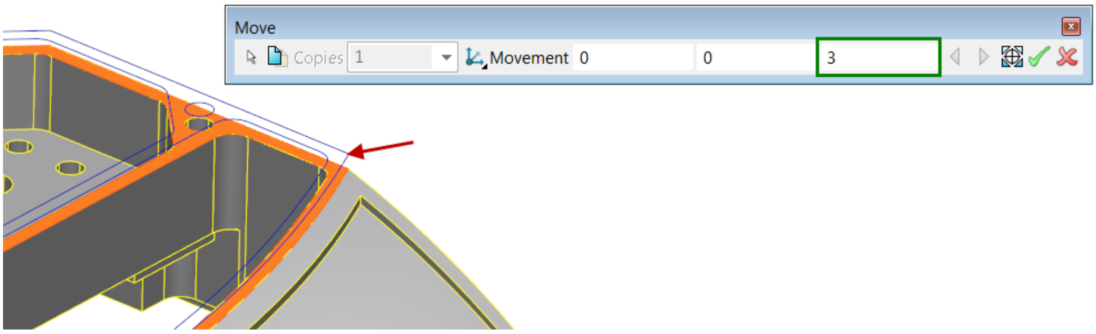
The **face** has been **offset**. More importantly however, **all surrounding vertical faces** have been **retrimmed** and **reintersected automatically**.

1 **Select** and **highlight** the **top horizontal face**.



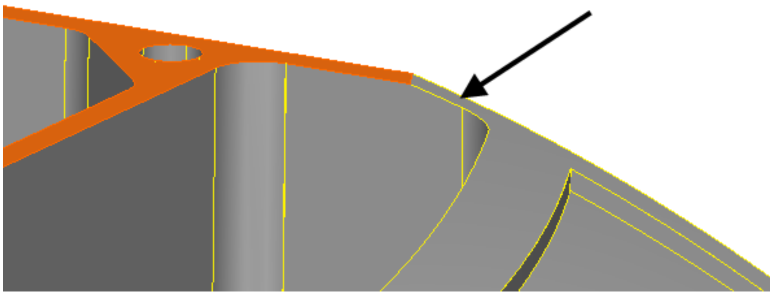
2 Select **Move**  from the **General Edits** toolbar. 

3 Enter a distance of **3mm** in the **Z axis** and press **Enter** (for preview).



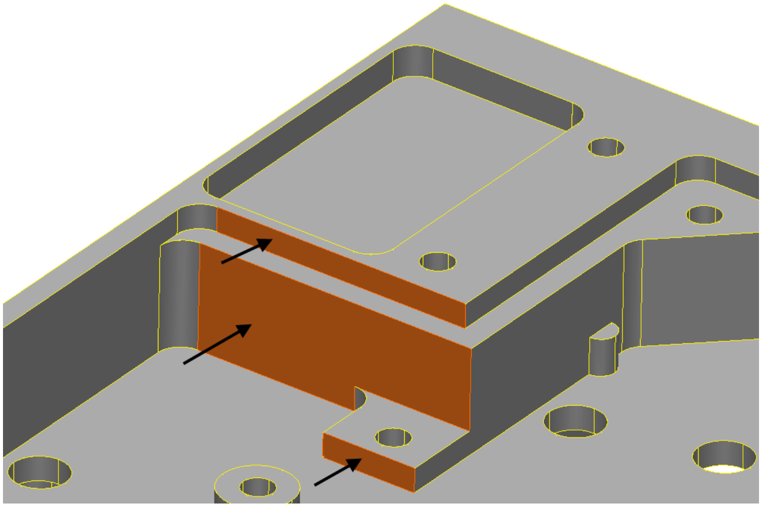
As expected, the **preview** of the calculated result is displayed.

4 Select the green tick  to accept. Note the effect in the region shown above.



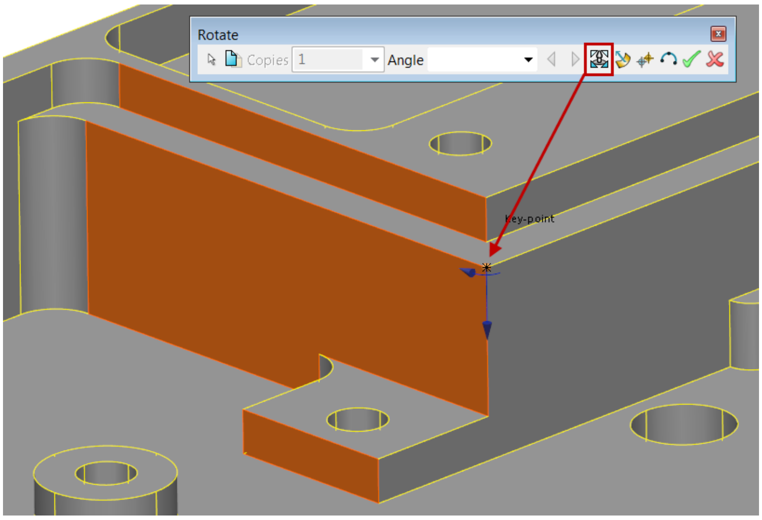
The top face has successfully moved by **3mm** in the **Z axis.** All related vertical faces have reintersected and maintained the solid closure. The sloping face achieved this by extending its faces.

1 **Rotate** the part to view the bottom side and zoom into the region shown.



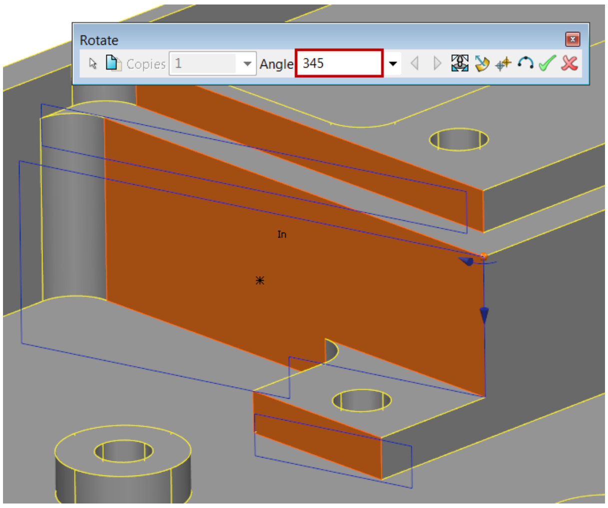
2 **Shift** select the three faces highlighted.

3 Select **Rotate**  from the **General Edits** toolbar. 

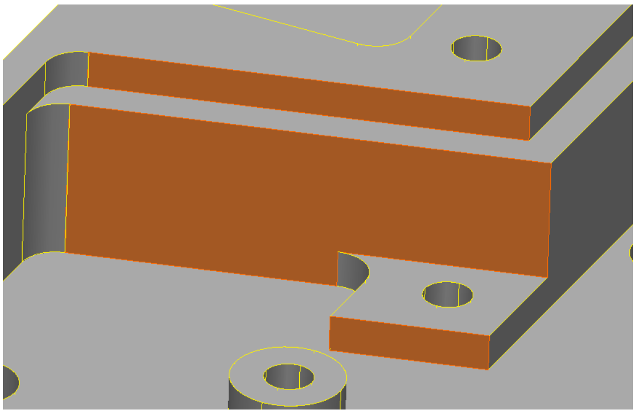


4 From the rotate form select **Reposition**  and click on the **keypoint** to specify the rotate origin.

5 Select and drag only of the selected faces to dynamically preview the faces rotating about the origin.



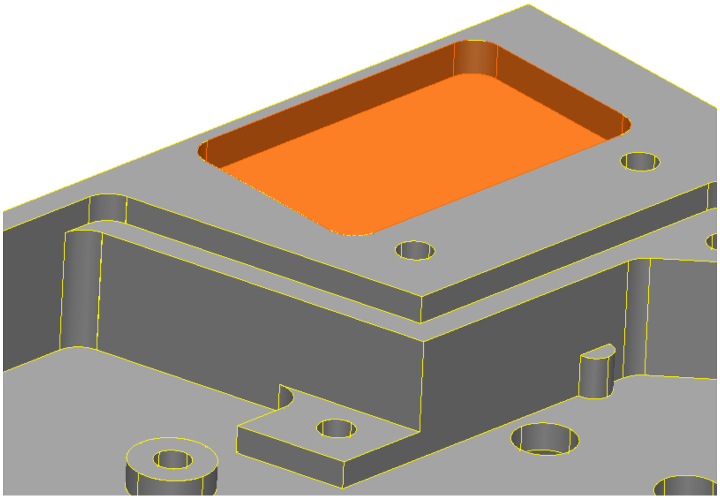
6 Release the mouse and enter an **angle of 345** before accepting the result. 



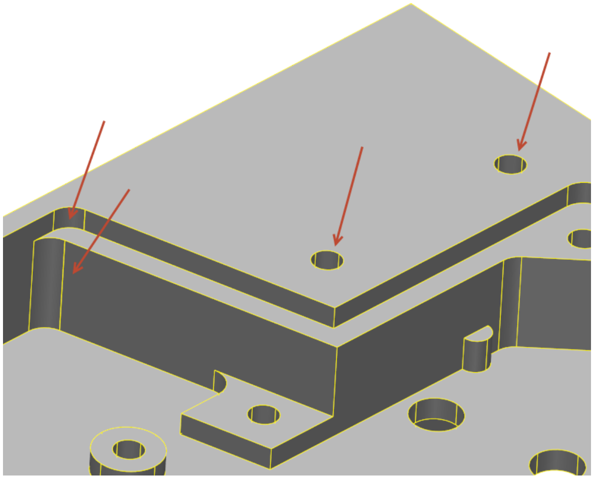
Only the selected faces have rotated to the specified angle.

1 From the **solid edit** toolbar change the selection mode to faces of **concave regions**. 

2 Single click in the pocket to highlight all relevant faces.

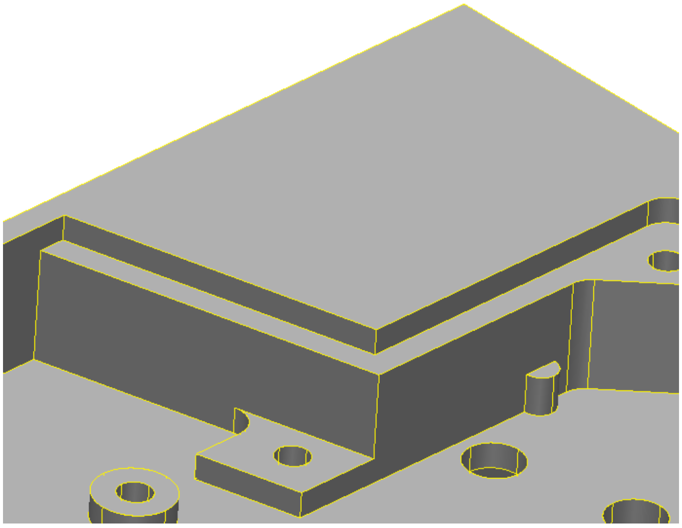


3 From the **Solid Editing** toolbar, Select **Remove and Heal**. 

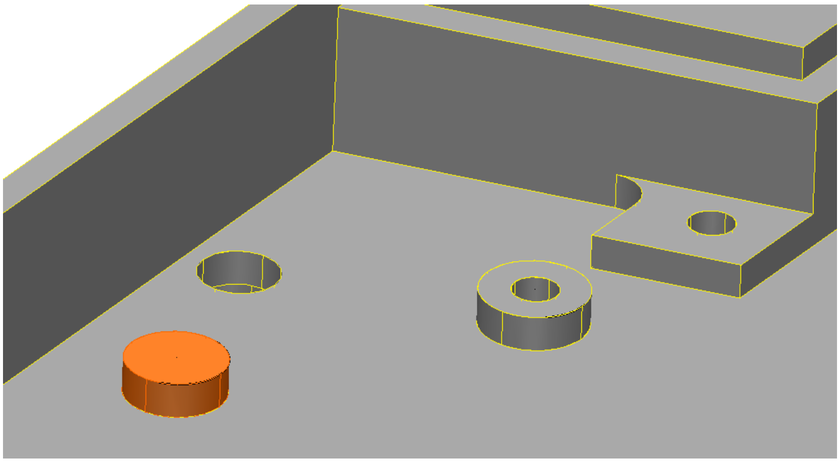


This tool **removes all selected faces** and then **closes** the **resulting gap** by **modifying** the **surrounding faces**.

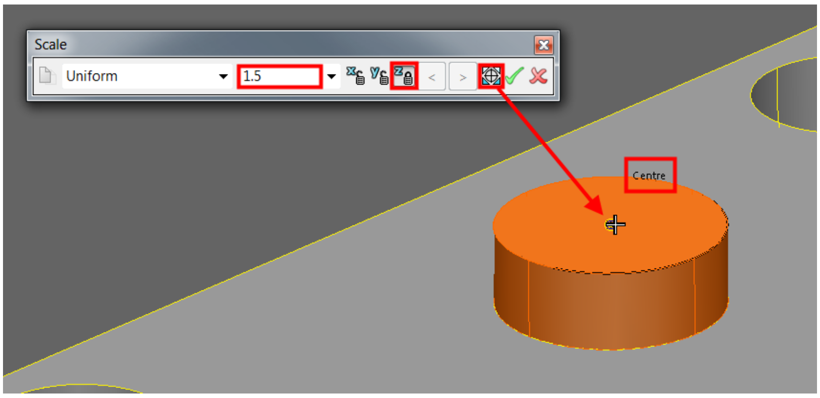
4 Remove the **two holes** and **two fillets** (indicated in the **previous image**) using **Remove and heal**. 



5 Select the **top** and **cylindrical faces** which define the **boss feature**.

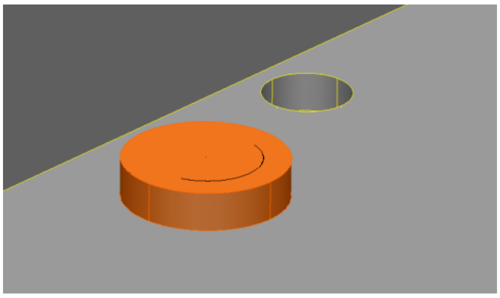


6 Select **Scale**  from the **General Edits** toolbar. 



7 Select **reposition scale origin**  and snap to the **centre** of the **face** using the **Arc-Centre keypoint**.

8 Lock the **Z axis**  and enter a **scale factor of 1.5**. Select the green tick  to confirm.

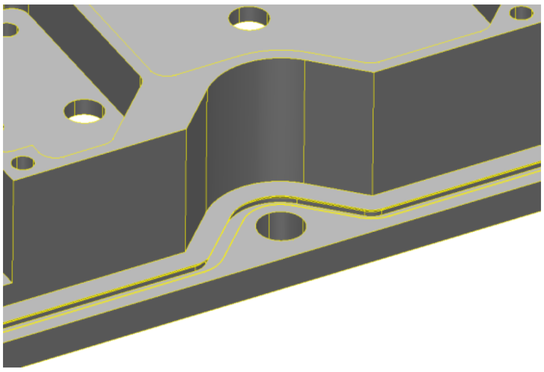


The boss feature has scaled in the **X** and **Y axis** by a **factor of 1.5**.

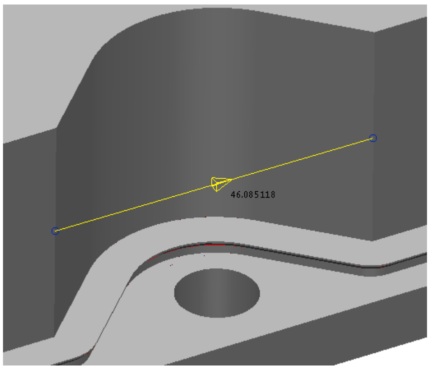
*If the **Z** axis was **not locked PowerSHAPE** would still have automatically limited back the **Z intersection** and provided the same result.*

PowerSHAPE provides the ability to **divide** the **default faces**, thus providing **more control** over specific regions that require editing.

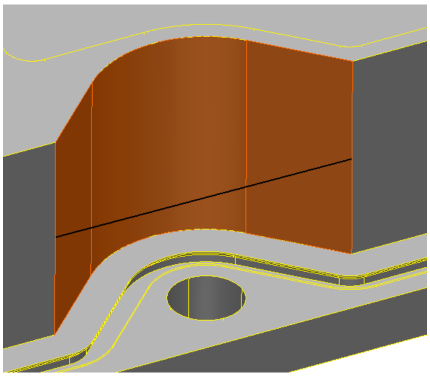
1 **Rotate** the model to view the top and zoom into the area show.



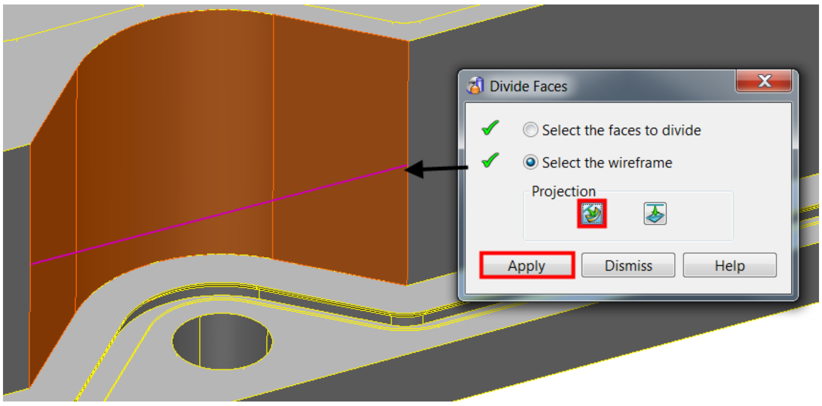
2 Create a **single line** approximately half way up the **concave face**, **snapping** the **outside edges**.



3 Select the **three vertical faces** which define the **concave recess**.

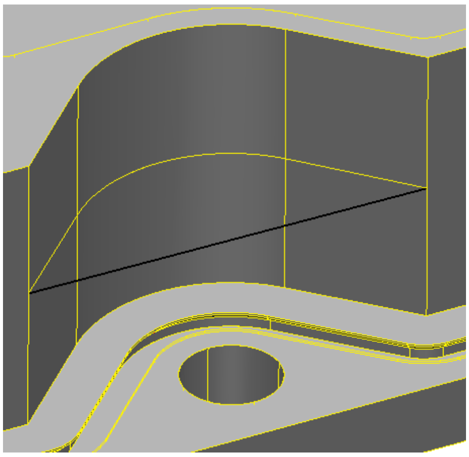


4 From the **Solid Editing** toolbar, Select **Divide faces**. 



5 Select the single wireframe to nominate and update the **Divide faces** form.

6 Keep the default projection mode of along face normal  then **Apply**.

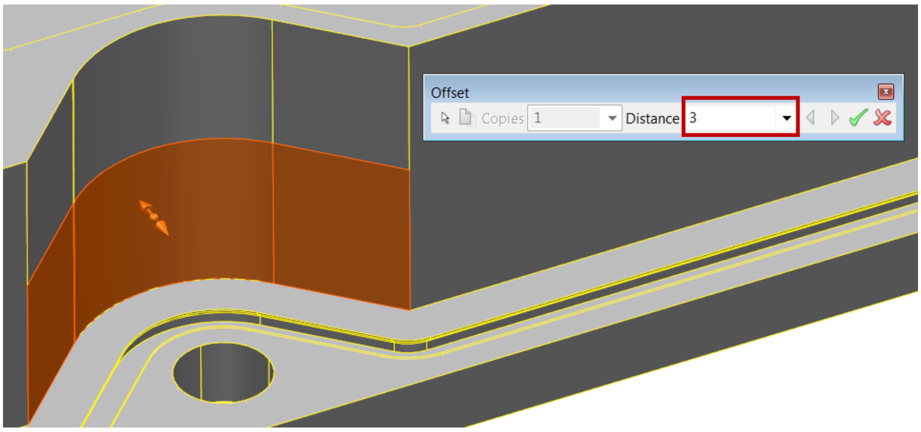


The curve is projected onto the selected faces and now the three faces become six.

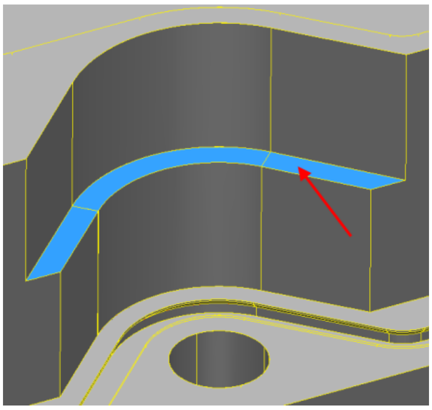
New** to **PowerSHAPE 2014 R2** is the option to **merge faces  *on a **solid**. This will merge **adjacent continuous faces** into one continuous face, essentially the **reverse** of the operation to **Divide Faces** described above.*

7 **Dismiss** the form and **delete** the single line.

8 Select the lower three faces then **Offset**  from the **General Edits** toolbar. 

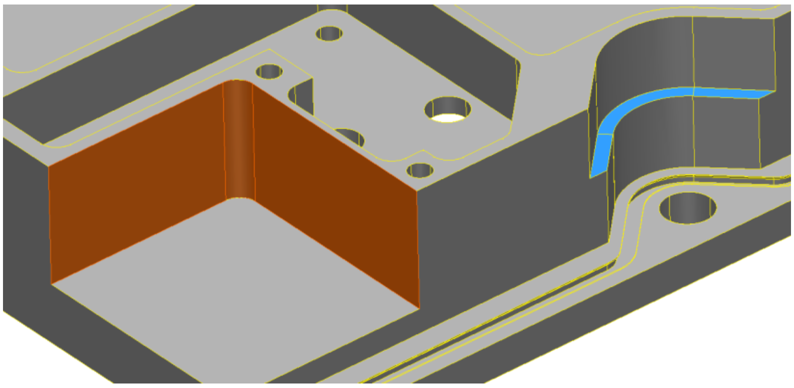


9 Enter an **Offset** distance of **3mm.**



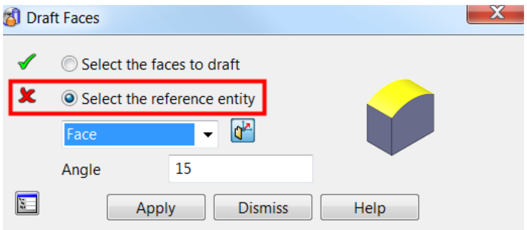
A **3mm** deep **step** has been generated with relevant **faces trimmed correctly**. A further **three faces** have been **created** to define the **horizontal step face(s)**.

1 **Zoom** into and **select** the three faces defining the corner pocket



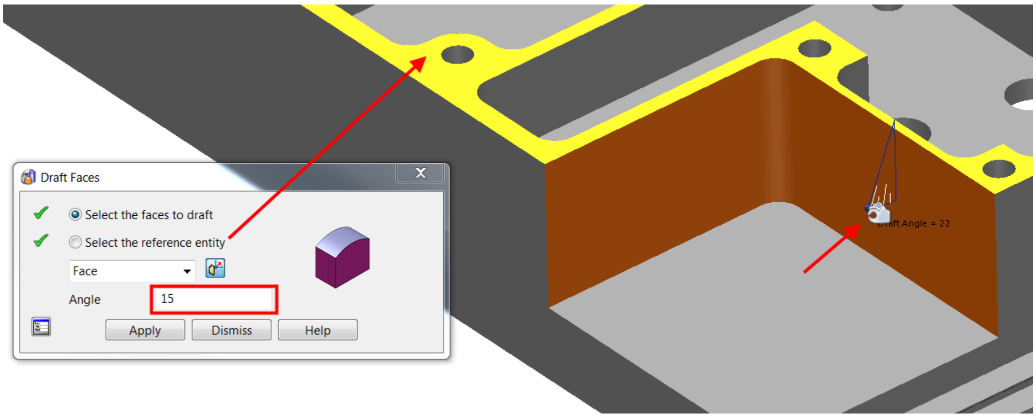
Direct modelling allows **draft angles** to be quickly applied to **selected faces**.

2 From the **Solid Editing** toolbar, Select **Draft faces.** 



The red cross  indicates a required reference entity. This can be a face, Surface or a Workplane.

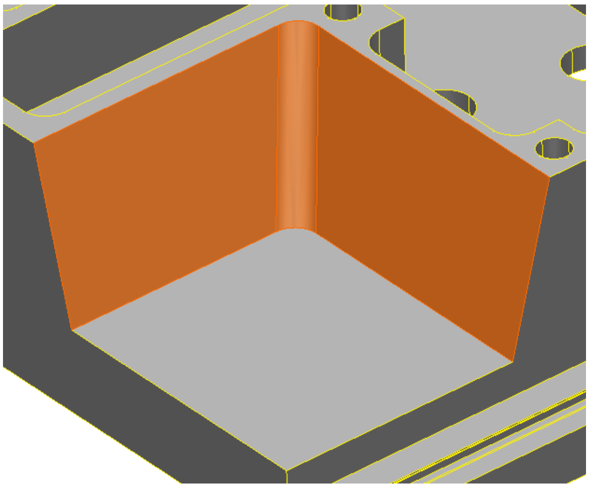
3 Leave the default **Face** entity and **select** the **top horizontal face**.



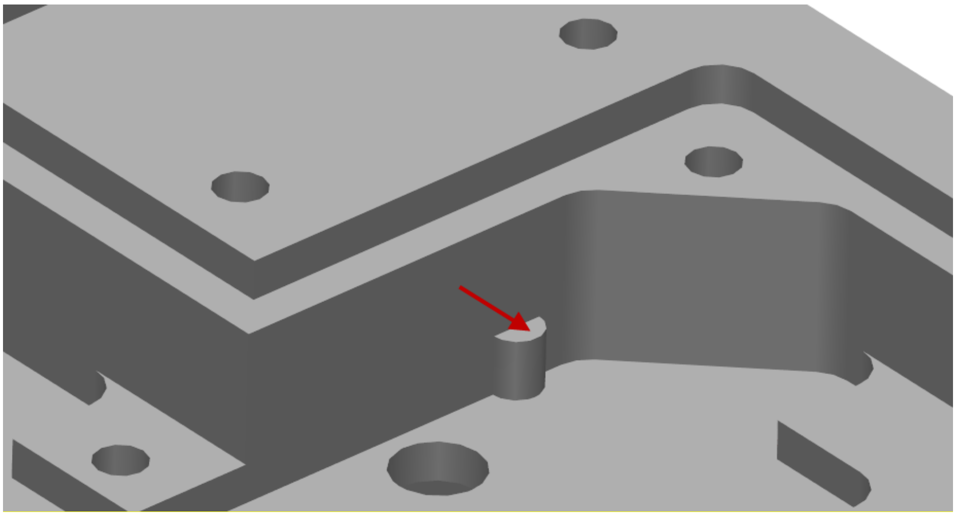
The **draft angle** can be **dynamically pulled** to **set** and preview the required angle.

4 **Dynamically** set or **enter** an **Angle of 15°** in the Draft faces form before selecting **Apply**.

5 **Dismiss** the **form** to view the result.



1 **Rotate** the part to view the bottom side and zoom into the region shown.

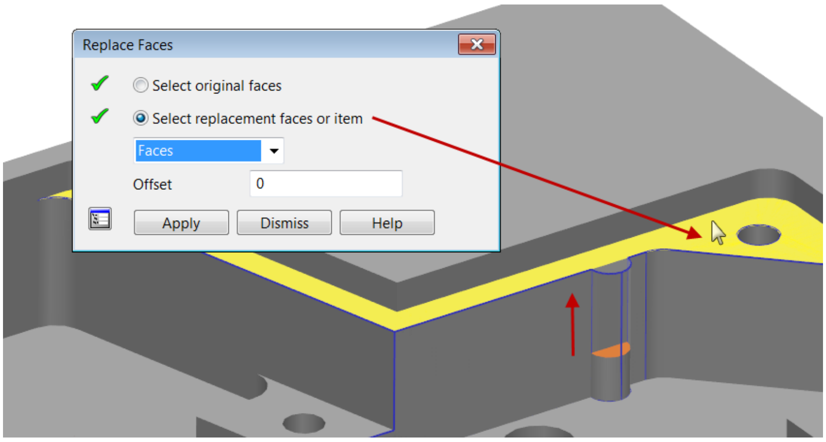


2 Select the small flat surface defining the lug feature, as indicated.

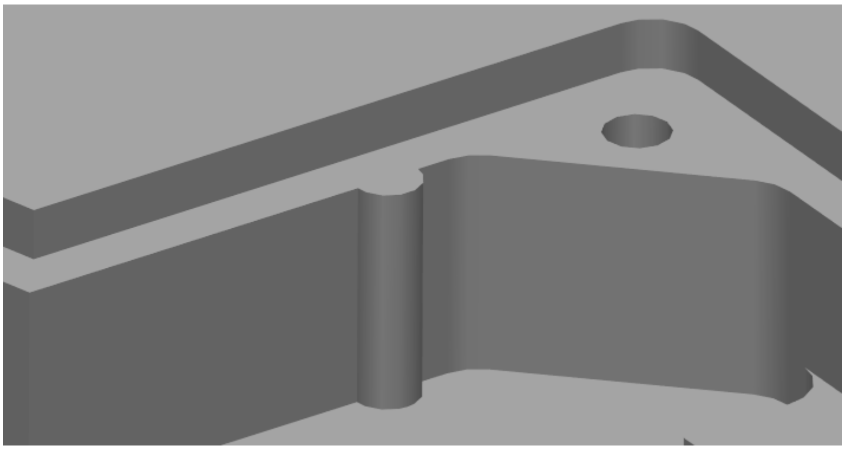
3 From the **Solid Editing** toolbar, Select **Replace faces.** 

This allows faces of a solid to be replaced with existing faces (as a reference). In this example, the length of the reinforcement lug will be extended.

4 **Select** the **face** indicated to **nominate** as the **replacement face**.



5 Select **Apply** to confirm the preview then **Dismiss** the form.

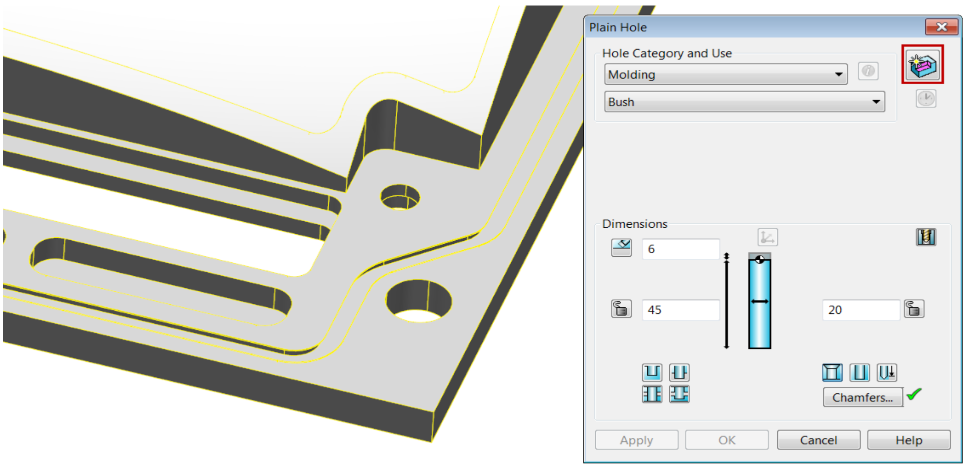


6 **Repeat** the above **process** to **extend** the **lug** on the **opposite side**.

Feature Recognition

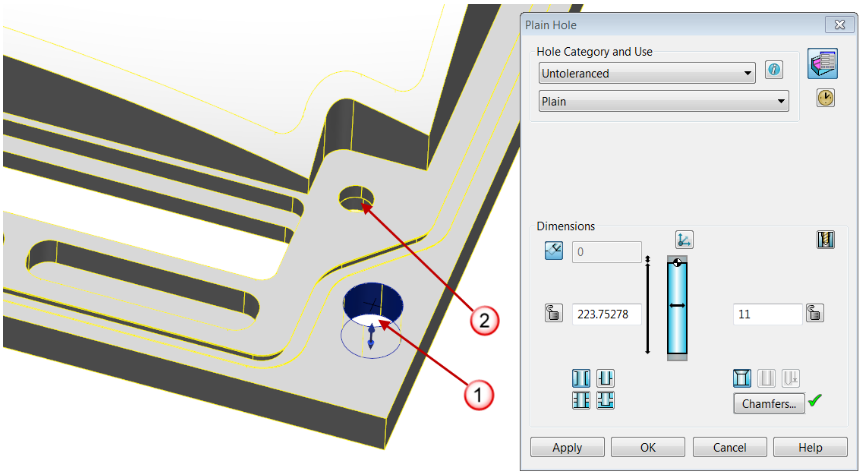
The following **Direct Modelling** tools **extract features** and **create history** in the **Solid Feature** tree.

1 **Rotate** the part and **zoom** into the area shown, then select **Create a hole**  from the **Solid features** toolbar. 



2 From the top right hand corner of the Plain hole dialog, select **Feature creation**  to toggle to **Feature recognition** mode. 

In this mode **PowerSHAPE** is capable of **recognising** the **hole**, the **parameters** that define it and then **add** it to the **feature tree**.



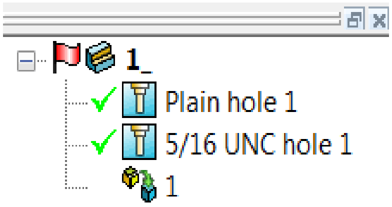
3 Select **hole**. 1

The **hole** is **highlighted** in **blue** and the **dialog form** displays its **properties**.

4 Select **Apply** to accept the results.

5 Repeat for hole. 2

A **counterbored tapped hole** is recognised.

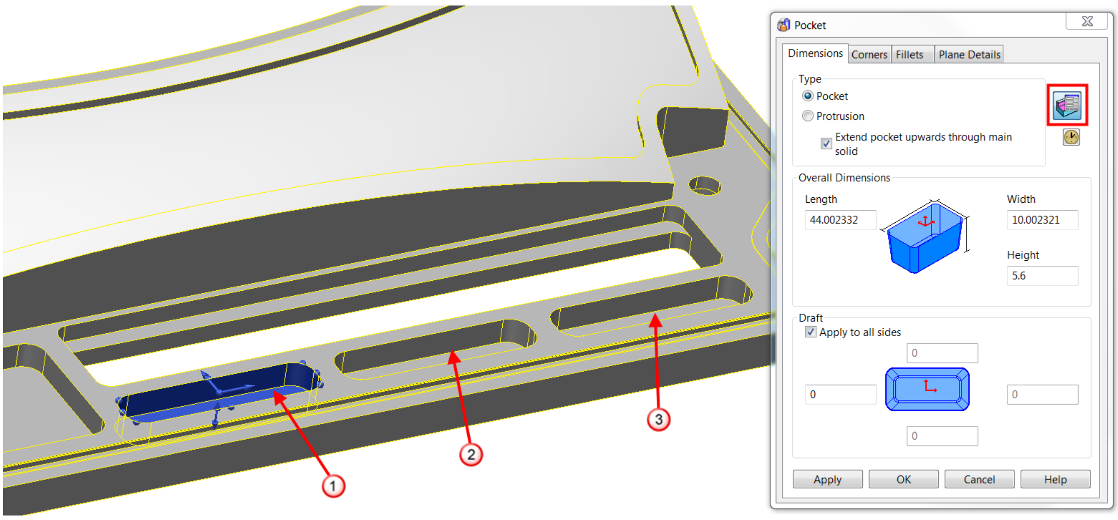


The **two hole features** are now **visible** in the **feature tree** and can now be **modified** as a standard solid feature.

6 Select Create a **Solid pocket**  from the **Solid features** toolbar. 

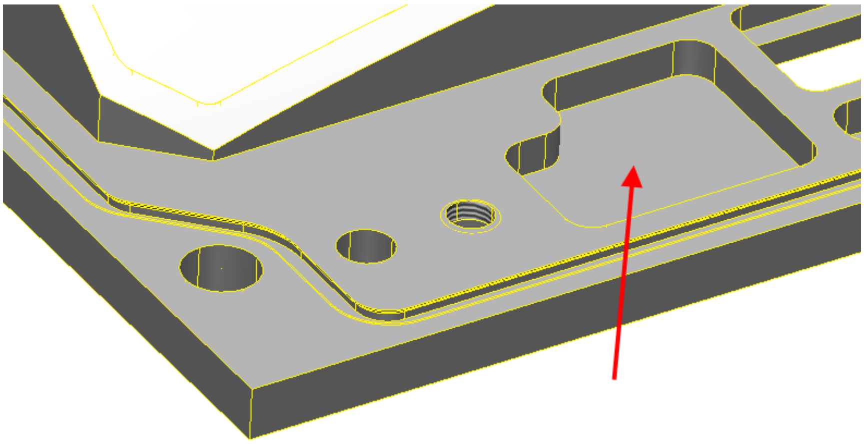
7 Toggle to **Feature recognition** mode. 

8 Zoom into the area shown and select the first pocket shown. 1



9 Select **Apply** to generate the pocket, then repeat for **pockets** 2 and 3.

10 **Do not close** the pocket dialog.

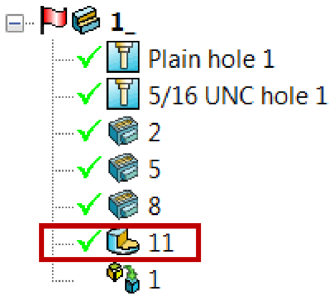


11 **Attempt** to select the **adjacent pocket** shown above.

PowerSHAPE does not recognise the **feature** as a **standard pocket**. In this instance the feature will be extracted as a **solid cut feature**.

12 Close the **Solid pocket** form then select **Solid cut**. 

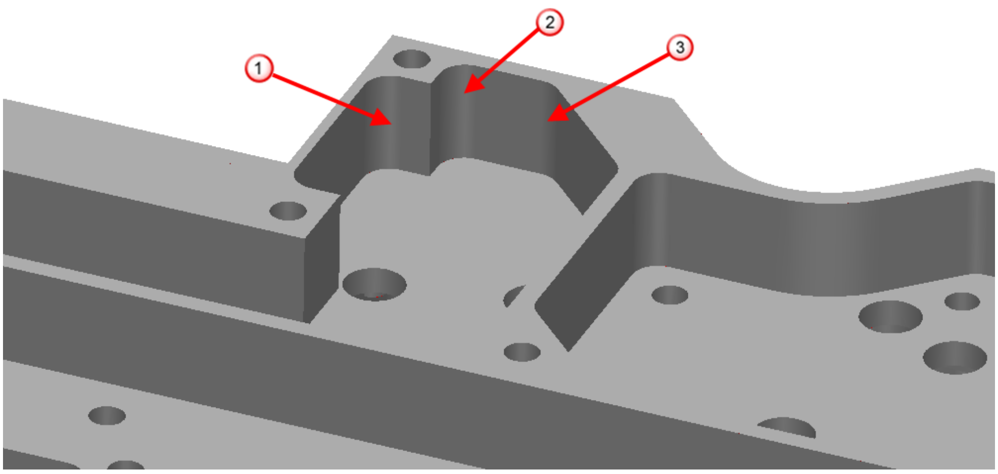
13 **Extract** the **feature** to add it to the **feature tree**.



The **Solid cut** feature is added to the bottom of the solid feature tree below the three solid pockets created earlier.

The **feature recognition** of **fillets** is similar in method.

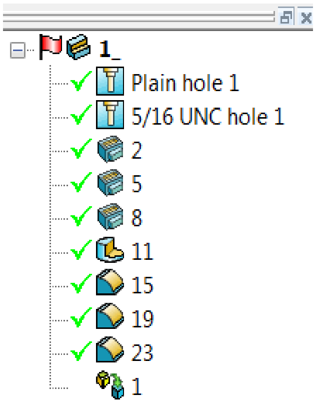
1 Select Create a **Solid fillet**  from the **Solid features** toolbar. 



2 Extract and create the three fillet features shown above.

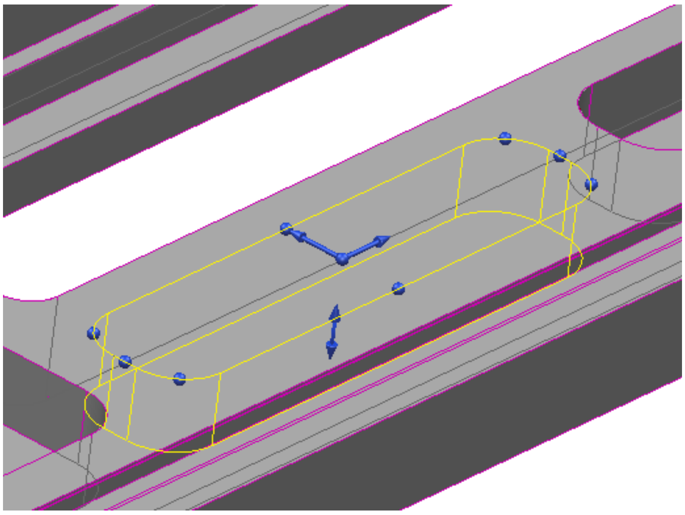
Modifying extracted features

Recognised features are modified in the same manner as if they were initially created directly in **PowerSHAPE**. All current features are displayed in the **Solid feature tree.** 

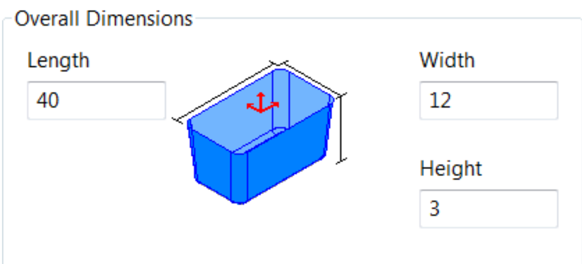


3 Display the dialog for the first pocket by **double-clicking** the **item** in the **feature tree.** 

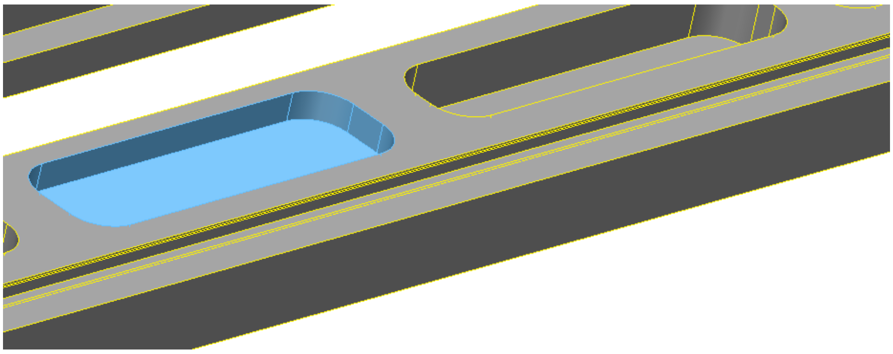
 *Or **right-click>Modify...***



The **pocket** can now be **modified dynamically** using the **drag handles** or **updating** the **Pocket dialog values** themselves.

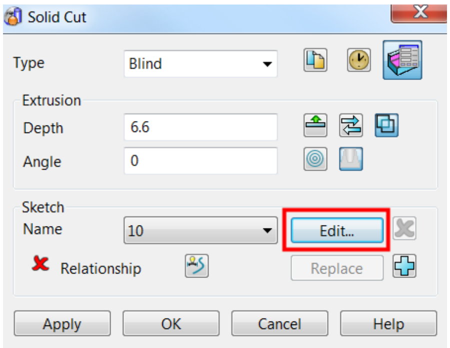


4 **Modify** the **overall dimensions** as above and click **OK** the **form** to **accept changes**.



All other **features** can be **edited** using the same method.

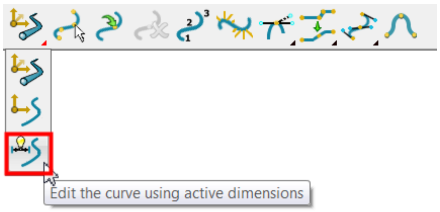
5 Display the **Solid Cut** dialog. 



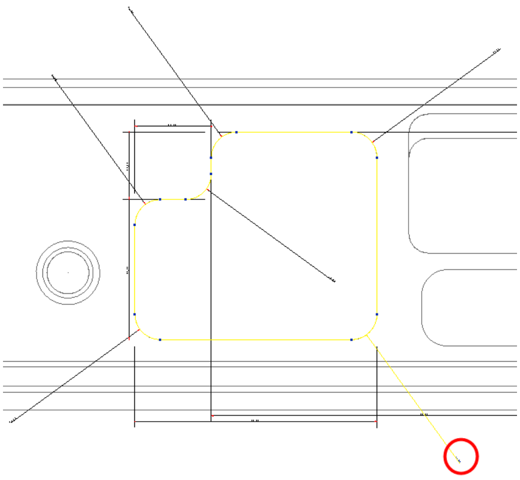
In this example the **actual profile** of the **cut** will be **modified** by altering the **sketch** which defines it. This process was also demonstrated in the **Surface Modelling chapter**.

A **wireframe sketch** was **automatically created** when the **feature** was **generated**.

6 Select **Edit** on the **Solid cut** dialog.

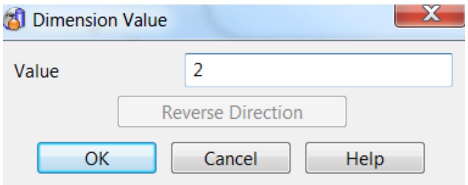


7 From the **curve editing** toolbar, Select **Edit with active dimensions**. 

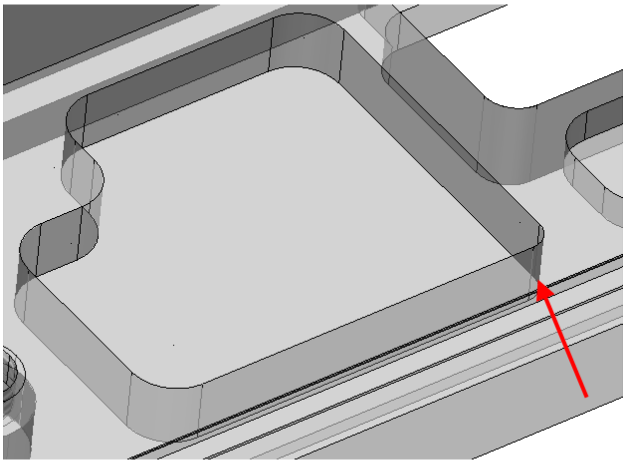


8 **Double-click** on the **bottom right corner dimension** (**circled red above**) to display the **Dimension value** form.

9 Change the dimension from **4mm** to **2mm**.



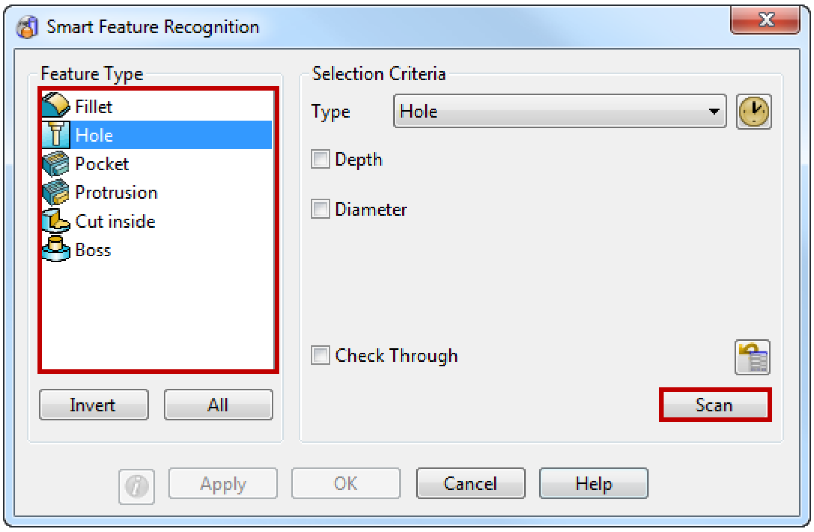
10 Select **OK** on the dimension form and the **Solid cut** dialogs.



The solid cut feature has updated with the new **2mm corner radius**.

Smart Feature Manager

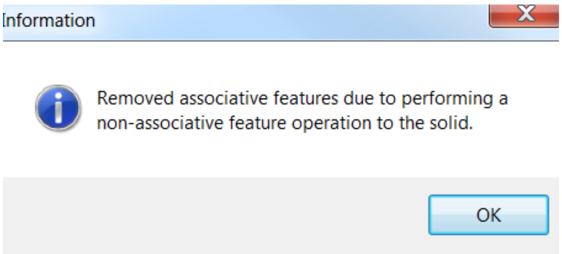
PowerSHAPE also allows **multiple features** and **feature types** to be recognised at once to speed up the process of recreating the solid tree. The **Smart Feature Manager**  is found in the **Solid Feature**  sub-menu. **Selecting** the **feature type** (which can be filtered down to specific sizes, sub-types etc) and then choosing **Scan** will search the model for features, adding them into the tree. These features can then be grouped automatically.



*The **smart feature manager** is a very powerful tool however it uses a lot of processing power. It is advisable to scan for **single feature types** at one time.*

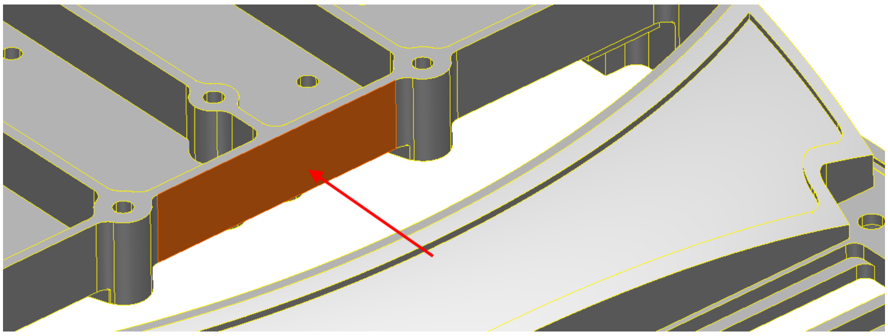
Maintaining Solid history

At this stage, If any direct editing tools were to be applied (e.g. Offset, Draft face), then **PowerSHAPE** displays a warning to the user.

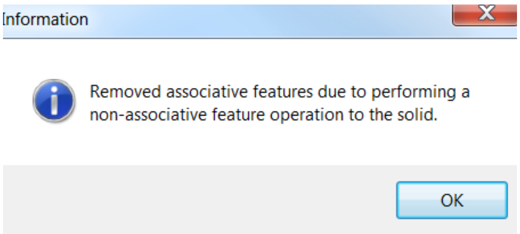


Applying the **edit** now would now **remove all previous feature history generated**. However the **following procedure** shows how to **apply** the **edit** and **maintain feature history**. A **rib feature** will be **thickened** to outline this important step if required.

1 **Select** the **single face** shown.

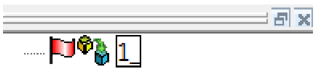


2 Select **Offset**  from the **General Edits** toolbar  and enter a value of **2mm**.



The information dialog is displayed.

3 Select **OK** to accept.



The offset has been applied but associated history removed.

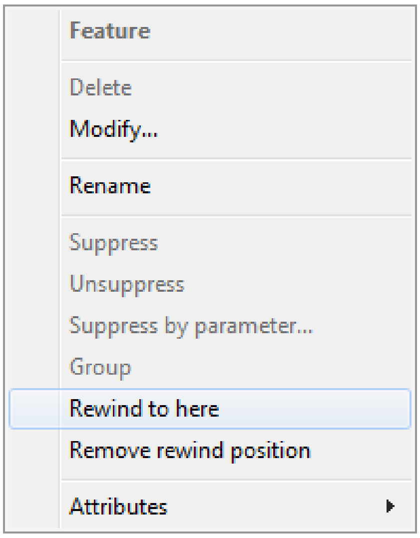
 *This is because a **non history edit** was applied to a part containing history.*

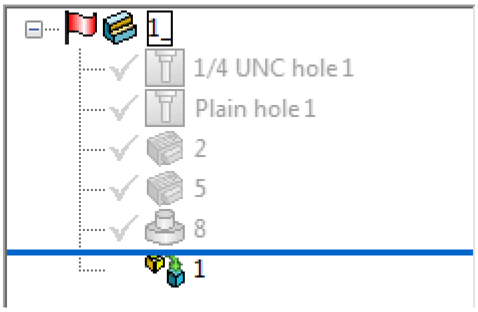
4 Select **Undo**  to revert to previous condition.

At the bottom of the feature tree, the original imported solid is displayed. 

Maintaining the Solid Tree: Method 1

5 Right-click on the original imported solid at the bottom of the history tree and select the option to **Rewind to Here**.



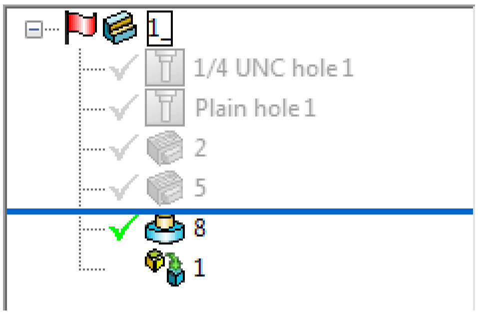


This will rewind the **recreated build history** until the original featureless solid. **Note** this will also suppress the features that are greyed out in the solid tree.

6 Select the **face** on the copied base solid and create on **offset** of **2mm** to **thicken** the **rib**.

The same warning message will still appear however anything greyed out above the ***blue rewind position*** *line will be unaffected.*

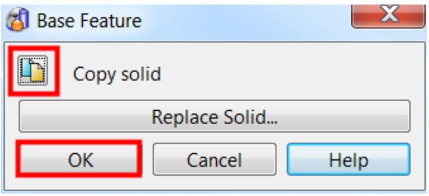
If a ***feature*** *in the* ***solid tree*** *needs to be edited using a non-history edit you can* ***move*** *that feature to the bottom of the tree, just above the base solid and* ***rewind*** *to just above to perform the edit. This will only remove a single feature which can be re-recognised easily.*



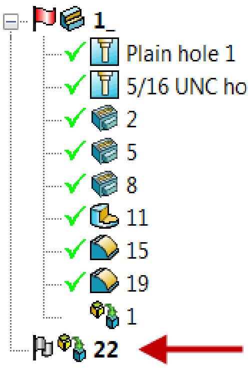
To remove the **Rewind Position** simply right-click in the solid tree on any feature and select the option to **Remove Rewind Position**.

Maintaining the Solid Tree: Method 2

7 Double click to **modify** the base solid 1. 1



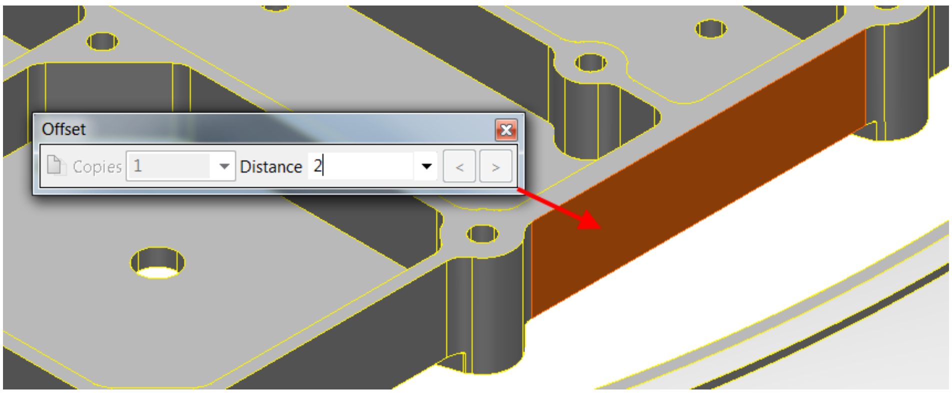
8 Select **Copy solid** then **OK.**



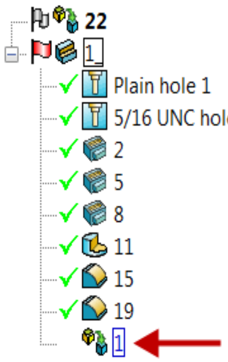
A copy of the base solid 22 has been copied out into the feature tree. Note this base solid copy has no history including all the feature recognition items created earlier (holes, pockets...).

9 Select and **Blank** the main solid with feature history. 1

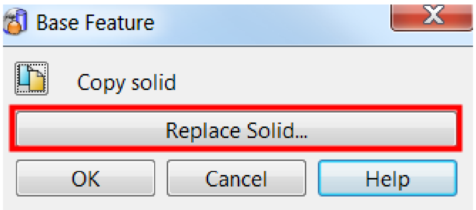
10 Select the face on the copied base solid and create on offset of 2mm to thicken the rib.



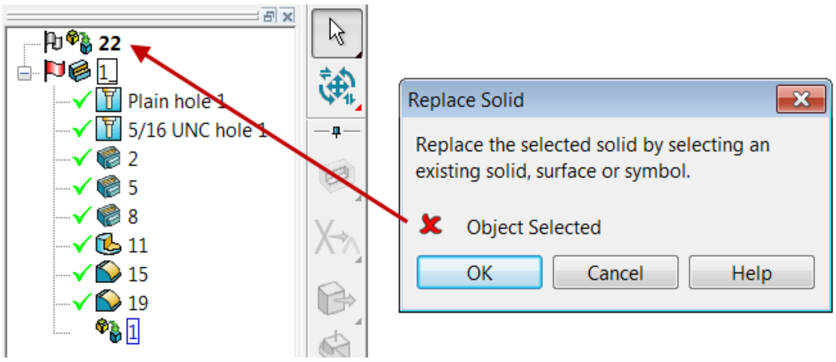
11 Unblank to bring back the main original solid with history.



12 As before, double click to modify the original base solid 1. 



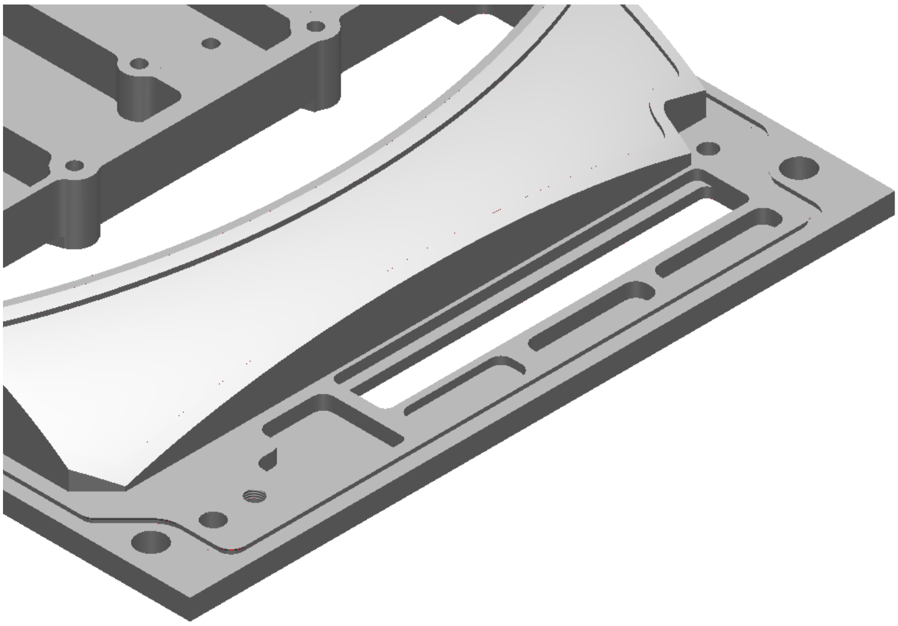
13 Select **Replace Solid**.



A replacement solid must be selected.

14 Select the copied out base solid from the feature tree. 

15 Select **OK.**



The **original base** solid has been **replaced** with the **modified solid** (**thickened rib**) whilst maintaining the previous history.

This shows the **powerful ability** of **PowerSHAPE** to **combine direct** and **feature recognition features** on a **component** which was **imported** with **no history**.

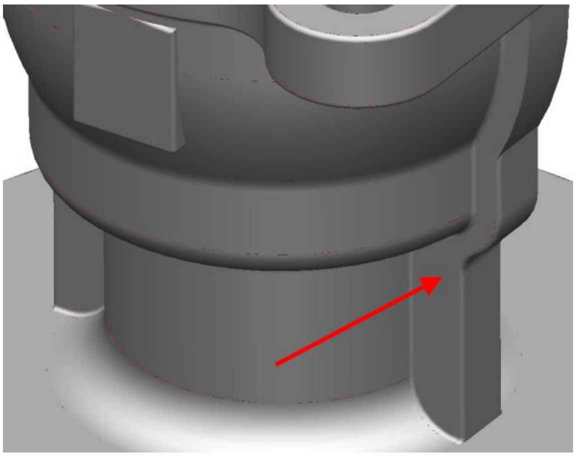
16 **Save** the **model** as.....

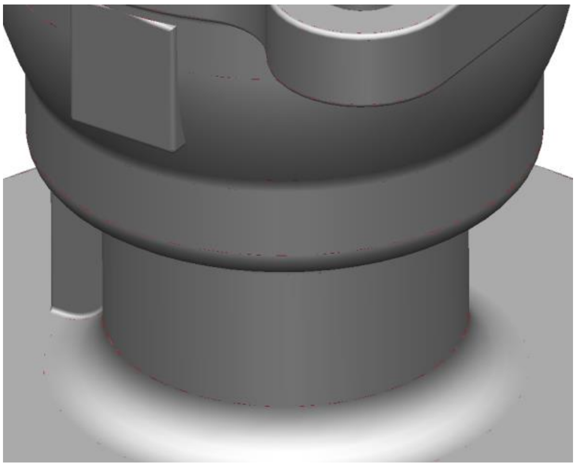
....**PowerSHAPE Data****Direct modelling plate.psmodel**

Exercise 10: Pump housing

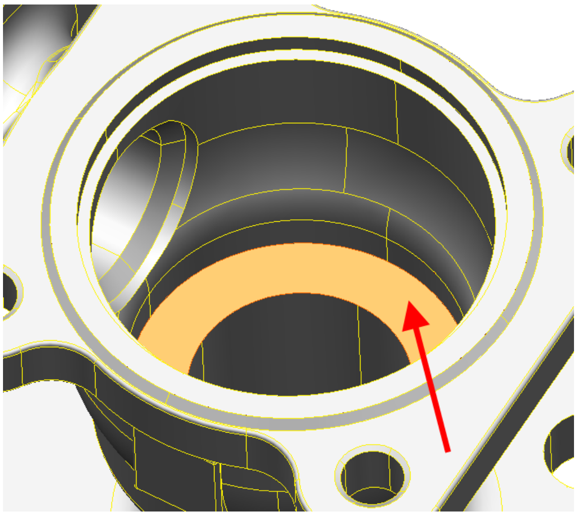
1. **Open** the surface model
   .....\PowerSHAPE_Data**pump housing.psmodel**

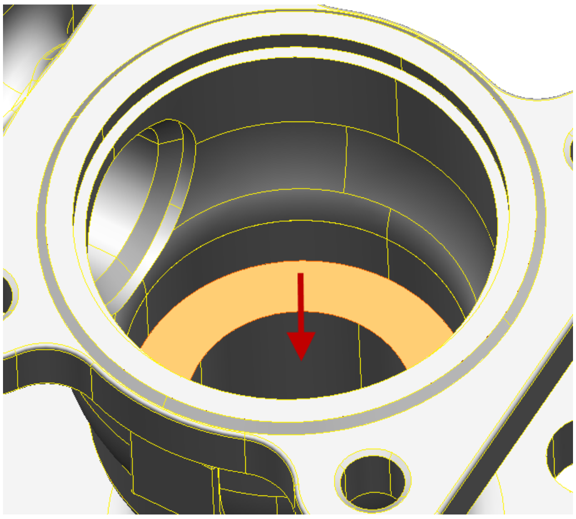
2. *Select* all *the surfaces and convert to a solid.* 
3. Using **direct modelling** to apply the following design changes to the pump housing.
4. **Remove** the complete outer Rib feature.



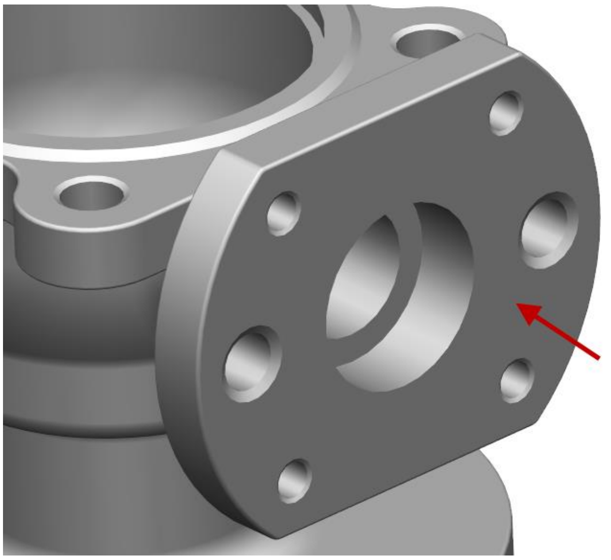


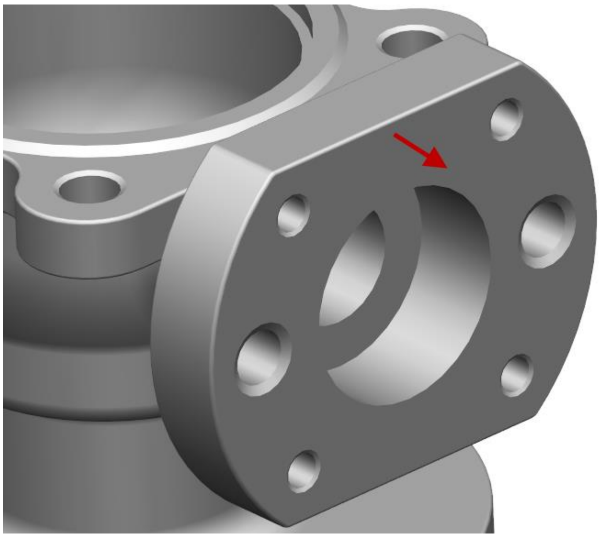
5 Lower the inner step face by **5mm**.



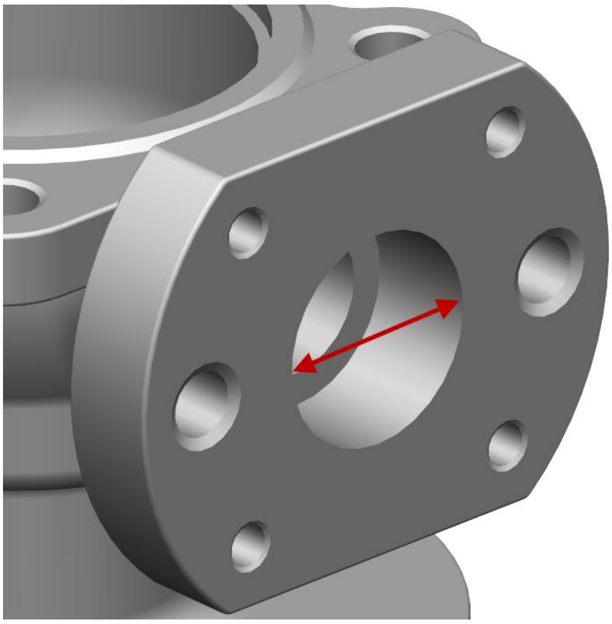


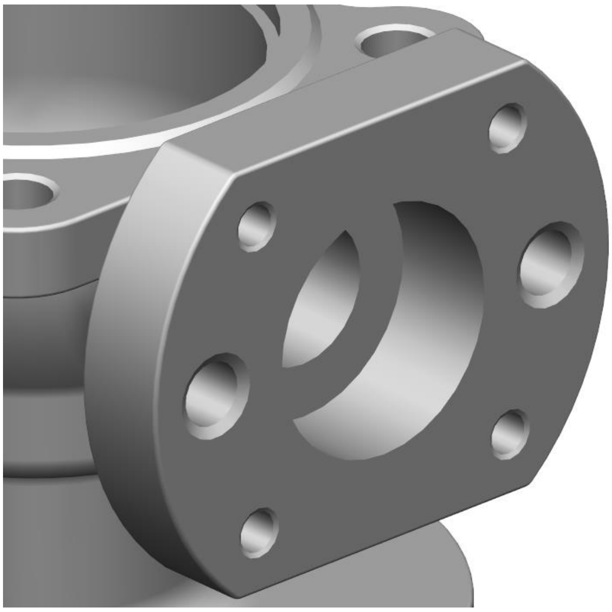
6 Thicken the end plate by an additional **5mm** whilst maintaining the edge fillets and hole chamfers.



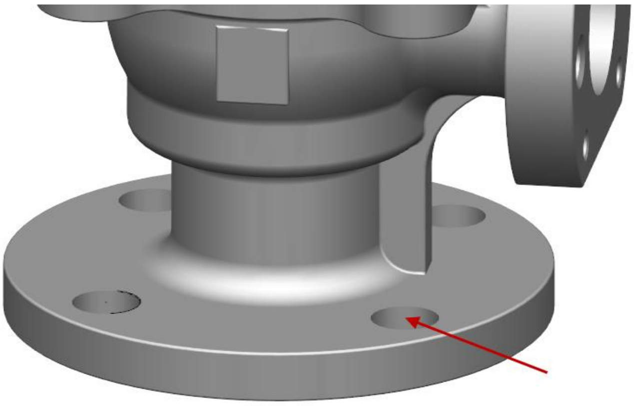


7 **Change** the outer bore diameter to **40mm**.

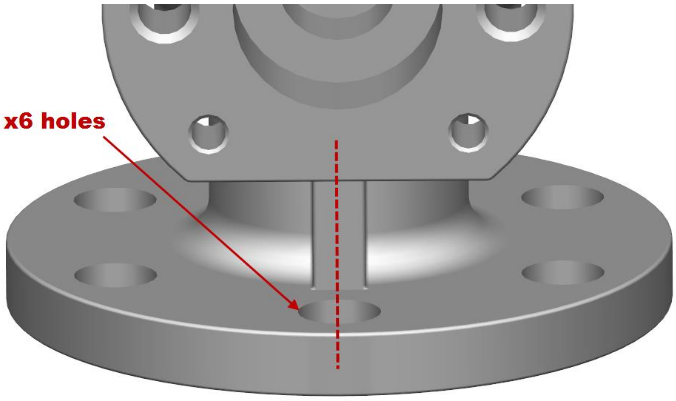




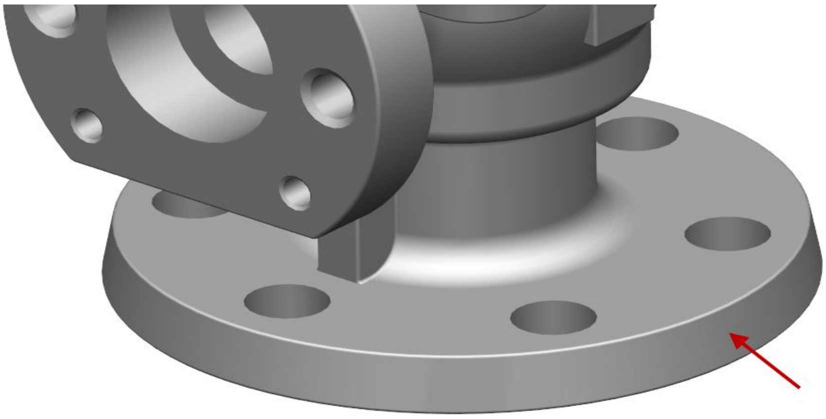
8 Modify the base plate to have **six holes** (from four) on the same **PCD** (Pitch circle diameter).



Ensure **one** of the **holes** is **centrally located** to the **main vertical rib** as shown below.



9 Finally, apply a **draft angle of 15 degrees** to the **bottom face plate** as shown.



10 Save the model as.....

…/PowerSHAPE-Models/**Ex10 pump housing.psmodel.**

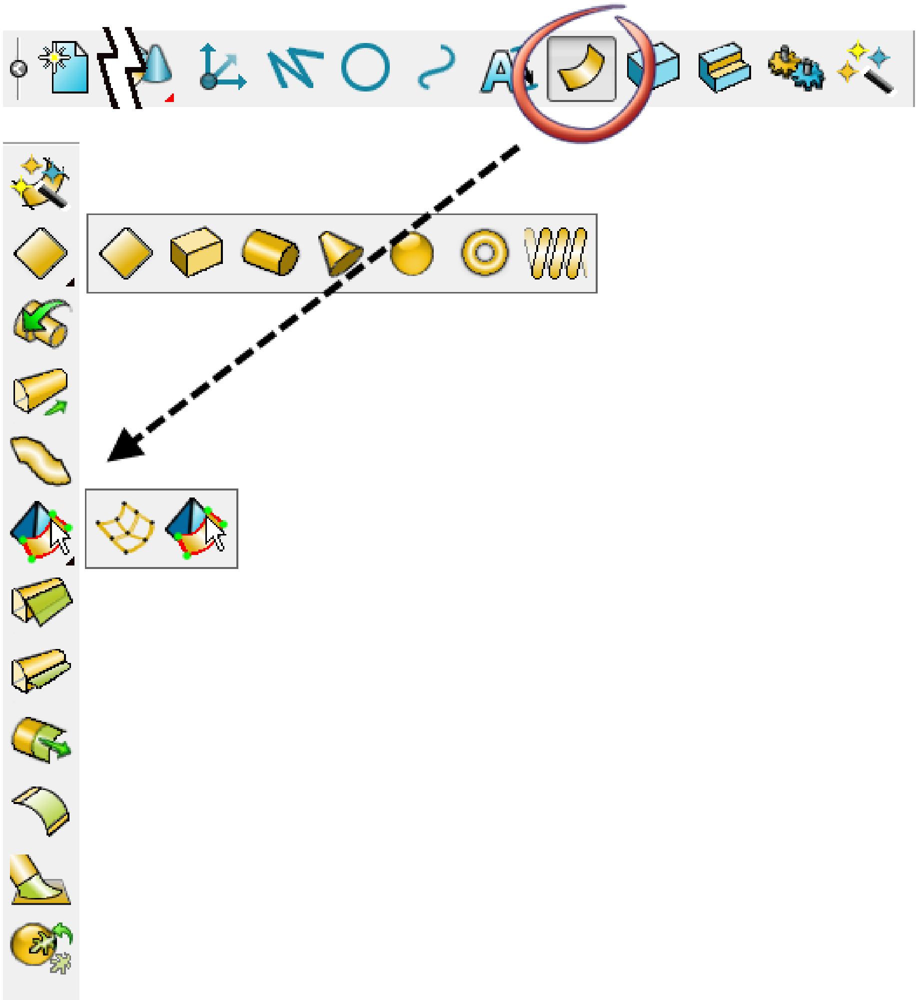
8. Surface Modelling

What is a Surface?

A **surface** is a **skin** of **negligible thickness** stretched across a defined **2D** or **3D area**.

There are **3 main types** of **surface** supported by **PowerSHAPE**: **Primitives**, **NURBS** and **Power Surfaces**. Each surface type has **different constructional attributes** and **editing capabilities**.

When **Surface**  is selected from the **main menu**, the following buttons are displayed to the left of the **graphics area**.



A **definition** of **each function** is as follows.

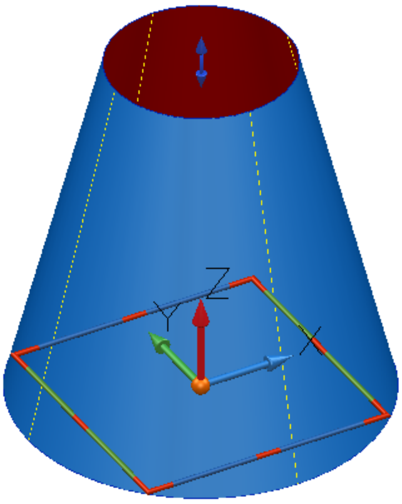
- Smart Surfacer
- Create a primitive surface:
  - plane
  - block
  - cylinder
  - cone
  - sphere
  - torus
  - spring
- Create a surface of revolution
- Create an extruded surface
- Create a bead surface from wireframe
- Create a surface from patches
- Create a surface from a mesh
- Create a draft surface
- Create a split surface
- Create an extension of a surface
- Create a fillet surface
- Create a blend surface
- Wrap Triangles

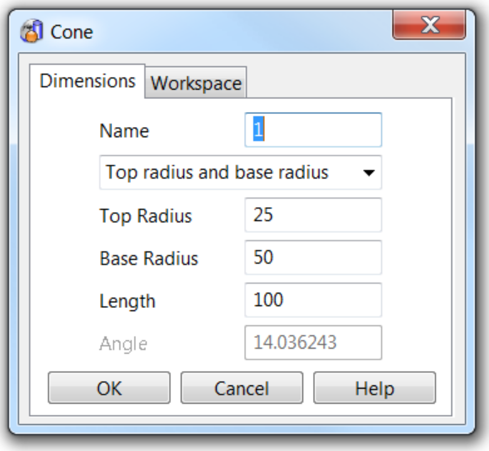
Primitive Surfaces

PowerSHAPE can quickly create a range of simple surfaces defined by a **few basic parameters**. These are **Primitives**; they include **seven standard shapes**, **extruded surfaces** and **surfaces of revolution**. Using **primitive surfaces**, only the **basic dimensional parameters** can be **modified** with the original **defined shape** being fixed. Other types of surfaces used in PowerSHAPE include **NURB surfaces** (frequently obtained from imported data) and **Power Surfaces. NURB surfaces** also have **limited editing capability** and along with **primitive surfaces** must be **converted** to **Power Surfaces** for more **powerful** and **complete editing capability**.

The **PowerSHAPE 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 Surface** is shown selected with the **Cone editing** dialog displayed.

The Surface Cone is a skin with the top and base 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 **PowerSHAPE**. For this to be possible, **PowerSHAPE** supports other types of **surface definitions** such as **NURB surfaces**. **PowerSHAPE** also creates **NURBS Surfaces** during certain applications or if specified by the user in **PowerSHAPEs Surface options**.

Conversion of Primitives and NURB to Power Surfaces

If you want to perform more **complex modifications** to the shape of a **Primitive** or **NURB Surface**, you must first **convert** it to a **Power Surface**. With a **Power Surface**, you have access to a **full range of editing options** through a comprehensive **Surface/curve editing** toolbar or **Dynamic** operations.

*You **cannot convert** a **Power Surface** to a **Primitive** or **NURB Surface**.*

Select **Menu > Tools > Options > Objects > Surfaces** ensure that in the section labelled **Primitives** that **Create as NURBS** is **deselected** if direct **conversion** from **Primitives** to **Power Surfaces** is required. Otherwise, when **Convert Surface** is applied to a **Primitive**, the surface becomes a **NURBS** which, in turn, will have to be **converted** to a **Power Surface**.

Several of the surface creation options directly create a Power Surface.

Power surfaces

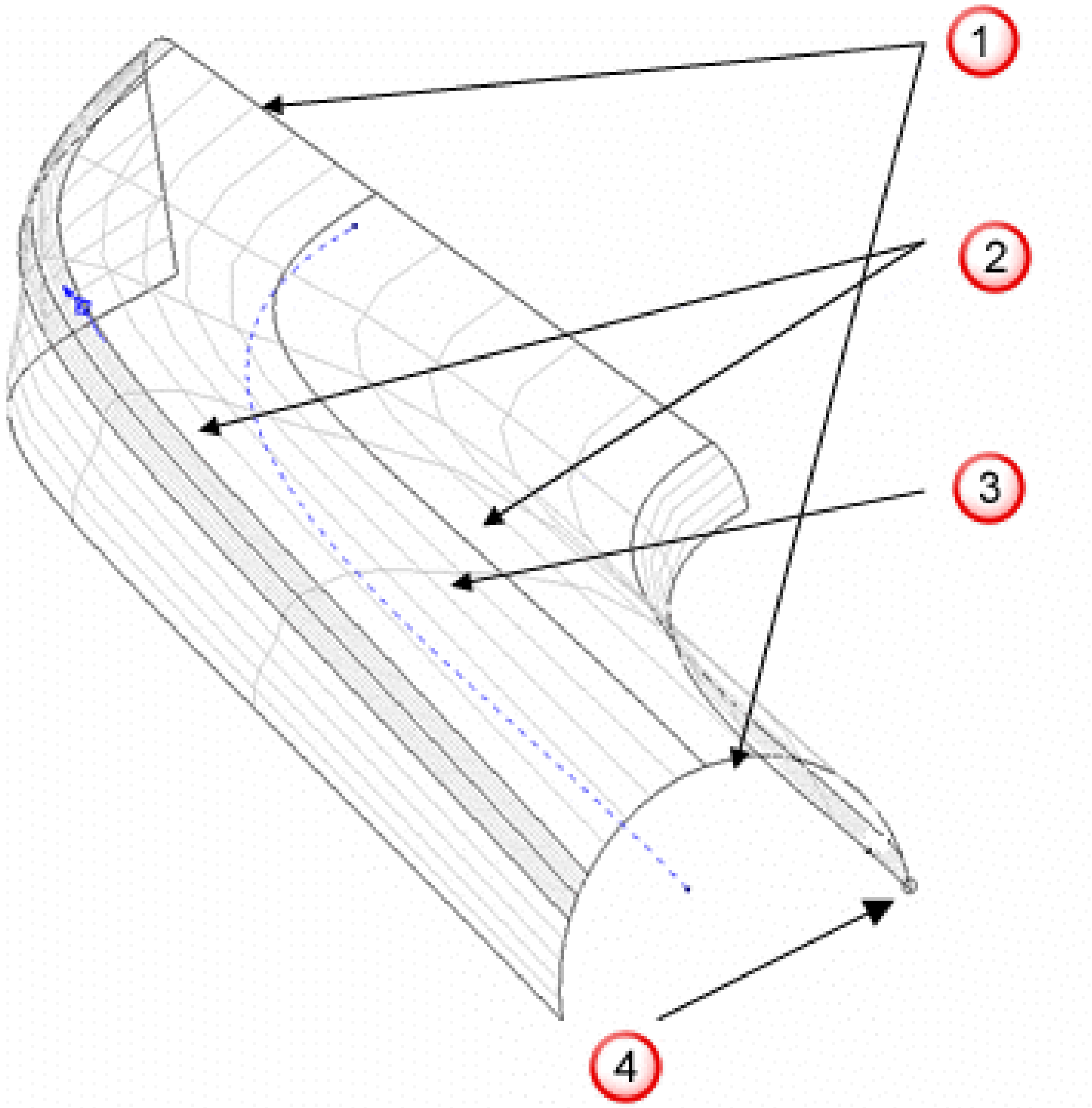
A **Power Surface** is based on a network of 4 sided wireframe elements appearing as curves along **(longitudinals)** and across **(laterals)** the surface.

A **Power Surface** can generate **complex forms** while **retaining full editing capability** including the direction and magnitude through **surface curve intersections** (**Control Points**).

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 1 This surface contains the minimum, **2 laterals** that run **across** the **surface direction**.

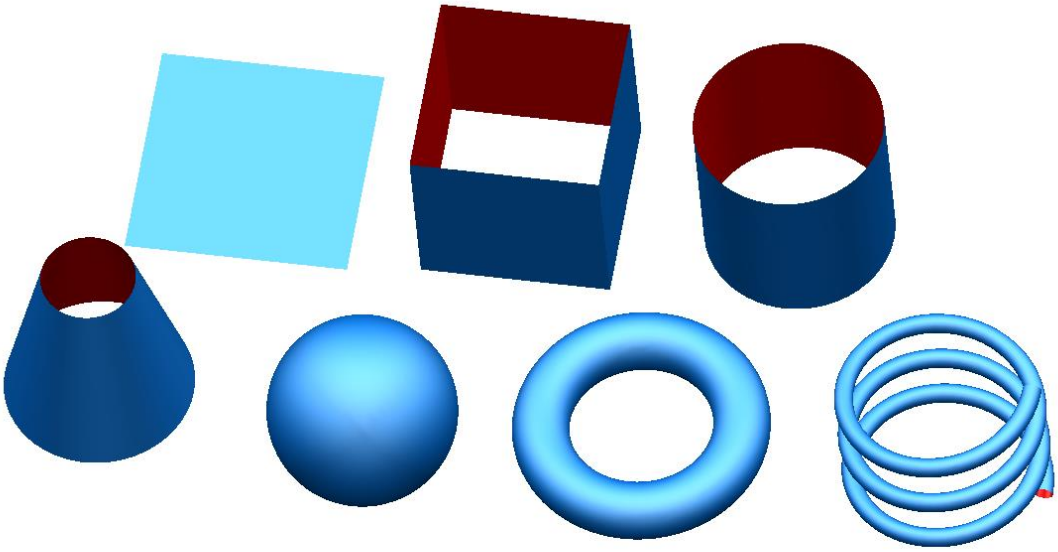
Longitudinals 2 This surface contains a total of **7 longitudinals** flowing from **corresponding points** from the **first** to the **second lateral**.

Spine (Drive Curve) 3 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 4 This 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**.

Standard Primitive Surfaces

There are seven standard **primitive surfaces** in **PowerSHAPE**: **Plane, Block, Sphere, Cylinder, Cone and Torus and Helix**. Primitive surfaces are generated with minimal data input and can be an ideal starting point for many applications. Primitives are coloured blue or gold in the toolbar to distinguish them from other surface creation options.



When you create a primitive, it is given a size that is proportional to the zoom of the screen. It can be modified as required. A primitive can be moved, copied, rotated, intersected and filleted. However, if you need to alter the defined shape by actions such as moving surface points or adding extra sections, you must first convert the primitive to a Power Surface.

All primitives are created in the direction of the active Principal Axis (by default this is along the Z-Axis).

Extrusion Surfaces

This type of surface is formed as an **extrusion** of a wireframe entity, normal to the planer base of the wireframe (default). If required, it is possible to change the settings in **Tools > Options > Objects > Surfaces** for the extrusion surface to generate along the active X, Y, or Z axis. If several items are selected for extrusion, a series of separate surfaces are generated.

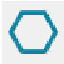
As with standard primitives, you can change only **basic parameters** for an **extrusion surface**, unless you **convert** it to a **Power Surface**.

By default, the wireframe used to create the extrusion surface is deleted. To retain the wireframe, select **Tools > Options > Object > Surfaces** and select- **Keep wireframe (extrusions and revolutions)**.

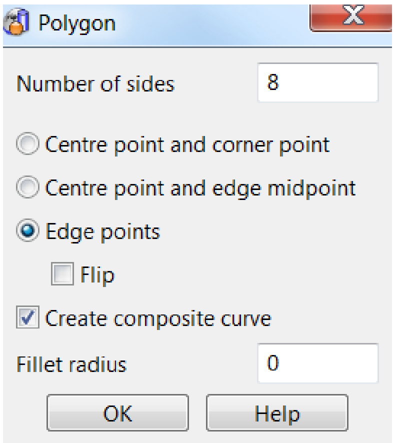
Extrusion Example 1

This example demonstrates creating a **wireframe polygon** which is then used to generate an **octagonal extrusion surface**.

1 Open **New Model**. 

2 Select **Polygon**  from the **Line** toolbar. 

The **Polygon** dialog is displayed. This allows you to specify the **Number of sides** in the polygon. The option to **Create composite curve** provides the ability to create a polygon as a single entity. If this option is deselected, the sides of the polygon will consist of single lines.



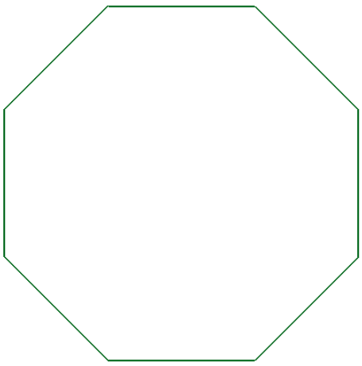
3 In the **Number of sides** box, type **8**. Select **Edge points** and select **Create composite curve**.

The **start** and **end points** of the **first span** need to be defined before closing the form.

4 Enter **0** in the **Command Input** box and press **Enter**.

5 Enter **50** in the **Command Input** box and press **Enter**.

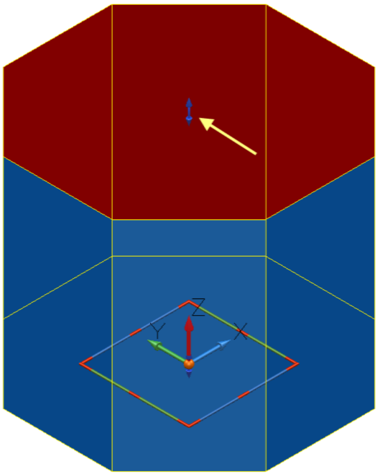
6 Select **OK** to close the **Polygon** dialog.



The **wireframe** is **completed** as a **single closed composite curve**. This will be used as the shape of the **Extrusion Surface**.

7 Select the composite curve and switch to **View Iso1**. 

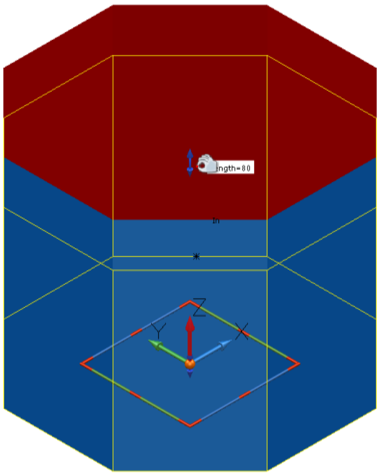
8 Select **Extrusion**  from the Surface toolbar. 



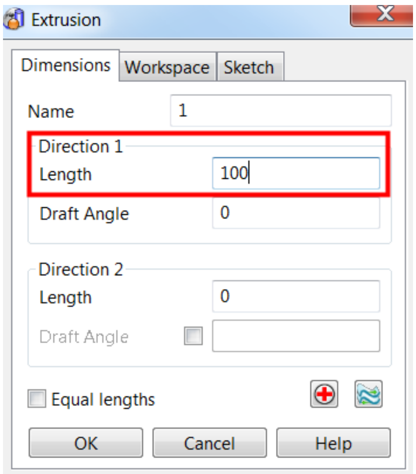
This produces an **extrusion surface** from the selected **composite curve** up the **Z-axis** by a default value.

The **extrusion** has a set of **double arrows** (indicated by **yellow arrow above**), which are used to change the **length up** or **down** the **Z axis** and, if required, to create a **negative extrusion** (lower set). The **workplane handles** can be used to **dynamically edit** the **position** and **orientation** of the surface.

9 Drag the arrows to a length of **50**.



10 Alternatively Double-click on the **surface** to display the **Extrusion** dialog. This can be used to enter or **modify** the **dimensional values**. The extrusion can be given a positive or negative draft angle.



Using the **Workspace Tab** the extrusion can be **repositioned** by entering new **origin coordinates**, and aligned to, or **rotated** about an **axis**.

11 Enter a length of **100mm** and select **OK**.

12 **Deselect** the **surface** by clicking away from it. The **extrusion surface** is **completed**.

PowerSHAPE can also **simultaneously extrude several separate geometry** items to make **individual surfaces**.

13 Select the extrusion surface.

14 Right-click and select **Delete** from the popup menu. Alternatively, press the **Delete** key.

The **surface** has now been **deleted**; the original composite curve does not exist either. This was deleted by default when the extrusion was created. To automatically retain curves, select **Tools > Options > Object > Surfaces** and select **Keep wireframe** (**extrusions and revolutions**).

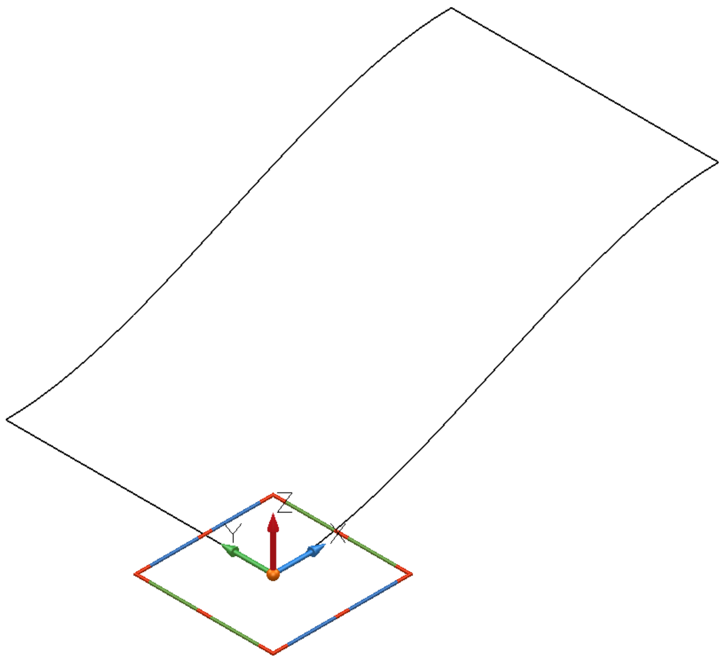
Extrusion Example 2

A **3D wireframe** can be used to create an **extrusion surface**.

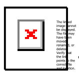
1 **Import** the model from:

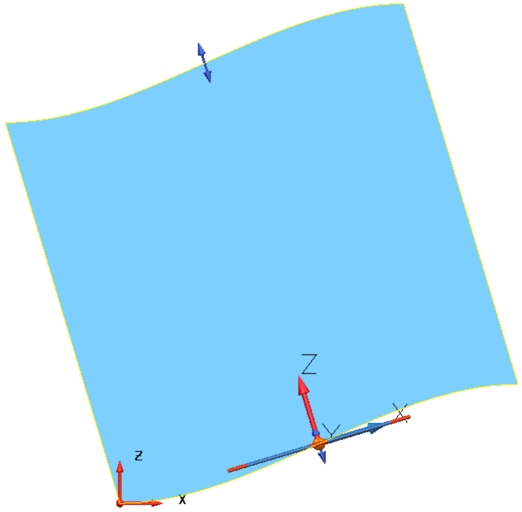
.....*__PowerSHAPE Data__**__3D_Wireframe.dgk__*

2 Activate **Workplane 1** from the **Workplane selector**.



An **extrusion surface** will be created from the **imported 3D composite curve**.

3 Select the **composite curve** and create a **surface extrusion**  to a **length of 50mm.**

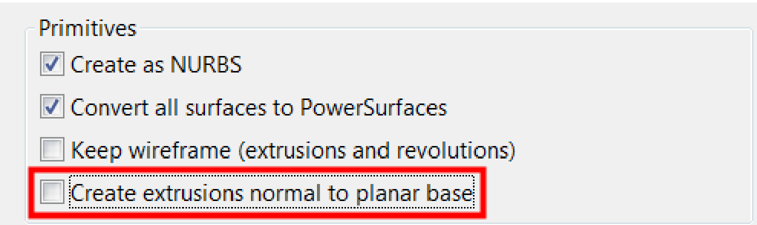


The **extrusion direction** is **normal** to the **planar base**. This default can be changed.

4 Select **Undo**  to return to the stage prior to creating the extrusion surface.

5 Select **Tools > Options > Object > Surfaces**.

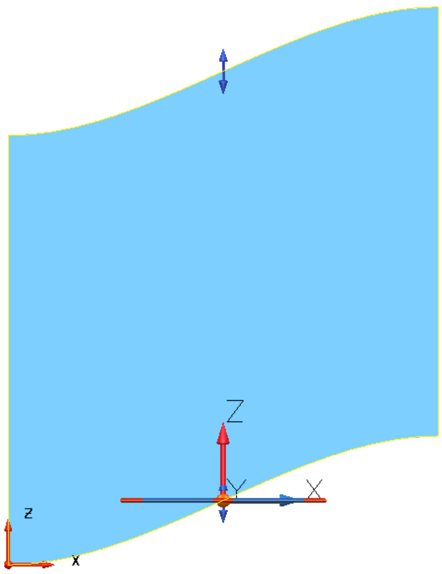
The **Primitives** section controls the default behaviour when primitives are generated. By default, **Create extrusions normal to planar base** is selected.



6 Deselect **Create extrusions normal to planar base** and select **OK**.

7 Select the **Z Axis**  , and select the **composite curve.**

8 As before, create a surface extrusion.

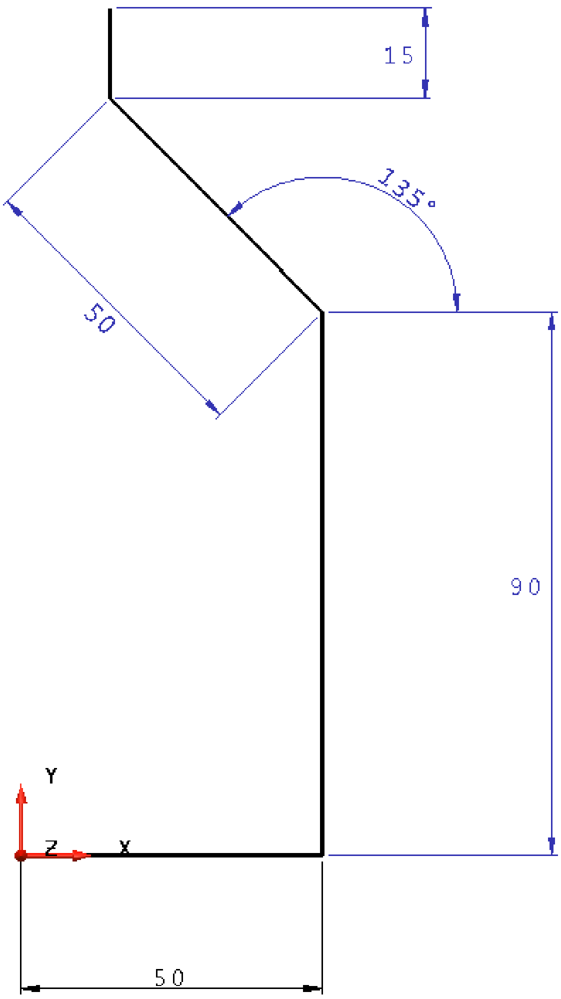


9 Select **File > Close** without saving the model.

Surfaces of Revolution Example

A **surface of revolution** is created by **rotating single geometry** or a **composite curve** around a specified **Principal Axis**.

1 Open **New Model**  and create a **Workplane at 0** on the **XY 'face'**.

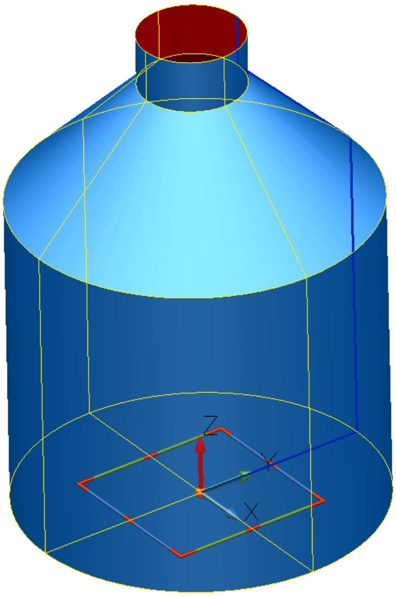


This shape will be connected with a **composite curve**. This curve will then be rotated **360°** around the **workplane** to generate a **surface**. The **correct axis** for **rotation** must be **pre-selected** using the **principal axis selector**. In this case, it can be clearly seen the **Y axis** is required for the **correct revolution**.

2 Select the **Y-Axis** .

3 Create a **composite curve** of the section geometry, and select the wireframe.

4 Select **Surface of Revolution**  from the **Surface** menu. 

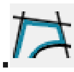


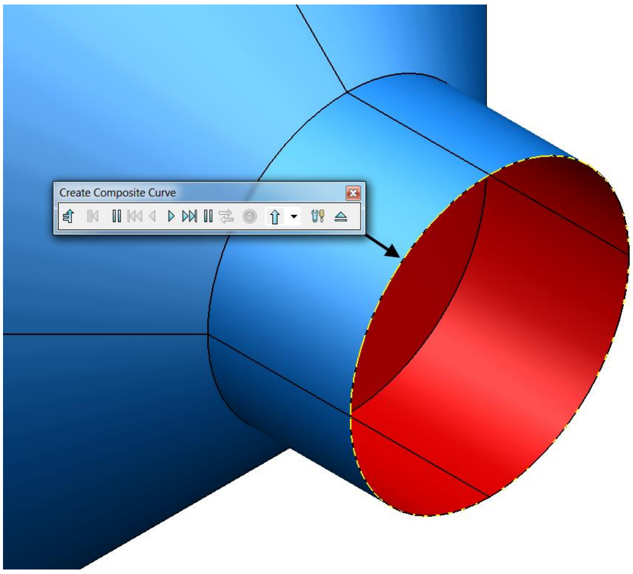
 *The selected **composite curve** is **spun** around the **active axis** (**Y**) producing the **bottle shape**.*

The **red side** of the **surface** represents the **negative side.** If the cylinder **appears red** following the **revolution**, then **reverse** the **surface** by selecting  from the **Surface Edit** toolbar, or alternatively **right-click>reverse**.

An end cap will be created by generating a **curve** around the end profile, and applying the **Fill-In surface** option.

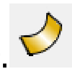
5 Select the **view Iso3** (**Ctrl+7**). 

6 Create a **Composite Curve** on the **open edge.** 

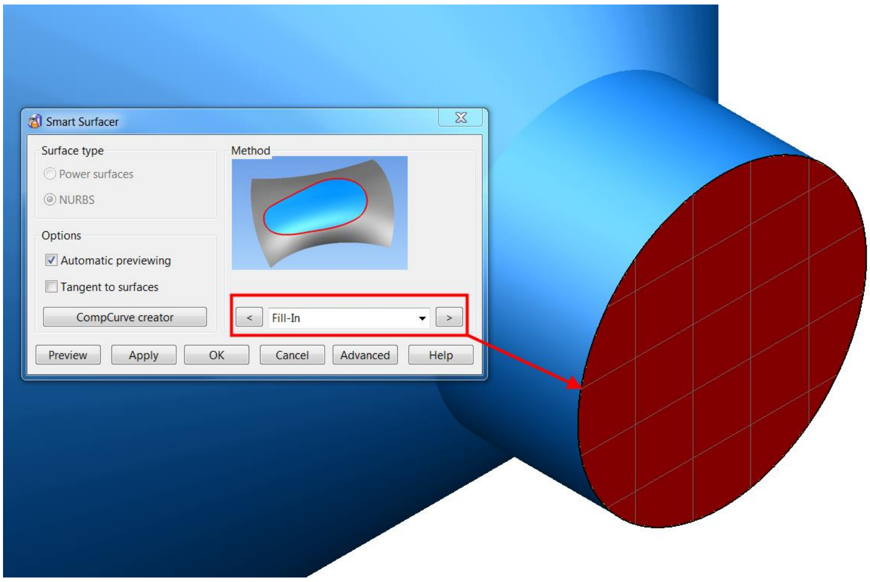


7 Select the new **composite curve**.

A fill-in surface can be generated from a composite curve or a series of wireframe entities. It is useful for filling in gaps in models.

8 Select the **Smart Surfacer**  from the Surface toolbar. 

9 In the drop-down list, select **Fill In** (if not already selected).



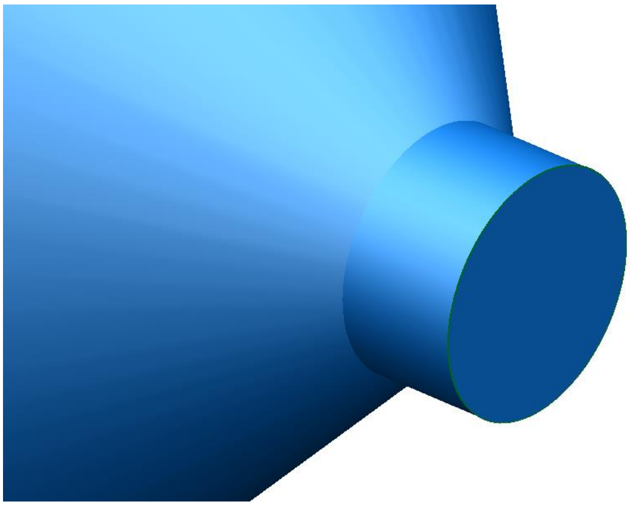
This Wizard will be used in detail later in the chapter.

10 Select **OK**.

The surface is generated but negative side up.

11 **Right-click** and select **Reverse** from the menu.

The surface is reversed showing the user-defined, coloured side, which is the **outside**.



12 Select **File > Save As** and save the file as:

.....***PowerSHAPE Models******Primitive cylinder.psmodel***

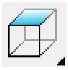
*When you **save** a model the **undo/redo history** is **reset**. Only changes made after the last save can be undone or redone.*

13 **Close** the **model**.

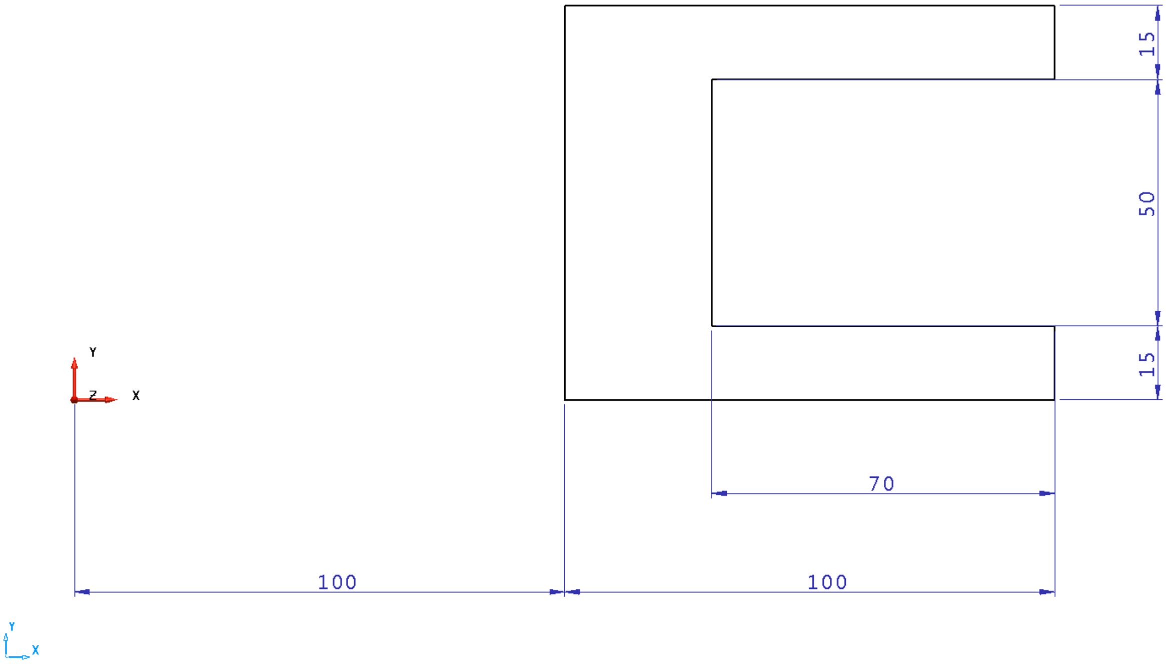
Wheel Example

Create the following **wheel shape section**. This will then be altered using various commands.

1 Open a **New Model** and **Create a Workplane** at **0**.

2 Select **View from top**. 

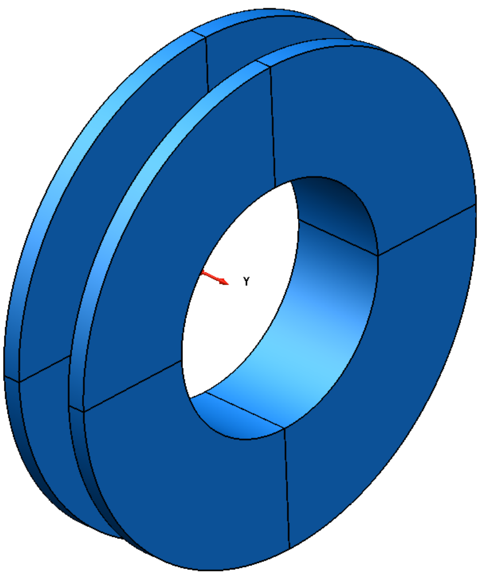
3 Generate the **basic wireframe shape**.



4 From the **Curve** menu create a **Composite Curve** of the **wireframe**.

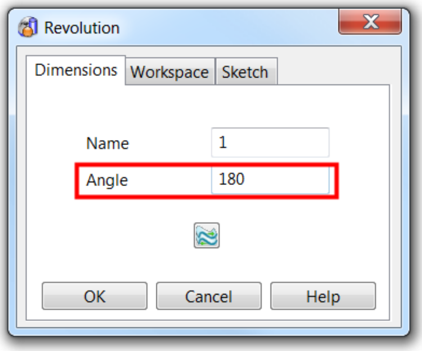
*With **simple composite curves** with an obvious route, holding the **Alt** key and **clicking** the **wireframe** will create the **composite curve**.*

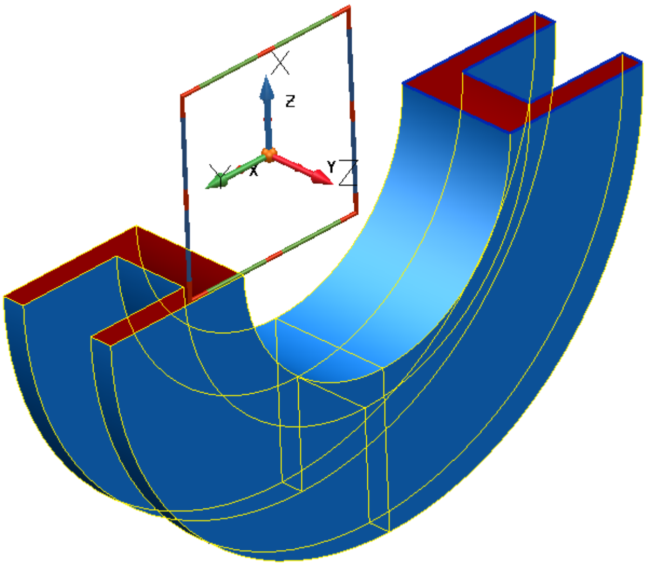
5 Select the **composite curve** and create a **surface of revolution**. 



6 **Double-click** the **surface**. The **Revolution** dialog appears showing the workplane options as well as the **Angle of Rotation**.

7 Change the **Angle** to **180°** and select **OK**. The surface is generated through an angle of 180 degrees.





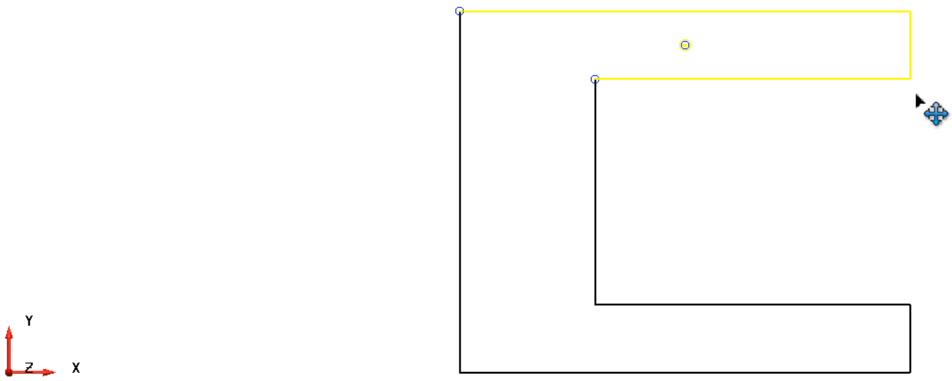
*Due to the properties of **surface models** as described earlier, the ends of the **primitive surface** will not be **capped**.*

Changing the shape of the surface of revolution

To alter the physical shape of the surface of revolution, the original wireframe must be altered. The wireframe can edited beforehand in the normal way or after the surface has been generated. Both methods will be shown.

1 Click on **Undo** twice to return to the original wireframe curve.

2 Select **Stretch Object** from the **General Edits** toolbar.

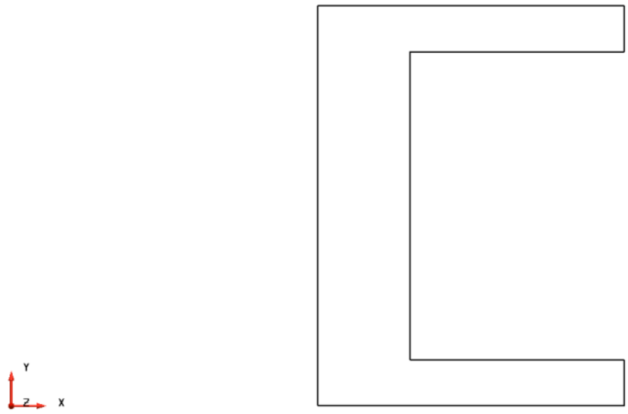


3 Select the **top half** of the shape, by **dragging a box** over it as shown.

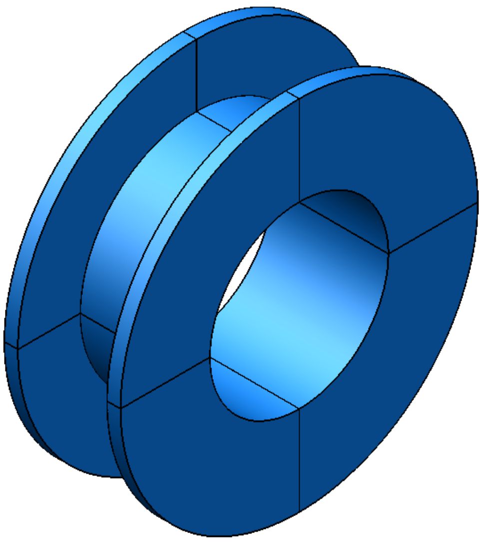
*Ensure the geometry to select is **completely** is within the **drag box**.*

4 Enter **0 50** to move the selected parts **50** mm along **Y**, stretching the back and inner wall.

The top half is dragged upwards and the connecting geometry is stretched.



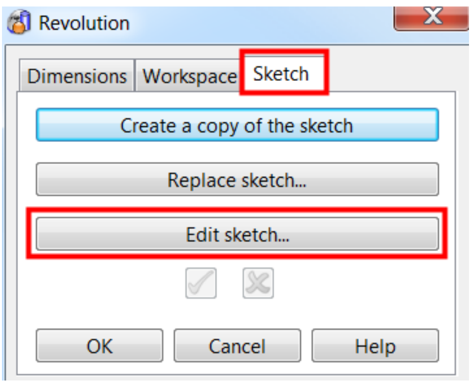
5 Select all the **wireframe** and **recreate** a **surface of revolution**. 



6 **Double-click** the **surface** to display the **Revolution** dialog.

