PowerSHAPE 2015 R2

Reference Help

Surface modelling

PowerSHAPE

Acknowledgements

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Patent Information

Emboss functionality is subject to patent number GB 2389764 and patent applications US 10/174524 and GB 2410351.

Morphing functionality is subject to patent application GB 2401213.

PowerSHAPE 2015 R2. Published on 03 March 2015

Contents

Surface modelling

Use the following sections to find information on surface modelling:

Primitive surfaces

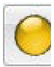
The following sections contain information on primitive surfaces:

Creating a primitive surface

1 Click the required primitive type on the Surface Creation toolbar.

 **Plane**

 **Block**

 **Sphere**

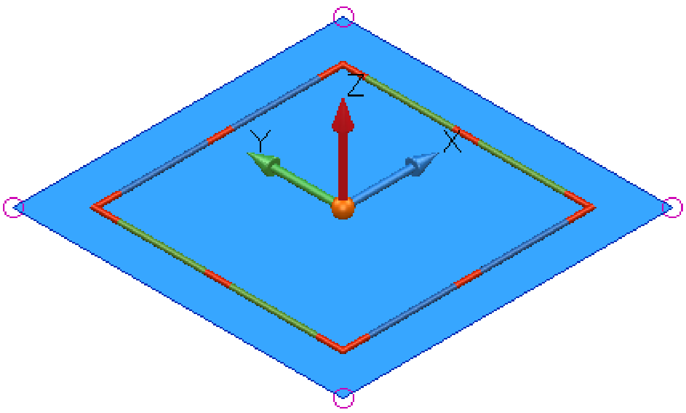
 **Cylinder**

 **Cone**

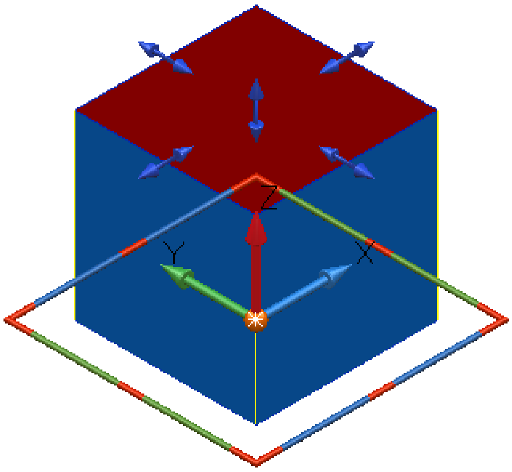
 **Torus**

 **Spring**

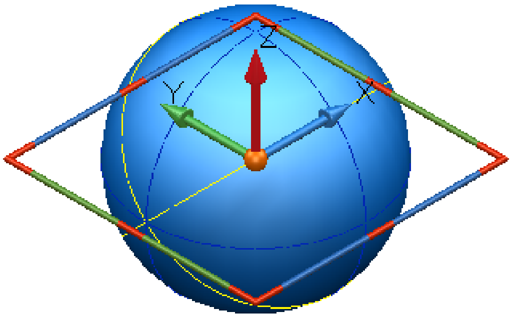
Plane — This creates a rectangular, planar surface.



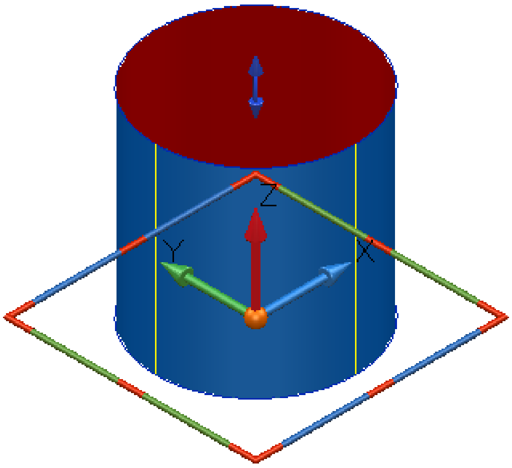
Block — This creates an extruded rectangle (with open ends).



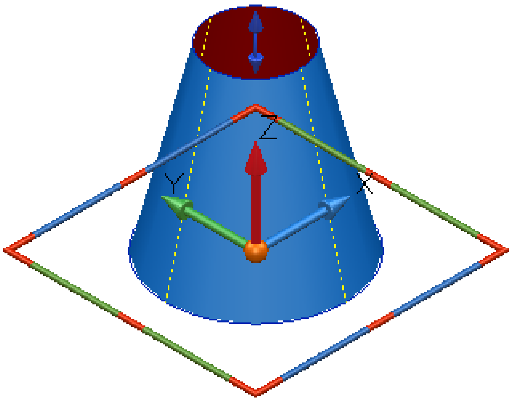
Sphere — This creates a spherical surface by revolving a semi-circle around the sphere's axis.



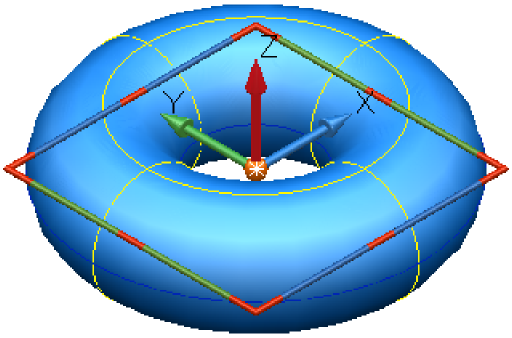
Cylinder — This creates a cylindrical surface (an extruded circle with open ends).



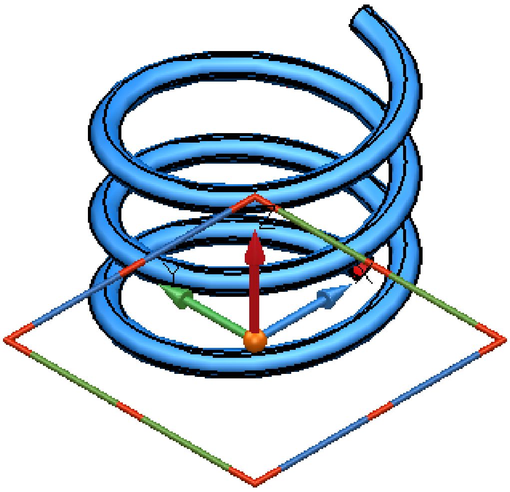
Cone — This creates a conical surface (with open ends).



Torus — This creates a toroidal surface by revolving a circle around an axis outside the circle.



Spring — This creates a **Surface Spring Primitive**.



2 Input a position on the screen to create a default primitive of the required type.

You can exit surface creation mode at any time by clicking **Select**  on the main toolbar. A surface is not created until its origin is defined; if you click **Select**  before a surface is positioned, no new surface is created.

3 To edit the new surface:

- use the graphical handles.
- right-click on the surface and select **Modify** from the menu.
- double-click the surface.

If you click on the surface just created, you start editing that surface; if it is clicked anywhere else, a new surface of the same type is created in that location.

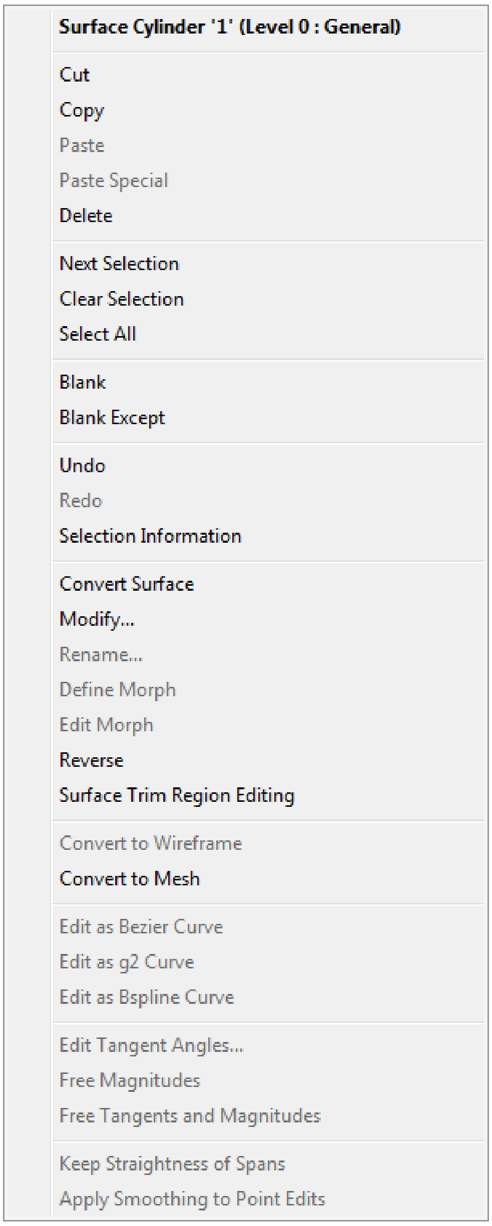
If you wish to create a different primitive surface, select it from the **Surface** toolbar and the cursor changes accordingly to indicate which type of surface is created next.

Editing a primitive surface

You can edit a primitive surface in one of the following ways:

- With the primitive selected, use its graphical handles to change the dimensions and orientation.
- Use one of the operations from the **Edit** menu:
  - **Edit > Modify** — displays the primitive's dialog.
  - **Edit > Convert > Surface** — converts a primitive into a power surface. This implies that any future edits to the surface must be performed using methods for power surfaces and not those used for primitives.
- Double-click the primitive to display its dialog.
- You can also simultaneously edit multiple primitive surfaces (see page 21).

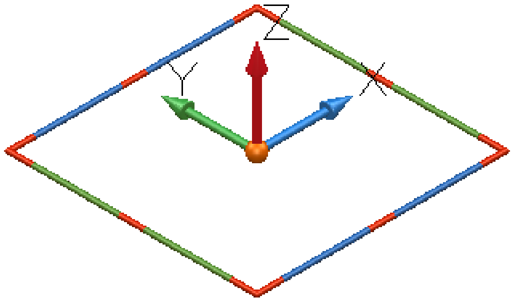
- Right-click the primitive to display edit menu available for primitives. At the top of the menu, you can see the type of the surface, its name and the level on which it lies.



Graphically editing primitives

Select the primitive to display its graphical handles.

Each primitive has handles similar to a workplane's graphical handles, as shown below. Other graphical handles which appear are unique to each primitive.



These workplane type handles are used to:

- move the primitive.
- change its direction.
- twist it about its axis.

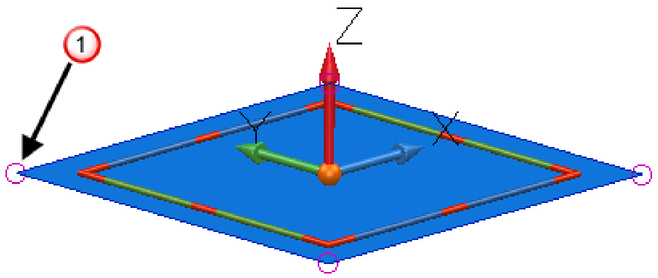
The unique graphical handles for each primitive are described below. In general:

1. Select the handle of the dimension you wish to edit.
2. Drag the handle to the required dimension.

   As you drag the handle, the new dimension is displayed on the screen. How the dimension value increases and decreases depends upon the zoom factor. To work with small increments, zoom in; to work with large increments, zoom out.

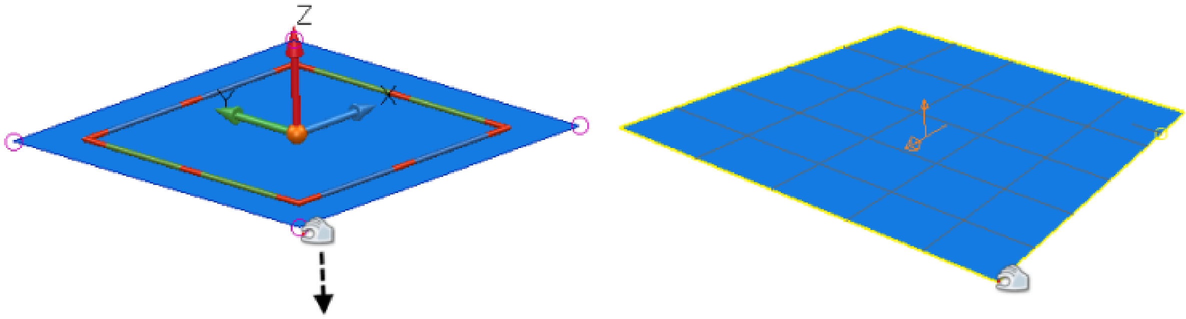
   The drag handles of primitive surfaces snap to other geometry in the model

Plane graphical handles

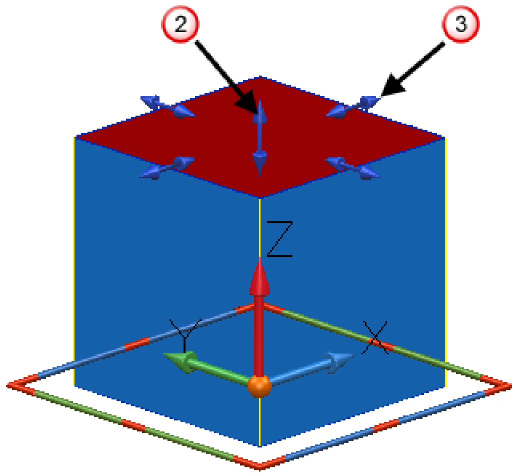


Select an edge and drag to change the width or length of a plane.

Select a corner handle 1 and drag to move the corner to a new position. This will automatically convert the surface to a **Power Surface**.



Block graphical handles

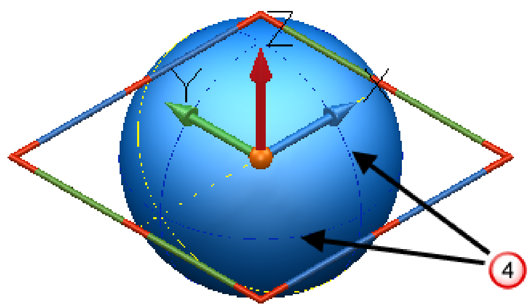


Select the top handle 2 and drag to edit the height of the block.

Select the edge handles 3 and drag to edit the draft angle of a side.

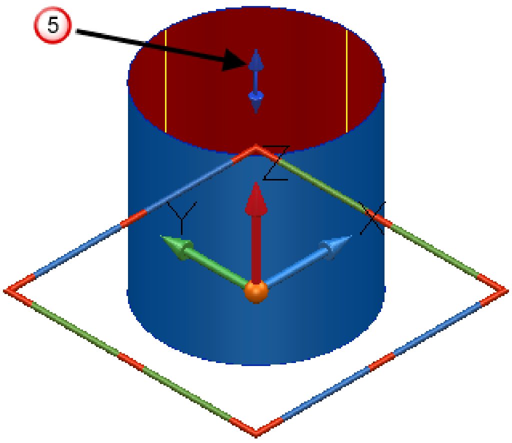
Select an edge and drag to change width or length of the block.

Sphere graphical handles



Select and drag any of the blue curves 4 to edit the radius of the sphere.

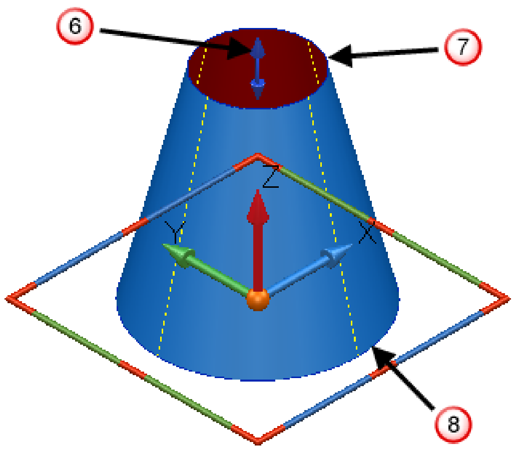
Cylinder graphical handles



Select the top handle 5 and drag to change the height of the cylinder.

Select the edge and drag to change the radius of the cylinder.

Cone graphical handles

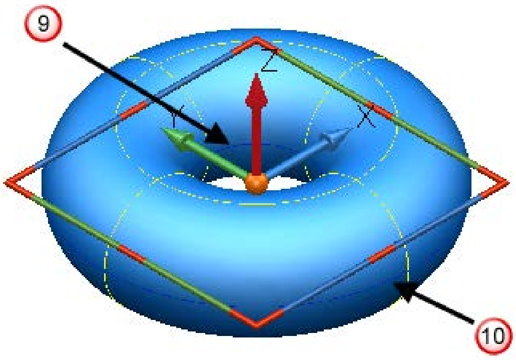


Select the top handle 6 and drag to change the height of the cone.

Select the top edge 7 and drag to change the top radius of the cone.

Select the bottom edge 8 and drag to select the base radius of the cone.

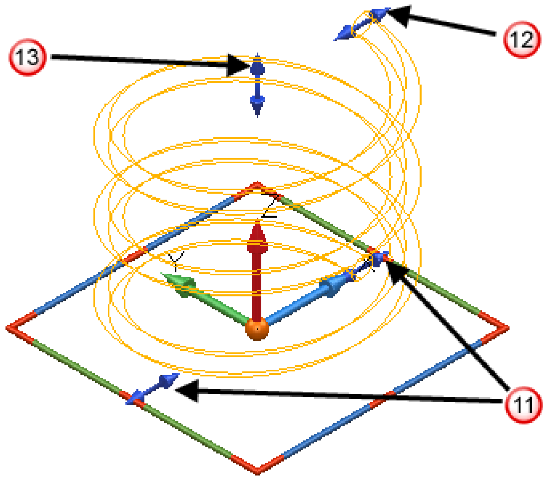
Torus graphical handles



Select the inner curve 9 and drag to change the minor radius of the torus.

Select the outer curve 10 and drag to change the major radius of the torus.

Spring graphical handles



Select either of these handles 11 and drag to change the base radius of the spring.

Select this handle 12 and drag to change the top radius of the spring

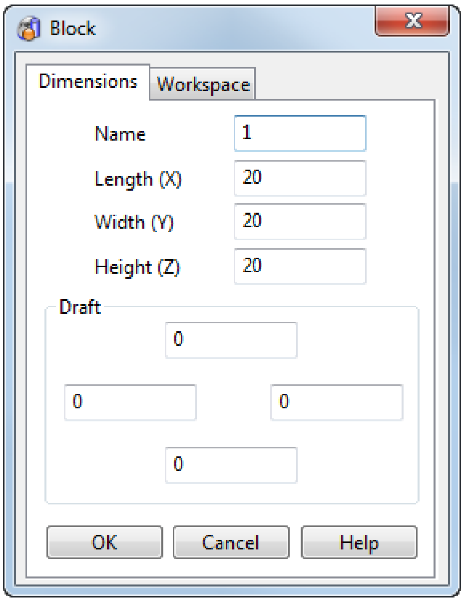
*If the **Constant Radii** option is ON (default), the top and bottom radii will be adjusted simultaneously.*

Select the top handle 13 and drag to change the height of the spring.

Primitive surface dialog

To edit a primitive surface using the dialog:

1 Double-click on the primitive you want to edit to display the following dialog:

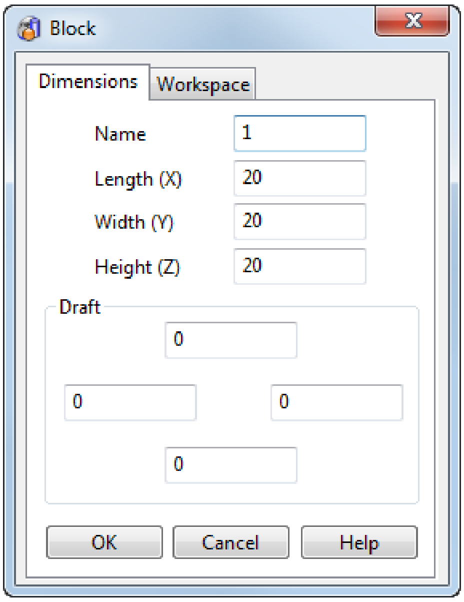


2 Use the two pages of the dialog to define the changes you want to make.

- Dimensions (see page 13)
- Workspace (see page 15)

Primitive surface dialog - Dimensions

Use the **Dimensions** page of the dialog to edit a surface primitive.

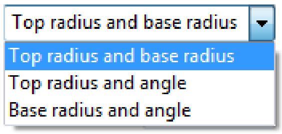


Name - This is the name of the selected primitive. You can edit the name.

Specify the following dimensions for the selected primitive:

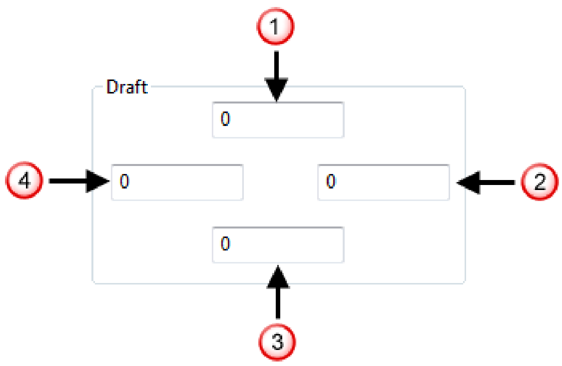
- **Plane** - Specify the **Width (X)**, **Length (Y)**

- **Block** - Specify the **Length (X)**, **Width (Y)**, **Height (Z)**. You can also apply **Draft** angles to four sides of a block.
- **Cylinder** - **Radius** or **Diameter** is selected from the drop down list. You can also specify the **Length**.
- **Cone** is specified as follows:
  a Enter the **Length** of the cone.
  b Select one of the options from the drop-down list.



  c Your selection determines the other options that are available on the dialog. It also affects the dimensions of the cone that can be changed using graphical editing.
  d Enter the other parameters.
  e Click **OK**.
- **Sphere** - **Radius** or **Diameter** is selected from the drop down list.
- **Torus** - Specify the **Major Radius**, **Minor Radius**.

Draft - Enter draft angles for the following sides, when looking down the Z axis:



1 - Top

2 - Right

3 - Bottom

4 - Left

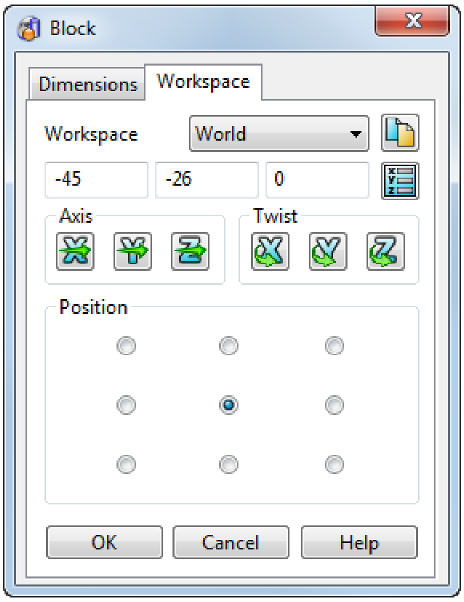
You can also define the draft angles by using the drag handles that are displayed in the centre of each top edge of the block.

OK - Saves the edits carried out on the primitive and removes the dialog from the screen.

Cancel - Closes the dialog and discards any edits carried out on the primitive whilst it was displayed.

Primitive surface dialog - Workspace

Use the **Workspace** page of the dialog to specify the primitive.



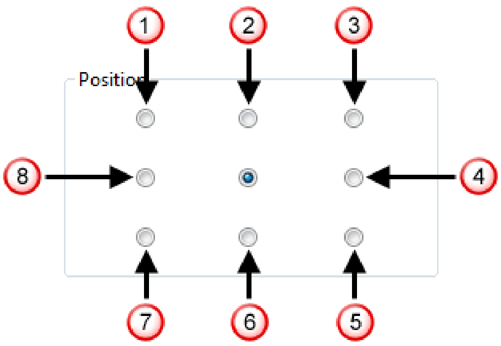
You can:

- define the **Workspace** in which to edit the primitive.
- create a copy of the workplane by clicking  on the dialog.
- define a new origin of the primitive by entering new **X**, **Y**, **Z** values or clicking  to display the **Position** dialog.
- change the direction of its **Axis.**
- **Twist** it about its axis.

Position

 *This option is only displayed when editing a block.*

Use this option to position the origin of the block. The default option is centre of the base of the block. Other available options are shown below. All positions are relative to the default.



 Top left corner.

 Middle of top edge.

③ Top right corner.

④ Middle of right edge.

⑤ Bottom right corner.

⑥ Middle of bottom edge.

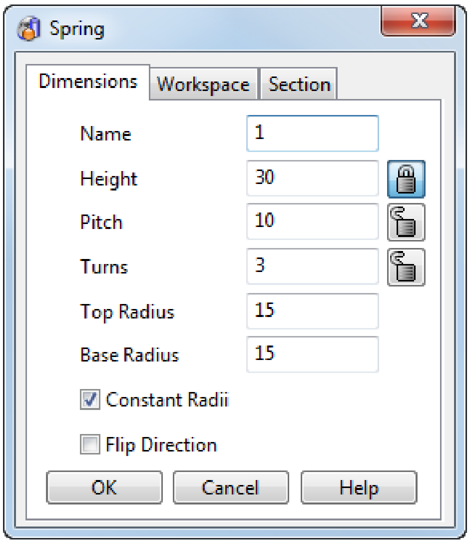
⑦ Bottom left corner.

⑧ Middle of left edge.

Spring dialog

Use this dialog to specify or edit the dimensions for a solid spring or a surface spring primitive.

1 Double-click on an existing spring to display this dialog:

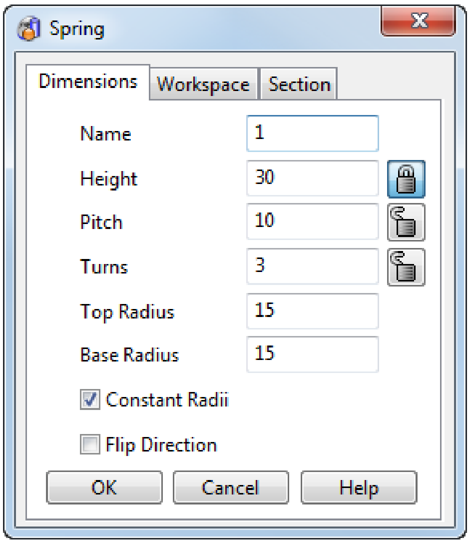


2 Click on the tabs to define the spring:

- Dimensions (see page 17)
- Workspace (see page 18)
- Section (see page 18)

Spring dialog - Dimensions

Use this page of the dialog to specify the dimensions of the spring.



Name - Enter a name for the spring.

Height - Enter the overall vertical height of the spring. The height can also be changed by dynamically dragging the arrow handle at the top of the spring.

*The **Height**, **Pitch** and number of **Turns** are inter-related. If one value is changed another value must also change in order to keep the definition consistent.*

Pitch - Enter the vertical distance between consecutive turns.

Turns - Enter the number of turns in the spring.

 - Lock or unlock the **Height**, **Pitch** and number of **Turns** for the helix. When a dimension is locked it will not change when another dimension is changed. For example, if the height is locked and the user changes the pitch, the number of turns will change to keep the definition consistent.

Top Radius - Enter the radius for the top of the spring. The top radius can also be changed by dynamically dragging the arrow handle at the top of the spring.

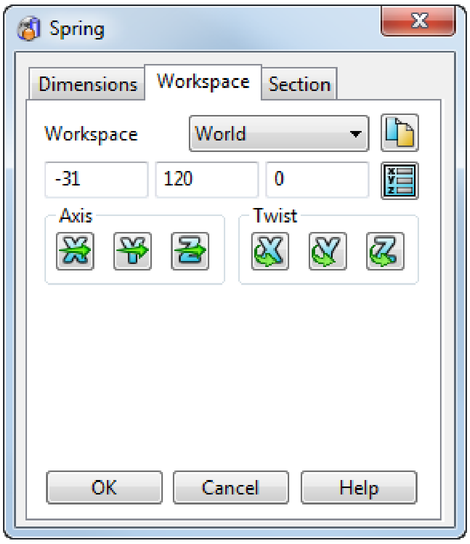
Base Radius - Enter radius for the bottom of the spring. The base radius can also be changed by dynamically dragging the arrow handle at the bottom of the spring.

Constant Radii - Select this option to keep the base and top radii equal. The default setting is *ON.*

Flip Direction - Select this option to flip the direction of the spring between clockwise and anti-clockwise.

Spring dialog - Workspace

Use this page of the dialog to specify the **Workspace** details of the spring.

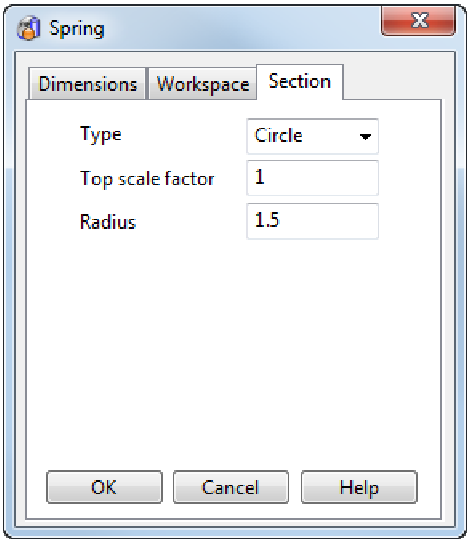


Workspace options - These options allow you to:

- define the workspace in which to edit the primitive.
- change the primitive's workspace.
- move the primitive's origin.
- change the direction of its axis.
- twist it about its axis.

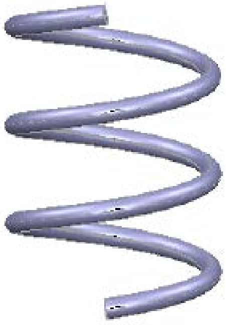
Spring dialog - Section

Use this dialog to specify the Section details for a solid spring or a surface spring primitive.

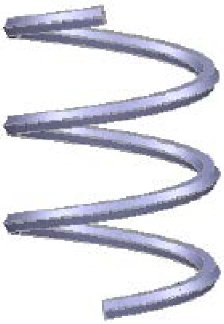


Type - Select one of the options:

- **Circle** - Select this option to create a circle type spring:

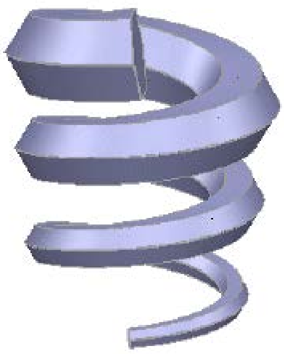


- **Polygon** - Select this option to create a polygon type spring:



*Additional options for polygon type springs are displayed. For further details see **Polygon Options** (see page 20).*

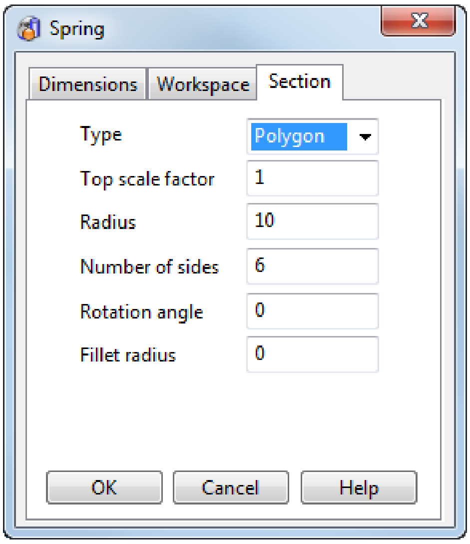
Top scale factor - Enter a factor by which the scale of the section radius increases along the height. The example below shows a polygon type solid spring with a section radius of **35** and a **scale factor** of **5**.



Radius - Enter the radius for the section.

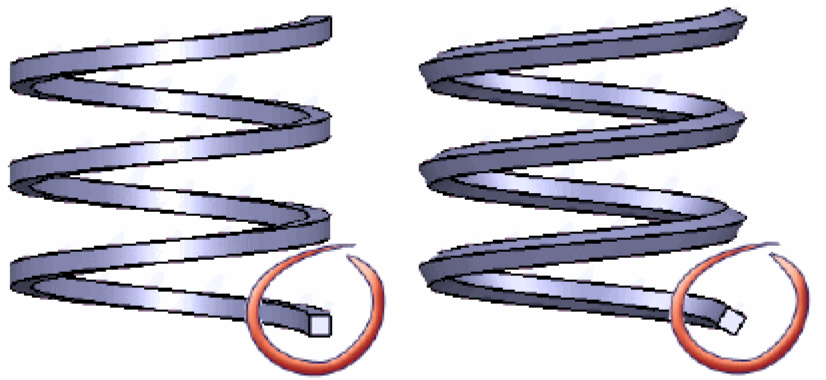
Spring dialog - Section - Polygon options

When you select a **Polygon** type spring on the **Section** page of the **Spring** dialog, additional options are displayed.



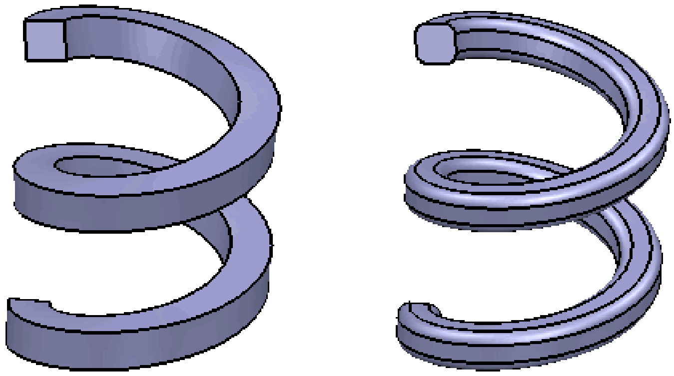
Number of sides - Specify the number of sides for the polygon type spring.

Rotation angle - Specify the rotation angle for the section. The example below shows two, four-sided polygon solid springs. The spring on the left was created with a rotation angle of *0*. The spring on the right was created with a rotation angle of *30*.



Fillet radius - Enter a radius to create fillet arcs between each straight edge of the polygon type spring. If set to zero, no arcs will be created.

The example below shows two, four-sided polygon type solid springs. The spring on the left was created with a **Fillet radius** of *0*. The spring on the right was created with a **Fillet radius** of *3* and is highlighted to show the fillets.



Editing multiple primitive surfaces

You can edit multiple primitive surfaces of the same type at the same time.

This function is available for surface:

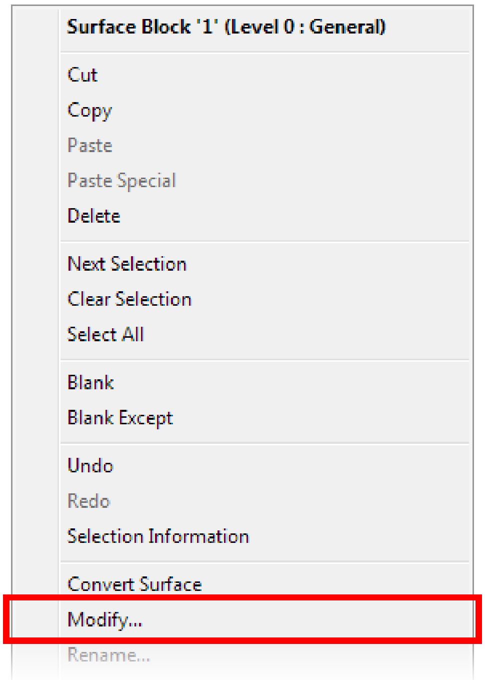
- planes
- blocks
- cylinders
- cones
- spheres
- tori
- springs

To edit multiple primitive surfaces of the same type:

This example shows how to edit multiple surface blocks, however the same concepts apply for all surface primitives.

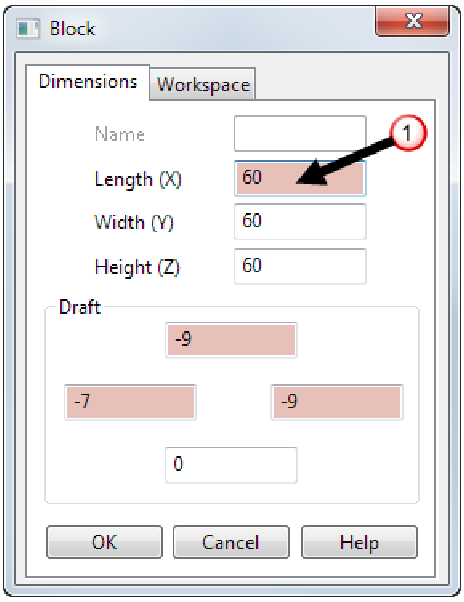
1. Select a number of primitive surfaces that you want to edit (for example blocks) with one of the following methods:
   - Press and hold the **Shift** key and click the blocks.
   - Press and hold the **Ctrl** key and click the blocks.
   - Click and drag the cursor over the blocks.

2 Right-click one of the selected primitives, and select **Modify**.

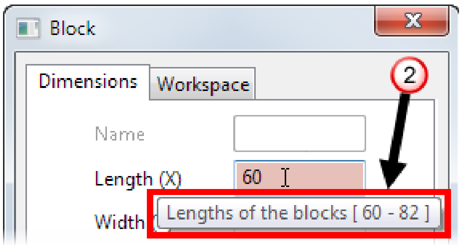


The dialog to edit the appropriate selected primitives is displayed (for example, the **Block** dialog). The dialog is similar to that of editing a single primitive of the same type, with the following differences:

1 — The properties that differ between the selected primitives are highlighted in pink.



2 — The range of values is displayed when you hover the cursor over a property.



You can also double-click one of the selected primitives to display the dialog.

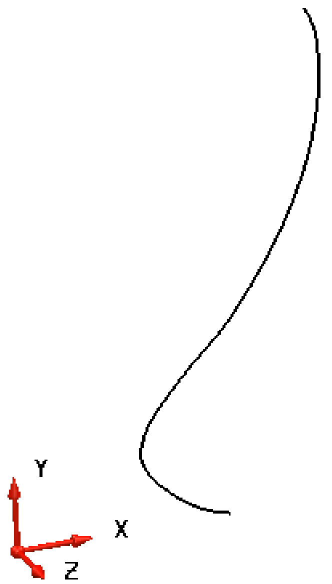
3 Edit the properties.

4 Click **OK** to apply the changes to the selected primitives and close the dialog.

Creating a surface of revolution

A surface of revolution is created from a wireframe object which is rotated around one of the axes of the current workspace. We recommend you create a workplane to make sure the surface of revolution is created around the correct axis.

1 Select a wireframe object.

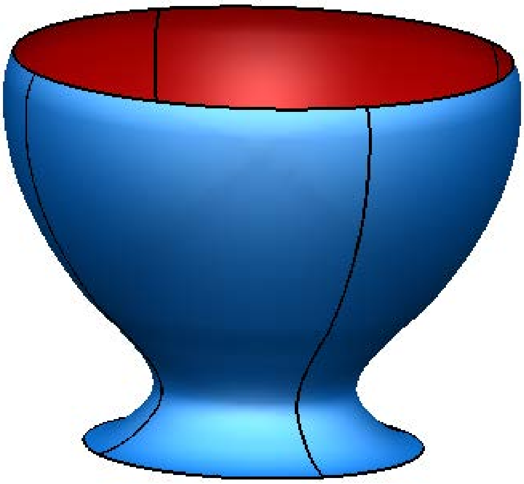


2 In the status bar, select the **Principal plane** button  for the axis you want to rotate around.

To rotate the curve around the **Y** axis, select the **Y Principal plane** button.

3 Click  (*Surface toolbar*).

The new surface is created.

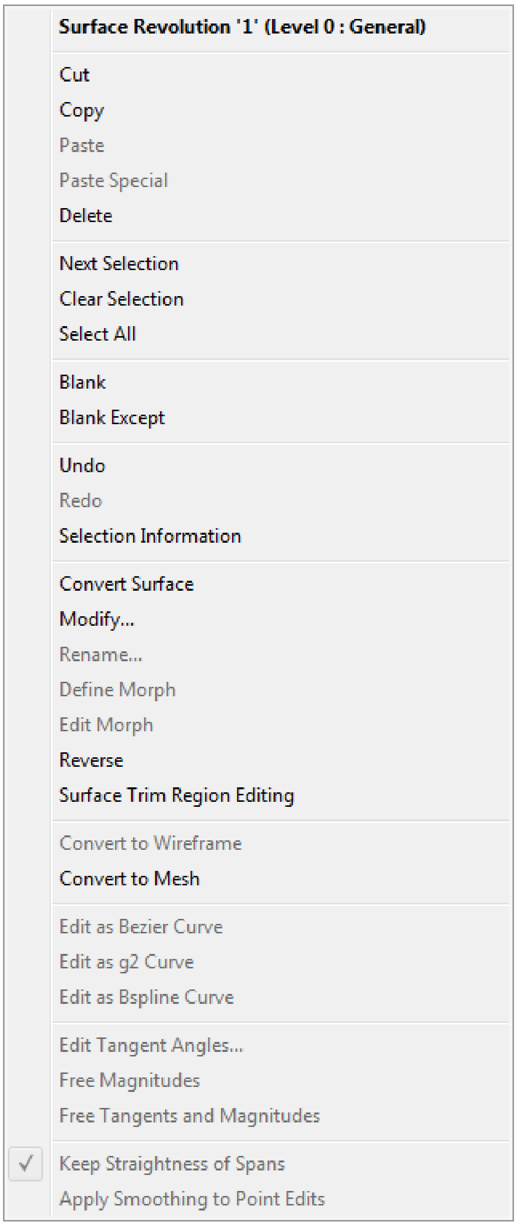


Editing a surface of revolution

You can edit a surface of revolution in one of the following ways:

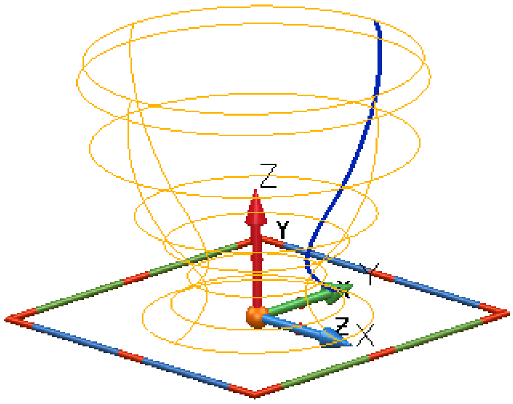
- With the surface of revolution selected, use its graphical handles to change the orientation.
- Use one of the operations from the **Edit** menu:
  - **Edit > Modify...** - displays the surface of revolution's dialog.
  - **Edit > Convert > Surface** - converts a primitive surface into a power surface. This implies that any future edits to the surface must be performed using methods for power surfaces and not those used for primitives.
- Double-click the surface of revolution to display its dialog.

- Right-click the surface of revolution to display edit options available for surfaces of revolution. At the top of the context menu, you can see the type of the surface, its name and the level on which it lies.



Graphically editing a surface of revolution

Select the surface of revolution to display its graphical handles.

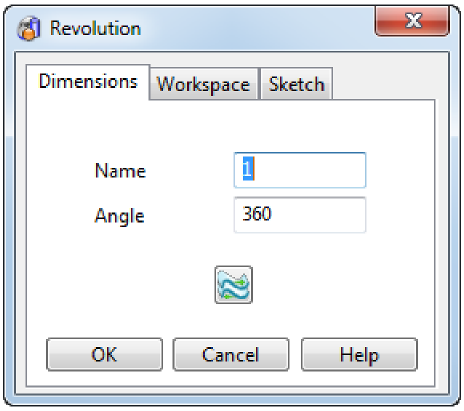


These handles are used to:

- move the revolution.
- change its direction.
- twist it about its axis.

Revolution dialog

Use this dialog to edit a surface of revolution:

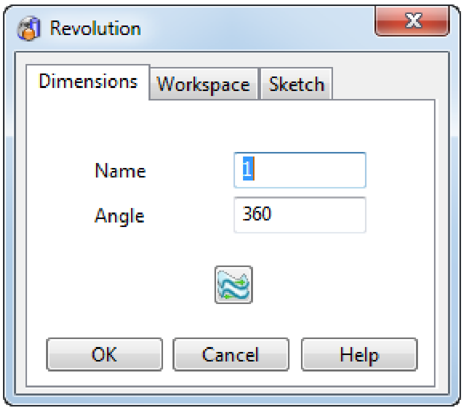


Click on the tabs to define the revolution:

- Dimensions (see page 27)
- Workspace (see page 27)
- Sketch (see page 28)

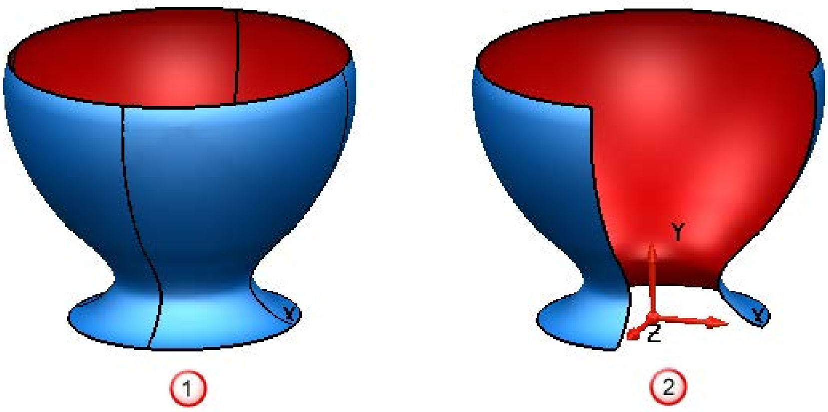
Revolution dialog - Dimensions page

Use this page of the dialog to define the dimensions for the surface of revolution.



Name - This is the name of the selected surface of revolution. The name can be edited.

Angle - This is the angle that the wireframe object is rotated around the axis.



1 Angle = 360°

2 Angle = 270°

 - **Reverse direction.** This reverses the surface so that the outside of the surface becomes the inside, and the inside becomes the outside.

OK - Saves the edits carried out on the surface of revolution and closes the dialog.

Cancel - Closes the dialog and discards any edits carried out on the surface of revolution whilst it was displayed.

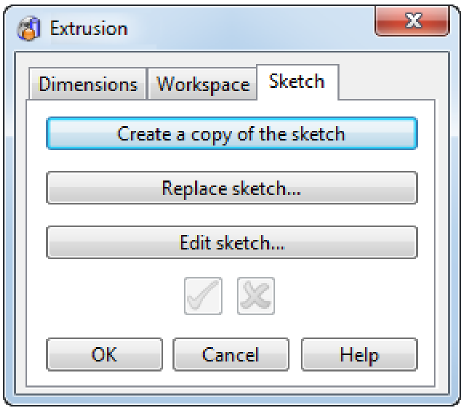
Revolution dialog - Workspace

Use this dialog to edit the workspace options for the surface of revolution.

It is the same as Primitive Surface dialog - Workspace (see page 15)

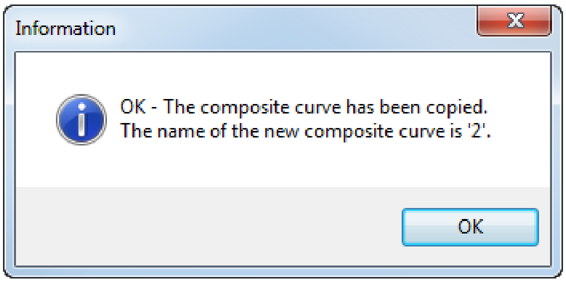
Revolution dialog - Sketch page

Use this page of the dialog to edit the wireframe for the surface of revolution.



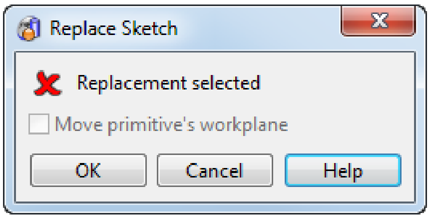
Create a copy of the sketch

Click this button to add a copy of the composite curve to the model. The **Information** dialog is displayed to confirm the copy.



Replace sketch

Click this button to replace the sketch curve. Use the dialog to select the curve to replace.



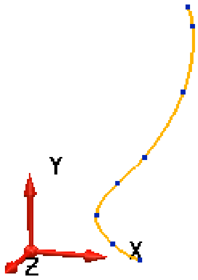
Edit sketch

Click this button to edit the wireframe. The example below is used to show you how to edit the wireframe.

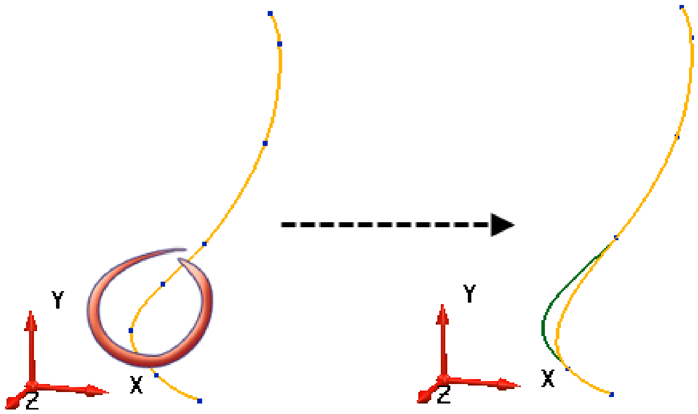
1 Click **Edit sketch**. The following is displayed:

- wireframe.
- **Curve Edit** toolbar.

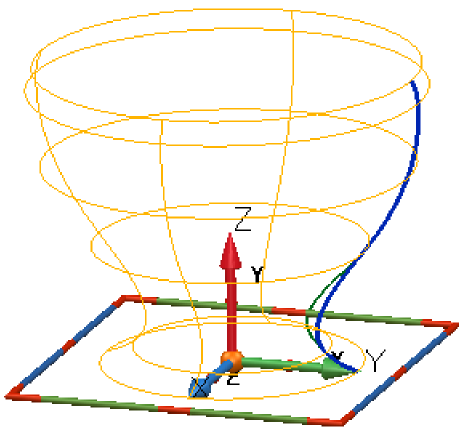
- is displayed on the dialog.



2 Delete two of the points, by using the graphical handles or the options on the **Curve edit** toolbar.



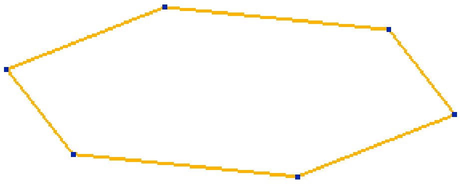
3 Once you have finished editing the wireframe, click to accept the edit or to cancel the edit.



Creating an extruded surface

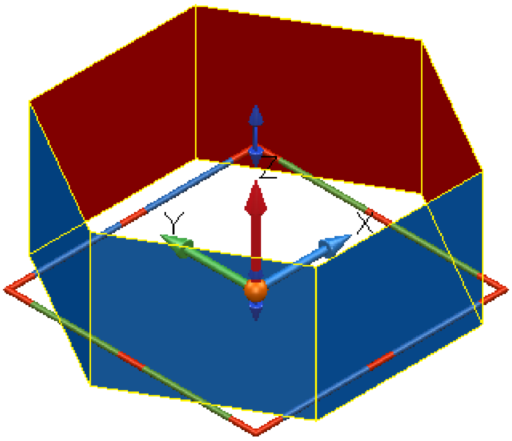
You can create extruded surfaces from single or multiple wireframe objects.

1 Select one or more wireframe objects.

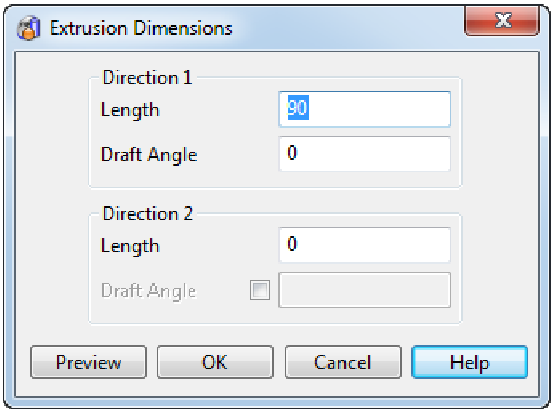


2 Click  (*Surface toolbar*).

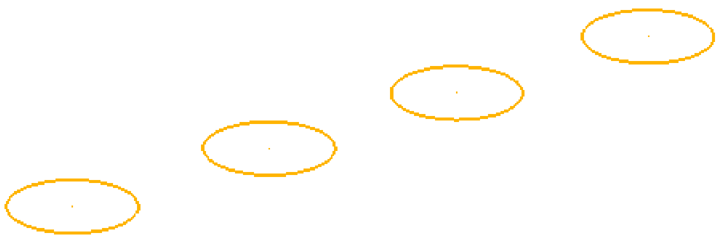
3 If a single wireframe object was selected, it is extruded to form a surface.



4 If multiple wireframe objects were selected, the **Extrusion Dimensions** dialog is displayed. This dialog allows you to set the dimensions of the extruded surfaces.

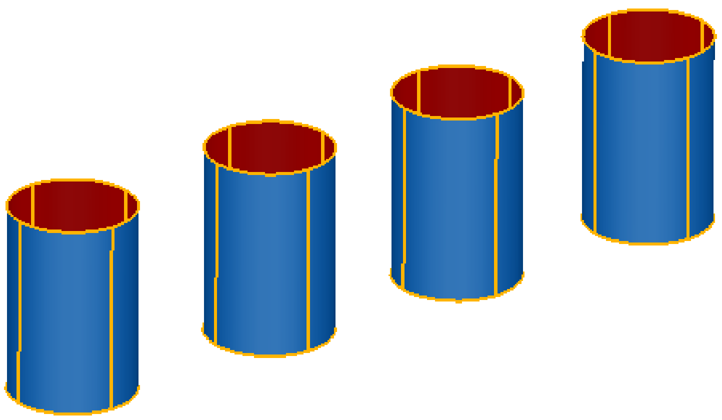


Select the following wireframe objects:

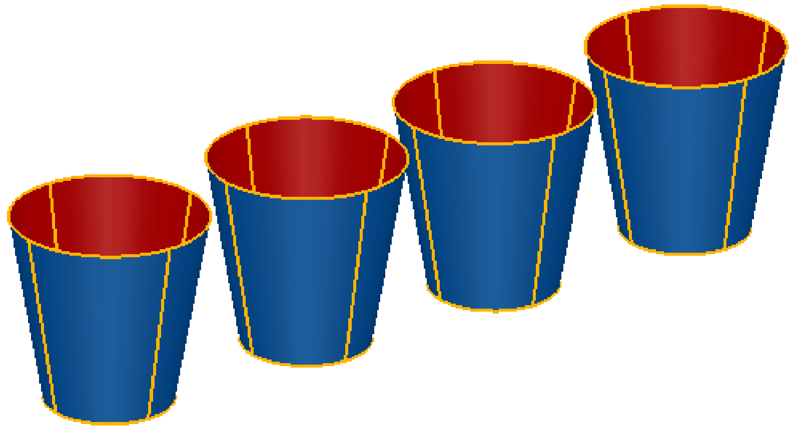


Input the values for the length and draft angle for the extruded surfaces. You can create an extrusion in both directions by entering values for **Direction 1 - Length** and **Direction 2 - Length**.

Click **Preview** to see the extruded surfaces.



You can change the values and click **Preview** again to update the surfaces.



5 Click **OK** to accept the surfaces that you have previewed and close the dialog.

To edit an extruded surface see Editing an extruded surface (see page 32)

If the wireframe is non-planar, the extrusion is in the positive direction of the axis normal to the principal plane.

If the wireframe is planar, the extrusion is normal to the plane in which the wireframe lies. You can change this default by deselecting the **Create extrusions normal to planar base** option on the **Tools > Options > Object > Surfaces** dialog.

The wireframe used to create the extruded surface is removed from the model by default. You can add the wireframe to the model again using the **Extrusion** dialog. This is displayed by double-clicking the surface.

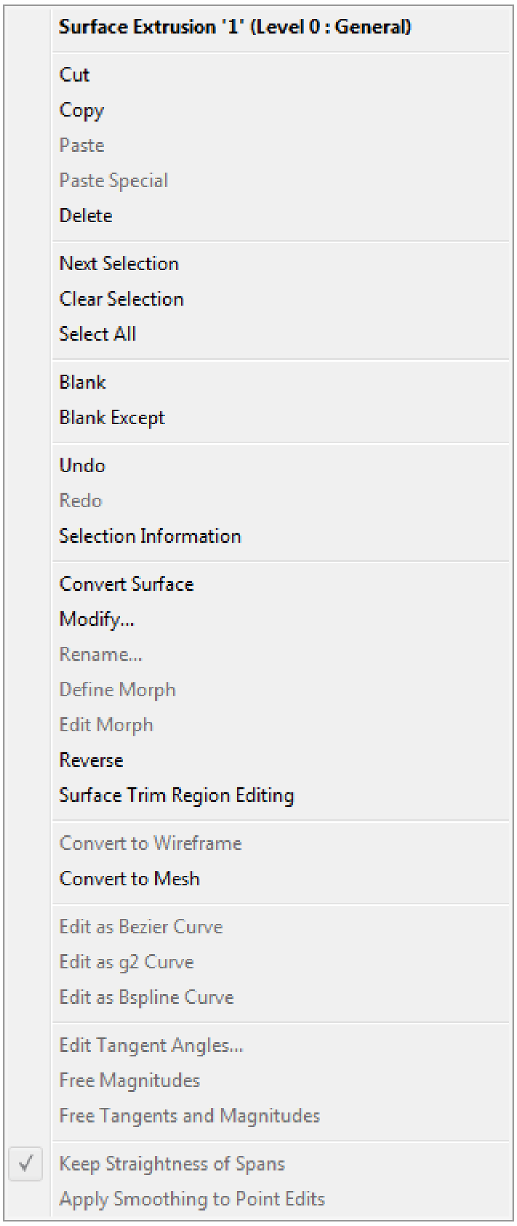
To keep the wireframe in the model whenever you create an extruded surface, select the **Keep Wireframe** option on the **Tools > Options > Object > Surfaces** dialog.

Editing an extruded surface

You can edit an extruded surface in one of the following ways:

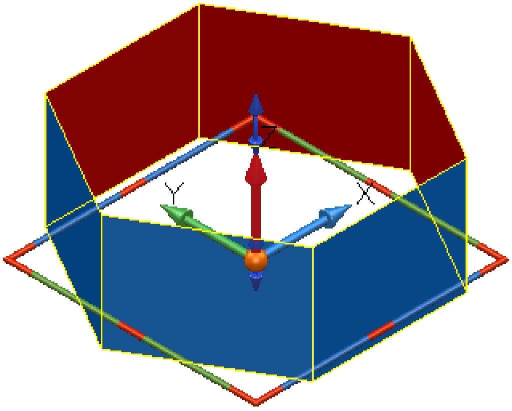
- With the extruded surface selected, use its graphical handles to change the orientation.
- Use one of the operations from the **Edit** menu:
  - **Edit > Modify** - displays the extruded surface's dialog.
  - **Edit > Convert > Surface** converts an extruded surface into a power surface. This implies that any future edits to the surface must be performed using methods for power surfaces and not those used for extruded surfaces.
- Double-click the extruded surface to display its dialog.

- Right-click the extruded surface to display edit options available for extruded surfaces. At the top of the context menu, you can see the type of the surface, its name and the level on which it lies.



Graphically editing an extruded surface

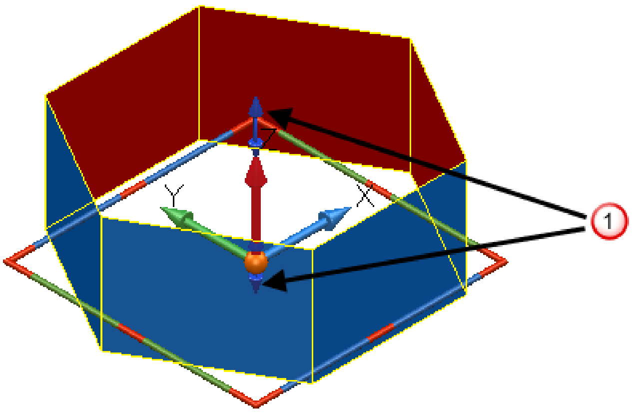
1 Select the extruded surface to display its graphical handles.



The workplane type handles are used to:

- move the extrusion.
- change its direction.
- twist it about its axis.

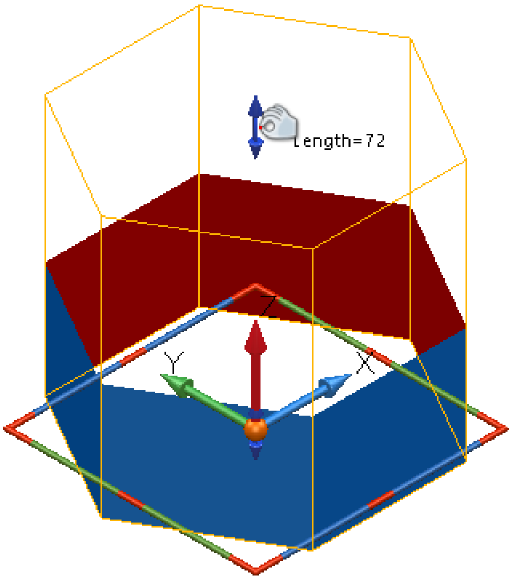
The blue handles 1 edit the **length** and **negative length** of the extruded surface.



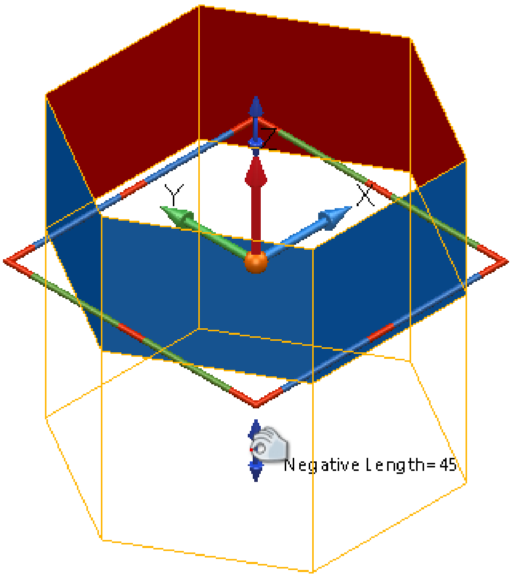
1 To edit the length, select the top handle and drag it to the required height.

As you drag the handle, the new dimension is displayed on the screen.

How the dimension value increases and decreases depends upon the zoom factor. To work with small increments, zoom in; to work with large increments, zoom out.

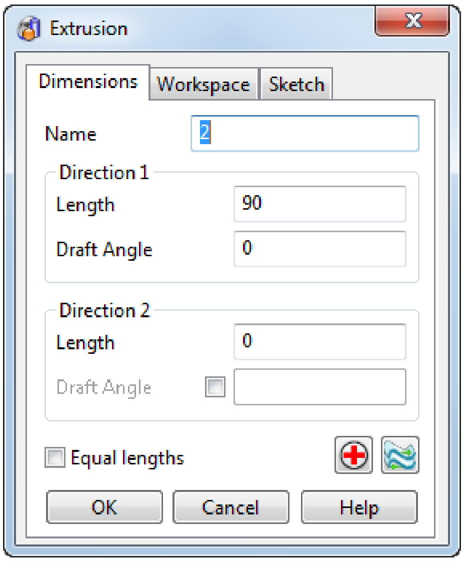


You can also drag the handle in the negative Z direction.



Extrusion dialog

This dialog is used to edit an extruded surface.

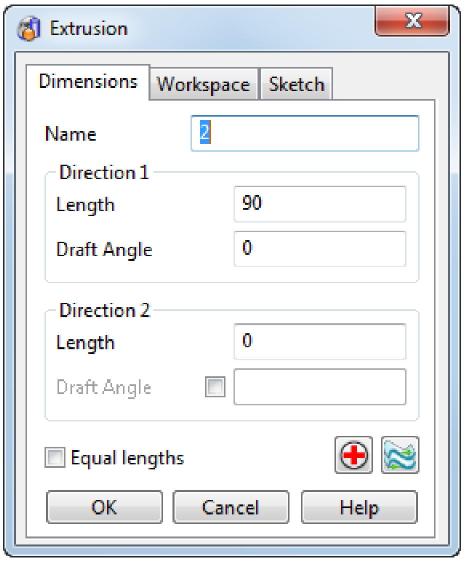


The dialog has three pages:

- Dimensions (see page 36)
- Workspace (see page 38)
- Sketch (see page 39)

Extrusion dialog - Dimensions

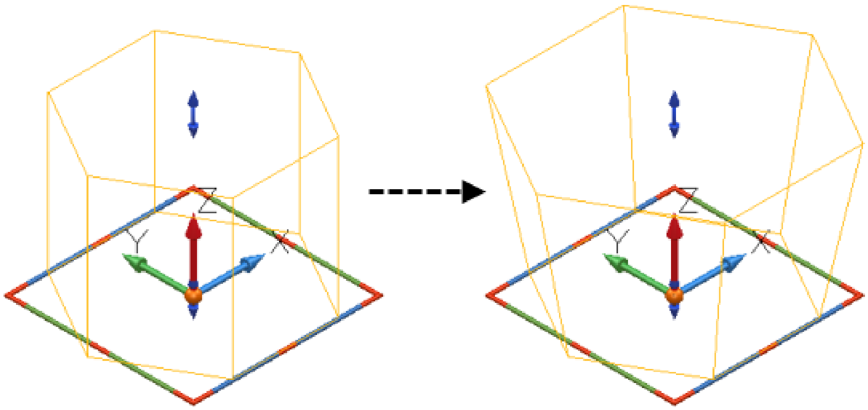
Use this page of the dialog to specify the dimensions for the surface extrusion.



Name - This is the name of the selected extrusion. The name can be edited.

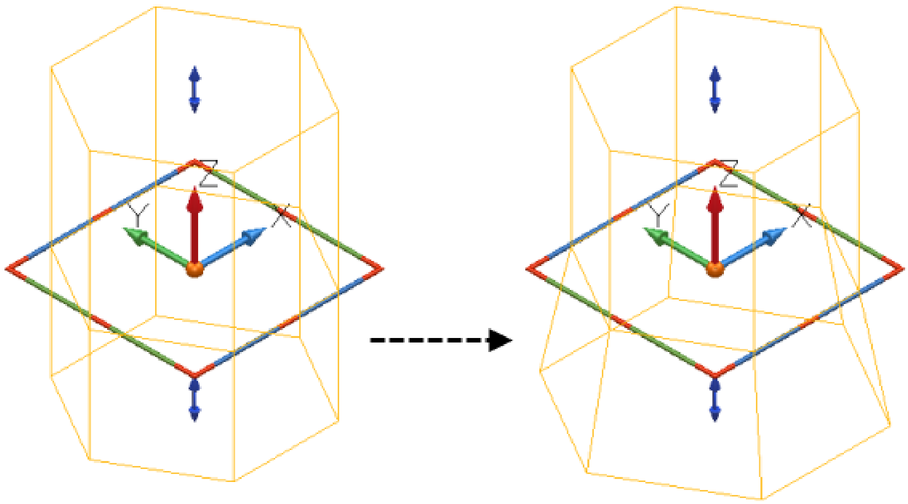
Direction 1 - Length - This is the **Length** of the extrusion. The length can be zero if the length specified in **Direction 2** is non-zero.

Direction1 - Draft Angle - This is the **Draft Angle** from the base of the extrusion. The angle is measured from the axis normal to the principal plane.



Direction 2 - Length - This is the negative length of the extrusion.

Direction 2 - Draft Angle - This is the draft angle of the extrusion in the negative length.



After you have entered a value, the model updates dynamically to reflect the change.

If required, click  to check that the draft angle is constant.

Select **Equal lengths** to set the same **Length** for **Direction 1** and **Direction 2**.

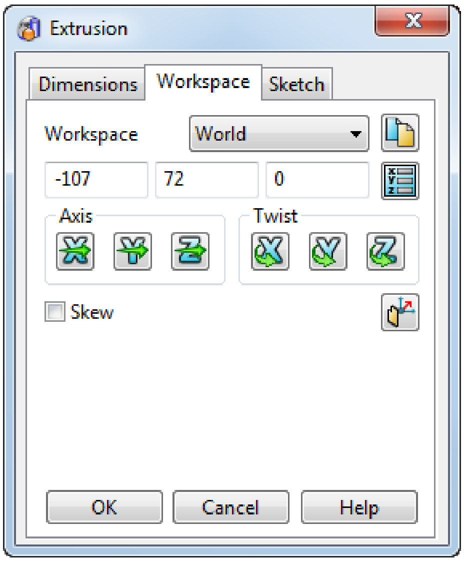
Click  to **Reverse direction**. This reverses the surface so that the outside of the surface becomes the inside, and the inside becomes the outside.

Click **OK** to save the edits carried out on the extruded surface and remove the dialog from the screen.

Click **Cancel** to close dialog and discard any edits carried out on the extruded surface whilst it was displayed.

Extrusion dialog - Workspace

Use this dialog to edit the workspace of an extrusion.



Workspace options - Use these options to:

- define the workspace in which to edit the extrusion.
- change the extrusion's workspace.
- move the extrusion's origin.
- change the direction of the extrusion's axis.
- twist the extrusion about its axis.
- create a copy of the workplane by clicking on the dialog.

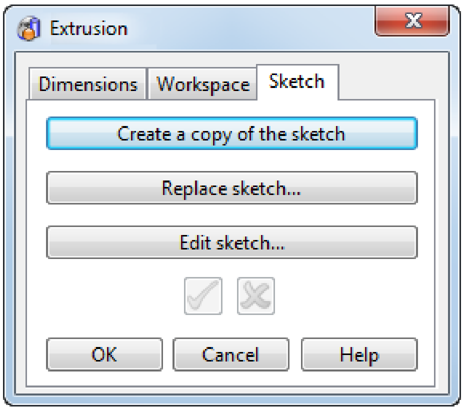
Enter the **X Y Z** coordinates or click the **Position** button to open the **Position** dialog where you can use position entry tools.

Skew - If selected, any changes to the workplane will change the direction of the extrusion without affecting the base curve. If deselected, then changes to the workplane will change the orientation and position of the whole extrusion.

- This makes the Z axis of the workplane perpendicular to the plane in which the base curve lies. If the base curve is non-planar then the plane of best fit is calculated.

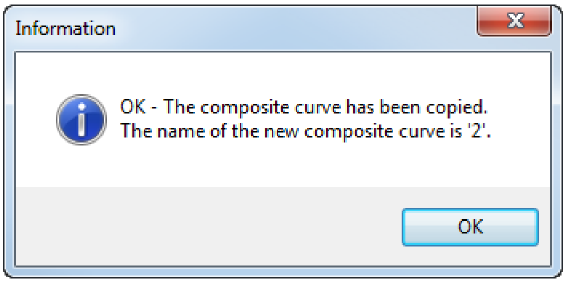
Extrusion dialog - Sketch

Use this page of the dialog to edit the wireframe for the extrusion.



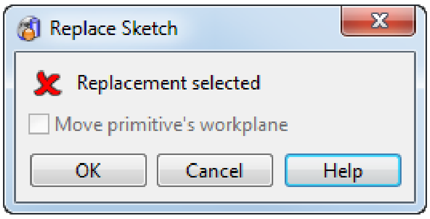
Create a copy of the sketch

Click this button to add a copy of the composite curve to the model. The **Information** dialog is displayed to confirm the copy.



Replace sketch

Click this button to replace the sketch curve. Use the dialog to select the curve to replace.



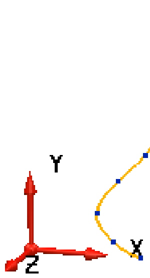
Edit sketch

Click this button to edit the wireframe. The example below is used to show you how to edit the wireframe.

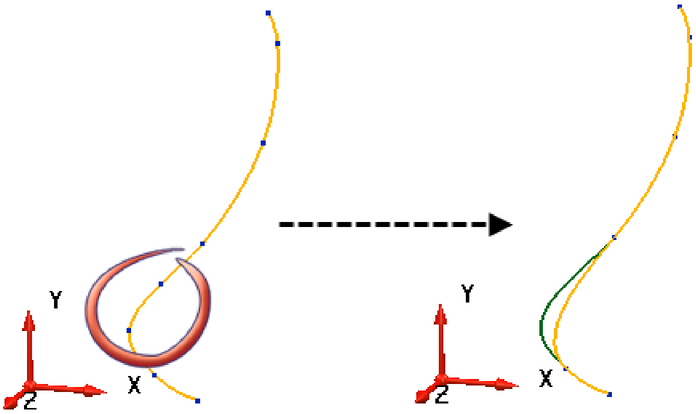
1 Click **Edit sketch**. The following is displayed:

- wireframe.
- **Curve Edit** toolbar.

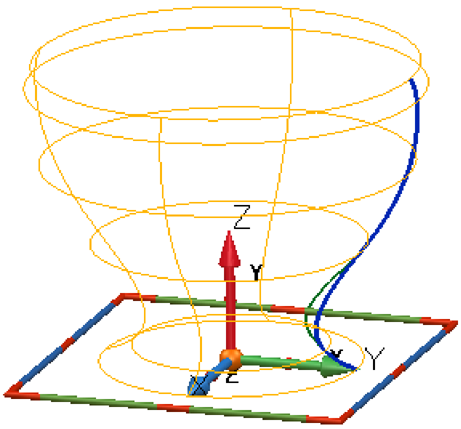
- is displayed on the dialog.



2 Delete two of the points, by using the graphical handles or the options on the **Curve edit** toolbar.

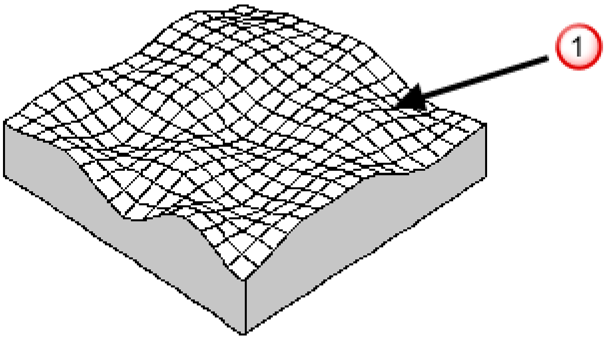


3 Once you have finished editing the wireframe, click to accept the edit or to cancel the edit.



What is a surface?

A surface can be defined as the exposed layer of a material 1.

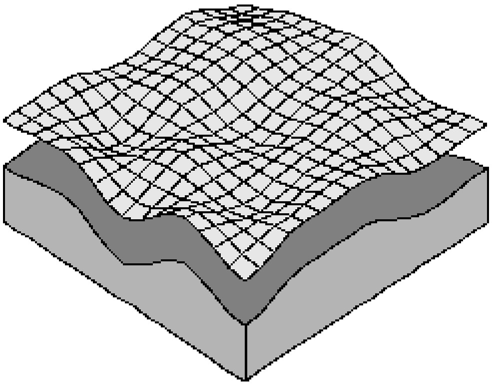


A surface model is a representation of a surface. A geometric modeller is an instrument used to model a surface.

PowerSHAPE is a geometric modeller with wireframe and surface modelling capabilities. It is particularly suitable for the design and analysis of components with complex shapes and surfaces.

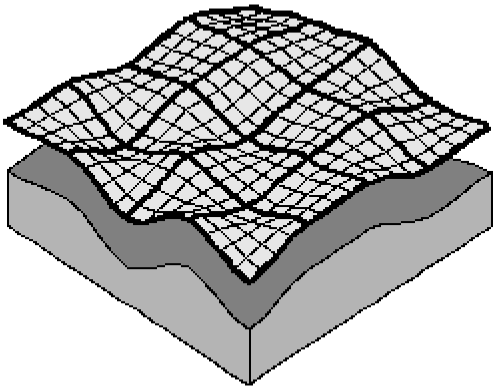
A surface modelled using PowerSHAPE is known as a *power surface*. You can imagine a power surface to be like a skin with no thickness, placed on a material.

The following figure shows how a power surface can be viewed as a skin. The skin is lifted up from the material.

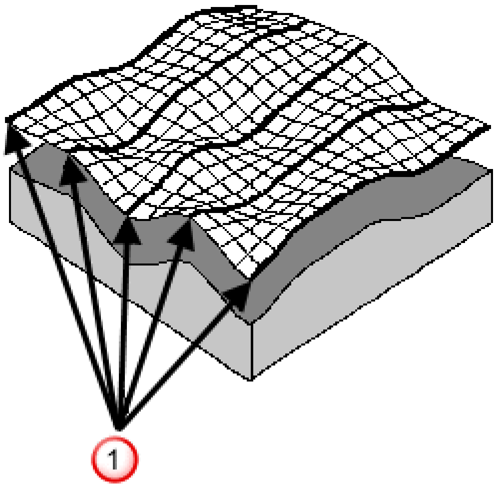


Power Surface

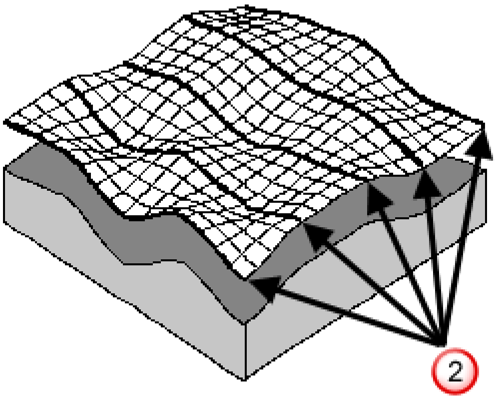
A Power Surface is made up of a network (or wireframe) of curves.



Some curves run along the surface. These curves are called longitudinals 1.

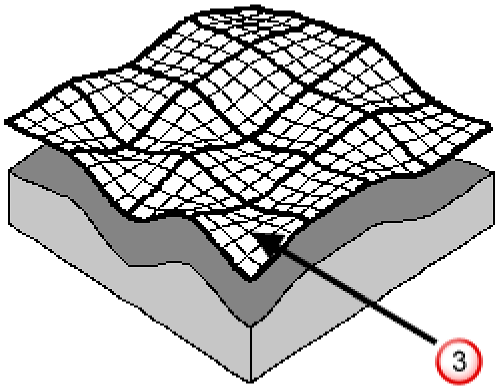


Others run across the surface. These are called laterals 2.



The surface will always pass exactly through these curves, which can each be individually edited. The curves control the key points which the surface must pass through, and the tangent directions at those points. To add finer control to a surface, create an extra lateral or longitudinal control curve.

The area of the surface between adjacent pairs of laterals and longitudinals is called a *patch* 3.



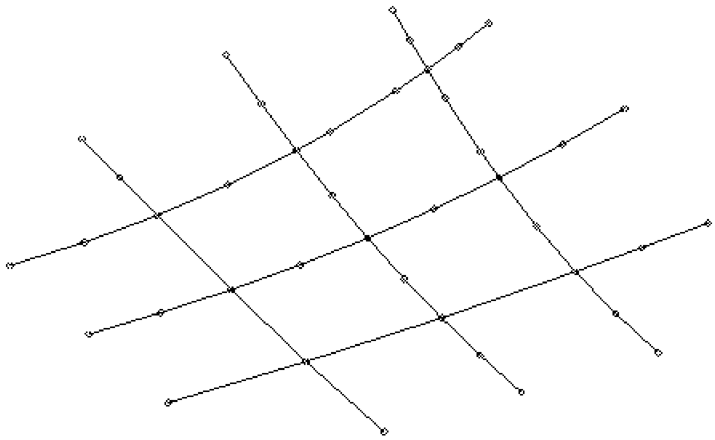
The control curves are cubic Bezier curves. This type of curve can be specified as a series of points through which the curve must pass, together with optional tangent directions into and out of each point.

Power Surfaces are mainly created from a network of curves. Fillet, split and draft surfaces are also Power Surfaces.

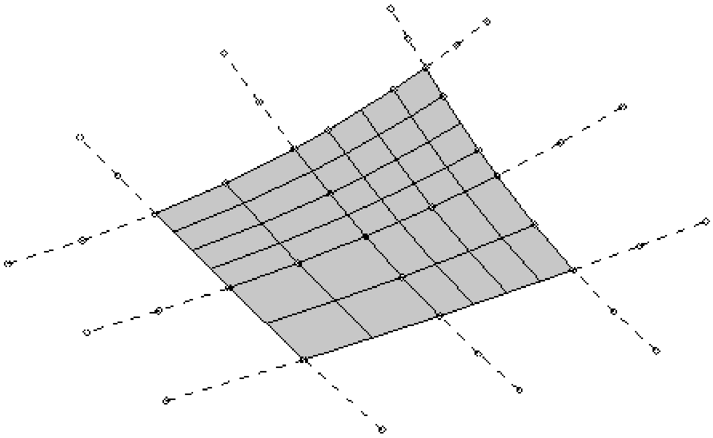
For further details, see Editing Power Surfaces (see page 112).

Surfaces from wireframe

Surfaces can be created directly from wireframe. For example, this wireframe:

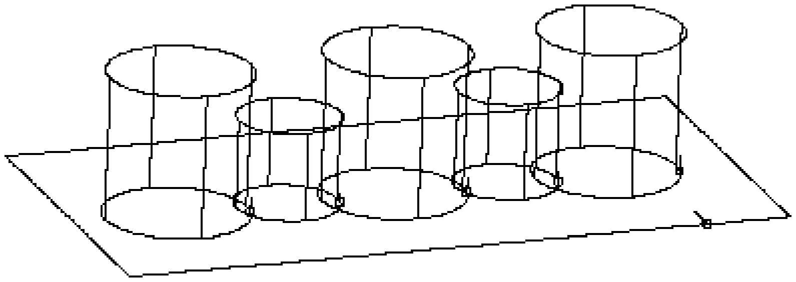


gives you the following surface:

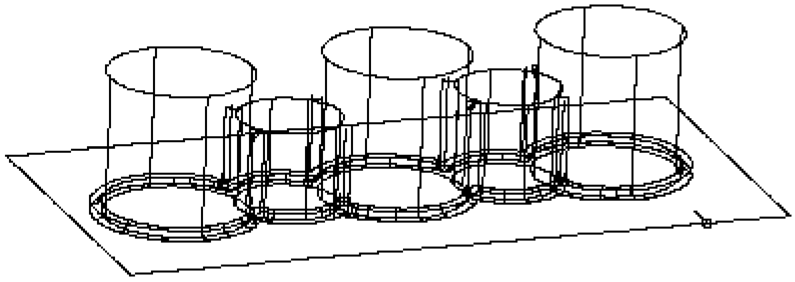


Fillet surfaces

Filleting allows you to take a selection of surfaces:



and create fillet surfaces between them to give you this:



For further details see How do I create a fillet surface? (see page 186)

Split and draft surfaces

For further details, see Preparing surfaces for manufacture. (see page 307)

Complex surfaces

More complicated surfaces are made by combining several power surfaces together. It is often convenient to define such surfaces by power surfaces which intersect or overlap. The power surfaces may be trimmed by the lines of intersection, known as trim boundaries. The power surfaces, together with their defining curves and boundaries, comprise the data for a model stored in the database.

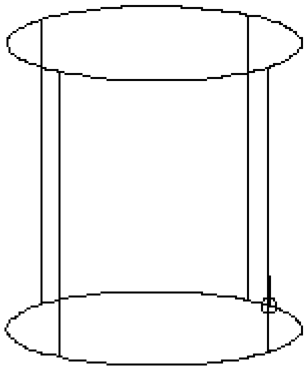
Joining surfaces

In real life, there's usually more to surface design than handling single surfaces. Most design jobs involve defining, and matching up, two or more surfaces which have to be joined together by a smoothly curved surface: For example, to avoid the risk of stress concentrations, or merely for the sake of aesthetics.

Parameterised (primitive) surfaces

In PowerSHAPE, you can quickly create surfaces that are defined by a few simple parameters, rather than a network of curves.

For example, the cylinder shown below is a parameterised surface. You may only edit its parameters: length, radius and orientation. To edit one of its curves to a new shape, it must first be converted to a power surface.

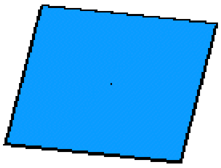


PowerSHAPE provides the following parameterised surfaces:

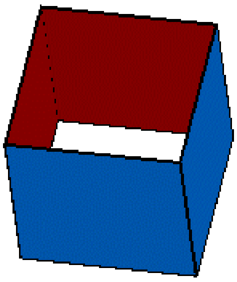
- primitives: planes, blocks, spheres, cylinders, cones, springs and tori.
- extruded surfaces.
- surfaces of revolution.

Primitive surfaces

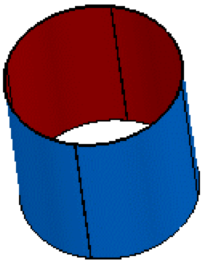
Each of the primitive surfaces is shown below.



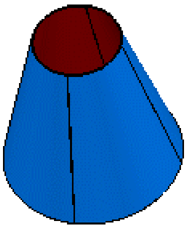
- Plane



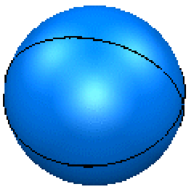
- Block



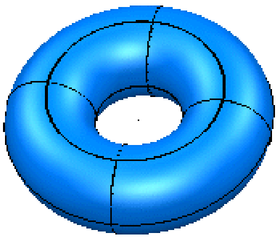
- Cylinder



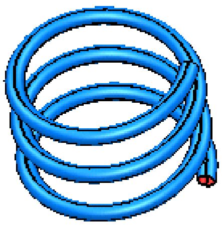
- Cone



- Sphere



- Torus

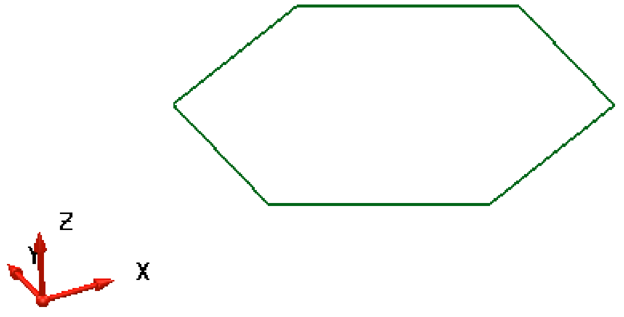


- Spring

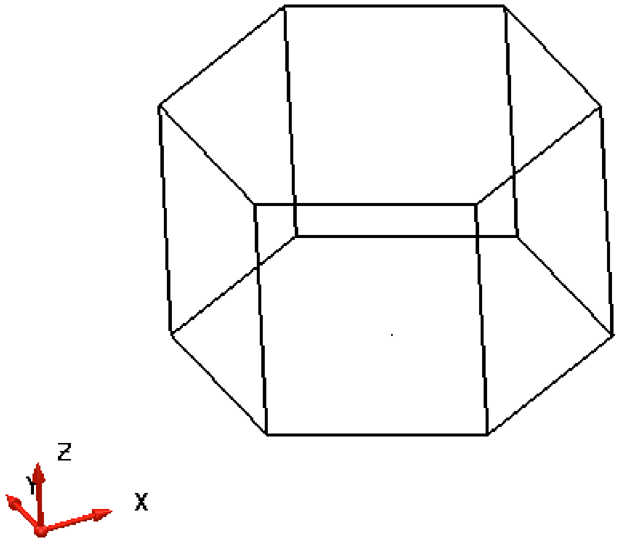
Extruded surface

You can create surfaces by extruding composite curves as shown below.

This composite curve:

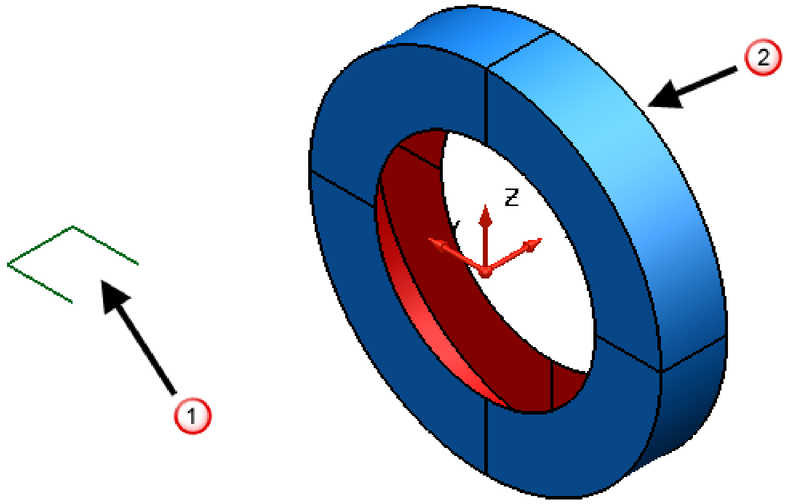


gives the following extruded surface:



Surface of revolution

A surface of revolution 1 is created from a wireframe object 2.



Other surface types

As an alternative to re-designing existing objects within the program, surface data may be transferred from another system into PowerSHAPE. Various interface standards are supported, and surfaces read in this way may be kept in a new model and edited to correct errors and design flaws.

NURBS surfaces

Non-Uniform Rational B-Spline (NURBS) surfaces are a powerful way of representing surfaces internally in a CAD program, but users find it difficult to understand and control surfaces using a complex array of knots, weights, and so on. Power surfaces are much easier to control.

NURBS surfaces can be read from other CAD systems.

BCP surfaces

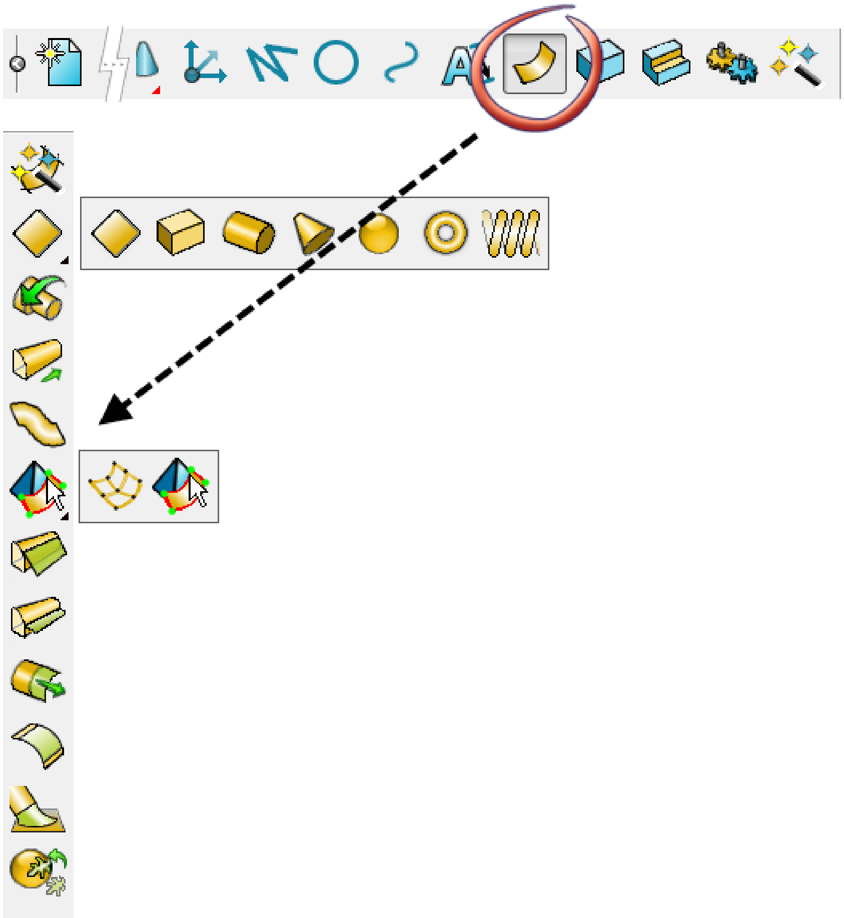
Rational Bezier Control Point (BCP) surfaces can be read from other CAD systems.

