Delcam

Advanced Manufacturing Solutions

Powering your productivity

PowerSHAPE

Training Course

Important Notice

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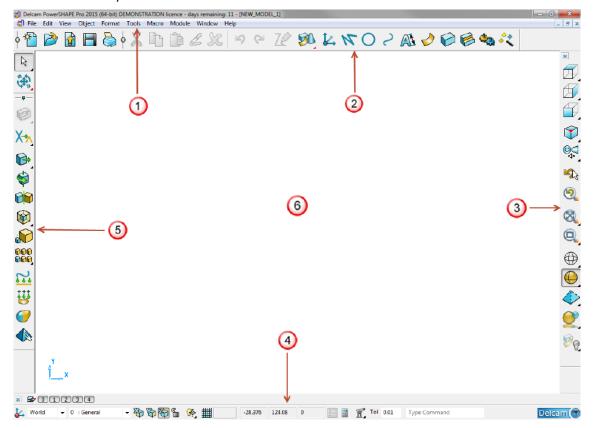## Chapters   ## Page Number

1. Introduction

PowerSHAPE Introduction
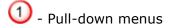
PowerSHAPE is a **powerful CAD modelling package,** which comprises core modules (**wireframes**, **surfaces**, **solids**, **drafting** and **assembly**) plus **optional specialised modules**. These include **PowerSHAPE Pro** for **Reverse Engineering** (meshes), **Toolmaker** (mold design) and **Electrode** (Electrode wizard).

1 To open **PowerSHAPE**, double click on the desktop icon.

Once loaded, the screen looks like this:

① - Pull-down menus

② - Main toolbar

③ - Viewing and shading options

④ - Status bar

⑤ - Command toolbar

⑥ - Graphics window

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

Delcam PowerSHAPE Pro 2015 (64-bit) [NEW_MODEL_1]

The model can 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 session.

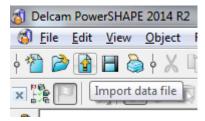At the top of the screen there are a series of pull-down menus. ①

File   Edit   View   Object   Format   Tools   Macro   Module   Window   Help

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

File	Edit	View	Object	Format	Tools	Ma
New					Ctrl+N	
Open...					Ctrl+O	
Open Drawing...						
Open Component Library...						
Close					Ctrl+F4	
Close and Zip						
Save					Ctrl+S	
Save As...						
Save Thumbnail						
Properties						
Examples...						
Print...					Ctrl+P	
Print Preview						
Page Setup...						
Print to File...						
Reset						
Delete...						
Import...						
Export...						
Recent Files					▸	
Exit					Alt+F4	

In the **File** menu, model files can be created, saved, imported, exported and printed.

Sub-menus are accessed by clicking on ▸ symbol.

If the mouse is held over a button for a short period, a box containing a description of the command (tooltip) is displayed.

To the bottom left of the graphics area is the **Levels** menu. Entities can be assigned to a level for long term, group selections.

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

The **Main** toolbar ② provides access to the Workplane, Line, Arc, Curve, Annotation, Surface, Solid, Feature, Assembly, and Wizard toolbars.

Beneath the pull-down menus are the **Main** toolbar buttons②. When selected, these buttons either open up a dialog or display the relevant functions on the

Command toolbar ⑤ to the left of the graphics window. ⑥

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

3   Select the **Line** creation button, the following toolbar is displayed to the left of the graphics window.

The top two icons are fixed and comprise **Selection** **General editing**
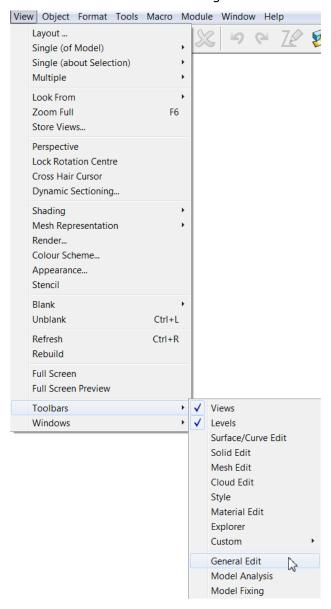
Model fixing and **Model analysis** .

Toolbars
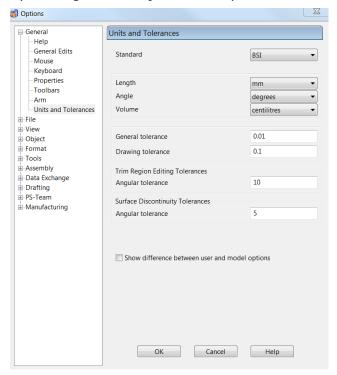
All the toolbars can be shown or hidden from the **View** pull-down menu by selecting **Toolbars** ▸ and then left clicking on the toolbar name.
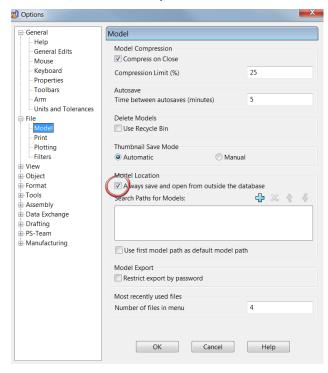
PowerSHAPE Options

The **Tools > Options** page allows the user to configure PowerSHAPE 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 sessions.

1 In **Tools > Options >File > Model** select **Always save and open from outside the database,** if not already selected.
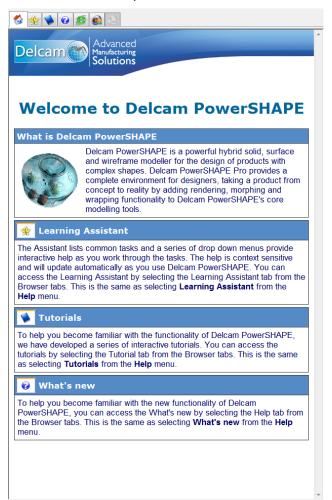
2 Select **Cancel**.

PowerSHAPE Help

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.
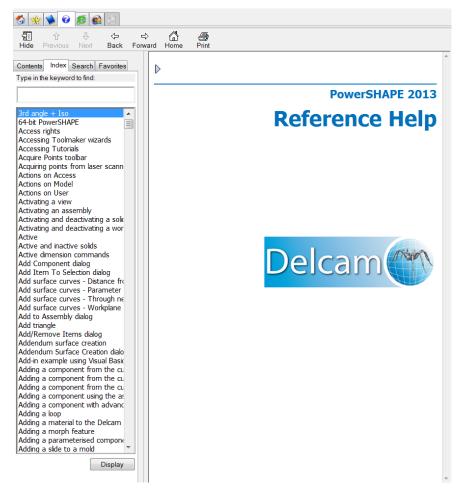
The window displayed above is the welcome page and can be accessed by

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.

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 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.
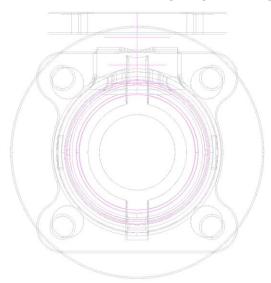
Exercise 1: Dynamic mouse controls.

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).
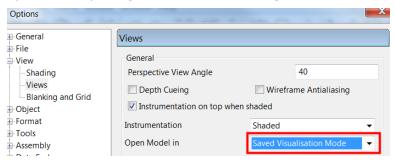
2   **Open** the model:

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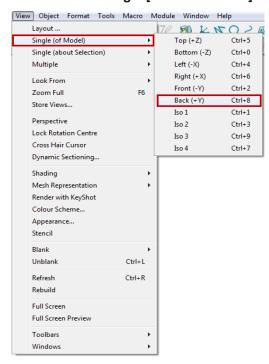
> *When scrolling the middle button (wheel) to zoom, the position of the cursor is the focal point.*

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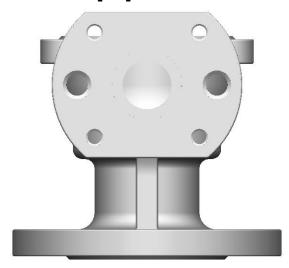
8   Select **View > Single [about selection]**.

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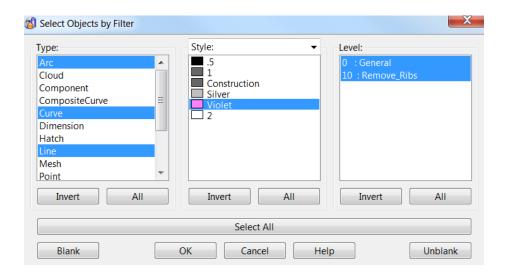
The **Select** flyout (above command toolbar) reveal the following options.

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**
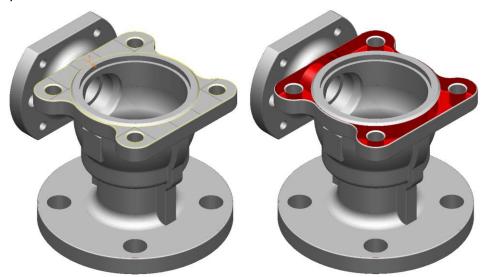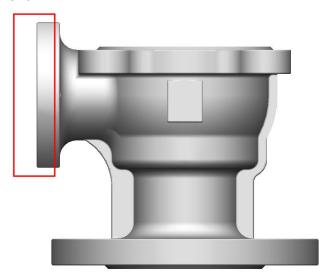
Each will be demonstrated using the current model

1 Select the upper surface of the pump housing and its outline will change to a yellow colour.
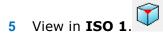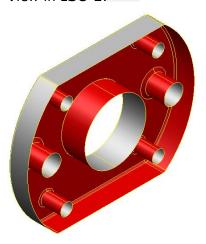
1 Select **Ctrl+J** to temporarily remove the upper surface from the graphics area as shown to the right (above).

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

3 View the model from Left (−X) , then box select the flange surfaces as shown.

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

5 View in **ISO 1**.
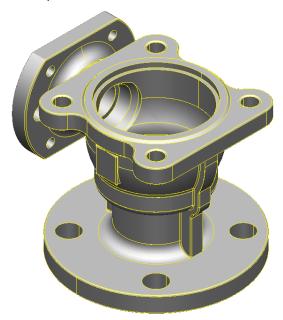
6 Select **Ctrl+Y** to blank **toggle** and view the blanked surfaces
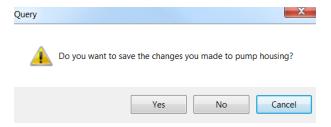
7 Select **Ctrl+Y** to toggle the view again.

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

9 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.

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

2. Wireframe Modelling
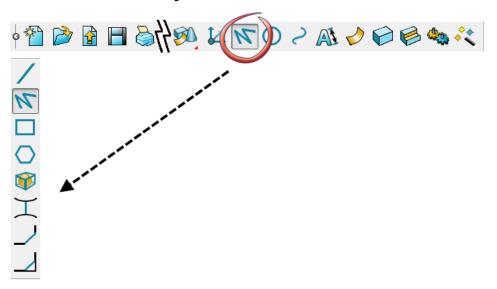
Introduction

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.
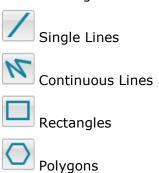
The buttons for generating **Wireframe objects** are located within the **main** toolbar.
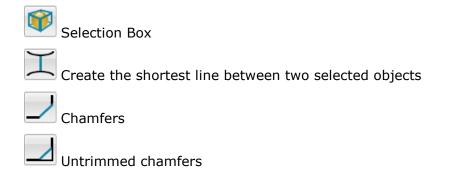
1   Select **Line** from the **Object** menu.
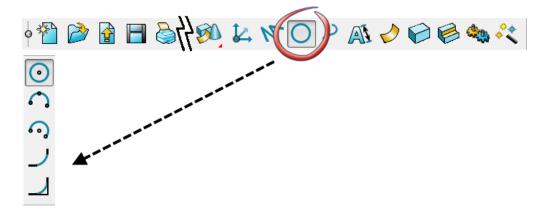
The following **Line** creation options are available.

 Single Lines

 Continuous Lines

 Rectangles

 Polygons

Selection Box

Create the shortest line between two selected objects

Chamfers

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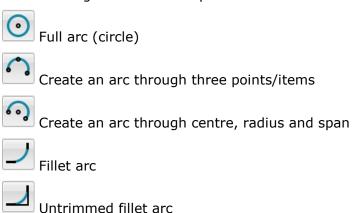
2   Select **Arc** from the **Object** menu.

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**
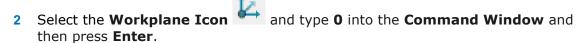
Construction lines

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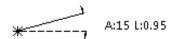
1   From the top Menu bar, select **File > New**, or click on **Open New Model** on the Main toolbar. A **New Model** is opened.

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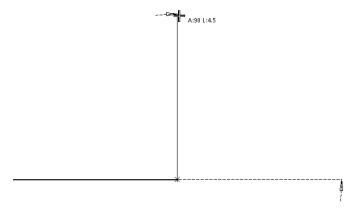
3   Select a **View in (+Z)**.

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**.

A:15 L:0.95
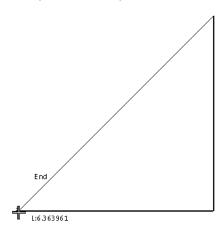
6   Drag along a **horizontal** construction line (A:0) and then click to create it.

A:0 L:4.5
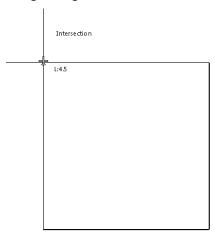
7   Drag along a **90°** construction line (A:90) and click. The line snaps to give a square intersection.

A:90 L:4.5
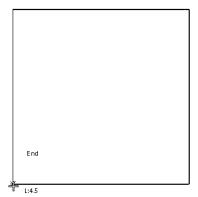
8   Drag the cursor down to the original start point and hover. The cursor displays **End** (do not click).

End

L:6.363961

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

Intersection

L:4.5
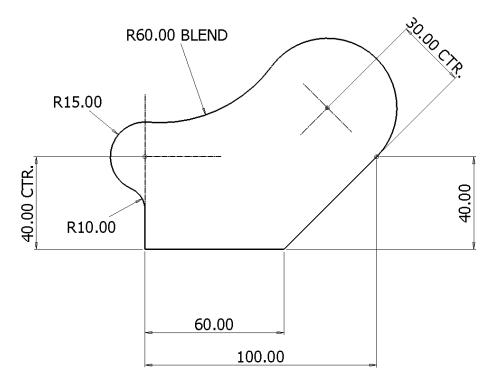
10 Click to snap at the intersection point then finally back down to the start point when **End** is displayed.

End

L:4.5

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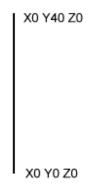
To begin, a single line will be created from **0** to **Y40** to coincide with the **R 15 arc centre** shown above.

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

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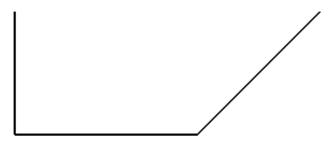
4   From the **Line** toolbar, select **create a continuous line**.

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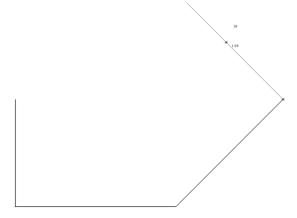
7   Input the value **40 40** in the **Command input** box, then **Enter**.

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**.

| Tol | 0.01 | r 15 |
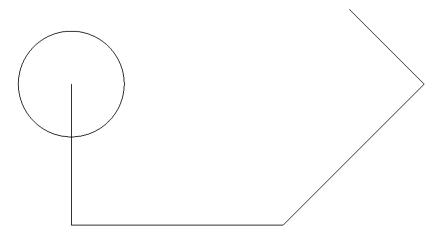
This input redefines the default radius

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.
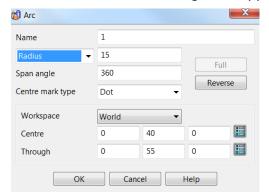
4 Press **Esc** to exit.

Wireframe objects can be edited by:
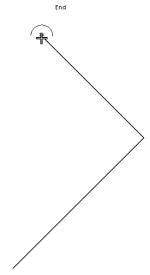
- Double left mouse clicking on the object.
- Right mouse select on the object then Modify from the local menu.

In both methods the editing form appears

Arc	
Name	1
Radius ▼	15
Span angle	360
Centre mark type	Dot ▼
Workspace	World ▼
Centre	0 40 0
Through	0 55 0

Full
Reverse

OK   Cancel   Help
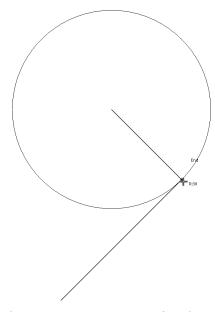
5 From the **Arc** toolbar ⭕ , select **Arc through centre, radius and span**. ⌢

---

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

End

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

End
R:30
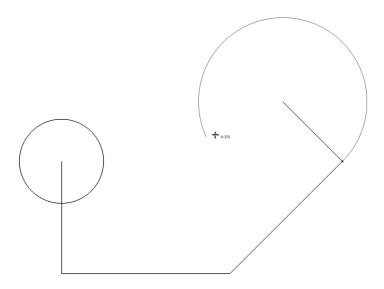
The position is now set for the start of the arc.

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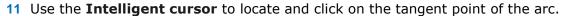
9   Left-click when the arc reaches about **250** degrees.

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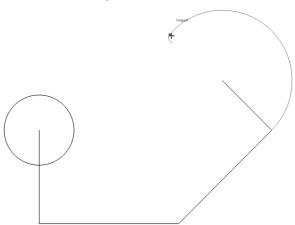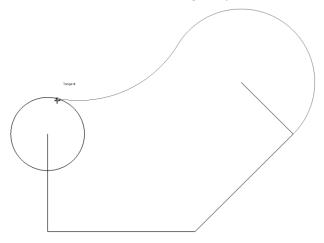
Next, a blend radius (**R 60**) will be generated between the arc and circle using **Create an arc through three points**. The first and second points will be snapped as tangencies to the existing full circle and arc with the third point being entered as the specified blend radius of **60**.

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.
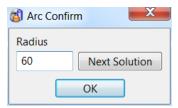
12 Locate and click on the tangent point of the circle.
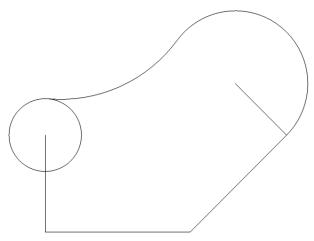
13 Drag the third radius point to the required shape and to a value as close as possible to **60,** then left-click.
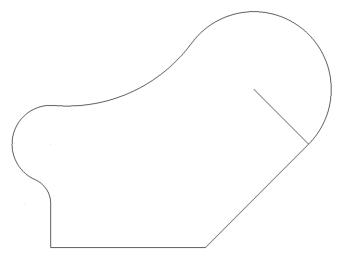
14 If necessary, in the **Arc Confirm** dialog, modify the **Radius** value to **60** and toggle through **Next Solution** until the correct shape is achieved. Select **OK**.

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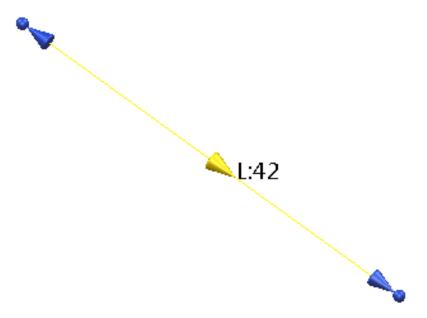
Once **Lines** or **Arcs** have been created there are two main ways which they can be then edited. This can be done **Graphically** from within the **graphics window** or using the **Line/Arc Editor** dialog.

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.
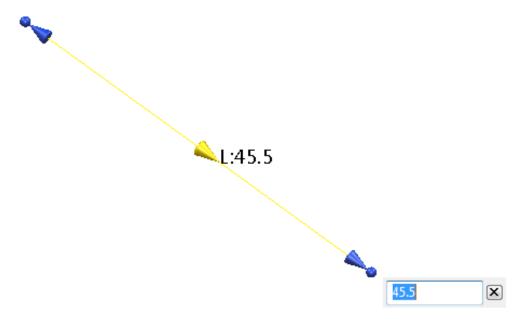
Within this mode the **Line** is highlighted yellow with an arrow indicating the **Direction** of the line and text showing its current **Length**.
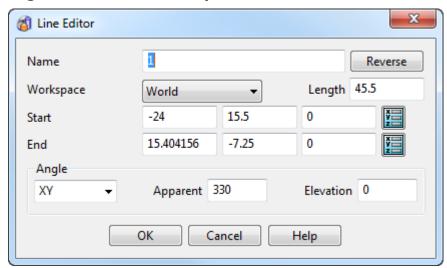
The blue **Arrow** and **Sphere** at either end of the **line** allows you to **edit** the characteristics quickly and easily while the **intelligent cursor** shows the **original position** and allows you to **snap** to other objects.

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 pa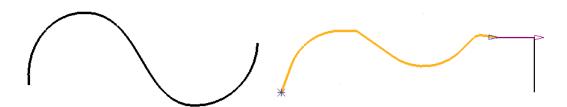rticularly useful for naming and editing the **relative Azimuth (Apparent)** and **Elevation angles** of the line **accurately**.

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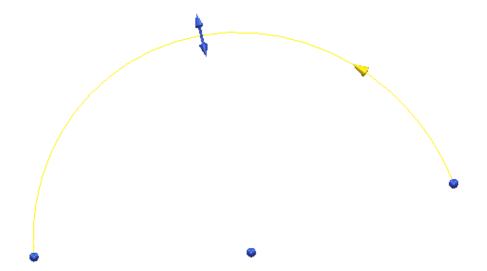
Arc Edit

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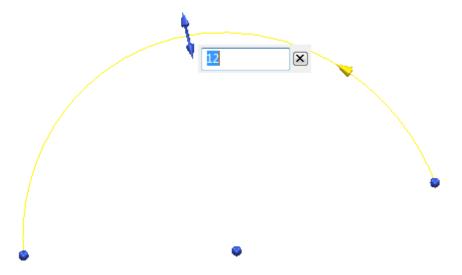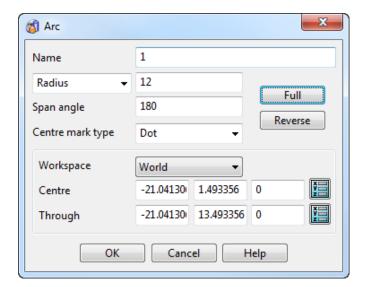
Clicking and **dragging** the **spheres** will allow either end of the **arc** to move in a **fixed circular motion** dictated by the **radius** in order to **graphically edit** the **span angle** of the **arc**.

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**.
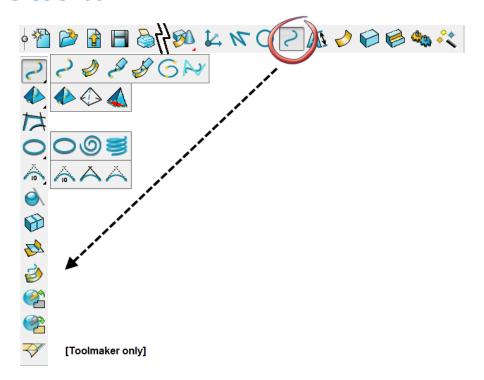
When using the **arrows** or **spheres edit** the **arcs characteristics**, when the mouse button is **released**, **PowerSHAPE** will give the opportunity to **fine tune** these values such as **span angle** and **radius**.
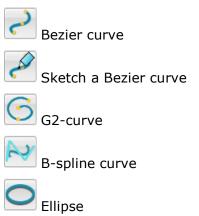
To fully fine tune the **Arc**, the most accurate option will be to enter the **Arc Editor** dialog. Again, as with all **Objects** in **PowerSHAPE** a double left mouse click on the **Object**, in this case an **Arc** enters the **Editor** dialog.
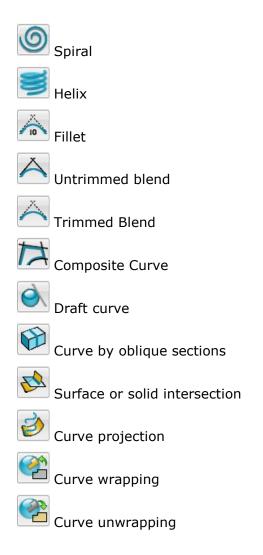
Curve toolbar

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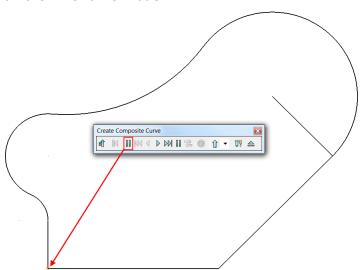
A composite curve can only be created along existing model entities.

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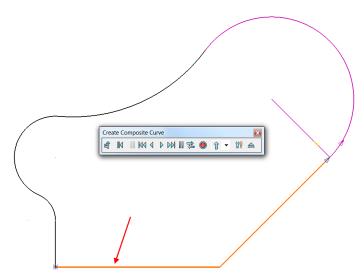
2   Select the **Start Point** option in the toolbar and left-click the lower-left corner on the wireframe model.

3   Left-click on the bottom horizontal line.
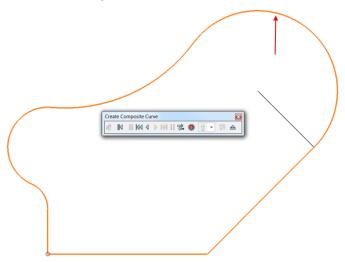
The composite curve is generated along the route until it hits a branch (intersection) point where optional directions will be arrowed for you to manually select the required route.

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

4 Left-click along the **R60** arc.
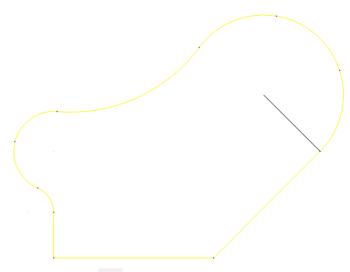
The composite curve has continued around the wireframe back to the defined start point.

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).
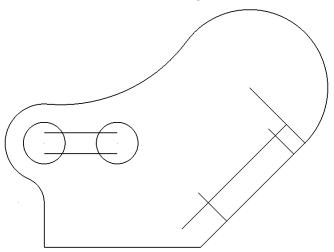
Exercise 1: Composite Curves

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**
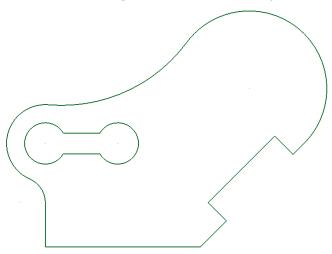
3 Use Composite curve to generate the result shown below.

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**.
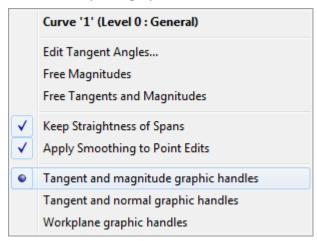
Single Curve Point Edits

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**.
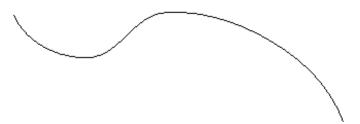
Tangent and Magnitude graphic handles

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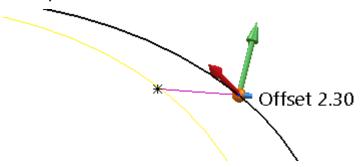

2 Select a **point** on the curve and right-click and ensure that **Tangent and Magnitude graphic handles** is selected in the menu.

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.

	Curve '1' (Level 0 : General)
	Edit Tangent Angles...
	Free Magnitudes
	Free Tangents and Magnitudes
✓	Keep Straightness of Spans
✓	Apply Smoothing to Point Edits
	Tangent and magnitude graphic handles
●	Tangent and normal graphic handles
	Workplane graphic handles

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.

Offset 6.10

3.4

Similarly, th**e 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.

Offset 2.00

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.

<3.20,2.90,0.00>

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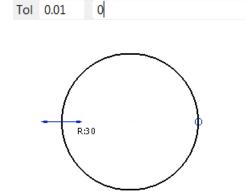
When this button is selected, the centre position of the circle is defined first. This can be entered as a value or by snapping using the **Intelligent Cursor**.

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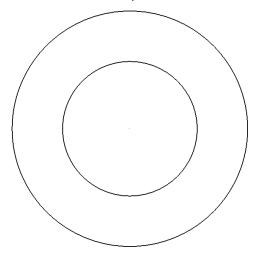
7 In the **command input** box enter **r 20** then **Enter**.

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.
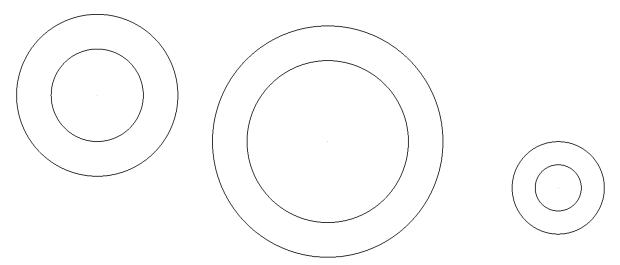
Creating additional circles

Use the command input box to enter the circle centre if it cannot be snapped.
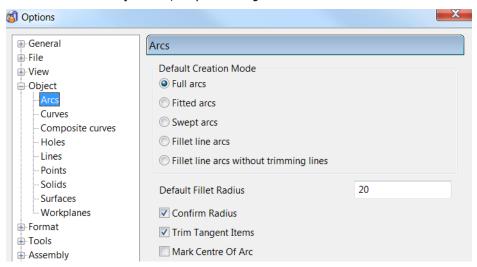
10 Create a circle at **centre 100 −20** with a **radius** of **50**.

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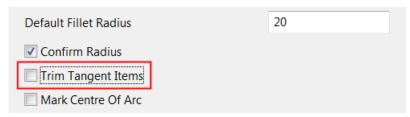
The default option when creating tangent arcs is to trim back the associated geometry. In this case, it is not required, so the option (**Trim Tangent Items**) needs to be deselected.

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

16 Deselect **Trim Tangent items**.
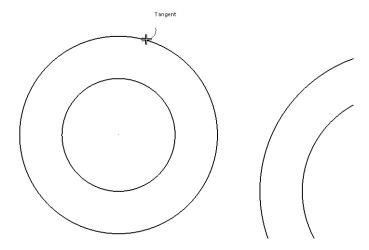
17 Select **OK**.

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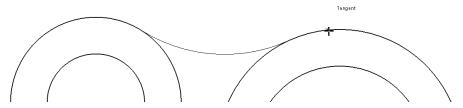
For the tangent arcs, the fitted arc option is the most suitable as it provides full dynamic control through all possible combinations.

1 From the **Arcs** toolbar, select **Create an arc through three points/items.**
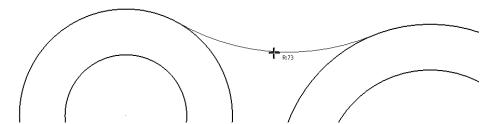
2 Move the cursor over the outer left circle until the word **Tangent** appears and left-click.

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.
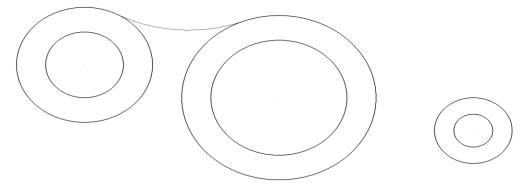
The **Arc Confirm** dialog is displayed. This can be used to enter an exact radius.
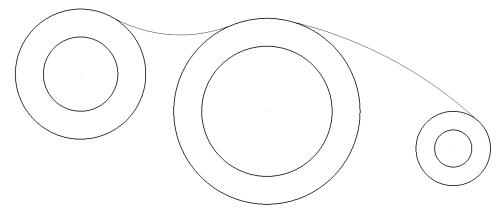
5   Enter a **Radius** of **75** and select **OK**.

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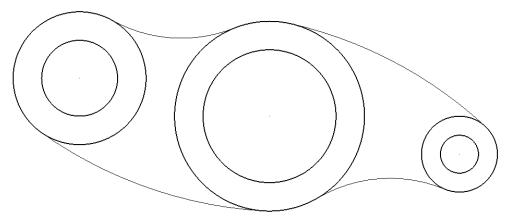
6   Create an **arc through three points/items** of radius 225 between the middle and outer circles shown below.

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**.
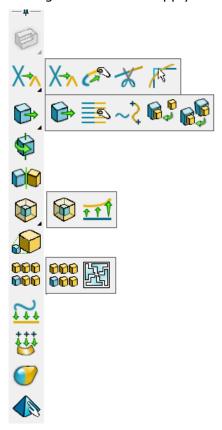
The model is complete

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**.
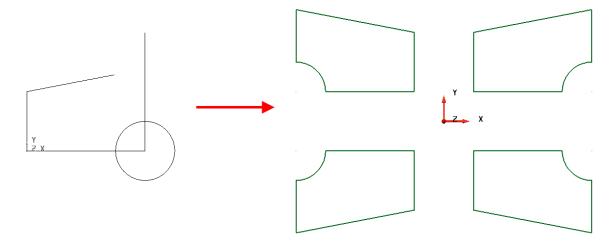
General Edits example

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.
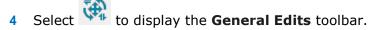
1   Start a **new model**.

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

.....\PowerSHAPE_Data\GeneralEdit_Wireframe.dgk
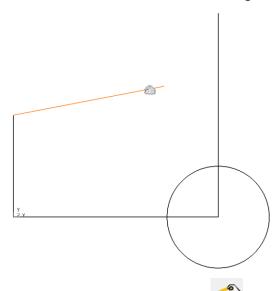
The imported wireframe (shown above, left) will be modified using the **General Edits** options, to produce the 4 finished wireframes (shown above, right).
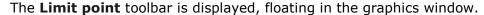
3   Click in the graphics area to deselect the wireframe.

4   Select [icon] to display the **General Edits** toolbar.

5   Select the line that needs extending as shown
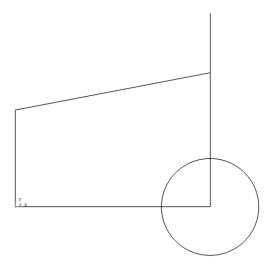
6   Select the **Point limit** option. [icon]

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

[Limit point toolbar image showing: Edge/End Point 1, Distance 0]
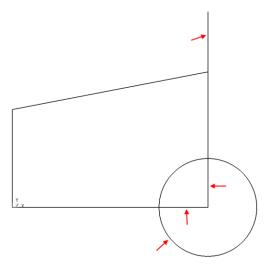
The cursor also changes to a magnet symbol. [icon]

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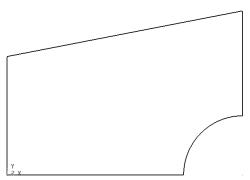
9   Select the **Interactively limit wireframes** option [icon] from the **General Edits** toolbar. The **Limit point** toolbar will be closed automatically.

10 Left-click the mid-span of any sections of the wireframe network that are to be trimmed away (as illustrated below).
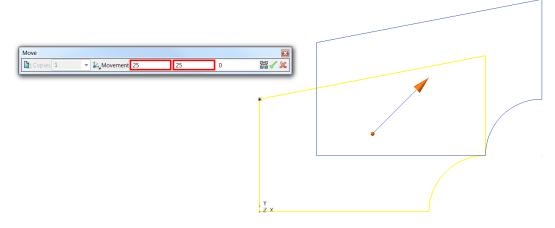
11 Create a composite curve from the network of lines and arc wireframes.

> *A quick way to create a composite curve is to hold down the **Alt** key and left-click anywhere along the wireframe network.*
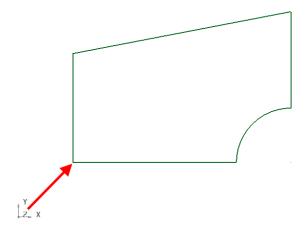
12 With the composite curve selected, select **Move** from the **General Edits** toolbar.

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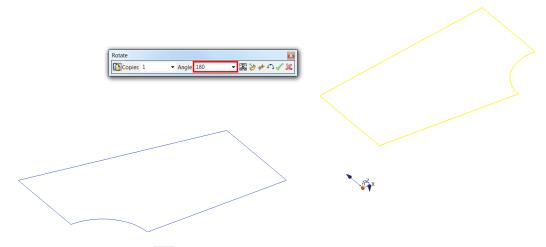
The Rotate toolbar is displayed. In addition, the blue arrow on the
datum shows the current rotational axis (Z). In this example the wireframe will
require rotation about the Y axis, so this must be changed.

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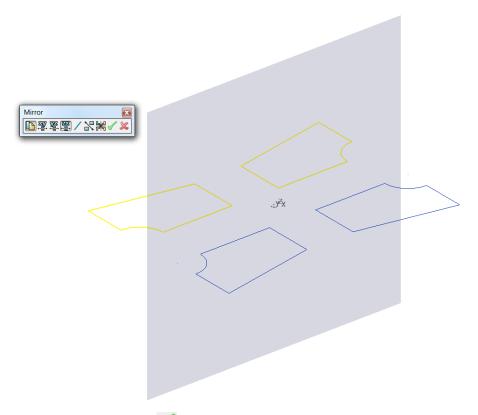
A copy of the composite curve is rotated by **180** degrees about the active **Y** axis.

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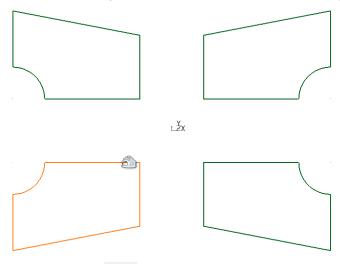
Further General Edits options
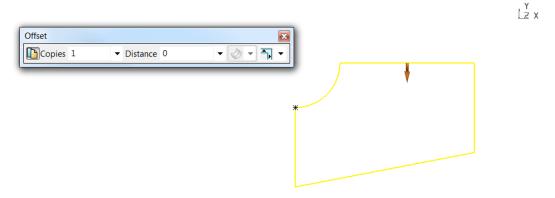
In addition to the General Edits options used, other options are used later in the course with other PowerSHAPE entities, such as surfaces and solids.

Offset
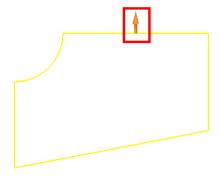
1   Select the composite curve located in the **–X –Y** quadrant.
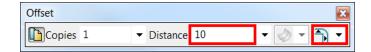
2   Select **Offset** from the **General edits** toolbar.
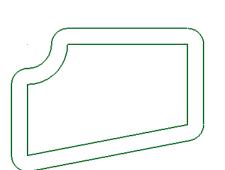
3   Select the small red arrow on the curve to switch the positive direction outwards.

4   Select **Round discontinuities** and enter a **Distance** of **10.**
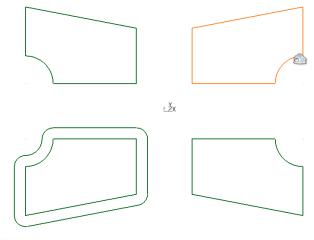
5 Close the **offset** form by selecting the small cross in the corner.

A 10mm offset with round corners has been applied.
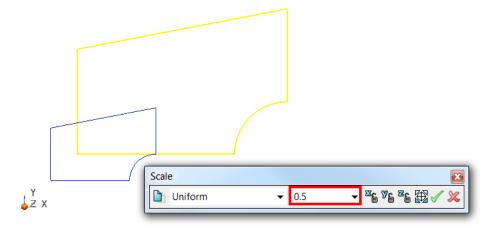
Scale

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.
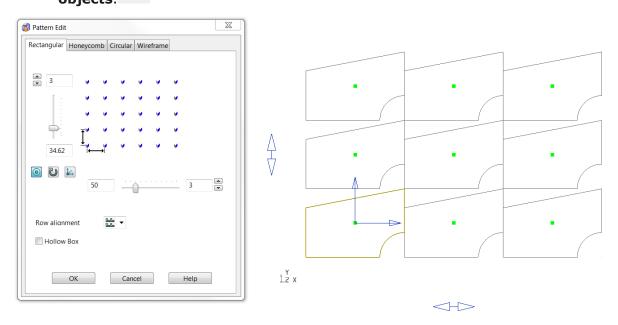
4   Select the green tick ✔ to accept the preview.

> *Scaling is from active workplane, therefore vectors to the active workplane origin also scale. This can be repositioned as required within the scale dialog*
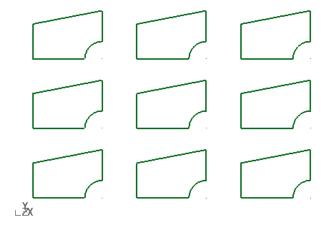
Create Pattern of objects

1   With the scaled down composite curve selected, select **Create Pattern of objects**.
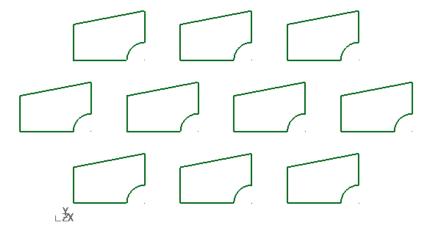
The current settings are immediately previewed.

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

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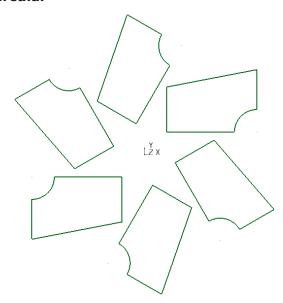
Rectangular is one of 4 main functions in the Pattern Edits dialog. The other three include Honeycomb, Circular, and along Wireframe.
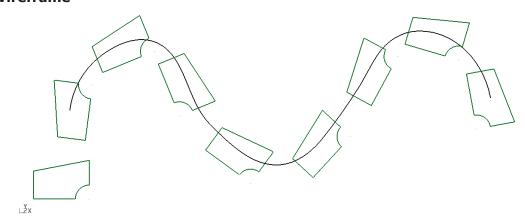
Honeycomb

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:
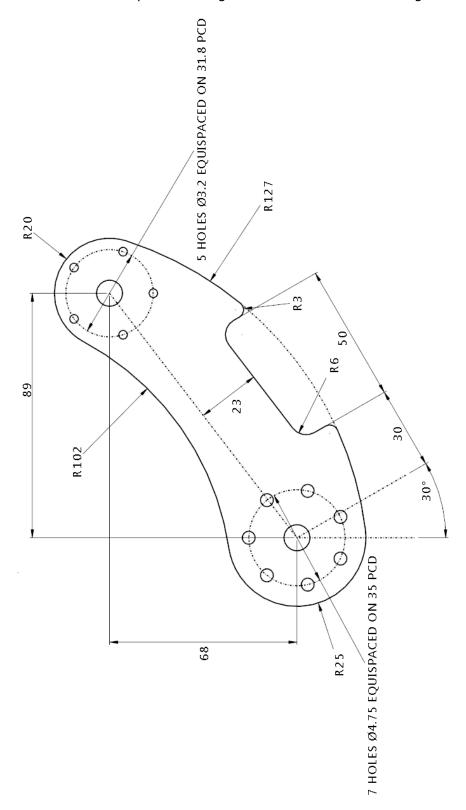
.....\PowerSHAPE_Models\General Edits

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**
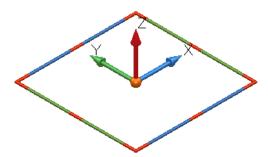
3. Workplanes

Workplanes

Workplanes are **user-defined datums**. They can be **positioned** and **aligned** as required to simplify model creation.
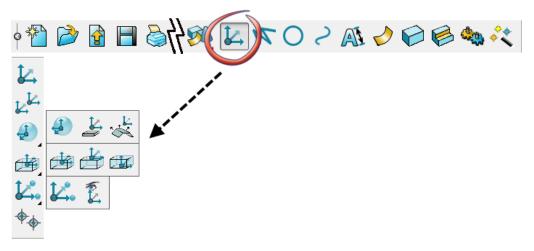
An active **workplane** represents the current **workspace** and its **origin**.

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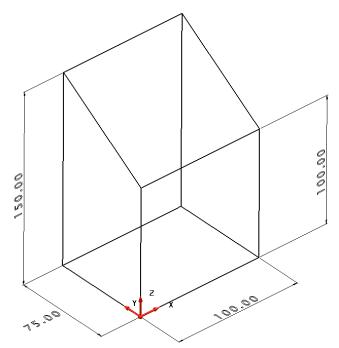
A simple wireframe box with angled top will be initially created to demonstrate the basic applications of workplanes.
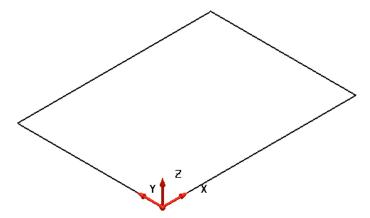
Box Example

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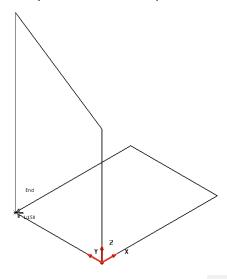
9   Enter **0 0 100** in the **Command input** box and press **Enter**.

10 Enter **abs 0 75 150** in the **Command input** box and press **Enter**.
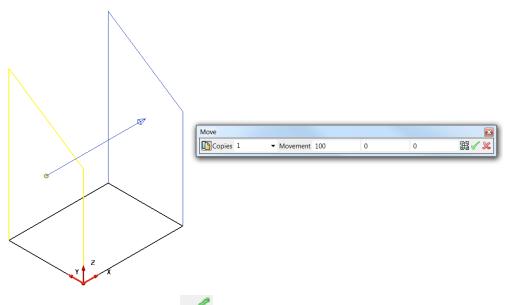
Adding the prefix abs changes the input from the default relative to absolute coordinates.

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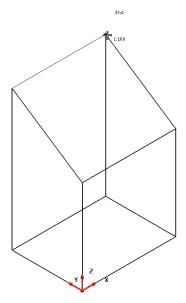
12 Press the **Esc** key or click to end creation mode.

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**:
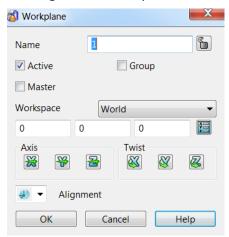
▪ Snapping onto the screen or geometry using the left mouse button.
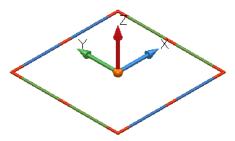
- Entering the coordinates using the command input box
- Generated from the geometry and option type selected.

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.
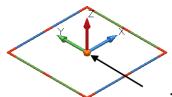
Editing using Graphical handles
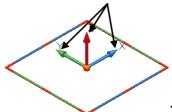
Single selecting a Workplane will display its graphical handles.

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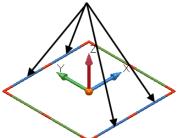
To change the direction of the X, Y or Z axes, select the end arrow head and release over an item. The selected axis will point in that direction.
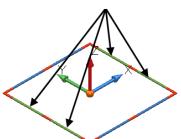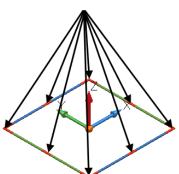
To move the workplane along the X, Y or Z axes, select the handle of the axis itself and drag.

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

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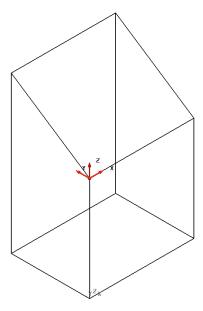
To twist around the Z axis, select any red handle

18 Select **Single Workplane** , and snap a new workplane to the top end of the line running up **Z** from the original **Workplane**.
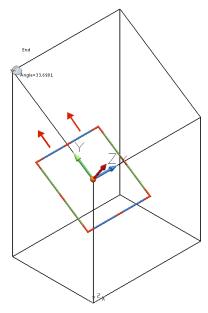
The **new workplane** is now **active** displayed in **red**, with the **previous workplane** now **deactivated** (shown in **grey**).

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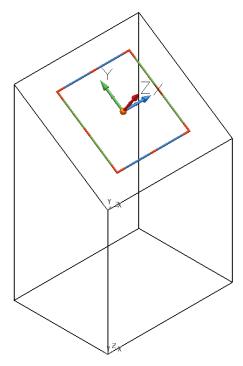
With the **workplane** now in the **correct orientation**, it will now be centred within the **top wireframe boundary**.

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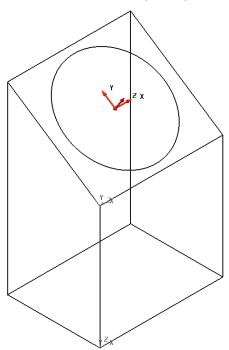
A **new workplane** is **created**, **central** to the **four selected lines**. This workplane is both **active** and **selected**.

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.

Use the XY 'face' of the workplane

-277.35
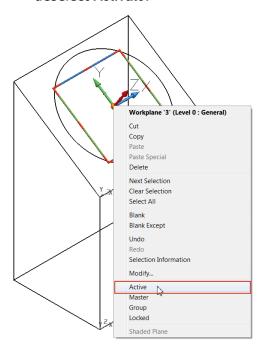
Master Workplane

1   **Right-click** on the **active workplane** to open the popup menu and then deselect **Activate**.
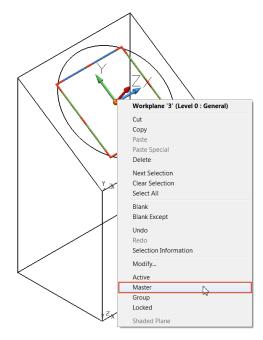
There is **no active workplane**; all **coordinates** are now defaulted to the **World** coordinate system.

