PowerSHAPE 2015 R2

Reference Help

Menus and toolbars

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.

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Menus

The **Main** menu is located at the top of the main window. Use the options on this menu to control PowerSHAPE.

File Edit View Object Format Tools Macro Module Window Help
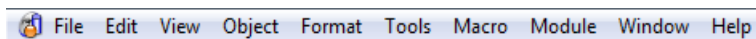
Use the following sections to find information on PowerSHAPE menus:

File (see page 7)

Edit (see page 22)

View (see page 112)

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File menu

The file menu enables you to access and manage the CAD model files. It also enables you to import and export files and exit the application.
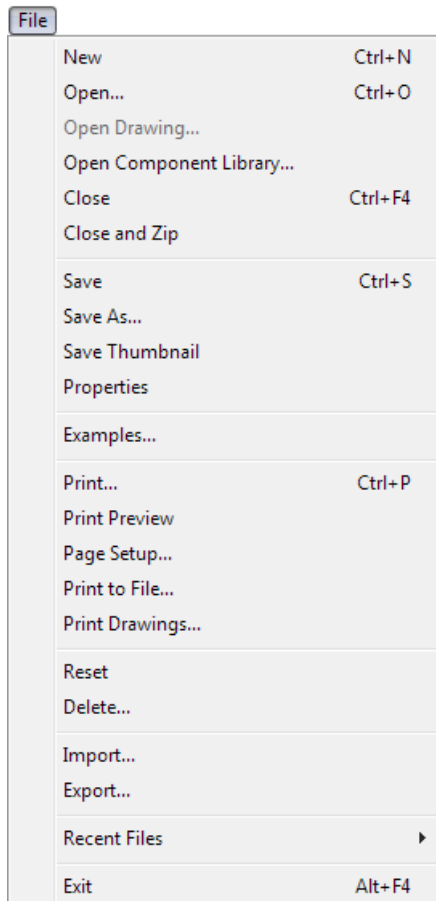
For further details, see:

New (see page 8)

Open (see page 8)

Open Drawing (see page 16)

Open Component Library

Close (see page 8)

Close and Zip (see page 8)

Save (see page 8)

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Delete (see page 20)

Import (see page 20)

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Recent Files (see page 21)

Exit (see page 21)

File > New

Use this option to creating a new empty model and open a window to display it.

File > Open

Use this option to choose the model to open. By default the file is opened in read/write mode. To ensure that the original file does not get overwritten, select **Open as read-only** on the **Open Model** dialog.

For further details, see Opening a model.

File > Open Drawing

Use this option to open drawings within a model.

File > Close

Use this option to close the active model.

File > Close and Zip

Use this option to close and zip a saved model.

File > Save

Use this option to save the active model.

*Occasionally there is a problem saving the model using **File > Save** and it is not possible to overwrite the existing version of the model. In this case, **Save** is disabled. Use **File > Save As** to save the current model. We recommend that you then use **File Doctor** to check the model.*

File > Save As

Use this option to save the current model with a new name.

File > Save Thumbnail

Use this option to save a thumbnail image of the active model. This enables you to preview the model without opening it.

File > Properties

Use this option to display the **File Properties** dialog. This shows how many objects, details of current levels, and levels of objects are contained in the selected model. You can also change the password of a model.

The **File Properties** dialog has the following tabs:

- **Details**
- **Properties**
- **Custom**

File > Examples

Use this option to open the **Run Macro** dialog.

If you want some data to experiment with in PowerSHAPE, you can import sample data as follows:

1 From the **File** menu, select **Examples** to open the following dialog:
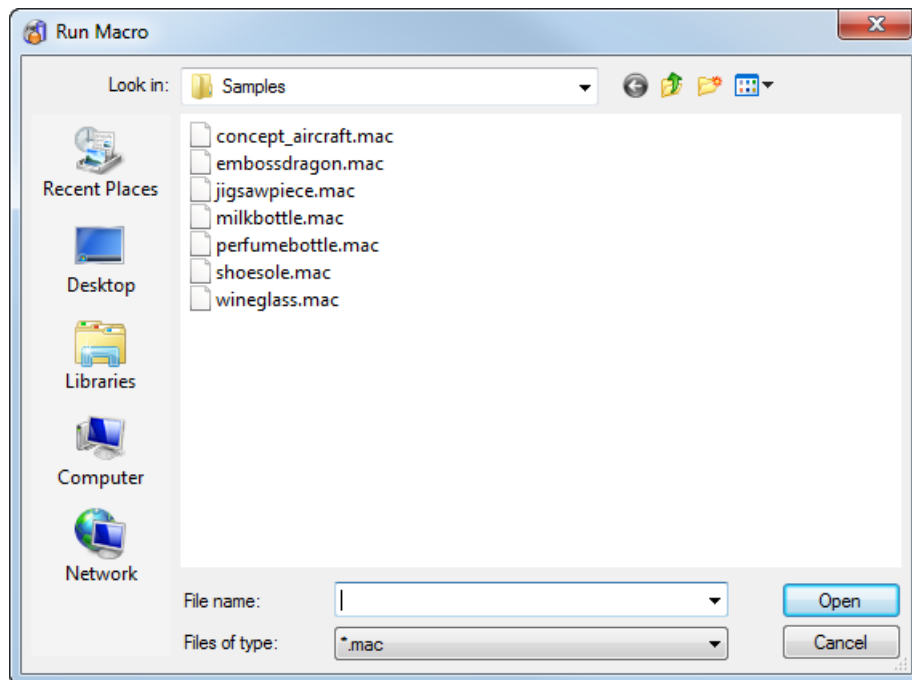
2 Select a file.

3 Click **Open** to import the data.

In some cases, a dialog is displayed to use with the data.

File > Print

This option enables you to print the contents of your graphics window. Selecting this option is the same as clicking the **Print** on the **Main** toolbar.

1 Select the graphics window.

2 Ensure your graphics window displays precisely what you want to print.

3 Select **File** > **Print** to display the **Print** dialog.
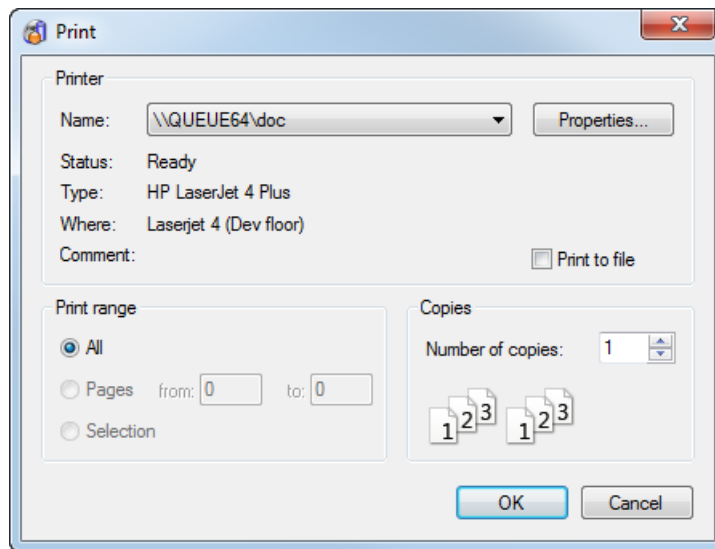
Use the **Print** dialog to print the contents of your graphics window. For further details, see Print dialog (see page 11).

Print dialog

Use the options on this dialog to print the contents of your graphics window.
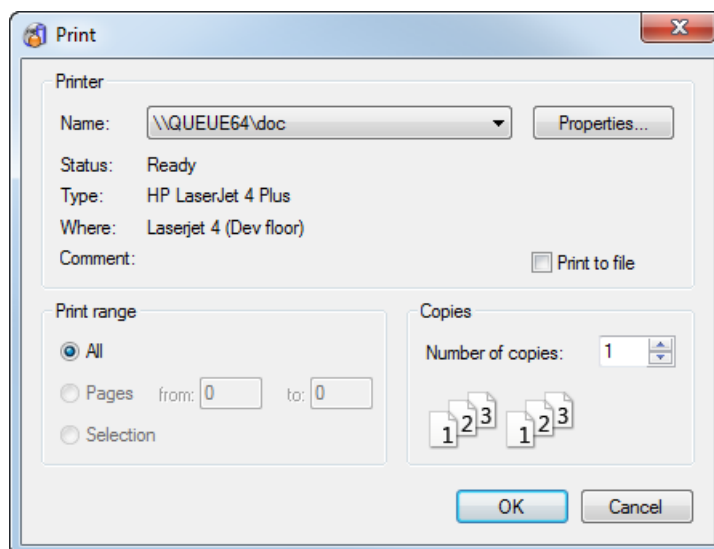
1 Select the **Name** of the printer installed on your computer. If you have more than one printer installed, select the required printer from the drop-down list.

2 Click **Properties** to display a dialog with the current settings for your printer. Change the settings to adjust the paper type and paper orientation.

3 Select **Print to file** to print the image to a printer file.

4 Enter the number of **Copies**.

5 Select a **Print Range** option:

  ▪ **All** — Print all the pages.

  ▪ **Pages** — Print a range of pages. Enter the **from** and **to** page numbers.

  ▪ **Selection** — Print a highlighted section.

6 Click **OK** to send the image to the printer.

7 If you selected **Print to file**, a dialog is displayed. Enter a filename for the image.

Printing high resolution images

PowerSHAPE uses two methods for printing windows: one for views containing only wireframe and the other for shaded images.

If the view contains only wireframe, printing is done at the full resolution of the printer.

If the view contains colour shading, OpenGL is used to render it into a bitmap and the bitmap is sent to the printer. Making a bitmap at the full resolution of the printer can be impractical as it could be very large. Instead, PowerSHAPE makes a bitmap which has the same aspect ratio as the printer but is limited in resolution so that its size doesn't exceed a limit.

The limit is set by a resource in the following file:

powershapexxxx\sys\misc\powershape.con

The relevant section of code for setting the size is:

> \#
>
> **# Printing colour shading.**
>
> \#
>
> **# This resource controls the size (in Mb) of the intermediate bitmap**
>
> **# which is used to print colour shaded images. Increasing its value**
>
> **# increases the printed resolution at the cost of temporary memory**
>
> **# usage and file size. Decreasing it reduces print resolution and**
>
> **# memory and file size requirements. 15Mb has been found to be a good**
>
> **# compromise on plain A4 paper in a typical inkjet printer.**
>
> **# print_opengl_bitmap_size: 15**

To set the size

1 Open the resource file:

powershapexxxx\sys\misc\powershape.con

2 Locate the relevant section of code (shown above)

3 Remove the # from the command line so that it reads

print_opengl_bitmap_size: 15

4 Save the resource file.

You can experiment with larger values for this resource. Each pixel of the image requires 3 bytes, so if you want a print resolution of X by Y dots, then set this resource to:

X * Y * 3 / 1,000,000

File > Print Preview

This option displays a preview of the printed image.
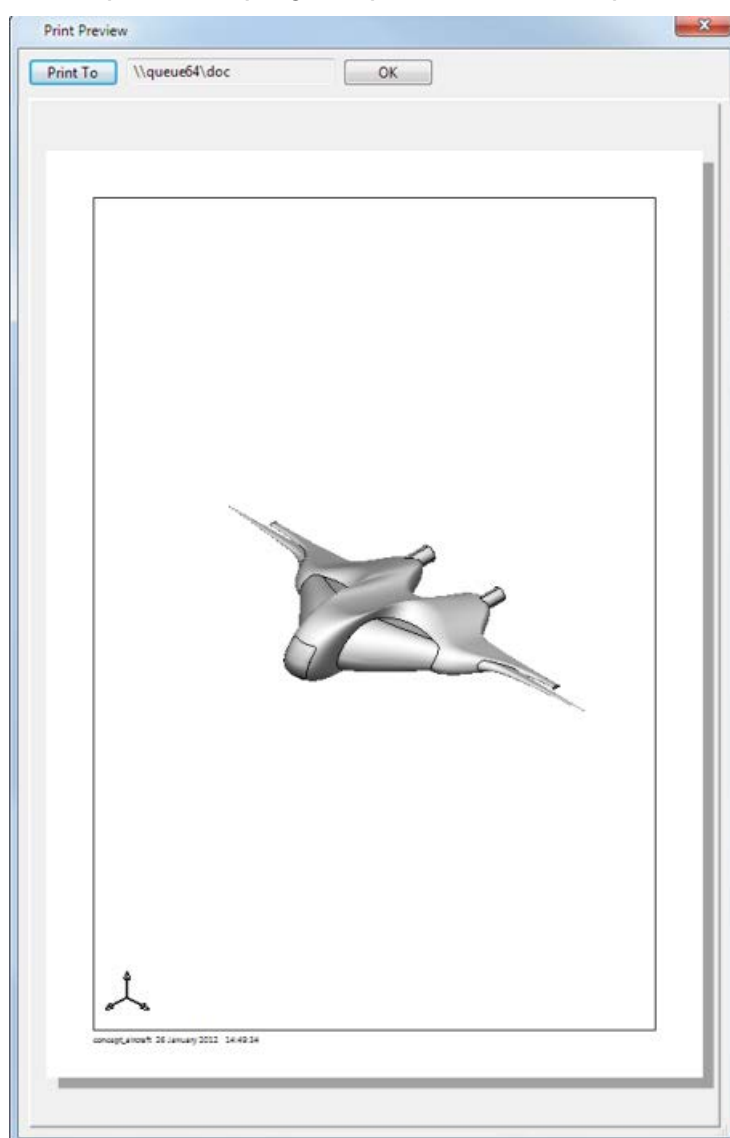
Print To — Click this button to send the image to the named printer and close the dialog.

OK — Click this button to close the dialog.

File > Print drawings

This option enables you to print your drawings using the **Print Drawings** dialog.

To print drawings:

1 Select **Print Drawings** from the **File** menu. This option is unavailable if your model contains no drawings.
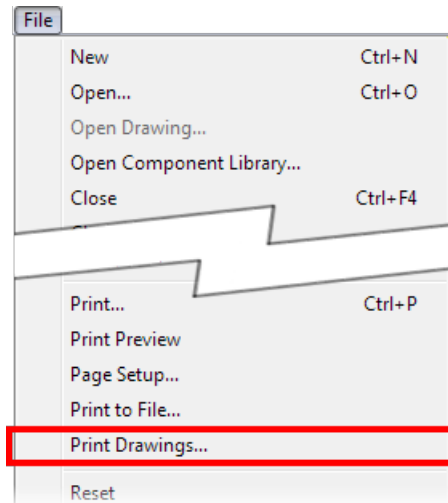
The **Print Drawings** dialog is displayed.
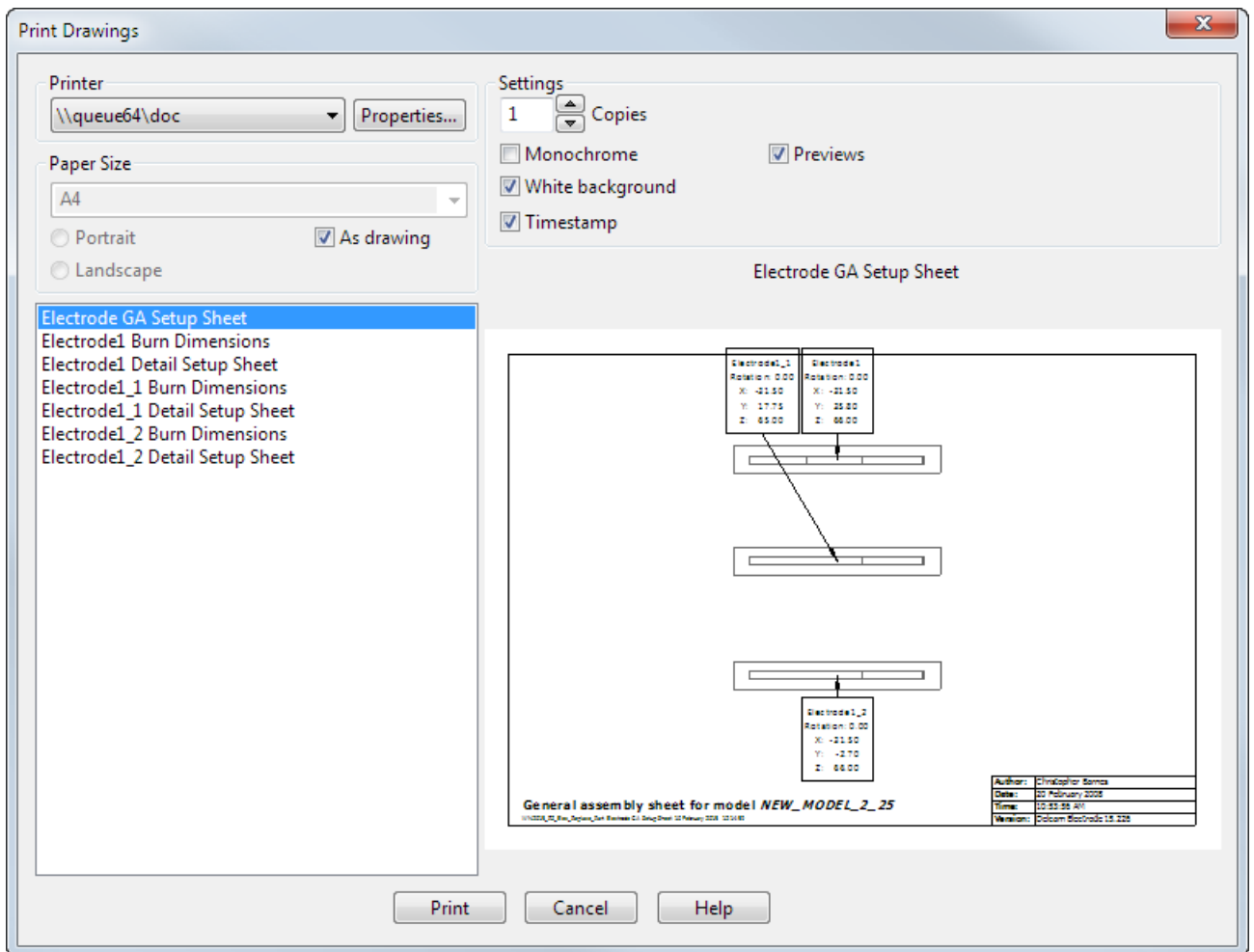
2   Select the drawing you want to print from the list. The drawing is displayed in the dialog.

3   Use the options on the dialog to modify your print settings:

- **Printer** — Select the printer location from the drop-down list.

- **Properties** — Click this button to display the properties dialog for the selected printer.

- **Paper Size** — Select the size of the drawing you want to print from the drop-down list and specify its orientation by selecting Portrait or Landscape. If **As drawing** is selected these options are unavailable.

- **As drawing** — Select this option to print the drawing to its dimensions specified in its template.

- **Copies** — Select the number of copies you want to print.

- **Monochrome** — Select this option to print your drawings in black, on a white background.

- **White background** — Select this option to print the drawing with a white background, independent of your background colour defined in PowerSHAPE. This option is unavailable if Monochrome is selected.

- **Timestamp** — Select this option to print the drawing with a timestamp.

4 Click **Print** to close the dialog and print the drawing.

File > Page Setup

This option enables you to adjust the print settings.
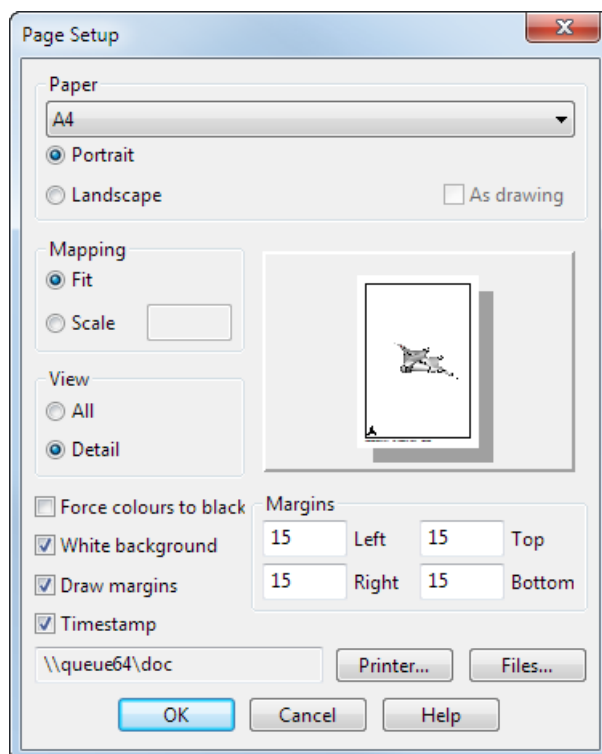
The default settings on this dialog depend on the mode you have selected from the **Module** menu.

Paper

These options enable you to choose the size of the paper.

Portrait — Orientate the image on the paper so that the height of the paper is greater than its width.

Landscape — Orientate the image on the paper so that the width of the paper is greater than its height.

As drawing — This option is selected by default if you are in drafting mode. PowerSHAPE selects the best paper size to suit the drawing from those available on the selected printer. **As drawing** is not available in modelling mode.

A preview image is shown on the dialog.

> *If you click this image, the **Print Preview** dialog (see page 13) is displayed showing this image. (see page 13)*

Mapping

These options fit your image onto the paper.

Fit — Scale the image to the size of the paper chosen.

Scale — Scale the image to fit a specified size.

> *Your image may not fit the paper. In drafting mode, if the drawing is smaller than the paper, it is not scaled up.*

View

These options determine what is printed from the model or drawing window.

Detail — Print only the visible part of the screen.

All — Print the whole model or drawing.

In drafting mode, the default is **All**, which maps the whole drawing onto the paper. In modelling mode, the default is **Detail**, which maps only the visible part of the model in the graphics window onto the paper.

Margins

This gives the values to offset the margins from the edge of the paper.

Force colours to black — Print all coloured objects as black.

White background — Print the background as white. White objects are printed as black.

Draw margins — Draw a border around the image using the **Margins** values.

Timestamp — Add the following to the image:

- Name of the model.

- Name of the drawing, if you are printing a drawing.

- Date and time the model was printed.

Printer — Open the **Print Setup** dialog, which you can use to change the current printer settings in PowerSHAPE.

Files... — Open the **Print to File Settings** (see page 19) dialog. This is used to set the image size.

OK — Save any changes and close the dialog.

Cancel — Discard changes and close the dialog.

File > Export

Use this option to display the Export settings dialog where you can choose to save models in file formats other than the PowerSHAPE model format. For further details, see Exporting data from a model.

File > Print to File

Use this option to print the contents of your graphics window to the following file types:

- jpeg (.jpg)

- png (.png)

- tiff (.tif)

- bitmap (.bmp)

- SGI rgb (.rgb)

- DUCT image (.img)

- neutral plot file (.npf)

To print to a file:

1 Select the graphics window. Make sure your graphics window displays precisely what you want to print.

2 Select **File > Print to File** to display the **Print to File** dialog.
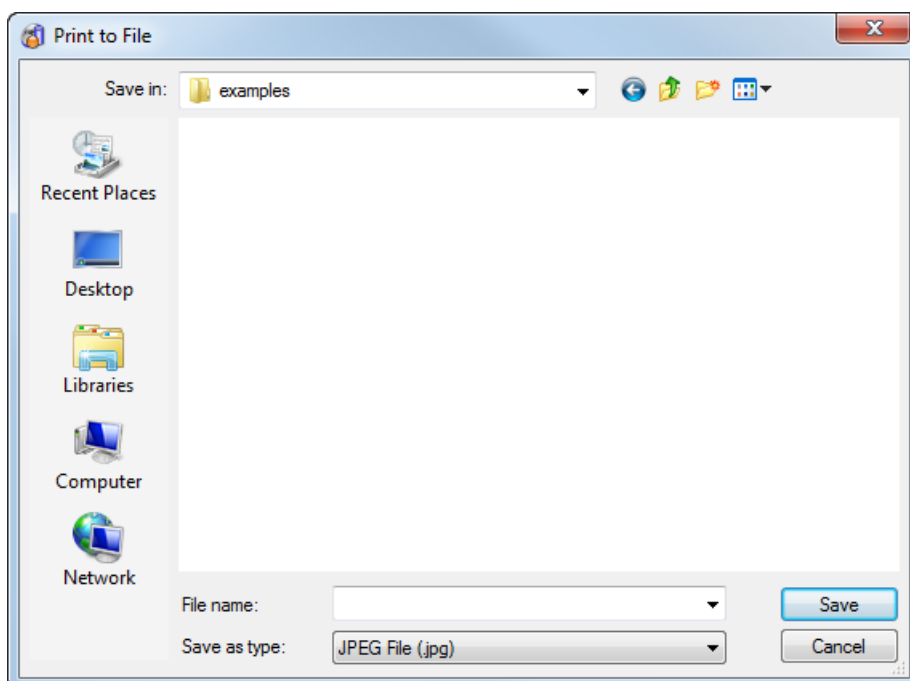
3 Use the **Print to File** dialog to select save location, the file type.

4 Click **Save** to save the file.

You can set options when printing an image to a file on the Print to File Settings dialog.

Print to File Settings dialog

Use this dialog to set the options to be used when printing an image to a file. To print to a file, you must select the **Print to File** option on the Print dialog (see page 11).
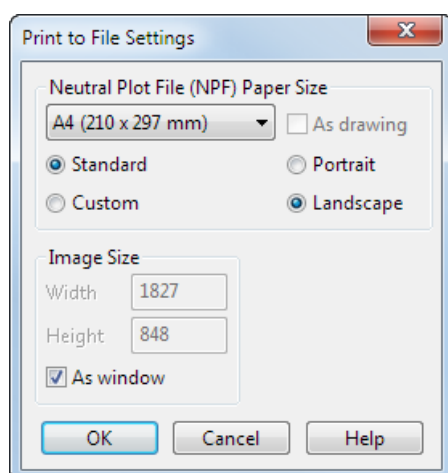
1 Use the **Neutral Plot File (NPF) Paper Size** options to set the size and orientation of the paper used by the neutral plot file:

- Select **Standard** and then select a standard paper size from the drop-down list.

- Select **Custom** to define your own paper sizes. Two boxes are displayed instead of the drop-down list. Enter maximum width and height values. The contents of your graphics window are scaled to fit the sizes specified. It is possible that your image may not fit the paper.
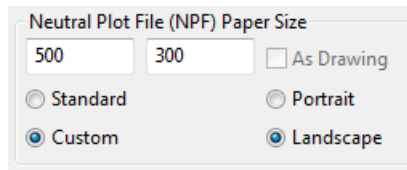
- If you are in drafting mode, then by default, **As Drawing** is selected and PowerSHAPE selects the best paper size to suit the drawing from those available on the selected printer. **As drawing** is not available in modelling mode.

2 Select the orientation of the image on the paper:

- **Portrait** orientates the image on the paper such that the height of the paper is greater than its width.

- **Landscape** orientates the image on the paper such that the width of the paper is greater than its height.

3 Use **Image Size** options to set the size of the image:

- Enter maximum **Width** and **Height** values (in terms of pixels) . The contents of your modelling window are scaled to fit the sizes specified. These options are only available if **As Window** is deselected

- Select **As Window** to set the size of the image from the size of the modelling window.

4 Click **OK** to save any changes made to the options on the dialog. The dialog is removed from the screen.

File > Reset

Use this option to remove all unsaved edits from the active model.

File > Delete

This is a powerful option, which enables you to permanently remove unwanted models.

File > Import

Use this option to display the Select a file to import dialog.

For further details, see Importing data into a model.

File > Recent Files

Recently opened models are listed in the **Recent Files** submenu at the bottom of the **File** menu. The read or write status when they were last opened is also shown.

To open a recently opened model

1   Select the **File > Recent Files**.

2   Select the name of the file to be opened. It is opened with the same read or write status as the last time it was opened.

If the model is not listed or you want a different read/write status, use the **File > Open** (see page 8) option.

Four recently opened models are listed by default. You can specify the quantity of recently opened models up to a maximum of 20 on the Models options dialog.

File > Exit

Use this option to close PowerSHAPE.

1   Save any changes that you want to keep.

2   Select **File > Exit**.

▪   If you have not made changes to the model, a dialog is displayed. Click **Yes** to close PowerSHAPE.

▪   If any open models contain unsaved changes, a dialog is displayed.

3   To save your work, click **Yes** and the dialog closes. You must now save any models and reselect **Exit** from the **File** menu. If you do not want to save your work, click **No** and wait for the software to shut down. Any unsaved edits are lost.

Edit menu

The **Edit** menu contains options that modify aspects of the active model.

Undo	Ctrl+Z
Redo	
Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Paste Special	Ctrl+E
Paste Attributes	
Paste Style	
Paste Level	
Select	▶
Delete	
Convert	▶
Add to Active Workplane Group	
Remove from Workplane Group	
Modify...	
General Edits	▶
Surface and Curve Edits	▶
Solid Edits	▶
Active Dimensions	▶
Fillet Corner...	
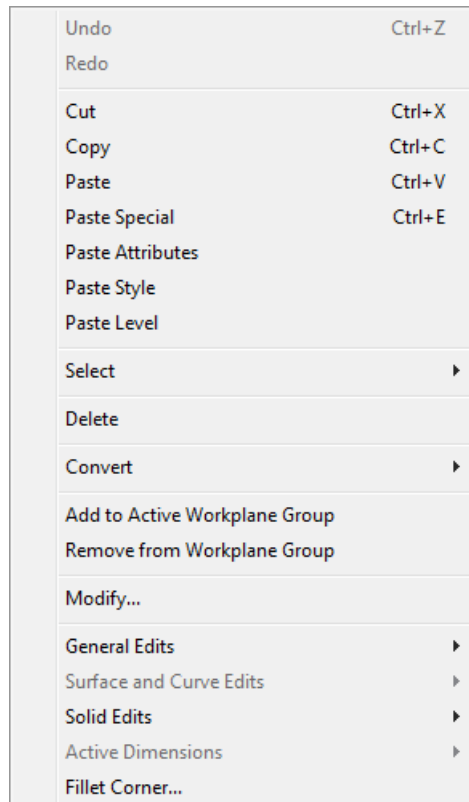
Many of these options are only effective on items that are selected. This is indicated by the select colour and possibly displaying edit handles. If no object is selected, some of the options are unavailable.

Edit > Undo

Use this option to reverse the last **Edit > Redo** command.

Alternatively, click **Undo** .

You can undo multiple commands, back to when you last saved the model. To undo multiple commands, keep selecting **Undo**.

For dialog operations, **Undo** reverses the overall action of a dialog. It does not undo individual operations within the dialog.

*The **Undo** command does not undo changes made in the **Options** dialog.*

A set of last commands for each model opened are remembered; You can undo and redo commands in one model, then go to another model and undo and redo its commands too.

When creating **Bezier**, **G2**, and **Spline** curves, **Undo** undoes individual curve points. You can undo curve points back to the first point created, but once the curve is completed, **Undo** undoes the whole curve.

*To undo all of your changes since you last saved the model, use the **File > Reset** command. If you are working on a new, unsaved model, the **File > Reset** command removes the entire model.*

Edit > Redo

Use this option to reverse the last **Edit > Undo** command.

Alternatively, click **Redo**   .

Edit > Cut

Use this option to add the clipboard contents into the current model. You can paste as many times as you like, but old clipboard data is lost each time you make a new **Cut** or **Copy**.

1   Select some objects.

2   On the **Main** toolbar (see page 196), click the **Cut**   button.

   A copy of the selected objects is put onto the clipboard, overwriting anything already there. The original objects are deleted from the model.

3   Paste the clipboard contents onto another model or the same model. The objects remain on the clipboard until you make another **Cut** or **Copy**.

The memory area used by **Cut** (and **Copy**) to hold copied data is called the **clipboard**. It enables you to easily re-use objects in the current model or in other models.

Objects on the clipboard are not permanent. Each cut or copy erases any data that is already on the clipboard. The data is also lost when you close the program or when the model you copied or cut from is closed or reset. However, other editing tools which create copies of objects do not use the clipboard, so they do not overwrite data you have previously placed there.

Edit > Copy

Use this option to copy selected objects to the **clipboard**.

1 Select one or more objects.

2 Click (*Main toolbar*).

The selected objects are copied to the clipboard, overwriting anything that is already there. The originals remain in the model.

You can now **Paste** the clipboard contents onto another model or the same model.

The objects will remain on the clipboard until you make another **Cut** or **Copy**.

The clipboard is used by **Copy** (and **Cut**) to hold copied data. This lets you re-use objects in the current model or in other models.

> *Objects on the clipboard are not permanent. Each cut or copy erases any data that is already on the clipboard. The data is also lost when the program shuts down or when the model you copied or cut from is closed or reset. However, other editing tools which create copies of objects do not use the clipboard, so they will not overwrite data you have previously placed there.*

To copy objects using Drag copy

1 Select one or more objects.

2 Use **Ctrl+drag** to move the selection to another position. The originals remain in place, and copies appear when you release the mouse button.

3 Confirm the action in the **Confirm Copy** dialog. Ths dialog also displays the distance (in terms of coordinates) of the copied objects from the original objects.
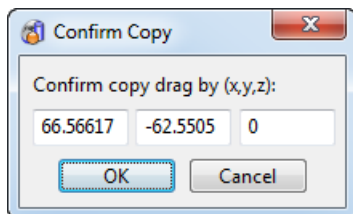
4 Click **OK** to confirm the copy drag.

> *These copies do not affect objects already on the clipboard.*

Edit > Paste

Use this option to add the clipboard contents into the current model.

1 **Cut** or **Copy** at least one object.

2 Click (*Main toolbar*).

You can paste as many times as you like, but the old clipboard data is lost each time you make a new **Cut** or **Copy**.

The pasted objects are inserted at the exact X, Y, Z co-ordinates of the originals and automatically selected. This is most convenient when pasting between models. However when pasting copies into the same model, it can appear that nothing has happened because the copies fall at exactly the same coordinates as the originals.

*Pasting objects into the same model is easier if you use **Paste Special** or drag copies using the mouse.*

Edit > Paste Special

This option is similar to **Edit > Paste**, except that pasted objects are attached to the cursor and pasted when you click.

Paste Special is particularly useful on complex models where you can easily lose sight of the pasted objects.

To paste objects using Paste Special

1   **Cut** or **Copy** at least one object.

An anchor point is used to position the pasted objects. PowerSHAPE chooses an anchor point when you select the objects.

2   To change the anchor point, hold down **Ctrl+Alt** and click a position.

3   Select **Paste Special** from the **Edit** menu.

When you move the cursor over the current graphics window, the objects appear attached to the cursor at their anchor point.

4   Position the cursor where you want the objects to fall and left-click.

When you left-click, the objects are pasted with their anchor point at the click position.
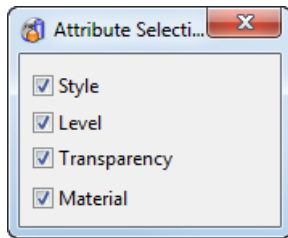
Objects remain selected after pasting.

Edit > Paste Attributes

This option pastes the attributes of a selected object to another object. You can paste the **Style**, **Level** and **Material** attributes.

1   Select the source object.

2   Click (*Main toolbar*).

The **Attribute Selection** dialog is displayed.

3 Select the objects to which you want to paste the selected attributes. The attributes are pasted automatically.

> *If you want to paste attributes onto multiple objects, select them using a drag box.*

Edit > Paste Style

Use this option to assign the style of an object to a group of objects.

1 Select an object.

2 From the **Edit** menu, select **Cut** or **Copy**.

The selected object is now on the clipboard.

3 Select a group of objects.

4 From the **Edit** menu, select **Paste Style**.

The selected group of objects are assigned the style of first object.

# Edit > Paste Level

Use this option to put a group of objects on the same level as the object on the clipboard.

1 Select an object.

2 From the **Edit** menu, select **Cut** or **Copy**.

The selected object is now on the clipboard.

3 Select a group of objects.

4 From the **Edit** menu, select **Paste Level**.

The group of objects are put on the same level as the first object.

Edit > Select

Use the options on the submenu to select objects.
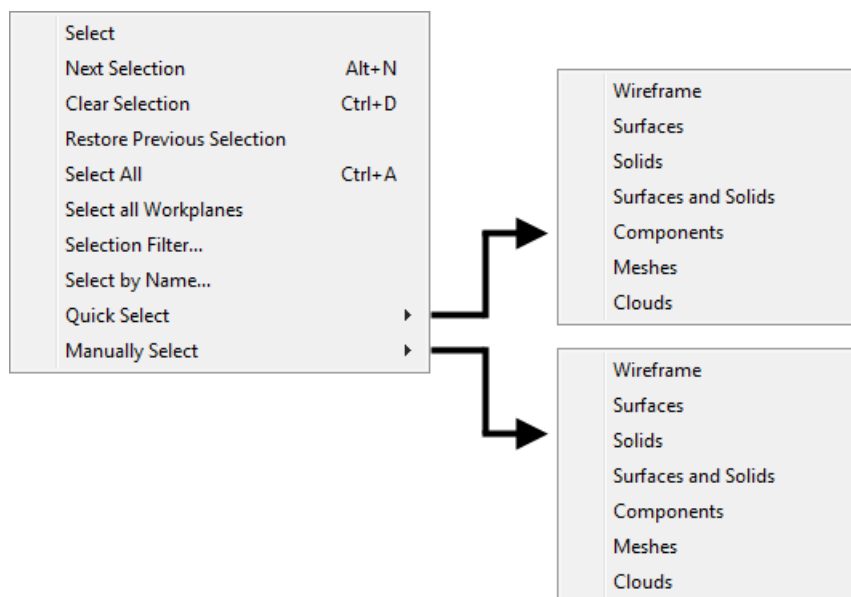
The following sections give details about the **Selection** options:

- Select (see page 27)
- Next Selection (see page 27)
- Clear selection (see page 28)
- Restore Previous Selection (see page 28)
- Select All (see page 28)
- Select All Workplanes (see page 28)
- Selection Filter (see page 28)
- Select by Name (see page 28)
- **Quick Select** is the same as using the options on the **Select** flyout (see page 193) on the toolbar to select objects.
- Use **Manually select** to select a particular type of object. For example, **Manually select > Wireframe** sets the selection filter to let you select just wireframe.

Edit > Select > Select

Use this option to enter normal selection mode.

For further details, see Selecting objects.

Edit > Select > Next Selection

Use this option to help you select the correct object when objects intersect.

1 Click on an intersection in a model.

2 Select **Edit > Select > Next Selection**. The next intersecting object created in the model is selected.

If **Show Next Selection** is selected on the **Mouse Options** (see page 80) dialog, a dialog is displayed containing buttons for each selectable intersecting object in the model. Click the buttons to select the corresponding intersecting object.

Next Selection is also available if you:

- Right-click to display the context menu.
- Enter **Alt + N.**

Edit > Select > Clear selection

Use this option to clear the current selection.

This option is also available on the **View** context menu, which you can display by right-clicking in the graphics window.

Edit > Select > Restore Previous Selection

Use this option to replace the current selection of items with the previous selection.

Edit > Select > Select All

Use this option to select all objects displayed on the model.

This option is also available on the **View** context menu, which you can display by right-clicking in the graphics window.

Edit > Select > Select All Workplanes

Use this option to select all the workplanes displayed in the model.

Edit > Select > Selection Filter

Use this option to display the **Selection Filter** dialog.

Edit > Select > Select by Name

Use this option to display the **Add Item to Selection** dialog.

Edit > Delete

Use this option to permanently remove selected objects from the model. You can also delete objects using the delete button on the toolbar.
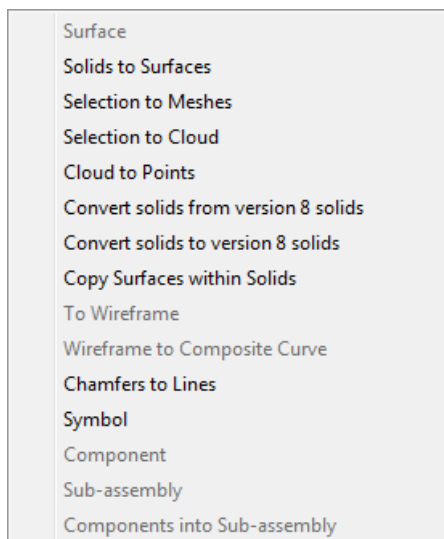
To delete objects using the delete button:

1 Select some objects.

2 Click (*Main toolbar*).

The objects are removed from the active model. If you want them back, select **Undo** , otherwise the objects will be lost permanently.

Edit > Convert

Use the options on the submenu to convert objects.

The following sections give details about the **Convert** options:

Surface (see page 30)

Solids to Surfaces (see page 30)

Selection to Meshes (see page 31)

Selection to Cloud (see page 31)

Cloud to Points (see page 31)

Convert solids from version 8 solids (see page 32)

Convert solids to version 8 solids (see page 32)

Copy Surfaces within solids (see page 33)

To Wireframe (see page 33)

Wireframe to Composite Curve (see page 33)

Chamfers to Lines (see page 33)

Symbol (see page 33)

Component (see page 34)

Sub-assembly (see page 34)

Components into Sub-assembly (see page 34)

Edit > Convert > Surface

This options enables you to convert non-Power Surfaces to Power Surfaces, for example, convert a NURBS surface to a Power Surface.

To convert a surface:

1 Select one or more surfaces.

2 Select **Edit > Convert > Surface.**

For further information see the following topics:

- Converting between surface types

- Primitive surfaces

- Power Surface

- NURBS surfaces

- BCP surfaces

- Editing Power Surfaces

Edit > Convert > Solids to surfaces

1 Select the solid to be converted.

2 Select **Edit > Convert > Solids to Surfaces** to convert a solid into the group of surfaces that define it.

Edit > Convert > Selection to Meshes

1 Select the surfaces or solids to be converted.

2 Select **Edit > Convert > Selection to Meshes.**

Edit > Convert > Selection to Cloud

1 Select the group of points or a mesh to be converted.

2 Select **Edit > Convert > Selection to Cloud** to convert the selection to a point cloud.
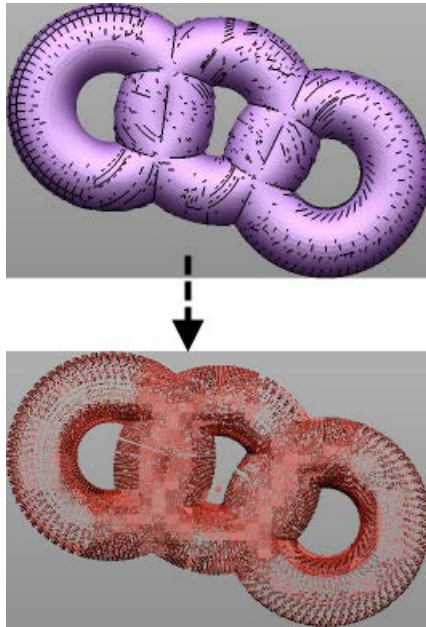
Use **Edit > Convert > Selection to Cloud** to convert:

- all selected lines, arcs, curves and composite curves to a single point cloud. The cloud has no normal or colour information.

- all points in the selection to a single cloud. This cloud is separate from the cloud that is created by converting lines, arcs, curves and composite curves.

- all meshes in the selection to individual point clouds; a cloud retains the normal and colour information from the mesh.
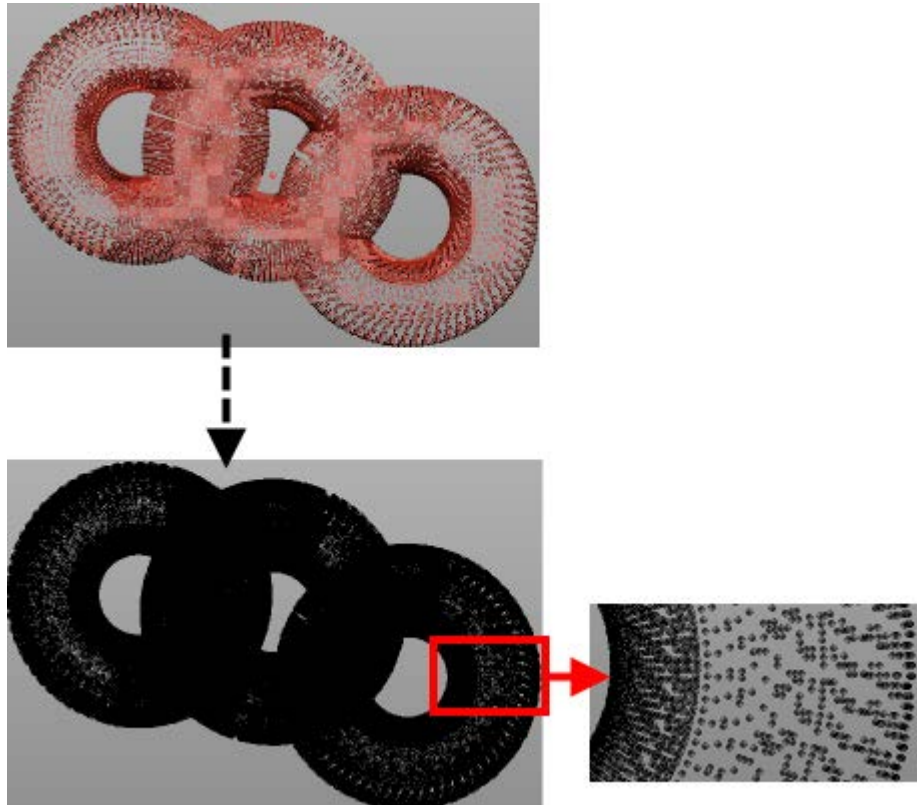
*You can cancel converting a selection to a cloud by pressing the **Esc** key on the keyboard, or by clicking the **Stop** ⊗ button on the progress bar.*

## Edit > Convert > Cloud to Points

1 Select a point cloud.

2 Select **Edit > Convert > Cloud to Points** to explode the selected cloud into points.

Edit > Convert > Solids from version 8 solids

Use this option to convert one or more selected version 8 solid to parasolids. This is the same as clicking ![icon] (*Solid Edit toolbar*).

Right-click on the converted solid to display the **Solid** context menu for the converted solid
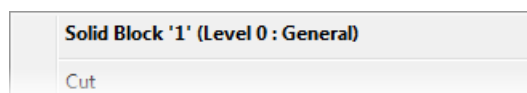
Edit > Convert > Solids to version 8 solids

Use this option to convert one or more selected post-version 8 solids to version 8 solids. This is the same as clicking ![icon] (*Solid Edit toolbar*).

Right-click on the converted solid to display the **Solid** context menu for the converted solid
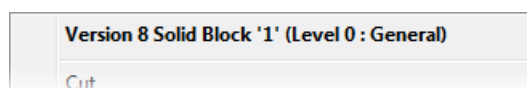
Edit > Convert > Copy Surfaces within solids

This option keeps a copy of the solid and creates a copy of all the surfaces.

Edit > Convert > To wireframe

Use this option to:

- convert composite curves into wireframe.
- create composite curves from surface curves (laterals, longitudinals and spine).
- create composite curves from trim boundaries.
- create curves from pcurves.
- convert drawing views into wireframe. For further details see Editing a view.
- convert text to wireframe.
- convert hatch lines to wireframe.

Convert to wireframe is only available if your selection contains an object that can be converted.

Edit > Convert > Wireframe to composite curve

Use this option to create composite curves from all the selected wireframe objects, attempting to connect adjoining pieces.

Edit > Convert > Chamfers to lines

This option converts chamfers (which contain dependency data to their associated objects) into simple lines at the same location. This conversion cannot be reversed.

To convert a chamfer to lines:

1 Select a chamfer or a group of objects which contain chamfers.

2 Select **Edit > Convert > Chamfers to Lines**.

The chamfers are converted.

3 Click **OK** to close the dialog.

The selected chamfers have now been converted into ordinary lines.

Edit > Convert > Symbol

Use this option to convert a symbol into one of the following:

- its individual objects (if **Keep items** was selected when its symbol definition was created).
- a series of lines. Links from the object definition to the symbol are lost.

Edit > Convert > Component

Use this option to convert a component into a solid.

Edit > Convert > Sub-assembly

Use this option to explode the currently selected sub-assemblies into their constituent parts.

For further details. see Converting a sub-assembly into its components.

Edit > Convert > Components into Sub-assembly

Use this option to convert the currently selected components into a sub-assembly.

For further details, see Converting components into a sub-assembly.

Edit > Add to active workplane group

Use this option to add objects to a grouped workplane.

For further details, see Grouped workplanes.

Edit > Remove from workplane group

Use this option to remove objects from a grouped workplane.

For further details, see Grouped workplanes.

Edit > Modify

Use this open to object-specific dialogs or toolbars so you can modify the selected objects. If no object is selected, this option is not available.

Modifying an object

1 Select an object.
2 Select **Edit > Modify**. The appropriate dialog or toolbar for the selected object appears.

You can also double-click on the object to display the appropriate dialog or toolbar.
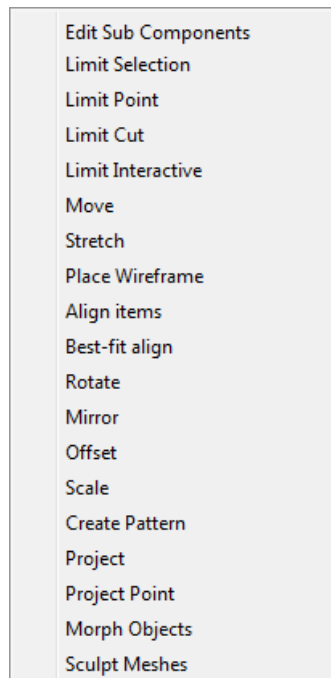
3 You can now edit the object.

For further details, see:

- Editing a workplane
- Editing a line
- Editing an arc
- Editing a curve
- Editing a Power Surface
- Editing a surface of revolution
- Editing an extruded surface
- Editing a primitive surface

Edit > General Edits

Use the options on this submenu to edit the model. The options correspond to the buttons on the **General Edits** toolbar (see page 204).

Edit Sub Components
Limit Selection
Limit Point
Limit Cut
Limit Interactive
Move
Stretch
Place Wireframe
Align items
Best-fit align
Rotate
Mirror
Offset
Scale
Create Pattern
Project
Project Point
Morph Objects
Sculpt Meshes

For further details see:

Edit sub components (see page 36)	Limit Selection (see page 37)
Limit Point (see page 52)	Limit Cut (see page 54)

Limit Interactive (see page 55)	Move (see page 56)
Stretch (see page 59)	Place Wireframe (see page 61)
Align items (see page 63)	Best-fit align (see page 63)
Rotate (see page 63)	Mirror (see page 69)
Offset (see page 73)	Scale (see page 78)
Create Pattern (see page 86)	Project (see page 103)
Project Point (see page 104)	Morph Objects (see page 105)
Sculpt Meshes (see page 107)	

Edit > General Edits > Edit Sub-components

This allows you to move, rotate, scale and offset sub-components of certain objects.

Sub-components can be:

- laterals and longitudinals on surfaces.
- points on curves (including composite curves and surface curves).

To edit the sub-components using the tools on the Edit toolbar:

1 Select the sub-components. The sub-component that is displayed on the toolbar will be determined by the type of sub-component you select.

2 From the **Edit** menu, select **General Edits** followed by **Edit Sub-Components**.

This is the same as selecting **Show general edits options** from the toolbar. The **General edits** toolbar is displayed:

3 From the **Edit** menu, select **General Edits** and then one of the following **Move**, **Rotate**, **Scale**, **Offset** and **Project**. These options are also available on the **Edit** toolbar.

The **Move**, **Scale**, **Offset** and **Project** commands work in exactly the same way as for other objects except:

- The **Keep Original** and **No. of copies** options are not available.

- If you move the rotate direction to a spine point, it will take the spine direction (laterals and longitudinals only).

- **Offset** does not allow continuous or discontinuous options.

- **Scale** does not allow scale to volume.

Edit > General Edits > Limit Selection

You can use this option to limit/trim the following:
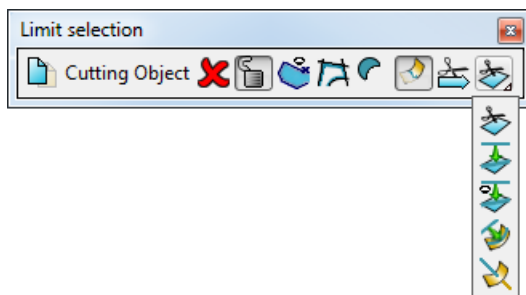
- wireframe objects to:

    - wireframe object (see page 41)

    - surface (see page 43)

    - mesh.

- surfaces to:
  - surface (see page 44)
  - mesh.
- solid to a mesh.
- mesh to:
  - mesh
  - wireframe
  - surface using the **Project** option. The wireframe gets projected through the mesh along the principal plane of the active workplane.
- clouds to:
  - mesh
  - surface (see page 51)
  - workplane
  - wireframe.
- group of surfaces and solids to a wireframe object (see page 46).
- objects to a plane (see page 49).

To limit objects:

Use the following technique to limit objects:

1 Select the cutter (see page 39).

2 Click the **General Edits** ⬚ flyout to display the **General Edits** toolbar.

3 Click the **Limit** ⬚ flyout (*General edits toolbar*).

4 Click **Limit selection** ⬚ (*Limit flyout*) to display the **Limit selection** toolbar (see page 206).

5 Select the appropriate options from the toolbar to limit the selected objects.

6 Select the object to limit.

The following sections give further information and examples on using this toolbar:

- Cutting objects (see page 39)
- Limiting wireframe objects to a wireframe object (see page 41)
- Limiting wireframe objects to a surface (see page 43)
- Limiting surfaces to a surface (see page 44)
- Limiting a group of surfaces and solids to a wireframe object (see page 46)
- Limiting objects to a plane (see page 49)
- Limiting a cloud to a surface (see page 51)

Cutting objects

Cutting objects are used to limit (extend or shorten) other objects. A cutting object may be:

- an object that is already part of the model.
- a composite curve or polygon that is created specifically as a cutting object.

Selecting a cutting object

Select a cutting object in one of the following ways:

- Select an object before selecting **Limit selection** from the toolbar.
- If no object is selected before selecting **Limit selection**, select one immediately after entering the command. To select a group of objects, you must use box selection.

As soon as the limit operation is completed, the cutting object becomes deselected unless it is locked.

> *To use the same cutting objects many times, click **Lock*** 🗐 *next to **Cutting Object** on the **Limit selection** toolbar.*

To change the cutting objects:

1 If the cutting objects are locked, click the **Lock** button 🔒 to unlock them.

2 Deselect the current cutting objects by either clicking the **Tick** button next to **Cutting object** or clicking in empty space in the graphics window.

3 Select a new cutting object. To select a group of objects, use box selection.

Creating cutting objects

Create a cutting object in one of the following ways:

- Using **Create composite curve** (*Limit selection toolbar*) to create a temporary composite curve. This composite curve becomes the cutting object and is automatically deleted when it is no longer required.

- Using **Limit to polygon** (*Limit selection toolbar*).

  1 Select a mesh.

  2 Select **Limit selection** (*Limit flyout, General Edits toolbar*) to display the **Limit selection** toolbar.

  3 Click (*Limit selection toolbar*).

  4 Click the start point and click additional points to create a polygon on the mesh. Click the start point again.
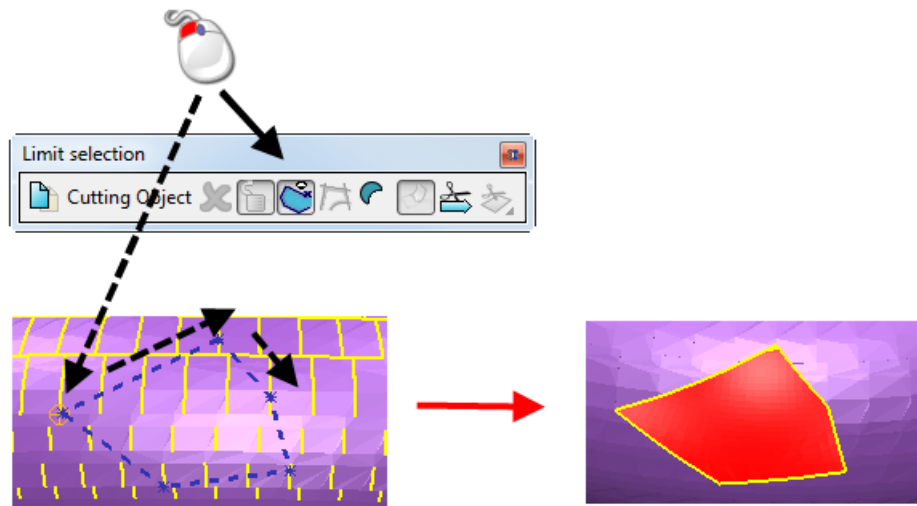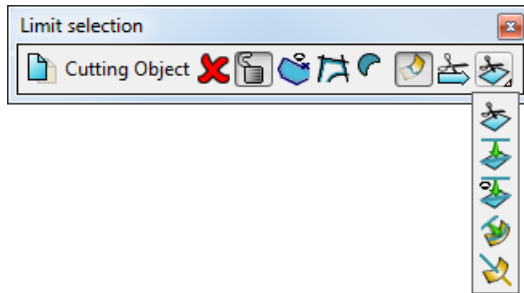

The polygon is used as the cutting tool.

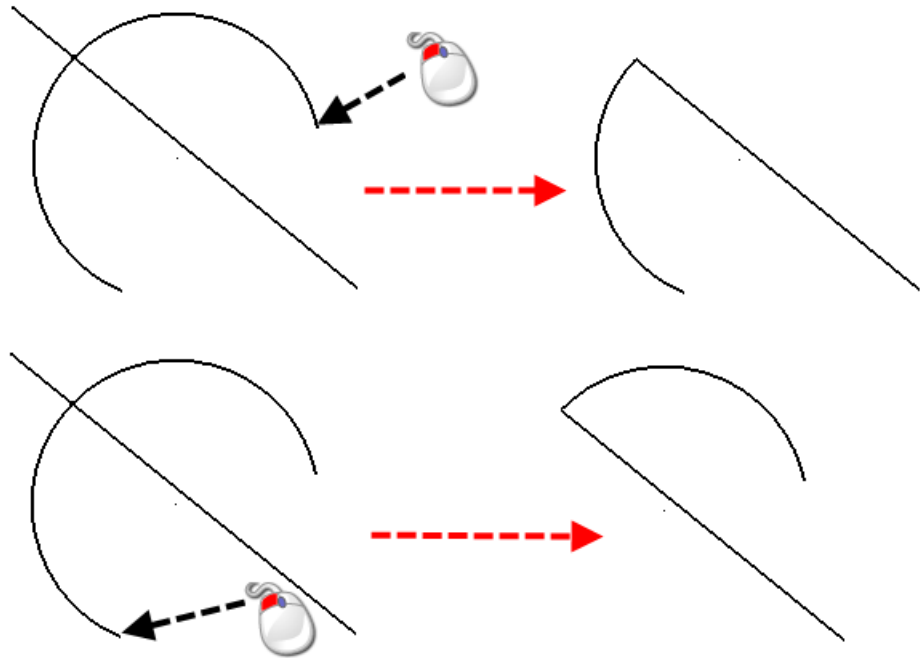## *Limiting wireframe objects to a wireframe object*

1 Select the wireframe object to use as the cutting object ①.
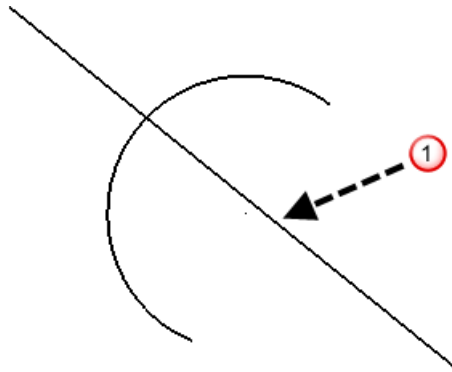
2 Click **Limit selection** [icon] to display the **Limit selection** toolbar.

3 To keep a copy of the wireframe to be limited, click [icon] to select **Keep original**. The button changes to [icon] to reflect the selection.

4 Click [icon].

5 Select the wireframe object you want to limit. The selected object automatically extends or trims back to the cutting object.

  ▪ Where you select the object determines how it is limited:

- You can limit multiple wireframe objects to the cutting object by box-selection. Click **Keep both** to select this option. The button changes to .
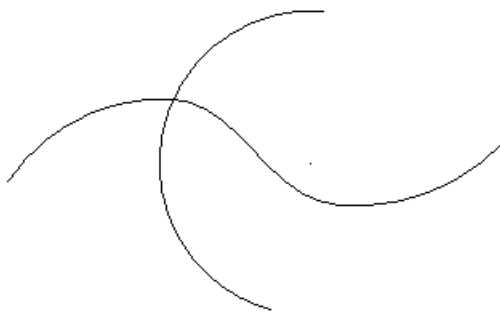
Limiting non-intersecting wireframe objects

You can use the **Limit selection** command to limit two pieces of wireframe even though they do not intersect:

1 Create two objects that do not intersect.

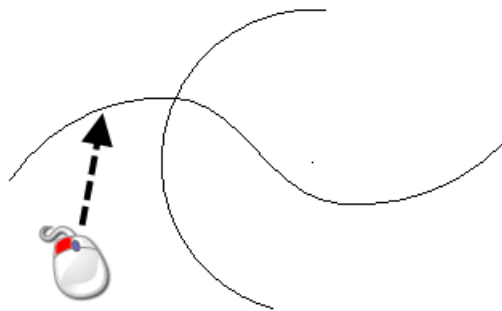**2** Rotate the view so that the wireframe objects appear to intersect.
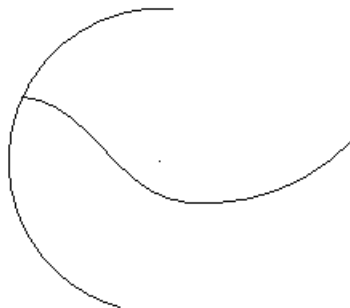
3 Click **Limit selection** to display the **Limit Selection** toolbar.

4 Select one of the objects as the cutting item.
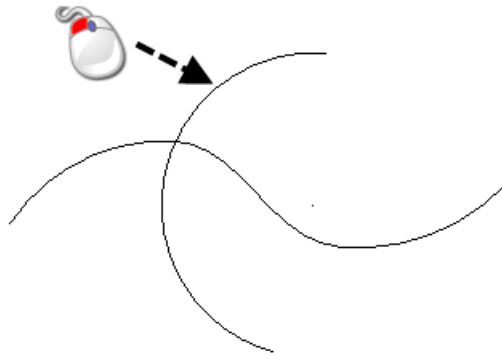
5 Select the other object to limit.

The second object is trimmed to the intersection in the view.

Limiting wireframe objects to a surface

1 Select the surface to use as the cutting object.
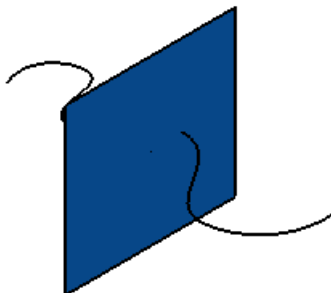
2 Click **Limit selection** [icon] to display the **Limit Selection** toolbar.
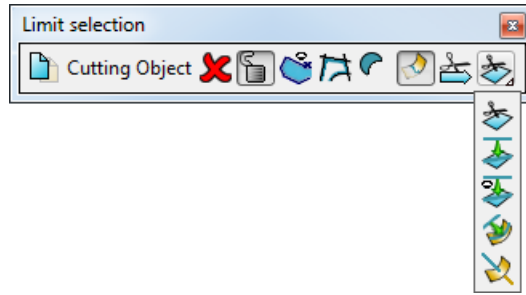
3 Click the **Intersect curve mode** button [icon] (*Limit selection toolbar*).

4 Select the wireframe object you want to limit.

The portion of the wireframe where you clicked is removed.
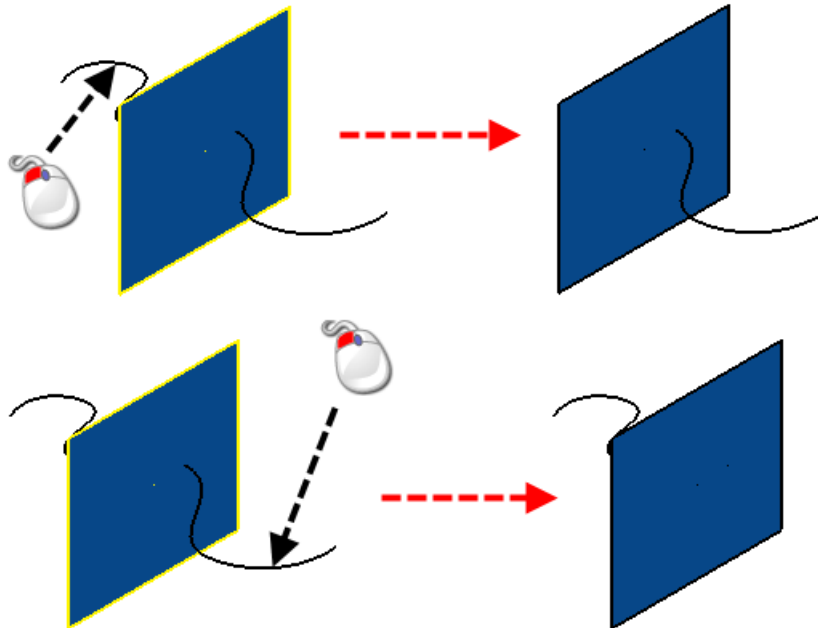
You can also box-select a group of wireframe objects.

Limiting surfaces to a surface

You can limit surfaces to the intersection of a cutting surface.

1 Select the cutting surface.
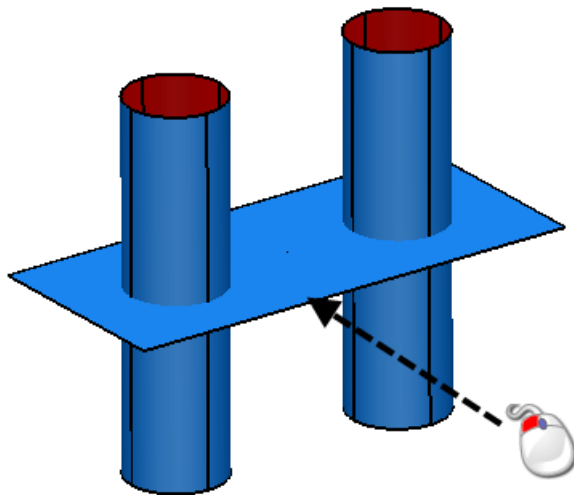
2 Click **Limit selection** ⬚.
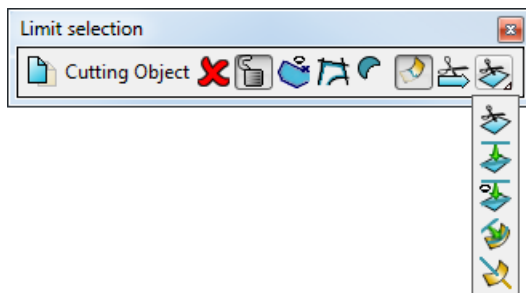
3 To keep a copy of the surfaces to be limited, select the **Keep original** ⬚ button. The button changes to ⬚.

4 Once limited, surfaces are cut into two portions. If you want both portions of the limited surfaces, click the **Keep both** button ⬚ to turn it *ON* ⬚.

5 If you do not want to trim the cutting surface to the other surfaces, the **Keep both** ⬚ button should be *OFF*.

6 Select the surfaces you want to limit. The surfaces automatically trim back to the cutting object.

If the top cylindrical surfaces are selected, the trim is as shown below:
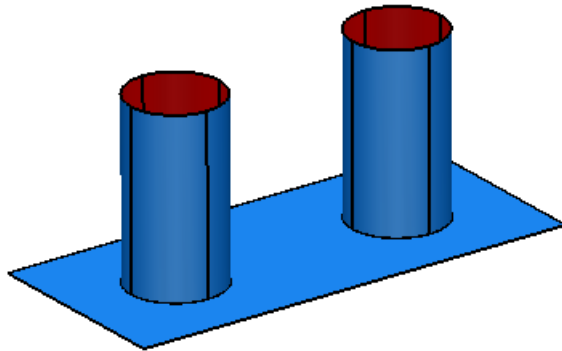
Limiting a group of surfaces and solids to a wireframe object

You can limit a group of surfaces and solids to a wireframe object. If the wireframe object does not lie on the surfaces and solids, the wireframe object is projected onto the surfaces and solids and they are limited to it.

There are two options:

- wireframe object lies on surfaces and solids (see page 46).
- wireframe object does not lie on surfaces and solids (see page 47).

Wireframe object lies on surfaces and solids

1   Select the wireframe item as the cutting object.
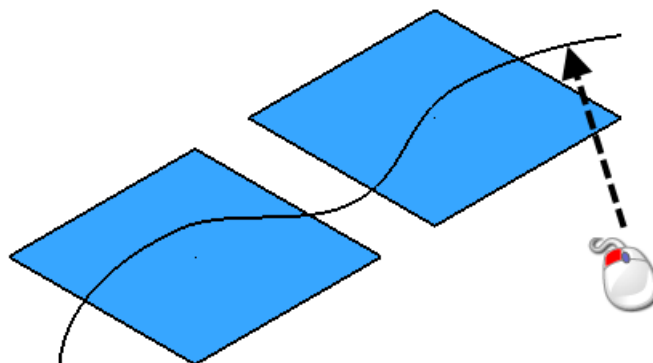
2   Click **Limit selection** .
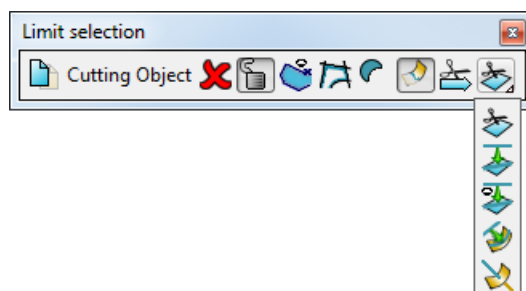
3 On the **Limit selection** toolbar, click **Limit mode** 🖈.