Creating surfaces

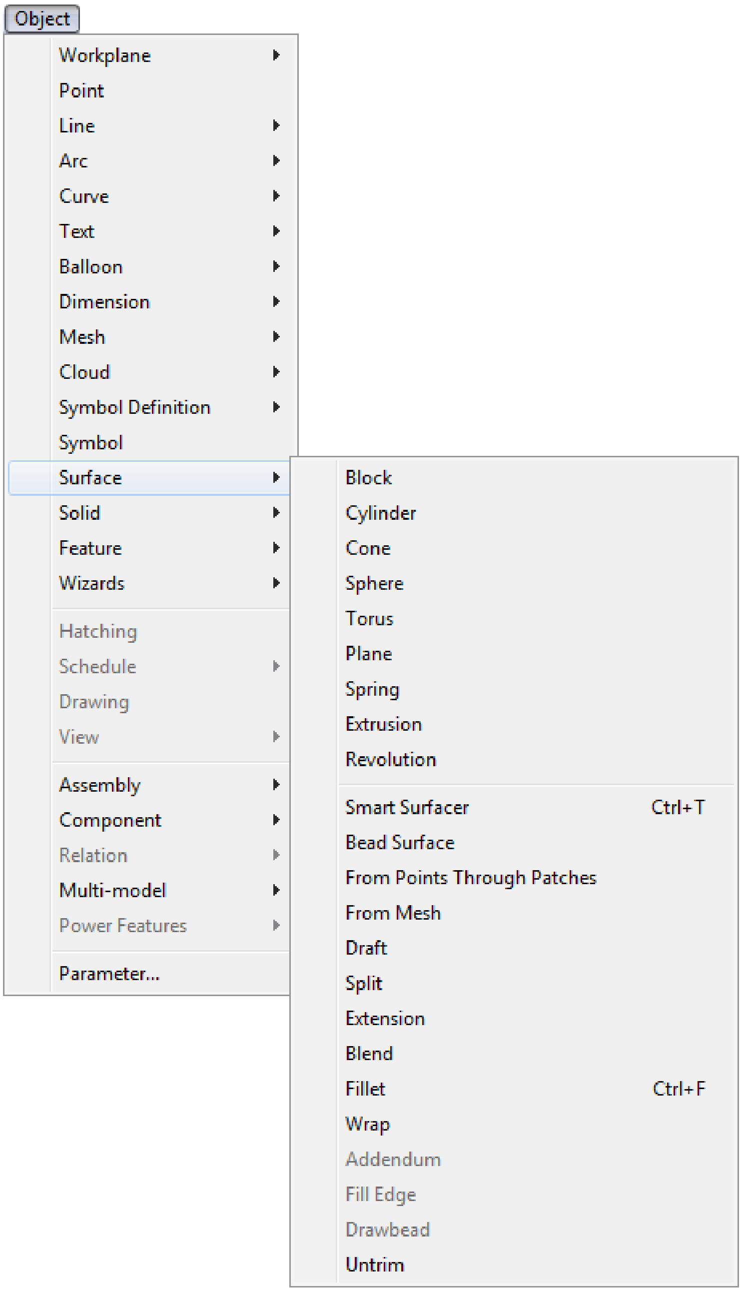
To use Surface modelling:

1 Ensure that you have a model displayed.

2 Select **Surface**  from the main toolbar. The **Surface** toolbar will be displayed.



This is the same as selecting **Surface** from the **Object** menu to see the surface creations options.



When you first start PowerSHAPE and click Surface , PowerSHAPE automatically starts primitive plane creation. If you want a different surface creation option, select the option you want from the **Surface creation** menu. PowerSHAPE remembers the last surface creation option used when you next click Surface .

Each button in the **Surface creation** toolbar is described below.

 **Smart Surfacer** (see page 54) — Click this button to open the **Smart Surfacer**.

Primitive surface (see page 4) flyout toolbar:

- **Plane**
- **Block**
- **Cylinder**
- **Cone**
- **Sphere**
- **Torus**
- **Spring**

Revolution (see page 23) — Click this button to create a surface of revolution.

Extrude (see page 29) — Click this button to create an extruded surface.

Bead (see page 103) — Click this button to create a bead surface from wireframe.

From Patches (see page 106) — Click this button to create a surface from patches

From Mesh (see page 107) — Click this button to create a surface from a mesh

Draft (see page 324) — Click this button to create a draft surface

Split (see page 309) — Click this button to create a split surface

Extend (see page 332) — Click this button to create an extension of a surface

Fillet (see page 186) — Click this button to create a fillet surface.

 **Blend** (see page 214) — Click this button to create a blend surface

 **Wrap Triangles** (see page 219) — Click this button to wrap triangles.

 **Addendum** (see page 344) (Toolmaker only) — Click this button to create an addendum surface.

 **Fill edge** (see page 340) (Toolmaker only) — Click this button to create a fillet edge surface.

 **Drawbead** (see page 353) (Toolmaker only) — Click this button to create a drawbead surface.

 **Untrim** (see page 355) (Toolmaker only) — Click this button to untrim a trimmed surface.

Creating a surface

There are various ways to create a surface according to the type of surface you require. For further details, see

What is a surface? (see page 41)

Surface creation (see page 49)

Primitive surfaces (see page 4)

Power surfaces (see page 54)

Automatic surfacing (see page 54)

Creating a surface primitive using the command window

You can specify the dimensions of a primitive before you input the origin point.

To create a block with origin at 0 0 0, you could type the following in the command window:

create surface block
length 45
width 67.89
height 43
0 0 0

If you don't specify a particular dimension, then the default value is used. You can specify the following dimensions:

radius

width

length

height

minor_radius

major_radius

base_radius

top_radius

Surfaces from wireframe (Power Surfaces)

The following sections contain detailed information on creating surfaces from wireframe:

The Smart Surfacer

The **Smart Surfacer** selects the optimum surface creation method, displays the best solution in the **Smart Surfacer** dialog and applies it to the selection. All the surface creation methods are ordered in priority of best solution and displayed in the **Smart Surfacer** dialog.

See Also:

What is Smart Surfacer? (see page 54)

Using Smart Surfacer (see page 55)

Smart Surfacer dialog (see page 57)

What is the Smart Surfacer?

The Smart Surfacer uses a single dialog to create a surface from a selection of items. The following surface creation methods are used by the Smart Surfacer:

- From network (see page 61)
- From separate (see page 73)
- Fill-in (see page 77)

- Drive-curve (see page 84)
- Two rails (see page 98)
- Plane of Best Fit (see page 83)
- Developable (currently under development)

Getting the best from the Smart Surfacer

The **Smart Surfacer** is designed to automatically create a surface from selected items using the optimum surface creation method. This is ideal for new users to create a surface quickly and easily and for less complex items.

For more advanced users, or for more complex models you may wish to use advanced options to fine tune the surface creation solution further.

We recommend you follow the sequence described below to create the surface you require.

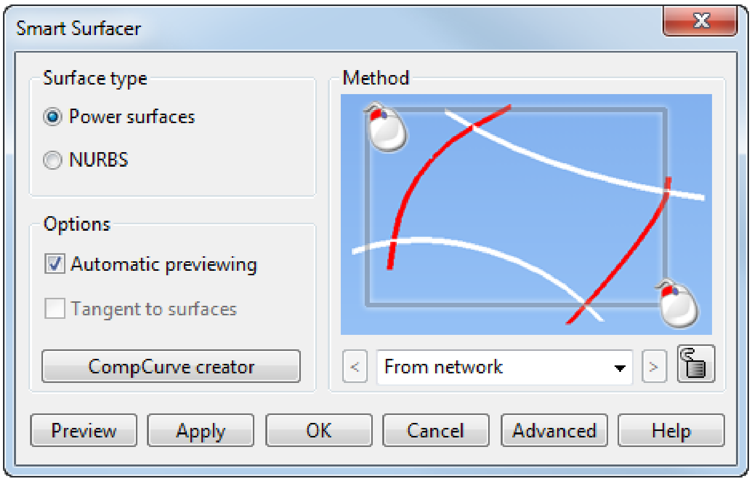
1. Start **Smart Surfacing** to create a surface from your selection. For further details see Using the Smart Surfacer (see page 55). If the created surface meets your requirements, click **OK** on the **Smart Surfacer** dialog to complete the Smart Surfacer.
2. If you wish to see solutions from other surface creation methods, click the **next** and **previous** buttons  on the Smart Surfacer dialog. This cycles through the other optimum surface creation methods and displays their surface creation solutions on the selection. If the displayed surface creation method meets your requirements, click **OK** to complete Smart Surfacer.
3. If you wish to force surface creation using another method, click on the drop-down list on the Smart Surfacer dialog to list all the surface creation methods available. Select a method to try to force it to create a surface from the selection. If surface creation is not possible using the method selected, an error message is displayed.
4. Enhance surface creation by improving the selection. Create additional Guide-Curves on the wireframe where required and start the Smart Surfacer again to see the results.
5. Select **Advanced** on the Smart Surfacer dialog to specify advanced options for the current surface creation method.

Using the Smart Surfacer

1. Click the **Smart Surfacer**  button on the Surface toolbar.

This is the same as selecting **Smart Surfacer** from the **Object/Surface** menu.

The **Smart Surfacer** dialog is displayed.



2 Select the required items to automatically create a surface.

The **Smart Surfacer** selects the optimum surface creation method, displays the best solution in the **Smart Surfacer** dialog and applies it to the selection. All the surface creation methods are ordered in priority of best solution and displayed in the **Smart Surfacer** dialog.

3 Click the **next** and **previous** buttons < > to cycle through the other optimum surface creation methods and see the effects of solutions on the selection.

4 Click **OK** to complete the **Smart Surfacer**.

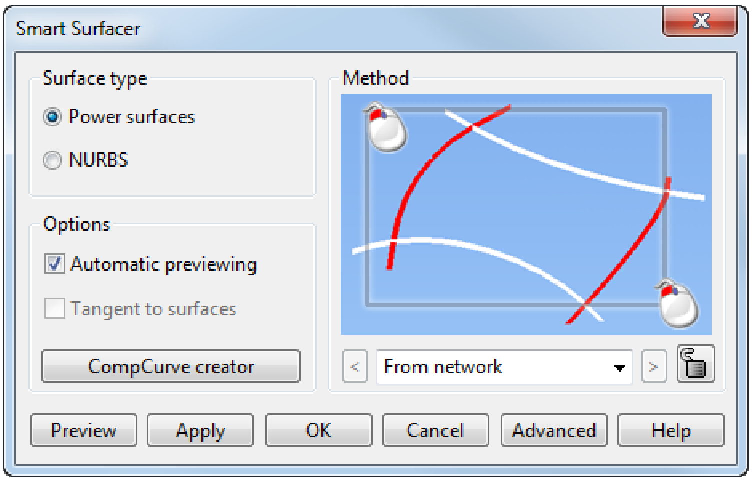
Smart Surfacer - Changing the wireframe selection

You can add or remove wireframe items in the selection easily by pressing and holding the **Shift** or **Ctrl** keys as you select them.

The Smart Surfacer automatically applies the optimum surface creation method to the new wireframe selection and grows or reduces the surface creation accordingly. The new method used is displayed in the **Method** section of the dialog and the results are displayed in the graphics window.

Smart Surfacer dialog

This dialog is used to create a surface automatically using the best solution from the available surface creation methods.



Surface Type

Select one of the surface types to change the surface type creation, if required.

PowerSurface(s) — This option is automatically selected to create a PowerSurface from the selected items, with the exception of the **Fill-In** surface creation method which is a NURBS surface.

If a fill-in surface fit is outside of the working tolerance, the words *Fit:- 0.abcd* is displayed in red text next to NURBS instead of a warning dialog. You may wish to change the fitting by using the **Advanced** options.

NURBS — Select this option to create a NURBS surface from the selected items. The result is previewed automatically if **Automatic Previewing** is ON. This option is only available for the following surface creation methods:

- **From Network**
- **From Separate**

Options

Automatic Previewing — Select this to preview the results of the surface creation method automatically. The default is ON

Tangent to Surface(s) — This option is greyed out if none of the selected curves have surface dependencies. If the selected curves have dependencies, select this option to create a surface that is tangent continuous to the surrounding surface.

*The **Tangent to Surfaces** option is automatically remembered for the next surface creation of the same type.*

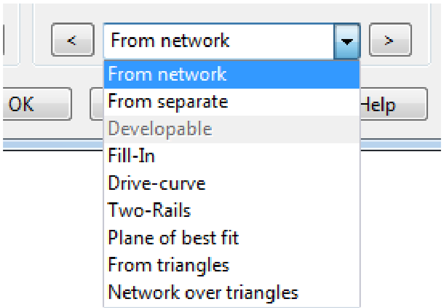
CompCurve Creator — Select this option to display the **Create Composite Curve** toolbar and to create composite curves dynamically. For further details see, Creating Composite Curves during Smart Surfacer (see page 59).

Method

— Click to cycle through the optimum surface creation methods. If the **Automatic Previewing** option is selected, the surface creation solution is displayed for the selected items. The currently selected surface creation method is displayed. The following surface creation methods can be selected:

- From Network (see page 61)
- From Separate (see page 73)
- Fill-In (see page 77)
- Drive Curve (see page 84)
- Two Rails (see page 98)
- Plane of Best Fit (see page 83)
- From triangles (see page 90)
- Network over triangles (see page 90)
- Developable (currently under development)

If the surface creation method selected by **Smart Surfacer** is not the one you want, you can force a different method by selecting it from the drop-down list.



An error message is displayed if the forced surface creation method is not possible for the selected items.

Lock — Click the **Lock** button to set a default method of surfacing. A locked method is used to create surfaces using the **Smart Surfacer** dialog, unless that method is not applicable to the selection.

The following limits apply to locking a method:

- You can lock and unlock a method at any time.

- Using the **Previous** or **Next** buttons will cancel the lock when multiple methods are available for your selection.
- Selecting a method from the drop-down list in the **Method** area will cancel the lock if that method is different to your locked method

Preview — Select to preview the results. You may make changes and preview again.

Apply — Select to apply the results.

Advanced — This displays the method **Options** dialog for the current surface creation method. These options are for the individual surface creation methods used to create the surface.

Press Del when the main page of the Smart Surfacer is displayed as a shortcut to enter the guide-curve deselection mode. Press Del again to exit the deselection mode.

For further details on the **Advanced** options for the Smart Surfacer creation methods, see:

- From Network (see page 62)
- From Separate (see page 74)
- Fill-In (see page 79)
- Drive Curve (see page 85)
- Two Rails (see page 99)
- Plane of Best Fit (see page 83)
- From triangles (see page 91)
- Network over triangles (see page 95)

Creating Composite Curves during Smart Surfacer

You can create additional composite curves dynamically during automatic surfacing, by selecting the **CompCurve Creator** button on the **Smart Surfacer** dialog.

1. Click the **CompCurve Creator** button. The **Create Composite Curve** toolbar is displayed. Use the **Composite Curve** toolbar to create the composite curve.
2. To create another composite curve use additive picking.

   Additive picking is the item selection method where you press and hold the Shift key as you select objects to add them to the current selection of objects.

   The newly created composite curves remain in the selection.

As new composite curves are added to the current selection or to the current surface, Smart Surfacer automatically uses the optimum surface creation method to create a new surface from the new selection of wireframe items, depending on the composite curves in the item selection.

Creating Guide Curves dynamically

A **Create A Guide** button on the **Options** dialogs lets you create guide-curves during **Smart Surfacer**. To display these dialogs, click **Advanced** on the Smart Surfacer dialog.

- From Network (see page 62)
- From Separate (see page 74)
- Drive Curve (see page 85)

Creating a new guide curve during surface creation

Create a guide curve using one of the following methods:

- Using **Create a Guide button:**

  1 Click the **Create A Guide** button. The button changes to **Finish The Guide**.

  *Click in the graphics area with the right mouse button to activate **Create A Guide**.*

  1 Click the positions on the wireframe to create the new guide curve. Guide-curves can only snap the the primary creation curves of the wireframe.

  2 Select **Finish The Guide** to complete the new guide curve creation.

  ***Guide-Curve** creation can be halted manually by clicking the **Finish The Guide** button.*

  ***Guide-Curve** creation also halts automatically when the number of separate selected curves is reached.*

- Using the keyboard:

  1 Press Ins on the keyboard.

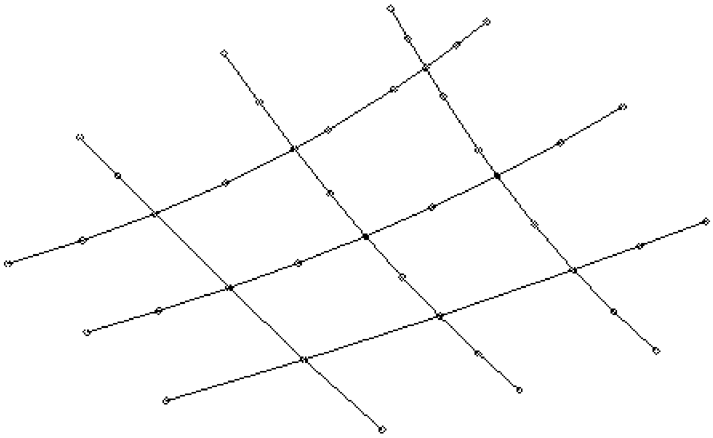
  2 Click the positions on the wireframe to create the new guide curve.

  3 Press Ins again to exit guide curve creation.

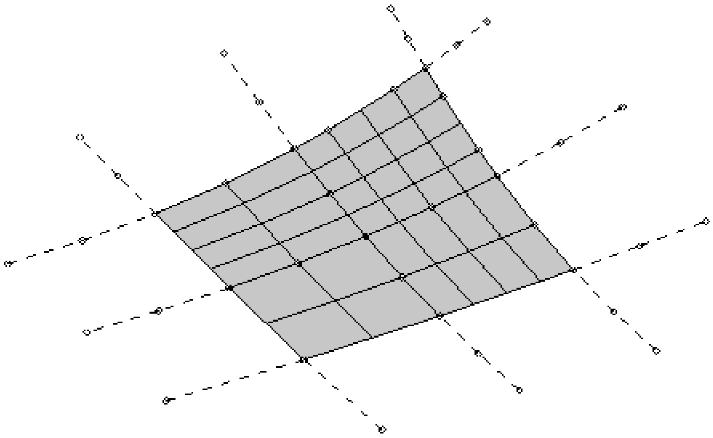
The new guide curves are added to the selection automatically, but can be deselected if required.

Creating a surface from a network of wireframe

You can construct laterals and longitudinals of the new surface from a network of wireframe objects.



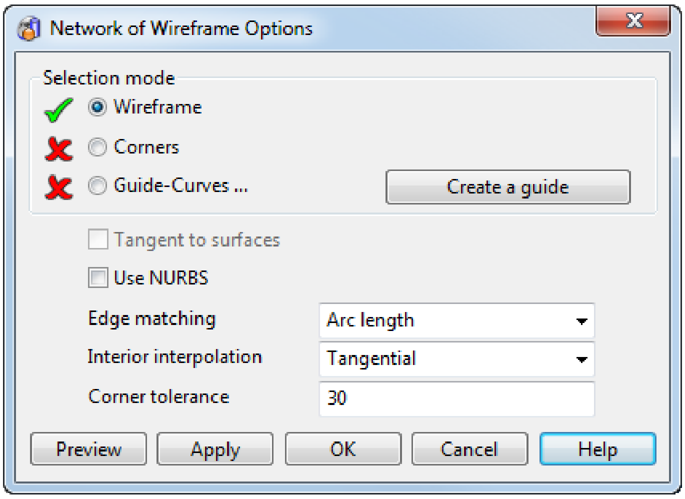
The surface is created within the outer boundary formed by the wireframe objects. PowerSHAPE sorts the wireframe into laterals and longitudinals, reversing or renumbering them as required.



1. Select the wireframe.
2. Start the **Smart Surfacer** (see page 55) to create your surface.
3. Ensure that the required creation method is displayed in the drop-down list.
4. Select **Advanced** on the Smart Surfacer dialog to change the surface creation options.
5. Use the Network of wireframe options dialog (see page 62) to change the surface options and click **OK**.
6. Click **OK** on the **Smart Surfacer** dialog to create the surface.

Network of Wireframe Options dialog

This dialog is used to change the advanced surface creation options.



Selection mode — This allows you to select objects to define your surface. Since objects can be used for different reasons, different options are provided to reflect this. You can specify whether the objects selected are to be wireframe, corners or guide curves.

- **Wireframe** — You can select the wireframe to create the surface before displaying this dialog, however this option allows you to select extra wireframe or change the selected wireframe objects.
- **Corners** — This allows you to enter graphically the corners of the surface. To do this, switch on the option. Click a position on the wireframe where you want a corner. A small circle with a cross in it marks the position. If no corners are defined, then PowerSHAPE decides where the corners of the surface are depending on the discontinuities in the wireframe selected.
- **Guide-Curves(s)...** — Sometimes none of the edge-matching options give you the shape you require because the wrong points on successive curves are joined. You can define which points should be linked by defining a wireframe object between them. In the new surface, these points will be joined by a smooth curve, which does not necessarily follow the shape of the wireframe object.

  To use this option, select the option and then select wireframe objects. To use an existing curve as a guide-curve, select this option and select the curve.

  A wireframe object which links points on laterals must cross all laterals. The wireframe must already exist in your model before using this surface creation command.

- **Create a guide** — Use this button to create guide-curves dynamically (see page 60) during Automatic surfacing.

A wireframe object that links points on laterals must cross all laterals; a wireframe object that links points on longitudinals must cross all longitudinals. The wireframe must already exist in your model before using this surface creation command.

Tangent to surfaces — This option is available when a composite curve lying on a surface is one of the selected wireframe objects. If this option is selected, the new surface is made tangent-continuous with the underlying surfaces of the selected composite curves only along its outer edges.

Use NURBS — If selected, this option creates a **NURB** surface instead of a power surface.

This option is only available with the Tangential option from the Interior Interpolation option menu. If the Linear option from the Interior Interpolation option menu is already selected when you select this option, it changes to Tangential and you are warned. Similarly, if Tangent to surfaces is selected when you select the Linear option from the Interior Interpolation option menu, Tangent to surfaces is turned off and you are warned.

Edge matching — Edge matching options (see page 64) determine how surface curves are fitted to your new surface. Choose an option from the following:

- Arc length (see page 69)
- Tangent direction (see page 66)
- Width of curve (see page 66)
- None (see page 66)
- Repoint (see page 70)

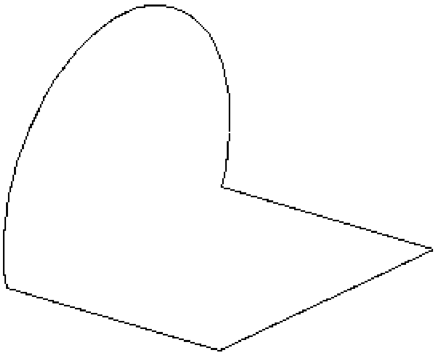
To control exactly where the points lie on the surface, design each curve with the same number of points. Use the Edge Matching option None. The surface will then be created with points on each lateral only at the points you supply.

Interior interpolation — This determines the curvature of new laterals and longitudinals added to the network.

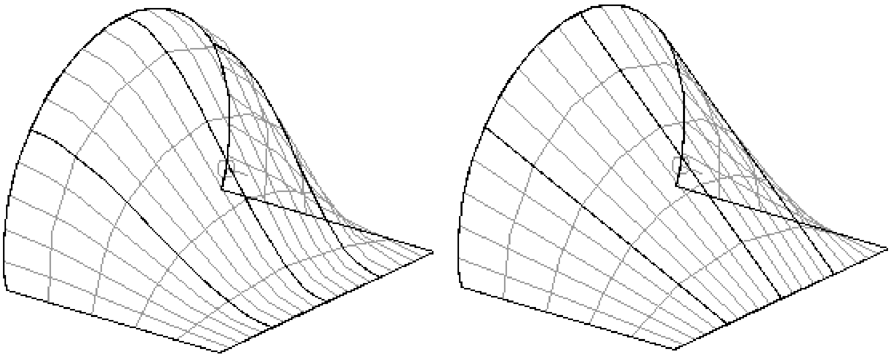
- **Tangential** preserves tangent directions as much as possible across spans. If tangents at start and end of spans are parallel, then intermediate tangents are parallel too. If they lie in the same plane, then the intermediate tangents are in the same plane too. This option is automatically used when **Tangent to surface** is ON, and is recommended for most other cases.

- **Linear** links the opposite edges of patches by lines which are similar in shape to the existing curves on the sides of the patch. This option is recommended for twisted surfaces with straight edges.

For example, consider the following wireframe.



The following surfaces are created with **Tangential** and **Linear** options.



Tangential Linear

Corner tolerance — Enter a value that is used to decide whether a point, where two wireframe objects meet, is a corner point of a surface curve. For further details, see Corner tolerance (see page 72).

Preview — Display the surface created using the current settings in the dialog. You may continue to change the settings in the dialog until you are satisfied with the previewed surface.

Apply — Save the surface. The dialog remains open for you to select more wireframe objects and continue creating surfaces.

OK — Save the surface created and close the dialog.

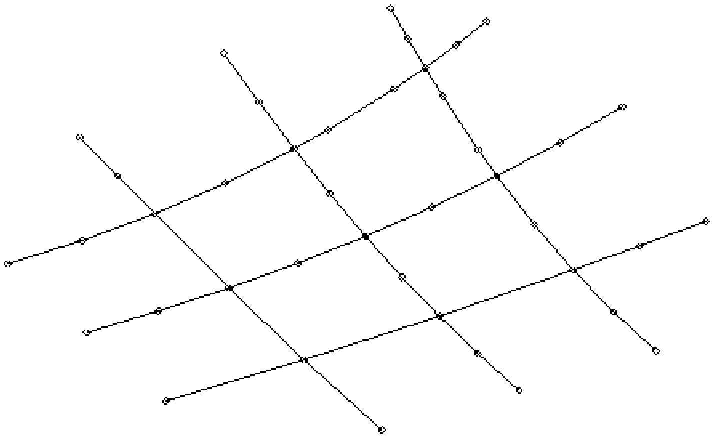
Cancel — Close the dialog and do not create a surface from the selected wireframe.

Edge matching options

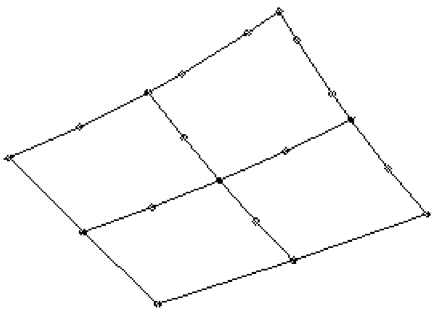
Edge matching options determine how surface curves are fitted to your surface from your selected wireframe.

PowerSHAPE creates an initial network of laterals and longitudinals based on the selected wireframe.

Consider the following wireframe.



The initial network for this wireframe is given below.



PowerSHAPE then checks that each point lies at the junction of a lateral and longitudinal.

PowerSHAPE adds extra surface points and curves to the initial network so that each point lies at the junction of a lateral and a longitudinal. How PowerSHAPE does this depends on the edge matching option you select.

You can choose from the following options:

- None (see page 66)
- Tangent direction (see page 66)
- Width of curve (see page 66)
- Arc length (see page 69) (default)
- Repoint (see page 70)

If the spans have different numbers of guide curves, the surface is created as if you had selected the default **Arc Length** option.

Edge Matching > None

This matches points if corresponding spans have the same number of points.

If the spans have the same number of points, then the new curves pass through the corresponding points. No new points are added to the surface.

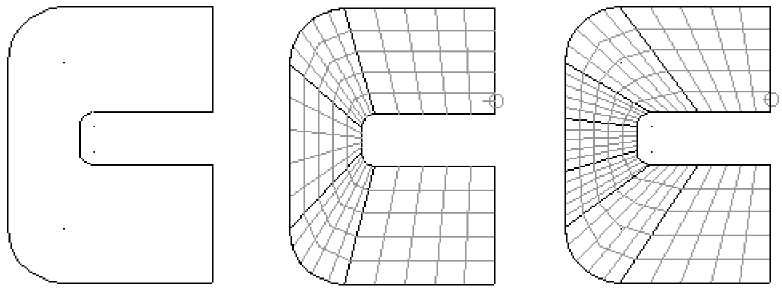
Edge Matching > Tangent direction

New surface curves are added by matching points (which do not lie at the junction of a lateral and longitudinal) with the same tangent directions.

PowerSHAPE checks points similar to the arc length option (see page 69), except the tangent directions at the points are compared.

The effect is that new curves follow features such as corners, rather than simply being regularly spaced between the given wireframe.

In the figure below, you can see the original wireframe 1 and then the surfaces created using the tangent direction 2 and arc length options 3. With the tangent direction option, the points are matched where the tangent directions are equal.

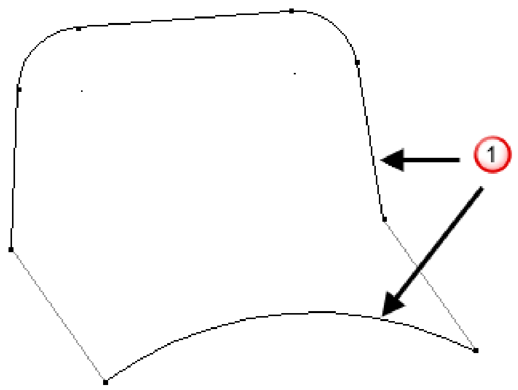


1 2 3

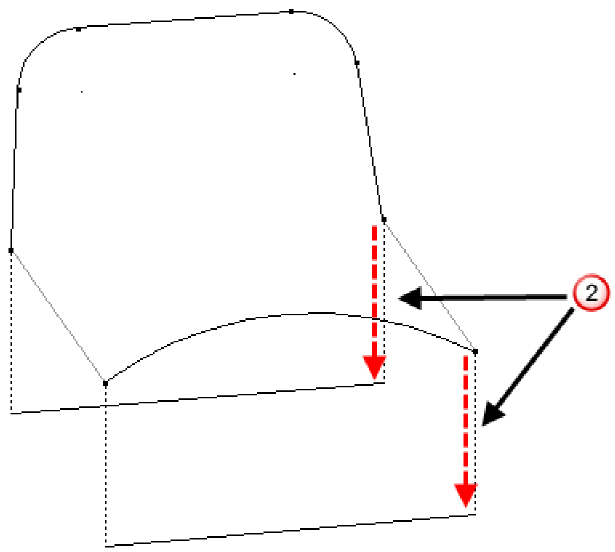
Edge Matching > Width of curve

This option is similar to the arc length option. The arc length compared points along the spans. This option projects the spans and points onto a straight line from the start to the end of each curve, and then compares the points along the lines instead of the span.

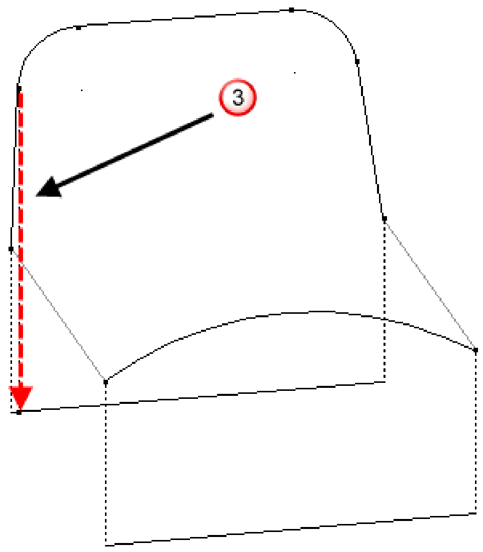
Consider the spans 1 in the following network of wireframe.



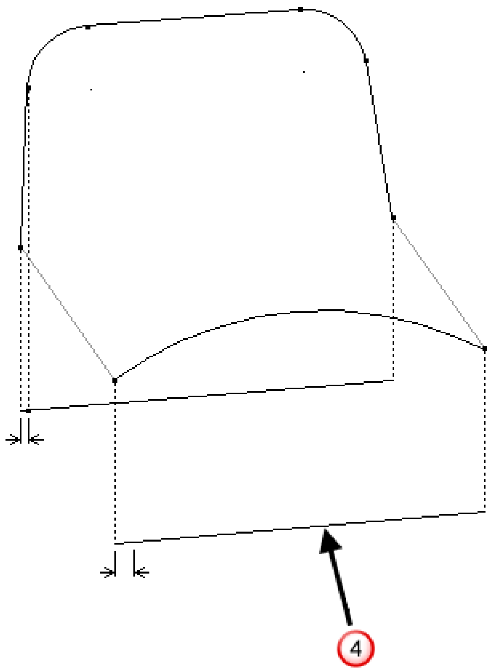
The spans are projected onto lines in the same plane 2.



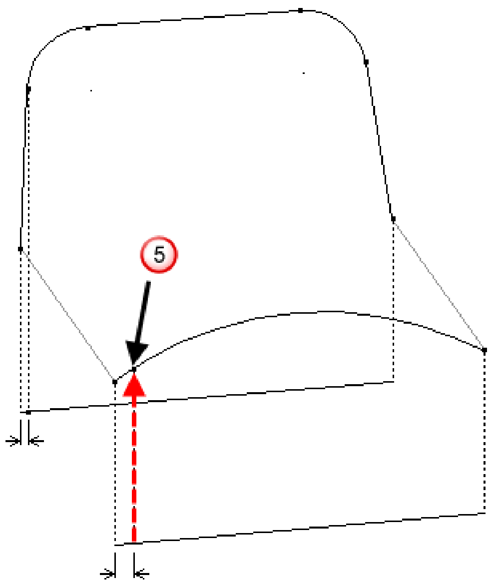
Each point is projected onto the line 3.



PowerSHAPE checks to see if points exist at approximately the same percentage along the line of a corresponding span 4.

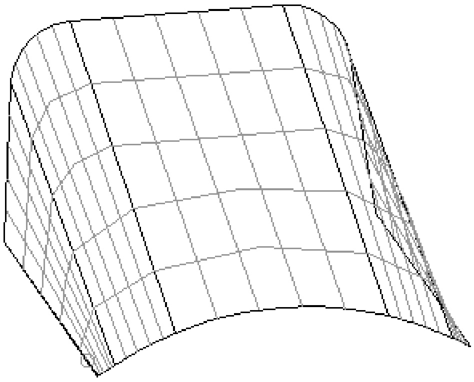


If a point exists at approximately the same percentage along the line of a corresponding span, then no point is added, otherwise a new point 5 is projected from the line and added to the span.

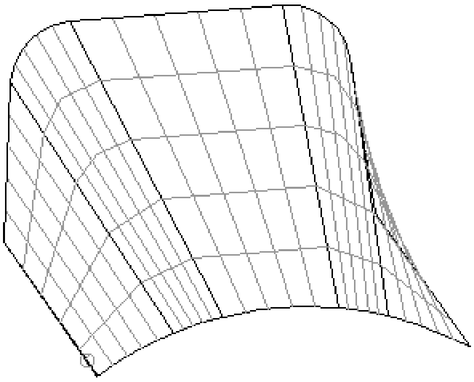


Surface curves are added to the matching points.

The complete surface generated from the network of wireframe is shown below.



You can compare the above surface with the surface (shown below) created using the arc length option.

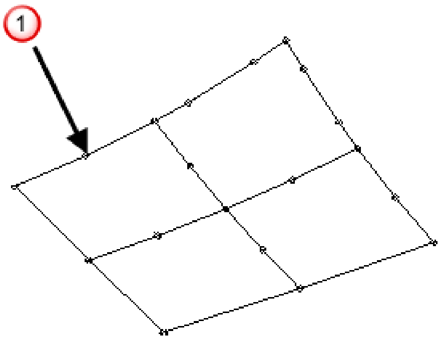


Edge Matching > Arc length

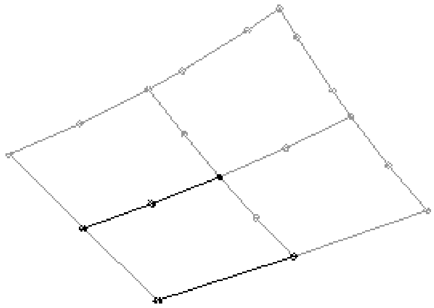
New surface curves are added by matching points (which do not lie at the junction of a lateral and longitudinal) along the length of the span.

If a point lies at x percent along a span, then PowerSHAPE checks if points lie at approximately x percent along the corresponding spans.

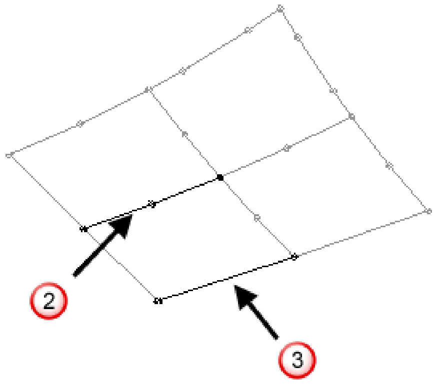
Consider the point on the surface below.



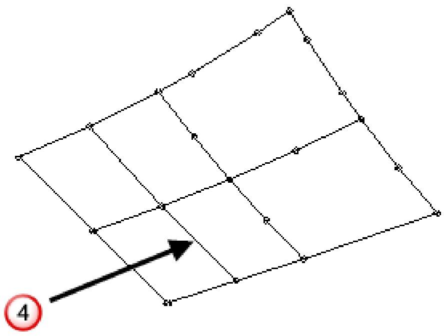
Then, the corresponding spans shown below are checked.



If a point exists at approximately x percent along a corresponding span 2, then no point is added, otherwise a point is added 3.



All the points at approximately x percent join together to define a new surface curve 4.



PowerSHAPE continues adding curves to the surface until each point is at the junction of a lateral and a longitudinal.

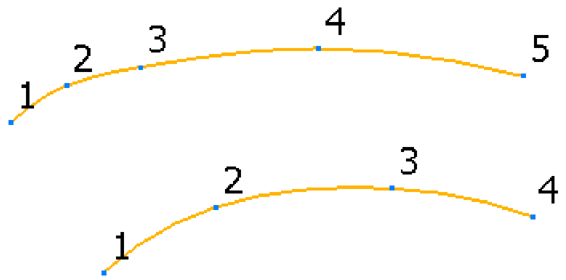
Edge Matching > Repoint

Sometimes the default option results in a surface that has too many internal surface curves close together, so that the surface is rippled. This is a problem often occurs when the surface is created from many curves, each with a different distribution of points.

When you select **Repoint:**

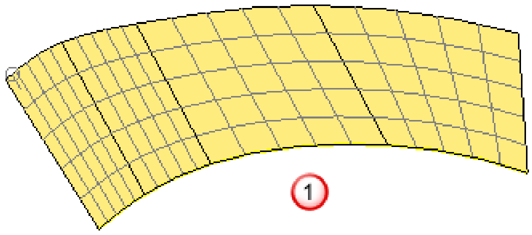
- The points within each span are redistributed so that the spans are approximately the same shape.
- The number of points on each curve is increased so that each curve has the same number of points as the curve with the most points.

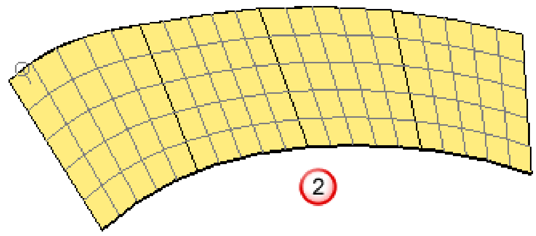
The effect of using **Repoint** can be seen using the following example:



① Using **Arc Length** (default) option.

② Using **Repoint** option.

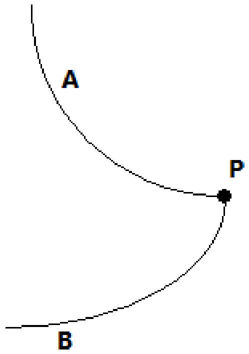




Although the resulting surface will generally be smoother, with fewer internal surface curves, repointing can alter the shape of the curve so that the surface may deviate from the original geometry.

Corner tolerance

Corner tolerance is a value defined by you. It is used to decide whether a point where two wireframe objects meet is a corner point of a surface curve. To explain this graphically, suppose we have two curves **A** and **B** as shown below:

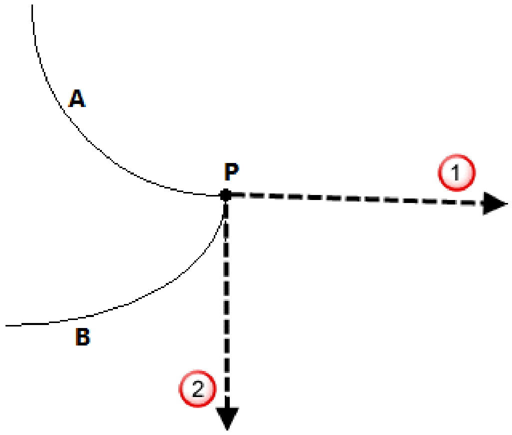


Point **P** is where they meet. Corner tolerance is used to decide whether:

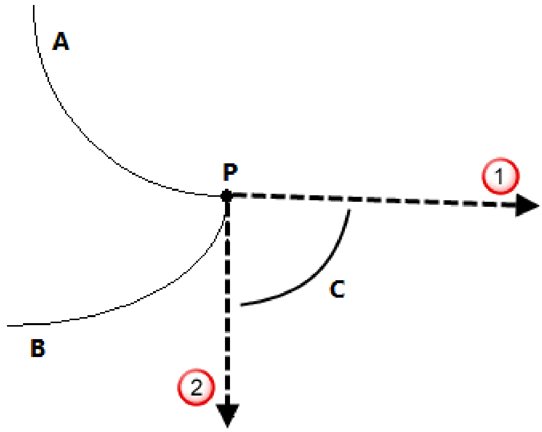
- Point **P** is a corner point of a surface curve where curve **A** defines one surface curve and curve **B** another.
- Point **P** joins curves **A** and **B** to define the same surface curve.

Corner tolerance is compared with an angle between the two curves. This angle, which we will call **C**, is defined as follows.

Two tangent vectors are defined at point **P**: one for curve **A** 1 and the other for curve **B** 2.



Angle **C** is the angle between these two vectors.

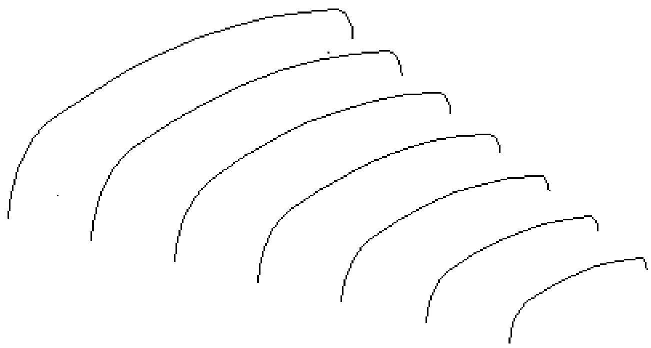


If the corner tolerance is greater than angle **C**, then point **P** is classed as a corner point. This implies that curve **A** defines one surface curve, and curve **B** another.

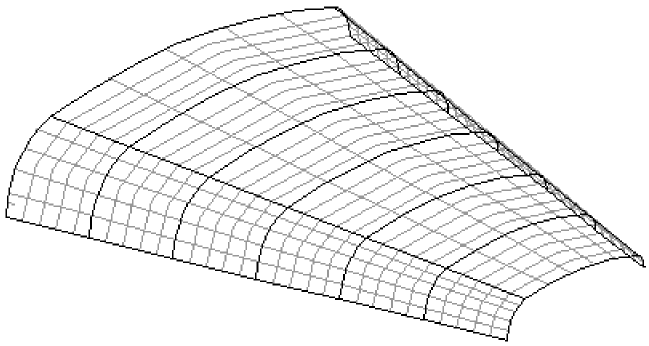
If the corner tolerance is less than or equal to angle **C**, then curves **A** and **B** are taken to be a composite curve which passes through point **P**.

Creating a surface from separate wireframe curves

You can create a surface from a set of separate curves.



The curves become laterals on the new surface. Longitudinals are added to join up the laterals.

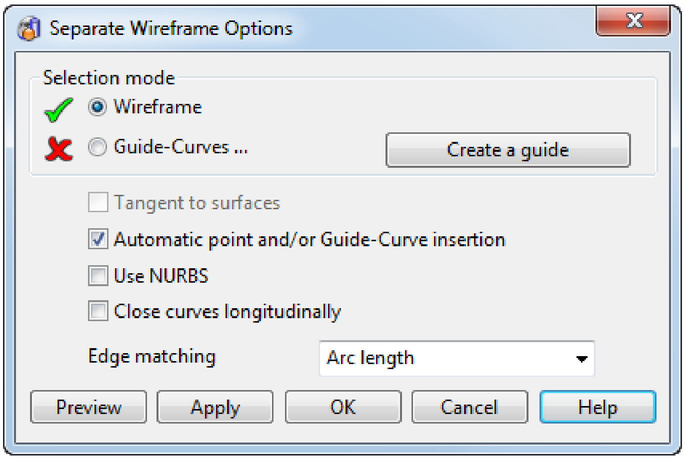


This method also creates B-spline surfaces from B-spline curves by default.

1. Use the **Separate wireframe selection** dialog to select the wireframe or change default creation options.
2. Select the wireframe.
3. Start the **Smart Surfacer** (see page 55) to create your surface.
4. Ensure that the required creation method is displayed in the drop-down list.
5. Select **Advanced** on the Smart Surfacer dialog to change the surface creation options.
6. Use the Separate wireframe options dialog (see page 74) to change the surface options and click **OK**.
7. Click **OK** on the **Smart Surfacer** dialog to create the surface.

Separate Wireframe Options dialog

This dialog is used to change the surface creation options.



Selection mode - This allows you to select objects to define your surface. Since objects can be used for different reasons, different options are provided to reflect this. You can specify whether the objects selected are to be wireframe or a guide curve.

- **Wireframe** - A tick is displayed if wireframe objects are selected, otherwise a cross is displayed. Select the wireframe objects, if none are already selected. You can also change the selection.
- **Guide-Curve(s) ...** - Sometimes none of the edge-matching options gives you the shape you require, because the wrong points on successive curves are joined. You can define which points should be linked by defining a wireframe object between them. In the new surface, these points will be joined by a smooth curve, which does not necessarily follow the shape of the wireframe object.

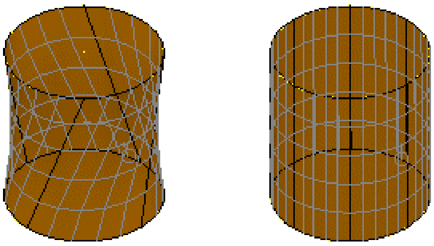
To use an existing curve as a guide-curve, select this option and select the curve.

The wireframe object which links points on laterals must cross all laterals. Also, the wireframe must already exist in your model before using this surface creation method.

- **Create a guide** - Use this option to create guide-curves dynamically during **Automatic-Surfacing**. For further details see Creating Guide Curves dynamically (see page 60).

Tangent to surface - This option is available when a composite curve lying on a surface is one of the selected wireframe objects. If this option is selected, the new surface is made tangent-continuous with the underlying surfaces of the selected composite curves only along its outer edges.

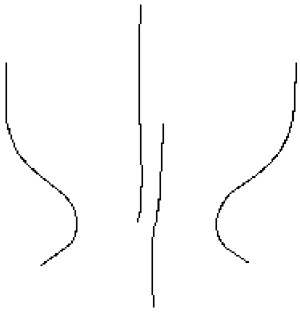
Automatic point and/or guide-curve insertion - Attempt to insert guide-curves automatically. This option may help untwist the surface as in the case below.



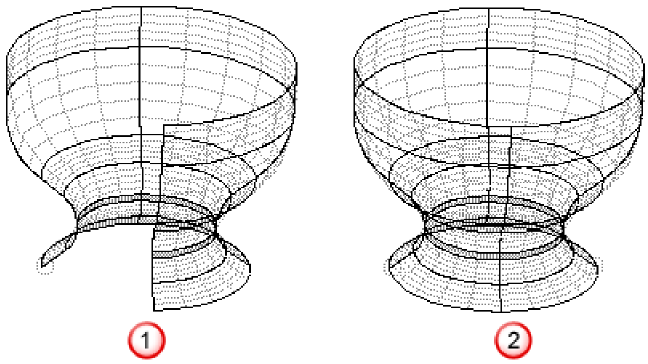
If this option is *ON*, any previously selected guide-curve will be deselected. If the option is *ON* and you manually select a guide-curve, the option will be turned off.

Use NURBS - If ON, this option creates a **NURB** surface instead of a power surface.

Close curves longitudinally - This creates a closed surface from wireframe. For example, consider the following wireframe.



With this option turned *OFF*, the surface on the left 1 is created from the wireframe. With the option *ON*, the surface on the right 2 is created.



Edge matching - Edge matching options determine how surface curves are fitted to your new surface. Choose an option from the following:

- Arc length (see page 69)
- Tangent direction (see page 66)
- Width of curve (see page 66)
- None (see page 66)
- Repoint (see page 70)
- B-Spline (only available when B-spline curves are selected)

If you are not sure which option to use, try each one to find which one gives you the most suitable surface.

The **B-spline** option creates a B-spline surface from the selected B-spline curves. All B-spline curves must have the same number of points.

To control exactly where the points lie on surface, design each curve with the same number of points. Use the Edge Matching option None. The surface will then be created with points on each lateral only at the points you supply.

Preview - Displays the surface created using the current settings in the dialog. You may continue to change the settings in the dialog until you are satisfied with the previewed surface.

Apply - Saves the surface. The dialog remains on the screen for you to select more wireframe objects and continue creating surfaces.

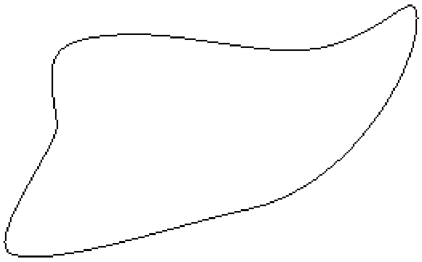
OK - Saves the surface created and removes the dialog from the screen.

Cancel - Removes the dialog from the screen and does not create a surface from the selected wireframe.

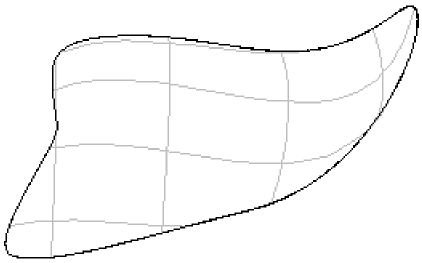
Creating a fill-in surface

You can create a surface to fill selected wireframe.

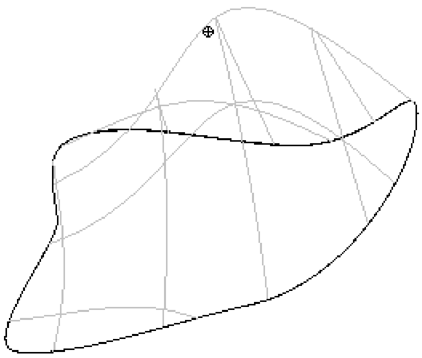
An example is shown below.



The surface fills the area enclosed by the wireframe.

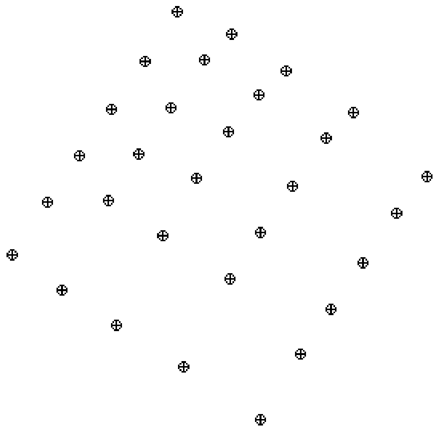


You can also make the surface pass through point objects.

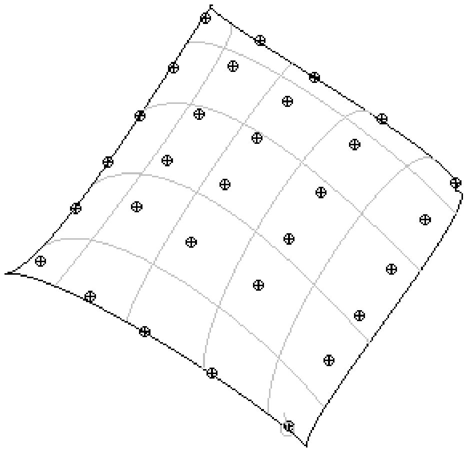


If only point objects are selected, the surface fits the points.

An example is shown below.



The surface fits the points.



Unlike most surface creation options, this command creates a trimmed surface when wireframe objects are selected. In other words, the command generates a larger surface and then creates trim boundaries to trim this surface back to the required region. This makes this method of surface creation very suitable for filling complex shaped boundaries such as those with many sides or even spikes. Conventional untrimmed surfaces cannot be easily created for such boundaries.

This can also be used to fill holes. To fill a hole, create a composite curve of the hole. Create a surface to fill the hole.

1 Select the wireframe.

2 Start the **Smart Surfacer** (see page 55) to create your surface.

3 Ensure that the required creation method is displayed in the drop-down list.

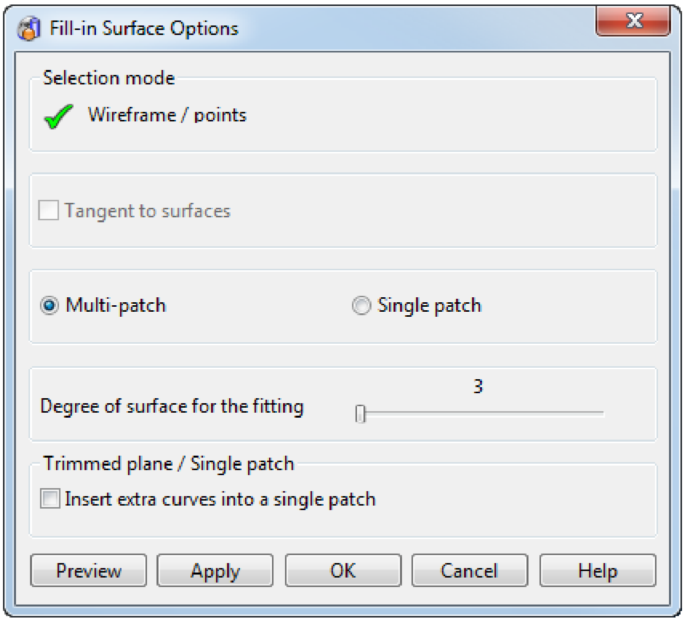
4 Select **Advanced** on the Smart Surfacer dialog to change the surface creation options.

5 Use the Fill-in surface options dialog (see page 79) to change the surface options and click **OK**.

6 Click **OK** on the **Smart Surfacer** dialog to create the surface.

Fill-in Surface Options dialog

This dialog is used to change the surface creation options.

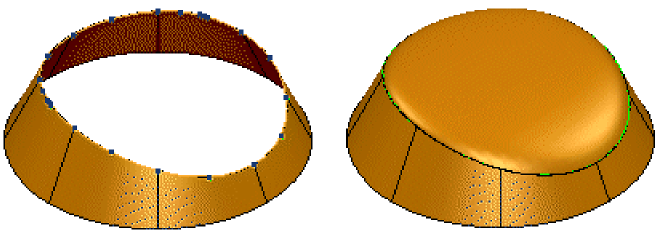


Selection mode - This allows you to select objects to define your surface.

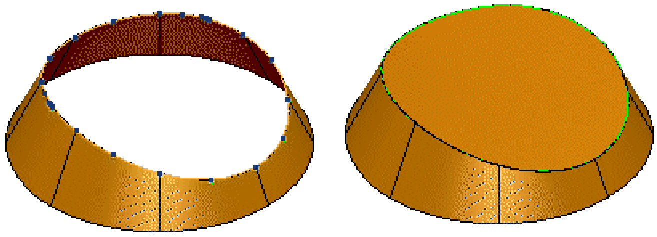
- **Wireframe/Point(s)** - A tick is displayed if wireframe or point objects are selected, otherwise a cross is displayed. Select wireframe or point objects, if none are already selected. You can also change the selection.

Tangent to surfaces - This option is available when one of the selected wireframe objects is a composite curve with dependencies on an underlying surface.

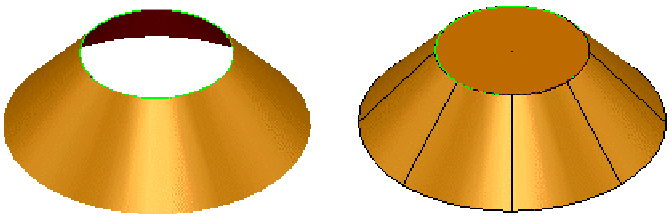
For a non-planar composite curve with dependencies, the fill-in surface will be tangent continuous by default.



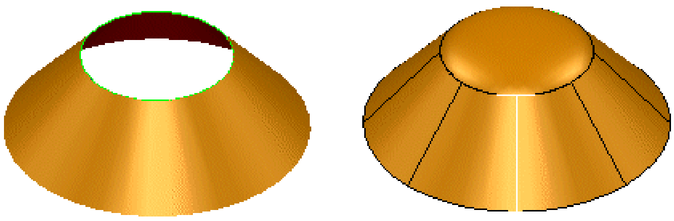
Turn off the **Tangent to surfaces** option on the **Fill-in Surface options** dialog to produce a fill-in surface that is non-tangent continuous.



For a planar composite curve with dependencies, the fill-in surface will be non-tangent continuous by default.

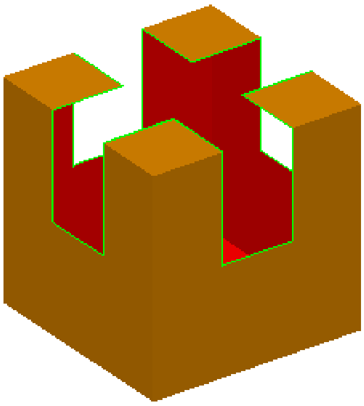


Turn on the **Tangent to surfaces** option on the **Fill-in Surface options** dialog to produce a fill-in surface that is tangent continuous.

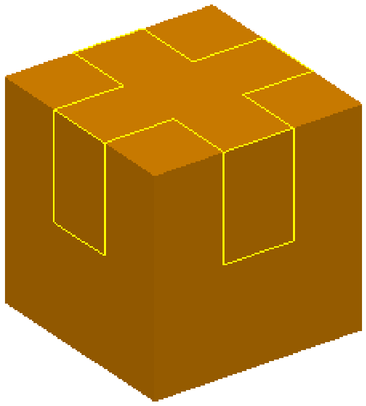


The area is filled with multiple surfaces if a single surface would be naturally creased.

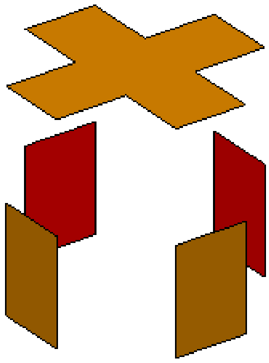
In the example below, we will add a fill in surface.



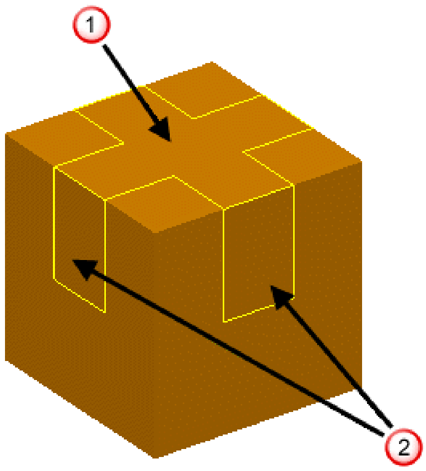
The result is shown below.



The command creates the following surfaces.

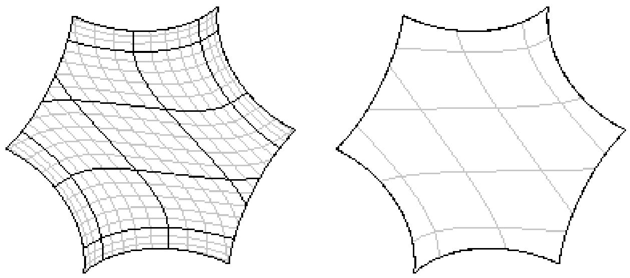


This only works if the area to be filled can be divided into a central continuous region 1 with neighbouring continuous regions 2.



It includes the case where the central region has only one neighbouring region.

Multi-patch / Single patch - This creates either a single or multiple patch surface. A single patch surface 4 is made of only two laterals and two longitudinals, whereas a multiple patch surface 3 can have many laterals and longitudinals.



3 4

*The **Single patch** option is generally better for internal smoothness and the **Multi-patch** option for tangent continuous fill-ins. The **Multi-patch** option is usually quicker and fits the wireframe more accurately than the **Single patch** option.*

Degree of surface for the fitting - You can change the degree of the new surface. The value you choose will depend on your selected objects and what you want the resulting surface to look like. For high values, the surface:

- is flexible;
- fits the selected objects more accurately;
- may have ripples.

For low values, the surface:

- is more rigid;
- flatter;
- may not fit the selected objects accurately.

Insert extra curves into a single patch - If the surface is either a trimmed plane or a single patch, you can insert a lateral and a longitudinal at the mid-points of the bounding curves using this option. You will only see the extra curves when you click **Apply** or **OK**.

*You must turn on the **Draw continuous interior laterals on NURBS** option on the **Surface** page of the **Options** dialog to see the interior laterals and longitudinals on a NURB surface.*

Preview - Displays the surface created using the current settings in the dialog. You may continue to change the settings in the dialog until you are satisfied with the previewed surface.

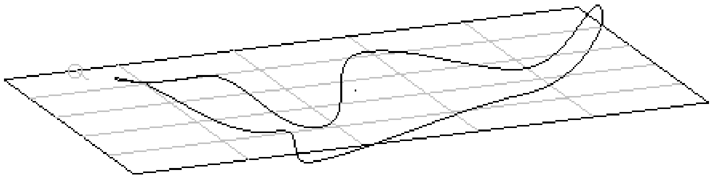
Apply - Saves the surface. The dialog remains on the screen for you to select more objects and continue creating surfaces.

OK - Saves the surface created and removes the dialog from the screen.

Cancel - Removes the dialog from the screen and does not create a surface from the selected objects.

Creating a plane of best fit

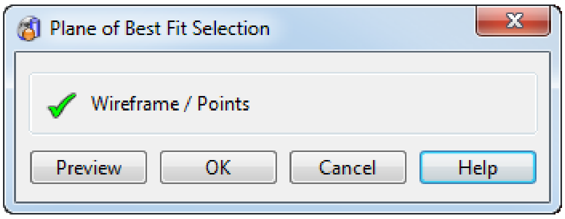
You can create a plane of best fit through selected wireframe and points.



1. Select the wireframe.
2. Start the **Smart Surfacer** (see page 55) to create your surface.
3. Ensure that the required creation method is displayed in the drop-down list.
4. Select **Advanced** on the Smart Surfacer dialog to change the surface creation options.
5. Use the Plane of best fit selection dialog (see page 83) to change the surface options and click **OK**.
6. Click **OK** on the **Smart Surfacer** dialog to create the surface.

Plane of Best Fit Selection dialog

This dialog is used to select wireframe or point objects and creates the plane.



Wireframe/Point(s) — A tick is displayed if wireframe or point objects are selected, otherwise a cross is displayed. Select wireframe or point objects, if none are already selected. You can also change the selection.

Preview — Displays the plane created using the current selection. You may change your selection until you are satisfied with the previewed plane.

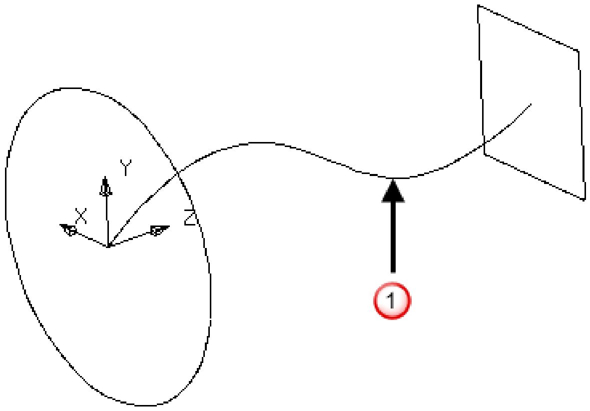
OK — Saves the primitive plane created and removes the dialog from the screen.

Cancel — Removes the dialog from the screen and does not create a plane from the selected objects.

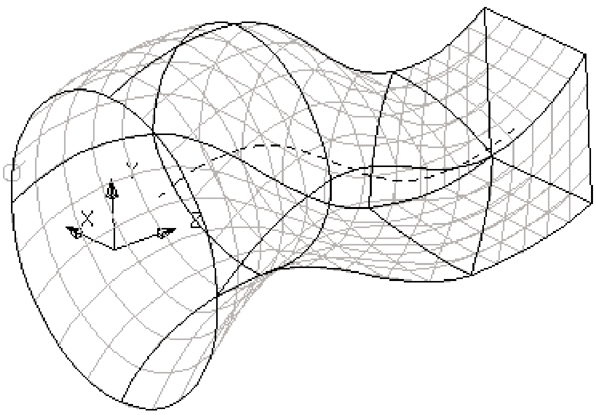
Creating a surface from a drive curve

You may want your longitudinals to follow a particular path. PowerSHAPE allows you to specify a curve to define this path. This curve is known as the drive curve. You can select a line, arc, curve or composite curve from your model to define the drive curve.

An example of a drive curve is shown below 1.



The surface is created along the drive curve.



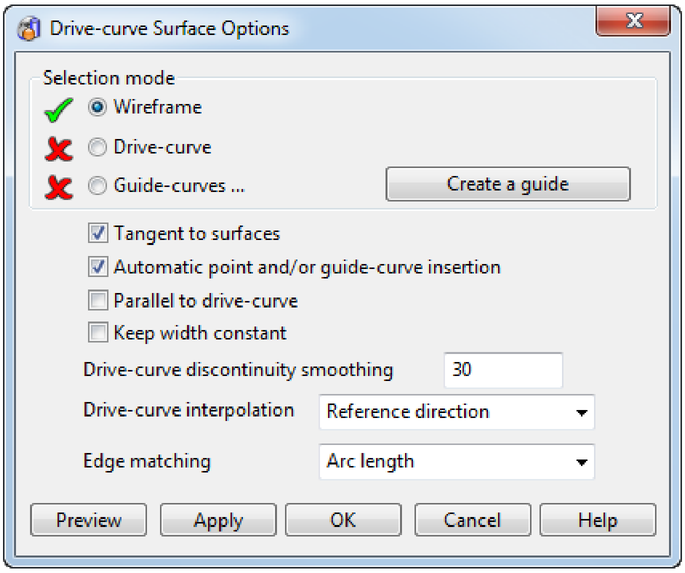
The drive curve can be open or closed. You can use a single lateral or any number of laterals. If the drive curve is longer than where the laterals lie, extra laterals are created at the end of the drive curve.

If you want the same shape for the laterals along the whole surface, use a single lateral.

1 Select the wireframe.
2 Start the **Smart Surfacer** (see page 55) to create your surface.
3 Ensure that the required creation method is displayed in the drop-down list.
4 Select **Advanced** on the Smart Surfacer dialog to change the surface creation options.
5 Use the Drive curve surface options dialog (see page 85) to change the surface options and click **OK**.
6 Click **OK** on the **Smart Surfacer** dialog to create the surface.

Drive-Curve Surface Options dialog

This dialog is used to change the default options and create the surface.



Selection mode — This allows you to select objects to define your surface. You can specify whether the objects selected are to be wireframe or a drive curve.

- **Wireframe** — A tick is displayed if wireframe objects are selected, otherwise a cross is displayed. Select the wireframe objects, if none are already selected. You can also change the selection.
- **Drive-curve** — Switch on and select a drive curve. You can select a line, arc, curve or composite curve from your model to define the drive curve.

- **Guide-curves(s)...** — Sometimes the wrong shape is created, because the wrong points on successive curves are joined. You can define which points should be linked by defining a wireframe object between them. In the new surface, these points will be joined by a smooth curve, which does not necessarily follow the shape of the wireframe object.

  The wireframe object which links points on laterals must cross all laterals. Also, the wireframe must already exist in your model before using this surface creation command.

  To use an existing curve as a guide-curve, select this option and select the curve.

- **Create a guide** — Use this to create guide-curves dynamically during Automatic-Surfacing. For further details see Creating Guide Curves dynamically (see page 60).

Tangent to surface — This option is available when a composite curve lying on a surface is one of the selected wireframe objects. If this option is selected, the new surface is made tangent-continuous with the underlying surfaces of the selected composite curves only along its outer edges.

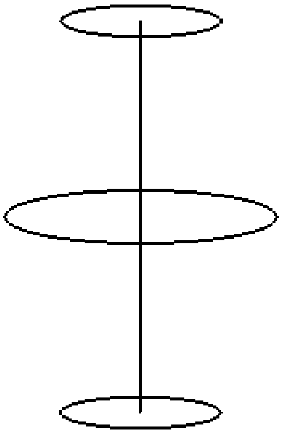
Automatic point and/or guide-curve insertion — Attempt to insert guide-curves automatically. This option is available for use with separate closed curve surfaces and drive-curve surfaces with closed sections. This is *ON* by default.

If *ON*,

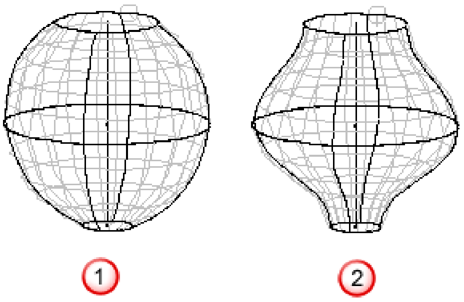
- any previously selected guide-curve will be deselected.
- the option will be turned off if you manually select a guide-curve.

This option and the manual Guide-Curve creation option are mutually exclusive.

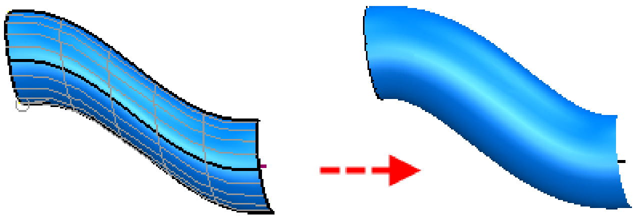
Parallel to drive-curve — This option is available when a drive curve is selected. When on, longitudinals leave and enter laterals with the same tangent direction as the drive curve. For example, consider the following wireframe.



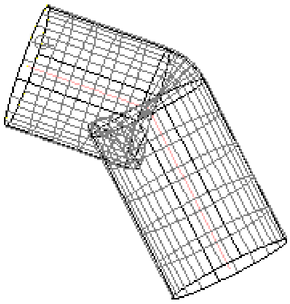
The following surfaces are created with **Parallel to drive-curve** off 1 and on 2.



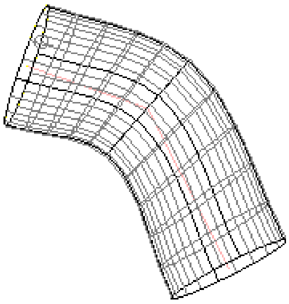
Keep width constant - If *ON*, the thickness of the surface shape is kept throughout, by adding in extra points for the complex region of the drive curve.



Drive-curve discontinuity smoothing — If the angle of discontinuity is less than the value given here, all tangent discontinuities are smoothed when generating the drivecurve surface. The model below shows the result if the discontinuity of the drive curve is greater than the **Drive curve discontinuity smoothing** angle.



If you increase the **Drive-curve discontinuity smoothing** angle, the tangent discontinuities are smoothed.



Drive curve interpolation — This controls how laterals are orientated relative to the drive curve.

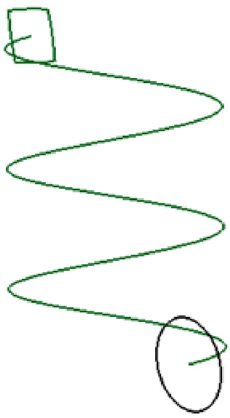
Reference direction — Laterals are orientated relative to a reference direction, which is set to be approximately at right angles to the whole drive curve. Portions of each lateral, which are at right angles to the drive curve, are made to correspond, by linking them with longitudinals. New laterals are aligned to match the orientation. This option is best for a planar or only slightly 3D drive curve.

Curvature — Successive laterals are orientated so that they twist as little as possible as we move along the drive curve. This option is best for a drive curve that is straight or lies entirely in one plane.

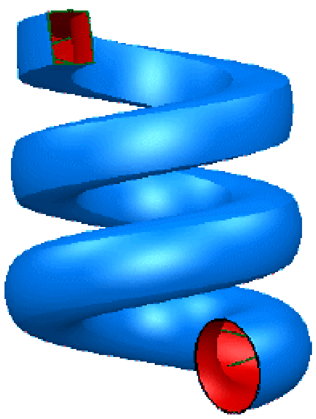
Manual reference direction — The reference direction is aligned with the Z axis of the active workplane without reference to the drive curve. This option is recommended for advanced users only.

Helical — Use this option to create helical sweep surfaces.

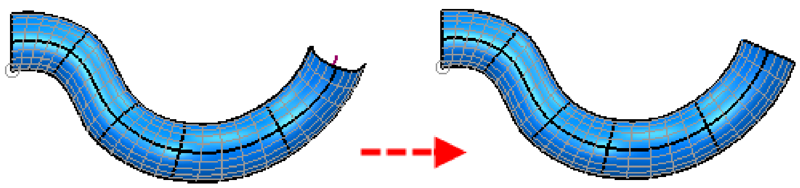
For example, consider the following wireframe.



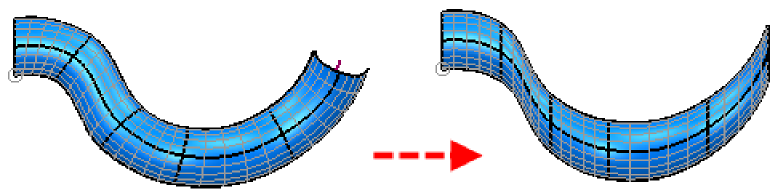
The surface below is created with **Helical** Interpolation.



Manual alignment direction — Aligns to, and rotates around Z-axis of active workplane.



Minimal lateral variation — Minimises the changes of alignment from one lateral to the next



Edge matching — Select an appropriate option from the drop-down list (see page 64).

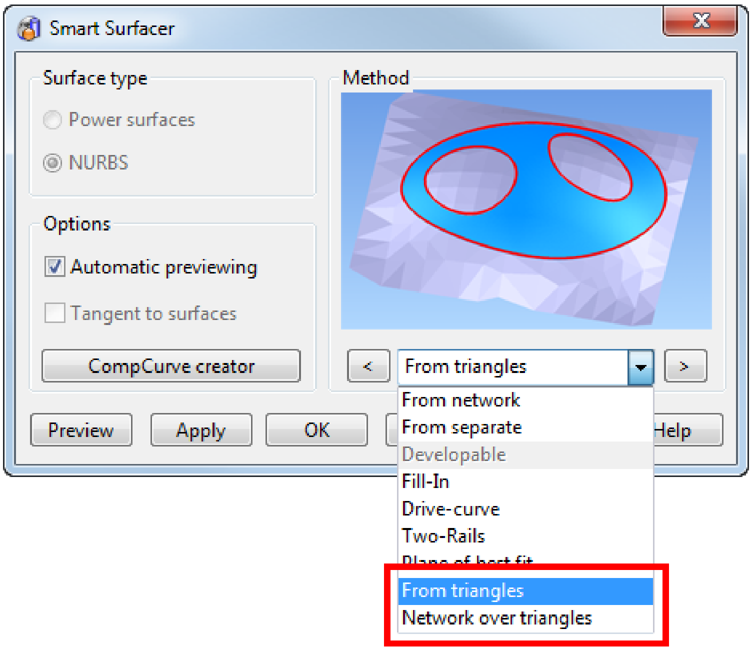
Preview — Display the surface created using the current settings in the dialog. You may continue to change the settings in the dialog until you are satisfied with the previewed surface.

Apply — Save the surface. The dialog remains on the screen for you to select more wireframe objects and continue creating surfaces.

OK — Save the surface created and close the dialog.

Cancel — Close the dialog and do not create a surface from the selected wireframe.

Creating a surface from triangles



To create a surface from triangles:

1 Select a model that includes a mesh and wireframe.

2 Click  (*Surface toolbar*) to start **Smart Surfacer**.

3 Select one of the following methods from the drop-down list:

From triangles

Network over triangles

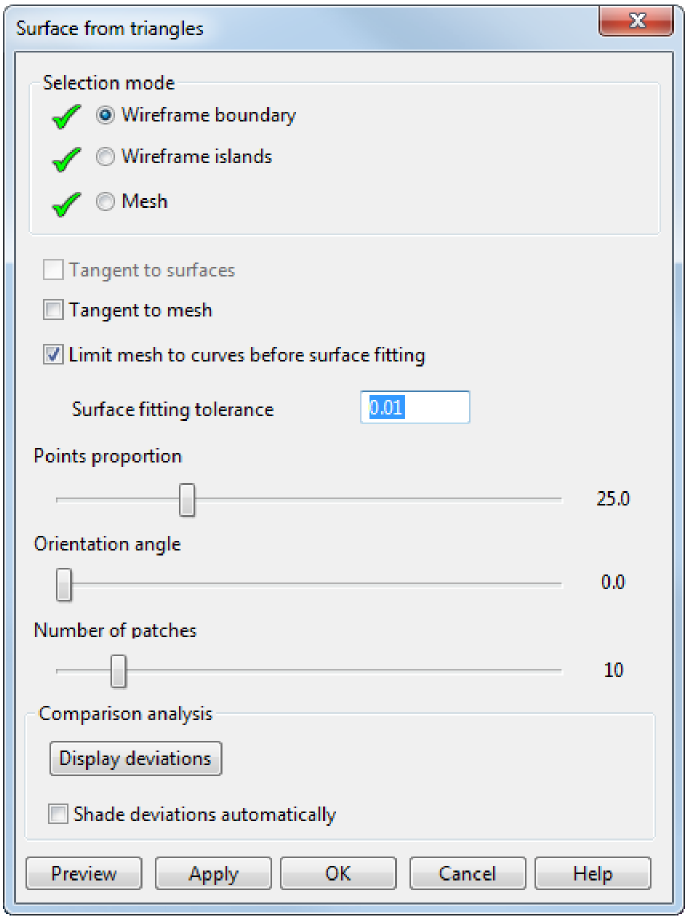
4 Click **Advanced** to display the options page for the method you have chosen:

Surface from triangles dialog (see page 91)

Network of wireframe over triangles dialog (see page 95)

Surface from triangles options

This option uses selected wireframe and mesh points to create a surface.

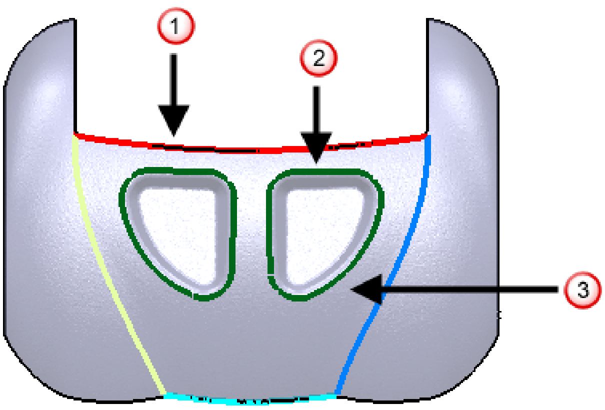


1 Using the example below as a guide, select the following on your model:

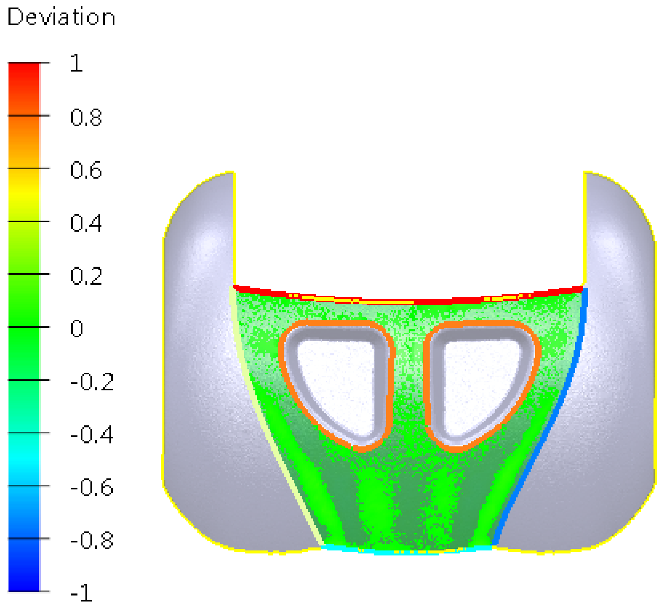
1 Wireframe boundary

2 Wireframe islands

3 Mesh



Once the mesh is selected, the **Comparison analysis** colour map and deviations are displayed by default on the preview of the surface:



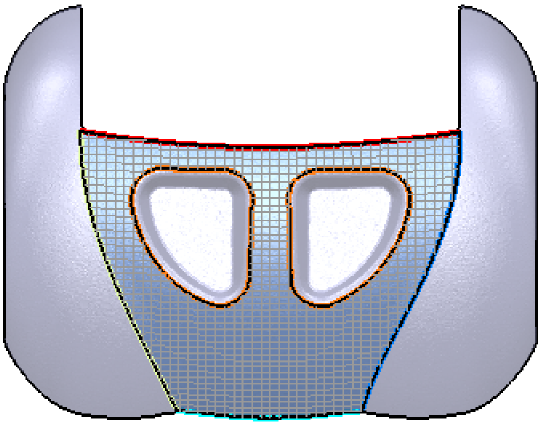
2 Select the following options as appropriate:

- **Tangent to the surrounding surfaces** to create the surface tangent to the surrounding surfaces on which the selected curves are dependent.
- **Tangent to mesh** to create the surface tangent to the mesh.
- **Limit mesh to curves before surface fitting** to create a surface from a mesh that has been limited. This is the default setting and will create a surface that ignores any triangles that are outside the limiting curve. If this option is deselected, the surface that is created is fitted to the whole mesh.

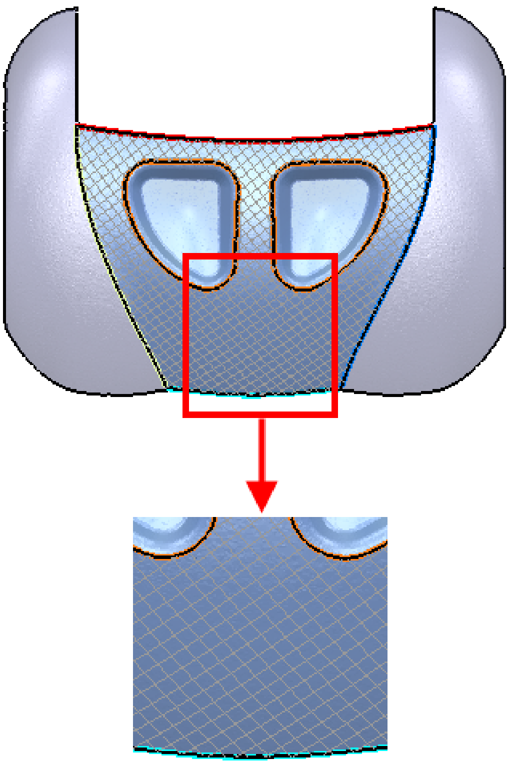
3 Enter the **Surface fitting tolerance** to indicate the allowable discrepancy between the mesh and the created surface. A large tolerance will produce a surface that does not closely match the underlying mesh.

4 Move the slider to change the **Points proportion**.This sets the percentage of points inside the boundaries that are used to fit the surface.

5 Move the slider to change the **Orientation angle** that defines the direction of the surface laterals and longitudinals. In the following example, shade deviations and the colour map have been deselected so that the surface laterals and longitudinals are visible.

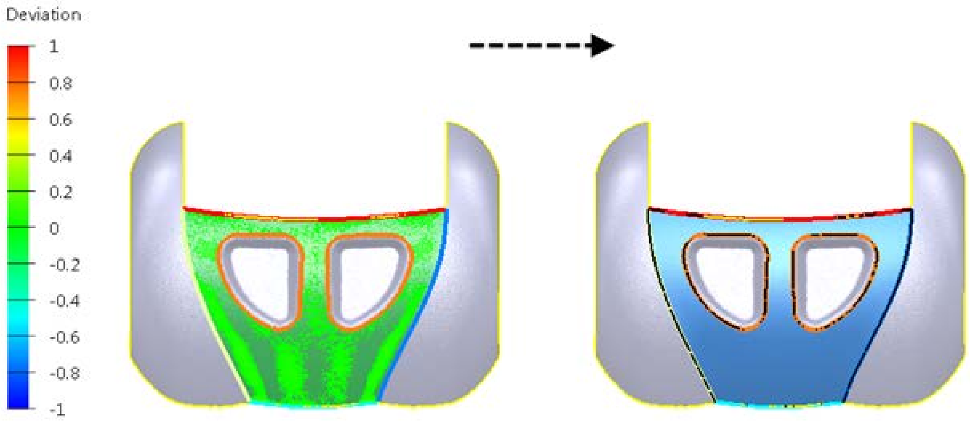


The default angle is *0.0*; changing the angle may produce a surface that is a better fit for the triangles. In the example below, the **Orientation angle** has been changed to *45*°.



6 Move the **Number of patches** slider to adjust the number of patches. Increasing the number of patches will improve the fit of the surface to the triangles.

7 Click **Hide deviations** to display the model without the colour map.



If the colour map is not displayed, click **Display deviations** to show the deviations and the colour map.

8 **Shade deviations automatically** is deselected by default. Select this option to change the default setting when the dialog is next displayed.

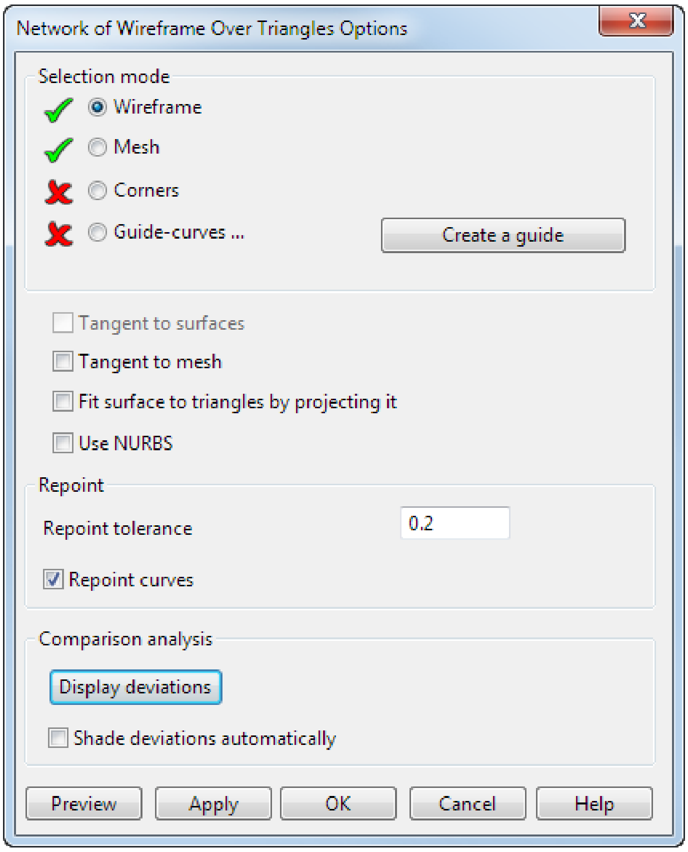
9 Click **Preview** to see the results. If necessary, make changes and click **Preview**.

10 Click **Apply** to create the surface. The dialog continues to be displayed. Create additional surfaces as required.

11 When all the surfaces have been created, click **OK**. To cancel the surface creation, click **Cancel**.

__Cancel__ will cancel all the surfaces that you have created whilst the dialog has been displayed.

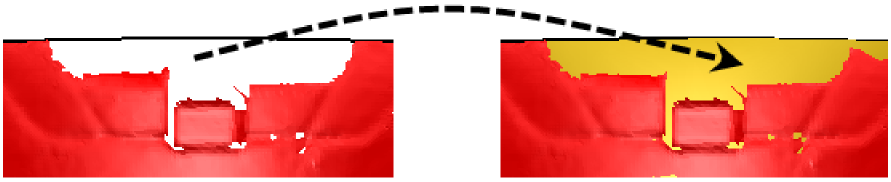
Network of wireframe over triangles



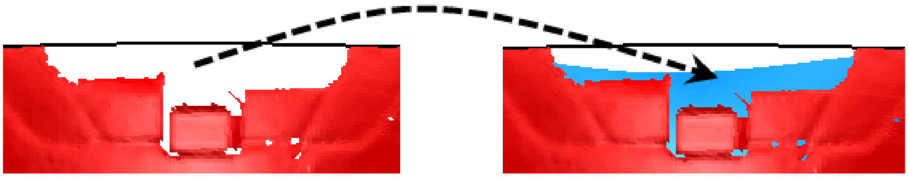
1. Select the following on your model:
   - Wireframe
   - Mesh
   - Corners (optional)
   - Guide curves (optional)

   Use **Corners** and **Guide curves** to help define the shape of the surface.
2. Select the following options as appropriate:
   - **Tangent to surface** to create the surface tangent to the surrounding surfaces on which the selected curves are dependent.
   - **Tangent to mesh** to create the surface tangent to the mesh.

- **Fit surface to triangles by projecting it**. Use this option to project the surface to the curve as shown in the following example:



Selecting this option is useful when a mesh contains a damaged edge. It uses a curve that bridges the gap in the damaged mesh, where there are missing triangles. When the surface is created, this option causes the surface to be projected to the curve that was created.

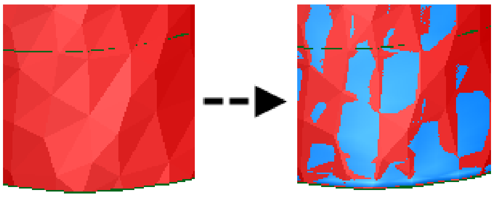


If the option is deselected (default), the edge of the surface is matched to the damaged edge of the mesh, as shown above.

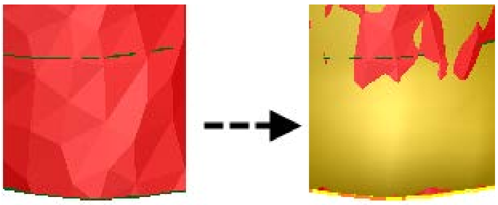
- **Use NURBs** to create a **NURB** surface instead of a power surface.

3 Enter **Repoint tolerance**.

4 Deselect **Repoint curves** to produce a surface that closely matches the triangles, including any triangles that might contain errors. In the example below, this produces an unsmooth surface that is almost identical to the mesh.



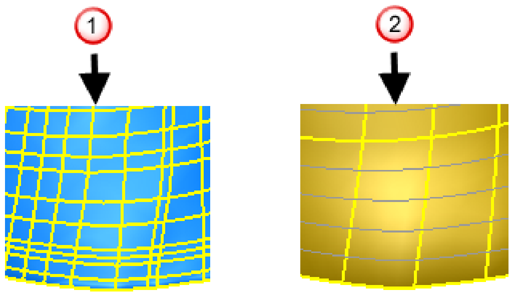
Selecting **Repoint curves** is recommended if the curves have a lot of randomly distributed points. Repointing the curves will produce a smoother surface, as shown below:



The following diagram shows the surface construction that result from the two options:

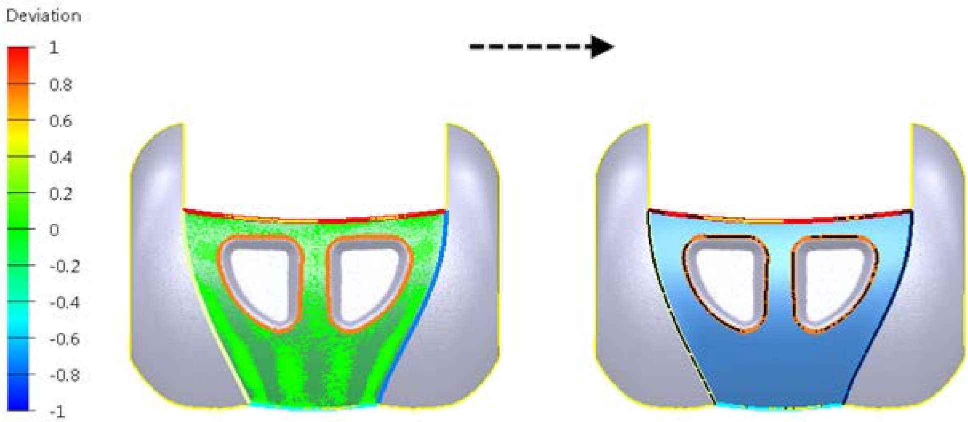
1 With **Repoint curves** deselected, all the internal surface curves are displayed.

2 With **Repoint curves** selected, the surface curves are evenly distributed as the result of repointing.



Repointing can alter the shape of the curve so that the surface may deviate from the original geometry.

5 Click **Hide deviations** to display the model without the colour map.



If the colour map is not displayed, click **Display deviations** to show the deviations and the colour map.

6 **Shade deviations automatically** is deselected by default. Select this option to change the default setting when the dialog is next displayed.

7 Click **Preview** to see the results. If necessary, make changes and click **Preview**.

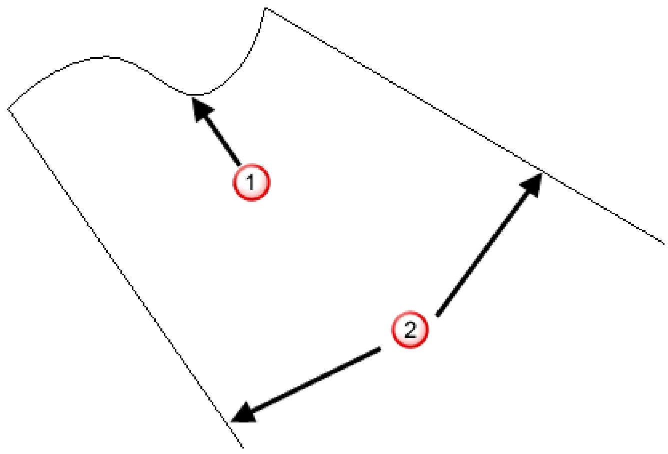
8 Click **Apply** to create the surface. The dialog continues to be displayed. Create additional surfaces as required.

