PowerSHAPE 2015 R2

Reference Help

Solid 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

Solid modelling

Use the following sections to find information on solid modelling:

Introduction to solids

The following sections provide general information on using solids in PowerSHAPE:

What is a solid?

A PowerSHAPE solid is a single object that represents the outer shape of a physical body. It is made up of a number of surfaces that join together and match at their edges within a given tolerance.

The shaded model below shows a PowerSHAPE solid.

The solid is displayed in the solid feature tree (see page 254).

This solid is made of many surfaces that join together as shown below.

You can operate on a solid in the following ways:

- As a single entity. For example, if you move a solid, all the surfaces which define the solid move.
- On a solid with another solid. For example, you can intersect a solid with another solid. The intersecting solid defines a feature in the solid feature tree.

Active and inactive solids

Many solid operations are performed on the currently active solid.

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

- If you activate an inactive solid, the currently active solid (if any) becomes inactive
- In a wireframe or shaded wireframe view, the wireframe of active and inactive solids are coloured differently. The exact colour combinations will depend on the colour scheme you are using. If there is currently no active solid in your model, then all solids are shown in their true colour.

For further details, see Activating and deactivating a solid (see page 269)

Parasolid in PowerSHAPE 2015 R2

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

In almost every way, the user interface remains the same. You should notice that the commands work faster and more reliably. The kernel will also make it easier for Delcam to develop new solid modelling commands in the future.

The Parasolid kernel is only used with the following products:

Delcam Estimator
Delcam Crispin SoleEngineer
Delcam PowerSHAPE
Delcam PowerSHAPE Pro
Delcam PowerSHAPE-e
Delcam Draft
Delcam Toolmaker
Delcam Electrodemaker
Delcam Orthotics
Delcam Designer
Delcam Designer-e
Delcam PowerMILL Modelling
PartMaker Modeling

Use the information in the following sections to understand more about using Parasolid:

Parasolid - the benefits (see page 7)

Parasolid - do I need to model differently? (see page 8)

Parasolid - overview of model conversion (see page 10)

Parasolid - overview of fixing models (see page 10)

Exporting to earlier versions of Parasolid (see page 11)

Parasolid - the benefits

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

- Faster and more robust solid modelling operations. Tests have shown that using the parasolid kernel is up to four times faster than using the PowerSHAPE kernel.
- Faster and more robust drawings.
- Faultless data conversion to and from other parasolid-based modellers. Higher quality transfer to and from other solid modellers because you can ask for parasolid data from customers instead of IGES or STEP.
- *The .psmodel* file size is much smaller if the part has been created wholly using the parasolid kernel. If there has been any conversion from version 8 solids or to/from surfaces, this is not the case.
- Import and export of Parasolid file formats will conserve the parasolid data with no loss of quality. At present the recommended formats are:

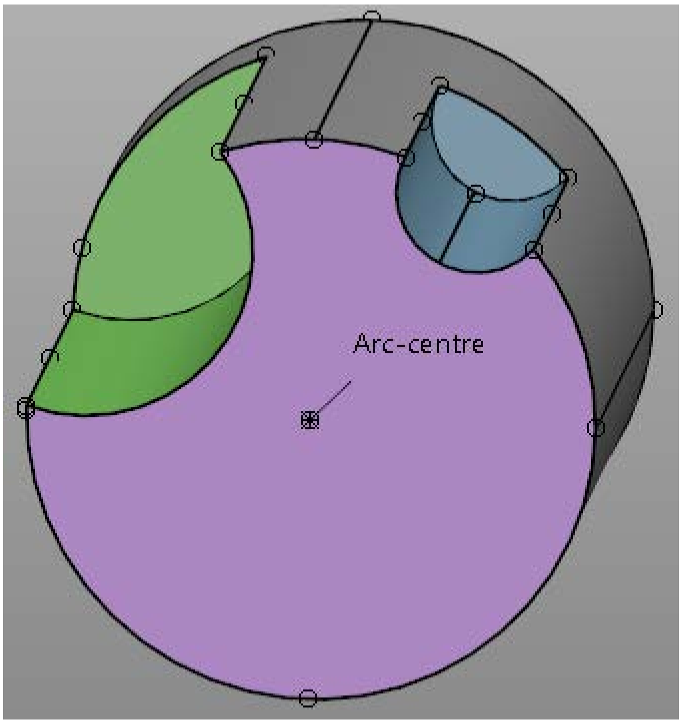
  Parasolid (x_t, x_b, xmt_txt, xmt_bin)

  The following formats that support Parasolid are available with Delcam Exchange installed.

  SolidWorks (sldprt)

  UG NX (prt)
- Crisper, tidier graphics.
- Automatically calculated snapping points for the following geometric points:
  - Corners.
  - Centres of planar faces.
  - Centres of arc edges.
  - Mid-points of straight line edges.
  - Tangent to arc edges.

The snapping points become visible when you enter creation mode.



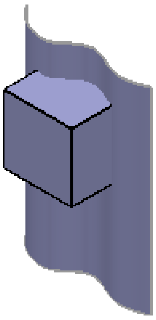
Parasolid - do I need to model differently?

Generally, you do not need to change the way you model in PowerSHAPE, but there are a few exceptions:

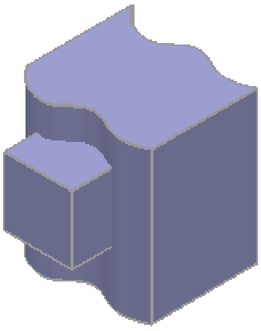
- If you want to boolean an open solid (sheet) to an active solid, and re-trim the active solid to the edges of the sheet,

  do one of the following:

  - Extend the sheet through the active solid.



    - Form a closed solid from the sheet.



- If you want to fill a hole in the active solid, use the **Sew** operation rather than a boolean addition.

- A scale transform causes a complete replay of the history tree. If the part has a very complex tree, this may take some time.
- Parasolid is particular about the wireframe you use for creating solids:
    - Ensure that there are no zero-magnitude points and that any point that is supposed to be tangent continuous really is continuous. If not, surfaces may be divided unnecessarily and operations (particularly thickening) may fail.
    - When creating extrusions or cuts/bosses from planar wireframe, always check that the wireframe really is planar. Parasolid uses a much tighter tolerance than PowerSHAPE when testing wireframe for planarity (flatness). If the wireframe is not planar, extrusions/cuts/bosses will not be capped and may fail.
    - Always try to create wireframe from lines and arcs rather linear or arc-like Bezier spans.

Parasolid - overview of model conversion

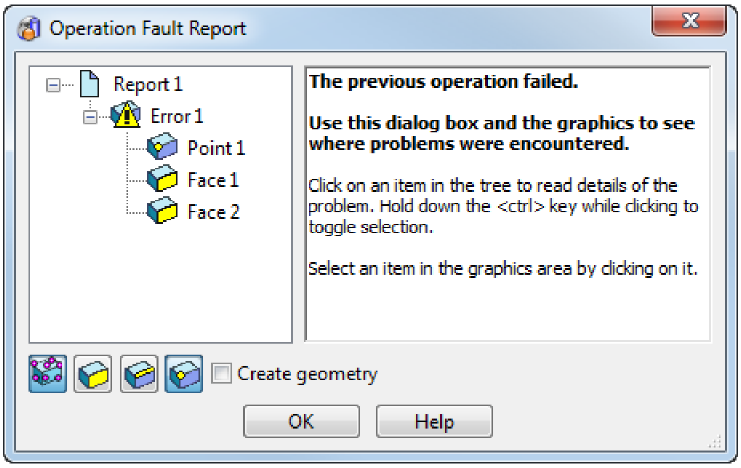
Separate, detailed information is available on converting models to and from an earlier version of PowerSHAPE (see page 21). The following is a summary:

- Existing models that were created using the PowerSHAPE kernel continue to work in PowerSHAPE 2015 R2
- New models use parasolid, unless you change the default by selecting the **Version 8 solids** option on the **Tools > Options > Object > Solids** dialog.
- Parasolid and version 8 solids co-exist in the same model.
- It is not possible to perform boolean operations between solids of different kernels. You must convert one solid before attempting the operation.
- It is possible to convert solids in existing models to parasolid and parasolid to version 8 solids.

Parasolid - overview of fixing models

There is separate and detailed information on fixing parasolid models. The following is a summary:

- Parasolid has stricter rules on 'valid' geometry. Geometry that is valid in version 8 models is not necessarily valid in a parasolid.
- **Solid Hints** may help you identify the areas of a model that are causing problems. If an operation fails when using a Parasolid solid, the **Operation Fault Report** dialog may be displayed.



The dialog indicates any problem areas when performing the operation. Use the options on the dialog to identify the area that requires attention.

- Faults in a parasolid body can be viewed and fixed using the Solid Doctor.

- Generally, you must fix a fault only if you are going to perform a modelling operation through the faulty area; for example, performing a boolean subtraction.

Use of colour when importing/exporting Parasolid files

Colours are a useful way to define machining intent, or pass other useful information between systems. Colours are used when importing and exporting Parasolid files:

- The wireframe colour of each face is exported to .dgk and Parasolid formats.
- Colours defined in other systems are retained when Parasolid files are imported into PowerSHAPE.

Exporting to earlier versions of Parasolid

You can export the following file formats to lower versions of Parasolid than the one currently used by PowerSHAPE.

- *x_t* (parasolid text)
- *x_b* (parasolid binary)

The table below indicates the parasolid export versions that are available from PowerSHAPE.

Parasolid version	Comment
7.0	The oldest version that can be used for export from the current version of Parasolid.
8.0	First version supported by Solid Works.
9.0	First version to support more than one solid per file.
9.1	
10.0	Latest version which Pro E Wildfire 4 can read/write
11.0	
11.1	
12.0	
13.0	
14.0	Major change to *x_t* file format. Use a later version than this if possible.
15.0	

15.1	
16.0	
16.1	
17.1	
18.1	
19.1	
20.0	
21.0	Used by PowerSHAPE 8.2
22.0	Current version of parasolid used by PowerSHAPE. This is displayed as the **Current version** option in the **Options** dialog.

PowerSHAPE exports to all version of parasolid that are supported by Solid Works.

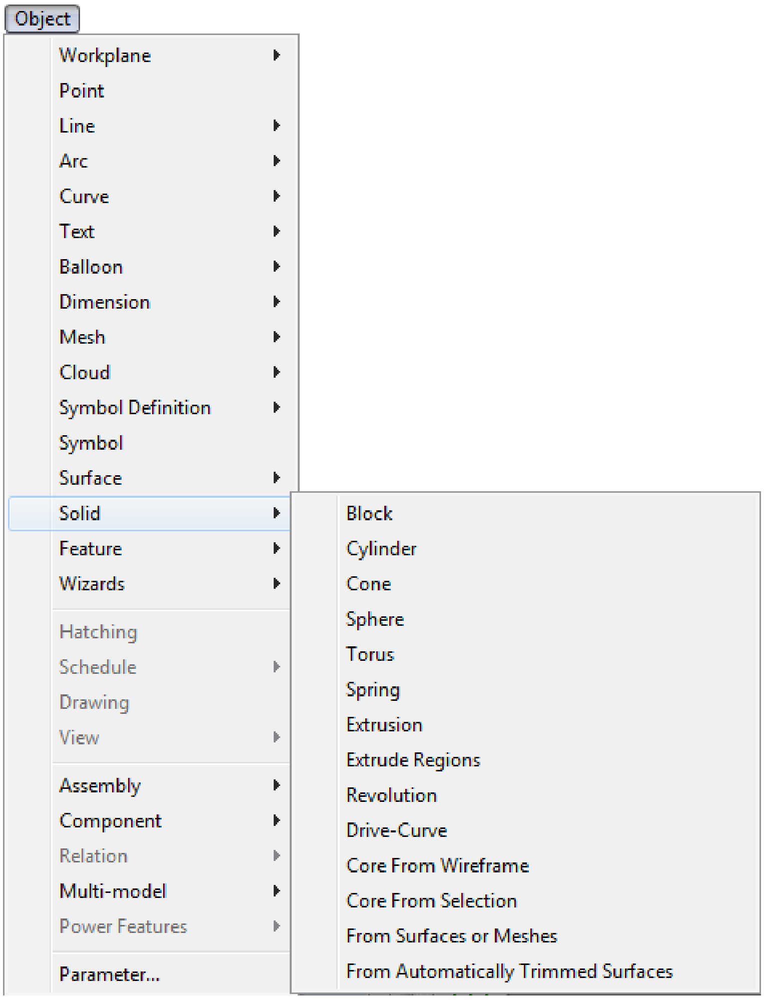
Creating a solid

You can create solids using one of the following methods:

- Using the solid creation options (see page 13)
- Using the solid modelling toolbar (see page 13)
- Using the feature creation options (see page 16)
- Using the wizard options (see page 18).

Using the solid creation options

1 From the **Object** menu, select **Solid** to display the **Solid creation** options.



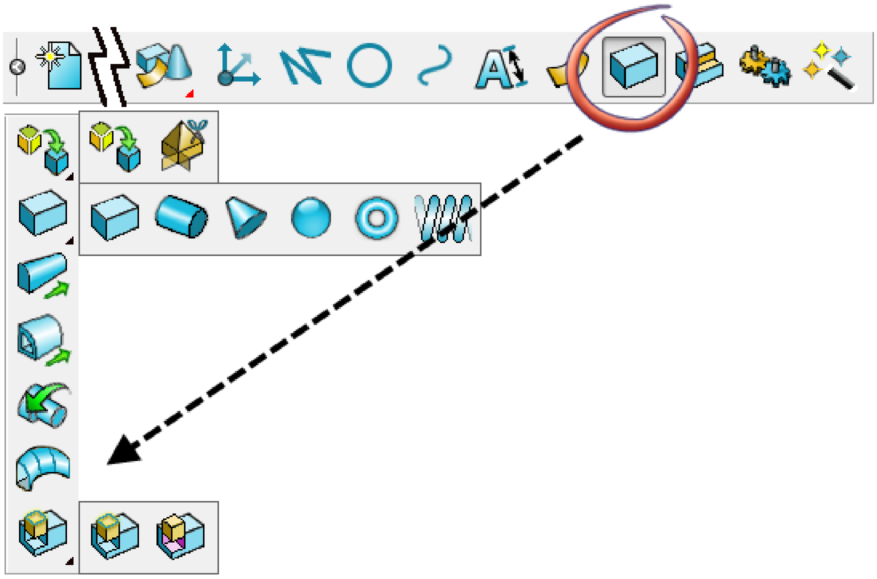
2 Select the option you require.

This is the same as clicking the **Solid**  button on the main PowerSHAPE toolbar and selecting an option from the **Solid** toolbar (see page 13).

Using the solid modelling toolbar

1 Ensure that you have a model displayed.

2 Select the **Solid** button on the Main toolbar. The Solid Modelling toolbar is displayed.



3 Select an option from the toolbar:

Creating a solid from surface and meshes (see page 37)

Creating a solid by automatically trimming surfaces (see page 38)

Create solid block (primitive) (see page 42)

Create solid cylinder (primitive) (see page 42)

Create solid cone (primitive) (see page 42)

Create solid sphere (primitive) (see page 42)

Create solid torus (primitive) (see page 42)

Create a solid spring (primitive) (see page 42)

Create solid extrusion (see page 56)

Extrude regions (see page 68)

Create a solid of revolution (see page 73)

Create solid from drive curve and sections (see page 79)

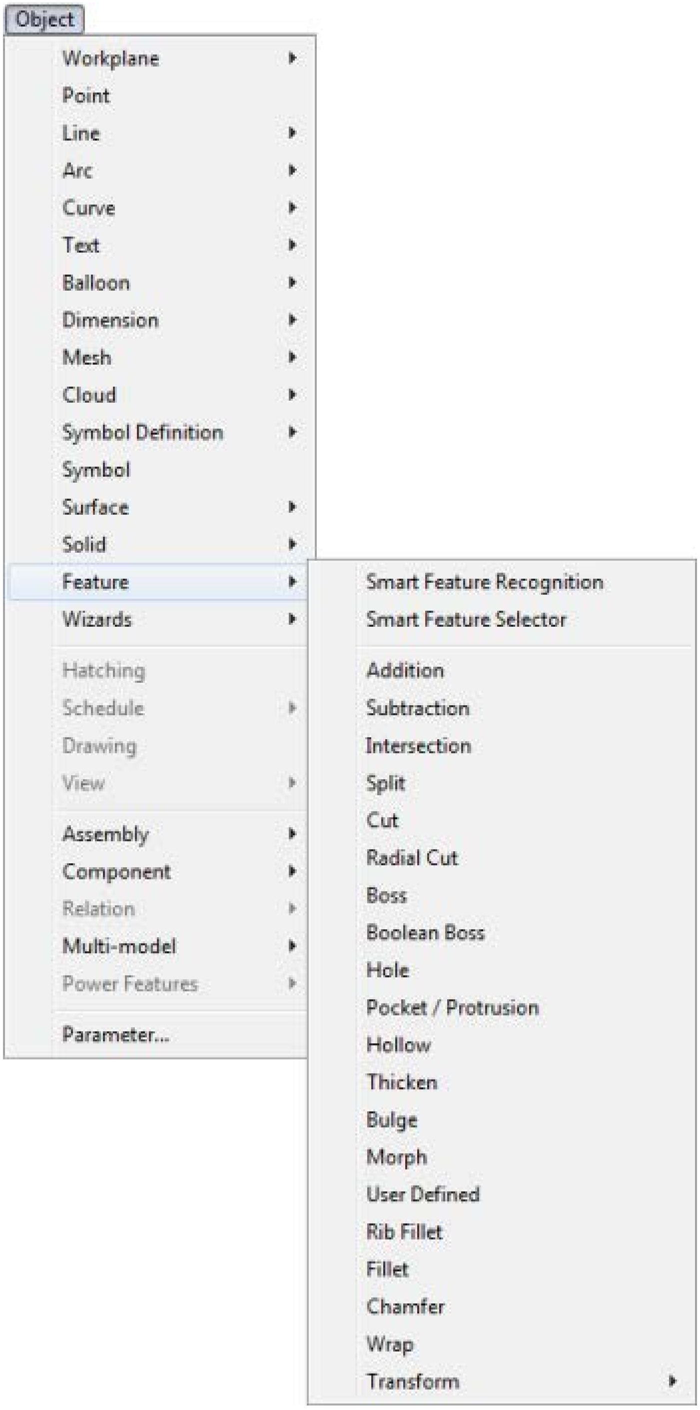
 Create solid core (see page 86)

 Create a solid core from selection (see page 87)

When you start PowerSHAPE, selecting the **Solid**  button automatically starts primitive block creation.

Using the feature creation options

1 From the **Object** menu, select **Feature** to display the **Solid Feature** options.



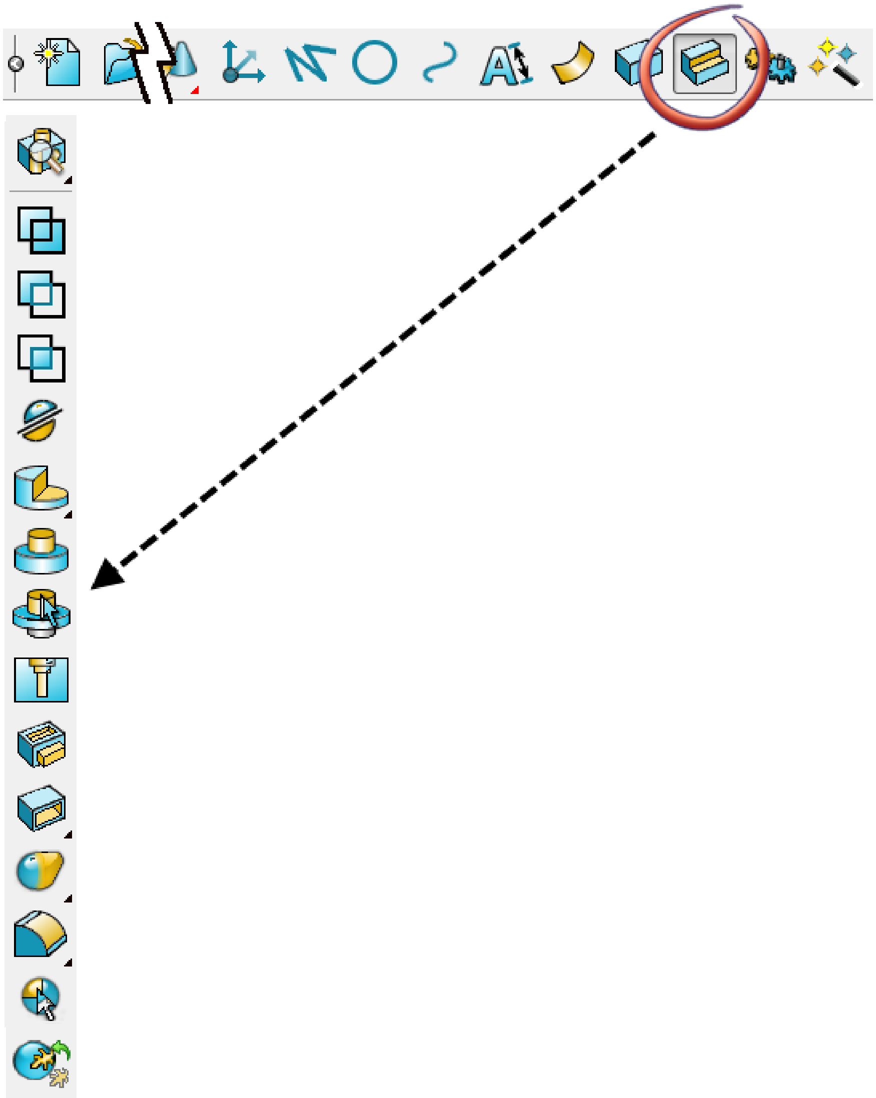
2 Select the option you require.

This is the same as clicking the **Feature**  button on the main PowerSHAPE toolbar and selecting an option from the **Solid Feature** toolbar. Transform options (see page 243) do not appear on the **Solid Feature** toolbar.

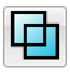
Using Features in solid modelling

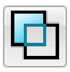
1 Ensure that you have a model displayed.

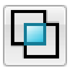
2 Click **Feature**  (*Main toolbar*). The **Solid Feature** toolbar will be displayed.



3 Click one of the following buttons to create the required feature:

 Solid from addition (see page 108)

 Solid from subtraction (see page 110)

 Solid from intersection (see page 112)

 Solid split (see page 117)

 Solid cut (see page 122)

 Radial cut (see page 134)

 Solid boss (see page 138)

 Solid boolean boss (see page 148)

 Solid hole (see page 151)

 Pocket or protrusion (see page 227)

 Hollow solid (see page 167)

 Thicken solid (see page 174)

 Bulge (see page 176)

 Morph feature (see page 311)

 Rib fillet (see page 225)

 Solid fillet (see page 180)

 Solid chamfer (see page 206)

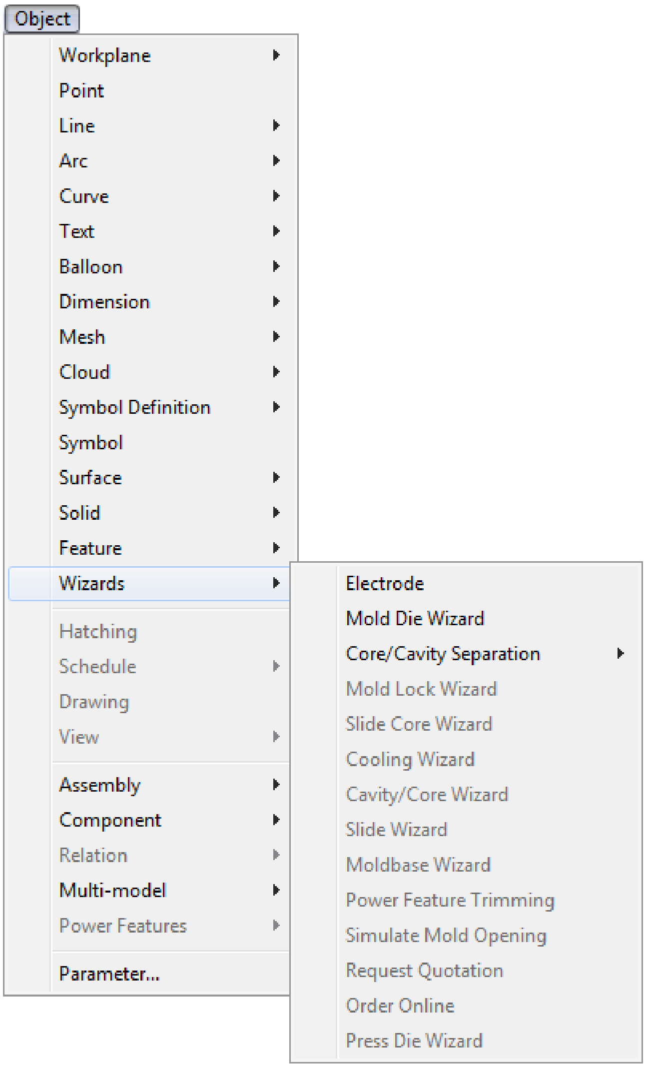
 User defined feature (see page 239)

 Solid wrap feature (see page 242)

*When you start PowerSHAPE, clicking **Solid Feature***  *will automatically start the hole creation option if there is an active solid present. If there is not an active solid, the **Solid** toolbar is displayed, but does not start the hole creation option.*

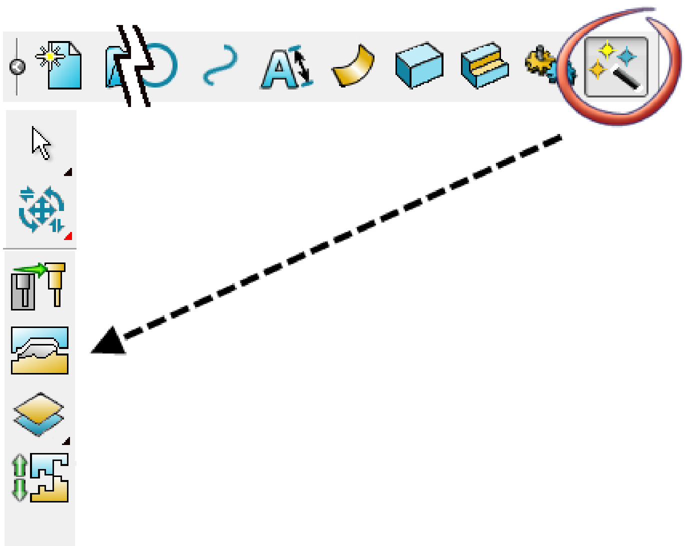
Using the wizard options

1 From the **Object** menu, select **Wizards** to display the available wizards options.



2 Select the option you require.

This is the same as clicking the **Wizard**  button on the main PowerSHAPE toolbar and selecting an option from the **Wizards** toolbar.



Create an electrode using **Delcam Electrode**.

Start the Mold Die Wizard using **Delcam Toolmaker**.

Surface Core/Cavity Separation. Use this option to find either the visible or hidden surfaces of a model.

When importing a model (e.g. via IGES), from which a mold tool will be designed, it is necessary to split the surfaces into those which form the outer face of the component (the cavity side of the mold) and those which form the inner face of the component (the core side of the mold). Some models (especially IGES) can contain thousands of surfaces, some of which will be in close proximity, but each describing different parts of the mold.

Separating them manually would be a difficult and tedious task, but this option calculates the split quickly and easily.

The split is shown either by selecting all the visible surfaces or all those which are not visible. The view point is down the Z-axis of the active workplane (or if there is not an active workplane, the Z-axis of the global workspace). Use the options as follows:

1 Open the model.

2 Add a workplane if the split is not going to be down the Z-axis of the global workspace and position it accordingly.

3 Click **Wizards** to display the wizards toolbar.

4 Click **Core/Cavity** flyout (*Wizards toolbar*)

5 Click the appropriate option on the flyout.

   - **Visible Surfaces**. This finds and selects all the visible surfaces of a model.

   - **Hidden Surfaces**. This finds and selects all the hidden surfaces of a model.

   You can now move the selected surfaces to a new level. Any surfaces that are entirely parallel to the point of view will not be selected.

Use the **Solid Core/Cavity Separation** (see page 302) wizard to split a solid into core, cavity, sliders and internal features (such as holes)

*Additional Toolmaker options are available from the Wizards toolbar when **Toolmaker** is selected from the **Module** menu.*

Converting solids

If a model contains post-version 8 solids (Parasolids), the solids need to be converted manually before exporting back to an earlier version that doesn't support Parasolids.

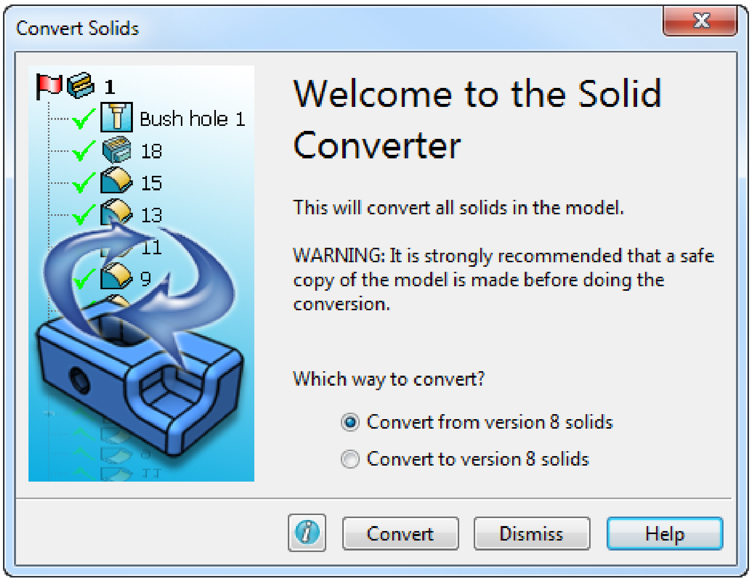
Use the following tools to convert solids to and from version 8 solids

- Use **Tools > Convert all solids** to convert all solids in a model (see page 21).
- Use the **Convert** button  on the **Solid Edit** toolbar to convert a single solid (see page 23).
- Use **Edit > Convert > Solids to version 8 solids** and **Edit > Convert > Solids from version 8 solids** to convert multiple solids.

Convert all solids

1 Click the model that contains the solids.

2 Select **Tools > Convert all solids** to display the **Convert Solids** dialog.

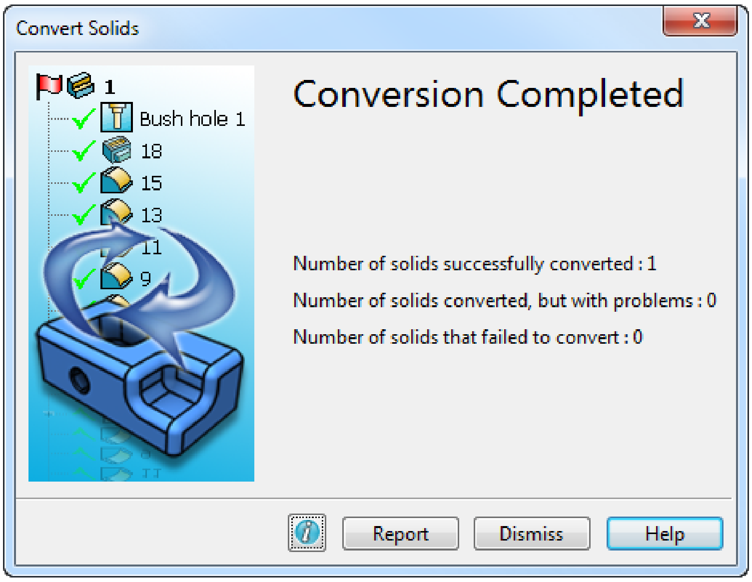


3 Select the conversion option you require.

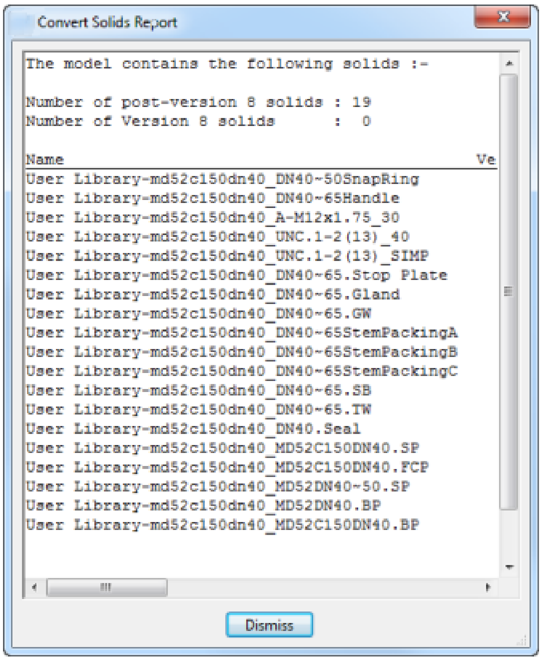
*To check the type of solid in your model, right click on a solid. If a solid is version 8, **Version 8** is displayed in the title of the solid context menu.*

4 Click **Convert**.

5 Click **Yes** to confirm that you want to convert all the solids in a model. A summary of the results of the conversion is displayed in the dialog.



6 Click to display the **Convert Solids Report**.



For each solid, the report displays:

- model name.
- whether the solid is a version 8 solids or a post version 8 solid.
- level the solid is on.
- component the solid belongs to.

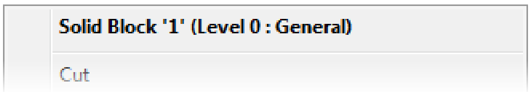
This option helps identify version 8 solids in old library models.

7 Click **Report** to display the **Convert Solids Report** detailing any issues with the conversion.

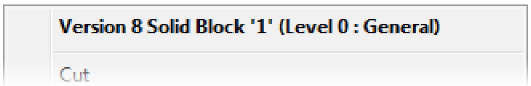
8 Right click on the converted solid to display the solid context menu.

This will say:

Solid for post-version 8 solids.



Version 8 Solid if the solid is version 8.



Convert selected solids

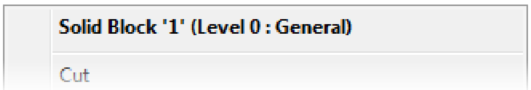
1 Click the solid to be converted to display the **Solid Edit** toolbar.

2 Click .

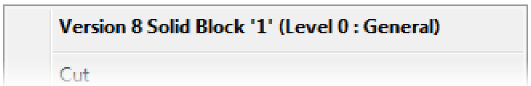
3 Right-click on the converted solid to display the solid context menu.

This will say:

Solid for post-version 8 solids.



Version 8 Solid if the solid is version 8.



Editing a solid

Use one of the following techniques to edit a solid:

- From the main menu:

  1 Select the solid.

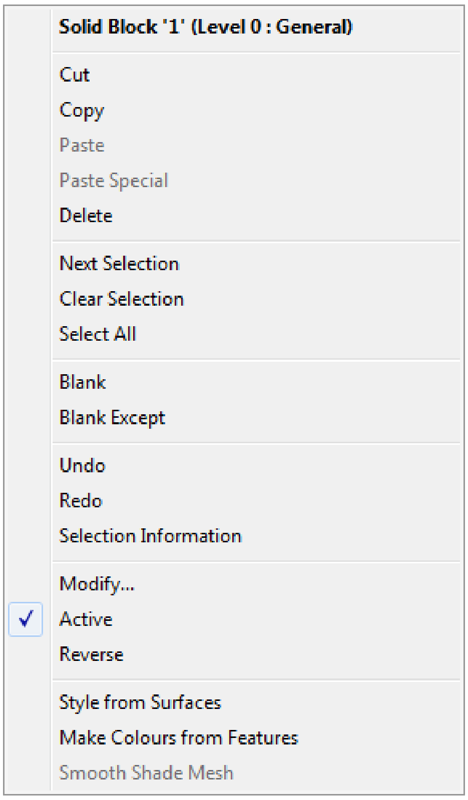
  2 Select one of the following:

  Edit > Convert > Solids to surfaces converts a solid into surfaces that define it. You can then edit the surfaces.

  Edit > Modify displays the solid feature tree.

- Click  (*Solid Edit toolbar*) to display the solid feature tree.

- Use the options on the **Solid Edit** toolbar (see page 24). The **Solid Edit** toolbar is displayed automatically when you
  - start to edit a solid.
  - select one or more solids.
  - selecting **View > Toolbars > Solid Edit**.
- If required, create a workplane at the centre of a face of a solid (see page 27).
- Right-click the solid to display the edit options available for solids. The name of the solid and the level on which it lies is at the top of the context menu.



When you reverse a solid, it loses its feature tree.

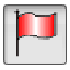
Using the Solid Edit toolbar

To edit a solid using the **Solid Edit** toolbar:

1. Select a solid to edit. The **Solid Edit** toolbar is displayed:

2. Click one of the following buttons to edit the solid:

 **Show/Hide Tree** — Click this button to control the display of the solid tree. The solid tree window is hidden by default. Clicking this button is the same as **View > Windows > Tree**.

  **Activate/Deactivate** — Toggle the button to activate or deactivate the solid.

 **Fix** — Click this button to find and fix faults. One of the following is displayed:

- **Solid Doctor** if the solid is a Parasolid (post-version 8 solids).
- **Make Watertight** if the solid is a PowerSHAPE solid (version 8 solids).

 **Heal/Loosen Edges** — Click this button to heal or loosen edges in the solid using the **Solid Doctor Healing** dialog (see page 293).

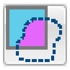
 **Select Features** — Select this button and click on the solid to select the feature (see page 34).

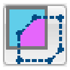
 **Select Faces** — Select this button and select an individual face of a solid (see page 34). Use this option with box selection to select faces that are completely inside the selection box.

 **Select Continuous Faces** — Select this button to highlighted faces that are continuous with the selected face.

 **Select Convex Faces** — Select this button and click on a face in the solid to select all the faces in the convex region of the face.

 **Select Concave Faces** — Select this button and click on a face in the solid to select all the faces in the concave region of the face.

 **Continuous Lasso** — Select this button to select faces of the solid by drawing a lasso around the required faces. After making the selection, move the mouse over the selected faces to highlight the single faces.

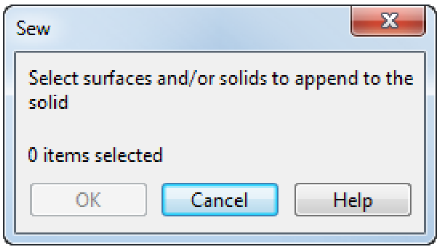
 **Discrete Lasso** — Select this button to select faces of the solid by drawing a lasso around the required faces. The cursor changes to  to show that you can complete the lasso. After making the selection, move the mouse over the selected faces to highlight the single faces.

 **Copy** — Click this button and select one or more faces to create surface copies.

 **Cut** — Click this button and select one or more faces to create surface copies and remove them from the solid.

 **Sew** — (Available only for post-version 8 solids).

a Click this button to display the **Sew** dialog.



b Select one or more surfaces or solids and click **OK**.

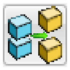
 **Remove and Heal** — Click this button to remove the selected faces and close the gap by modifying the surrounding faces. This option is part of Direct Modelling functionality.

 **Merge** — Click this button to merge selected faces of a solid into one face (see page 27).

 **Divide Faces** — Click this button to divide the faces of the selected solid using wireframe (see page 28).

 **Draft** — Click this button, to display the **Draft Faces** dialog. This option is part of Direct Modelling functionality.

 **Replace** — Click this button to use **Solid Replace Faces** to select faces of a solid and replace them with other existing faces or surfaces.

 **Divide Solid** — Click this button to divide an existing solid into separate solids (see page 31).

Convert — Click this button to convert to/from version 8 solid (see page 21).

Solid to Surface — Click this button to convert one or more solids to surfaces.

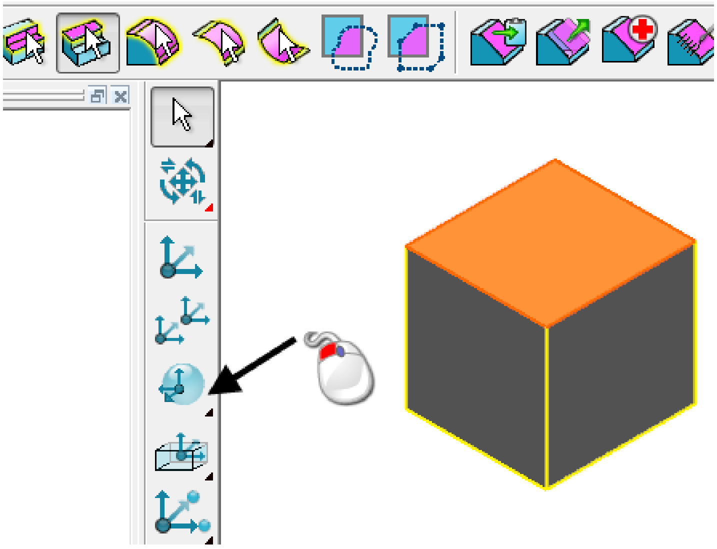
Solid to Mesh — Click this button to convert one or more solids to meshes.

Use normal selection methods: *to add/remove a face from the selection;* *to add a face to the selection.*

Creating a workplane at the centre of a face of a solid

You can create a workplane at the centre of a face of a selected solid.

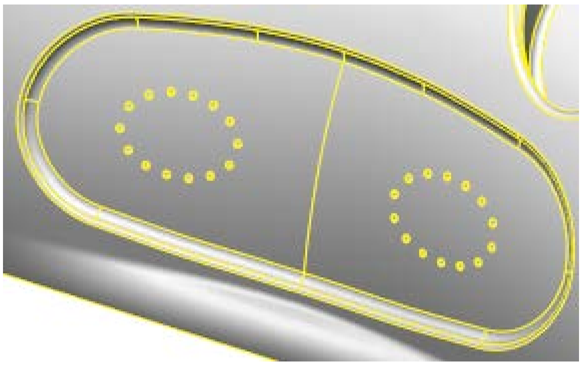
Click **Create a single workplane aligned to geometry** *(Workplane toolbar)* to create a workplane in the centre of the selected face.



Merge selected faces

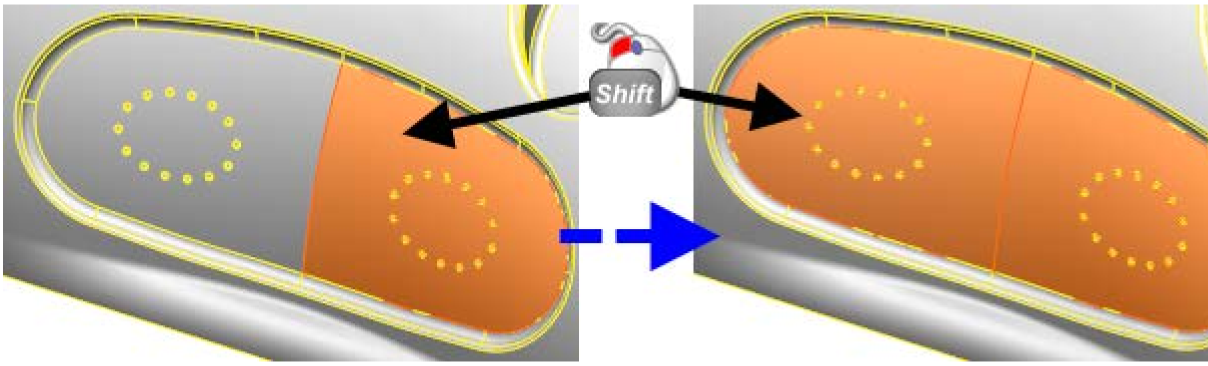
Use the **Merge selected faces** functionality to join multiple continuous solid faces together, into a single face.

1 Select a solid.

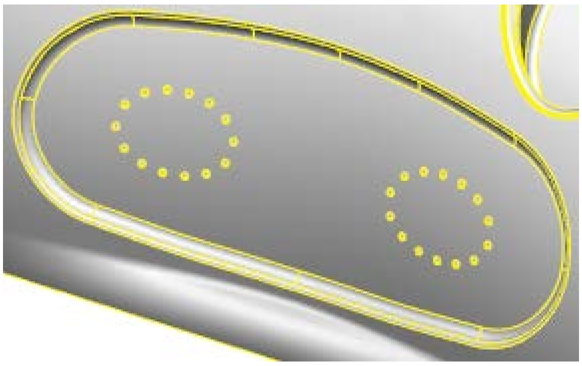


2 Select **Select individual faces**  from the **Solid Edit** toolbar.

3 Use  to select two or more continuous faces.



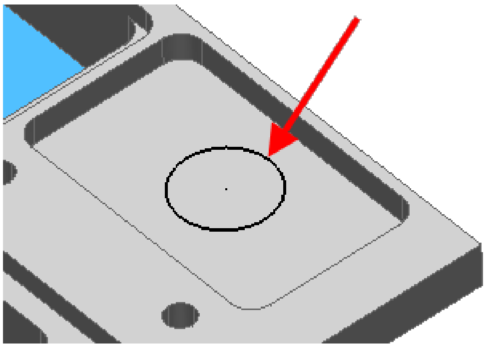
4 Click  on the **Solid Edit** toolbar to merge the faces together.



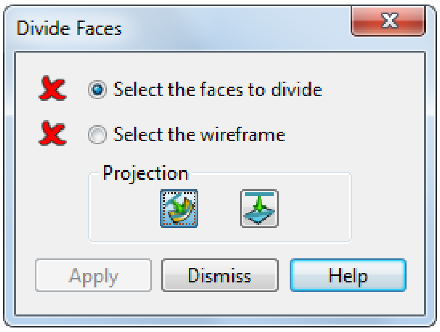
Divide face

Use **Divide face**  (*Solid Edit toolbar*) to divide the faces of the selected solid using wireframe. This option is useful if you need to make small, localised changes, or to limit the effect of Direct Modelling editing operations.

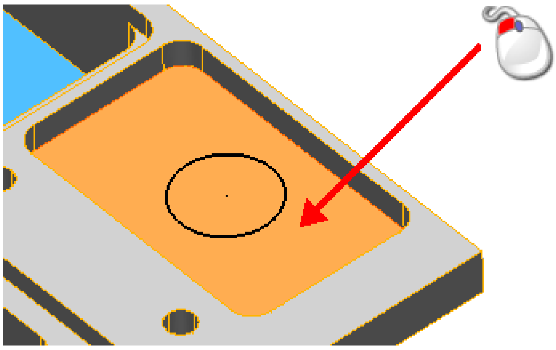
1 Create the wireframe that will be used to divide the face.



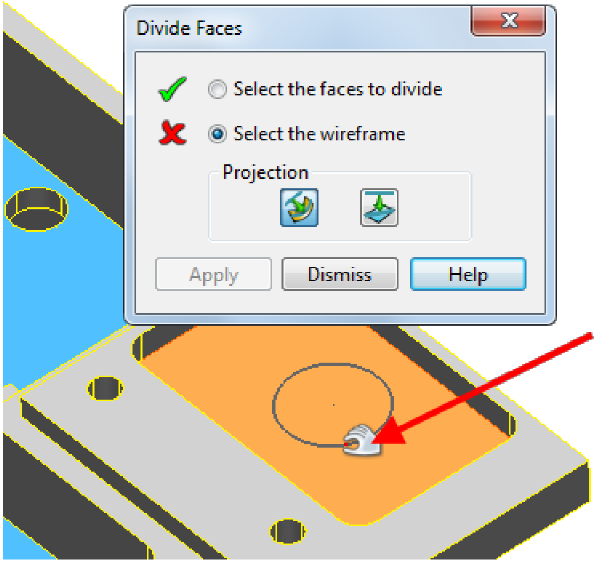
2 Click  (*Solid Edit toolbar*) to display the **Divide Faces** dialog.



3 Select the face to divide.



4 Select the wireframe.

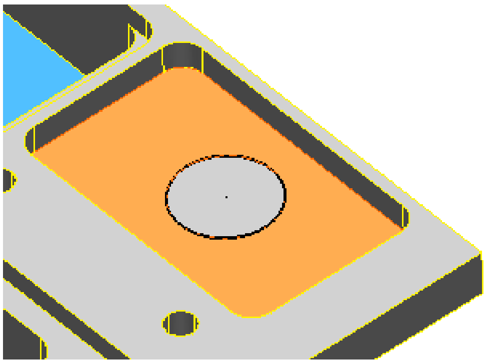


5 Select an appropriate projection option. **Project wireframe along face normal** is the default setting.

6 Click **Apply**. If you want to split another face, you can do this without closing the dialog.

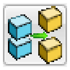
7 Click **Dismiss**.

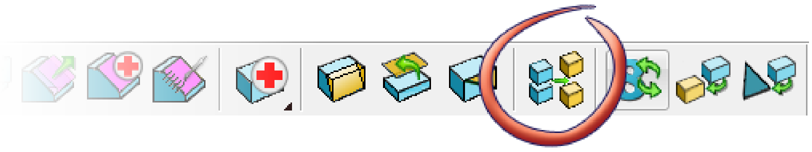
8 Move the cursor over the face to show the divided face or click one of the faces.



If you have an example where the wireframe does not completely divide the face, the Operation Fault report will be displayed. Add additional wireframe and repeat the divide face operation using both the original and additional wireframe.

Divide solid

Use the **Divide solid**  button (*Solid editing toolbar*) to divide an existing solid into separate solids.

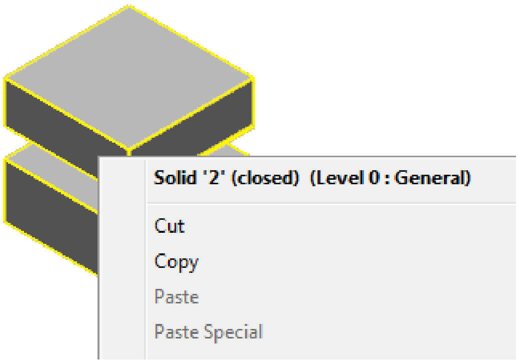


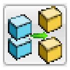
This is useful for dividing:

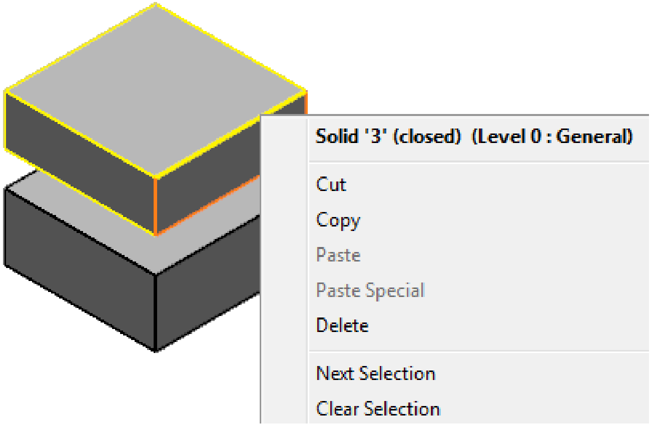
- a disjoint solid into two solids. (see page 31)
- a solid into inner and outer solids (see page 32).

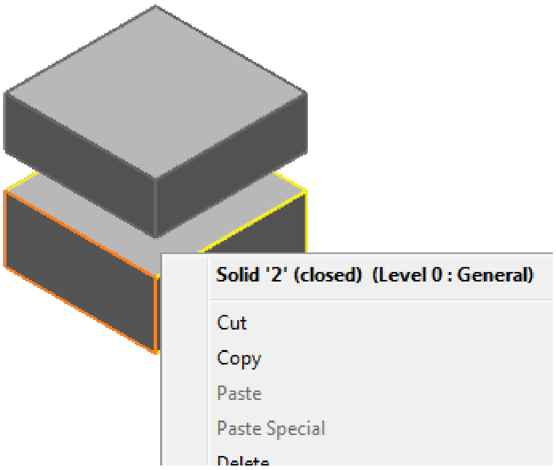
Example — Dividing a disjoint solid into two solids

1 Select the disjoint solid



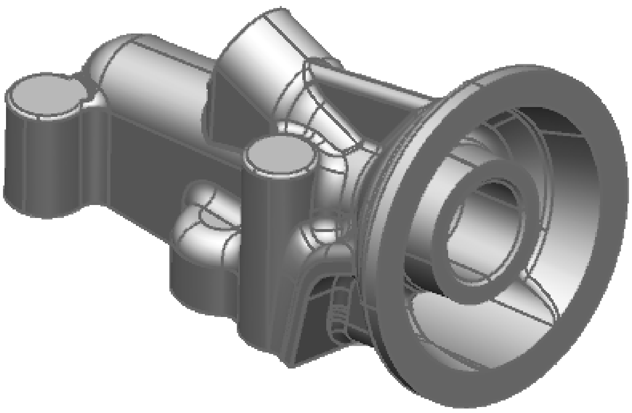
2 Click  (*Solid edit toolbar*). The disjoint solid is divided into two separate solids.



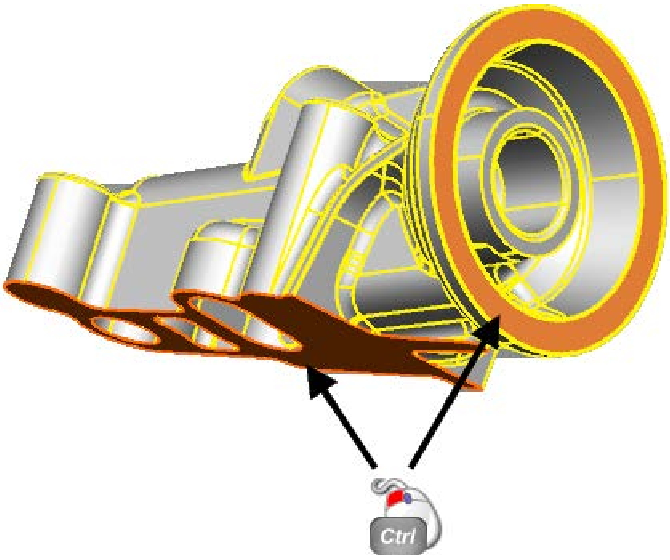


Example — Dividing a solid into inner and outer solids

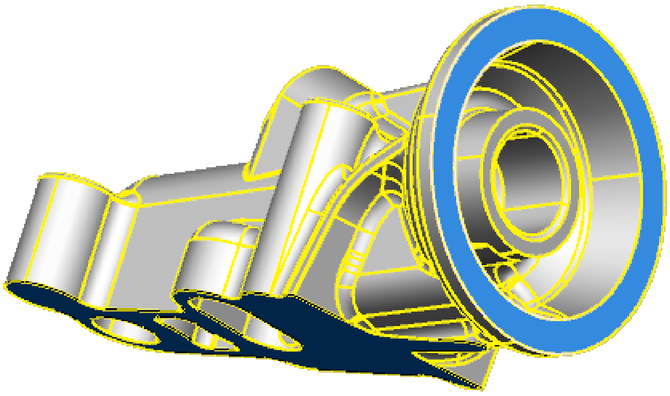
You can use **Divide solid** to divide a solid into an inner solid and an outer solid.



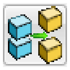
1 Select the two faces that join the inner and the outer portions of the solid.

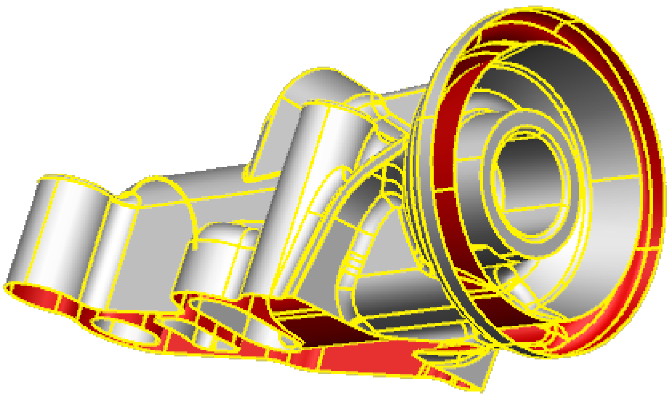


2 Click  (*Solid edit toolbar)* to extract the faces.



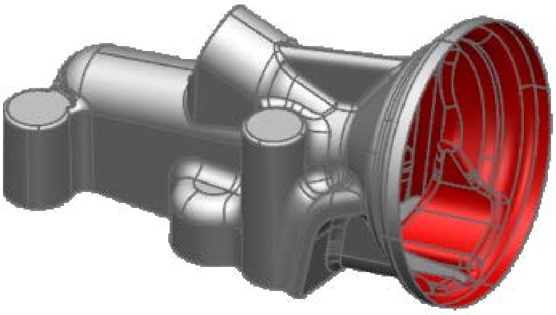
3 Blank the faces you have extracted.

4 Click  (*Solid edit toolbar*) to divide the solid.

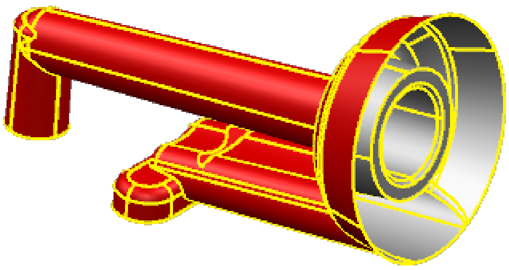


5 Use **Blank Except** to display the solids separately:

- Outer solid.



- Inner solid.



Selecting solid features and faces

Selection modes on the **Solid Edit** toolbar highlight the areas under the cursor.

Selecting solid features and faces of a solid are mutually exclusive.

1 Click one of the following buttons on the **Solid Edit** toolbar:

 faces only.

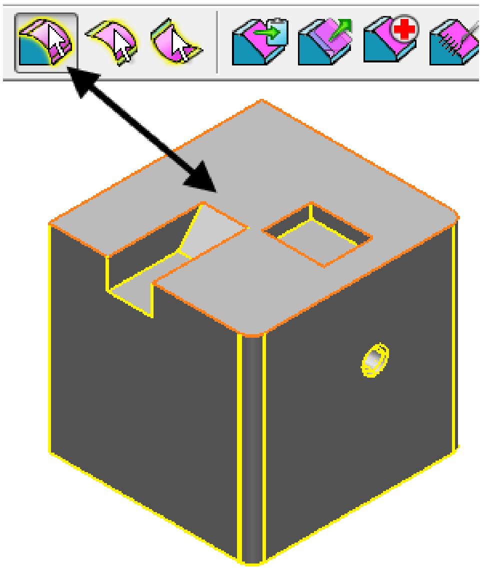
 features only.

 continuous regions.

 convex regions.

 concave regions.

2 Move the cursor over a model to see the faces that will be selected.



Feature selection

- When a solid is selected, the **Feature selection** mode is automatically used.
- If the solid has no history tree, single-face selection mode is used and  is unavailable.
- Selecting features from the tree browser switches the mode to **Feature selection** mode.
- Use the mouse to make/amend selection as follows:

 replaces the current selection.

toggles the item into/out of selection.

 adds the item to the selection.

displays a feature edit dialog.

*As well as using the **Solid Edit** toolbar buttons to switch between feature and face selection modes, there is a* click-again *option. Click-again is a slow double-click. It is set to four times the MS Windows time for a double-click. So if the MS Windows double-click speed is 0.5 seconds, a* click-again *is 2 seconds).*

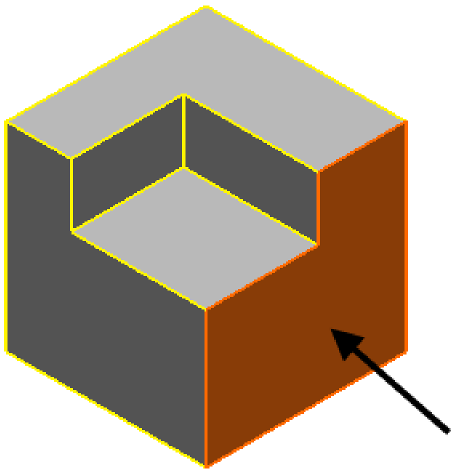
Editing surfaces of solids

You can edit, remove and copy a face of a solid without converting it to surfaces.

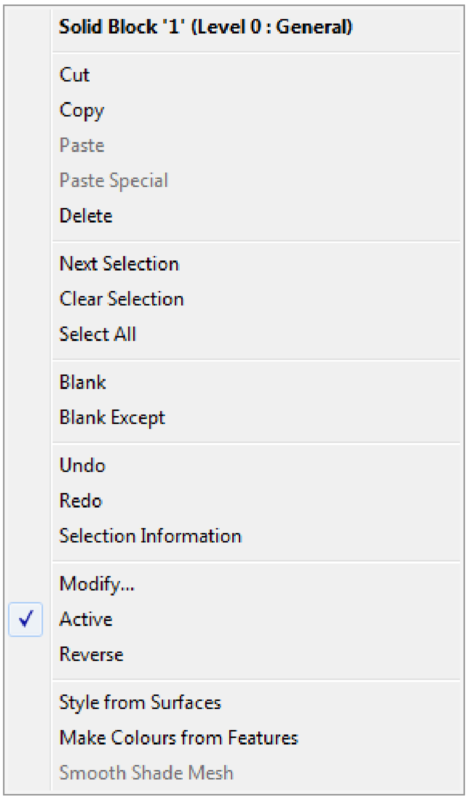
1 Select the solid.

2 Click twice, slowly, to select the face of the solid you wish to edit. The selected face will be highlighted.

   If the selected face is part of a feature with a single visible face, it will be highlighted in a different colour. The colours that are used will depend on the colour scheme you are using.



3 Right-click the highlighted surface and select the required operation from the **Solid** context menu.



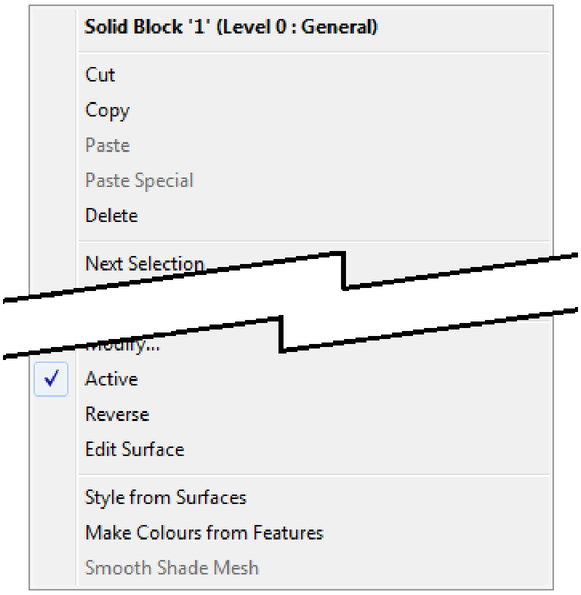
You can use the options to change the appearance of the faces of the solid. (see page 37)

Changing the appearance of the faces of a solid

To change the appearance of the faces of a solid:

1 Right-click on a selected solid.

2 Select the required option from the **Solid** context menu.



Style from Surfaces - If deselected, the style and material of faces in a solid comes from the solid. If selected, the style and material of the faces in the solid is stored with each face of the solid.

Make colours from features - Sets the material of each feature to a particular material colour. The colour depends on the type of feature. If **Style from Surfaces** is deselected, selecting **Make colours from features** will also select **Style from Surfaces**.

Copy All Surfaces

You can create copies of all the surfaces within a selected solid. The solid and the history tree are unchanged.

1 Select the solid

2 Select **Edit > Convert > Copy surfaces within solids**.

Creating a solid from surfaces

You can create a solid from a single surface or a group of surfaces.

1 Select the surfaces you want to change into solids.

2 Click (*Solid toolbar*). The surfaces are created into solids and the solids appear in the solid feature tree.

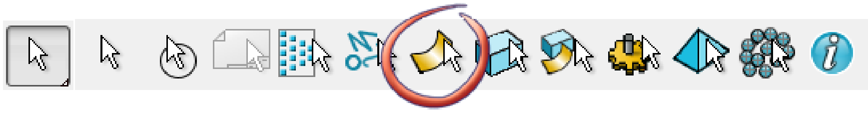
- Any surface with width less than the creation tolerance is excluded from the solid.
- If trim boundaries exist on a surface, then they define trim boundaries on the solid too.
- If there is no active solid in your model and only one solid is created from the selected surfaces, the new solid becomes active.
- If your new solid has either small holes or contains more than 20 surfaces, you are asked if you want to repair the solid. If you click **Yes**, the **Solid Doctor** dialog (Parasolids) or **Make Watertight Wizard** (see page 296) dialog (Version 8 solids) is displayed to help you repair the solid.

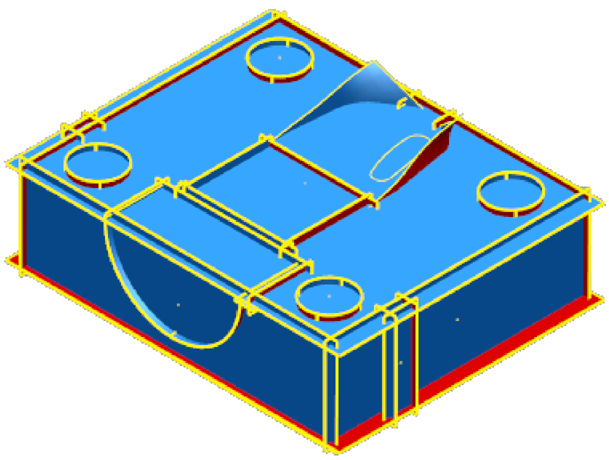
Creating a solid from untrimmed surfaces

Use the automatic trimming functionality to generate a solid from a group of intersecting untrimmed surfaces.

This functionality is available only with PowerSHAPE Pro and Toolmaker.

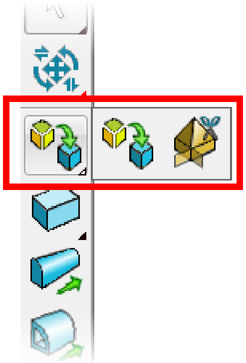
1 Click **Quick select all surfaces** .



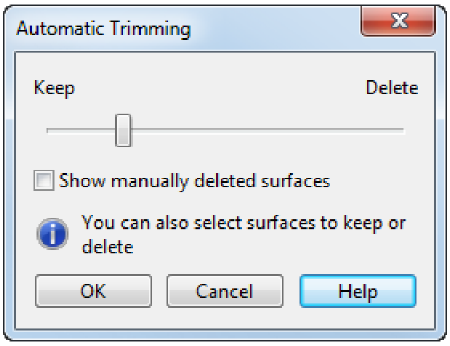


2 Click **Solid** .

3 Click **Automatically trim surfaces and create a solid**  from the flyout to display the **Automatic Trimming** dialog.



4 Use the slider on the **Automatic Trimming** dialog to control how many surfaces are used for trimming and generating a solid.



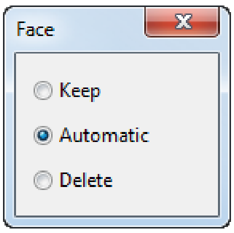
PowerSHAPE estimates the proportion of surfaces to keep and delete. This is indicated by the initial position of the slider when the dialog is displayed.

- Additionally, you can manually select surfaces to display the **Face** dialog. Use the following options to include or exclude surfaces from trimming:

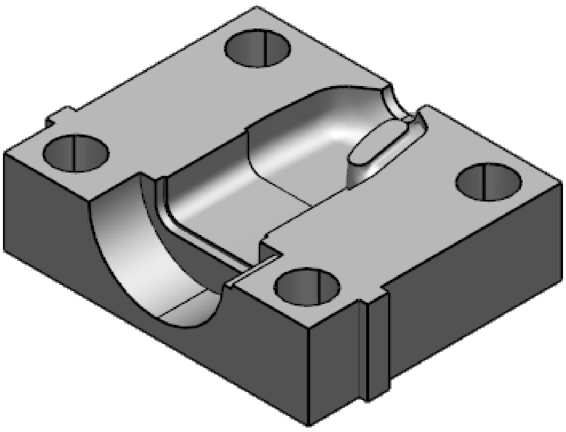
  Keep — select this to include the surface when trimming. The surface is highlighted in green to indicate this option has been selected.

  Automatic — select this to let PowerSHAPE calculate the option to keep or delete the surface with respect to your slider setting.

  Delete — select this to exclude the surface when trimming. The surface is hidden unless **Show manually deleted surfaces** is selected.



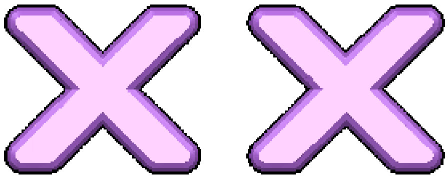
5 Click **OK** to close the dialog and generate the solid.



The surfaces used to create the solid are moved to a new level when the trimming has finished. Additionally, the created solid is made active if no other active solids exist in the model at the time of creation.

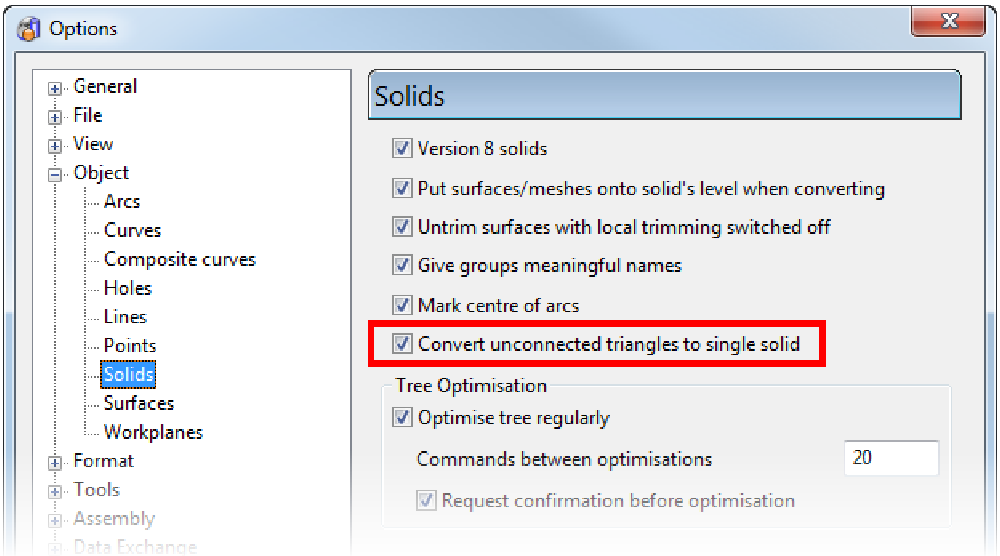
Creating a single solid from unconnected triangles

Unconnected triangles can be converted to a single solid. Using the example symbols below:

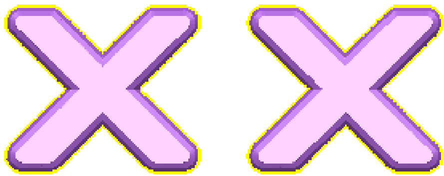


1 Select **Tools > Options > Object > Solids** to display the solid modelling options.

2 Select **Convert unconnected triangles to single solid**.

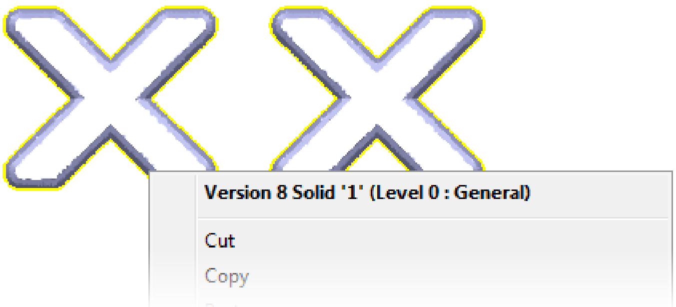


3 Select the symbols.



4 Click  (*Solid toolbar*).

The two symbols are converted to a single solid.



Primitive solids

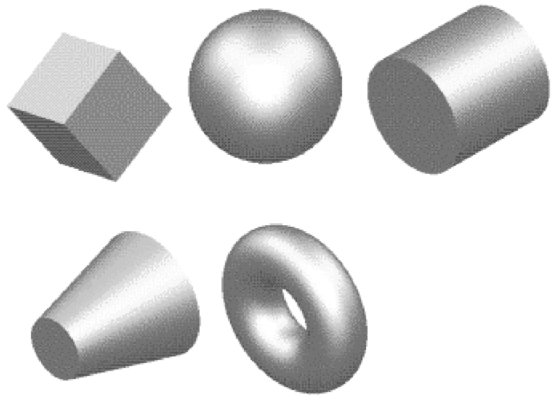
The following sections contain information on modelling with primitive solids:

Creating a primitive solid (see page 42)

Editing a primitive solid (see page 43)

Creating a primitive solid

You can create solid primitives similar to the surface primitives.



1 Click the required primitive type (*Solid toolbar*).

 block

 sphere

 cylinder

 cone

 torus

 spring

2 Enter a position on the screen to create a default primitive solid of the required type. The solid appears in the solid feature tree as a solid and the instrumentation is visible.

If there is no active solid in your model, this solid becomes active. Otherwise it becomes an inactive solid.

You can create more solids of the same type by entering more positions.

If, at this stage, you wish to create a different primitive solid, select it from the **Solid** toolbar and the cursor will change to indicate the type of solid that will be created. To exit primitive creation, click **Select**  .

3 If required, edit the solid you have created by using the graphical handles that are displayed or by using the appropriate primitive dialog that is displayed by selecting **Modify** from the context menu or double-clicking the primitive solid.

Creating a solid 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*, type the following in the command window:

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

If you don't specify a particular dimension, the default value is used.

You can specify the following dimensions using the command window:

radius

width

length

height

minor_radius

major_radius

base_radius

top_radius

Editing a primitive solid

Use one of the following techniques to edit a primitive solid:

- Use the graphical handles to change the dimensions and orientation of the primitive.
- Use one of the options from the **Edit** menu.

  Edit > Modify displays the dialog for the solid.

  Edit > Convert > Solids to surfaces converts the solid into surfaces for editing.
- Double-click the primitive solid to display the dialog for the solid.
- You can simultaneously edit multiple primitive solids (see page 54).
- Select one or more faces on a selected primitive solid by clicking one of the face selection buttons (*Solid Edit toolbar*) and selecting the required faces.

- Double-click the icon of the primitive solid in the tree to display its dialog and graphical handles. Double-clicking a block, sphere, cylinder, cone and torus displays a solid dialog (see page 45) that has two tabs. Double-clicking a spring will display the Spring dialog (see page 49) that has three tabs.

  The icon is different if it has features.

   = primitive without features. This is the icon of the solid (block, sphere, cylinder, cone, torus or spring)

   = primitive with features. This is the icon of the whole solid.

  If the solid has features, the primitive icon appears at the bottom of the tree for the solid and will be one of the following icons.

   = block

   = sphere

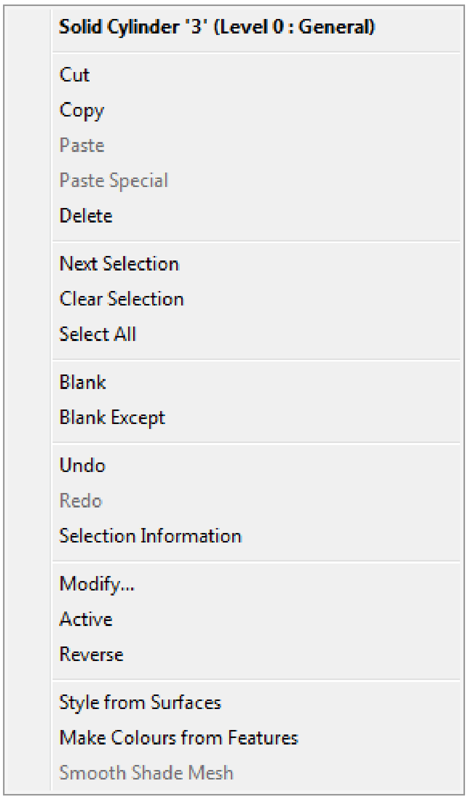
   = cylinder

   = cone

   = torus

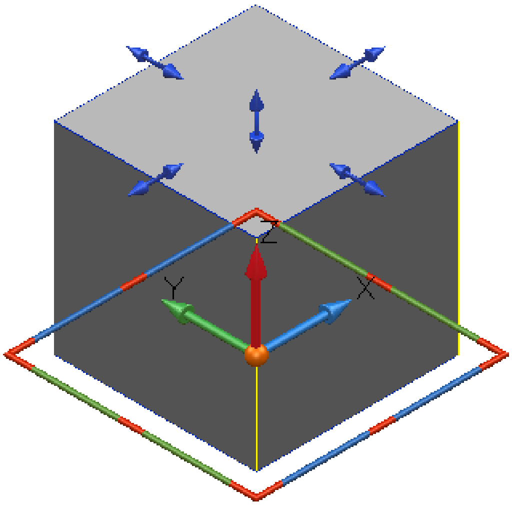
   = spring

- Right-click the primitive solid to display the edit options available for primitive solids. The top of the context menu shows the type of the solid, its name and the level on which it lies.



Graphically editing primitive solids

Select the primitive solid to display its graphical handles.



The workplane type handles are used to:

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

The blue handles are used to edit the dimensions of the primitive solid.

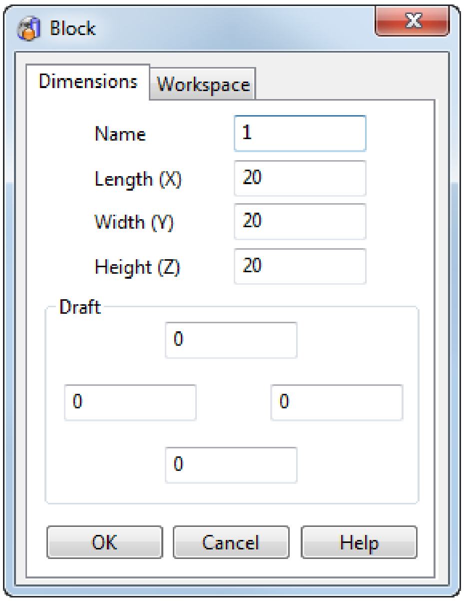
The unique graphical handles for each primitive solid are exactly the same as for primitive surfaces. The drag handles of primitive solids snap to other geometry in the model.

Primitive solid dialog

The title on the dialog reflects the primitive that you are editing.

To edit the primitive solid:

1 Double-click on the primitive you want to edit.

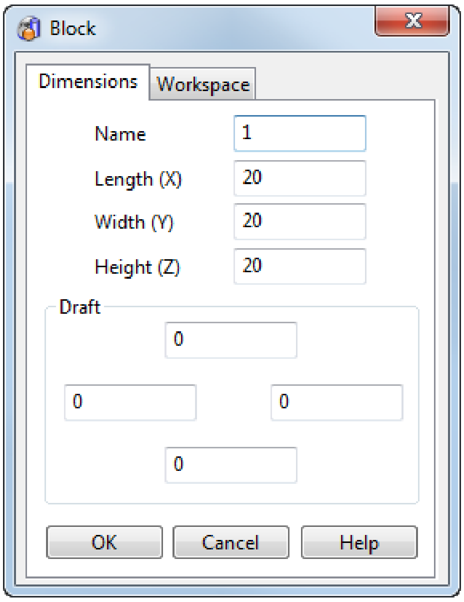


2 Use the two pages of the dialog to define the changes you want to make.

- Dimensions (see page 46)
- Workspace (see page 48)

Primitive Solid dialog - Dimensions

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

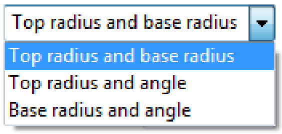


Name - This is the name of the selected primitive. You can edit the name.

Specify the following dimensions for the selected primitive:

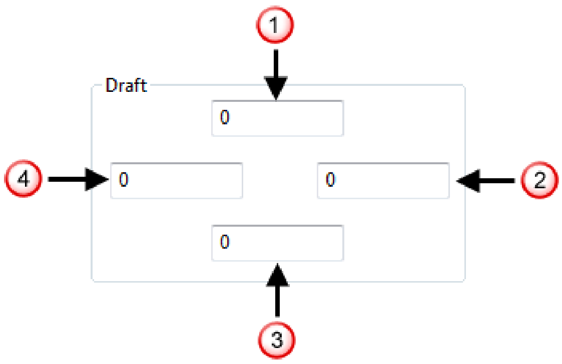
- **Block -** Specify the **Length (X)**, **Width (Y)**, **Height (Z).** You can also apply draft angles to four sides of a block.

- **Cylinder** - **Radius** or **Diameter** is selected from the drop-down list. You can also specify the **Length**.
- **Cone** - 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**.
- **Spring** - Specify the **Height**, **Pitch**, **Turns**, **Top Radius** and **Base Radius**

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



1 - Top

2 - Right

3 - Bottom

4 - Left

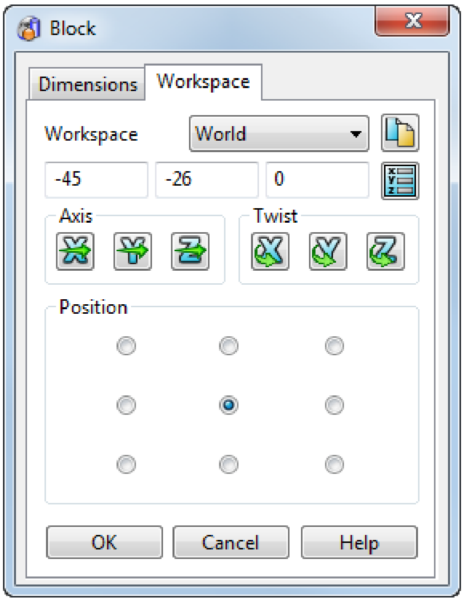
You can also define the draft angles by using the drag handles that are displayed in the centre of each top edge of the block.

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

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

Primitive Solid dialog - Workspace

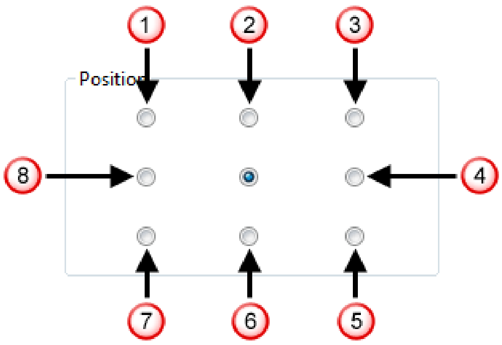
Use the **Workspace** page of the dialog to specify the primitive.



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 coordinates or click to display the Position dialog.
- Change the direction of its **Axis.**
- **Twist** it about its axis.

Position - Select an 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.



1 Top left corner

2 Middle of top edge

3 Top right corner

4 Middle of right edge

5 Bottom right corner

6 Middle of bottom edge

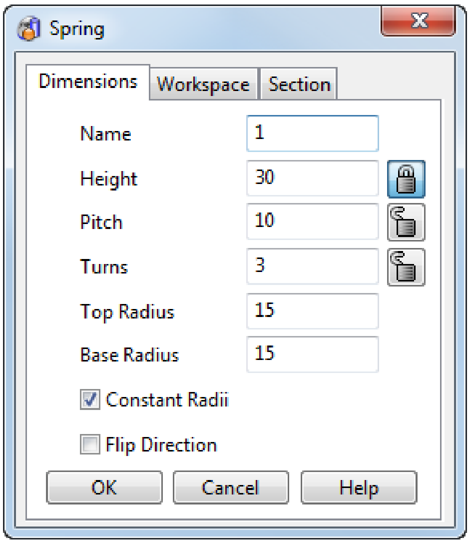
7 Bottom left corner

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:

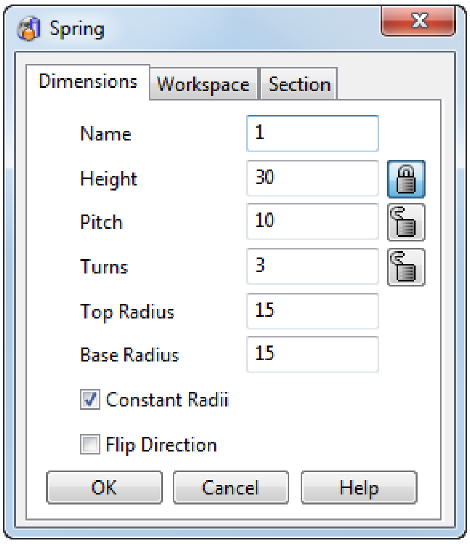


2 Click on the tabs to define the spring:

- Dimensions (see page 50)
- Workspace (see page 51)
- Section (see page 51)

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 of the spring 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.

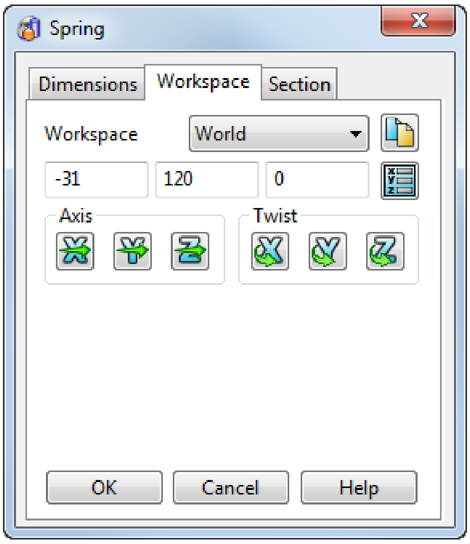
Base Radius - Enter radius for the bottom of the spring. The base radius can also be changed by dynamically dragging the arrow handle at the bottom of the spring.

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.

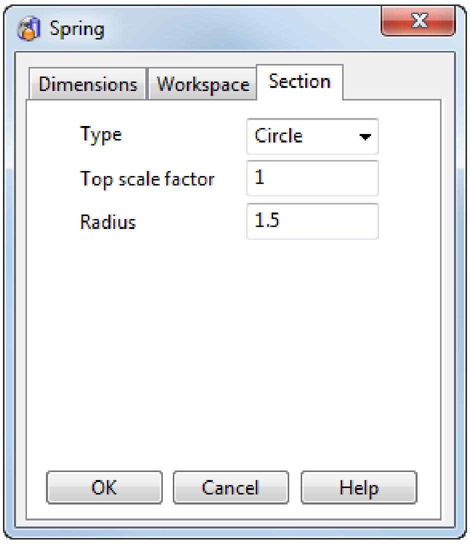


Workspace options - These options allow you to:

- 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.