4 To keep a copy of the surfaces and solids to be limited, select the **Keep original** 🗋 button. The button changes to 🗋.

5 Once limited, surfaces and solids are cut into two portions. To keep both portions of the limited surfaces, click **Keep both** 🌙. The button changes to 🔵. Both pieces of the solids are always kept.

6 Select the surfaces and solids you want to limit. The surfaces and solids limit to the wireframe object, and trim boundaries are created along them.

Using the above example, box select the surfaces.

The surfaces trim as shown:
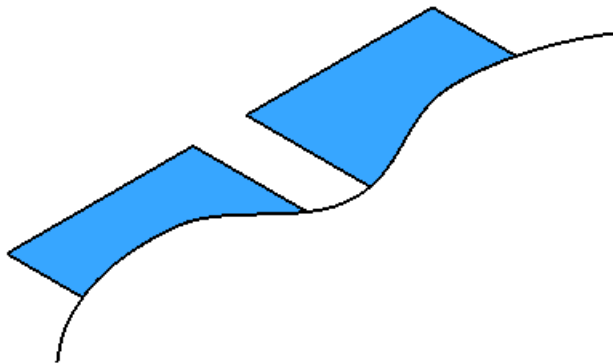
If the projected wireframe object does not meet the edges of the surfaces and solids, the trim boundaries are automatically extended to meet the edges.

7 If the limited surfaces are not as expected, click the **Next Solution** button 🖈 until the required solution is displayed.

> 💡 *Next Solution cycles through the range of solutions the trim boundaries can offer for your limited surfaces.*

Wireframe object does not lie on surfaces and solids

The wireframe object is projected onto the surfaces and solids and they are then trimmed.

You can project the wireframe onto the surfaces and solids along either:

- the axis normal to the principal plane; or
- the surface normal.

The methods for both are very similar, except in the first method you must make sure that one of the axes of the current workspace lies in a suitable direction to project the wireframe object. If it does not, add a workplane and orientate it to a suitable angle.

1 Select the wireframe object as the cutting object.

Consider the following objects.
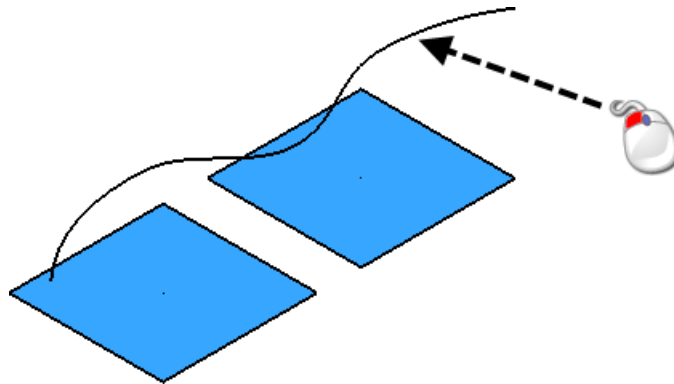
2 Click **Limit selection** to display the **Limit Selection** toolbar.
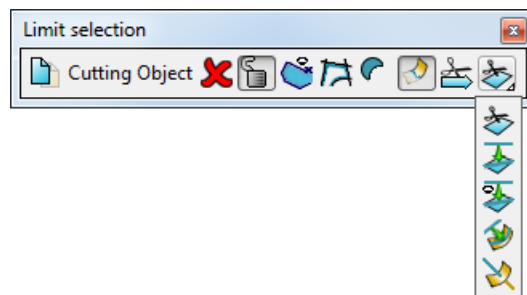
3 On the toolbar, click one of the projection options:

- **Project mode** projects the wireframe onto the surfaces and solids along the axis normal to the principal plane.

- **Project surface normal mode** projects the wireframe onto the surfaces and solids along the surface normal.

4 If you chose **Project mode** , change the principal plane so that the axis you want to project along is normal to the principal plane.

 The principal plane is changed using the selector in the Status bar.

5 To keep a copy of the surfaces and solids to be limited, select the **Keep original** button. The button changes to .

6 Once limited, surfaces and solids are cut into two portions. To keep both portions of the limited surfaces, click **Keep both** . The button changes to . Both pieces of the solids are always kept.

7 Select the surfaces and solids you want to limit.

The surfaces and solids limit to the wireframe object and trim boundaries are created along them.

Using the above example, box-select the surfaces.

The surfaces trim as shown:
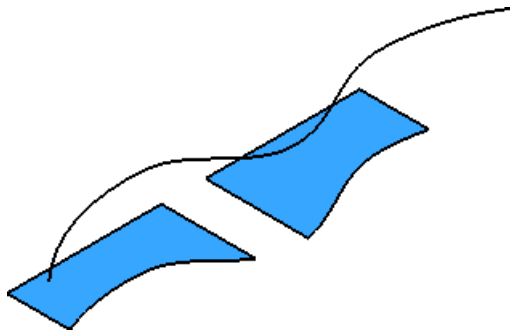
If the projected wireframe object does not meet the edges of the surfaces and solids, the trim boundaries are automatically created extending to meet the edges.

8 If the limited surfaces are not what you expected, click the **Next Solution** button until you see the required solution.

Next Solution cycles through the possible solutions given by the trim boundaries on your limited surfaces.

Limiting objects to a plane

You can use the principal plane of any workplane as a cutting object. The workplane does not have to be active.

1 Select any workplane as the cutting object.
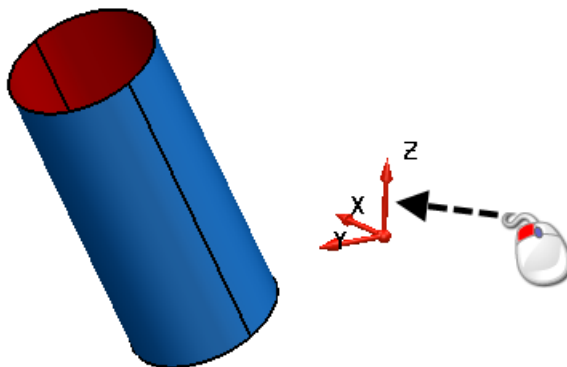
2 Click **Limit selection** [icon] to display the **Limit Selection** toolbar.
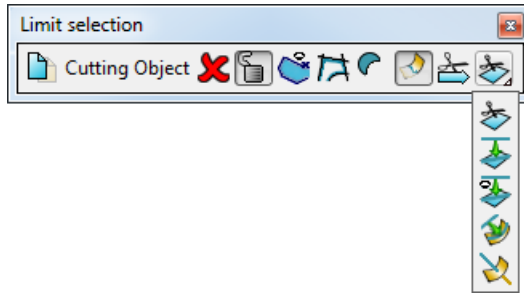
3 To keep a copy of the object to be limited, select the **Keep original** [icon] button. The button changes to [icon].

4 Once limited, surfaces are cut into two portions. To keep both portions of the limited surface, click **Keep both** [icon]. The button changes to [icon].

5 Change the principal plane, as required.

6 Select the objects you want to limit to the principal plane, which in this case is the XY plane.
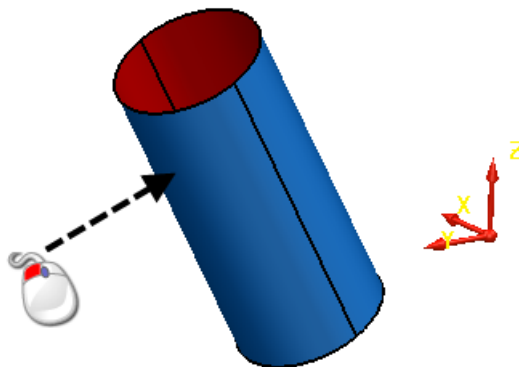
The object is trimmed to the principal plane.
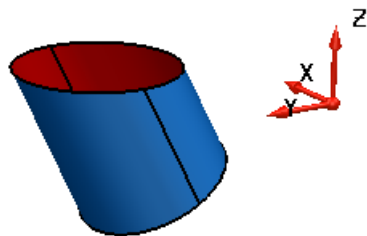
7 If the solution is not what you expected, click the **Next Solution** button [icon] until you see the required solution.

[note icon] *Next Solution cycles through the possible solutions.*

Limiting a cloud to a surface

Limit a cloud using a surface that cuts through the cloud.

1 Select the surface to use as the cutting object.
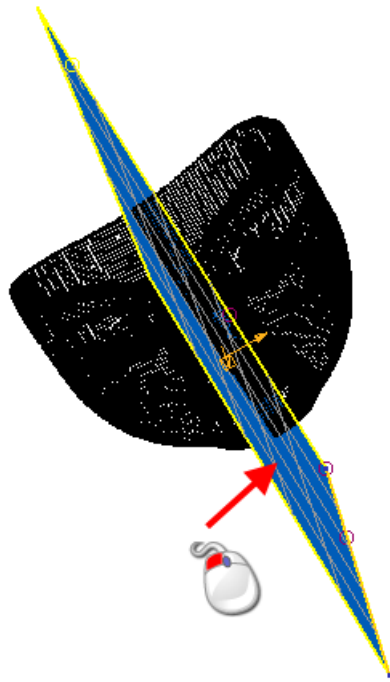
2 Click **Limit selection** to display the **Limit Selection** toolbar.
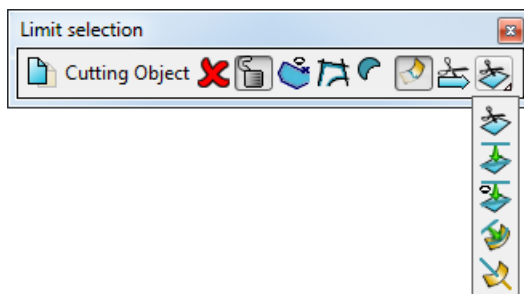
3 Click

4 Select the cloud to limit. The cloud is automatically trimmed back to the cutting object.
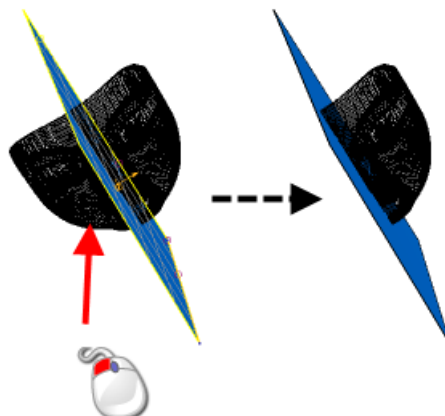
5 Click the **Next Solution** button ⬚ to see the alternative solution.
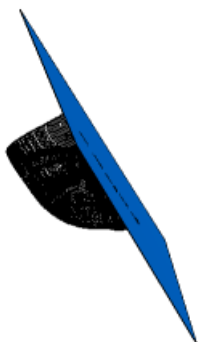
Edit > General Edits > Limit Point

Use this option to trim or extend objects to a point. The operation behaves slightly differently for various objects.

1 Select an object.

2 Click the **General Edits** ⬚ flyout to display the **General Edits** toolbar.

3 Click **Limit** ⬚ flyout (*General Edits toolbar*)

4 Click **Point limit** ⬚ (*Limit flyout*)

The cursor icon changes to a magnet and a circle is drawn on the end point or edge to be limited.
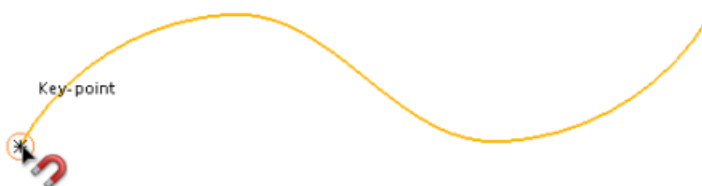
5 Do one of the following:

- Drag an end of the object to a new position. The end of the object extends or trims to where you release the mouse.

- Click a position on a wireframe object. The object trims back to that position from the nearest end key point.

- Click a curve on a surface. The surface trims back to that curve from its nearest end.

  If there are only two curves in a particular direction, then you can't trim in that direction unless you drag the end of the surface.

You can extend surfaces and curves either linearly or along their curvature.

Consider the following surface.
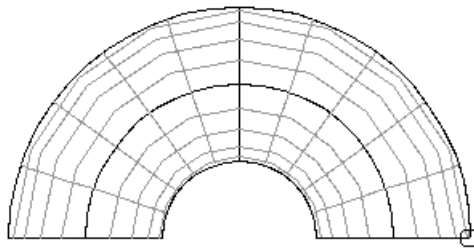
This surface extends linearly as follows:
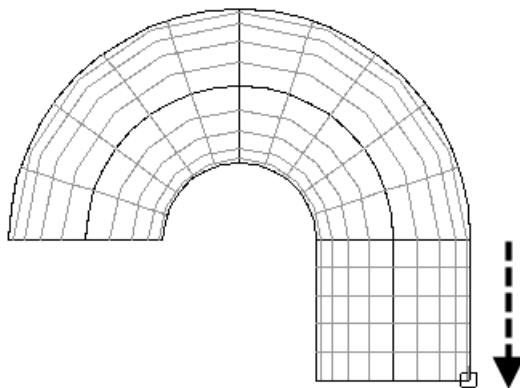
It extends along its curvature as shown below.
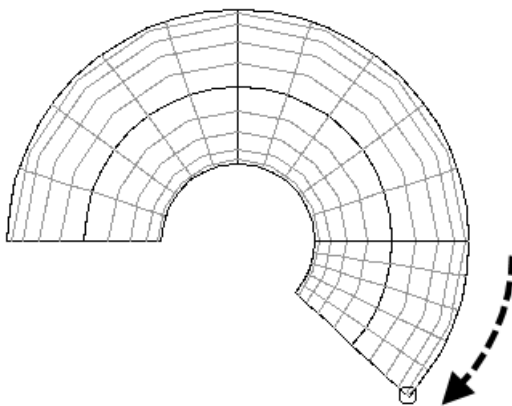
By default, surfaces and curves extend linearly. To change from linear to curvature mode, click the **Extend linearly** button to toggle to the **Extend along curvature** button .

The **Next Solution** button cycles through the possible solutions given by the trim boundaries on your limited surfaces.

- Enter a value in the **Distance** text box and press *enter* to limit the object.

  The edge or end point with the marker is limited.

  You can change the edge or end point using the **Edge/End Point** drop-down list.

Edit > General Edits > Limit Cut

You can cut lines, arcs and curves into smaller pieces as follows:

- an object at any position
- an object in a number of equal sized parts
- a hole into an object

To cut an object

1 Select an object.

2 Click the **General Edits** ⬚ flyout to display the **General Edits** toolbar.

3 Click **Limit** ⬚ flyout (*General Edits toolbar*)

4 Click **Cut object** ⬚ (*Limits flyout*).

> *If you have not selected the correct object, you can change the selection while* **Limit Cut** *toolbar is displayed.*

5 Do one of the following:

- **To cut the object at any position,**

   Click where you want to cut the object. If you cut a circle or a closed curve, it is cut into two pieces from its start point to its cutting point.

- **To cut the objects into equal sized parts,**

   Enter a value in **No. of pieces** box, or select a number from the drop-down list. The object is cut equally into the required number of parts.

- **To cut a hole in the object,**

   Left-click on the selected object at the point where you want to create the hole.
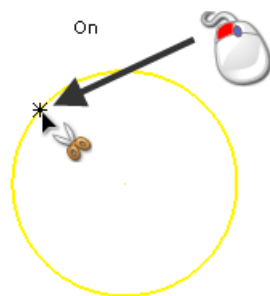
Click+drag the cursor to the end of the hole.
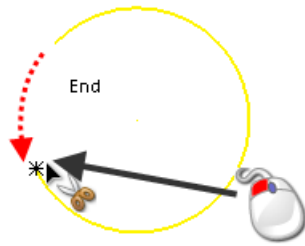
Release the mouse.
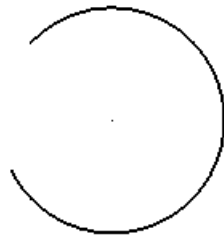
When you create a hole in an open object, such as a line, you are effectively splitting the object into two.

Edit > General Edits > Limit Interactive

This option allows you to remove portions of wireframe objects by simply selecting them. The portion is removed to the position where another wireframe object intersects it.

1   Click the **General Edits** flyout to display the **General Edits** toolbar.

2   Click **Interactively limit wireframes** (*General Edits toolbar*).

The cursor changes to the selection hand icon when you move it over objects that can be selected.

3   Click to select a line, arc, curve or composite curve.
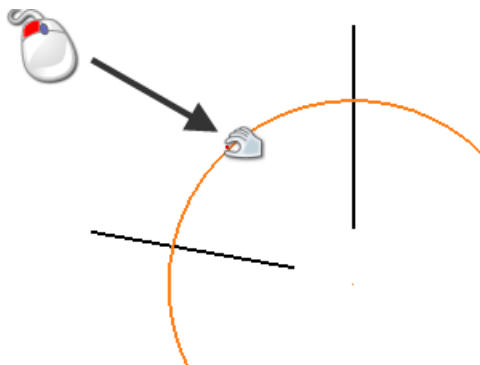
The selected portion is automatically limited to the intersection of any wireframe objects.

If no wireframe object intersects the selected object, then the object is deleted.

Edit > General Edits > Move

The **Move** option allows you to move a group of objects or copy multiples of them by a specified offset.

1 Select a group of objects.

2 Click the **General Edits** flyout to display the **General Edits** toolbar.

3 Click (*General Edits toolbar*)

4 Click **Move/copy object** (*Move/copy flyout*) tro display the **Move** toolbar.

In the graphics window, the move origin marker is placed at the centre of the selection of objects.
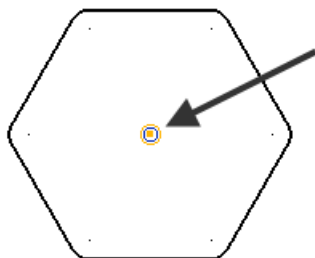
The move-origin is used to move the objects. When you input the position to move the objects, the move origin is moved to the new position and the objects are moved such that they are placed relative to the move origin.

If you are moving a single object and you select it at one of its key points, then the move origin is at the key point.

5 Enter values for *X*, *Y* and *Z* to define the **Movement**. As you enter the coordinates, a preview of the transformed items is displayed to allow further modification before applying the change.
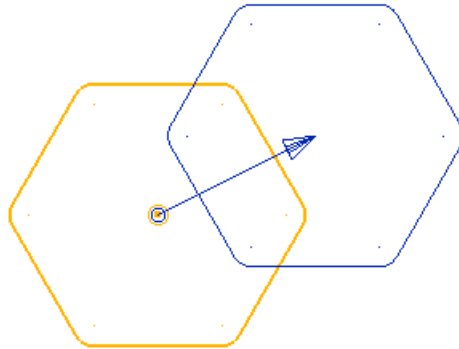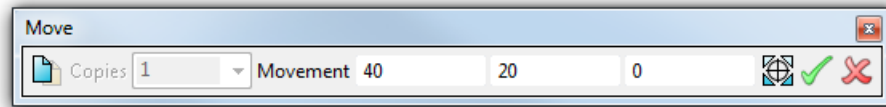
6 You can move the marker to change the move origin using one of the following techniques:

- drag the marker to a new position
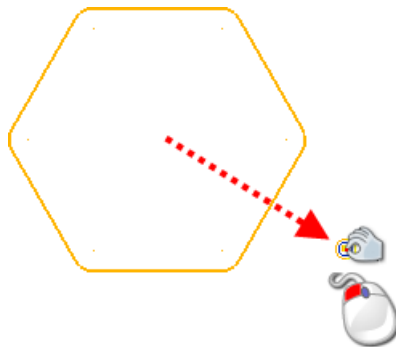
- click **Move origin** ⊞ and click anywhere in the graphics window or enter a new position using the **Status bar**.

7 You can create copies of the objects. Click **Keep original** 📄 and enter the number of copies you want in the **No. of copies** box.

8 Input a position to move the **Move origin**.

If you are just moving the objects, they are moved relative to the move origin.

Input position to
move objects
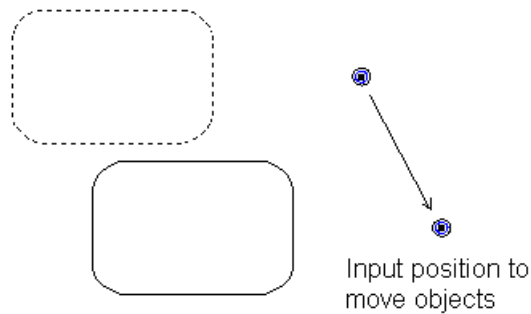
If you are creating multiple copies, the offset of the move origin defines the offset of the copies from each other.

Offset of the
move origin
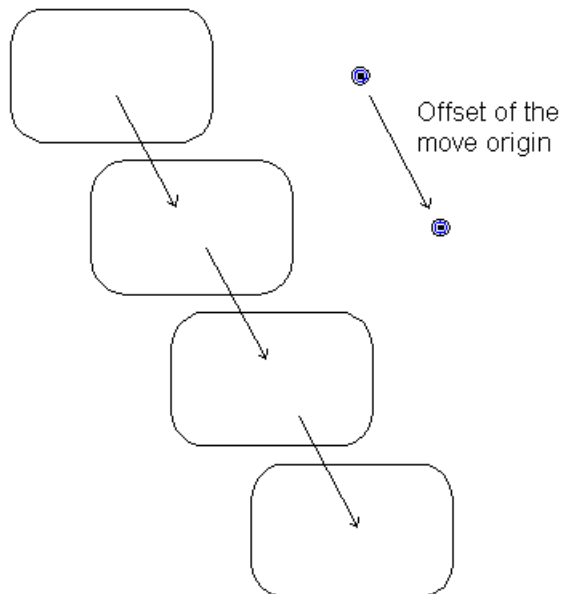
9 Select new objects using box-selection. Box-selection is when you hold down the left mouse button and drag the mouse to form a box around a group of objects.

Add further objects to the selection by pressing the **Shift** key and box-selecting. You can remove objects from the selection by pressing both the **Shift** and **Ctrl** keys and box-selecting.

Pressing the **Ctrl** key and box-selecting objects removes them from the selection if they are already selected, otherwise they are selected.

10 Click **Apply** to apply the preview, or click **Cancel** to cancel the operation.

You can also move and copy objects using **Drag move (see page 59)** and **Drag copy** (see page 59).

Drag move

1 Select one or more objects.

2 Drag the selection to another position. The objects move to where you release the mouse button.

A dialog is displayed asking you to confirm the drag move. It also displays the distance (in terms of coordinates) the objects have moved.
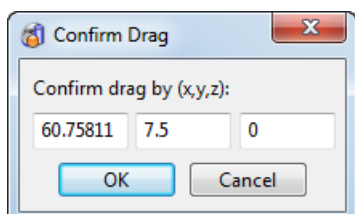
3 Click **Yes** to confirm the drag move.

Drag copy

1 Select one or more objects.

2 Hold down **Ctrl** and drag the selection to another position. The originals remain in place, and copies appear when you release the mouse button.

A dialog is displayed asking you to confirm the drag copy. It also displays the distance (in terms of coordinates) of the copied objects from the original objects.
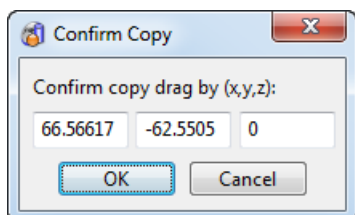
These copies do not affect objects already on the clipboard.

3 Click **Yes** to confirm the drag copy.

Edit > General Edits > Stretch

Use this option to stretch wireframe objects.

To stretch wireframe objects

1 Click the **General Edits** flyout to display the **General Edits** toolbar.

2 Select the points on the wireframe objects by drag-selecting them. All points within the box are used to stretch the objects.
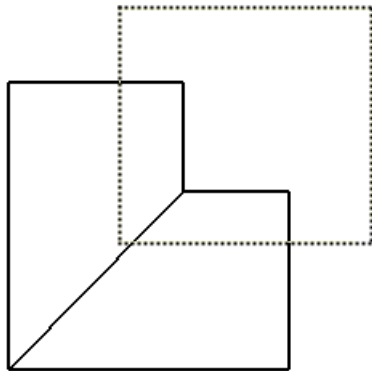
For each object type, the points used to stretch the object are given below.

> **Line** — The start and end points are used to stretch a line.

> **Arc** — The start and end points are used to stretch an arc

> **Curve** and **composite curve** — All key points are used to stretch a curve.

3 Click (*General Edits toolbar*)

4 Click **Stretch object** .

In the graphics window, you will see the stretch origin marker ①. It is placed at the centre of the selection of objects. The stretch points are also marked ②.
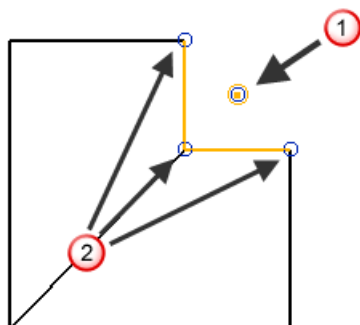
The stretch origin is used to stretch the objects. When you input the position to stretch the objects, the stretch origin is moved to the new position and the stretch points are moved such that they are placed relative to the stretch origin.

If a whole object is selected, it is highlighted in the selection colour and is moved rather than stretched.

5　To move the relative position of the stretch origin, click the **Stretch origin** ⊞ button and enter a new position in the **Status** bar.

6　To change the selection of stretch points, drag a box around a new set of points. You can also add to the selection by using **Shift+drag** and dragging a box around the required points.

7　To stretch the objects, enter a position, drag a stretch point or click in the graphics window.

If you enter a position, the stretch origin moves and the stretch points move relative to it.

In this example, the stretch origin is moved by dragging the top stretch point.
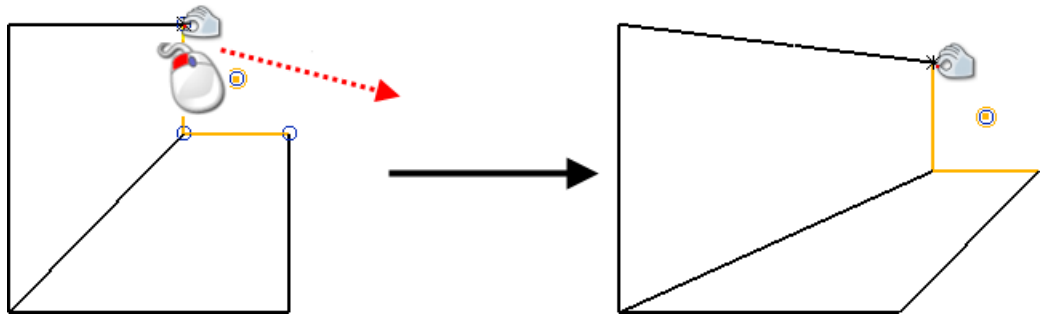
Edit > General Edits > Place Wireframe

This option places a curve between two points on a single object or on separate objects. The curve is copied from a selected wireframe item. **Place wireframe** ⟿ is also available on the **Move/Copy** flyout ⬆ (*General Edits toolbar*)

Placing wireframe between two points

To place wireframe between two points:

1　Select the wireframe item. You can use any open curve or line.
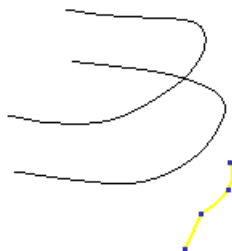
2　Click the **General Edits** ⊕ flyout to display the **General Edits** toolbar.

3 Click the **Move/copy object** flyout 🔲 (*General Edits toolbar*).

4 Click **Place Wireframe** 〰️ (*Move/copy flyout*).

Two points are displayed on the selected wireframe and the **Position curve** (see page 62) dialog is displayed.

5 Use the dialog to select the two points where you want to place the curve.

Position curve dialog

This is used to place a curve between two points on a single object or on separate objects. The curve is copied from a selected wireframe item.
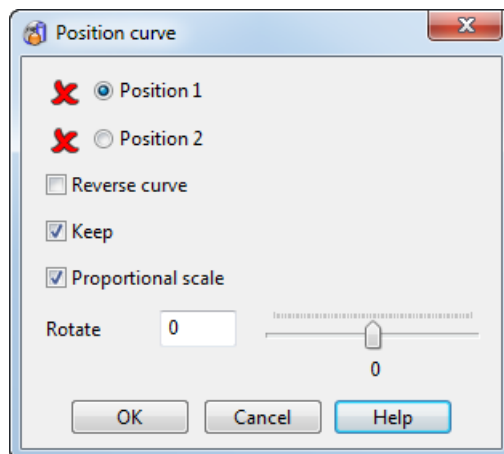
1 Select the wireframe item you want to place.

2 Select **Position 1;** the first position on an object where you want the new curve to touch.
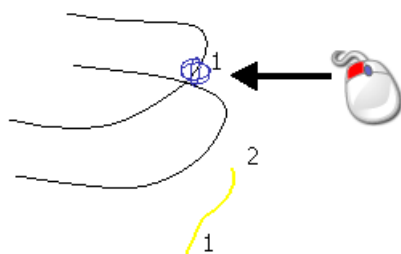
3 Select **Position 2;** the second position on an object where you want the new curve to touch.
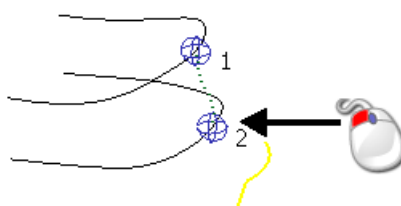
4 Use the other options to adjust the position of the curve.

If *ON*, **Reverse curve** reverses the new curve.

If *ON*, **Keep** keeps the original wireframe. If *OFF,* the original wireframe is deleted.

If *ON*, **Proportional scale** resizes the new curve by keeping its ratio of width to height.

Rotate allows you to specify an angle of rotation or you can use the slider to rotate the new curve. The curve rotates between the two points.

The example below shows the curve rotated to -90°.
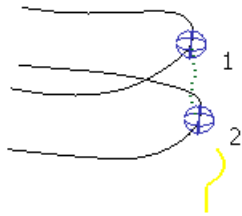
5   Click **OK** to create the new curve with the specified options .
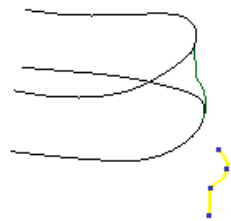
Edit > General Edits > Align items

Use this option to adjust the alignment of the model (see page 208).

Edit > General Edits > Best-fit Align

Use this option to align multiple items (see page 211) that are already roughly aligned.

Edit > General Edits > Rotate

Use this option to rotate a group of objects by a specified angle. The objects are rotated around an axis normal to the principal plane, for example, if the principal plane is YZ, the objects are rotated around the X axis.

You can rotate the objects to give one or more copies. The following picture shows how three copies have been rotated from the original, each separated by an angle of 30°.
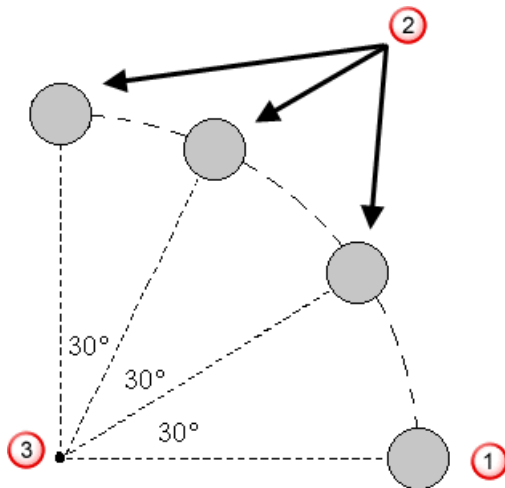
① Original object.

② Copies.

③ Rotate axis.

To rotate a group of objects:

1 Click the **General Edits** ⟳ flyout to display the **General Edits** toolbar.

2 Select a group of objects.
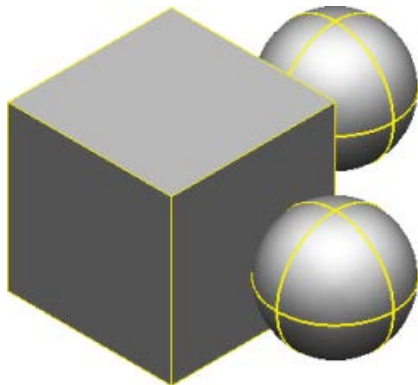
3 Click **Rotate** ⟳ (*General Edits toolbar*).

In the graphics window, you will see the rotate axis marker. It is an arrow which is placed at the origin of the current workspace and points along the axis normal to the principal plane. Objects are rotated around the rotate axis.
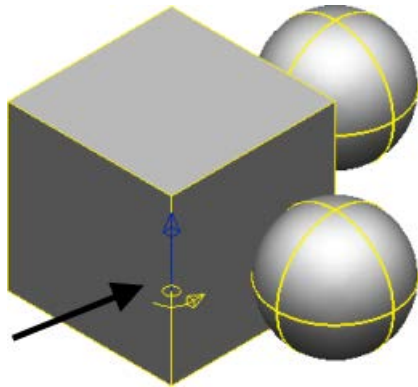
4   To move the marker to change the axis, do one of the following:

  ▪   Drag the marker to a new position. You can modify the angle value after dragging the selected item.
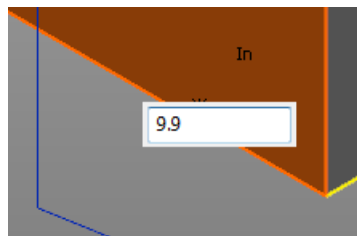
  If you don't wish to enter a value, you can ignore the box; it will disappear when you continue with your work.

  ▪   Click **Rotate axis** and enter a position.
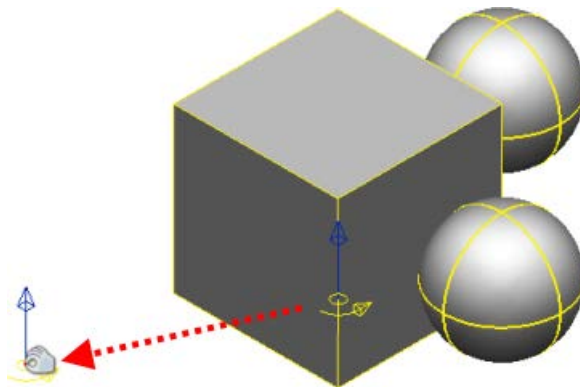
If you change the principal plane, the rotate axis automatically points in the direction of the axis normal to the new principal plane.

5   If you want to rotate the objects and keep a copy of the original objects, select the **Keep original** button. The button changes to ..

If you want to create copies of the objects and the **Copies** box is greyed out, select the **Keep original** ![button] button. The button changes to ![button]. Enter the number of copies you want in the **Copies** box.

Enter the angle of rotation to rotate the objects or drag the objects around the rotate axis. A graphical preview of the transformed items is displayed when you enter an **Angle**. The example below shows the preview that is generated when the solids are selected and an **Angle** of **60** is entered.
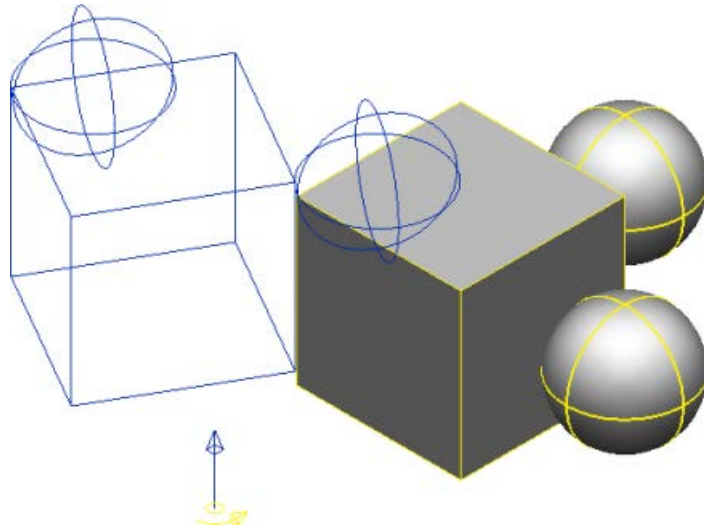
If you are creating many copies, the angle of rotation defines the angle of rotation of the copies from each other.

If we wanted 3 copies with an angle of rotation of 60°, we get the following where each surface is rotated 60° from the one next to it.
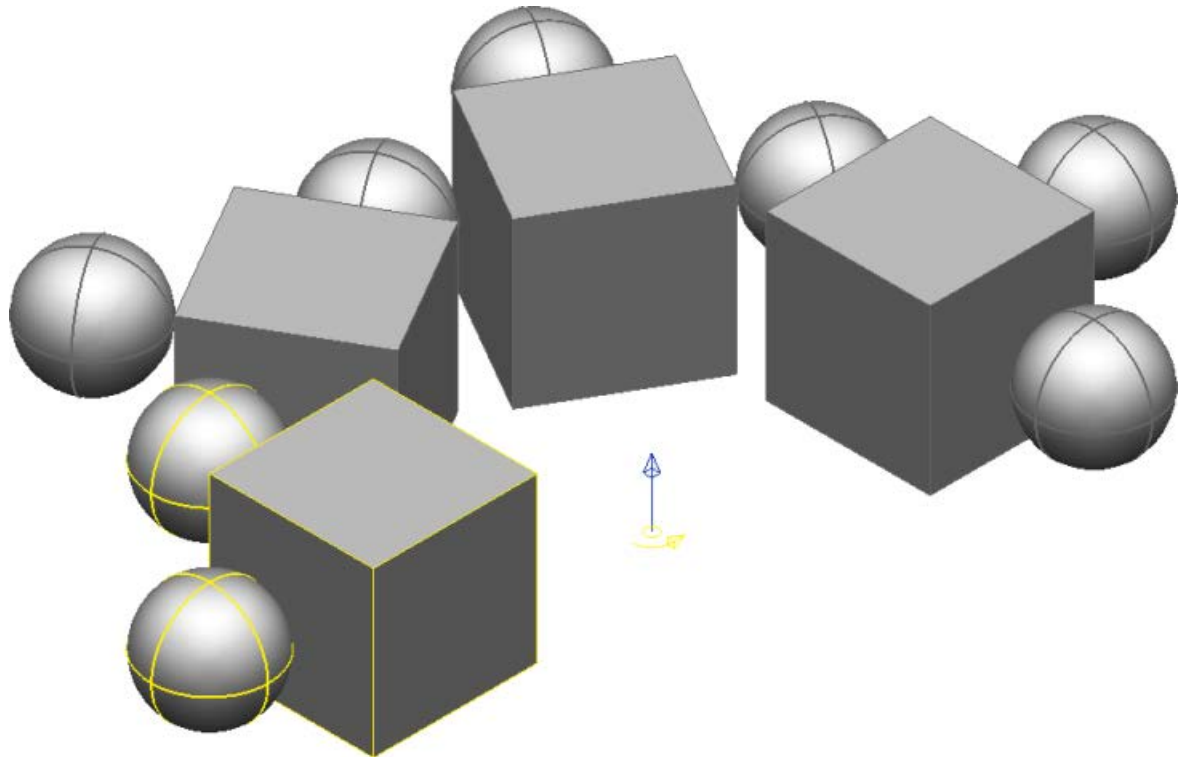
6   You can select new objects to operate on using box selection.

If you want to add further objects to the selection, use **Shift+drag** to select them. If you want to remove objects from the selection, use **Ctrl+Shift** and click or drag-select the objects.

Ctrl+drag removes objects from the selection if they are already selected, otherwise they are added to the selection.

7   Click **Apply** ✅ to accept the changes or **Cancel** ❎.

Rotating by dragging

You can define the rotation angle by dragging the selected objects.

1   Left-click any point on the selected objects and hold down the mouse button.
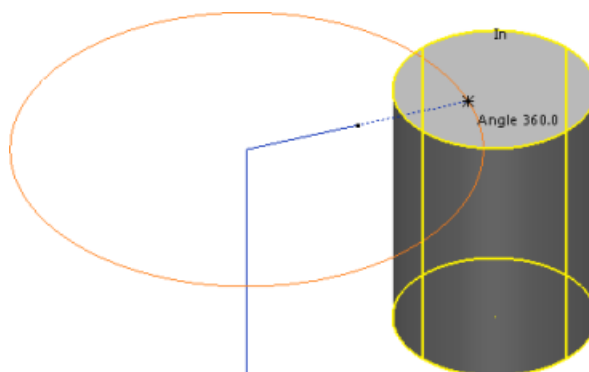
Extra instrumentation is drawn to help you rotate the objects; The angle of rotation displayed, line is drawn from the selected objects to the rotate axis marker and a circle is drawn around the rotate axis from the position the line touches the objects.

Both the line and the circle are drawn in the plane parallel to the principal axis.

The line is half solid and half dashed. If the cursor is over the solid part when you drag the objects ①, the rotation angle changes by increments of $15°$. With the cursor over the dashed part ②, the angle changes by $1°$. If you move the cursor so that it lies outside the circle ③, the angle changes by $0.1°$.
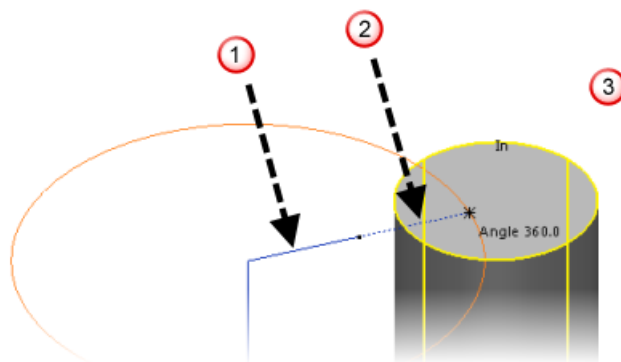
The circle shows the rotation path around the axis.

2 Drag the objects to the rotation angle you want. As you move the mouse, the angle updates.
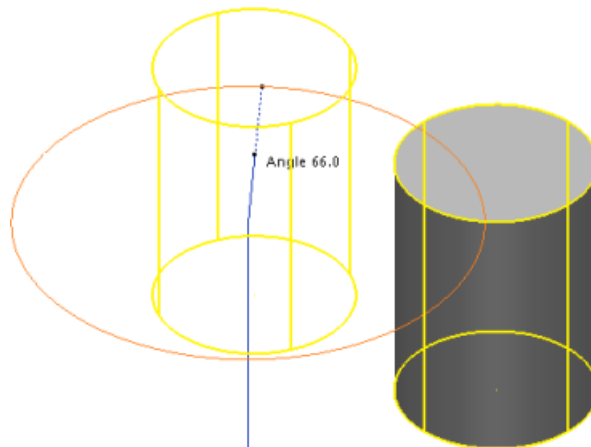
3 Release the mouse button when you reach angle you want. The objects are drawn at the new position.
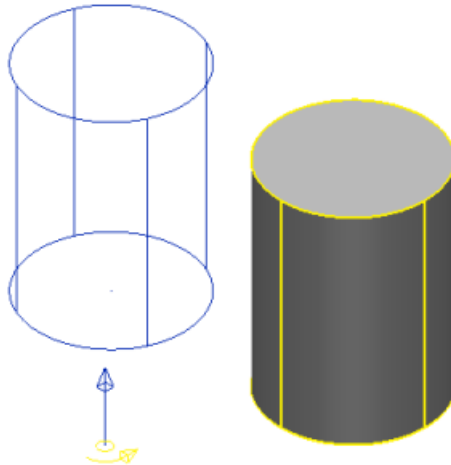
4 You can select new objects to operate on using box selection.

If you want to add further objects to the selection, use **Shift+drag** to select them. If you want to remove objects from the selection, use **Ctrl+Shift** and click or drag-select the objects.

Ctrl+drag removes objects from the selection if they are already selected, otherwise they are added to the selection.

Edit > General Edits > Mirror

Use this option to create objects that mirror selected objects through a particular plane.

There are three ways of defining the mirror plane:

- Selecting a line on your model.
- Creating a temporary user-defined line by entering two points.
- Using one of the plane buttons from the **Mirror** toolbar.

To mirror objects:

1 Click the **General Edits** flyout to display the **General Edits** toolbar.

2 Select the objects to be mirrored. If you use box selection, you can:

- add further objects to the selection by holding down the **Shift** key and box selecting.
- remove objects from the selection by holding down the **Shift** and **Ctrl** keys and box selecting.
- add/remove objects by holding down the **Ctrl** key and box-selecting objects.

3 Click **Mirror items** ◰◰ (*General Edits toolbar*) to display the **Mirror** toolbar. The cursor changes to ▸.

> 📝 *By default **Keep Original** ▱ is selected. If **Keep Original** is deselected, mirrored objects will be created, but the original objects will be deleted.*

4 Use **Create multiple copies** ⊞ to copy the selected items three times, mirrored in the two non-principal planes.

5 Use the buttons on the **Mirror** toolbar to mirror an object.

Using a line to define the mirror plane

1 Select the object to mirror.

2 Click **Mirror Items** ◰◰ (*General Edits toolbar*)

3 Click **Mirror in a wireframe line** ▱ (*Mirror toolbar*) to mirror objects across the mirror plane.
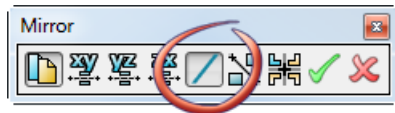
4 Click a line.
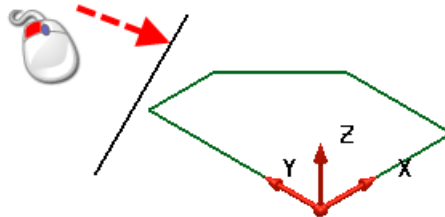
A preview of the mirrored object is displayed, together with the mirror plane.
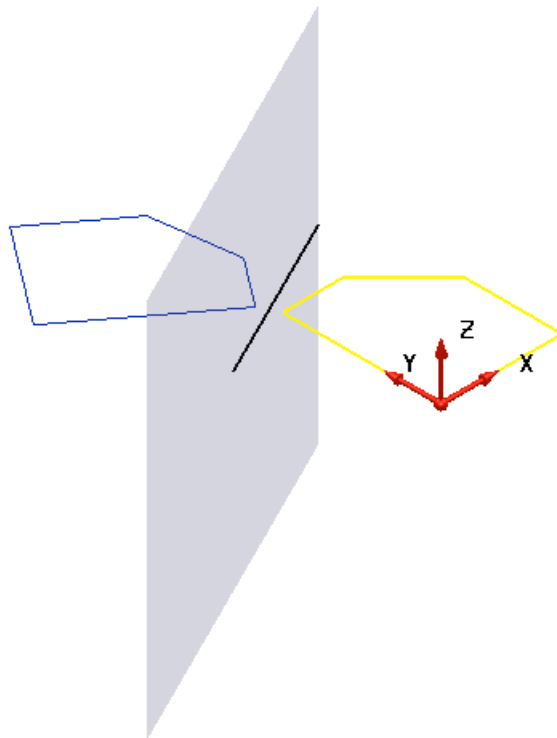
5 Click ✅ to confirm the edit or ❌ to cancel.

Using a user-defined line created from two points

Define a mirror plane by creating a line on the principal plane.

1 Click 🔲. The cursor changes to 🖱️.

2 Click a position on the screen to define the start point of the line ①.

3 Move the mouse to position ②. As you move the mouse, the line and the mirrored objects are displayed on the screen to guide to you.
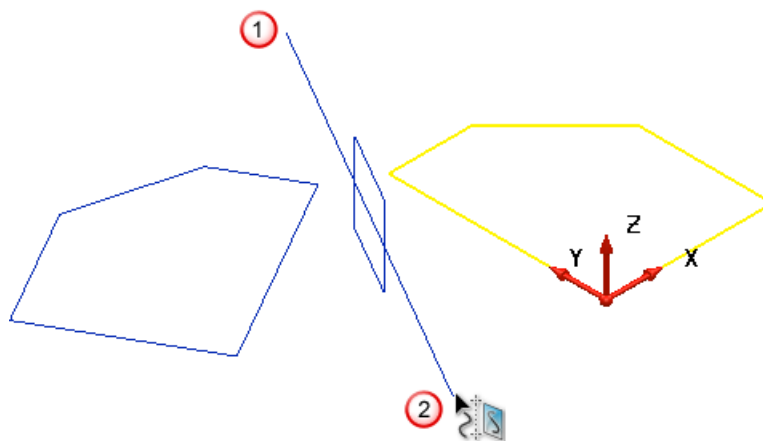
4 Click to define the point ② and mirror the objects. A preview of the mirrored object is displayed, together with the mirror plane.
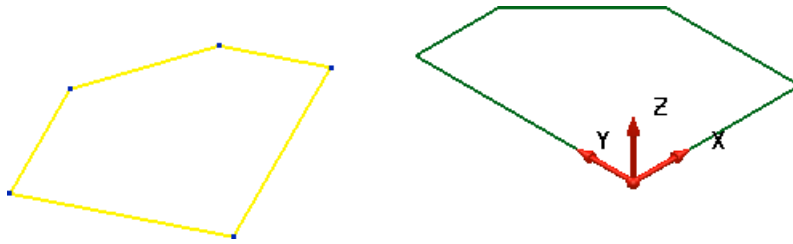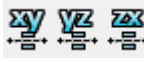
5 Click ✅ to confirm the edit or ❌ to cancel.

Using a plane button

For some applications, such as creating symmetrical designs it can be quicker to use the plane buttons.

> *The workplane defines the origin of the mirror plane. The example uses a workplane on the object. Defining a different workplane will give different results.*

1 Select one of the plane buttons XY YZ ZX to mirror the objects on the opposite side of your chosen plane. A preview of the mirrored object is displayed, together with the mirror plane.

With **YZ** selected:
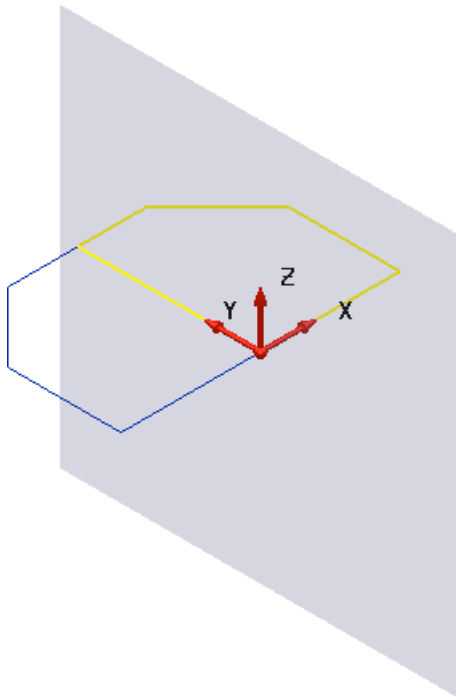
With **ZX** selected:
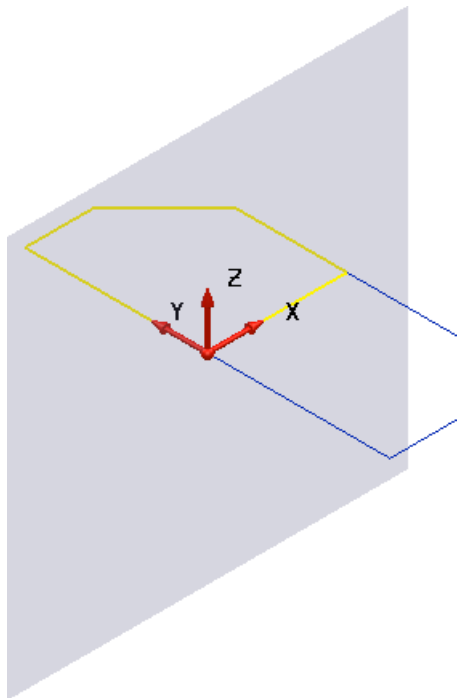
2 Click ✅ to confirm the edit or❌ to cancel.

📝 *Boolean features can be mirrored in the same way as other solid features.*

Edit > General Edits > Offset

Use this option to offset objects by a set distance:

1 Select the object or at least one surface that you want to offset.

2 Click the **General Edits** 🔄 flyout to display the **General Edits** toolbar.

3 Click **Offset** 📦 flyout (*General Edits toolbar*).

4 Click **Offset object** 📦 (*Offset flyout*).

5 Use the **Offset** toolbar to offset the objects.

Surfaces and wireframe

An arrow appears on each object, indicating the direction of the offset. If you want to offset an object in the opposite direction click its arrow or enter a negative value in the **Distance** box.

Solids

Solids do not display arrows to indicate the direction of the offset. The outside material indicates the positive direction and the inside material indicates the negative direction. To reverse the offset direction, you need to reverse the whole solid's direction. (Note that the history tree is lost when a solid is reversed.)

3D curves

The offset direction of a 3D curve is not fixed. The principal plane will determine the shape of the offset curve. Try offsetting the curve using different principal planes. You may need to create a workplane.

Meshes

Meshes do not display arrows to indicate the direction of the offset. The outside material indicates the positive direction and the inside material indicates the negative direction. Use the **Smoothing** (see page 77) option to apply a boundary smoothing factor along the mesh edge.

Selecting objects for offsetting

You can select new objects using box-selection:

- Add further objects to the selection using to box select the required objects.

- Remove objects from the selection using to box select the required objects

- Use **Ctrl** to box-select objects to remove them from the selection if they are already selected, otherwise they are selected.

 Box-selection is when you hold down the left mouse button and drag the mouse to form a box around a group of objects.

Offsetting objects with a discontinuity

Select one of the following options from the **Offset** menu:

- **Retain discontinuities** preserves any discontinuities in the objects that are offset.
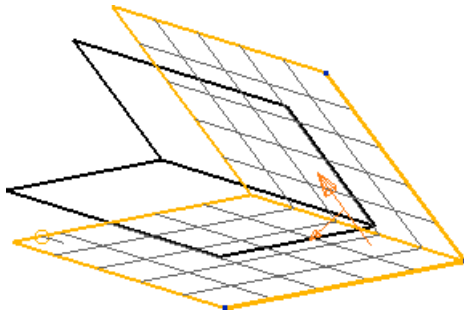
- **Round discontinuities** rounds any discontinuities when the objects are offset.
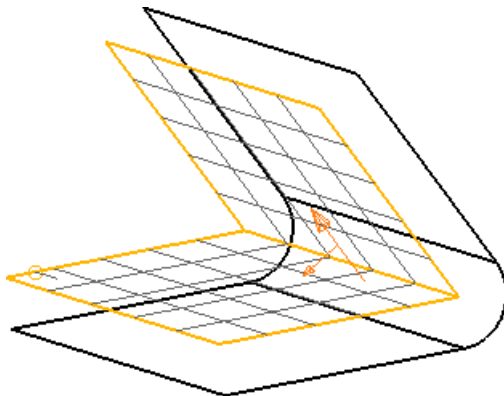
Offsetting surfaces

Select one of the following options from the **Offset** menu:

- **Always create an offset surface** 🍌 preserves any discontinuities in surfaces during surface offset creation.
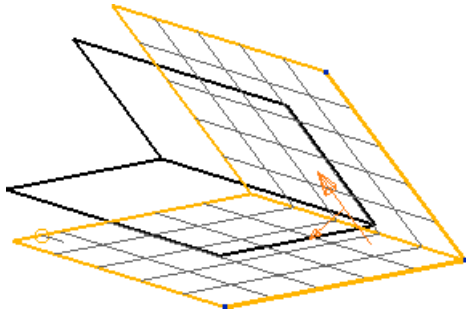
- **Split surface at discontinuities** 🔵 splits the suface at discontinuities during offset creation. The offset surface will be a solid, even if you started with a surface.

  The edges at the discontinuities are extended until they intersect.
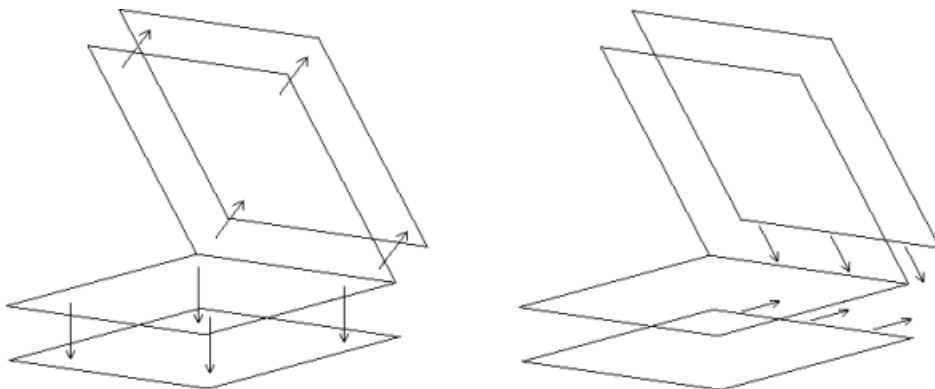
  A solid is created from the set of offset surfaces.
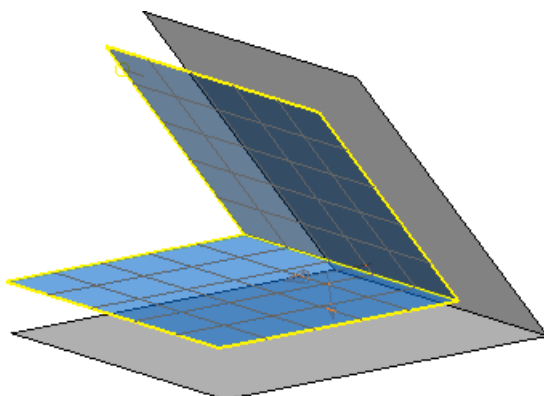
  A discontinuity in a surface normal is defined by the **Angular tolerance** value in the **Surface Discontinuity Tolerance** section on the **Units and Tolerance** page of the **Options** dialog.

  📝   *Primitive surfaces are automatically converted into power surfaces.*

  If a curve or composite curve is dependent on a surface, its offset will lie on the surface.

Offsetting meshes

You can use the **Smoothing** option on the **Offset** dialog (*General Edit toolbar*) when you are offsetting one or more meshes.
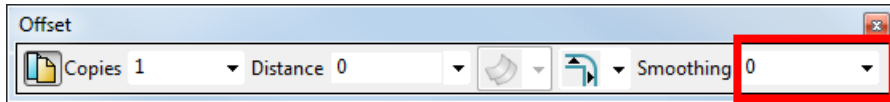
The boundary smoothing factor recalculates and moves the points of the mesh boundary when it is being offset. This reduces the number of jagged edges and creates a boundary with smooth edges.
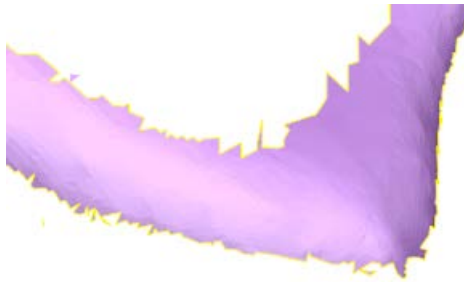
Smoothing = 0   **Smoothing** = 2

Enter a numerical value into the **Smoothing** field to define how the mesh boundary is smoothed. The magnitude of the smoothing is determined by the following:

- 0 — no smoothing, the mesh boundary is unchanged (default).

- Low values — the mesh boundary keeps most detail, but it retains some jagged edges.

- High values —the mesh boundary loses most detail, but it becomes much smoother.

> *The **Smoothing** function only accepts values greater than or equal to 0.*

Offsetting specification options

Use the following options to specify how you want to use the **Offset surfaces** and **Offset objects with discontinuities options**.
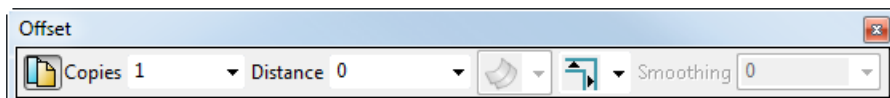
- **Keep Original** is selected by default. Deselect this option to offset the object and delete the original.

- **No. of Copies** is set to **1**. To offset multiple copies of the objects, enter the quantity and press *Enter*.

- **Distance** is set to **0.** Enter an offset value and press *Enter* to offset the objects. To offset the selected objects in the opposite direction, enter a negative value.

Edit > General Edits > Scale

Use this option to expand or reduce objects in size.

1   Select the objects to scale (see page 79).

2   Click the **General Edits** flyout to display the **General Edits** toolbar.

3   Click (*General Edits toolbar*).

4   Use the **Scale** toolbar to expand or reduce the objects in size.

The position of the scale origin controls the way the model is scaled. By default, the scale origin is placed at the origin of the current selection and objects are scaled relative to that position. It is important to position the scale origin correctly to achieve the required result.

5   Change the scale origin by moving the marker in one of the following ways:

- Drag the marker to a new position.

- Click the **Scale origin** button and enter a position.

6   Use the drop-down list to select one of the following scale options.

- **Uniform** (see page 79)

- **Non-uniform** (see page 80)

- **To projected Volume** (see page 81)

- **Enclosed Volume**. Scale the volume enclosed by the selected solid and face of the active workplane. Enter the required volume in centilitres.

You can also scale objects by dragging them on the screen; this method works best when used to scale wireframe.

1   Enter the scaling factor. A preview of the scaled item is displayed.

2   Click one of the following:

- ✔ to scale the selected items.

- ✖ to close the toolbar without scaling the selected items.

The display of these buttons and the graphical preview is controlled by **Show a preview of transformed items** on the **General Edits** options dialog (see page 78).

Selecting objects to scale

Use the following techniques to select the objects to be scaled:

- Select new objects to operate on using box selection.

- Add further objects to the selection by holding down **Shift** and box-selecting. You can remove objects from the selection by holding down the **Shift+Ctrl** and box-selecting.

- Remove objects from the selection by holding down **Ctrl** and box-selecting objects that are already selected.

Uniform scaling

This option lets you resize selected objects in a uniform manner, making them larger or smaller equally in all directions. You can also lock the scaling factor in any of the directions to 1.

1 Select the objects you want to scale.

   To scale a workplane with the objects, select it after entering the scale command.

2 Click **Scale items** (*General Edits toolbar*).

   In the graphics window, you will see the scale origin marker. It is placed at the origin of the current workspace. Objects are scaled relative to the scale origin.

3 If you want, move the marker to change the scale origin. You can either drag the marker to a new position or click the **Reposition scale origin** and enter a position.

4 By default, **Keep Original** is deselected. To keep the original objects, click .

5 To lock the scaling factor of any axes to 1, click on the required **Axis Lock** button on the toolbar. For example, click to lock the scaling factor of the X axis.

   If you scale an arc and lock the scaling factor of any axis to 1, then the arc is converted to a curve.

6 Enter a scale factor and press **Enter**. The model now shows the scaled objects.

   Primitive surfaces are converted to power surfaces and primitive solids to general ones.

7 Select new objects to operate on using box-selection. Box-selection is when you hold down the left mouse button and drag the mouse to form a box around a group of objects.

Add further objects to the selection by holding down **Shift** and box-selecting. You can remove objects from the selection by holding down the **Shift+Ctrl** and box-selecting.

Holding down **Ctrl** and box-selecting objects removes them from the selection if they are already selected, otherwise they are selected.

Non-uniform scaling

This resizes selected objects in a non-uniform manner, allowing you to choose the scaling to be applied for each axis. Arcs are converted to curves and primitive surfaces to power surfaces by default.

1 Select the objects to scale (see page 79).

To scale a workplane with the objects, select the workplane after starting selecting the scale option..

2 Click **Scale items** .

The scale origin marker is displayed in the graphics window. It is placed at the origin of the current workspace. Objects are scaled relative to the scale origin.

3 Move the marker to change the scale origin in one of the following ways:

- Drag the marker to a new position.
- Click the **Scale origin** button and enter a position.

4 By default, **Keep Original** is deselected. To keep the original objects, click .

5 From the drop down list, select **Non-Uniform**. The toolbar is updated to show scale factor boxes for X, Y and Z.
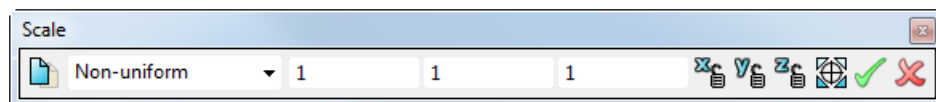
6 Enter a scaling factor for each axis. To lock the scaling factor of any axis to **1**, click on the appropriate lock button.

*If scaled arcs are to remain as arcs, use the **Uniform** option*

7 Click ✔ to scale the selected objects.

Scaling to projected volume

This option resizes surfaces and solids according to their volume as projected on to the active workplane. Closed surfaces scale as you would expect because there is no ambiguity about the volume. However, the volume of open surfaces is ambiguous and can cause unexpected results. For example, the volume of a sphere which has a non-circular curved shape cut from it cannot be defined unless you apply some rules.

To avoid ambiguity PowerSHAPE projects the surface on to the current active plane and then runs the volume calculations. The scale volume algorithm is explained in the next section.

To scale to projected volume:

1 Select a surface.

2 Click **Scale items** .

   In the graphics window, you will see the scale origin marker. It is placed at the origin of the current workspace. Objects are scaled relative to the scale origin.

3 If you want, move the marker to change the scale origin. You can either drag the marker to a new position or click the **Scale origin** button and enter a position.

4 By default, **Keep Original** is deselected. To keep the original objects, click . When the original object is retained and a copy scaled, any dependent items (for example the solid history tree) will be present in the scaled copy.

5 From the **Scale** option menu, choose **Projected Volume.** The drop-down list shows the present projected volume.

6 Enter a new volume in the drop-down list box. The surface and solid is scaled when you press **Enter**. Primitive surfaces are converted to power surfaces and primitive solids to general ones.

Change your selection using the following techniques:

▪ You can select new objects to operate on using box-selection (hold down the left mouse button and drag the mouse to form a box around a group of objects).

▪ You can add further objects to the selection by holding down **Shift** and box-selecting. You can remove objects from the selection by holding down **Shift+Ctrl** and box-selecting.

Holding down **Ctrl** and box-selecting objects removes them from the selection if they are already selected, otherwise they are selected.

## *PowerSHAPE projected volume calculations*

PowerSHAPE calculates the volume of surfaces as follows:

1 Looking perpendicular to the active plane, the projected volumes (*Vol A*) under the outside surface are calculated.

2 Looking perpendicular to the active plane, the projected volumes (*Vol B)* under the inside surface are calculated.

3 *Vol A* is deducted from *Vol B* to give the projected volume.

With the following closed surface example, there is no ambiguity, so the calculated value is the actual volume contained by the surface.
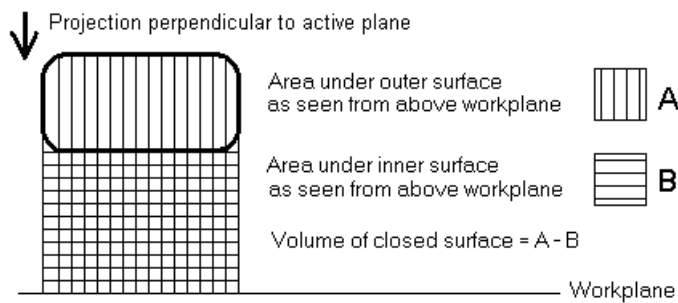
However, the volume of an open surface is ambiguous and varies according to how it is projected on to the workplane. The illustration below shows how the calculated values can vary depending where the active workplane falls in relation to the surface.

A bottle is a good example of how the projections can affect the results. When standing upright above the workplane, the volume under the upward facing outer surface does not include the neck. When the volume under the upward facing inner surface is subtracted from it, the result is not the capacity of the bottle.
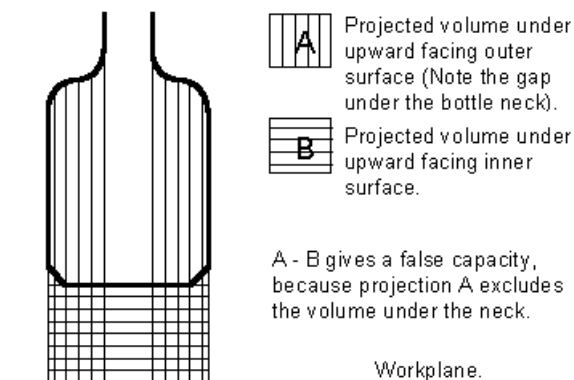
Inverting the bottle causes a similar problem. However, if the bottle is placed on its side, the open neck does not distort the calculations because it cannot be seen from above the workplane. The capacity is calculated correctly.

Projected volume A minus
Projected volume B gives
the correct capacity
because the projection
cannot see the open neck.
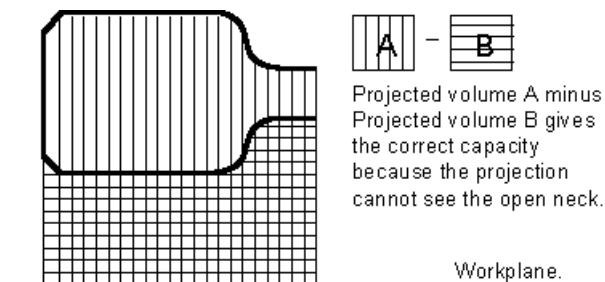
Workplane.