9 When all the surfaces have been created, click **OK**. To cancel the surface creation, click **Cancel**.

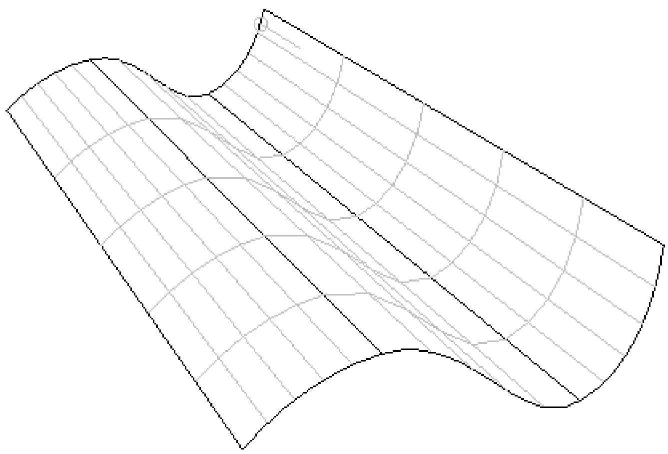
***Cancel** will cancel all the surfaces that you have created whilst the dialog has been displayed.*

Creating a surface from two rails

You can create a surface from a section curve 1 and two rail curves 2.



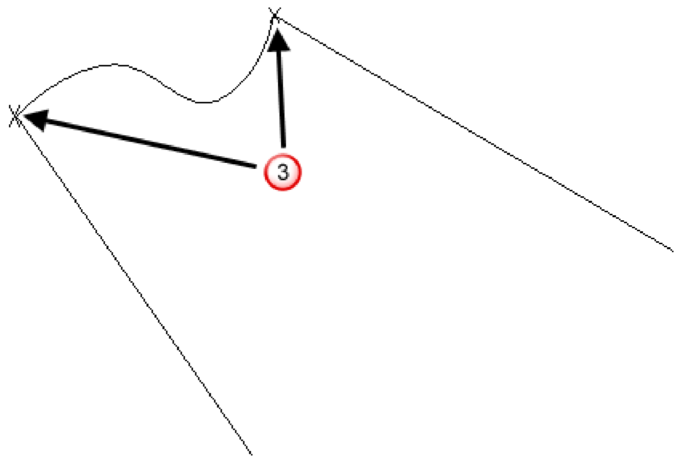
Copies of the section are positioned at adjacent points along the two rails. These copies are scaled and rotated versions of the original one. The surface is created from the net.



To create a surface using this method, the section and rails must satisfy the following:

- The section must be a single, open wireframe object.
- Each rail must be a single wireframe object.
- The two rails must have the same number of points and both be either closed or open.

- The ends of the section must touch the ends of the rails 3.

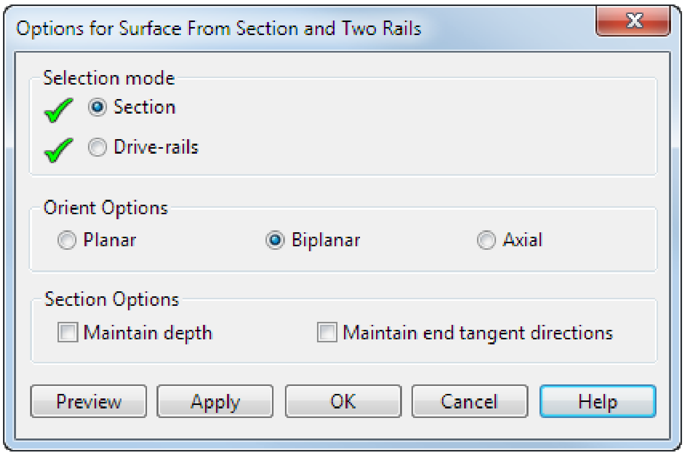


To create a surface from a section and two rails:

1 Select the wireframe.
2 Start the **Smart Surfacer** (see page 55) to create your surface.
3 Ensure that the required creation method is displayed in the drop-down list.
4 Select **Advanced** on the Smart Surfacer dialog to change the surface creation options.
5 Use the Options for surface from section and two rails dialog (see page 99) to change the surface options and click **OK**.
6 Click **OK** on the **Smart Surfacer** dialog to create the surface.

Options for surface from section and two rails dialog

Use this dialog to set the options that will be used to produce the surface.



Selection Mode

Section - is displayed if there is no wireframe selected. Select a wireframe object for the section. changes to when a wireframe object is selected 1. You can also change the wireframe selection.

Drive-rails - is displayed if there is no section selected. Select the option, then select two wireframe objects for the rails. changes to when two rails are selected 2.

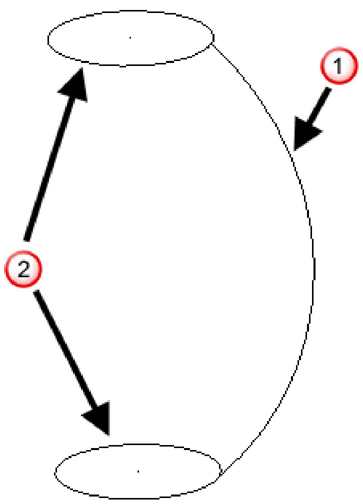
Orient Options

Select one of the following options:

- **Planar**
- **Biplanar**
- **Axial**

In most cases, the default option should give you the correct result. If you click **Preview** and the resulting surface is not what you expect, use one of the other options.

Example 1

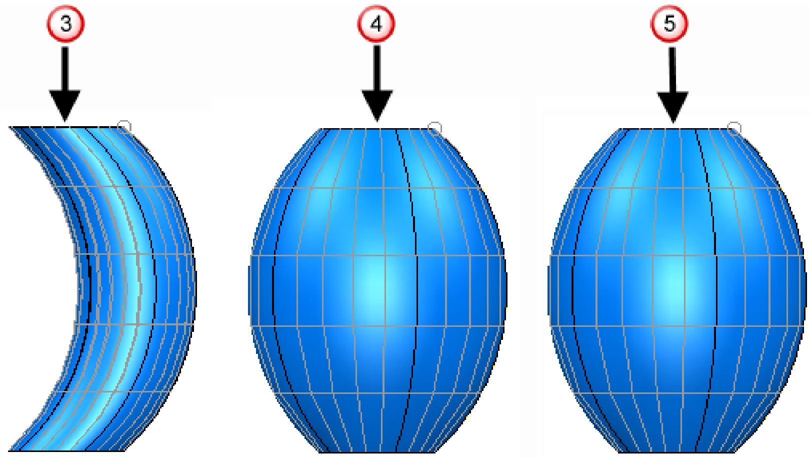


The example above produces the following surfaces:

3 Planar option

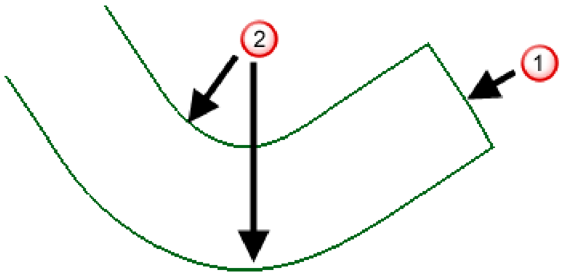
4 Biplanar option

5 Axial option

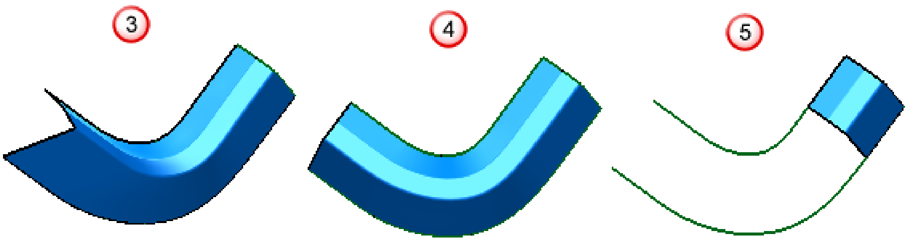


The results for **Axial** and **Biplanar** look the same.

Example 2



The example above produces the following surfaces:

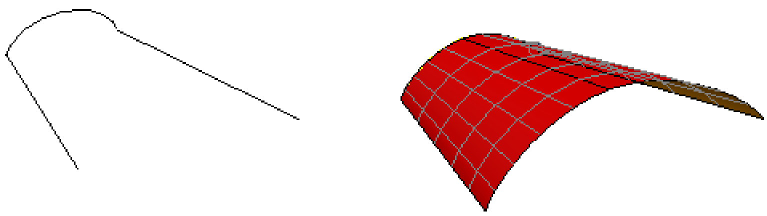


The three options produce three different results.

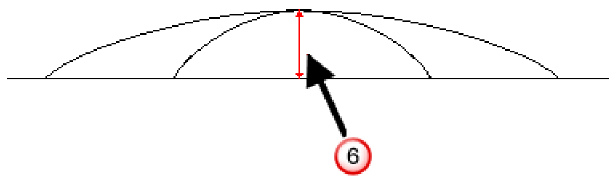
Section Options

These options let you apply different criteria to the section that is used to generate the surface.

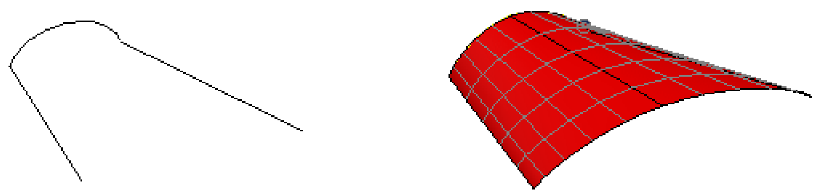
The example below shows that if both options are unchecked, the height of the profile will increase along the length of the drive rails.



Maintain depth - The height of the profile will be maintained for the length of the drive rails. In the image below, 6 shows the depth:



The example below illustrates the surface that will be produced if **Maintain depth** is used.

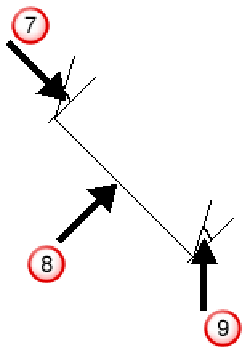


Maintain end tangent direction - The angle of the tangent of the drive rail and the surface is maintained for the length of the drive rails. To use this option there needs to be at least one point on the profile. In the image below:

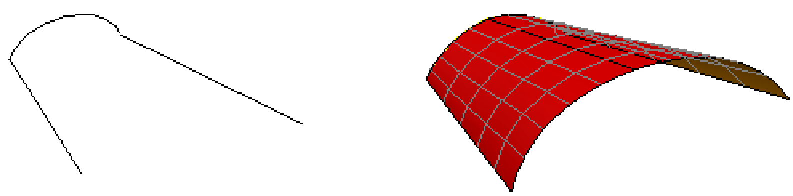
7 Tangent angle at the intersection of the profile and drive curve.

8 Drive curve.

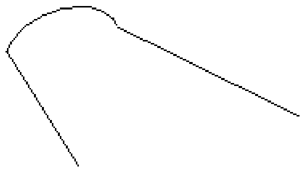
9 Tangent angle at the intersection of the surface and drive curve.

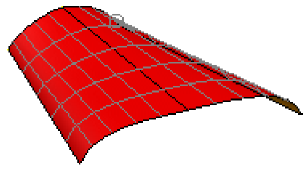


The example below illustrates the surface that will be produced if **Maintain end tangent direction** is used.



You can select **Maintain depth** and **Maintain end tangent direction** together. The example below illustrates the surface that will be produced if both options are chosen.

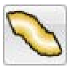




Creating a bead surface

You can create a surface that bulges using any wireframe.

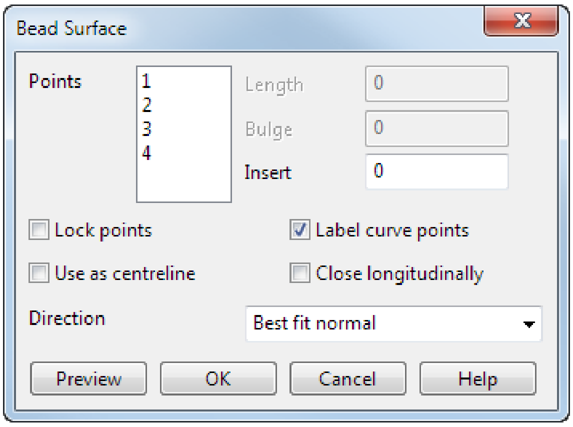
1 Select the wireframe.

2 Click  (*Surface toolbar*).

3 Use the Bead surface dialog (see page 103) to create a bulge.

Bead surface dialog

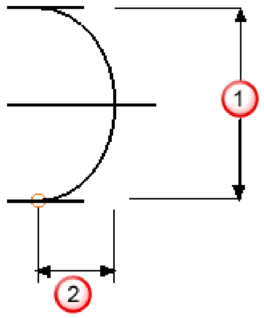
This dialog creates a bead surface from the selected wireframe.



Points — The number of points in this list corresponds to the points on your wireframe. At each point you can specify the chord length of the arc and the bulge distance. You can select more than one point and specify the parameters for the selection.

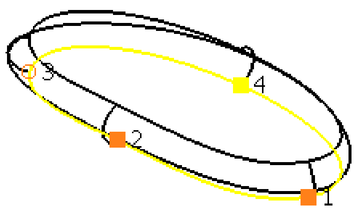
Length — Enter length of the chord for selected point(s).

(2) **Bulge** — Enter bulge distance for selected point(s).

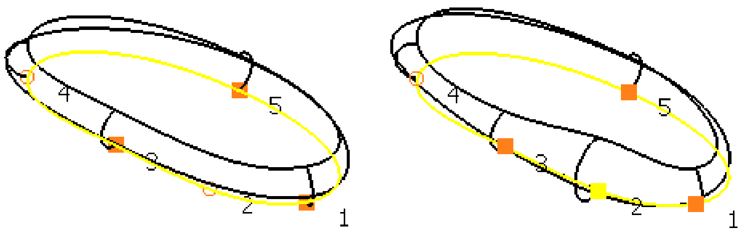


Insert — Insert a point between two existing points. The **Points** list and the labels on the curve are renumbered accordingly. This means that you can specify more precisely the shape of the bead surface between two points.

In the example, there are four points. Values have been entered to define points 1, 2 and 4. The position of point 3 has been obtained by interpolation.



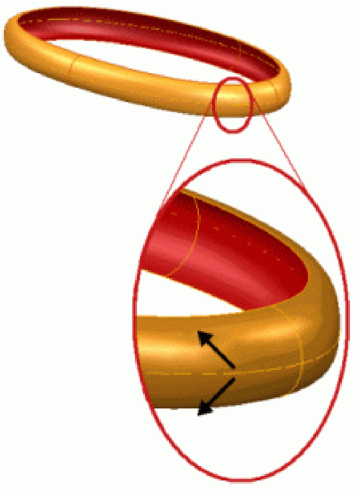
Entering a value of *1.5* into **Insert**, will create another point between point 1 and point 2. The points are then relabelled, so that the inserted point is point 2. You can modify the **Length** and **Bulge** parameters for the new point to define the shape of the surface more precisely.



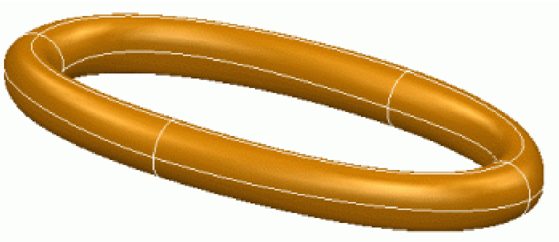
Lock points — This option is *OFF* by default until you enter a **Length** or **Bulge** value. If *OFF* the **Length** and **Bulge** at this point will be obtained by interpolating the values of the locked points. The end points of open curves are locked by default.

Label curve points — If *ON*, the original curve is displayed, with point numbers by each point marker. If *OFF*, point numbers are not displayed. The default setting is *ON*.

Use as centreline — Select this option to create a bead surface either side of the centreline along the curve.



Close longitudinally — Select this option to create a closed bead surface in the longitudinal direction.

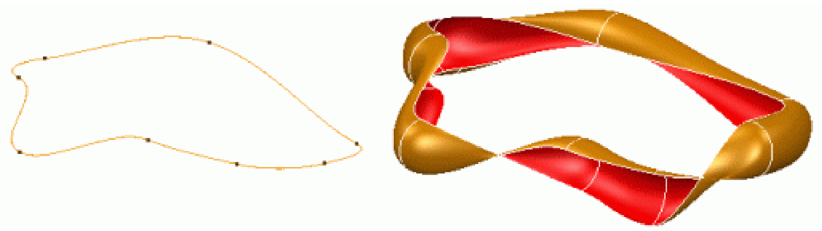


Direction — Select an option from this list to specify the direction of the surface creation.

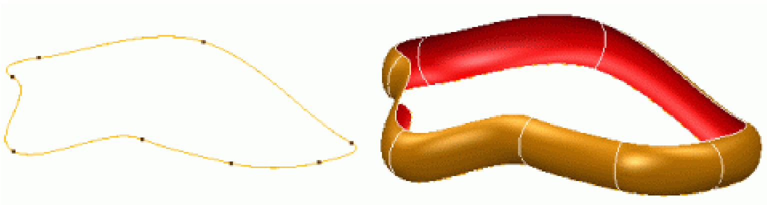
Normal — Align surface to underlying wireframe normals.

Best fit normal — Select this option to create a bead surface from a non—planar curve that is the best fit for the normal of the curve.

The example below shows the bead surface with the option not selected (OFF).



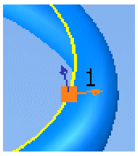
This example shows the difference when the option is selected (ON).



Radially — Set direction radially outwards from wireframe.

Align to axes — Set direction to axis in principal plane.

Preview — This displays the surface using the given parameters. If the resulting surface is not correct, change the parameters and click **Preview** again. The instrumentation at point **1** can be used to reverse the direction of the bulge.

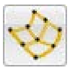


OK — Click this button to create the surface.

Cancel — Click this button to quit the command without creating the surface and remove the dialog from the screen.

Creating a surface from patches

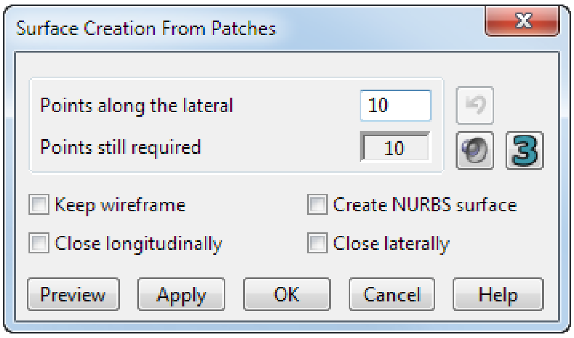
You can generate a surface from patches. The surface will be created from a series of patches and you enter the points at the intersection of the lateral and longitudinal.

1. Click  on the **Surface** toolbar.
2. Complete the Surface Creation From Patches dialog. (see page 106)
3. Create the points on the first lateral.
4. Create the points on the second lateral. The surface will be automatically generated.

   You can create as many laterals as you like, varying the number of points in every lateral if required.

Surface Creation From Patches dialog

This dialog is used to create a surface from points on laterals.



Points along the lateral - Enter the number of points on the lateral.

 — Undo the last entered point.

Points still required - The number of points that still need to be entered to complete the current curve. The number of points still required is reduced each time you enter a point. This option becomes active when you click the ARM button.

 — Use these buttons to turn the sound prompting on and off.
 — Open a window showing number of points that are still required to complete the current lateral.

Keep wireframe - If *ON*, the wireframe is retained when **OK** is selected. If *OFF*, only the surface is displayed.

Create NURBS surface - If ON, this option creates a **NURB** surface instead of a power surface.

Close longitudinally - Closes the curves longitudinally when generating the surface.

Close laterally - Closes the curves laterally when generating the surface.

Preview - Displays a preview of the surface that has been created.

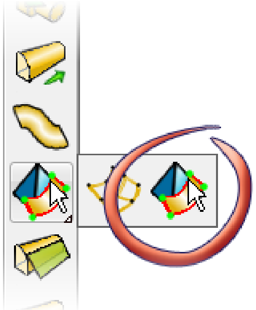
Apply - Click to save and apply the created surface so that another surface can be created without exiting the dialog. Multiple surfaces can be created as required.

OK - Save the surface that has been created and exit **Surface Creation From Patches**.

Creating a surface from a mesh

Use the **Mesh Surfacing** toolbar to generate a surface from a mesh.

1 Click  to display the **Surface** toolbar.



2 Click  to display the **Mesh surfacing** toolbar (see page 109).

3 Use the buttons on the toolbar to select the options to be used when creating the surface from a mesh.

4 Sketch a curve on the mesh using the following mouse controls:

 Add the first and intermediate points to a surface edge. This option is also used to finish a surface contour, when the  is used to indicate the start point of the contour.

Insert discontinuity to a surface edge.

Finish a surface edge.

5 Use  to undo points when creating the curve. Once the surface has been created,  cancels the whole surface creation.

Changing the surfacing algorithm

In PowerSHAPE <NEW>, a new surfacing algorithm was introduced to improve performance and fix occasional problems will inverted tangency. The new algorithm, CUBIC_PLUS, is used by default.

If you get poor tangent matching between the surface and the mesh, you can switch to the older surfacing algorithm. With the **Surface from mesh** toolbar displayed, enter a command **ALGORITHM CUBIC**. The change to the algorithm is maintained for the duration of the session, but it reverts to **CUBIC_PLUS** when you restart PowerSHAPE.

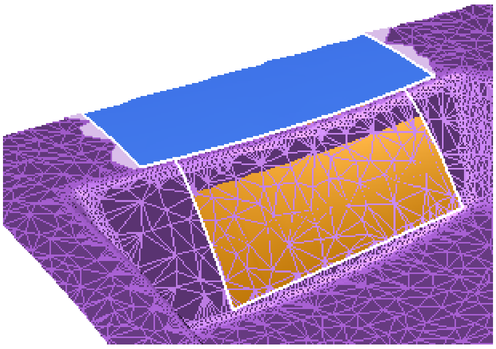
Use the command **ALGORITHM CUBIC_PLUS** to revert to the new alogoritm within a PowerSHAPE session.

Use the command **ALGORITHM SHOW** to see which surfacing algorithm is in use.

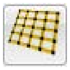
Mesh surfacing toolbar

Use the buttons on the toolbar to select the options to be used when creating the surface from a mesh:

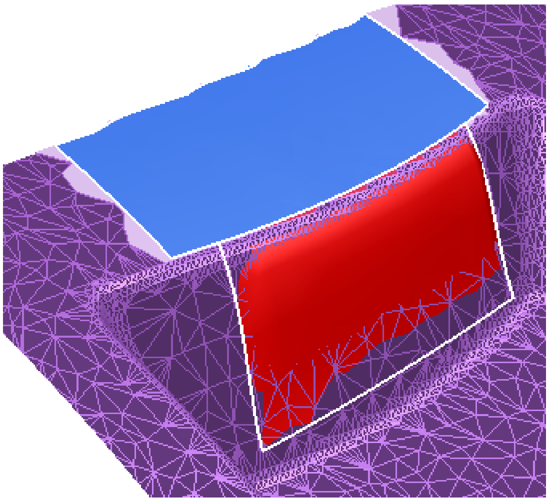
 Click to produce a trimmed (NURB) or untrimmed surface.



Enter the contour sketching tolerance to define the quality of fit between the contour and the mesh.

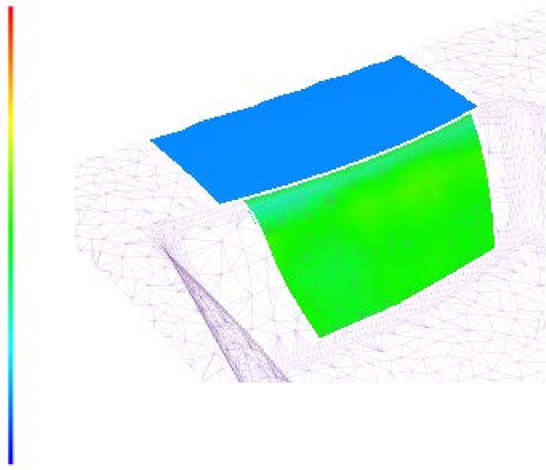
 Click to create a surface with a pre-defined number of laterals and longitudinals (see page 111).

Click to create a surface tangent to existing surfaces (deselected by default). The surface will match the existing surface, without reference to the underlying triangles. This option can be applied separately to each contour segment.



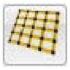
Click to turn a point of a mesh into a corner. Use  to create a point when picking the point.

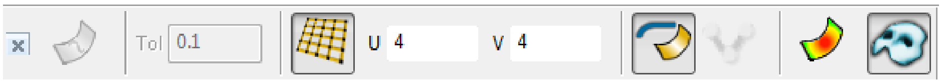
Click to view the deviation of the created surface from the underlying mesh. Create the surface by picking contour points on the mesh. The following shows the deviations on a surface created tangent to the underlying mesh:



Click to toggle whether the triangles covered by the new surface are shown or hidden.

Pre-defined number of laterals and longitudinals

1 Click  to create a surface with a pre-defined number of laterals and longitudinals. The **U** and **V** options are activated.



2 Enter the number of laterals (**U**).

3 Enter the number of longitudinals (**V**).

The surface will be automatically generated with the required number of laterals and longitudinals.

*Enter a value of **0** for **U** and **V** to choose to insert zero laterals and longitudinals.*

Editing Power surfaces

A Power Surface is a surface created from wireframe items. Editing curves and points within a surface uses the same methods as editing wireframe curves.

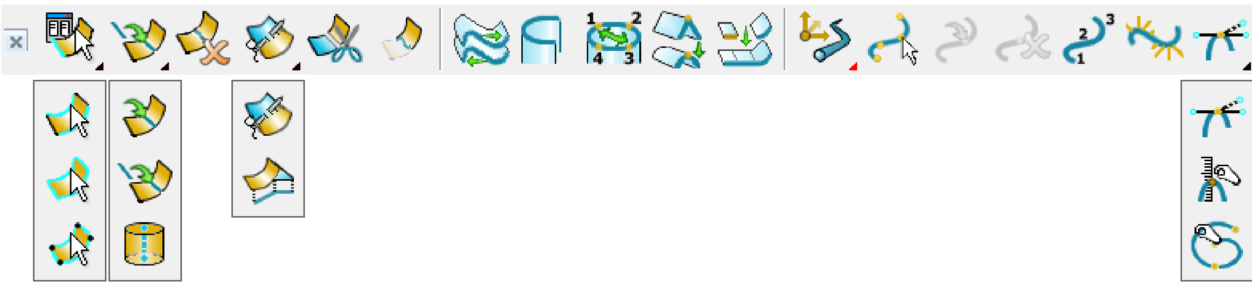
You can edit a Power Surface in the following ways:

- Double-click the surface to display the **Surface/Curve Edit** (see page 112) toolbar.
- Use the options available from the **Edit** menu (see page 150).
- Use the graphical surface editing features (see page 163).
- Right-click the surface to open the **Power Surface** context menu (see page 178). The top of the context menu displays the type of the surface, its name, and the level on which it lies.

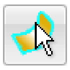
You can also find details on fixing surfaces (see page 179) and inspecting surfaces (see page 182) in this section.

Editing a Power Surface using the toolbar

When you create or select a Power Surface, the **Surface/Curve Edit** toolbar is displayed, and the options that are appropriate for your surface are made available.



Use the following table to identify the buttons on the toolbar:

	Name	Edit menu	Function
	**Select Curves**	**Surface and Curve Edits > Surface Edits > Select Curves/Points > Select Curves...**	Select curves on a surface (see page 114)
	**Select Curves in Active Direction**	**Surface and Curve Edits > Surface Edits > Select Curves/Points > Select Curves in Active Direction**	Select all curves in current direction (see page 115)
	**Select All Curves**	**Surface and Curve Edits > Surface Edits > Select Curves/Points > Select All Curves**	Add all curves to selection (see page 116)

	Select Points	**Surface and Curve Edits > Surface Edits > Select Curves/Points > Select Points...**	Select pattern of points (see page 116)
	Add Curve	**Surface and Curve Edits > Surface Edits > Add Curves > Add Curve...**	Add curve (see page 118)
	Add Curve from Wirefram e	**Surface and Curve Edits > Surface Edits > Add Curves > Add Curve from Wireframe**	Insert curve from wireframe (see page 124)
	Add Spine	**Surface and Curve Edits > Surface Edits > Add Curves > Add Spine**	Create spine (see page 126)
	Delete Curve	**Surface and Curve Edits > Surface Edits > Delete Curve**	Delete the curve (see page 128)
	Stitch	**Surface and Curve Edits > Surface Edits > Edit Curves > Stitch**	Stitch the surface (see page 129)
	Match Lateral	**Surface and Curve Edits > Surface Edits > Edit Curves > Match Lateral**	Match lateral to curve projection (see page 137)
	Break Surface	**Surface and Curve Edits > Surface Edits > Break Surface**	Break surface (see page 140)
	Trimmin g	**Surface and Curve Edits > Surface edits / Show Trimmed Surface**	Trimming — click to toggle *ON*/*OFF* (see page 142)
	Reverse	**Surface and Curve Edits > Common Edits > Reverse**	Reverse the surface (see page 143)
	Close/Op en	**Surface and Curve Edits > Common Edits > Close**	Open (see page 144)/close (see page 145) surface — click to toggle *ON*/*OFF*
	Renumb er	**Surface and Curve Edits > Common Edits > Renumber**	Renumber surface points (see page 146)
	Spline	**Surface and Curve Edits > Common edits > Spline**	Spline surface (see page 146)
	Append	**Surface and Curve Edits > Common edits > Append**	Join two surfaces (see page 148)
	Select Points	**Surface and Curve Edits > Curve Edits > Select Points...**	Select points on a curve

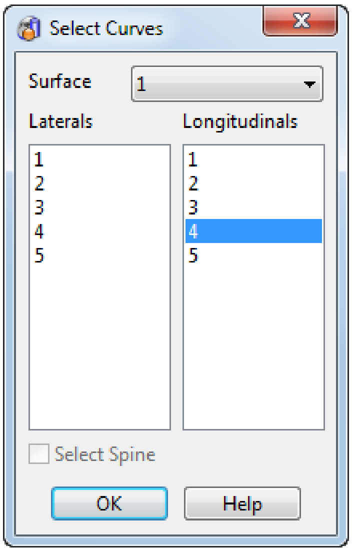
	Show Labels	**Surface and Curve Edits > Curve Edits > Show Labels**	Turn point labels *ON/OFF*
	Show Curvature Combs	**Surface and Curve Edits > Curve Edits > Show Curvature Combs**	Turn curvature combs *ON/OFF*
	Edit Tangent	**Surface and Curve Edits > Curve Edits > Curve Point Adjustment > Edit Tangent Angles...**	Edit tangent angles
	Fine Tune	**Surface and Curve Edits > Curve Edits > Curve Point Adjustment > Fine Tune Points**	Fine tune points
	G2 edit	**Surface and Curve Edits > Curve Edits > Curve Point Adjustment > G2 Edit Fairing**	G2 edit
	Repoint Curve	**Surface and Curve Edits > Curve Edits > Curve Point Reordering > Repoint Curve**	Repoint curve
	Edit as Bézier	**Surface and Curve Edits > Curve Edits > Curve Type > Edit as Bezier Curve**	Edit as a bezier curve
	Edit as G2	**Surface and Curve Edits > Curve Edits > Curve Type > Edit as G2 Curve**	Edit as a G2 curve
	Edit as B-spline	**Surface and Curve Edits > Curve Edits > Curve Type > Edit as Bspline Curve**	Edit as a B-spline curve
	Smoothing	**Surface and Curve Edits > Curve Edits > Apply Smoothing to Point Edits**	Apply smoothing to point edits — click to toggle *ON/OFF*

Selecting curves on surfaces

1. Select all the surfaces from which you want to select curves.
2. Click the **Select Curves** button on the Surface Edit toolbar.
3. Use the **Select Curves** dialog (see page 115) to select the curves on the surfaces.

Select Curves dialog

Use this dialog to select curves on surfaces.



Surface - the name of the surface whose curves are displayed in the **Laterals** and **Longitudinals** lists. You can select another surface and its curves are displayed in the two lists of curves.

Whilst the dialog is displayed, you can change the surfaces selected on the screen. This also automatically updates the surfaces in the **Surface** selector.

Laterals and **Longitudinals** - these lists are the curves on the surface named in the **Surface** selector. You can select curves from the two lists by clicking to select single curves, **Shift-click** to add a curve to the selection and **Ctrl-click** to toggle a curve from the selection. As you select the curves, they are highlighted on the screen.

When you choose another surface from the selector, PowerSHAPE remembers the selected curves of the previously selected surfaces.

Select Spine - selects the spine on the surface. This option is only available when a surface with a spine is selected.

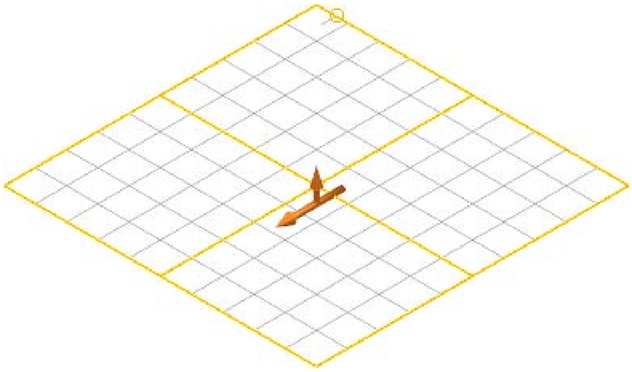
OK - closes the dialog. You can keep the dialog displayed while you edit the surface curves.

Selecting all the curves in the current direction

You can select all the curves in the current direction on a surface.

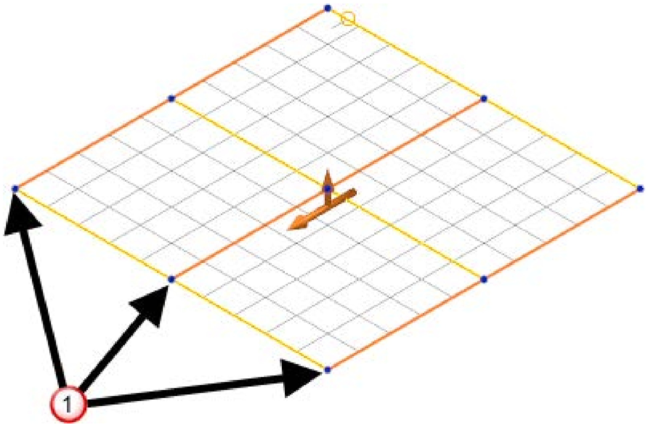
1 Use the **Direction** arrow on the surface to select the curve direction.

In the following example, the lateral direction is selected.



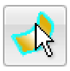
2 Click **Select all curves in the current direction**  (*Surface Edit toolbar*).

All the laterals on the surface are now selected 1.



Selecting all the curves on a surface

You can select all the curves on a surface.

1 Click **Add all curves to selection**  (*Surface Edit toolbar*).

All the curves on the surface are now selected.

Selecting a pattern of points

You can select points on a surface using simple pattern sequences.

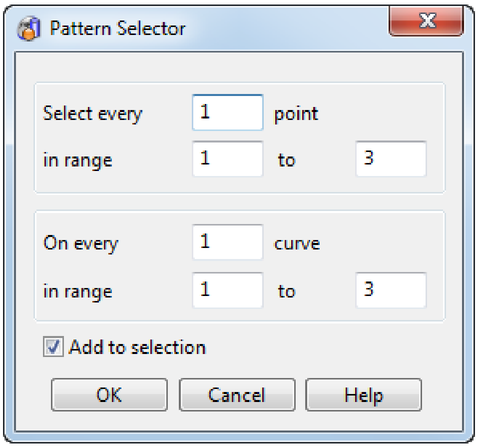
1 Select the direction arrow on the surface to select the direction for the pattern of points.

2 Click  (*Surface Edit toolbar*).

3 Use the Pattern Selector dialog (see page 287) to select points.

Pattern Selector dialog

Use this dialog to select surface points using simple pattern sequences.



Select every *n* **point** - selects every *nth* point starting at the first point in the range, where *n* is the number you input. If you enter **3**, then the first point in the range is selected followed by every 3rd point.

in range *a* **to** *b* - the range of the points to select from.

On every *m* **curve** - selects points from every *mth* curve starting at the first curve in the range, where *m* is the number you input. If you enter **3**, then points are selected on the first curve in the range followed by every 3rd curve.

in range *x* **to** *y* - the range of the curves from which the points are selected.

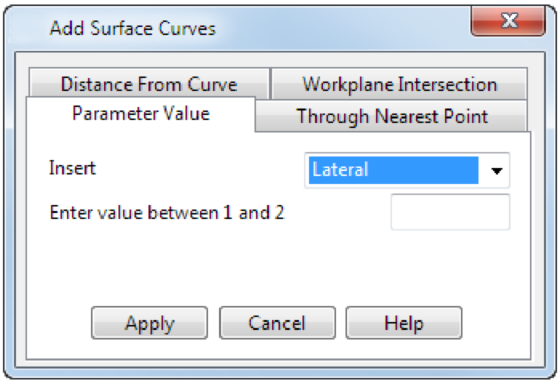
Add to selection - if selected, you can add the selection to the existing selection of points. Deselect this option to make a fresh seection.

OK - accepts the changes and closes the dialog.

Cancel - discards the changes and closes the dialog.

Creating a curve on a surface

Use the **Add surface curves** dialog to add laterals and longitudinals to the surface.

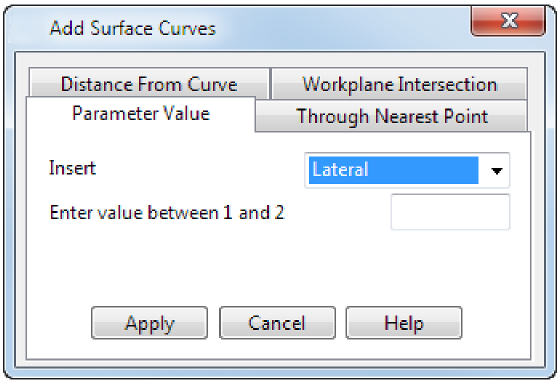


The dialog contains four different methods for adding surface curves. Select a tab to display the relevant page.

- Parameter value (see page 118)
- Through nearest point (see page 120)
- Distance from curve (see page 121)
- Workplane intersection (see page 122)

Add surface curves - Parameter Value

Use this page of the **Add surface curves** dialog to create a curve at a specified parameter value.



Insert — select the surface direction for the new curve from the following:

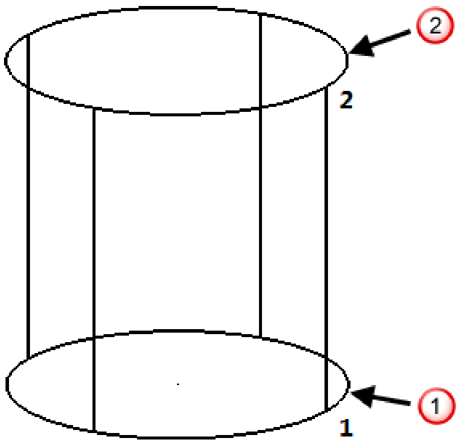
- Lateral
- Longitudinal

Enter value between 1 and 2 — enter the parametric value of where you want to position the new curve.

Apply — inserts the new curve. This also renumbers the other curves to reflect the change.

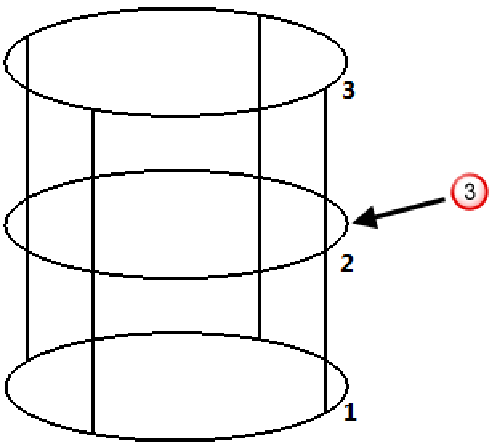
Cancel — Saves the surface and closes the dialog.

To add a new lateral half way between laterals 1 1 and 2 2:



1 Click **Add Curve**  (*Surface Edit toolbar*)
2 Select the **Parameter Value** tab.
3 Select lateral from the **Insert** drop-down list.
4 Enter a **Value** of 1.5. This is the parametric value half way between laterals 1 and 2.

   A new lateral 3 is added half way between laterals 1 and 2. This lateral is numbered 2 and lateral 2 has become lateral 3. The new lateral becomes the current lateral.

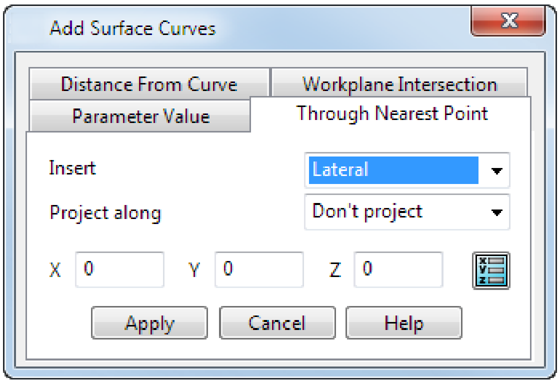


   If you now want to add another lateral equidistant between 2 and 3, you need to enter 2.5. To put one between 1 and 2 you would enter 1.5.

Each time you add a lateral or longitudinal curve to a surface, the parametric value changes for a particular position on the surface.

Add surface curves - Through nearest point

Use this page of the Add surface curves dialog to add a curve at a point on the surface nearest a specified point. You may choose an axis along which the specified point is projected on to the surface.



Insert — select the surface direction for the new curve from the following:

- Lateral
- Longitudinal

Project along — choose the axis of projection from the following:

- Don't project
- X
- Y
- Z

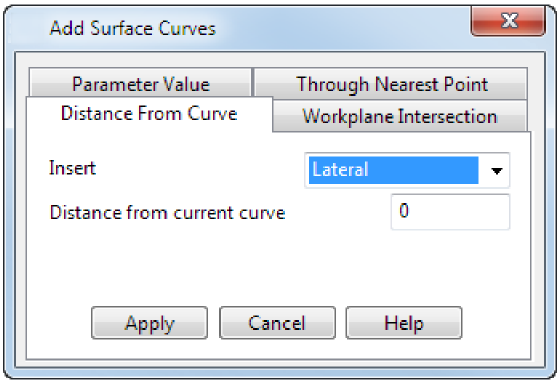
Enter the **X Y Z** coordinates or click the **Position**  button to open the **Position** dialog where you can use position entry tools.

Apply — inserts the new curve. This also renumbers the other curves to reflect the change.

Cancel — saves the surface and removes the dialog from the screen.

Add surface curves - Distance from curve

Use this page of the Add surface curves dialog to add a curve at a specified distance from the current curve. If there is no curve selected, then curve 1 is used by default.



Insert — select the surface direction for the new curve from the following:

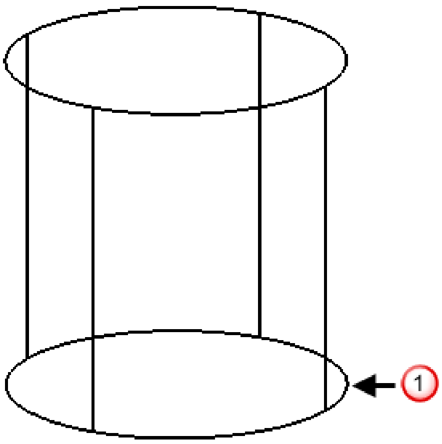
- Lateral
- Longitudinal

Distance from current curve — enter the distance from the current curve where you want to position the new curve.

Apply — inserts the new curve. This also renumbers the other curves to reflect the change.

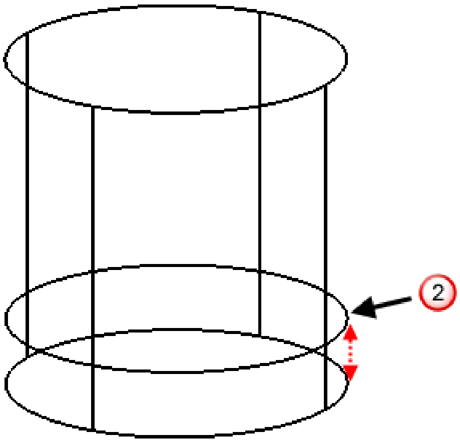
Cancel — saves the surface and closes the dialog.

To add a new lateral 10 units from lateral 1 1:



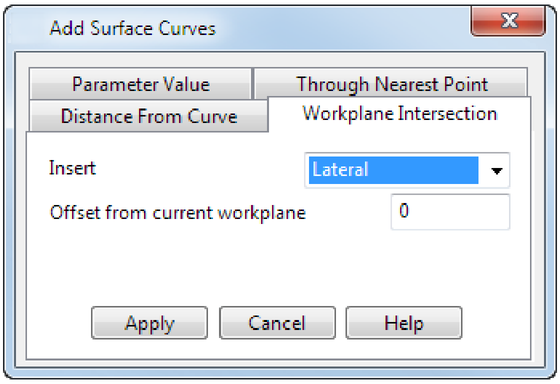
1 Click **Add Curve**  (*Surface Edit toolbar*)
2 Select the **Distance from curve** tab.
3 Select lateral from the **Insert** drop-down list.
4 Enter a **Distance** of **10**.

A new lateral ② is added to the surface. This lateral is numbered 2 and lateral 2 has become lateral 3. The new lateral becomes the current lateral.



Add surface curves - Workplane intersection

Use this page of the Add surface curves dialog to add a curve where the principal plane of an active workplane intersects a surface.



Insert — select the surface direction for the new curve from the following:

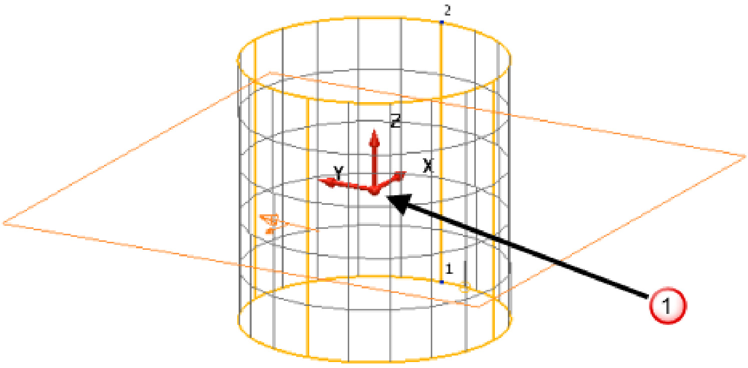
- Lateral
- Longitudinal

Offset from curent workplane — enter a value from the workplane to postion the new curve. The plane graphic updates to show the new value.

Apply — inserts the new curve. This also renumbers the other curves to reflect the change.

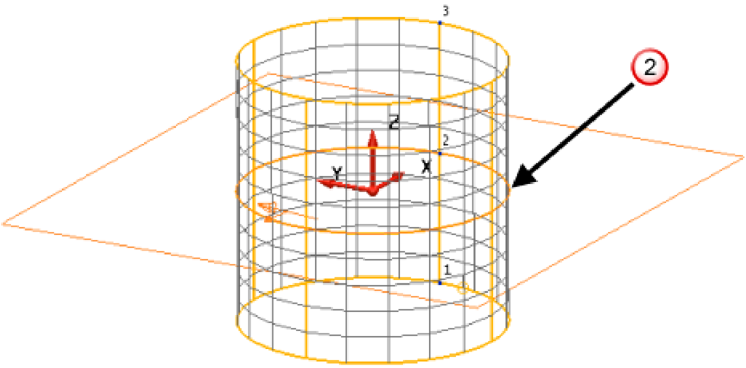
Cancel — saves the surface and closes the dialog.

To add new lateral at the postion of the active workplane 1:

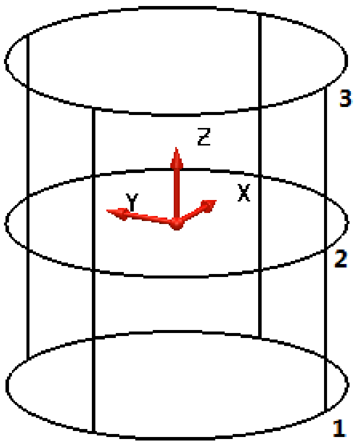


1 Click **Add Curve** (*Surface Edit toolbar*).
2 Select the **Workplane intersection** tab.
3 Select lateral from the **Insert** drop-down list.
4 Enter an **Offset** of **0**.

A new lateral 2 is added where the XY plane of the active workplane intersects the surface.

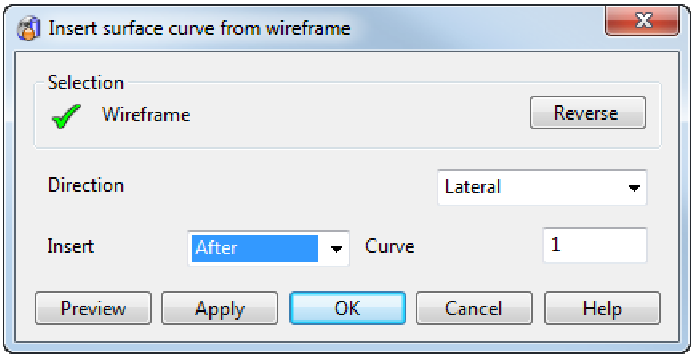


The new lateral is numbered 2 and lateral 2 has become lateral 3.

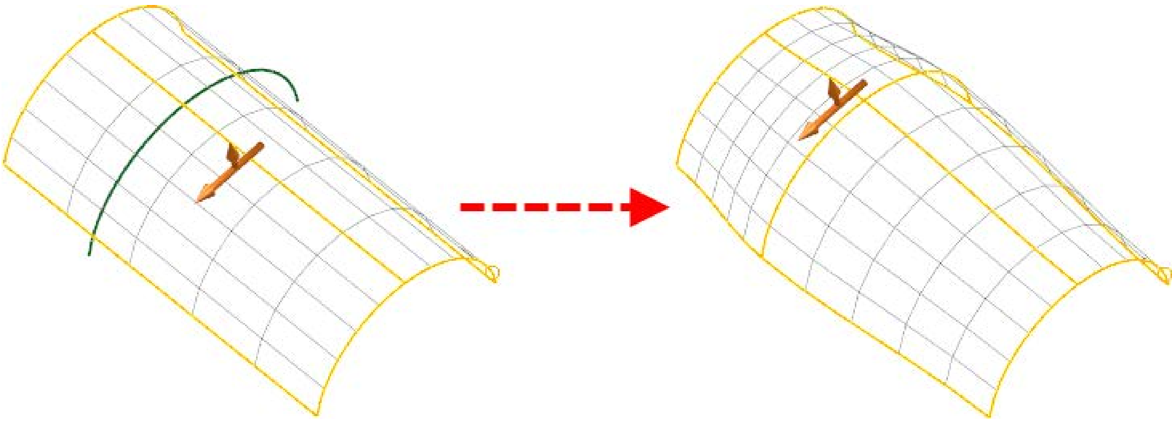


Inserting a surface curve from wireframe

Use the **Insert surface curve from wireframe** (see page 124) dialog to create laterals or longitudinals on a surface by inserting wireframe curves.

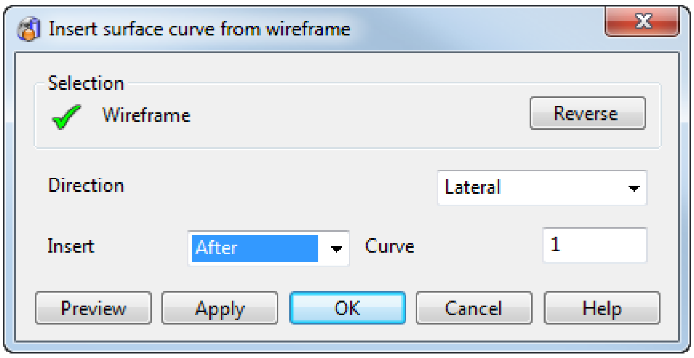


The wireframe curve must have the same number of points and be in the same open/closed state as the laterals/longitudinals on the surface. The surface cannot contain a spine.



Insert surface curve from wireframe dialog

Use this dialog to insert curves from wireframe into the surface.



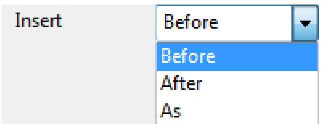
Selection

Wireframe - when you select a valid wireframe curve, the The cross icon changes to a tick icon..

Reverse - reverses the direction wireframe curve.

Direction - select whether to insert the wireframe as a lateral or longitudinal.

Insert - indicates where to position the wireframe on the surface. You can insert the wireframe **Before**, **After** or **As** the curve given in **Curve**. The **As** option replaces the existing curve with the wireframe.



Curve - enter the number of the curve used by **Insert**.

Preview - displays the new curve in the surface from the selected wireframe object. You may change your wireframe object or settings on the dialog until you are satisfied with the previewed surface.

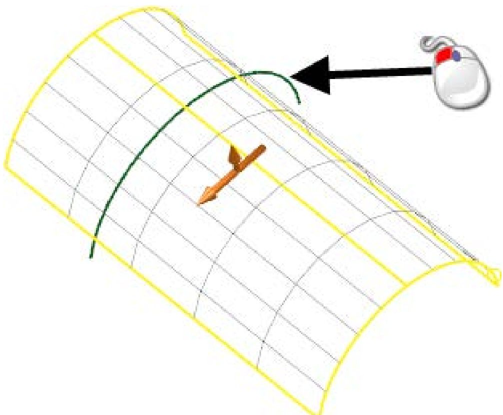
Apply - saves the surface. The dialog remains open for you to insert more wireframe objects into the surface.

OK - saves the surface and closes the dialog.

Cancel - closes the dialog and dismisses any changes that have not been applied.

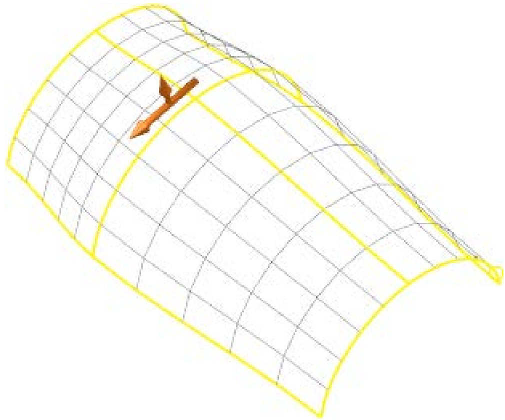
To insert a new curve in a surface:

1. Click **Insert curve from wireframe** (*Surface Edit toolbar*).
2. Select the wireframe.



3. Select **Lateral** from the **Direction** drop-down list.
4. Select **After** from the **Insert** drop-down list.

5 Click **Preview** to view the new curve in the surface.



Why doesn't my surface look as I expected?

If your surface looks different from the one you expected, it could be that it changed when you inserted a lateral. If this is the case, the following may help you understand why.

In a **Power surface**, the interior of a patch is mathematically more complex (6th order) than the laterals and longitudinals. The laterals and longitudinals are composite cubic curves (3rd order). In general cases, it is impossible to represent a general parameter curve across the surface with these cubic curves. They can only be an approximation to the original surface.

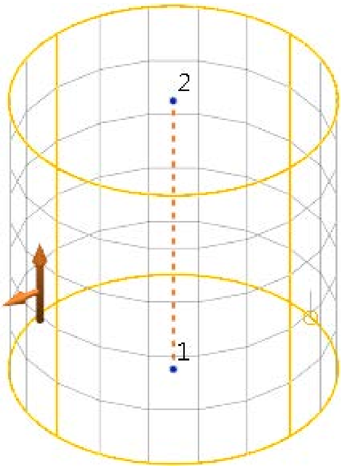
The tolerance is not relevant in this case. Any freedom is used to try to match positions at midpoints in the span. In other parts of the curve it may be well out of tolerance.

Adding a spine

A spine is a reference line on which laterals hang. You can edit the spine to change the shape of the surface.

1 Click  (*Surface Edit toolbar*).

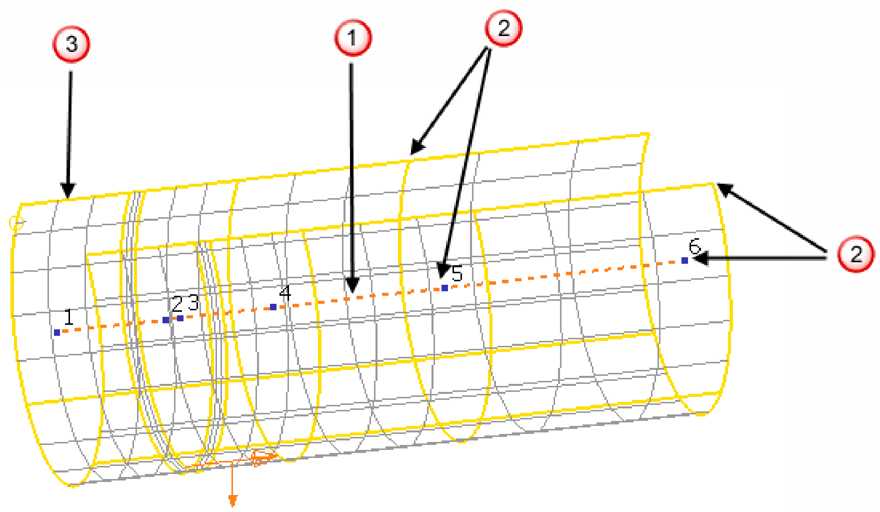
A spine is added to the surface in the current direction. This is not always a longitudinal direction. If the current direction is lateral, the laterals and longitudinals will be swapped. This also results in the surface being reversed.



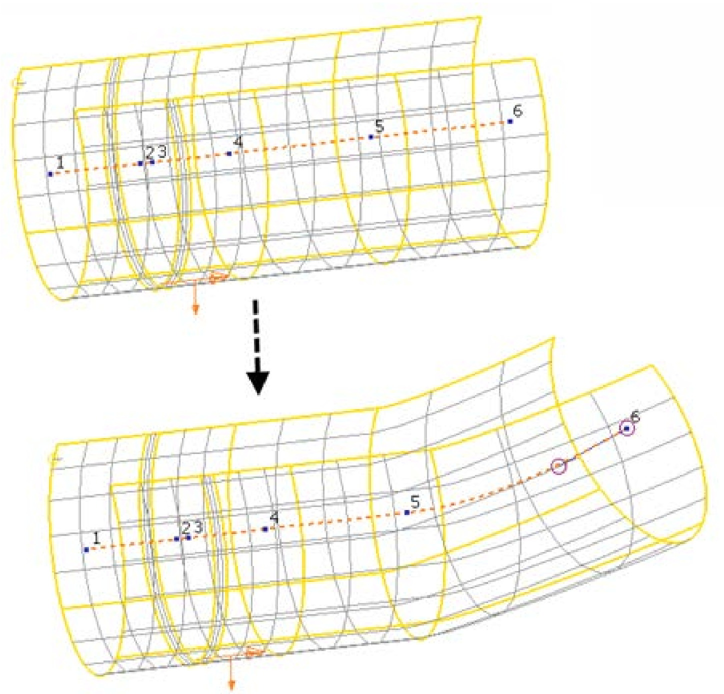
What is a spine?

The spine is a reference curve that may be used to define the overall shape of a surface. It was widely used in DUCT, but PowerSHAPE users need not normally be concerned with it.

The spine 1 is the reference line that determines where the data sections (laterals) are positioned in space. The spine is defined in terms of points. Every spine point must have an associated section 2 and every section must have the same number of points. The section is orientated to lie in a plane at right angles to the tangent to the spine. Corresponding points on successive sections are linked by longitudinal lines 3 to make up a network of curves. If you add a spine point, a new section is automatically created for it.



Similarly, when you delete a spine point, the associated section is deleted too. More usefully, if the data at one or more spine points is modified, the orientation of the section relative to the spine is preserved.

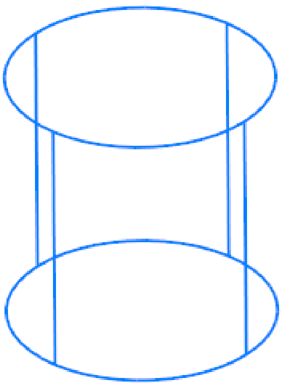


While the spine is naturally the centre-line, it may be any convenient line inside, on or outside the surface. The shape of the surface is easier to control if the spine is straight or two-dimensional, but this is not necessary. The sections and the spine may be open or closed, so that the surface may form an open region, a tube or a ring. The spine data should be chosen carefully to avoid unnecessary inflexions or concentrations of curvature.

In PowerSHAPE a spine is generated automatically for the centre-line of a fillet surface, but otherwise is rarely used.

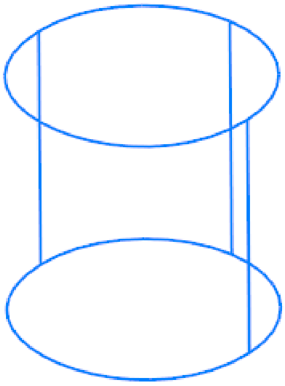
Deleting a curve on a surface

To delete one of the longitudinals on the following surface:



1 Select the longitudinal you want to delete.

2 Click  (*Surface Edit toolbar*) to remove the curve.



A surface must have at least 2 laterals and 2 longitudinals. You cannot delete any curves that would reduce the number of curves in that direction to be less than 2.

Stitching surfaces

Stitching is a method of closing small gaps between surfaces and curves. It moves the points on a selected surface curve onto another surface or curve.

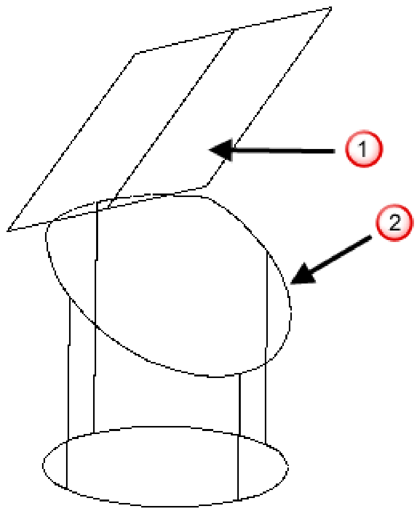
You stitch the surface curve onto surfaces and curves known as **targets** using the Stitch dialog. (see page 131) Targets for stitching are one of the following types:

- surfaces
- surface curves
- wireframe curves (including composite curves)

You cannot use targets that contain a mixture of surfaces and curves, as each uses different stitch options.

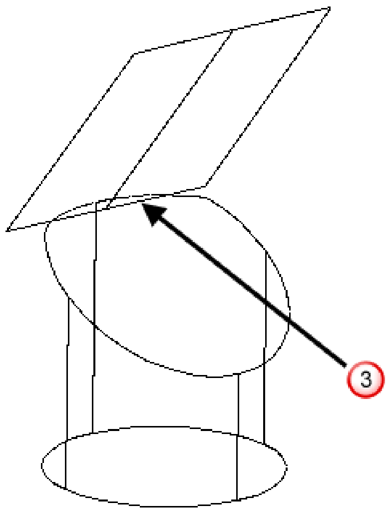
When stitching onto a surface, you can edit the tangent angles along the curve of the surface you are stitching to smoothly match those of the surface you are stitching onto.

Suppose we have the following surfaces. We want to stitch the lower curve on the planar surface 1 onto the cylindrical (target) surface 2.



1 Select the curve on the surface to be stitched 3.

We selected the surface curve shown below:

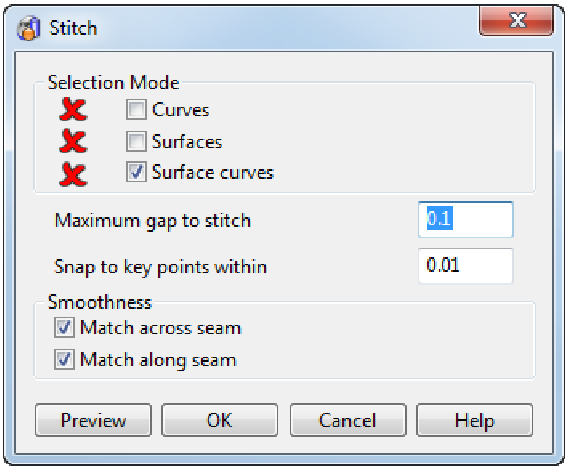


If you only want certain points on the curve to be stitched, select those points.

If no points are selected, then all points on the curve are stitched.

1 Click **Model Fixing** on the **General edits** flyout.

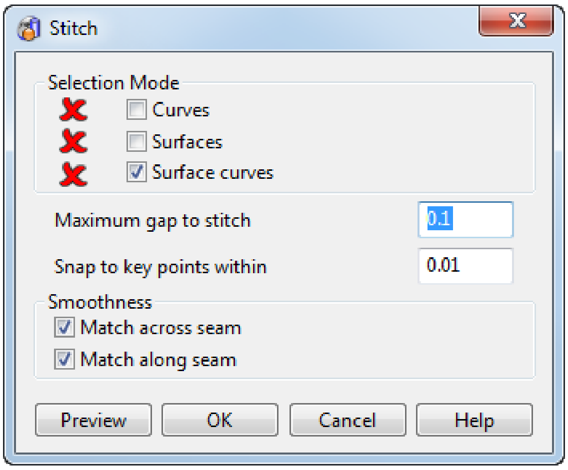
2 Click  (*Model fixing toolbar*) to display the **Stitch** dialog.



3 Change the values and options to achieve the required stitched surface.

Stitch dialog

This dialog is used to define the stitching.



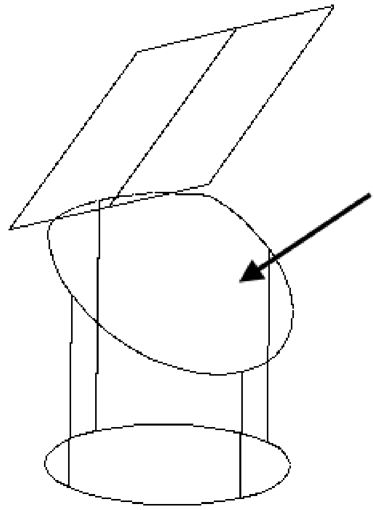
Selection Mode - This allows you to choose the type of the target from one of the following:

- wireframe curves (including composite curves)
- surfaces
- surface curves

Toggle the option of the target type you want to choose, then select the target. The cross changes to a tick if you select the correct target.

A mixture of surfaces and curves is not allowed.

In this example, the **Surfaces** option is toggled and the cylindrical surface is selected.



Each stitch point is moved onto the nearest position on the target surface or curve. The points on the curve are only stitched if they are within the distance specified in **Max gap to stitch** of the target surface.

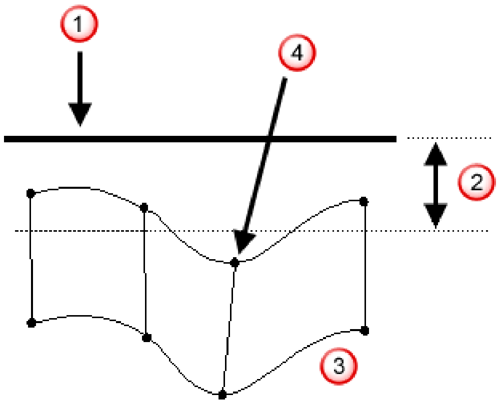
Maximum gap to stitch - This value is the size of the largest gap to stitch. If the stitch point has to move a distance greater than this value, then this point is not moved onto the target.

1 Target surface.

2 Maximum gap to stitch.

3 Stitch surface.

4 Stitch point does not move because its projected distance is greater than the **Maximum gap to stitch**.



Snap to key points within - This value determines whether a stitch point snaps onto a key point of the target.

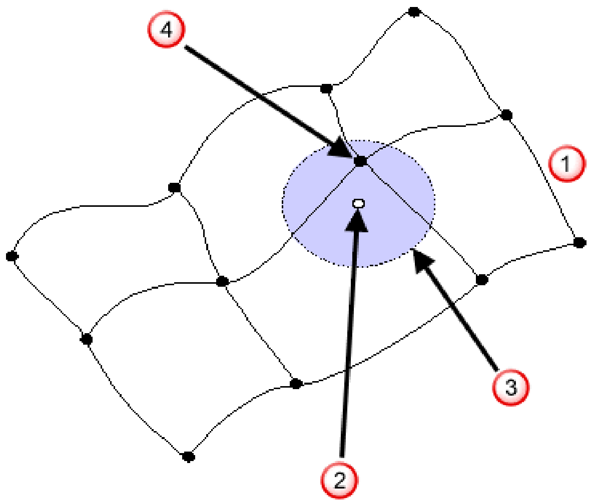
Once a stitch point moves onto a target, PowerSHAPE looks for any key points on the target within the radius defined by **Snap to key points within** of the stitch point. If a key point is found, the stitch point snaps onto it.

1 Target surface.

2 Stitch point.

3 Area around stitch point.

4 Stitch point snaps on this key point on the surface.



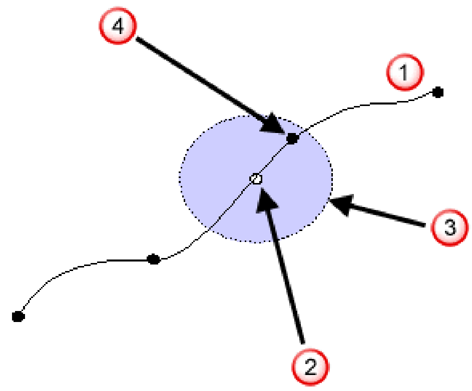
If the target is a curve, nothing happens. (In future releases of PowerSHAPE, the stitch point will snap to a key point within the given radius.)

1 Target curve.

2 Stitch point.

3 Area around stitch point.

4 Stitch point does not snap to this point on the curve.



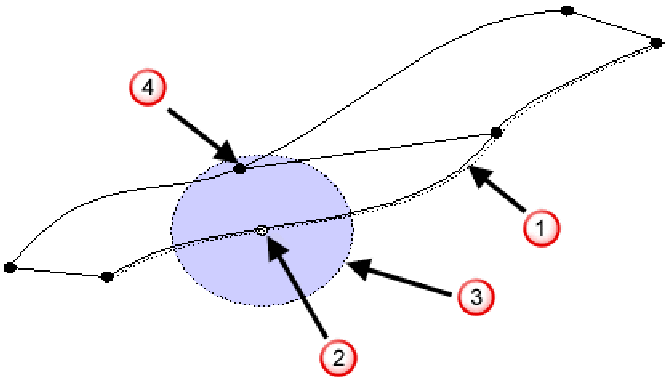
If the target is a surface curve, the stitch point may snap to a key point that does not necessarily lie on the target surface curve.

1 Target surface curve.

2 Stitch point.

3 Area around stitch point.

4 Stitch point snaps to this key point even though the point does not lie on the target surface curve.

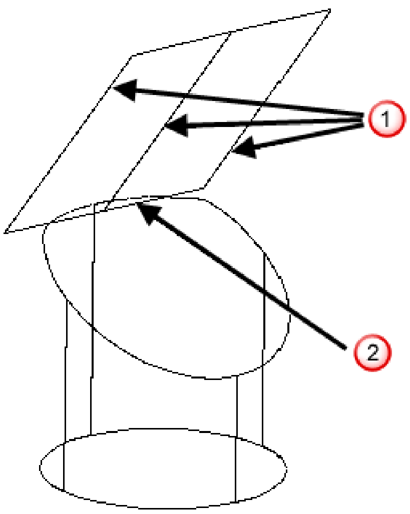


Smoothness

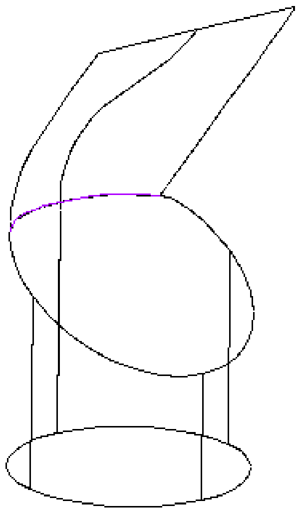
Smoothness - When the target is a surface or a curve on a surface, you can join the stitch surface smoothly onto the target surface using **Match across seam** and **Match along seam**. The seam is where the selected curve of stitch surface meets the target.

Match along seam smoothes the selected curve onto the target. **Match across seam** smoothes the curves in the opposite direction to the selected curve onto the target.

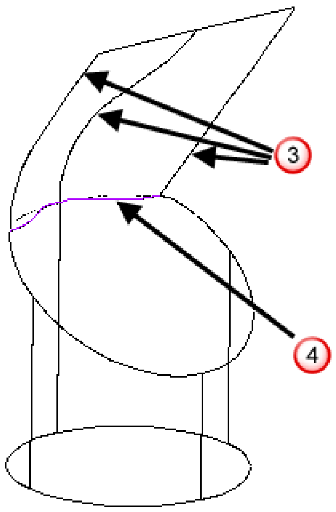
In this example, the curves across 1 and along 2 the stitch surface are shown below.



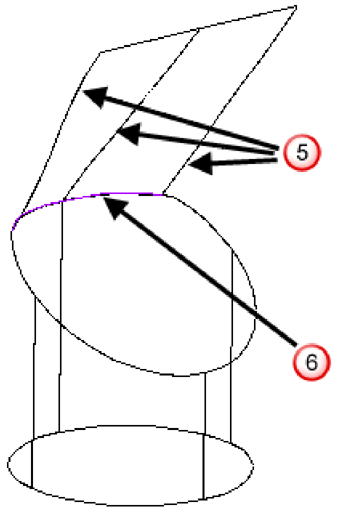
If **Match across seam** and **Match along seam** are both on, the following result is given when the surfaces are stitched together.



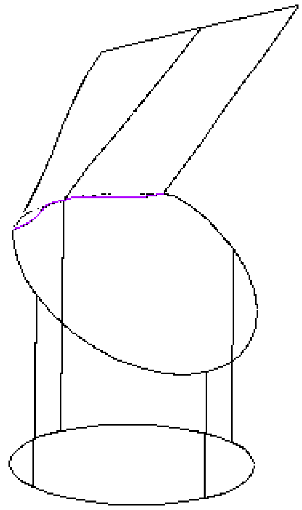
If **Match along seam** is OFF and **Match across seam** is ON, the curves across the surface join the cylindical surface smoothly 3, but the curve along the surface does not join smoothly 4.



If **Match along seam** is ON and **Match across seam** is OFF, the curves across the surface do not join the cylindrical surface smoothly 5, but the curve along the surface does join smoothly 6.



If both **Match along seam** and **Match across seam** are OFF, none of the curves on the planar surface join smoothly onto the cylindrical surface:



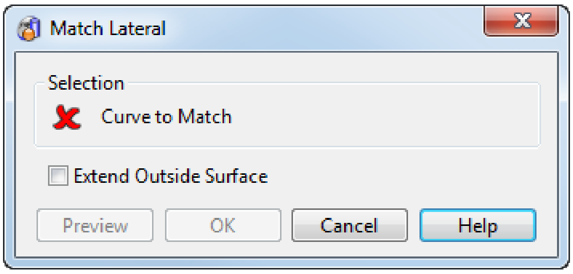
Preview - Displays the stitched surface as defined by the settings on the dialog. Continue to change the settings until you are satisfied with the previewed stitched surface.

OK - Saves the surface and removes the dialog from the screen.

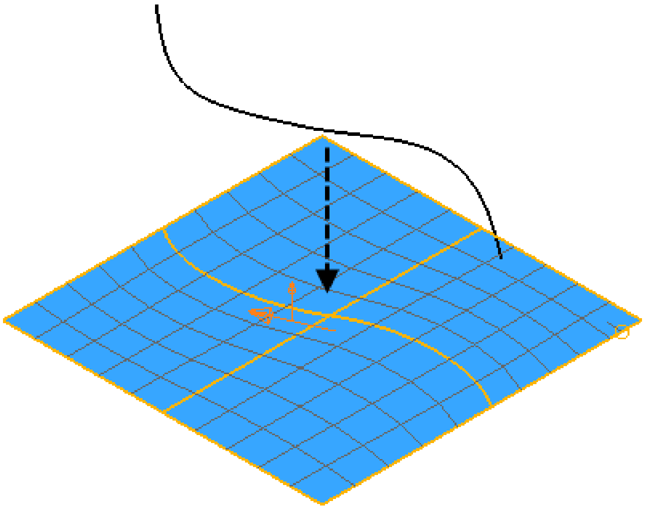
Cancel - This removes the dialog from the screen without making any changes.

Matching a lateral to a curve

Use the **Match Lateral** (see page 137) dialog to match the shape of a lateral or longitudinal to a projected wireframe curve.

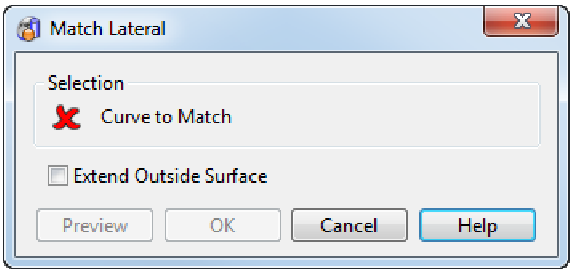


The curve is projected along the axis normal to the principal plane.

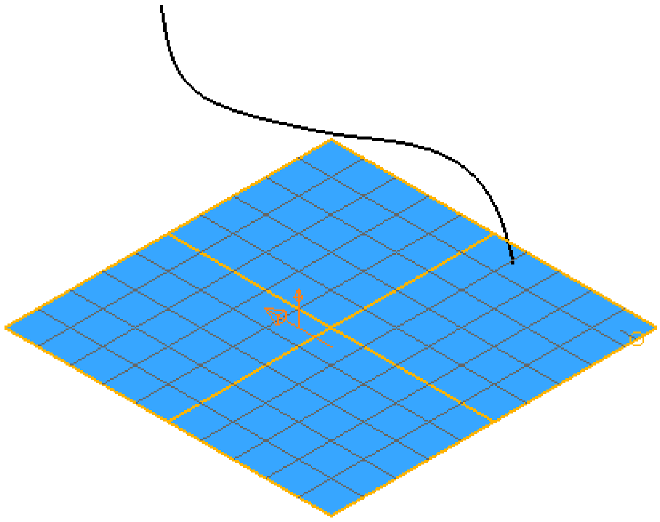


Match Lateral dialog

Use this dialog to match a lateral to the shape of a projected curve.

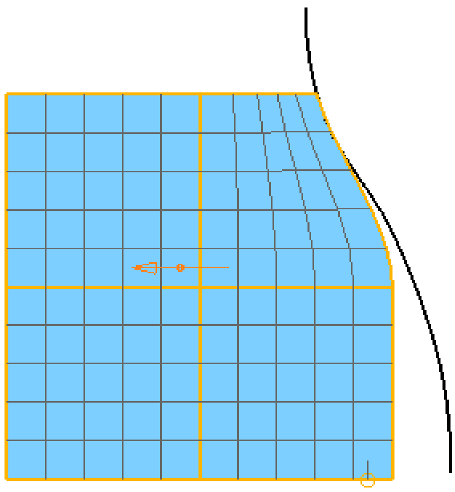


Curve to match - when a valid wireframe curve is selected, The cross icon changes to a tick icon..

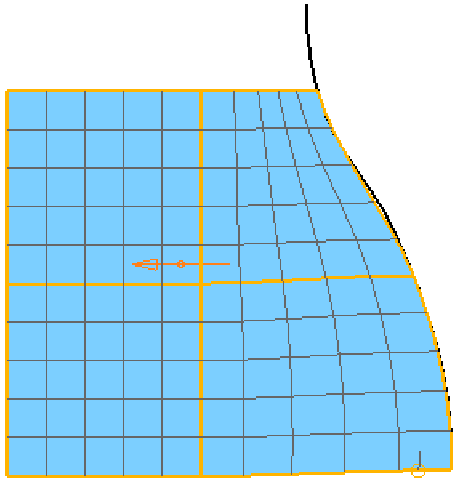


Extend Outside Surface - when the projection of the curve lies outside the surface:

- if the **Extend Outside Surface** option is deselected, only part of the lateral matches the curve.



- if the **Extend Outside Surface** option is selected, the points on the lateral are extended to match the nearest points on the curve.



Preview - displays the new shape of the lateral in the surface. The projected curve is along the axis normal to the principal plane.

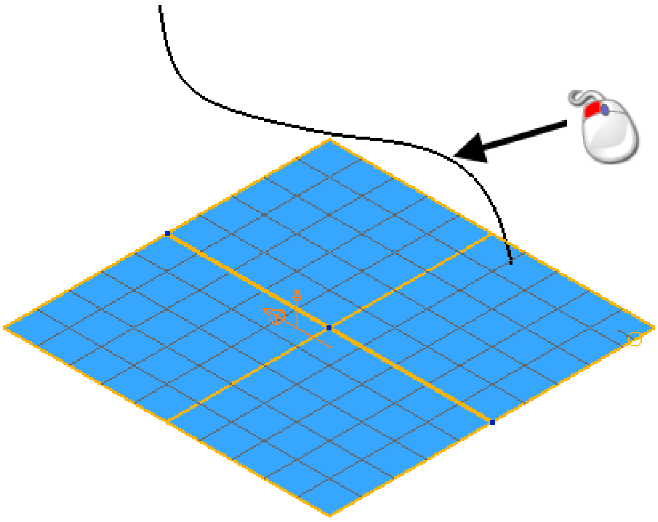
OK - saves the surface and closes the dialog.

Cancel - closes the dialog and dismisses any changes.

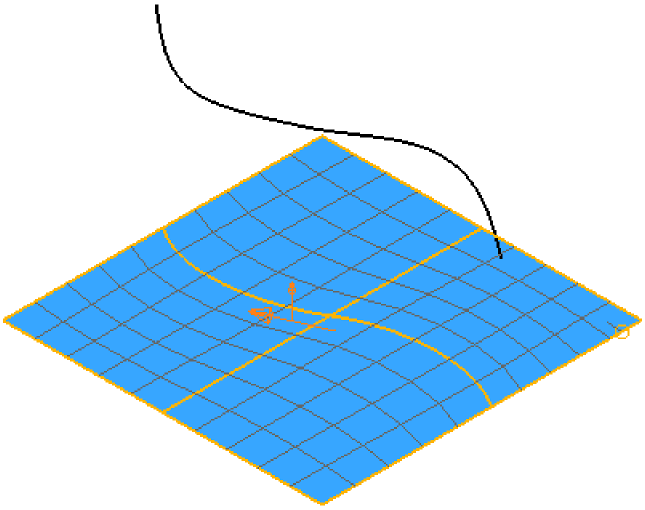
To match a lateral to a curve:

1 Click **Match lateral to curve projection**  (*Surface Edit toolbar*)

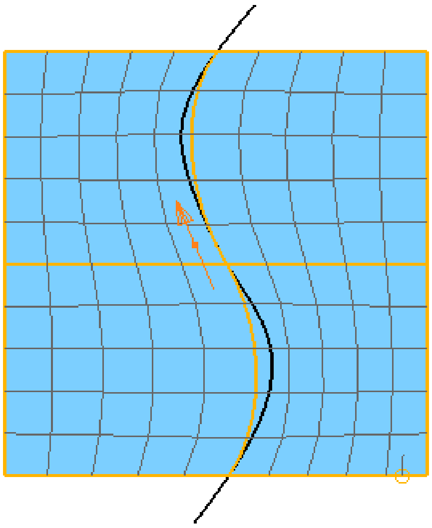
2 Select the curve.



3 Click **Preview**.



If you view the surface from the top, you can see that the lateral matches the shape of the curve.



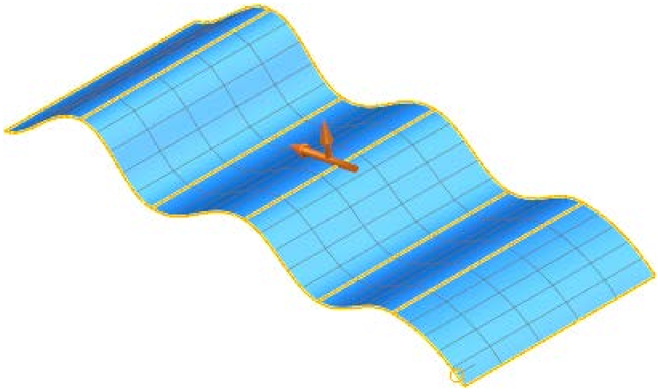
You can change your wireframe object, settings on the dialog, or principal plane until you are satisfied with the previewed surface.

Breaking a surface

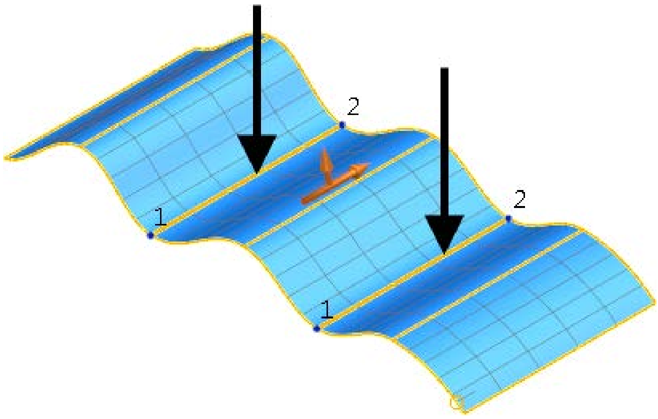
A surface can be broken into smaller pieces at selected curves.

1 Select the curves where you would like to break the surface.

2 Consider the following surface.



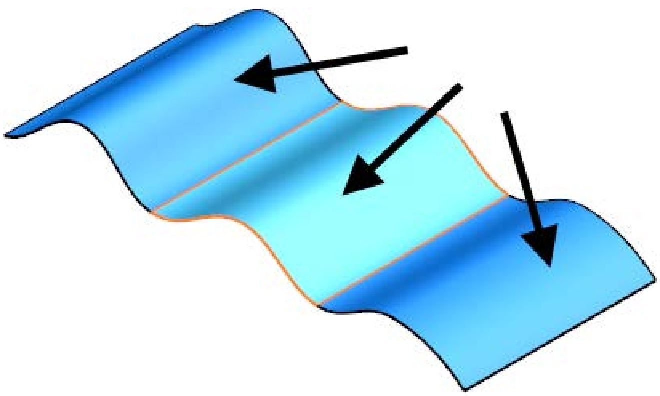
The curves are selected as shown below.



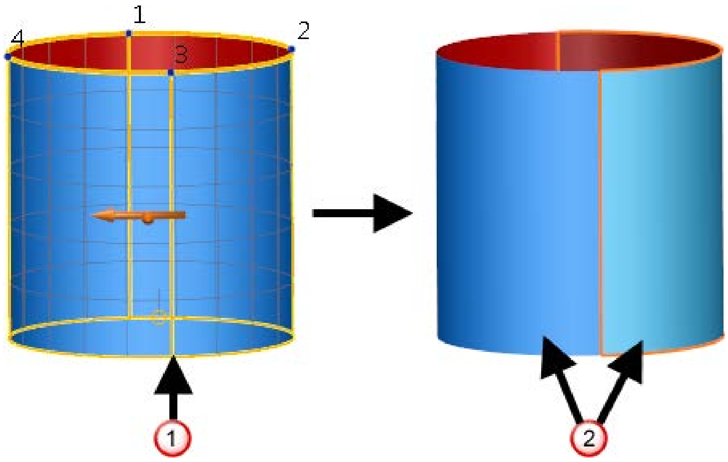
You can select a mixture of lateral and longitudinal curves.

3 Click  (*Surface Edit toolbar*).

The surface breaks into smaller pieces at the selected curves.



If you select a curve (other than curve number 1) on a surface that is closed in the other direction, then two surfaces are created. One surface is from curve 1 to the selected curve and the other is from the selected curve to the original closure. In the following example, we break the surface at curve 3 1 *to give two surfaces* 2.



When a trimmed surface is broken at selected curves, the trim data is mapped onto the new surfaces thus preserving the original trim result. If you break a trimmed surface that is displayed as trimmed, any resulting surfaces that fall entirely within the trimmed away region are discarded. To keep these discarded surfaces, you can turn the local or global trim off before breaking. (Use the **Trim ON/OFF** buttons on the **Surface Edit** toolbar to turn local trim *OFF*, and the **Trimming** option on the **Surface** page of the **Options** dialog to turn global trim *OFF*).

Toggling the trimming drawing

You can change the trim settings of a single surface to display or not display its trimmed area.

Two trim flags are provided:

- The global flag affects all surfaces.
- The local flag affects only the surface for which it is set.

If the global trim flag is *OFF*, then all surfaces regardless of whether their local flag is set *ON*/*OFF*, their trimmed boundaries are ignored and their whole shape is displayed. If the global flag is *ON*, then the whole shape of a surface is only displayed when its local flag is *OFF*.

The global trim flag is set on the **Surface** page of the **Options** dialog.

We recommend that you normally work with the global trim flag *ON*, because untrimmed surface shapes can be confusing. You can always set the trimming *ON* or *OFF* locally.

If the whole shape of a surface is displayed, then any operation on the surface ignores its trim boundaries.

To display or not display the trimmed area of a surface:

1 Click (*Surface Edit toolbar*) to turn trimming *OFF*.

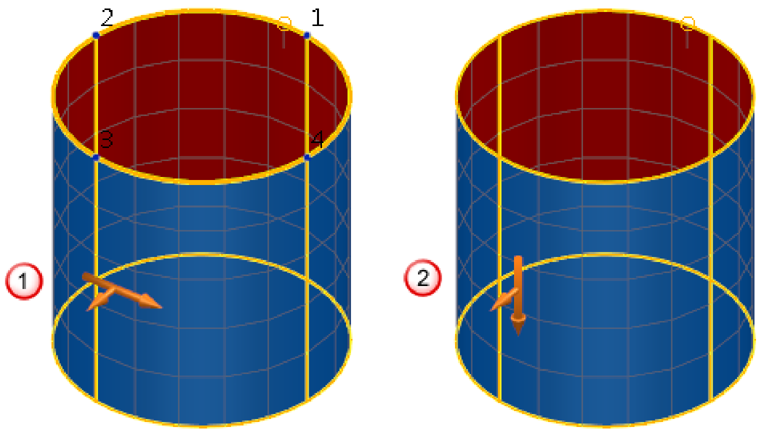
2 Click (*Surface Edit toolbar*) to turn trimming *ON*.

Reversing the surface

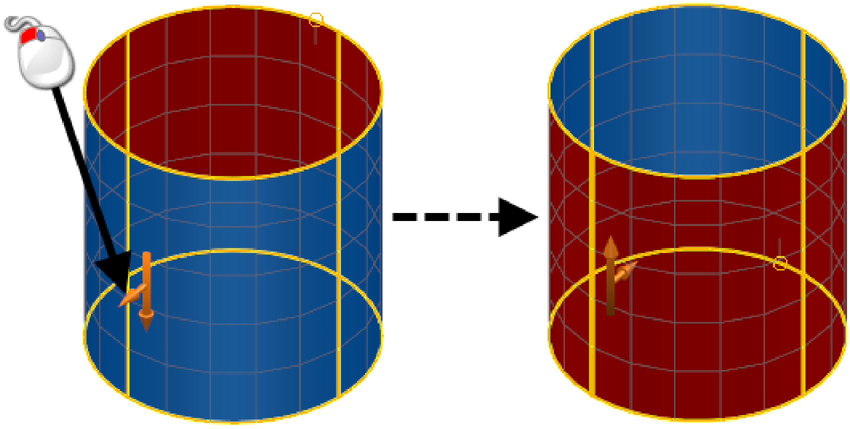
To reverse a surface:

- Click the direction arrow on the surface.

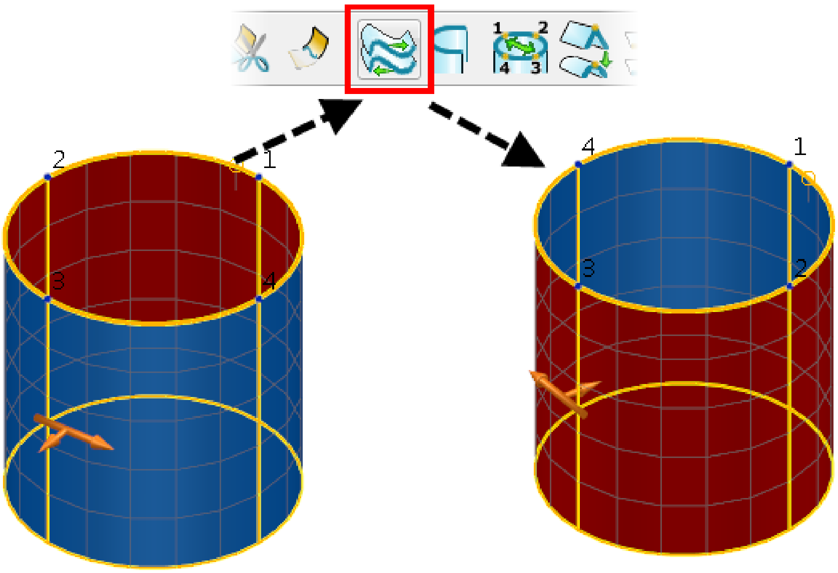
  1 Click the large arrow handle to choose between lateral 1 or longitudinal 2.