 *The surface will be **modified** by directly editing the wireframe.*

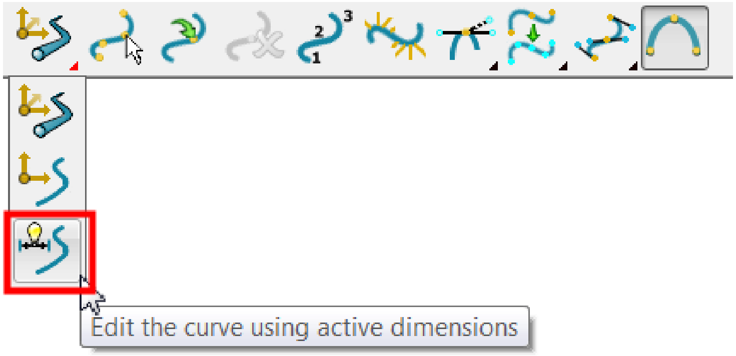
7 Select the **Sketch** tab.



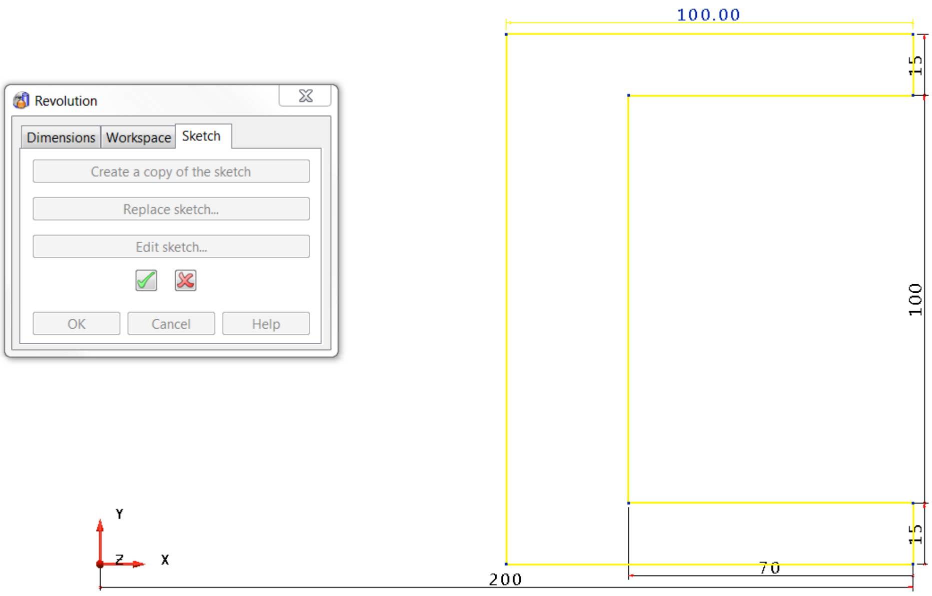
 *If required, a copy of the original wireframe can be created.*

8 Select **Edit Sketch...**

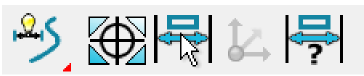
9 From the **Surface/Curve editing** toolbar, select **Edit the curve using active dimensions**. 

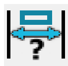


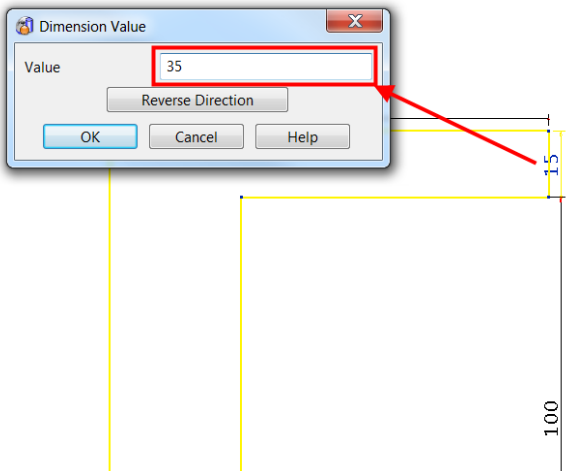
PowerSHAPE undraws the **surface** and displays the **dimensions** for the **wireframe**.



These **dimensions** can be **double-clicked** and **editted**. The **surface editing toolbar** now **displays additional functionality**.



10 Select the upper **15** dimension then **Edit active Dimension.** **.**

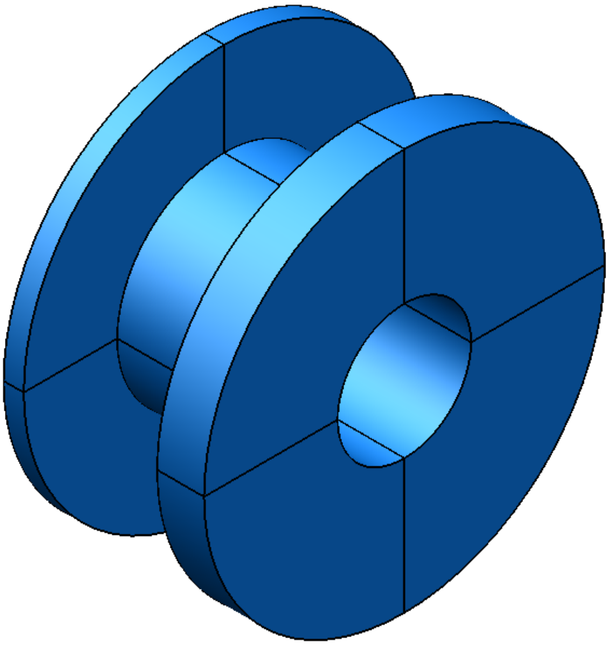


11 In the **Value** box, enter **35** and select **OK**.

12 Change the **200mm dimension to 150mm**.

13 In the Revolution dialog, select the **Finish** button .

14 Select **OK** to accept and close the form.



The surface revolution is updated to reflect the wireframe changes.

15 Select **File > Save As** and save the file as:

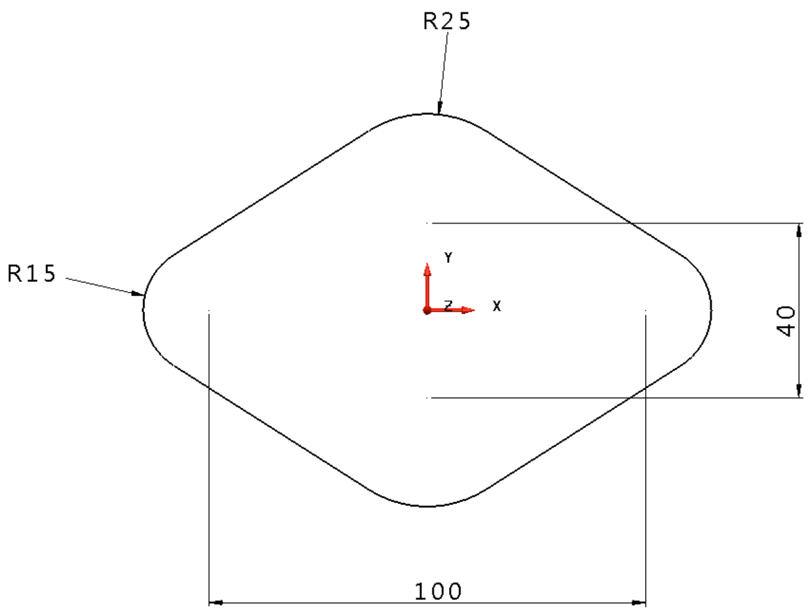
.....\PowerSHAPE-Models\Primitive wheel.psmodel

16 **Close** the model

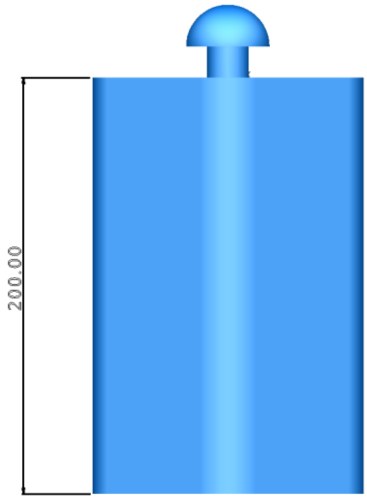
Exercise 7: Surface bottle

1 Using the information provided, create the surface bottle shown.

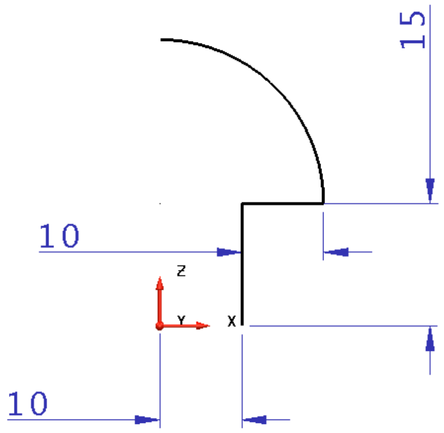
- The following wireframe defines the sectional shape of the body.

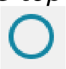


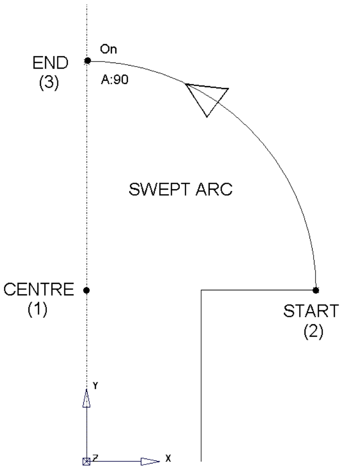
- The main bottle body is **200mm in height.**



- The following wireframe defines the profile of the bottle top.



Create the bottle top arc using  *arc through centre, radius and span from the circle menu.* 



- Ensure fill in surfaces are applied to both top and bottom of the body.

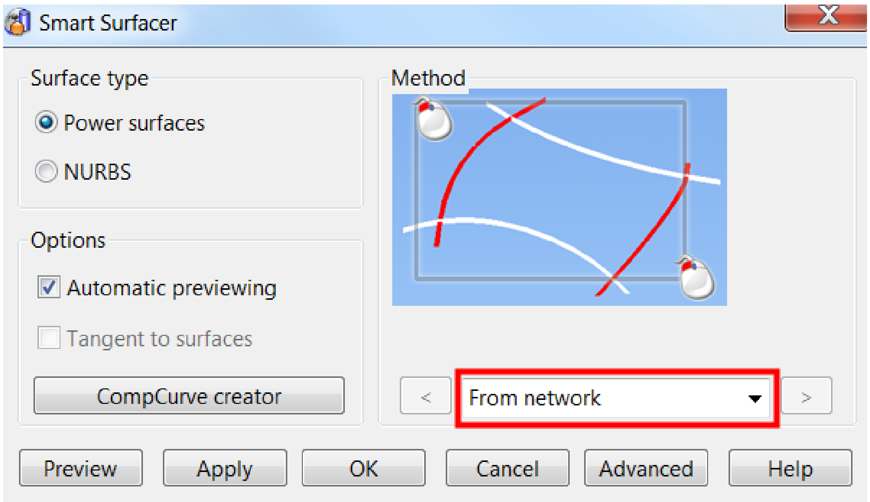
2 **Save** the **model** as:
.....**PowerSHAPE Data\Ex7 Surface bottle.psmodel**

3 **Close** the **model**.

Power Surfaces from Wireframe

Smart Surfacer 

Several of the PowerSHAPE surface creation options have been included into the **Smart Surfacer**. If you select 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, you can toggle to it by opening the list of options and clicking on the downward pointing chevron.



Automatic Surfacing options covered in this section are:

- **Fill-In**
- **From-Network**
- **Drive-Curve**
- **From Separate**
- **Two Rails**

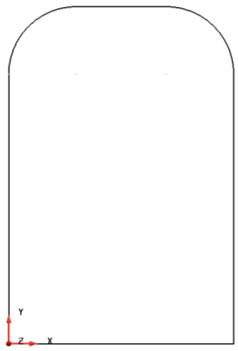
Fill-In Surface

A Fill-in surface is usually created from one or more enclosed wireframes. A Fill-in surface can also be created from open-ended wireframes and point data.

1 Create a **new model**.

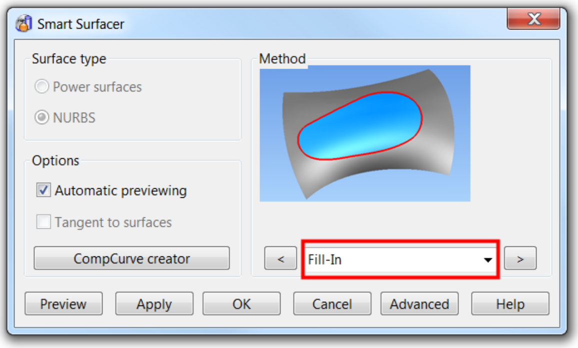
2 Create a **Rectangle** from the **0** datum with sides of **X50** and **Y75**.

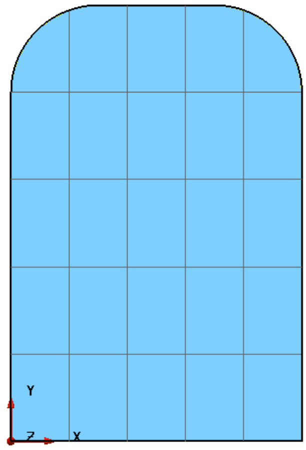
3 Create **R 15 fillets** at both of the top corners.



4 Select all the wireframe entities.

5 Select the **Smart Surfacer**  from the **Surface toolbar**. 

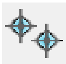
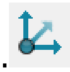




The dialog automatically selects the most likely surface creation options for the selected wireframes. In this case, a Fill-in surface.

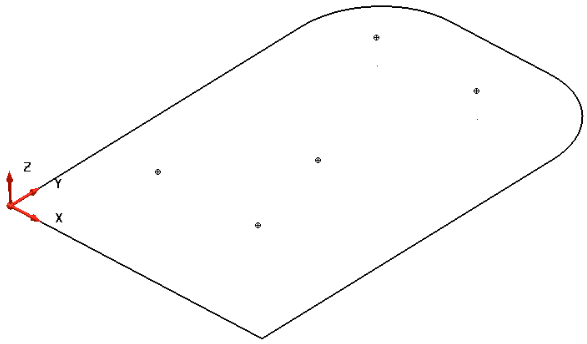
6 Select **OK** to accept the surface and close the form.

7 Select and delete the **fill-in** surface.

8 Select **Create a point**  from the **Workplane** menu. 

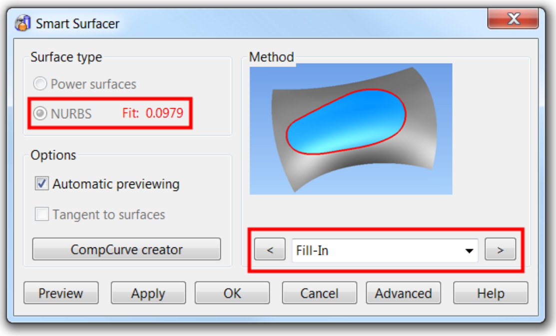
9 In the **Command input** box enter the following **5** coordinate positions, pressing the **Enter** key after each one to save them.

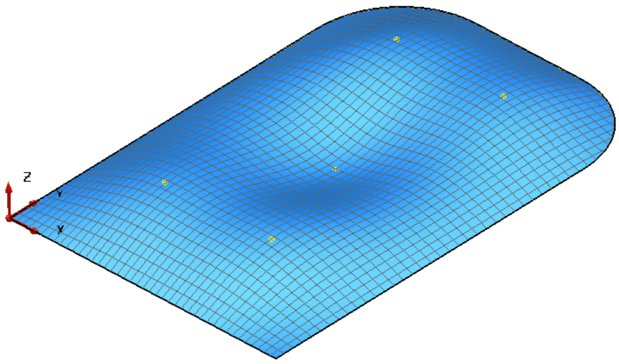
- **25 37.5 0**
- **15 15 5**
- **15 60 5**
- **35 60 5**
- **35 15 5**



10 Reselect all the wireframe, including the five 3D points.

11 Select the **Smart Surfacer**  from the **Surface** toolbar. 





A smooth Fill-in surface has been generated to include the point data. In this case, PowerSHAPE created a NURB surface. If required, the surface can be converted to a PowerSURFACE after.

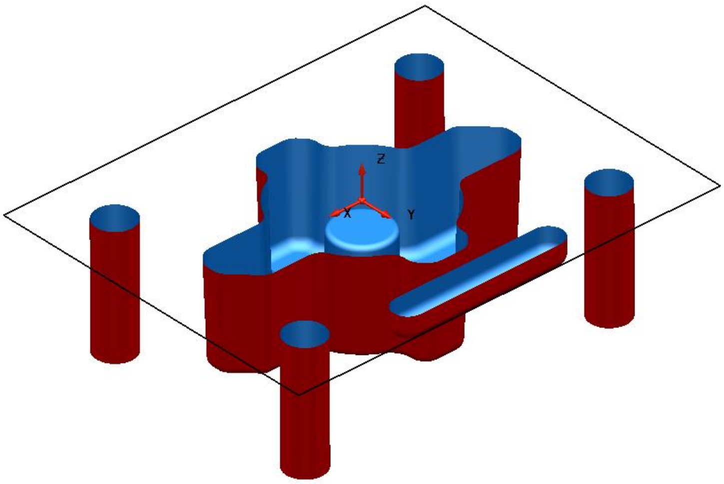
12 Select **cancel** then close the model

Fill in Surface - Die Example

A fill in surface can also be made up from several wireframes. When a fill-in surface is generated, the composite curve is used to create the trimmed area within a surface.

1 **Create** a new model and **Import** the following data;

.....***PowerSHAPE Data******Fill-in_Die.dgk***

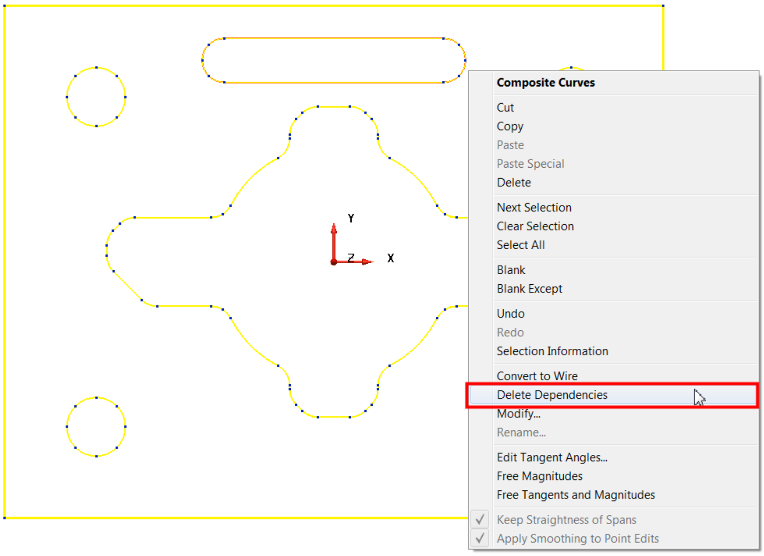


A new **Fill-in surface** will be generated **inside** the **rectangular wireframe** and from **composite curves** traced around the top edges of the surface forms.

2 Hold down the **Alt** key, and left-click the upper edge of each surface form to create a new composite curve for each one.

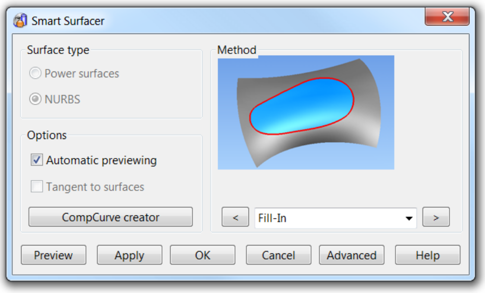
3 From the **Selection** toolbar, click **Quick select all surfaces** and then **Blank** (**Ctrl+J**) the selected surfaces.

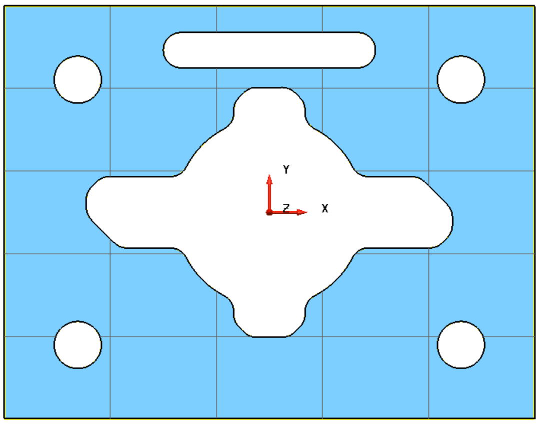
4 Select all the composite curves and right-click on one of them to open the popup menu. Select **Delete Dependencies.**



When a curve or composite curve is created on an object (for example, a surface), it may be dependent upon that object. This option removes any such relationships that may exist between the curve and in this case, the surfaces.

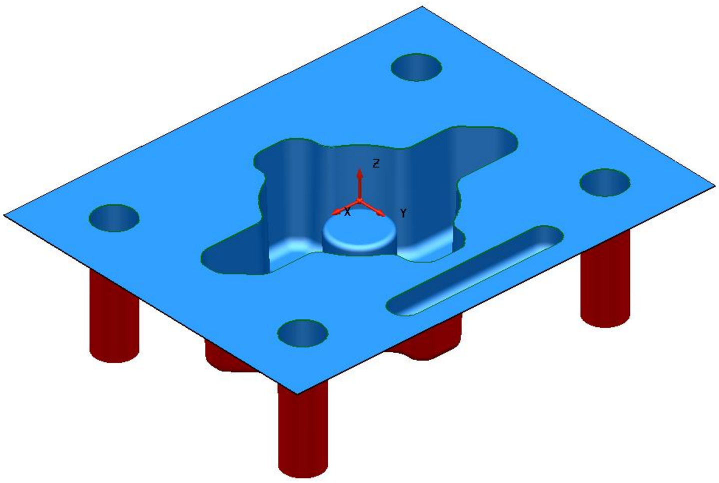
5 Select all the composite curves, then the **Smart Surfacer.** 





6 Select **OK** to accept the fill in surface.

7 **Unblank** (CTRL+L) the model.



8 Save the model as

.....\PowerSHAPE-Models\Surface Die example.psmodel

9 Close the model

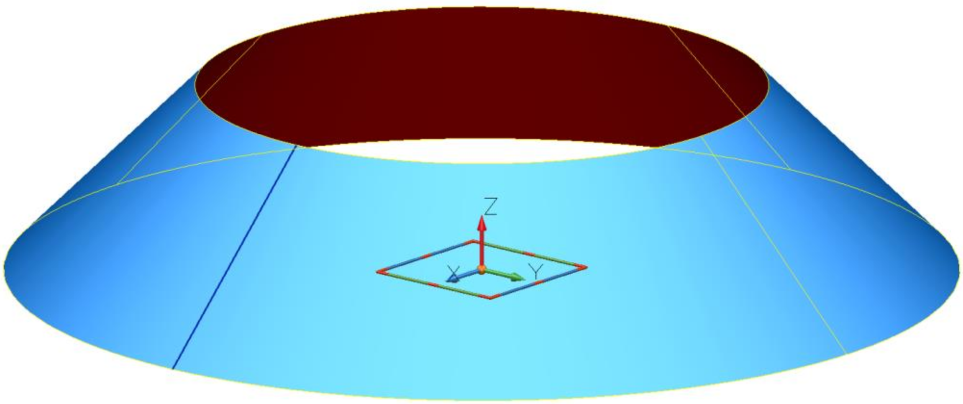
3D Fill-in Surface Example

A composite curve does not have to be created in advance because it can be generated within the **Fill-in Surface** form.

1 **Create a New Model** and Workplane at **0**.

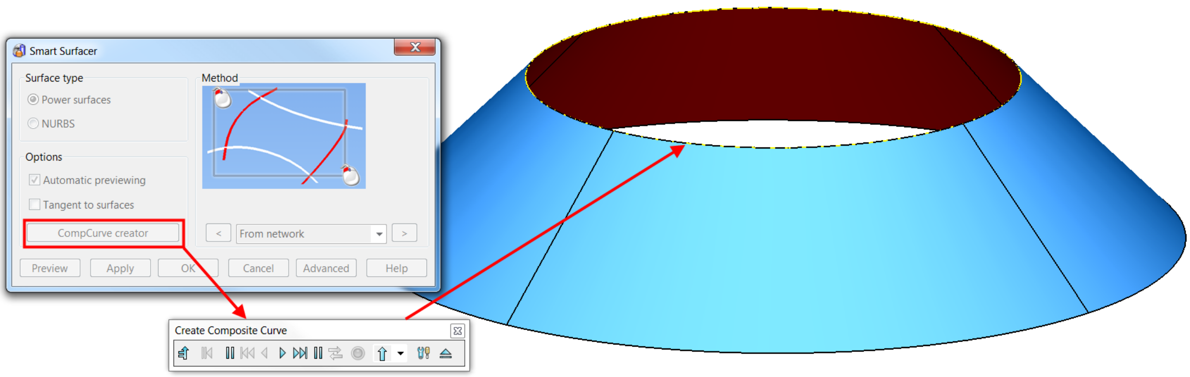
2 Start a single line at **50 0 0** and enter a relative end point using the coordinates **-20 0 20**.

3 Create a **surface of revolution**  around the **Z axis**.



4 Select the **Smart Surfacer** 

5 Select **Composite Curve Creator** and trace a composite curve around the top edge of the surface.

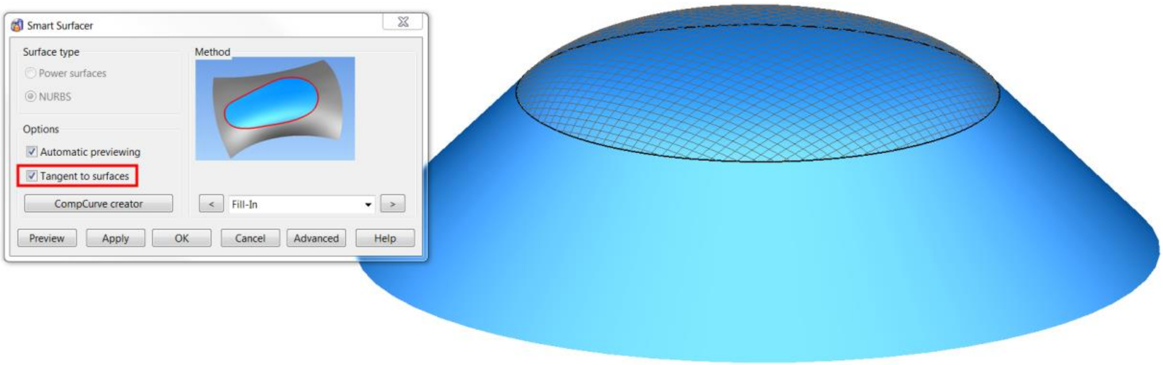


6 Select **Save** and **Eject** to save the curve.

When the **composite curve** is completed, the **Wizard** recognises that the curve is suitable for producing a **fill-in surface**. The surface creation dialog is updated, and a **preview of the surface is displayed**. In this case, the surface is flat, but in many cases, you need a new surface that is tangential to the surrounding surfaces.

Creating a tangential surface

1 Select **Tangent to surfaces** in the Smart Surfacer dialog.



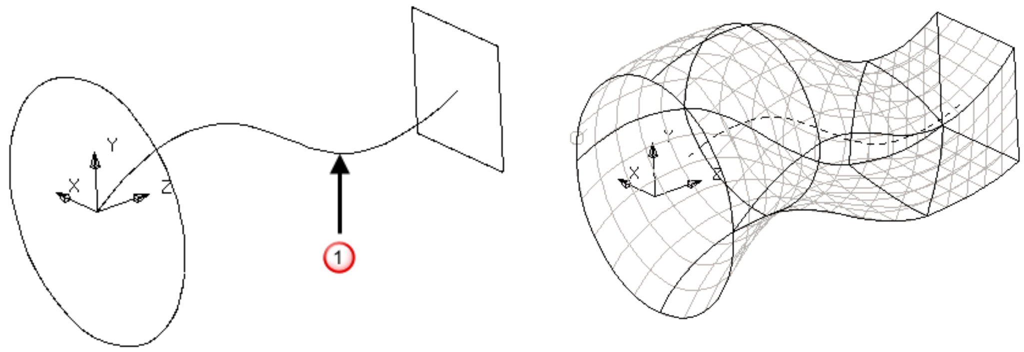
The preview is tangential to the adjacent surface.

2 Select **OK** to accept the surface.

3 **Close** the model without saving.

Drive-curve Surface

A **drive-curve surface** consists of a **spine** curve along which **section** curves, are **aligned perpendicular**, relative to the spine points. Below is a drive-curve surface using a two sections running along the whole length of the drive-curve. 1 The **minimum requirement** is **one sectional wireframe**, positioned anywhere along the **single Drive-curve**.

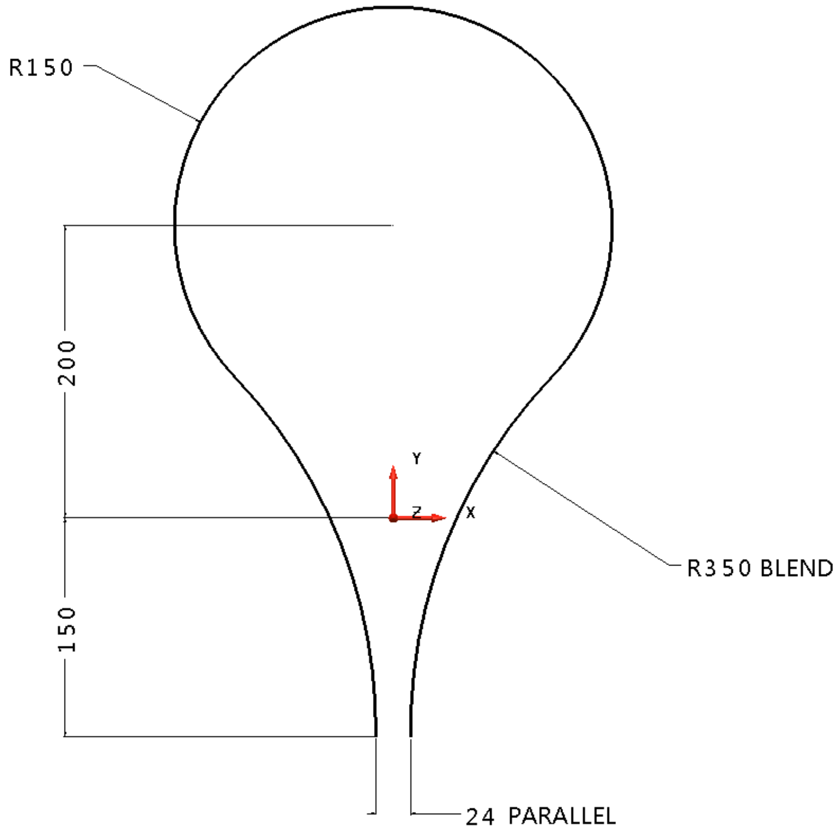


Racket Example

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

1 Open a **New model.**

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

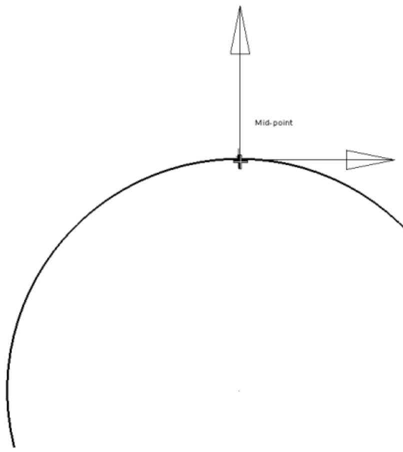


This shape will form the single **drive-curve** along which the surface will be generated.

In this case, wireframe sections will be defined perpendicular to and at strategic positions along the drive-curve.

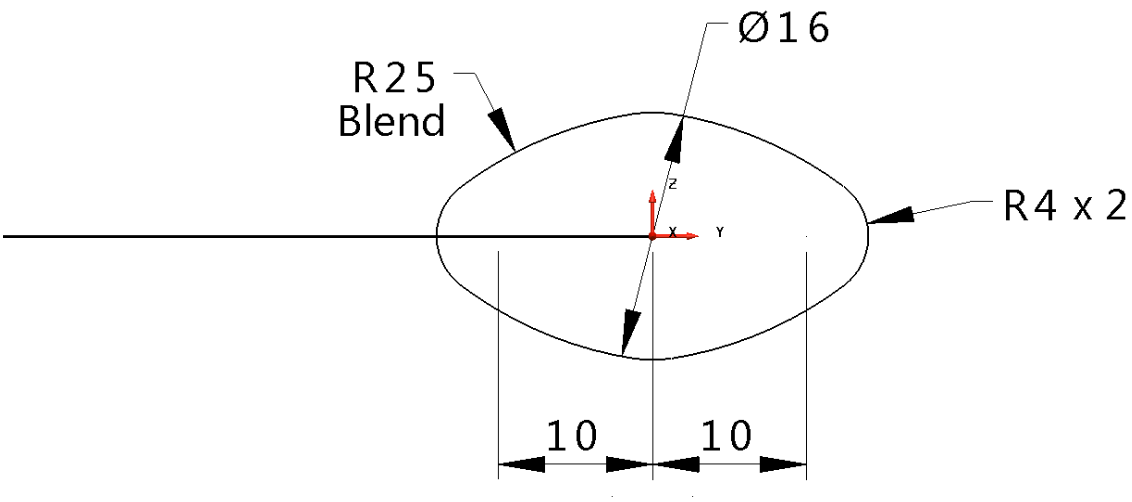
Upper cross Section

3 Create a new Workplane and position it at the midpoint at the top of the racket



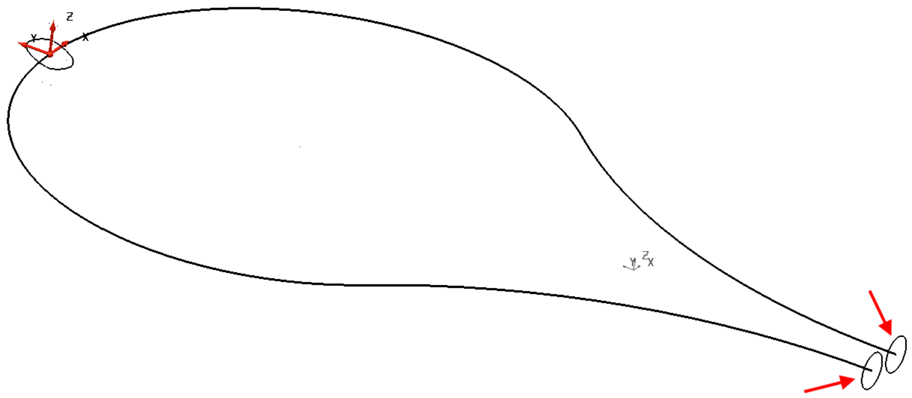
4 **View from the Right (+X)** and zoom in to the new Workplane.

5 Create the following wireframe section on the new Workplane.



6 Select the **Y** principal plane.

7 Generate two circles of radius **10** with their centres snapped to the end of the open lines.

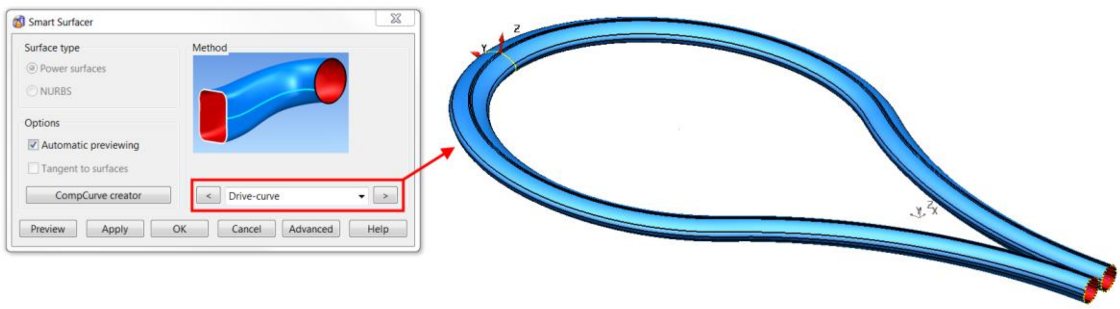


The **drive-curve** and each **cross-section** must be **single entities** before the required Drive-curve surface can be produced. The sections are already single **Full Arc** entities, but the drive-curve is a series of arcs and lines which must first be made into a **single composite curves**.

8 Create a **single composite curve** out of the **main drive-curve** and **upper cross section entities**.

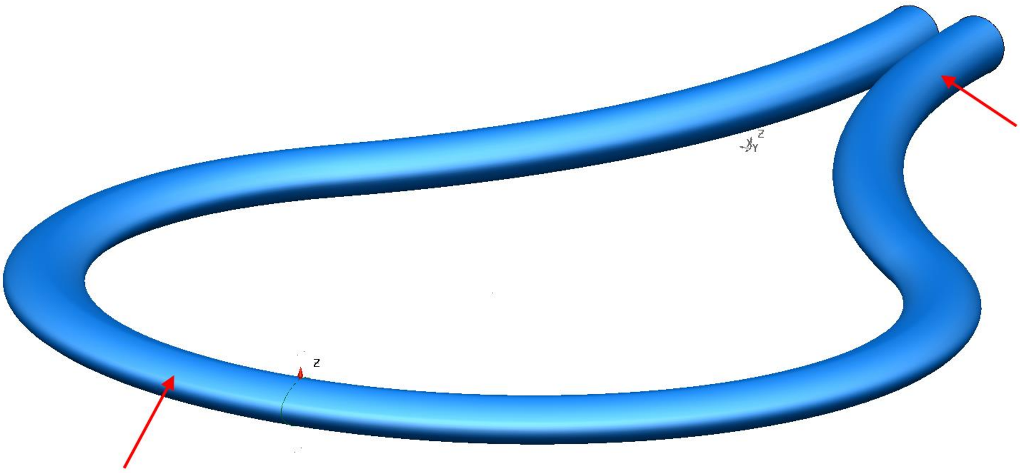
9 Select **all three sections** and the **drive-curve**.

10 Select the **Smart Surfacer.** 



The **Smart Surfacer** recognises that the wireframe selection is suitable for creating a **drive-curve surface** and changes the option in the drop-down list accordingly. A preview of the surface is displayed.

11 Click **OK** to accept



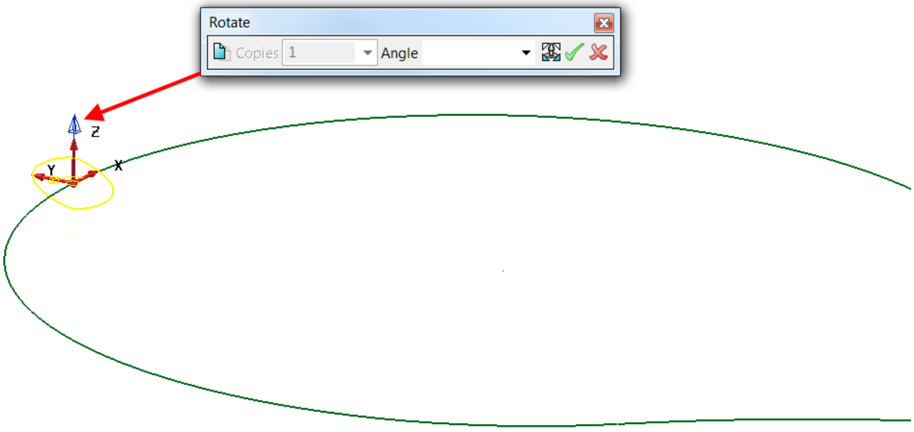
This generates a surface with a **section that changes** from the **first circle** to the **defined section**, and then back to the **last circle**.

To maintain a particular cross-section at a position on the drive-curve**, extra sections can be added**.

12 **Select** and **delete** only the **surface**.

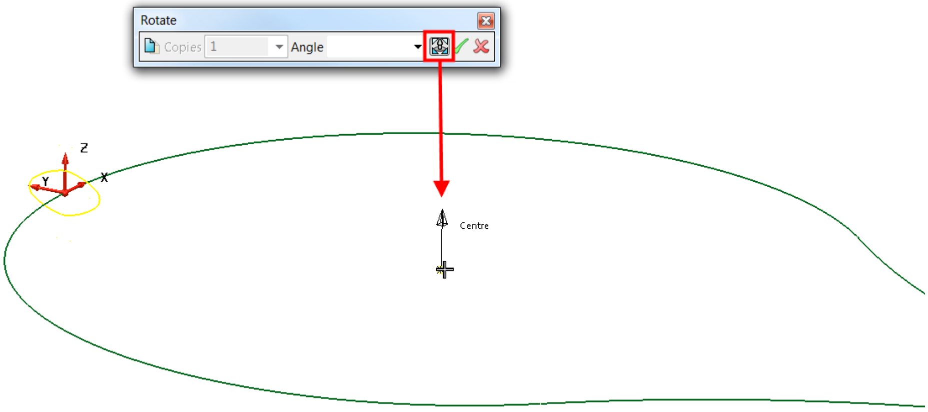
13 Select the **composite curve section** at the top of the racket and select the **Z** principal plane.

14 Select **Rotate items**  from the **General Edits** toolbar. 



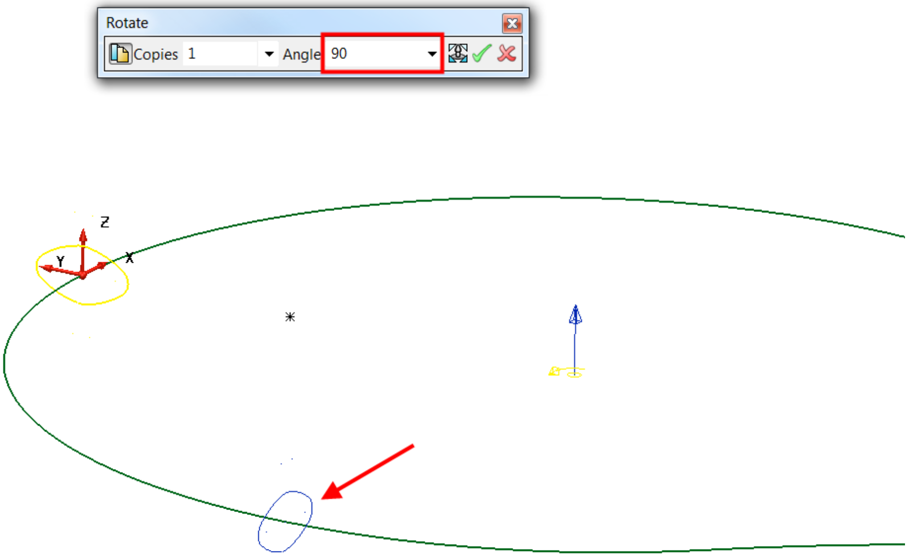
A **blue arrow** indicates the **directional axis** and **position of rotation**. Although the **direction** is **correct**, the **position** will need to be **moved** to the **centre** of the **racket** if the section is to be copied to the correct position.

15 Select **Reposition rotation axis**  from the Rotate toolbar then snap to the centre key point of the circle racket as shown.



The **Rotational axis** has moved to the centre of the racket.

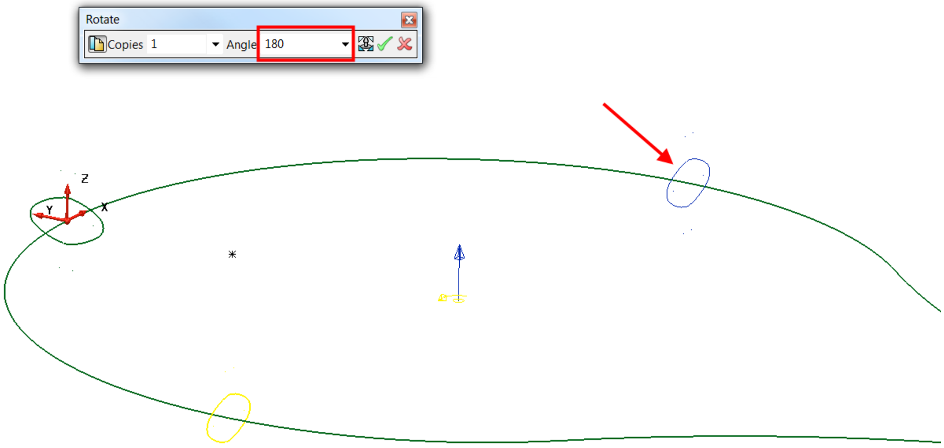
16 Select Copy the selected item button, enter **Angle** as **90** then Enter.



A copy of the original section is previewed.

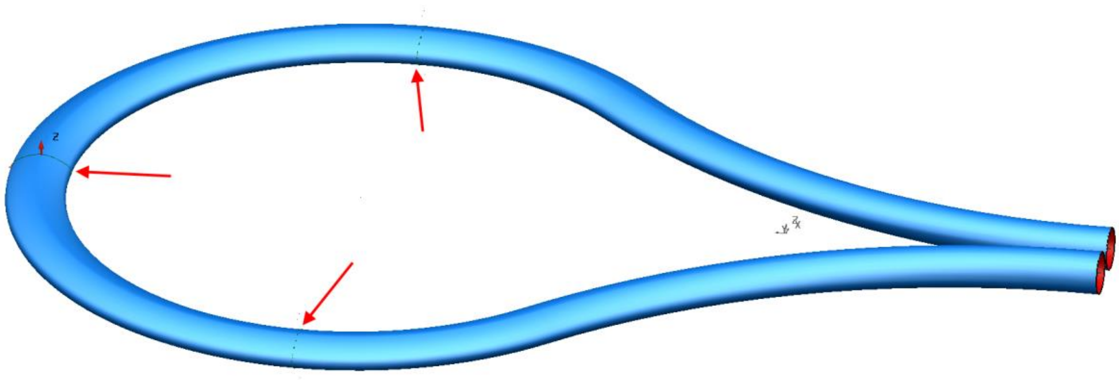
17 Select the green tick to confirm.

18 Enter **180** in the rotate form then green tick to accept.



19 Close the **rotate** form and reselect the **drive curve** and **5 section entities**.

20 Generate a surface from drive-curve using all five sections.



The elliptical shape is now constant between the three sections indicated.

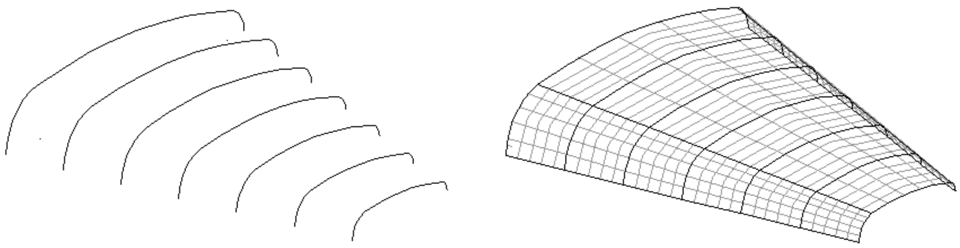
21 Select **File > Save As**:

.....\PowerSHAPE-Models\surface racket

22 Select **File > Close**.

Surface from separate curves

If separate curves are selected, the **Smart Surfacer** uses the **Create a surface from separate curves** option.

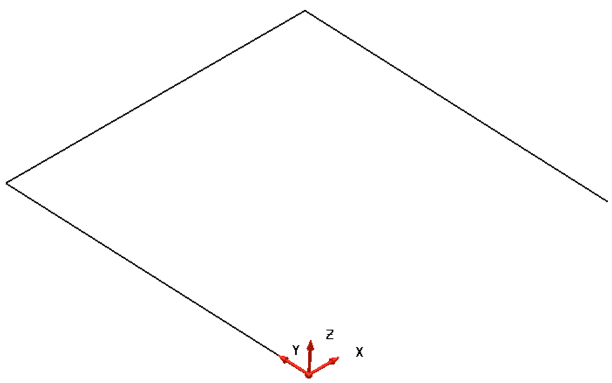


The **separate curves** are defined across the surface (**laterals**), which are then linked with curves of best fit along the surface (**longitudinals**).

Lateral Curve Example

1 Create a **new model** and workplane at **0**.

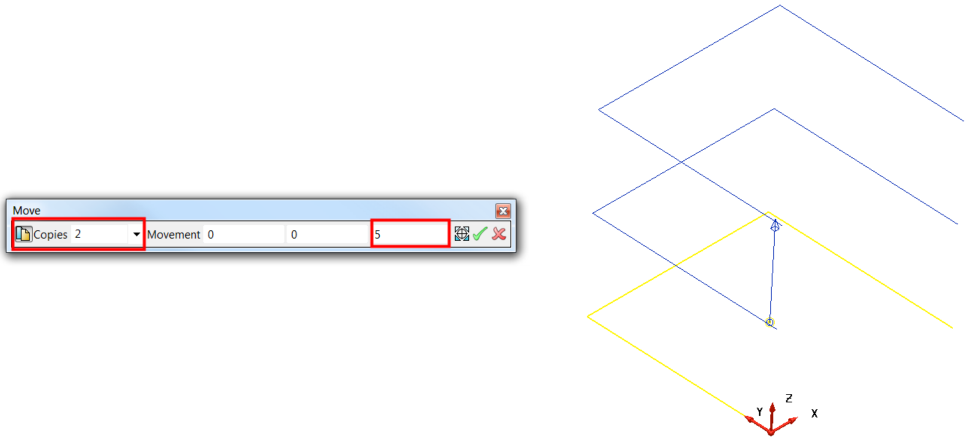
2 Create a continuous line from the workplane, using the distances **10** in **Y**, **10** in **X**, and **10** in **-Y** to produce an n-shape.



This **section** is the **basis** for the **other sections**, so the next step is to **copy** it up the **Z-axis** as individual lines:

3 Select the three lines, then **Move**/Copy items  from the **General Edits** toolbar. 

4 Enter **2** for the **No.```` of copies** and the value of **0 0 5** in the position window.



5 Confirm the preview with the green tick.

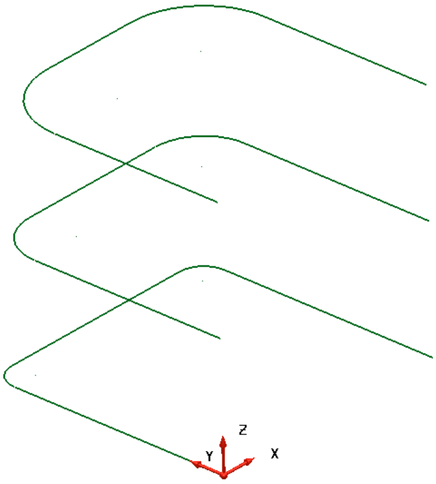
Each **composite curve** will be **filleted** by a different radius.

6 Make composite curves from each set of lines (**Alt** key and left-click).

7 Create a **fillet radius** of **1** on the bottom curve.

8 Create a **fillet radius** of **2** on the middle curve.

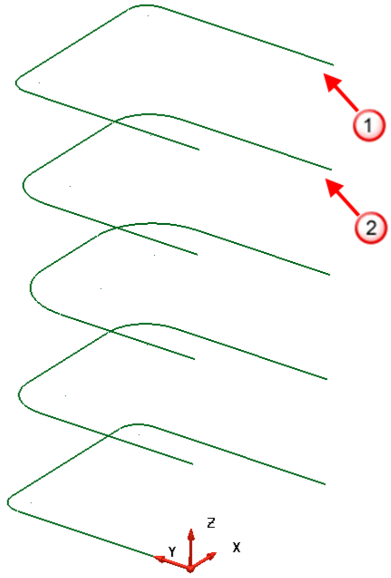
9 Create a **fillet radius** of **3** on the top curve.



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

10 Select the lowest composite curve and **Move/Copy** it up the Z-axis by **20**. 1

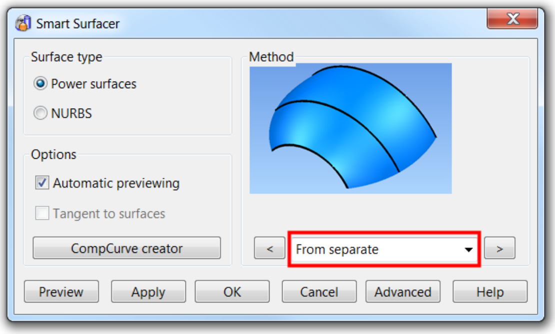
11 Select the second from bottom composite curve, and **Copy** it up the Z-axis by **10**. 2

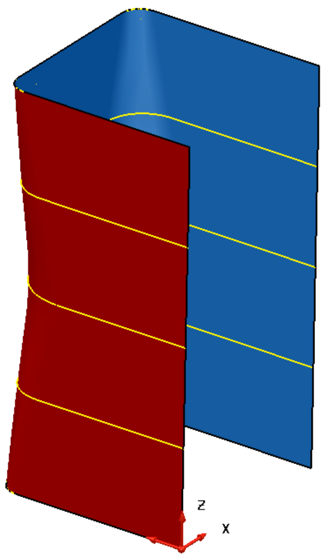


All of the shapes have been turned into **composite curves**, ready for **surface generation**. Each of the composite curves will be turned into a **lateral** on the surface.

12 Select *all* of the composite curves then **Smart Surfacer**. 

13 Select **Surface** and select **Smart Surfacer** from the Surface toolbar.





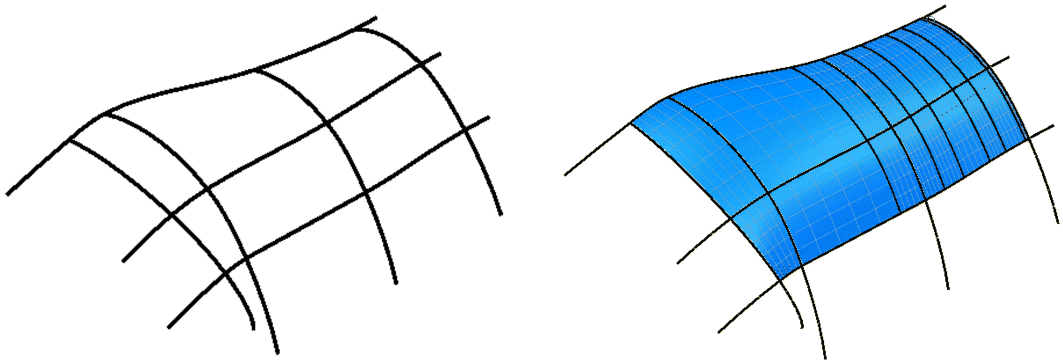
The Wizard analyses the selected wireframe and selects the surface type to be created as **From Separate**.

14 Select **OK** to create the surface from separate curves**.**

15 **Close** the file without saving.

Surface From Network of Curves

Surface from Network creates **one or more** four-sided surface patches over a network of wireframe. Surface definition will not occur on any open ended parts of wireframe that extend beyond the network.



Surface Gap Example

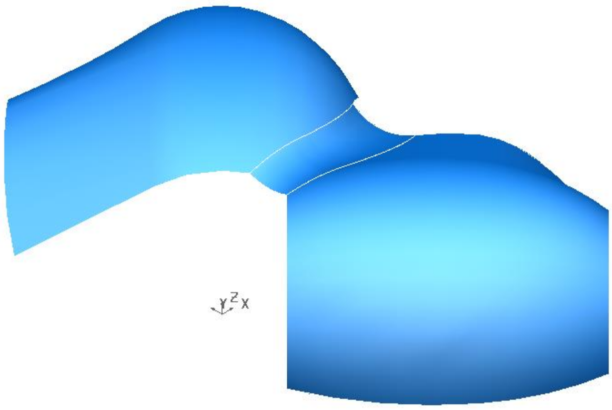
The following example uses Automatic surfacing and applies a Surface From Network to blend across a significant gap in the model.

1 Open a **new model**.

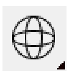
2 Import the model.

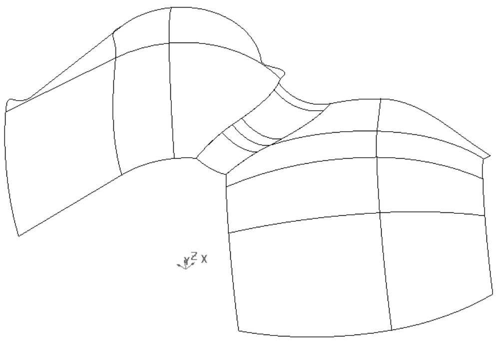
.....\PowerSHAPE_data**network_example.dgk**

3 View in **ISO1**  to show the gap to be filled.



To fill the gap with a smooth surface, additional wireframe curves will be created. By default, the **tangencies of the existing surfaces** are used by composite curves defined along exiting surface edges.

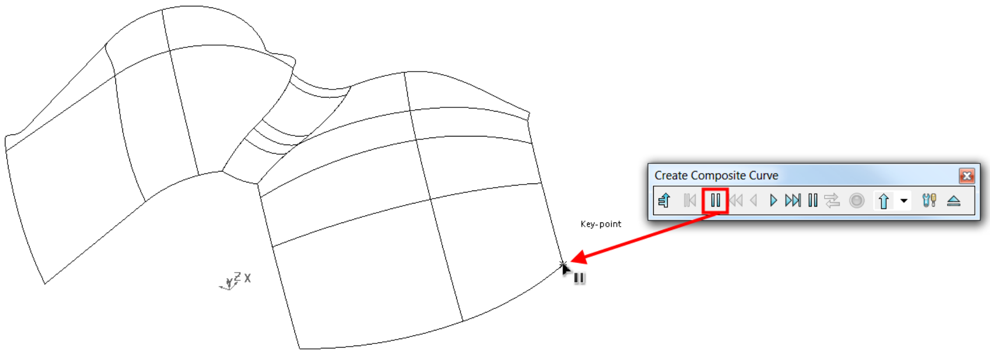
4 Switch to **Wireframe view**  so the internal geometry can be seen.



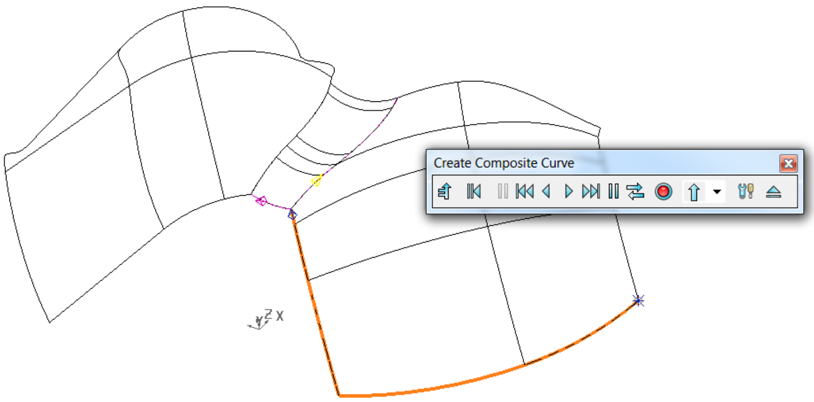
5 Select Create Composite Curve. 

The Composite Curve toolbar is displayed.

6 Select **Define Start Point**, then click on the key-point on the corner of the surface shown.

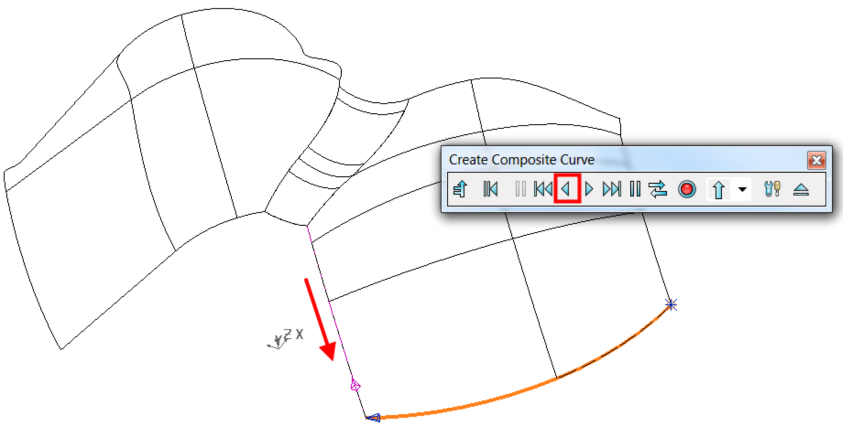


7 Select the bottom edge with the left mouse button to create the first part of the composite curve.

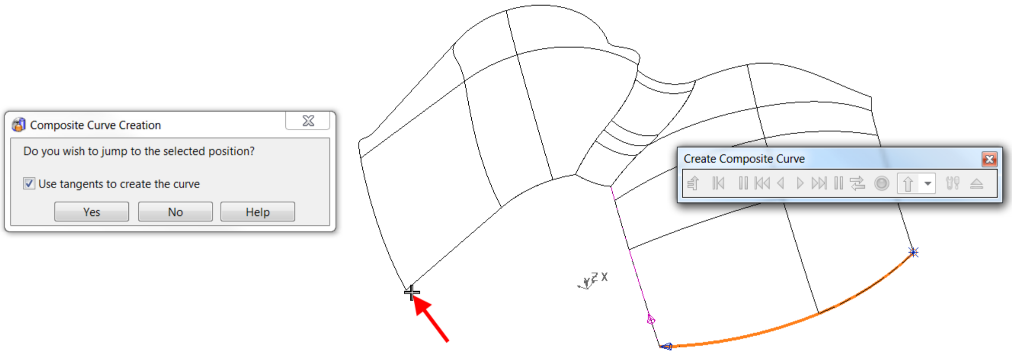


The curve continues to the first branch point awaiting further interaction from the user. In this case, too far.

8 On the **Composite Curve** toolbar, select **Backwards**  once to go back one span.

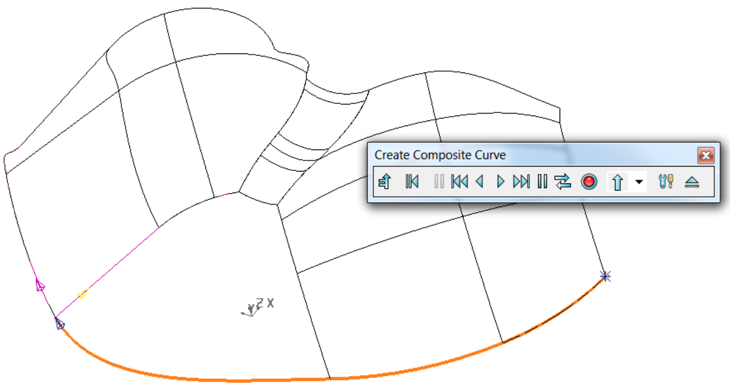


9 **Left-click** on the opposite corner indicated by the **red arrow below**.



A dialog requests if the composite curve jump to this position?

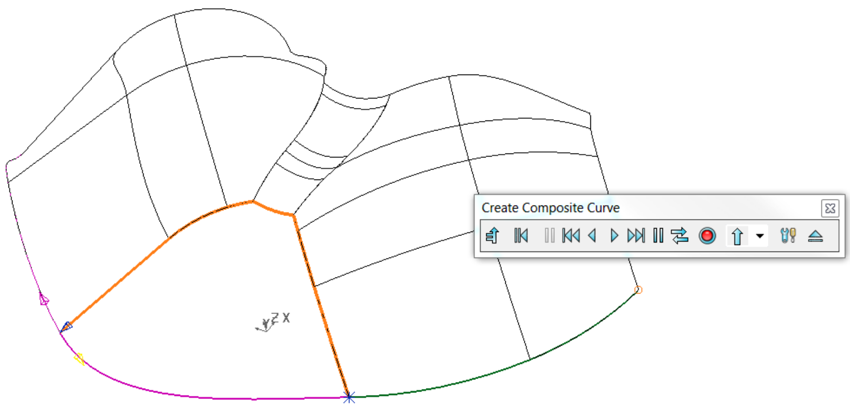
10 Select **Yes**.



The **composite curve** jumps to the **selected point** while **maintaining** the **tangencies** relative to both surfaces.

11 Select **Save**  on the **Create Composite Curve** toolbar.

12 Create a second composite curve along the three edges shown.

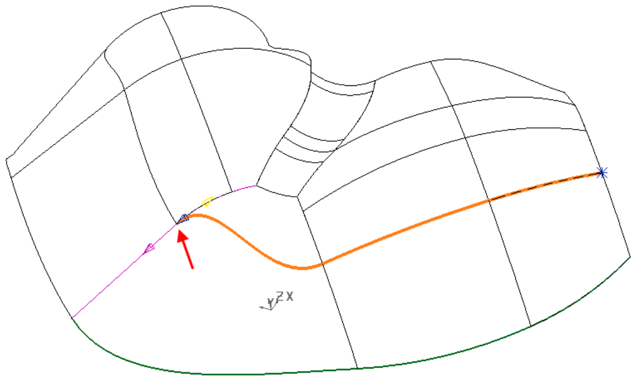


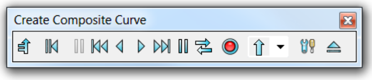
When a **composite curve** is created along a **surface edge**, it inherits the **tangencies of the surface**. These are reproduced if the curve is used to create a new surface from appropriate **Smart Surfacer** options.

Although enough wireframe is now available to create a **Surface - from Network**, a more accurate alignment will be achieved by creating additional curves across the void. Existing lateral geometry will be used.

13 Select and **Blank** the last composite curve.

14 Create a **new composite curve** along one of the **intermediate curves** on the right surface and as before attempt to bridge the gap to the corresponding curve on the left surface (as shown below).



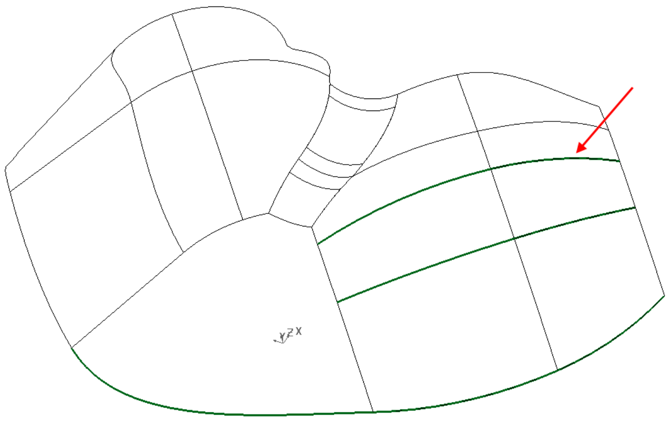


The end of the composite curve is aligning to the surface edge and not, as required, along the intermediate curve!

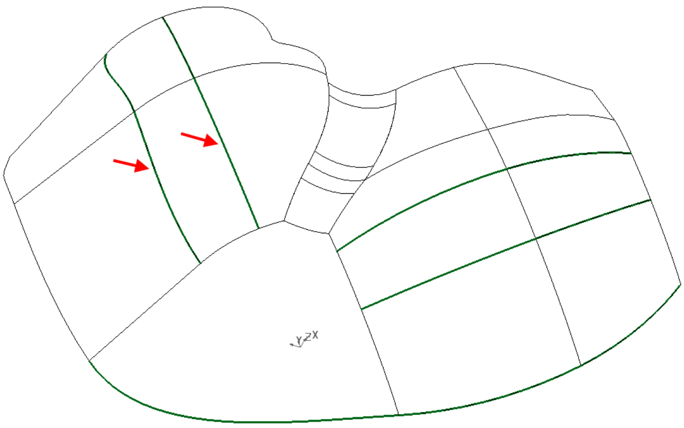
15 On the **Composite Curve** toolbar, select **Backwards**  once to go back one span.

16 Select **Save**  on the **Create Composite Curve** toolbar.

17 Similarly, generate another composite curve along lateral above on the same surface as shown.

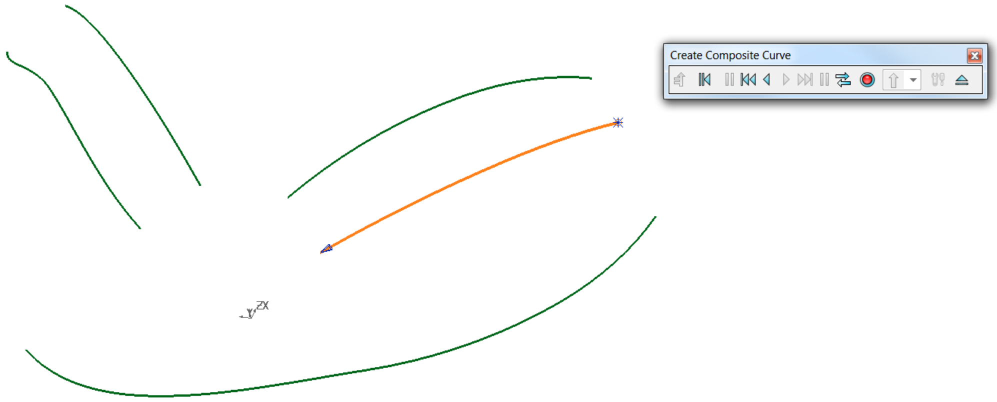


18 Create **two further composite curves** along the corresponding surface curves on the left surface as shown below.



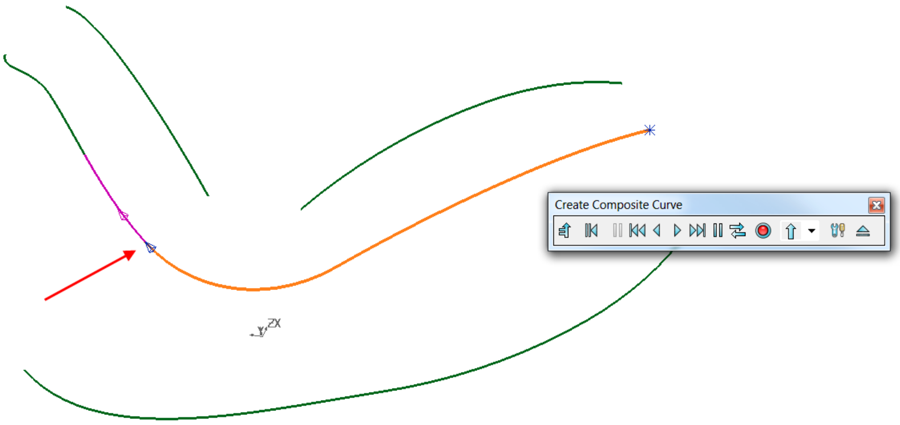
19 Select and **Blank** all surfaces on the model.

20 Create a **new composite curve** with the start point at the start of the existing composite curve as shown.



21 Click on the curve to include it as part of a new one and then click on the continuation at the end of the corresponding curve across the gap.

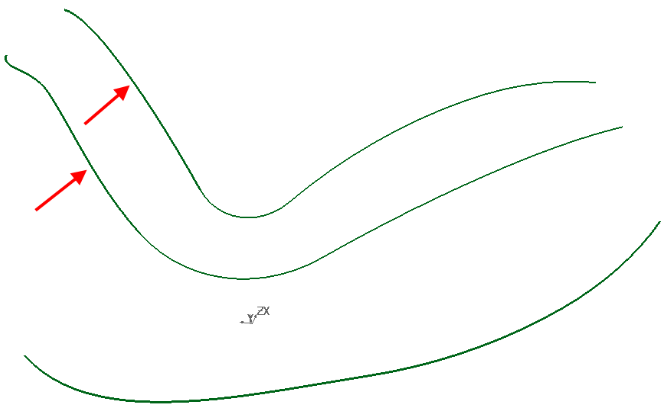
22 Select **Yes** to use tangents.



23 **Save**  the curve.

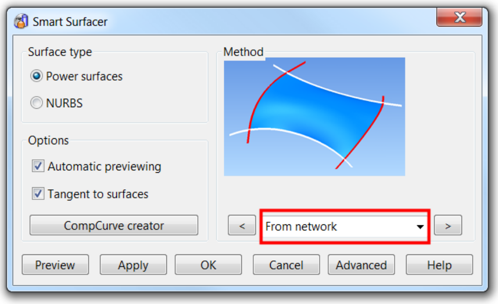
24 Repeat on the curve(s) above.

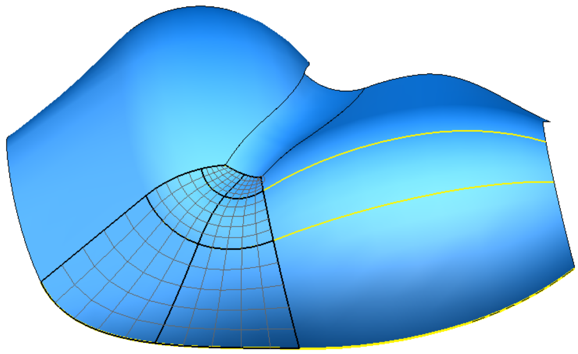
25 Delete the two curves that are not part of the required surface network.



26 **Unblank (CTRL+L)** to bring back the surfaces and composite curve created earlier.

27 **Quick select all wireframe**  and then **Smart Surfacer**. 



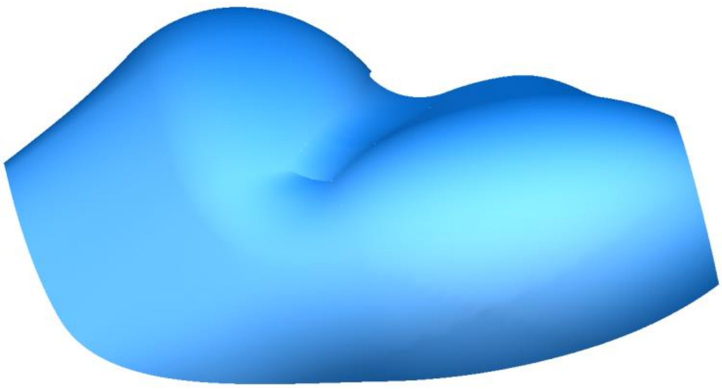


The Wizard selects **From Network** as the most likely option to create a surface from the selected wireframe.

28 Select **OK**.

29 Select the new central curve running down the surface. Right-click the surface and select **Free tangents and magnitudes** in the local menu to provide a smoother transition.

30 Deselect the curves.



The **gap** in the model is **filled** in with the **new surface** which **flows smoothly** into the **adjoining surfaces**.

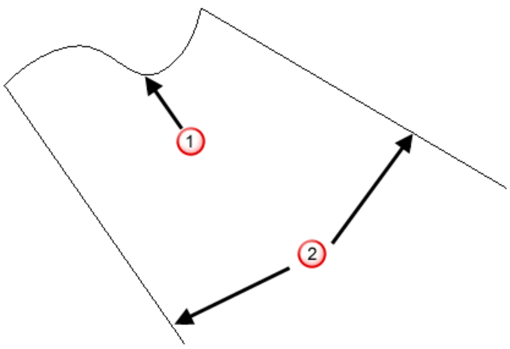
31 Select **File > Save As**:

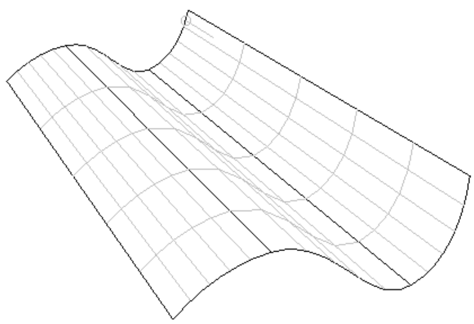
.....***PowerSHAPE Data******network curve example.psmodel***

32 Select **File > Close**.

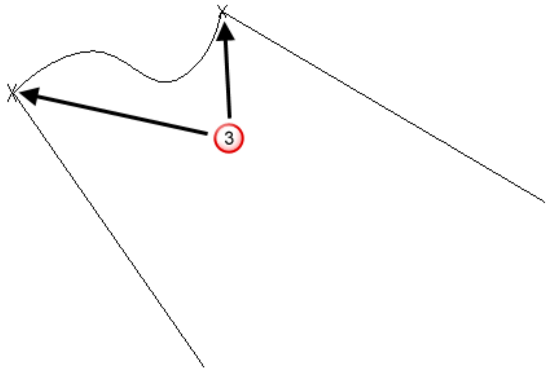
Surface from Two Rails

A surface can be generated from a section 1 that is scaled along two drive rails 2, defining the edge of the surface.





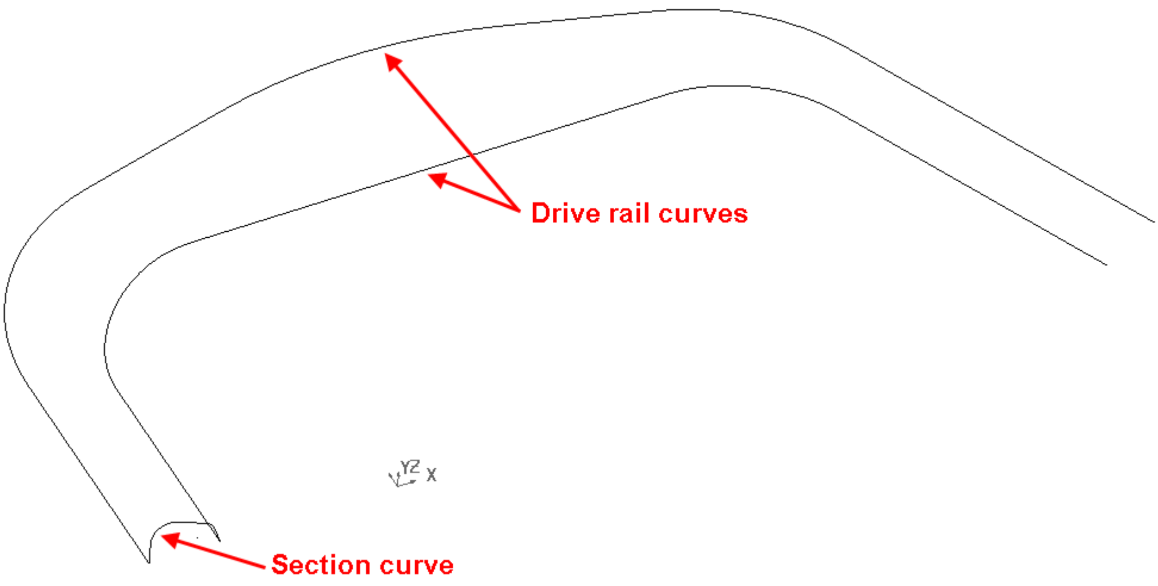
The section is scaled across corresponding points along the drive rails to generate the surface. Drive rail pairs must contain the same number of points and can both, if required, form a closed loop. The end points on an open section must be positioned exactly on the start point of each drive rail .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



Surface from Two Rails Example

1 **Import** the model:

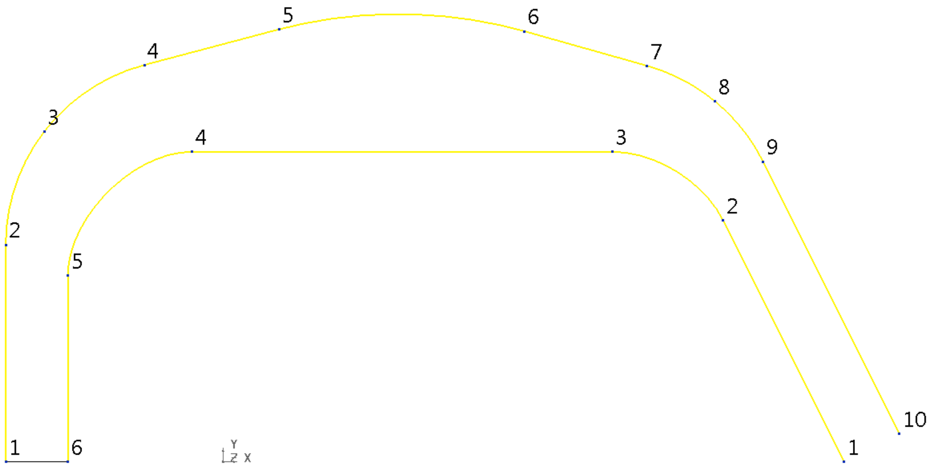
.....\PowerSHAPE_data**two_rail_data.dgk.**



2 Select one of the drive rail curves to display the **Curve** Editing toolbar.

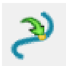
3 Shift-select the other drive rail curve.

4 Select **Show point labels.** 

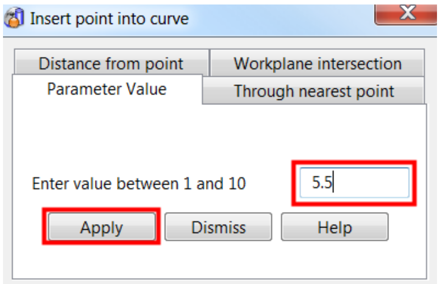


Each drive rail must have the same direction and number of points. In this this example, the opposite.

5 Select just the top curve (10 points).

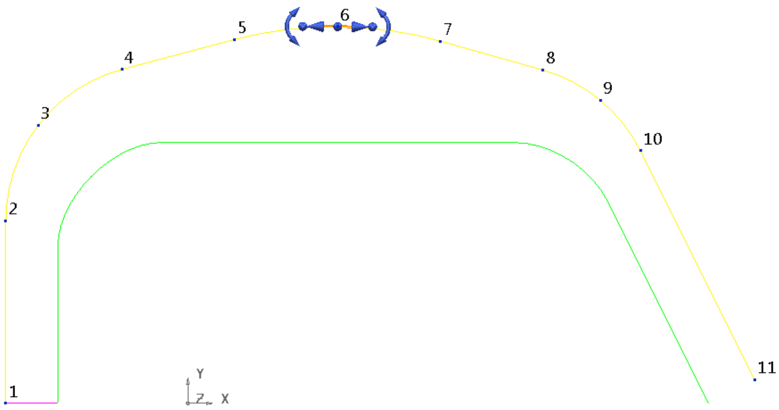
6 From the **Curve Editing** toolbar, select **Create a point.**  .

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



7 Select the **Parameter** tab and enter **5.5**. Select **Apply** and then Dismiss.

It is easier to add points from the last number backwards because the curve is renumbered when each point is added.



A point is added to the curve

8 Select the bottom curve (6 points) then Reverse surface/curve. 

9 With the lower curve still selected, select **Create a point.**  .

10 Select the **Parameter** tab and enter **4.5** then **Apply**.

11 Enter **3.5** and select **Apply**.

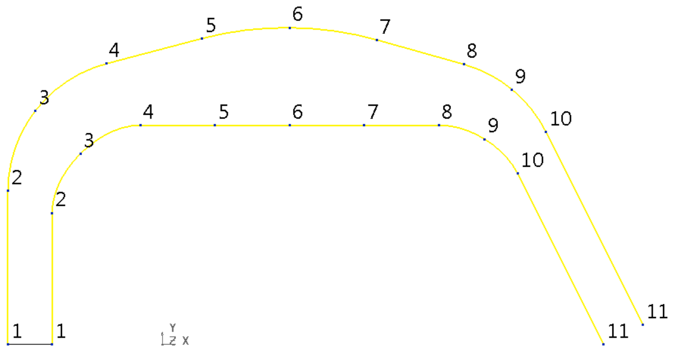
12 Enter **4.5** and select **Apply**.

13 Enter **3.5** and select **Apply**.

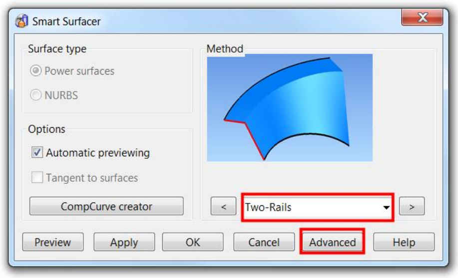
14 Enter **2.5** and select **Apply** and then Dismiss.

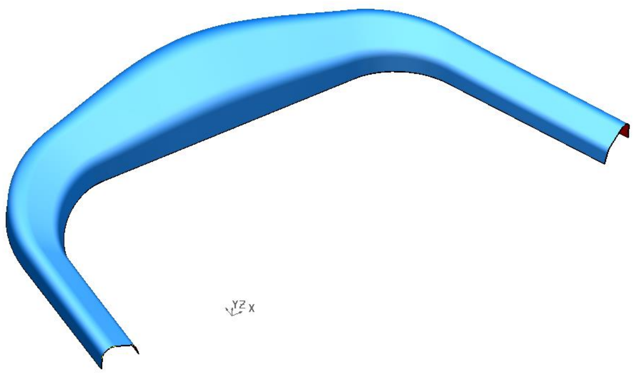
15 Select both curves.

Each curve has the same number of points so the generated surface aligns smoothly.



16 Select all three composite curves then **Smart Surfacer**. 

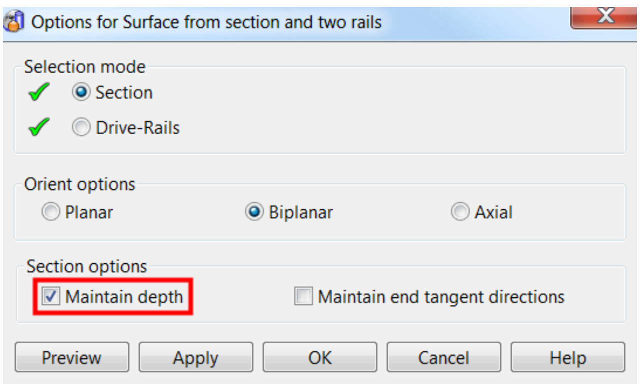




The Wizard recognises that a **Two Rails** surface is the best choice for surface creation.

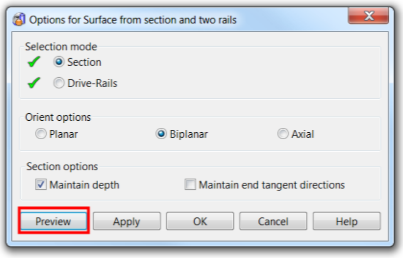
However, the **section height** has **scaled** in proportion to the distance between the drive-curves at each section position.

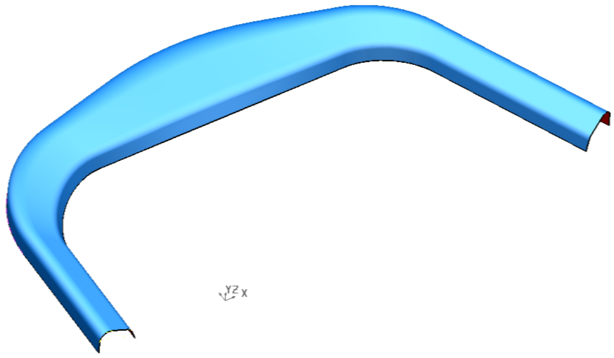
17 Select the **Advanced** button at the bottom right of the **Smart Surfacer** form.



The **Options for Surface from section and two rails** dialog is displayed. It provides more control over the way in which the surface is defined from the selected wireframes.

18 Select the **Maintain Depth** option and select **Preview**.





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

19 Select **OK** twice to accept.

20 Select **File > Save As**:

.....\PowerSHAPE-Models\two rail surface example

21 Select **File > Close**.

Limit selection

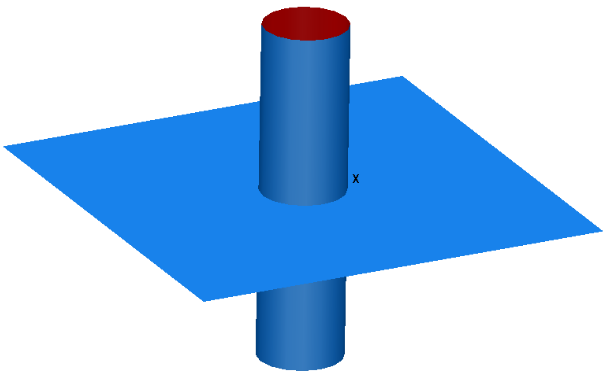
Limit Surface using a Surface as the Cutting Object

When the **Limit selection**  option is applied with surfaces, they are trimmed back to their common intersection. The **Next Solution** option enables you to toggle through all possible trim combinations. When a surface has been included in the limit selection process, a boundary is created, and only the part of the surface either inside or outside is displayed. Several surfaces can be limited with a single cutting object.

1 Create a **new model** and **workplane at 0**.

2 Create a **plane primitive** surface  on the **0** datum of size **X 50**, **Y 50**.

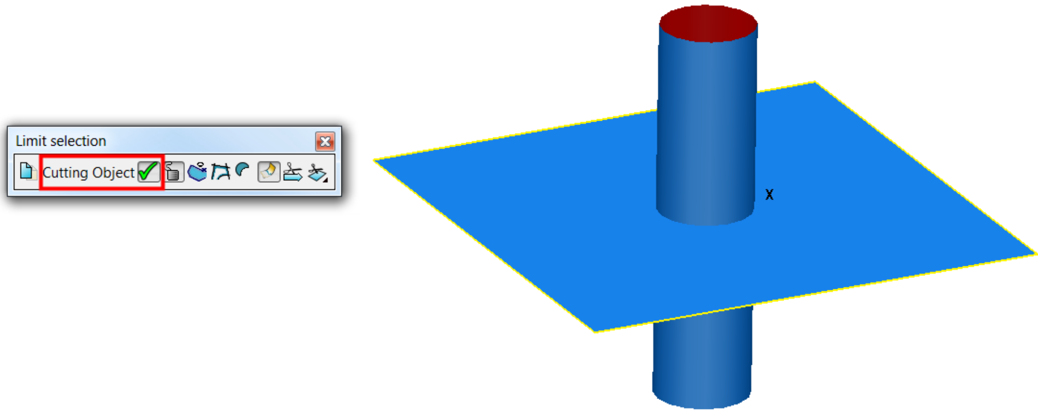
3 Create a **cylinder primitive** surface  of **Length 40** and **Radius 5** positioned at **Z-20**.



The cylinder will be limited by the plane.

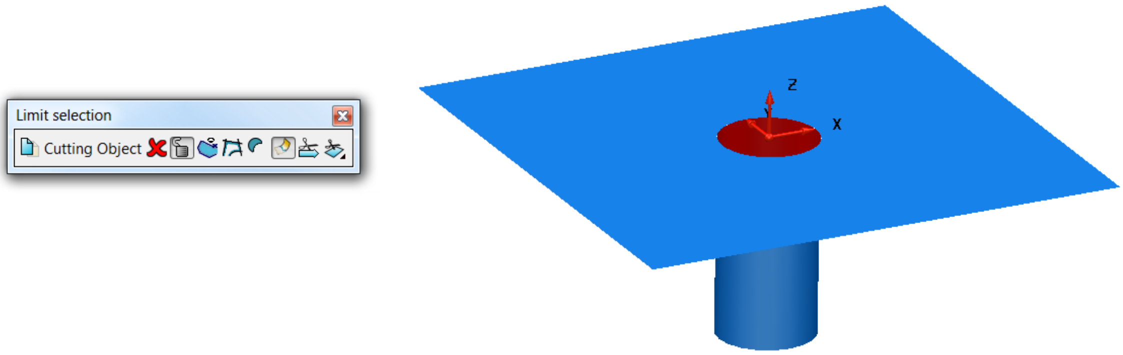
4 Select **Limit selection**  from the **General Edits** toolbar. 

5 Select the Plane surface.



The Plane has been assigned as the cutting object. 

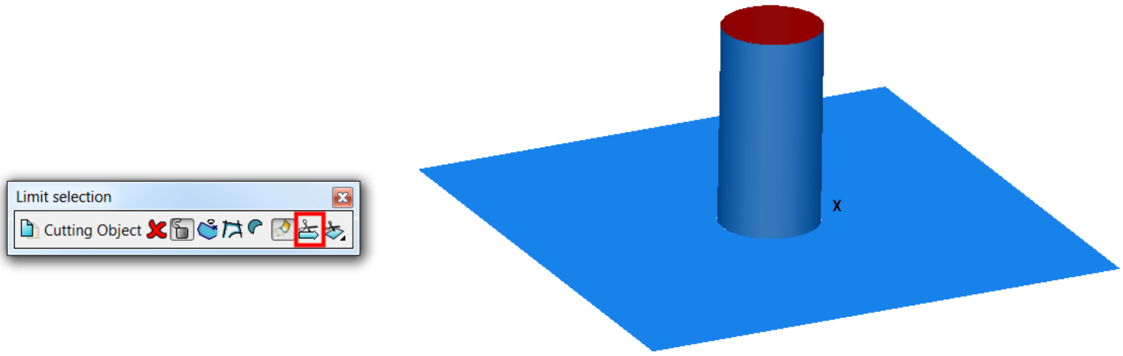
6 Click anywhere on the top side of the cylinder surface.



The **plane** and **cylinder** are **automatically trimmed** back to their common intersection.

This is one of several solutions; the others can be obtained by clicking ***Next Solution****.* 

7 Keep selecting **Next Solution**  until the following trim combination is displayed.



8 Close the **Limit selection** form. 

If the lower half of the cylinder is clicked during the original selection process, it would be the initial solution.

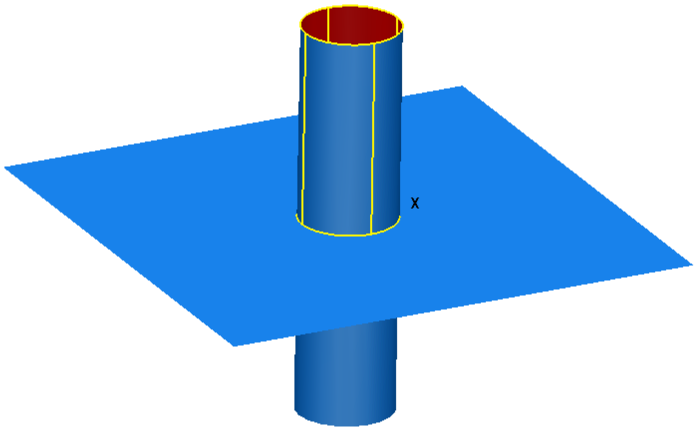
9 Select **Undo**. 

10 Select the plane surface first and then Select **Limit selection.**

By selecting a surface first, the form opens with it assigned as the cutting object.

11 Click to change to the **Keep both** option.

12 Select the cylinder surface.



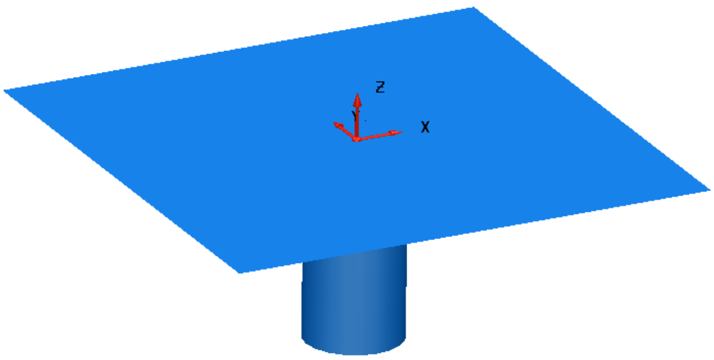
*When **Keep both** is active, **Limit selection** splits the secondary surface selection into two surfaces, one on each side of the Cutting Object.*

13 As before, close the form then Undo.

14 Select the plane surface first and then Select **Limit selection.**

15 Click to disable **Trim both**.

16 Select the cylinder surface.



*When **Trim both** is disabled, the **Cutting Object** (primary selection) is not trimmed back to the secondary selection.*

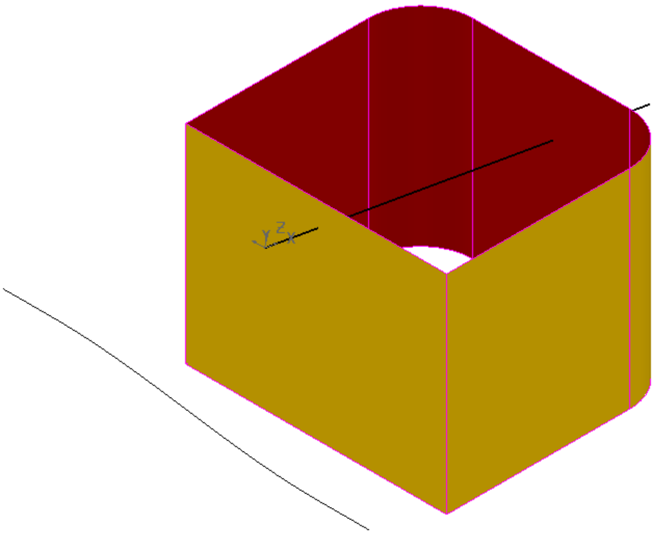
17 **Close** the model without saving.

Limit Surface using a Curve as the Cutting Object

A **wireframe entity** can also be used as the **Cutting object**, This as shown in the following example.

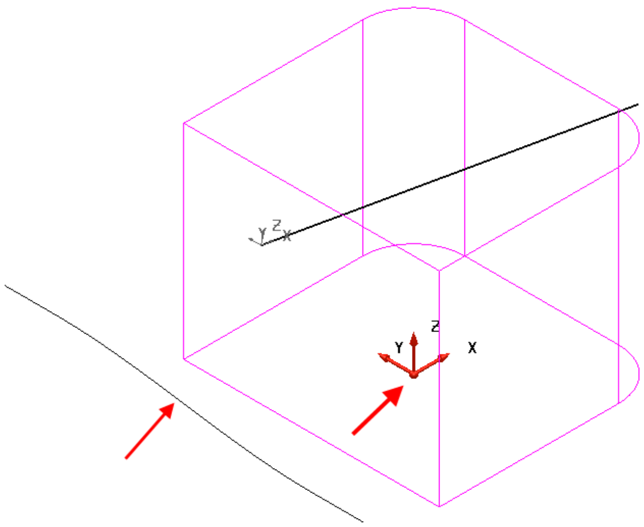
1 Open a **new model** and **import** the following data;

.....\PowerSHAPE_Data**Limit2.dgk**



The imported model contains a component made up of six separated surfaces and some wireframes defining the 2D trimming requirements.

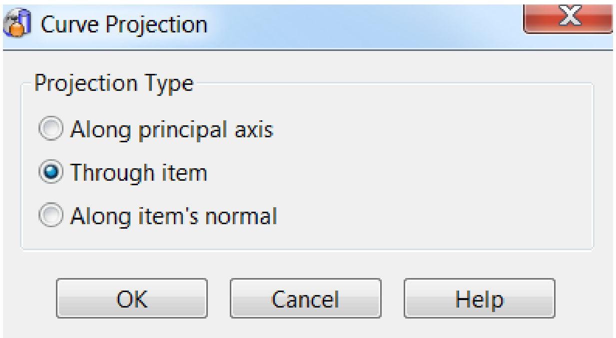
2 Activate **Workplane 1** located at the base of the component.



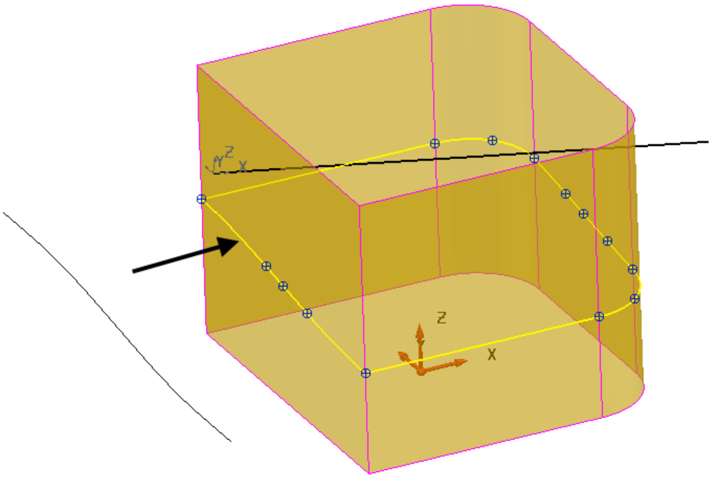
3 Select **X** as the operational direction.

4 Pre-select all the surfaces and the composite curve aligned on to the left (as shown above).

5 Select **Curve projection**  from the **Curve Toolbar.** 



6 Select the option **Through item** then OK.



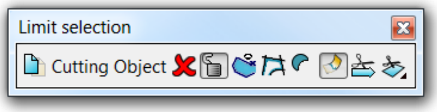
A composite curve has been generated through the component in the X axis.

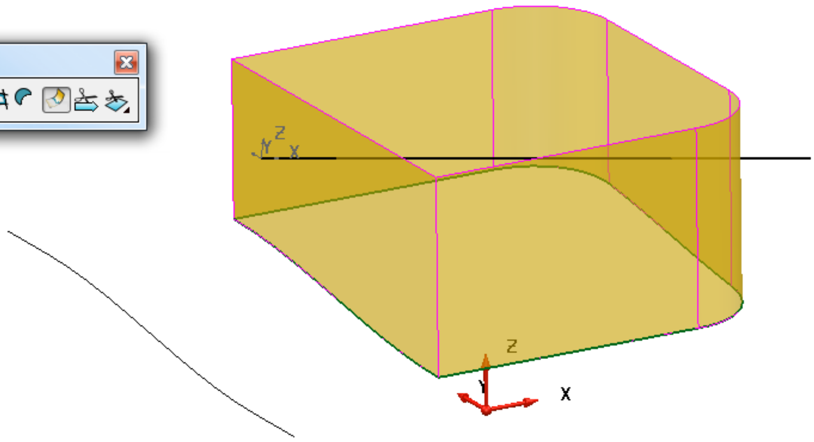
7 Select **Z** as the operational direction.

8 Select the new composite curve then **Limit selection.** 

*The **curve** is the cutting object.*

9 Drag a box to select all the surfaces and, *if required*, click **Next solution** to produce a trim retaining the parts of the surfaces above the split **curve**.



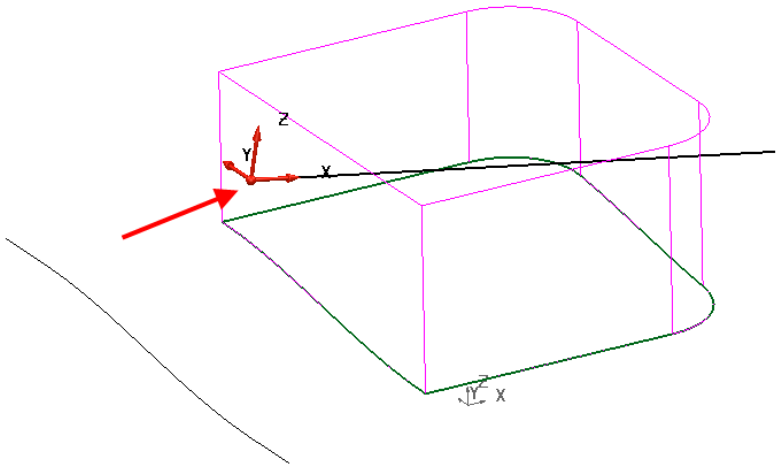


10 **Close** the form.

Limit Surface using a Workplane as the cutting object

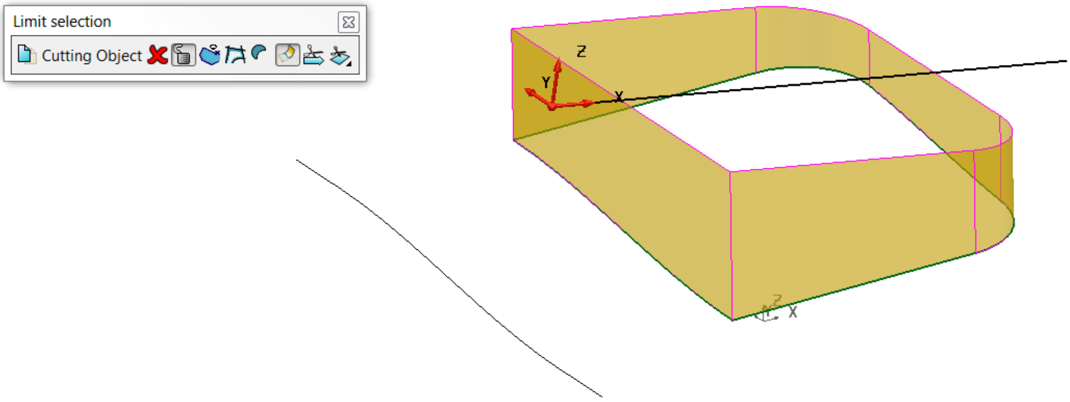
A **Workplane** can also be used as the **cutting object** as shown in this example.

1 Activate **Workplane 2** as shown.



2 Select the **Workplane** itself then **Limit selection.** 

3 **Box select** all the surfaces and, *if required*, click **Next solution** to produce a trim retaining the parts of the surfaces below the workplane (-Z).



4 Click the cross at the top-right corner to close the **Limit selection** toolbar.

5 **Close** the model without saving.

Surface Filleting

As with **Solids**, **Surface filleting** is mainly used to create a smooth join between two or more surfaces.

A fillet surface is the developed outside contour of a ball if rolled between two or more surfaces. **PowerSHAPE** can create both fixed and variable radius fillets between surfaces. Where the ball makes contact defines the outer edges of the fillet to where, by default the original surfaces are trimmed back to the fillet edge. It is also possible to create a fillet between a composite curve and a selection of surfaces.

PowerSHAPE creates concave fillets running from the outside (user-defined colour side) or **convex** fillets from the inside (red side) of the host surfaces. The **concave** filleting direction for each surface is marked by an arrow which, if clicked, reverses the surface causing a **concave fillet** to run from the other side. Otherwise, the **convex** option is applied.

Basic Fillet Example

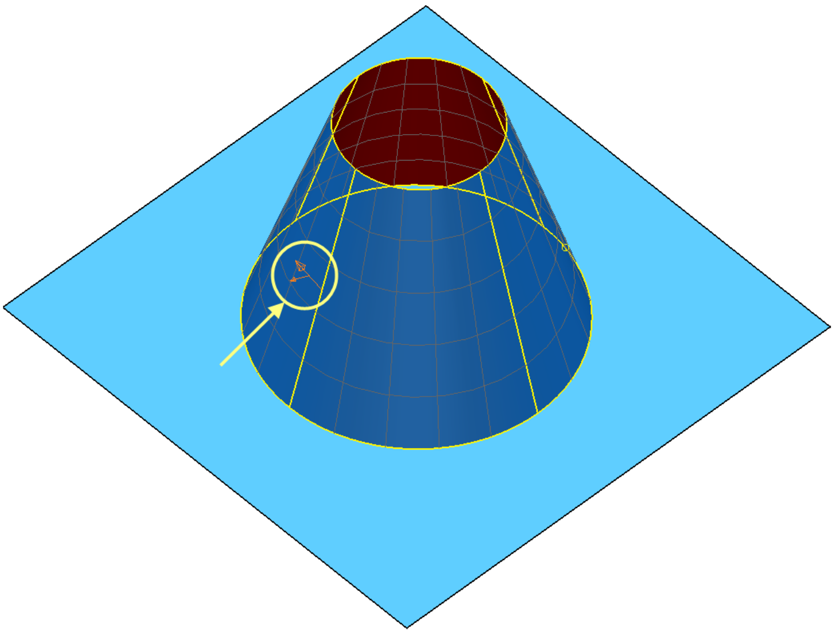
This **example** demonstrates the **basic filleting** options between **2 surfaces**.

1 Create a **new model.**

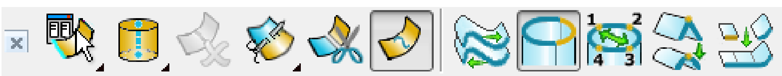
2 Create a **plane primitive** surface  of length **100** and width **100** at **0 0 0**.

3 Create a **Cone Primitive**  of **base radius 30, top radius 15** and **length 50** at **0 0 0**.

4 Right-click over the cone and select **Convert Surface** (converts to Power Surface).



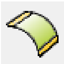
A selected Power Surface displays more information graphically and editing options the Surface editing toolbar.

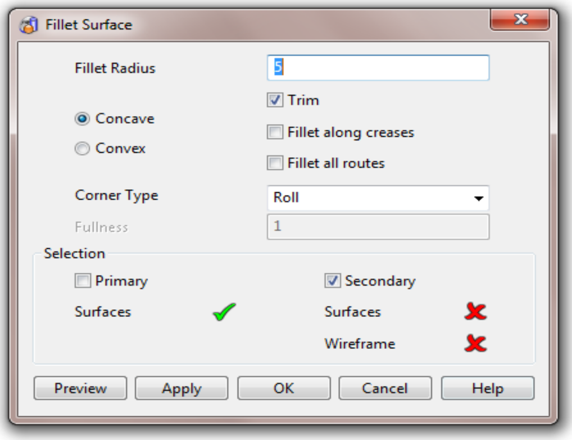


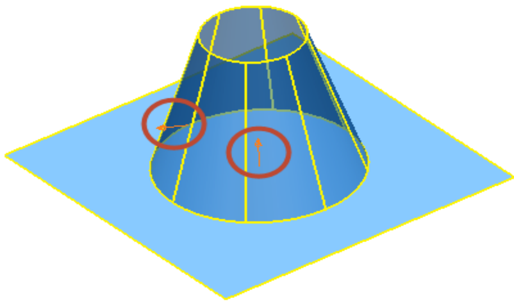
The small arrow points outwards from the surface, indicating that this is the outside edge of the surface. The large arrow indicates the operational direction for editing surface points

The large arrow does not affect filleting.

5 Select both surfaces and switch to **Transparent Shading**. 

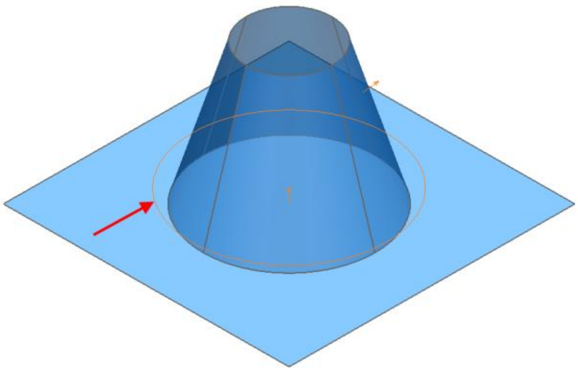
6 Select **Surface fillet**  from the **Surface** toolbar. 





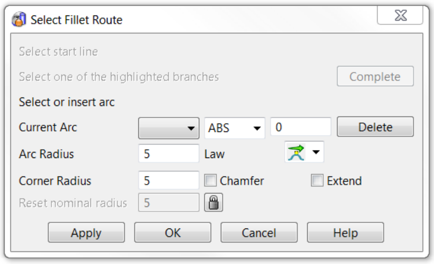
The surface direction arrows are displayed. If clicked, these turn the surface inside out. The small arrow points away from the outside (user-defined colour) face of surface.

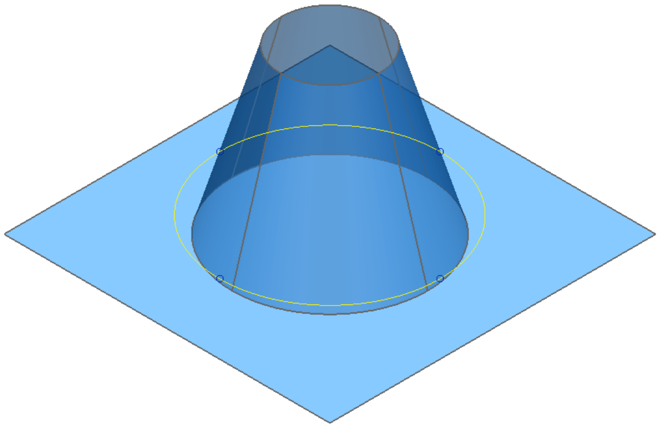
7 Leave the default **5mm radius** and concave setting, and select **Preview** on the **Fillet Surface** dialog.



The fillet track is displayed as the centreline of the rolling ball as it runs along the selected surfaces. If it is acceptable, select OK, otherwise enter alternative values and select Preview again until the end result is acceptable.

8 Select **OK** on the dialog.



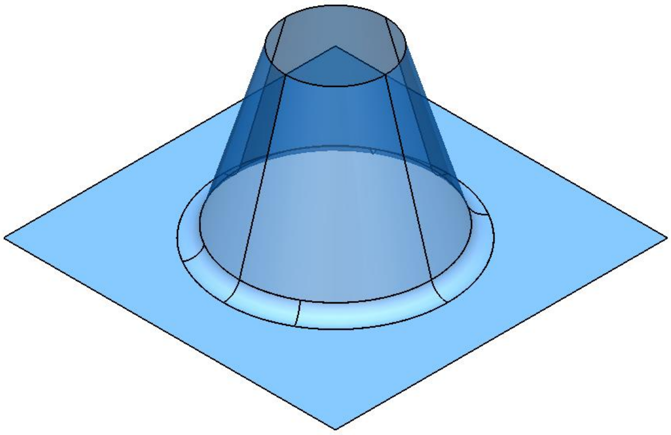


The Select Fillet Route dialog is displayed.

The single drive curve now turns yellow to show that it is selected. As there is only one track available in this case, select OK to create the fillet.

For examples with more than one track, no track is pre-selected. Select each one and Apply in turn, until all the required fillets have been created. Then, select OK to finish.

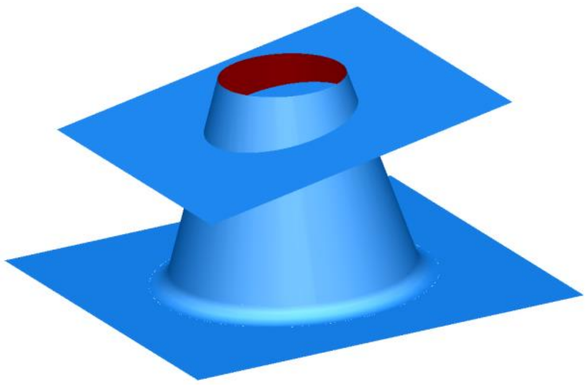
9 Select **Apply** followed by **OK** in the dialog.



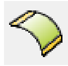
The fillet surface is produced. The cone and plane are trimmed back to a 5mm concave fillet.

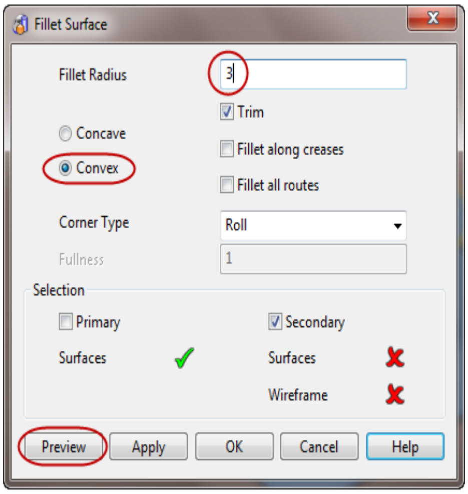
10 Create a Plane Primitive at **0 0 40** with a **length** and **width** of **80**.