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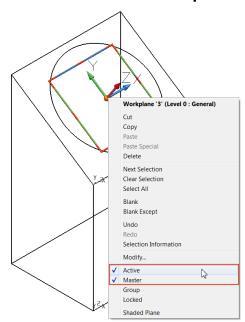
The **Workplane** is not **currently active** but has now been assigned as the **Master Workplane**.

3   **Re-activate** this **Workplane**.
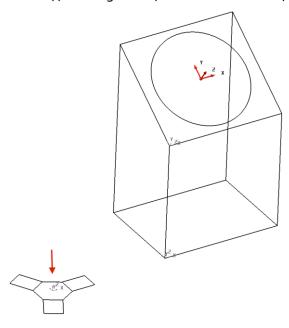
A previously created **2D model** will now be **copied** and **pasted** onto the **active workplane**.

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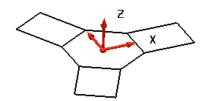
Simply selecting the workplane in this menu makes it **active**. In addition, **Workplanes** can be renamed from the default number provided.

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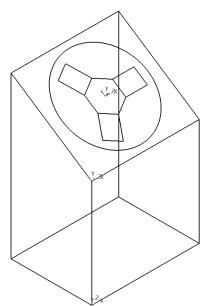
The **selected wireframe** is **copied** to the **clipboard**.

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.
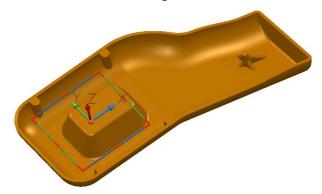
The **imported model** of a **plastic molding** is **not** in a **suitable orientation** from which to create a **mold tool**. A **suitably positioned workplane** will be **created** to provide an **appropriate tooling datum**.
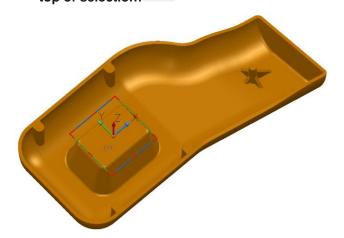
The most obvious choice for a **tooling alignmen**t is **normal** to the **base** of the pocket.

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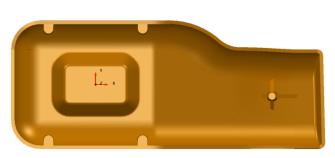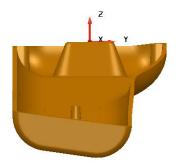
The **workplane** is **positioned central** to the **selected surface**.

7 Select and **delete** the original workplane used to create the alignment with the base of the pocket.
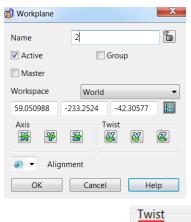
8 View the model in all directions to check that a suitable tooling alignment has been achieved.
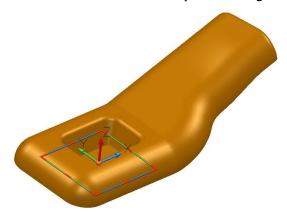
It is possible that the model may need to be rotated 180 degrees about the **X** axis at some stage. This can be performed using the **Workplane** editing dialog.

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.
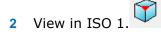
Exercise 3: Workplane creation

1 **Open** the surface model:
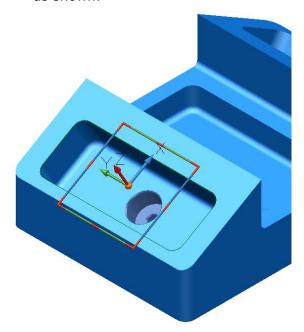
.....\PowerSHAPE_Data**Exercise 3 model.psmodel**

2 View in ISO 1.
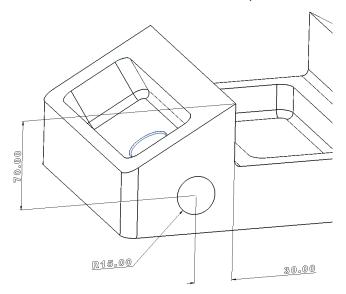
3 Create a workplane at the **centre of the pocket** aligned to the top sloped face as shown.
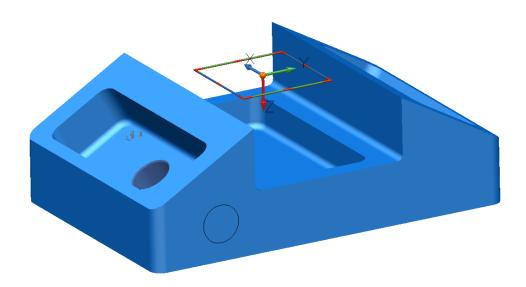
4 **Rename** the Workplane **Angled left pocket**.

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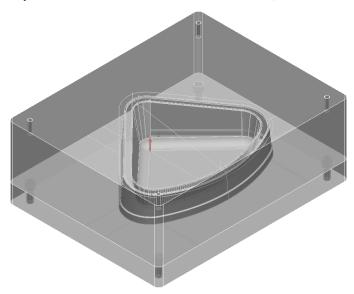
Levels enable you **to group model entities** and help **simplify the display** and selection of items. Levels can be **distinct groups of component surfaces**, or **different types of entity**, and they can be **renamed** to identify the items they contain.

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 **session**. A typical example use is to assign the **inner wall surfaces** of a **component** to a **one level**, and the **outer wall surfaces** to **another**.

PowerSHAPE supports up to **1000 levels**, but levels **998** and **999** are **reserved** for internal use.

Example
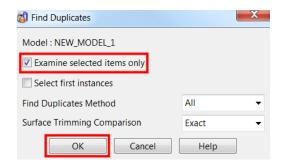
1   **Create a New Model** and **Workplane** at **0**.

2   **Import** the model......PowerSHAPE_data/**Corner Bowl.dgk**
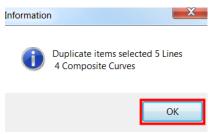
The **model** contains a **mixture** of **wireframe** and **solids**. It **may** also contain **duplicate entities** that could be **wireframe**, **solids**, or a mixture of both. It is advisable to check and move these items to a **level**, or to **delete** them if not required.

3   Select everything within the **graphics area**.

4   Select **Find duplicates**   from **Model Fixing**   toolbar.

5 Select the option **Examine selected items only** then **OK**.
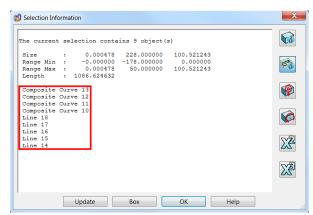
PowerSHAPE examines the model and selects duplicate entities An information box is displayed, showing the number of duplicate entities. Click OK to close the dialog.

The duplicated items are also selected in the model.

6 Select **Selection information** from the **Selection** toolbar.

A list of the current selected items is also summarised here.

These duplicate entities will be **moved to another level**.

7 Select **OK**.

8 From the lower-left side of the window, open the **Levels** form.
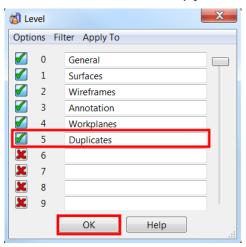
Each level has a **number**. **Visible levels** are indicated by ✅. **Hidden levels** are indicated by ❌. You can turn *ON* a level by clicking ❌ and *OFF* by clicking ✅.

A **new level** is created simply typing a **name** into an **empty box adjacent** to the **level number**.

Level names can be edited by just modifying the existing name.

9   In level **5**, enter the name **Duplicates**, and select the red **X** next to the number **5**.

10 Select **OK**.
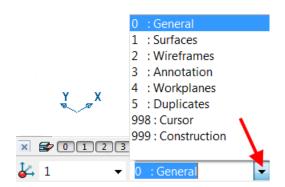
The duplicate items previously found are still highlighted. To transfer data to a level, the middle mouse button is used.

11 Click the **middle mouse button** over the level **5** icon itself, to move the selected items to that level.

12 Left-click on the **Level 5** button to switch off visibility.

By default, all new entities created or imported are placed on the default

0:General layer. The **'active'** layer can changed by simply by selecting the required layer (if created) from the drop down list shown.
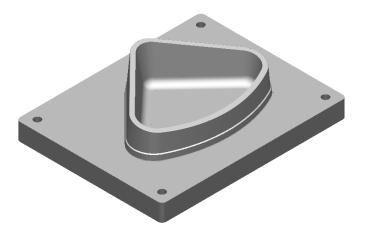
13 Open the **Levels** form and create two further levels. Name level **6** as *Core* and name level **7** as *Cavity*.

14 Select **OK** on the form.

This model contains solids. These are a different type of component, which can be used to make complex shapes. To select a solid, clicking on any part of it.

15 Select the bottom solid and press **Ctrl+K** (**Blank Except**).
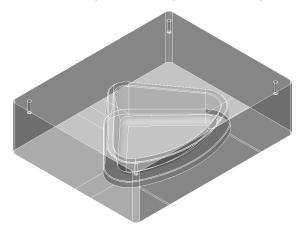
Adding the cavity and core to level 6 and 7
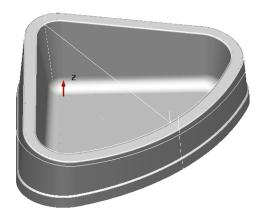
1   With only the **bottom solid selected**, click the **middle mouse button** over level **6**.

The **solid** has **disappeared** as the **level** was **switched off**.

2   Press **Ctrl+L** (**Unblank**).

3   Select the top solid and press **Ctrl+K** (**Blank Except**). Only the top solid is shown.

4   Transfer it to **level** 7.

5   Select **Ctrl+L** to **Unblank**.

The component solid and construction wireframe remain on level **0** (**General**).

Grouping levels

PowerSHAPE has the ability to group levels. 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 **8**, enter the name *Components : Wireframe* and select the red **X** button next to the level number.

3 In level **9**, enter the name *Components : Solid* and select the red **X** button next to the level number.

✓	8	Components : Wireframe
✓	9	Components : Solid
	OK	Help

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

4 Select **OK**.

The two levels now belong to the group called *Components*.

× 🖊 Com   ⓪①②③④⑤ 6 7 **8 9**

On the **Levels** toolbar, the buttons of levels **8** and **9** have changed to blue. In addition, a new Levels Group button is displayed, also in blue Com . You can use this Levels Group button to turn both levels on and off simultaneously

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

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

6 With all the wireframes highlighted, middle mouse button over level **8.**

7 Select the component solid and place it onto level **9**.

8 Select Com to switch off both the associated components layers.

Using Copy and Paste to make copies of entities

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

1 Turn on level **6** (**Core**) and select the **solid**.

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

Two identical solids are on level 6

3 Select a solid and move it to level **0:General**.

4 Switch off level **6**. Toggle level **0** on and off to see the new copy.

5 **File >Save as**
*…\PowerSHAPE-Models**Corner bowl levels.psmodel***
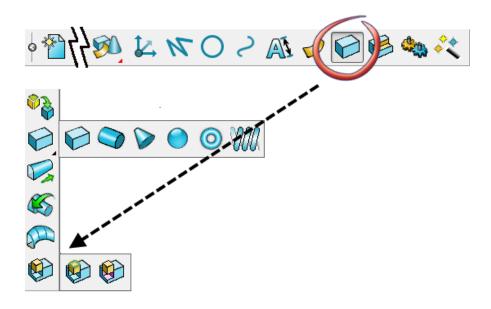
5. Solid Modelling

Introduction

Solid modelling applies a different approach to the creation of a **CAD** model compared to surface modelling studided 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.
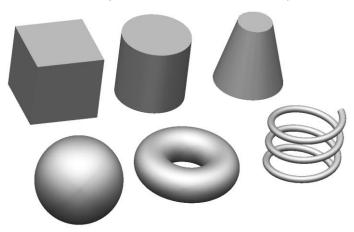
When **Solid options** is selected, it displays the following buttons in the toolbar to the left of the graphics window.

 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
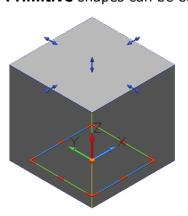
Solid Primitives

In addition to the **primitive solids** which are generated from wireframe items (e.g. Extrude, Rotate), the toolbar contains six pre-defined shapes.
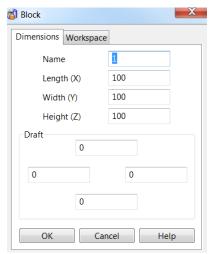
Primitive shapes can be edited dynamically with a single mouse click (select)
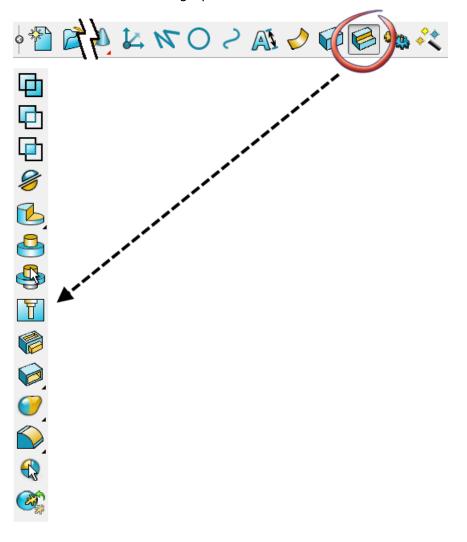
The **workplane 'type' handles** can be used to **move**, **change orientation** and **twist** about its **axis**. The **blue handles** can be **dragged** to **edit the dimensions** of the **primitive solid**.

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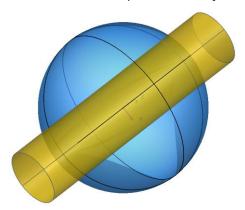
Boolean Operations
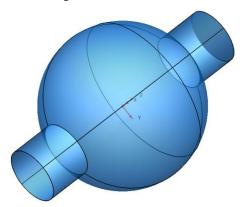
Boolean operations define the relationship between entities. As a **solid** is built up, **additional solids** and **surfaces** can be **absorbed** into it. This behaviour is illustrated below using the **three Boolean operations** available from the **solid features** toolbar.

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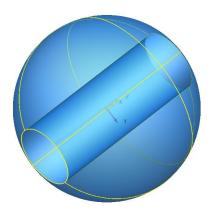
Selecting **Add** will **combine** the **selected solids**.
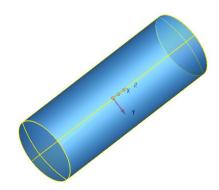
The **two separate solids** have merged into **one solid object**. The cylinder is joined onto, and becomes part of the solid sphere.

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.
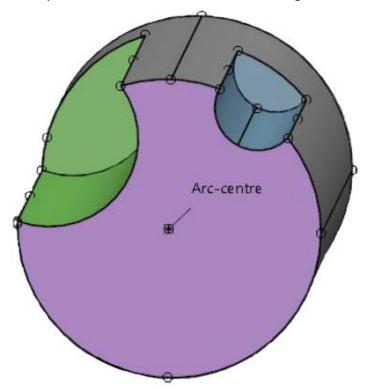
Snapping to Parasolids

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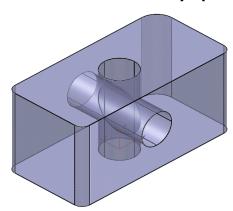
- **Corners**
- **Centres of Planar Faces**
- **Centres of Arc Edges**
- **Midpoints of Straight Line Edges**
- **Tangent to Arc Edges**

Examples of these can be seen on the image on the next page.
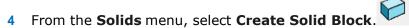
Solid Modelling Example 1

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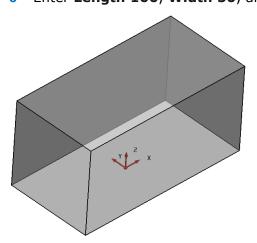
2 Create a **workplane** at **0** and set the **Principal Axis** to **Z**.

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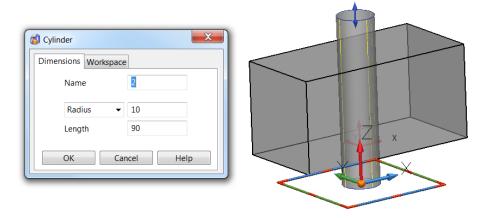
9 Select **OK** to apply the **changes** and **close** the **dialog**.

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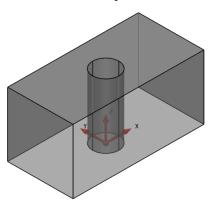
12 Press the **Esc** key to **break out** of the **command**.

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**.*
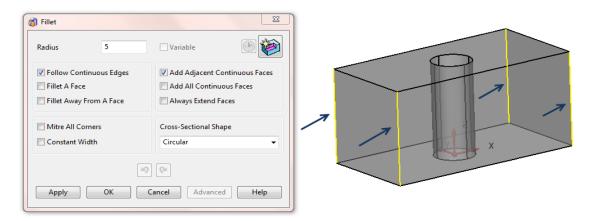
17 From the **Solid Features** toolbar, select **Create solid fillet**.
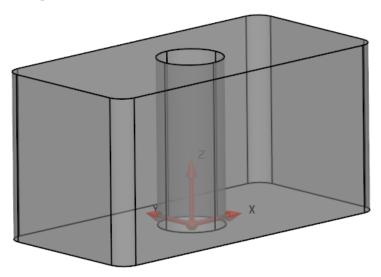
18 In the Fillet dialog, enter a **Radius** of **5mm**. Hold down the **Ctrl** key and click all the **vertical edges** of the **solid block**.

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
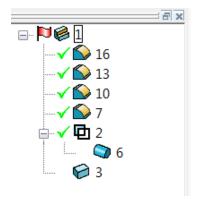
graphics area. This can be **viewed** by selecting **Show History Tree** window from the **Solid Edits Toolbar**, which automatically launches when any **solid** is **selected**.

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
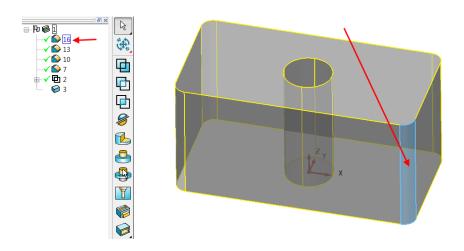
Red Flag indicates an **active solid** and **grey flag** a **inactivate** solid. Flags can be toggled from within the **feature tree** or from the **Solid Edits toolbar**.
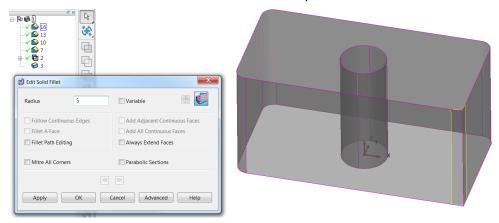
If **sub-items** are **not shown**, click to display them. To hide the sub-items, click.

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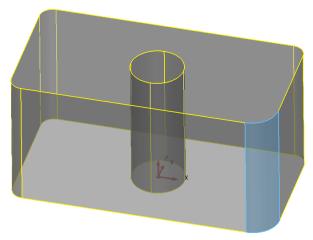
2   Double-click on the solid fillet at the top of the tree.

*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**.*
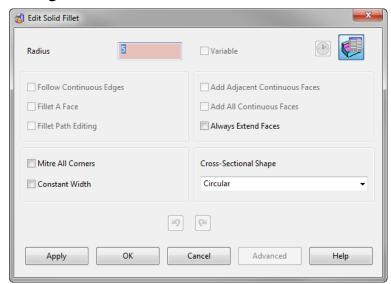
Multiple Feature Edits

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**.

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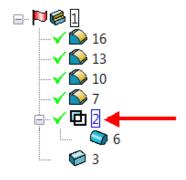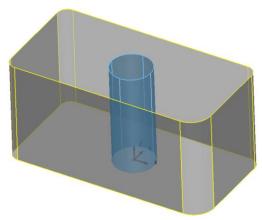
*This have **converted** both the **5mm** and **10mm fillets** to **8mm fillets** in one operation.*

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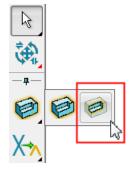
1   Select only the **Boolean Subtraction** operation in the **History Tree** as indicated.

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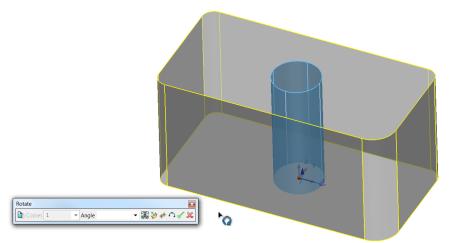
This ensures the next general edit will now apply to the selected sub-items only. In this case the **cylinder** and the **Boolean Remove** which created the hole through the block.

4   Set **X** as the principle axis.
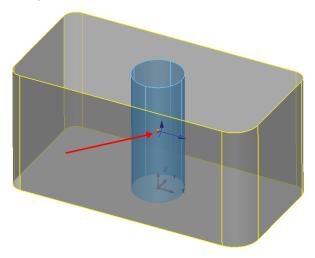
5 Select **Rotate** from the **General Edit** toolbar.

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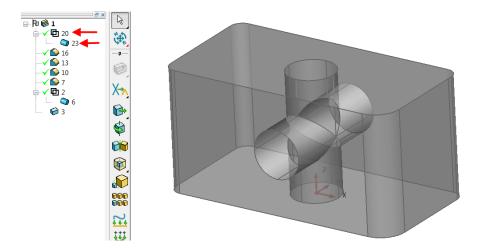
6 On the **rotate dialog** select **reposition rotation axis**.

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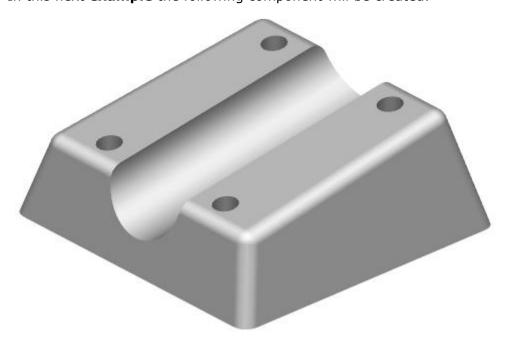
The original **sub-solid cylinder** is **copied** and **rotated** within the **solid**. The new **sub-solid cylinder** appears at the **top** of the **History Tree**.

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

Solid modelling example 2

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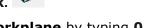
The **Mounting Block** base dimensions are **Length (X) = 100mm, Width (Y) = 75mm** and **Height (Z) = 50mm** centrally positioned relative to the **workplane** at The **Draft Angle** on all **4** walls is **5°**.

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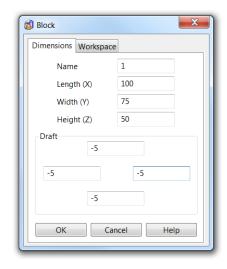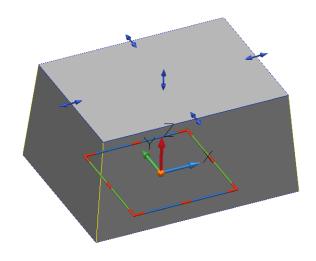
2   Set the **principal plane** to **Z** and create a **workplane** at **0**.

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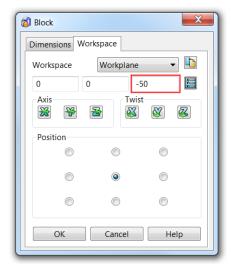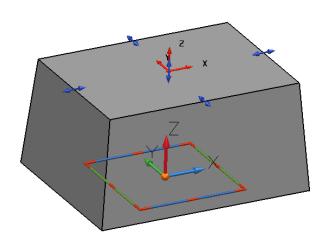
> *A solid **primitive** block allows the **draft angles** to be applied immediately.*

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**.*
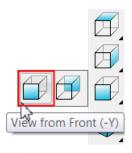
Applying fillets to the block

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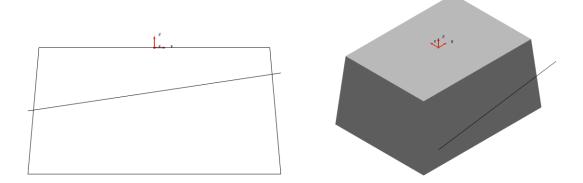
> *By default the **Principal Axis automatically** switches to the **selected view**.*

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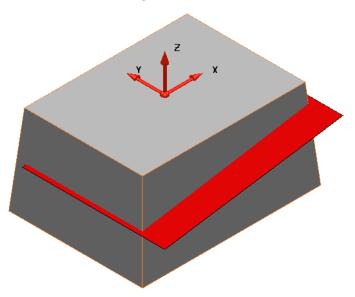
7 From **Surface** toolbar, select **Create a Surface** Extrusion.

> *This **training manual** will cover **surface modelling** in the next chapter.*
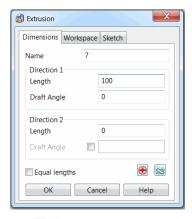
With the **line pre-selected**, an **extrusion surface** is created **immediately**.
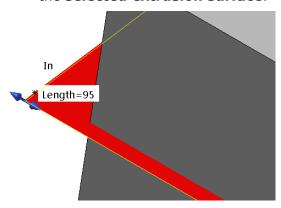
> *The **upper side** of the **surface** is **shaded red** (**inside**) while the **underside** is the **current default colour** for **surfaces** (**outside**).*

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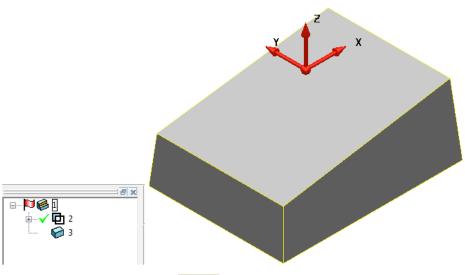
> *The **Split Solid** button can also be used to same affect.*

The **area of the solid** on the **red side** of the **surface** is **removed**, with the surface removal registered in the **Feature Tree**.
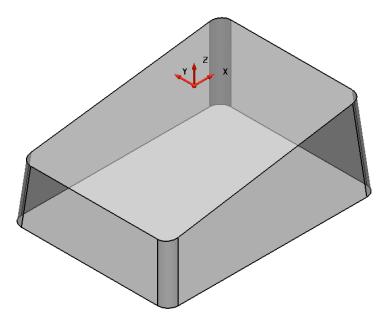
12 Apply a **5mm solid fillet** on each **draft edge** of the **solid block**.
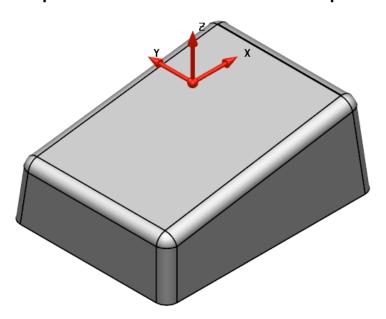
13 **Repeat** the **fillet** of **Radius 5mm** on the **sloped rectangular face**.
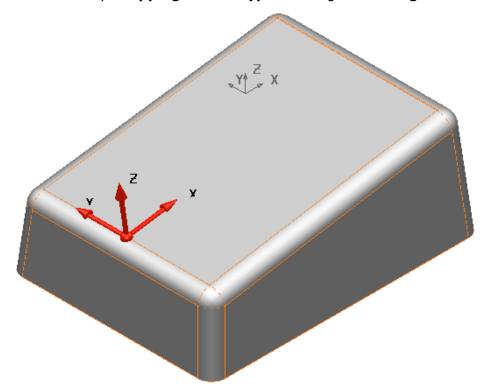
*By creating the **fillets** in **two separate operations**, the **sidewall** and **top surface fillets** are **independent** which makes it easier to make changes to the individual fillet groups at a later stage by control the grouping in the **history tree**.*

Creating a cylinder

A **new solid cylinder** will be **created** then **removed** from the **existing solid**.
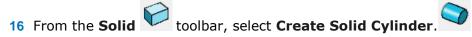
Before creating the cylinder a **new workplane** is **required aligned** to the **sloping face** and **positioned** at a **known keypoint**.

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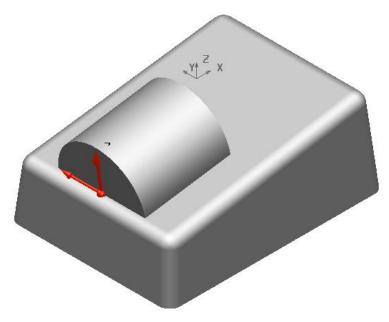
15 From the **bottom-left** of the **graphics area**, select X as the **Principal Axis**.

16 From the **Solid** toolbar, select **Create Solid Cylinder**.
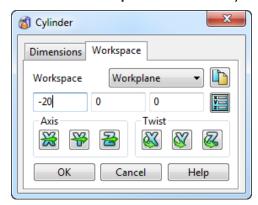
17 **Left-click** on the new **workplane** to locate the new **solid cylinder** on to it, or type **0** into the type **command box** and press the **Enter** key.
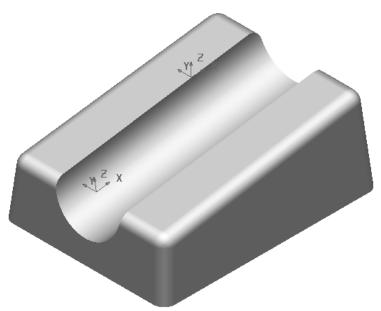
18 Modify the cylinder parameter to a **Radius of 15mm** and **length of 125mm.**

19 In the **Workspace** tab of the cylinder dialog, modify the **X coordinate to -20.**
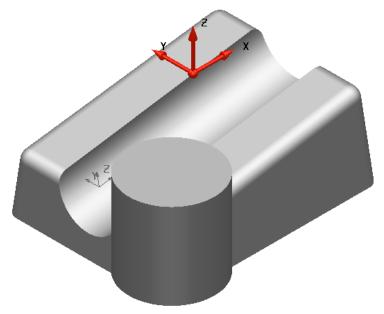
Using a Boolean operation to remove the cylinder

1   Boolean **Remove** the **solid cylinder** from the **main block solid**.
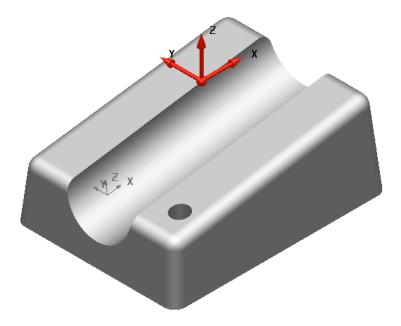
2   **Activate** the original workplane (**Workplane 1**) at the **top centre** of the **component**.

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**.
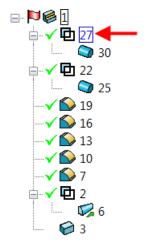
Mirroring the hole feature

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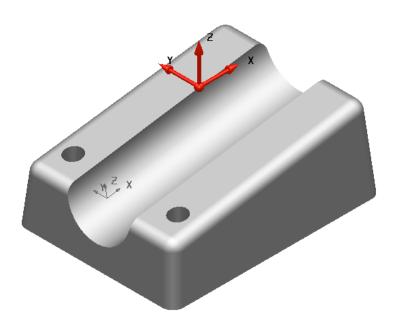
3 Select **Edit Sub Item** from the flyout.

4 **Mirror** the hole across the **ZX plane**.
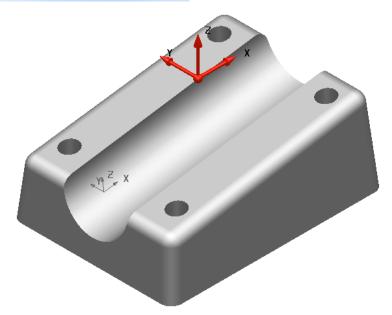
The new hole is displayed at the top of the **Feature Tree**.

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.**
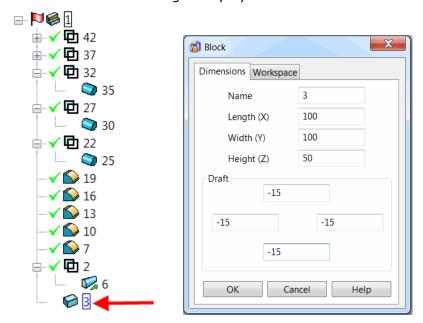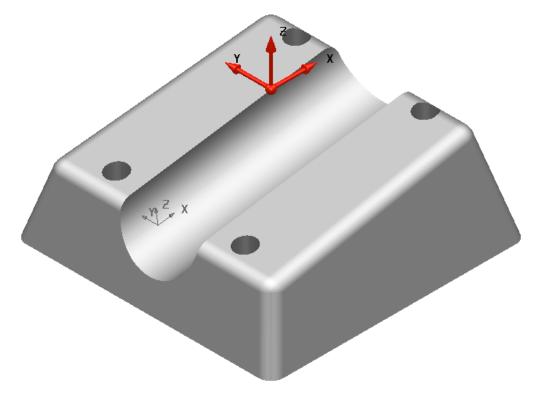
Basic Solid Editing

1   Double left-click on the original solid block symbol at the bottom of the **Feature Tree.** The **Block** dialog is displayed.
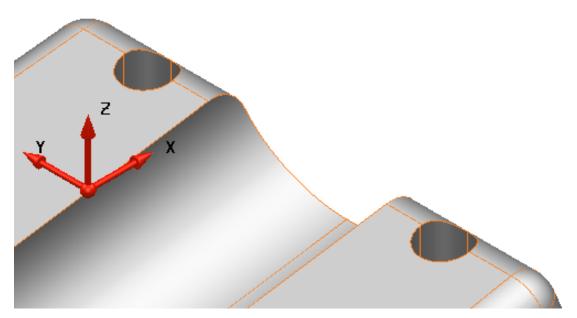
2   **Modify** the value for **Width** (**Y**) from **75mm** to **100mm**, and all four **Draft angles** to **-15°**

3   Select **OK**.

The **solid component** is updated with the changes.
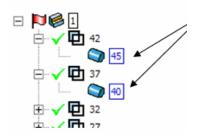
The **holes** must now be moved **clear** of the **fillets** running around the **upper sloping** face.

4   **Zoom** into the **area containing** the **two latest holes** in the **+X** direction.
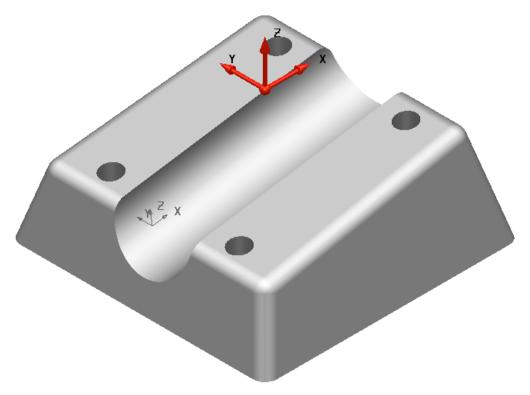
5   Hold down **Ctrl** and **click** the two **Boolean operations** (**holes**) to **select** them **both**. The corresponding **cylinders** are **highlighted** in the **CAD model**.

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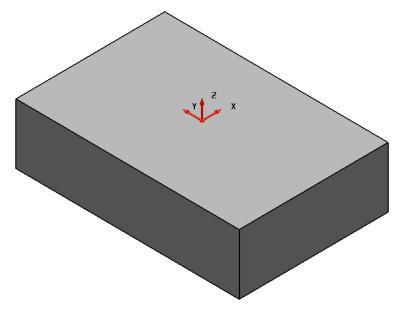
.....\Training Data\Coursework\Solid Modelling Example2

11 **Close** the model.

Solid Modelling Example 3 – Die block

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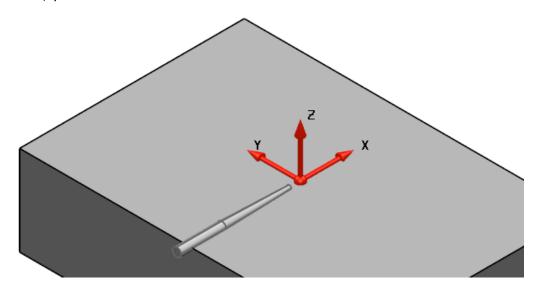
2   Create a **workplane** at **0** and ensure the **Principal Axis** is set to **Z**.

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**.
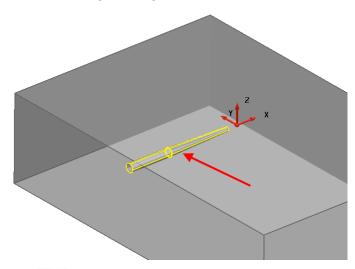
10 **Select** and **Blank (Ctrl+J)** the **solid block**.
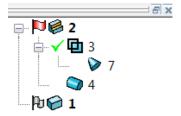
11 **Boolean Add** the **Solid cone** to the **solid cylinder.**

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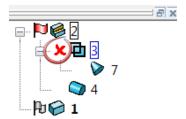
*The **Feature Tree** shows that the '**sprue' solid** comprises the **solid cone** and the **solid cylinder**.*

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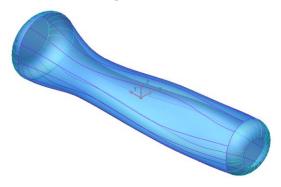
The **suppressed solid feature** can be **unsuppressed** by clicking on the **red cross** which will a **green tick** and the **feature** is **visible**.

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**
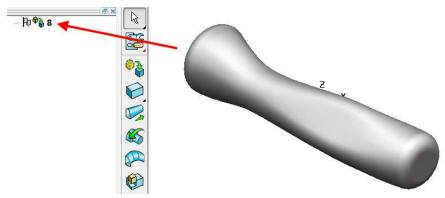
The **imported model** contains various **surfaces**, which will be used to create a **single solid**.
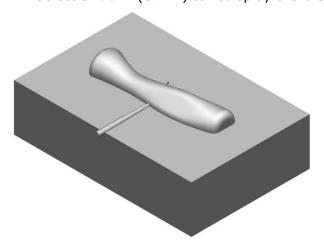
2 **Quick Select All Surfaces** from the **Selection** menu.

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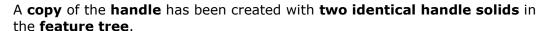
6   Name **level** 5 as **Handle copy** and leave the **level switched off**. Select **OK**.

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**.
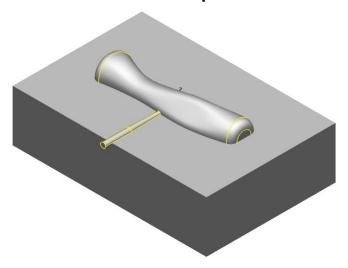
9   With **one** of the **handles selected**, **middle mouse click** over the **level 5** icon in the **levels toolbar**.
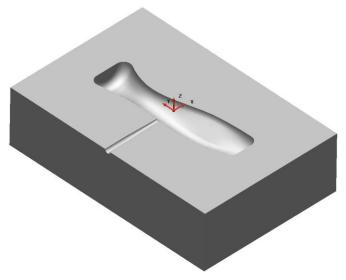
     A **solid handle** has been transferred to the new level. With the **level 5 switched off** the copy of the **handle solid** is now **not visible**.

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. .
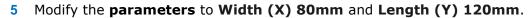
The ability of a **surface** to **split** a **solid** will now be shown.
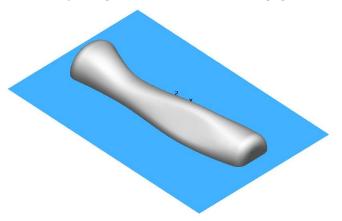
3 From the **Main** toolbar, select **Surface** to display the **Surface** menu.

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**.
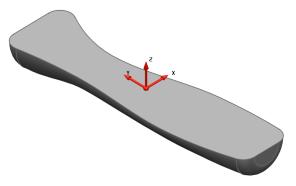
6 **Blank ALL Solids EXCEPT** the **handle**.
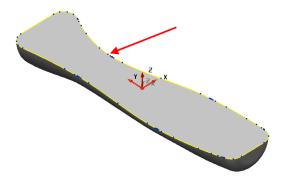
> *Select the **handle** and press **Ctrl+K**.*

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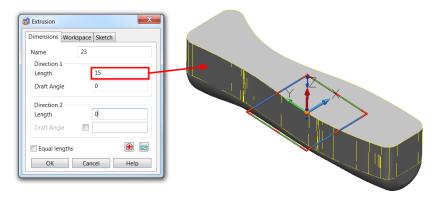
10 From the **Solids** menu, **Create a Solid Extrusion** .

11 **Modify** the **extrusion** and change its **length** to **15mm**.
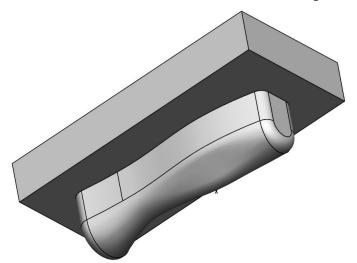
12 Using the **Boolean Addition** operation **add** the extrusion to the handle solid.
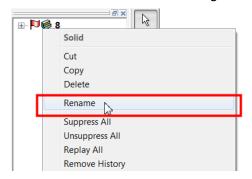
> *The **solid extrusion** forms a **run off** to provide **clearance** between the **electrode** and the **back plate**.*

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**.
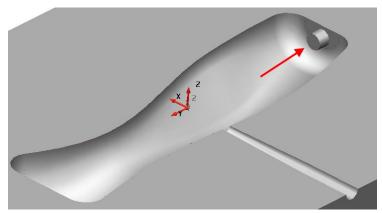
16 Switch **OFF level 5**.

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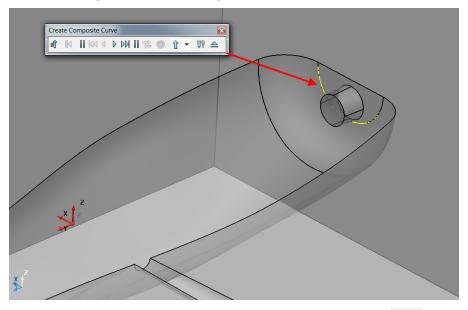
The **solid** now has a **localised undercut core** that is to be **separated** onto a **retractable slide**. A **workplane** is required at the **end** of the **core** to **assist** with the **separation of the slide**.

An **extruded surface** will be **created** and **utilised** again to **split** the **solid** and provide the **required component**.
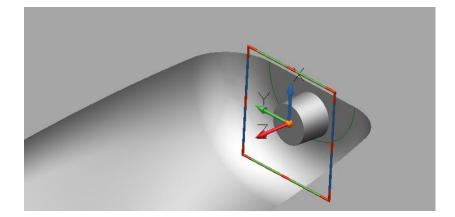
6 Zoom into the small **cone solid** and create a **composite curve** on the edge of the bottom flat as shown.

> *Switching to **wireframe shaded** or **transparent** view will **display** the **edge clearer** making it easier to **select** and **trace**.*
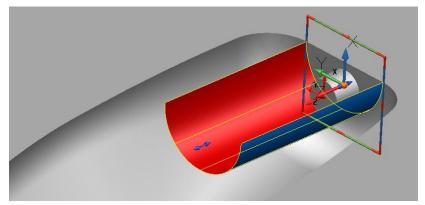
7 **Create a single Workplane aligned to geometry** and click and snap at the centre of the **outer planar face** of the **small cone**.

---

8 Select the **composite curve** and create a **Surface extrusion**.
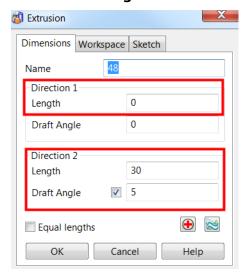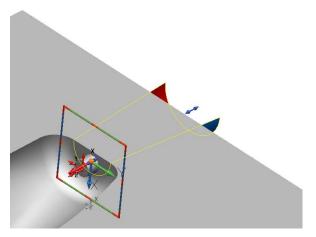
The **surface** needs to be **extruded through** the **model** including a **taper angle**. Both can be achieved simply from the **extrusion form**.

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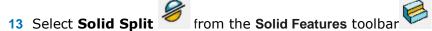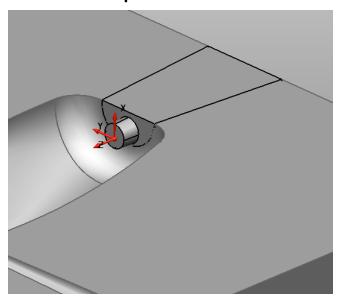
The **extrusion surface** will be used to **split** the **main solid extracting** the **slide** as a **separate solid**.

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

13 Select **Solid Split** from the **Solid Features** toolbar .
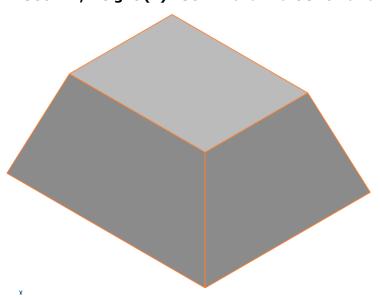
Both resultant solids retain their separate associations within the **History Tree**.

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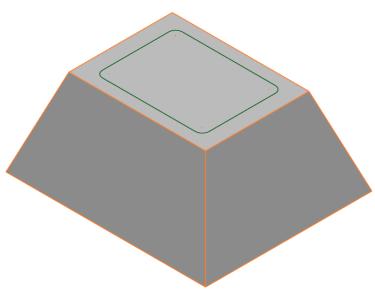
This has formed the **basic shape** for the **alarm box**.

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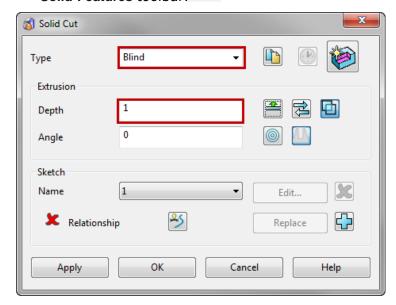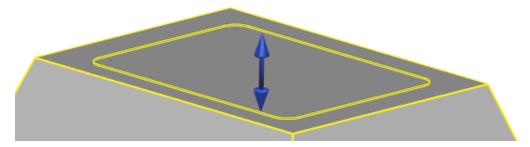
4 With the **composite curve selected**, select **Create Solid cut** from the

Solid Features toolbar.

5 In the **Type** drop-down list, select **Blind** and enter **Depth** of **1mm** and select **OK**.
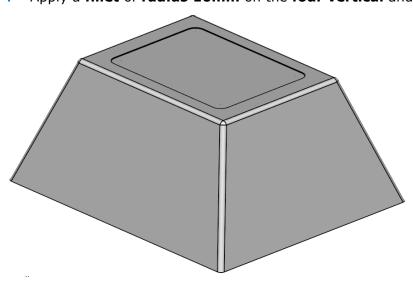
A recess of **depth 1mm** is created in the **solid** downwards from the composite curve into the **solid**.

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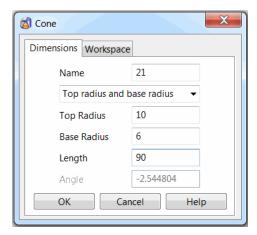
Create four screw apertures
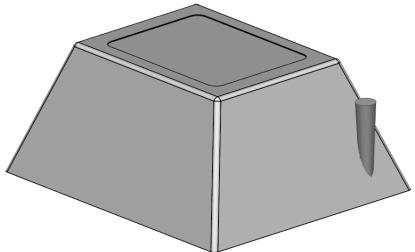
1   Create a **solid cone** at **80 -135 5**.

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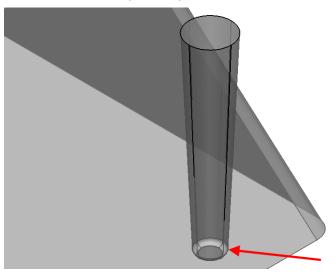
3   Make the new solid cone **active**.

4   Apply a **solid fillet of radius 2** around the **base** of the **cone**.
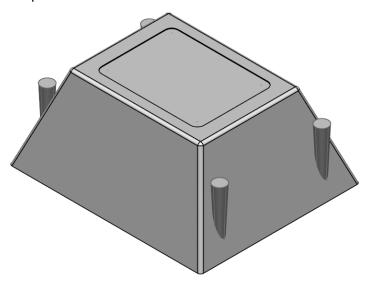
> *You may need to use a **transparent shaded view**, or alternatively **Blank** (**Ctrl+J**) the **main block**.*
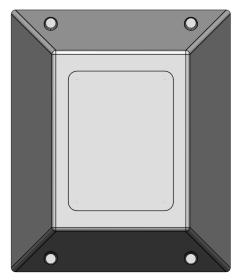
5   **Mirror** the **cone** across the **XZ** and **YZ planes** of the worklplane 'Datum' to produce **4 cones**.
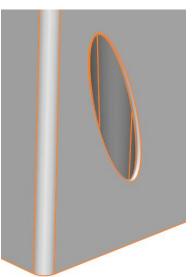
6   Use **Boolean Subtraction** to **subtract** the **4 cones** away from the **block**.

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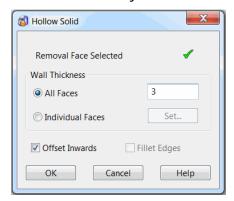
Hollowing out the base

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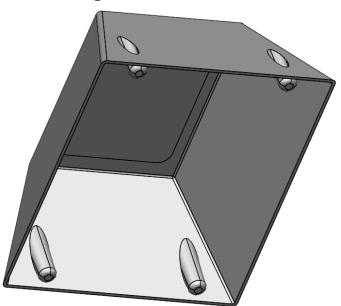
The **red cross** adjacent to **Removal Face Selected** is replaced by a **green tick**.