  2 Click the small arrow handle to reverse the surface.



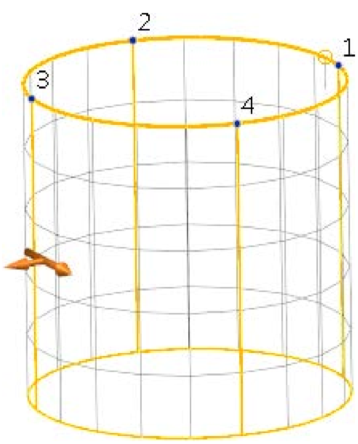
- Select a lateral or longitudinal and click  (*Surface Edit toolbar*).



Opening a surface

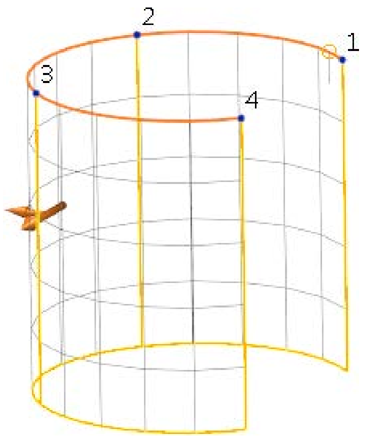
1. Use the direction arrow on the surface to select the direction you wish to open the surface.

   In this case, the lateral direction is selected:

2. Click  (*Surface Edit toolbar*).

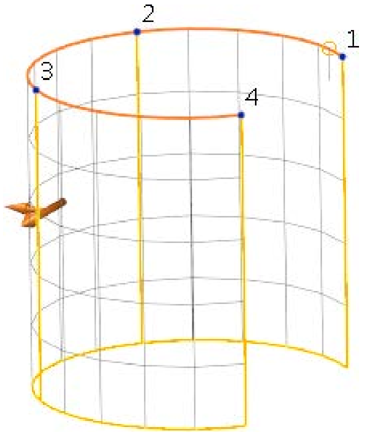
The surface opens between the first and the last point on each lateral.



Closing a surface

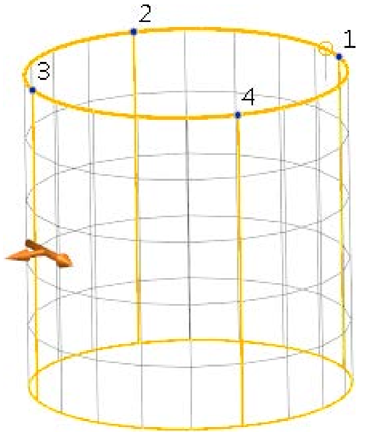
1 Use the direction arrow on the surface to select the direction you wish to close the surface.

In this case, the lateral direction is selected.

2 

3 Click  (*Surface Edit toolbar*).

This closes the surface between the first and last points on each lateral.

4 

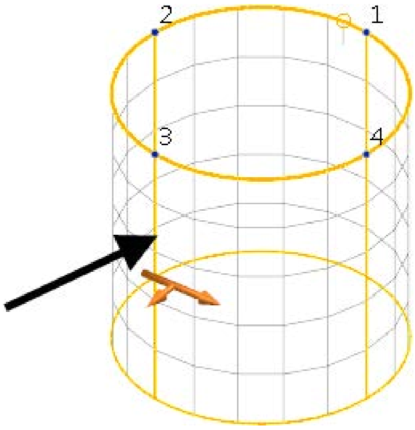
Renumbering curves

Curves may be renumbered. One reason for this may be to open a surface at a different location.

You can only renumber the curves if the surface is closed in that direction.

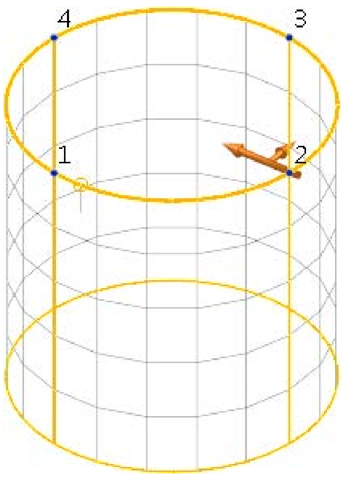
1 Select the curve you wish to number as 1.

In this case, longitudinal 3 is selected.



2 Click  (*Surface Edit toolbar*).

The longitudinals have been renumbered.



Splining surfaces

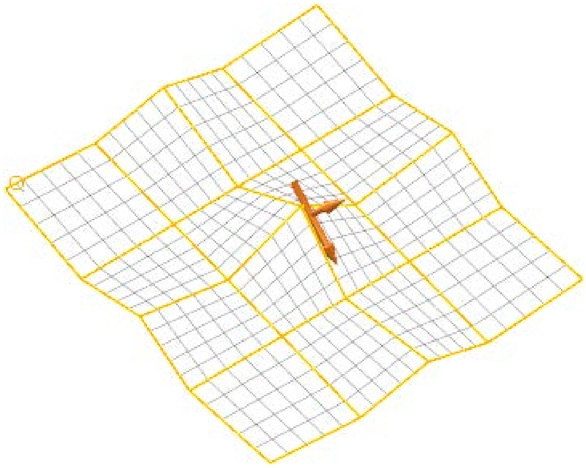
This smoothes surfaces to give tangency continuity on selected points. The positions of the selected points are changed during this operation.

Tangent continuity is a smooth change in tangent direction as you move across the surface.

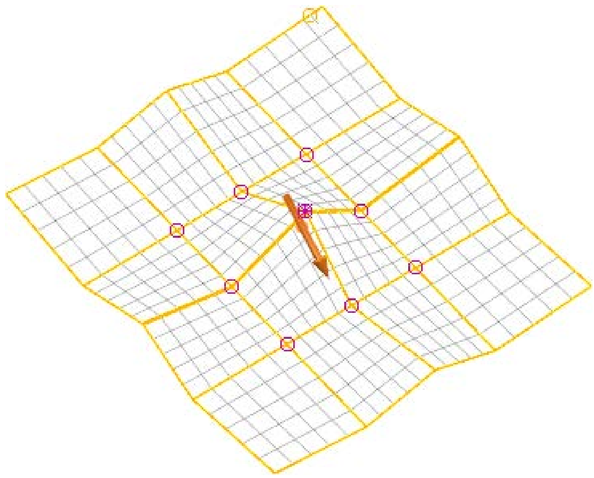
1 Select specific points on the surface.

Selected curves are ignored.

The following surface willl be used as an example.

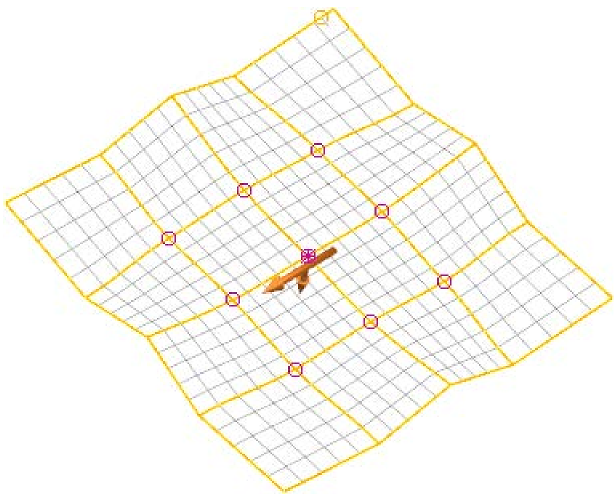


The selected points are marked as circles.



Points along the edges of surfaces cannot be splined.

2 Click (*Surface Edit toolbar*).



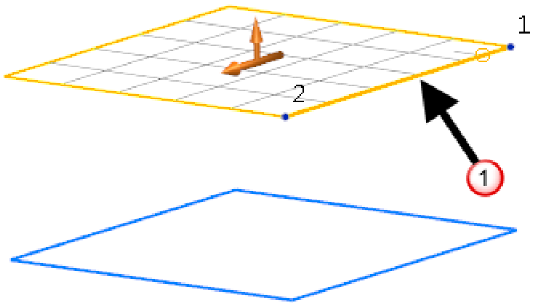
Selected points are splined in both the lateral and longitudinal directions, giving a smooth surface at these locations.

Appending surfaces

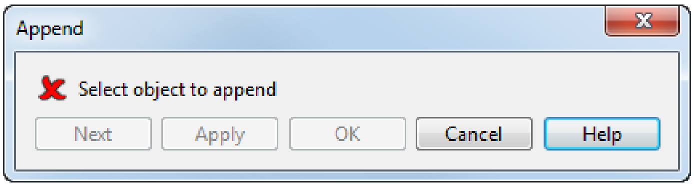
You can join two power surfaces together to produce a single continuous surface. It is necessary for both surfaces to be either open or closed in the direction perpendicular to the connection. Additionally, the curves along the joining edges of the surfaces must have the same number of spans.

Appending two surfaces — example

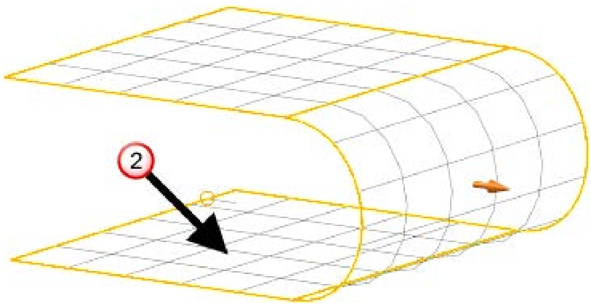
1 Select an edge curve on the power surface you want to append *from* 1.



2 Click  (*Surface Edit toolbar*) to display the **Append dialog** (see page 149).



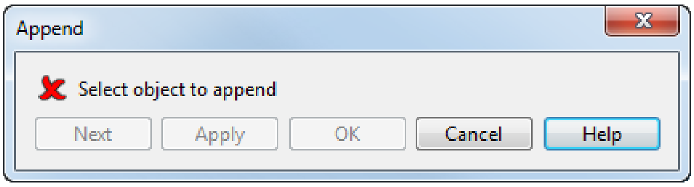
3 Select the power surface you want to append *to* 2.



4 Click **OK** to accept the append and close the dialog.

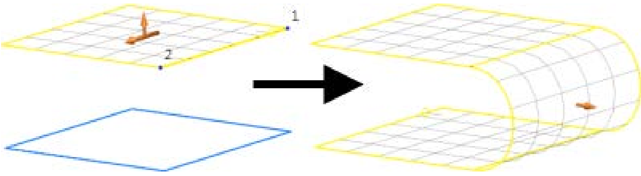
Append dialog

Use this dialog to select and append secondary surfaces to a primary selected surface.



When you select the secondary surface:

- changes to .
- The selected curve of the primary surface connects to the closest matching edge curve of the secondary surface, by adding a row of patches between the two surfaces.



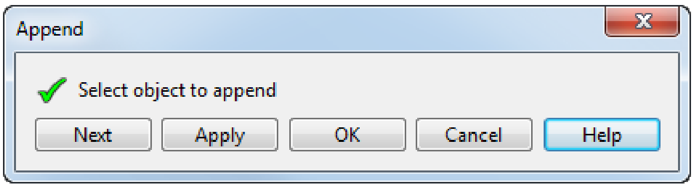
Click a different secondary surface to undo the append and select a new surface. An appended surface using the new secondary surface is shown.

Click the arrow normal to the surface to reverse the direction of the newly appended surface.

To deselect the secondary surface, select any surface curve on either of the surfaces. changes back to .

Dialog options

After you have selected the surfaces to be appended, use the following options on the **Append dialog**:



- **Next** — Click to cycle through the solutions for appending the two surfaces. For each solution, PowerSHAPE selects a different matching edge curve on the second surface.
- **Apply** — Click to append the surfaces and automatically select the next open edge curve, in preparation for further appending, without closing the dialog. If the automatically selected edge curve is not correct, select a new edge curve.

*Right click the model to perform the same action as selecting **Apply.***

- **OK** — Click to save the appended surface and exit the dialog. The first surface becomes the appended surface and the second surface is deleted. Spines and trim boundaries (and their pcurves) on either surface are deleted.
- **Cancel** — Click to exit the dialog and undo any editing, including the reversal of surfaces and the removal of any spines or trim boundaries.

Editing a Power Surface using the Edit menu

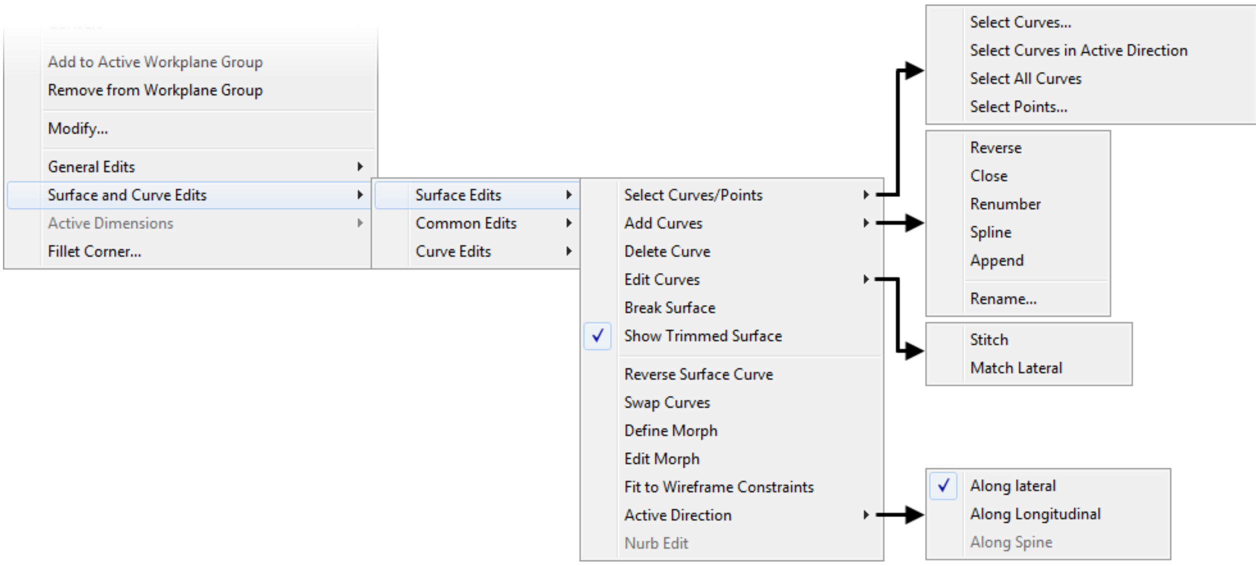
When you have a Power Surface selected, you can edit it using the options available from **Edit > Surface and Curve edits**. Most of these options are also available from the **Surface/Curve Edit toolbar**.

For details on editing Power Surfaces using the Edit menu, see:

- Surface Edits (see page 150)
- Common Edits (see page 162)
- Curve Edits (see page 163)

Surface Edits submenu

Select **Edit > Surface and Curve Edits > Surface Edits** to access the surface edit options.



For details on the options that appear on the **Surface/Curve Edit** toolbar, see Editing a Power Surface using the toolbar (see page 112).

The following options do not appear on the **Surface/Curve Edit** toolbar:

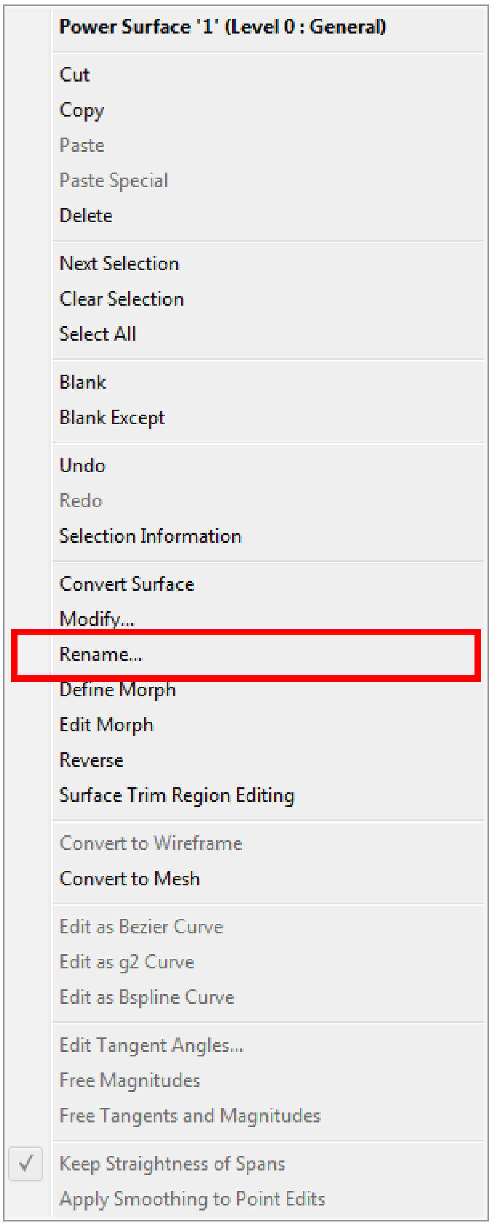
- Rename (see page 151)
- Swap Curves (see page 152)
- Define Morph (see page 237)
- Edit Morph (see page 266)
- Fit to Wireframe Constraints (see page 153)
- Active Direction (see page 160)
- NURB Edit (see page 162)

Renaming a surface

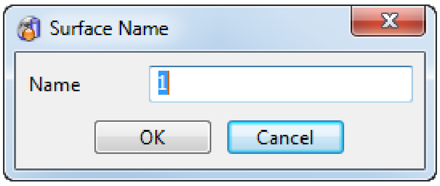
PowerSHAPE by default gives each surface a unique number. You can change this to a name of your choice in the same way as you can for other objects.

1 Right-click on a power surface.

2 Select **Rename** from the context menu.



The **Surface Name** dialog is displayed.



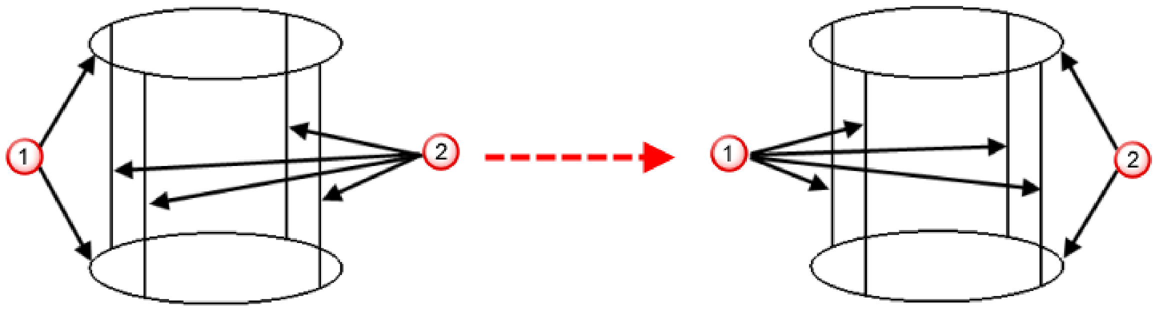
3 In the **Name** text box, type the new name for the surface.

4 Click **OK**.

Swapping curves

You can swap which curves are laterals and longitudinals.

Select **Edit > Surface and Curve Edits > Surface Edits > Swap Curves** and the laterals 1 and longitudinals 2 swap round.



Fitting a surface to wireframe and point objects

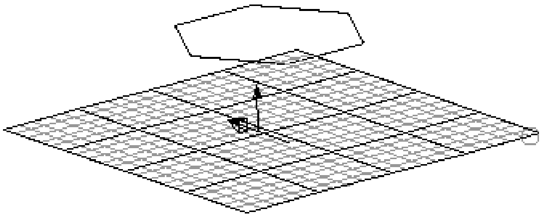
You can edit a surface such that it passes through existing wireframe and point objects. (This is also known as deformation modelling.)

This command treats the original surface as if it is a sheet of elastic rubber that can be deformed to pass through any wireframe and point objects you supply. A 'thin' sheet of elastic rubber can be easily stretched to form a complex shape, but may 'pucker' in areas where the surface changes direction rapidly. A 'thick' sheet of rubber may not be able to be deformed to a complex shape, but will resist 'puckering' and will be generally smooth (curvature continuous). The rubber may be very elastic (trying to minimise its surface area), or less elastic. As with the 'thickness' of the rubber, strong elasticity will tend to make the surface smoother but less able to deform to complex, rapidly changing shapes.

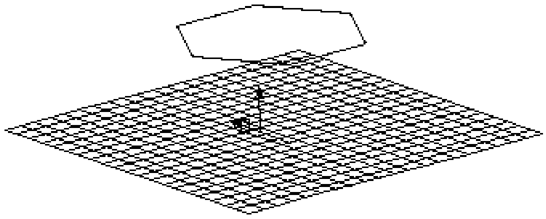
This command allows you to convert a surface to a 'rubber' surface and to specify the elasticity and smoothness ('thickness') of the 'rubber'. It then allows you to specify wireframe and point objects that act as constraints ('magnets' pulling the surface towards the wireframe and point objects). With low strength elasticity and low smoothness the surface should match the constraining curves/points fairly well. With higher elasticity and smoothness, the surface will be smoother but may not pass through the constraining curves/points at all.

This type of deformation is only possible on NURBS surfaces. PowerSHAPE will automatically convert power surfaces to NURBS surfaces as part of the operation, but will not automatically convert them back again at the end. In general, it is better to leave surfaces modelled in this way as NURBS surfaces as conversion back to a power surface may result in a surface with a large number of lateral and longitudinal curves.

Consider the following wireframe and surface.



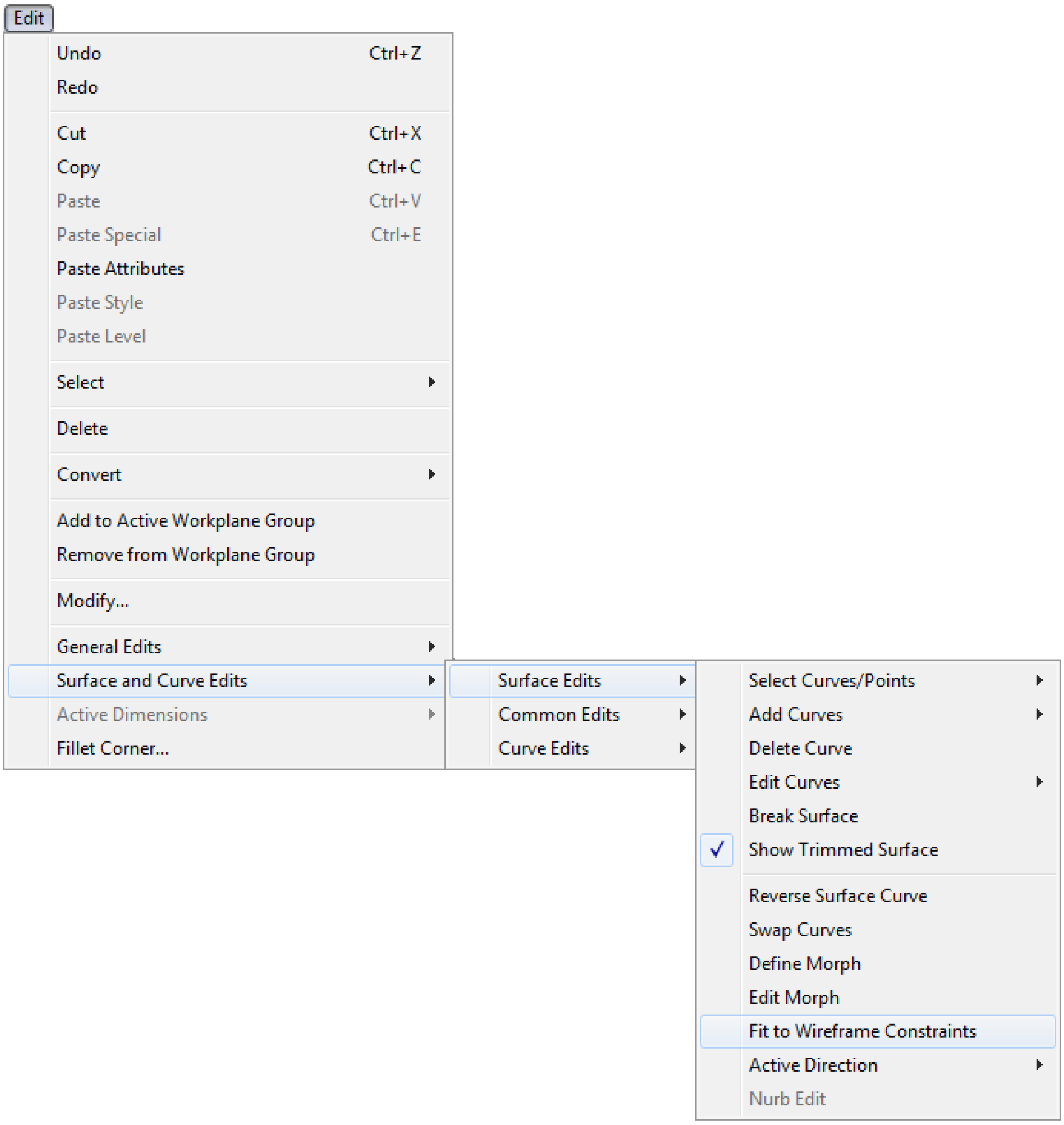
The detail lines of the surface become bold to help show the preview of the fitted surface clearly.



You can control the number of detail lines displayed using the Surfaces page of the **Options** dialog.

1 Select the surface.

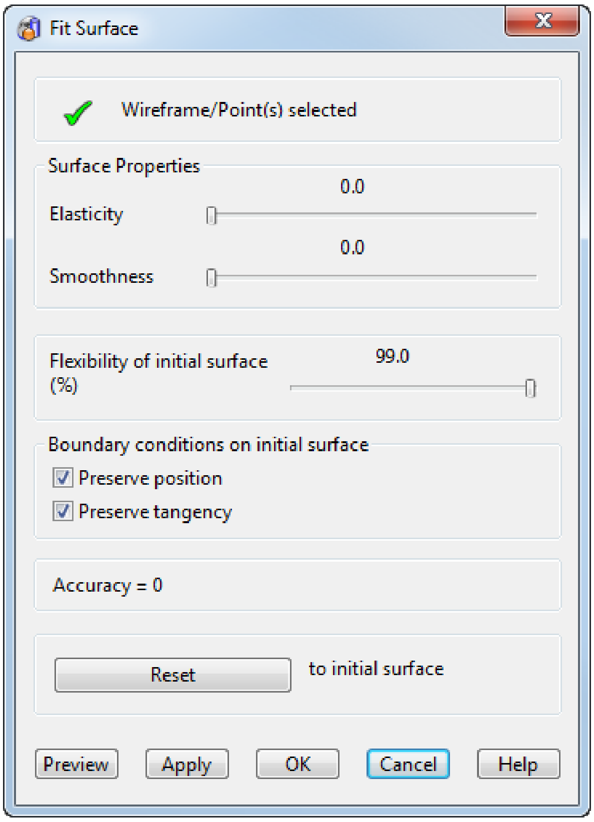
2 Select **Fit to wireframe constraints** from the Surface and Curve Edits submenu from the **Edit** menu.



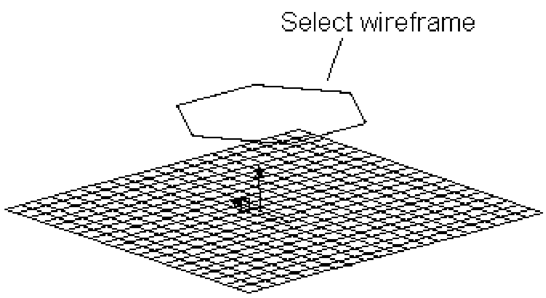
3 Use the Fit Surface dialog (see page 156) to fit the surface to the wireframe.

Fit Surface dialog

Use this dialog to fit a surface to wireframe and point objects.



1 When you select the wireframe and point objects you want to fit the selected surface to, the ✘ changes to ✔ next to the **Wireframe/Point(s) selected** option.

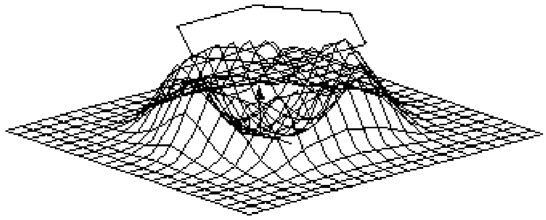


If a composite curve is selected and is created from an underlying surface, then the composite curve contains information on the tangent data of the surface. This tangent data is automatically used as an additional constraint.

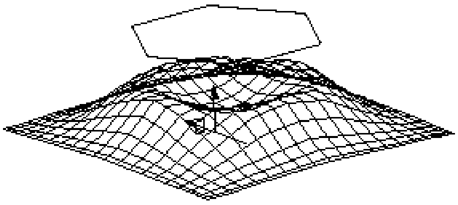
When illustrating a setting, we have not used the default values of the other settings to help you to see clearly the results of changing the setting being discussed.

Elasticity - This determines the elasticity of the material. If set to zero, the surface is not elastic and fits the constraints without stretching. When set at 10, the surface is stretched through its constraints and its area is minimised.

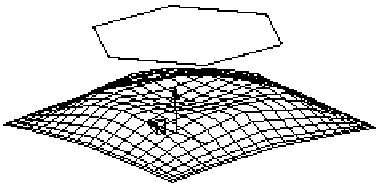
In our example, the fitted surface with an elasticity of zero is shown below.



If we increase the elasticity of the surface, its body shrinks to show it has become elastic and its area decreases.

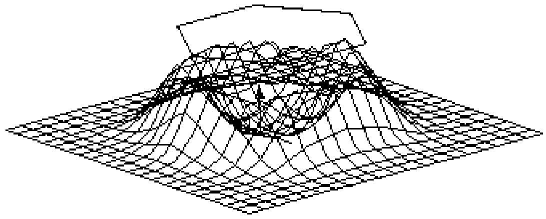


If we increase the elasticity further, the surface becomes tenser. Notice that the outer edges of the surface are shrinking.

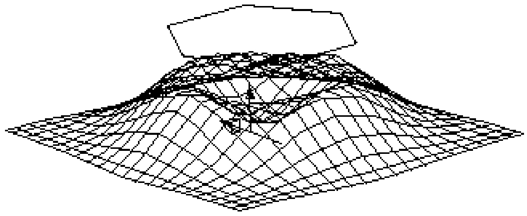


Smoothness - This slider is used to determine how smooth the edited surface is.

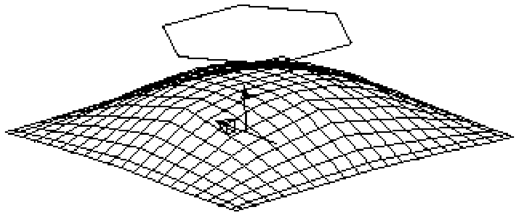
If we have a setting of zero, PowerSHAPE makes no attempt to smooth the surface.



As we increase the smoothness, ripples on the fitted surface are made smoother.

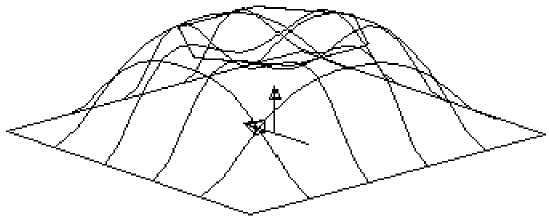


Increasing the smoothness further, makes the fitted surface smoother.

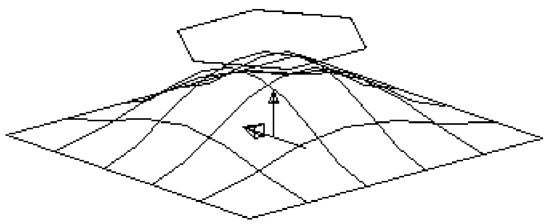


Flexibility of initial surface - This slider determines how flexible the initial surface is.

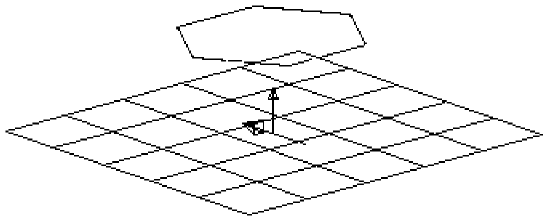
If the flexibility is set to 100%, the surface is totally flexible. In our example, we have the following fitted surface.



If the flexibility is reduced, the surface is reluctant to change its original shape.



If flexibility is set to zero, the body of the surface is rigid and remains fixed in its original position.

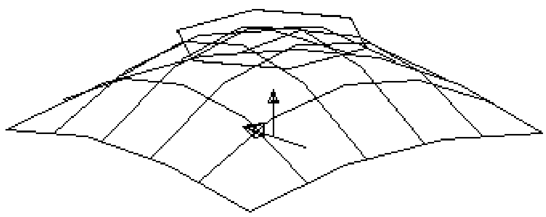


Boundary conditions on initial surface - You can constrain the fitted surface to data from the boundary of the initial surface using **Preserve position** and **Preserve tangency**.

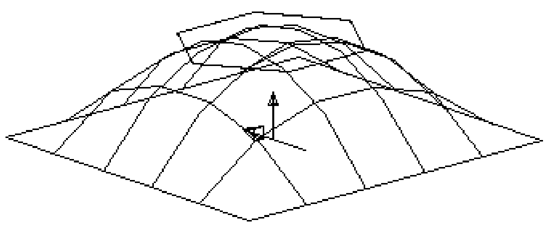
Preserve position - If on, the boundary positions of the initial surface are also boundary positions of the fitted surface.

Preserve tangency - If on, tangencies at the boundary of the initial surface are also tangencies at the boundary of the fitted surface. When Preserve tangency is ON, Preserve position is also ON.

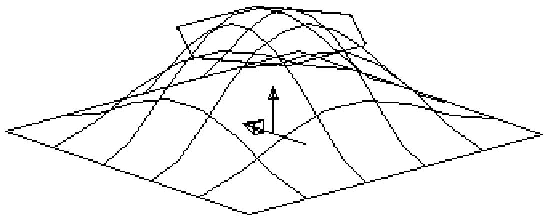
If both **Preserve position** and **Preserve tangency** are turned off, the fitted surface makes no attempt to preserve the boundary of the initial surface.



If **Preserve position** is on and **Preserve tangency** off, the fitted surface preserves the boundary of the initial surface and makes no attempt to preserve the tangents at its boundary.



If both **Preserve position** and **Preserve tangency** are on, the fitted surface preserves both the tangency and boundary of the initial surface.



Accuracy - This displays the maximum distance of the fitted surface from the wireframe and point objects.

You can improve the accuracy by saving the surface when it is close to the correct shape (by clicking the Apply button) and refitting.

Reset to initial surface - This resets the surface to its original shape. This is the shape of the surface when you first display the dialog.

Preview - Once wireframe and point objects are selected, clicking **Preview** displays the fitted surface as defined by the settings on the dialog. As you change any of the settings, the previewed surface automatically updates. To change the selection of wireframe and point objects, select the objects and click **Preview** to update the preview surface.

Apply - Saves the edited surface. You can fit the new surface to wireframe and point objects, again. You can improve the accuracy by fitting the surface to the same wireframe and point objects again.

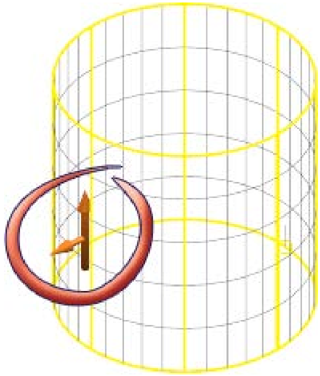
OK - Saves the edited surface and removes the dialog from the screen.

Cancel - Does not edit the surface and removes the dialog from the screen.

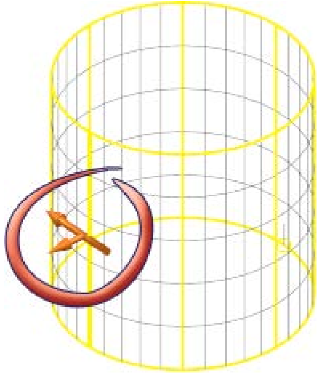
Selecting the direction

You can choose the lateral or longitudinal direction using the direction arrow on the surface.

In the diagram below, the direction arrow shows that the direction is longitudinal.



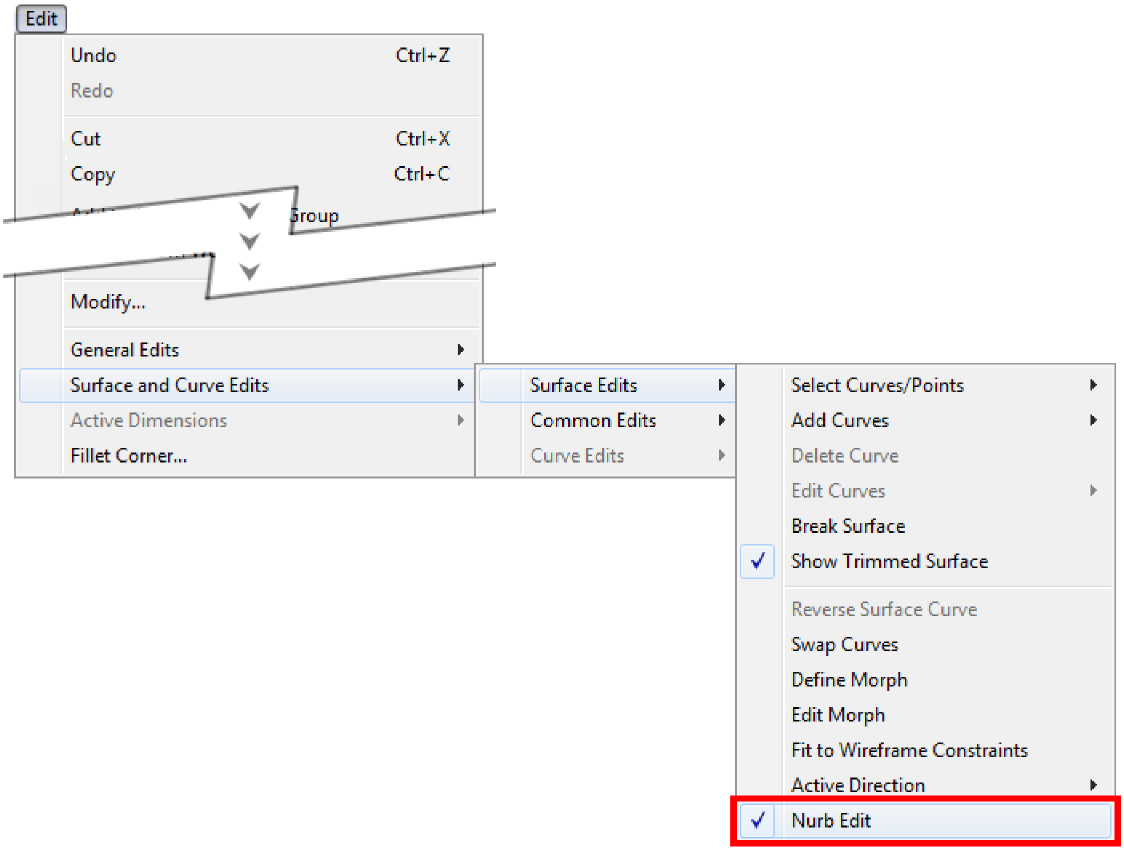
Click the arrow to change the direction. The selected direction becomes lateral.



Some edits, such as opening a surface, require only the direction of the curves. Other edits, such as renumbering the curves, require a current curve to be defined too.

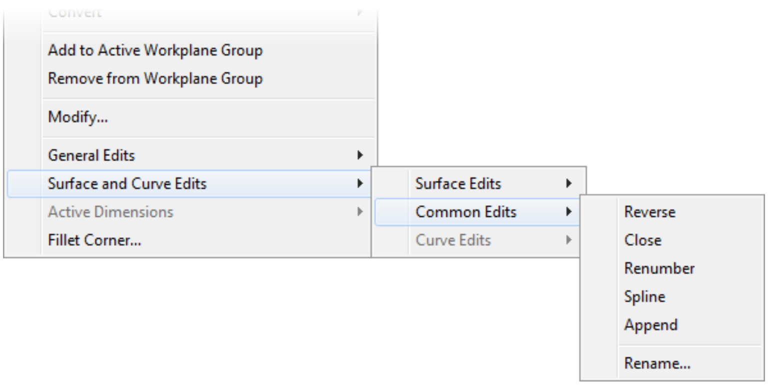
NURB editing mode

From the **Edit** menu, select **NURB Edit** to enter NURB editing mode.



Common Edits

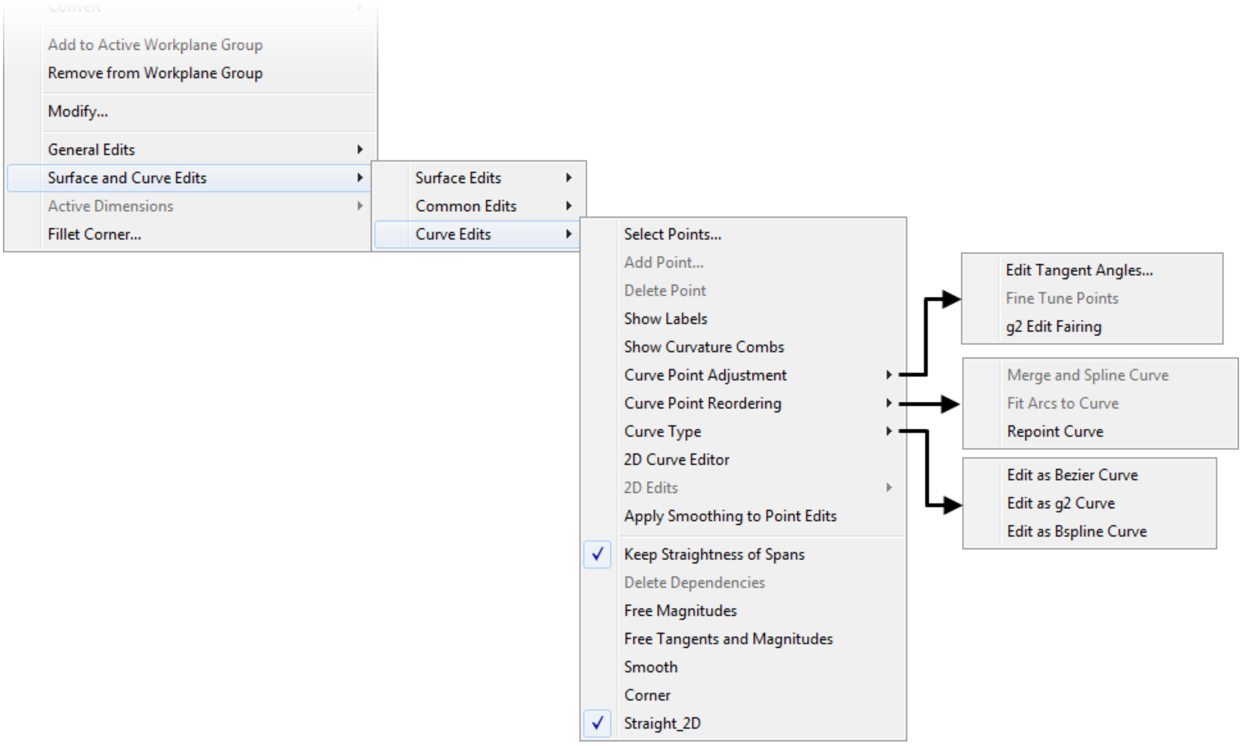
Select **Edit > Surface and Curve Edits > Common Edits** to access the common edit options for surfaces and curves.



The common edits options are available from the Surface/Curve Edit (see page 112) toolbar.

Curve Edits

Select **Edit > Surface and Curve Edits > Curve Edits** to access the curve edit options. The **Curve Edits** options are only available if you have a curve selected.



For details on the options that appear on the **Surface/Curve Edit** toolbar, see Editing a Power Surface using the toolbar (see page 112).

Graphically editing power surfaces

The following sections contain information on graphically editing power surfaces:

Graphically inserting a curve on a surface (see page 163).

Graphically moving a point along a surface curve (see page 164).

Graphically editing curves and points using handles (see page 166).

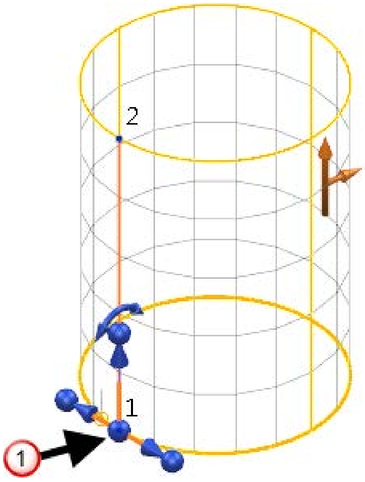
Graphically selecting multiple curves on a surface (see page 174).

Graphically selecting multiple curves on multiple surfaces (see page 175).

Graphically inserting a curve on a surface

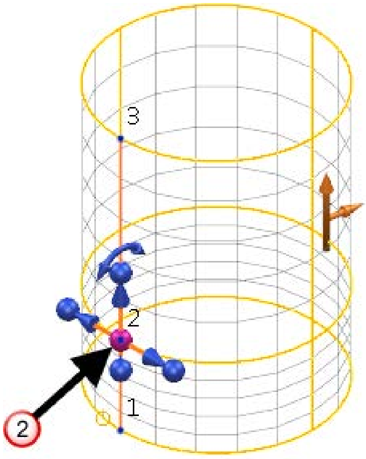
1 Select the surface.

2 Select any point on the surface. The handles of the point will be displayed 1.



3 Hold down **Ctrl**, click on the instrumented point and drag the cursor to the position where you want to add the new curve.

4 Release **Ctrl** and the mouse button to input the curve. This also renumbers the other curves to reflect the change 2.



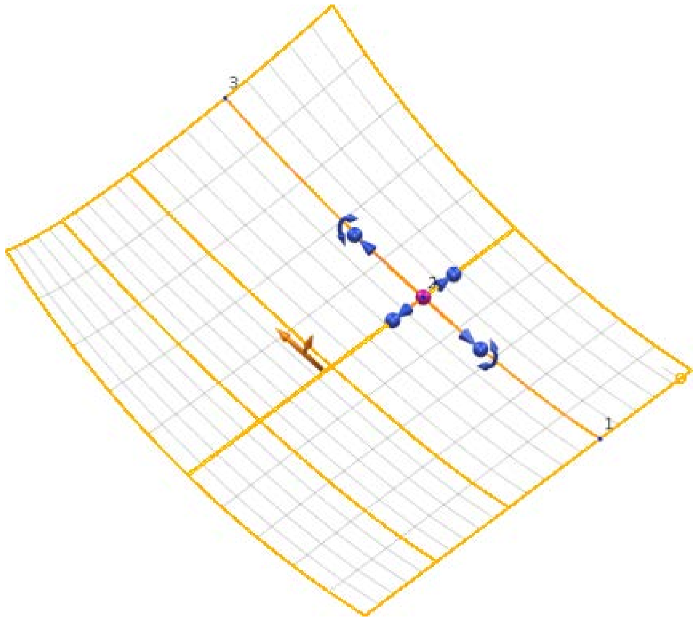
If you release the mouse over a lateral, only a longitudinal is inserted. If you release over a longitudinal, only a lateral is inserted. If you release the mouse at a position not on the surface, then no curve is inserted.

Graphically moving a point along a surface curve

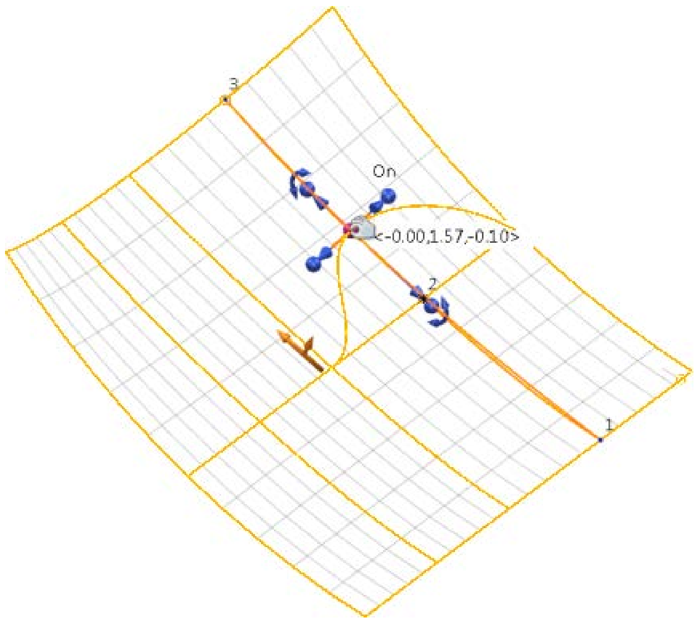
You can adjust the position of a point along its adjacent curves.

1 Select the surface.

2 Select any point on the surface. The handles of the point will be displayed.

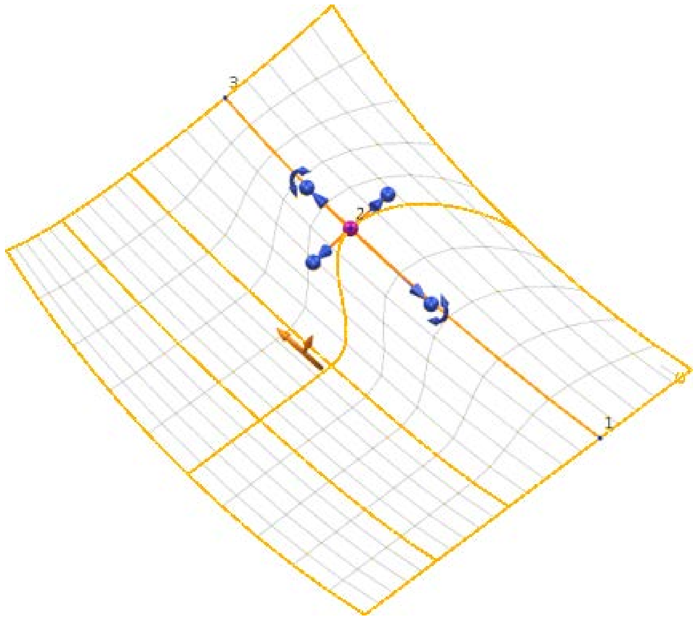


3 Hold down the **Shift** key and drag the instrumented point along its adjacent surface curves.



4 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.

Graphically editing curves and points using handles

You can modify a surface by using graphical handles to edit:

- Surface curves (see page 166).
- Surface curve points (see page 173).

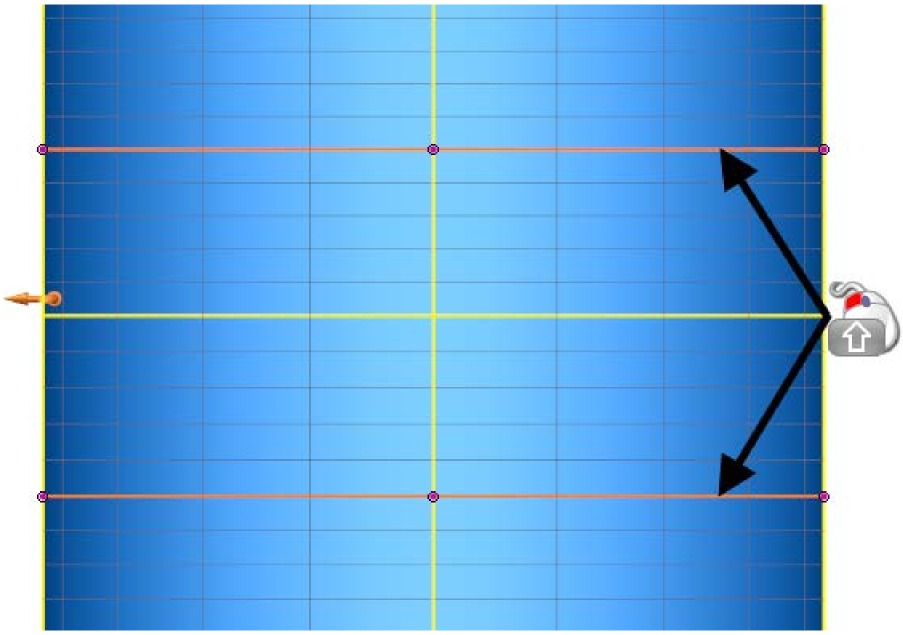
Graphically editing surface curves using handles

With a single curve or multiple curves selected, you can select the different curve handles from the following menus:

- **Edit > Surface and Curve Edits > Surface Edits > Selected Curves Graphical Handles**.
- Surface curve context menu.

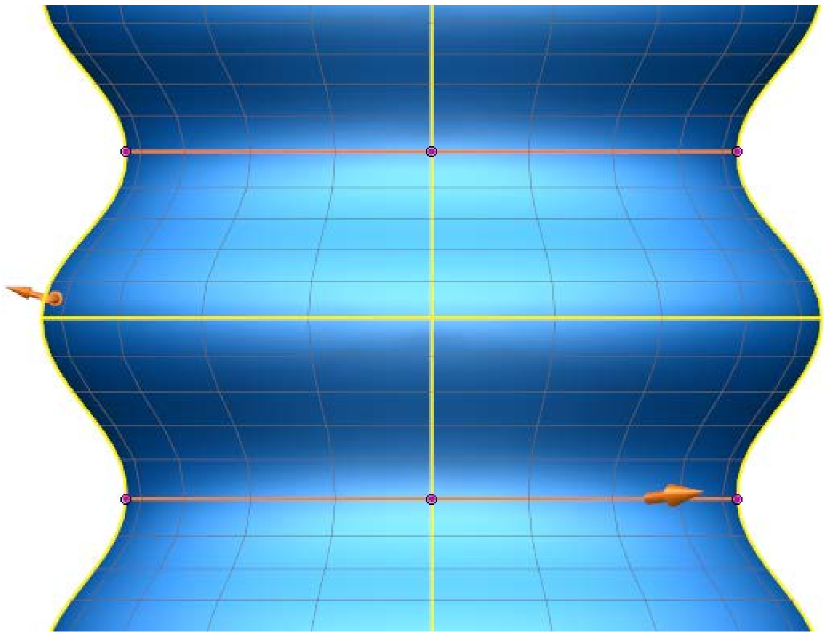
Graphically editing multiple surface curves - Example

1 Press and hold the **Shift** key, and click multiple curves on the surface to select them.

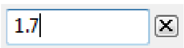


2 Right click a curve and select a graphic handle from the **Selected Curves Graphical Handles** flyout of the **Power Surface** context menu.

3 Drag the handle to move the curves.



*If you are using **Workplane graphic handles** or **Offset graphic handles,** a movement confirmation field is displayed. Enter the exact value of the movement into the field, and press the **Enter** key to confirm.*

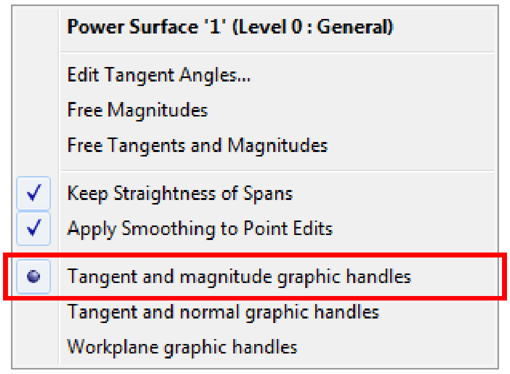


The following handles are available:

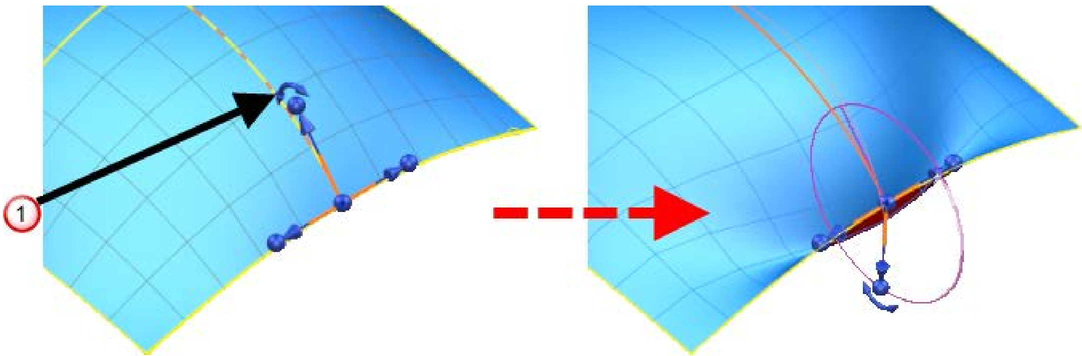
- **Workplane graphic handles**
- **Offset graphic handles**

Tangent and magnitude graphic handles

1. Select a point on the surface curve.
2. To display the context menu  the point.
3. Select **Tangent and magnitude graphic handles** from the context menu.

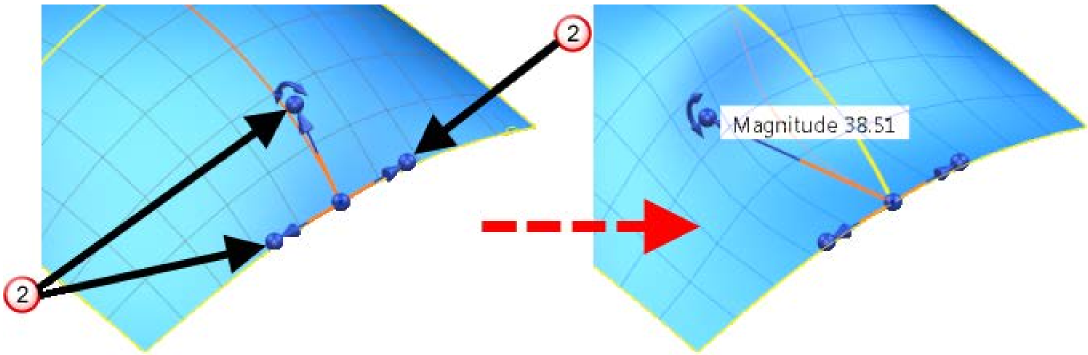


4. To edit the surface curve  the various handles:
   - Use the arced handle ① to modify the tangent angle of the curve, without changing the magnitude.

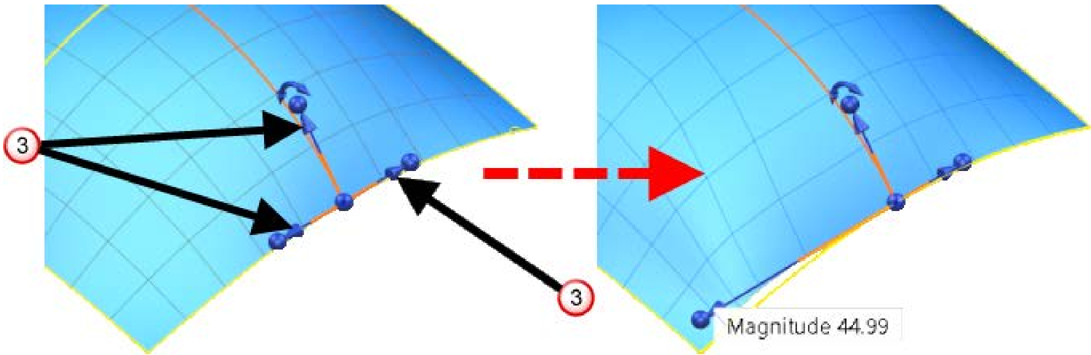


A circle is drawn in a fixed plane relative to its starting position

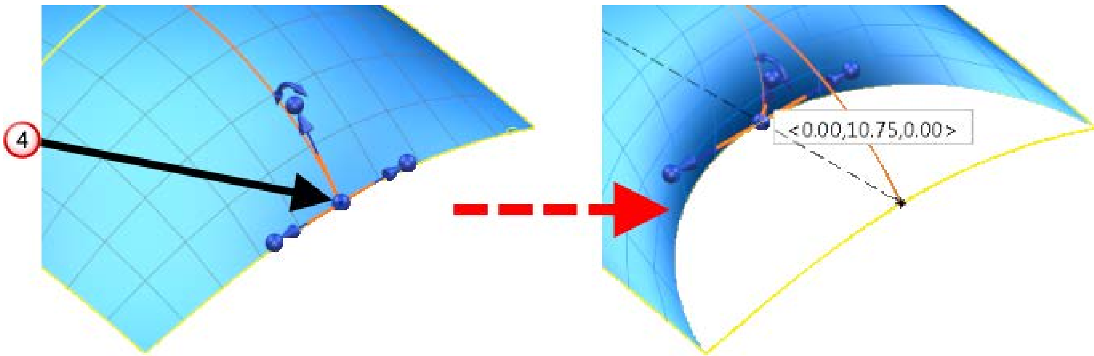
- Use the spherical handles (2) to freely modify the magnitude of the curve in any direction.



- Use the arrow tip handles (3) to modify the magnitude of the curve along the fixed tangent.



- Use the centre handle (4) to move the curve point freely.

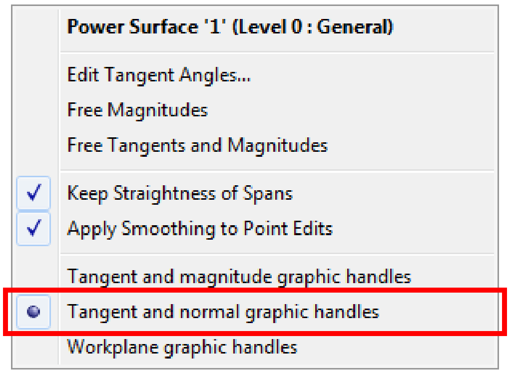


Moving the cursor over a handle highlights the handle, indicating that you can move it.

Tangent and normal graphic handles

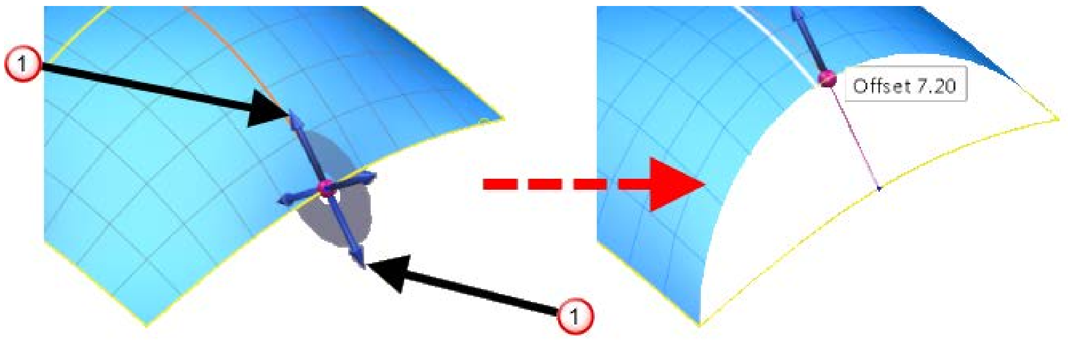
1. Select a point on the surface curve.
2. To display the context menu the point.

3 Select **Tangent and normal graphic handles** from the context menu.

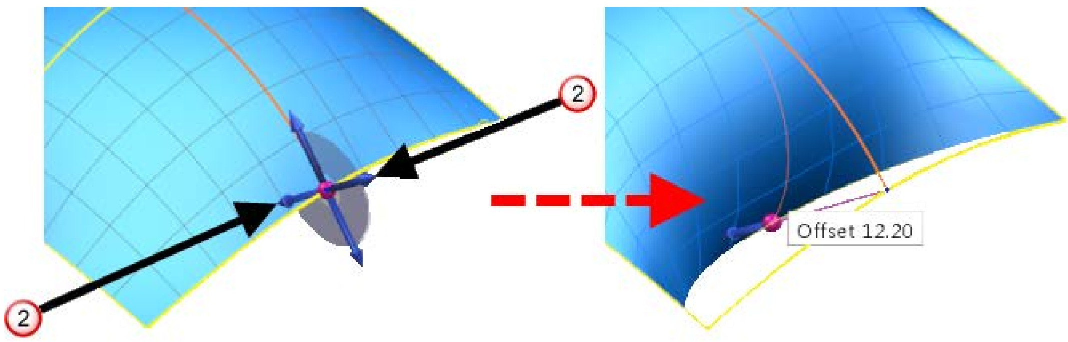


4 To edit the surface curve  the various handles:

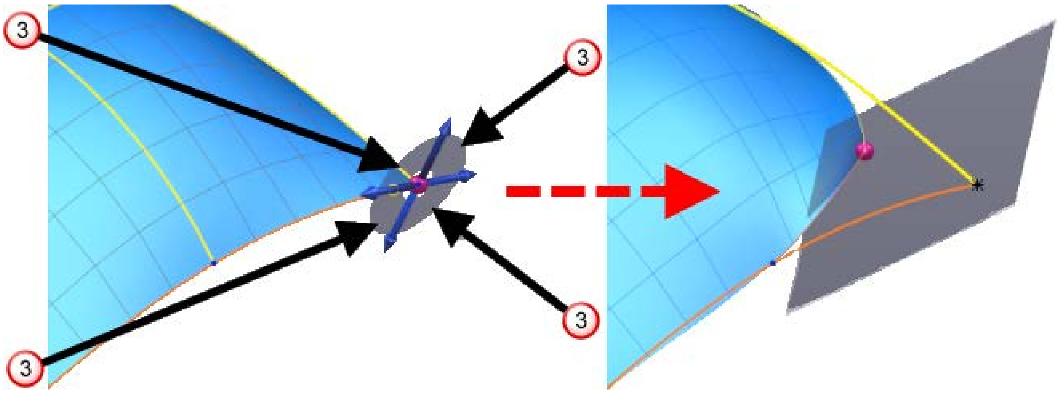
- Use the arrow handles *tangent* to the surface curve ① to move the point, in a fixed direction, along the tangent.



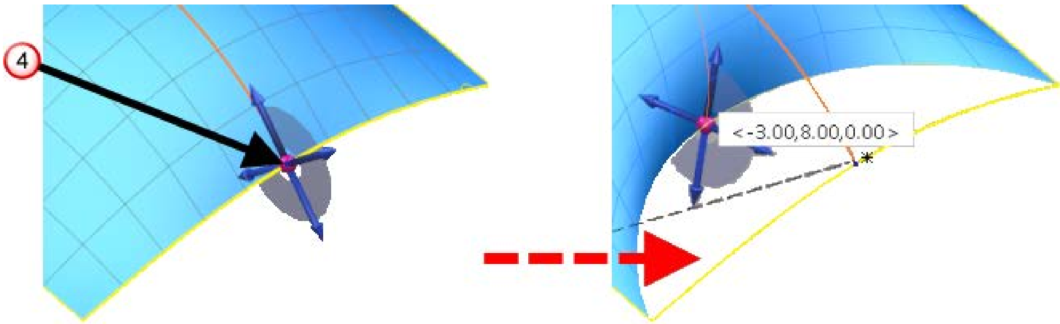
- Use the arrow handles *normal* to the surface curve ② to move the point, in a fixed direction, along the normal.



- Use the planar handles 3 to move the point across a 2D plane.



- Use the centre handle 4 to move the point freely.



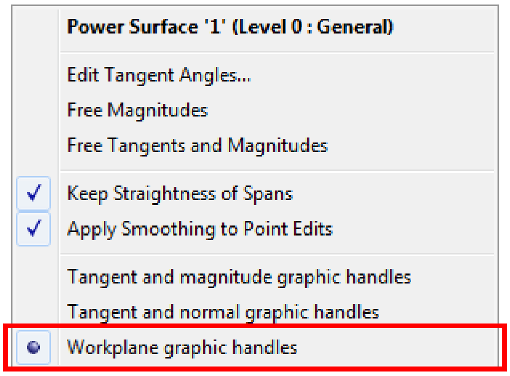
Moving the cursor over a handle highlights the handle, indicating that you can move it.

Workplane graphic handles

1 Select a point on the surface curve.

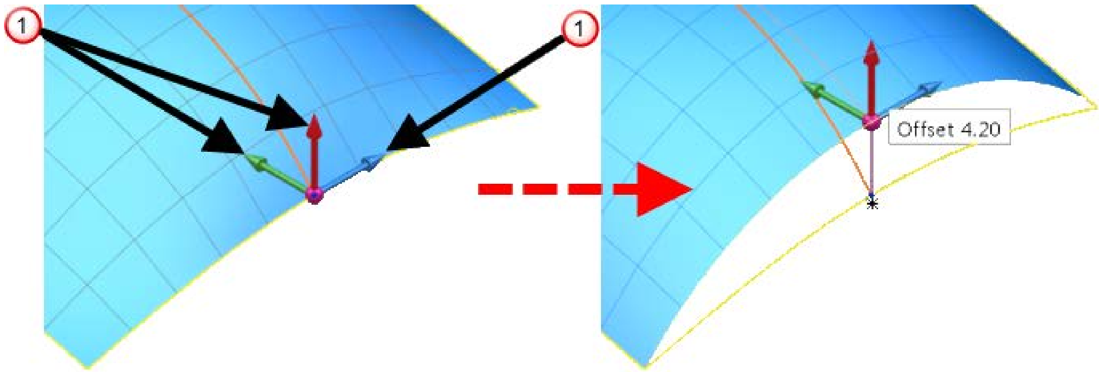
2 To display the context menu  the point.

3 Select **Tangent and normal graphic handles** from the context menu.

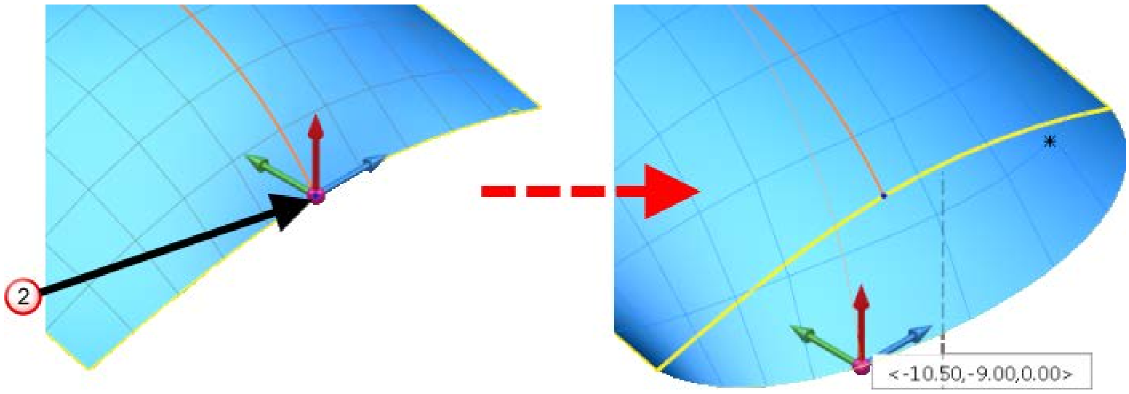


4 To edit the surface curve  the various handles:

- Use the workplane arrow handles  to move the point along a fixed axis.



- Use the workplane centre handle  to move the point freely.



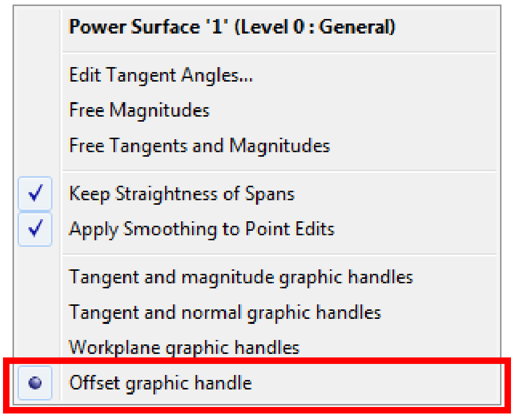
Moving the cursor over a handle highlights the handle, indicating that you can move it.

Offset graphic handle

Use the **Offset graphic handle** option to graphically modify surface curve points.

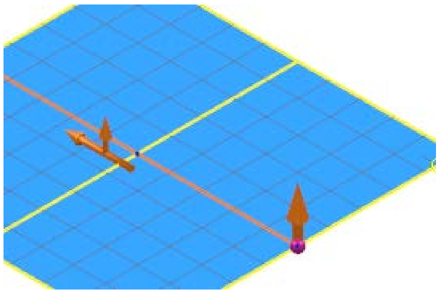
1 Select a point on the surface curve.

2 Right-click the point and select **Offset graphic handle** from the **Power Surface** context menu.

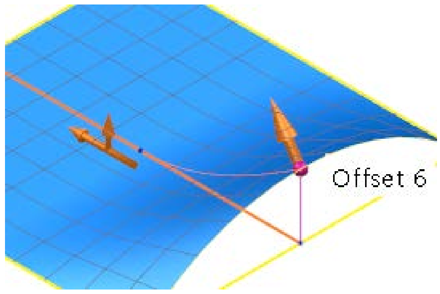


3 Click and drag the handle to edit the surface curve:

Before



After



The point is offset a set distance away from its original position.

Moving the cursor over a handle highlights the handle, indicating that you can move it.

Graphically editing surface curve points using handles

You can use graphic handles to edit a single curve point, or multiple curve points at the same time.

Single points

With a single surface curve point selected, you can select the different curve point handles from the following menus:

- **Edit > Surface and Curve Edits > Common Edits > Selected Point Graphical Handles**.
- Surface curve point context menu.

The following handles are available:

- **Tangent and magnitude graphic handles** (see page 168).
- **Tangent and normal graphic handles** (see page 169).
- **Workplane graphic handles** (see page 171).
- **Offset graphic handle** (see page 172).

Multiple points

With multiple surface curve points selected, you can select the different curve point handles from the following menus:

- **Edit > Surface and Curve Edits > Common Edits > Multiple Selected Point Graphical Handles**.
- Surface curve point context menu.

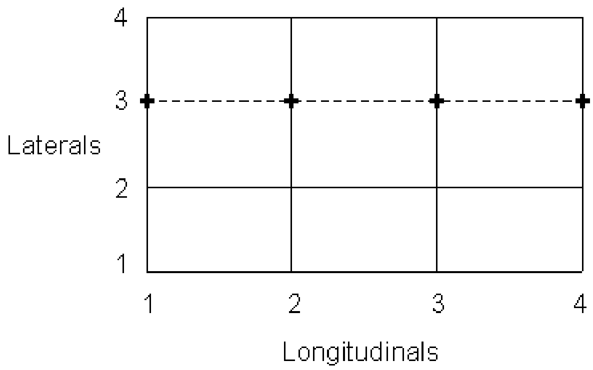
The following handles are available:

- **Simple graphical marks**.
- **Workplane graphic handles** (see page 171).
- **Offset graphic handle** (see page 172).

Graphically selecting multiple curves on a surface

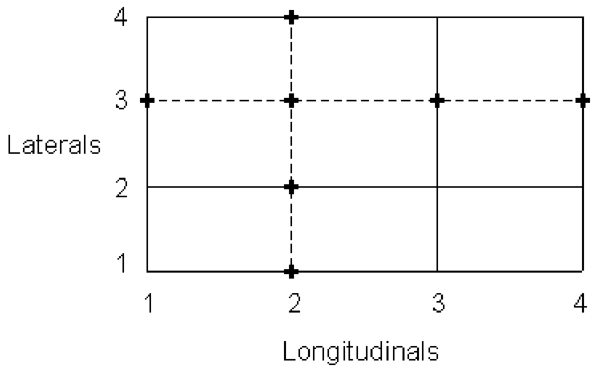
Clicking a surface curve selects it. Any other selected object becomes deselected.

Click a lateral to select it. It changes colour to show it is selected.



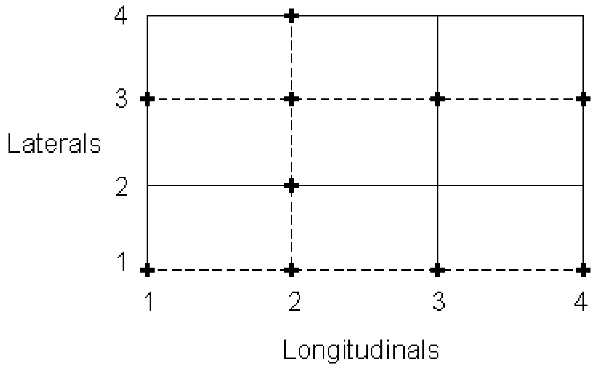
Shift-click adds a surface curve to the current selection of surface curves. If you **Shift-click** any other object, the curves are no longer selected. The object and the surface become the current selection.

If we now **Shift-select** longitudinal 2, it is added to the current selection and changes colour to show it is selected.

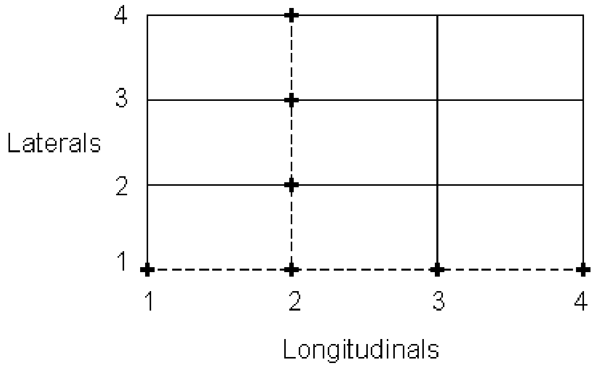


Ctrl-click adds a surface curve to the current selection of surface curves or removes the curve from the current selection if it is already selected.

If we **Ctrl-click** lateral 1, it is added to the selection.

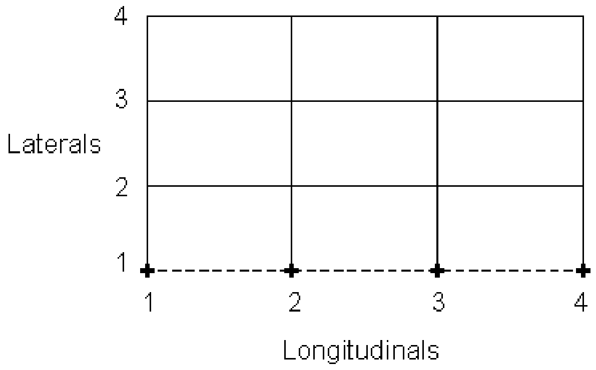


If we **Ctrl-click** lateral 3, it is removed from the selection.



Ctrl-Shift-click removes surface curves from the current selection.

If we **Ctrl-Shift-click** longitudinal 2, it is removed from the selection.



The **Shift** key with box selection adds points to the current selection.

Graphically selecting multiple curves on multiple surfaces

Select all the surfaces from which you want to select curves.

Use **Shift-click** to add a surface curve to the selection containing sub-components.

Use **Ctrl-Shift-click** to remove a surface curve from the selection containing sub-components.

Use **Shift** with box selection to add surface curves to the selection containing sub-components.

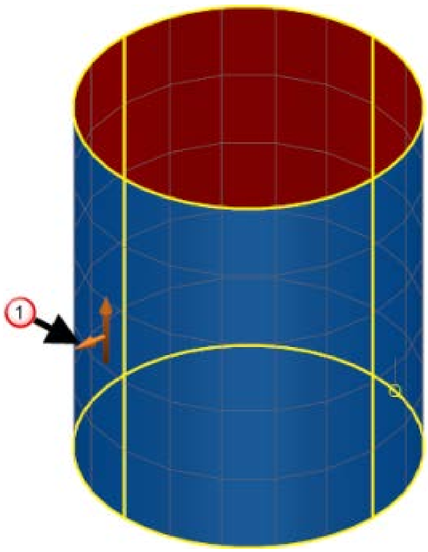
Use **Ctrl-Shift** with box selection to remove surface curves from the selection containing sub-components.

You can edit the selected curves using the **Edit sub-component** tool on the **Edit** toolbar.

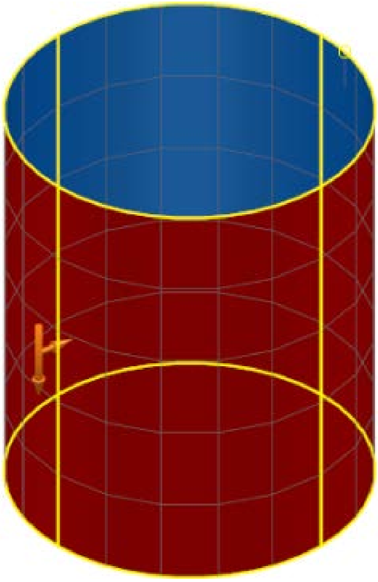
Graphically reversing a surface

Turn a surface inside out by changing the direction of the surface normal. This reverses the points of the laterals in the surface.

1 Double-click the surface to display the surface normal arrow 1.

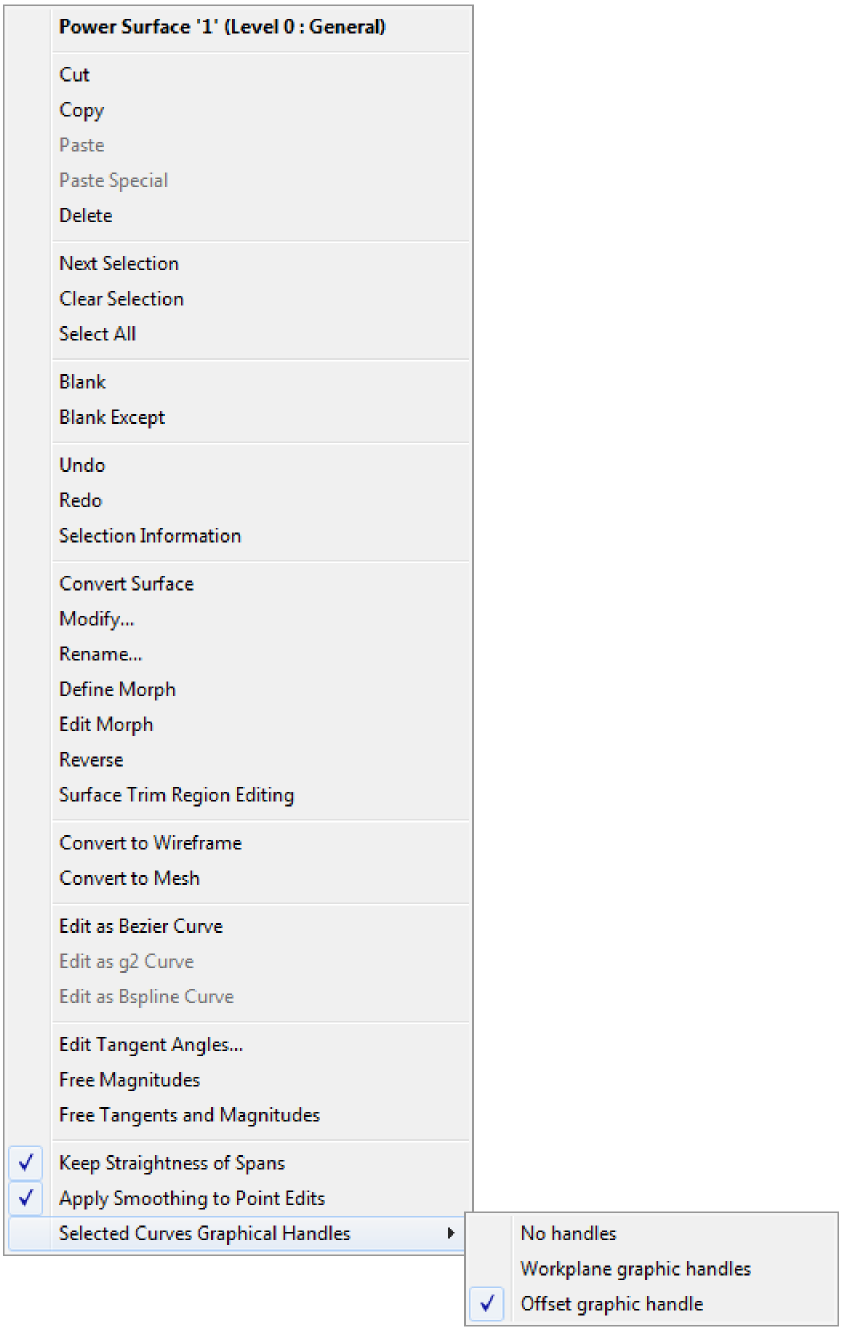


2 Click the arrow to change the direction of the surface normal.



Editing curves on a power surface

Right-click on a Power Surface to open the context menu. The top of the context menu displays the type of the surface, its name, and the level on which it lies.



You must have curves selected for the following option to be available:

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 is the same as clicking 

on the Tangent Editor dialog. Tangent magnitudes are freed through selected points along the selected surface curve. If no points are selected, the whole curve is freed.

Free Tangent and Magnitudes — This the same as clicking 

on the Tangent Editor dialog. The tangent directions and magnitudes are freed through selected points along the selected curve. If no points are selected, the whole curve is freed.

Converting between surface types

A power surface has a variety of editing functionality that is not available for BCPs or NURBS. In order to access this functionality, the BCP or NURB needs to be converted.

You can also convert primitive surfaces to power surfaces.

To convert your surface:

1 Select the surface.

2 Right-click the selected surface. From the context menu, select **Convert.**

This is the same as selecting Edit > Convert > Surface.

Fixing surfaces

The following section contains information on fixing surfaces using options that are available from the **Model Fixing** toolbar.

Orient surface normals (see page 179).

Approximating a surface to tolerance (see page 180).

Simplifying outside trim (see page 181).

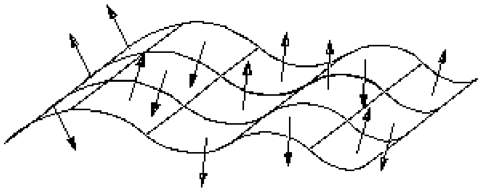
Orient surface normals

You can orient the normals of multiple surfaces such that the outside of two adjacent surfaces match.

A quick way of checking if surface normals are oriented is by shading the model. By default, the outside of a surface is shaded blue and the inside red (using the default colour scheme).

Two surfaces are adjacent if their edges match within general tolerance. Otherwise, they are treated as disconnected.

1 Select a group of surfaces.

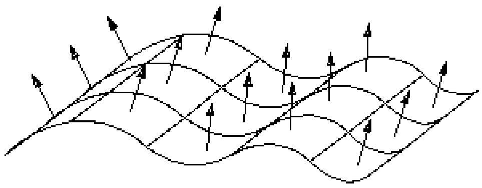


When you select multiple surfaces, by default,their surface normals are not displayed. You can display the surface normals by turning on ***Display normals for multiple surfaces*** *on the* ***Surfaces*** *page of the* ***Options*** *dialog.*

2 Click **Model Fixing**  on the **General edits**  flyout.

3 Click  (*Model Fixing toolbar*).

The surface normals all point such that the outside of two adjacent surfaces match.



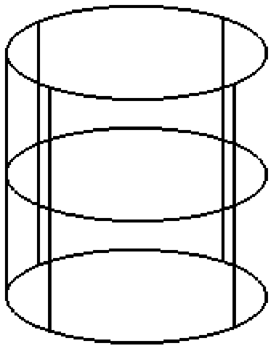
As a general rule, you should always keep the surface normals pointing out from the material.

Approximating a surface to tolerance

This removes laterals and longitudinals that do not support the surface structure within tolerance.

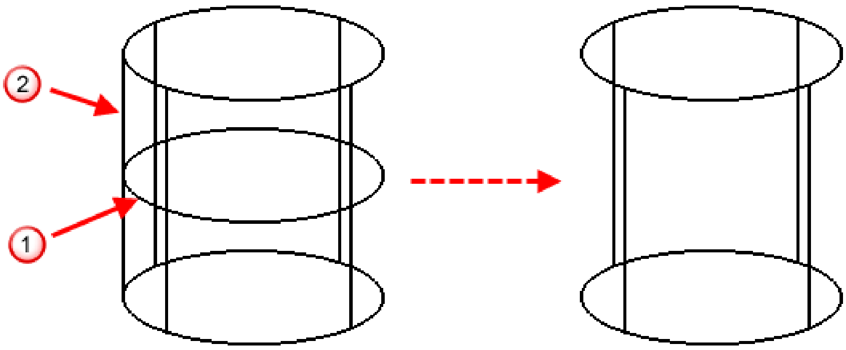
1 Click **Model Fixing**  on the **General edits**  flyout.

2 Select a surface.



3 Click  (*Model fixing toolbar*).

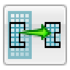
In this case, a lateral 1 and longitudinal 2 are removed.



Simplifying outside trim

This removes laterals and longitudinal made entirely invisible by being outside the trim boundaries.

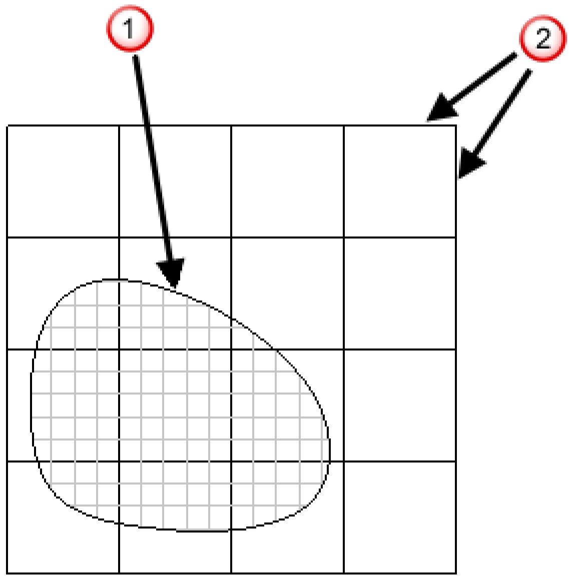
1 Click **Model Fixing**  on the **General edits**  flyout.

2 Click  (*Model fixing toolbar*).

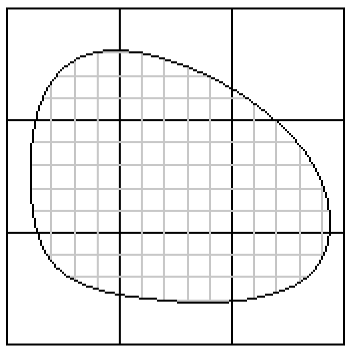
Let us use the following surface as an example.

1 Trim boundary.

2 Curves outside the trim boundary.



In this case, a lateral and longitudinal are removed.



Inspecting surfaces

You can inspect a solid, surface or component and display the following inspection data:

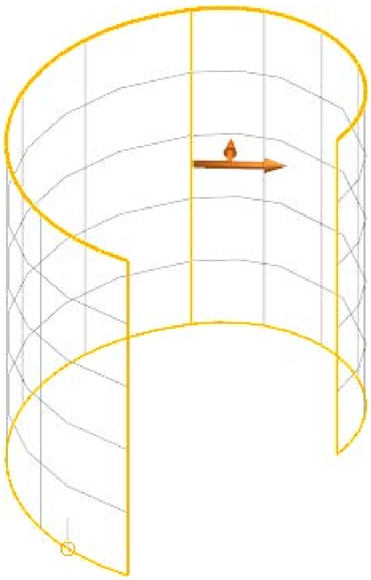
- coordinates
- surface normal
- draft angle (see page 184)
- wall thickness (see page 184)
- curvature (see page 184).

Surface inspection data can also be captured and sent to the clipboard to be pasted into any text editor. For details see Capturing surface inspection data (see page 185).

Displaying surface inspection data

Select **Model Analysis** on the **General edits** flyout.

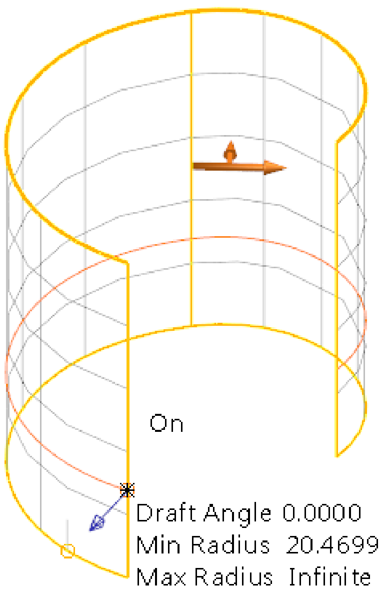
1 Select a solid, surface or component. For example:



You can select more than one item.

2 Click (*Model analysis toolbar*).

3 Left-click on a position on the selected surface. You can click anywhere on the surface, not just on a curve. By default, the surface normal arrow, draft angle and curvature of the position is displayed.



4 If you want to change the data displayed, select **Tools > Options > Tools > Analysis > Surface Analysis** to display the **Surface Analysis** options page.

5 Drag the cursor across the surface to display data at different positions.

If multiple items are selected, you can drag the cursor from one item to another. If the first item is no longer under the cursor, data from the second item is displayed.

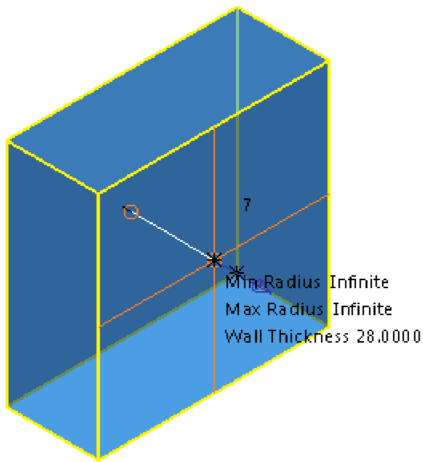
6 Click  to turn inspection mode *OFF*.