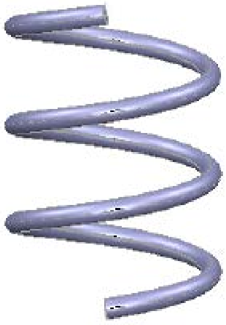
Spring dialog - Section

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

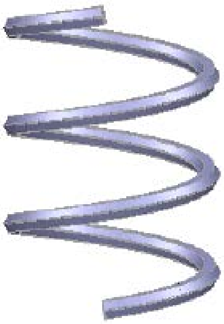


Type - Select one of the options:

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

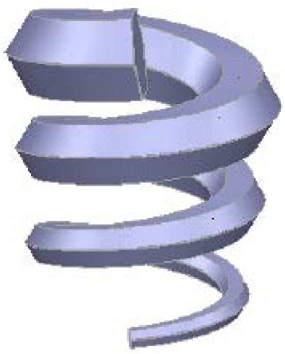


- **Polygon** - Select this option to create a polygon type spring:



*Additional options for polygon type springs are displayed. For further details see **Polygon Options** (see page 53).*

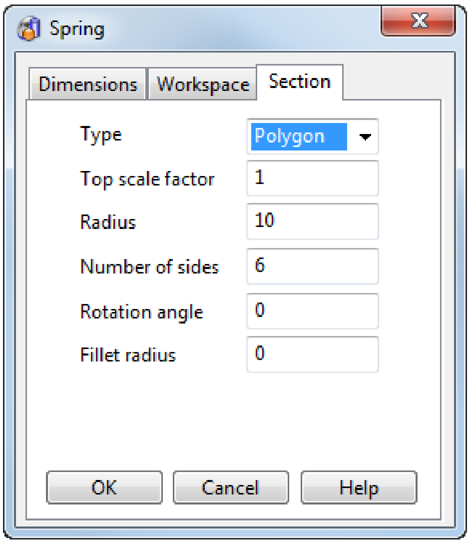
Top scale factor - Enter a factor by which the scale of the section radius increases along the height. The example below shows a polygon type solid spring with a section radius of 35 and a **scale factor** of 5.



Radius - Enter the radius for the section.

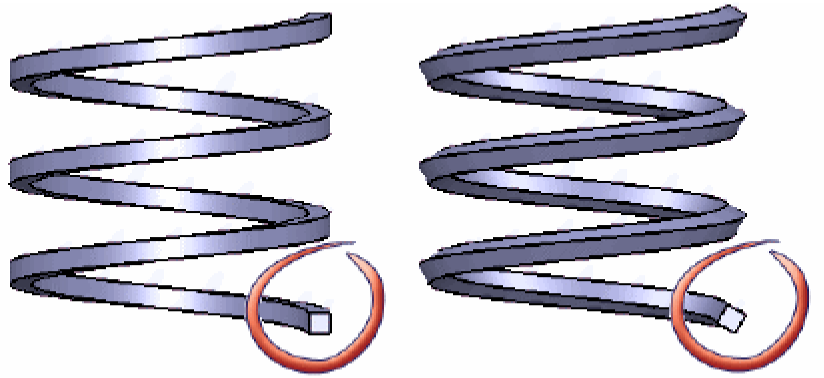
Spring dialog - Section - Polygon options

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



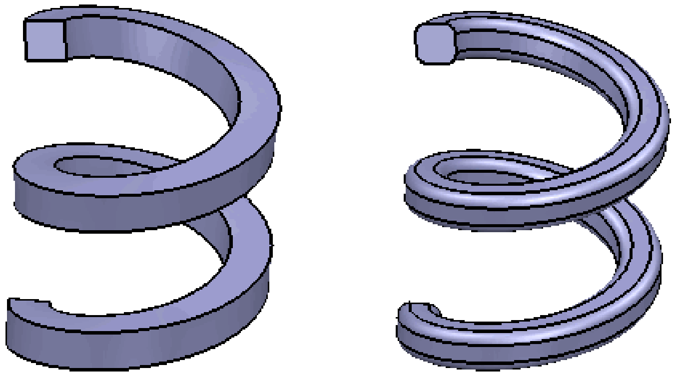
Number of sides - Specify the number of sides for the polygon type spring.

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 solids

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

This function is available for solid:

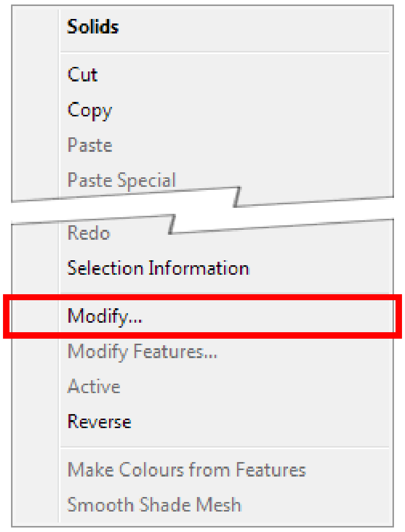
- blocks
- cylinders
- cones
- spheres
- tori
- springs

To edit multiple primitive solids

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

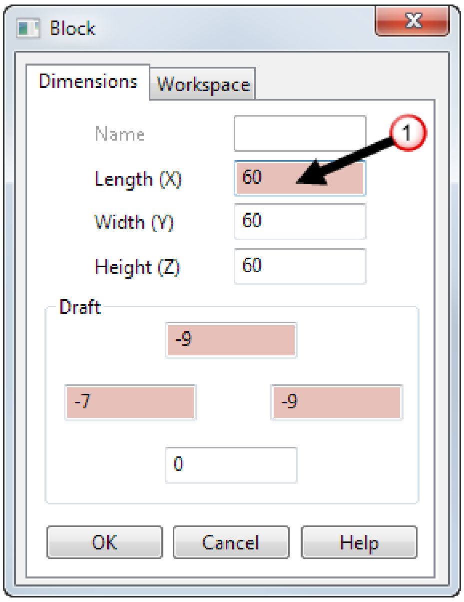
1 Select a number of primitive solids that you want to edit (for example blocks) with one of the following methods:
    - Press and hold the **Shift** key and click the blocks.
    - Press and hold the **Ctrl** key and click the blocks.
    - Click and drag the cursor over the blocks.

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

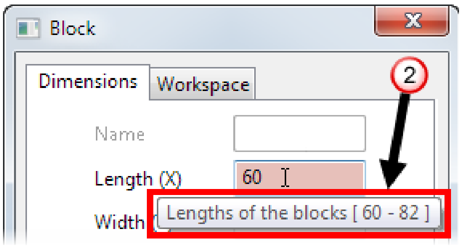


The dialog to edit the appropriate selected primitives is displayed (for example, the **Block** dialog). The dialog is similar to that of editing a single primitive of the same type, with the following differences:

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.

Other solids

The following sections contain information on other PowerSHAPE solids:

Creating an extruded solid (see page 56)

Editing an extruded solid (see page 59)

Creating a solid of revolution (see page 73)

Editing a solid of revolution (see page 74)

Creating a solid drive-curve from wireframe (see page 79)

Creating a solid core (see page 86)

Creating an extruded solid

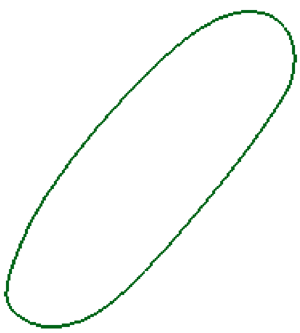
Solids can be created by extruding single or multiple wireframe objects.

If you intend to cut extruded solids into another solid, we recommend you use the **Cut** command instead of extruded solid.

Similarly, if you intend to create a boss by adding an extruded solid to an existing solid, we recommend you use the **Boss** command.

1 Select the wireframe object you want to extrude.

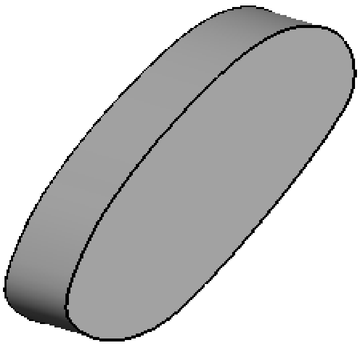
The example below shows a composite curve:



2 Click  (*Solid toolbar*).

If a single wireframe objected is selected, it is extruded to form a solid.

In our example, the solid shown below is created.



A solid is created by extruding the wireframe object. This is similar to creating an extruded surface except caps are added to the solid if the wireframe object is planar and closed.

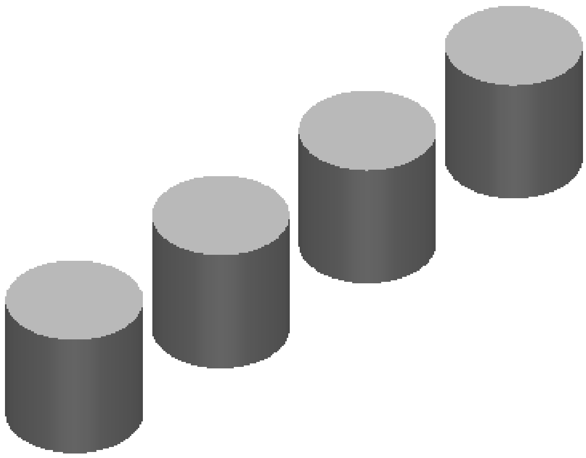
The solid is displayed in the solid feature tree.

If there is no active solid in your model, this solid becomes active. Otherwise it becomes an inactive solid.

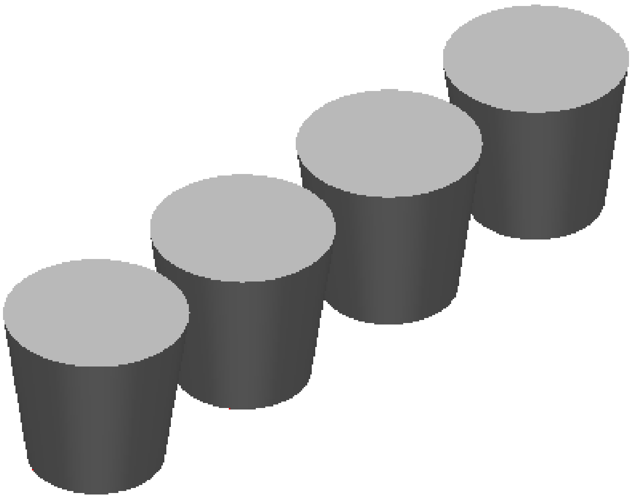
3 If multiple wireframe objects are selected, the Extrusion Dimension dialog (see page 58) is displayed where you set the dimensions of the extruded solids.

Enter the length and draft angle values for the extruded solids.

4 Click **Preview** to see the extruded solids.



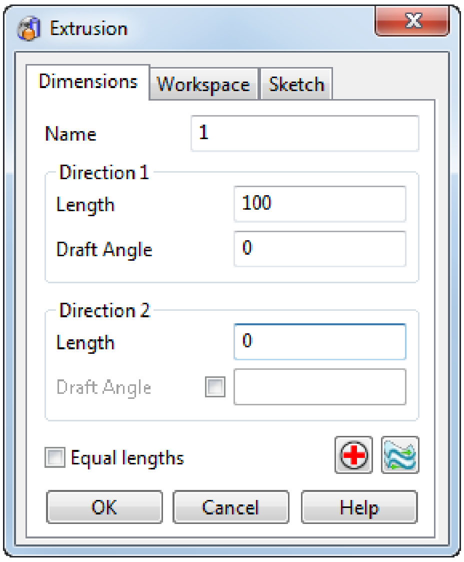
You can change the length and draft angle values and click **Preview** again to update the solids.



5 Click **OK** to accept the preview and close the dialog.

Extrusion Dimensions dialog

Use this dialog to set the dimensions of extruded solids from multiple wireframe objects.



Direction1 - Length - This is the length of the extrusion. The **Length** of the extrusion can be zero if the **Direction 2 Length** has a value other than zero.

Draft Angle - This is the draft angle between the base curve of the extruded solids and the curve at the other end. The angle is measured from the axis normal to the principal plane or the plane of best fit of the curve.

Direction 2 - Length - The length of the extrusion below the active plane.

OK - Click to accept the preview and close the dialog.

Editing an extruded solid

Edit a selected extruded solid in the following ways:

- Use its graphical handles to change the orientation of the extruded solid.
- Use the options on the **Solid Edit** toolbar (see page 24).
- Use one of the options on the **Edit** menu.

  Edit > Modify displays the **Extrusion dialog.**

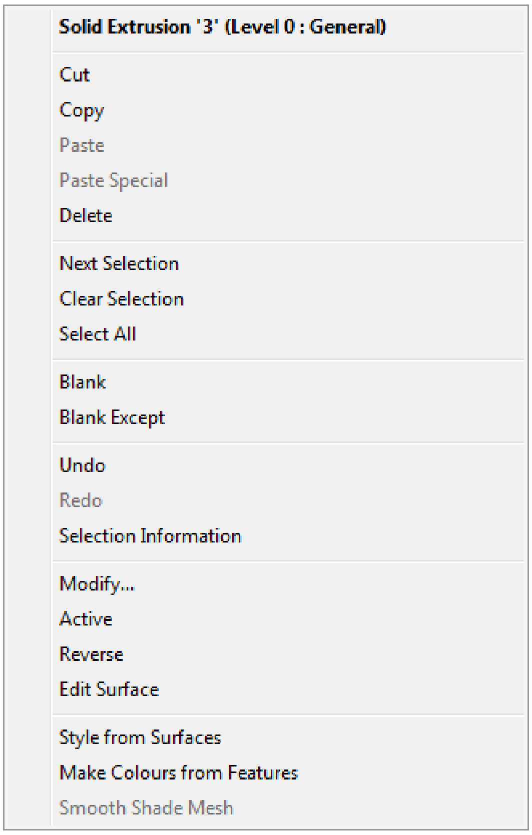
  Edit > Convert > Solids to surfaces converts the solid into surfaces. You can then edit the surfaces.
- Double-click the extruded solid to display the **Extrusion dialog**.
- Double-click the icon of the extruded solid  in the tree to display its dialog and graphical handles.

  The icon is different if it has features.

   = solid with features. This is the icon of the solid.

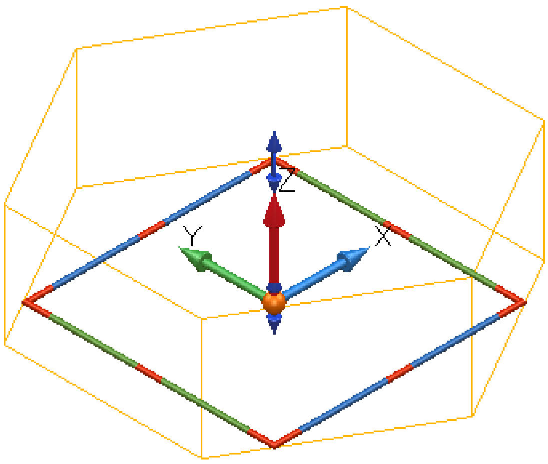
   = solid without features. This icon appears as a feature at the bottom of the tree for the solid.

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



Graphically editing an extruded solid

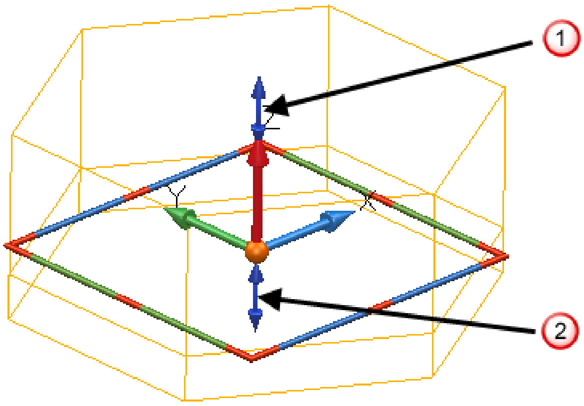
1 Select the extruded solid to display its graphical handles.



Use these workplane type handles to:

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

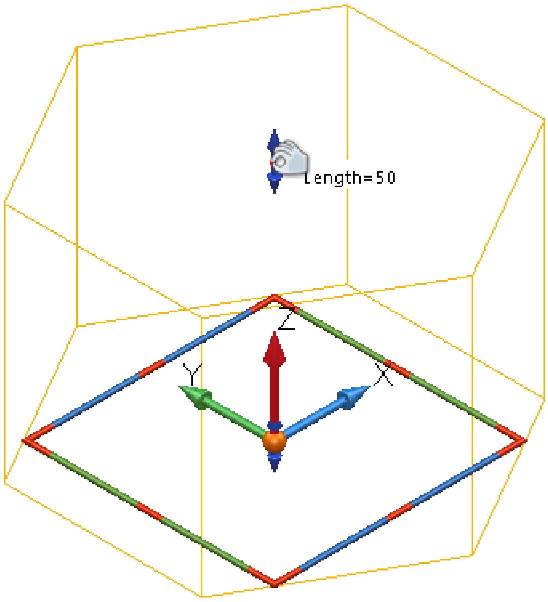
Two other handles are displayed for editing the height of the extruded solid. These are the **Length** 1 and **Negative Length** 2 parameters. **Negative length** is set to zero by default, so is not visible in the following models.



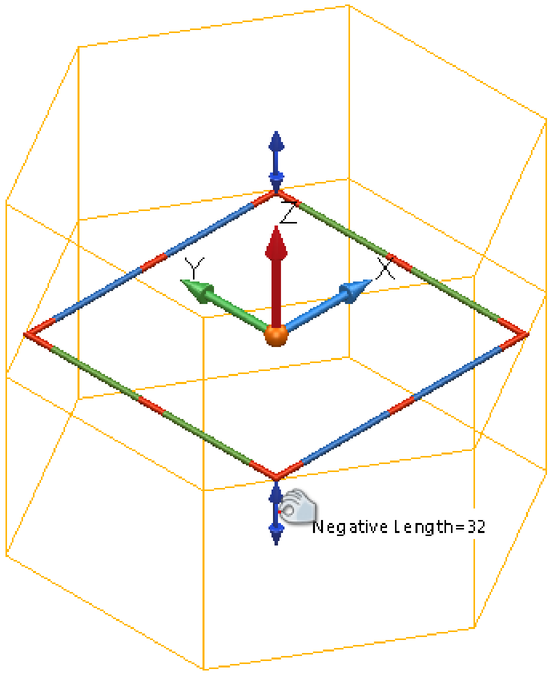
2 To edit the height, select the handle and drag upwards or downwards until you reach the required height.

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

Dragging the handle upwards increases the height of the extruded solid. How the dimension value increases and decreases depends upon the zoom factor. *Zoom in* to work with small increments and *zoom out* to work with large increments.



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



Extrusion dialog

Use the three tabs to edit the extrusion.

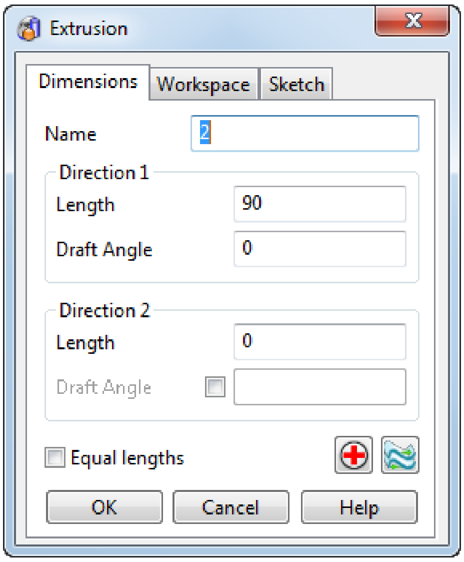
Extrusion dialog - Dimensions (see page 62)

Extrusion dialog - Workspace (see page 65)

Extrusion dialog - Sketch (see page 66)

Extrusion dialog - Dimensions

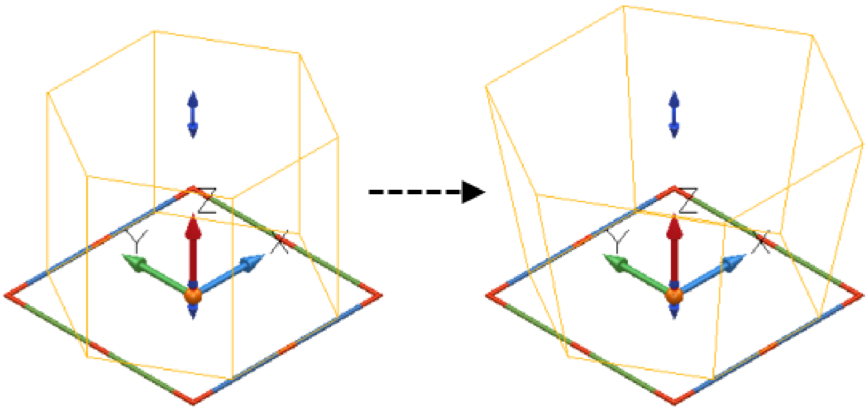
Use this dialog to edit the dimensions of an extruded solid.



1 Enter a **Name** for the extrusion (if required). Alternatively, use the default **Name** allocated by PowerSHAPE.

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

3 Enter the **Draft Angle** (*Direction 1*). This is the draft angle between the base curve of the extruded surface and the curve at the other end. The angle is measured from the axis normal to the principal plane.



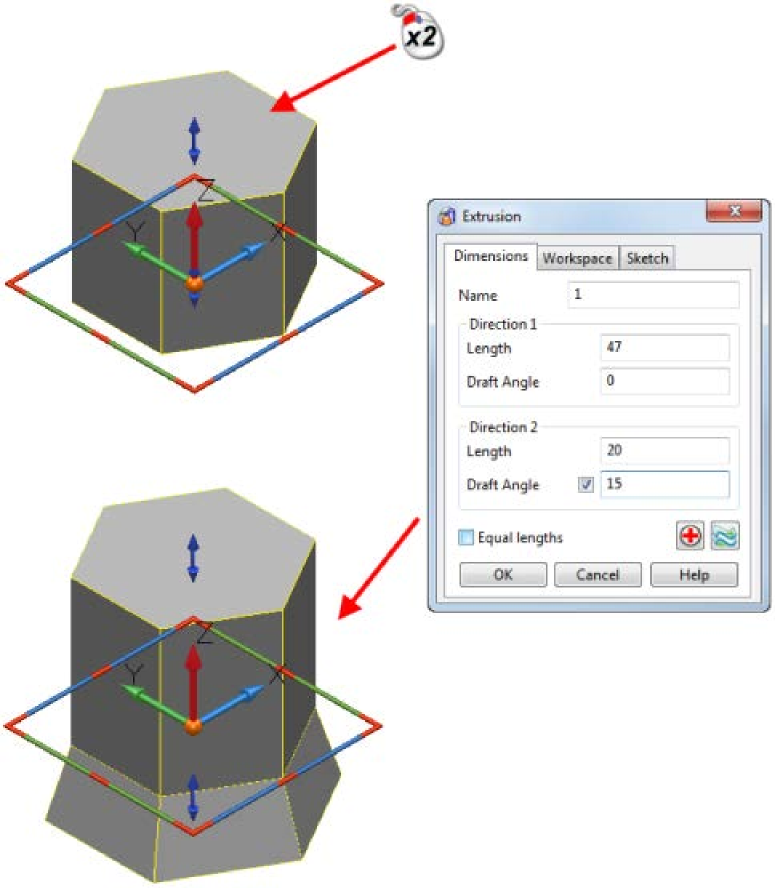
4 Enter the **Length** of the extrusion (*Direction 2*).

5 Enter the **Draft Angle** (*Direction 2*).

The model below uses the following dimensions:

- Direction 1: **Length** = 47, **Draft Angle** = 0.
- Direction 2: **Length** = 20, **Draft Angle** = 15

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



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

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

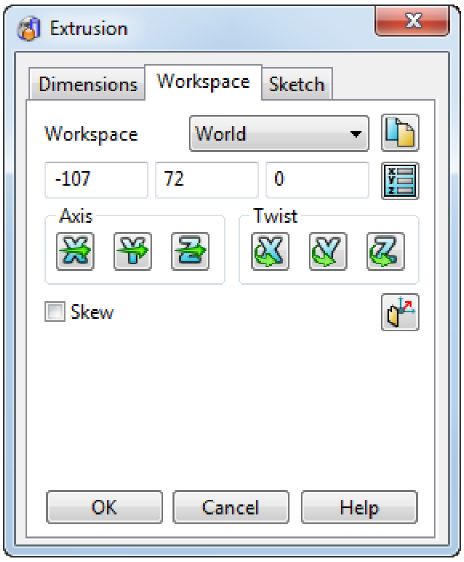
8 Click  to **Reverse direction**. This reverses the solid so that the outside of the solid becomes the inside, and the inside becomes the outside.

9 Select one of the following:

- **OK** to save the edits carried out on the extruded surface and removes the dialog from the screen.
- **Cancel** to remove the dialog from the screen and discards 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.

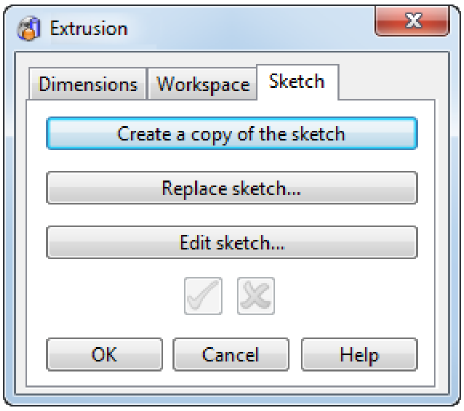
Enter the **X Y Z** coordinates or click the **Position** button to open the **Position** dialog where you can use position entry tools.

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.

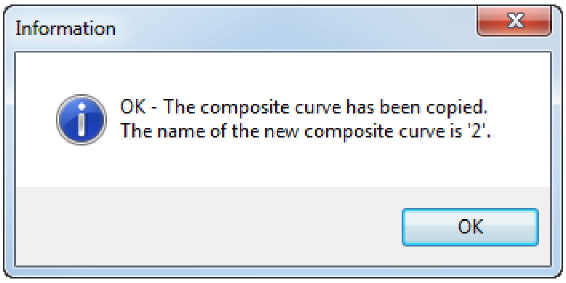
Extrusion dialog - Sketch

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



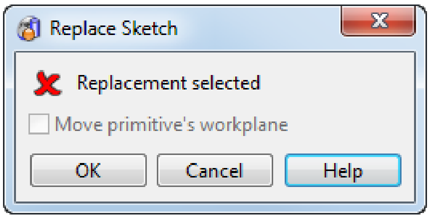
Create a copy of the sketch

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 (see page 67). Use the dialog to select the curve to replace.



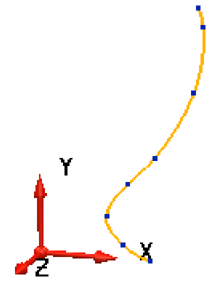
Edit sketch

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

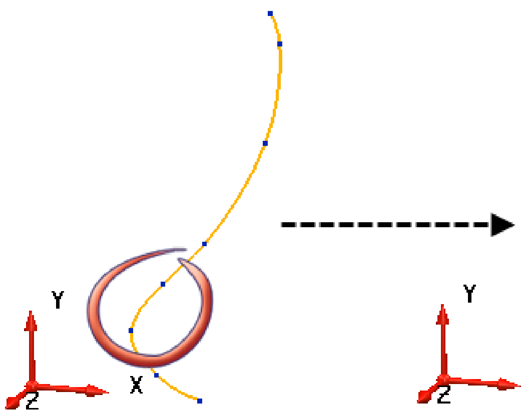
1 Click **Edit sketch**. The following is displayed:

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

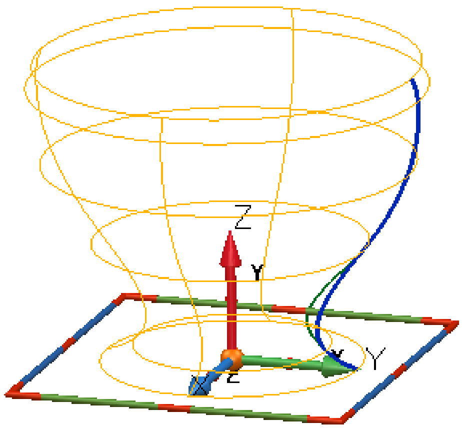
- is displayed on the dialog.



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

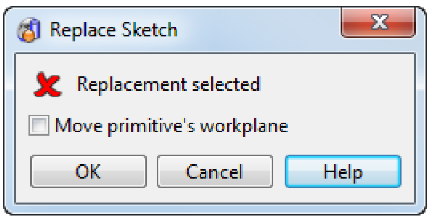


3 Once you have finished editing the wireframe, click to accept the edit or to cancel the edit.



Replace Sketch dialog

Use this dialog to replace wireframe in an extruded solid.



Replacement selected - Select the replacement wireframe. The will be replaced with a .

Move primitive's workplane - Select this to move the primitive's workplane to the centre of the replacement wireframe.

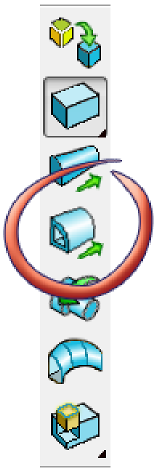
OK - The original composite curve is replaced with the selected wireframe.

Cancel - The **Replace Sketch** dialog is closed without replacing the curve.

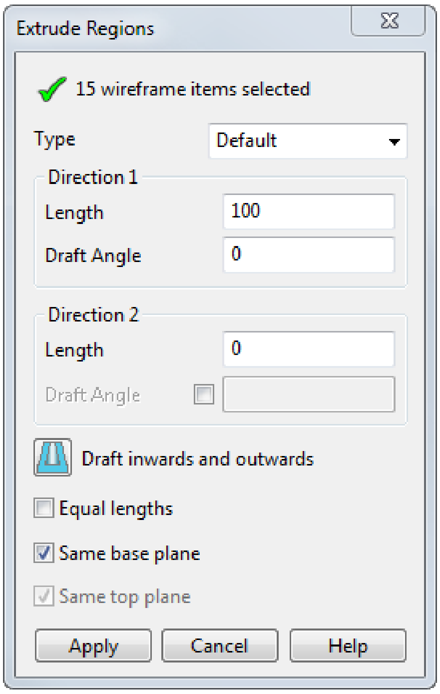
Extrude regions

Use the new **Extrude Regions** button to extrude multiple nested regions in a single operation using the **Extrude Regions** dialog.

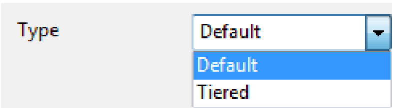
1 Click (*Solid creation toolbar*).



The **Extrude Regions** dialog is displayed.

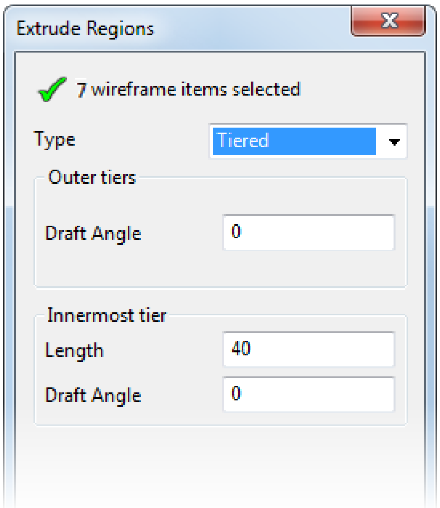


2 Select the type of extrusion from the **Type** drop-down list.



Default creates extrusions from the wireframe items (see page 71).

Tiered uses the height of the wireframe items to define the height of the extrusions. A modified dialog box is displayed.

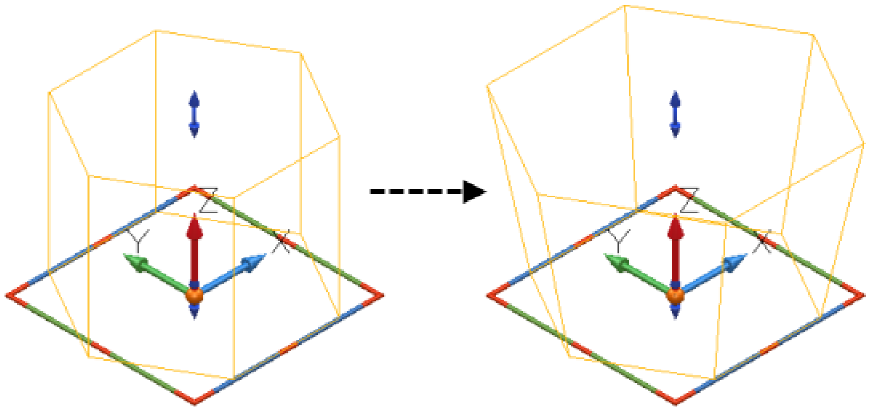


The **Innermost tier** can be:

- positive, to define a boss-like tier.
- negative, to define a cut-like tier.
- zero, if you want a neutral tier.

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

4 Enter the **Draft Angle** (*Direction 1*). This is the draft angle between the base curve of the extruded surface and the curve at the other end. The angle is measured from the axis normal to the principal plane.



5 Enter the **Length** of the extrusion (*Direction 2*).

6 Enter the **Draft Angle** (*Direction 2*).

7 Click  or  to specify if the draft angle is to go in or out. If selected, in/out draft is added to each region alternately.

8 Select **Equal lengths** to set the same **Length** for **Direction 1** and **Direction 2**. If this option is selcted, updating either length will also cause the other length to update.

9 Define the extrusion plane:

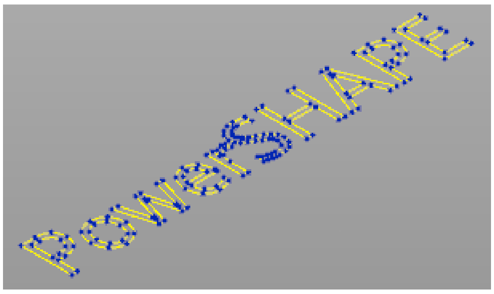
- If **Same base plane** is selected, the inner and outer wireframe are extruded from the same plane. This option is selected by default.
- If **Same top plane** is selected, each region of the solid is extruded up to the height of the outer wireframe.

10 Click **Apply** to extrude the regions

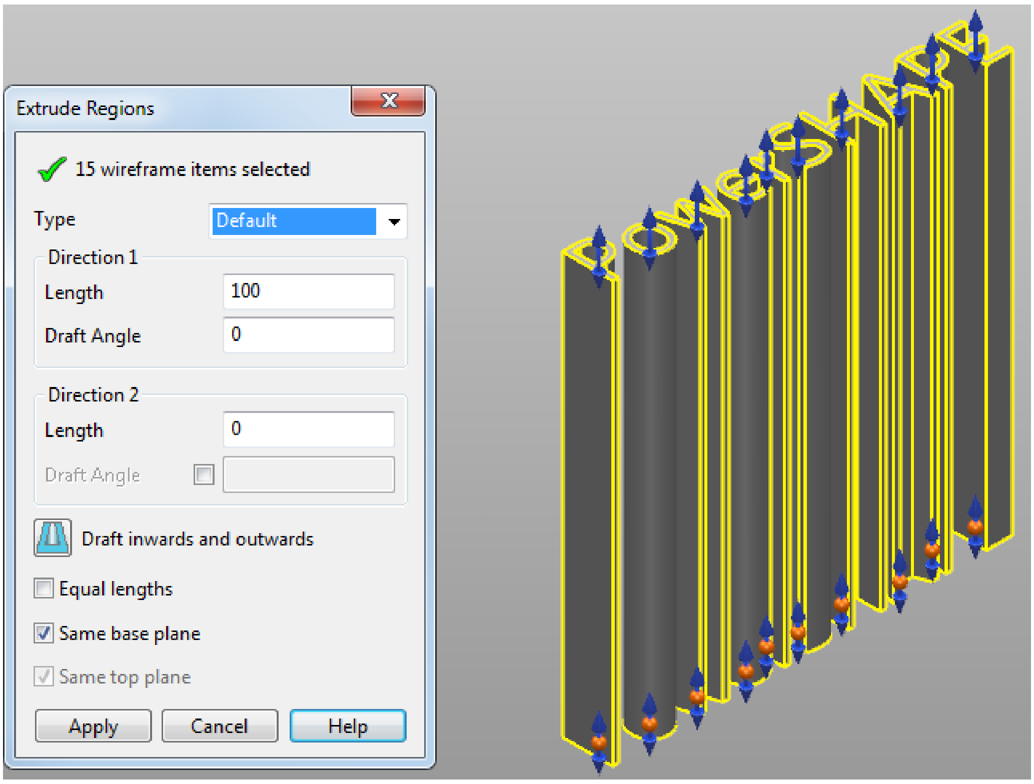
Example — Extruding regions

Extruding regions is useful when you need to extrude a hollowed out shape.

1 Select the wireframe.

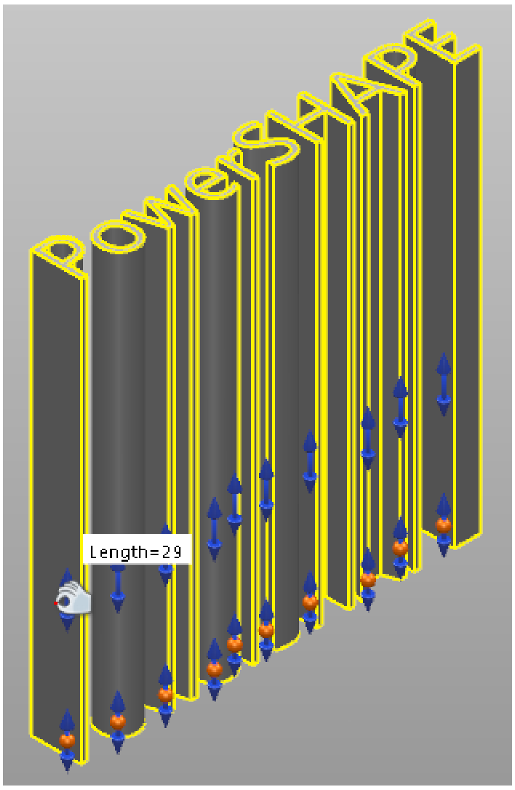


2 Click  *(Solid creation toolbar)* to display the **Extude Regions** dialog and a preview of the extrusion.

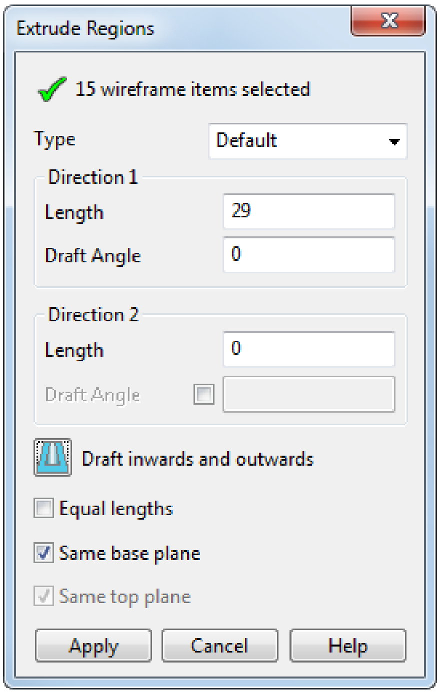


3 Edit the extruded regions by:

- using the handles on the model to adjust the extrusion. Any changes you make will update the values in the Extrude Regions dialog.



- entering a new value in the **Extrude Regions** dialog and clicking **Apply**.

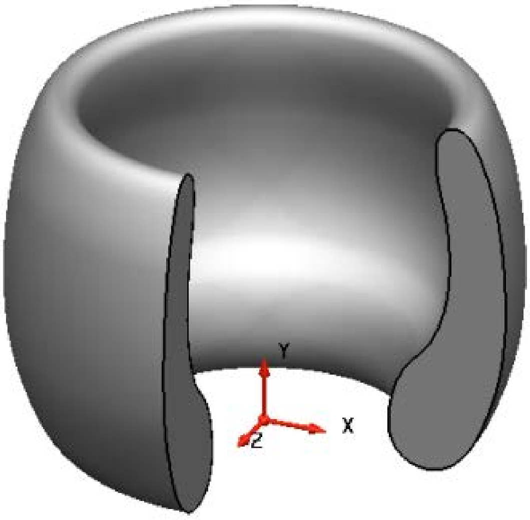


How do I create a solid of revolution ?

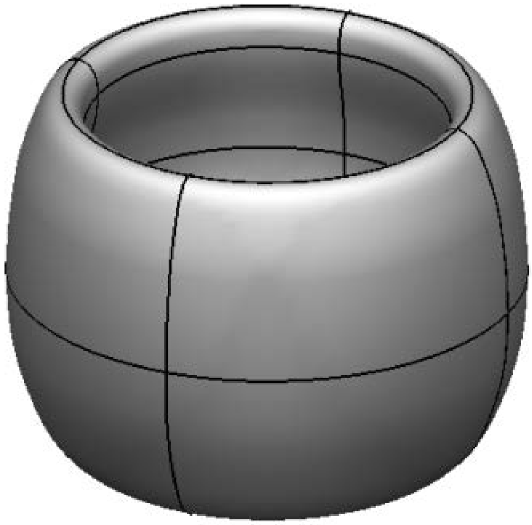
A solid of revolution is created from a wireframe object which is rotated around one of the axes of the current workspace. Create a workplane to ensure that the solid of revolution is created around the correct axis.

The solid of revolution can be either open or closed:

- In an open solid of revolution, the wireframe object is rotated around the axis by an angle of 270° 1.
- In a closed solid of revolution, the wireframe object is rotated around the axis by an angle of 360° 2.

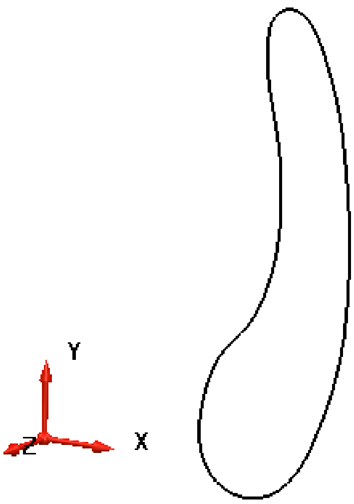


1


2

Once the solid is created, you can edit the angle of the open solid of revolution.

1 Select a wireframe object, for example :

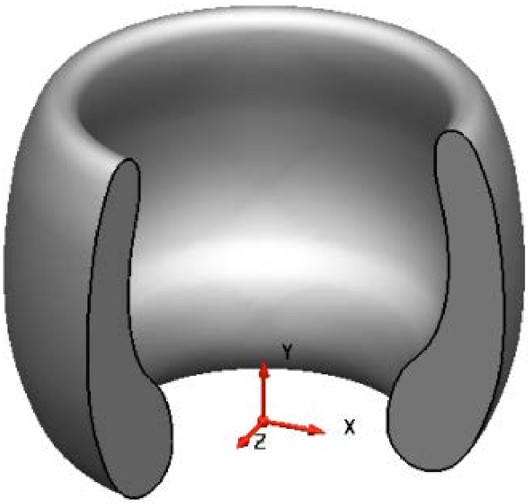


2 Select the **Principal plane** button (*Status bar*) of the axis you want to rotate around. In this case, select the **Principal plane** button with **Y** on it.  to rotate our curve around the Y axis,

3 Click  (*Solid toolbar*) to display the solid. The solid appears in the solid feature tree as a solid.

4 Create an open solid by:

- dragging the handles.
- double-clicking the solid to display the **Revolution** dialog (see page 76). Enter a new **Angle**.



For an open solid of revolution, caps are only added to the ends if the wireframe object is planar and closed.

Editing a solid of revolution

Use one of the following techniques to edit a solid of revolution:

- Select the solid of revolution and use its graphical handles to change the orientation of the solid of revolution.
- Select the solid of revolution and use one of the operations from the **Edit** menu.

  Edit > Modify displays the Revolution dialog (see page 76).

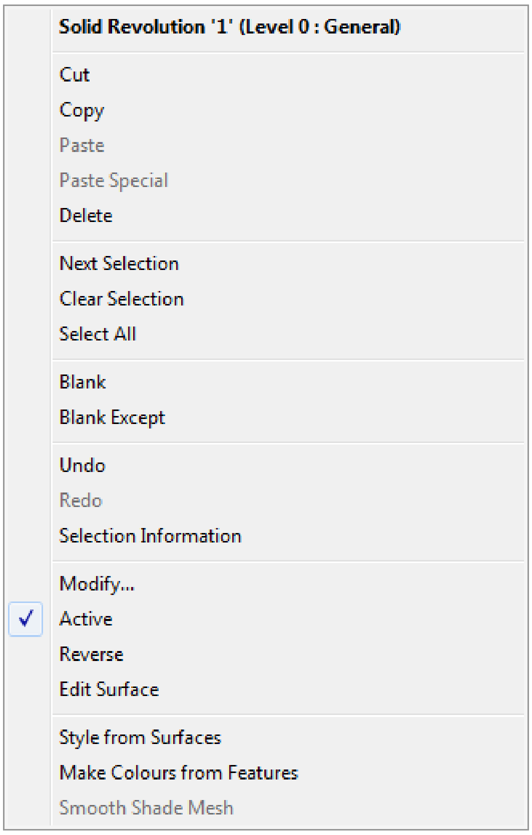
  Edit > Convert > Solids to surfaces converts the solid into surfaces for editing.
- Double-click the solid of revolution to display the Revolution dialog (see page 76).
- Use the options on the **Solid Edit** toolbar (see page 24).
- Double-click the icon of the solid of revolution in the tree to display its dialog and graphical handles.

The icon is different if it has features.

 = solid with features. This is the icon of the solid.

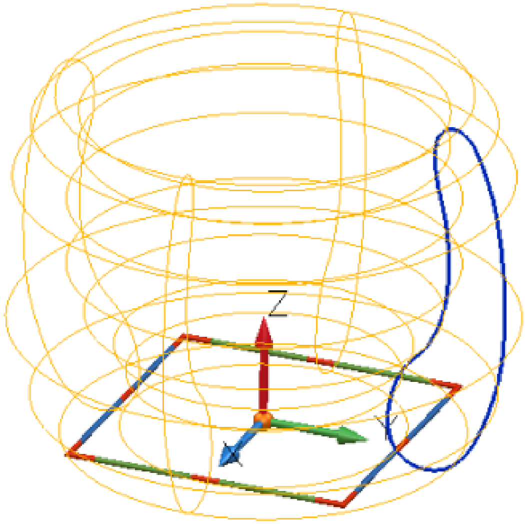
 = solid without features. This icon appears as a feature at the bottom of the tree for the solid.

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



Graphically editing a solid of revolution

Select the solid of revolution to display its graphical handles.



These workplane handles are used to:

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

Revolution dialog

Use the three tabs on this dialog to edit the solid revolution

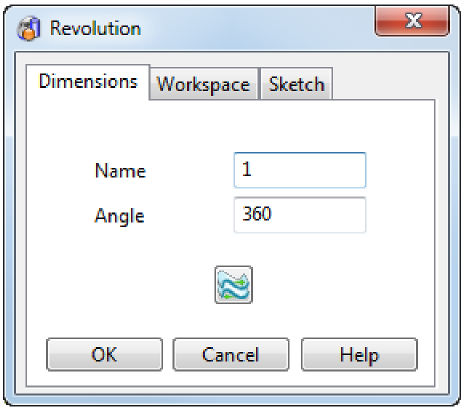
Revolution dialog - Workspace (see page 77)

Revolution dialog - Sketch (see page 78)

Revolution dialog - Dimensions (see page 76)

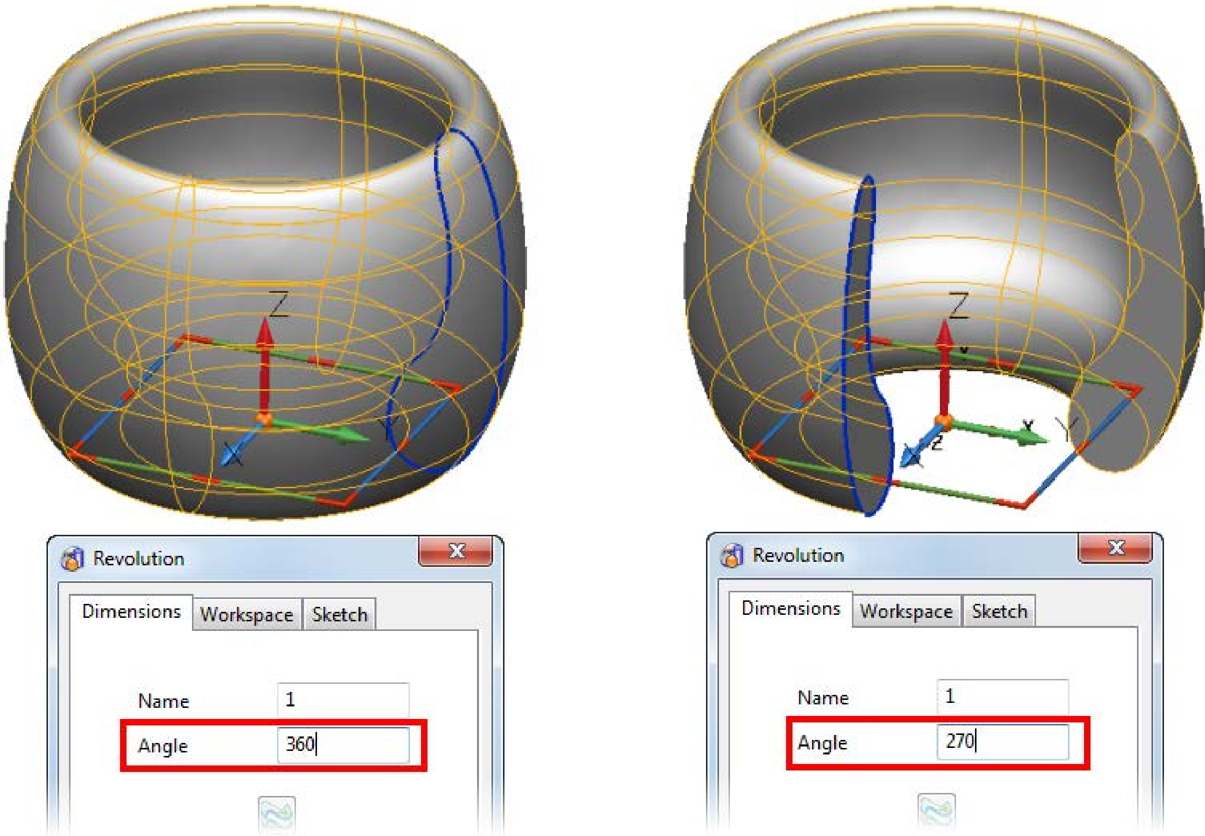
Revolution dialog - Dimensions

Use this dialog to edit a solid of revolution.



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

Angle - This is the angle that the wireframe object is rotated around the axis. You can change this angle to make a closed solid of revolution into an open one.



 - Flips the orientation of the solid, such that the outside of the solid becomes the inside and the inside becomes the outside.

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

Cancel - Closes the dialog and discards any changes made.

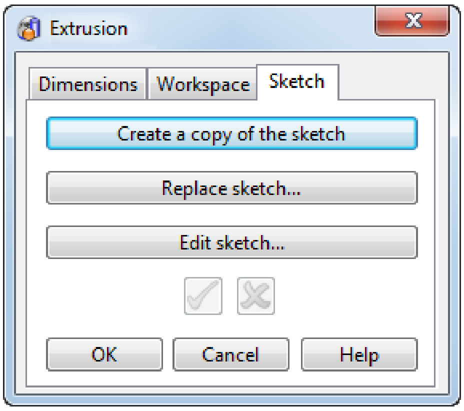
Revolution dialog - Workspace

Use this dialog to edit the workspace of a solid of revolution.

It is the same as Primitive Solid dialog - Workspace (see page 48).

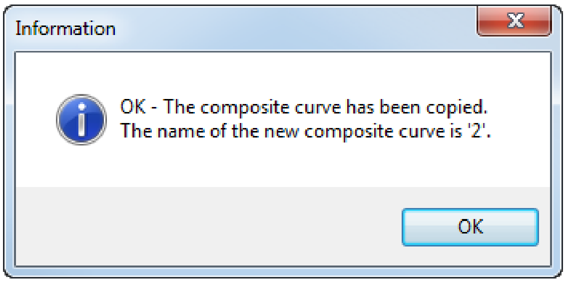
Revolution dialog - Sketch

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



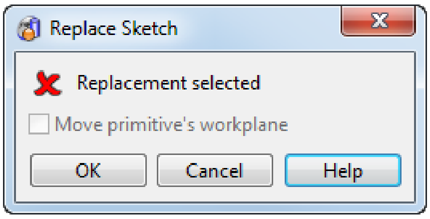
Create a copy of the sketch

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 (see page 67). Use the dialog to select the curve to replace.



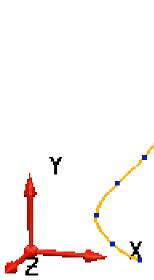
Edit sketch

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

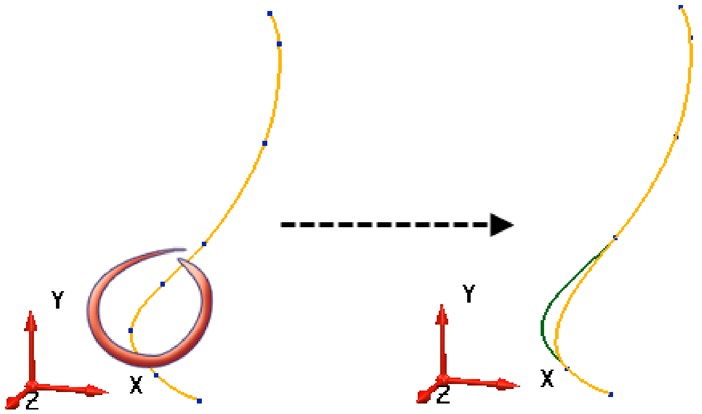
1 Click **Edit sketch**. The following is displayed:

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

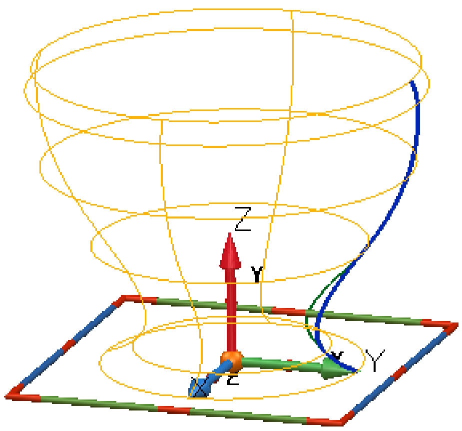
-   is displayed on the dialog.



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

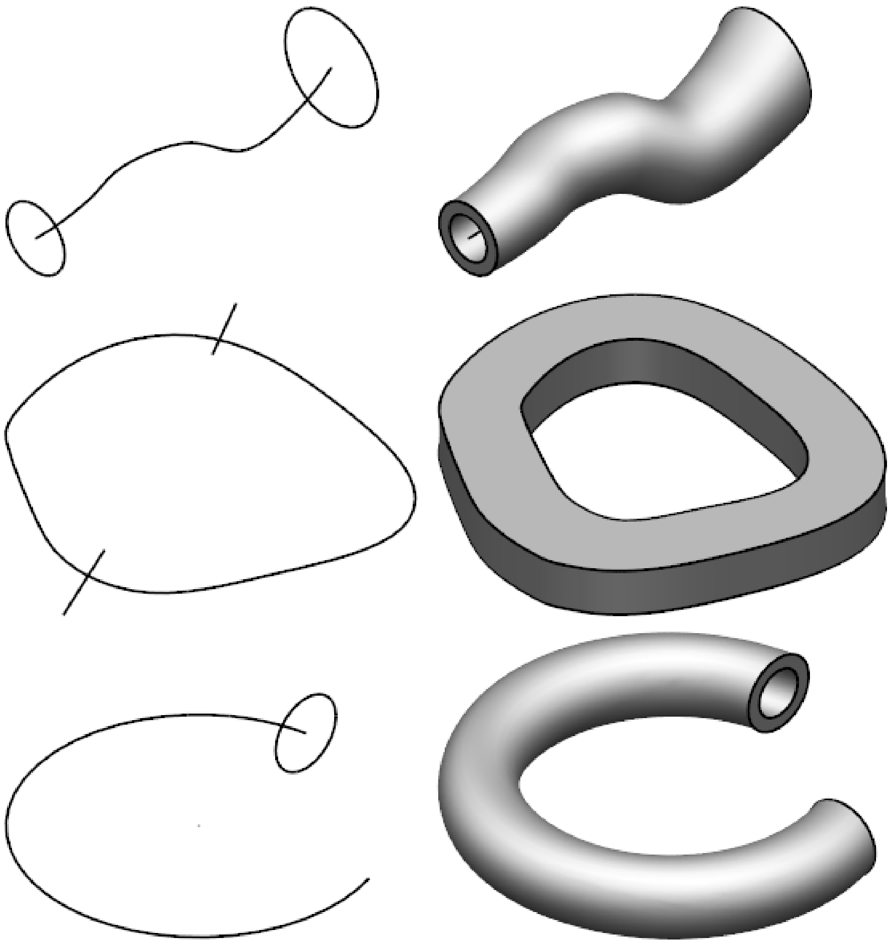


3 Once you have finished editing the wireframe, click  to accept the edit or  to cancel the edit.



Creating a Solid Drive Curve from wireframe

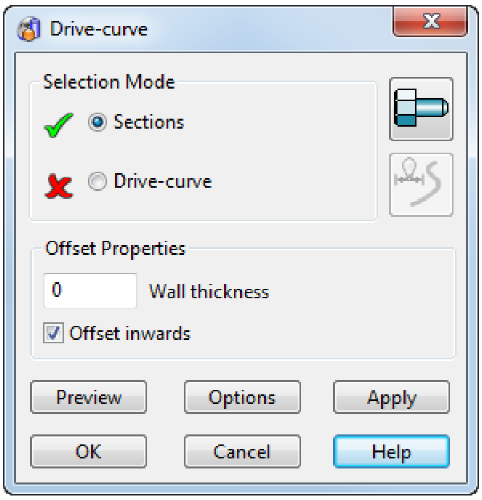
You can create a drive-curve solid from wireframe. This means that you can create elaborate solids from a very basic starting point. You can also create standard tubes and rods using this functionality.



1. Click  (*Solid toolbar*)
2. Complete the options on the **Drive-Curve** dialog.

Drive-Curve dialog

Use this dialog to create a solid from a drive-curve and sections.



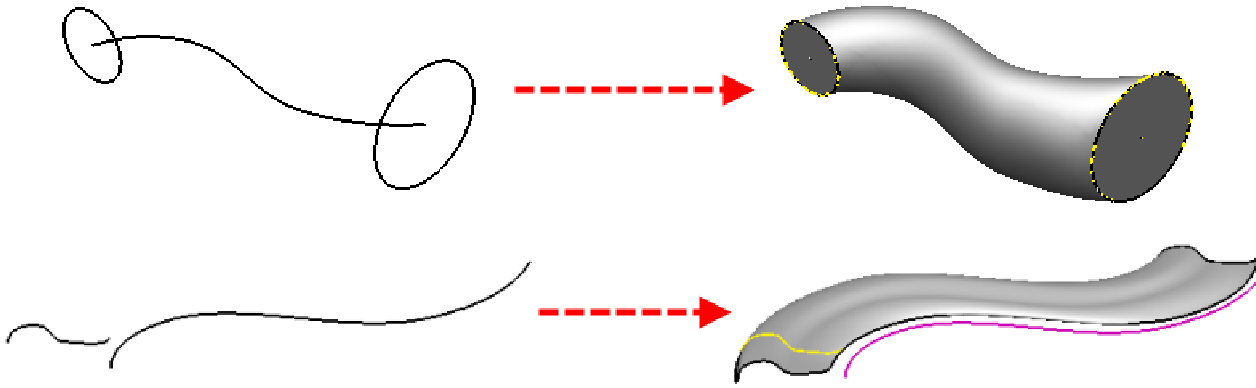
Selection Mode

Sections - Select the section(s) to use to create the solid. The ✗ changes to ✓. The section does not have to be planar.

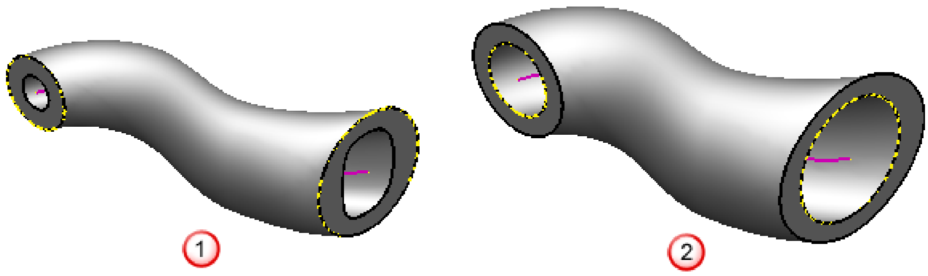
Drive-Curve - Select the **Drive-Curve** option and then select the drive curve to use to create the solid. The ✗ changes to ✓.

Offset Properties

Wall Thickness - A thickness of zero will result in solids like the ones below:



Offset Inwards - Non-zero wall thickness can be inwards 1 or outwards 2 from the section.



Preview - Displays the solid created using the current selection of objects and the default options. You may change your selection until you are satisfied with the previewed solid. You can change the default options by clicking the **Options** button.

Options - This displays the **Drive-Curve options** dialog, for changing the default options. The options on this dialog are the same as on the Drive-curve surface options dialog.

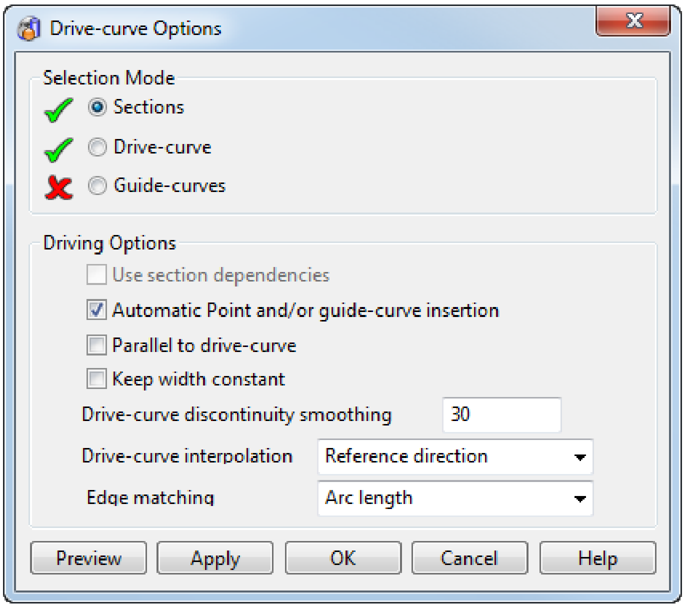
Apply - Saves the solid. The dialog remains on the screen for you to select more objects and continue creating solids.

OK - Saves the solid created and closes the dialog.

Cancel - Closes the dialog and discards any changes made.

Solid Drive-Curve options

Use this dialog to change the default options used to create the solid.



Selection mode - Select objects to define your solid. 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 drive-curve.

Sections - Select the section. The ✗ changes to ✓. You can also change the selection.

Drive-Curve - Select this option and select a drive curve. You can select a line, arc, curve or composite curve from your model to define the drive-curve. The ✗ changes to ✓.

Guide-Curves... - Sometimes the wrong shape is created because the wrong points on successive curves are joined. You can define the points that should be linked by defining a wireframe object between them. In the new solid, these points will be joined by a smooth curve, that 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 solid creation command.

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

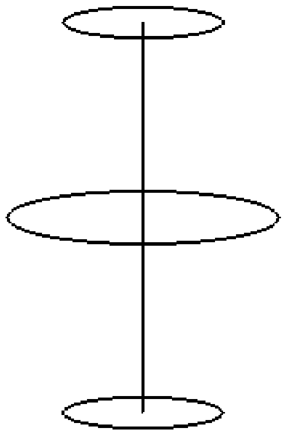
Driving Options

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 option is selected by default.

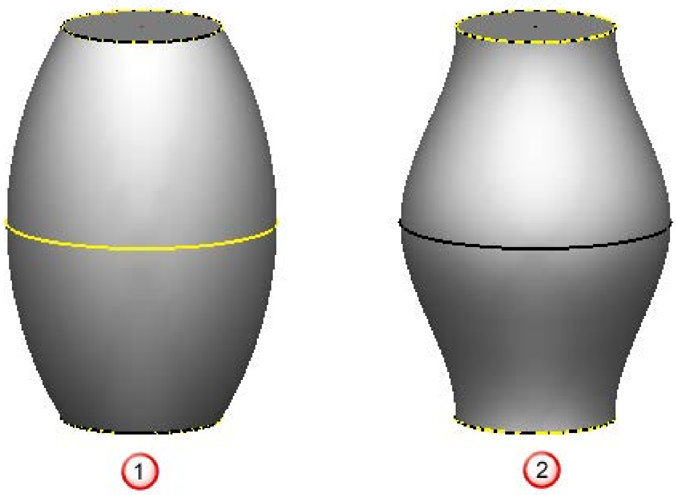
If selected, any previously selected guide-curve will be deselected. The option will be deselected if you manually select a guide-curve.

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

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

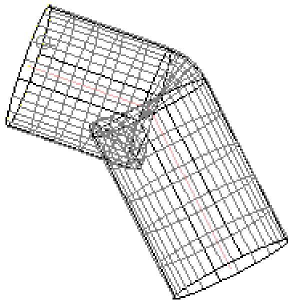


The following show the effect of deselecting 1 and selecting 2 **Parallel to drive curve**.

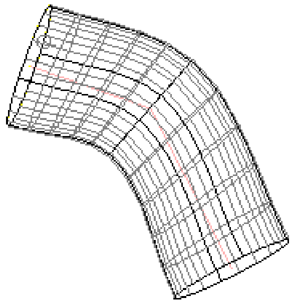


Keep width constant - If *ON*, the thickness of the solid is kept throughout, by adding in extra points for the complex region of the drive-curve.

Drive Curve Discontinuity Smoothing Angle – If the angle of discontinuity is less than the value given here, all tangent discontinuities are smoothed when generating the drive-curve solid. The model below shows the result if the discontinuity of the drive-curve is greater than the **Drive Curve Discontinuity Smoothing Angle**.



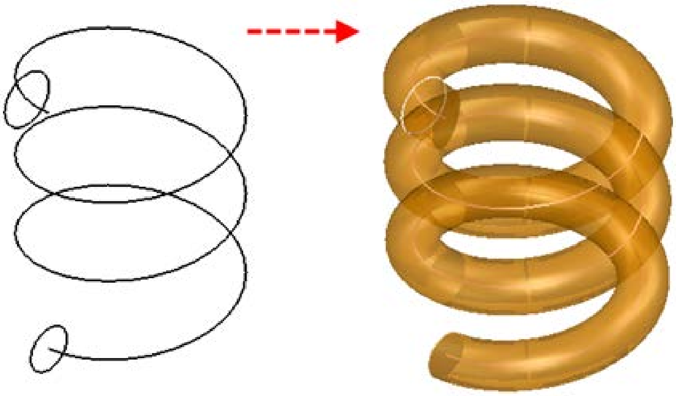
If you increase the **Drive-Curve Discontinuity Smoothing Angle,** the tangent discontinuities are smoothed.



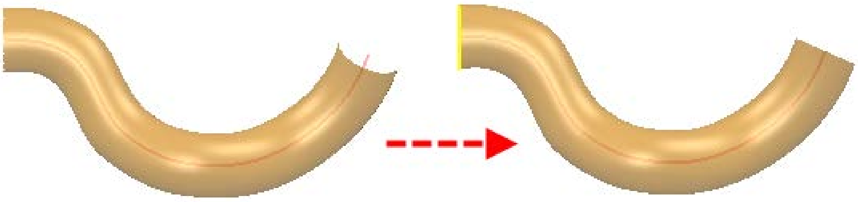
Drive-Curve interpolation - This controls how laterals are orientated relative to the drive-curve.

- **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.

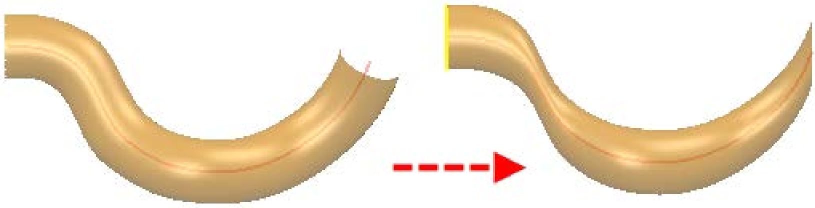
- **Helical** - Use this option to create helical sweep solids. The solid below is created with **Helical** Interpolation.



- **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.

Preview - Displays the solid 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 solid.

Apply - Saves the solid. The dialog remains open for you to select more wireframe objects and continue creating solids.

OK - Saves the solid and closes the dialog.

Cancel - Closes the dialog and discards any changes made.

Creating a solid core

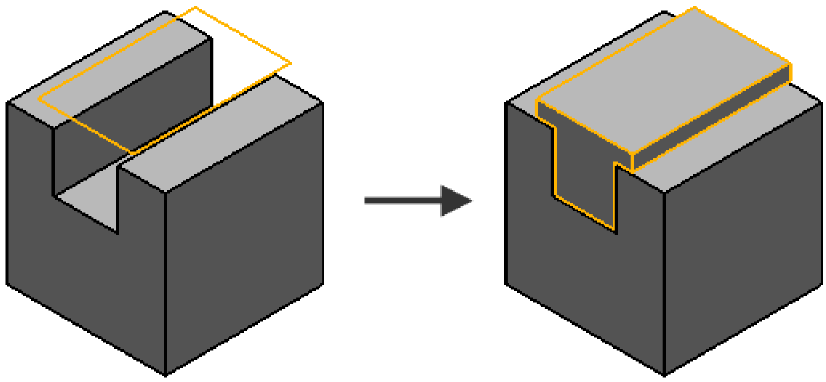
Use this option to produce a male form to match a female cavity. This can be useful when modelling electrodes or when generating cores for casting.

You can use solid core functionality to create of the following:

- solid core using wireframe (see page 86).
- solid core from a selected face (see page 87).

Creating a solid core using wireframe

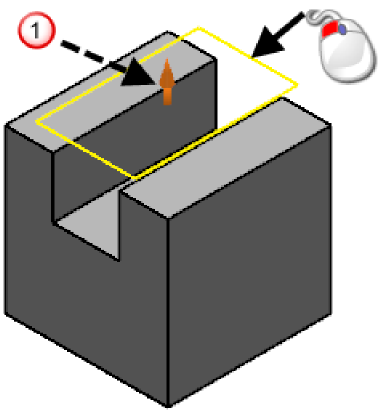
To create the solid core using wireframe, define the cross-section of the new solid and an offset. The core is automatically generated and fitted into the relevant feature on the model.



The cross-section of the new solid is defined by a closed and planar wireframe object.

1. Make sure that the solid is either active or selected.
2. Select a closed and planar wireframe object.

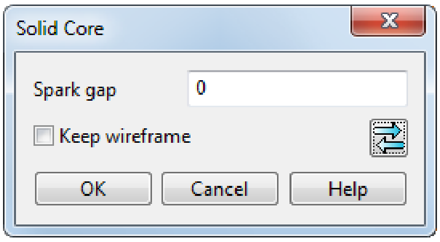
   The direction of extraction is shown by the instrumentation 1.



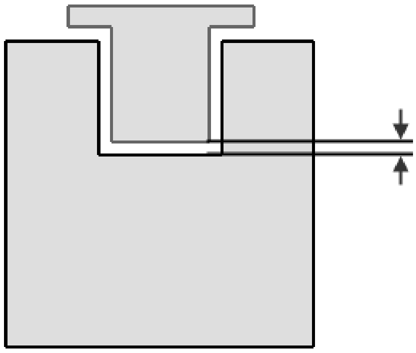
3. Click  (*Solid toolbar*).
4. Use the Solid Core dialog (see page 87) to create a solid core.

Solid Core dialog

This dialog is used to define a solid core (for example, for an electrode).



Spark gap — This is the distance between the core and the cavity. Usually, this is zero because any offsetting can be done more easily by using a negative thickness when machining.



If you specify an offset, make sure that the original solid is faultless because solids with faults will not offset correctly. Click **Solid Doctor**  (*Solid Edit toolbar*) to fix faults in the solid.

Keep wireframe — When selected, the original wireframe object is kept in the model.

 — Click this button to reverse the extrusion direction. This is the same as clicking on the wireframe instrumentation on the model.

OK — Creates the core and closes the dialog.

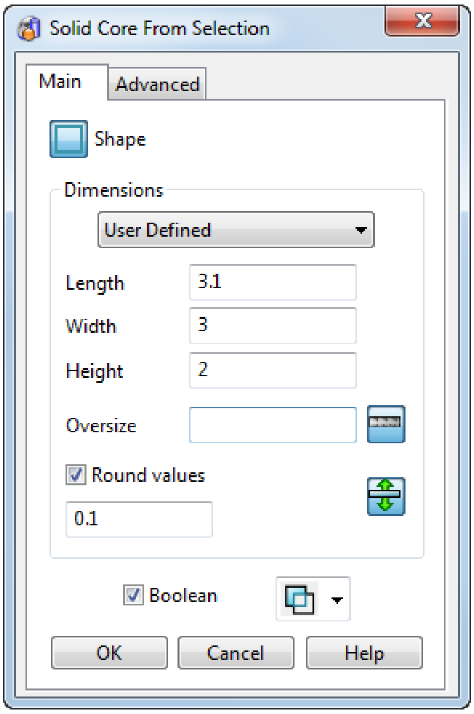
Cancel — Closes the dialog without creating the core.

Creating a solid core using a selection

To create a solid core from a selection:

1 Select the faces.

2 Click  to display the **Solid Core From Selection** dialog.



3 Use the options on the **Main** tab to specify the main settings for the core.

4 Use the options on the **Advanced** tab to specify additional settings.

*When the **Solid Core from Selection** dialog is displayed, you can change the face selection using the **Solid Edit** toolbar selection buttons and **Restore selection** (Views toolbar).*

Main tab

Use the settings on this tab to specify the main settings for the core.

1 Click the **Shape** button to select one of the following shapes:

-  If you select this option, you can define the length, width and height of the core.
-  If you select this option you can define the diameter and the height of the core.

2 Use the options in the **Dimensions** section to specify the dimensions of the core.

- Choose **User Defined** (default setting) to use set the dimensions of the core to the tightest values enclosing the selected items.

Alternatively, to use previously defined shapes, select an option from the drop-down list. The options on this list are defined in blanks.csv.

Information on blanks.csv is included in the Electrode section of the PowerSHAPE on-line help.

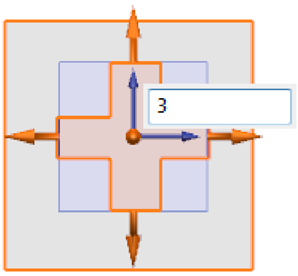
- Enter an **Oversize** value to increase the dimensions by the amount you enter. Use to toggle between specifying **Oversize** as a measurement or a percentage.

  Entering an oversize value of **0** reverts the dimensions back to the tightest values enclosing the selected items. This is the same as choosing **User Defined** in the drop-down list

- Select **Round values** to round the dimension values.

  For example, if the length of the core is **10.367** and **Round values** is selected and a rounding value of **0.5** is entered, the length will be rounded to **10.5**.

- Use the drag handles to change the size of the core. You can enter an exact value in the box that is displayed when you have dragged instrumentation when creating a **Solid core from selection**.

  You can define how use of the graphical drag handles is applied. If is displayed, the opposite side of the core will also be updated when the handle is dragged.

  If is displayed, the opposite side of the core is not updated.

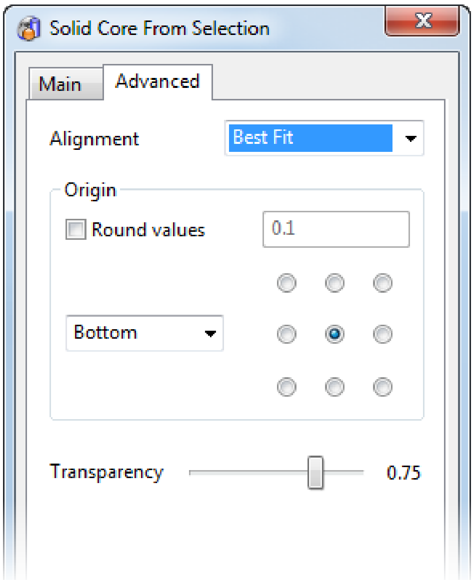
3 Use the **Boolean** option to control whether or not the core should be Boolean subtracted or Boolean intersected with the selected faces.

- If deselected, the operation will create a solid core as a primitive solid.
- If selected, the operation will create a solid with a Boolean feature.

The Boolean section does not appear if the selected items are not faces of a solid.

Advanced tab

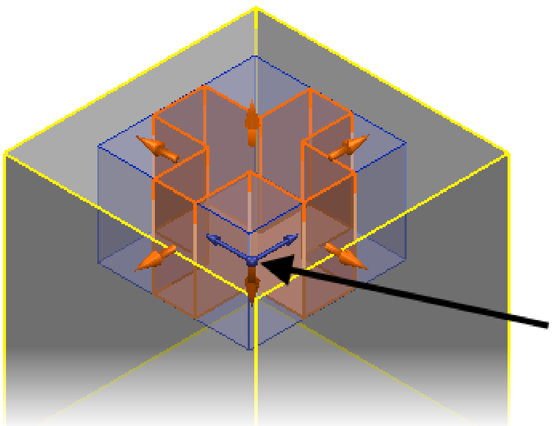
Use the options on the **Advanced** tab to specify additional settings.



1 Select **Best Fit** or **Workplane** from the **Alignment** drop-down list.

If you select **Best Fit**, the Z-axis of the core is aligned with the Z-axis of the currently active workplane, but it is rotated to produce the tightest bounding box around the selected items.

2 Use the **Origin** options to specify the origin position of the primitive solid:



- **Round values** — Select this option to round the values and enter the rounding factor.
- Use the grid to position the origin of block.
- Use the drop-down list to specify the location of the origin:

  Bottom — The origin will be at the minimum Z coordinate of the solid.

Top — The origin will be at the maximum Z coordinate of the solid.

3 Use the **Transparency** slider to specify the transparency of the faces of the graphical preview of the core.

To avoid the need to re-enter the value, the rounding value that you enter is remembered for your PowerSHAPE session.

Using solids

The following sections contain information on finding open boundaries and using scaling constraints:

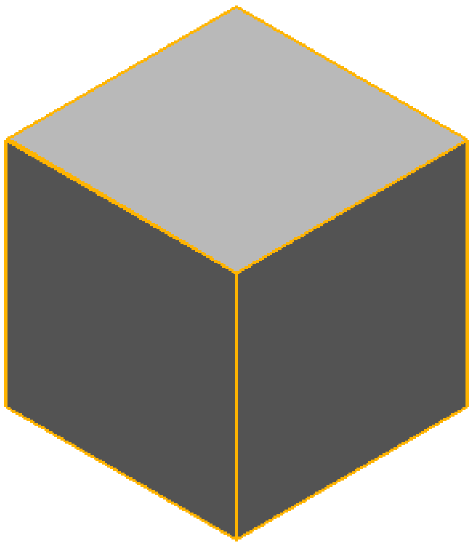
Finding open boundaries on solids (see page 91)

Using Scaling Constraints (see page 92) (*version 8 solids only*)

Finding open boundaries on solids

You can highlight any segments of selected solids' boundaries which are open (that is, not properly linked) within tolerance.

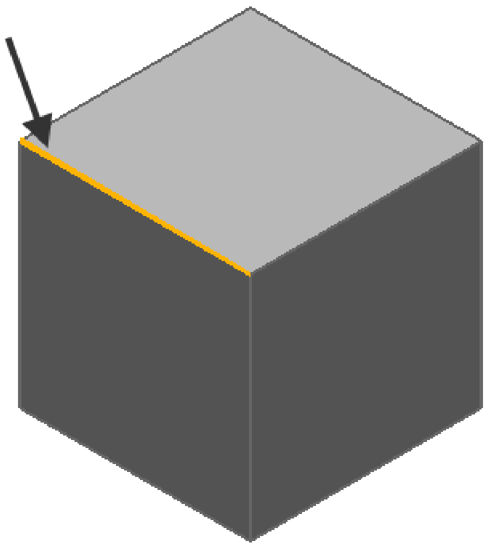
1 Select the solids you want to check.



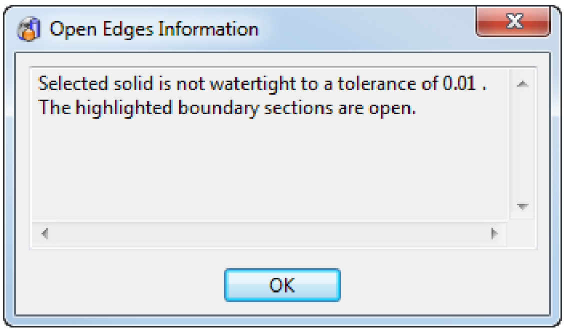
2 Click **Model Fixing** on the **General edits** flyout.

3 Click **Show open edges in solid** (*Model fixing toolbar*) to display the **Open Edges Information** dialog.

Any open segments of the solids' boundaries are highlighted.



The dialog tells you whether the solids are fully linked.



4 Click **OK**. Any open segments of solids' boundaries are no longer highlighted.

If there are small holes in the solid, fix these using one of the following methods:

- Convert the solid it back to surfaces, increasing the tolerance and re-make the solid.
- Select **Tools > Model Fixing > Repair Solid**.

Using Scaling Constraints

Scaling constraints can only be applied to version 8 solids. If your model contains post-version 8 solids, they will need to be converted manually *(see page 21)* *before constraints can be applied.*

You can apply constraints to solids that you wish to scale. It is advisable to use small scaling factors (for example, 1.2, 1.3) when you are using scaling to ensure that the stitch margin is able to produce a smooth and acceptable solution.

Scaling constraints let you keep certain entities locked whilst scaling the rest. You can apply scaling constraints to the following:

- solids (see page 93)
- groups of adjoining surfaces in a solid that have been defined as a User Defined feature (see page 94)
- symbols.

Scaling constraints are used where graded copies of some geometry are required but portions of the geometry are to be kept constant irrespective of the scale factor that is being applied.

Examples

You could use scaling constraints in the following situations:

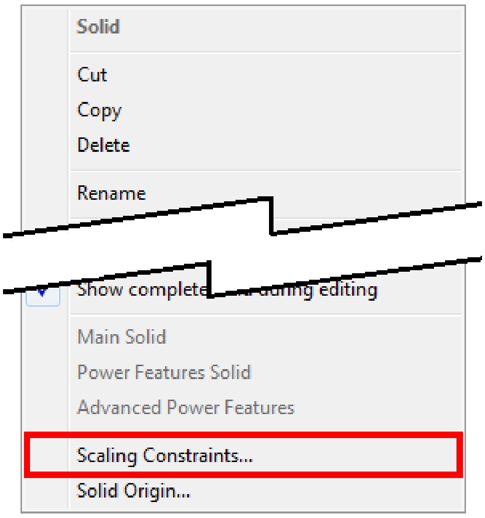
- Bottles; scale the whole bottle but keep the neck of the bottle the same size.
- Shoes; scale the shoe sole but keep the airbag or logos the same size.

Applying scaling constraints will not make any immediate change to the model that is displayed, but will be used when you scale the model.

Setting constraints for solids

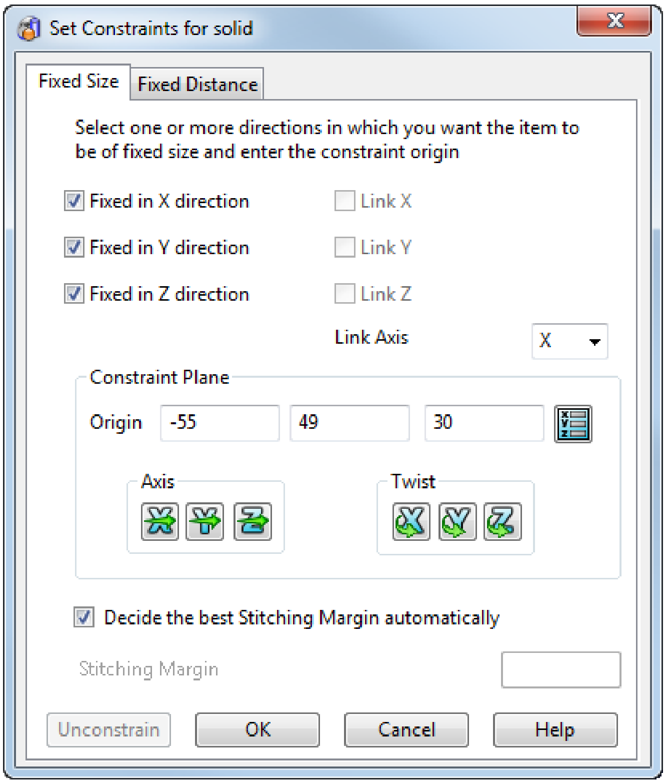
1 Select the solid to be constrained.

2 Select **Scaling Constraints...** from the solid context menu in the solid tree.



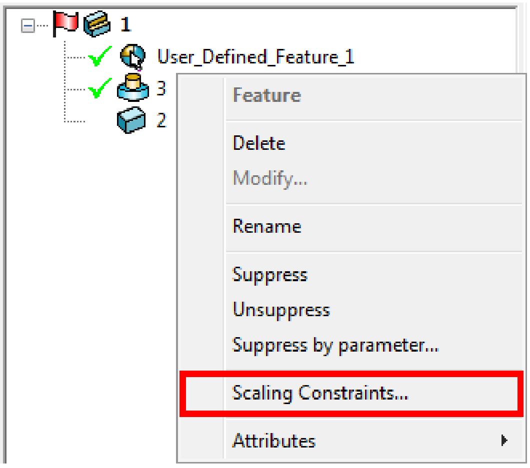
*The **Scaling Constraints** option is also displayed on the **Symbol** context menu to allow scaling constraints to be set for symbols.*

The **Set Constraints for solid** dialog will be displayed.