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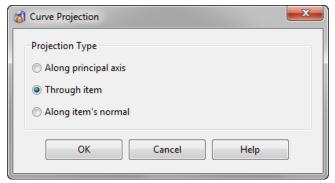
4   **Fillet** both left corners with **radius 10mm**.

5   Create a **composite curve** of the profile.
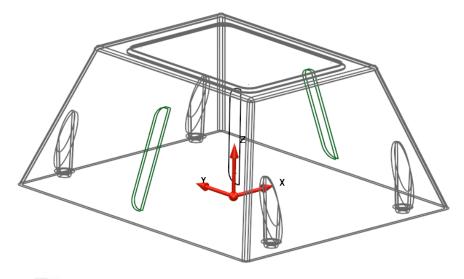
6   **Box-Select both** the **solid** and the **composite curve**.

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)**.
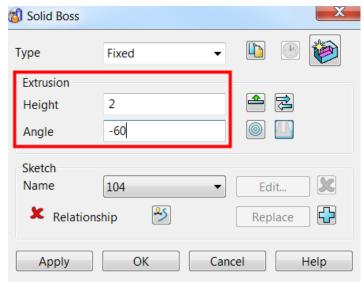
Only the *curves* on the *outside face* are required.
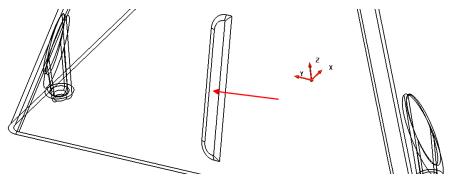
1   **Select** and **Delete** the **two curves** which were **projected** onto the **inner face** and the **original central curve**.

2   Select the **remaining outside curves**.

3   Select **Create solid boss** from the **Solid Features** toolbar .
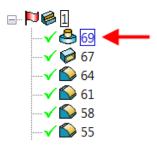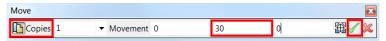
4   Change the **Extrusion height to 2mm** and **enter an angle** (**applies draft**) of - **60°**.

Select the **new boss feature** (created as one) from the **feature tree** as shown.
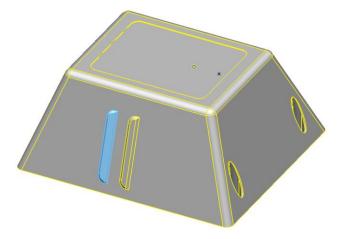
5   Select **Move** from the **General Edits** toolbar.

6   Select **Copy selected item** and enter **30mm** in the **Y input box**, then teh **green tick** to **confirm**.
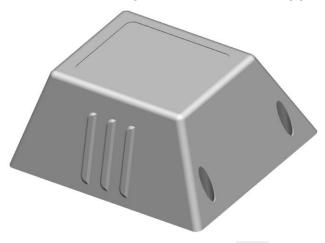
A **copy** of the **boss features** have been created to the side of the original.
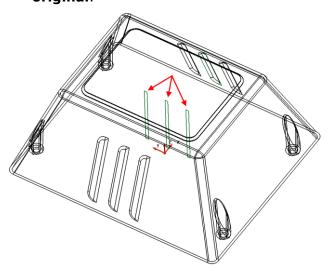
7   With the **Move** dialog still **open** and the new **boss features highlighted** enter **-60** in the **Y input box** to create a **copy** on the **opposite side**.
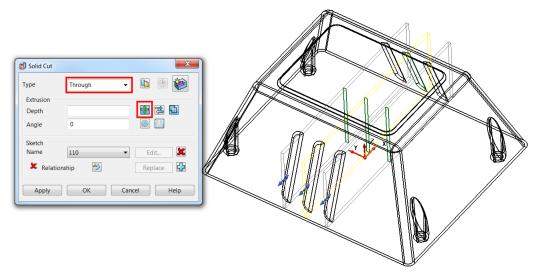
8   Select **View down from left (-X).**

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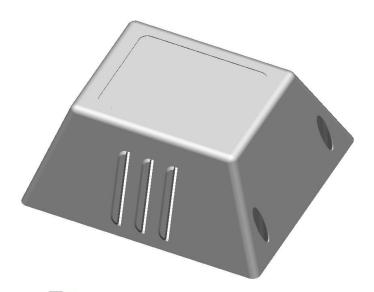
12   Select all 3 rectangular **composite curves**.

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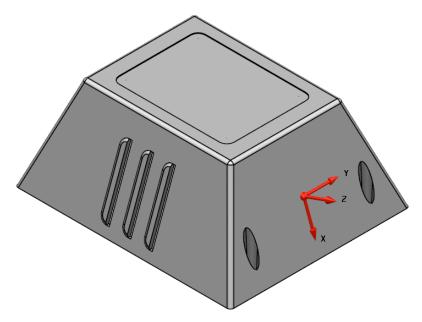
The **next stage** is to create a **mounting** for the **strobe light** on the **end face** of the alarm box. To **assist alignment** of the **mounting**, a **workplane** will be **created aligned normal** to the **face**.

1 Select the **Iso1** view.

2 Select **Workplane Aligned to Geometry** and select the **face centre keypoint** shown.
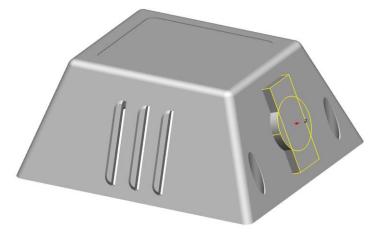
3   **If necessary**, **Twist** the workplane about **Z** so that its **X** axis is aligned towards the top of the alarm box.

The **Z axis** of the **new workplane** is **centrally aligned normal** to the selected **face**.
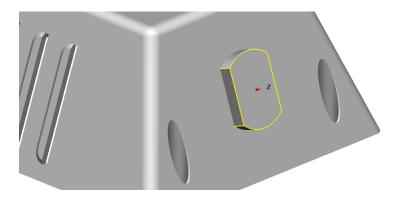
Creating a boss to provide access to the mounting

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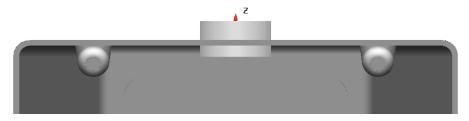
4   Make the circular extrusion **active** and **select** the rectangle block.

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**.
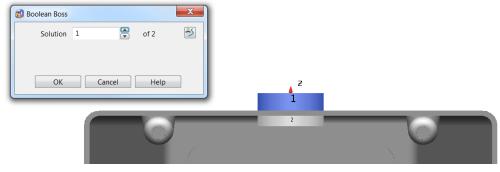
1   Select the **main casing** solid and make it **active.**
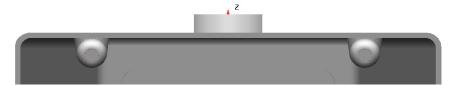
2   Select the intersected solid, and then select **Solid Boolean Boss** from the

   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**.
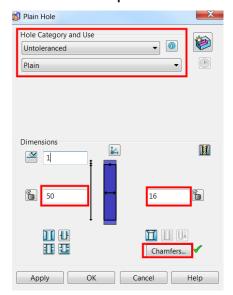
The **solid boss** has been **trimmed** back.
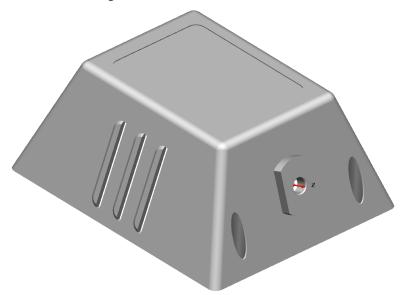
Creating a hole for the wiring

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**.
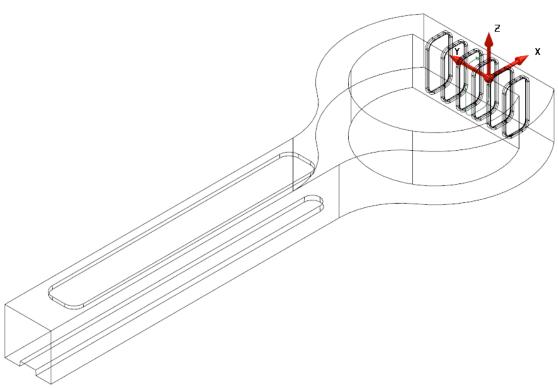
8 Select **File > Save As**:

.....\Training Data\Coursework\Alarm Box.psmodel

9 **Close** the model.

Plastic Handle Example

1   **Open** the model:
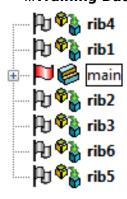
…**PowerSHAPE Data/toy_handle.psmodel**

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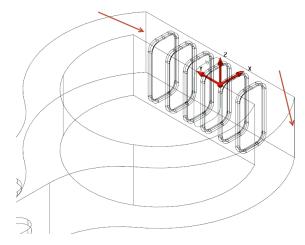
*The **Solid Fillet** dialog is displayed. Unlike surface filleting, **solid filleting** uses the **sharp edges** on the **solid** as the **filleting track**. The **order** and **extent** to which **solid fillets** are **created** is **important** if the correct design specification is to be achieved.*

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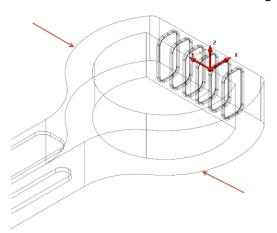
The **fillet path** runs until it finds a **sharp edge** (**Discontinuity**).

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**.
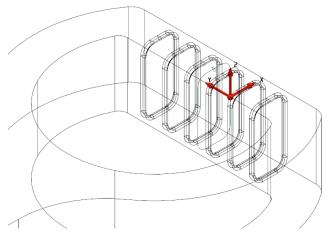
Modifying the D-shape

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**.*
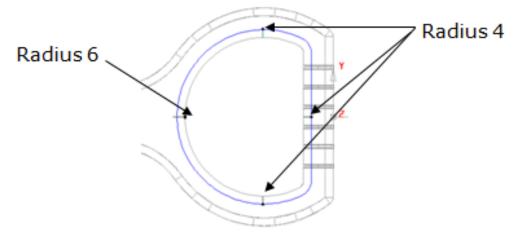
10 Select the **top inner edge**.

11 Select the **Advanced** option in the **Fillet** dialog to display the **Variable Radius Fillet** dialog.
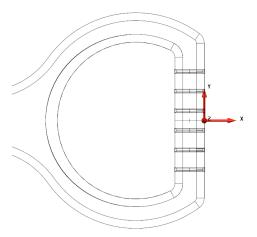
This **dialog** allows you to use the **mouse** to **select** an **arc** that represents the **radius of the fillet** at a location around the **curve** representing the shape. This works in a similar way to **variable radius filleting** of **surfaces** covered later. When you move the mouse along the 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**.
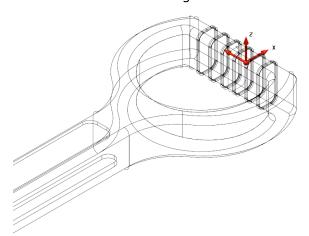
14 **Modify** the **arc radii** as indicated on the **image above** and select **OK** to produce the **variable radius fillet**.

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.
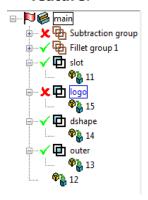
18 Select the **Solid** in the **graphics area**.
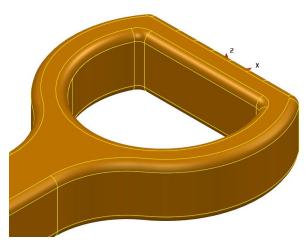
The outer must be cored with a **3 mm minimum wall thickness**. Due to **dimensional restrictions**, some of the **features** must be **suppressed** before attempting to create a **new solid** to represent the **inner**. It is also necessary to perform **additional material removal** on the **inner** 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.
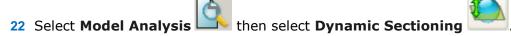
20 In the **History Tree** click on all **red crosses** adjacent to the **suppressed features** to **reinstate** them onto the **outer solid**.
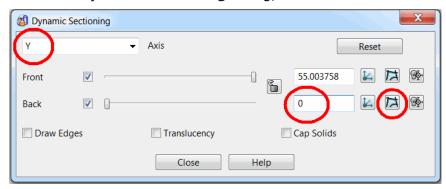
Changes to the inner core solid

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** [icon] then select **Dynamic Sectioning** [icon].

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**) [icon] and set the **Prinicipal Axis** as **Y**.

[icons]

26   **Quick select ALL wireframe** [icon] 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.

> [note icon] *As you should hopefully see at the **logo recess** the **minimum wall thickness** his **reduced** to **1mm** due to the **depth** of the **recess**.*
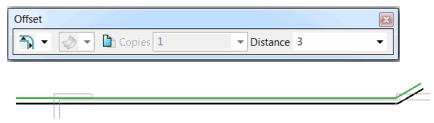
28   **Snap** a **wireframe line** to the outer **composite curve** as shown above, then **edit** the **length** to pass through the **end wall** (for example, increase the **length** to **200mm**).

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**.
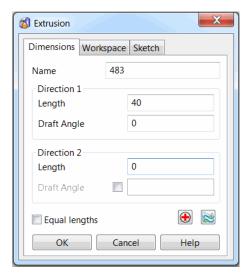
30 **Select** and **Blank** (**Ctrl+J**) the **two composite curves**.

31 Create a **composite curve** along both **lines**.
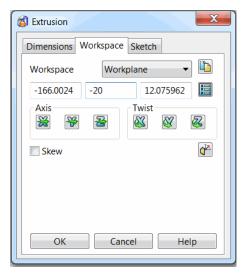
32 From the **General Edits** toolbar, select **Offset**. Specify **Round discontinuities**, **No. of Copies 1**, and **Distance 3mm**, then press the **Enter** key.

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**.
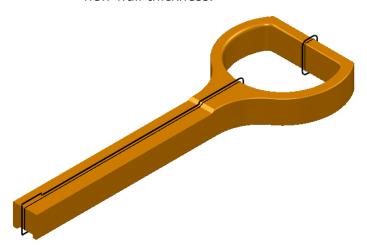
*This may be easiest in the **Solid History Tree**.*

39 From the **Feature** toolbar, select **Boolean Subtracction** to remove everything to the **red side** of the **surface** from the **solid**.
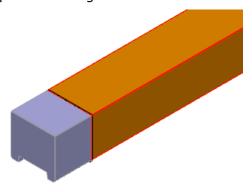
*The **core solid** illustrated **below** also includes the **revised wireframe** obtained from a further **dynamic sectioning operation** to show the new wall thickness.*

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** .
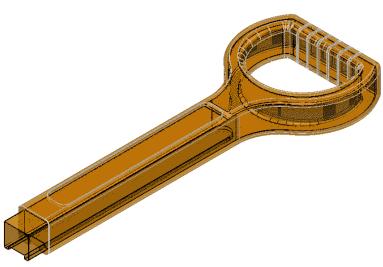
> *You may find that **PowerSHAPE** will not initially add the **extrusion** to the **inner core solid**. This may be because the extrusion is of **Version 8** format (mainly found in older **PowerSHAPE** sessions). To fix this select the extrusion a from the **Solid Edit** toolbar, click on the **Version 8** Converter icon.*
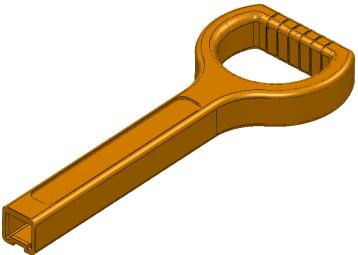
45 Make the **outer solid active**.

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**.
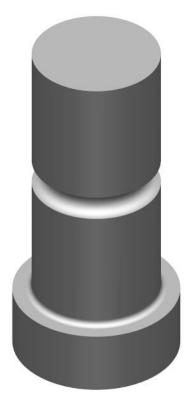
Radial Solid Cut

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**.
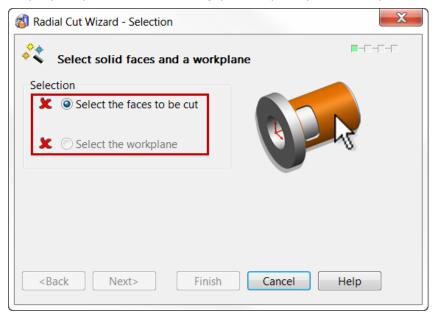
> The **Radial Solid Cut** feature works on a **selected** solid, however it is good practice to have this as the **Active** solid when you are editing it. The selected solid is represented by the **yellow** outlining of the solid.

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.
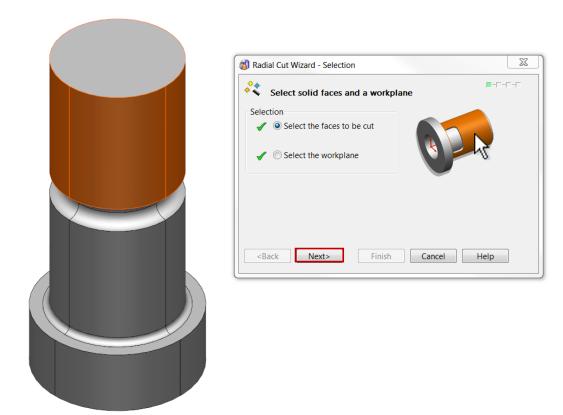
To make a **selection** simply **click** your chosen **face(s)** in the **graphics window**, these will be highlighted in orange. In this first example we will create a **rectangular pocket** on the **uppermost cylindrical face** of the solid.

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**.
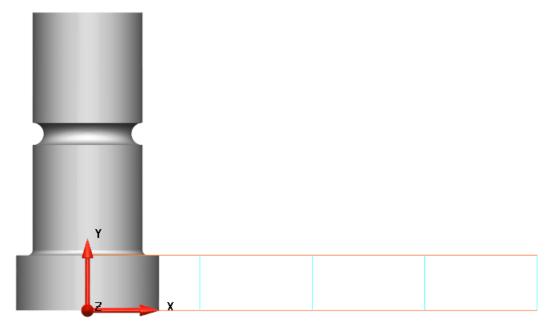
> *The **Z Axis** of the **Active/Selected Workplane** needs to be **parallel** to the axis of the cylindrical face(s) to be cut.*
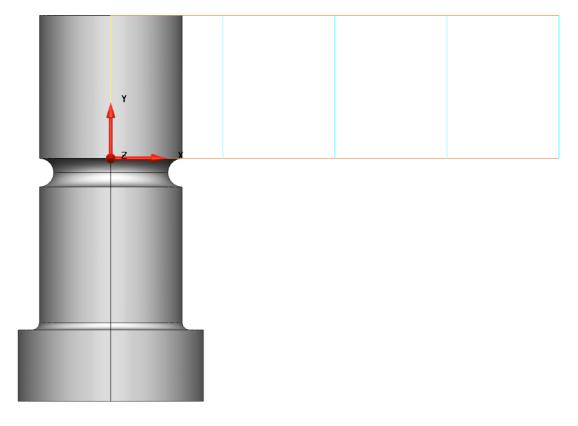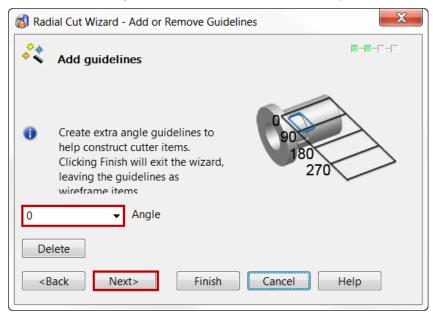
PowerSHAPE now unwraps the selected face of the solid into a plane. Its upper and lower edges outlined by **horizontal lines**, while also creating guidelines at 0, **90°**, **180°**, **270°** and **360°** (same as 0° start and end) to help visualise the distance **around** the face.
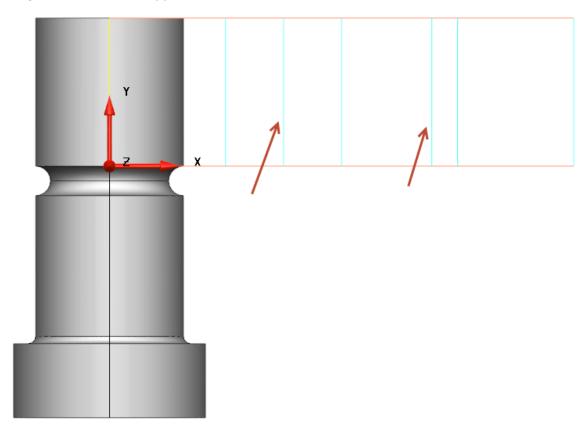
58 Select **Next**.

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.
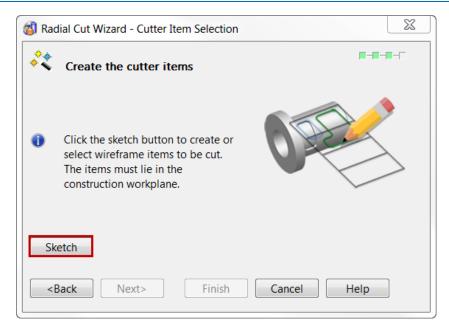
> *Note: Selecting **Finish** in the wizard at this point will simply create the **guidelines** as wireframe allowing the user to create the wireframe at a later stage and re-enter the wizard to create the cut. However **in this example** we will use the wizard throughout to create a **Radial Cut**.*

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.
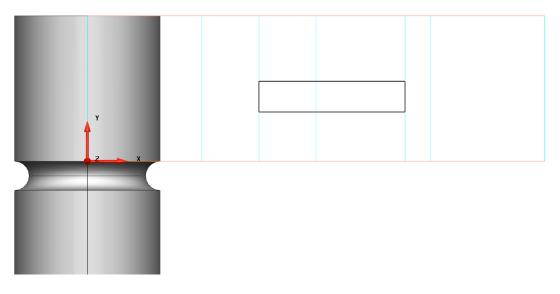
> *The **angle** is measured in the **XY plane** from the **positive X axis** in a **counter clockwise** direction.*

> *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**.*
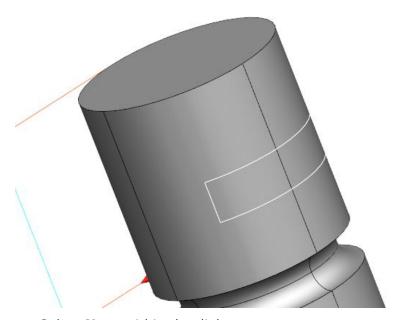
64 Create a **composite curve** from the **rectangle**. As with other solid feature function the use of composite curves is required to perform the function. The

Composite curve creator [icon] from the **Curves** menu [icon] is required. Trying to use the **ALT + Click** shortcut in this case will likely fail in this case.
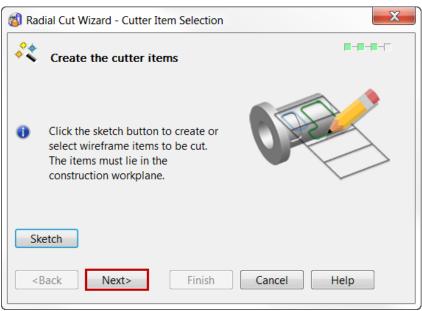
65 Once you are happy with the **shape** and **position** of you **composite curve**, **SELECT all required curve(s) that you wish to wrap/cut** and click the **green tick** to accept the changes and **re-enter** the **wizard**.
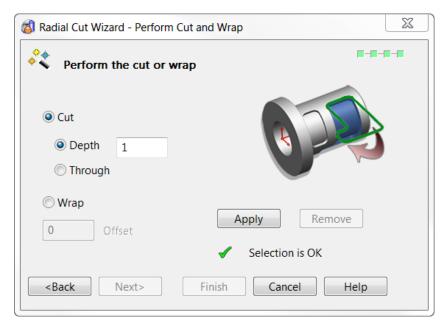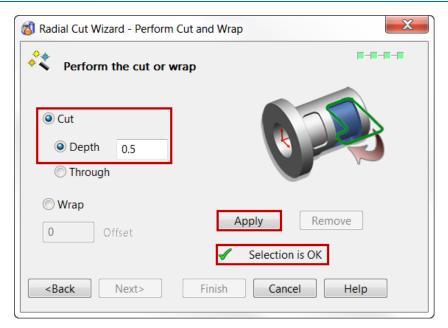

Accepting this form will also create a view of the **wrapped curve(s)** onto the **model** to show there **position** and **size** in direct relation to the solid model.

66 Select **Next** within the dialog.
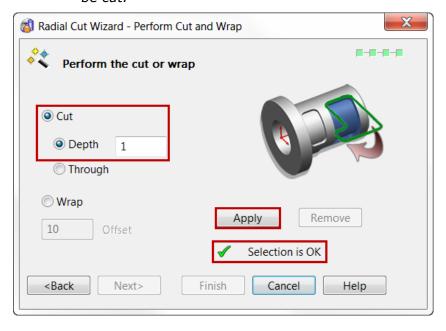
67 Within the **final window** of the **wizard** there are **two** main options; **Cut** or **Wrap**.

▪   **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**.
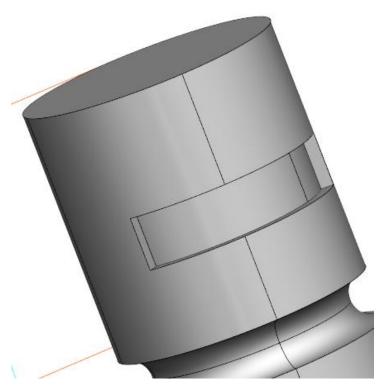
   *If after you select **Apply**, changes are required and you are still in the wizard the **Remove** button will delete the current cut, allowing edits to be made.*

   *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**.
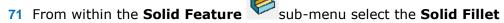
Looking to the **left** at the **Solid History Tree** we can see that this feature has been created by effectively creating a **solid** from the **wrapped curve** and the using a **Boolean Remove** function to create the cut.

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**.
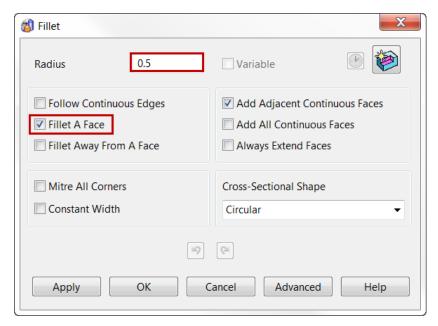
> *The **Fillet a Face** allows you to select faces of the solid and automatically select and fillet **all** the face edges to save time multi-selecting faces.*
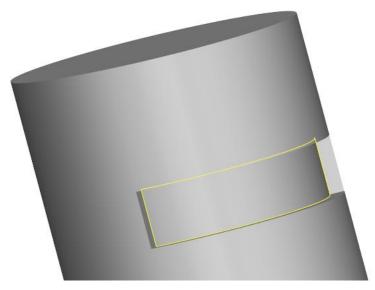
> *Currently **solid edits** to the **radial cut feature** are only supported **after** the feature has been **created** and once the **wizard** has been **finished**.*
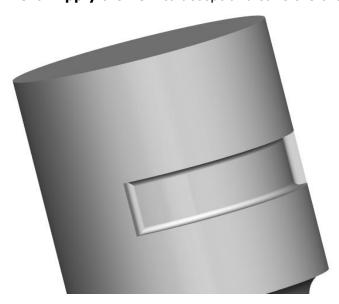
73 Select the **face** at the bottom of the **radial cut pocket**.

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.
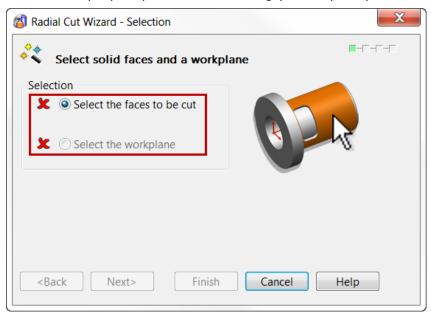
> *This should already be the in place from the **previous example**, however it is **good practice** to develop a process of checking everything is in place **before** you start the feature wizard.*

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**.
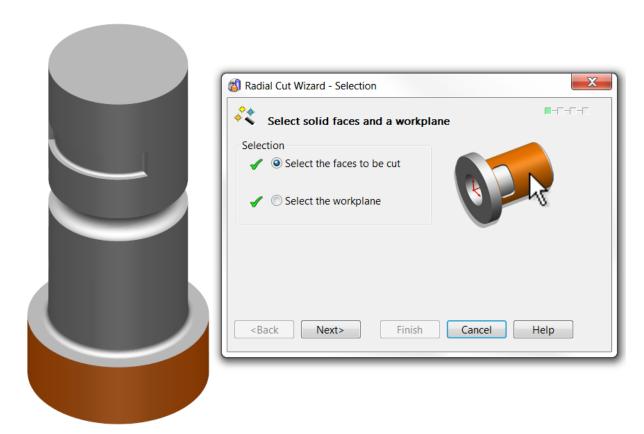
80 Select the **Lowest "Base" Cylindrical Face** on the solid as shown in the image on the next page.

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.*
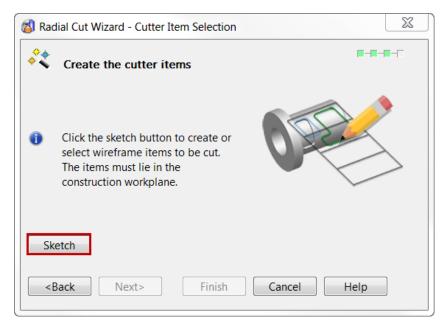
82 Click **Next**.

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**.
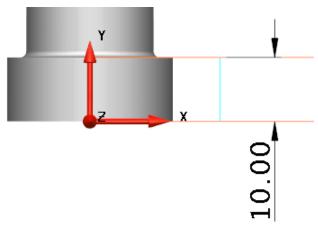
85 Select **Sketch** to **hide** the main wizard **dialog** and enter the **sketch mode**.

> *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**.
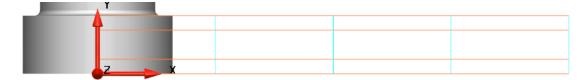
87 Select and **Delete** the **dimension**.

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**.*
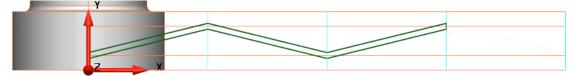
Next we will create a zig-zag line with a **height** varying between **2.5mm** & **7.5mm** from the **lower edge** of the face changing direction every **90°**.

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.
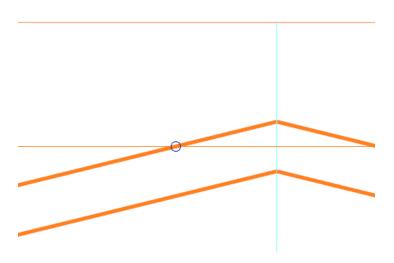
94 Using the vertical guidelines that are already in place and the two **composite curves**, **track** a **single closed composite curve** around both zig-zag profiles.

*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.
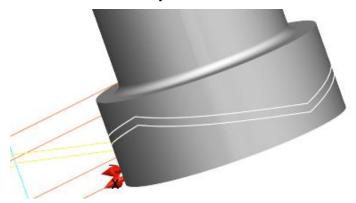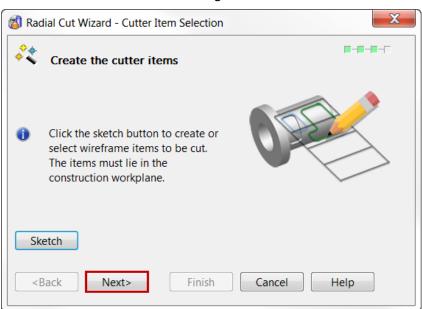
96 Once you are happy with the **shape** and **position** of you **composite curve**, **SELECT all required curve(s) that you wish to wrap/cut** and click the **green tick** on the **sketch dialog** to accept the changes and re-enter the **wizard**.

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.
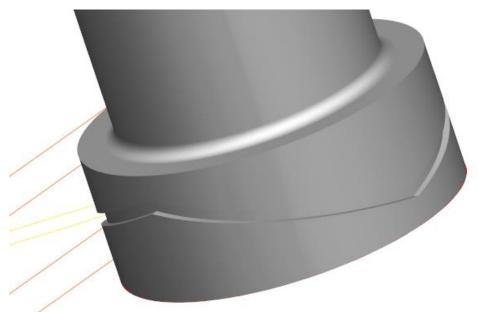
98 Select **Cut** and a **Depth** of **0.5mm**, then select **Apply**.

> *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**.
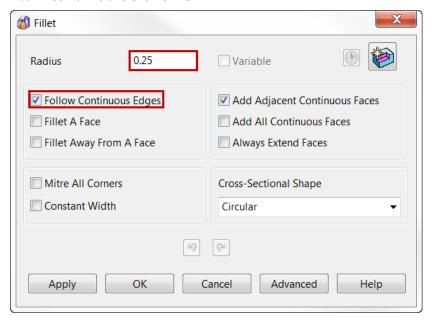
102 From within the **Solid Feature** sub-menu select the **Solid Fillet** .
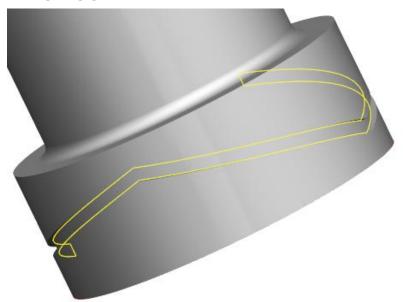
103 Enter a **Radius** of **0.25mm**.

104 Select the **ALL** the edges surrounding the face at the bottom of the **radial cut zig-zag groove**.
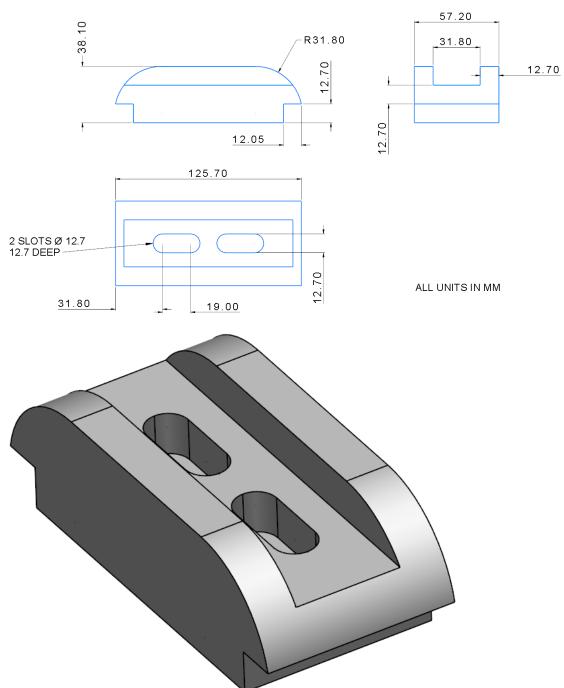
105 Click **Apply** then **OK** to accept and save the changes and **create** the **fillet**.

Exercise 4: Solid model

1   Using the basic dimensions provided generate a solid model.
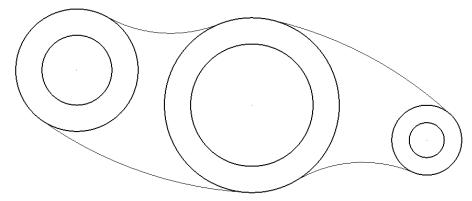
1   Select **File > Save As**:

.....\Training Data\Coursework\Exercise 4.psmodel

Close the model.
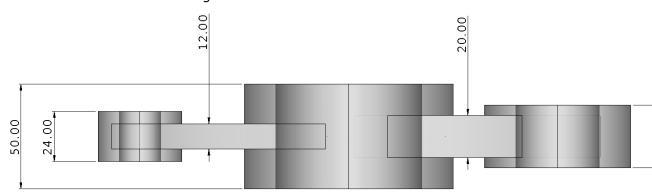
Exercise 5: Solid lever

1   Open the previously completed wireframe model from Chapter 2
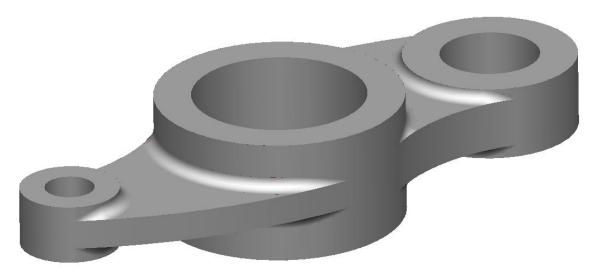
.....\PowerSHAPE_Models**lever-example.psmodel**

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**:
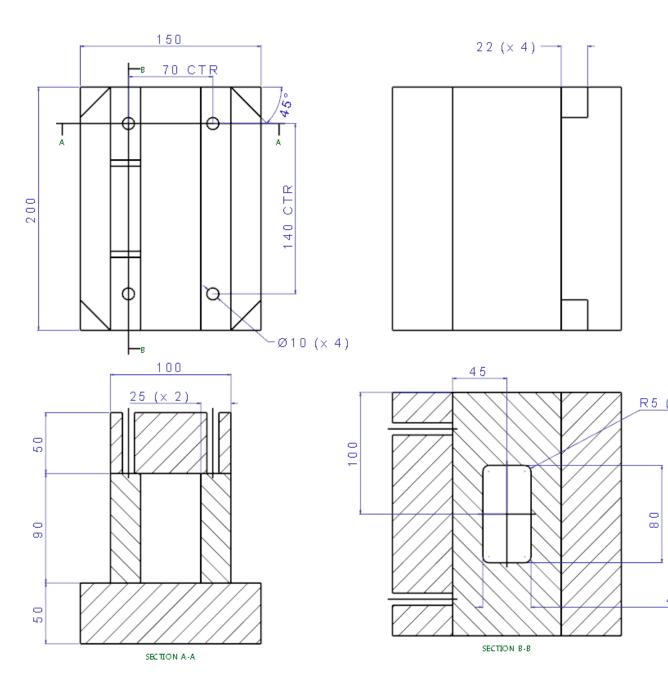
.....\Training Data\Coursework\Exercise 5.psmodel

5   **Close** the **model**.

Exercise 6: Assembly plate
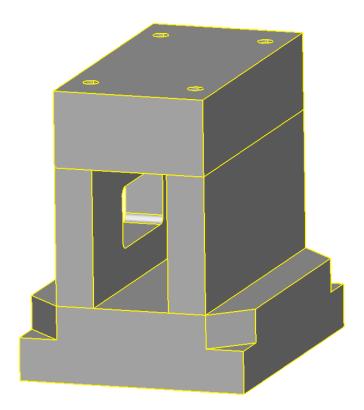
1   Generate a solid model using the dimensions provided.

SECTION A-A

SECTION B-B

1   Select **File > Save As**:

   .....\Training Data\Coursework\Exercise 5.psmodel

2   **Close** the **model**.

6. Surface Modelling
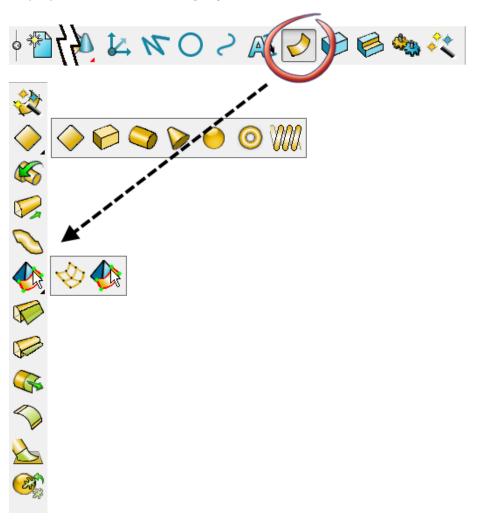
What is a Surface?

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
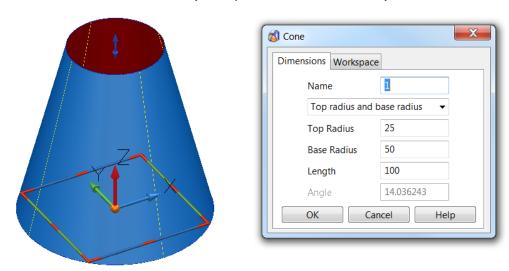
Primitive Surfaces

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
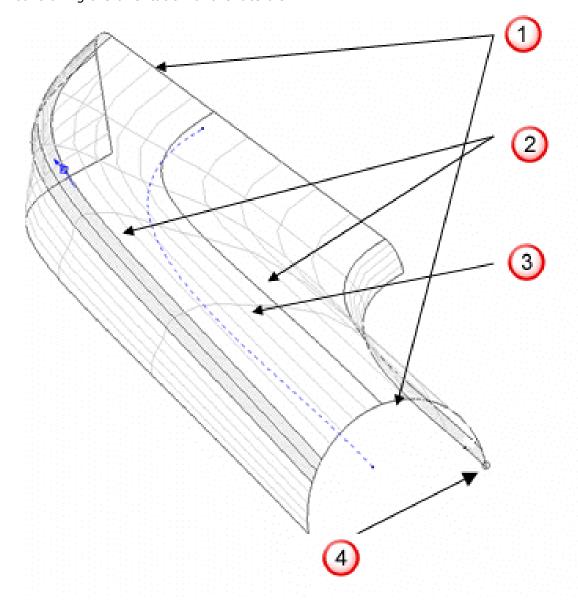
A **Power Surface** is based on a network of 4 sided wireframe elements appearing as curves along **(longitudinals)** and across **(laterals)** the surface.

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.
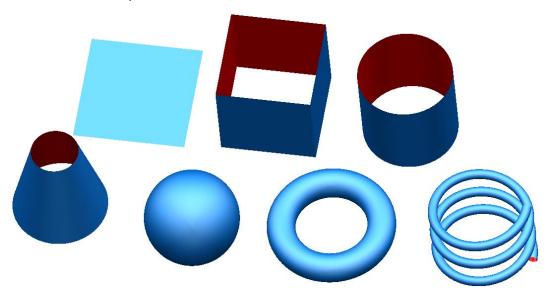
The Cato mark (4) This identifies the **start point** for **Laterals** and **Longitudinals** on a **Power Surface**. It is positioned a **short distance** from **point 1 along lateral 1** with a short line pointing from it, representing the **Longitudinal direction**.

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)**.
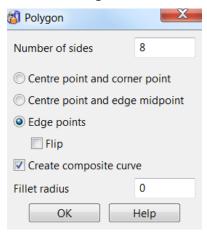
Extrusion Example 1

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.
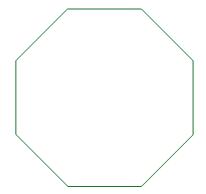
The **Polygon** dialog is displayed. This allows you to specify the **Number of sides** in the polygon. The option to **Create composite curve** provides the ability to create a polygon as a single entity. If this option is deselected, the sides of the polygon will consist of single lines.

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.
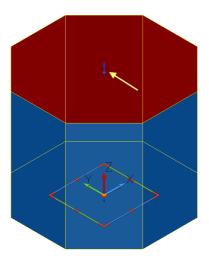
The **wireframe** is **completed** as a **single closed composite curve**. This will be used as the shape of the **Extrusion Surface**.

7   Select the composite curve and switch to **View Iso1**.

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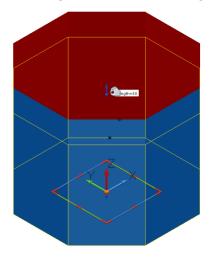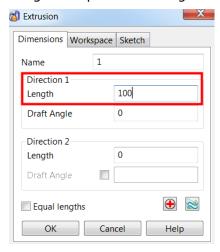
This produces an **extrusion surface** from the selected **composite curve** up the **Z-axis** by a default value.

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**.
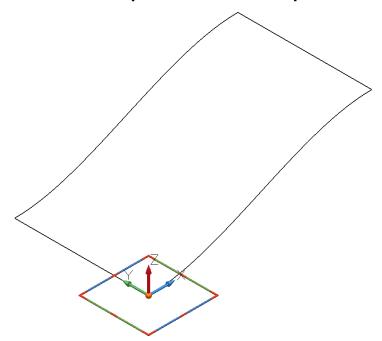
13 Select the extrusion surface.

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**.
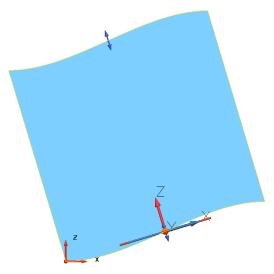
An **extrusion surface** will be created from the **imported 3D composite curve**.

3 Select the **composite curve** and create a **surface extrusion** to a **length of 50mm.**
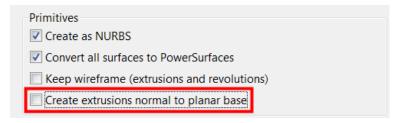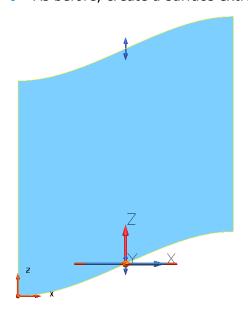
The **extrusion direction** is **normal** to the **planar base**. This default can be changed.

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.
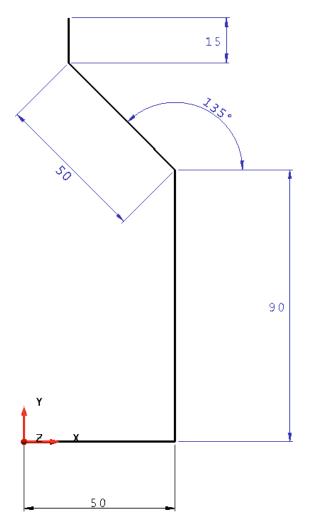
9   Select **File > Close** without saving the model.
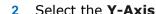
Surfaces of Revolution Example

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** .
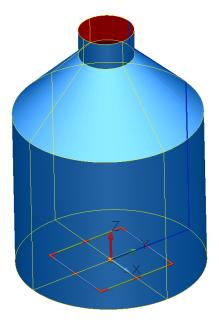
3 Create a **composite curve** of the section geometry, and select the wireframe.

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**.
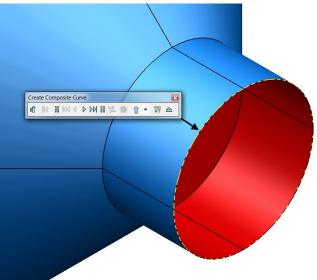
An end cap will be created by generating a **curve** around the end profile, and applying the **Fill-In surface** option.

5   Select the **view Iso3** (**Ctrl+7**).

6   Create a **Composite Curve** on the **open edg**e.

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.
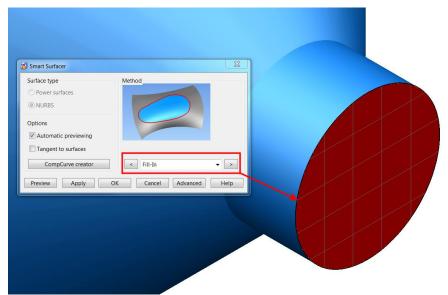
8   Select the **Smart Surfacer** from the Surface toolbar.

9 In the drop-down list, select **Fill In** (if not already selected).

📝 ***This Wizard will be used in detail later in the chapter.***
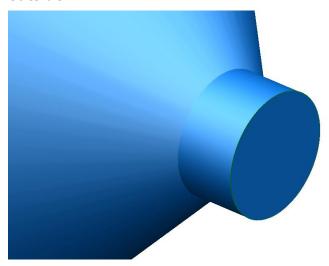
10 Select **OK**.

📝 *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**.
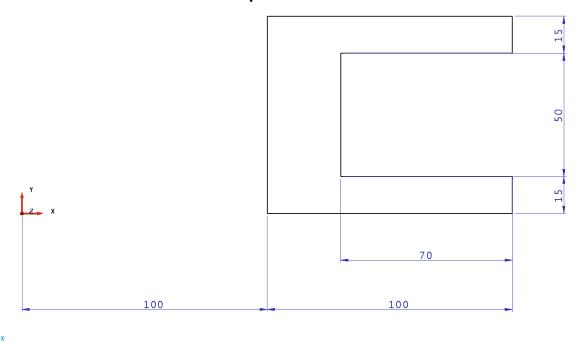
Wheel Example

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**.
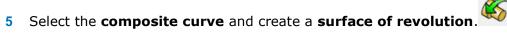
3 Generate the **basic wireframe shape**.
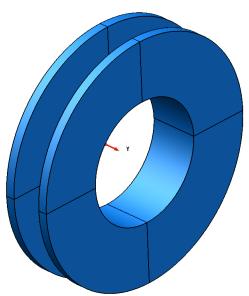
4 From the **Curve** menu create a **Composite Curve** of the **wireframe**.

> 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**.
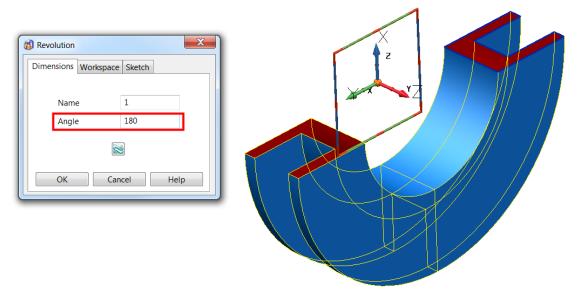
6   **Double-click** the **surface**. The **Revolution** dialog appears showing the workplane options as well as the **Angle of Rotation**.