11 In the **Workspace Tab** of the Plane form, apply a **Y Twist** of **10** degrees.



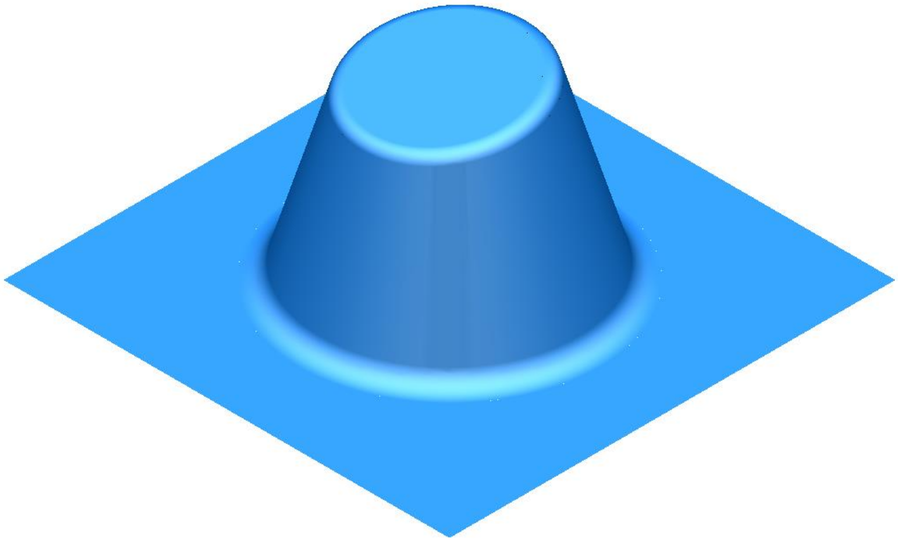
This top surface will be filleted to the cone, but this time the fillet is on the inside of the cone.

12 Select just the cone and the angled plane surface, then **Surface fillet**. 



13 Enter a **Fillet Radius** of **3** mm, select **Convex** and select **Preview**.

14 Select **OK** (fillet surface) and then **OK** (Select fillet route).



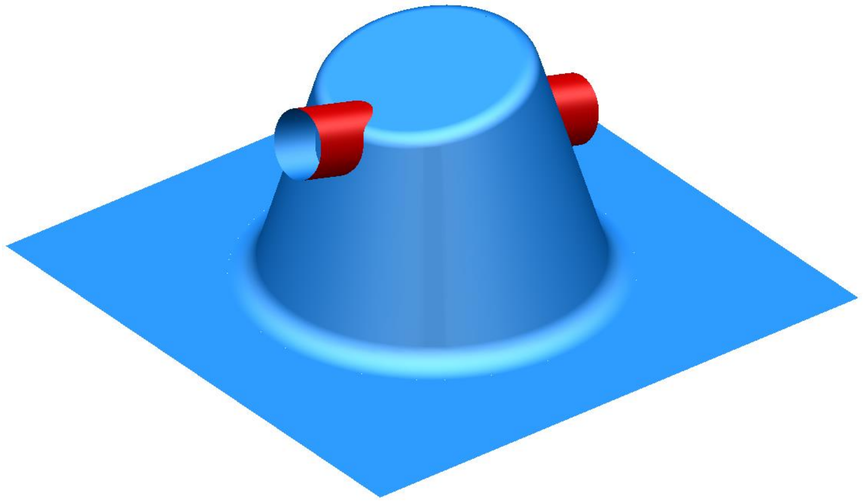
The fillet is produced trimming back the top of the cone and the plane.

Fillets can also be produced with several surfaces.

15 Select **X** as the **operational axis** and create a Cylinder Primitive positioned at **–30 0 20**.

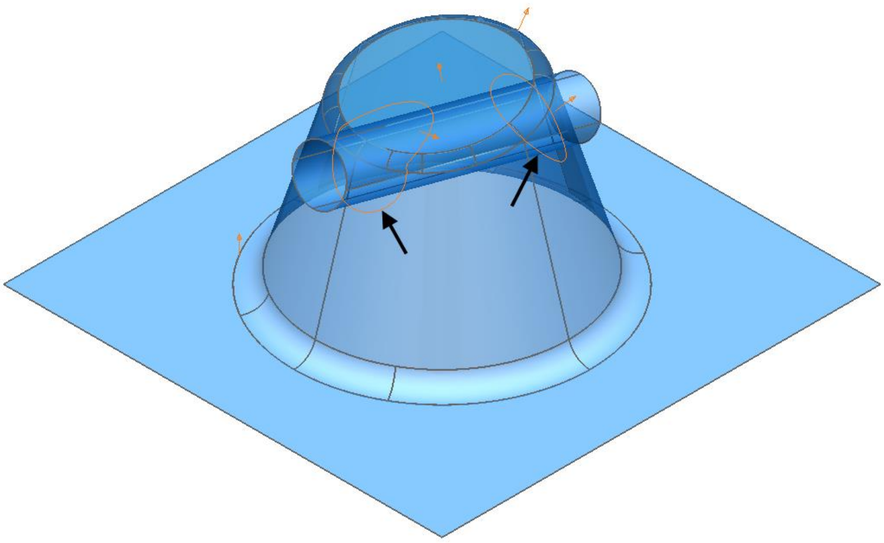
16 Change the **radius** to **6**, **length** to **60** and rotate it with an **X-Twist** of **–15**.

17 Select and reverse the cylinder surface only (red on outside).



18 Select all of the surfaces except the lower base fillet and plane. Then **Surface fillet**. 

19 Enter a **Fillet Radius** of **2 mm**, select **Convex**, and select **Preview**.



The available fillet tracks are displayed. The cylinder was initially reversed to control where the fillet tracks appeared.

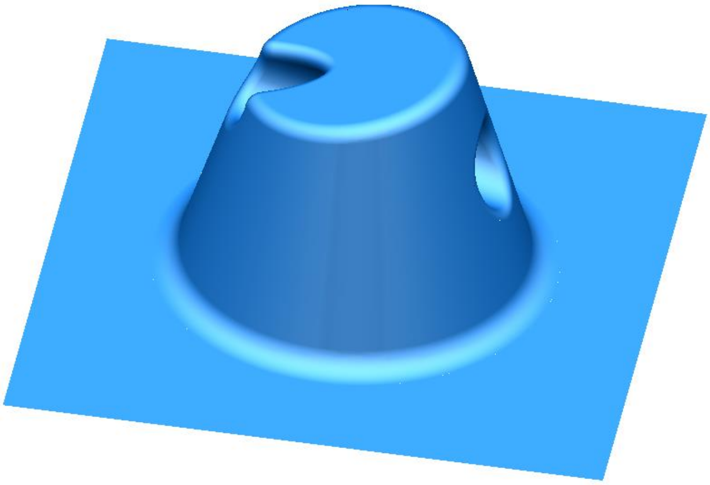
20 Select **OK**.

In this case, PowerSHAPE has identified two complete fillet tracks. PowerSHAPE allows you to specify which track you wish to use. Fillet tracks are created only between the two separate groups and not surfaces within the same group.

21 Select the first fillet track nearest the plane (lowest one), which turns yellow.

22 Select **Apply**.

23 Select the second track nearest the top, select **Apply.**



The surface cylinder is limited back to the fillets. By reversing the cylinder, the fillets are produced internally and not on the outside.

Filleting using Primary and Secondary sets of surfaces

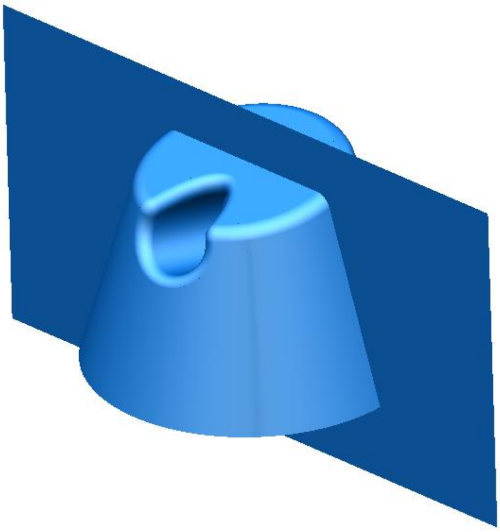
The **Filleting** option checks every selected surface to find all the potential tracks. Where there are a large number of surfaces involved, this can take some time, and it may be awkward to identify the required fillet tracks.

By using the **Secondary Selection** option, it is possible to register the two separate sets of surfaces for the fillet to run between. With this option, the first pre-selection of surfaces (yellow) is registered to the Primary (selected) set. If the **Secondary** box is then selected, any further surface selection is registered as the Secondary set of surfaces (pink). PowerSHAPE only attempts to create the fillet tracks between the separate groups.

1 Select the lower plane and fillet and delete them.

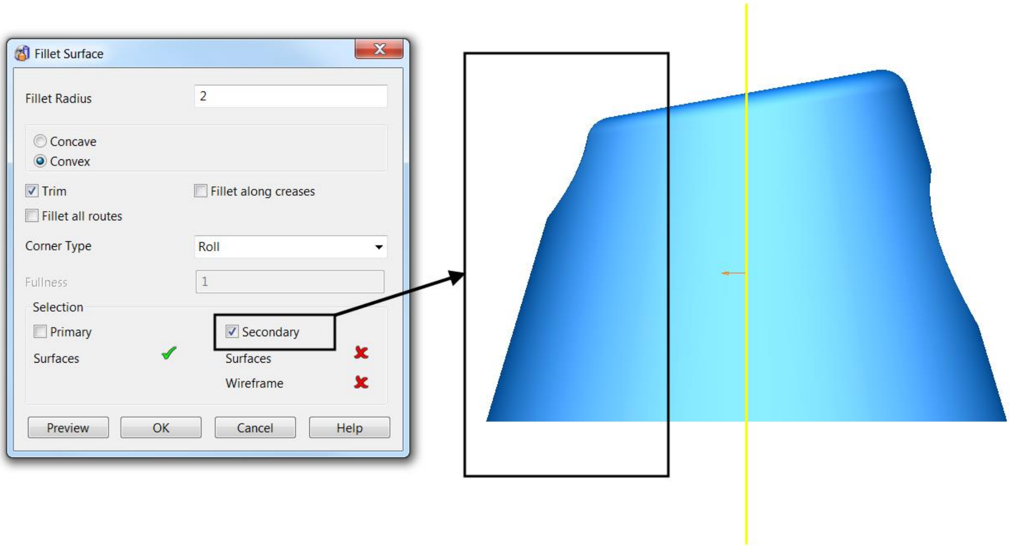
2 Select **X** as the **operational axis** and create a Plane Primitive at **0 0 20** with a **width** of **110** and a **length** of **60**.

3 Ensure the positive side of the surface is facing **(+X)** as shown.



4 Select the Primitive plane surface then **Surface fillet**. 

This surface is now assigned as the primary surface.



This set of selected surfaces will become the secondary set. The fillet routes are calculated only for the intersection between the primary and secondary surfaces. This results in a quicker calculation for the fillet routes. As fewer routes are produced, it is also easier to select the correct ones.

5 Drag a box over the surfaces on the left as shown, to assign as the secondary set.

6 Set a **Fillet radius** of **2** mm and select **Convex**.

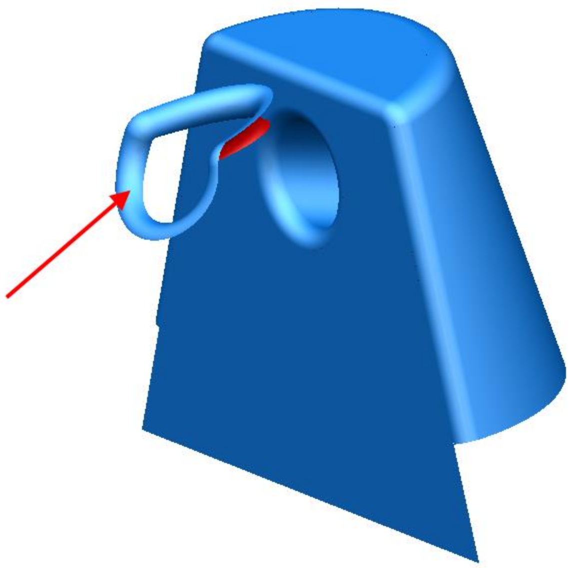
7 Select **OK**.

8 Select the first track and then select **Apply**.

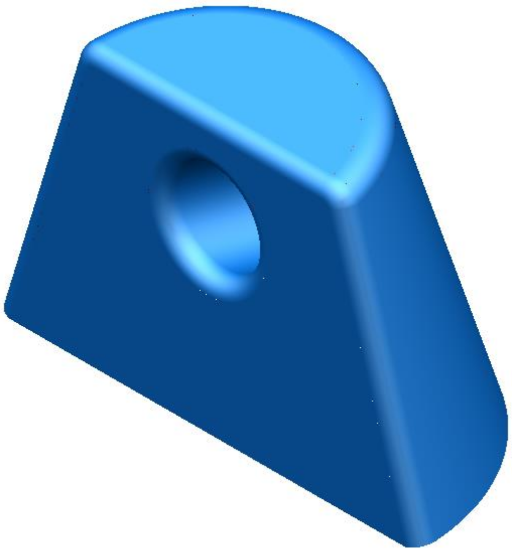
9 Select the last track and then select **Apply**.

The fillets are produced and most of the other surfaces have been trimmed back.

10 Delete the detached surface.



11 Use a suitable method to limit the bottom of the plane back to complete the model.



12 Select **File > Save As**:

.....\PowerSHAPE-Models\fillet example.psmodel

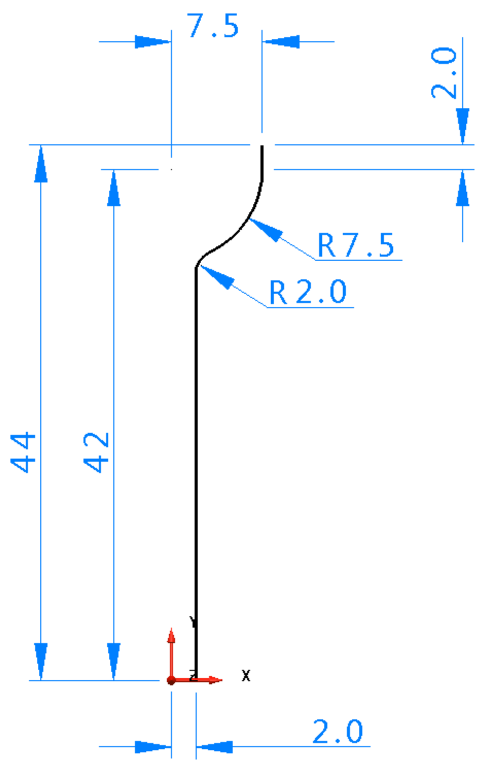
13 Select **File > Close**.

More filleting using Primary and Secondary surfaces

The **following example** uses a simple **tap design** to demonstrate **fillet** creation between a **Primary** and **Secondary set** of **surfaces**.

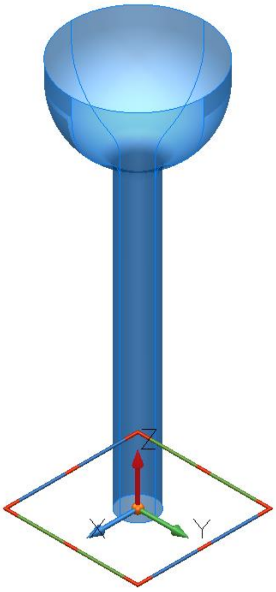
1 **Create a New Model and Workplane at 0.**

2 Generate the following wireframe on the XY Plane.



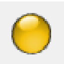
3 Create a **surface of revolution**  around the **Y axis**.

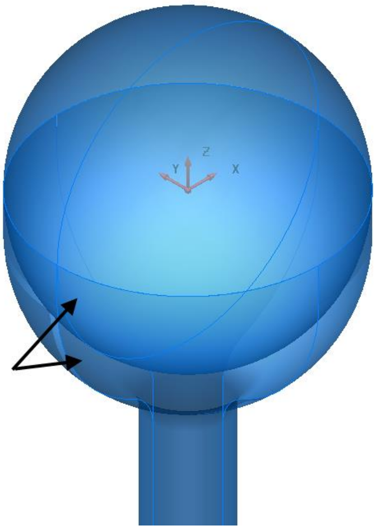
4 **Rotate**  the new surface **90** degrees about the **X Axis** to align it to the **Z** direction.



5 Switch the operational axis to **Z.**

6 Create a new **workplane** at **0 0 44**.

7 Generate a **Sphere Primitive**  of **radius 7.5** at the new workplane.

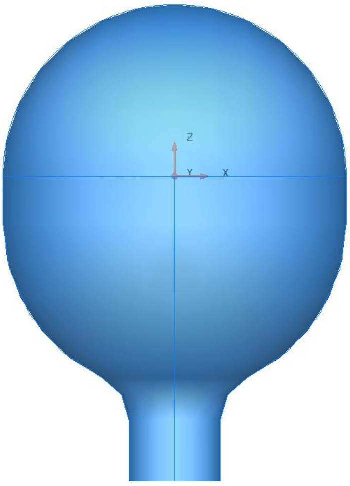


The lower half of the sphere will be limited to remove the overlapping surfaces.

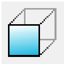
8 Select the workplane, then **Limit selection.** 

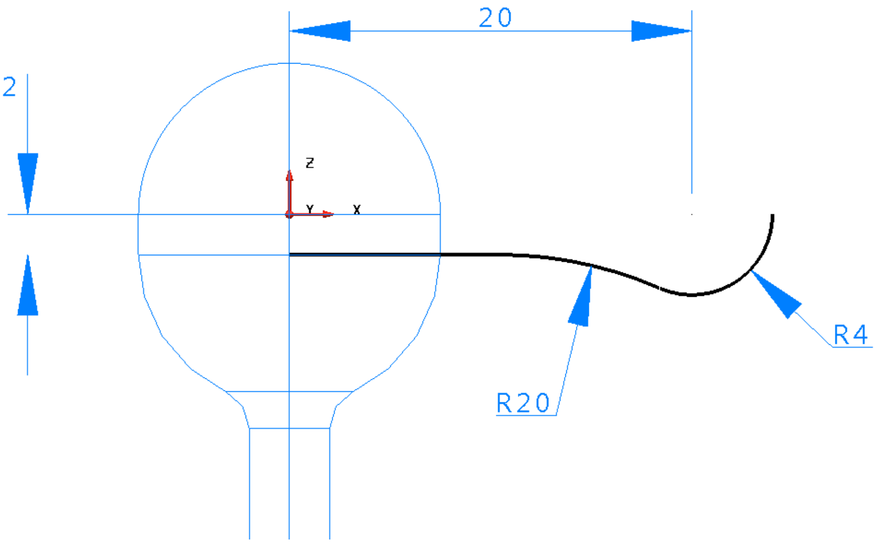
9 Select the Primitive Sphere, and, if necessary, select **Next Solution** to retain the top half before closing the dialog.

Ensure the correct operation axis is selected first! (Limiting occurs in the direction of the active Principal Plane)



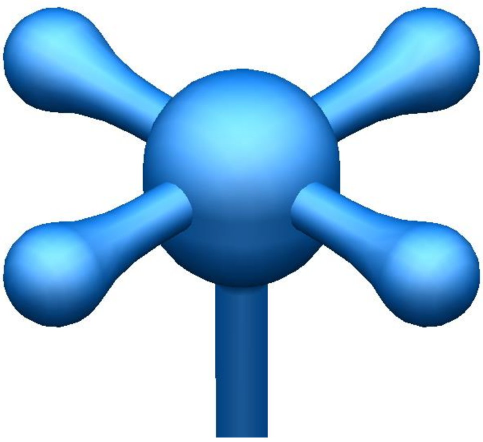
The Primitive Sphere is limited back to the workplane with the top half being retained

10 Select a **view from Front (-Y)**  then create the following wireframe geometry for the tap lever shape around the workplane.

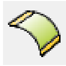


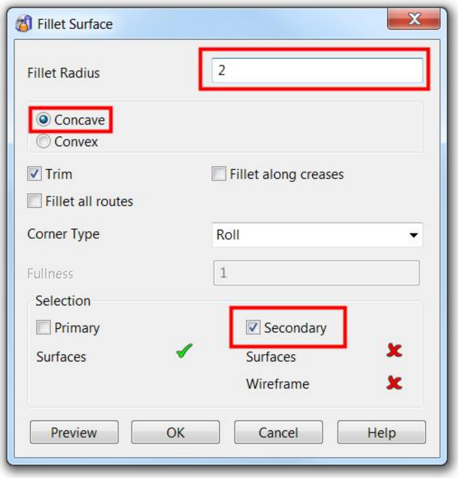
11 Create a **surface of revolution** around the **X** Principal Plane.

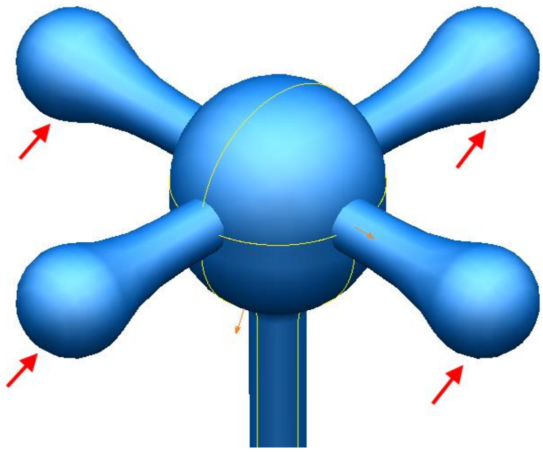
12 Rotate and copy this surface 3 times by **90** degrees around the Z Principal Plane to make the four levers.



The **four arms** need to be **filleted** to the **central section**. Make sure that when **shaded**, all the **surfaces** are **positive**. We do not want to **fillet** each arm to each other or to generate the route, so we are going to use the Secondary surfaces option.

13 Select both the shaft and sphere surfaces then **Surface fillet**. 



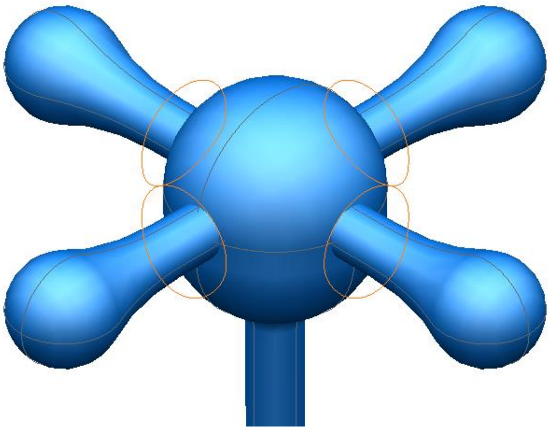


With the **primary surfaces pre-selected**, the dialog now awaits the **secondary surface selection**.

14 Select the **four lever surfaces**.

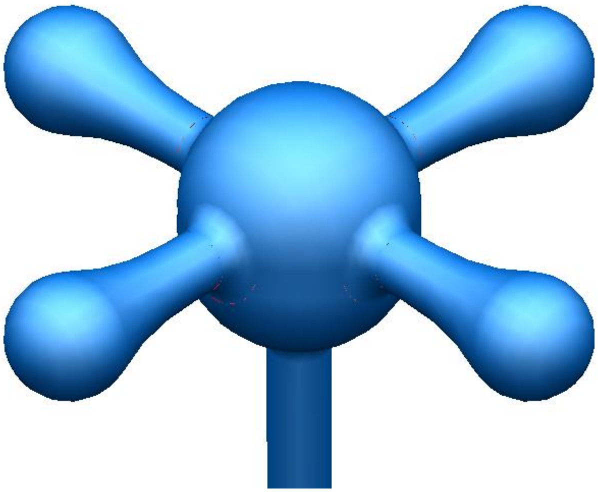
15 Enter a **Concave Fillet** of **Radius 2** then **OK** the **fillet dialog**.

16 Select **OK**.



The **fillet tracks** are **displayed**.

17 **Select** and **Apply** each one to complete the exercise.



18 Select **File > Save As**:

.....\ ***PowerSHAPE Data\surface tap.psmodel***

19 Select **File > Close**.

Variable Radius Fillets

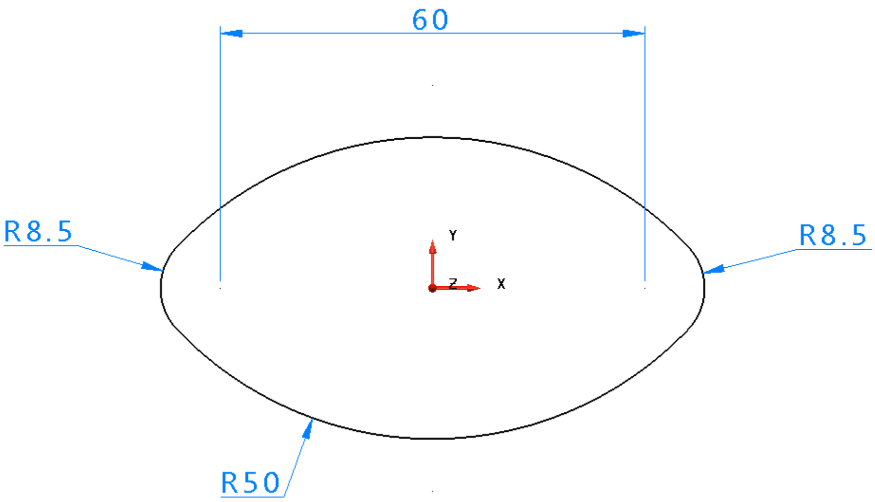
Variable radius fillets can be created along the **fillet track** using the **mouse** to **identify key points** or by **applying specialist options** to position the **fillet arcs** either by **parametric**, **relative** or **absolute values**.

Example

20 Select **Create New Model** and workplane at **0**.

21 Create two circles with **radius 8.5** at **X 30** and **X -30**.

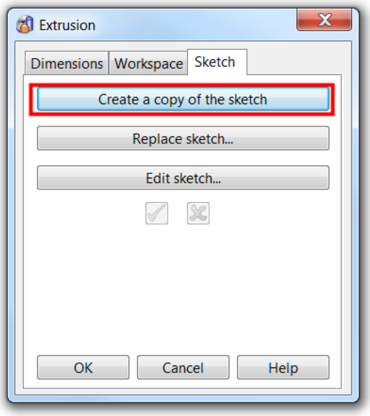
22 Create an **arc through three points** of **radius 50** on both sides of the two circles to form the following shape.

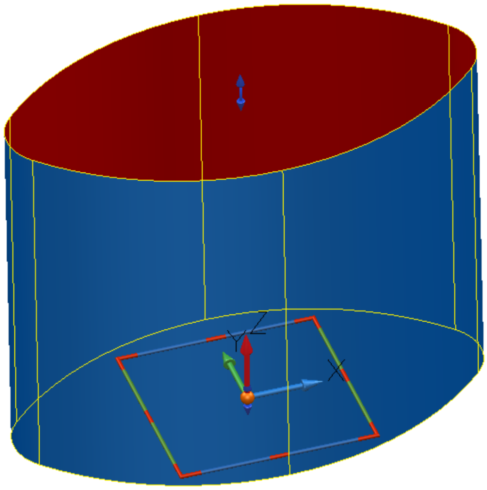


23 Create a Composite Curve from the wireframe.

24 Generate a **Surface Extrusion** of **length 60**.

When an **Extrusion surface** is created, the default option causes the wireframe used to be deleted. This default status can be altered to retain wireframes by selecting **Tools > Options > Object > Surfaces>Keep wireframe (extrusions and revolutions).**





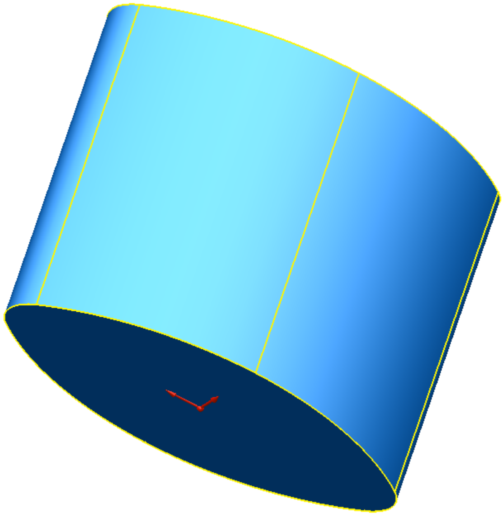
Alternatively, the individual wireframe for an extrusion can be retrieved directly.

25 From the sketch Tab, select **Create a copy of the sketch** then OK.

This returns a copy of the original composite curve to be used to create a Fill in surface at the base of the extrusion.

26 Select the new composite curve and create a **Fill in** surface.

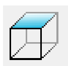
27 Reverse the surface so the red side is facing up the **Z** axis.

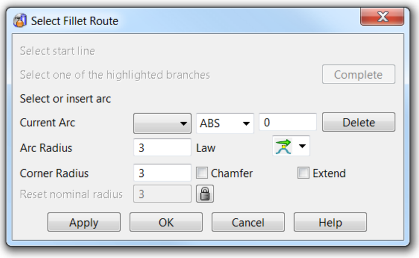


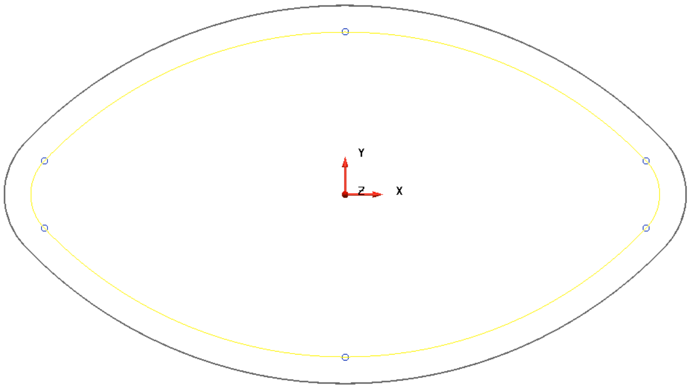
28 Delete the composite curve.

29 Select both surfaces then then **Surface fillet**. 

30 Input a **Fillet radius** of **3,** select **Convex** and select **OK.**

31 Select a **View down +Z**  and switch to **wireframe** view.





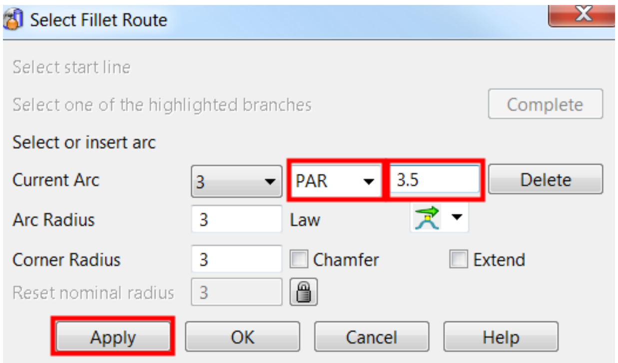
The **fillet track** is displayed (in **yellow**) with circles (in **blue**) showing **Key points** that can be **snapped** to using the mouse. By snapping at these **Key points**, an **arc** of the **initial radius** is created. The **value** of this **radius** can be **modified** in the **Arc Radius** area of the form.

32 **Snap** to each of the **6 circles** when the word ***Key*** appears.

33 **6 arcs** have been **created** and can be **selected** from the **Current Arc** drop down list.

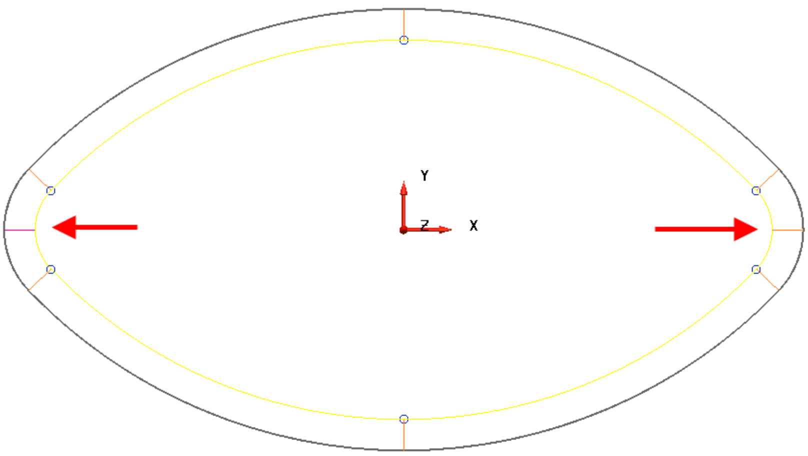
A **radius** appears for each point and this becomes numbered in the current arc part of the form. Individual arcs can be selected from this form or by manually clicking on them. The radius can be changed in the form or dragged manually.

Graphically, selected arcs are displayed in blue with unselected in red.



New laterals are added either by clicking on the track, or, more accurately, by entering in the form as a parametric position. To add a lateral at a parametric position, select the adjacent laterals, and check their number on the form in the **Current Arc** box. Switch the **ABS** box to **PAR**, enter the required parametric position (such as **4.5** for midway between **4** and **5**) and press **Return**.

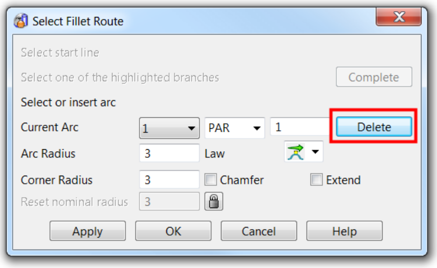
34 Select the correct arcs (displayed in blue) and insert two new radii between the two at each side, as shown.

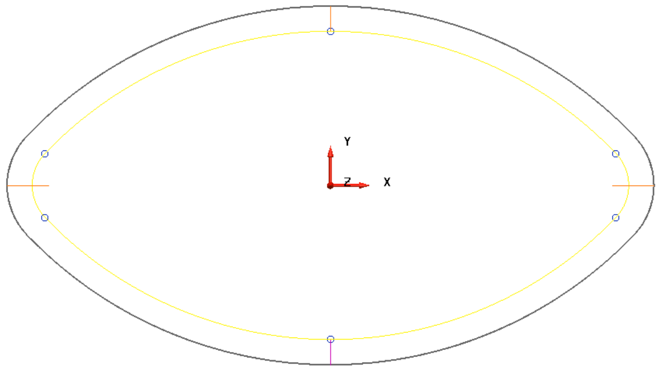


While the form is **active**, the values for **individual fillet arcs** can be **edited** or **deleted**. The **radius progressively changes** between the **two arcs** assigned with **different values**.

35 Select the **new arcs** in **turn** and **change** the **radius** to **5 mm**.

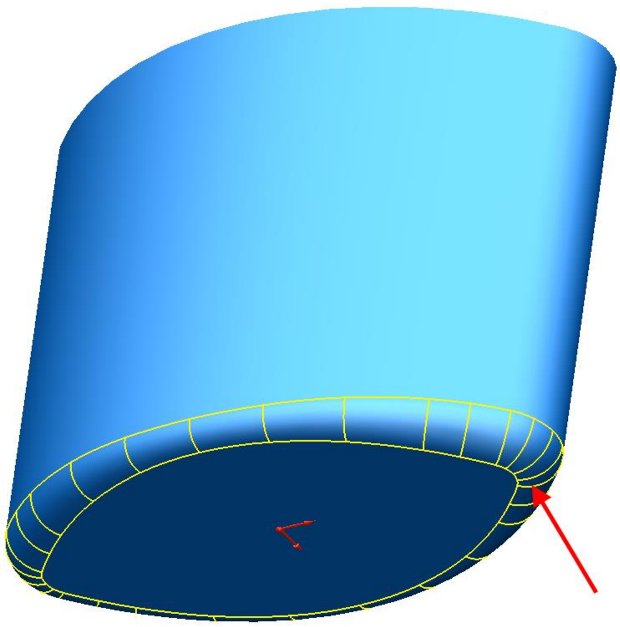
36 Select the **arcs** (in **turn**) on **each side** of the **new larger arcs** and select **Delete** on the dialog (4 arcs in total).





When there is an arc, the fillet value is fixed at that point. If the next radius value is different, the radius changes smoothly along the distance.

37 Select **Apply** to create a variable radius fillet.



38 **Close** the model without saving.

9. Surface Editing

Editing a Power surface

A **Power Surface** is a surface created from wireframe items.

If you create or select a power surface, the **Surface Editing toolbar** will be displayed and the options that are appropriate for your surface are made available.

There are three entities that you can edit on a surface:

- the **surface** itself.
- the **curves within** the **surface**.
- the **points on** the **surface within** the **curves**.

Editing curves and points on a surface

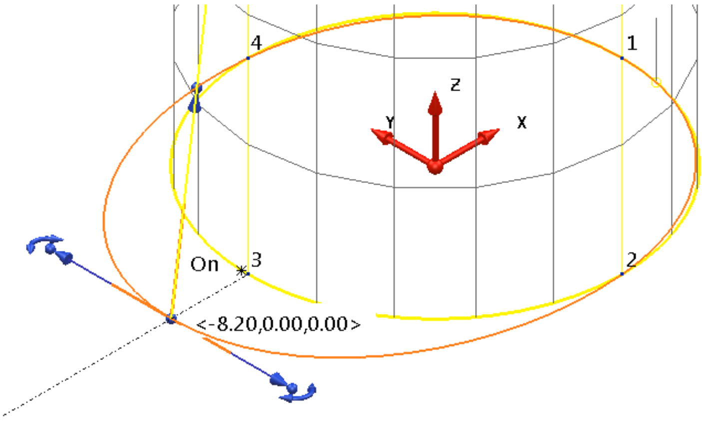
Editing **curves** and **points** within a **surface** use the **same methods** as **wireframe curves**.

You can use **Edit > Convert > To Wireframe** to create wireframe curves from selected surface curves (laterals, longitudinals and spines).

Editing the power surface

You can edit a power surface in a number of ways, most simply:

- With the power surface selected, use the **graphical surface editing features** to edit it. For example below, a control point is being dragged.



- Select the surface to display the **Surface** and **Curve Editing** toolbars.

You must have curves selected for the following options to be available from the right mouse local menu.

Convert to wireframe - A composite curve is created from the selected curve.

You must have curves or points selected for the following options to be available.

Edit tangent angles - This displays the **Tangent Editor** dialog, which allows you to edit the tangent directions and magnitudes of curves.

Free magnitudes - This frees tangent magnitudes through selected points along the selected surface curve. If no points are selected, the whole curve is freed.

Free tangent and magnitudes - This frees the tangent directions and magnitudes through selected points along the selected curve. If no points are selected, the whole curve is freed.

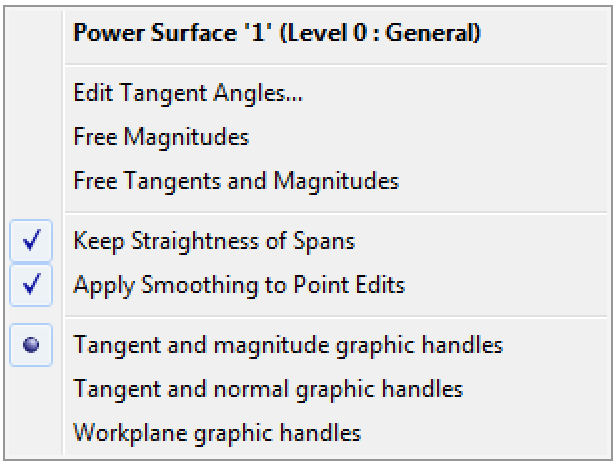
Single Surface Curve Point Edits

In the same way as curve editing covered earlier, **single points** on **surface curves** can be edited directly in the **graphics window** using a variety of different **graphical edit handles**, as well as using the **Command Box** on the **Status Bar**. Use the handles on curve points to graphically modify single surface curve points. The handles will display only when a single point is selected.

It is possible to change they different curve point handles within **Edit>Surface and Curve Edits>Common Edits> Selected Point Graphical Handles** or by right-clicking with a single point selected.

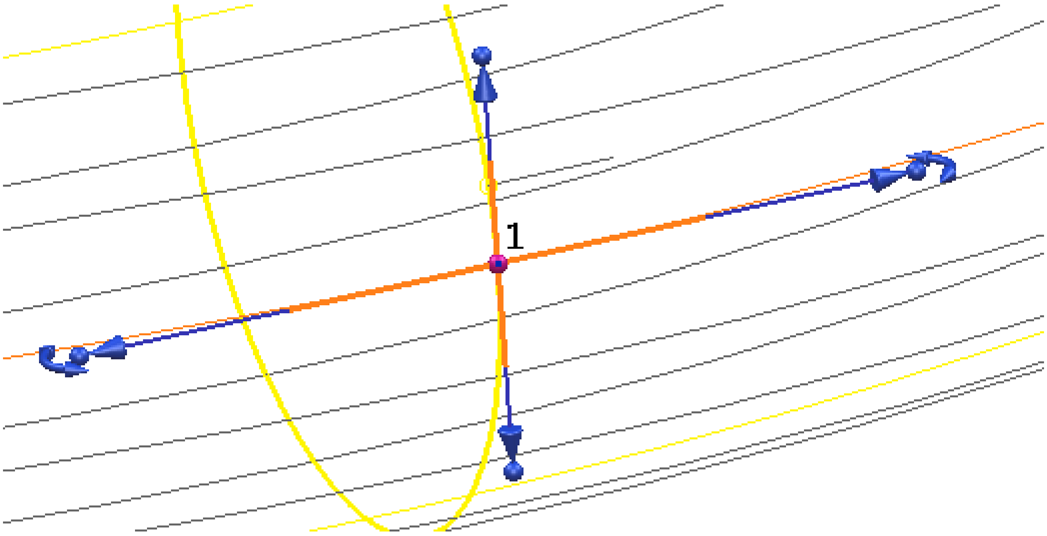
The following types of handles are currently available to use:

- **Tangent and Magnitude graphic handles**.
- **Tangent and Normal graphic handles**.
- **Workplane graphic handles**.



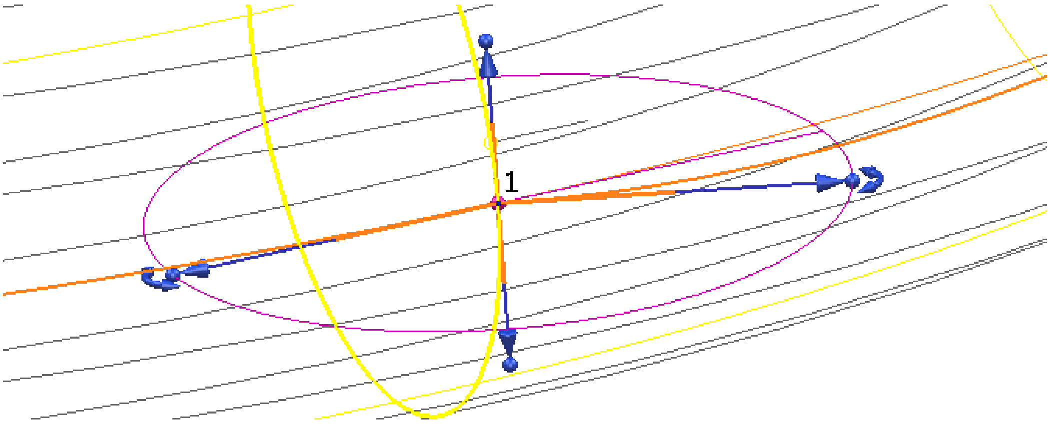
Tangent and Magnitude graphic handles

1 Select a **point** on the curve and right-click and ensure that **Tangent and Magnitude graphic handles** is selected in the menu.

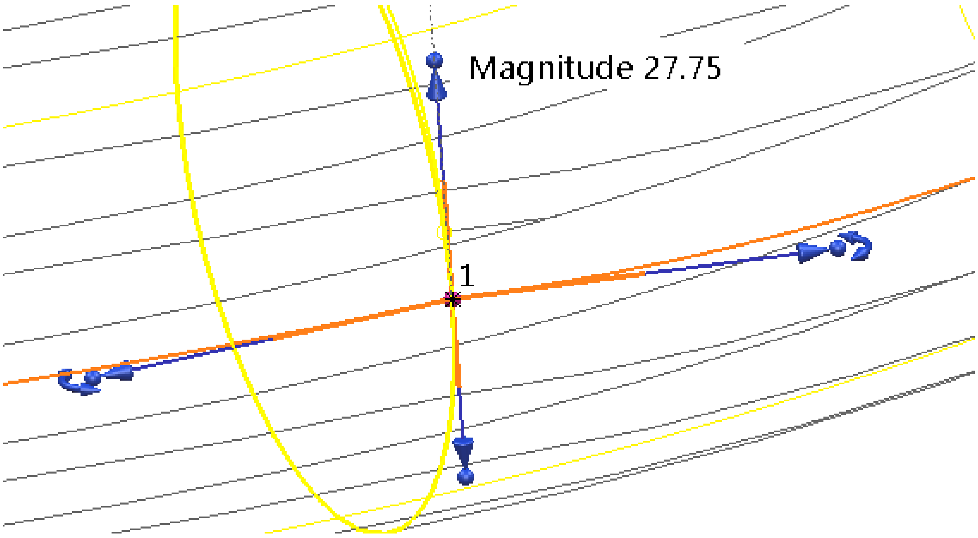


These handles allow the user to graphically edit the **Tangency** and **Magnitude** of the curve point.

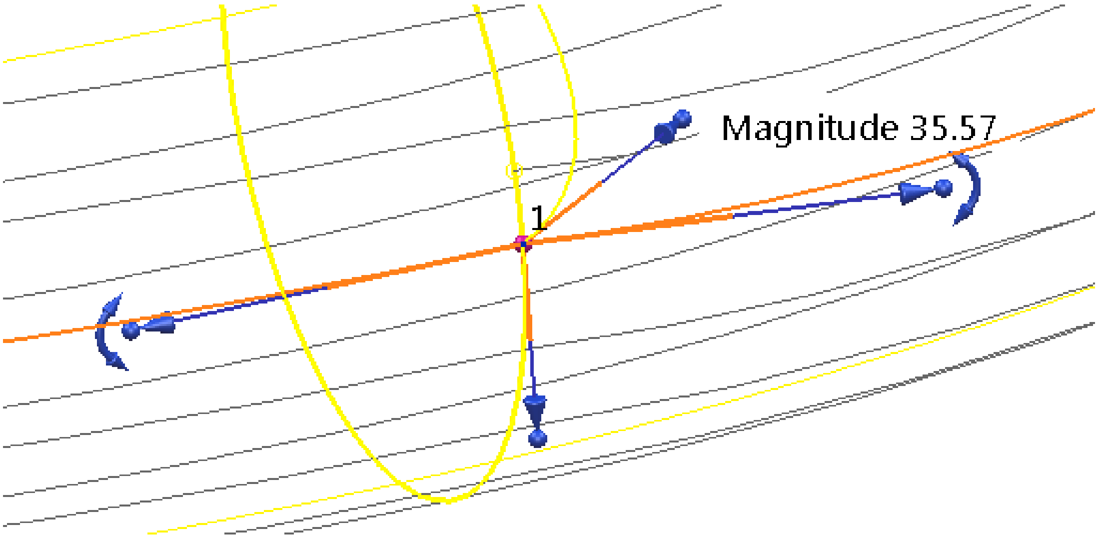
The **arced handles** offset from the end will modify the **tangent angle** of the **curve** in a **fixed circular motion**, without affecting the magnitude as can be seen below.



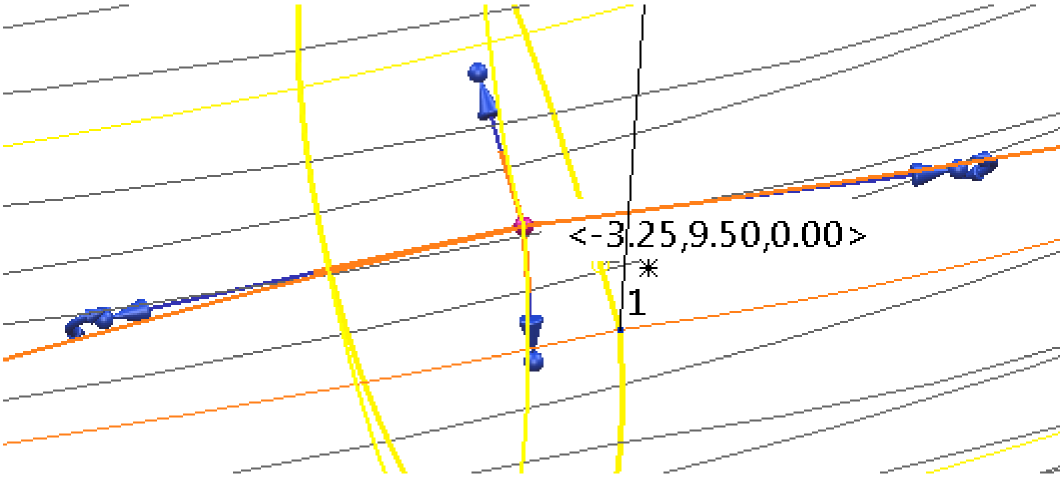
The **arrows** at the end of each handle will **graphically modify** the **magnitude** of the curve along a **fixed tangent angle**, in this case to help the user, PowerSHAPE will give interactive feedback of the magnitude.



The **Spherical** handles will **modify** both the **Tangent** and **Magnitude** freely in **2D** or **3D** space. Again, **PowerSHAPE** will give interactive feedback of the magnitude to the user.

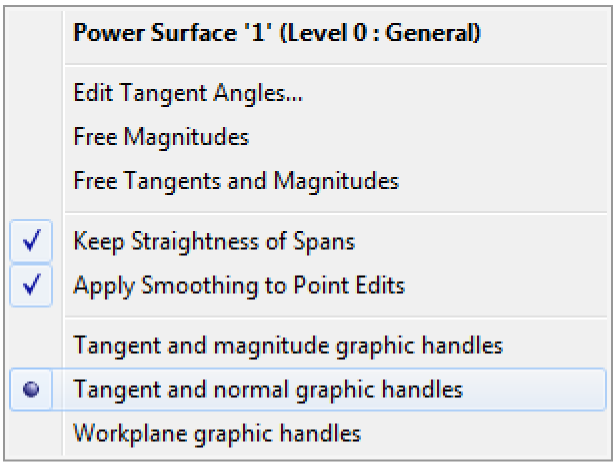


Finally, the **spherical handle** at the selected curve point position at the **centre** will **edit** the **position** of the **point**, without affecting the **direction** or **magnitude** of the point. For increased usability the intelligent cursor will remember the **original position**, and give coordinates (**X, Y, Z)** of the movement.

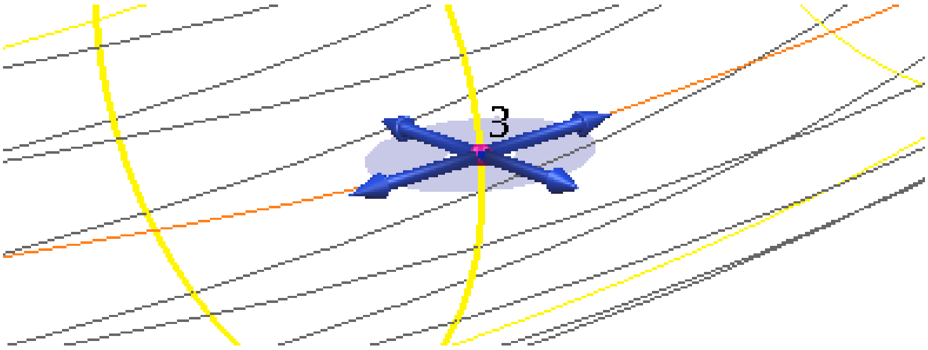


Tangent and Normal graphic handles

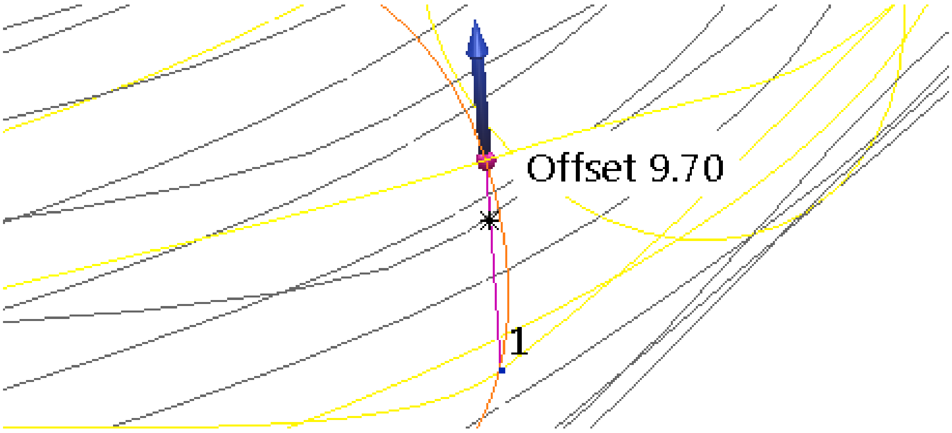
1 Select a **point** on the curve and right-click and ensure that **Tangent and Normal graphic handles** is selected in the menu.

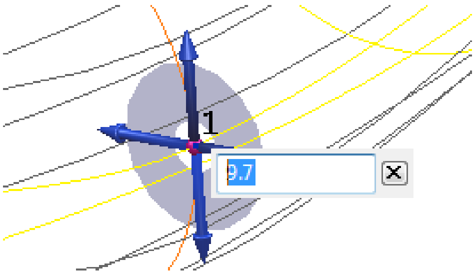


These handles allow the user to graphically edit the position of the selected point in fixed **Normal** or **Tangent** directions, and alternatively **freely in a 2D plane**.

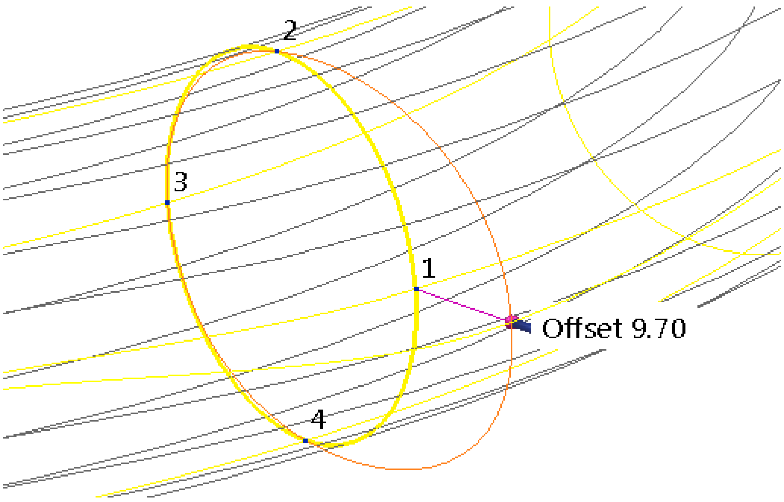


The **arrow** handles **tangent** to the curve will modify the **position** of the **point** in a **fixed direction** along the **tangent direction.** For accuracy the offset distance from the **orginal location** will be visible, as well as **editable** once the mouse button has been released.

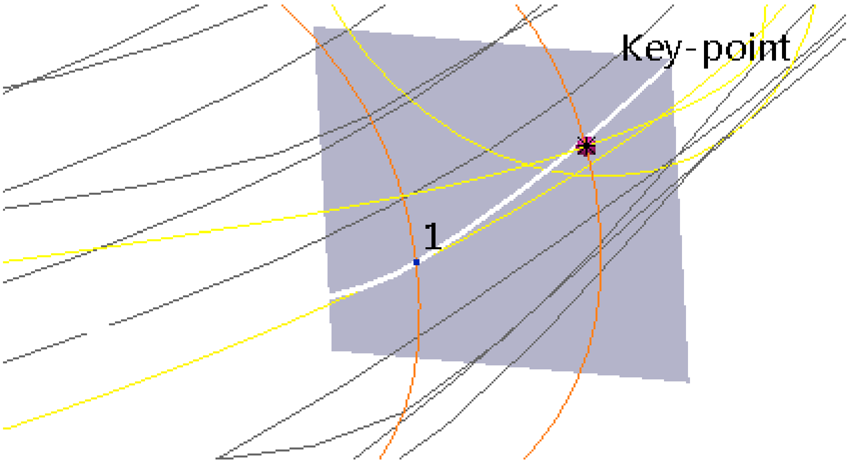




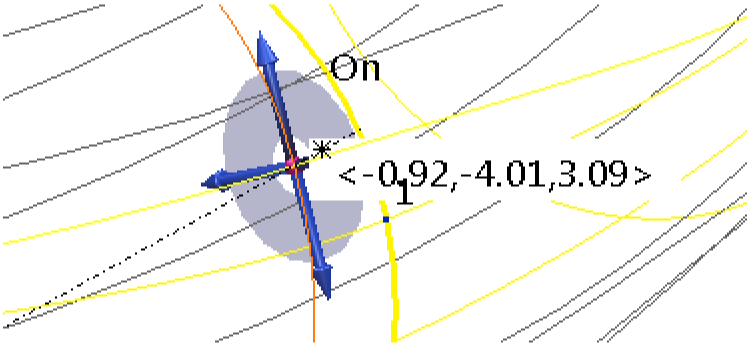
Similarly, the **arrow** handles **normal** to the curve will modify the **position** of the **point** in a **fixed direction** along the **normal direction.** For accuracy the offset distance from the **orginal location** will be visible, as well as **editable** once the mouse button has been released.



The **4 Planar Handles** surrounding the point will allow you to move the **point** freely in a **2D plane** while again the **original location** is kept visible throughout (**1**).

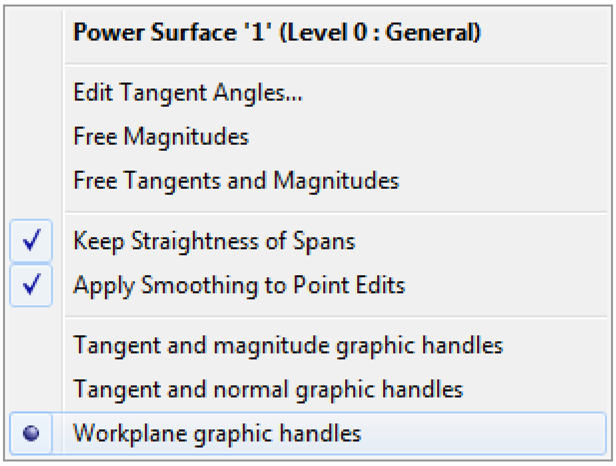


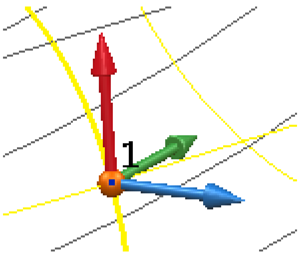
Again, the central **spherical handle** at the centre will move the point **freely** in **3D space** while showing **X**, **Y**, **Z** coordinates offset from its original location.



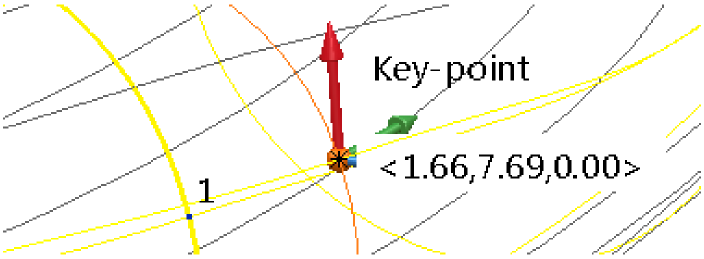
Workplane graphic handles

2 Select a **point** on the curve and right-click and ensure that **Tangent and Normal graphic handles** is selected in the menu.



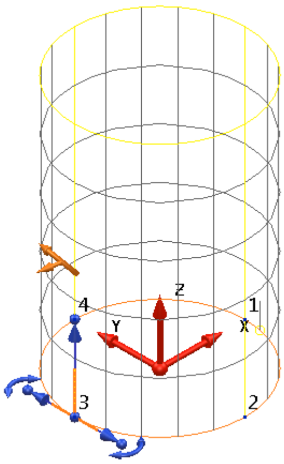


Clicking and **dragging** the **workplane axis** arrow handles will move the point in a **fixed axial** (**X**, **Y** or **Z**) direction with a **visible offset distance** from the original position, with the option to **further edit** or fine tune the offset once the mouse button has been released. The **workplane origin** in this case allows the point to be moved freely in **3D space**.

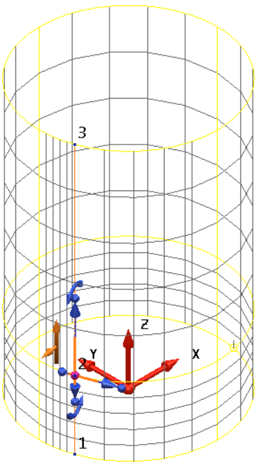


Graphically inserting a curve on a surface

When any point on a surface is selected, the handles of the point will be displayed. 1



- Hold down **Ctrl**, click on the instrumented point and drag the cursor to the position where you want to add the new curve. 2
- Release **Ctrl** and the mouse button to input the curve. This also renumbers the other curves to reflect the change.

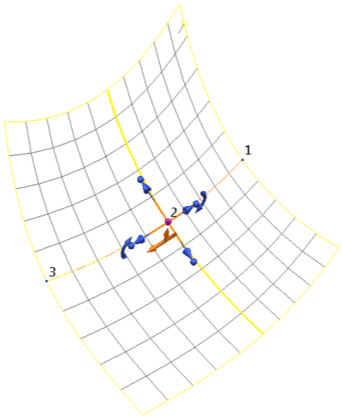


If you release the mouse over a lateral, only a longitudinal is inserted. If you release over a longitudinal, only a lateral is inserted. If you release the mouse at a position not on the surface, then no curve is inserted.

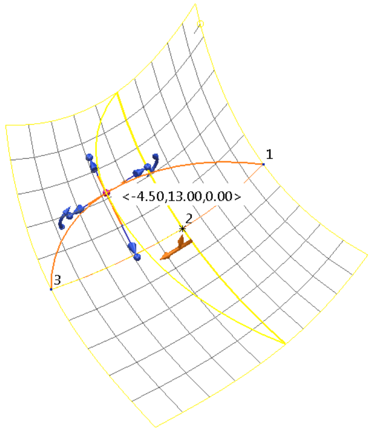
Graphically moving a point along a surface curve

You can adjust the position of a point along its adjacent curves.

When any point on a surface is selected, the handles of the point will be displayed.

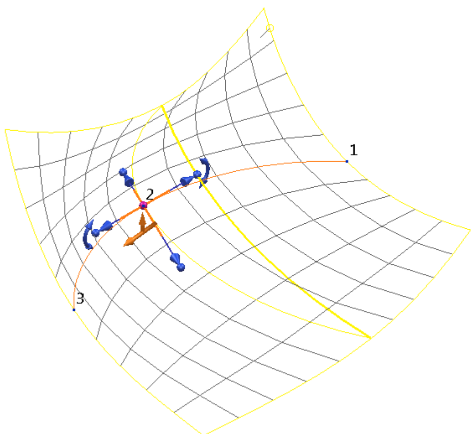


- Hold down the **Shift** key and drag the instrumented point along its adjacent surface curves.



- Release the **Shift** key and the **mouse button** to move the point.

  The shape of the surface does not change.



You can drag the point along a curve up to the next point on the surface. If you drag the point to any other position, the point is moved but the shape of the surface changes.

Editing a surface using the toolbar

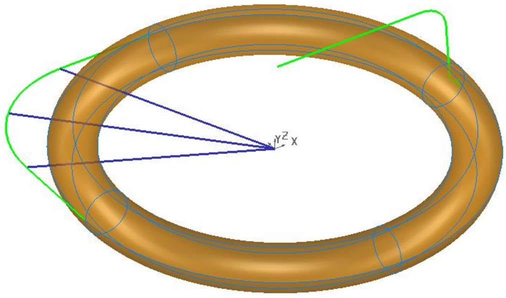
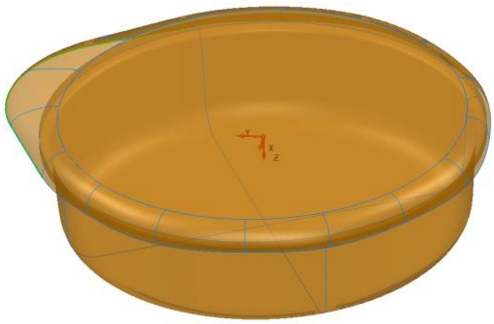
When you create or select a power surface, the **Surface Editing toolbar** will be displayed and the options that are appropriate for your surface are made available.

Use the following table to identify the buttons on the toolbar.

Option	Function
	Select curves on a surface
	Select all curves in current direction
	Add all curves to selection
	Select pattern of points
	Add surface
	Insert curve from wireframe
	Create spine
	Delete the curve
	Stitch the surface
	Match lateral to curve projection
	Break surface
	Trimming on - click to turn off
	Trimming off - click to turn on
	Reverse the surface.
	Surface is open - press to close
	Surface is closed - press to open
	Renumber surface points
	Spline surface
	Join two surfaces

Food Container Example

The project starts by importing the model shown below left, and progresses to the finished outer form of the food container. Key surface editing techniques will be shown.

1 **Import** the pre-defined CAD data located in:

.....\PowerSHAPE_Data**SurfaceEditStart.dgk**

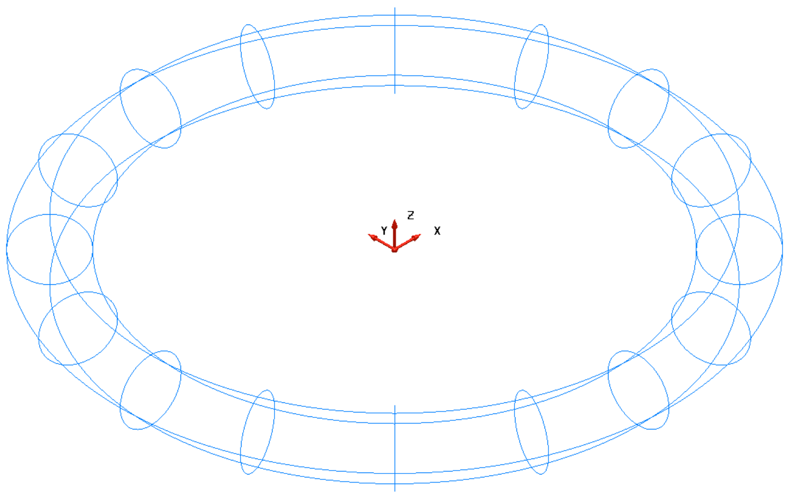
Conversion of primitive to Power Surface

2 Select and **Blank** the **wireframe** only .

3 Make **Workplane 1 active**.

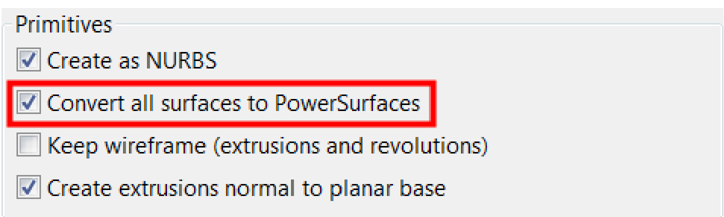
Before any changes can be made, this primitive surface will be converted to a Power surface.

4 **Right-click** on the surface and select **Convert Surface** in the local menu.

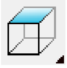


Depending on the settings, the surface is converted to a NURBS surface (the default) or directly to a Power surface.

The option can be changed in ***Tools>Options>Object>Surfaces.***

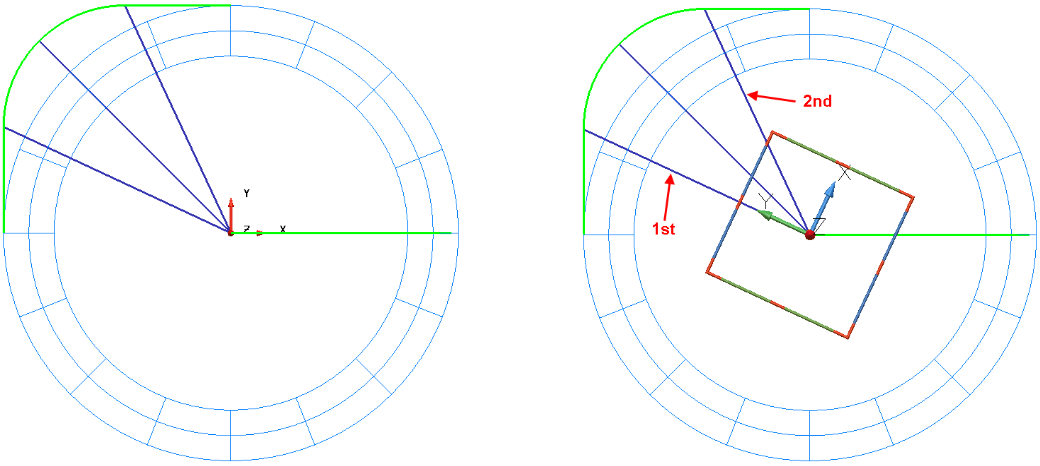


With the default setting above, a **Power Surface** has been created with all of the surface editing toolbar options available.

5 Select **View From Top (+Z).** 

6 **Unblank** to display all entities.

Dynamic editing of a surface curve



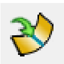
Two additional **surface lateral curves** will be added, using the lines above as a reference. Curves can be added in a number of ways. In this example curves will be added using the Workplane intersection option. The new curved will then be manipulated and aligned to the end points of the fillet radius along the route of the lip.

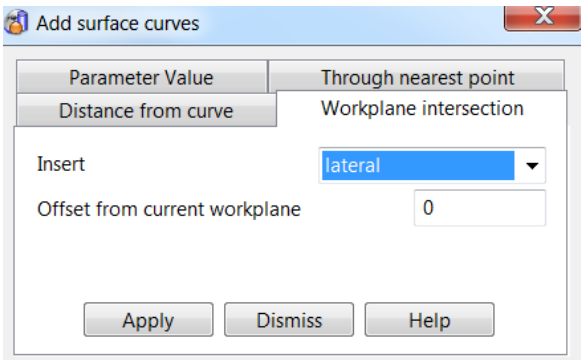
1 Ensure that **Workplane 1** is **active**.

2 As shown on the right hand side image above, **dynamically rotate** it to **align** the **Y Axis** with the **first** of the **construction lines** that run from the centre to the ends of the radius on the modified corner geometry.

3 Set the operational direction to **X** .

4 Select the surface to display the **Surface editing** toolbar.

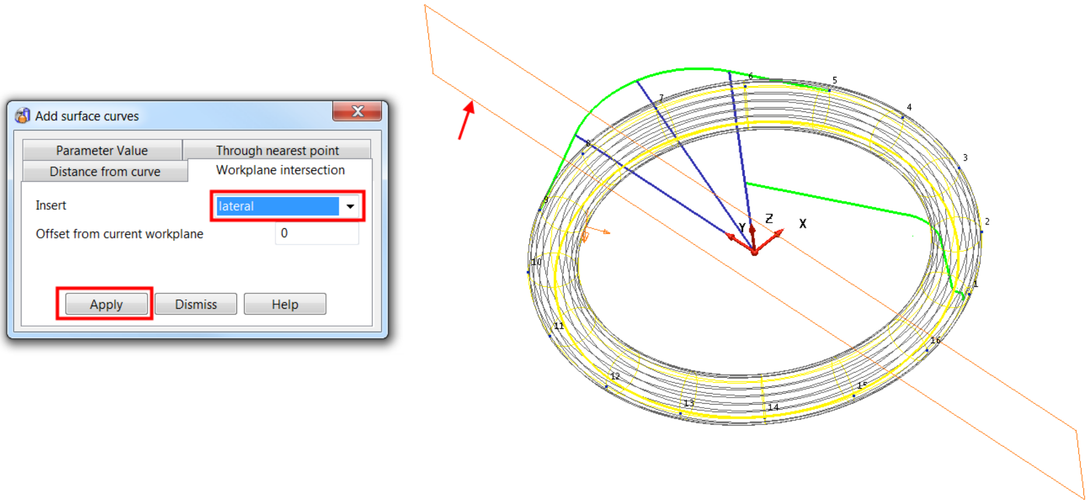
5 Select **Add curve.** 



Four methods are available to add laterals or longitudinal curves.

6 Select the **Workplane intersection** then lateral from the drop down list.

*If a cross-sectional surface curve itself was selected (not surface) prior to opening the form, the **Insert** option would have pre-set to **Lateral**.*



A red outline indicates the position of the new lateral. In this case on the XY plane of the active workplane.

7 Select **Apply** to insert an additional lateral curve.

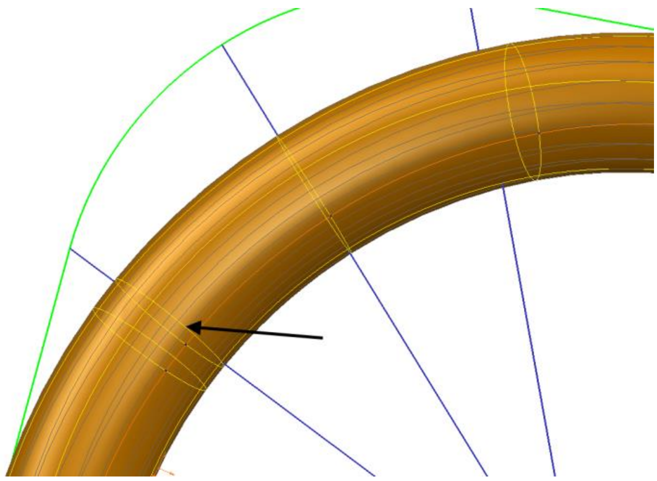
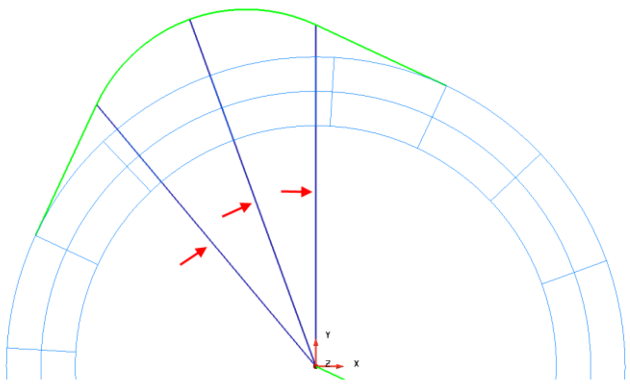


Image shaded here to show new curve generated.

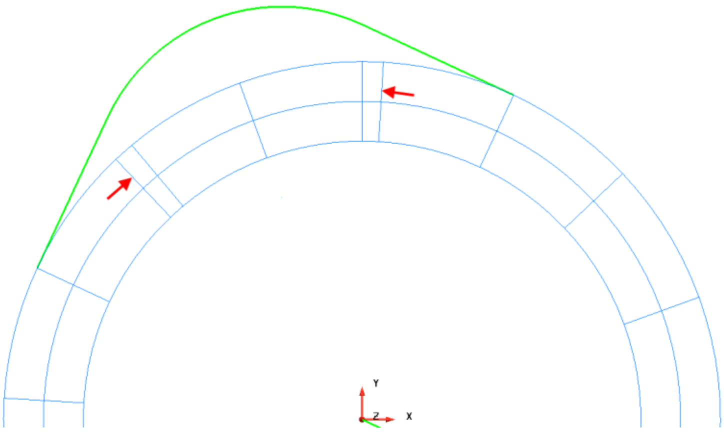
8 **Rotate** the workplane to align the **Y Axis** with the other of the construction lines that run from the centre to the ends of the radius on the modified corner geometry.

9 **Insert** a **second additional lateral**, flush with the workplane **YZ** plane.

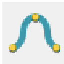
10 **Select** and **delete** the three straight construction lines.

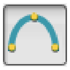


11 Select the two original **laterals** that are **not aligned** to key points on the lip-shaped composite curve.

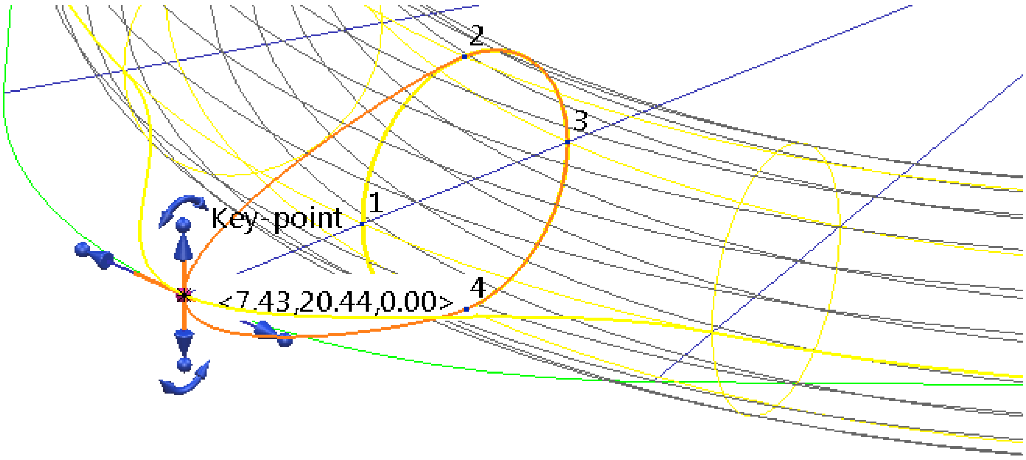


12 Select **Delete** curve  from the **Surface/curve editing** toolbar.

13 Select any curve and ensure **smoothing** if **OFF**  from the **curve editing** toolbar.

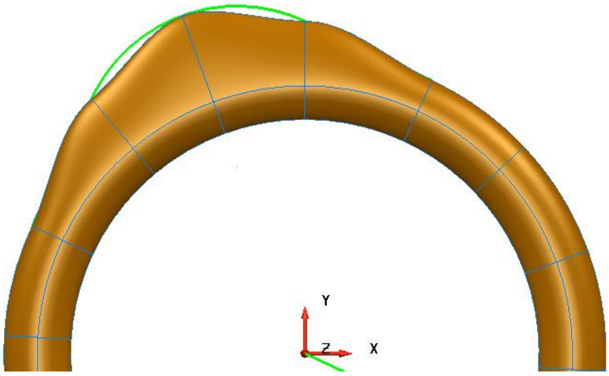
*If the option is **ON***  *then smoothing is applied through all points including Powersurface editing.*

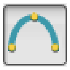
14 **Select** the **middle surface point** on the **outer longitudinal** in the **top-right quadrant** ,and **dynamically drag** it to the corresponding keypoint on the new lip-shaped composite curve (as shown below).

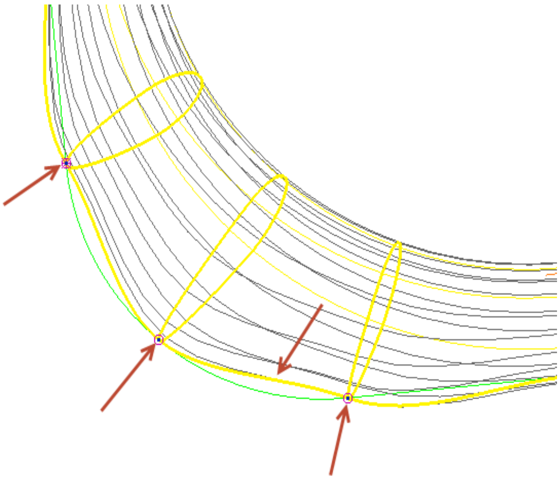


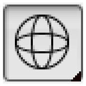
15 **Repeat** the **drag** move on the **adjacent** surface points to create the following.

*Ensure a **keypoint** (**End**) is **selected** on the **compositive curve**.*



The **tangency** and **magnitude** through the **3 points** requires freeing up to achieve the more **natural transition** through the points. If **smoothing** was **ON**  then this would have been achieved immediately. However this example will show how to smooth a surface after creation.

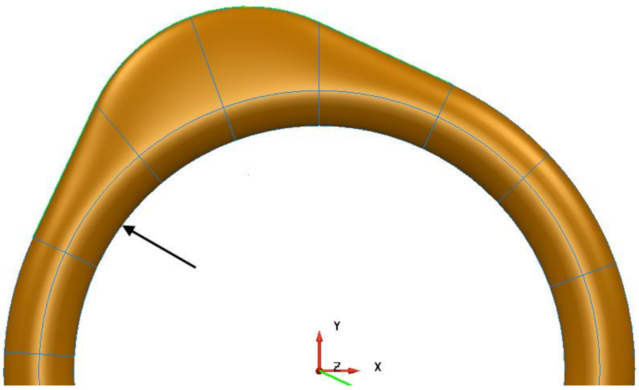


16 Switch to **Wireframe** View .

17 Use **Shift Select** to select the **outer longitudinal line**, followed by the **3 points** through which correct alignment is required.

18 **Right-click** on the longitudinal and select **Free tangents and magnitudes** in the menu. This creates the most **natural geometric flow** through the selected surface points taking into account the adjacent non-selected points.

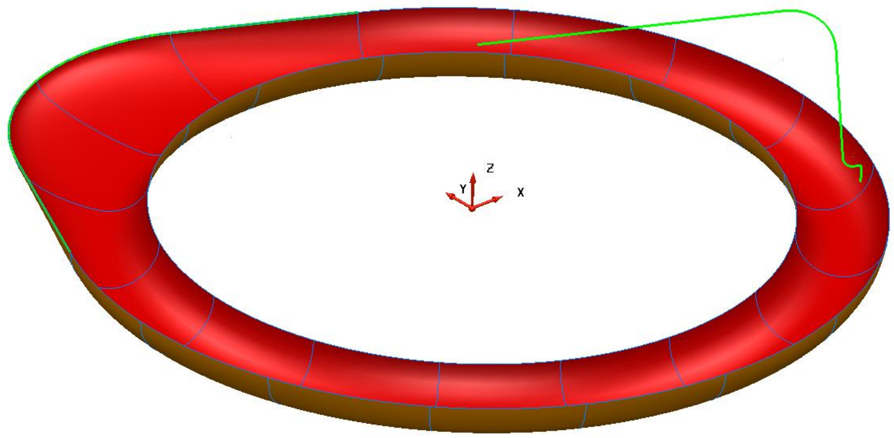
19 **Shade** the **model**.



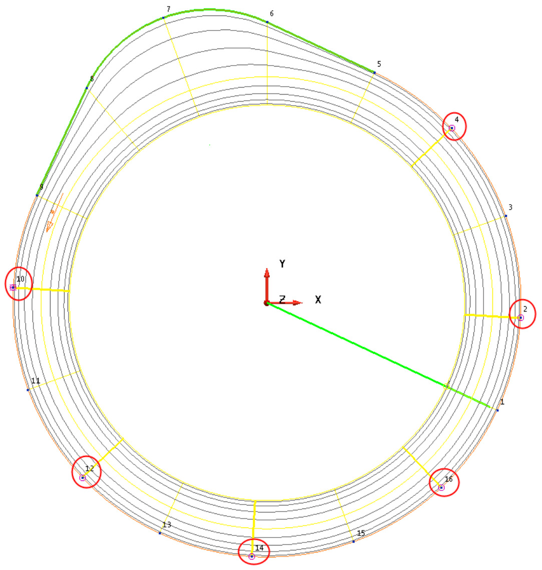
20 Select the **longitudinal** running around the **inside** of the rim (indicated above)

21 Select **Break surface**  to split the surface into two separate upper and lower halves.

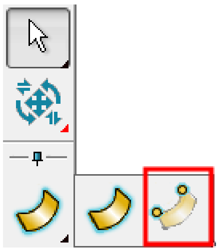
22 Select and **delete** the **upper surface**.



23 Select a **view from top** (down **Z**), then select the alternate six surface points around the outside of the lid as shown below.

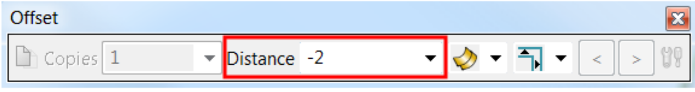


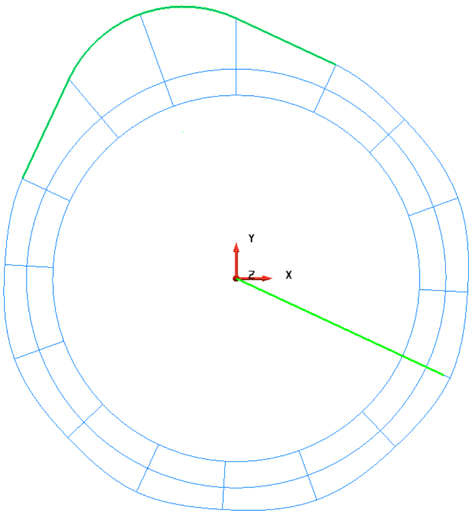
If one or more surface curves or surface points are selected, it is possible to select the **Edit sub-item** option. This lets you apply the **General Edit** options to the selected sub items and not the surface as a whole.



24 Select **Edit sub items**  from the **General Edits** Toolbar.

25 Select **Offset** , enter a value **-2** and press the **Enter** key.

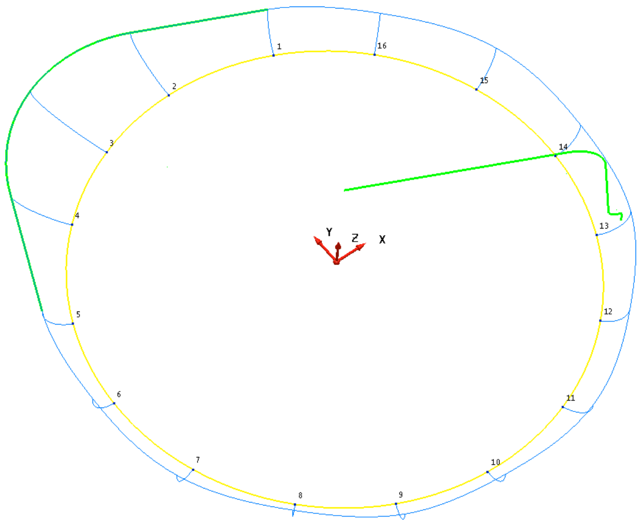




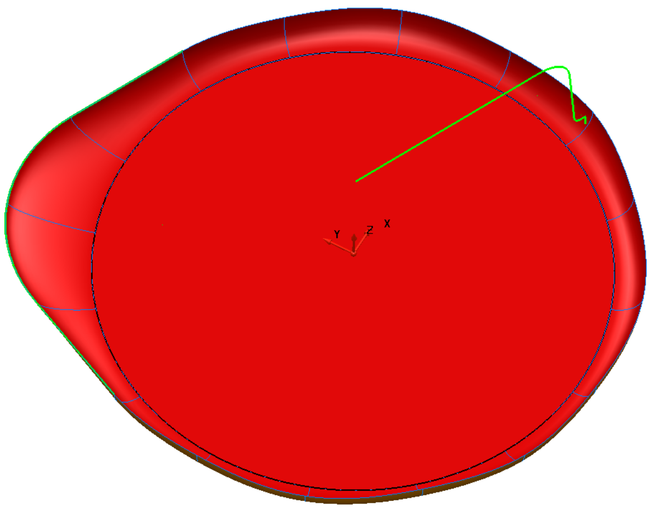
The selected surface points have been offset inwards, normal to the surface.

26 Select and **Delete**  the inner most surface **longitudinal** curve.

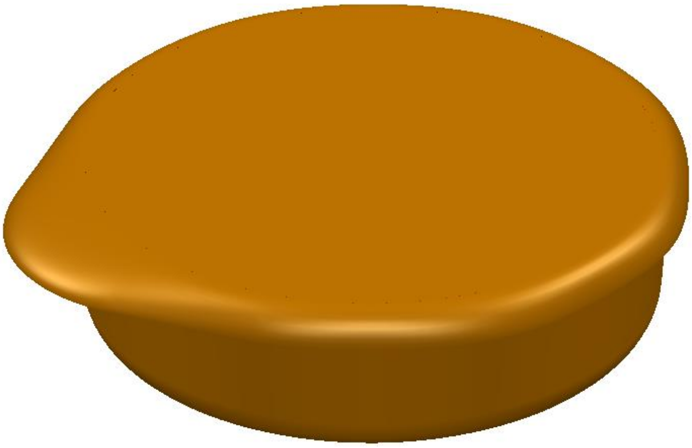
27 Create a composite curve along the **new inner most surface** curve.



28 Generate a **Fill in** surface using the **Smart Surfacer**.



29 Use the remaining composite curve defining the outer profile, and generate a **surface of revolution.** 



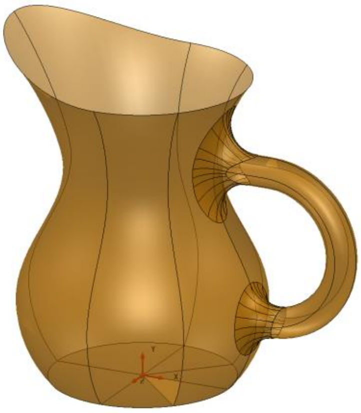
30 **Save** the model as

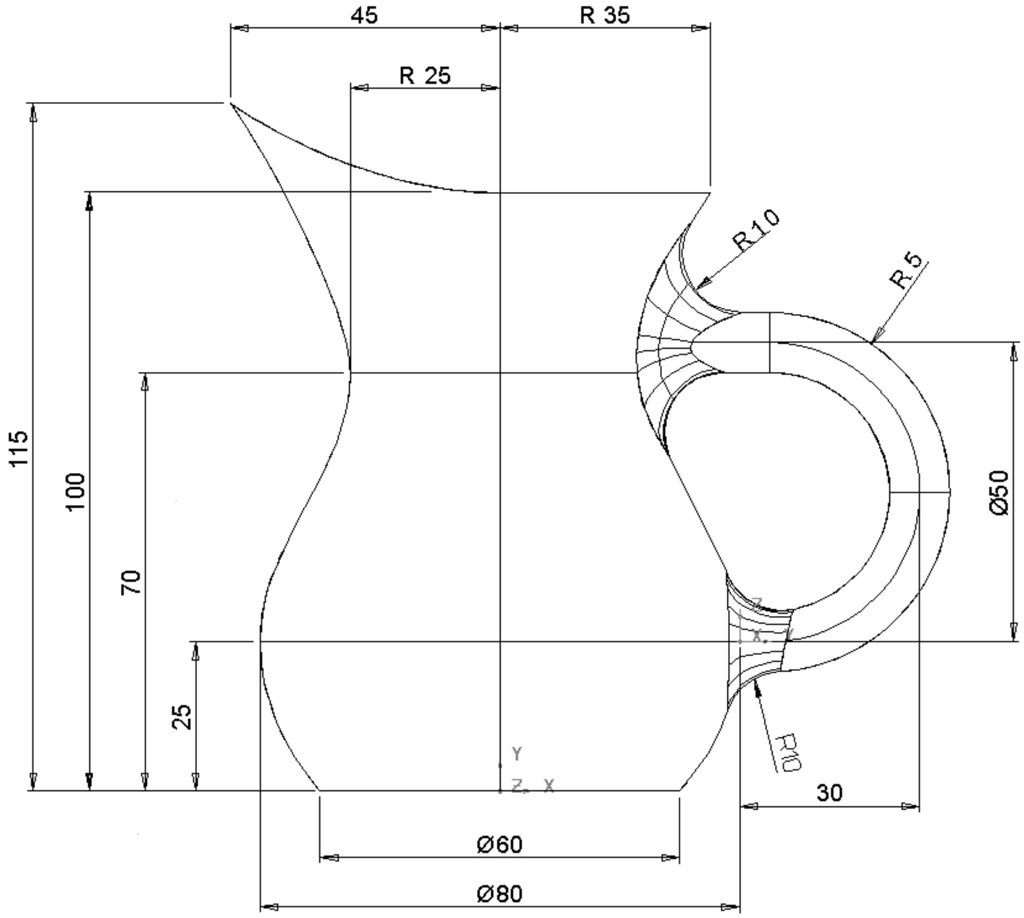
.....\PowerSHAPE-Models\food container.psmodel

31 **Close** the model.

Exercise 8 : Surface Jug

1 Create a surface model of the outside of the jug shown below.





2 Save the model as

.....\PowerSHAPE-Models\Ex8 Surface Jug.psmodel

3 Close the model

Surface Trim Region Editing

Trim Boundaries

A **trim boundary** on a surface defines the trim region of a surface.

Trim boundaries are created automatically when:

- creating **fillet surfaces**.
- creating **draft surfaces**.
- limiting surfaces using **Limit Selection** and **Limit Point.**

A trim boundary consists of **parameter curves (pcurves)** and ultimately **parameter points (ppoints)** joined together to form a closed region of a surface.

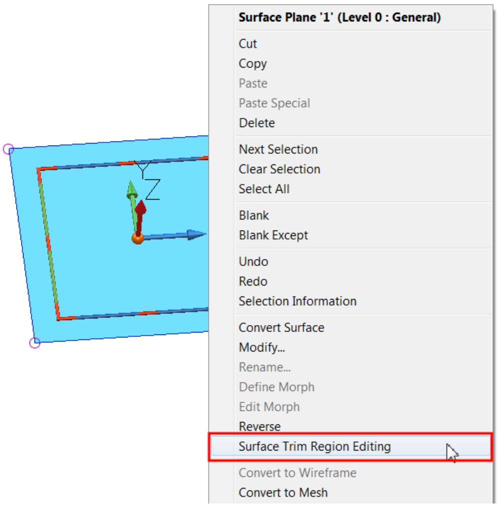
PowerSHAPE provides powerful tools to help you define and edit your trim region accurately.

Parameter Curves (Pcurves) and Points (Ppoints)

Parameter Curves (pcurves) are made up of linear spans that run between **Parameter Points (ppoints)**. As moves between **ppoints** are linear, any curvature is controlled by the proximity of adjacent ppoints within the tolerance setting.

Ppoints are defined as a proportional distance between surface points. A ppoint defined at **2.5 1.5** is half way between points **2** and **3** along the longitudinal (T) direction and halfway between points **1** and **2** along the lateral (U) direction.

Individual **pcurves** and **Trim boundaries** are unique to a surface and can be accessed from the **Surface Trim Region Editing** toolbar, which can be opened by right-clicking on a surface for menu options or from the **Surface Edits** toolbar.



The Previous Surface editing Toolbar is now replaced with two toolbars to edit the associated **boundary** or **Pcurve** on the **selected surface.**

Selecting **Boundary mode**  displays the following trim boundary options.

 Generates diagnostics on boundaries.

 Removes one-point spikes from the selected boundary.

 Removes loops from the selected boundary.

 Removes coincident points to simplify the selected boundary.

 Closes a boundary.

 Recreates boundaries. Deletes any selected boundary and enters boundary creation mode.

 Automatically creates boundaries from the pcurves on the surface. Any existing boundaries are deleted.

 Swaps the trim region defined by a set of trim boundaries (for example, it converts a "hole" into an "island").

 Selects all the boundaries on the selected surface.

 Displays/removes the boundary selector.

 Deletes the selected boundary (without deleting its pcurves).

Selecting **Pcurve mode**  displays the Pcurve edit options.

 Generates diagnostics on pcurves.

 Removes one-point spikes from the selected pcurve.

 Removes loops from the selected pcurve.

 Removes surplus points to simplify the selected pcurve.

 Labels the ppoints on pcurves.

 Turn point labels off.

 Makes pcurves from wireframe.

 Extends the selected pcurve.

 Cuts the selected pcurve at the selected point into two pcurves.

 Opens the pcurve.

 Closes the pcurve.

 Selects all pcurves on the selected surface.

 Selects all unused pcurves on the selected surface.

 Displays/removes the pcurve selector.

 Deletes the pcurve.

 Displays/removes the ppoint selector.

 Edits the parametric value at the selected ppoint.

 Inserts ppoints into a pcurve.

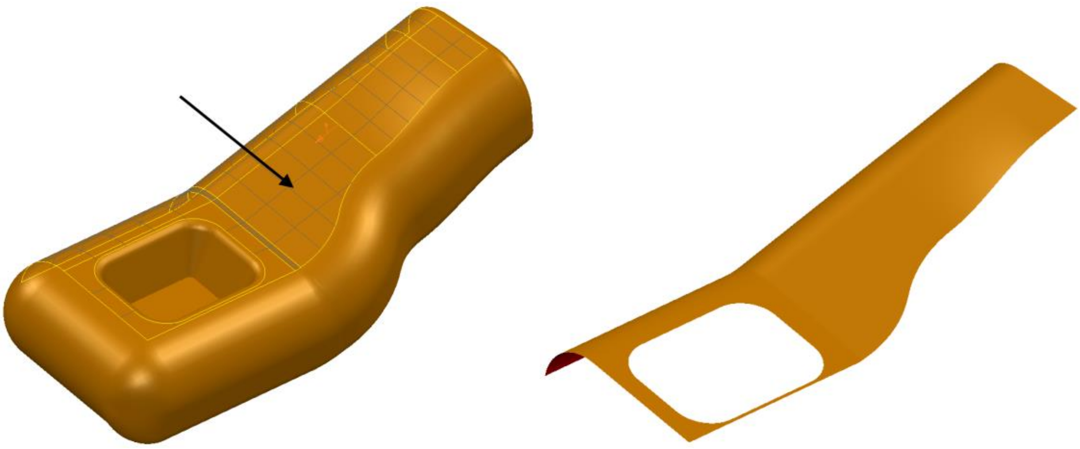
 Deletes the selected ppoint on a pcurve.

The following examples will help demonstrate the above.

Switch housing

1 **Open** the model

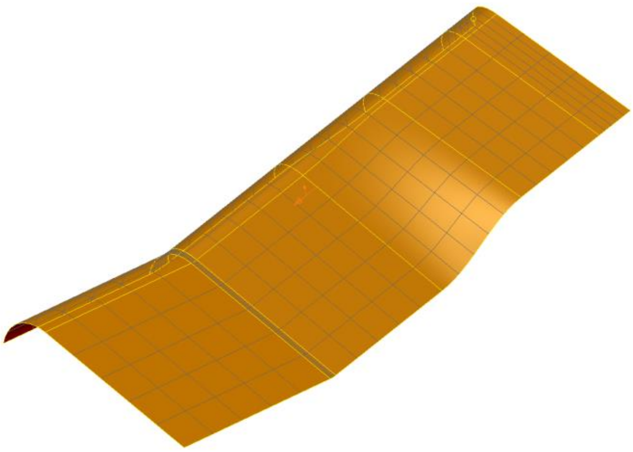
.....\PowerSHAPE_Data**switch_housing.psmodel**



2 Select the upper surface and then **Blank Except**.

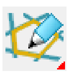
3 Select the surface to display the **Surface Edits** toolbar.

By default, trimming **ON**  displays the surfaces in its trimmed state. To view a selected surface untrimmed, toggle this icon to Trimming **OFF**. 

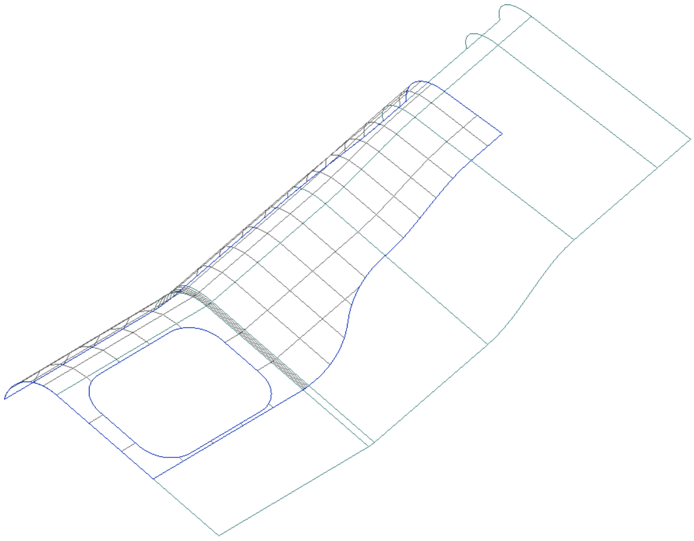


4 Ensure Trimming is **ON**. 

5 Right-click on the surface and select **Surface Trim Region Editing**. The **Trim Region Editing** toolbar is now displayed.

6 Select **Boundary Edit Mode**. 

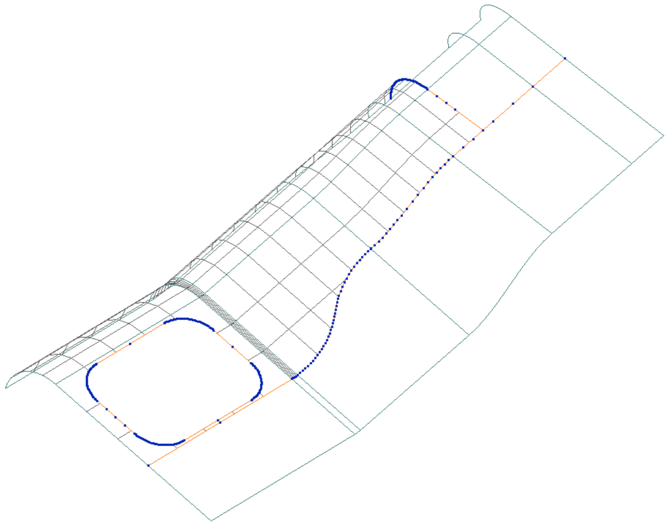
The whole surface is displayed with the trimmed area identified by a mesh of **grey** detail lines. All boundaries are displayed, marking the edge of the trimmed, surface area. The pcurves are not displayed in this mode.



 *If a boundary is deleted*  *from a surface, the associated pcurves remain.*

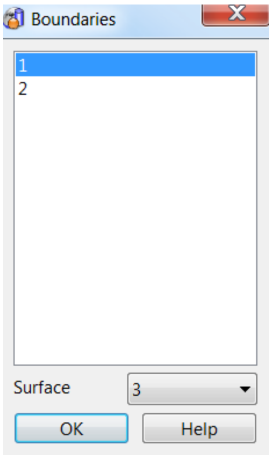
7 Select **Pcurve Edit Mode.** 

The pcurves are displayed along with trimmed areas. The boundaries are not displayed in this mode.



If a pcurve is deleted  from a surface, any associated boundary is also deleted.

8 Toggle back to **Boundary Edit Mode,**  then select the **Boundary Selector.** 



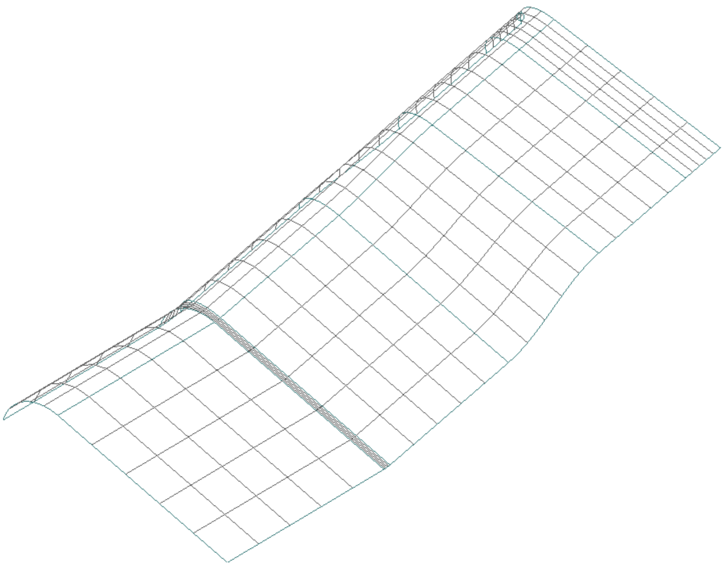
When a **boundary** is highlighted in the form, it is displayed in **orange** on the surface.

Several boundaries can be selected by holding down the **Ctrl** key while left-clicking each boundary. Hold down the **Up/Down** keys to toggle the selection of a boundary. Boundaries can also be selected from within the graphics area.

9 Select **both boundaries** by selecting them with the **Ctrl** key depressed.

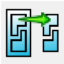
10 Select **OK** on the Boundaries form.

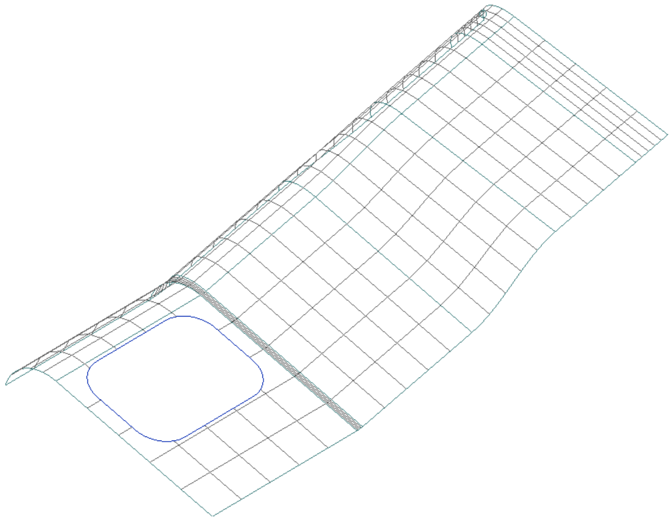
11 Select **Explode**  to delete both boundaries.



With the boundaries exploded, the trimming is removed to show the full original surface area. The pcurves remain intact, but are no longer in use as part of a boundary definition.

*The pcurves still exist and can be displayed by toggling into **Pcurve Edits**.*

12 Select **Autocreate**  regenerate the boundaries from the existing pcurves.



One of the two boundaries has been **regenerated**.

The second boundary did not appear due to the existence of a branch point along the network of pcurves that makes it impossible for PowerSHAPE to determine which route to take (similar to composite curve creation).

Define the route

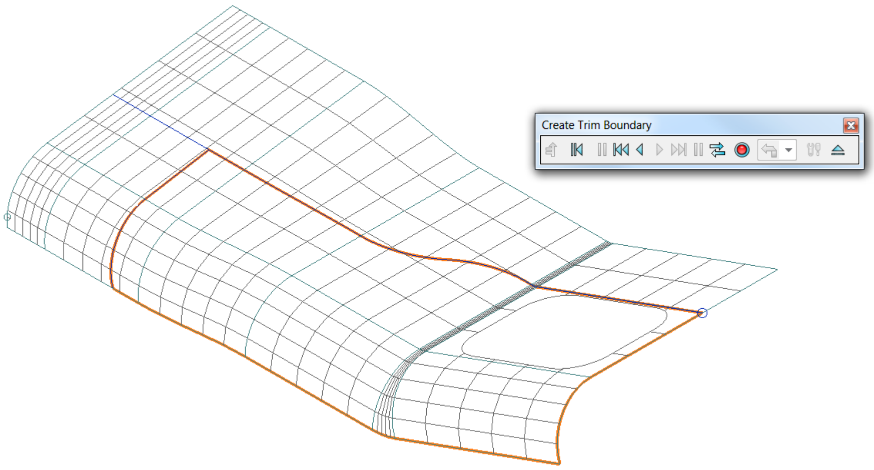
The easiest way to create the required trim option is to **manually** define the route around the **pcurve network** as demonstrated in the following section.

13 Select **Recreate**  to open the toolbar and display all unused pcurves on the surface.

The **Create Trim Boundary** toolbar is displayed. Similar to Composite curve creator, this is used to trace along the pcurves to define a boundary.

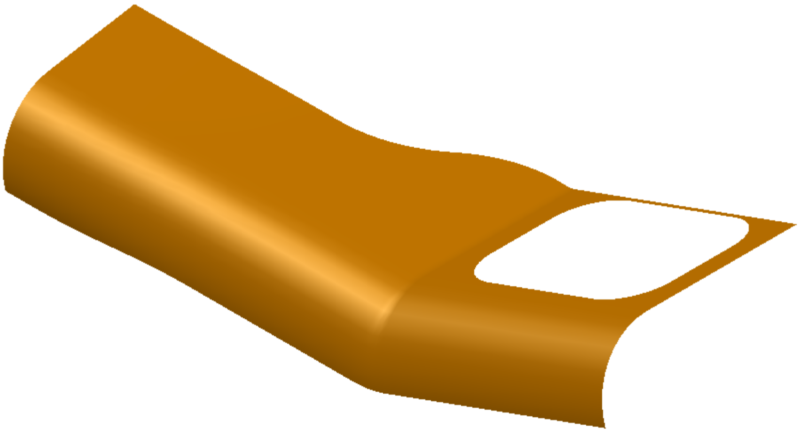
All of the pcurves are displayed on the surface and are available for selection. To form a boundary, the **pcurve** must form a **closed route**. The edges of a surface can also be used as part of the route of a boundary.

14 Click on a pcurve along the require route and sequentially, click on the direction arrows to produce the required closed boundary.



15 Select **Save** and then select **Eject.**

16 Deselect the surface to see the trimmed surface more clearly.



Removing excess pcurve data

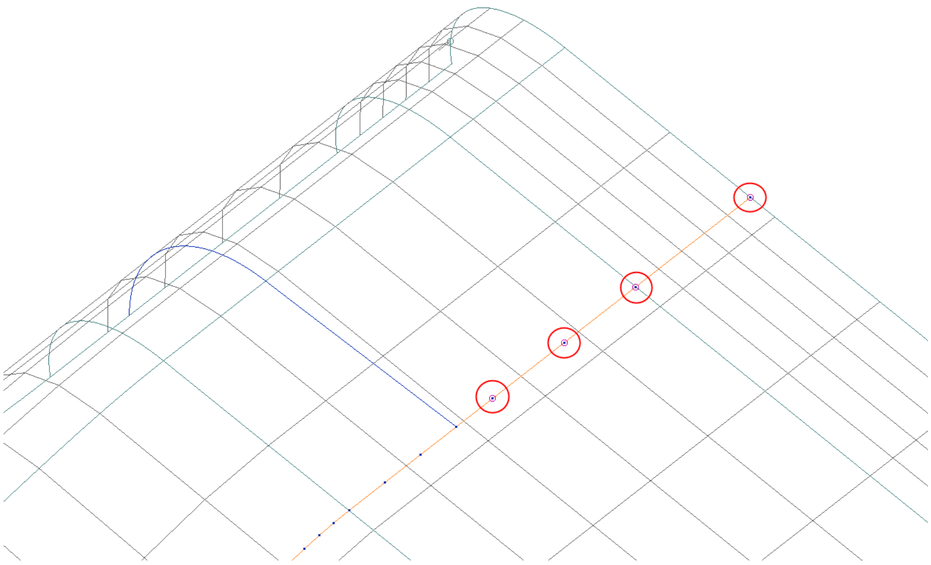
The next stage is a repeat of the above, but this time, excess pcurve data is removed, allowing **Autocreate** to make the trim boundaries automatically.

1 Select the surface.

2 Select all boundaries then Explode.

3 Select **Pcurve Edit Mode.**

4 Select the **long pcurve** along the part not required for boundary definition.

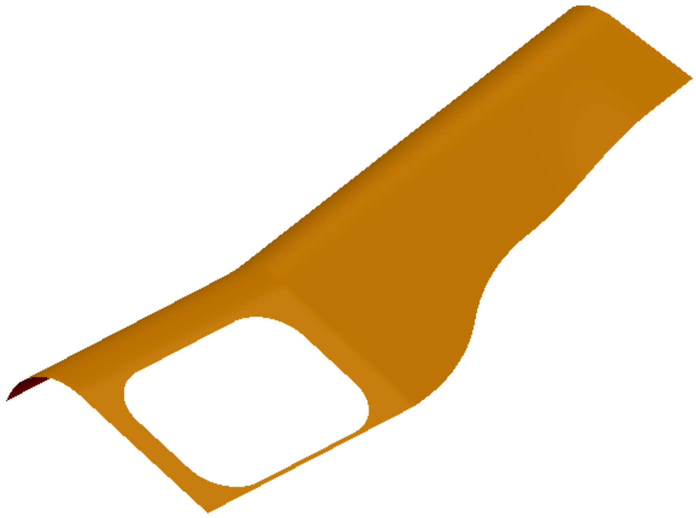


When a pcurve is selected, the ppoints are displayed. These can be labelled, selected, repositioned or and deleted, as required.

5 Shift-Select the four extra ppoints circled above.

6 Click **Delete Point.**

7 Switch back to **Boundary Edit Mode** then **Autocreate.**



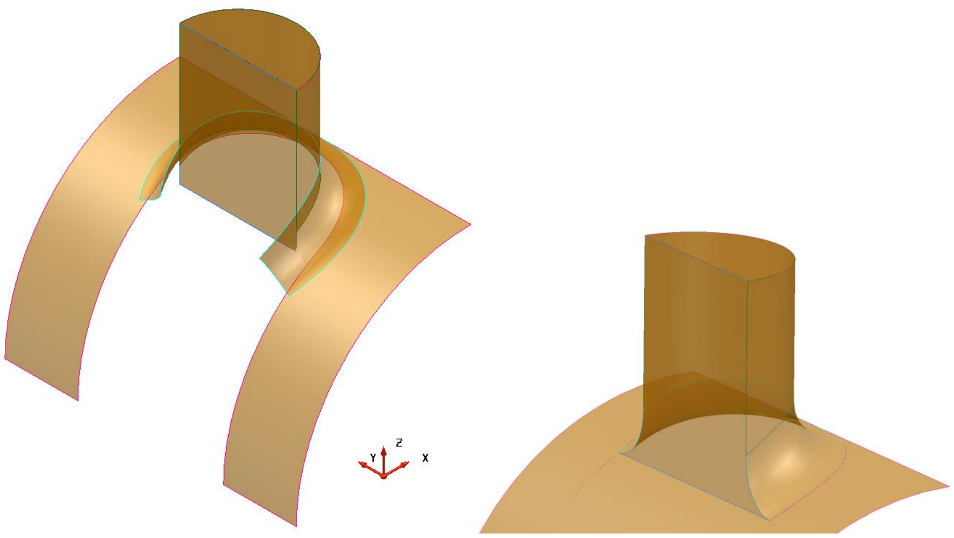
Removing the unused ppoints enables future automatic trimming operations to immediately produce the desired results and not stray along a path defined by redundant pcurve data.

8 **Close** the model without saving.

Repair Incorrectly Trimmed Surfaces

1 **Open** the model

*.....\PowerSHAPE_Data**locationlug**.psmodel*

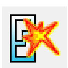


The model includes several **incorrectly trimmed surfaces** that require a combination of **pcurve** and **boundary editing** to correct the faults.