With complex curved open surfaces, which do not have a flat open area, you need to be aware that the calculated values vary according to the surface position relative to the workplane. If possible, close all surfaces before calculating the projected volume or arrange the active workplane so that any openings are on the workplane or perpendicular to it.

The following diagram shows how a bottle model can be arranged above a workplane ① to produce a "correct" volume.
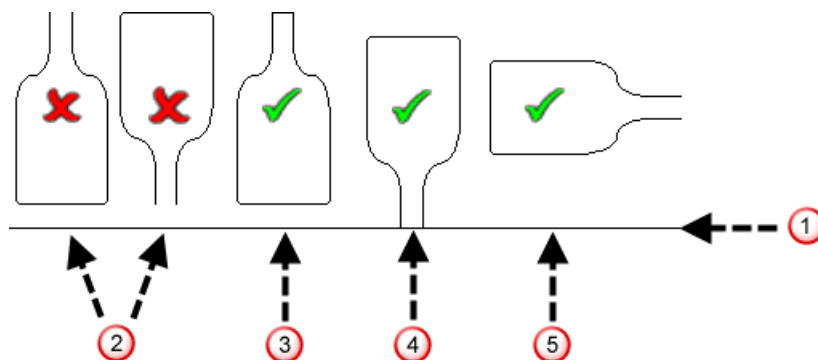
② Central cylinder missing from parts of calculations.

③ Closed surface.

④ Open edge is on a workplane.

⑤ Open edge is perpendicular to the workplane.

Dynamic scaling

You can scale objects by dragging them on the screen. The selected objects are made larger or smaller in any direction. You can also lock the scaling factor in any of the directions to 1.

Dynamic scaling only works in uniform scaling mode because of conflicts with the interface for non-uniform and volume scaling. However, it does essentially do a non-uniform scale.

1 Select the objects you want to scale.
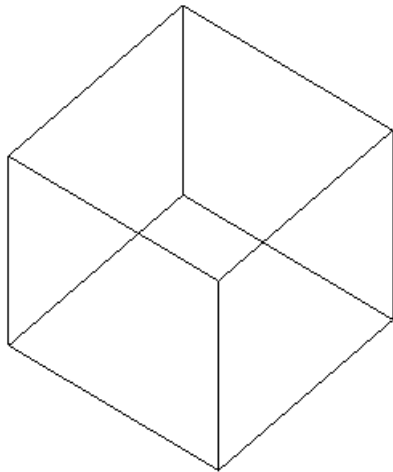
2 Click **Scale items** .

In the graphics window, you will see the scale origin marker ①. It is placed at the origin of the current workspace. Objects are scaled relative to the scale origin.
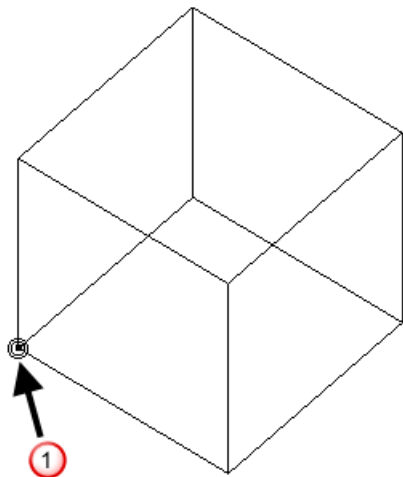
3 If you want, move the marker to change the scale origin. You can either drag the marker to a new position or click the **Scale origin** button and enter a position.

4 By default, **Keep Original** is deselected. To keep the original objects, select **Keep Original**.

5 If you want to lock the scaling factor of any axes to 1, click on the required **Axis Lock** button on the toolbar. For example, click to lock the scaling factor of the X axis.

6 Drag the objects on the screen.

The line from the scale origin to where you first press the mouse button defines a scale of 1 in each axis.

Press the mouse button on the selected objects.

Mouse button
pressed here
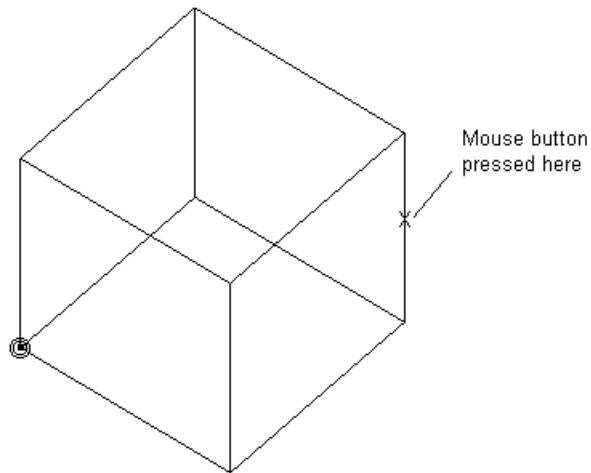
The line, which defines the scale in our example, is drawn below.

Mouse button
pressed here

This line defines
a scale of 1 in
each axis
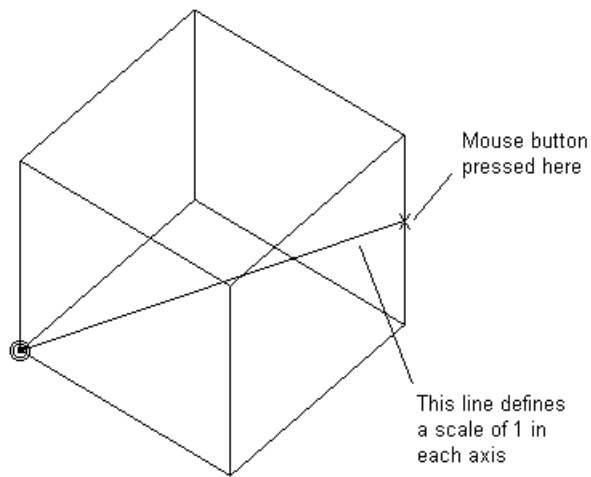
When the mouse is dragged, the objects are scaled and the scale factors are displayed on the screen.
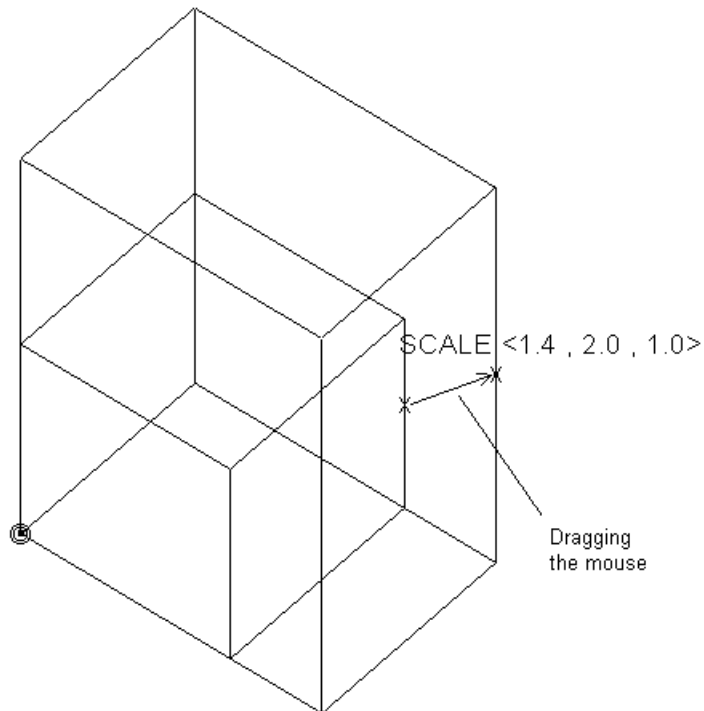
When you release the mouse, the objects are scaled. A scale axis is the line through the scale origin, parallel to the axis of the current workspace.

Change your selection using the following techniques:

- You can select new objects to operate on using box-selection (hold down the left mouse button and drag the mouse to form a box around a group of objects).

- You can add further objects to the selection by holding down **Shift** and box-selecting. You can remove objects from the selection by holding down **Shift+Ctrl** and box-selecting.

  Holding down **Ctrl** and box-selecting objects removes them from the selection if they are already selected, otherwise they are selected.

Edit > General Edits > Create Pattern

Use this option to create rectangular, honeycomb and circular patterns of items and to create patterns on wireframe items. You can also create patterns using the option on the **General edits** toolbar.

Creating a pattern of items

To create a pattern of items:

1 Click the **General Edits** flyout to display the **General Edits** toolbar.

2 Select the object you want to create a pattern from.

3 Click **Create pattern of items** . The **Pattern Edit** dialog is displayed.

4 Use the options on the dialog to create the pattern.

Pattern Edit dialog

The dialog has four tabs. Each tab is associated with a pattern style.

- Rectangular (see page 87)
- Honeycomb (see page 90)
- Circular (see page 93)
- Wireframe (see page 98)

Create Rectangular pattern

This page enables you to create rectangular patterns of objects by creating multiple copies of the selected item.
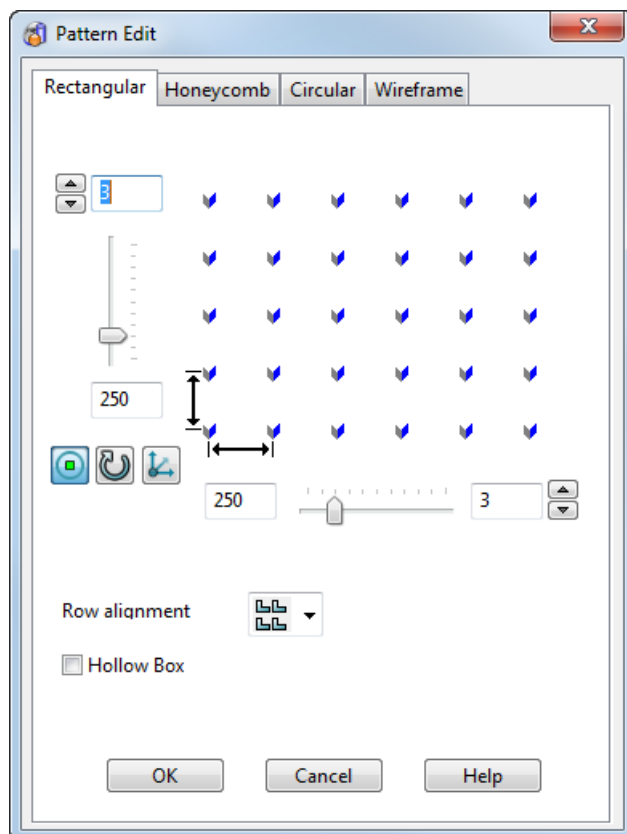
To create rectangular patterns of objects, you need to select the pattern objects you want to create the pattern with. Use the options to determine how the elements are created.
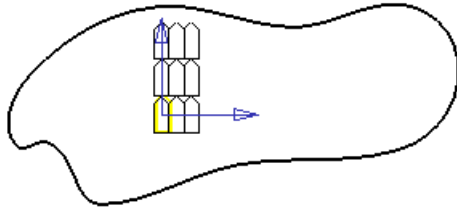
Number of rows — Enter the number of rows in the pattern or use the the arrow buttons to increase or decrease the value.

Distance between rows — Enter a value in the box or use the slider to increase or decrease the value.

Display suppress markers — Display a marker on each pattern element. Click on the markers to turn off individual elements in the created pattern.
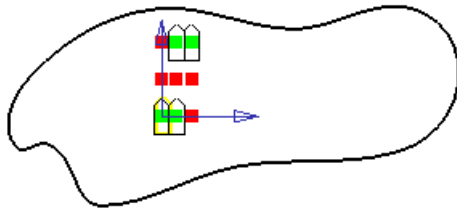
Rotate pattern axes — Rotates the whole pattern 90° around the Z axis.

Set plane of pattern — Display the **Pattern Plane** dialog where you set the coordinates for the origin of where the pattern will be positioned.

Row alignment

Choose an alignment suitable for the pattern effect you want:

 Create all rows with the same alignment.

 Create even rows with rotation.

 Create even rows with mirror.

Hollow Box — Only the outer elements of the pattern are created.
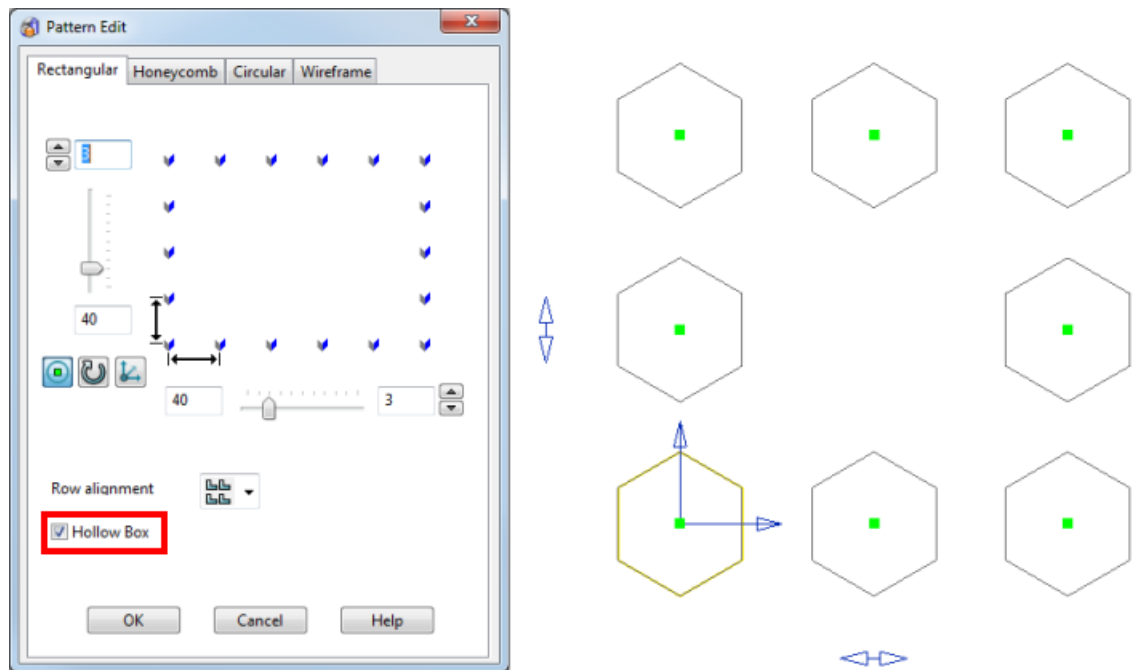
The following additional options are displayed on the dialog if the item you are using to produce the pattern is a feature. They will not be displayed if the selected object just contains solid features.

Remove feature — Control how the original feature is handled when the pattern is created. If *ON*, when the pattern is created, the original feature is absorbed by the pattern and they become one feature. In the solid tree, the feature icon is replaced by a pattern icon

If *OFF* when the pattern is created, the pattern is created as a separate feature and both the original feature and the pattern feature can be edited separately. The solid tree will show both the feature icon and a pattern icon

📝 *This option is not displayed on the dialog if you are editing an existing pattern feature.*

Tile — This option is not currently available.

Edit Definition — This option is only available when you are editing an existing pattern feature. When you select this option, you can edit the pattern feature using the appropriate dialog.

OK — Create the pattern and closes the dialog.

Create Honeycomb pattern

This page enables you to create honeycomb-shaped patterns of objects by creating multiple copies of the selected item.
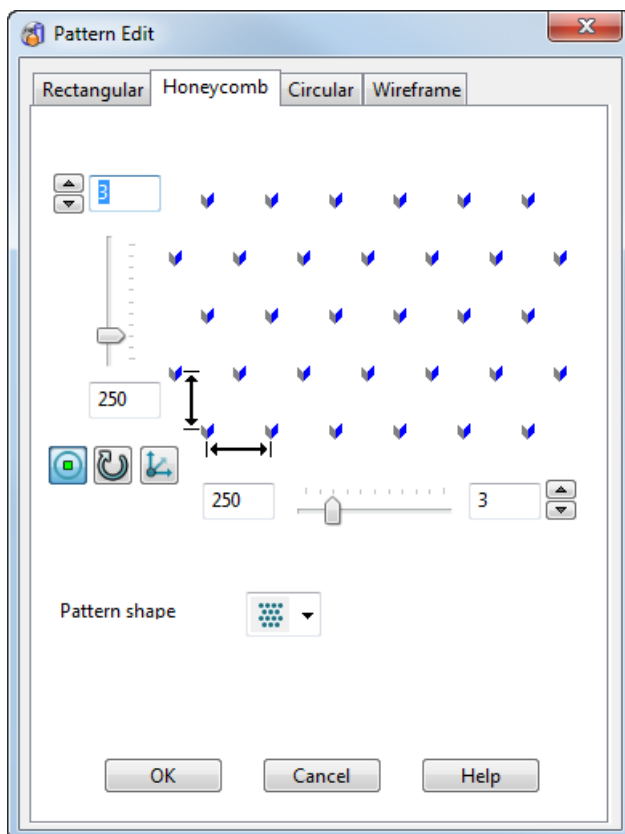
To create honeycomb shaped patterns of objects, you need to select the pattern objects you want to create the pattern with. Use the options to determine how the elements are created.
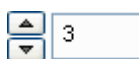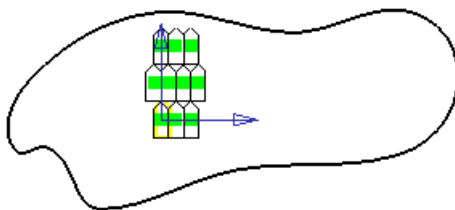
Enter number of rows in the pattern — Use the arrow buttons to increase or decrease the number of rows.

183.58 **Enter the y direction spacing** — You can also use the slider to set the row spacing.

Display suppress markers — This displays a marker on each pattern element. Click on the markers to turn off individual elements in the created pattern.
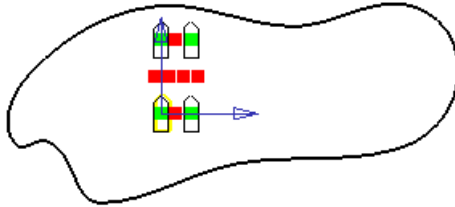
Rotate pattern axes — This rotates the whole pattern through 90° around the Z axis.

Set plane of pattern — This displays the **Pattern Plane** dialog where you set the coordinates for the origin of where the pattern is positioned.

98.859 **Enter the x direction spacing** — You can also use the slider to set the column spacing.

3 **Enter the number of columns in pattern** — Use the arrow buttons to increase or decrease the number of columns.

Pattern shape

Choose a pattern shape suitable for the pattern effect you want:

Create offset rows outwards.

Create offset rows to the right.

Create offset rows to the left.

Create offset rows inwards.

The following additional options are displayed on the dialog if the object you have selected to produce the pattern is a feature. They will not be displayed if the selected object just contains solid features.

Remove feature — Controls how the original feature is handled when the pattern is created. If *ON,* when the pattern is created, the original feature is absorbed by the pattern and they become one feature. In the solid tree, the feature icon is replaced by a pattern icon

If *OFF* when the pattern is created, the pattern is created as a separate feature and both the original feature and the pattern feature can be edited separately. The solid tree will show both the feature icon and a pattern icon

This option is not displayed on the dialog if you are editing an existing pattern feature.

Tile — This option is not currently available.

Edit Definition — This option is only available when you are editing an existing pattern feature. When you select this option, you can edit the pattern feature using the appropriate dialog.

OK — This creates the pattern and closes the dialog.

Create Circular pattern

Use this page to create circular patterns of objects by creating multiple copies of the selected item.
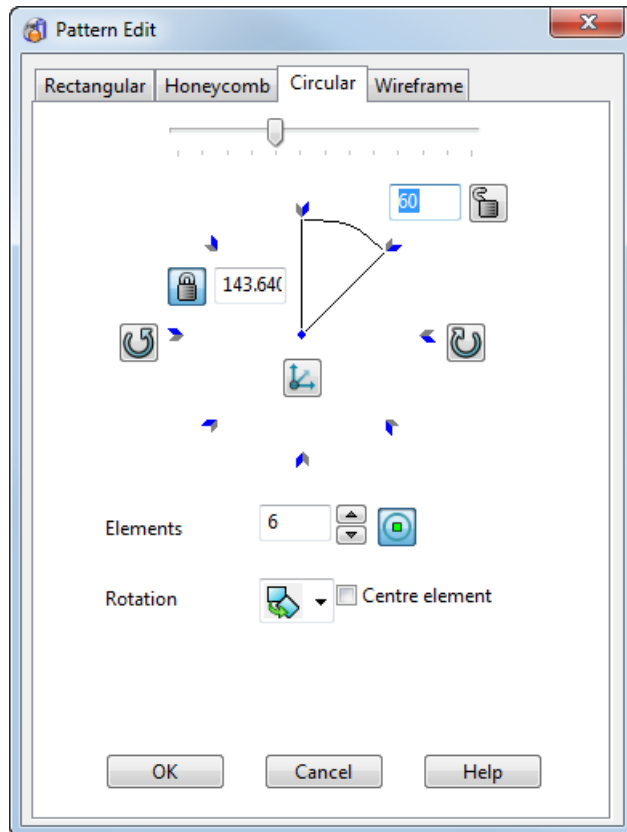
Creating a circular patterns of objects

1 Select the pattern objects you want to use to create the pattern.
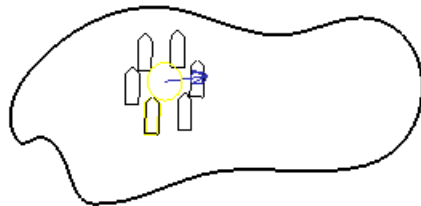
2 Use the options to determine how the elements are created.

Options when creating a circular pattern of objects

Angle — Use the slider to set the angle between pattern elements.

Set angle between pattern elements — Enter the angle required between pattern elements. If you change the angle, the number of pattern elements is adjusted to create the pattern.

In the example below, the angle is set to **40** and the quantity of elements has increased.
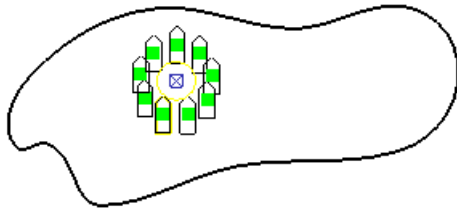
🔓 **Lock the angle** — When you have entered an value, click this to lock the angle. The button changes to 🔒 to indicate that the angle is locked.

In the example below, the angle is set to **60** and locked. The number of elements has been reduced to **4**. The pattern elements remain fixed.

If the angle is unlocked, the spacing between the pattern elements is adjusted.

🔒 55 **Set pattern radius** — Enter the required radius of the elements around the pattern origin axis and press the **Enter** or **Tab** key. The pattern is adjusted to reflect the value you entered. This option is only available when 🔒 is displayed (default). Click 🔒 to swap mode and specify the pattern radius as the distance between the origin and the original object.
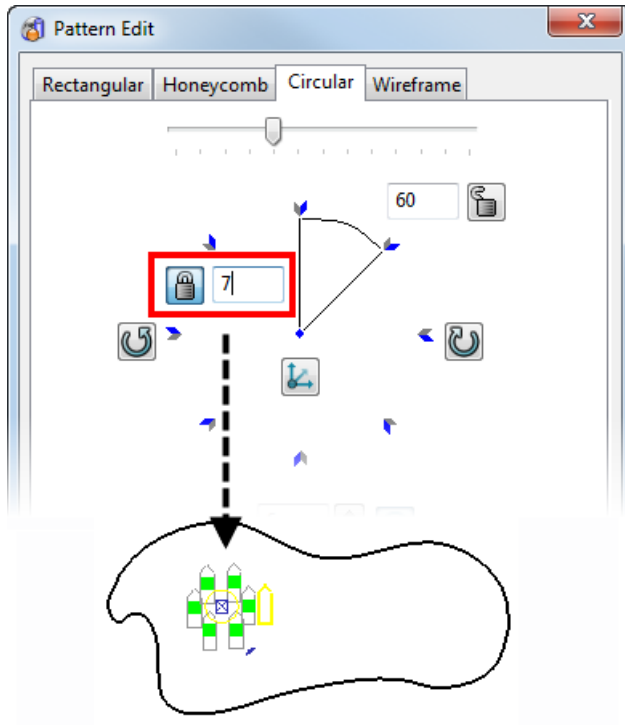
📋 283.72⁹ **Set pattern radius** — The pattern radius is specified as the distance between the origin and the original object. You cannot enter an alternative value. Click 📋 to swap mode and enter a radius.
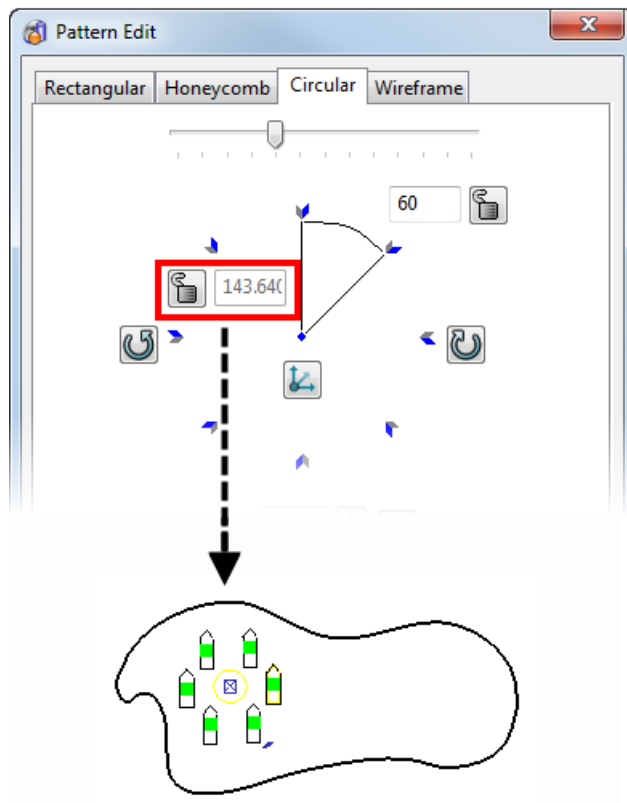
📋 **Rotate pattern anticlockwise** — This rotates the pattern through half of the specified angle.

📋 **Set axis of pattern** — This sets the coordinates for the centre of the circle.

📋 **Rotate pattern clockwise** — This rotates the pattern through half of the specified angle.

3 ▲▼ **Elements** — Enter the fixed number of elements you want around the pattern. Use the arrow buttons to increase or decrease the number of elements.

📋 **Display suppress markers** — This displays a marker on each pattern element. Click on the markers to turn off individual elements in the created pattern.

Rotation

Choose the rotation style for the pattern effect you want. In the examples below the angle is set to **60**.

 Rotate around circle.

 Move around circle.
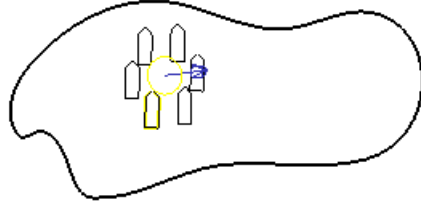
Centre element — This creates a pattern element at the centre of the circle.

Additional options

The following additional options are displayed on the dialog if you selected a feature to produce the pattern. These options are not displayed if the selected object just contains solids.

Remove feature — Controls how the original feature is handled when the pattern is created. If selected when the pattern is created, the original feature is absorbed by the pattern and they become one feature. In the solid tree, the feature icon is replaced by a pattern icon.

If deselected when the pattern is created, the pattern is created as a separate feature and both the original feature and the pattern feature can be edited separately. The solid tree will show both the feature icon and a pattern icon.

Edit Definition — This option is only available when you are editing an existing pattern feature. When you select this option, you can edit the pattern feature using the appropriate dialog.

OK — This creates the pattern.

Create pattern from Wireframe

Use this page to create patterns of objects on wireframe items.
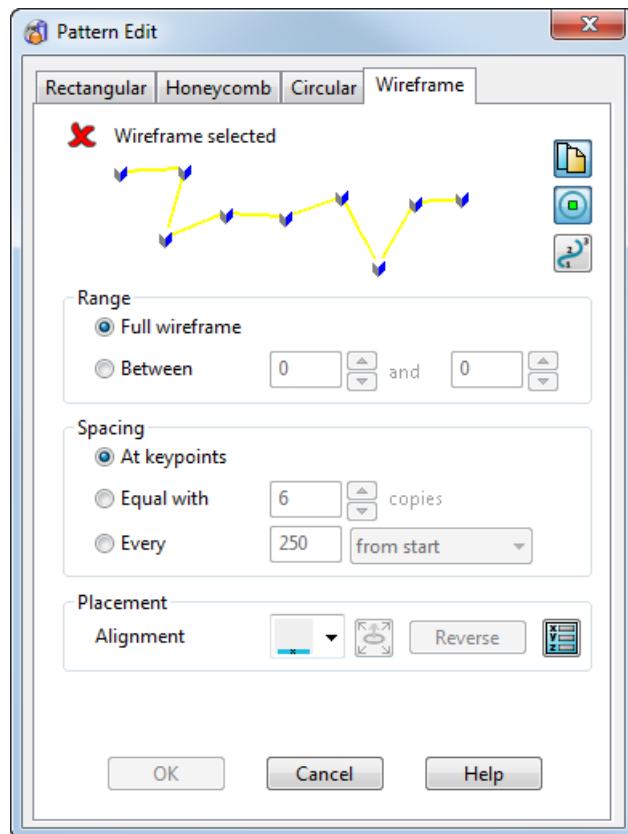
To create patterns of objects on wireframe items, you need to:

1 Select the pattern objects.

2 Display the **Wireframe** page of the dialog.

3 Select the wireframe item that you want to create the pattern on.

4 Use the options to determine how the elements are created on the wireframe.
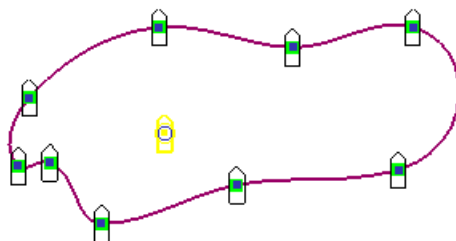
Keep the pattern wireframe — You can choose to keep the wireframe or delete it. Click this button to toggle between the two options.

Display suppress markers — This displays a marker on each pattern element. Click on the markers to turn off individual elements in the created pattern.

 Toggle curve point display — This turns the display of point numbers on the wireframe on or off.
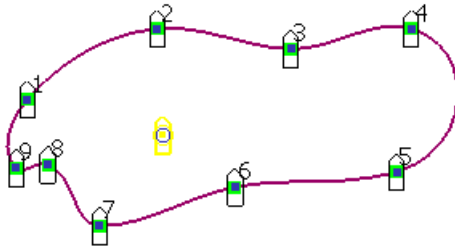
Range

Set the range of the wireframe to create the pattern effect that you want:

Full wireframe — Select this option to include the whole wireframe item when creating the pattern elements, as shown in the example above.

Between — Select this option and specify the range of point numbers on the wireframe you want the pattern to be created between. In the example below the pattern is created between points 3 and 5.
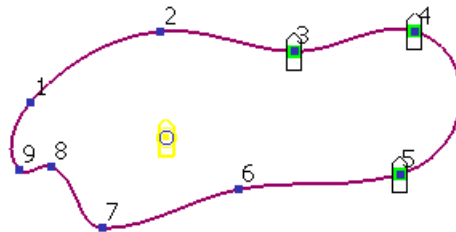
Click to display the Position dialog to specify the position of the present point.

Spacing

Set how the wireframe will be used to position elements:

At keypoints — Select this option to create the pattern at the key points on the wireframe, subject to the range you have specified.

Equal with — Select this option and specify the fixed number of copies of pattern elements to be created within the range specified. The example below shows the pattern created with 20 copies.
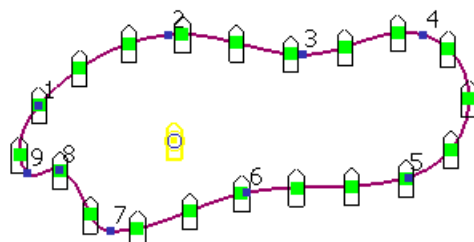
Every — Select this option and specify the spacing distance for the elements in the pattern. Choose an alignment option from the drop down list:

- **from end**

- **from middle**

- **from start**

You can also open the Position dialog to specify the position of the present point.

Placement

Choose an alignment suitable for the pattern effect you want.

Copy along wireframe — This creates a copy of the pattern element at the points and range specified. This is the default setting.

Rotate copies to axis — This displays two points on the original pattern element, initially in the same position, approximately at the centre of the original pattern object. You position these points to create an axis that can be used to align with the normal of the curve.

If the curve does not have surface dependencies, the pattern will align to the normal of the curve. If the curve has surface dependencies, the pattern will align to the normal of the surface.

The **Set plane of pattern** option becomes available.

1 Select the pattern element.

2 Use to toggle between the selection of the points. The points are different colours.
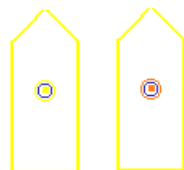
3 Click on each point and move them into the positions where you want to create an axis to align with the normal of the curve.
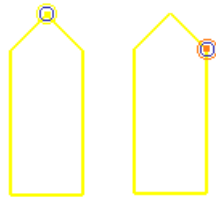
The example below shows the pattern element aligned with the normal on the curve at the axis between the two points.

4 Click the **Reverse** button to change the normal and create the pattern on the other side of the curve at the same axis.
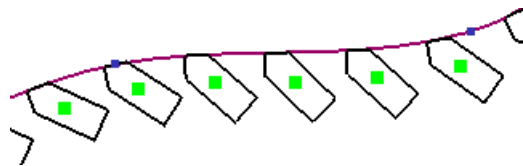
When reversed, the pattern changes as shown below:
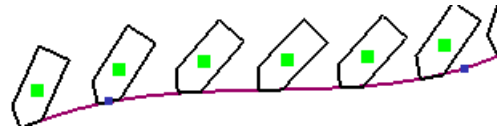
Click ▦ to display the Position dialog to specify the position of the present point.

▦ **Place two points on wireframe** — This displays 2 points on the pattern element. Use ⟳ to toggle between the selection of the points and position them to choose the part of the pattern element you want to lie on the wireframe.

The example below shows the points positioned on the edge of the pattern element, and the pattern elements created on the wireframe.
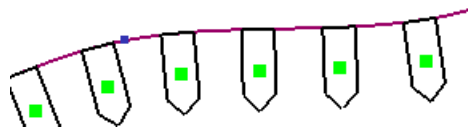
⟳ **Set the point to be repositioned** — This is used with the Alignment options to move the point markers on the pattern element.

Reverse — This flips the side of the wireframe where the pattern will be created, corresponding with the curve normal. For example inside or outside.

 Set plane of pattern — This option becomes available when the **Rotate copies to axis** option is selected from the **Alignment** drop down list. Use the **Pattern Plane** dialog to set the coordinates for the alignment of the pattern elements on the wireframe. The workplane can also be edited dynamically. The normal of the pattern is along the Y axis and the tangent of the pattern is along the X axis.

The following additional options are displayed on the dialog if the object you have selected to produce the pattern is a feature. They will not be displayed if the selected object just contains solid features.

Remove feature — Controls how the original feature is handled when the pattern is created. If *ON,* when the pattern is created, the original feature is absorbed by the pattern and they become one feature. In the solid tree, the feature icon is replaced by a pattern icon

If *OFF* when the pattern is created, the pattern is created as a separate feature and both the original feature and the pattern feature can be edited separately. The solid tree will show both the feature icon and a pattern icon.

This option is not displayed on the dialog if you are editing an existing pattern feature.

Edit Definition — This option is only available when you are editing an existing pattern feature. When you select this option, you can edit the pattern feature using the appropriate dialog.

OK — This creates the pattern and closes the dialog.

Pattern Plane dialog

This dialog is used to set the coordinates for the origin of where the pattern is positioned.
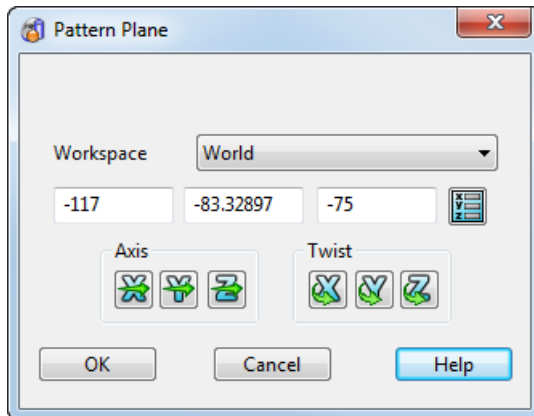
These options allow you to:

- Define the workspace in which to edit the workplane for the pattern.

- Move the workplane's origin.

- Change the direction of its axes.

- Twist it about its axes.

Use the **Relationship** button to define the relative position of the pattern with respect to the solid.

Edit > General Edits > Project

This projects the following objects onto a plane at a particular point along an axis:

- wireframe objects

- points on curves, composite curves and surface curves

- whole laterals and longitudinals on surfaces

To project objects:

1 Select some objects.

2 Click the **General Edits** flyout to display the **General Edits** toolbar.

3 Click **Project items onto plane** (*General Edits toolbar*).
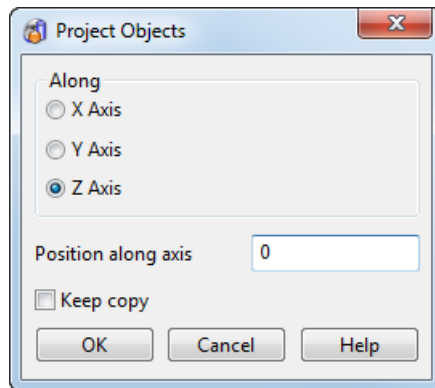
4 Select the axis you want to project along.

5 In the **Position along axis** box, type the coordinate of plane along the axis.

6 If you want to keep a copy of the original, select **Keep copy**.

7 Click **OK** to project the objects onto the defined plane.

Edit > General Edits > Project Point

You can also project points onto a surface or a mesh using the **General edits** toolbar.

To project points along the principal axis onto surfaces

1 Click the **General Edits** flyout to display the **General Edits** toolbar.

2 Ensure the principal axis of the current workspace is pointing in the correct direction.

3 Select points and surfaces.

4 Click **Project points onto surface** (*General Edits toolbar*).
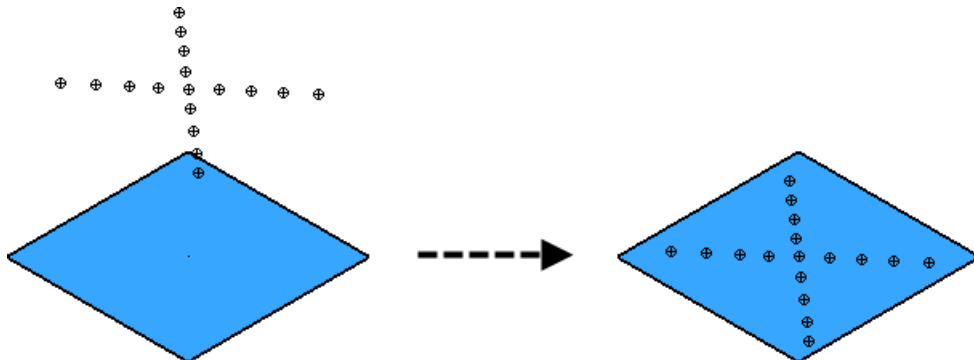
Edit > General Edits > Morph objects

Select this option to morph the object you are editing.

1 Ensure that you have an active solid.

2 With the **General Edits** toolbar displayed, click to display the **Morphing** dialog.

3 Use the **Morphing** dialog to edit the solid.

Edit > General Edits > Morph to Cloud or Mesh

Select this option to morph a surface or solid model, with respect to a separate cloud or mesh.

Before morph
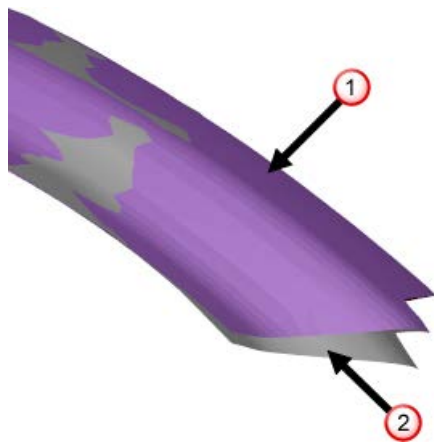
After morph
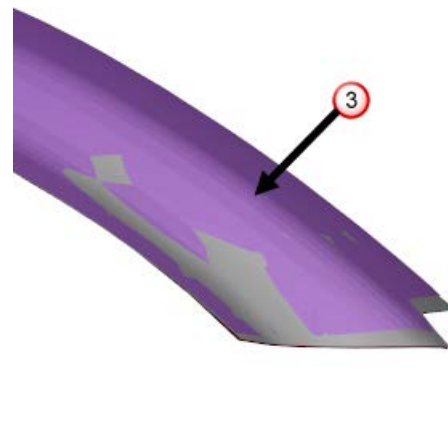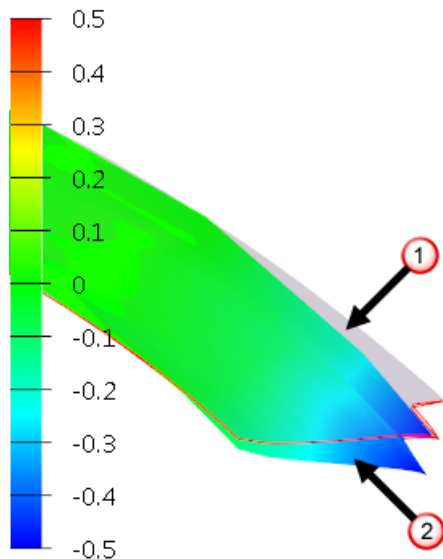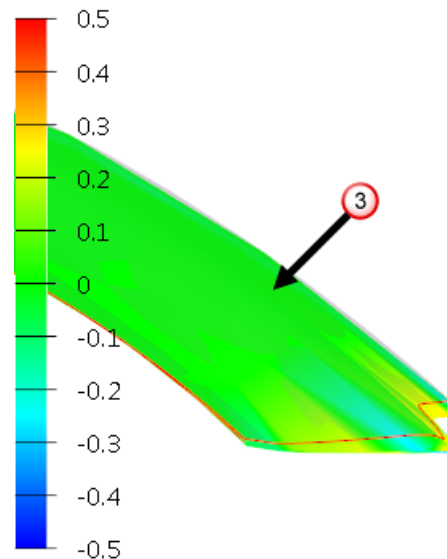
① — Mesh measurement data.

② — Solid model.

③ — Model morphed towards data.

To morph to a cloud or mesh:

1 Ensure you have a solid or surface visible.

2 With the **General Edits** toolbar displayed, click to display the **Morph to Cloud/Mesh** dialog.

3 Use the **Morph to Cloud/Mesh** dialog (see page 106) to edit the object.

Morph to Cloud or Mesh dialog

The **Morph to Cloud/Mesh** dialog is used to morph solids or surfaces with respect to a point cloud or mesh.
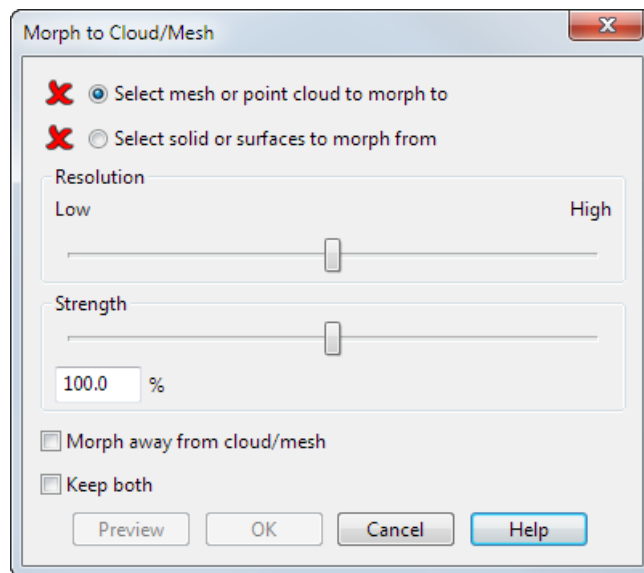
Use the following options on the dialog to modify the morph:

- **Select mesh or point cloud to morph to** — Select this option and then select the reference cloud or mesh to morph to. When an appropriate cloud or mesh is selected the ✗ changes to a ✔.

- **Select solid or surfaces to morph from** — Select this option and then select the solid or surface model that you want to perform the morph on. When an appropriate solid or surface is selected the ✗ changes to a ✔.

- **Resolution** — Use the slider to control the accuracy of the morph.

  At a low resolution the morph is faster, however small details are smoothed out. At a high resolution the morph is slower, but small details are captured.

- **Strength** — Use the slider, or enter a value to control the morph movement.

At **100%** the morph moves the solid or surface up to the mesh or cloud. If desirable you can move the solid or surface by a smaller or larger amount. For example, at **50%** the CAD geometry is morphed half way to the measurement data.

- **Morph away from cloud/mesh** — Select this option to apply an inverse morph, and move the CAD geometry in the opposite direction to the measurement data. This allows you to compensate for artefacts in the manufacturing process, such as springback.

- **Keep both** — Select this option to keep both of the CAD models, before and after the morph.

- **Preview** — Click this to display a preview of the morph.

Edit > General Edits > Sculpt Meshes

Use this option to change the shape of meshes and solids.

Edit > Surface and Curve Edits

Select the appropriate option from the flyout to display the edit options for surfaces and curves.

Surface Edits	▶
Common Edits	▶
Curve Edits	▶
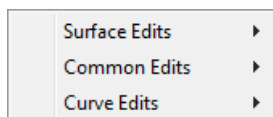
Edit > Surface and Curve Edits > Surface edits

Use this option to display options used to edit surfaces.

Select Curves/Points	▶
Add Curves	▶
Delete Curve	
Edit Curves	▶
Break Surface	
✓ Show Trimmed Surface	
Reverse Surface Curve	
Swap Curves	
Define Morph	
Edit Morph	
Fit to Wireframe Constraints	
Active Direction	▶
Nurb Edit	
Selected Curves Graphical Handles	▶
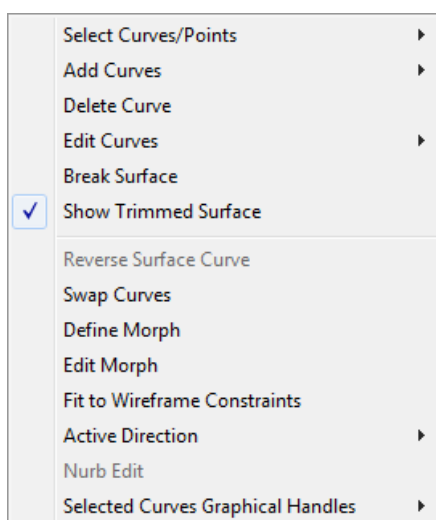
For further details, see Editing Power Surfaces and Editing a surface using the toolbar.

Edit > Surface and Curve Edits > Curve edits

This displays options used to edit curves.

Select Points...
Add Point...
Delete Point
Show Labels
Show Curvature Combs
Curve Point Adjustment ▶
Curve Point Reordering ▶
Curve Type ▶
2D Curve Editor
2D Edits ▶
Apply Smoothing to Point Edits
✓ Keep Straightness of Spans
Delete Dependencies
Free Magnitudes
Free Tangents and Magnitudes
Smooth
Corner
✓ Straight_2D
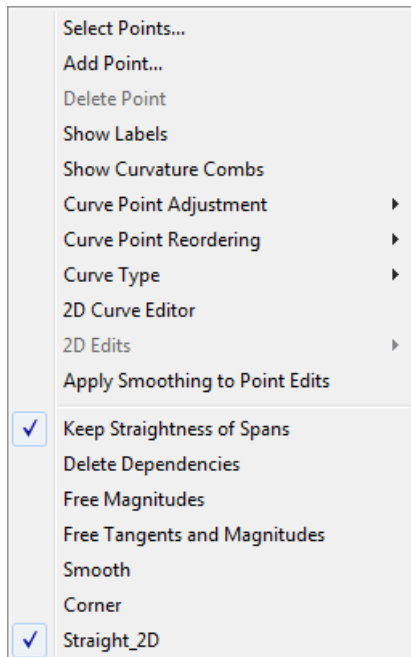
For further details, see Editing a curve.

Edit > Surface and Curve Edits > Common edits

This displays edit options that are common to editing surfaces and curves.

Reverse
Close
Renumber
Spline
Append
Rename...
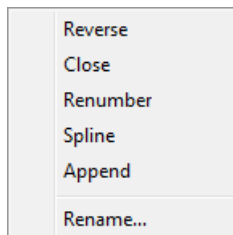
For further details, see:

- Editing a curve .
- Editing a Power Surface

Edit > Solid Edits

Use the options on this submenu to edit the model. The options correspond to the buttons on the **Solid Edit** toolbar.
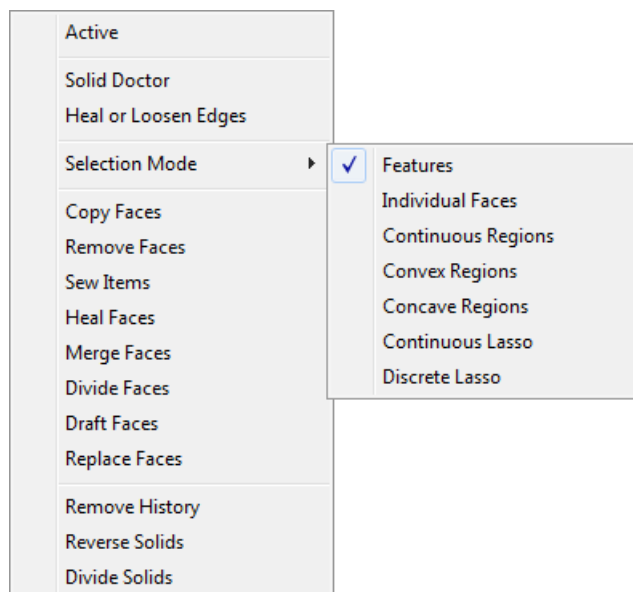
Active — Select this to make the selected solid active.

Solid Doctor — Select this to find and fix faults using the Solid Doctor.

Heal or Loosen Edges — Select this to heal or loosen edges in the solid using the **Solid Doctor Healing** dialog.

Selection Mode:

- **Features** — Select this and click on the solid to select features.

- **Individual Faces** — Select this and click and individual faces on a solid to select them. Use this option with box selection to select faces that are completely inside the selection box.

- **Continuous Regions** — Select this to highlight faces that are continuous with the selected face

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

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

- **Continious Lasso** — Select this 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 tos elect 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 Faces — Select this and select one or more faces to create surface copies.

Remove Faces — Select this and select one or more faces to create surface copies and remove them from the solid.

Sew Items — Select this to display the **Sew** dialog. Select one or more surfaces or solids and click **OK**.

Heal Faces — Select this to heal and remove faces from the solid.

Merge Faces — Select this to merge selected faces of a solid into one face.

Divide Faces — Select this to divide the faces of the selected solid using wireframe.

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

Replace Faces — Select this to select faces of a solid and replace them with other existing faces or surfaces.

Remove History — Select this to remove the history of the solid from the solid feature tree.

Reverse Solids — Select this to reverse the faces of the solid.

Divide Solids — Select this to divide an existing solid into separate solids.

Edit > Active dimensions

You can use **Active dimensions** to alter the geometry of a composite curve by changing the dimension value.
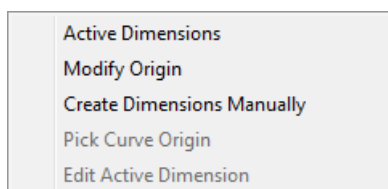
1   With a composite curve selected, select **Active dimensions** from the menu.

    The appropriate options for editing the curve are displayed on the **Surface/Curve Edit** toolbar.

It is the same as selecting **Edit the curve using active dimensions**
 (*Curve Editing flyout*).
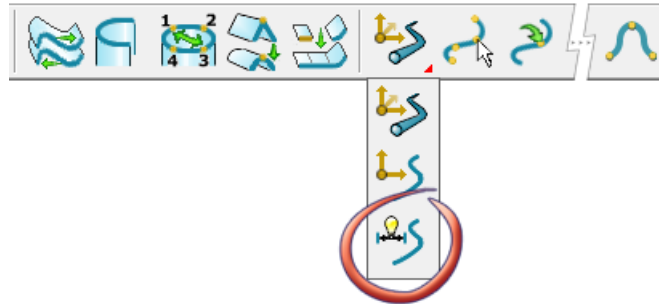
2   Double-click on the **Dimension value** you wish to change and enter a new value. The geometry will be modified accordingly.

For further details, see Editing using active dimensions.

Edit > Fillet Corner

This is used to edit the corner of a fillet.

For further details, see How do I edit a corner fillet?

View menu
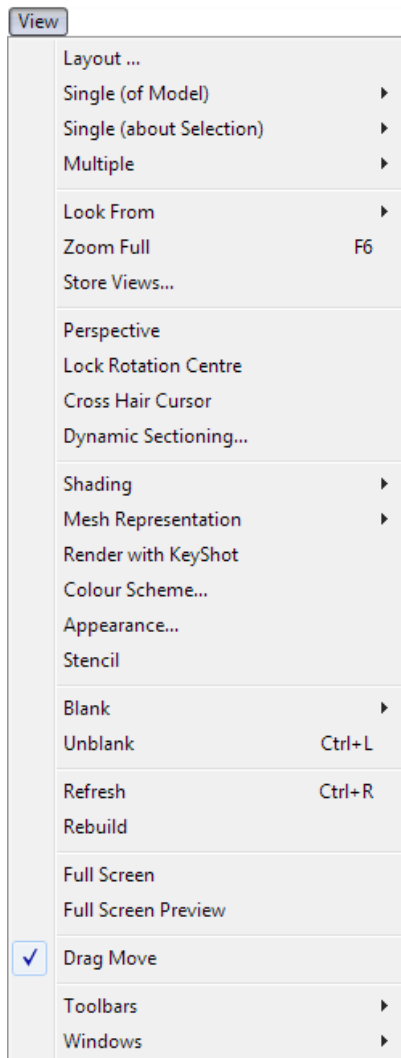
PowerSHAPE initially displays with a view from above (that is, down the global Z-axis). With the middle mouse button and certain keyboard keys, you can turn and roll the view (rotate), zoom in or out and pan across the view.

The **View** menu lets you to choose from a selection of single and multiple views shown from specific angles. It also includes the **View Edit** dialog which allows you to create *customised* views.

You can have more than one window open for the same model, any changes you make to objects within the model are shown in all the open windows.

Remember that any changes to the view affect only the active window. This allows you to maintain views from various angles in separate windows.

PowerSHAPE is designed to provide fast, intuitive view controls. These are all centred around the middle mouse button.

For further details, see Middle mouse button in Modelling Concepts.

View > Layout

Use this option to edit the way your model is displayed in the graphics window.

1 From the **View** menu, select **Layout** to display the **View Edit** dialog (see page 113).

2 Use the dialog to change how your model is displayed in the graphics window.

View Edit dialog

This dialog lets you to edit how your model is displayed in the graphics window.
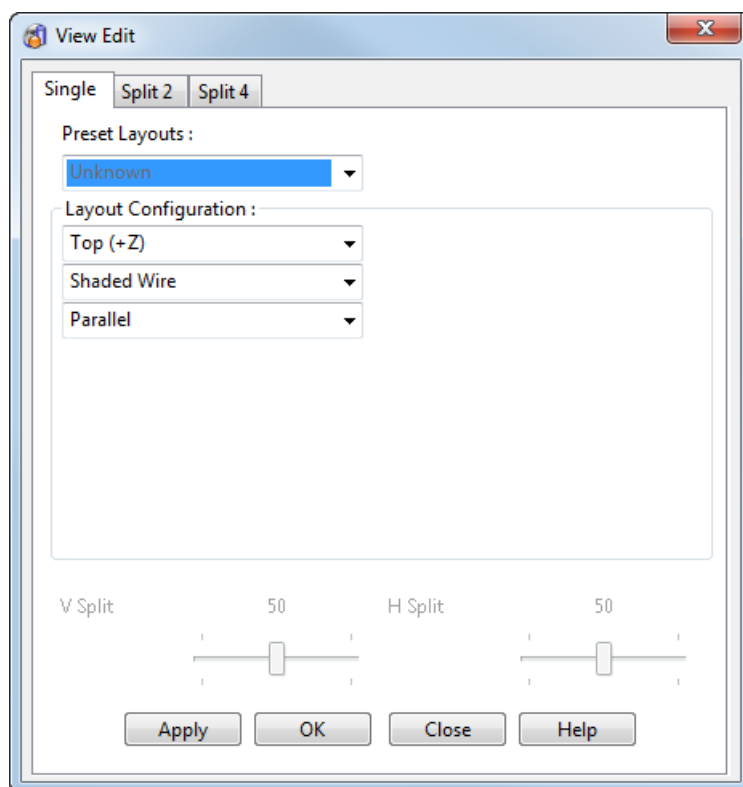
Preset Layout — You can choose one of the following options from this drop-down menu:

Top looks down on the Z-axis.

Bottom looks up on the Z-axis.

Left and **Right** look along the X-axis.

Front and **Back** look along the Y-axis.

Iso offers four isometric options.

Third Angle and **First Angle** provide views where the model appears in plan and various elevations within the same window.

Unknown is shown when the current window contains a custom view layout, or has been rotated using the middle mouse to an arbitrary viewpoint.

The above options are the same as those available from the **View > Single (of model)** and **View > Multiple** menus. When you select an option, the **View Edit** dialog updates to show the configuration for the selected option.

You can choose one of the preset layouts and then change it to your requirements.

The isometric and perspective views have a default viewing angle that defines the visual depth of the model. Most people find the default 40° to be acceptable, but if required the angle can be changed having an effect similar to altering a camera lens focal length. For further details, see Perspective View Angle on the Views options (see page 121) dialog.

Layout Configuration

You can split the graphics window into 1, 2 or 4 views. Select one of the tabs:

- **Single** gives a single view.
- **Split 2** splits the view vertically into two separate views.
- **Split 4** splits the view vertically and horizontally to give four separate views.

When you select one of these tabs, the dialog updates to show the options available for each view. For example, when you select **Split 2** tab, the dialog updates to show options for the two views:
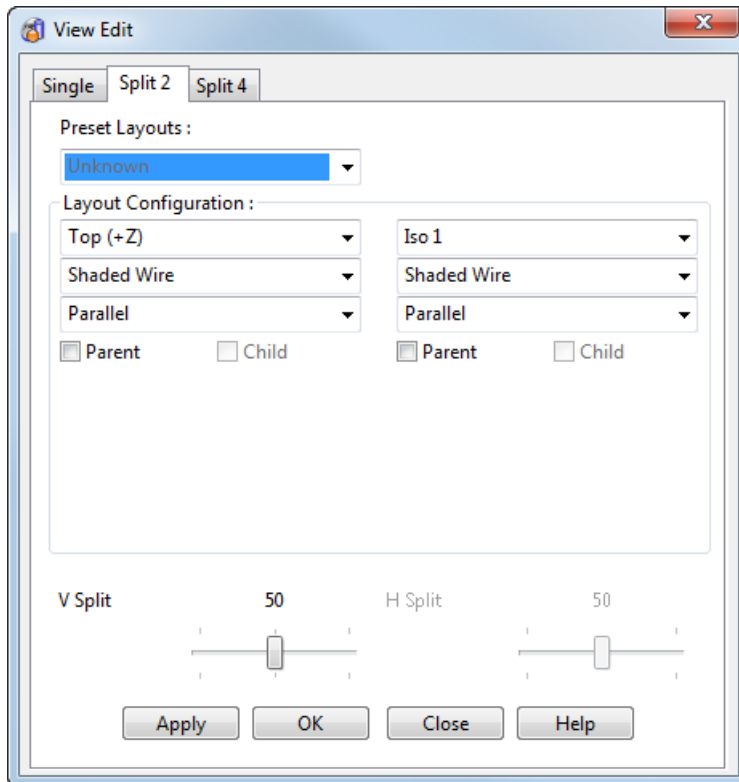
For each view, you can set:

- the direction of the view (using the options on the option menu given below);

```
Top (+Z)
Bottom (-Z)
Left (-X)
True Left
Right (+X)
True Right
Front (-Y)
Back (+Y)
True Back
Iso 1
Iso 2
Iso 3
Iso 4
```
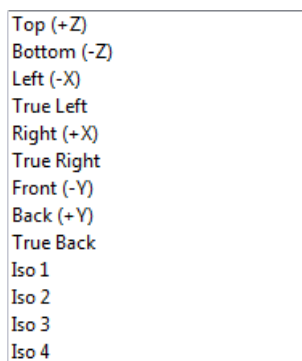
- how the surfaces and solids are displayed;

```
Wireframe
Shaded Wire
Shaded
Dynamic Hidden Line
Transparent Shaded Wire
Transparent Shaded
```
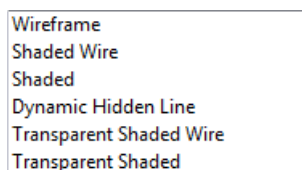
- whether the view is parallel or perspective.

```
Parallel
Perspective
```

Parent and **Child** — (Only available for **Split 2** and **Split 4** views.) These control how the views rotate. When you rotate a parent view, the child also rotates in an appropriate manner.

V-Split — Adjust the position of the vertical split between views.

H-Split — Adjust the position of the horizontal split between views.

Apply — Display your choice in the active window, but keep the dialog open.

OK — Save any changes and close the dialog.

Close — Discard changes and close the dialog.

View > Single (of model)

Use the options on the submenu to set the orientation of the single view of the model.

1   From the **View** menu, select **Single (of Model)** to display the following menu:

Top (+Z)	Ctrl+5
Bottom (-Z)	Ctrl+0
Left (-X)	Ctrl+4
Right (+X)	Ctrl+6
Front (-Y)	Ctrl+2
Back (+Y)	Ctrl+8
Iso 1	Ctrl+1
Iso 2	Ctrl+3
Iso 3	Ctrl+9
Iso 4	Ctrl+7

Each view looks towards the origin, down the axis given in the brackets. For example, the **Top** view is looking at the model down the Z axis.

2   Select the view you require from the options.

The view of the model changes to the selected orientation and the view is zoomed and panned so that the whole model fits centred in the window.

Sometimes, for example when zoomed in on a feature of interest, this rescaling of the view is not desirable. Instead, you would like the feature of interest to remain fitted in the view after you change the view direction. To achieve this, select **View > Single (about selection)** (see page 116).

View > Single (about selection)

Use the options on the submenu to set the orientation of the single view of the selected objects.

1. Select your objects.

2. Select **Single (about Selection) > View** to display the following menu:

Top (+Z)	Alt+5
Bottom (-Z)	Alt+0
Left (-X)	Alt+4
Right (+X)	Alt+6
Front (-Y)	Alt+2
Back (+Y)	Alt+8
Iso 1	Alt+1
Iso 2	Alt+3
Iso 3	Alt+9
Iso 4	Alt+7

Each view looks towards the origin, down the axis given in the brackets. For example, the **Top** view is looking at the model down the Z axis.

3. Select the view you require from the options.

The view of the model changes to the selected orientation and the view is zoomed and panned so that the selected objects fit centred in the window. If no objects are selected, then the whole model fits the window.

This command is useful when you are zoomed in on a selected feature of interest and you want to view this feature from a different direction.

View > Multiple

Use the options on the submenu to provide orthographic style images, each showing three or four separate views of the model.

1. From the **View** menu, select **Multiple** to display the following menu.

Third Angle
Third Angle + Iso
First Angle
First Angle + Iso

2. Select the view you require from the options.

When an option is selected, the active window changes to show a number of views of the model within the window.

The views are linked, so, if you rotate the top left hand view, all the other views follow the rotation.

All the views can be rotated with the middle mouse button, so they may no longer remain true orthographic projections.

View > Look From

Use the options on the submenu to change the view to look from a direction relative to the current view.

1   From the **View** menu, select **Look From** to display the following menu.

Behind	
Above	
Below	
Left-side	
Right-side	
Last View	F5
Stored View	Ctrl+F5
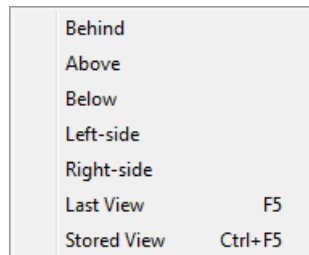
Each option changes orientation relative to the current view direction. For example, if you are looking at the top view of a model (that is down the Z axis) and you select **Above**, the view of the model changes to look down -Y axis.

2   Select the view you require from the options.

The view of the model changes to the selected orientation and the view is zoomed and panned so that the whole model fits centred in the window.

View > Look From > Last View

This displays the view before it was changed (for example, by rotating or zooming) providing a toggle between the current view and the previous view.

1   From the **View** menu, select **Last View**. The previous view is displayed in the graphics window in which the view was last changed.

2   Select **Last View** again from the **View** menu, and the original view is shown in the graphics window.

View > Look From > Stored View

Use this option to restore the previously stored view.

1   Select **View > Stored View**.

The last view that you stored previously using **Store View** will be displayed.

View > Zoom Full

Use this option to fit the view of the model to the graphics window.

1   Select **View > Zoom full**.

The whole model fills the active graphics window.

View > Store Views

Use this option to store a view to use at a later date.

1 Rotate your model until you have the view that you wish to store.

2 Select **View > Store Views** to display the **Stored View** dialog.
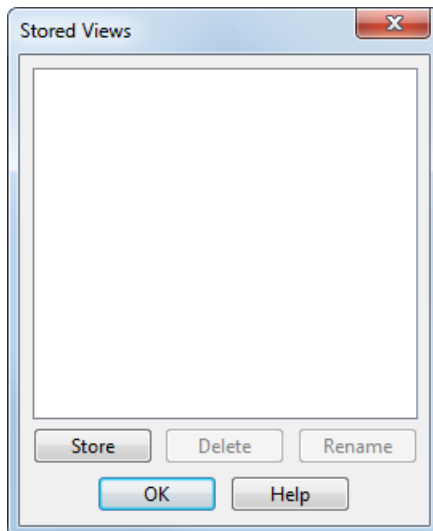
3 Click **Store** and *View 1* will be added to the list of views.

4 Click **OK**.

5 Rename this view by selecting **Rename** and using the **Rename Stored View** dialog
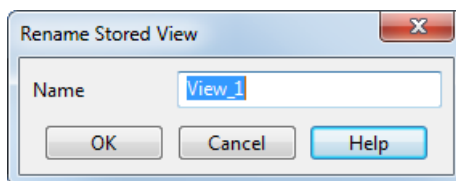
View > Perspective

Use this option to toggle the view from parallel to perspective.

By default, the view is set to parallel ①. Objects are drawn to the same scale regardless of whether they are near or far away.

A perspective view adds depth to the model. Objects are drawn as they appear to the human eye so that objects which are further away appear smaller than those which are close.
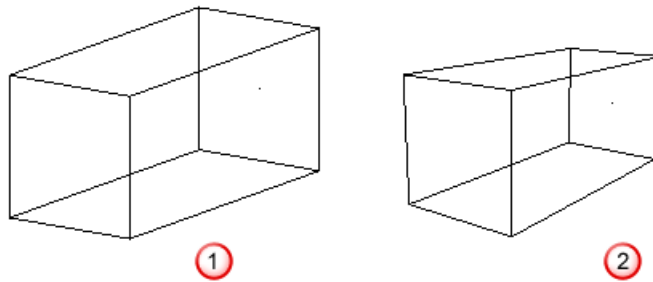
View > Lock Rotation Centre

This locks the rotation of the view to the centre of the current selection.

1 Select one or more objects.

2 Select **Lock Rotation Centre** from the **Views** menu

3 Rotate the model. It will rotate around the centre of your selection.

4 Change your selection. The view will still rotate around the centre of the original selection.

5 If you unlock the rotation centre by selecting **Lock Rotation Centre** again, the view will revert to rotating as defined by the **Rotate Around** option on the **Views** page of the **Options** dialog.

View > Cross Hair Cursor

Use this option to change the display of the cursor to cross hairs.

Select this menu option to toggle the cross hair cursor *ON* and *OFF*.
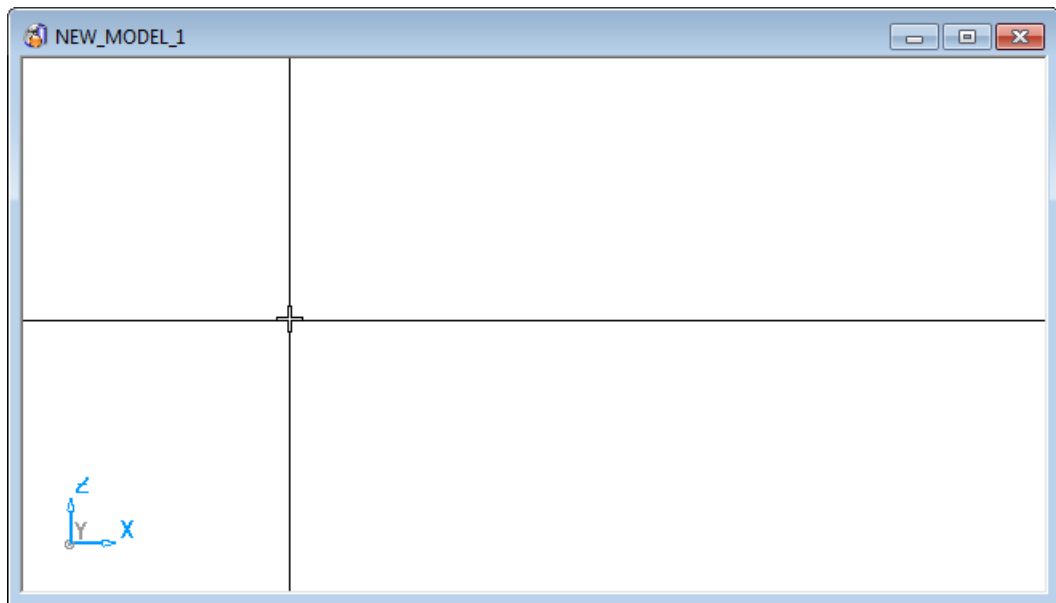
View > Dynamic Sectioning

Use this option to take sections through the front and back of a model using the **Dynamic Sectioning** dialog (see page 43)

View > Shading

Use the options on the submenu to change the appearance of the model.