Setting constraints for solid features

The **Scaling Constraints** option is only available on the solid tree for a **User-defined** feature. It does not appear on the context menu for other solid features.



Set Constraints for solid dialog

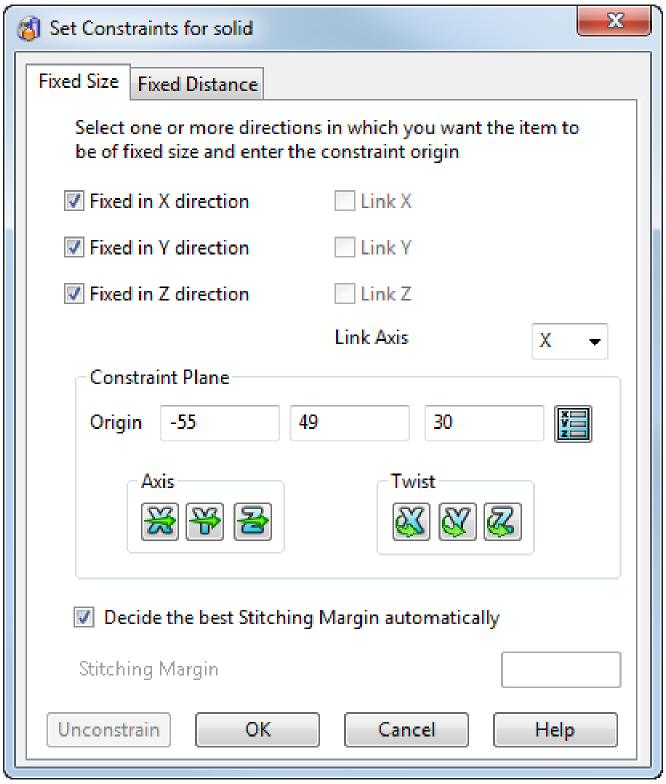
You can define the scaling constraints in one of the following ways:

- Fixed Size (see page 95) - Use this method to select the directions that are to remain unaltered by scaling.

- Fixed Distance (see page 97) - Use this method to maintain the relative position of two solids or two user-defined features.

Set Constraints dialog - Fixed Size

This page lets you modify the **Constraint Plane** and select the directions to remain unaltered by scaling.



Use the check boxes to select the directions to remain unaltered by scaling.

Fixed in X direction - Constrain the item so that it is of fixed size in the X direction with respect to the constraint workplane.

Fixed in Y direction - Constrain the item so that it is of fixed size in the Y direction with respect to the constraint workplane.

Fixed in Z direction - Constrain the item so that it is of fixed size in the Z direction with respect to the constraint workplane.

Use the **Link Axis** and **Link** options to create linked scaling constraints. For example, link the X and Y dimensions as follows:

1. Deselect **Fixed in X direction**.
2. Select **Link X**.
3. Select **Y** axis from the **Link Axis** drop-down list. This links X and Y dimensions so any change that you make to the Y dimension will also be applied to the X dimension.

4 Use **Scale** (*General edits toolbar*) to scale the solid. The scaling constraints that you set are applied and a confirmation message is displayed.

Constraint Plane

When you scale the constrained object, it is the constraint plane origin that is scaled. The constraint plane origin moves with the scaled origin and the constrained entity repositions itself around the updated constraint plane origin.

The default origin of the constraining entity is displayed on the model. Change the **Origin**, **Axis** and **Twist** options to modify the constraint plane

Origin - The X, Y, Z coordinates of the centre point of the constraining entity. If you need to alter the origin, enter the new coordinates in the relevant box.

 - Display the Position dialog so that you can define the constraint plane.

Axis - To change the direction of the constraint plane's axes, click the button of the axis you wish to change. This displays the **Direction** dialog. Use this dialog to input the axis information and click **Accept**.

Twist - To twist the constraint plane about its axis, click the button of the axis you wish to twist. This displays the **Calculator** dialog. Use this dialog to input the twist value and click **Accept**.

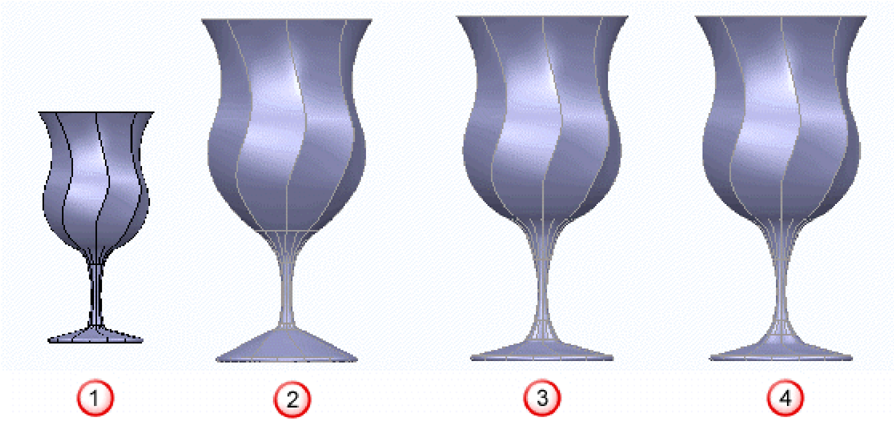
Decide the best Stitching Margin automatically - If you want to keep the user defined feature the same size, scaling will result in a gap around the feature. A stitch margin stretches the portions of the solid around the user defined feature to blend the gap generated around the feature and maintain the closure of the solid.

If *ON*, PowerSHAPE will decide the size of the stitching margin, depending upon the size of the gap created.

If *OFF*, the **Stitching Margin** option is active.

Stitching Margin - Enter the distance around the periphery of the item being constrained, where stretching would occur when you scale the item. This option is only available if **Decide the best Stitching Margin automatically** is *OFF*.

The following shows the effects of using the automatically calculated stitch margin and using different sizes of user defined stitch margins.



1 - Original.

2 - Stitching margin decided automatically.

3 - Userdefined stitching margin of 5.

4 - User defined stitching margin of 10.

Unconstrain - Remove all the constraints on the selected item and close the dialog.

OK - Accept the constraints settings and close the dialog.

Set Constraints dialog - Fixed Distance

This dialog is used to maintain the relative position of two solids or two user-defined features.

You can constrain a solid to be a fixed distance from:

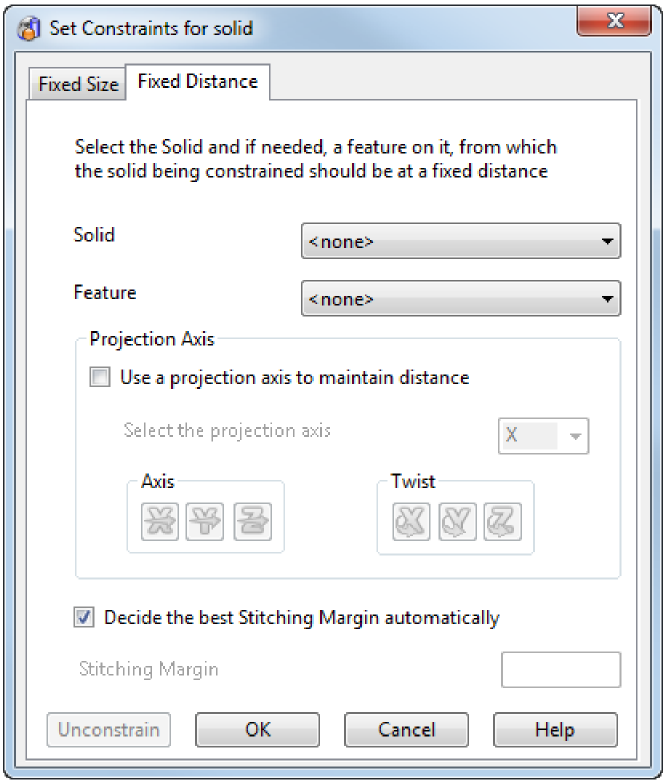
- another solid.
- a user defined feature on another solid.

You can constrain a user defined feature to be at a fixed distance from:

- Another user defined feature that is in the solid tree.
- A solid that is not the parent of the user defined feature being constrained.
- A user defined feature on a solid that is not the parent of the user defined feature being constrained.

*You cannot define circular constraints. The following constraints would **not** be valid:*

(feature 1 should be a fixed distance from feature 2*)* ***AND***
(feature 2 should be a fixed distance from feature 1*)*



Use the **Solid** and **Feature** drop-down lists to define the reference for the object to be constrained. The reference can be another solid or a user defined feature on another solid. When scaling, the constrained object will be scaled such that it will maintain its distance from its reference object.

Solid - Select one of the available solids from the drop-down list.

Feature - Select one of the available features from the drop-down list.

Projection Axis

Use a projection axis to maintain distance - Click to use an axis of the workplane to maintain the distance. When this option is selected, modify the projection axis by changing the **Origin**, **Axis** and **Twist** options.

Select the projection plane - Select the axis from the drop-down list. The distance is maintained along the selected axis.

Axis - Click the button of the axis to be changed. Enter the axis information using the **Direction** dialog.

Twist - Click the button of the axis you wish to twist. Enter the twist value using the **Calculator** dialog.

Decide the best Stitching Margin automatically - If you want to keep the user defined feature the same size, scaling will result in a gap around the feature. A stitch margin stretches the portions of the solid around the user defined feature to blend the gap generated around the feature and maintain the closure of the solid.

If selected, PowerSHAPE will decide the size of the stitching margin, depending upon the size of the gap created.

If deselected, the **Stitching Margin** option is active.

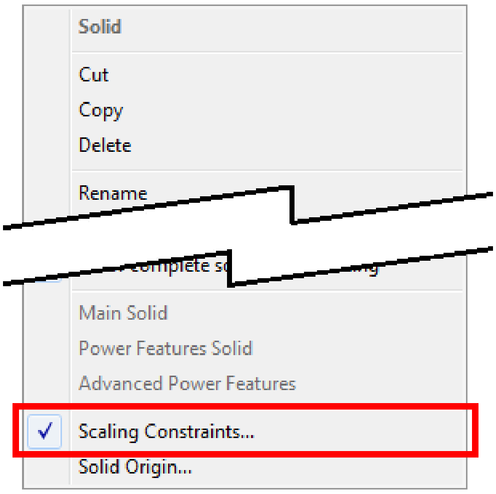
Stitching Margin - Enter the distance around the periphery of the item being constrained, where stretching would occur when you scale the item. This option is only available if **Decide the best Stitching Margin automatically** is selected.

Unconstrain - Remove all the constraints on the selected item and close the dialog.

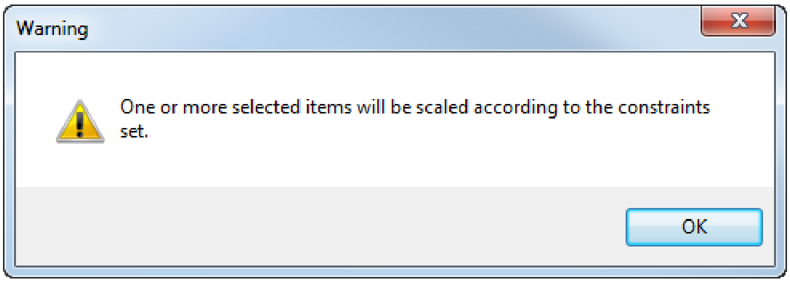
OK - Accept the constraints settings and close the dialog.

How do I know if scaling constraints are set?

When you set up a scaling constraint,  will appear against **Scaling Constraints** in the context menu.



*The warning message shown below is displayed when you use the **Scale** option (General edits toolbar) to scale multiple objects if any of the selected items has scaling constraints already set.*



Editing and Removing constraints

You can edit the constraints on a version 8 solid or symbol as follows:

1 From the solid tree context menu, select **Scaling Constraints**.

2 Use the Set constraints for solids dialog (see page 94) to choose different options.

You can remove the constraints on a version 8 solid or symbol as follows:

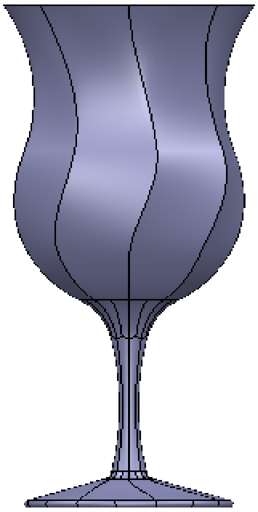
1 From the solid tree context menu, select **Scaling Constraints**.

2 Select **Unconstrain** on the Set Constraints for solids dialog (see page 94).

Scaling constraints are not available for post-version 8 solids (Parasolids)

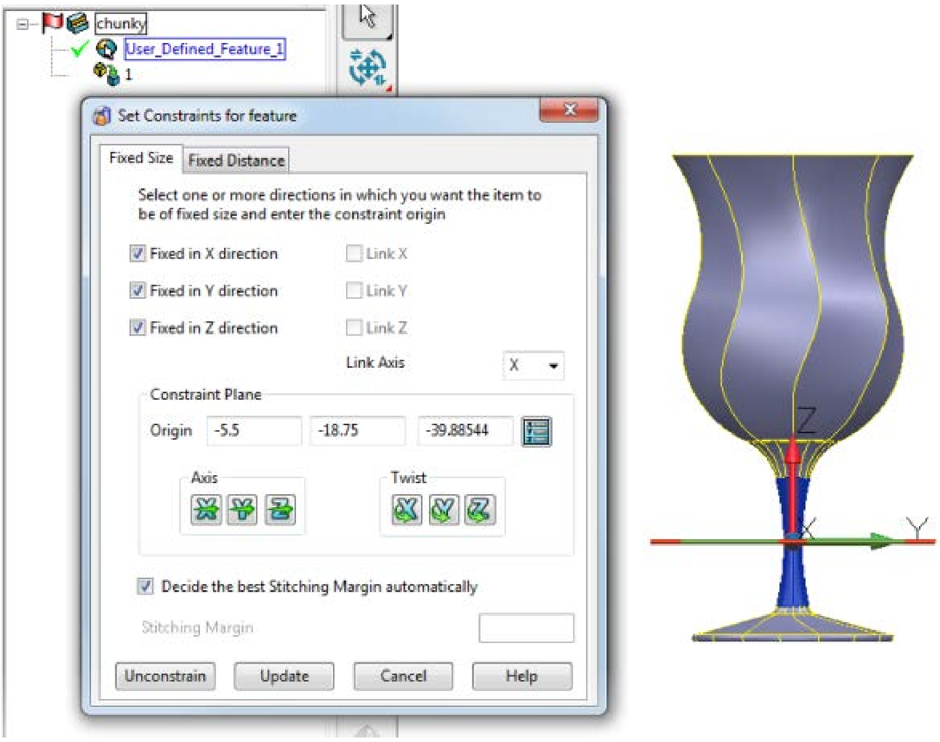
Scaling using Fixed Size scaling constraints

The following example shows you how to create a scaled version of a wine glass, whilst keeping the stem of the glass the same length.

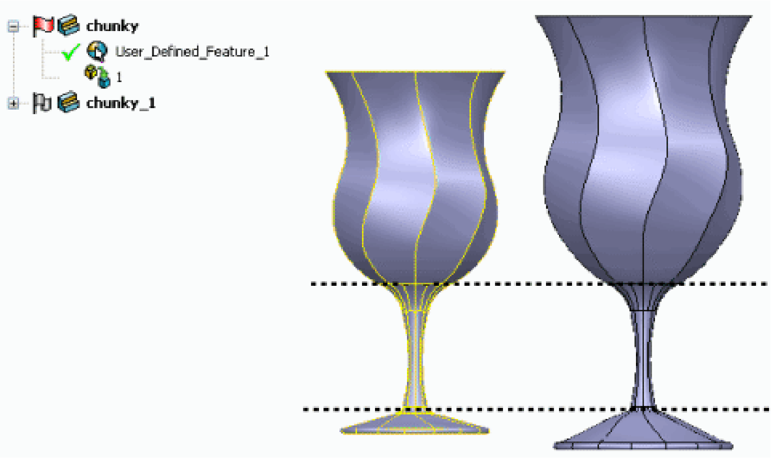


1 Define the stem surfaces as a user-defined feature.

2 Using the **Set Constraints for feature** dialog, add a **Fixed Size** constraint on the user-defined feature in X, Y and Z direction, moving the Z coordinate of the origin of the **Constraint Plane** to -*33.72899*



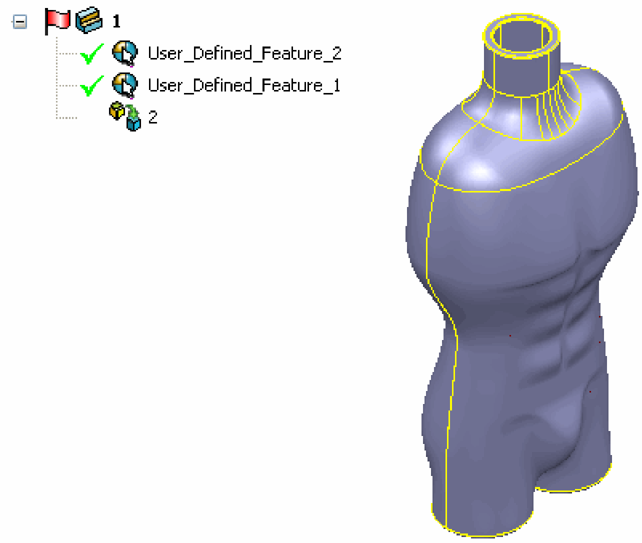
3 Scale the wine glass uniformly by a factor of *1.2*



You will see that geometry of the body of the wine glass has been scaled, but the stem of the scaled version of the wine glass is the same size as that of the original.

Scaling using Fixed Distance scaling constraints

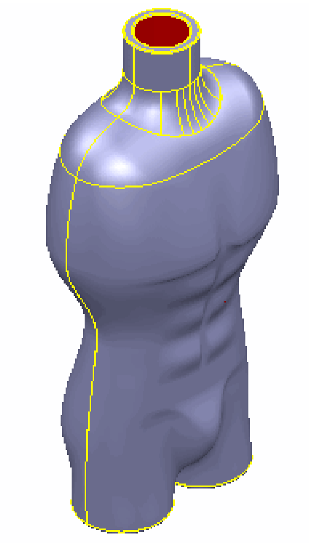
The following example shows you how to create a scaled version of a bottle, whilst maintaining the thickness of the bottle.



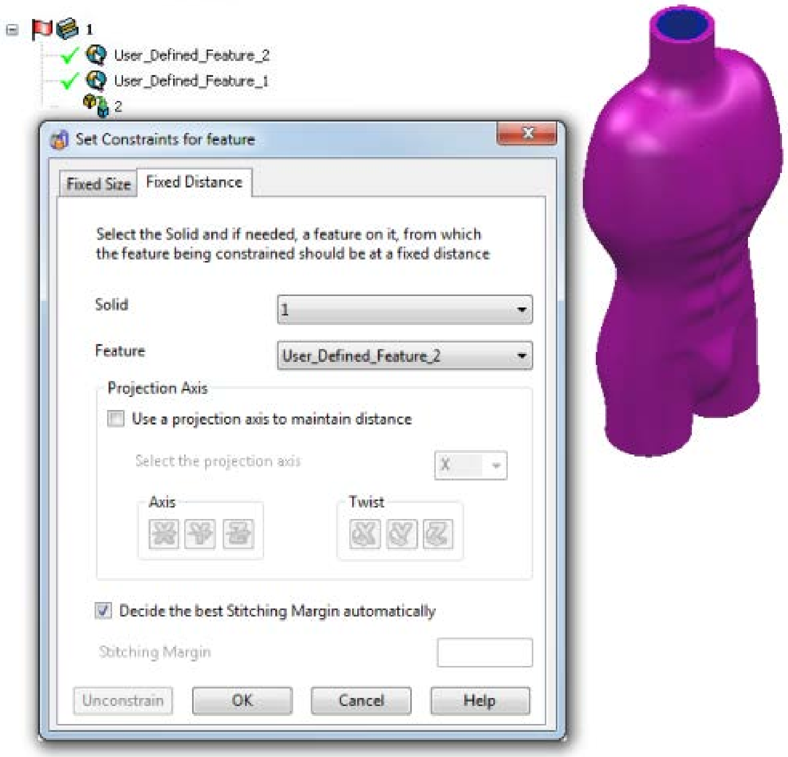
1 The model above has two user defined features:

User_Defined_Feature_1 - the bottle inner.

User_Defined_Feature_2 - the bottle outer.



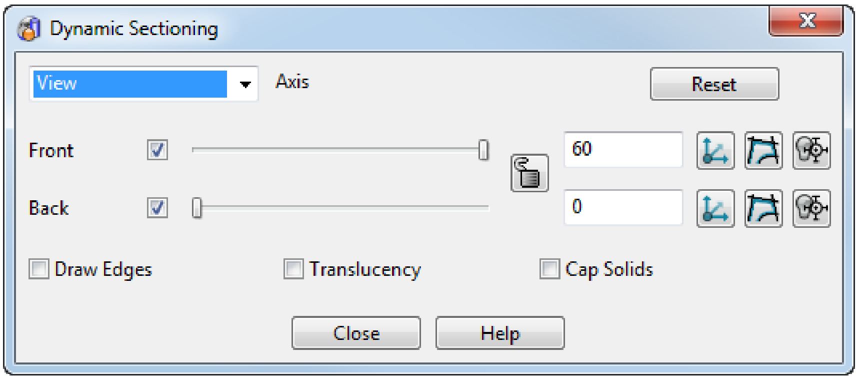
2 Using the **Set Constraints for feature** dialog, add a **Fixed Distance** constraint on the user-defined feature, such that *User_defined_Feature_2* will remain a fixed distance from *User_Defined_Feature_1*.



3 Scale the bottle uniformly with a factor of *1.2*

The bottle will be scaled in size but the thickness of the bottle will remain the same because of the fixed distance constraint.

You can compare the thickness of the two bottles by using dynamic sectioning.



Scaling with constraints applied completely changes the shape of the model. The stretching process may well affect any features you have used. It is therefore advisable to have only user defined features in your model when using constrained scaling.

Solid features

A solid feature is an operation that has been applied to a solid. By remembering the details of the operation, a feature can be redefined, reordered, deleted and suppressed. The history of solid operations performed on a solid is stored in the solid feature tree (see page 254). Each solid operation is defined as a feature on the solid.

Creating solid features

1 On the **Main** toolbar, select the **Feature** button to display the **Feature** toolbar

2 Use the following buttons to create solid features.

   Smart Feature Recognition

   Smart Feature Selection

   Adding onto a solid (see page 108)

   Subtracting from a solid (see page 110)

   Intersecting using a solid (see page 112)

   Split a solid (see page 117)

   Solid cut (see page 122)

   Radial cut (see page 134)

   Solid boss (see page 138)

   Boolean boss (see page 148)

   Solid hole (see page 151)

   Hollow solid (see page 167)

   Thickened solid (see page 174)

   Solid bulge (see page 176)

- Morph (see page 312)
- Solid fillet (see page 180)
- Solid chamfer (see page 206)
- Rib fillet (see page 225)
- Pocket or protrusion feature (see page 227)
- User-defined feature (see page 239)
- Wrap feature (see page 242)

Editing a feature

You can edit a feature in one of the following ways:

- Using the feature tree. (see page 257)
- Using the options on the **General Edit** toolbar or the corresponding **General Edits** options on the **Edit** menu.

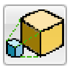
Editing a feature using the General Edit toolbar

You can use the **Move**, **Rotate**, **Scale**, **Mirror** and **Create pattern** options from the **General Edit** toolbar on most features. You can also use the corresponding **General Edit** options on the **Edit** menu.

You cannot apply these commands to some features, such as the fillet feature. PowerSHAPE automatically deselects such features when you begin the edit command.

To edit a feature using the General Edit toolbar:

1. Select a feature in the tree.
2. Click **General Edits** .
3. The following options are available on the toolbar:
   - Edit selected item (solid)
   - Edit selected sub-item (solid feature)
   - Move
   - Rotate

-  Scale *(Uniform scaling only)*
-  Mirror
-  Pattern of items

Exporting solid features

The following solid features can be exploited to PowerMILL, using the **Delcam Geometry + Features (.dgk)** option on the drop-down list (*Export dialog*):

- holes
- pockets/protrusions
- cuts/bosses

Fillet and chamfer information at the bottom and top of a feature is also exported, but is not currently used by PowerMILL.

Adding onto a solid

You can add the following onto a solid:

- Inactive solids — after the operation, the inactive solid no longer exists.
- Surfaces — PowerSHAPE automatically makes the surface into a solid before adding it to the solid.
- Symbols— all solids and surfaces within the symbol are copied and added onto the solid.T

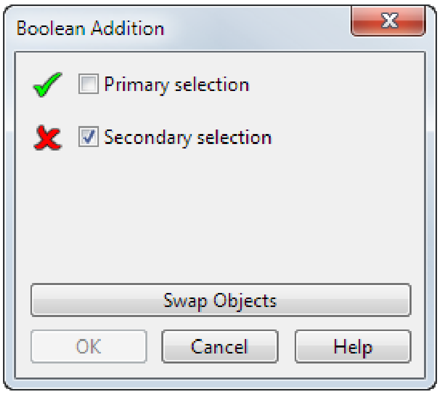
The symbol itself does not change or get deleted. If you use a symbol, the add feature of the symbol is non-scalable in the solid.

You can use Boolean addition on:

- The active solid.
- A pre-selected inactive solid, when the active solid is hidden.

To add objects to a solid:

1 Click (*Solid feature toolbar*) to display the **Boolean Addition** dialog.



If there is a visible active solid, this is automatically be used as the **Primary selection**. If you pre-select an inactive solid while the active solid is hidden, the inactive solid is used as the **Primary selection**.

2 Select a secondary object. The changes to .

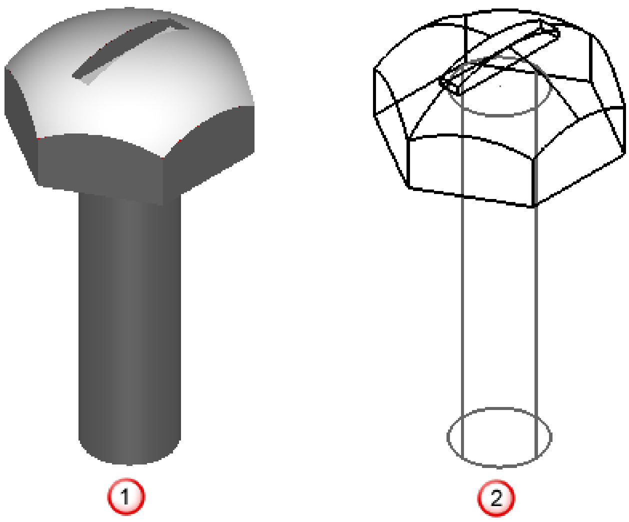
If necessary, click **Swap Objects** to reverse the **Primary** and **Secondary selection** objects.

3 Click **OK** to confirm your selection and close the dialog.

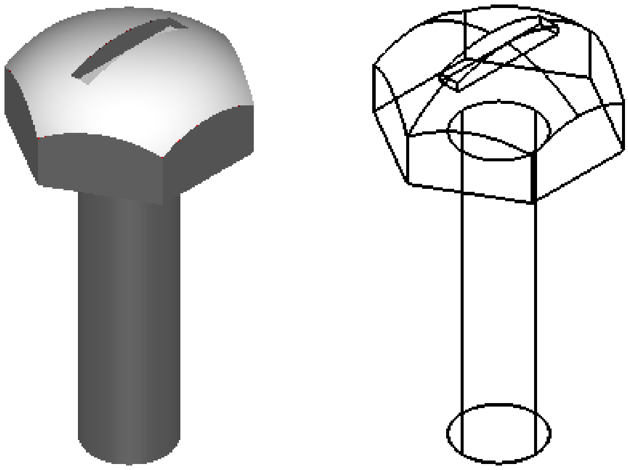
An **Add feature** icon representing the operation appears in the solid feature tree.

If the added solid has features, they are added as a sub-tree next to the **Add feature** icon .

Below is an example of two intersecting solids, displayed as shaded 1 and as wireframe 2.



In this example, the following solid is created:



3 4

The shaded view 3 looks identical to the original, but in the wireframe view 4 you can see the two separate solids have been merged into one solid.

Subtracting from a solid

You can subtract the following from a solid:

- Inactive solids — after the operation, the inactive solid no longer exists.
- Surfaces — PowerSHAPE automatically makes the surface into a solid before subtracting it from the active solid.
- Symbols —all solids and surfaces within the symbol are copied and subtracted from the active solid.

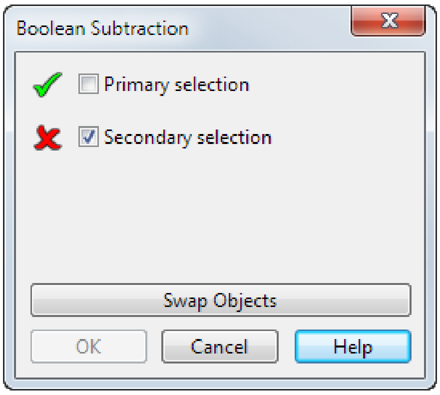
Note the symbol itself does not change or get deleted. If you use a symbol, the subtract feature of the symbol is non-scalable in the solid.

You can use Boolean subtraction on:

- The active solid.
- A pre-selected inactive solid, when the active solid is hidden.

To subtract from a solid:

1 Click (*Solid feature toolbar*) to display the **Boolean Subtraction** dialog.



If there is a visible active solid, this is automatically be used as the **Primary selection**. If you pre-select an inactive solid while the active solid is hidden, the inactive solid is used as the **Primary selection**.

2 Select a secondary object. The ✘ changes to ✔.

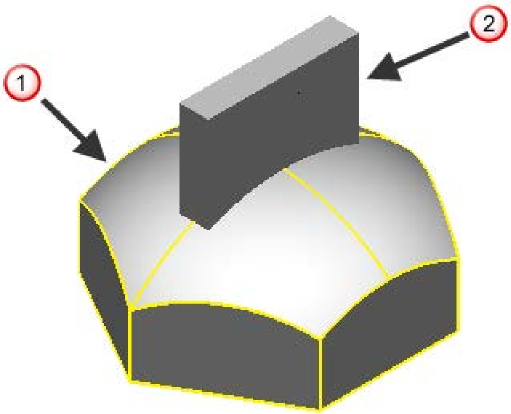
If necessary, click **Swap Objects** to reverse the **Primary** and **Secondary selection** objects.

3 Click **OK** to confirm your selection and close the dialog.

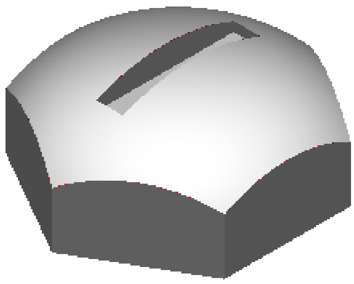
A **Subtract feature** icon representing the operation appears in the solid feature tree.

If the subtracted solid has features, they are added as a sub-tree next to the **Subtract feature** icon

Below is an example an active solid 1 intersected by an inactive solid 2:



In this example, the new active solid is given below:



Intersecting using a solid

You can intersect the following with a solid:

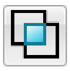
- Inactive solids — after the operation, the inactive solid no longer exists.
- Surfaces — PowerSHAPE automatically makes the surface into a solid before intersecting it with the solid.
- Symbols — all solids and surfaces within the symbol are copied and intersected with the solid.

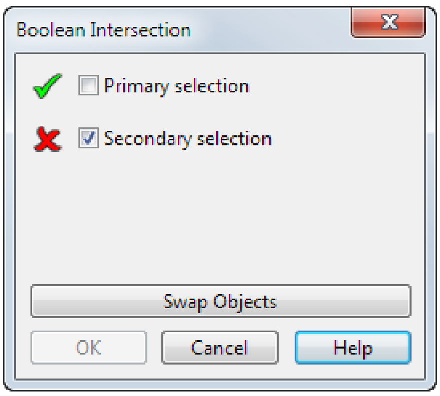
The symbol itself does not change or get deleted. If you use a symbol, the intersect feature of the symbol is non-scalable in the solid.

You can use Boolean intersection on:

- The active solid.
- A pre-selected inactive solid, when the active solid is hidden.

To intersect with the active solid:

1 Click  (*Solid feature toolbar*) to display the **Boolean Intersection** dialog.



If there is a visible active solid, this is automatically be used as the **Primary selection**. If you pre-select an inactive solid while the active solid is hidden, the inactive solid is used as the **Primary selection**.

2 Select a secondary object. The changes to .

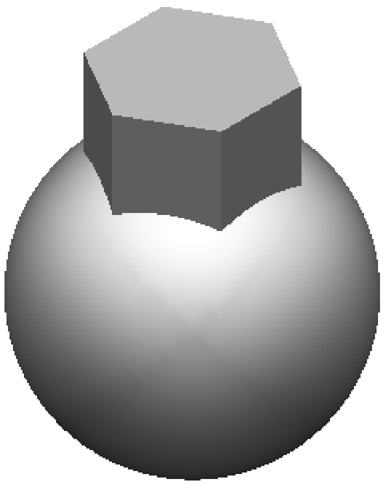
If necessary, click **Swap Objects** to reverse the **Primary** and **Secondary selection** objects.

3 Click **OK** to confirm your selection and close the dialog.

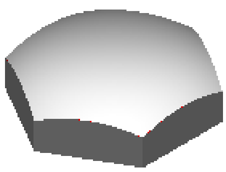
An **Intersect feature** icon representing the operation appears in the solid feature tree.

If the intersected solid has features, they are added as a sub-tree next to the **Intersect feature** icon .

Below is an example of two intersecting solids:



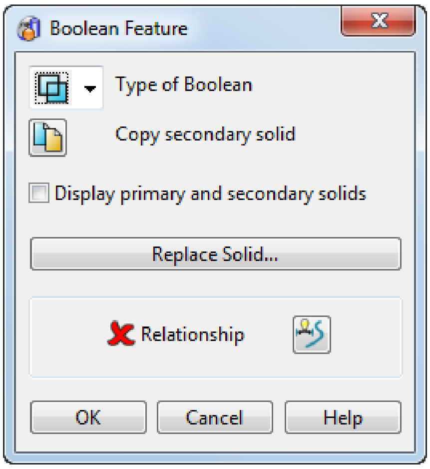
In this example, the following solid is created:



Editing an add, subtract or intersect feature

You can replace or edit the solid used in the add, subtract and intersect features.

1 Double-click the **Add feature**, **Subtract feature** or **Intersect feature** icon in the tree to display the **Boolean Feature** dialog.



2 Use the dialog to edit the feature.

- Use the drop down list to change the type of Boolean operation.

- Creates a copy of the selected solid used in the feature. When you select **OK**, the entire sub-branch (the solid and its history) will be copied. If the sub-branch only contains one primitive feature, the new solid will be a primitive solid.

Display primary and secondary solids - If *ON*, the original solids are displayed. If *OFF*, the original solids are not displayed. The default setting is *OFF*.

If this option is OFF *and you click to display the Solid Feature Relationship dialog (see page 251), the key points on the solids (corners and centres of planar faces) will not be available.*

Replace Solid - Select this if you want to replace the solid with another one. The **Replace Solid** (see page 115) dialog will be displayed.

- Displays the Solid Feature Relationship dialog (see page 251). This allows the relative position of the feature to be defined with respect to the solid. The current status of the relationship is indicated by one of the following

- No relationship is currently defined.

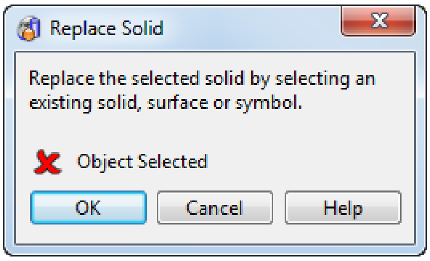
- A valid relationship is defined.

- a relationship is defined, but there is a problem with the definition.

3 Click **OK** to accept the changes.

Replace Solid dialog

This replaces the solid used in an add, subtract and intersect feature.

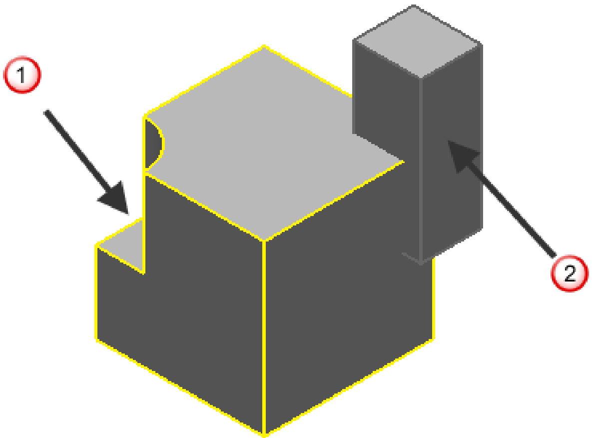


Object selected - This indicates whether a replacement object is selected for the feature. The replacement object can be a solid, surface or a symbol.

OK - Accepts the changes and closes the dialog.

Cancel - Closes the dialog without saving any changes.

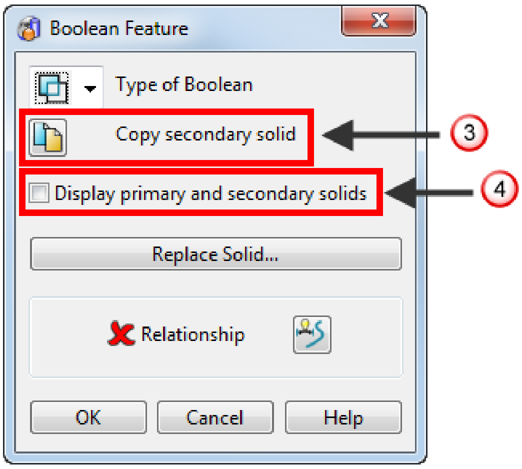
In the following example, the subtracted solid 1 will be replaced by the block 2.



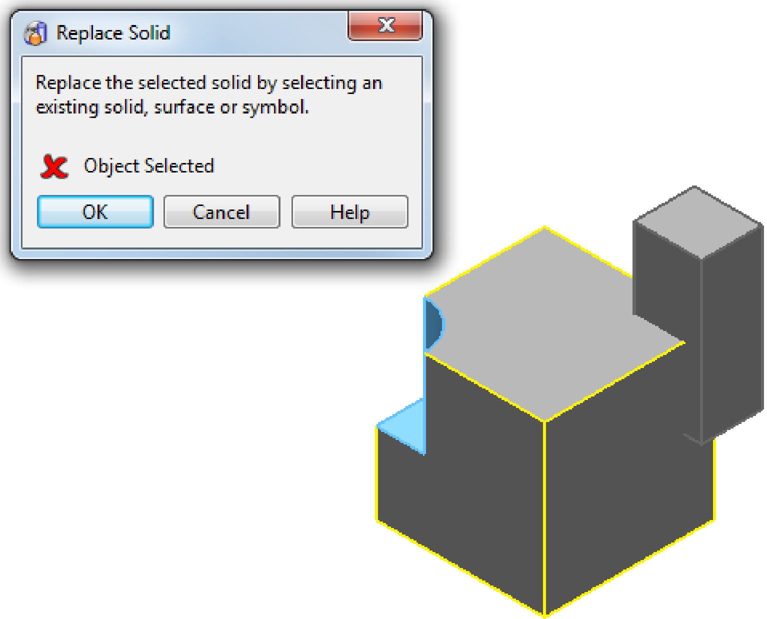
1 Double-click the **Subtract feature** icon in the solid feature tree to open the **Boolean Feature** dialog.

If you want to keep a copy of any subtracted solids, click **Copy secondary solid** 3 before clicking **Replace Solid**.

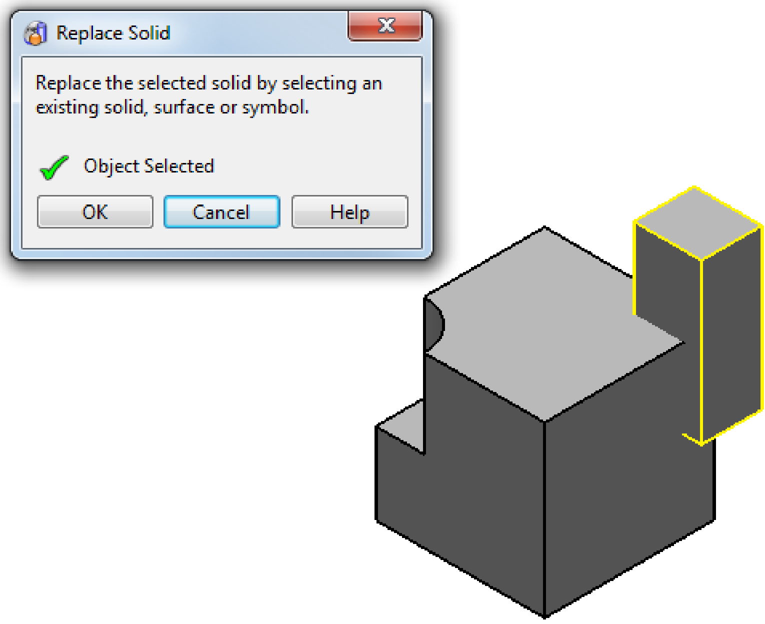
If you want to display any subtracted solids, select **Display primary and secondary solids** 4 before clicking **Replace Solid**.



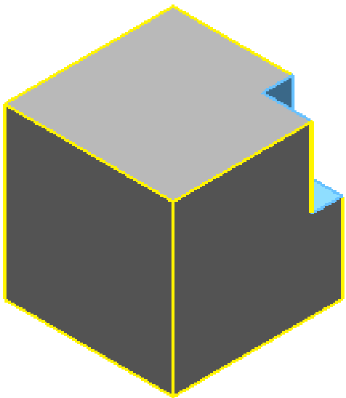
2 Click **Replace Solid** to open the **Replace Solid** dialog.



3 Select the object that you want to use to replace the original solid. The changes to .



4 Click **OK** to accept the changes. The cylindrical solid is removed and the block is subtracted from the active solid.



Spliting a solid

You can split a solid using one of the following methods:

- Using a pre-selected cutter (see page 118). If you have an active solid and a pre-selected cutter, the **Solid Split** dialog will not be displayed and the solid will be split according to default settings.

  The following can be pre-selected as the cutter:

  - an inactive solid. After the operation, the inactive solid no longer exists.
  - a solid face.

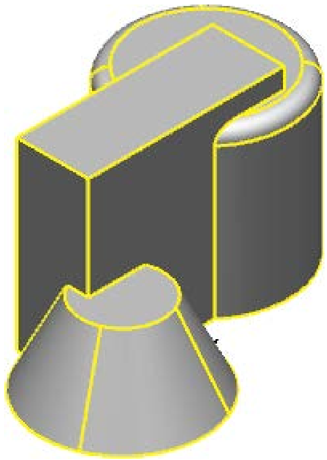
- a surface. PowerSHAPE automatically makes the surface into a solid before using it to split the active solid. After the operation, the surface no longer exists.
- a symbol. All solids and surfaces within the symbol are copied and used to split the active solid. The symbol itself does not change or get deleted.
- a workplane.

- Without a pre-selected active solid or cutter (see page 119).
- Using multiple faces (see page 119).

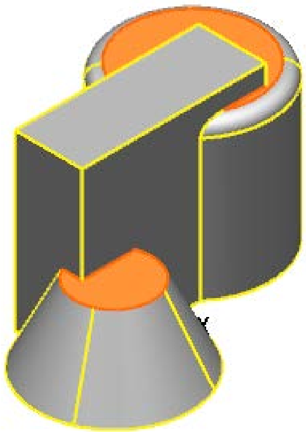
*The default **Split** operation produces unconnected solids with copied history trees (see page 121).*

Splitting a solid using a pre-selected cutter

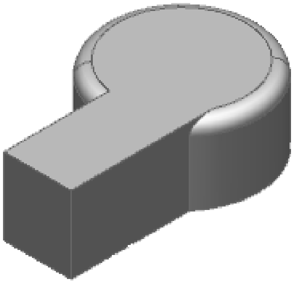
You can split a solid using multiple selected faces as the cutter. When multiple faces are selected, the single solid is split and no extra solids are created



1 **Ctrl** + click the faces to select them to be used as cutters.

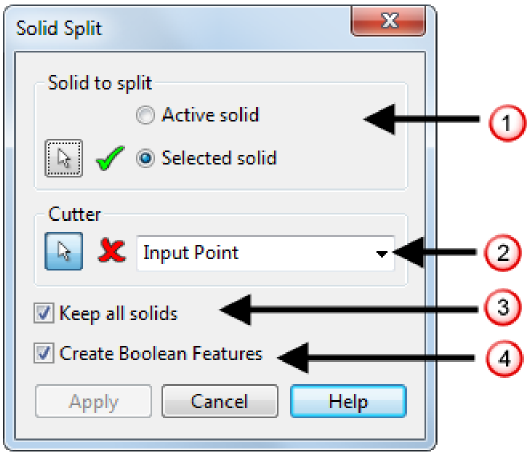


2 Click  (*Solid Feature toolbar*) to split the solid.



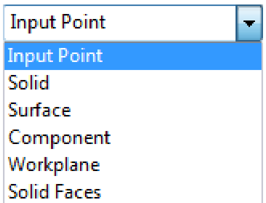
Splitting a solid without a pre-selected cutter

If you select **Solid split** without having a suitable cutter item selected or without having an active solid to work on, the **Solid Split** dialog is displayed.



1 Select the solid to be split ①.

2 Select the cutter type from the drop-down list ②.



3 Click  and select the cutter.

4 Select **Keep all solids** to retain all the new solids created by the split operation ③.

5 Select **Create Boolean features** to create Boolean features ④.

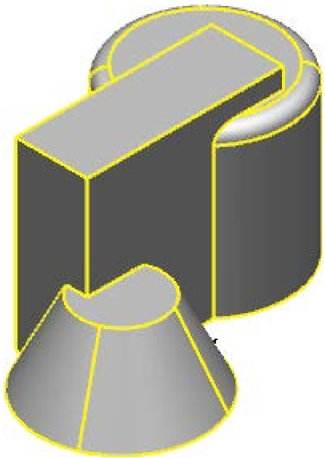
6 Click **Apply** to split the solid.

Splitting a solid using multiple faces

You can split a solid using multiple selected faces.

This example does not use a pre-selected cutter.

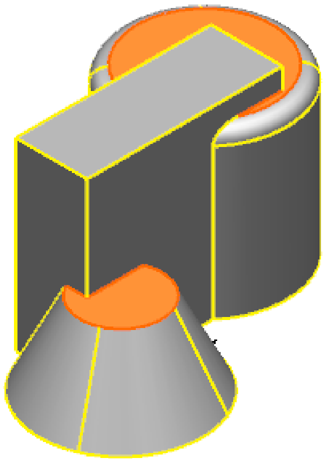
1 Select the solid to be split.

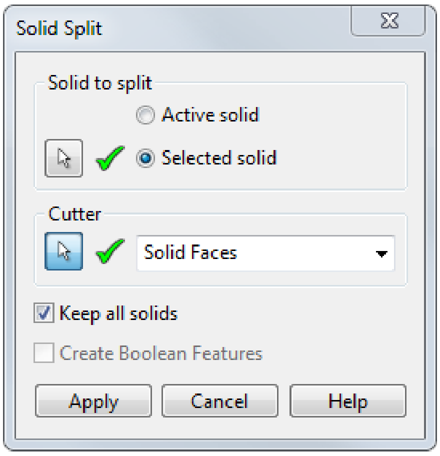


2 Select **Solid Faces** from the **Cutter** drop-down list.

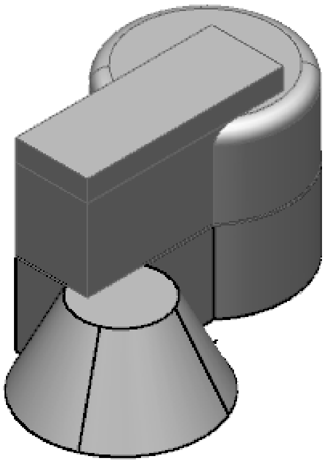
3 

to select the faces to be used as cutters.



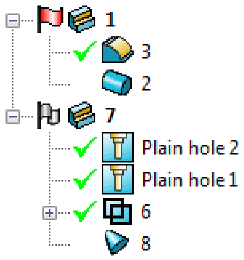


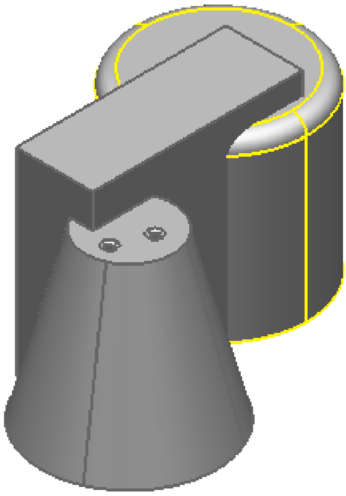
4 Click **Apply** to split the solid.



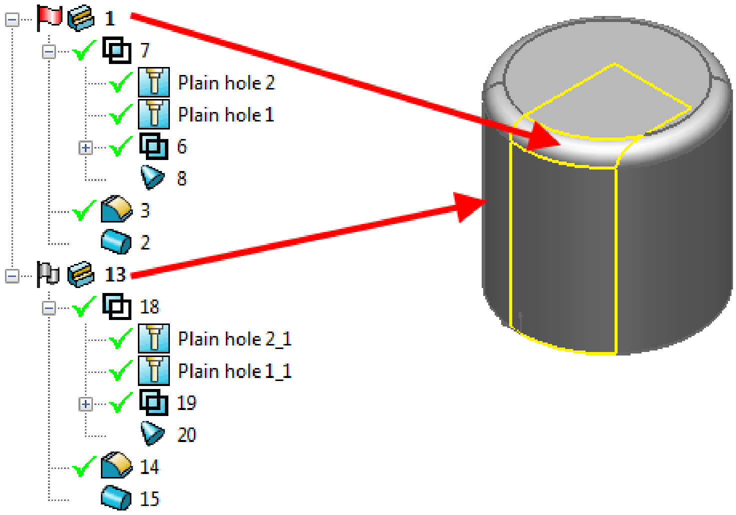
Split history

The **Split** operation produces unconnected solids with copied history trees (with the addition of the top boolean feature). The example below shows two solids:





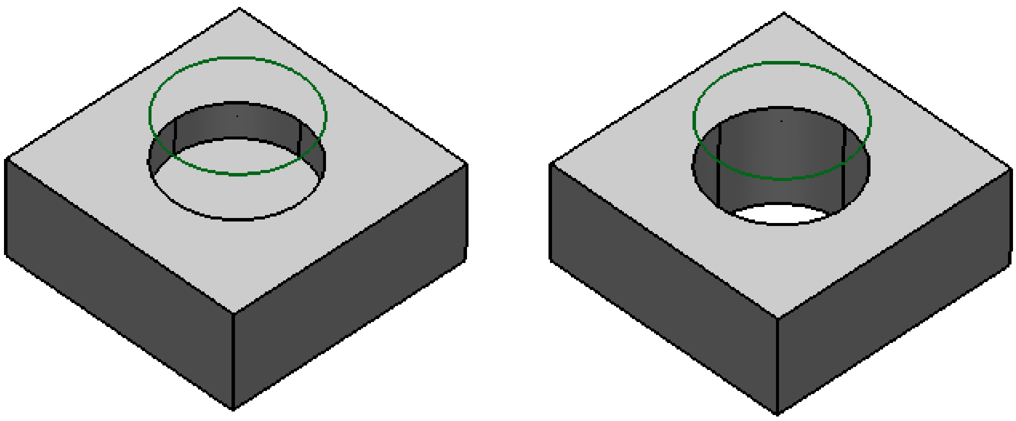
- Use **Split**  to split a solid and produce unconnected solids with copied history trees.



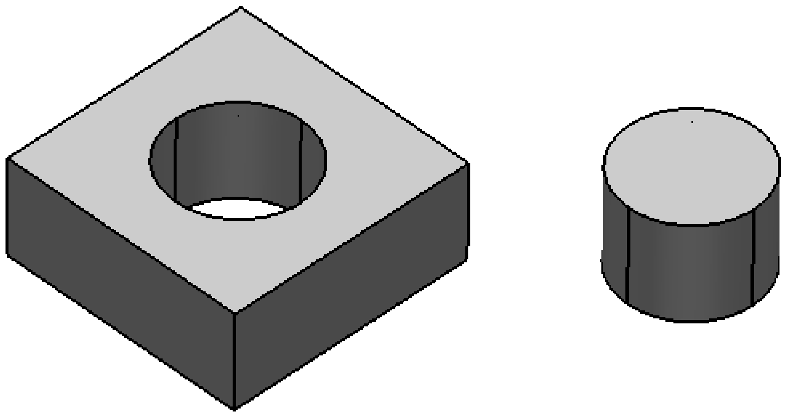
Creating a solid cut

You can cut shapes into a solid using wireframe objects. Behind the scenes, PowerSHAPE creates extruded solids from the wireframe objects and either subtracts or intersects them with the solid.

You can partially cut the solid or cut straight through.



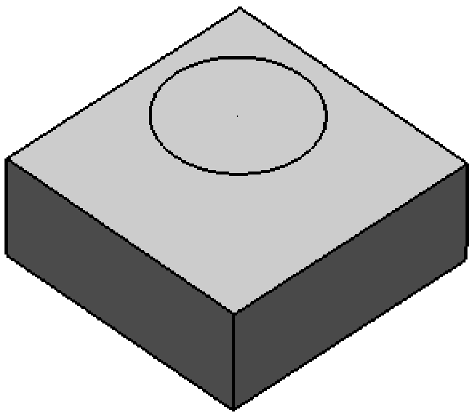
You can keep either the outside or inside material of the cut.



To create a solid cut use one of the following methods:

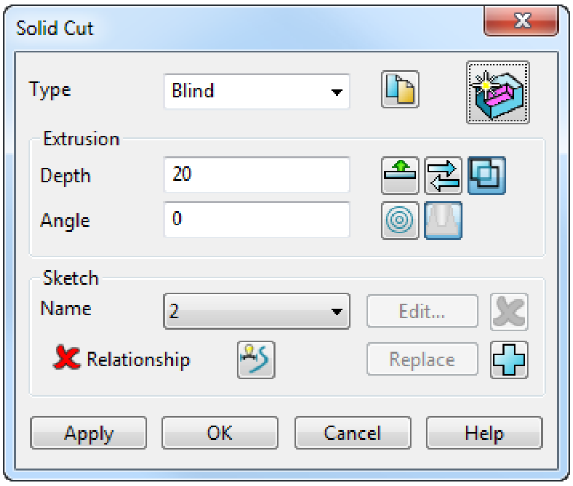
- Make sure you have an active solid and then select the wireframe objects.
- Drag your cursor to select an inactive solid and the wireframe objects together.

  The wireframe objects must be closed and planar.



Your selection can contain other objects. PowerSHAPE filters the valid wireframe from the selection.

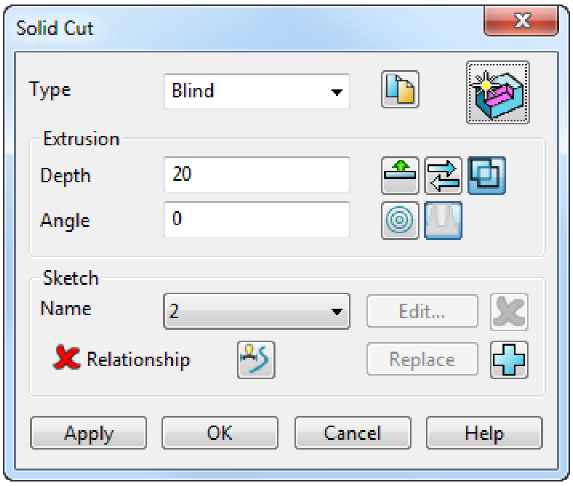
1 Click  (*Solid feature toolbar*).



2 Use the Solid Cut dialog (see page 124) to cut the wireframe objects into the solid.

Solid Cut dialog

Use this dialog to cut wireframe into a solid.

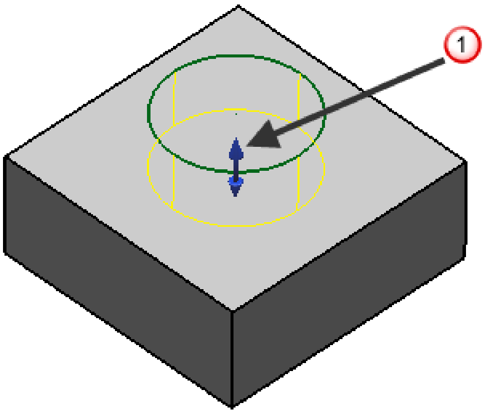


You can add a solid cut to:

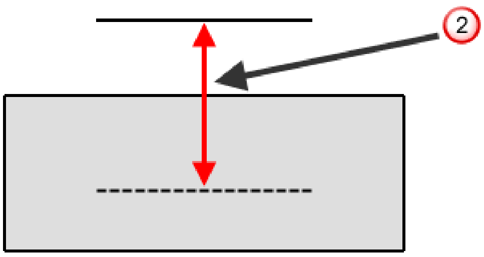
- The active solid.
- An inactive solid that has been preselected with the wireframe.

Each wireframe object is extruded along the axis normal to the plane in which it lies to create the extruded solid.

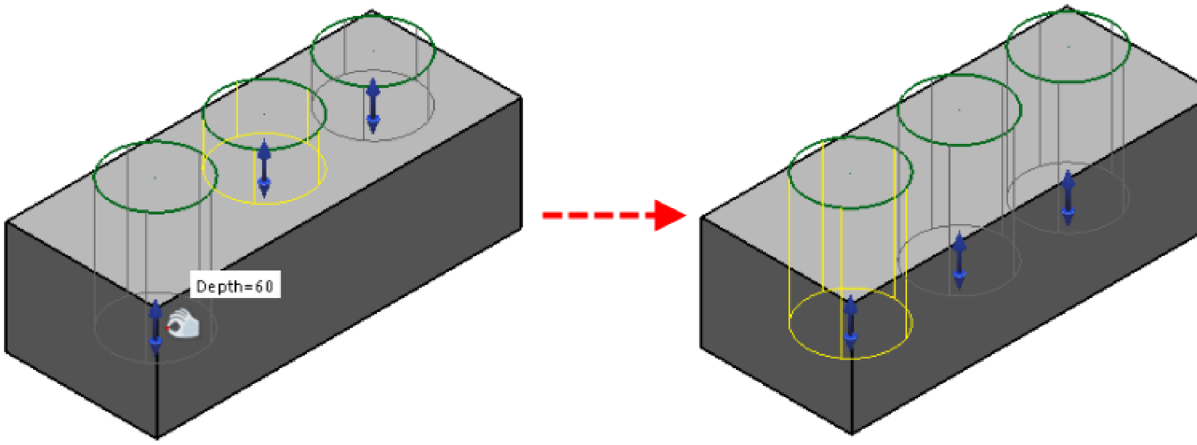
The extruded solid has a handle 1, which can be used to edit the length.



1 Select **Type** from the drop-down list:
   - Select **Blind** to set the depth to cut into the solid.
   - Select **Through** to cut all the way through the solid.
2 Click to create a copy of the original wireframe used in the construction of the cut.
3 Enter the **Depth**. This is the length of the extruded solid, measured from the plane of the wireframe 2.

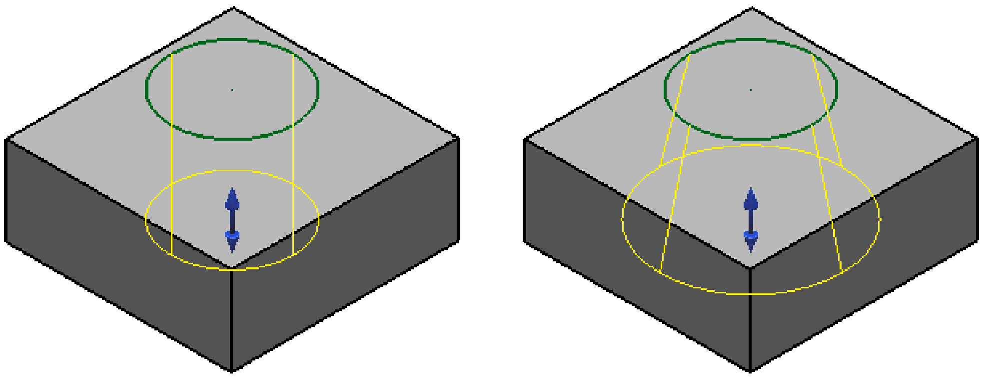


The depth can also be adjusted using the handle. If you are creating a solid cut using more than one wireframe, dragging the handle of an extruded solid will change the lengths of all the extruded solids. The value of **Depth** in the dialog updates to reflect the change.



To cut all the way through the solid, select the **Through** option from the **Type** option menu.

4 Enter the **Angle** to define a draft angle the extruded solid.



5 Click to extend the extrusion in both directions from the sketch. The button changes to to show that the extrusion will be extended in both directions.

6 Click to reverse the direction of the cut.

7 Click to display the **Solid Feature Relationship** dialog. This allows the relative position of the feature to be defined with respect to the solid. The current status of the **Relationship** is indicated by one of the following:

no relationship is currently defined.

a valid relationship is defined.

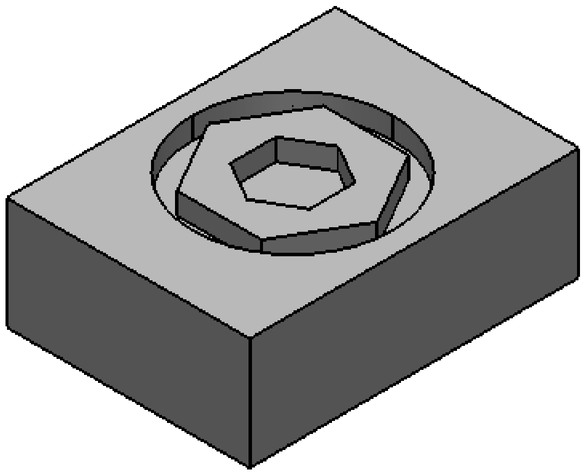
a relationship is defined, but there is a problem with the definition

8 Click to add the selected sketch (arc, curve or composite curve) to the feature.

9 Click to group nested sketches (see page 147).

If this option is selected, any sketches that lie on the same plane are nested and are used to produce an island-type extrusion.

- Using two concentric circular sketches would produce a tube with a thickness.
- Using three concentric circular sketches, the tube would have a cylinder inside it.



If this option is deselected, only the selected wireframe will be used to produce the solid boss.

10 Click or to specify if the draft angle is to go in or out.

11 Use and to remove material as required. The default setting will remove material outside the sketches.

- Click to remove material inside sketches.
- Click to remove material outside sketches.

12 Click **Apply** to create the cut, leaving the dialog displayed. This lets you

- create another cut.
- edit the existing cut.

Alternatively, click **OK** to create the cut and close the dialog.

A **Cut feature** icon appears in the solid feature tree.

13 Make changes to the cut as required:

a Click to switch to **Edit** mode. The button changes to .

b Click the cut.

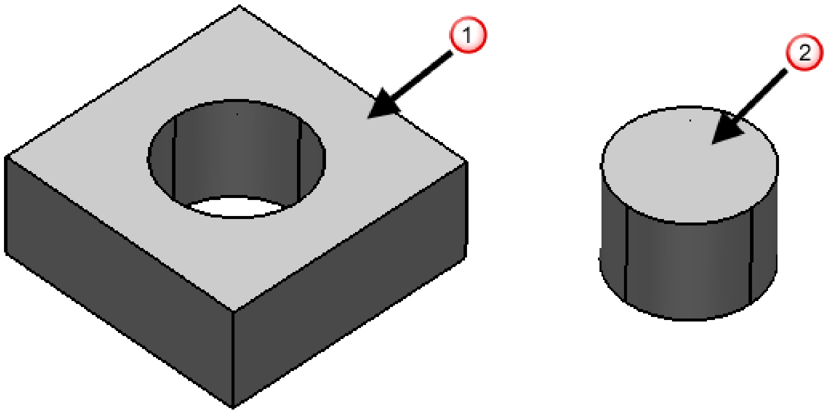
c Use the options on the dialog to edit the cut.

d Click **Apply** to save the changes you have made.

14 When the cut creation is complete, click **Cancel** to close the dialog.

Name - The drop down list contains the names of the sketches. If the feature has more than one sketch, the **Delete** button is active. You can remove a sketch from the feature by selecting a profile from the **Name** drop down list and clicking **Delete**.

Remove inside material - When on, the extruded solids are subtracted from the solid 1. When off, the extruded solids are intersected with the solid 2.



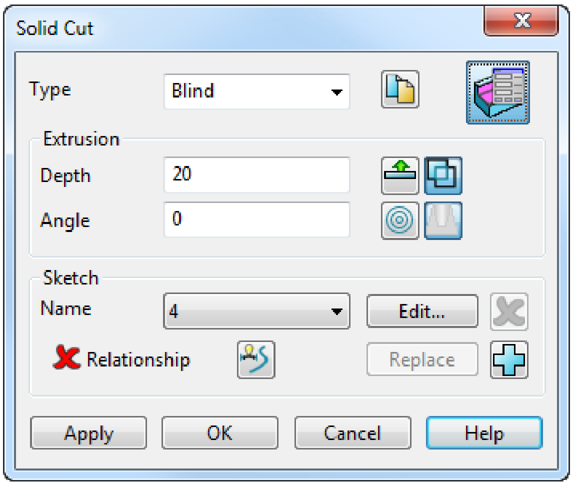
Solid Cut dialog (Edit options)

Additional options are available when you edit a cut feature. You can simultaneously edit multiple cuts (see page 247).

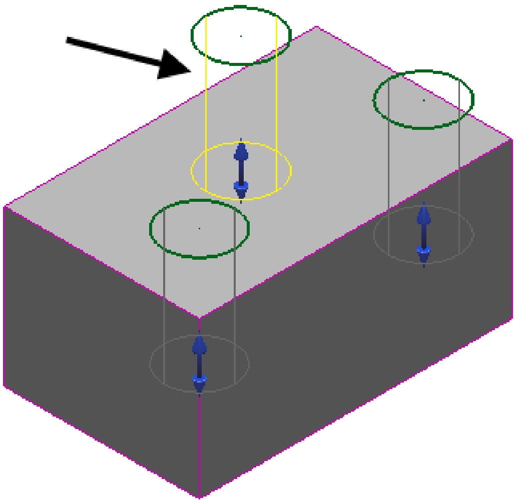
1 Enter editing mode in one of the following ways:

- Double-click the **Cut feature**  icon in the tree to display the **Solid Cut** dialog (see page 124) with the edit options available.
- Double-click the cut feature on the model to display the **Solid Cut** dialog (see page 124) with the edit options available.
- Click  (*Solid feature toolbar*) to display the **Solid Cut** dialog (see page 124). Click  and select the cut to make the edit options available.

- If the **Solid Cut** dialog is already displayed, click  and select the cut. The edit options will become available on the **Solid Cut** dialog.

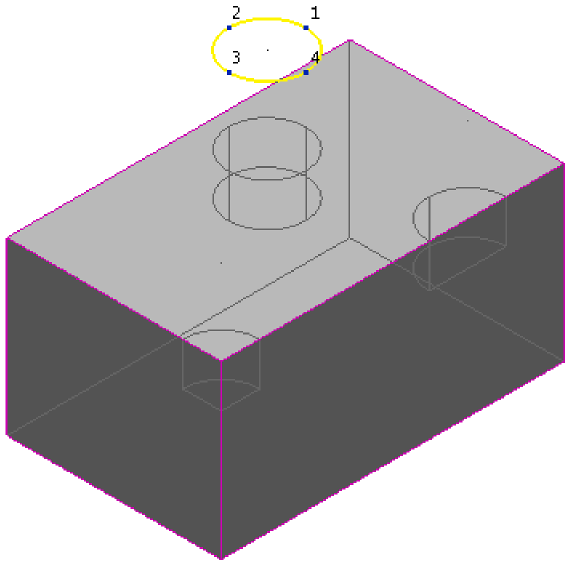


2 Select the wireframe of the existing cut by clicking on it or selecting it from the **Name** drop-down list. The extruded solid of the selected wireframe is highlighted.

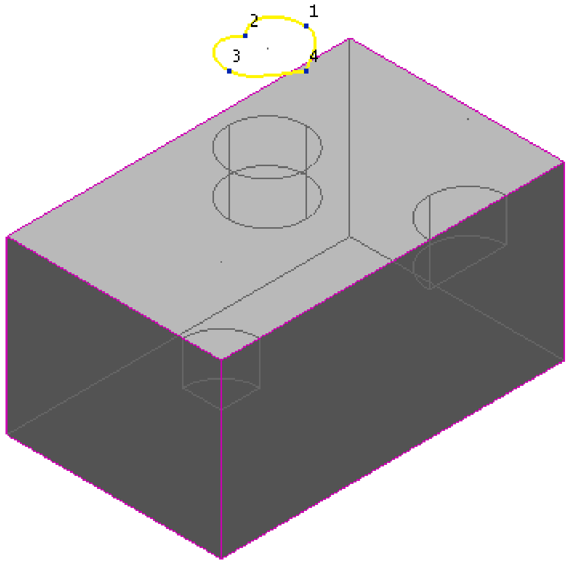


3 **Edit** the wireframe as required:

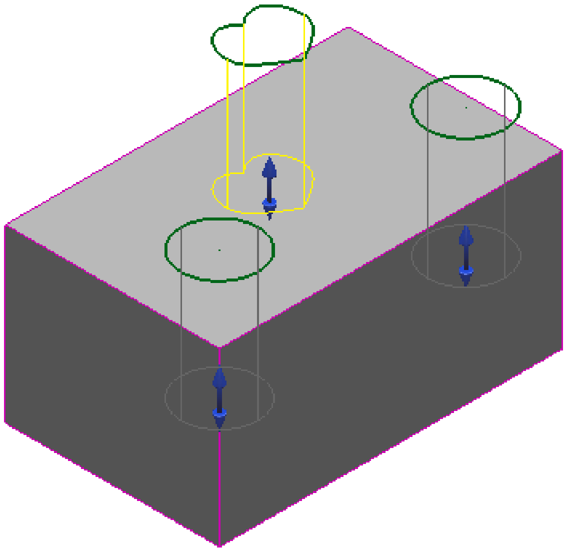
a Click the **Edit** button to make the **Edit curve** toolbar available. The label on the **Edit** button changes to **Finish**. The extruded solid for the selected wireframe is no longer visible.



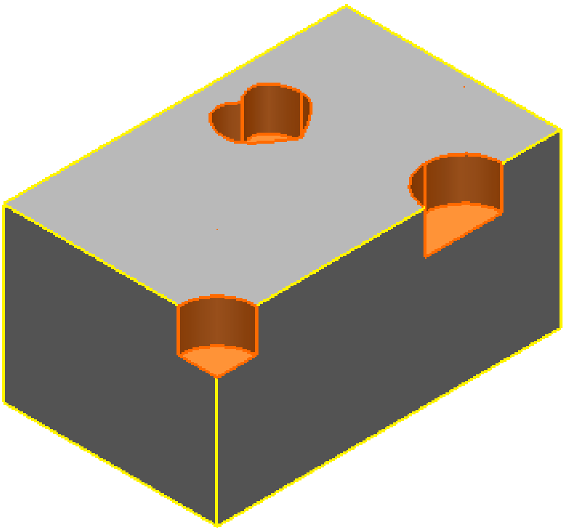
b Edit the wireframe either graphically or using the commands on the **Edit curve** toolbar.



c Once you have finished editing the wireframe, click the **Finish** button.

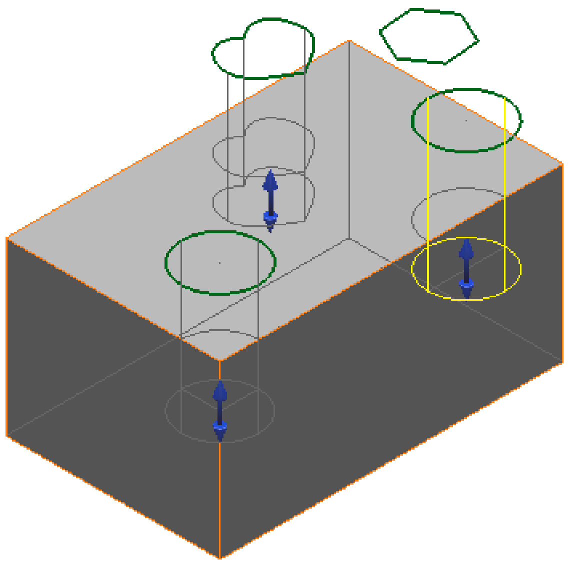


d Click **Apply** to recreate the cut on the solid using the edited wireframe.

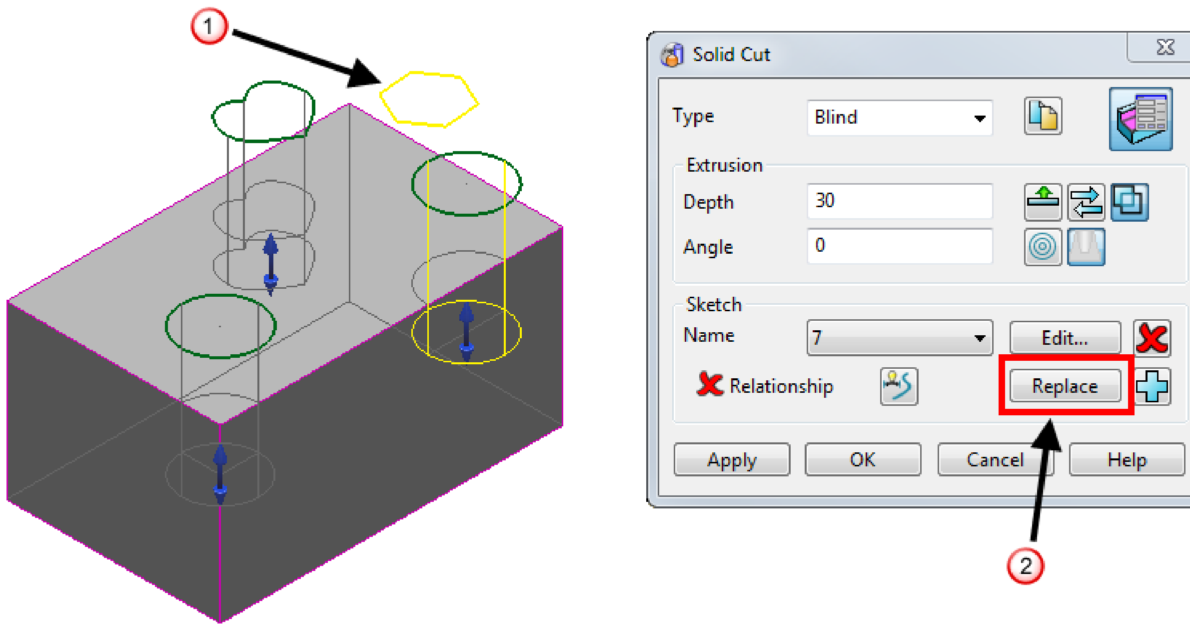


4 If required, **Replace** the selected wireframe of a cut with another piece of wireframe:

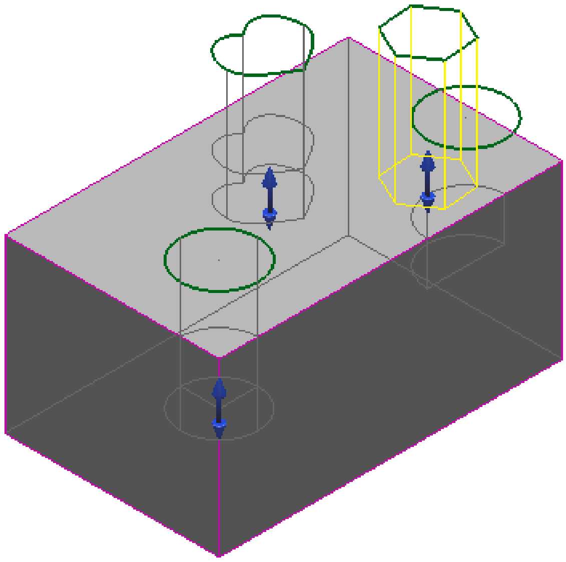
a Select the wireframe of the cut to be replaced.



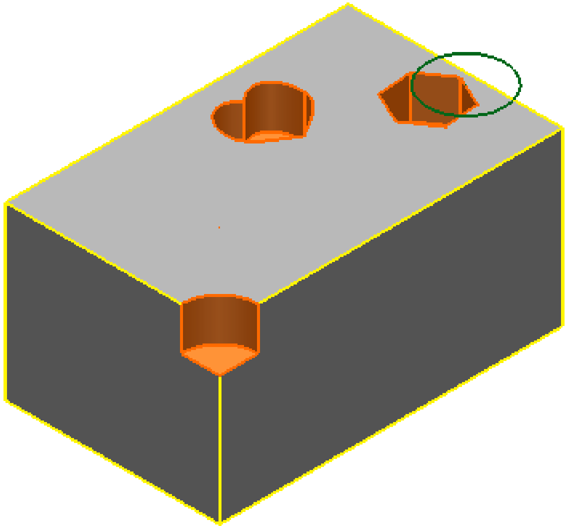
b Select the replacement wireframe 1. The **Replace** button becomes available 2.



c Click the **Replace** button. An extruded solid is created from the replacement wireframe and the extruded solid of the replaced wireframe is deleted. The wireframe of the replaced extruded solid is now visible.



d Click **Apply** to recreate the cut using the replacement wireframe. The wireframe of the replaced extruded solid is left in the model.

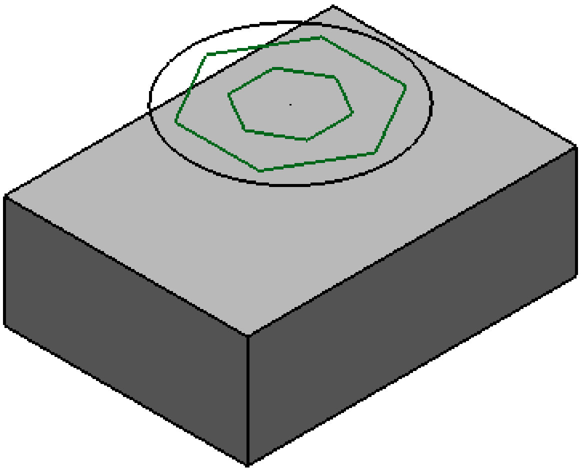


*You can edit the wireframe using the mirror, move, rotate, offset and scale commands on the **Edit** toolbar.*

Example: Creating a solid cut using nested sketches

1 Create a block and ensure that it is active

2 Create a full arc and two polygons in the same plane as the block (the **Create Composite curve** option on the **Polygon** dialog should be *ON*). Your model should similar to the one below.



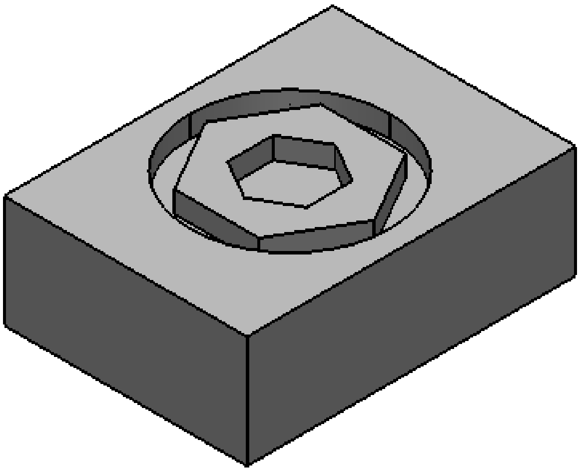
3 Select the block, arc and polygons

4 Click  (*Main toolbar*).

5 Click  (*Solid feature toolbar*) button to display the Solid Cut dialog (see page 124).

6 Click  to **Group sketches into regions**.

7 Click **OK**.



Radial Cut

Use the **Radial Cut** feature to create a pocket in a solid, that can be milled on a turn-mill machine.

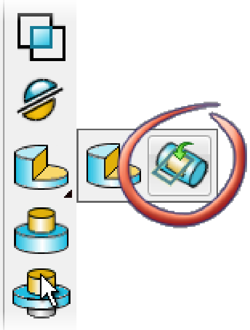
This functionality is available only with PowerSHAPE Pro, PartMaker Modeling, and Toolmaker.

Creating a radial cut feature

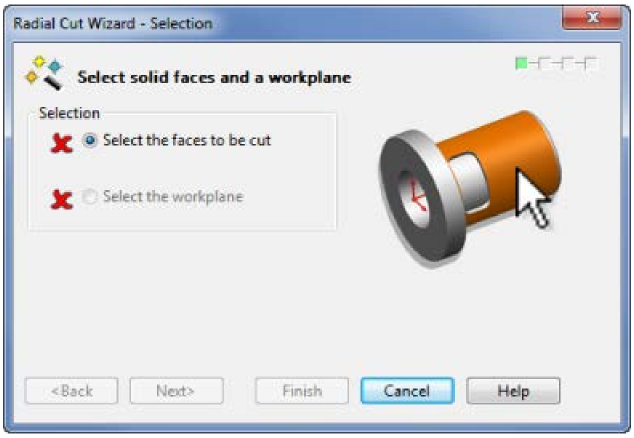
1 Create a workplane on the solid, that is centred along the axis of revolution.

2 Select the solid and click  to display the **Feature** toolbar.

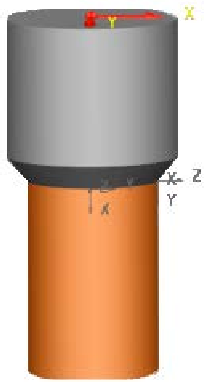
3 Select **Create radial solid cut feature**  from the **Solid cut feature** flyout .



4 The **Radial Cut Wizard** is displayed.



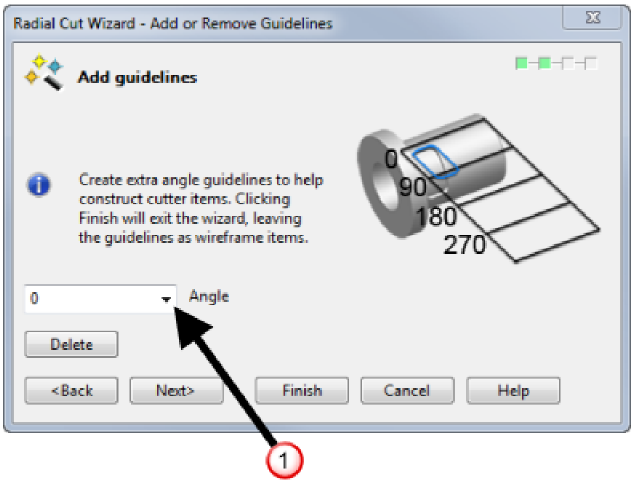
5 Select the cylindrical faces to cut, and the workplane created previously, then click **Next**.



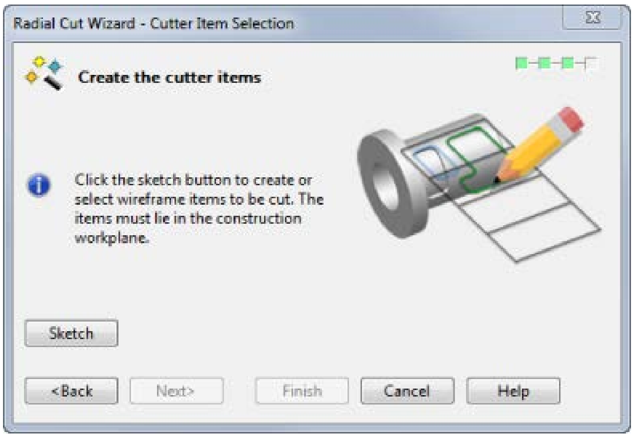
6 Use the **Add or Remove Guidelines** page of the wizard to insert extra angle guidelines.

- Enter an angle into the **Angle** field 1, and press [Enter] to insert a guideline.
- Select an angle from the Angle field 1, and click **Delete** to remove a guideline.

Guidelines are useful as markers when sketching the wireframe used to cut the solid.



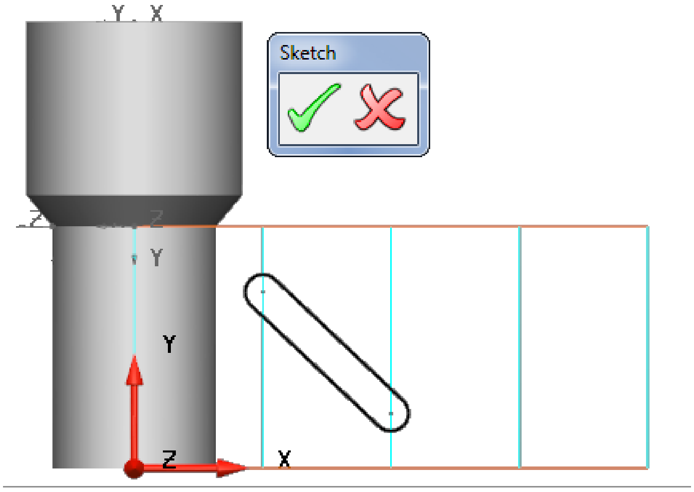
7 Click **Next** to display the **Cutter Item Selection** page of the wizard. Use this page to define the shape of the cutting object.