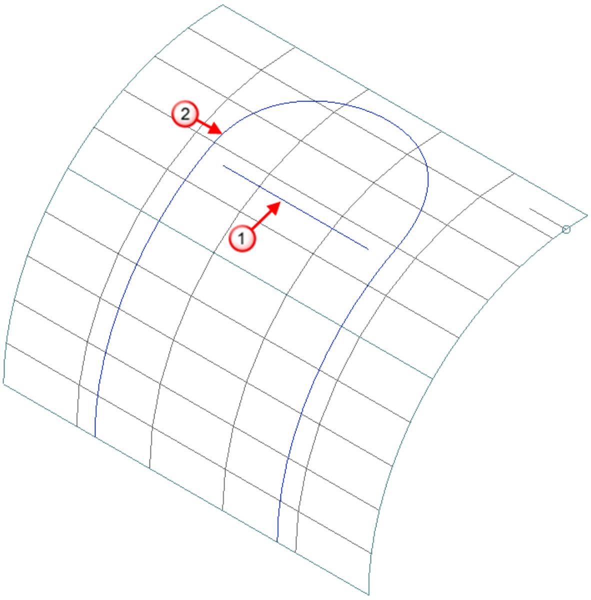
The more complex the network of pcurves on a surface, the less likely it is to obtain a trimmed surface automatically. The following worked example shows the stages of pcurve and boundary editing required to produce the correctly trimmed model shown above right.

2 Select the large curved base surface and **Blank Except** (**Ctrl+K**).

3 Enter **Trim Region Editing.**

4 **Explode**  the only existing boundary.

5 Switch to **Pcurve Edit Mode**. 



The surface contains **two pcurves**: one correctly defines the intersection with the vertical face of the location lug 1 ; the other, U-shaped pcurve 2 defines the path of an old fillet prior to an amendment, and needs to be deleted.

6 Select the U-shaped pcurve 2 and **Delete a pcurve**. 

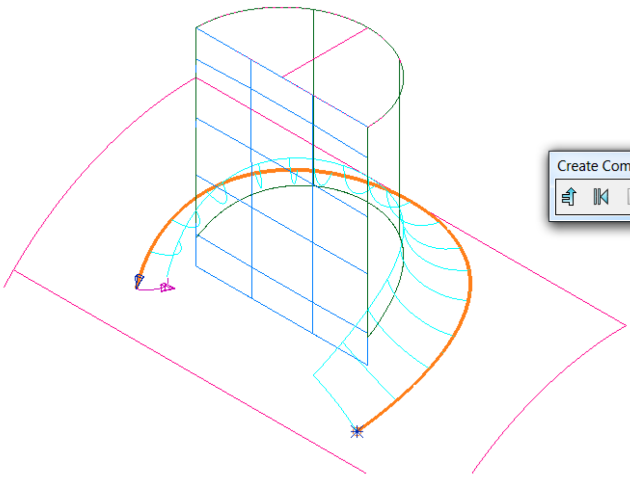
7 Select **Unblank** (**Ctrl+L**).

Creating a pcurve from a composite curve 

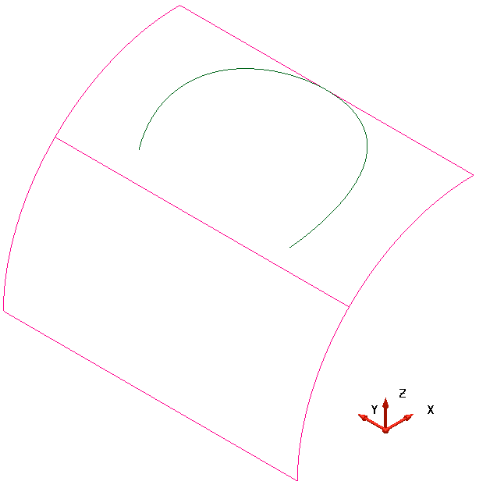
Pcurves can also **be created** from standard **wireframe** and **composite curves.**

In this case, a **composite curve** will be created from the lower edge of the fillet. This new **composite curve** is then turned into a **pcurve** and projected onto the original curved surface to help trace a new boundary.

8 Create a composite curve on the lower edge of the fillet surface as shown.

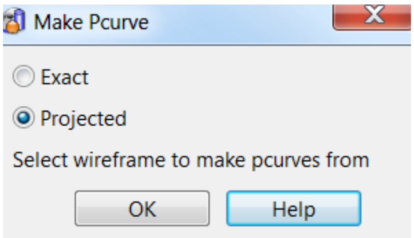


9 **Blank all** except the new composite curve and the curved surface.

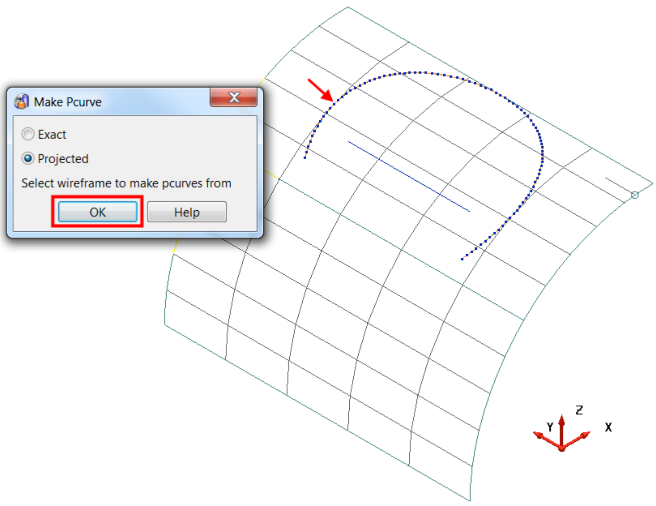


10 Select the curved surface and switch to **Pcurve Edit Mode.** 

11 Select **Make pcurves from projecting wireframe.** 



12 Select **Projected** then the composite curve. Select **OK.**

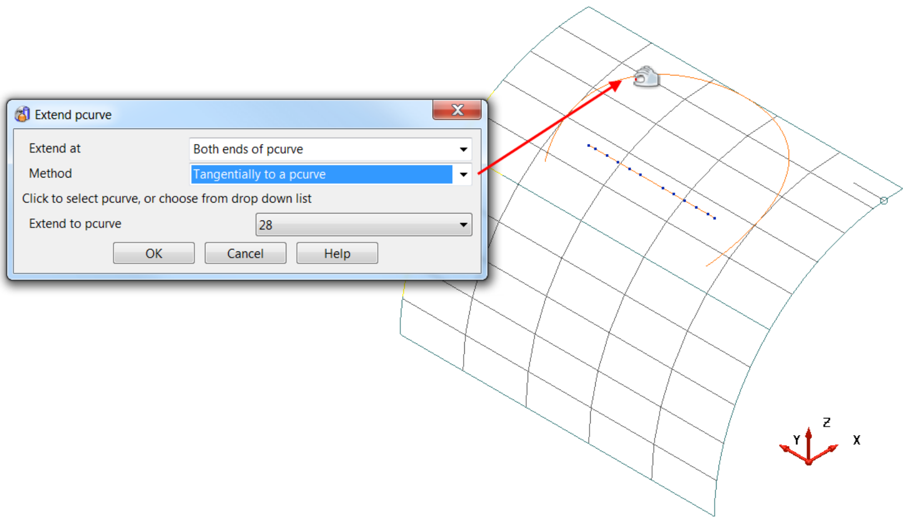


13 **Deselect** the **surface** and **delete** the **composite curve.**

14 Select the **lower curved surface** to display the pcurves on the surface.

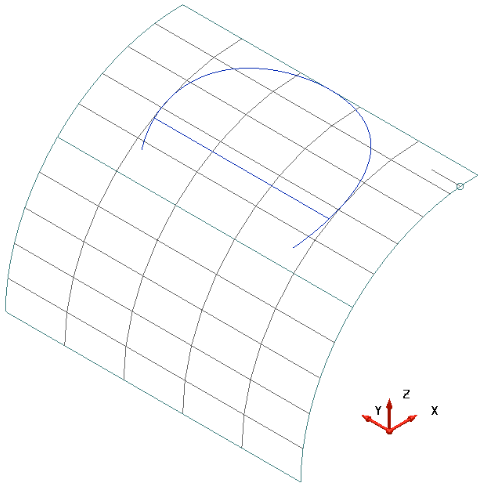
The linear pcurve needs to be extended to create a closed boundary. This will ultimately be traced as a D section.

15 Select the straight pcurve then **Extend pcurve.** 



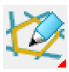
16 In the **Extend at** list, select **Both ends.** In the **Method** list, select **Tangentially to a pcurve.**

17 Select the other curved pcurve then **OK.**

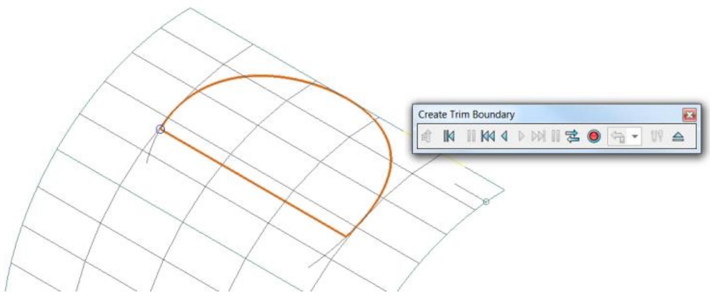


 *The straight pcurve has extended to effectively create a closed boundary.*

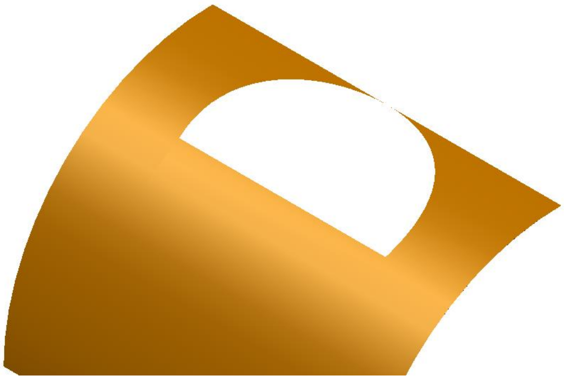
Manually create a new boundary

1 Switch back to **Boundary Edit Mode**  then **Recreate.** 

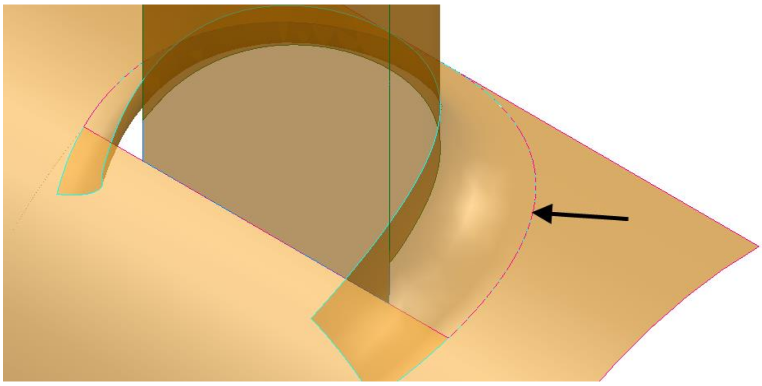
2 Trace the pcurves to define the new boundary as shown below.



If the trimmed surface appears within the D-shape, select ***Reverse the Boundary*** *to create the result shown.*



3 Select **Unblank** (**Ctrl+L**).

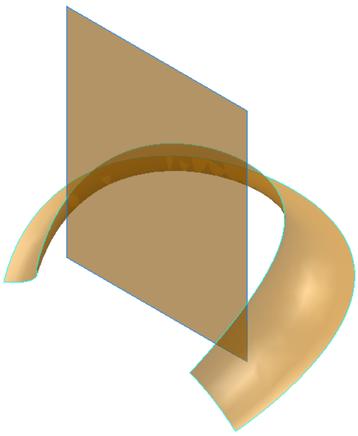


The new boundary has removed the overlap between the fillet and curved surface which previously existed.

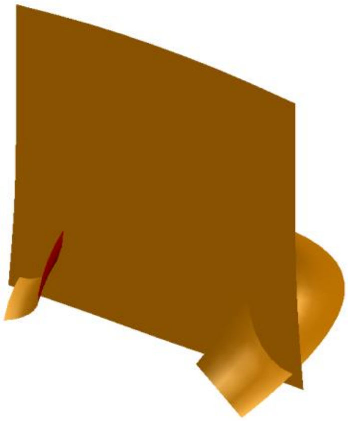
4 **Blank** all surfaces except the front vertical and fillet surfaces.

Trimming the planar surface

The planar surface needs to be correctly trimmed with the fillet.

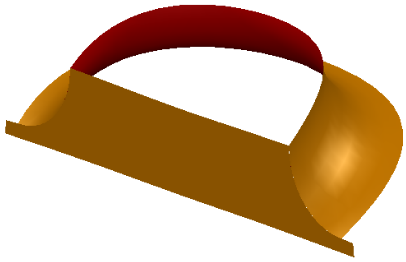


1 **Explode** all boundaries on the **vertical surface.**



The vertical surface is untrimmed to its original size and will be limited to the fillet surface.

2 Use **limit select**  to trim the surfaces and leave the solution below.

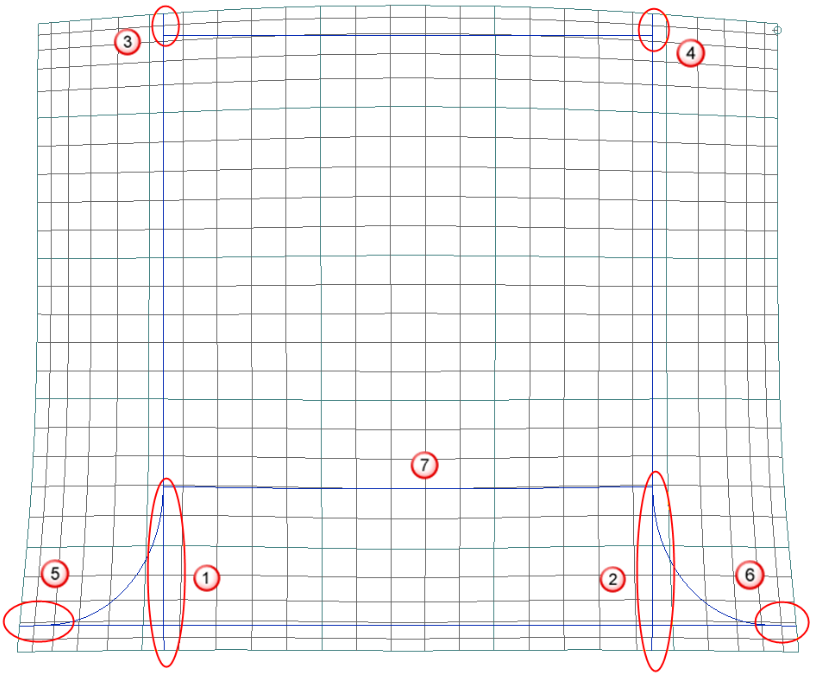


The vertical surface to its full size was not available as a solution due to the complexity of the pcurve network in the surface. However a new boundary will be traced to include this.

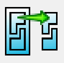
3 Select and **Blank Except** (**Ctrl+K**) the vertical surface.

4 **Explode**  all boundaries on the vertical surface

5 Switch to **Pcurve Edit Mode**  to display all existing pcurves.



If required, a new boundary can simply be **recreated**  to include the vertical surface.

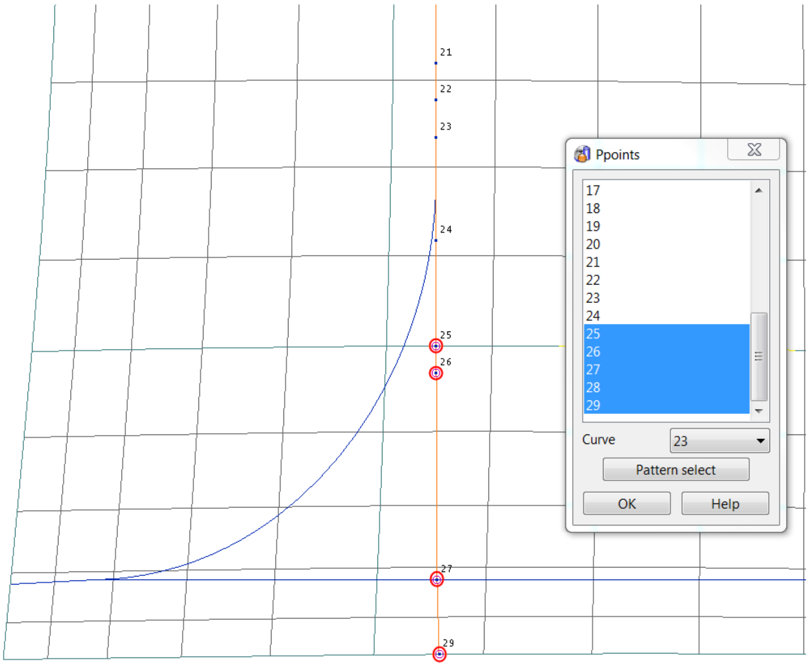
However in this case the pcurves circled in red will be edited to leave a single closed path. With this, **Autocreate**  can then be used effectively.

6 Select and **delete**  the horizontal pcurve. 7

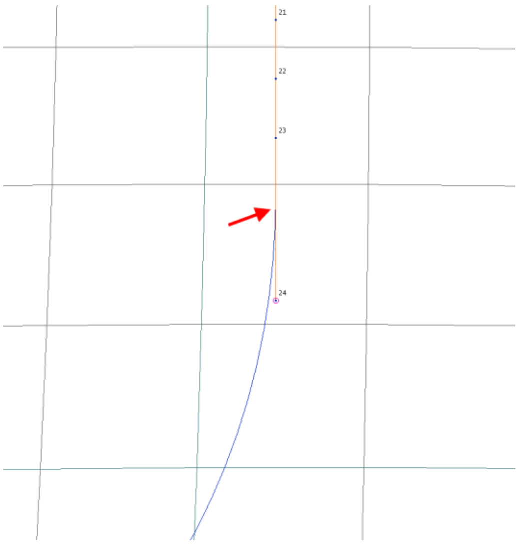
7 Zoom into region 1 select the vertical pcurve.

8 Select **Turn point labels on/off**  to display the ppoint numbers.

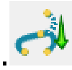
9 Shift select or use ppoint selector  to select points 25 to 29 inclusive.

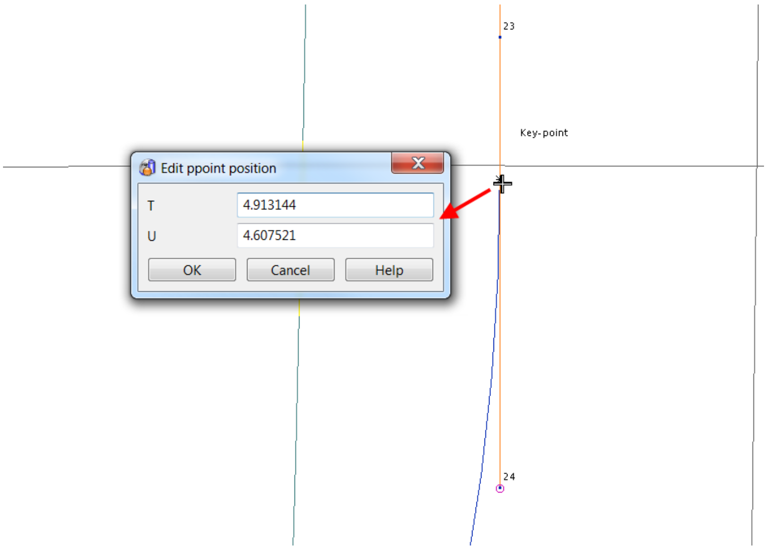


10 **Delete** the ppoints 



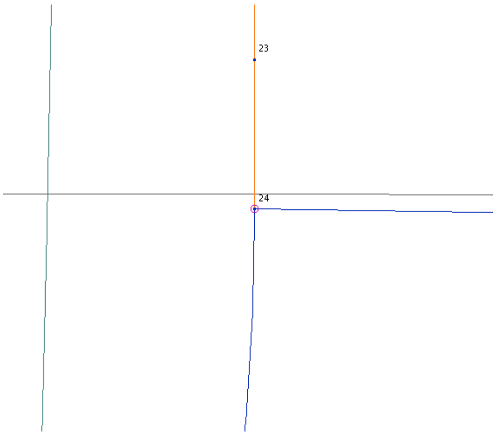
Point 24 was not deleted. This will be moved up to the start of the fillet arc.

11 Select ppoint 24 and select **Edit parametric value at ppoint.** 



12 Click the point situated before ppoint 24 (shown as keypoint above) and note the updating of the values in the **Edit ppoint position** form (zoom in as close as possible).

13 Select **OK.**

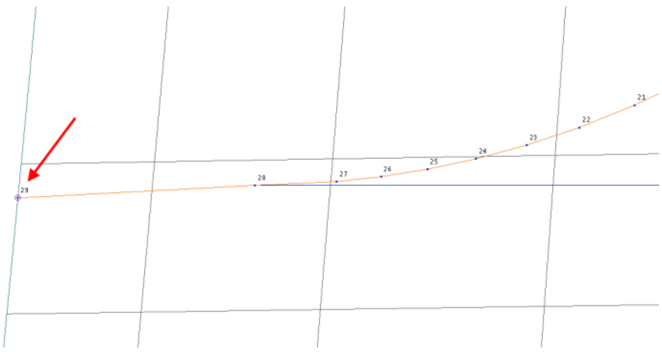


The position of ppoint 24 updates to the new parametric values. The ppoint is now repositioned to join the start of the fillet arc.

14 Repeat the above for the opposite side. 2

15 Zoom into the bottom left corner region. 5

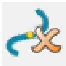
16 **Delete**  just point 29 on the fillet pcurve.

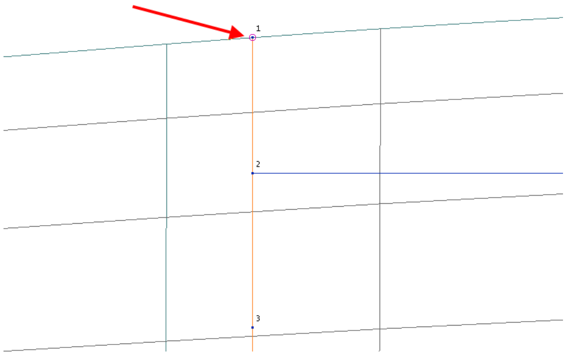


The pcurve is trimmed back to point 28 which is also the intersection point with the adjoining pcurve.

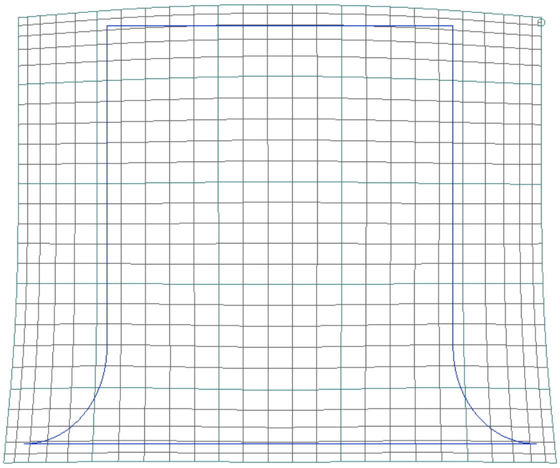
17 Repeat the above for the opposite side. 6

18 Zoom into the top left corner region. 3

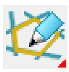
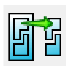
19 **Delete**  just point 1 on the vertical pcurve.

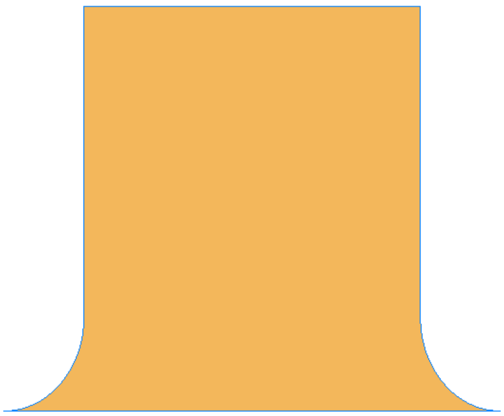


20 Repeat the above for the opposite side. 4



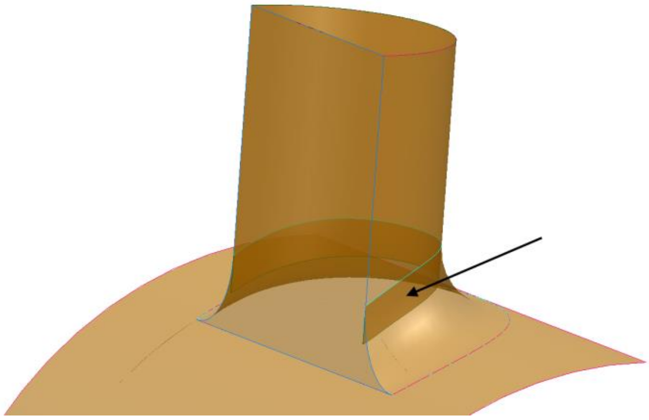
A single pcurve path remains.

21 Switch back to **Boundary Edit Mode**  then **Autocreate**. 



A single clean surface is generated.

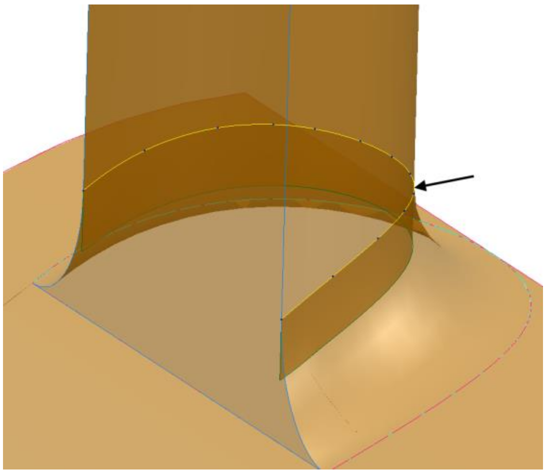
22 Select **Unblank** (**Ctrl+L**).



A final problem remains indicated by the arrow above. The vertical D section surface requires trimming back to the top edge of the fillet surface. An overlap can be clearly seen in transparent view.

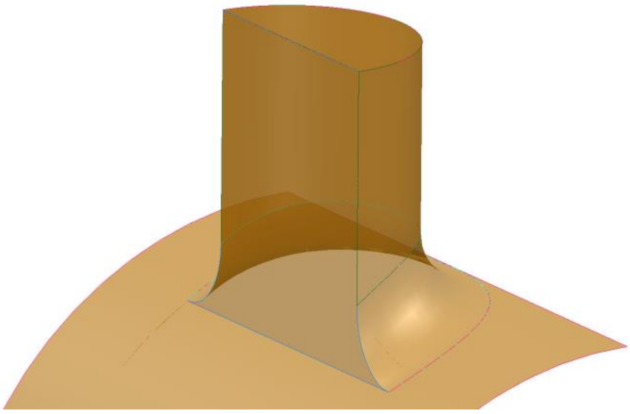
Trimming the vertical D-section

It is possible in this instance that the trimming can be simply applied using **Limit selection.** 



23 Trace a composite curve along the top edge of the fillet surface.

24 Using this curve as the cutting object, **limit**  the surface to leave the required result.

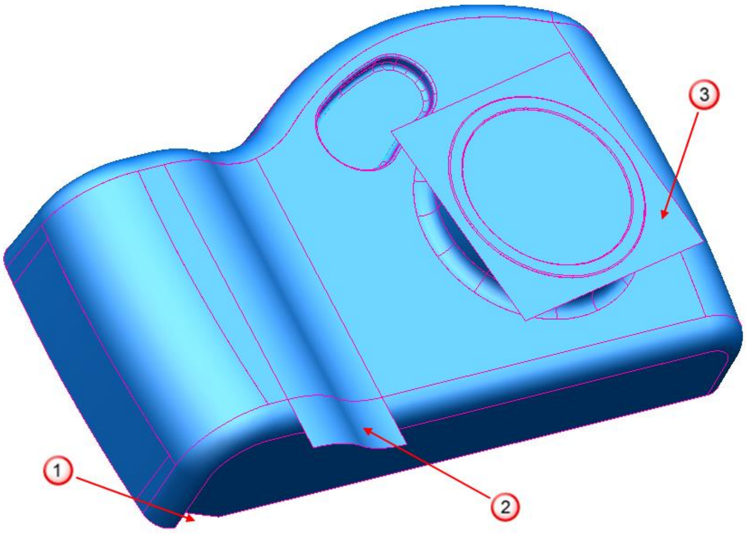


25 Save the model as.....\PowerSHAPE-Models\Location Lug repaired.psmodel

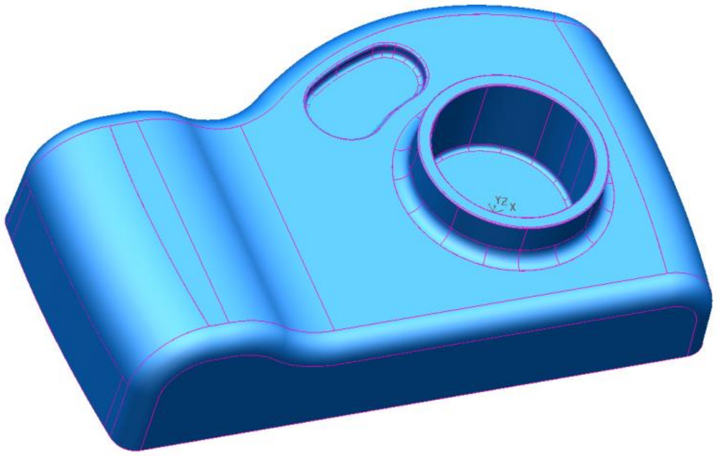
Exercise 9: Region trim camera

1 **Open** the model

.....\PowerSHAPE_Data**camera repair.psmodel**



2 Use Trim Region editing ONLY * to repair the three areas highlighted on the surface model.



1 **Save** the model as

.....\PowerSHAPE-Models**Ex9 region trim camera.psmode**l.

2 **Close** the **model**.

**Try not* to use Limit Selection  or Smart Surfacer!! *

10. Basic Resurfacing

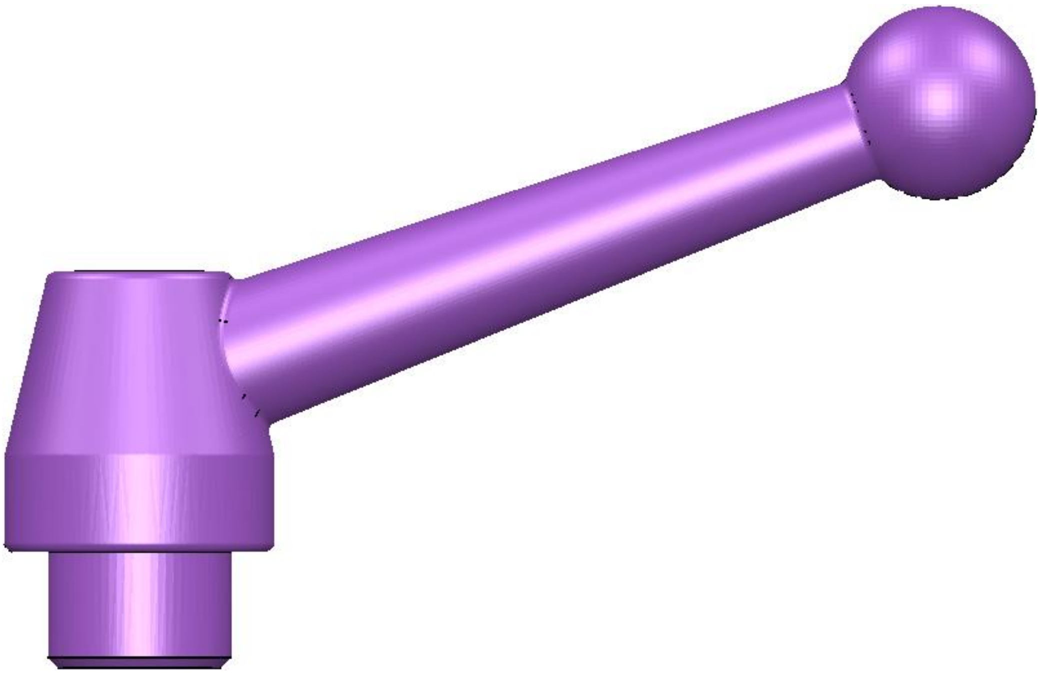
Introduction – Method 1

The **following exercise** will **resurface** the **mesh** of a **handle** using the **alternative techniques** using **curve** and **surface creation** rather than the newer, simpler **Automatic** and **Manual mesh segment tools**, which may not always produce desired results on **vague** or **coarse mesh.** Knowledge of these methods is still of great importance to enable you to successfully **reverse engineer** a mesh. This exercise will cover:

- **Creating curves from a mesh.**
- **Creating surfaces from curves** – from **Revolution**, from **Separate** and **Drive Curve** using the **Smart Surfacer**.
- Complete the model by converting the **surfaces** into **solids** and **create solid fillets**.

Resurfacing the Handle Mesh

1. Select **File>Import…**
2. Browse to: **C:\Training Data\PowerSHAPE Pro Data** and **import** the file **handle.stl**.
3. Select a **View from Top** (**Ctrl+5**).



We will now create a **workplane** at the **centre** of the **hole** so we can correctly create a **curve** and use the **workplane** to **revolve** around the axis.

4 Select the **mesh** by clicking on it in the **graphics area** which will highlight the edges **yellow** and activate the **Mesh Edit toolbar**.

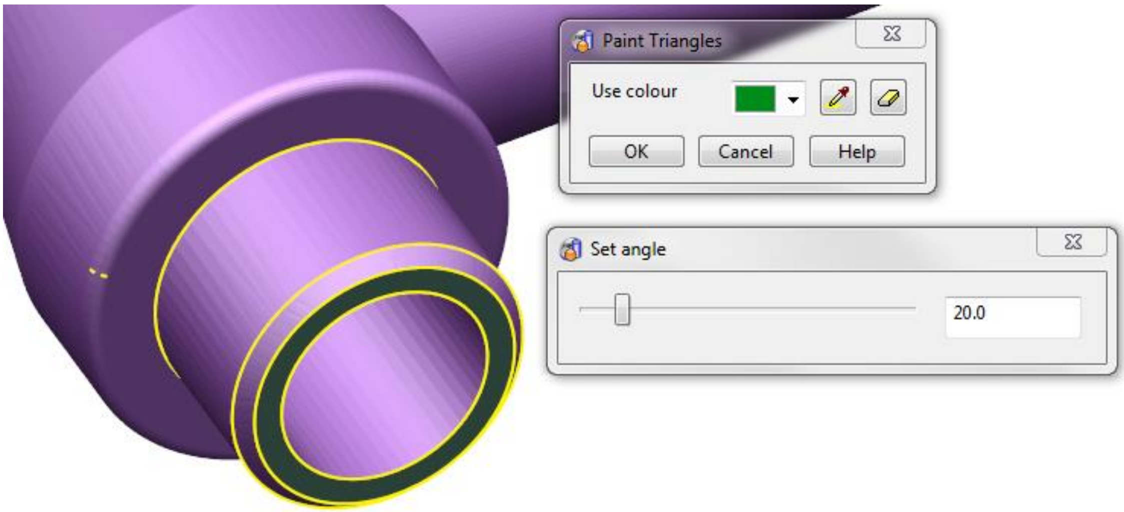
5 Select the **Paint Triangles**  option from the **Mesh Edit toolbar** and from the **drop down menu** in the dialog select a **green colour**.

*This option will **apply** a **chosen colour** to any **selected triangles** on the mesh which can be used to **highlight specific areas** of the mesh and also can be used as an **alternative** to **divide the mesh** into **multiple mesh**.*

6 Select the **Select triangle area to discontinuity area**  option from the **Mesh Edit toolbar**.

7 Select the **bottom** of the **mesh** with the **discontinuity angle** set as **20 degrees as default** as seen below which will **apply** the **green** to the **bottom face** of the **mesh**.

*The **discontinuity angle** in this instance will act as the **threshold value** when selecting **adjoining triangles**. In this case we can see that the bottom face is flat therefore a very small angle here would suffice, while a much higher angle will begin to select the **adjoining chamfered edge** if the **discontinuity angle** is set **higher** than the **chamfer angle**.*



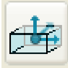
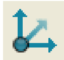
8 Select **OK** on the **Paint Triangles** dialog.

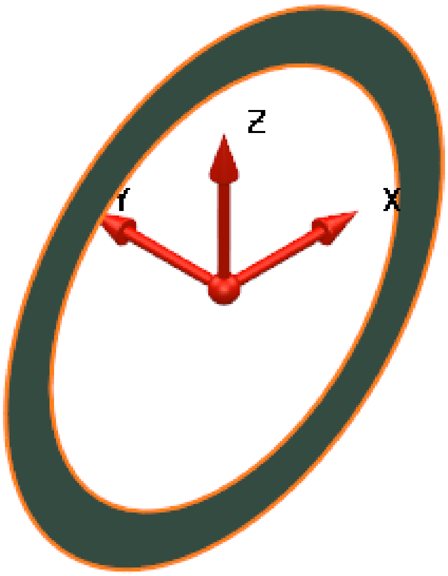
9 Select the **Divide mesh into multiple meshes by colour**  option.

*The mesh has now been split into **two separate meshes** using the **different colours** applied to **divide**, this can be used as an **alternative** way than splitting by selection, and is particularly **efficient** when **splitting** into **more than two mesh**.*

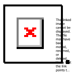
The next step will be creating a **c**at the **centre** of the **green mesh**. It was important to **divide** the mesh into two, to enable us to only select the new **separate mesh**.

10 **Select** and **Blank** (**Ctrl+J**) the main handle mesh leaving only the green bottom face mesh.

11 With the visible **mesh** selected in the **graphics area**. Create a **single workplane at the centre of selection**  from the **Workplane**  menu.

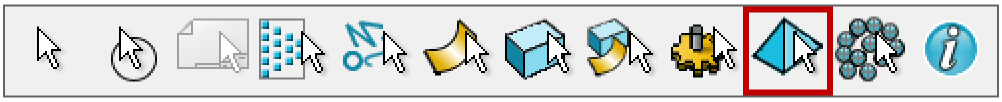


12 **Unblank** (**Ctrl+L**) the other mesh.

13 Select **View From Top** .

Next, we will **combine** the two mesh into one complete mesh.

14 From the **selection flyout** menu choose **Quick select ALL mesh**.



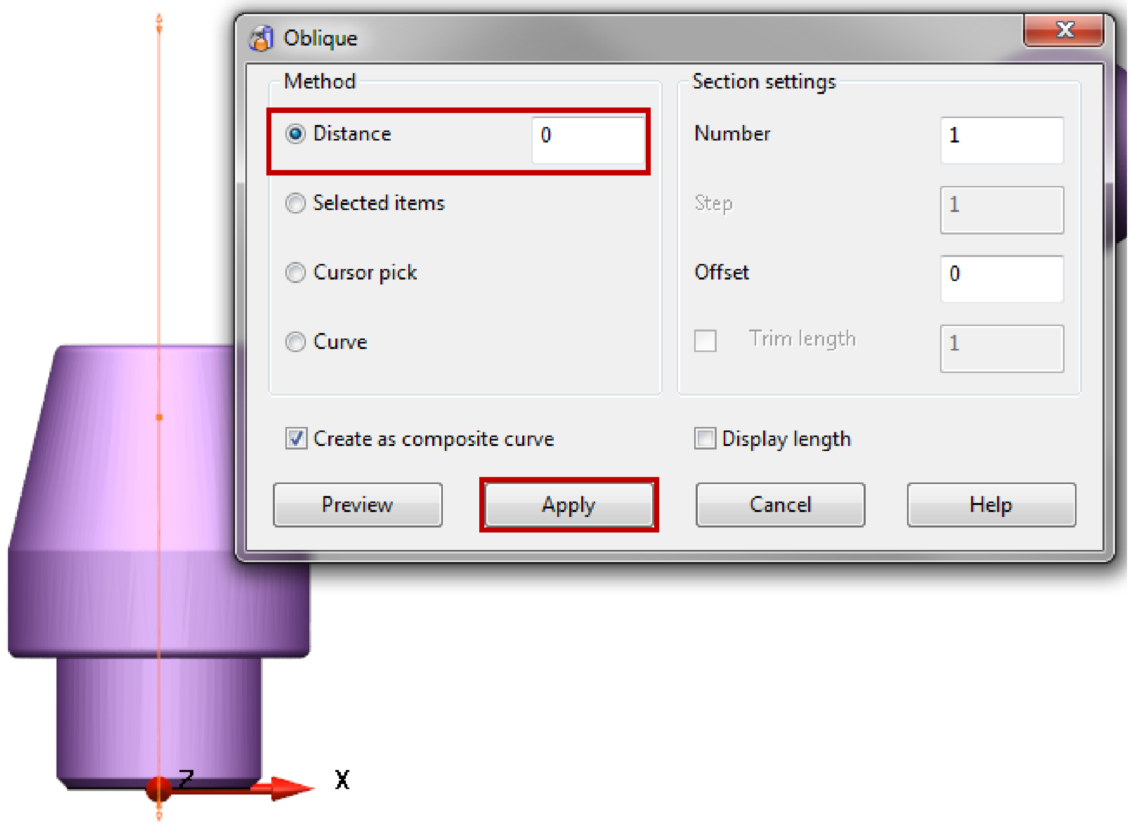
15 Select **Object>Mesh>Combine**.

We will now create an **Oblique Curve** through the mesh at the **workplane** using the **YZ plane**.

16 Select **X** as the **Principal Axis**.

17 Select **Create an Oblique Curve**  from the **Curve**  menu.

18 Within the dialog choose a **Distance** of **0** (which will be from the **principal plane 'YZ'**) and ensure that **Create as composite curve** is **ticked**.



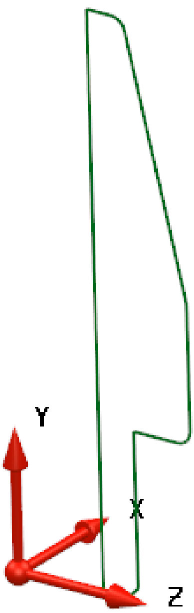
19 Select **Apply** to **create** the **curve**, and then **cancel**.

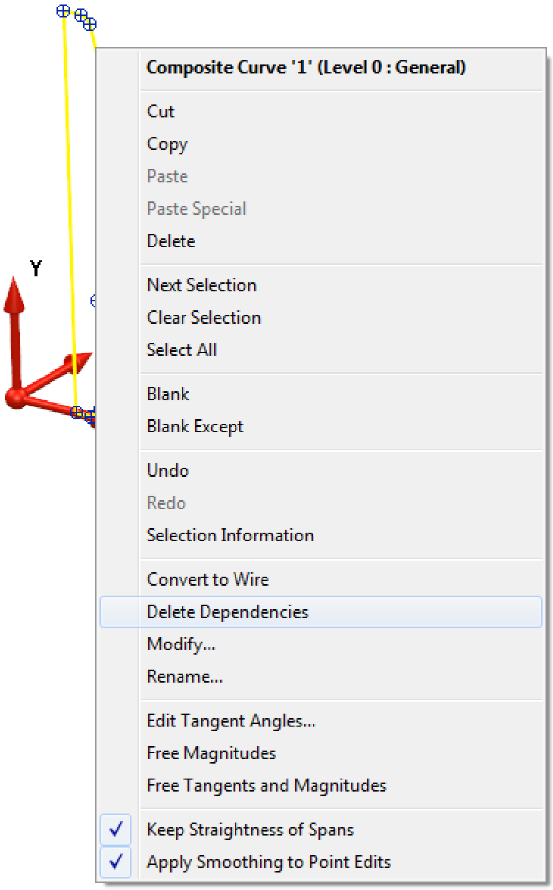
This has create a section curve through the mesh at the point where it intersects the chosen section, in this case the YZ plane of the active workplane.

20 **Select** and **Blank** (**Ctrl+J**) the **mesh** and **Rotate** the **view** to see the **curves**.

As you can see there are currently two curves where the mesh intersected the plane. To create a ***surface of revolution*** *only one will be necessary.*

21 **Select** and **delete** one of the **section curves**, as they are both **indentical** for our requirements it is **not important** which **curve** is **kept**.

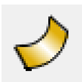




22 Right click on the curve and from the menu options, select **Delete Dependencies**.

***Dependencies** are **relationships** between **geometrical objects**, therefore this option will **remove** any of these **relationships** taken from the **mesh**.*

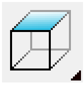
23 Select **Y** as the **Principal Axis**.

24 With the **curve selected**, create a **Surface of Revolution**  from the **Surface**  menu.

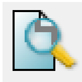
25 **Unblank** the **Mesh** (**Ctrl+L**).

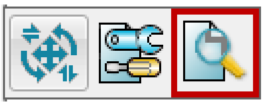
26 **Select** the **Surface** in the **graphics area**.

27 Using the **middle mouse button** place the **surface** on **Level 1 – Surfaces** and then **Turn OFF** the level.

28 Select **View From Top** .

The next step we will take is to create the curves to form the shaft of the handle.

29 In the **Model Analysis**  menu select **Dynamic Sectioning** .



*The **Dynamic Section** tool is also accessible through*

***View>Dynamic Sectioning**.*

***Dynamic Sectioning** is a tool that enables the user to **interactively view sections** through an **object** in the **graphics area**. It can be done relative to the **current View**, or **through** or **around** the **X**, **Y** or **Z axes**. It also allows the user to **create curves** of the **sections** in the **object**.*

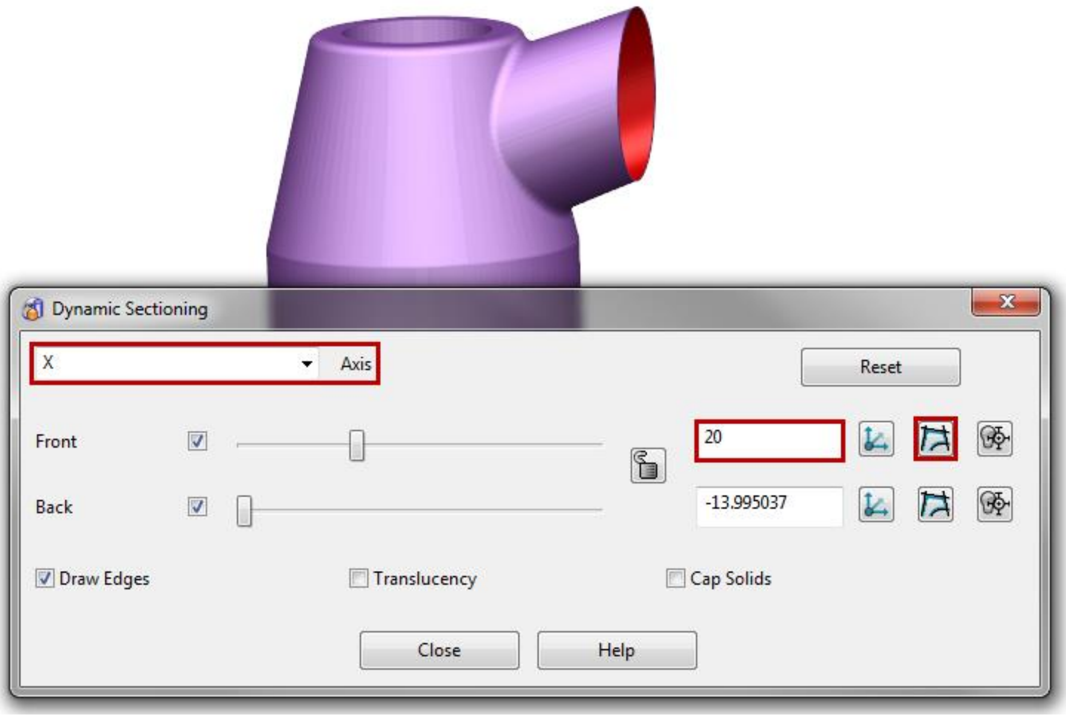
30 In the **dynamic section** dialog choose to take a section through the **X Axis**.

31 Either using the **Front slider** or the **adjacent text box enter** a **value** of **20**.

*As you can see this **sections** the part down the **X Axis** at a location of **20mm**.*

Aswell as viewing the section PowerSHAPE Pro also has the ability to create curves of the sections using the create wireframe button highlighted in the image below.

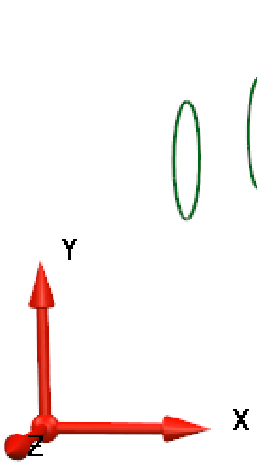
32 **Select** the **create wireframe of selected item on the front plane**  button in the dialog to create a section curve.

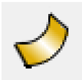


33 Repeat this method and create **section curve** at **30**, **40**, **50** and **60** in the **X Axis** on the **Front** plane.

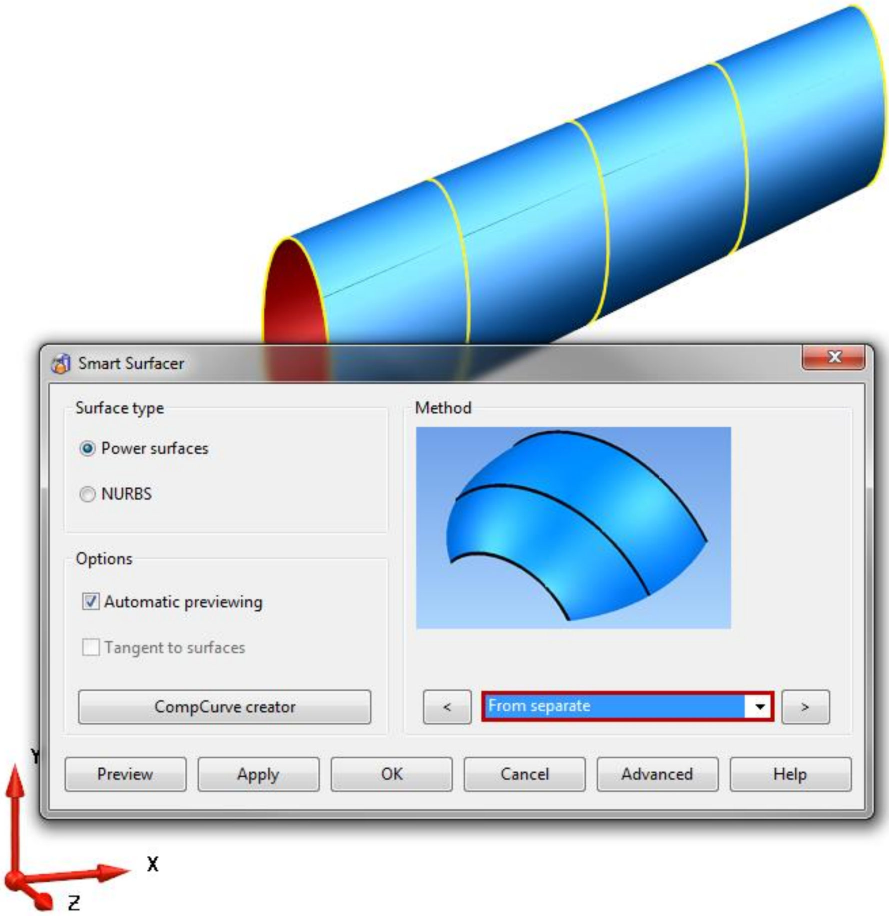
34 **Blank** (**Ctrl+J**) the **mesh**.

35 With **ALL** the curves selected, **right click** on the **curves** and select **delete dependencies**.



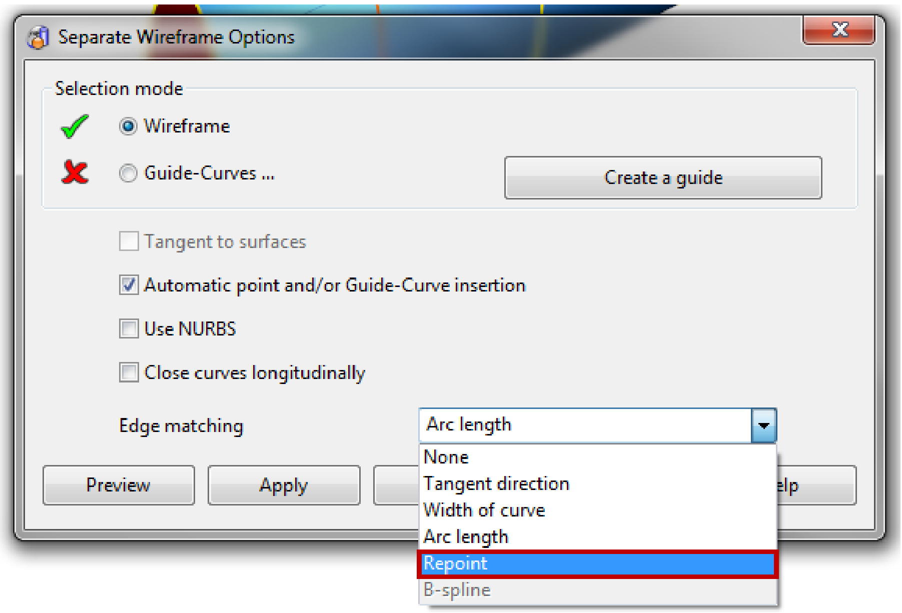
36 Box select the **5 curves** and open the **Smart Surfacer**  from the **Surface**  menu.

37 In the **dialog** select that **surface creation method** as **From Separate**.



As you can see the **smart surface tool** has created a **conical surface** from the **section curves**. However, as the **structure** of the **curve points** varies greatly between each **curve**, this has created a **'rippled' surface** and **uneven structure** to the **surface**. To combat this we need to **repoint** the **curves** so that there is the **same number of points** on each, **evenly spread** apart.

38 Select the **Advanced** button in the dialog to open the **advanced options** for a **From Separate** surface.



39 In the **Edge Matching** drop down menu choose **repoint**, then select **Preview** to see the effect of the changes.

*In the **graphics area hover** the **mouse** over the **surface** and notice how the **structure** has changed to a much more **even spread** of **longitudinal curves** running the length of the **surface**.*

40 Select **Apply** to both the **advanced options** and **smart surfacer** to **create** the surface.

41 **Unblank** the mesh (**Ctrl+L**).

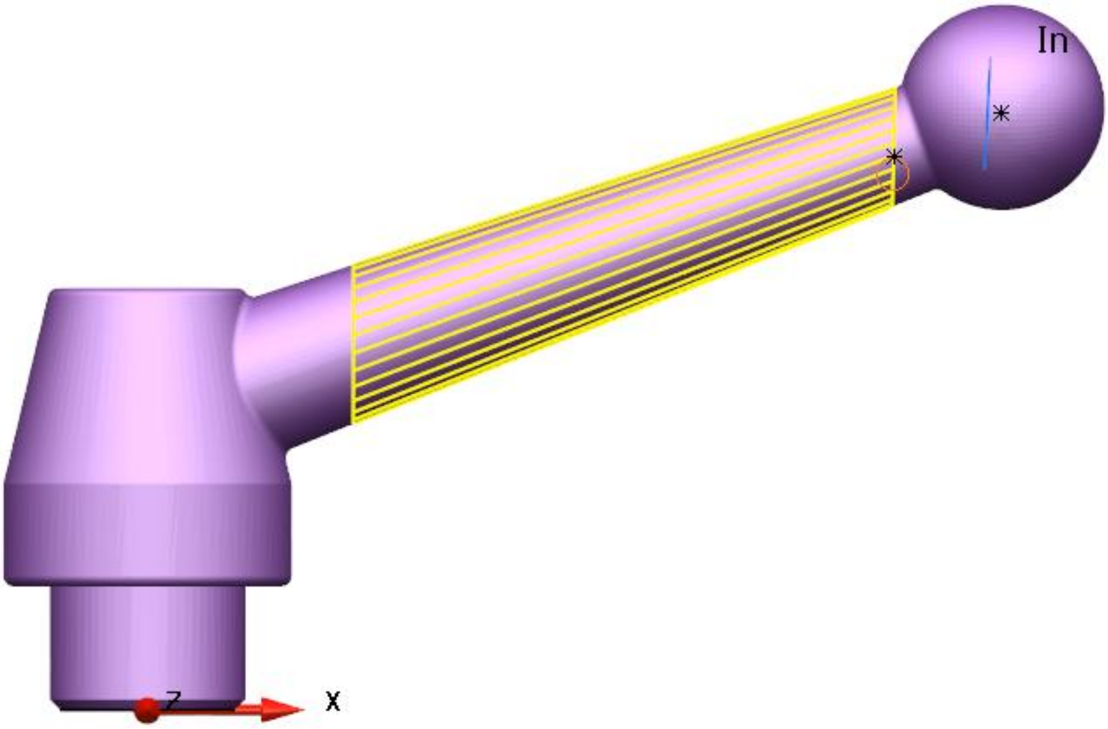
As you can see the **surface** has been created using the **section curves derived** from the **mesh** and has now produced a **satisfactory result**. However the current surface does not fully extend the length required to **complete** the **model**. Next, we will use the **point limit function** to achieve this.

42 Select a **View from Top** (**Ctrl+5**).

43 With the surface selected and the mesh still visible, from the **General Edits** menu select **Point Limit**. 

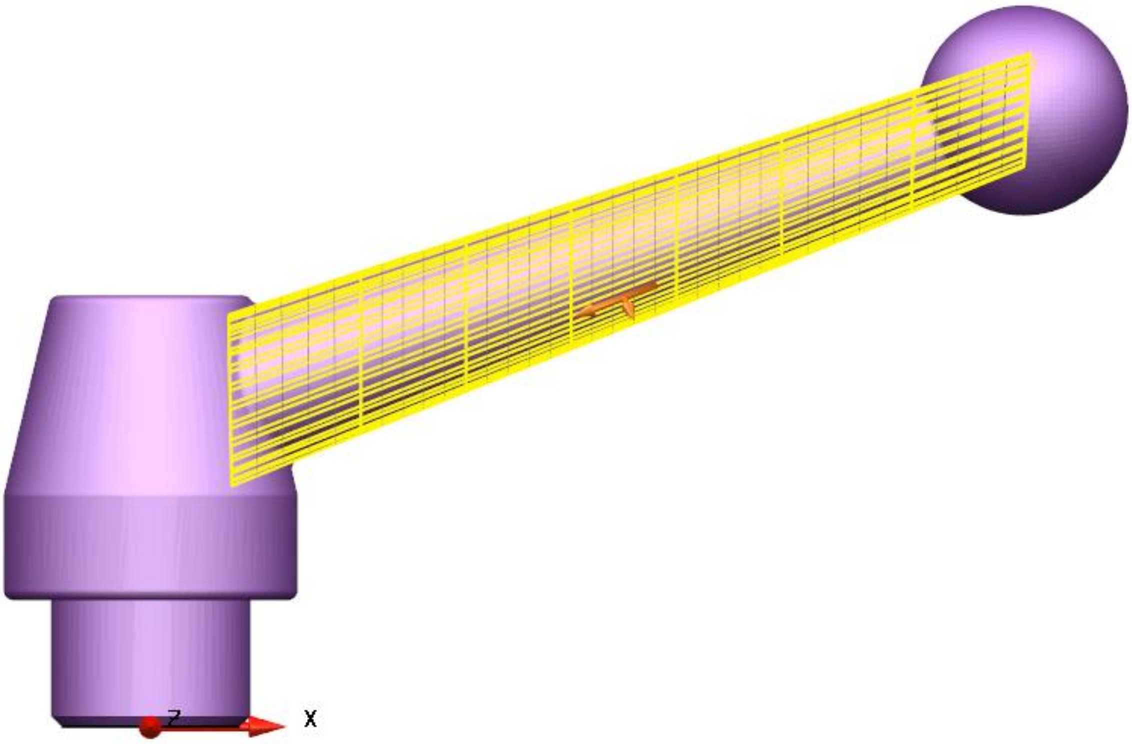
44 **Click** and **drag** the **right hand end curve** of the **surface** to **extend** it inside the **spherical end** of the **handle** as shown in the image below.

*In this example we have extended the surface slightly further than necessary to ensure we can further **trim** it back at a later stage. The **blue preview curve** shows where the surface will **extend** to.*



45 **Repeat** the above **process** for the other end of the **cylindrical surface**. In this case it is also **important** to **ensure** that the **surface does not extend too far** through to the hole through the part.

*If you know exactly the **distance** required it is also possible to insert the **value** (in **mm**) into the **dialog**.*

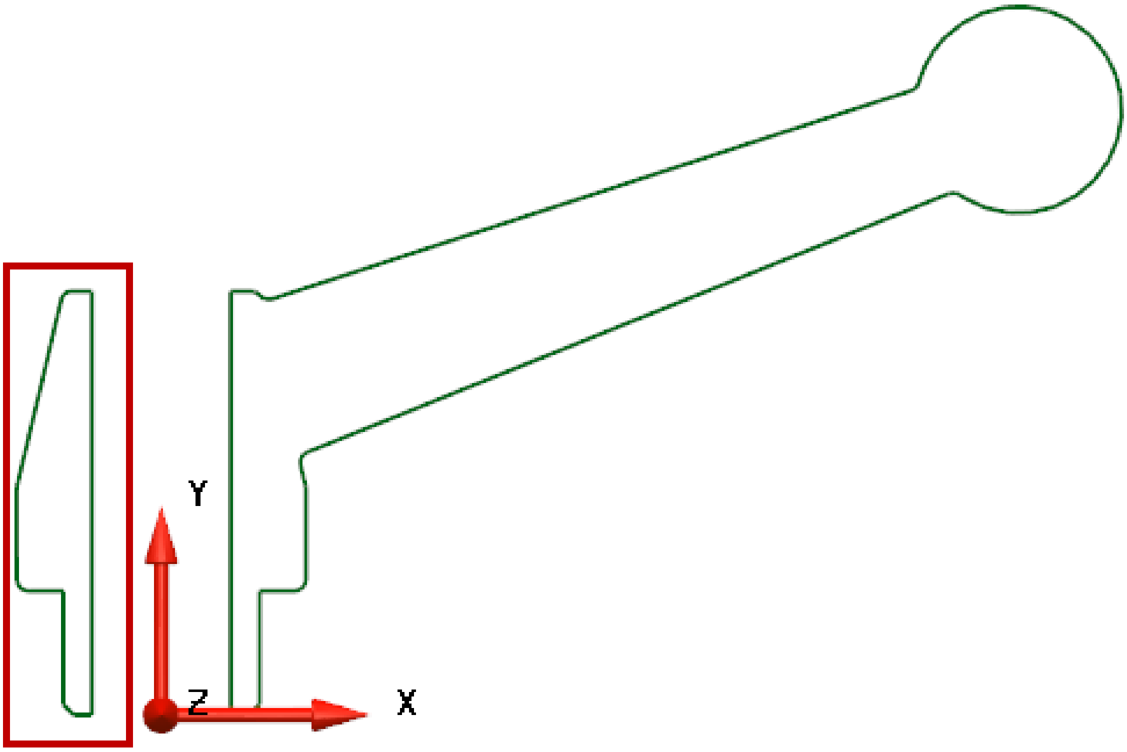


46 Select the **surface** and place it on **Level 1 – Surfaces** using the **middle mouse button**.

47 Open the **Dynamic Sectioning**  tool from the **Model Analysis** menu.

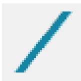
48 Choose to take a **section** down the **Z Axis** and use the **slider/text box** to set a value of **0** on the **Front** face with the **create wireframe button**. 

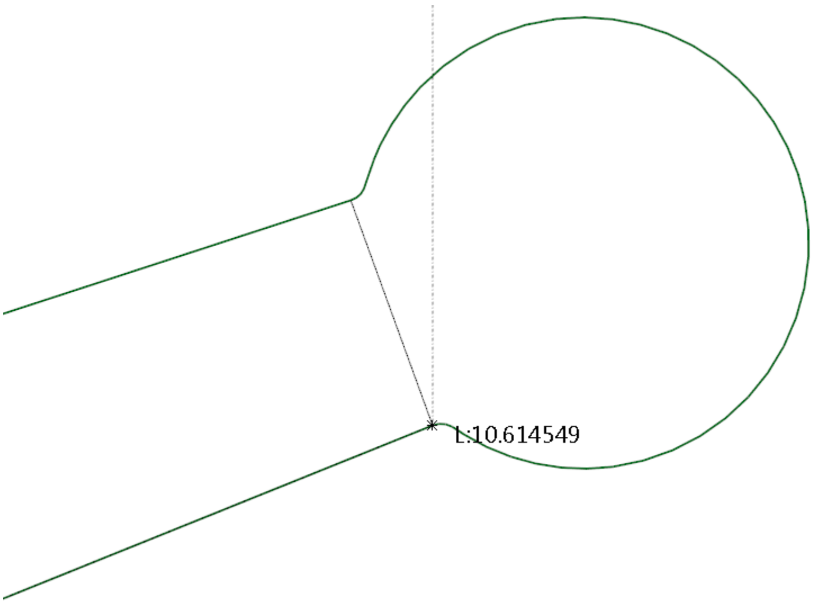
49 **Blank** the **Mesh** (**Ctrl+J**).



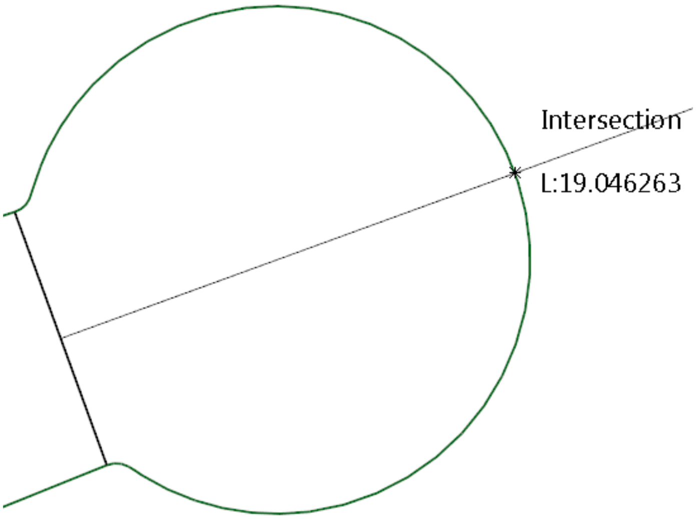
50 **Select** and **Delete** the **left hand side curve** that we do not require for the next operation marked by the **red box** in the **image above**.

51 Zoom into the circular area of the curve representing the end of the handle.

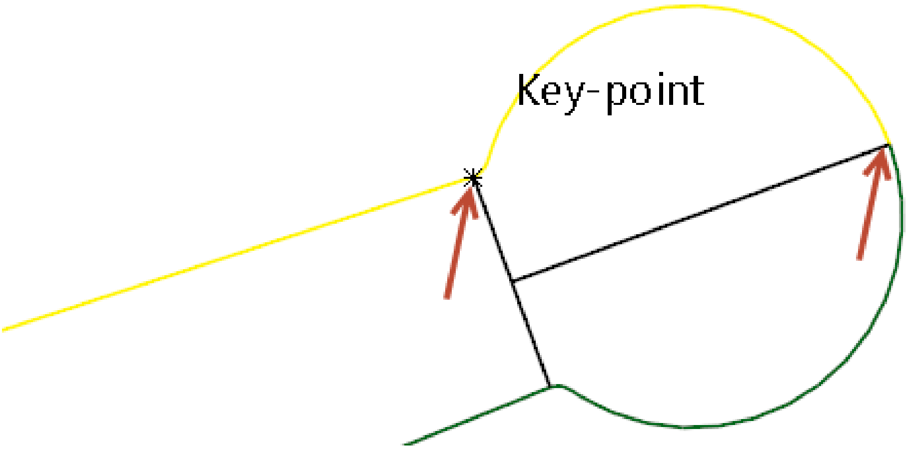
52 **Create a Single Line**  connecting the **two Keypoints** defined by the edge the the **radiused corner** meets the **handle shaft**.



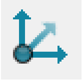
53 From the **midpoint** of this **new line**, **create** a further **single line running perpendicular** until it **intersects** the **arc**.

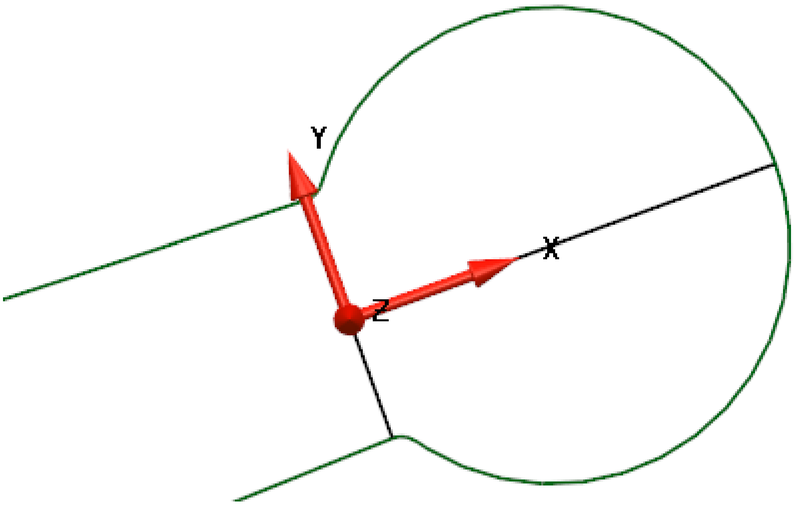


54 **Select** the **section curve** and then open the **Cut Object**  function from the **General Edits** menu.

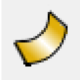


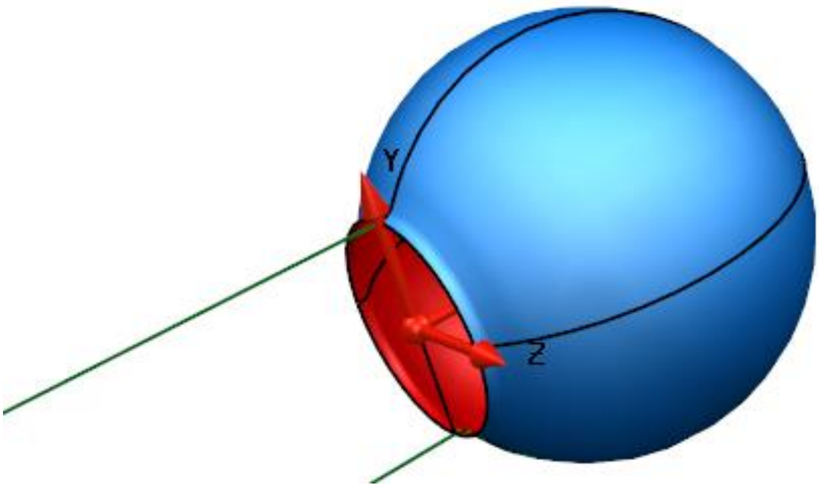
55 Using the **Intelligent Cursor snap** to and **cut** the **section curve** in the **two locations** specified in the **above image**.

56 At the point where the **two straight lines intersects** place a **single workplane**  and **realign** it to the lines. We will use this **workplane** to create a **surface of revolution**.



57 Set **X** as the **Principal Axis**.

58 Select the **Arc shaped curve** above the **new workplane**. From the **Surface**  menu select **Create a Surface of Revolution**. 

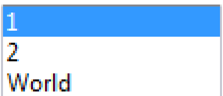


59 Select the **Surface of Revolution** and place it onto **Level 1 – Surfaces**.

60 From the **Selection** flyout menu choose **Quick Select ALL Wireframes**. 

61 Place **all** the **wireframe** onto **Level 2 – Wireframes**.

62 From the **Workplane selector** drop down menu **activate workplane 1**.

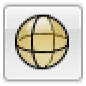


63 **Turn ON Level 1 – Surfaces** to show all of the surfaces we have recreated over the mesh.

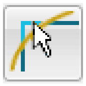
*Note that the mesh is still **blanked from view** at this point.*

64 **Turn OFF Level 0 – General**.

This will allow us to work solely with the created surfaces.

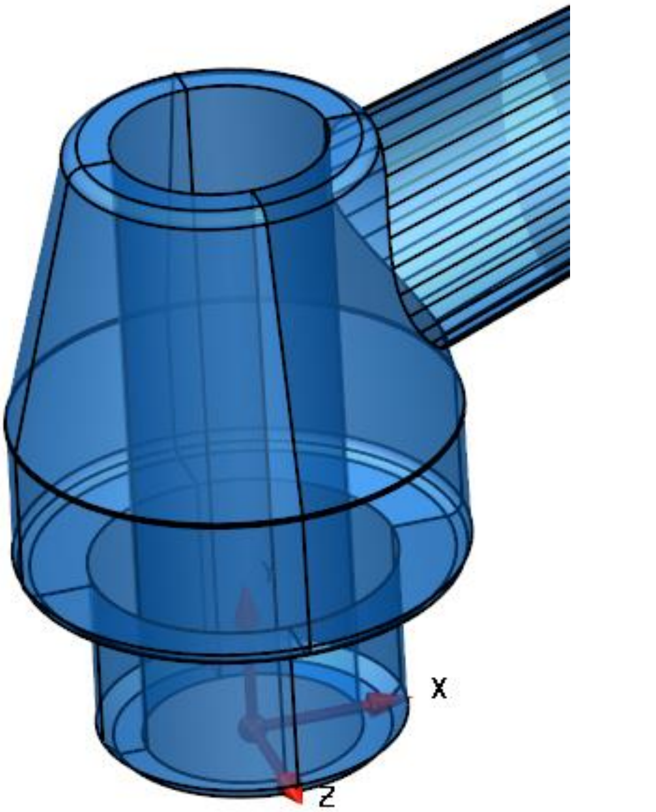
65 Switch to **Transparent View**. 

*This will give the ability to **select objects** that are **inside** or **hidden by** others.*

66 From **General Edits** select **Limit Selection**. 

67 Select the **Surface of Revolution** as the **Cutting Object** and then cut the end of main shaft of the handle that is **inside**.

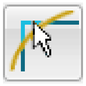
*The **Next Solution** option is always available within the dialog if your selection is incorrect which will cycles through **all possible solutions**.*



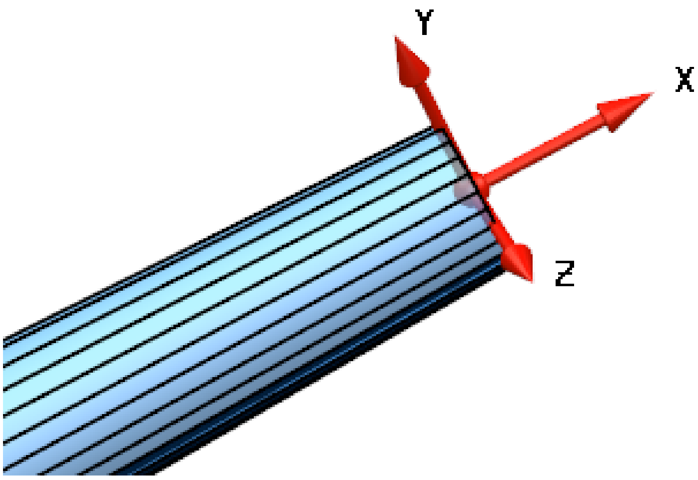
68 **Activate Workplane 2**.

69 **Select** the **cylindrical shaft surface** and then **Blank Except** (**Ctrl+K**).

70 Set **X** as the **Principal Axis**.

71 Open the **Limit Selection**  dialog from the **General Edits** menu.

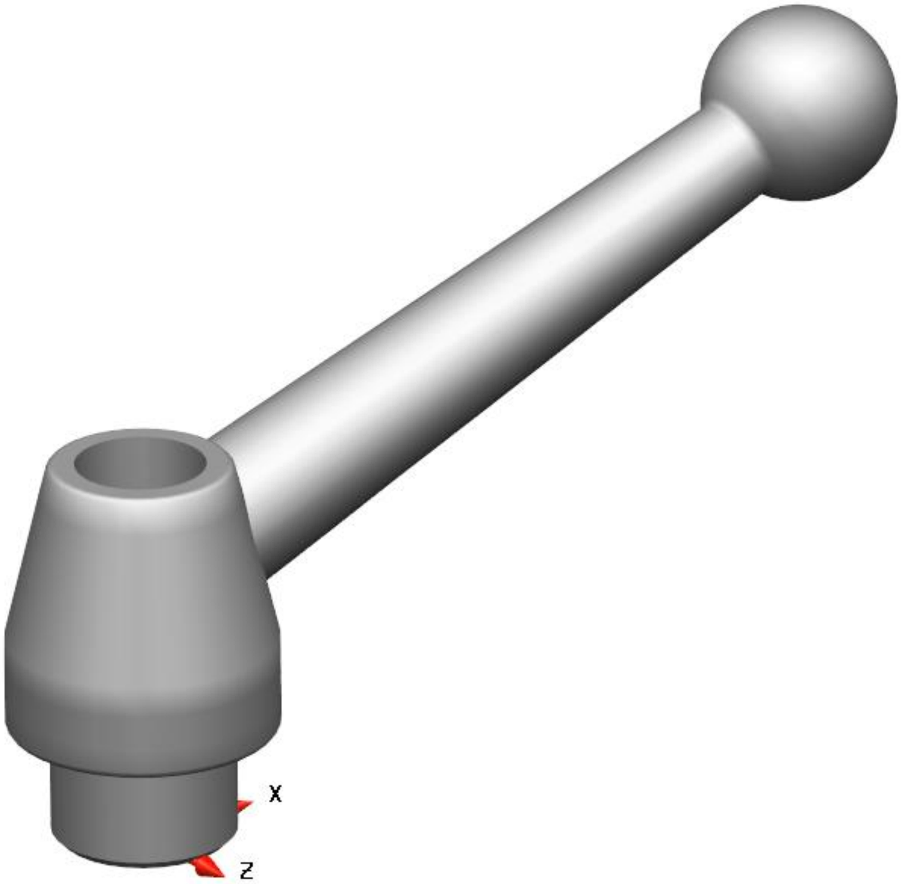
72 Using the **workplane** (and appropriate principal axis) as the **cutting object** trim back the other end of this **surface**.

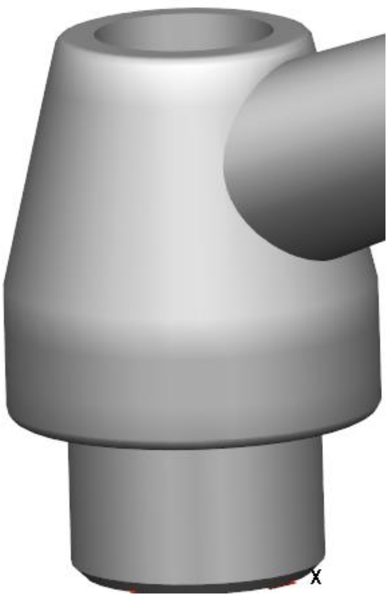


73 **Unblank ALL** the **Surfaces**.

74 From the **Selection Flyout** menu **Quick Select ALL Surfaces**. 

75 Select **Create Solid from Surfaces**  in the **Solid Creation**  menu.





Next we will create a **0.5mm fillet** around the join shown above.

76 From the **Feature**  menu select **Solid Fillet**. 

77 Input a **Radius** of **0.5mm** and **select the edge** in the **graphics area**, then select **Apply**.

We have now completed the resurfacing of the model.

78 **Save** the **model** as **ResurfaceHandle.psmodel**.

Introduction – Method 2

The **following exercise** will **resurface** the **mesh** of a **handle** using the new **Automatic** and **Manual mesh segment tools** available in **PowerSHAPE Pro**. This exercise will cover:

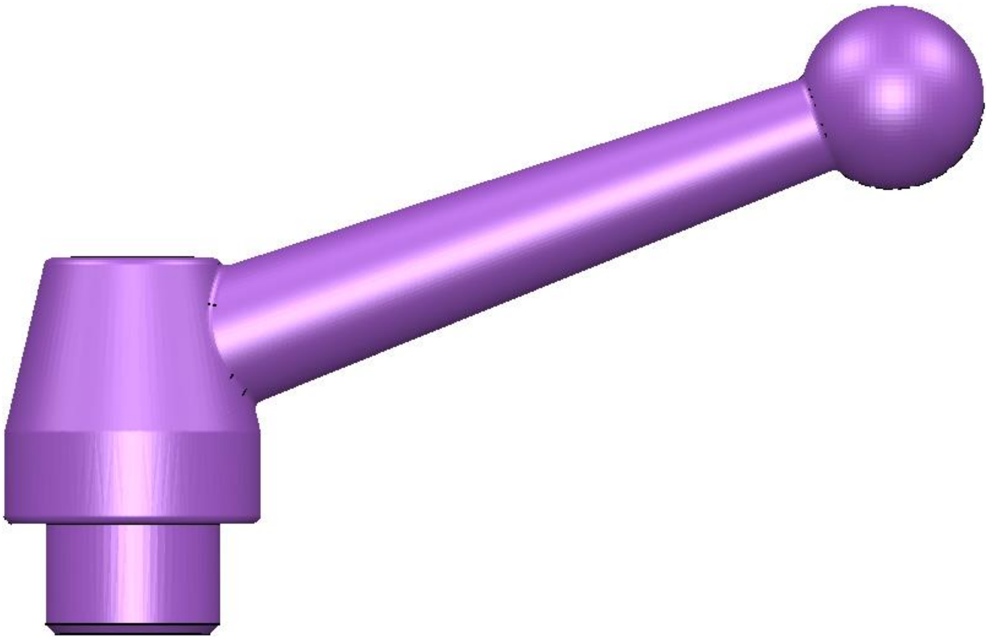
- Creating **Surfaces** using the **Automatic Mesh Segment** tool.
- Creating **Surfaces** using the **Manual Mesh Segment** tool.
- Creating **Solids from Untrimmed Surfaces**.

Resurfacing the Handle using the Mesh Segment tools

1 Select **File>Import...**

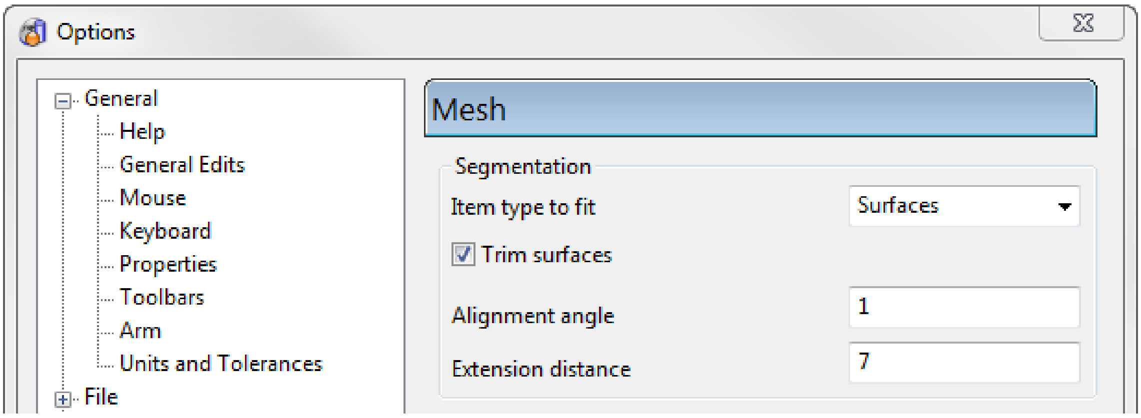
2 Browse to: **C:\Training Data\PowerSHAPE Pro Data** and **import** the file **handle.stl**.

3 Select a **View from Top** (**Ctrl+5**).



The **Mesh Segment** tools can be used **Automatic** (where possible) or **Manually** to create **surfaces** ond/or **solids** different areas of the mesh to later be combined to **recreate a model** from the **mesh.** There are a variety of **options** relating to these tools which will allow us to further understand how we can successfully **utilise** this **functionaility**.

4 **Browse** to **Tools>Options>Object>Mesh**.



The options in this menu relate to the Mesh Segmentation tools and are defined as follows:

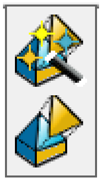
- **Item type to fit:** Select **Surfaces** or **Solids** to be the **default** item type to fit to the mesh when segmenting, this can be changed also within the tool dialogs.
- **Trim surfaces:** Select this option to toggle between creating **trimmed** or **untrimmed surfaces** when **segmenting**.
- **Alignment angle**: Enter a **value** to define the **tolerance** for **aligning primitives** with a **principal plane**. If an **axis** of a **primitive** is within the **tolerance** of a **principal plane** the axis is snapped to the **principal plane**.
- **Extension distance:** Enter a value to define the **distance** by which a **primitive** is **extended** beyond the **edge** of its **region**.

For this example we will leave the default options the same.

5 **Select** the **Mesh** to **activate** the **Mesh Edit** toolbar.

Automatically Segment Mesh and Fit Primitives

The **Automatically** and **Manually Segment Mesh** options are located on the **Mesh Edit** toolbar organised into a **flyout menu**, shown below.



As you would expect it is obviously worth using the **Automatic Mesh Segment** tool for all **available** and **relevant surfaces** before using the **manual tool**. However, as you will see on some **mesh** this may not find all **objects** you require to **recreate** the **model**, although a finer mesh may help.

6 Select **Refine Mesh**  from the **Mesh Edit** toolbar.

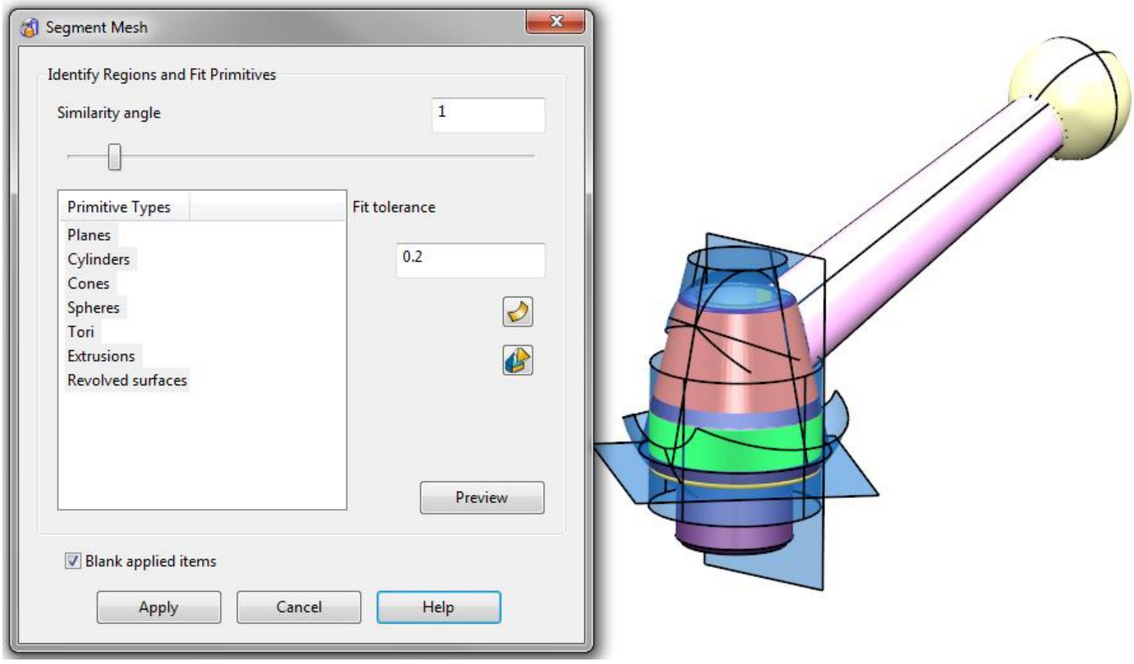
7 Using the option **Split Each Edge**, select **Apply Twice**.

8 Click **Cancel** to **Close** the dialog.

*This has simply **split each edge** of every triangle to quickly and easily produce a **finer mesh** to aid the use of the **Automatic Segment** tool in this case, obviously this is a **compromise** between working with a **larger number of triangles** and should be **avoided** on already **large mesh**.*

*Feel free to undo the refine mesh operations and then have a look at the **primitive surfaces** that are found with far less triangles. In this case it is because there are a **small number of triangles** in the mesh **compared** with the **size of the mesh**.*

9 Select the **Automatically Segment Mesh**  button to automatically search for **primitives** and open the **Segment Mesh** dialog.

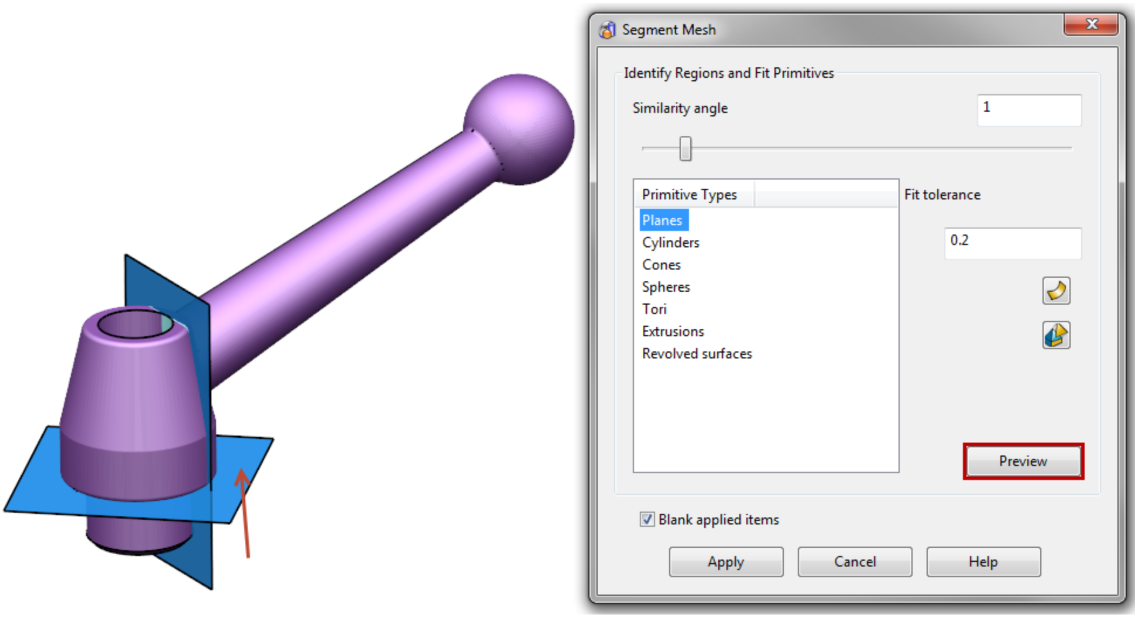


As you can see now the **Automatic Segmenter** find and previews a large number of **different primitives** all over the mesh. While the dialog allows you to define a threshold **similarity angle**, of which a **large value** will suit **more coarse mesh** with **stronger discontinuities** and a **smaller angle** will generally reduce the **unsuitable surfaces** found on **finer mesh**.

Within this dialog you can also choose which **primitives** you would like to look for at any **instantaneous moment** which can make this process simpler when there are a large amount of **different primitives** found by **reducing** the **view**.

It is good practice to take a **logical step by step** approach to efficiently work through the **Automatic Segment** tool, therefore we will work through **each primitive type individually**.

10 In the **Segment Mesh** dialog select **Planes** from **Primitive Types**, and then select **Preview**.



*With a **similarity angle** of **1 PowerSHAPE Pro** has found **two planar surfaces**, only **one is relevant** in this case.*

11 Select the **Plane** marked in the **above image** and then select **Apply**.

*With the **surface selected**, **right-clicking** on the **surface** will perform the same function as **Apply**.*

*Once applied, the **area** that have already been **covered** will be **highlighted** in **white** and will stop you from **creating duplicate surfaces**.*

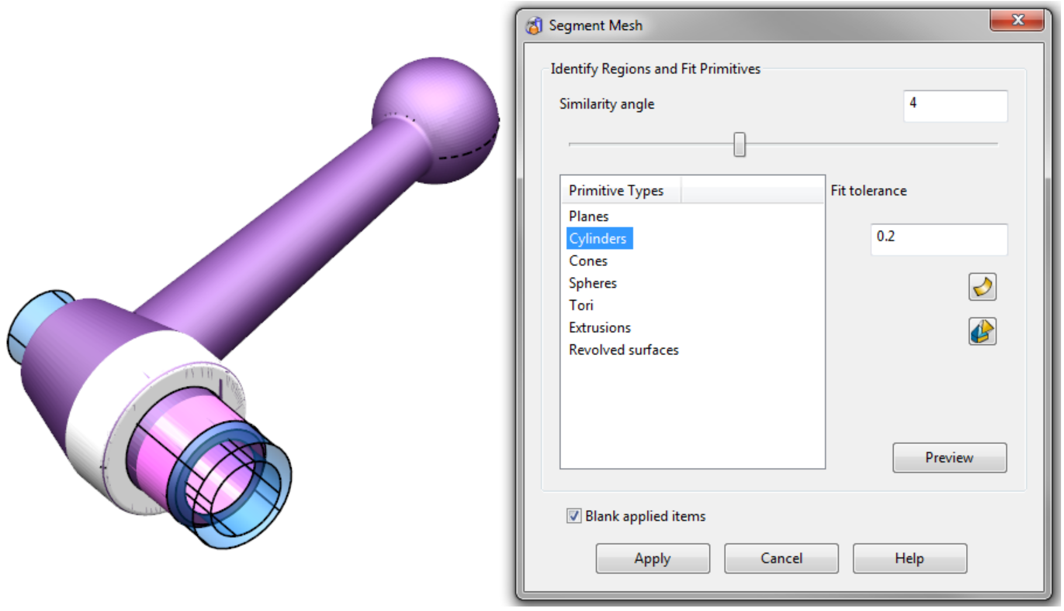
12 Next, select **Cylinder** in the **Primitive Types** area.

13 **Select** and **Apply** the **Cylinder** found on the **mesh** in the **graphics area**.

*There are other **cylinders** that currently **have not** been **recognised** in the **model**.*

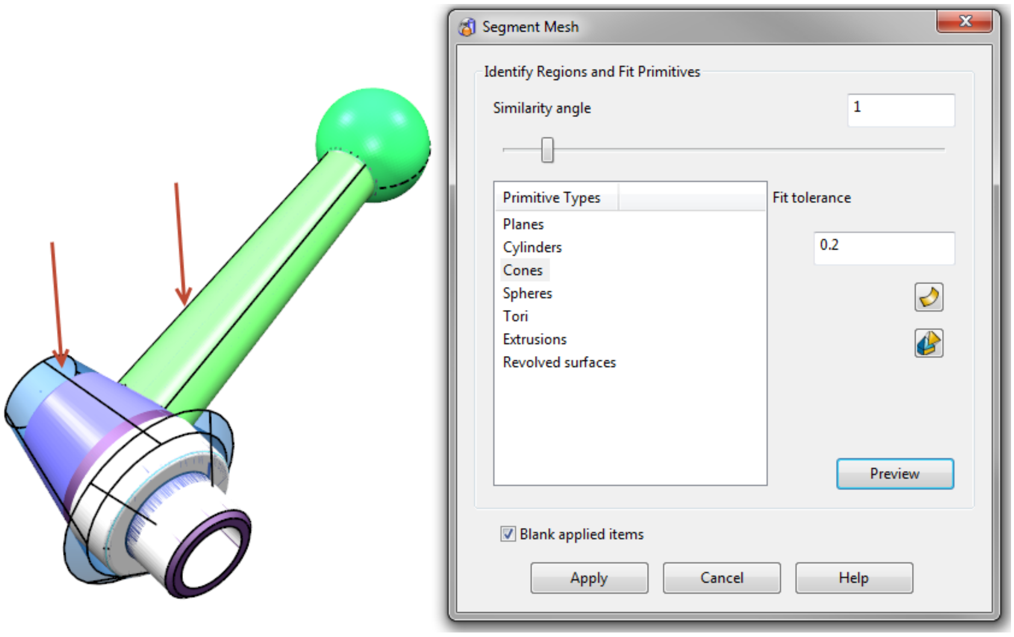
14 **Increase** the **similarity angle** to **4** using the **slider**, then select **Preview**.

15 **Select** and **Apply** the **two remaining cylinders** on the **mesh**.



16 **Slide** the **Similarity Angle** back to a **value** of **1**.

17 Select **Cones** from the **primitive types**.



18 **Create** the **two** correct **Cones** found by the **Automatic Segment** tool on the **mesh**.

*The other cone has been found due to the **fillets** and **chamfers** around the model, there is no great need to create these now as we can simply create these as **solid features** later while also gaining a **feature history** in the **process**.*

19 **Select** to find **Spheres** in the **primitive types**, then **Apply** and **Create** the **sphere** at the **end of the handle**.

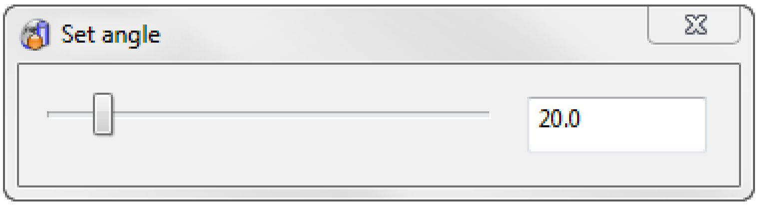
Looking around the model the **Automatically Segment Mesh** tool has found nearly all the required surfaces. In this case there are only **two planar surfaces** (aswell as associated **chamfers** and **fillets**) that haven't been found on the **model**. We will no use the **Manually Segment Mesh** tool to attempt to complete the **surface creation** for this **model**.

Manually Segment Mesh and Fit Primitives

The **Manually Segment Mesh** tool is available directly from the **flyout menu** on the **Mesh Edit** toolbar described previously, and is also available directly from within the **Automatic Segment** dialog.

20 **Select** the **Manually Segment** mesh option inside the dialog which will open its dialog below.

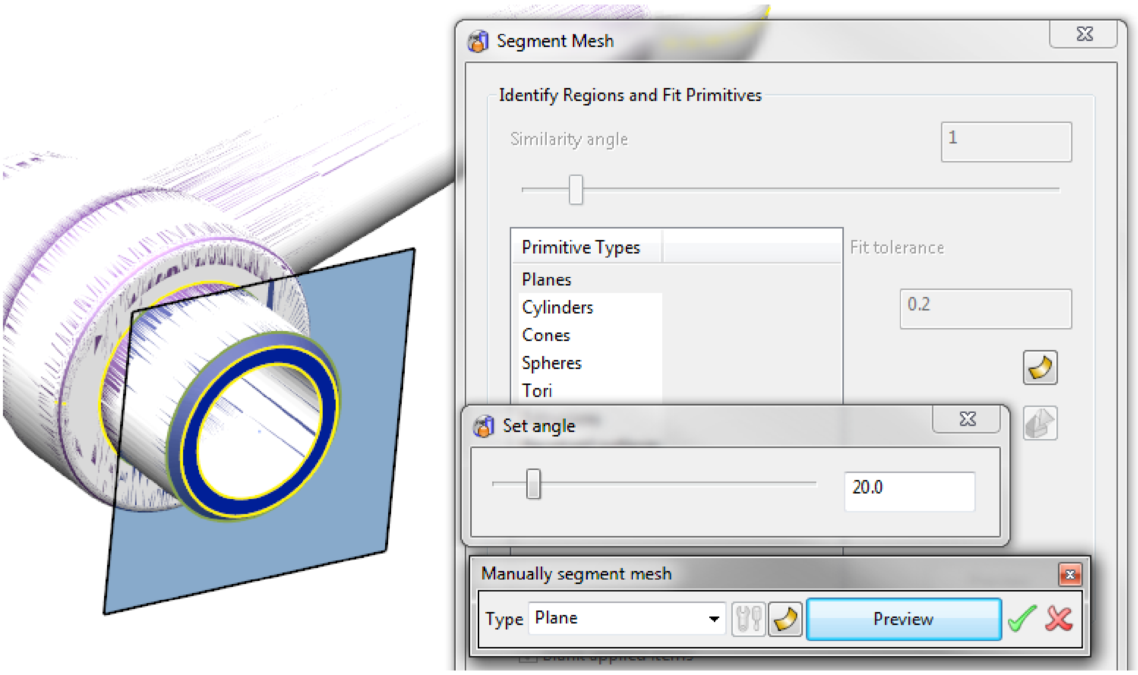
This will also open the **Set Triangle Area to Local Horizon** menu to allow to select **individual (multiple) triangles** in the **mesh**. This dialog allows you to manually seek primitive object on the mesh using the drop down menu provided.



*A **discontinuity angle slider** allows the user to change the **threshold value** which will **increase** the selection of triangles in a selection for a **larger value** by increasing the **allowable discontinuity between triangles**. In this case we are only selecting **planar surfaces** so we require a **low value** to avoid selecting further than required.*

21 Using the **default values** select the **bottom face** of the **handle** shown on the **next image**.

Hoevering your mouse over the preview will show you the **fit** (in **mm**) between the **preview surface** and the **selected area** of the **mesh**.

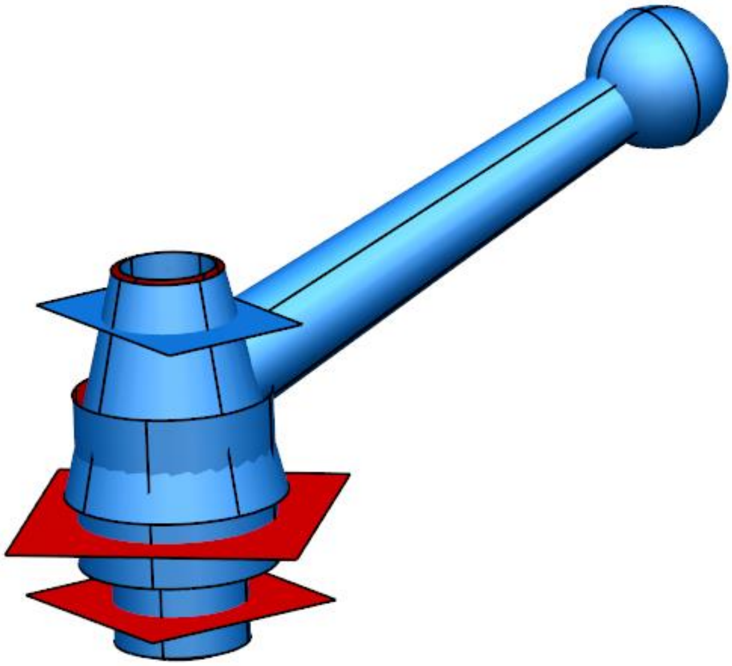


22 Once you are happy with the **preview** select the **green tick** in the dialog to create the surface. Again, as with the **automatic segment tool**, the **manual tool** will create and **immediately blank** the surfaces until you are done inside the dialog while highlight the area already covered in **white**.

23 **Rotate** the **view** and create the **final planar surface** on the **top** of the **model** using the same **process**.

24 This has **completed** the process of creating the **primitive surface required** to create the model. **Select** the **Red Cross** to **close** the **manual segmenter**, and then **cancel** to **close** the **automatically segment mesh** dialog.

25 **Select** and **Blank** (**Ctrl+J**) the **Mesh**.



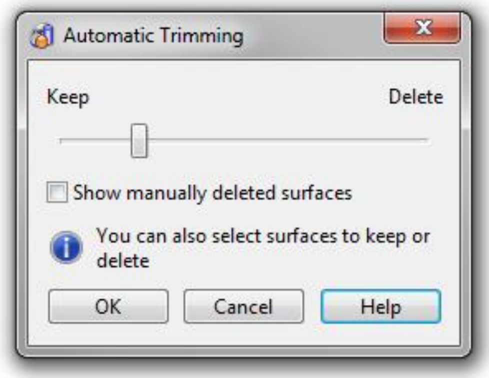
Above is an image of the **surfaces** that have been created using the **Automatically Segment Mesh** tool complimented later by the **Manually Segment Mesh** tool. We can see that it forms a complete inner shape while the extension distance has ensured that the untrimmed surfaces has extended far enough to create a complete inner close shape.

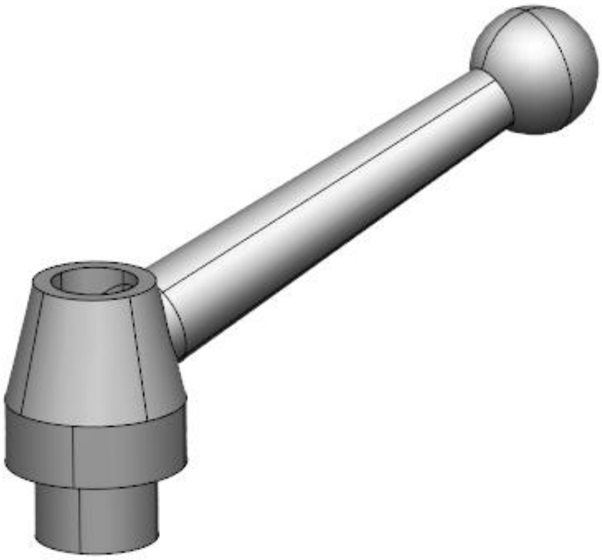
It is often **important** to pay close attention where possible to the **orientation** of the **surface before** we try and create a **solid model** from these surfaces, and ensure that they **correctly overlap without forming any gaps** in the model**.** We are ideally looking to see the **blue** '**positive**' side of the **surfaces** as what we would consider as the **outside** of the **solid**.

26 **Quick Select ALL Surfaces**. 

27 From the **Solid**  menu select **Create Solid from Untrimmed Surfaces**. 

28 This **Automatic Trimming** tool will attempt to **create** the **most relevant solid** from the **selection** of **untrimmed surfaces**.

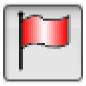
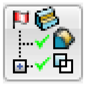




If the solid is not quite what you were expecting the slider will **keep more of the surfaces** or further **delete more surfaces**. **Alternatively**, it is possible to move the **slider** further towards '**Keep**' and then **manually delete as required**.

29 Press **OK** to create the **Solid**.

30 Select the **Solid** in the **graphics area** to **activate** the **Solid Edit** toolbar.

31 Ensure the new solid is the **active solid**  and then open the **Solid History Tree**. 

*Currently there will be **no solid history** but as we create further **features** this will build up as **fully editable history** within this **window**.*

32 **Unblank** the **Mesh** (**Ctrl+K**).

33 From the **Levels** toolbar **create** a new **Level 10 – Mesh**.

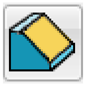
34 Place the **Mesh** on **Level 10**.

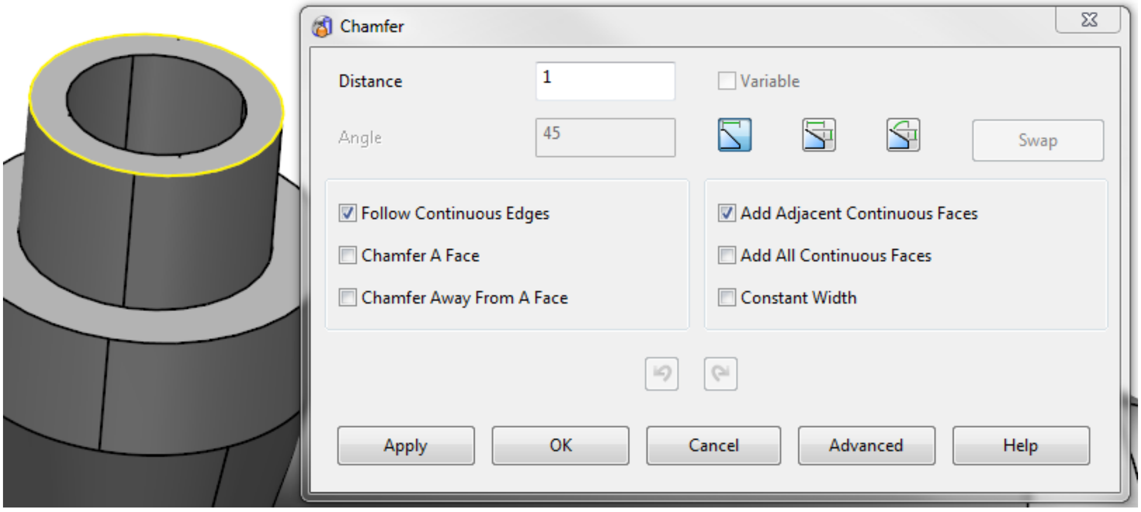
35 From the **Levels** toolbar **create** a new **Level 20 – Solid**.

36 Place the **Solid** on **Level 20**.

*The **surfaces** created earlier are already backed up on **Level 800** as a result of the **operation** so they are **accessible** in the **future** if required.*

37 **Turn ON Level 20 – Solid**.

38 Choose to create a **1mm Solid Chamfer**  within the **Solid Feature**  menu on the **bottom face** of the **solid**, as shown in the **image**.



39 Select **Apply** to create the **Chamfer** then **Cancel** to **close** the form.

40 From the **Feature** menu select **Create a Solid Fillet**. 

41 **Create** a **1mm Solid Fillet** on the face below the previous chamfer as seen in the **image** below.

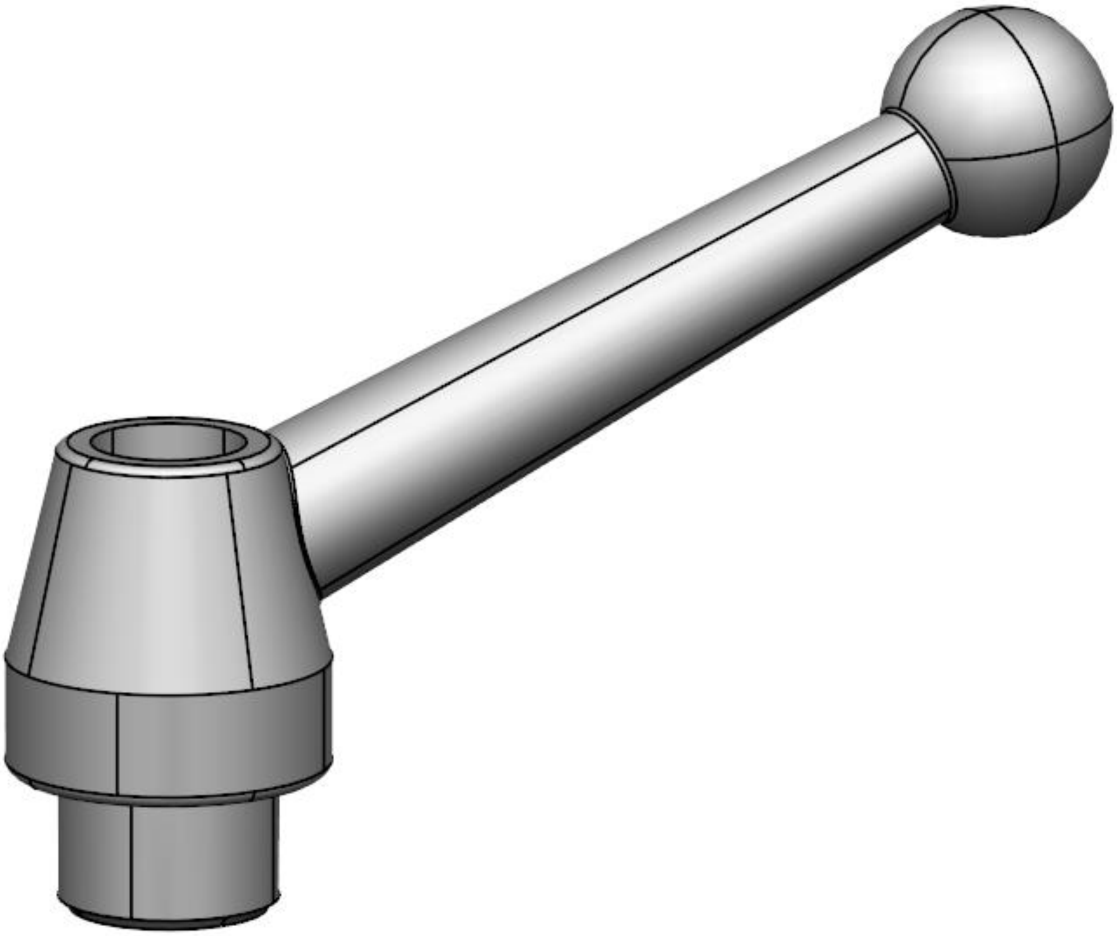
42 Press **Apply** but **do not close** the **fillet dialog**.

43 Create **0.5mm radius fillets** at each end of the **shaft** of the **handle**.

44 Again, **Apply without closing** the **dialog**.

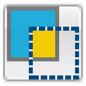
45 Finally, create a **1mm radius fillet** on the **top planar face** of the handle.

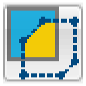
We have now **completed the model** using the **Automatic** and **Manually Segment Mesh** tools. Now you have the knowledge to **recreate solid** and **surface** models from **mesh** using a number of **different methods**.

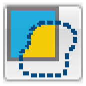


The Mesh Edit Toolbar

The **Mesh Edit Toolbar** will be **activated** above the **grahpics area** with a **mesh selected** in the session and its features are shown and described below. The toolbar is organised into **groups of functionality** and then **sub groups** using the **flyout menus** seen below. Throughout the course we will be using this toolbar heavily, some basic functions will be explained next.

 Pick triangles on a mesh by box

 Pick trianglles on a mesh using a continuous lasso

 Pick triangles on a mesh using a discrete lasso

 Select all triangles on top

 Select visible triangles on top

 Select all triangles

 Select triangle area by colour

 Select triangle area to discontinuity angle

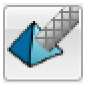
 Select triangle area to local horizon

 Select triangle area to distance

 Thicken mesh

 Reduce mesh

 Refine mesh

 Smooth mesh

 Paint triangles on a mesh

 Morph trianglles general

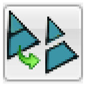
 Morph triangle regions defined by planes

 Morph triangle regions defined by curves

 Automatically segment mesh and fit primitives

 Manually segment mesh and fit surfaces or solids

 Shrink-wrap a surface on to the select mesh

 Divide the mesh into multiple mesh

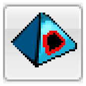
 Divide the mesh into multiple mesh by colour

 Divide the mesh into multiple by selection

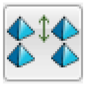
 Reverse the mesh

 Open the mesh doctor

 Fix bad connections

 Fix bad boundaries

 Fix triangles with bad aspect ratio

 Fix small triangles

 Fix intersecting triangles

 Show fixing status of the mesh

 Fill holes in the mesh

 Stitch mesh

 Resolve overlapping triangles

 Apply Z-compensation to triangles

 Mesk selected triangles on a mesh

 Unmask select triangles on a mesh, allowing them to be used

 Unmask all triangles on a mesh, allowing them to be used

 Add a single triangle

 Delete selected triangles

In the following example we will look at the various **mesh shading** options available in **PowerSHAPE Pro** while **reducing** and **refining** a **complete mesh** to help improve the **smoothness** of the mesh, and also **selecting triangles** within a mesh using the **Mesh Edit** toolbar.