1 From the **View** menu, select **Shading** to display the following menu:

Wireframe	F2
Shaded	F3
Transparent Shaded	F4
Transparent Shaded Wire	F10
Shaded Wire	F11
Dynamic Hidden Line	F12
Enhanced Shading	
✓ Inside Material	
Use Wireframe Colour	

Each option changes the appearance of the model when it is displayed.

2 Select the shading option you require.

View > Shading > Wireframe

This shows surfaces and solids as wireframe constructions.

1 From the **View** menu, select **Wireframe.** This changes any surfaces and solids that are shaded or have hidden lines to be shown as wireframe.

This is the same as clicking the **Wireframe view** button on the **Views** toolbar.

All views in the currently selected window as wireframe are displayed. The objects are fully transparent with all the lines visible. This option requires the least processing power, so image rotation is fast and smooth.

View > Shading > Shaded

Select this option to use colour shading to make surfaces appear solid.

1 Display the surfaces on the screen you want to shade.

2 From the **View** menu, click on **Shaded**. The surfaces will now be shaded making them appear solid.

This is the same as clicking the **Shaded view** button on the **Views** toolbar.

When you first select **View > Shaded**, the view becomes shaded. If you select it again, the view becomes shaded wire. Pressing it again, toggles the view between shaded and shaded wire.

The surface colours and lighting effects highlight any bumps, creases or ripples in a surface. This allows you to better visualise how the final product will look. The effects change as you rotate the image.

Rotations can be slow on complex models, because shaded images require considerable processing power.

The shading is achieved by filling the surface with shaded triangles of varying colour and brightness to represent the solid surface. PowerSHAPE automatically chooses triangle sizes which best suit the model being shaded. However, where surfaces meet, gaps can appear between the triangles. This tendency can be minimised by changing the shaded surface options.

For further details, see Changing the material of selected surfaces and solids.

On operating systems that do not support hardware graphics acceleration, redraws are slow during rotations and shading of complex models can take a considerable time.

For further details see Shading options (see page 97).

View > Shading > Transparent Shaded

This displays transparent colour shading on surfaces and solids.

1 Display the surfaces and solids you want to shade on the screen.

2 From the **View** menu, select **Transparent Shaded**.

You can also use the **Toggle Transparent Views** button on the **Views** toolbar. This button toggles the view from **Transparent Shaded Wire** to **Transparent Shaded**.

The surfaces and solids are now shaded using transparent material.
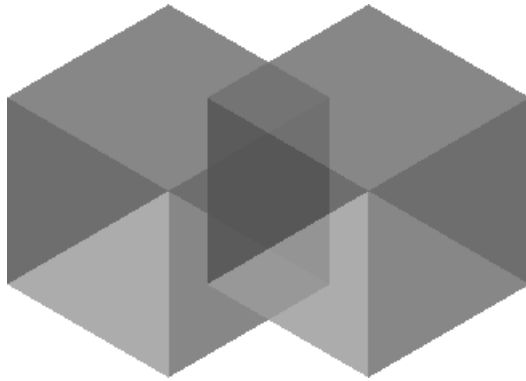
View > Shading > Transparent Shaded Wire

This displays transparent colour shading on surfaces and solids, but the wireframe is not hidden.

1   Display the surfaces and solids you want to shade on the screen.

2   From the **View** menu, select **Transparent Shaded Wire**.

You can also use the **Toggle Transparent Views** button on the **Views** toolbar. This button toggles the view from **Transparent Shaded Wire** to **Transparent Shaded.**

The surfaces and solids are now shaded using transparent material.
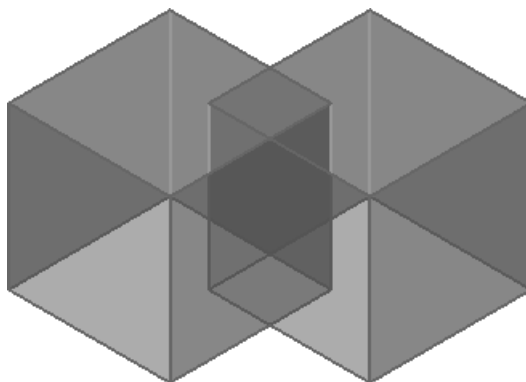
View > Shading > Shaded Wire

This uses colour shading and lighting effects to make surfaces appear solid, but the wireframe is not hidden.

This shades your surfaces and draws the wireframe on top.

To use the Shaded Wire option:

1 Display the surfaces you want to shade on the screen.

2 From the **View** menu, select **Shaded Wire**.

This is the same as clicking the **Shaded and wireframe** button on the **Views** toolbar.

The surfaces will now be shaded with the wireframe appearing on top.

On operating systems which do not support hardware graphics acceleration, redraws are slow during rotations and shading of complex models can take a considerable time.

View > Shading > Dynamic Hidden Line

This option shows the wireframe view but all lines which fall behind surfaces are hidden.

To use the Dynamic Hidden Line option:

1 Display the surfaces you want to process on the screen.

2 From the **View** menu, select **Dynamic Hidden Line**.

This is the same as clicking the **Hidden Wire View** button on the **Views** toolbar.

The surfaces will now show only those lines which would be visible when viewed from the screen. When the model is rotated, these lines are removed as they go out of view and redrawn as they come into view.
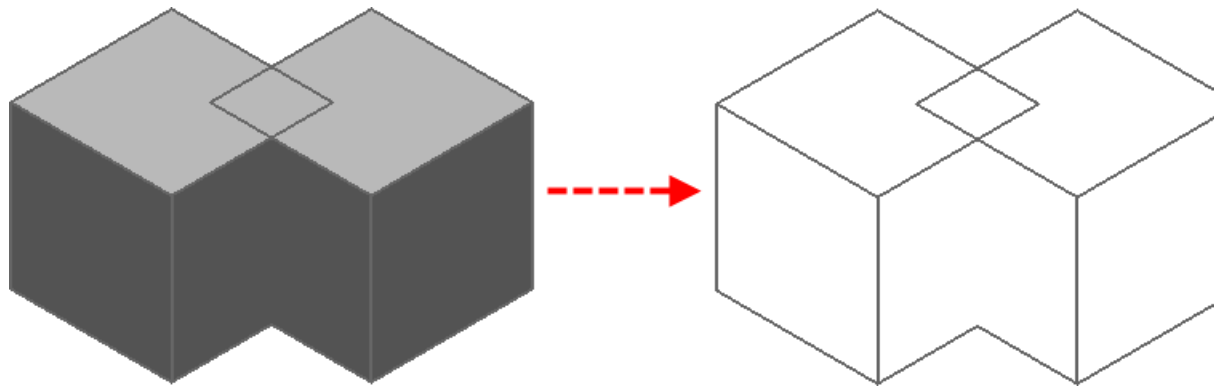
View > Shading > Enhanced Shaded

Use this option to shade the selected object in a reflected environment, producing a more realistic image.

This is the same as clicking the **Enhanced Shading** button on the **Views** toolbar.

The following options are automatically turned on:

- **View > Perspective** to produce a more realistic image.
- **View Spinning** on the **Tools > Options > Views** dialog.
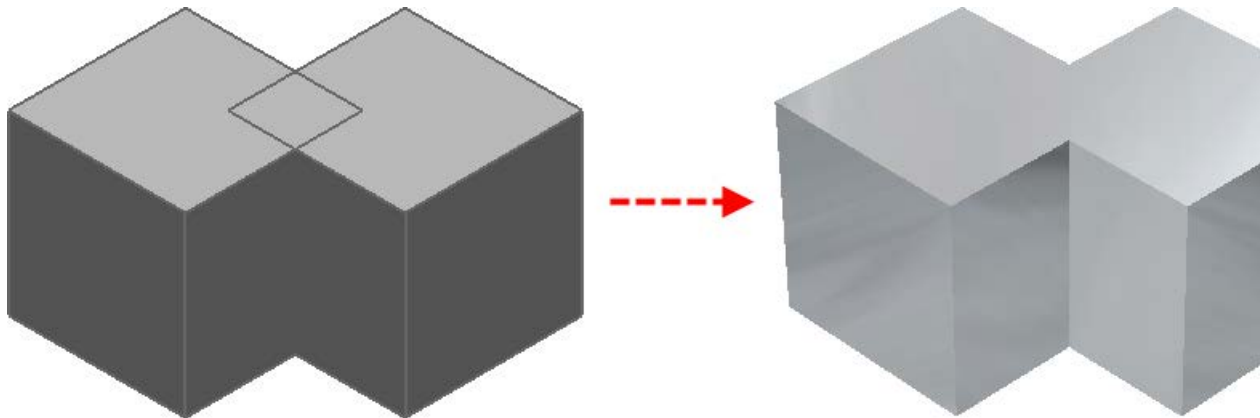
View > Shading > Inside Material

This toggles the inside material to match the outside one when shading.

View > Shading > Use wireframe colour

Select this option to shade surfaces using their wireframe colour. This functionality is also available on the **Materials selection** dialog.

View > Render

Use this option to render a model to create a photo-realistic image.

Before rendering a scene, make sure the solids and surfaces have the correct materials assigned.

> *Rendering a scene is a very easy operation, but it can be time-consuming.*

1 Orientate and position your model as required.

2 Click **Render** to render the image. The image is rendered using KeyShot in a separate window.

Rendering using the Views toolbar

1 Ensure your model is shaded. Select the **Shaded** button on the Views toolbar.

2 Click the **Render with KeyShot** button on the Views toolbar.

   A new window is opened which contains the image.

3 You can go back to your original model using the **Window** menu.

4 KeyShot is the software that produces photo-realistic ray-traced images. KeyShot needs to be installed separate to Delcam software.

If it is not already installed, download and install (free of charge) by clicking **KeyShot (http://www.delcam.com/keyshot/download/)** or by running the installation again. Ensure you install the appropriate version (32-bit or 64-bit) according to the type of computer you are using.

View > Mesh Representation

Use the options on this submenu to specify the display mode for triangles in a mesh.

✓	No Edges
	Solid with Edges
	Solid with Nodes
	Hollow with Edges
	Hollow with Nodes
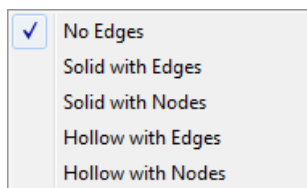
Use the following options to specify the way the mesh is displayed:

- **No Edges** displays the mesh as open edges only. Selecting this option is the same as clicking on the **Views** toolbar.

- **Solid with Edges** displays the mesh as open and interior edges over shaded triangles. Selecting this option is the same as clicking on the **Views** toolbar.
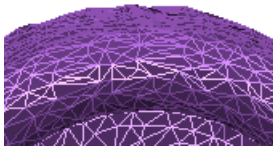
- **Solid with Nodes** displays the mesh as nodes and open edges over shaded triangles. Selecting this option is the same as clicking on the **Views** toolbar.

- **Hollow with Edges** displays the mesh as open and interior edges over hollow triangles. Selecting this option is the same as clicking on the **Views** toolbar.
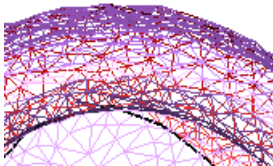
- **Hollow with Nodes** displays the mesh as nodes and open edges over hollow triangles. Selecting this option is the same as clicking on the **Views** toolbar.
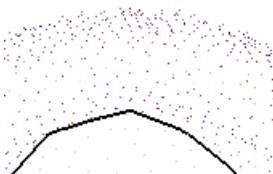
View > Colour Scheme

Use this option to set the colour scheme for all views.

Selecting this option displays the **Properties** dialog where you define the colour scheme that is used.

View > Appearance

Use this option to set the background of the modelling window and the environment which is reflected in a shaded model.

1 From the **View** menu, select **Appearance**.

2 Use the **View Appearance** dialog (see page 129) to set the options.

View Appearance dialog

This sets the background of the modelling window and the environment that is reflected in a shaded model.
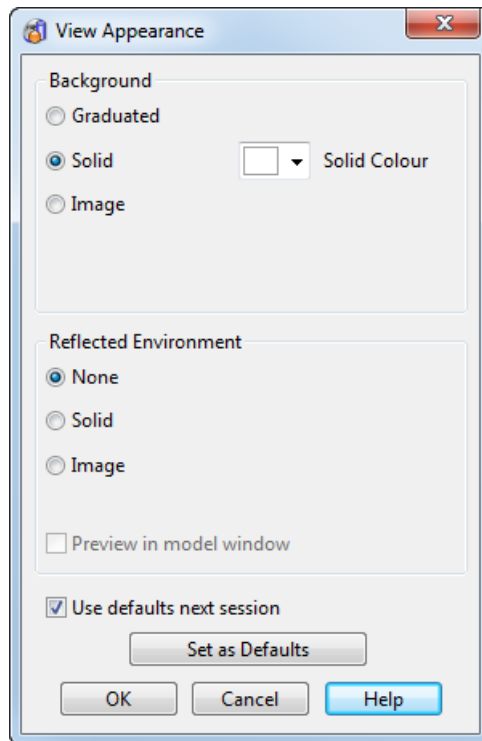
Background

These options set the background of the modelling window. You can set the background to:

Graduated — Gradually change the background from one colour to another. Select this option and then select the top and bottom colours using the two drop-down colour selectors.

> **Top Colour** — This is the colour displayed at the top of the graduated background.

> **Bottom Colour** — This is the colour displayed at the bottom of the graduated background.

Solid — A single colour is used for the background. Select this option and then select the colour from the drop-down menu.

Image — Select this option to display the **Image Selector**. Use this dialog to select an image to display in the background. For further details, see Image Selector dialog.

When you have selected an image, click **OK**. The **Background Image Properties** dialog is displayed. This contains the controls to fit the image on the screen. For further details, see Background Image Properties dialog (see page 132).

The selected image and the **Properties** button appear on the **View Appearance** dialog. Click the **Properties** button to display the **Background Image Properties** dialog.

The default background ensures that model objects do not merge into the background. The background options are not intended for normal modelling use, but:

- to allow colour shaded images to be generated with good colour contrast to the shading colour.

- to add a background image for tracing.

> *When you choose different colours, be aware that objects of the same colour as the background will be invisible. If you need to change the colour of your objects, see Setting up a colour scheme for graphics.*

Reflected Environment

These options set an environment which is reflected in the shaded model. The following options are available:

None — No reflected environment is set.

Solid — A solid colour is used. This is only visible in PowerSHAPE Render. Select this option and then select a solid colour.

Image — An image is used as the reflected environment. Select this option to display the **Image Selector** dialog where you can select an image. For further details, see Image Selector dialog.

When an image is selected, it is reflected in shiny materials in the shaded model. Shiny materials are those which show reflections in the **Material Selection** dialog, and include Metal, Cast Metal, Mirror (in Coated Metal), Glass and Metal25.

The amount by which a material reflects the environment is controlled by its polish. A value of 15 gives a small amount of reflection, and 50 gives a mirror-like material.

Reflecting images can seriously slow your graphics card, although most modern cards can handle it. If it is too slow, then either do not use the **Image** option or turn off the **Preview in model window** option. If the **Preview in the model window** option is off, the image only appears in the rendered view without slowing down the model window too much.

Use defaults next session — If you do not want the settings to use default, turn this option *OFF*. PowerSHAPE will then use the program's original settings. To change the default colour scheme, see the Properties page of the **Tools > Options** dialog for further details.

Set as Defaults — Set the current view appearance for the rest of the session. The settings are also stored in the registry so that they will be used in the next session.

OK — Apply the changes. If you set the reflected environment options, you will need to render the model to see the results.

Cancel — Discard any changes and close the dialog.

*To cancel all your view appearance settings, choose the default colour scheme from the **Properties** page of the **Options** dialog.*

Background Image Properties dialog

This dialog contains the controls to fit the image on the screen.
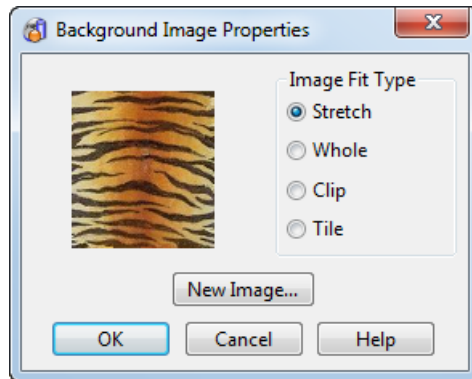
Image Fit Type

These options determine how the image is displayed in the graphics window.

Stretch — The image is stretched to fit the modelling window.

Whole — Display the whole image on the screen and preserves the aspect ratio of the image. It is most useful for tracing.

Clip — Clip the image to the view and preserves the aspect ratio of the image.

Tile — This repeats and tiles the image to cover the entire area of the view.

New Image... — Displays the **Image Selector** dialog so that you can select another image. For further details, see Image Selector dialog

OK — Accept the changes and close the dialog.

Cancel — Discard any changes and close the dialog.

View > Stencil

You can transfer a 2D image onto a surface using the **Stencil** functionality. This displays the image and lets you use the standard curve creation tool to trace the image onto the surface. This is particularly useful for tracing a design onto a shoe last.

Use the following sections to explore the **Stencil** functionality:

- Stencil toolbar (see page 134)
- Creating a stencil (see page 133)
- Editing a stencil (see page 135)
- Hiding and displaying a stencil (see page 137)

Creating stencils

Use a stencil to transfer a sketch to an object.

To create a stencil:

1 Ensure that you have an image file to use as the stencil. The image file can have one of the following extensions:

   jpg, bmp, png, tiff, obj

2 Select the object to have the stencil applied.

3 Select **View > Stencil** to display the **Stencil** toolbar.

4 Click to display the **Load Image** dialog

5 Select an image file using the **Load Image** dialog.

6 Click **Open** to display the image file on top of the model.
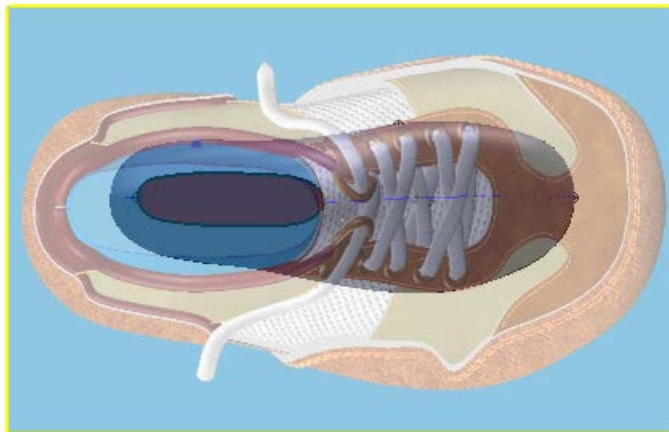
7 Click **True Size** to show the real size of the image.
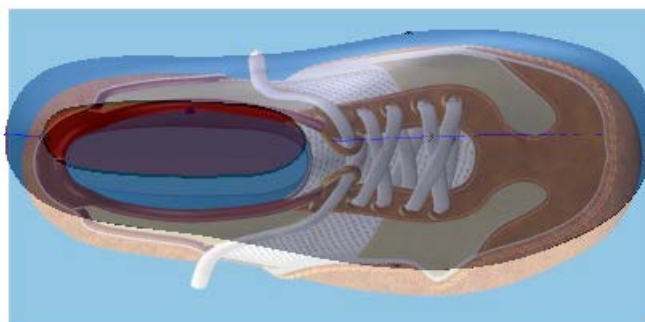
8 Use the Editing tools (see page 136) to create the stencil you require.
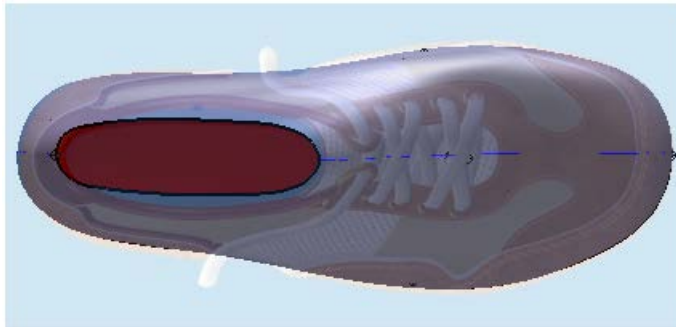
9 Save the model. Any stencils that you have created are saved with the model.

10 An additional **Stencil** flyout is added to the **Views** toolbar. The flyout includes a separate button for each stencil that has been saved with the model.

11 When you have created the required stencil, use wireframe functionality to sketch through the transparent stencil onto the last. This transfers a 2D sketch onto a 3D last.

Creating a stencil the size of your image

You can create a **Stencil** that is the same size as the image you are using.

1 Create an image to sketch onto.

2 Create and select a workplane that is to be the origin of your image.

3 Create a stencil using the image. The surface to which the image is applied takes the size of the image. The image size is retained.

Stencil toolbar

1 Select **View > Stencil** to display the **Stencil** toolbar.

2 Click to load an image file. The other options become available on the toolbar.

The following buttons are displayed on the stencil toolbar:

![icon] Display the **Load Image** dialog.

![icon] Delete the selected stencil from the drop-down list.

![icon] Make the stencil **True size**; the stencil size will be determined by the attributes of the image (width and height).

![icon] **Wrap** the stencil onto the last or selected object (surface. solid or mesh).

![icon] **Edit** the stencil using the tools on the Editing section of the toolbar (see page 136).

![icon] Flip the stencil horizontally.

![icon] Flip the stencil vertically.

![icon] Display the Editing section of the toolbar (see page 136).

![slider] Adjust the **Transparency** of the stencil.

Editing stencils

You can edit a stencil in the following ways:

- Adjust the transparency level of the stencil.
- Adjust the horizontal and vertical location of the stencil.
- Rotate the stencil.
- Scale the stencil image.
- Mirror the stencil image.
- Delete the stencil (see page 137).

To edit a stencil:

1 Open a model that includes one or more stencils.

2 If the stencil is not displayed, click **View > Stencil.**

3 Select **Edit.** If necessary, click to display the **Stencil Editing** toolbar (see page 136).

4 Use the buttons on the **Stencil Editing** toolbar to position and size the stencil as required. Alternatively, use one of the following key combinations:

Move the stencil - **Shift+middle mouse button**.

Scale the stencil - **Ctrl+middle mouse button**.

Rotate the stencil - **R+scroll middle mouse button**.

5 Save the model. The updated stencils will also be saved.

Stencil toolbar (editing)

With the **Stencil** toolbar displayed:

1 Click .

2 If the Editing section of the toolbar is hidden, click to display the following:
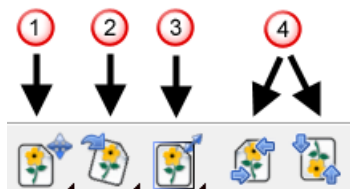
3 Use the flyouts and buttons to edit the stencil:

① **Move**

 Move in both directions.

 Move horizontally.

 Move vertically

② **Rotate**

 Rotate 1°.

 Rotate 5°.

 Rotate 90°.

③ **Scale**

 Scale in both directions.

 Scale horizontally.

 Scale vertically.

 Scale, maintaining aspect ratio.

④ **Flip**

 Flip horizontally.

 Flip vertically.

Deleting a stencil

Delete a stencil in one of the following ways:

- Delete the currently selected stencil by clicking on the **Stencil** toolbar. The active stencil is deleted, the next stencil is displayed and made active.

- Delete a stencil from the drop-down list as follows:

  1 Select the stencil to be deleted.
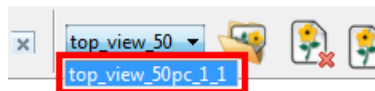
  2 Click **Delete Stencil** . The selected stencil is removed from the drop-down list.

Hiding a stencil

Hide a stencil that is displayed on a model by clicking on (*Views toolbar*)

Displaying stencils on a model

When you open a model that includes stencils, is automatically included on the **Views** menu.

1 Click .

2 Select the required stencil from the flyout.

The relevant view of the model is displayed with the selected stencil superimposed.

When you view the model for each of the stencils, the image is projected over the selected model. If you twiddle or select an ISO view, the stencil will stay in the original position.

View > Blank

Use these options to set the parts of the model to blank (temporarily undraw).

1 From the **View** menu, select **Blank** to display the following menu:

Blank Selected	Ctrl+J
Blank Except	Ctrl+K
Blank Toggle	Ctrl+Y

2 Select the option you require:

Blank Selected (see page 138)

Blank Except (see page 139)

Blank Toggle (see page 139)

View > Blank Selected

This allows you to temporarily remove selected objects from the screen.

1 Select the objects on the screen that you want to blank.

2 Click **Blank Selected** (*Views toolbar*). The selected objects are removed from the screen.

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

This option allow you to remove **selected** objects from the screen. They are not deleted, but merely hidden and can be unblanked when required. This allows you to concentrate on the objects you need to work on without being hindered by others close by.

View > Blank Except

This option blanks all objects on the screen apart from those which are selected.

1   Select the objects you want to remain on the screen.

2   Click **View > Blank > Blank Except**.

3   All those which are not selected are blanked from the screen.

This allows you to remove from view, everything **except** the selected objects. They are not deleted, but merely hidden and can be unblanked when required. This allows you to concentrate on the objects you need to work on without being hindered by others close by.

View > Blank Toggle

This option blanks all visible objects and unblanks all previously blanked objects.

1   Click **View > Blank > Blank Toggle**.

All visible objects become blanked and previously blanked objects become visible.

View > Unblank

This option displays all objects which were previously blanked. This option is the same as clicking (*Views toolbar*)

1   Click **Unblank** . If any objects have been blanked from the screen, they become visible.

When this option is used, all objects that were hidden with either Blank option are displayed selected. If you only want to unblank a few, deselect them (hold down **Shift** while clicking on them) and re-blank the remaining selected objects with **Blank Selected** .

View > Refresh

Sometimes, pieces of the display can remain on screen after the actual objects have been moved, deleted or are no longer displayed.

Selecting **Refresh** redraws all graphics windows, cleaning up the display.

A more powerful redraw function is available by using the Rebuild (see page 140) option.

View > Rebuild

Use this option to clear the graphics window of the current model and redisplay the items using data from the database.

> *Rebuild redisplays any blanked items and does not remember selected items.*

1   From the **View** menu, click on **Rebuild**.

This clears the graphics window of the current model, recalls the definitions of the items from the database and redraws the items again.

> *Since **Rebuild** recalls the items' definitions from the database, **Rebuild** takes longer to perform than **Refresh**. We recommend that you first use **Refresh** to correct your view and if that does not work, use **Rebuild**.*

View > Full Screen

Select **Full Screen** to increase the visible modelling area by removing the banner and the menu bar from the top of the display. Menus can still be accessed by moving the mouse across the top of the screen area.

View > Full Screen Preview

Select **Full Screen Preview** to increase the visible modelling area by removing the banner, menu bar and all the toolbars from the display. Menus can sill be accessed by moving the mouse across the top of the screen.

View > Drag Move

Select **Drag Move** to toggle the ability to select and move and object, on or off.

- If **Drag Move** is on, you can move an object to a new position by dragging it with the cursor.

- If **Drag Move** is off, you can select an object, but not move it with the cursor.

View > Toolbars

Use the options on the submenu to control the toolbars that are displayed on the screen.

1 Select **View > Toolbars** to display the following menu:
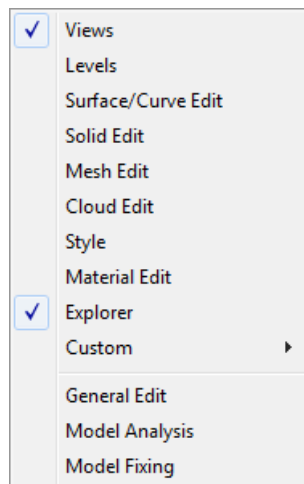
If an option is selected, the toolbar will be displayed

2 Select the option you require.

View > Toolbars > Views

This displays and removes the **Views** toolbar from the screen.

Most buttons on the **Views** toolbar are a shortcut to an option on the **View** menu. and are given below with their corresponding **View** menu option.

	View > Single (of model) > Right
	View > Single (of model) > Front
	View > Single (of model) > Top
	View > Single (of model) > Left
	View > Single (of model) > Back
	View > Single (of model) > Bottom
	View > Single (of model) > ISO1
	View > Single (of model) > ISO2
	View > Single (of model) > ISO3
	View > Single (of model) > ISO4
	View > Multiple > 3rd Angle + ISO (see page 144)

	View > Multiple > 1st Angle + ISO
	Restore selection * Use this option to toggle between the previous and current selection. It also works on sub-item selection of a solid.
	Last view (see page 118)
	View > Zoom Full
	Zoom in (see page 145) *
	Zoom out (see page 145) *
	Actual size (see page 145) *
	Zoom to fit box (see page 146) *
	Zoom (see page 146) *
	Rotate (see page 148) *
	Pan (see page 147) *
	Blank Selected (see page 138)
	Unblank (see page 139)
	View > Shading > Wireframe (see page 121)
	View > Shading > Shaded (see page 122)
	View > Shading > Transparent Shaded (see page 123) (click to toggle Transparent Shaded Wire *ON* or *OFF*)
	View > Shading > Shaded Wire (see page 124)
	View > Shading > Dynamic Hidden Line (see page 125)
	View > Shading > Enhanced Shaded (see page 126)
	Not available (Under development)
	Not available (Under development)

	Display meshes as nodes and open edges (hollow) (see page 127)
	Display meshes as open and interior edges (hollow) (see page 127)
	Display meshes as nodes and open edges (shaded) (see page 127)
	Display meshes as open and interior edges (shaded) (see page 127)
	Display meshes as open edges only (see page 127)
	Render an image (see page 127)
	Display the Render dialog *

* Not on **View** menu.

3rd angle + Iso

Click (*Views toolbar*). This displays four views.

When viewing or sketching onto objects using the **3rd angle + Iso**
 view, each of the four views has its own different principal
plane which is at right angles to the view. The principal plane is
automatically selected appropriate to each view.

This excludes the ISO1 view which will have the XY principal plane.

For best results to see the orientation of the principal plane in the 4 views, click the **Grid on/off** ▦ option on the **Status Bar** to display the grid.
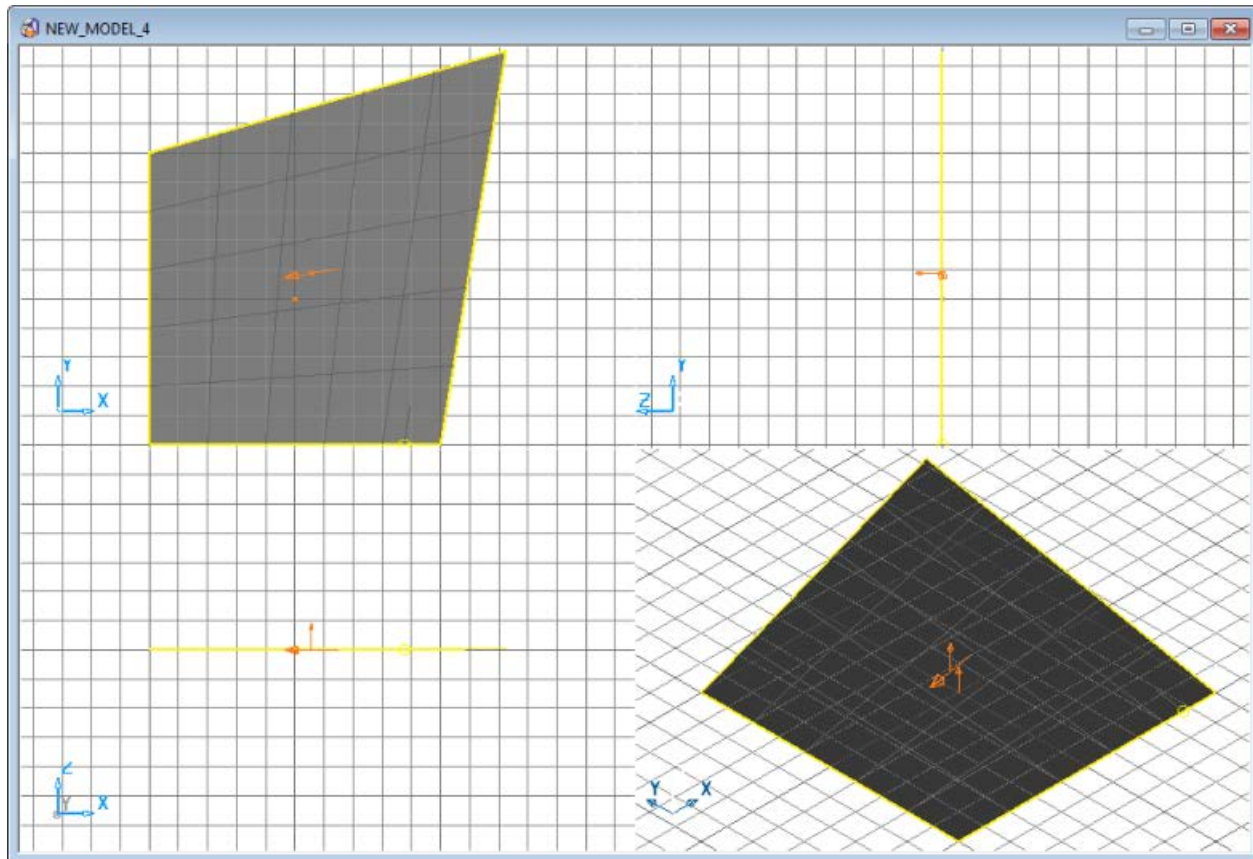
Zoom in

Click ⊕ (*Views toolbar*). The new scale is double the original scale.

You can also zoom into your model by holding down **Ctrl** and either the middle or right mouse button, and dragging the mouse up the screen.

Zoom out

Click ⊖ (*Views toolbar*). The new scale is half the original scale.

You can also zoom out from your model by holding down **Ctrl** and either the middle or right mouse button, and dragging the mouse down the screen.

Zoom to actual size

Click 🔍(1:1) (*Views toolbar*).

The model is panned to the centre of the screen and drawn to its actual size.

Zoom to box

Use this button to select an area of the model that will be magnified to fill the graphics window.

1 On the **View** toolbar click (*Views toolbar*). The cursor

changes to . This is the same as selecting **View > Zoom to box**.

2 Using the left mouse button, click a point on the model, drag a box to enclose the required area and release the left mouse button. The area of the model will be magnified to fill the graphic window.
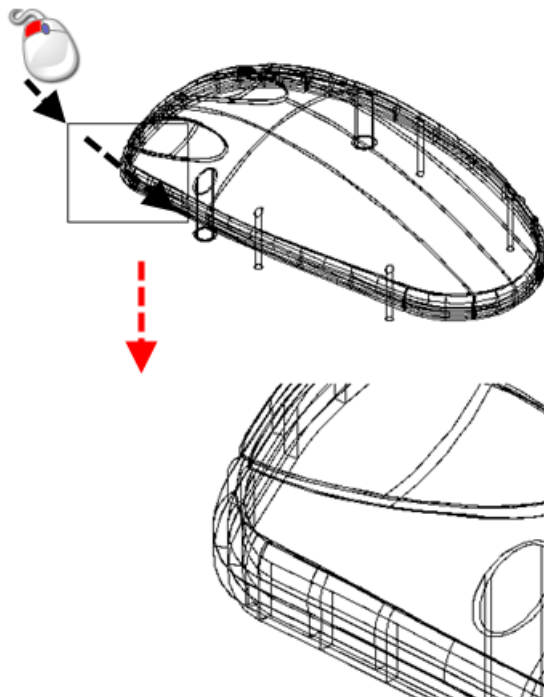
You can also zoom to a box by:

▪ holding down **Shift+Ctrl** and dragging a box using the middle mouse button.

▪ holding down **Alt** and using the right mouse button.

Zoom in and out

1 Click (*Views toolbar*).

2 Click the left mouse button in the graphics window and keep the mouse button pressed.
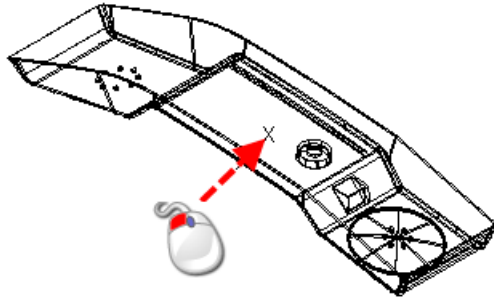
3 Drag the cursor up and down the screen.
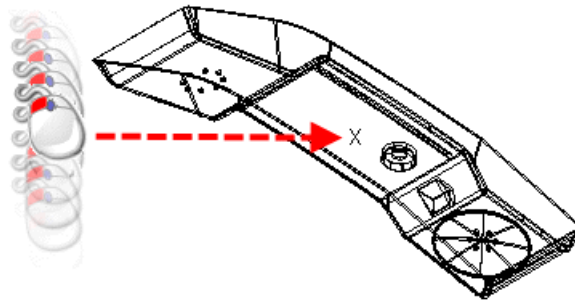
Drag the cursor up the screen to zoom into the model.
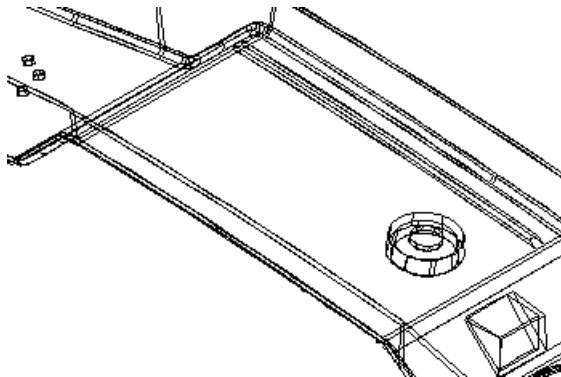
Drag the cursor down the screen to zoom out.
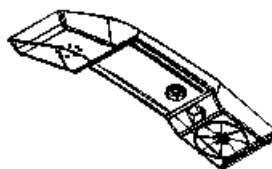
4 When you have the required view, release the mouse.

Panning the view

Use this option to move the model in the graphics window.

1 From the **View** menu, select **Pan**. The cursor changes to ✋

2 Hold down the left mouse button and drag the mouse to move the view. The view of the model is temporarily attached to the cursor.

3 When you have moved the view to the required position, release the mouse button .
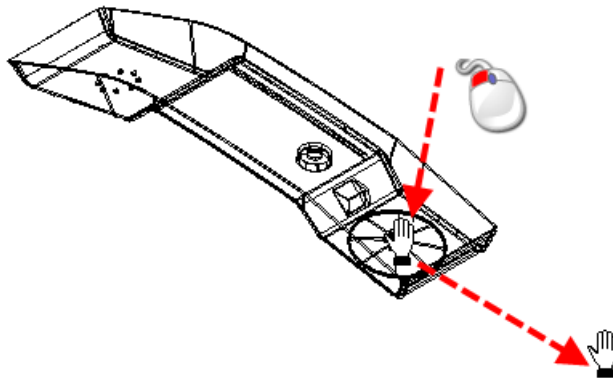
Rotating the view

1 Click (*Views toolbar*).

2 Drag the mouse in the graphics window to rotate the view.

As soon as you click the graphics window, a roller ball appears on the screen.
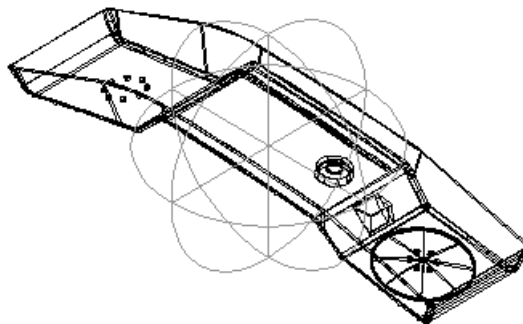
You can drag the cursor in any direction.
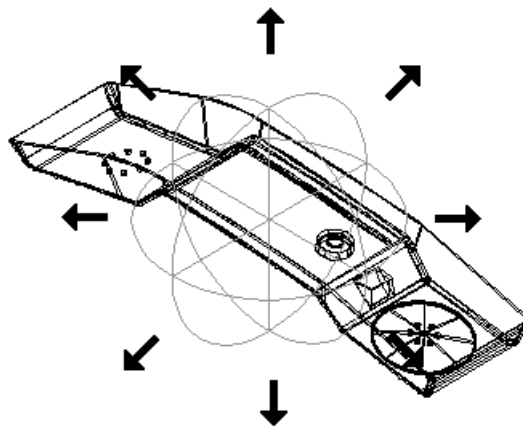
As you drag the cursor, the view of the model changes.
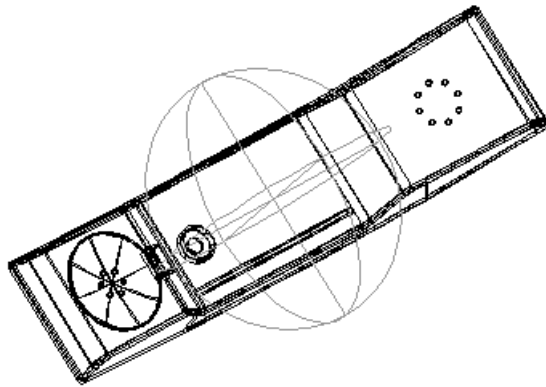
3 Release the mouse when the model is viewed at the required angle.

View > Toolbars > Levels

This displays and removes the **Level** toolbar from the bottom of the screen.

The **Levels form** button opens the **Levels** dialog. For further details, see Format > Levels (see page 29).

The **Level** toolbar displays buttons for used and named levels. If you move the mouse over a button, the name of the level is displayed as dynamic help.

- A level is *ON* if its button is pressed in.
- To turn a level *OFF,* push out its button. You can also turn levels on or off using the **Level** dialog. For further details, see Level dialog (see page 29).
- To turn a level *ON*, press in its button.

To add objects onto a level:

1 Select the group of objects.

2 Click the middle mouse button on a level button on the **Levels** toolbar.

The objects are put on that level.

Grouping levels

You can group levels together. To do this, put the name of the group in front of the name of the level and separate the names using a colon ':'.

For example, suppose we have the following levels:

5 : bottle

6 : handle

To group these levels together, we rename the levels as follows:

5 : top : bottle

6 : top : handle

You can name a level in the following ways:

- Using the **Levels** drop-down list.
- Using the **Level** dialog (see page 29).
- Using the **Edit groups** dialog (see page 151).

The two levels now belong to the group called **top**.
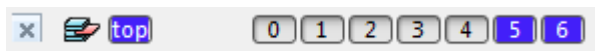
On the **Levels** toolbar, the following changes can be seen:

- the buttons of the two levels have changed to the same colour.
- a new button appears representing the group of levels. It displays the first part of the name of the group.
- the colour of the group button matches that of the two levels.

You can turn *OFF* and *ON* a group of levels by turning on or off the group button. When a level in a group or a group is *OFF*, its button has a border containing the colour of the group to indicate it is off. In the example below, level 5 has been switched *OFF* .
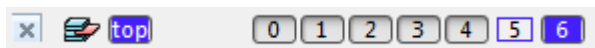
Each time you create another group, another group button is added with a different colour.

You can add and remove levels from groups by adding and removing the group name from the level name.

Levels context menu

1 Right-click on a button on the **Level** toolbar to display the context menu.
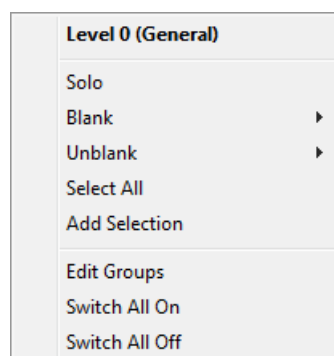
2 Select an option to apply to the level.

Solo — Turn on that level and turns off all other levels.

Blank — Open a submenu with options to blank particular objects on that level.

Unblank — Open a submenu with options to unblank particular objects on that level.

Select All — Select all items on the level.

Add selection — Move the current selection to the level.

Edit groups — Display the **Edit groups** dialog (see page 151).

Switch all on — Turn on all levels.

Switch all off — Turn off all levels.

> *For group buttons, the options apply to all the levels in that group.*

Edit groups dialog

This lets you to switch levels from one group to another and rename levels.
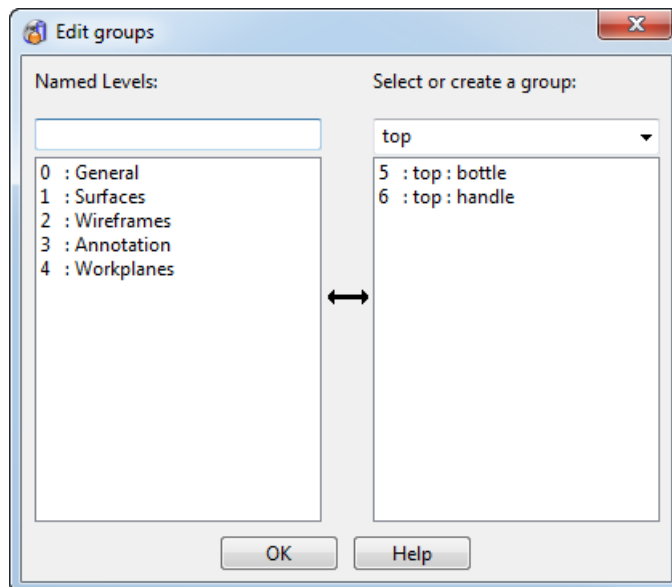
Named levels

You can use this box to name a level.

For example, to name level *35* as **handle** and add it to the group called **core**, type:

35 core: handle

The format of the information you enter should be in the following format:

level_number group_name: level_name

If no group name is given, just the level is named.

The column below the **Named levels** box displays the other named levels not in the selected group. Click a level to remove it from this column and add its name to the other column.

Select or create a group

This is the name of the selected group. You can create a new group by typing a new name in this drop-down list.

The column below this box displays the names of the levels in the selected group. Click a level to remove it from the group and add its name to the other column.

If a group has no levels, it is automatically deleted.

OK — Close the dialog.

View > Toolbars > General Edit

The **General Edits** toolbar is automatically displayed when you open PowerSHAPE.

If you have the **Model Analysis** or **Model Fixing** toolbar *ON*, you can swap to the **General Edit** toolbar by clicking the **General edits** button on the **General edits** flyout.
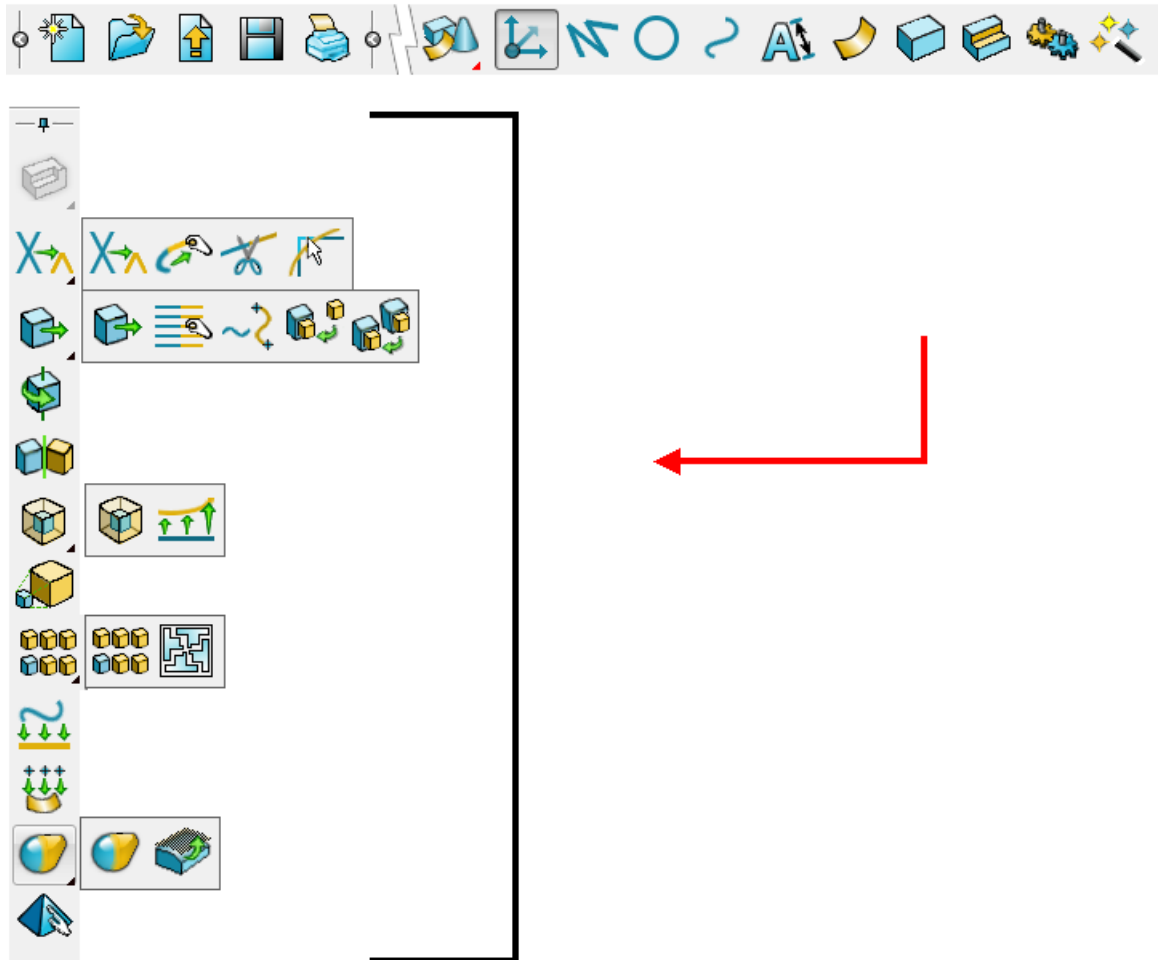
The type of item and sub-item that appear on the toolbar depend on your selection. Options for wireframe, surface and solids are included here.

Edit selected item (wireframe)

Edit selected sub-item (wireframe point)

Edit selected item (surface curve)

Edit selected sub-item (surface point)

Edit selected sub-item (surface curve)

Edit selected item (solid)

	Edit selected sub-item (solid feature)
	Interactively limit wireframe
	Point limit
	Limit cut
	Limit selection
	Move/copy object
	Stretch object
	Place wireframe
	Rotate object
	Mirror/symmetrise object
	Offset object
	Variable offset
	Scale the object(s)
	Create pattern of objects
	Project items onto a plane
	Project points onto surface
	Morph objects

View > Toolbars > Surface/Curve Edits

Use this option to display the **Surface/Curve Edit** toolbar.

For further details, see:

- Editing a curve.
- Editing a surface using the toolbar.

View > Toolbars > Model Fixing

This toolbar groups together the tools for fixing a model.

1 Select **Model Fixing** from the **Toolbars** options on the **View** menu.

This is the same as clicking the **Model Fixing** button on the

General edits flyout.

The **Model Fixing** toolbar is displayed. The options that are available depend on your model and selection.
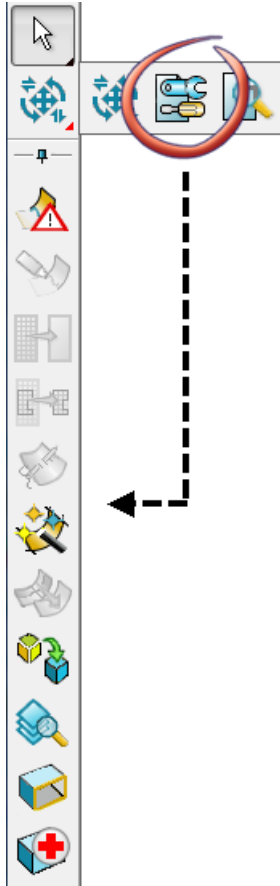
2 Select an option from the toolbar

Identify badly trimmed surfaces from selection

Open the **Surface trim region** editor

Approximate surfaces

Simplify surfaces outside trim

Stitch the current surface to either another surface or a curve (**Stitch surface**)

Automatic surfacing (**Smart Surfacer**)

	Orient surface normals
	Create solid from selected surfaces
	Find and select duplicate attributes (**Find duplicates**)
	Display open edges of solid
	Repair selected solid.

3 When required, turn *OFF* the **Model Fixing** toolbar by selecting **General Edit** or **Model Analysis** from the **Toolbars** menu.

View > Toolbars > Model Analysis

This toolbar groups together the tools for analysing a model.

1 Select **Model Analysis** from the **Toolbars** options on the **View** menu. This is the same as clicking the **Model Analysis** button on the **General edits** flyout.

The **Model Analysis** toolbar is displayed.
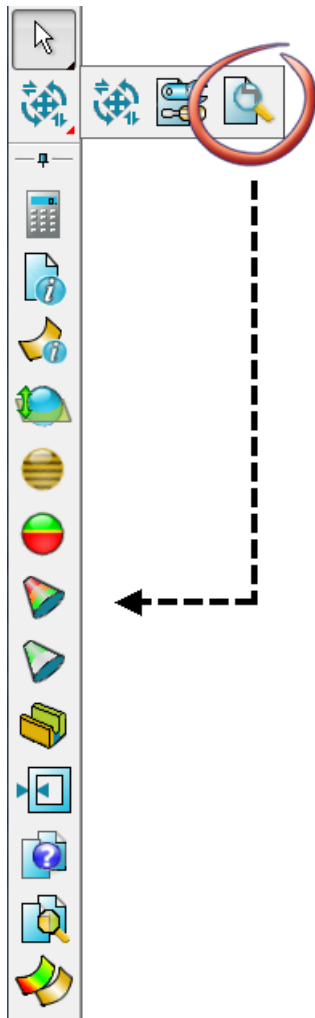
2   Select an option from the toolbar.

 Open the calculator (**Calculator**)

 Show model contents list

 Turn surface inspection on

 Turn surface inspection off

 Dynamic sectioning

 Smoothness shading

 Undercut shading

 Curvature shading

 Minimum radius shading

	Display thickness of a solid as a colourmap (**Thickness shading**)
	Display minimum wall thickness of solid
	Compare two models and select the differences (Model compare)
	Model visual difference
	Comparison analysis

3 When required, turn off the **Model Analysis** toolbar by selecting **General Edit** or **Model Fixing** from the **Toolbars** menu.

View > Toolbars > Style

This displays and closes the **Style** toolbar. To use the **Style** toolbar, you must either have an object selected or be in an object creation mode.

From the **Format** menu, select **Style** to display the **Style** toolbar. This is the same as clicking **Style Toolbar** on the **Item colour** drop-down menu (*Main toolbar*).

If the **Style** toolbar is already raised, it is lowered.
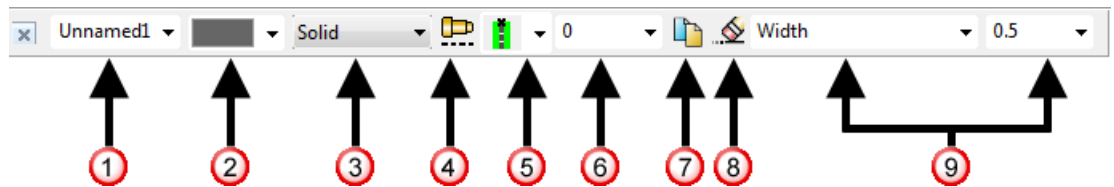
① Style name (see page 3).

② Select colour (see page 3).

③ Line pattern (see page 4).

④ Create user defined line pattern (see page 4).

⑤ Line justification (see page 8).

⑥ Plotter pen (see page 10).

⑦ Copy style (see page 10).

⑧ Delete unused styles and pattern (see page 11).

⑨ Line width and weight/gap options (see page 11).

Use the **Style** toolbar to change:

- the style of a selected object.
- the default creation style of an object.

How do I change the style of selected objects?

1   Select one or more objects.

2 Select a style by name using the **Style name** drop-down list or change settings (such as colour) on the **Style** toolbar.

The objects use the new style or setting. You may need to unselect the objects to see the new style.

The styles on the objects are renamed to existing styles with matching settings. If no existing style matches the settings, then a new one is created.

How do I change the default creation style of objects?

1 Enter any object creation mode.

2 Select a style by name using the **Style name** drop-down list or change settings (such as colour) from the **Style** toolbar.

When you change the settings on the **Style** toolbar, the style matching the new settings is displayed in the **Style name** drop-down list. If no existing style matches the settings, then a new style is created.

New styles are given a default name, which will not appear in the **Style name** drop-down list unless an object using the style is selected or the settings defining the style are selected. To allow a new style to be selected by its name, rename it by typing in the required name into the **Style name** drop-down list.

By default, all objects created from now onwards will use the new settings.

You can change the default so that the new settings are only used on objects of the selected creation mode or on objects belonging to a particular group (for example, if the object is wireframe then all wireframe use that style). To change the default, select the appropriate option on the **Format** page of the **Options** dialog.

Style toolbar - Solid style

 This drop-down list lets you choose how a style is applied to a solid.

 Whole solid — Style is applied to whole solid.

 Selected features — Style applies to selected features of solid. Select the features on the solid or by using the solid feature tree

 Selected face — Style applies to selected face of solid. Select the face on the solid.

Style toolbar - Style name

This drop-down list stores a list of names of the styles. When you create new styles, you can change their names using this drop-down list. You can also select styles from this drop-down list to use on objects.

When you select an option from the **Style name** drop-down list, a preview of the style is displayed.
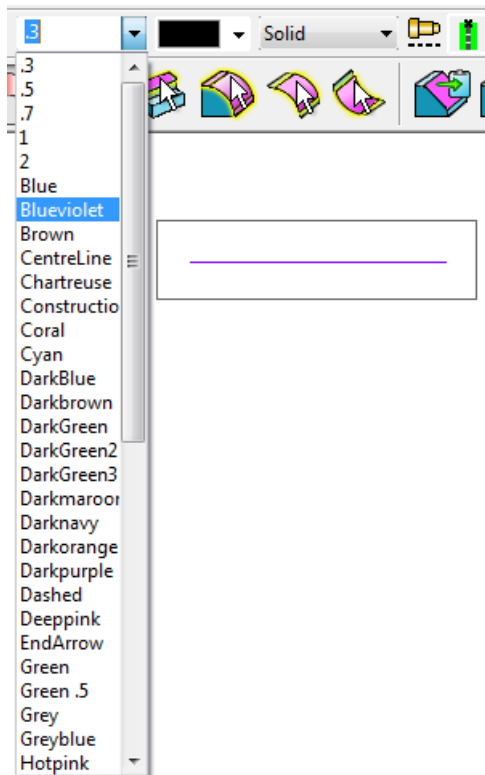
A style with a name of the format *Unnamedn* will not appear in the **Style name** drop-down list unless an object using the style is selected or the settings defining the style are selected. To allow such a style to be selected by its name, select the object displaying this style and rename the style by typing in the required name into the **Style name** drop-down list.

Style toolbar - Style colour

The **Colour** drop-down menu changes the colour of the style.

Style toolbar - Line pattern

The **Line pattern** drop down list on the toolbar provides a selection of patterns.
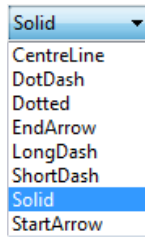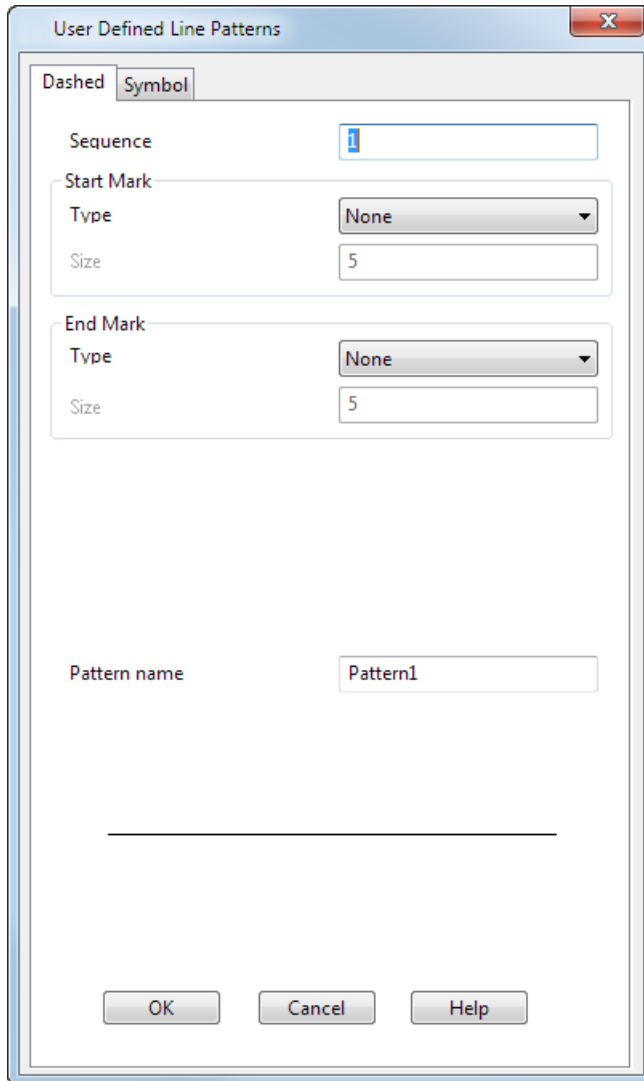
Create user defined line pattern

To create your own line patterns:

1 Click the **Create user defined line pattern** button.

2 Select the appropriate tab on the **User defined line patterns** dialog to create your own line pattern using

- dashed lines (see page 5)

- symbols (see page 7)

Line patterns from Dashed tab

This dialog is used to create user defined line patterns.

Sequence — In this box, input the pen down and up sequences. For example, *1 1 3 3* creates a pattern where the pen is down for 1 unit, up for 1 unit, down for 3 units and up for 3 units. This pattern is repeated in the line.
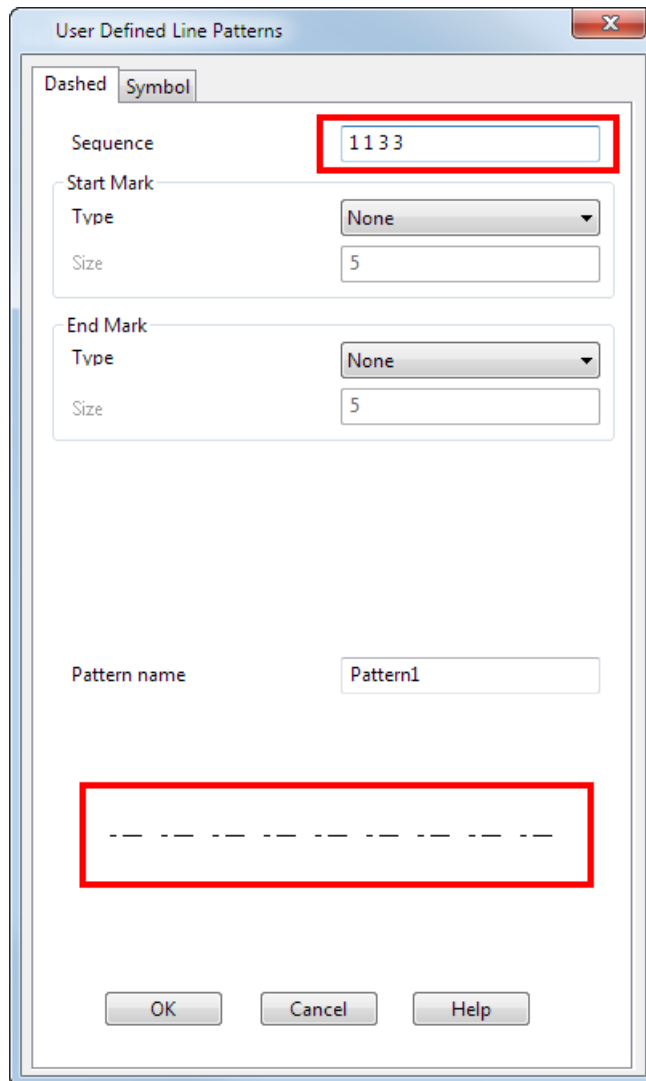
Start Mark and **End Mark** — Enables you to add a mark to the start or end of the line.

Type — Select a mark to add to the end of the line pattern. There are standard marks in the list plus all the symbol definitions stored in the model. Symbol definitions are prefixed with *. This helps you to distinguish between the standard and the user-defined markers. For further details, see Symbols.

Size — For the standard marks, you can set the size of the mark.

Pattern name — This is the name of the line pattern. A unique default name is created. You can change it to your own requirements.

The graphics window shows the pattern of the line as you define it.

OK — Creates the line pattern and adds its name to the **Line pattern selector**. A new style is also created using this pattern and given a default name.

Line patterns from Symbols tab

This page uses a symbol definition to define the line pattern. You must have a symbol definition in your model to use this page of the dialog.

For further details see Symbols.
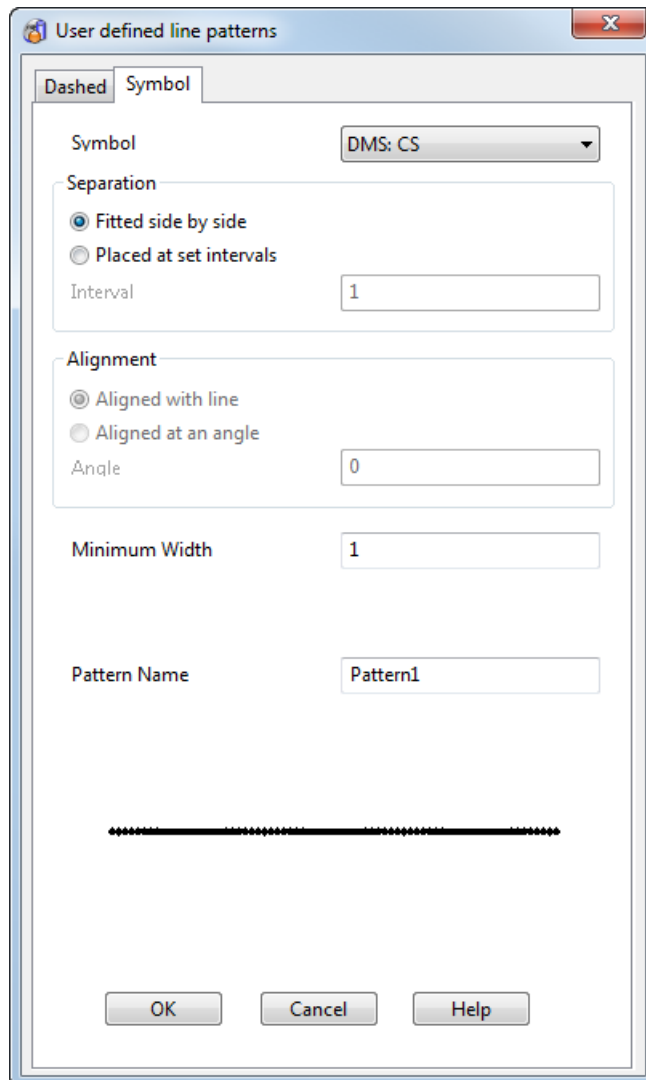
Symbol — This contains the list of symbol definitions in your model. Select the name of the symbol definition.

We will use the symbol below to show what each option means.

Fitted side by side — This fits the symbol side by side in the line pattern.
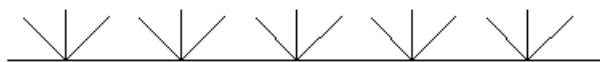
Placed at set intervals — The symbol is placed at set distances apart in the line pattern. The distance is set using the **Interval** box.
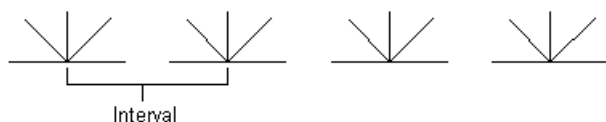
Aligned with line — The base of the symbol aligns with the line.

Aligned at an angle — The base of the symbol is set at an angle relative to the current workspace. The angle is set in the **Angle** box.
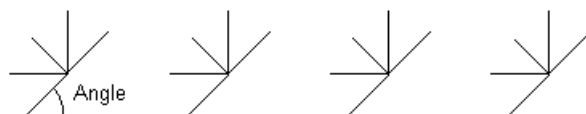
Minimum width — This is the width of the symbol in the line pattern.
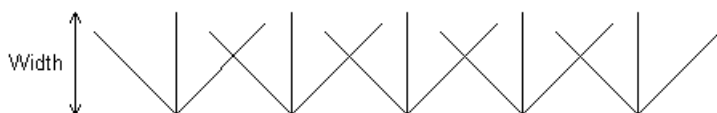
Pattern name — This is the name of the line pattern. A unique default name is created. You can change it to your own requirements.