8 Click **Sketch** to hide the wizard and enter a sketching mode. Use the sketching mode to:

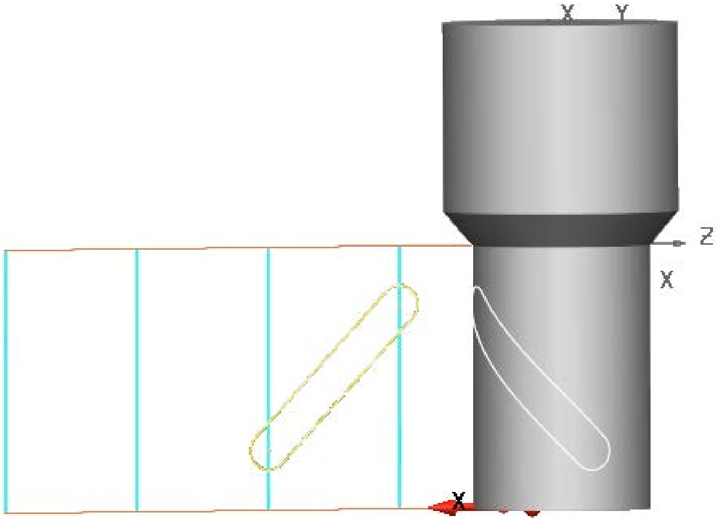
a create new wireframe, where appropriate.

b select new or existing wireframe.



9 Use  to select any wireframe to use as a cutting object.

10 Click  to re-open the wizard and display a projection of the wireframe onto the model. Alternatively click  to discard the wireframe and return to the wizard.



11 Click **Next** to display the **Perform Cut and Wrap** page of the wizard with the following options:

- Select **Cut** 2 to use the wireframe as a cutting object. Specify the cutting distance using:

  Depth — enter a numerical value to define the cutting depth.

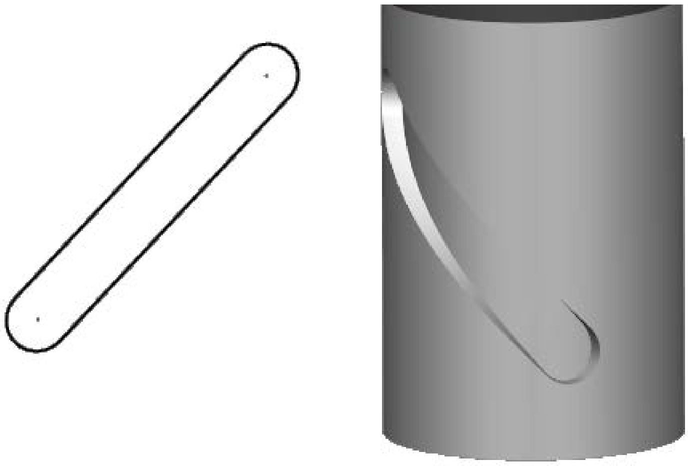
  Through — cut through to the cylinder centre-line.

- Select **Wrap** 3 to draw the wireframe onto the model, without cutting into the solid.

Enter a value into the **Offset** field to move the wrapped wireframe away from the solid.

- Click **Apply** to perform the cut or wrap.
- Click **Remove** to undo a cut or wrap.

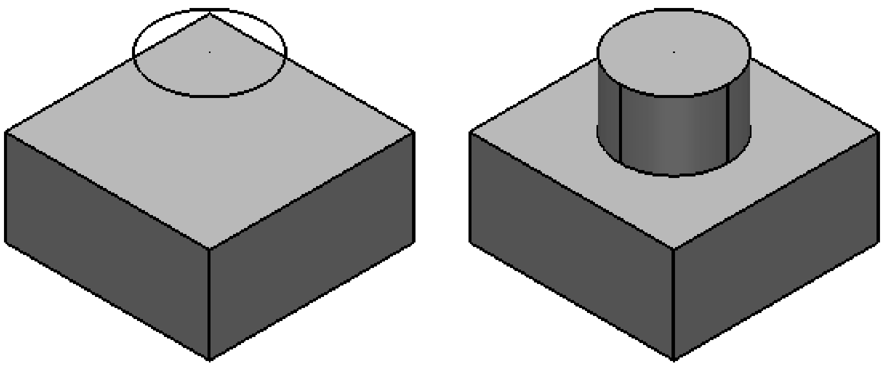
12 Click **Apply** to perform the cut or wrap.



13 Click **Finish**.

Creating a solid boss

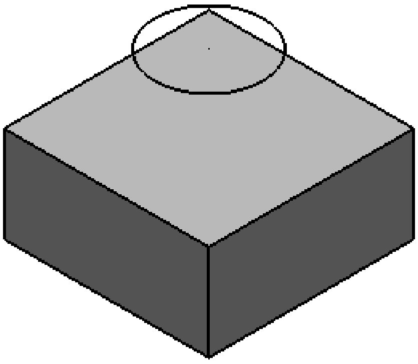
You can add material onto a solid using wireframe objects. Behind the scenes, PowerSHAPE creates extruded solids from the wireframe objects and adds them onto the active solid.



To create a solid boss use one of the following methods:

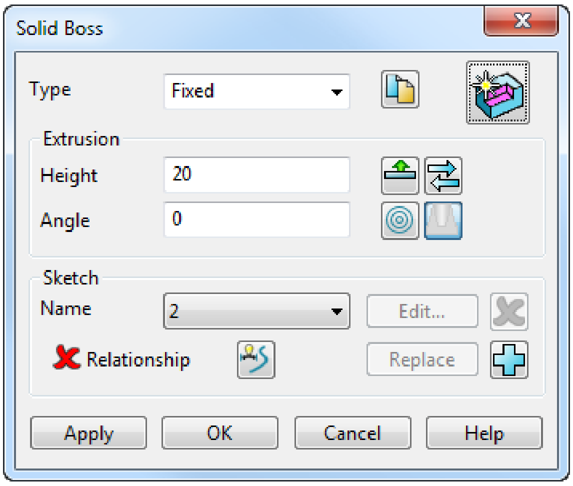
- Make sure you have an active solid and then select the wireframe objects.
- Drag your cursor to select an inactive solid and the wireframe objects together.

The wireframe objects must be closed and planar.



Your selection can contain other objects. PowerSHAPE filters the valid wireframe from the selection.

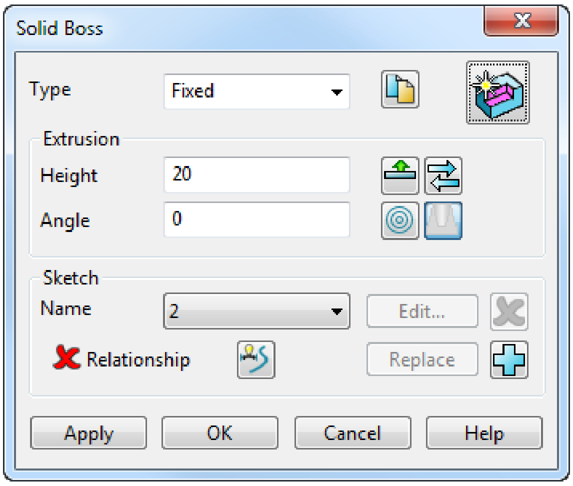
1 Click (*Solid feature toolbar*).



2 Use the dialog to create bosses onto the active solid using the wireframe objects.

Solid Boss dialog

Use this dialog to add solid bosses a solid, using selected wireframe.

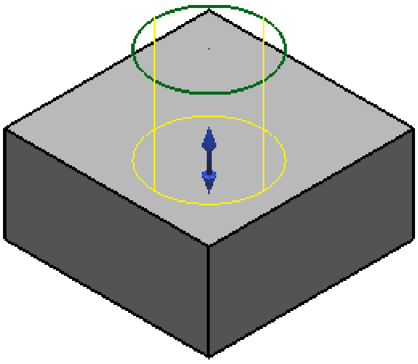


You can add a solid boss to:

- The active solid.
- An inactive solid that has been preselected with the wireframe.

Each wireframe object is extruded to create the extruded solids. The direction of extrusion is defined by the position of the composite curve.

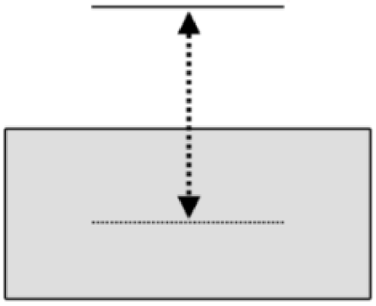
The extruded solid has a handle, which edits the length of all the extruded solids.



1 Select **Type** from the drop-down list.
   - If you select **Automatic**, the wireframe will be extruded just far enough to join to the solid. You can specify the angle of the extrusion using the **Angle** option.
   - Selecting **Fixed** lets you specify the **Height** of the extrusion as well. The extruded wireframe is added to the solid. The **Extrude both directions** option also becomes active if **Fixed** is selected.

2 Click  to create a copy of the original wireframe used in the construction of the boss.

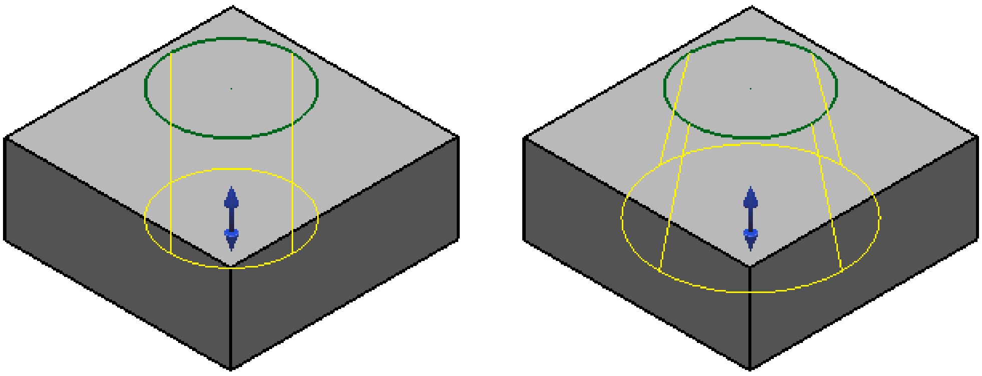
3 Enter the **Height**. This is the length of each extruded solid, measured from the plane of the wireframe.



If you drag the handle of an extruded solid, the lengths of all the extruded solids change. The value of **Height** in the dialog updates to reflect the change.

To create bosses all the way through the solid, make sure the height extends beyond the base of the solid.

Enter the **Angle** to define a draft angle on the extruded solids.



To reverse the direction, drag the handle on the extruded solids to the opposite side of the wireframe.

4 Click  to extend the extrusion in both directions from the sketch.The button changes to  to show that the extrusion will be extended in both directions.

5 Click  to reverse the direction of the boss.

6 Click  to display the **Solid Feature Relationship** dialog (see page 251). This allows the relative position of the feature to be defined with respect to the solid. The current status of the **Relationship** is indicated by one of the following:

 no relationship is currently defined.

 a valid relationship is defined.

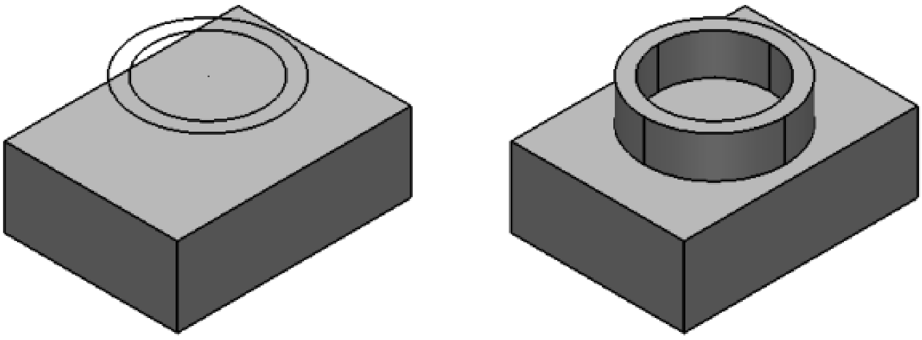
 relationship is defined, but there is a problem with the definition

7 Click to add the selected sketch (arc, curve or composite curve) to the feature.

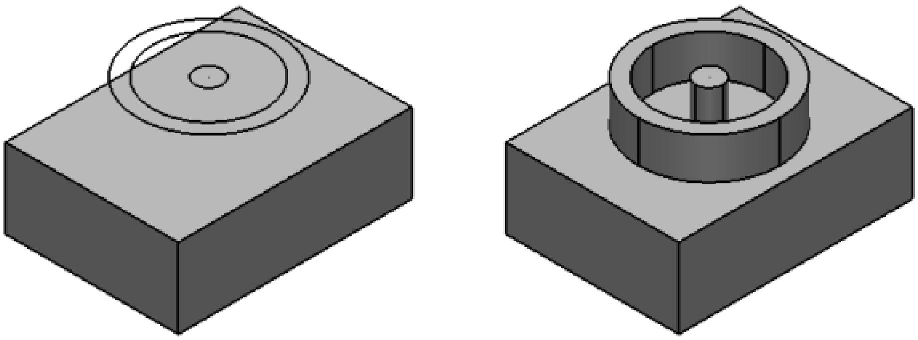
8 Click to group nested sketches (see page 147).

If this option is selected, any sketches that lie on the same plane are nested and are used to produce an island-type extrusion.

- Using two concentric circular sketches would produce a tube with a thickness.



- Using three concentric circular sketches, the tube would have a cylinder inside it.



If this option is deselected, only the selected wireframe will be used to produce the solid boss.

9 Click or to specify if the draft angle is to go in or out.

10 Click **Apply** to create the boss, leaving the dialog displayed. This lets you:

- create another boss.
- edit the existing boss.

Alternatively, click **OK** to create the boss and close the dialog.

A **Boss feature** icon representing the operation appears in the solid feature tree.

11 Make changes to the boss as required:

a Click to switch to **Edit** mode. The button changes to .

b Click the boss.

c Use the options on the dialog to edit the boss (see page 143).

d Click **Apply** to save the changes you have made.

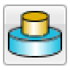
12 When boss creation is complete, click **Cancel** to close the dialog.

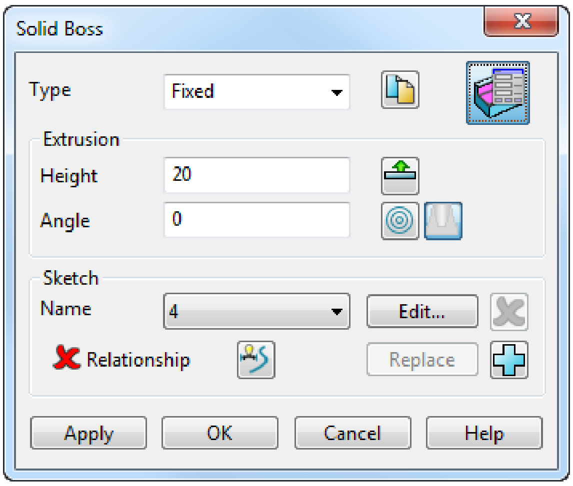
Solid Boss dialog (Edit options)

You can edit an existing solid boss or simultaneously edit multiple bosses (see page 247).

To edit a single solid boss:

1 Enter editing mode in one of the following ways:

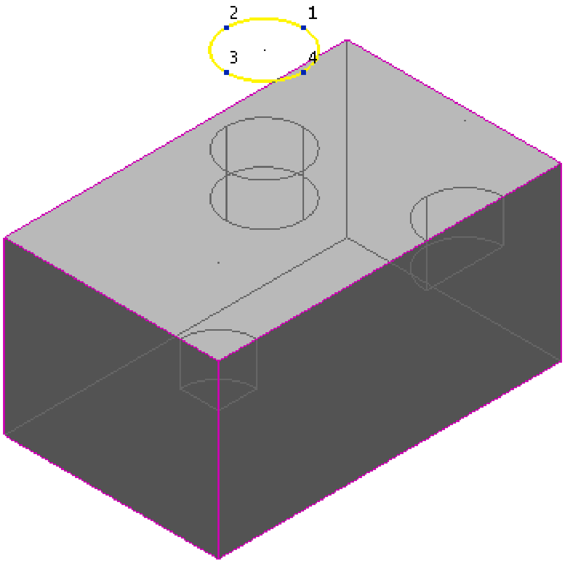
- Double-click the **Boss feature**  icon in the tree to display the **Solid Boss** dialog (see page 143) with the edit options available.
- Double-click the boss feature on the model to display the **Solid Boss** dialog with the edit options available.
- Click  (*Solid feature toolbar*) to display the **Solid Boss** dialog (see page 143). Click  and select the boss to make the edit options available.
- If the **Solid Boss** dialog is already displayed, click  and select the boss. The edit options will become available on the **Solid Boss** dialog.



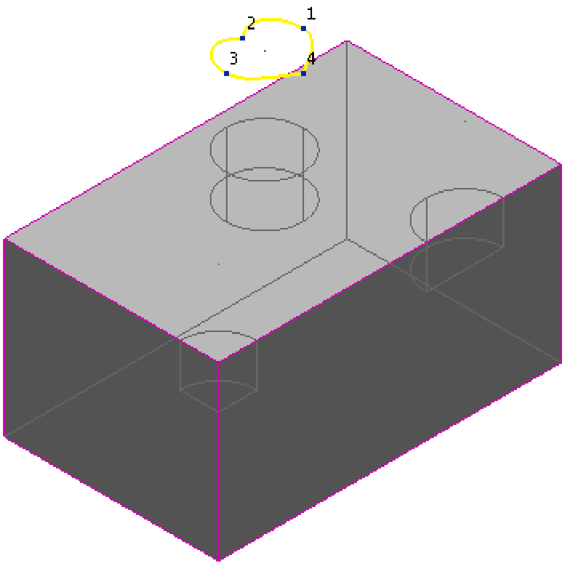
2 Select the wireframe of the existing boss by clicking on it or selecting it from the **Name** drop-down list. The extruded solid of the selected wireframe is highlighted and all the others are greyed.

3 **Edit** the wireframe as required:

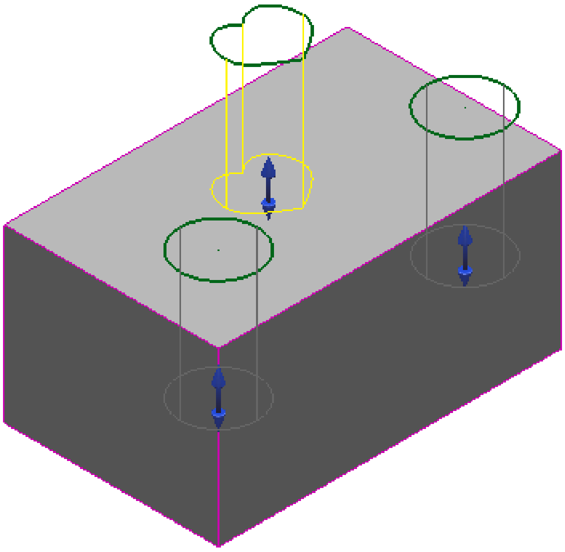
a Click **Edit** button to make the **Edit curve** toolbar available. The label on the **Edit** button changes to **Finish**. The extruded solid for the selected wireframe is no longer visible.



b Edit the wireframe either graphically or using the commands on the **Edit curve** toolbar.



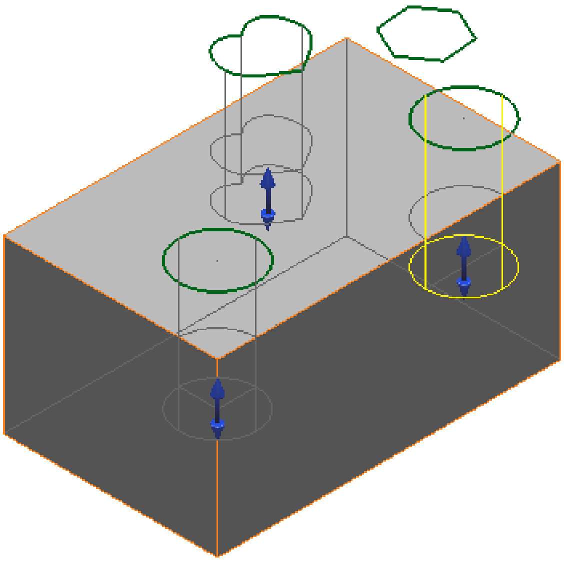
c Once you have finished editing the wireframe, click the **Finish** button.



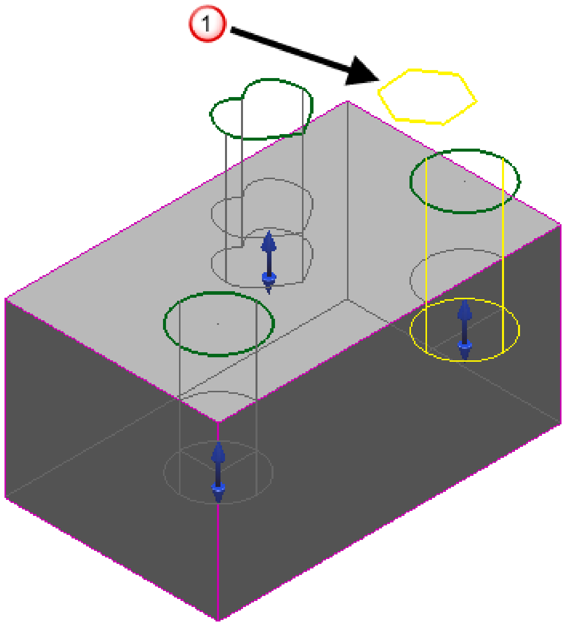
d Click **Apply** to recreate the boss on the solid using the edited wireframe.

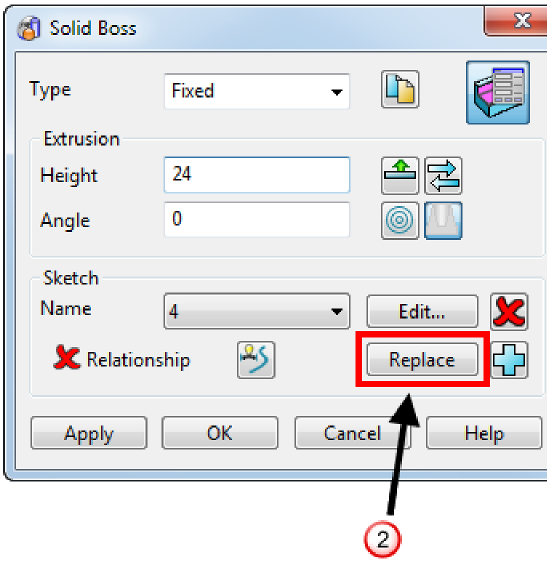
4 If required, **Replace** the selected wireframe of a boss with another piece of wireframe:

a Select the wireframe of the boss to be replaced.

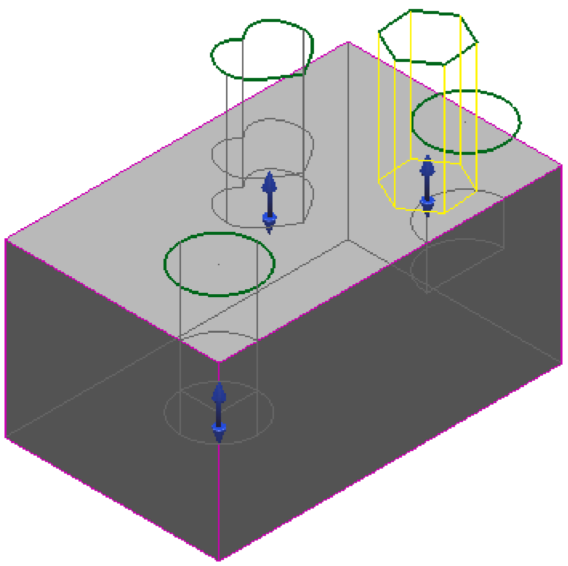


b Select the replacement wireframe 1. The **Replace** button becomes available 2.





c Click the **Replace** button. An extruded solid is created from the replacement wireframe and the extruded solid of the replaced wireframe is deleted. The wireframe of the replaced extruded solid is now visible.

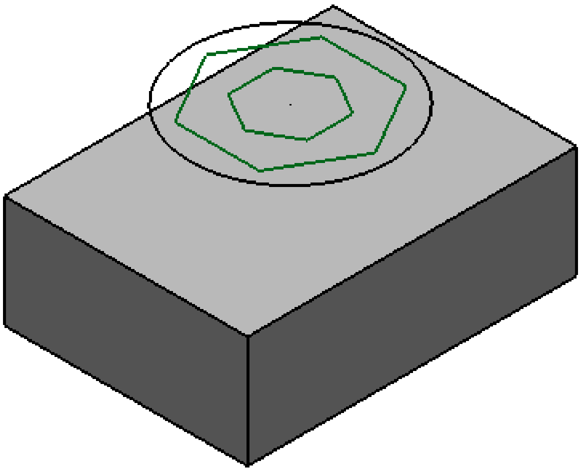


d Click **Apply** to recreate the boss using the replacement wireframe. The wireframe of the replaced extruded solid is left in the model.

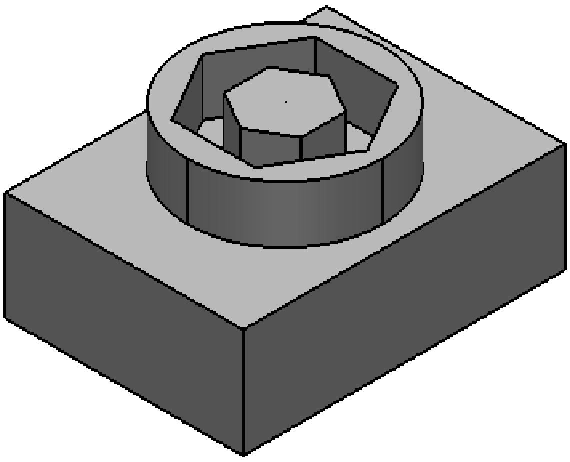
You can edit the wireframe using the mirror, move, rotate, offset and scale commands on the ***Edit*** *toolbar.*

Example: Creating a solid boss using nested sketches

1. Create a block and ensure that it is active
2. Create a full arc and two polygons in the same plane as the block (the **Create Composite** curve option on the **Polygon** dialog should be selected).



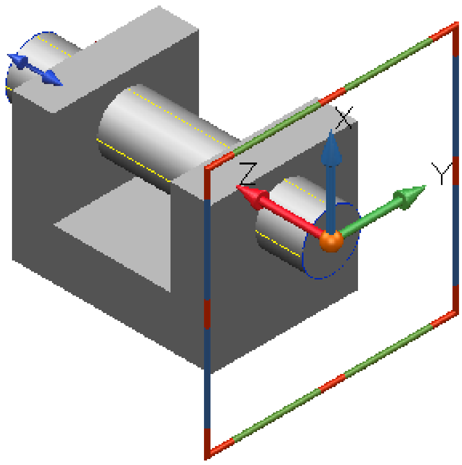
3. Select the block, arc and polygons
4. Click  (*Main toolbar*) to display the Solid Feature toolbar.
5. Click  (*Solid feature toolbar*) button to display the **Solid Boss** dialog.
6. Click  to **Group sketches into regions**.
7. Click **OK.**



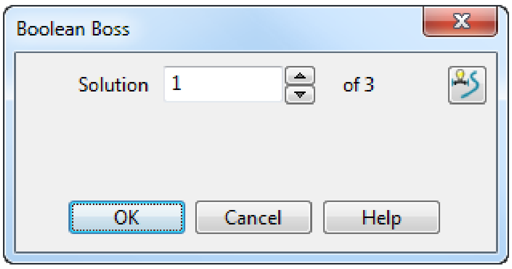
Boolean boss

Use Boolean boss functionality to create a Boolean addition between the active solid and another solid; the other solid will not pass all the way through the active solid. The feature tree will be updated to include a sub-branch to the boss feature showing the history of the secondary solid.

1 Select the solid to be added to the active solid.



2 Click  (*Solid feature toolbar*) to display the **Boolean Boss** dialog. (see page 148)



3 Use the options on the dialog to create the Boolean boss feature.

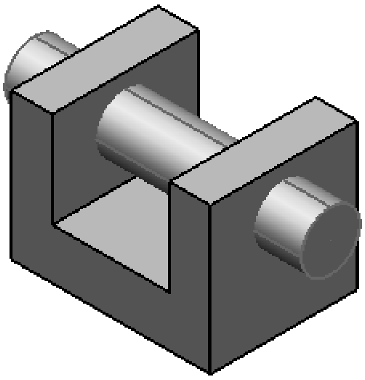
4 Select **OK**.

5 Edit the Boolean boss (see page 150) as required.

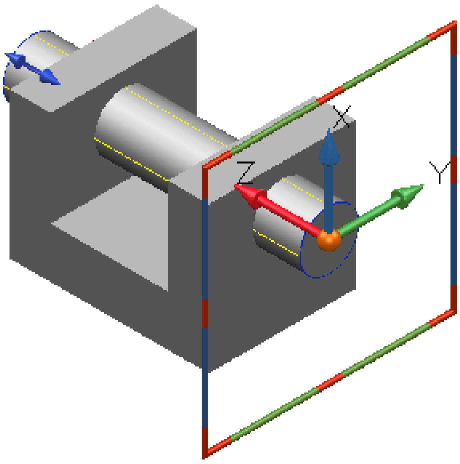
Creating a Boolean boss

Create a Boolean addition between the active solid and another solid; the other solid will not pass all the way through the active solid. The feature tree will be updated to include a sub-branch to the boss feature showing the history of the secondary solid.

In the example, the cylinder will be added to the active solid.



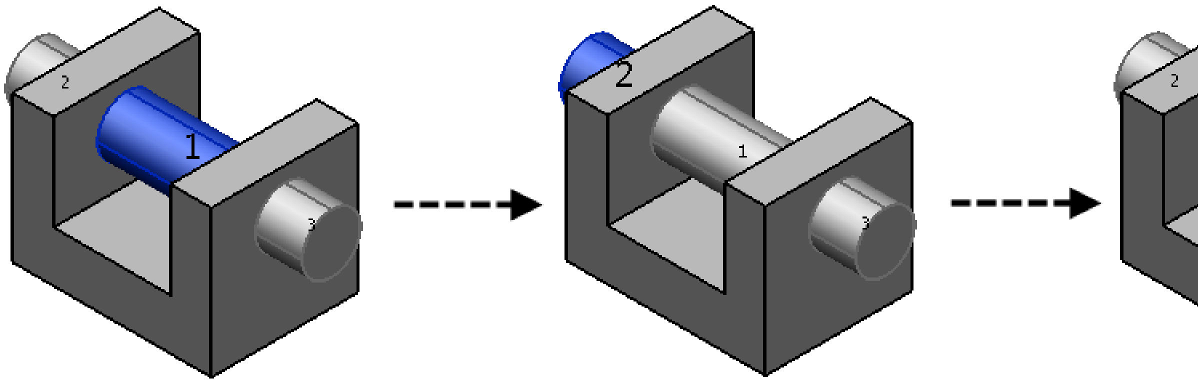
1 Select the solid to be added to the active solid.



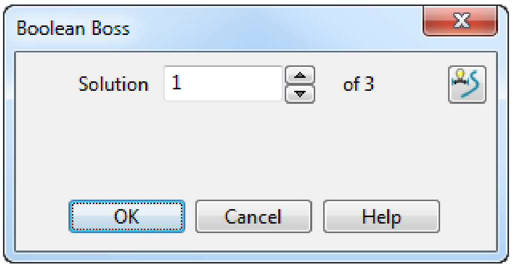
2 Click  (*Solid feature toolbar*).

A Boolean addition will add the selected solid to the active solid without making it pass all the way through the active solid.

If more than one solution is available, all possible solutions are displayed on the model. The faces of the current solution are highlighted and the number of the current solution is displayed prominently. Additional solutions are also numbered.



3 Scroll through the possible solutions using the **Boolean Boss** dialog, or click on one of the numbers to highlight that solution and update the **Boolean Boss** dialog.



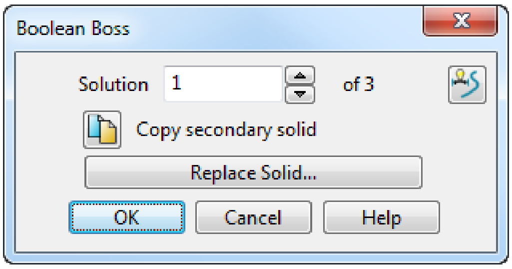
4 Define the relationship of the feature during creation using the **Solid Feature Relationship** button. For further details, see Creating a relationship between a solid feature and a solid (see page 249).

5 Select **OK**.

Editing a Boolean boss

1 Enter editing mode in one of the following ways:

- Double-click the **Boss feature** icon in the tree window to display the **Boolean Boss** dialog with the edit options available.
- Double-click the boss feature on the model to display the **Boolean Boss** dialog with the edit options available.

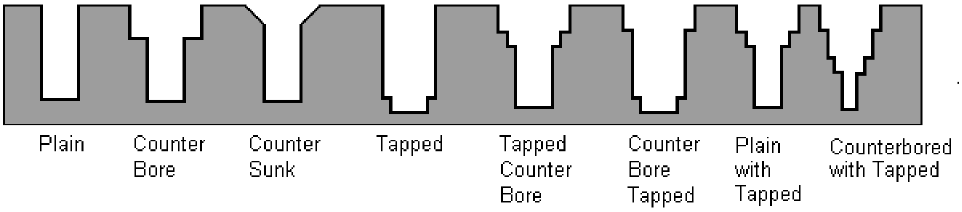


2 Use the following options to make changes to the boolean boss:

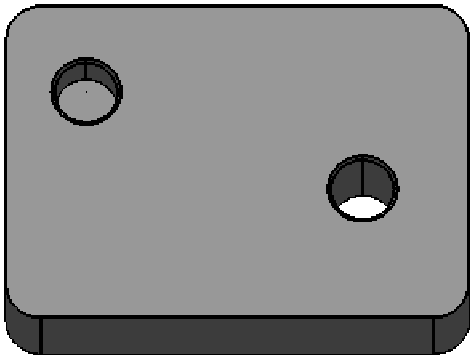
- Use the up and down arrows to choose the **Solution** to edit.
- Click to create a copy of the selected solid used in the feature. When you select **OK**, the entire sub-branch (the solid and its history) will be copied. If the sub-branch only contains one primitive feature, the new solid will be a primitive solid.
- Select **Replace Solid** to replace the solid with another one. The **Replace Solid** dialog will be displayed.

Creating a solid hole

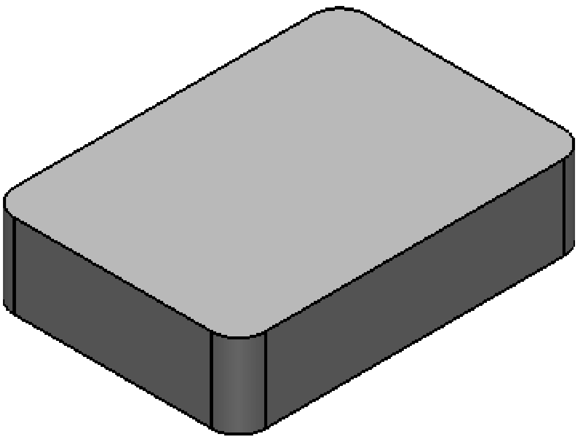
You can add the following types of holes to a solid by specifying the use of the hole.



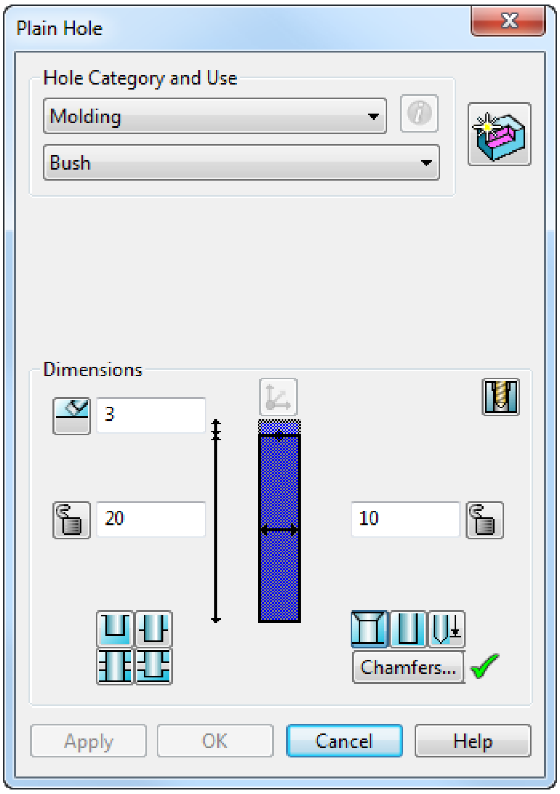
You can create holes in or through a solid:



1 Create a solid or open a model with a solid:



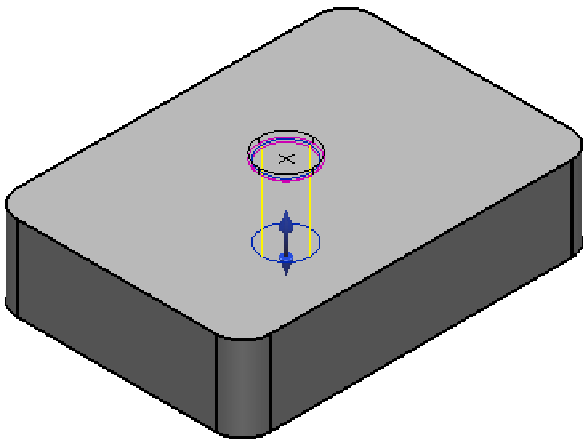
2 Click  (*Solid feature toolbar*) to display the Plain Hole dialog (see page 154).



3 Use the options on the Plain Hole dialog to create a hole (see page 154).

4 Click a position on the solid where you want to create the hole.

A shape representing the hole is drawn on the solid.



5 To finish creating the hole, click one of the following:

- **Apply** to create the hole and continue to display the dialog.

When you use **Apply**, you are still in hole creation mode. You can click the solid again to add another hole. The new hole will automatically use the same definition as the last one created and the **Hole Feature** dialog is displayed for you to modify the hole.

The next hole will be created using the default settings, unless you turn on the Reuse Last Hole Definition option on the Holes page of the Options dialog.

- **Cancel** to remove the dialog from the screen without operating on the solid.
- **OK** to create the hole as defined and remove the dialog from the screen.

A **Hole feature** icon representing the operation appears in the solid feature tree. After 20 holes are created in the solid, you are asked by default whether you want to optimise the tree (see page 278).

6 Make changes to the hole as required:

a Click to switch to **Edit** mode. The button changes to .

b Click the hole.

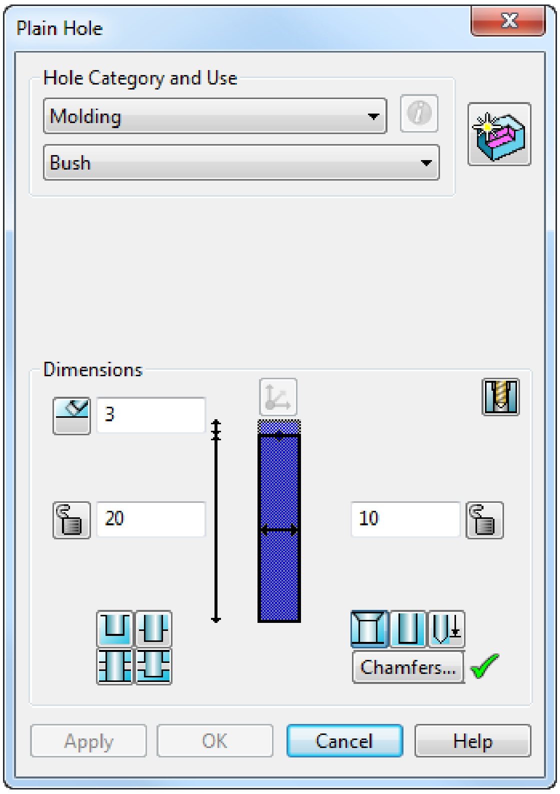
c Use the options on the dialog to edit the hole.

d Click **Apply** to save the changes you have made.

Set the Hole Export options on the Manufacturing page of the Options dialog to export the holes as PowerMILL hole features and automatically generate their toolpaths .

Plain Hole dialog

Use this dialog to define the hole that is cut into the solid.



1 Select the **Hole Category** from the drop-down list:

- **General Machining (ISO)**

  Precision Machining (ISO)

  Molding

  Untoleranced

2 Select the specific **Use** of the hole for the category you have chosen.

  With the exception of untoleranced holes, machining tolerances based on the intended use are also stored when the hole is created.

   *You can add your own hole categories and uses on the **Holes** page of the **Options** dialog.*

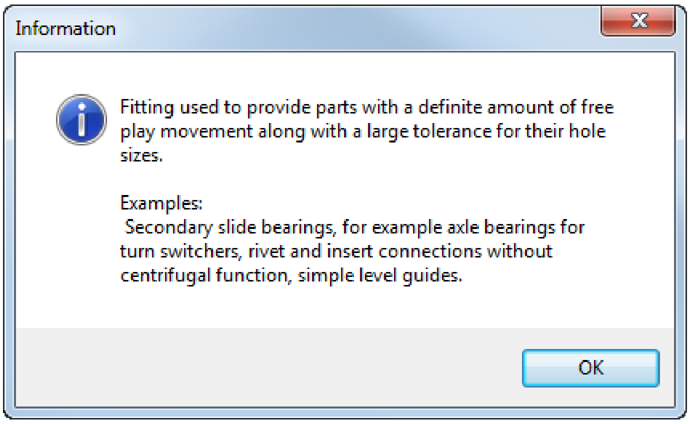
  The options that are displayed on the dialog reflect the **Hole Category** and **Use** that you have chosen.

3 Use  to swap between creation mode and editing mode without closing/reopening the dialog.

- When in creation mode, click  to swap to editing mode and recognition mode.

- When in editing mode, click to swap to creation mode.

4 If available, click to display information about the selected hole use. This is a general description on usage and an example of the application.



If available, click to display the Plane Details dialog (see page 162). This option is only available once you have positioned the hole.

5 When is displayed, the hole is considered for machining in PowerMILL. Clicking changes the button to .

When is displayed, the hole is ignored when exporting to PowerMILL.

6 Use the following sections to set type-specific options for:

- Counterbored holes
- Tapped holes (see page 160)
- Countersunk holes (see page 161)

7 Use the Dimensions (see page 156) options to define the size and trim attributes.

8 To finish creating the hole, click one of the following:

- **Apply** — Create the hole and keep the dialop open.

  When you use **Apply**, you are still in hole creation mode. You can click the solid again to add another hole. The new hole will automatically use the same definition as the last one created and the **Hole Feature** dialog is displayed for you to modify the hole.

  The next hole will be created using the default settings, unless you turn on the Reuse Last Hole Definition option on the Holes page of the Options dialog.

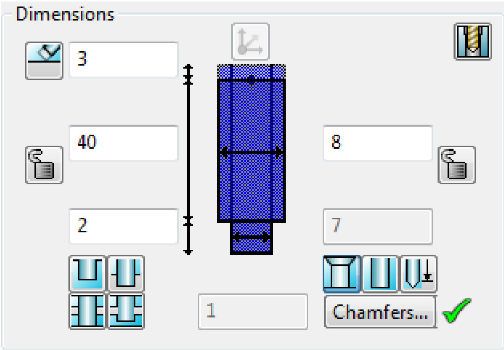
- **Cancel** — Close the dialog without operating on the solid.

- **OK** — Create the hole as defined and close the dialog.

*If changing a value on the **Hole** dialog causes the definition of the hole to be invalid, the status bar help for the greyed out **Accept** and **Apply** buttons now indicates that the values are incorrect.*

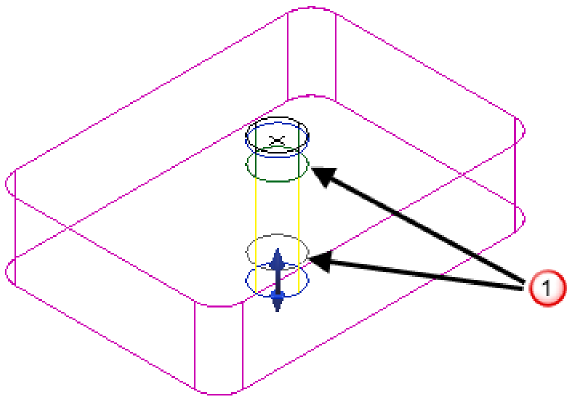
Dimensions

For each type of hole, an image of a hole is displayed in the dialog.



1. Change the dimension in one of the following ways:
   - Drag the edit handles on the hole. It is only possible to use this method if you have already positioned the hole on the solid.
   - Enter values in the dimension boxes.
2. Use the following buttons to specify the way the hole is extended or limited:

    - If selected, the hole cuts all the way through the solid. On the screen, the length of the hole extends beyond the solid. Safety margins are drawn at both ends of the hole.

    - If selected, the length of a hole that is longer than the thickness of the solid is limited to that thickness. If deselected, the length of a hole is not limited.

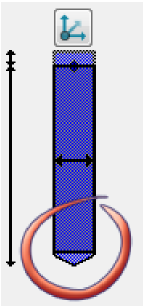
limits the length of the through hole to the final forward intersection. If you want to limit the length of the through hole to the first intersection, click to display .

limits the backward length of the hole to the final intersection. If you want to limit the length of the backward hole to the first intersection, click to display .

3. Use the following buttons to specify how the depth is measured and the end of the hole:

   - If selected, the depth of the hole is measured to the shoulder of the hole. If deselected, the depth of the hole will be measured to the tip of the cone. If this button is selected, the **Display the end of the hole** button becomes inactive.

   - If selected, a cone will be displayed at the end of the hole.

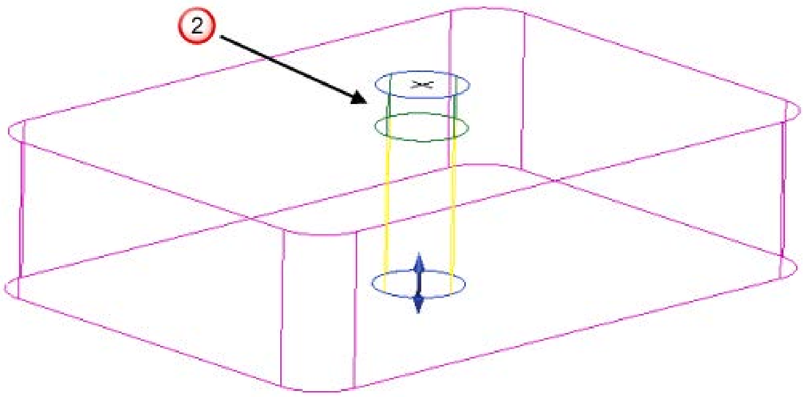
   If is displayed, chamfers are added to the hole sections as defined in the Chamfer Hole dialog (see page 165). Click Chamfers... to display the Chamfer Hole dialog (see page 165).

   If no chamfers are required, click . This removes ✓ and no chamfer is displayed.

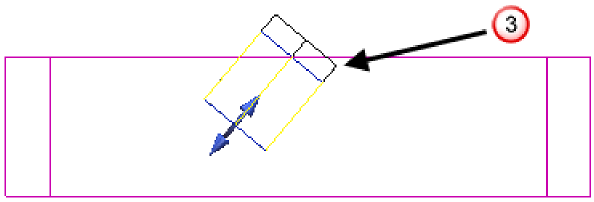
4. Use the following option to specify the safety margin:

   - If selected, safety margins are set to an automatically generated value that is large enough to ensure that the hole completely passes through any surface intersections. The offset safety distance text box is inactive. If deselected, set the offset safety distance manually.

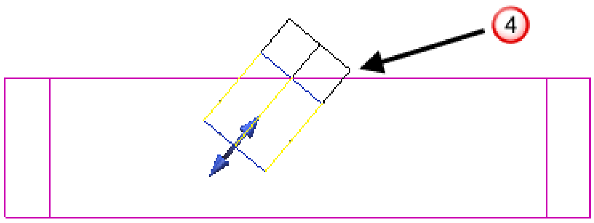
The safety margin is the length above the hole which ensures that the hole cuts into the solid 2. If you can see that the safety margin lies inside the solid, increase it.



In the figure below, the safety margin lies inside the solid 3.



The safety margin value is increased so that it lies outside the solid 4.

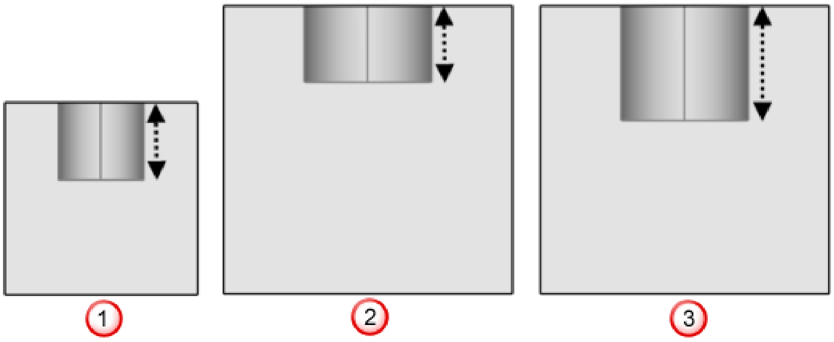


5 Use to lock and unlock the depth and the diameter.

 indicates that the dimension is locked. Click to unlock.

 indicates that the dimension is unlocked. Click to lock.

- If the depth is locked, the depth of the hole does not change when the whole solid is uniformly scaled from the **Edit** toolbar. If the depth is unlocked, the depth is scaled too.



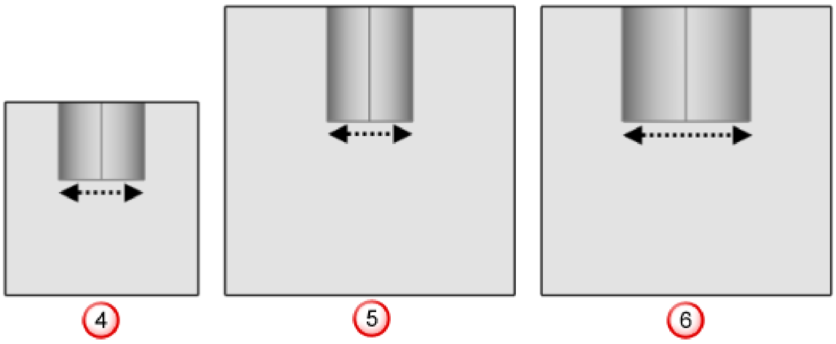
1 Unscaled solid shows the original depth.

2 Scaled solid with locked depth shows the original depth.

3 Scaled solid with unlocked depth shows the scaled depth.

If you non-uniformly scale the solid or define the depth of the hole using parameters, the depth will not be scaled regardless of whether it is locked or unlocked.

- If the diameter is locked, the diameter of the hole does not change when the whole solid is uniformly scaled from the **Edit** toolbar. If the diameter is unlocked, the diameter is scaled too.



4 Unscaled solid shows the original diameter

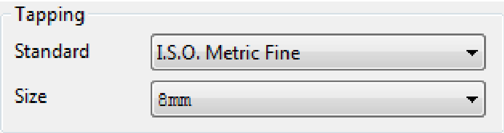
5 Scaled solid with locked diameter has original diameter

6 Scaled solid with unlocked diameter has scaled diameter

If you non-uniformly scale the solid or define the diameter of the hole using parameters, the diameter will not be scaled regardless of whether it is locked or unlocked.

Tapped holes

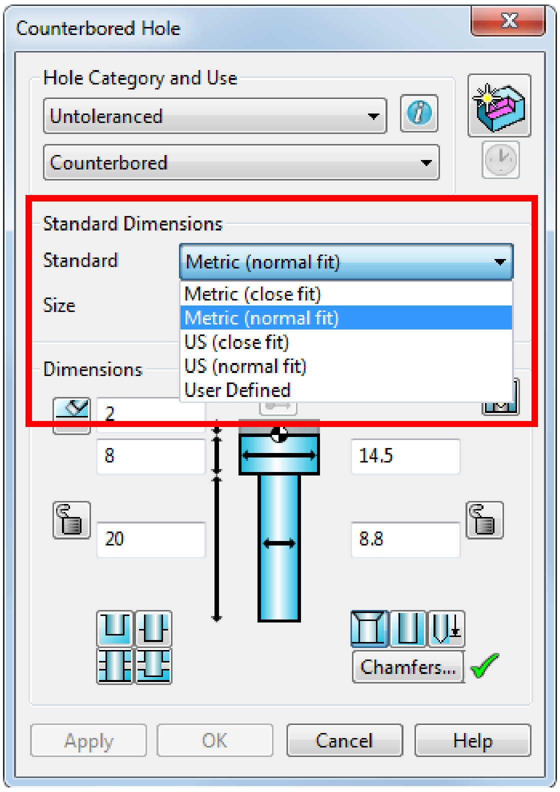
If the hole is tapped, the **Tapping** options are displayed.



1. Select the **Standard** from the drop-down list.
2. Specify the **Size** in one of the following ways:
   - Select the size from the drop-down list. The size of the pre-defined tapped holes correspond to a drill size in the appropriate units.
   - Select **User Defined** from the **Standard** menu and enter your own values in the dimension boxes.
   - Use a parameter to define the diameter of a tapped hole. The hole is set to the tapping size that is closest to the value of the parameter.

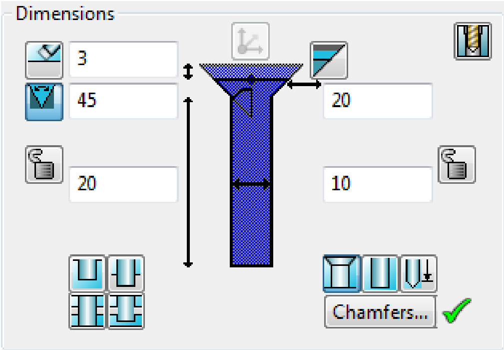
Counterbored holes

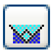
Use the **Standard Dimensions** section for **Counterbored** holes to set the dimensions to match standards that fit screw sizes easily.



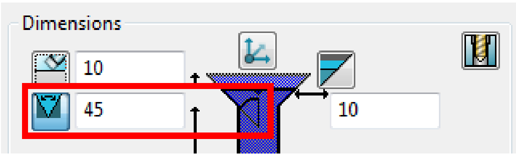
Countersunk holes

If you are using countersunk hole, there are additional options to define the countersink angle and diameter.

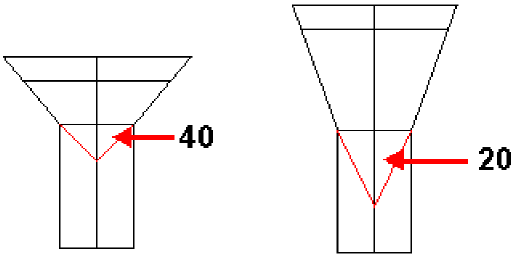


 - This option toggles between  and .

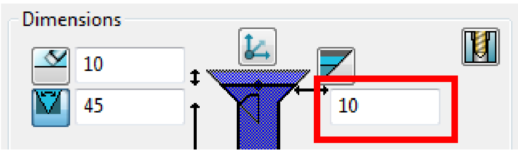
If  is displayed, the hole has a user-defined countersink angle. You can change the angle of the countersink by entering a different value for the angle of the countersink cone.

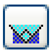


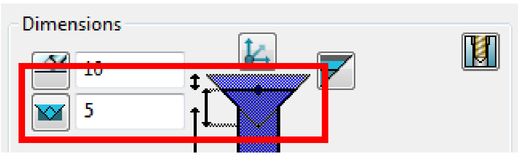
The diagrams below show the effect of entering different angles for the countersink, whilst keeping the other values constant.



You can also change the maximum diameter of the countersink by entering a different value as shown below.

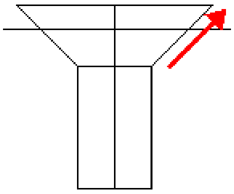


If  is displayed, the hole has a fixed 45 degree countersink angle and the option to define the maximum diameter of the countersink is not available. You set the size of the countersink by entering countersink size in the adjoining box.

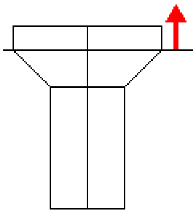


- This option toggles between and .

If is displayed, the safety margin is projected out of the hole at the countersink angle.

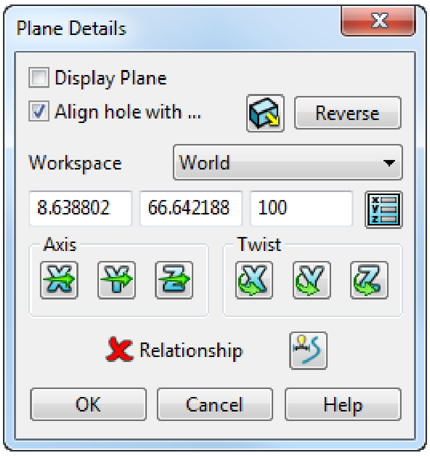


If is displayed, the safety margin is projected straight out of the hole.



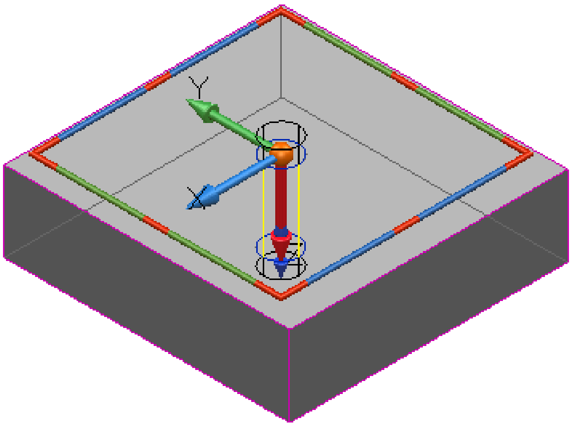
Plane Details dialog

This sets the plane details on the hole.



1 Click displays the **Solid Feature Relationship** dialog. This lets you define the relative position of the hole with respect to the solid. A cross is displayed by default. A tick is displayed if a valid relationship is defined.

2 Select **Display Plane** to display the workplane handles on the hole.



3 Select **Align hole with ...**to rotate the hole to match the selected alignment option:

 aligns hole to the solid face.

 aligns hole to the active workplane.

4 Click 

to reverse the hole direction and origin to the opposite side of the solid.

5 Use the **Plane options** to:

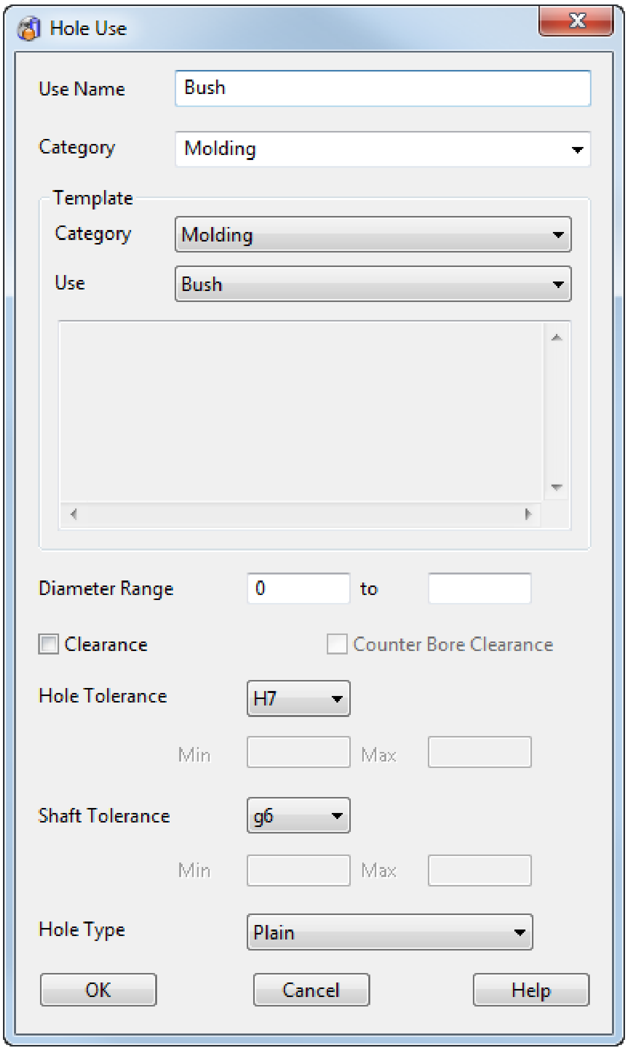
- Move the origin of the hole. Enter the **X Y Z** coordinates or click the **Position**  button to open the **Position** dialog where you can use position entry tools.
- Change the direction of its axis (**Axis** settings)
- Twist it about its axis (**Twist** settings)

6 Click one of the following:

- **OK** to set the plane details for the hole and close the dialog.
- **Cancel** to ignore any changes made to the plane and close the dialog.

Hole Use dialog

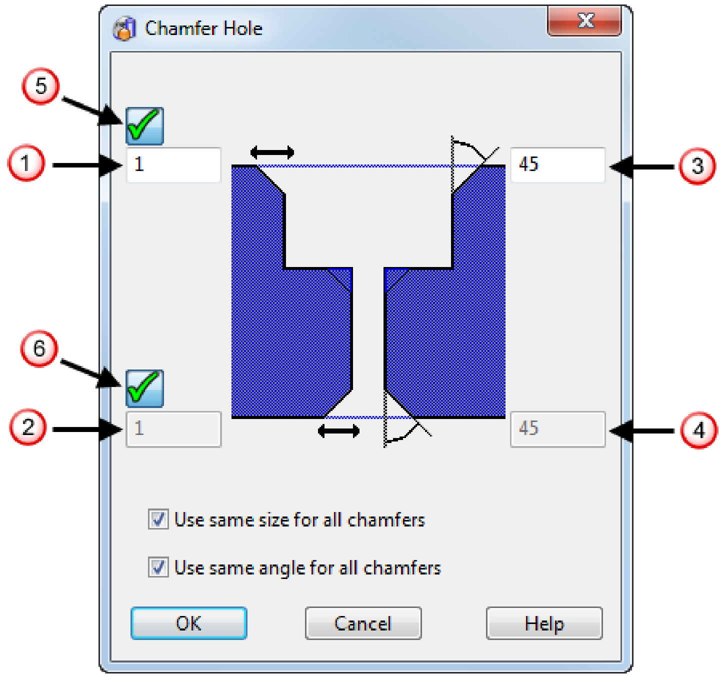
This dialog is used to create and edit hole uses. It is displayed by clicking **Create** on the **Holes** page of the **Options** dialog.



1. Enter a **Name** for the hole use. The name must not already exist within the selected category.
2. Enter the name of a **Category**. You can select an existing category or enter in a new name.
3. Define the **Template** using the **Category** and **Use** drop-down lists to select an existing hole use. This hole use is used to set the values (mainly the tolerances) for the rest of the dialog.
4. Change the settings on the remaining options to define the new hole use.

Chamfer Hole dialog

Use this dialog to define the chamfer angle and size for a hole.



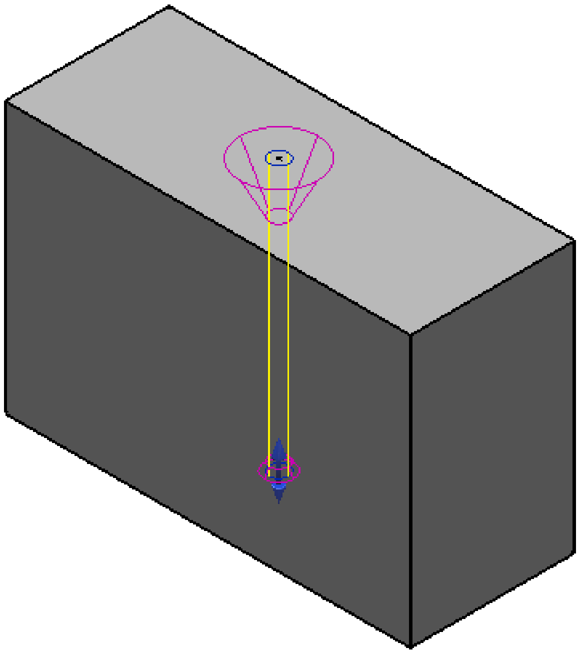
1 and 2 - Enter size of the chamfers on the hole. If **Use same size for all chamfers** is *ON*, every chamfer added to the hole will be set to the same size.

3 and 4 - Enter the angle of chamfers on the hole. If **Use same angle for all chamfers** is *ON*, every chamfer added to the hole will be set at the same angle to the hole.

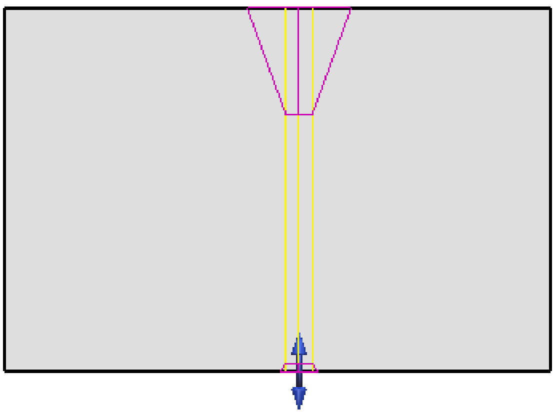
5 - Click on to indicate that there is no base chamfer on the hole. ✓ changes to ✗ and the base chamfer options are hidden.

6 - Click on to indicate that there is no through chamfer on the hole. ✓ changes to ✗ and the through chamfer options are hidden.

Use same size for all chamfers - If *ON*, every chamfer added to the hole will be set to the same size. If OFF, you can define different chamfer sizes for the hole base and the through hole, by entering different values in 1 and 2. The example below shows a chamfer size of *3* for the hole base and *0.5* for the through hole



Use same angle for all chamfers - If *ON*, every chamfer added to the hole will be set at the same angle to the hole. If *OFF,* you can define different chamfer angles for the hole base and the through hole by entering different values in 3 and 4. The example below shows a chamfer angle *20* for the hole base and *45* for the through hole.



Editing a solid hole

You can edit the dimensions of the hole or change to another hole type using the **Plain Hole** dialog (see page 154).

1 Open the dialog in one of the following ways:

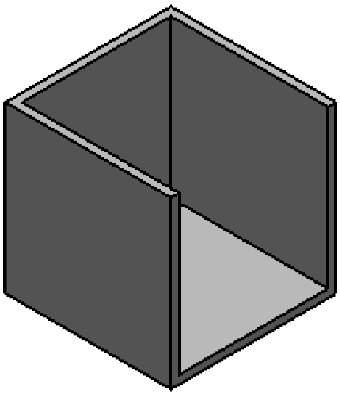
- Double-click the **Hole feature** icon  in the tree.
- Double-click the hole on the model.

2 Use the **Plain Hole** dialog to edit the hole.

You can also simultaneously edit multiple holes (see page 247).

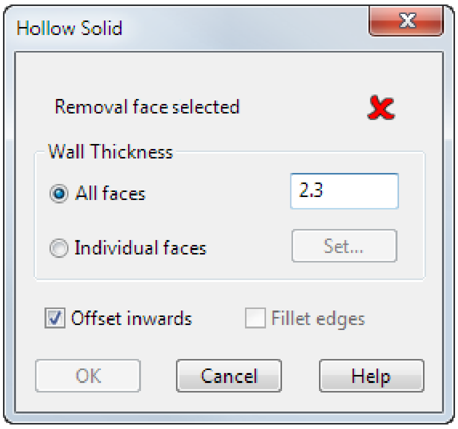
Creating a hollow solid

When creating a hollow solid (often called a 'shell' solid because the thin wall is like a shell), faces of a solid are removed and copies of the other faces are thickened to create a solid wall.



To create a hollow solid:

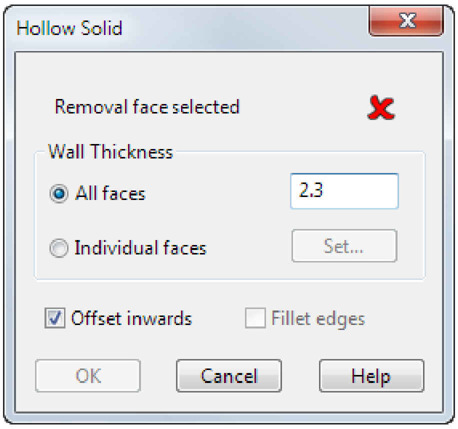
1 Click  (*Solid feature toolbar*).



2 Use the dialog to create the hollow solid.

Hollow Solid dialog

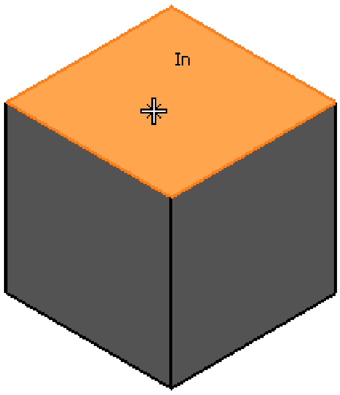
This dialog removes a face of a solid and offsets it to create a hollow solid.



Removal face selected — This indicates if you have selected any faces to remove. To select a removal face:

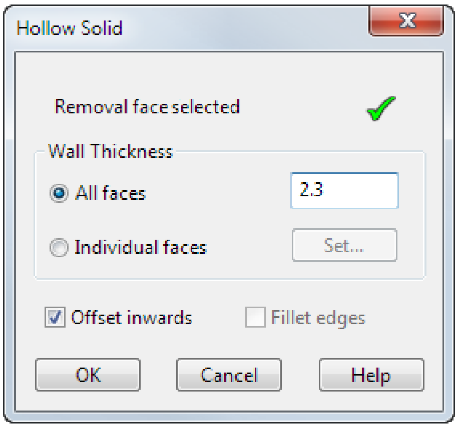
1 Move the cursor over a solid.

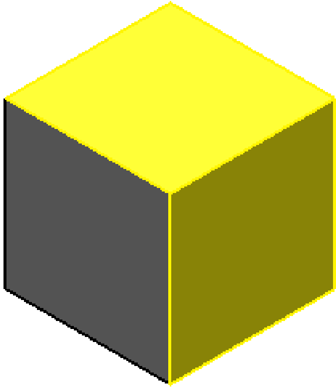
Valid faces are highlighted.



2 Click the face of the solid you want to remove. The cross changes to a tick and the face is highlighted.

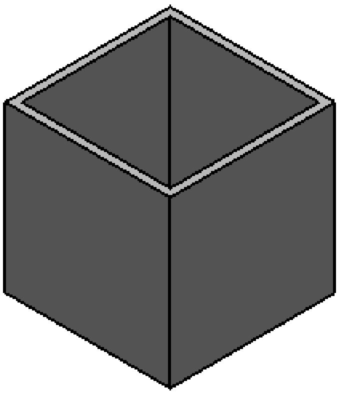
You can select multiple removal faces. You can use either **Ctrl**+click or **Shift**+click to add and remove faces from the selection.

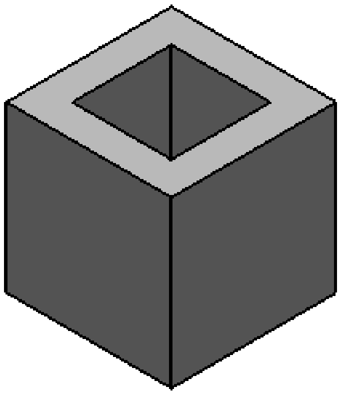


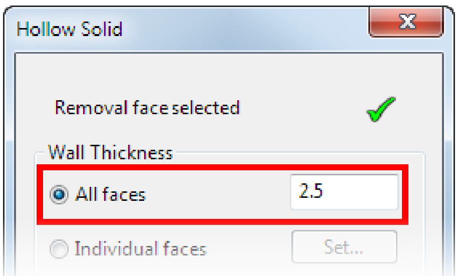


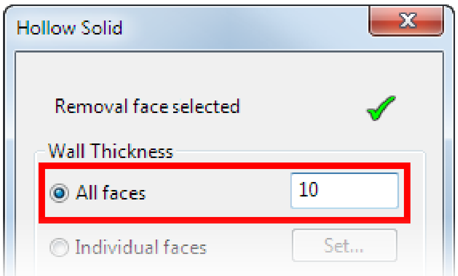
Wall Thickness — This enables you to set the thickness of the faces in the hollow solid.

All faces — If on, all the faces in the solid offset uniformly using the value in data box.

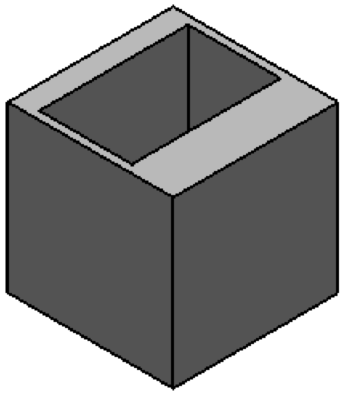






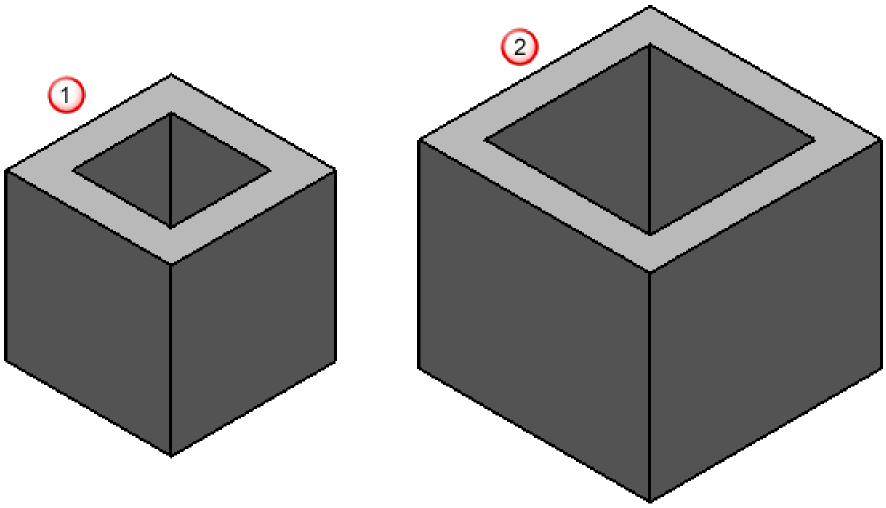


Individual faces — This allows you to have a different wall thickness value for each face.

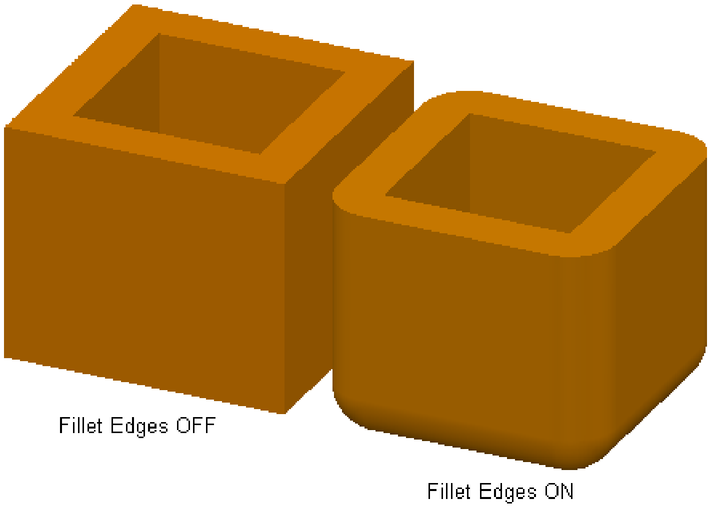


Select this option, then click **Set** to display the **Individual Wall Thickness** dialog.

Offset Inwards — If *ON*, faces are offset inwards 1, otherwise they are offset outwards 2.



Fillet Edges — You can make corners rounded by switching this option on, if faces are offset outwards each face has the same offset value

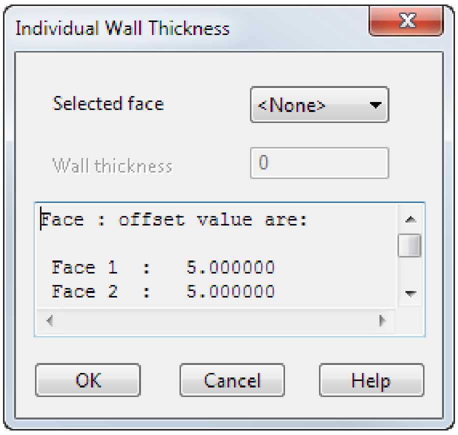


OK — Creates the hollow solid as defined and removes the dialog from the screen. A **Hollow feature** icon  representing the operation is displayed in the solid feature tree.

Cancel — Closes the dialog without operating on the solid.

Individual Wall Thickness dialog

This is used to change the wall thickness of an individual face.



1 Select a face from the **Select face** drop-down list.

   You can also select a face by clicking on its surface rather than its edge. The face nearest to you is selected and highlighted. Its name is displayed in this data box.

   All faces, which are tangent continuous to each other, are highlighted when one of them is selected. They all have the same wall thickness.

2 Enter a **Wall thickness** for the selected face.

The offset value of each face is listed in the scrolling list .

3 Click one of the following buttons:

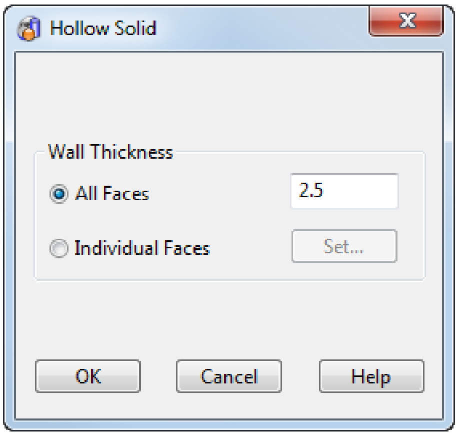
- **OK** to set the wall thickness values of the faces and removes the dialog from the screen. You are returned to the **Hollow Solid** dialog.
- **Cancel** to close the dialog without changing the wall thickness values of the faces.

Editing a hollow

You can edit the wall thickness of the hollow feature.

1 Double-click the **Hollow feature** icon in the tree to display the **Hollow Solid** dialog.

2 Use the dialog to edit the hollow.



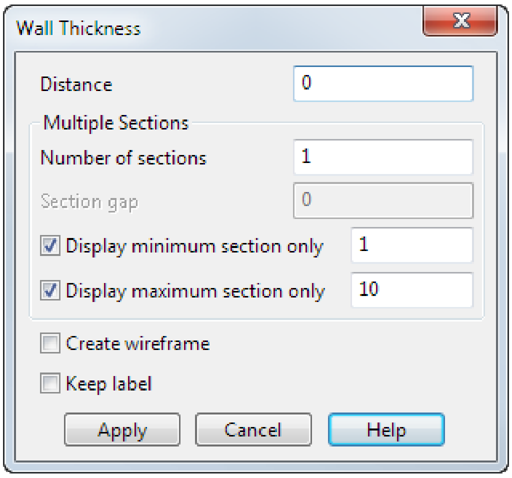
This is a modified version of the **Hollow Solid** dialog used to create the hollow.

Thickness of a hollow solid

You can find the wall thickness of a hollow solid by selecting **Solid Wall thickness** from **Tools > Model Analysis**. This displays the Wall thickness dialog (see page 173) so that you can define a series of sections relative to the active workplane.

Wall thickness dialog

Use this dialog to define a series of sections relative to the active workplane.



1. Enter the **Distance** of the section along the current axis
2. Use the following options to define **Multiple Sections:**

   Number of sections — Number of sections to create.

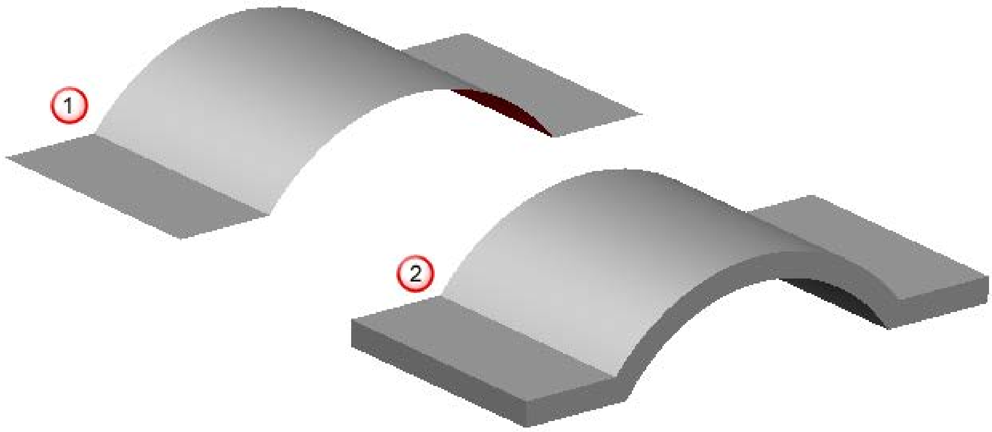
   Section gap — Gap between sections.

   Display minimum section only — The input box displays the minimum thickness. If the option is selected, only the section with the minimum thickness will be displayed.

   Display maximum section only — The input box displays the maximum thickness. If the option is selected, only the section with the maximum thickness will be displayed.
3. Select **Create wireframe** to keep the sections with the minimum/maximum thickness as wireframe dimension after selecting **Apply** or **OK**.
4. Select **Keep label** to keep the labels on the sections with the minimum/maximum thickness as a dimension after selecting **Apply** or **OK**.
5. Click one of the following:
   - **Apply** to apply the wall thickness parameters; the smallest thickness is marked with arrows and a text label and the dialog continues to be displayed.
   - **OK** to accept the smallest thickness is marked with arrows and a text label and close the dialog.

Creating a thickened solid

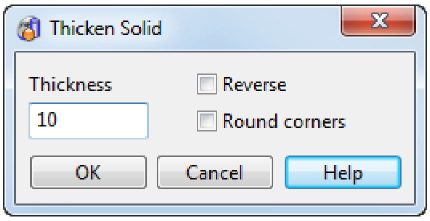
You can apply thickness to an open 1, flattish solid to create a closed solid 2.



The opposite face of the closed solid is created by offsetting the open face.

To create a thickened solid:

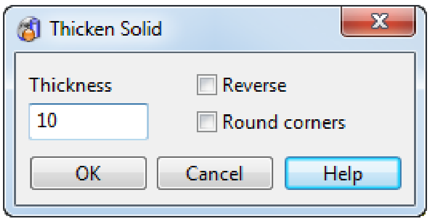
1. Select an open solid.
2. Click  (*Solid feature toolbar*).



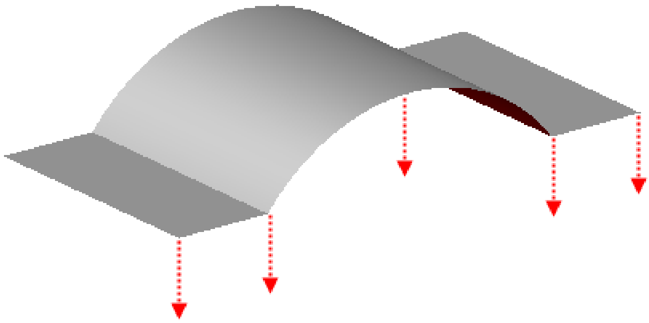
3. Use the **Thicken Solid** (see page 174) dialog to control how the solid is thickened.
4. Click **OK** to close the dialog and thicken the solid.

Thicken Solid dialog

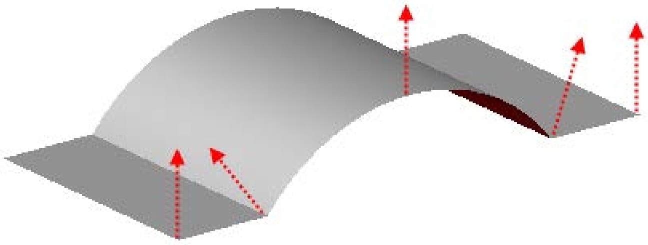
The **Thicken Solid** dialog is used to thicken an open solid and create a closed solid.



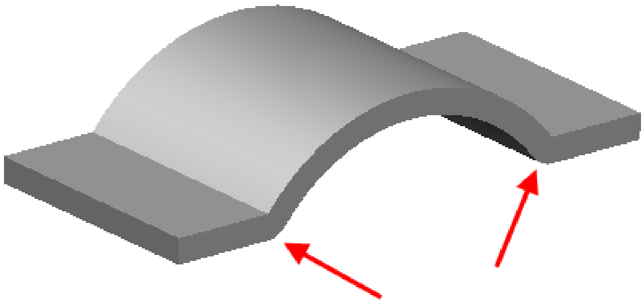
Thickness of solid — This is measured from the inside of the solid, along the negative axis of the surface normal.



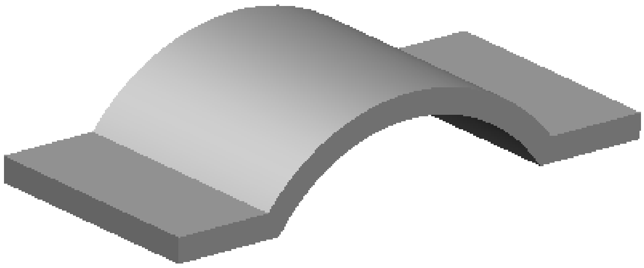
Reverse — The thickness is measured from the outside of the solid.



Round corners — This makes the corners round in the new face of the thicken solid.



OK — Creates a thickness on the solid. A **Thicken feature** icon representing the operation appears in the solid feature tree.



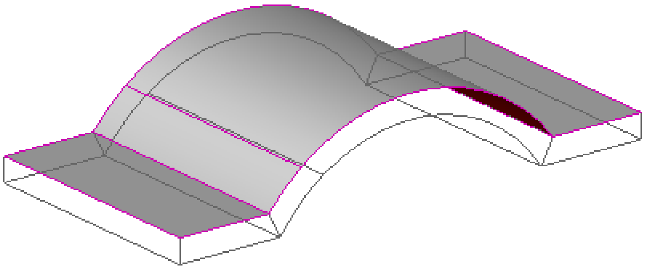
When you thicken a solid, any unused large holes are filled.