7   Change the **Angle** to **180°** and select **OK**. The surface is generated through an angle of 180 degrees.
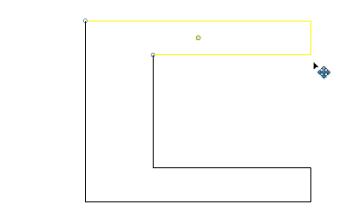
📝   *Due to the properties of* ***surface models*** *as described earlier, the ends of the* ***primitive surface*** *will not be* ***capped****.*

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.
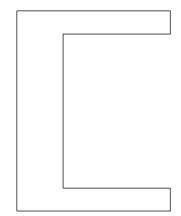
📝   *Ensure the geometry to select is* ***completely*** *is within the* ***drag box****.*

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.
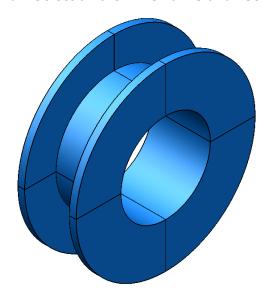
5   Select all the **wireframe** and **recreate** a **surface of revolution**.
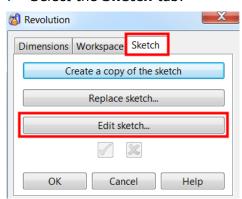
6   **Double-click** the **surface** to display the **Revolution** dialog.

>   *The surface will be **modified** by directly editing the wireframe.*
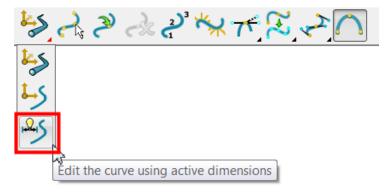
7   Select the **Sketch** tab.
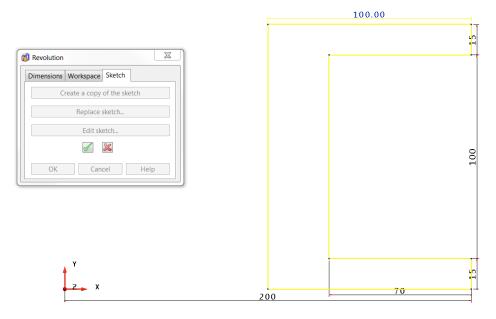
>   *If required, a copy of the original wireframe can be created.*

8   Select **Edit Sketch…**

9   From the **Surface/Curve editing** toolbar, select **Edit the curve using active dimensions**.

Edit the curve using active dimensions
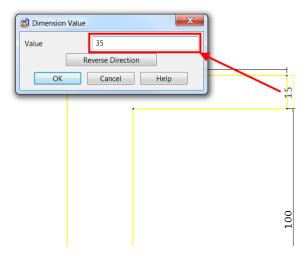
PowerSHAPE undraws the **surface** and displays the **dimensions** for the **wireframe**.

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.** .
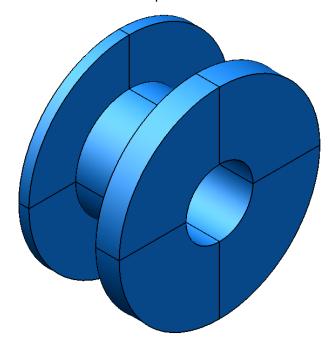
11 In the **Value** box, enter **35** and select **OK**.

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.
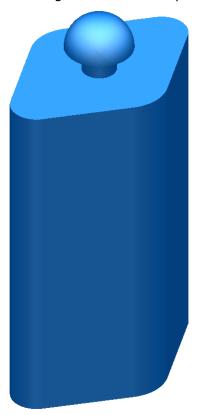
15 Select **File > Save As** and save the file as:

   .....\PowerSHAPE-Models\Primitive wheel.psmodel
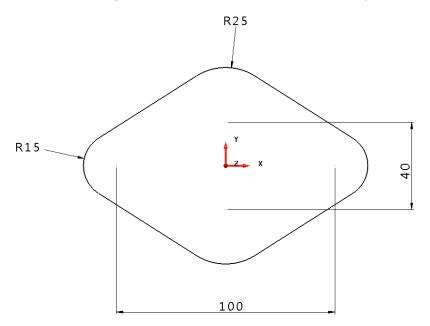
16 **Close** the model

Exercise 7: Surface bottle

1   Using the information provided, create the surface bottle shown.
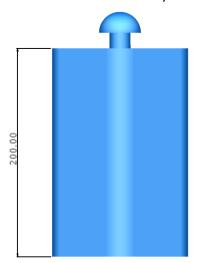
▪   The following wireframe defines the sectional shape of the body.
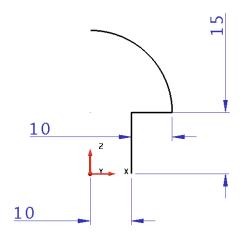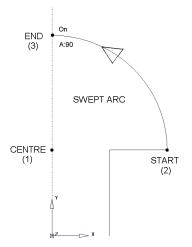
▪ The main bottle body is **200mm in height.**

▪ 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.*
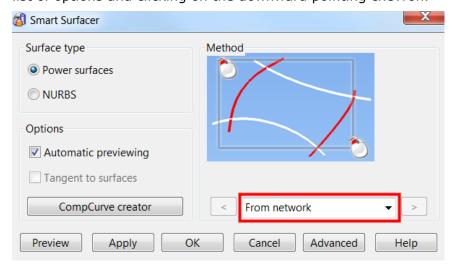
▪ Ensure fill in surfaces are applied to both top and bottom of the body.

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.
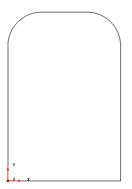
Automatic Surfacing options covered in this section are:

- **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.
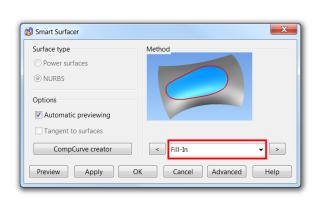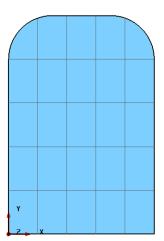
4   Select all the wireframe entities.

5   Select the **Smart Surfacer** ![icon] from the **Surface toolbar**. ![icon]
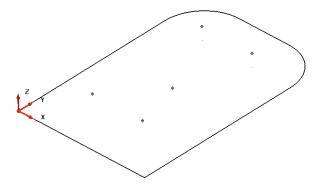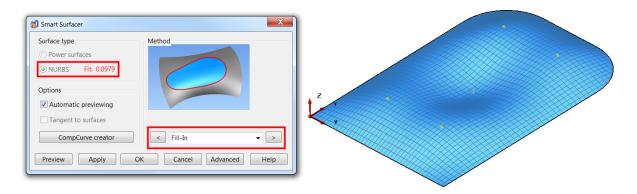
The dialog automatically selects the most likely surface creation options for the selected wireframes. In this case, a Fill-in surface.

6   Select **OK** to accept the surface and close the form.

7   Select and delete the **fill-in** surface.

8   Select **Create a point** ![icon] from the **Workplane** menu. ![icon]

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** ![icon] from the **Surface** toolbar. ![icon]

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.
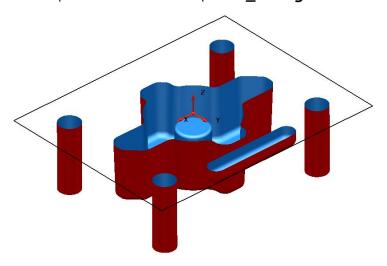
12 Select **cancel** then close the model

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
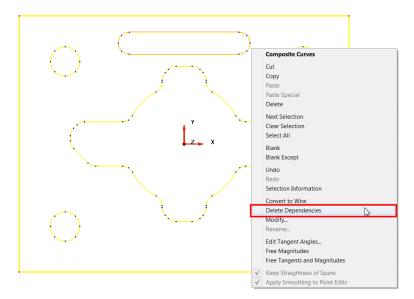
A new **Fill-in surface** will be generated **inside** the **rectangular wireframe** and from **composite curves** traced around the top edges of the surface forms.

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.**
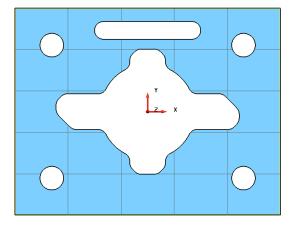
When a curve or composite curve is created on an object (for example, a surface), it may be dependent upon that object. This option removes any such relationships that may exist between the curve and in this case, the surfaces.
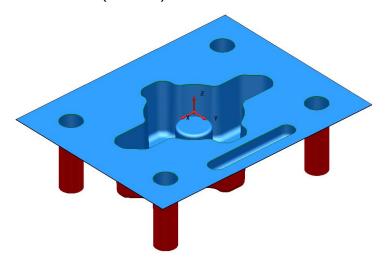
5   Select all the composite curves, then the **Smart Surfacer.**

6   Select **OK** to accept the fill in surface.

7   **Unblank** (CTRL+L) the model.

8   Save the model as
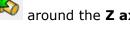
.....\PowerSHAPE-Models\Surface Die example.psmodel
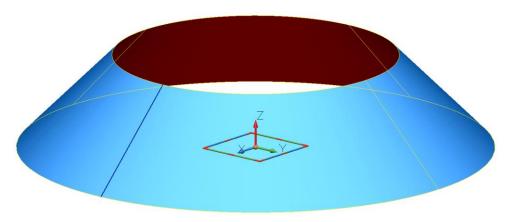
9 Close the model

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**.
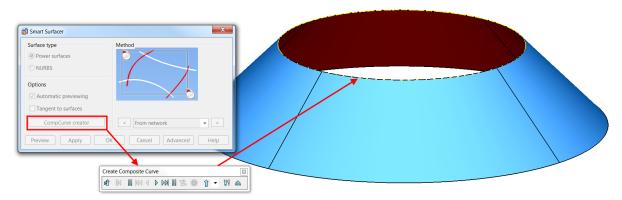
2 Start a single line at **50 0 0** and enter a relative end point using the coordinates -**20 0 20**.

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.
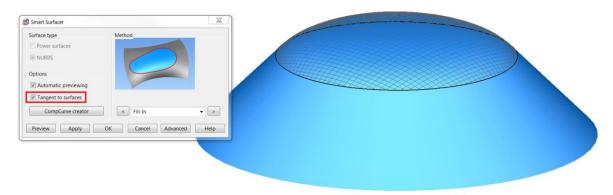
6 Select **Save** and **Eject** to save the curve.

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.
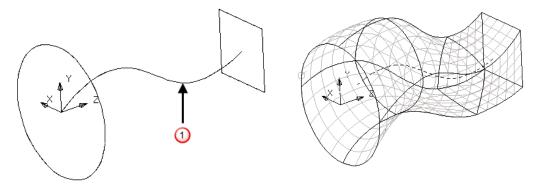
The preview is tangential to the adjacent surface.

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
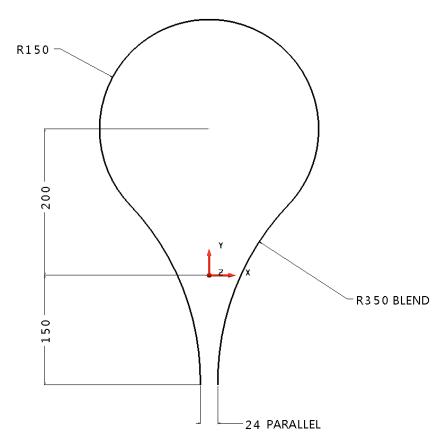
using a two sections running along the whole length of the drive-curve. ① The **minimum requirement** is **one sectional wireframe**, positioned anywhere along the **single Drive-curve**.

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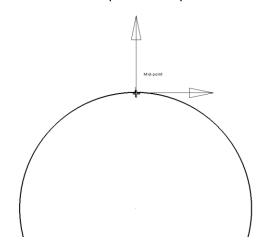
This shape will form the single **drive-curve** along which the surface will be generated.

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
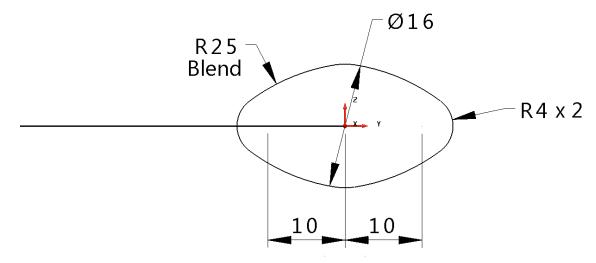
4   **View from the Right (+X)** and zoom in to the new Workplane.
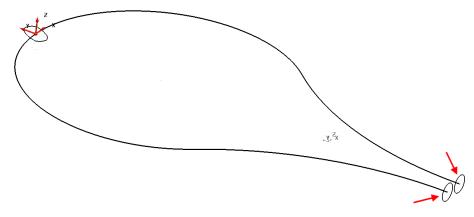
5 Create the following wireframe section on the new Workplane.
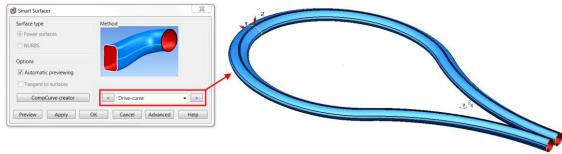
6 Select the **Y** principal plane.

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**.
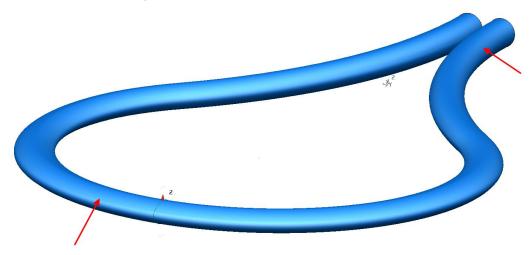
8 Create a **single composite curve** out of the **main drive-curve** and **upper cross section entities**.

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.
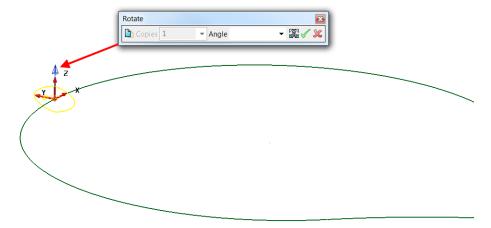
11 Click **OK** to accept

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**.
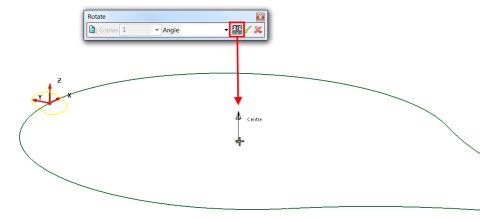
12 **Select** and **delete** only the **surface**.

13 Select the **composite curve section** at the top of the racket and select the **Z** principal plane.

14 Select **Rotate items** [icon] from the **General Edits** toolbar. [icon]

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** [icon] from the Rotate toolbar then snap to the centre key point of the circle racket as shown.
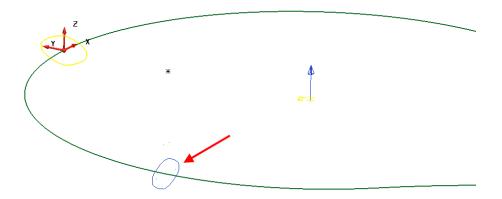
The **Rotational axis** has moved to the centre of the racket.
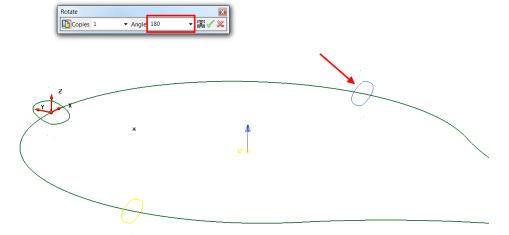
16 Select Copy the selected item button, enter **Angle** as **90** then Enter.
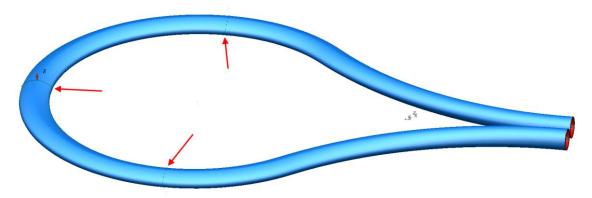
A copy of the original section is previewed.

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.
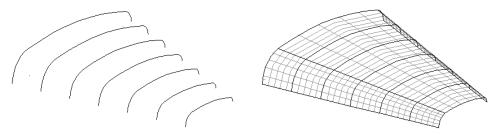
The elliptical shape is now constant between the three sections indicated.

21 Select **File > Save As**:

.....\PowerSHAPE-Models\surface racket

22 Select **File > Close**.

Surface from separate curves
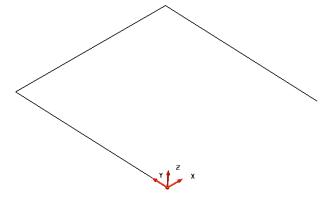
If separate curves are selected, the **Smart Surfacer** uses the **Create a surface from separate curves** option.

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.
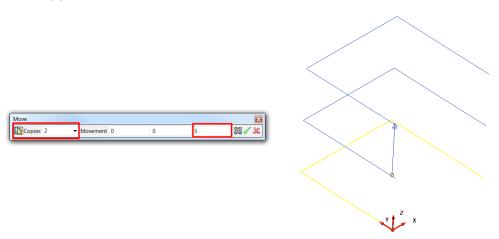
This **section** is the **basis** for the **other sections**, so the next step is to **copy** it up the **Z-axis** as individual lines:

3 Select the three lines, then **Move**/Copy items from the **General Edits** toolbar.
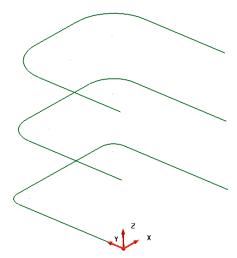
4 Enter **2** for the **No.```` of copies** and the value of **0 0 5** in the position window.

5 Confirm the preview with the green tick.
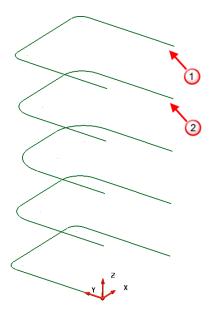
Each **composite curve** will be **filleted** by a different radius.

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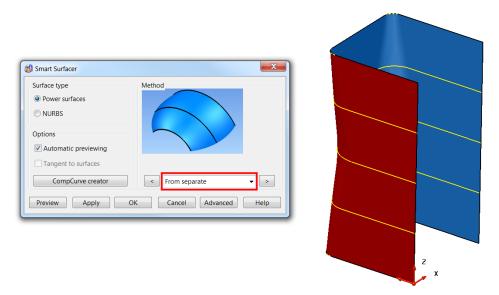
All of the shapes have been turned into **composite curves**, ready for **surface generation**. Each of the composite curves will be turned into a **lateral** on the surface.

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**.
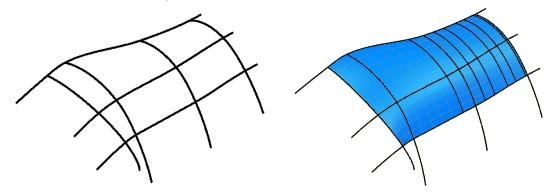
14 Select **OK** to create the surface from separate curves**.**

15 **Close** the file without saving.

Surface From Network of Curves
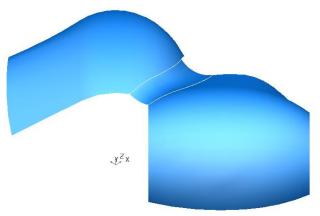
Surface from Network creates **one or more** four-sided surface patches over a network of wireframe. Surface definition will not occur on any open ended parts of wireframe that extend beyond the network.

Surface Gap Example
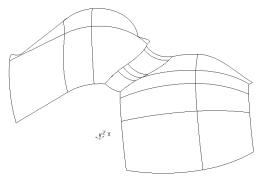
The following example uses Automatic surfacing and applies a Surface From Network to blend across a significant gap in the model.

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.
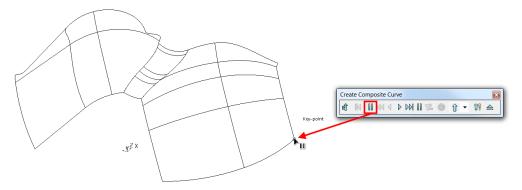
5   Select Create Composite Curve.
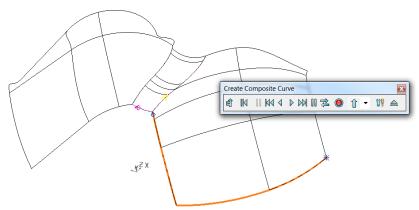
The Composite Curve toolbar is displayed.

6   Select **Define Start Point**, then click on the key-point on the corner of the surface shown.
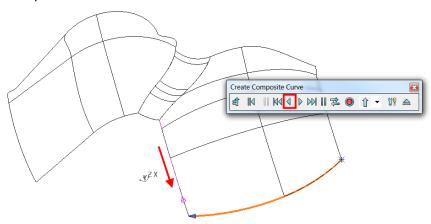
7   Select the bottom edge with the left mouse button to create the first part of the composite curve.
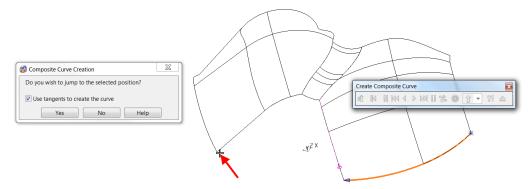
The curve continues to the first branch point awaiting further interaction from the user. In this case, too far.

8   On the **Composite Curve** toolbar, select **Backwards** once to go back one span.
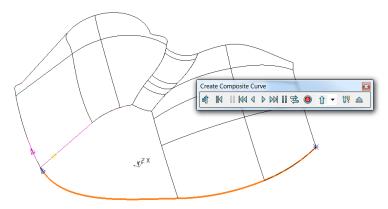
9   **Left-click** on the opposite corner indicated by the **red arrow below**.
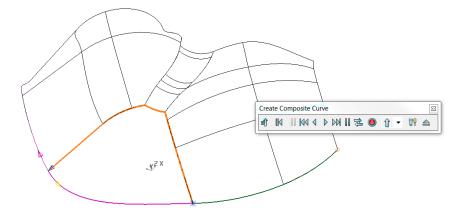
A dialog requests if the composite curve jump to this position?

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.
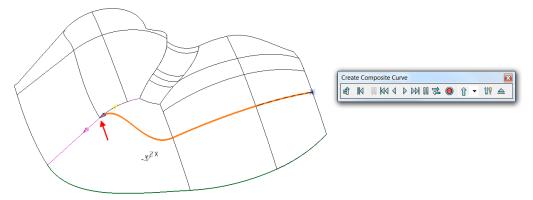
When a **composite curve** is created along a **surface edge**, it inherits the **tangencies of the surface**. These are reproduced if the curve is used to create a new surface from appropriate **Smart Surfacer** options.

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.
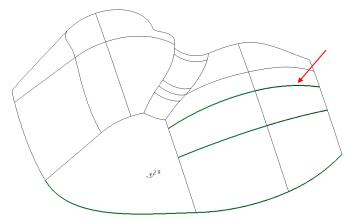
13 Select and **Blank** the last composite curve.

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).
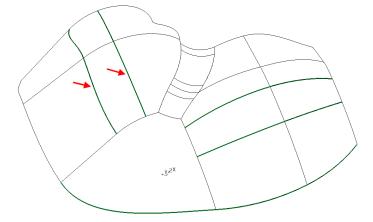
The end of the composite curve is aligning to the surface edge and not, as required, along the intermediate curve!

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.
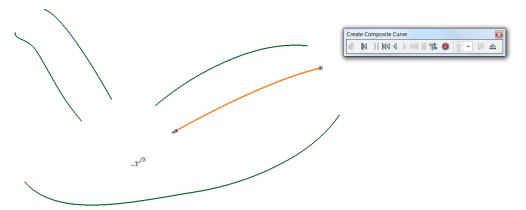
18 Create **two further composite curves** along the corresponding surface curves on the left surface as shown below.
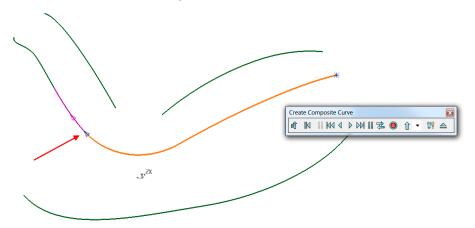
19 Select and **Blank** all surfaces on the model.
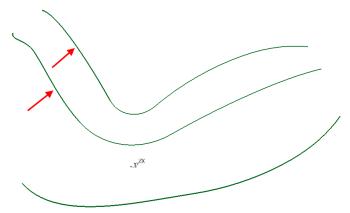
20 Create a **new composite curve** with the start point at the start of the existing composite curve as shown.
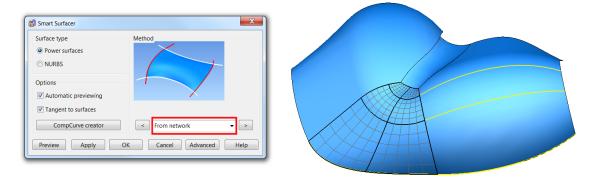
21 Click on the curve to include it as part of a new one and then click on the continuation at the end of the corresponding curve across the gap.

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.
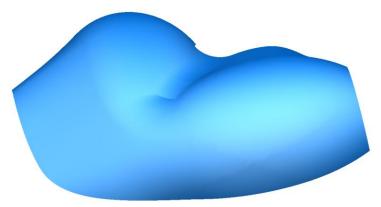
26 **Unblank (CTRL+L)** to bring back the surfaces and composite curve created earlier.

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.
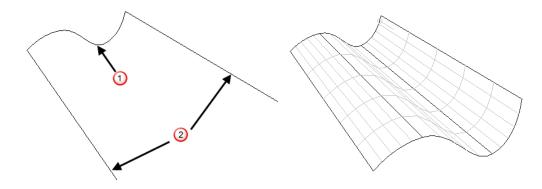
The **gap** in the model is **filled** in with the **new surface** which **flows smoothly** into the **adjoining surfaces**.

31 Select **File > Save As**:

   .....\PowerSHAPE Data\network curve example.psmodel

32 Select **File > Close**.

Surface from Two Rails
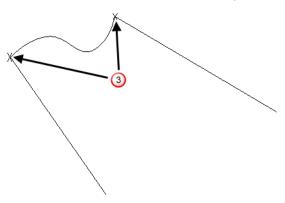
A surface can be generated from a section ① that is scaled along two drive rails ②, defining the edge of the surface.

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
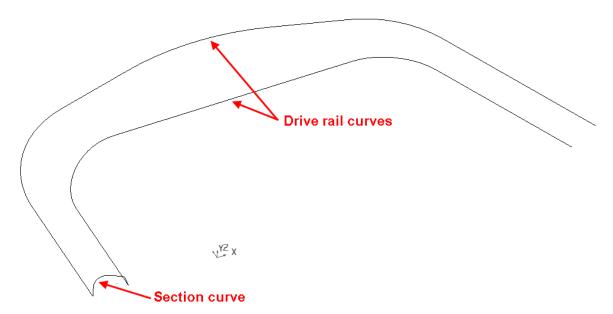
positioned exactly on the start point of each drive rail ③. Both the section and the drive rails must be defined as composite curves.

Surface from Two Rails Example

1   Import the model:
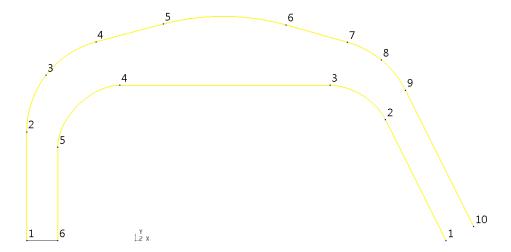
.....\PowerSHAPE_data**two_rail_data.dgk.**

2   Select one of the drive rail curves to display the **Curve Editing toolbar.
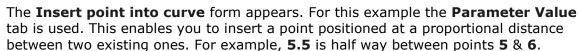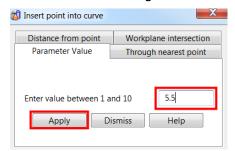
**3   Shift-select the other drive rail curve.

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**.
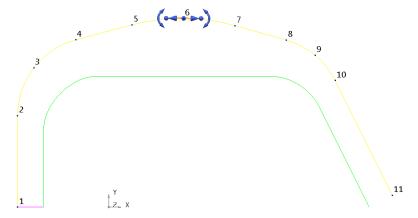
7   Select the **Parameter** tab and enter **5.5**. Select **Apply** and then Dismiss.

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.
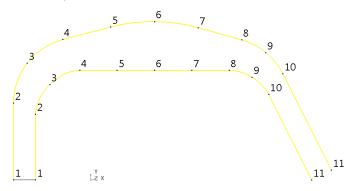
9   With the lower curve still selected, select **Create a point.**   .
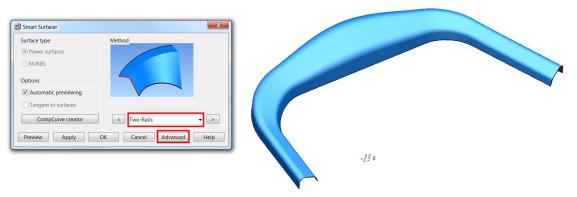
10 Select the **Parameter** tab and enter **4.5** then **Apply**.

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.
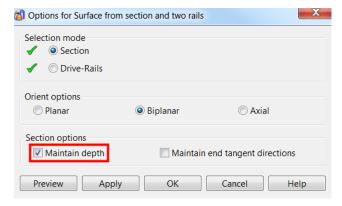
16 Select all three composite curves then **Smart Surfacer**.

The Wizard recognises that a **Two Rails** surface is the best choice for surface creation.
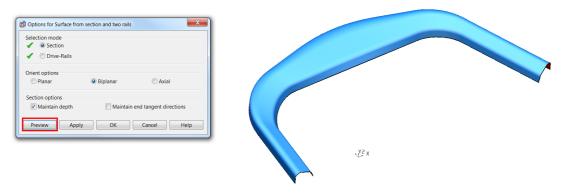
However, the **section height** has **scaled** in proportion to the distance between the drive-curves at each section position.

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.
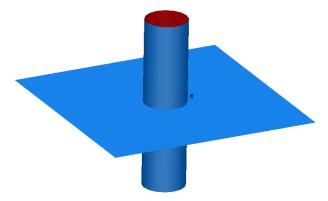
19 Select **OK** twice to accept.

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**.
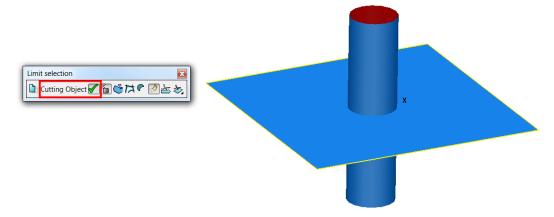
The cylinder will be limited by the plane.

4 Select **Limit selection** from the **General Edits** toolbar.
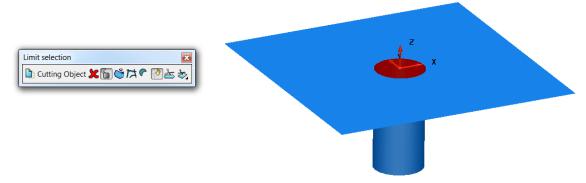
5   Select the Plane surface.
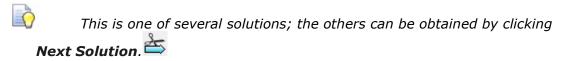
The Plane has been assigned as the cutting object.
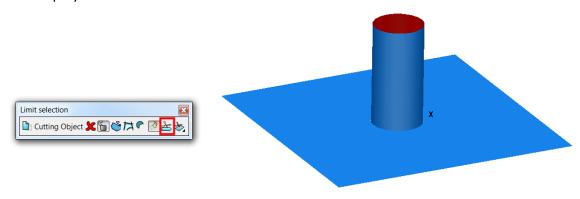
6   Click anywhere on the top side of the cylinder surface.

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.
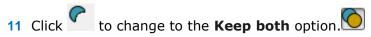
9   Select **Undo**.

10 Select the plane surface first and then Select **Limit selection.**
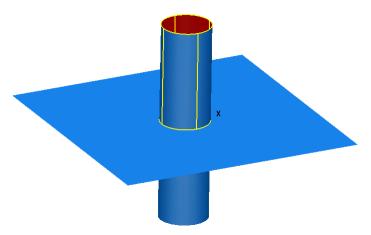
By selecting a surface first, the form opens with it assigned as the cutting object.

11 Click [icon] to change to the **Keep both** option.

12 Select the cylinder surface.
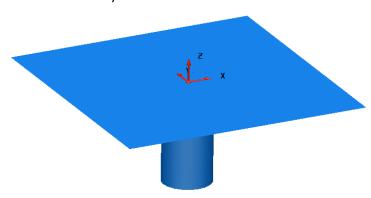
*When **Keep both** is active, **Limit selection** splits the secondary surface selection into two surfaces, one on each side of the Cutting Object.*

13 As before, close the form [icon] then Undo.

14 Select the plane surface first and then Select **Limit selection.**

15 Click [icon] 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.
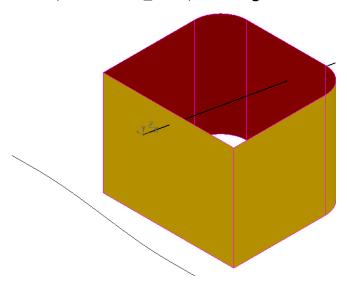
Limit Surface using a Curve as the Cutting Object

A **wireframe entity** can also be used as the **Cutting object**, This as shown in the following example.
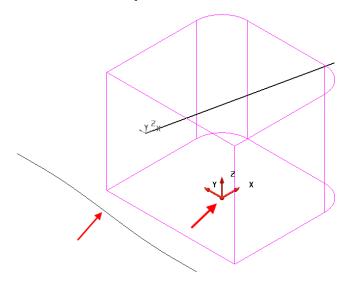
1   Open a **new model** and **import** the following data;

.....\PowerSHAPE_Data**Limit2.dgk**
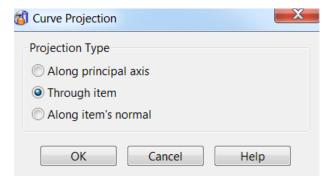
The imported model contains a component made up of six separated surfaces and some wireframes defining the 2D trimming requirements.
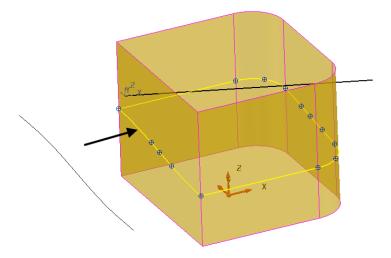
2   Activate **Workplane 1** located at the base of the component.

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.
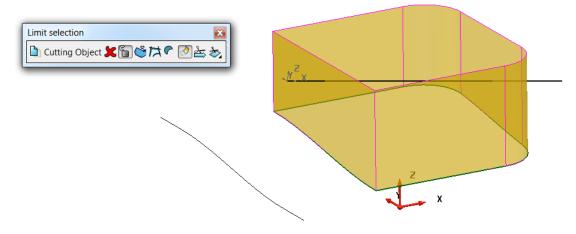
8   Select the new composite curve then **Limit selection.**

> 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**.
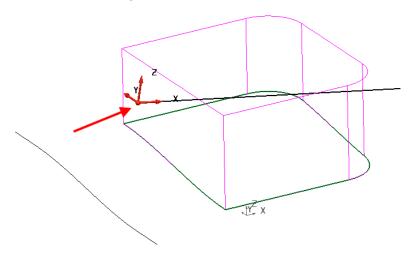
10 Close the form.

Limit Surface using a Workplane as the cutting object

A **Workplane** can also be used as the **cutting object** as shown in this example.
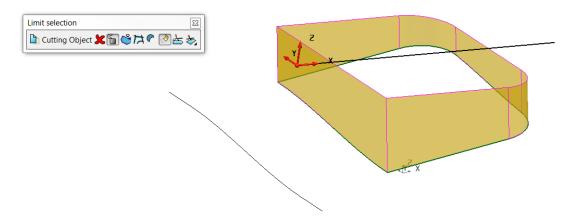
1   Activate **Workplane 2** as shown.

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.
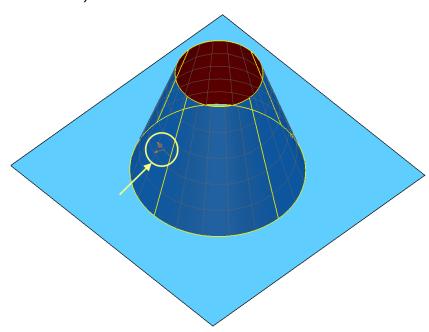
A fillet surface is the developed outside contour of a ball if rolled between two or more surfaces. **PowerSHAPE** can create both fixed and variable radius fillets between surfaces. Where the ball makes contact defines the outer edges of the fillet to where, by default the original surfaces are trimmed 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.
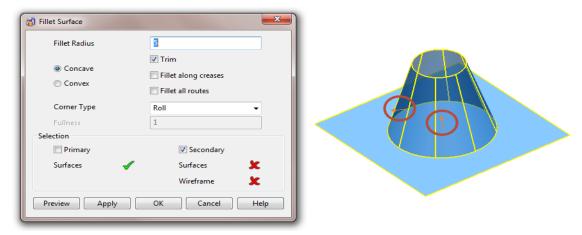
The small arrow points outwards from the surface, indicating that this is the outside edge of the surface. The large arrow indicates the operational direction for editing surface points

The large arrow does not affect filleting.
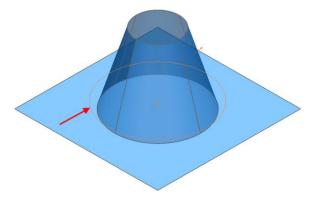
5 Select both surfaces and switch to **Transparent Shading**.

6 Select **Surface fillet** from the **Surface** toolbar.
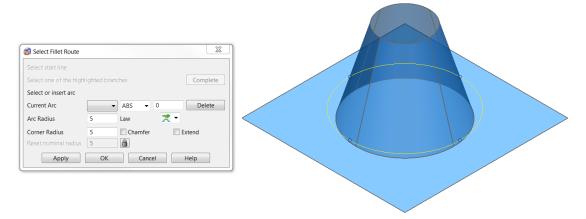
The surface direction arrows are displayed. If clicked, these turn the surface inside out. The small arrow points away from the outside (user-defined colour) face of surface.

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.
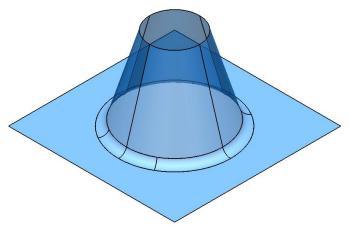
The Select Fillet Route dialog is displayed.

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.
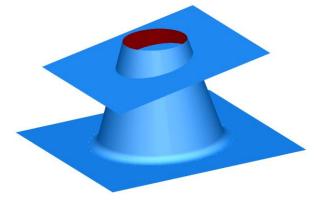
For examples with more than one track, no track is pre-selected. Select each one and Apply in turn, until all the required fillets have been created. Then, select OK to finish.

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.
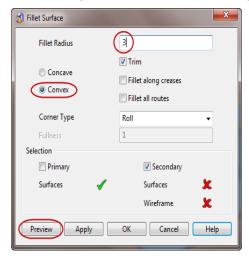
This top surface will be filleted to the cone, but this time the fillet is on the inside of the cone.
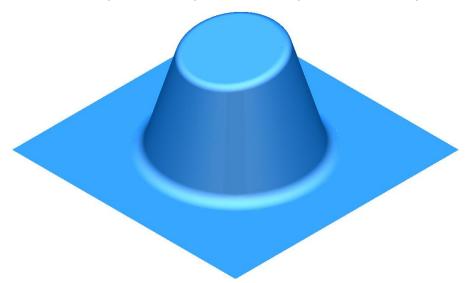
12 Select just the cone and the angled plane surface, then **Surface fillet**.

13 Enter a **Fillet Radius** of **3** mm, select **Convex** and select **Preview**.

14 Select **OK** (fillet surface) and then **OK** (Select fillet route).
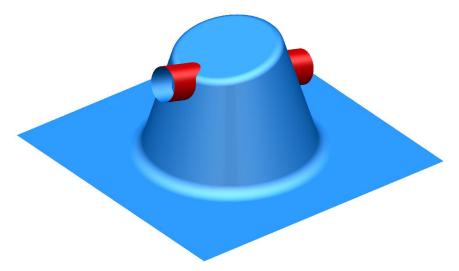
The fillet is produced trimming back the top of the cone and the plane.
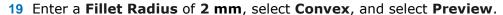
Fillets can also be produced with several surfaces.
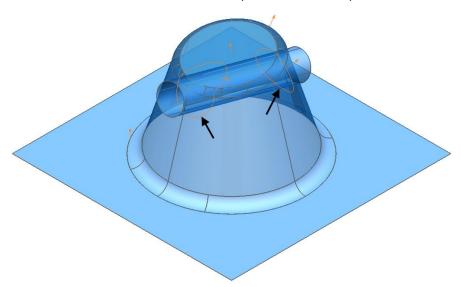
15 Select **X** as the **operational axis** and create a Cylinder Primitive positioned at **–30 0 20**.

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.
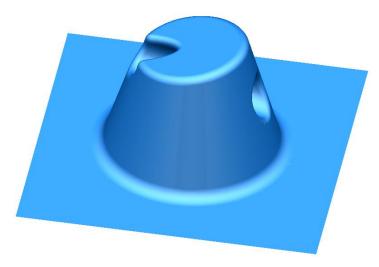
20 Select **OK**.

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.
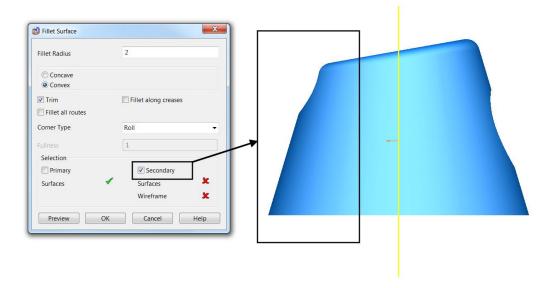
4   Select the Primitive plane surface then **Surface fillet**.
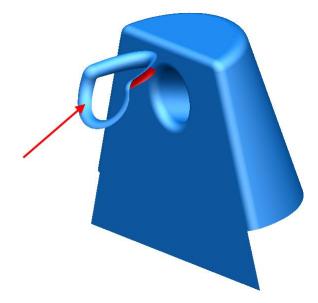
This surface is now assigned as the primary surface.

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.
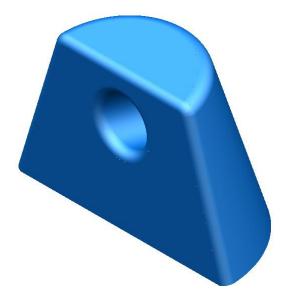
5 Drag a box over the surfaces on the left as shown, to assign as the secondary set.

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**:
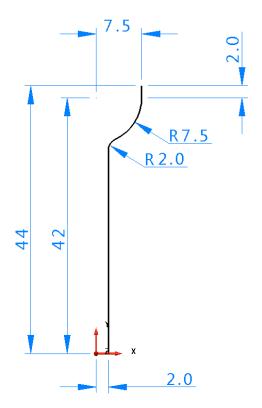
.....\PowerSHAPE-Models\fillet example.psmodel

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**.
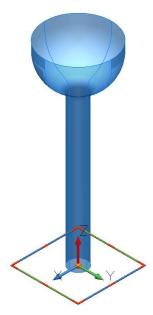
1 **Create a New Model and Workplane at 0.**

2 Generate the following wireframe on the XY Plane.

3   Create a **surface of revolution** around the **Y axis**.
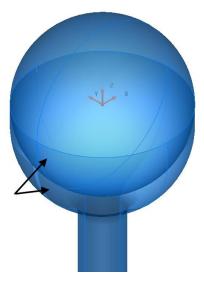
4   **Rotate** the new surface **90** degrees about the **X Axis** to align it to the **Z** direction.
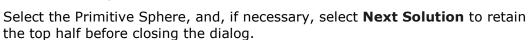
5   Switch the operational axis to **Z.**

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.
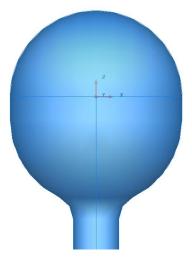
8   Select the workplane, then **Limit selection.**

9   Select the Primitive Sphere, and, if necessary, select **Next Solution** to retain the top half before closing the dialog.
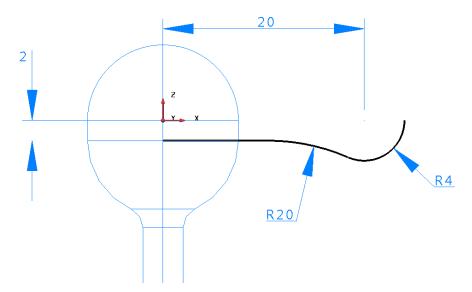
> *Ensure the correct operation axis is selected first! (Limiting occurs in the direction of the active Principal Plane)*
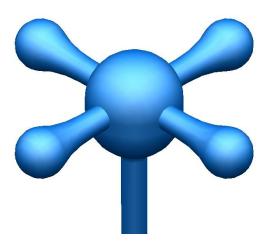
The Primitive Sphere is limited back to the workplane with the top half being retained

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.
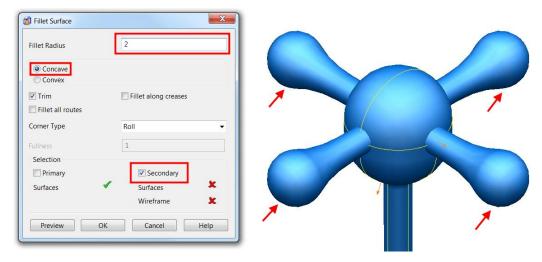
The **four arms** need to be **filleted** to the **central section**. Make sure that when **shaded**, all the **surfaces** are **positive**. We do not want to **fillet** each arm to each other or to generate the route, so we are going to use the Secondary surfaces option.
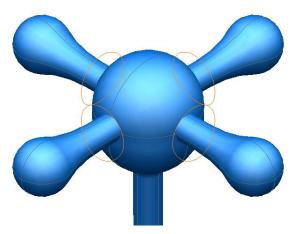
13 Select both the shaft and sphere surfaces then **Surface fillet**.

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**.
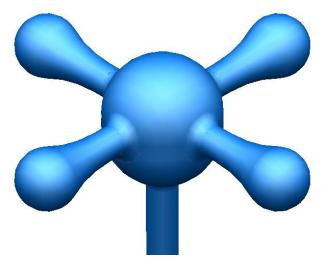
The **fillet tracks** are **displayed**.

17 **Select** and **Apply** each one to complete the exercise.
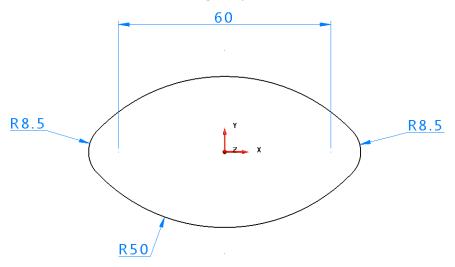
18 Select **File > Save As**:

.....\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
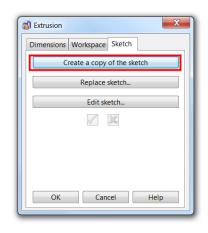
20 Select **Create New Model** and workplane at **0**.

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.
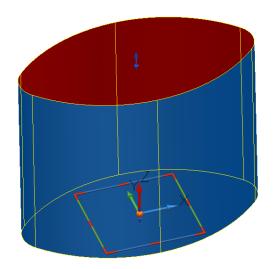
23 Create a Composite Curve from the wireframe.

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).**
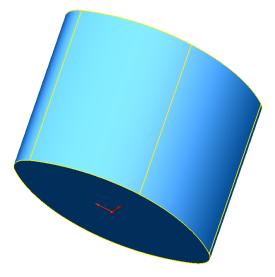
Alternatively, the individual wireframe for an extrusion can be retrieved directly.

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.
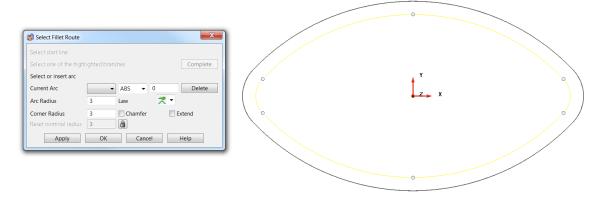
28 Delete the composite curve.

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.
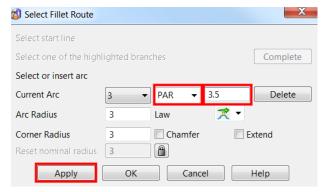
The **fillet track** is displayed (in **yellow**) with circles (in **blue**) showing **Key points** that can be **snapped** to using the mouse. By snapping at these **Key points**, an **arc** of the **initial radius** is created. The **value** of this **radius** can be **modified** in the **Arc Radius** 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.
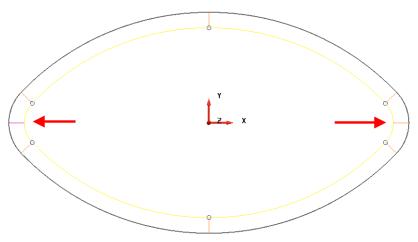
A **radius** appears for each point and this becomes numbered in the current arc part of the form. Individual arcs can be selected from this form or by manually clicking on them. The radius can be changed in the form or dragged manually.