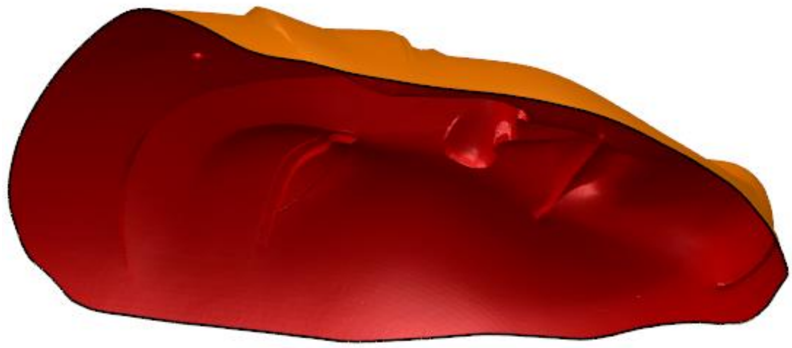
1 **Open** the model **Mask.psmodel** from the **PowerSHAPE Pro Data** folder.

2 If prompted, select **close** on the **File Doctor**.

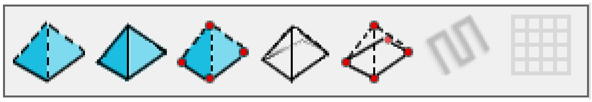
3 Select an **Isometric View** (**Ctrl+1**).

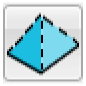
4 Select the **Mesh** by clicking on it in the **graphics area**.

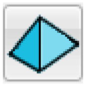
*Note that **selecting** the **mesh** should **show** and/or **activate** the **Mesh Edit** toolbar above the **graphics area**.*

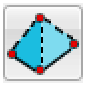


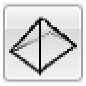
Similar to a surface model, within **PowerSHAPE Pro mesh** have a **negative/reverse** side coloured in **red** by default. While the **positive/normal** side is set to shade as a predefined colour. The mesh is currently set to **Shade Open Edges only** which means that only the 'open edges' of each triangle (without adjoining triangles) are shaded and outlined. This **triangle shading** can be changed using the **Mesh Shading** flyout on the **Views** toolbar. These options will be discussed below:

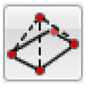


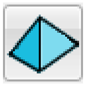
 Open edges only, shaded

 Open and interior edges, shaded

 Nodes and open edges, shaded

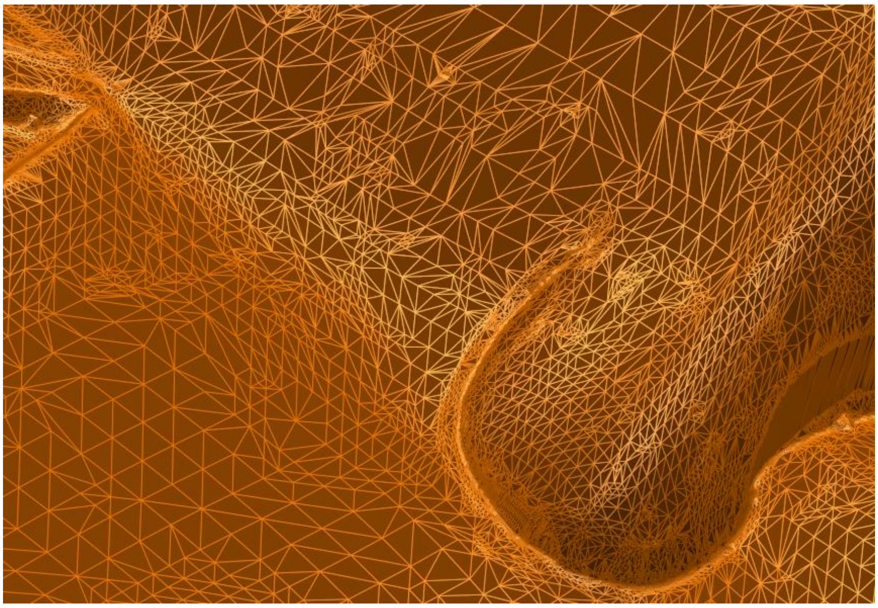
 Open and interior dgse, hollow

 Nodes and open edges, hollow

5 Select the **shading option** to shade and outline **Open and Interior edges**  from the **Mesh Shading** flyout on the **Views** toolbar.

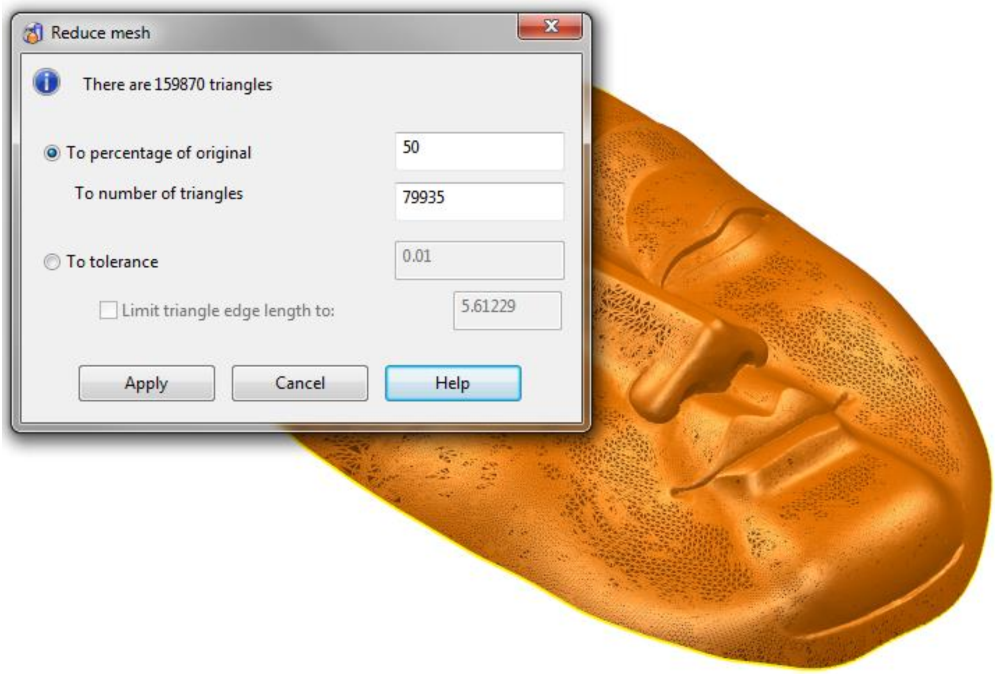
*This option will **outline ALL triangles** in the mesh to give a clearer view of the scale and coarseness of the mesh visually and show the density of the triangles over the mesh.*

6 **Zoom closer** into the **Mesh** to see the detail.



Reduce and Refine Mesh

7 From the **Mesh Edit** toolbar select **Reduce Mesh**  to raise the dialog to reduce the number of triangles in the mesh.

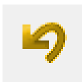


It is possible to reduce the number of triangles in a mesh firstly **To Percentage** or **To Number of Triangles** where entering either value will change the other accordingly.

8 Select to **reduce** the number of triangles in a mesh by **percentage** to **50%**, then click **Apply**.

Take note of the change in the number of triangles at the top of the dialog and the visual difference of the interior shading of the mesh.

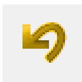
Alternatively to reduce the mesh By Tolerance by entering a defined tolerance to be used while reducing the mesh, aswell as this you can also **limit the triangle edge length** to specifiy the maximum edge length of the triangles to be used during the reduce operation.

9 Select **Cancel** on the dialog and **Undo**  the **last operation**.

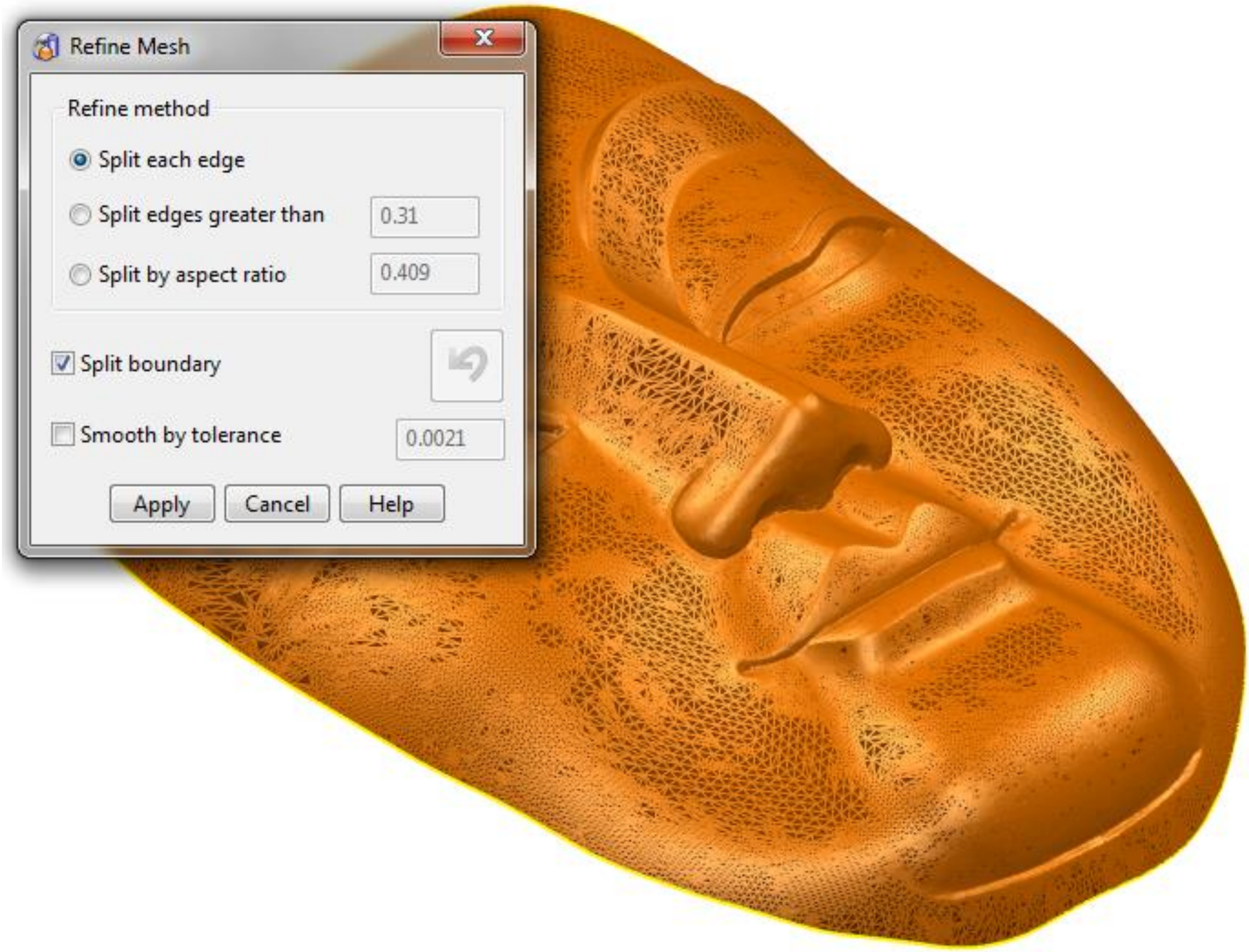
10 Again, select **Reduce Mesh** to reopen the dialog.

11 Select to **Reduce Mesh to Tolerance** of **0.01** and select **Apply**.

This has reduced the number of triangles in the mesh from ***approximately 159k*** *to* ***90k*** *keeping within a* ***tolerance*** *of* ***0.01****.*

12 Select **Cancel** on the dialog and **Undo**  the **last operation**.

13 Select the **Refine Mesh**  option from the **Mesh Edit** toolbar.



The **Refine Mesh** option can be used to increase the **density** of the triangles in the mesh by **dividing** the **existing triangles** and refining the triangles by **curving** the original **triangles** to create a **smoother mesh**.

14 Ensure the **Split Each Edge** option is selected in the dialog, then select **Apply**.

*The **Split each edge** option subdivides **each triangle** into **four new triangles** by inserting **new vertices** at the **midpoint** of **each edge**.*

15 Select the **Undo** button located **within** the **Refine Mesh** dialog.

16 Change the **Refine Method** to **split edges greater than** and enter a threshold value of **0.5**, then select **Apply**.

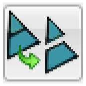
*The **Split edges greater than** option subdivides the triangles by inserting a new vertex at the **midpoint** of an **edge** when the **length** of the edge is **greater than** the **specified threshold value** leaving a very fine mesh.*

17 Select the **Undo** button located **within** the **Refine Mesh** dialog.

18 Finally change the **Refine Method** to **Split by Aspect Ratio** and enter a **threshold value** of **0.1**, then select **Apply**.

*The **Split by aspect ratio** option subdivides triangles with an aspect ratio of less than the specified value. The aspect ratio is the ratio of the height of the triangle divided by the length of the edge.*

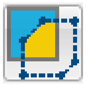
Next, we will look at the dividing the mesh into multiple mesh. For this we have 3 options as seen earlier in this chapter, and repeated below.

 **Divide the mesh into multiple mesh**: This option will divide largely unconnected regions of mesh into multiple mesh.

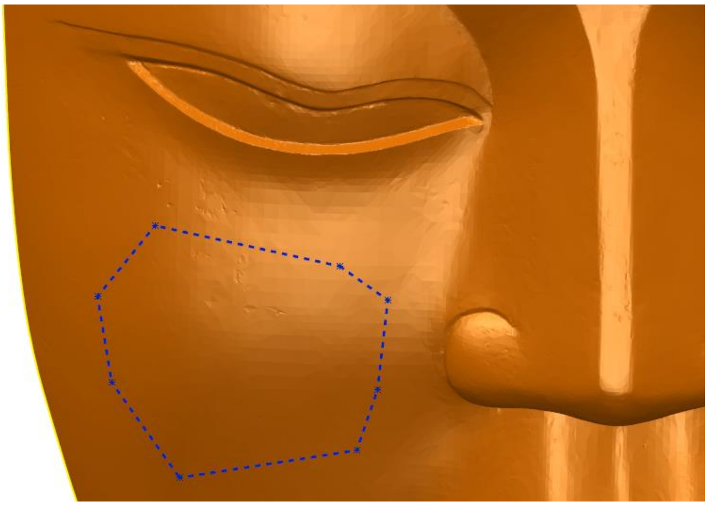
 **Divide the mesh into multiple mesh by colour**: This option will divide a mesh into **different mesh** by **colour**, where colour can be added to **selected triangles** using the **Paint Triangles**  function.

 **Divide the mesh into multiple by selection**: This option will divide a mesh into multiple mesh by **selecting triangles** in the mesh.

19 Select a **View from Top** (**Ctrl+5**).

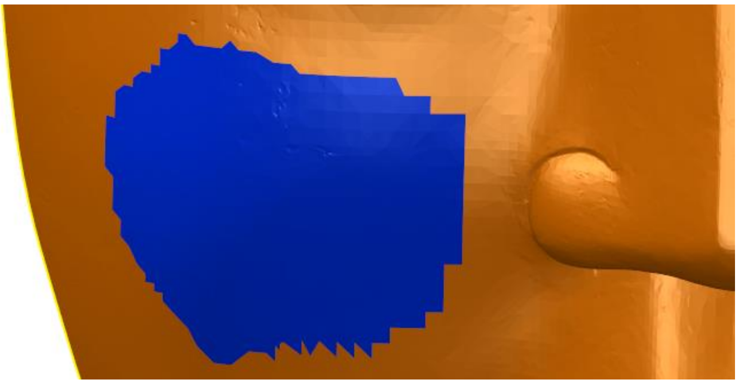
20 From the **Mesh Edit** toolbar select **Pick Triangles by Discrete Lasso** 

21 Using a number of **clicks**, **select** an **area** of the mesh.

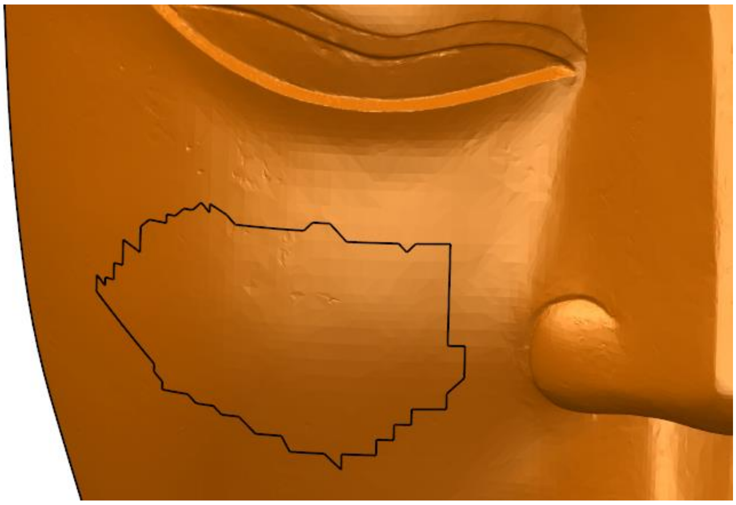


22 **Close** the **bounding box** formed by the lasso by clicking onto the **first point**.

*This will then create the **selection**, and the **selected triangles** of the mesh will become **highlighted blue**.*



23 **Divide** the Mesh by choosing the option to **Divide Mesh by Selection**  from the **Mesh Edit** toolbar.

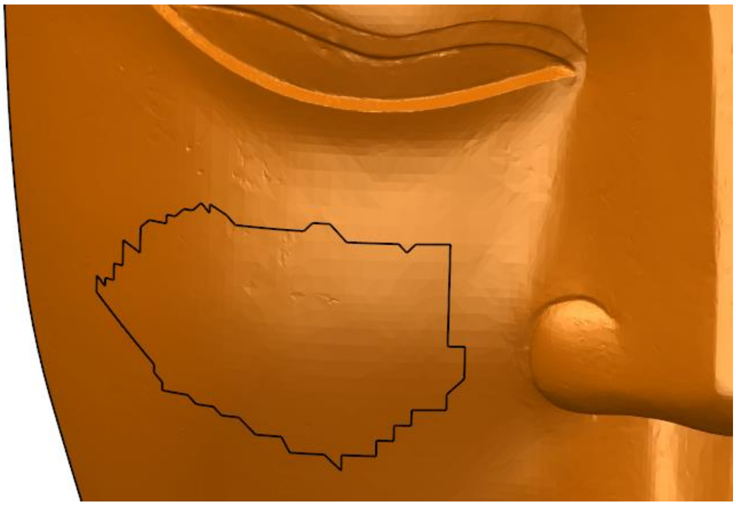


As we know the dark outlines represent the open edges of the mesh which shows that these are now two separate mesh. We will now select and blank the larger main mesh.

24 In the **graphics area** select the larger outer part of the mesh and select **Blank** (**Ctrl+J**).

As we have successfully divided a **single mesh** into **separate mesh** means it is **now possible** to perform the **mesh edits** on **small areas of mesh** by highlighting a **specific area** and using the **specialist select** and **divide mesh tools**.

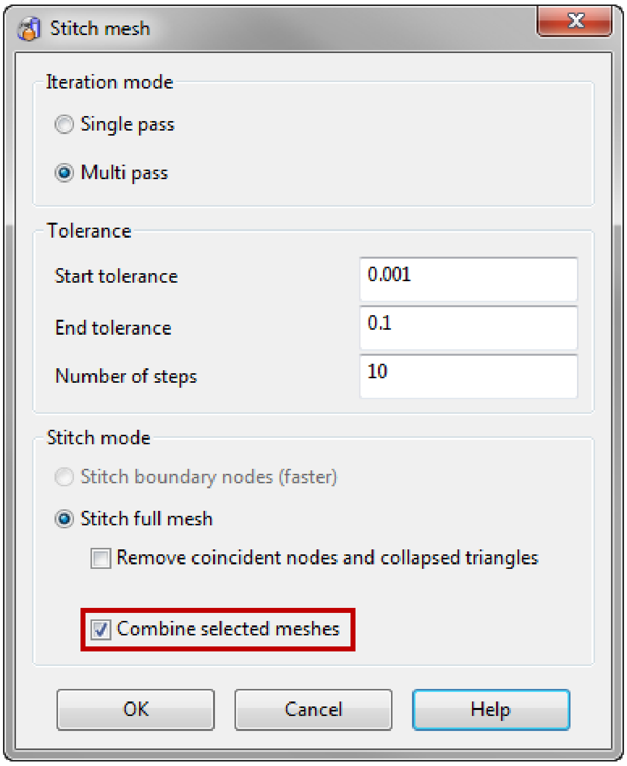
25 **Unblank** (**Ctrl+L**) the other mesh.



Next, we will **stitch** both mesh back together to again form a **single mesh**.

26 In the **graphics area** select both mesh, or use the **Quick Select ALL Mesh** option from the **quick select flyout menu**.

27 From the Mesh Edit toolbar select **Stitch Mesh**  which will raise the **Stitch Mesh dialog**.



As these two mesh were originally one complete mesh, we know that there is a perfect fit between them and therefore we can use the default values successfully.

28 Ensure that the option to **Combine Selected Meshes** at the bottom of the window is selected and then click **OK** to **stitch** and **combine** the **two** mesh together.

*This option to **Combine Selected Mesh** will not only stitch any slight gaps in the mesh, it will then combine all the selected mesh into **one complete mesh**.*

The **Reverse Mesh** option enables the "positive" side of the mesh to be quickly and easily be reversed, for instance to change a **'Male'** cast to a **'Female'** mold.

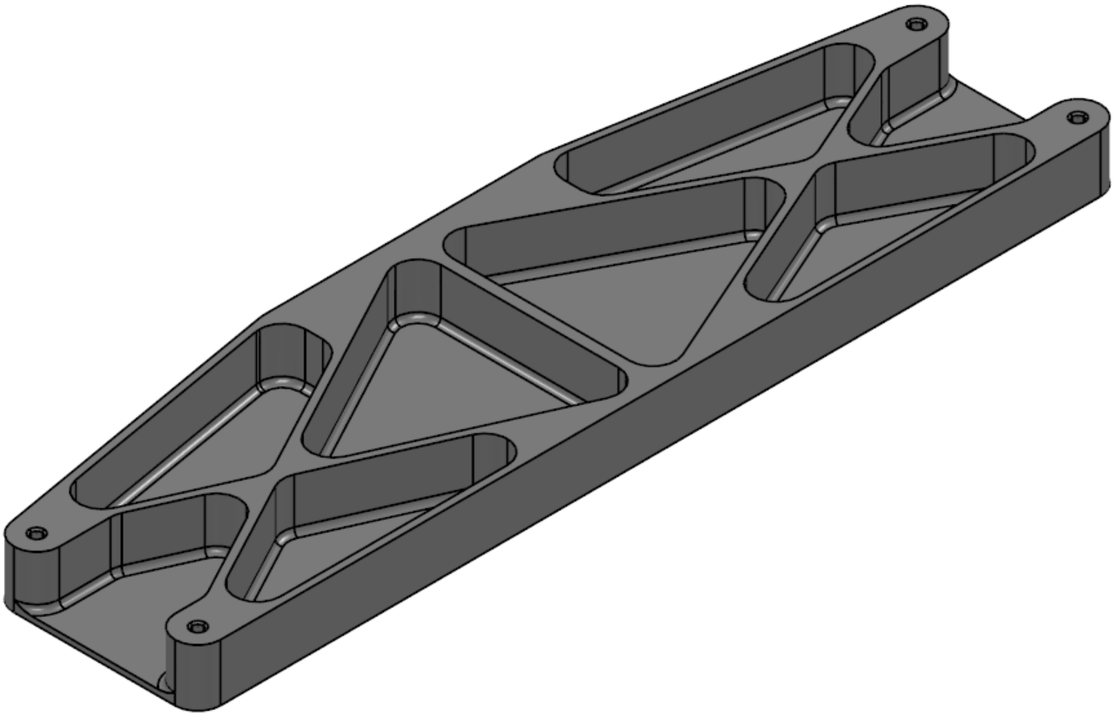
29 Select an **Isometric View** (**Ctrl+1**).

30 From the **Mesh Edit** toolbar select **Reverse Mesh**  to reverse all of the triangles of the mesh.

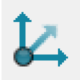
12. Reverse Engineering
Bloodhound Steering Support

Introduction

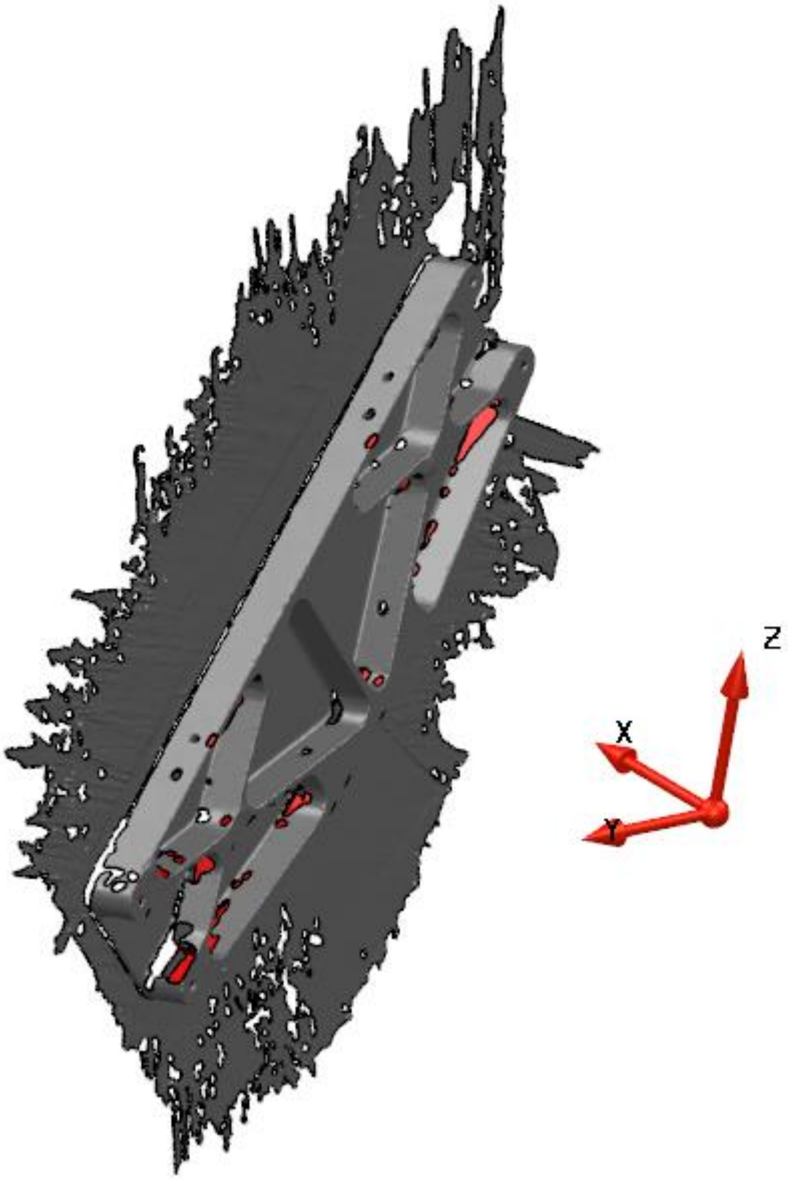
In this **tutorial** we will look at the many tools available within **PowerSHAPE Pro 2015** introduced previously for the purpose of, and to aid with the **Reverse Engineering process** of a **part** from the **steering support assembly** used in the **Bloodhound SSC**.



1 **Open** the **Bloodhound_steering_column.psmodel** from the **PowerSHAPE Pro Data** folder.

2 **Create a Single Workplane**  at **0** by typing **0** in the **command box** and then press the **Enter** key.

*This allows us to **visualise** where the **origin** of the **World Coordinate System** is **located** in **comparison** to the **mesh**. This is important as our first step will be to try and realign the mesh with this **coordinate system**.*



You should hopefully see that the **mesh** needs to **change location** and **orientation** to **realign** with the **coordinate system**. If in any future session your **mesh** and **workplane** are **far apart** you can **select** and **drag** the **mesh** closer, and this will help you to **align** the part **quicker** and **easier**.

Reverse Engineering Notes

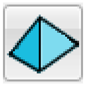
At this point it is **useful** to look at the **mesh**, the **physical part** and **any other information** as a whole and make notes and **plan** the **process** to take while **Reverse Engineering** the **part**.

Aligning the Mesh

To aid the **Reverse Engineering** process and make the best use of the **mesh tools** available within **PowerSHAPE Pro 2015** the **mesh** should be **aligned** to a **workplane X**, **Y** and **Z axes**, in most cases the most relevant will be the **World workplane**.

Correctly **aligning** the **mesh** is of importance to the entire Reverse Engineering process and ideally we should you the **largest area** and most **simple shape** to give the **best** and **easiest results**. In doing this the **Mesh Segmentation** tools will then have the ability to automatically adjust **primitive shapes** to the **planes** of the **active workplane**.

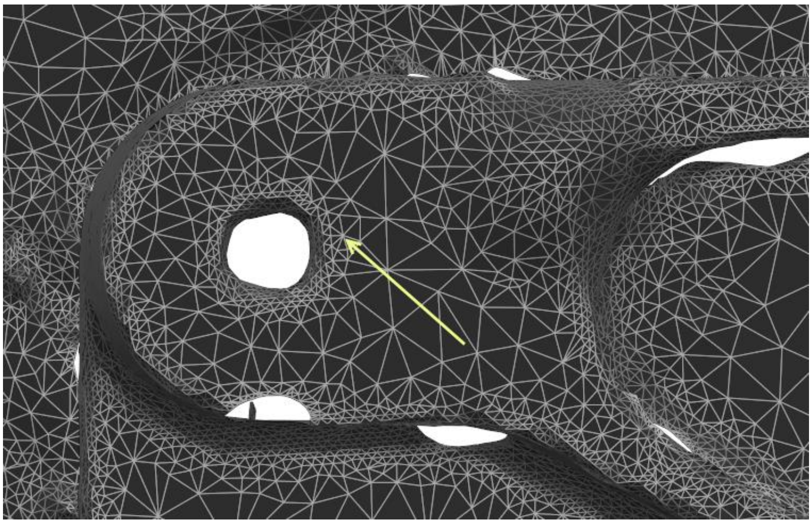
Although this isn't always possible it is best to **align** using the **largest** and **most basic shape** on the **mesh**. This will help to **reduce** and **average** any slight **errors** in the **alignment** over the whole model, preparing for the **Best-fit alignment**.

3 From the **Mesh Shading** options on the **View/Shading** menu on the **right hand side toolbar** of the **graphics area** select the option to show **Open and Interior edges, shaded triangles**. 

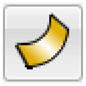
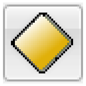
*This will show the **edges** of the **inner triangles** allowing **better clarity** when **selecting** areas **inside** the **mesh**.*

Rotating the **view** and looking at the part from above shows a clear **pattern** of **4 Holes** which make a **rectangular shape** which we could use to **align** to a **Surface Plane**. Although we cannot use the **actual holes**, having to **select points** on the **mesh**, we could use locations **close**, or **next to** these **features**.

In this example I am going to use **memorable locations** to the **inside edge** of the **holes** as shown in the **image below**. Showing the **interior edges** will help this **greatly** as we have to **select each location** multiple **times**.



4 Ensure the **Principal Axis** is set to **Z**.

5 From the **Surface**  menu select **Surface Plane**. 

6 Centre the **Surface** on the **active workplane origin** by typing **0** in the **command box** and then press the **Enter** key.

7 **Double-click** on the **surface** to raise the **surface edit** dialog.

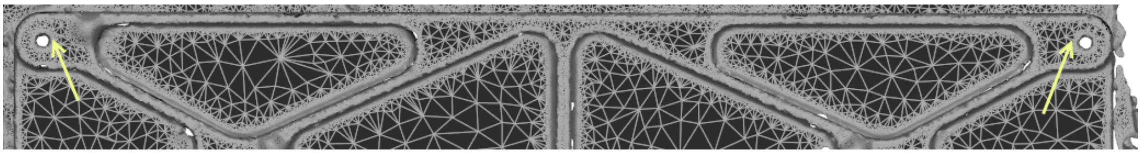
8 **Rotate** and **Zoom** the **View** to look down on the **top face** of the **part** in the **graphics area**.

As before, to **resize** the **surface** using the **measuring tool** directly we can **right-click** inside the **text box** adjacent to the **width** or **height** in the dialog**.** This will then open the **calculator** and allow **measurement** of the **correct length** and **automatically fill** the **respective box**.

9 **Right-click** in the **text box adjacent** to the **Width**.

*This will raise the **Measure** tool dialog and allow you to quickly **measure objects** inside the **graphics area**.*

10 **Measure** the **Width** (**X**) by clicking on the **position shown** on the **previous image** and its **counterpart** at the other end of the part to the **inside** of the **hole**, shown **below**.

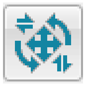


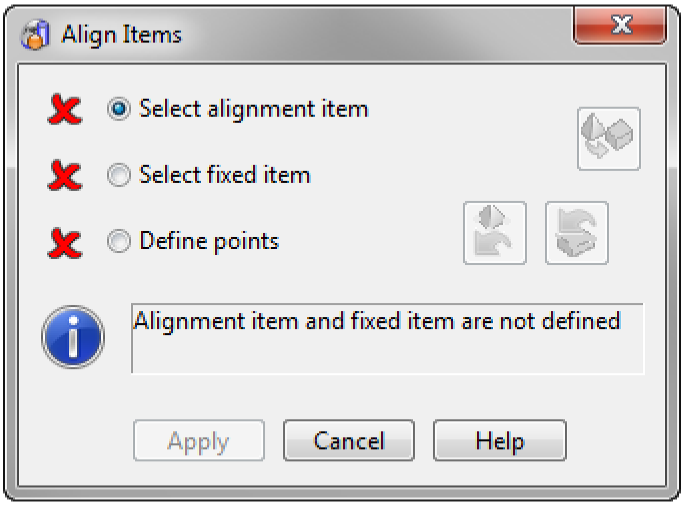
*Try to remember which **corner triangles** you selected as clicking these exactly again during the **alignment phase** will produce the **best result**.*

11 Click **OK** in the **Calculator** to use the **measured value** as the **Width**.

12 **Repeat** the same **process** for the **Length** of the same face to fully **resize** the **planar surface** to **fit**.

*If you're **values** are **slightly different** to the **values** in the image above **do not worry**. Again, the **best alignment** will rely on selecting the **same positions** as you did to **resize** the **surface** when **aligning**.*

13 Select **Align Items**  from the **General Edits**  menu.



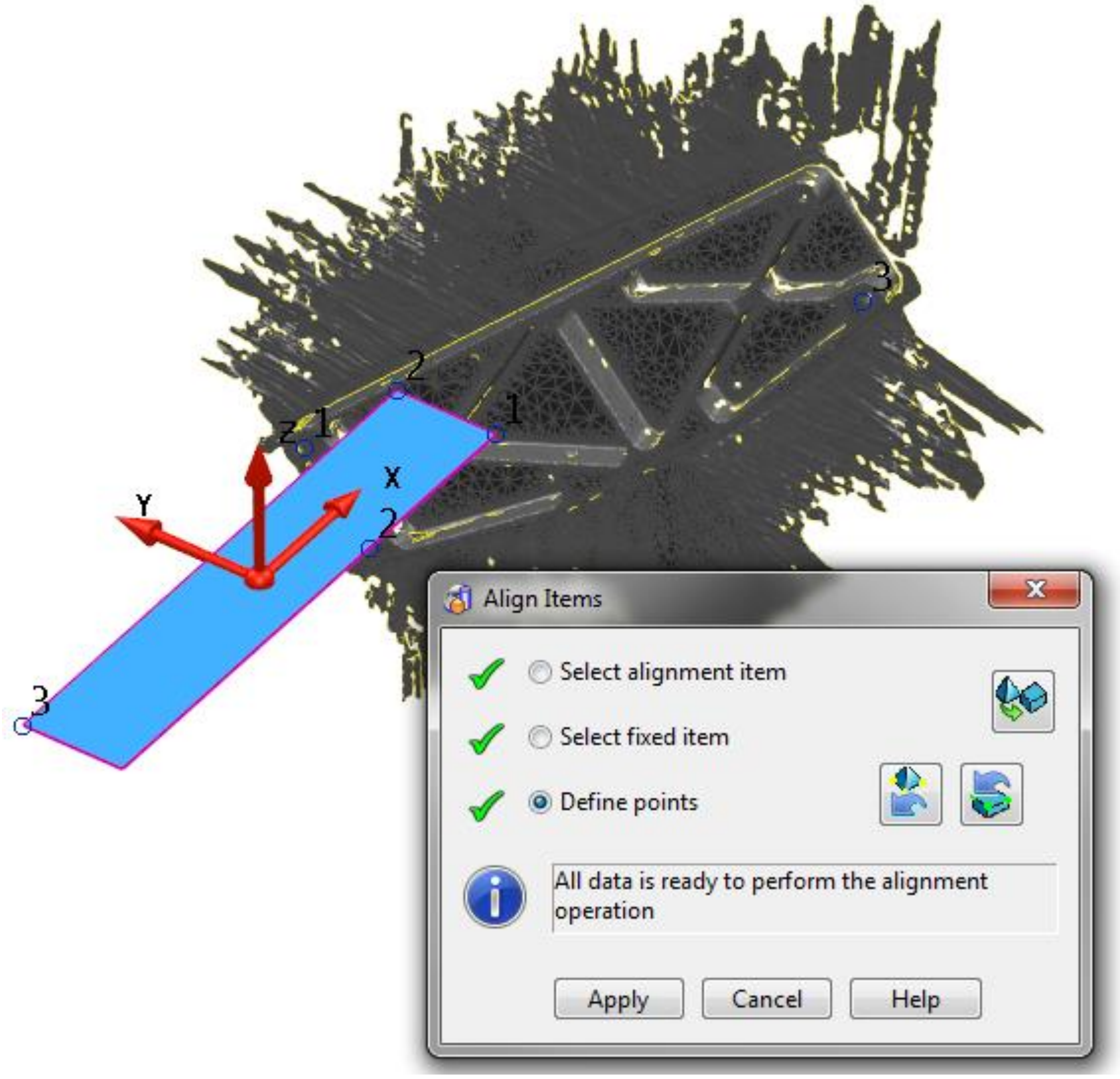
*If the **alignment** and **fixed objects** are far apart in the **graphics area** this may cause problems while **navigating the view**. It is possible to simply **drag** move the **alignment object**, in this case the **mesh closer** to the **surface**.*

14 In the **graphics area** select the **mesh** as the **Alignment Item**, then select the **surface plane** as the **Fixed Item**.

Next, we have to select a **number of points** on the **alignment object**, and then repeat the **same selection** on the **reference object**. Note that the **order** and **amount** of the points selected must be the same for each object. For a **rectangle** a good selection order is **"clockwise around width first then the length"** this way you know that the **selection** will **always** be the **same**.

15 Select the **3 points** around the face of the **Alignment Object**, one near **each hole** as you selected to **measure** the **surface earlier**, and then repeat the selection on the **Reference Object**, the **surface**.

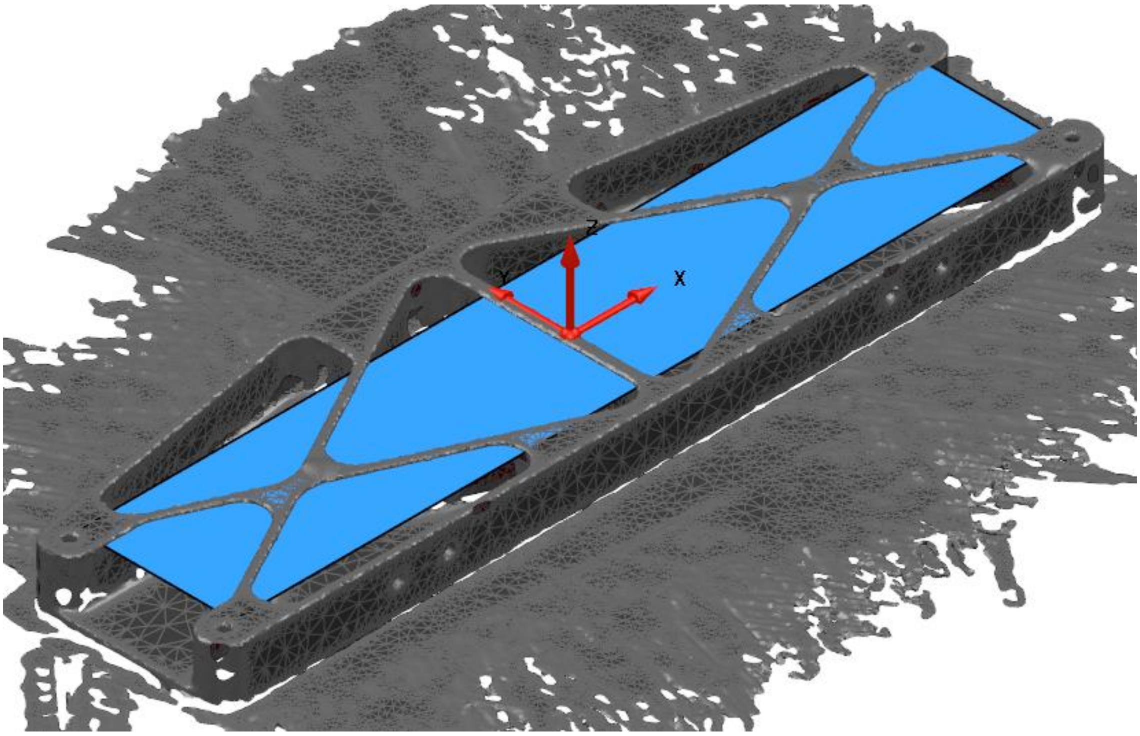
*Note: The **order** and **direction** of your **selected points** must be the **same** for **each object**.*



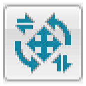
16 When your **selection** is **complete** and **correct** all the **3 red crosses** will become **green ticks** and the dialog will tell you that <u>you are ready</u> to perform the **alignment operation**.

17 Select **Apply** and **Cancel** to **close** the dialog and **view the results**.

18 Select an **Isometric View** (**Ctrl+1**).

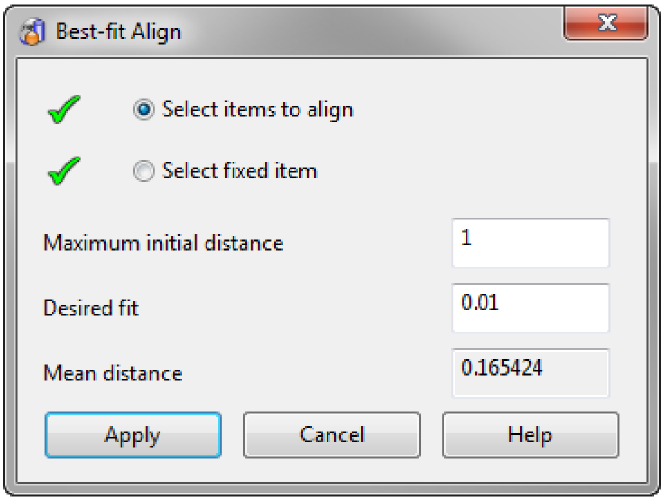


The **mesh** has been **aligned** with the **surface plane** and due to its **location**, the **world coordinate system**. Further to this we can **fine tune** our **alignment** using the **best fit alignment**  tool from the same menu.

19 From the **General Edits**  menu, select **Best Fit Items.** 

20 Again, select the **mesh** as the **item** to **align** and the **surface** as the **fixed item**.

21 **Edit** the **desired fit** to **0.01mm** and then select **Apply**.



22 Press **Cancel** to **close** the **dialog**.

*Using the **View from**.... toolbar to the right of the **graphics area** or using the **keyboard shortcuts** will allow you to **visually inspect** the **alignment**.*

23 Create a **New Level** on **Level 997** called "**Alignment Surface**".

*Here we have used such a **high number** as this is data we need to keep but are **unlikely to need** again **unless** we have an **issue**. Therefore it will always be at the **end** of the **levels toolbar**.*

24 In the **graphics area** select the **surface** we used to **align** the **mesh**, and place it on **level 997** using the **middle mouse button**.

25 Ensure **Level 997** is **switched OFF**.

Cleaning up the Mesh Data

Next we will remove any **disconnected areas** of the **mesh** in an effort to begin to clean up the **mesh** and **remove unwanted data** to use the **graphics** most **efficiently**. This will also involved removing the areas of data were the **workbench/fixture underneath** has been **captured** also.

*Note: A **small portion** of the **scanned workbench/fixture** will be **useful** to provide us with an **overall depth** of the **part**.*

26 Select the **mesh** to **activate** the **Mesh Edit** toolbar.

27 Choose to **Divide the Mesh into Multiple Meshes**.

28 Select and **Blank** (**Ctrl+J**) the **main mesh**.

*You may have noticed the **mesh** has been **split** into **3 separate mesh** by any **disconnected outlying regions**. By **blanking** the **main mesh** we can remove the **extra disconnected regions** easily.*

29 **Quick Select ALL Mesh** from the **Selection** flyout menu.

30 **Delete** the **selected objects.**

31 **Unblank** (**Ctrrl+L**) the **mesh**.

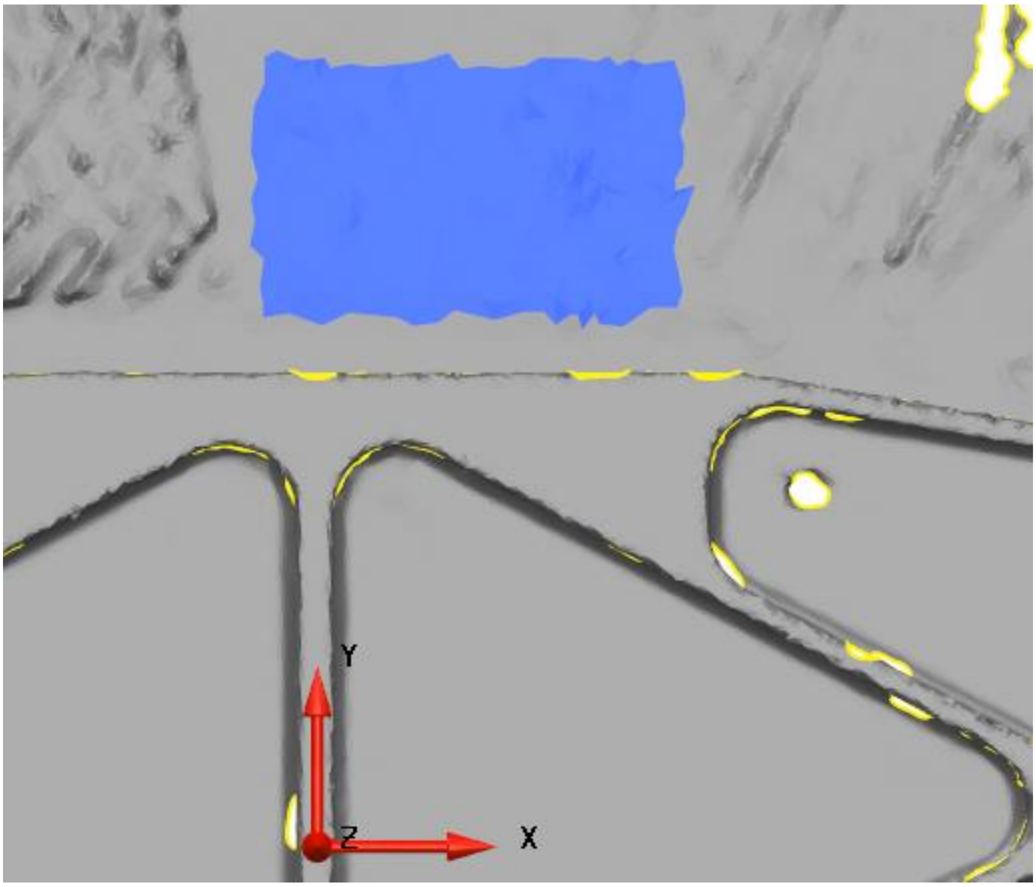
32 Take a **View from Top** (**Ctrl+5**).

*We are going to **select and divide** a **small area** of the **data captured** of the **workbench** to allow us to to create a **bottom face** for the **part**.*

33 From the **Mesh Edit** toolbar select **Pick Area of Triangles by Box**.

34 **Drag select** an **area of triangles** from consistent **region of triangles** representing the **workbench**.

*These **selected triangles** will become **highlighted blue** inside the **graphics area**.*

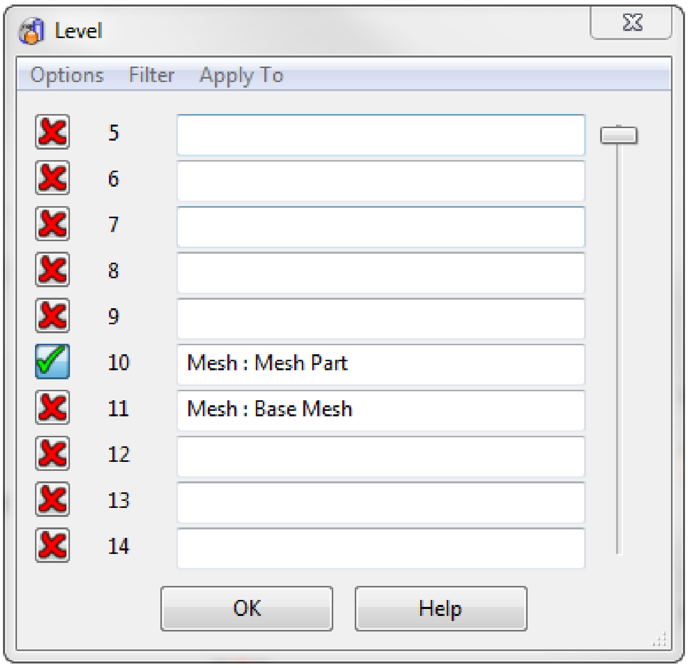


35 **Divide the Mesh by Selection**  located on the **Mesh Edit** toolbar.

*This will **divide the mesh** into the **main mesh** and a **small area** consisting of the **triangles selected***

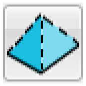
36 Create a **New Group of Levels** called **Mesh** to help us to further **organise** our data.

- **Level 10 – Mesh : Mesh Part**
- **Level 11 – Mesh : Base Mesh**



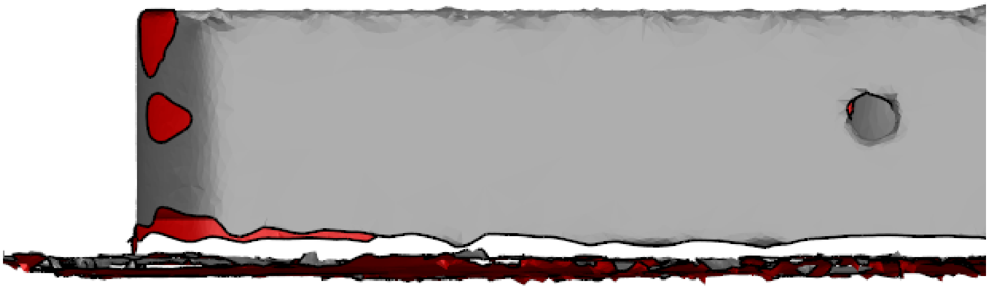
37 Select the **main mesh** and place it on **Level 10**, then ensure the **level** is **turned ON**.

38 Select the **small mesh square** and place it on **Level 11**, then ensure the **level** is **turned OFF**.

39 Select to draw **Open Edges Only**  from the **mesh shading options** to the right of the **graphics area**.

40 Take a **View from Front** (**Ctrl+2**).

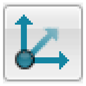
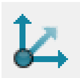
41 **Zoom** into the **left of the part**, as shown below.



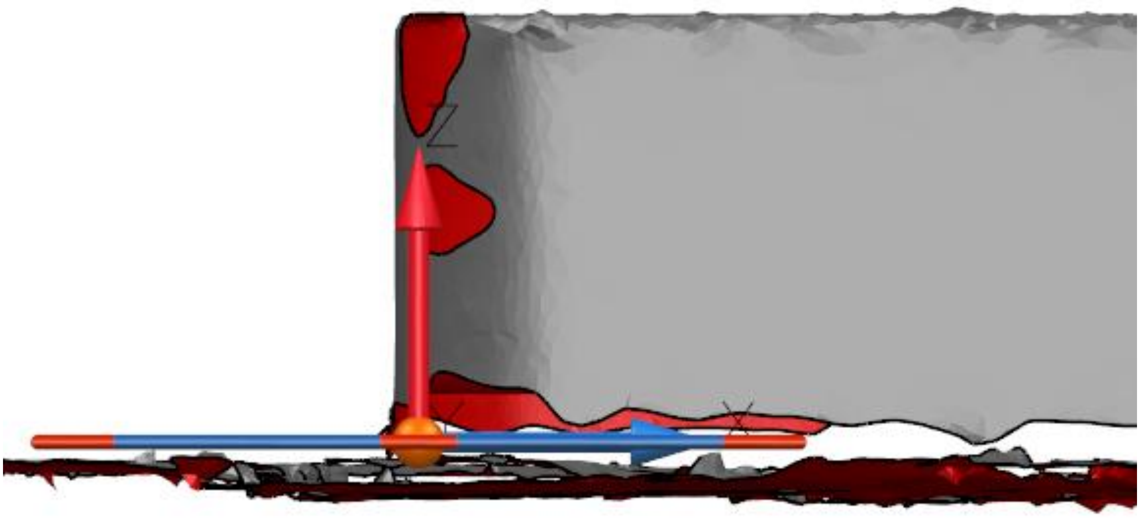
Removing any **unwanted data** in the **mesh** will **reduce** the **overall mesh size** and **complexity** and **speed** up the **software graphics**. We will do this using **limit selection** from a **new workplane**, and as we already have **isolated** a **small section** to provide the **overall part depth** and placed it on a **separate level**, we can **remove** the rest of the **extra data**.

42 Ensure the **Principal Axis** is set to **Z**.

*This will make sure that the **new workplane** we will create is in the same **orientation** as the **current active workplane**.*

43 Select to **Create a Single Workplane**  form the **Workplane**  menu.

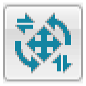
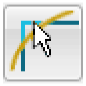
44 Create the **new workplane** in the **location shown** below. This will allow us to use the **limit selection function** to trim the **excess data** across the **principal plane**.



45 Ensure the **Principal Axis** of the **new workplane** is set to **Z**.

46 Select the **workplane** in the **graphics area**.

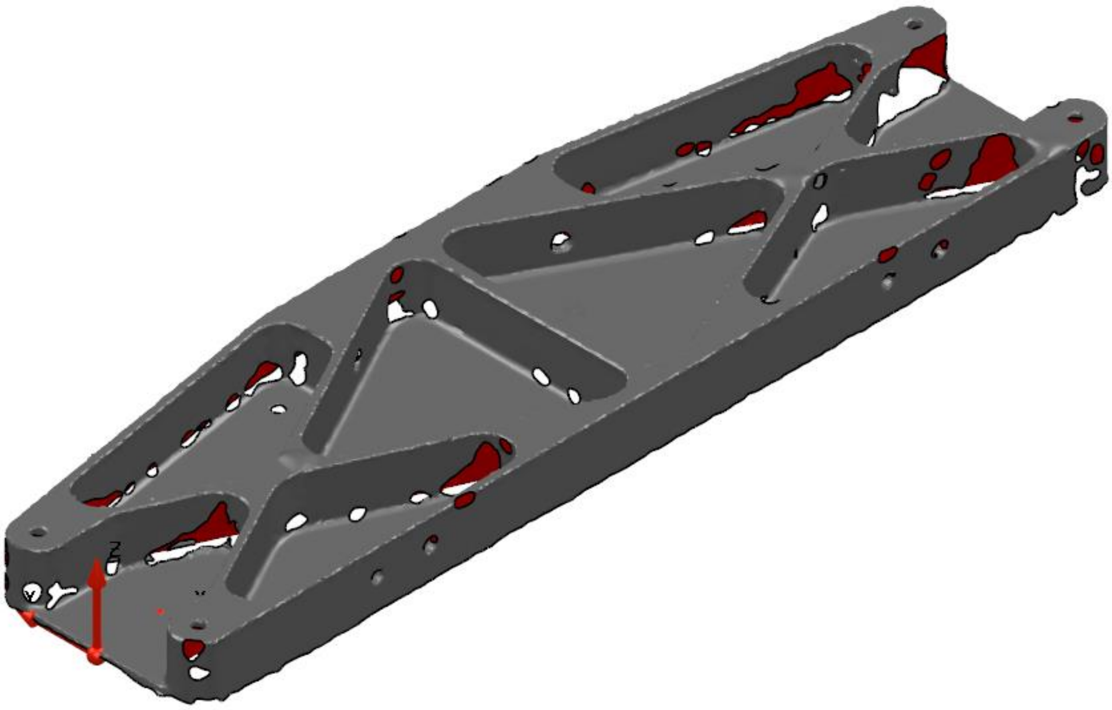
*This will show the **workplane manipulation handles**.*

47 From the **General Edits**  select the **Limit Selection** tool. 

*As the **Principal Axis** is set to **Z** and the **workplane** is selected the **cutting object** has **automatically** become the **XY plane** of the **workplane**.*

48 Select the **area** of the **mesh below** the **workplane** (**cutting object**).

49 Take an **Isometric View** (**Ctrl+1**).



We have **successfully removed** all the **excess data** that we do not require from the **fixture underneath the part** during the scanning process. However, we do still have the small section kept on **Level 11 : Base Mesh** in order to create a **base surface** to give the **overall depth** of the **part**.

Now, we will begin to **Reverse Engineer** the part using **PowerSHAPE Pro**. As with the previous example we will use a **variety** of **methods** to **best suit each step** in the **process**. **Working methodically**, initially we will aim to create the **main solid** and then **add/remove** the **features**.

Reverse Engineering the Part

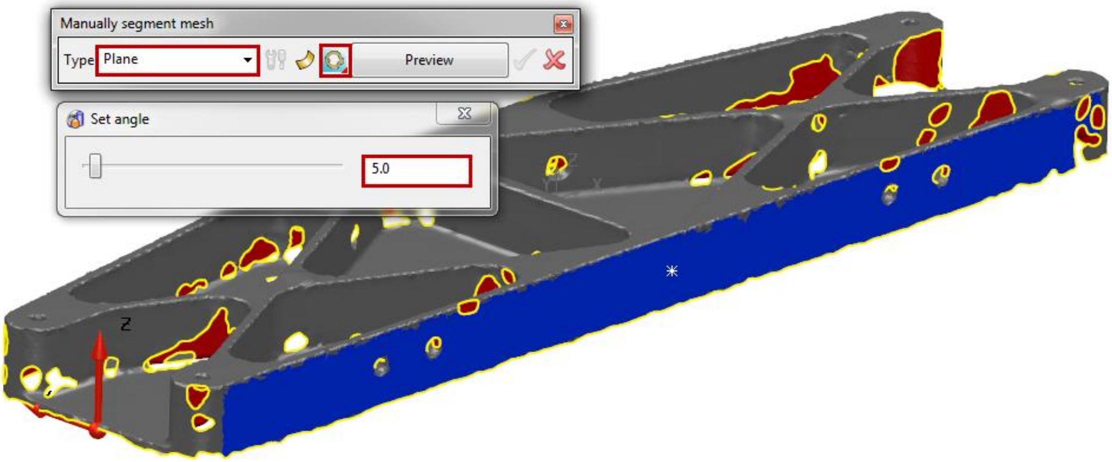
1 Select the **mesh** in the **graphics area** to **activate** the **Mesh Edit** toolbar.

2 From the **Mesh Edit** toolbar select to **Manually Segment mesh**. 

3 In the **manually segment mesh** dialog select to fit **Plane primitive surfaces**, **fit through the middle** of the **mesh**.

4 Next, in the **Set Angle** dialog (**Local Horizon**) set the **angle** to **5°** and select the **longest front face** of the **part** as shown in the **below image**.

*This **5° value** was found using the **slider** in the **dialog** as the **minimum value** to **select triangles** the **full length** of the **face**.*



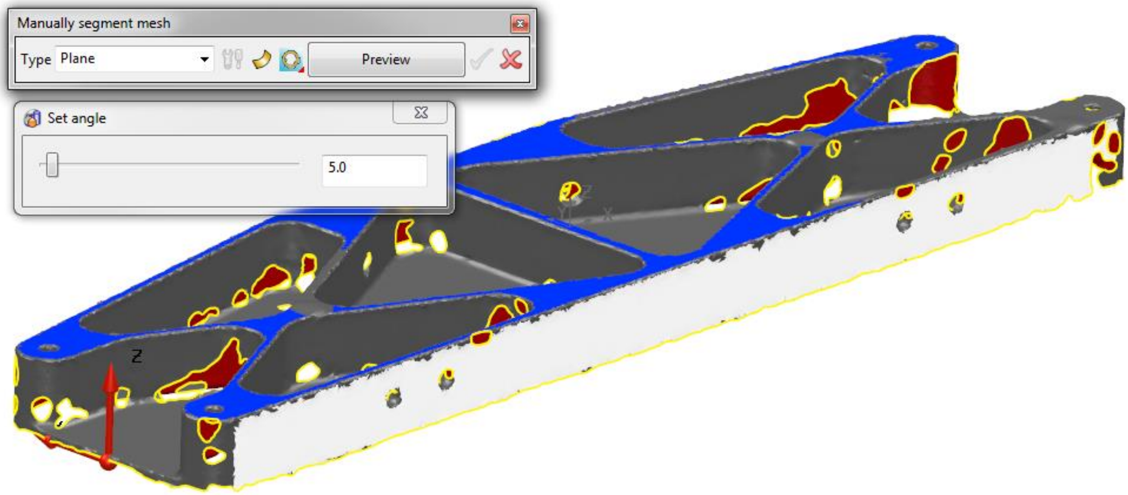
5 Select **Preview** and then **select** the **green tick** to **create** the **Plane primitive**.

***Do not** close the **dialog**. After the **surface** is created it will not be **visible** while inside the **manually segment mesh** dialog to **avoid obstructing the view**, however the **triangles** used will be **highlighted** in **white**.*

6 Using the same **values**, **select** the **triangles** on the **top face** of the **mesh**.

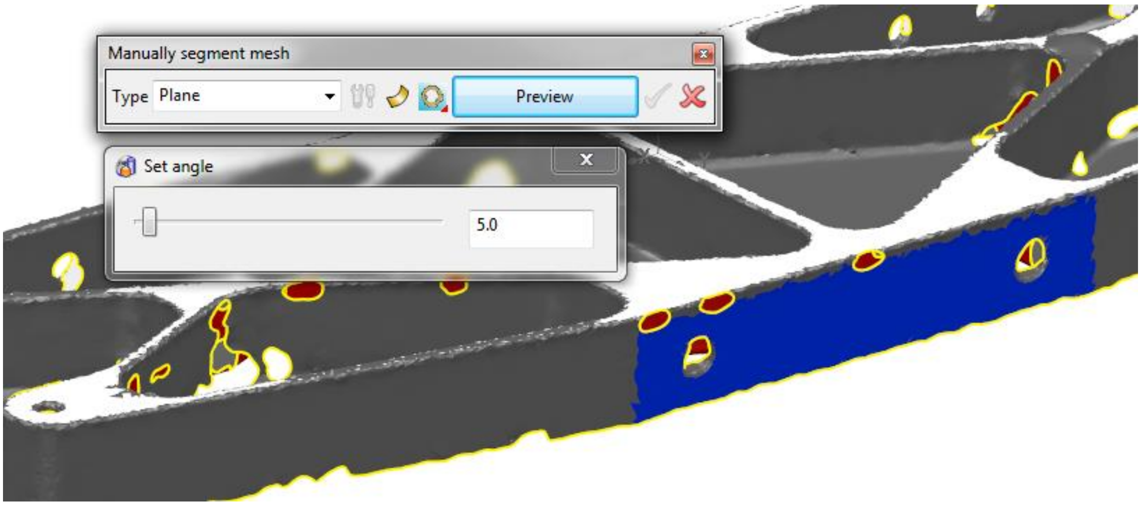
7 Select **Preview** and the **green tick** to create the **surface**. **Do not close** the **dialog**.

*This may require **multiple selections** on the **top face** to select the **entire area**. Using the **Ctrl+Click** will enable you to make **multiple selections**.*



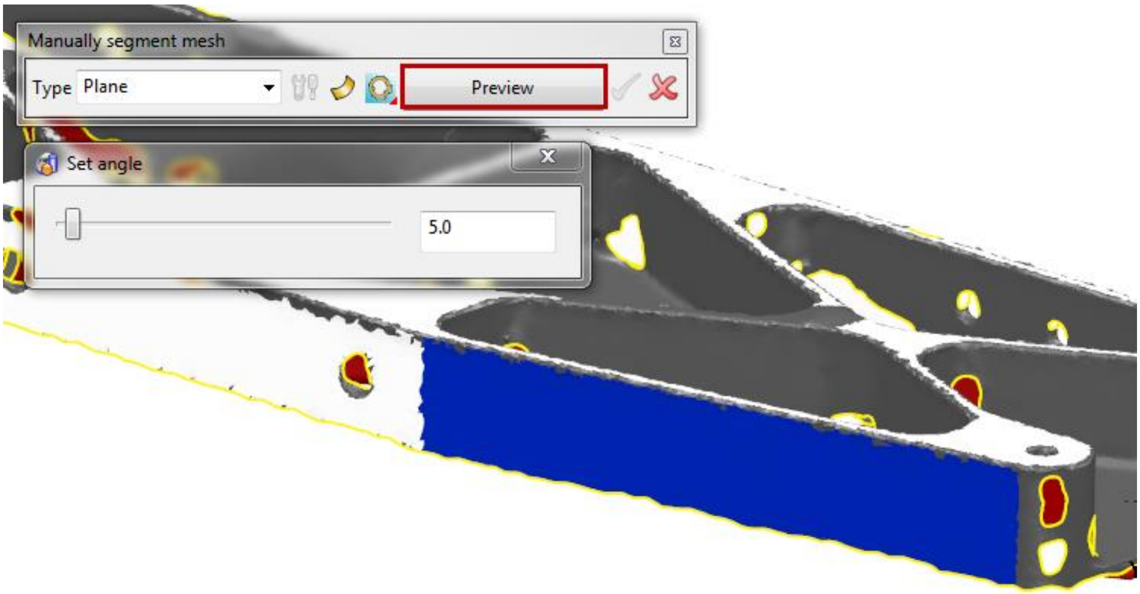
8 **Rotate** the **part** around to enable us to see and select the **rear face(s)** of the **mesh** to **create** the **required surfaces**.

9 This time select the **centre-face** of the **rear** of the **mesh** that will be **parallel** with the **front face** as shown in the **next image**.



10 Select **Preview** and then the **green tick** to create the **surface**.

11 Next **select** the triangles on **one** of the two other **adjoining faces** using the **same values** as before. We will **only require one of the two** as the **part** is **symmetrical**.



12 Select **Preview** and then the **green tick** to create the **surface**.

*We can later find the **axis of symmetry** in the part and **Mirror** this **surface** to quickly create an **exact matching face**.*

13 Zoom into one of the **open sides** of the **part**.

14 **Next**, we need to **fit** a **surface** onto one of the **side faces** of the **mesh**.

*Again, due to the **symmetry** on the **part** we only **require one** which we can later **Mirror** across.*

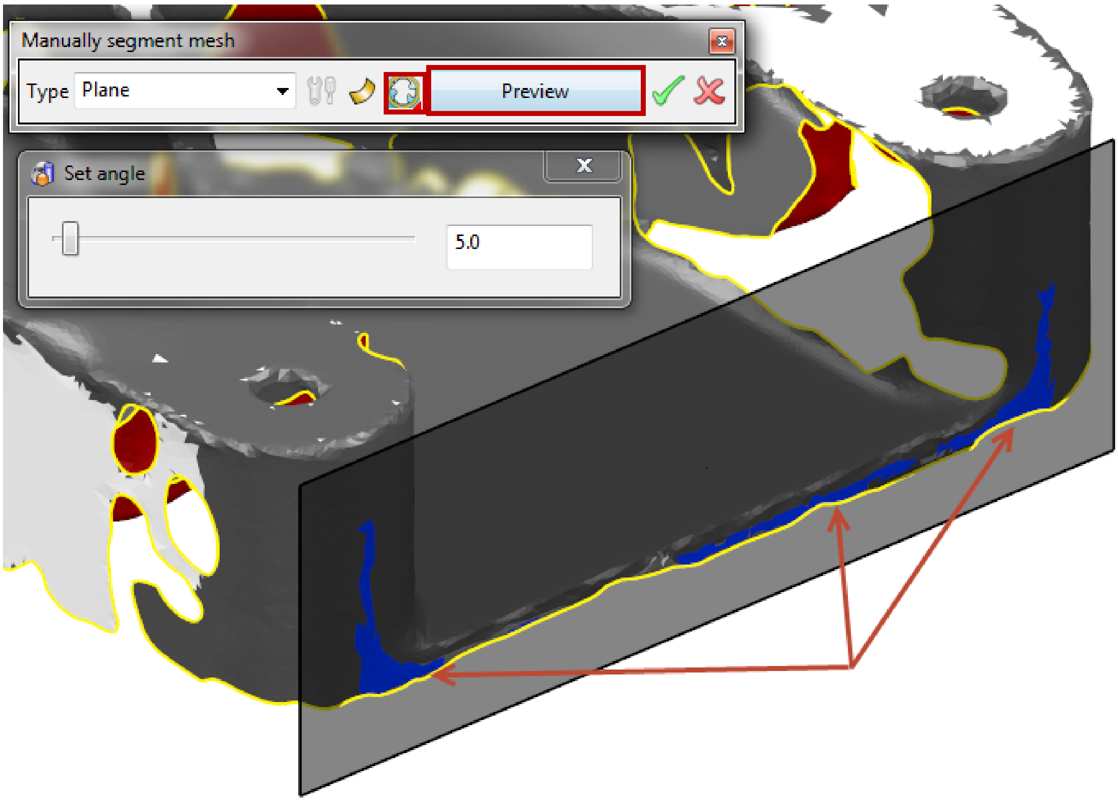
15 In the **dialog** choose **fit outside** as the **mesh fit**.

*This will **fit** the **plane primitive slightly outside** the **mesh** which will allow us to **fully round** the **edges** and create the **open pocket shape later**.*

16 Using the **Ctrl+Click multiple selection** select the **remaining small side edges** that represent the **side face**.

*As we **do not** have much data for this particular side it is best to use **as much** of the **limited data** as **possible**.*

17 Select **Preview** within the **dialog**.



18 Click the **green tick** to **fit** the **surface**.

*This is also a **surface** which we can **Mirror** across the **axis of symmetry** to create the **other end surface** as **required**.*

We now have **one further surface** we need to create using the **Manually Segment** mesh tool, the **bottom face**. We will create this using the **small mesh** we divided from the **fixture** earlier that is hidden on **Level 11 : Mesh:Base**.

To complete this we need to perform a **similar operation** with the **manually segment mesh** tool on the **other mesh**.

19 In the **dialog** select the **red cross** to **dismiss** the **form**.

*This will show all the **surfaces** we have just **created**.*

20 **Turn ON Level 11 : Mesh:Base**.

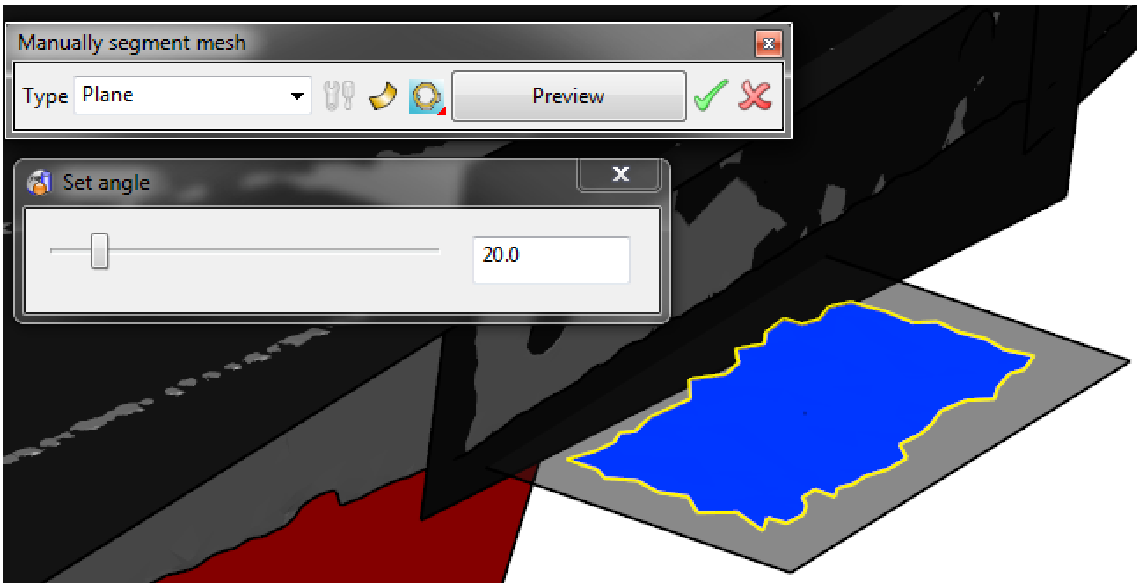
21 In the **graphics area** select the **small mesh** we have just **switched on**.

22 From the **Mesh Edit** toolbar select **Manually Segment mesh** tool. 

*We have had to **exit** out of this **tool** and **show then select** the **other mesh** as this tool will act on the **selected** **mesh**.*

23 Using the **default values** within the dialog **select the triangles** in the **mesh**.

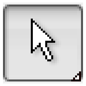
24 Select **Preview** and then the **green tick** to **accept** and create the **surface**. Then **close** the **dialog**.



25 **Turn OFF the Level Group : Mesh.** 

26 **Open** the **Levels** dialog.

27 **Create** a **New Level 20 : Model Surfaces**. Ensure it is **Switched OFF**.

28 Using the **Selection**  flyout menu, **Quick Select ALL Surfaces**. 

29 Using the **middle mouse button**, place the **surfaces** onto **Level 20 : Model Surfaces**.

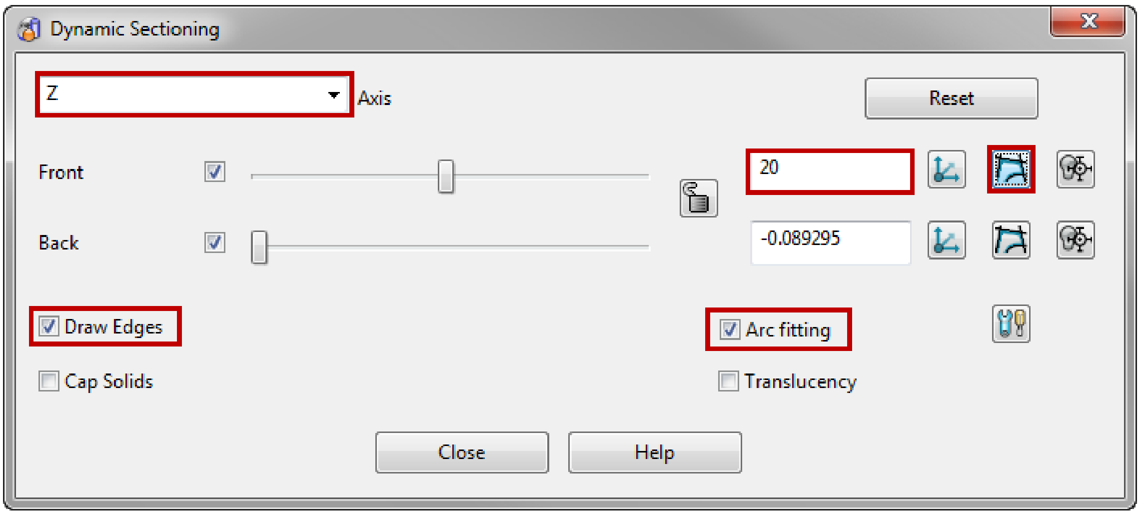
We are currently **missing two surfaces** that we need to **Mirror** across to complete the **main model block**. To do this we need to find the **centreline/axis of symmetry** of the part. To find this we will use **Dynamic Section** to create **section curves** in order to work out the **midpoint** of the **width**, which will be the **axis of symmetry**.

30 **Turn ON Level 10 : Mesh:Main**.

31 Select an **Isometric View** (**Ctrl+1**).

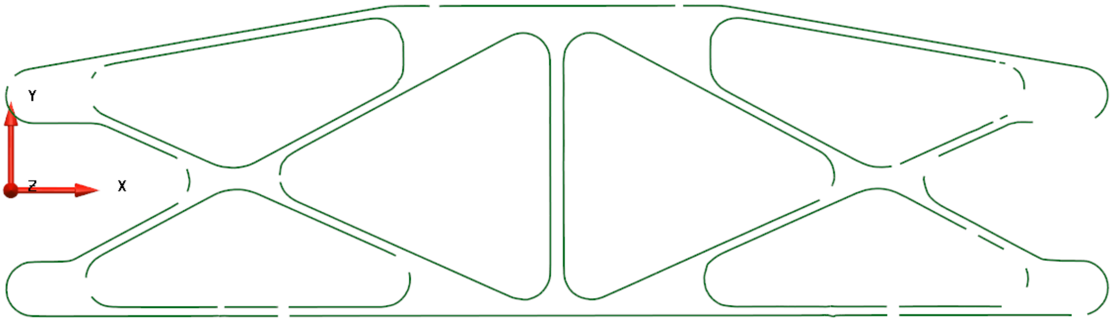
32 Navigate to **View>Dynamic Sectioning**.

33 Inside the dialog choose to take a **section** through the **Z Axis** and slide the **slider** to a **value** of **-20mm** and select to **create wireframe from section**. . Select the **Arc Fitting** tick box.



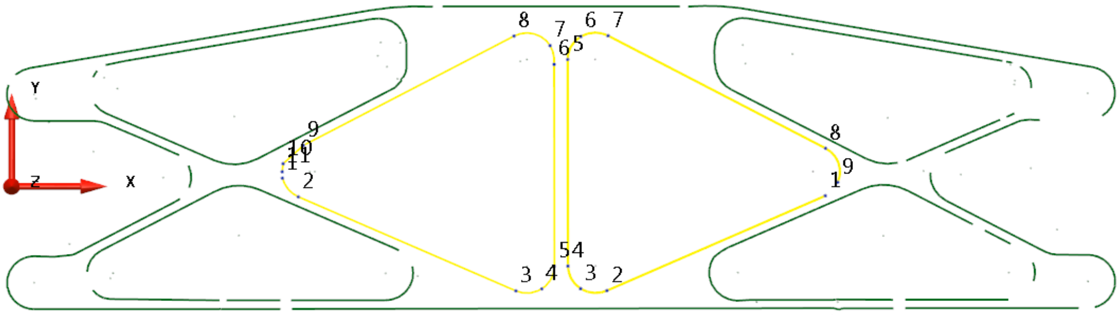
34 **Turn OFF Level 10 : Mesh:Main**.

35 Take a **View from Top** (**Ctrl+5**).



*Above shows an **image** of the **section curves** created using the **dynamic section** tool. To find the **axis of symmetry** from these curve we can use an **equivalent point** from **each side**, draw a **line** in between and use its **midpoint**. This method will be **described next**.*

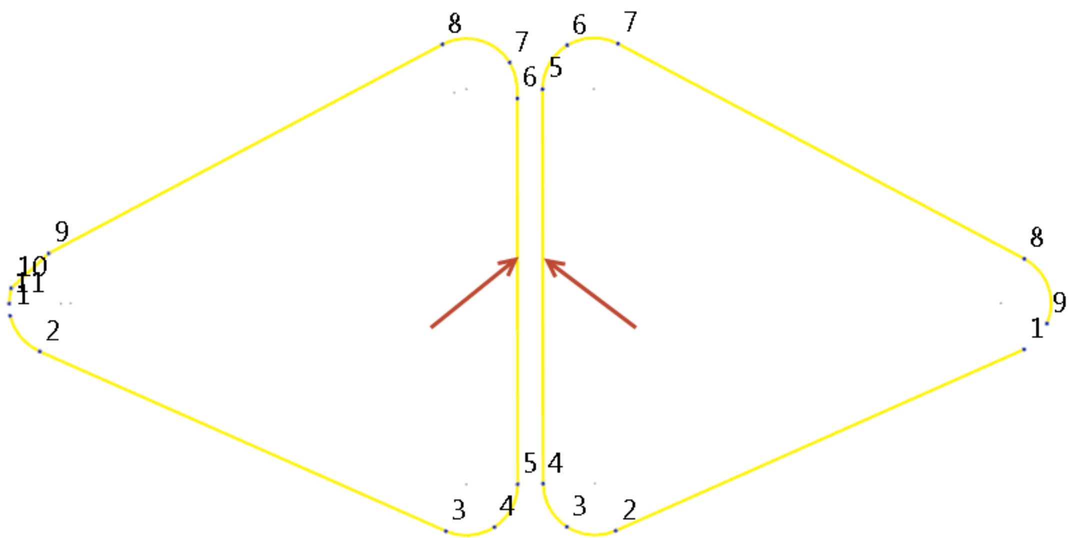
36 **Select** and **Blank** (**Ctrl+J**) the **two centre triangle feature shapes**.



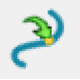
37 **Select ALL** (**Ctrl+A**) and **Delete** the other **section curves**.

38 **Unblank** (**Ctrl+L**) the **hidden curves**.

To find the **Midpoint** in between these **two curves**, first, we will **create a point** on the **straight area** of one of the curves using a **Parameter Value**, and then a **point closest** to this on the **other curve**.

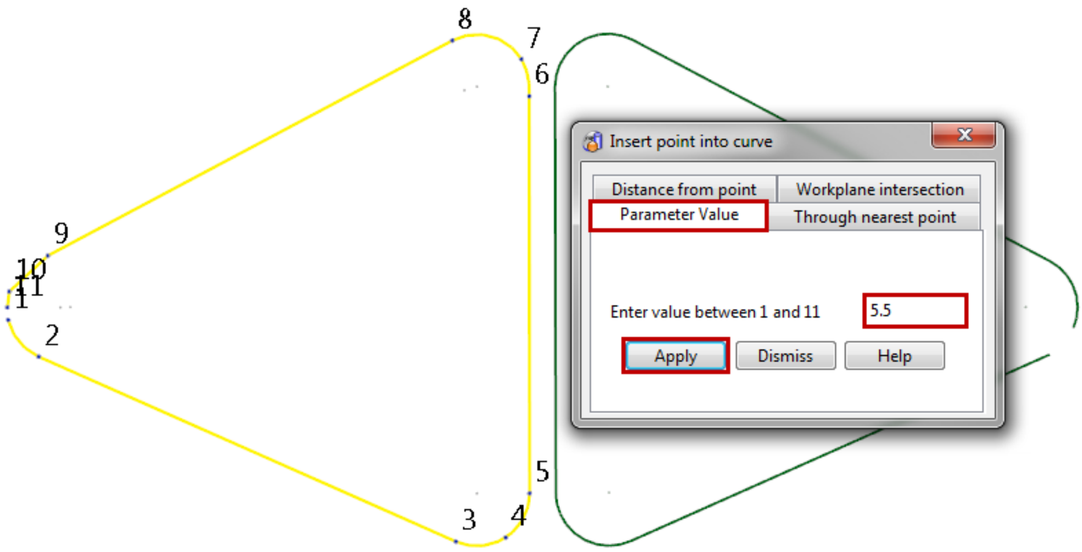


39 Select the **left hand curve**.

40 From the **Curve Edit** toolbar, ensure **Curve Numbering**  is **switched ON** select **Create a Point**. 

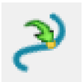
41 **Create a point** using the **Parameter Value** method **<u>approximately</u> halfway along** the **right hand straight edge**. Click **Apply**.

> ***In this case** a **value** of **5.5** was used, although this **may differ** in your **session**.*



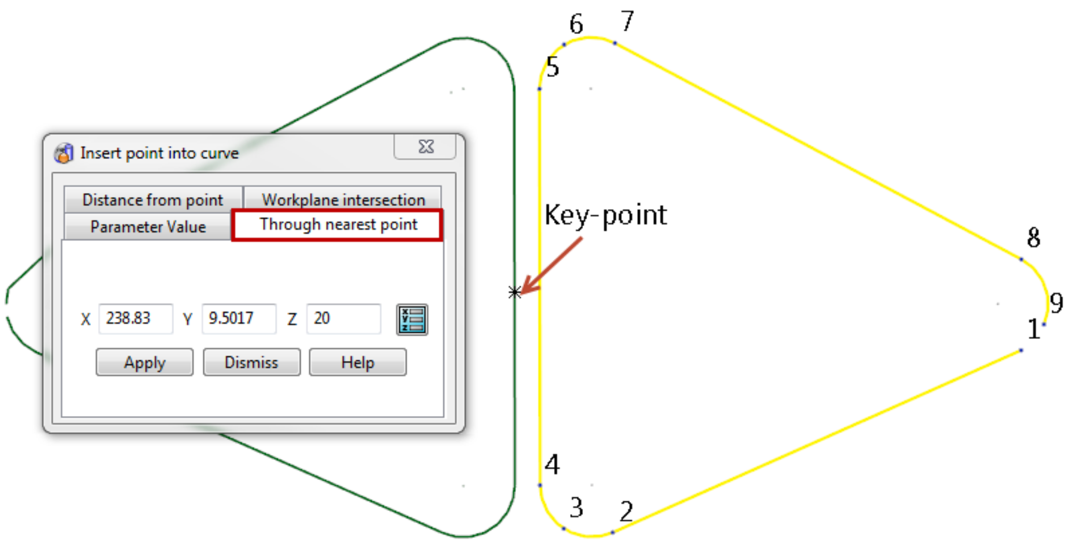
42 **Dismiss** the **dialog**.

43 Now **select** the **right hand section curve**.

44 From the **Curve Edit** toolbar select **Create a Point**. 

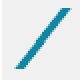
45 In the dialog select the **Through Nearest Point** tab and then in the **graphics area select** the **location** of the **point** we have just created (on the **left hand curve**).

> *This will **automatically fill** in the **X**, **Y** and **Z values** in the dialog and when you click **Apply** and **point closest to this location** will be created.*

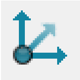
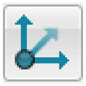


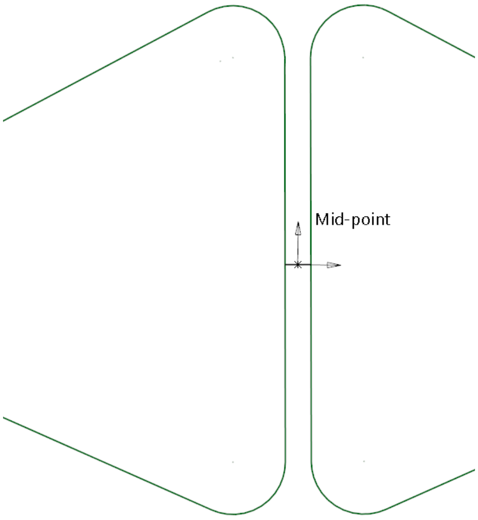
46 Click **Apply** to **create the point** and then **Dismiss** to **close** the dialog.

We know have to points, one on each curve, directly next to each other. The **midpoint** of a **line** between these **two points** will be on the **axis of symmetry**.

47 From the Line  menu, create a **Single Line**  joining the **two curve points** we have just **created**.

48 Ensure the **Principal Axis** is set to **Z**.

49 Next, from the **Workplane**  menu, **create a single workplane**  and **snap** it to the **Midpoint** of the line using the **Intelligent Cursor**.



This **new workplane** is now our **Active Workplane** and its **Y Axis** is the **axis of symmetry**.

50 **Quick Select ALL Wireframe**  from the **Selection** flyout menu.

51 Place the selected items on **Level 2 : Wireframe** and **Switch** the **Level OFF**.

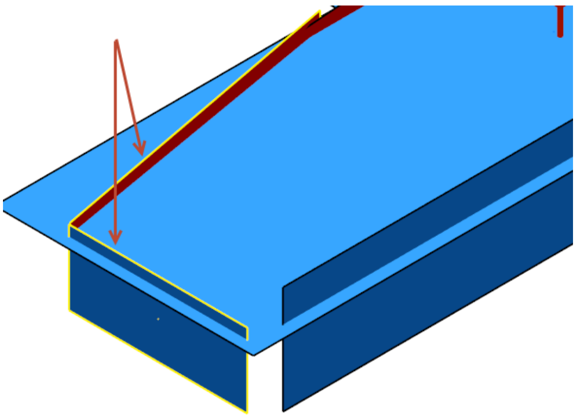
52 **Turn ON Level 20 : Model Surfaces**.

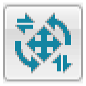
We will now **Mirror** the **two required surfaces** across the **YZ plane** of the **current active workplane**.

53 Ensure the **Principal Axis** is set to **X** (To use the **YZ plane**).

54 Select the **two surfaces** in the **graphics area**.

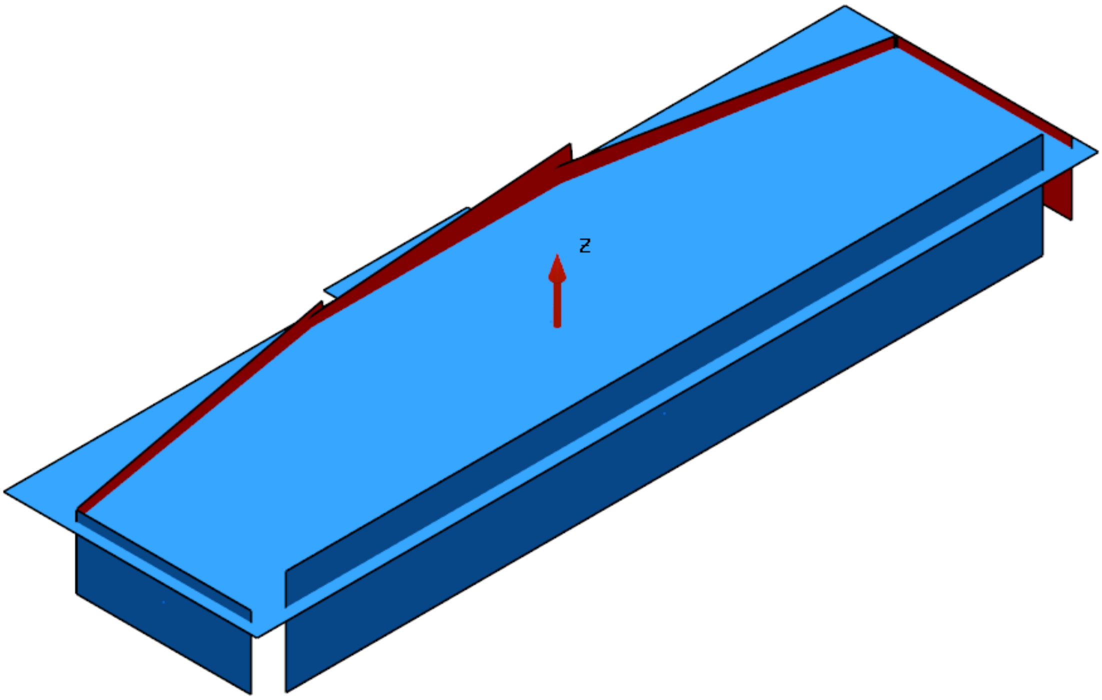
 *Holding the **Ctrl** key will allow you to **select multiple objects**.*



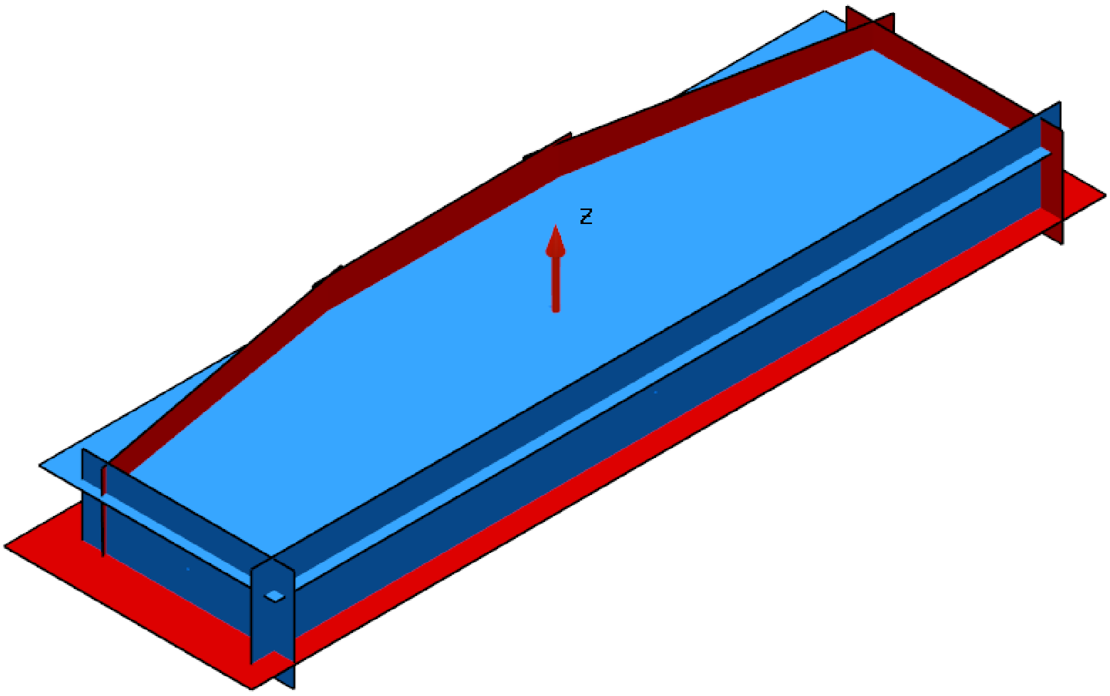
55 From the **General Edits**  menu select **Mirror objects**. 

56 Ensure **YZ plane** is **selected**, this will be **automatic** if the **Principal Axis** is **X**.

57 Select the **green tick** to **mirror** the **objects**.

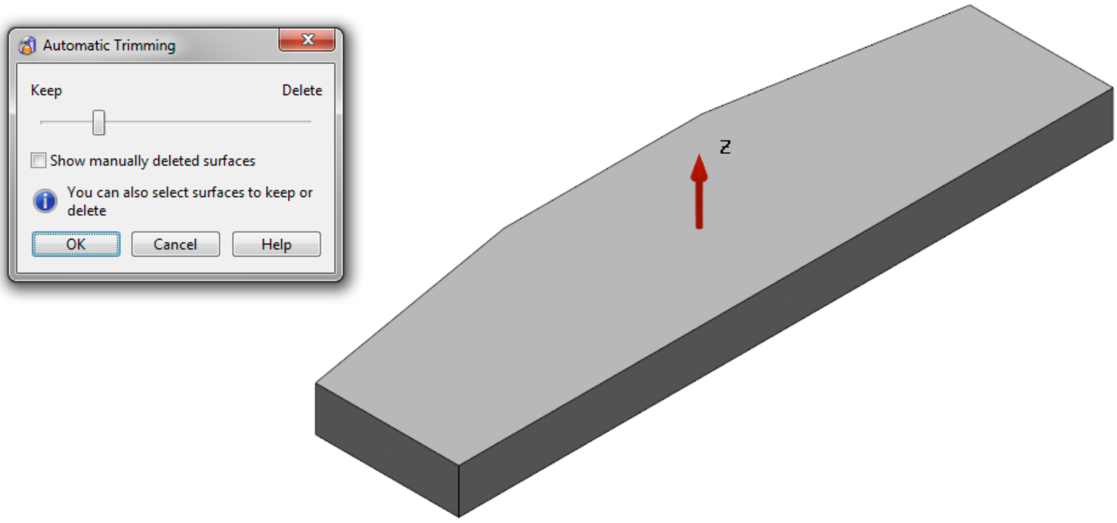


58 Using **Primitive Surface editing** techniques described in the Surfaces chapter, edit all the surfaces to **ensure adjoining surfaces overlap** to create an **enclosed internal cavity**, and **each surface** is **orientated** with the **positive** (**blue**) **side facing outwards** in preparation for the **Solid from Untrimmed Surfaces** tool.



59 From the **Selection** flyout **Quick Select ALL Surfaces**. 

60 Next, select **Create a Solid by Automatically Trimming Surfaces**  from the **Solid**  menu.



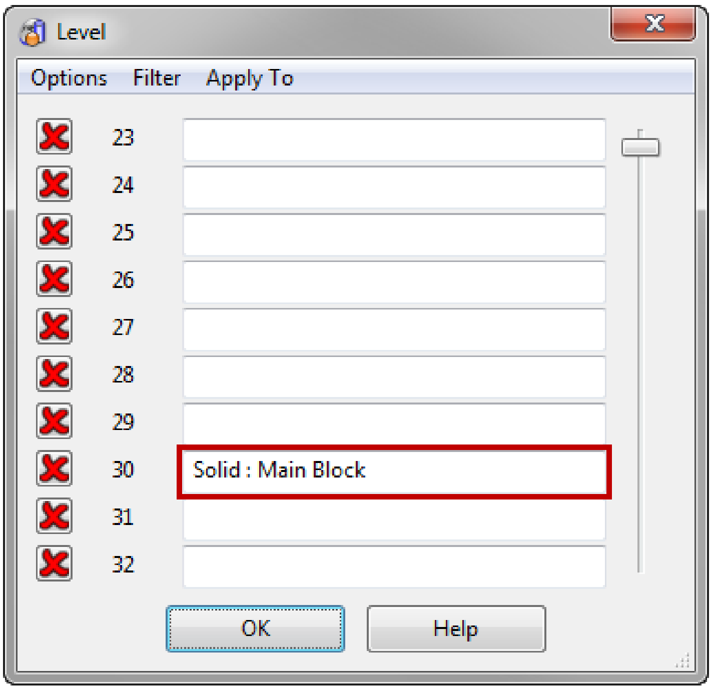
61 **Check** to make sure the **solid** is **complete** and **closed** by **rotating** the **view**. Click **OK** in the dialog to create the **solid**.

We now have a created a **solid** forming the **main block** of the part. From this and the **mesh** we can now start to **create** and **remove** the **features**. First we will create a **new solid level** and **group** from the **block**.

62 Raise the **Levels** dialog.

63 Create a **New Level 30 : Solid:Main Block**.

This will create a ***level*** *called* ***main block*** *and a* ***group named solid****. This will help later when we create and use* ***further solids****.*



64 Select the **solid** in the **graphics area** and **place** it on **Level 30 : Solid:Main Block**.

65 Ensure **Level 30 : Solid:Main is turned ON**.

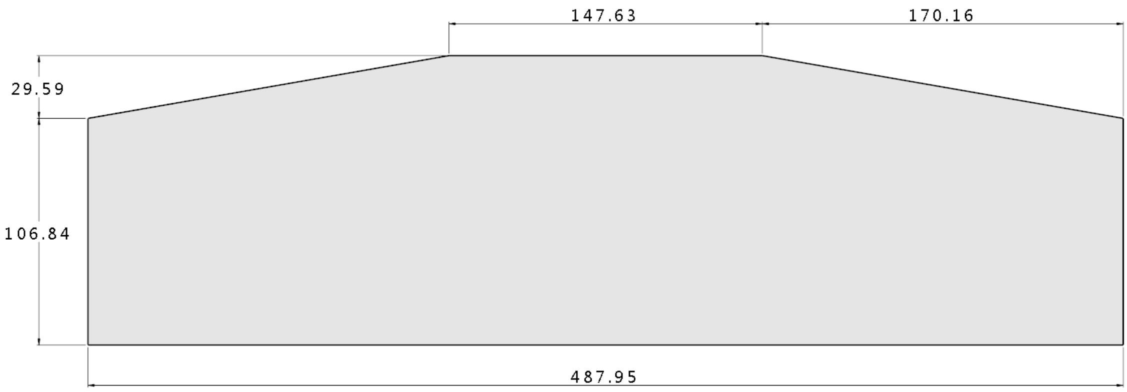
66 Select a **View from Top** (**Ctrl+5**).

The **above image** shows the **top down view** of the **main block** before any features have been created. Before creating the features we will **refine** the **dimensions** of the **block** to fully **Reverse Engineer** the **part**. For this **stage** the more data we have including the **physical part** and the **desgn intent** of the **part** and **each feature** will ease this **process**. **Scanning** a part from **raw data** through to **solid model** may introduce **minor deviations** in **size** and **location**, and this process will allow us to **react** to this.

To do this we will initially **dimension** the **model** and then use **Direct Solid Modelling** techniques to make the **minor changes required** to the **solid model**.

67 Open the **Annotation**  menu and then select the **Automatic Dimension** tool. 

68 **Dimension** the **Top Face** as **shown below**.



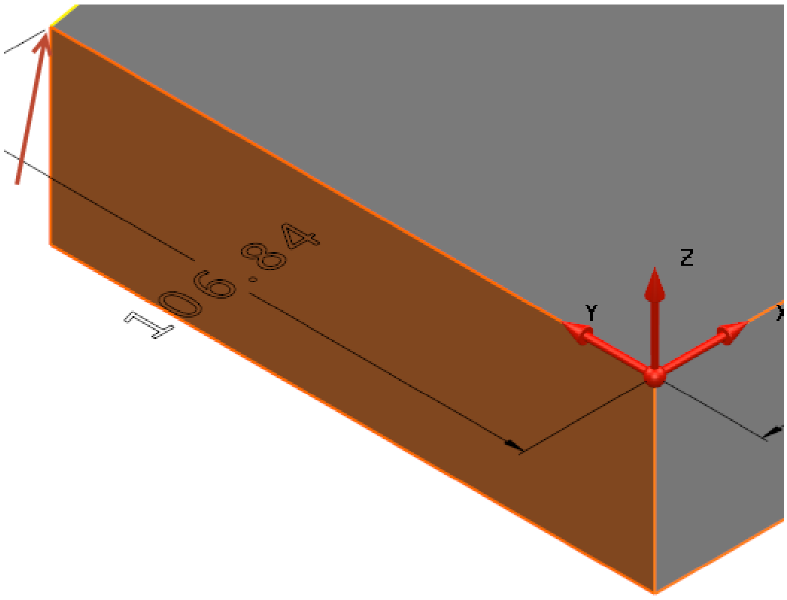
Now we will **resize** the part to the **exact dimensions** from the **original** (**if known**) using **Direct Solid Modelling**. These **minor errors** can stem from **part wear** to **errors inherent in the scanner**.

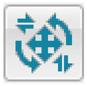
69 Select the **solid** and then **Select Inidividual Faces**  from the **Solid Edit** toolbar.

We know the overall width of the part is **488.00mm** and we currently have measured it as **487.95mm**. Therefore we need to extend the width by **0.05mm**. However due to the part **symmetry** that will require **each side face** to be **extended equally** at **0.05/2mm**.

70 Take an **Isometric View** (**Ctrl+I1**) and **zoom in** on the **end face**.

71 Select the **face**, this should **highlight orange** on the **solid**.

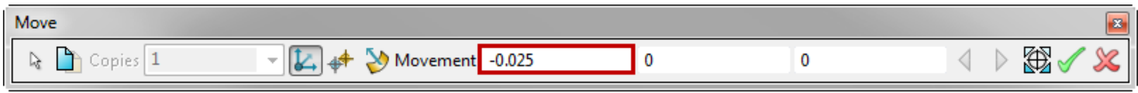


72 From the **General Edits**  menu select **Move**. 

*Taking a look at the **Active Workplane orientation** we will need to **move** this **face** in the **–X direction**.*

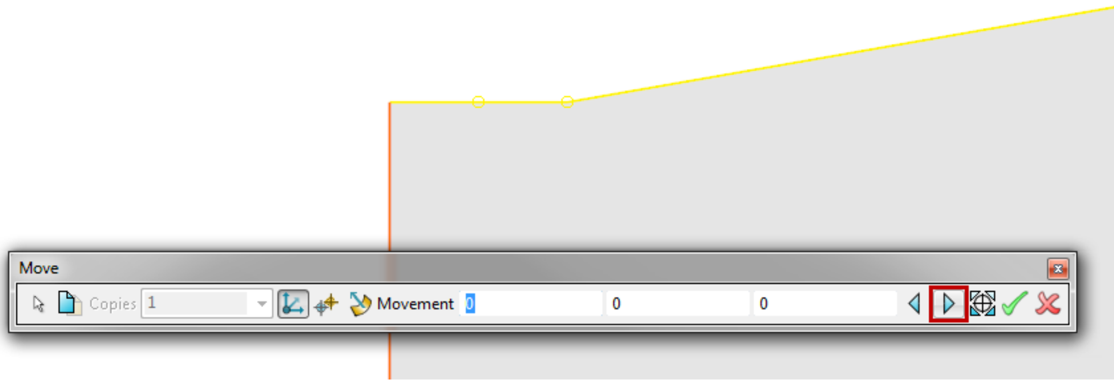
73 In **move** dialog **X distance** box type **-0.025mm**.

*This value was derived from **–(488.00-487.95)/2**, it is also possible **type** the **mathematical equation** in also to create the **value**. **Press Enter** to **Preview**.*



74 **Select** the **green tick**. **Do not close** the **dialog**.

75 Take a **View from Top** (**Ctrl+5**) and **zoom** into the **area highlighted** in the image at the **top of the page**.



76 You may see that instead of following on the angled face, the **software** has created a **different solution**. If this is the case select the **Next Solution** icon **highlighted above**.

*This will **automatically update** any **associated dimensions**.*

77 **Repeat** this **process** for the **opposite end face** (**+X direction**).

*This has now given an overall **width** (**X**) of **480.00mm**.*

Next, we will address the **length** of the **two side faces** we have just moved. Currently they have a **length** of **106.84mm**, whereas the **actual length** is **107.00mm**. The **front face** of the **part** will be moved to **edit** these **faces**.

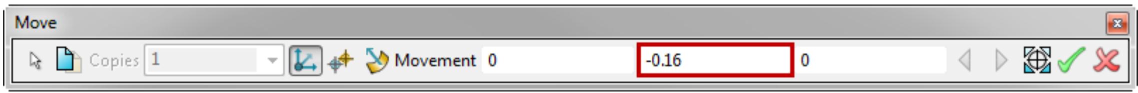
78 Select a **View from Front** (**Ctrl+2**).

79 Select the **front face** of the **solid**, it will be **highlighted orange** in the **graphics area**.

80 From the **General Edits** menu select **Move**.

*Taking a look at the **Active Workplane orientation** we will need to **move** this **face** in the **–Y direction**.*

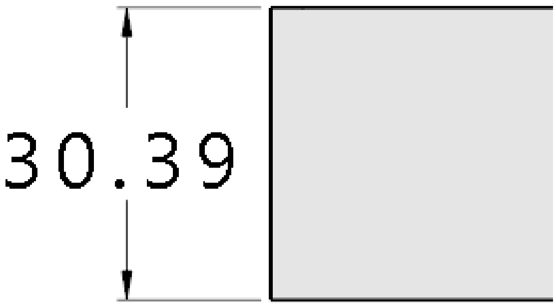
81 In **move** dialog **Y distance** box type **–(107.00-106.84)** and then press **Enter**.



*This will **automatically calculate** the **movement distance** from the **equation**.*

82 Select the **green tick** to **move** the **face**.

83 Open the **Annotation** menu and then select the **Automatic Dimension** tool. **Dimension** the **height** (**Z**) of the **model**.



*The **dimension** is **measured** as **30.39mm** however after **measuring** the **physical part** we can see it is **30.00mm deep**.*

84 Select an **Isometric View** (**Ctrl+1**).

85 Select the Top Face of the part and then from the **General Edits** menu select **Move**.

86 In **move** dialog **Z distance** box type **–(107.00-106.84)** and then press **Enter**.

87 Select the **green tick** to **move** the **face**.

*Again, the **dimension** will **automatically update**.*

88 Select all the **dimensions** in the **graphics area** by holding the **Ctrl** key and select **each dimension independently**.

89 Place the **annotation** on **Level 3 : Annotation** and ensure it is **Switched OFF**.

90 **Turn OFF Level 30 : Solid:Main Block** and **Turn ON Level 10 Mesh:Main**.

91 Select a **View from Top** (**Ctrl+5**).

Looking at the **mesh model** we have a **mirrored pattern** of **4 features**, we will create the **features** as **separate solids** and **remove** them from the **main block** using the **Boolean Subtraction** tool. We will create a **solid** for **each feature separately**.

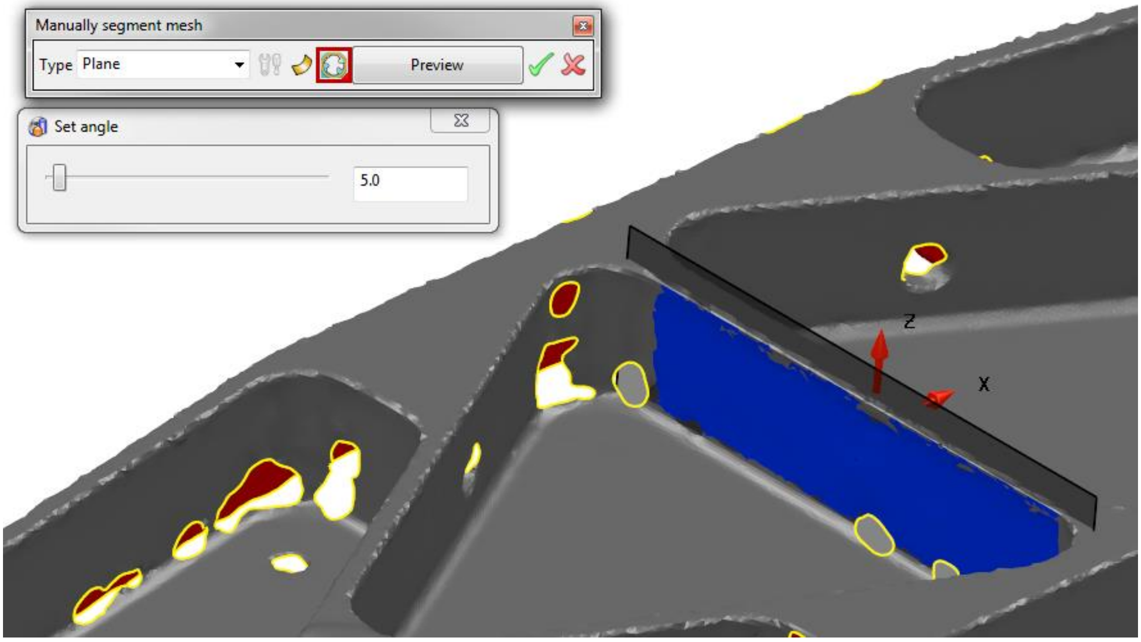
92 Select the **mesh** and then from the **Mesh Edit** toolbar select **Manually Segment Mesh**. 

93 In the **Set Angle** dialog reduce the angle to **5.0°**.

*We achieved this **value** using **trial and error** as the **minimum value** to achieve triangle selected across the **length of each face**.*

94 In the **Manually Segment** mesh dialog choose to **Fit Mesh Outside**.

95 Select the **face** shown in the **below image**. Click **Preview**.



96 Click the **green tick** to **create** the **surface. Do not close** the **dialog**.

*This will create the **surface** and **hide** it until we **close** the **dialog** to allow us to **continue fitting surfaces** to the mesh **without obstructing the view**.*

97 Using the **same method** create the other **two vertical surfaces** that **define** this **pocket feature**.

98 Using the **Ctrl** key to **multi select** areas on the **mesh** select the **bottom face** of **each pocket**.

*This will provide an **average** of the **entire area** and will create a **'master' surface** that we can use to create **each feature**.*

99 Click the **green tick** to **create** the **surface**.

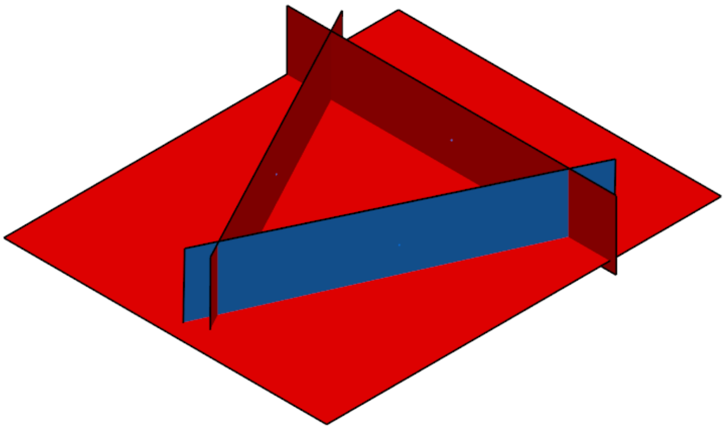
100 Click the **red cross** to **close** the **dialog**.

*Although we have not created the **'top' surface** to **close the model**, we can use an **open solid** to **remove the feature**.*

101 **Turn OFF Level 10 : Mesh:Main**.

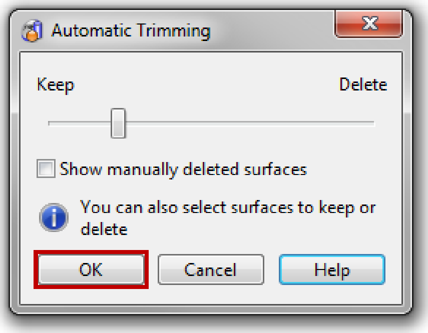
102 **Reverse** the **surfaces** so that the **blue** (**selected colour**) will form the **outer faces** of the **open solid**.

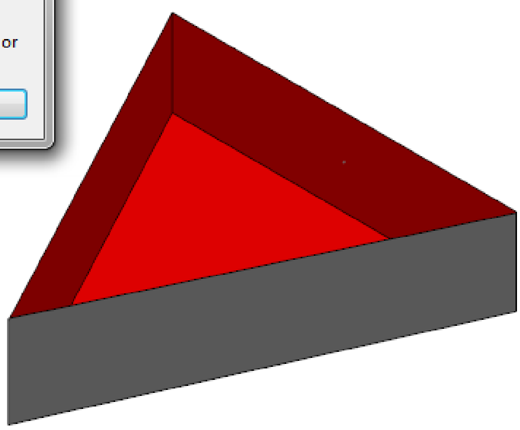
103 **Next**, **dynamically edit** the **size** of each **surface** to create the **required adjoining overlaps**. Shown below.



104 **Quick Select ALL Surfaces**. 

105 From the **Solid**  menu select **Create Solid by Automatically Trimming Surfaces**. 





This will create an **open solid shape as above.** You may have to **refine** your **solid** by **moving** the **slider** and **manually deleting some segments.** Alternatively, you can click **Cancel** to go **back** and **edit** your **surfaces** and **try again**. We will apply **corner** and **bottom radius fillets later**.