Inspecting surfaces - Draft angle

The draft angle is measured from the principle plane to the normal of the point on the surface under the cursor.

Inspecting surfaces - Wall thickness

Wall thickness is determined by measuring the distance between the cursor's position on a face, and the closest point to this on the back face. You must select all the faces to be considered before clicking the **Surface inspection** button . The positions are displayed graphically by a circle on the back surface joined by a line to the cursor position.



Inspecting surfaces - Curvature

You can display the curvature as either:

- maximum and minimum values

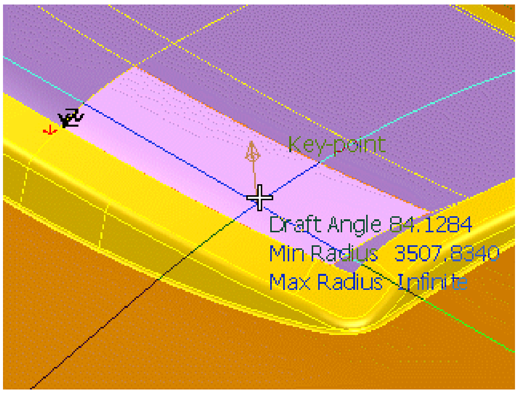
  or

- values along the T or U parametric directions

by selecting the **Curvature** type on the **Surface Analysis** page of the **Options** dialog. You can also display the curvature as either radius or curvature values by selecting the relevant **Display As** option also on the **Surface Analysis** page.

Capturing surface inspection data

1. Select the required solid, surface or component. More than one object can be selected. If nothing is selected at this stage, the tool will not return any values.
2. Click **Surface Inspection**  (*Model analysis toolbar*).
3. Click in the graphics window and drag the mouse over the surface, solid or component to be analysed. Values will be displayed below the cursor.



4. Click a position on the selected surface, solid or component.
5. Hold down the left mouse button and press *Enter*. Repeat to capture data for multiple positions.
6. Click  (*Model analysis toolbar*) to turn inspection mode *OFF*.
7. Select the **Paste** option in any text editor to paste the captured data.

Data cannot be pasted until Inspection mode is turned OFF.

Filleting and blending

The following sections contain information on filleting and blending surfaces.

What is a fillet surface? (see page 188)

Creating a fillet surface (see page 190).

How do I edit a fillet surface? (see page 211)

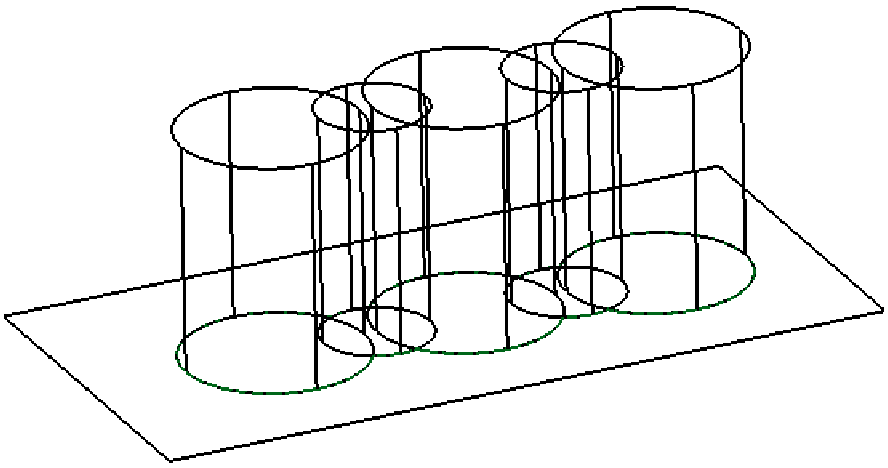
How do I edit a corner fillet? (see page 211)

Creating a blend surface (see page 214).

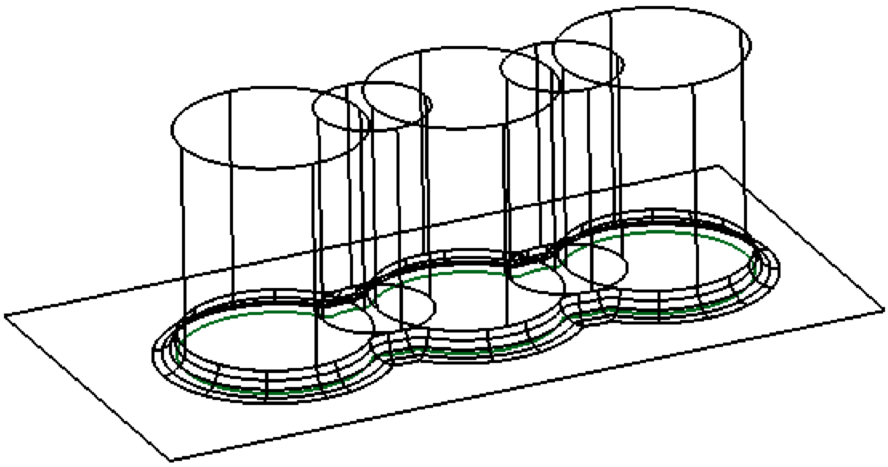
What is filleting?

Filleting is used mainly to create a smooth join between two or more surfaces.

Filleting allows you to take a selection of surfaces and create fillets between them.

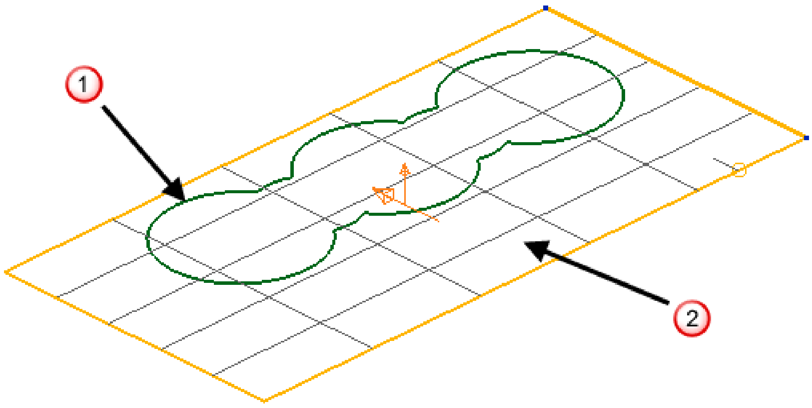


In your model, you may see two distinct sets of surfaces where you wish to create a fillet. PowerSHAPE allows you to create a fillet between these two lists.

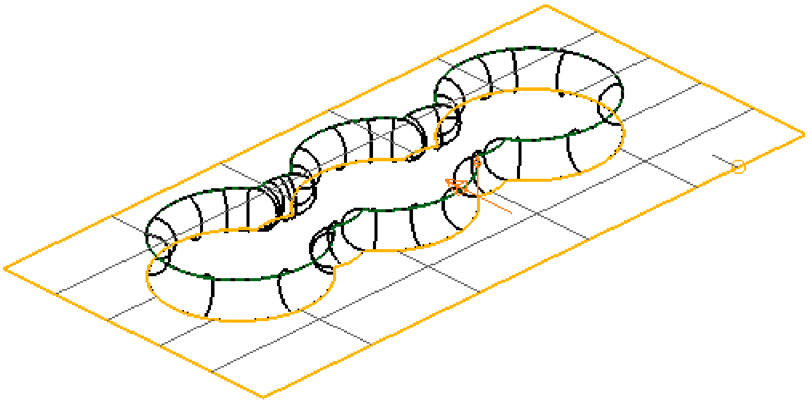


This method of filleting works faster than the more general case.

You can also create a fillet between a single wireframe object 1 and a set of surfaces 2. This is illustrated below.

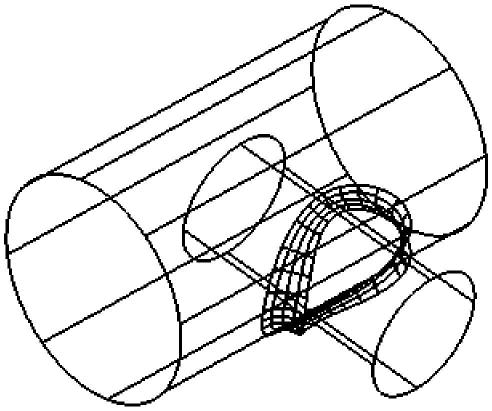


The fillet, created between the surface and the curve, is shown below:



What is a fillet surface?

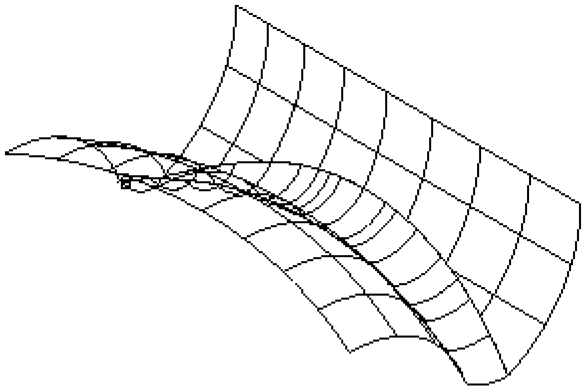
If you think of the indented shape which would result from rolling a ball around the intersection between two surfaces (that is, a circular arc of the same radius as the ball), you get a good idea of what a fillet looks like. PowerSHAPE can produce a constant-radius fillet surface between any pair of surfaces which intersect.



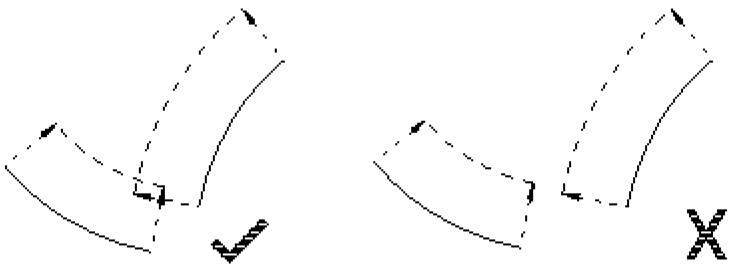
A typical fillet surface between two intersecting surfaces.

A constant-radius fillet can therefore be machined in a single pass using a ball-nosed cutter of the same radius as itself.

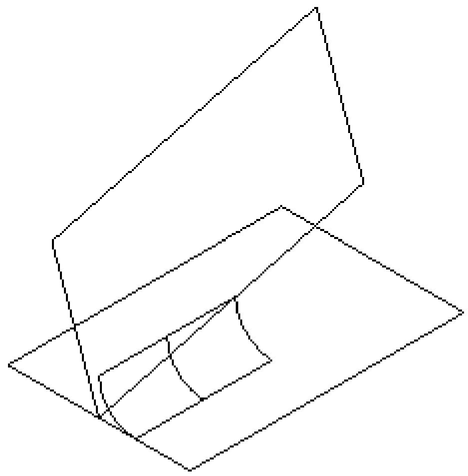
A fillet surface will follow both surfaces exactly (within tolerance), even if each is doubly curved (see figure below).



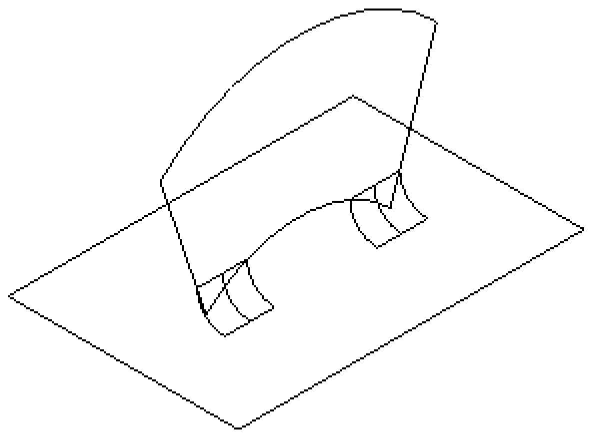
You can even produce a fillet when two surfaces do not nominally intersect - the only condition is that, if offset by the fillet radius, they would intersect (see figure below).



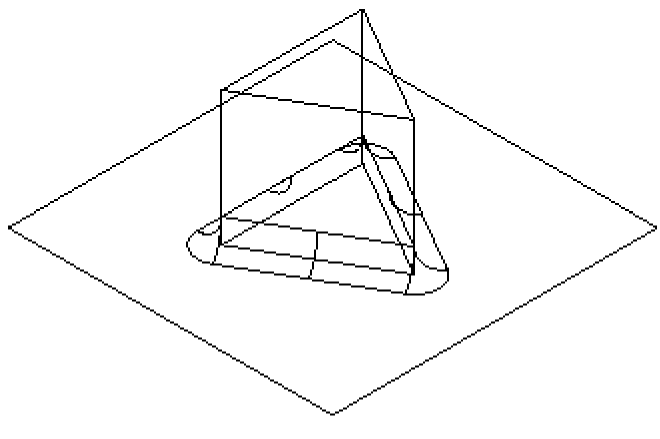
In the case of two surfaces which are divergent, the fillet will be produced between the portions of the two surfaces which do intersect when offset by the fillet radius (see figure below).



Other fillets of this type are shown below.

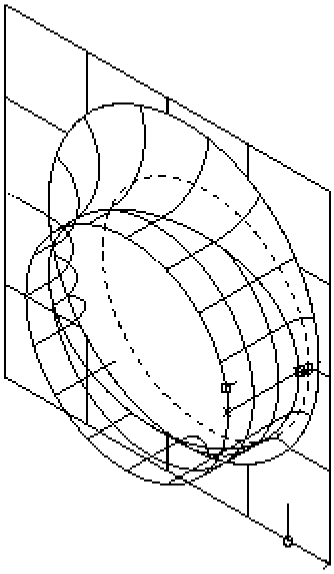


If the intersection line between two surfaces contains discontinuities (for example, as a result of sharp corners, or because some of the inside radii are smaller than the fillet's radius), PowerSHAPE will construct a fillet from each unbroken portion of line. In other words, it will produce a series of fillets. For example, the fillet illustrated below incorporates six separate fillet surfaces, one for each corner and one for each side to create a single fillet surface.



Variable radius fillets

A variable radius fillet may be created with different radius values at specified points.



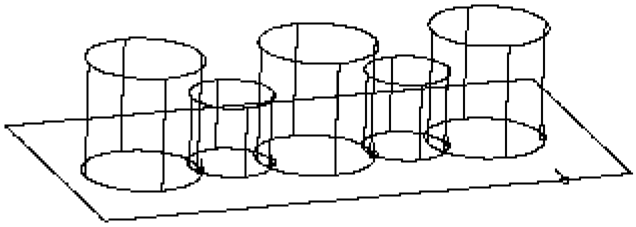
As you can see the variable fillet is used to good effect here. A constant fillet would only have allowed the smallest radius to be used along the whole length of the fillet, but here we specified a larger radius on one side to improve strength and avoid run-out of the fillet on the narrower side.

Creating a fillet surface

You can create fillet surfaces between a set of surfaces as follows:

- Along creases of surfaces. For further details see Creating a fillet along creases of surfaces (see page 204).
- Between two lists of surfaces. For further details see Creating a fillet between two list of surfaces (see page 205).
- Between surfaces and a 3D curve. For further details see Creating a fillet between surfaces and a 3D curve (see page 207).
- A variable fillet between curves surfaces. For further details see Creating a variable fillet between surfaces and a curve. (see page 209)

The following surfaces are used to illustrate filleting.

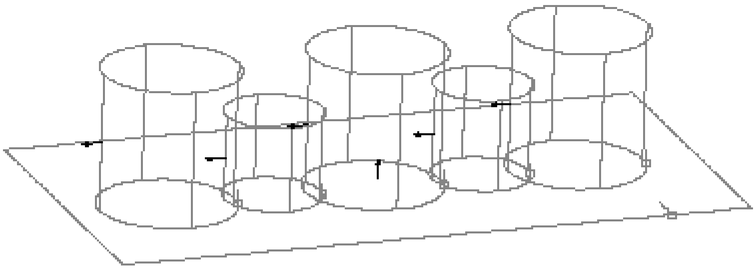


1 Click  (*Surface toolbar*).

2 Select at least two surfaces in your model.

The surface normal arrows of the selected surfaces are displayed. Also, ✘ next to the **Primary Surfaces** changes to ✔ in the **Selection** part of the dialog.

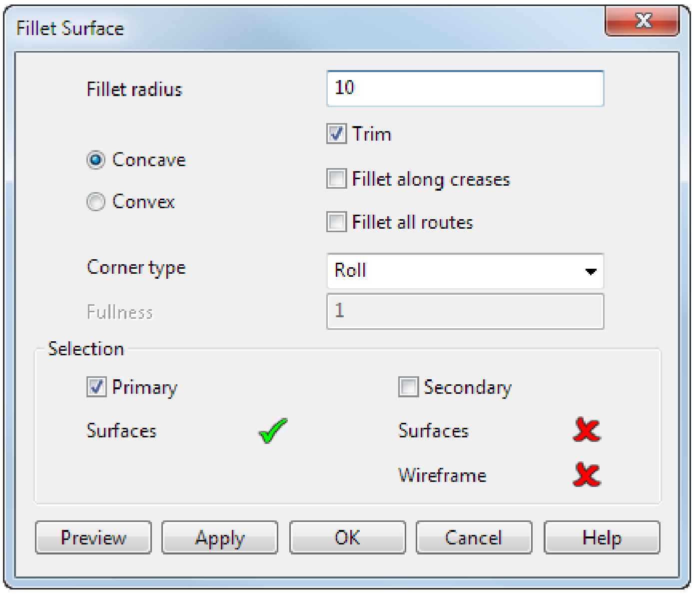
In the example shown below, all the surfaces are selected.



3 Use the Fillet surface dialog (see page 191) to provide initial data to create a fillet surface.

Fillet Surface dialog

Use this dialog to provide initial data to create a fillet surface.



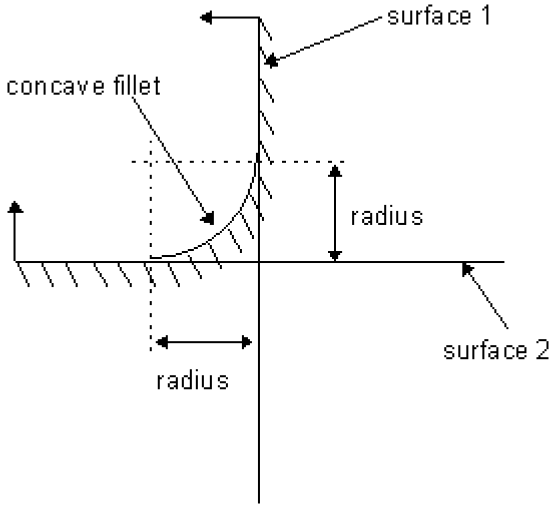
Fillet radius — This is the nominal radius of the fillet. Generally, set the radius to the smallest radius on the fillet. If the fillet runs around a corner, try a radius approximately equal to the fillet radius in the corner.

Concave/**Convex** — Determines how the fillet is created between the two surfaces.

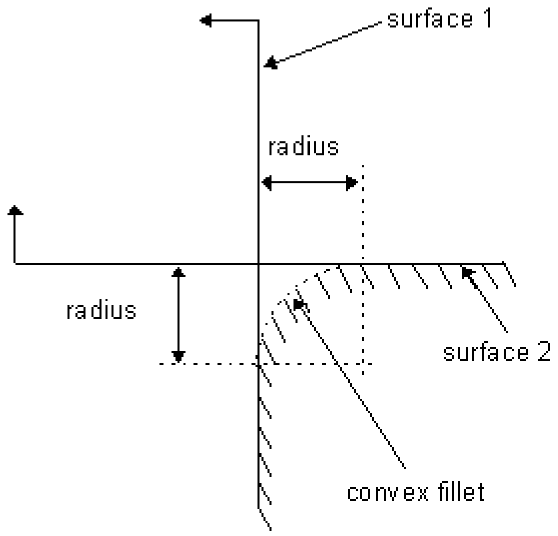
A fillet produced between two surfaces depends on:

- whether **Concave** or **Convex** is selected
- the direction of the surface normals on both surfaces

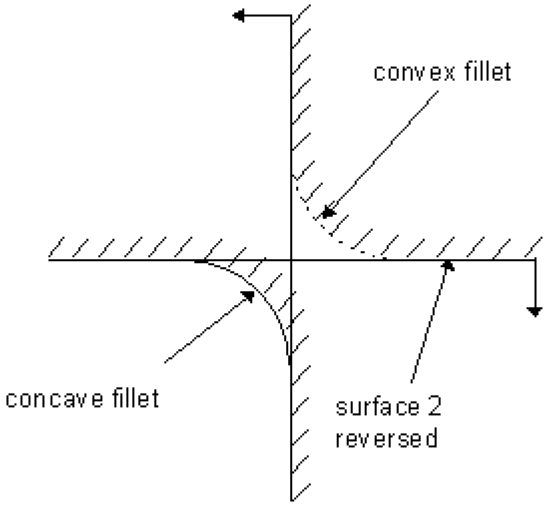
The example below shows a fillet solution between two surfaces using the **Concave** option. The arrows pointing out from the surfaces indicate the surface normals.



The example below shows a fillet solution between two surfaces using the **Convex** option.The arrows pointing out from the surfaces indicate the surface normals.



To obtain fillets in the bottom left and top right, one of the surfaces must be reversed, as shown in the example below. If you wish to change the direction of the surface normal of a surface, click the surface normal arrow.



You should always keep the surface normal pointing out from the material. This ensures surfaces are correctly combined into solids.

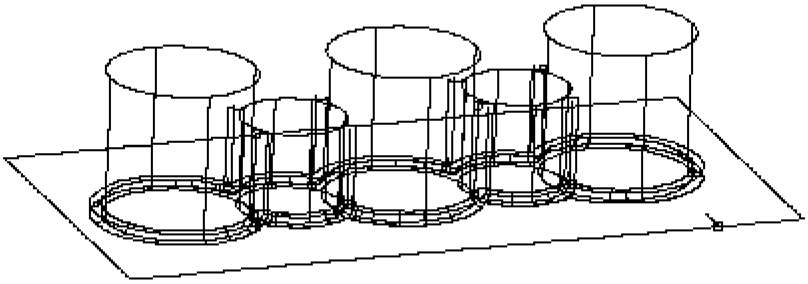
Therefore, for any two surfaces, four possible fillet solutions can be obtained.

Trim — Select this option to trim surfaces to the fillet surface when the fillet is generated. If deselected, you can still generate the pcurves and not the trim boundaries by turning on the **Keep pcurves generated by filleting** option on the **Surfaces** page of the **Options** dialog (this dialog is displayed by selecting **Options** from the **Tools** menu).

For further details on pcurves and trim boundaries, see Trimming surfaces (see page 272).

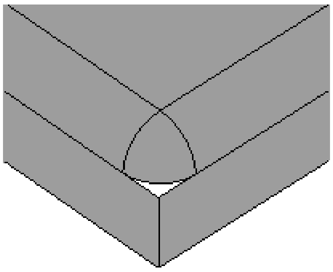
Fillet along creases — If selected, you can fillet along creases in the selected surfaces. For further details, see Creating a fillet along creases of surfaces (see page 204) .

Fillet all routes — If selected, fillets are created along all the fillet routes between the selected surfaces.

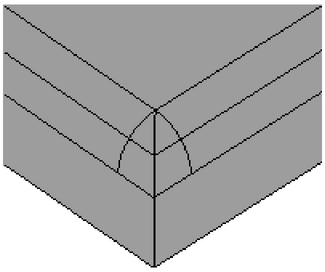


Corner Type — You can select the corners on the fillet as one of the following options:

Roll — Creates a round corner.

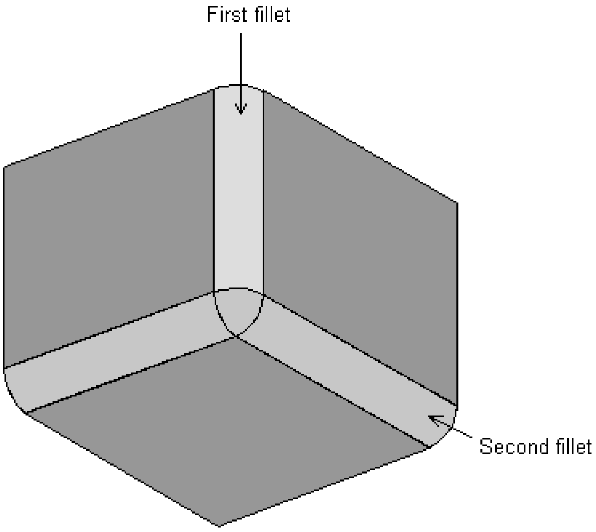


Mitre — Creates a sharp corner.

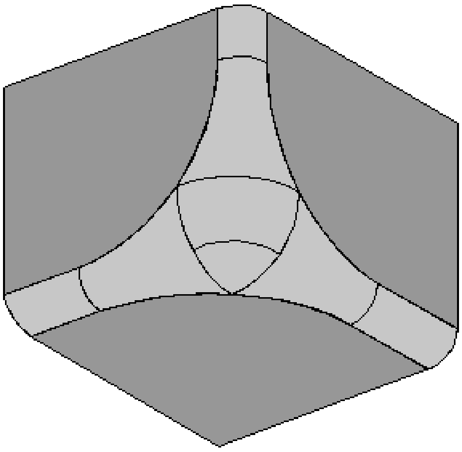


Round — This uses the **Fullness** value to make rounder corner fillets with branches.

When you click **OK** or **Cancel** on the Select Fillet Route dialog (see page 197), PowerSHAPE searches for any corner fillets as shown below.



It makes them rounder using the **Fullness** value.



Selection — You can add objects to either the primary or secondary selections. The primary selection can only contain surfaces, whereas the secondary selection can contain either a set of surfaces or a single wireframe object.

Objects in Primary selection	Objects in Secondary selection	What PowerSHAPE does with the selection
At least two surfaces	Empty	Fillet route is calculated between all surfaces.

Empty	At least two surfaces	Fillet route is calculated between all surfaces.
At least one surface	Empty	Fillet route is calculated between all surfaces and along the creases of the selected surfaces, providing **Fillet along creases** is ON. For further details, see Creating a fillet along creases of surfaces (see page 204).
Empty	At least one surface	Fillet route is calculated between all surfaces and along the creases of the selected surfaces, providing **Fillet along creases** is ON. For further details, see Creating a fillet along creases of surfaces (see page 204).
Surfaces	Surfaces	Fillet route is calculated between the two lists of surfaces. For further details, see Creating a fillet between two list of surfaces (see page 205).
Surfaces	A single wireframe object	Fillet is created between the surfaces and the wireframe object. For further details, see Creating a fillet between surfaces and a 3D curve (see page 207).

To add objects to a particular selection, select the appropriate selection toggle (**Primary** or **Secondary**) and select the objects. You can add more objects to a selection by pressing the Control key and selecting the objects. Surfaces in the primary selection are coloured differently to those in the secondary.

*You can select objects before displaying this dialog. If you select a set of surfaces and then display the dialog, the **Selection** toggle changes to **Secondary** and waits for you to add the secondary list of objects if any. If you select a wireframe before displaying the dialog, you are expected to enter the primary selection of surfaces.*

Preview — Displays the calculated fillet routes using the current settings in the dialog. You can continue to change the settings in the dialog until you are satisfied with the previewed fillet routes. Note: When creating a fillet between a set of surfaces and a wireframe, the fillet surface is displayed.

Apply — Creates the selected fillets.

OK — If the **Fillet all routes** option is on, this creates fillets along all the fillet routes. If the option is off, this calculates the fillet routes, removes the **Fillet** dialog and displays the **Select Fillet Route** dialog. For further details on how to use this dialog and continue creating the fillet, see Select Fillet Route dialog (see page 197).

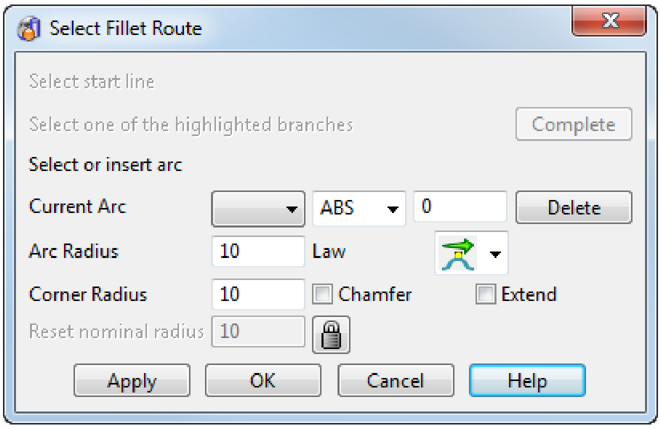
*When creating a fillet between a set of surfaces and a wireframe, the fillet surface is created and the **Select Fillet Route** dialog is not displayed.*

Cancel — Removes the dialog from the screen and does not continue creating the fillet.

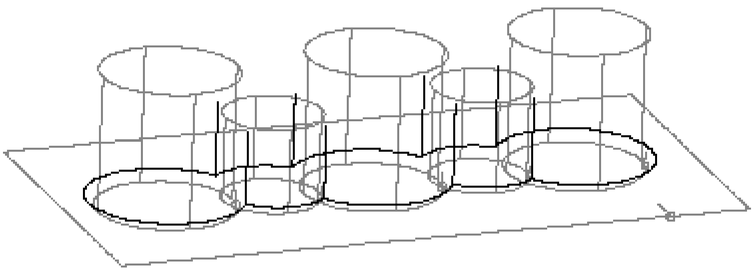
Select Fillet Route dialog

Use this dialog to:

- select the fillet routes.
- control the shape of the fillet.



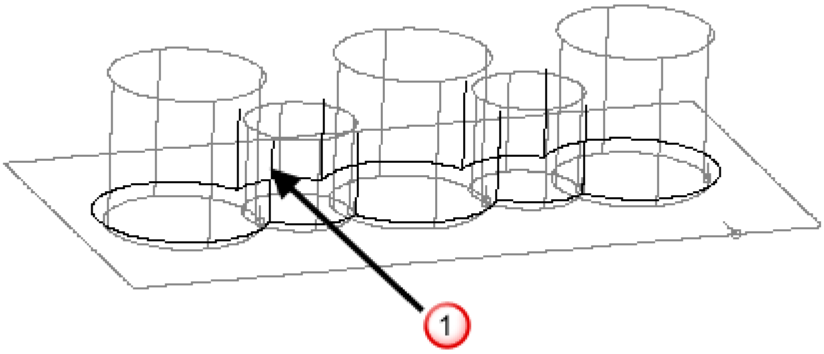
When this dialog is displayed, candidate lines display to help you determine the fillet route.



The candidate lines are the spines of the possible fillets. These lines are selected to create fillets to your requirements.

Selecting the fillet routes

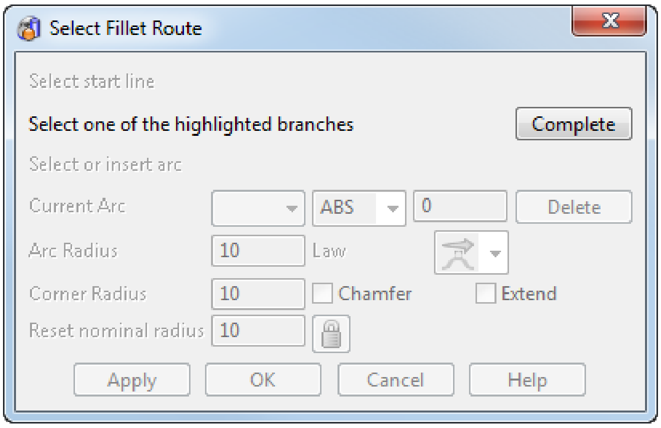
To start creating a fillet, select a candidate line as the start line 1.



Once a start line is selected, PowerSHAPE tracks the next line which joins the end of the start line. Once this line is found, it continues to track the joining lines until the end of the current route or a branch is reached.

If a single route is possible, it is automatically detected.

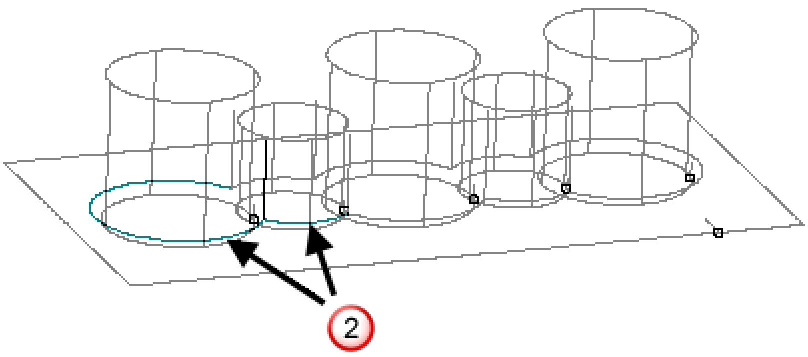
If a branch is reached, the **Select Fillet Route** dialog highlights this:



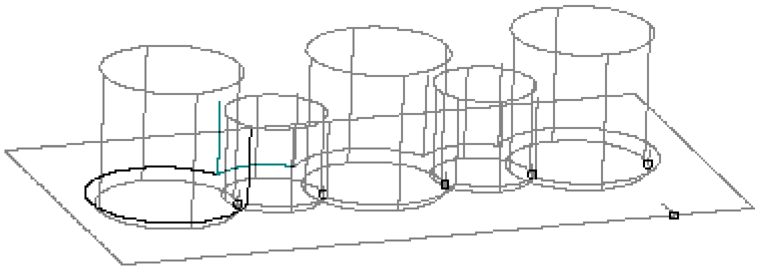
You now have two choices:

- To stop selecting candidate lines, click **Complete** on the dialog.
- To continue selecting candidate lines, click one of the branches highlighted.

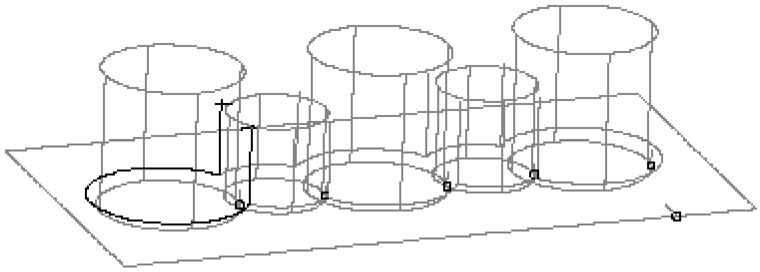
In the example below, two branch candidate lines 2 are displayed.



We have selected the branch candidate line on the left. This leads the route to reach another branch line as shown below.



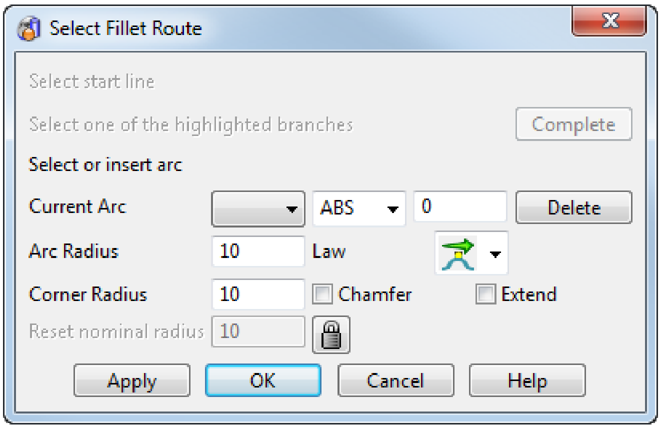
To complete the first route, the vertical branch is selected.



You can now control the shape of the fillet (see page 199).

Controlling the shape of the fillet

Once a route is complete, the **Select Fillet Route** dialog highlights the fillet arc editing features.



On the ends of the fillet route, arcs appear which represent the radius values at those points. These values are the nominal radius as set in the **Fillet** dialog.

For a closed route, no arcs are displayed.

Current Arc — This has two drop-down lists and a text box.

The first drop-down list displays the selected fillet arc. You can use this drop-down list to select existing fillet arcs. To find the number of an existing arc, simply select the arc in graphics window. Its number is displayed in this drop-down list.

Use the second drop-down list and text box to enter new fillet arcs as follows:

1 Select one of the following options from the second drop-down list to determine where to input the new fillet arc:

   ABS - A new arc is inserted along the fillet at a distance from the first fillet arc.

   REL - A fillet arc is inserted along the fillet at a distance from the selected fillet arc in the first drop-down list.

   PAR - A new arc is input at a parametric distance between two existing arcs. For example, a parametric distance of 3.5 inserts an arc half way between arcs 3 and 4.

2 Input a distance in the text box and press *Enter* to input a new fillet arc.

   The fillet arcs are also relabelled.

You can also click a position on the fillet route to enter a new arc.

If you hover the intelligent cursor over the fillet route, a distance is displayed. The option selected in the second drop-down list will determine what the value means. For example if **ABS** is selected, the value is the absolute distance along the fillet route.

The word KEY is displayed where the fillet crosses a curve on one of the selected surfaces.

Delete — This deletes the selected arc.

End arcs cannot be deleted.

Arc Radius — This displays the radius of the selected arc. You can change its value by

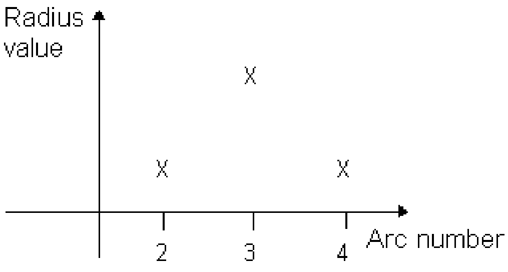
- entering a new value.
- dragging the arc on the screen to the required value.

Law — This displays the variation of the radius at the current arc position. Use the **Law** option to indicate how you want to vary the fillet radius at the selected arc.

Three variations exist:

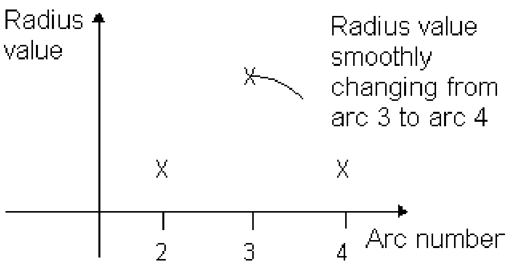
- Free
- Horizontal
- Sharp.

To explain what each of these variations mean, suppose we have the arcs as in the following diagram.

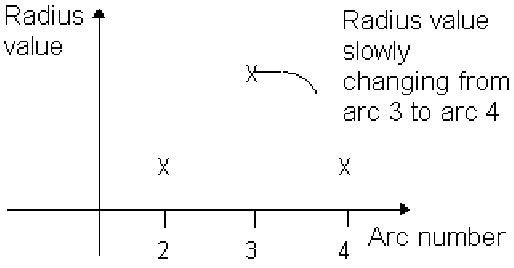


We will look at how the radius changes from arc 3 to arc 4 with respect to the three variations:

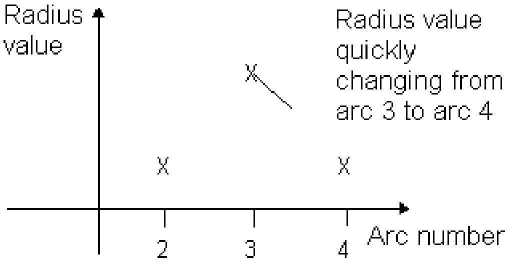
Free - the radius varies freely from arc 3 and then takes on the conditions dictated by arc 4



Horizontal - the radius remains constant for a short distance away from arc 3 and then takes on the conditions dictated by arc 4



Sharp - the radius changes size quickly as soon as it leaves arc 3 and then takes on the conditions dictated by arc 4



The **Law** option has nine selections:

- Free Free

- Free Horizontal

- Free Sharp

- Horizontal Horizontal

- Horizontal Free

- Horizontal Sharp

- Sharp Sharp

- Sharp Free

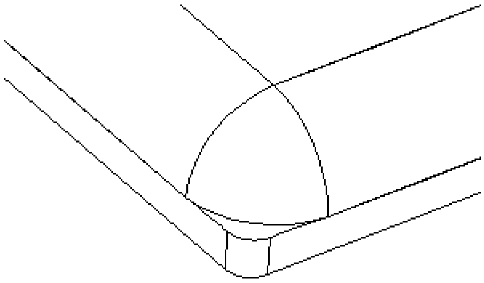
- Sharp Horizontal

The first word dictates how the radius varies as it enters an arc and the second how it leaves an arc.

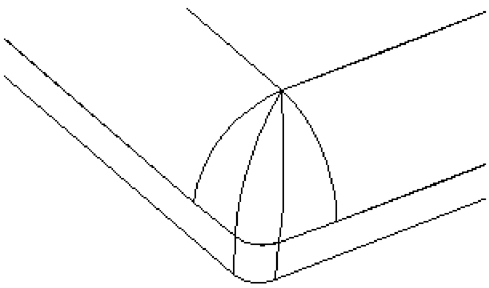
Corner Radius Type the required value in the **Corner Radius** text box.

The corner radius must not be greater than the nominal radius.

You can use the corner radius to fit a fillet into a corner. In the following figure, the fillet does not fit into the corner; its corner radius is too large.



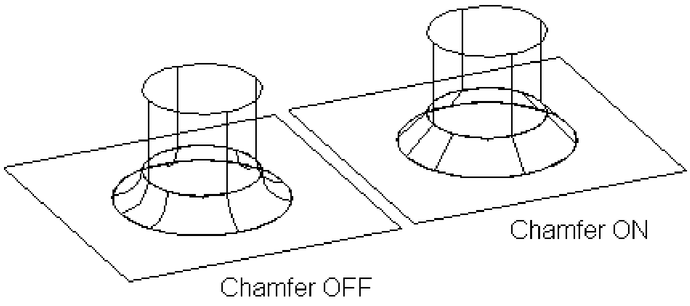
By entering a corner radius equal to the corner, the fillet fits the corner.



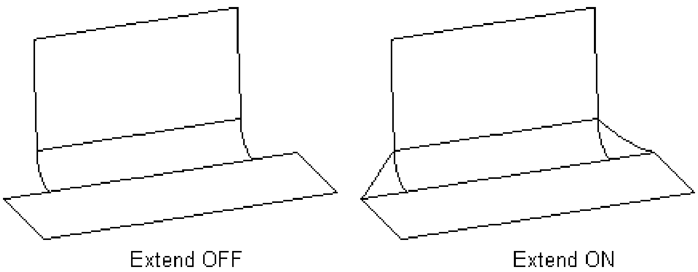
Here are some more tips on what **Corner Radius** values to use and when:

- If you create several fillets, all with the same radius, which meet in one or more concave corners, set the corner radius equal to the fillet radius. Spherical corner pieces will then be created in the corners.
- If you create several fillets, with two different radii, which meet in one or more concave corners, create the fillet with the larger radius first and set its corner radius equal to the smaller radius. Then when you create further fillets with the smaller radius, they will match the first fillet exactly in the corners.
- If you create a single fillet with longitudinal edges that follow the main surfaces closely and go into corners which are either sharp or have a smaller radius than the fillet, set the corner radius to zero.
- Once you create a corner, you can edit it to make it rounder. For further details, see How do I edit a corner fillet? (see page 211)

Chamfer — When selected, this creates straight fillets.



Extend — If **Extend** is deselected, the fillet stops when it reaches a boundary or an open edge on one of the surfaces. When selected, the fillet extends to the edge of the longer surface.



 — This locks the nominal radius. By clicking this **Lock** button to open , you can change the radius in the **Reset** text box. Once the lock is open, clicking **Apply** will cancel any fillets created and takes you back to the stage where you begin to select candidate lines.

Reset — This displays the nominal radius. It can only be changed by opening the **Lock** button.

Preview — Click this button to see the fillet path.

Apply — Once you edit the fillet to your requirements, click **Apply** on the dialog to create the fillet. Continue creating more fillet surface, if necessary, by selecting a start candidate line.

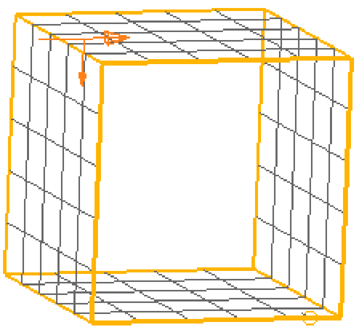
OK — Creates the required fillets and closes the dialog.

Cancel - Removes all the fillets created whilst the **Select Fillet Route** dialog was displayed and closes the dialog.

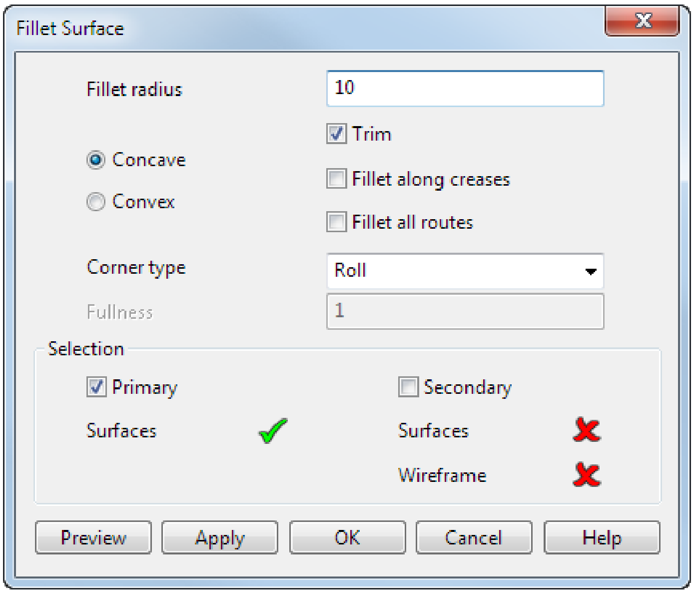
Creating a fillet along creases of surfaces

You can create fillets along creases of a surface without having to break the surface first.

We will use the following surface to show how to create a fillet along creases of a surface.



1 Click  (*Surface toolbar*).

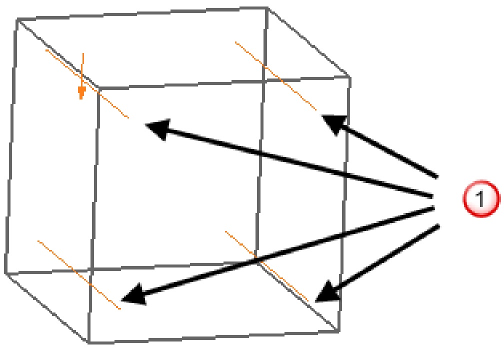


2 Fill in this dialog as described in Fillet surface dialog (see page 191).

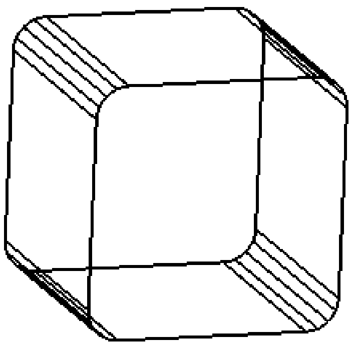
3. Select the surfaces.
4. Select the **Fillet along creases** option.
5. Make sure the surface normals are pointing in the correct directions. For further details on surface normals, see Fillet surface dialog (see page 191).
6. The single surface is treated as two surfaces at the crease.
7. Click **OK** to calculate the fillet routes, remove the **Fillet** dialog and display the Select Fillet Route dialog. (see page 197)

   The fillet routes lie along the creases of the surface 1.

   The example shown below shows four fillet routes.



8. Use the Select Fillet Route dialog (see page 197) to continue creating the fillets.

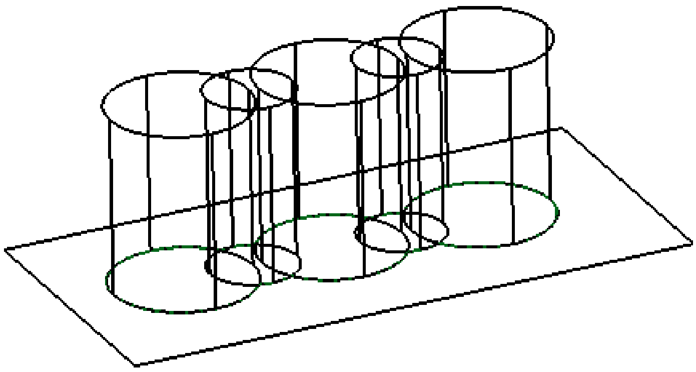


When you create the fillets along the creases of a surface, the surface breaks at the corresponding fillet route, even if you don't select the route for the fillet to lie.

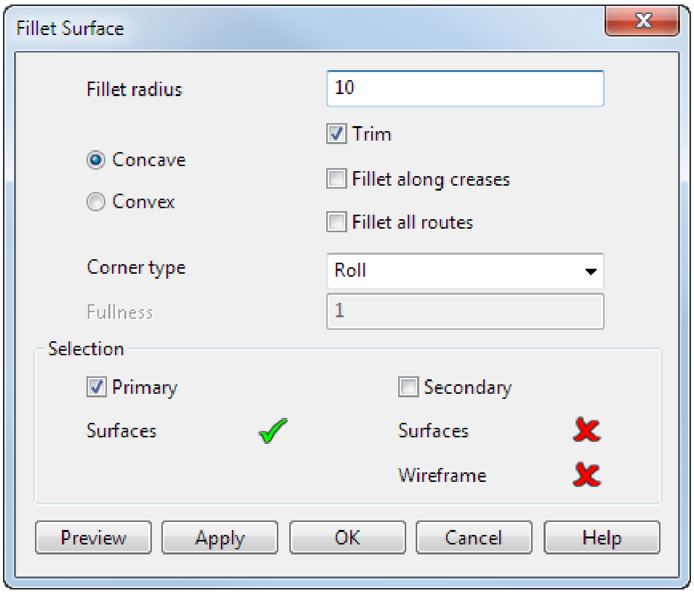
Creating a fillet between two lists of surfaces

Because you provide more information about the fillet required, PowerSHAPE creates the fillet much more quickly. As this method is very similar to creating a fillet discussed above, we only show you how to select the two lists.

The example surfaces shown below illustrate creating a fillet between two lists of surfaces. All the cylindrical surfaces will be in one list and the planar surface in the another.



1 Click  (*Surface toolbar*).



2 Fill in this dialog as described in Fillet surface dialog (see page 191).

Select the surfaces for one of the lists.

3 In our example, all the cylindrical surfaces are selected.

4 In the **Selection** part of the dialog, The cross icon changes to a tick icon.next to **Primary Surfaces**.

5 Click the **Secondary** toggle in the **Selection** part of the dialog.

6 Select the second set of surfaces.

In our case, we select the plane. Therefore, one list contains the plane and the other list all the cylindrical surfaces.

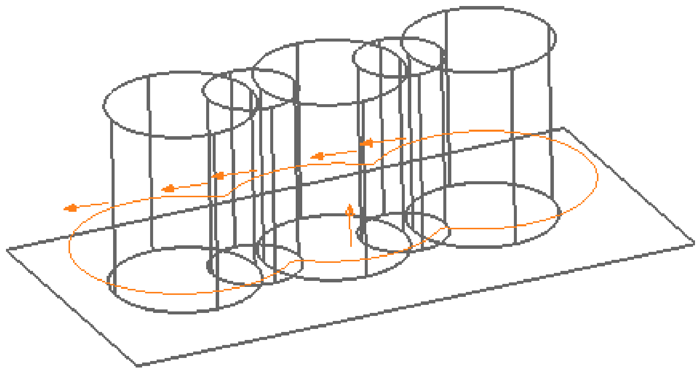
When a surface is selected, The cross icon changes to a tick icon. next to **Secondary Surface**.

If you have selected the wrong surface, simply deselect using Ctrl-click.

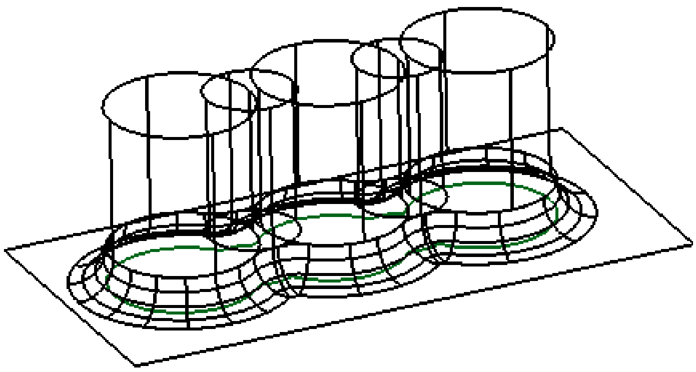
If you have no surfaces selected, then the selection is taken as a single list.

7 Click **OK** to calculate the fillet routes, remove the **Fillet** dialog and display the **Select Fillet Route** dialog.

Notice how the fillet routes separate the two lists of surfaces.

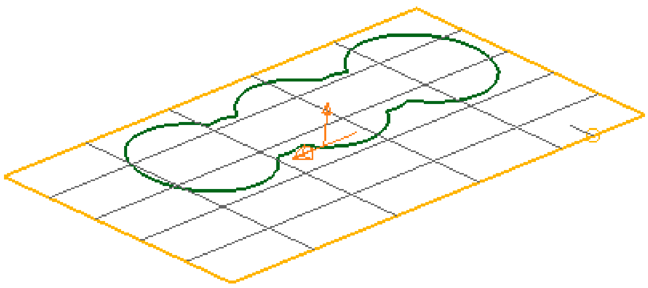


8 Use the Select Fillet Route dialog (see page 197) to continue creating the fillets.

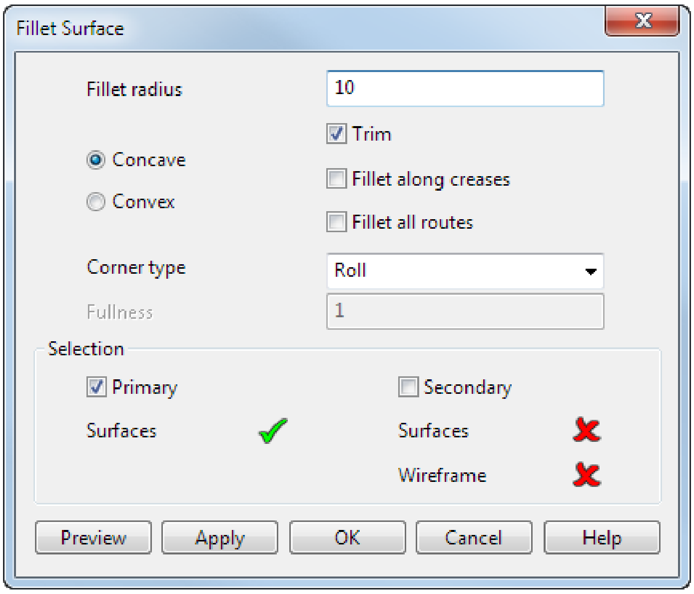


Creating a fillet between surfaces and a 3D curve

You can create a fillet between a set of surfaces and a 3D curve. The 3D curve can be any single wireframe object, for example, a line or a composite curve.



1 Click  (**surface creation** menu).



2 Fill in this dialog as described in Fillet surface dialog (see page 191).

3 Select the surfaces.

The next to the **Primary Surfaces** changes to in the **Selection** part of the dialog.

4 Click the **Secondary** toggle in the **Selection** part of the dialog.

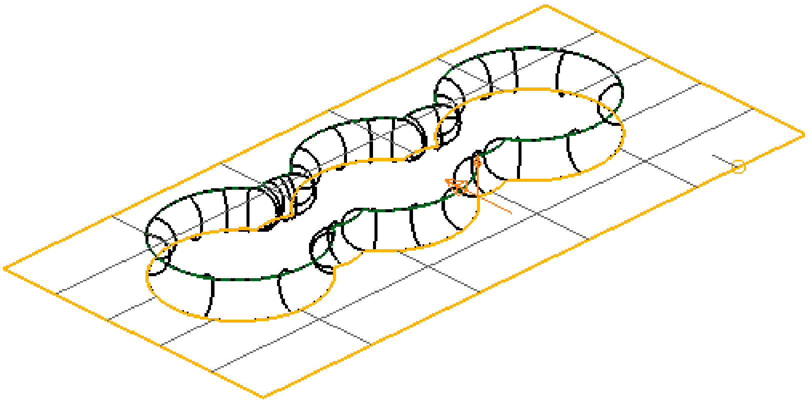
5 Select the curve.

When the curve is selected, The cross icon changes to a tick icon. next to **Secondary Wireframe**.

6 You will see an arrow on the curve. This determines which side of the curve the fillet is created. We advise you to see the fillet by clicking the **Preview** button. If the fillet is on the wrong side of the curve, click the arrow on the curve and preview the fillet again.

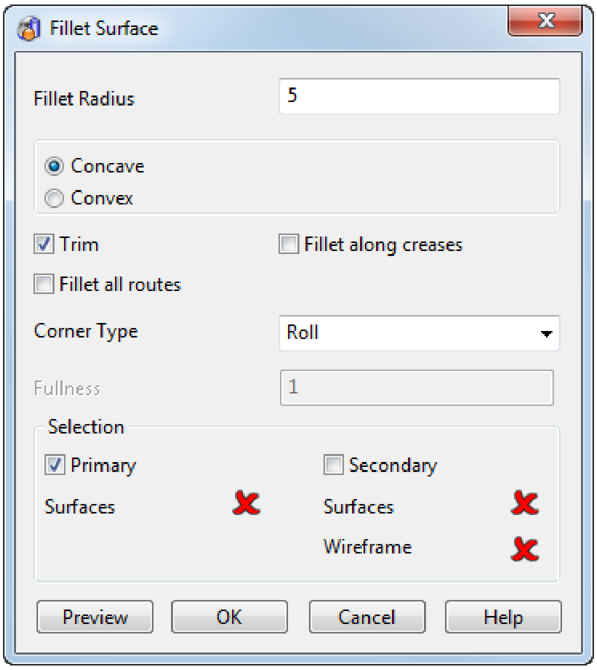
7 Click **OK** to create the required fillet.

Notice how the fillet smoothly meets the surface.



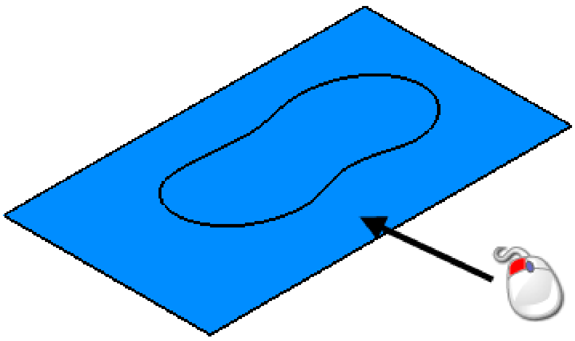
Creating a variable fillet between surfaces and a curve

1 Click **Fillet Surface**  (**surface creation** menu).



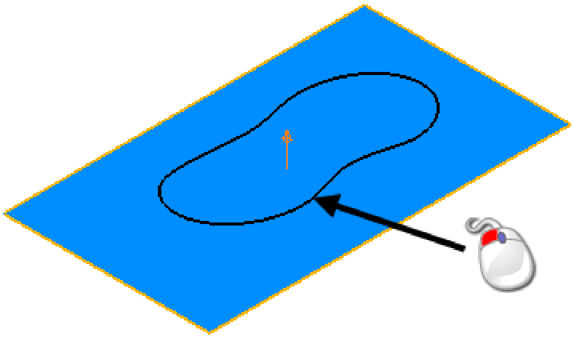
2 Fill in this dialog as described in Fillet surface dialog (see page 191).

3 Select the surface.



4 In the **Selection** part of the dialog, click the **Secondary** option

5 Select the curve.

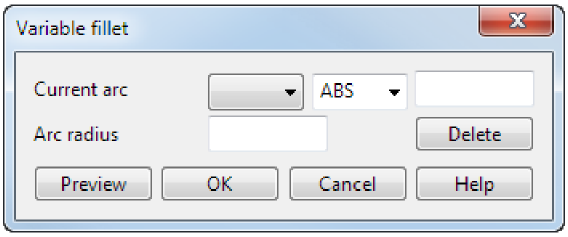


6 Click **OK**.

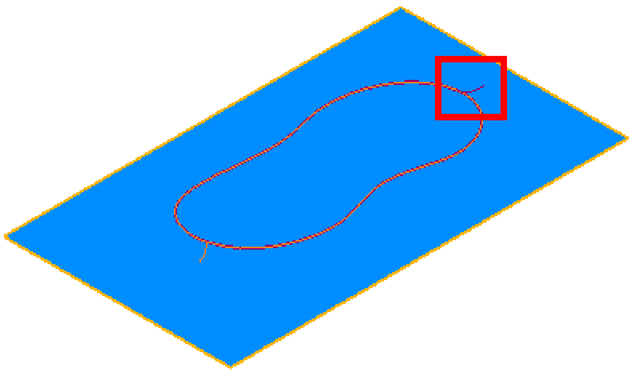
7 The Variable Fillet dialog (see page 210) is displayed, where you specify the details for the variable fillet. For further details, see Variable Fillet dialog (see page 210).

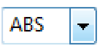
Variable Fillet dialog

Use this dialog to specify the variable fillet between a surface and a curve.



Current arc - Click on the curve where you want to create a variable fillet. A small arc is displayed on the curve, as shown in the example below. The corresponding number for the arc is added and displayed in the **Current arc** drop-down list.





 - Select one of the following options to specify the distance to be calculated for the variable fillet at the selected arc:

- **ABS** - Absolute distance
- **REL** - Relative distance

- **PAR** - Parametric distance.

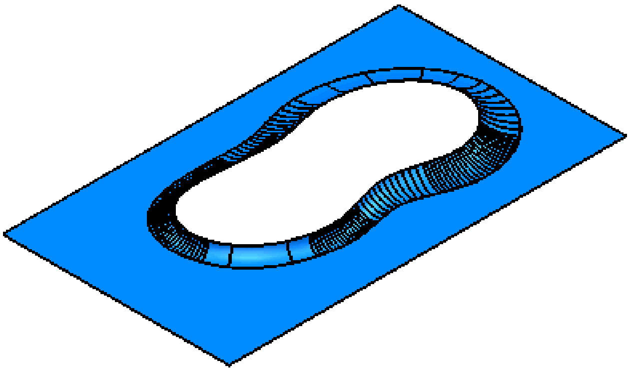
- Enter the distance for the selected arc.

Preview — Click to display a preview of the variable fillet.

Arc radius - Enter a radius for the selected arc.

Delete - Delete the selected arc.

OK - Click to create the variable fillet.



Editing a fillet surface

You edit a fillet surface in the same way as a Power Surface. For further details, see Editing Power Surfaces (see page 112).

A fillet surface is made of laterals and longitudinals, where laterals run across the fillet spine and longitudinals along it.

Editing a corner fillet

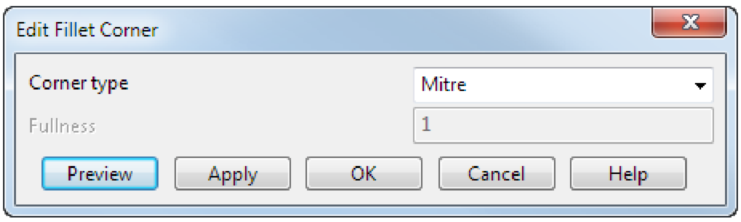
You can edit a single fillet containing a corner into a mitre or make it rounder in the corner.

For a region where many fillets meet, you can create a mitre or a smooth n-sided corner surface.

1. From the **Edit** menu, select **Fillet Corner** to display the Edit Fillet Corner dialog. (see page 211)
2. Use the Edit Fillet Corner dialog. (see page 211) to edit the corner of the fillet.

Edit Fillet Corner dialog

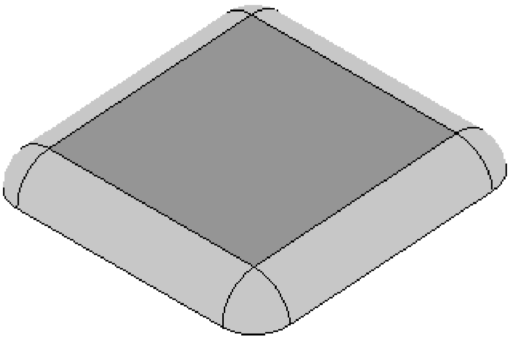
Use this dialog to edit the corner of the fillet.



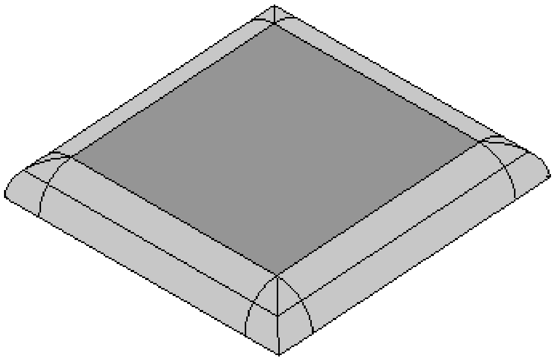
Corner type — This gives the operations you can perform on corner fillets. Select from the following options:

Mitre — This creates mitred corners on the selected fillet.

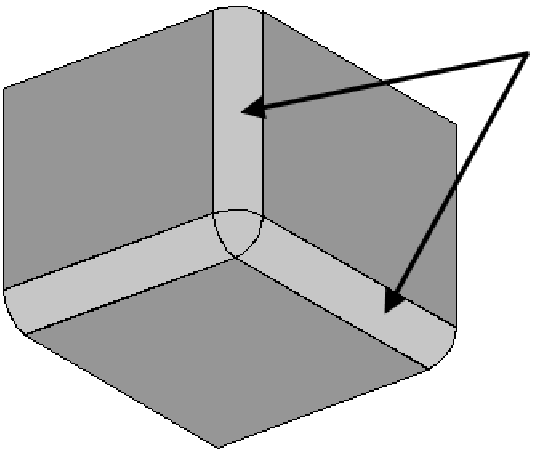
We will edit the corners on the following fillet.



The result is shown in the example below.



Round — This makes the corner fillet rounder in the corner. Select the fillets containing corners and those which branch onto the corner fillets.

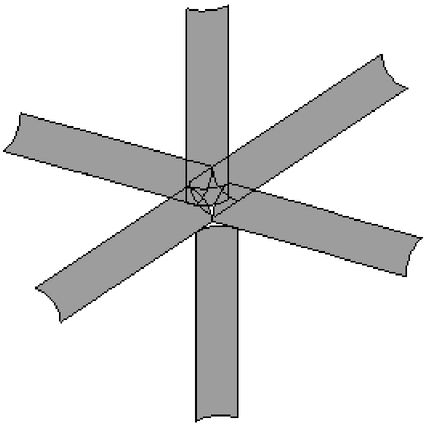


You can select multiple surfaces containing fillets with corners.

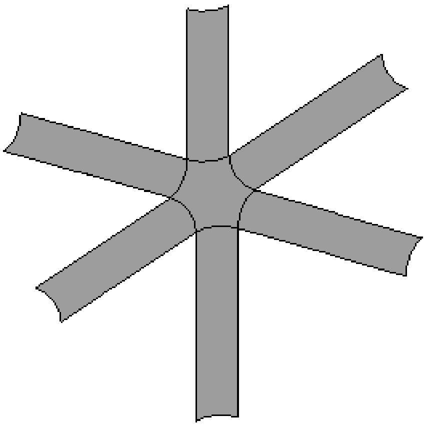
Enter a number in the **Fullness** box, if required.

N-sided corner — In a region where many fillets meet, this option trims the fillets back to each other (forming an n-sided hole) and automatically fills the hole with a smooth surface.

We will edit the corners on the following fillet.



The result is given below.



Preview — Displays the edited corner fillets using the current settings in the dialog. You can change the settings in the dialog and the selected surfaces until you are satisfied with the previewed results.

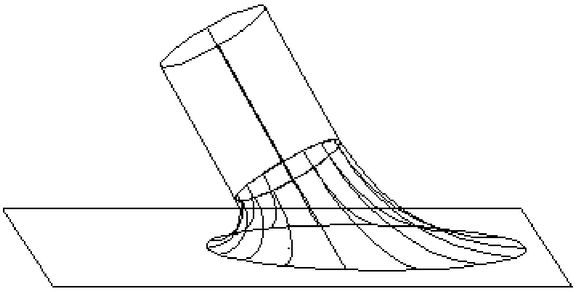
Apply — The corners are edited.

OK — The corners are edited and the dialog is removed from the screen.

Cancel — Closes the dialog without applying the corner edits.

How do I create a blend surface?

A blend surface creates a smooth joint from one surface onto a group of surfaces. It tangentially leaves the one surface and joins the group of surfaces to ensure a smooth transition between the surfaces.

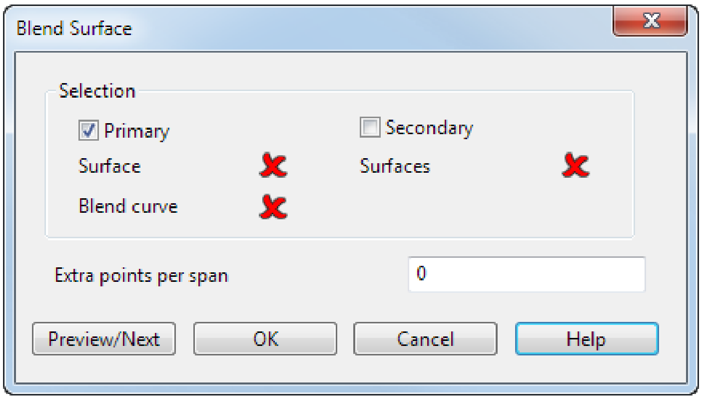


1 Click  (*Surface toolbar*) to display the Blend Surface dialog (see page 214).

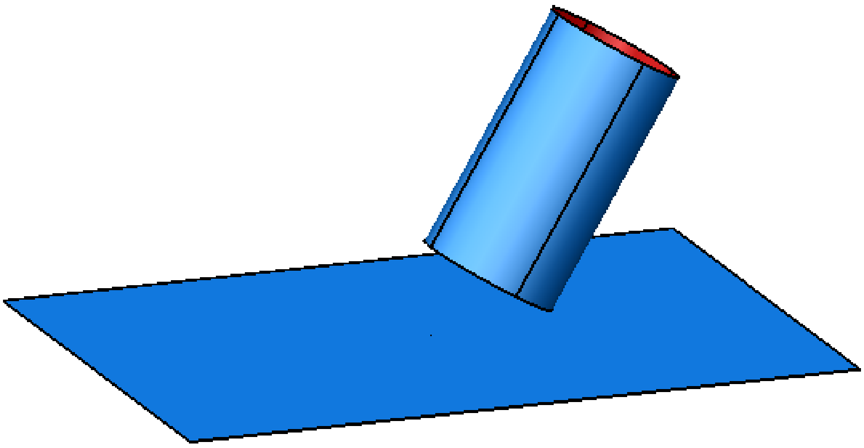
2 Use the Blend Surface dialog (see page 214) to create the blend surface.

Blend Surface dialog

Use this dialog to create a blend surface from one surface onto a group of surfaces.



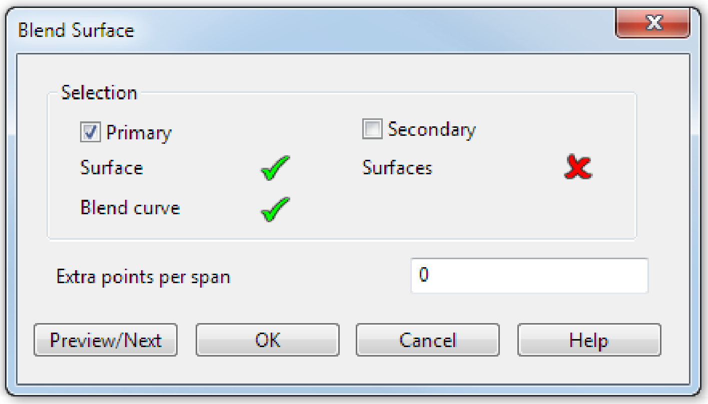
Consider the following surfaces.

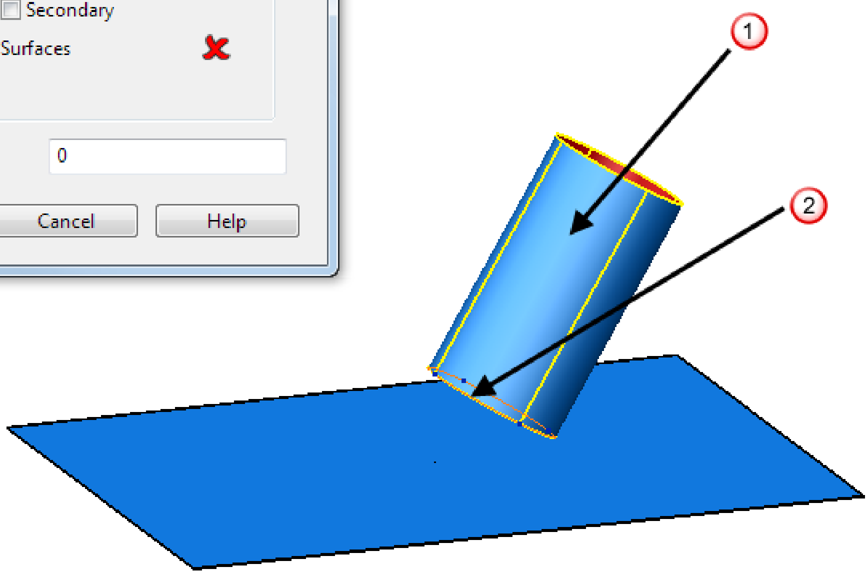


Selection

Select the surfaces to blend.

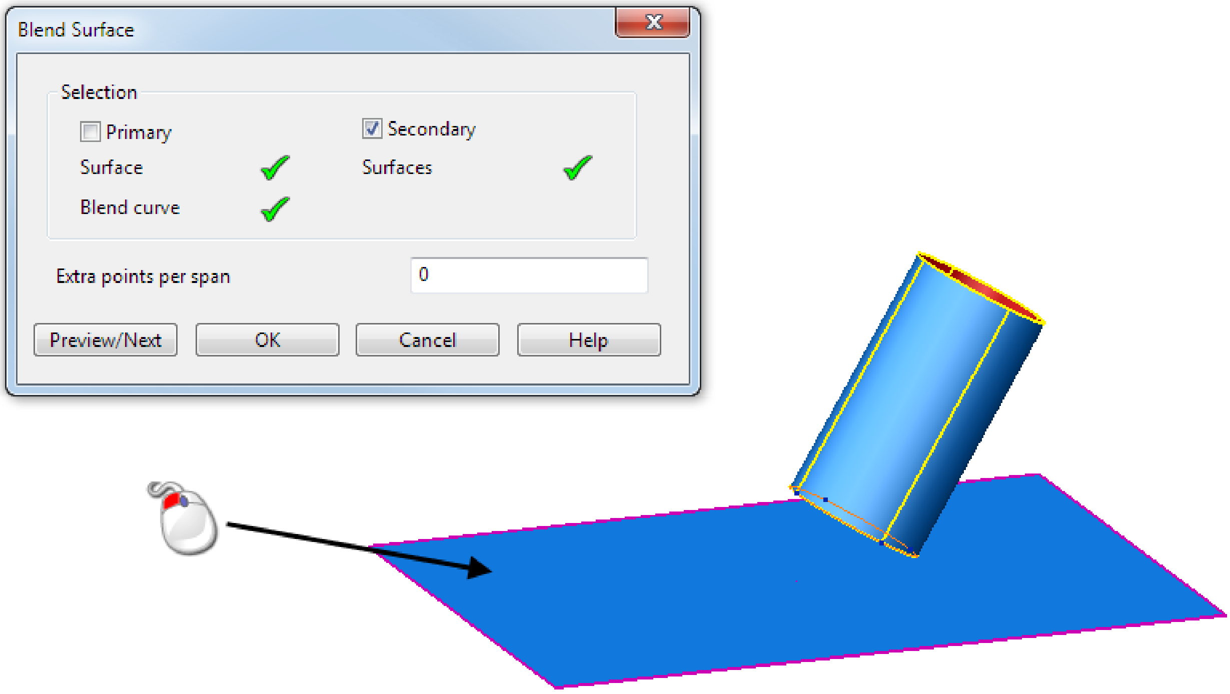
Primary - This is used to select the surface to blend from and one of its curves from where the blend surface starts. Select the surface you want to blend from 1 and a curve 2.





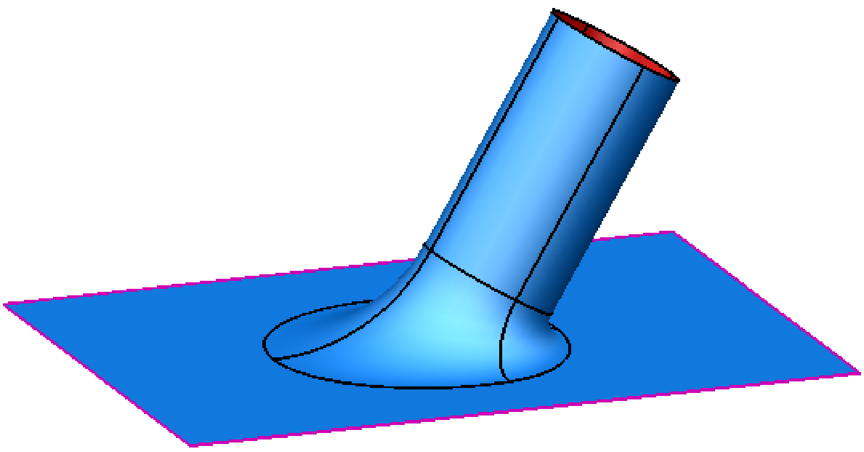
The curve can be any curve on the surface, not necessarily the end curve.

Secondary - This is used to select the group of surfaces to blend onto. Select this option, select one or more surfaces and The cross icon changes to a tick icon..

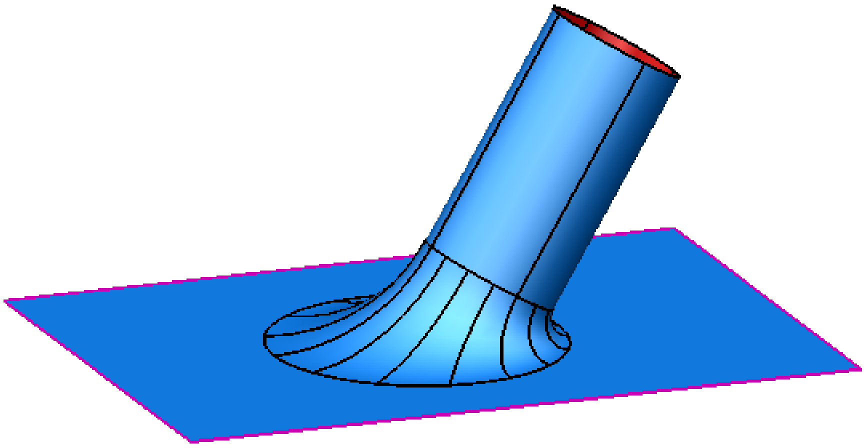


Extra points per span - The blend curve defines one of the end curves on the blend surface. You can add extra points on each span of this end curve.

In the example below, no extra points per span are added to the blend.



In the next example, three extra points per span are added to the blend.



With no extra points per span, the blend surface is created to tolerance and extra points may be added. If you specify a value other than zero, you restrict the number of points per span to that value and the blend surface may not be to tolerance.

The number of points chosen should reflect the complexity of the surfaces being blended on to. If the surfaces are smooth, you'll get a good match with relatively few points. On the other hand, if the surfaces are convoluted, corrugated, or lumpy, you'll need a correspondingly greater number of points to achieve an adequate surface match.

Preview/Next - This displays the blend surface as defined on the dialog. If you change the selection of objects or the number of **Extra points per span** and click this button, the preview of the blend surface is updated. If you click this button without making any changes, the next solution of blend surface is displayed. This blend surface tangentially leaves the blend curve in the opposite direction to the first solution.

OK - This creates the blend surface and closes the dialog.

Cancel - Closes the dialog without applying the blend.

Wrapping

The following sections provide information on wrapping.

What is wrapping? (see page 218)

Using the Wrap Wizard (see page 219).

What is wrapping?

Wrapping allows you to add extra detail, complex 3D logos and artwork onto surface or solid models by wrapping the detail onto or around the shape of the model. For example, adding artwork to standard jewellery molds, adding grip texture to plastic moldings or adding detail and logos to shoe soles. For areas where an electrode will be used for manufacture, the wrap detail can be applied either to the actual part or directly onto a previously extracted electrode body.

The type of wrapping strategy you use depends on the shape of the base model and how you want the relief to appear.

The following menu options start the **Wrap Wizard**:

- **Curve Unwrapping** (*Curve toolbar)* unwraps curves from surfaces and solids.
- **Curve Wrapping** (*Curve toolbar*) wraps wireframe objects onto surfaces and solids.
- **Wrap Triangles, Surfaces or a Solid** (*Surfaces toolbar*) wraps onto a triangle, surface or solid (see page 219) creating a wrap with no dependencies.
- **Create a Solid Wrap Feature** (*Solid toolbar*) wraps onto a solid creating a solid feature in the solid feature tree. This feature can be modified as required.

The **Wrap Wizard** changes automatically into the **Rewrap Wizard** depending on the stage of the wrapping process you have reached.

1 For best results begin by **unwrapping** from a surface or solid. You may have a model that already contains detail that can be unwrapped, or you may need to create the geometry first by sketching curves onto the solid or surface.

   Unwrapping gives you back the wireframe edge curves so that when you rewrap the object, the wrap is a perfect match and positioned precisely.

2 When the curves are unwrapped from the object you can add more detail using PowerSHAPE or ArtCAM.

Designing with PowerSHAPE - If necessary, use the tools on the **Mesh Editing** toolbar to fix the mesh.

You can smooth triangle symbols in PowerSHAPE to improve the appearance of the faceted triangle mesh.

Designing with ArtCAM - export the unwrapped wireframe as a .PIC file into ArtCAM. Export the completed design back into PowerSHAPE as a DMT file ready for wrapping.

For best results, the triangle mesh of an imported DMT file needs to be a watertight C (boat - open topped) rather than a O (submarine - closed shape). If it is a O (closed shape) the target surface will not be trimmed correctly.

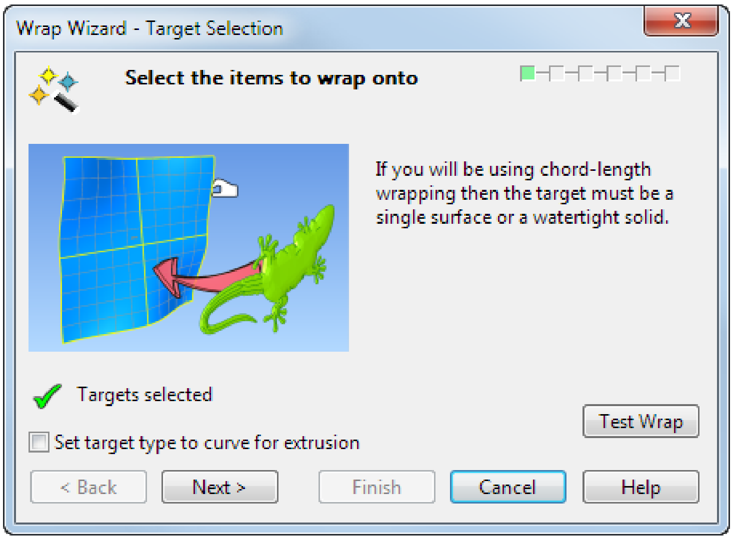
3 When the wrap design is completed, you **Rewrap** onto the target object.

Using the Wrap Wizard

1 Select **Wrap Triangles** (*surface toolbar*) to start the **Wrap Wizard**.

If the model has been previously unwrapped, the wrap wizard detects this and turns into the **Rewrap Wizard**.

2 Use the **Target Selection** page of the **Wrap Wizard** to select the target surfaces and/or solids for wrap.



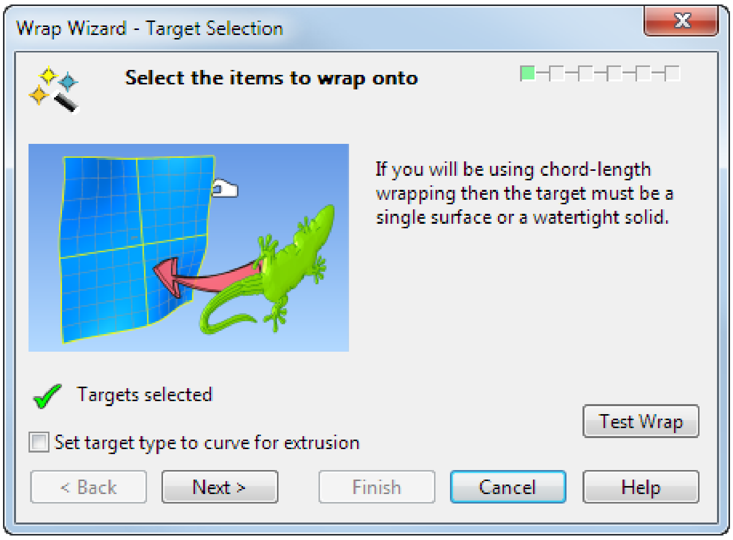
3 Use the **Test Wrap** option to create a test wrap for the selected wireframe and target object to let you see the viability of the results before proceeding through the **Wrap Wizard**.

4 Work through the steps of the **Wrap Wizard** by using **Next** to move to subsequent pages of the wizard.

Wrap Wizard - Target Selection

Use this page of the **Wrap Wizard** to select the target surfaces and/or solids for the wrap.

*If you intend to select the **Chord** wrapping method on the Wrap method (see page 223) page of the **Wrap wizard**, the target selected must be a single surface or a watertight solid.*



Targets selected - When the target for the wrap is selected, The cross icon changes to a tick icon..

Set target type to curve for extrusion - This option is used for wrapping curves more easily in shoe design where you have a curve representing a cross section of the shoe sole instead of an existing sole surface. Any type of curve can be used as long as it is a single, planar curve.

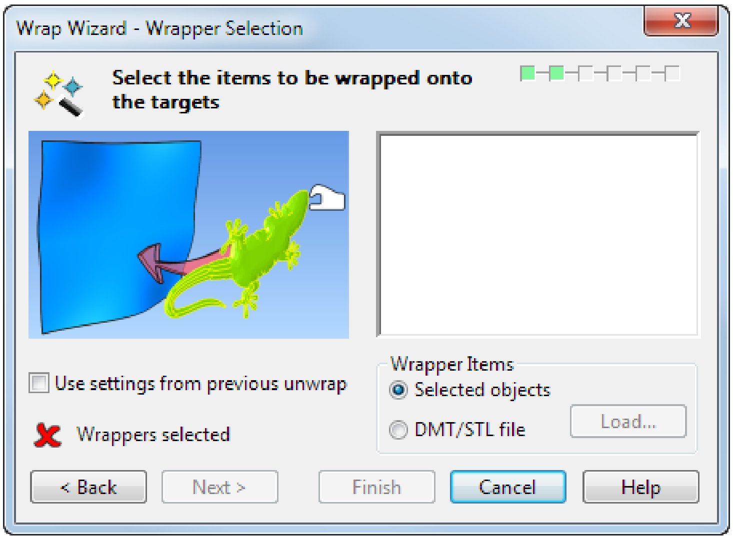
Select this option on the dialog then select the cross section curve as the target. An extruded surface is created automatically from the cross section curve which is then used as the target for wrapping.

If you wish to wrap curves directly onto an existing shoe sole surface, leave this option blank and select the shoe sole surface as the target. For further details, see How do I create a shoe sole wrap? in the Wireframe modelling manual.

Test Wrap - Select to create a test wrap for the selected wireframe and target object to see the viability of the results before proceeding through the **Wrap Wizard**. This option becomes available when the target is selected.

Wrap Wizard - Wrapper Selection

Use this page of the **Wrap Wizard** to select one or more surfaces, a solid, or a symbol to be wrapped onto an object.



Wrapper Items

- **Selected objects** — Select this option and click on the required items displayed in the graphics window.

  The wrappers must be:

  - one solid
  - all surfaces
  - all symbols
  - all wireframe.

  The selection will be displayed in the preview window of the **Wrapper Selection** page of the **Wrap Wizard**.

- **DMT/STL File** — Select this option to load a **DMT (.dmt)** file or a **STL (.stl)** file for wrapping. The **Load DMT File** dialog is displayed. where you select the file you wish to load. The image of the contents of the **DMT** or **STL** file or symbol is displayed in the preview window on the **Wrapper Selection** page.

For best results, the selection should be an open shape, with its outline lying in a plane.

Load — Select this option to load another DMT file when a DMT file has already been loaded. The **Load DMT File** dialog is displayed.

Use settings from previous unwrap — This option becomes available and is selected if the selected surface or solid has been previously unwrapped using **Curve Unwrapping** .

Wrappers selected — The cross icon changes to a tick icon. when the wrapper is selected.

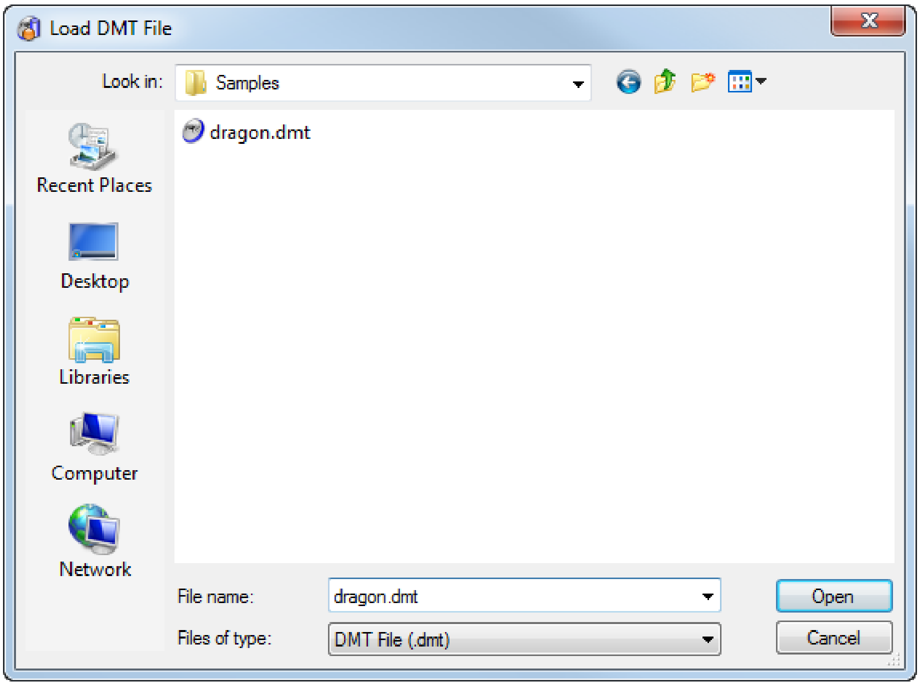
Next — Different pages of the wizard are displayed depending on which method is selected:

Selected Objects — the **Reference Plane** page is displayed.

DMT/STL File — the **Wrap Method** page is displayed.

Load DMT File dialog

Locate and select a DMT or STL file for wrapping onto a target object.



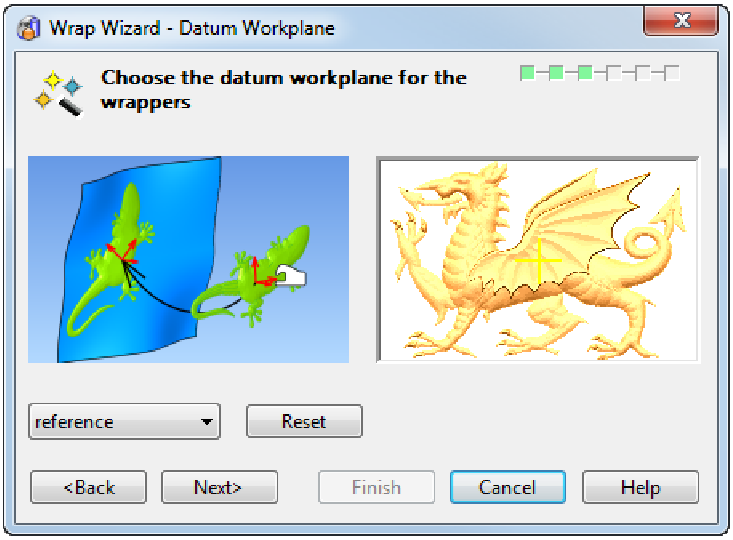
Files of type - Select either a **DMT (.dmt)** or **STL (.stl)** file type from the drop down list.

You can smooth triangle symbols in PowerSHAPE to improve the appearance of the faceted triangle mesh.

Open - Displays the selected DMT file. The image of the contents of the DMT file or symbol is displayed in the preview window of the **Wrapper Selection** page of the **Wrap Wizard**.

Wrap Wizard - Datum Workplane

Select the workplane you want to use for the wrapper.



Reference — This is a reference workplane that is created automatically at the centre of the wrapper at Z0. The wrapper will be wrapped onto the target solid or surface in a position relative to this reference workplane.