The graphics window shows the pattern of the line as you define it.

OK — Create the line pattern and add its name to the Line pattern selector. A new style is also created using this pattern and given a default name.

Style toolbar - Line justification

The **Line justification** drop-down menu has four justification options:

- Left
- Centre
- Right
- Round Ends.

Justification controls how lines are thickened. This can happen at either side or both sides of the "track" between the start and end key points.

The following examples use thick lines to illustrate the point on a bold box shape:
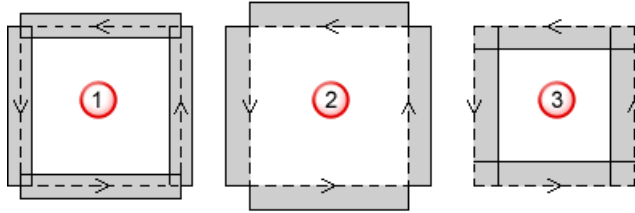
① Justification centre.

② Justification right.

③ Justification left.

You can see how the line justification can be used to produce sharp corners for square boxes. However, for lines which meet at other than 90°, none of the left, right or centre justification options can produce neat corners. The round-ends option is provided to solve this problem:
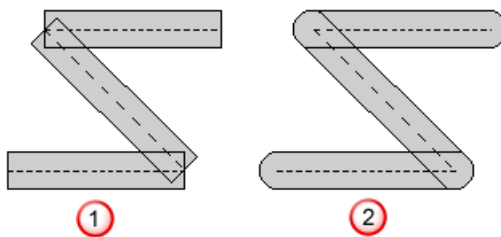
① Square ends.

② Round ends.

The lines on the right have round ends. These are automatically centre justified and produce much neater corners than the square-ended lines on the left.

Justify left

puts the line thickness to the left of the track.
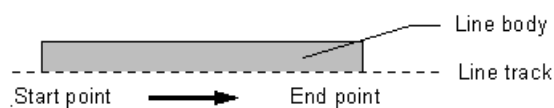
Justify centre

![icon] puts the line thickness either side of the track.
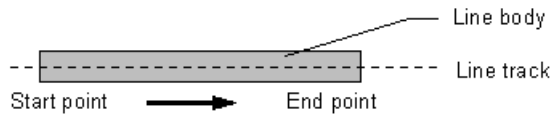
Justify right

![icon] puts the line thickness to the right of the track.
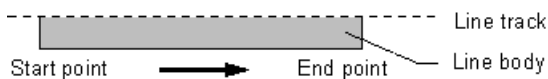
Round ends

![icon] draws round instead of square ends on selected lines. All round-ended lines are centre justified. Their actual end points are inside the end radii as shown below.
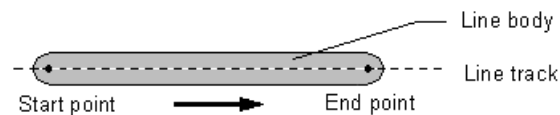
Style toolbar - Plotter pen

When printing to a plotter, this allows you to specify the plotter pen to be used for a specific style. If the pen is set to **0**, PowerSHAPE automatically chooses the most appropriate pen, based on the nearest match between the styles and available pens.

Style toolbar - Copy style

This option allows you to copy the current style.

To copy a style:

1   Click the **Copy** button ![icon] on the **Style** toolbar to display the **New Style Name** dialog.
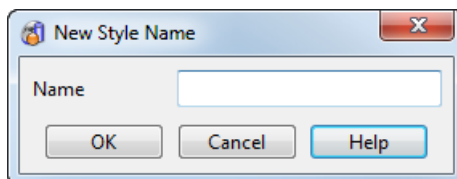
This dialog is used to name the copy of the selected style.

2   In the **Name** text box, enter a new name for the style.

3   Click **OK.**

Style toolbar - Deleting unused styles and patterns

This option is used to delete unused styles and patterns.

1 Click the **Delete style** ![icon] button to display the **Delete unused styles and patterns** dialog.

2 Use the Delete unused styles and patterns dialog (see page 11) to remove unused styles and patterns.

Style toolbar - Delete unused styles and patterns dialog

This dialog is used to delete unused styles and patterns.
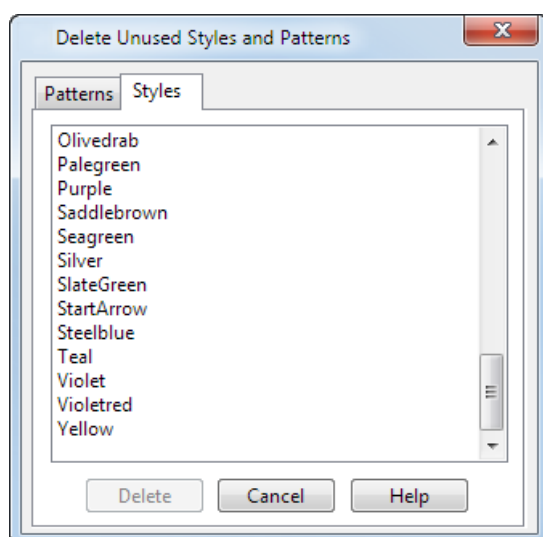
The **Styles** tab lists all the styles that are not used in the current model. Select the styles from the list. A single click on an item highlights it as selected. Clicking a second time removes it from the selection.

The **Patterns** tab lists all the patterns that are not used in any of the styles. Select the patterns you want to delete from the list. A single click on an item highlights it as selected. Clicking a second time removes it from the selection.

If you want to delete a pattern and it is not listed, it must be used in one of the styles. You must delete the style first and then delete the pattern

Delete — Deletes the selected item.

Close — Closes the dialog.

Style toolbar - Line Width and Weight/Gap option

This is a drop-down menu showing:

▪ Width.

▪ Weight/Gap.

Line Width

This controls the width of the line strand.

1   On the **Style** toolbar, select **Width**.

2   Type a value into the text box.

3   Press **Enter** to update the style.

*If you alter the line width and there is an existing matching style of a different width, the new style name has the new line width added as part of the style name. For example, if the original style was **Red** and you alter the line width to .**7**, a new style called **Red .7** will be created.*

Line Weight/Gap

This allows you to create lines with multiple strands and set the gap between the strands. The weight of a line is the number of strands.

The overall line width is determined by the sum of strands ① and gap widths ②. Note that the default thickness of a strand is 0.3.
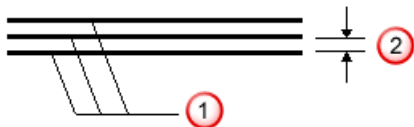
1   If **Weight/Gap** is not already displayed on the **Style** toolbar, from the option menu with **Width** select **Weight/Gap**.

2   Type values into the two data entry boxes.

Weight controls the number of strands.

Gap controls the spacing between strands.

3   Press **Enter** to update the style.

View > Toolbars > Explorer

This displays the **Explorer** toolbar in the browser window. It appears on the Browser and PS-Team pages of the browser window and contains tools, such as back and forward, for viewing web pages.

http://www.delcam.com/info/info1.htm
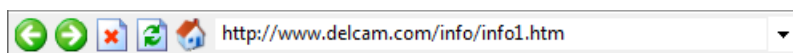
View > Windows

Use the options on the submenu to control the windows that are displayed on the screen.

1 From the **View** menu, select **Windows** to display the following menu:

| Command |
| Tree Window |
| Browser |
| PS-Team |
| Component Library |
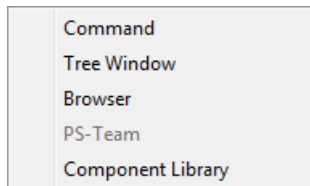
If an option is selected, the window will be displayed.

2 Select the option you require.

View > Windows > Command

This displays the **Command Window**.


```
Command Window                                    [x]
dbsetr() Set all tracing off
The dbase core size (Mb) is  = 64
All tracing now off.
No test print name_sets being traced.
Hello PowerSHAPE User
Getting Model NEW_MODEL_1_23, please wait ...
Parasolid undo memory size (Mb) is 1000

Select >
```
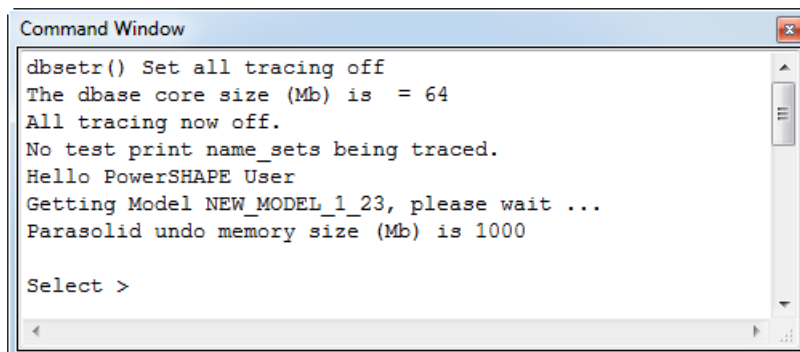
The command window allows you to type commands directly. It also lets you view commands and macros as they are run.
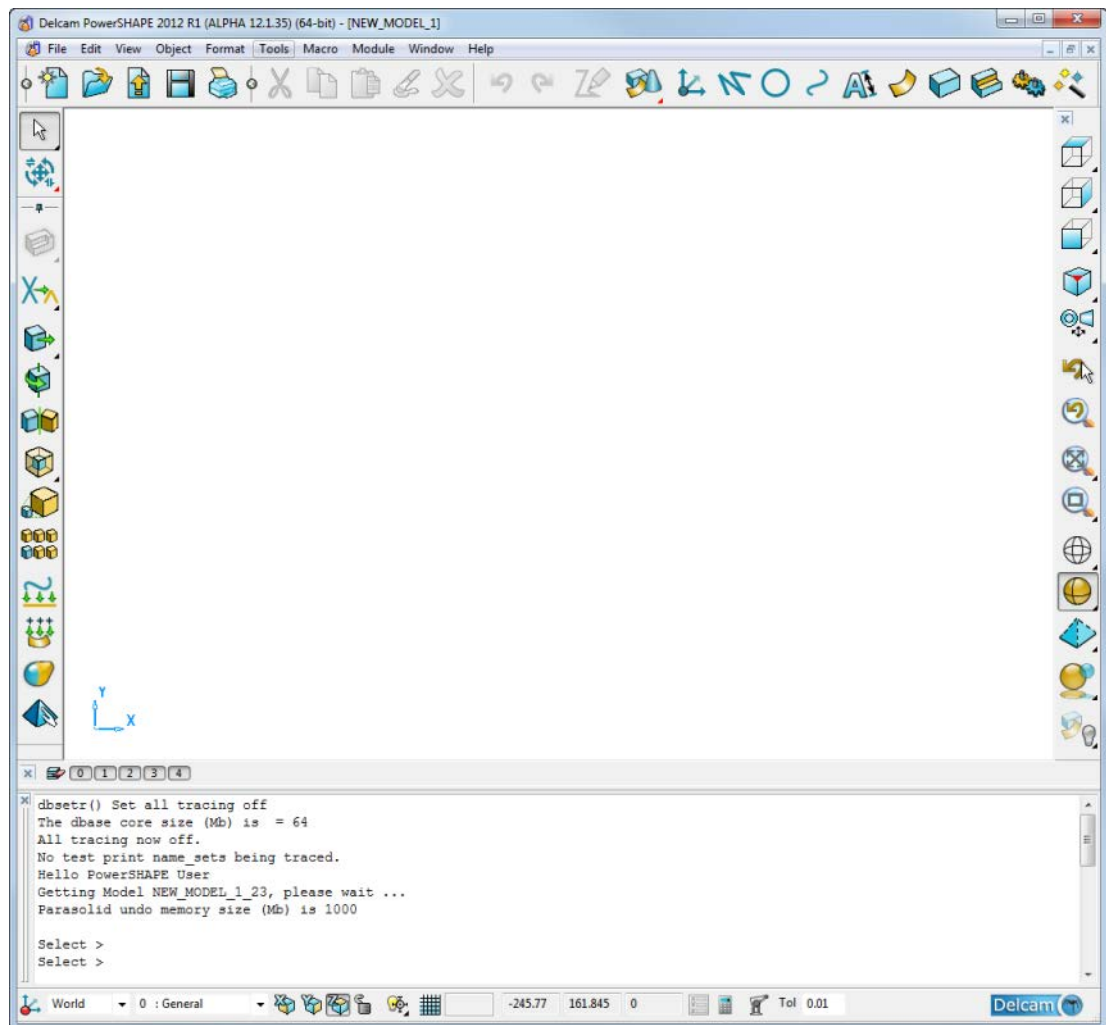
The command window can appear in two different forms

- **Docked** — The window is formatted to appear at the edge of the graphics window. One possible position is shown above. You dock the command window by dragging the floating window to the bottom of the graphics window.

- **Floating** — The position of the command window is independent of the graphics window.

  You can float the **Command Window** using one of the following methods:

  - dragging the two parallel lines next to the ✖ button on the edge of the frame.

  - holding down the **Control** key and clicking the two parallel lines.

  When dragging the floating window, you can prevent it from docking by holding down **Ctrl.**

You can remove the **Command Window** from the screen by either pressing the button on its frame or selecting **View > Windows > Command.**

View > Windows > Tree Window

You can display and remove the window containing the solid tree by selecting and deselecting this option. The window will be opened automatically when you edit a solid.

View > Windows > Browser

You can display and remove the **Browser** window by selecting and deselecting this option.

To shrink the browser window into the left side of the window:

1   Click .

The window is replaced by a bar with an arrow on it in the left side of the window.

2   Click this bar to restore the browser window.

The **Browser** displays the following tabs:

  Home.

Learning Assistant. This option becomes inactive when a Delcam Toolmaker wizard is running in the Browser.

Tutorials.

Help.

Delcam.com browser

PS-Team - currently under development.

Delcam Toolmaker. This tab is only active when a Delcam Toolmaker wizard is running in the Browser.

View > Windows > Component Library

You can display and remove the Component Library window by selecting and deselecting this option.

Object menu

Use this menu to select the object to be created.
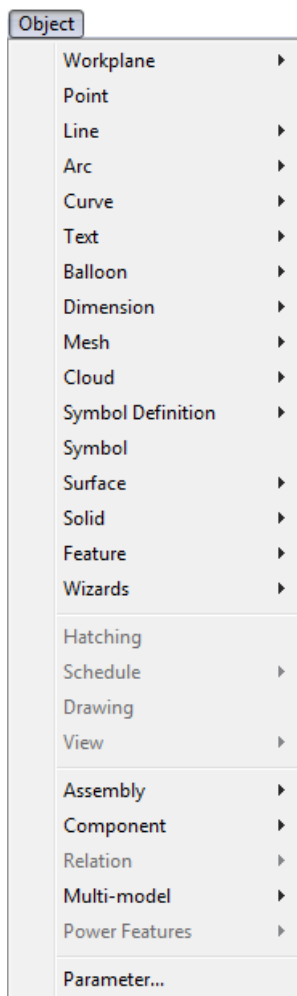
The submenu for each object is explained in the relevant section.

Object > Workplane

Use the options on the submenu to create a workplane in your model.
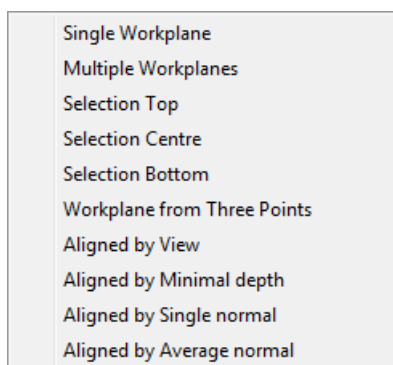
For further details, see Creating a workplane.

Object > Point

Use this option to create a point in your model.

For further details, see What is a point?

Object > Line

Use the options on the submenu to create lines.

Single
Continuous
Rectangle
Polygon
Selection Box
Minimum Distance
Chamfer
Chamfer Untrimmed
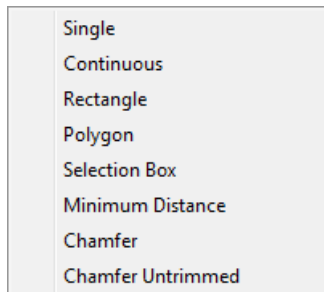
For further details, see What is a line?

Object > Arc

Use the options on the submenu to create arcs and circles.

Full
Fitted
Swept
Fillet Arc
Fillet Arc Untrimmed
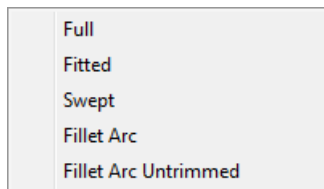
For further details, see What is an arc?

Object > Curve

Use the options on the submenu to create curves.

```
Curve
Snapped
Sketch
Sketch Snapped
Curve Snapped to Mesh
Discontinuities in a Mesh
From Mesh Boundaries
g2-curve
Bspline
Composite Curve          Ctrl+H
Ellipse
Spiral
Helix
Fillet
Untrimmed Blend
Trimmed Blend
Draft
Oblique
Intersection
Curve Projection
Curve Wrapping
Curve Unwrapping
Curve Trimming Line
```
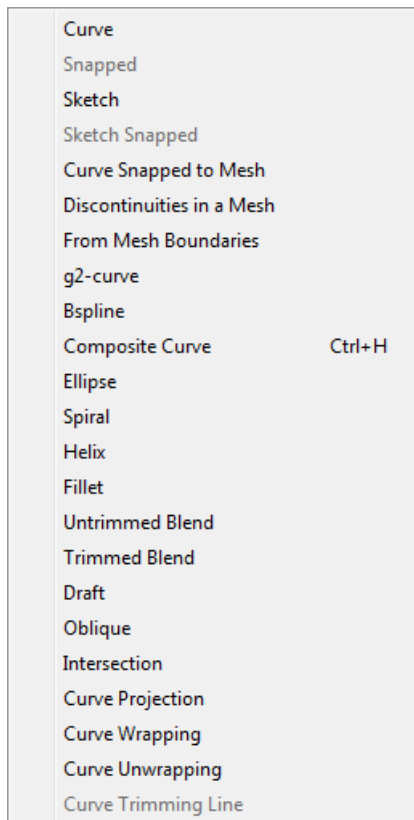
For further details, see What is a curve?

Object > Text

Use the options on the submenu to create text in the Drafting module.

```
Horizontal
Vertical
```
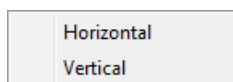
For futher details, see Entering annotation creation mode.

Object > Balloon

Use the options on the submenu to create balloons in the Drafting module.

Default Balloon
Counter Balloon
Label Balloon
Enclosed Label Balloon
Point Balloon (as Options Page)
Point Balloon (Boxed)
Schedule Balloon
Underline Balloon
Split Circle Balloon
Geometric Tolerance
Datum Balloon
Surface Texture
Weld Symbol
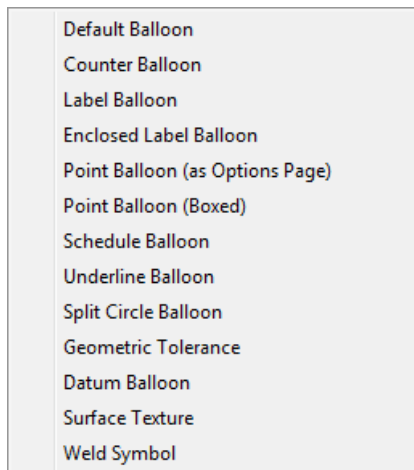
For further details, see Entering balloon creation mode.

Object > Dimension

Use the options on the submenu to create dimensions in the Drafting module.

Automatic
Parallel
Radius
3pt Radius
Diameter
Minor Angle
Major Angle
Basepoint
Chain
Ordinate
View
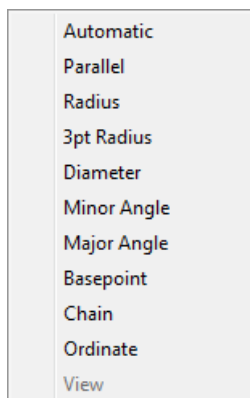
For further details, see Entering dimension creation mode.

Object > Surface

Use the options on the submenu to create surfaces.

Block	
Cylinder	
Cone	
Sphere	
Torus	
Plane	
Spring	
Extrusion	
Revolution	
Smart Surfacer	Ctrl+T
Bead Surface	
From Points Through Patches	
From Mesh	
Draft	
Split	
Extension	
Blend	
Fillet	Ctrl+F
Wrap	
Addendum	
Fill Edge	
Drawbead	
Untrim	
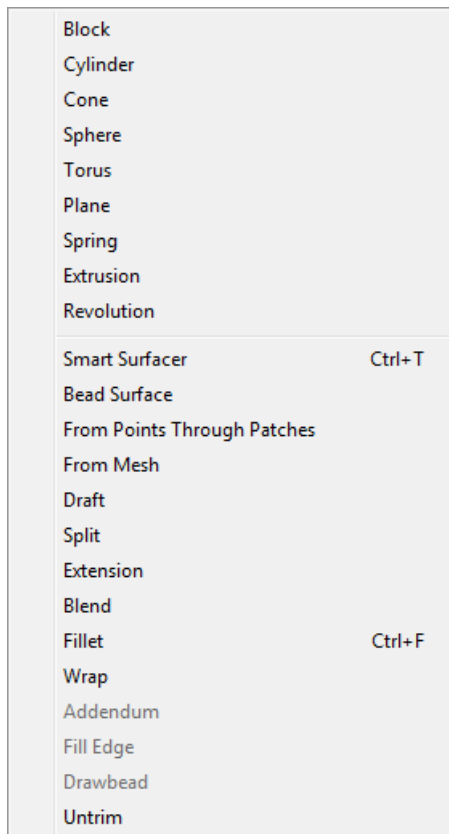
For further details, see Surface creation.

Object > Mesh

Use the options on the submenus to edit meshes.
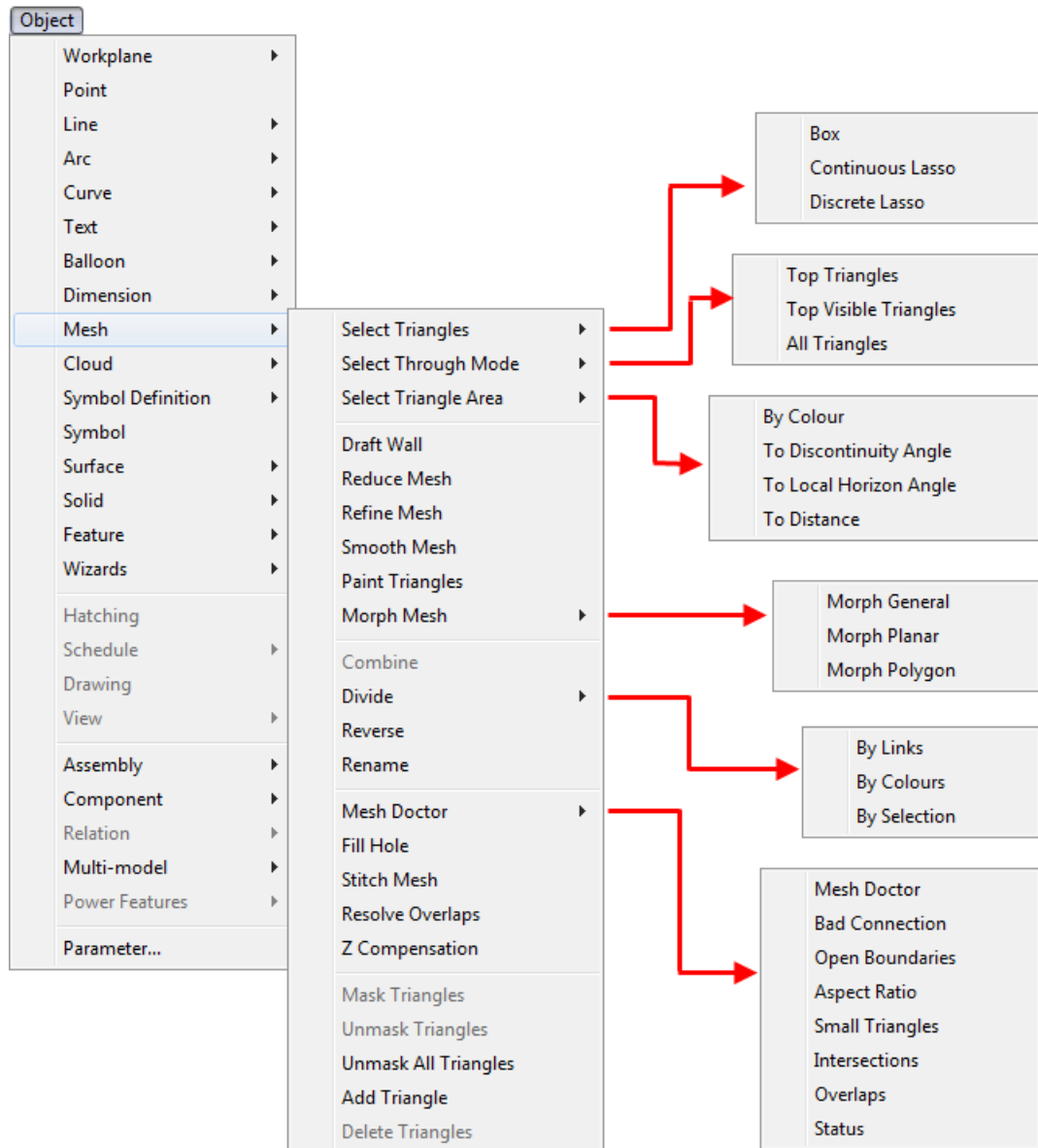
For further details see Editing a mesh.

Object > Solid

Use the options on the submenu to create solids.

Block
Cylinder
Cone
Sphere
Torus
Spring
Extrusion
Extrude Regions
Revolution
Drive-Curve
Core From Wireframe
Core From Selection
From Surfaces or Meshes
From Automatically Trimmed Surfaces
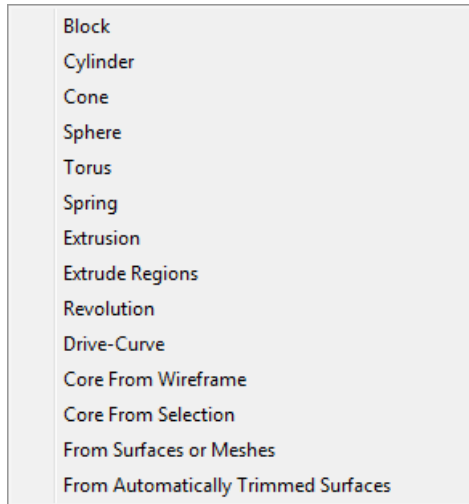
For further details, see Using the solid creation options.

Object > Feature

Use the options on the submenu to create solid features.

Symbol Definition ▸	
Symbol	
Surface ▸	
Solid ▸	
Feature ▸	Smart Feature Recognition
Wizards ▸	Smart Feature Selector
Hatching	Addition
Schedule ▸	Subtraction
Drawing	Intersection
View ▸	Split
	Cut
Assembly ▸	Radial Cut
Component ▸	Boss
Relation ▸	Boolean Boss
Multi-model ▸	Hole
Power Features ▸	Pocket / Protrusion
	Hollow
Parameter...	Thicken
	Bulge
	Morph
	User Defined
	Rib Fillet
	Fillet
	Chamfer
	Wrap
	Transform ▸ Move
	Rotate
	Mirror
	Scale

For further details, see Using the feature creation options.

Object > Wizards

Use the options on the submenu to access the wizard options.

Electrode
Mold Die Wizard
Core/Cavity Separation ▸
Mold Lock Wizard
Slide Core Wizard
Cooling Wizard
Cavity/Core Wizard
Slide Wizard
Moldbase Wizard
Power Feature Trimming
Simulate Mold Opening
Request Quotation
Order Online
Press Die Wizard

For further details, see Using the wizard options.

Object > Hatching

Use this option to create hatching in the Drafting module.

For further details, see Creating hatching.

Object > Schedule

Use the options on the submenu to create and append schedules in the Drafting module.

Hole Schedule
Ejector Schedule
Bill Of Materials
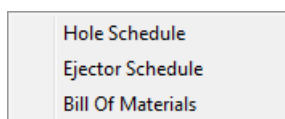
For further details, see What is a schedule?

Object > Symbol Definition

Use the options on the submenu to access the create and edit options for symbol definitions.
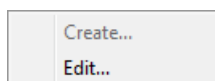
Create...
Edit...

For further details, see Creating a symbol definition and Editing a symbol definition.

Object > Symbol

Use this option to create symbols in the Drafting module.

For further details, see What is a symbol?

Object > Drawing

This creates drawings in the Drafting module.

For further details, see Creating a drawing in Delcam Draft..

Object > View

Use the options on the submenu to create views in the Drafting module.

```
Single
Relative
Section
Section with Background
Limited
OLE
```
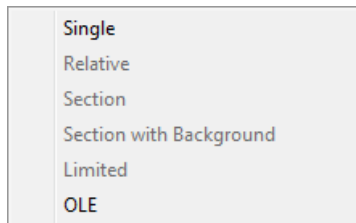
For further details, see Entering view creation mode.

Object > Assembly

Use the options on the submenu to create an assembly when in Assembly modelling mode.

```
Create
Explode          ▶
Intersection
```
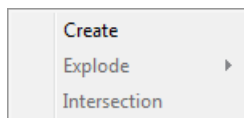
For further details, see Creating a new assembly.

Object > Component

Use the options on the submenu to create components when in Assembly modelling mode.

```
From Solids
From Solids with Power Features...

Add to Assembly
Add to Assembly Advanced          ▶

Component Wizard
```
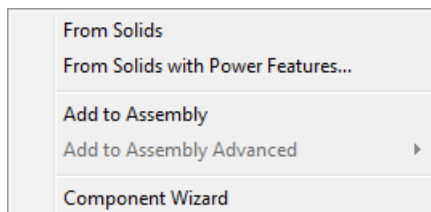
For further details, see Creating components.

Object > Relation

Use the options on the submenu to create a relation and manage attachments when in Assembly modelling mode.

Create
Manage Attachments
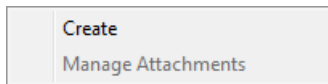
For further details, see Adding relationships between components.

Object > Multi-model

Use the options on the submenu to manage multiple models when in Assembly modelling mode.

Manage Source Models
Refresh Imported Components
Convert to Local
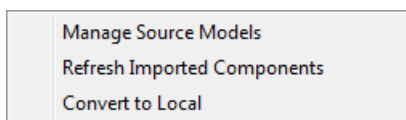
For further details, see Assembly modelling.

Object > Parameter

Use this option to open the **Parameter Editor** dialog which enables you to calculate and input parameters for objects in your model.
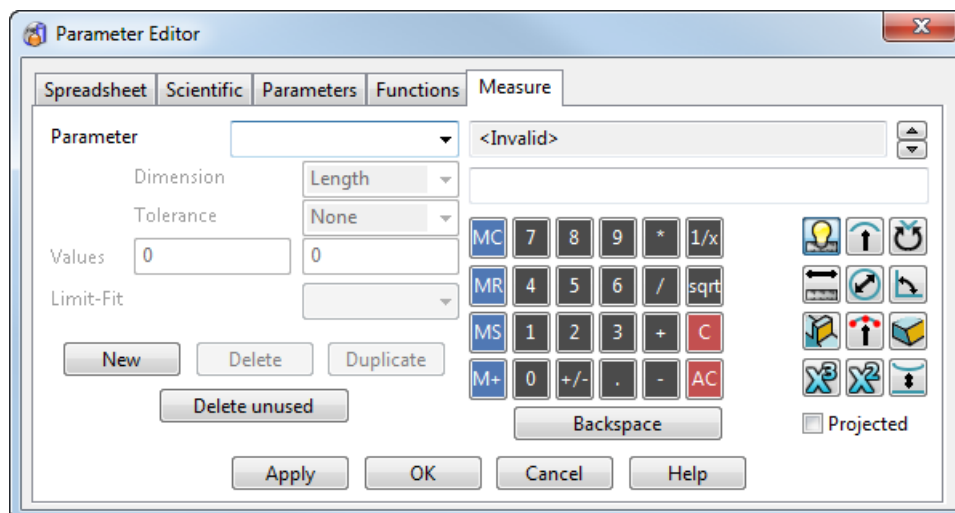
For further details, see Creating a parameter.

Format menu

The options in this menu let you alter the format of the non-geometric properties of objects.
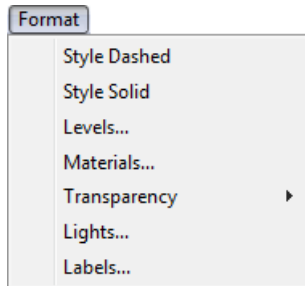
Format > Style Dashed

Use this option to change the line style of selected lines and arcs to dashed. It is particularly useful in drawing views for displaying lines and arcs as hidden.

If you use the **Style** toolbar to change the style of an object, the style affects all views of the object in all model and drawing windows. In drawing windows, the style of an object sometimes needs to be view dependent because an object may be hidden (and hence dashed) in one view, yet visible (hence solid) in another.

The **Format > Style Dashed** option can be used to set the style of pattern of a line or an arc to dashed, in a particular drawing view only.

You can change dashed lines and arcs to use solid lines using Format > Style Solid (see page 28).

When you change the lines and arcs in drawing views to dashed, the lines and arcs in the model don't change.

*In model windows, the command simply does the same as changing the pattern on the **Style** toolbar.*

Format > Style Solid

Use this option to change the line style of selected lines and arcs to solid after it has been set to dashed with the Format > Style Dashed (see page 28).

Format > Levels

As your models become more complex, you may find that some areas become cluttered. The **Levels** option enables you to assign objects to different drawing levels and then choose whether to show or hide them.

Levels are purely a display feature. They have no effect on any other aspect of the model. However, good use of levels can add to the logical structure of a model. This makes it easier for you and others to understand your model.

To create levels for your model

1   From the **Format** menu, select **Levels** to display the Level dialog.

2   Use the dialog to add different components to different levels.

Level dialog

Use this dialog to name and display levels.
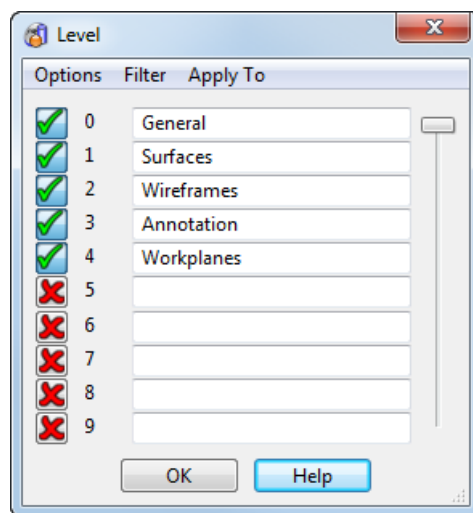
The above example shows the pre-set level options in their default view settings. You can change these, but as the program can handle up to 1000 levels, it is recommended that you leave the pre-sets in place and create more levels when you need them.

The cursor/construction level (located by default on Level 998/999) is used by the intelligent cursor to create the construction lines.
We strongly recommend that you do not remove or rename this level. You can switch it on, but the program can become confusing to use when all the construction objects are displayed.

Apply to — Use this option to specify where you want to change the levels. For example you might want to have the same model viewed in one window with all objects visible, but in another window with some objects hidden. Select one of the following:

- **Model** — Change levels in all the windows of the model.
- **Window** — Change levels in only the selected window of the model.
- **View** — Change levels in only the selected view of a drawing.

Level — Each level has a number. Visible levels are indicated by ✔. Hidden levels are indicated by ✘. You can turn *ON* a level by clicking ✘ and *OFF* by clicking ✔. You can type in a new name for a level or edit an existing name.

If the model is shaded and you turn on a level, you can view the model in wireframe mode by pressing the ⊗ button next to the progress bar on the status bar whilst the level is opening.

You can also turn levels on and off using:

- the **Levels** toolbar.
- the Drop-down list on the status toolbar.

    If you type **15** in the drop-down list, level 15 is switched on.

    If you type **15 B surfaces**, level 15 is switched on and named. This happens even if level 15 is not currently named or used and is therefore not displayed in the drop-down list.

Options — This menu has three options:

- **Set All** — Set all levels to be visible (apart from the **Cursor** level).
- **Unset All** — Set all levels to be hidden.
- **Copy Names** — Copy level names from another open model (see page 31).

Filter — This menu is at the top of the dialog. It controls which levels are shown in the dialog. Each option has a toggle, allowing you to select any of the options.

- **Used** — Shows only the levels being used in the current PowerSHAPE window.
- **Named** — Shows only the levels with names, even if your model is not using them.
- **On** — Shows only the levels which are set to *ON* (that is the levels that are visible).

*Any changes you make to this filter, do not affect the filter used on the **Levels** drop down list on the **Main** toolbar. The toolbar filter is controlled from the **Options** dialog.*

Copying level names from another model

Copy level names from another open model by selecting **Copy Names** from the **Options** menu on the **Level** dialog.
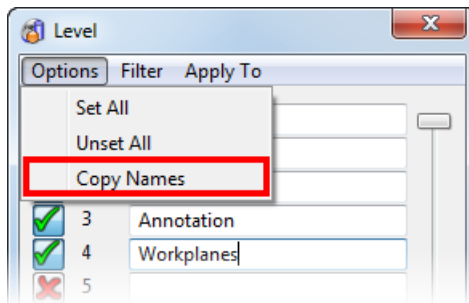
To copy a level name:

1 Ensure that you have two open models: the model with the level to be copied (source) and the model that the level will be copied to.

2 Click ![icon] to open the **Levels** dialog.

3 Select **Options > Copy Names** to display the **Copy Level Names** dialog.
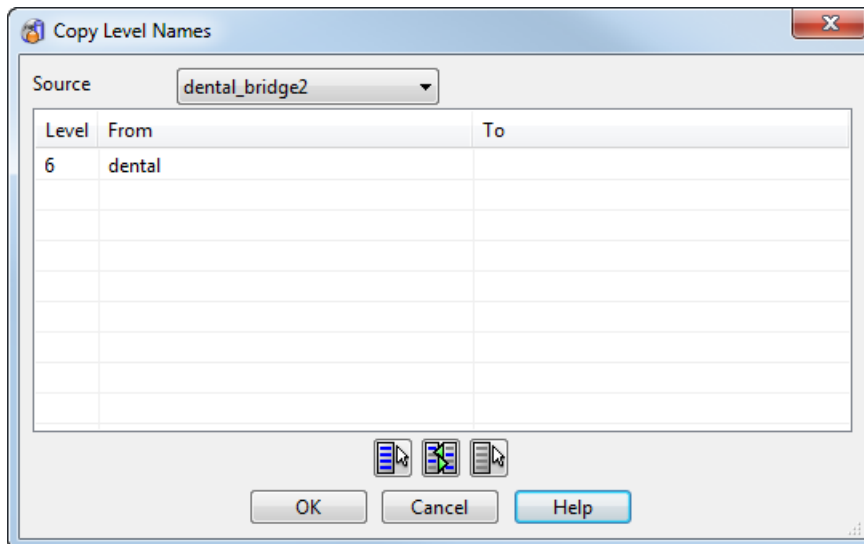
4 Select the **Source** model from the drop-down list.

5 Select the levels to be copied from the list using the following tools:

- **Shift+click** and **Ctrl+click** selection technique.

- Button on the dialog:

  ![icon] Select all levels

  ![icon] Invert selection

  ![icon] Clear selection

6 Click **OK**.

Format > Materials

This option lets you edit the shading effects applied to the surfaces and solids in your model using **View > Shading > Shaded**. Shading does not provide fully rendered images. It allows you to visualise your model as a finished article.

The **Material Selection** dialog can be used in two ways:

- To change the material of selected surfaces and solids
- To change the default material

Changing the material of selected surfaces and solids

1 Shade the surfaces and solids by selecting **View > Shading > Shaded**.

2 Select the surfaces and solids whose material you want to change.

3 From the **Format** menu, select **Materials** to display the **Material Selection** dialog.

4 Select a new material.

5 Click **OK** to change the material of the selected objects.

Changing the default material

1 Ensure that no object is selected.

2 From the **Format** menu, select **Materials** to display the **Material Selection** dialog.

3 Select a new material.

4 Click **OK**. The new material will be used for subsequent surface colour shading.

Material Editor dialog

This lets you edit existing options or create your own material effects.
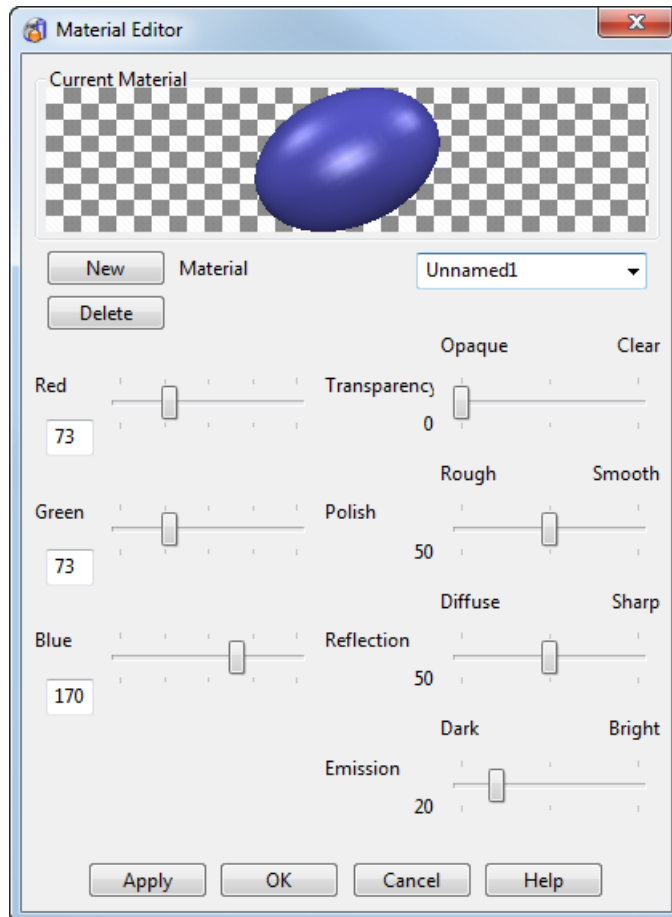
New — Creates a new material as defined by the current slider settings. A default name is given to the material, which is displayed in the **Material** text box. You can use the **Material** text box to edit the name.

Material — This displays the name of the selected material.

Delete — If a user-defined material is selected, the material is deleted.

Sliders — If a user-defined material is selected, adjust the sliders until the effect you want is displayed.

For the colours, **Red**, **Blue** and **Green**, you can also type in the values in the text boxes.

Apply — Save the changes. The dialog remains open so you can continue making changes.

OK — Save the changes and close the dialog.

Cancel — Discard any unsaved changes and close the dialog.

Transparency

Use the options on the submenu to create transparency on a shaded object.
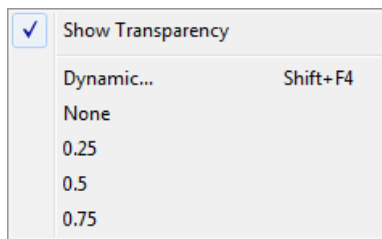
1   Select the objects that are to be given transparency.

2   Select **Format > Transparency** to create transparency on a shaded object, even if the material is not transparent. This 'quick' transparency is saved with the object and overrides any setting in the material.

3   Select the appropriate option from the menu. **Show Transparency** is selected by default. If deselected, the following transparency options are not available:

- **Dynamic** displays the **Set Object Transparency** toolbar (see page 34). Use the slider on this toolbar to adjust the transparency.

    *Selecting **Dynamic** is the same as selecting the objects and pressing **Shift + F4**.*

- **None** gives no transparency for selected objects.

- **0.25** sets 25% transparency for selected objects.

- **0.5** sets 50% transparency for selected objects.

- **0.75** sets 75% transparency for selected objects.

Using the different transparency settings gives the following results:
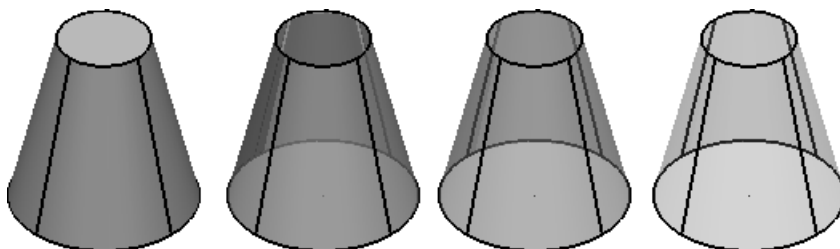
Set Object Transparency toolbar

The **Set Object Transparency** toolbar is displayed when you select **Dynamic** from the **Transparency** (see page 34) options on the **Format** menu. You use the toolbar to set the transparency on selected objects.

*You can also display this toolbar by selecting the objects and pressing **Shift+F4**.*
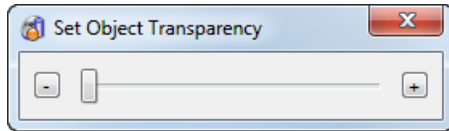
You can use the toolbar in the following ways:

- Use the buttons to cycle through the different increments of transparency, shown on the **Transparency (see page 34)** options on the **Format** menu.

- Use the slider to gradually apply transparency as required.

Format > Lights

Use this option to set up the lights in the shaded and render views.

1   From the **Format** menu, select **Lights** to display the **Light Studio Selector** dialog.
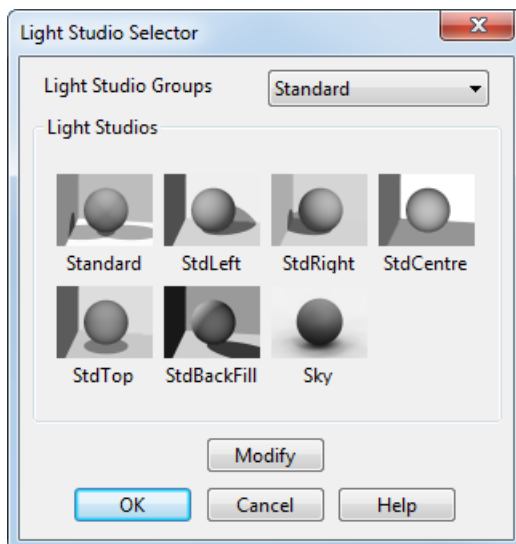
2   Use this dialog to set up the lights.

    Light Studio Groups — You can choose either standard or spot lights. You can also set up custom lights

    Light Studio — This contains the direction of the lights.

3   Select an option.

4   Click **OK** to apply the changes to the shaded view and remove the dialog from the screen.

For further details, see Changing lights in the lights studio.

Format > Labels

Use this option to wrap labels onto shaded objects.

For further details, see Wrapping a label onto objects.

Tools menu

This menu contains options that affect the operation of the program. It also controls how some of the functionality is applied.

Tools
Model Fixing ▶
Model Analysis ▶
Selection Information
Component Drawings...
Create Titleblock
Create Movie...
Login...
Preview Mode
Release Licences
Vouchers ▶
File Doctor
Compress Model
Convert all Solids...
Customise...
Options...
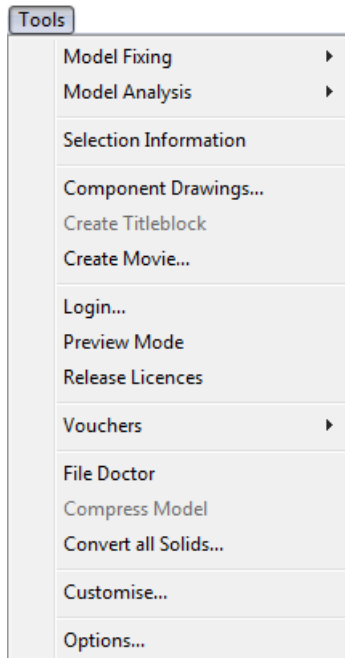
For further information, select from the following:

Model Fixing (see page 37)

Model Analysis (see page 41)

Selection information (see page 67)

Component Drawings (see page 69)

Create Titleblock (see page 69)

Create Movie (see page 69)

Login (see page 71)

Preview Mode (see page 72)

Release Licences (see page 72)

Vouchers (see page 72)

File Doctor (see page 72)

Compress model (see page 73)

Convert all Solids

Customise (see page 73)

Options (see page 75)

Tools > Model Fixing

Use the options on this submenu to fix your model.

1   From the **Tools** menu, select **Model Fixing** to display the following menu:

Identify Badly Trimmed Surfaces	
Surface Trim Region Editing	
Find Duplicates	
Stitch	
Approximate	
Simplify Outside Trim	
Smart Surfacer	Ctrl+T
Orient Normals	
Solid From Surfaces	
Show Open Edges in Solid	
Repair Solid	
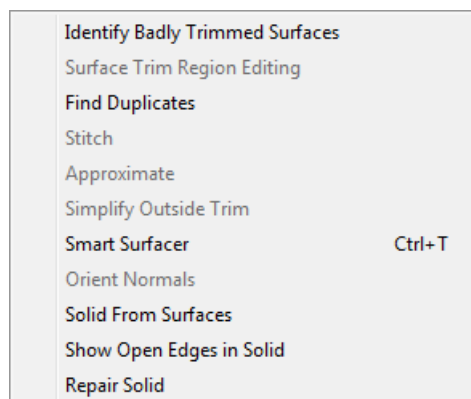
2   Select the option you require.

Identify badly trimmed surfaces (see page 38)

Surface trim region editing

Find duplicates

Stitch

Approximate

Simplify outside trim

Smart Surfacer

Orient normals

Solid from surfaces

Show open edges in solid

Repair solid (see page 40)

Tools > Model Fixing > Identify badly trimmed surfaces

Badly trimmed surfaces are usually drawn or shaded as untrimmed.

To identify these surfaces in the current selection

1 Select the surfaces/solids you want to check.

You can select a group containing any type of objects. Surfaces are filtered out from the group.

If the selection contains solids, the surfaces that make the solids are checked and you can decide how solids containing badly trimmed surfaces should be handled.

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

3 Click **Identify badly trimmed surfaces** (*Model Fixing toolbar*).

4 If any of the selected solids are found to have badly trimmed surfaces:

- For version 8 solids, the **Identify Badly Trimmed Surfaces** dialog (see page 39) is displayed. This lists the names of any bad surfaces, identifies the solids with bad surfaces and lists the bad surfaces in the solid. The text is also displayed in the command window so you can still see the surfaces that are bad after the dialog has closed.

- For Parasolids, you are warned that badly trimmed surfaces have been found. Use **Trim region editing** to fix the surfaces.

Identify Badly Trimmed Surfaces dialog

This dialog is only displayed if your solid is a version 8 solid.
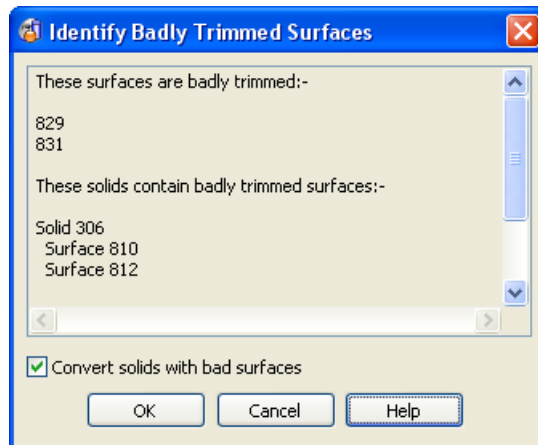
Convert solids with bad surfaces — If selected, the solids with bad surfaces are converted to surfaces and added to the selection when **OK** is selected. If deselected, solids with bad surfaces will remain as solids. The option is selected by default.

Cancel — Discard changes and close the dialog.

OK — Any badly trimmed surfaces that do not belong to solids are selected. If **Convert solids with bad surfaces** was selected, any solids with bad surfaces are converted to surfaces and the bad surfaces added to the selection.

If all the solids contain correctly trimmed surfaces, or if there are only badly trimmed surfaces, no dialog is displayed.

You can investigate the trim data in detail using the **Generate diagnostic** button on the **Trim region editing** toolbar.

Tools > Model Fixing > Surface Trim Region Editing

Use this option to display the trim region editing options on the **Boundary editing** toolbar. The **Trim region editing** option is only active when you select a surface that has trim boundaries.

Tools > Model Fixing > Find duplicates

Use this option to select all duplicate objects in a model.

For further details, see Finding duplicate objects in a model.

Tools > Model Fixing > Stitch

Use this option to close small gaps between surfaces and curves

For further details, see Stitching surfaces.

Tools > Model Fixing > Approximate

Use this option to remove laterals and longitudinals that do not support the surface structure within tolerance.

For further details, see Approximating a surface to tolerance.

Tools > Model Fixing > Simplify outside trim

Use this option to remove laterals and longitudinal made entirely invisible by being outside the trim boundaries.

For further details, see Simplifying outside trim.

Tools > Model Fixing > Automatic Surfacing

Use this option to create a surface from a selection of wireframe.

For further details, see Using Automatic Surfacing.

Tools > Model Fixing > Orient normals

Use this option to orient the normals of multiple surfaces such that the outside of two adjacent surfaces match.

For further details, see Orient surface normal.

Tools > Model Fixing > Solid from surfaces

Use this option to create a solid from a surface or group of surfaces.

For further details, see Creating a solid from surfaces.

Tools > Model Fixing > Show open edges in solid

Use this option to highlight any segments of selected solids' boundaries which are open within tolerance.

For further details, see Finding open boundaries on solids.

Tools > Model Fixing > Repair solid

This lets you repair a solid by filling large gaps and sealing gaps. One of the following tools is used to repair the solid:

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

For further details see Solid modelling.

Tools > Model Analysis

Use the options on this submenu to analyse your model.

1 From the **Tools** menu, select **Model Analysis** to display the following menu:

Calculator
Properties
Surface Inspect
Dynamic Sectioning...
Smoothness Shading   F7
Undercut Shading   F8
Curvature Shading
Minimum Radius Shading   F9
Thickness Shading
Solid Wall Thickness...
Model Compare...
Model Visual Difference...
Comparison Analysis...
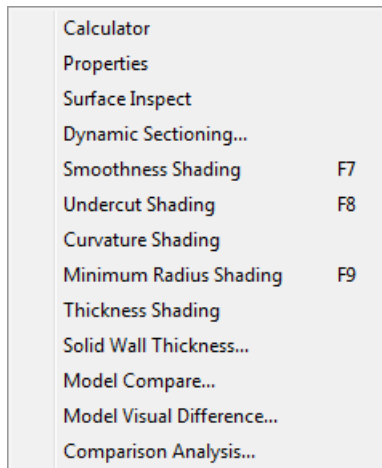
2 Select the option you require.

Calculator

Properties

Surface inspection

Dynamic sectioning (see page 42)

Smoothness shading (see page 45)

Undercut shading (see page 49)

Curvature shading (see page 51)

Minimum radius shading (see page 53)

Thickness shading (see page 54)

Solid wall thickness (see page 56)

Model compare

Model visual difference (see page 57)

Comparison analysis (see page 59)

Tools > Model Analysis > Calculator

Use this option to display the **Calculator** dialog.

For further details, see Calculator dialog.

Tools > Model Analysis > Properties

Use this option to find out how many objects are stored, details of current levels and levels of objects in the selected model.

For further details, see File Properties dialog.

Tools > Model Analysis > Surface Inspect

Use this option to inspect a position on a surface to display its coordinates, surface normal, draft angle, the wall thickness and curvature. Surface inspection data can also be captured and pasted into any text editor.

For further details, see Inspecting surfaces.

Tools > Model Analysis > Dynamic Sectioning

Use **Dynamic sectioning** to take sections through your model. Use the sliders to move the clipping planes backwards and forward through the model. This lets you view the interior detail of the model.
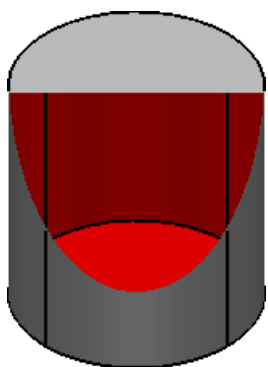
The orientation of the front and back clipping planes is defined by selecting an **Axis** from the drop-down list in the Dynamic Sectioning dialog (see page 43).

1 Shade your model.

2 If you have more than one view of your model displayed, click the view in which you want to move the section plane.

3 Select **Model Analysis** on the **General edits** flyout.

4 Click the **Dynamic Sectioning** (*Model Analysis toolbar*).

5 Use the Dynamic Sectioning dialog (see page 43) to dynamically take sections through your model.

Dynamic Sectioning dialog

Use the **Dynamic Sectioning** dialog to take sections through the model.
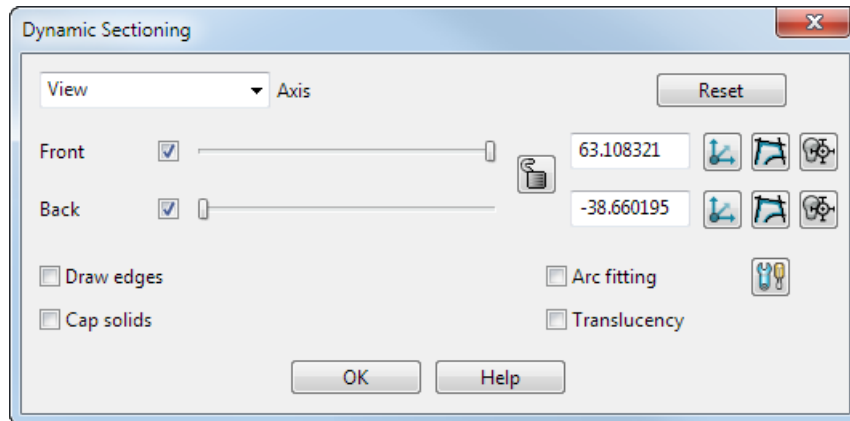
1   Select the **Axis** from the drop-down list. This determines the orientation of the **Front** and **Back** planes.
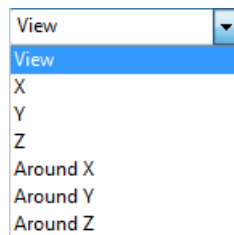
Around Y means rotate the clip plane around the Y axis of the world or active workplane. This is used for circular items that are centred on the workplane or world origin. In this case, the clip plane will give views of the cross section at particular angles.

① **Y** axis selected.

② **Around Y** axis selected.
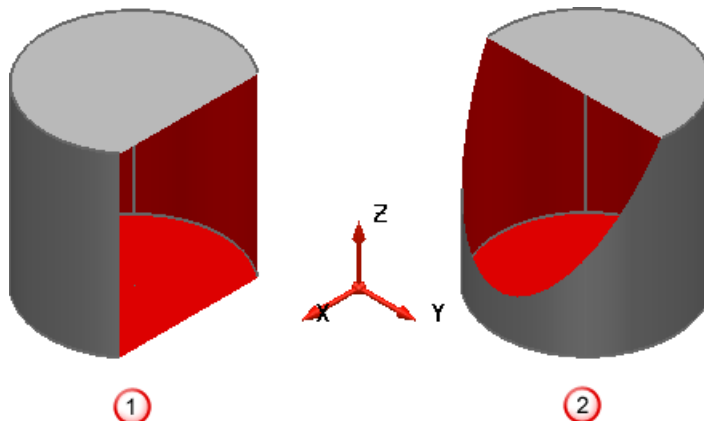
2   Specify the clipping planes to be used by selecting **Front**, **Back** or both.

3 Holding down the left mouse button, drag the mouse to move a slider along. As you move the slider, the position of the sectioning plane is moved to display a section through the model.
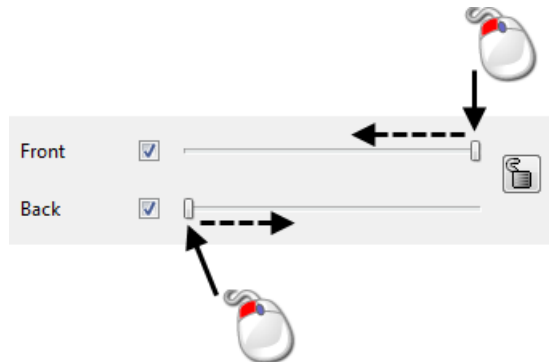
If necessary, click **Reset** to reset the clipping planes to the default position for the current view.

4 Click to lock the clipping planes, so that the **Front** and **Back** planes maintain the same relative distance. indicates that the planes are locked.

5 If necessary, change the distance of the **Front** or **Back** plane from the origin.

6 If required, click to create a workplane on the clipping plane. There are separate buttons for the **Front** and **Back** planes.

7 You can also click to position the front and back planes at a position chosen using the Intelligent cursor.

8 Click to create wireframe of the intersection of the model and the clipping plane ③. There are separate buttons for the **Front** and **Back** planes.
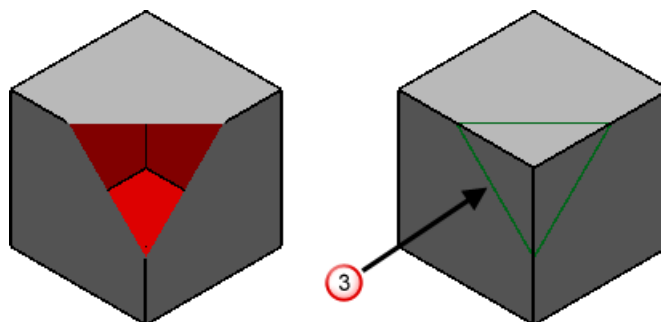
9 The following options determine the appearance of the clipped plane:

Draw edges — Select this option to draw the edges of the section, as shown in ④.
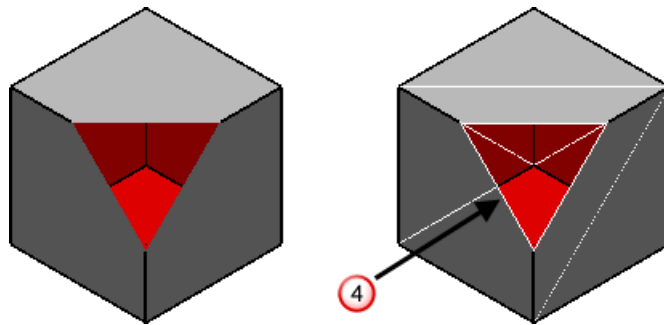
Translucency — Select this option to draw the section translucent, as shown in ⑤.
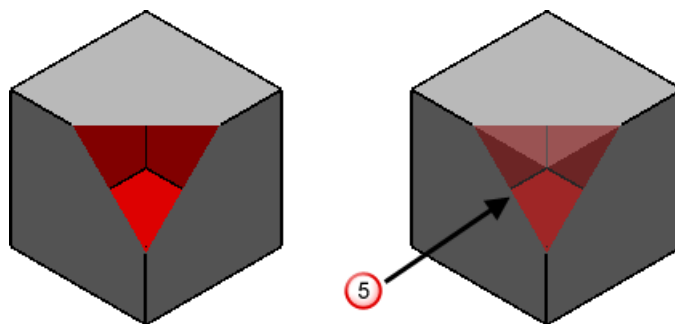
Cap solids — Select this option to display a cap on the solid where it is intersects with the clipping plane ⑥.
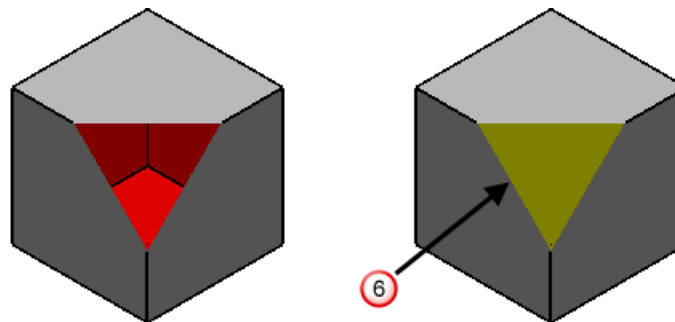
10 Select the **Arc fitting** option to automatically apply Arc fitting to any curves created using the dialog. You can also click the **Arc fitting options** button to open the Arc Fitting dialog.

11 Click **Close** to remove the dialog and reset the clipping planes.

Tools > Model Analysis > Smoothness Shading

This option lets you check the smoothness of surfaces.

Smoothness Shading shades surfaces in bands of colour to show how smooth surfaces are and how smoothly they join together. This type of shading will show up the following discontinuities in the surface.

- **Positional continuity**. Surfaces have positional continuity if there is a smooth change in position as you move across the surface.

- **Tangent continuity**. Surfaces have tangent continuity if there is a smooth change in tangent direction (surface normal direction) as you move across the surface. Sharp corners have positional continuity but not tangent continuity.

  Where there is a tangent discontinuity, the bands of colour will not join up (the bands do not have positional continuity).

- Curvature continuity. Surfaces have curvature continuity if there is a smooth change in curvature as you move across the surface. A rounded corner (as below) has tangent continuity but not curvature continuity as the curvature changes suddenly at point B from one value to another.
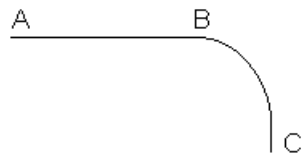
  Where there is a curvature discontinuity, the bands of colour will not have tangent continuity.
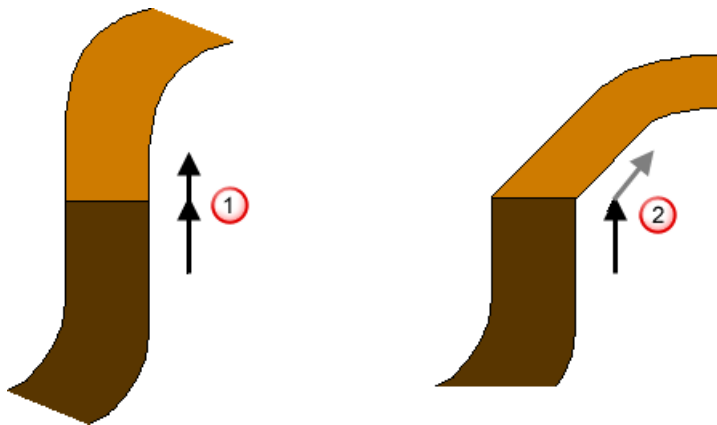
  ① Band has positional and tangent continuity at the join, so the surface has curvature continuity at the join.

  ② Band has positional continuity but not tangent continuity at the join, so the surface is tangent continuous, but not curvature continuous at the join.

Suppose we have the following surfaces on the screen.
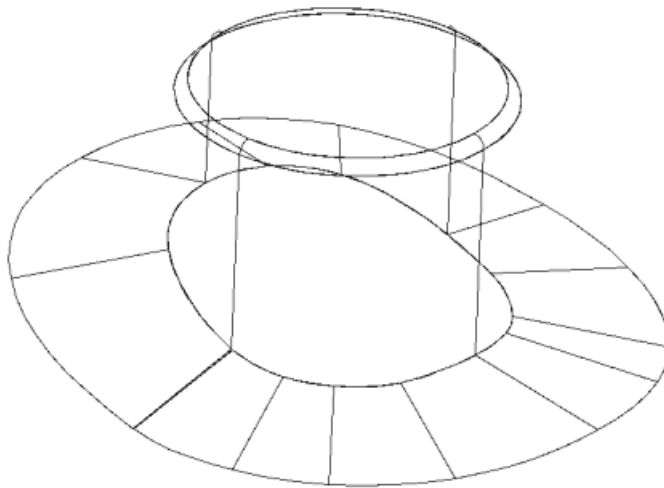
We will now orient the surfaces with respect to the light source. It is best to orient the surfaces when they are already shaded.

We will use **Smoothness Shading** to shade the surfaces and orient them in the required position.

1   From the **Tools** menu, select **Model Analysis** to display the following menu:
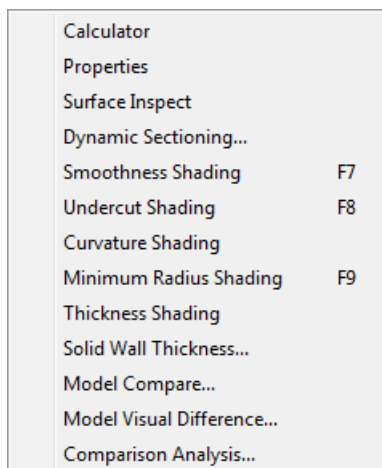
2   Click on **Smoothness Shading** to shade the surfaces.

3   Dynamically rotate the view using the middle mouse button.

Remember that the contours will show the angle between the surface normal and a light shining from the top of the screen. Try and orient the model so that the surfaces you are interested in are pointing towards this imaginary light source.

The shading will update, showing the bands of colour moving over the surfaces.