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**.
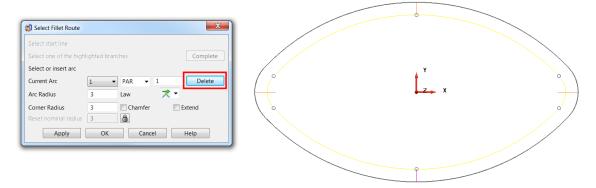
34 Select the correct arcs (displayed in blue) and insert two new radii between the two at each side, as shown.
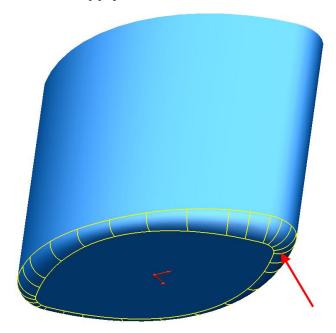
While the form is **active**, the values for **individual fillet arcs** can be **edited** or **deleted**. The **radius progressively changes** between the **two arcs** assigned with **different values**.

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.

7. 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
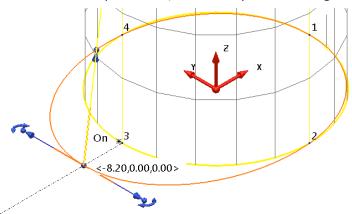
Editing **curves** and **points** within a **surface** use the **same methods** as **wireframe curves**.

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.
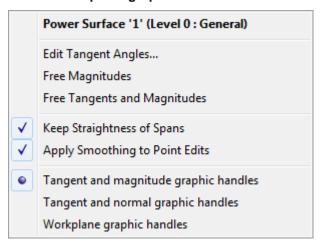
Single Surface Curve Point Edits

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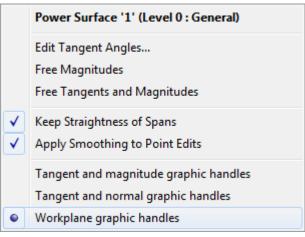
Again, the central **spherical handle** at the centre will move the point **freely** in **3D space** while showing **X**, **Y**, **Z** coordinates offset from its original location.

Workplane graphic handles
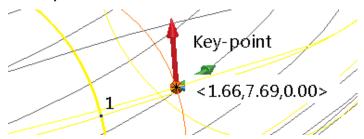
2   Select a **point** on the curve and right-click and ensure that **Tangent and Normal graphic handles** is selected in the menu.
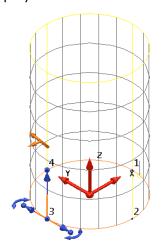
Clicking and dragging the workplane axis arrow handles will move the point in a fixed axial (X, Y or Z) direction with a visible offset distance from the original position, with the option to further edit or fine tune the offset once the mouse button has been released. The workplane origin in this case allows the point to be moved freely in 3D space.
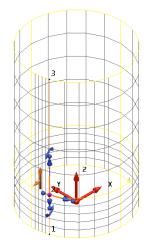
Graphically inserting a curve on a surface

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.
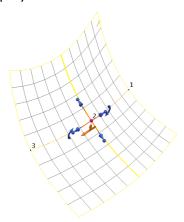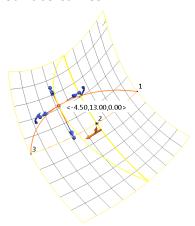
If you release the mouse over a lateral, only a longitudinal is inserted. If you release over a longitudinal, only a lateral is inserted. If you release the mouse at a position not on the surface, then no curve is inserted.

Graphically moving a point along a surface curve
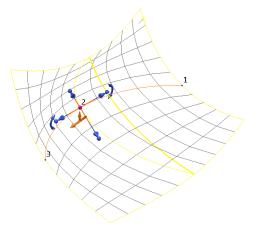
You can adjust the position of a point along its adjacent curves.

When any point on a surface is selected, the handles of the point will be displayed.①

- 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.
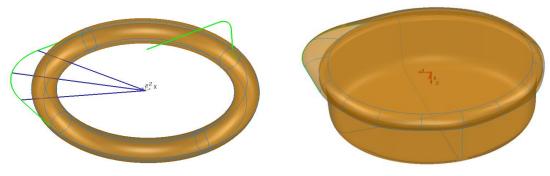
Use the following table to identify the buttons on the toolbar.

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 .
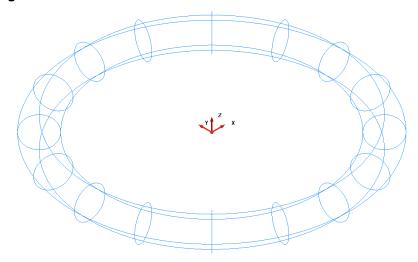
3 Make **Workplane 1 active**.

> *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.***
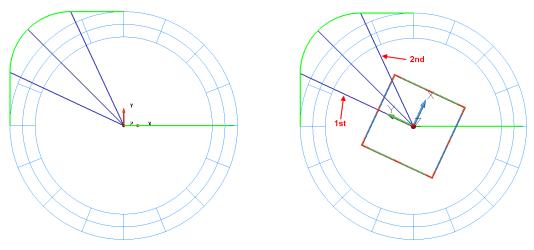
With the default setting above, a **Power Surface** has been created with all of the surface editing toolbar options available.
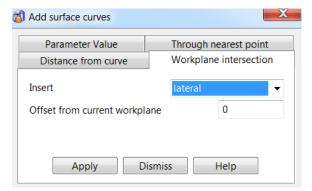
5   Select **View From Top (+Z)**.

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.**
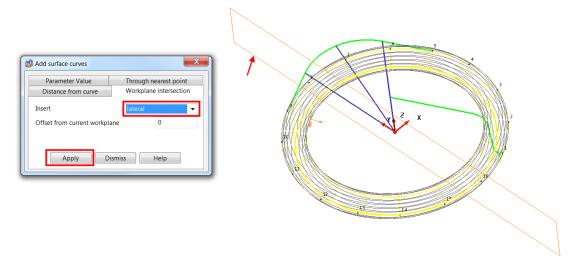
Four methods are available to add laterals or longitudinal curves.

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.
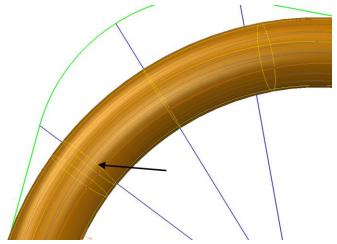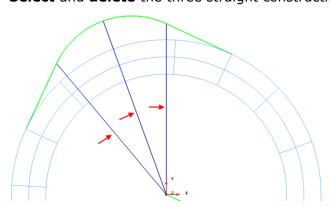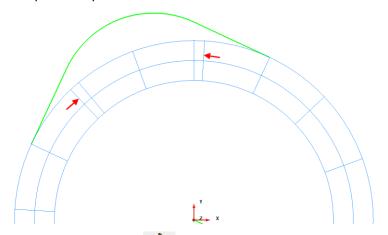
Image shaded here to show new curve generated.

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.
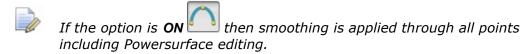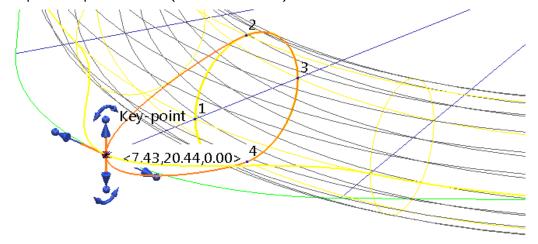
11 Select the two original **laterals** that are **not aligned** to key points on the lip-shaped composite curve.

12 Select **Delete** curve ![icon] from the **Surface/curve editing** toolbar.

13 Select any curve and ensure **smoothing** if **OFF** ![icon] from the **curve editing** toolbar.

![note icon] *If the option is **ON** ![icon] then smoothing is applied through all points including Powersurface editing.*
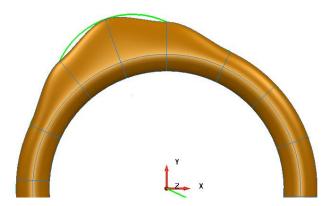
14 **Select** the **middle surface point** on the **outer longitudinal** in the **top-right quadrant** ,and **dynamically drag** it to the corresponding keypoint on the new lip-shaped composite curve (as shown below).

15 **Repeat** the **drag** move on the **adjacent** surface points to create the following.
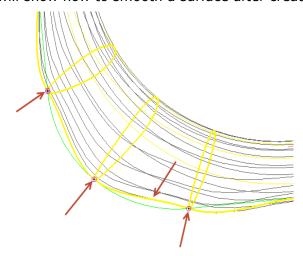
![note icon] *Ensure a **keypoint (End)** is **selected** on the **compositive curve**.*
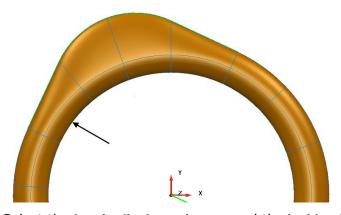
The **tangency** and **magnitude** through the **3 points** requires freeing up to achieve the more **natural transition** through the points. If **smoothing** was **ON**

 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**.
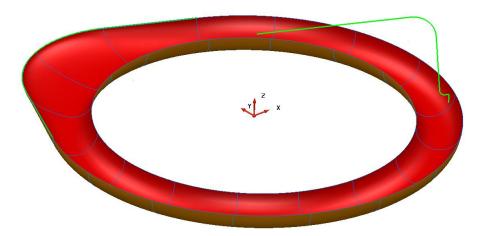
20 Select the **longitudinal** running around the **inside** of the rim (indicated above)
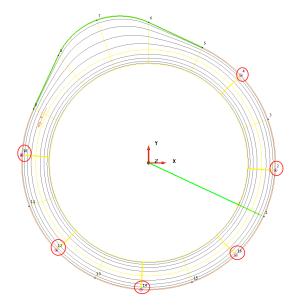
21 Select **Break surface** to split the surface into two separate upper and lower halves.

22 Select and **delete** the **upper surface**.
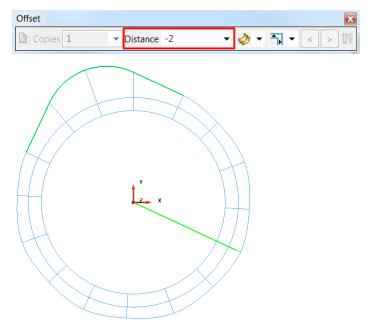
23 Select a **view from top** (down **Z**), then select the alternate six surface points around the outside of the lid as shown below.

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.
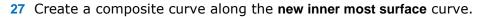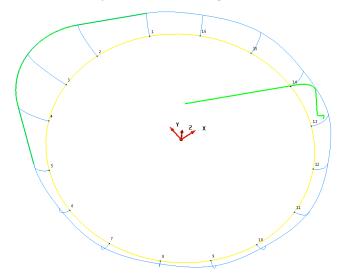
24 Select **Edit sub items** from the **General Edits** Toolbar.

25 Select **Offset** , enter a value -**2** and press the **Enter** key.
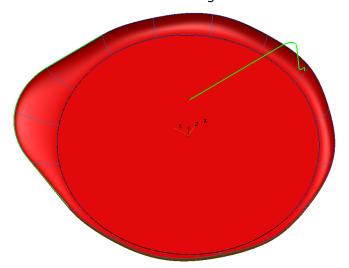
The selected surface points have been offset inwards, normal to the surface.

26 Select and **Delete** the inner most surface **longitudinal** curve.

27 Create a composite curve along the **new inner most surface** curve.
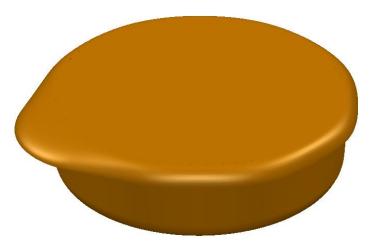
28 Generate a **Fill in** surface using the **Smart Surfacer**.

29 Use the remaining composite curve defining the outer profile, and generate a

surface of revolution.
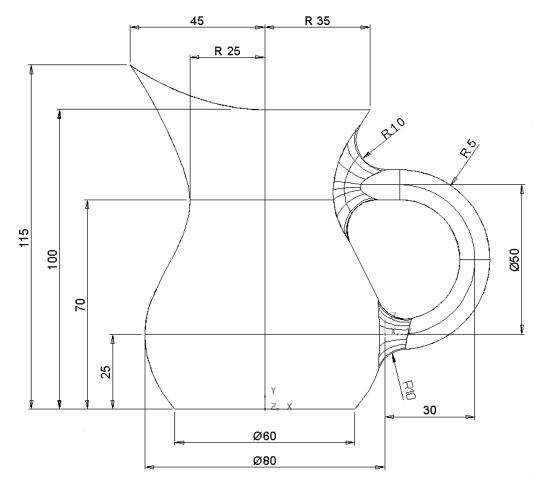
30 **Save** the model as

.....\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.
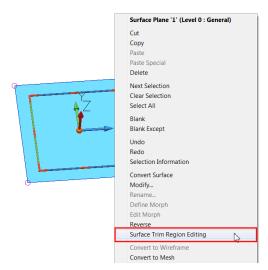
PowerSHAPE provides powerful tools to help you define and edit your trim region accurately.

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.
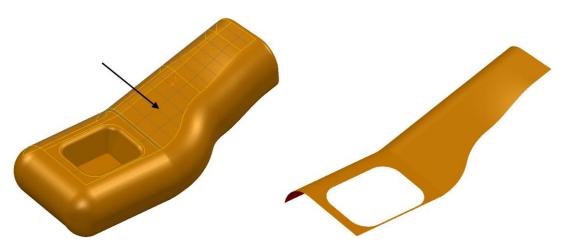
Deletes the selected ppoint on a pcurve.

The following examples will help demonstrate the above.

Switch housing

1 **Open** the model

.....\PowerSHAPE_Data**switch_housing.psmodel**
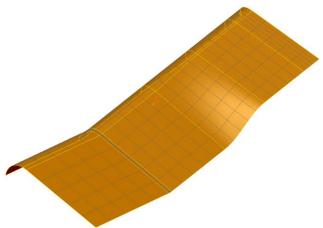
2 Select the upper surface and then **Blank Except**.

3 Select the surface to display the **Surface Edits** toolbar.

By default, trimming **ON** displays the surfaces in its trimmed state. To view
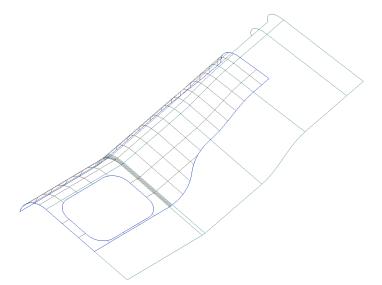
a selected surface untrimmed, toggle this icon to Trimming **OFF**.

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.
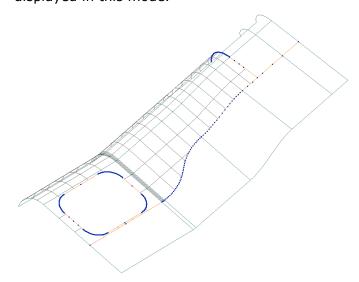
If a boundary is deleted 📋 *from a surface, the associated pcurves remain.*

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.**
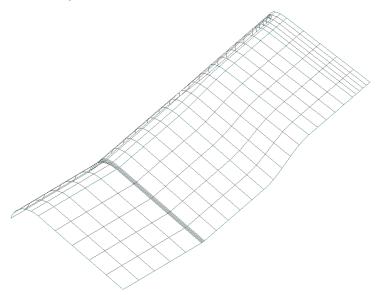
When a **boundary** is highlighted in the form, it is displayed in **orange** on the surface.

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.
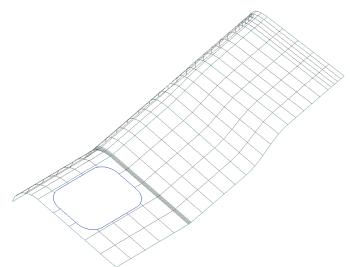
*The pcurves still exist and can be displayed by toggling into **Pcurve Edits**.*

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.
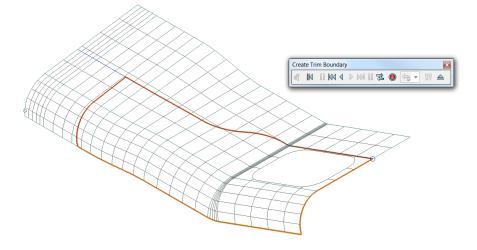
13 Select **Recreate** to open the toolbar and display all unused pcurves on the surface.

The **Create Trim Boundary** toolbar is displayed. Similar to Composite curve creator, this is used to trace along the pcurves to define a boundary.
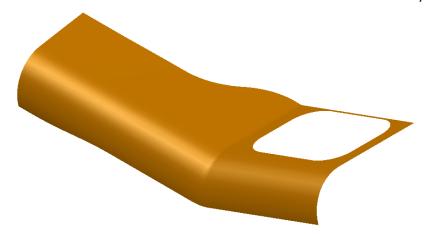
All of the pcurves are displayed on the surface and are available for selection. To form a boundary, the **pcurve** must form a **closed route**. The edges of a surface can also be used as part of the route of a boundary.

14 Click on a pcurve along the require route and sequentially, click on the direction arrows to produce the required closed boundary.
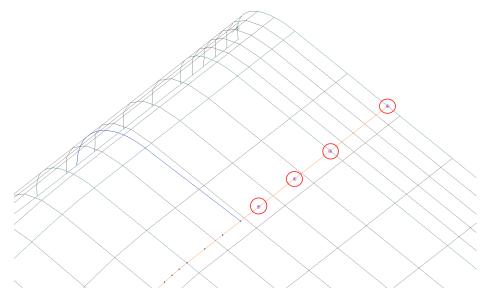
15 Select **Save** and then select **Eject.**

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.
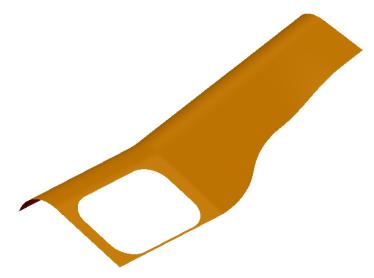
When a pcurve is selected, the ppoints are displayed. These can be labelled, selected, repositioned or and deleted, as required.

5 Shift-Select the four extra ppoints circled above.

6 Click **Delete Point**.

7 Switch back to **Boundary Edit Mode** then **Autocreate.**
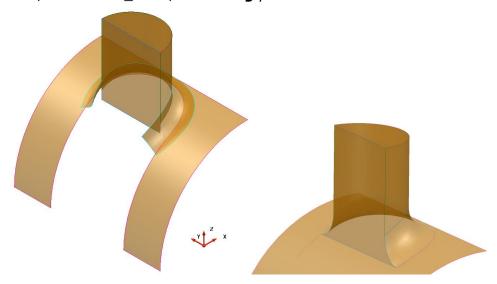
Removing the unused ppoints enables future automatic trimming operations to immediately produce the desired results and not stray along a path defined by redundant pcurve data.

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.
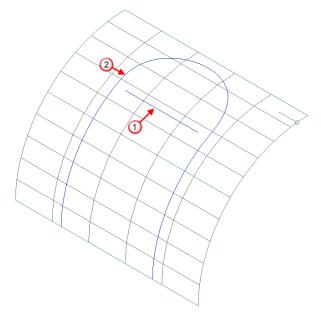
The more complex the network of pcurves on a surface, the less likely it is to obtain a trimmed surface automatically. The following worked example shows the stages of pcurve and boundary editing required to produce the correctly trimmed model shown above right.

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**).
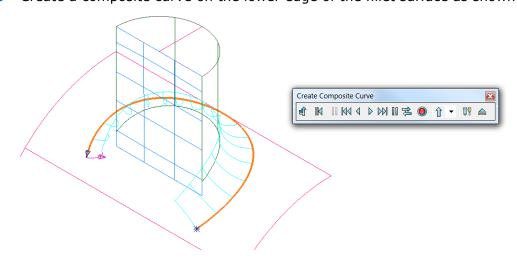
Creating a pcurve from a composite curve

Pcurves can also **be created** from standard **wireframe** and **composite curves**.
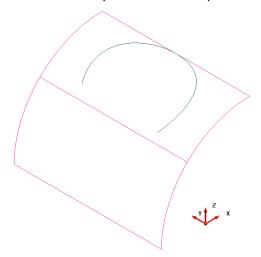
In this case, a **composite curve** will be created from the lower edge of the fillet. This new **composite curve** is then turned into a **pcurve** and projected onto the original curved surface to help trace a new boundary.

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.
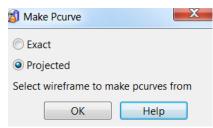
10 Select the curved surface and switch to **Pcurve Edit Mode**.
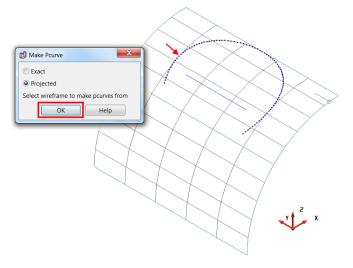
11 Select **Make pcurves from projecting wireframe**.
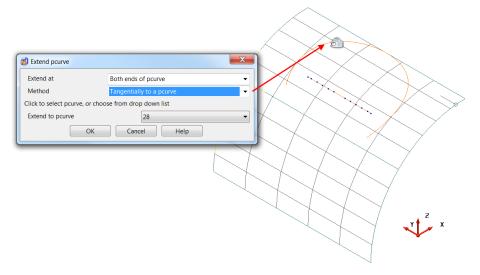
12 Select **Projected** then the composite curve. Select **OK**.
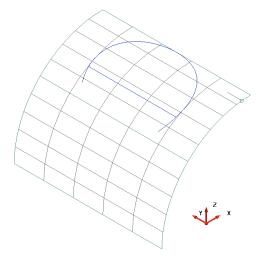
13 **Deselect** the **surface** and **delete** the **composite curve**.

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**.
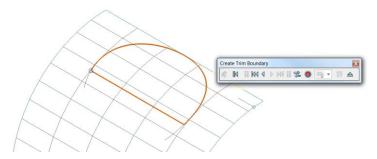
 The straight pcurve has extended to effectively create a closed boundary.

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.
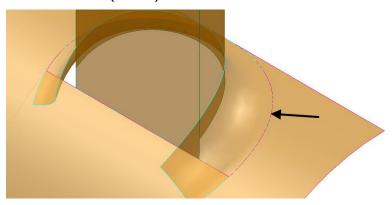
*If the trimmed surface appears within the D-shape, select **Reverse the***

*   **Boundary** 🔲 *to create the result shown.*

3   Select **Unblank** (**Ctrl+L**).
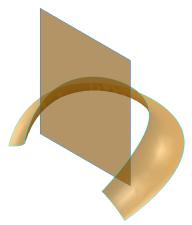
The new boundary has removed the overlap between the fillet and curved surface which previously existed.

4   **Blank** all surfaces except the front vertical and fillet surfaces.
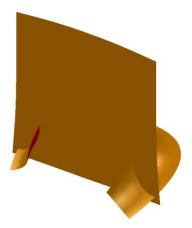
Trimming the planar surface

The planar surface needs to be correctly trimmed with the fillet.
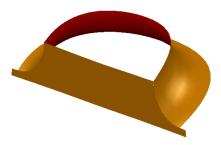
1   **Explode** 🗒 all boundaries on the **vertical surface**.
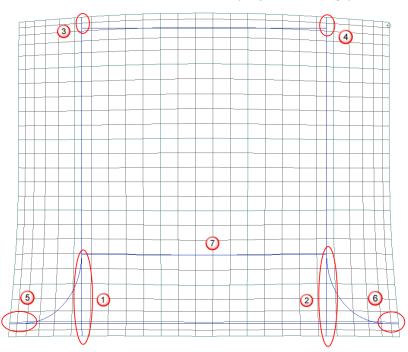
The vertical surface is untrimmed to its original size and will be limited to the fillet surface.

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.
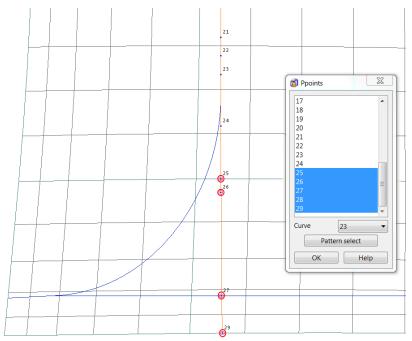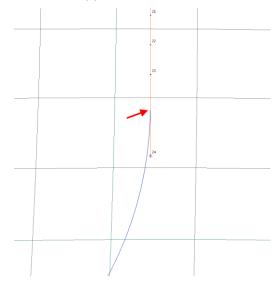
If required, a new boundary can simply be **recreated** to include the vertical surface.

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
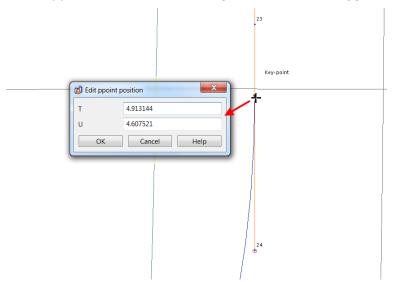
Point 24 was not deleted. This will be moved up to the start of the fillet arc.
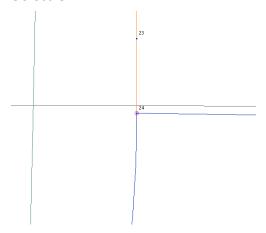
11 Select ppoint 24 and select **Edit parametric value at ppoint**.

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.
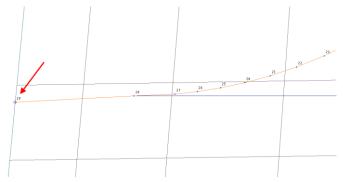
14 Repeat the above for the opposite side. ②

15 Zoom into the bottom left corner region. ⑤
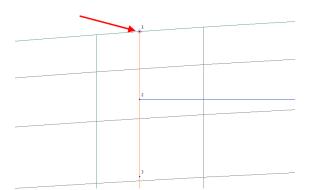
16 **Delete** just point 29 on the fillet pcurve.
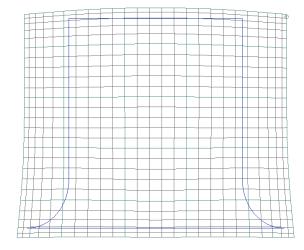
The pcurve is trimmed back to point 28 which is also the intersection point with the adjoining pcurve.

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
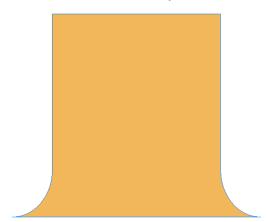
A single pcurve path remains.

21 Switch back to **Boundary Edit Mode** then **Autocreate**.
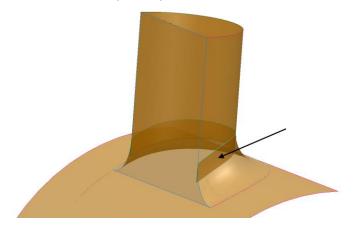
A single clean surface is generated.
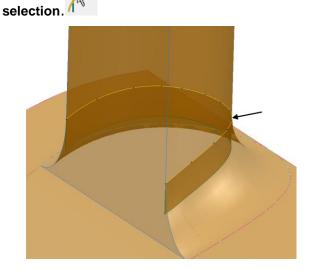
22 Select **Unblank** (**Ctrl+L**).
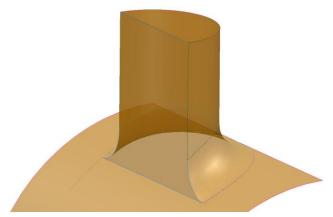
A final problem remains indicated by the arrow above. The vertical D section surface requires trimming back to the top edge of the fillet surface. An overlap can be clearly seen in transparent view.

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
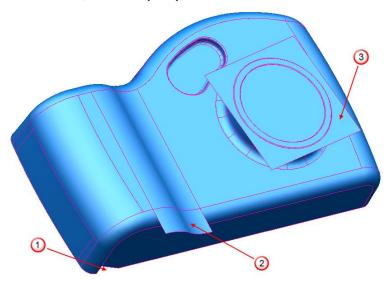
Exercise 9: Region trim camera

1 **Open** the model

.....\PowerSHAPE_Data**camera repair.psmodel**

2 Use Trim Region editing <u>ONLY</u> * 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!! ✨

8. Direct Solid Modelling

Introduction

Direct Solid Modelling provides tools to:
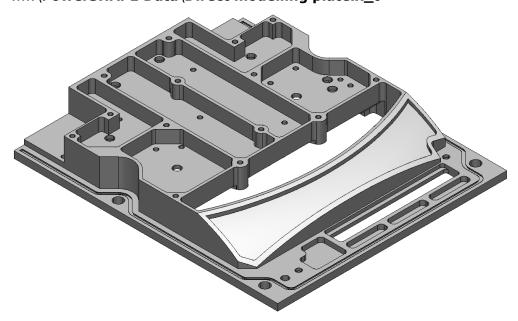
- Make **rapid edits** to **history-free solids**.

- **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**.
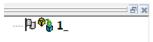
- **Convert** between **Surfaces** and **Parasolid**.

- **Convert** between **Mesh** and **Parasolid**.

3 Select [icon] 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** [icon] can be applied.

- **Move** - [icon]

- **Rotate** - [icon]

- **Mirror** - [icon]

- **Offset** - [icon]

- **Scale** - [icon]
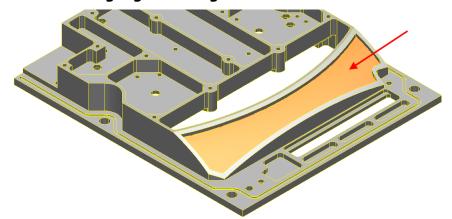
4   **Select** and **highlight** the **large curved face** shown in the **next image**.
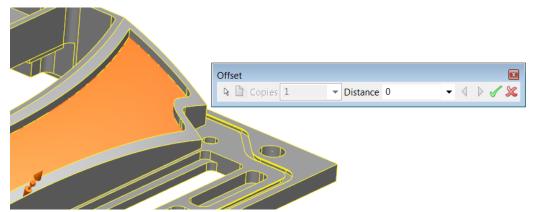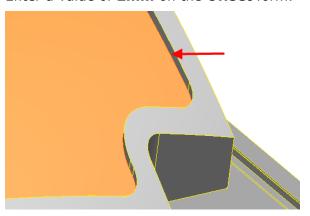
   *By default, the surface select mode is **Select individual faces**.*

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

6   Enter a value of **2mm** on the **Offset** form.
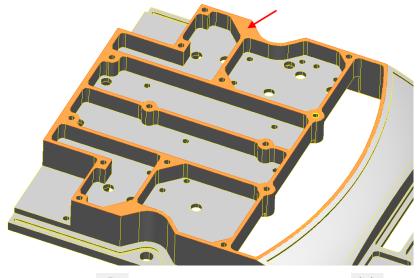
The **face** has been **offset**. More importantly however, **all surrounding vertical faces** have been **retrimmed** and **reintersected automatically**.
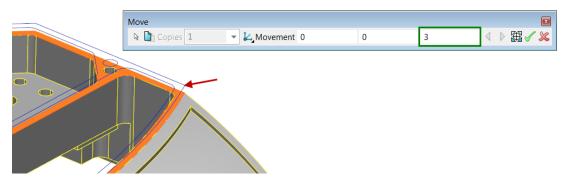
1 **Select** and **highlight** the **top horizontal face**.
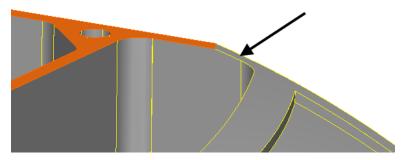
2 Select **Move** from the **General Edits** toolbar.

3 Enter a distance of **3mm** in the **Z axis** and press **Enter** (for preview).
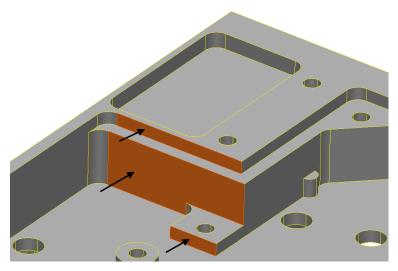
As expected, the **preview** of the calculated result is displayed.

4 Select the green tick to accept. Note the effect in the region shown above.
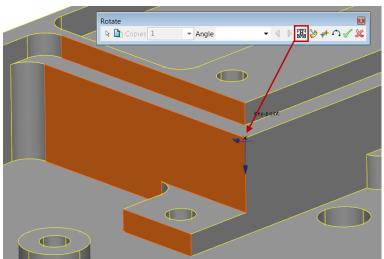
The top face has successfully moved by **3mm** in the **Z axis**. All related vertical faces have reintersected and maintained the solid closure. The sloping face achieved this by extending its faces.
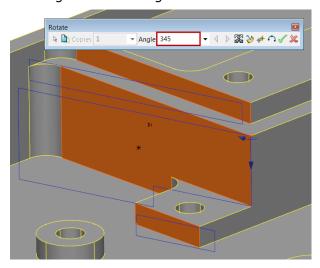
1 **Rotate** the part to view the bottom side and zoom into the region shown.

2 **Shift** select the three faces highlighted.

3 Select **Rotate** from the **General Edits** toolbar.
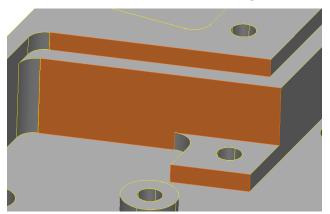
4 From the rotate form select **Reposition** and click on the **keypoint** to specify the rotate origin.

5 Select and drag only of the selected faces to dynamically preview the faces rotating about the origin.
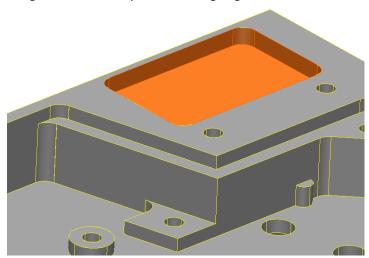
6 Release the mouse and enter an **angle of 345** before accepting the result. ✓

Only the selected faces have rotated to the specified angle.
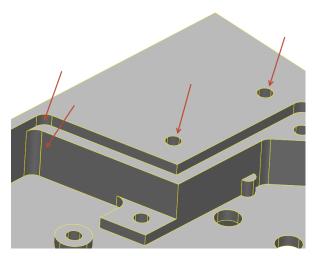
1 From the **solid edit** toolbar change the selection mode to faces of **concave regions**.

2 Single click in the pocket to highlight all relevant faces.

3 From the **Solid Editing** toolbar, Select **Remove and Heal**.
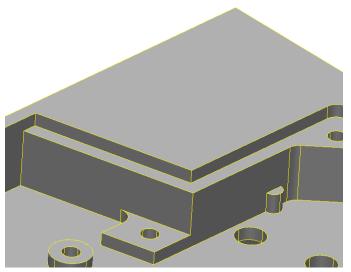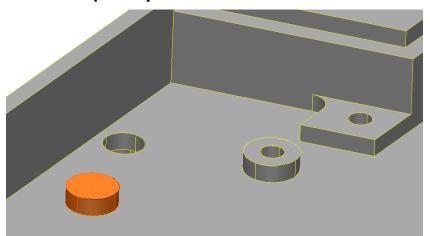
This tool **removes all selected faces** and then **closes** the **resulting gap** by **modifying** the **surrounding faces**.

4 Remove the **two holes** and **two fillets** (indicated in the **previous image**) using **Remove and heal**.
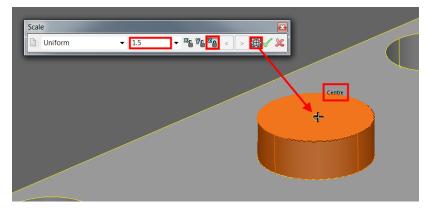
5 Select the **top** and **cylindrical faces** which define the **boss feature**.
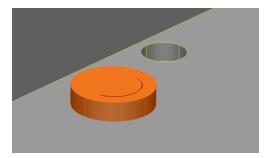
6 Select **Scale** from the **General Edits** toolbar.

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.
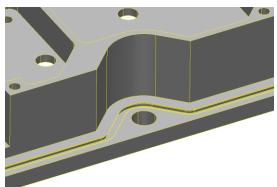
The boss feature has scaled in the **X** and **Y axis** by a **factor of 1.5**.
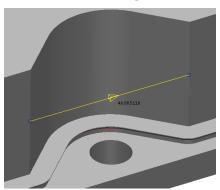
> *If the **Z axis** was **not locked PowerSHAPE** would still have automatically limited back the **Z intersection** and provided the same result.*

PowerSHAPE provides the ability to **divide** the **default faces**, thus providing **more control** over specific regions that require editing.
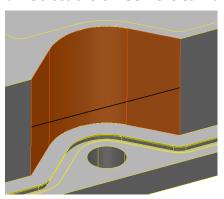
1   **Rotate** the model to view the top and zoom into the area show.
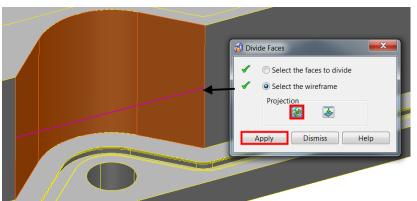
2   Create a **single line** approximately half way up the **concave face**, **snapping** the **outside edges**.
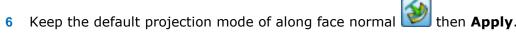
3   Select the **three vertical faces** which define the **concave recess**.
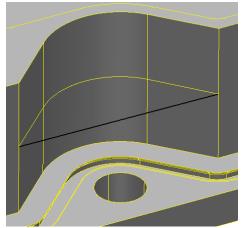
4   From the **Solid Editing** toolbar, Select **Divide faces**.

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.
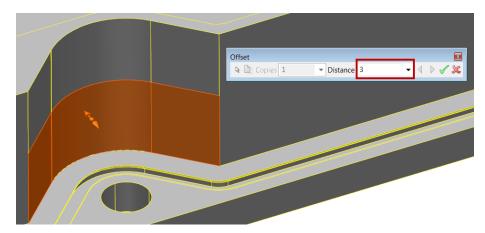
> *New to **PowerSHAPE 2014 R2** is the option to **merge faces*** *on a solid. This will merge **adjacent continuous faces** into one continuous face, essentially the **reverse** of the operation to **Divide Faces** described above.*
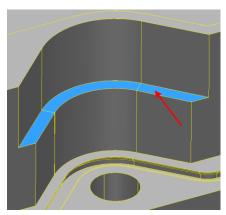
7   **Dismiss** the form and **delete** the single line.

8   Select the lower three faces then **Offset** from the **General Edits** toolbar.
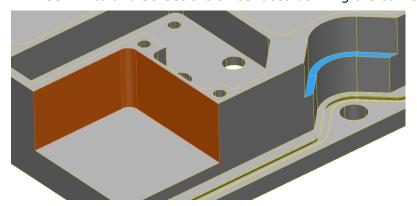
9 Enter an **Offset** distance of **3mm.**

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)**.
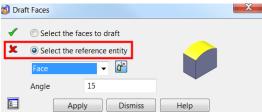
1 **Zoom** into and **select** the three faces defining the corner pocket

Direct modelling allows **draft angles** to be quickly applied to **selected faces**.
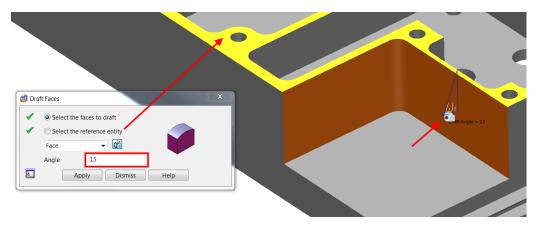
2 From the **Solid Editing** toolbar, Select **Draft faces.**

The red cross indicates a required reference entity. This can be a face, Surface or a Workplane.
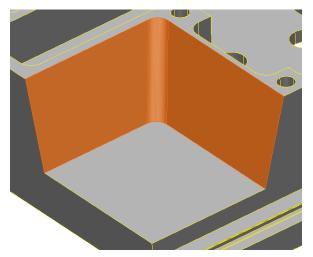
3 Leave the default **Face** entity and **select** the **top horizontal face**.
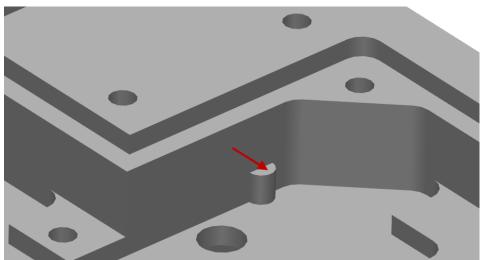
The **draft angle** can be **dynamically pulled** to **set** and preview the required angle.

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.
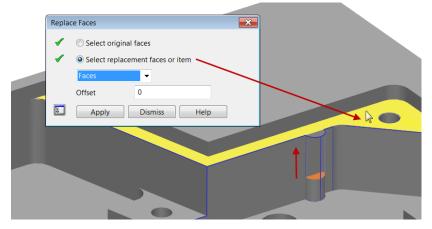
3   From the **Solid Editing** toolbar, Select **Replace faces.**
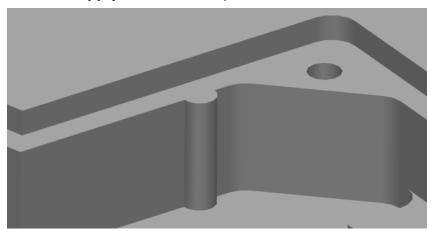
This allows faces of a solid to be replaced with existing faces (as a reference). In this example, the length of the reinforcement lug will be extended.

4   **Select** the **face** indicated to **nominate** as the **replacement face**.

5   Select **Apply** to confirm the preview then **Dismiss** the form.
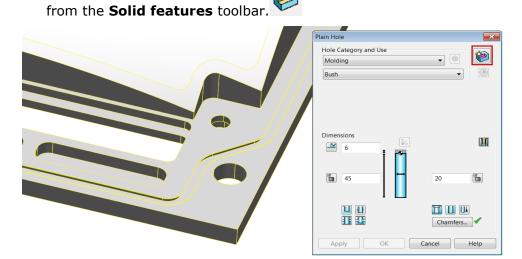
6   **Repeat** the above **process** to **extend** the **lug** on the **opposite side**.

Feature Recognition

The following **Direct Modelling** tools **extract features** and **create history** in the **Solid Feature** tree.
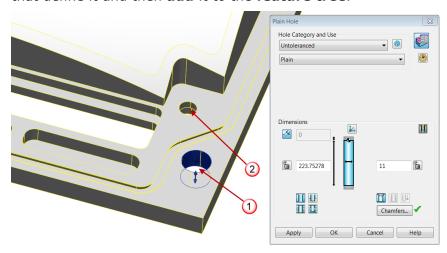
1   **Rotate** the part and **zoom** into the area shown, then select **Create a hole**
from the **Solid features** toolbar.

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
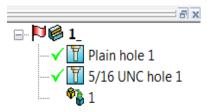
The **hole** is **highlighted** in **blue** and the **dialog form** displays its **properties**.

4   Select **Apply** to accept the results.

5   Repeat for hole. ②

A **counterbored tapped hole** is recognised.
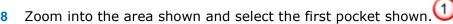
The **two hole features** are now **visible** in the **feature tree** and can now be **modified** as a standard solid feature.
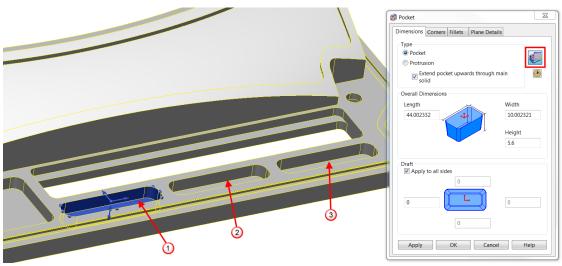
6   Select Create a **Solid pocket** from the **Solid features** toolbar.

7   Toggle to **Feature recognition** mode.

8   Zoom into the area shown and select the first pocket shown. ①
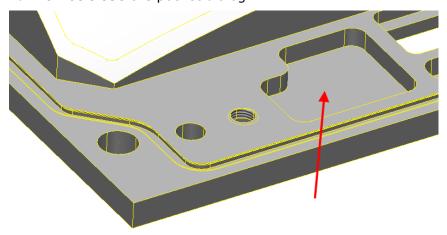
9 Select **Apply** to generate the pocket, then repeat for **pockets** ② and ③.

10 **Do not close** the pocket dialog.
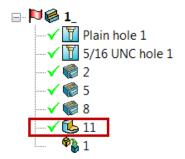
11 **Attempt** to select the **adjacent pocket** shown above.

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**.
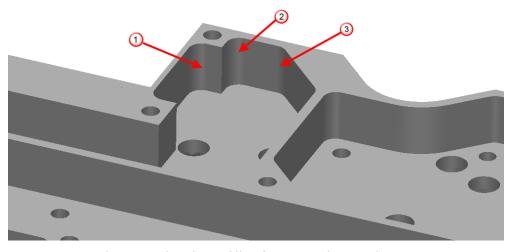
13 **Extract** the **feature** to add it to the **feature tree**.

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
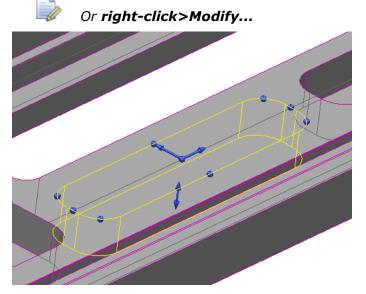
Recognised features are modified in the same manner as if they were initially created directly in **PowerSHAPE**. All current features are displayed in the **Solid feature tree**.
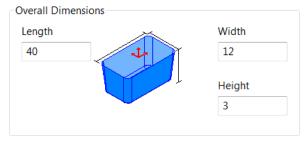
3   Display the dialog for the first pocket by **double-clicking** the **item** in the **feature tree**. ✔ 🗃 2

📝   *Or right-click>Modify…*
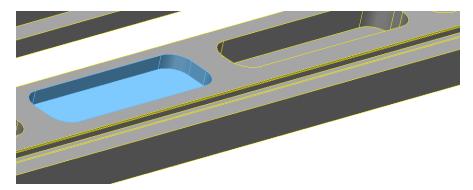
The **pocket** can now be **modified dynamically** using the **drag handles** or **updating** the **Pocket dialog values** themselves.

4   **Modify** the **overall dimensions** as above and click **OK** the **form** to **accept changes**.
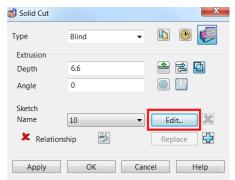
All other **features** can be **edited** using the same method.

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**.
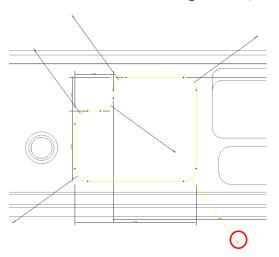
A **wireframe sketch** was **automatically created** when the **feature** was **generated**.

6   Select **Edit** on the **Solid cut** dialog.

Edit the curve using active dimensions
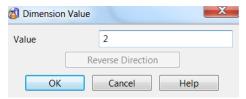
7   From the **curve editing** toolbar, Select **Edit with active dimensions**.
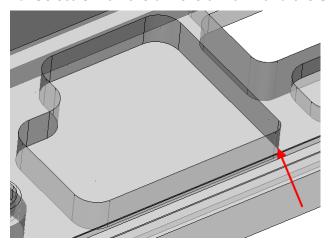
8   **Double-click** on the **bottom right corner dimension** (**circled red above**) to display the **Dimension value** form.

9   Change the dimension from **4mm** to **2mm**.

10   Select **OK** on the dimension form and the **Solid cut** dialogs.
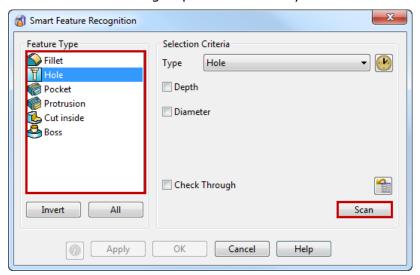
The solid cut feature has updated with the new **2mm corner radius**.

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.*
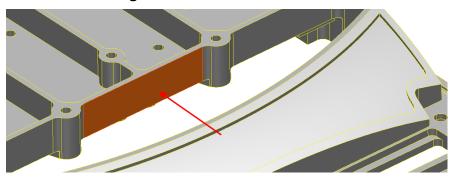
Maintaining Solid history

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.
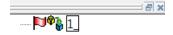
2   Select **Offset** from the **General Edits** toolbar and enter a value of **2mm**.

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.*
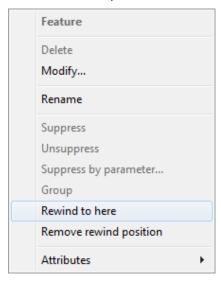
4   Select **Undo** to revert to previous condition.
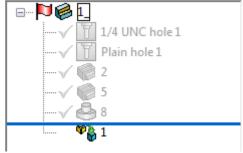
At the bottom of the feature tree, the original imported solid is displayed.

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**.
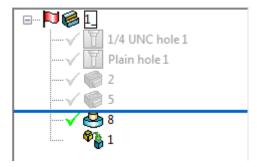
> *The same warning message will still appear however anything greyed out above the **blue rewind position** line will be unaffected.*

> *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.*
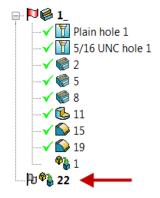
To remove the **Rewind Position** simply right-click in the solid tree on any feature and select the option to **Remove Rewind Position**.