Editing a thickened solid

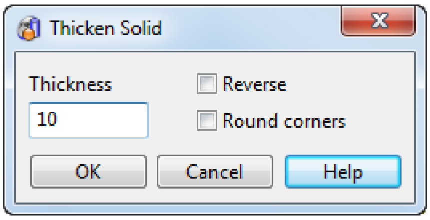
You can edit the thickness of the thicken feature.

1 Double-click the **Thicken feature** icon in the tree to display the **Thicken Solid** dialog.

The solid is displayed on the screen without its thickness.

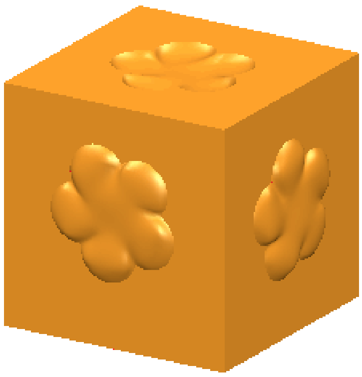


2 Use the Thicken Solid dialog (see page 174) to edit the thickness.



Creating a solid bulge

You can create a bulge using wireframe to wrap a solid.

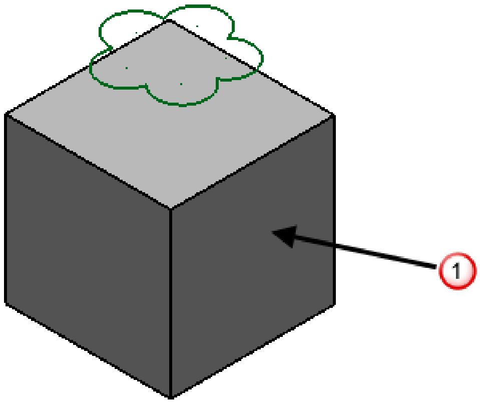


The wireframe is projected onto the solid, along the surface normal of the solid. The solid is then bulged in the area enclosed by the wireframe.

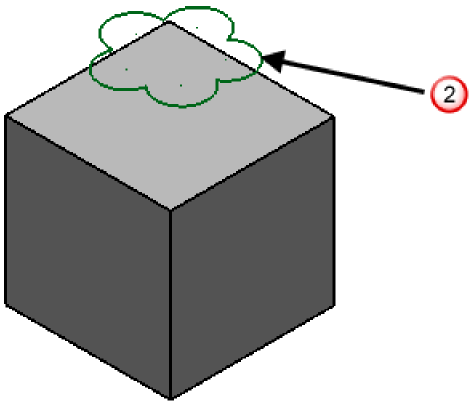
Create a solid bulge using one of the following methods:

Using an active solid

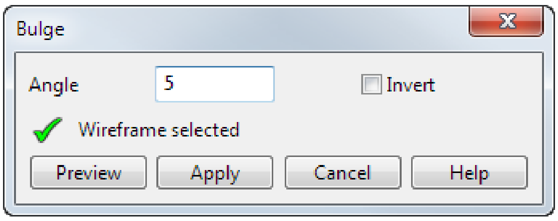
1 Make sure you have an active solid 1.



2 Select one closed wireframe object 2.



3 Click  (*Solid feature toolbar*).

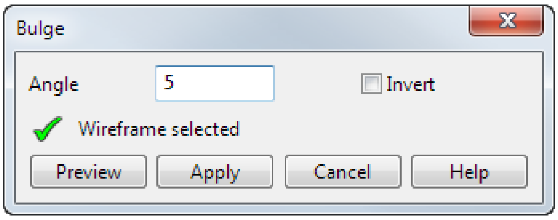


4 Use the **Bulge** dialog (see page 178) to add a bulge onto the solid using the wireframe object.

Using an inactive solid

1 Drag your cursor to select an inactive solid and the closed wireframe.

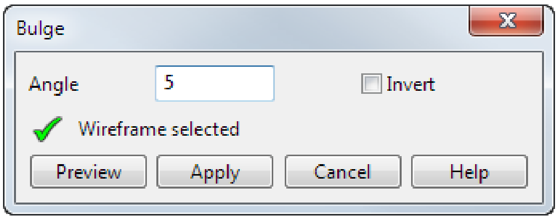
2 Click  (*Solid feature toolbar*).



3 Use the **Bulge** dialog (see page 178) to add a bulge onto the solid using the wireframe object.

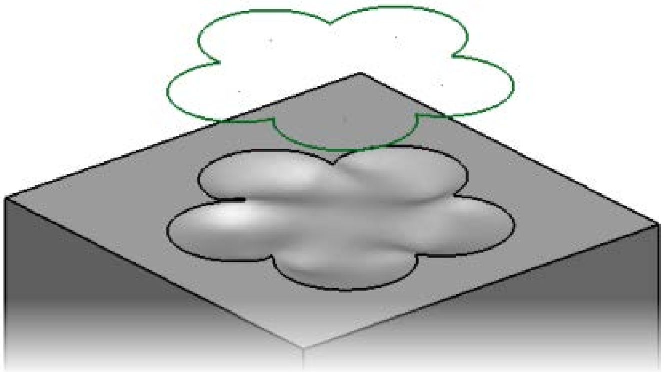
Bulge dialog

Creates a bulge on an active, or pre-selected inactive, solid using wireframe.



1 Enter an **Angle** . This angle defines the amount the bulge lifts from the solid.

2 Select **Invert** to invert the bulge into the solid.

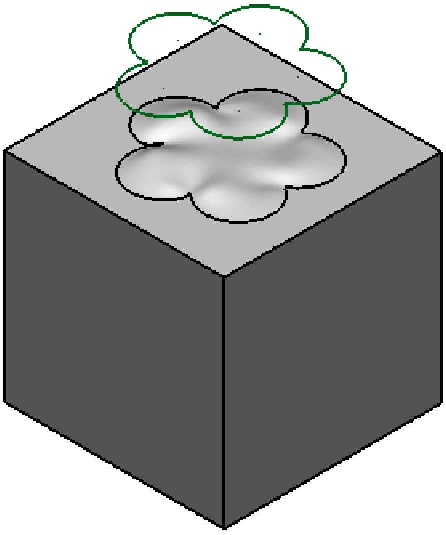


3 **Wireframe selected** — The wireframe defines the outline of the bulge on the solid. When a wireframe object is selected, a tick is shown.

4 Click one of the following:

- **Preview** to display the bulge on the solid using the selected wireframe. You can change the settings on the dialog or the selected wireframe until you are satisfied with the previewed solid.

- **Apply** to create a bulge in the solid. The dialog remains on the screen for you to select more wireframe and continue creating bulges. A **Bulge feature** icon representing the operation appears in the solid feature tree.



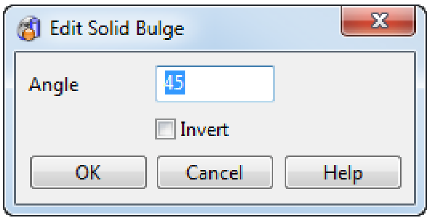
- **Dismiss** to close the dialog.

Editing a bulge feature

Edit the bulge feature as follows:

1 Double-click the **Bulge feature** icon in the tree to display the **Edit Solid Bulge** dialog.

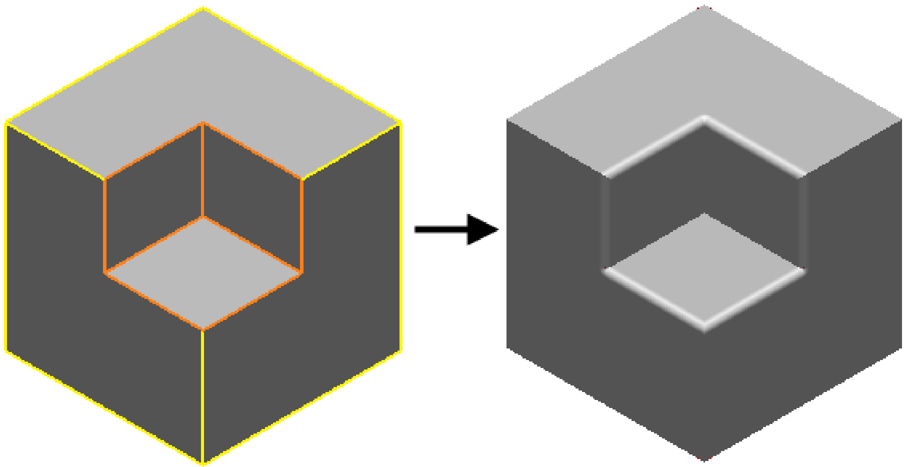
2 Use the dialog to edit the bulge.



This is a modified version of the Bulge dialog (see page 178) used to create the bulge.

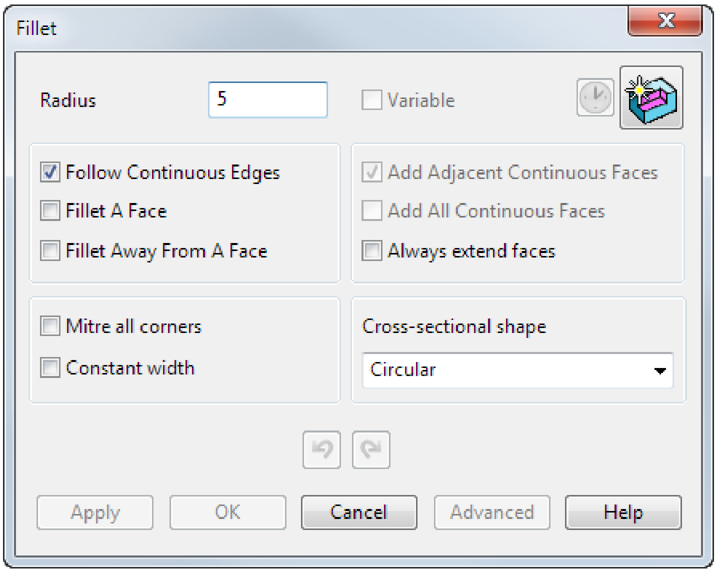
Creating a solid fillet

You can create fillets on the edges of a solid.



To create a solid fillet:

1 Click  (*Solid feature toolbar*).



If your solid contains more than 20 surfaces and is not watertight, you are asked if you want to make it watertight. If you say yes, the **Solid Doctor** is displayed to help you make the solid watertight (within tolerance).

2 Use the Fillet dialog (see page 184) to create fillets on the solid.

Selecting the solid fillet route

Solid filleting uses the dominant face (the one under the cursor) to allow more intelligent picking when filleting the face of a solid.

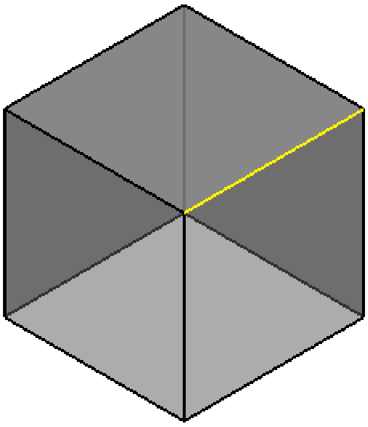
If you are working in wireframe view, you can hover the cursor over the edges of the solid to find out the ones that can be used to create a fillet. If an edge or a path of edges highlights, it can be used. Otherwise, it cannot.

If one of the options **Follow Continuous Edges**, **Fillet a Face** or **Fillet Away From a Face** are on and you hover over a valid edge, a path may highlight. Otherwise, only the single edge highlights. Full details on these three options are given below.

You can select, append or de-select edges to an existing path as follows:

-  selects an edge or a path. If a path is already selected, it is de-selected and a new one is started.

  The selected edge is highlighted. This is the first edge in the path.

- You can select any number of independent tracks.

  is used to add a track to the current selection.

  is a toggle to add/remove tracks from the current selection.

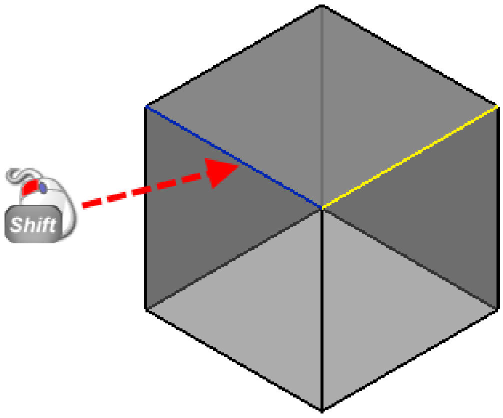
  

  may also be used to join separate tracks into a single track and to split a single track into two separate tracks.

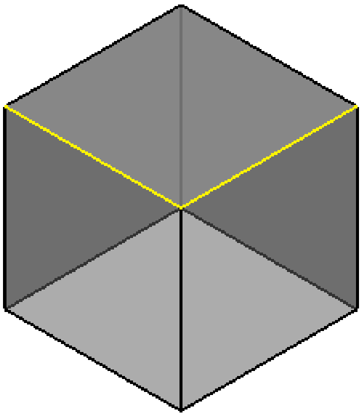
  If the **Follow Continuous Edges** option is on and you  or 

  the selected path, the whole path is de-selected.

In the example below, deselect the **Follow Continuous Edges** option and select an edge at the end of the path using .

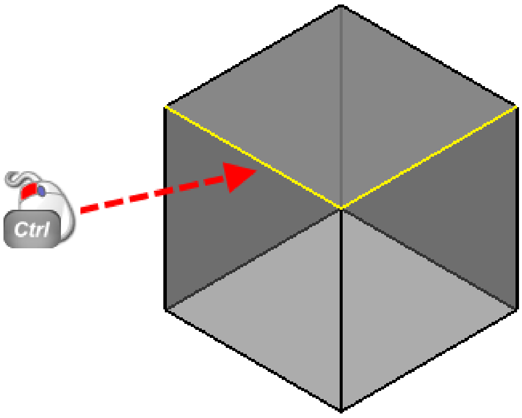


The edge is now highlighted as part of the path.

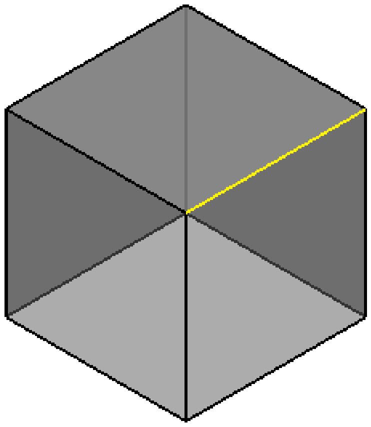


Now 

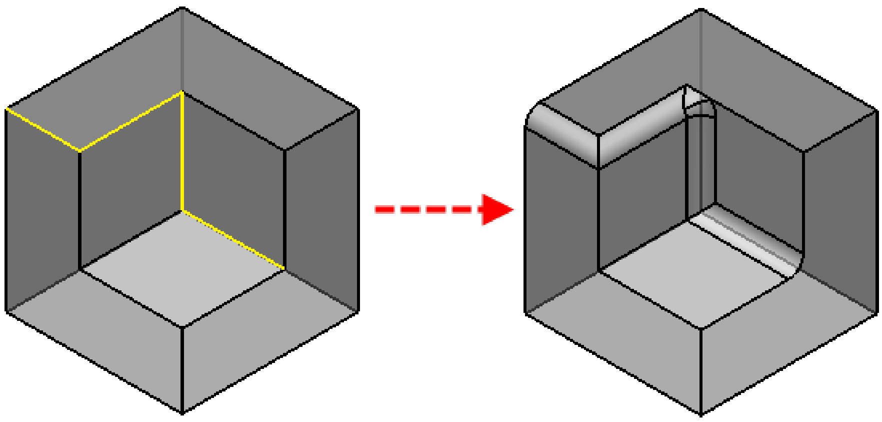
this edge.



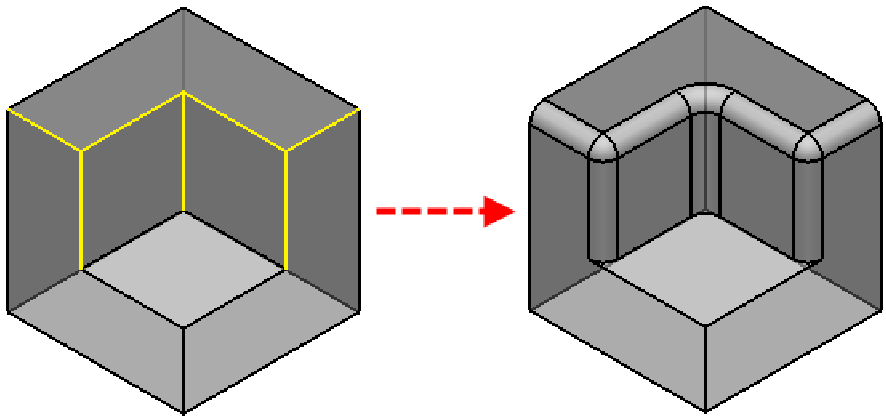
This removes the edge from the path and we are left with the following.



Mixed single path filleting - You can fillet a path that involves both concave and convex fillets. Use  to select the edges to be filleted and click **Apply**.



- **Branch path filleting** - You can fillet a path that has branches. The branches can be any combination of concave and convex. Use  to select the edges on longest path first followed by the edges on the branches.

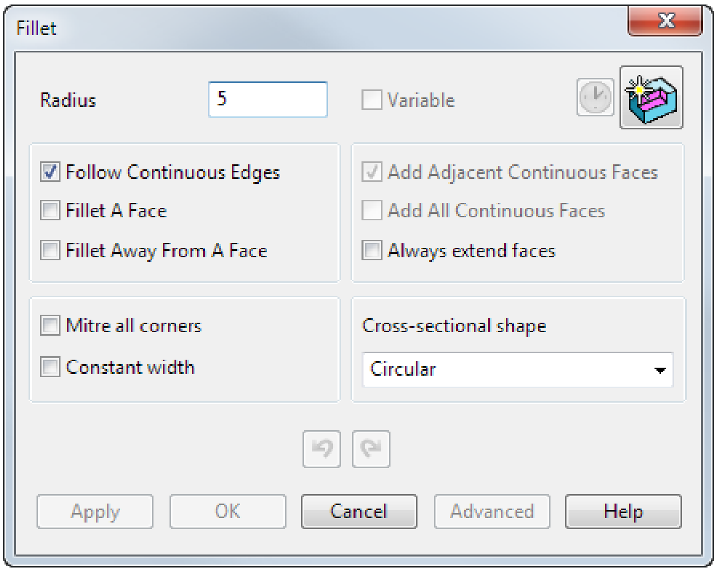


- **Creation of several separate fillets at once** - You can create several separate fillets at the same time. The fillets can be a combination of concave and convex.
- **Parameterised filleting** - You can use a parameter instead of a value to fillet a group of independent tracks. All the parameterised tracks will appear as separate, editable features in the tree.

  As with all current multiple fillet track functionality, using a parameter will parameterise all of the tracks, and all fillets appear as separate editable features in the tree.

Fillet dialog

This dialog is used to fillet edges on a solid.



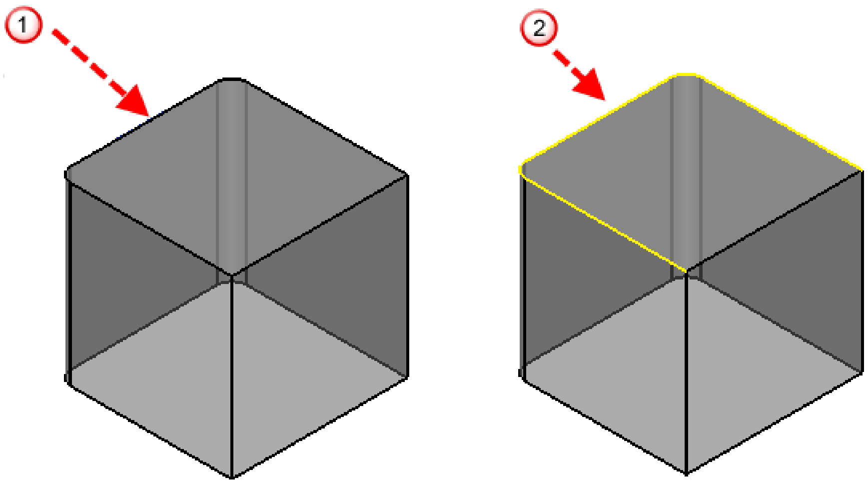
Radius — Enter the radius of the fillet.

Selecting filleting routes

Follow Continuous Edges — If this option is selected and you hover over a valid edge before a path is selected, a path is highlighted which contains that edge and all edges which are tangent continuous with that edge.

When you select an edge, a new fillet path is selected containing that edge and all edges that are tangent continuous with that edge.

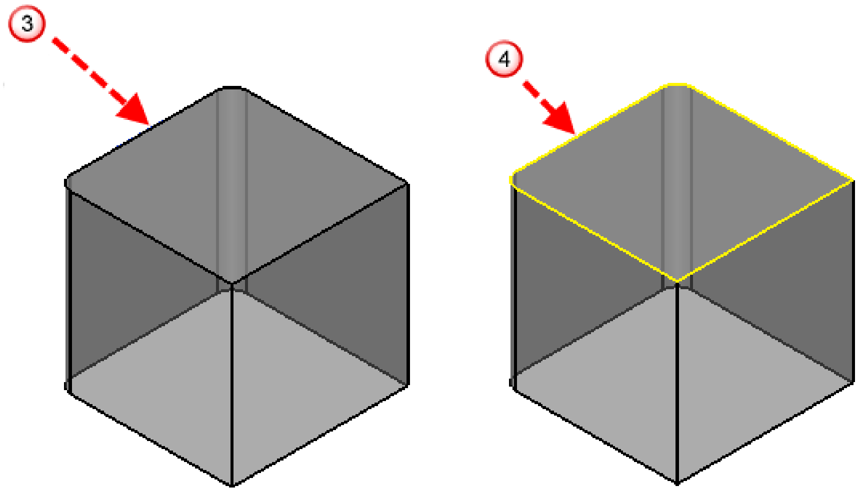
If you select **Follow Continuous Edges** and the edge 1, the fillet path 2 is selected.



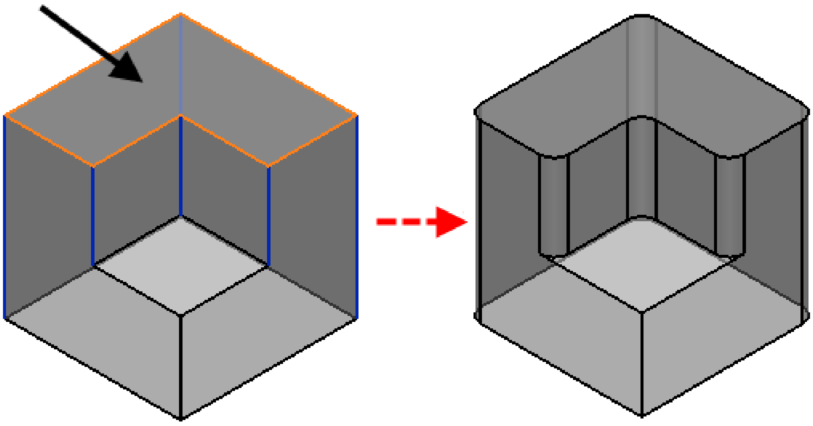
Fillet A Face —- If this option is selected and you hover over a valid edge (before a path is selected), a path is highlighted containing all edges of a face of the solid which contains that edge.

When you select an edge, a new fillet path is selected containing all edges of a face of the solid that contains that edge.

If you select **Fillet a Face** and the edge 3, the fillet path 4 is selected.



Fillet Away From A Face — With this option selected, selecting a face on the model all the edges that will be filleted are highlighted. The highlighted path will be filleted when you click **Apply**.



If required, this option can be used in combination with **Follow Continuous Edges**.

Always extend faces — If this option is deselected, a fillet may partially overlap additional non-tangential geometry, giving an imperfect fillet. Select **Always extend faces** to maintain the fillet radius. An example of using this option is shown in Example 7 (see page 205).

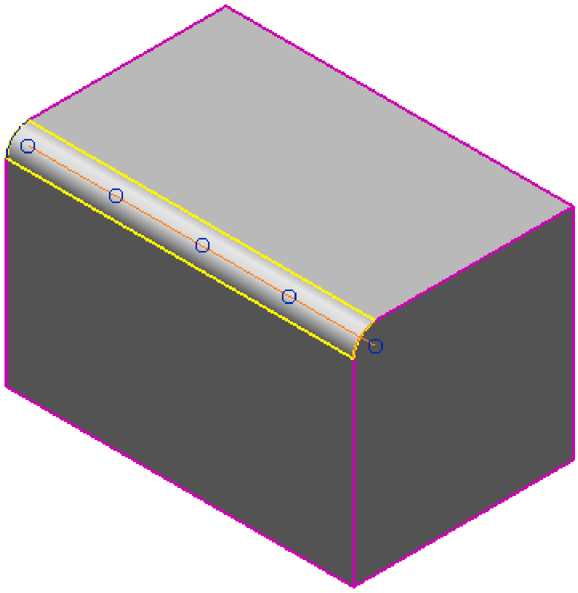
Creating a variable fillet

Use the **Variable** option to edit the radius along the last fillet created. This option is only available once you have clicked **Apply** to create a fillet.

*If you have only selected the fillet path, but not created the fillet yet, you can select the **Advanced** button to create a variable radius fillet using the **Variable Radius Fillet** dialog* *(see page 196)**.*

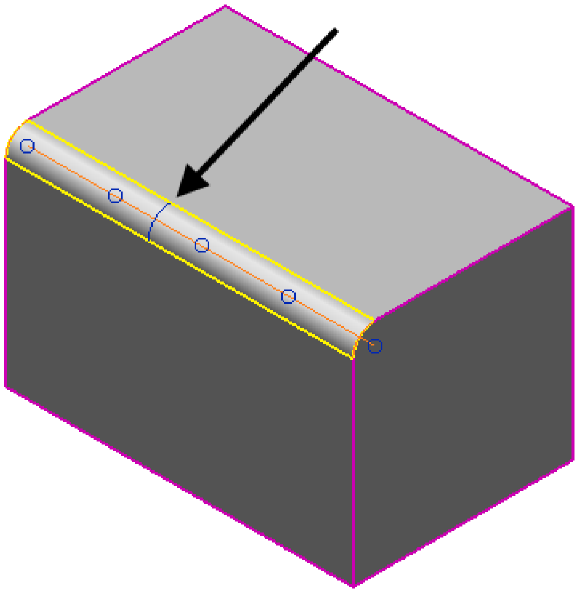
1 Create a fillet and click **Apply**.

2 Select **Variable**. The route of the fillet is displayed.



If you have a closed fillet, this option will remain deselected until you insert an arc.

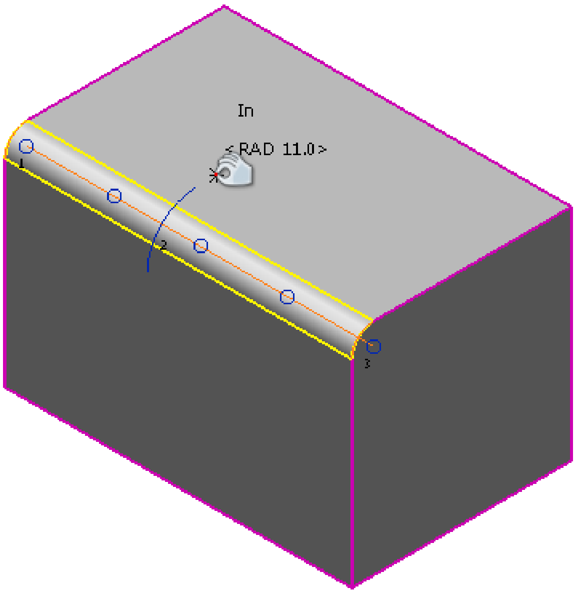
3 Click a position on the fillet to insert an arc that is used to define the radius of the fillet at that point on the fillet path. The inital radius of the arc is **Radius** value.



4 Change the radius of the selected arc by:

- dragging the arc

- entering a new **Radius** value.

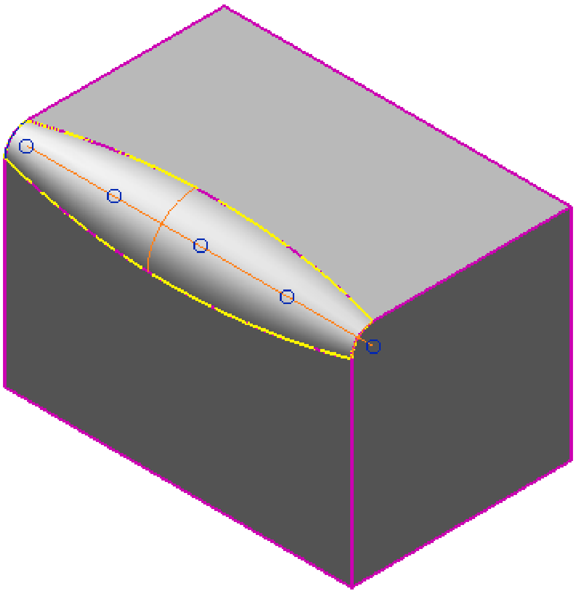


To select an arc on the fillet, click the arc.

To delete the selected arc, select **Delete**.

End arcs cannot be deleted.

5 Click **Apply** to update the fillet.

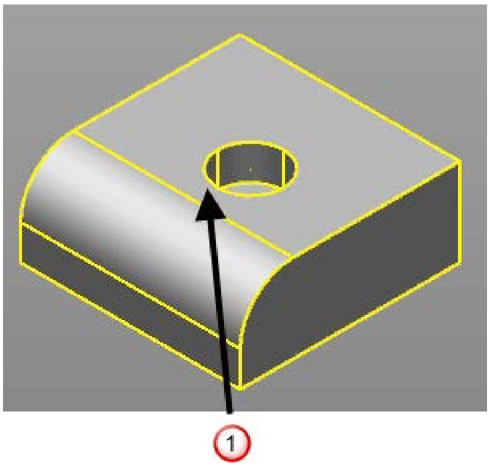


6 Continue to edit the fillet or select another fillet path.

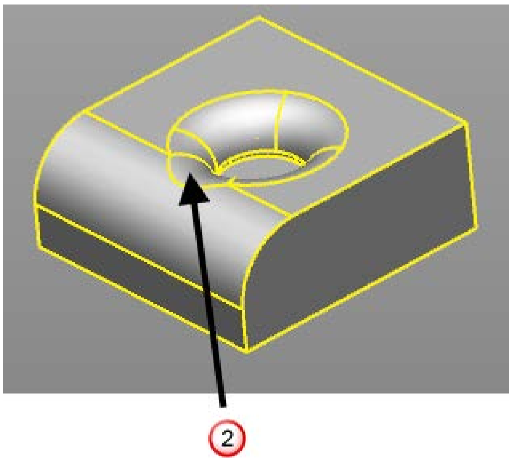
Adding neighbouring surfaces

- If a fillet cannot be created because it spreads onto neighbouring surfaces, select **Add Adjacent Continuous Faces**.

- The fillet will not spread beyond the neighbouring surfaces, so you cannot create a fillet at position ①.



To spread the fillet beyond onto all surfaces, use the **Add All Continuous Faces** option.This option will allow you to create the fillet onto the continuous surfaces ②.



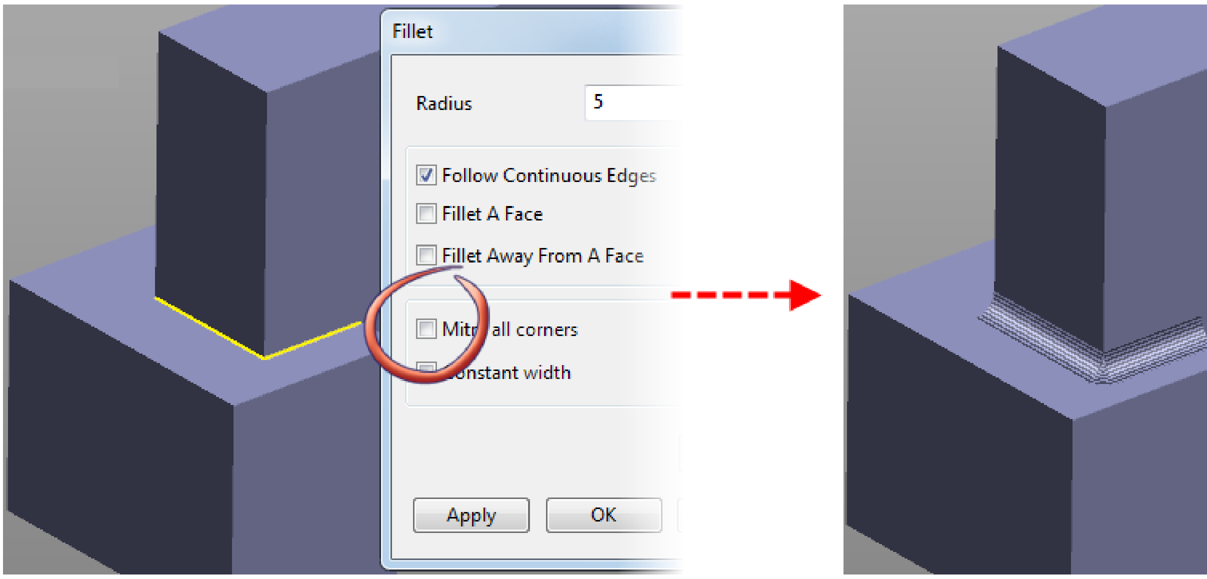
- Select **Add All Continuous Faces** when you need the fillet to spread onto all surfaces that are tangent plane continuous.

  This option may be much slower when creating fillets. We recommend you use the **Add Adjacent Continuous Faces** if you recognise that a fillet will not spread beyond its neighbouring surfaces. When you have finished using this option, we advise you to turn it off.

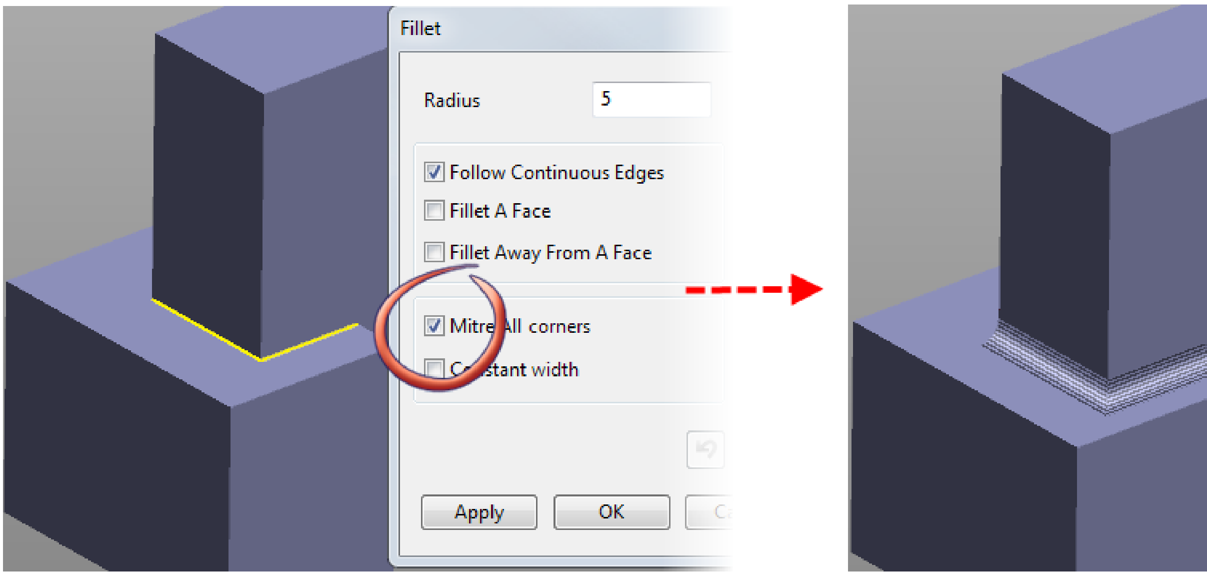
Corners and Sections

- Click **Mitre all corners** to create a fillet with mitred corners or to redefine the fillet to have mitred corners.

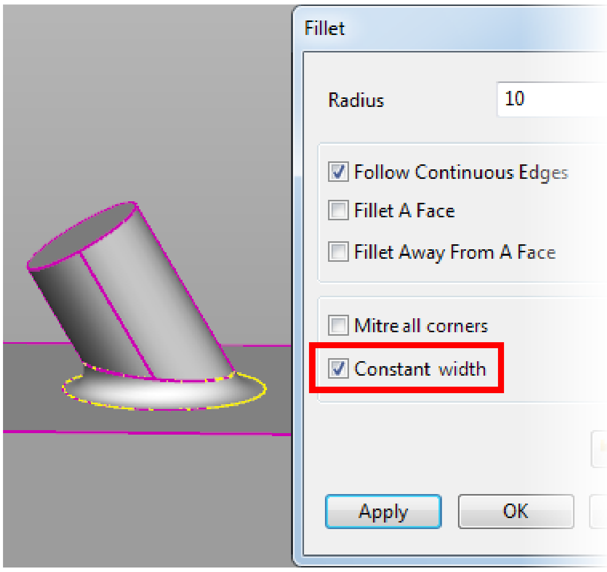
With **Mitre all corners** deselected:



With **Mitre all corners** selected:

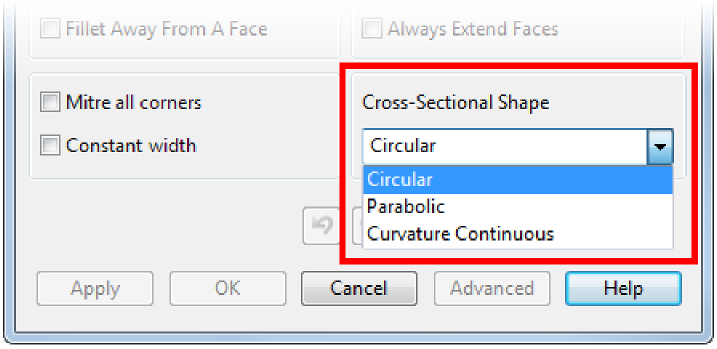


- Click **Constant width** to create a fillet that has a constant width. When this option is selected, the distance between the edges of the fillet is always the same.



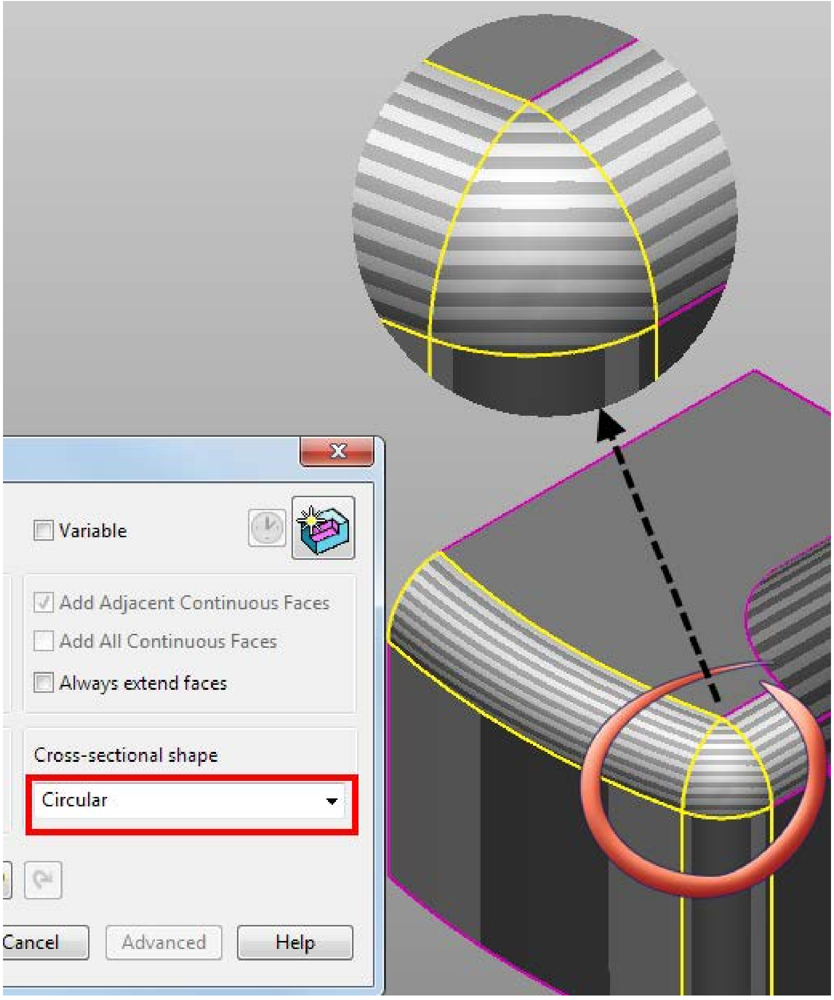
Cross-sectional shape

Select a **Cross-sectional shape** from the drop-down list to change the shape of the cross-section of the fillet.

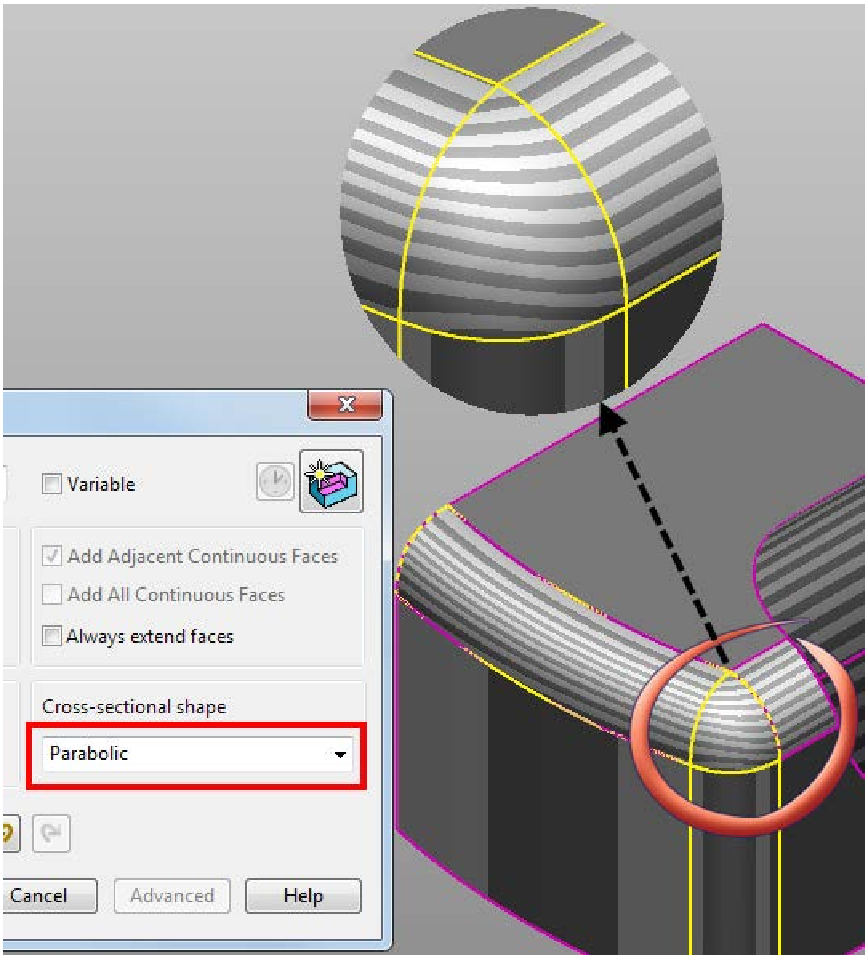


The option that you use depends on your model; choose the option that gives the smoothest fillet for your model.

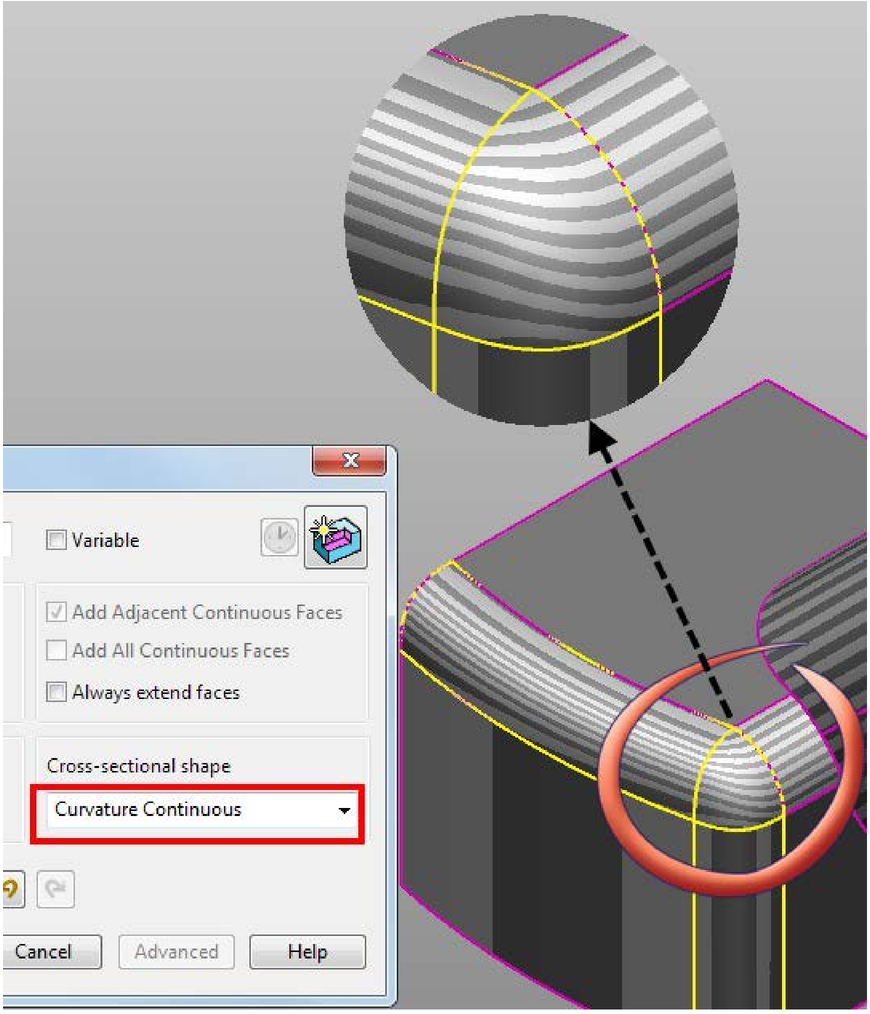
- **Circular** — This is the default cross-sectional shape. It can give a slight change of curvature where the fillet meets another curved edge.



- **Parabolic —** Select this option to create a fillet with parabolic sections.



- **Curvature continuous** - This option creates a fillet without the sharp change in curvature that is created using the **Circular** option. Sections of the fillet are not circular; the curvature varies continously as the blend is created between the adjoining faces.



 — **Undo / Redo** a fillet. The normal undo behaviour has been retained once the dialog is removed from the screen. That is, selecting  **Undo** from the main toolbar immediately after creating a fillet or group of fillets will remove all the fillets that were created.

Apply — Creates a fillet along the selected path and the dialog remains open. This also applies any changes you make to the last fillet created. A **Fillet feature** icon  representing the operation appears in the solid feature tree.

 lets you swap between creation mode and editing mode without closing/reopening the dialog.

- When in creation mode, click  to swap to editing mode.

- When in editing mode, click  to swap to creation mode.

  Using the creation/edit mode button in association with **Apply** lets you create and edit any number of fillets without closing the dialog. It is therefore possible to:complete the following sequence:

  1 Create several fillets
  2 Select edit mode
  3 Select any editable fillet
  4 Modify the fillet by changing its properties
  5 **Apply** the changes
  6 Select and edit another fillet
  7 **Apply** the changes
  8 Enter creation mode
  9 Create another fillet

Advanced — While a path is selected, click **Advanced** to create a variable radius fillet using the **Variable Radius Fillet** dialog (see page 196). Use this dialog to insert arcs along the path to define the different radius values.

OK — Closes the dialog.

Editing a solid fillet during creation

Use the **Fillet** dialog to edit a fillet.

*Remember that you can keep editing the fillet after you click the **Apply** button.*

To edit the radius of a fillet with constant radius:

1. Enter a new value in the **Radius** box.
2. Click **Apply**.

To make a fillet with constant radius into one with a variable radius:

1. Click the **Variable** button.
2. See the **Variable** option for details on how to insert new radius values along the fillet.
3. Click **Apply**.

To make a fillet with variable radius into one with a constant radius:

1. Turn off the **Variable** option.

2 Enter the new radius value in the **Radius** box.

3 Click **Apply**.

To change the radius of a fillet with variable radius:

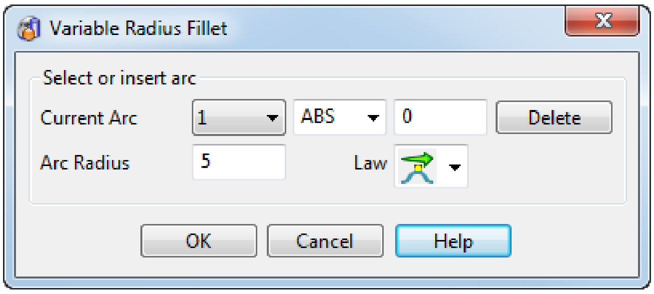
1 Keep editing the radius by inserting arcs along the fillet.

2 Click **Apply**.

Variable Radius Fillet dialog

Use the options on this dialog to insert arcs along the fillet and allows you to edit their radius values. It also sets how the radius varies from one arc to another.

*Display the dialog by clicking the **Advanced** button on the **Fillet** dialog.*



Specifying the Current Arc

Use the drop-down lists and a text box to specify the Current Arc as follows:

1 The first drop-down list box displays the selected fillet arc. You can use this drop-down list box 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 box.

2 Use the second drop-down list box and text box to enter new fillet arcs as follows:

   a Select one of the following options from the second drop-down list box 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 box.

   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.

3 Enter a distance in the text box and press *Enter* to enter 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 box will determine what the value means. For example if **ABS** is selected, the value is the absolute distance along the fillet route.

Delete - This deletes the selected arc.

End arcs cannot be deleted.

Specifying the Arc Radius

The **Arc Radius** option 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.

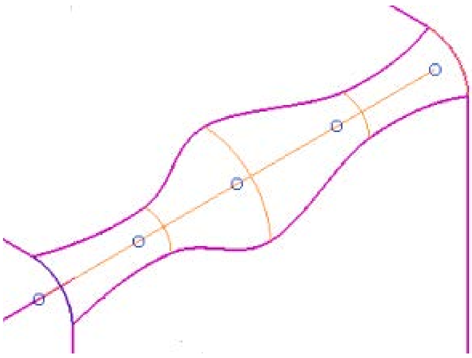
Specifying the Law

The **Law** option 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.

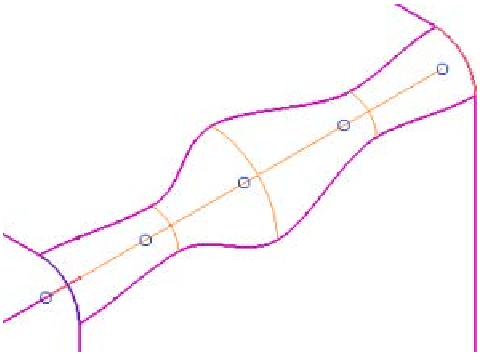
There are three ways of varying the arc:

- Free
- Horizontal
- Sharp.

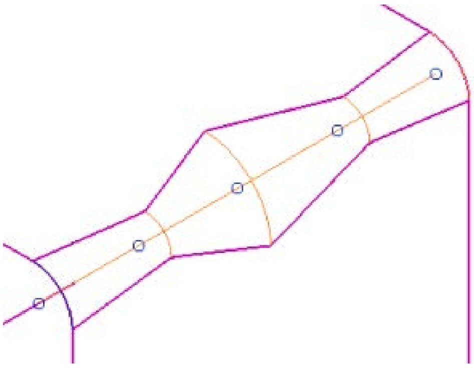
Free — the radius varies freely from the arc and then takes on the conditions of the neighbouring arc.



Horizontal — the radius remains constant for a short distance away from the arc and then takes on the conditions dictated by the neighbouring arc.



Sharp — the radius changes size quickly as soon as it leaves the arc and then takes on the conditions dictated by the neighbouring arc.



The **Law** option has nine selections, where the first word dictates how the radius varies as it enters an arc and the second how it leaves an arc.

 - Free-Free

 - Free-Horizontal

 - Free-Sharp

 - Horizontal-Horizontal

 - Horizontal-Free

 - Horizontal-Sharp

 - Sharp-Sharp

 - Sharp-Free

 - Sharp Horizontal

OK — This creates the variable radius fillet and takes you back to the Fillet dialog (see page 184).

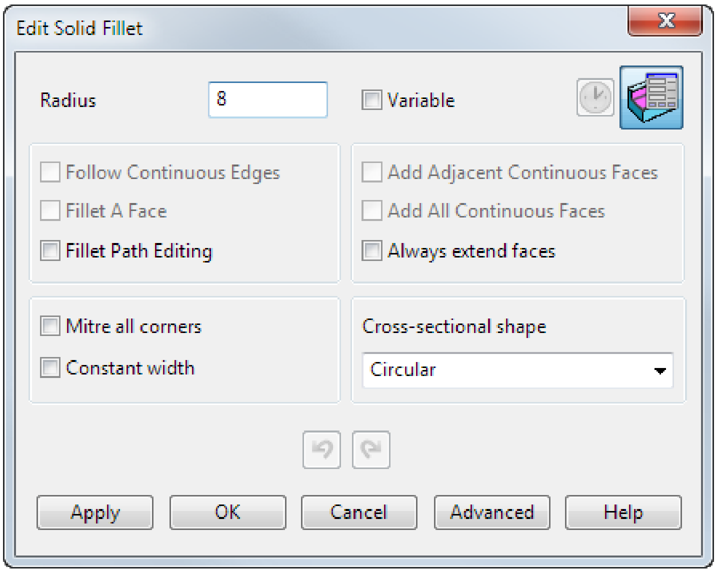
Cancel — The fillet is not created and you return to the **Fillet** dialog. The path is still selected.

Editing a solid fillet

You can edit an existing fillet or edit multiple fillets (see page 247).

To edit a fillet:

1 Double-click the **Fillet feature** icon  in the tree to display the **Edit Solid Fillet** dialog.



2 Use the dialog to edit the fillet, you can:

- Change a variable radius fillet to a constant one. Deselect the **Variable** option and enter a new value in the **Radius** box .
- Change a constant radius fillet to a variable one. Select **Advanced** to display the **Variable Radius Fillet** dialog or select the **Variable** option and insert new radius values along the fillet.
- Edit the path of the fillet. Select **Fillet Path Editing** to highlight the original path of the fillet. Adjust the existing path or create a totally new one.
- Edit the fillet to have a constant width by toggling the **Constant Width** option.
- Select a new **Cross-sectional shape**.
- Select a new path for the fillet as if creating a new fillet.

Some filleting examples

'Rolling-ball' filleting is used when generating fillets would otherwise be inaccurate. The examples shown here illustrate different cases where this methodology has been automatically incorporated into the generation of fillets. In every case, the fillet on the updated model has been created using the **Fillet** option on the solid menu.

Example 1 - Filleting side walls with cut-outs (see page 200)

Example 2 - Filleting a face with a hole (see page 201)

Example 3 - 'Rolling ball' technique (see page 202)

Example 4 - Filleting two blocks (see page 203)

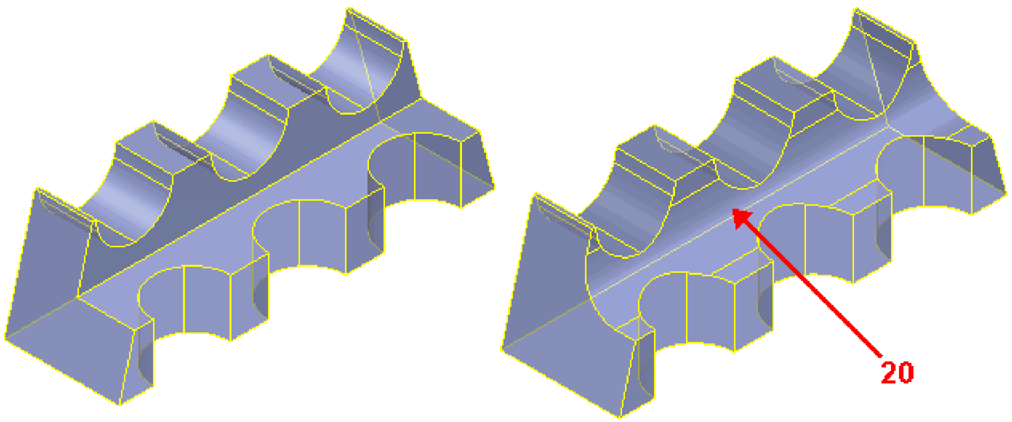
Example 5 - Maintaining the fillet curvature (see page 203)

Example 6 - Flexibility of the 'Rolling ball' technique (see page 204)

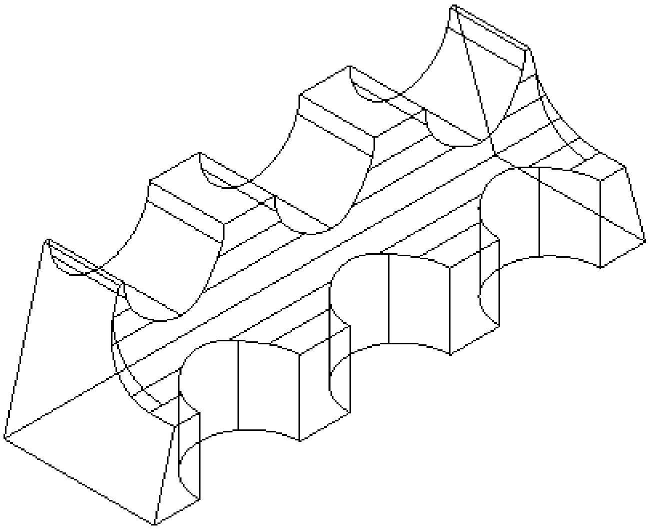
Example 7 - Using Always Extend Faces (see page 205)

Example 1 - Filleting side walls with cut-outs

In this model, a fillet of *20* is added. The cut-outs intersect with the fillet, maintaining the fillet curvature throughout.

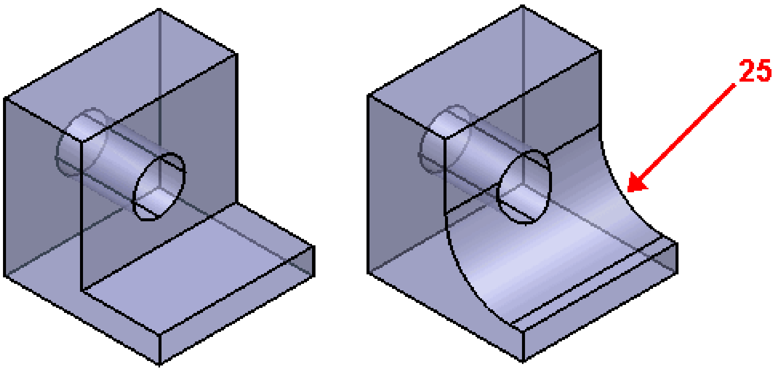


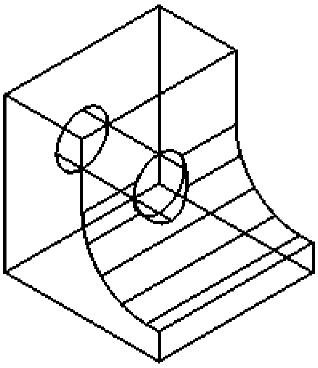
Looking at the wireframe view below, you can see that the fillet is carried right up the side walls of the block.



Example 2 - Filleting a face with a hole

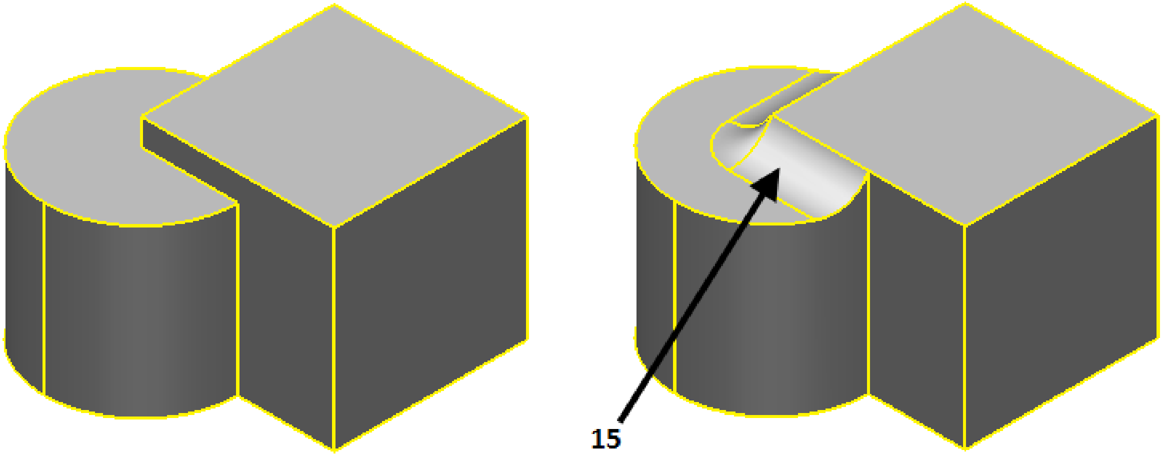
The model below shows two blocks with a hole in the middle of one block face. If you add fillet of *25* as shown, the hole is extended to maintain the curvature of the fillet, as shown in the wireframe version .





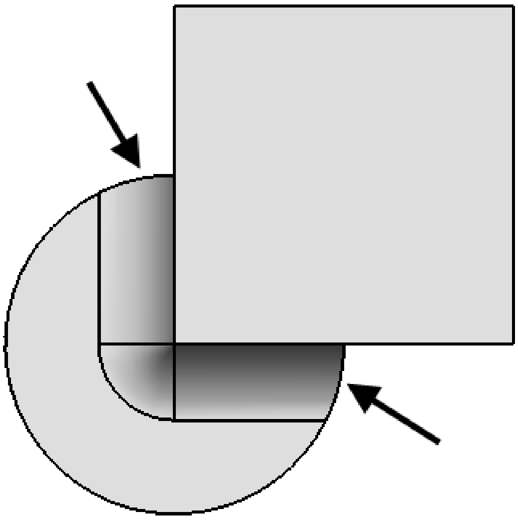
Example 3 - 'Rolling ball' technique

Use 'rolling-ball' filleting to produce fillets like the one shown on the model below.

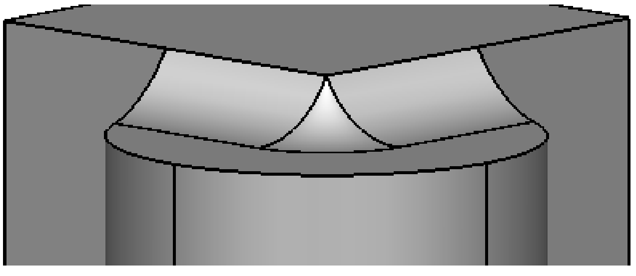


A fillet of *15* has been added at the intersection of the block and the cylinder. The important things to note about this fillet are:

1 The smooth end caps, where the surface of the fillet is curved to match the radius of the cylindrical part of the solid.

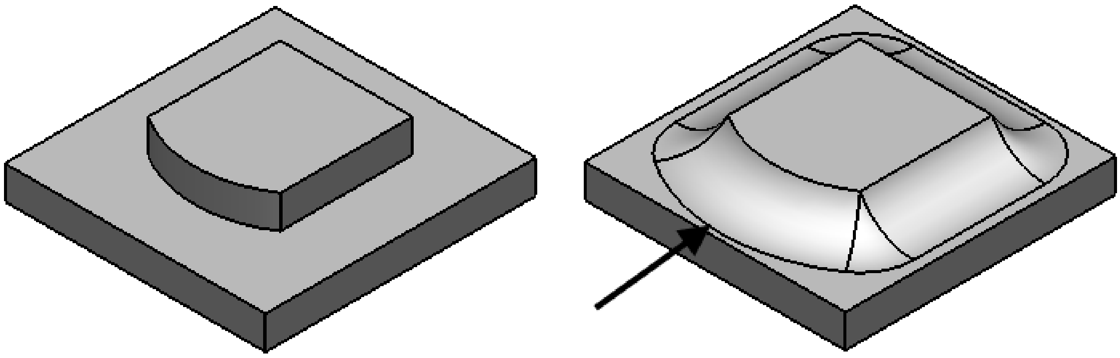


The bottom of the fillet is tangential to the cylindrical part of the solid and the top of the fillet touches the edge of the block at an angle.



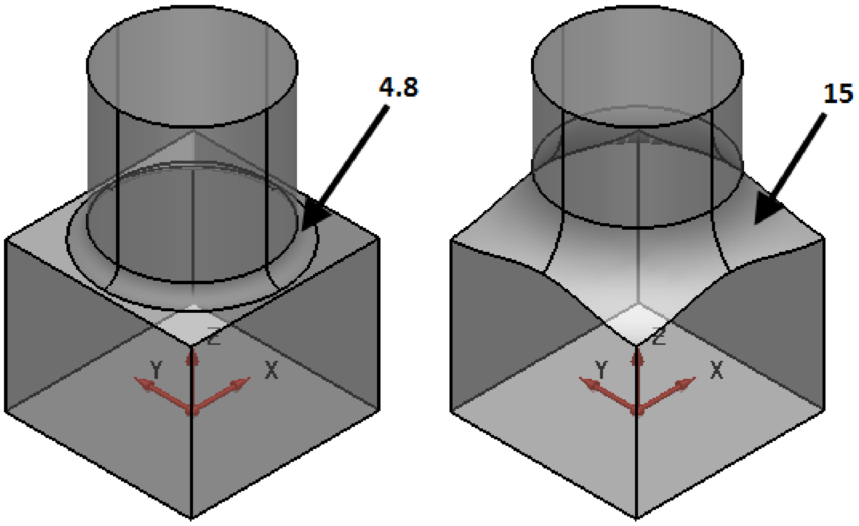
Example 4 - Filleting two blocks

In this model, a fillet of *18* has been created all the way round the top block. The bottom of the fillet that is created is tangential to the bottom block and the top of the fillet touches the edge of the top block at an angle.



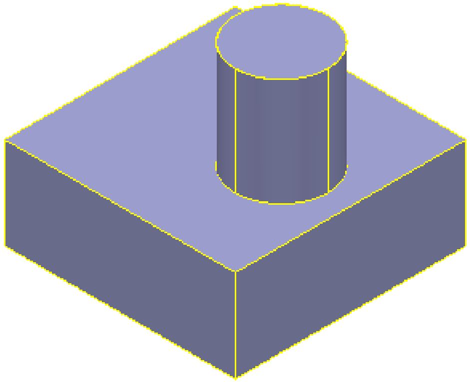
Example 5 - Maintaining the fillet curvature

On the original model, the edge of the fillet of *4.8* touches the sides of the block. You can increase the existing fillet radius to *15.* Again, the fillet curvature has been maintained by extending and re-intersecting surfaces.

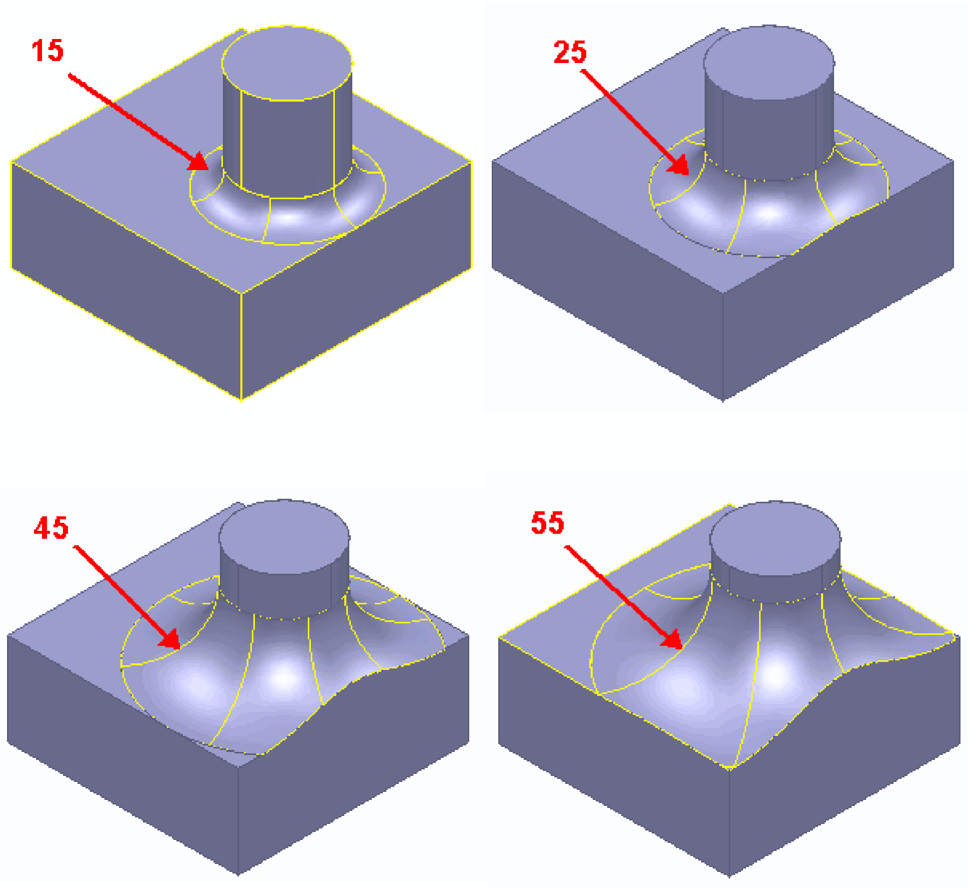


Example 6 - Flexibility of the 'Rolling ball' technique

This example illustrates the flexibility that the 'rolling ball' technique provides.



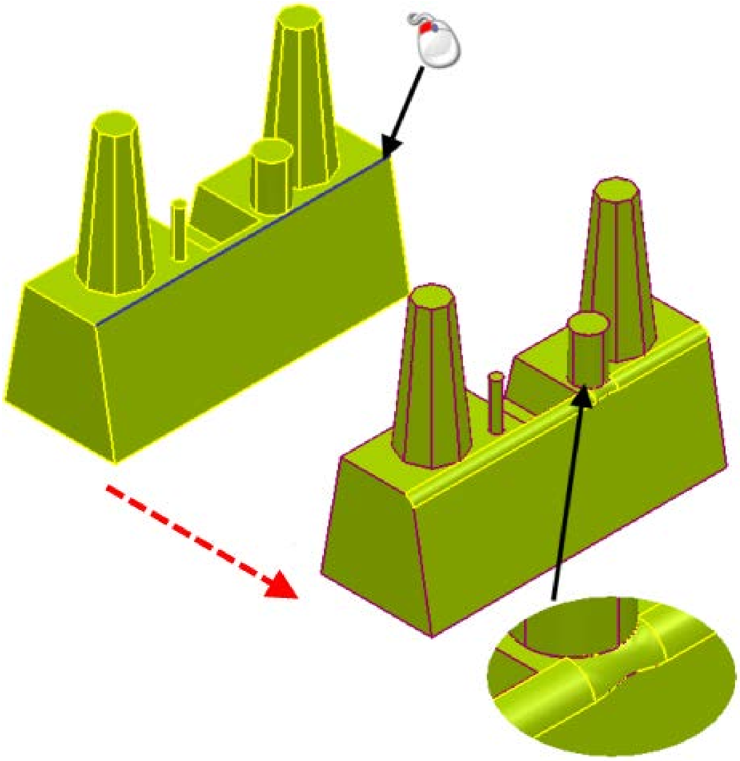
Each of the models below shows the effect of a different radius. The numbers in red are the radius of the fillet that was used to produce the model. Note how the edge of the block is modified so that the filleting is smooth at the intersection between the block and the fillet.



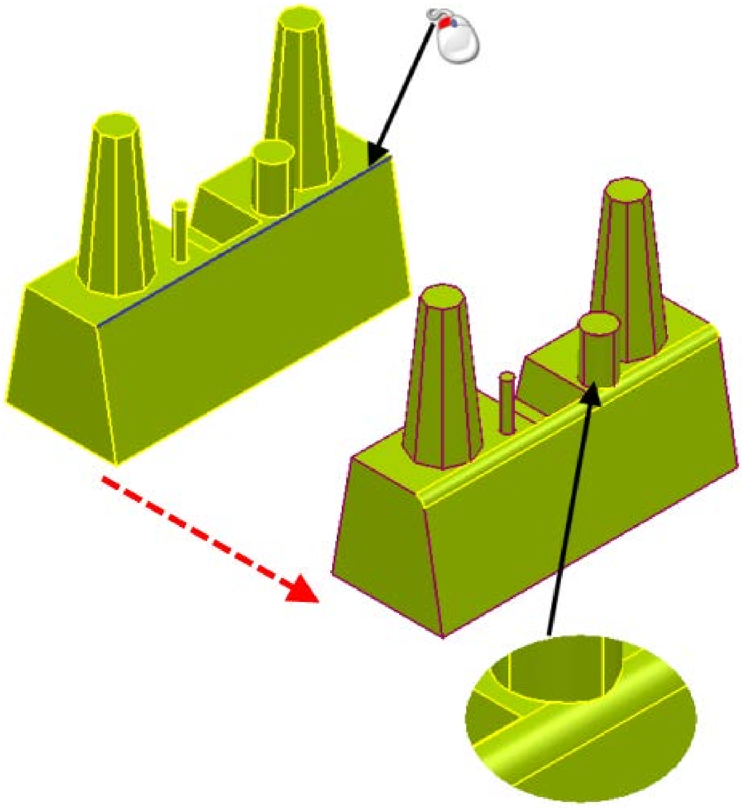
Example 7 - Using Always Extend Faces

This example illustrates the use of **Always Extend Faces** to maintain the fillet radius.

In this example, a fillet partially overlaps additional non-tangential geometry, giving an imperfect fillet.

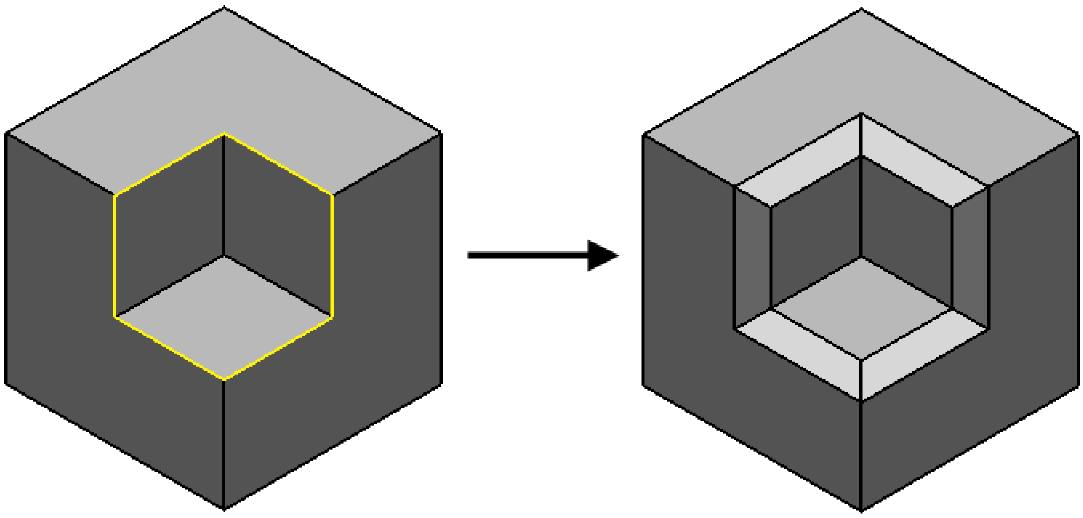


Select **Always Extend Faces** to maintain the fillet radius.



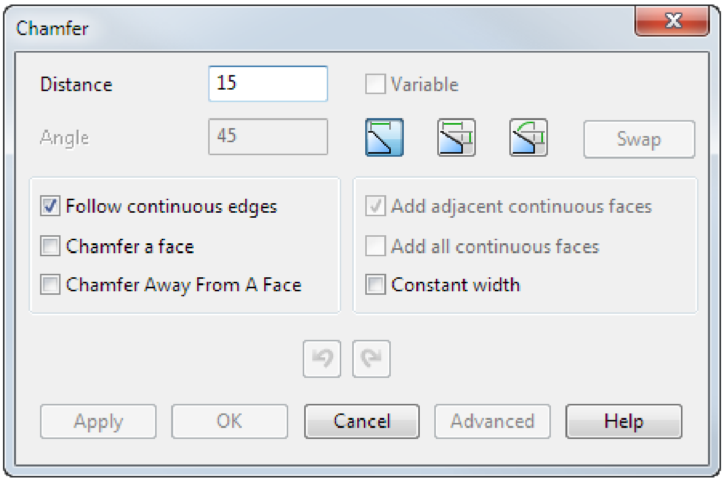
Creating a solid chamfer

You can create chamfers on the edges of a solid.



To create a solid chamfer:

1 Click  (*Solid feature toolbar*).

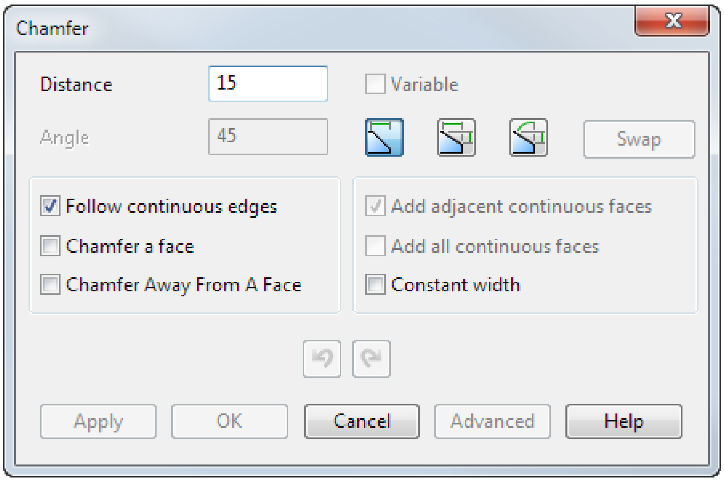


If your version 8 solid contains more than 20 surfaces and is not watertight, you are asked if you want to make it watertight. If you say **Yes**, the **Make Watertight Wizard** (see page 295) appears to help you make the solid watertight (within tolerance).

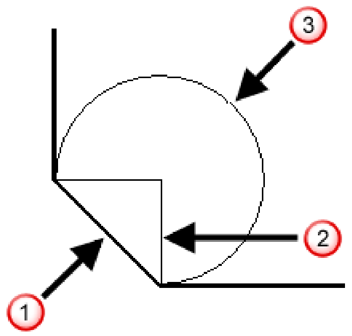
2 Use the **Chamfer** dialog (see page 207) to create chamfers on the solid.

Chamfer dialog

This dialog is used to chamfer edges on a solid.



Enter the **Distance** to be used to create the chamfer as shown below.



1 Chamfer.

2 Distance.

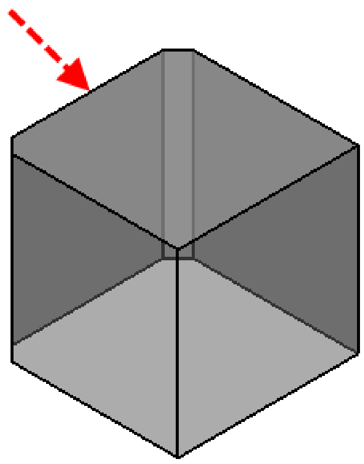
3 Arc with radius equal to distance.

Selecting the chamfering route

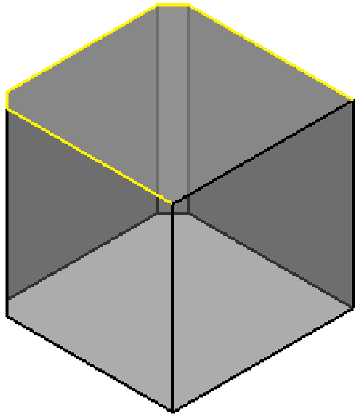
Follow continuous edges - If this option selected and you hover over a valid edge, before a path is selected, a path is highlighted which contains that edge and all edges which are tangent continuous with that edge.

If you select an edge, a new path is selected containing that edge and all edges which are tangent continuous with that edge.

The edge shown below is selected.



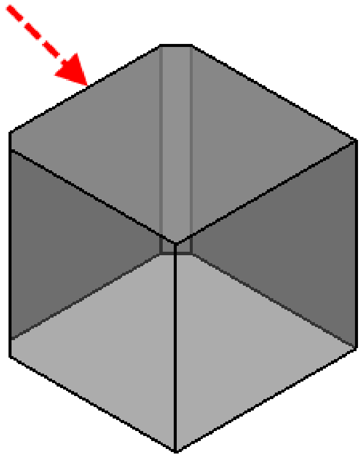
The following path is selected.



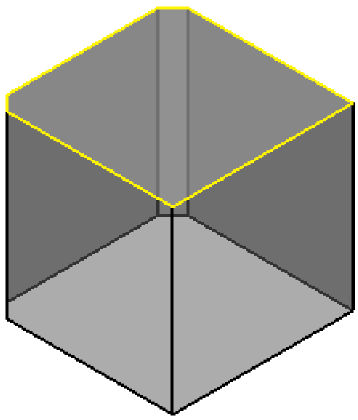
Chamfer a face — If this option is selected and you hover over a valid edge, before a path is selected, a path is highlighted containing all edges of a face of the solid which contains that edge.

If you select an edge, a new path is selected containing all edges of a face of the solid which contains that edge.

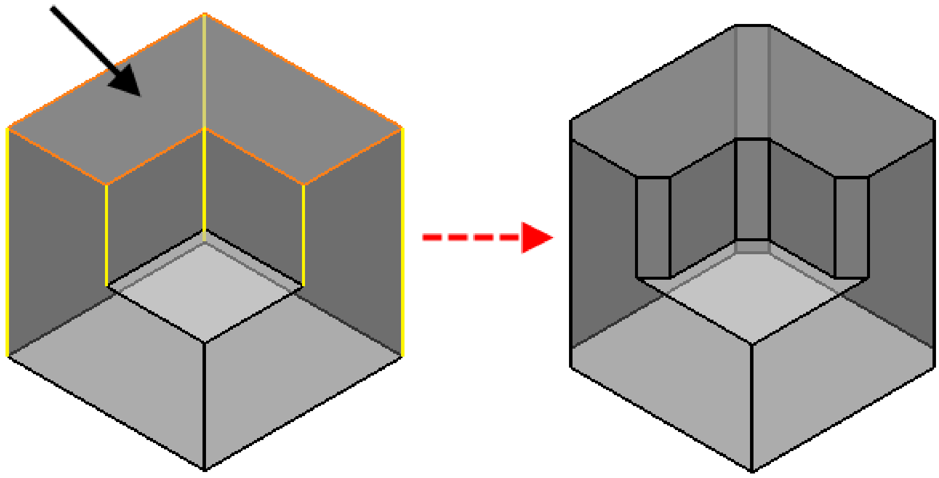
The edge shown below is selected.



The following path of face edges is selected.



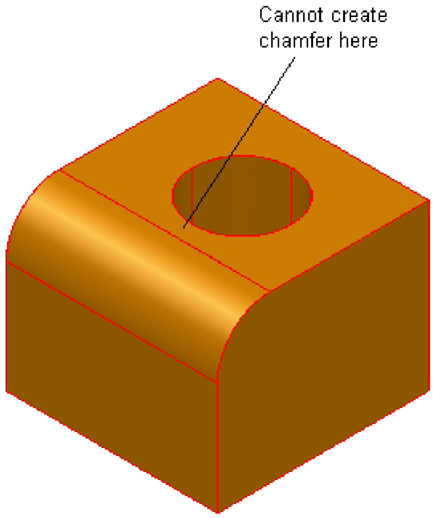
Chamfer Away From A Face — With this option selected, selecting a face on the model will cause all the edges that will be chamfered to be highlighted. The highlighted tracks will be chamfered when you click **Apply**.



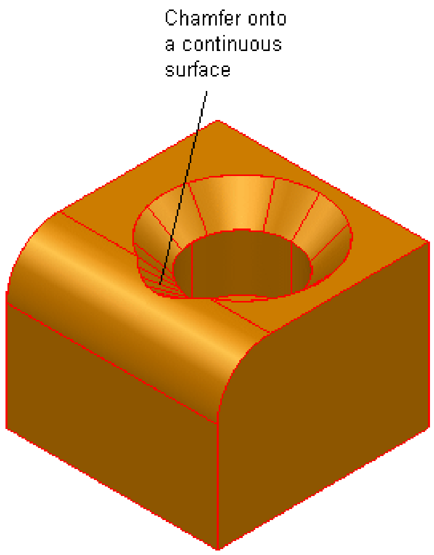
Variable — This option is only available once you have clicked **Apply** to create a chamfer (see page 217). If you have only selected the path, but not created the chamfer, you can select the **Advanced** button to create a variable distance chamfer (see page 215).

Add adjacent continuous faces — If a chamfer cannot be created because it spreads onto neighbouring surfaces, turn this option on.

The chamfer will not spread beyond the neighbouring surfaces. To spread the chamfer beyond onto all surfaces, use the **Add all continuous faces** option.



This option will allow you to create the chamfer onto the continuous surfaces.



Add all continuous faces — This is the same as the **Add adjacent continuous faces**, except the chamfer can spread onto all surfaces that are tangent plane continuous.

This option may be much slower when creating chamfers. We recommend you use the **Add adjacent continuous faces** if you recognise that a chamfer will not spread beyond its neighbouring surfaces.