The surfaces will be shaded similar to those shown below.
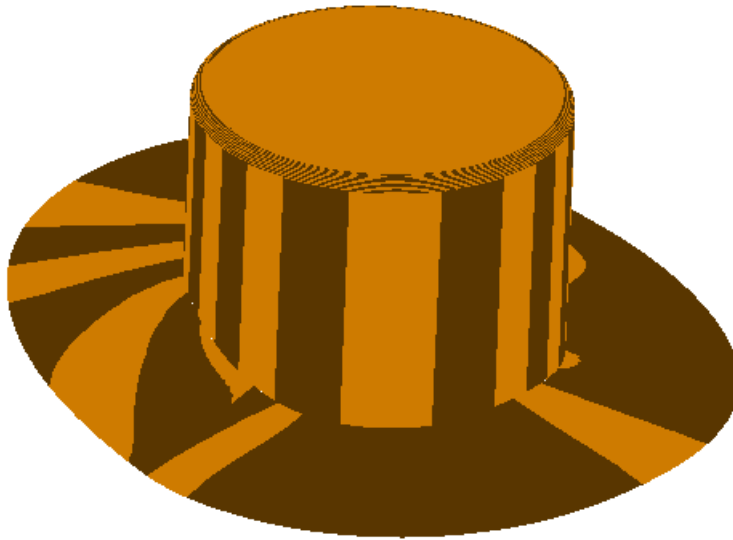
The surfaces are shaded with two bands of colour. These define how smooth the surfaces are and how smooth the join between surfaces is.

If the bands of colour flow into one another but the bands change direction, it shows the surfaces are tangent continuous (smooth) but have a discontinuity in curvature.
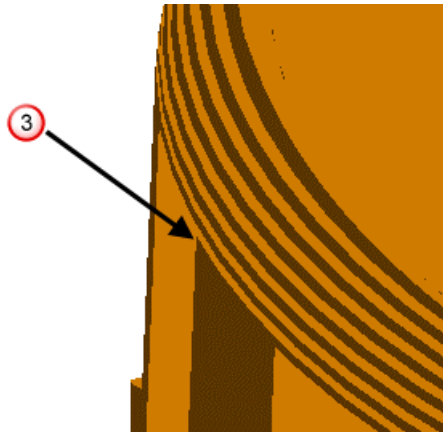
③ The bands join, but there is a change in direction.

When surfaces do not meet smoothly (with tangent continuity), you will see bands similar to the following.
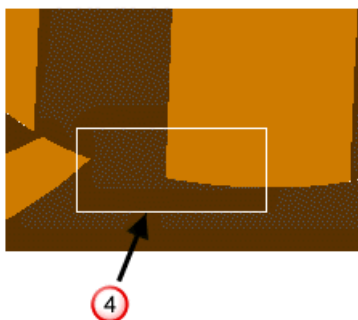
④ This area shows that the surfaces are not smoothly joined.

Surfaces which join with tangent and curvature continuity show bands similar to the following:

The colours of the bands shown are shades of colour of the default material for surfaces. If you change the colour of the material of the surface, the colours of the bands change too.

For further details, see Format > Materials (see page 32).

Tools > Model Analysis > Undercut Shading

This lets you check if a mold design has any undercut areas.Undercut areas show where it would be difficult to separate the part from the mold.

By using **Undercut Shading**, you can help reduce costs because you can check the model for any areas of undercut before manufacturing the model. If any areas of undercut are found, the surfaces can be adjusted to remove them.

Undercut shading visualises the draft angle at all points on the surface. The draft angle is measured between the surface normal and the split plane. The split plane is the XY plane of the current workspace.

Suppose we have the following mold design.
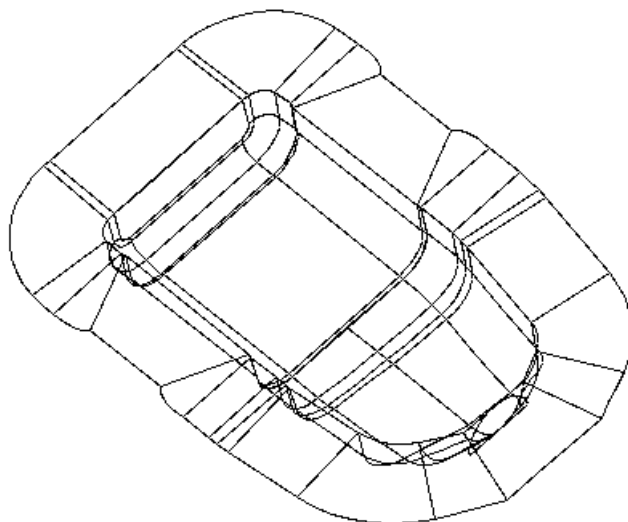
We will now use **Undercut Shading** to show areas where it will be difficult or impossible to extract our mold tool.

1 Set the required minimum draft angle using the **Draft Angle** text box on the **Surface Analysis** page of the **Options** dialog.

The **Options** dialog is displayed by choosing **Options** from the **Tools** menu.

2 From the **Tools** menu, select **Model Analysis** to show the shading options.

3 Click on **Undercut Shading**.

The model is now shaded in three colours to show areas of undercut:

Red - This warns you that the area will be undercut, because the draft angle on the model is less than the required minimum angle.

Yellow - This warns you of possible undercutting, because the draft angle is greater than, but within a given warning angle of, the required minimum angle. You can set the warning angle on the Surface Analysis options (see page 128) dialog.

Green - This area is well clear of undercutting.

Our mold is shaded as follows.
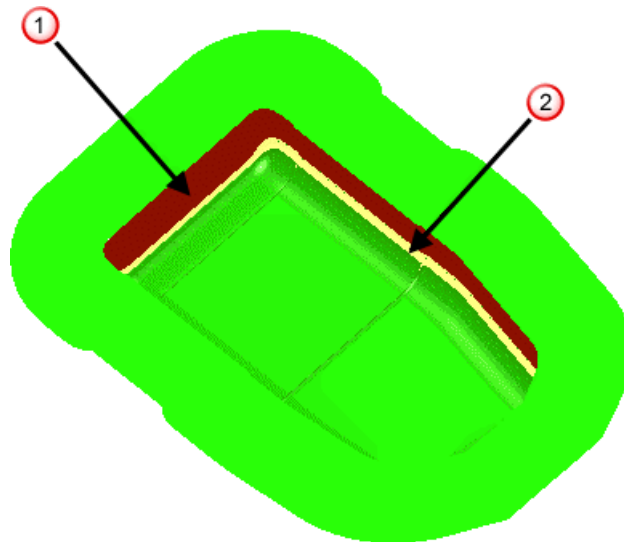
We need to adjust the surfaces in the red ① (and possibly yellow ②) areas to avoid undercutting.

Tools > Model Analysis > Curvature Shading

Curvature shading displays the curvature of a surface to help detect any defects. It can also help decide the optimum tool cutter size. Curvature shading can be considered as a form of surface inspection. It allows you to check the surface of a model in a number of ways. These include:

- highlighting surface ripples.

- showing the smoothness of the surface.

- showing how surfaces blend with one another.

This type of shading helps when designing products that need to have an aesthetic appearance. The shading indicates the level of curvature of the surface and whether the surfaces merge well with one another.
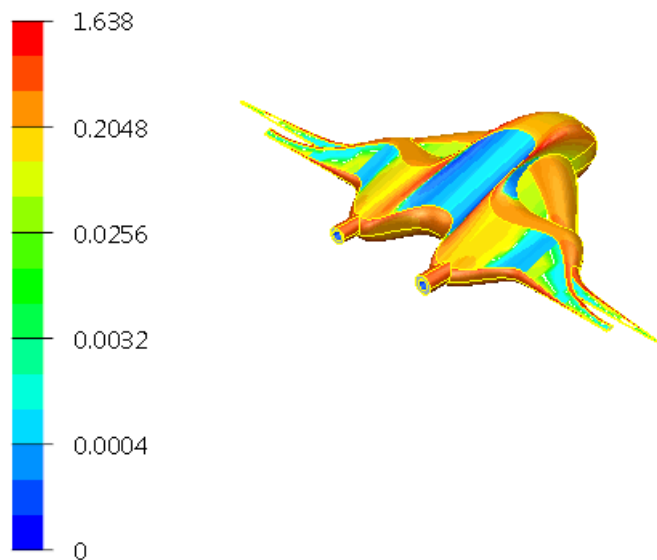
1   Select **Model Analysis** on the **General edits** flyout.

2   Click (*Model Analysis toolbar*). This is the same as selecting **Tools > Model Analysis > Curvature Shading.**

The surfaces in the model are painted using curvature shading, different colours are used to represent the different curvature on the surfaces.

Curvature at a point on a curve is defined as:

Curvature = 1/r

where *r* is the radius of the arc which fits the curve at that point. Radius *r* is also known as the radius of curvature.

Curvature maps can be hard to interpret, and it is usually the case that the variations in curvature rather than absolute values are of more interest when viewing a curvature shaded surface. Irregular variations in curvature tend to indicate poor quality surfaces.

Curvature of a point on a surface is obtained by intersecting the surface with a plane through the point to give a planar curve. The curvature of the point is then taken as the curvature of the point on the planar curve. At any point, an infinite number of planes can intersect the surface, therefore giving an infinite number of curvatures. So for each point, there will be a maximum and minimum curvature. These values are used for the curvature.

3   Specify the display using **Options** dialog. Select **Tools > Options > Tools > Analysis > Surface Analysis** to display the following options.

Type - This sets the type of curvature displayed. You can choose from one of the following:

- **Minimum curvature** - This is the minimum curvature at a point.

- **Maximum curvature** - This is the maximum curvature at a point.

- **Gaussian curvature** - This is the product of the minimum and maximum curvatures.

- **Mean curvature** - This is the average value of the minimum and maximum curvatures.

Representation Type - This controls the representation of the curvature. You can choose from one of the following:

- **Banded** - The curvature at each point is represented by a colour in a spectrum. The colours are in bands.

- **Spectrum** - The curvature at each point is represented by a colour in a spectrum. The colours change smoothly.

- **Bicolour** - This shows the positive and negative curvature using two colours, regardless of whether **Signed Curvature** is selected. The colour gets lighter as the magnitude of the curvature get higher.

In **Banded** and **Spectrum** shading, only the magnitude of the curvature is shown (unless the **Signed Curvature** option is selected). The maximum magnitude shown will be the larger of the absolute values of the minimum and maximum curvatures. The minimum magnitude shown will be the smaller of the absolute values of the minimum and maximum curvatures. The mean magnitude shown will be the mean of the absolute values of the minimum and maximum curvatures.

Scaling - Select **Log** or **Linear**. With a linear curvature scale you would see a change in the colour of the shading more clearly if the curvature of a surface changed suddenly. To see a wider range of curvatures on the surface of a model, use the logarithmic curvature scale.

Draw Scale - If selected, a colour-coded key is displayed in the lower left of the view.

Signed Curvature - If selected, the negative and positive curvature is displayed. If deselected, then only the magnitude of curvature is considered, and the sign is ignored.

Tools > Model Analysis > Minimum Radius Shading

This shading lets you specify a radius for a tool, The model is then compared with the tool size and areas where the model would be machined to size or where material would be left are highlighted.

Suppose we have the following model on the screen.
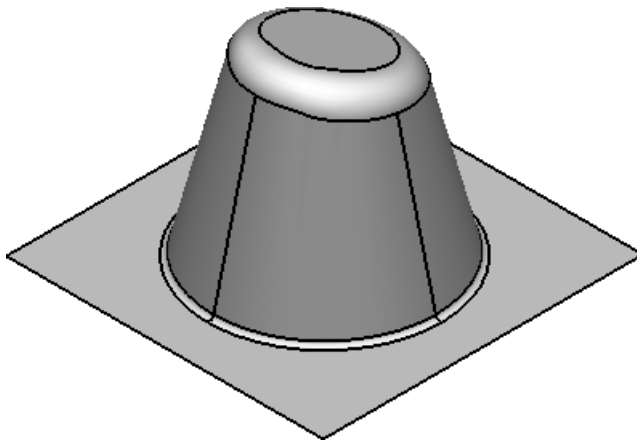
We will shade it using **Minimum Radius Shading** to show any areas where material would be left on.

1 Specify the **Cutter Tip Radius** of the tool you want to use to check the model using the **Surface Analysis** page of the **Options** dialog. (The **Options** dialog is displayed by choosing **Options** from the **Tools** menu.)

2 Click ![Tools flyout icon] (*Tools flyout*)

3 Click ![Model Analysis icon] (*Model Analysis toolbar*). This is the same as selecting **Minimum Radius Shading** from the **Model Analysis** of the **Tools** menu.
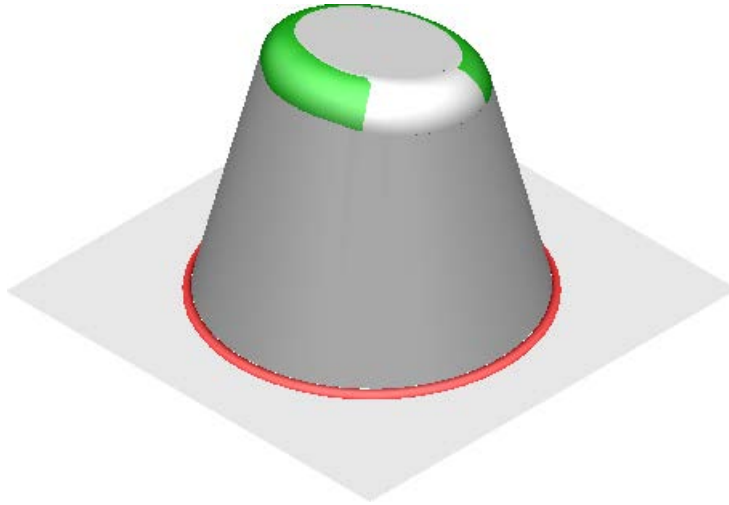
The model is now shaded to show areas that can be machined to size and any areas where material would be left on. Each colour of the shading is discussed below.

- *Red* - This shows areas where the concave radius is less than the cutter radius. Machining here leaves material on. However if you decide to machine the model from the underside then these areas will be machined to size.

- *Green* - This shows areas where the convex radius is less than the tool radius. These areas will be machined to size. However, if you decide to machine the model from the underside (for example, machine a cavity as opposed to a punch) these areas will leave material on.

- *White* - These areas will machine to size.

Tools > Model Analysis > Thickness Shading

Use this option to apply thickness shading to your model. Thickness shading enables you to visualise the thickness of solids. This is useful for identifying thin regions of a model.

The thickness is measured by a ball that is placed inside the solid and made as large as possible without passing through any of the surfaces. As this ball rolls around inside the solid it grows or shrinks as appropriate in order to fill the available space. The front face of the model below has been cut away to show the measuring ball rolling around inside.
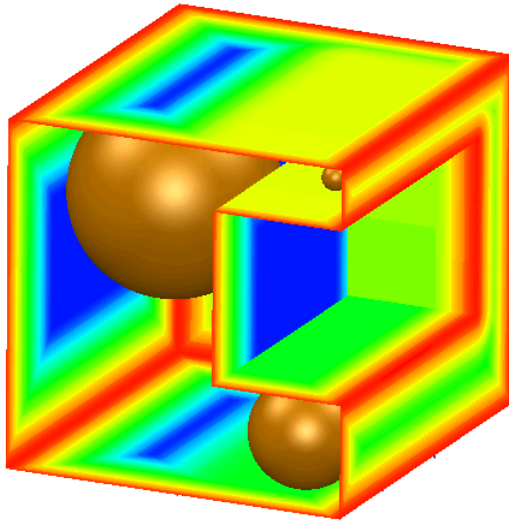
The colour displayed at each point on the surface of the solid represents the diameter of the ball when it was touching that point.

Using Thickness Shading

1   Make sure you have the relevant solid selected.

2   Click (*Tools flyout*)

3   Click (*Model Analysis toolbar*)

   This is equivalent to selecting **Thickness shading** from **Model Analysis** option of the **Tools** menu.

   There will be a pause whilst the thicknesses are calculated; the length of time taken will depend on the size of the model.

4   Use the Thickness Colour Scale dialog (see page 56) to dynamically rescale the colour thickness display.

Thickness Colour Scale dialog

Use this dialog to dynamically rescale the colour thickness display
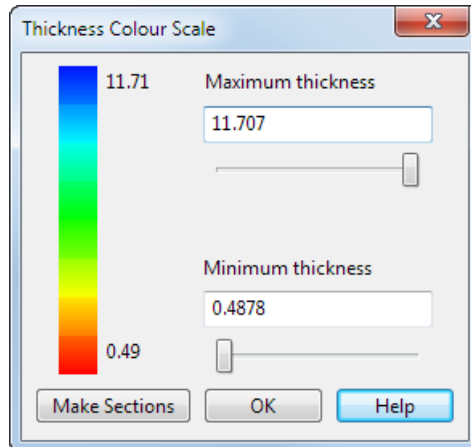
Maximum thickness — Regions of the model that are thicker than the value shown in this box are displayed in blue. Adjusting this value by using the slider will be reflected in the colouring of the model.

Minimum thickness — Regions of the model that are thinner than the value shown in this box are displayed in red. Adjusting this value by using the slider will be reflected in the colouring of the model.

Intermediate thicknesses are represented according to the colour scale in the dialog.

Make Sections — You can create dynamic wireframe sections of the areas of the model.

Before creating the sections, ensure that you have the relevant plane defined. The plane through which the sections are created will be either the active workplane (if selected), or the plane of the graphics window when you click **Make Sections.**

Selecting **Make Sections** displays the Section dialog (see page 43). The position of the section can be varied using the slider on the dialog. In the example on the right, the slider was set at 50%.
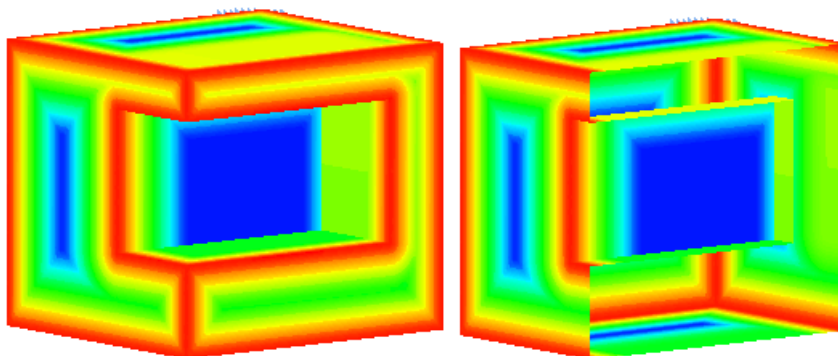
Tools > Model Analysis > Solid wall thickness

Use this option to find the wall thickness of a hollow solid.

This displays the **Wall thickness** dialog so you can define a series of sections relative to the active workplane.

For further details see Wall thickness dialog.

Tools > Model Analysis > Model Compare

Use this option to compare two models and select the differences in one of the models.

For further details see Comparing two models.

Tools > Model Analysis > Model Visual Difference

Model Visual Difference is a tool that gives a visual representation of the differences between two versions of a model. It considers surfaces and solids and also symbols created by importing triangle files

The visual differences are worked out using the total shape of the two models and do not depend on how the shapes are constructed. This can be shown by creating the two models below.
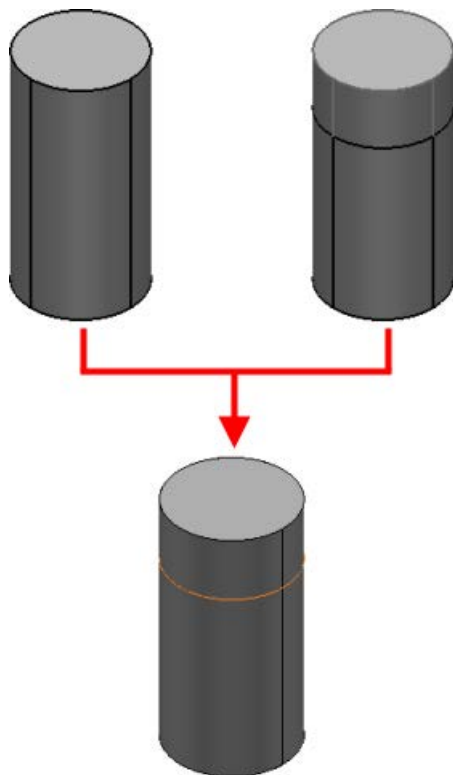
The first model is a single solid cylinder. The second model was constructed by joining two solids to create a cylinder of the same size. If you use **Model Visual Difference** on these models, no difference is detected.

For further details, see Using Model Visual Difference (see page 58).

Using Model Visual Difference

1 Create the model below. Save the model as **model-visual-diff.**
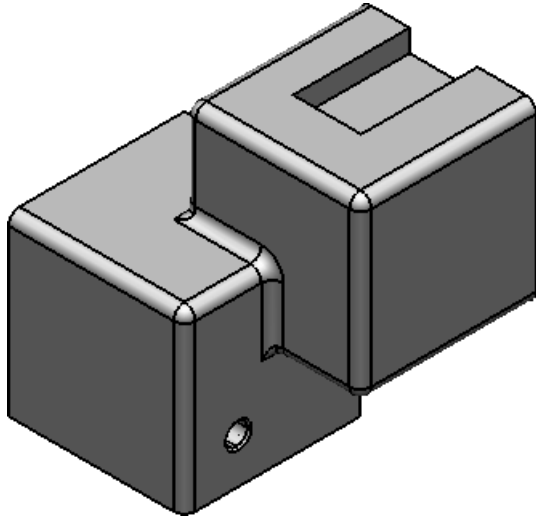
2 Copy the model into a new model. Save this model as **model-visual-diff2.**

3 On the second model,

- Delete the fillet indicated below.
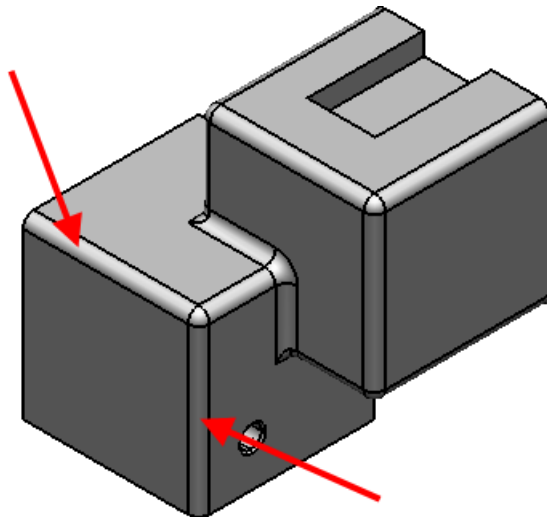
- Increase the size of the hole to *15.*

4 Select **Model Visual Difference** from the **Model Analysis** toolbar.

The **Model Visual Difference** dialog is displayed:
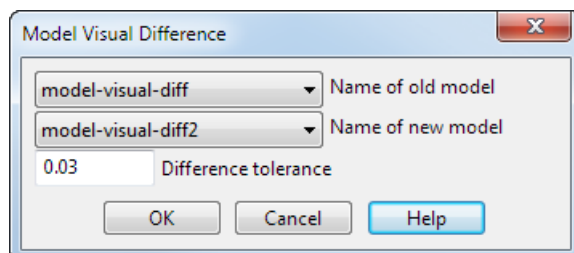
5 Select the **Name of old model** from the drop down list.

6 Select the **Name of new model** from the drop down list.

7 Click **OK**.

The **Visual Difference** window is displayed. This window shows the differences between the models. Although you can navigate around the window, you cannot select anything in it and any change that you make to the models at this point are not reflected in existing Visual Difference window.

> *If you modify either model and want to get a new view of the difference, you must create another Visual Difference window. There can only be one Visual Difference window, so creating a new window will destroy the old window.*

1 Rotate the model in this window until your model looks like the one below.
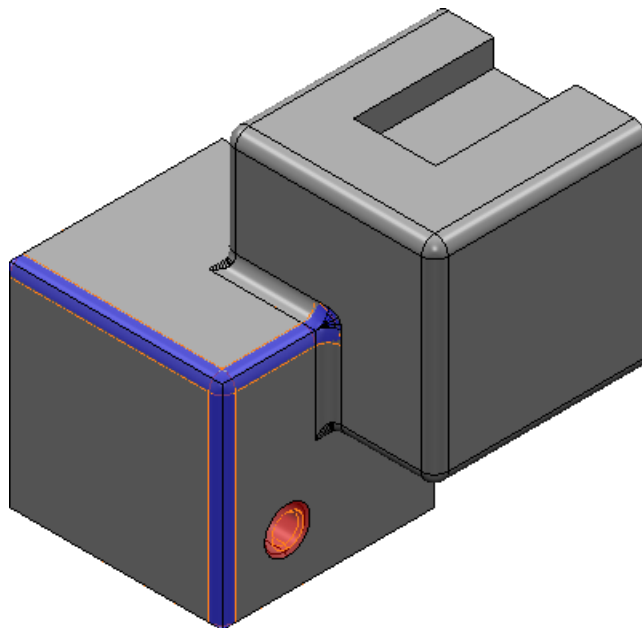
You will notice the following:

- The area from where the fillet was removed is shown in blue. This indicates that material has been added to the model.

- The change in the size of the hole is shown in red. This indicates that material has been removed from the model.

These colours are the default colours for addition and removal of material. You can change the default settings using the **Options** page. For full details, see Model Visual Difference options (see page 127).

Tools > Model Analysis > Comparison Analysis

(available with PowerSHAPEPro licence only)

Use **Comparison Analysis** to find differences between any two objects, using the Comparison Analysis dialog (see page 60). The functionality is also available from the **Model Analysis** toolbar, but the button is only displayed if you have a PowerSHAPE Pro licence.

Comparison Analysis dialog

1 Select the **Model Analysis** button from the **Editing and Analysis** flyout to display the **Model Analysis** toolbar.
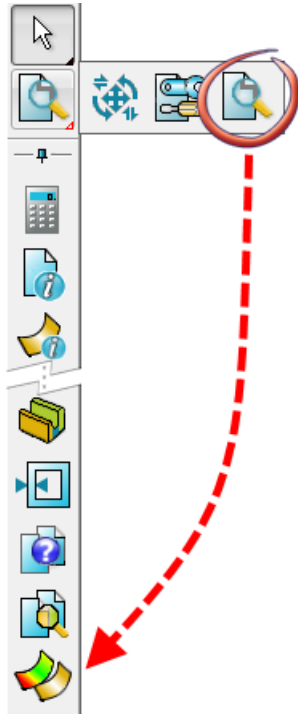
2 Click the **Comparison Analysis** button to display the **Comparison Analysis** dialog.

You must have a PowerSHAPE Pro licence.
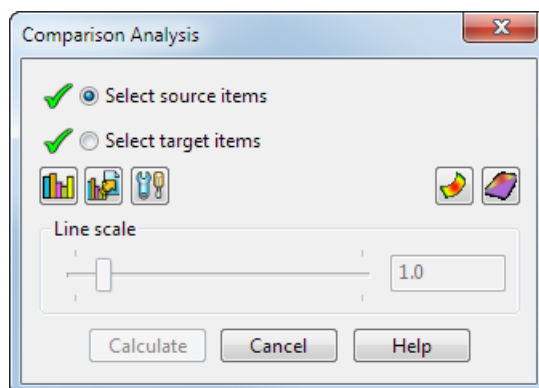
3 Select the first item to compare against (source item).

4 Select the second item to compare to (target item).

5 Click **Calculate**.

The differences between the objects are shown as an error map.
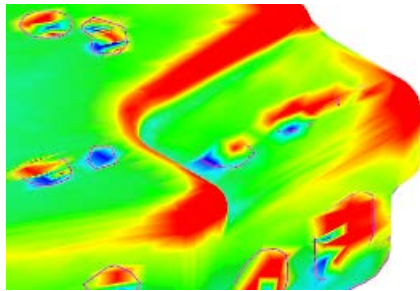
6 Use the options on the dialog as follows:

- Click the **Statistics** button to display the deviation statistics.
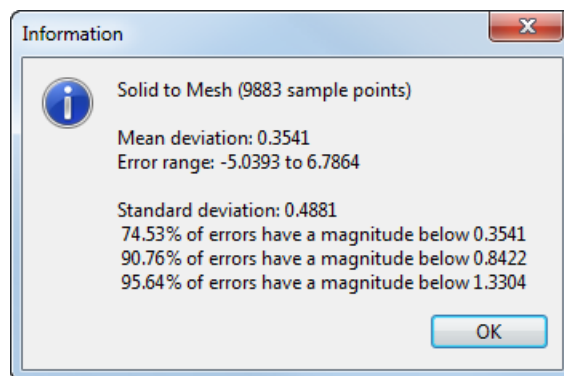
- Click the **Export** button to export differences to using the **Export To** dialog:

  You can export differences between objects to ASCII text file or STL file.

- Click the **Options** button to change the comparison analysis options (see page 64).

- Click the **Curve Deviations** button to display the curve deviations (see page 62). The option is deselected by default.

- Use the **Line scale** slider or enter a value in the field to define the curve deviation line widths. This option is only available if you have clicked the **Curve Deviations** button.

- Click the **Shade**   buttons to toggle shading on or off. The model below shows:

  ① Difference shading off.

② Difference shading on.
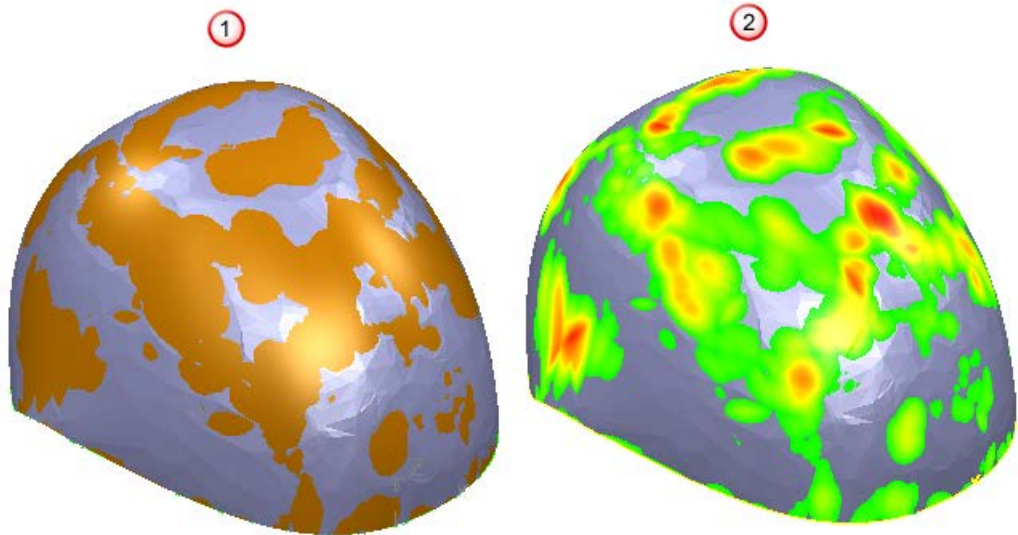
Display curve deviations

1. Select one of the following:
   - curves dependent on an object
   - curves and the object
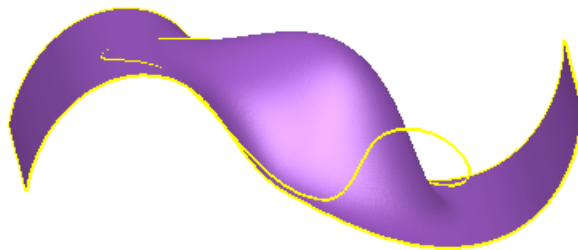
2. Click the **Comparison Analysis** 🔷 button to display the **Comparison Analysis** dialog.

   📝 *You must have a PowerSHAPE Pro licence.*
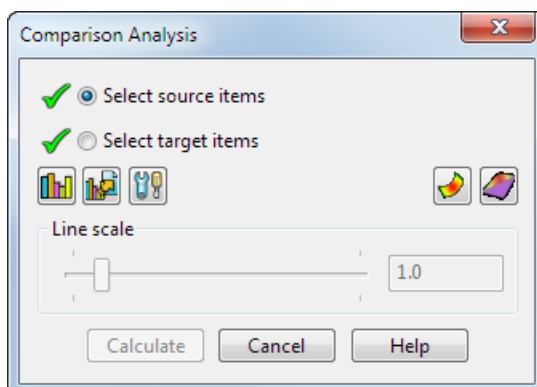
3 Click the **Curve Deviations** button to display the deviations.
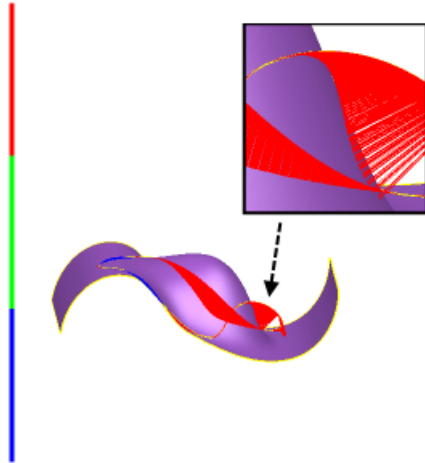
4 Use the **Line scale** slider or enter a value in the field to define the width of the curve deviation lines.

*This option is only available if you have clicked the **Curve Deviations** button.*

5 Click the **Curve Deviations** button again to hide the curve deviation lines.

Comparison Analysis options

Use this page to specify the parameters to be used in **Comparison Analysis**.

- **Display legend** — Select this option to show or hide the error legend in the graphics window.

- **Always display errors** — Select this option to automatically show or hide deviations after they have been calculated.

- **Display error under cursor** — Select this option to show or hide the numerical deviation beneath the cursor when you hover your cursor over the comparison.

- **Colour bands** — Select an option from the list to change how the deviations are displayed on the model:

    Continuous — The deviations are displayed using a continuous scale.

    Two — The deviations are displayed using two distinct colours.

Three — The deviations are displayed using three distinct colours.

Five — The deviations are displayed using five distinct colours.

Ten — The deviations are displayed using ten distinct colours.

Twenty — The deviations are displayed using twenty distinct colours.

Forty — The deviations are displayed using forty distinct colours.

*Colour bands is not available if **Display errors to tolerance** is selected.*

- **Width of error lines** — Enter a value to define the width of the error bands shown on the model when curve deviations are displayed.

- **Display errors to tolerance** — Select this option to display the deviations in two or three colour bands. The bands are set according to the general tolerance.

  This option is deselected by default.

  If **Display errors to tolerance** is deselected, select the required number of **Colour bands** from the list.

Colour bands	Continuous ▾
	Continuous
	Two
	Three
	Five
	Ten
	Twenty
	Forty

- **Display absolute range** — Select this option to display the absolute (modulus) values of deviation. That is the range of errors from **0** to the maximum value.

Deviation

1
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0

- **Set maximum value to maximum error** — Select this option to set the maximum value of the error range to the largest deviation.

- **Maximum value of error range** — Enter a value into the field to define the maximum value of the error range. In the following example :

  - **Maximum value of error range** is set **1**.

  - **Colour bands** are **Continuous**.

Deviation

1
0.8
0.6
0.4
0.2
0
-0.2
-0.4
-0.6
-0.8
-1

*This option is unavailable if **Set maximum value to maximum error** is selected.*

- **Sample Point Density** — Use the slider to control the number density of points used to calculate differences between your models. A larger **Sample Point Density** produces more accurate results, with a longer computation time. A smaller **Sample Point Density** produces less accurate results, with a shorter computation time.

Tools > Selection information

Use this option to open the **Selection Information** dialog, which displays information about selected objects. You can find out the number of items in a selection by hovering the mouse over the **Select** flyout to display the tooltip.

1   Select one or more objects.

2   Click ![info icon] (*Selection flyout*) or select **Tools > Selection Information**.

If a single object is selected, then information about that object is displayed in the Selection Information dialog (see page 68). The information is also displayed in the **Command** window.
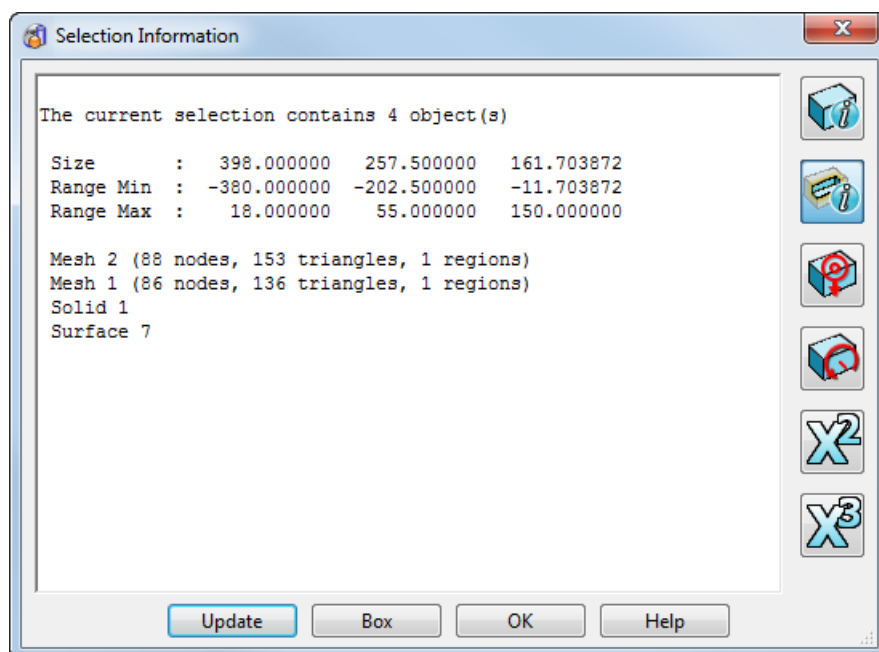
Selection information dialog

Use this dialog to display information about the objects that are selected.
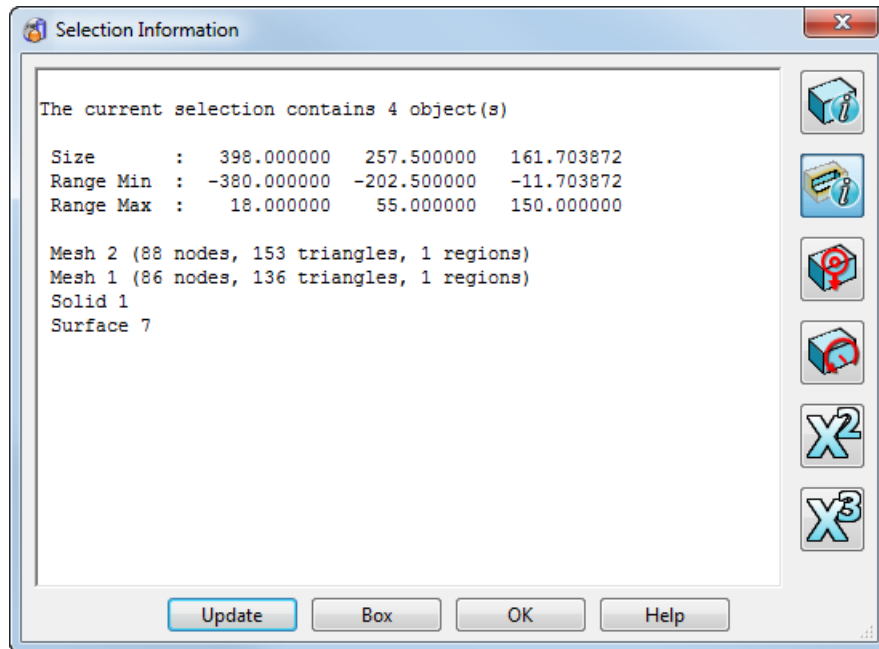
As well as basic information about the selection, the following information will be displayed (where appropriate):

- If the object is a surface, further detailed information will be given about any sub-selection (such as curves or p-points).

- If a group of objects is selected, the dialog tells you how many objects are selected and the type and name of each object. You can find out the number of items in a selection by hovering the mouse over the **Select** flyout to display the tooltip.

- The **Size** of the bounding box containing the objects, and the minimum and maximum coordinates of the bounding box.

- Any selected that is badly trimmed.

- The centre of gravity for a multiple selection of components.

The following buttons allow you to see additional information:

 Toggle detailed information. When a single solid is selected, the names of the currently selected solid faces and features is displayed

 Toggle sub item information.

 Toggle centre of gravity.

 Toggle moment of inertia.

X^2 Toggle the display of the area of the selection.

X^3 Toggle the display of the volume of the selection.to display the volume of the selection.

Update — You can change the selection of objects and click **Update**. This updates the information to the new selection.

Box — This displays a bounding box around the current selection. If a bounding box is already displayed, it undraws. You can create the bounding box around the current selection using the Selection box option on the line creation menu.
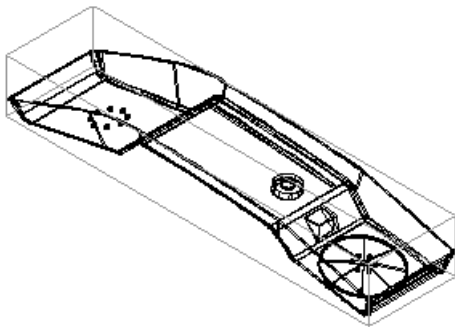
OK — Close the dialog. If a bounding box has been drawn, this is removed.

> *Use the Selection information context menu to copy, print or save the information displayed in the dialog.*

Tools > Component Drawings

Use this option to create component drawings for standard and non-standard components using a template.

For further details, see Creating components drawings.

Tools > Create Titleblock

Use this option to add a title block to a drawing.

For further details, see Creating a title block on a drawing.

Tools > Create Movie

Use this option to create an *.avi* file that represents a 360° rotational view of the objects you have selected. If nothing is selected, the movie is created using everything in the graphics window.

To create a movie:

1   Select **Create movie** from the **Tools** menu.

2   Use the options on the **Movie Creation** (see page 70) dialog to set options that control options such as the frame rate, axes of rotation and movie duration. You can also choose to view the object from different angles.

Movie Creation dialog

Use this dialog to adjust the options associated with creating a movie.
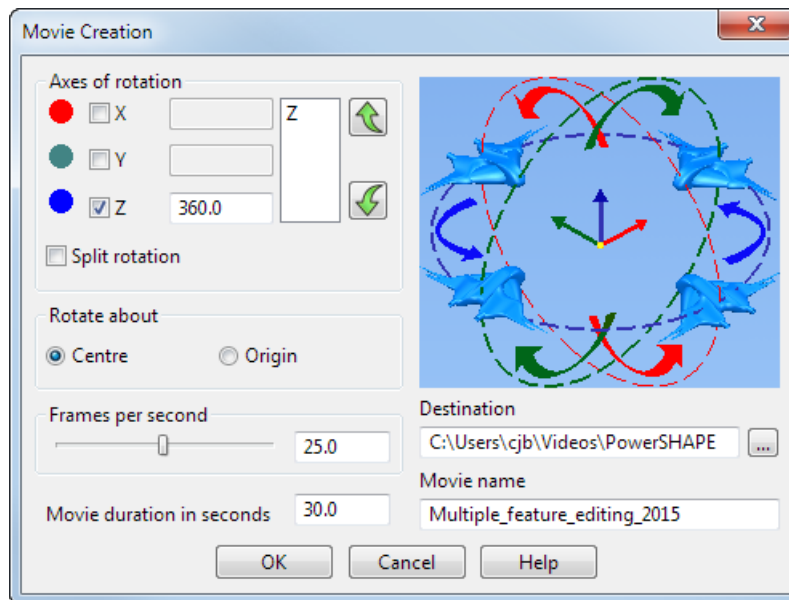
1   Use the options in the **Axes of rotation** section of the dialog to define the axes and angle of rotation as follows:

   a   Select the required axes of rotation (any combination of X, Y, Z).

   b   For each selected axis, the default angle of rotation is 360°. Enter a different angle if required.

   c   Set the sequence of the selected axes using ⬆ and ⬇

   d   select **Split rotation** to half the value entered as the angle of rotation and rotate the model by that amount in the positive direction and the negative direction.

   For example, if you enter 120° as the rotation angle and this option is *ON*, the model will be rotated 60° in the positive direction and 60° in the negative direction.

2   Select the **Rotate about** option to define the point of rotation:

   Centre rotates around the centre of the bounding box enclosing all the selected objects.

Origin rotate about the origin of the active workplane.

3 Use the slider to specify the number of **Frames per second**. Alternatively, enter a value. The higher the number, the better the quality of the movie, but the longer it takes to create the movie. A rate of 25 provides a good balance between quality and creation time.

4 Enter the **Movie duration in seconds**.

5 Enter the **Destination** for the saved .avi file or click ⊡ to open the **Browse for folder** dialog.

6 Enter the **Movie name**. The default name is the name of the model.

7 Click **OK** to create the movie using the values entered in the dialog. If no values have been entered, default values will be used.

Tools > Login

This option lets another user log in to the program without closing the application. If any models are open, you are prompted to save and close them.

1 From the **Tools** menu, select **Login** to display the **Login** dialog.

2 Fill in the Login dialog *(see page 71).*This dialog also appears when you start the program if you are not a registered user or need to enter a password.

Login dialog

This dialog is used to log in.
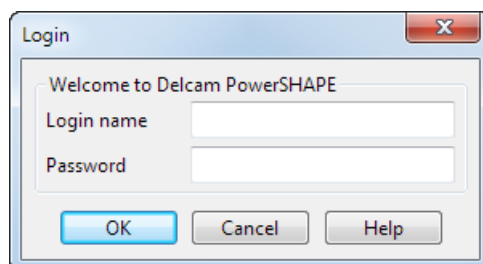
Login name — Type in your username.

Password — Type in your password.

OK — Start the program. The username and password you have entered are checked. If the username and password do not match, you are prompted. Type in your **Login Name** and **Password** again. If, after six attempts, you still cannot provide a valid username and password, the login is automatically aborted.

Tools > Preview Mode

This option lets you use functionality that is still under development and not released.

You can only preview functionality if you have security levels 1 and 2. Usually, system administrators have security level 1 and design office managers have security level 2.

> *Warning: Be careful when using the additional functionality that is available in preview mode. This is functionality that is under development.*

Tools > Release Licences

This allows you to release licences.

A floating licence system is used, so licences for particular areas of functionality (for example, Drafting) are "grabbed" by the user when needed.

Release licences allows you to "free-up" a licence when finished with it. The functionality covered by the licence is then available for another user. This allows a network of users to share a (relatively) small number of licences.

When you have finished with a licence, select **Release licences** from the **Tools** menu.

After a short delay, the licence will be free for others to use.

> *Licences are released when the program closes.*

Tools > Vouchers > Manage account

If you are are importing from or exporting from PowerSHAPE-e, use this option to manage your voucher account.

Tools > File Doctor

Use this option to check and repair model files and the shared database.

> *File Doctor contains powerful tools. If it is used incorrectly, serious damage can be caused to your model or to your shared database. Consult Delcam Customer Support before using File Doctor on the shared database and always make a Full system backup before proceeding. Ensure you have back-up copies or duplicate models before using File Doctor on models.*

For further details, see File Doctor.

Tools > Compress Model

This lets you manually compress a model and can free disk space. This option compresses the space used by surface data and frees space allocated to deleted objects. It is useful if your model contains many surfaces or you have deleted many objects from the model.

By default, a model is automatically compressed when it is closed. If you wish to turn off automatic compression, the **Compress on close** on the **Tools > Options > File > Model** dialog (see page 91) needs to be *OFF*.

1   Open the model with write access.

2   Make sure there are no unsaved edits.

3   From the **Tools** menu, select **Compress Model**.

4   Once the model is compressed, a dialog appears.
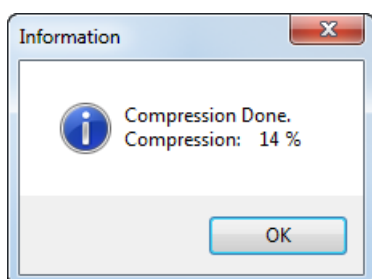
This dialog gives the percentage of compression in terms of the reduction in size of the model after compressing the space used by surface data. The percentage does not include freed disk space, that was allocated to deleted objects.

The value for maximum compression is set in the **Tools > Options > File > Model options** dialog

Tools > Customise

Use the options on this submenu to customise toolbars, menu shortcuts, and file paths.

1   From the **Tools** menu, select **Customise Toolbar** to display the following menu:
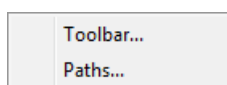
2   Select the option you require:

▪   Toolbar (see page 74).

▪   Paths (see page 74).

Tools > Customise > Toolbar

Use this option to create user-defined toolbars and menu shortcuts using the **Customise** dialog.

For further details see:

- Customise dialog - toolbars (see page 189).
- Customising menu shortcuts (see page 180).

Tools > Customise > Paths

Use this option to customise the default locations where PowerSHAPE looks for additional configuration files, and add your own locations using the **Customise Paths** dialog (see page 74).

Customise Paths dialog

Customise Paths enables you to define the search paths that are used to find custom toolbars, bill of materials and schedules, shortcuts, macros, models, and electrodes.

Use this dialog to add, delete, and re-order the pathnames that are searched for within each type.
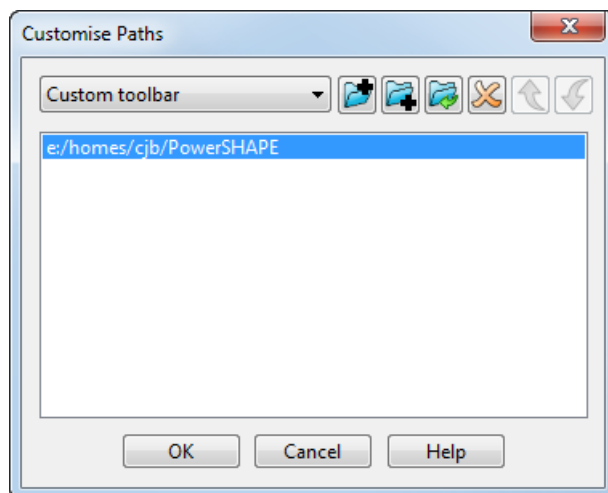
Use the following options on the dialog to customise the paths:

Path List — Select the type of path.

Add Path to Top of List — Click to add a new path to the top of the path list.

Add Path to Bottom of List — Click to add a new path to the bottom of the path list.

Edit selected path — Click to edit the path name.

Delete — Click to delete the selected path from the path list.

![Move Up icon] **Move Up** — Click to move the selected path up the path list.

![Move Down icon] **Move Down** — Click to move the selected path down the path list.

Tools > Options

The **Options** dialog enables you to set the options that affect the overall operation of the program.
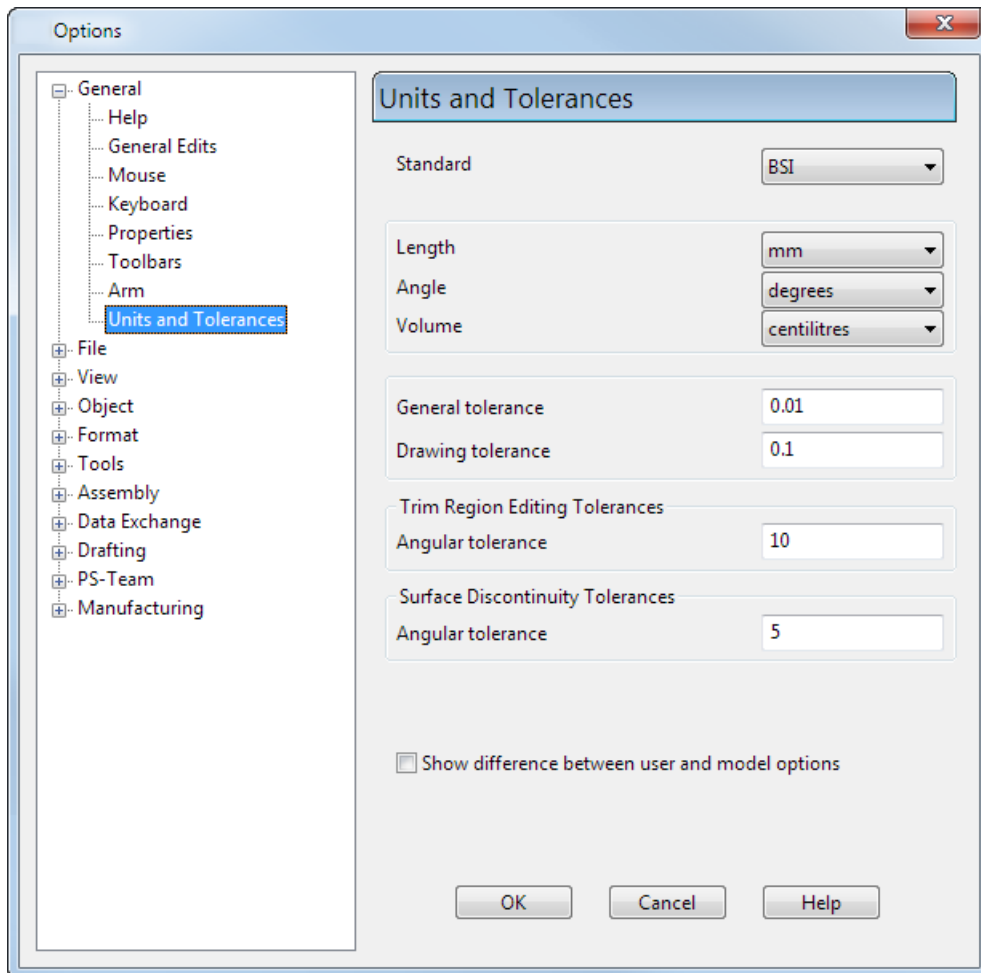
The **Options** dialog contain many options that you can select and adjust to suit your needs.

Use the tree (on the left of the dialog), to select a page. You can expand and collapse branches of the tree by clicking the + and – icons. Each branch of the tree has a separate page.

The **OK**, **Cancel**, and **Help** buttons are common to all pages.

OK — Click this button to apply the settings and close the dialog. The current model and any new model you create use these settings. Existing models use the settings that were saved with the model.

Cancel — Click this button to close the dialog without saving any changes.

*The **Undo** command does not undo settings of the options on the **Options** dialog.*

General options

The following options pages are grouped under the **General** category:

- **Help** (see page 77)
- **General Edits** (see page 78)
- **Mouse** (see page 80)
- **Keyboard** (see page 82)
- **Properties** (see page 83)
- **Toolbars** (see page 86)
- Arm (see page 87)
- **Units and Tolerances** (see page 88)

Help options

Specify the help options using the **Help** dialog.
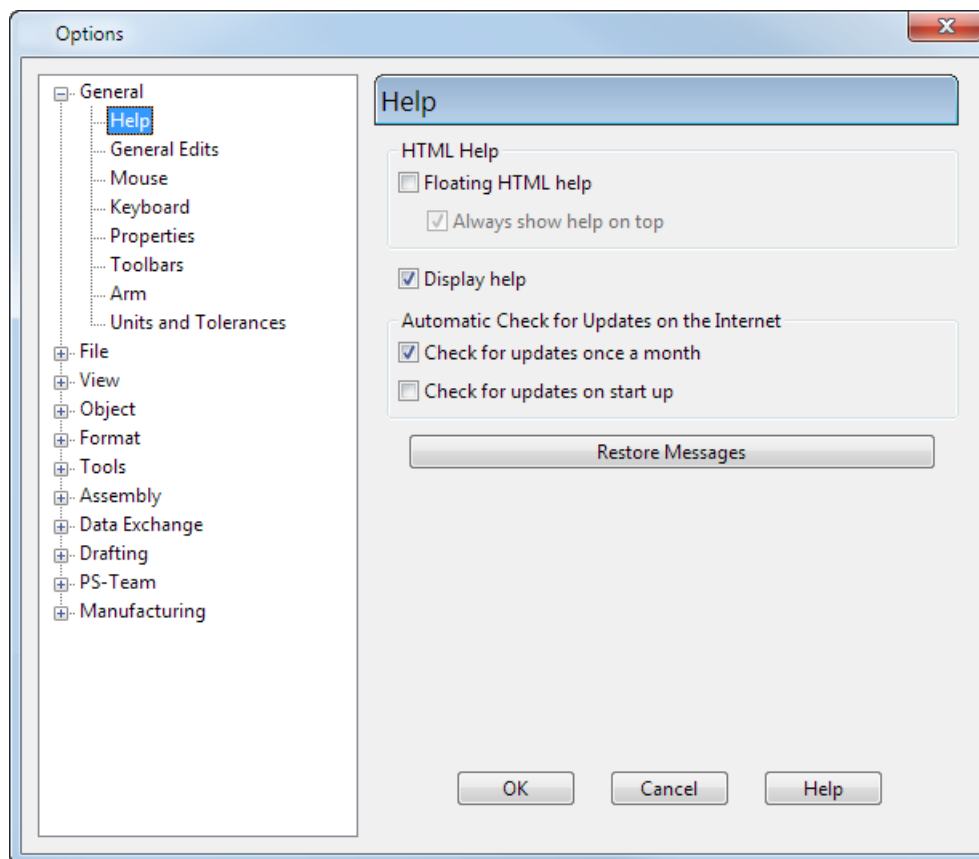
Floating HTML Help — If on, the help is displayed in a separate floating window. If off, the help is displayed in the browser window in PowerSHAPE.

Always show help on top — (Only available if **Floating HTML Help** is on) If on, the floating window is displayed in front of the PowerSHAPE window. If off, the floating window can be hidden behind the PowerSHAPE window.

Display help — If on, help tips are displayed when you hover the cursor over items on the user interface.

Automatic Check for Updates on the Internet

Check for updates once a month — If on, a check is made on *updates.delcam.com* each month. The date of the check for updates will be based on the date of the installation of the PowerSHAPE release.

When the relevant date is reached,

- You receive a message that the Delcam website will be accessed to look for updates.

- A check is made on your entitlement to updates, based on system security information.

- The list of updates to which you are entitled is displayed. Only the updates that have been issued since your current version are available to you.

If you prefer, leave this option unchecked and do a manual check for updates by selecting **Check for Updates on the internet** from the **Help** menu.

Check for updates on start up - If *ON*, a check is made on the Delcam website for any available updates every time you start the product.

Restore Messages — This option will allow messages to be displayed that have previously been suppressed by clicking on the **Do not show this message again** option in a dialog.

General Edits options

Use this options page to specify how to handle dependencies and parameters when copying items.

1   Select **Tools > Options > General > General Edits.**
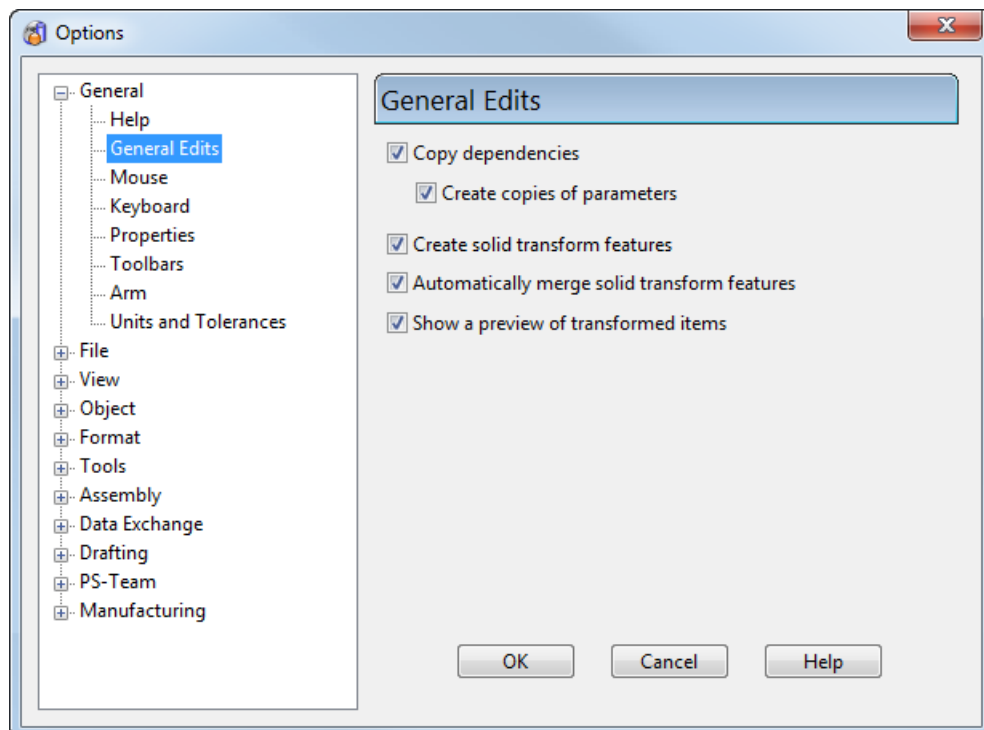
2   Specify the **Copy dependencies** and **Create copy of parameters** options:

- **Copy dependencies** — If selected, associated dependencies are copied when copying items. If deselected, associated dependencies are not copied when copying items.

- **Create copies of parameters** — If selected, a copy of the parameters is used when dependencies are copied. If deselected, existing parameters are used when dependencies are copied. This option is only available when **Copy dependencies** is selected.

3 Use **Create solid transform features** to specify if transform features will be created when using **Move**, **Mirror** or **Rotate** when one or more solids are selected.

- If selected (default), transform features will be created when using **Move**, **Mirror** or **Rotate** if one or more solids are selected.

  If **Copies** are not created, the new transform feature appears at the top of the solid's history tree; a history tree is created if the solid has no existing history tree.

  If **Copies** are created, the original solids are left unchanged; the new solids have history trees with transform features at the top.

- If deselected, transform features are not created.

4 Use **Automatically merge transform features** to mergeadjacent transform features automatically. This option is selected as default.

5 Use **Show a preview of transformed items** to control the way transformed items are displayed. This option is selected by default.

When the option is selected:

- ✔ and ✘ buttons are displayed on the editing toolbars.

- a graphical preview of the transformed items is displayed.

When the option is deselected:

- ✔ and ✘ are not displayed on the editing toolbars.

- no graphical previews are displayed.

- the transform is applied immediately when the user inputs a value or a point.

Mouse options

This controls the mouse defaults.
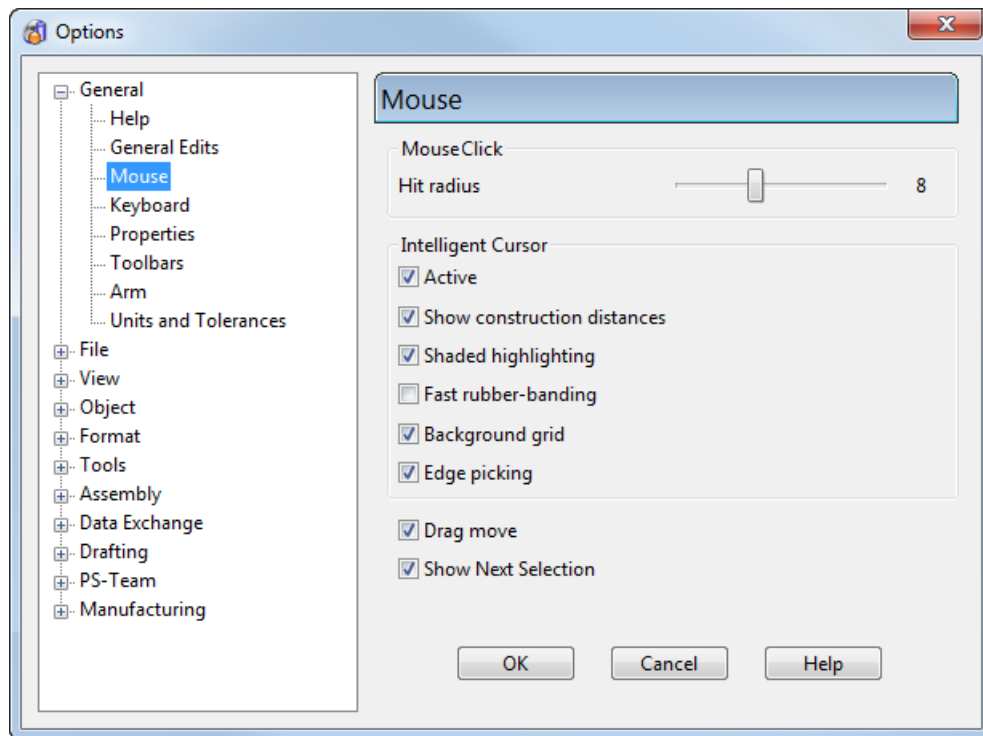
Mouse Click

Hit Radius — This is the region (in pixels) around the cursor that recognises when the cursor has hit an object. Setting it to zero means you have to accurately hit the object before you can select it. Setting it to a high value makes it easier to hit an object. However, on complex models, you may have to zoom more often to avoid selecting the wrong object.

Intelligent Cursor

Active — Select this option to turn the **Intelligent Cursor** *ON* (default). The **Intelligent Cursor** provides dynamic assistance when you are selecting existing objects and/or positioning new objects. When active, PowerSHAPE highlights objects and snap points, and generates temporary construction lines as the cursor moves over the model. The **Intelligent Cursor** also:

- finds intersections between wireframe items and a surface or solid.

- snaps to the projected intersection of wireframe items and surfaces.

Show Construction Distances — This is only available when the **Intelligent Cursor** is active. It provides stepped graduations along objects such as lines and arcs to which the cursor can snap. It displays a dimensional value which changes as the cursor moves along objects. You can then snap to the displayed dimension.

The units change according to the zoom level. Zoom in to see finer graduations. Zoom out for more coarse graduations.

Shaded highlighting — If *ON*, a shaded object is highlighted when you move the cursor over it. If *OFF*, a shaded object is never highlighted.

Fast rubber-banding — Use this to switch between faster rubber-banding and higher quality graphics. Deselect this option to improve the rubber-band graphics and use the correct construction colours.

Background Grid — If *ON*, an invisible grid is used to round any points that are entered using the cursor. The amount that a position is rounded depends on the scale of the view. If this is unchecked, the position of the point is not rounded.

Edge picking — If *ON*, you can select edges of surfaces, solids and wireframe. The edges can then be used instead of a composite curve for creation and editing. This option is *ON* by default.

Drag Move — This allows you to select and move an object. If *ON*, you can move an object to a new position by dragging it with the cursor. If *OFF*, you can select an object but not move it. This is useful when dealing with complex models where you do not want to accidentally move objects when selecting them.

Show Next Selection — This option is used to aid selection of objects that intersect. This displays a dialog showing the name of each selectable object in the model that intersects with the currently selected object.

In the example below, there are two possible surfaces that can be selected when an intersection point is selected.
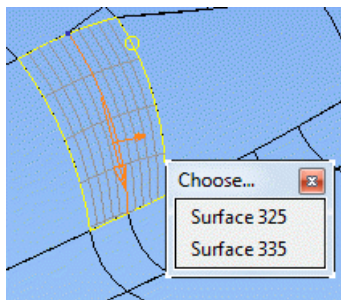
When you hover over the option, the corresponding item is highlighted. Select the required surface from the list. If no selection is made, the selection options are removed from the screen.

A variant of this, the **Next Selection** option, is displayed on context menus and is available on the **Select** options of the **Edit** menu. Choosing this option does not immediately display the dialog, but automatically selects the next item in the selection list and then displays the dialog. The shortcut **Alt+N** also activates **Next Selection**.

See Also

Turning the Intelligent cursor on or off .