Maintaining the Solid Tree: Method 2

7   Double click to **modify** the base solid 1.
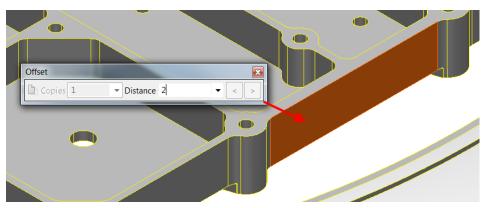
8   Select **Copy solid** then **OK.**

A copy of the base solid 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...).
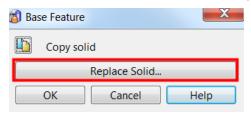
9   Select and **Blank** the main solid with feature history.

10   Select the face on the copied base solid and create on offset of 2mm to thicken the rib.
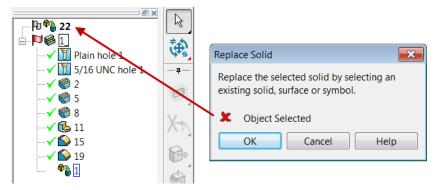
11 Unblank to bring back the main original solid with history.

12 As before, double click to modify the original base solid 1.

13 Select **Replace Solid**.
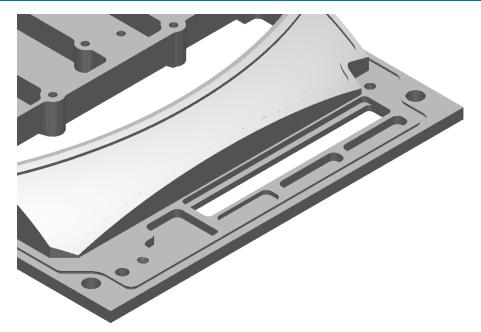
A replacement solid must be selected.

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**
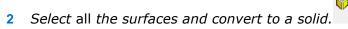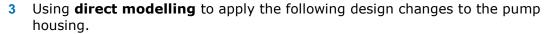
Exercise 10: Pump housing
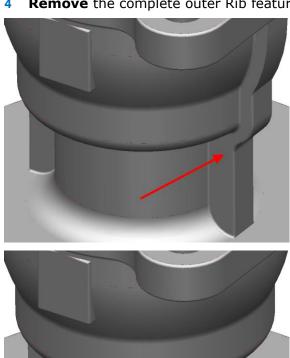
1   **Open** the surface model

.....\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.
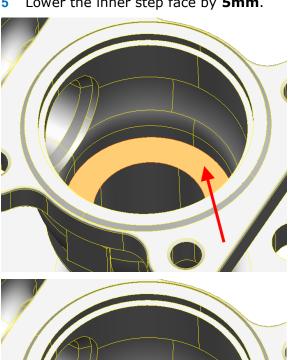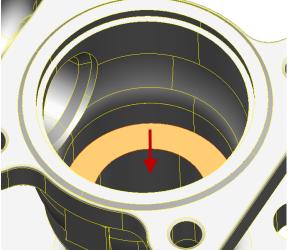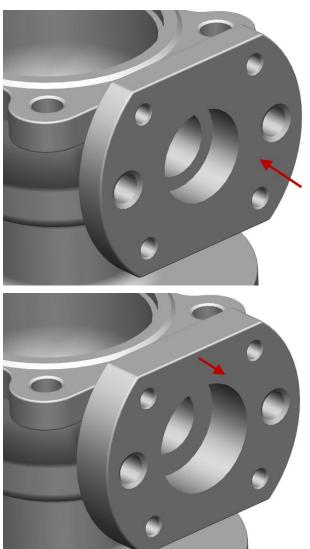
5   Lower the inner step face by **5mm**.
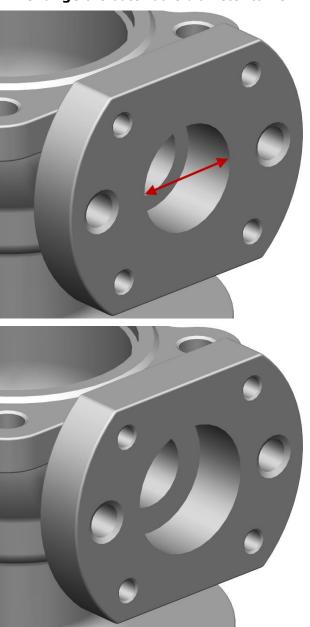
6 Thicken the end plate by an additional **5mm** whilst maintaining the edge fillets and hole chamfers.
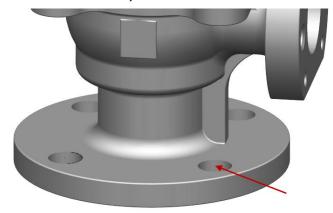
7 **Change** the outer bore diameter to **40mm**.
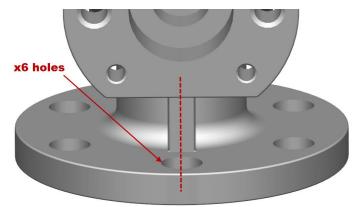
8   Modify the base plate to have **six holes** (from four) on the same **PCD** (Pitch circle diameter).
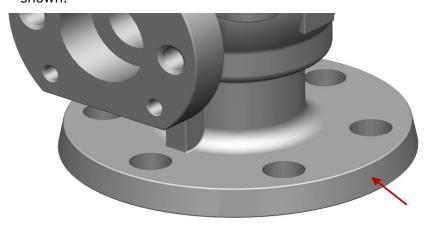
Ensure **one** of the **holes** is **centrally located** to the **main vertical rib** as shown below.

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.**

9. Fixing a solid

Parasolids

PowerSHAPE uses **Parasolids** as the solid modelling engine (kernel); all new solids are created using Parasolids. This changeover was implemented from **PowerSHAPE 2010**. All solids created in PowerSHAPE before that release, are now known as **V8 Solids.**

Parasolid is becoming an industry-standard, and is used as the backbone of many of the leading solid modellers, including SolidWorks (Dassault Systèmes SolidWorks Corp), Solid Edge and NX (Siemens).

Parasolid benefits

The following are benefits you will get from PowerSHAPE using the parasolid kernel:

- Faultless data conversion to and from the other parasolid-based modellers. Higher quality transfer to and from other solid modellers because you can ask for parasolid data from customers instead of IGES or STEP.

- Faster and more robust solid modelling operations.

- Faster and more robust drawings.

- Import and export of Parasolid file formats will conserve the parasolid data with no loss of quality. At present the recommended formats are:

  Parasolid (x_t, x_b, xmt_txt, xmt_bin)

- Crisper, tidier graphics.

Model conversion

- **Version 8 solids (pre 2010)** will continue to work in PowerSHAPE (post 2010).

- All new solid models will use parasolid, unless you change the default by selecting the **Version 8 solids** option on the **Tools > Options > Object > Solids** dialog.

- Parasolid and version 8 solids can co-exist in the same model.

- It is not possible to perform boolean operations between solids of different kernels. One solid must be converted before the operation is attempted.

- It is possible to convert solids in existing models to parasolid and parasolid to version 8 solids.

When a solid is selected the **Solid Editing** toolbar is displayed.

The **Find and fix faults** wizard and **conversion** tool are located here.

Identify solid type

The type of solid can be identified in two ways.

1   Right mouse over the solid to display the local menu and note the name.

Version 8 Solid '1' (Level 0 : General)
Cut
Copy

‘**Version 8 Solid**’ prefixes the solid name

Solid '1' (open) (Level 0 : General)
Cut
Copy

A **Parasolid** is simply prefixed as Solid.

2   With the Solid highlighted

With the icon selected, it identifies a **V8 solid**.

With the icon not selected, it identifies a **parasolid**.

A solid can be converted to or from a Parasolid by simply selecting this icon. The default conversion for a surface model is directly to a Parasolid. In both cases, it is likely that the new Parasolid will require work to remove gaps and other inaccuracies.
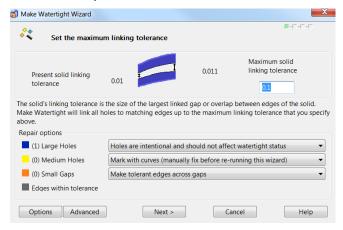
Solid Model fixing

The **find and Fix faults** Wizard can be selected to verify the solid for common problems. It will also request the user to run the wizard if problems are found during conversion between solid types and from surfaces.

If the **Find and fix faults in solid** option is selected, the **Solid Doctor** dialog opens for Parasolids, or the original **Make Watertight Wizard** for version 8 solids.

If **V8 solids** require fixing then the **Make watertight Wizard** is displayed automatically.

If **Parasolids** (post V8) require fixing then the **Solid Doctor wizard** is launched.
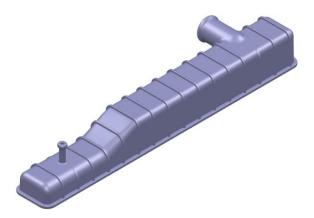
The Solid doctor to repair parasolids, will be demonstrated in this chapter

V8 Solid to Parasolid - convert and fix

A Version 8 solid will be imported and converted to a Parasolid. After the conversion process, some areas of the model will require fixing for it to be fully compatible as a Parasolid. These problem areas will be identified and fixed using the **Solid Doctor**.

1 **Import** the model:
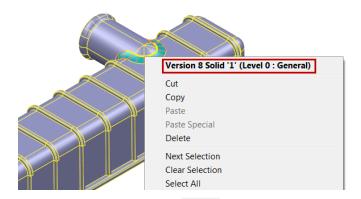
   *.....PowerSHAPE_Data**RadTank-V8solid.dgk***

2 Select **File >Save as**:
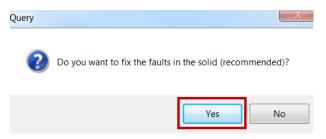
   *\\PowerSHAPE-Models**RadTank-Parasolid***

3 Right-click on the solid to open the popup menu; the first entry confirms it is a **Version 8 Solid**.
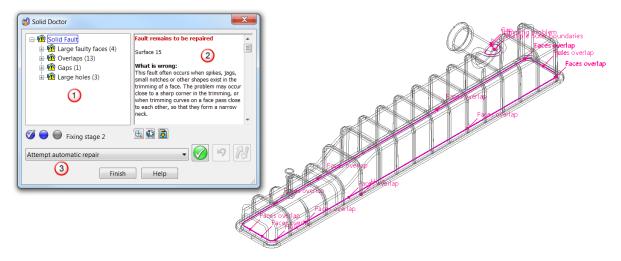
4 Select the **Convert Solid** option. When the conversion to a Parasolid is completed, the following message is displayed.

5 Select **Yes** to launch the **Solid Doctor** wizard.

The Solid Doctor wizard is launched and all faults with the solid are organised and displayed into folders (braches). ① To reveal the individual the faults on a branch, click ⊞ to expand the branch. The following are classified as faults:

- Holes.
- Overlaps.
- Self-intersections.
- Bad trimming of surfaces.
- New faults created by a previous repair option.

  The right hand window ② summarises the fault highlighted with suggestions to help repair it.
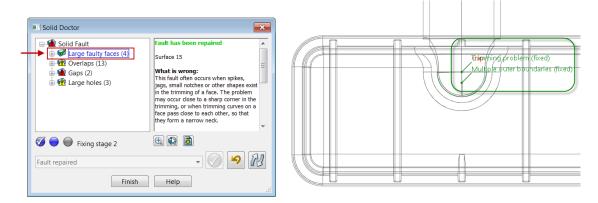
  The drop down list ③ offers the relevant repair methods which can be applied to the select fault(s).

  In the graphics area, the faults are labelled on the solid in purple.

6 Highlight the root folder **Solid fault** then select ✓ to attempt an **Automatic repair.**
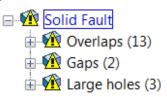
It is recommended the Automatic repair option is initially applied to let PowerSHAPE fix as many faults possible itself.

One fault is repaired and is now labelled in green instead of purple. The repair is identified with a green tick ✓ in the **Solid Doctor Explorer**.

7   Select the **Re-check the solid for faults** option.

The repaired gap is removed from **Solid Doctor**, and is no longer identified by a green label on the Parasolid.
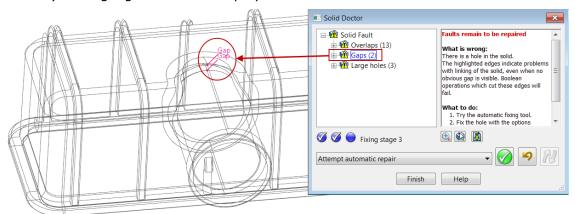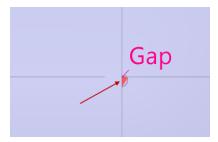
However note an extra Gap fault has been added following this Re-check.

Fix gap (2)

1   Select **Gaps (2)** in the **Solid Doctor Explorer**. Only the related faults (two in this case) are highlighted on the displayed solid.
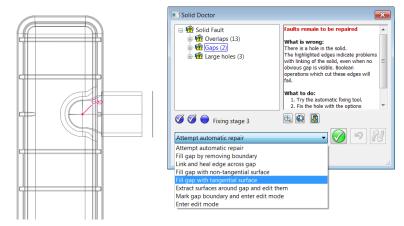
Upon close inspection a small gap is evident where the cylindrical and spherical surfaces intersect on both top and bottom surfaces.

In many circumstances a Gap can be repaired quickly using the Solid Doctor Wizard itself.

2   With the **Gaps (2)** highlighted expand the drop down list and select **Fill gap with tangential surface.**
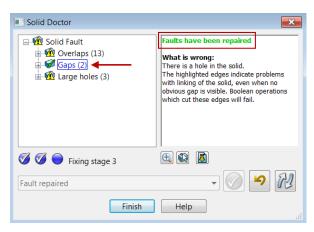
Only valid repair options for the selected faults are listed in the drop down list.

3   Select **Process selected operation**.

The Solid doctor wizard updates the changes and in this case reports both faults have been repaired.

4   Select the **Re-check the solid for faults** option to confirm and remove from the list.
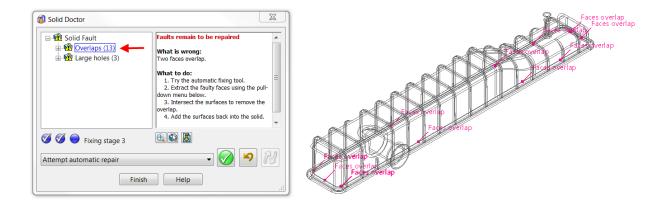
It is recommended rechecking the solid after each fix to highlight any new errors have might have been created following the last repair!
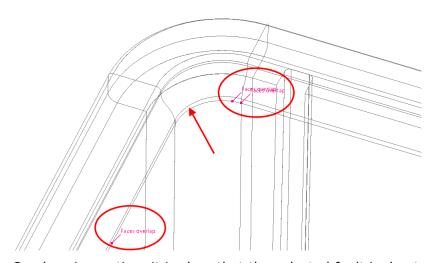
Fix Overlaps (13)

The 13 overlaps are repaired next. Although a high number of instances are reported, it seems they all originate from a single issue.

1   Select **Overlaps (13)** in the **Solid Doctor** tree.
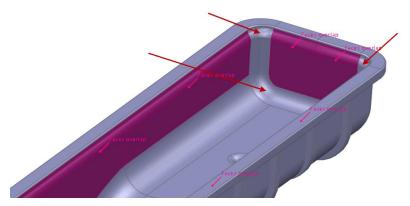
2   Zoom into a region highlighting the overlaps.
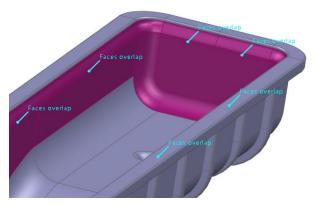
On close inspection, it is clear that the selected fault is due to the inside wall of the solid overlapping beyond the fillet at the base. This must be repaired manually.

3   Select the option **Extract surfaces and edit them.** (Do not process ✅ yet)

Faces that will be extracted are displayed in purple. However there are two upper corner and one lower fillet surface which require inclusion (arrowed above).
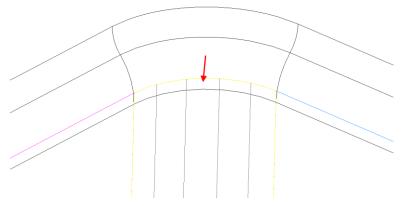
4   Shift select the two corner and lower fillet faces then **Apply selected operation.**
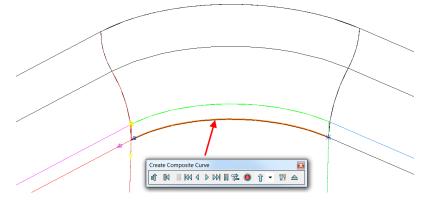
> *Surfaces can be deselected using the **CTRL** key.*

5   Select **Blank Except (Ctrl+K)** to hide all except the surfaces highlighted.

   The faulty surfaces are the only selected items in the graphics area.
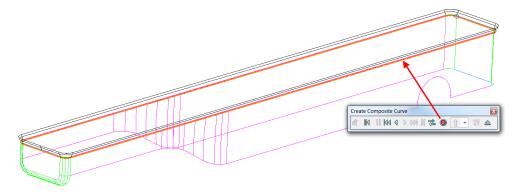
6   Switch to **wireframe** view and zoom into a corner to have a closer look.
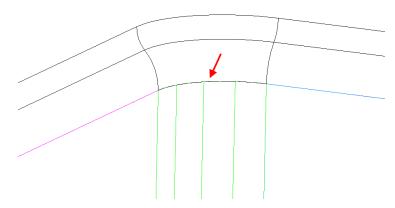
   The main, inner-sidewall surfaces need to be trimmed back as they are extending upwards, beyond the tangency with the blend surfaces.

7   Create a **Composite Curve** and select the lower edge which defines the fillet base.

8   Select **Fast Forward** to quickly trace the complete lower edge around the component.

9   **Save** and **close** the form to create the **composite curve**.

10 Use the composite curve as the cutting object to **Limit** back the vertical surfaces and remove the overlap.

11 Delete the **composite curve**.

The overlap has been removed.
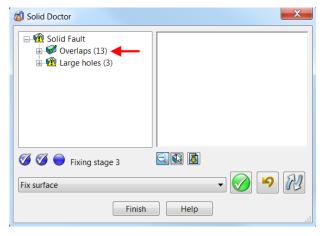
12 **Unblank** (**Ctrl+L**) to redisplay the solid and as before, quick select all the Surfaces and solids.

 *In the selection menu you can **quick select all solids and surfaces**.*
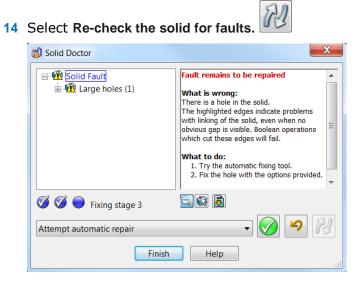
13 Select **Accept all surface edit changes** from the floating **Solid Doctor** Toolbar

The **Solid Doctor** confirms that the all 13 faults have been repaired.

14 Select **Re-check the solid for faults.**

Fix large hole (1)
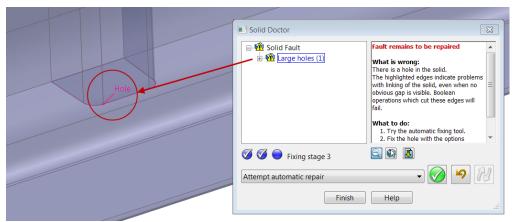
1 Select the remaining set of surfaces registered in **Large holes (1)** in the **Solid Doctor Explorer**.
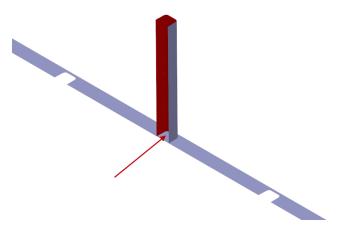
The process has exposed a surfaces-overlap problem at the base of the fifth rib, from the non-pipe end of the tank.

2 Select the option **Extract surfaces around the hole and edit them.**
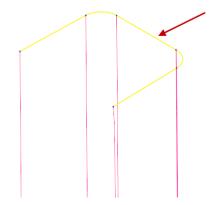
3 Select **Apply selected operation.**

4 **Blank Except (Ctrl+K)** to hide all entities apart from the extracted surfaces.
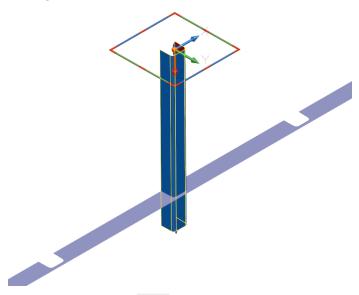
The fact that the base surface is not trimmed back to the inside of the rib suggests that there is likely to be a small gap.

5 Select a **wireframe** view and zoom into the problem area.

6 With the **Alt** key depressed, left-click a composite curve on the top edge of the rib surfaces.
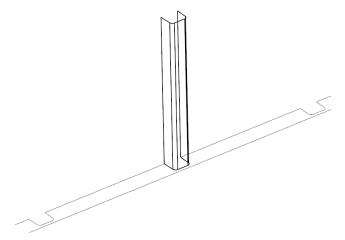
7 **Select** and **delete** the original **rib surfaces** (keep the base form surface).

8 Select the **composite curve** and generate a **surface extrusion** which passes through the base surface as shown.

9 Use **Limit Selection** to trim back the new rib form surface to the base surface.
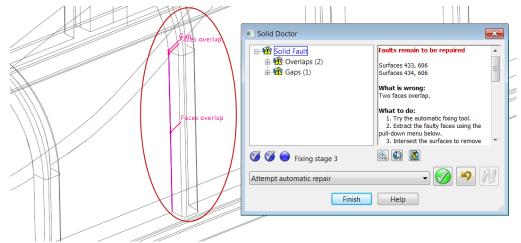
10 **Unblank** (**Ctrl+L**) to redisplay the solid and as before, **quick select all the Surfaces and solids**.

11 Select **Accept all surface edit changes** from the floating **Solid Doctor** Toolbar

Although the hole fault was repaired, it seems the last operation caused additional faults.

12 Select **Re-check the solid for faults** to confirm.

The new rib surface has created minor overlaps and a gap.
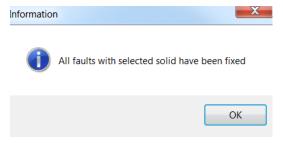
13 Highlight Solid Fault then Attempt Automatic Repair.

The two overlaps have been repaired along with a related gap concern.

14 Select **Re-check the solid for faults** to confirm.
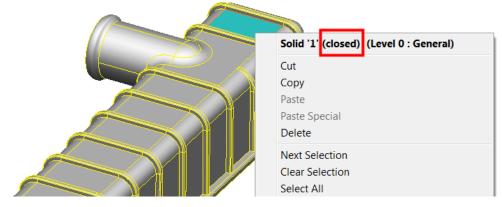
The solid model is reported repaired.

15 Select **OK** to accept.

If the solid doctor was applied again, then the following message appears.

16 Right mouse select on the Solid to display the local menu.

A closed solid also indicates the faults have been repaired.

17 Save to update the model. (Do not close).

Export Wizard

PowerSHAPE can **export** models to formats other than DELCAM Geometry (.dgk).
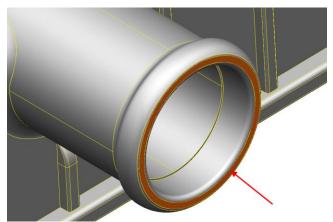
The Export Wizard in PowerSHAPE allows the user to export data but providing the additional options.

▪ Export selected model items, visible items or all items.

▪ Export relative to an active workplane or World.

▪ Export drawings (Delcam drafting)
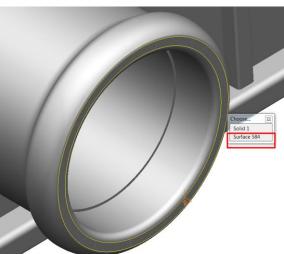
The above repaired solid will now be exported as a Parasolid relative to an active workplane.

This Workplane will be created first.

1 Select and highlight the end solid face of the outer pipe as shown.
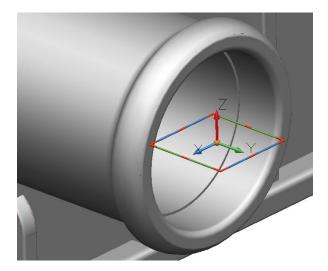
2 From the **Solid edits** toolbar, **Copy the selected face**.
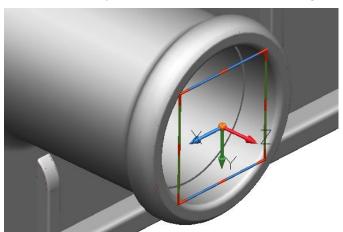
A surface of the selected face has been copied out.

3 Just select the surface and then create a Workplane at the **centre of selection**.
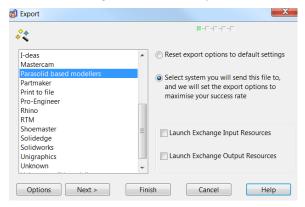
4   Twist the Workplane in **X** so the **Z axis** is along the pipe axis.

With the correct orientation, the model will now be exported.

5   Select **File>Export….**from the pull down menu.
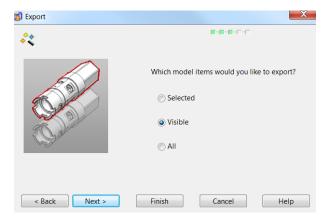
The export wizard allows PowerSHAPE geometry to be exported to most common formats (depending on licence).

6   Highlight **Parasolid based modellers** then **Next>.**

7   *Browse to and save as \PowerSHAPE-Models\RadTank*
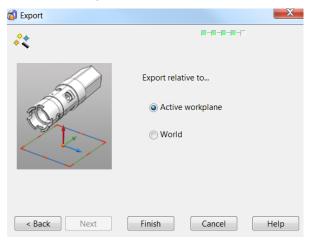
The save as type has automatically set to **Parasolid (*.x_t)**

With only a single model on the screen, all three options would be applicable.

8   Select **Visible**, then **Next>.**

The last workplane created in currently active.

9   Select **Active Workplane** then **Finish.**
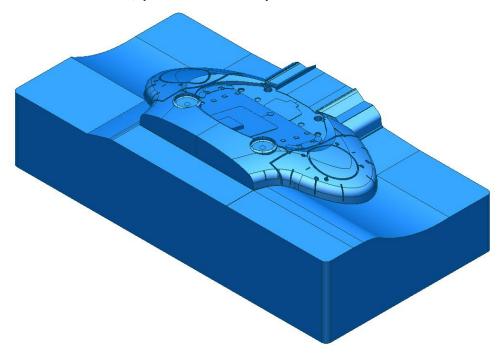
10 Select **OK** to complete the process and export the data as a Parasolid.

11 **Save** to update the psmodel then close.

Exercise 11: Solid Doctor Speaker Core

1   Open the PowerSHAPE model:

.....PowerSHAPE_Data**Speaker Core_Surfs.psmodel**

2   Convert the surface model to a parasolid with no faults.

3   Once completed, Select **File >Save as**

…/PowerSHAPE-Models/**Ex 11 Solid Doctor spkrCore.psmodel**

A more difficult challenge is available by opening:

...../PowerSHAPE_Data**Speaker Core_surfs_advanced.psmodel**
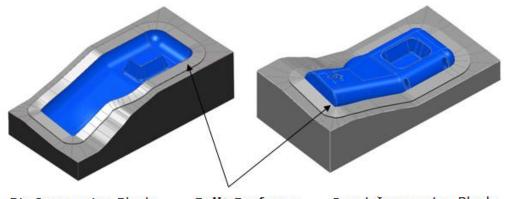
10. Surfaces for Manufacture

Introduction

Split and **draft** surfaces play an important role in designing components and tooling for material-forming applications. In most mold or die applications the establishment of the natural split line is crucial, and so is the need to design a split surface for machining. In most cases too, a draft angle has to be incorporated into the surface of a mold or die to enable the product to be withdrawn.

Split Surfaces

Also known as a '**run-off**' surface, split Surfaces are an essential feature required in applications such as **mold tools**. The component wall thickness is formed by the gap between the separate Punch and Die Impression Blocks. The edges of the plastic component are formed by a Split Surface common to both halves, accurately bedding out when the Punch and Die Impressions are pushed together.
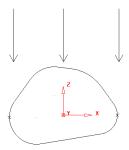
Die Impression Block   **Split Surfaces**   Punch Impression Block

A surface's natural split line or occluding boundary is the line around a surface where the surface, when looked at along the draw axis (that is, the direction in which the mold or die is to open), just dips out of sight.

Split Surfaces are generated from split lines previously created around a component model.
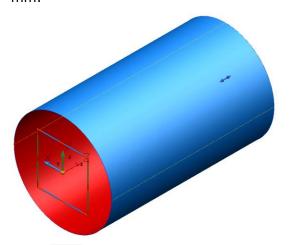
In areas where the surface model is vertical or does not reach the vertical angle, you must manually construct a suitable curve. This is not always evident until after the **Create a Draft Curve** with the **Draft Angle** value as **0** is applied. These areas are then identifiable as gaps along the Draft Curve.

In the following example, the Split Line is obtained using **Create a Draft Curve** to obtain a curve for the natural split. This is then used with the **Split Surface** options.
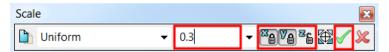
Split Surface Example 1

In this example, a primitive cylinder is modified to produce a simple part. A split line for the part is calculated for the surfaces and a new split surface is made.

1   **Create a New Model** and **Workplane at 0**.

2   Set the **X** as the active Principal Plane.

3   Create a **Surface cylinder** Primitive at **0** with a **radius** of **30** mm, and **length 100** mm.
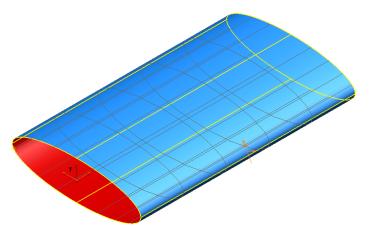
4   **Scale** the surface by **0.3** in the **Z-axis** (Lock the X & Y values).

5   **Right-click** on the **surface**, and, from the **popup menu**, select **Convert Surface** to create a **Power Surface**.
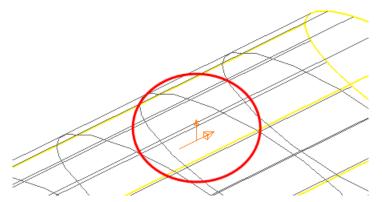
The surface will undergo further modifications to illustrate split surface creation.

To give a curved flow along the surface, a spine will be added. A spine is a curve that can be used to control the alignment of sections on a surface. A spine has a spine point for each section along the surface. If a spine point is moved, the corresponding section moves with it, retaining the same tangency if **Apply smoothing to point edits** is disabled.
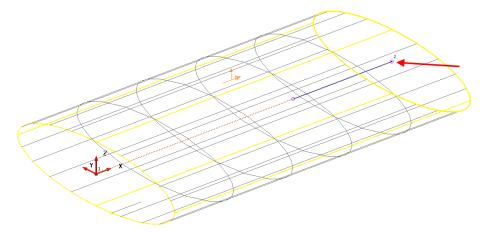
6   **Select** the **surface** to display the **Surface edit** toolbar.

7   Select **one** of the surface **curves** to enable the **Curve & Point Edit** toolbar.

8   Select to enable labelling of points on a selected curve.
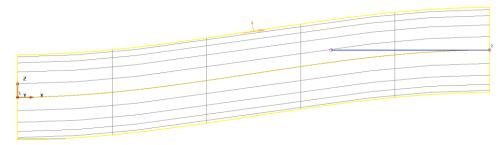
9   **Ensure** the **long arrow** displayed on the selected **surface** is pointing **along** the surface. This sets the **operational direction**. This can be left-clicked to switch from **across** to **along** the selected surface.

10 From the **Surface edit** toolbar, select **Create Spine** (**Add curve options)**.
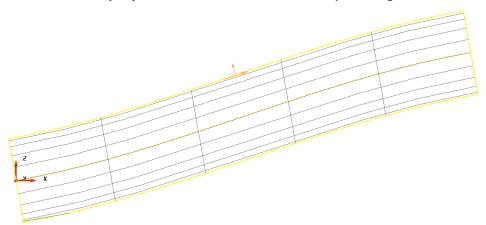
This creates a **spine** running through the cross sections of the surface, shown as a dotted line. This **spine** can be deleted without affecting the surface.

11 Select **Spine Point 2** on the spine by left-clicking on it (shown above)

12 In the **Surface/Curves editing** toolbar, ensure **Apply smoothing to point edits** is disabled.

13 With **Spine Point 2** selected, type **0 0 10** in the **Command Input** window; both the Spine Point and corresponding Section move up Z by **10** mm. The tangent angle along the surface remains the same, as shown below.

14 Set the **Y Principal plane** and rotate the surface by **-10** degrees.

15 Generate a **composite curve** for each end lateral.

16 **Right-click** over each **composite curve** in turn and select **Delete Dependencies**.

> *When composite curves are traced from surfaces, the curve remembers the tangency of the underlying surfaces. When a further surface is generated from these composite curves, they match perfectly. To remove this relationship, **Delete dependencies** is used on the composite curve before a new surface is generated.*

17 Create a **Fill-in** surface for each composite curve.
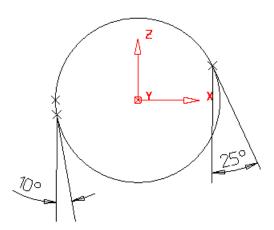
18 **Delete** the **composite curves** and **reverse** any **surfaces** to have the **gold/blue outside**.

19 Create a **Convex surface fillet** on each end of **Radius 2 mm**.

The model now is complete but before a split surface can be generated the appropriate **Draft Curve** must first be created.

For a **Draft Curve** suitable for **split surface** creation, the **Z** principal plane must be set.

For a **draft angle** of **0** degrees (as marked by a cross), it appears exactly half way down the sphere (equator).

For a positive angle of **25** degrees, the draft curve appears on the upper half of the sphere as indicated.

For a negative angle of −**10** degrees, the draft curve appears on the lower half of the sphere.

20 Select the **Z** principal plane.

21 Select **all surfaces**.

22 Select **Draft Curve** from the **Curve** menu.

The draft angle is specified in this dialog. If the composite curve option is selected, PowerSHAPE generates composite curves of the natural draft; otherwise, ordinary curves are generated.
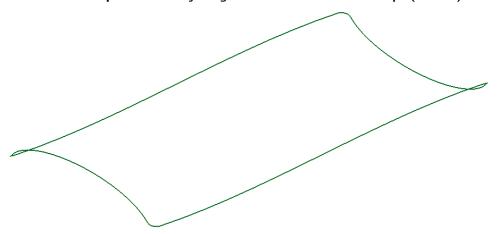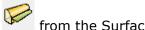
23 Leave the **Draft Angle** at **0** and **Create Composite curves selected,** then select **OK**.

24 Select the **composite curve** just generated and **Blank Except** (**Ctrl+K**).
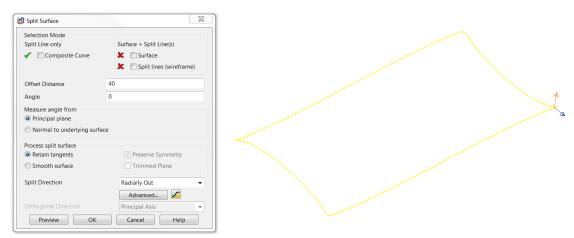
The split surface can now be created from this composite curve.

25 Select the **composite curve** then **Split Surface** from the Surface

menu.

The **Split Surface** dialog is displayed and two arrows appear on the split line.

The blue arrow shows the direction the split surface will flow from the curve. The brown arrow indicates the direction any optional Draft Angle will be applied.

26 Click on the arrows if necessary to change the **Split Surface** directions to be as shown in the illustration above.

27 Enter an **Offset distance** of **40** and select **Preview**.
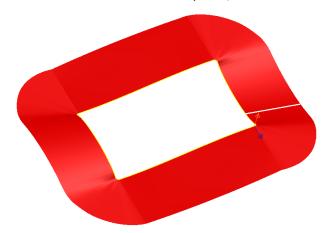
A **40 mm** wide split surface is previewed, tangentially outwards form the **curve**.

It is likely that the new split surface contains **localised ripples**, especially where there is a group of surface curves that are close together. This is not an acceptable condition to achieve a satisfactory bed-out between the two halves of a mold. To achieve a **smooth surface form**, the ripples must be prevented or edited out.

28 Select the **Smooth Surface** option, select **Preview** and then select **OK**.
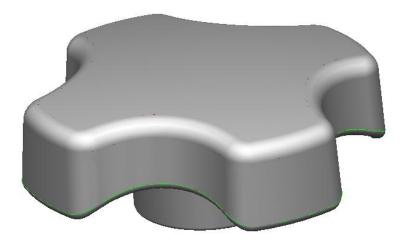
This produces a smoother surface on the outside edge by allowing the surface curves to rotate away from the original perpendicular direction from the draft curve. This helps improve the smoothness, but the split surface may still benefit further from further editing, this can be carried out retrospectively by moving or inputting points, removing or inputting curves, and editing tangent angles.

29 **Cancel** the **form**.

30 Select **Unblank** and delete the surfaces and workplane.

31 **Close** the **Model**
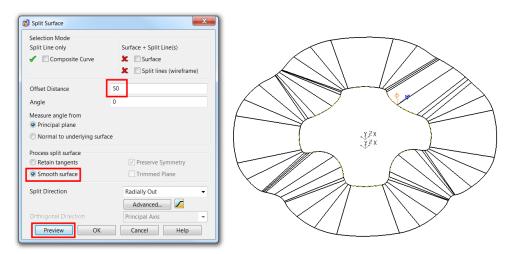
Split Surface Example 2

1 **Open a New Model** and **Import** .....PowerSHAPE_data/**Split_surf_example.dgk**

The file contains a solid model and a predefined Composite Curve, which will be used to generate the split surface.
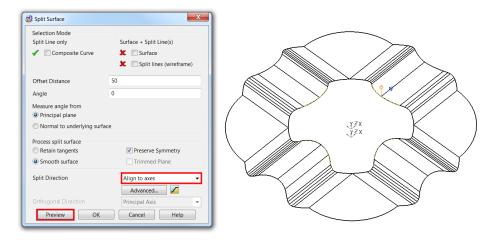
2 Select the **Solid** and then **Blank** (**Ctrl+J**).

3 Select the pre-defined **composite curve** running around the handle.

4 Select **Split Surface** from the **Surface** menu.

5 On the **Split Surface** dialog, enter an **Offset distance** of **50** and set the horizontal arrow to point outwards from the surface (if not already the case).

6 Select **smooth surface** and the **Split direction** to **Radially Out** and then **Preview**.

The surface curves are converging in 4 places due to the shape of the draft curve and the in-out length of the split surface. One way of improving this is to use the **Align to Axes** option. This aligns appropriate sections of the split surface along the X or Y directions.
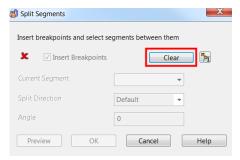
7 In the **Split direction** list, select **Align to axes** and select **Preview**.
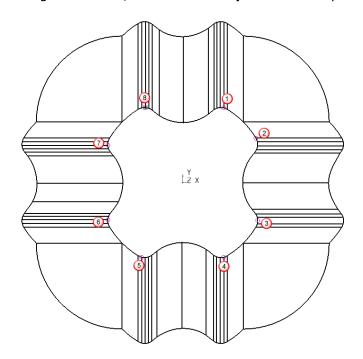
The split surface is now aligned to the X & Y axes giving an overall smoother split surface. Further improvements can be made to the surface by using a combination of **Radially out** and **Aligned to axes**.
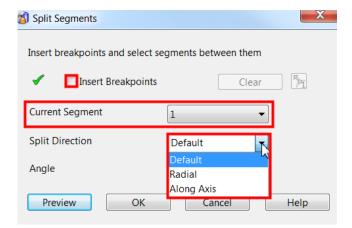
8   Select **Advanced** on the dialog to display the **Split Segments** dialog.
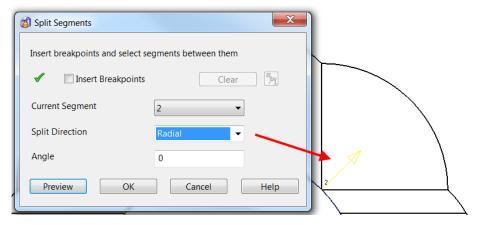
9   Select **Clear** to remove any existing segments from the curve.

10 Using the mouse, click the **breakpoints** at the positions indicated.
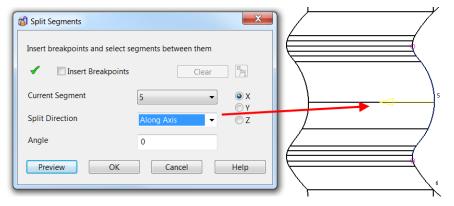
11 Deselect the **Insert Breakpoints** check box.

12 Use the drop down list to toggle through each segment and then select a suitable split direction.

For example in this instance...segment 2 would require a Radial split direction.
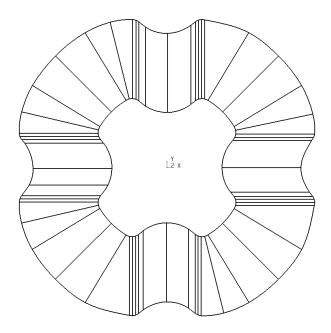
Segment 5 would require a split direction along the Axis (X automatically selected)

13 **Repeat** for **all segments** assigning a suitable split direction.

14 Select **OK** to accept and close the dialog.

15 Select **Preview** from the **Split Surface** dialog. The **Preview** indicates a combination of radial and align to axis directions.

 Do not cancel the dialog!

Automatic Stepped Split Surfaces
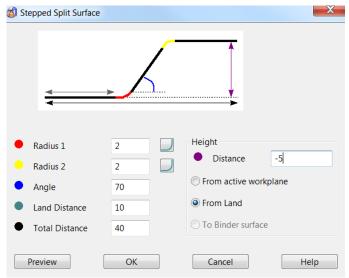
Stepped Split Surfaces can be applied to reduce the extent of the *shut out* required to ensure correct matching of the two halves of a mold. Split Surfaces can be defined with:

- **Land Distance**, which is the length from the outer edge of the part to the step.

- **Angle** of inclination for the step.

- **Total Distance** across the split surface.
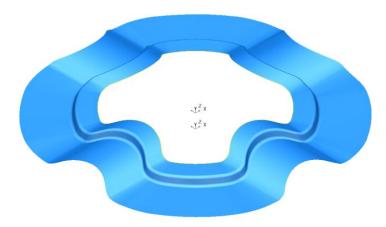
In addition, you can specify different radii from the joins between the various segments, or use chamfers.

1   Select the **Stepped Split Surface** option.

The form allows values to be defined for the stepped split.
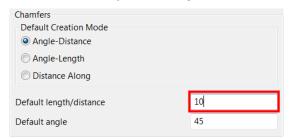
2   Enter the **values** in the dialog, as **shown above**, and select **OK**.
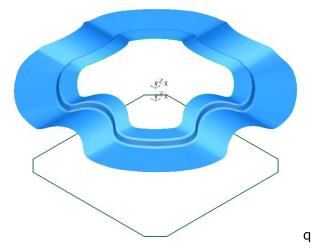
The Split Surface is produced with the defined step. An impression block will be created to be trimmed back to the split surface.

3 Create a **Rectangle** from **-60 -60 -60** of **Width 120** and **Length 120**.

4 Select **Tools > Options >Object > Lines**.

Chamfers are created from the **Default Creation Mode** setting in the **Options** dialog. If required, a chamfer can be retrospectively edited both dimensionally or by any of the available creation modes.

5 Set the Default length/distance to **10** and then **OK**.

6 Select **Create a chamfer** ⌐ from the Lines menu. N

7 Click each corner in turn to apply a chamfer.

8 Generate a **composite curve**.

q
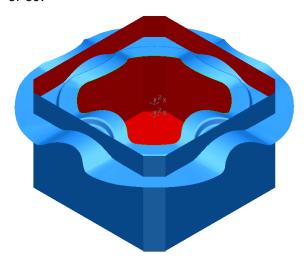
The curve will be used to generate the side walls of the block.

9 Select the lower **composite curve** and create a **Fill-in surface**.
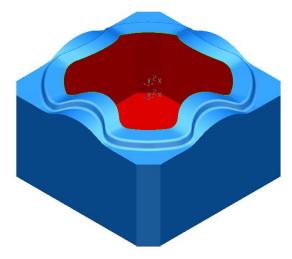
10 Select the **composite curve** and create a surface of **extrusion** with a length of **80**.

The sidewalls will be trimmed back to the split surface as shown above.

11 Limit the side walls back to the split surface as shown below.

12 Select **Unblank** (**Ctrl+L**) to redisplay handle solid.

The basic Impression block design is complete.

13 Save the model as

.....**PowerSHAPE Data\\Impression block.psmodel**

14 Close the **model**.

Mold Die Wizard

An option has been included in **PowerSHAPE**, called **Mold Die Wizard**. Using the **active solid**, the **Mold Die Wizard** will automatically **split** and **trim** the model creating **cavity** and **core** inserts.

15 **Open** a **New Model**.

16 **Import** the file **Phone_Cover.dgk** in the modelling window.

The **Phone_Model.dgk** contains the **surface data** of a mobile phone cover. For **PowerSHAPE** to process the data through the **Die Wizard**, the model needs to be in the form of a **solid**. After conversion from a **Surface** model to a **solid** it is strongly advised to **check** for any potential **faults**.

17 **Box-select ALL surfaces**.

18 From the **solids menu**, select **create solid from selected surfaces**.

>   *In doing this, **PowerSHAPE** will attempt to convert **all** the **selected** surfaces in to **solids**.*

19 Click onto the newly converted **solid** to unlock the **Solid Edit Toolbar**.

20 Select the option to **"Find and Fix faults in the solid"** highlighted above.

21 A form will appear informing you that the **selected solid** (**phone**) contains **no faults**.

> *If there are any **faults** in the solid you may have to follow the **solid doctor** wizard in order to fix the solid. Please refer to the **Solid Doctor** help file.*

22 Select the **solid**.

23 From the **Main Toolbar** select **Wizards** and then select **Start Mold Die wizard**.
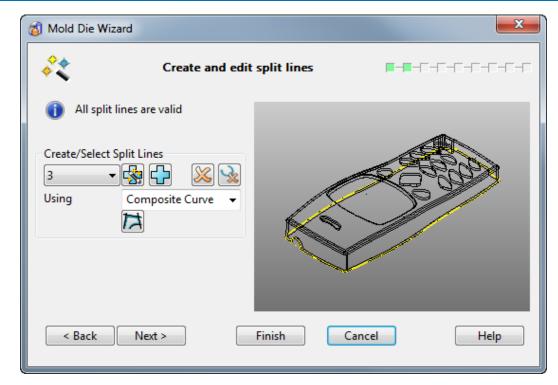
The **first page** of the **Mold Die Wizard** allows the user to **select** the **objects** to be used for **die creation**. These can be in the form of previously generated **Split lines**, **Split Surfaces** and **Die blocks**. In this example, **Split lines**, **Split Surfaces** and **Die Blocks** will be **automatically** created by the wizard and the object **information area** highlights the selection of the **product only**.

24 Select **Next**.
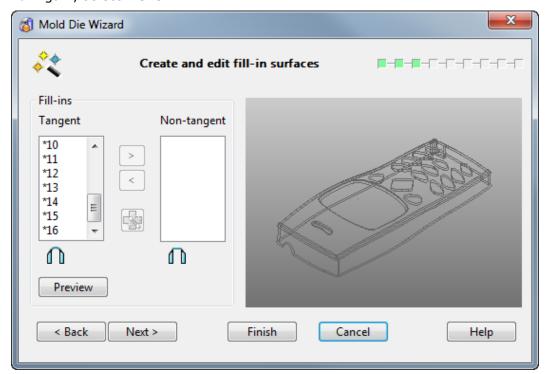
The **second page** allows the user to **create and edit split lines**. The wizard has **automatically** generated the **split lines** to be used for this model.

Each **split line** is **numbered** and can be **selected**, and **modified** if necessary from the **Pull Down** list. As a **split curve** is **selected**, it is displayed in the **Die Wizard** graphic window.

The model can be **dynamically manipulated** in the **Die Wizard graphic window** in the same way it can be manipulated in the **PowerSHAPE** graphic window.
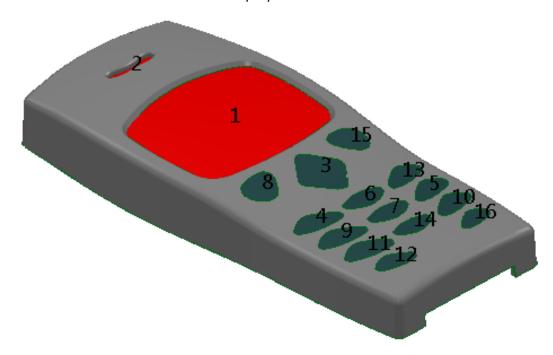
25 Again, select **Next**.
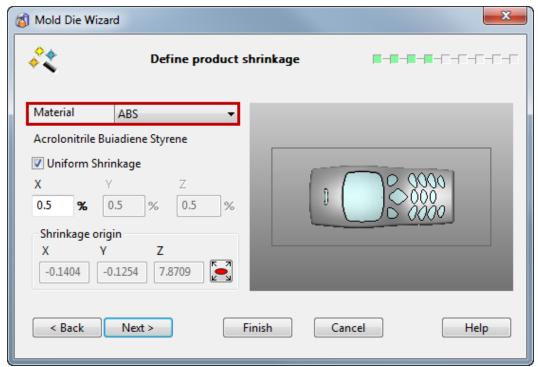
This page gives the user the opportunity to **change** or **create fill-in surfaces**. In this case the **surfaces** created by the **wizard <u>do not</u>** need to be modified.