If this reference plane is not in the right place, you may need to define one explicitly with its XY plane aligned with the open face of the wrappers.

World — Refers to the world workplane. The wrapper will be wrapped onto the target object in a position relative to the world workplane.

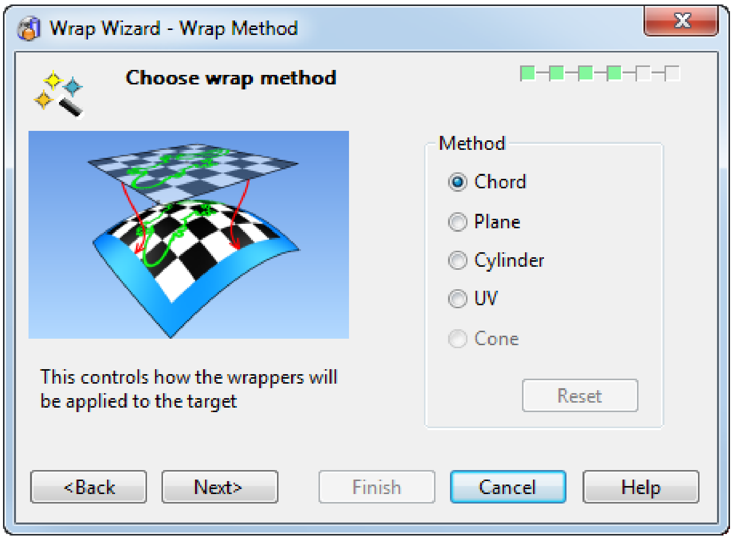
Reset — Click to reset the reference workplane to align with the workplane of the selection.

Next — Displays the **Wrap Method** page of the **Wrap Wizard**.

Wrap Wizard - Wrap Method

Use this page of the **Wrap Wizard** to select the method for applying wrappers to target objects.

A wrapper is the item selected for wrapping onto a target object.

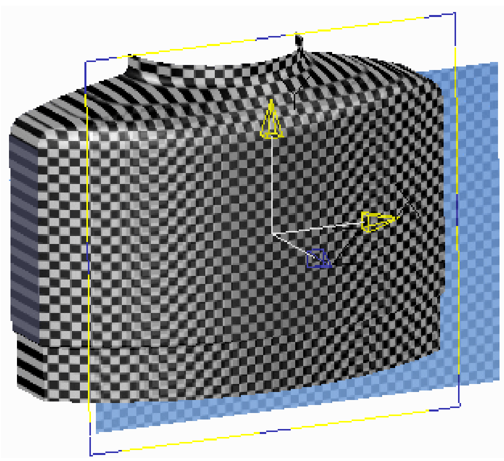


Wrap Method: Choose one of the following methods:

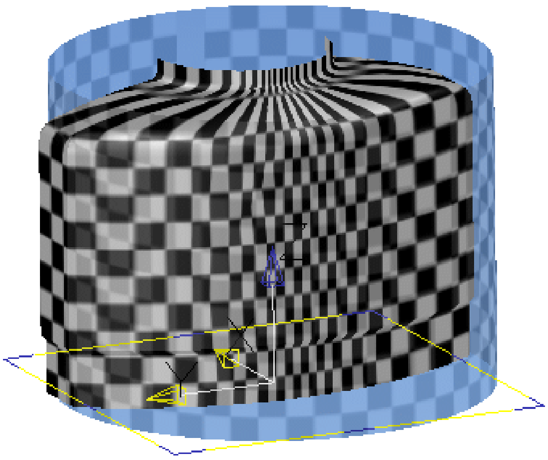
Chord — This method wraps wireframe triangles onto the target object, minimising distortion.

You define how the **Chord** wrapper is applied to the target object on the **Chord Length Wrap Map Creation** page of the **Wrap Wizard**.

Plane — This projects the wrapper from a plane onto the target object. You can modify the plane in the graphics window. A chequerboard visual effect is displayed on the target object to show the effect of the wrap method.



Cylinder — This projects the wrapper from a cylinder onto the target object. You can modify the cylinder in the graphics window. A chequerboard visual effect is displayed on the target object to show the effect of the wrap method.



UV — Uses the U, V (or T, U) parametric spacing of the surface or solid. A chequerboard visual effect of the result is displayed on the target object.

Reset — This reverts the settings back to the default values.

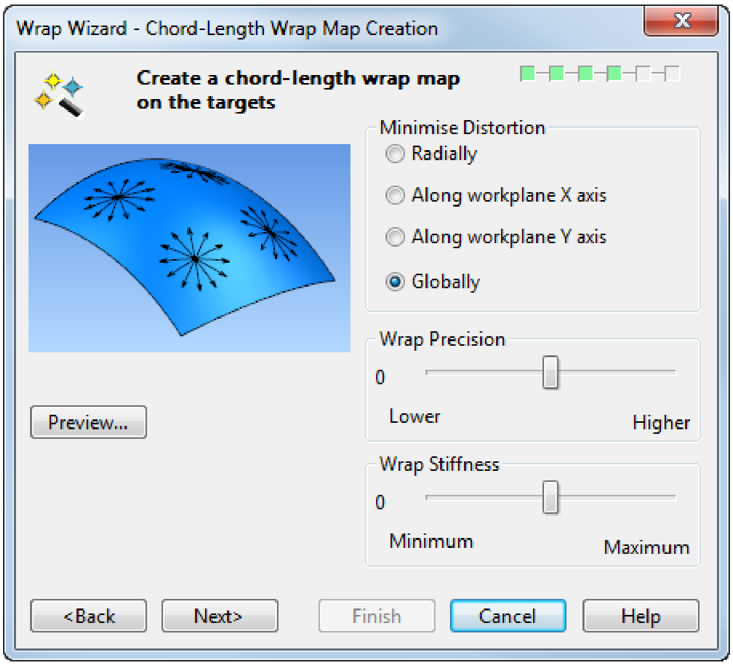
Next — Different pages of the wizard are displayed depending on which method is selected:

- **Plane**, **Cylinder** or **UV** — the **Wrapper Layout** page of the **Wrap Wizard** is displayed.
- **Chord** — the **Chord Length Wrap Map Creation** page of the **Wrap Wizard** is displayed.
- **Cone** — This option is selected when creating a 360 degree conical wrap and is greyed out when other wrap methods are in progress. For further details, see How do I create a 360 degree conical wrap? (Wireframe modelling).

Wrap Wizard - Chord Length Wrap Map Creation

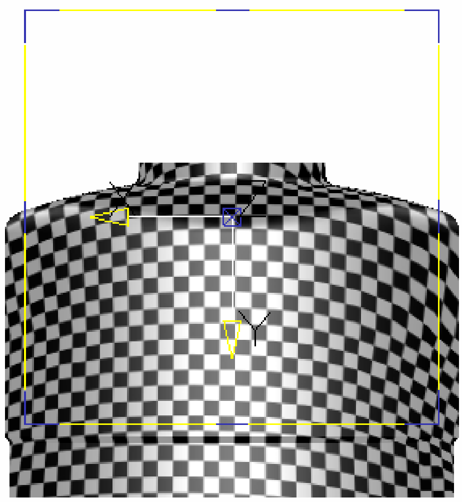
Use this page of the **Wrap Wizard** to define how the Chord wrapper is applied to the target object.

The wrap map is a 2D coordinate system for a triangulated surface. It is displayed as a black and white chequerboard effect on the target object.

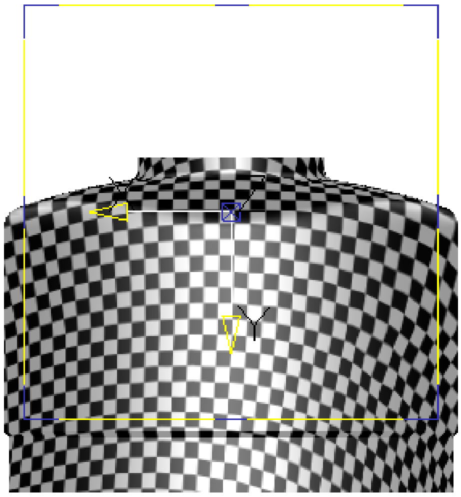


Minimize Distortion — Allows you to select where to minimize the distortion caused by the wrapping process. Select one of the following:

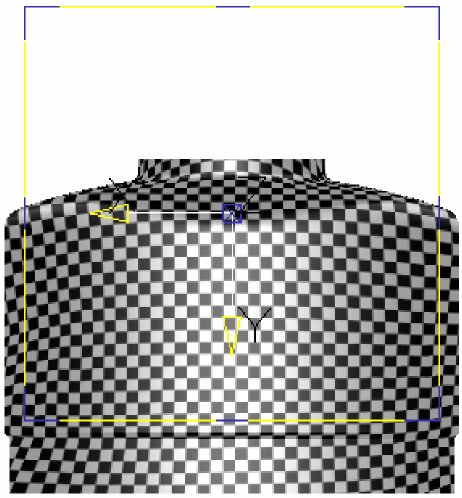
Radially — This option works most successfully with square-shaped target objects.



Along workplane X axis — This is the default option. This provides straighter lines and more even spacing along the X axis of the target object.



Along workplane Y axis — This provides straighter lines and more even spacing along the Y axis of the target object.



Globally (*recommended*) — minimises the distortion equally over the whole surface, preserving the shape of the wrapper as well as possible. When this option is selected, **Wrap Precision** is unavailable.

This method is only suitable for open targets; the target cannot be closed or tubular. The target may contain holes.

Wrap Precision — Move the slider to adjust the precision of the wrap map to reduce distortion if necessary. **Higher** gives the best wrap quality but the wrap map takes longer to create. The default position is **Normal**.

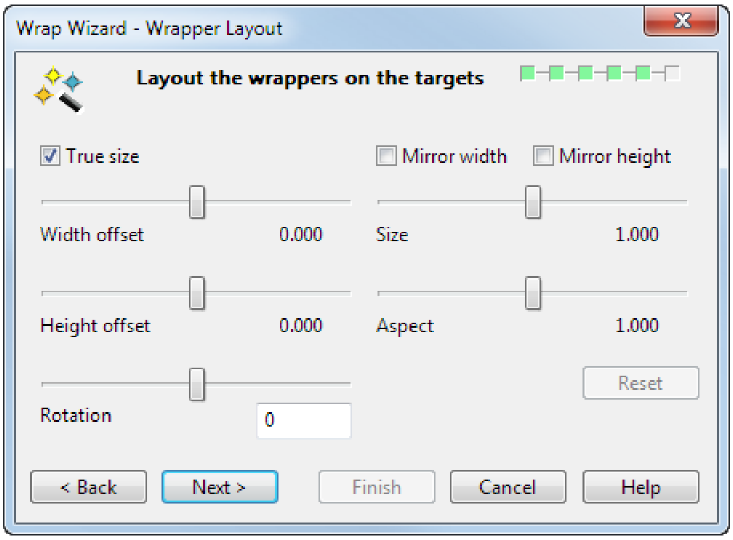
Wrap Stiffness — Move the slider to adjust the preservation of large triangle areas, by changing their wrapping angle. **Maximum** stiffness reduces wrapping angles and preserves large triangle areas. **Minimum** stiffness increases wrapping angles of triangles for greater curvature. This is useful to prevent the flipping of triangles during the wrapping process. The default position is Normal.

Preview — This creates the wrap map and displays it as a chequerboard effect on the target object.

Next — The Wrapper Layout page (see page 228) of the **Wrap Wizard** is displayed.

Wrap Wizard - Wrapper Layout

Use this page to manipulate the position of the wrapper on the target object.



True size — If ON, this preserves the aspect ratio and size of the triangles in the DMT file. If OFF, the triangles in the DMT file will be scaled to fill the target object.

Mirror width — Mirrors the wrapper along its x axis.

Mirror height — Mirrors the wrapper along its y axis.

Use the following sliders to position and orientate the wrapper precisely on the target object:

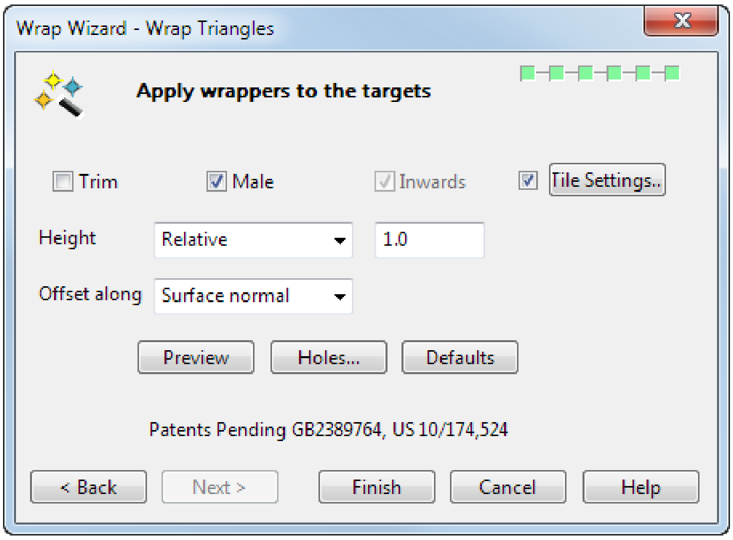
- **Width offset**
- **Size**
- **Height offset**
- **Aspect**
- **Rotation** — You can also enter the angle of rotation required.

To finely adjust the sliders, use the arrow keys on the keyboard or the mouse wheel.

Next — The **Wrap Selection** page of the **Wrap Wizard** is displayed.

Wrap Wizard - Wrap Triangles

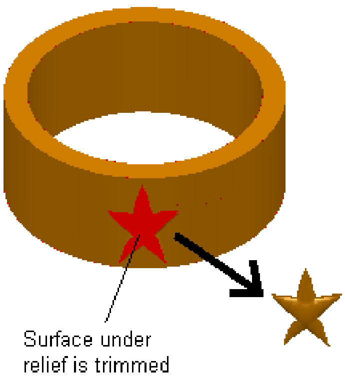
Use this page to wrap the relief onto the surfaces and solids.



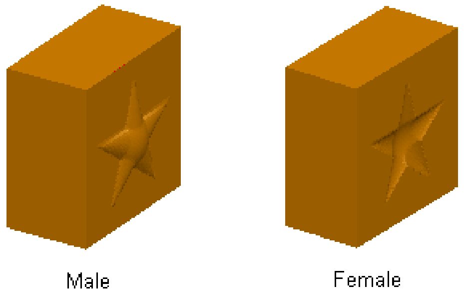
If the selected wrapper is a solid or one or more surfaces, then the result of wrapping the selection onto the target is also a solid or one or more surfaces.

If the target is not smooth, the result may not match accurately or become folded.

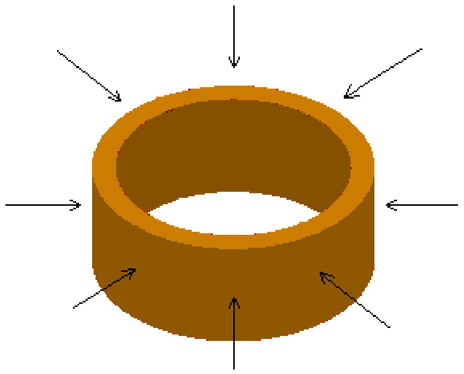
Trim — If on, the objects are trimmed to the outer boundary of the relief.



Male — If on, the offset is along the normal or workplane vectors and creates a male wrap. If off, the offset is in the opposite direction and creates a female wrap.

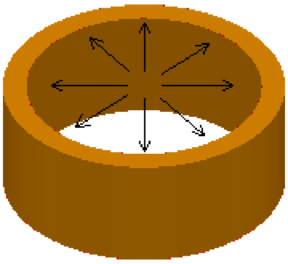


Inwards — This option is most likely needed when using cylindrical projections. If it is on, the relief is projected from outside the model, inwards towards the centre of the cylinder.

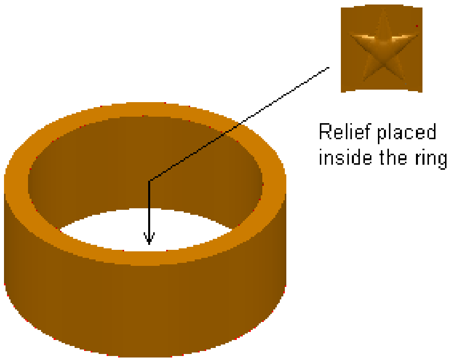


A relief node is placed on the first surface it encounters.

If off, the relief is projected outwards from the centre of the cylinder.



The relief is placed inside the ring.



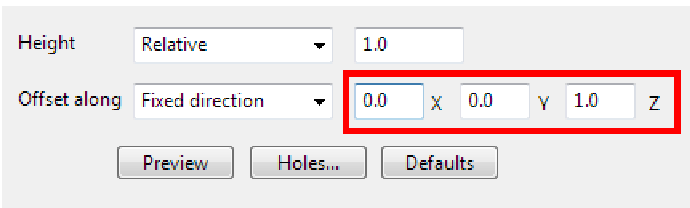
Tile Settings — This creates a tiling pattern on the target using the selected wrapper. For further details, see Wrap Wizard Tiling (see page 233). Select this option to activate the **Tile Settings** button. Click the **Tile Settings** button to display the Tile dialog (see page 235).

Height — This controls the height of the wrap. There are three options:

- **Relative** — You need to enter a value in the text box. A "natural" height for the relief is multiplied by the number in the text box.
- **Preserve** — The height of each node is as given in the dmt file.
- **Maximum** — You need to enter a maximum value. The height of the highest node is set to this value. The heights of all the other nodes are set proportionally to preserve the appearance of the relief.

Offset along — This controls the offset direction of the wrap. There are two options:

- **Surface normal —** Select this option to offset each node of the wrapper triangles in the direction of the surface normal by the height specified in the **Height** box.
- **Fixed direction —** Select this option to offset each node of the wrapper triangles in the fixed direction, entered in the **X**, **Y**, **Z** boxes, to the height specified in the **Height** box.



Preview — This previews the wrapping of the triangles from the dmt file onto the objects.

Holes — This displays the **Hole Filling** dialog (see page 232) dialog, which sets options for filling holes caused by gaps in the model.

This option is not available when:

- *creating solid wrap features*
- *if* **Globally** *is selected as the method to minimise distortion. The resulting wrap goes round the holes without distortion and the holes remain unfilled.*

Defaults — This changes the settings on the dialog back to their default values.

Cancel — No wrap is created.

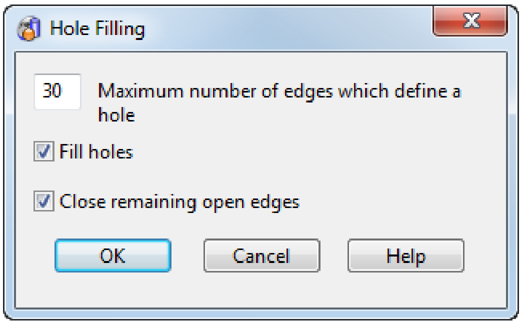
Finish — This wraps the surfaces, solids or triangles from the dmt file onto the objects and the **Wrap Wizard** completes.

If you create a wrap using the **Wrap Triangles** option on the **Surfaces** menu, the wrapper will have no dependencies on the target surface and the two objects will be unconnected.

If you create a wrap using the **Create a Solid Wrap feature** option on the **Solids** menu, dependencies will be created between the wrapper and the target object and a wrap feature will be created in the solid feature tree. This allows the wrap feature to be modified. For further details, see What is the Solid Feature Tree? (Solid modelling)..

Hole Filling dialog

This dialog sets options for filling holes caused by gaps in the model.



During wrapping, nodes in the relief can miss the target surfaces if there are gaps in the model or the relief falls off the edges of the surfaces. Such nodes are removed from the relief and their edges trimmed back. This can create holes and open edges in the wrap.

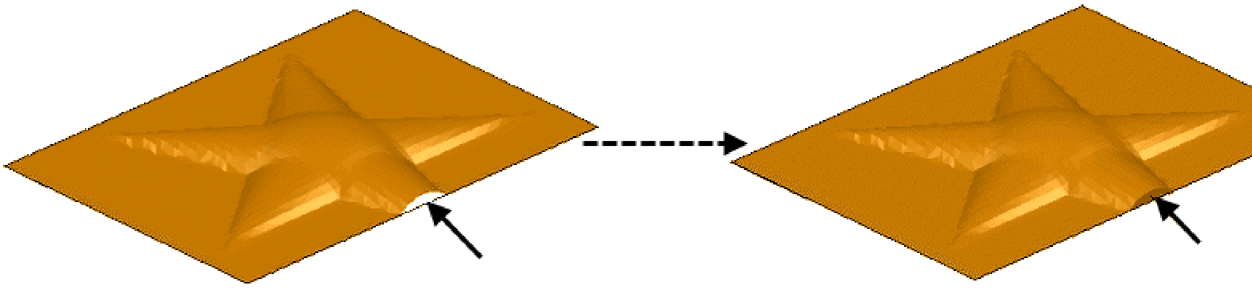
Maximum number of edges which define a hole - This defines a hole in the relief. The edges are those of the triangles in the relief.

Fill holes - Fills a hole with triangles if it has fewer edges than the maximum number.

PowerSHAPE uses a simple method for filling holes. For minor problems, it should do the job. In some cases, it may not give the required result. In these cases, you can do one of the following:

- Correct the surfaces underneath the wrap and create the wrap again
- Switch off the **Fill holes** option to create the wrap. Use **Mesh Editing** toolbar to fix the triangle and improve the results.

Close remaining open edges - Open edges are gaps between the model and the boundary of the relief. These are usually caused by the relief spilling off the edge of the surface. This option closes any open edges.



OK - Changes the options to the new settings and removes the dialog from the screen.

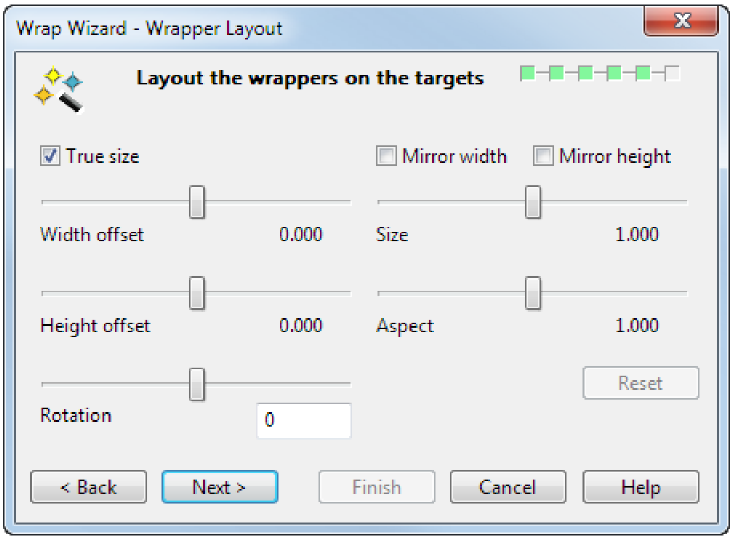
Cancel - The options are not changed and the dialog is removed from the screen.

Wrap Wizard tiling

You can create a tiling pattern from a surface, DMT file or symbol on your target object. Tiling creates copies of the original tile in rows and columns. The rows are separated by vertical spacing and the columns by horizontal spacing.

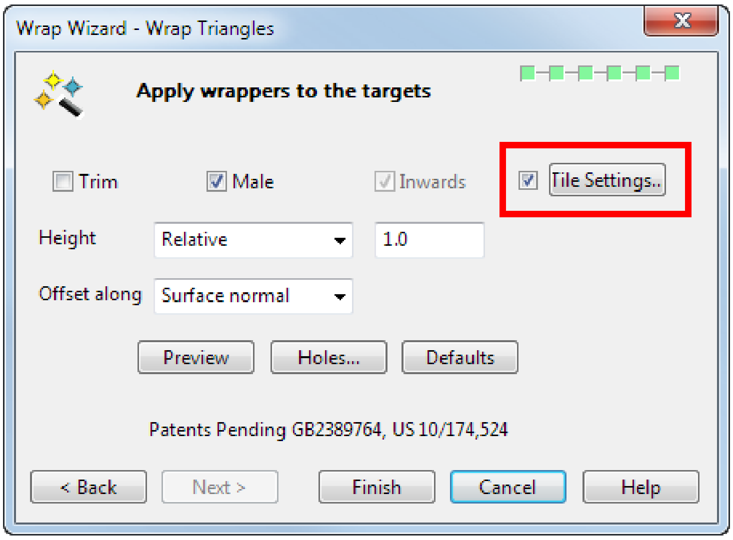
To create a tiling pattern:

1 Specify the start position for the first tile in the tiling pattern on the target object using the layout controls on the **Wrapper Layout** page of the **Wrap Wizard.**



2 Select the option next to **Tile Setting** to activate the tiling functionality.

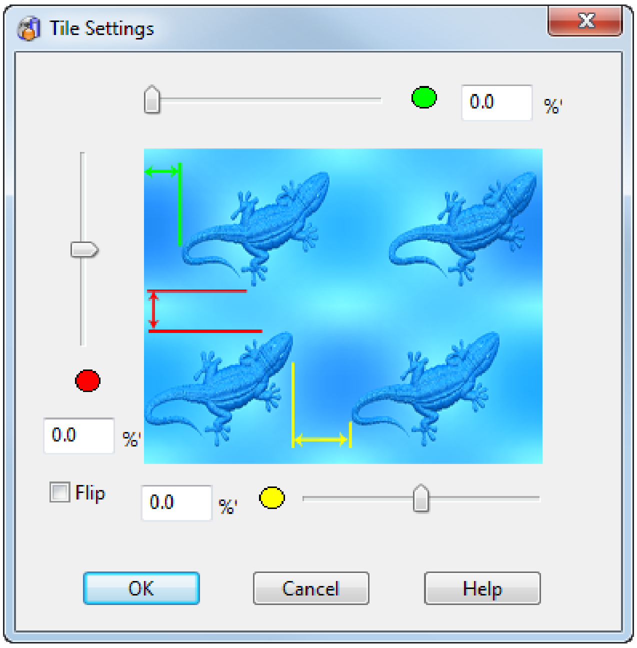
3 Click on the **Tile Setting** button.



The **Tile** dialog is displayed, containing the tiling controls for creating a tiling pattern.

Tile dialog

The **Tile** dialog contains four tiling controls for creating a tiling pattern using the wrapper in the **Wrap Wizard**.



You can use a combination of the four tiling controls to create the desired tiling effect, or leave the values at zero to create tiling with the default values. You can enter the values directly or use the sliders to set the values.

\- **Tile with offset.** Enter a percentage of the corresponding dimension of a tile to offset each row to the right. This results in a stretcher-like pattern similar to that of a brick wall.

\- **Tile with vertical spacing**. Use the slider to increase or decrease the vertical spacing between the tiles, or enter a value to specify the percentage of the total spacing in the vertical direction.

\- **Tile with horizontal spacing.** Use the slider to increase or decrease the horizontal spacing between the tiles, or enter a value to specify the percentage of the total spacing in the horizontal direction.

Flip - If *ON*, the alternate tiles are flipped. **Flip** is not available when wrapping a surface.

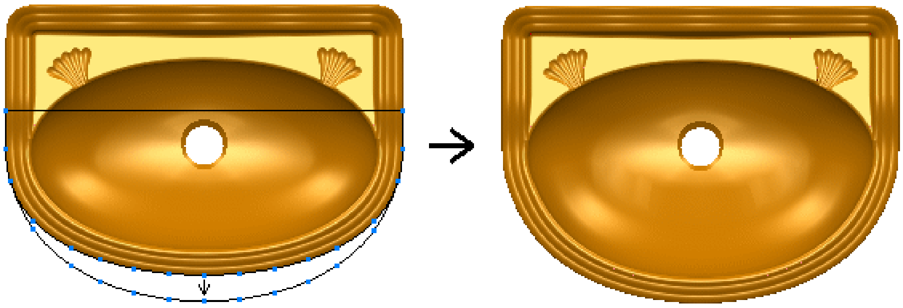
OK - Click to create a tile using the values displayed on the dialog or accept the default zero values. Tiling is created when you select **Finish** on the **Wrap Selection** page of the **Wrap Wizard**.

*Only click **OK** if you are sure you want to create a tile pattern.*

Cancel - Click to exit **Tile Settings** and return to the **Wrap Wizard**.

Surface morphing

Morphing allows you to deform a group of surfaces into a different shape.



All the detail on the surfaces, such as fillets and trimming remain intact, even though the shape may be totally changed.

Morphing can be used for a number of applications:

- Adding draft to a model.
- Compensating for distortion that occurs during the manufacturing process, for example, sag and spring-back.
- Adding design features.
- Local modification such as closing a gap.
- Shoe design, wrapping a sole onto a new size.
- Making artistic and aesthetic changes to a model to allow you to improve and rejuvenate existing models.

It can be controlled by using points, curves or surfaces around the model. For best results, use morphing to make fairly minor modifications rather than substantial changes to the shape.

The point, curve and surface morphing functionality is subject to a patent application. Patent pending: GB2401213 Altering a CAD model.

Creating a surface morph

1. Select the surfaces you want to morph.
2. Select  (*General edits toolbar*) This is the same as selecting **Edit > General Edits > Morph Objects**.

   *You can also select the surface with the right mouse button and choose **Define Morph**.*
3. Use the **Surface morphing** dialog to define the morph.

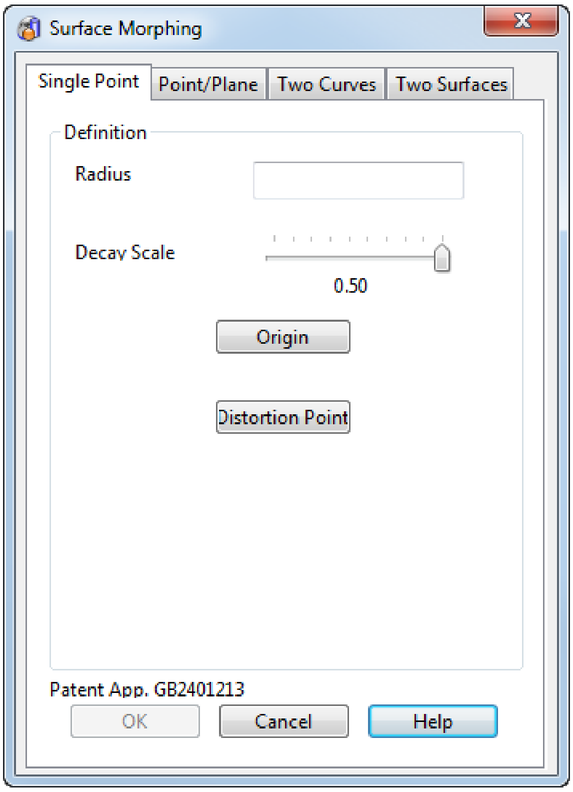
Surface morphing dialog

The **Surface Morphing** dialog has four tabs. Select one of the following for further information:

- Surface Morphing dialog - Single Point (see page 237)
- Surface Morphing dialog - Point/Plane (see page 240)
- Surface Morphing dialog - Two Curves (see page 243)
- Surface Morphing dialog - Two Surfaces (see page 249)

Surface Morphing dialog - Single Point

Use this morphing method to make local changes to a shape, while keeping it smooth.

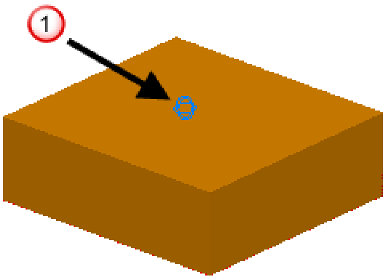


This morphing method uses two points:

- **Origin**. This usually lies on the surface.
- **Distortion point**. This can lie anywhere in space.

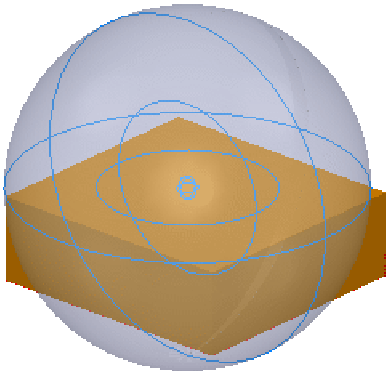
The origin point moves to distortion point.

When this page on the dialog is displayed, a point is marked on the objects to show where the origin and distortion points lie. Both points initially lie in the same position 1 until one of them is redefined.



1 Enter a **Radius**. Two spheres are drawn around the origin point.

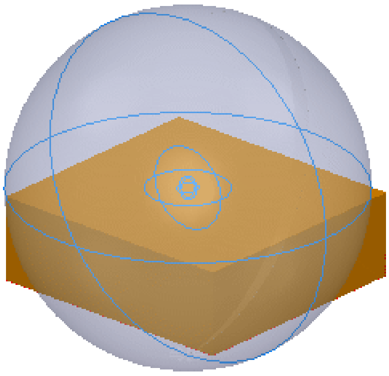
- The outer sphere shows the bounded region affected by the morph.
- The inner sphere reflects the **Decay Scale.**



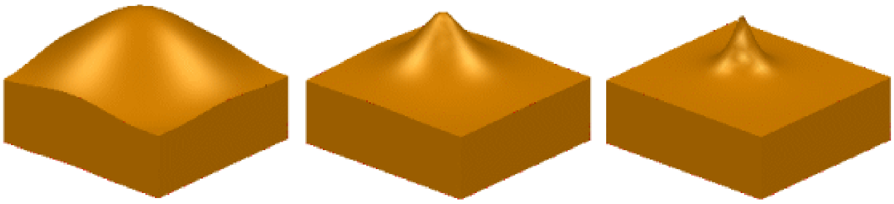
Points within the sphere move in the direction defined by the origin and the distortion point. The amount they move depends on the distance from the origin and the decay scale. Points at or beyond the sphere remain unchanged.

To avoid introducing folds in the surface, the radius should generally be at least twice the distance from the origin to the distortion point.

2 Use the slider to set the **Decay Scale**. This controls how rapidly the shape changes as you move away from the origin point. If you change the value of the **Decay Scale**, the inner sphere changes size. The limit of this sphere is where the distortion will have been reduced by half. So, the smaller the inner sphere is in comparison to the outer sphere, the more rapid the shape change.

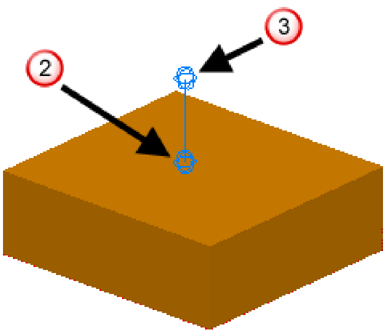


You can set a value between *0.05* (rapid change) and *0.5* (slow change). The diagram below shows the changes to the shape as the decay value decreases. The decay defines the fraction of the radius at which the distortion is halved.

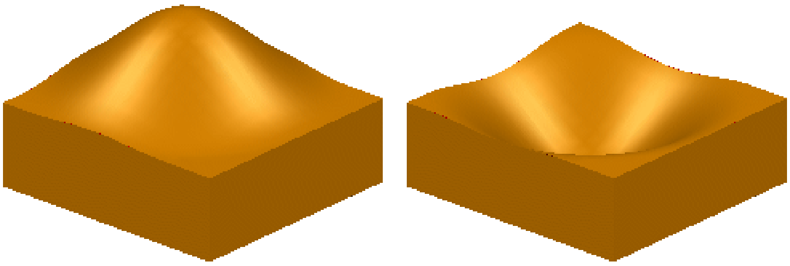


3 Click **Origin** to display the **Position** dialog. Use this to enter a new origin by selecting a point on the surface or solid. When you enter a new origin, the mark moves to the new position 2. You can also drag the origin point to move it.

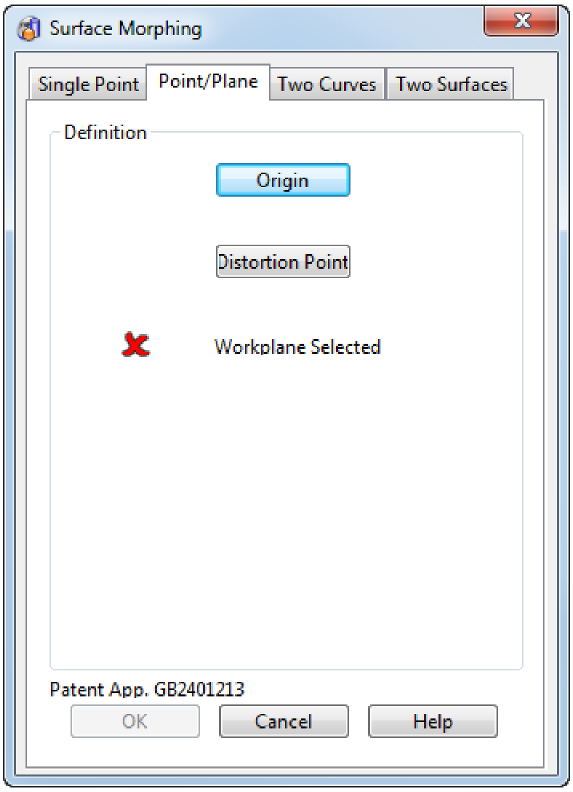
4 Click **Distortion Point** to display the **Position** dialog. Use this to enter the new position of the origin point. A point is drawn to mark where the distortion point lies 3 and a line is drawn from the point to the origin. You can also drag the distortion point to move it. The distortion point can lie on either side of the surface.



5 Click **OK** to create the morph.



Surface Morphing dialog - Point/Plane

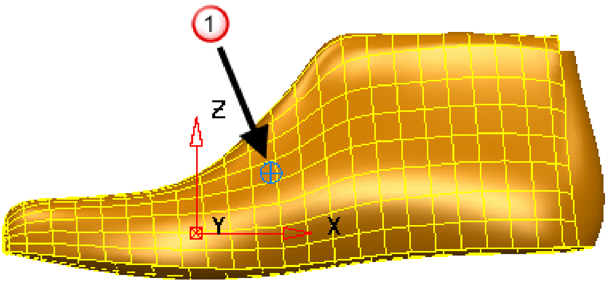


This morphing method is ideal for altering the toe-spring of a shoe, and similar applications where you want to bend a portion of a model.This morphing method uses two points:

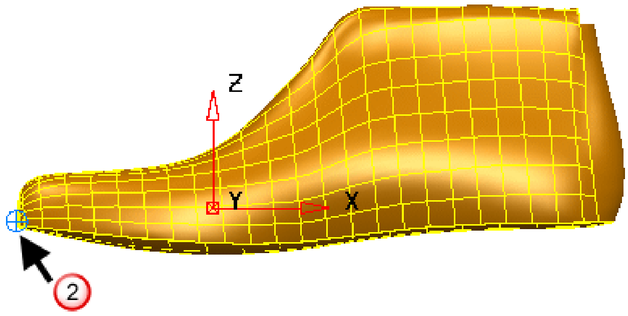
- **Origin**. This usually lies on the surface.
- **Distortion point**. This can lie anywhere in space.

The origin point moves to distortion point.

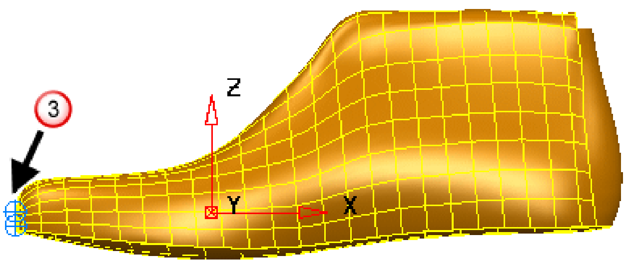
When this page on the dialog is displayed, a point is marked on the objects to show where the origin and distortion points lie. Both points initially lie in the same position 1 until one of them is redefined.



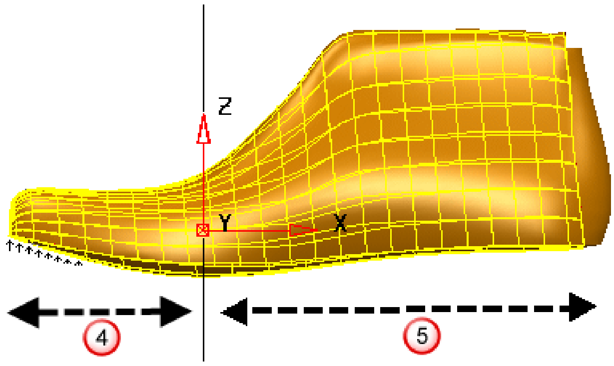
1 Click **Origin** to display the **Position** dialog. Use this to enter a new origin by selecting a point on the surface or solid. When you enter a new origin, the mark moves to the new position 2. You can also drag the origin point to move it.



2 Click **Distortion Point** to display the **Position** dialog. Use this to enter the new position of the origin point. A point is drawn to mark where the distortion point lies 3 and a line is drawn from the point to the origin. You can also drag the distortion point to move it



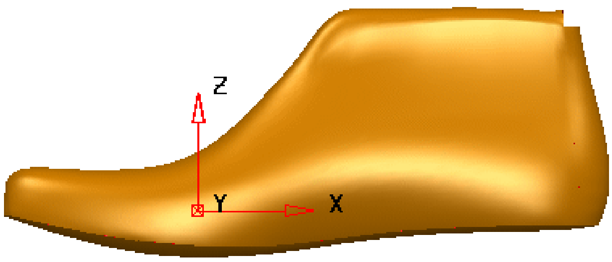
3 Create and select a workplane on the surface. The cross icon changes to a tick icon.. This selection determines the region of the objects to alter. If a region of the object lies on the same side of the plane as the origin point, its points are moved 4. Otherwise the region remains unchanged 5. If no workplanes exist in the model, the world workspace is automatically used.



Points are moved in the same direction by an amount depending on their distance from the plane, in order to keep a smooth shape. All points that are the same distance from the plane are moved by the same amount.

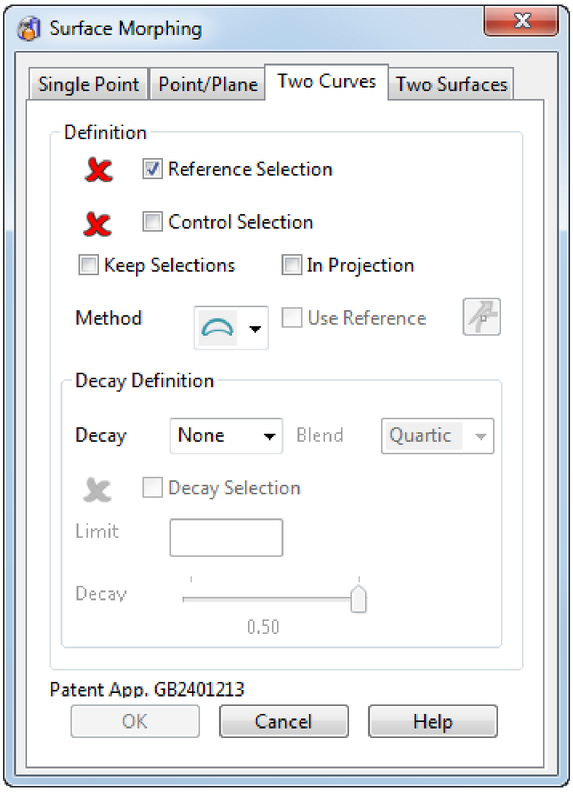
*The principal plane is selected by choosing one of the buttons from the **Status** bar. For example, if you want to select the YZ principal plane, select the button with **X** on it.*

4 Click **OK** to create the morph.



Surface Morphing dialog - Two Curves

Define the morph using a control and reference curve.

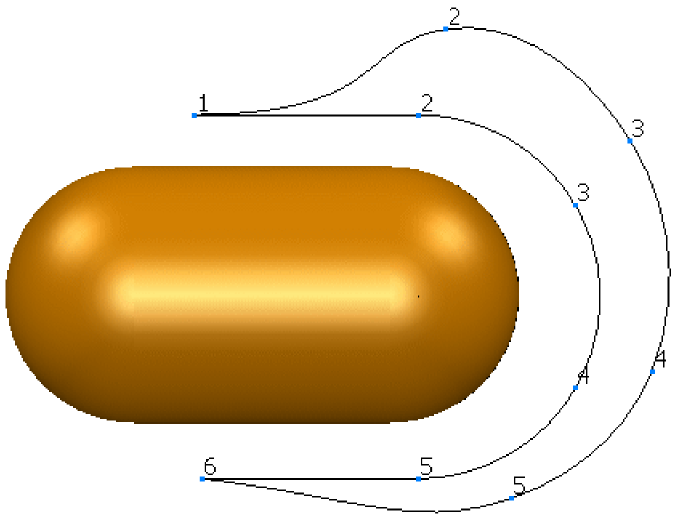


The two curve method provides more control over the shape of the result than the single point methods.

When morphing with curves:

- Both the reference and control curve must have the same number of points.
- Both the reference and control curve must be open or both closed.
- Keep the reference and control curves simple.
- Limit curves or surfaces should not cross the reference curve.
- Keep the limit curve or surface simple.
- If the curves are open, the start and end points of the curves must coincide.

- If open curves are also tangential at the start and end, you will get smoother results in cases where the model to be morphed extends beyond the ends of the curve.

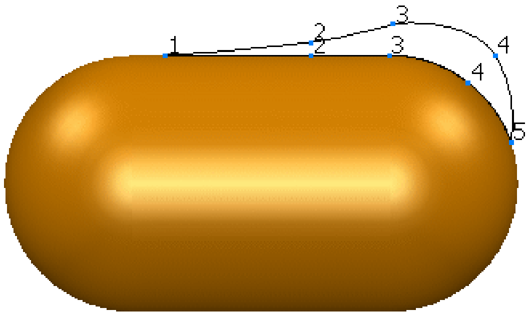


- Avoid discontinuities and sharply curved corners in the reference and control curves and surfaces, as they may introduce kinks or ripples into the morphed model.
- As you move over the model, it is important that there is always a unique closest point on the curve, and that the closest point does not jump from one place to another. Otherwise the morphed model may contain kinks.
- The model being morphed may be as complicated as you like.

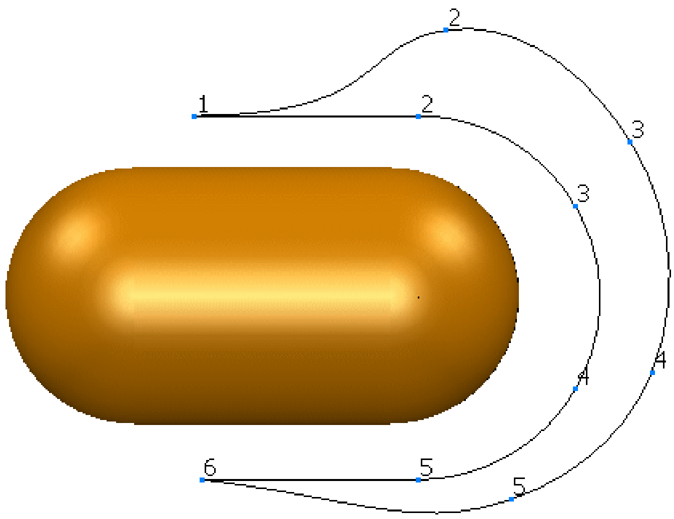
Two curves dialog

1. Select **Reference Selection** and select the reference curve on the screen.
2. Select **Control Selection** and select the control curve on the screen.
3. By default, the reference and control curves are deleted from the model after you have used them for morphing. If you want to keep them, select **Keep Selections**.
4. Select a **Method** suitable for the shape of your reference and control curves.

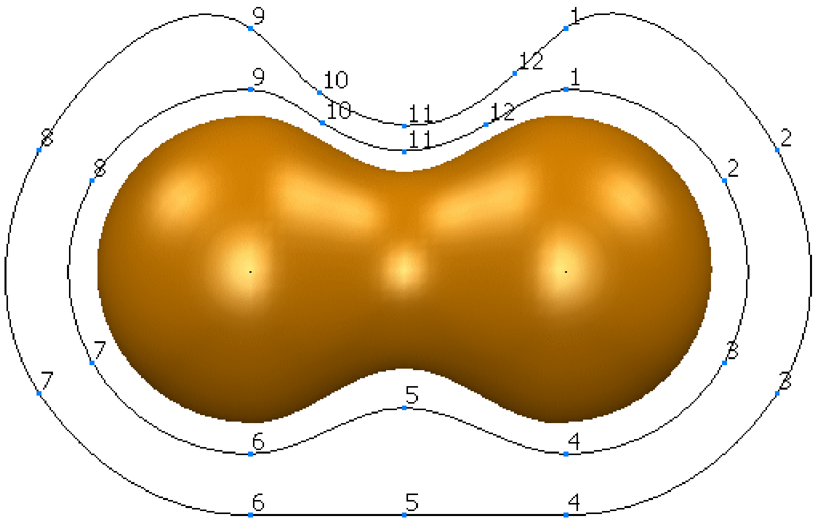
 - Select this method if your curves are open and have only a small turning.



 - Select this method if your curves are open and have a large turning.



 - Select this method if your curves are closed.



If your curves are open and you are not sure which method to use, try methods and and select the one that gives the best results.

In Projection - This option projects certain objects onto the principal plane so that PowerSHAPE can calculate how to morph the surfaces.

To fully understand this option, you need to know how objects are morphed.

The principal plane is selected by choosing one of the buttons from the **Status** bar. For example, if you want to select the YZ principal plane, select the button with **X** on it. If no workplanes exist in the model, the world workspace is automatically used.

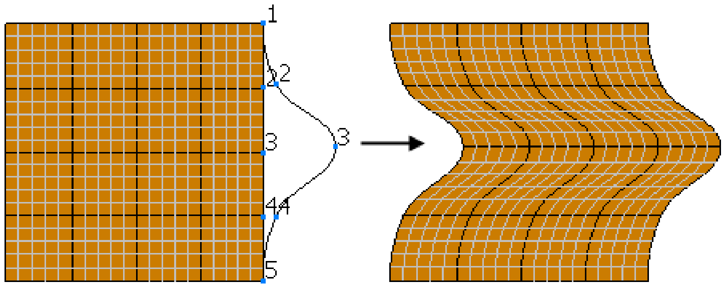
Use Reference - (Only available if you use the closed curves method and the **In Projection** option is on.) This allows you to select the reference axis that is used to calculate the corresponding points on the reference curve. When using closed curve morphing, it is recommended that you use a reference axis if the object to be morphed lies mainly inside the curve. Click the button to display the **Direction** dialog. Use this dialog to select the axis.

Moving points using the two curves morphing method (see page 251) gives details on how objects are morphed.

5 Select the **Decay** option. This decides how much the points on the selected surfaces move as you move away from the reference curve. If the points to be morphed lie on the reference curve, they are moved onto the corresponding point on the control curve regardless of the decay option. Points that are not on a reference curve are moved according to the selected **Decay** option.

The following options are available:

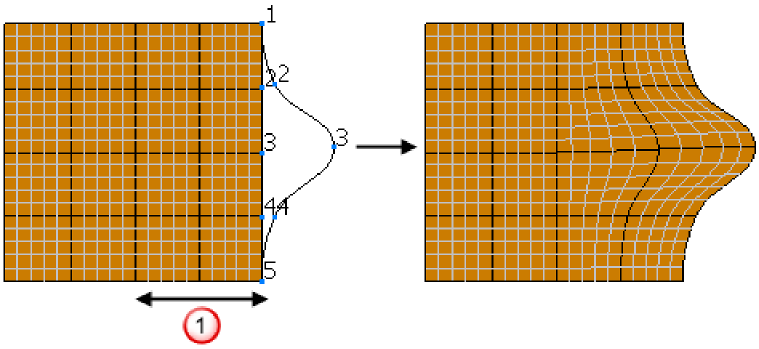
None - Points are moved by the distance between the corresponding points on the two curves.



Distance - Points are moved depending on the distance from their corresponding point on the reference curve 1.

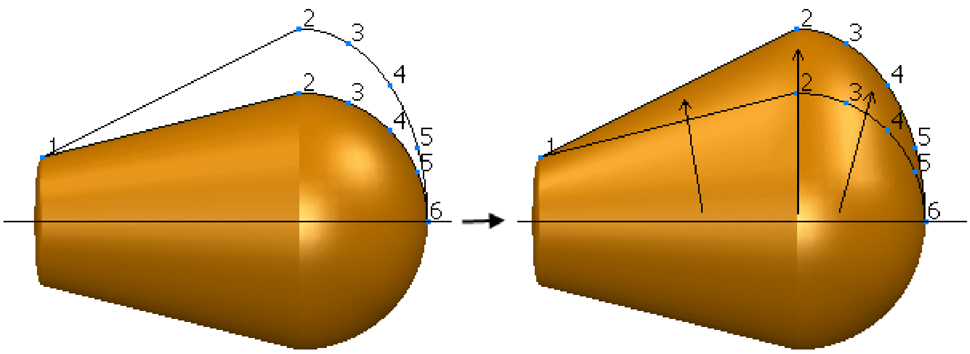
If the distance is less than the value given in the **Limit** box, a point moves by the distance defined by its corresponding points on the two curves and the **Decay** scale. The closer the distance is to the value given in the **Limit** box, the less the point moves.

If the distance is equal to or more than the value given in the **Limit** box, the points are not moved.



Curve or **Surface** - Points are moved depending on the distance from their corresponding point on the reference curve and distance from the closest point on the selected limit curve or surface.

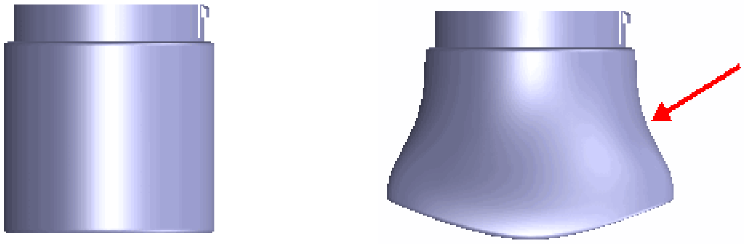
The closer the distance is to the limit curve or surface, the less the point moves. Beyond the selected curve or surface, the points are not moved. The curve or surface should not cross the reference curve.



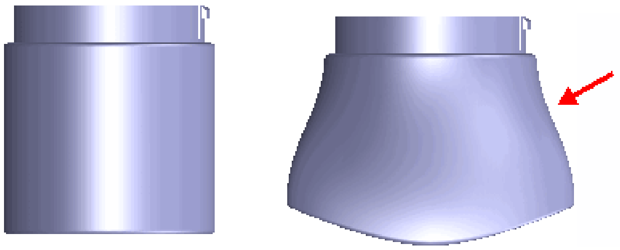
The **Curve** and **Surface** options take longer to process, and should only be used when other options fail to produce the required solution..

6 Select a **Blend** from the drop-down list. For all types of decay other than **None**, you can define how the shape of the morphed region blends into its surroundings using the following options:

Quartic - This is the default option It provides a smooth blend and join to the surroundings and a smooth shape as it crosses the reference curve. This option should give the correct result in most cases.

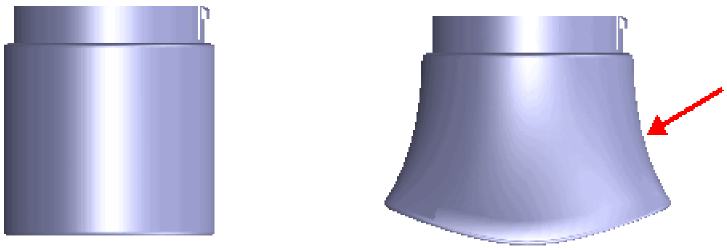


Cubic - It provides an S-shaped blend, with a smooth join to the surroundings, and a smooth shape as it crosses the reference curve.



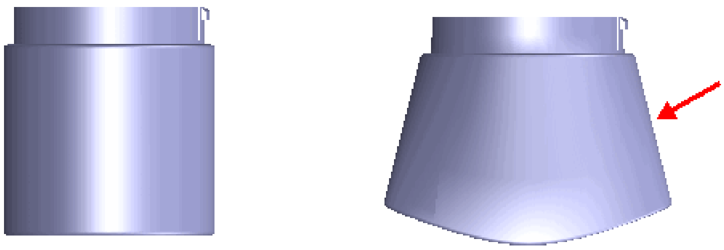
The **Quartic** and **Cubic** options give very similar results. so it is best to try each option in turn and use the option that produces the best solution for your model.

Parabolic - This varies smoothly with the surroundings, but comes to a sharp peak at the reference curve.



It is suitable when the reference curve lies on an open edge of the model, else a sharp edge will be introduced along the curve.

Linear - This varies in a straight line from the maximum on the reference curve to zero at the limit. Therefore the morph does not blend smoothly with its surroundings.



It is suitable if the entire surface is being morphed, or if there is a limit curve defined along a discontinuity on the model.

7 Select **Decay Selection** if you selected **Curve** or **Surface** as the **Decay** option. Select the required object.

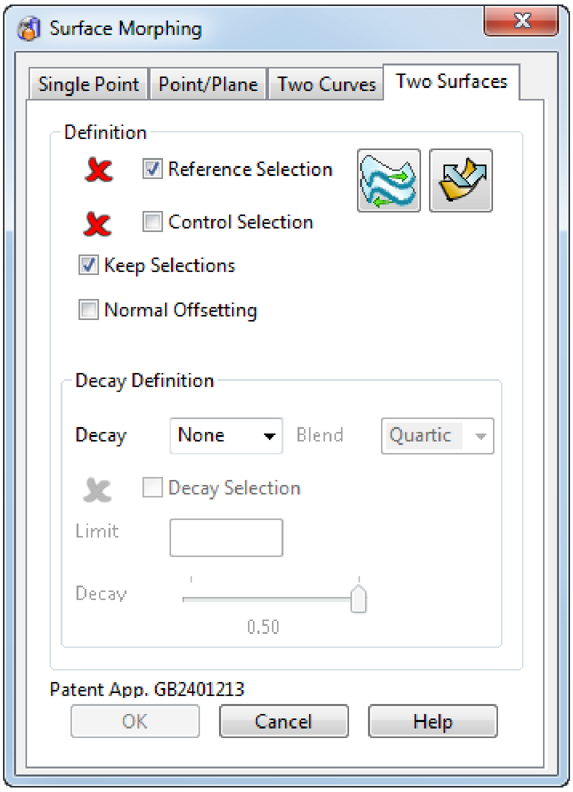
8 Enter the **Limit**. This is the distance used by the **Distance** option in the **Decay** menu.

9 Select the **Decay** scale if you selected the **Distance** as the **Decay** option. This controls how rapidly the shape changes as you move away from the reference curve. You can set a value between *0.05* (rapid change) and *0.5* (slow change).

   The decay defines the fraction of the limit distance at which the distortion is halved.

Surface Morphing dialog - Two Surfaces

This uses a control and reference surface to define the morph.



The two surface method provides more control over the shape of the result than the single point methods. It can be used to make substantial changes to a model, for example to stretch or bend in a controlled manner, or to project one shape onto another.

You should bear the following important points in mind when working with surfaces.

- Both the reference and control surface must have the same number of laterals and longitudinals.

- The laterals on both surfaces must be open or both closed.
- The longitudinals on both surfaces must be open or both closed.
- Keep the reference and control surface simple.
- Keep the limit curve or surface simple.
- Limit curves or surfaces should not cut the reference surface.
- Avoid discontinuities and sharply curved corners in the reference and control surfaces, as they may introduce kinks or ripples into the morphed model.
- As you move over the model, it is important that there is always a unique closest point on the reference surface, and that the closest point does not jump from one place to another. Otherwise the morphed model may contain kinks.
- The model being morphed may be as complicated as you like.

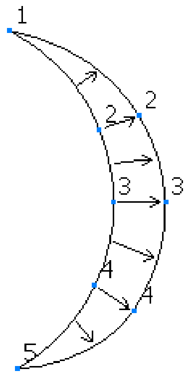
Two surfaces dialog

1. Ensure that **Reference Selection** is selected and choose the reference surface on the model. Use **Reverse surface**  and **Swap laterals and longitudinals**  as required.
2. Select **Control Selection** and choose the control surface on the model.
3. If you want to keep the reference and control surfaces after you have used them for morphing, select **Keep Selections**.
4. Select **Normal Offsetting** so the surfaces to be morphed are bent to follow the control surface. If you want to know precisely how the selected objects are morphed using **Normal Offsetting**, see Moving points using the two surface morphing method (see page 251).

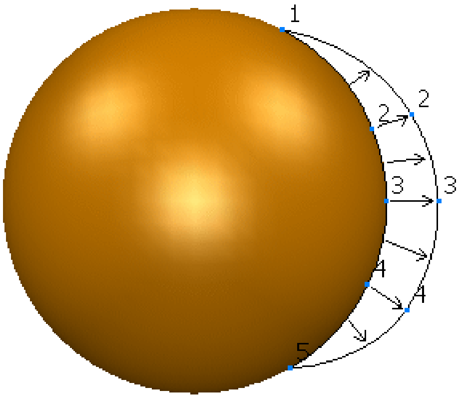
   If **Normal Offsetting** is deselected, you can specify the **Decay Definition** options. The options that are used to define the decay are described in Surface Morphing dialog - Two curves (see page 243)

Moving points using the two curves morphing method

The reference and control curve define the morph. For each point on the reference curve, there is a corresponding point with the same number on the control curve.



The distance and direction between two corresponding points on the two curves defines the morph. If the points on the selected objects lie on the reference curve, they are moved to the corresponding point on the control curve.



1. The position of corresponding points on the curves is important. In the example above, the keypoints correspond, producing the expected morph. If the keypoints do not correspond, the resulting morph may have an unexpected twist.

2. If the points on the selected objects don't lie on the reference curve, the ***Method*** *option determines how the points will move.*

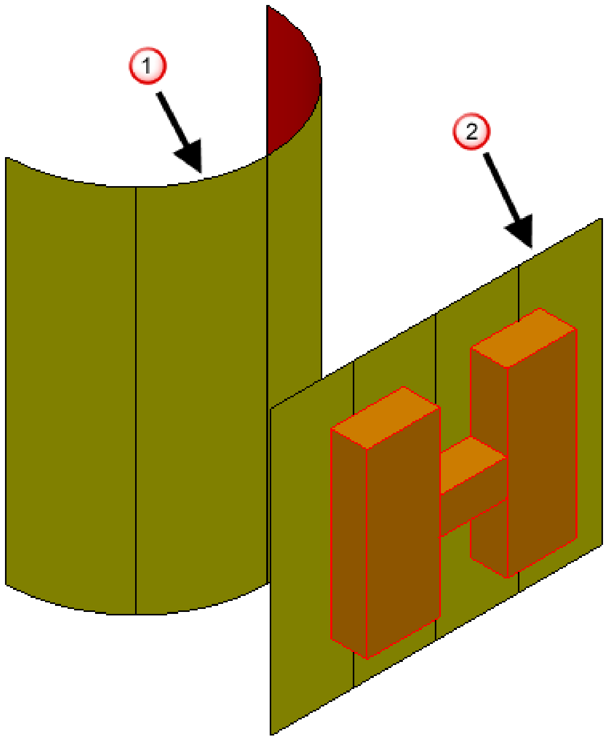
Moving points using the two surface morphing method

The following example shows how selected objects are morphed using the two surface morphing method.

The morph is defined by:

1 Control surface

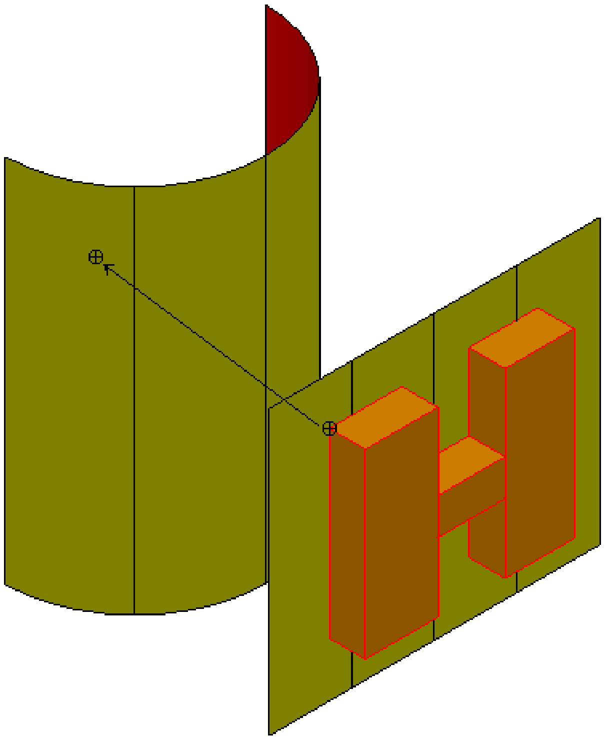
2 Reference surface



The extruded **H** surface is morphed by the difference between the flat reference surface and the cylindrical control surface. For each point on the reference surface, there is a corresponding point with the same 2D parametric coordinates on the control surface. This is used to define how to reshape the **H**.

2D parametric coordinates are measured using the laterals and longitudinals.

If the points on the selected objects lie on the reference surface, they are moved to the corresponding point on the control surface.



If the points on the selected objects don't lie on the reference surface, the **Normal Offsetting** option determines where the points will move.

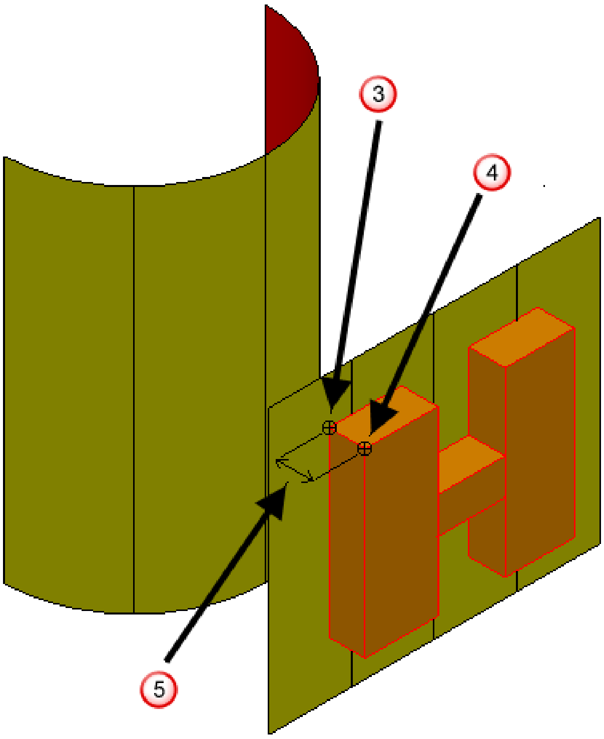
The morphing method finds the closest point on the reference surface for each point on the selected objects and the offset between the two points.

The positions of corresponding points on the surfaces are important. If the key points do not correspond, the resulting morph may have an unexpected twist.

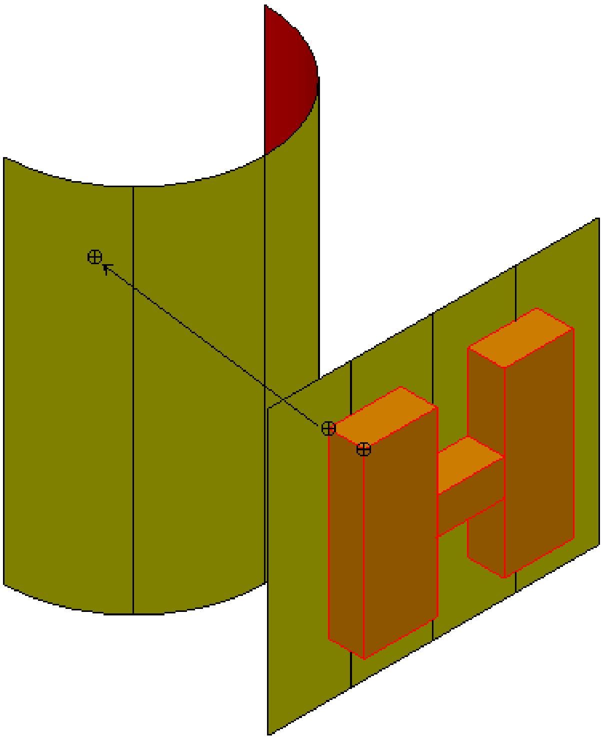
3 Closest point on the reference surface.

4 Point to be morphed.

⑤ Offset.



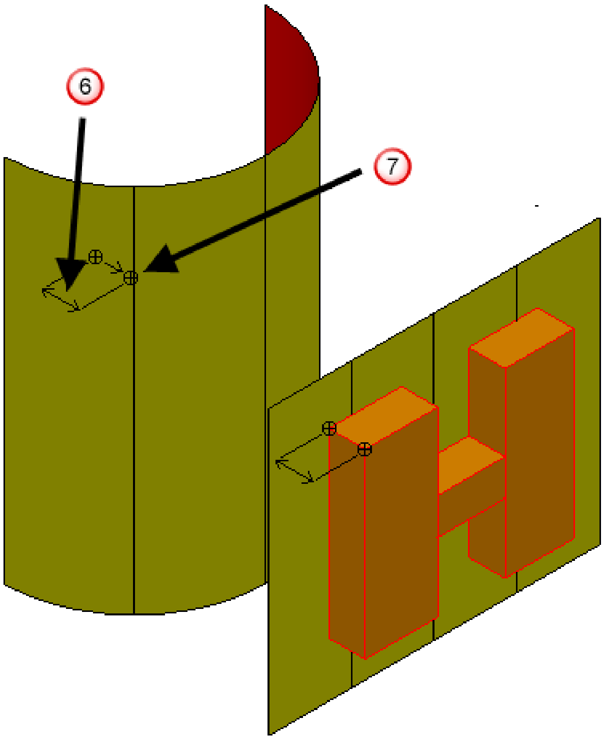
The point on the reference surface has a corresponding point with the same number on the control surface.



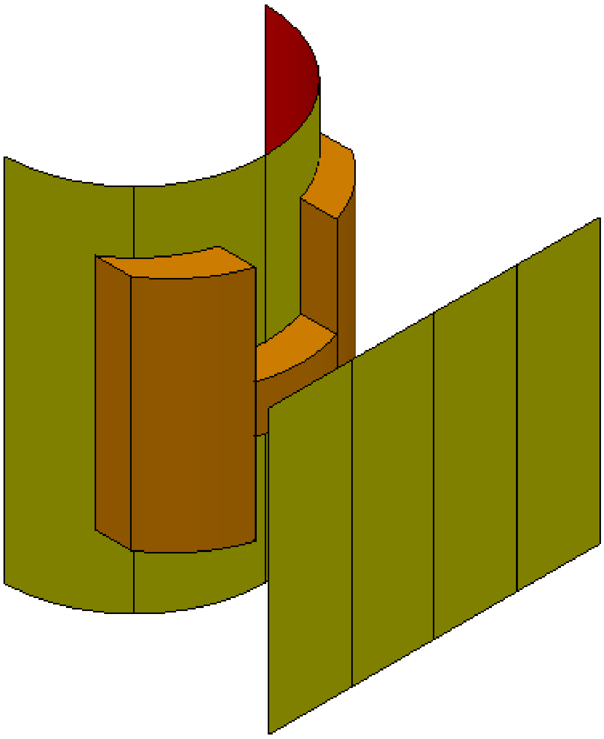
If the **Normal Offsetting** option is deselected, the point is moved so that it is offset by the same amount from the point on the control surface, reduced by any decay that you may have defined.

⑥ Offset on reference surface.

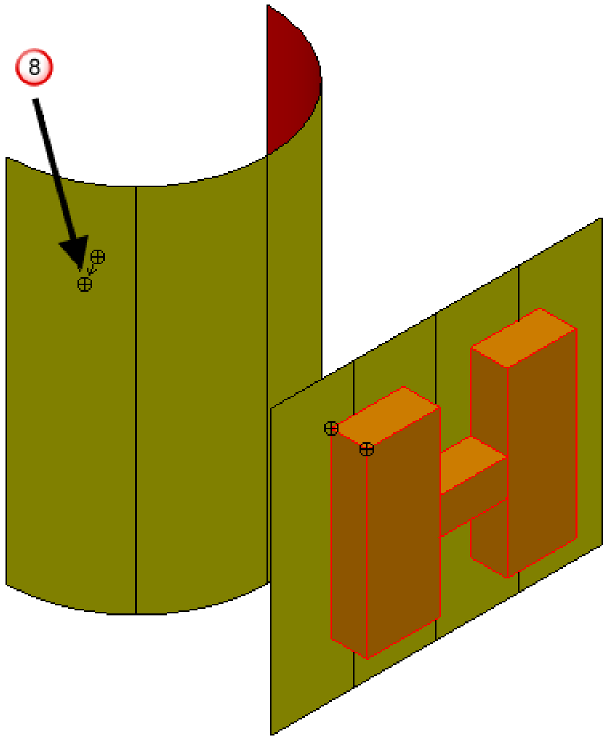
⑦ Position of point if there is no decay.



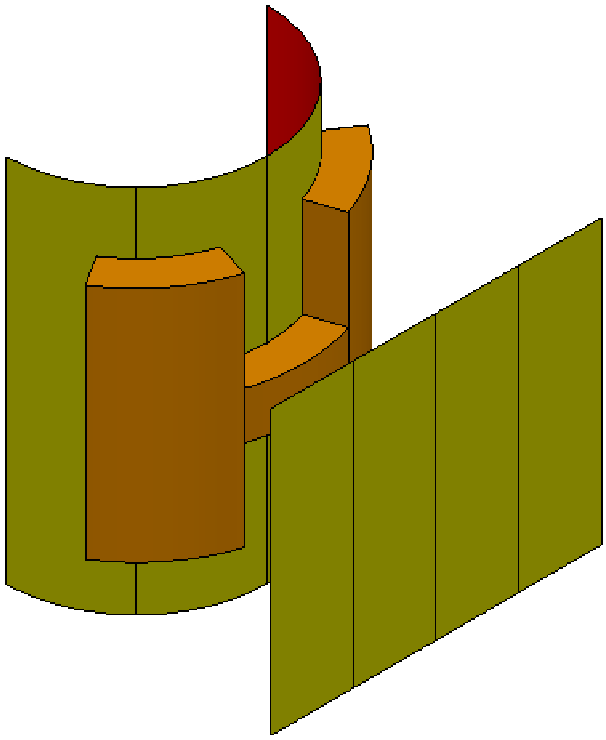
In this case, the model appears as follows:



If the **Normal Offsetting** option is on, points are offset along the surface normal on the control surface by their distance from the reference surface, so that the model appears to be bent to follow the control surface 8.



In this case,the surface appears as follows:

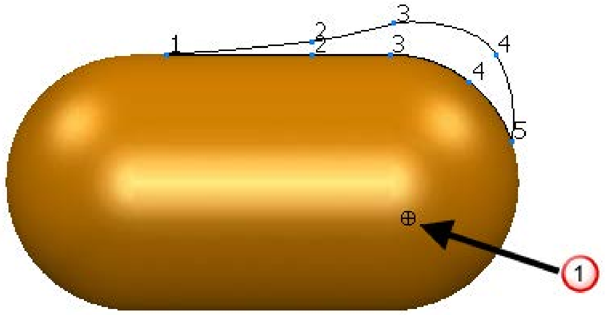


The positions of corresponding points on the surfaces are important. If the keypoints do not correspond, the resulting morph may have an unexpected twist.

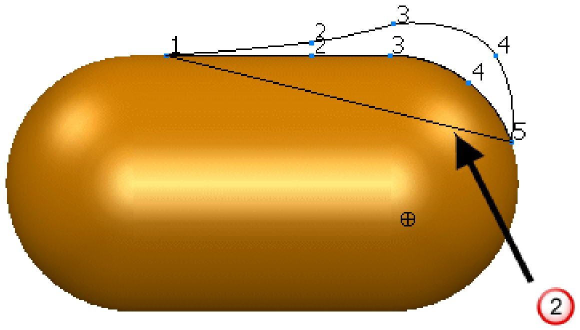
Small turning method

This section shows how points on the selected objects that don't lie on the reference curve are morphed using the small turning method.

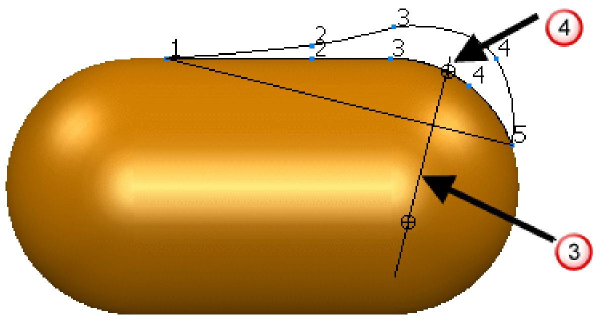
In the example below, the point 1 does not lie on the reference curve. We will find its corresponding point on the reference curve.



The reference axis 2 is along that line which joins the two ends of the reference curve.

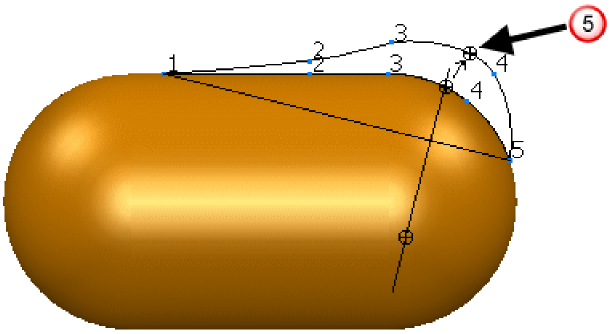


If a point to be morphed does not lie on the reference axis, a plane is defined through the point, normal to the reference axis 3. A corresponding point is defined on the reference curve where the reference curve intersects this plane 4.

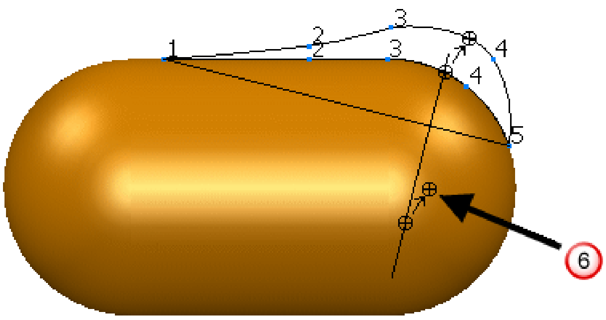


The curve must not be so highly curved that it crosses the plane more than once, or the morph will fail.

A distance and direction is defined from the point on the reference curve to the point with the same number on the control curve 5.



The point on the surfaces to be morphed is moved by this distance and direction, reduced by any decay that you may have defined . On the example, the point will move to 6 if there is no decay.

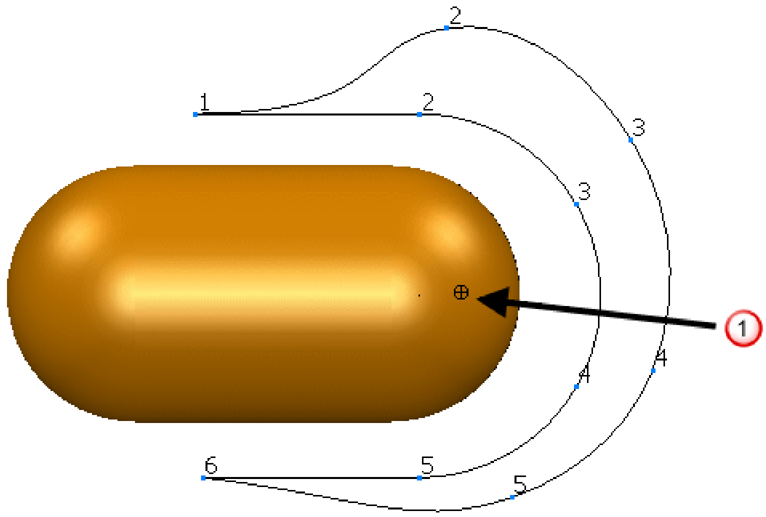


If **In projection** option is selected, the reference axis is projected onto the principal plane before the plane is defined to identify points on the reference curve.

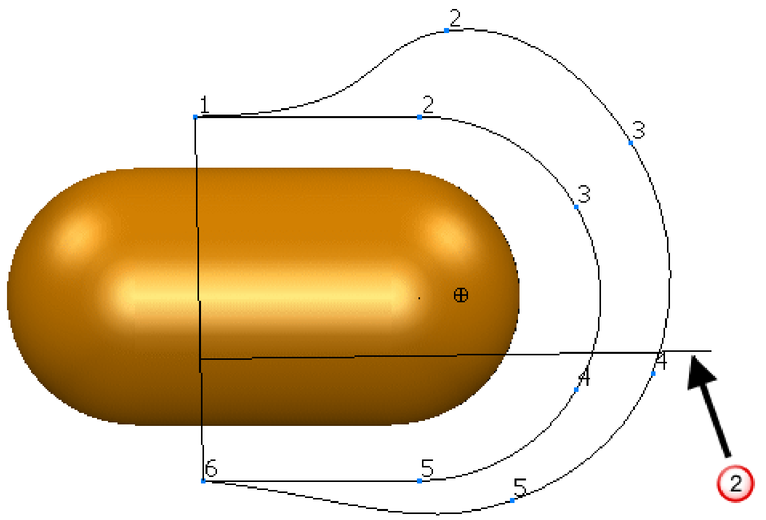
Large turning method

Points on the selected objects that do not lie on the reference curve are morphed using the large turning method .

In the example, the point 1 does not lie on the reference curve and the reference curve has a large turning angle



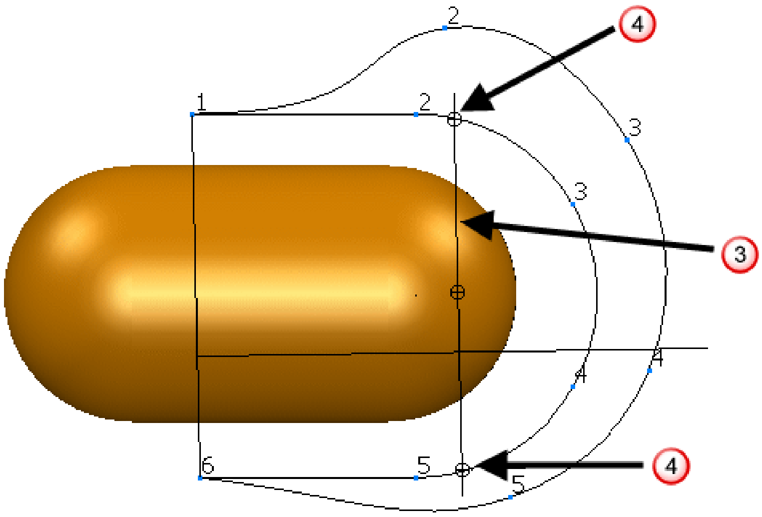
The reference axis 2 is defined at right angles to the line that joins the two ends of the reference curve. It lies in the plane of the curve if it is planar, else approximately so.



For each point to be morphed:

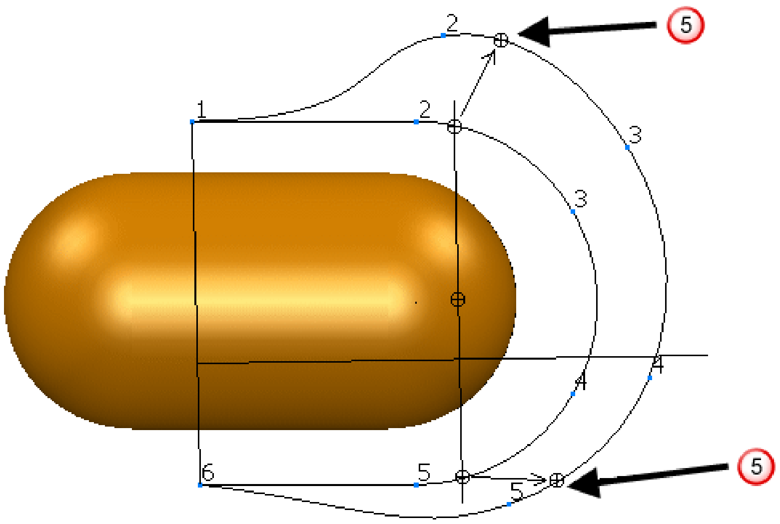
- a plane is defined through the point, normal to the reference axis 3.

- two points on the reference curve are defined where it intersects this plane 4.

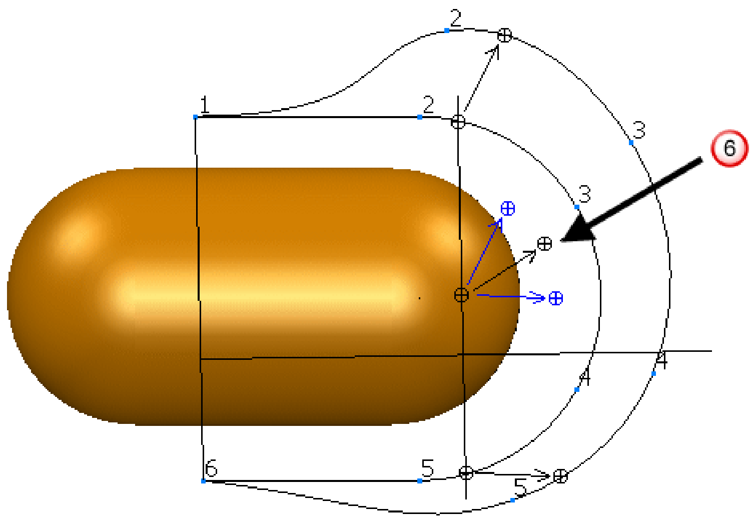


The curve must not be so highly curved that it crosses the plane more than twice, or the morph will fail.

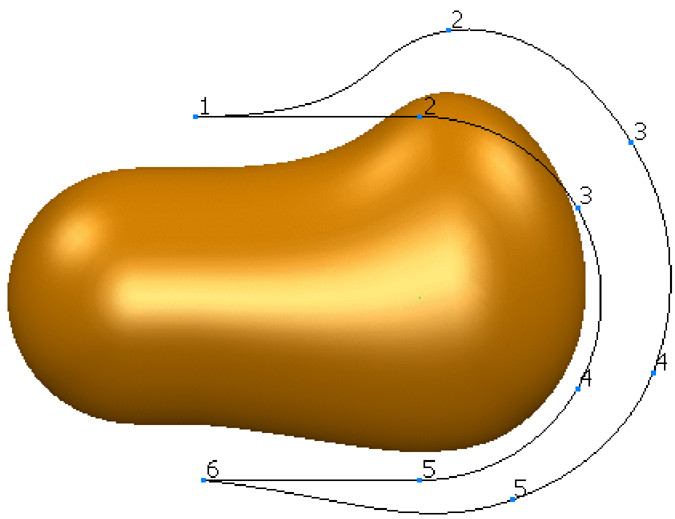
For each of the two points on the reference curve, a distance and direction is defined from it to the point with the same number on the control curve 5.



If the point to be morphed lies between the two points on the reference curve, it is moved by a combination of their distances and directions6.



Many points to be morphed can be identified with the same two points on the reference curve. They are not all moved by the same amount. The closer they are to one of the two points, the more influence its distance and direction has on their movement.



If the point to be morphed lies outside the two points on the reference curve, the closest point on the reference curve is found instead, in projection if a projection axis is defined. The point is moved by the distance and direction from the closest point to the corresponding point on the control curve, reduced by any decay.

Closed curve method

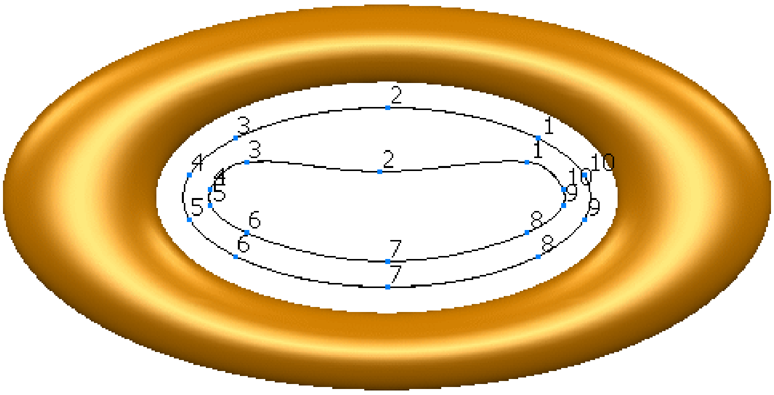
Points on the selected objects that don't lie on the reference curve are morphed using the closed curve method.

For each point to be morphed, the corresponding point on the reference curve is identified by one of three methods. Each method is selected by using a combination of the **In Projection** and **Use Reference** options:

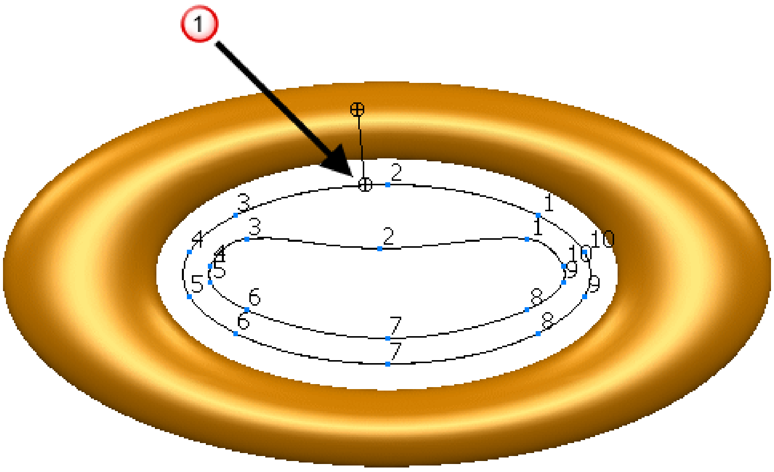
- Closed curve method - In Projection deselected (see page 262)
- Closed curve method - In Projection selected, Use Reference deselected (see page 264)
- Closed curve method - In Projection selected, Use Reference selected (see page 264)

Closed curve method - In Projection deselected

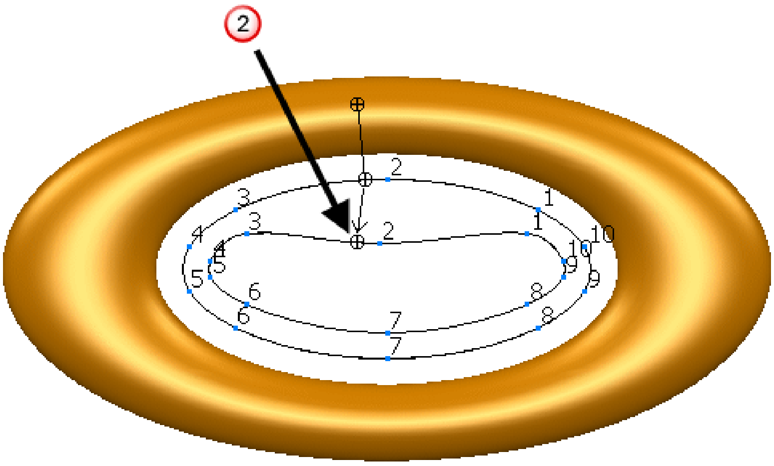
This morphing method works best for surfaces that lie entirely outside or on the reference curve.



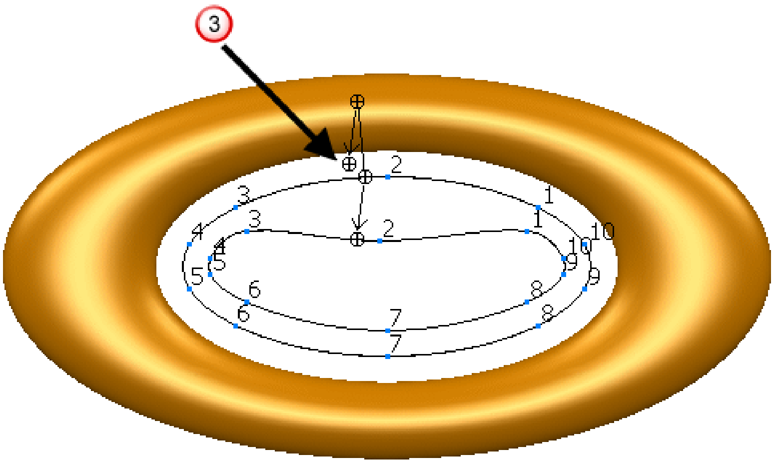
1 For each point on the selected curve, the closest point on the reference curve is identified 1.



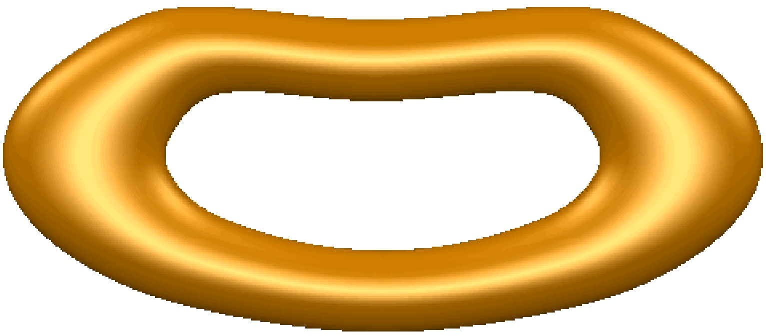
2 A distance and direction is defined from the point on the reference curve to the point with the same number on the control curve 2.



3 The point on the surfaces to be morphed is moved by this distance and direction, reduced by any decay that you may have defined 3.