If you have finished using this option and it is no longer needed, we advise you to turn it off.

Constant width — When you create a chamfer, the width of the chamfer varies to reflect the angle between the surrounding faces. Select **Constant width** to create a constant width for a chamfer.

— **Undo / Redo** a chamfer. The normal undo behaviour has been retained once the dialog is removed from the screen. That is, selecting **Undo** from the main toolbar immediately after creating a chamfer or group of chamfers will remove all the chamfers that were created.

Apply — Creates a chamfer along the selected path. The dialog remains on the screen for you to select more edges and continue creating chamfers.

A **Chamfer feature** icon representing the operation appears in the solid feature tree.

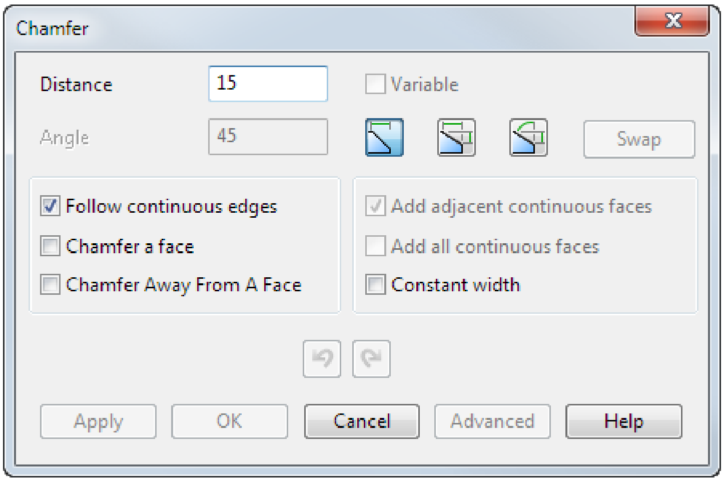
You can now either edit the distance of the chamfer just created or select more edges to create another chamfer.

Advanced —- While a path is selected, you can create a variable distance chamfer. Click this button to display the **Variable Distance Chamfer** dialog. Use this dialog to insert lines along the path to define the different distance values.

OK — Removes the dialog from the screen.

Selecting the solid chamfer route

If you are working in wireframe view, you can hover the intelligent cursor over the edges to discover which ones can be used to create a chamfer. If an edge or a path is highlighted, you can use it.

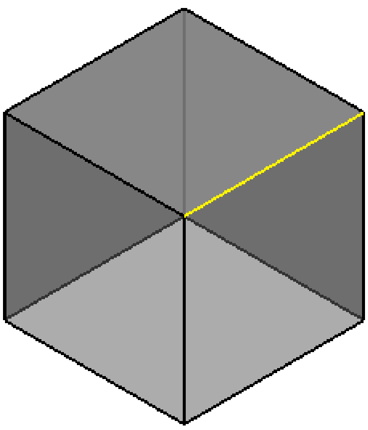


If one of the options **Follow Continuous Edges**, **Chamfer a Face** or **Chamfer Away From A Face** are selected and you hover over a valid edge, a path may highlight. Otherwise, only the single edge highlights. Full details on these options are given below.

You can select, append, or de-select edges to an existing path as follows:

- A single click selects an edge or a path. If a path is already selected, it is de-selected and a new one is started.

  The selected edge is highlighted. This is the first edge in the path.



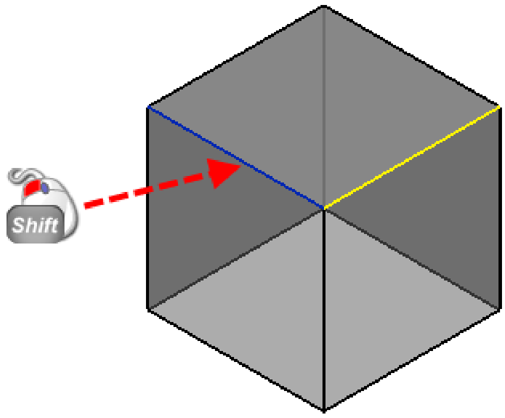
- You can select any number of independent tracks.

  Press and hold the **Shift** key and click to add a track to the current selection.

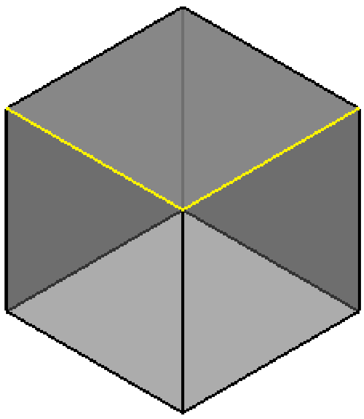
  Press and hold the **Ctrl** key and click to add/remove tracks from the current selection. You can also use **Ctrl+click** to join separate tracks into a single track and to split a single track into two separate tracks.

  If the **Follow Continuous Edges** option is selected and you use **Shift+click** or **Ctrl+click** on the selected path, the whole path is de-selected.

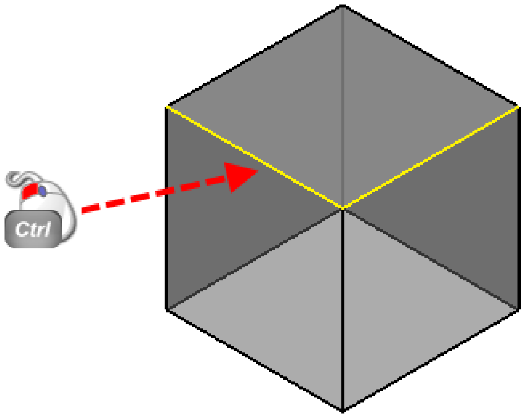
  In the example, deselect the **Follow Continuous Edges** option and select an edge at the end of the path using **Shift+click**.



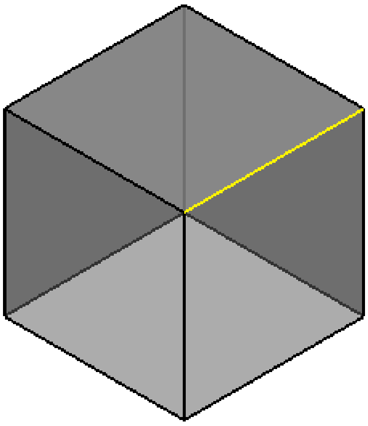
  The edge is now highlighted as part of the path.



Now **Ctrl+click** the edge again.



This removes the edge from the path and we are left with the following.

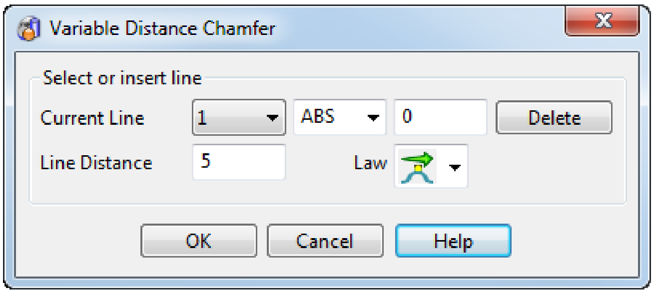


- **Creation of several separate chamfers at once** — You can now create several separate chamfers at the same time. The tracks to be chamfered must be independent, but they can be a combination of concave and convex.
- **Parameterised chamfering** — You can use a parameter instead of a value to chamfer a group of independent tracks. All the parameterised tracks are displayed as separate, editable features in the tree.

  As with all current multiple chamfer track functionality, using a parameter will parameterise all of the tracks and all chamfers appear as separate editable features in the tree.

Variable Distance Chamfer dialog

Use this dialog to inserts lines along the chamfer and edit the length values. This also determines how the width of the chamfer varies from one line to another.



Current Line - This has two drop-down lists and a text box.

The first drop-down list displays the selected line. You can use this to select existing lines. To find the number of an existing line, select the line in graphics window. Its number is displayed in this drop-down list box.

Use the second drop-down list and text box to enter new lines as follows:

1 Select one of the following options from the second drop-down list to determine where to input the new line:

   ABS — A new line is inserted along the chamfer at a distance from the first line.

   REL — A line is inserted along the chamfer at a distance from the selected line in the first drop-down list box.

   PAR — A new line is input at a parametric distance between two existing lines. For example, a parametric distance of 3.5 inserts a line half way between lines 3 and 4.

2 Enter a distance in the text box and press *Enter* to specify a new line. Alternatively, click a position on the chamfer route to enter a new line.The lines are relabelled.

If you hover the intelligent cursor over the chamfer route, a distance is displayed. The option selected in the second drop-down list box will determine what the value means. For example if **ABS** is selected, the value is the absolute distance along the chamfer route.

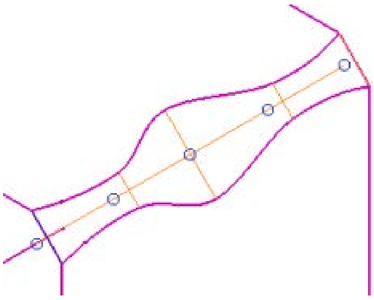
Delete — This deletes the selected line. Note that end lines cannot be deleted.

Line distance — This displays the distance of the selected line. You can change its value by

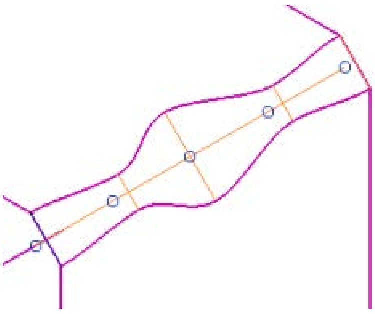
- entering a new value
- dragging the line on the screen to the required value

Law — This displays the variation of the distance at the current line position. The **Law** options use one of the following variation methods for the distance at the selected line.

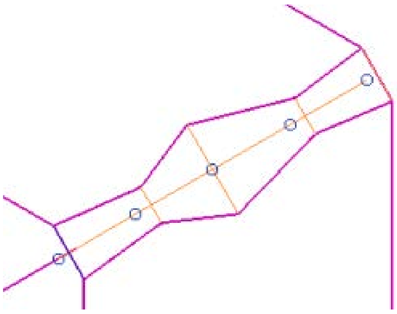
- **Free** — The distance varies freely from the line and then takes on the conditions of the neighbouring line.



- **Horizontal** — The distance remains constant for a short distance from the line and then takes on the conditions of the neighbouring line.



- **Sharp** — The distance changes size as soon as it leaves the line and then takes on the conditions of the neighbouring line.



The **Law** option uses a combination of the distance methods to define nine options. The first word specifies how the distance varies as it enters a line; the second word specifies how the distance varies as it leaves a line:

 — Free Free

 — Free Horizontal

 — Free Sharp

 — Horizontal Horizontal

 — Horizontal Free

 — Horizontal Sharp

 — Sharp Sharp

 — Sharp Free

 - Sharp Horizontal

OK — This creates the variable distance chamfer and takes you back to the **Chamfer** dialog.

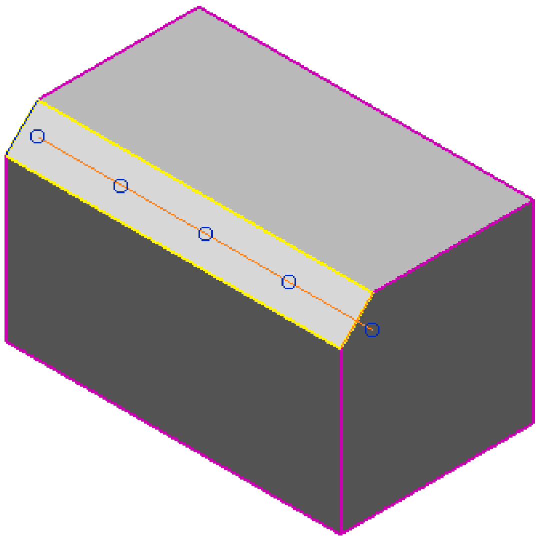
Cancel — The chamfer is not created and you return to the **Chamfer** dialog. The path is still selected.

Creating a variable chamfer

The **Variable** option is only available once you have clicked **Apply** to create a chamfer. If you have only selected the path, but not created the chamfer yet, you can select the **Advanced** button to create a variable distance chamfer. If you select this option, you can now edit the distance along the last chamfer created.

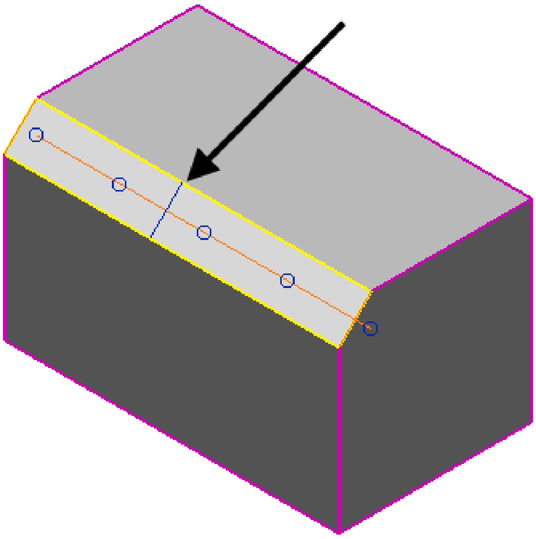
1 Create a chamfer and click **Apply**.

2 Select **Variable**. The route of the chamfer is now displayed.

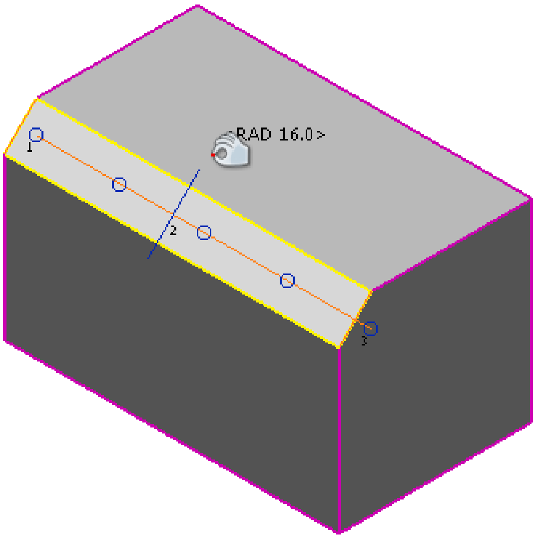


If you have a closed chamfer, then this option will remain deselected until you insert a line.

3 Insert new distance values along the chamfer by clicking a position on the chamfer to insert a line. This line uses the **Distance** value that is specified in the Chamfer dialog.



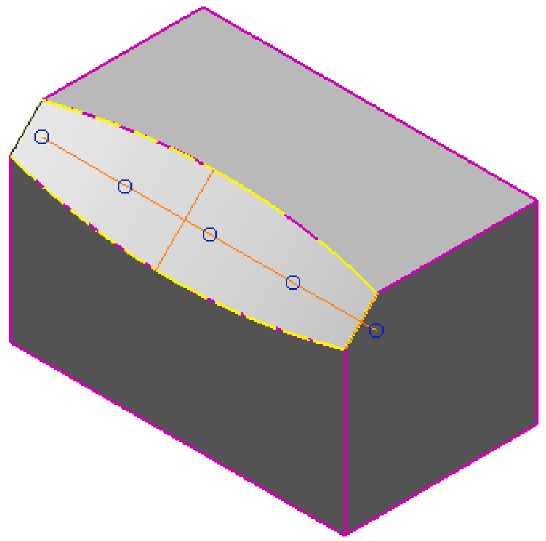
To change the distance of the selected line, drag the line or enter a **Distance** value.



To delete the selected line, press **Delete.** End lines cannot be deleted.

To select a line on the chamfer, simply click it.

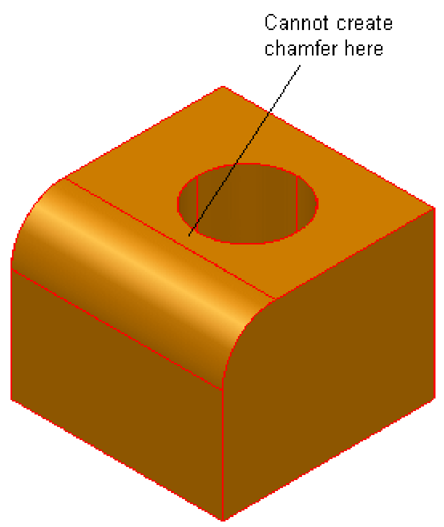
4 Click **Apply** to update the chamfer.



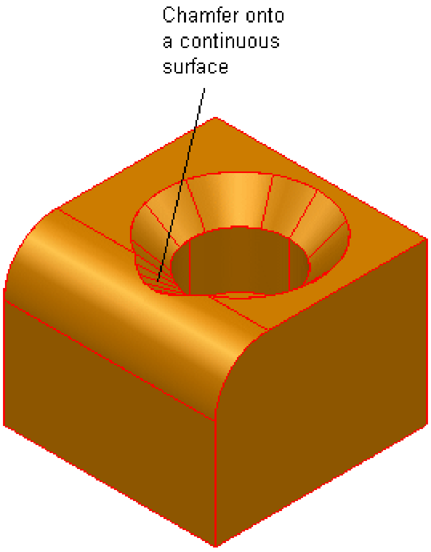
You can continue to edit the chamfer if you want or select another chamfer path.

Add Adjacent Continuous Faces — If a chamfer cannot be created because it spreads onto neighbouring surfaces, turn this option on.

The chamfer will not spread beyond the neighbouring surfaces. To spread the chamfer beyond onto all surfaces, use the **Add All Continuous Faces** option.



This option will allow you to create the chamfer onto the continuous surfaces.



Add All Continuous Faces - This is the same as the **Add Adjacent Continuous Faces**, except the chamfer can spread onto all surfaces that are tangent plane continuous.

This option may be much slower when creating chamfers. We recommend you use the **Add Adjacent Continuous Faces** if you recognise that a chamfer will not spread beyond its neighbouring surfaces.

If you have finished using this option and it is no longer needed, we advise you to turn it off.

  — Undo / Redo a chamfer. The normal undo behaviour has been retained once the dialog is removed from the screen. So, selecting  **Undo** from the main toolbar immediately after creating a chamfer or group of chamfers will remove all the chamfers that were created.

Apply — Creates a chamfer along the selected path. The dialog remains on the screen for you to select more edges and continue creating chamfers.

A **Chamfer feature** icon  is added to the solid feature tree to represent the chamfer opertaion.

You can now either edit the distance of the chamfer just created or select more edges to create another chamfer.

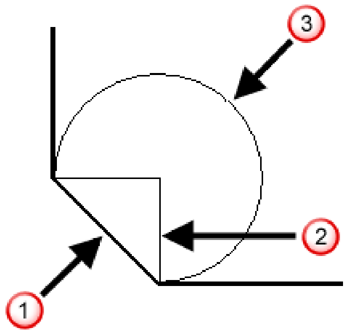
Advanced — While a path is selected, you can create a variable distance chamfer. Click this button to display the **Variable Distance Chamfer** dialog. Use this dialog to insert lines along the path to define the different distance values.

OK — Removes the dialog from the screen.

Creating a chamfer using a single distance

1 Click .

2 Enter the **Distance** to be used to create the chamfer as shown below.



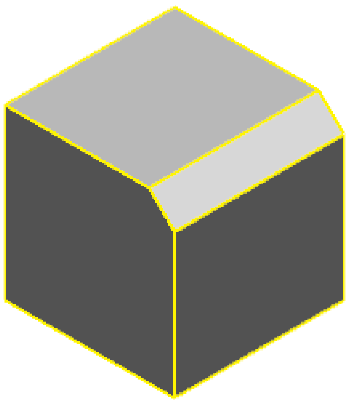
1 Chamfer.

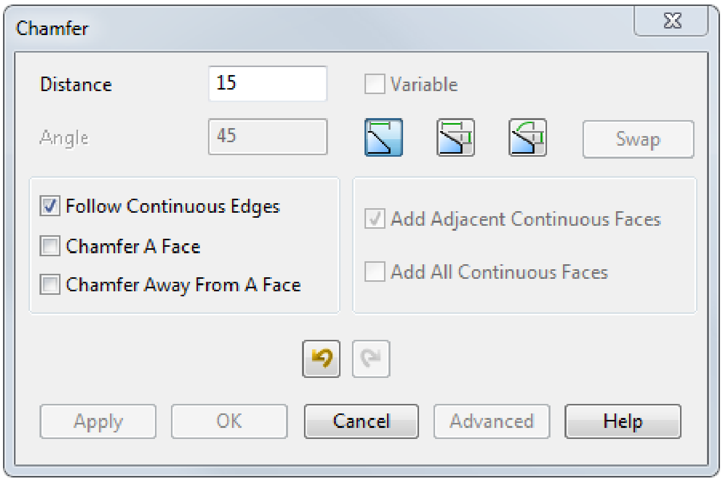
2 Distance.

3 Arc with radius equal to distance.

3 Select an edge of a solid.

4 Click **Apply** to create a chamfer with a single distance and a radius of 45°. The chamfer on the following model was created by entering a **Distance** of 15.





Creating a chamfer using two distances

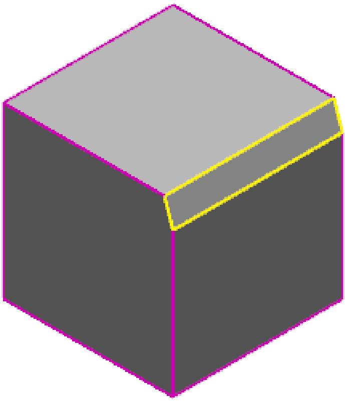
1 Click 

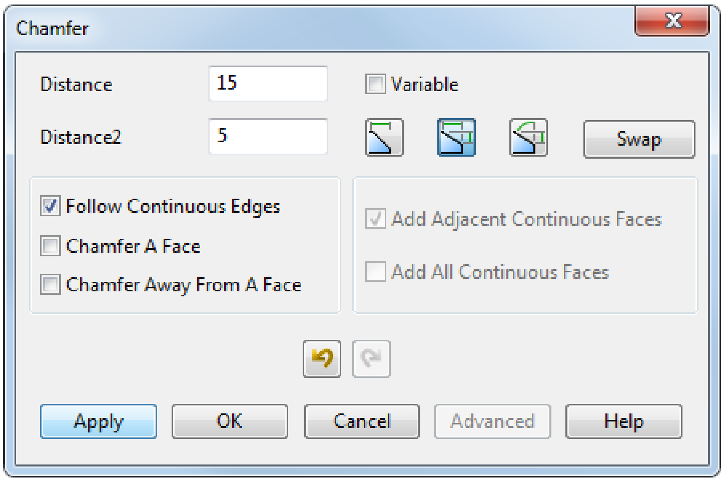
2 Enter a value for **Distance**.

3 Enter a value for **Distance 2**.

4 Select an edge of a solid.

5 Click **Apply** to create a chamfer with two distances. The chamfer on the following model was created by entering a **Distance** of 15 and **Distance 2** of 5.





 *Click **Swap**  to swap the effect of unequal distances.*

Creating a chamfer using a distance and an angle

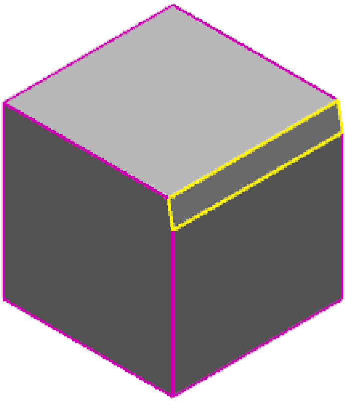
1 Click 

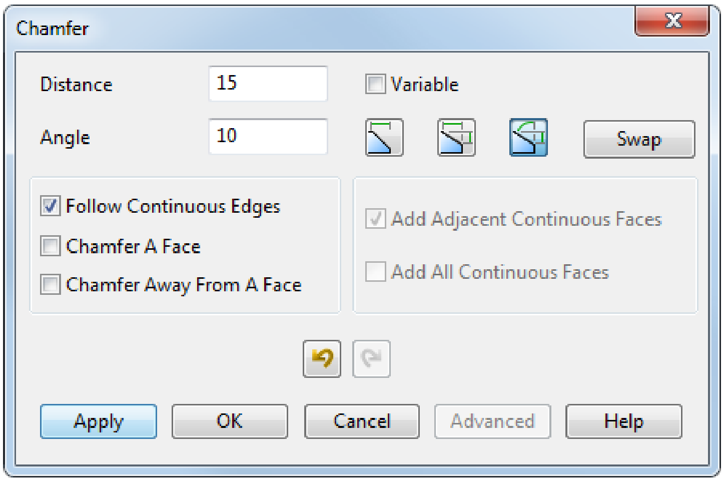
2 Enter a value for **Distance**.

3 Enter an **Angle**

4 Select an edge of a solid.

5 Click **Apply** to create a chamfer with a distance and an angle. The chamfer on the following model was created by entering a **Distance** of 15 and **Angle** of 10.





 *Click **Swap**  to swap the effect of unequal distances.*

Editing a solid chamfer during creation

You can edit a chamfer using the **Chamfer** dialog. Remember that you can keep editing the chamfer after you click the **Apply** button.

To edit the distance of a chamfer with constant distance:

1 Enter a new value in the **Distance** box.

2 Click **Apply**.

To make a chamfer with constant distance into one with a variable distance:

1 Click the **Variable** button.

2 See the **Variable** option above for details on how to insert new distance values along the chamfer.

3 Click **Apply**.

To make a chamfer with variable distance into one with a constant distance:

1 Turn off the **Variable** option.

2 Enter the new distance value in the **Distance** box.

3 Click **Apply**.

To change the distance of a chamfer with variable distance:

1 Keep editing the distance by inserting lines along the chamfer.

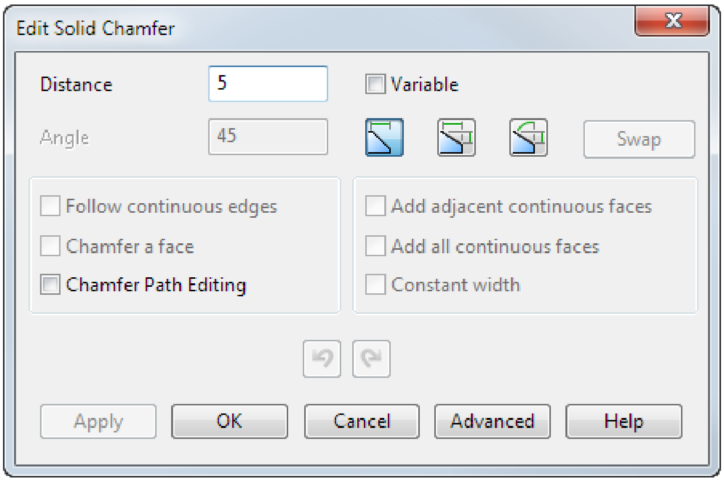
2 Click **Apply**.

Editing a solid chamfer

You can edit an existing chamfer or edit multiple (see page 247)chamfers.

To edit a fillet:

1 Double-click the **Chamfer feature** icon  in the tree to display the **Edit Solid Chamfer** dialog.



2 Use the dialog to edit the chamfer, you can:

- Change a variable distance chamfer to a constant one, simply turn off the **Variable** option and enter a new value in the **Distance** box.
- Change a constant distance chamfer to a variable one, select **Advanced** to display the Variable Distance Chamfer dialog (see page 215) or select the **Variable** option and insert new radius values along the fillet.
- Select one of the following options to recreate the chamfer using:

   — a single distance

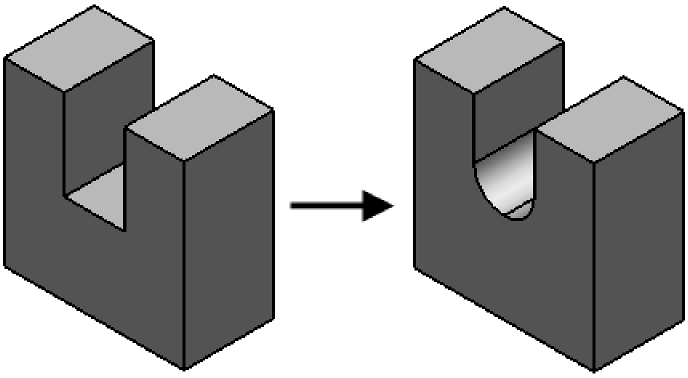
   — two distances

   — a distance and an angle

- Edit the path of the chamfer, click the **Chamfer Path Editing**. The original path of the chamfer is highlighted; adjust the existing path or create a totally new one.
- Edit the chamfer to have a constant width by toggling the **Constant width** option.
- Select a new path for the chamfer as if creating a new chamfer.

Creating a rib fillet

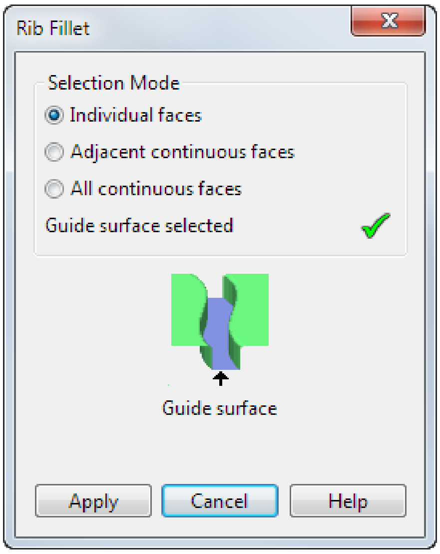
If you have a face of a solid surrounded by walls, you can replace the face by a rib fillet.



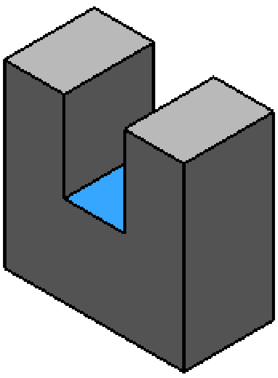
The new fillet is tangent to the walls and its base touches where the face used to lie.

To create a rib fillet:

1 Click  (*Solid feature toolbar*).



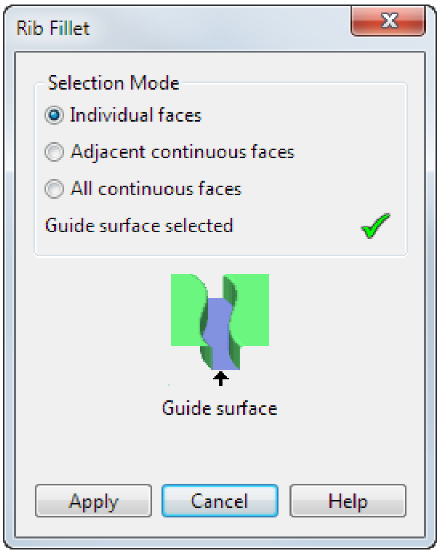
2 Select the faces you want to fillet:



3 Click **Apply** to close the dialog and create a rib fillet on the solid.

Rib Fillet dialog

This creates a rib fillet on a solid.



Selection Mode - Select a face on the solid as a guide surface. The fillet is created along this surface.

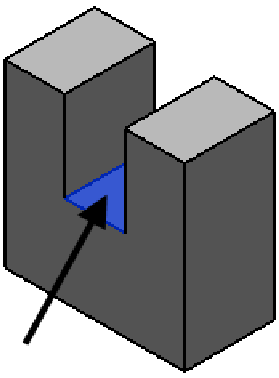
Individual faces - Only selects the face clicked.

Adjacent continuous faces - Selects the face clicked and all faces which are adjacent and tangent plane continuous to the clicked face.

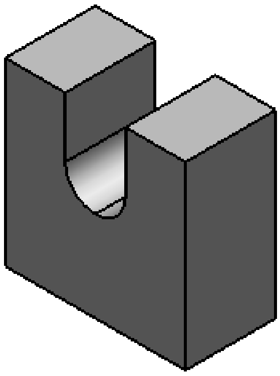
All continuous faces - Selects the face clicked and all faces which form a tangent plane continuous surface with the clicked face.

*Hold down the **Ctrl** key and click faces to add and remove them from the current selection.*

Select the following face on the solid. The ✘ changes to ✓. This is the guide surface.



Apply - Saves the rib fillet feature along the selected face.



A **Rib fillet feature** icon representing the operation appears in the solid feature tree.

Creating a Pocket or Protrusion Feature

1 Click **Solid Pocket** or **Protrusion** (*Solid feature toolbar*).

2 Use the cursor to position the origin of the feature, or enter coordinates into the status bar.

3 Complete the options on all pages of the **Pocket** dialog. Whilst the dialog is displayed, 3D instrumentation is displayed on the feature.

   As an alternative to entering the information into the dialog, you can use the handles to define the length, width and height of the feature. You can also define the corner fillets using the handles in each corner of the feature. The value boxes on the dialog are updated to reflect the settings defined by the instrumentation.

4 Click **OK** to create the pocket or protrusion. The pocket/protrusion feature icon is added to the solid tree.

When creating a pocket or protrusion, the values you enter in the ***Pocket/Protrusion*** *dialog are remembered when you click* ***Apply*** *or* ***OK****. This means that the next time you create a protrusion or pocket, these values will be the default values.*

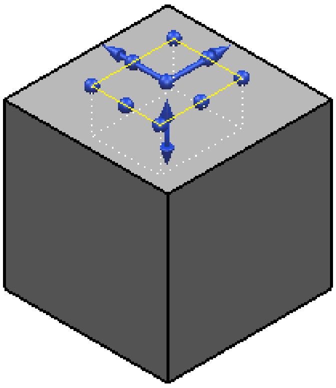
Pocket dialog

The four pages of the **Pocket** dialog let you define the pocket/protrusion feature. Each page is described separately.

- Dimension page (see page 229)
- Corners page (see page 232)
- Fillets page (see page 234)

- Plane Details page (see page 237)

*If the definition of the feature is invalid (for example if the fillet radius is too large), the **Apply** button is greyed out and the graphic will show a dotted outline of the feature.*

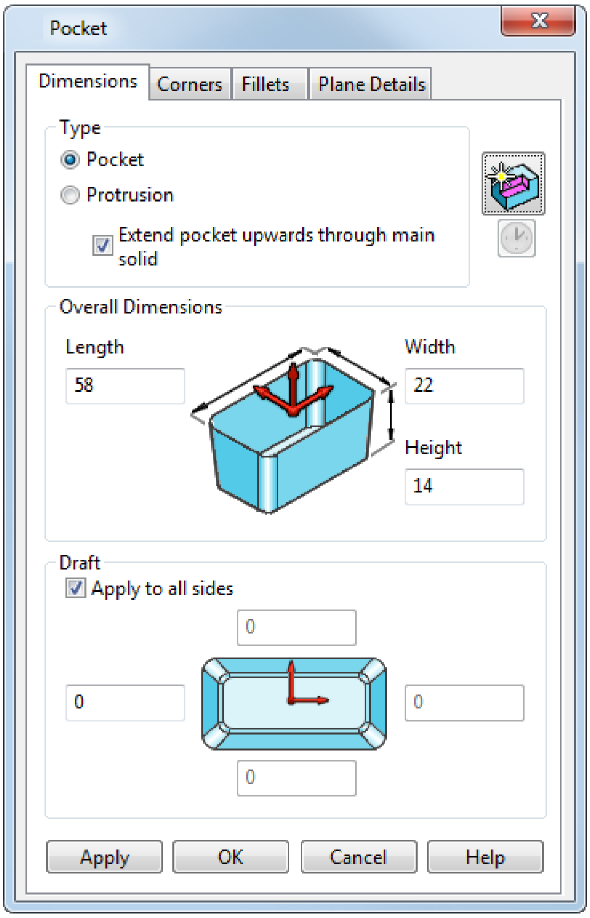


*If you are creating or editing a protrusion, the dialog will be renamed **Protrusion**.*

You can also simultaneously edit multiple pockets or protrusions (see page 247).

Pocket dialog - Dimensions page

This page lets you define the dimensions of the pocket or protrusion.

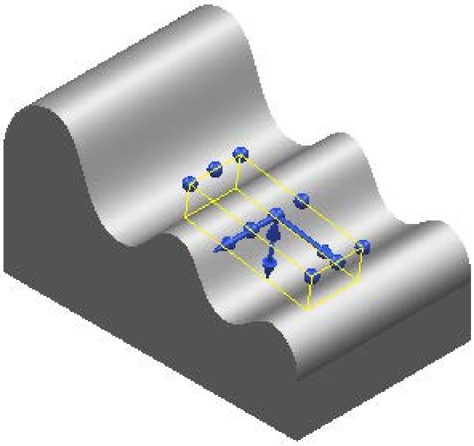
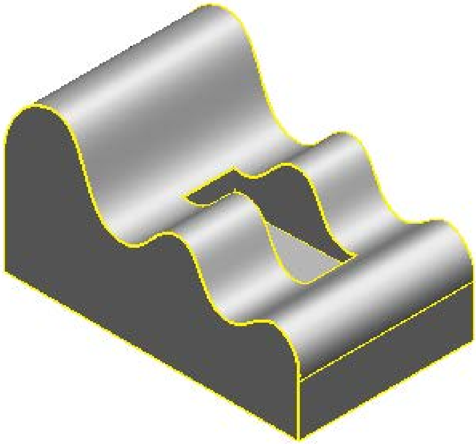


Type - Select the option to define the type of feature you want to create.

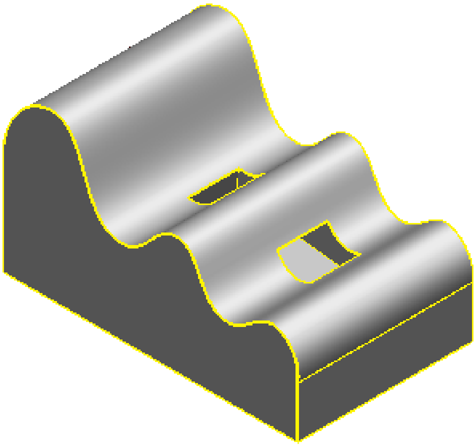
Pocket - Select this option to create a pocket feature.

Protrusion - Select this option to create a protrusion feature.

Extend pocket upwards through main solid - If *ON*, the pocket solid will be extended upwards, so that it cuts through all surfaces in the main solid as shown below.

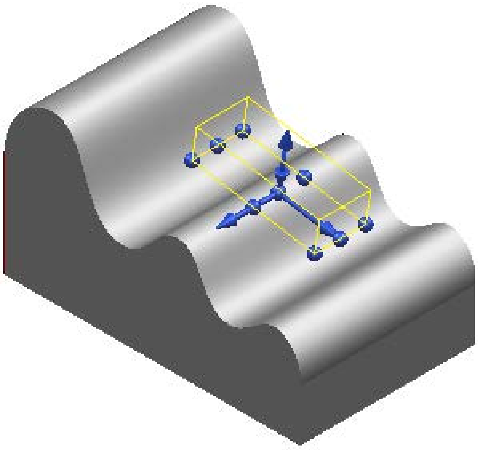
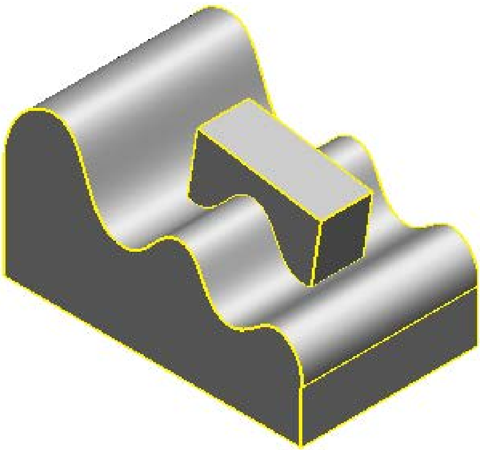



If *OFF*, the pocket is not extended upwards and the same model would look like this:

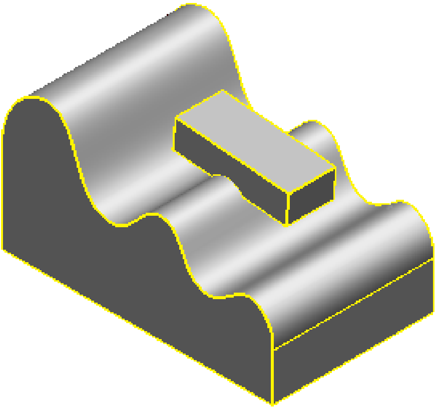


*If you are creating a protrusion, the option is called **Extend protrusion onto main solid**.*

Extend protrusion onto main solid - If *ON*, the protrusion solid is extended downwards so that it meets all surfaces below it in the main solid.

If *OFF*, the protrusion is not extended onto the main solid and the same model would look like this:



If you deliberately place a protrusion over a hole in a solid, set the **Extend protrusion onto main solid** *to* OFF *so that the protrusion does not attempt to fill in the hole.*

 lets you swap between creation mode and editing mode without closing/reopening the dialog.

- When in creation mode, click  to swap to editing mode.
- When in editing mode, click  to swap to creation mode.

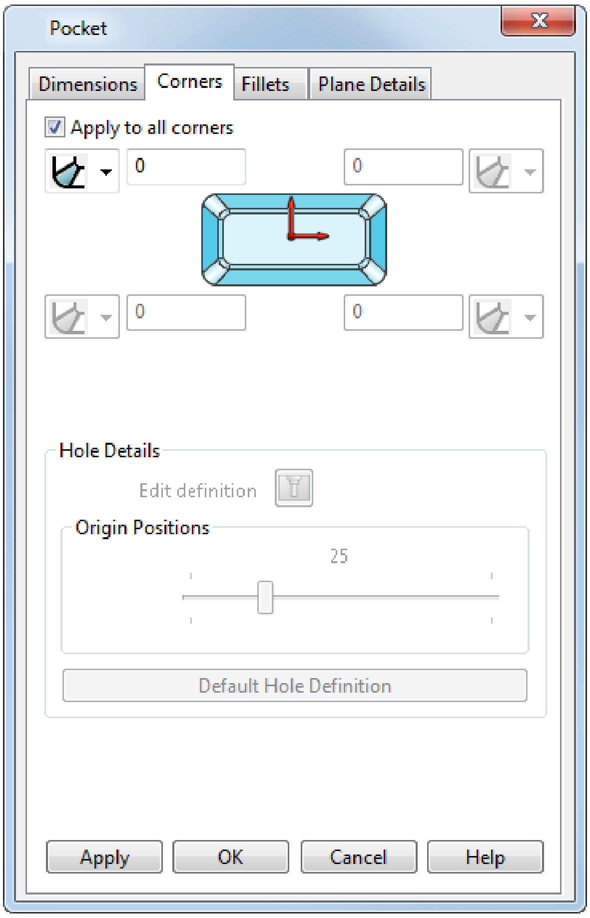
Overall Dimensions - The **Length**, **Width** and **Height** of the feature can be entered in the boxes.

Draft - Each side face of the feature can have a draft angle.

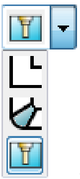
Apply to all sides - If *ON*, enter one draft angle to be applied to each side wall. If *OFF*, enter separate draft angles for each side wall.

Pocket dialog - Corners page

Use this page to define the type of corner finishing on the pocket.



Apply to all corners - If *ON*, the corner details you have specified are applied to all corners. If *OFF*, each corner is defined separately.



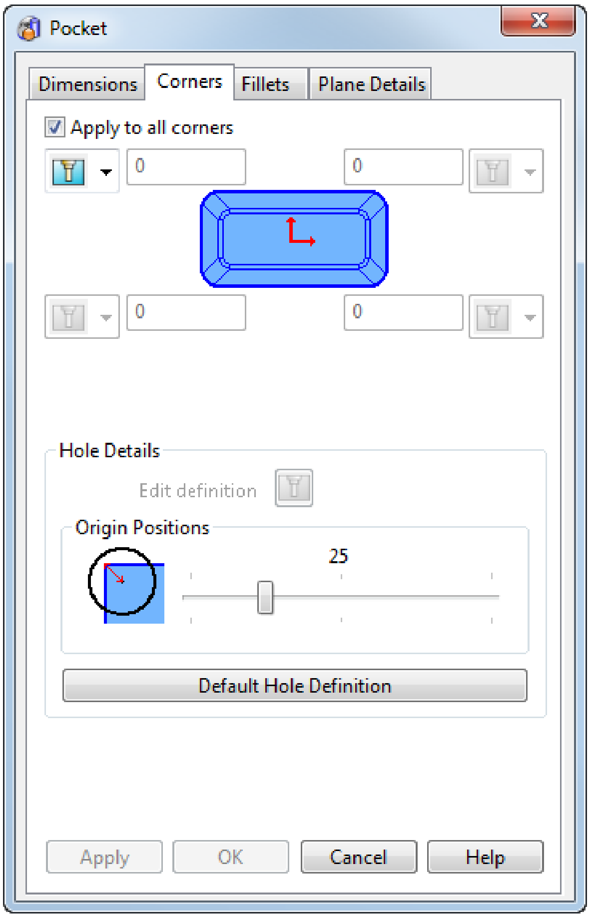
Select the type of corner finishing from the drop down list.

 - Do not apply any corner finishing.

 - Fillet the corner edge. Enter the fillet radius for the corner in the text box.

 - Create a corner hole. When you select this option, the Hole Details section of the dialog becomes active so that you can define the size, type and position of the hole.
You can only create a corner hole if you are defining a pocket.

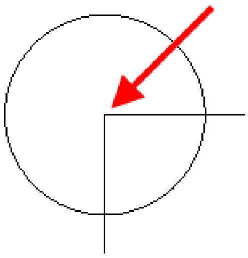
Hole Details - If you have created one or more corners of a pocket as a hole, you can define the size and position using this section of the **Corners** page.



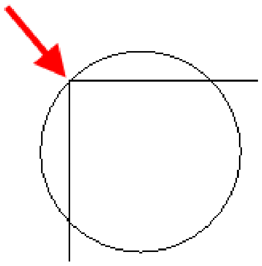
Edit definition - Click on  to make changes to the size and type of hole using the **Hole** dialog.

Origin Positions - Move the slider to position the origin point of the hole in relation to the corner of the pocket. The position is expressed as a percentage of the hole's largest radius.

If the slider is positioned at *0*, the origin of the hole is centred on the corner of the pocket as follows:



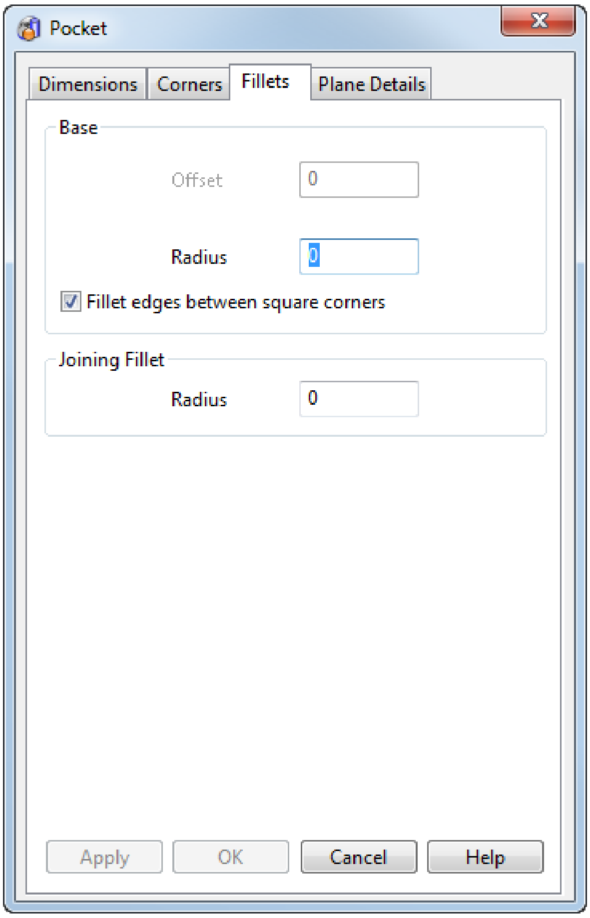
If the slider is positioned at *100*, the origin of the hole is positioned at the length of the radius away from the corner of the pocket as follows:



Default Hole Definition - Creates a default plain hole, based on the current height of the pocket.

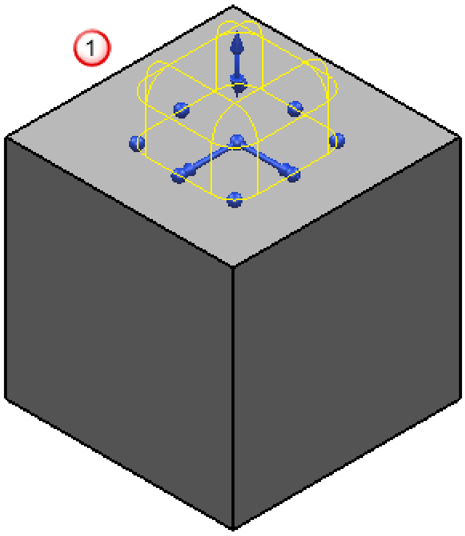
Pocket dialog - Fillets page

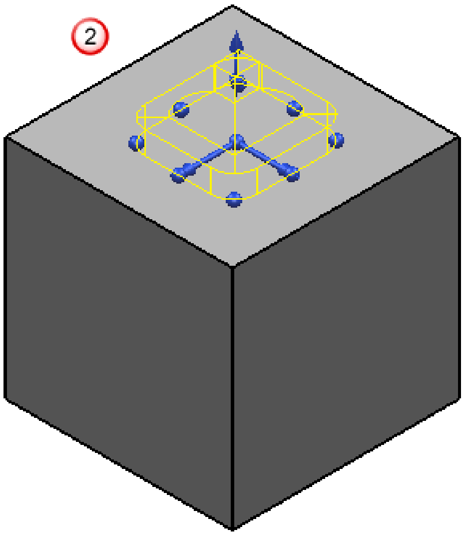
This page lets you define the fillet running along the base of the pocket or top of the protrusion.



Base / Top - The option is called **Base** if you are creating a pocket and **Top** if you are creating a protrusion.

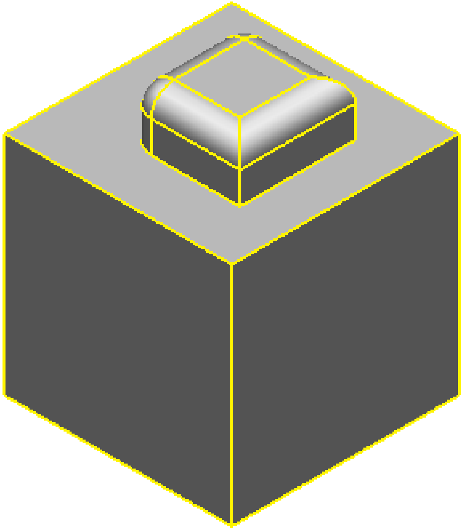
Offset - This option is only active if you are creating a protrusion. You can use it to create an offset fillet around the top of the protrusion. The fillet is offset from the top of the protrusion by the value entered in the text box. The models below show the results when **Offset** is *0* 1 and **Offset** is *5* 2.



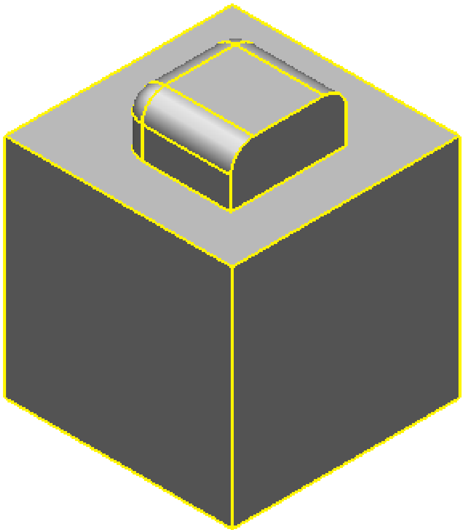


Radius - Enter the radius of the fillet around the base of the pocket or the top of the protrusion.

Fillet edges between square corners - If *ON*, fillets will be created around the whole of the base of the pocket, or top of the protrusion.

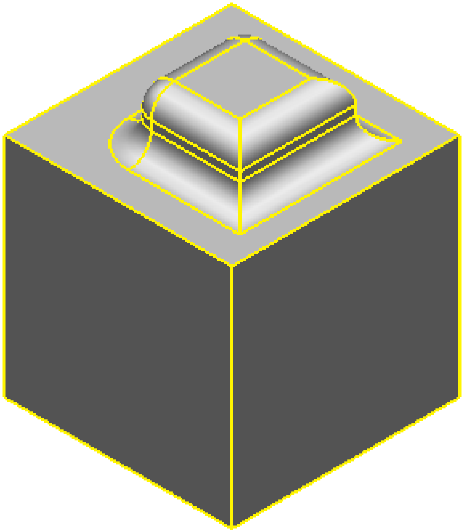


If *OFF*, edges with square corners will not be filleted.



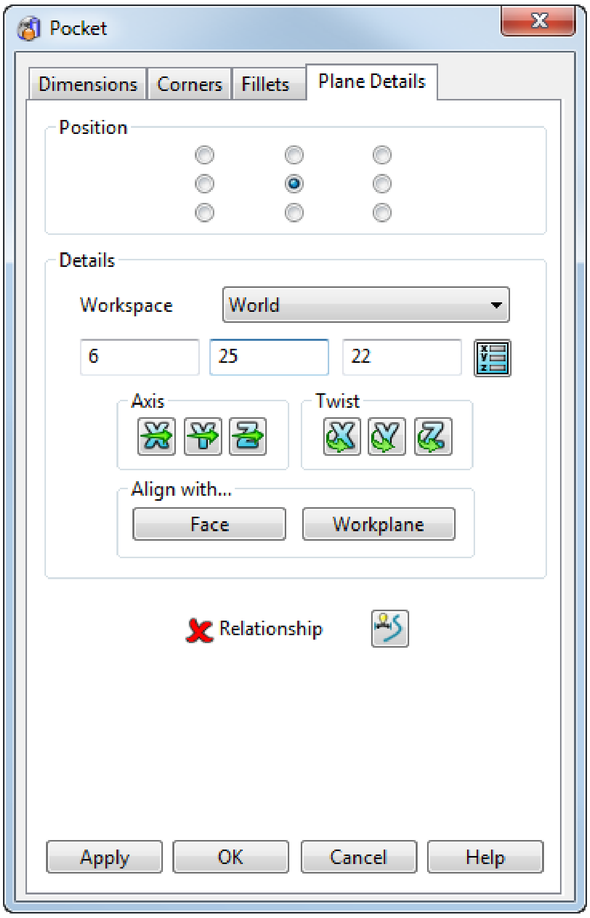
Joining Fillet

Radius - Enter a fillet radius to define the radius where the pocket/protrusion joins the solid.



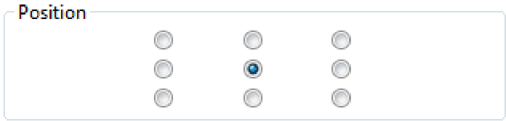
Pocket dialog - Plane Details page

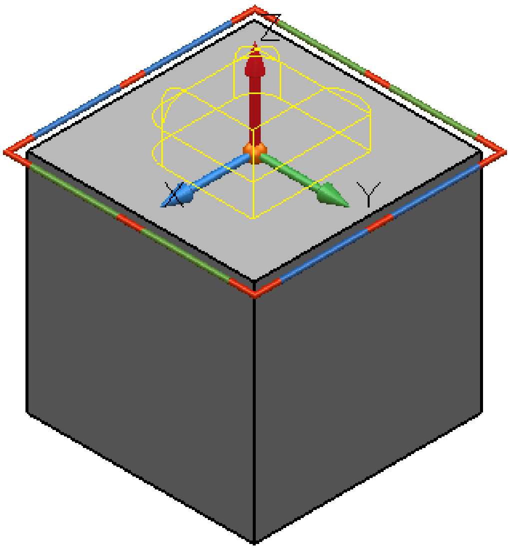
This page lets you define the workplane for the feature.



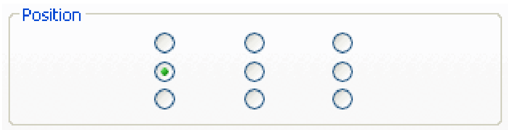
Position - The location of the workplane on the feature is defined using this panel.

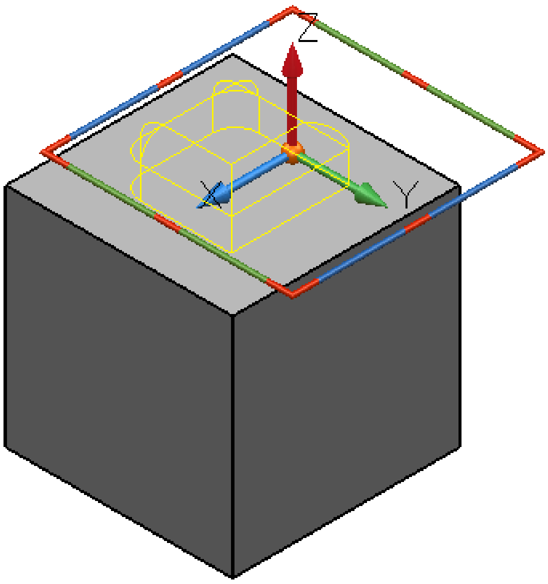
The default setting will position the workplane in the centre of the feature.





Selecting an alternative setting will define the workplane accordingly. The example below defines the workplane at the middle of the left edge of the pocket/protrusion.





Details - The workplane for the feature is defined using these options. The usual workplane instrumentation is displayed whilst this page of the form is selected.

Relationship - Define the relative position of the pocket or protrusion with respect to the solid by selecting the **Solid Feature Relationship** button.

Creating a User Defined Feature

User Defined Feature can be used to create a solid feature from faces of a solid. This feature is included in the solid feature tree and can be suppressed/unsuppressed and renamed.

To create a User Defined Feature,

1 Create a solid.

2 Select **User Defined Feature** (*Solid feature toolbar*). The **Create User Defined Feature** dialog is displayed.

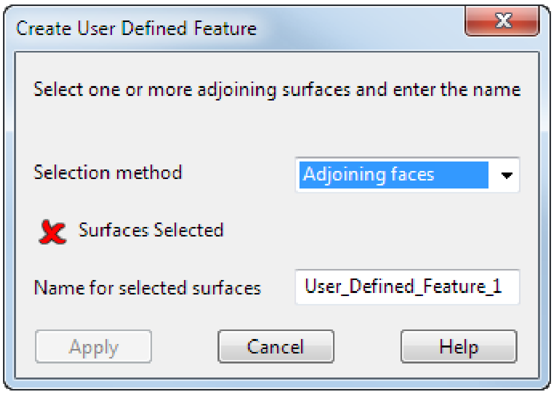
3 Select the area of the solid that you want to turn into a user defined feature.

4 Select your options on the dialog.

5 Click **OK**.

Create User Defined dialog

Use this dialog to specify the surfaces to be used to create the user-defined feature.



Selection method - Select from the following options:

Adjoining Faces

Concave Faces

Convex Faces

Tangent Continuous Regions

Single Face

Surfaces Selected - When you select the area of the solid to turn into the user-defined feature, the ✘ changes to a ✔.

Name for selected surfaces - Enter the name of the feature.

Apply - Create a user-defined feature from selected surfaces.

Scaling constraints for user defined features

Scaling constraints can only be applied to version 8 solids. If your model contains post-version 8 solids, they will need to be converted manually (see page 21) before constraints can be applied.

You can apply constraints to user-defined features that you wish to scale. It is advisable to use small scaling factors (for example, 1.2, 1.3) when you are using scaling to ensure that the stitch margin is able to produce a smooth and acceptable solution.

Scaling constraints let you keep certain entities locked whilst scaling the rest. They are used where graded copies of some geometry are required but portions of the geometry are to be kept constant irrespective of the scale factor that is being applied.

Examples

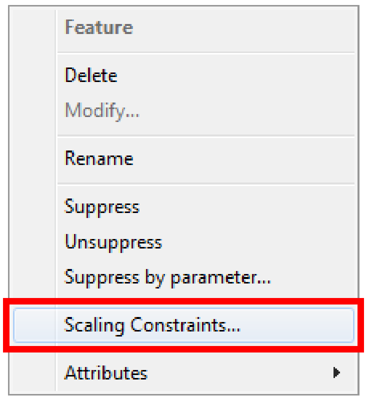
You could use scaling constraints in the following situations:

- Bottles; scale the whole bottle but keep the neck of the bottle the same size.
- Shoes; scale the shoe sole but keep the airbag or logos the same size.

Applying scaling constraints will not make any immediate change to the model that is displayed, but will be used when you scale the model

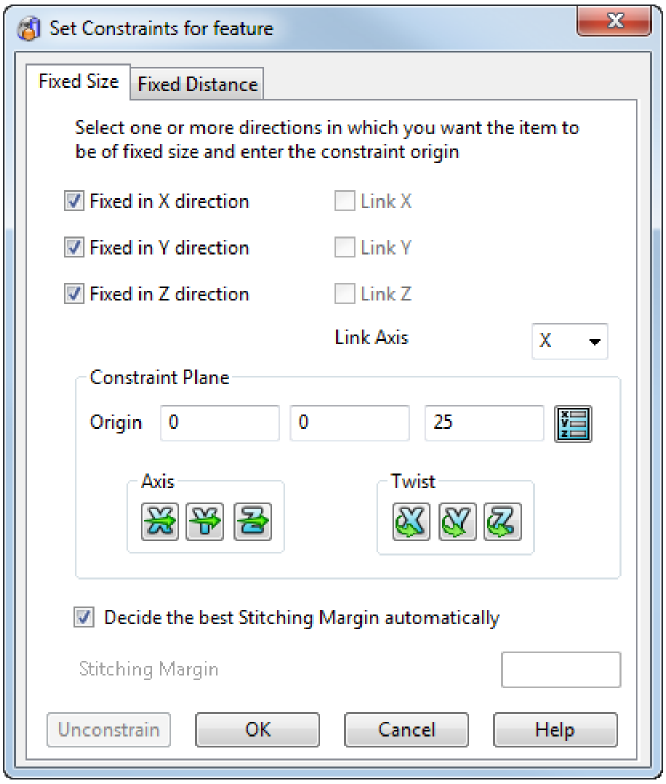
Defining scaling constraints for a user defined feature

1. Right-click on a **User Defined feature** in the tree window.
2. Select **Scaling constraints** from the feature context menu.



The Set constraints for feature dialog (see page 241) is displayed.

Set constraints for feature dialog



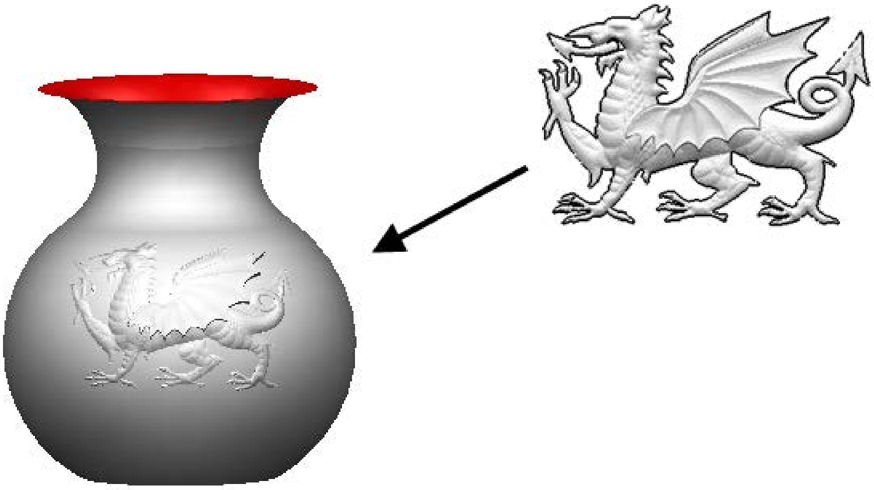
The options on this dialog are the same as those on the Set Constraints for solid dialog (see page 94).

You can define the scaling constraints in one of the following ways:

- Fixed Size (see page 95) - Use this method to select the directions that are to remain unaltered by scaling.
- Fixed Distance (see page 97) - Use this method to maintain the relative position of two solids or two user-defined features.

Creating a wrap feature

You can wrap surfaces and solids by wrapping a triangle mesh onto them using the **Wrap Wizard**.



The wrapping method is the same as **Wrap Triangles**  from the **Surfaces** toolbar, except the wrap is created as a solid feature displayed in the feature tree. This enables the wrap feature to be edited.

Wrap features are:

- always placed at the top of the feature tree.
- preserved when you offset a solid.

To wrap objects,

1. Use the Mesh Creator in ArtCAM Pro to create a *dmt* file of the relief you want to wrap onto objects.

   ArtCAM Pro is a Delcam software program which allows you to easily create impressive, high quality 3D products starting from 2D bitmap or vector based artwork.

2. In PowerSHAPE, create the surfaces or solids on to which you want to wrap the feature.

3. If necessary, edit the image to be wrapped so that it fits on the surfaces and solids.

4. Click **Create a Solid Wrap Feature**  (*Solid feature toolbar*).The **Wrap Wizard** starts and displays the Wrapper Selection page.

   The wrappers must be all symbols or all wireframe.

For further details, see What is wrapping? in the Surface modelling manual.

How do I edit a wrap feature?

To edit a wrap feature:

1 Double-click the **wrap feature**  icon in the solid tree. This starts the **Wrap Wizard**.

2 Use the **Wrap Wizard** to edit the wrap feature.

For further details, see What is wrapping? in the Surface modelling manual.

Solid transform feature

Create solid transform features on solids so that values can be changed, or the transform suppressed, unsuppressed or deleted. Information about changes of position, rotation, scaling or mirroring are stored with the solid.

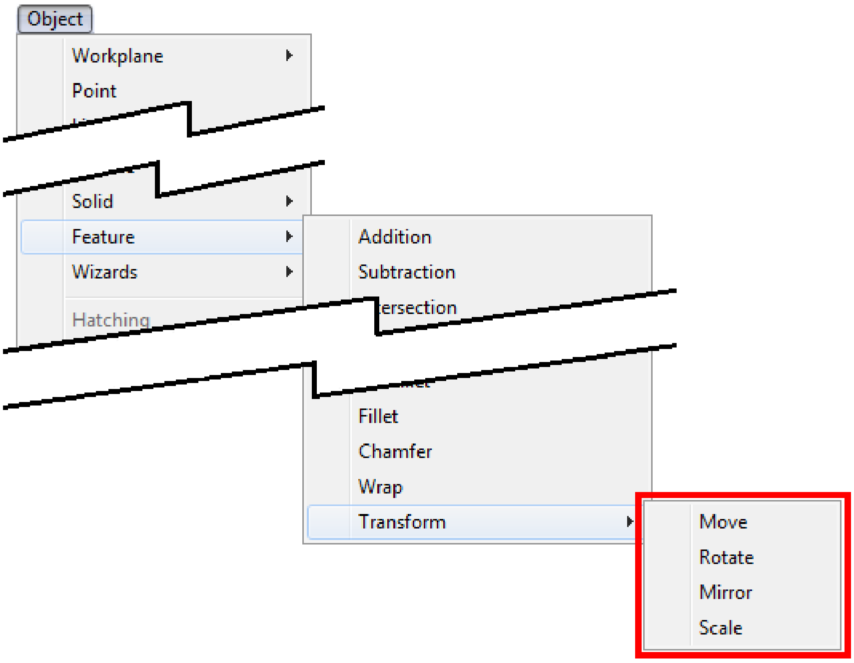
Creating transform features

Transform features can be created by:

- Selecting one or more solids and selecting **Move**, **Rotate** or **Mirror** from the **General Edits** toolbar when the option **Tools > Options > General Edits > Create solid transform features** is selected.

Scale transform feature will not *be created if you select Scale from the General Edits toolbar, even if the **Tools > Options >General Edits > Create solid transform features** is selected.*

- Selecting one of the options on the **Object > Feature > Transform** menu. These options will create a transform feature on the active solid and let you enter the parameters and show a preview of the transformed solid.



The following specific dialogs are displayed when using this method to create the transform feature:

Move feature (see page 245)

Rotate feature (see page 245)

Mirror feature (see page 246)

Scale feature (see page 246)

Editing a transform feature

1. Use one of the following methods to display the relevant transform feature dialog.
   - Double-click the transform feature in the solid tree.
   - Select **Modify** from the feature's context menu.

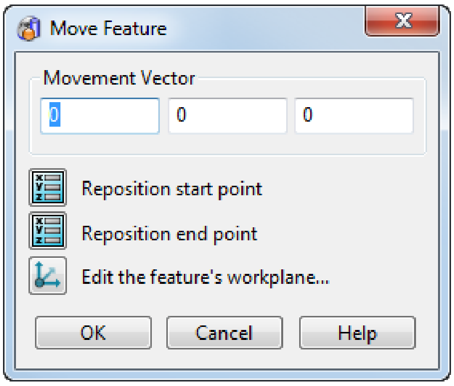
   The transform feature dialogs will be displayed irrespective of how the transform feature was created.
2. Use the dialog to modify the feature.
3. Click **OK** to make the changes.

Tips when using transform features

- When using the **General Edits** toolbar, transform features will not be created on solids that do not have history trees, whatever the setting on **Tools > Options > General Edits**.

- If you use **Copy original**, the original won't have a feature associated with it, but the copies will.
- Features below a transform feature in the solid tree are not transformed. When modifying an earlier feature it will appear in its original state.
- Transform features can be suppressed, deleted and reordered. If the transform is suppressed or deleted, the solid will revert to its original state (before transformation).

Move feature



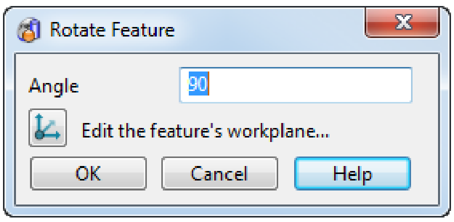
1 **Movement Vector** defines the distance to be moved along each axis. These axes refer to the feature's workplane.

2 To define the translation by clicking on the model, use  to display the **Position** dialog to **Reposition start point** and **Reposition end point**.

3 Click  to edit the feature's workplane using the **Workplane** dialog. The feature's workplane acts as a reference point for the transform.

4 Click **OK** to create the transform feature. The move feature  is added to the feature tree.

Rotate feature

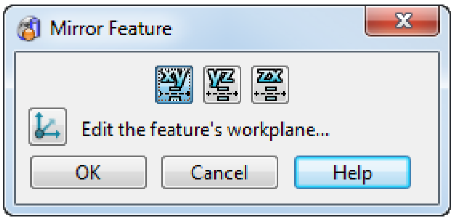


1 Enter an angle of rotation as measured around the Z axis of the feature's workplane. A preview of the rotated solid is displayed with the existing position.

2 Click to edit the feature's workplane using the **Workplane** dialog. The feature's workplane acts as a reference point for the transform.

3 Click **OK** to create the transform feature. The rotate feature is added to the feature tree.

Mirror feature



1 Use to change the plane in which the mirror transform takes place.

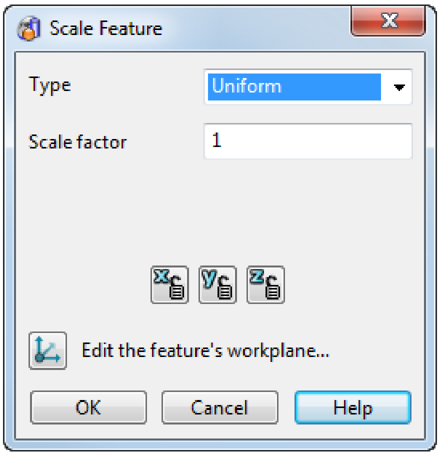
2 Click to edit the feature's workplane using the **Workplane** dialog. The feature's workplane acts as a reference point for the transform.

3 Click **OK** to create the transform feature. The mirror feature is added to the feature tree.

Scale feature

A scale feature can only be created if you select **Object > Feature > Transform > Scale** to display the **Scale Feature** dialog. It will *not* be created if you select from the **General Edits** toolbar, even if the **Tools > Options >General Edits > Create solid transform features** is selected.

Use this dialog to specify the scale feature:



1 Use the drop-down list to select the **Type** of scale to be used (**Uniform**, **Non-uniform** or **Projected volume**).

2 Enter a single **Scale** factor for **Uniform** scaling. For **Non-uniform** scaling enter scale factors for X, Y and Z. For scaling **Projected volume**, enter values for **Original volume** and **Desired volume**.

3 Use to lock the scale factor to 1 for a particular axis. The X, Y and Z scale factors refer to the axes of the feature's workplane.

4 Click to edit the feature's workplane using the **Workplane** dialog. The feature's workplane acts as a reference point for the transform.

5 Click **OK** to create the transform feature. The scale feature is added to the feature tree.

*The results that are achieved using transform features may not necessarily be as expected. This is because functionality such as **Scaling Constraints** is not applied. Therefore, if it is essential that the original behaviour is ensured, deselect the **Create solid transform feature** option on the **Tools > Options > General edits** dialog.*

Editing multiple features

You can make identical changes to multiple features of the same type in a solid. This is useful if you want to quickly edit a large project that contains many features.

This function is available for :

- cuts (see page 128) and bosses (see page 143)
- fillets (see page 199) and chamfers (see page 223)
- pockets (see page 227) and protrusions (see page 227)
- holes (see page 154)

Editing of multiple variable radius fillets is not supported.

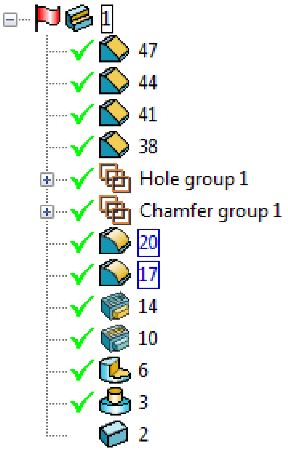
Editing multiple features - example

This example shows how to edit multiple fillets, however the same concepts apply for all families of solid features.

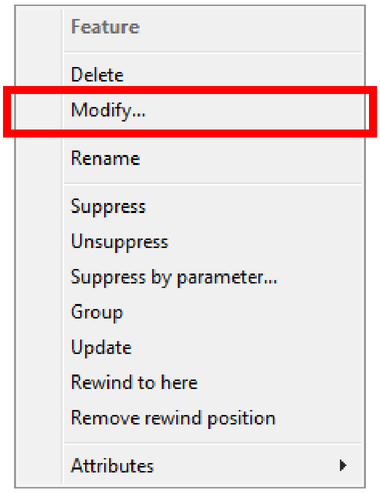
1 Select multiple features you want to edit (for example fillets) using one of the following methods:

- The **Smart Feature Selector** *(Solid Feature toolbar)*

- Hold the control key and select the features in the **solid feature tree**



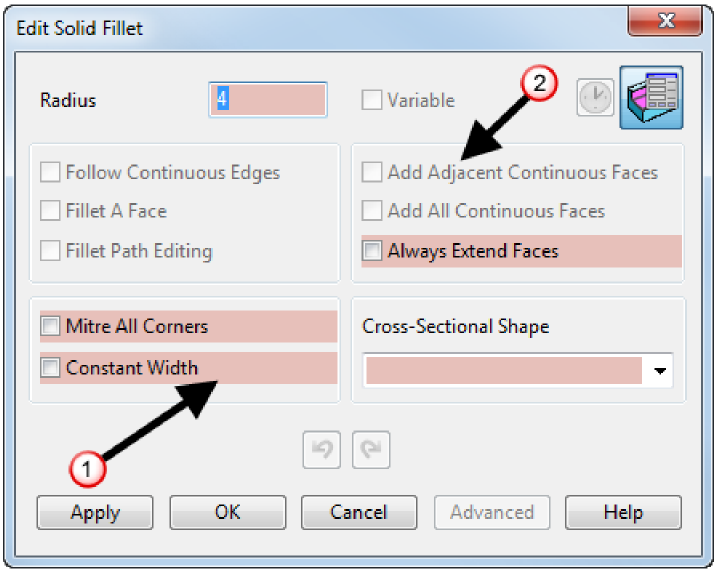
2 Right click one of the highlighted features in the tree and select **Modify**.



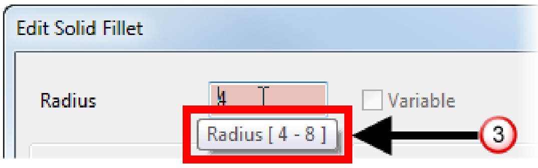
The dialog to edit the appropriate selected features is displayed (for example, the **Edit Solid Fillet** dialog). The dialog is similar to that of editing a single feature of the same type, with the following differences:

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

2 — The properties that cannot be edited for *all* the selected features are unavailable.



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



3 Edit the properties.

4 Click **OK** to apply the change to the selected features and close the dialog.

Creating a relationship between a solid feature and a solid

The relationship between a solid feature and a solid is defined using the **Solid Feature Relationship** dialog. The dialog is displayed by selecting  on the appropriate solid feature dialog.

To create a relationship between a solid feature and a solid, you need to define the following:

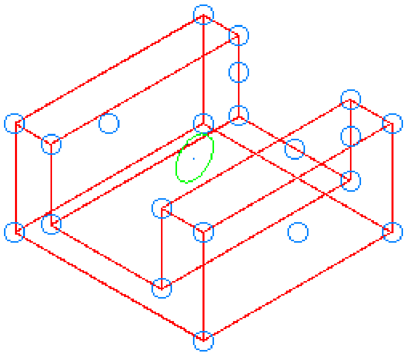
1 A keypoint on the solid.

2 A keypoint on the feature.

3 The distance between the two keypoints.

Defining a keypoint on the solid

1 Click  to attach a workplane to the cursor and display the keypoints on the solid as blue circles.

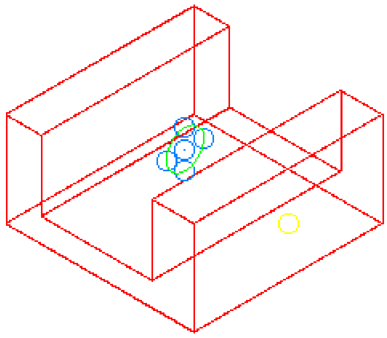


2 Select the keypoint. The unselected points will be removed, leaving only the selected keypoint.

The  changes to a  to indicate that the relationship workplane has been correctly defined.

Defining a keypoint on the feature

1 Click  to display the keypoints on the feature as blue circles.



The keypoints that are displayed will depend on the feature.

- Solid cut and solid boss - the keypoints are points on the sketch and the average centre point of the sketch.
- Boolean features - the keypoints are keypoints on the secondary solid.
- Hole features - the keypoint is always the origin point of the hole. The point button is greyed-out because it cannot be changed.
- Pocket/Protrusion features - the keypoint is always the origin point of the pocket/protrusion. The point button is greyed-out because it cannot be changed.

2 Select the keypoint. The unselected points will be removed, leaving only the selected keypoint.

The  changes to a  to indicate that the keypoint on the feature has been correctly defined.

Defining the distance between the two keypoints

The X, Y and Z values that define the distance between the keypoint on the solid and the keypoint on the feature are shown:

- graphically as dimensions on the model.
- as values on the dialog. The values can be numbers or parameters. If you alter these values, the feature will move.

Solid Feature Relationships

You can specify the position of a some types of feature with respect to the active solid by using the Solid Feature Relationship dialog box (see page 251). You can create a relationship for the following solid features:

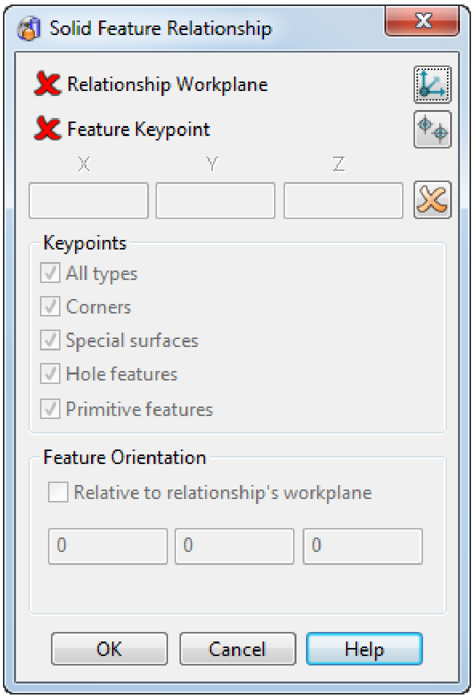
- Solid cut
- Solid boss
- Boolean
- Boolean-boss
- Hole
- Pocket/Protrusion

Solid Feature Relationship dialog

This dialog can be used to define the relative position of the feature with respect to the solid for

- Solid cut
- Solid boss
- Boolean
- Boolean-boss
- Hole
- Pocket/Protrusion

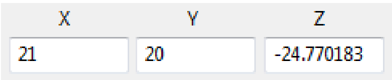
1 Select  on the appropriate solid feature dialog.



2 Use the following options on the dialog to define the relative positions with respect to the solid.

 Position the relationship workplane.

 Specify the point on the feature to use.



- X, Y and Z values to define the distance between the two keypoints.

 Delete the relationship.

The current status of the relationship is indicated on the feature dialog by one of the following:

 no relationship is currently defined.

 a valid relationship is defined.

3 Once you have selected the **Relationship workplane** button , the following options can be used to help define the keypoint on the solid:

All types - If *ON*, all key points are displayed. If *OFF*, other options are used to define the keypoints that are displayed. Turning off any of the other keypoint options will automatically set **All Types** to *OFF*

Corners - If *ON*, corner points on the solid are displayed.

Special surfaces - If ON, keypoints on special shaped surfaces (plane, cylinder, cone, torus and sphere) are displayed.

Hole Features - If *ON*, keypoints of hole features are displayed.

Primitive features - If *ON*, keypoints of primitive features are displayed.

Relative to the relationship's workplane - If *ON*, the feature will be aligned so that it's axes are parallel to the relationship workplane. If *OFF*, the orientation of the feature is not defined by the relationship workplane.

X, Y, Z boxes - Enter the X, Y and Z angles that the feature is twisted around each axis. The twists are applied in the order that they are entered. The order that the twists will be applied is indicated by the twist transformation order that is displayed at the bottom of the dialog when you enter values in the boxes.

4 Click **OK** toaccept all changes and return to the appropriate feature dialog, or **Cancel** to discard all changes and return to the appropriate feature dialog.

Solid feature tree

The solid feature tree contains the history of solid operations performed on a solid. Each solid operation is defined as a feature on the solid.

Use the following sections for more details on using the solid feature tree:

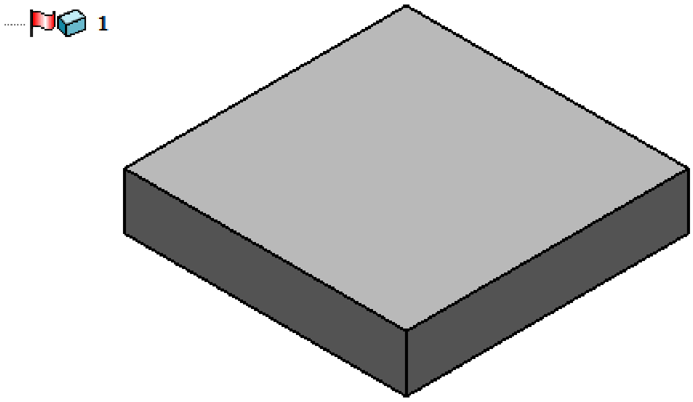
- Creating a solid feature tree (see page 254)
- Displaying the solid feature tree (see page 257)
- Editing a solid using the feature tree (see page 257)
- Editing a feature using context menus (see page 262)
- Selecting a feature (see page 265)
- Activating and deactivating a solid (see page 269)
- Reordering the features in the tree (see page 270)
- Setting a Rewind Position (see page 271)
- Supressing a feature (see page 275)
- Unsupressing a feature (see page 276)
- Deleting a feature (see page 276)
- Speeding up operations on solids (see page 276)
- Removing the history of features for a solid (see page 276)
- Deferring updating the solid (see page 277)
- Optimising the tree (see page 278)
- Changing the name in the feature tree (see page 282)
- Hiding and viewing features in the tree (see page 282)
- Editing the base solid (see page 283)

Creating a solid feature tree

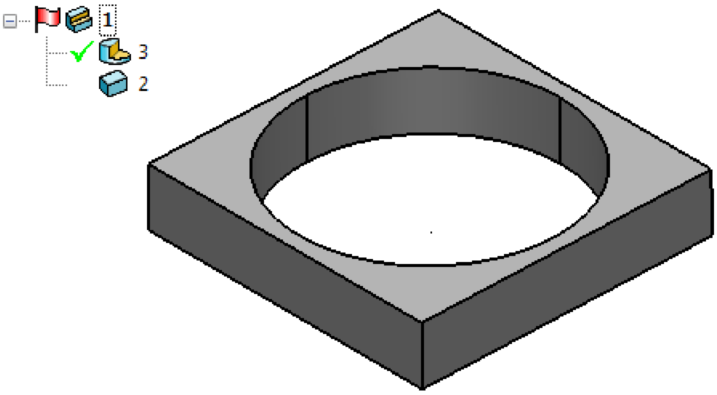
The solid tree is created automatically when you add a solid feature to a solid.

1 Create a solid block it is automatically added to the solid feature tree. The tree is hidden by default.

2 Display the solid tree (see page 257).

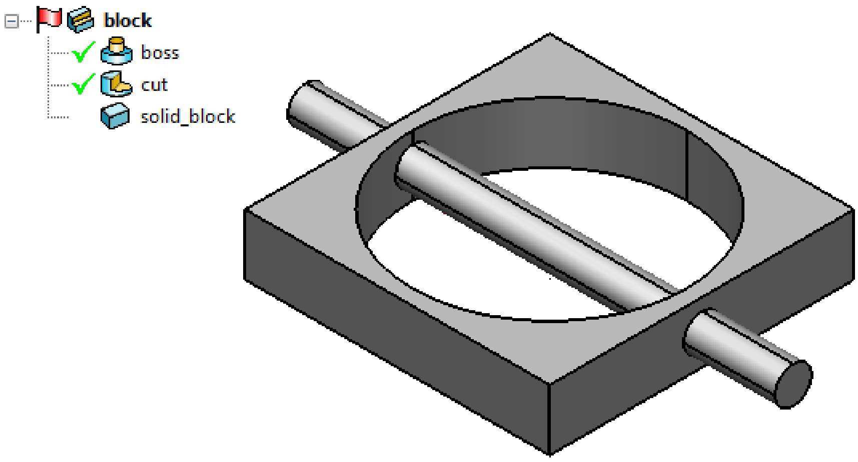


3 Add a cut feature to the solid. The cut feature is displayed in the tree; the block at the bottom of the tree to show that the solid was created from a primitive block.