26 Click the **Preview** button to display the **fill-in surfaces** on the **model**.
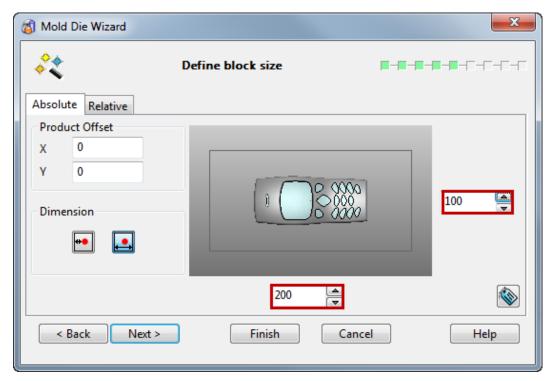
27 Select **Next** to continue through the **wizard**.

This page allows the user to specify **product shrinkage**; the **amount** by which the **mould cavity increases** to allow for **shrinkage**.
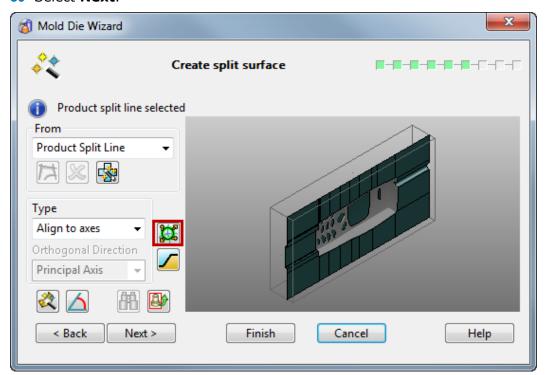
The **drop down list** displays the **abbreviated names** of the **materials**; when a **material** is **selected**, the **full name** of the material is displayed below. A **predefined value (**dependent on the material selected) will be **automatically inserted** into **X, Y** and **Z**. **Uniform Shrinkage** can be **switched off** to enable the user to set different shrinkage factors for **X**, **Y** and **Z.**

28 From the **Material** drop-down menu select **ABS**.

This page of the wizard allows the user to specify **block size** for the **die inserts**. **Size** can be set using **overall dimensions** for the **block** or by setting a **minimum land dimension**.

29 Set a **block length** of **200mm** and a **width** of **100mm**.

30 Select **Next**.
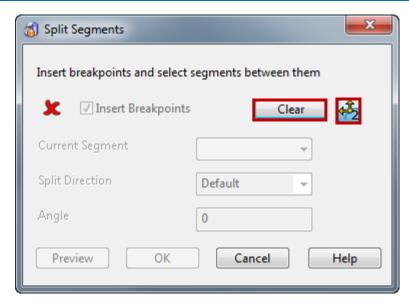
The next page allows the user to **edit** the created **split surface**. In this example the **split surface** is required to use a combination **run off** of **radially** and **along axis**.

31 Select the **Advanced Split Surface Options** button highlighted above with a red border.
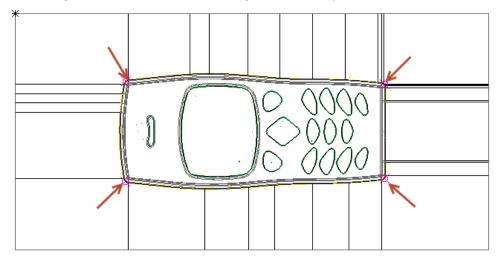
> The **split segments** form will appear.

Split Segments

Insert breakpoints and select segments between them

☒ ☑ Insert Breakpoints   Clear

Current Segment

Split Direction   Default

Angle   0

Preview   OK   Cancel   Help
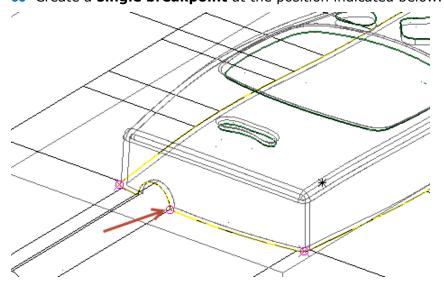
32 Select **Clear** to remove any existing segments from the curve and set the **Corner Type** to **run in both directions**.

33 Using the **mouse** create **breakpoint** at the positions indicated below.
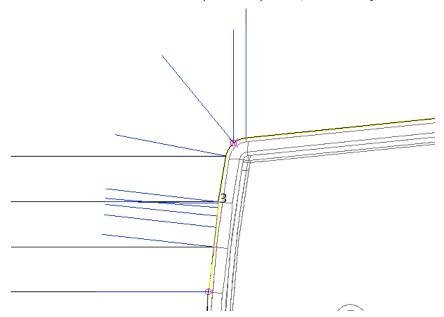
34 **Reset** the **corner type** to **run in a single direction**.

35 Create a **single breakpoint** at the position indicated below.
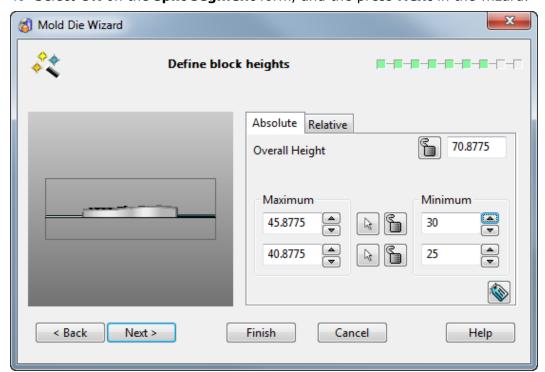
36 Untick the **Insert Breakpoints** box to toggle the curve **segments option**.

37 Select **segment 1** from the drop-down menu and set the **split direction** to be **Along Axis**. Note that the **relevant axis** will **automatically** be set upon selecting this option, but can be **overridden** if necessary.

38 **Repeat** this procedure for the segments **2**, **4** and **5**.

39 For, in this case **Segment 3,** which refers to the **breakpoint** created **last** around the hole in the top of the phone, set the **split direction** to be **Radial**.

40 Select **OK** on the **split segment** form, and the press **Next** in the wizard.
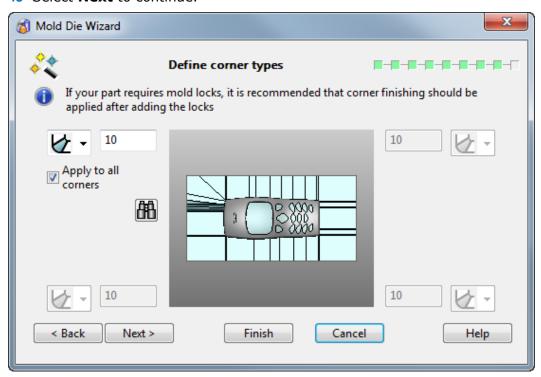
This page of the **wizard** allows the **user** to define **block heights** for the **die inserts**.

Values can be determined using **absolute** or **relative dimensions**, specifying an **overall height** or by fixing **maximum** and **minimum** values of the **upper** and **lower parts**.

41 Set a Minimum **Upper** value of **30**.
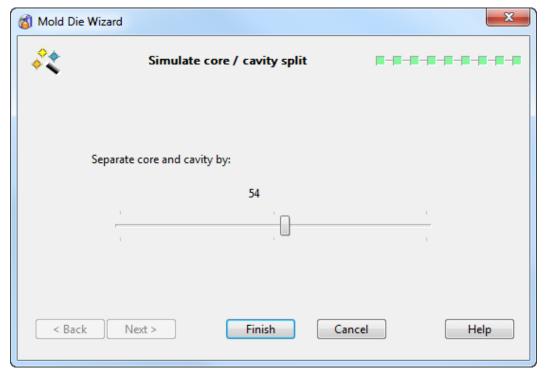
42 Set a Minimum **Lower** value of **25**.

43 Select **Next** to continue.
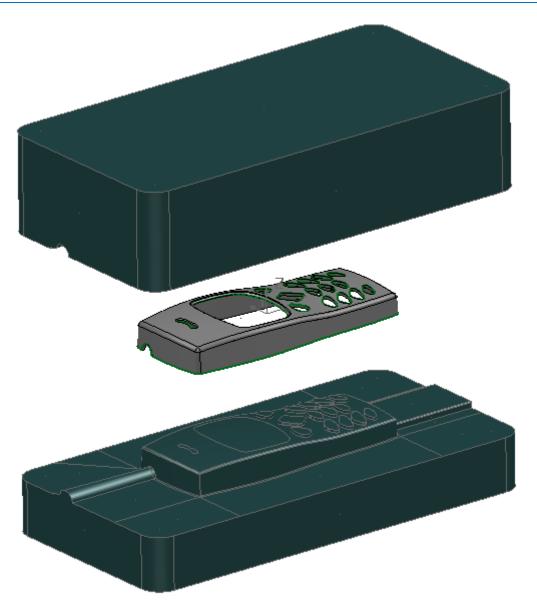
This page of the wizard allows the user to **define corner types**. There are three types of corner definition, **Radial**, **Chamfer** and **none**. These can be applied either **individually** or **globally** to the **insert corners**.

44 Set a **10mm Chamfer** and tick **Apply to all corners**.

45 Select **Next**.

The final page of the wizard allows the user to **simulate core** and **cavity separation** using the slider.
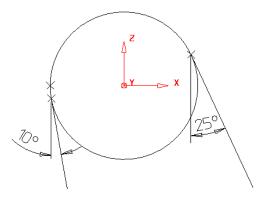
46 Move the **Slideer** to **simulate** separation.

47 Select **Finish** to **exit** and **complete wizard**.
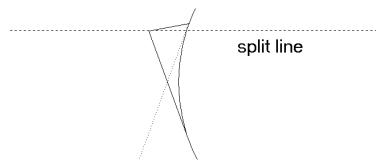
Draft Surfaces

A **Draft Surface** can be created from a Composite Curve or from one or more Surfaces. A Draft Surface is projected relative to the active Principle Axis, either onto the 0 plane, or onto one or more additional Power Surfaces, selected after the original data has been taken into the open **Draft Surface** creation dialog.
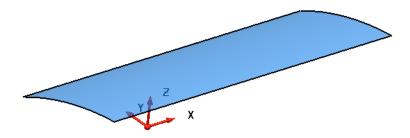
The **Draft Surface** is generated tangentially from the original surface dependent upon the angle selected, similar to the **Draft Curve** command.

Where the split line is generated with an angle above or below 0 degrees, the resultant split surface edges on both the top and bottom may be mismatched. PowerSHAPE increases the lower draft angle so the split surfaces edges match as shown.

Draft Surfaces from Composite Curves Example

1   **Create a New Model** and Workplane at 0.

2   Create a **Three point arc** ![arc icon] using the following coordinates:

    -15 10   4

    -15 -10 10

    -15   0   8

3   Create an **extruded surface** along the **X-axis** of **Length 50** mm.

This extruded surface will be used later to project a draft surface onto.
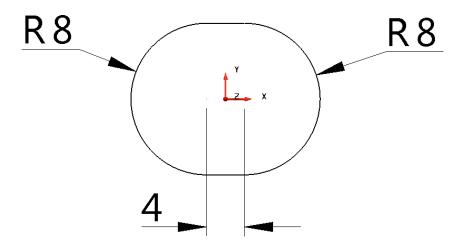
4   **Convert** the surface to a **Power Surface**.

5   **Blank** the surface and select a **view from the top**.
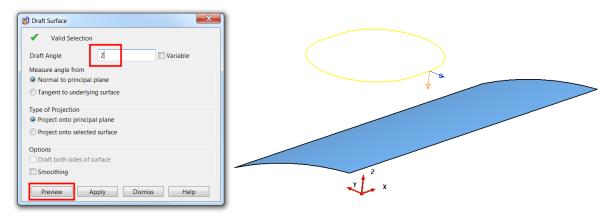
6   Generate the following **wireframe**.

This section will be used to generate a Draft Surface.

7   Create a **composite curve** and move it up in **Z** axis by **20** mm.

8   Select **Unblank** (**Ctrl+L**) to display the extruded surface.

9   **Ensure** the **composite curve** is selected.

10 Select **Draft Surface** from the **Surface** toolbar.
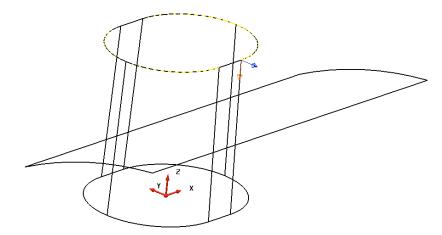
Draft Angle allows you to specify the taper angle of the surface to be produced. As with split surfaces, the direction in which the surface is created is determined by the two arrows displayed on the composite curve or surface selection.

11 Set the **Draft Angle** to **2** degrees.

12 If required, change the arrows by clicking on them so they point **down** and **outwards**.

13 Select **Preview**. The draft surface is generated and projected onto the **XY** plane of the active workplane.

14 Select the original **extruded surface**.
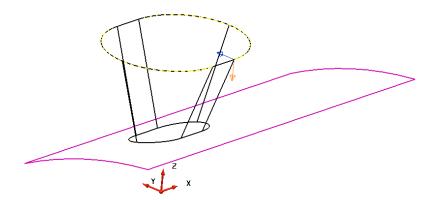
When selected, the surface appears pink and the **Split Surface** option on the **Draft Surface** form changes to **Project onto selected surface**.

15 Change the **Draft Angle** to **20** degrees and the projection arrows to inwards and downwards.

16 Select **Preview**. The draft surface is automatically trimmed to the lower curved surface.
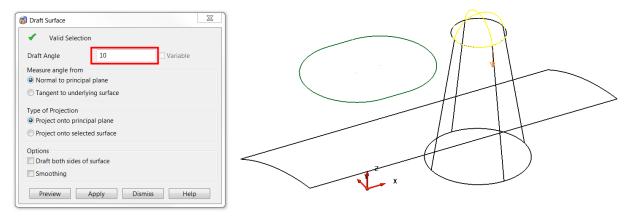
17 **Dismiss** the dialog.

Draft Surfaces from Existing Surfaces
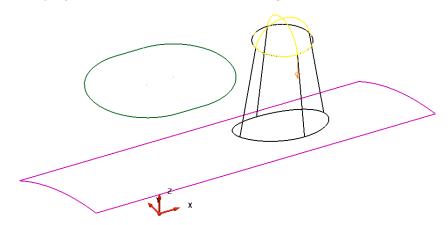
Draft surfaces can be created tangential to existing surfaces.

1 Create a Primitive **Sphere** of **radius 4** at **X18 Y0 Z20**.

2 Right-click over the **Sphere** and select **Convert Surface**.

3 Select the Sphere.

4 Open the **Surface Draft** dialog and enter an **angle of 10 degrees** with the direction arrow pointing downwards.

5 Select **Preview**.

The draft surface that would be produced is displayed.

6 Select the **extruded surface** and then select **Preview**. A new **draft surface** that project onto the curved surface is generated.
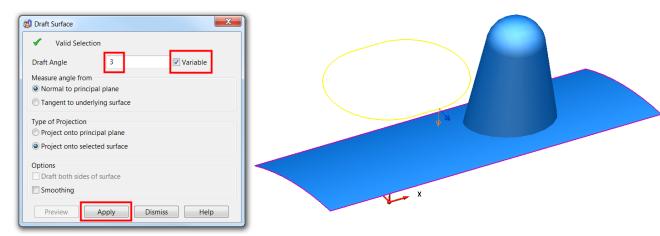
7 Select **Apply** and dismiss the dialog.

Variable Draft
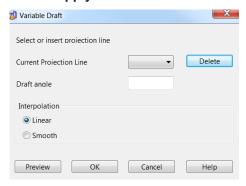
There is the ability to generate a variable draft surface from a selected profile. The variable draft surface may not be always tangential to the original surface, as it has to generate the draft from a fixed defined profile.

1 Select the **composite curve**.

2 Select **Draft Surface** from the **Surface** toolbar.

3 Select the extruded surface and make sure that the arrows are pointing **down** and **out**.
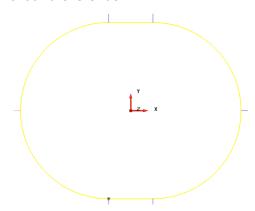
4   Enter a **Draft Angle** of **3** and select **Variable.** ☑ Variable

5   Select **Apply**.

The **Variable Draft** dialog is displayed. This is an interactive command where lines to display the draft surface are generated by clicking on the draft curve. The draft angle is changed at these lines.

6   Select a **View from Top** (down Z).
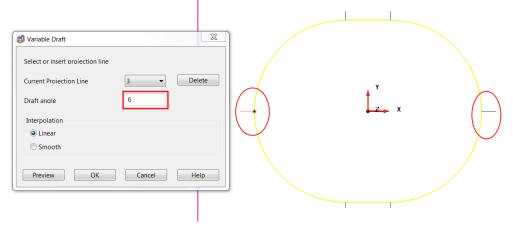
We will create a variable draft of **5** degrees along the sides and **6** degrees around the ends.

7   Click **6 points** around the draft curve as shown above using the intelligent cursor to snap onto **End and Key Points**.
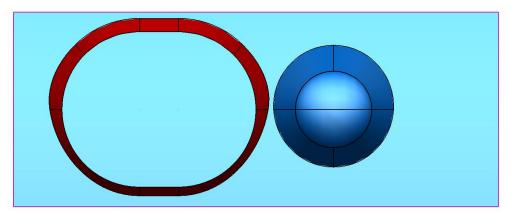
The draft angle for each line can be changed by selecting the line from the drop down list and then changing of the draft angle in the dialog. Selected lines are highlighted in Red.

8   Set both **left** and **right** end projection lines to **6** degrees.

9   Set the remaining **4 lines** to 5˚.

10 Select **Interpolation** as **Smooth,** select **Preview** and select **OK**.

11 Dismiss the dialog.

The Draft Surface is shown.

12 Close the **model** without saving.

Surface Extension
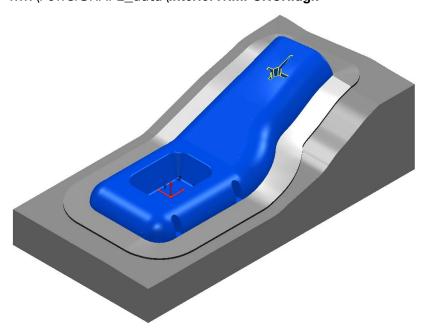
A **surface extension** can only be created from a composite curve that has been driven along, and is still in contact with, the edge of one or more surfaces. This causes the composite curve to inherit the tangencies of the surfaces, which will then control the directional flow of the surface extension. A typical application for the surface extension is in the creation of electrodes or slides, which must extend tangentially, outwards from the main, tool model.

Electrode Example

The imported model of a punch insert will be used to illustrate the application of surface extension as the main option to create an electrode model.

1 Select **File > Import** and select the model:
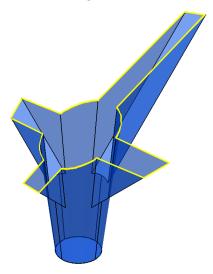
.....\PowerSHAPE_data**InteriorTrimPUNCH.dgk**

2 To **re-orientate** the model in the correct tooling alignment, activate the workplane named **datum**.
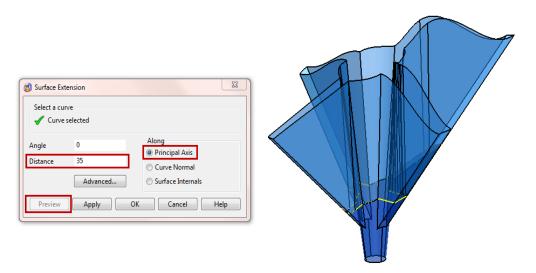
3   Zoom into the rib area and **Blank** all surfaces outside the rib detail.

4   Drive a **composite curve** around the **top edge** of the rib detail.

The composite curve inherits the tangencies from the host surfaces. For this reason it is essential that the actual rib recess surfaces are used and not the main body surfaces when creating the composite curve.

5   Select **Create a surface extension**.
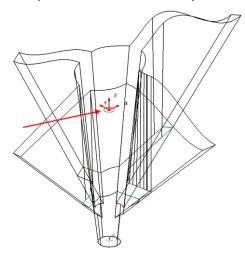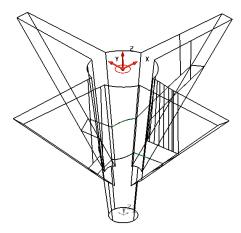
6   In the **Surface Extension** form, select **Along Principle Axis** and enter **Distance 35**.

7   Select **Preview** to display the proposed surface extension.

8   Select the circular fill-in surface at the bottom of the rib form and create a **workplane at centre of selection**.

9   Name the new active workplane as *Electrode*.

10 Create a **Temporary Workplane** at **0 0 35** (relative to the currently active workplane named *Electrode*).
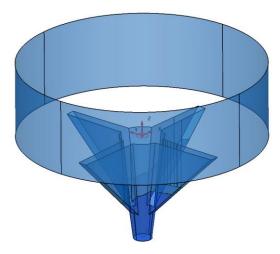
11 Limit the extension surface using the temporary workplane to leave the result below.
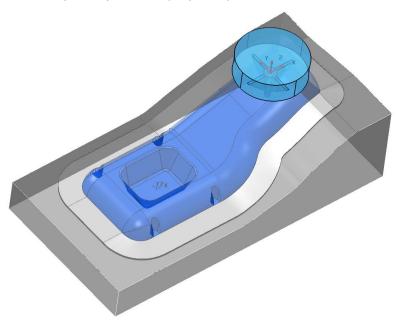
The electrode will be completed.

12 Create a **Primitive Cylinder** surface of **Radius 40** and **Length 25** from the temporary workplane (top of the extension surface) and **central to the circular fill-in surface** at the bottom of the rib.

13 Create **Fill-in** surfaces for both **top** and **bottom** of the cylinder.

14 **Unblank** (**Ctrl+L**) to redisplay the punch insert.
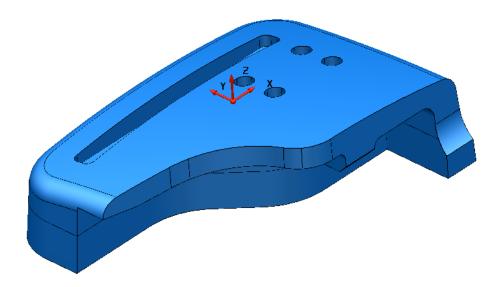
15 **File >Save as**

...../PowerSHAPE-Models/**Trim punch.psmodel**

Model Preparation for Machining

1   **Create** a **New Model** and **Import**l:-

...../PowerSHAPE_Data/**MC-Preparation1.dgk**
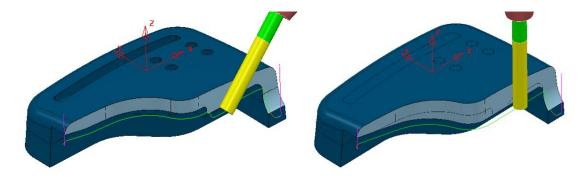
2   **Save** the **model** as:-

...../COURSEWORK/PowerSHAPE-Models/**MC-Preparation-EX1**

In this example the imported **model** will undergo modifications to make it easier to apply machining strategies in **PowerMILL**:-

a   The *Slot* and *Holes* in the upper form will be capped with new, individual **surfaces** to make it easy to prevent cutting tools from plunging into these pockets.

b   The separate, front face surfaces will be replaced with 3 more suitable **surfaces** designed to provide the *PowerMILL* 5-axis **Swarf Finishing** strategy with a smoother tool flow.
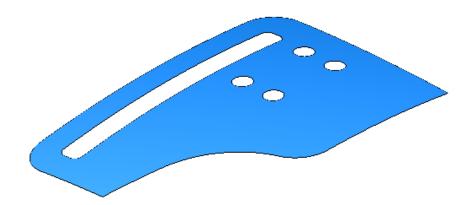
Swarf M/C using Original Surfaces   *Swarf M/C using modified Surfaces*
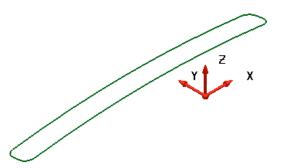
Creating Capping Surfaces

3   Select the upper form **surface** and apply **Blank Except** (*Ctrl K*).

4   Position the cursor on the edge of the slot and then press the **ALT** key to create a **Composite Curve**.
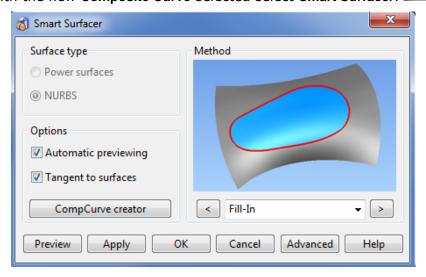
*When a **Composite Curve** is created along a **surface** edge it inherits the tangencies. These tangencies will subsequently be applied when the capping **surface** is created.*

5   From the **main** toolbar activate the **Surface** options toolbar.
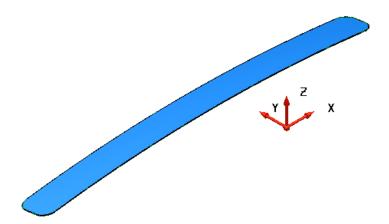
6   With the new **Composite Curve** selected select **Smart Surfacer**.
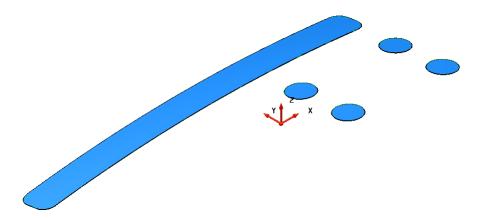
The **Smart Surfacer** will automatically select the most likely valid option which in this case is **Fill-In**.
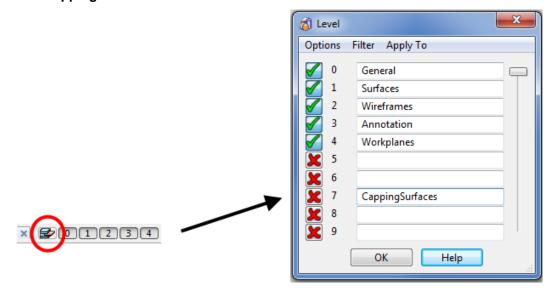
7   With the **Fill-In** option selected **Apply** followed by **OK**.

8   Use the same process to create separate capping **surfaces** for each of the 4 circular holes in the upper form.

9   Select and **Delete** all 5 **composite curves** used to create the capping **surfaces**.

10 Open the **Levels** dialog (bottom left of graphics area) and input the name **CappingSurfaces** on the row for **level 7**.

11 Select **OK** to close the **Level** dialog.

12 With the new capping **surfaces** selected *middle mouse* click on the **Level 7** in the *levels toolbar* located below the graphics area.
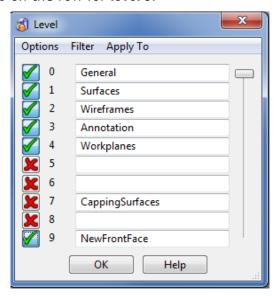
13 Left mouse click on the *level 7* box to re-display the newly acquired *capping* **surfaces**.
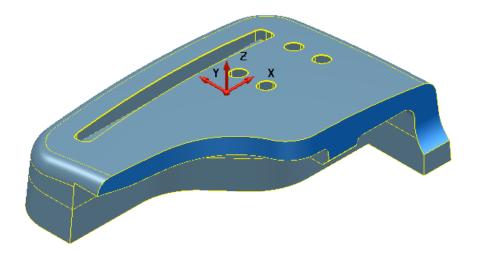
*On import to **PowerMILL** the **capping surfaces** will already exist on a separate **level** named **CappingSurfaces**.*

Creating improved Swarfable Surfaces
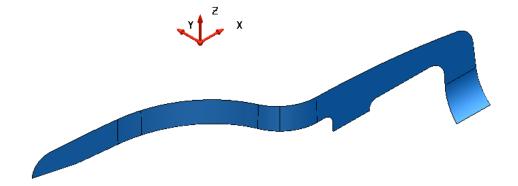
14 Open the **Levels** dialog (bottom left of graphics area) and input the name **NewFrontFace** on the row for **level 9**.
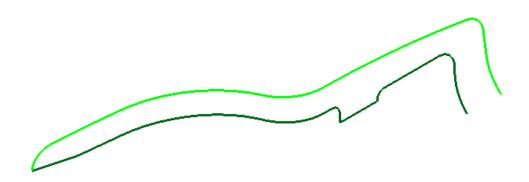
15 Switch *Level 9* on (green tick) and select **OK** to close the **Level** dialog.

16 Select all the **surfaces** except those that define the front edge of the model.

17 Select **Blank selected** (**Ctrl+J**).
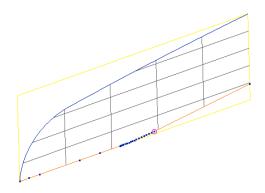
18 Create separate **Composite Curves** along the upper and lower edges of the *Front Edge* **surfaces**.

19 Select all of the *Front Edge* **surfaces** and acquire them to **Level 9** (In the *Level toolbar*, middle mouse click on **9**).

20 Right mouse click on the left hand **surface** and from the local menu select **Surface Trim Region Editing**.

21 In the dual toolbar select **Pcurve edit mode**.
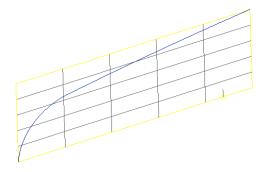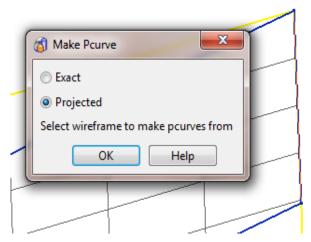
22 Select all the **lower edge Pcurves** and select the **Delete a pcurve** option, as above.
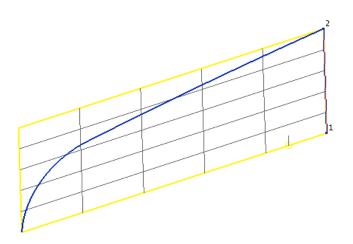
23 Currently there is no **Pcurve** running up the right hand edge of this face. We can create one using a wireframe, and then convert it to a **Pcurve**.

24 Create a line connecting the upper and lower edges of the surface.

25 With the **face** selected, **make Pcurve from projecting wireframe** from the Pcurve toolbar.

26 Choose **projected** and select the wireframe line, then select **OK**.
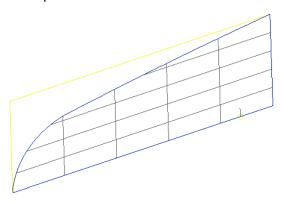
27 Select the new **Pcurve** running up right hand edge and **Turn point labels on**.
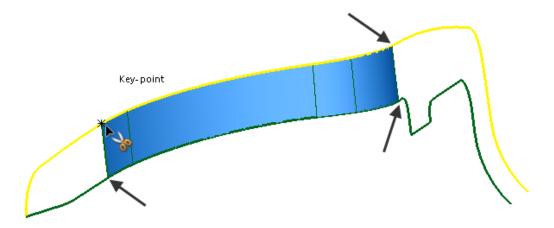
28 In the dual toolbar select **Boundary edit mode**.
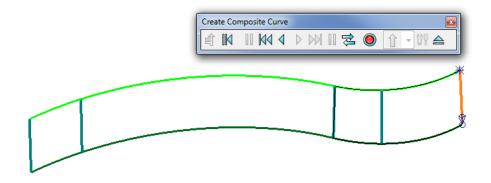
29 Select the **Autocreate** option and then close the toolbar.

30 **Blank** all items except the two *Front Edge* **composite curves** and the **2nd** and **3rd surfaces** from the left.

31 From the **general edits** toolbar use **Cut object** to split the 2 **composite curves** at the 4 positions shown in the above illustration.
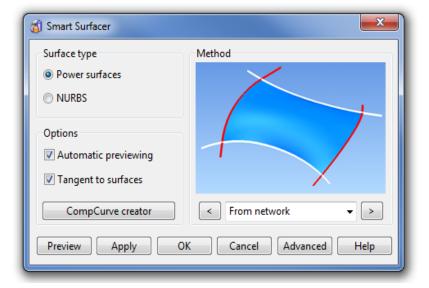
32 Create **composite curves** down each of the existing **surface curves** (on the original **surfaces**) running from the upper edge to the lower edge.

33 **Delete** the 2 existing **surfaces** and then select all the remaining **composite curves**.
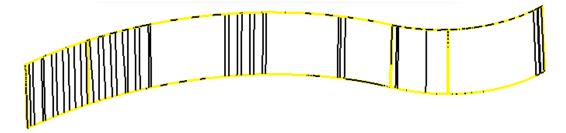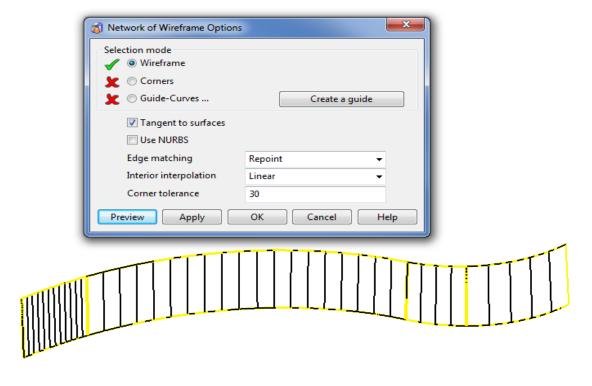
34 Select the **Smart surfacer** and **Preview** the **From network** option.
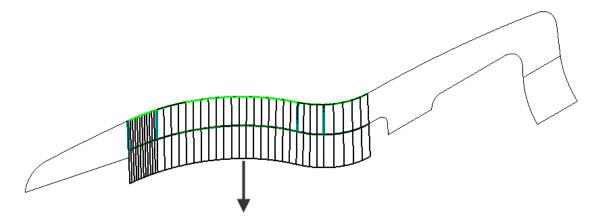
35 Select the **Advanced** tab and in the new dialog use *Edge matching* as **Repoint** and *Interior interpolation* as **Linear**.
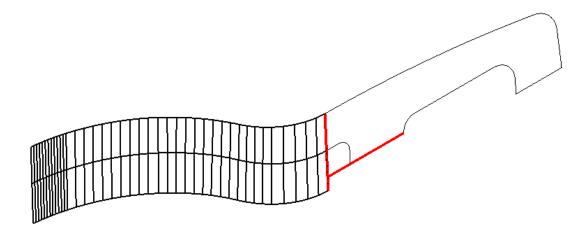
36 Select **Apply** and **OK** to create the new **surface.**

37 From the **general edits** toolbar use **Point limit** to drag (extend) the lower edge of the new **surface** downwards (below the downward step in the original form).
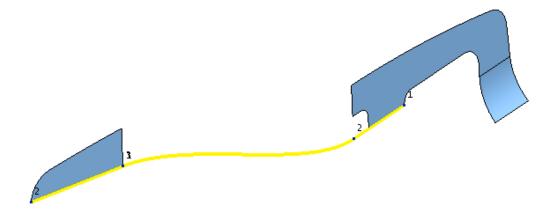
38 Select **Unblank** (Ctrl L).

39 Create a **line** down the right hand side of the middle surface.

40 Create and extend a **line** along the lower edge of the step in the right hand **surface**.

41 Trim the horizontal **line** back to the **line** running up the right of the angled, middle front face, **surface**.

42 Select **Unblank** (Ctrl L) and then **Blank** all items that will get in the way of the next action.
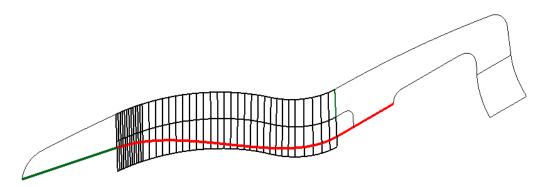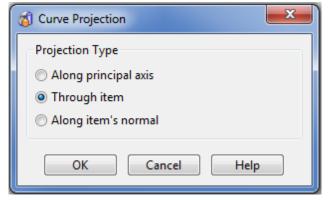
43 Drive a **composite curve** along the lower edge **line** along the step in the right hand surface, jump the gap *tangentially* onto the **curve** running along the lower edge of the recently modified, left hand **surface**.
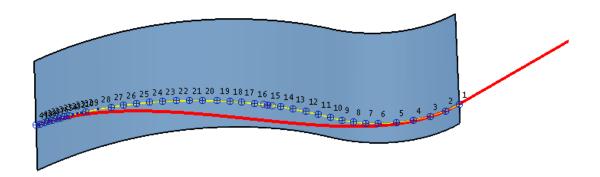
*No need to drive the new **curve** along the lower edge of the left hand surface.*
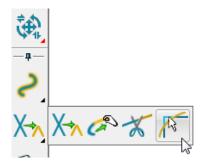
44 Set **Y** as the operational axis.

45 Select both the new **composite curve** and the new central **surface**.

46 In the **Curves** toolbar select **Curve Projection** .

47 Select **Through item** and *Project* a copy of the new **composite curve** along **Y** onto the new central **surface**.
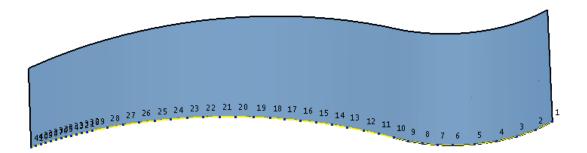
48 With the new **curve** selected (Cutting Object) open the **general editing** - **Limit Selection** toolbar.
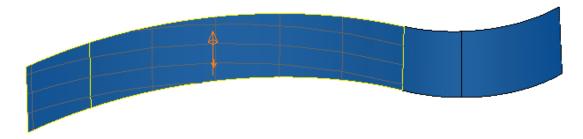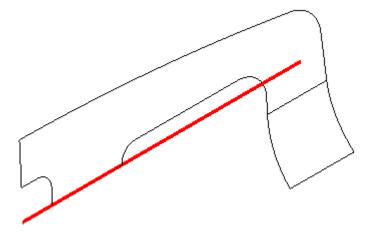
49 Using the left mouse key, click on the **surface** somewhere below the **curve**.
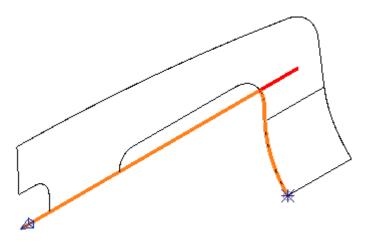
50 If you haven't already, select and **Delete** the 2 original, front face **surfaces** (shown below).
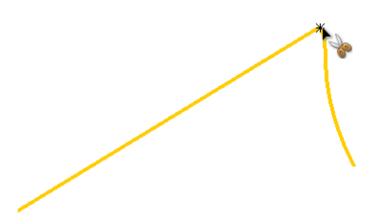
51 At the bottom of the downward step on the right hand, front face surface extend the **line** to the right (as shown above).

52 Create a new **composite curve** (as shown above).

53 Cut the new **composite curve** (as shown below).
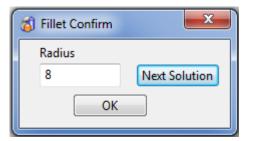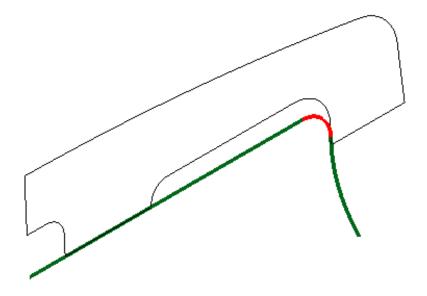
Key-point
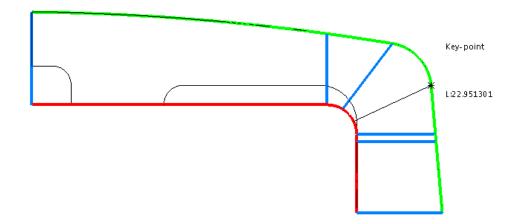
54 In the **Curves** toolbar select **Create a fillet** , **click** on each **curve** in turn near to the **intersection**, and input **Radius 8** before selecting **OK**.
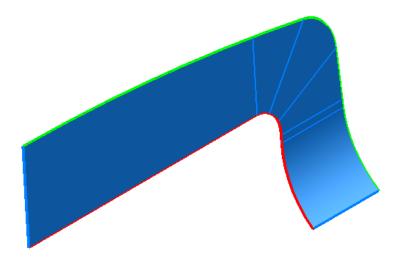
55 Create a single **composite curve** along the above 3.

56 **Unblank** (Ctrl L) all items.

57 Select the original 2 right hand **surfaces**, the **composite curve** running along the outer edge (green), and the new **composite curve** (red)**.**
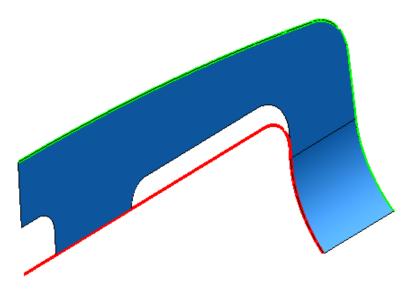
58 Input strategically positioned **lines** (shown blue) to enable the alignment of a cutting tool to the **surface curves**, during a *PowerMILL* **Swarf Finishing** strategy.
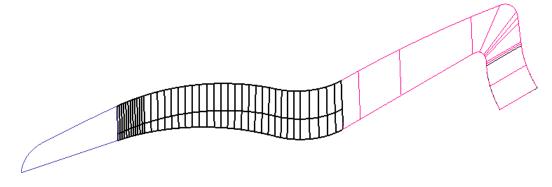
59 Select the above 2 **composite curves** and **lines** and use the **Smart Surfacer** to create a *From network* surface.

60 Select and **Delete** the original 2 **surfaces** (shown below) that will be replaced by the above new one.

61 **Unblank** (Ctrl L) all items.

62 Select and **Blank** (Ctrl J) all **wireframes**.
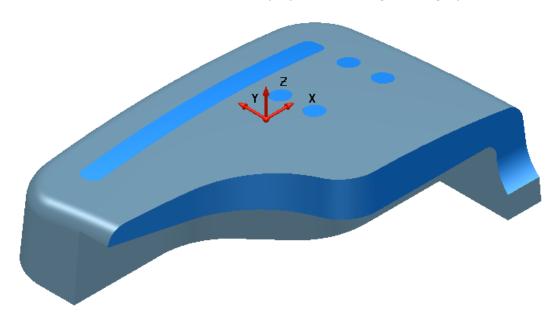
*The above selection of 3 new **surfaces** will provide a far better solution if they are to be used in a **PowerMILL Swarf Finishing** strategy.*

63 Select these 3 surfaces and add them to the **NewFrontFace** level using the **middle mouse button** over Level 9.

64 Switch all **Levels** back on to restore the rest of the original **surface model** back to the graphics area.

65 Select and **Blank** all **wireframes** to display the following in the graphics area.

66 **Save** the **model** to update:-

.... /PowerSHAPE-Models/**MC-Preparation-EX1**

Surface Model Fixing

Model Fixing Toolbar

The **Model Analysis** toolbar is opened by clicking the second button along the row in the toolbar opened by moving the cursor over the **General Edits** option.

The left toolbar contains the **Model Fixing** options. It is shown below repositioned to a horizontal alignment.
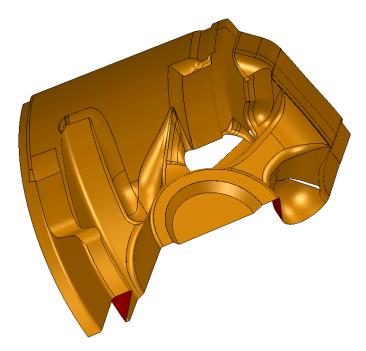
Piston Crown Example

67 **Create** a **New Model** and **import**:

…/PowerSHAPE_Data/**PistonCrown-Start.igs**

The imported model contains surfaces representing a quarter segment of a piston crown. The model is incomplete: it requires a few gaps to be filled, some re-trimming of existing surfaces, and the identification and transfer of missing data from another model.

Before applying further surface modelling, it is necessary to identify and delete any duplicate surfaces and to identify and repair any badly trimmed surfaces

68 **Shade** the model.

69 the model as:
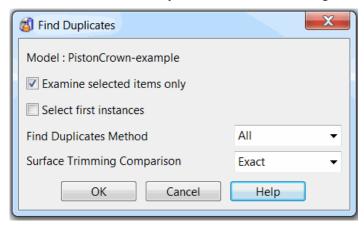
…/PowerSHAPE-Models/**PistonCrown-Example**

Find Duplicates
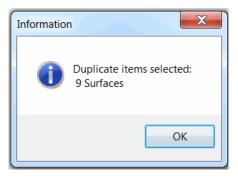
1 **Select** all surfaces. Move the cursor over the **General Edits** option to open the local toolbar and left-click **Model Fixing** to activate the toolbar options.

From the **Model Fixing** toolbar, select **Find Duplicates**.

The **Find Duplicates** dialog appears. This provides the option to choose what to compare and the ability to compare all, exact, trimmed, or untrimmed surfaces. Select the **Help** button for a full listing.

2 Select the **Examine selected items only** box.

3 In the **Find duplicates method** list, select **All**.

4 In the **Surface trimming comparison** list, select **Exact**.

5 Select **OK**. The **Information** dialog displays the number of duplicate items that have been identified and selected. In this case, 9 duplicate surfaces have been found.

6 Select **Blank Except** (**Ctrl+K**). Only the selected duplicate surfaces are shown in the graphics area.

7 Select **Delete** to remove the selected duplicate surfaces.

8 **Unblank** (**Ctrl+L**) to display all surfaces hidden from the view earlier.
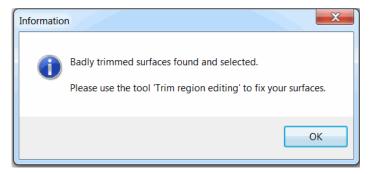
Identify badly trimmed Surfaces

The next action is to check the model for any badly trimmed surfaces. These are surfaces that contain invalid boundaries. These could be due to pcurves not forming a closed loop, or geometrical faults along one or more of the pcurves.
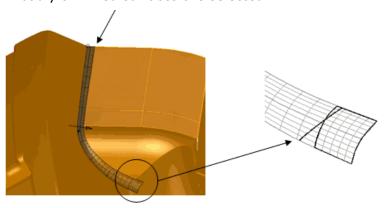
9 **Select** all surfaces.

10 From the **Model Fixing** toolbar, select **Identify Badly Trimmed Surfaces**.

The **Information** dialog opens advising of surfaces with an incorrectly defined trim boundary.

All badly trimmed surfaces are selected:

11 Select **Blank Except** (**Ctrl+K**).
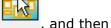
12 From the **Model Fixing** toolbar, select **Surface trim region editing.**

The boundaries are now visible, but require rebuilding because they have failed to trim the surface.

13 From the **Boundary Edits** toolbar, select **Select all boundaries** , and then
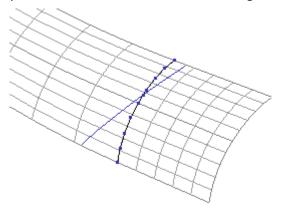
select **Explode** to delete all boundaries.

14 Toggle into the **Pcurve Edits** toolbar.

The pcurve shown as a dotted line does not follow the correct trim, and it must be deleted from the surface.
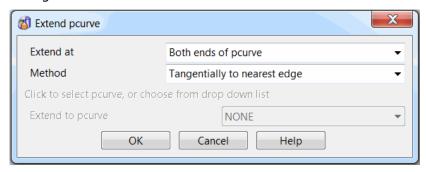
The other pcurve follows the correct trim path, but does not reach the top edge of the surface. Because this would result in a gap in the trim path, the pcurve must be extended to the edge before a boundary is created.

15 Select the pcurve shown above as a dotted line and select **Delete**.
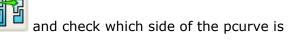
16 Select the remaining pcurve and select   to open the **Extend pcurve** dialog.

Extend pcurve		
Extend at	Both ends of pcurve	▼
Method	Tangentially to nearest edge	▼
Click to select pcurve, or choose from drop down list		
Extend to pcurve	NONE	▼
	OK Cancel Help	
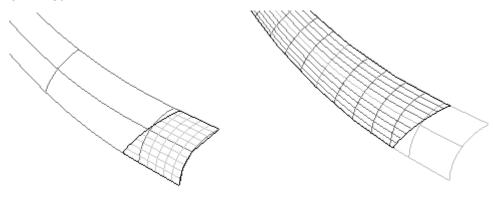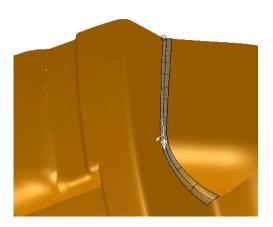
17 In the **Extend at** list, select **Both ends of pcurve** and in the **Method** list, select **Tangentially to nearest edge**.