4 Click on **OK** to produce the result below.



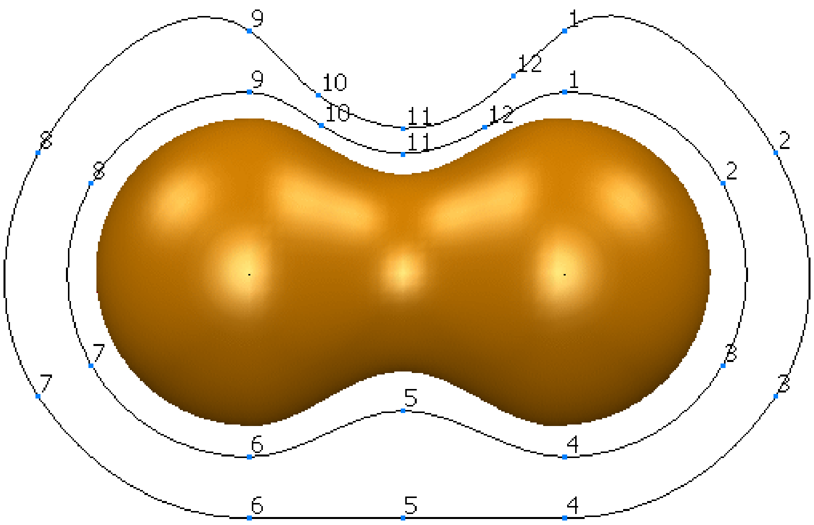
If a point lies inside and close to the centre of the reference curve, the closest point may lie on either the upper or lower side of the reference curve, leading to a gap in the morphed surface.

Closed curve method - In Projection selected, Use Reference deselected

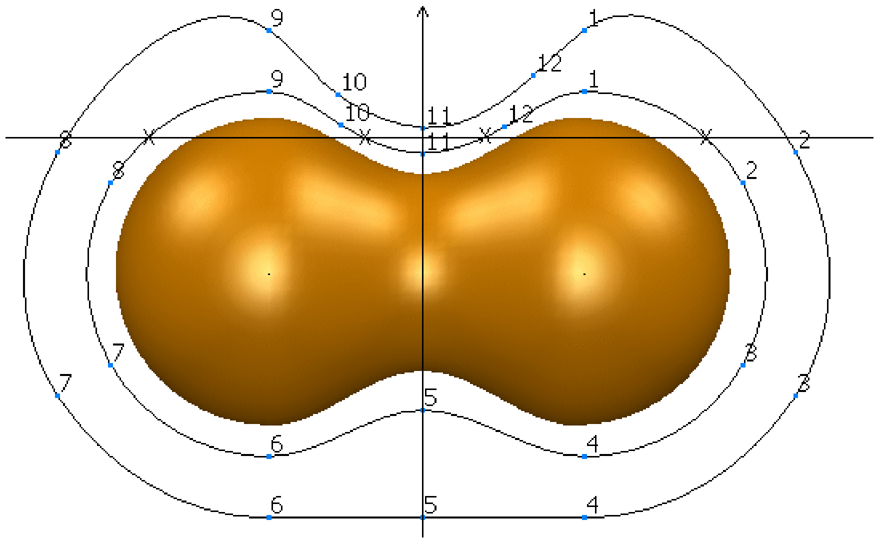
When In Projection is selected and **Use Reference** is deselected, the closest point on the reference curve is identified by first projecting the point and reference curve onto the principal plane. Apart from this, the method works in the same way as Closed curve method - In Projection selected (see page 262).

Closed curve method - In Projection selected, Use Reference selected

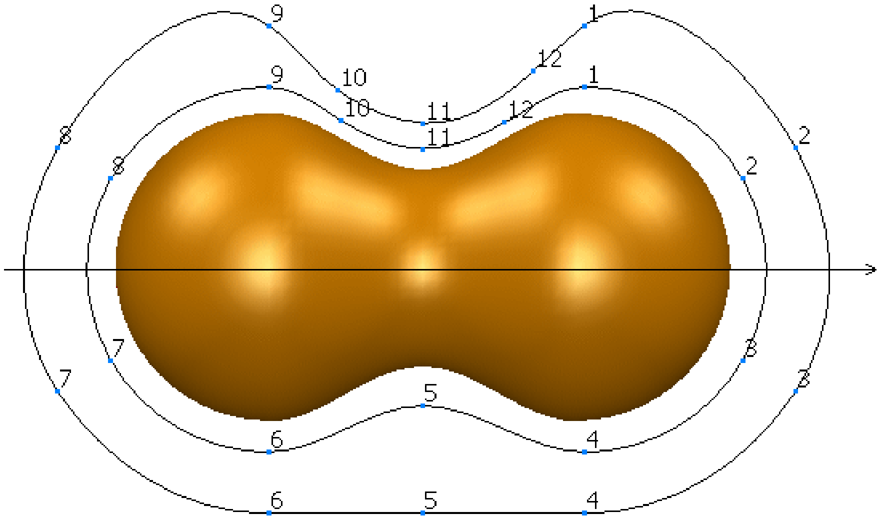
This method works in the same way as  on an open curve, but you must define a reference axis. Before the reference axis is used, it is projected onto the principal plane.



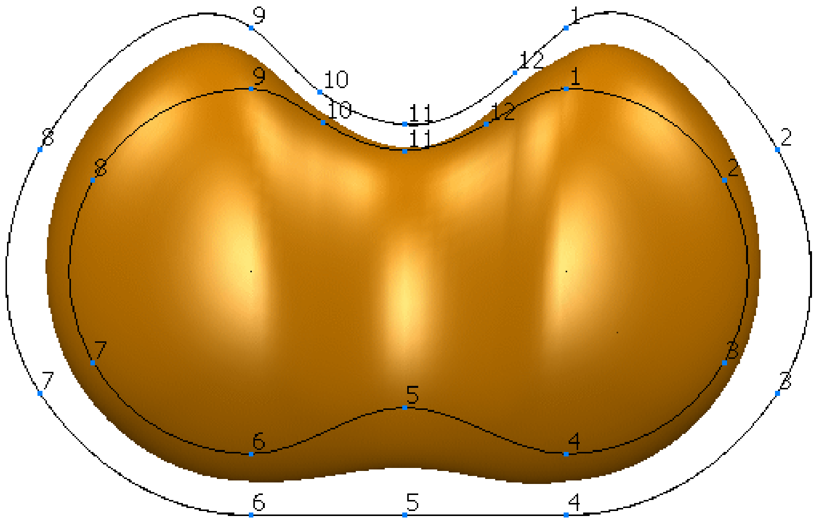
Define a suitable reference axis. In this example, the surfaces will fail to morph with a vertical axis. This is because when you define the plane that is normal to the reference axis, it goes through the reference curve more than twice for some points on the surfaces to be morphed.



For this example, use a horizontal axis. No plane normal to this axis crosses the reference curve more than twice.



This example gives the following result.



See Also:

Surface Morphing dialog - Two curves (see page 243)

Editing morph surfaces

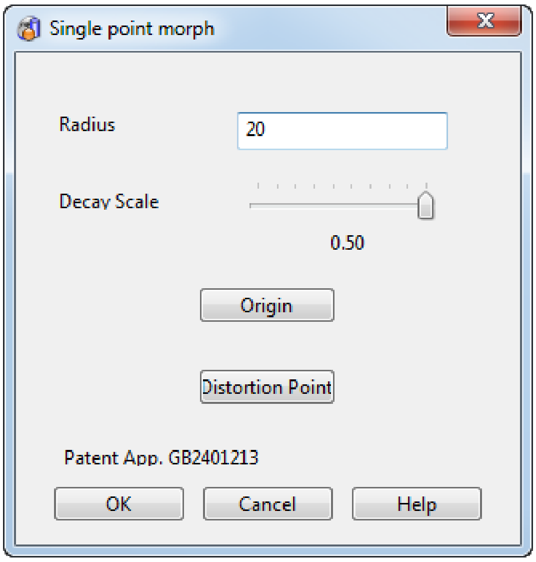
Edit an existing morph surface in one of the following ways:

- Use the tools on the Surface editing toolbar but only some of the options are available for morphed surfaces.
- Select the morph and select **Edit > Surface and Curve Edits > Surface Edits > Edit Morph** to displays the dialog for the particular type of morph.
- Select the morph with the right mouse button and choose **Edit Morph** from the context menu. This displays the dialog for the particular type of morph.
- Click the **Morph**  (*General Edits toolbar*) to display the dialog for the type of morph.

You cannot change the type of morph once the morph is defined.

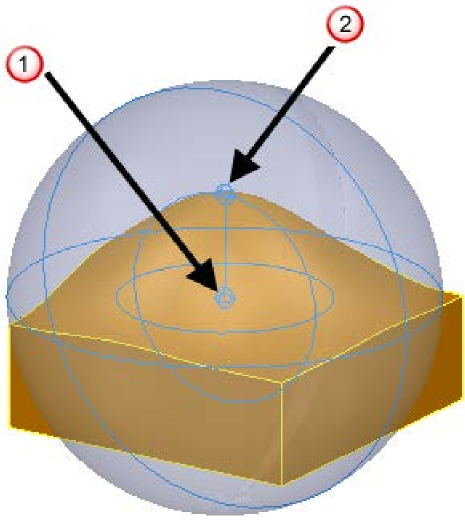
Single point morph dialog

Use this dialog to edit a single point morph.



When this dialog is displayed, the following are shown:

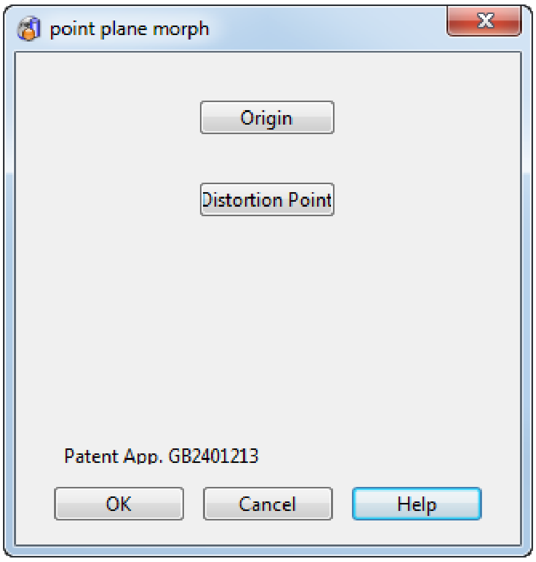
- Origin point 1
- Distortion point 2
- Two spheres around the origin point. The outer sphere shows the region enclosed by the radius and the inner sphere defines the **Decay Scale**.



Use the options on the dialog as described in Surface Morphing dialog - Single Point (see page 237)

Point/Plane morph dialog

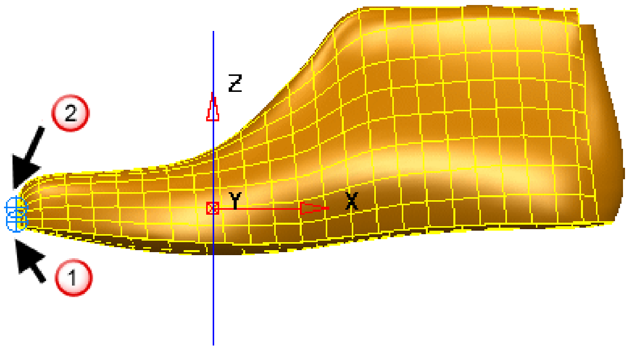
Use this to edit a morph that was created using a point and a plane.



When this dialog is displayed, the following are drawn:

- the origin 1.
- the distortion point 2.
- the workplane.

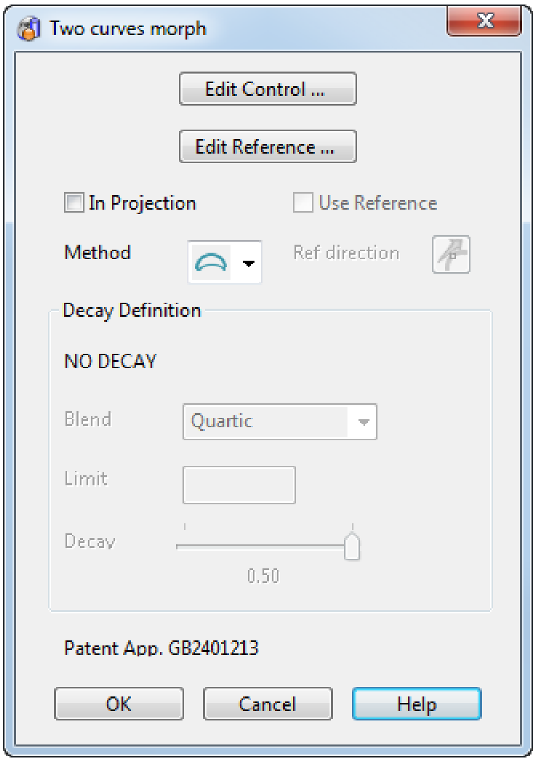
You edit the positions of the origin and distortion points, but not the workplane.



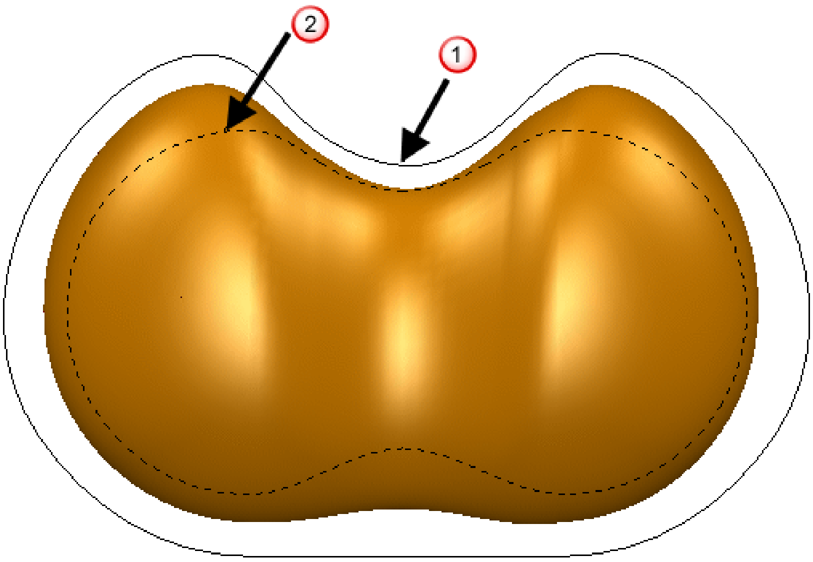
Use the options on the dialog as described in Surface Morphing dialog - Point/Plane (see page 240).

Two curves morph dialog

This lets you edit a two curves morph.



The reference curve is drawn as a dotted line and the control curve as a solid line.



If a decay curve or surface is used in the morph definition, it is also drawn as a dotted line. In shaded mode, the decay surfaces are shaded in a transparent material.

- Click **Edit Control** to edit the control curve 1.
- Click **Edit Reference** to edit the reference curve 2.

When you select **Edit Control** or **Edit Reference**, the relevant curve becomes selected and the **Curve Edit** toolbar is displayed. You can edit the curve graphically or using the toolbar. Some curve edits are not allowed as they destroy the morph definition.

To exit curve editing mode, select another item on the dialog.

- **Decay Definition** - You can only edit the decay if it was defined when you created the morph. The method cannot be changed.

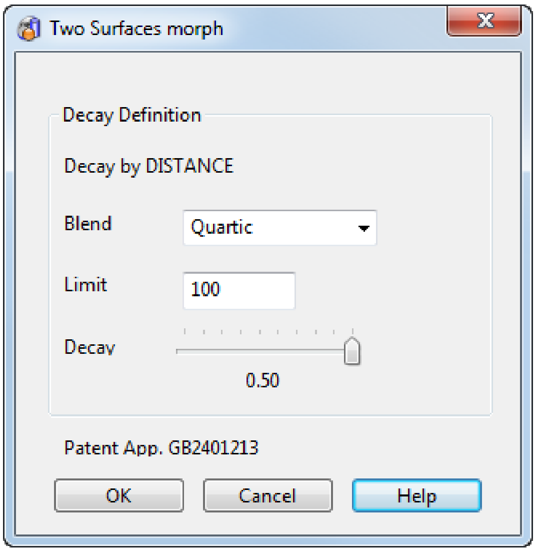
  For **Curve** and **Surface** decays, you can only change the blend. The selected objects cannot be changed.

  For the **Distance** method, you can change the blend, limit and decay scale.

The remaining options on this page appear on the Surface Morphing dialog - Two Curves (see page 243).

Two surfaces morph dialog

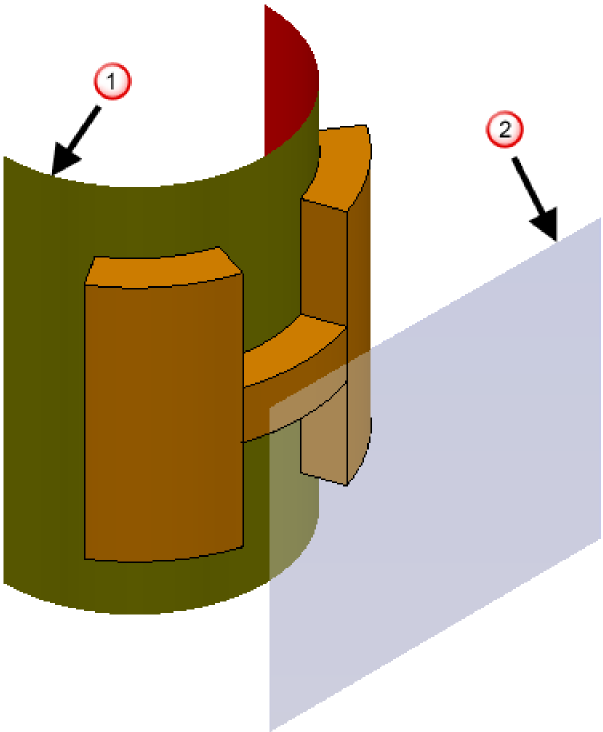
Use this dialog to edit a two surfaces morph.



When the dialog is displayed, the following are shown:

1 Control surface.

② Reference surface. In wireframe mode, the reference surface is drawn using dotted lines. In shaded mode, it is drawn in a transparent material.



If a decay curve or surface is used in the morph definition, it is also drawn as a dotted line. In shaded mode, the decay surfaces are shaded in a transparent material.

The **Decay Definition** can only be edited if it was defined when you created the morph. The method cannot be changed.

- For **Curve** and **Surface** decays, you can only change the blend. The selected objects cannot be changed.
- For the **Distance** method, you can change the blend, limit and decay scale.

The **Decay** options appear on the Surface Morphing dialog - Two Surfaces (see page 249)

Trimming surfaces

Use the following sections to trim surfaces:

How do I trim surfaces?

Many real objects consist of several surfaces, some of which are trimmed. It is convenient to describe such objects by regular surfaces, together with a set of trim boundaries for each surface, so that when the object is drawn, the trimming is performed automatically.

You can trim surfaces using the following options on the **Limiting** flyout (*General Edits toolbar*):

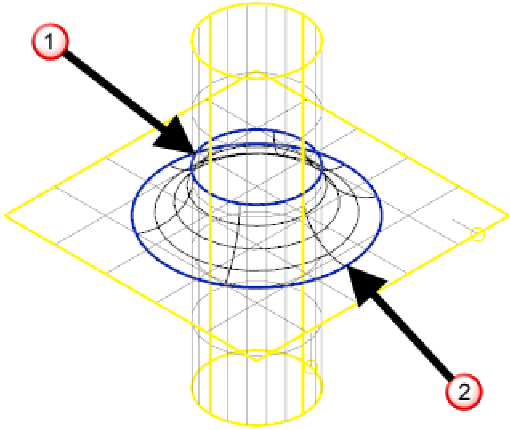
 **Limit Selection**

 **Limit Point**

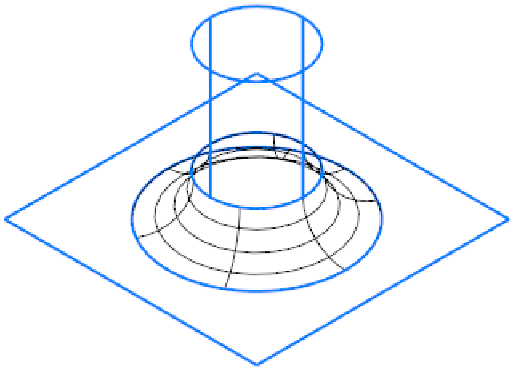
What is a trim boundary?

A trim boundary or a set of trim boundaries on a surface define the trim region of a surface.

On the surfaces shown, two trim boundaries exist: one on the cylinder 1 and the other on the plane 2.



These surfaces can be trimmed back to give the following:



A trim boundary consists of pcurves and surface edges joined together to form a closed region of a surface.For further details, see What is a pcurve? (see page 282).

How do I create a trim boundary?

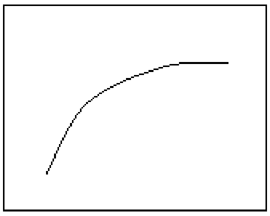
Trim boundaries are created automatically when:

- creating fillet surfaces.
- creating draft surfaces.
- limiting surfaces using **Limit Selection** and **Limit Point.**

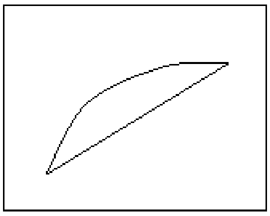
Editing a trim boundary

A trim boundary must form an enclosed region on a surface, otherwise the trim region is not defined.

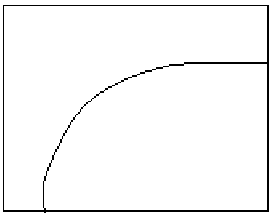
When you generate a trim boundary, you may end up with one as shown below.



From this boundary, it is difficult to define the trim region. Is the trim region the following?



Or is it?



PowerSHAPE provides tools to help you define your trim region accurately. Sometimes, the pcurves may cause spikes or even loops in the trim boundaries. These tools can also be used to remove spikes and loops.

1. Select the surfaces on which the trim boundary lies.

   The Boundary editing toolbar (see page 274) is displayed.

2. Use the **Boundary Editing** toolbar to edit the trim boundary.

Boundary editing toolbar

Each button on the **boundary editing** toolbar is described below.

 Display the boundary editing toolbar

 Display pcurve editing toolbar (see page 283)

 Generates diagnostics on boundaries (see page 275)

 Selects all the boundaries (see page 279) on the selected surface.

 Displays/removes the boundary selector (see page 278).

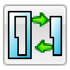
 Removes one-point spikes (see page 279) from the selected boundary.

 Removes loops (see page 279) from the selected boundary.

 Closes a boundary (see page 280).

 Recreates boundaries (see page 280). Deletes any selected boundary and enters boundary creation mode.

 Automatically creates boundaries (see page 280) from the pcurves on the surface. Any existing boundaries are deleted.

 Swaps the trim region (see page 281) defined by a set of trim boundaries (for example, it converts a "hole" into an "island").

 Deletes the selected boundary (see page 281) (without deleting its pcurves).

Most operations are available when multiple surfaces are selected.

Generating diagnostics on boundaries

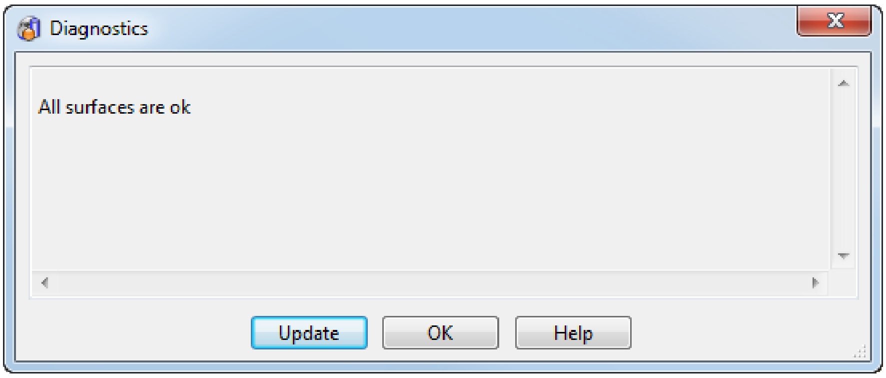
Badly defined pcurves (that can lead to badly trimmed surfaces) can be identified using **Identify badly trimmed surfaces** on the **Model Fixing** toolbar. You can closely analyse boundary data to see where any problem areas exist and then manually correct them.

1 Select the surfaces.

2 Click  (*boundary editing toolbar*). This displays the **Diagnostics** dialog.

Diagnostics dialog

This dialog displays details about trim boundaries (if in boundary editing) and pcurves (if in pcurve editing) on the selected surfaces.



In the window, a warning is displayed if the boundaries cannot form a trim region. If a corrupt boundary is found, you are told to either delete and recreate it or use the **File Doctor**.

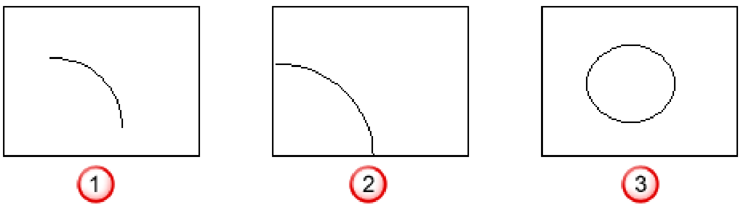
This window also displays the following information about each boundary:

- whether the boundary is open or closed;
- whether the boundary has self intersections;
- whether the boundary crosses any other boundaries;
- whether the boundary has a region with zero area.

Each of the above is discussed in more detail.

Closed or open - An open boundary cannot define a trim region. You must edit the boundary to close it.

A closed boundary can be defined using the outer edges of its surface.

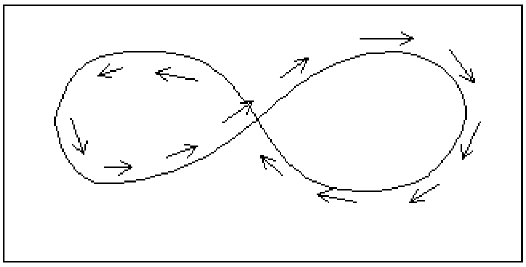


1 Open boundary.

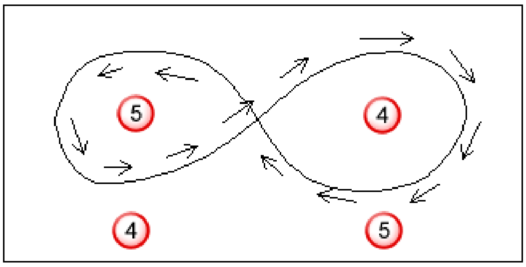
2 Closed boundary (using surface edges to define a valid trim region).

3 Closed boundary.

Self intersections - If a boundary intersects itself, a loop exists (see figure below). The visible region of the surface lies on the left of the boundary. Arrows are drawn to show the left of the boundary.



As you can see in the figure below, we have a region that is both invisible 4 and visible 5. Therefore, such a boundary cannot define a trim region.



You must edit the boundary to remove the loop.

Crossing other boundaries - If a boundary crosses another, the trim region cannot be defined. The reason is similar to the self intersecting boundary. You must edit the boundaries to stop them from crossing.

Region with zero area - PowerSHAPE attempts to find regions where the area is zero. This is where the boundary goes back on itself. In such cases, the following warning may display

A section of boundary 2 has zero area

check pcurve 3 around point 12

This is an approximate position of where the error is. The named point may not be the problem itself but the surrounding points will be.

*Only the first error is reported. We recommend you fix the problem by editing the necessary points. Then click **Update** on the **Diagnostic** dialog to find any further problems.*

Similarly, you can analyse the pcurves on the surface. For further details, see Generating diagnostics on pcurves (see page 287).

Update - You can manually edit the boundary and then click **Update**. This updates the information to the new selection. If you change to pcurve editing and then click this button, information about the pcurves is displayed. For further details, see Generating diagnostics on pcurves (see page 287).

OK - Removes the dialog from the screen.

Selecting boundaries

Use the boundary selector to select multiple boundaries on multiple surfaces.

1 Select the surfaces.

2 Click  (*Boundary editing toolbar*).

3 Use the dialog to select boundaries.

Select the boundaries from the list. A single click selects a single item. Pressing the **Shift** key when clicking a single item, selects all the items from the last selected item to the clicked item. Pressing the **Control** key when clicking a single item, adds/removes the clicked item from the current selection.

You can also select boundaries by clicking them on the screen.

Surface - This contains the names of the selected surfaces. You can choose another surface from the drop down list and its boundaries are displayed in the list.

Any selected boundaries remain selected when you use the drop-down list to select another surface.

4 Click **OK** to confirm the boundaries and remove the dialog from the screen.

Selecting all boundaries

You can select all the boundaries on selected surfaces.

1 Select the surfaces.

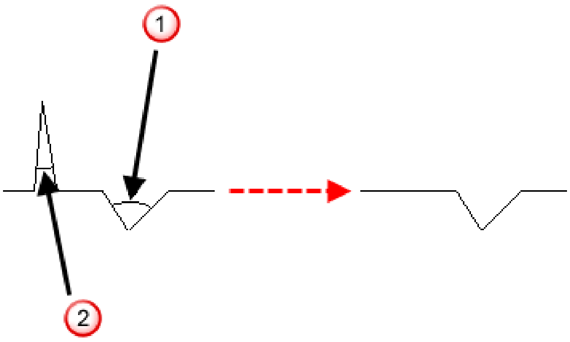
2 Click  (*Boundary editing toolbar*).

Removing spikes from a boundary

This deletes points in the selected boundary where the angle between the line segments is less than angular tolerance.

1 Select the boundary.

2 Click  (*Boundary editing toolbar*).



1 Angle greater than angular tolerance.

2 Angle less than angular tolerance.

To change the angular tolerance:

1 From the **Tools** menu, select **Options** to display the **Options** dialog.

2 Select **Units and Tolerances** options.

3 Change the **Angular tolerance** to the required value.

4 Click **OK**.

Removing loops from a boundary

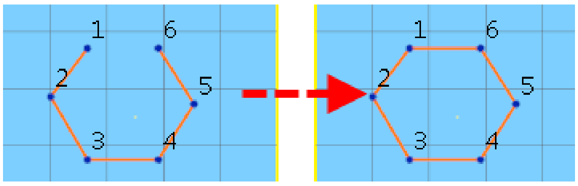
This deletes points in the selected boundary which cause loops.

1 Select the boundary.

2 Click  (*Boundary editing toolbar*).

Closing a boundary

This closes a boundary by adding a pcurve to link the end point of the last pcurve in the selected boundary to the start point of the first pcurve.



1 Select a boundary.

2 Click  (*Boundary editing toolbar*).

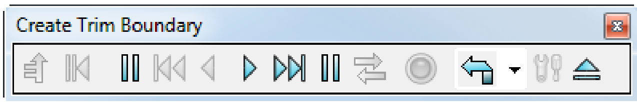
Recreating a boundary

You can recreate trim boundaries on surfaces using a direct method similar to composite curve creation.

1 Select the boundary if you wish to delete it.

2 Click  (*Boundary editing toolbar*).

3 Use the **Create Trim Boundary** control panel to create and delete the boundary



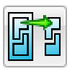
This control panel works in exactly the same way as the **Create Composite Curve** control panel.

Automatically creating boundaries

You can automatically create boundaries from the pcurves on a surface. Any existing boundaries are deleted before creating new ones.

Ideally, the pcurves should lie in sensible positions to define unique boundaries, otherwise you'll have to use the boundary editing tools to change which pcurve belongs to which boundary.

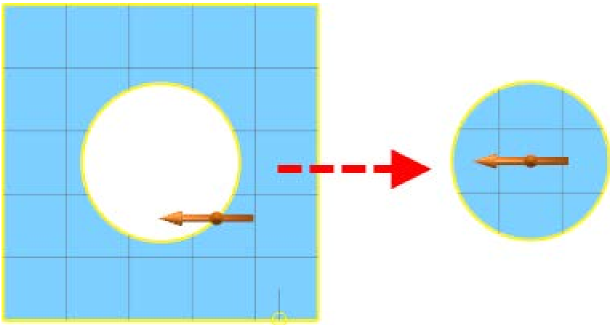
1 Select the surfaces.

2 Click  (*Boundary editing toolbar*).

If boundaries already exist, you are warned that they will be deleted.

Swapping the trim region

This swaps the trim region by switching the visible and invisible regions of the surface.



1 Select the boundary.

2 Click  (*Boundary editing toolbar*).

Deleting a boundary

On the **Boundary editing** toolbar, two commands exist for deleting a boundary: one command deletes the boundary and takes you into boundary creation mode and the other command just deletes the boundary. These commands do not delete the pcurves. If you delete the boundary using the **Main** toolbar, then the boundary and its pcurves are deleted.

Just deleting a boundary

1 Select the boundary.

2 Click  (*Boundary editing toolbar*).

If no boundaries are selected, but the surface has exactly one boundary then the boundary is exploded.

Deleting a boundary and entering boundary creation

1 Select the boundary.

2 Click  (*Boundary editing toolbar*).

The selected boundary is deleted. You can now create a new boundary using the **Create Trim Boundary** control panel.

Deleting the boundary and its pcurves

1 Select the boundary.

2 Click the **Delete** button  on the main toolbar or press **Delete**.

Creating composite curves from boundaries

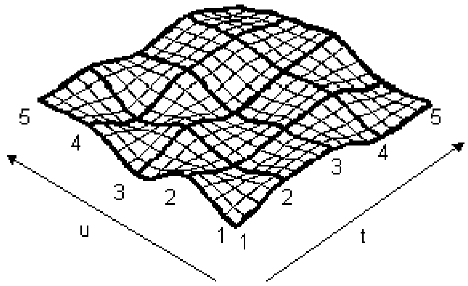
While the **Boundary editing** toolbar is displayed, you can use **Edit > Convert > to wireframe** to create composite curves from selected boundaries.

What is a pcurve?

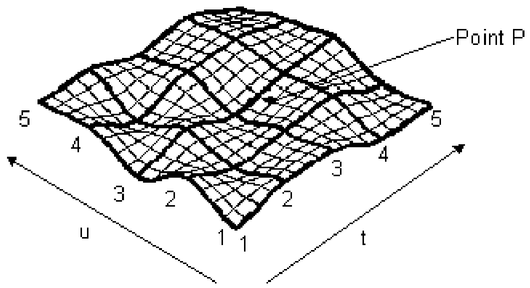
A pcurve is a parameter curve which is generated on a surface by certain operations. These operations include:

- creating fillet surfaces
- creating draft surfaces
- limiting surfaces using the **Limit Selection** and **Limit Point** commands

A pcurve is used to define the trim boundary of a surface and is made up of a series of line segments which match the surface to within tolerance. A pcurve is a parameter curve because the curve is defined in the 2D parameter space of a surface. A pcurve contains points which are called ppoints. Every surface is defined by parameters **t** and **u** which vary across the surface.



The values of **t** and **u** uniquely define a point, **p**, on the surface. Thus point **p** can be defined by 3D world space coordinates (x, y, z) or by 2D parameter space coordinates (**t**, **u**). In the figure below, the 2D parameter space coordinates of **p** are (3.5, 2.75).



World coordinates are not guaranteed to lie on the surface, whereas parameter space coordinates must by definition lie on the surface (the third dimension is the surface). Curves used as part of trim boundaries of surfaces must lie exactly on the surface being trimmed, and so pcurves are defined within the 2D parameter space of a surface. They cannot therefore be operated on in quite the same way as other curves (for example, they cannot be dragged off a surface) and a special editor is provided to manipulate them.

How do I create a pcurve?

Pcurves are usually generated automatically when performing certain operations on surfaces and not created from scratch. These operations include:

- creating fillet surfaces
- creating draft surfaces
- limiting surfaces using the **Limit Selection** and **Limit Point** commands.

How do I edit a pcurve?

Occasionally when you generate a pcurve, it may need adjusting slightly. Editing a pcurve also changes the boundary containing it.

1. Select the surfaces on which the pcurve lies. The **pcurve editing** toolbar will be displayed.

   All pcurves on the selected surfaces are displayed.

2. Select the appropriate button from the toolbar.

Pcurve editing toolbar

The **pcurve editing** toolbar consists of the following buttons:

 Display the boundary editing toolbar (see page 274).

 Display pcurve editing toolbar.

 Generates diagnostics on pcurves (see page 287).

 Selects all pcurves (see page 285) on the selected surface.

 Selects all unused pcurves (see page 285) on the selected surface.

 Displays/removes (see page 284) the pcurve selector.

Removes one-point spikes (see page 288) from the selected pcurve.

Removes loops (see page 289) from the selected pcurve.

Removes surplus points (see page 289) to simplify the selected pcurve.

Labels the ppoints (see page 289) on pcurves.

Turn point labels off (see page 289).

Makes pcurves from wireframe (see page 290).

Copy pcurves (see page 293).

Extends the selected pcurve (see page 294).

Cuts the selected pcurve (see page 297) at the selected point into two pcurves.

Opens the pcurve (see page 297).

Closes the pcurve (see page 297).

Deletes the pcurve (see page 298).

Displays/removes the ppoint selector (see page 286).

Edits the parametric value (see page 298) at the selected ppoint.

Inserts ppoints into a pcurve (see page 298).

Deletes the selected ppoint (see page 305) on a pcurve.

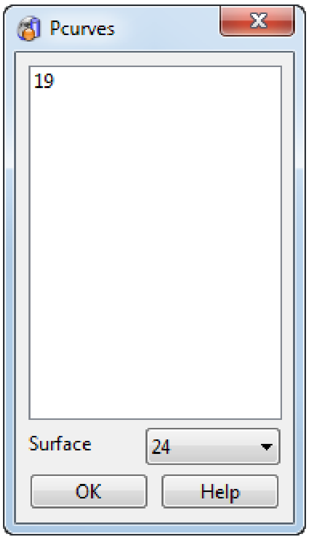
Most operations are available when multiple surfaces are selected.

Selecting pcurves

You can use the pcurve selector to select multiple pcurves on multiple surfaces.

1 Select the surfaces.

2 Click  (*Pcurve editing toolbar*).



3 Use the dialog to select pcurves.

List of pcurves - This is the list of pcurves on the surface displayed in the **Surface** drop down list. You can use this list to select the pcurves. A single click selects a single item. Pressing the **Shift** key when clicking a single item, selects all the items from the last selected item to the clicked item. Pressing the **Control** key when clicking a single item, adds/removes the clicked item from the current selection.

You can also select pcurves by clicking them on the screen.

Surface drop down list - This contains the names of the selected surfaces. You can choose another surface from the drop down list and its pcurves are displayed in the list.

Any selected pcurves remain selected when you use the drop down list to select another surface.

4 Click **OK** to confirm the changes and remove the dialog from the screen.

Selecting all pcurves

You can select all the pcurves on selected surfaces.

1 Select the surfaces.

2 Click  (*Pcurve editing toolbar*).

Selecting unused pcurves

You can select all pcurves on surfaces which are not used to define a trim boundary.

1 Select the surfaces.

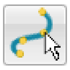
2 Click  (*Pcurve editing toolbar*).

All unused pcurves are selected.

Selecting points on pcurves

You can use the ppoint selector to select multiple ppoints on surfaces.

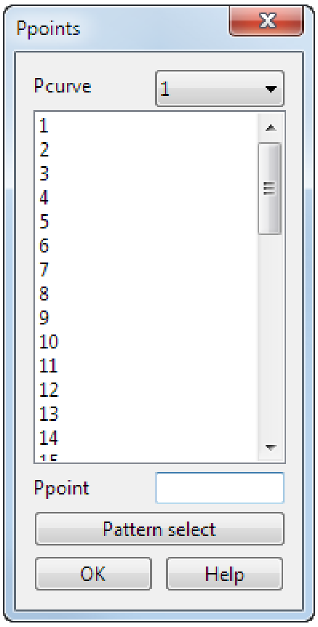
1 Select the surfaces.

2 Click  (*Pcurve editing toolbar*).

3 Use the Ppoints dialog (see page 286) to select ppoints.

Ppoints dialog

Use this dialog to select ppoints.



A single click selects a single item. Press the **Shift** key when clicking a single item selects all the items from the last selected item to the clicked item. Pressing the **Ctrl** key when clicking a single item, adds or removes the clicked item from the current selection.

PCurve — Select a different pcurve using the drop-down list.

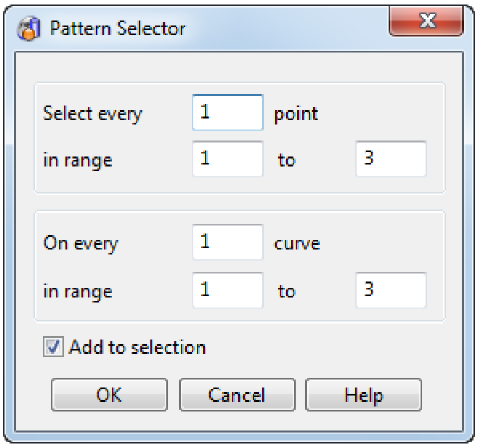
*Any selected ppoints remain selected when you use the **PCurve** drop down list to select another pcurve.*

Ppoint — Enter a ppoint number into the field and press the **Enter** key to select it.

Pattern select - Click to display the Pattern selector dialog (see page 287).

Pattern Selector dialog

Use this dialog to select ppoints (on the pcurve currently in the curve drop down list on the Ppoints selector dialog (see page 286)) using simple pattern sequences.



Select every ***n*** **point** - This selects every nth ppoint starting at the first point in the range, where **n** is the number you input. If you enter **3**, then the first ppoint in the range is selected followed by every 3rd ppoint.

In range ***a*** **to** ***b*** - This is range of the ppoints to select from.

For example, to select all even ppoints between 1 and 50, you would enter 2 in the **Select every Point** text box, and values 2 and 50 in the **Range** text boxes.

Add to selection - You can add the selection to the existing selection of ppoints by turning this option on. To make a fresh selection, turn this option off.

OK - Makes the selection.

For further details, see Selecting pattern of points (see page 116).

You can also select ppoints by clicking them on the screen.

Generating diagnostics on pcurves

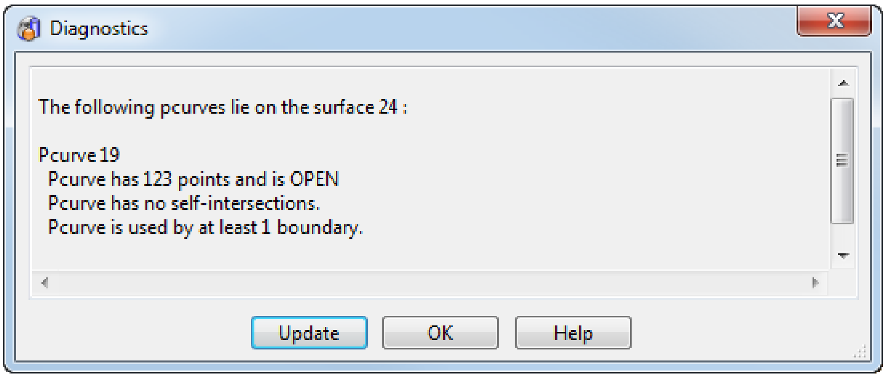
Badly defined pcurves (that can lead to badly trimmed surfaces) can be identified using the **Identify badly trimmed surfaces** (model fixing toolbar). You can closely analyse pcurves to see where any problem areas exist.

1 Select the surfaces.

2 Click  (*Pcurve editing toolbar*). This displays the **Diagnostics** dialog.

Diagnostics dialog (pcurves)

This dialog displays details about pcurves (if in pcurve editing) and trim boundaries (if in boundary editing) on the selected surface.



In the main window, the following information is displayed about each pcurve:

- whether the pcurve is open or closed;
- whether the pcurve has self intersections;
- whether the pcurve is used in any boundary.

Each of the above is given for information purposes and does not necessarily indicate a problem.

Closed or open - Open pcurves are not a problem, providing they link with other pcurves to create a closed boundary.

Self intersections - If a pcurve self intersects, it contains a loop. This may be the cause of loops within a boundary.

Used in any boundary - If a pcurve is not used in a boundary, you may delete it or use it to create a new boundary.

Similarly, you can analyse the boundaries on the surface. For further details, see Generating diagnostics on boundaries (see page 275).

Update - Click to update the edited pcurves. This updates the information. If you change to boundary editing and click this option, information about the boundaries is displayed. For further details, see Generating diagnostics on boundaries (see page 275).

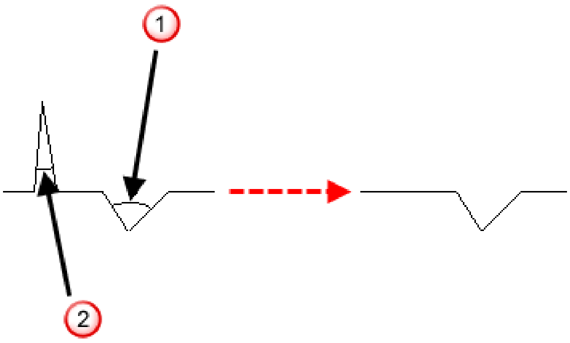
Dismiss - Removes the dialog from the screen.

Removing spikes from a pcurve

This deletes points in the selected pcurve where the angle between the line segments is less than angular tolerance.

1 Select the pcurve.

2 Click  (*Pcurve editing toolbar*).



1 Angle greater than angular tolerance.

2 Angle less than angular tolerance.

To change the angular tolerance:

1 From the **Tools** menu, select **Options** to display the **Options** dialog.
2 Select the **Units and Tolerances** option.
3 Change **Angular tolerance** to the required value.
4 Click **OK**.

Removing loops from a pcurve

This deletes points in the selected pcurve which cause loops.

1 Select the pcurve.
2 Click  (*Pcurve editing toolbar*).

Removing surplus points from a pcurve

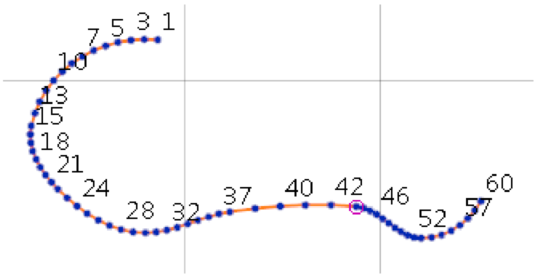
This deletes points in the selected pcurve which are coincident (within tolerance).

1 Select the pcurve.
2 Click  (*Pcurve editing toolbar*).

Labelling points on a pcurve

1 Select the pcurve.
2 Click  (*Pcurve editing toolbar*).

Points on the pcurve are now labelled.



Only a subset of points are labelled. This helps you to see the labels, because typically points are very close together.

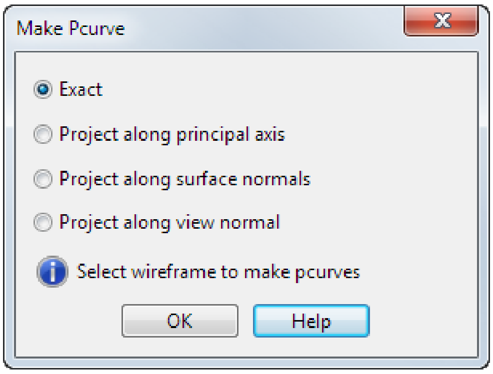
Making pcurves from wireframe

This allows you to make pcurves from wireframe lying on a surface, within tolerance, or by projecting wireframe onto a surface.

1. Select the surface.
2. Click  (Pcurve editing toolbar).
3. Use the Make Pcurve dialog (see page 290) to create the pcurves.

Make Pcurve dialog

Use this dialog to create pcurves from wireframe.

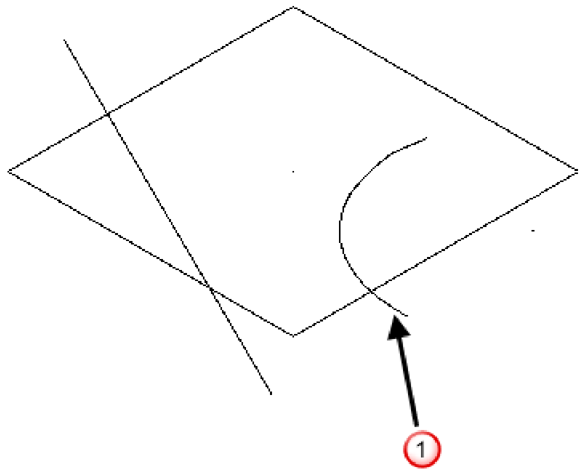


1. Select how you want to create the pcurves.
2. Select the wireframe to create the pcurves from.
3. Create pcurves.
4. Use the **Undo** or **Redo** buttons on the **Main** toolbar. to undo or redo any pcurves you create.
5. Click **OK** when you have finished creating pcurves.

   The pcurves are then created automatically.

Exact — This allows you to make pcurves from wireframe lying on the surface.

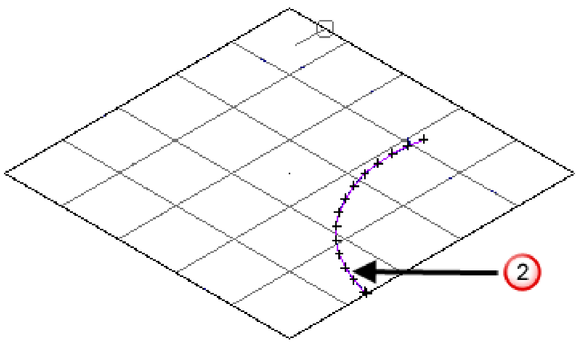
After you have selected **Exact**, you need to select the wireframe 1.



Only select wireframe which is lying on the surface as any wireframe lying above or below the surface cannot be used to make pcurves with this method.

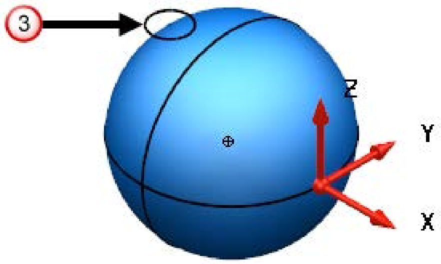
The pcurves created are only made with the wireframe touching the surface and any part of the wireframe which extends beyond the surface is ignored. This is shown below 2.

The wireframe has not been drawn to show the pcurves more clearly.

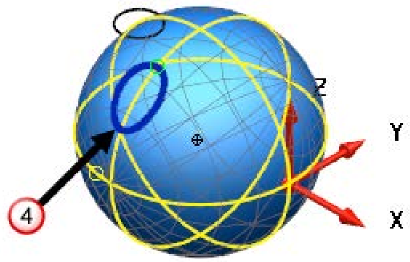


Project along principal axis — Select this option to create pcurves on a surface by projecting the selected wireframe along the current principal axis, independent of view direction. This can be thought of as shining a light onto the wireframe down the principal axis and creating a pcurve from its shadow.

Once you have selected **Project along principal axis**, you need to select the wireframe to project onto the surface 3. You can select any wireframe, regardless of whether it lies on the surface.

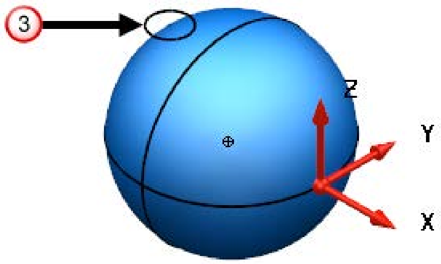


The pcurves are created by projecting the wireframe selection onto the surface, along the principal axis 4. In the example below the principal axis was set in the Z direction.

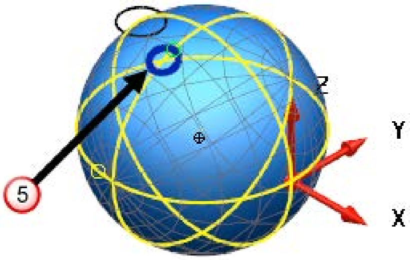


Project along surface normals — Select this option to create pcurves on a surface by projecting the selected wireframe along the surface normal, independent of view direction. This can be thought of as shining a light onto the wireframe down the surface normal axis and creating a pcurve from its shadow.

Once you have selected **Project along surface normals**, you need to select the wireframe to project onto the surface 3. You can select any wireframe, regardless of whether it lies on the surface.

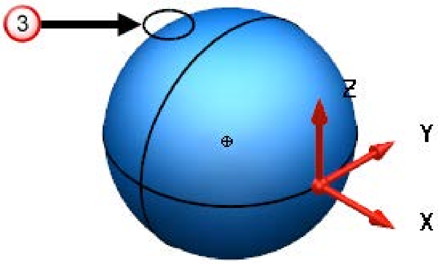


The pcurves are created by projecting the wireframe selection onto the surface, along the surface normal 5.

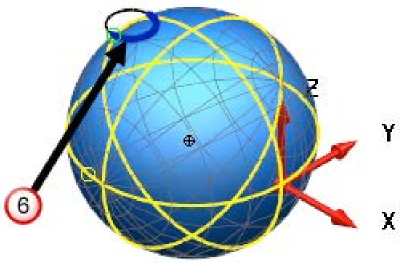


Project along view normal — Select this option to create pcurves on a surface by projecting the selected wireframe along the view normal, independent of the principal axis. This can be thought of as shining a light onto the wireframe through the screen and creating a pcurve from its shadow.

Once you have selected **Project along view normal**, you need to select the wireframe to project onto the surface 3. You can select any wireframe, regardless of whether it lies on the surface.



The pcurves are created by projecting the wireframe selection onto the surface, along the view normal 6. In the example below the view direction is ISO 1.



OK — Click this to close the dialog.

Copy pcurves

This allows you to copy pcurves across surfaces.

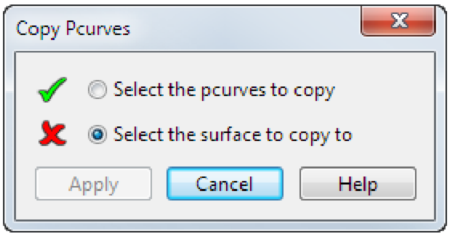
1 Select the surface.

2 Click  (Pcurve editing toolbar).

3 Use the **Copy Pcurves** dialog (see page 294) to copy the pcurves.

Copy Pcurves dialog

Use this dialog to copy pcurves across surfaces.



To copy pcurves across surfaces:

1. Select the pcurves you want to copy. You can select multiple pcurves.
2. Select the surface you want to copy to.
3. Click **Apply**.

   The pcurves are copied to the new surface.

Extending a pcurve

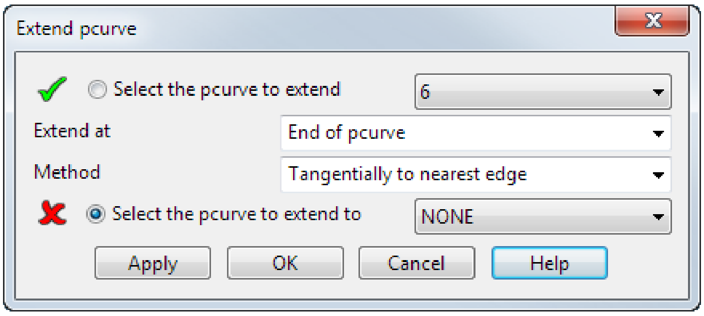
This extends the selected pcurve to another position on the surface.

1. Select the pcurve.
2. Click  (*Pcurve editing toolbar*).
3. Use the dialog to extend the pcurve.

Extend pcurve dialog

This provides tools to extend the selected pcurve to another position on its surface.

Point labels are automatically displayed when the **Extend pcurve** dialog is displayed. The button to toggle **Turn point labels** on and off is available on the toolbar; click  to hide the labels.



*The curve extension options cannot work if the selected pcurve is closed. Closed curves can be opened using the **Cut the pcurve** option.*

Select the pcurve to extend — Select this option then graphically select a pcurve on your surface, or select a pcurve from the drop-down list to extend from.

Extend at — This allows you to choose which end of the pcurve to extend. The point numbers shown on the selected pcurve allow you to see which are the start and end points.

You can graphically set the Extend at option by clicking the graphic handles at the end of each pcurve. Click the handles to toggle extension of or off.

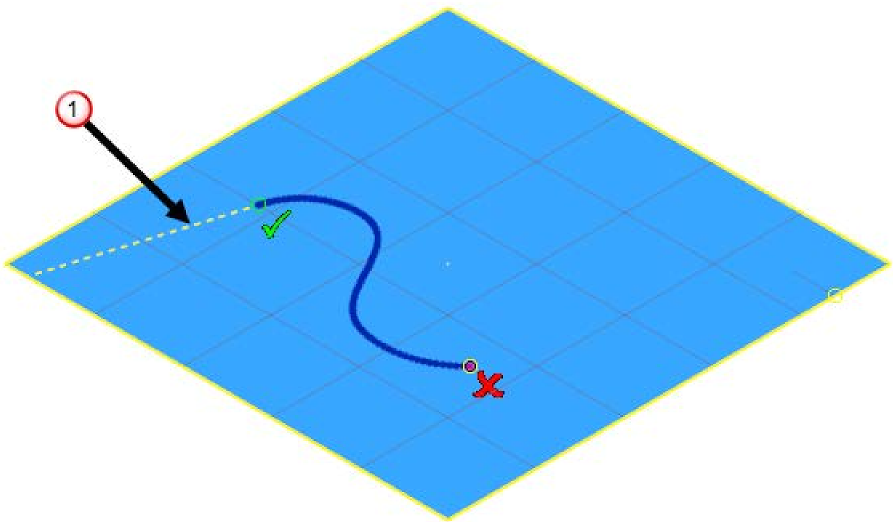
- ✓ — This handle indicates that the Pcurve is extended from this end.
- ✗ — This handle indicates that the Pcurve is not extended from this end.

You can initialise the Extend at option by pre-selecting points on the pcurve before opening the dialog:

- If the start point is pre-selected, the option is set to **Start of pcurve**.
- If the end point is pre-selected, the option is set to **End of pcurve**.
- If both the start and end points are pre-selected, the option is set to **Both ends of pcurve**.
- If no points are selected, the option defaults to the last set value.

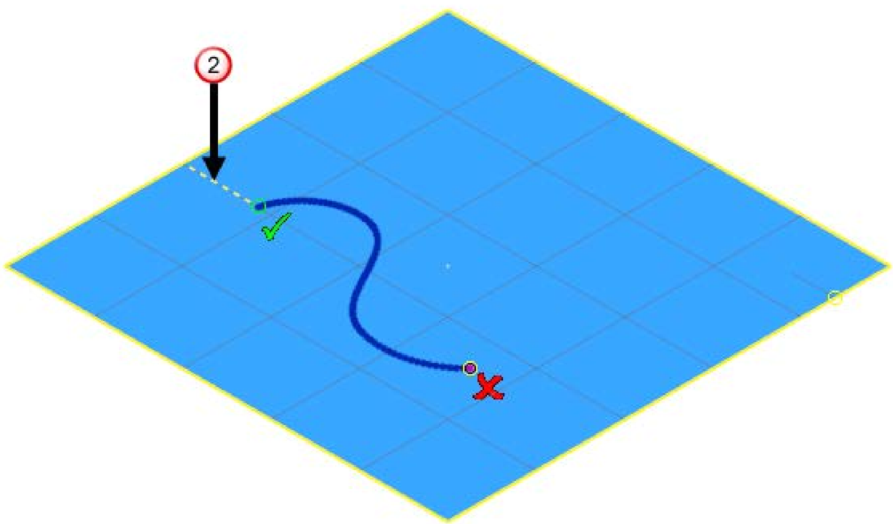
Method — This allows you to choose how and where to extend the pcurve. The option menu contains the following options:

- **Tangentially to nearest edge** — Extends the pcurve along the tangent(s) at its chosen end point(s) to the nearest edge(s) of the surface 1.



*The picture shows one end extended, but both ends can be extended if the **Both ends** option is selected from the **Extend At** menu.*

- **By shortest route to nearest edge** — Extends the pcurve along the shortest route to the nearest edge of the surface 2.



*The picture shows one end extended, but both ends can be extended if the **Both ends** option is chosen from the **Extend At** menu. Each will take the shortest route, which may not be to the same edge of the surface.*

- **Tangentially to a pcurve** — Extends the selected pcurve, along the tangent at its end point, to another pcurve within the same surface. If the surface contains more than two pcurves, the **Extend to pcurve** drop—down list becomes active, allowing you to choose.
- **By shortest route to a pcurve** — Extends the selected pcurve by the shortest distance to another pcurve within the same surface. If the surface contains more than two pcurves, the **Extend to pcurve** drop—down list becomes active, allowing you to choose a pcurve.

Select the pcurve to extend to — Select this option then graphically select a pcurve on your surface, or select a pcurve from the drop down list to extend your initially selected pcurve to.

Apply — Extends the pcurve.

OK— Extends the pcurve and closes the dialog.

Cancel — Closes the dialog and does not extend the pcurve.

Cutting a pcurve into two

This cuts the selected pcurve into two at the selected point.

1 Select a pcurve.

2 Select a point on the pcurve.

3 Click  (*Pcurve editing toolbar*). The names of the pcurves are defined as follows:

   If the name of the selected pcurve was 3, then one pcurve is 3 and the other 3A.

Opening a pcurve

This removes the span which joins the first and last points of the pcurve.

1 Select a pcurve.

2 Click  (*Pcurve editing toolbar*).

Closing a pcurve

This adds a span to the first and last points of the pcurve.

1 Select a pcurve.

2 Click  (*Pcurve editing toolbar*).

Deleting a pcurve

1. Select the pcurve.
2. Click  (*Pcurve editing toolbar*).

   Any trim boundary defined by this pcurve is also deleted.

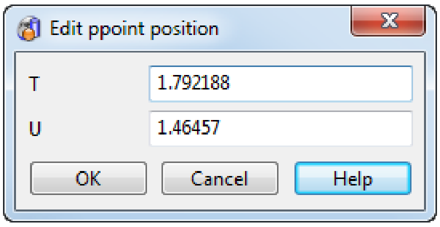
Editing a point on a pcurve

This allows you to edit the t and u parameters of a point on a pcurve.

1. Select the pcurve.
2. Select the point you want to edit.
3. Click  (*Pcurve editing toolbar*).
4. Use the dialog to edit the selected point.

Edit ppoint position dialog

This allows you to edit the position of a point.



T — Enter a value to define the t parameter value for the selected point.

U — Enter a value to define u parameter value for the selected point.

A preview is displayed on the model when you edit a ppoint position.

For further details, see What is a pcurve? (see page 282).

If you click any position on the surface, its t and u values are read into the dialog.

OK — Click apply any changes and move the point to the new position.

Cancel — Click to close dialog without making any changes.

Inserting a point in a pcurve

You can insert ppoints which either preserve or change the shape of the pcurve.

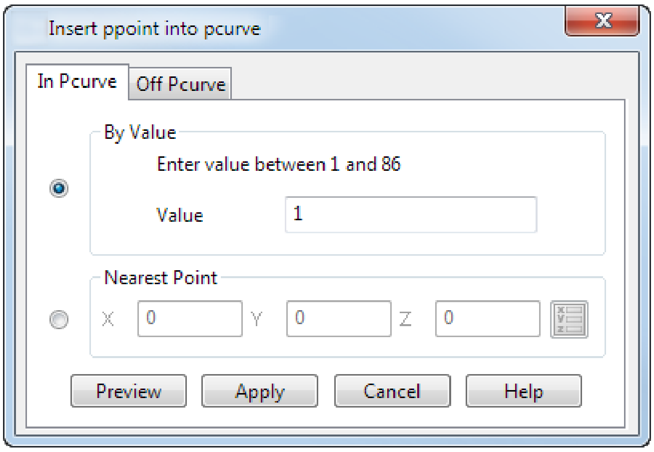
1 Select one pcurve.

2 Click  (*Pcurve editing toolbar*).

3 Use the Insert ppoint into pcurve dialog (see page 299) to insert ppoints.

Insert ppoint into pcurve dialog

Use this dialog to insert ppoints.



There are two tabs:

In PCurve — Inserts ppoints without changing the shape of the pcurve. See In PCurve tab (see page 300).

Off PCurve — Inserts ppoints to change the shape of the pcurve. See Off PCurve tab (see page 302).

Each of the above is discussed in a separate section below.

Other buttons on the dialog are:

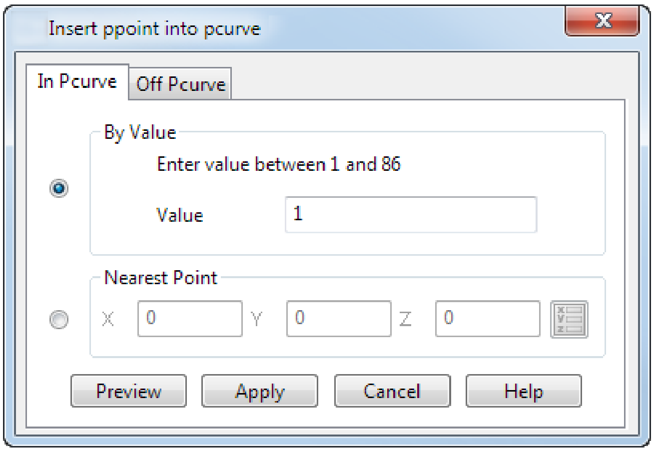
Preview — Displays the new ppoint in the pcurve. You can change the settings on the dialog until you are satisfied with the previewed pcurve.

Apply — Inserts a new ppoint into the pcurve.

Dismiss — Removes the dialog from the screen.

In PCurve tab

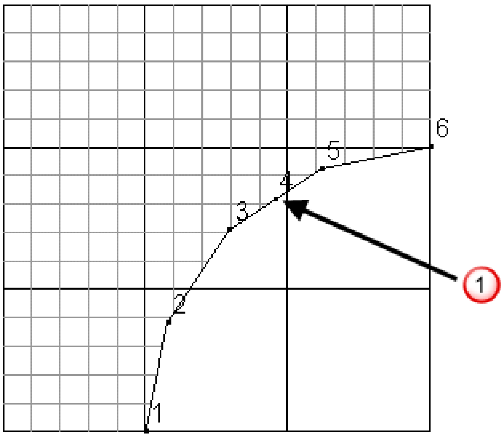
Use this page of the dialog to insert ppoints without changing the shape of the pcurve.



There are two ways to insert ppoints using this page of the dialog:

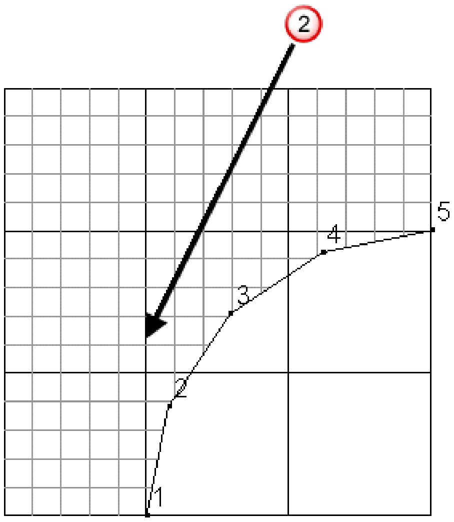
- **By Value** — This inserts a ppoint at a specific parameter value. Enter the parametric value of where you want the new ppoint to lie in the pcurve.

  If you enter 3.5, a new ppoint with label 4 1 is added to the pcurve and other ppoints are renumbered.

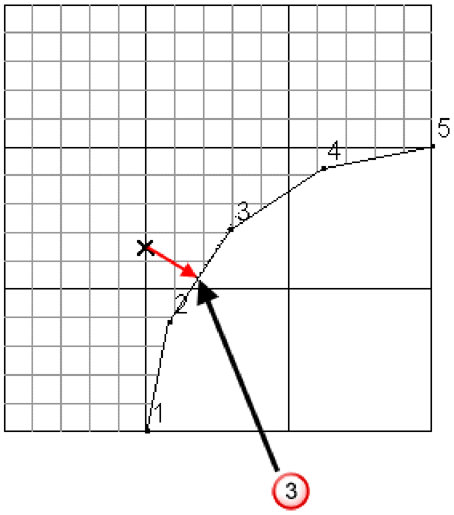


- **Nearest Point** — This inserts a ppoint on the pcurve nearest a specified point. Enter the **X Y Z** coordinates or click the **Position** button to open the **Position** dialog where you can use position entry tools.

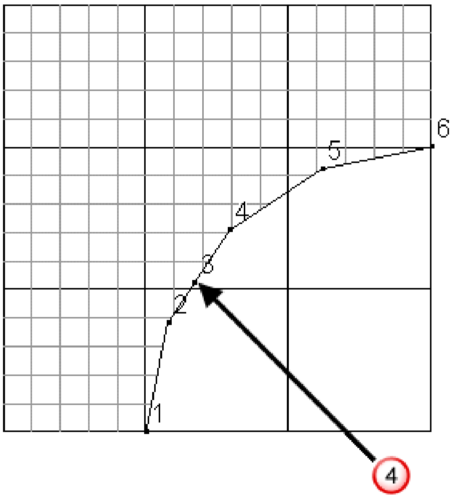
1 Enter the X, Y, Z coordinates. In the following example, the coordinates match the indicated point 2.



2 The position is found on the pcurve nearest to these coordinates 3.



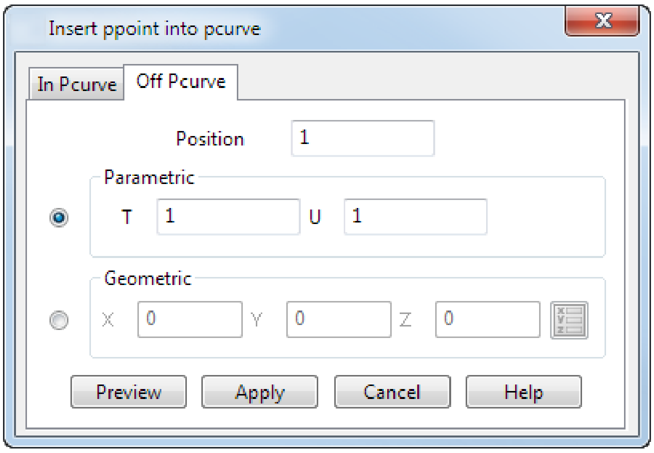
3 A ppoint is inserted at that position. The new ppoint has label 3 and other ppoints are renumbered 4.



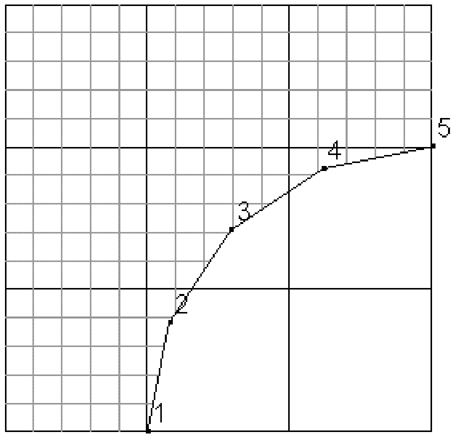
You can also enter a position by clicking on the screen.

Off PCurve tab

Use this page of the dialog to insert ppoints by changing the shape of the pcurve.



The following example illustrates the options on this dialog.

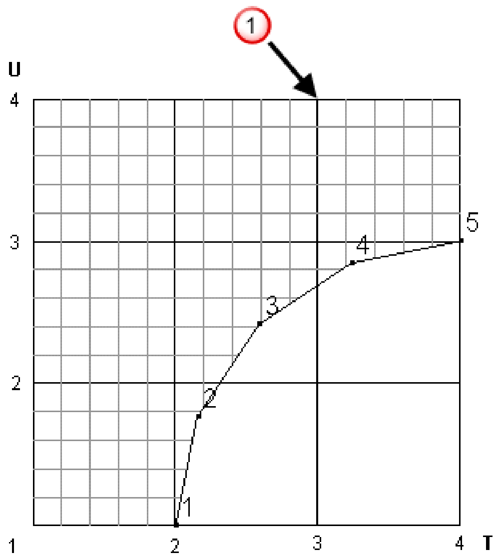


Position - This defines the position of the new ppoint in the pcurve. For example, if you enter 3, the new ppoint becomes 3 and all the original ppoints from 3 onwards are renumbered.

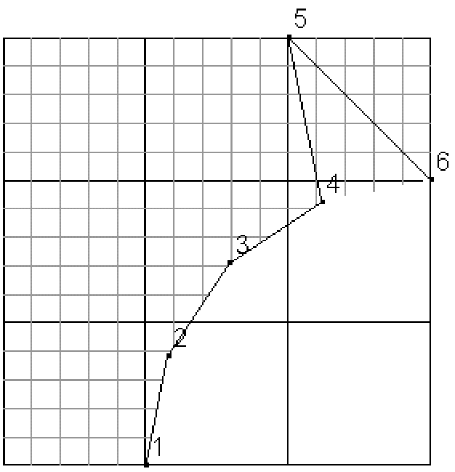
There are two ways to insert ppoints using this page of the dialog:

- **Parametric** - This inserts a ppoint into the pcurve at the **T** and **U** parameters on the surface. See What is a pcurve? (see page 282) for further details about t and u parameters.

  In this example, you want to insert a new ppoint at **T** = 3 and **U** = 4 1.

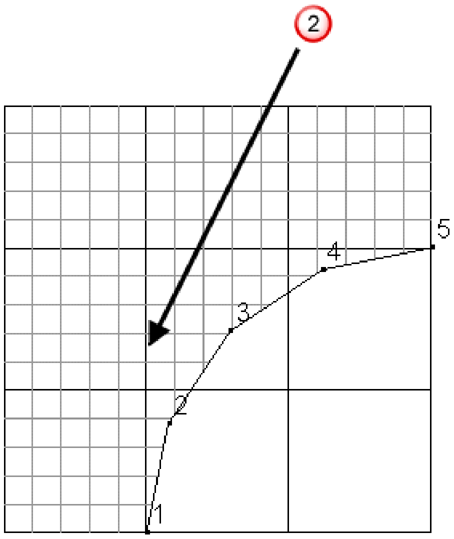


To make the new ppoint lie between ppoints 4 and 5, you need to enter **5** in the **Position** box. The result is shown in the image below:

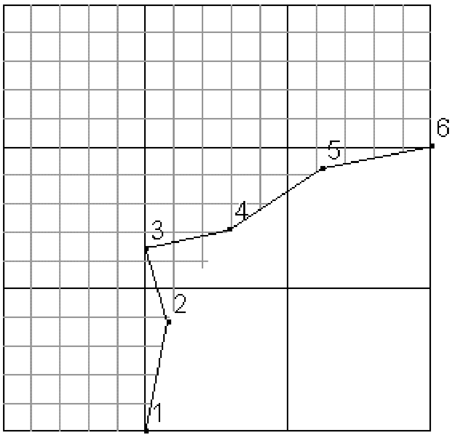


- **Geometric** - This enters a ppoint at the position on the surface nearest to the specified point. Enter the **X Y Z** coordinates or click the **Position**  button to open the **Position** dialog where you can use position entry tools. You can also enter a position by clicking on the screen.

  1 Enter the X, Y, Z coordinates. In the following example, the entered coordinates match the following position :



2 To make the new ppoint lie between ppoints 2 and 3, enter **3** in the **Position** box. The result is shown in the image below:



Deleting a point on a pcurve

1 Select the pcurve.

2 Select the point on the pcurve.

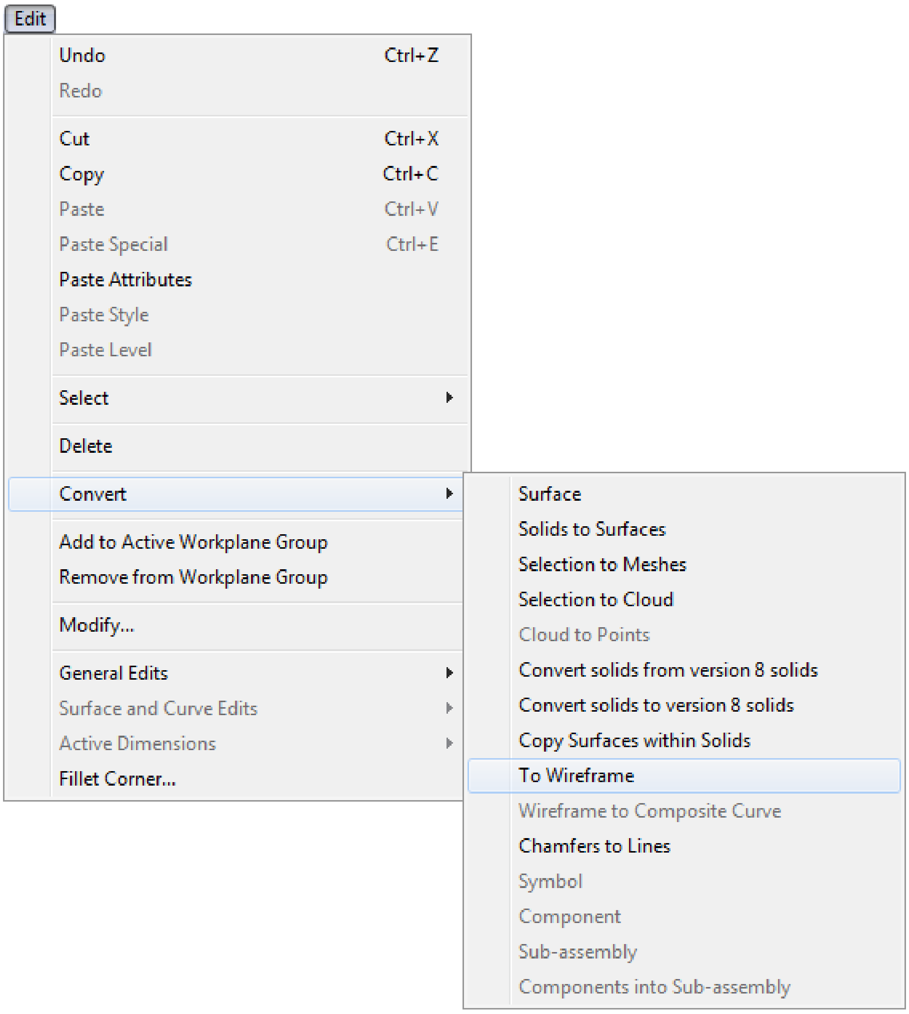
3 Click  (*Pcurve editing toolbar*).

Creating wireframe curves from pcurves

With the **pcurve editing** toolbar displayed,

1 From the **Edit** menu, select **Convert.**

2 From the **Convert** sub-menu, select **To wireframe** to create curves from selected pcurves.

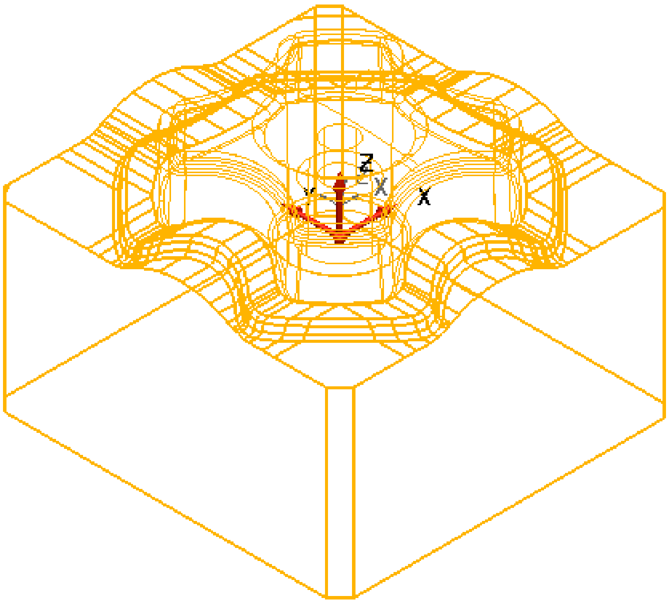


Preparing for manufacture

The following sections provide information on preparing for manufacture.

Introduction to preparing for manufacture

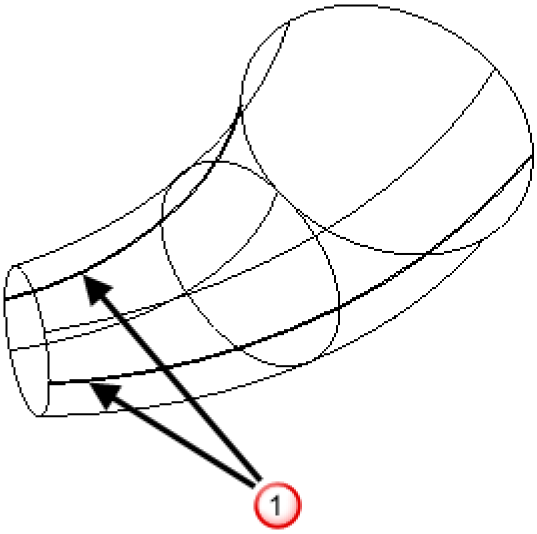
Split and draft surfaces play an important role in designing components and tooling for material-forming applications. In most mold or die applications it is vital to establish the natural split line, and to design a split surface for machining. In most cases, you must use a draft angle in the surface of a mold or die to enable the product to be withdrawn.



In PowerSHAPE, you can create split and draft surfaces. There is also a wizard to help you create a mold for a single, connected solid.

What is a split line?

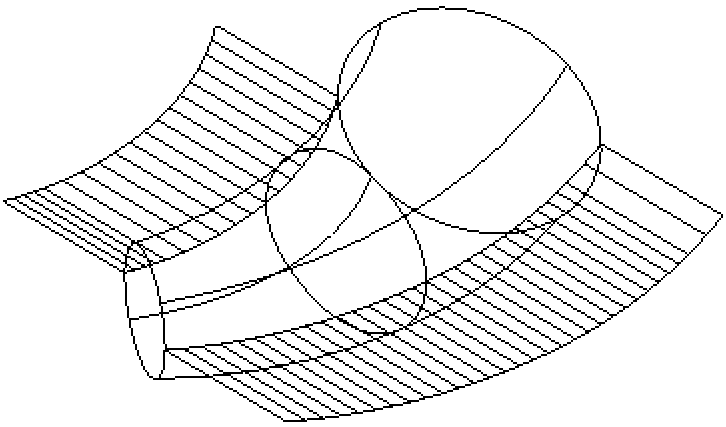
A surface's natural split line or occluding boundary is the line around a surface where the surface, when looked at along the draw axis (that is, the direction in which the mold or die is to open), just dips out of sight 1.



PowerSHAPE can find the natural split line automatically and generate new curves to represent this line. For further details see Creating a draft curve in the **Wireframe modelling Manual**. Alternatively, you can specify a curve by creating a composite curve from existing geometry, for example, edges of surfaces. For further details see Creating a composite curve by tracing in the **Wireframe modelling Manual**.

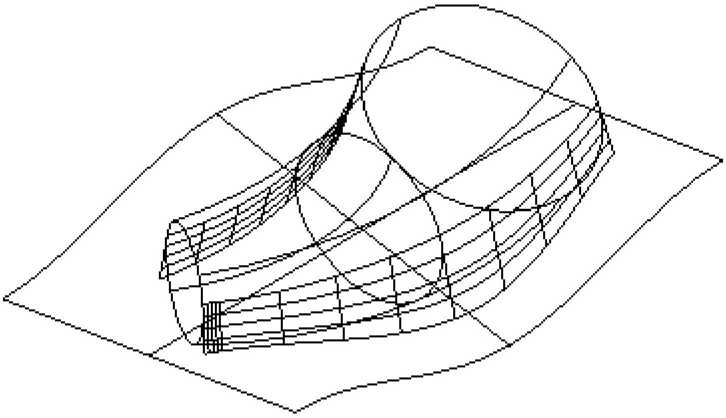
What is a split surface?

A split surface is a surface created from the split line and is also known as a 'run-off' surface.



What is a draft surface?

A draft surface is one which projects from a model onto a split surface at a small angle, as shown below. The draft angle ensures that the component will separate from the mold cavity.



Creating a split surface

You can create split surfaces using two methods:

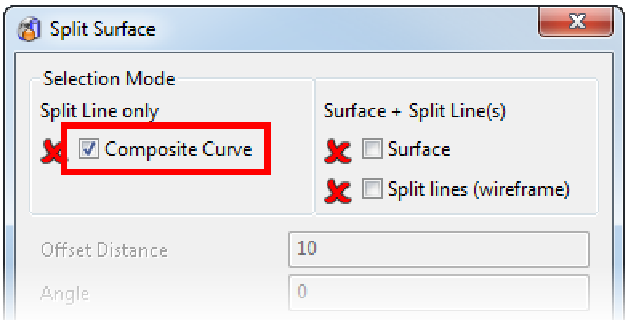
- offsetting a single split line
- fitting an existing surface to split lines

Creating a split surface by offsetting a single split line

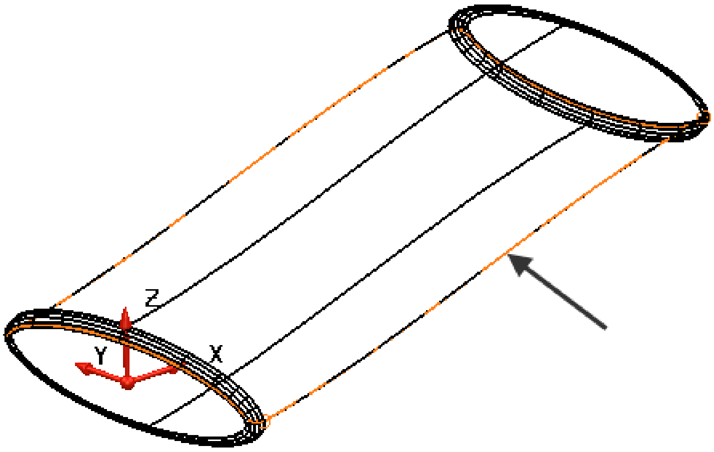
The split surface is created by selecting a composite curve as the split line. This split line is then offset by a specified distance and along a specified angle. One side of the split surface is the split line and the other the offset of the split line.

1 Click  (*Surface toolbar*).

2 Select the **Composite curve** option from the **Split Surface** dialog.



3 Select the composite curve which defines the split line.



4 Fill in the dialog.

Creating a split surface by fitting a surface to split lines

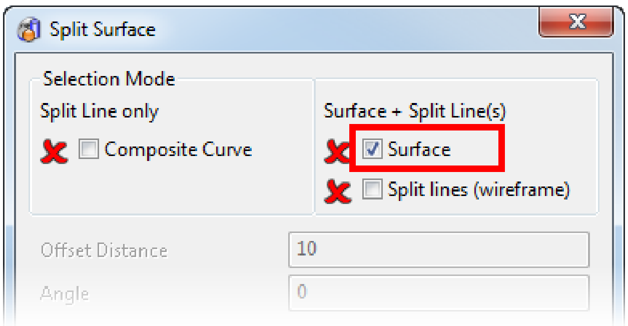
The split surface is created by modifying an existing surface to fit split lines. The split surfaces:

- preserve the outline of the surface and the tangency at the outline.
- fit the wireframe at directions parallel to the principal plane.

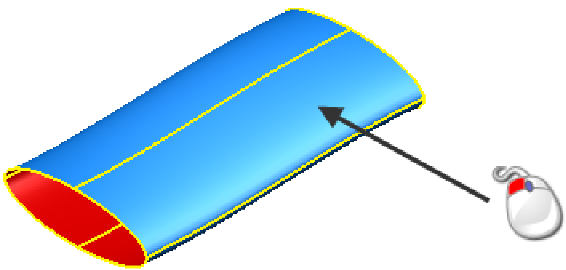
PowerSHAPE will also attempt to trim off the surface inside of the wireframe.

1 Click  (*Surface toolbar*).

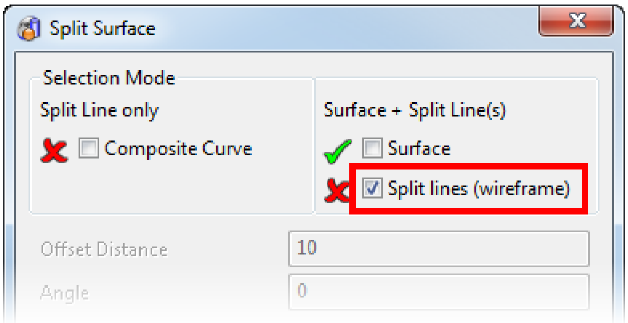
2 Select the **Surface** option from the **Split Surface** dialog.



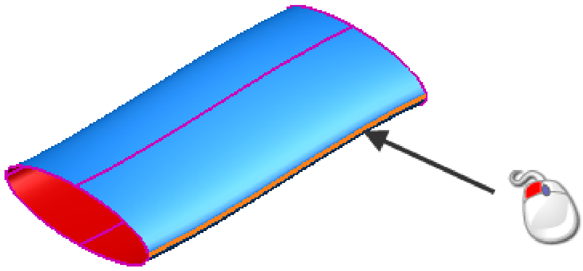
3 Select the surface.



4 Select the **Split lines** option from the **Split Surface** dialog.

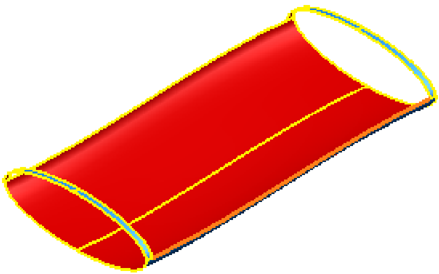


5 Select the section of wireframe along which you want to split the surface.



6 Click **Preview** to see the result.

PowerSHAPE generates the split surfaces and attempts to trim off the surface inside of the wireframe.

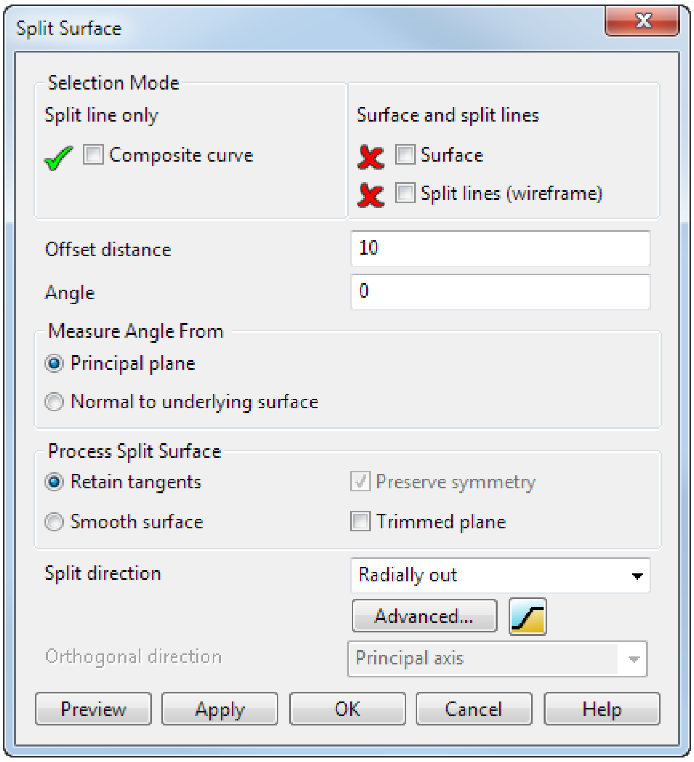


7 Change the selection if necessary.

8 Click **OK** when you have the required results.

Split Surface dialog

Use this dialog to create a split surface.



Selection Mode - To add objects to a selection mode, turn on a mode and then select the objects.

> **Split line only** - Use the **Composite curve** mode to select the composite curve for the method of creating a split surface by offsetting a split line. The **Composite curve** option is preselected if a composite curve is present in the model. You can then select the required composite curve, if it is not already selected.
>
> **Surface and split Lines** - Use the **Surface** and **Split lines (wireframe)** modes to select the surface and wireframe for the method of creating a split surface by fitting a surface to split lines. There are no other options on this dialog for this method.

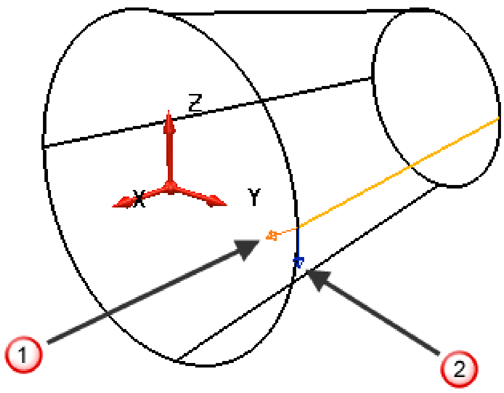
The rest of the dialog contains options for the method of creating a split surface by offsetting a split line.

Offset distance - This is the distance of the second side of the split surface from the split line.

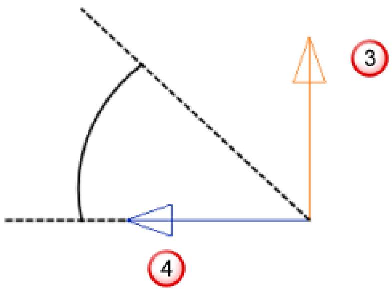
Angle - This is the angle of the second side of the split surface from the split line. Two arrows are drawn at right angles (normal) to the composite curve and are used to measure the angle. The direction of the arrows is determined by the **Measure Angle From** options.

Measure Angle From - You can create the split surface relative to either the principal plane or the normal of the underlying surface. The directions of the two arrows on the split line are determined by these options:

Principal plane - One arrow points parallel to the principal plane ① and the other parallel to the axis normal to the principal plane ②.