106 Click **OK** in the **dialog**.

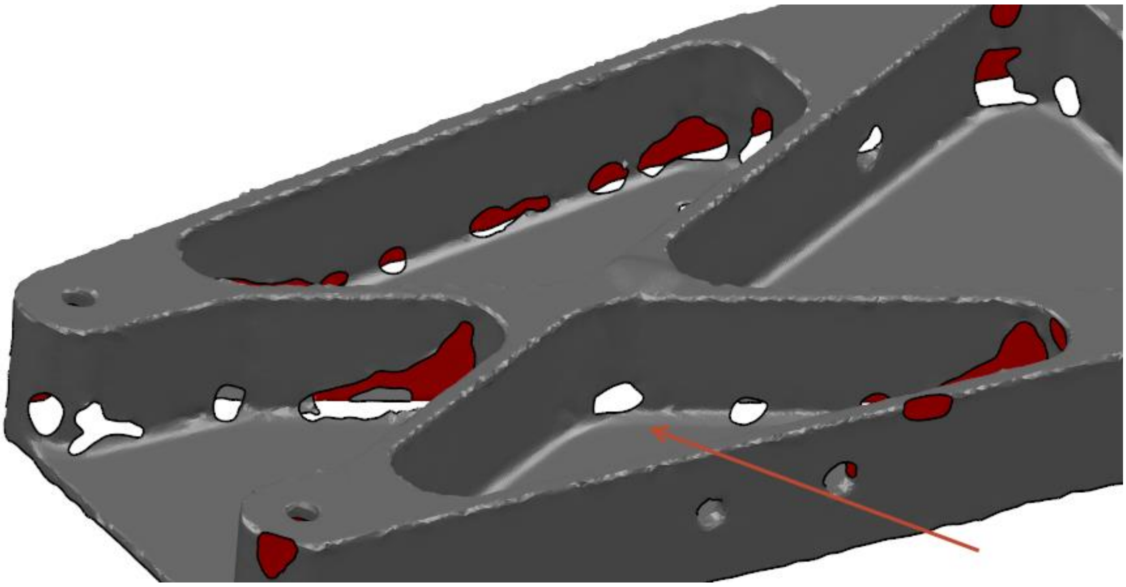
107 Raise the **Levels dialog** from the **Levels** toolbar.

108 Create a **New Level 31 : Solid:Open Solids** and ensure it is **Switched OFF**.

109 Place the **open solid** on the **New Level 31 : Solid:Open Solids**.

110 **Turn ON Level 10 : Mesh:Main**.

111 Using the **Manually Segment mesh**  tool create the **3 vertical surfaces** for the **feature highlighted** in the **image below** using a **Set Angle** of **5.0°** and **Fit Mesh Outside**.



112 **Close** the **dialog**.

*Although we only have the **3 side surfaces**, we can **retrieve** the **bottom surface** that we created earlier from **Level 799**, which was **automatically created** when using the **Solid from Untrimmed Surfaces** tool.*

113 **Turn OFF Level 10 : Mesh:Main** and **Turn ON Level 799**.

114 **Select** the **horizontal** (**bottom face**) **surface** and place it on **Level 0 : General**.

115 **Turn OFF Level 799**.

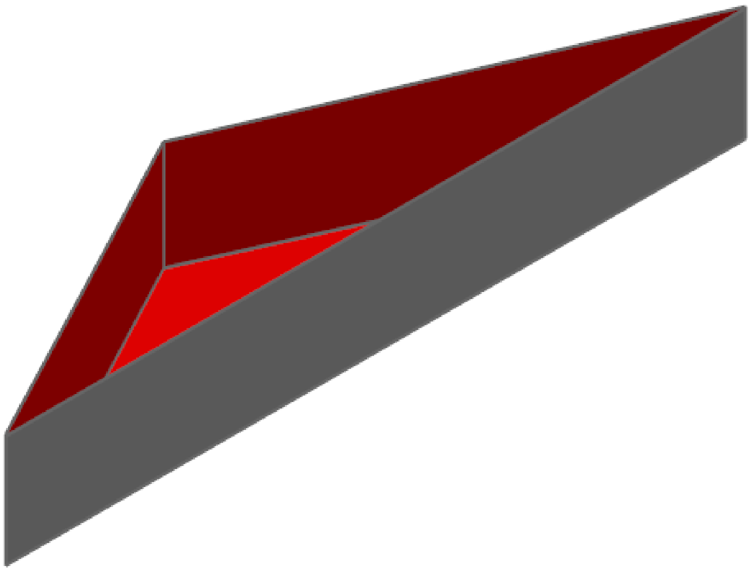
116 As before, **Reverse** the **surfaces** so that the **blue** (**selected colour)** will form the **outer faces** of the **open solid**.

117 **Next**, **dynamically edit** the **size** of each **surface** to create the **required adjoining overlaps**.

118 **Quick Select ALL Surfaces**. 

119 From the **Solid**  menu select **Create Solid by Automatically Trimming Surfaces**. 

This will create an **open solid shape as seen below**. You may have to **refine** your **solid** by **moving** the **slider** and **manually deleting some segments**. Alternatively, you can click **Cancel** to go **back** and **edit** your **surfaces** and **try again**. We will apply **corner** and **bottom radius fillets later**.



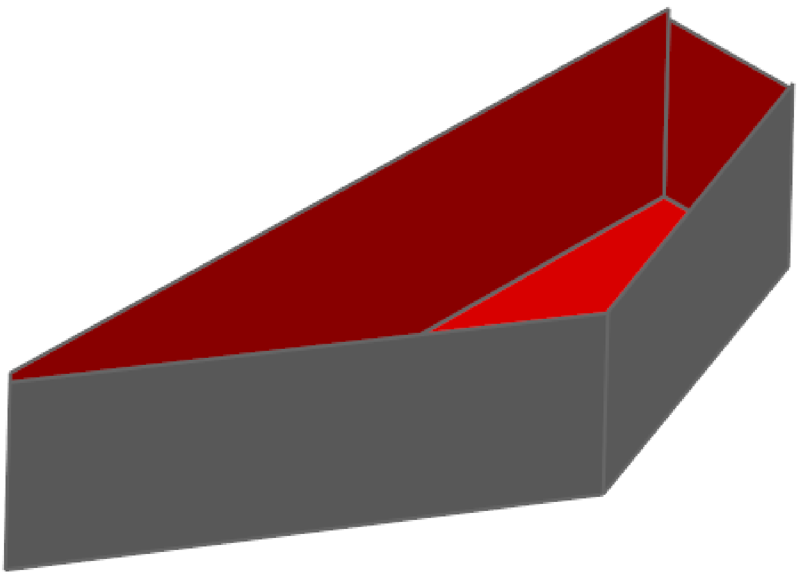
120 Place the new **open solid** on the **New Level 31 : Solid:Open Solids**.

121 **Turn ON Level 10 : Mesh:Main**.

122 Using the **same method** create the other **open solid** that define the **basic shape** of the other **pocket** on the **left hand side** of the **part**.

 *Note: This **pocket** has **4 sides**.*

123 **Turn OFF Level 10 : Mesh:Main**.



124 Place the new open solid on the **New Level 31 : Solid:Open Solids**.

Finally, we need to create the **open solid** to represent the **open side feature**. This will be completed in a similar method, however this will have **more than** the **one open face** we have used **earlier**.

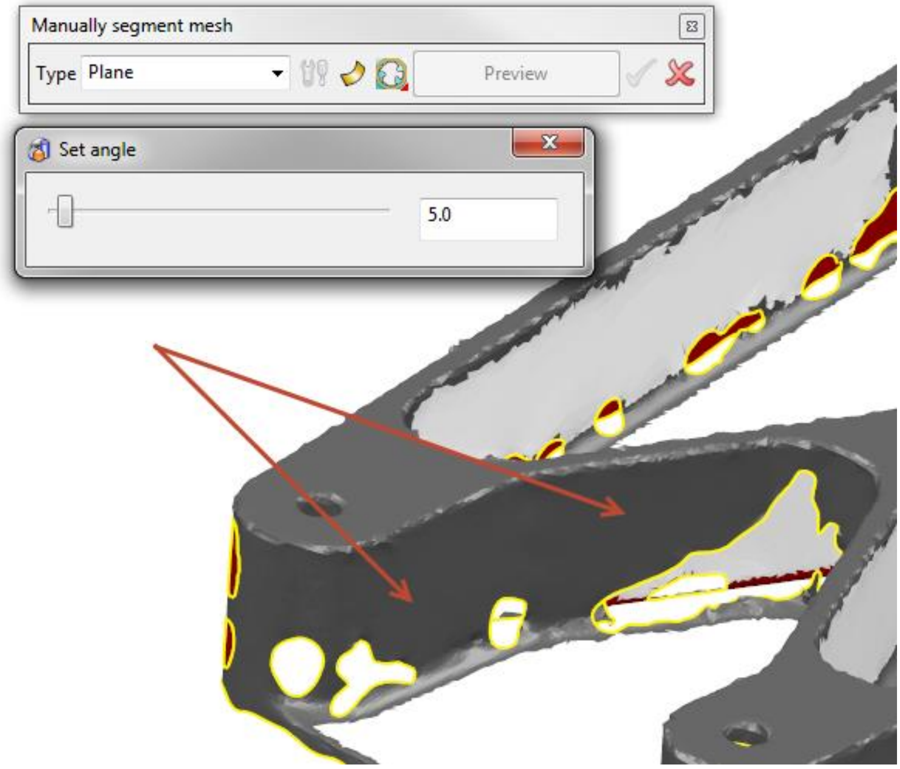
125 **Turn ON Level 10 : Mesh:Main**.

126 Select the **mesh** and then from the **Mesh Edit** toolbar select **Manually Segment Mesh**. 

127 In the **Set Angle** dialog reduce the angle to **5.0°**.

128 In the **Manually Segment** mesh dialog choose to **Fit Mesh Outside**.

129 **Select** and **create surfaces** for the **two faces indicated** in the **below image**.



130 Next, create the **opposing two faces** within the **feature**.

131 **Close** the **dialog** by **clicking** the **red cross**.

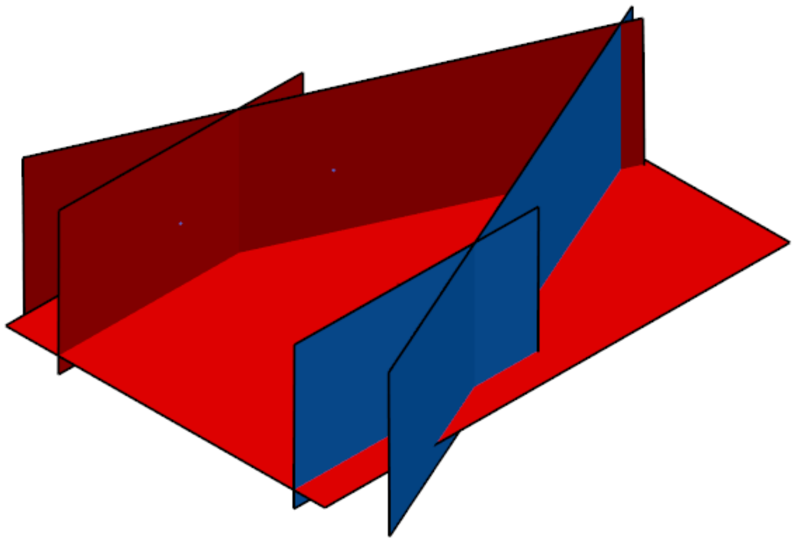
132 **Turn OFF Level 100 : Mesh:Main** and **Turn ON Level 797**.

133 **Select** the **horizontal** (**bottom face**) **surface** and place it on **Level 0 : General**.

134 **Turn OFF Level 797**.

135 As before, **Reverse** the **surfaces** (**if required**) so that the **blue** (**selected colour)** will form the **outer faces** of the **open solid**.

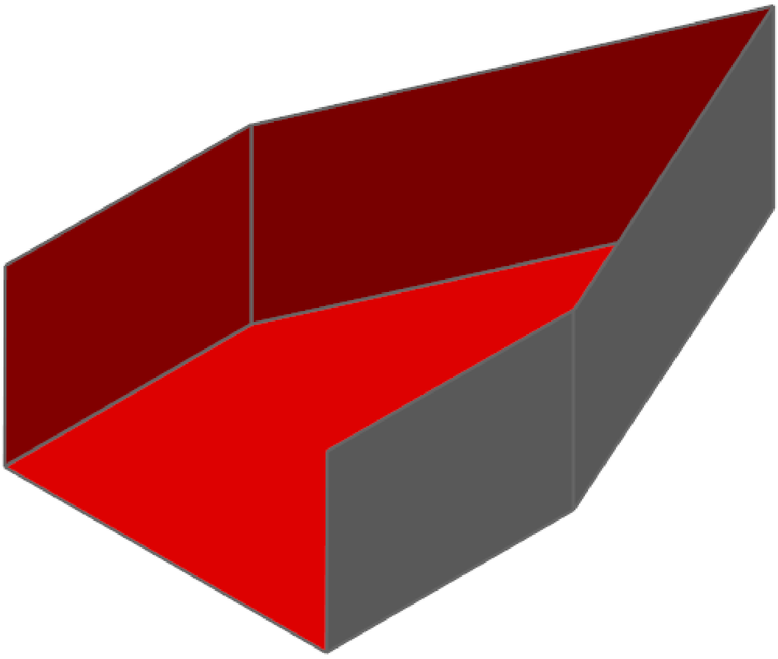
136 **Next**, **dynamically edit** the **size** of each **surface** to create the **required adjoining overlaps**.



 *As with the **actual feature** on the **mesh**, this will have an **open end**.*

137 **Quick Select ALL Surfaces**. 

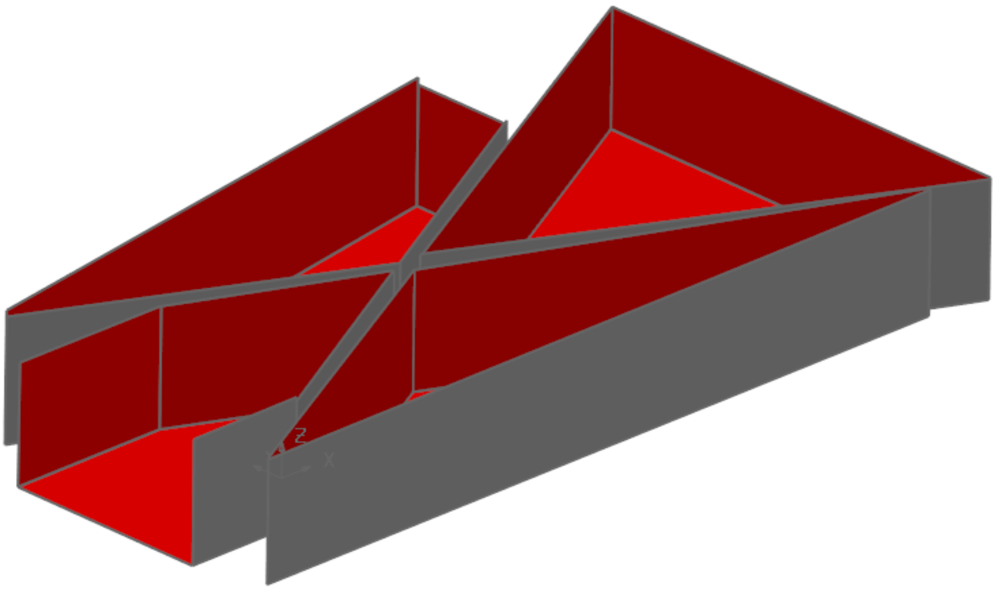
138 From the **Solid**  menu select **Create Solid by Automatically Trimming Surfaces**. 



This will create an **open solid shape as seen above**. You may have to **refine** your **solid** by **moving** the **slider** and **manually deleting some segments**. Alternatively, you can click **Cancel** to go **back** and **edit** your **surfaces** and **try again**. We will apply **corner** and **bottom radius fillets later**.

139 Place the new open solid on the **New Level 31 : Solid:Open Solids**.

140 **Turn ON Level 31 : Solid:Open Solids**.



141 **Turn ON Level 10 : Mesh:Main** and **Turn OFF Level 31 : Solid:Open Solids**.

Looking at the **mesh** we can see that the **features** have a **large corner radius** joining **each vertical face** as well as **bottom radius fillets**. Using the **Solid Fillet** tool we can **fillet** our **open solids** to exactly represent the required feature shape **before removing** the **features** from the **main block**.

First, the **size** of the **fillets** need to be determined by **creating** and **measuring section curves** using the **dynamic section tool**.

142 **Navigate** to **View>Dynamic Sectioning**.

*The **Dynamic Section***  *tool is also available from the **Model Analysis** menu.* 

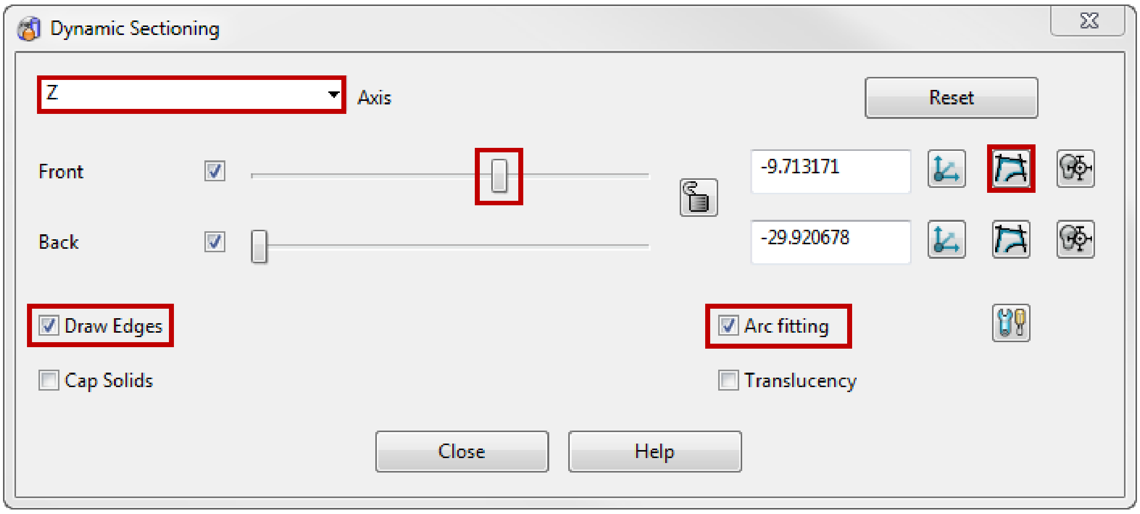
143 Inside the **Dynamic Section** dialog, select to **section** through the **Z Axis** and tick the **Draw Edges** and **Arc Fitting** check boxes.

***Draw Edges** will **highlight** the **edges** of the **section** in **white** for **better visibility** of the **section** before **section curves** are **created**.*

***Arc Fitting** will create a **smoother curve fit** to the **section**.*

144 Using the **Front** slider find a **location** where we have at least **one complete radius** to **measure** from, as all the **radii** in the case are the **same size**.

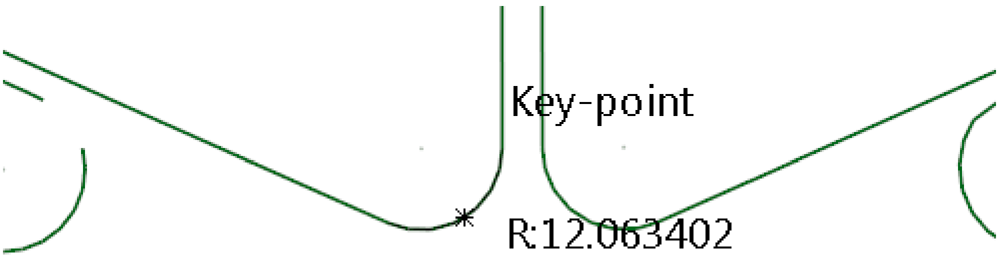
145 Click **Create Wireframe** from **section**  to create the **section curves**.



146 Select and **Blank** (**Ctrl+J**) the **mesh**.

147 Take a **View from Top** (**Ctrl+5**).

148 Using the **3 Point Arc** from the **Circle** menu **measure** any one of the **corner radii** to **determine** the **size** of the **corner fillets** to apply.



Having measured a **radius** of approximately **12.00mm** we will apply this **fillet size** to each of the **vertical corners.** You can check the **radius** on more of the **corners** to **double check the sizing if required**.

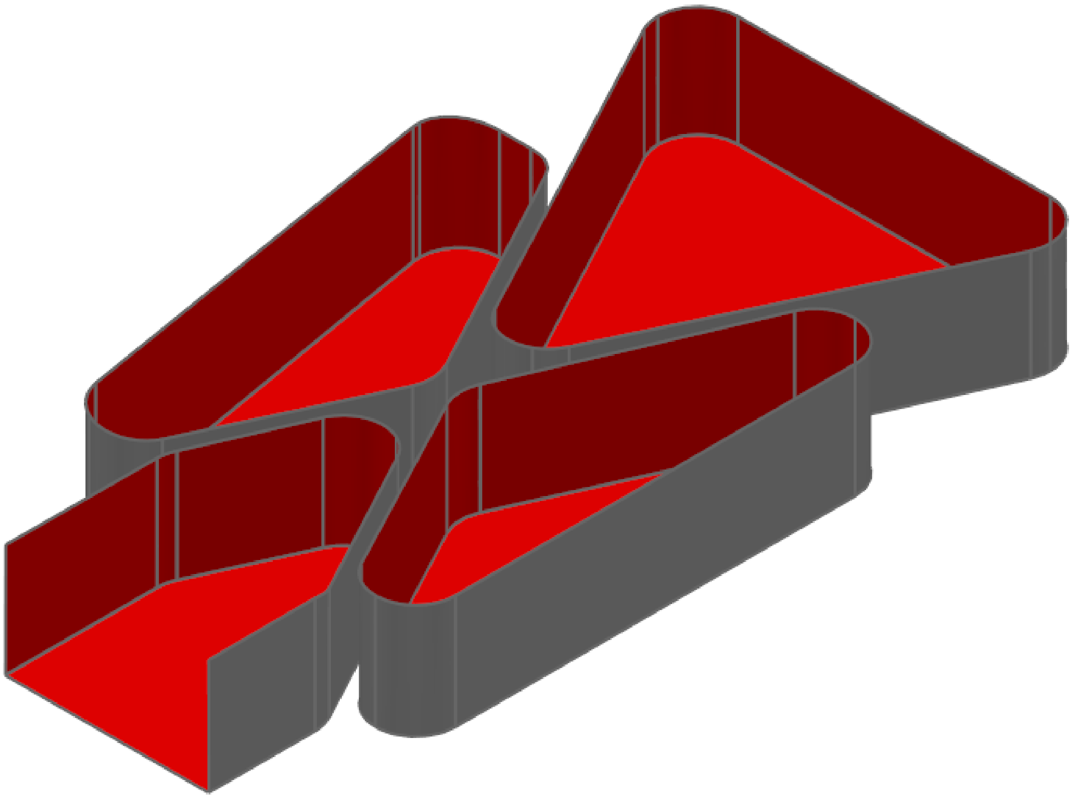
149 Select **ALL** the **section curves** using **Quick Select ALL Wireframe**.

150 **Delete** the **curves**.

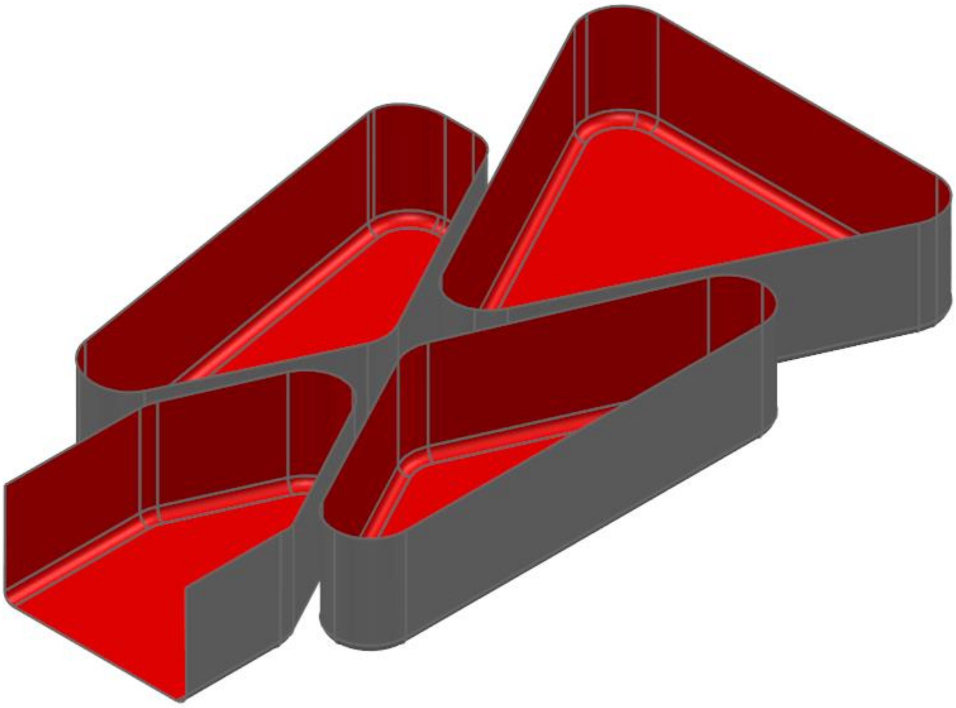
151 **Unblank** (Ctrl+L) the **mesh** and then **Turn OFF Level 10 Mesh:Main**.

152 **Turn ON Level 31 : Solid:Open Solids**.

153 Using the **Solid Fillet** tool from the **Solid Feature** menu, apply a **fillet** of **12.00mm** to **all** of the **vertical corners** on the **4 open solids**.



154 Using the **same method** as above (**142 – 153**) by taking a **dynamic section** down the **X Axis**, determine the **bottom radius** of the **pocket features** on the **part** (**3.00mm**) and then **apply** this **value** to the **bottom** of **each** of the **open solids**.



155 **Turn ON Level 30 : Solid:Main Block**.

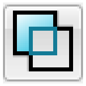
156 Open the **Solid History Tree**  from the **Solid Edit** toolbar.

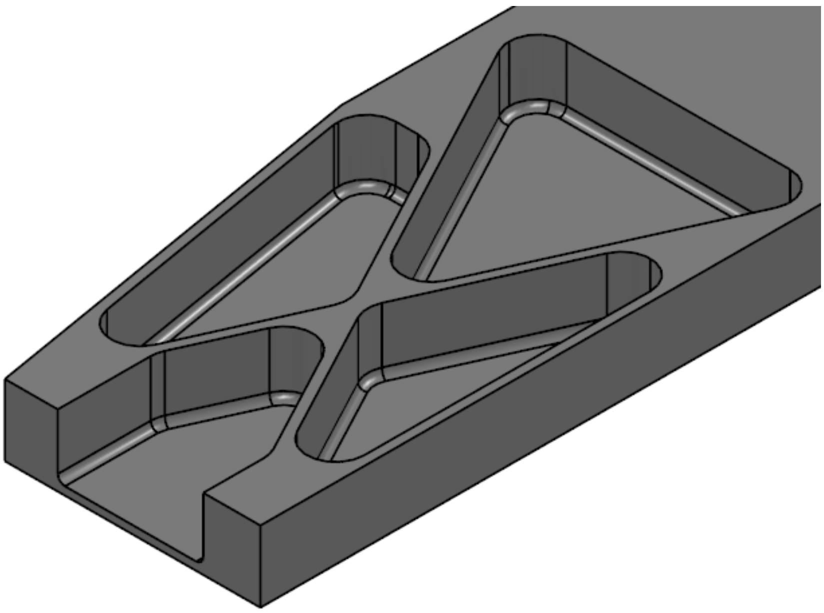
157 Ensure the **main block** is **active**.

*This will **automatically** select the **main block** as the **primary selection** in the **Boolean Subtraction** operation.*

158 **Quick Select ALL Solids**. 

159 From the **Solid Feature**  menu select to perform a **Boolean Subtraction**.

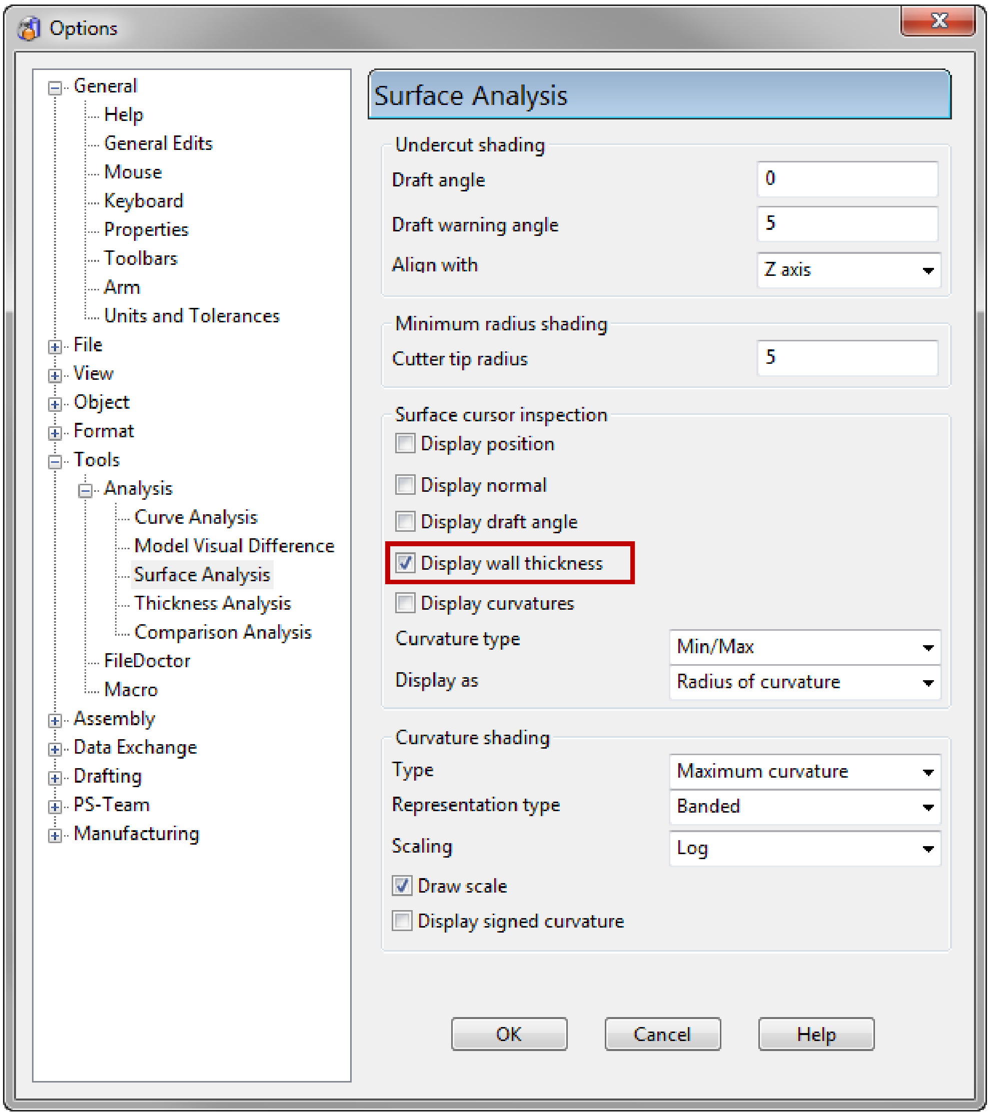




Next, we will **measure** and **refine** the **depth** of the **pockets**, we know from the **physical model** that the **wall thickness** at the **bottom** of the **pocket** is **3.00mm**. We can check this on the **solid model** in the **graphics area** using **Surface Inspection/Model Analysis** tools. To do this we need to set up the **surface inspection options** in the **tools>options area**.

160 Navigate to **Tools>Options>Tools>Analysis>Surface Analysis**.

161 In the **surface cursor inspection** area ensure **Display Wall Thickness** is **ticked**. Click **OK**.

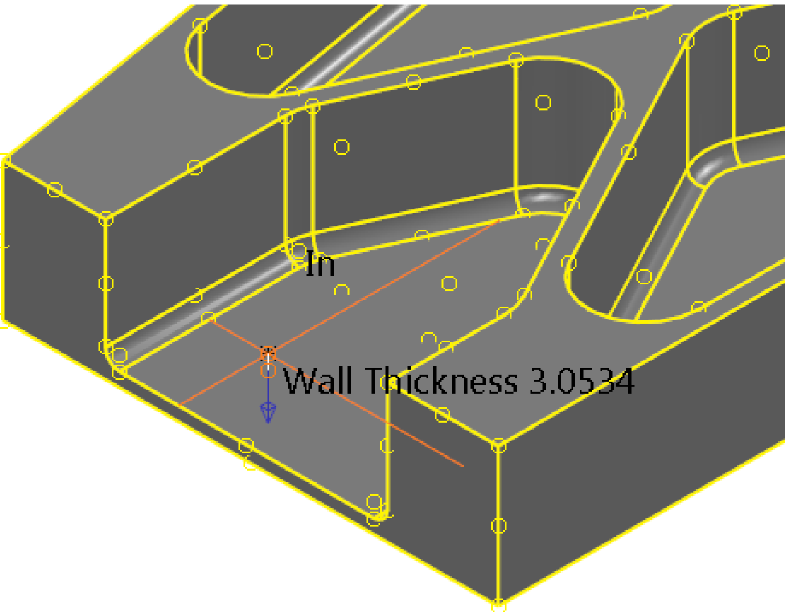


162 Select the **solid** in the **graphics area**.

163 From the **Model Analysis**  menu select **Surface Cursor Inspection**.

164 Place your cursor over the bottom of any of the pockets and click & hold to left mouse button to display the wall thickness at that location.

*As we used the **same surface** for **each feature** the **wall thickness** will be **constant throughout**.*



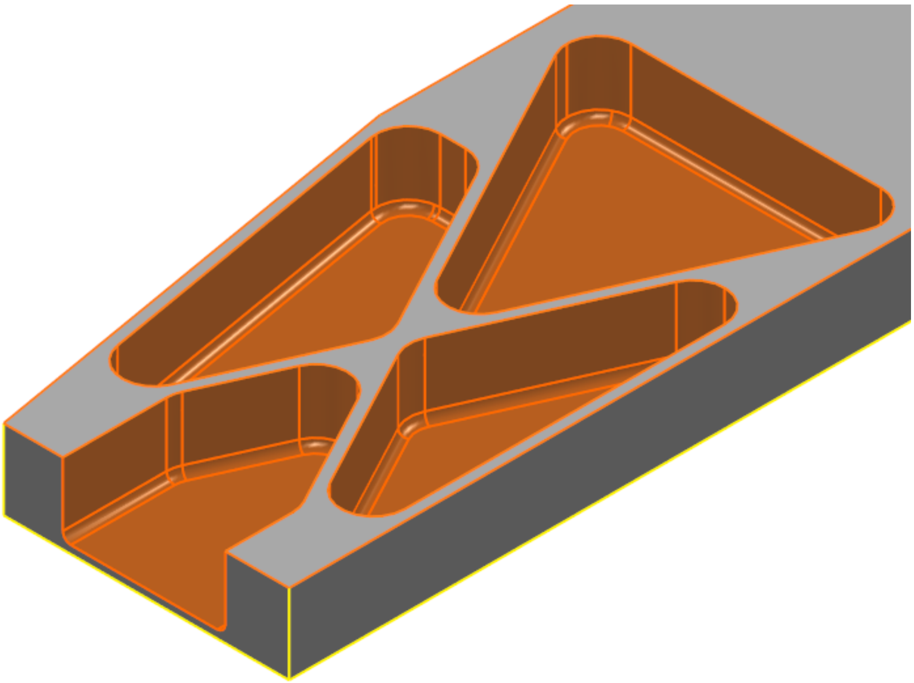
*This has given a **wall thickness** of **3.0534mm**. Therefore we need to increase the **depth** of **each feature** by **0.0534mm** using **Direct Solid Modelling** tools.*

165 Press the **Esc** key to **exit surface inspection**.

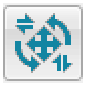
166 From the **Solid Edit** toolbar choose to **Select Concave Regions**. 

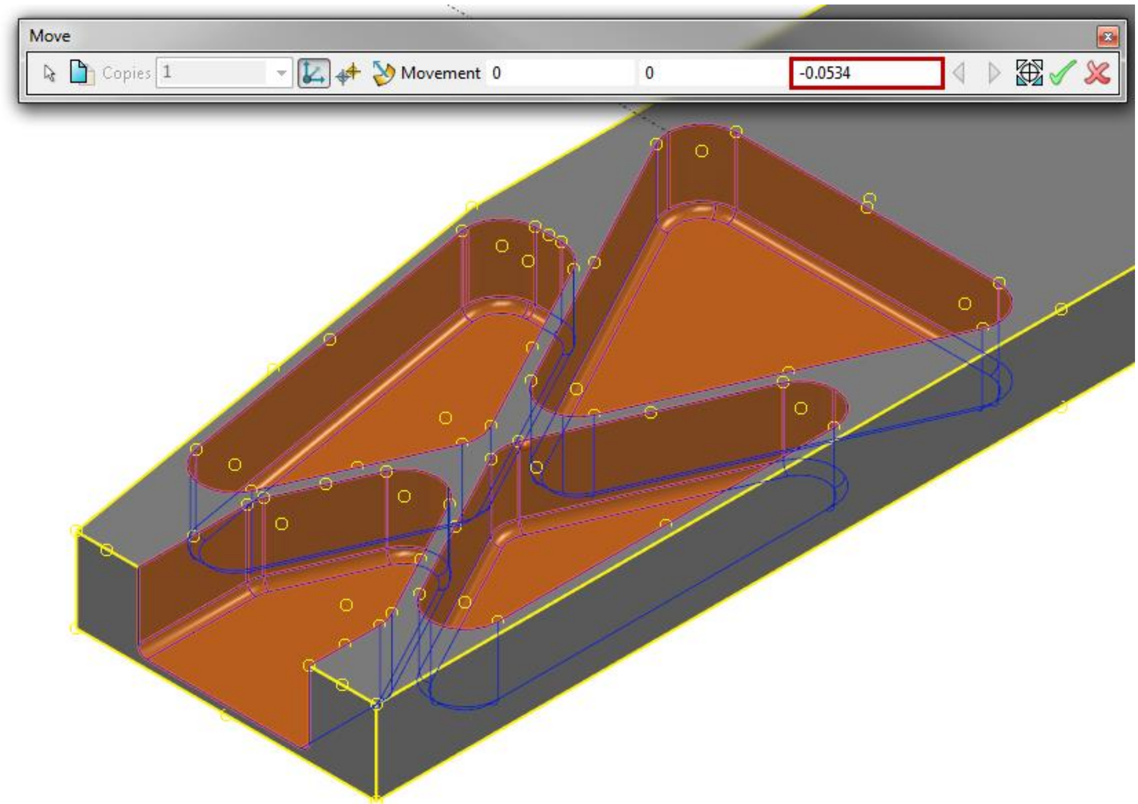
*This will work in a **similar way** to the **Select Individual Faces** option by **selecting faces** and **highlighting orange** rather than **Selecting Features**. The upside to this is we can quickly and easily select **entire concave regions** (**multiple faces**)*

167 By holding the **Ctrl** key to multi select, **select all** the **faces** of the **four pocket features**.



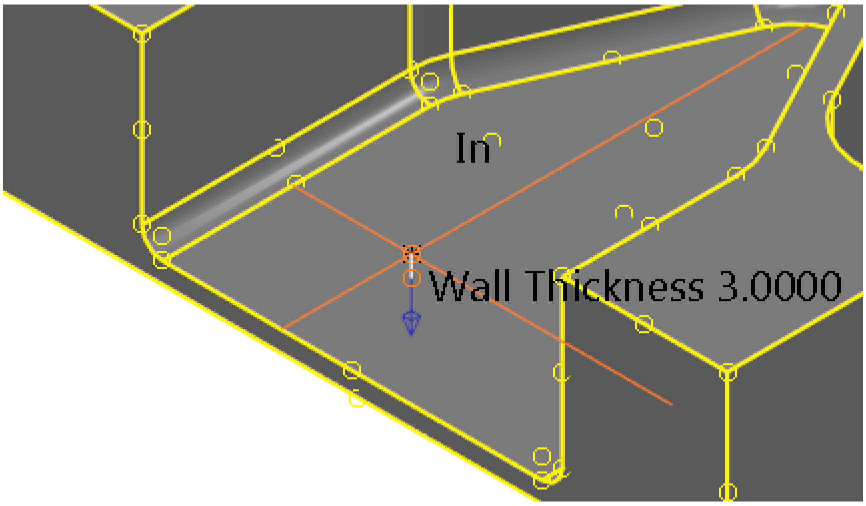
The next step will be to **Move** these **faces** in **Z** by a **distance** of **-0.0534mm** to give a **constant 3.00mm wall thickness**.

168 From the **General Edits**  menu select **Move**. 



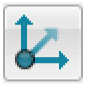
169 Select the **green tick** to **move** the **faces** and allow the **software** to **retrim** a **closed solid form**. **Close** the **dialog**.

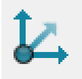
170 Check the **wall thickness** now using the **Surface Cursor Inspection**  tool in the **Model Analysis** menu. 



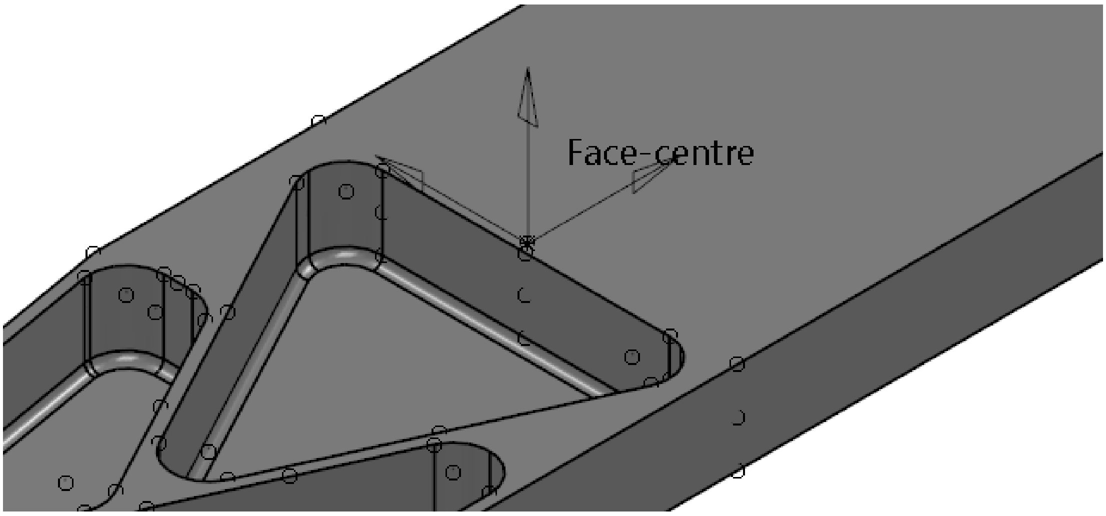
To create the **features** on the **right hand side** of the **model** we will **mirror** the faces across the **axis of symmetry**. To complete this we will create a **new workplane** which can form our **Mirror plane**.

171 Ensure the **Principal Axis** to **Z**.

172 Select to **Create a Single Workplane**  from the **Workplane** menu.

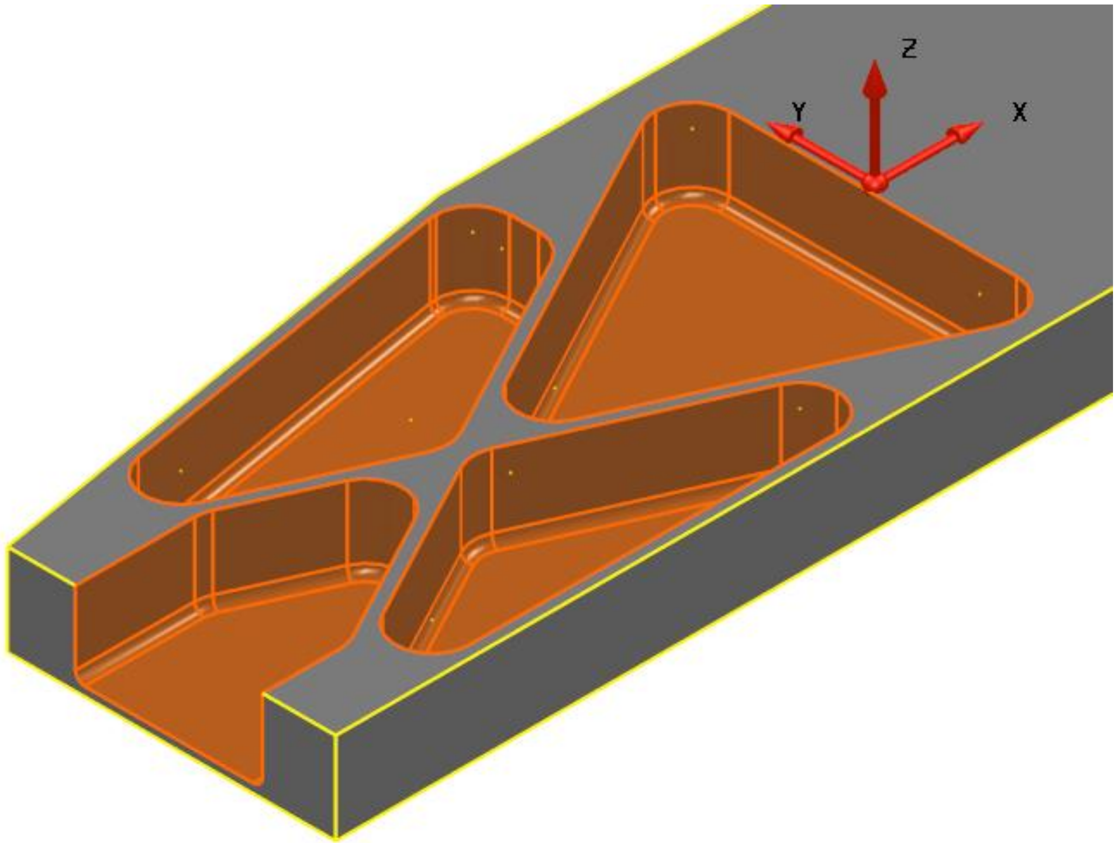


173 Create a **single workplane snapped** to the Face**-Centre** of the **Top Face** of the part as **shown** in the **image below**.



174 From the **Solid Edit** toolbar choose to **Select Concave Regions**. 

175 By holding the **Ctrl** key to multi select, **select all** the **faces** of the **four pocket features**.

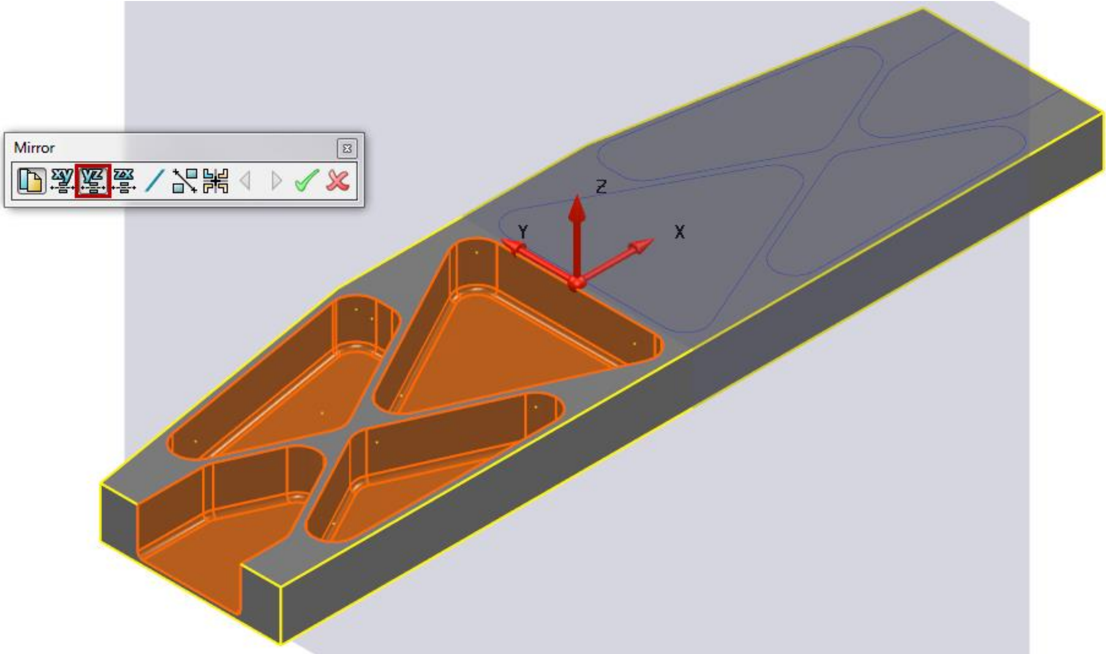


176 Ensure the **Principal Axis** is set to **X**.

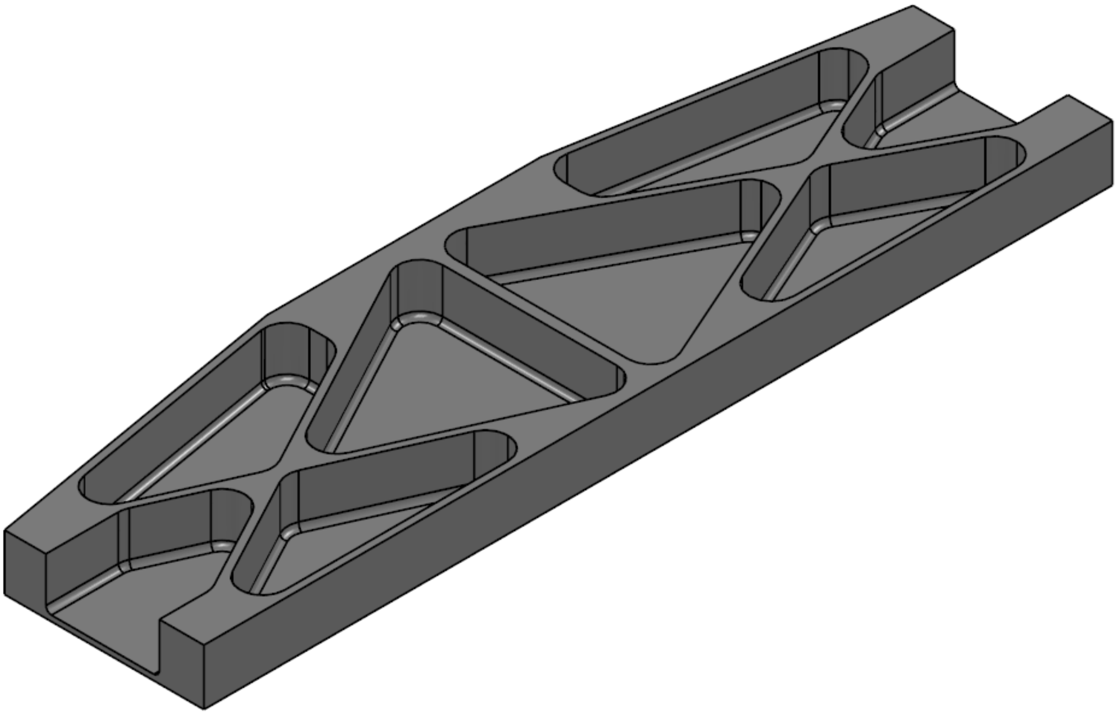
*This may differ due to the **workplane orientation** in your **session**.*

177 From the **General Edits** menu select **Mirror Objects**.

178 In the dialog ensure the **YZ plane** is selected as the **Mirror plane**.



179 Click the **green tick** to **accept** and **mirror** the **pocket features**. **Close** the **dialog**.



*The **feature faces** have successfully **mirrored** across the **axis of symmetry** to create the **equivalent** on the **right hand side** of the part.*

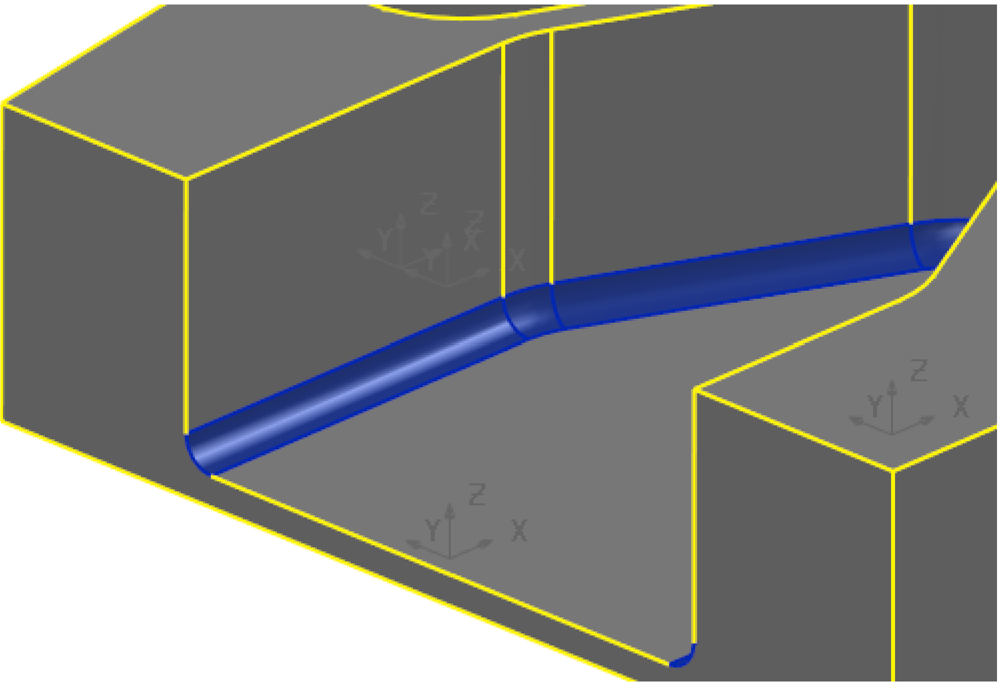
The **next step** we will look at is to create the **fillets** around the **outer edge** of the **block** and into the **open side pockets**. Because we have already created the **bottom radius fillets** in these **open pockets**, we will need to **suppress** them in order to create the **fillets** on the **vertical edges**, this is to avoid the **radii** creating a **continuous region**. To **suppress** these **features** we need to **re-recognise** them as features, due to performing **non-associative feature operations** and losing our **feature history**.

180 From the **Solid Feature**  menu select **Solid Fillet**. 

181 In the **dialog** choose to swtich to **Feature Editing** mode. 

182 In the **graphics area** select the **3mm bottom radius fillet** on one of the **open side pockets**. Select **Apply**.

*This will **recognise** the **fillet size** and **track** and create a **solid feature** inside the **history tree**.*



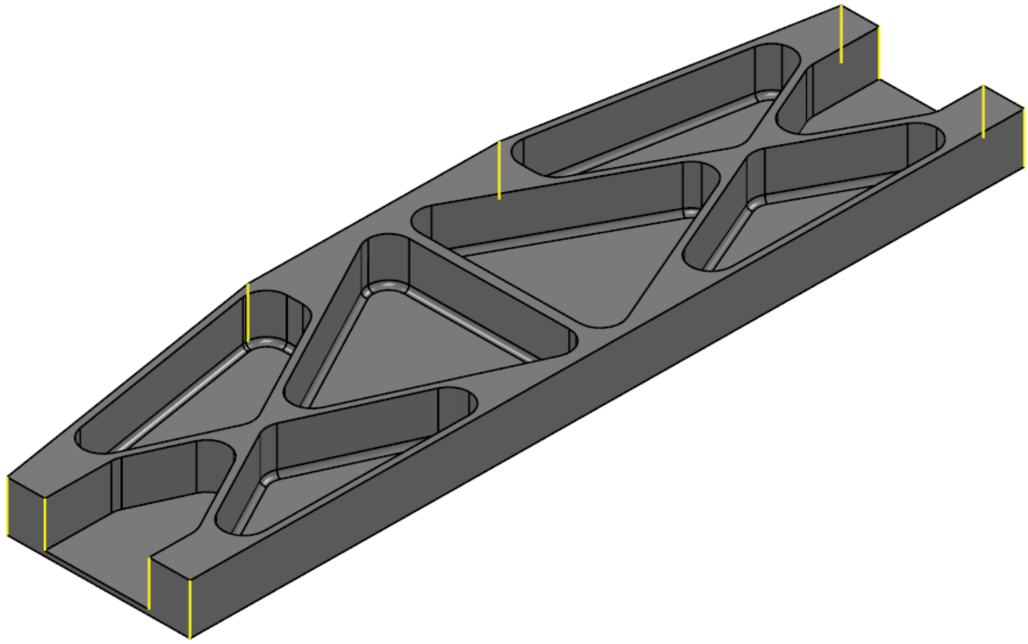
183 **Repeat** the **previous step** for the **other open side pocket**.

184 In this **Solid History Tree**, **delete** both **fillet** features by **clicking** the **adjacent green ticks**.

*We could have also use the **Remove and Heal Selected Faces** tool from the **Solid Edit** toolbar to **remove** these fillets **without the need** to **re-recognise** the **features**.*

185 Using the **Solid Fillet**  tool from the **Solid Feature**  menu create **fillets** on each of the **outer vertical edges** of radius **12.00mm**. Select **Apply** and **OK** to **create** and **close** the **dialog**.

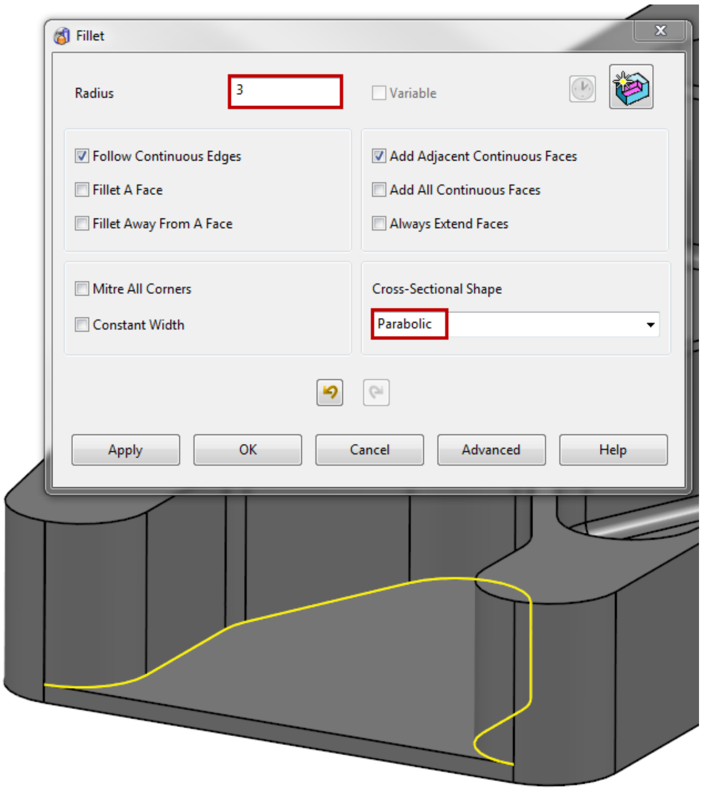
*There are **10 edges**, highlighted **below**, which also include the **small angle changes of direction** on the **back faces** of the **part**.*



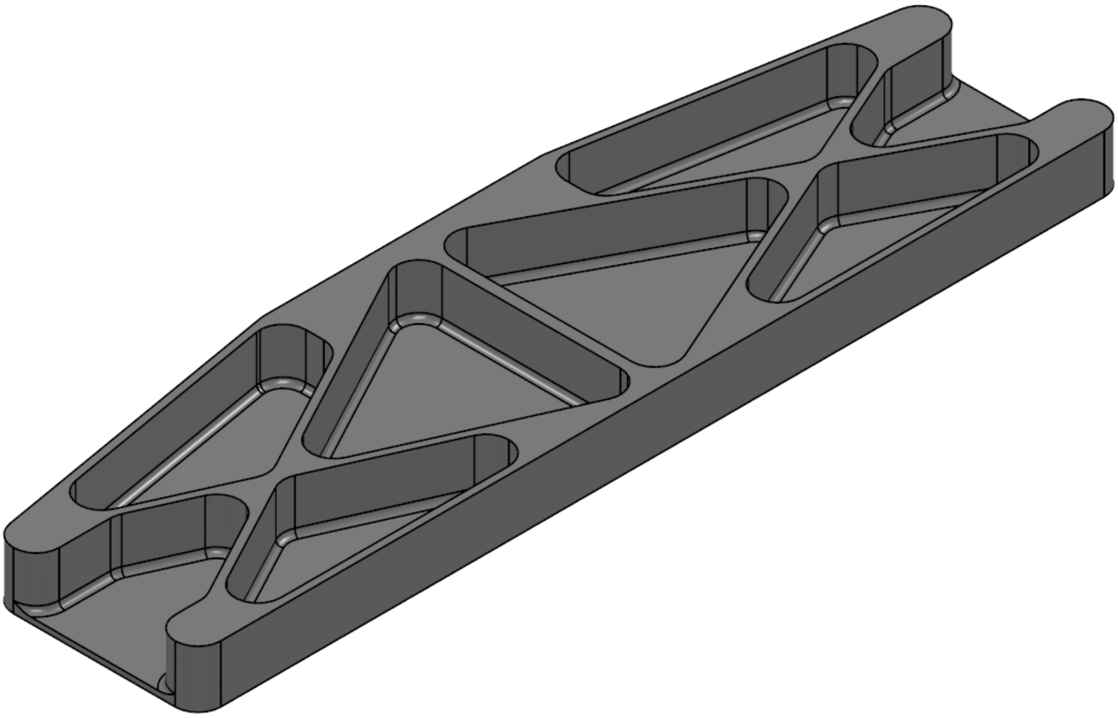
186 We will now **recreate** the **two bottom radius fillets** we **removed earlier**.

From the **Solid Feature**  menu select **Solid Fillet**. 

187 Create a **3mm fillet** following the track we removed earlier using a **Parabolic Cross-Sectional Shape**, as **shown below**. Select **Apply** but **do not close** the **dialog**.



188 **Repeat** the previous **step** for the **same feature** at the **opposite end** of the **part**.

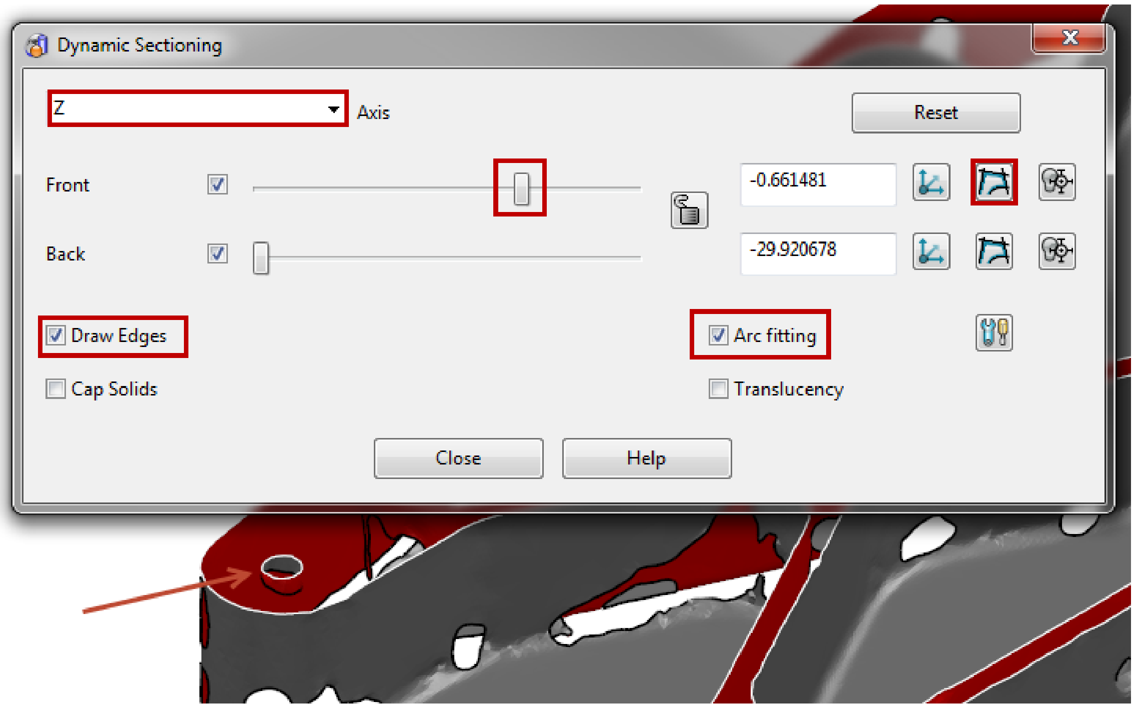


*The **main solid** is now **complete** and **awaiting finishing** with **holes**.*

189 **Turn OFF Level 30 : Solid:Main Block** and **Turn ON Level 10 Mesh:Main**.

190 Navigate to **View>Dynamic Sectioning**.

191 Inside the dialog choose to take a **section** through the **Z Axis** and slide the **slider** to a location allowing to take a **complete curve** of **one** of the **pattern of holes** and select to **create wireframe from section**. Select the **Arc Fitting** and **Draw Edges** tick box.



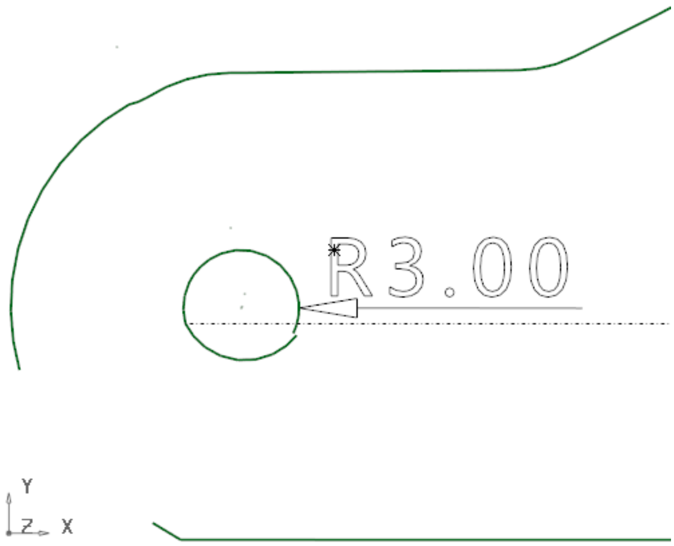
192 **Blank** (**Ctrl+J**) the **mesh**.

193 Select **a View from Top** (**Ctrl+5**).

194 **Zoom** into **any one** of the **circular arcs** created to define the **holes** from the **dynamic section**.

195 Using the **3 point radius**  **dimensioning** tool from the **Annotation**  menu, **measure** the the **arc**.

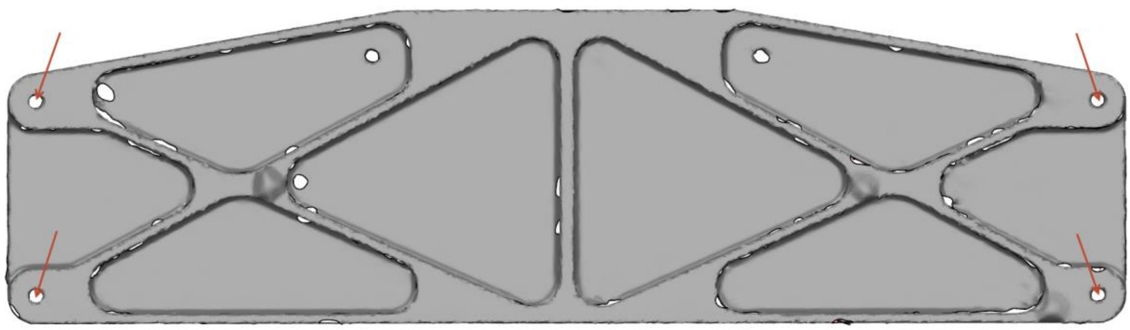
*This tool works in the same way as the **3 point arc tool** used earlier but will **measure** and create a **dimension instead**.*



*This tells us that the **pattern** of **4 holes through** the part at the **centre-arc** of each of the **corners** of the **top face**.*

196 **Quick Select ALL Wireframe**  and then **Delete**  to **remove** the **section curves** we **no longer need**.

197 **Unblank** (**Ctrl+L**) the **mesh**.



*The image above **highlights** the **location** of the **holes**. They are located at a **keypoint**, the **Arc-Centre** of the **radius** of the **corner**.*

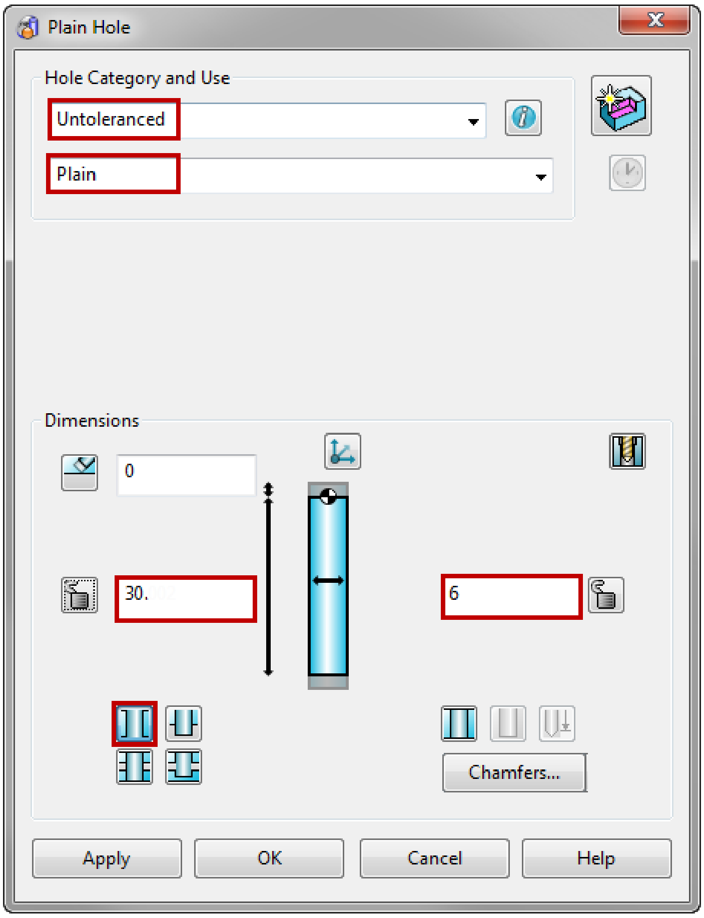
Next, we will create a **6.00mm plain hole** feature and then **pattern** it around the **model**.

198 **Turn OFF Level 10 Mesh:Main** and **Turn ON Level 30 : Solid:Main Block**.

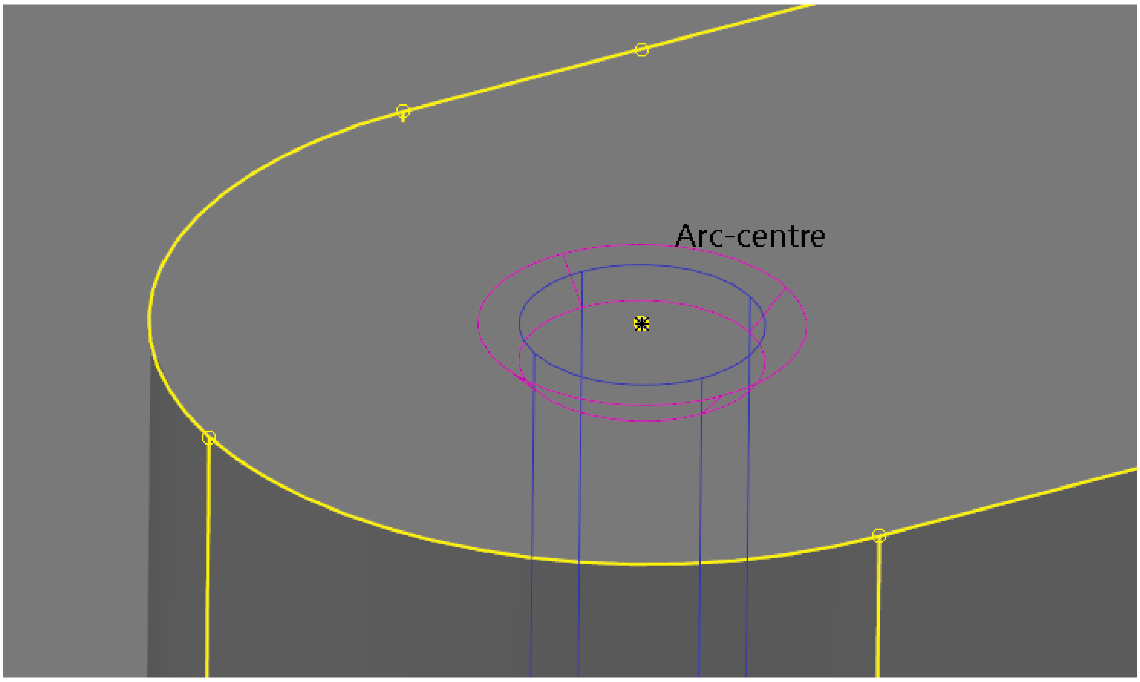
199 From the **Solid Feature**  menu select **Create a Hole**. 

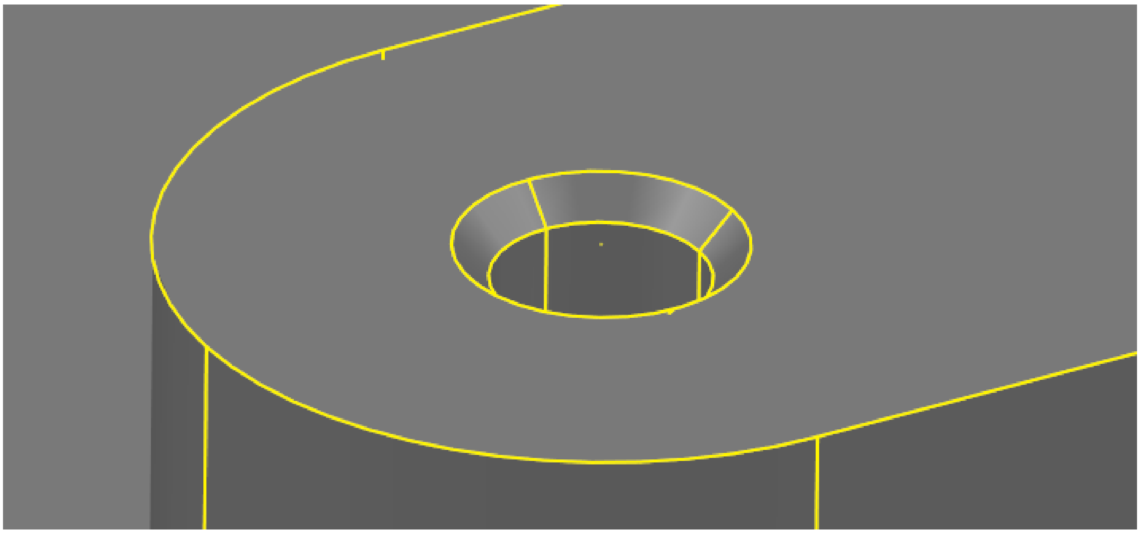
200 In the dialog select **Untoleranced > Plain hole** as the **Hole** type. Select the **Through Hole** option with a **Diameter** of **6.00mm**.

201 Select **Chamfers** and in the next dialog **apply** a **0.25mm chamfer** to the **top edge**.



202 Snap to the **Arc-Centre** of the **lower left hand corner radius** as below, and then select **Apply** in the dialog to create the **feature** in the **Solid History Tree**.



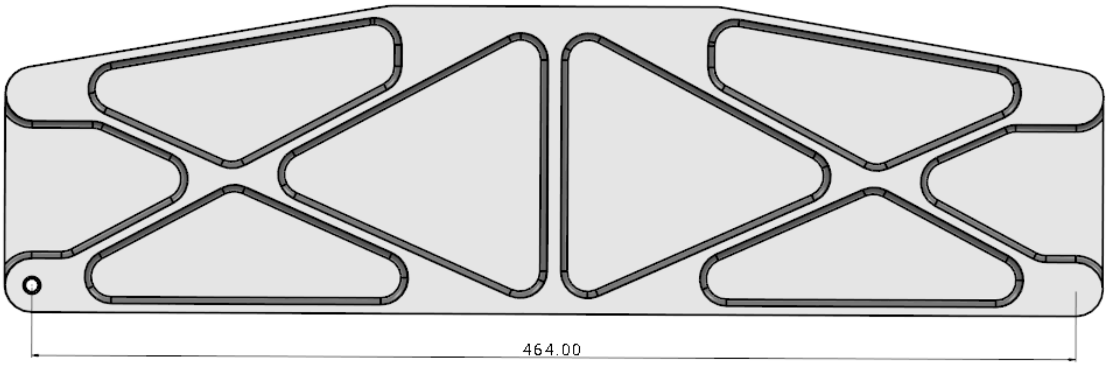


To create the other holes in the **pattern** we could **navigate** around the **model** and apply **3 further holes** of the **same dimensions.** This would create **4 separate features** in the **Solid History Tree**. Instead, we will create a **Pattern** of **features** using **General Edits**, this method will only hold **one feature**, and therefore in the future any edits will be much quicker.

To achieve this we first need to determine the **X** and **Y translations** for **the 2 x 2 pattern**.

203 Using the **Automatic Dimension**  tool from the **Annotation**  toolbar, determine the **distance** between the **Hole-Centre** and the **Arc-Centre** for the other **Hole locations**.

 *The **X distance = 464.00mm** and the **Y distance = 85.00mm**.*

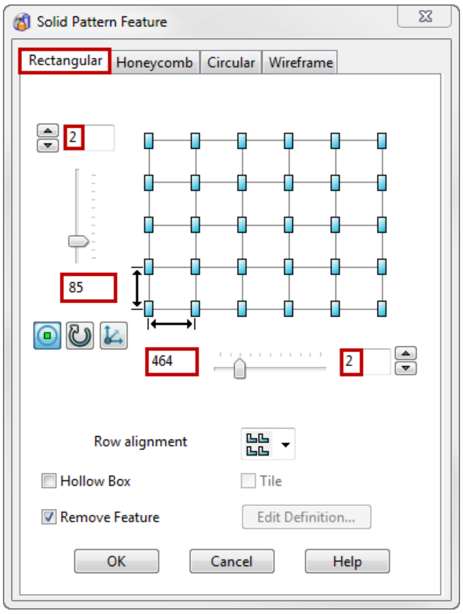


204 In the **Solid History Tree** select the **Hole** feature.

 *This will also **highlight** the **feature blue** in the **graphics area**.*

205 From the **General Edits**  tool choose to **Create a Pattern**. 

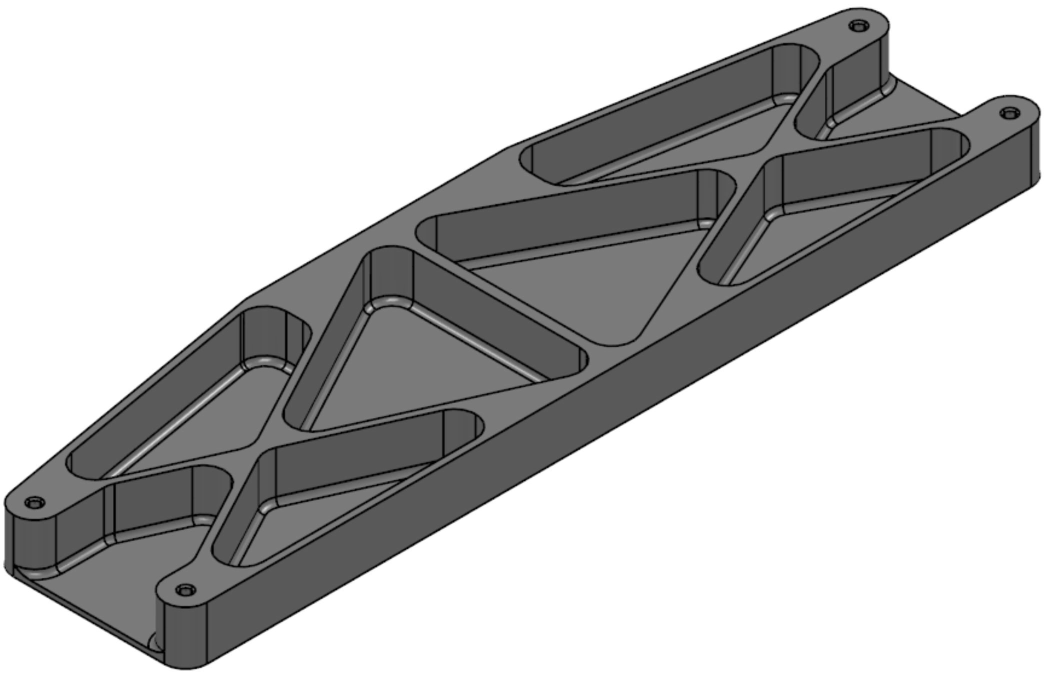
206 In the Pattern dialog select the **Rectangular** tab and create a **2 x 2 pattern** using the **dimensions** we **calculated previously**.



207 Click **OK** in the **dialog** to create the **pattern** of **features**.

*This has created a **pattern** based around the **Hole feature** we previously had made. This means any **edits** in the future can be done on **one feature** and **applied to all** in the **pattern**.*

208 Select an **Isometric View** (**Ctrl+1**)



We have now completed the solid model.

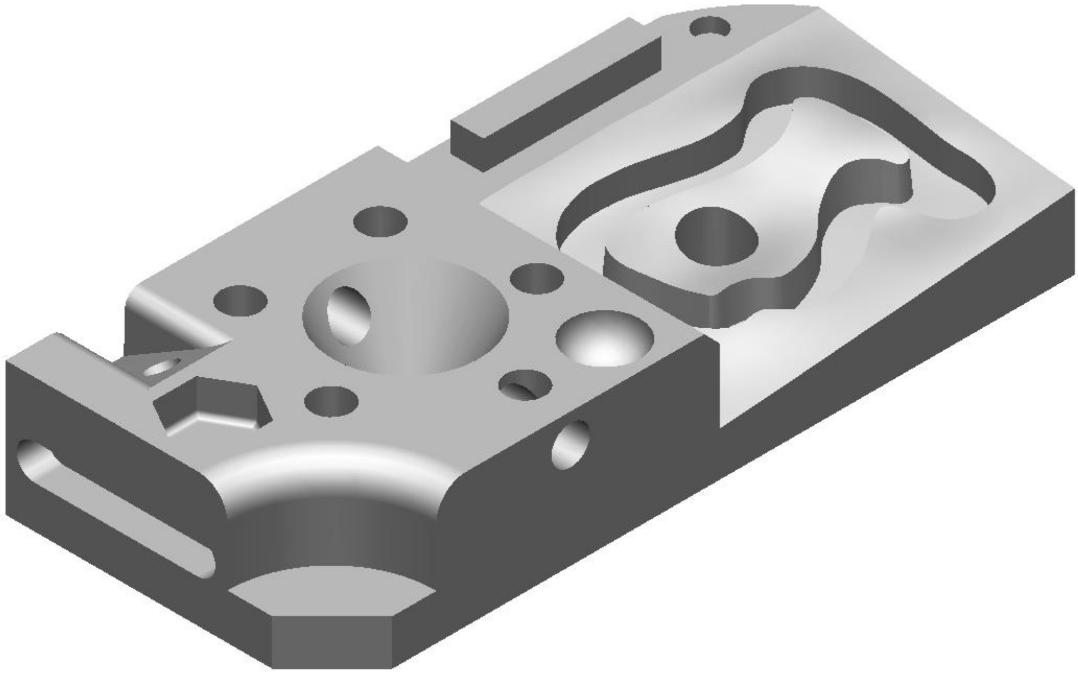
Save the model as **bloodhound steering support.psmodel** in your **desired location**.

13. Reverse Engineering

Introduction

In this tutorial we will look at the many tools available within **PowerSHAPE Pro 2015** for the purpose of, and to aid with the **Reverse Engineering** of a part from **scan** to a **complete solid model**.

The part used for this **tutorial** is a **Delcam demonstration block** which will allow us to **highlight** all the **tools** available while guiding the user through most **features** and **issues** commonly seen during the **reverse engineering process**.



Importing the Point Cloud

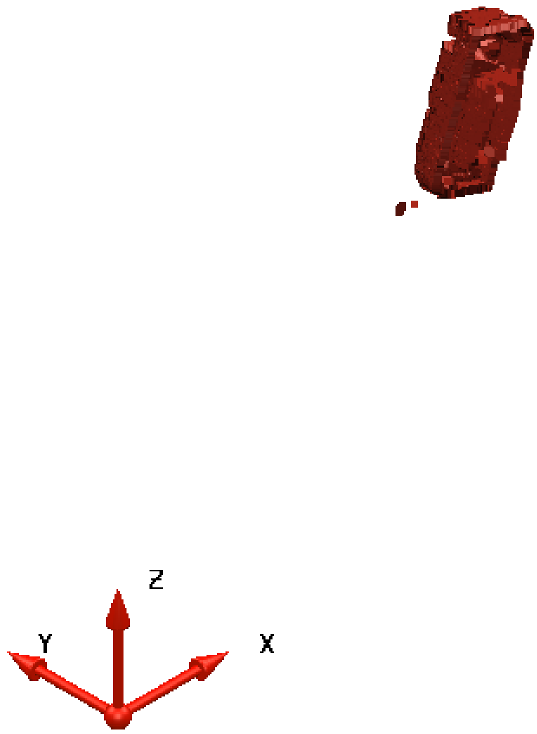
We will start with **point cloud data** just as would be **standard**, **exporting** from the **metrology software** used to **collect** the **data**, or by using the **PowerSHAPE Pro laser scanning** and **probing** ability to collect point data.

1 Import the file **re eng point cloud.asc** located in:
C:\\Training Data\PowerSHAPE Pro data

2 Select the **Workplane Icon** from the **Main** toolbar, and then in the **command box** type **0** and press **Enter**.

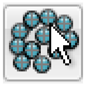
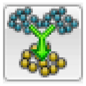
*This will place a **workplane** at **0, 0, 0** (**World**) to highlight where the **world datum** is **located**. We can either **align** the part back to the **world** or to **any other location** and **export** back to the **part datum**. We will look at **workplanes** and **alignments** shortly in the course.*

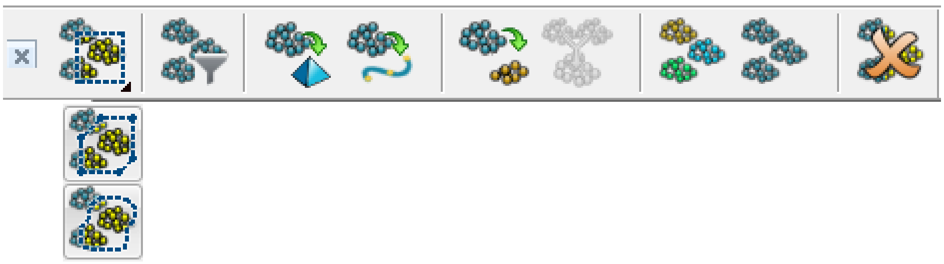
3 Select an **Isometric View** (**Crtl+1**).



*As you can see the **scan data** of the part is positioned away from the **workplane origin** and orientated off axis. Both these issues will require consideration by aligning the part, this is easier once we have created a **mesh** from the **point cloud**.*

4 **Zoom** into and **select** the **point cloud**, this will **highlight** as **green** in the **graphics area** and also will activate the **Cloud Edit** toolbar.

*The data may consist of more than one point cloud, if this is the case use the **Quick Select ALL Clouds***  *function from the **selection flyout menu**, and then the **Combine Clouds***  *from the **Cloud Edit** toolbar, below.*



 Pick Points in a cloud by box

 Pick Points in a cloud by discrete lasso

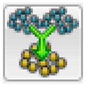
 Pick Points in a cloud by continuous lasso

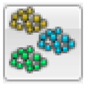
 Filter points in the selected cloud

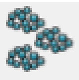
 Generate Mesh

 Generate Curve

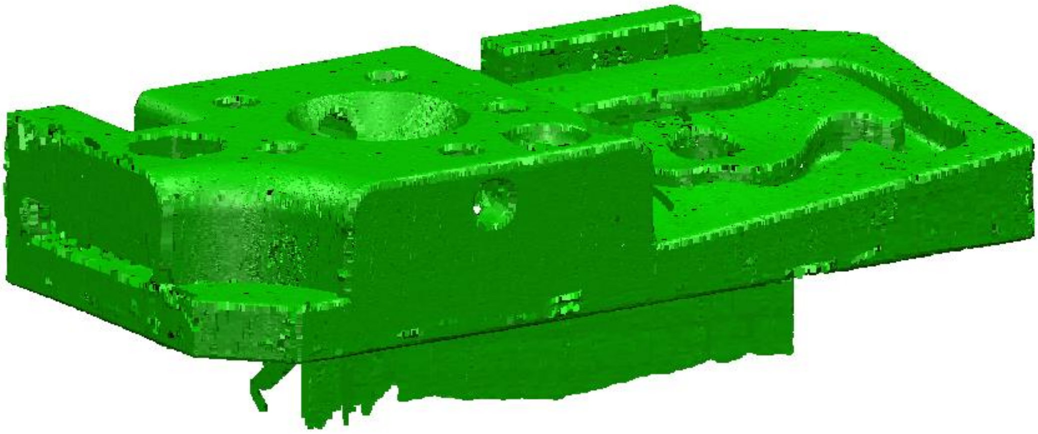
 Divide cloud by selection

 Combine clouds

 Colour clouds uniquely

 Reset to default colour

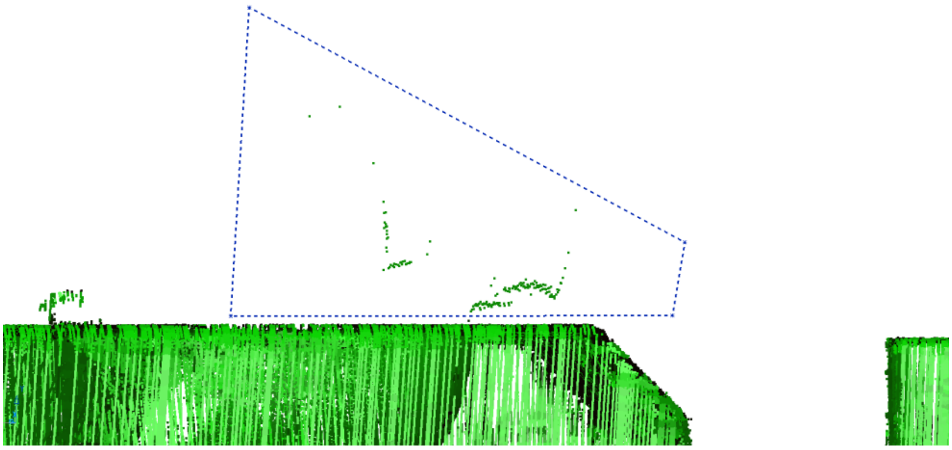
 Delete selected points’



As you can see the **cloud** contains some **noisy data** and **extra data** from the **fixture** used to raise and hold the part to allow a more **complete scan** to be taken. We will now **remove** any **data** that we do not want to take forward.

5 **Rotate** and **Zoom** into the **anomalous data** above the **hemispherical cut** on the **top face** of the part.

6 From the **Cloud Edit** toolbar select **pick points using discrete lasso**  (any preferred pick option may be used at this point). Then carefully **select** the **extra data** to be **deleted**.

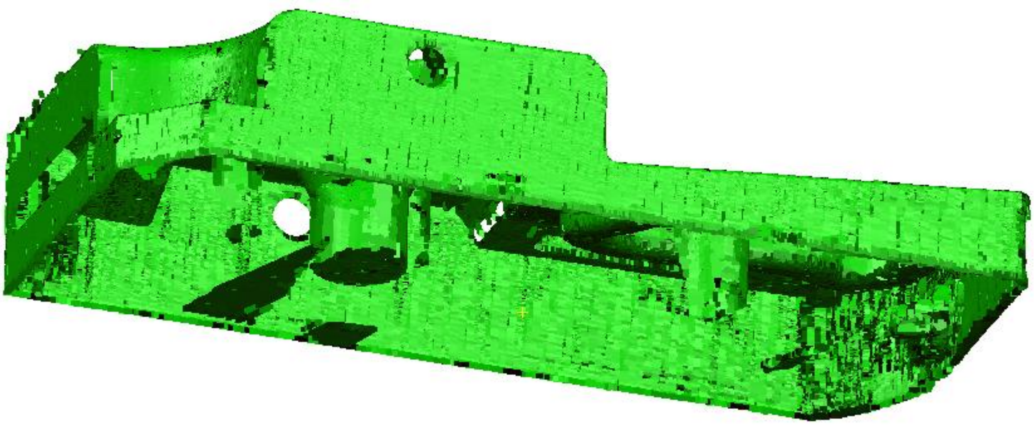


7 Select the **Delete Selection**  icon from the **Cloud Edit** toolbar.

8 **Repeat this process** for **all other extra data** that would you not like to be used in the **generation** of the **mesh**. Try using a variety of the different **point selection methods**.

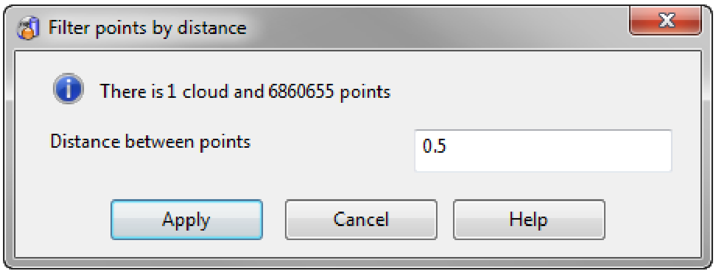
> ***NOTE: Limit Selection** may also be used to remove the **cloud data** underneath the part caused by the **fixture**.*

9 **Rotate** the **part** to allow us to view the **underside**. Note that we can still see a small amount of data that shows **capture** of the **bottom face** of the part. There is **no need to delete** this data as it is useful to the **reverse engineering process** as it allows us to **measure** the **depth** of the **part**.



10 From the **Cloud Edit** toolbar select **Filter points in the selected cloud**.

> *This dialog is used to **reduce the density** of **points** in the cloud by introducing a **minimum distance between each point**. There are currently slightly over **6.8 Million points** in the cloud.*



11 Enter a **minimum distance between points** of **0.5mm** and then select **Apply**.

*Note in the dialog that the **cloud** has been **dramatically reduced** down to **140,000** points **without any noticeable loss of detail**. This will help **reduce** the **complexity** and **processing time** required to **generate the mesh**.*

12 **Cancel** the **dialog**.

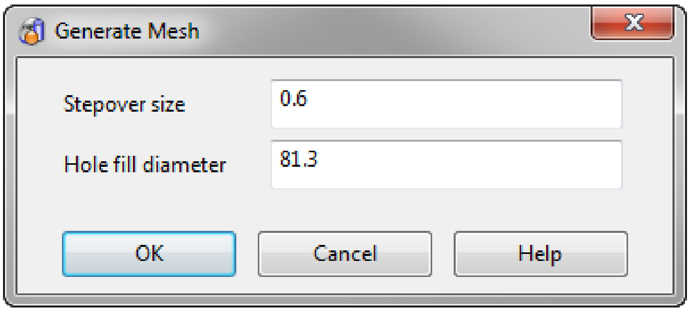
Next we will **generate** a **mesh** from the **point cloud**. However first it is **good practice** to **save the session**.

13 **File>Save As...** Choose a name relevant to the **part** & **stage** we are currently at **in the model**. **Eg Demo block point cloud.psmodel**

14 Select **Generate Mesh**  on the **Cloud Edit** toolbar.

*The **default values** in the **Generate Mesh** dialog are designed to **minimise processing time** and may **not provide satisfactory results**.*

15 Inside the **generate mesh** dialog change the **stepover size** to **0.6mm** (just large than the filter points distance). Leave the **hole fill diameter** as default in this case.

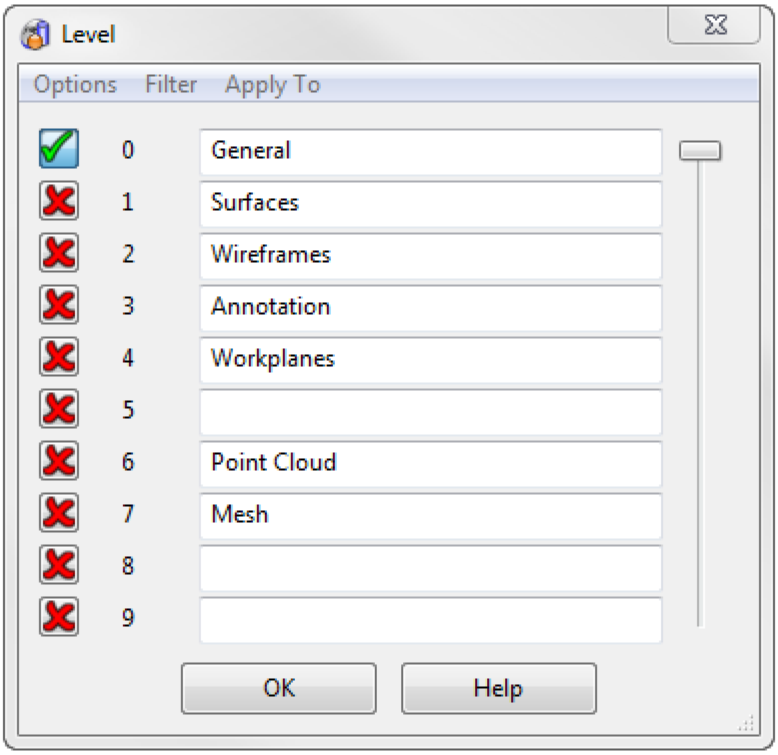


*If the **mesh generation** creates a **Reversed** (**Red**) mesh. It is possible to **reverse the mesh** by selecting the **Reverse Meshes***  *option from the **Mesh Edit** toolbar.*

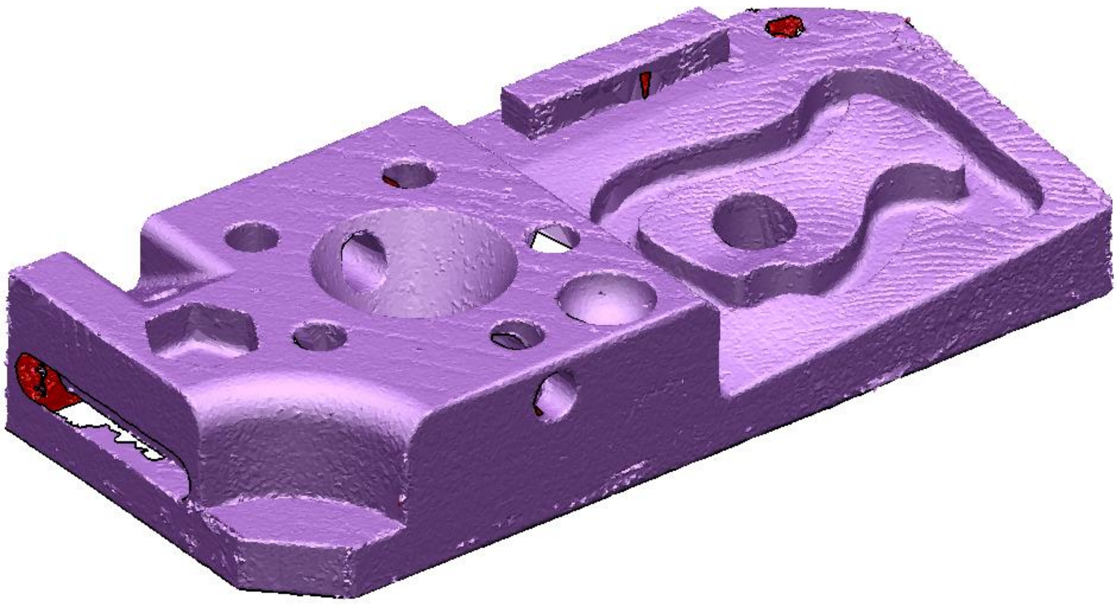
*Feel free to **experiment** with these **values** to get a feel for their response to edits.*
***Stepover: -** specifies the **size of the triangles in the mesh**. Smaller **Stepover** values give a finer and smoother mesh.*
***Hole fill: -** specifies the **maximum diameter** of **holes** that should be **filled automatically**. The mesh generator will attempt to fill holes that have a diameter equal to or smaller than the **Hole fill** value.*

16 From the **Levels** toolbar at the **lower left hand corner** of the window raise the **levels dialog**. 

17 Create a **New Level** on number **6** called "**Point Cloud**" and another on number **7** called "**Mesh**" and then click **OK** to **close** the dialog.



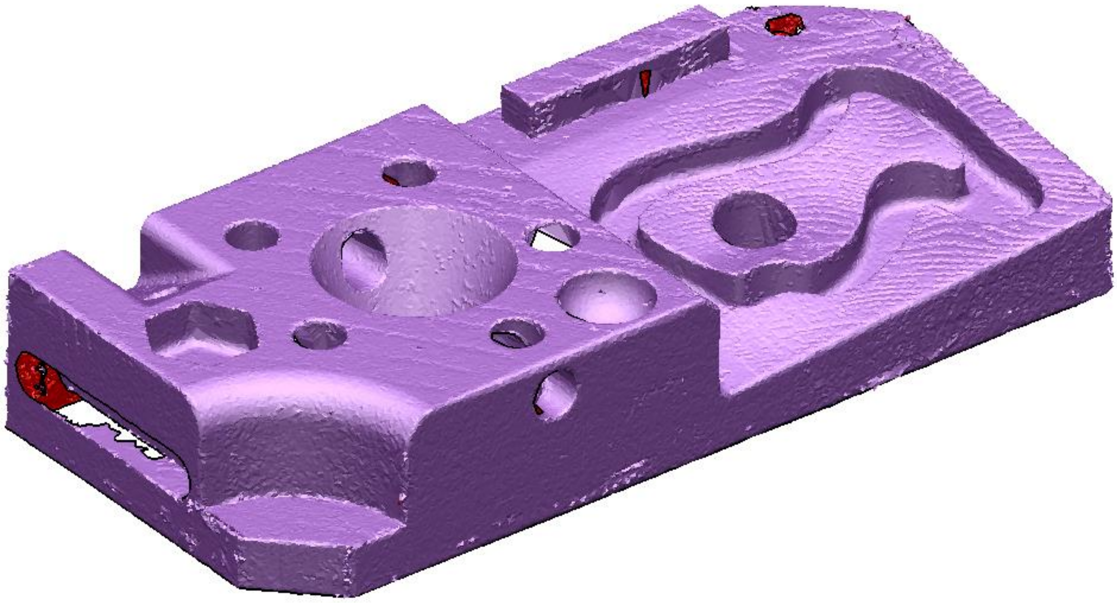
18 Place the **objects** on the **respective levels** and then switch the **mesh** (**level 7**) **ON**.



A **mesh** has been created from the **filtered point cloud** using a **stepover size** of **0.6mm** and a **hole fill diameter** size of **81.3mm**. As you can see **PowerSHAPE Pro** has created a **fine** & **clear mesh** which will make the entire **reverse engineering** process much simpler. There are still some relatively small areas of **noise** and **missing data** on the **mesh.** This is mostly locate inside the holes. This issue is **very common** on **scanned data** and at this point it is necessary to decide whether any of this will cause errors later, while considering the **design intent of the model**.

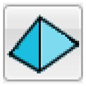
Reverse Engineering Notes

Before we go any further on how to use the **reverse engineering** & **modelling** tools available in **PowerSHAPE Pro**, use this page to have a look at the **mesh** and make **notes** on what you believe, using your knowledge, is the most **efficient** way to **remodel this part**.



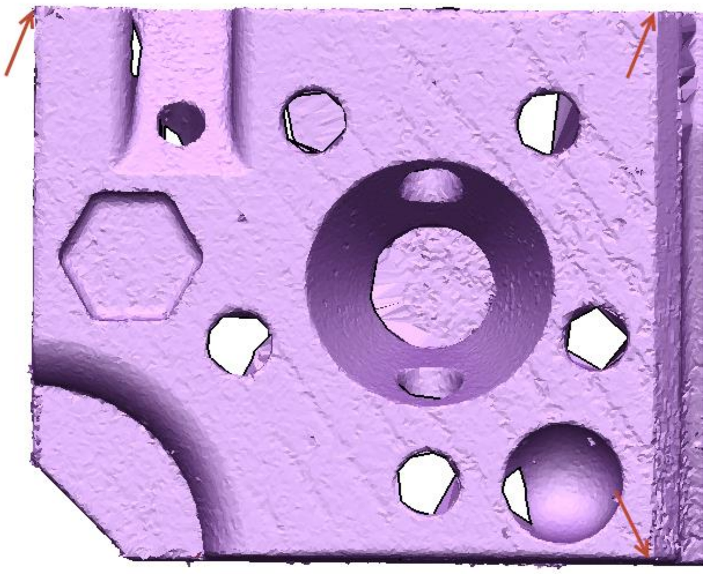
Aligning the Mesh

Correctly **aligning** the **mesh** is important to the entire **Reverse Engineering** process and ideally we should use the **largest area** and **simplest shape** to give the **best results**. This will allow the **Mesh Segmentation** tools to automatically adjust **primitive shapes** to the **axes** of the **active workplane**.

1 From the **Mesh Shading** options on the **View/Shading** menu on the **right hand side** of the **graphics area** select the option to show **Open and Interior edges, shaded triangles.** 

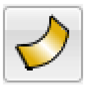
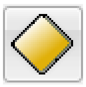
*This will **show** the **edges** of the **inner triangles** allowing better **clarity** when **selecting** areas **inside** the **mesh**.*

In this example we will align with the **raised step face** on the **left hand side** of the **mesh**. We have **3 clear** and **definable corner points**, which we can **easily align** with a **planar surface**. We can align objects using many **primitive shapes**. However the **defined corner points** of a **plane** makes it the ideal tool in this case. As this time we are using a much **larger area**, but only have **3 corners** on the **mesh** we should produce an **accurate alignment**.



*The image above highlights the **3 available corner points** located on the **upper face** of the **mesh** as a **rectangular formation**. The **alignment tools** use a **minimum** of **3 points** for each **object** to create an **alignment** so this will suffice.*

2 Ensure the **Principal Axis** in set to **Z**.

3 From the **Surface**  menu select **Create a Planar Surface**. 

4 Place the **surface** on the **workplane origin** by typing **0** in the **command box** and then press **Enter**.

5 **Double click** on the **surface** to open the **surface edit** dialog.

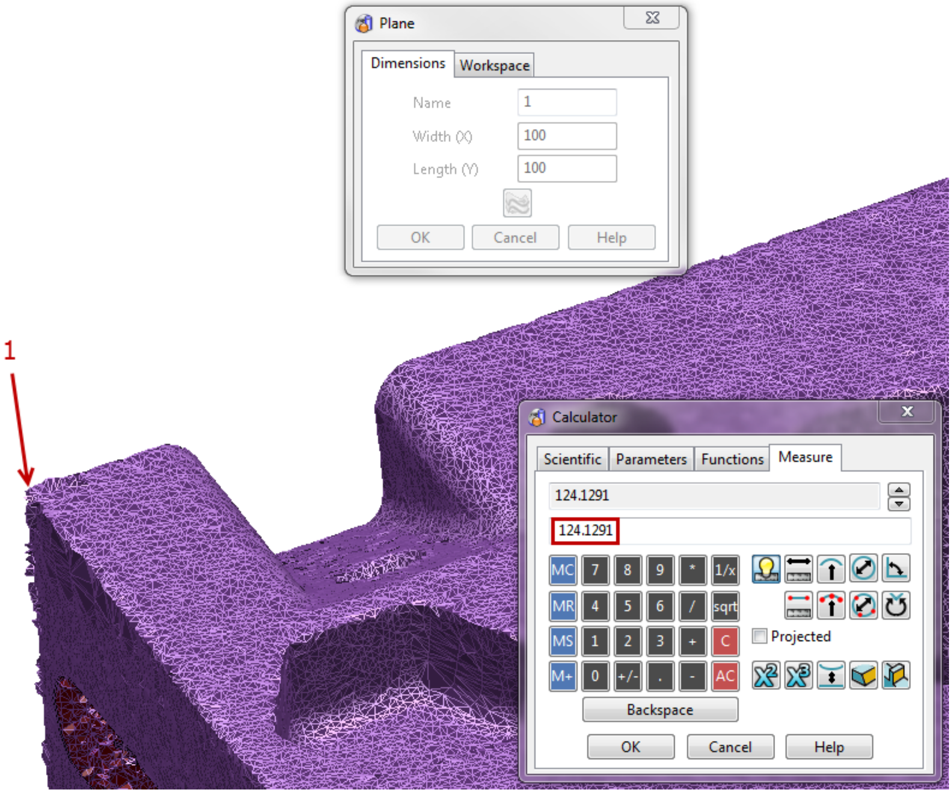
6 **Zoom** into the **top face** of the **mesh** shown above.

To **resize** the **surface** using the **measuring tool** directly we can **right-click** inside the **text box** adjacent to the **width** or **height** in the dialog. This will then open the **calculator** and allow **measurement** of the **correct length** and **automatically fill** the **respective box**.

7 **Right-click** in the **Width text box**.

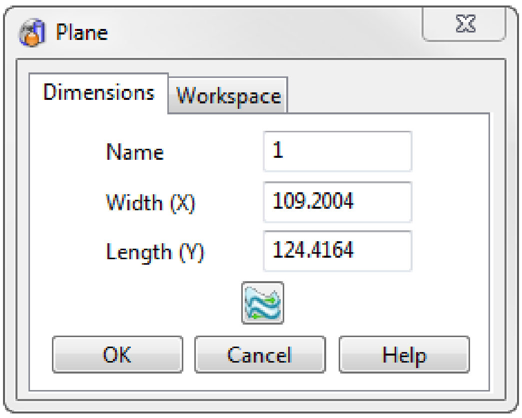
8 **Measure** the **Width** by clicking on the **corners** of the **face** in the **graphics area**.

*Try to remember which **corner triangles** you selected as clicking these exactly again during the **alignment phase** will produce the **best result**.*

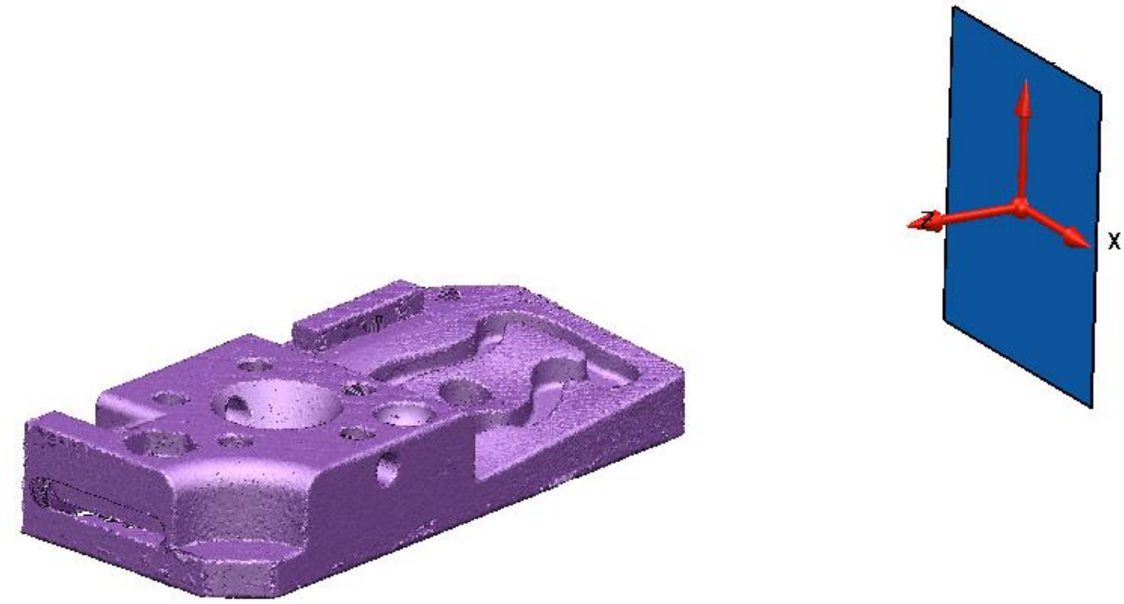


9 Click **OK** in the **Calculator** to use the **measured value** as the **Width**.

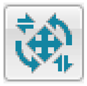
10 **Repeat** the same **process** for the **Length** of the same face to fully **resize** the **planar surface** to **fit**.

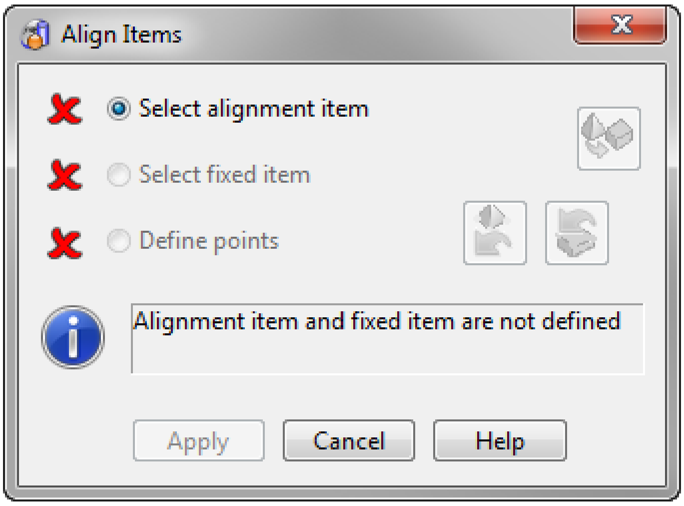


*If your **values** are **slightly different** to the **values** in the image above **do not worry**. Again, the **best alignment** will rely on selecting the **same corner positions** as you did to **resize** the **surface** when **aligning**.*



In the **graphics area** we now have the **mesh** created from the **imported/scanned point cloud** and an **alignment surface** sized to fit the **face** we have chosen to use to help **align** with **planar surface**, and therefore the **origin** of the **active workplane/world datum position**.

11 Select **Align Items**  from the **General Edits**  menu (This is located in the **flyout** menu with **Move**).



- The **Alignment Item** is defined as the **object** that **will move** during the **alignment operation**.
- The **Fixed Item** is defined as the **object** that will **NOT move** during the **alignment operation**.