The features are shown in chronological order in the tree. The most recent one applied is shown at the top and the earliest at the bottom.

4 Add a boss to our example. The boss feature appears above the cut feature.



The following icons can be displayed in the solid tree:

- Solid with a history

- General solid without a history

- Active solid

- Inactive solid

- Extrusion or Extruded solid feature

- Solid of revolution or Solid of revolution feature

- Block primitive or Block primitive feature

- Sphere primitive or Sphere primitive feature

- Cylinder primitive or Cylinder primitive feature

- Cone primitive or Cone primitive feature

- Torus primitive or Torus primitive feature

- Add feature

- Subtract feature

- Intersect feature

- Cut feature

- Boss feature

- Hole feature

- Pocket/Protrusion feature

- Hollow feature

- Thicken feature

- Bulge feature

- Fillet feature

- Chamfer feature

- Rib fillet feature

- Wrap feature

- Sculpt feature

- Suppressed feature

- Unsuppressed feature

- Error suppressed feature

- Defer update

- Group feature

- Pattern feature

- Edited face

- Unrecognised. This is used to indicate that the feature is not recognised by the version of PowerSHAPE. It is displayed in the feature tree when you create a model in a future version of PowerSHAPE that contains a new type of feature and then open that model in an earlier version of the software that cannot recognise the new feature type. When the feature tree is replayed, an **Unrecognised** feature is error-suppressed.

- Associated features hidden in tree

- Associated features visible in tree

Displaying the solid feature tree

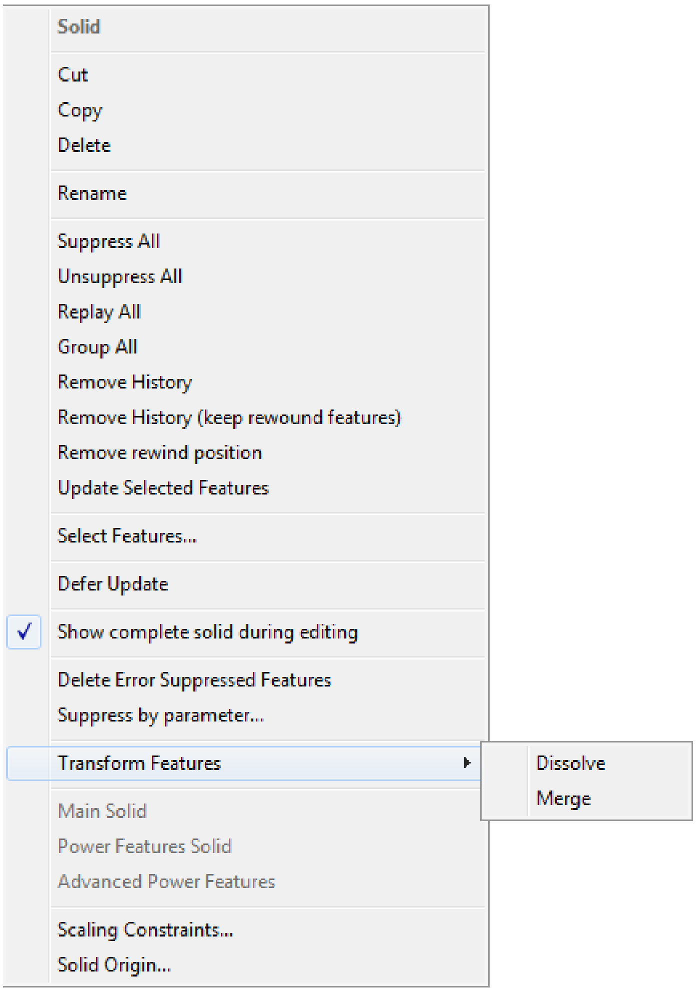
Use one of the following techniques to display the solid feature tree:

- Click the solid to display the **Solid Edit** toolbar and click the **Show/Hide tree window** button .
- Select **View > Windows> Tree Window**.

Editing a solid using the tree

When you select a single solid in the graphics window or solid tree, it is highlighted as selected and a black box is drawn around its name in the solid tree.

1 In the feature tree, right-click the **Feature** icon to display the solid popup



2 Use the options on the context menu to edit the selected solid.

Rename — Edit the name of the solid.

Suppress All — Suppress all the features on the solid. All features are no longer displayed on the solid.

Unsuppress All — Unsuppress all the features on the solid. All features are displayed on the solid.

Replay All — Redefine all the features in the history tree of the solid. No changes are made to the definition of any features (for example, suppressed features will remain suppressed).

This option may help you fix a problem that was present in a model created in an earlier version of PowerSHAPE, for example if the solid contains badly trimmed surfaces.

Group All — Select this option to group sequential features throughout the whole tree.

Remove History — Select this option to remove all the history for a solid.

Remove History (keep rewound features) — Select this option to remove the history below the rewind bar and combine the features into a single general base feature. The greyed out rewound features are left intact.

This option is only available when the solid has a history tree that has a rewind bar.

Remove rewind position — Removes the rewind position. All the greyed-out rewound features are applied to the solid.

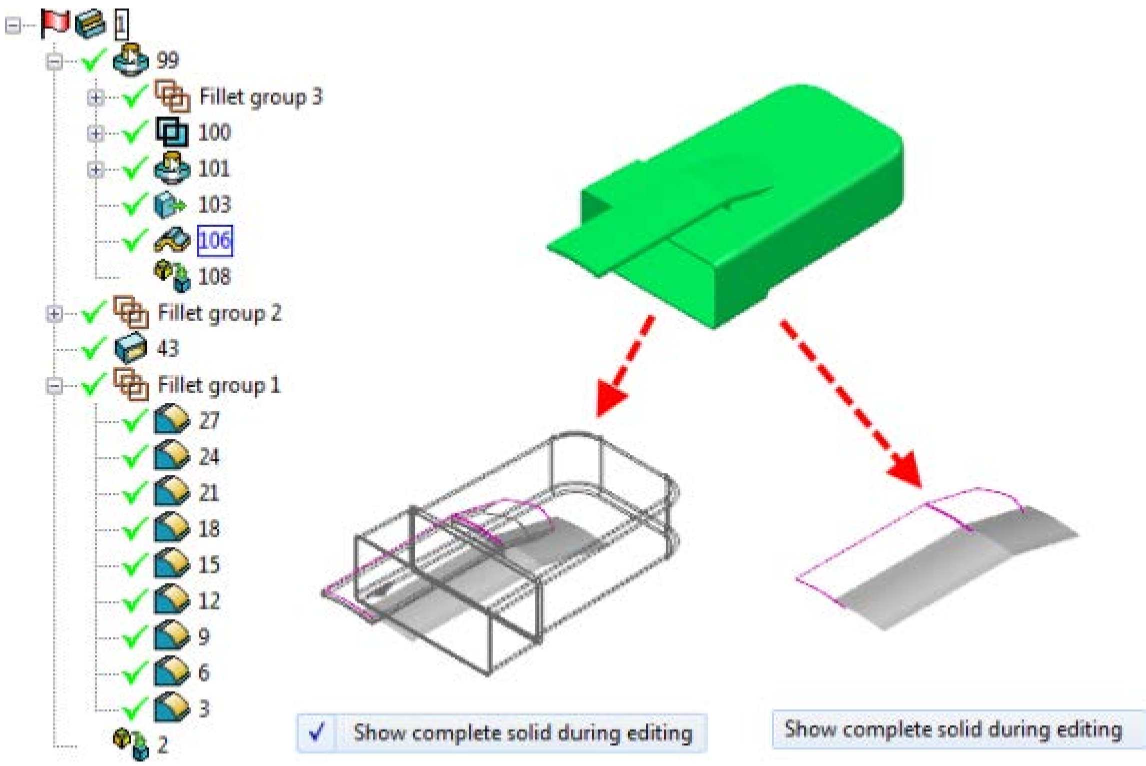
Update Selected Features — Replays the selected features in a solid using the latest version of Parasolid.

Try using this option on error-supressed features to see if the latest version of Parasolid fixes them.

Select Features... — Use this option to use the **Select Features** dialog (see page 268) to select features in the solid tree that comply with the filter.

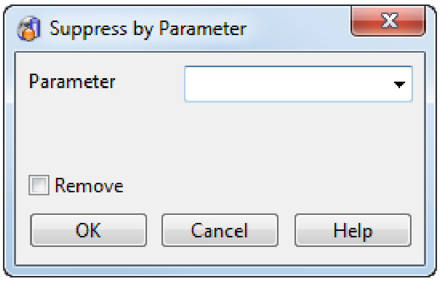
Defer Update — Defers updating the solid while editing it.

Show complete solid during editing — Use this option to display the complete solid whilst editing a feature. This means that you can see the features that are further up the solid tree whilst you are editing a feature. This option is selected by default. The example below shows the effect of using this option when editing the chamfer *45.*



Delete Error Suppressed Features — Delete all error-suppressed features in the history tree for the solid.

Suppress by parameter... — Select this option to display the **Suppress by Parameter** dialog.



Parameter — Use the drop-down list to select the parameter to control suppression of the feature. The parameter can also be used to control multiple, selected features.

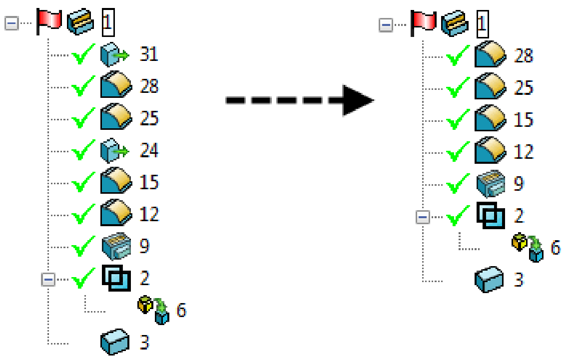
If the parameter value is non-zero, the feature will be suppressed.

If the parameter value is zero, the feature will be unsuppressed.

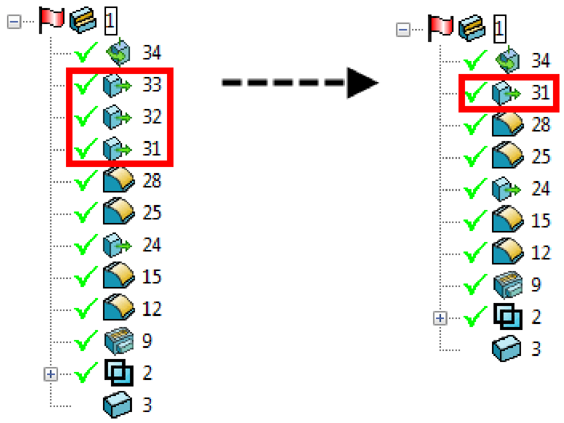
Remove — Select this option to remove the use of a parameter to control suppression. If this option is selected, suppression/unsuppression will no longer be controlled by the parameter.

Transform Features — Use **Dissolve** and **Merge** to rationalise the solid tree:

Dissolve — Select **Dissolve** to remove the transform features from the solid tree. An **Information** dialog is displayed to tell you how many transforms have been removed.



Merge — Use this option to merge adjacent transform features of the same type. For example, if you move a solid three times, selecting **Merge** will merge the three transform features into one.



If the Automatically merge solid transform features option is set on the Tools > Options > General > General Edits dialog, the transform features will be automatically merged as they are created.

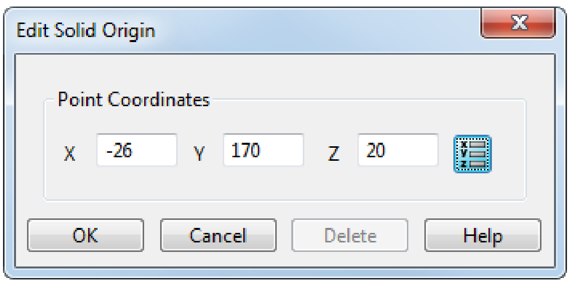
Main Solid — If this is deslected, click this option to make the selected solid the **Main Solid**. *This option is only active when you are using an assembly.*

Power Feature Solid — If this option is deselected, click this optionto make the selected option a **Power Feature Solid**. *This option is only active when you are using an assembly.*

Advanced Power Features — Select this option to use the **Set advanced property for Power Features** dialog. *This option is only active when you are using an assembly.*

Scaling Constraints — Use this option to Apply constraints (see page 92) to solids that you wish to scale.

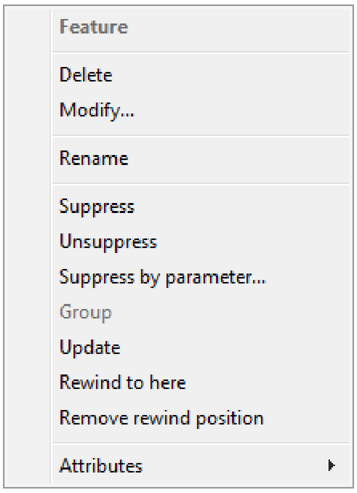
Solid Origin... — Select this option to display the **Edit Solid Origin** dialog.



Use this dialog to define a reference point for a solid, to be used as a reference point during in grading.

Editing a feature using the context menu in the feature tree

In the feature tree, right-click a feature to display the feature pop up menu.



Delete - Select this option to delete the feature from the tree.

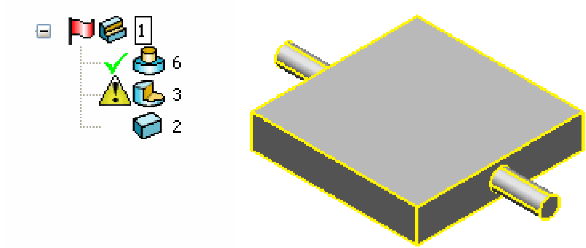
Modify opens the feature-specific dialog.

Rename lets you edit the name of the feature.

Suppress the feature on the solid. The feature is no longer displayed on the solid.

Unsuppress the feature on the solid. The feature is displayed on the solid.

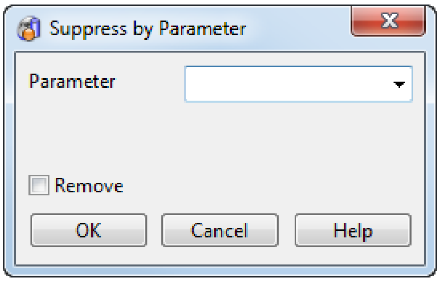
If a feature is no longer valid, it is suppressed from the solid and the **Error suppressed feature** icon appears next to its icon in the tree. For example, if you edit a cut feature and define a depth that doesn't cut the solid, an **Error suppressed feature** icon appears.



You can click the **Error suppressed feature** icon to find out why a feature is invalid.

You cannot edit a suppressed feature, but you can edit an error suppressed one.

Suppress by parameter displays the **Suppress by Parameter** dialog.



Parameter lets you select the parameter to control suppression of the feature. The parameter can also be used to control multiple selected features.

If the parameter value is non-zero, the feature will be suppressed.

If the parameter value is zero, the feature will be unsuppressed.

Remove the use of a parameter to control suppression. If this option is selected, suppression/unsupression will no longer be controlled by the parameter.

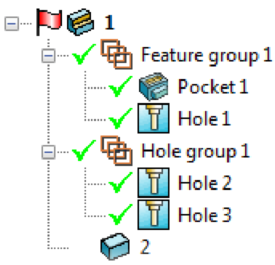
Convert to general base feature converts the selected primitive features to base features. This option is hidden if you have not clicked on a primitive feature.

In addition:

- *If the solid is a primitive solid,* **Convert to general solid** *is displayed on the context menu.*
- *If the solid has a history tree, then* **Remove history** *is displayed on the context menu.*
- *If the solid has no history and is not a primitive, then neither option is displayed on the context menu.*

Group collects a number of features together into a group.

- If the features are of different types, the group is given a generic name, for example *Feature group 1.*
- If the features are of the same type, the group is named after that type, for example *Hole group 1.*



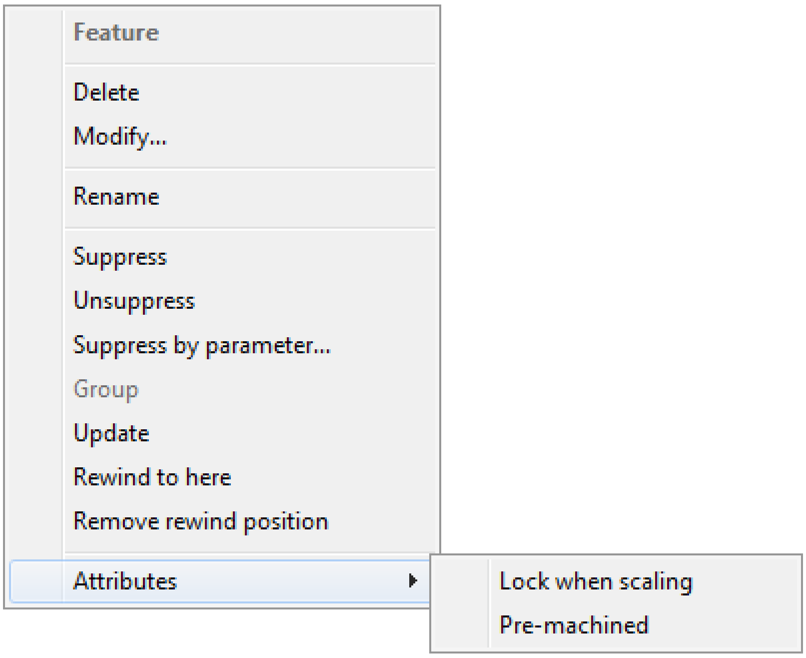
Update Replays the features in a solid using the latest version of Parasolid.

Try using this option on error-supressed features to see if the latest version of Parasolid fixes them.

Rewind to here sets the rewind position immediately above the feature you clicked.

Remove rewind position removes the rewind position. All the greyed-out rewound features are applied to the solid.

Attributes displays the **Attributes** submenu.



Lock when scaling behaves slightly different for each feature.and is not available for some features.

Generally, if this option is selected, the original dimensions of the feature are unchanged when the whole solid is uniformly scaled from the **Edit** toolbar. If deselected, the feature is scaled too.

If you non-uniformly scale the solid or define a dimension of the feature using parameters, the feature will not be scaled regardless of whether the option is selected or deselected.

Pre-machined lets you specify whether or not the feature is already machined.

Editing a feature using the feature tree

Double-click the icon of a feature in the tree to display the dialog for that feature. When you do this, only the following aredisplayed on the edited solid:

- the feature you are editing.

- the solid before that feature was applied.

The following icons may appear in the tree.

- Extruded solid feature

- Solid of revolution feature

- Block primitive feature

- Sphere primitive feature

- Cylinder primitive feature

- Cone primitive feature

- Torus primitive feature

- Add feature

- Subtract feature

- Intersect feature

- Cut feature

- Boss feature

- Hole feature

- Hollow feature

- Thicken feature

- Fillet feature

- Chamfer feature

- Bulge feature

- Wrap feature

- Group feature

- Pattern feature

- Sculpt feature. This feature cannot be edited.

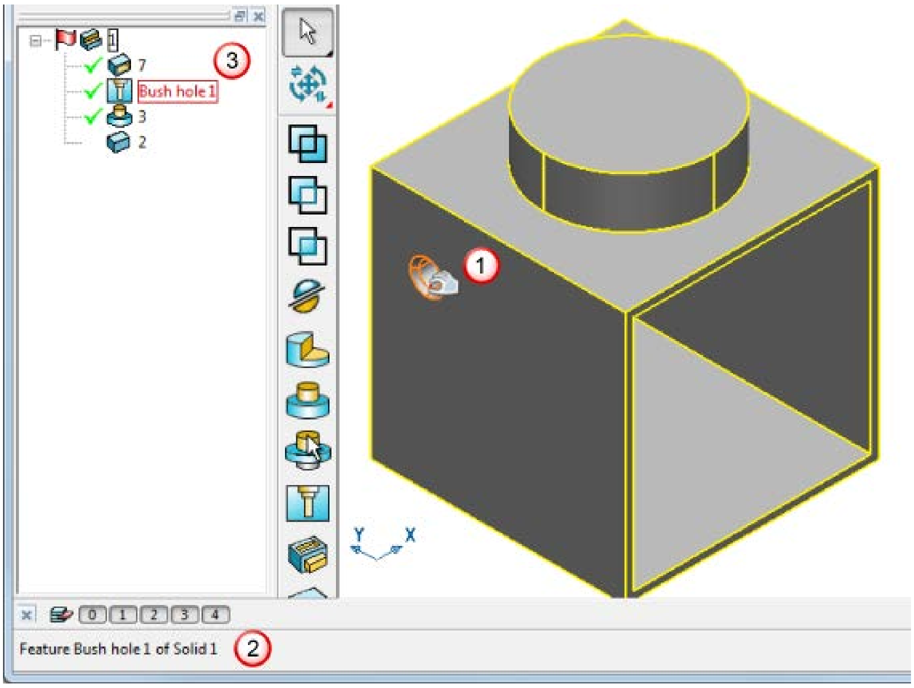
Details about editing the different features are included with the feature creation infomation.

Selecting a feature

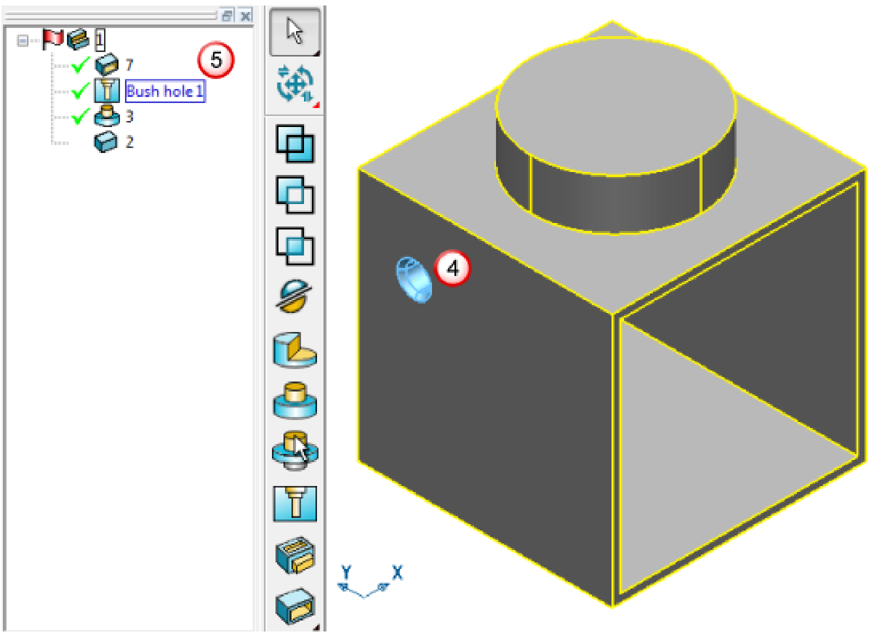
Use one of the following techniques to identify the feature was used to create a particular face.

- If a single solid is selected, the edges of the faces are highlighted orange (default colour scheme) as you move the cursor over them 1.

  If a face originated from a feature, the name of the feature is displayed in the **Status** bar 2 and highlighted with a red box in the feature tree 3.



- If you click a feature in the solid feature tree, the faces of this feature are highlighted blue (default colour scheme) 4 and the name of the feature is highlighted with a blue box in the feature tree 5.



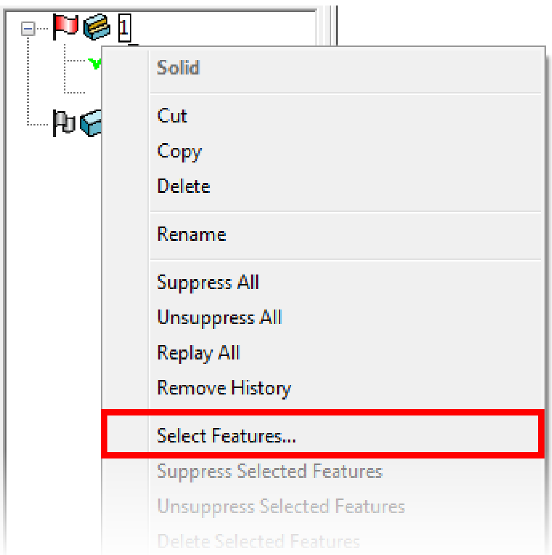
- Use  to select a number of individual features. All selected features are highlighted in blue.

  If you click the face of the solid that was created from a feature, the selected face will be coloured pink. If the face is part of a feature with a single visible face it will be coloured blue (default colour scheme). The name of the feature is highlighted with a blue box around it in the feature tree. This feature will remain highlighted even if the cursor is moved over other faces of the solid.

Selecting one or more features in the tree

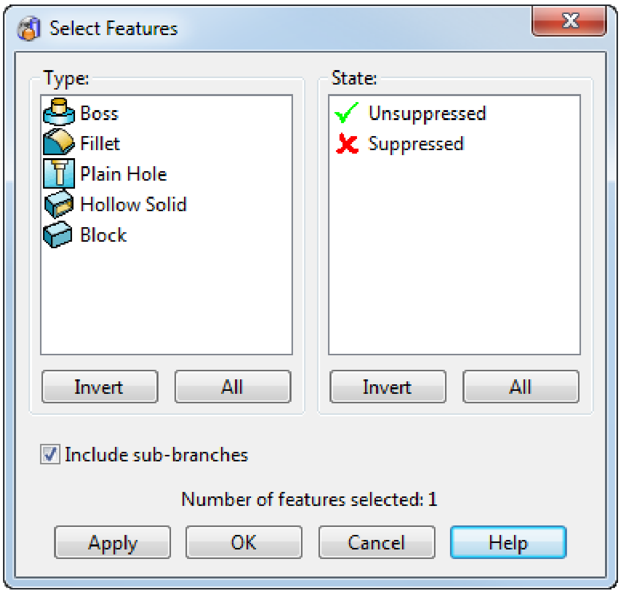
Use one of the following techniques to select features in the tree

- Use  to select the name of the feature. The feature is added or removed from the list of selected features. If you don't hold down **Ctrl**, the previous selection is replaced by the single feature. The name of each selected feature has a blue box around it and the surfaces that make up each feature are highlighted in the solid.
- In the solid tree, right-click a solid to display the context menu. Choose **Select Features...** to open the **Select Features** dialog (see page 268).



Select Features dialog

Choose **Select Features...** (*Solid tree context menu*) to display the **Select Features** dialog. This option will be unavailable if the solid has no features.



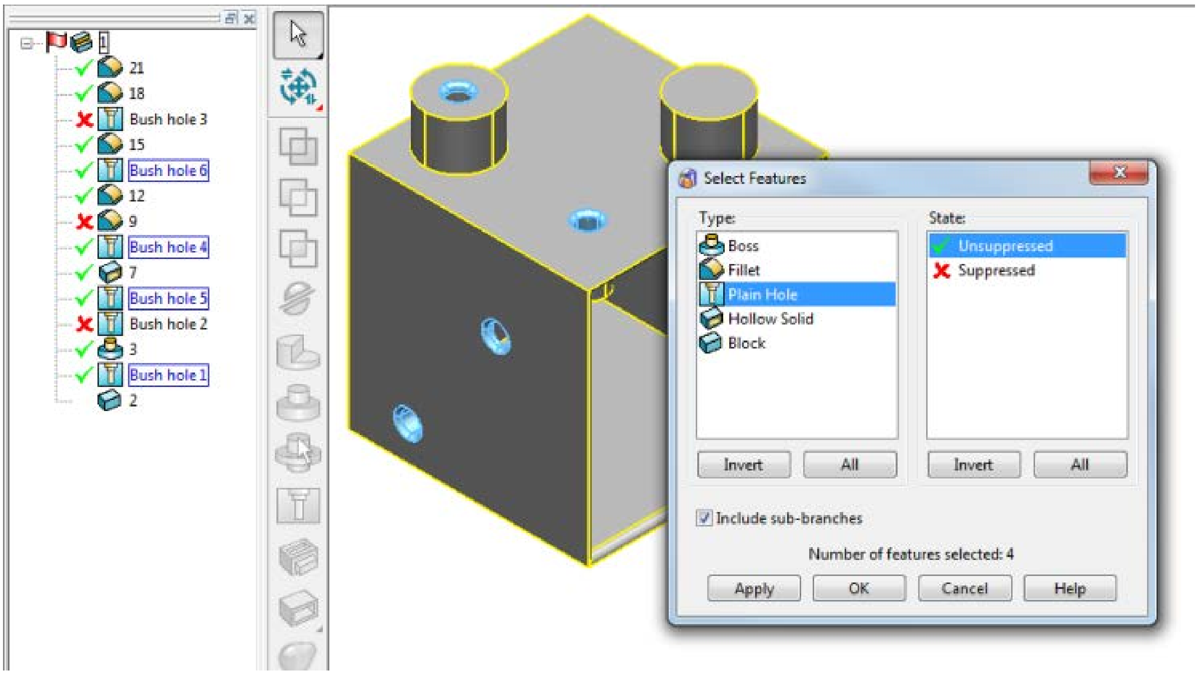
Type lists the different features that can be highlighted on the model. Click items in the list to select/deselect.

State lists the state of the features. Click to select/deselect.

Invert inverts the current selection; features that are selected become deselected and features that are deselected become selected.

All selects all items in the list.

Apply highlights the features that match the filters you have applied.



OK -saves the changes you made and closes the dialog. You can only select the items you specified on the dialog.

Cancel closes the dialog and discards any changes.

Activating and deactivating a solid

- Double-click the solid to display the solid feature tree. In the solid feature tree, each solid has an icon next to it to indicate whether it is active or not.

  indicates the solid is active.

  indicates the solid is inactive.

  Click the **Active solid** icon next to the solid in the tree to make it inactive.

  Click the **Inactive solid** icon next to the solid in the tree to activate the solid.

- Right-click the solid in the view to display its context menu. Select **Active** to activate the solid and turn off to deactivate the solid.

A solid can also be activated and deactivated using the ***Solid Edit*** *toolbar.*

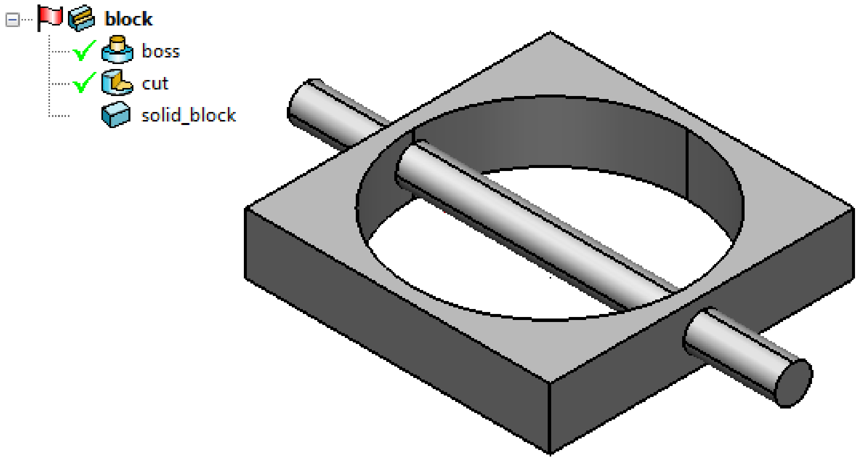
Reordering the features in the tree

You can redefine the solid by changing the order of the features in the tree.

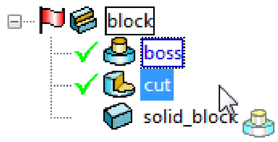
1 In the solid feature tree, select the icon of the feature.

2 Drag the selected icon onto the icon of the feature you want it to lie below.

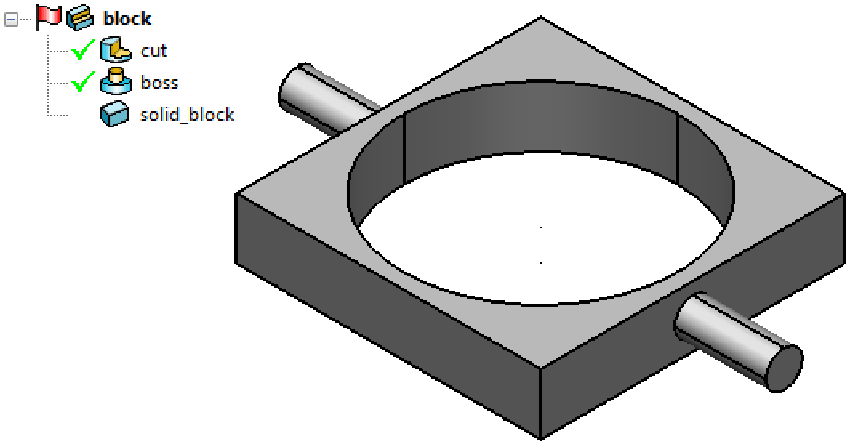
In the tree shown below, we will switch the positions of the *cut* and *boss* features.



Drag the icon of *boss* feature  onto the icon of the *cut* feature .

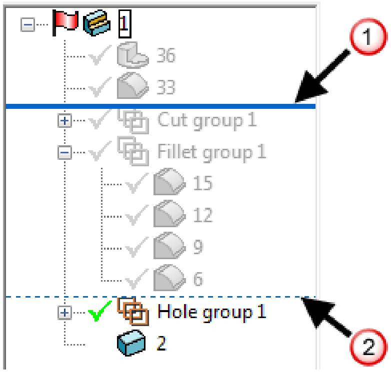


This reorders the features and redefines the solid.



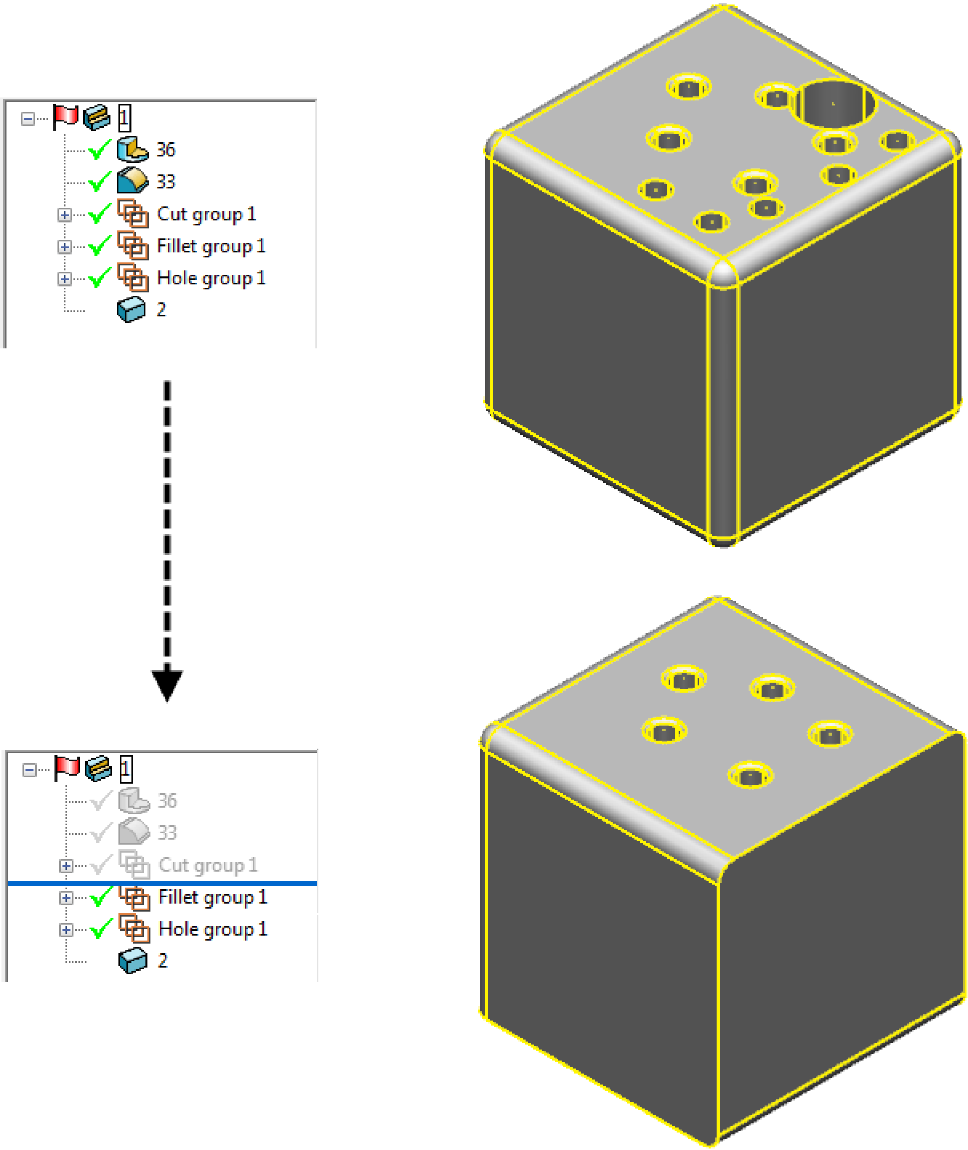
Setting a Rewind Position

You can set a **Rewind Position** in the solid tree.



The rewind position appears as a horizontal bar on the solid tree 1.

This bar can be dragged up and down the solid tree using  . A dashed line 2 is displayed to show the potential new position of the bar during the drag operation.



The **Rewind Position** lets you:

- create a new feature anywhere in the tree (see page 273). If you create a new feature, it will appear at the rewind position. This means you do not have to create it at the top of the tree and then re-order it to where you want it.
- preserve features that would be automatically removed (for example, Direct Modelling operations). Only features that are below the rewind position are replaced, They will be replaced by a single base feature, without history.

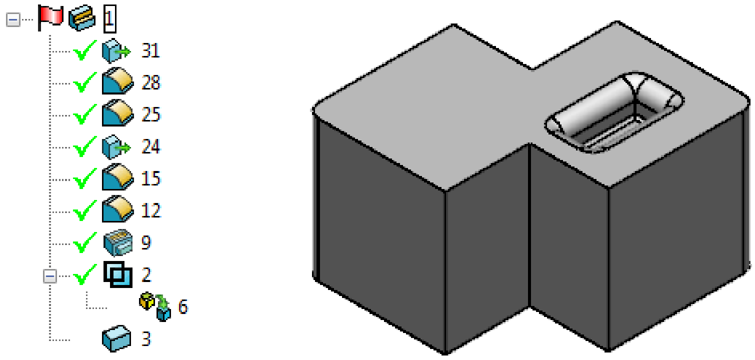
- export solids in an earlier state; for example, prior to a part revision.

Features that are rewound appear greyed-out in the solid tree. They are unavailable for selecting, or editing.

Creating a feature at a specified place in the tree

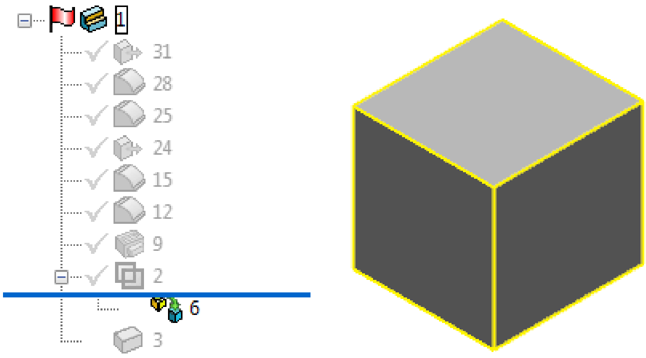
Use **Rewind Position** to create a new feature anywhere in the tree.

This example reduces the height of the general solid and creates a hole feature on the general solid.

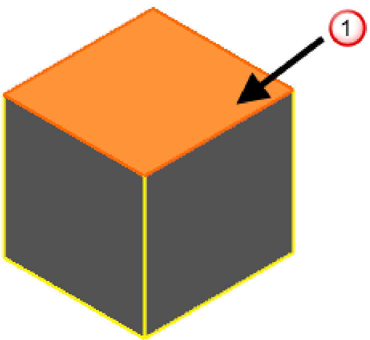


1 Right-click 

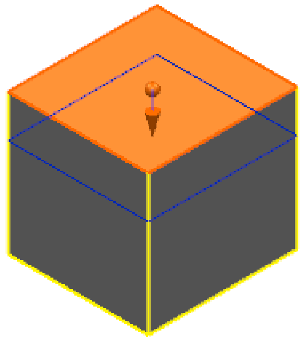
2 Click **Rewind to here** to suppress the features of the solid and leave the general solid displayed.

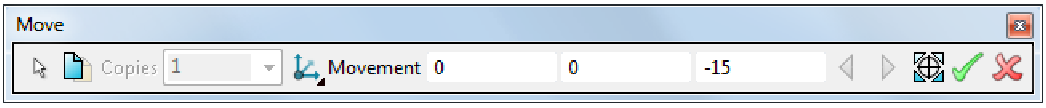


3 Click  (*Solid edit toolbar*) and select the individual face 1.



4 Move the face by entering **-15** in the Z value on the **Move** toolbar.

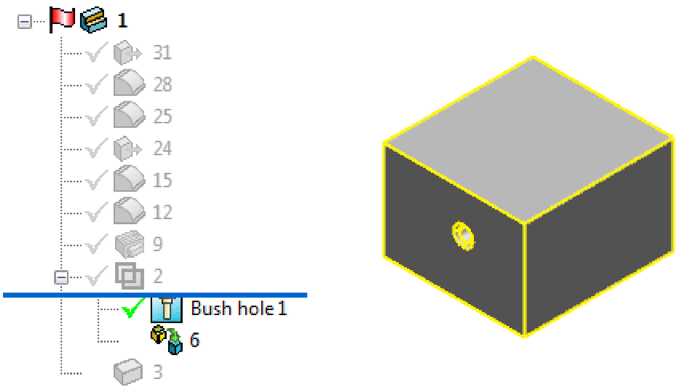




5 Click **Apply** .

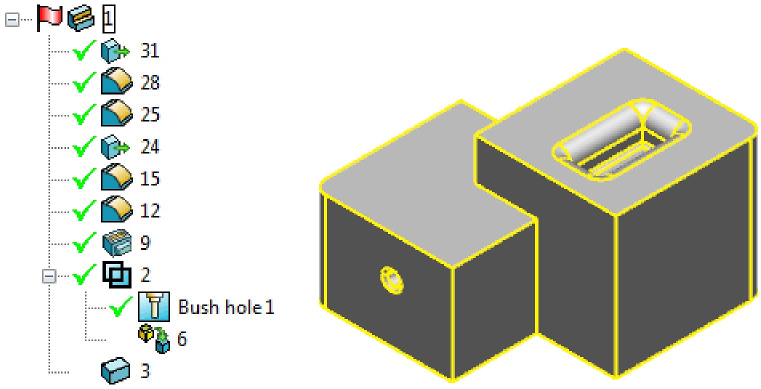
6 Click **Dismiss** . The solid still has a history tree.

7 Add a hole feature to the solid. The hole is added to the tree in the sub-branch above the general solid feature.



8 Move the cursor over the **Rewind bar** so that the cursor changes to a double-headed arrow.

9 Right-click and select **Remove rewind position** from the context menu.



Suppressing a feature

You can suppress a feature to remove it from the solid. PowerSHAPE remembers the details of the suppressed feature.

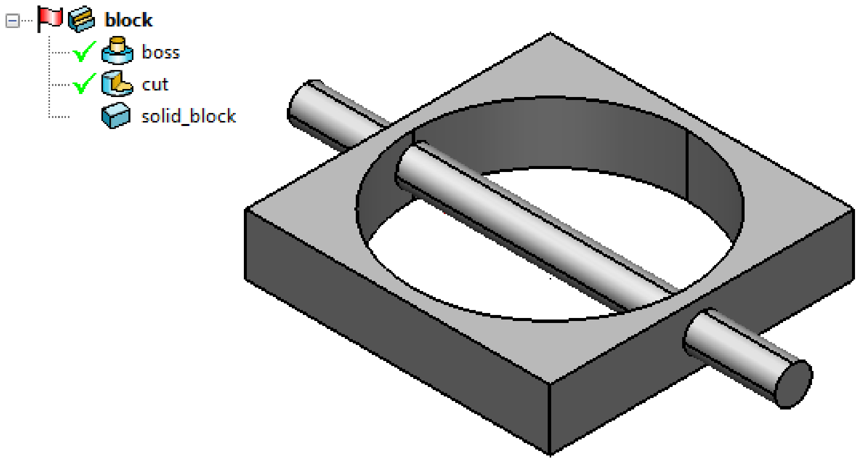
To suppress a feature:

1 Double-click the solid to display the solid feature tree, if the tree is not already displayed.

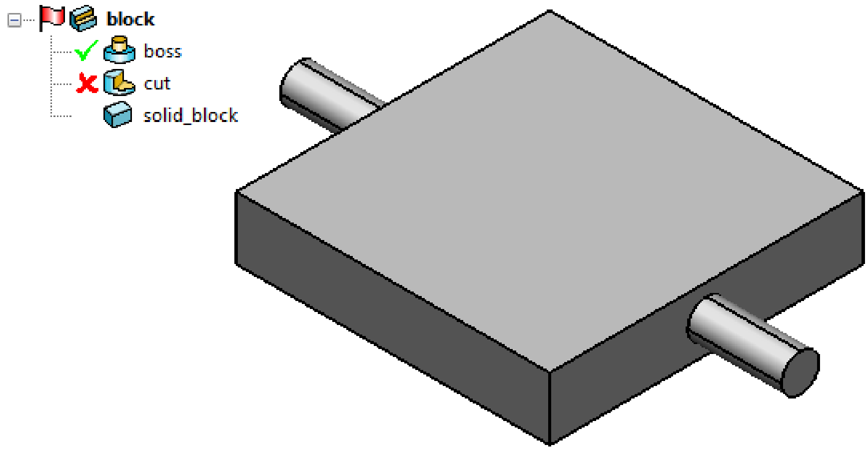
2 Click the **Unsuppressed feature** icon ✓.

3 The icon changes to the **Suppressed feature** icon ✗ to indicate that the feature is now suppressed.

In the example below, we will suppress the *cut* feature.



The feature is removed from the solid.



Unsuppressing a feature

To unsuppress a feature, click the **Suppressed feature** icon . The icon changes back to the **Unsuppressed feature** icon and the feature is visible on the solid again.

Deleting a feature

To permanently delete a feature, select it and then press **Delete** on the keyboard.

Speeding up operations on solids

When there are a lot of features on a solid, you may find it takes longer to update your solid.

To speed up operating on solids, do one of the following:

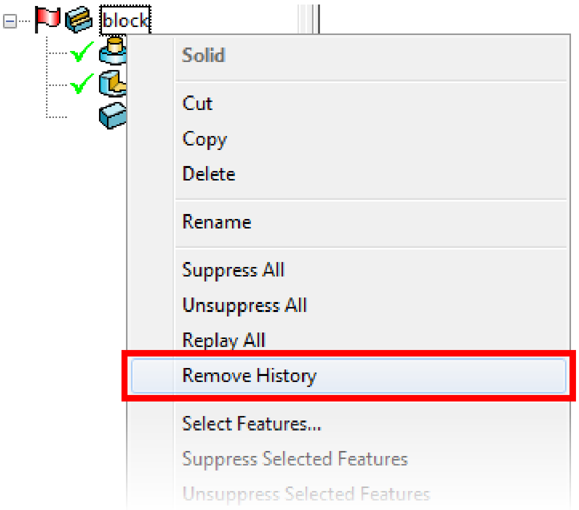
- Remove the history of the features (see page 276).
- Defer updating the solid (see page 277).

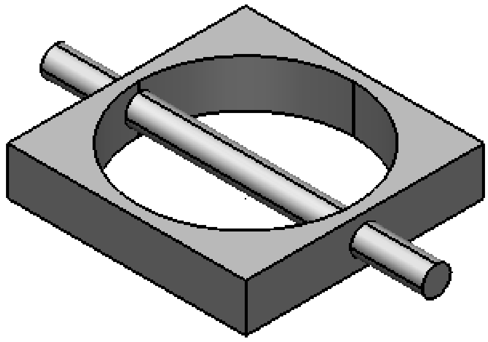
Removing the history of features for a solid

If you remove the history of features from a solid, it is quicker to operate on the solid.

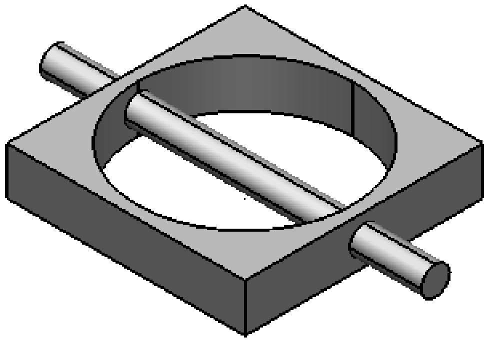
1 Right-click the icon of the solid in the feature tree to open the context menu.

2 Select **Remove History**.





The features are permanently removed from the tree.



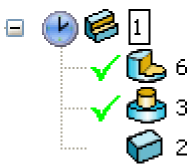
Deferring updating the solid

To speed up editing a solid, you can defer updating the solid. This allows you to apply several edits to a feature tree of a solid, without having to wait for the solid to redefine after each edit. All the edits are treated as one command and applied when you exit defer update mode.

1 Right-click the **Feature** icon of the solid in the feature tree

2 Select **Defer Update** from the context menu.

The icon next to the **Feature** icon in the tree changes to the **Defer Update** icon .



3 Make the required changes to the solid. While you are in defer update mode, the following are true:

- All edits on the features are available, for example, suppressing, re-ordering and using the dialogs to edit features.
- After each edit, the feature tree will update, but the solid will not graphically change.

  When you edit some features using their dialogs, their correct shape will display. Once you finish editing them, their original shape is displayed and they look unedited.

If you delete a feature, the **Deleted feature** icon is placed next to the icon of the feature in the tree.

The feature remains visible in the solid and is only removed when you exit defer update mode.

4. Exit defer update mode. The edits will then be applied to the solid.

   To exit defer update mode, do one of the following:

   - Click the **Defer update** icon .
   - Right-click the **Feature** icon of the solid in the feature tree and select **Defer Update** from the context menu.
   - Deselect the solid by clicking in free space.
   - Stop editing the solid and begin a different operation, for example create a line.

__Undo__ is not available while you are in defer update mode. Once you exit defer update mode, the __Undo__ command treats all the edits in defer update mode as a single command.

Optimising the tree

Speed up the redefinition of multiple features in a large feature tree by optimising the solid feature tree. This groups together multiple occurrences of the same features:

Optimise the tree in one of the following ways:

- Automatic optimisation uses the **Tree Optimisation** options on **Tools > Options > Object > Solids** dialog. For example, using the default settings, you are asked if you wish to optimise the tree after 20 holes are created in the solid.

- Use the solid feature tree to optimise the tree (see page 279) by creating feature groups.
- Edit feature groups (see page 280) to make identical changes to a large number of features.

Use **Explode** (see page 281) to remove the groups that are created by optimising the tree.

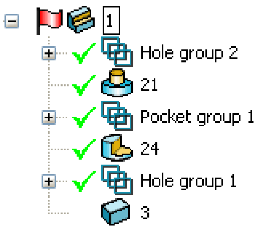
Creating group features

To optimise the tree:

1 Right-click **Feature** to display the solid context menu.

2 Select **Group Features**.

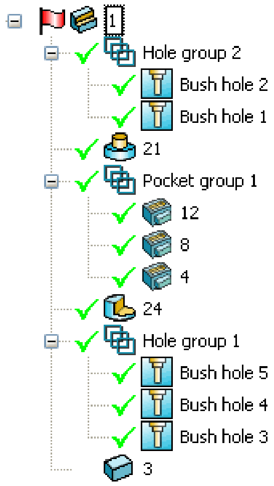
The holes are put into two different **Hole group** features and the pocket features are put into a **Pocket group**. Each group feature is marked with the .



In the example, all the holes next to each other in the solid tree are put in the same group.

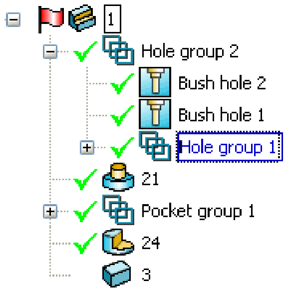
Using a Group feature

Each group feature should only contain hole features that do not interact with one another. This allows all the features in the group to be applied in a single operation.



Using a group lets you:

- edit the hole features in a group as you normally would. If you edit a hole feature and as a result it interacts with other features in the group, it is suppressed and an error message is displayed.
- move hole and group features as you normally would by dragging their icons. For example, you can move hole features from one group to another.
- move a group feature inside another group to create a sub-group. If you can't move a hole feature, it will be left where it is.



If you delete a group feature, the hole features are moved up a level in the tree and the group feature icon is removed from the tree.

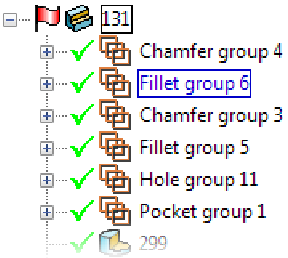
If you suppress a group feature, all the holes in the group are suppressed.

Editing group features

You can simultaneously modify a group of features of the same type in a solid using the solid feature tree.

To modify a group of features:

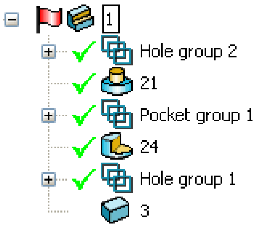
1 Select a group in the solid feature tree.



2 Right-click the selected group and select **Modify** from the **Feature** context menu.

The appropriate dialog for editing multiple features (see page 247) of the same type is displayed (for example, the **Edit SolidFillet** dialog).

Exploding a group feature

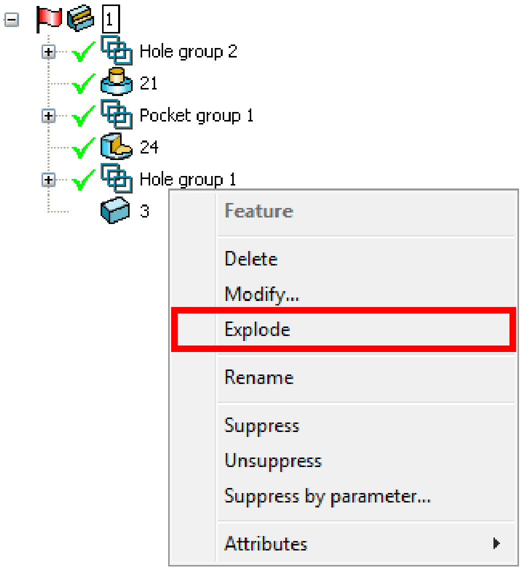


In the solid feature tree:

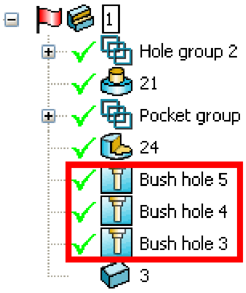
1 Select the group to ungroup.

2 Right click **Feature**  to display the solid pop up menu.

3 Select **Explode**.



The group is removed and the features are moved into the main tree.

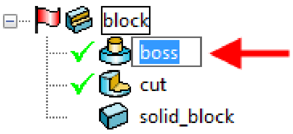


If the group was part of a parent group, the features are moved into the parent group.

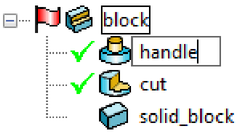
Change a name in the feature tree

1 Click the name to select it.

2 Click the name again to make it editable.



3 Enter the new name in the text box.



*You can also edit the name pressing **F2** or by right-clicking a selected name and selecting **Rename** from the context menu.*

Hiding and viewing features in the tree

If a solid has features, it will have either a **Plus** ⊞ or **Minus** ⊟ icon to its left.

⊟ - Features are visible in the tree.

⊞ - Features are hidden in the tree.

Click the **Plus** icon ⊞ to expand the tree and show the features. The icon will change to a **Minus** icon ⊟.

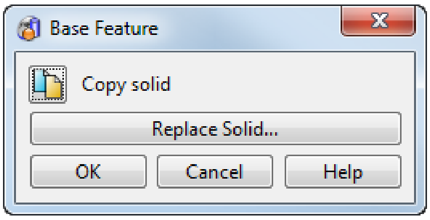
Click the **Minus** ⊟ icon to collapse the tree and hide the features.

Some features can have branches too. For example, when you add a solid with features to the active solid, the new ***Add*** *feature will have branches representing the features from the original solid.*

Editing the base solid

Copy and replace the base (general) solid as follows:

1 Select **Modify** from the base solid popup in the solid feature tree. The **Base Feature** dialog is displayed.



2 Click to create a copy of the selected solid used in the feature. When you select **OK**, the entire sub-branch (the solid and its history) will be copied. If the sub-branch only contains one primitive feature, the new solid will be a primitive solid.

3 Click **Replace Solid** to replace the solid with another one using the **Replace Solid** dialog (see page 115).

4 Click **OK** to complete the operation.

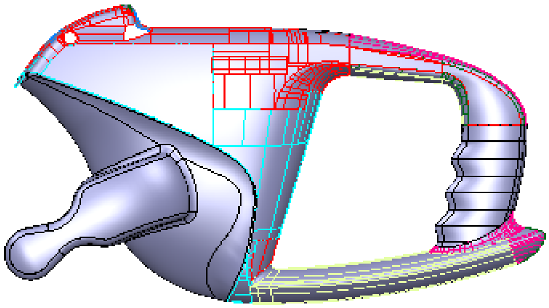
Using the Solid Doctor

Use the **Solid Doctor** to find and fix the faults in a solid. The following are classed as faults:

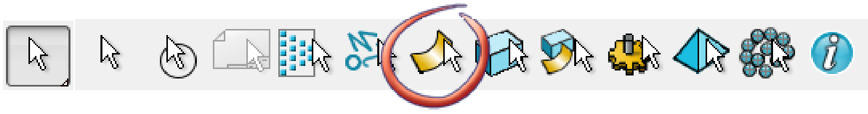
- holes
- overlaps
- self-intersections
- bad trimming of surfaces

These faults need to be removed as they can cause further solid feature operations to fail.

The following model is used to illustrate creating a faultless solid.

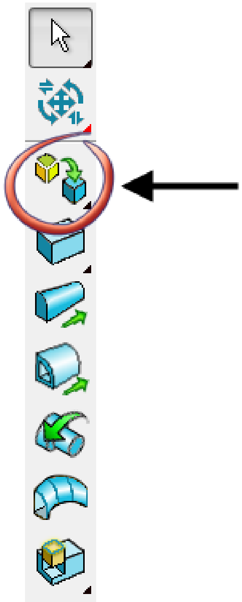


1 Click **Quick select all surfaces** .



2 Click **Solid** .

3 Click **Create solid from selected surfaces or meshes** .

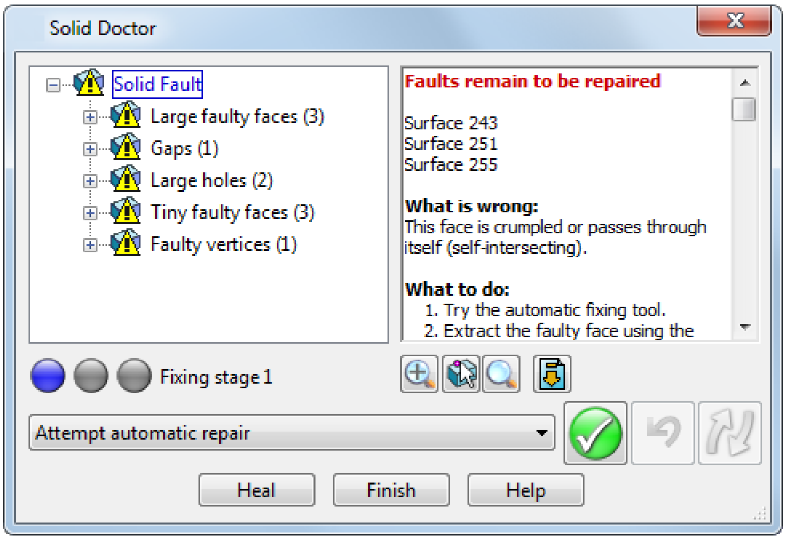


4 Click **No** to the **Make Watertight Wizard**.

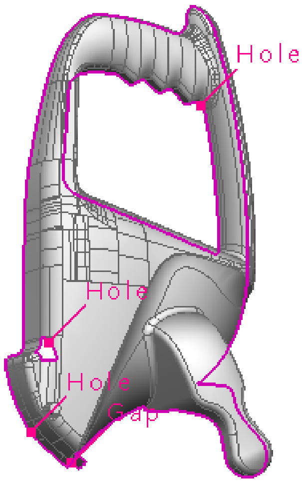
This will eventually be replaced by the Solid doctor.

5 Click **Tools > Model Fixing > Solid Doctor**.

The Solid Doctor dialog box (see page 288) is displayed.



The first group of faults is selected in the fault tree and displayed on the model.

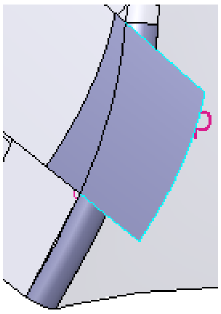


6 Select **Gaps (1)**.

7 Click **Extract surfaces around gap and edit them** from the **Repair Options** drop down list

8 Click **Process the selected faults** .

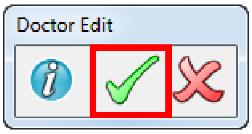
9 Zoom around the gap in the model window and select the 3 surfaces shown below.



10 Click **Delete** .

11 Click **Quick select all surfaces** .

12 Click **Accept** on the Doctor Edit toolbar (see page 293).



13 Click **Tiny faulty surfaces (3).**

14 Click **Delete surface(s)** from the **Repair Option** drop down list.

15 Click **Process the selected faults** .

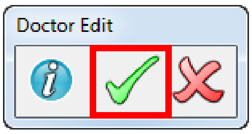
16 Click **Re-check the solid for faults** .

17 Click on the **Gap** label of the fault in the model.

18 Click **Extract surfaces around gap and edit them** from the **Repair Option** drop down list.

19 Click **Process the selected faults** .

20 Click **Accept** on the Doctor Edit toolbar (see page 293).



21 Click **Gap 1** in the **Fault tree**.

22 Click **Fill gap with tangential surface** from the **Repair Option** drop down list.

23 Click **Process the selected faults** .

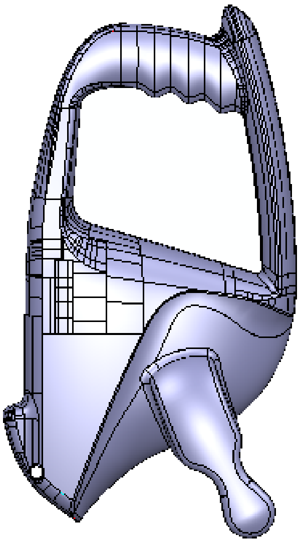
24 Click **Re-check the solid for faults** .

The outer edge of the solid is now marked as a hole.

25 Click **Ignore intentional large hole** from the **Repair Option** drop down list.

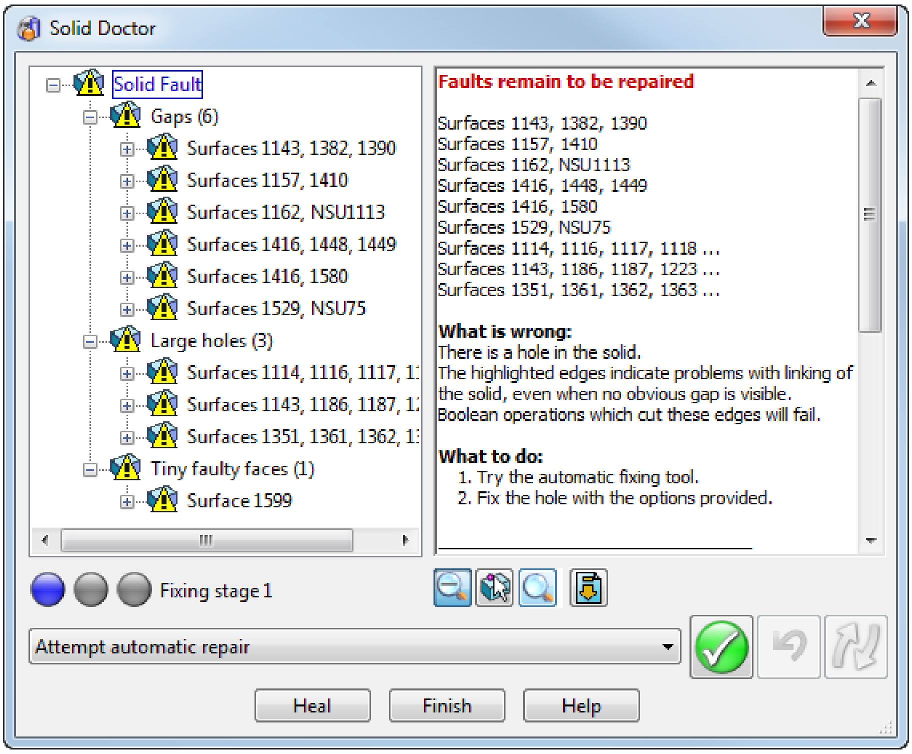
26 Click **Process the selected faults** .

27 Click **Finish**.

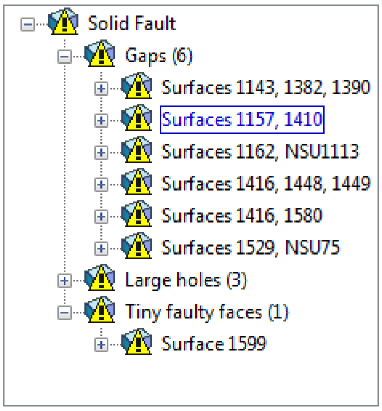


Solid Doctor dialog box

Use the dialog box to fix all the faults in a solid by removing holes and repairing the trimming of the faulty surfaces.

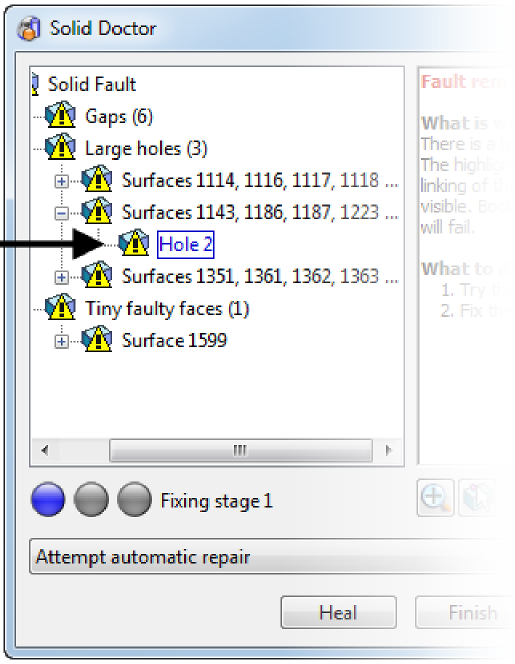


Fault Tree - This lists the faults in the solid. Faults can be selected in the Fault Tree or the model window. If you select a fault in the tree, it will be highlighted in the model. You can also select a fault in the model to highlight it in the tree.

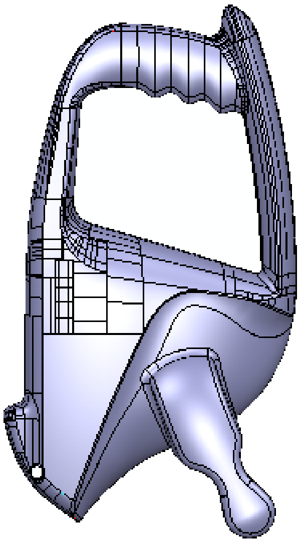


For example:

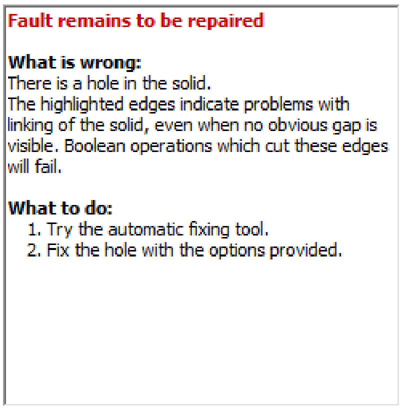
1 Click on the fault in the **Fault tree**.



2 Click on the fault annotation in the model.



Fault Information - This describes the selected fault.



The following fault information is provided:

- **What is wrong:** - This shows a description of the fault.
- **What to do:** - This gives a suggested repair procedure.
- **Failed repair operations** - This lists the fixing operations that have failed.

Zoom  - Click to zoom into the model to enlarge the view of the selected fault. Zoom out to view the whole solid.

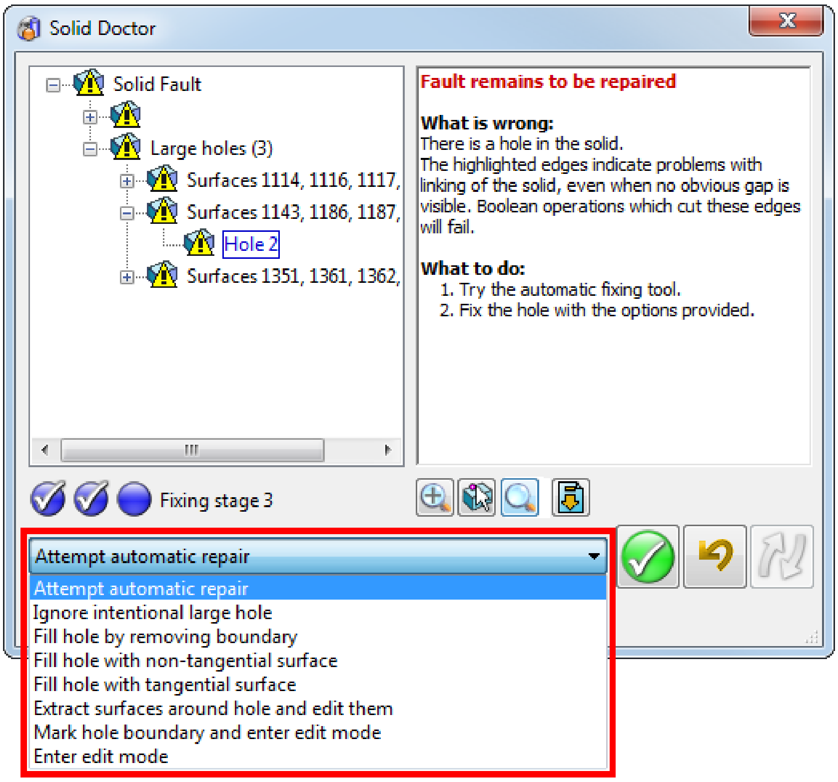
Fault Display - Choose how you want to display the faults in the model window from the options in the drop down list.

Display selected faulty areas - This displays the selected fault in the model window.

Display all faulty areas - This displays all the faults in the model window.

Show affected areas - Not currently available. For future development.

Repair Options - Select a repair option from the list of available repair operations. For further details see the List of fault options (see page 291). The default option is **Attempt automatic repair**. Depending on which repair option you have selected, the Doctor Edit toolbar (see page 293) is displayed.



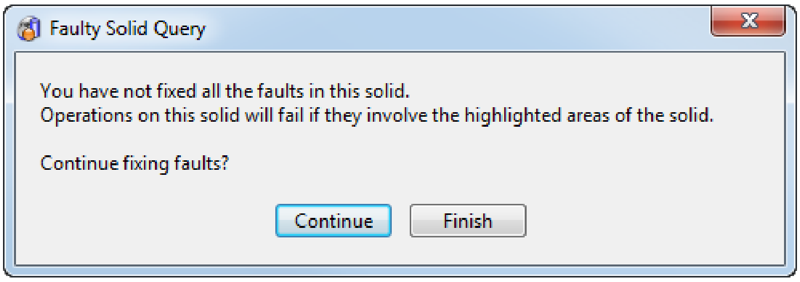
Apply  - Click to apply the repair option to the selected faults.

Undo  - Click to undo the last repair operation.

Re-check - Click to check the solid for faults.

Advanced - Not currently available. For future development.

Finish - Click to close the dialog box and check for faults. If faults are found the **Faulty Solid Query** dialog box is displayed.



Continue - Click to open the **Solid Doctor** dialog box to continue fixing faults.

Finish - Click to accept the changes made by the Solid Doctor.

List of repair options

This lists the repair options displayed on the **Solid Doctor** dialog (see page 288). The repair options relevant to the selected fault are displayed in the **Repair Options** dropdown list.

Repair option	Action
▪ Attempt automatic repair	Automatically fixes the fault. This combines other standard listed options to try to repair the solid.
▪ Extract surfaces and edit them ▪ Extract surfaces around gap and edit them ▪ Extract surfaces around hole and edit them ▪ Extract surfaces at edge and edit them	Removes surfaces of the fault. Enters surface edit mode to fix the surfaces.
▪ Delete surfaces	Deletes the faulty surface.
▪ Delete selected surfaces	Deletes highlighted surfaces
▪ Fix surface	Enters surface edit mode when a surface is selected.
▪ Add selected surfaces to solid	Adds the selected surfaces and/or solids to the main solid to fill the open hole.

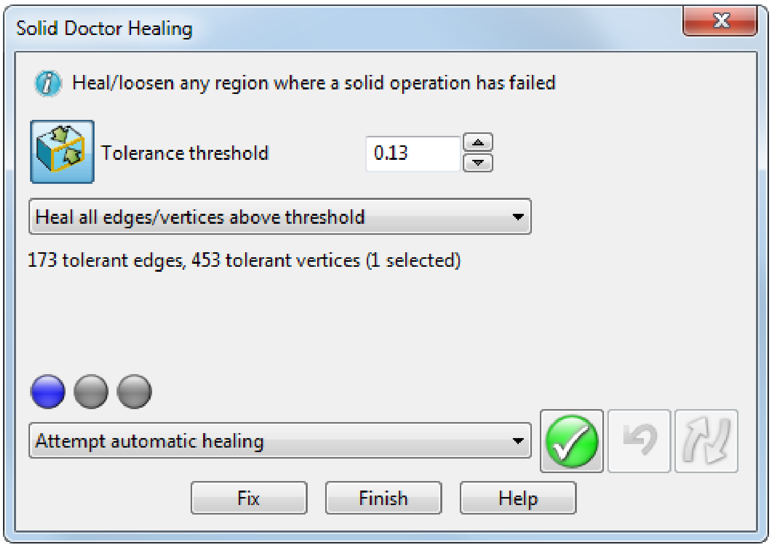
▪ Fill gap with tangential surface ▪ Fill gap with non-tangential surface ▪ Fill hole with tangential surface ▪ Fill hole with non-tangential surface	Creates a fill-in surface for the selected hole in the solid.
▪ Fill gap by removing boundary ▪ Fill hole by removing boundary	Extends the surfaces adjacent to the hole to try to fill the hole.
▪ Link and heal edge across gap ▪ Link and heal edge across overlap	Heals the open edge. This may increase the solid tolerance.
▪ Ignore intentional large hole	Marks the selected large hole as intentional.
▪ Mark problem area and enter edit mode ▪ Mark hole boundary and enter edit mode ▪ Mark gap boundary and enter edit mode ▪ Mark problem areas for later repair	Creates composite curves around the faulty surfaces and holes.
▪ Global fault with unknown fix	Not currently available. For future development.
▪ Enter edit mode	Enters surface edit mode without changing the solid.
▪ Fault repaired	Displays if the fault has been repaired.
▪ Heal edges at vertex	Heal edges of the surfaces that meet at a point.
▪ Replace vertex	Extracts, heals and replaces the surfaces.

Solid Doctor Healing dialog

Use this dialog to heal or relax a region where a solid operation has failed.

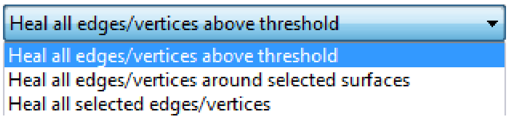
Edge tolerances can be healed (*tightened*) or relaxed (*loosened*). If a solid operation is failing, try loosening the edge tolerances in that area.

Healing and relaxing is optional. Normally it is only necessary to heal or relax locally if a solid operation fails and the solid has no faults.

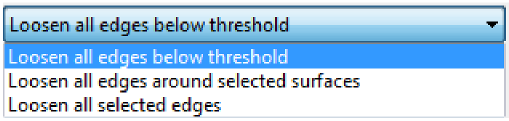


Use  and  to toggle the threshold that you are using.

When  is displayed, select one of the options form the drop-down list to heal a region.



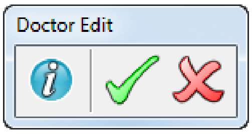
When  is displayed, select one of the options form the drop-down list to loosen a region.



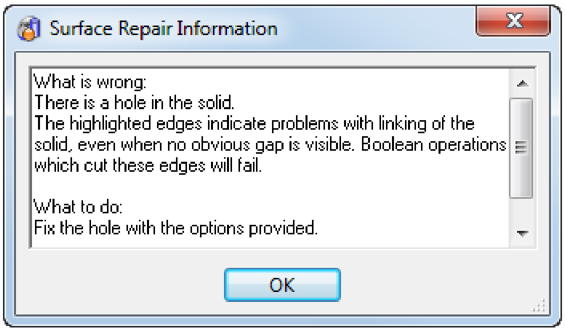
Doctor Edit toolbar

Use this toolbar with the selected repair option to use the surface creation and editing tools to fix the surface before adding it back into the solid.

The necessary curve and surface creation and editing tools are displayed.



Information  - Click to display the **Surface Repair Information** dialog box, that displays details of the suggested repair operations.



OK - Click to dismiss the dialog box.

Accept  - Click to accept the surface edit changes.

If a surface is selected, Solid Doctor tries to add the surface back into the solid.

Cancel  - Click to cancel any surface edit changes.

Tips on using Solid Doctor

It is useful to remember the following points when using **Solid Doctor**:

- When a fixing problems, look at the problem first and ask yourself how you would fix that problem using surfaces. Then make/repair the surfaces that way and sew them back into the solid.
- Run automatic fixing as the first step. Select the top entry in the fault tree and click  It won't fix all the problems, but it will usually drastically reduce the number of faults that need fixing using other options.
- Use  often to re-check the solid.
- Only selected surfaces are sewn into the solid when you leave **Doctor Edit** mode. You will be prompted if you forget to select a surface to be used.

- Gaps can be overlaps.
- If there are any tiny gaps between surfaces, consider removing the surfaces that are adjacent to the fault. Use one of the hole filling options to fix the fault.
- Use **Shift** and **Ctrl** to add and remove surfaces from the selection.

Make Watertight Wizard (version 8 solids only)

Make Watertight *is only used with version 8 solids. Post-version 8 solids use the* ***Solid Doctor****.*

This wizard makes a solid or mesh watertight by filling large gaps and sealing small gaps.

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

2 Click  (*Model Fixing toolbar*)

3 Use the pages of the wizard to make your solid watertight.

The **Make Watertight Wizard** consists of the following pages:

Set the maximum linking tolerance - This is the first page of the **Make Watertight Wizard**

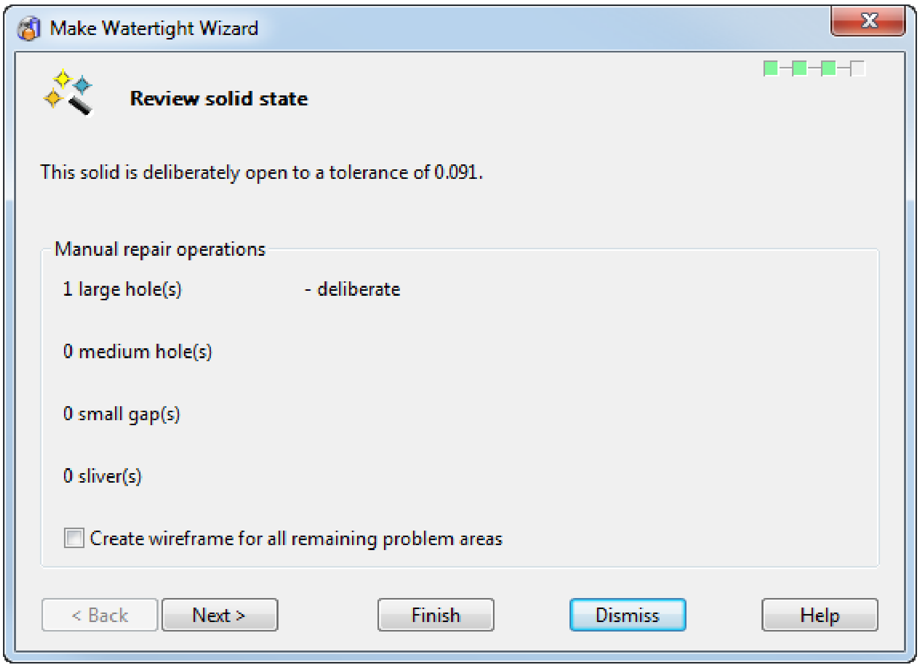
Options - There are additional controls on this page. It is accessed from the first page of the **Make Watertight Wizard**.

Repair holes in solids - This page is the second page of the **Make Watertight Wizard** and gives you additional control when repairing holes.

Review solid state - This is the third page of the **Make Watertight Wizard**.

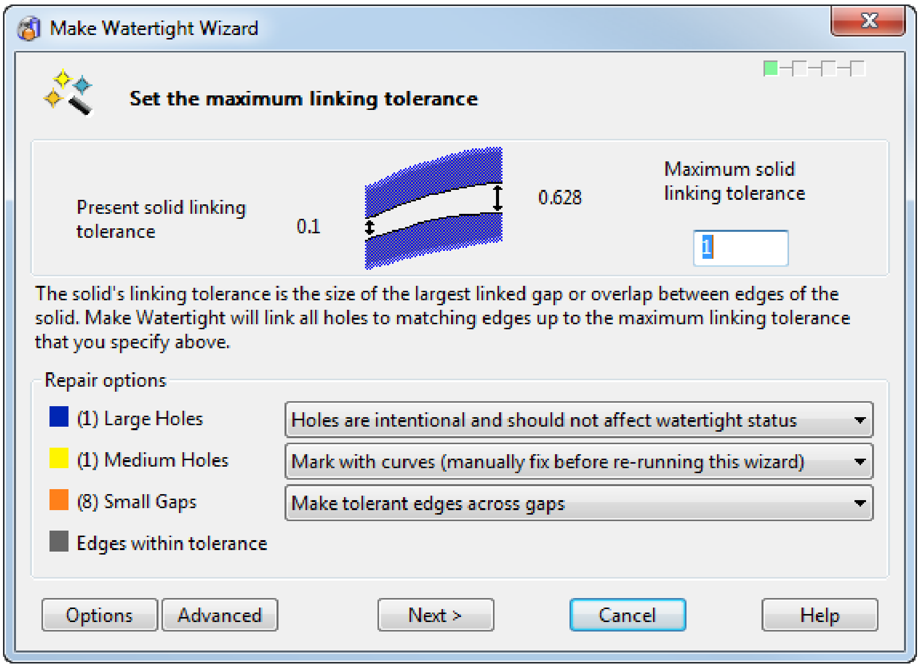
Heal existing surface links in the solid - This is the fourth page of the **Make Watertight Wizard**.

Wizard result - If the solid becomes watertight, a dialog is displayed that shows the number of holes and gaps that remain in the solid at a particular tolerance.



First page of Make Watertight Wizard - Set maximum linking tolerance

This page sets the maximum tolerance of the sealing operation.



Present solid linking tolerance - This is the latest tolerance to which the solid has been linked.

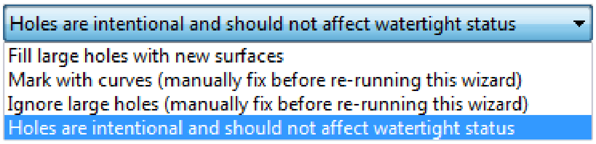
Maximum solid linking tolerance - The initial maximum tolerance set for the wizard will be the size of the largest small gap.

Repair Options

This section indicates the number of large holes, medium holes and small gaps in the model. You can either accept the default operations or select an option from the appropriate drop-down list.

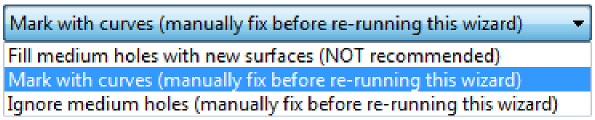
Large Holes - Large holes are highlighted blue on the model. A large hole is any hole greater than one hundred times the initial solid tolerance.

The options that are available from the **Large Holes** drop-down list are:



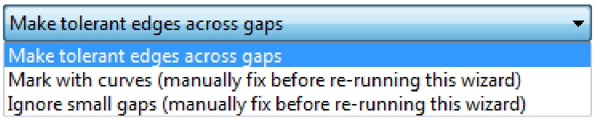
Medium Holes - Medium holes are highlighted yellow on the model. A medium hole is any hole larger than ten times the initial solid tolerance, but less than one hundred times the initial solid tolerance.

The options that are available from the **Medium Holes** drop-down list are:



Small Gaps - Small gaps are highlighted orange on the model. A small gap is any gap that is above the initial solid tolerance, but less than ten times the initial solid tolerance.