The angle is measured from one arrow to the other as shown below:



3 Arrow pointing along the axis normal to the principal plane.

4 Arrow pointing parallel to the principal plane.

You can change the direction of the arrows by clicking them or typing one of the following commands in the command window:

REVERSE PROFILE - reverses the 'normal' direction instrumentation arrow.

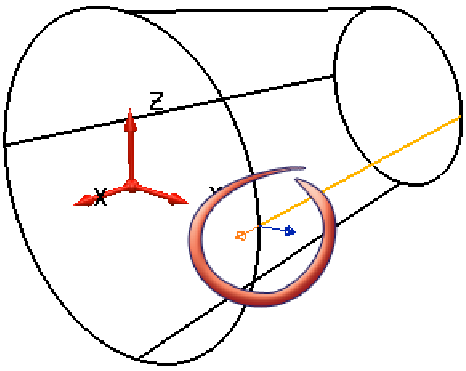
REVERSE WITHDRAWAL_DIRECTION - this reverses the withdrawal direction instrumentation arrow.

The arrow parallel to the principal plane determines the direction of the laterals in the split surface. You can set its direction using the **Split direction** options.

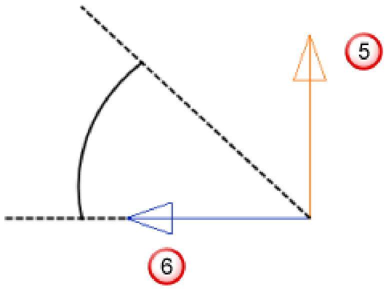
Normal to underlying surface - For this option to be available, the composite curve must be created from the underlying surface.

One arrow points along the normal of the underlying surface and the other parallel to the tangent of the underlying surfaces.

In this example, if you click **Normal to underlying surface**, the arrows change as shown below.



The angle is measured from the arrows as shown below.



5 Arrow pointing along the tangent of the surface.

6 Arrow pointing along the normal to the surface.

Click the arrow pointing along the surface normal to change its direction.

*The arrow parallel to the tangent of the underlying surfaces will affect the direction of laterals in a split surface created at an angle. You can set its direction using the **Orthogonal direction** options.*

Process Split Surface - You can choose whether to retain the tangency across the split surface or smooth out any ripples in the surface.

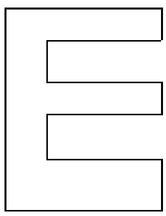
Preserve symmetry - This preserves the symmetry of the split surface when using the **Align to Axes** option in the **Split Direction** menu.

This option only preserves existing symmetry. It does not guarantee a symmetric result if the split curve is not itself symmetrical.

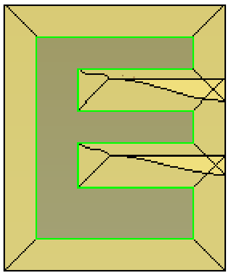
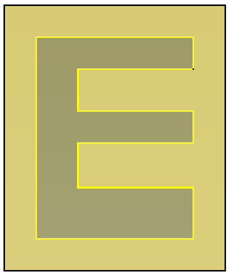
In some cases, the curve may look symmetrical, but the points are distributed differently, especially around the corners where the decisions to change direction are made. This can cause an asymmetric result even with this option enabled.

Trimmed plane – This option is active if the split curve is planar and the composite curve is closed. By default it is set to OFF. If you select Trimmed Plane, **Offset distance** is used as the distance the bounding box of the split curve is enlarged to create the edge of the trimmed plane.

*If **Trimmed plane** is ON, all options apart from Offset Distance become inactive as they not required when using this method.*



Without the **Trimmed plane** option, it would be impossible to create a sensible split surface for the curve shown above. This result is shown by the model on the left. The model on the right shows the result when the **Trimmed plane** option is ON.

Split direction - If you measure the angle from the principal plane, you can set the direction of the laterals within the split surface using one of the following options:

Align to axes - The laterals of the split surface are parallel to the axes of the principal plane (prior to smoothing). For example, if principal plane is the XY plane, the laterals of split surface will run parallel to the X or Y axis. Use this option when the split line is nearly at right angles to the axes.

Radially out - The laterals of the split surface are normal to the composite curve. Use this option when the model is mostly convex.

Surface normal - (Option only available if the composite curve has an underlying surface) The laterals of the split surface lie along the normal of the underlying surface projected onto the principal plane. Use this option when the split line is mostly convex.

Surface internals - (Option only available if the composite curve has an underlying surface) The laterals of the split surface lie in the direction of the internal curves of the underlying surface projected onto the principal plane. This option works well for most shapes, but should be avoided for near vertical edges.

Advanced - This displays the **Split Segments** dialog. You can use this dialog to break the split line into segments. For each segment, you can then specify the direction of the laterals.

 - Create a **Stepped Split Surface** by specifying a number of parameters. Selecting this button displays the Stepped Split Surfaces dialog (see page 322).

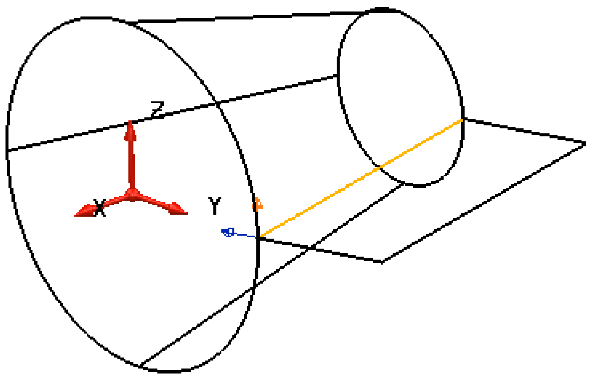
Orthogonal direction - When the **Normal to underlying surface** option is selected and the **Angle** value is greater than zero, **Orthogonal direction** affects the direction of laterals of the split surface using one of the following options:

Principal axis - Uses the axis normal to the principal plane as the direction of the laterals. This option is most useful for surfaces with edges that are trimmed at an angle. You must align the principal axis to the required direction of the split surface. This is done by creating a workplane before creating the split surface and pointing the workplane's principal axis in the direction you want to create the split surface.

Curve normal - The direction of the laterals lie along the normal of the composite curve. This is the best option for creating split surfaces radially around flat surfaces, or when the composite curve is in the plane perpendicular to the required direction of the split surfaces.

Surface internals - The direction of the laterals lie along the direction of the laterals and longitudinals of the underlying surface. This option can sometimes get the required result when the other two options fail.

Preview - Displays the split surface created using the current settings in the dialog. You may continue to change the settings in the dialog until you are satisfied with the previewed split surface.



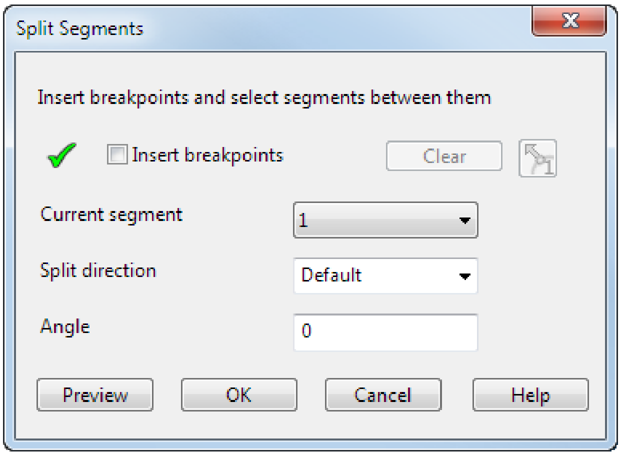
Apply — Creates the split surface.

OK - Creates a split surface and closes the dialog.

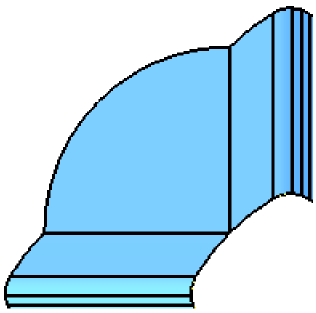
Cancel - Closes the dialog and does not create a split surface.

Split Segments dialog

Use this dialog to break the split line into smaller segments and use them to define the direction of the laterals in the split surface.

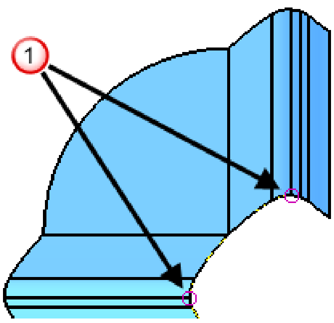


In the diagram below, you can see a segment of a split surface at a rounded corner.



Insert breakpoints - While this option is selected, you can insert and remove split segments from the split line.

To insert breakpoints, click positions on the split line. In the example below, two breakpoints have been inserted at the corner 1.

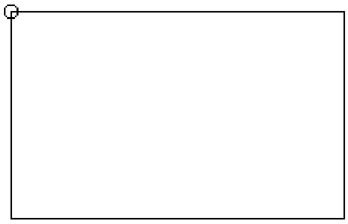


To remove a breakpoint, hold down the **Ctrl** key and click the breakpoint.

Once you have entered the breakpoint, turn off the **Insert breakpoints** option.

When  is showing, the breakpoint is selected as a single breakpoint.

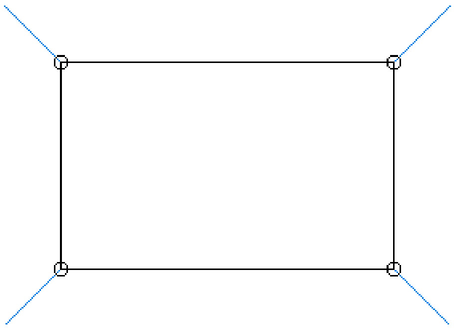
If you add a single breakpoint, it will look as shown below:



The lateral at this point follows the direction of only one of the segments.

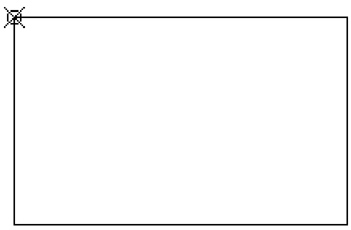
To find out the direction of the lateral at the breakpoint, set the split direction of the segments on either side of the breakpoint and then select **Preview**.

In the example below, you can see one lateral at each single breakpoint.



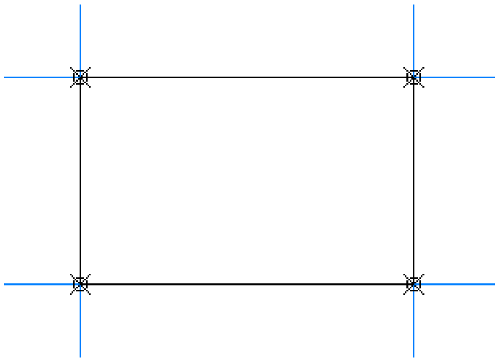
When  is showing, the breakpoint is selected as a split point that can go in two directions.

If you add a split breakpoint, it will look as shown below:



This breakpoint represents two points which lie on top of each other. A lateral is created from each point. One lateral follows the split direction of the segment on one side and the other lateral follows the split direction of the segment on the other side.

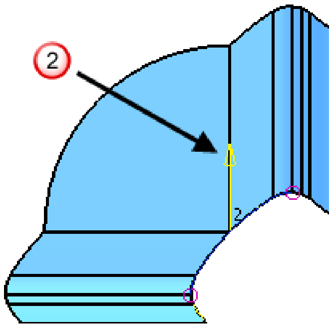
In the example below, you can see two laterals at each split breakpoint.



If you select the **Along axis** option on the **Split surface** dialog, breakpoints are automatically added where the curve changes direction. You can edit these breakpoints like the ones you insert manually.

Clear - Removes all selected breakpoints.

Current segment - Select a segment from the drop down list. The segment is highlighted in the model and has an arrow in it giving the default direction of the laterals 2.

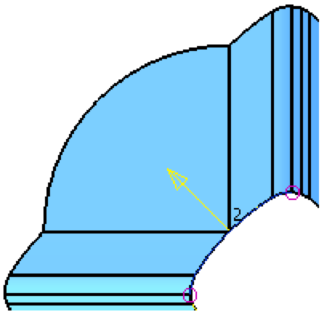


Split direction - Choose a direction from the menu. This menu contains three directions:

> **Default** - Uses the direction as specified on the **Split Surface** dialog. You can reverse the direction of the laterals.
>
> **Radial** - Uses the direction along the normal of the split line.
>
> **Along axis** - Uses the direction along the axis specified on the dialog. Select an axis from the options displayed.

When you select an option, the arrow changes to the specified direction.

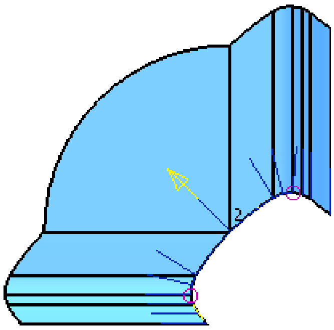
In this example, the **Radial** option was selected.



To reverse the direction of the laterals to the opposite side of the composite curve, click the arrow.

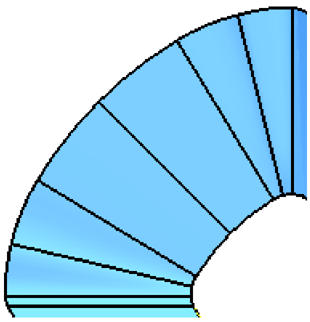
Angle - Defines the angle for the segment of the split surface. The default setting is the default angle for the split surface as a whole.

Preview - This displays the direction of the laterals in the whole of the split surface.



OK - Saves the changes and takes you back to the **Split Surface** dialog.

To preview the new split surface, you need to click **Preview** on the **Split Surface** dialog.

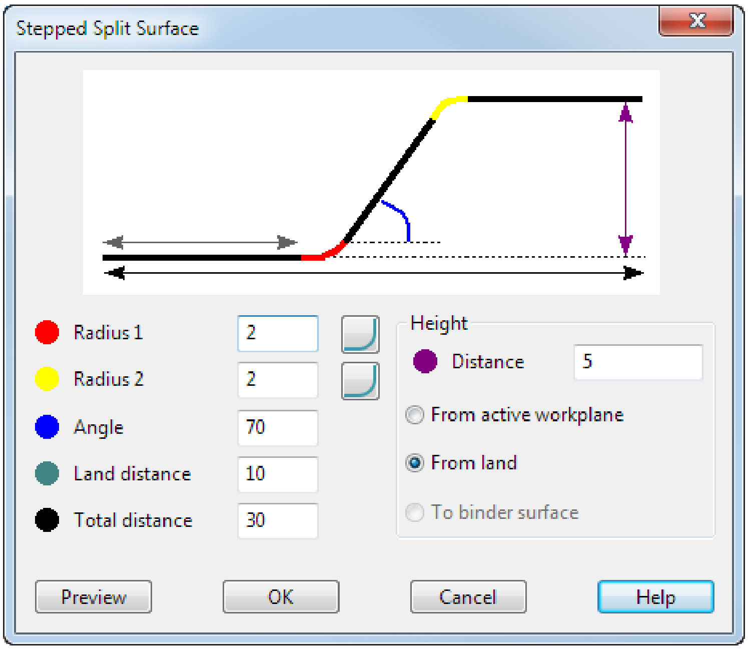


The instrumentation arrow is placed at the midpoint in the following cases

- if the span has only two points.
- if the existing midpoint is coincident with an endpoint.

Stepped Split Surface dialog

Use this dialog to create a stepped split surface.



Colour coding has been used alongside the labels to help identify the values.

Radius 1 — Enter a value for the angle of the first radius.

Radius 2 — Enter a value for the angle of the second radius.

If you enter zero radii in the **Stepped Split Surface** dialog, the result is a sharp discontinuity with no fillet. This is needed in some complex cases where the fillet may need to be added manually later.

 — Toggle between chamfered or filleted edges. The corners of the step can be either filleted or chamfered. The option can be set independently for **Radius 1** and **Radius 2**.

Choosing the chamfer option does not create a chamfer of the length in the radius box, but keeps the distance between the points the same as they were for the fillet, joining them with a straight line.

*The **Land distance** stays constant at the given value. Increasing the **Radius 1** moves the draft part of the step out, rather than keeping it in the same place and reducing the land distance.*

Angle — Enter a value for the draft angle.

Land distance — The initial flat split distance.

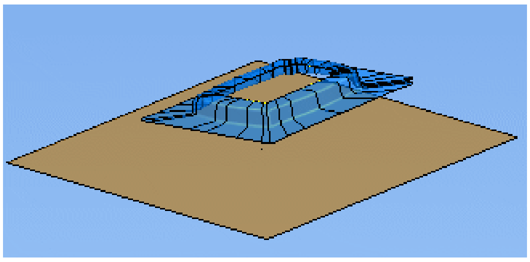
Total distance — This value indicates the total extent of the stepped split surface.

Height

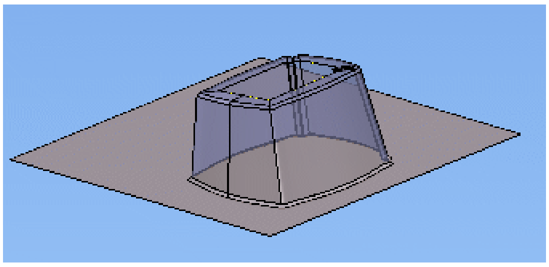
Distance — Height of the step. This value can be specified in three ways:

From active workplane — The height is the distance along the principal axis from the principal plane in the current workplane. This option means that the run-off part of the surface (the bit after the step) will always be planar.

From land — The run off will be an offset continuation of the land and follow the 3D shape of the original split line.



To Binder surface — This option is active when you select a binder surface. The value entered for **Distance** is ignored. The step is effectively a draft onto the binder surface that is then filleted/chamfered back.



Preview - View the stepped split surface.

OK - Accept the values and close the dialog.

Cancel - Return to the **Split Surface** dialog. As the direction of the laterals in the stepped split are identical to those in the basic split, it is useful to be able to return to the **Split Surface** dialog to change the **Split Direction** or to use the **Advanced** options before returning to the stepped split surface page.

Editing a split surface

A split surface is a Power Surface and is edited accordingly. For further details, see Editing Power Surfaces (see page 112).

Creating a draft surface

Draft surfaces can be generated from either a set of surfaces or a composite curve. The creation method of the draft surfaces is very similar for both starting objects.

When draft surfaces are generated from a set of surfaces, PowerSHAPE automatically generates composite curves known as draft curves on the surfaces. Each draft curve generates a draft surface as follows. One side of the draft surface is the draft curve. The draft surface projects along the draft angle until it touches either:

- the principal plane.
- a group of surfaces.

If you are not entirely satisfied with the automatically generated draft curves, you can generate draft curves from a surface by using the **Create a draft curve** option (C*urve toolbar*) and edit them to your requirements.

When a draft surface is generated from a composite curve, PowerSHAPE treats the composite curve as a draft curve. In this case, it also allows you to vary the draft angles along the draft surface.

1 Click  (*Surface toolbar*).

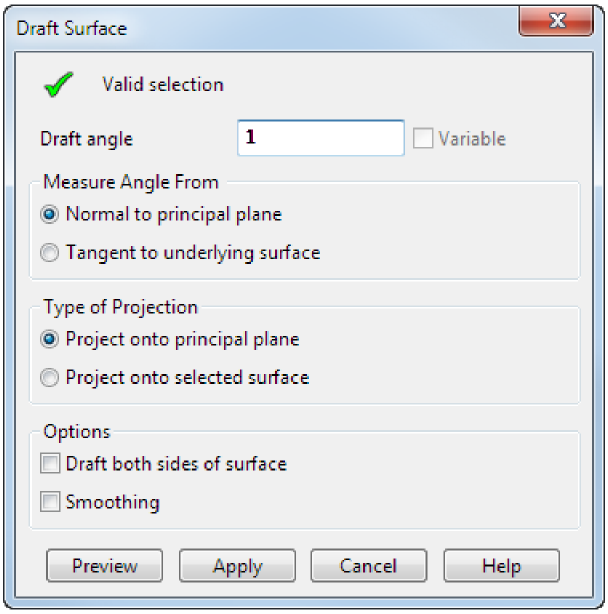
   The Draft Surface dialog (see page 325) is displayed.

2 Complete the dialog.

*PowerSHAPE checks that you have an appropriate licence when you try to use **Draft surfacing**. An error message is displayed if the appropriate licence is not available.*

Draft Surface dialog

Use this dialog to create a draft surface.



Valid selection — If valid objects are selected, a tick is displayed. Select one of the following:

- A composite curve.
- One or more surfaces.
- One or more solids.

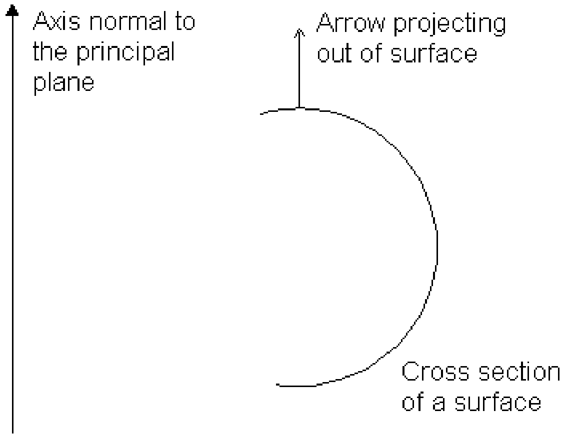
Draft angle — This is the angle the draft surface will be projected from the selected objects. You can also use a negative angle when creating a draft surface.

Variable — This option is only available if you are creating a draft surface from a composite curve. It enables you to create a draft surface with variable draft angles.

Once this option is selected, the **Preview** button is greyed out. When you click **Apply**, the **Variable Draft** (see page 331) dialog is displayed. Use this dialog to input the variable draft angles.

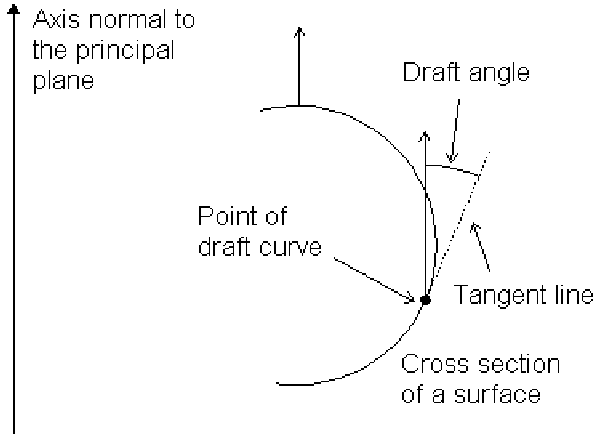
Measure angle from — The draft angle is measured (by default) from the axis normal to the principal plane. The **Tangent to underlying surface** option is only available if you are creating a draft surface from a composite curve that lies on one or more surfaces.

If you are creating a draft surface from surfaces or solids, an arrow is projecting out from the surfaces. It is parallel to the axis normal to the principal plane and is used to calculate where the draft curves lie on the surfaces.



Remember that one side of the draft surface is defined by the draft curve.

The points of the draft curves are positions on the surfaces where the angle between the tangent and the arrow equals the draft angle.

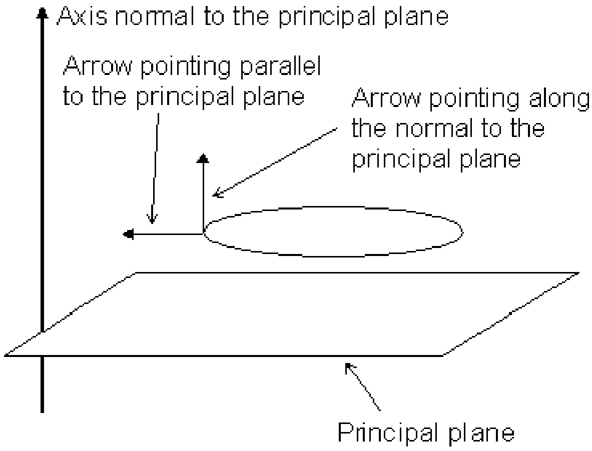


The draft surface is projected along this tangent until it touches whatever you select as the **Type of Projection** on the dialog.

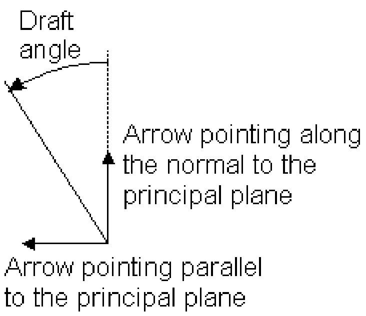
Click the arrow to change its direction if you want the draft surfaces to project in the other direction.

If you are creating a draft surface from a composite curve, two arrows are drawn on the composite curve. Both arrows determine the direction of projection of the draft surface.

With **Normal to principal plane** selected from **Measure angle from** options, one arrow is parallel with the principal plane and the other with the axis normal to the principal plane (see below).



The draft angle is measured between the arrows as shown below.



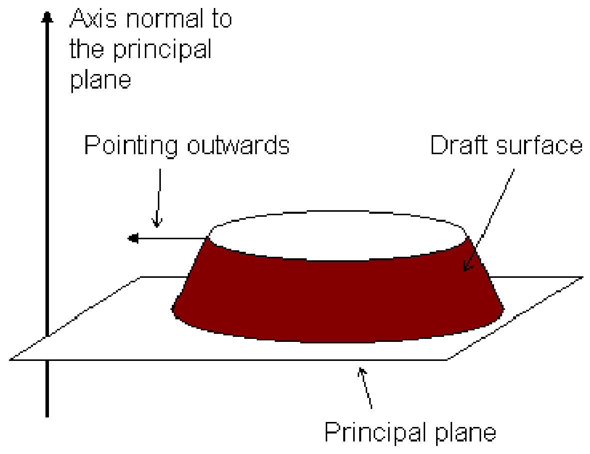
You can change the direction of the arrows by clicking them or typing one of the following commands in the command window:

REVERSE PROFILE — reverses the 'normal' direction instrumentation arrow.

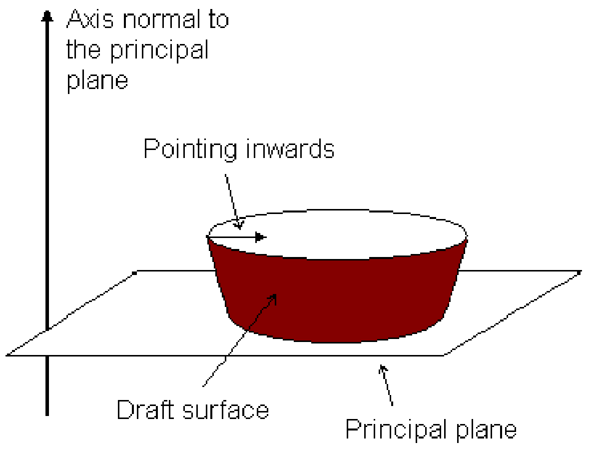
REVERSE WITHDRAWAL_DIRECTION — this reverses the withdrawal direction instrumentation arrow.

Ensure that the arrow along the normal to the principal plane points in the correct direction, usually in the direction of the split surface.

If you require a draft surface that projects outwards, ensure the arrow parallel to the principal plane points outwards:



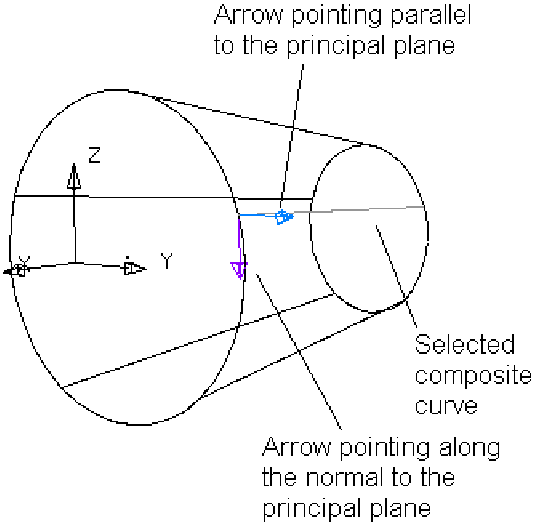
If you want a draft surface that projects inwards, ensure the arrow parallel to the principal plane points inwards:



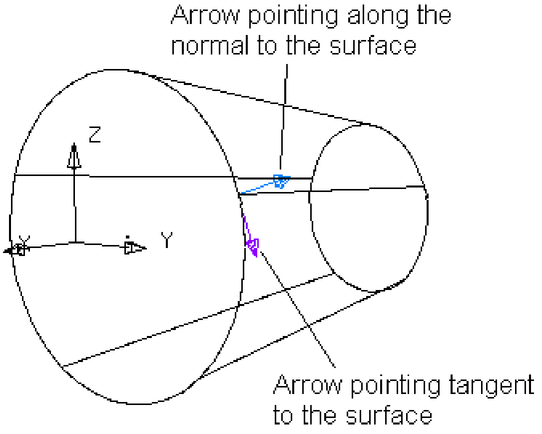
If your composite curve is created from the underlying surface, you can use the tangent data of the surface to define the draft angle. To use the tangent data, select **Tangent to underlying surface** from the **Measure angle from** options. The arrows change direction such that:

- the arrow pointing parallel to the principal plane now points along the normal of the underlying surface;
- the arrow pointing normal to the principal plane now points tangent to the surface.

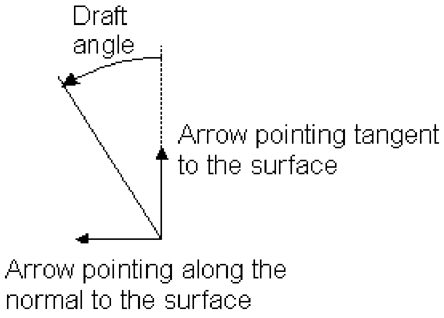
Consider the following composite curve on a surface. With **Normal to the principal plane** selected (by default) from the **Measure angle from** options, the arrows point as shown below.



With **Tangent to underlying surface** selected, the arrows point as shown below.

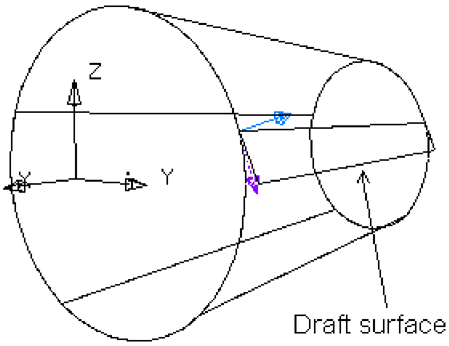


The draft angle is measured between the arrows as shown below.



You can click the arrows to change their direction.

In our example, the following draft surface is created when we enter a draft angle of 5°.



Type of projection — This allows you to choose where to project the draft surface.

Project onto principal plane — This option projects the draft surface onto the principal plane.

Project onto selected surface — You can project the draft surface onto a number of surfaces. Select the surfaces and a TICK appears in the box. If the surfaces overlap one another, the draft surface is projected onto the surface nearest to it.

If the draft surface does not entirely project onto the surface or the plane, a draft surface is created on the portion of the surface or plane that it can project onto.

Options: Draft both sides of the surface — (Available only when creating draft surfaces from surfaces.) This ignores the arrow projecting out of the surfaces and creates draft surfaces from both sides of the surfaces.

If the surfaces are not symmetrical about the surfaces or principal plane onto which the draft surfaces are projected, PowerSHAPE matches the draft surfaces by increasing the draft angles along the draft surfaces, where necessary.

Smooth — This smooths out any ripples in the surface. It frees projected tangents where appropriate and preserves straight spans.

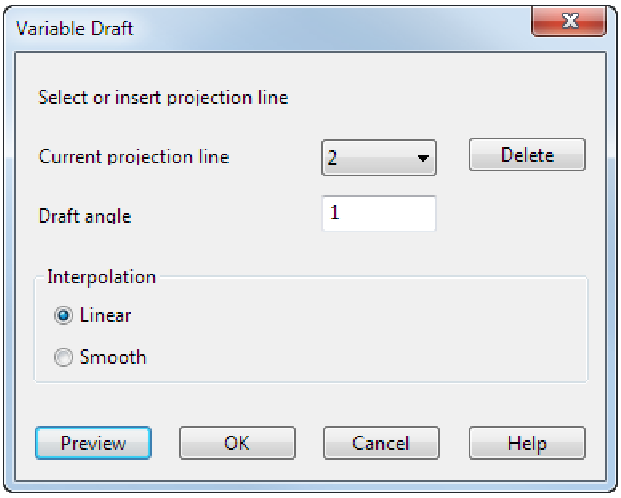
Preview — Displays the draft surface using the current settings on the dialog. You can continue to change the settings on the dialog until you are satisfied with the previewed draft surface.

Apply — Creates draft surfaces in the model, trims the set of selected surfaces to the draft curves. The dialog remains on the screen for you to select more objects and continue creating draft surfaces.

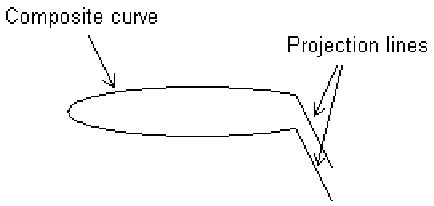
Cancel —Closes the dialog.

Variable Draft dialog

This dialog adds variable draft angles on the draft surface.

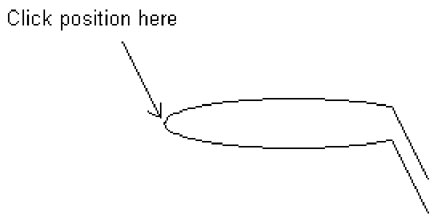


Projection lines are drawn at the ends of the composite curve to mark the draft angles of the draft surface.

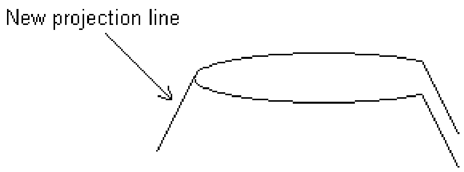


If the composite curve is closed, no projection lines are drawn.

Click positions on the composite curve where you want to input specific draft angles.



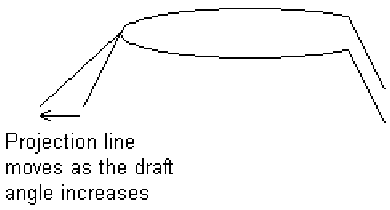
A projection line is inserted at each position.



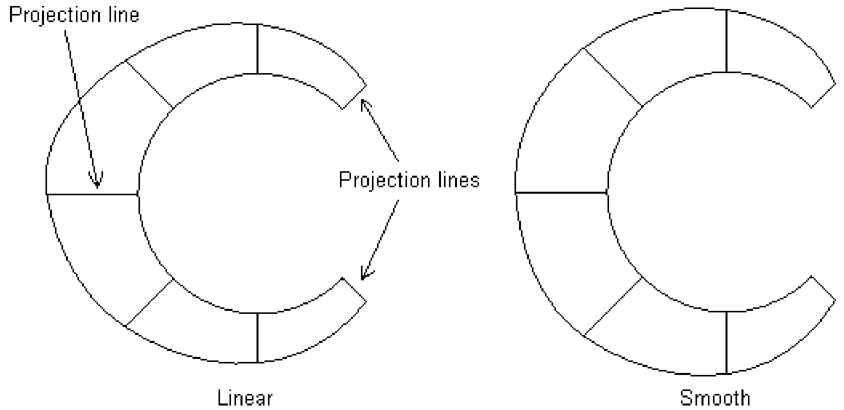
Current projection line - The number of the current line is given here. You can select a projection line from either the drop down list or the screen.

Delete - Deletes the current projection line. You can't delete the projection lines at the ends of the composite curve.

Draft angle - This is the draft angle of the current projection line. If you change the angle, the current projection line moves to the new position. If the preview surface is displayed and you insert a new projection line, the value of the draft angle is taken from the preview surface.



Interpolation - You can vary the draft angles either linearly or smoothly in between projection lines.



Preview - Displays the draft surface using the current settings on the dialog. You can continue to change the settings on the dialog until you are satisfied with the previewed draft surface.

OK - Creates the draft surface in the model. The dialog is removed from the screen and you are returned to the **Draft Surface** dialog.

Cancel - No draft surface is created. The dialog is removed from the screen and you are returned to the **Draft Surface** dialog.

Editing a draft surface

You may wish to edit the outer boundary of the draft surface. A draft surface is a power surface and is edited accordingly. For further details, see Editing Power Surfaces (see page 112).

Creating an extension of a surface

You can extend a group of surfaces by creating a new surface which is tangent to the other surfaces along a common edge.

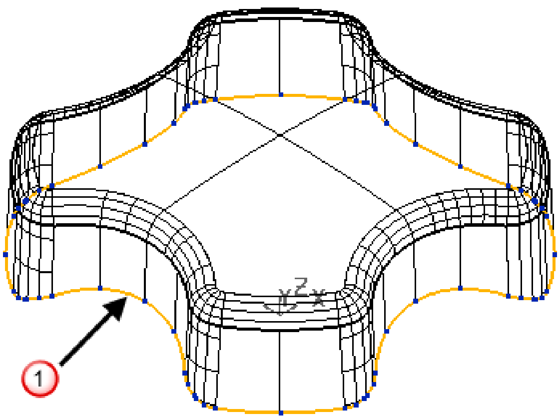
1 Orientate the surface normals of your surfaces.

As a general rule, you should always keep the surface normals pointing out from the material.

Make sure the outside of two adjacent surfaces match by reversing the surface normals if necessary.

The surface normals of the surfaces determine the direction of the extended surface.

2 From the edges of your surfaces, create a composite curve 1.



When you create a composite curve from a surface, the tangent data of the surface is stored with the definition of the composite curve.

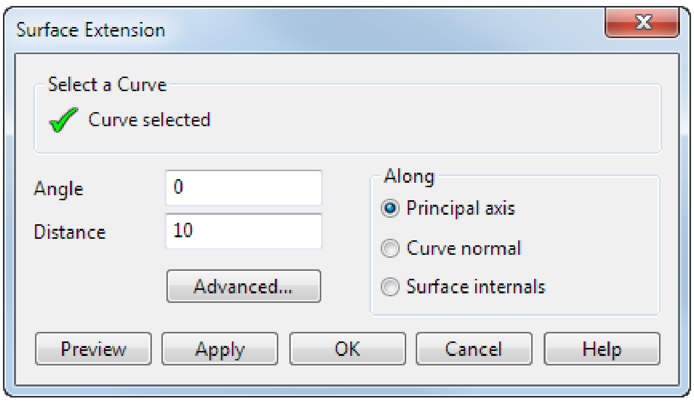
3 Select the composite curve.

4 Click  (*Surface toolbar*).

5 Use the Surface Extensions dialog (see page 333) to extend the surface.

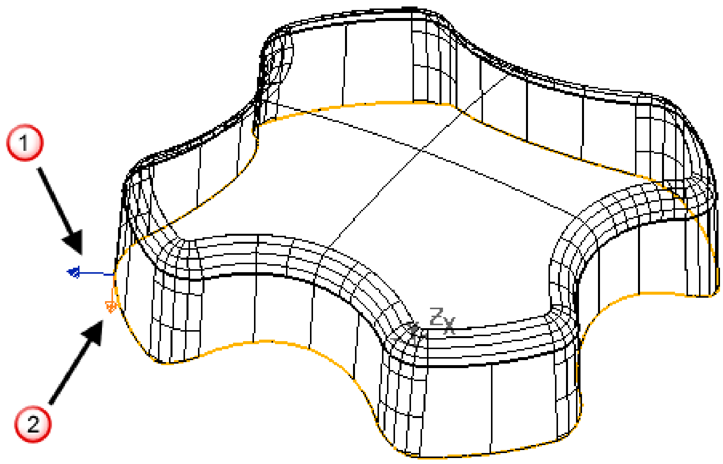
Surface Extension dialog

Use this dialog to create a surface tangent to the surface used to create the composite curve.

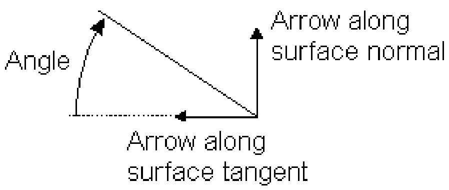


Angle — This is the angle the new surface is projected from the existing surface.

Two arrows are drawn on your composite curve. One arrow points along the surface normal 1 and the other along the surface tangent 2.



Both arrows are used to calculate the angle of the surface extension. The angle is measured from the arrow along the surface tangent to the one pointing along the surface normal (see below).



You can change the direction of the arrows by clicking them or typing one of the following commands in the command window:

> **REVERSE PROFILE** — reverses the 'normal' direction instrumentation arrow.
>
> **REVERSE WITHDRAWAL_DIRECTION** — this reverses the withdrawal direction instrumentation arrow.

For our example, we want the new surface to point downwards along the surface tangents. The arrow pointing along the surface tangent is in the correct direction. We leave the angle as zero.

Distance — This is the distance the new surface is projected from the composite curve.

In our example, we enter 5.

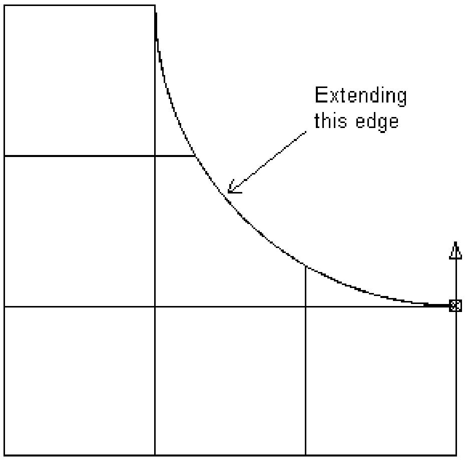
Along — This sets the direction of the extension using one of the following options.

> **Principal axis**

Curve normal

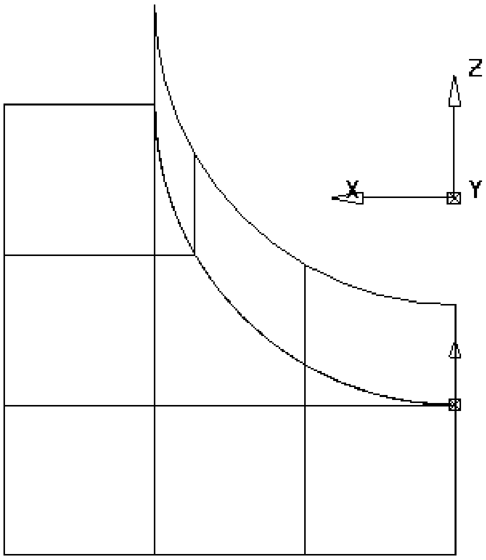
Surface internals

We will use the following surface to illustrate these options.



Principal axis — Uses the axis normal to the principal plane as a guide to extend the surface.

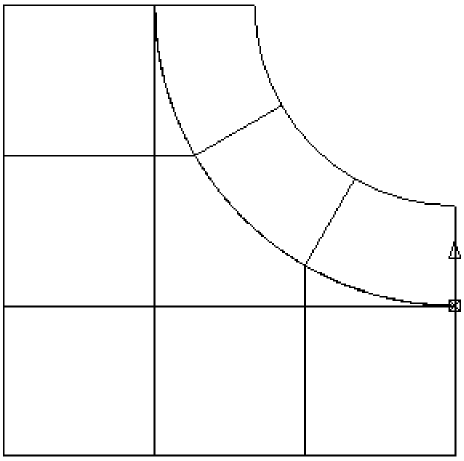
In the example below, the Z axis is the principal axis.



This option is most useful when extending an edge of a surface that is trimmed at an angle.

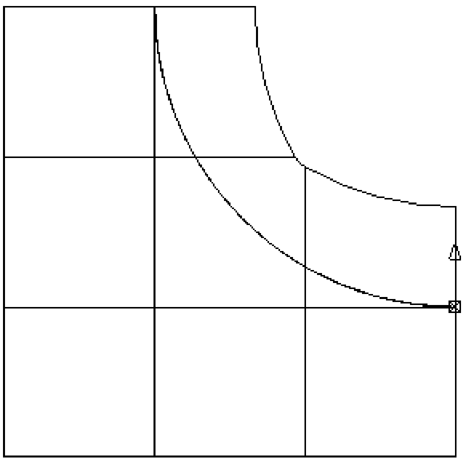
You must align the principal axis to the required direction of the extension. This is done by creating a workplane before creating the extension and pointing the workplane's axis, which is normal to the principal plane, in the direction you want to create the extension.

Curve normal - Extends the surface along the normal of the composite curve.



This is the best option for extending radially around flat surfaces, or when the composite curve is in the plane perpendicular to the required direction of the extension.

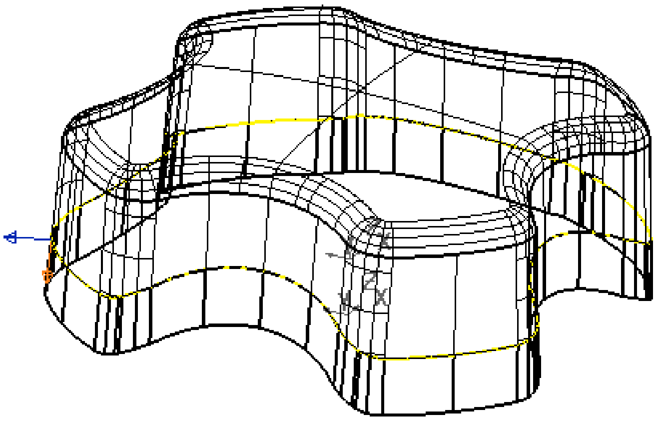
Surface internals - Extends the surface along the direction of the existing laterals and longitudinals of the surface.



This option can sometimes get the required result when the other two options fail.

Advanced - This displays the **Extension Segments (see page 337)** dialog. You can use this dialog to break the composite curve into segments. For each segment, you can then specify the direction of the extension.

Preview - Displays the surface created using the current settings on the dialog. You can continue to change the settings in the dialog until you see the required solution on the screen.

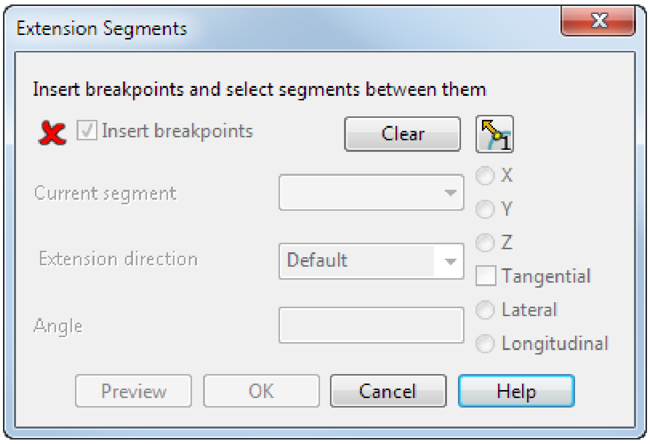


OK - Creates the tangent surface and removes the dialog from the screen.

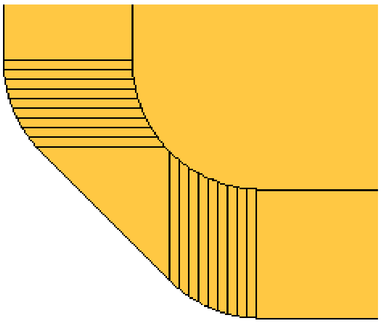
Cancel - Does not create the tangent surface and removes the dialog from the screen.

Extension Segments dialog

Use this dialog to break the composite curve into smaller segments and use them to define the direction of the laterals in the extension surface.



In the diagram below, you can see a segment of an extension surface at a rounded corner.

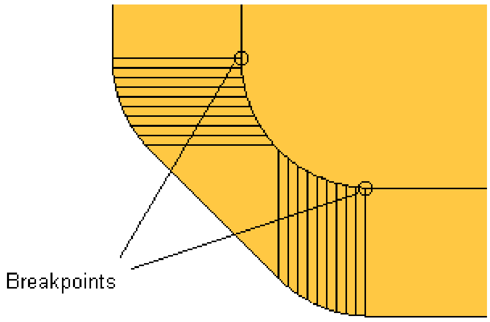


We will show you how to improve this extension surface at the corner using the dialog.

Insert breakpoints - While this option is selected, you can insert and remove breakpoints from the composite curve.

To insert breakpoints, click positions on the composite curve.

We inserted two breakpoints at the corner.



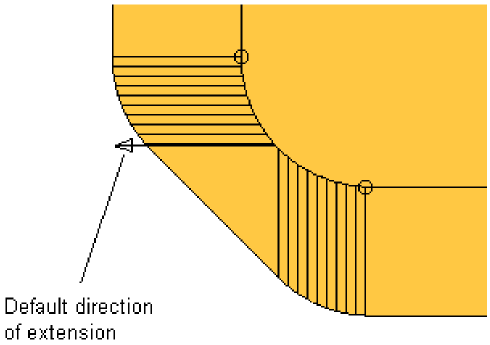
To remove a breakpoint, hold down the **Ctrl** key and click the breakpoint.

Once you have entered the breakpoints, turn off the **Insert breakpoints** option.

Clear - Click to clear all guide-curves.

 and  toggle - Click to toggle the direction of the breakpoints. For further details, see Split Segments dialog (see page 318).

Current segment - Select a segment from the combo box. The segment is highlighted in the model and has an arrow in it giving the default direction of the extension.

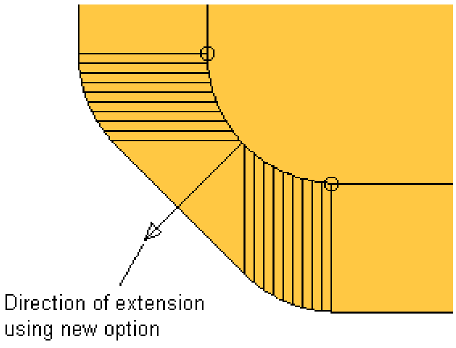


Extension direction - Choose a direction from the menu. This menu contains the following directions:

- **Default** - Uses the direction as specified on the **Surface Extension** dialog. You can reverse the direction of the laterals.
- **Curve normal** - Uses the direction along the normal of the composite curve.
- **Along axis** - Uses the direction along the specified axis on the dialog.
- **Surface internals** - Uses the direction along the internal curves of the surface.

When you select an option, the arrow changes to the specified direction.

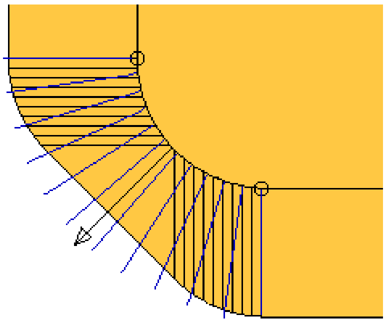
In our example, we choose the **Curve normal** option.



To reverse the direction of the laterals to the opposite side of the composite curve, click the arrow.

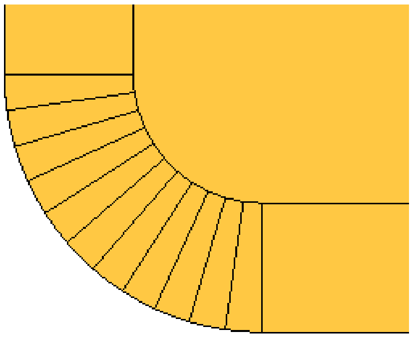
Angle - This is the angle the new surface segment is projected from the existing surface.

Preview - This displays the direction of the laterals in the whole of the extension surface.



OK - Saves the changes and takes you back to the **Extension Surface** dialog.

To preview the new extension surface, you need to click **Preview** on the **Extension Surface** dialog.



The instrumentation arrow is placed at the midpoint in the following cases

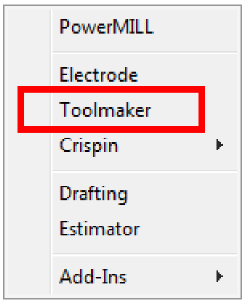
- if the span has only two points.
- if the existing midpoint is coincident with an endpoint.

Fill edge surface creation

This option is only available in Toolmaker mode.

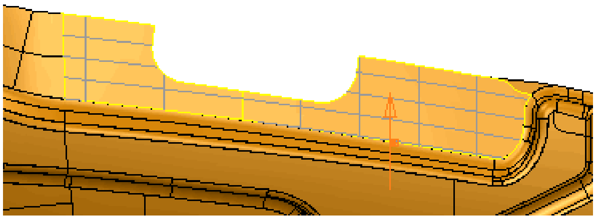
You can create a surface that will fill the gap between selected points on an existing surface.

1 With PowerSHAPE running, click **Module > Toolmaker**

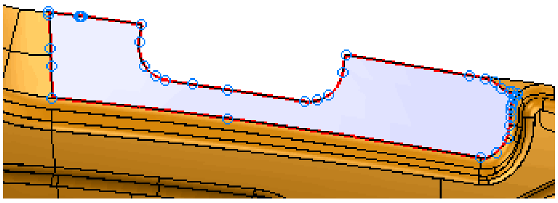


2 Select **Surface**  from the main toolbar to display the surface modelling toolbar. This includes **Addendum surface** and **Fill edge regions** buttons.

3 Select the surface to be used.

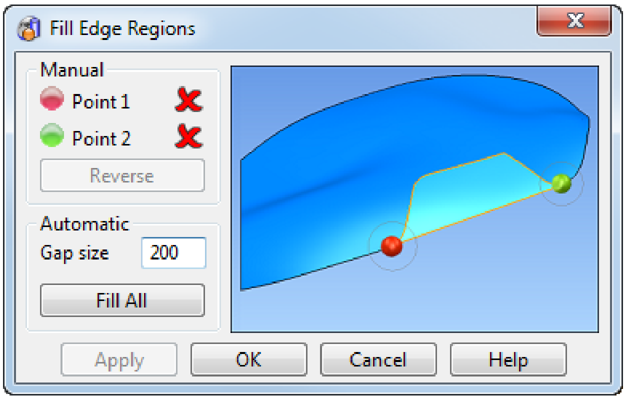


4 Click on **Fill edge regions**  to display the Fill Edge Regions dialog (see page 342). The points on the edge region are displayed on the model.



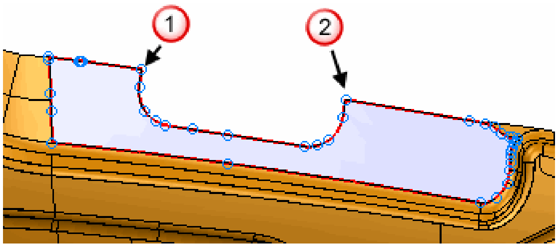
Fill Edge Regions dialog

Use this dialog to create a surface using the fill edge regions.

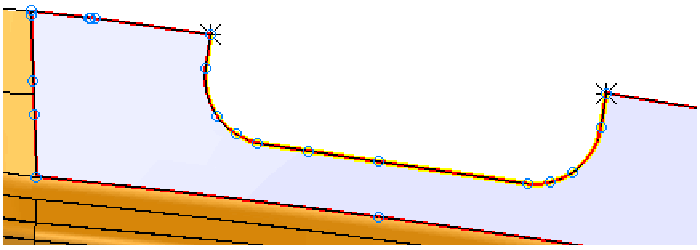


To use the Manual option

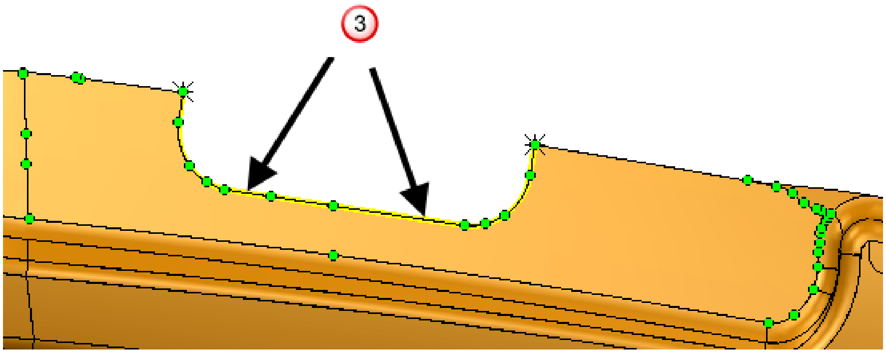
1 Select **Point 1** and **Point 2** on the surface, as shown by ① and ② .

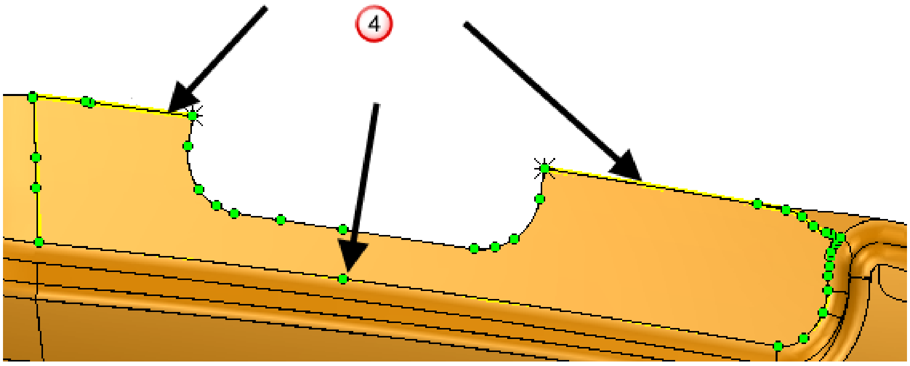


When you have made valid selections, both the ✘ change to ✔ and the fill edge is highlighted.

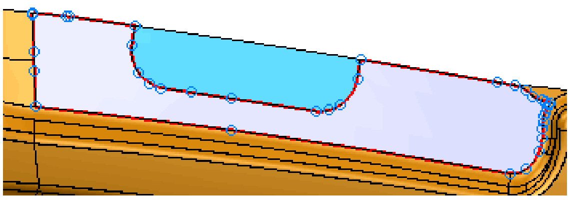


Use the **Reverse** option on the **Fill Edge Regions** dialog to switch between the shortest path 3 and the longest path 4 for selected points. The surface that is created depends on the path that you select.



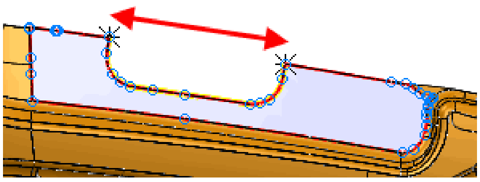


2 Click **Apply**



To use the Automatic option

1 Adjust the **Gap size** to the maximum size of the gaps to be filled. Any gaps that are larger than the **Gap size** will not be filled.



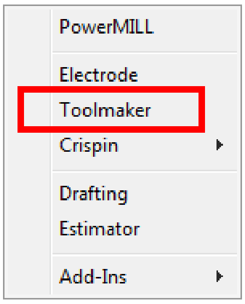
2 Click **Fill all** to Fill all the gaps the gaps that are smaller than the **Gap size**.

3 Click **OK**.

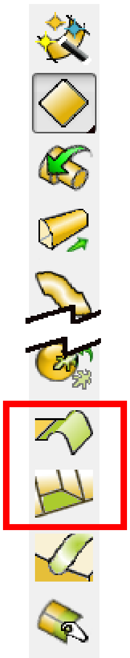
Addendum surface creation

This option is only available in Toolmaker mode.

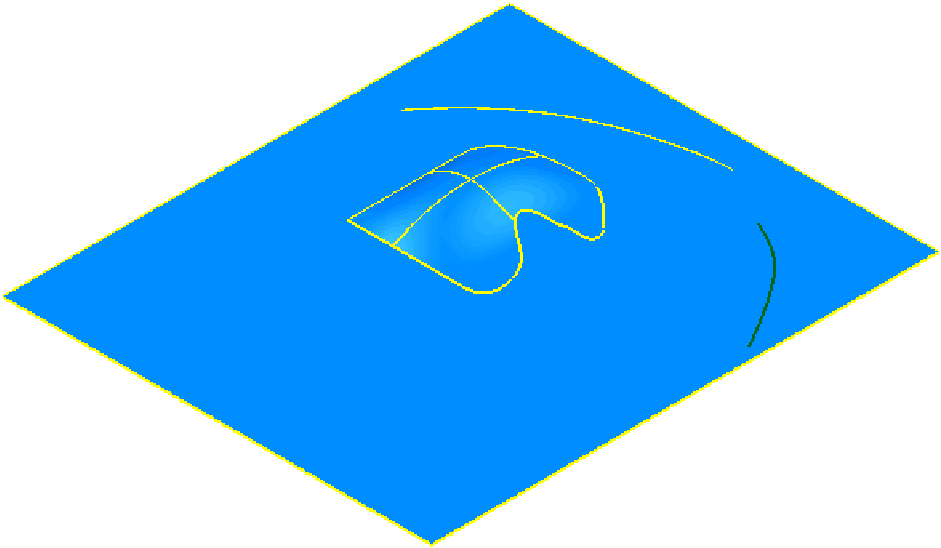
1 With PowerSHAPE running, click **Module - Toolmaker**



2 Select **Surface**  from the main toolbar. The surface modelling toolbar is displayed. This includes **Addendum surface** and **Fill edge regions**.



3 Select the Binder surface and the surface that is to be used to create the addendum surface. This step is important as it is not possible to select the binder surface later.



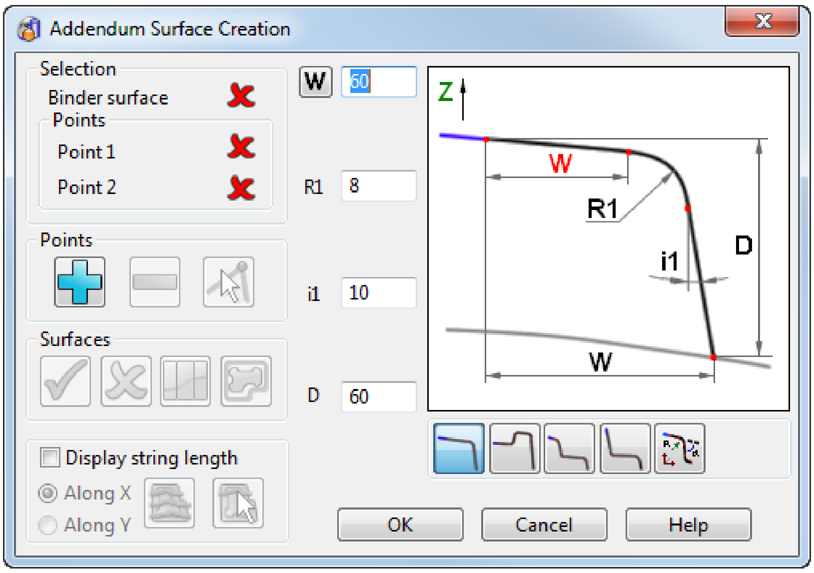
4 Click **Addendum surface**  to display the Addendum Surface Creation dialog (see page 345), with the Binder surface pre-selected.

5 Use the options on the dialog to enter the settings to be used when creating the surface.

6 Click **OK** on the **Addendum Surface Creation** dialog to create the Addendum surface.

Addendum Surface Creation dialog

Use this to specify the parameters when creating an addendum surface.

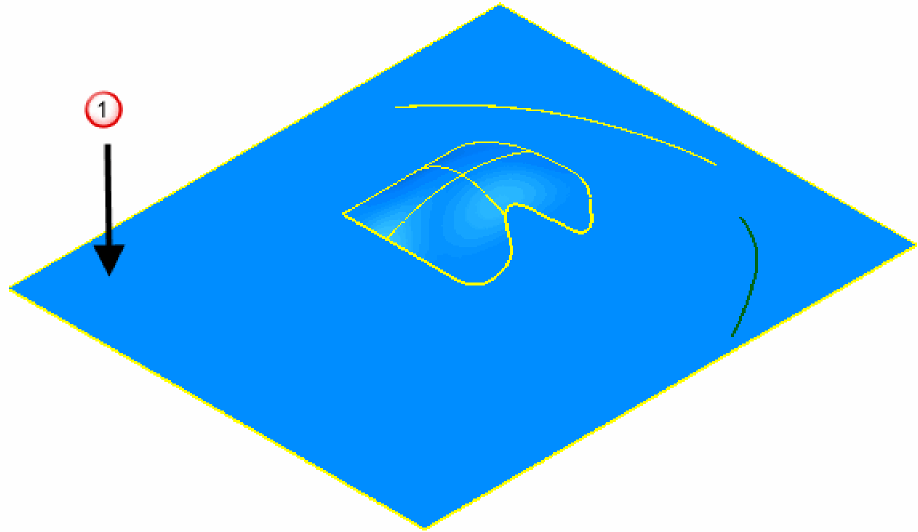


You cannot select the Binder surface during the creation process - it must be selected before you select the Addendum creation button from the surface modelling toolbar.

Selection

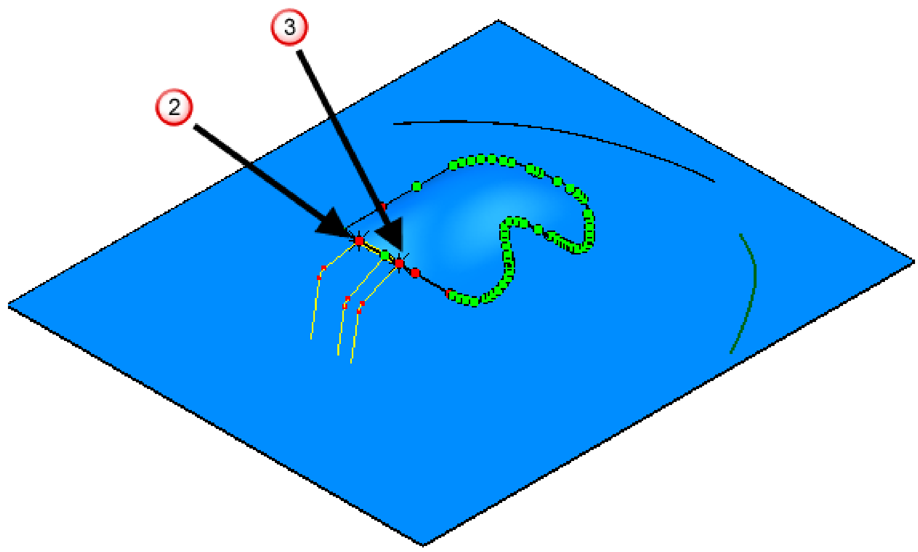
Use the options in this section to identify the surface and points that are to be used.

1 **Binder surface** or *Blank holder surface*. The Binder surface must be larger than the part and situated below the part. The Binder surface ① was selected before the dialog was displayed.



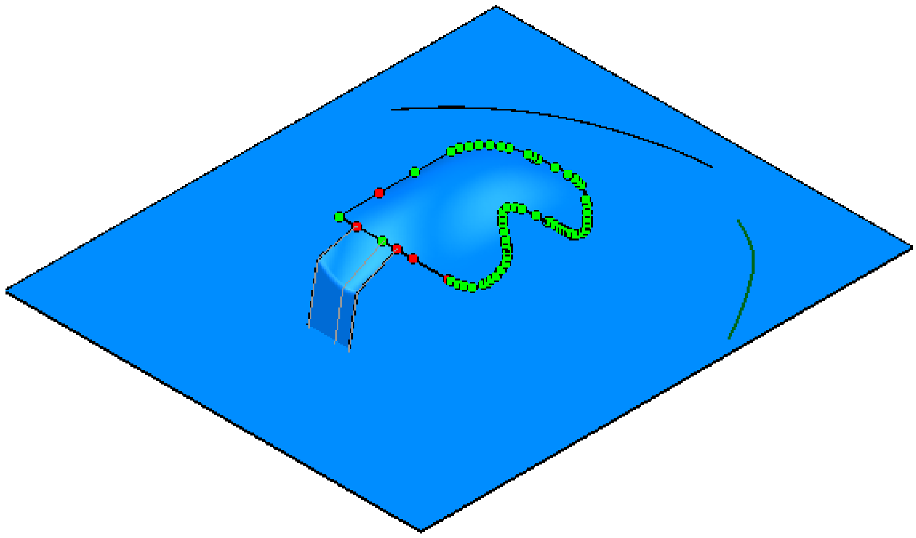
2 Select **Point 1** to be used to create the addendum surface. The ✘ changes to ✔. The selection is indicated by ② on the following model.

3 Select **Point 2** to be used to create the addendum surface. The ✘ changes to ✔. The selection is indicated by ③ on the following model.



Points

Use this section to add and delete points.



On the model, green points are required to hold the edge of the addendum onto the part. If you remove a green point, you will end up with an overlap or a gap. Red points are redundant points. These can be deleted without causing any problem.

To insert an additional boundary point:

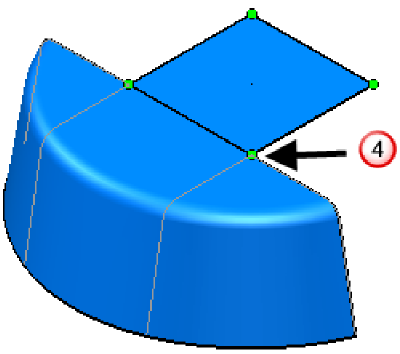
1. Click .
2. Click on the boundary curve where the new point is to be created.

To delete a point, or a selection of points from the boundary curve:

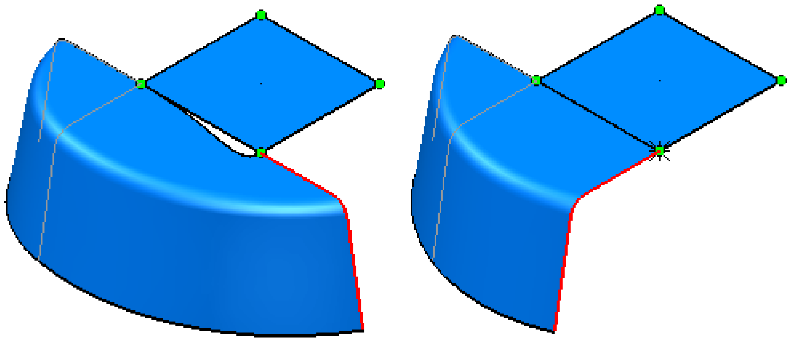
1. Select your point(s).
2. Click .

To see an alternative solution for a surface that has sharp corners:

1. Select 



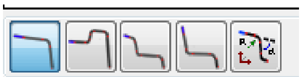
2 Click and cycle through the possible solutions.



*You can add or remove points either before or after creating your addendum surface. If you edit the points after creating an addendum surface, you will need to select **Point 1** and **Point 2** and click to create the amended surface.*

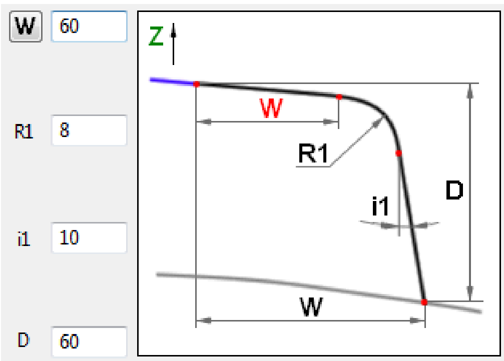
Selecting a profile

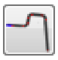
Use the profile buttons to select an appropriate profile.

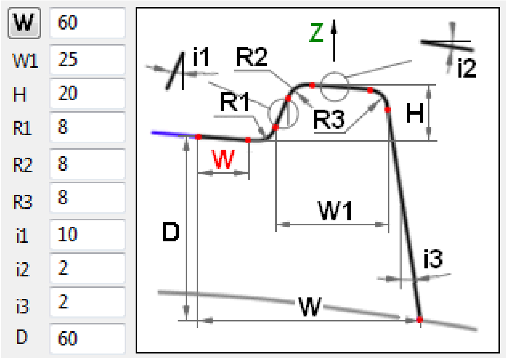


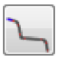
If the profile is blue, you cannot use the profile type you have selected. If the profile is yellow, the profile type you have selected can be used. The dimension options that are available is based on the profile you select. The Depth (**D**) is calculated automatically to ensure that the addendum surface intersects with the binder surface.

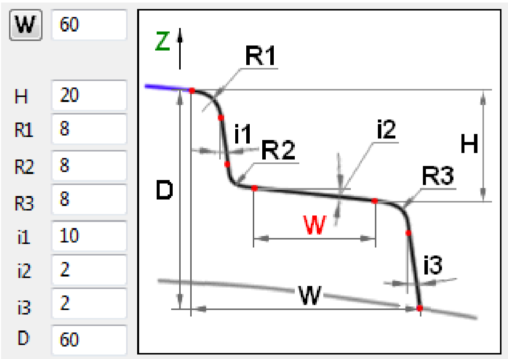
- Selecting this profile makes the following dimension options available

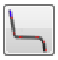


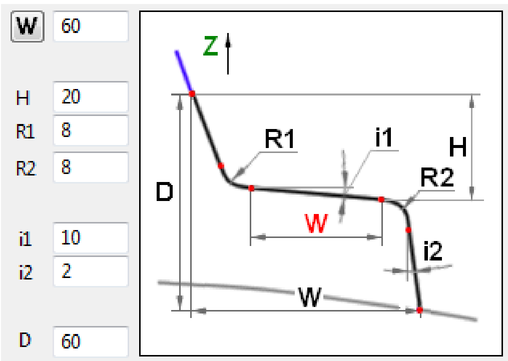
 - Selecting this profile makes the following dimension options available



 - Selecting this profile makes the following dimension options available



 - Selecting this profile makes the following dimension options available

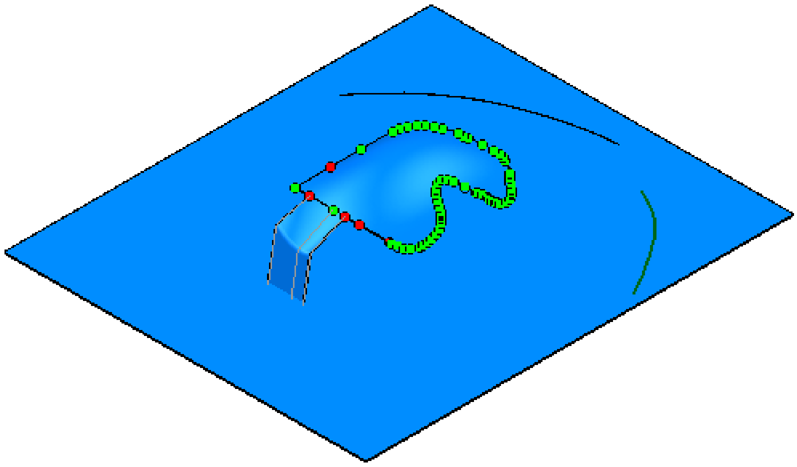


 - Opens the Parametric curve editor. Use the options in the Parametric curve editor to edit the selected profile. The buttons on the Parametric curve editor also appear on the 2D curve editing toolbar.

 - Click to toggle between black and red width. This provides two solutions using different widths but keeping the remaining parameters the same.

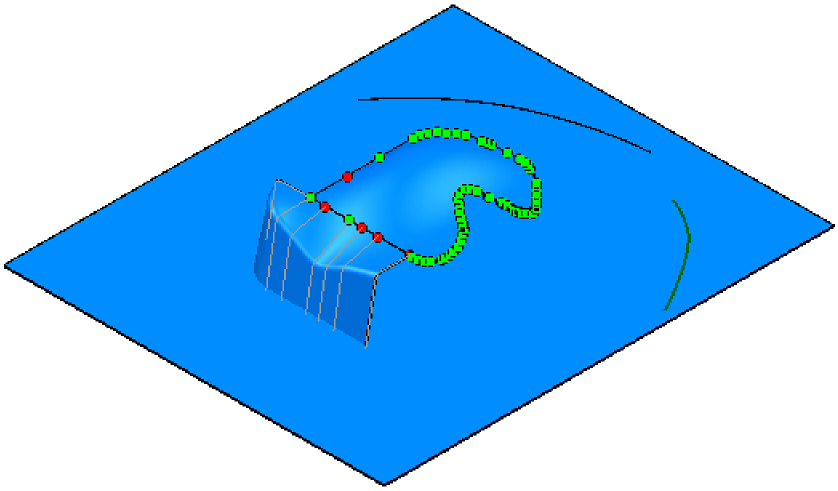
Surfaces

- Create the addendum surface.



- Delete the currently selected addendum surface. You can select the addendum surface to delete by clicking on the surface itself, or by clicking on the points on the laterals. You can also delete the selected addendum surface using the **Delete** key.

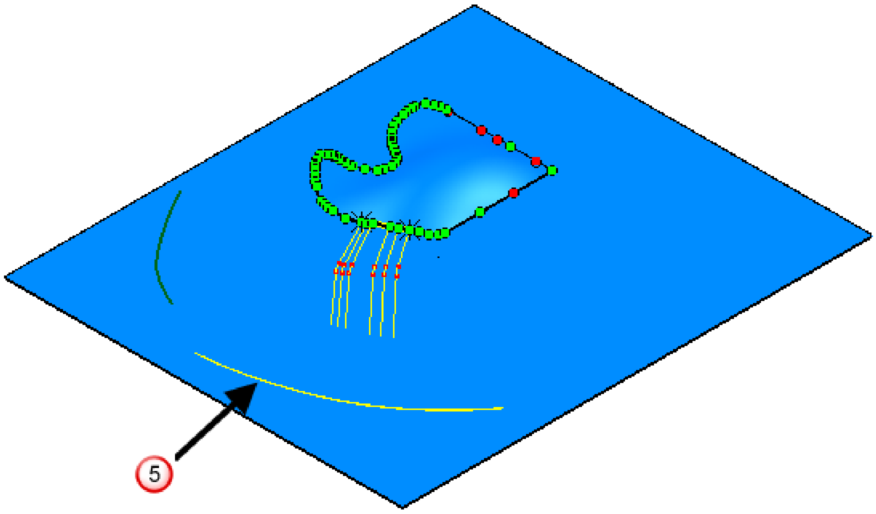
- Click on any boundary point between the two surfaces you want to connect. Click to create the connected surface. You can delete the connection surface using the button on the dialog or by using the **Delete** key.



- The **P**unch **O**pen **line** is the intersection of the addendum surface and the binder surface and is typically a curve or a composite curve

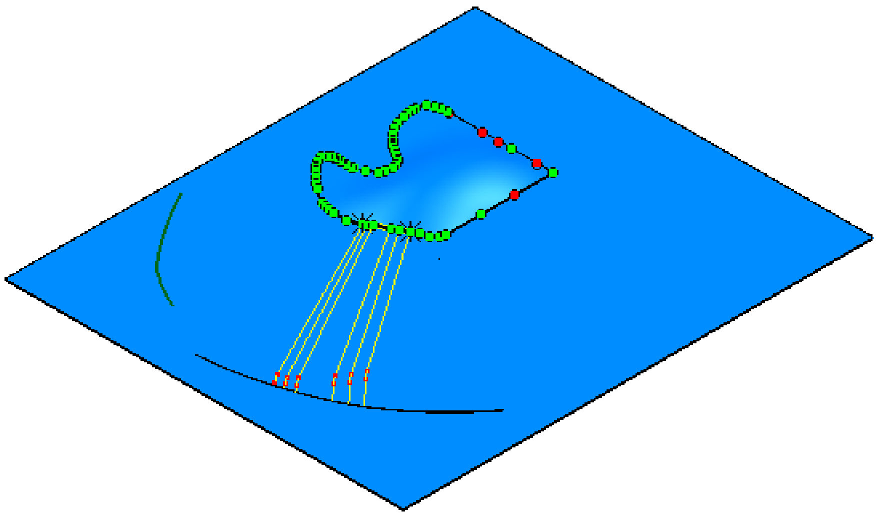
1 Select **Point 1** and **Point 2**.

2 Select the curve or composite curve to be used as the PO line 5

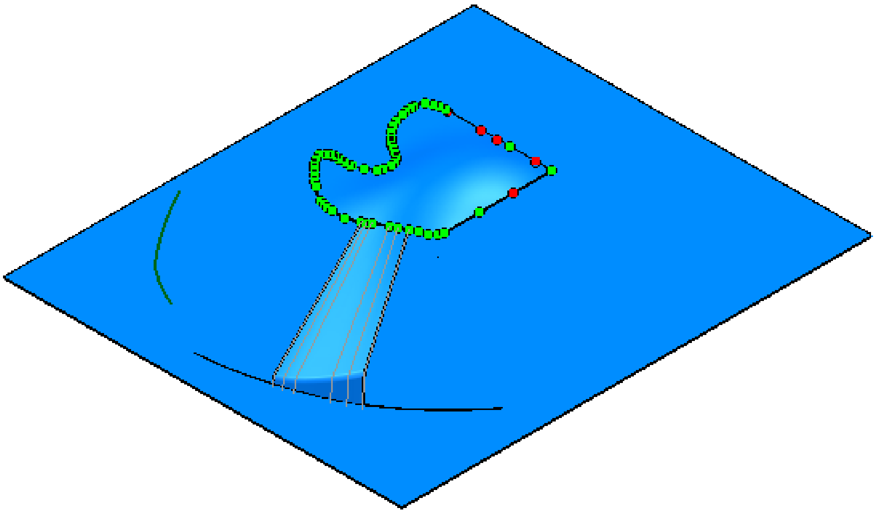


3 Click .

The laterals that are used when creating the addendum surface will be extended to the PO line.

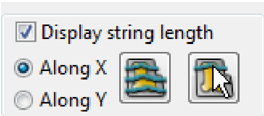


4 Click  to generate the addendum surface.



String Length

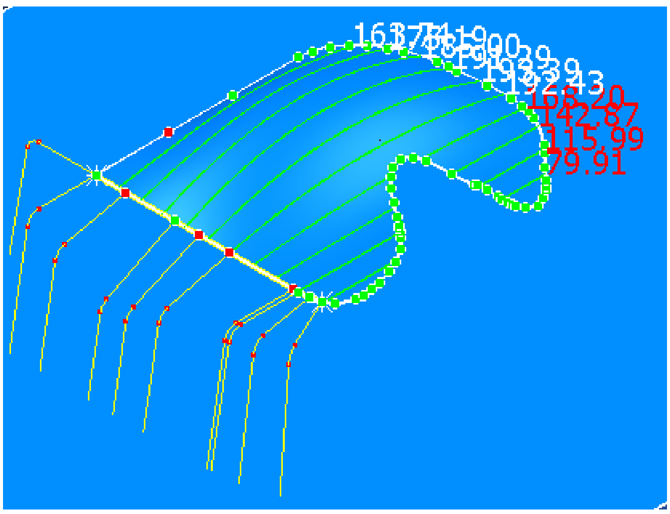
1 Click **Display string length** to activate the string length functionality. This controls the way oblique strings are displayed.



2 Click **Along X** or **Along Y** to define the orientation of the oblique strings.

3 Click on of the following:

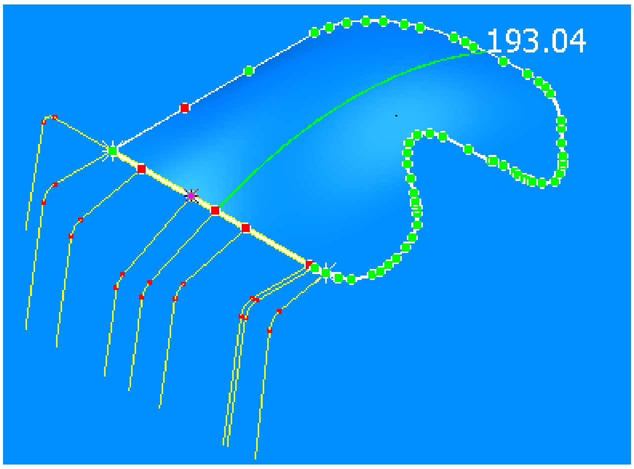
- **Area** to create 10 oblique strings with a sensible interval through all items (a main part + created addendum surfaces). The model below was created by selecting **Along X** and **Area**.

- **Point** to create an oblique string through a picked on an item point.

  Pick a point.

The following model was created by selecting **Along X** and **Point**.

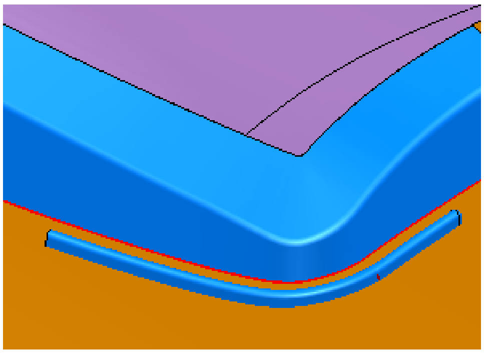


Switching off the Display string length option or closing the Addendum form removes all the oblique strings and labels.

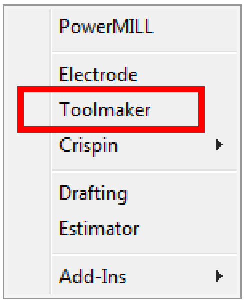
Creating a Drawbead surface

This option is only available in Toolmaker mode.

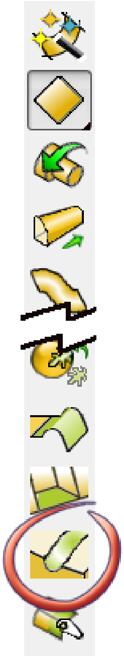
A **Drawbead** surface is a ridge constructed to hold the metal in position during the forming process. It is often created at the corners of the part.



1 With PowerSHAPE running, click **Module > Toolmaker**.



2 Select **Surface**  from the **Main** toolbar. The surface modelling toolbar is displayed.



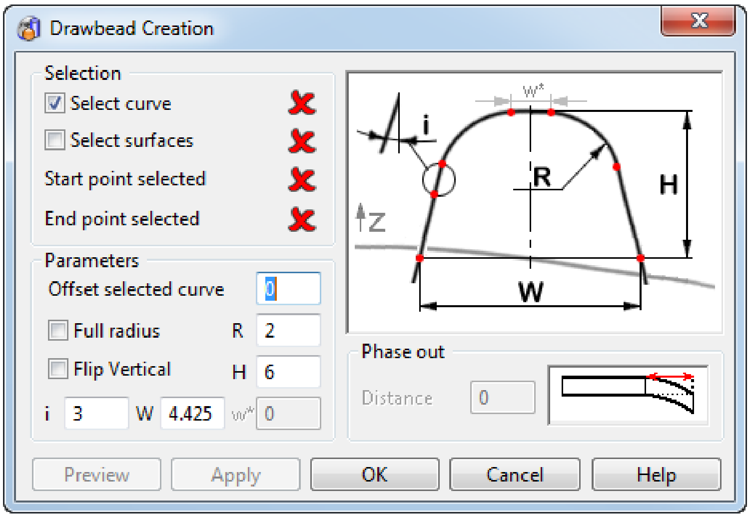
3 Click  to display the Drawbead Creation dialog (see page 354).

4 Use the options in the dialog to define the **Drawbead**.

5 When the **Drawbead** is correct, click **OK**.

Drawbead Creation dialog

This dialog lets you define the parameters to be used when creating the Drawbead surface.



Selection - The options in this section let you choose the curve, surface and points to be used to create the Drawbead surface.

Curve selected - Select the curve to be used and The cross ✘ icon changes to a tick ✔ icon.

Surface selected - Select the surface to be used and The cross ✘ icon changes to a tick ✔ icon.

Start point selected - Select the start point and The cross ✘ icon changes to a tick ✔ icon.

End point selected - Select the end point and The cross ✘ icon changes to a tick ✔ icon.

Parameters - The options in this section let you change the shape and size of the Drawbead surface by adjusting parameters.

Offset selected curve - The selected curve will be offset by the value you enter.

Full Rad - If *ON*, the radius **R** is applied to the profile and **w*** is automatically set to zero.

R - Radius of the profile.

Flip Vertical - If *ON*, the Drawbead is created in the -Z direction.

H - Height of the profile.

i - Angle of the side of the profile.

W - Width of the base of the profile.

w* - Width of the top of the profile. This is automatically set to zero if **Full Rad** is set to *ON*.

Preview - Select to preview the results. You may make changes and preview again.

Apply - Select to apply the results.

Phase out - This tapers off the end of the Drawbead surface over a specified distance.

Distance - Distance over which the phase out will occur.

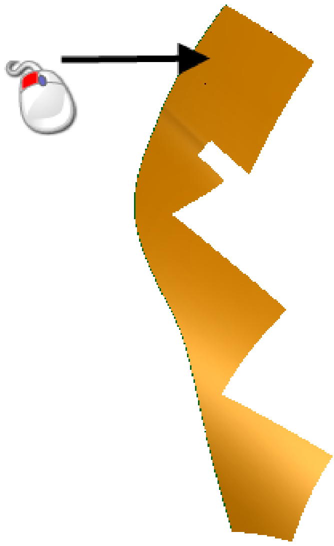
Untrimming a trimmed surface (Toolmaker mode only)

This option is only available in Toolmaker mode.

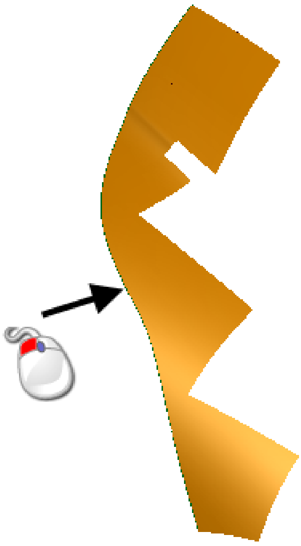
Create an untrimmed surface by selecting edges of existing trimmed surfaces and altering the boundaries out to make the surface larger. This is particularly useful for creating tangential split surfaces or addendum die faces.

To create an untrimmed surface from existing trimmed surfaces:

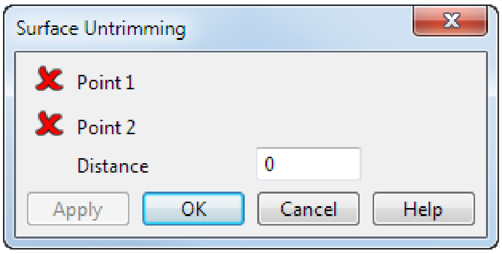
1 Select a previously trimmed surface or surfaces.



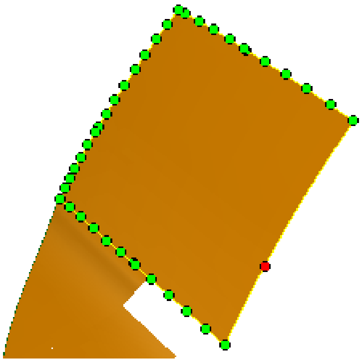
You can also select the composite curve that lies on the surface edges.



2 Click  to display the **Surface untrimming** dialog.



Green and red markers have been added to the edges of the surface, as shown below:



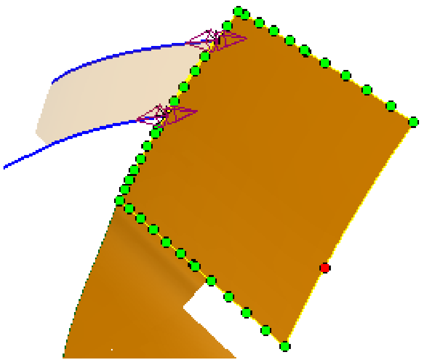
indicates that there is trimmed edge available.

indicates an untrimmed edge. Selecting a red marker will not allow you to create a surface.

3 Select **Point 1**.

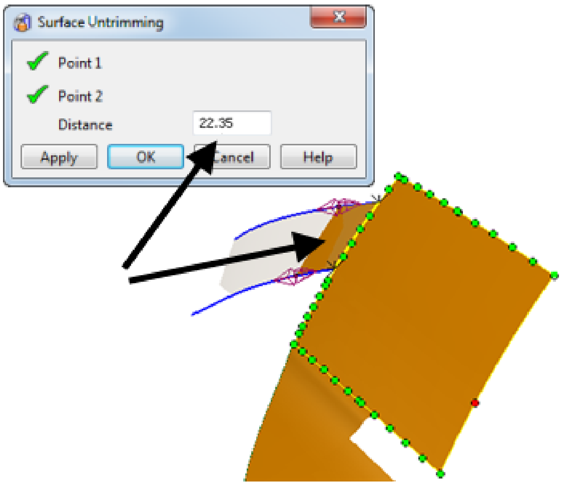
4 Select **Point 2**.

The surface extension and drag handles are displayed on the model.



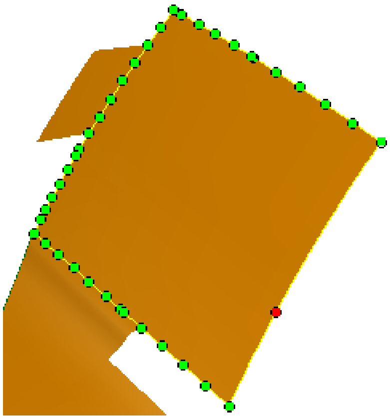
5 Define the extent of the untrimmed surface in one of the following ways:

- Use the drag handles that are displayed on the model. As you drag the handles, the value in **Distance** and the model are updated.



- Enter a **Distance** in the dialog. **Distance** is the actual distance measured along the extension surface, not the projected distance.

6 Click **Apply** to create the surface. The dialog is still displayed to let you create additional surfaces as required.



7 When you have created the required surfaces, click **OK**.