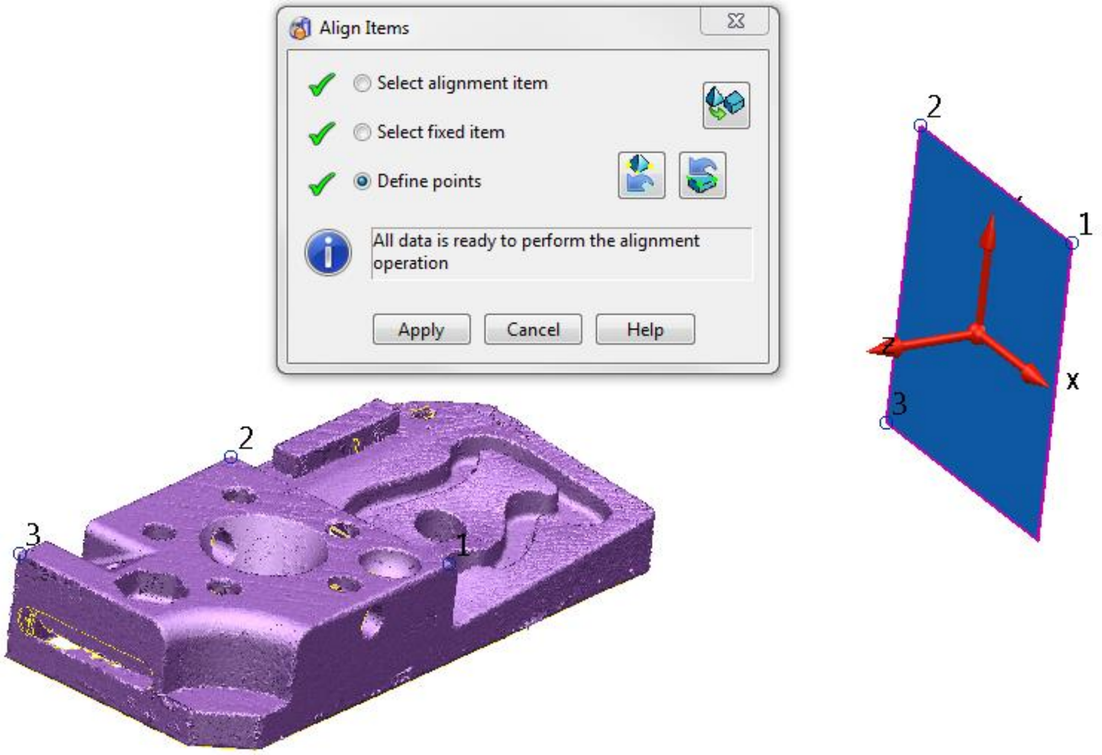
*If the **alignment** and **fixed items** are far apart in the **graphics area** this may cause problems while **navigating the view**. It is possible to simply **drag** move the **alignment item**, in this case the **mesh closer** to the **surface**.*

12 Click on the **mesh** to select it as the **alignment item**, then select the **surface** as the **fixed item** in the same way.

Next, we have to select a **number of points** on the **alignment item**, and then repeat the **same selection** on the **reference item**. Note that the **order** and **amount** of the points selected must be the same for each object. For a **rectangle** a common selection order is **"clockwise, width first then the length"** this way you know that the **selection** will **always** be the **same**.

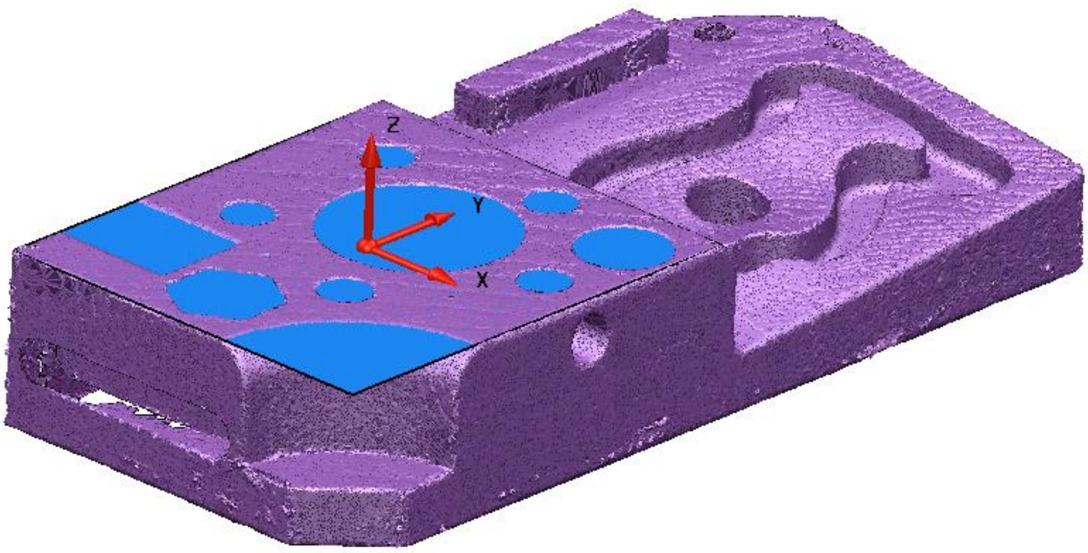
13 Select **3 points** around the face of the **Alignment Itme**, one in **each corner** on the mesh and then repeat the selection on the **Reference Item**, the **surface**.

*Note: The **order** and **direction** of your **selected points** must be the **same** for **each object**.*

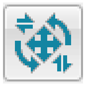


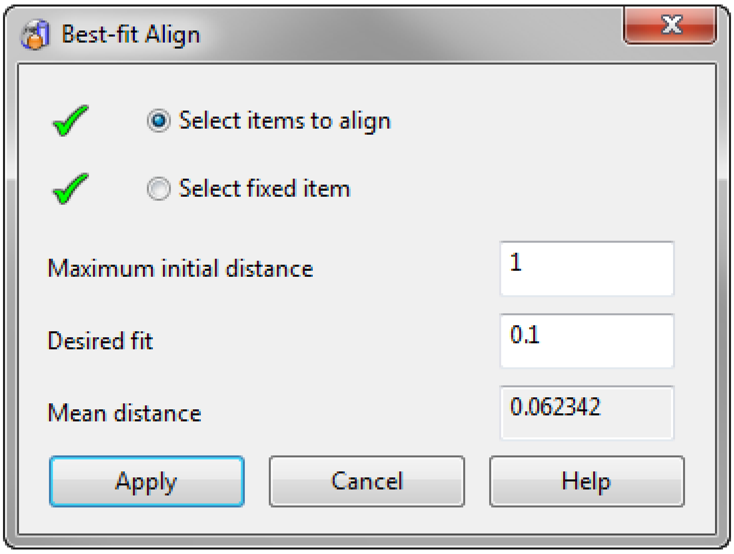
14 When your **selection** is **complete** and **correct** all the **red crosses** will become **green ticks** and the dialog will tell you that you are **ready to perform** the **alignment operation**.

15 Select **Apply** and **Cancel** to **close** the dialog and view the **results**.



As you can see we have **successfully aligned** the **mesh** with the **surface** and the **origin** of the **active workplane.** Further to this we can **fine tune** our **alignment** using the **best fit alignment**  tool from the same menu.

16 From the **General Edits**  menu select **Best Fit Items.** 



17 In the **Best-fit Align** dialog select the **Mesh** as the **item to align**, select the **surface** as the **fixed item**.

18 Click **Apply**, the **mean distance** is the **average movement** of the **mesh** to further **best fit** with the **surface.**

19 Select **Cancel** to close the dialog.

20 Raise the **Levels** dialog to create a **new level** on **Level 997** called "**Alignment Surface**".

21 Select the **surface** we have used **align** the part in the **graphics area** and place it on **level 997 : Alignment Surface**.

22 **Turn OFF Level 997 : Alignment Surface**.

*The surface we have used for the **alignment** will unlikely to be required again, but is **useful to keep** as backup. Placing it on the **final available level** will mean that although still there, it will always appear at the end of the **Levels** toolbar, out of the way.*

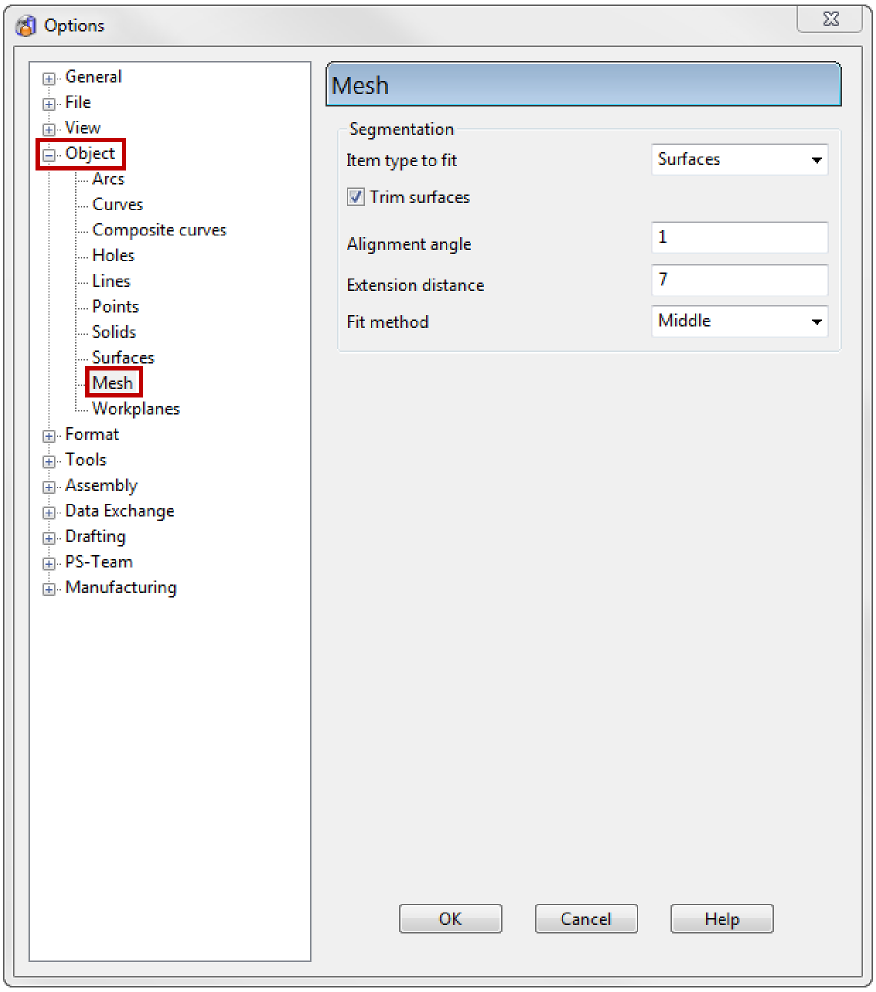
As the **mesh** has been successfully **aligned** and further **best fit** to the **surface** we can now begin to **Re-Engineer** the **part**.

Reverse Engineering the Block

We will start by using the **Automatic & Manual Mesh Segment** tools to find all the **planar surfaces** to enable the **main shape** of the **model** to be **created**. This and other tools will be used to create the **part**.

First we will look at and **setup** the **default options available** to aid us using the **Mesh Segment** tools inside the **Tools>Options** area.

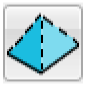
23 Navigate to **Tools>Options>Object>Mesh**.



*This area holds the **mesh options**, more specifically the **options** relating to the **Mesh Segmentation** tools and how they interact with the **mesh** and **workspace**.*

- **Item type to fit**: This option allows the user to fit **surfaces** or **solid models** when using the **Automatic** or **Manual Mesh Segmentation** tools.
- **Trim Surfaces**: This option allows you to toggle between creating **trimmed** or **full surfaces** when **segmenting** the **mesh**.
- **Alignment Angle**: This value will define the **threshold tolerance** (in degrees) for **aligning primitives** with a **principal plane**. If an **axis** of a **primitive** is **less** than this angle away from a **plane** of the **active workplane**, the axis of the **primitive** is **snapped** to that plane.
- **Extension Distance**: The value to define the distance by which a primitive is **extended** beyond the **edge** of the **selected region**.
- **Fit Method**: This defines the **location** in which the **primitives** are **fitted** to the average depth of the selected region; **Inside, Outside** or **Middle**.

1 From the **Mesh Shading** options on the **View/Shading** menu on the **right hand side** of the **graphics area** select the option to show **Open Edges Only**.



2 Select an **Isometric View** (**Ctrl+3**).

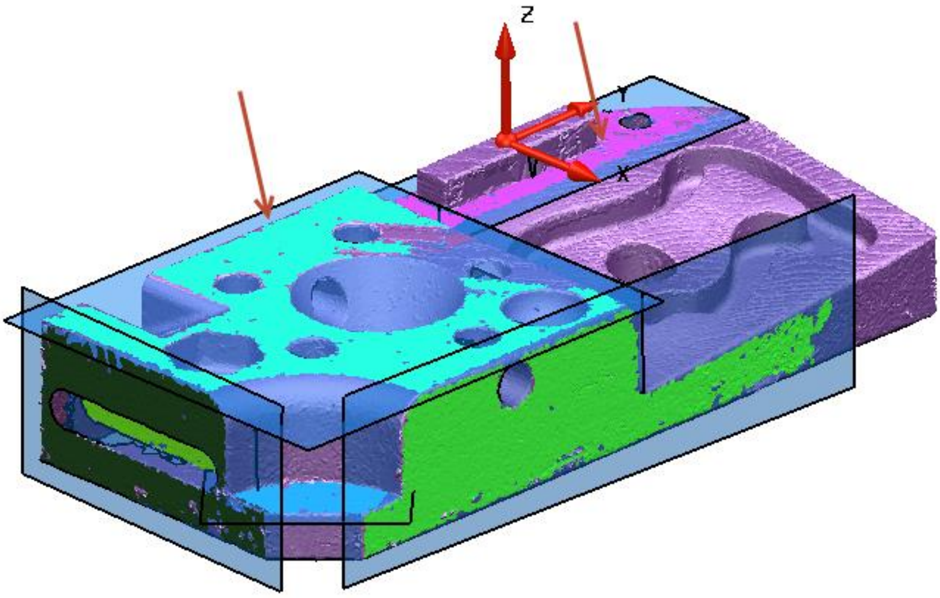
3 Select the **mesh** in the **graphics area** to **activate** the **Mesh Edit** toolbar above, and then open the **Automatic Mesh Segment** tool. 

4 As we are currently only looking to create a **basic shape** to work from, in the **primitive types** area select only **Planes**.

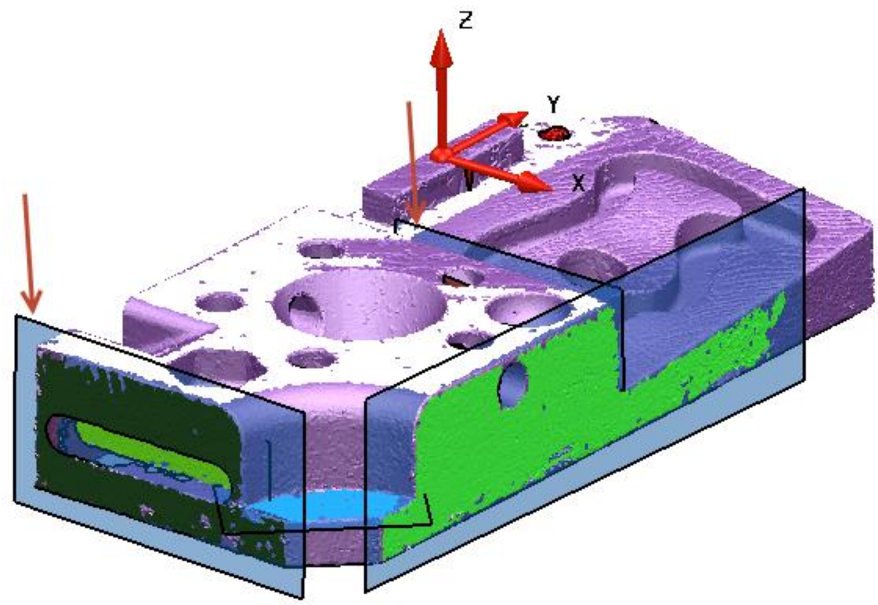
5 With the **default values** select **Preview**.

6 Select and **Apply** the **two upper faces** of the mesh indicated in the **next image**.

In this case we will look at the **freeform surface** and other **features** later as currently we are only looking to create a **basic block shape**. As **primitive surfaces** are **created** we can later **extend** any **shorter** surfaces to **suit our requirements**.

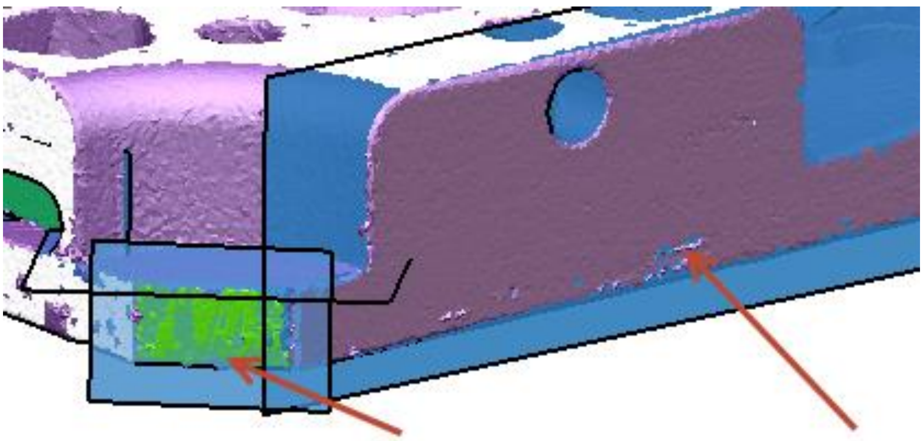


7 Next, select and **Apply** the next **two vertical surfaces** shown in the **next image**.



8 Inside the **automatic segmenter** dialog **Adjust** the **Similarity angle**, until you have **selected all** the **triangles** for the **face**. A **value** of **2** gives a good result in this case, and then select **Preview**.

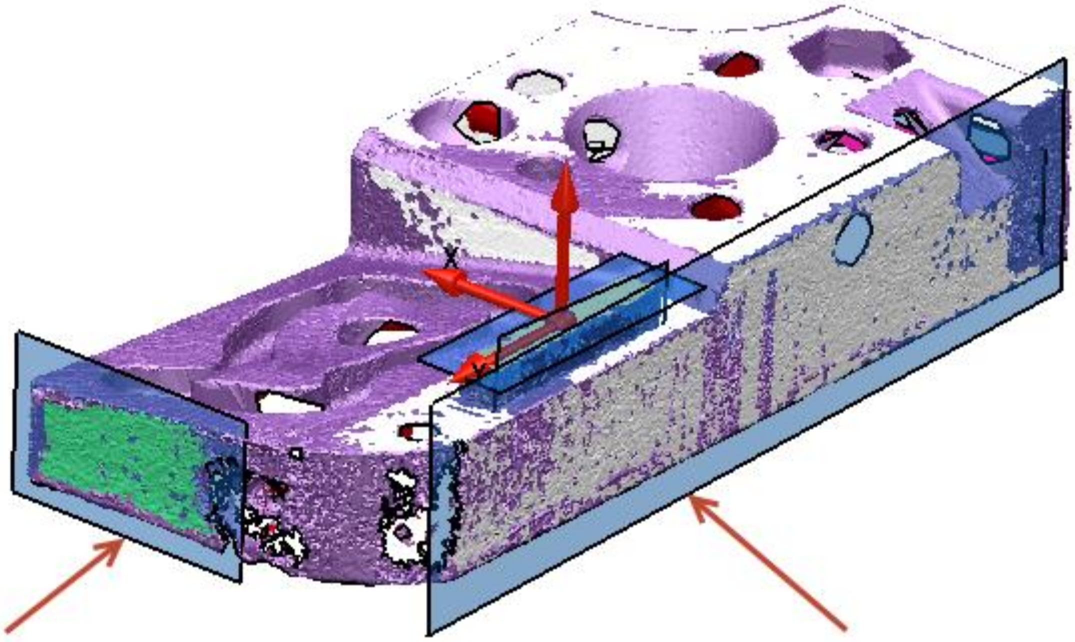
9 Select and **Apply** the **long edge surface** as well as the **intersecting cut edge surface indicated below**.



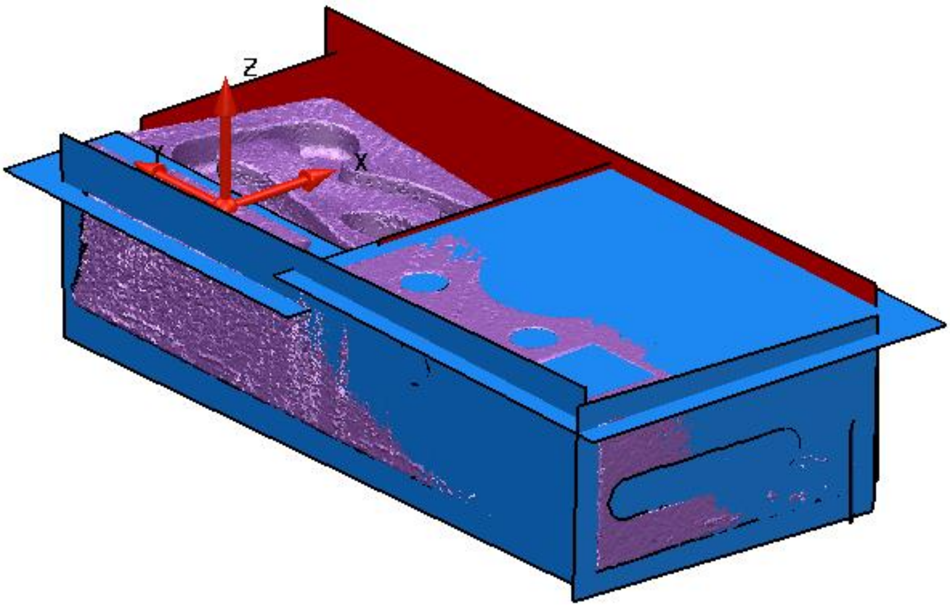
10 **Adjust** the **Similarity angle**, until you have **selected all** the **triangles** for the **face**. A **value** of **6** gives a good result in this case, and then select **Preview**.

11 In the **image below** are **two further surfaces** to create the **main outer shape** of the **model** from the **mesh**. **Select** them and click **apply**.

*These surfaces **may not intersect currently**, however as mentioned earlier, we can simply **dynamically drag edit** them in the **graphics area** later to suit. Due to the **coarse area** of the **mesh** and the **cut corner** we will have to compromise in this case.*



12 In the **dialog** select **Cancel** to **close**.



The **only surface** for the **main shape** we are currently **missing** is the **bottom face** of the part. In the case of this **mesh** we have some **data** collected from the **scan** through the **large hole** through the part. To **extract** this **surface** we will use the **Manually Segment Mesh**  tool.

If you are **missing any** of the **surfaces** the example has created above, you can use this tool to **manually create** them too.

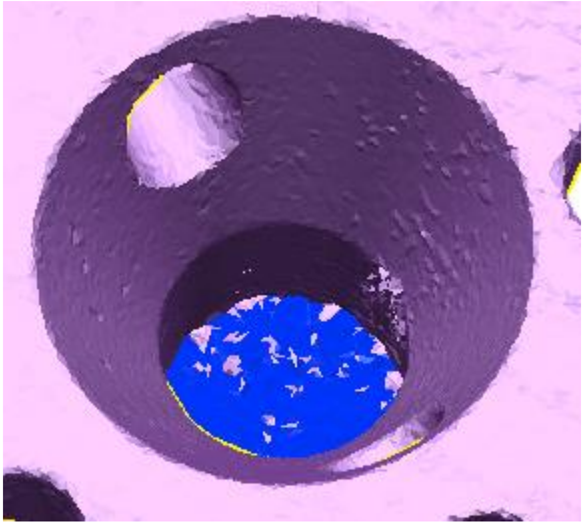
13 **Quick Select ALL Surfaces**  from the **Selection** flyout menu.

14 **Blank** (**Ctrl+J**) the **selected surfaces**.

15 **Rotate** the **view** to look **down** to the **bottom** of the **hole**.

16 Select the **mesh** and then open the **Manually Segment Mesh**  tool from the **Mesh Edit** toolbar.

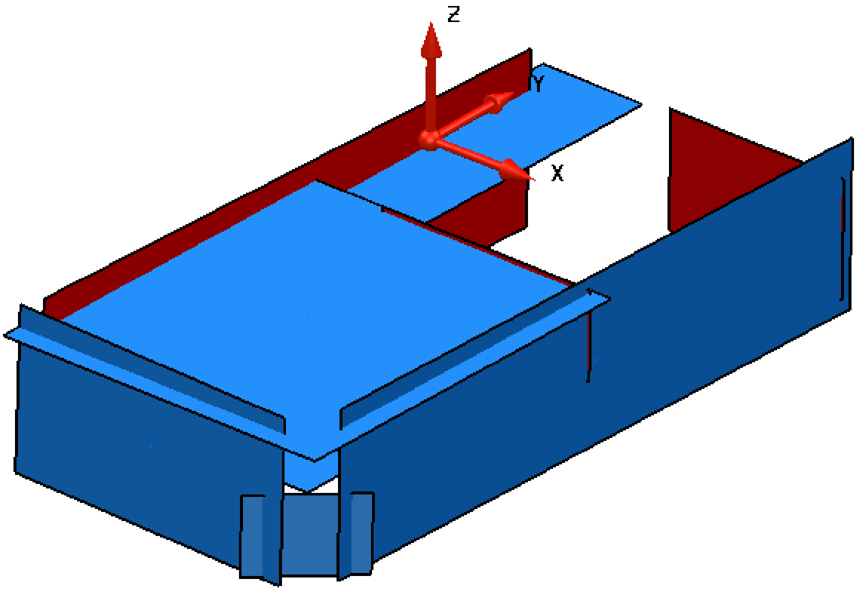
17 With **Type: Plane** chosen and the **default horizon angle** select the area at the **bottom** of the **hole**.



18 Click **Preview** and **Apply** to create the **surface** and then **Dismiss** to **close** the dialog.

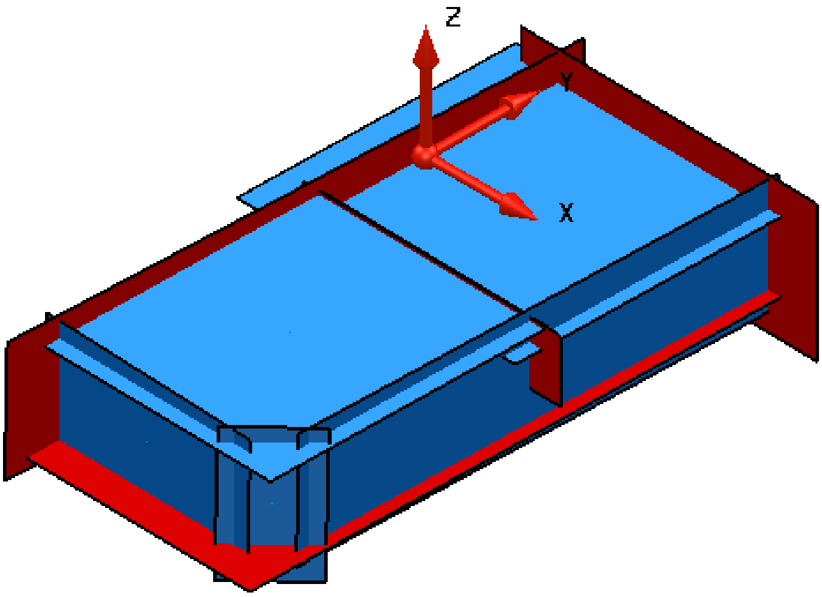
19 **Unblank** (**Ctrl+L**) the other **surfaces**.

20 **Turn OFF** the **Level 7 – Mesh**.



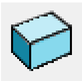
These are currently all the **surfaces** required to form the **main outer block** using the function to **Create Solids from Untrimmed Surfaces** however we need to ensure that the model has **no gaps** by **extending** the **primitives** and **orienting** them **correctly**.

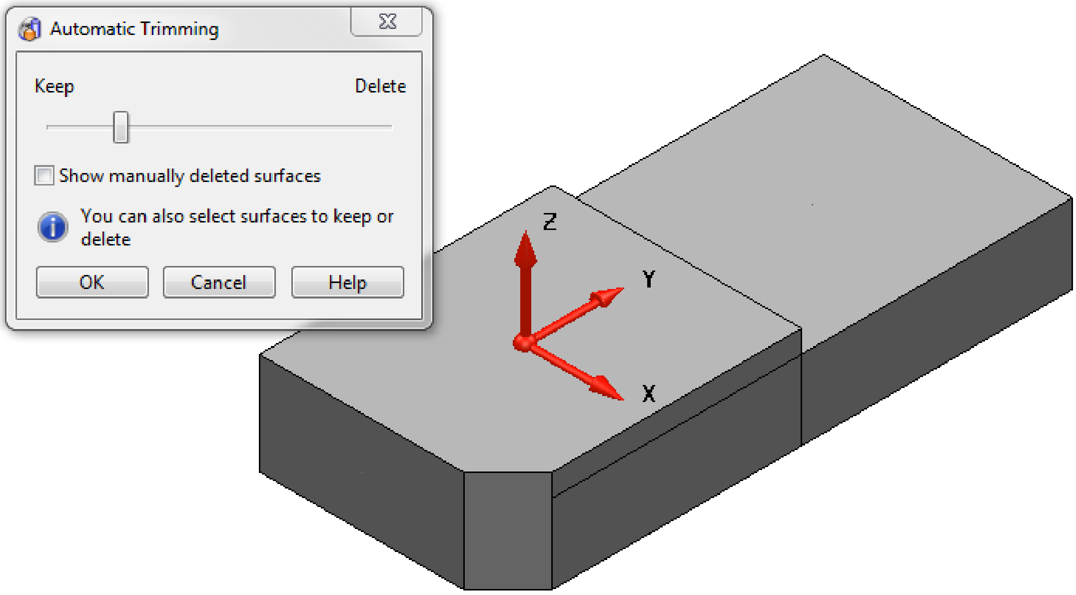
21 Extend the **surfaces** in the **graphics area** to cover the **gaps** to create a **closed region** and **orientate** the **surfaces** with the **positive "blue" side facing outwards**.



*Ideally we are looking to see a group of **untrimmed surfaces** similar to above. The **positive side** of the **surface** is **facing outwards** and all **adjoining surfaces** are **slightly overlapping**.*

22 **Quick Select ALL Surfaces**. 

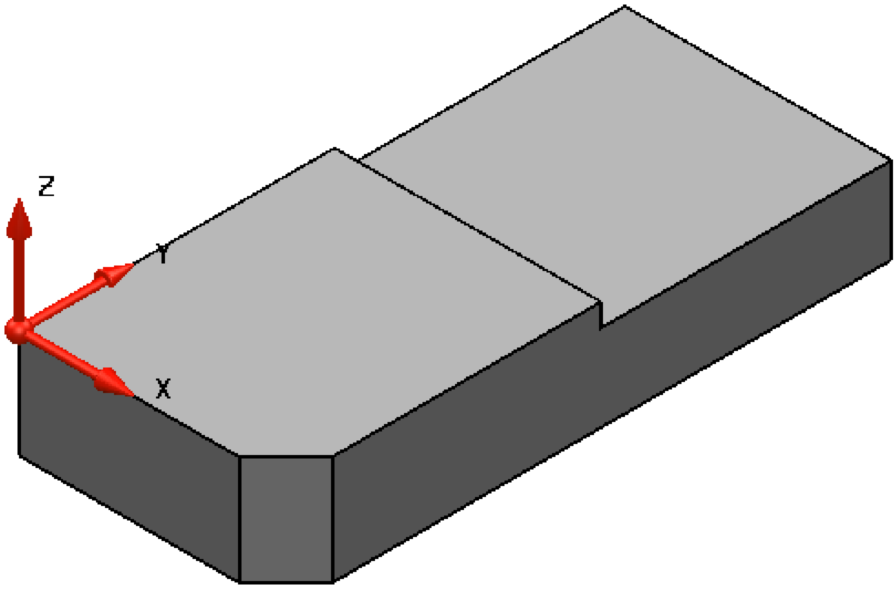
23 From the **Solid**  menu select **Create Solids from Untrimmed Surfaces**.



24 Select **OK.**

*This creates a **Solid Model** from the **cavity** inside the **untrimmed surfaces**. If **your model** is **not a complete** and **closed solid** this will be due to the **orientation** of one or more of the **surfaces**, or a **missing overlap**. **Select Undo** and **check** the **affected area** of the **model**.*

25 **Select** and **move** the **active workpane** into the **corner shown below** and then **reorientate** it to ensure that the **X** & **Y axis** are **parallel** with **both edges**, which in turn will mean that these **edges** are **perpendicular** to each other.



26 From the **Levels** toolbar create a new **Level 10 – Solid Model**.

27 Place the **Solid** on **Level 10** and ensure the **level** is **OFF**.

28 **Turn ON** the **Mesh** on **Level 7**.

As we now have created a **basic solid shape** it is **good practice** to **dimension** the part to allow us to **ensure** it is **correctly sized**. **Reverse Engineering** from the **raw data** scan of the part to an initial **solid model** will inherently introduce small errors which we can easily rectify using **Solid Modelling** and **Direct Solid Modelling** techiniques introduced earlier.

Dimensioning the Solid

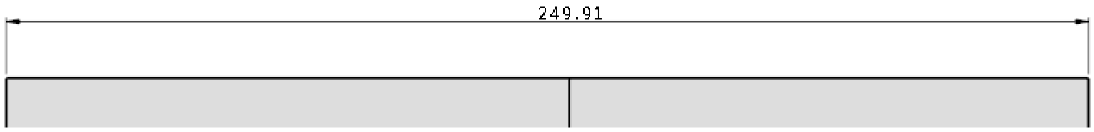
We will now **dimension** the **major elements** of the **solid** to check the **sizing**, and further **resize** the part to suit requirements. In this step, having the **physical part** and **background information** including **design intent** will help produce an **accurate part**.

29 Select the **Automatic Dimension**  tool located in the **Annotation**  menu.

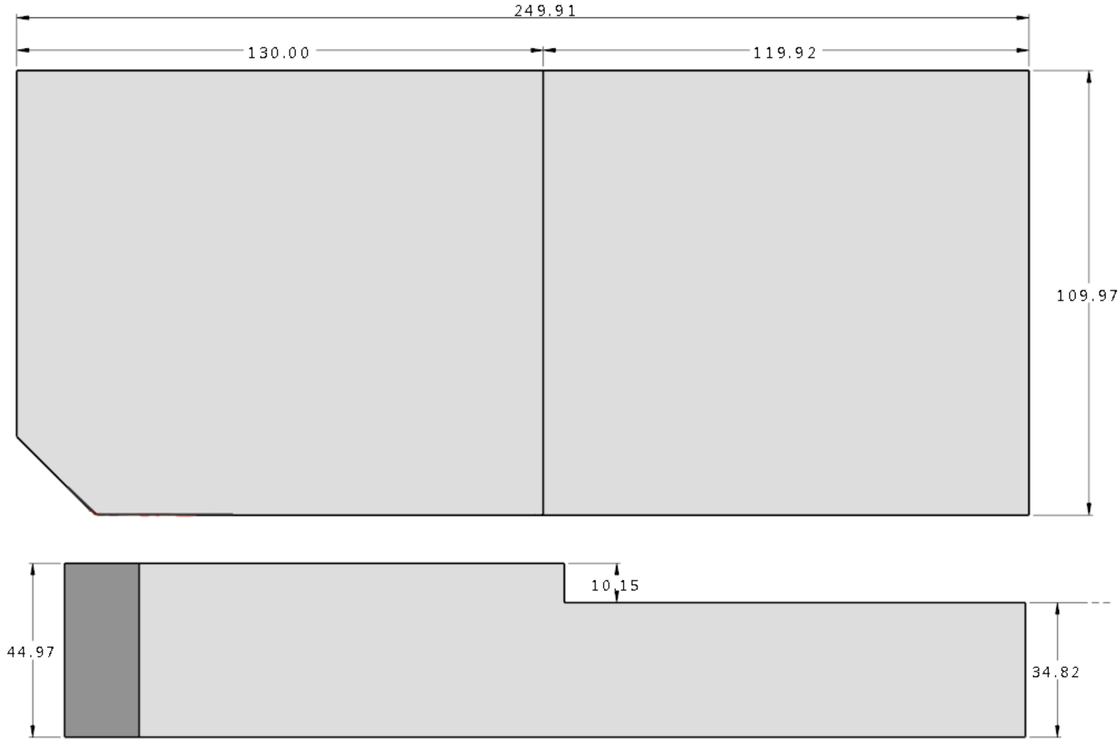
*The **Automatic Dimension** tool is a **multi-purpose dimensioning tool** allowing dimensions to be created **quickly** and **easily** in a **PowerSHAPE** session. This **tool** picks up **dimensions** by **hovering** the **mouse** over the key areas such as **solid edges**.*

30 Select a **View from Top** (**Ctrl+5**).

31 Once the **dimensions** are shown as a **preview** (in **yellow**, as above) a **single mouse click** will produce a **dimension**, while a **further click** will place the **dimension** under the **cursor**.



32 **Continue** to **dimension** the rest of the **major elements** of the **model** using the **Automatic Dimension** tool.



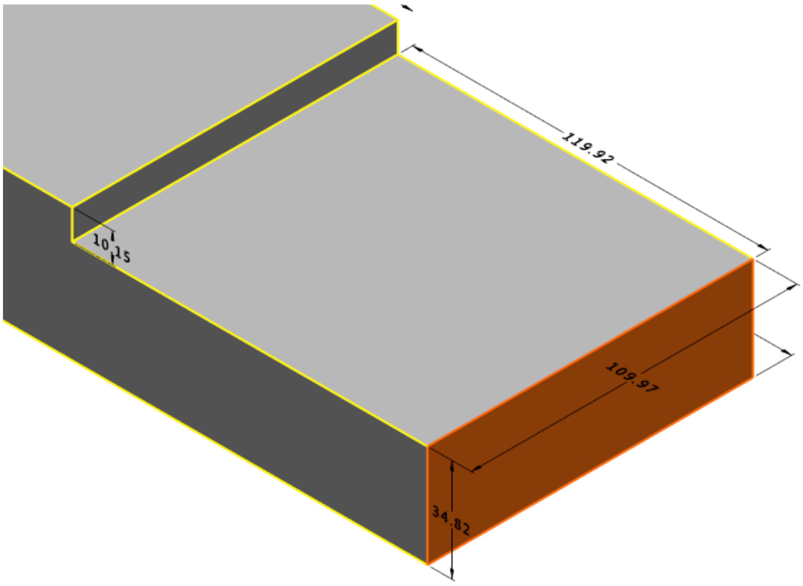
As you can see from the **above images** the data has provided a mesh that has **closely represented** the part. Any number of variables can cause these small descrepencies in **sizing.** However if we are **reverse engineering** to **recreate** the exact model we can now use the **Direct Solid Modelling** techniques shown earlier to perform **General Edits** to **individual faces** of the **solid model**.

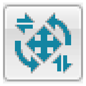
*With a **length** of, for example, **249.91mm** we can **expect** that the **orginal length** would have been **250.00mm** unless further information can **prove otherwise**.*

*We can also see that a split length of **130.00mm** and **119.92mm** would require **material** to be added at the **right hand edge face** (looking from above).*

33 Select an **Isometric View** (**Ctrl+3**) to show the **face** we need to **add material**.

34 **Select** the **Face** which will become highlighted in **orange**.



35 With the **face selected** select **Move**  from the **General Edits**  menu.

36 Inside the **Move** dialog insert a **Y movement** distance of **120.00-119.92** (**0.085mm)**.

*Dependent on the **orientation** of your **workplane**.*

37 View the **Preview** of the **Move** operation by pressing the **Enter** key. **Select** the **green tick** to **move** the **face**.

Next we will **refine** the **Width** of the **Model**. It is currently measured to be **109.97mm**. Therefore we expect an **actual value** of **110.00mm**.

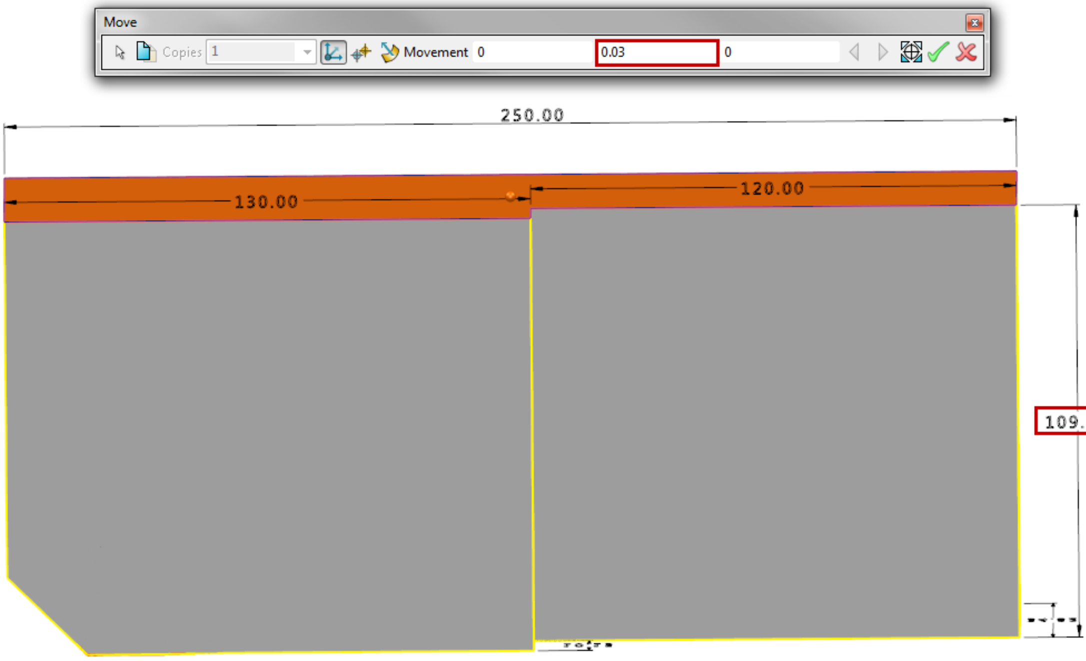
38 Select the correct **face** to move in the **graphics area**.

*This will again be highlighted **in orange**.*

39 Inside the **Move** dialog insert an **X movement** distance of **110.00-109.97** (**0.03mm)**.

*Dependent on the **orientation** of your **workplane**.*

40 View the **Preview** of the **Move** operation by pressing the **Enter** key. **Select** the **green tick** to **move** the **face**.



*The **dimensions** will **automatically update** upon completing any of the **Direct Solid Modelling** operations.*

41 To **complete** this process we need to **refine** the **block dimensions** in **Z** (**height**) also. Attempt to do this **yourself** for the **step height** and **overall height** we **dimensioned** earlier using the same **method described above**.

***Hint**: **Move** the **lower side** of the **step face** to create a **step height** of **10.00mm** and **then move** the **bottom face** of the **solid** to **refine** the **overall height** to **45.00mm**.*

This action is an important part of the **reverse engineering process** and should be completed right the way through to **feature level size** and **positioning** as will be **demonstrated later** in this **tutorial**.

Now we have the **main block correctly sized** we can start to build up **features** using the **Automatic Mesh Segment** tool where possible and then the **Manually Segment Mesh** tool.

First we will **remodel** the **main large through holes** through, one with a **conical opening**.

42 Open the **Manually Segment Mesh** tool and select to fit **primitive <u>SOLID</u> cylinders**. Choose to **recognise solid primitives** by **clicking** the **surface icon** in the dialog, which will **change** to a **solid**.

*By selecting to **recognise primitive solids** the **Mesh Segment** tools will create **primitive solids** in the **graphics area**, rather than the **surfaces** as we have seen before.*

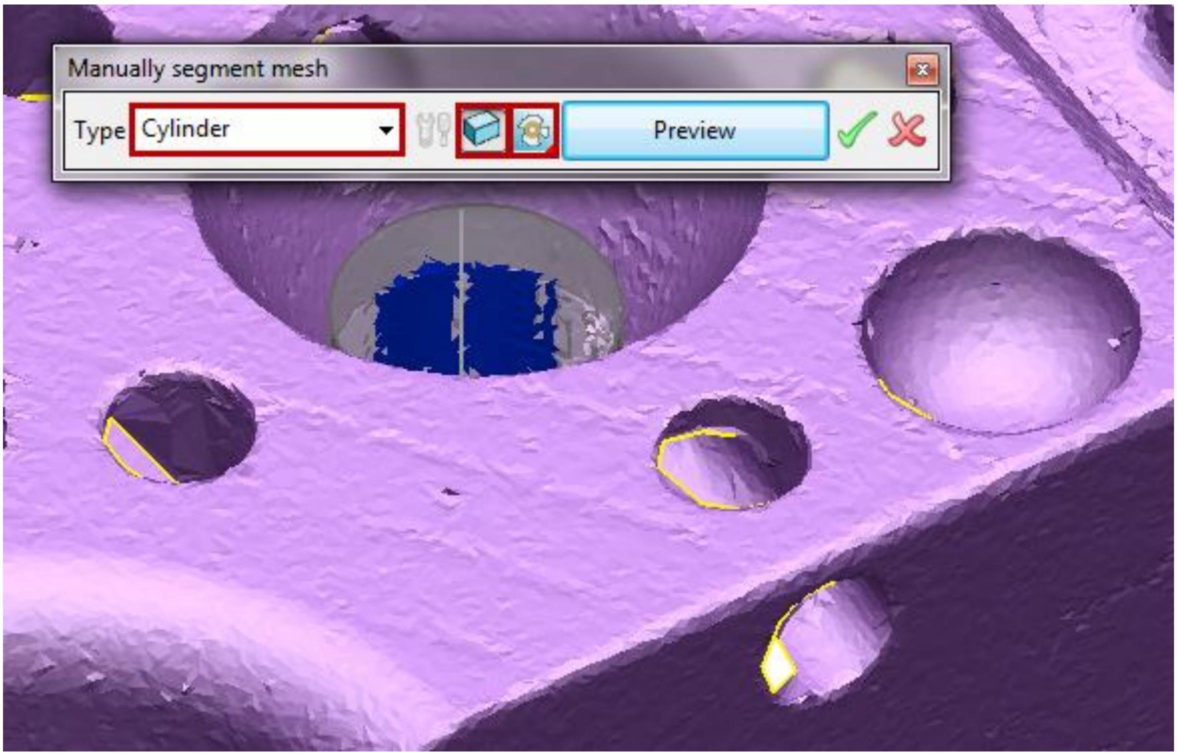
43 Select to **Fit Inside** in the dialog. This will **fit** the **primitives** just below the **surface** of the **mesh**, most useful for areas such as **holes** where the **data** can create **undersized holes**.

The options in this **flyout menu** are:

- **Fit Inside** - Select this option so that the **primitive** is fitted to lie **just below** the **inside surface** of the **mesh**.
- **Fit through Middle** - Select this option so that the **primitive** is fitted as close as possible to the **centre of the mesh**.
- **Fit Outside** Select this option so that the **primitive** is fitted to lie just **above** the **outside surface** of the **mesh**.

44 Select the **cylindrical area** of the **large countersunk hole** in the **centre** of the **raised face**. Select **Preview**.

*Holding the **Ctrl** key while selecting the **triangles** will allow you to **multi select regions**.*



45 Select the **green tick** to **Apply** and **create** the **sold cylinder**. **Right-clicking** the **solid** in the **graphics area** will also perform the **same action (Apply)**.

46 With **type** set to **Cylinder**, now select the **area of triangles** that represent the **through hole** on the **formed face** of the **part**. Select **Preview**.

47 Click the **green tick** in the dialog to create the **Solid Cylinder**.

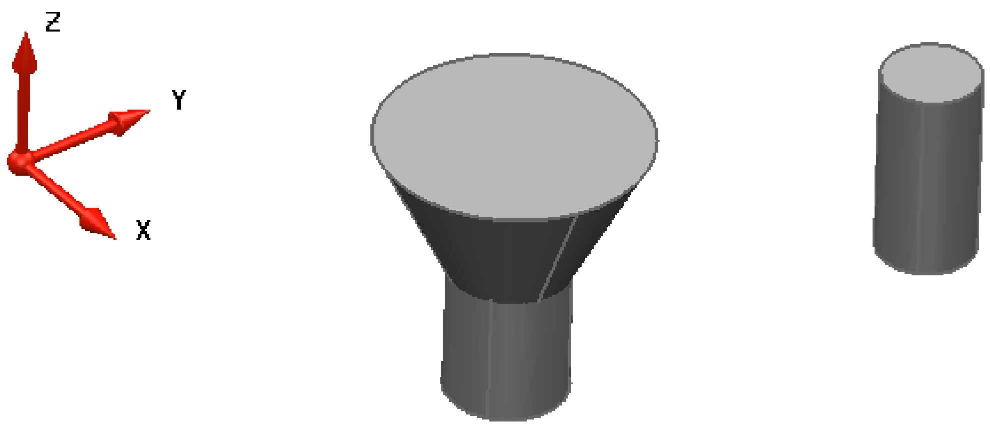
Next, we are going to create the **Solid Cone** to represent the **countersink** using the same **process** as above in the **Manual Mesh Segment** dialog.

48 In the **Manually Segment Mesh** dialog change the **Type** to **Solid Cone**. Continue to **Fit Inside**, **Solid Primitives**. Select **Preview**.

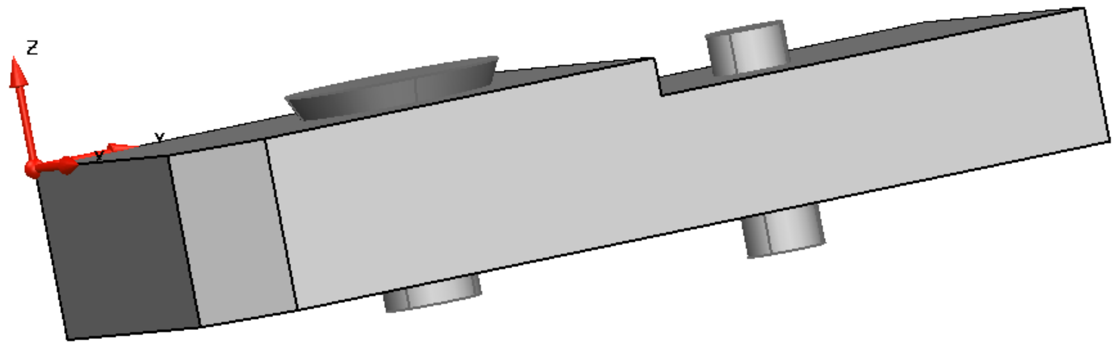
49 Select the **green tick** to **Apply** and **create** the **sold cone**.

50 Click the **red cross** in the **dialog** to **dismiss** the **form** and show the **solids created**.

51 **Turn OFF Level 7 : Mesh**.



52 **Turn ON Level 10 : Solid Model** and **ensure** the **primitive shapes extend beyond** the **outside** of the **main solid block**. If at this point they **<u>do not</u>** you can **extend** the **length** of the **primitives** to make sure they do, as below.

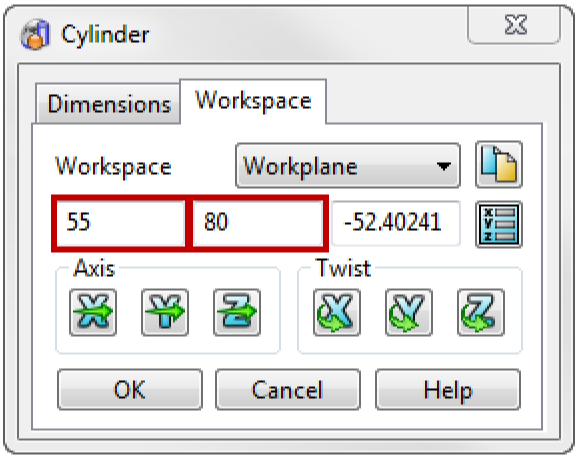


53 **Turn OFF Level 10 : Solid Model**.

The next step is to refine the **size** and **location** of the **primitive shapes** in relation to the **datum location**, and **each other**.

54 **Double-click** on the **Solid Cylinder** below the **Cone**. In the **Dimensions** tab refine the **Radius** to **12.5mm** (**Diameter 25mm**)

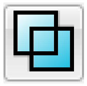
55 In the **Workspace** tab refine the **X** and **Y coordinates** to **55.00mm** and **80.00mm respectively**.

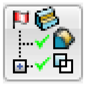


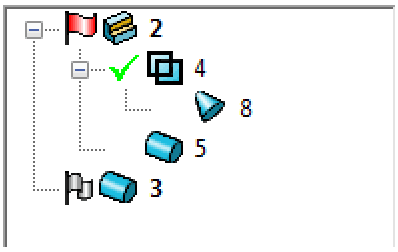
It is important that the **Solid Cone** has the same **X** and **Y location** (as the previous cylinder) to provide a **consistent countersink shape**.

56 **Double-click** on the **Solid Cone** and **navigate** to the **Workspace** tab of the **Cone Edit** dialog. **Edit** the values to **X=55.00mm** and **Y=80.00mm**.

57 **Select both** these **solids** in the **graphics area** (**Cone** and **Cylinder**).

58 From the **Solid Feature**  menu choose **Boolean Addition**  to **combine** the **two solids**.

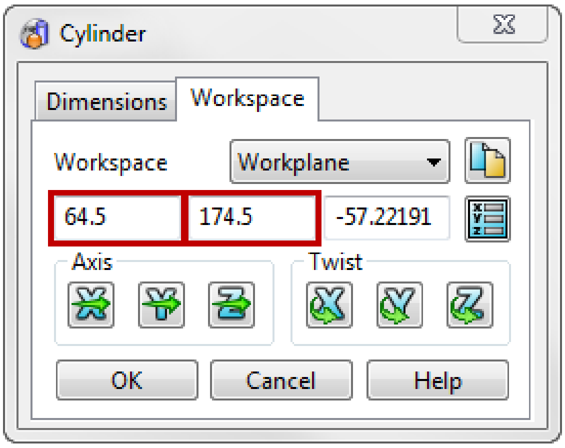
59 **Open** the **Solid History** tree. 



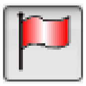
60 **Double-click** on the other **Solid Cylinder** in the **graphics area**.

61 In the **Dimensions** tab refine the **Radius** to **10.00mm**.

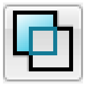
62 Navigate to the **Workspace** tab and using the **values below**, refine the **X** and **Y coordinates** of the **datum** of the **cylinder**.

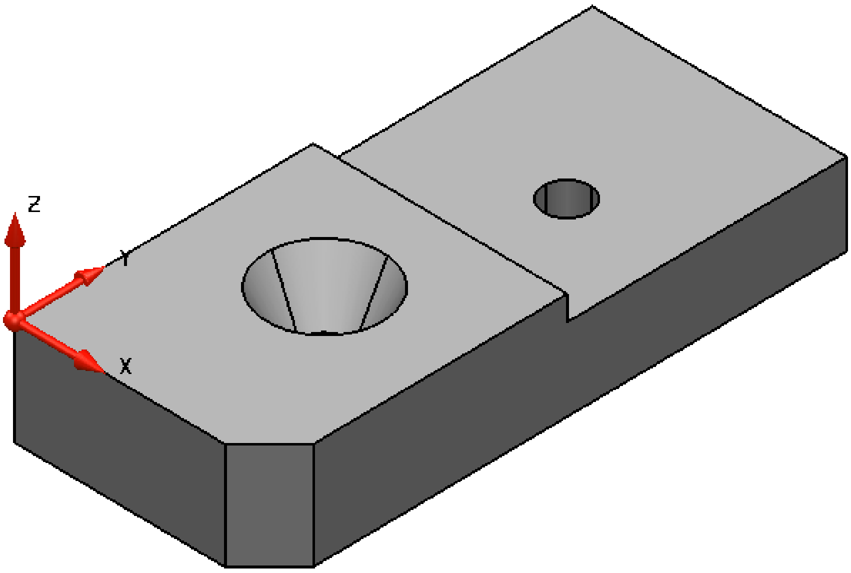


63 **Turn ON Level 10 : Solid Model**.

64 **Ensure** the **main solid block** is **Active**. 

65 **Quick Select ALL Solids**  from the selection flyout.

66 Perform a **Boolean Remove**  operation from the **Solid Feature**  menu on the **multiple solids** selected using the **main solid** as the **active.**

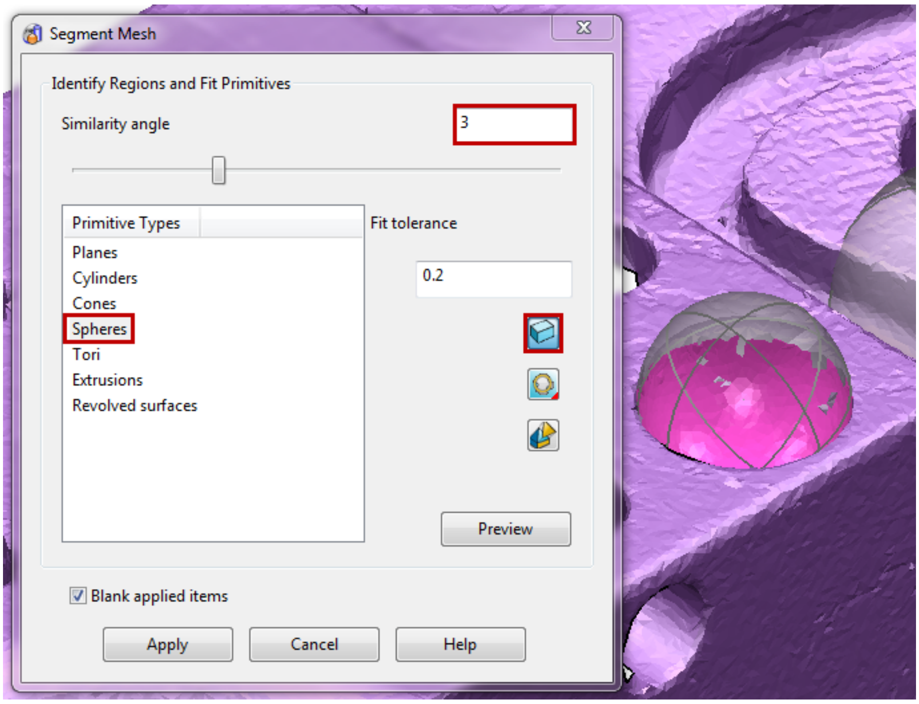


67 **Turn ON Level 7 – Mesh** and **Turn OFF Level 10 – Solid Model**.

68 Select the **mesh** in the **graphics area** to **activate** the **Mesh Edit** toolbar and **open** the **Automatic Mesh Segment**  tool.

69 In the dialog set the **Similarity Angle** to **3** and **Primitive Types** to **<u>Solid</u> Spheres.** Select **Preview**.

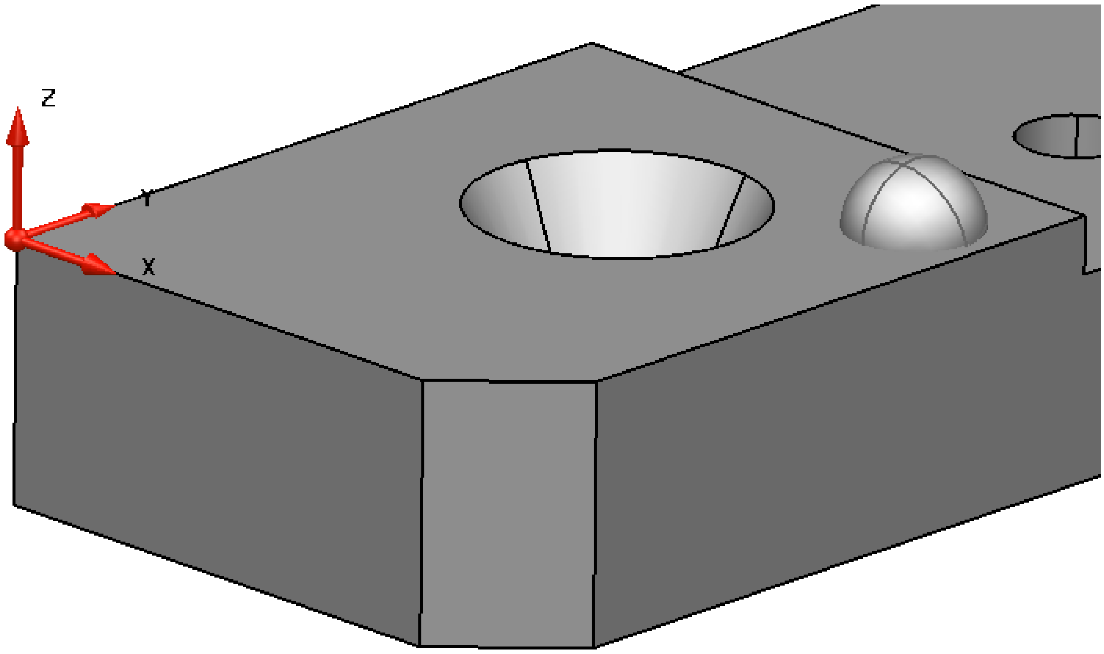
 *As before we have changed the **similarity angle** using the **slider** until we **fitted** this **sphere**, in this instance **3** gave a good result.*



70 The **hemispherical cut** from the **top face** of the **mesh** has been recognised as a **solid sphere**. **Select** and **Apply** this **primitive shape**.

71 Click **Cancel** to **close** the **form**.

72 **Turn OFF** the **Level 7 : Mesh** and **Turn ON** the **Level 10 : Solid Model**.

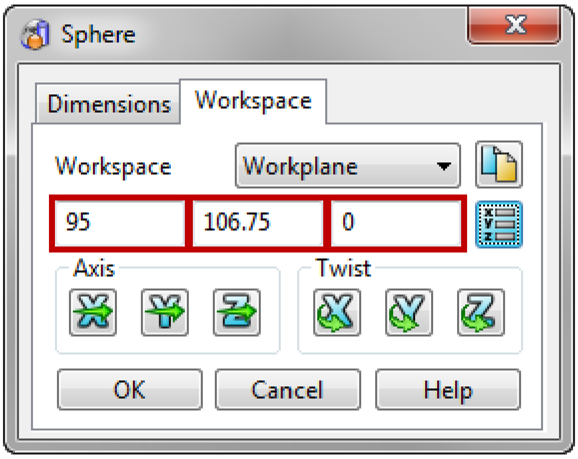


The image above highlights how the **solid sphere** we have created **interacts** with the **solid block**. We can now again **refine** the **size** and **location** of the **solid sphere** to **Reverse Engineer** the **part**.

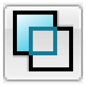
73 **Double-click** on the **solid sphere** to raise the **sphere edit** dialog.

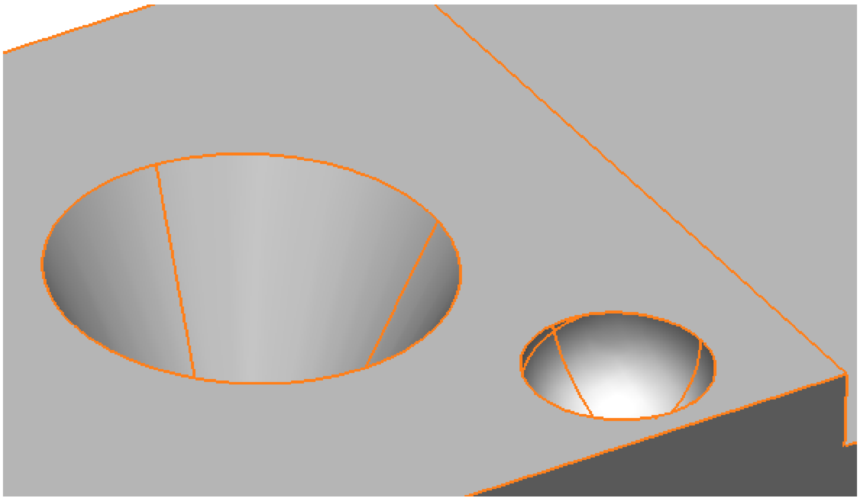
74 In the **Dimensions** tab **edit** the **radius** to **12.00mm**.

75 Select the **Workspace** tab and fill in the **values** from the **image** below.



*This time it is important to set the **sphere centre Z location** to **0**. This will ensure the **centre** of the **sphere** is **flush** with the **top face** of the **part**.*

76 Select the **solid sphere**, and then select **Boolean Subtract**  from the **Features**  menu.



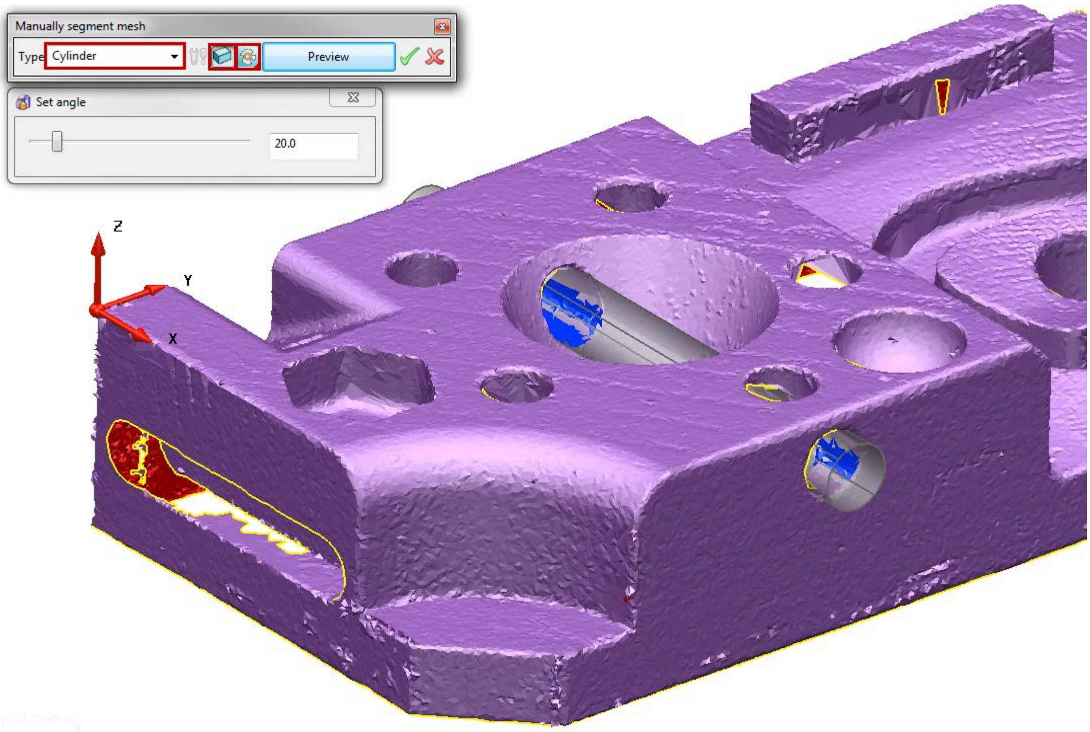
There **hemispherical cut** has been created by using a **Boolean Subtraction** between the **active solid (primary selection)** and the **solid sphere (secondary selection)**.

77 **Turn ON Level 7 – Mesh** and **Turn OFF Level 10 – Solid Model**.

78 Select the **mesh** in the **graphics area** to **activate** the **Mesh Edit** toolbar and **open** the **Manually Mesh Segment**  tool.

79 In the dialog choose to fit **Solid Cylinders** using the **Fit Inside** method.

80 Within the **graphics area** select the **hole** running **horizontally** through the **width** of the **part**. Use the **Ctrl** key to **multi-select triangles** to get a **spread of data** along the **length** of the **hole**. Select **Preview**.

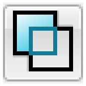


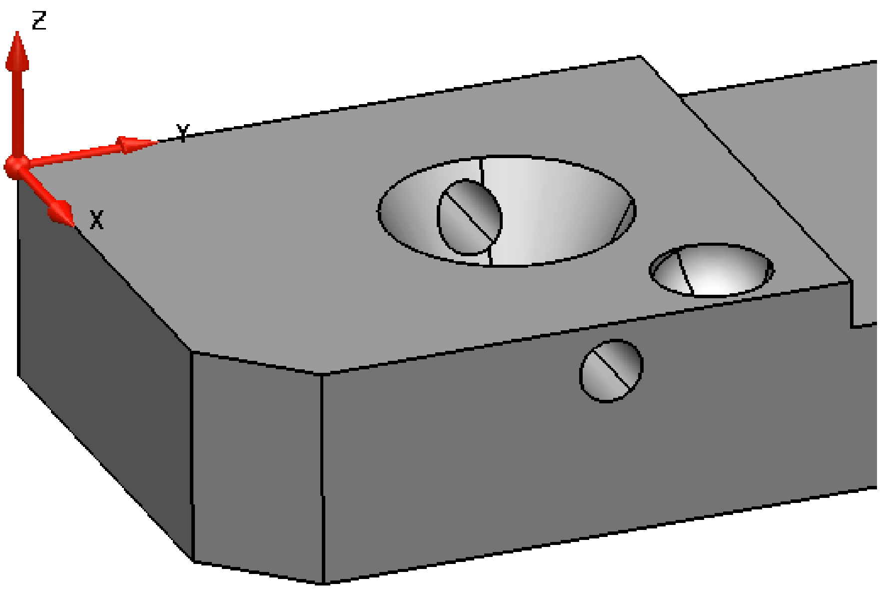
81 Select the **green tick** in the dialog to create the **solid cylinder**, and then the **red cross** will **dismiss** the **form**.

82 **Double-click** on the **solid cylinder** and then **refine** its **radius** to **6.5mm**.

*The **alignment angle** set inside our **options file** has, as with the other **primitives**, **aligned** the **axis** of this **cylinder** to an **axis** of the **active workplane**.*

83 **Turn OFF Level 7 : Mesh** and then **Turn ON Level 10 : Solid Model**.

84 From the **Solid Features**  menu select **Boolean Subtraction** . The **main solid block (active)** will be the **primary selection** and select the **cylinder** as the **secondary selection**.



85 **Turn ON Level 7 : Mesh** and then **Turn OFF Level 10 : Solid Model**.

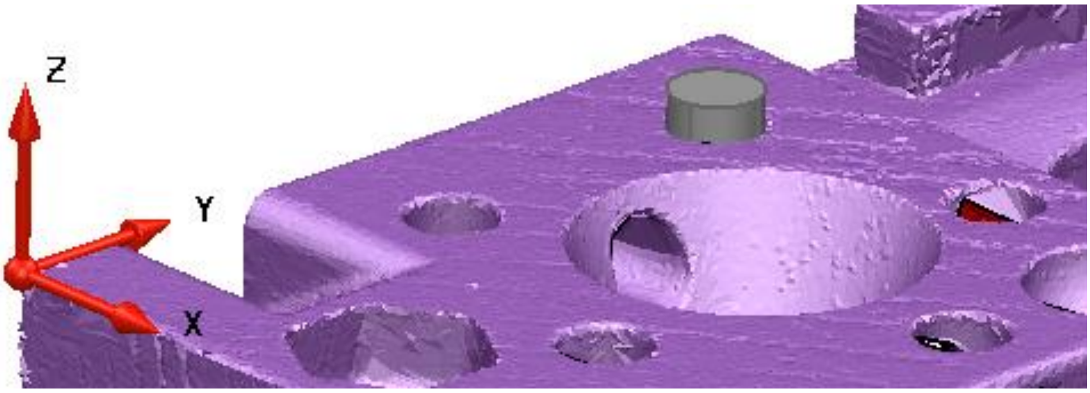
86 Select an **Isometric View** (**Ctrl+3**).

87 Select the **mesh** in the **graphics area** to **activate** the **Mesh Edit** toolbar and **open** the **Manually Mesh Segment**  tool.

88 In the dialog choose to fit **Solid Cylinders** using the **Fit Inside** method.

We are now going to **fit primitives** in order to create the **PCD** of **holes** on the **part**. To do this we need to **fit** a **number** of **primitive cylinders** to **calculate** the **diameter** of the **pattern** using a **3 point arc**, which will also give us the **centre point** of the **pattern**.

89 Using the **Manual Mesh Segment** tool **fit** a **solid cylinder** using **one** of the **holes**.



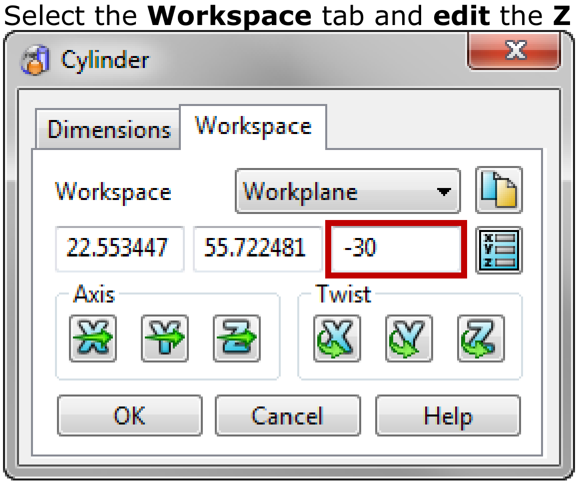
90 **Turn OFF Level 7 : Mesh** and then **Turn ON Level 10 : Solid Model**.

91 As you can see the **scan data** has not reached the **bottom** of the **holes**. However, we know from the **physical model** and its **design intent** that these **holes** have a **Depth** of **30mm**. We can **refine** the **location** of the **bottom** of the **cylinder** to reflect this.

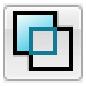
92 **Double-click** on the **cylinder** in the **graphics area**. In the **Dimensions** tab edit the **Radius** to **6.5mm** and the **Length** to **35mm**.

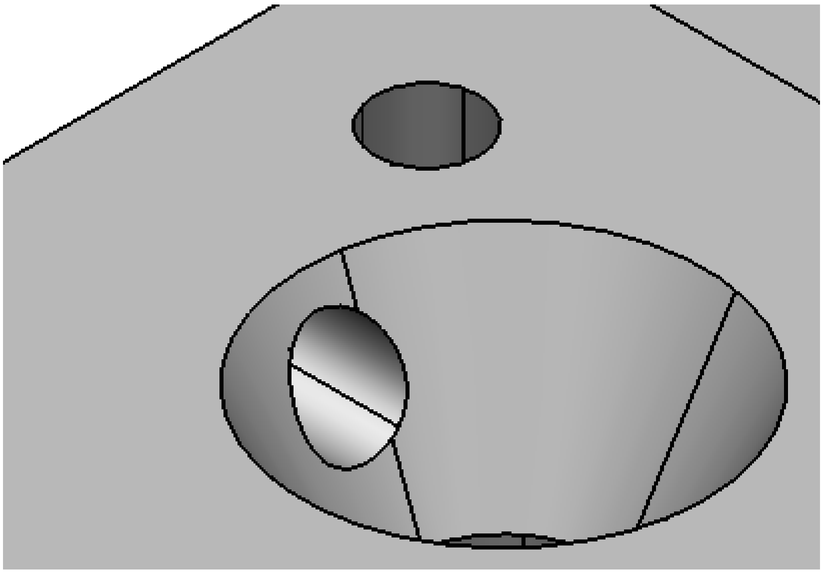
*The **length** of **35mm** will ensure that when we include a **Depth** of **30mm** the **solid cylinders** will **break through** the **top face** of the **solid block**.*

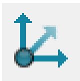
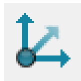
93 Select the **Workspace** tab and **edit** the **Z location** to **-30mm**.



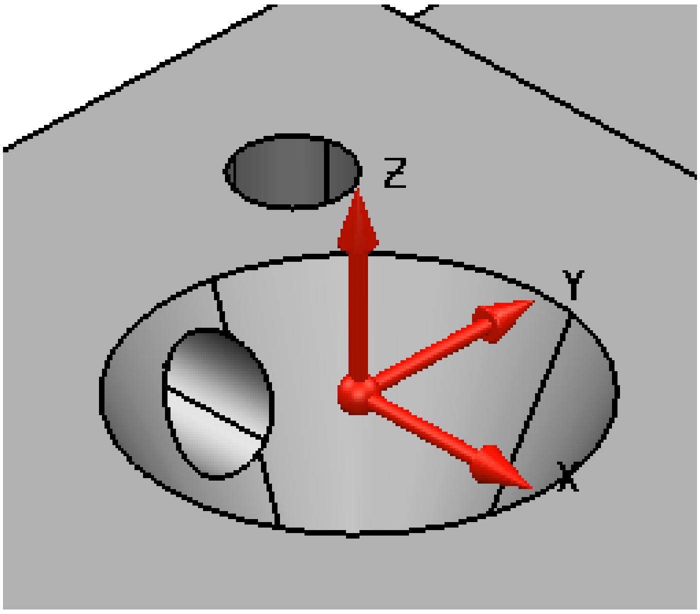
The centre of the **countersunk hole** is the **centre point** of the **PCD**. Therefore, we will create a new **workplane** at this **location** and then **pattern** the **hole** around the **workplane** using the **Pattern Feature**.

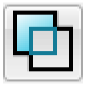
94 Select the **solid cylinder** and select to perform a **Boolean Subtraction**  from the **Solid Feature** menu. 



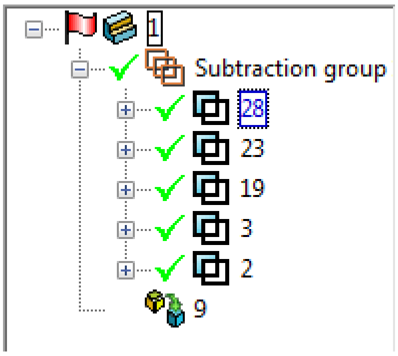
95 From the **Workplane**  menu select **Create a Single Workplane**. 

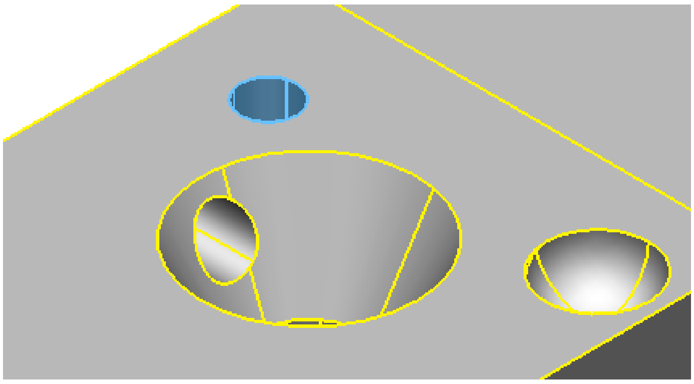
96 Create a **workplane** at the **Arc-Centre** of the **countersunk hole** by **automatically snapping** to it using the **intelligent cursor**.

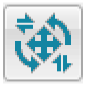


97 Select the **Boolean Subtraction**  operation of the **13mm Diameter hole** we have just created in the **History Tree**.

*This will also **highlight** the **hole** in **blue** in the **graphics area** as shown below.*



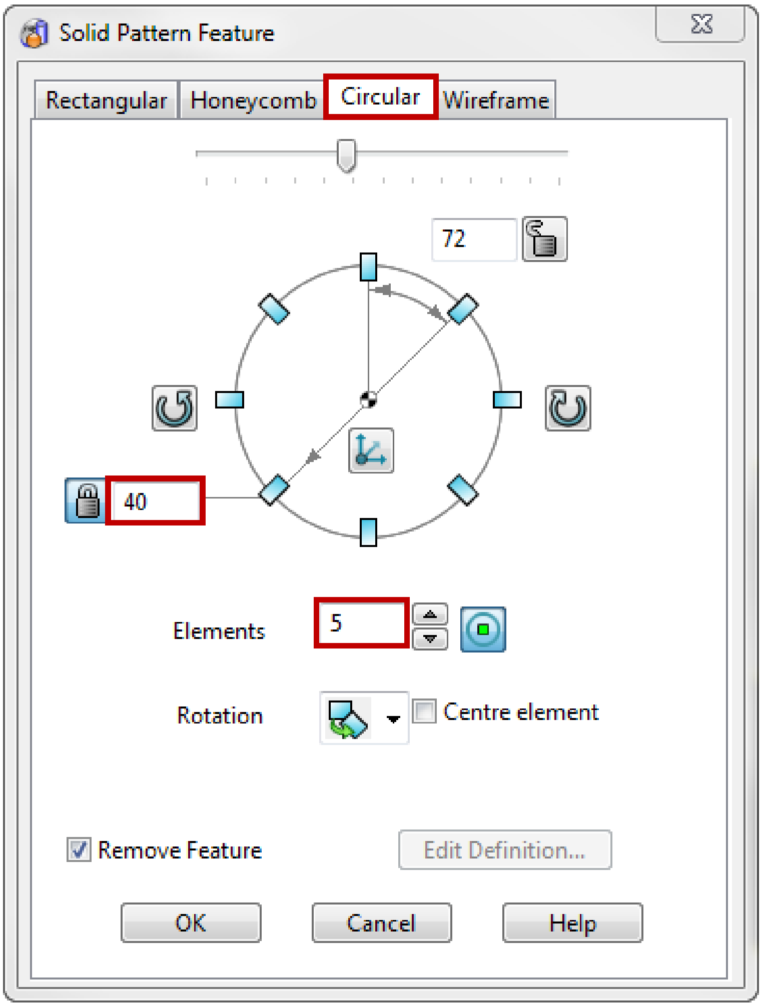


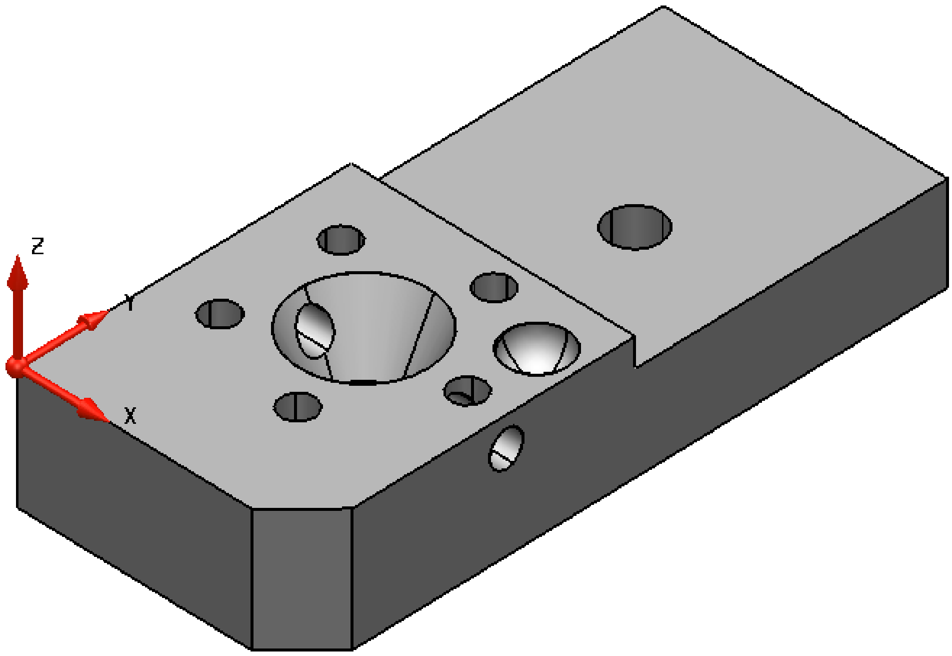
98 From the **General Edits**  menu select **Create a Pattern**. 

99 Within the **Pattern** dialog navigate to the **Circular Pattern** tab.

100 Edit the **Number of Elements** to **5** and the **Radius** to **40mm**.

*The **angle** will **automatically update** in this dialog to create the **pattern** with **equidistant spacing**.*





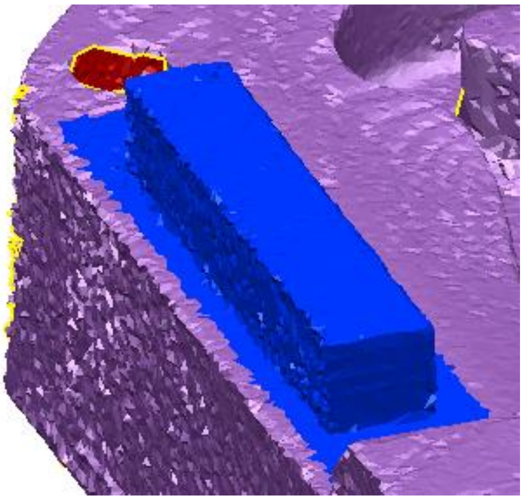
We have successfully create the **PCD** of **holes** of **Depth 30mm** and **Diameter 13mm**. Next, we will **divide the mesh** to help us **remodel** the **rectangular protrusion** on the **right hand side** of the part, shown below.

101 **Turn OFF Level 10 – Solid Model** and **Turn ON Level 7 – Mesh**.

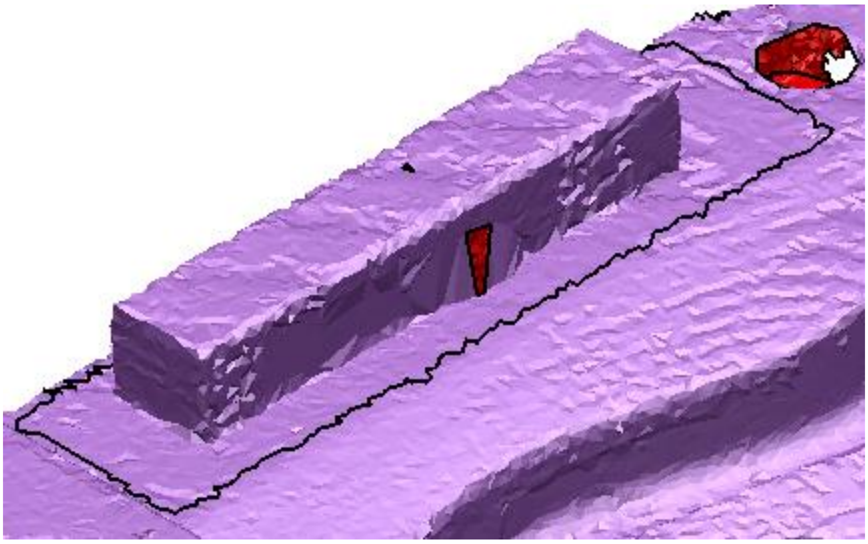
102 Take a **View from Top** (**Ctrl+5**).

103 Select the **Mesh** to **activate** the **Mesh Edit** toolbar.

104 Using the option to **Pick Triangles in a Mesh by Box** drag a **box** around the **protrusion** while also **selecting extra triangles** around the **area as shown** below to allow a **complete solid** to be **created**.



105 With the **triangles selected** choose to **Divide the Mesh by Selection**  from the **Mesh Edit** toolbar.



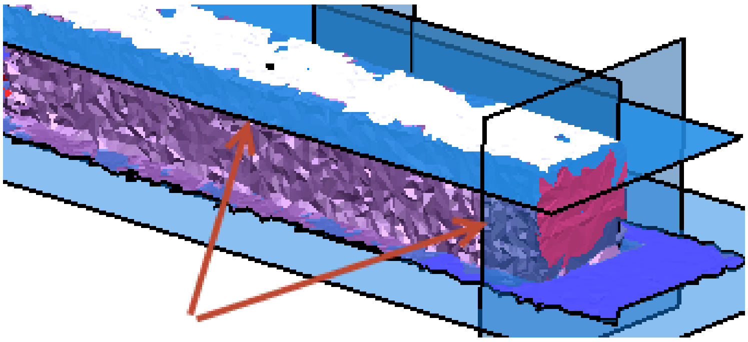
The **major benefit** for **dividing** this **mesh** is that we can **blank** the rest of the **mesh** and **concentrate** on this **particular area of interest** using the **PowerSHAPE Pro Mesh Segment tools** to **fit primitive surfaces** and then **combine** the **two meshes** back together later.

106 **Select** and **Blank** (**Ctrl+J**) the rest of the **mesh** leaving only the **protrusion area**.

107 Select **Automatically Segment Mesh**  from the **Mesh Edit** toolbar.

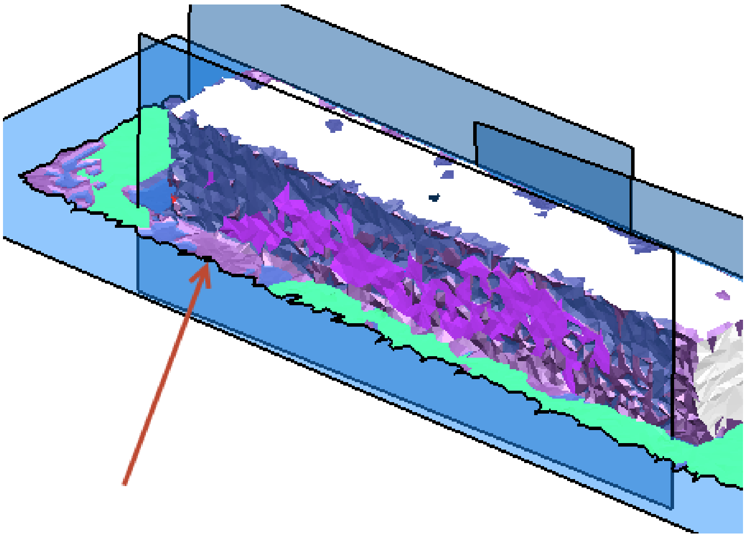
108 In the dialog select **Planes** as the only **Primitive Type**, choose a **Similarity Angle** of **2**, and then click **Preview**.

109 Select and **Apply** the **two surfaces** identified in the **next image**.



This will give us the **Top face** and one of the **end faces.** We also already have the **bottom face** which will join the **main solid model** automatically saved on **Level 800** from the **Solid from Untrimmed Surfaces operation.** We can create the other end face by **offsetting** the above **surfaces** the **length** of the **protrustion**.

110 Change the **Similarity Angle** to **3** and click **Preview** in order to **recognise** and then **Apply** the **side face shown below**.



Although it is clear that the **surface plane** recognised here doesn't cover the **full length** of the face we can simply **extend** it as a **primitive shape** later.

111 Click **Cancel** to **close** the dialog.

112 **Quick Select ALL Surfaces**  that we have created (**3**).

113 Place these **surfaces** on **Level 1 – Surfaces** and **turn OFF** the **level**.

114 **Turn ON Level 800** and **select** the **large face** which will form the **bottom** of the **protrusion** to join with the **main solid model** and place it on **Level 1 – Surfaces**.

115 **Turn OFF Level 800**.

116 From the **status bar** at the **bottom** of the **window**, **open** the **Measure**  form.

*With this form open, **measurements** can quickly and easily be taken **directly from the model**, in this case a **mesh**.*

117 **Measure** the **Length** (**60mm**) and **Width** (**11mm**) of the **protrusion**.

*These are the **distances** we will need to **offset** the **current surfaces** to create the **missing faces** to **create** the **solid**. Again **design intent** or the **physical model** will give us a better idea of these **measurements if critical**.*

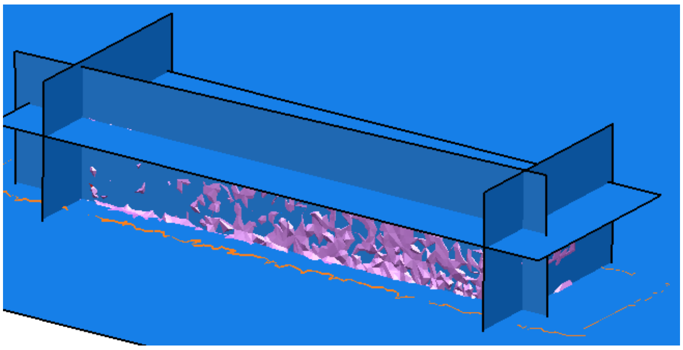
118 **Turn ON Level 1 – Surfaces**.

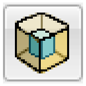
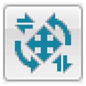
119 Select the **End surface** and from the **General Edits**  menu select **Offset**. 

*If required click the **orange arrow** to **reverse** the **direction** of the **offset**.*

120 In the **Offset** dialog type a **distance** of **60mm** and press **Enter**, and then **close** the **form**.

121 Where the **side surface** is **too short**, select and **dynamically extend** the **surface** to ensure an **overlap** between this and the **adjoining surfaces**.

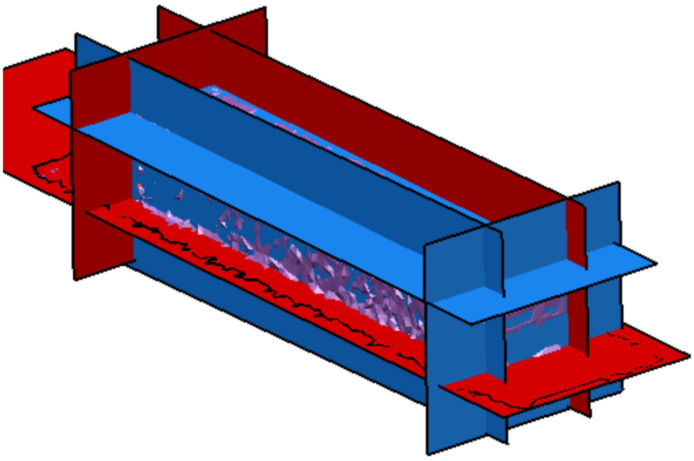


122 Select this **surface** and choose to **Offset**  from the **General Edits**  menu.

123 As before, if it is required click the **orange arrow** to **offset** through to the other side of the **mesh**.

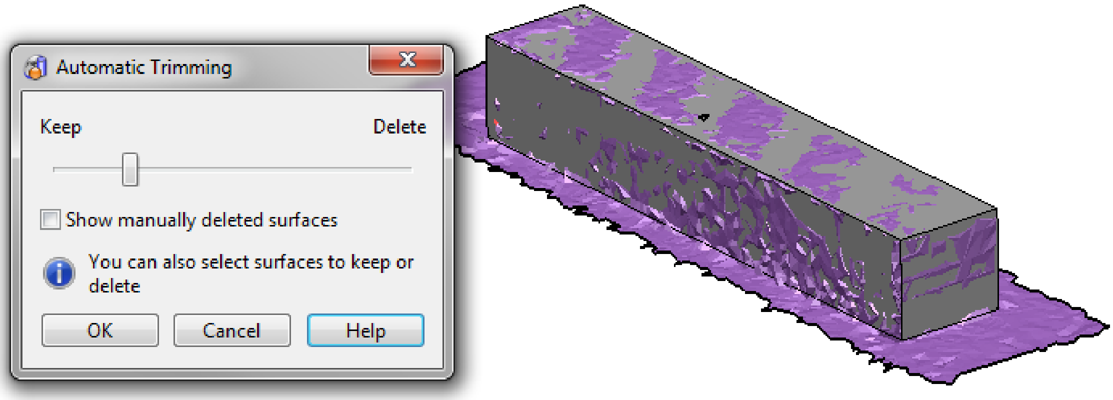
124 In the **Offset** dialog type a **distance** of **11mm** and press **Enter**, and then **close** the **form**.

125 Next, **reorientate** the **positive 'blue' side** of the **surfaces** to **face outwards** before we attempt to create the solid.



126 **Quick Select ALL Surfaces**. 

127 From the **Solid**  menu select **Create Solid from Untrimmed Surfaces**. 



128 Once you are happy with the **solid** created click **OK** in the **Automatic Trimming** form.

*If the selection of surfaces **<u>does not</u>** create a **closed solid model** select **Cancel** in the dialog and **go back and check** the **orientation** and **overlap** of the **surfaces**.*

129 Select the **solid** and place it on **Level 10 – Solid Model** using the **middle mouse button**.

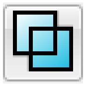
130 **Unblank** (**Ctrl+L**) the rest of the **mesh**.

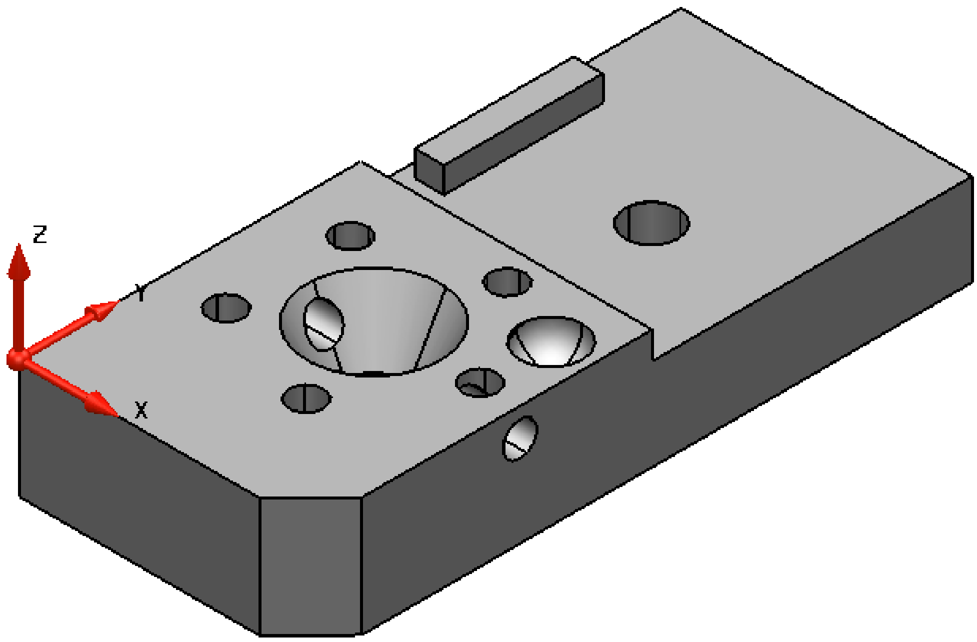
131 **Select both mesh** inside the **graphics area**.

132 Select **Stitch Mesh**  from the **Mesh Edit** toolbar.

133 Using the **default values** click **OK** to **stitch** the two **mesh** back **together**.

134 **Turn OFF Level 7 – Mesh** and **Turn ON Level 10 – Solid Model**.

135 Using **Boolean Addition**  **add** the **two solid models together**.



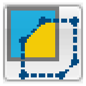
Hexagonal Cut

Next we will aim to create the **hexagonal feature cut** into the **top face** of the **model**. To do this we will again **divide the mesh** down to just the **area of interest** for **simplicity**.

136 Take a **View from Top** (**Ctrl+5**).

137 **Zoom in** to the **hexagonal feature**.

138 Select the **mesh** to **activate** the **Mesh Edit** toolbar.

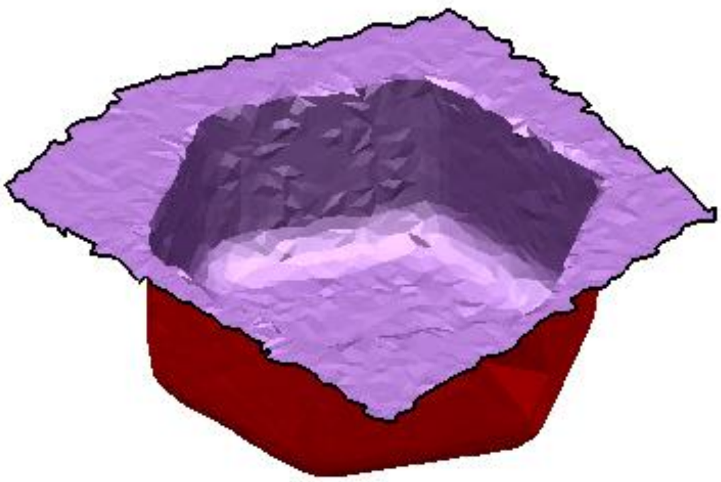
139 Choose the option to **Pick Triangles on a Mesh by Discrete Lasso**. 

140 Using the **lasso** select the **area around** and **including** the **hexagonal cut** similar to the **below image**.

141 From the **Mesh Edit** toolbar and with the **triangles selected** choose **Divide the Mesh by Selection**. 

142 Select and **Blank** (**Ctrl+J**) than **main mesh**, as before.

143 Select an **Isometric View** (**Ctrl+1**).



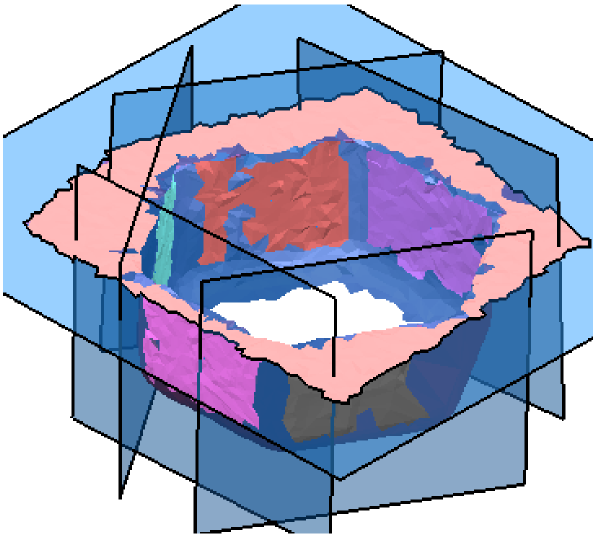
144 From the **Mesh Edit** toolbar choose to **Automatically Segment Mesh**. 

145 Inside the dialog change the **Similarity Angle** to **2** and search for only **Planes** as the **Primitive Type** then select **Preview**.

*Again this value of **2** for **similarity angle** works for this part, and was found using the **slider** to **give good results**.*

*This time we will be looking to create a **solid** to then **Boolean Subtract** from the **main block**.*

146 Select and **Apply** the **surface** representing the **bottom face** of the **cut**.



The **PowerSHAPE Pro Automatic Mesh Segment** tool has found **5** out of the **6 vertical** faces of the **hexagonal pattern**. If you rotate the view to see the face that is not found, you will see that a hole has been automatically filled which is likely the reason.

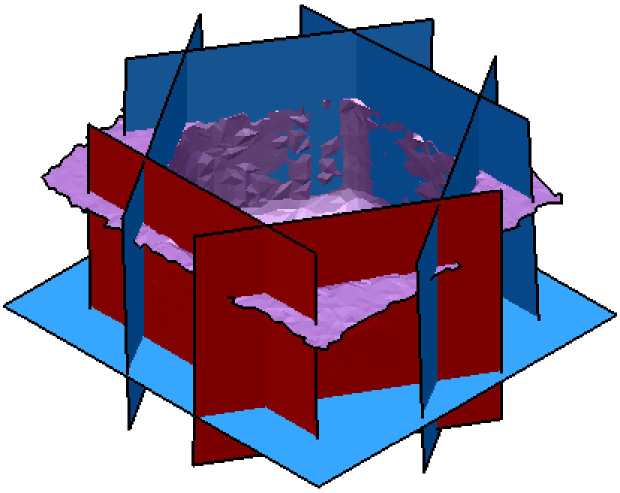
147 Select and **Apply** the **5 vertical faces** found of the **hexagonal pattern**.

148 Click **Cancel** to **close** the dialog.

149 Using the **Measuring**  tool found on the **status bar** below the **graphics area**.

***Face to face** we are looking to create a **26mm hexagonal solid** from results of using the **measuring tool**.*

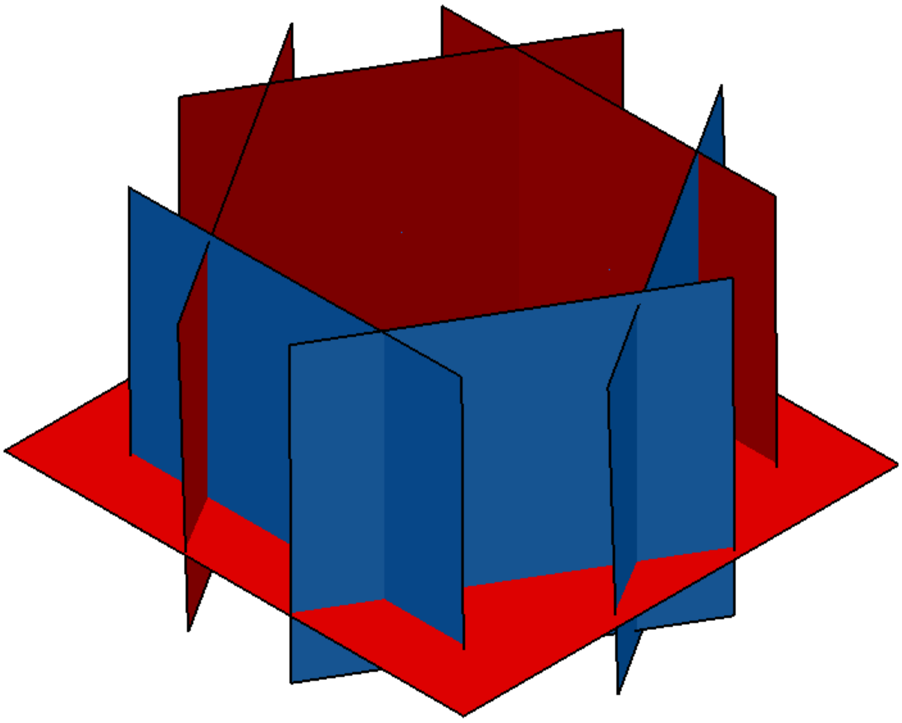
150 Using the **Offset**  tool (**26mm**) from the **General Edits**  menu and the **surface opposite** the **missing face** complete the **hexagonal pattern** of **surfaces**.

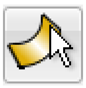


From here we can use an **open solid** by using a **solid from untrimmed surfaces** operation to enable us to **remove** the **hexagonal feature**.

151 **Unblank** the **main mesh** and then **Turn OFF Level 7 – Mesh**.

152 **Reorientate** the **untrimmed surfaces** so the **positive/blue side faces outwards** in preparation for the **Solid from Untrimmed Surfaces** function.

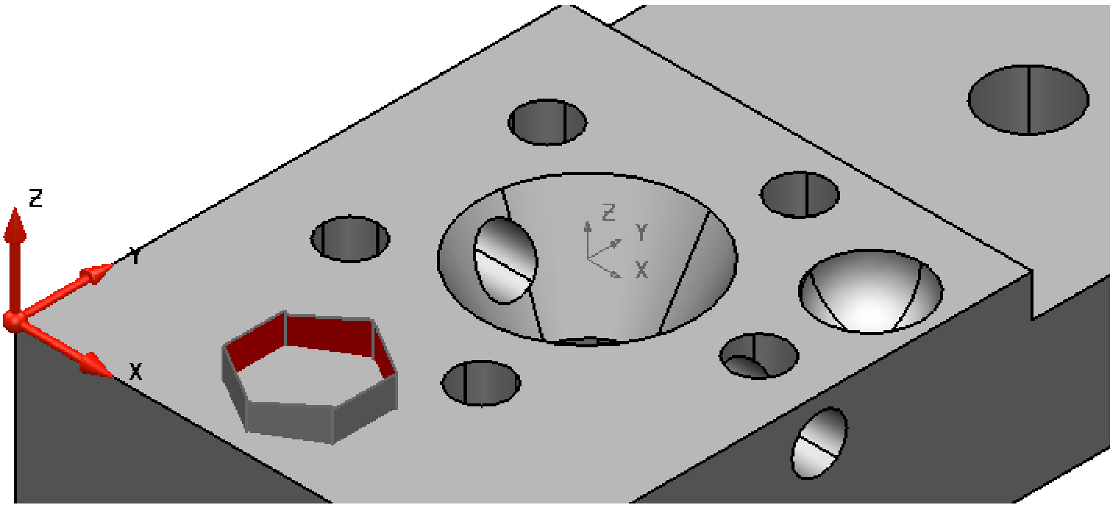


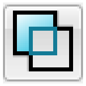
153 **Quick Select ALL Surfaces**  from the **Selection Flyout**.

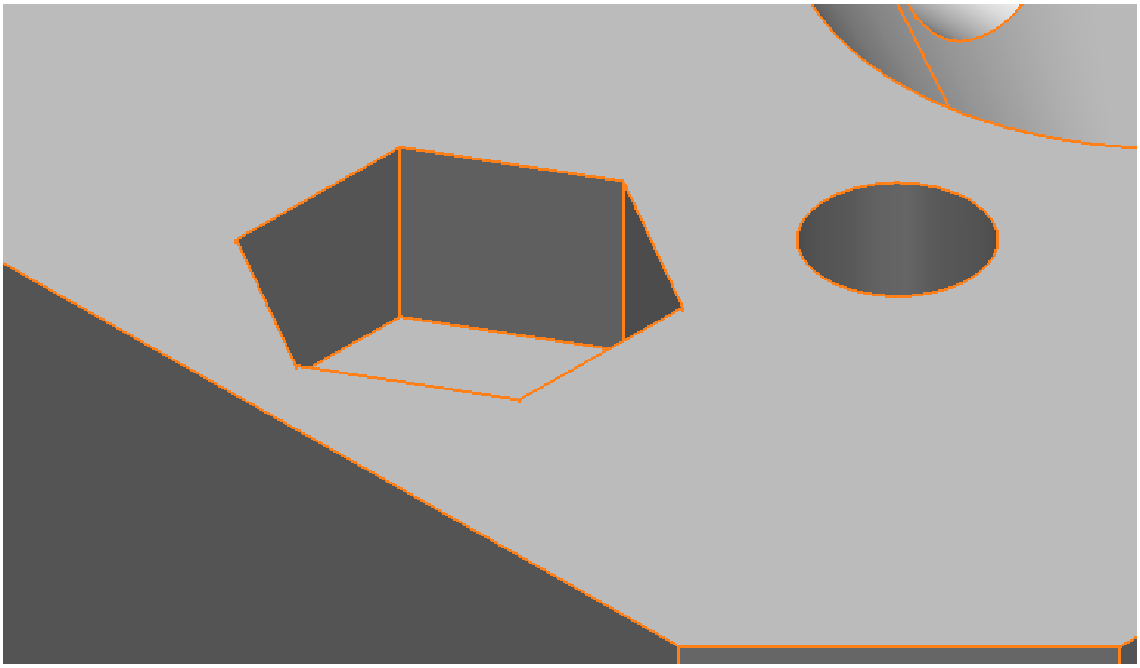
154 From the **Solids** menu select **Create Solid from Untrimmed Surfaces**. 

155 Using the **auto trimming dialog** create the **open hexagonal solid** shown in the next image.

156 **Turn ON Level 10 – Solid** and open the **Solid Feature Tree** from the **Solid Edit** menu.

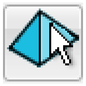


157 Using the **Boolean Subtract**  function, **remove** the **Hex** from the **Main Solid**.



158 **Turn OFF Level 10 – Solid** and **Turn ON Level 7 – Mesh**.

Next we need to **stitch** the **split segments** of **mesh** back together, as they are originally from the same connected mesh this should be a simple operation.

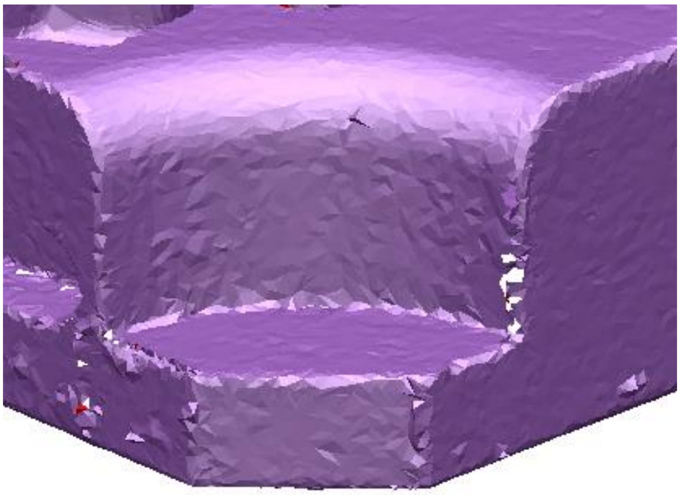
159 Navigate to, and select **Quick Select ALL Mesh**. 

160 From the **Mesh Edit** toolbar select **Stitch Mesh**. 

161 Within the **dialog** select **OK** using the **default values**.

Cylindrical Cut Corner

Next we will create the **cylindrical cut** from the **lower left hand** corner of the **top face** of the part. Note also that we have **already created** the **chamfered edge** below when we created the **main block**.

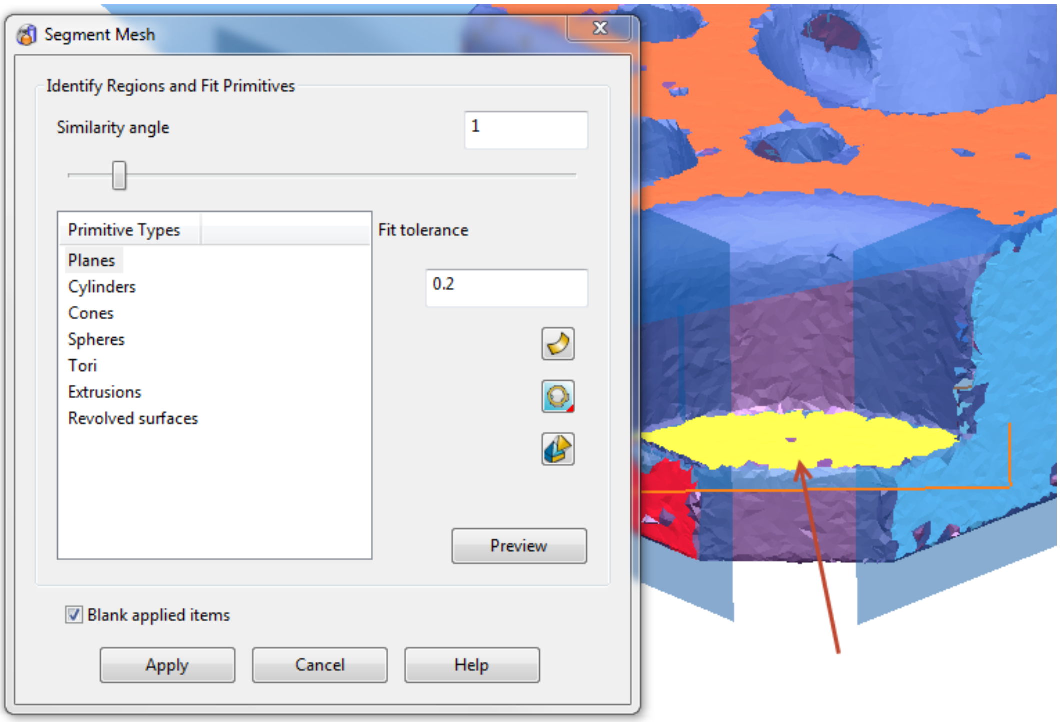


162 Activate the **Mesh Edit** toolbar by selecting the **mesh** in the **graphics area**.

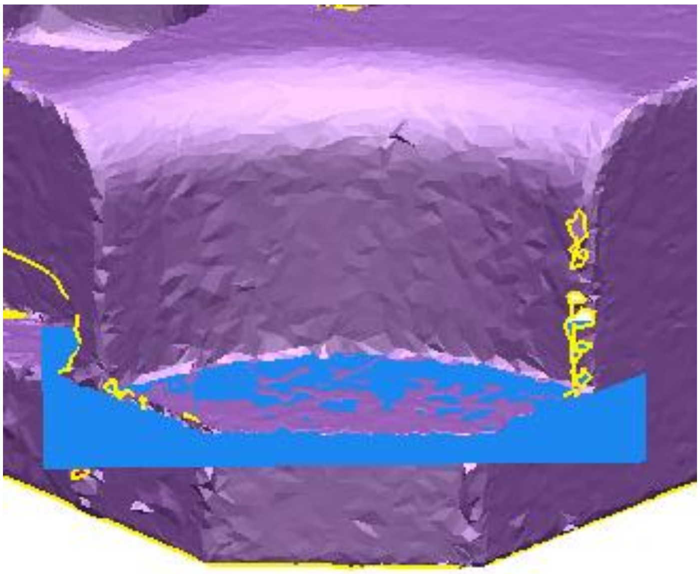
163 From the toolbar select to **Automatically Segment Mesh and Fit Primitives**.