The options that are available from the **Small Gaps** drop-down list are:



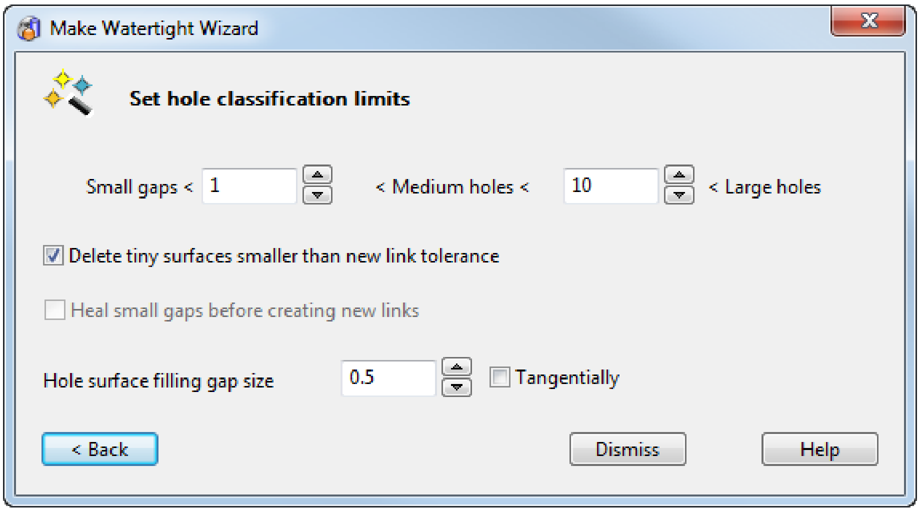
Edges within tolerance - This displays the default instrumention colour for edges within tolerance.

- Click this button to open the **Options** dialog (see page 298) to set the hole classification limits.

- Click this button to take you to the Second page of **Make Watertight Wizard** (see page 299) where you can set specific values in the options for filling holes.

Next > - If you select this option and the solid is watertight, the Fourth page of **Make Watertight Wizard** (see page 300) is displayed. In all other cases, a summary of the solid is displayed on the Third page of **Make Watertight Wizard** (see page 300).

Options dialog box



Small gaps < 1 - Use the up and down arrows to set the minimum hole width for medium holes.

< Medium holes < 10 - Use the up and down arrows to set the maximum hole width for medium holes.

Delete tiny surfaces smaller than new link tolerance - If *OFF*, limit the maximum linking tolerance to the smallest size. If *ON*, limit the maximum linking tolerance to the largest small gap size.

Heal small gaps before creating new links - Heal small gaps to the present solid tolerance before attempting to link the gap to a larger tolerance.

Hole surface filling gap size 0.5 - Use the up and down arrows to set the maximum permitted size of the gap in the boundary of any filled holes.

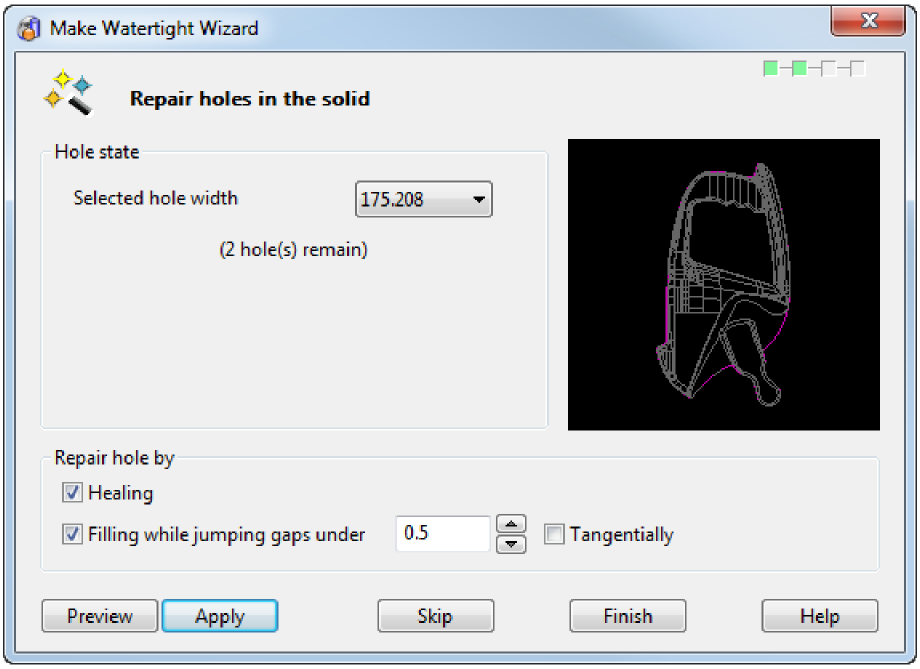
Tangentially - If *ON*, the surface is created tangent continuous to adjoining surfaces.

< Back - Return to the first page of the **Make Watertight Wizard**.

Dismiss - Close the **Make Watertight Wizard** without applying any changes.

Second page of Make Watertight Wizard - Repair holes in solid

This page of the wizard gives you additional control when repairing holes.



Hole state

The holes in the model are displayed in the drop-down list.

Selected hole width - Choose the hole to be repaired from the drop-down list.

Repair hole by

Use these options to control how the holes are repaired.

Healing - If *ON*, link hole within present tolerance.

Filling while jumping gaps - If *ON*, create a fill-in surface for the selected hole. If **Healing** is also *ON*, the fill-in surface will be created after any healing operation. Use the up and down arrows to set the maximum size of the gap in the boundary of the selected hole, or enter a value in the box.

Tangentially - If *ON*, the surface is created tangent continuous to adjoining surfaces.

Preview - Preview the repair of the selected hole, using the options you have selected.

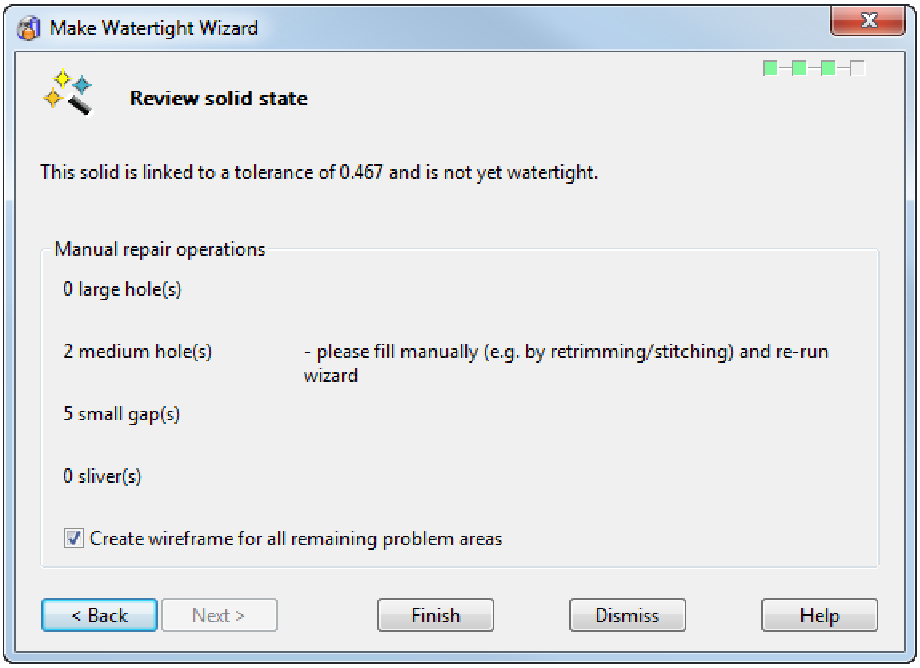
Apply - Repair the selected hole using the present repair settings, before proceeding to the next largest hole.

- Leave the selected hole in the solid and proceed onto the next largest hole.

- Leave **Advanced** options and return to the first page of **Make Watertight Wizard**.

Third page of Make Watertight Wizard - Review solid state

Use this page to review the current state of the solid you are attempting to make watertight. The **Manual repair operations** panel indicates the current state of the solid and your previous manual repair (wireframe creation) choices.



Create wireframe for all remaining problem areas - Create a wireframe for each area of the solid where there is a problem.

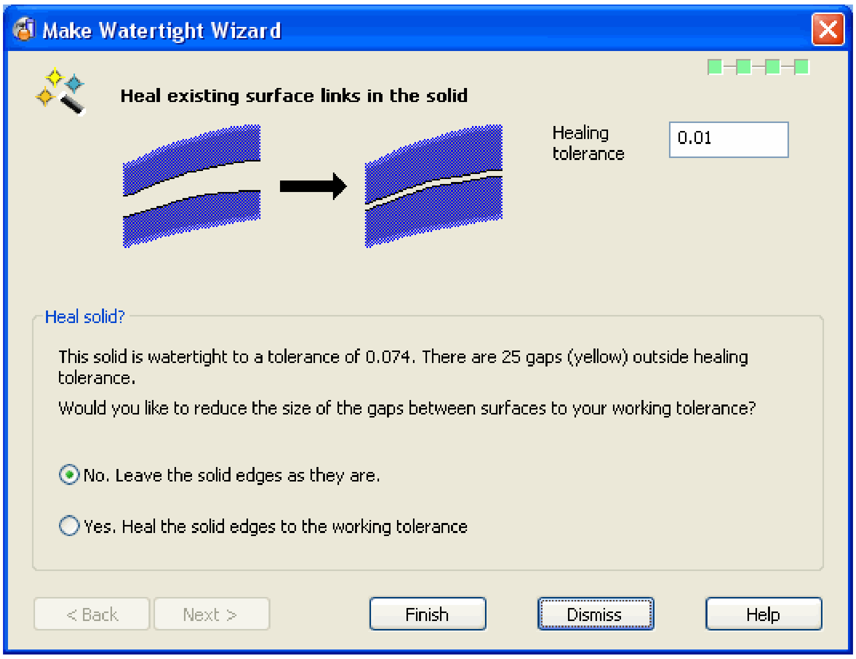
- Return to the first page of the **Make Watertight Wizard**.

- If the relevant option is *ON*, create wireframe for the selected problem areas and close the **Make Watertight Wizard**.

- Close the **Make Watertight Wizard** without applying any changes.

Fourth page of Make Watertight Wizard - Heal existing surface links

This page displays the number of gaps that are outside the healing tolerance. These gaps are highlighted in yellow on the model.



Healing tolerance - Enter a value to change the number of gaps outside the healing tolerance. The default value is the working tolerance.

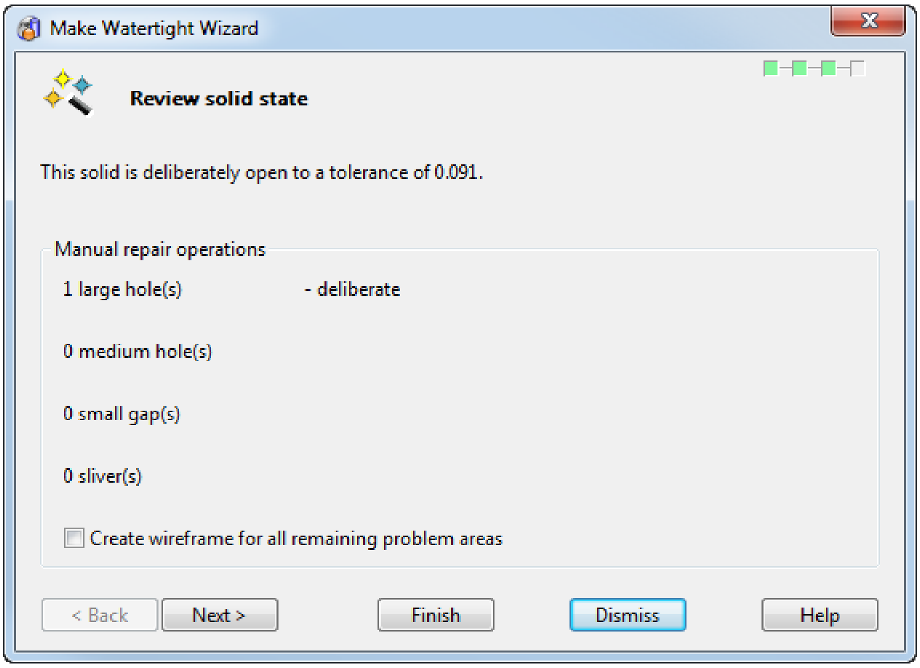
Heal Solid?

No, Leave the solid edges as they are - Do not alter the surfaces to reduce the tolerance of the solid edges.

Yes, Heal the solid edges to the working tolerance - Attempt to reduce the gaps between the linked solid edges by surface boundary manipulation.

- Perform the selected healing operation. If the healing process is unable to heal to the specified tolerance, a warning message is displayed.

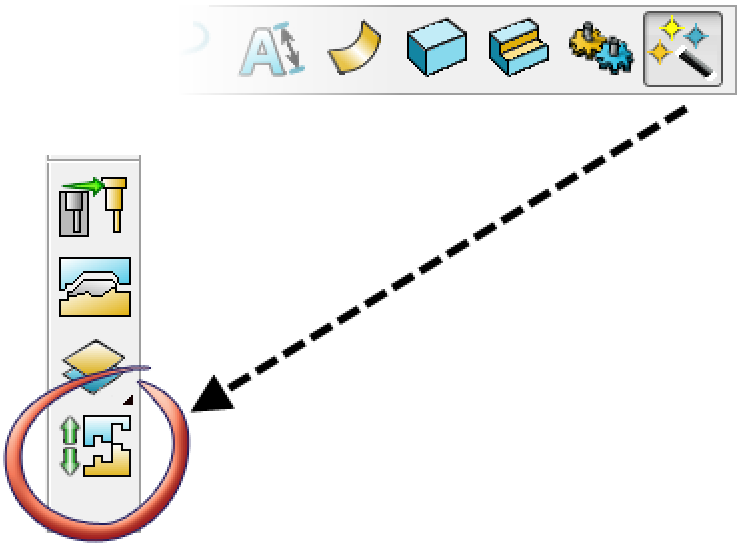
When you click **OK** on the warning dialog, you are returned to the wizard so that you can change the **Healing tolerance**. Selecting **Finish** again will then either display a further warning message (if the healing process again failed to heal to the specified tolerance), or display the **Make Watertight Wizard Result** page.



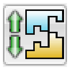
- Close the **Make Watertight Wizard**, creating any required wireframe.

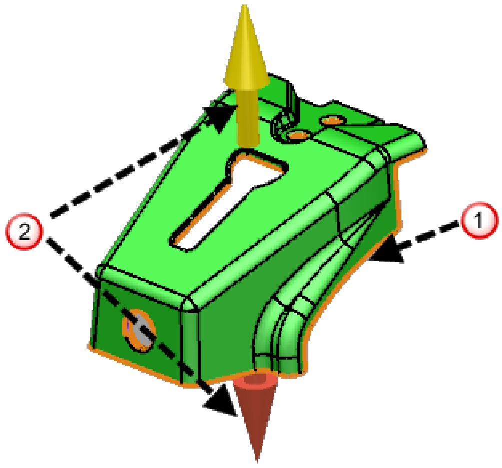
Solid Cavity/Core Separation

You can split a solid into core, cavity, sliders and internal features (such as holes).



1 Select the solid to separate.

2 Click  to display the automatic split line 1 and the draw directions for the part 2.



3 Use the **Solid Core/Cavity Separation** wizard (see page 303) to split a solid into core, cavity, sliders and internal features (such as holes).

When using **Solid Core/Cavity Separation** (see page 306):

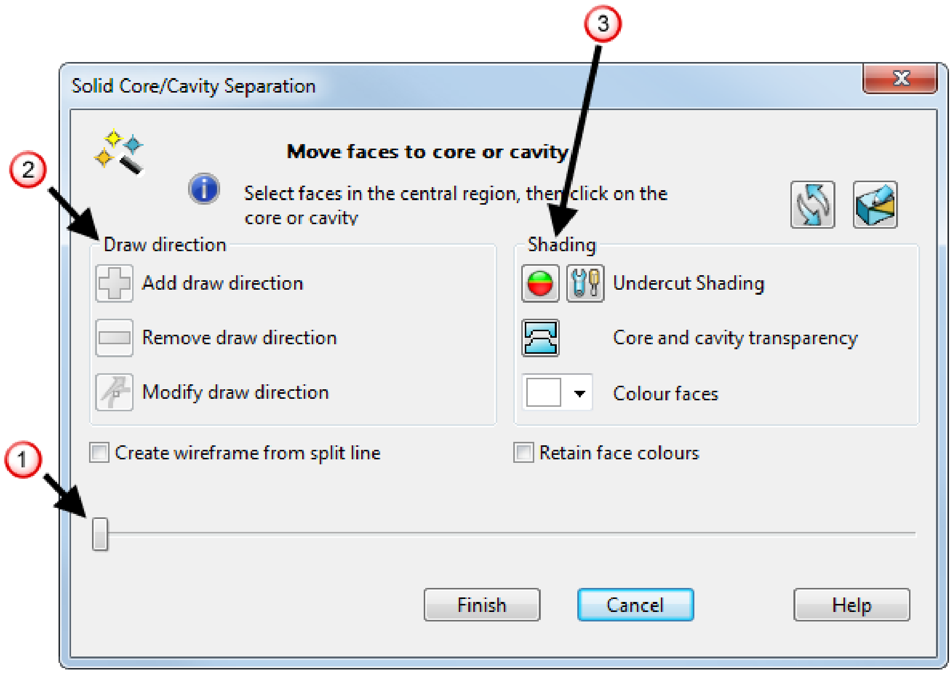
- solids are split based on the natural horizons from looking down the Z axis of the active workplane.
- parts are automatically split into core/cavity groups.
- you can move faces between different groups.
- any number of split directions can be used to define sliders.
- you can edit the split line by sketching new wireframe and dividing the relevant faces. The new faces can then be assigned to the relevant split group.

Solid Core/Cavity Separation wizard

Use the options on this wizard to:

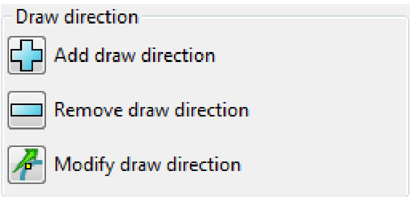
- simulate the separation of the core and cavity of a solid using the slider 1.
- control the draw direction 2.
- shade the core and cavity 3.

- create wireframe from the split line



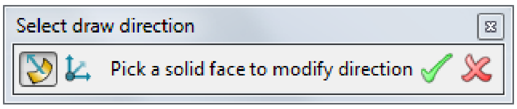
Draw direction

Use the options in this section to make changes to the draw direrction



 Add draw direction

1. Click a face to be used to define the draw direction.
2. Click  to display the **Select draw direction** toolbar.
3. Use the toolbar to define the draw direction.



 — Select a normal face to be used to define the draw direction.

 — Select a workplane and use one of the principal axes to define the draw direction.

 — Accept the current draw direction.

 — Cancel the creation of an additional draw direction.

Remove from draw direction — Click this button to remove the selected draw direction. Any faces being extracted along this draw direction become ambigous.

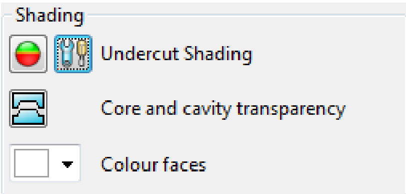
You cannot remove the Core/cavity directions.

Modify draw direction — Select a draw direction and click this button display the **Direction** dialog. Use this dialog to modify the selected draw direction.

You cannot modify the Core/cavity directions.

Shading

Use the options in this section to modify the shading of the core and cavity.



Undercut shading — Click to toggle undercut shading. In the Core/Cavity Separation wizard, undercut shading works slightly differently from standard undercut shading:.

> The areas highlighted in red are undercut with respect to the draw direction they will be extracted along; faces that cannot be extraced along either draw direction will be coloured red.

Core and cavity transparency — Click this button to toggle the transparency of the core/cavity faces. This lets you see internal features more clearly in relation to the core and cavity.

The transparent faces cannot be selected; only the internal features can be selected.

Colour faces — Use this selection drop-down list to choose the colour of the faces to be extracted along the current draw selection.

Create wireframe from split line — When this option is selected, wireframe will be created along the split line.

Retain face colours — Select this option to use the wizard colours on the faces after splitting.

— Click this button to re-calcualate the split line and face classification. This may be required if you have made a lot of changes to the model or the draw direction.

— Click this button to access other PowerSHAPE functionality whilst remaining inside the **Solid Core/Cavity Separation Wizard**. This means that you use a range of operations including:

- creating wireframe.
- using solid editing and Direct Modelling tools to divide faces.

Use the buttons on the **Split faces** toolbar to re-enter the wizard:

— Accept the changes you have made and re-enter the wizard.

— Cancel the changes you have made and re-enter the wizard.

Finish — Click this button to split the part and close the dialog.

Cancel — Click this button to undo all the changes you have made and close the dialog.

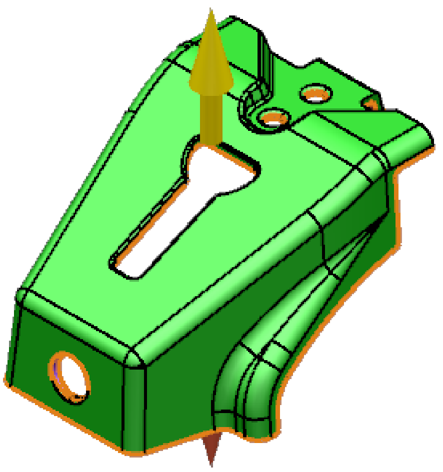
The part is not split and any wireframe you have sketched or changes you have made whilst using the Solid Core/Cavity Separation dialog.

Example — Separating a solid cavity and core

Use the **Solid Core/Cavity Separation** wizard to divide a part into its molding pieces.

1 Select the part.

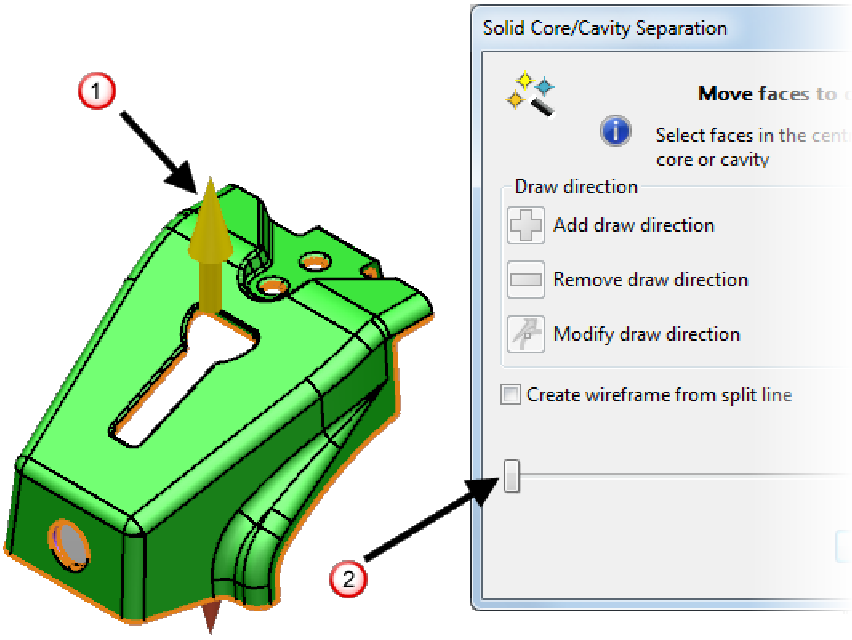
2 Select (*Wizards toolbar*).



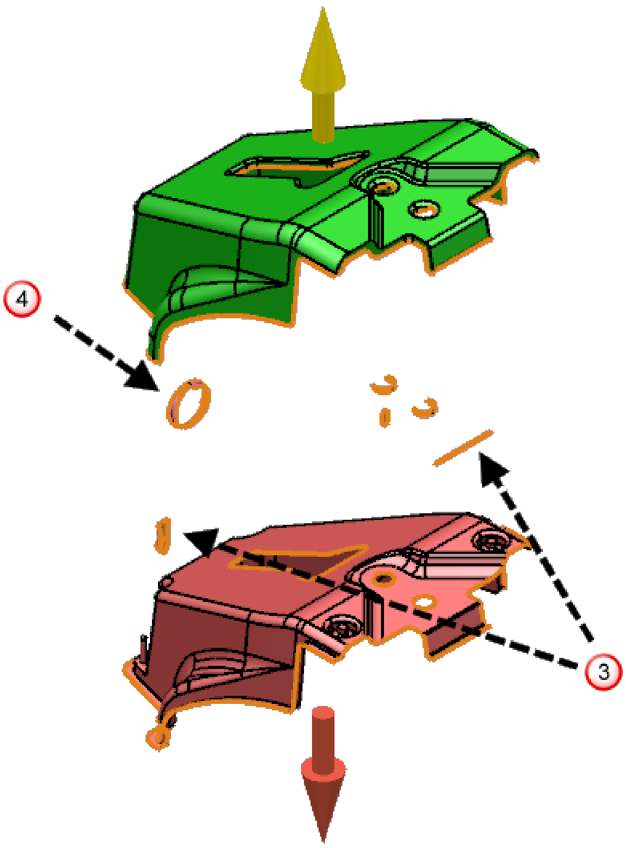
The part is split into the cavity and core sections.

3 Open the part using one of the following:

- draw direction arrows 1.
- slider on the dialog 2.



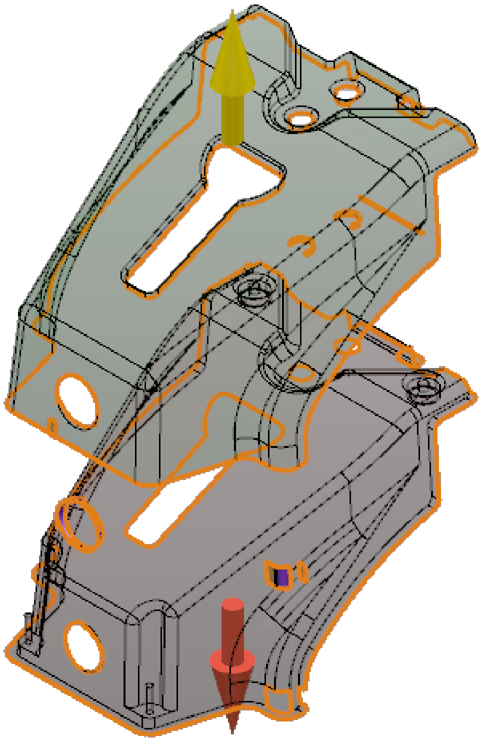
Any ambigous faces remain in the central region 3 and 4.



4 Click  to select **Undercut shading**; this will indicate faces that have insufficient draft when viewed from above or below the tool.

5 Click to deselect **Undercut shading**.

6 Click to select **Core and cavity transparency**. This lets you see clearly the faces that are ambigous, that is,they are not assigned to the cavity or the core.



You can assign ambigous faces to a part in translucent mode, but you can only assign a face by clicking the draw direction arrow.

If you are not in transluscent mode, you can also assign an ambiguous face by clicking on the core or cavity.

7 Click to deselect **Core and cavity transparency.**

8 Rotate the model so that you can see the individual ambiguous faces.

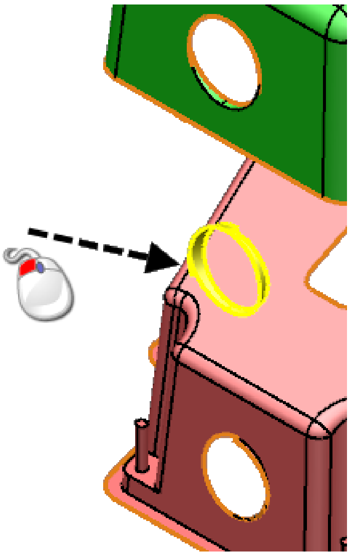
9 Assign the indicated ambiguous faces 3 to the core as follows:

a Select the ambigous face.

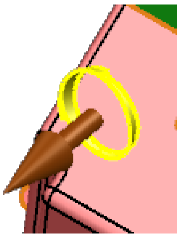
b Click either the face of the core or the draw direction arrow

10 To assign the ambigous faces 4, you need to create a new line of draw as follows:

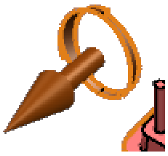
a Select the ambiguous face.



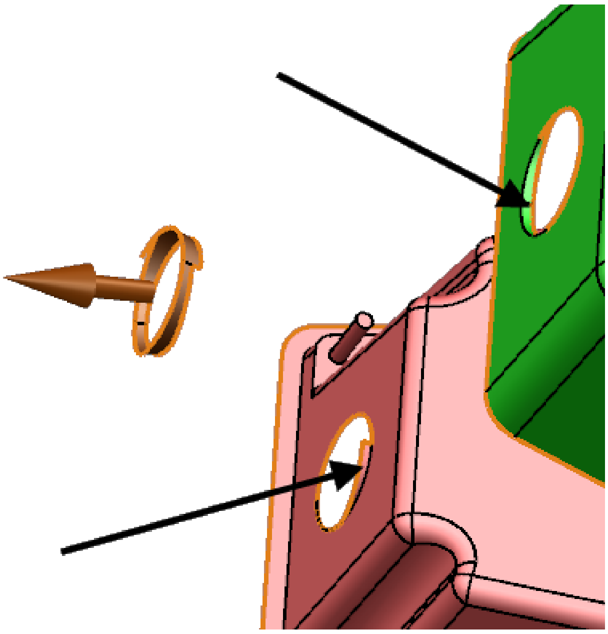
b Click  to display a default draw direction for the selected face.



c Click  (*Select draw direction toolbar*) to accept the default draw direction and assign the selected part to the new draw direction.



11 Assign the indicated the remaining ambigous faces to the new draw direction.



12 Close the part.

13 Select **Create wireframe from split line** create wireframe that represents the split line.

14 Click **Finish**. You now have

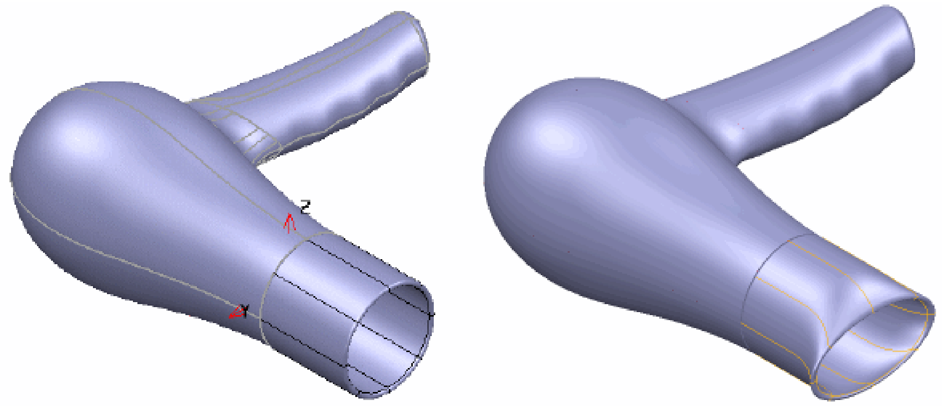
- three solids: the core, the cavity and the side core that you created in the new draw direction.
- wireframe that represents the split line.

Morphing

The following sections contain information on creating morphs.

Introduction to morphing solids

Morphing allows you to deform a solid into a different shape.



All the features on the objects such as fillets, trimming, walls, ribs and bosses 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, surfaces or a flexible box 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.

Adding a morph feature

Ensure you have an active solid displayed or pre-select an inactive solid and use one of the following methods to add a morph feature.

Creating a morph using the Solid Feature toolbar

1. Click **Feature** (*Main toolbar*) to display the **Solid Feature** toolbar.
2. Click **Create a morph feature** (*Solid Feature toolbar*).
3. Use the Morphing dialog (see page 313) to add the morph feature.

Creating a morph using the General Edits toolbar

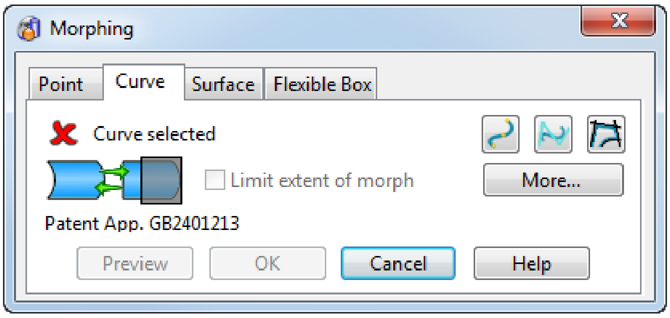
1. Click **show general edits options** to display the **General Edits** toolbar.
2. Select **Create a morph feature** (*General Edits toolbar*).
3. Use the Morphing dialog (see page 313) to add the morph feature.

Creating a morph using the Object menu

1. Select **Morph** from the **Object > Feature** menu.
2. Use the Morphing dialog (see page 313)to add the morph feature.

Morphing dialog

Use this dialog to morph a solid.



The **Morphing** dialog contains these tabs:

Point (see page 313)

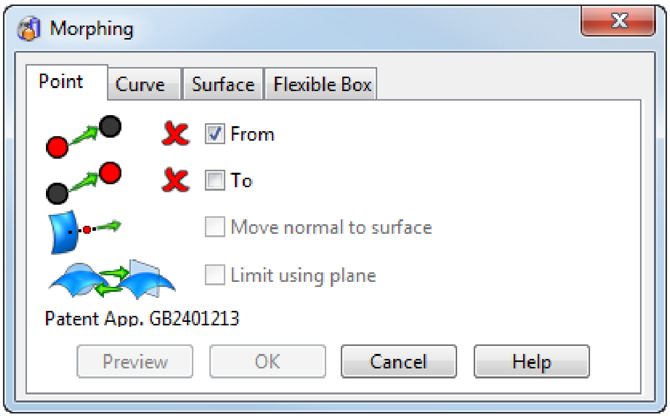
Curve (see page 317)

Surface (see page 325)

Flexible Box (see page 329)

Point tab

Use this tab of the **Morphing** dialog to define a morph using two points.



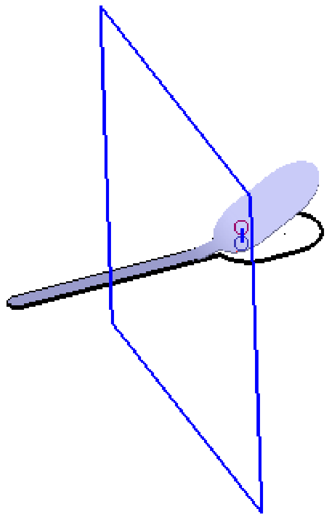
From — This allows you to select or modify the origin point that usually lies on the surface of the model. It is shown as a small blue circle.

To — This allows you to select or modify the distortion point that can lie anywhere in space. It will be indicated by a small red circle.

Move normal to surface — If selected, the position of the distortion point (**To**) can lie only along a line normal to the surface at the origin point. To select a the distortion point away from the line normal to the surface, deselect this option. If the origin (**From**) you selected lies on the surface of the model, this option is selected automatically.

Limit using plane — This option is ideal for altering the toe-spring of a shoe and similar applications where you want to bend a portion of a model. If selected, the bounding spheres are replaced with a limiting plane. The orientation of the plane is defined by the selected principle axis of the active workplane. You can drag the boundaries of the plane to adjust its position.

The model is bent so that the origin is moved to the distortion point. Points that are closer to the limiting plane are moved less and points at or beyond the limiting plane are not moved at all.



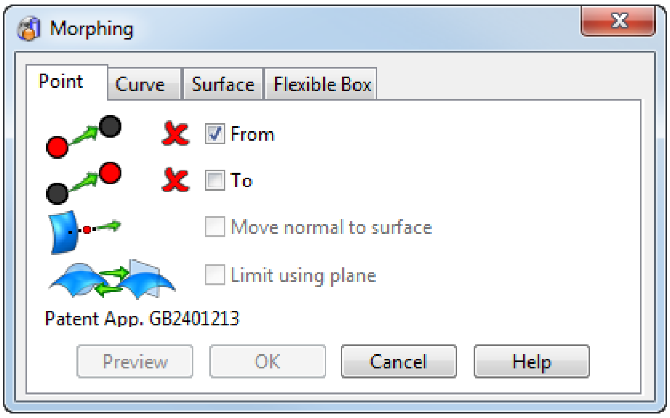
— The model is updated dynamically as you make changes. Click this button if the model is distorted by a large amount to ensure that the model reflects all the changes.

— Click OK to create the morph.

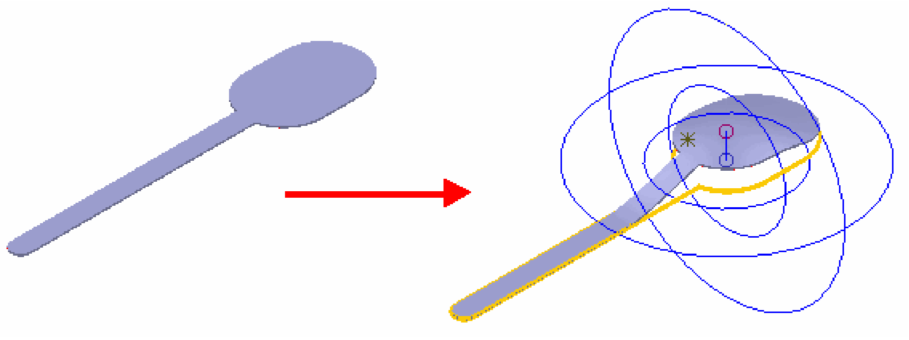
Creating a morph using the Point method

This morphing method uses two points and enables you to make bulges in the model or apply simple bending or stretching distortions.

When the **Point** tab (see page 313) on the **Morphing** dialog is selected, you select two points to define the morph. The **From** point is the origin and the **To** point is the distortion point.



When you create the morph, the bulge that is produced by the distortion is centred on the origin point and the height and direction of the bulge is determined by the distance from the origin point to the distortion point.

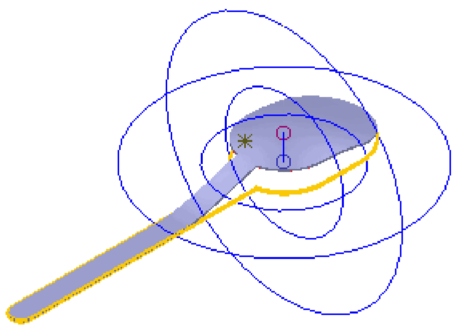


To create a morph

1 Select the **From** point to be used in the morphing process. This can be done by clicking on the model or using the **Position** dialog.

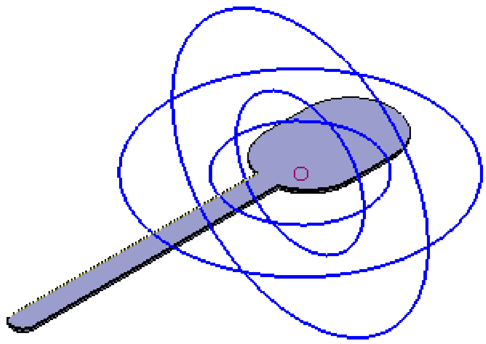
Depending on the selection you made for the **From** point, one of the following will happen:

- In many cases, when you select the **From** point, a **To** point will be automatically chosen and an initial bulge displayed.



You can modify the points (and therefore the bulge) using one of the methods described in the next section.

- If the selection you made for the **From** point does not automatically generate a bulge, the distortion point and the origin are in the same place (indicated by the red circle).

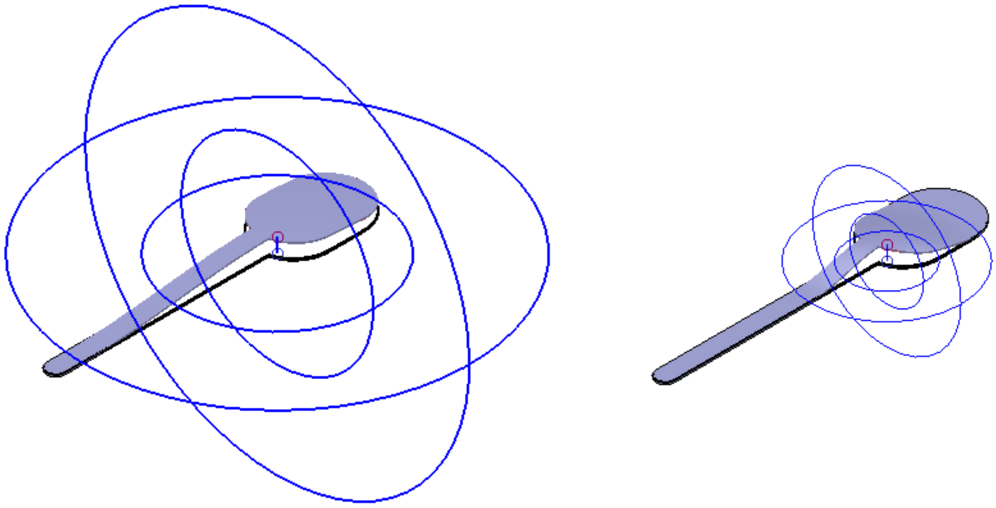


To create the bulge you require you must modify the distortion point, origin point, size and shape of the morph using the methods below.

2 Modify the size and shape of the distortion by using one of the following methods:

- Drag the point to be modified.
- Use the **Position** dialog to enter the position of the point. The point that is indicated by **From** and **To** in the dialog is the point that will be set.
- In the **Morphing** dialog, select either **From** (to modify the origin), or **To** (to modify the distortion point). Click the model to move the point.
- Use the two spheres to define the extent and shape of the bulge. You can change the bulge by using the mouse to drag a sphere. The graphics are updated when you release the mouse button.

Altering the size of the outer sphere changes the extent of the morph a shown below.

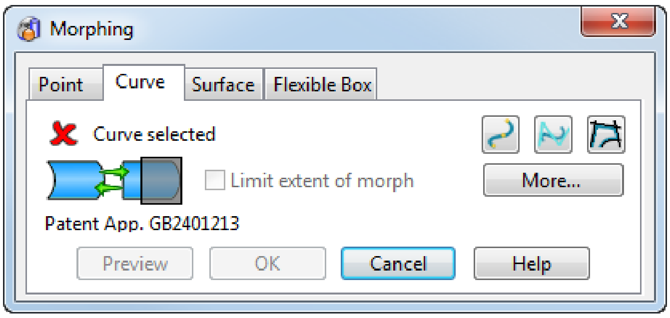


Altering the inner sphere affects the shape of the morph. Decreasing the size of the inner sphere means that the morph is sharper. Increasing the size of the inner sphere produces a more rounded bulge.

To avoid introducing folds in the solid, the radius of the outer sphere should generally be at least twice the distance from the origin to the distortion point.

If the ***From*** *point you selected was on the surface of the model, the* ***Move normal to surface*** *option will be ON. Whilst this option is ON, the distortion point can only lie along a line normal to the surface at that point. Turning the* ***Move normal to surface*** *option OFF will allow you to move the distortion point anywhere in space.*

Curve tab



Curve selected - When you have selected or created an appropriate curve, The cross icon changes to a tick icon..

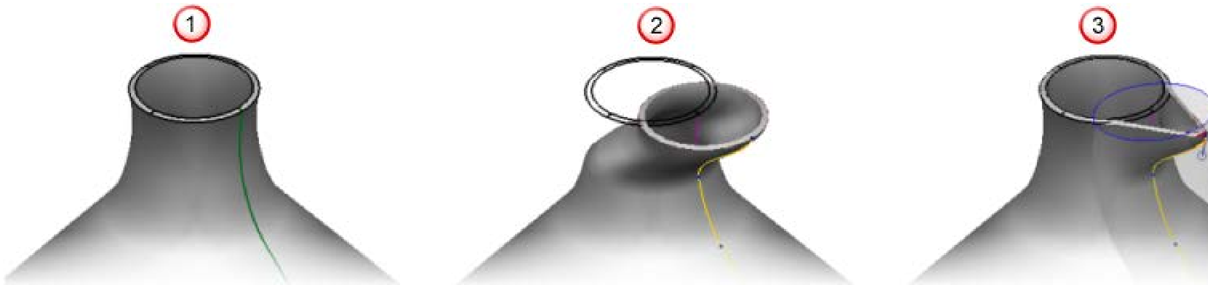
- Create a **Bezier** curve that will be used to define the morph.

- Create a **B-spline** curve that will be used to define the morph.

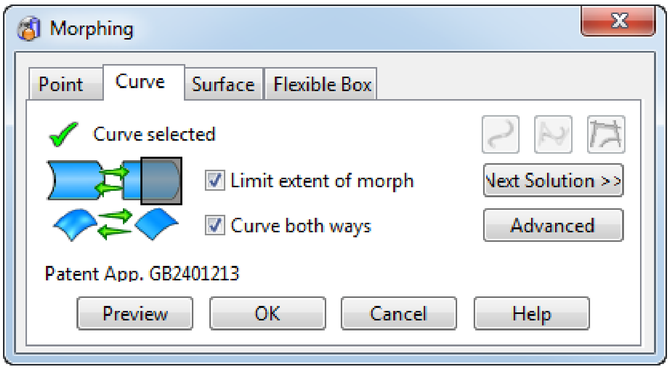
 - Trace a curve that will be used to define the morph.

Limit extent of morph - Allows you to morph a localised region of the model, while keeping the rest of the model unchanged. An indication of the region of the model that the morph affects is indicated by a translucent cylinder or a pair of parallel planes. These can be dragged to change the region that will be affected and hence the shape of the model. The model below shows the effect of using this option.

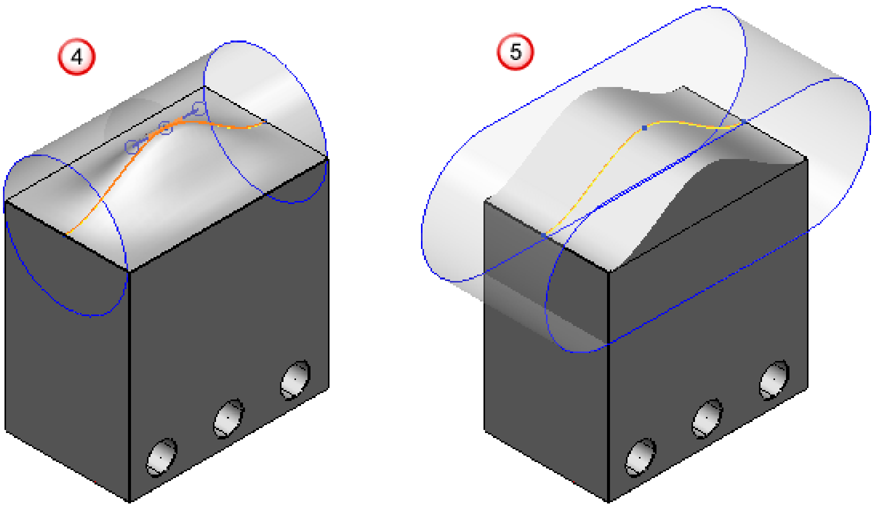
- The original model 1.
- Distortion with **Limit extent of morph** *OFF* 2.
- Distortion with **Limit extent of morph** *ON* 3.



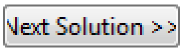
 - Display additional options on the dialog relating to the shape of the distortions created when the morph is limited.



Curve both ways - This option affects the extent the morph is limited. If *ON*, the model is curved in two directions relative to the curve 4. If *OFF*, the distortion will be kept constant along an axis controlled by the current principal plane 5. If no workplanes exist in the model, the **World** workplane is used.



*The principal plane is selected by choosing a button in the status bar. If you want to select the **YZ** principal plane, select the X button* .



- Cycle through a number of different possible shapes for the bulge that has been generated by applying limits to a distortion. Step through the four possible solutions and select the one that is best for your needs.

- Display additional options that let you separately select the two curves to be used in morphing. In addition, it allows you to use a curve or surface to limit the region that is affected by the morph.

— The model is updated dynamically as you make changes. Click this button if the model is distorted by a large amount to ensure that the model reflects all the changes.

— Click OK to create the morph.

Curve tab, advanced options

This page allows you to select two curves and then use them to define a morph. It also allows you to limit the morph using a decay curve of a surface. You should bear the following important points in mind when creating the two curves:

- Both the reference (**From**) and control (**To**) curve must have the same number of points.

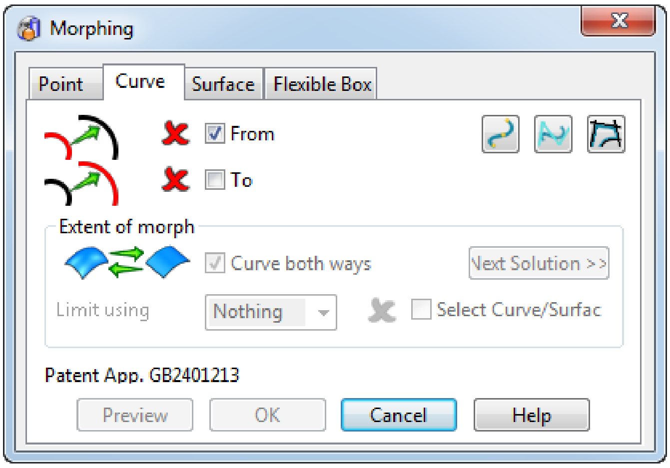
- Both the reference (**From**) and control (**To**) curve must be open or both closed.
- Keep the reference (**From**) and control (**To**) curves simple.
- If the curves are open, it is recommended that the start and end points of the curves coincide.
- If open curves are also tangential at the start and end, you will get smoother results in cases where the model to be morphed extends beyond the ends of the curve.
- Limit curves and surfaces should not cross the reference (**From**) curve.

You should bear the following important points in mind when creating the limit curve or surface.

- Keep the limit curve or surface simple.
- As you move over the model, it is important that there is always a unique closest point on the curve or surface and that the closest point does not jump from one place to another. Otherwise, the morphed model may contain kinks.

In contrast, the model being morphed may be as complicated as you like.

The **Advanced** options let you separately select the two curves to be used in morphing. In addition, it allows you to use a curve or surface to limit the region that is affected by the morph.



From - With this option selected, choose the reference curve and The cross icon changes to a tick icon..

To - With this option selected, choose the control curve and The cross icon changes to a tick icon..

 - Create a **Bezier** curve that will be used to define the morph.

 - Create a **B-spline** curve that will be used to define the morph.

 - Trace a curve that will be used to define the morph.

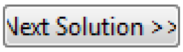
Extent of morph

Curve both ways - This option affects the extent the morph is limited. If *ON*, the model is curved in two directions relative to the curve. If *OFF*, the distortion will be kept constant along an axis controlled by the current principal plane. If no workplanes exist in the model, the **World** workplane is used.

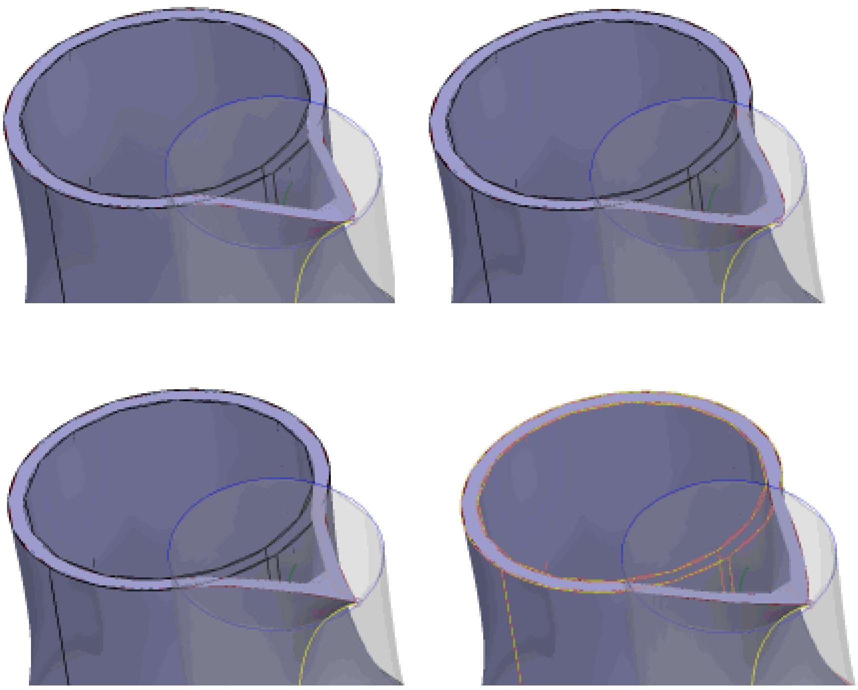
Limit using - Select one of the options from the drop-down list to use a curve or surface to limit the extent of the morph. You can choose from the following options:

- Nothing
- Distance
- Curve
- Surface

Select Curve/Surface - This option becomes active when you select **Curve** or **Surface** from the drop-down list. Select this option and then select the appropriate limiting curve or surface. If the selection is successful, The cross  icon changes to a tick  icon..



- Cycle through the four possible solutions and select the one that is best for your needs.

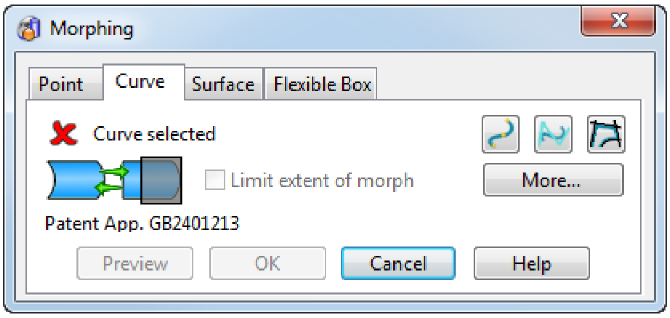


Points are moved depending on the distance from their corresponding point on the **From** curve and the distance from the closest point on the selected limit curve or surface. The closer the point is to the limit curve or surface, the less the point moves. Beyond the selected curve or surface, the points are not moved at all.

*The **Curve** and **Surface** options take longer to evaluate than **Distance** and so should only be used when necessary.*

Creating a morph using the Curve method

This method lets you sketch a curve on or around the model and then allows you to deform the curve to change the shape of the model. Alternatively, you can select two appropriate curves.



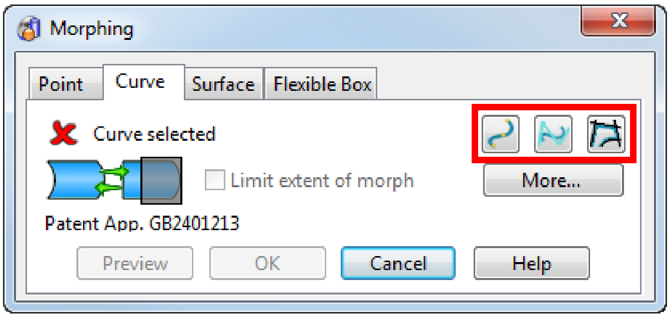
The curve method of creating morphs gives you more control over the shape of the resulting bulge than the point method.

To create a morph

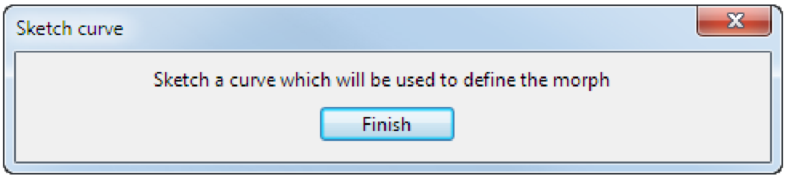
1 Select the solid to be morphed and click **Create a morph feature**  (*Solid Feature toolbar*).

2 Select the **Curve** tab on the **Morphing** dialog.

3 Select or create a curve on or around the model. This curve is called the reference curve and can be edited to bend or stretch the model. If a suitable curve already exists, it can be selected as normal. Otherwise, create a suitable curve using the curve buttons on the dialog

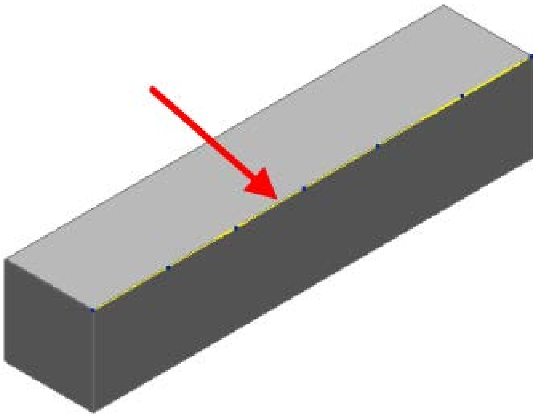


When sketching the curve, the **Sketch curve** dialog is displayed.

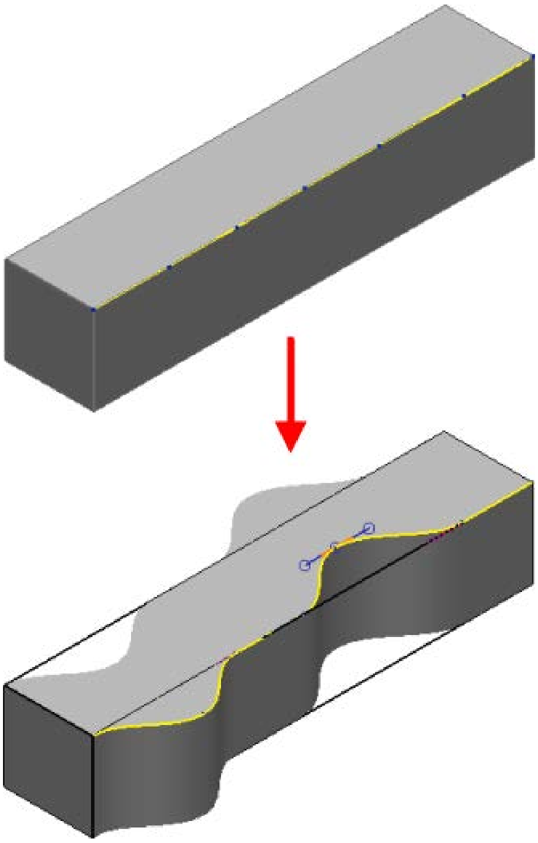


Select **Finish** on the **Sketch curve** dialog to complete the curve and start dynamic morphing.

The example below shows a simple block. A reference curve has been created along one edge of the block.



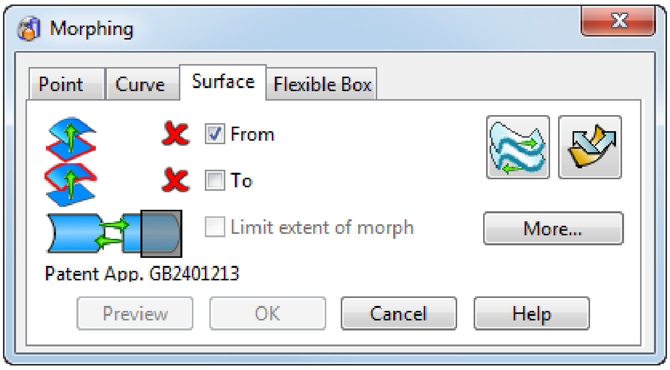
4 Click on a point and use the curve handles to change the shape of the curve. The distortion is applied so that the points on the original curve are moved to the corresponding point on the edited curve and the model is distorted in a similar way. The graphics will update when the curve handles are released. The original model is displayed in wireframe and the distorted model in shaded graphics.



When selecting or creating the reference curve, you should bear the following in mind:

- *Curves with fewer points are easier to edit and will generally give the best distortion results.*
- *Points on the curve should be placed so that they will give the curve flexibility in areas where you need to control the distortion.*
- *Discontinuities and sharply curved corners in the curve may introduce kinks or ripples into the morphed model.*

Surface tab



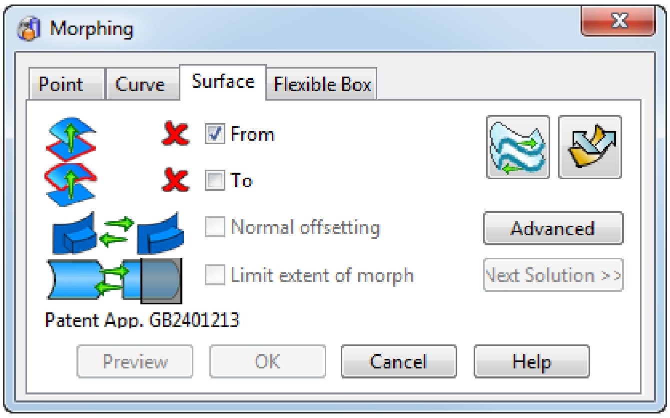
From - With this option selected, choose the surface to distort from and The cross icon changes to a tick icon..

To - With this option selected, choose the surface to distort to and The cross icon changes to a tick icon..

Once valid surfaces have been selected, the morph will be automatically previewed.

Limit extent of morph - Morph a localised region of the model, while keeping the rest of the model unchanged.

- Display additional options on the dialog relating to the way the morphed region is blended with the rest of the model.



Normal Offetting - If *ON*, the surfaces to be morphed are bent to follow the **To** surface. Full details on how selected objects are morphed when **Normal Offsetting** are provided in the Solid Modelling manual.

 - Reverse surface

 - Swap longitudinals and laterals

Next Solution >> - When **Limit extent of morph** is *ON*, this option allows you to cycle through the four available types of blend. Step through the available solutions to find the blend that produces the best result for you.

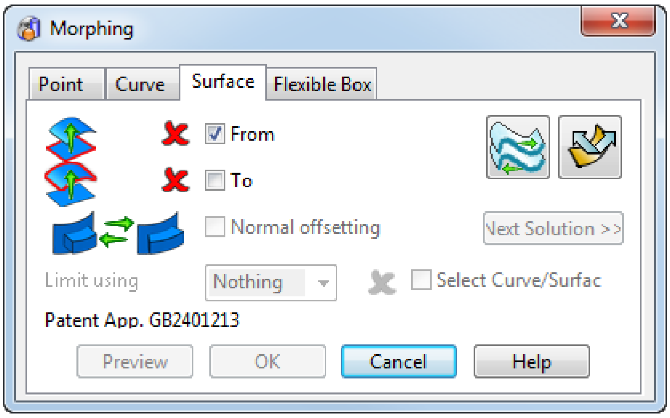
Advanced - Displays additional options on the dialog to modify the shape of the morph object.

Preview — The model is updated dynamically as you make changes. Click this button if the model is distorted by a large amount to ensure that the model reflects all the changes.

OK — Click OK to create the morph.

Surface tab, advanced options

The **Advanced** options let you to use a curve or surface to limit the region that is affected by the morph.



Limit using - Select one of the options from the drop-down list to use a curve or surface to limit the extent of the morph. You can choose from the following options:

- Nothing
- Distance
- Curve
- Surface

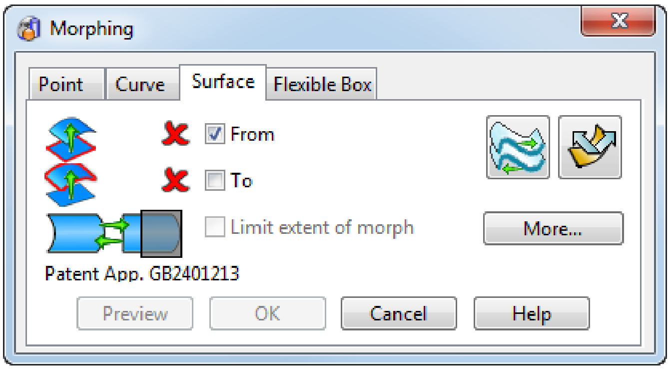
Select Curve/Surface - This option becomes active when you select **Curve** or **Surface** from the drop-down list. Select this option and then select the appropriate limiting curve or surface. If the selection is successful, The cross icon changes to a tick ✓ icon.

Next Solution >> - Cycle through the four possible solutions and select the one that is best for your needs.

Points are moved depending on the distance from their corresponding point on the **From** surface and the distance from the closest point on the selected limit curve or surface. The closer the point is to the limit curve or surface, the less the point moves. Beyond the selected curve or surface, the points are not moved at all.

Creating a morph using the Surface method

This uses two surfaces 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 surfaces you select must have the same number of laterals and longitudinals.
- The laterals on both selected surfaces must be open or both closed.
- The longitudinals on both surfaces must be open or both closed.
- Keep the surfaces simple.
- Keep the limit curve or surface simple.
- Limit curves or surfaces should not cut the surface that you have chosen as the **From** surface.

- Avoid discontinuities and sharply curved corners in the 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.

In contrast, the model being morphed may be as complicated as you like, without there being any problems.

To create a morph using the surface method

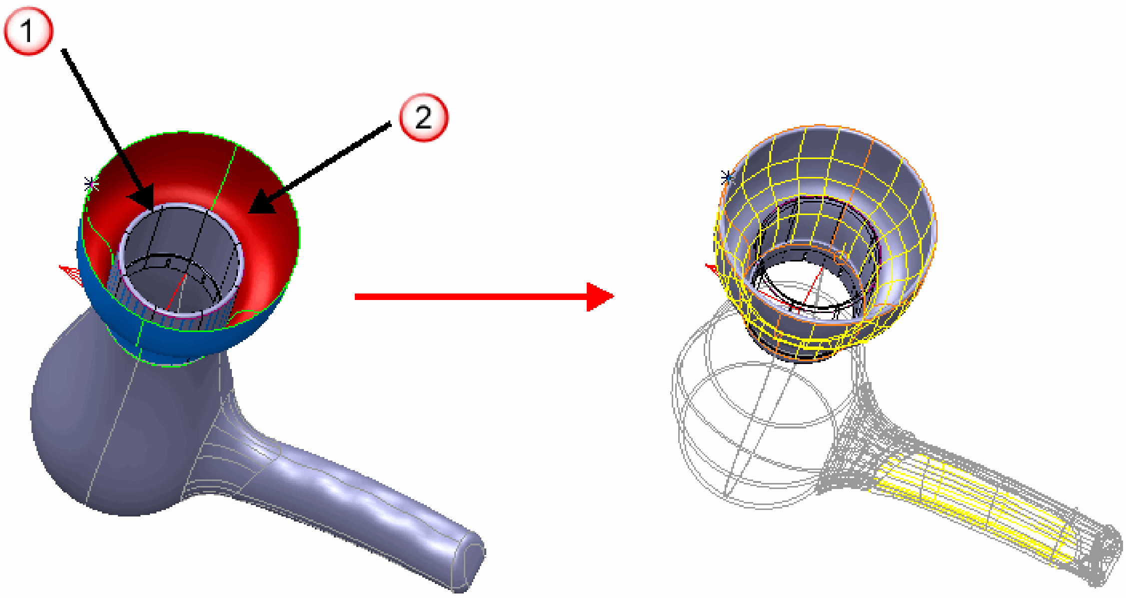
1 Select the solid to be morphed and click **Create a morph feature**  (*Solid Feature toolbar*).

2 Select the **Surface** tab.

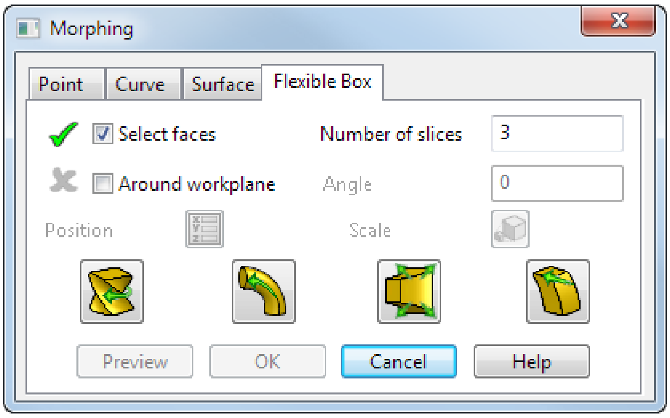
3 Select the **From** surface to be used in the morphing process 1. The ✗ changes to ✓.

4 Select the **To** surface to be used 2. The ✗ changes to ✓.

Once valid surfaces have been selected, the morph will be automatically previewed. The original model will be displayed in wireframe and the distorted model will be shown in shaded mode.



Flexible Box tab



Select faces - If *ON*, you can select the region of the model that you want the morph to affect. The entire model will be selected by default, but you can select smaller regions of the model. You can select multiple faces on a solid as follows:

- Hold down **Shift** and select a surface to add a surface to the selection.
- Hold down **Ctrl** and select a surface to add/remove a surface from the selection.

Number of slices - The initial number of control slices in the flexible box. The flexible box will be distorted by a moving drag handles attached to a the control slices through the box.

*You can create additional control slices at a later date by holding down **Ctrl** and dragging the active control slice. Twisting and bending the box may insert new control slices.*

Around workplane - If *ON*, you can select a workplane that defines an axis for bending or twisting the box. If you want to use this option, you should create a workplane that has an axis that will be used to twist or bend the box. Make sure that the principal plane you have selected is correct for the axis of the workplane you would like to use.

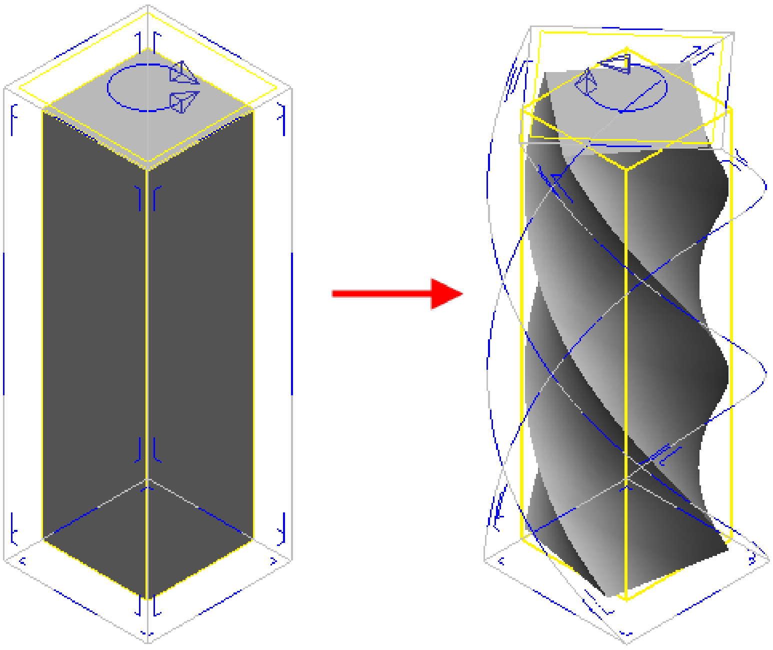
You can turn this option *OFF* at any time and continue free form editing of the box.

Angle - Enter the angle to be used when bending or twisting the box. This option is only active when the **Around Workplane** option is selected.

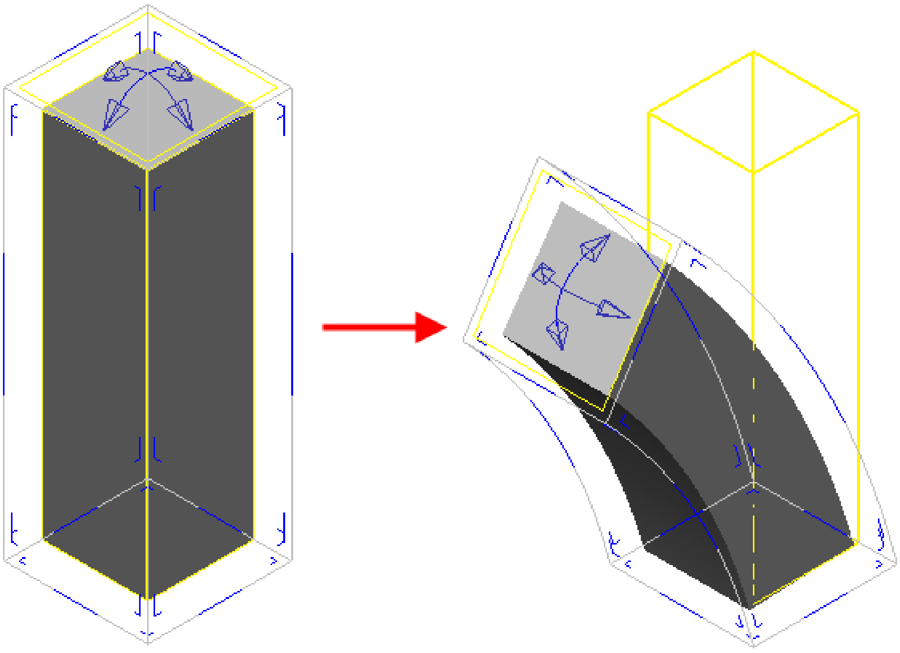
Once you are happy with the selected region of the model, you can select a button to show the flexible box and start editing the model dynamically. The initial box will be large enough to surround all the surfaces you have selected and will be oriented along either the axes of the current active workplane or the axes of the World workplane.

The following buttons let you change the type of operation you perform on the box.

 - **Twist** the model interactively.

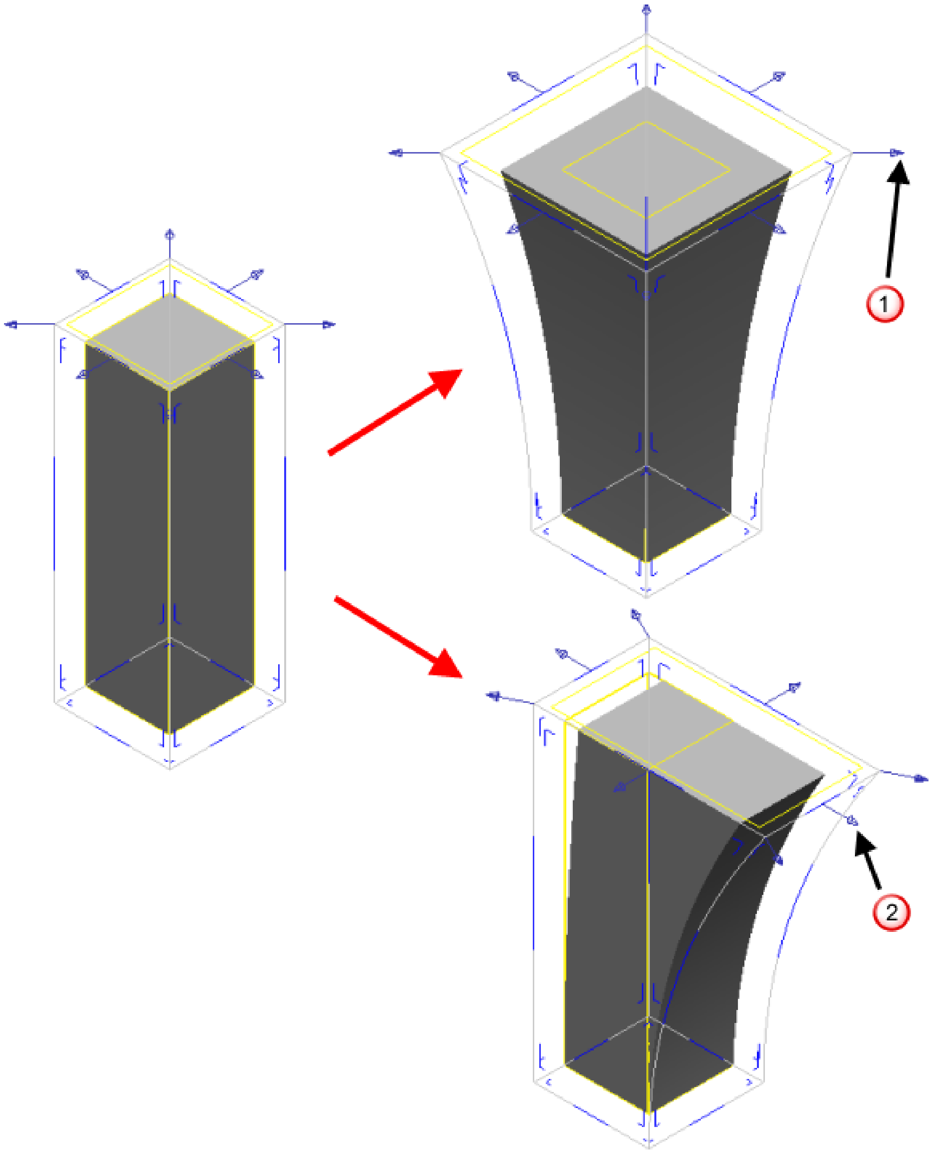


 - **Bend** the model interactively.

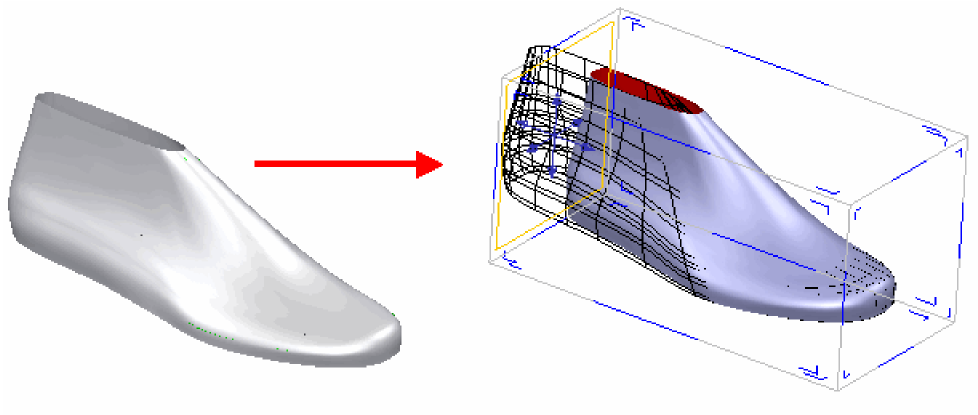


 - **Stretch** the model. Two kinds of stretching can be used:

- Symmetric - Scales each selected slice around its centre 1.
- Asymmetric - Stretches the slice only in the direction of the arrow 2.


1
2

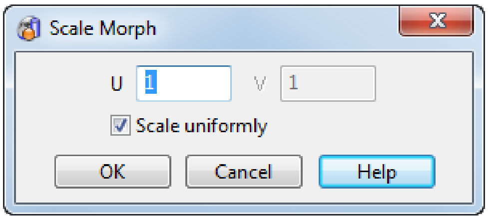
 - **Shear**. This lets you move control surfaces about in 3D. To make this easier, the handles in the plane of the control surface will only move the slice in that plane. The handles that are normal to the control surface will move the surface normal to itself.



You can swap between the different drag handles at any time and make as many different edits to the box as you like before applying the morph.

 - Display the **Position** dialog.

 - Open the **Scale Morph** dialog. Use this dialog to scale the morph:



1. Enter values to scale the morph in **U** and **V** direction, where **U** and **V** are the axis tangent to the edges of the morphing bounding box
2. Select **Scale uniformly** to scale the morph by the same amount in all directions. Deselect to scale by different amounts in each axis.

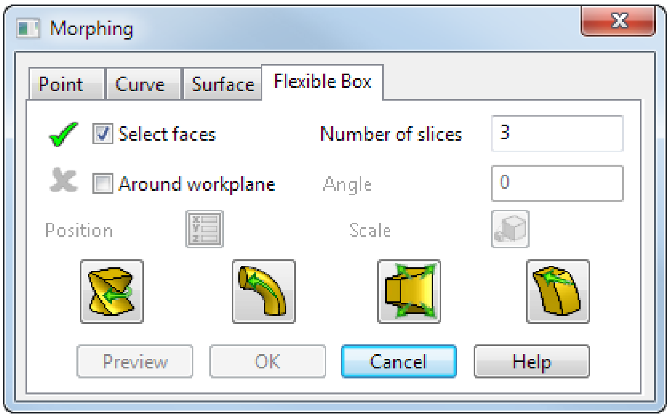
— The model is updated dynamically as you make changes. Click this button if the model is distorted by a large amount to ensure that the model reflects all the changes.

— Click OK to create the morph.

Creating a morph using the flexible box method

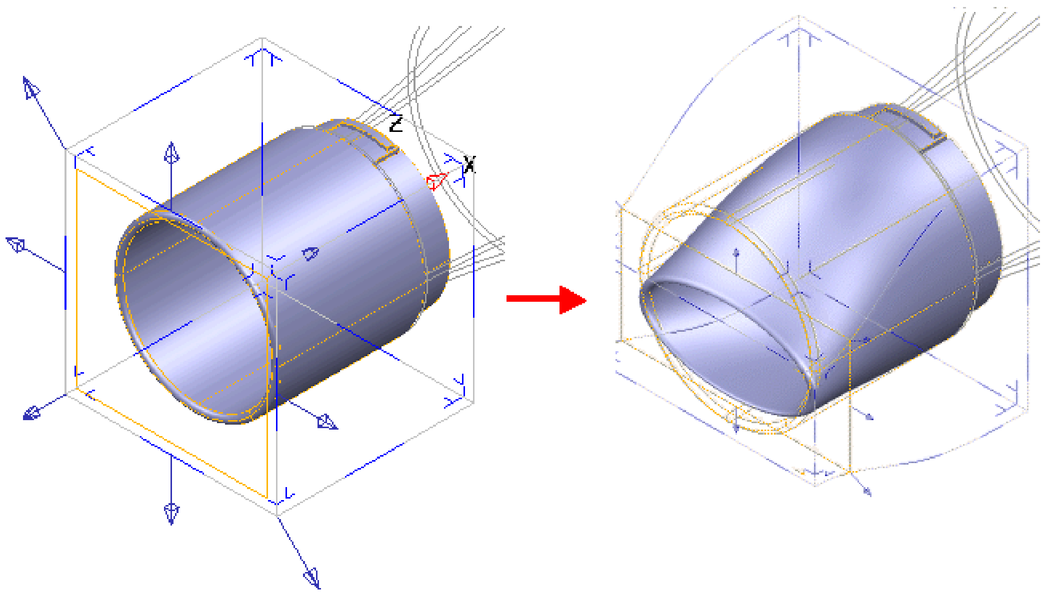
Flexible box morphing allows you to distort all or part of a solid model by manipulating a flexible box. The flexible box can also be used to bend or twist a model around an axis defined by the workplane.

When the **Flexible Box** page on the **Morphing** dialog is displayed, you select the faces of the solid that you want to distort.



The distortion buttons on the dialog let you edit the model dynamically by manipulating the drag handles on a box. As you move the drag handles, you distort the box. You can select different slices within the bounding box and morph the section using a combination of twist, bend, stretch and shear until you achieve the desired result.

The following example shows the change of shape that can be achieved by using the drag handles to manipulate the model.



For full details on all the options on this dialog, see Morphing dialog - Flexible box (see page 329)

To create a morph,

1 Select the solid to be morphed and click **Create a morph feature**  (*Solid Feature toolbar*).

2 Select the **Flexible box** tab on the **Morphing** dialog.

3 With **Select Faces** *ON*, select the faces of the solid that you want to distort. When you make your initial selection, all the faces of the solid are selected. You can modify this selection by clicking on the faces you would like to select. Use standard surface selection techniques to add and remove surfaces from the selection.

   The flexible box that is derived from your selection will be big enough to contain all the faces that you selected and will be aligned to the current workplane.

4 Enter the number of controlling slices in the **Number of slices** box. The number of slices is set to *3* by default.

5 **Bending or twisting around a known axis.** If you want to bend or twist your model around a known axis (defined by a workplane), select the **Around Workplane** option. The flexible box will appear.

   (a) Select the workplane and make sure that the principal plane you have selected is correct for the axis of the workplane you would like to use.

   (b) Enter the angle of twist or bend in the Angle box.

   (c) Use the buttons on the dialog to select the type of distortion.

6 **Distorting the box using the drag handles**. If you want to distort the box using the drag handles, select one of the buttons on the dialog. The flexible box instrumentation appears. Use the controls to create the required distortion

   There are a number of things to notice at this point:

   - One slice through the box is drawn in yellow. This is the **Active slice**.
   - The drag handles that will allow you to distort the box are drawn on the active slice.
   - Other slices in the box are hidden. The hidden slices are represented as blue sections of the outline of the box.
   - Slices that represent the faces of the box are drawn as the small 'L' shaped sections on the faces of the box.
   - You can change the active slice by clicking on this instrumentation.

- You can select more than one slice by holding down the **Shift** key and clicking on the instrumentation. The slices must all be in the same direction across the box. The last slice you click on will become the active slice.
- You can add new slices into the box by holding down the **Ctrl** key and dragging the active slice along the edge of the box
- If you previously selected **Around workplane**, you can turn this *OFF* and continue free form editing of the block.

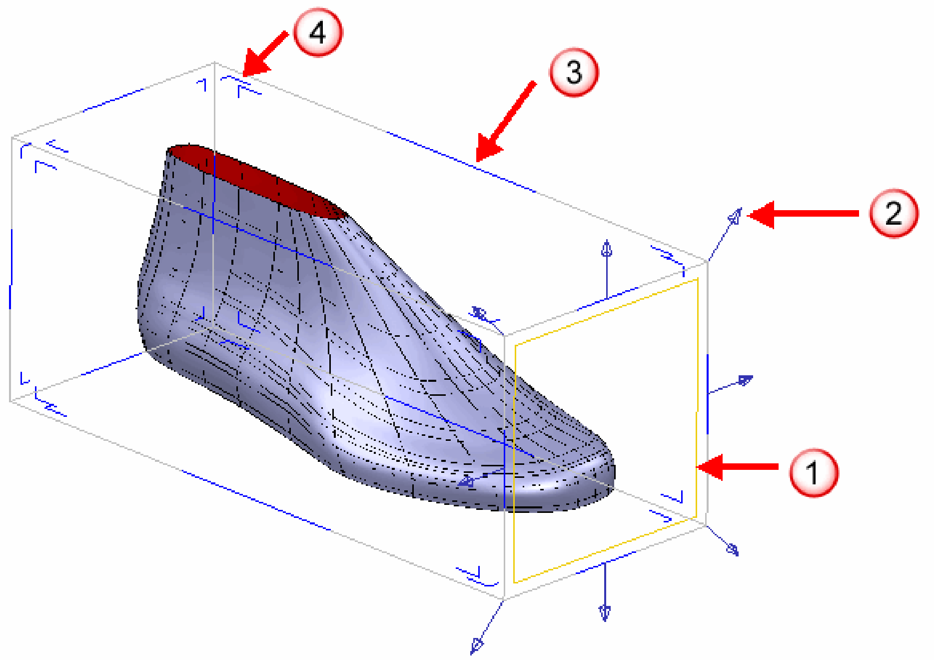
The numbers on the diagram below, correspond to the following features of the flexible box:

1 - Active slice.

2 - Drag handles for distorting the box. These are drawn on the active slice only.

3 - Hidden slice.

4 - For selection of slices at the edge of the box.



7 Select **OK** to save the changes you have made