18 Select **OK**. The pcurve is extended to reach the edge of the surface.

19 Toggle into the **Boundary Edits** toolbar.

20 Select **Auto Create boundary** and check which side of the pcurve is trimmed.

21 If the trim appears as above left, select **Reverse the boundary** to change the trim to that shown above right.

22 Close the **Boundary Edits** toolbar. The surface model is now correctly trimmed.
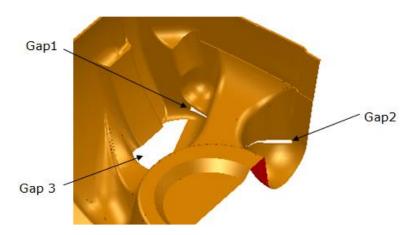
Repairing Gaps and Overlapping surfaces
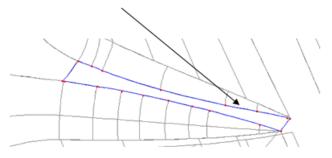
By shading and dynamically rotating the view of the cavity model, you will see three that require patching.
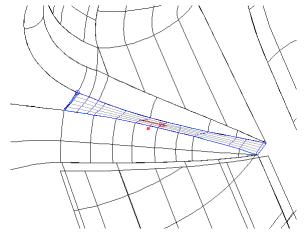
Filling in Gap1 – Surface from Network of curves

23 **Zoom** into the area labelled *Gap 1*.

24 Create **a composite curve** using the edges of the gap.

A patch surface from a network of curves is generated from the new composite curve.

25 Using the **Smart Surfacer** , create a fill-in surface.
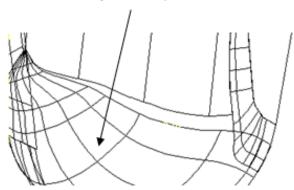
Filling in Gap2 – Stitch surface

26 **Zoom** into the area around *Gap 2*.

The bowl surface is a **NURBS surface**, which must first be converted to a **Power Surface** before editing can be performed. This gap will be filled by using the **Stitch surface** option.
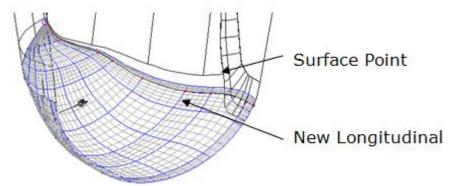
27 Right-click on the bowl surface, and, from the popup menu, select **Convert Surface** to change it to a power surface.

Stitching moves the selected surface points to match the edge of other surfaces or composite curves. It is not possible to apply **Stitch surface** from a trimmed edge. The process benefits if further laterals are inserted to provide a more accurate edge-match for the **Stitch surface** operation.
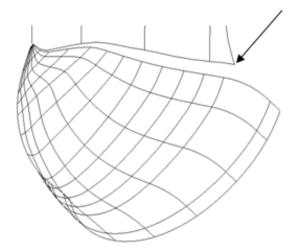
An easy way of inserting an additional lateral is to select a surface point, followed by holding down the **Ctrl** key and then left-clicking. A new lateral can then be dragged along an existing longitudinal. This is repeated until there are sufficient surface points along the edge to be stitched.

28 Select the **surface**. The **Surface Edits** toolbar is displayed.

29 Select a lateral/longitudinal point and drag a new longitudinal near to the end one using the **Ctrl** key and left mouse button, as shown below.

30 **Repeat** the process of dynamically adding laterals to provide an even point distribution along the surface edge to be stitched, without going beyond the end of the adjacent surface edge.
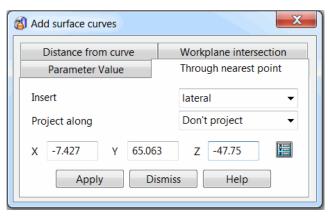
A more accurate method of adding a lateral will be used to match the end of the adjacent surface edge.

31 Select **Add Curve**.
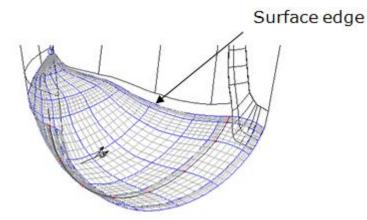
The **Through nearest point** tab allows you to pick a point on the model. When picked, the **XYZ** co-ordinates are loaded into the dialog.

32 Select the **Through nearest point** tab. In the **Insert** list, select **Lateral**.

33 Click on the surface point indicated earlier at the end of the surface to be stitched to.

34 In the dialog, select **Apply** and then select **OK**. The new lateral is generated.

Surface edge
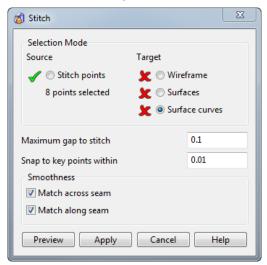
The **Stitch** command uses the selected surface curve and matches it along the surface edge of a selected secondary surface. It is also necessary to specify a value for the maximum gap to stitch across.

35 Select the edge (end longitudinal) of the lower surface and select **Stitch**.

The **Stitch** dialog appears. It is essential to estimate and specify a value for the **Maximum gap to stitch**. This is used to eliminate any ambiguity about which surface edge to stitch onto.
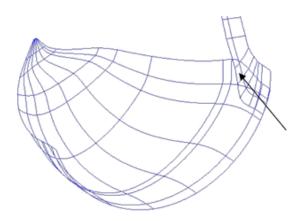
36 Enter **2** as the **Maximum gap to stitch**.

37 Pick the edge surface on the edge curve to stitch onto, **Surfaces** is selected.

38 Select **Preview** to view in advance, and then select **Apply** to perform the stitching.
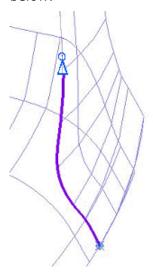
The fillet surface needs to be trimmed back to the stitched surface to finish off this area.

39 Select the two surfaces shown and **Blank Except** (**Ctrl+K**).
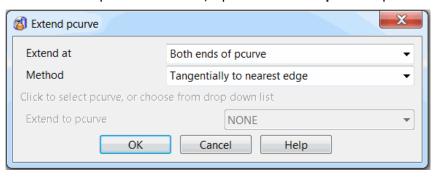
40 Create a composite curve from the edge curve of the flange surface as shown below.

41 Use **Define Start and End points** on the **Composite Curve** dialog to create the curve shown.

42 Select the lower surface and enter into **Trim Region Editing**.

43 In the **Pcurves** toolbar , select **Make pcurves from projecting wireframe** .

44 In the dialog, select **Projected,** and then click on the composite curve and select **OK**.

45 With the new pcurve selected, open the **Extend pcurve** option .

46 Select:

Extend at - Both ends of pcurve
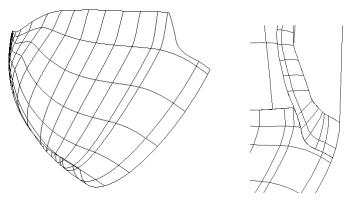
Method - Tangentially to nearest edge

47 Select **OK**.

48 In the **Boundary** toolbar , select **Autocreate** .

49 If required, select **Reverse the boundary** to achieve the following trim:
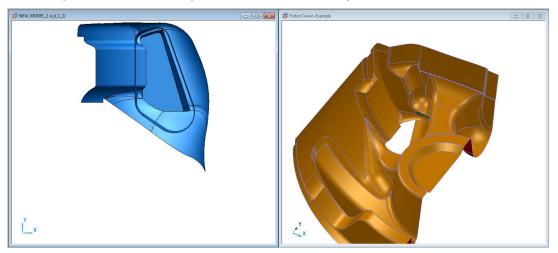
The bowl surface is now trimmed neatly to the flange.

Selection and Import of Surface Data for Gap 3

50 **Unblank** all of the surfaces.

Gap 3 is a large hole in the model, due to missing data that will be supplied later from an IGES file. By loading it into a new model, it can be checked before copying the data into the original piston model.

51 **Create** a **new model** and **import** the file …/PowerSHAPE_Data/**cyl_1_2.igs.**

52 Rearrange the models using **Window > Tile Vertically**.

Most of the surfaces in the new model already exist in the original PistonCrown-example.

Model Compare
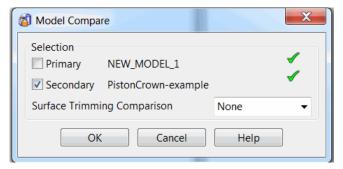
Model Compare analyses two models and selects items on the first model that are different or that do not exist on the second model. The trimming options can be specified in the dialog.

53 From the **Model Analysis** toolbar, select **Model Compare**.
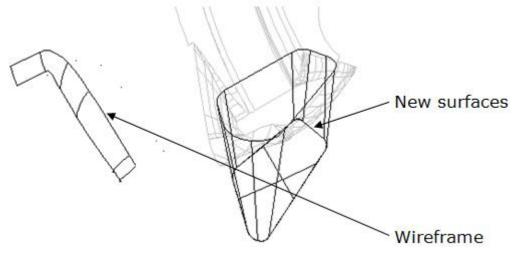
The **Model Compare** dialog is displayed. This allows you to choose which model is the primary (old) model, and which model is the secondary (new) model. The different model entities are then selected in the primary model.

54 Select **Primary** and then left click in the **New Model** window.

55 Select **Secondary** and then left click in the **PistonCrown**-**example** window.

56 In the **Surface Trimming Comparison** drop-down list, select **None**.

57 Select **OK**.

Some wireframe and two surfaces are highlighted in the **Primary - New Model** window.

58 **Blank Except** (**Ctrl+K**) the selected items.

59 Select and delete the wireframe.

60 **Select** the **extruded surfaces**, and then select **Copy Selection**.
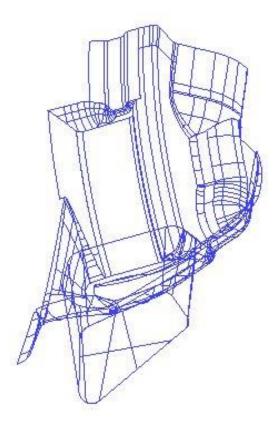
61 **Select** the model **PistonCrown** and select **Paste from clipboard**.

A copy of the new, different surfaces appears in the original model.

62 **Close** the new **model** leaving the **PistonCrown-example** model **open**.

Filleting

The deep pocket will be **filleted** to the main surfaces with a radius of **3** and the bottom-edge of the pocket will be filleted with a radius of **2**.
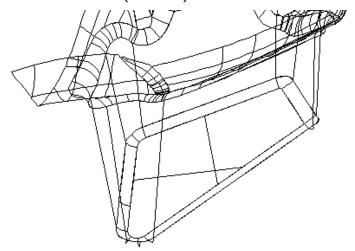
📝 *The creation of the **fillet** of radius **3** does not trim the tapered pocket or some of the adjacent surfaces correctly due to poor surface-edge matching across the original model. Surface editing of the fillet surface along with pcurve and boundary editing on the tapered pocket and some of the adjacent surfaces must be applied to fix the problems*
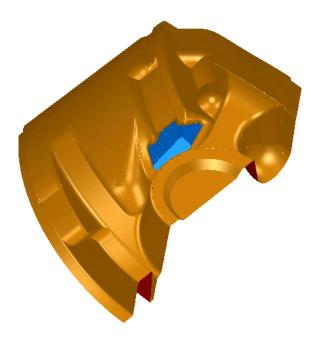
63 Select the newly copied surfaces and those that intersect them.

64 Select **Blank Except**.

65 Generate a concave fillet (radius **3**) between the original surfaces and the top of the new tapered pocket surface.

66 Select the taper surface and the surface defining the flat base of the pocket.

67 Create a convex fillet (radius **2**) between these surfaces.

Now that the quarter of the Piston Crown is complete, a workplane will be created to provide a datum to mirror a copy of the surfaces to end up with half of the full Piston Crown.
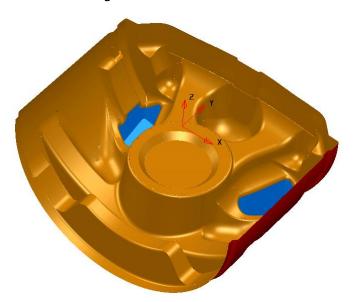
68 Select **Unblank** and create a workplane at 0.

69 Select all the surfaces.

70 From the **General Edits** toolbar, select **Mirror**.

71 From the **Mirror** toolbar, select **Mirror in YZ** and **accept** .

72 Close the dialog.

73 Select **File > Save** to update the model.

74 **Close** the **model**.

Further Model fixing

Imported IGES file of a PC Front Cover surface model

When you import a model saved in IGES format, it contains NURBS surfaces, which have a restricted editing capability. As will be shown later, when a NURBS surface is converted an excessive number of laterals and longitudinals can be generated on the resultant power surface. It will also be observed that several of the surfaces are in an orientation with the inner (red) side on the external faces of the model. Apart from the visual presentation, this causes problems if the model requires additional surfacing work, such as filleting.
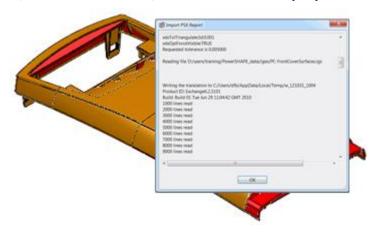
75 **Import** the model:

…/PowerSHAPE_Data/**PC_FrontCoverSurfaces.igs**

A dialog appears displaying the progress as an **Import IGES report**.

76 **Save** the model as:

…/PowerSHAPE_Data/**PC-FrontCover-example.psmodel**

77 Select **all** surfaces in the model.

78 From the **Editing** flyout, select the **Model Fixing**
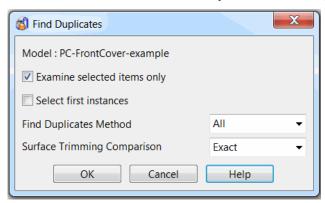
79 From the **Model Fixing** toolbar, select **Find Duplicates**.

The **Find Duplicates** dialog is displayed. Various options allow you to control the criteria of the comparison, and whether to consider exact, trimmed or untrimmed surfaces. Select **Help** for a full listing.
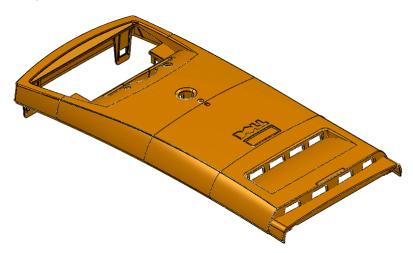
80 Select **Examine selected items only**.

81 Select **All** and **Exact** and then select **OK**.

In this case, no duplicate data is found, but it is still **essential** to perform the test

82 From the **Select** flyout, select the **Quick Select all Surfaces**.

83 From the **Model Fixing** toolbar, select **Orient surface normals** to arrange for the outer faces of the surfaces in the selection to be on the outside of the component.
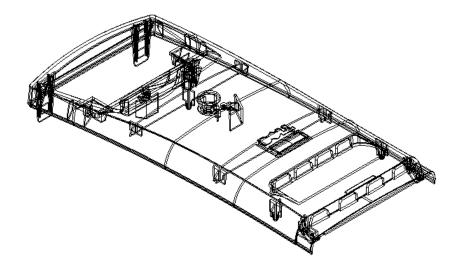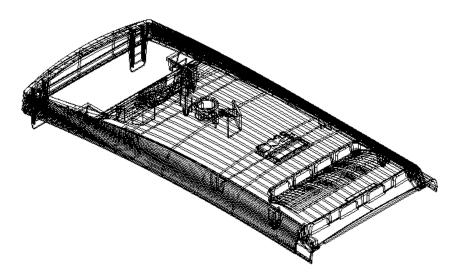
In some instances, it may not be obvious which way a surface should be orientated (such as when the model is missing data causing a surface to be isolated, or when the outer face of a surface exists on both the inside and outside of the model). If any surfaces remain in the wrong orientation, they can be selected and reversed using the local menu.

84 Display the **surface model** as **wireframe**.
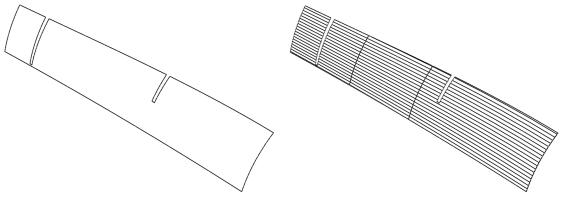
85 With all the surfaces still **selected**, right click over them and select **Convert Surface** from the popup menu to change the **NURBS** surfaces to **Power Surfaces**.

The following diagram shows a surface before and after conversion to a power surface.
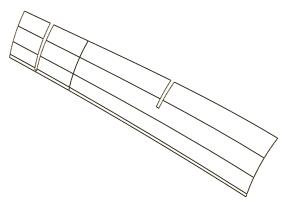
*Depending on the type of file originally translated into IGES format, power surfaces can be created with an excessive number of surface curves (as shown above right). This can result in increased processing times and excessive memory consumption. It is good practice to apply the **Approximate Surface** option to reduce the amount of curves while keeping the surface definition within a specified tolerance value.*

86 With all power surfaces selected, select the **Approximate Surfaces** option

 from the **Model Fixing** toolbar.
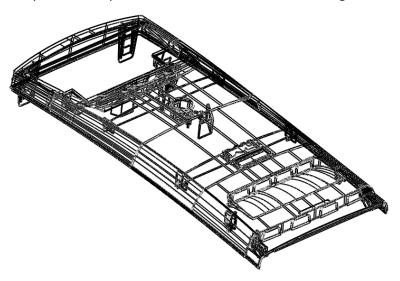
All surface curves are deleted that are not required to keep the model within the general tolerance value (default **0.01**). The tolerance value can be modified on the **Instrument** toolbar located below the graphics area.

The model is now ready for further operations such as constructing the components required for a suitable mold tool design.

Core/Cavity separation

This **Core/Cavity separation** option is located in the **Wizards** options of the **Main** toolbar. It operates on the selected surfaces and discriminates surfaces that are out of view if looking down the Z axis.
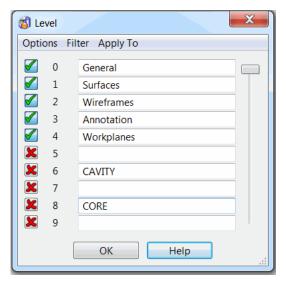
Depending on which option is selected, it selects the outer or inner surfaces so they can be transferred to another level.

87 Open the **Levels** dialog.

88 Enter the name *CAVITY* for level **6** and *CORE* for level **8**. Leave these two levels switched off.

> *If a name is assigned to a level, it appears in the **Levels** toolbar at the bottom left of the graphics area.*

89 Select **all** of the surfaces in the **model**.

90 Open the **Wizards** options, and click on **Core/Cavity separation split visible**.

The surfaces that are hidden from view when looking down the Z axis become deselected while the others remain selected.

91 While the upper surfaces are still selected, middle mouse click on level **6** on the **Levels** toolbar. Level **6** is switched off and the surfaces disappear from view.

92 **Select all** the **remaining surfaces** and middle mouse click on level **8**. Level **8** is switched off and the surfaces disappear from view.

93 **Toggle** levels **6** and **8** on and off while checking that the surfaces have been assigned to the correct side of the molding.

LEVEL 6 - CAVITY   **LEVEL 8 – CORE**

11. Shading & Model Analysis
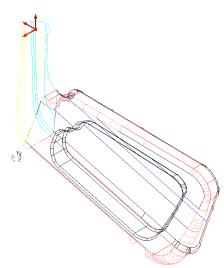
Basic Shading Options

Basic shading is displayed by clicking the various icons in the **Views** toolbar to the right hand side of the **graphics area**. As **surfaces** must be identified with **inside** and **outside**, the **outside** is displayed in a **user-defined colour**, and the **inside** as **red**. It is possible to switch the inside colour off so that both sides are shaded with outside colour.

> *This also works very similarly when working with **open solids**.*

1 **Import** the model.

   .....**Training Data\\PowerSHAPE Data\\golf_fin.dgk**

2 **Select** and **delete** all wireframe entities.

3 **Activate** workplane **2**.

4 Select **Wireframe** view.

The **model** is **displayed** with the **surface skin invisible**, enabling **visual access** to the **trimmed edges** and **surface curves**.

5 Select **Shaded view**.

The model is displayed with the **surface skin shaded**. The **outside** of a **surface** is the **default** or **user-defined colour**; the **inside** is **shaded red**. The use of shading **restricts visual access** to **internal detail**.

6 Select **Toggle Transparent wire**.

This view shows the **model partially shaded** (**transparent**), which provides a compromise with **visual access** to the inner and **wireframe detail**.
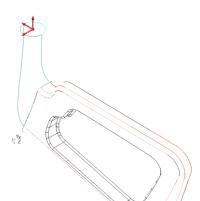
7 Select **Shaded wire view**.

This view is a **shaded view** with the **surfaces wireframe curves displayed**.

8   Select **Hidden wire view**.
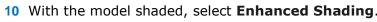
The model is displayed in **wireframe mode** with all the **internal wireframe removed** or **hidden**. This option helps you see the model clearly. This special view mode must be turned off for surface manipulation.

9   From the **View** menu on the main menu bar, select **Shading>Inside Material** (**if not already ticked**)

This view **enables/disables** the mode which displays **inside-out surfaces** as **red**. This allows the user to **shade** the **model without** having to **reverse inside-out surfaces**. Ensure the option is **ticked** to view **negative facing surfaces**.

10   With the model shaded, select **Enhanced Shading**.

This shows the model in **Enhanced shading** mode. This type of view includes **perspective** and **reflections** on the surface, giving a more **realistic appearance** with model **feature lines visually converging** towards vanishing points.

11 From the **Enhanced shading** toolbar, select **Render the View** . This displays a photo-realistic image of the model in a separate, newly-opened graphics window, using default light settings and render options.

To customise the way that the rendered image is generated, a dedicated options dialog is opened by clicking on the **Raise the render form** option.

12 **Closes** the **image** and **Keyshot** (rendering system used) **window appears**.

13 Select **Shaded view**.

Format > Shading Materials

Shadings materials are accessed from the **Format > Materials...** menu option.

By default, **all surfaces** are **shaded** using the **same material**. However, **individual surfaces** can be **modified** to use a **shading colour** and **material type** from the **standard range** or as **customised** by the user. There is a comprehensive range of pre-set **Material type** options available, such as metallic, glass, and neon.

To change the material of selected surfaces:

1   Ensure that shaded images are displayed

2   Select a surface or group of surfaces.

3   Open the **Materials** dialog and select a new material.

To change the default material:

1   Ensure that no surface is selected.

2   Open the **Materials** dialog and select a new material.

3   Select **OK**. The new material is used for subsequent surface colour shading.
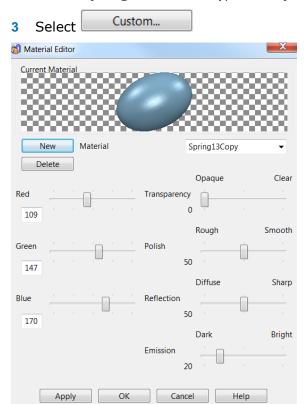
Customising Materials

You can create a **custom material colour** and **texture** to your **requirements**.

1   Select **Format > Materials...** to display the **Material Editor** dialog.

2   Select **Spring** from material type then **Spring 13**. Spring13

3   Select   Custom...
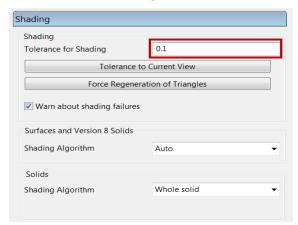
The **Material Editor** dialog is displayed. This allows you to change the shading effect using various sliders.

4   Select **New**. Leave the name as **Spring13Copy**.

5   Change the material using the sliders, and observe the effect on the preview bead at the top of the dialog.

6   When you have created a suitable material, select **Apply**.

7   Select **any surface** and the **material** is changed to the new one.

Shading Tolerance
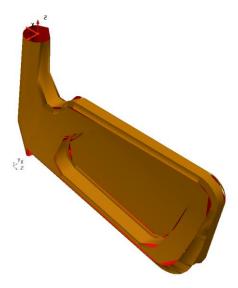
When models are shaded the surface is covered in a triangular mesh, which is set at a certain **tolerance**; the **smaller** the **tolerance value**, the **finer** the mesh. These tolerance values are in the **Shading** options, found in the **Tools** menu.

1   Select **Tools > Options>View>Shading**.

Tolerance to current view is a quick way of entering the ideal value for your screen resolution and current zoom. PowerSHAPE uses the new tolerance when the shading is next regenerated. If you want to see it immediately, select **Force regeneration of triangles.**

2   To exaggerate the effect, set the **Tolerance for Shading** to the value **5mm**.

3   Select **Force Regeneration of triangles** and select **OK**.

The **shaded surface model** appears very **faceted**. If you want to check a surface model for **tangencies** or **ripples along** the surface, it is essential to apply a **suitable shading tolerance**. This depends on the **physical size** of the model, but a more suitable value is **0.01mm** for an accurate visual representation.

4   Set the **Shading Tolerance** to **0.01** then **Regenerate.**

5   **OK** the options page.

Model Analysis toolbar

The **Model Analysis** toolbar is opened by clicking the option located along the row third from the top of the left toolbar.
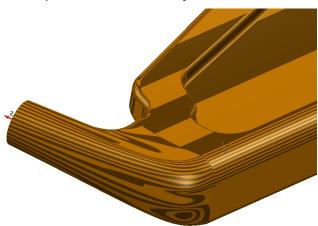
The new toolbar contains the **Model Analysis** options. It is shown below in a horizontal alignment.

Advanced Shading Options

PowerSHAPE has several **advanced shading** options that can be used to **check** and **analyse** models in preparation for programming.

- **Smoothness Shading** exaggerates the degree of smoothness over the model surface with dual coloured stripes. A **break in continuity** along the stripes indicates a **sharp transition** from one face to another.
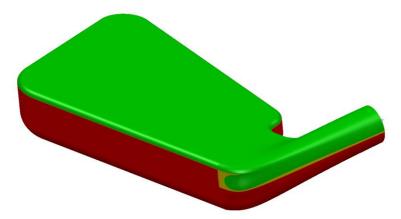
A sharp-cornered transition across adjacent surfaces is indicated by break in the continuity along the stripes.

*Select the **Smoothness Shading** option and take a closer look at the displayed model.*

6 **Undercut Shading** is used to check for correct **draft angle** on a **molding component** or to check for **undercuts**:

- **Green** shading indicates that area of the model surface does not exceed the draft angle alignment.

- **Yellow** shading indicates that area of the model surface is exactly aligned between the **draft angle** and **warning angle**.

- **Red** shading indicates that area of the model surface is undercut.
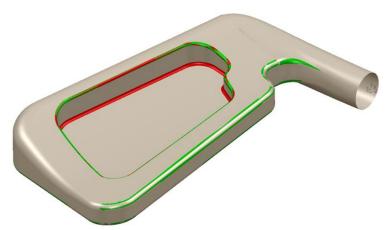
📝   *The above view is using the World datum.*
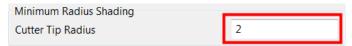
By default, the **Draft Angle** is set to **0° and Warning Angle** is set to **5°**. If you want to use a different value, you can modify the angle in **Tools > Options > Tools > Analysis > Surface Analysis**.

7   Select the **Undercut Shading** option and take a closer look at the displayed model.

▪   **Minimum Radius Shading** displays internal radii with values below the specified minimum radius in red, and external radii in green. The remainder of the model is shown in grey.

This illustration represents **Minimum Radius Shading** using a minimum **radius** of **2mm**.

By default, the **Minimum Radius Shading** value is **5mm**. If you want to use a different value, it can be modified in **Tools > Options > Tools > Analysis > Surface Analysis**.

8   Select the **Minimum Radius Shading** option and take a closer look at the displayed model.

9   **Close** the model without saving.
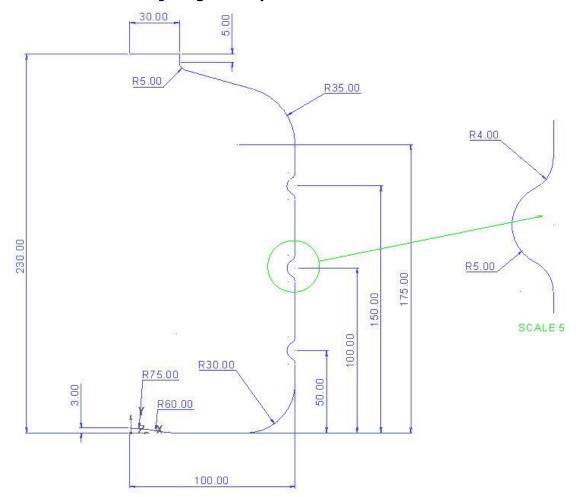
Volumes

PowerSHAPE calculates the **volume** of an **enclosed object** such as a sphere, based upon the direction of the principle plane.

With an **open surface**, **PowerSHAPE** generates a **projected volume**, from the **active workplane** or the **world origin**.
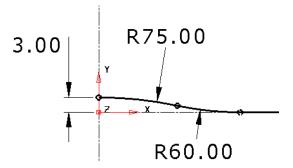
Bottle Volume Example

1   Create a **new model** and **workplane** at 0.

2   Create the following **2D geometry**.
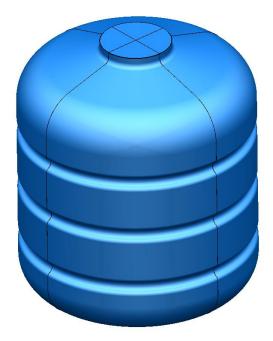
Use the following information for the bottom of the bottle.
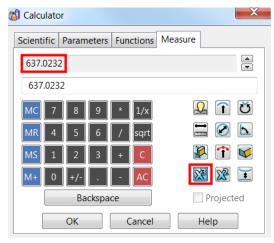
3   Generate a **composite curve** of the wireframe and make a copy.

4   Create a **new level** (**5**) named *Wire Copy* and place this copy onto it.

5 Switch OFF level 5.

6 Create a **surface of revolution** around the **Y** axis.

7 Shade the **surface** and, if necessary, **reverse** it to have the **positive outside**.

8 From the bottom of the window, select the **Calculator**.

9 Select the **Volume** option and then select the surface.

The calculator works out the volume as **637.0232mm³**. To change the volume to **700mm³**, we can expand the surface to the required volume.

10 OK to close the calculator.

11 Select the **surface** and from the **General Edits** toolbar, select **Scale**.

12 Select the **To Projected Volume** option and change the volume to **700**
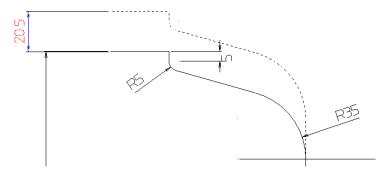
A **new surface** is produced, which is **scaled** in **all three axes**. If the **X** and **Z** movement had been **locked** (by clicking the padlocks), the scaling would occur only along the **Y axis**.

Another way to change the volume is to increase the length of the bottle from just beyond the top recess to the cap. By trial and error, you could get close to the desired volume. The following sequence attempts to reach the target by locally stretching by **20.5**.
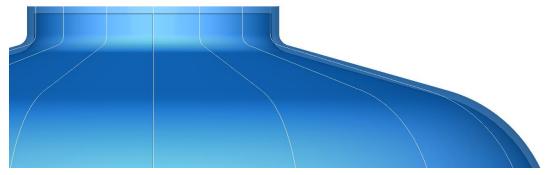
13 Delete the surface and switch on **Level 5 Wire Copy** (do not select it).

14 Select **Stretch Object** in the **Edits** toolbar.
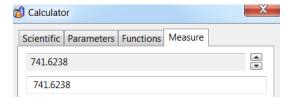
15 Box over all the elements of the composite curve including and above the **R35 Radius** (these will turn to yellow), and enter **0 20.5** in the **Command input** box to stretch this part of the wireframe (as shown dotted below).

16 Generate a new **surface of revolution** (and it reverse if required).

To find the volume between two surfaces and the wall thickness, we can use the calculator.

17 **Offset** the new bottle (keeping the original) outwards by **2**.
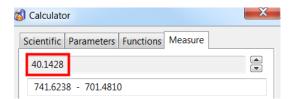
18 Select the **Calculator** and measure the volume of the new larger, offset surface.

19 Enter a minus sign (-) on the calculator.

20 Measure the volume of the original smaller surface by selecting it.

The answer, which is the volume of the wall thickness of the bottle, is displayed in the top box of the calculator.

> Volumes of solids can also be calculated in the same way. The method also works for multiple selected surfaces.

21 File >Save as

\PowerSHAPE-Models\Bottle volume.psmodel
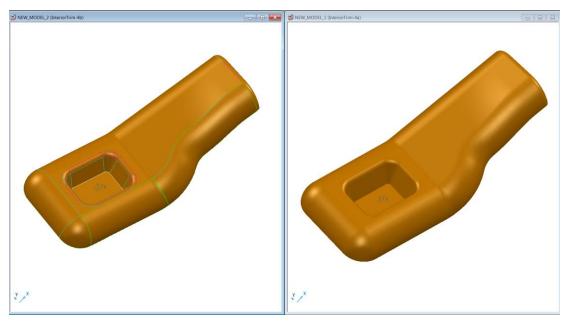
22 Close to close the model.

Identifying Modifications on a new issue model

1 Open a new model, and import:

.....PowerSHAPE_Data**InteriorTrim-4a.dgk**

2 Fill the model to screen to view it.

3 Create another **new model** and **import**

.....PowerSHAPE_Data**InteriorTrim-4b.dgk**

4 Fill the model to screen to view it.

5 From the **Main** menu, select **Window > Tile Vertically**.

The separate models are arranged to co-exist side by side in the graphics area. At the moment, there are no obvious differences, except that the central pockets appear to be different depths.

Dynamic Sectioning
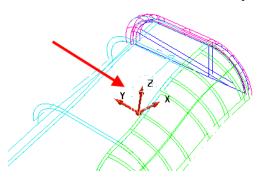
Dynamic Sectioning provides a fast means of taking a slice through a surface or solid model with the option of creating a 2D composite curve as a permanent record of the section.

1 Select the **New_Model_2** window to **activate** it.
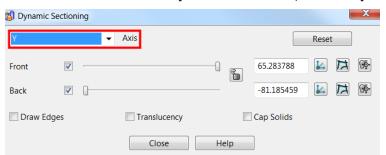
The window border itself turns a darker shade of blue.

2 **Activate** the **RibCentre workplane** as the datum.
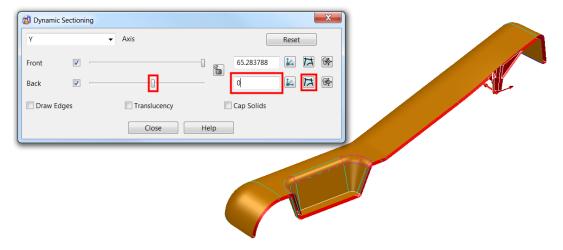
A similar workplane is positioned to a common position on each model, to enable a dynamic section to pass exactly through the same plane.

3 From the **Model Analysis** toolbar, select **Dynamic Sectioning**.
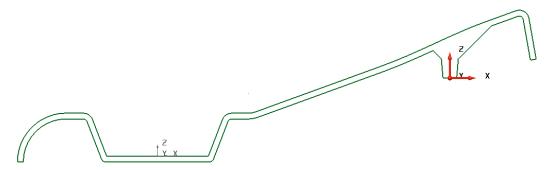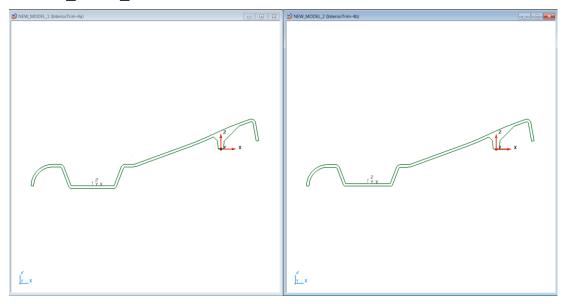
4 Select **X** from the **Axis** list. This controls the direction along which the dynamic section is created normal to.

5 Drag the **Back** slider. The model is cut back as the slider is moved.

6 Input **0** in the data input box at the right of the **Back** slider to make the section line up with the active **workplane**, **RibCentre**.

7 Select the **Create Wireframe** button on the back to generate a composite curve through the dynamic section.

8 **Close** the form and Blank all the surfaces.

9 Repeat the above to create a similarly positioned dynamic section on **New_Model_1.**

Comparing the two curves

At this stage, there is evidence that the central pocket is deeper in New_Model_1. To identify smaller differences, the two curves must be superimposed on each other. It is also necessary to apply other specialist model-analysis options

1   Select the composite curves stored in **New_Model_2** and open item colour.

2   Select a **red** colour and deselect on the page to see the change.

3   Ensure the **RibCentre workplane** is still active in both models.
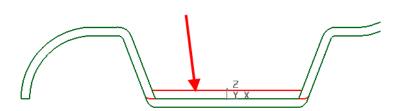
> *This is to make sure that the composite curve produced in New_Model_2 is copied exactly to the same world coordinates as in New_Model_1.*

4   Select the composite curves in **New_Model_2**, then select **Copy** on the **Main** toolbar.

5   Select the window in **New_Model_1** to activate it then **Paste**.
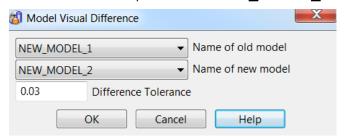
The view illustrates that the surface model stored in **New_Model_2** has a reduced depth central pocket and a slightly larger core form in the rib area to the right.

Model Visual Difference
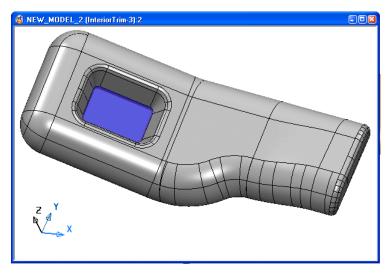
This option provides a fast means of viewing differences between two models. When two models are specified in the dialog, a new window displays the first model in light grey with areas of *material on* coloured blue and areas of *material off* coloured red.

1   In both windows delete the composite curves and unblank the surfaces.

2   From the **Model Analysis** toolbar, select **Model Visual Difference**.

1   In the **Model Visual Difference** dialog, enter *NEW_MODEL_1* in the **Name of old model** box, and enter *NEW_MODEL_2* in the **Name of new model** box.
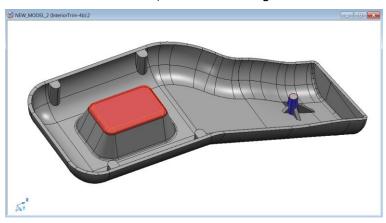
2   Select **OK** to apply the **Model Visual Difference** process. A new window appears containing a light grey image of **NEW_MODEL_2** with extra material shaded blue and removed material shaded red.

The base of the pocket is shaded blue, which confirms that extra material exists in this area.

3 In the new window, rotate the image to view the other side of the model.

4 Close this new window.
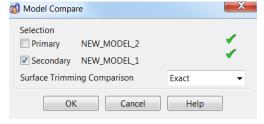
Model Compare

This option provides a means of **identifying** and **selecting** modified, model entities on an updated model while comparing it with the original model. The selected model data can then be copied and pasted into the original model without duplicating existing unchanged model entities.
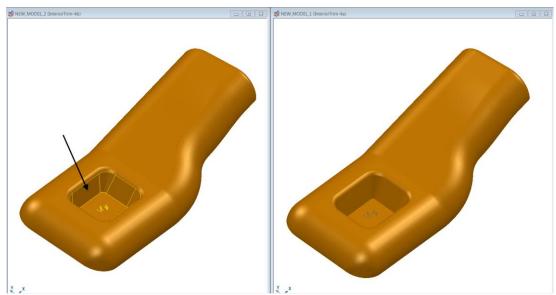
1 From the **Model Analysis** toolbar, select **Model Compare**.

The **Model Compare** dialog appears requiring you to identify the primary and secondary models on which to compare and select the different model entities.

2 Select the **Primary** box, and then click in the **NEW_MODEL_2** window to register it as the primary.

3 Select the **Secondary** box and then click in the **NEW_MODEL_1** window to register it as the secondary.

4   Click **OK** to apply the model compare process.

All entities in **NEW_MODEL_2** that do not exist or that are different from those in **NEW_MODEL_1** are automatically selected (highlighted) ready to be copied and pasted.
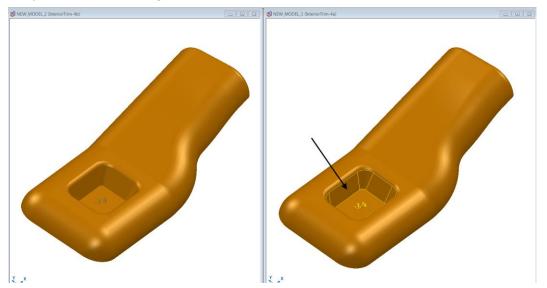
Copy and *Paste* options are included as buttons on the *Main* toolbar.

5   With the new items already selected in **New_Model_2**, select **Copy**.
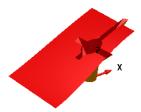
6   Click in **New_Model_1** to activate it, then select **Paste**. The new data is pasted into the original model.

7   Rename **Level 10** *as NewData* and assign the copied model entities to it.

8   Switch **off** all levels except **NewData**.

The inner Rib Data has also been modified and copied to New_Model_1 as well as the pocket.

To assess the manufacturing work required to modify any existing mold tool, the new and old data must be placed on different levels. Switching the levels on and off makes it more obvious whether the new data is material to be removed from or added to the existing tool.

9 Rename **Level 9** as **Old Data**, make sure it is switched off and then assign to it the original model entities to be replaced.

10 Transfer the new model items currently assigned to the level named **NewData** to the appropriate **Outer** (**level 6**) or **Inner** (**level 7**) levels.

The pocket requires more material to be machined off on the die half of a mold tool. On the punch half, material needs to be added to reduce the depth of the central pocket, and removed to create the increased size for the core, central to the 4 Ribs.

11 Close without saving **New_Model_2.**

12 Select **File > Save New_Model_1** as:

.....PowerSHAPE_Data\InteriorTrim-3.psmodel

13 Maximise the window
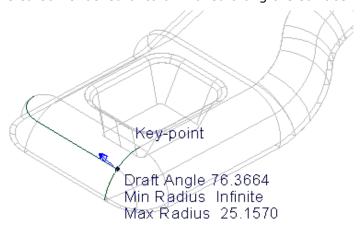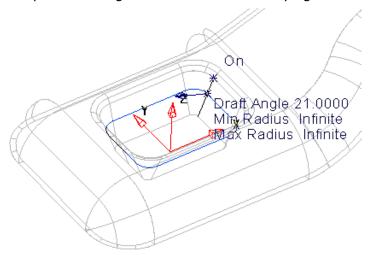
Do not close the model because it is used in the next section.

Surface Inspect

When active, the **Surface Inspect** option displays live dimensional information as the cursor is locked onto or moved along the surface model.

Key-point

Draft Angle 76.3664
Min Radius Infinite
Max Radius 25.1570

1   Switch off all levels except **6 - Outer**.

2   Select all displayed surfaces and then **Surface Inspect** on the **Model Analysis** toolbar.

3   Left click and drag (hold) on the model on or near the location where dimensional information is required.

4   Run the cursor along the model to lock on to key points or other, non-specific positions to give live feedback on varying dimensions.

On

Draft Angle 21.0000
Min Radius Infinite
Max Radius Infinite
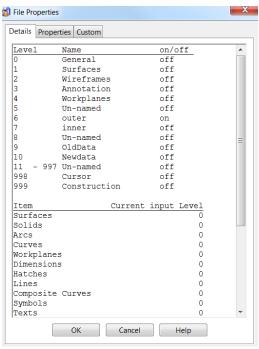
5   Select **File > Close** to close the model.

Other Model Analysis toolbar options
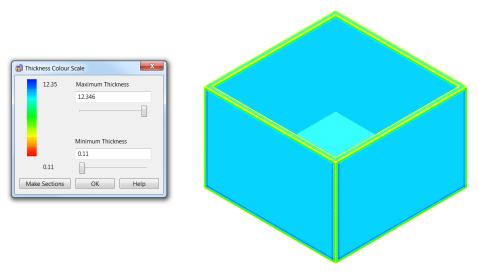
This section summarizes the remaining **Model Analysis** options.

- **Model Content list**

This displays details of the selected models contents and levels used.

- **Thickness Shading**

This displays the wall thickness (Solids only) as a colour map. The required Min/Max parameters can be defined.

These options operate on solid models only.
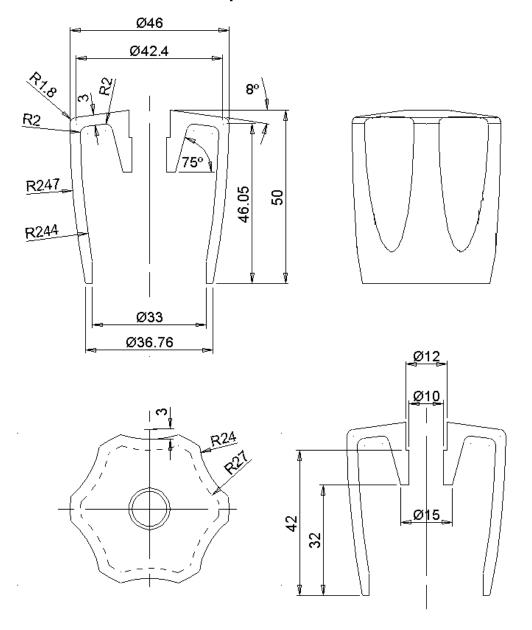
12. Modelling Exercises

Radiator Valve Cover

Model the **Radiator Valve component**.

Keyboard Button

Model the **surfaces** to form the **keyboard button** shown below. The button has a winding angle along one edge, which can be difficult to visualise. It is a constantly changing angle from one edge to the other.
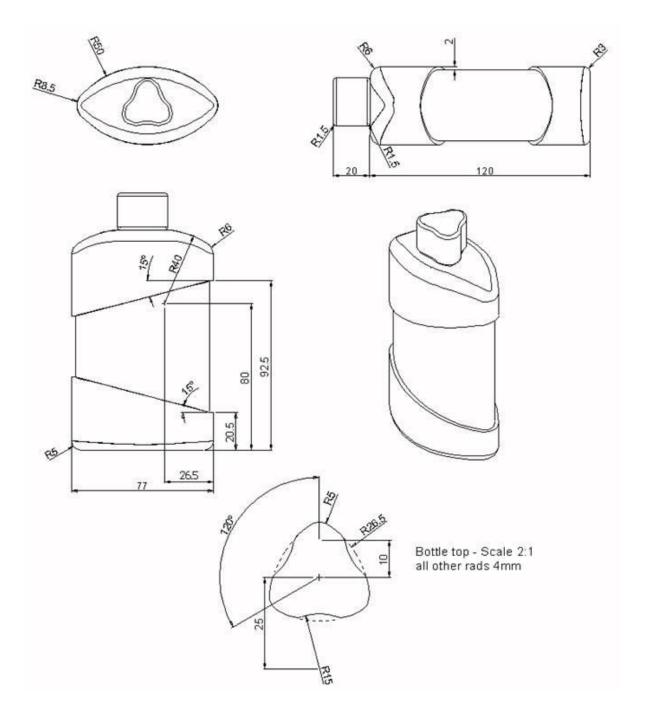
Plastic Bottle
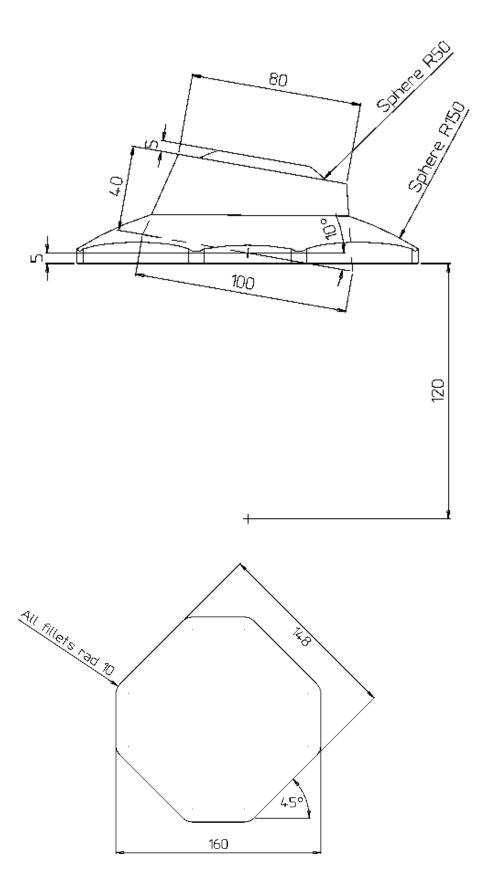
The **Plastic Shampoo Bottle** shown has a **variable radius fillet blend** at the base.

Bottle top - Scale 2:1
all other rads 4mm

Grommet

13. 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.

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.
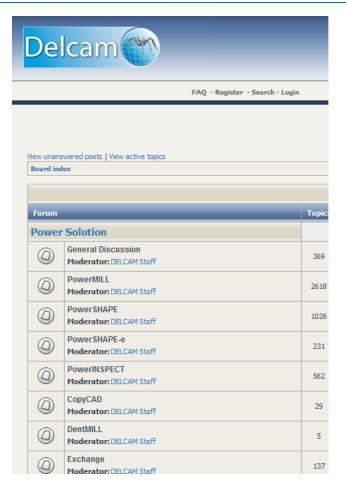
Help > Subscribe to the PowerSHAPE newsletter

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.
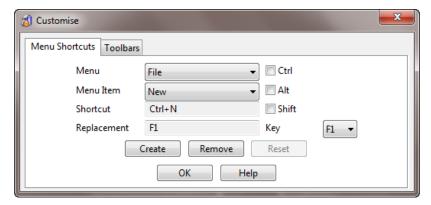
Other shortcuts

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.