164 In the **Segment Mesh** dialog select only to fit **primitive planes** as this will help clear the clutter from the screen slightly.

165 **Select** and **Apply** the **primitive plane** at the **bottom of the cut**.



166 **Cancel** the **Segment Mesh** dialog.



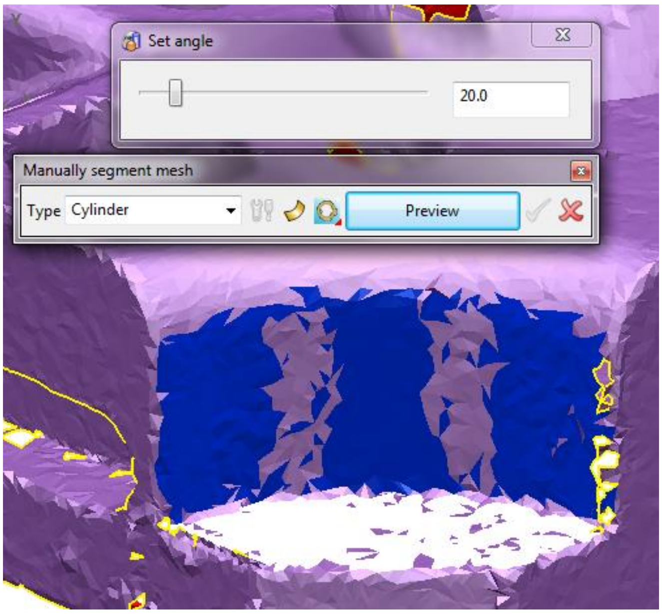
Next we will use the **Manually Segment Mesh and Fit Primitives**  option to fit a **primitive cylinder** above the plane we just created to cut away from the **main solid**.

167 From the **Mesh Edit** toolbar select **Manually Segment Mesh and Fit Primitives**. 

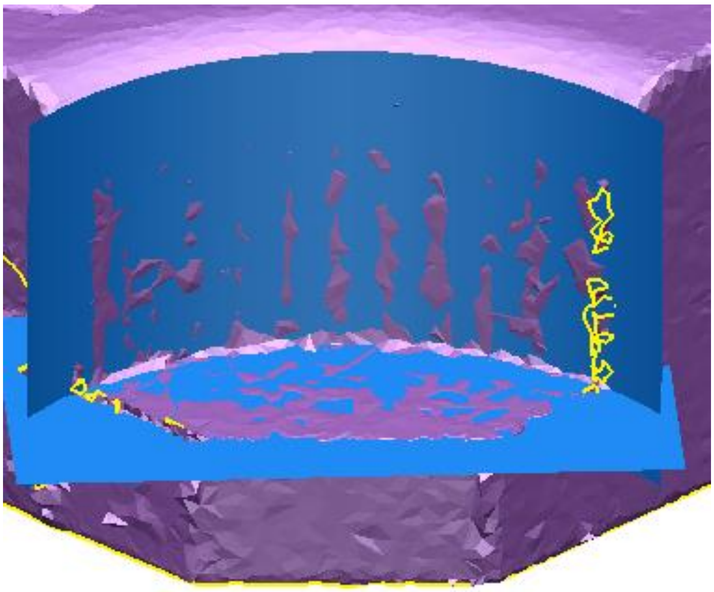
168 In the dialog, use the **drop-down** menu to select **Cylinder** as '**Type**'.

169 Select the **area on the mesh** to fit the **cylinder**.

*Holding the **Ctrl** key while clicking in the **graphics area** will allow you to make **multiple selections** to select the **full area** to fit.*



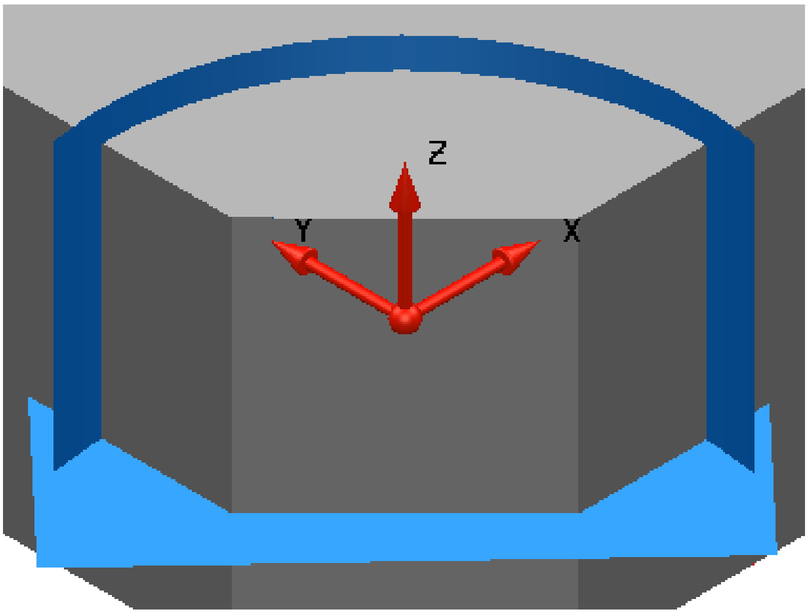
170 **Preview** the **Cylinder**. Once you are happy with the result select the **green tick** to create the **cylinder**, and then **dismiss** (red cross) the form.



We have now successfully **fitted** and **created** a **partial cylinder**. Next, we need to check the **location** of the **centre point**. On the model this should be where the two vertical edge faces would meet, therefore we will **create a workplane** at this location and then ensure that the **X** and **Y coordinates** of the **primitive cylinder** are **0**.

171 Turn **OFF Level 7 : Mesh** and **Turn ON Level 10 : Solid**.

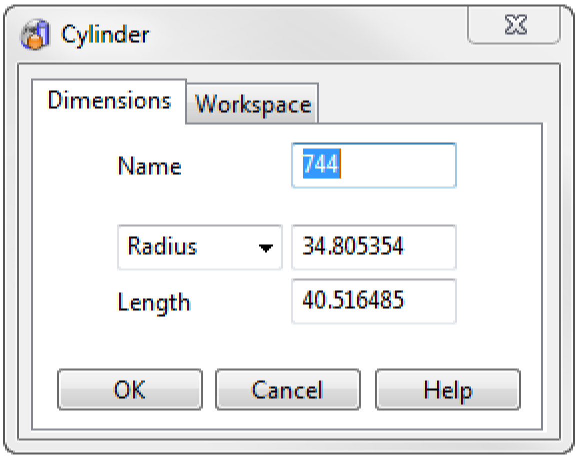
172 Using a **Workplane Aligned to Geometry**  and using the **Workplane manipulation handles** to move the workplane into the location below.



*If you are having trouble positioning the workplane, please refer back to the **workplanes chapter** or ask the **tutor**.*

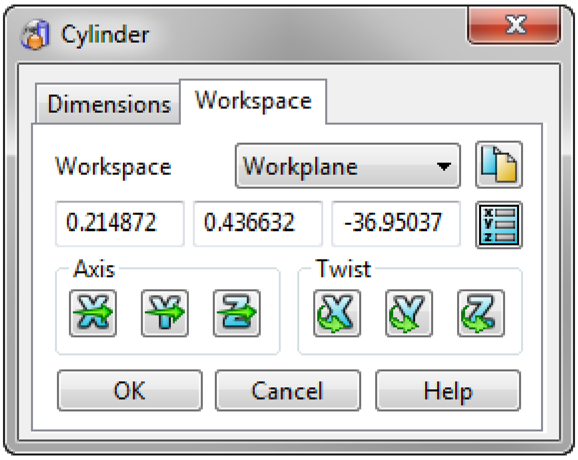
*The **Z Axis** of this **workplane** should line up exactly with the **axis** of the **cylinder**. Therefore this workplane will allow us to **refine** the **location** and **size** of the **primitive**.*

173 **Open** the **cylinder properties** dialog by double-clicking it in the **graphics area**.



174 Edit the **Radius** to **35mm**. The **length** currently is **not critical**, as long as it **extrudes above** the **top face** of the **solid**.

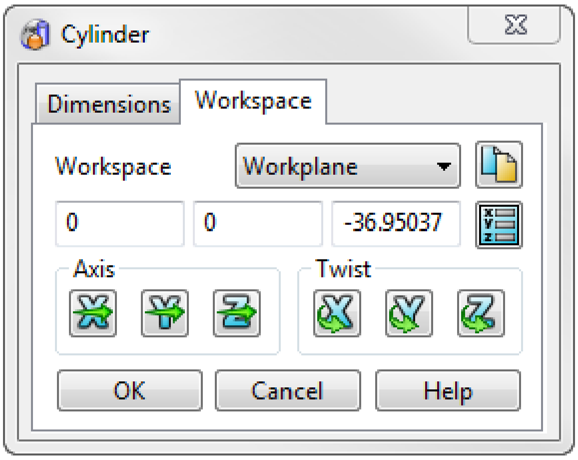
175 Select the **Workspace** tab to view the **coordinates** of the **cylinder datum**.



As described earlier, we are expecting the **X** and **Y values** to be **0** to indicate the **cylinder** is **centred** in the **correct location**. We can now simply move it in to location by **editing the values**.

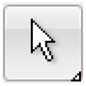
176 Ensure the **workspace drop-down** menu is set to **Workplane** and then change the **X** and **Y Coordinates** to **Zero**.

*As we are **aligning** the **Z Axis** with the **axis** of the **cylinder**, the **Z location is irrelevant** this situation.*



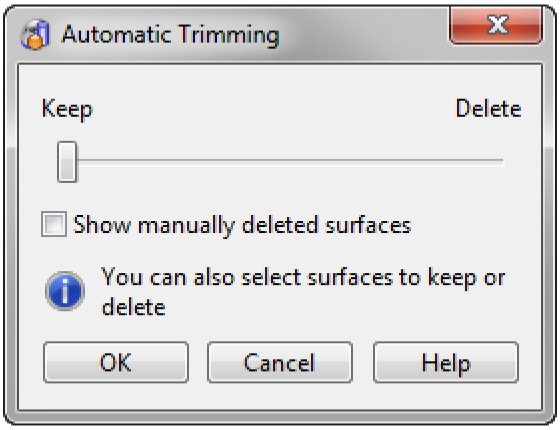
We currently have **two separate surfaces** that form the cut. To enable us to **remove** this area in one operation we can utilise the **Create a Solid by Trimming Surfaces**  to form an **open solid**.

177 **Select** and **Blank** (**Ctrl+J**) the **Main Solid**.

178 From the **Selection Flyout**  choose **Quick Select ALL Surfaces**. 

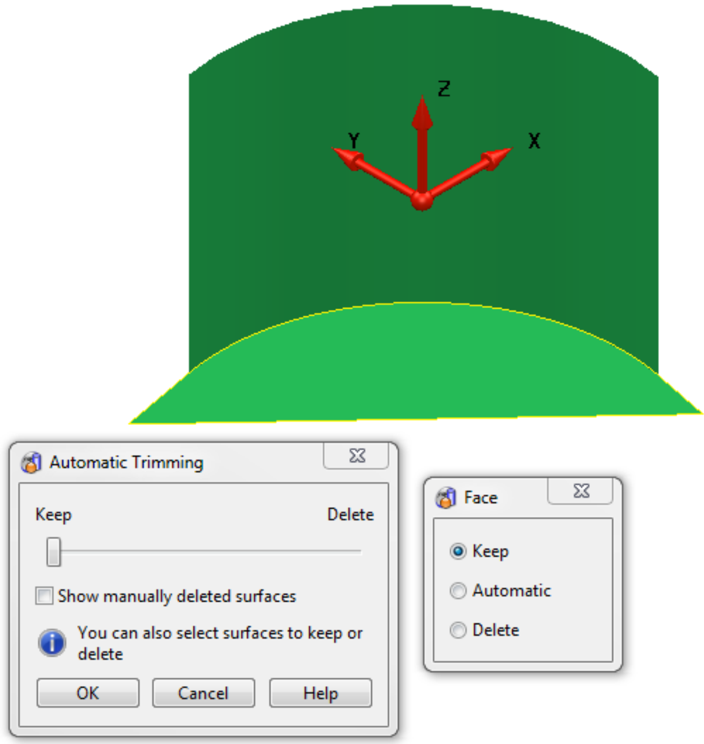
179 Go to **Create Solid by Trimming Selected Surfaces**  from the **Solid**  menu.

180 In the **Automatic Trimming** dialog move the **slider** across to **keep**, this will allow you to **control** which areas are **removed manually**.



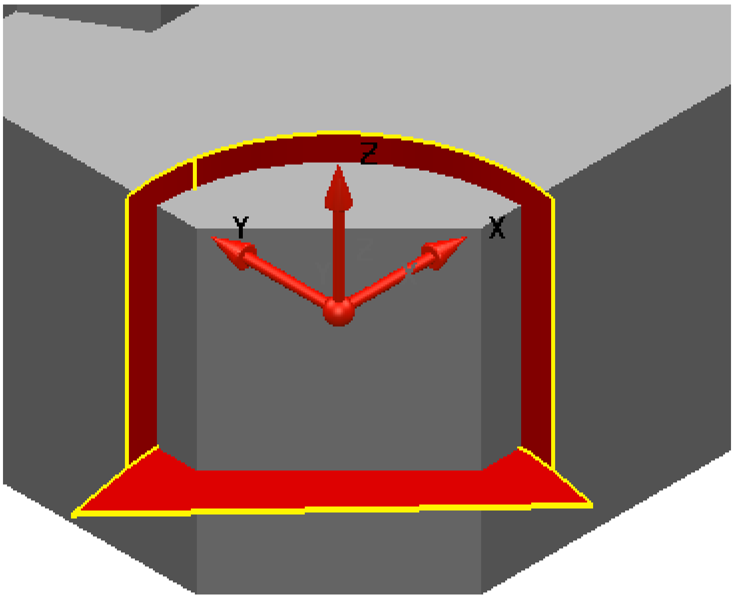
Now, we can go around the model **manually selecting areas** to **keep** or **remove** in the **popup dialog**.

181 **Select each segment** and choose **Keep/Delete** until the **solid** looks similar to below and then select **OK** to **accept** the solid and **close** the **dialog**.



182 **Unblank** (**Ctrl+K**) the **main solid**.

183 **Right-click** and **Reverse** the **new open solid**.

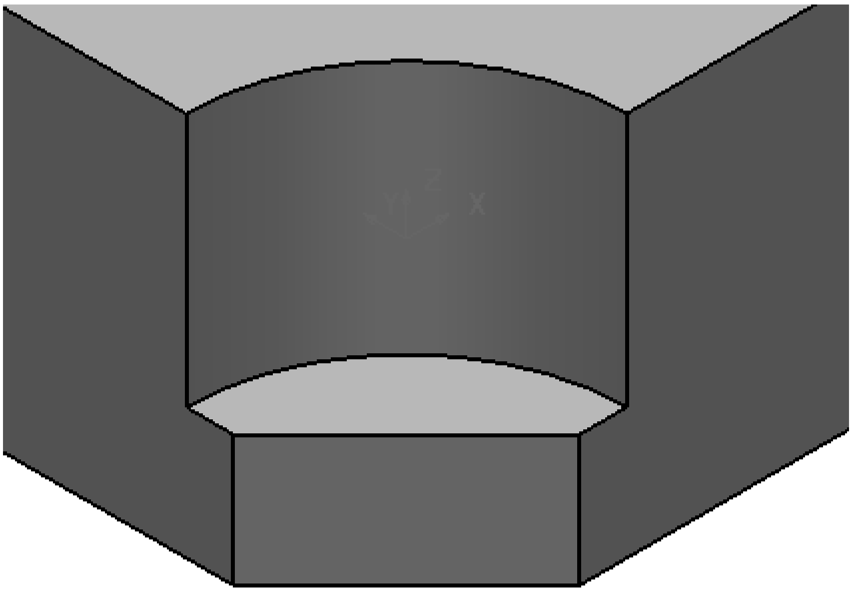


*The **solid** was **reversed** here because when we perform the **Boolean Subtraction** operation with an **open solid**, the **area** of the **primary solid** to the **red side** will be **removed**.*

184 Ensure the **Main Block** is the **active solid**.

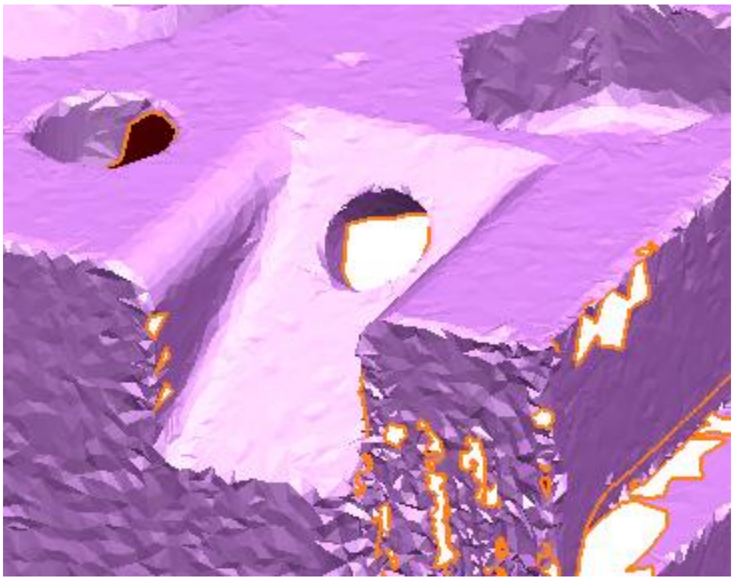
185 Select **Boolean Subtraction** from the **Solid Feature** menu.

186 The **Main Solid** will automatically become the **primary selection.** Select the **open solid** as the **secondary** by clicking on it in the **graphics area**, and then click **OK**.



*We have now created theis **cut** succesfully, all that remains to complete this section is the **fillet** around the **top edge** which we will **perform later**.*

187 **Save** the **Model**.

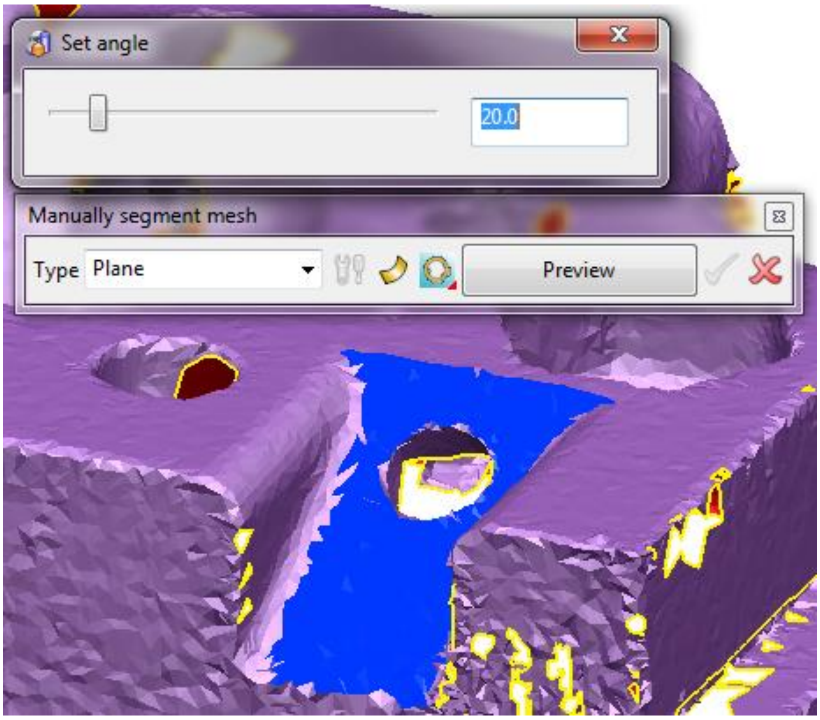


A **similar method** as the **previous cut** will be used to create this feature**.** A **number** of **plane primitive surfaces** will be fitted and **joined** to form an **open solid,** then removed use the **Boolean Remove** function.

Importantly, taking a look at the **right hand side vertical wall** of the cut (**above**), we can see that due to some **missing data** and the **hole** being '**filled**' when the **mesh** was created, the wall is not quite vertical. We can solve this issue by **Moving** a **copy** of the **surface** we generate from the **opposite wall**.

188 Select the mesh to **activate** the **Mesh Edit** toolbar and then select to **Manually Segment Mesh**. 

189 Choose to fit **Plane Primitives** and select the **sloped face** of the slot as **shown below**.

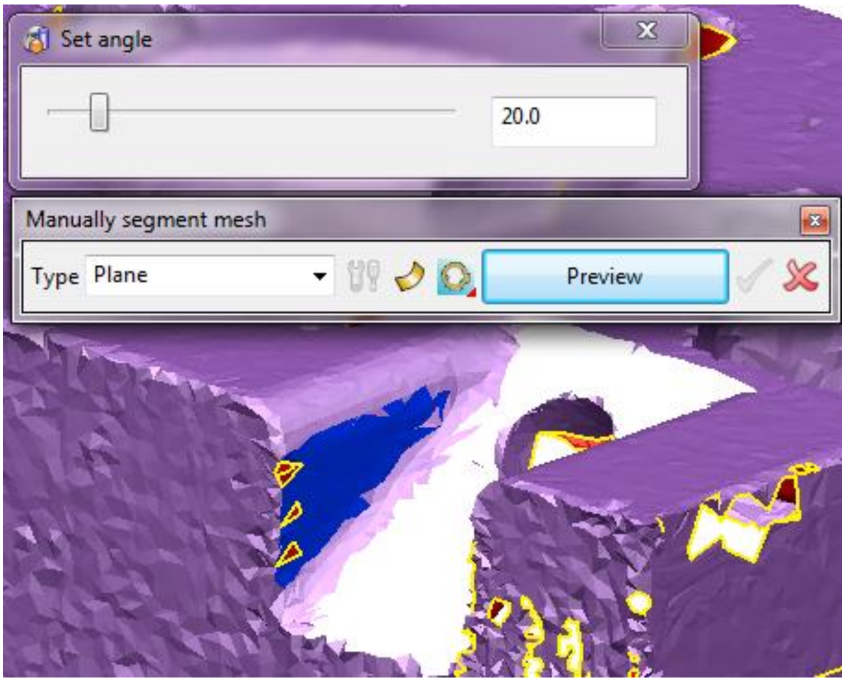


190 Select **Preview** and then when you are happy with the result, click the **green tick** to **create** the **surface**.

*As before, initially the **surface** will be **hidden** as to not obstruct the area you are working on, however the area you have created a **surface** over will be **shaded white**.*

191 While you are still in the **Manually Segment Mesh** tool select the **left hand side vertical wall** and **apply** a **primitive plane**.

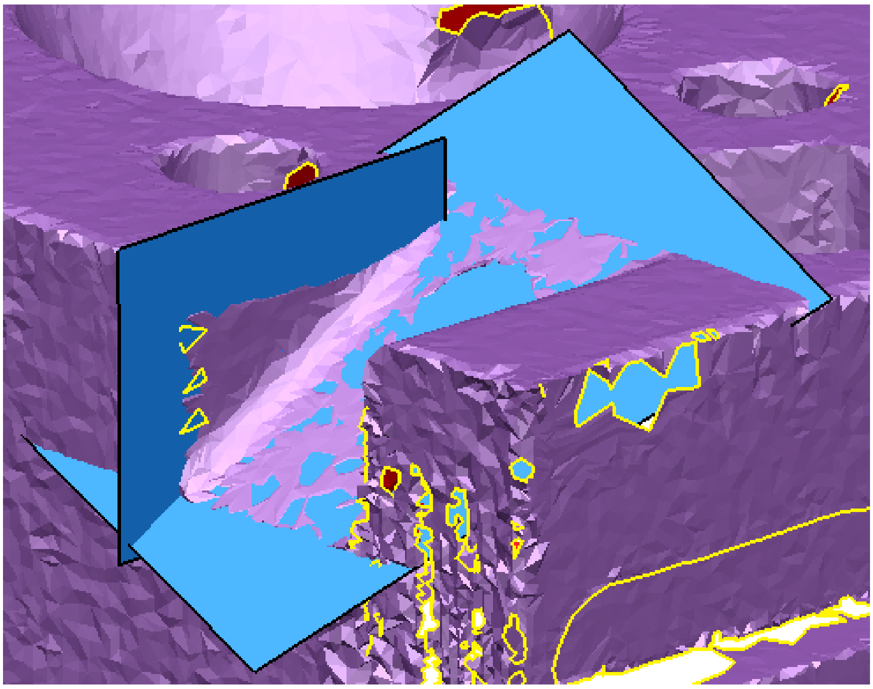
*As this is **near vertical,** the **plane** will **automatically** be **snapped vertical** with respect to the **Active Workplane**. This is done using the alignment angle in **Tools>Options>Objects>Mesh**. Its default value is set to **1°**, meaning any **primitive** created using these tools with an **angle** of **less than 1°** to a plane, will **snap** to the **plane**. This value can be **changed** to suit **your data**.*



192 **Click** the **red cross** to **close** the **form**.

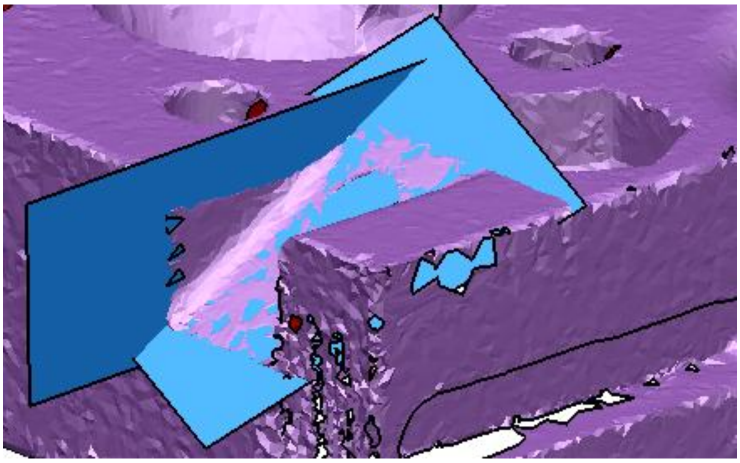
193 Take a **View from Top** (**Ctrl+5**).

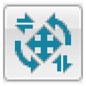
*This **view** will allow us to determine whether or not, with the **current settings,** the **vertical plane** we **expected to** has **snapped** to the **vertical**. If the **surface** hasn't, increase the **alignment angle** to around **2-3°**.*



To allow us to **create** a **solid** to **remove** from the **main block** we are going require the **Create Solid from Untrimmed Surfaces** function we used earlier. For this we need to **resize** the **left vertical surface** to reach the **complete length of the cut** and then **offset** a **copy** across **20mm** to act as the **right hand wall**.

194 Select the **vertical surface** in the **graphics area** and by **clicking** and **dragging** the **edge increase** its **width** to **fully intersect** the **angled face** of the **mesh**.



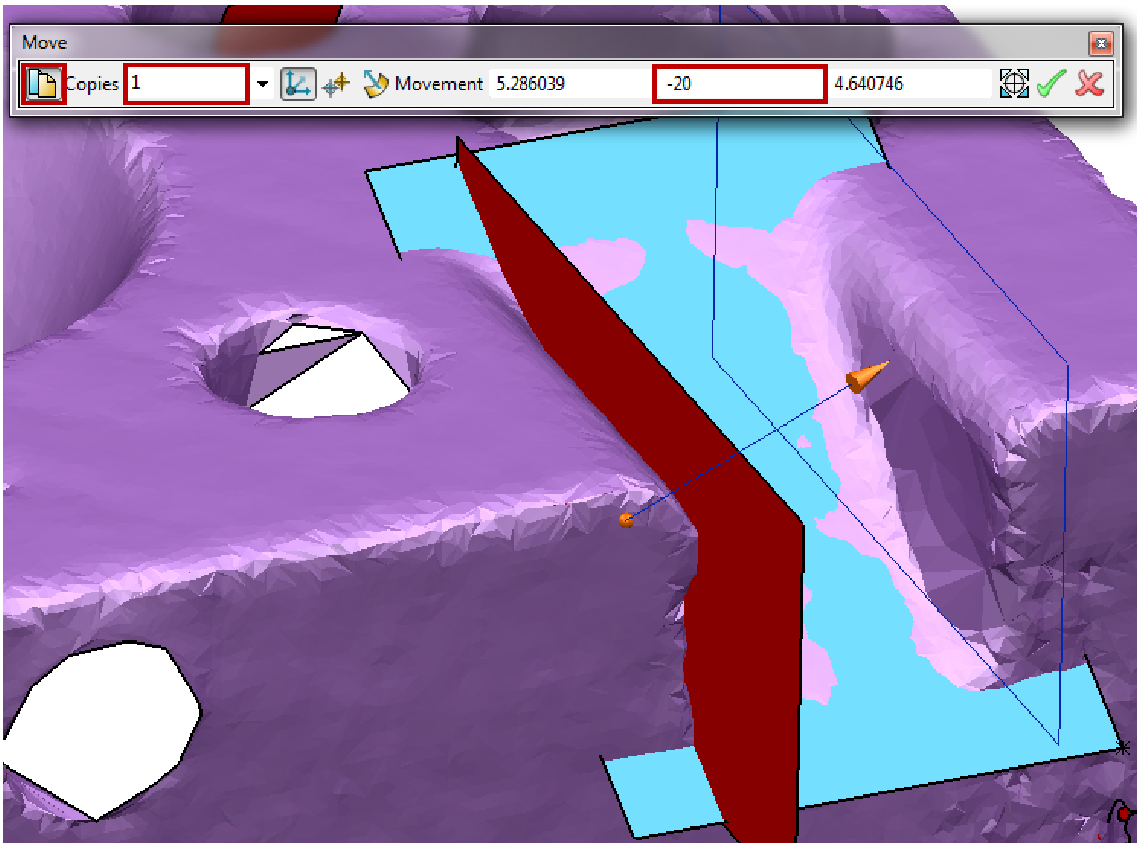
195 Select the **vertical plane surface** and **navigate** to the **General Edits**  menu and choose the **Move**  function.

196 Select create a **single copy**.

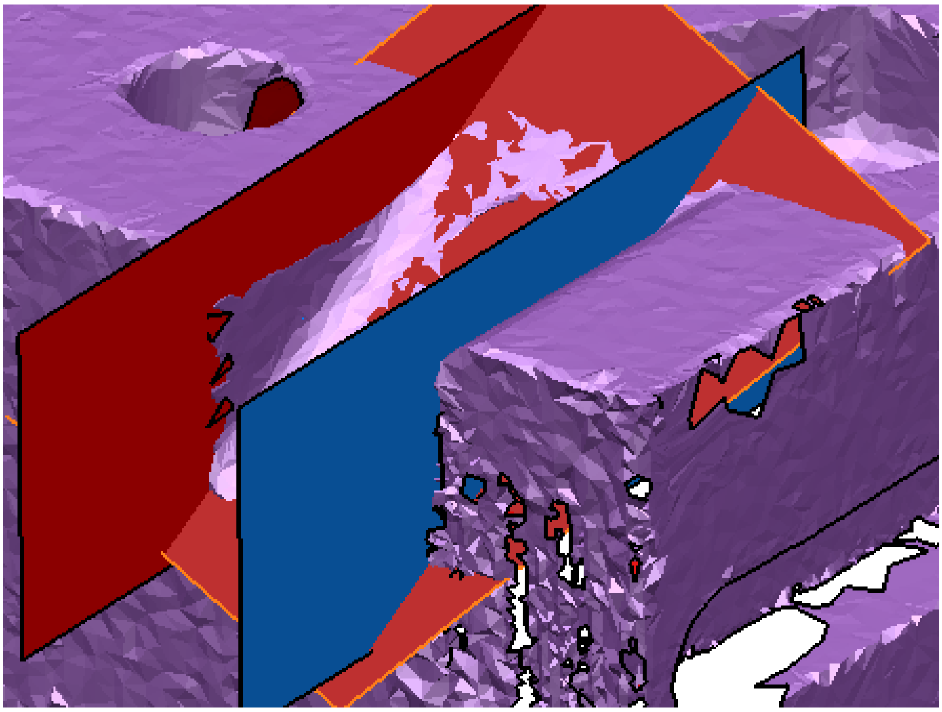
197 In the **graphics area dynamically click** against to **opposite vertical wall**. This will show a **preview** of the **new surface**.

198 **Edit** the **Y coordinate** to **move** a distance of **-20mm**.

*We are editing the **Y Coordinate** due to the **orientation** of the **active workplane**. This also makes the **X** and **Z component irrelevant**.*

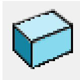


199 Next we need to **reorientate** the **surfaces** so the **negative 'red' side** of the **surfaces** are **facing inwards**. **Right-click** on the **two surfaces** that are currently in the opposite direction and **select reverse**.



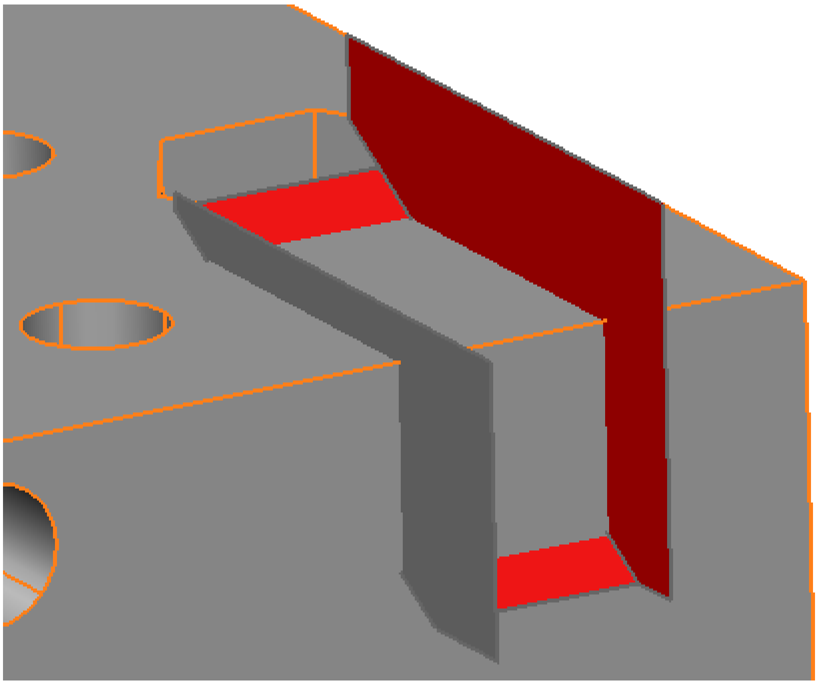
200 **Blank** the **Mesh** (**Ctrl+J**).

201 **Quick Select ALL Surfaces**. 

202 Select **Create a Solid from Untrimmed Surfaces**  from the **Solid**  menu.

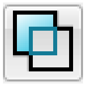
203 As before, move the **slider** across to **Keep** and **manually delete** the parts of the **faces** we **do not require** for this **open solid**.

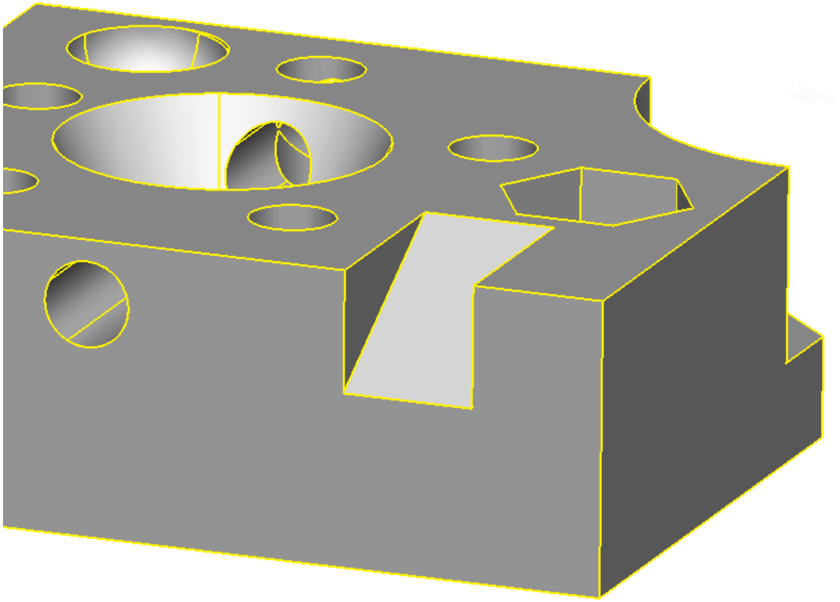
 *We are looking for an **open channel** similar to **below**.*



204 **Unblank** the **Mesh** (**Ctrl+K**).

205 **Turn OFF Level : 7 Mesh** and **Turn ON Level : 10 Main Solid**.

206 **Ensuring** the **Main Solid** is **Active** use the **Boolean Subtraction**  to **create** the **cut**.



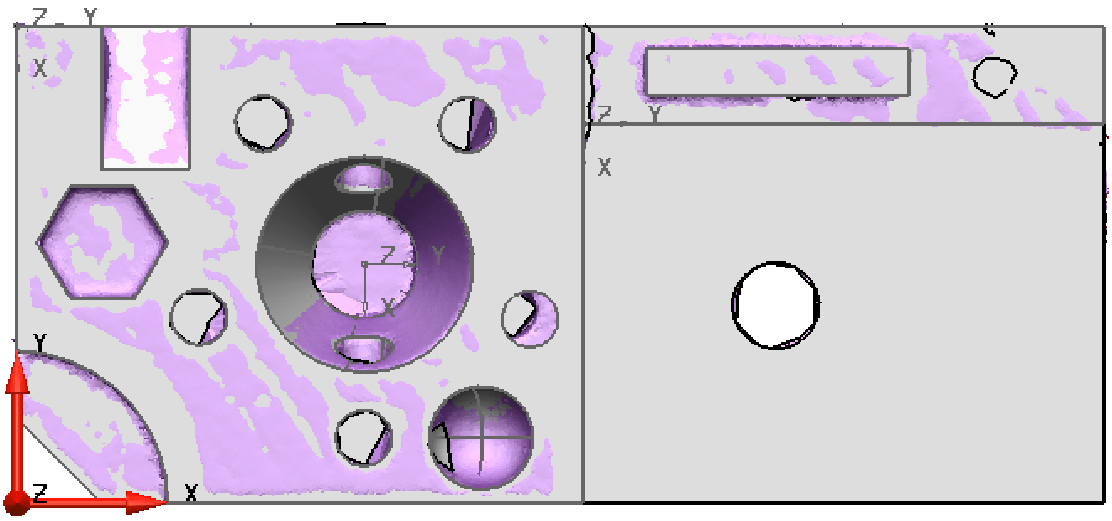
We have successfully created the **cut**. We can check the **dimensions** using the the **dimensioning tools** located in the **Annotation**  toolbar. To **finish** this **cut** we will create the **fillets** later.

207 **Turn ON Level 7 : Mesh**.

Creating the Organic Face

We have completed most of the **standard geometrical features** on this **part** and now we need to focus on the **organic formed faces** and **track** on the **right hand side** of the **model**.

Firstly, to avoid any of the next **surface/solid** operations interacting with the other parts of the **solid model** we will **split the solid** to isolate the particular **area of interest**, this in turn will make the process **much easier**. On this part we will **split the solid** at the **step** along the **Y Axis** direction.

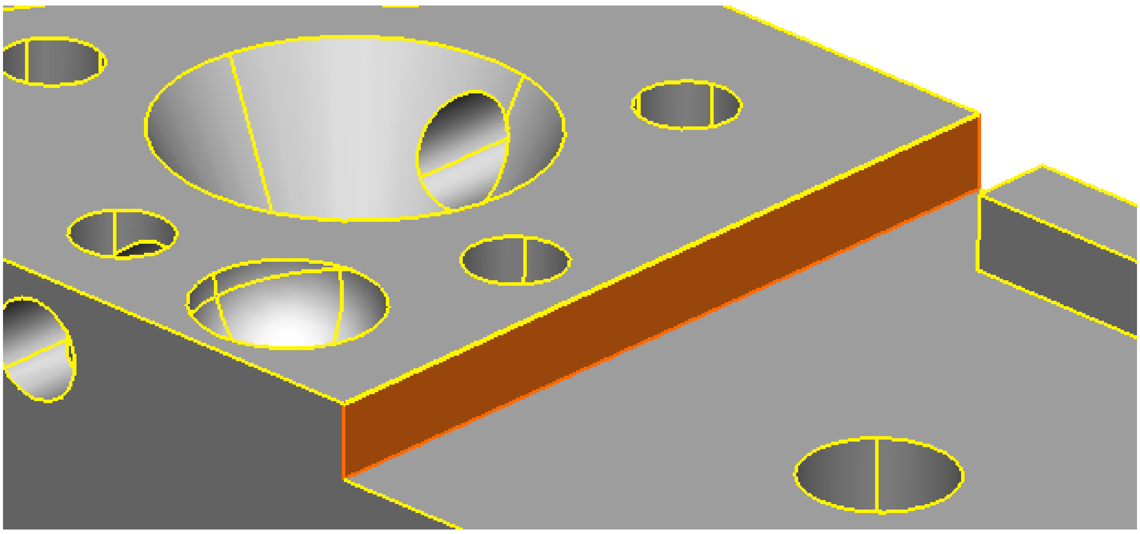


208 Turn **OFF Level 7 : Mesh**.

209 Select an **Isometric View** (**Ctrl+3**).

210 Select the solid and then **Select Individual Faces**  from the **Solid Edit Toolbar**. This will allow you to **select** and then **edit** the **faces** of the **solid**.

211 Select the **vertical face** forming the **raised step** in the **part**, as shown in the **below image**.



212 From the **Solid Feature**  menu select **Split Solid**. 

*This will **split** the **solid** using the **plane** of the **selected face**. As you have already made the **selection** the **dialog** will **not appear** and the **operation** will **automatically** be **completed**.*

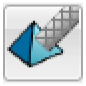
This operation has split the active solid into **two separate solids**. This has helped us to **isolate** the solid **area of interest,** while allowing us **greater control** of the **area affected** by further **solid operations**.

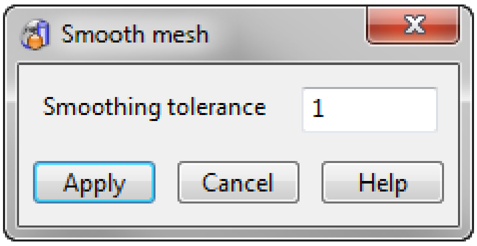
213 Raise the **Levels Dialog** and **Create** a new **Level 11** called "**Spare Solids**".

214 Select the **left hand solid** and place it on **Level 11 : Spare Solids**.

215 Ensure **Level 20** is switched **OFF**.

216 **Blank** (**Ctrl+J**) the **solid**.

217 Select the **Mesh**. Then select **Smooth Mesh**  from the **Mesh Edit Toobar**.



*This **tolerance** is a **distance** which determines the **nodes** that are **affected** by the **smoothing operation**.*

218 Using a **smoothing tolerance** of **1**, select **Apply**.

*This has **smoothed** the **mesh surface** which will allow us to **create smoother curves** using the **Oblique Curve** function from the **mesh**.*

219 Change the **Principal Axis** to **X**.

220 Select the **mesh** in the **graphics area**.

221 From the **Curve**  menu select **Create an Oblique Curve**. 

*The **Oblique Curve** tool allows us to easily create **multiple section curves** through a **selected mesh**, **solid** or **surface model**.*

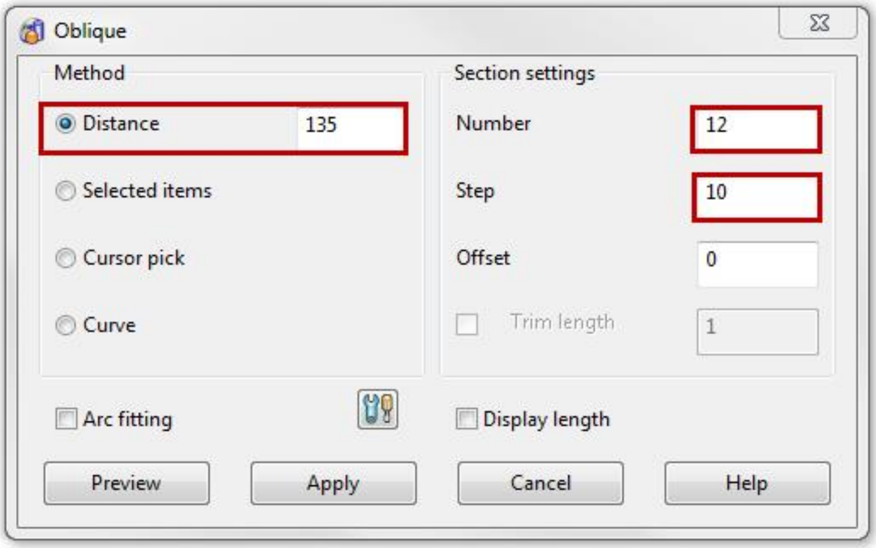
222 In the **Oblique Curve** dialog select **Distance** and enter and **distance** of **135mm**.

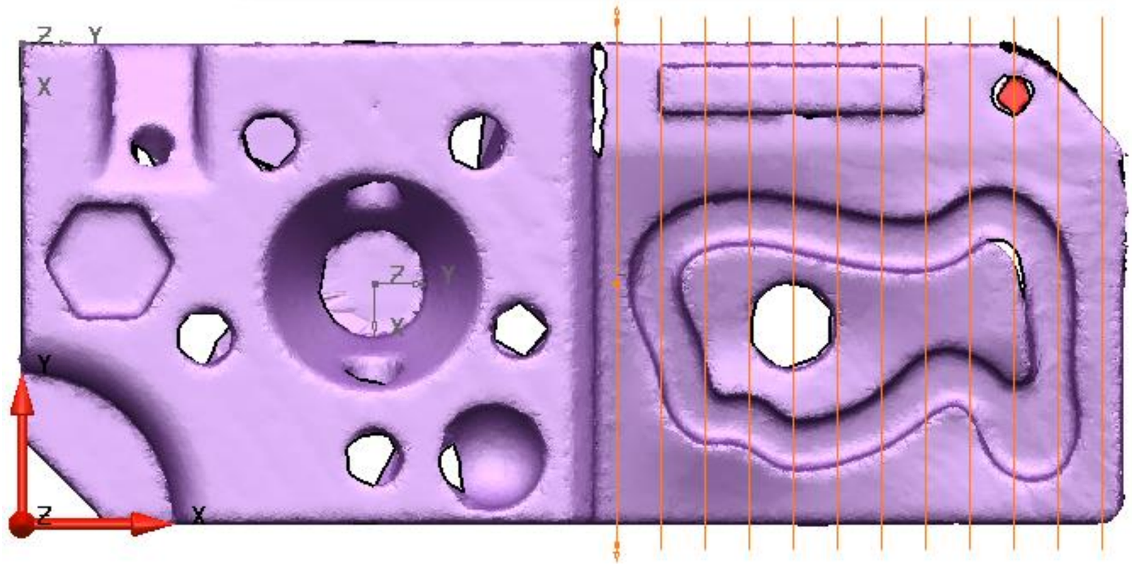
*This is the **offset distance** from the **Active Workplane** down the **Principal Axis** (**X**) until the **first section curve**.*

223 *In the **section settings** area of the dialog on the **right hand side** select a **Number** of **12** and a **Step** of **10mm**.*

This will create **12 section curves** at a **spacing** of **10mm** between each **curve**.

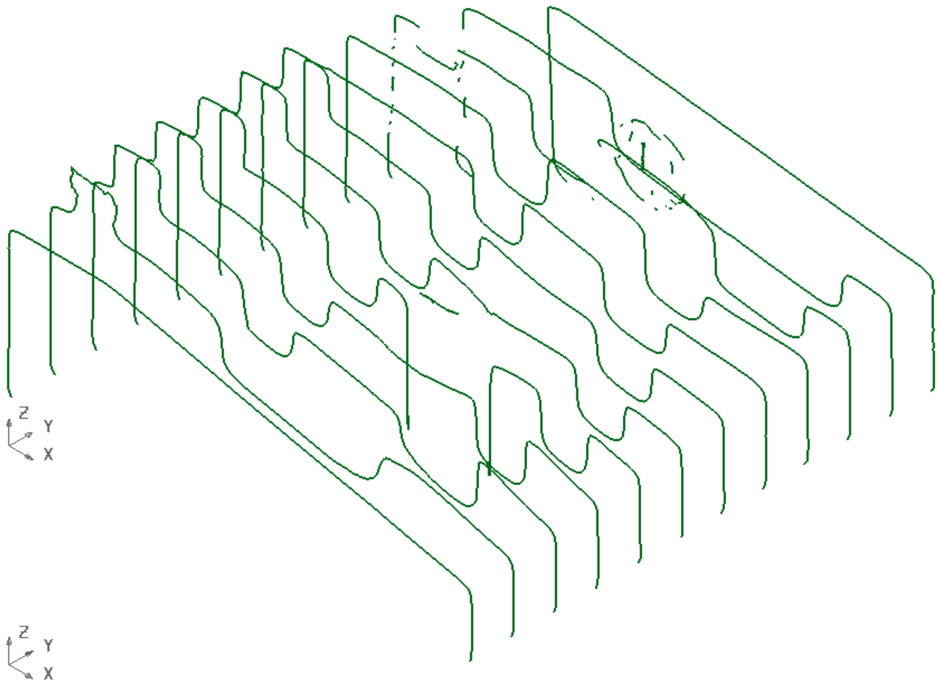
224 Select **Preview** to see the curves, the select **Apply**.





225 **Turn OFF the Level 7 : Mesh**.

226 Select an **Isometric View** (**Ctrl+1**).



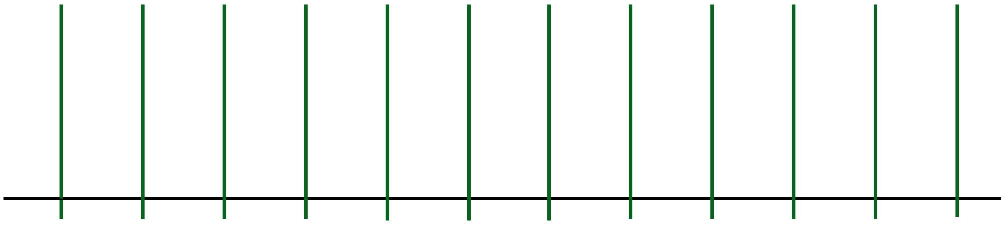
The curve method will create the **best fitting surface** for this type of **free form shape.** We have taken **12 section curves** across as much of the **surface** as possible to give as much data as possible to create the **best surface.**

In this example we will need to **edit** and **repoint each** of the **curves** in order to then create a **surface**. As we know from the **surfacing** chapter these **curves** and **curve keypoints** will become the **Longitudinal** and **Laterals curves** within the **surface.** We also require the **curves** to have **similar start/end point**, have the **same direction** and **number** of **keypoints**.

Firstly, we will trim the curves at the **start and end points** to **remove unwanted data** and produce even **start and end points**.

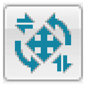
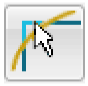
227 Select a **View from Top** (**Ctrl+5**).

228 Using a **Single Line** tool create a **line** that **intersects** (when viewed from above), and is **perpendicular** to the **section curves** slightly in from one end as shown in the **diagram**.



We will use this **line** and the **limit selection** tool to **trim** all the **section curves** and provide a **uniform curve start point**.

229 Ensure the **Principal Axis** is set to **Z**.

230 From the **General Edits**  toolbar select **Limit Selection**. 

231 In the dialog select the **Line** as the **Cutting Object** and click the **padlock** to **lock** the **cutting object**.

232 Next, from the furthest **right hand flyout menu** in the dialog select **Workplane Project Mode.** 

233 In the **graphics area**, **select** and **limit** the **ALL section curves** to this **line**. The **workplane projection mode** will **limit** using a **plane** created by the **line** and the **principal axis.**

*By **locking** the **Cutting Object** we will **not** have to keep reselecting it for **each limit**.*

234 Once complete **repeat this process** for at the other end to create **uniform curve end points**.

*At this end some of the **curves** may **not reach** the **line** towards the **right hand edge** of the part. Don't worry, we will **extend** these later.*

235 **Delete** the **two lines.**

We will now **edit each section curve independently**. In this example we will walk through **two**, and you can complete the others on your **own**.

236 From the **Levels** dialog, create a **New Level 20 : Section Curves**. For the moment leave it **Switched OFF**.

237 Select **ALL** the **section curves** in the **graphics area**, while **avoiding** any **noisy data**.

*Use the **CTRL key** to **multi select**.*

238 Place the **curves** on the **Level 20 : Section Curves**. They will **disappear** from view as the level is currently **OFF**.

239 **Select** and **Delete** the **noisy data** from **graphics area**.

240 **Turn ON Level 20 : Section Curves**.

241 Select the **first section curve** in the **graphics area** and **Blank Except** (**Ctrl+K**).

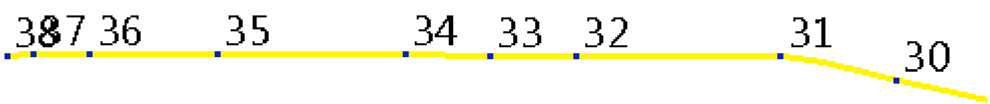
242 Take a **View from Left** (**Ctrl+4**).

243 From the **Curve Edit** toolbar **turn ON curve numbering**. 

*This will show the **direction** and **number** of **curve points**.*

As this **section** has created a **curve** with **good quality data**, we can use this to help create the further curves.

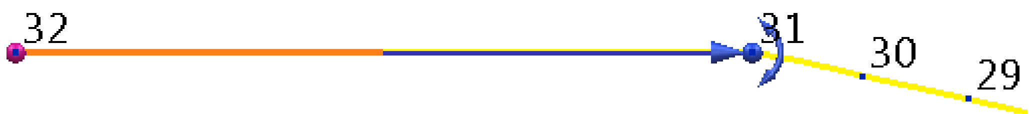
244 **Zoom** into the **flat area** of the **curve**.



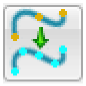
245 **Select** and **Delete** the **curve points 32 – 37**.

*Your data may have **different points** and **number of points**. Therefore **delete** the points **inbetween** the **change of direction** onto the **flat surface** up to (not including) the **end point**.*

246 Select the **end point** and ensure the **tangent** is pointing **directly towards** the **next point** using the **handles**.



247 **Repeat** the **above step** to ensure the **second last point** is **directed toward** the end point. This will create a **straight line** for the **top area**.

248 From the **Curve Edit** toolbar select **Repoint Curve**. 

249 In the dialog select **20** for the **Number of points in range**. Click **Apply**.

*This will **repoint** the **curve** to have an **evenly spread 20 points** along the curve. We will use this as a standard on the rest of the **curves** which will then form a **smooth surface**. This value is dependent on many variables including the **complexity** and **average length** of the **curves** used, in this case **20 points** as a **first interation of the surface**.*

We now need a **new level** to split our **original** (**Level 20**) and **edited section curves**.

250 Raise the **Level** dialog and create a **New Level 21 : Section Curves Edited**. Click **OK** to **create** and **close** the dialog.

251 **Ensure Level 21 is Turned OFF**.

252 In the **graphics area** select the **edited section curve** and place it **Level 21**.

253 **Unblank** (**Ctrl+L**).

254 **Select** the next **section curve** to **edit** and **Blank Except** (**Ctrl+K**).

*This will **blank everything** else from the **graphics area except** from the **single section curve**.*

255 Take a **View from Left** (**Ctrl+4**).

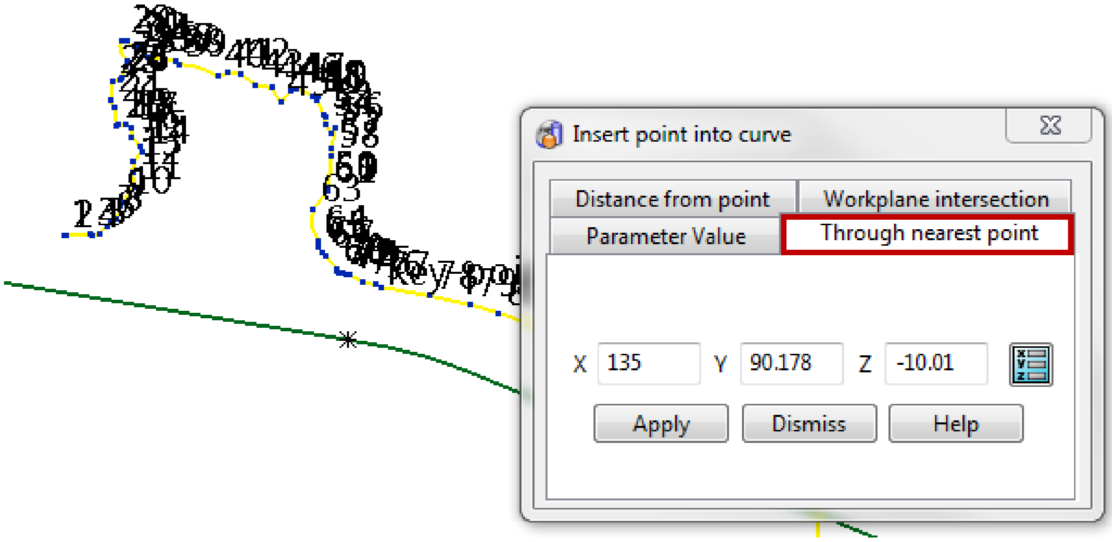
256 **Switch ON Level 21** so we can **compare** the **two curves**.

In this case we are **comparing** the **edited first section curve** with the **second one** which we are editing. Firstly, we need to create the **uniform flat area** at the **top of the face** and then need to **remove** the **unwanted data** from the **section curves** create by the **recess tracks** on the **formed face**.

Point 17 in the image on the **edited curve** shows where the **change** of **direction** from the **flat** onto the **formed face** occurs**.** Therefore we require a **curve point** at the **nearest location** on the **second curve** to flatten the **top face area**.

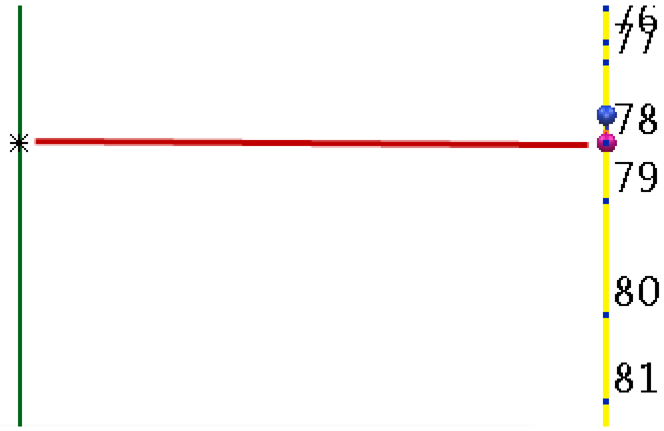
257 Select the **second curve**, then from the **Curve Edit** toolbar choose to **Create Point**. 

258 In the dialog select **Through Nearest Point** and then in the **graphics area** select the **location** of **Point 17** on the **first curve**, this should highlight as a **Keypoint**.



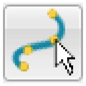
*Selecting this location has **automatically filled** the dialog **X**, **Y** and **Z** **Values** and a **new point** will be created at the **position** on the **second** **curve closest** to this **location**.*

259 Click **Apply** to **create the point** and then **dismiss** to **close** the **form**.



As you can see the **new point created** (**Point 78**) is **adjacent**. This means we can now create the **flat inbetween** this **point** and then end of the **curve**, which will **match** the **first curve**.

260 Select the **second curve** (the one we are **currently editing**) in the **graphics** **area**.

261 From the **Curve Edit** toolbar select the **Point Selector** option. 

*We will now **remove** all the **points** inbetween **1** and **78** (these points may differ in your **session**. These **points** represent either end of the **flat** **area** we wish to create to **match** the **first curve**.*

262 Inside the dialog select the **points 2 – 77**. Holding the **Shift** key and then **2**, then **77** will **select** the **entire continuous region**. Select **OK**.

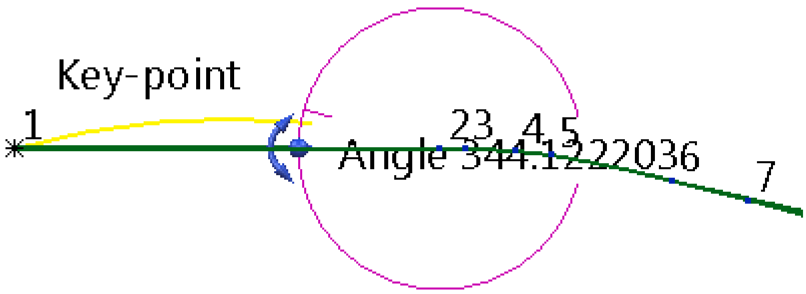
263 From the **Curve Edit** toolbar select **Delete Points**  to **remove** the **selection of points**.

Next we need to **flatten** the **curve** between this area (**Point 1 and 2**) using the **tangent handles** by selecting **each curve point** and **snapping** the **handles** to the **opposing point**, we will now go through this **next**.

264 Select the **first point** and then its **tangent** handle, **click** and **drag** it, **snapping** it to the **second point** to direct the **angle** of the **curve** from the **first point exactly towards** the **second point**.

*This single step may not straighten this part of the **curve straight away**. This is due to the **tangent** and **magnitude** from **point 2** to **point 1**, which will will **edit next**.*

265 **Repeat** the **above step** from the **second point** to the **first point** (**opposite** to **before**).

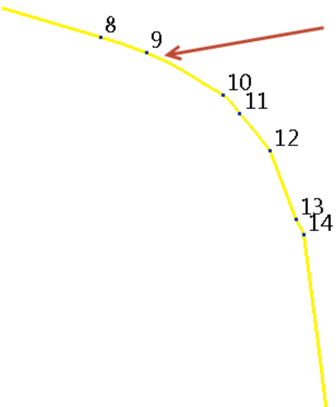


The **next step** in the **process** is to **remove** the **control points defining** the **area** created from the **recess track** on the **mesh**, this will be completed in a similar way as we have just done on the **previous operations**.

266 **Turn OFF Level 21 : Section Curves Edited**.

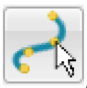
267 Select a **View from Left** (**Ctrl+4**).

268 **Zoom** into the **area** where the **curve drops** into the **track**.

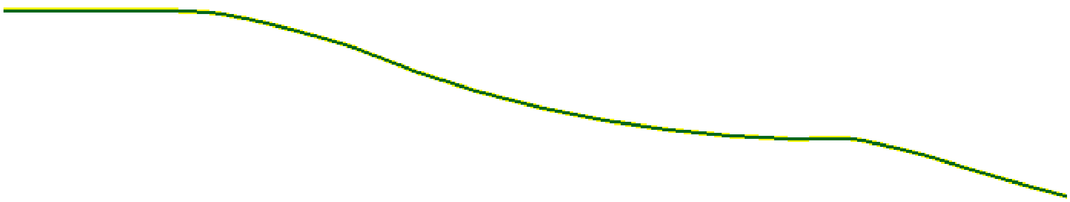


We can see that **Point 9** in this **image** is the **last point** before the **curve drops down** into the **recess section**, so we would remove **Point 10 onwards**.

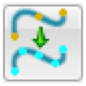
269 **Repeat** this **procedure** at the other end of this **section** of the **curve** to determine the **final point** we will **remove.** In the current session it can will be **Point 61** although you should use **your discretion** to **determine** this figure.

270 Using the **Point Selector** , select **Points 10 through to Point 60**.

271 From the **Curve Edit** toolbar choose to **Delete the Selected Points**. 



As you can see we have **removed** the **unwanted data** from the **second section curve** by **deleting** the **keypoints**. The **shape** of the **curve** where the **recess track** is formed in the **mesh** is **not critical**, however it is **important** that the **curve flows** to **avoid sharp discontinuities** in the **surface**. We will next **repoint** the **curve** to have the **same number** of **points** as the **first section curve**, and **ensure** the **same direction**.

272 Select the curve and then **Repoint Curve**  from the **Curve Edit** toolbar.

273 As before edit the **Number of Points in Range** to **20**. Select **Apply**.

274 Select the **curve** in the **graphics area** once again and place it on **Level 21 : Section Curves Edited**.

275 **Unblank** (**Ctrl+L**) the **orginal section curves**.

276 **Turn OFF Level 20 : Section Curves** and **Turn ON Level 21 : Section Curves Edited**.

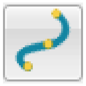
If we **compare** the **direction** (**start point** to **end point**) of the **first two completeted section curves**, you may see they are **different**. We will now **reverse** the **direction** to **ensure** they all follow the same route to **produce** a **surface**.

277 **<u>If your curve is reversed</u>**, select the **second curve** in the **graphics area** and select **Reverse the Direction**  from the **Curve Edit** toolbar.

*It is **not important which direction** the curves are for this example, as long as they are all the **same**.*

278 On your own using the **knowledge** you have **gained attempt** to **edit** and **recreate** the **remaining curves** in order the **complete the curves** for the **surface**.

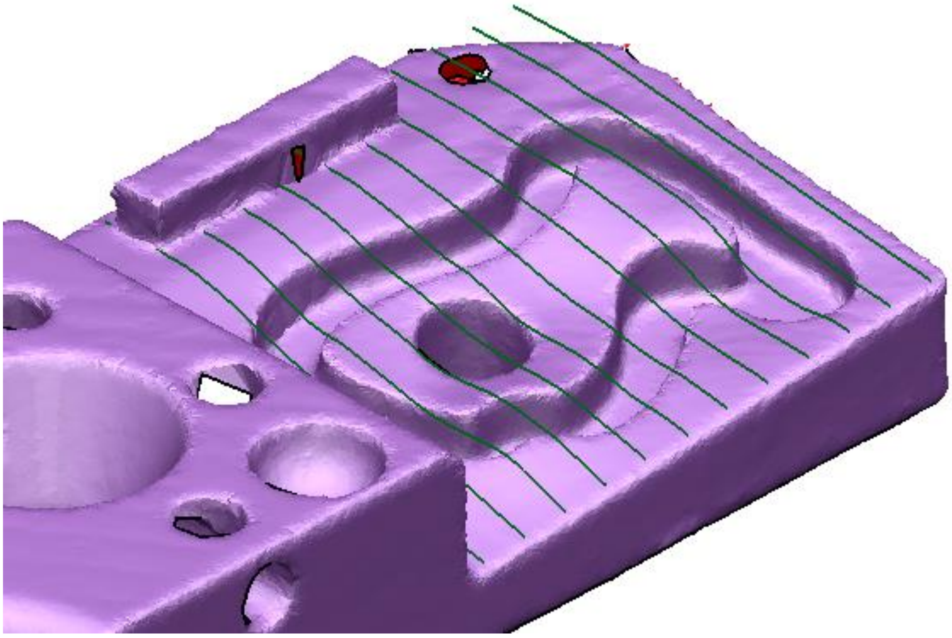
*Ensure all the **curves start** and **end** at the **same location** as the **edited versions**, have the **same direction** and have the same number of **keypoints**.*

*If the **curves** are not **joined together**, these can be **joined** using a **Bezier Curve***  *from the **Curve** menu.* 

279 Ensure **ALL** the **section curves** are on **Level 21 : Edited Section Curves**.

280 **Turn OFF Level 20**.

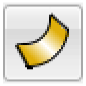
281 **Turn ON Level 7 : Mesh**.



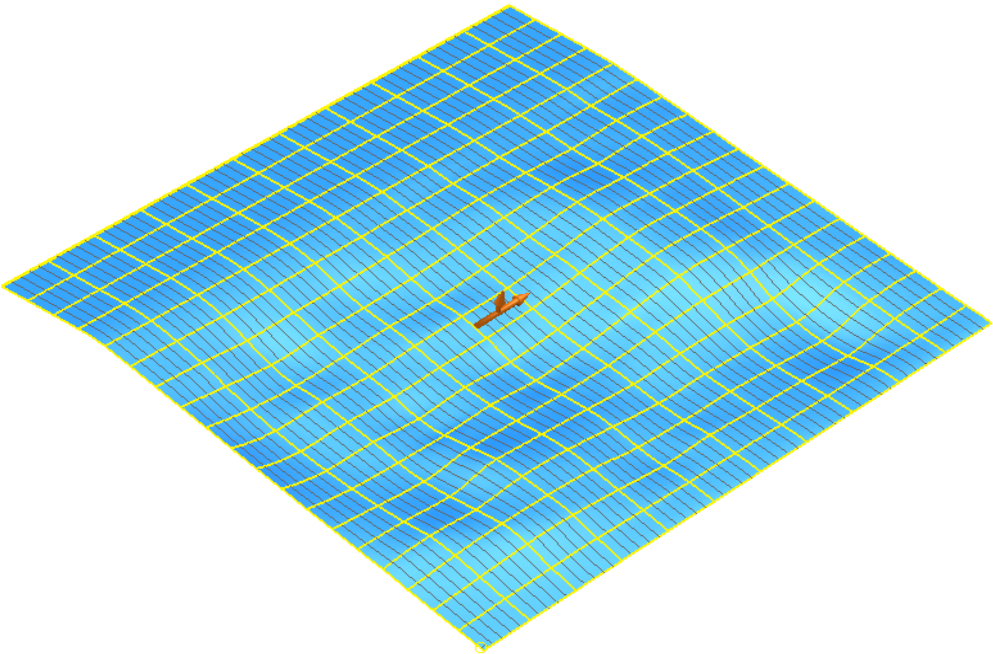
Once we have completed **ALL** the **curves**, as **shown above**, we are now ready to create the **surface** using the **Smart Surface** tool, we will do this **next**.

282 **Turn OFF Level 7 : Mesh**

283 From the **Selection Flyout** choose **Quick Select ALL Wireframe**. 

284 Select the **Smart Surface**  tool from the **Surface** menu. 

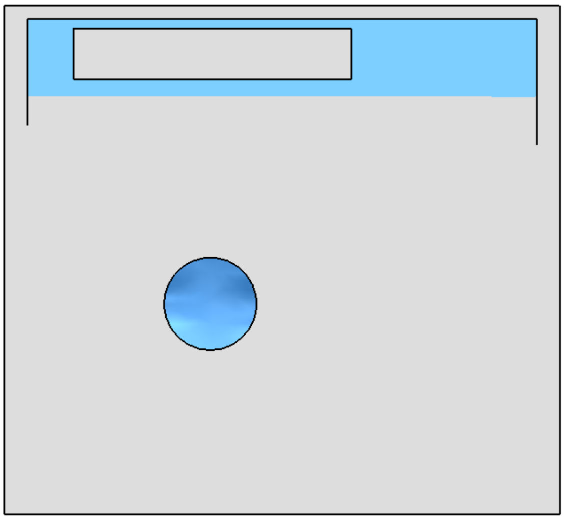
*The selection of **12 section curves** should **automatically** create a **preview** of a **surface 'from separate'**. **If not** ensure you have the **correct selection** and **all** the **curves** are **correct** and **try again**.*



The **surface** has been created by joining the **curve keypoints** together to create **Longitudinals** and the **curves** themselves as the **Laterals**.

285 **Turn ON Level 10 : Solid Model** and **Turn OFF Level 21**.

286 Select a **View from Top** (**Ctrl+5**).

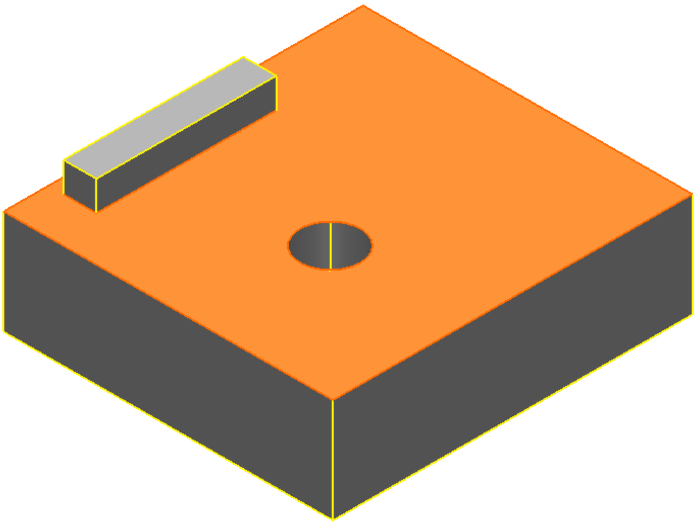


As you can see from the **image** above, as we create the **section curves slightly inwards** from **each edge** of the **surface**, it **does not extend fully outside** each **edge/face** of the **solid**. Next the **Limit Point** operation will be used to **extend** the **required surface edges beyond** the **edges** of the **solid**.

The **rectangular boss** will be **removed** when we use the **surface** and a **Boolean Remove** tool if **no action** is **taken**. To avoid this we can use **split solid** to create **two separate solids** before the **Boolean Remove** operation, this will **separate** it from the **solid** we are working on, and then **Add** them both **together** again.

287 Select the **Solid** and **Blank Except** (**Ctrl+K**).

288 Select the **top face** of the **solid**, it should **be highlighted** as **orange**.

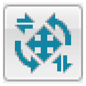


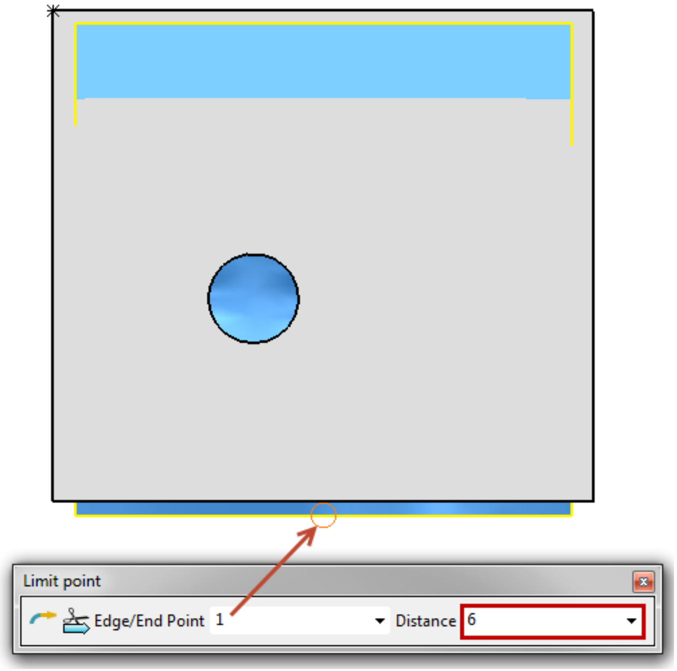
289 With this **face selected**, **navigate** to the **Feature**  menu and select **Split Solid**. 

 *As we have made the **selection** the **solid** will **automatically split without raising** the **dialog**.*

290 Select the **rectangular boss** and place it on **Level 11 : Spare Solids** with the other **solid** we have **split** earlier.

291 **Unblank** (**Ctrl+L**) to **show** the **surface**.

292 Select the **surface** in the **graphics area** and then navigate to **Point Limit**  within the **General Edits** menu. 

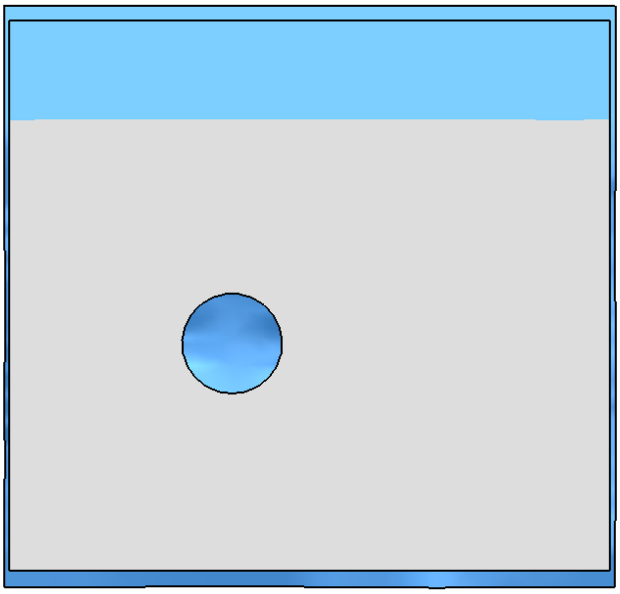


The **orange ring** attached to one of the **surfaces edges** links with the **selected Edge** in the **dialog**, whereas the **distance** is the **distance** to **extend** this selected **edge by**.

293 Extend **Edge 1** by a **distance** of **6mm** by typing **5** into the **distance box** and pressing the **Enter** key.

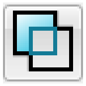
294 **Repeat** the above **process** for **edges 2-4** using a **distance** of **6mm**.

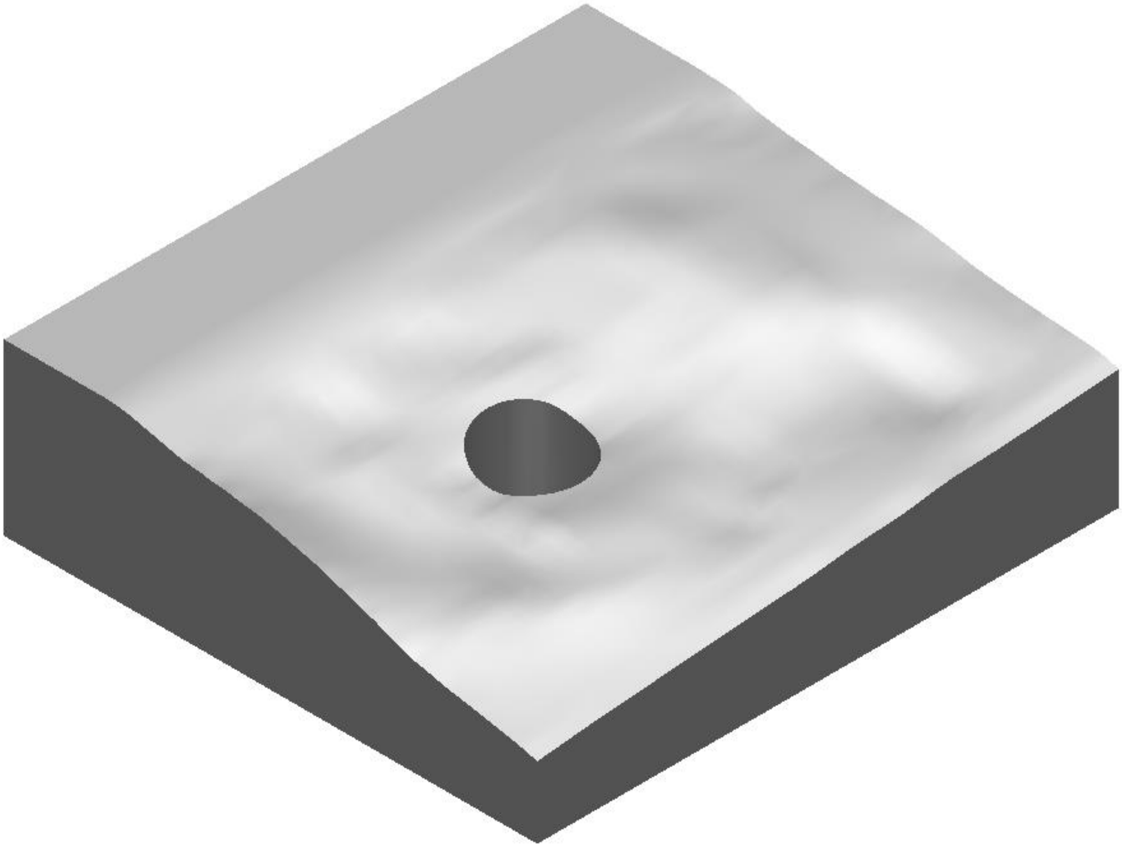
*This has ensured that **all** the **edges** of the **surface extend beyond** the **extremeties** of the **solid** in order to use the **surface** in a **Boolean Remove** operation. This figured was used after trial and error*



295 **Right-click** on the **surface** and select **Reverse**.

*From earlier we know that when using a **surface** in a **Boolean Subtraction** operation the area of the **solid** to the **red** (**inside**) of the **surface** will be **removed**.*

296 Select **Boolean Subtraction**  from the **Feature** menu  using the **Solid** as the **Primary Selection** and the **Surface** as the **Secondary Selection**.



We have created the **main formed face** of the part. To complete this area we need to **create** the **track recess**, we will do this next.

The **inner** and **outer curves** to define the **shape** of the **track** will be created using a **curve snapped to the mesh**.

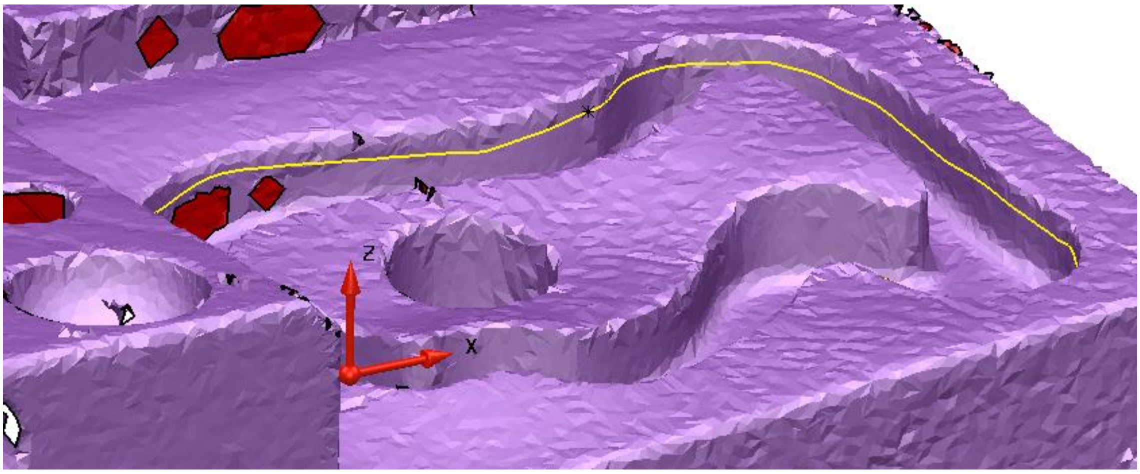
297 **Turn OFF Level 10 : Solid Model** and **Turn ON Level 7 : Mesh**.

298 From the **Curve**  menu select **Create a Curve Snapped to a Mesh**. 

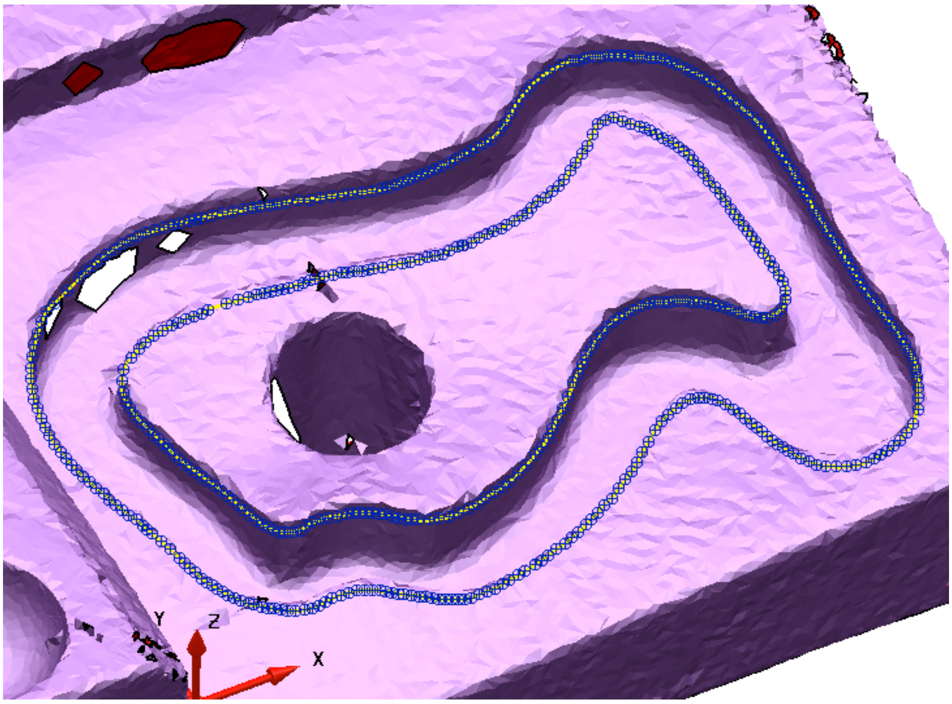
On the **inside** and **outside** of the **recess walls**, we will track a **curve snapped to the mesh** to provide the **outline** of the **shape**. It is **advisable** to keep the **curve** close to the **top of the walls** where possible to allow for any **draft angle** on the walls, where **varying** the **height** of the **curve** up the walls will change to **affect** the shape of the curve.

299 Track the curve by **clicking** around the **walls**, during this **process** your **mouse** and **keyboard** can be used to **reorientate the view**.

300 When almost complete, the **curve** can be **closed** and **created** to **double clicking** on the **first point**.



301 **Repeat** the **same process** as above for the **inner wall curve**.

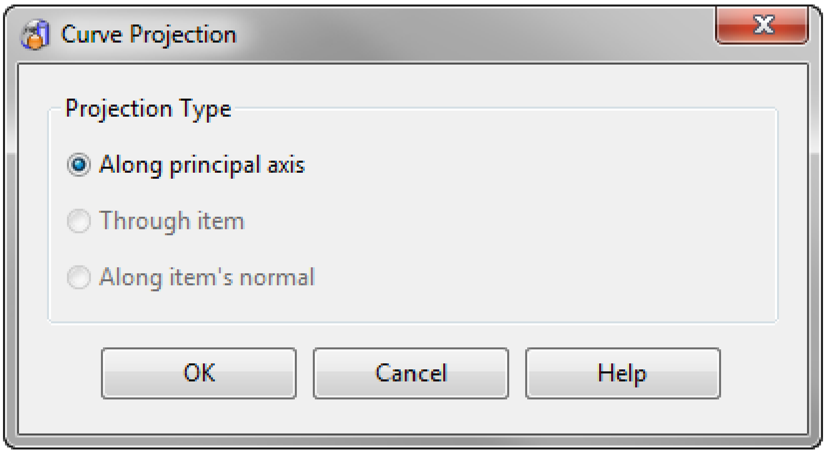


302 Switch to **activate Workplane 1** from the **workplane selector**.

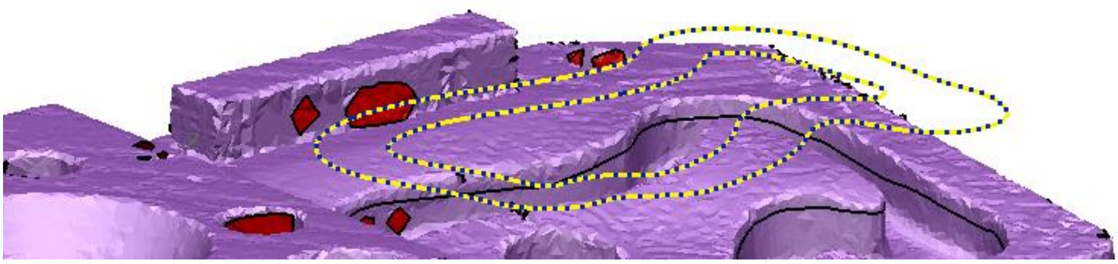
303 Ensure the **Principal Axis** is set to **Z**.

304 Select **both curves** forming the **inner** and **outer wall profiles**.

305 From the **Curve**  menu, select **Curve Projection**. 

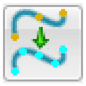


306 From the **Curve Projection** dialog select to project along the **Principal Axis** (**Z**).



 *This will project the **3D curves** and create **2D profile curves** on the **Principal Plane** of the **current active workplane**.*

307 **Select** and **Delete** the **curves snapped to the mesh**.

308 **On your own**, **Repoint**  both **curves** with **50 points** in **range**.

To enable us to create an **open solid** to **cut away** the **outer recess** into the **face** we also need to **shrink wrap** a **surface** forming the shape of the **bottom of the recess**.

309 Select a **View from Top** (**Ctrl+5**).

310 Select the **mesh** to **activate** the **Mesh Edit** toolbar and raise the **Manually Segment mesh** dialog. 

311 In the **Manually Segment** mesh dialog select **Shrink Wrap** as the **Type** and leave the **horizon angle setting** at **20.0°**.

312 Select **Preview** in the dialog to preview the **fit** of the **surface created**.

313 Click the **green tick** to create the **surface**, and then **dismiss** the **form**.

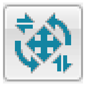
314 **Turn OFF Level 7 : Mesh**.

This should leave only the **surface** and **two curves.** Before we **extrude** the **outer curve** in order to create the **solid shape** we should ensure that the **bottom face surface extends beyond** where the **extrusion** will **intersect**.

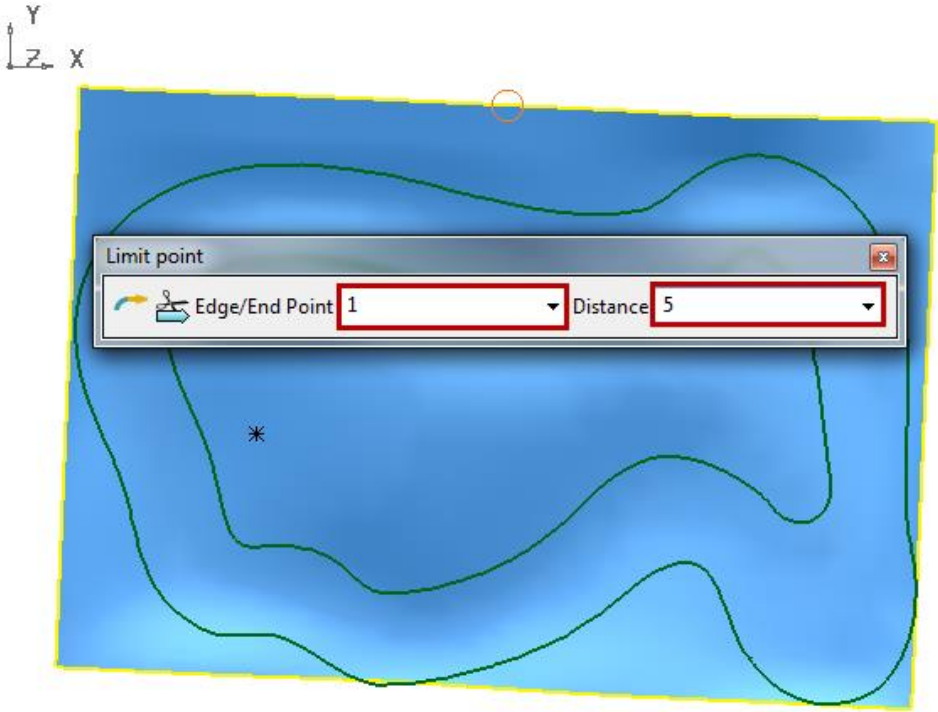
The **limit point** function will enable us to **extend** the **surface** in each **required direction**.

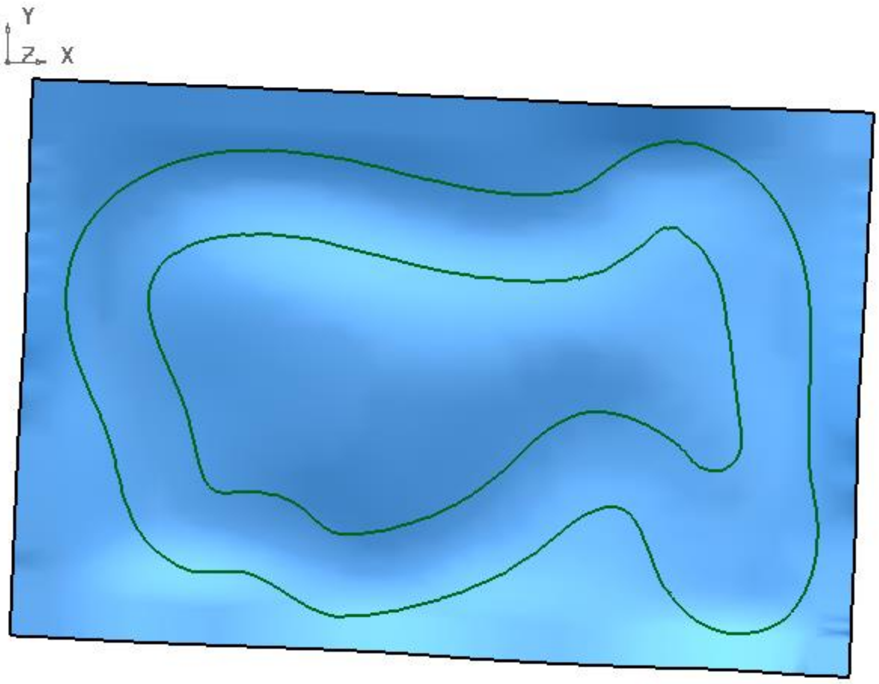
315 **Right-click** on the surface and **convert** it to a **Power Surface**.

As each edge of the surface is at the limits of the curve we will **extend each edge** by **5mm** to ensure the **surface extends beyond** the **outer curve**.

316 Navigate to the **General Edits**  menu and select **Limit Point**  from the **limits flyout menu**.

317 In the **Limit Point** dialog select each edge (**1-4**) in the **drop down** menu and then choose to **extend each edge** individually by **5mm** by pressing the **Enter** key.



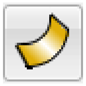
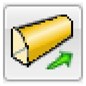


*Once **each edge** has been **extended** you should notice that, when taking a **View from Top** (**Ctrl+5**) the surface extends beyond the profile of the outer curve.*

Next we will **extrude** both **profile curves** as **surfaces** through the **shrink wrap surface** and use the **Create Solid from Untrimmed Surfaces** function to create an **'open' solid** to **cut** out the **recess** in the **face**.

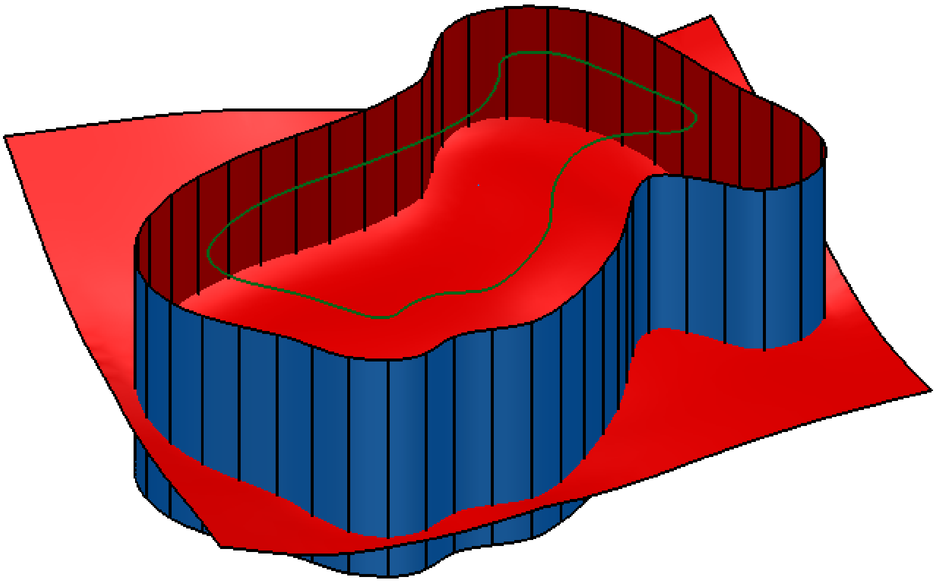
318 Ensure the **Principal Axis** is set to **Z**.

319 Select the **outer profile curve** in the **graphics area**.

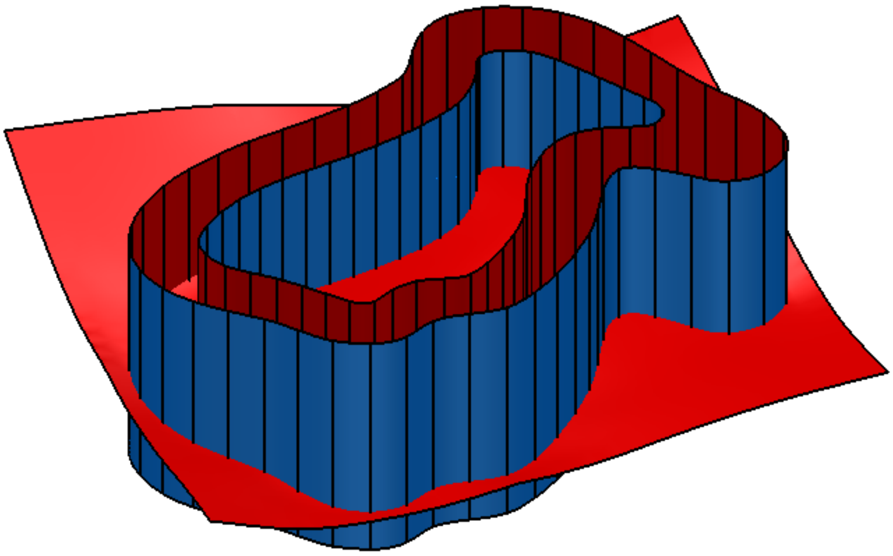
320 From the **Surface**  menu select **Surface Extrusion**. 

321 **Extrude** the **surface** down **Z** through completely through the **shrink-wrap** as shown in the below **image**. A **length** of **40mm** will suffice.

322 **Right-click** and **Reverse** the **Shrink Wrap** surface as it will be the area to the **red (negative) side** that will be removed.



323 **Repeat** the **same procedure** with the **inner profile curve**. **Reverse** this **surface if necessary** to look **similar** to the **below image**.



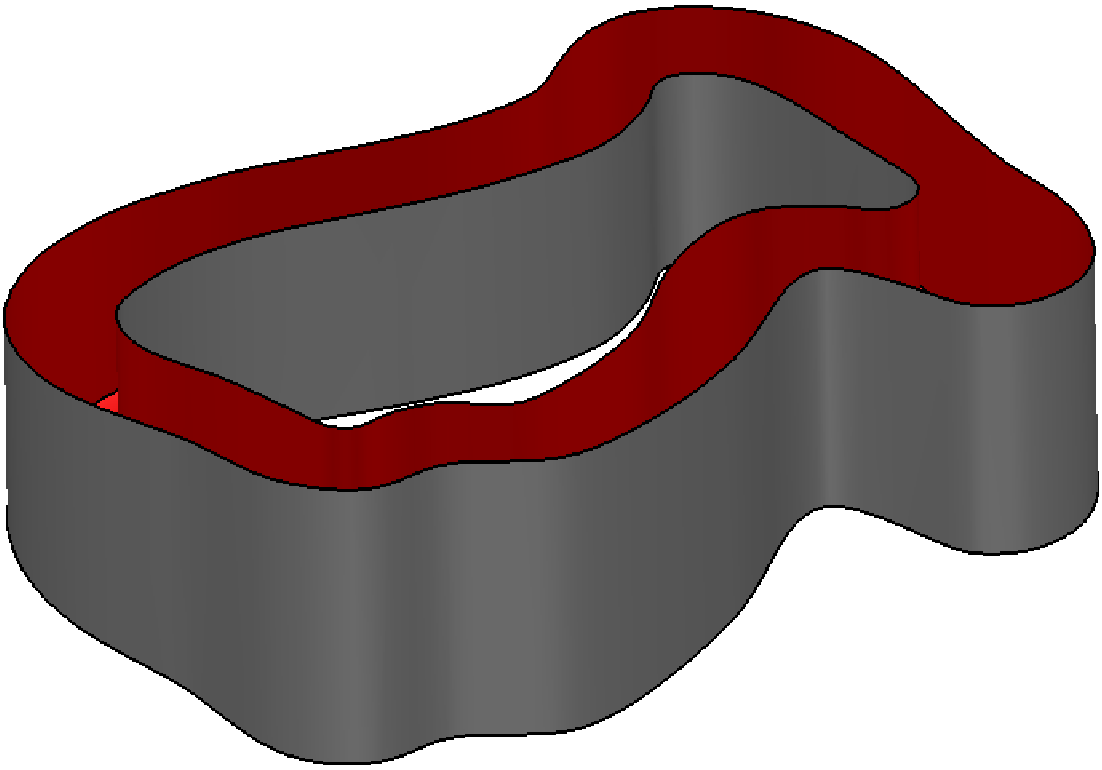
As we have **three separate untrimmed surfaces** that will form the shape, we need to combine them into **one complete surface**.

324 Select **ALL** (**3**) the **surfaces**.

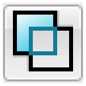
325 Navigate to the **Solid**  menu and select the option to **Create Solid by Trimming Surfaces**. 

326 Once you are happy with the **'open' solid** created select **OK** in the dialog to create the **shape**.

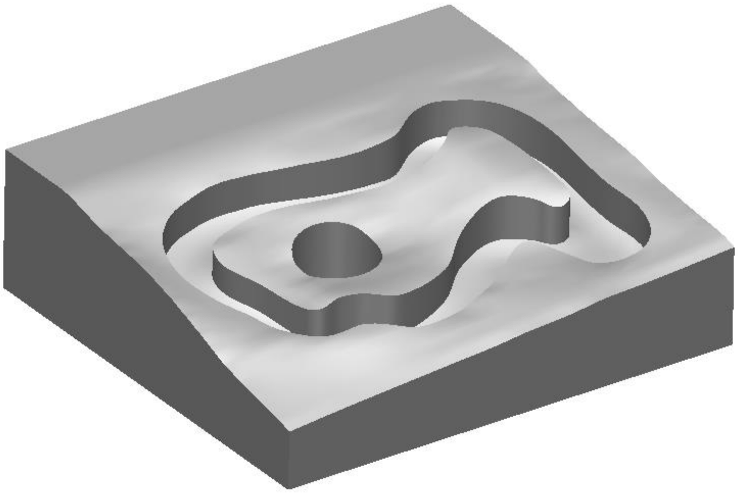
*The **solid shape** should look **similar** to below. You may have to move the **slider** to '**keep**' and then **manually select** and **delete each unwanted segment independently**. This will help us because we can use what is **effectively 3 separate surfaces** as **one complete object**.*



327 **Turn ON Level 10 : Solid Model.**

328 From the **Solid Feature**  menu select **Boolean Subtraction.** 

329 Select the **main block** as the **Primary Selection** and the **open solid cutter** as the **Secondary Selection**. Click **OK.**



*The **Boolean Subtraction** has been completed to create the **track** into the **formed face** of the part. Next we need to **add** together the **separate solids** to form **one complete solid** using the **Boolean Addition** tool.*

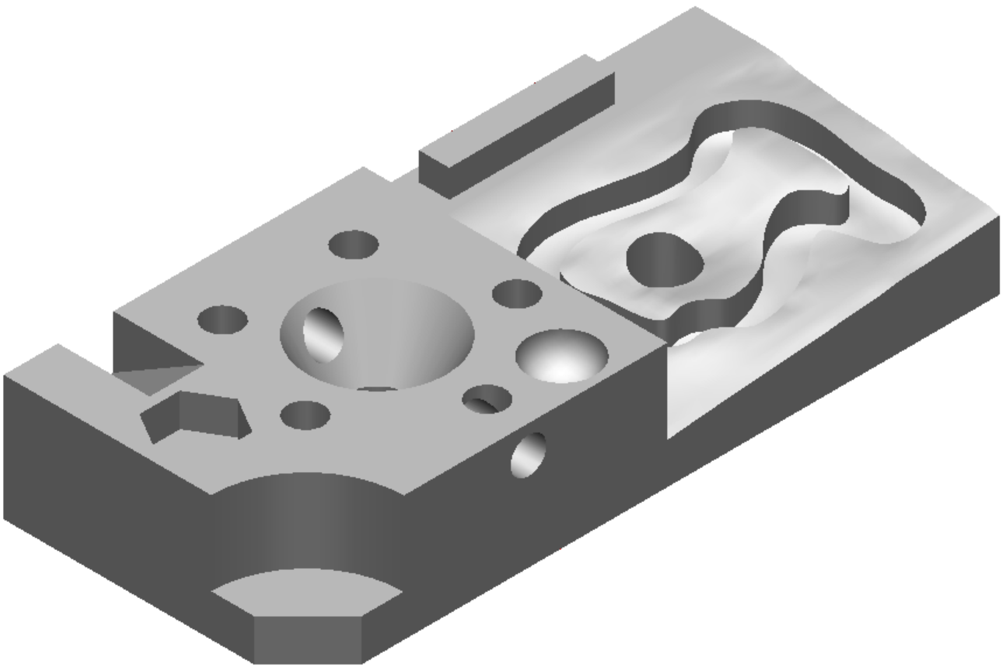
330 **Turn ON Level 11 : Spare Solids**.

331 From the **Selection** flyout choose to **Quick Select ALL Solids**.

332 Select **Boolean Addition** from the **Feature** menu. Select **OK** in the **dialog**.

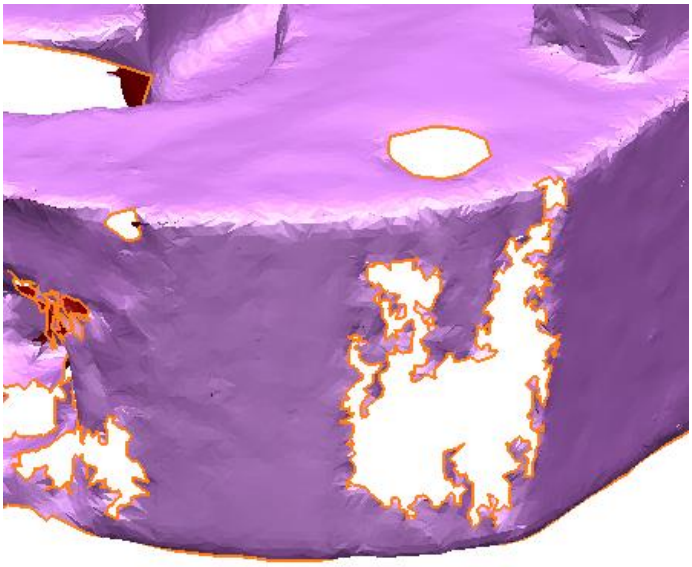
333 Select the **solid** and **ensure** the **Solid History tree** is shown.

*We have now **combined** all the **solids** into **one complete block**.*



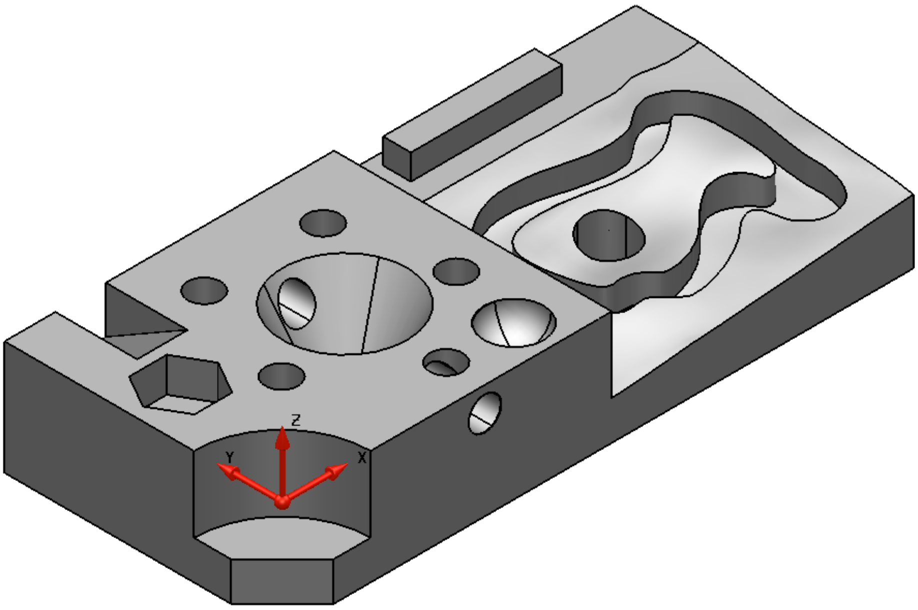
334 Ensure the **solid model** is placed on **Level 10 : Solid Model**.

335 **Turn OFF Level 10 : Solid Model** and **Turn ON Level 7 : Mesh**.



336 **On your own** use a **shrink wrap** surface along with the knowledge you have gained throughout this example to **remove** this **formed corner** from the **rear of the part**.

Finishing the Part (Fillets and Chamfers)



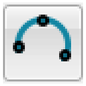
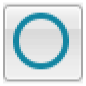
To **finish** the **model** we can create any additional required **fillets** and **chamfers** using the **solid feature operations**.

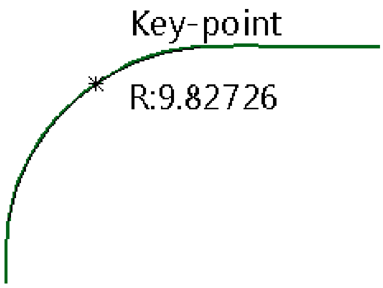
337 Ensure **Workplane 1** is **Active** from the **workplane selector**.

338 **Turn OFF Level 10 : Solid Model** and **Turn ON Level 7 : Mesh**.

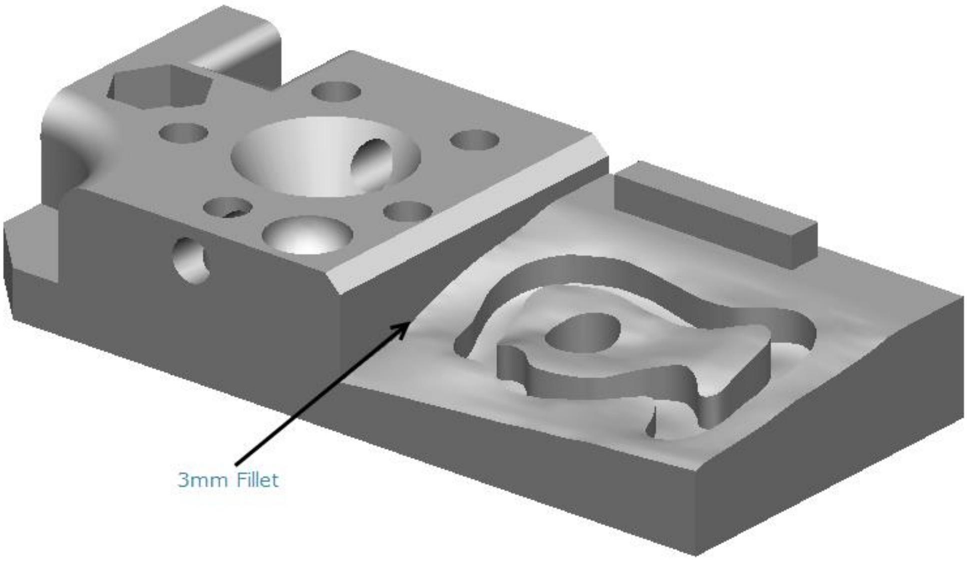
339 Select **View>Dynamic Sectioning** and within the dialog choose to take a **section Around Z** and then using the **back plane slider** choose an **angle** of **315°** and create **wireframe**.

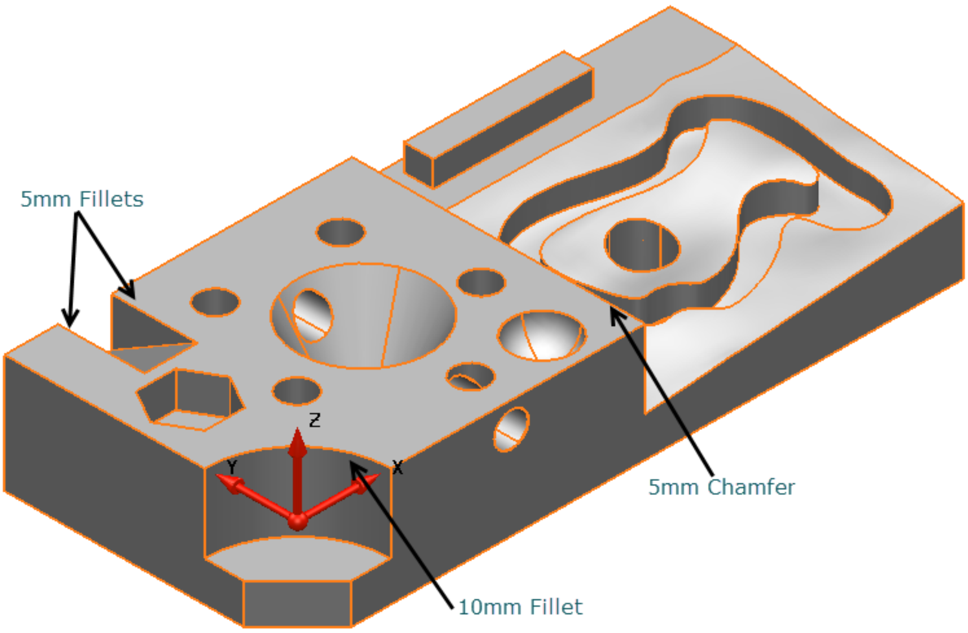
340 Select and **Blank** (**Ctrl+J**) the **mesh**.

341 Using a **3 point arc**  from the **Arc**  menu measure the **fillet radius** from the **cut** at the **lower left hand corner wireframe**. You should get a value close to **10.00mm**.



Using the same **dynamic sectioning** method down the **Y Axis** we can also measure the **5mm fillets** and **5mm chamfer** in the **location** shown **below**.





342 Create the **features** decribed above from the **Solid Feature**  menu.