Keyboard options

This page contains the options that are related to the behaviour of functionality controlled by the keyboard:

- Options related to using the up arrow ↑ and down arrow ↓ keys.
- The **Zoom factor** for the **Tab key**.
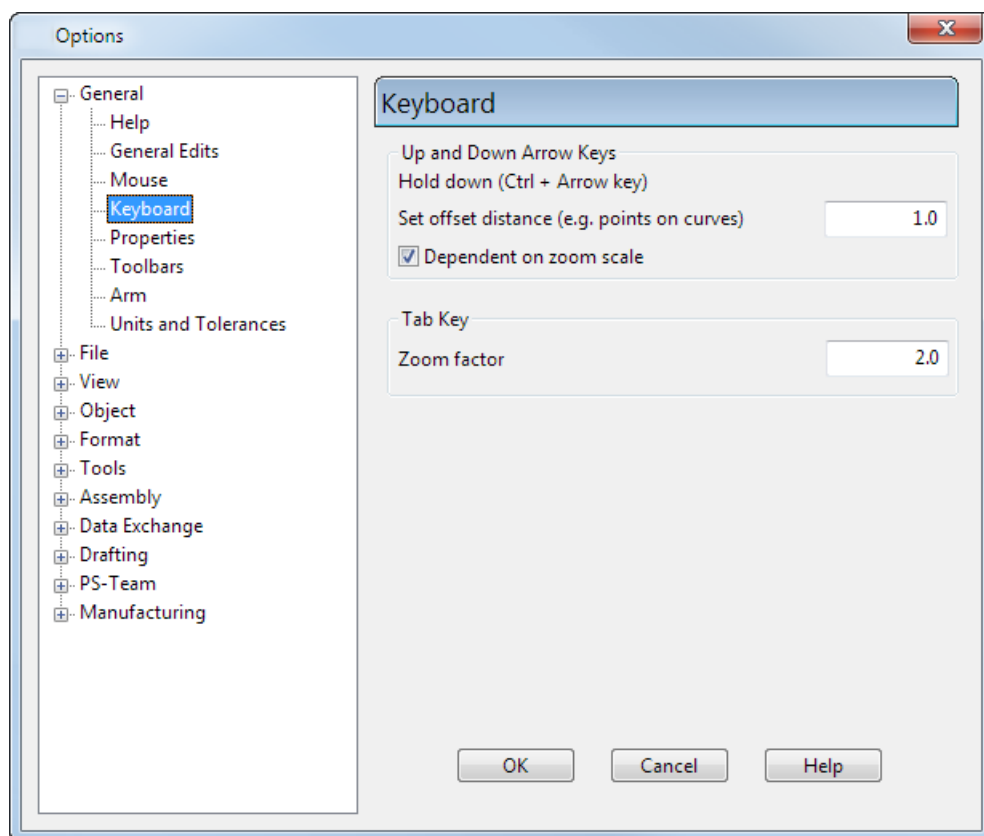
Properties

Specify various settings associated with operation and appearance of PowerSHAPE using the **Properties** dialog.
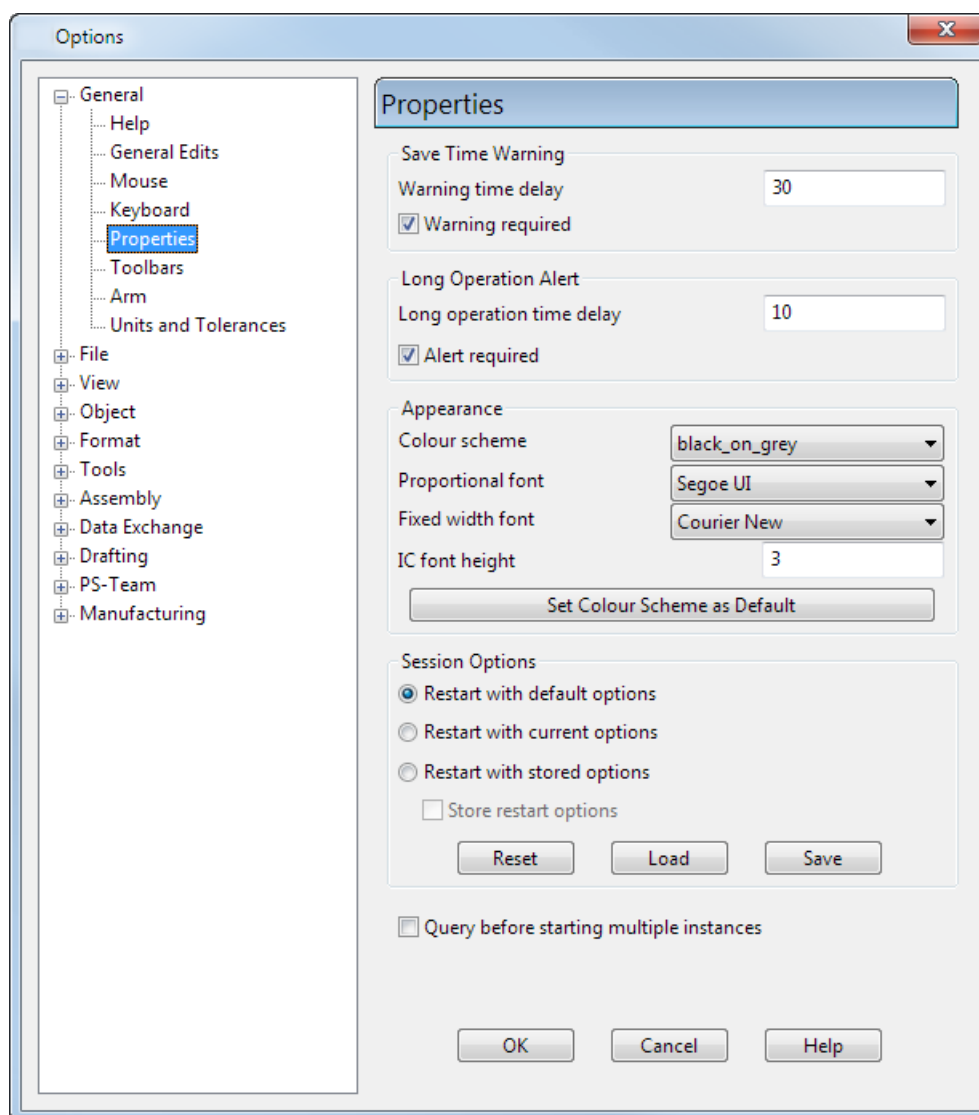
Save Time Warning

At regular intervals, a warning can be displayed to tell you when you last saved a model. You can set the time interval or turn off the warning.

Warning time delay — This defines the time interval in minutes for displaying the warning.

Warning required — If deselected, the warning is not displayed.

Long Operation Alert

PowerSHAPE beeps when long operations finish.

Long operation time delay — This defines the time in seconds that PowerSHAPE must be busy before the operation is classed as long.

Alert required — If deselected, the beep is turned off.

When PowerSHAPE is busy, an hourglass appears at the bottom of the screen.

Appearance

Colour scheme — The selector contains a list of colour schemes provided. Select a colour scheme from the drop down list and press **Set Colour Scheme as Default**. This will save the scheme as your chosen default and this scheme will be used each time PowerSHAPE is started. You can set up your own colour scheme.

Proportional font — This font is used by the intelligent cursor. It is also the default font for creating text and for text in balloons.

Fixed width font — This font is used by scrolled texts in dialogs such as the **Information** dialog.

IC font height — This is the height of the font used by the intelligent cursor.

Set Colour Scheme as Default — When you select this option, the selected **Colour Scheme** is stored as the default and will be used each time PowerSHAPE is started.

Session Options

You can store the settings on the pages of the **Options** dialog and use them in the next session.

Restart with default options — When you restart the program, the default values of the settings are used.

Restart with current options — When you exit the program, it remembers the latest settings on the **Options** dialog. When you restart the program, it will use those settings again.

Restart with stored options — When you exit the program, it will use your stored settings.

Store restart options — This stores the current settings on the **Options** dialog. Store the settings as follows:

1 Set the options on all the pages of the **Options** dialog.

2 Select **Restart with stored options.**

3 Select **Store restart options**. This option is automatically deselected after you click the **OK** button.

4 Click **OK**.

All the settings on the **Options** dialog are stored and will be used in the next session (providing **Restart with stored options** is turned selected).

The settings that are stored will apply to either:

- a model

- a session

as defined in **Show difference between user and model** *(Options > General > Units and Tolerances)*. This option changes the font to indicate if the setting applies to a model or session.

Template models can be used to store the model options.

Reset / Load / Save — These buttons let you:

- **Reset** the options to the state when PowerSHAPE was started.

- **Load** a previously saved options file.

- **Save** the options file.

Query before starting multiple instances — If this option is selected and you try to startup another copy of the program, you will get a message displayed.

Session options are stored for each product variant, so you can have different options for PowerSHAPE, Partmaker Modeling, and Dental.

Toolbars options

Use this dialog to specify the toolbars options.
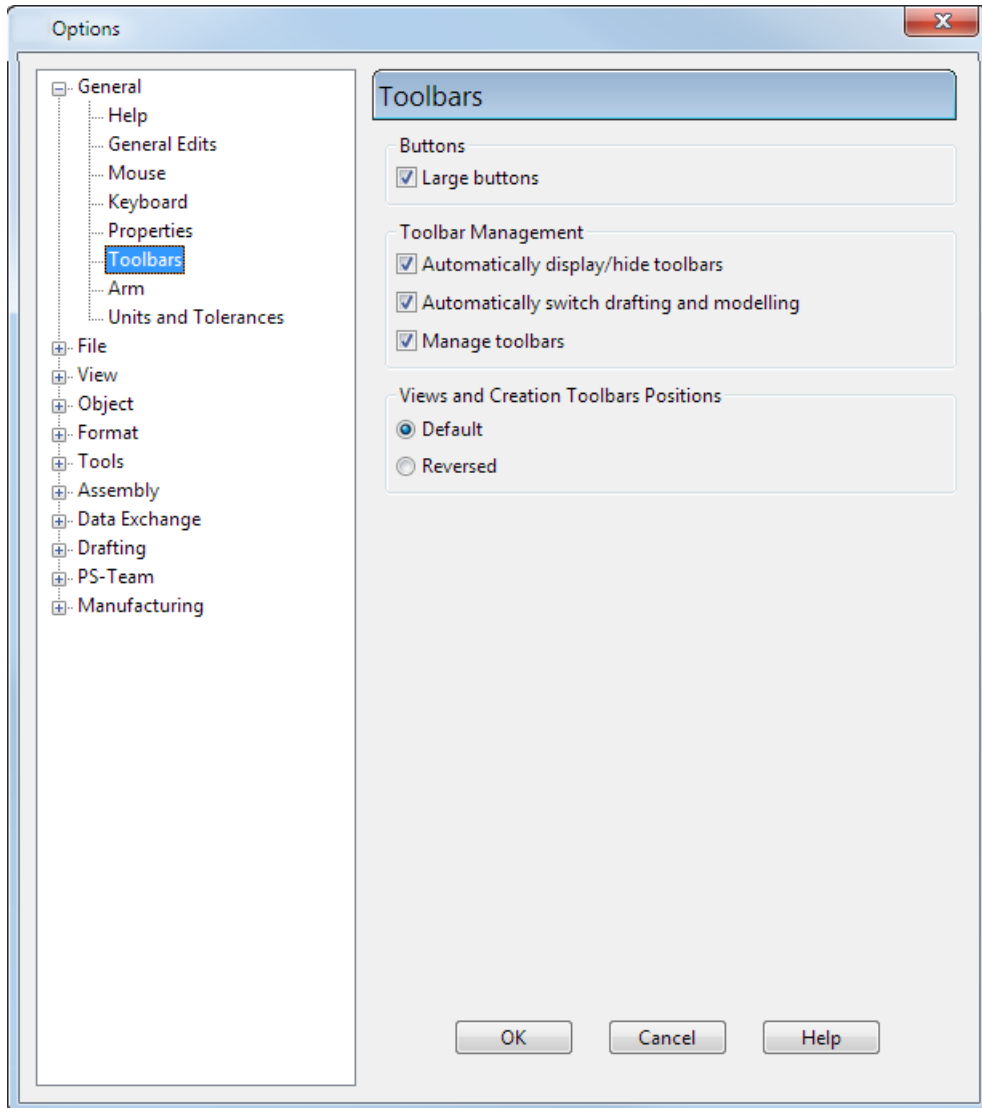
Buttons

Choose the size of buttons to be used on the toolbars.

Large buttons — Select this option to use large buttons on toolbars. Deselect it to use small buttons**.**

Toolbar Management

Automatically display/hide toolbars — If *ON*, the **Dimension**,**Text** and **Surface/Curve** toolbars display automatically when a corresponding object is selected and hide automatically when the object is deselected. If *OFF*, the **Dimension**,**Text** and **Surface/Curve** toolbars display automatically when a corresponding object is selected, but stay displayed once the corresponding object is deselected.

Manage toolbars — You can manage the position of the toolbars by selecting and deselecting this option.

If *ON*, toolbars are automatically positioned when you start the program. The toolbars will always appear in a standard order when docked and cannot be moved. You can float toolbars and dock them on another side of the screen. In practice, this still gives you some control over the toolbar layout but prevents accidental movement of toolbars and makes re-docking of floated toolbars easy (they will always re-dock in their correct position).

If *OFF*, the toolbars can be positioned anywhere.

Views and Creation Toolbars Positions

The options in this section let you select the positioning of your toolbars.

Default — Position the creation toolbar on the left and the views toolbar on the right.

Reversed — Position the views toolbar on the left and the creation toolbar on the right.

Arm options

This dialog lets you set the defaults for using the **Acquire Points** (see page 201) toolbar.
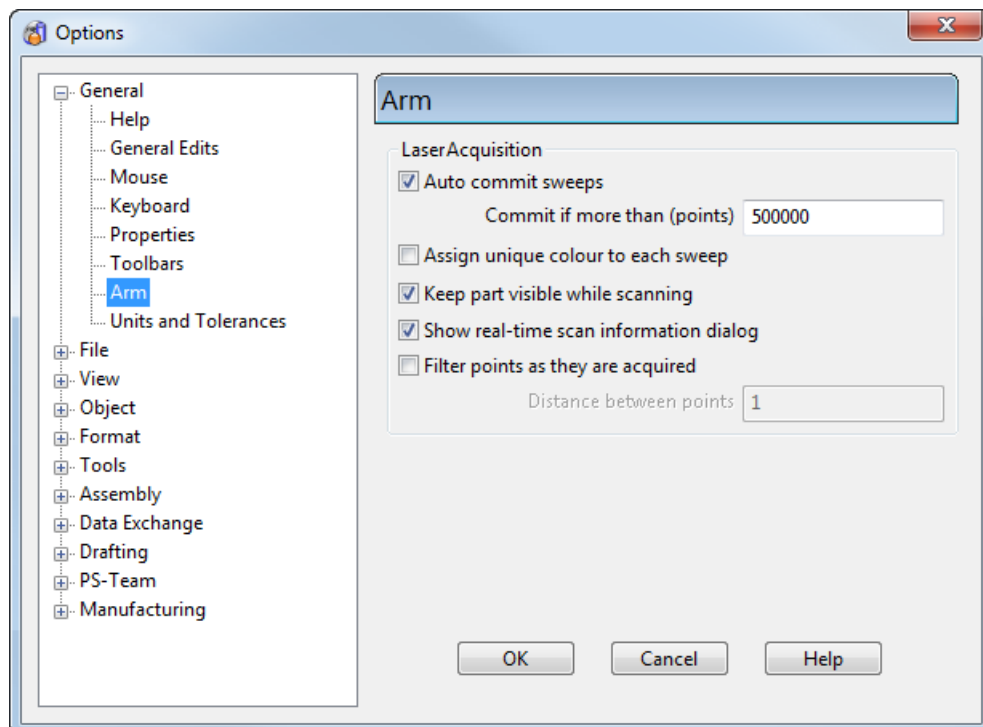
Auto commit sweeps — Sweeps are saved automatically when the number of points taken exceeds the number specified in the box. The clouds can be combined after all points have been acquired.

Assign unique colour to each sweep — Each sweep is assigned a unique colour. This is the same as clicking on the **Acquire Points** toolbar.

Keep part visible while scanning — The acquired points remain visible whilst you carry out further scanning.

Show real-time scan information dialog — The **Sweep Information** dialog is displayed when the **Acquire Points** toolbar is displayed. This is the same as clicking on the **Acquire Points** toolbar.

Filter points as they are acquired — Filter points during scanning. Enter a value for **Distance between points**. This distance is used to remove points that lie within a specified distance of another point.

*The distance is specified in the units that are defined in **Tools > Options> General > Units and Tolerances**.*

Units and tolerances options

Define the length and angle units and the tolerance defaults for the active model using the **Units and Tolerances** dialog.
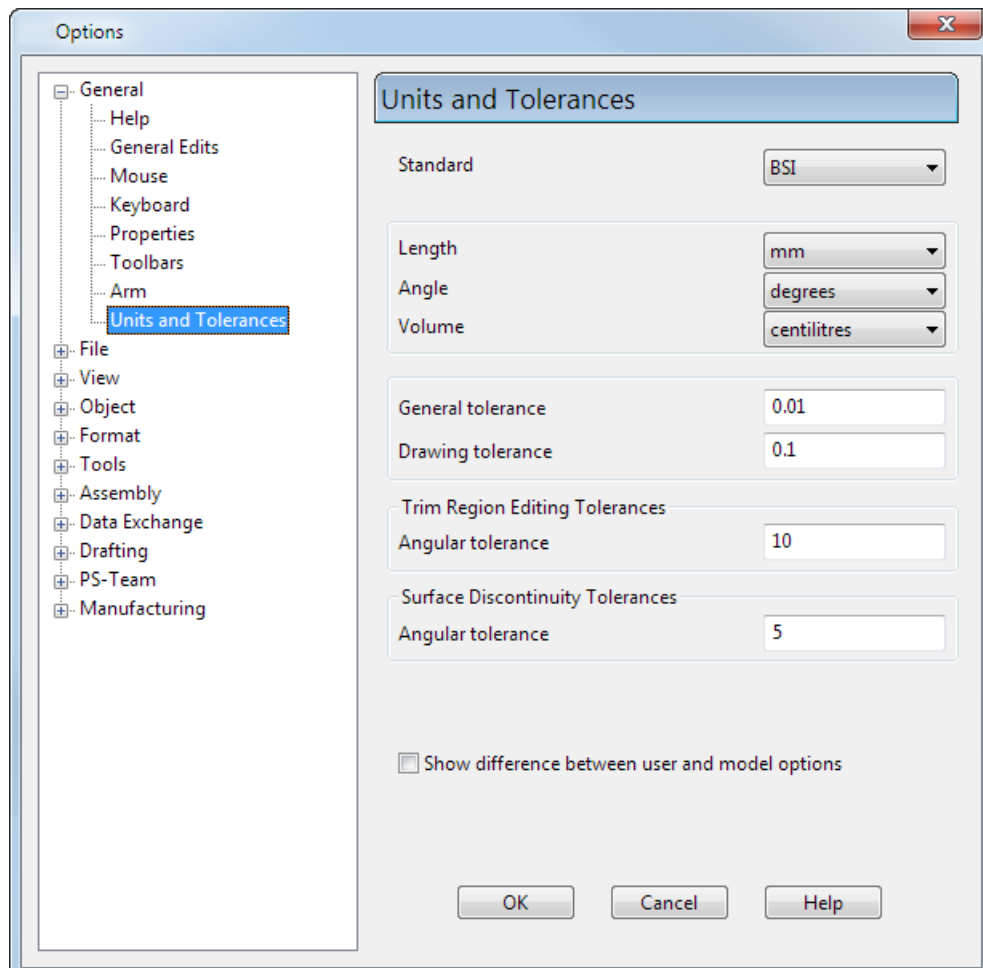
Standard — This list shows the standard used for the active model. You can select one of the following standards:

- ANSI
- BDC
- BSI
- DIN
- ISO
- JITS
- USDD

All models contain standard information. The System Administrator sets up the default for a particular installation. This controls the standard used by all new models.

To change the drawing standards for all models, see Changing the drawing standard

Length — Select the length units for the active model. The options are:

- mm (millimetres)
- inches
- thousandths (0.001 inches).

Angle — Select the angle units for the active model. The options are:

- degrees
- minutes
- seconds
- radians

Volume — Select the volume units for the active model. The options are:

- centilitres
- litres
- mm^3
- cm^3
- m^3
- gallons (UK)
- gallons (US, liquid)
- gallons (US, dry)
- ounces (UK, liquid)

- ounces (US, liquid)
- cubic inch

General tolerance — Select the default tolerance that affects all constructions which do not have a specific tolerance. This tolerance is also displayed in the **Tol** text box on the Status bar.

Drawing tolerance - This allows you to change the tolerance used for drawing objects. When you draw an object, this tolerance allows you to change how smoothly it is drawn.
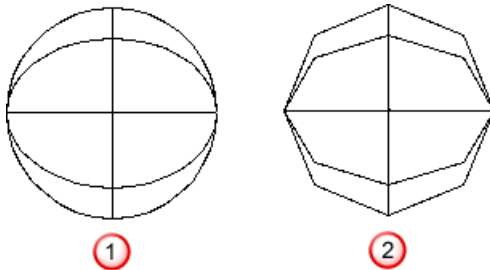
① Drawing tolerance = 0.1

② Drawing tolerance = 10

Trim Region Editing Tolerances

Angular tolerance - This tolerance is used when surfaces are trimmed.

Surface Discontinuity Tolerances

Angular tolerance - This tolerance is used to decide if a surface contains any discontinuities in the normal direction. This tolerance is compared with the angle between two normals at a point on a surface ③.
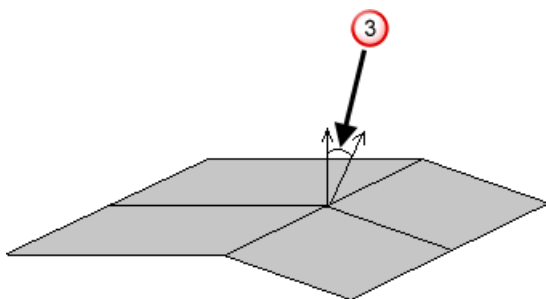
If the angle is greater than the angular tolerance, then the surface is discontinuous at that point.

Show difference between user and model options - If selected:
- Options that apply to all models are shown in italics.
- Options that apply to the current model only are shown in the standard font.

If deselected, all options are displayed in the standard font to make them easier to read.

File options

The following options pages are grouped under the **File** category:

- **Model** (see page 91)
- **Print** (see page 94)
- **Plotting** (see page 95)
- **Filters** (see page 96)

Model options

Use this dialog to specify model options.
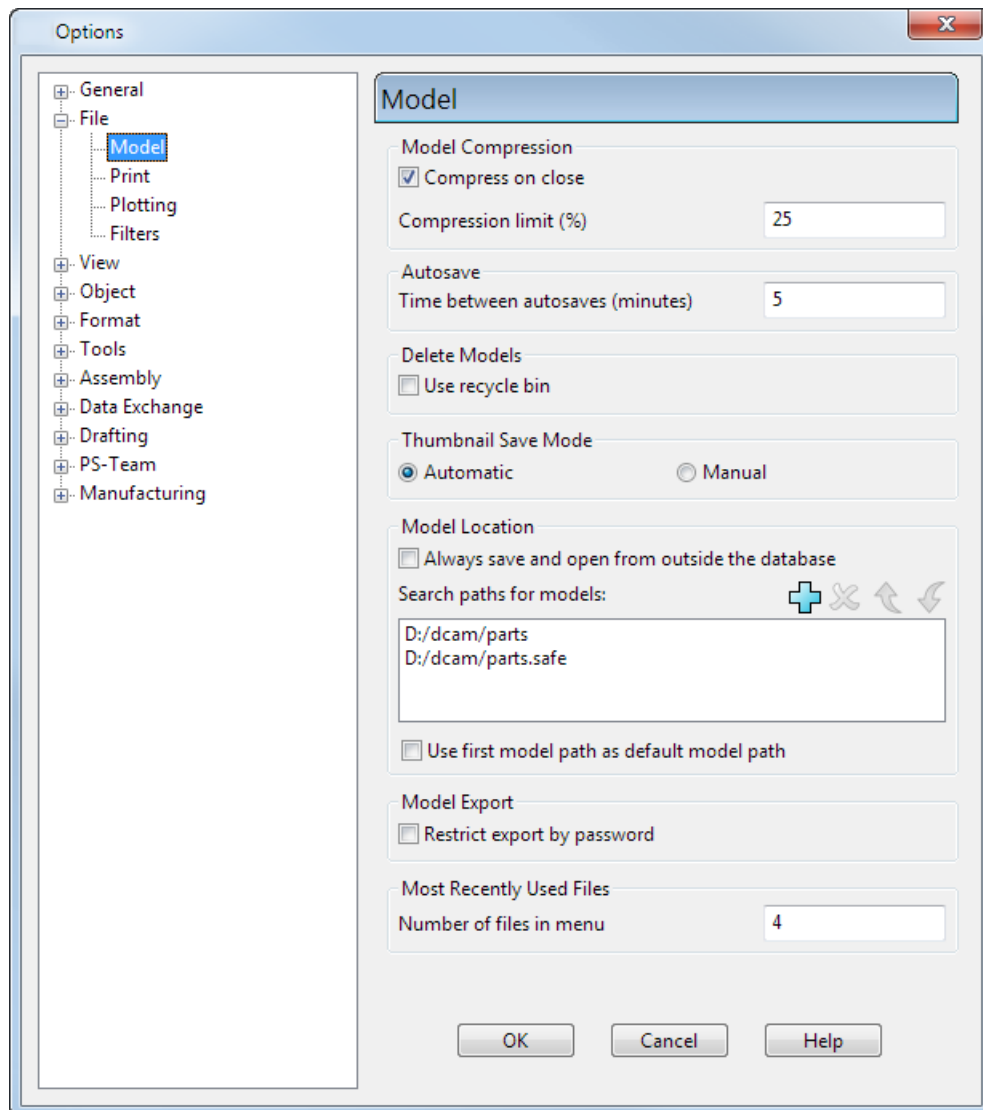
Model Compression

Compress on close — If selected, a model is compressed when the model is closed after saving.

Compression limit (%) — The model will reduce in size by at least the percentage entered. For example if **25%** is entered, models will only be compressed if they reduce in size by 25% or more.

Autosave

Set the **Time between autosaves (minutes)** to regularly save Parasolid (post-version 8) models. This allows model recovery up to the point of the last autosave.

The default setting is 5 minutes.

The autosave will only occur while you are creating or editing geometry. It will not occur if PowerSHAPE is idle for 5 minutes.

Delete Models

Use recycle bin — If selected, deleted models are sent to the recycle bin (providing the bin is not full). Otherwise, the models are permanently removed from the system.

Thumbnail Save Mode

This allows you to choose when the thumbnail is saved.

Automatic — A thumbnail is automatically saved when the model is saved.

Manual — This allows you to decide when you want to save the thumbnail.

Model Location

Always save and open from outside the database — If selected, the **Save** and **Open** commands will save and open models outside the model database and use the standard Windows dialogs.

Search paths for models - This window displays the list of paths that are searched for models. You can change the search list by using the associated buttons.

Use first model path as default model path — Select this option to set the default model location to the first entry in the list of model paths.

- If the option is selected, open and save operations use the first model path as the default location. If the model path list is empty, the default location is set to the current directory.

- If this option is deselected, the default path is specified in the **Start in** option on the PowerSHAPE shortcut

 — Add a new path to Search paths for component libraries. Clicking this displays the **Browse For Folder** dialog. Select the path to add to the list of search paths.

 — Removes selected path from **Search Paths for Models.**

 — Move selected path up in search order.

 — Move selected path down in search order.

You can replace a selected path by double-clicking on a path in the window and selecting a replacement path from the **Browse for Folder** dialog

Model Export

Restrict export by password — If selected, a password is required to export the model.

Most Recently Used Files

Number of files in menu — Enter the quantity of recently opened models to be listed at the bottom of the **File** menu in **Recent Files**. The default is 4 and the maximum number of files in the list is 20.

Saving a model with a thumbnail

Recent Files (see page 21)

Print options

Use this option to select the printing options.
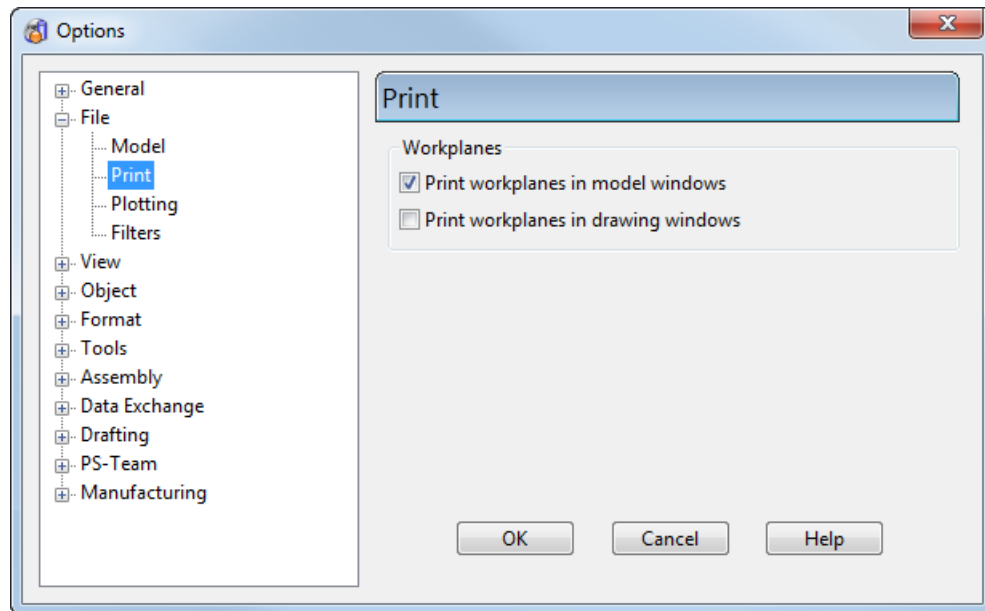
1 Click **Tools > Options > File > Print**.

2 Select the printing of workplanes in the model window.

3 Select the printing of drawing window.

Plotting options

This option lets you set the plotting options for the **File > Print** command.
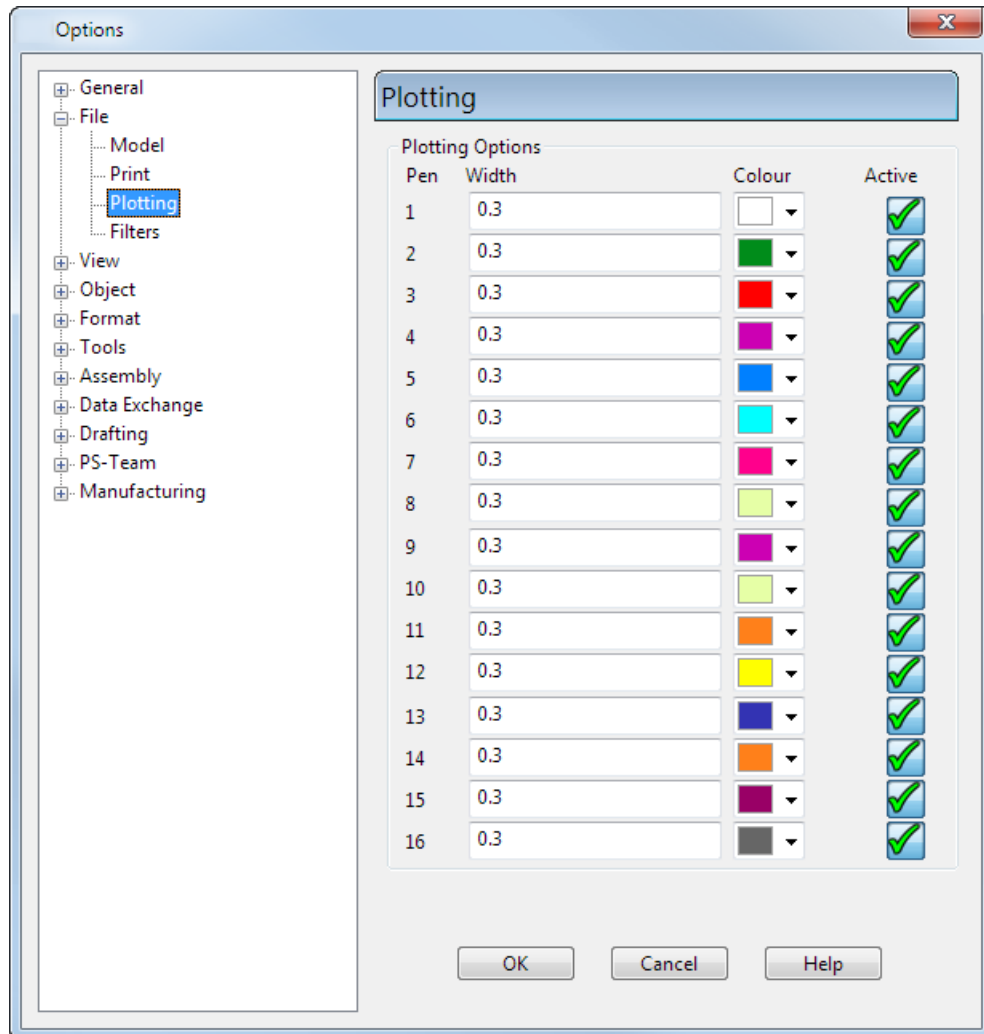
Plotting Options — Each printer is assigned a set of plotter pens, which are only considered when plotting to the printer. Plotting options allow you to set up the pens to your requirements. Each pen has a **Width**, a **Colour** and a status flag. The status flag indicates whether a pen is **Active** or not. At least one pen must be active at all times.

Filters options

This allows you to set the filters used in such commands as **File > Open.**
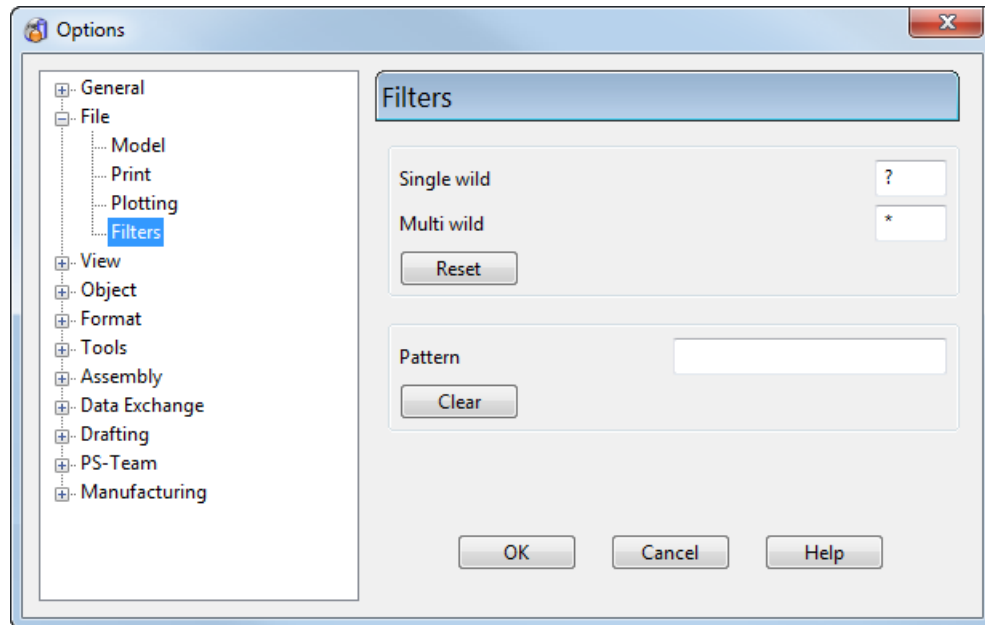
Wildcard Characters

This sets the defaults used by the filter controls in commands such as **File > Open**. Filters allow you to reduce lists so they show only the files similar to the one(s) you are looking for.

The default wild card characters are the same as used by UNIX operating systems, but if you want to change them, just click in the text box and type a new character.

Reset — This restores the PowerSHAPE defaults as shown.

Pattern — This allows you to store a character string as a preset filter. If you regularly use the same filter string, this option saves the need to retype it each time.

Clear — Empty the **Pattern** text box.

View options

The following options pages are grouped under the **View** category:

- **Shading** (see page 97)
- **Views** (see page 99)
- **Blanking and Grid** (see page 102)

Shading options

Control the settings for shading surfaces using the **Shading** page.
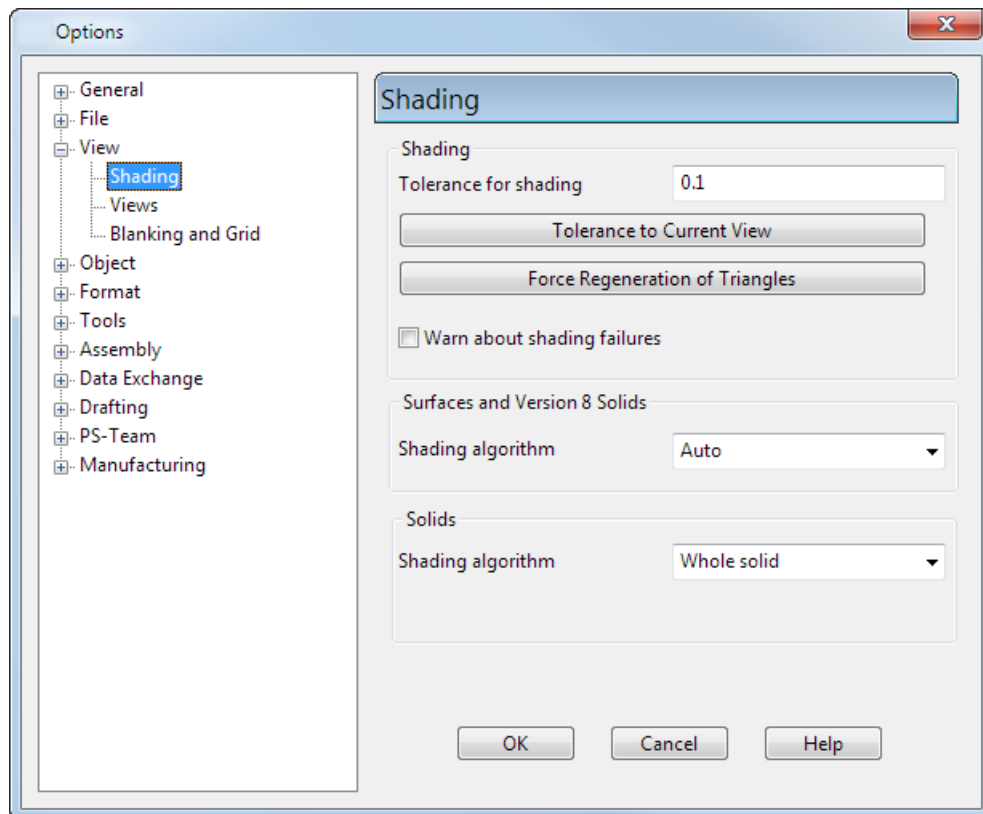
Tolerance for shading — This controls how accurately the triangles generated when shading objects fit the surfaces. The default is 0.1. The smaller the value the better the shading quality, however the time taken to shade the objects increases. When the value is large the objects are shaded quickly however gaps may appear between the triangles.

Tolerance to Current View — This sets the ideal value of Tolerance for Shading according to your current view and zoom level. Click the button and the value is displayed in the Tolerance for Shading text box. If you want a different tolerance, click in the box and enter a suitable value. The new tolerance will be used the next time the shading is regenerated. If you want to use it immediately, click **Force Regeneration of Triangles**.

Force Regeneration of Triangles — This forces the shading triangles to be regenerated, using any settings that have been changed.

Warn about shading failures — If selected (default), a warning is displayed if there is a problem when shading solids, surfaces and other items.

Surfaces and Version 8 Solids

Shading Aagorithm — Select the algorithm used to generate the triangles for shading. The following options are available:

Auto — This automatically choose between the **Rockwood** and **Subdivision** options, depending on the surface and trimming data in the model.

Rockwood — This is like the **Rockwood Patch** option but is faster and produces fewer triangles.

Subdivision — This produces fewer triangles on complex multi-patch surfaces than all the other options.

Rockwood Patch (older triangulation algorithm) — This is faster on complex curved objects than the **Duct Patch** option.

Duct Patch (older triangulation algorithm) — This is the quickest way of shading simple objects.

Solids

Shading algorithm — Select the algorithm to be used for solids and then select **Force Regeneration of Triangles**. The following options are available:

Whole solid — Use this option to close gaps in the shading of a Parasolid model.

Individual faces — Use this option to close gaps in the shading of faces on a solid.

Views options

Use the **Views** dialog to control the default view details.
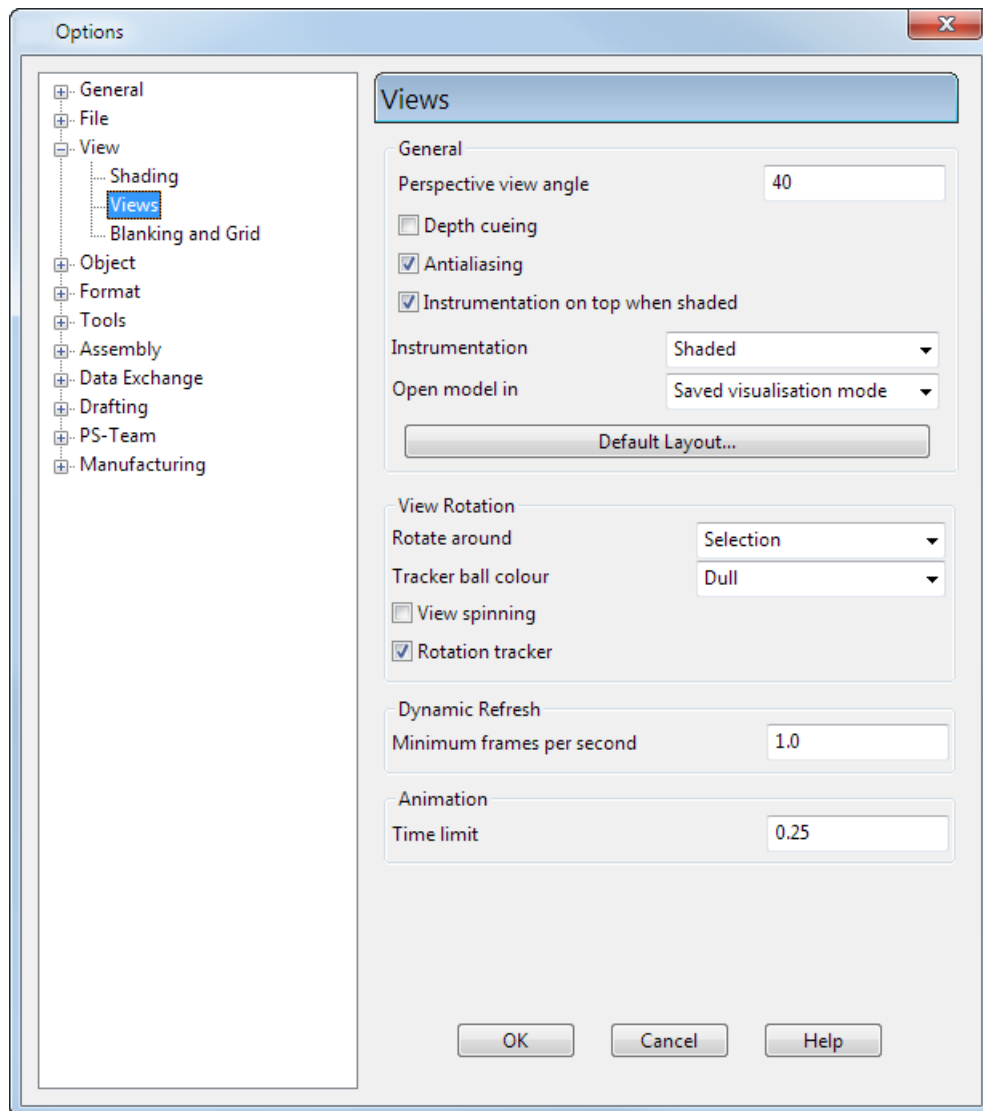
Perspective view angle — This affects how perspective views are displayed. Enter a value of 0° to put the vanishing point at an infinite distance, creating an orthographic view. Wider angles create a feeling of depth and distance, with really wide angles distorting the image to the point where there is no visible depth. The default angle is 40°.

Depth cueing — Select this option to draw objects and parts of objects progressively less brightly as they fall further from you. This gives a feeling of depth as if receding into the mists of distance.

Anti-aliasing — This option is now on by default, providing smoother outlines to rotated models.

Anti-aliasing provides greatly improved graphics display, but it can impact on performance. Consequently, this functionality is disabled automatically if the graphics card has less than 1Gb of video memory.

If you wish to disable the anti-aliasing functionality, use one the of the following methods:

- Make the following change to powershape.con and restart PowerSHAPE.

  enable_aa: false

- Use the control panel of the graphics card to turn off the anti-aliasing mode to of the graphics card.

- Deselect **Tools > Preferences > View > Views > Anti-aliasing**

  Changing this option affects subsequently created windows; it has no effect on currently open windows.

Instrumentation on top when shaded — When you select an object (for example a solid block) in shaded mode, its edit handles are displayed on top of the shading.

Instrumentation — Select the style of the instrumentation from:

- **Wireframe**
- **Shaded**
- **Old style**

Open model in — Select the default view to be used when opening a model, from:

- **Saved visualisation mode**
- **Wireframe**
- **Shaded wire**
- **Shaded**
- **Dynamic hidden line**
- **Transparent shaded wire**
- **Transparent shaded**

Default Layout — This defines the default view layout for new windows. Click the **Default Layout** button to display the **View Edit** dialog (see page 113). Use this dialog to define the default view layout.

View Rotation — When you drag the middle mouse button the view rotates in 3D as if within a ball. The two menus in this option control how the rotation behaves and the tracker ball graphic colour.

Rotate around — Select one of the following options from the drop-down list:

- **Selection** — The view rotates around the selected point on the selection. If nothing is selected, it reverts to the centre of the view.

- **Active workplane** — The view rotates around the currently active workplane origin. If no workplane is active, it reverts to the centre of the view.

- **View centre** — The view rotates around the centre of the view.

Tracker ball colour — Select one of the following options from the drop-down:

- **Dull** — The tracker ball lines are drawn in a colour which **does not** contrast heavily with the active window background colour.

- **Bright** — The tracker ball lines are drawn in a colour which **does** contrast heavily with the active window background colour.

- **Invisible** — The tracker ball lines are hidden during view rotations.

View spinning — When you drag the middle mouse button on a view, the view rotates. You can turn on **View Spinning** so that the view continues to rotate after you release the middle mouse button. The view is rotated in the direction that you moved the mouse.

Rotation tracker — If selected, axes are drawn in the bottom left corner of the view to show the direction of the X, Y, and Z axes.

If an axis points away from you, it is drawn dim.

The axes on the rotation tracker, which represent the active plane, are drawn in a different colour to the axis perpendicular to the plane. The colours are set in the scm file.

Minimum frames per second — Enter the minimum acceptable number of frames per second when dynamically rotating, zooming, and panning. If the view cannot achieve the requested minimum frames per second, objects are omitted while dynamically changing the view. Enter **0** for no minimum.

Time limit — Enter a maximum target time, in seconds, to animate between view orientations.

See Also

Colours used in PowerSHAPE

Blanking and Grid

Use this dialog to define the grid and blanking options.
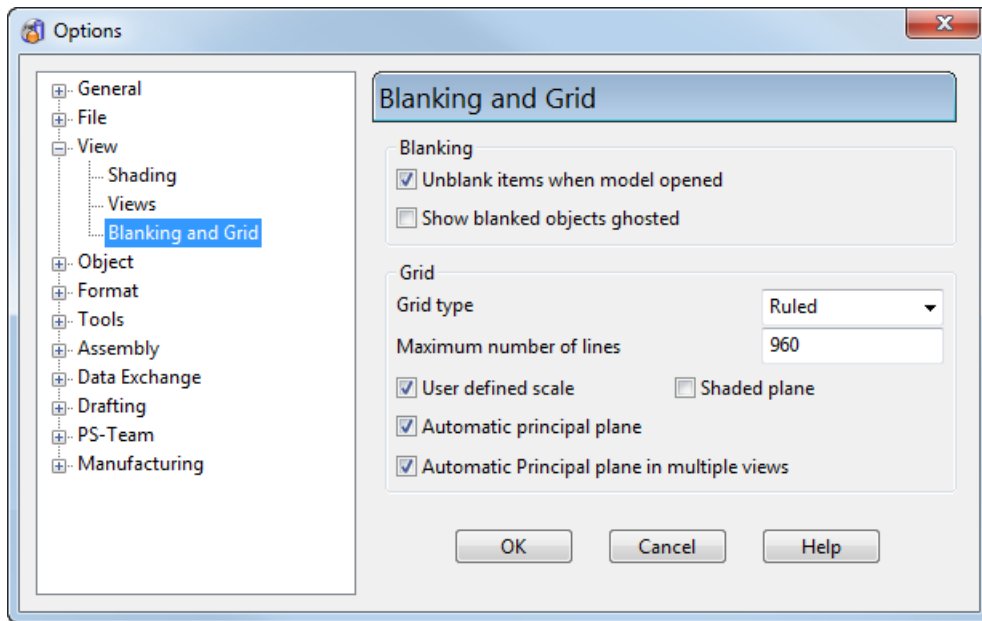
Blanking

Unblank items when model opened — Select this option to display blanked items when you open a model.

Show blanked objects ghosted — Select this option to display blanked items as grey instead of being fully hidden.

Grid

Grid type — Select whether to draw the grid as ruled or dotted lines.

Maximum number of lines — This value represents the maximum number of horizontal and vertical grid lines. The default number of grid lines in each direction is set to half the screen resolution. If you request a grid of more than 1000 lines in each direction, a warning is displayed to indicate the large number of lines may cause the grid to draw slowly.

User defined scale — If selected, the grid scale is locked to the current user defined value. If deselected, the grid scale adjusts as the view is changed.

Shaded plane — If selected, a shaded plane representing the current active workplane is displayed if the grid is switched off.

Automatic principal plane — If selected, the principal plane is set based on the viewing direction. The principal plane will be set to be at right angles to the axis you are looking down. So, if you are looking down the Z axis, the principal plane will be changed to XY.

Automatic Principal plane in multiple views — Select this option to have a principal plane related to the viewing axis as follows:

- Z - XY plane

- X - YZ plane

- Y - ZX plane

An ISO view has an XY principal plane.

Object options

The following options pages are grouped under the **Object** category:

- **Arcs** (see page 104)

- **Curves** (see page 105)

- **Composite curves** (see page 106)

- **Holes** (see page 108)

- **Lines** (see page 110)

- **Points** (see page 112)

- **Solids** (see page 112)

- **Surfaces** (see page 115)

- **Mesh** (see page 120)

- **Workplanes** (see page 121)

Arcs options

This page enables you to set the default arc creation tool, which is displayed in the arcs menu on the main toolbar.
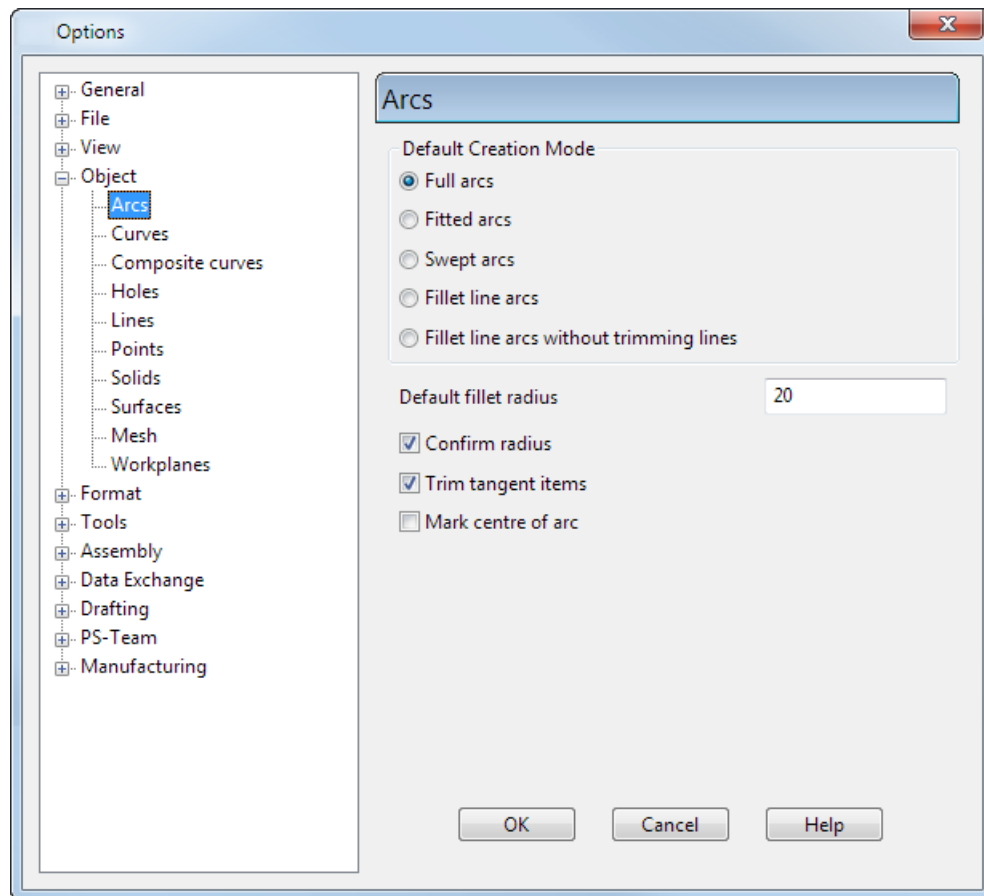
Default Creation mode — Set the default arc creation mode so that when you create an arc using the toolbar, this creation mode is automatically selected.

Default Fllet radius — When you create a fillet arc, it is created with a default radius. Enter the default fillet radius. The **Default Fillet radius** is stored with each model. The option is available only if a model is open.

Confirm radius — Select this option to display a dialog, which asks you to confirm or correct the radius value, when you create a new fitted arc.

Trim tangent items — Select this option to trim the object to the tangency point when you create an arc tangent to an object.

Mark centre of arc — Select this option to mark the centres of new arcs with a cross.

Curve options

This page enables you to set the curvature options for creating B-spline and g2 curves and to create composite curves automatically from a trimmed fillet solution.
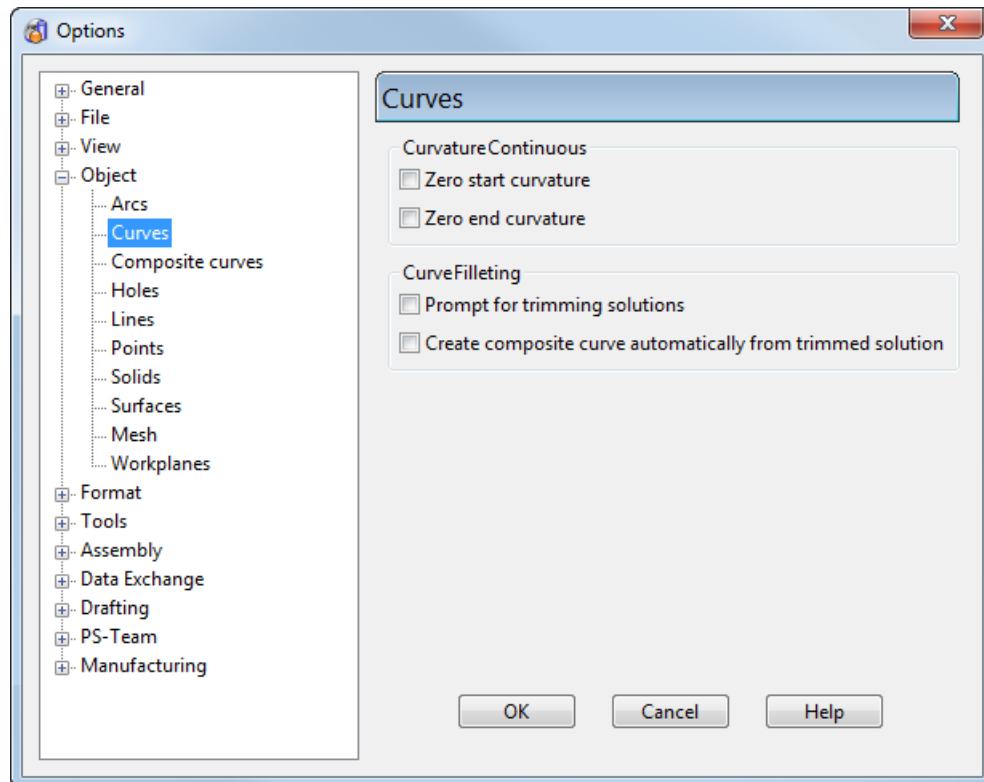
Curvature Continuous

These options set the curvature to zero at the start or end point when creating B-spline and g2 curves.

A zero curvature has a flattening effect on the curve.

① Non-zero curvature.

② Zero curvature.

Curve Filleting

The following two options are used with the **Create a Trimmed Bezier Curve Fillet** and **Create a Radii Bezier Curve Fillet** options on the **Curves** menu, where two adjoining items are trimmed and a fillet curve is created in between.

Prompt for trimming solutions — If this option is *ON* and the **Create profile automatically from trimmed solution** option is also *ON*, a composite curve is created for only 4 out of the 8 potential trim solutions that the **Create a Trimmed Bezier Curve Fillet** option offers. This is because a valid composite curve cannot be created automatically where there is a jump in the track in a trim solution.

Create composite curve automatically from trimmed solution — Select this option to automatically create a composite curve from the trimmed fillet solution.

Composite curve options

This controls the default settings for adding composite curves to models. This also contains the setting to zoom in when entering 2D Editing.

For further details, see:

- Creating a composite curve by tracing
- Setting composite curves options
- Using the 2D curve editor
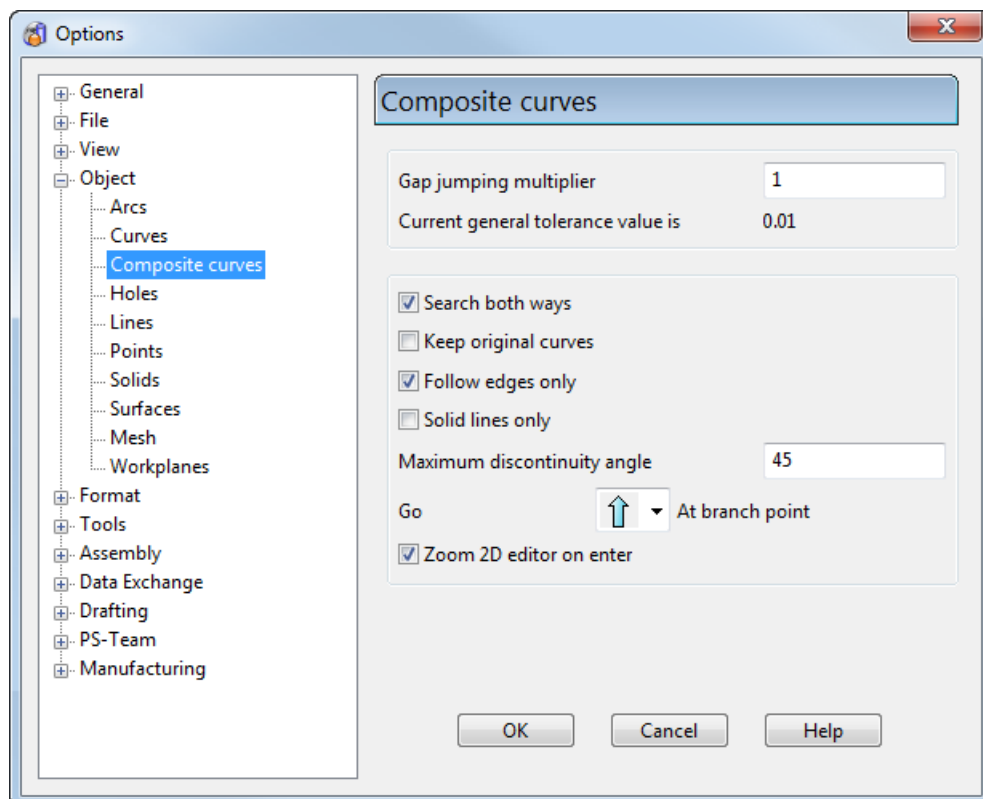
Gap jumping multiplier — Define the gap used to search for suitable paths during composite curve creation. The number is a multiple of the general tolerance and can be changed during composite curve creation. For your convenience, the current value of the general tolerance is displayed below this option.

Search both ways — Select this option to enable the control panel to select items from the start of the composite curve when it reaches the last item.

Keep original curves — This option is deselected by default and whole objects selected as part of a composite curve become part of the new composite curve (and can no longer be selected as single items). Items that are partially selected (for a portion of their length) cannot be partly consumed and so a copy of the selected portion is created and used in the new curve, and the original is left untouched. Select **Keep Original Curves** to copy all objects (whole or partial) to the new curve. The originals remain untouched.

Follow edges only — When selecting surface curves to be part of a composite curve, there are two options. You can select either curves only at the edge of the surface or all the curves (at surface edges or patch edges). The latter involves much more work and slows down the searching operation. Because it is also less commonly required, the default is normally to search following surface edge curves only.

> If the option is *ON*, it behaves as *OFF* when:
>
> - The surface is closed in either direction. If any trimming makes the closed surface appear open, the surface is treated as open.
> - The surface is a surface extrusion.

Solid lines only — This is used in hatching. For further details, see Hatching.

Maximum discontinuity angle — This angle affects the way the strategy of **Discontinuous** selects the path to follow at branch points when in **Fast forward** mode.

Go ⬆️▾ **At branch point** — Select how the control panel, when in **Fast forward** mode, automatically selects items when a branch point is reached:

Straight on — Select the path which is the most continuous (has the smallest change in direction) from the last object selected.

Left — Select the left most path. This is useful when choosing closed loops.

Right — Select the right most path.

Discontinuous — Select the path which has an angle that is less than the **Maximum discontinuity**. If there is more than one path which is less than the discontinuity angle, then the path which has the smallest discontinuity angle is chosen.

Zoom 2D editor on enter — Define how a composite curve is viewed when using the 2D Editor. The option is selected by default.

If this option is selected, when you select the **Edit the curve in 2D** button (*Curve Edits flyout*), the display zooms into the selected composite curve, ready for editing using the 2D Editor.

If this option is deselected, when you select the **Edit the curve in 2D** button (*Curve Edits flyout*), the selected composite curve is displayed for editing using the 2D Editor, but the display is not zoomed-in.

Holes options

Set the default hole options on the **Holes** page.

Hole Category and Use — Use the lists to specify the default settings to be used in the **Hole** dialog.

1 Select a hole category from the first list.

2 Select the default use for the selected hole category.

Delete category — (A*vailable only if a user-defined hole category is selected*). Click this button to delete the selected user-defined category. If the category is a standard one, all the user-defined hole uses within the selected category are deleted.

Create — Click this button to open the **Hole Use** dialog to define a new category and use.

Edit — (A*vailable only if a user-defined hole use is selected*) Click this button to use the Hole Use dialog to edit the hole use.

Delete — (A*vailable only if a user-defined hole use is selected*) Click this button to delete the selected hole use.

Delete category — Click this button to delete the selected user-defined category. If the category is a standard one, all the user-defined hole uses within the selected category will be deleted.

Mark centre of holes — If selected, the centres of all hole features are marked. When the centres are marked, you can snap to them and generate construction lines from them.

Reuse last hole definition — If selected, the definition of the last hole feature created is always used to create the next hole feature, even if it is on another solid. If deselected, the definition is only remembered as long as you are in hole creation mode.

Show hole threads — If selected, the thread detail for hole features is displayed.

Give holes meaningful names — If selected, each hole will be given a meaningful name based on its use or tapping standard. If deselected, the hole will be named according to the integer name system.

The following alignment options let you set initial options when placing a hole. You can reset the hole alignments for individual cases using the alignment options on the hole Plane Details dialog.

Align with any surface — If selected, new holes are aligned to any face normal. If deselected, new holes are only aligned to planar face normals.

Align with active workplane — If selected, the hole is aligned to the normal of the active workplane. If deselected, the hole is aligned to the face normals based on the **Align with any surface** option.

Create chamfers on holes — Select this option to create chamfers on the holes. If deselected, chamfers are not included.

>   **Chamfer size** — Enter the default size of all the chamfers on a hole.

>   **Chamfer angle** — Enter the default angle of all the chamfers on a hole.

Recognise tapped holes — This option is selected by default. Deselect this option to prevent holes from being recognised as tapped.

Line options

Use this page to control the default settings used to create lines.
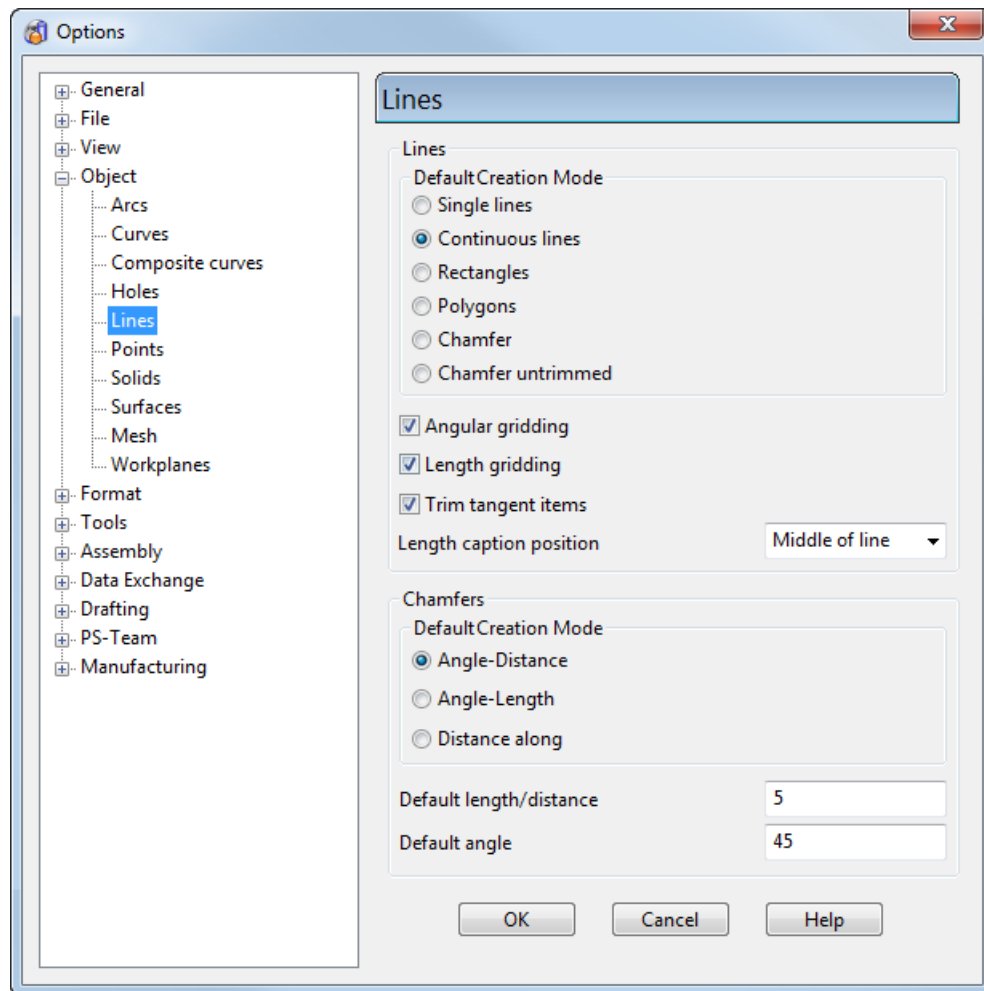
The page is split into two main sections:

- **Lines**
- **Chamfers**

Lines

Default Creation Mode — Select the default mode for creating lines from:

- **Single lines**
- **Continuous lines**
- **Rectangles**
- **Polygons**
- **Chamfer**
- **Chamfer untrimmed**

When you create a line using the toolbar, this creation mode is automatically selected.

Angular gridding — Select this option to use angular gridding when creating a continuous or non-continuous line. Angular gridding displays incremented angle values between the horizontal and the rubber band line displayed to the cursor, after the first point is input.

Length gridding — Select this option to use gridded values for the length when creating continuous lines, non-continuous lines, or rectangles.

Trim tangent items — Select this option to trim an object to the tangency point when you create a line tangent to an object.

Length caption position — Select where you want the line length measurement to be displayed. Length measurements are displayed when you create a line or select an existing line.

> **Off** — Select this option to disable the length measurement display.
>
> **End of line** — Select this option to display the length measurement at the end of the line.
>
> **Middle of line** — Select this option to display the length measurement in the middle of the line.

Chamfers

Default Creation Mode — Select the default mode for creating chamfers from:

- **Angle-Distance**
- **Angle-Length**
- **Distance along**

When you create a chamfer, this type of chamfer is automatically created.

Default length/distance — This is the length or distance used by the definition of the chamfer.

Default angle — This is the default angle of the chamfer with the first line when the **Default Creation Mode** is **Angle-Length** or **Angle-Distance**.

Points options

This sets the options for points.
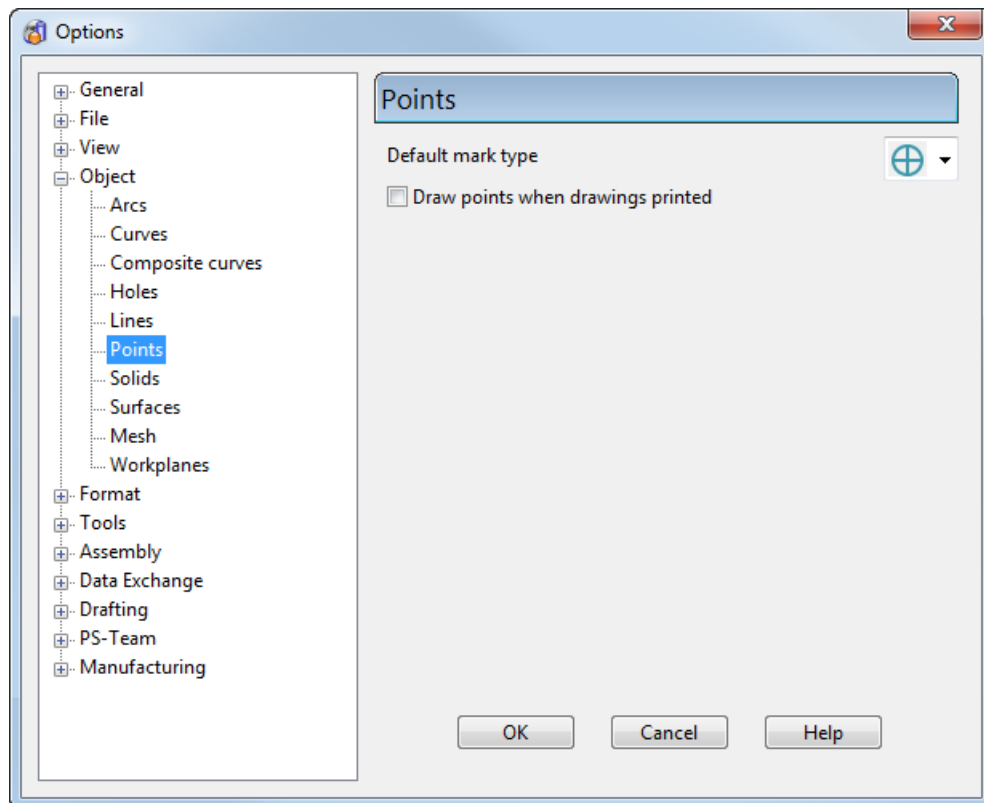
Default mark type — Select a symbol to represent the mark from the drop down list. The possible options are:

Draw points when drawings printed — Select this option to draw points when you print a drawing.

Solids options

This controls the default settings for solids within models.
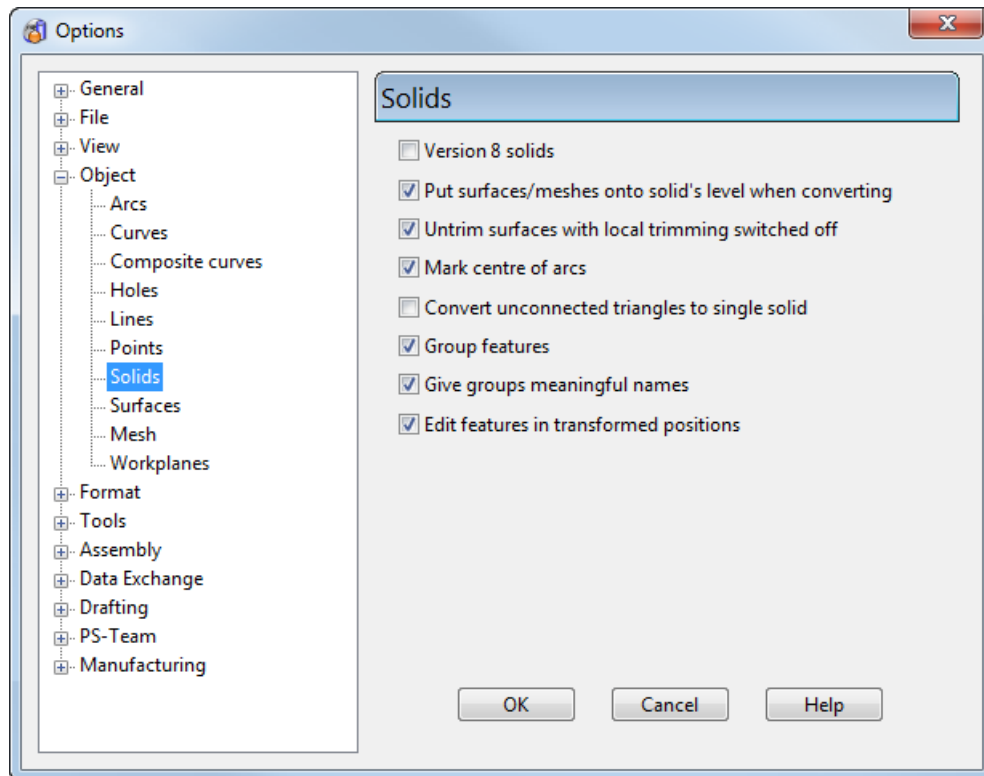
Version 8 solids — Select this option to create new solids as Version 8 solids.

Put surfaces/meshes onto solid's level when converting — Select this option to put surfaces and meshes on the same level as the solid. If deselected, the surfaces and meshes remain on their original levels.

Untrim surfaces with local trimming switched off — This option controls the behaviour of trimmed surfaces that have their local trimming flag switched off when they are made into a solid.

   If selected, the surfaces have their boundaries removed automatically when they are made into a solid. The surfaces in the solid therefore appear untrimmed.

   If deselected, these surfaces appear trimmed when made into a solid.

Mark centre of arcs — Select this option to mark the centre points of wireframe arcs used in solids and solid features. If selected, the centre point of any arc is marked by a dot. Any object dragged over this dot snaps to the centre of the arc. The default setting for this option is selected.

This option also marks the centre of the arcs used in composite curves when creating solid primitive extrusions, revolutions, cut features, boss features, bulge features, and network of curves features.

Convert unconnected triangles to single solid — Select this option to convert unconnected triangles to a single solid.

Group features — Select this option to create a group of features when you:

- sequentially create five or more features of the same type.

- select individual features in the tree and select **Group All** from the **Solid** tree menu.
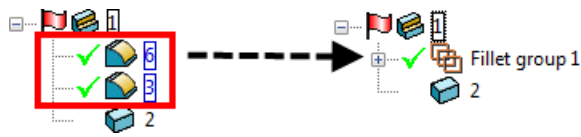
- select individual features on the model and select **Group All** from the **Solid** tree menu.
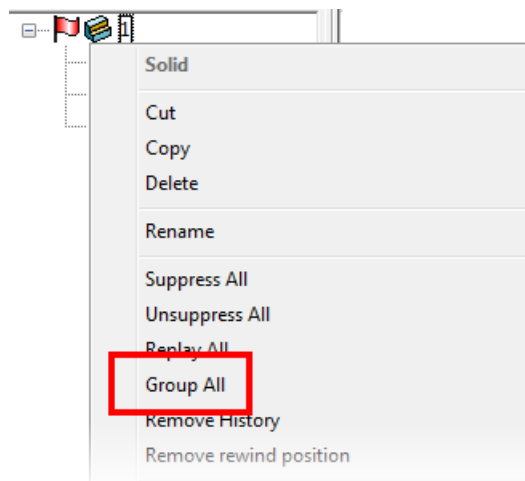
Give groups meaningful names — Select this option to give groups names that relate to their contents. This option is selected by default.

Edit features in transformed positions — Select this option to edit a feature in its transformed position. So, if you have moved a Boolean block, the block is edited in the current position.
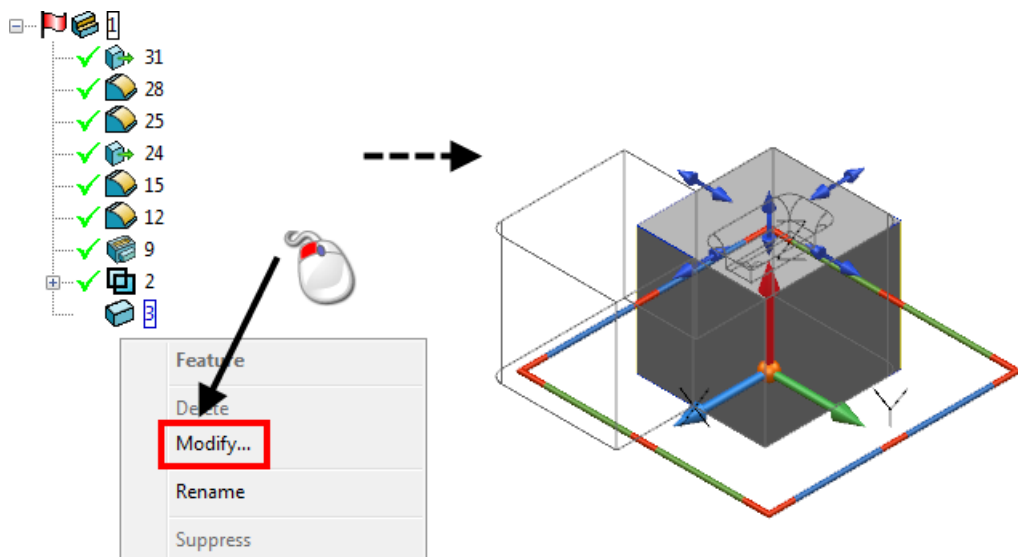
If **Edit features in transformed positions** is deselected, the feature is edited in its position before the transform was applied. So, if you move Boolean block, the block is edited in its original position.
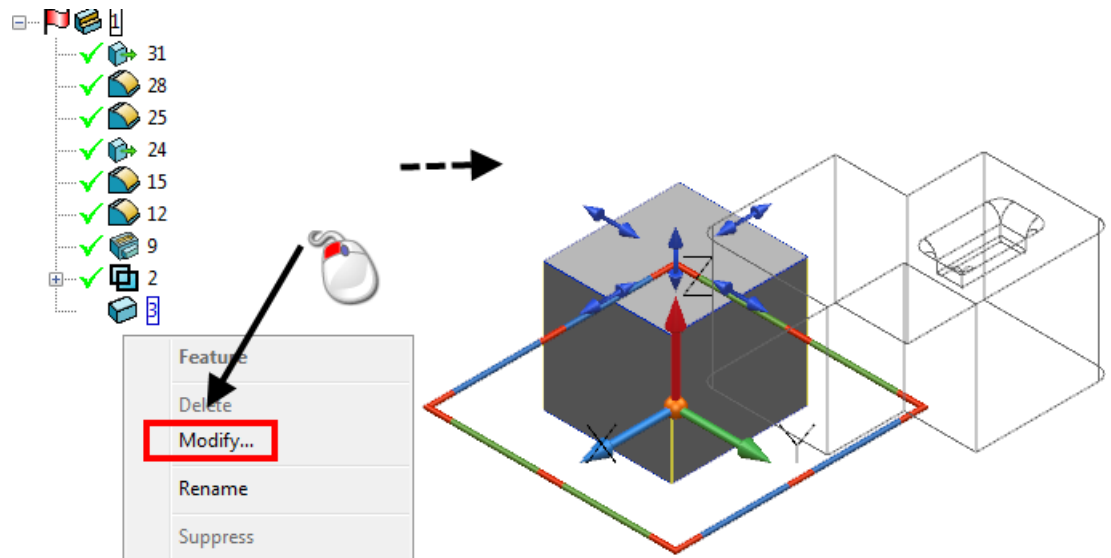
In PowerSHAPE 2015 R2, this option works for the following features in the tree:

- base and primitive features
- Boolean features
- Boolean boss features

Surfaces options

Use this page to control the default settings for surfaces within models.
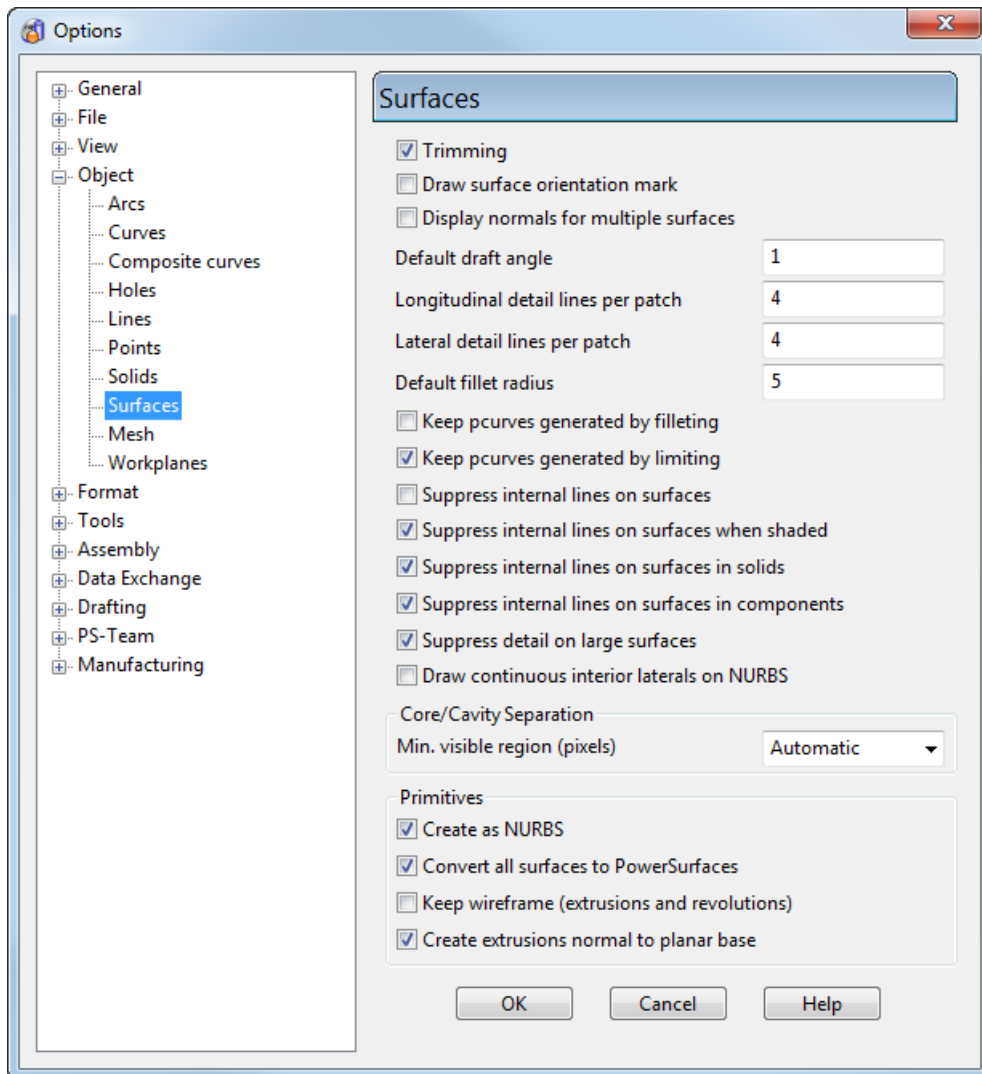
Trimming — Select this option to trim surfaces. Trimmed surfaces give a realistic shape. Untrimmed surfaces require less computer graphics processing, reducing screen redraw times. However, they can make the model confusing to look at.

Draw surface orientation mark — Select this option to mark a surface's outer face. Deselect this option to redraw complex models faster.

Display normals for multiple surfaces — Select this option to show surface normals when you select multiple surfaces.

Default draft angle — Enter the default angle to use when you create draft surfaces.

Longitudinal detail lines per patch — When you select a surface for editing, additional grey detail lines are displayed to indicate the surface shape. Enter the number of detail lines between each pair of longitudinal curves. Enter **0** to disable detail lines.

Lateral detail lines per patch — When you select a surface for editing, additional grey detail lines are displayed to indicate the surface shape. Enter the number of detail lines between each pair of sectional curves. Enter **0** to disable detail lines.

Default fillet radius — This sets the fillet radius which will be used by default when creating **fillet surfaces**.

For further details, see How do I create a fillet surface?

Keep pcurves generated by filleting — If *ON*, keeps the pcurves when **Trim** is turned off on the **Fillet Surface** dialog.

Keep pcurves generated by limiting — If *ON*, keeps the pcurves when generated by limiting. No pcurves will be generated if **Trim both** is set to *OFF* on the Limit selection dialog.

Suppress internal lines on surfaces — If *ON*, the laterals and longitudinals are only drawn:

- on the surface edges;
- in the direction where the surface is closed;
- along a discontinuity in a surface;
- on a single selected surface.

If *OFF*, all the laterals and longitudinals are drawn on the surfaces.

This option only applies to wireframe view.
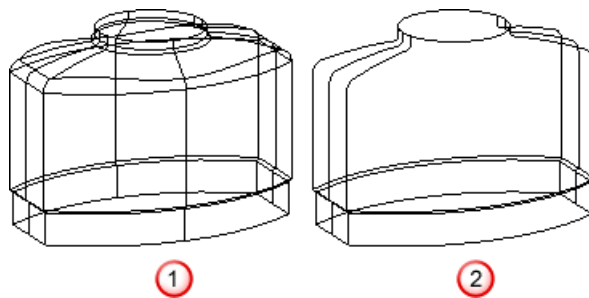
① **Suppress internal lines on surfaces** option *OFF*.

② **Suppress internal lines on surfaces** option *ON*.

Suppress internal lines on surfaces when shaded — This suppresses the drawing of internal lateral and longitudinal lines when shaded for clarity and efficiency.
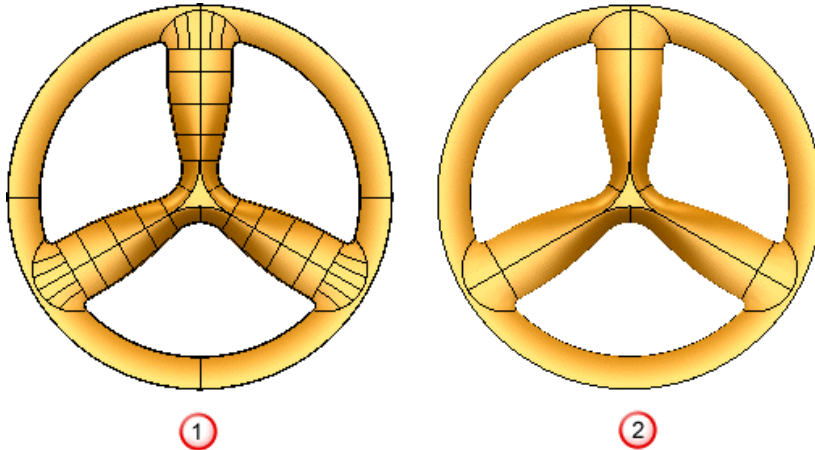
① **Suppress internal lines on surfaces when shaded** option *OFF*.

② **Suppress internal lines on surfaces when shaded** option *ON*.

Suppress internal lines on surfaces in solids — This allows you to set whether you want the internal lines to be drawn on surfaces in solids. If *ON*, internal lines are not drawn.

Suppress internal lines on surfaces in components — This allows you to set whether you want the internal lines to be drawn on surfaces in components. If *ON*, internal lines are not drawn.

Suppress detail on large surfaces — This allows you to set whether you want the detail lines to be drawn on single selected surfaces. If *ON* , detail lines are not drawn in a particular direction when there are more than 15 surface curves in that direction.

Draw continuous interior laterals on NURBS — If *ON*, you can display the interior laterals on a NURB surface.

By default, this option is *OFF*. So only surface curves at discontinuities and surface edges are displayed.
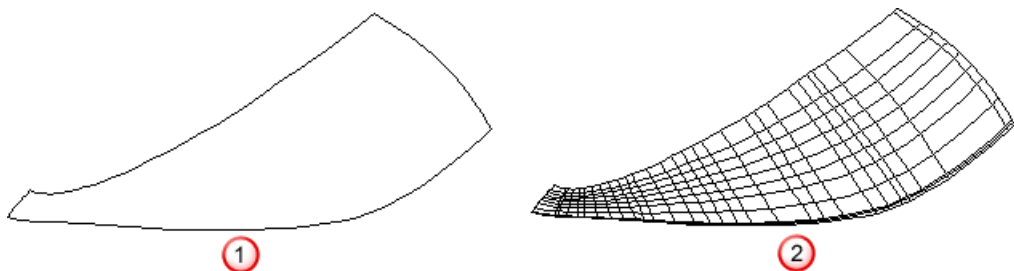
① Draw continuous interior laterals on NURBS option *OFF*.

② Draw continuous interior laterals on NURBS option *ON*.

You can only click curves on the screen if they are displayed. Even if the curves are not displayed, you can use the **Surface Edits** toolbar to select them.

Core/Cavity Separation

Min. visible region (pixels) — This tolerance is set to **Automatic** by default, but also provides a range of fixed values from 1 to 10.

When finding the core and cavity separation for a mold, the program looks at the model from the top and bottom (along the Z axis). Any surfaces which are visible in either direction are assumed to be part of either the core or cavity of the mold.

When calculating the separation, the surface is covered with triangles, however small gaps cannot be totally eliminated. Consequently, tiny areas (of an otherwise invisible surface) can be seen through these gaps. This option allows you to tell the core/cavity separation algorithm to ignore gaps (or other holes) of less than a set size.

Primitives

Use these options for creating primitives (such as blocks and cylinders), extrusions and revolutions.

Create as NURBS — When *ON*, the primitives (such as blocks and cylinders) are created as NURB surfaces.

Convert all surfaces to Power Surfaces — This option affects the behaviour of the **Edit > Convert > Surface** command. It controls how primitive, morph or other special surfaces (other than NURBS or Power Surfaces) are converted.

If *ON*, all types of surfaces are converted to a Power Surface.

If *OFF*, the surface is converted to a NURBS or PowerSurface according to the type of surface on which the surface is based.

Keep wireframe (extrusions and revolutions) — When *ON*, the wireframe is automatically kept in the model. When *OFF*, the wireframe is removed from the model. You can add the wireframe to the model again using the **Create a copy of the sketch** option on **Sketch** tab of the dialog for the extrusion or revolution. The dialog is displayed by double clicking the object.

Create extrusions normal to planar base — When *ON* and you create an extruded surface/solid using a planar base, then the extrusion is normal to the planar base. Otherwise the extrusion is in the positive direction of the axis normal to the principal plane.

Mesh options

This controls the default settings for meshes within models.
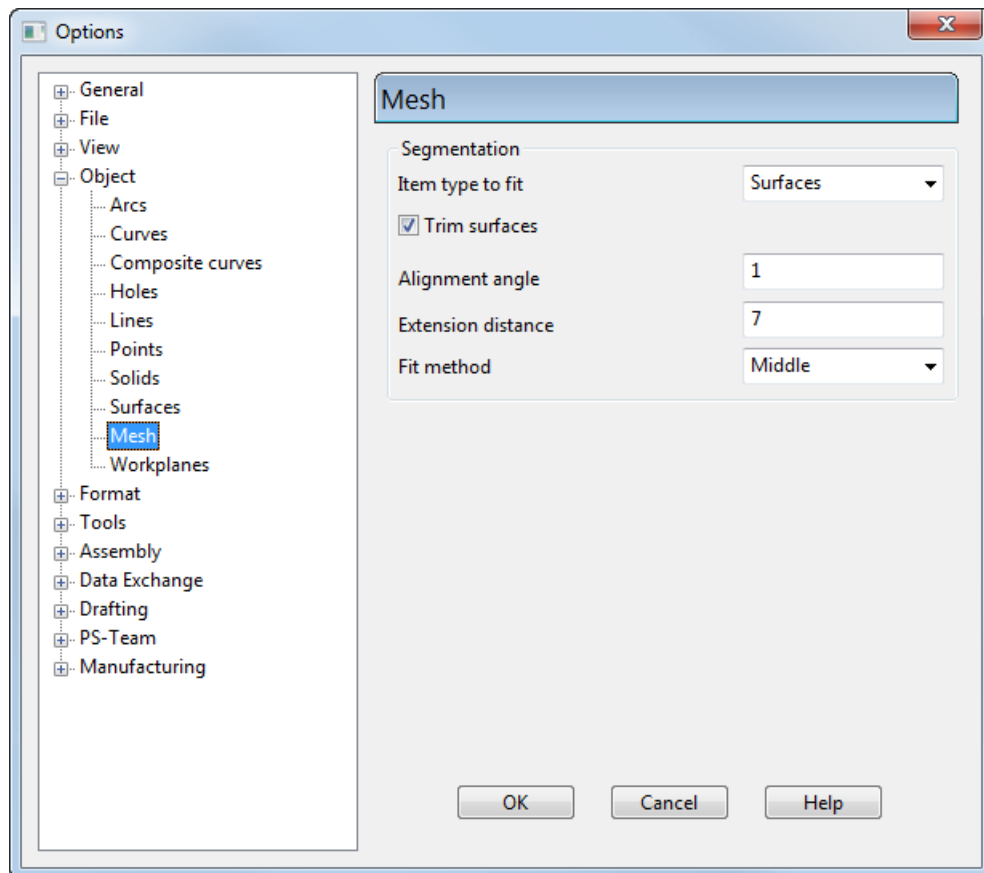
Use the following options to edit the **Segmentation** settings.

Item type to fit — Select **Surfaces** or **Solids** to be the default item type to fit to the mesh when segmenting.

Trim surfaces — Select this option to toggle between creating trimmed or full surfaces when segmenting.

Alignment angle — Enter a value to define the tolerance for aligning primitives with a principal plane. If an axis of a primitive is within the tolerance of a principal plane, the axis is snapped to the principal plane.

Extension distance — Enter a value to define the distance by which a primitive is extended beyond the edge of its region.

Fit method — Select an option from the list to set the default setting for fitting primitive surfaces to a mesh when you are automatically or manually segmenting a mesh.

Workplane options

The **Workplanes** page of the **Options** dialog enables you to set the default workplane options.
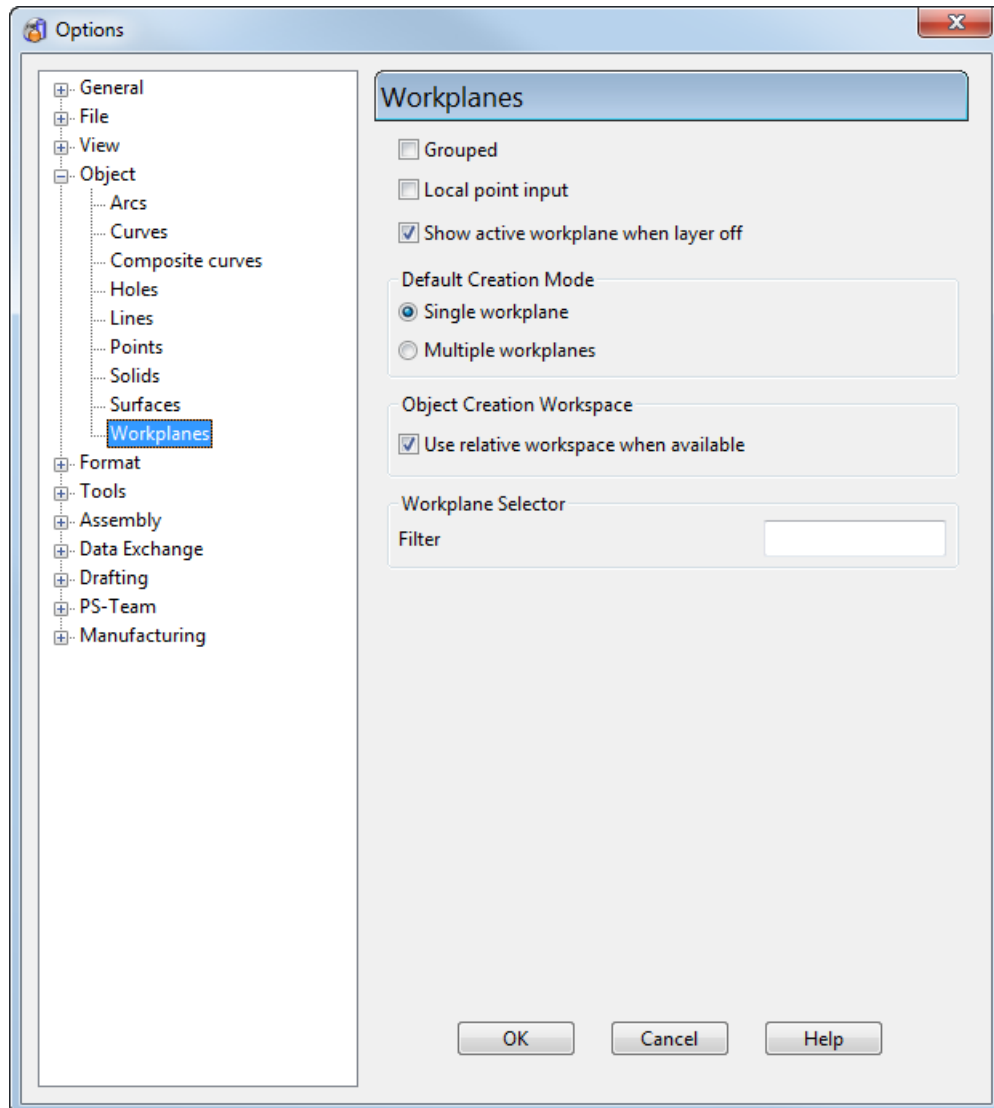
Grouped — When **Grouped** is on, associations are created between the workplane and objects created while the workplane is active. This means that if you reposition the grouped workplane, the associated objects move with it. If a grouped workplane is deleted, the objects remain in place, but their co-ordinates now relate only to the global workspace. If a grouped workplane is ungrouped, the object associations are lost. When **Grouped** is off, each workplane is treated as a simple 3D local coordinates system. These workplanes can be moved, but objects drawn previously remain in place.

Local point input — When this is off and you input a position into the coordinates text box on the status bar, then the position is input as XYZ coordinates. For example, if you type **20 0**, the position is X20 Y0 Z0. With **Local point input** on, the values are input relative to the current principal plane. For example, if ZX is the current principal plane, then 20 0 means a position of Z20 Y0 X0.

Show active workplane when layer off — If *ON*, the active workplane is displayed when its layer is turned off.

Default Creation Mode

You can set the default workplane creation mode so that when you create a workplane using the toolbar, this creation mode is automatically selected.

Object Creation Workspace

Use relative workspace when available — If *ON*, the relative workspace is used if available. If *OFF*, the absolute workspace is used.

Workplane Selector

Filter — You can use a filter to display only certain names in the **Workplane** combo box on the **Status** bar. Enter the filter pattern in the text box.

Format options

The following options pages are grouped under the **Format** category:

- **Levels and Styles** (see page 123)

Levels and styles options

Use this dialog to specify the options for levels and styles.
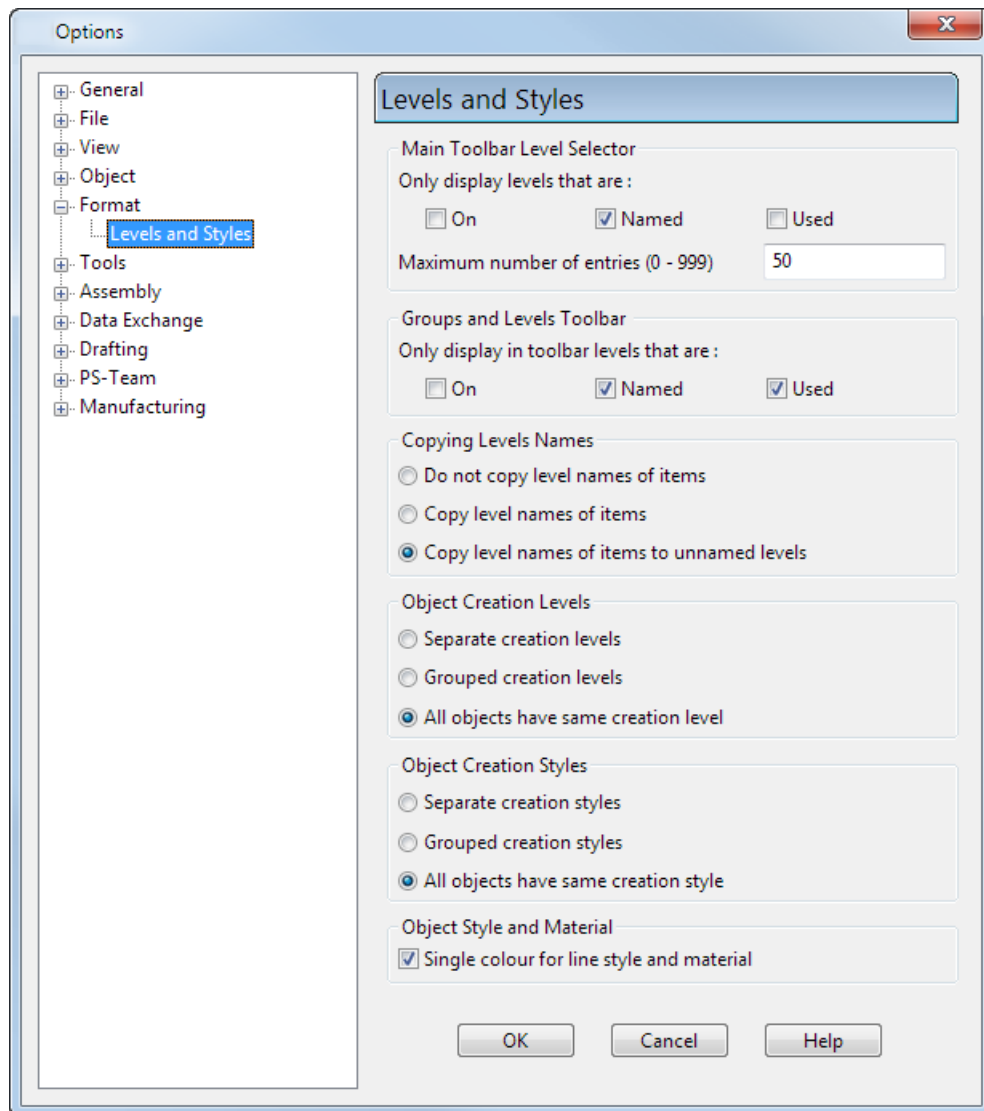
Main Toolbar Level Selector

Use these options to set the defaults for the **Levels** list on the **Main** toolbar (see page 196).
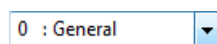
Select one or more of the following options to specify the levels that are displayed in the list on the **Main** toolbar. Use combinations of the options to keep the list of levels in the list to a reasonable size, even when there are many levels in a model.

- **Used** — Display only the levels that are currently in use within the model.

- **Named** — Display only the levels that are named.

- **On** — Display only the levels that are active within the model.

Deselect all these options to display all levels in the **Levels** list. This option does not affect the filter in the **Level** dialog

Max. number of. entries — Enter the maximum number of levels you want to display in the **Levels** list. The default is 50.

Groups and Levels Toolbar

Select one or more of the following options to specify the levels that are displayed in the list on the **Levels** toolbar. Use combinations of the options to keep the list of levels in the list to a reasonable size, even when there are many levels in a model.

- **Used** — Display only the levels that are currently in use within the model.

- **Named** — Display only the levels that are named.

- **On** — Display only the levels that are active within the model.

Copying Levels Names

Use these options to specify the copying of level names when items are copied between models. Select one of the following:

- **Do not copy level names** — When copying an item between models, do not copy its level name.

- **Copy level names of items** — When copying an item between models, copy its level name.

- **Copy level names of items to unnamed level** — When copying an item between models, copy its level name if the destination level is currently unnamed. This is the default setting.

Object Creation Levels

This determines which level an object is created on. You can choose from three options.

Separate creation levels — With this option, each type of object can be created on a different level. For example, if you change the creation level of lines to level 4, then all new lines are created on that level. Other wireframe objects such as arcs and curves are still created on the default level.

Grouped creation levels — With this option, if you change the level that an object is to be created on, then all objects of the same group are also created on this level. That is, wireframe objects are grouped together on the same level, solids and surfaces are grouped on the same level. For example, if you change the creation level of surfaces, then new solids are also created on the new level.

All objects have the same creation level — With this option, if you change the level that an object is to be created on, then all new objects are also created on this level. This is the default setting.

> *This functionality applies only when you change the level that a particular type of object is created on.*

Object Creation Styles

Select one of the following options to specify the style used to create an object:

- **Separate creation styles** — Create each type of object using a different style. For example, if you change the creation style of lines to **StartArrow**, all new lines are created with style **StartArrow**. Other wireframe objects such as arcs and curves are still created using the default style.

- **Grouped creation styles** — If you change the creation style of an object, all objects of the same group (so wireframe objects are created using the same style, solids and surfaces are created from the same style) are also created using that style. For example, if you change the creation style of surfaces, new solids are also created using that style.

- **All objects have the same creation style** — If you change the creation style of an object, all new objects are also created with that style. This is the default setting.

> *This functionality applies only when you change the style of a particular type of object.*

Example : A model has a level and style with the same name (for example *Top*); **Object Creation Level** and **Object Creation Style** are set to similar options. If you change the creation level of an object to *Top*:

- The creation style of that object will automatically becomes *Top*.

- Other objects may change to level and style *Top* depending on the options set for **Object Creation Level** and the **Object Creation Style**.

Tools options

The following options pages are grouped under the **Tools** category:

- **Analysis**
  - **Curve Analysis** (see page 126)
  - **Model Visual Difference** (see page 127)
  - **Surface Analysis** (see page 128)

- **Thickness Analysis** (see page 129)
- **Comparison Analysis** (see page 64)
- **FileDoctor** (see page 134)
- **Macro** (see page 137)

Curve Analysis options

Use this page to change the comb scale.
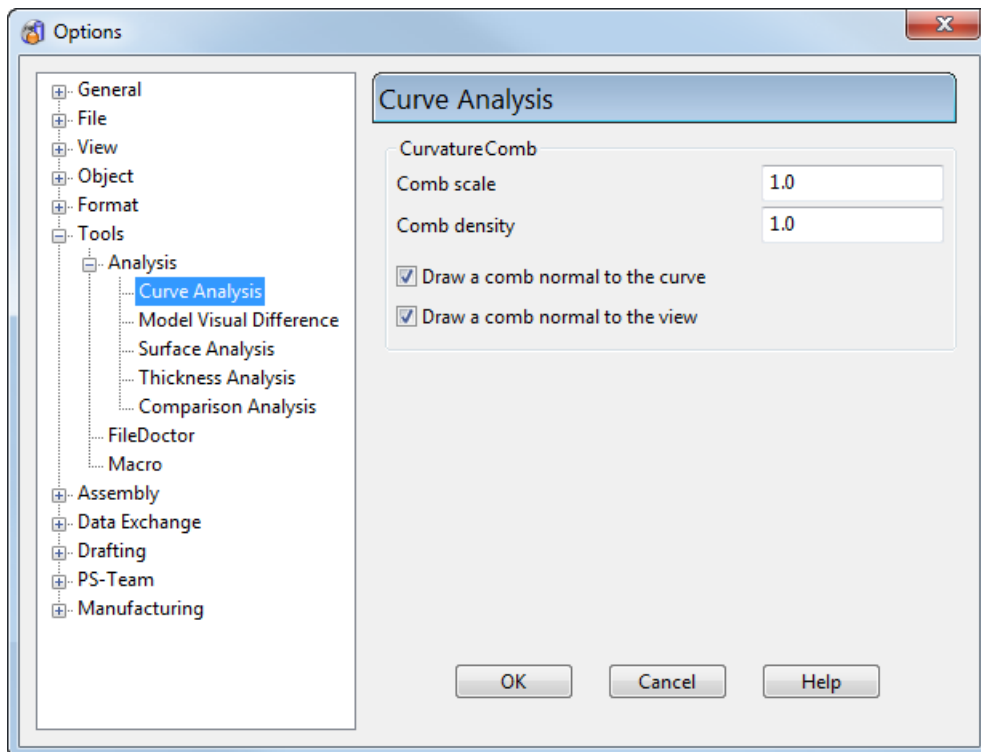
Comb scale — Enter the size of the curvature combs display. The default comb size is multiplied by this value.

Comb density — Enter the comb density. The default density is **1.0**. Increase the density to add more teeth on curvature combs, decrease the density to reduce the number of teeth. The density must be greater than **0.0001**.

> *A high comb density may take a long time to compute therefore it is recommended that you increase the density in small increments.*

Draw a comb normal to the curve — Select this option to draw the curvature combs normal to the curve.

Draw a comb normal to the view — Select this option to draw the curvature combs normal to the view.

Model Visual Difference options

The **Model Visual Differences** options dialog lets you control the appearance and colour of the model wireframes and shading displayed when you use this functionality.

You can display the dialog in one of the following ways:

- Select **Options...** from the **Visual Difference** context menu.
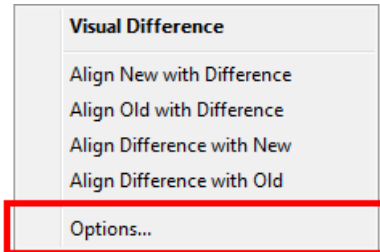
- Select **Model Visual Difference** from the **Analysis** options of the **Tools** branch in the tree of the **Options** dialog.
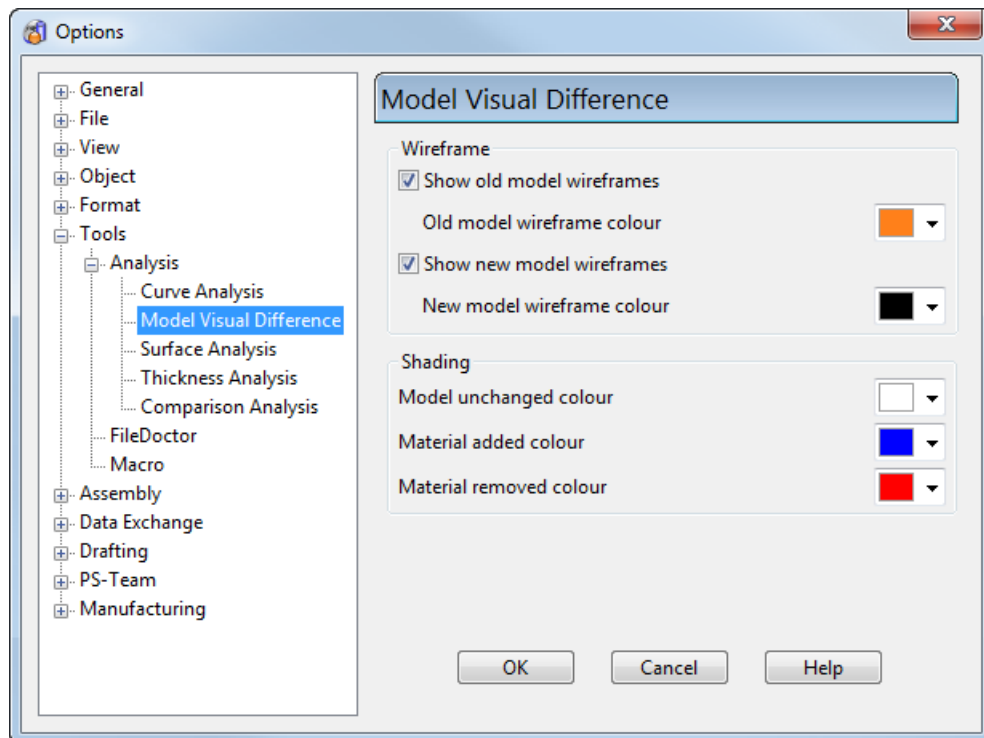
Wireframe

Use these options to control the display of model wireframes.

Shading

Select colours for shading unchanged, added, and removed material.

Surface Analysis options

Use this page to set the options for surface analysis.

Undercut Shading

Use these options to set the parameters that define if a surface is considered *undercut* (see page 49).

Draft angle — Enter a value between **+90°** and **-90°**. The default value is **0**.

Draft warning angle — Enter a value, between **0°** and **90°**, to show areas of the surface where the angle is close to the angular distance of the **Draft angle**.

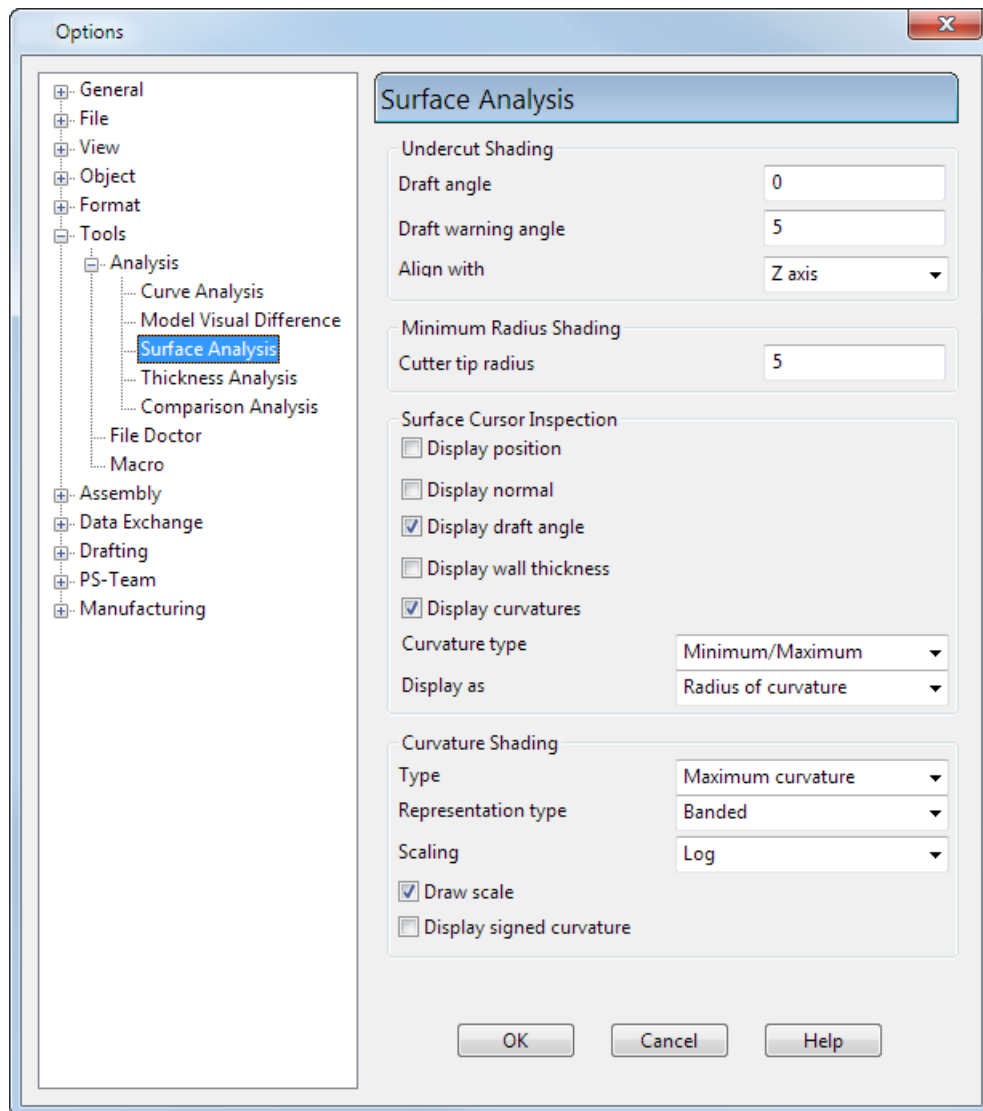**Align with** — Select an axis of the active workplane from the list or align the undercut shading with the **Principal axis** (controlled by the axis buttons 🗗🗗🗗 on the **Status** toolbar).

Minimum Radius Shading

Cutter tip radius — Enter the cutter tip radius between **0** and **20**. The default value is **5**.

Surface Cursor Inspection

Use these options to specify the data that is displayed when inspecting a surface.

Curvature Shading

These options determine what is displayed when using curvature shading (see page 51).

Thickness Analysis options

Use this dialog to specify options for thickness analysis.

Computation method — Select an option to define the way that the thickness is calculated.

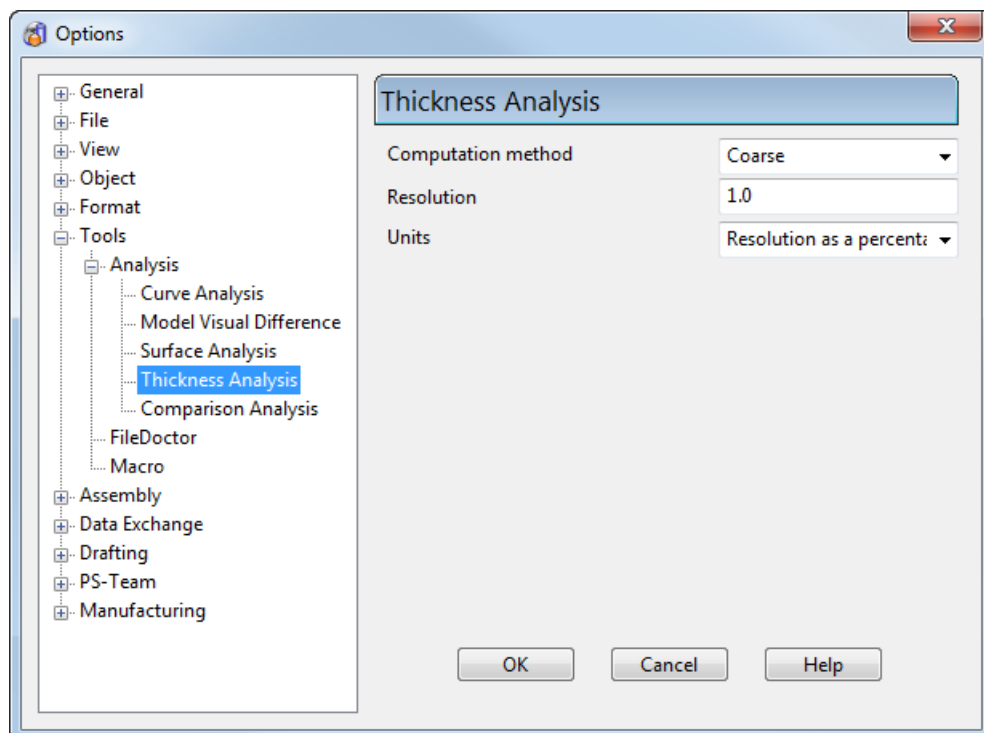- **Coarse** — Coarse, but fast calculation. This is the default setting and should be suitable for most users

- **Approximate** — Slower, but more accurate calculation.

- **Precise** — This option gives you micron accuracy, but this type of calculation takes longest to complete.

Resolution — This enables you adjust the resolution of the colourmap, by adjusting the number of points on the model for which the thickness is calculated. The resolution can be entered as a percentage of the model size or as a figure to represent the thickness. For best results, enter the wall thickness in the region you want to analyse, particularly for models where the wall thickness is thin. So, if you already know that the relevant wall thickness is 3mm, enter this as your resolution, with **Resolution in model units** selected from the **Units** options.

Units — Select from **Resolution as a percentage of the model size** (default), or **Resolution in model units** to define how you enter the **Resolution**.

Comparison Analysis options

Use this page to specify the parameters to be used in **Comparison Analysis**.

- **Display legend** — Select this option to show or hide the error legend in the graphics window.

- **Always display errors** — Select this option to automatically show or hide deviations after they have been calculated.

- **Display error under cursor** — Select this option to show or hide the numerical deviation beneath the cursor when you hover your cursor over the comparison.

- **Colour bands** — Select an option from the list to change how the deviations are displayed on the model:

    Continuous — The deviations are displayed using a continuous scale.

    Two — The deviations are displayed using two distinct colours.

Three — The deviations are displayed using three distinct colours.

Five — The deviations are displayed using five distinct colours.

Ten — The deviations are displayed using ten distinct colours.

Twenty — The deviations are displayed using twenty distinct colours.

Forty — The deviations are displayed using forty distinct colours.

*Colour bands is not available if **Display errors to tolerance** is selected.*

- **Width of error lines** — Enter a value to define the width of the error bands shown on the model when curve deviations are displayed.

- **Display errors to tolerance** — Select this option to display the deviations in two or three colour bands. The bands are set according to the general tolerance.

  This option is deselected by default.

  If **Display errors to tolerance** is deselected, select the required number of **Colour bands** from the list.

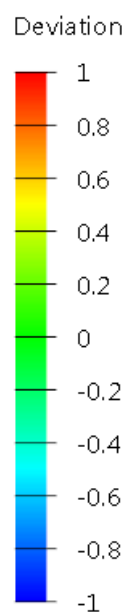- **Display absolute range** — Select this option to display the absolute (modulus) values of deviation. That is the range of errors from **0** to the maximum value.

Deviation

1
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0

- **Set maximum value to maximum error** — Select this option to set the maximum value of the error range to the largest deviation.

- **Maximum value of error range** — Enter a value into the field to define the maximum value of the error range. In the following example :

  - **Maximum value of error range** is set **1**.

  - **Colour bands** are **Continuous**.

Deviation

1
0.8
0.6
0.4
0.2
0
-0.2
-0.4
-0.6
-0.8
-1

*This option is unavailable if **Set maximum value to maximum error** is selected.*

- **Sample Point Density** — Use the slider to control the number density of points used to calculate differences between your models. A larger **Sample Point Density** produces more accurate results, with a longer computation time. A smaller **Sample Point Density** produces less accurate results, with a shorter computation time.

File Doctor options

This sets the options to run the **File Doctor** while working on a model or before it is saved. Depending on your level of security, some or all of the options on the **File Doctor** options dialog may be unavailable. This is intended to improve the chance of you repairing the model.
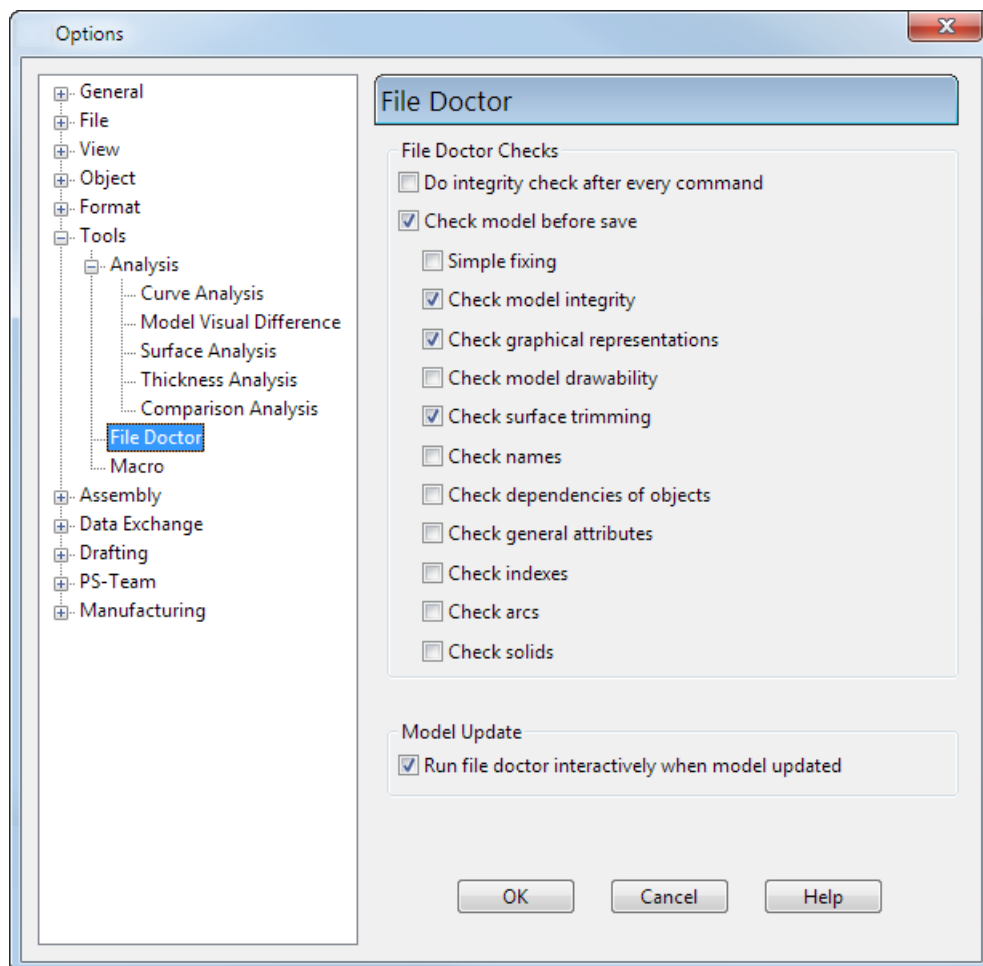
Do Integrity check after every command — PowerSHAPE checks the model for integrity after each command.

If a problem is found, the last commands are automatically undone until the model is in a state where the integrity check passes. This mode of checking is designed for use with problem models.

If **Do Integrity Check after every command** is selected, PowerSHAPE also does a **Check & Fix** when you open a model. As a result, the model may lose some data. If you want to recover this lost data, we suggest you immediately do the following:

1 Deselect the **Do Integrity check after every command** option.

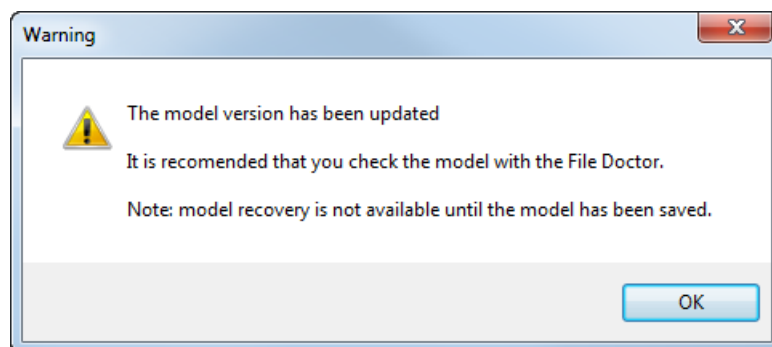**2** Close the model without saving it.

3 Open the model again.

Check model before save — PowerSHAPE uses the **File Doctor** to check your models each time you save using the selected **File Doctor** options. These checks are applied at file save, file reset, and when upgrading a model.

> *You can change the **Check model before save** options only if you have security levels 1 and 2. Usually, system administrators have security level 1 and design office managers have security level 2.*
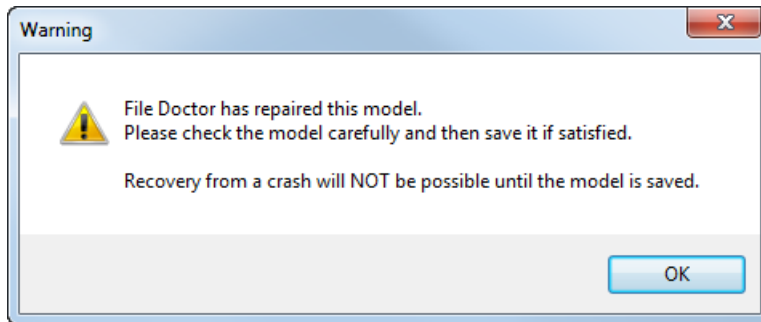
Run File Doctor interactively when model updated — Specify how the **File Doctor** is run.

- If **Run File Doctor interactively when Model updated** is selected (default) and the model version has been updated, the **File Doctor** is run interactively. The following warning dialog is displayed:

> *You are recommended to run **File Doctor** when opening an old version model in a version of PowerSHAPE that uses a later version model. However, if you select **Close** on the **File Doctor** dialog, the model is unchanged.*

- If **Run File Doctor interactively when Model updated** is deselected and the model version has been updated, the **File Doctor** is run automatically (as in previous versions of PowerSHAPE).

- If **Run File Doctor interactively when Model updated** is selected (default) and the model version has been updated, the **File Doctor** is run interactively. The following warning dialog is displayed:
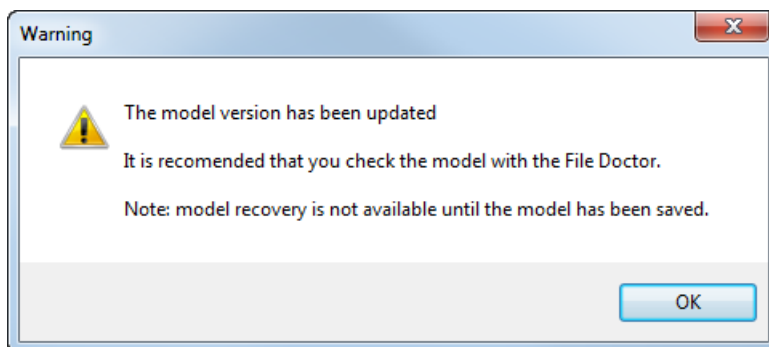

*You are recommended to run **File Doctor** when opening an old version model in a version of PowerSHAPE that uses a later version model. However, if you select **Close** on the **File Doctor** dialog, the model is unchanged.*

- If **Run File Doctor interactively when Model updated** is deselected and the model version has been updated, the **File Doctor** is run automatically (as in previous versions of PowerSHAPE)
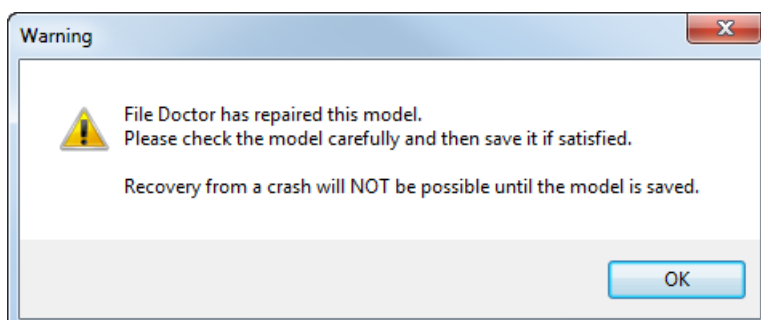
Macro options

Use this page to define the mouse options when recording macros.
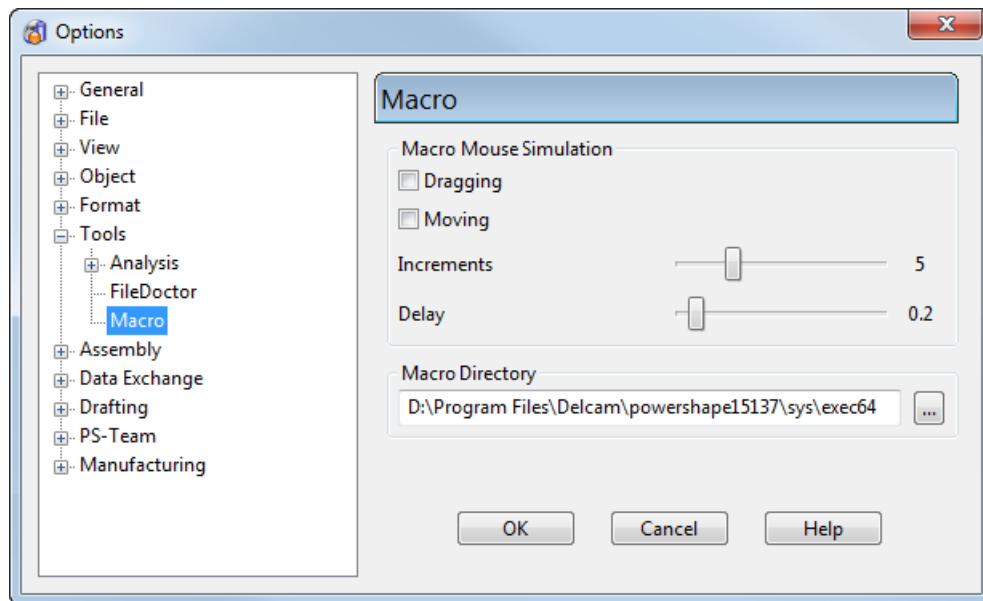
Macro Mouse Simulation

These options enable a macro to show mouse movements.

When you record a macro, the button clicks and their positions are recorded, but by default the mouse movements between clicks are not recorded. This keeps the macro file to a reasonable size and optimises performance. However, for demonstration and testing purposes, this option can be useful to see the dragging and rubber-banded mouse movements.

Selecting the **Dragging** and **Moving** options affects subsequently recorded macros. When you run macros, the mouse is shown dragging and moving between the button press and release points. The mouse movement characteristics are controlled as follows:

Increments affects the number of steps between points. A large value makes it move smoothly, but possibly more slowly. A low value means larger increments and faster movement, but possibly causing jerky motion. The default values provide a good compromise between speed and smoothness.

Delay controls the wait time at each incremental step.

Macro Directory

This is the default directory where recorded macros are stored and run from.

Assembly options

These options are part of the Assembly modelling functionality.

For further details, see Assembly options.

Data Exchange options

The following options pages are grouped under the **Data Exchange** category:

- **Version 8 / Surfaces** (see page 139)
- **Delcam Exchange** (see page 143)
- **IGES** (see page 144)
- **U3D / PDF** (see page 145)
- **HPGL** (see page 146)
- **Parasolid** (see page 148)
- **PIC** (see page 149)
- **Triangle/Mesh** (see page 149)
- **FBX** (see page 152)

Version 8 / Surfaces options

This page contains the options for data exchange.
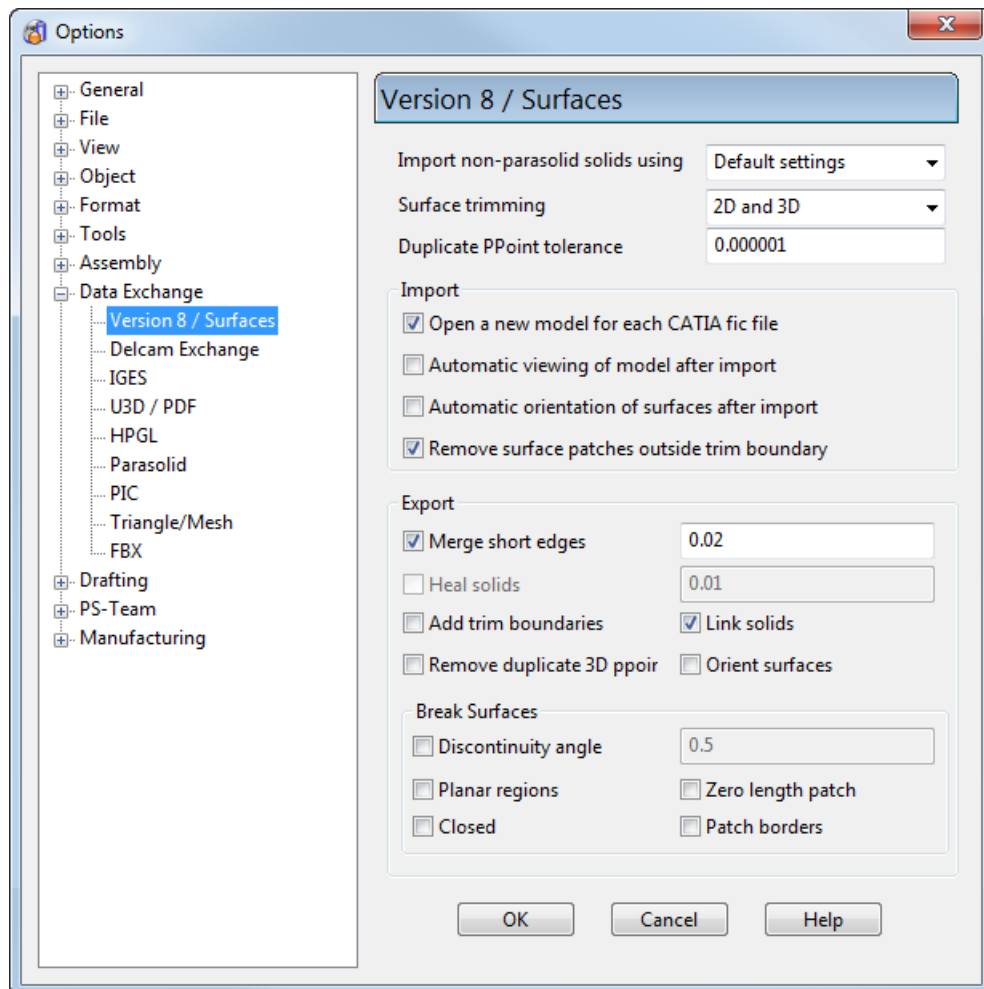
Import non-parasolid solids using — Select the method to use to import non-parasolid solids from:

- **Default settings** — Select this option to detect the file type and import solids as Parasolids or version 8 solids depending on the file type:

  - If the file is from a solid modeller, it is imported as a Parasolid.

  - If the file is not from a solid modeller, it is imported as a version 8 solid.

- **Parasolid** — Select this option to import a non-parasolid solid as a parasolid

- **Version 8** — Select this option to import a non-parasolid solid as a version 8 solid.

- **Surface** — Select this option to import a non-parasolid solid as surfaces.

Surface trimming — In an IGES file, a trimming curve can be defined as either 2D parameter or 3D coordinate values. The **Surface trimming** options enable you to set:

- whether a trimming curve is imported from an IGES file as either 2D parameter or 3D coordinate values.

- whether a pcurve is exported to an IGES file as 2D parameter or 3D coordinate values.

Select from:

2D — Only the 2D parameter values are imported or exported.

3D — Only the 3D coordinate values are imported or exported.

2D and 3D — The 2D parameter values are imported, but if the trimming curve cannot be drawn using the 2D curve values, the 3D coordinate values are imported. Both the 2D parameter and 3D coordinate values are exported.

None — No trimming curves are imported or exported, so untrimmed surfaces are imported or exported.

You are advised to use the **2D** option when importing Parasolid/Unigraphics, SolidWorks/SolidEdge.

Duplicate PPoint tolerance — This controls the tolerance for removing duplicate PPoints when importing files. If two PPoints lie within this tolerance, they are treated as duplicate points and one of the PPoints is removed when the file is imported.

Import

The following import options are available:

Open a new model for each CATIA fic file — Catia export files *.cat and *.exp are made up of a number of Catia fic files. By default, each fic file is imported into a separate new model. You can import the fic files into the same model by turning off this option.

Automatic viewing of the model after import — If selected, the model is viewed automatically when imported. This also turns on all the levels used and displays the model using the **zoom full** viewing option.

Automatic orientation of surfaces after import — If selected, normals are orientated automatically to surfaces when the model is imported. This points the normals of surfaces on each level in the same direction.

Remove surface patches outside trim boundary — If selected, surface patches outside the trim boundary are removed. If deselected, surface patches are retained.

Export

The following export options are available:

Merge short edges — Enter a tolerance for merging short edges. Edges that are shorter than the tolerance are merged.

Heal solids — *for future development.*

Add trim boundaries — Select this option to add trim boundaries to the edges of all surfaces that are untrimmed.

Link solids — Select this option to find and store the linked half-edges of a solid, where a half-edge is a segment of a boundary of a face.

Remove duplicate 3D ppoints — Select this option to remove all duplicate 3D ppoints within general tolerance.

Orient surfaces — Level by level, all the normals of multiple surfaces are orientated so that the outside of two adjacent surfaces match. The model is unaltered.

Break Surfaces

These options determine how to break a surface when it is exported.

Discontinuity angle — If a surface contains a tangent discontinuity, it is broken at that point. A tangent discontinuity in a surface is defined by angle given in the box next to this option.

Planar regions — Select this option to break surfaces at the edges of planar regions.

Zero length patch — If a surface contains a zero length patch, it is split on either sides of the patch.

Closed — Select this option to break a closed surface into two pieces.

Patch borders — Select this option to break all surfaces at all longitudinal and lateral curves.

See Also

Export dialog

Delcam Exchange options (see page 143)

DDX options

IGES options (see page 144)

STL/DMT options (see page 149)

Delcam Exchange options

This page contains the options for data exchange.
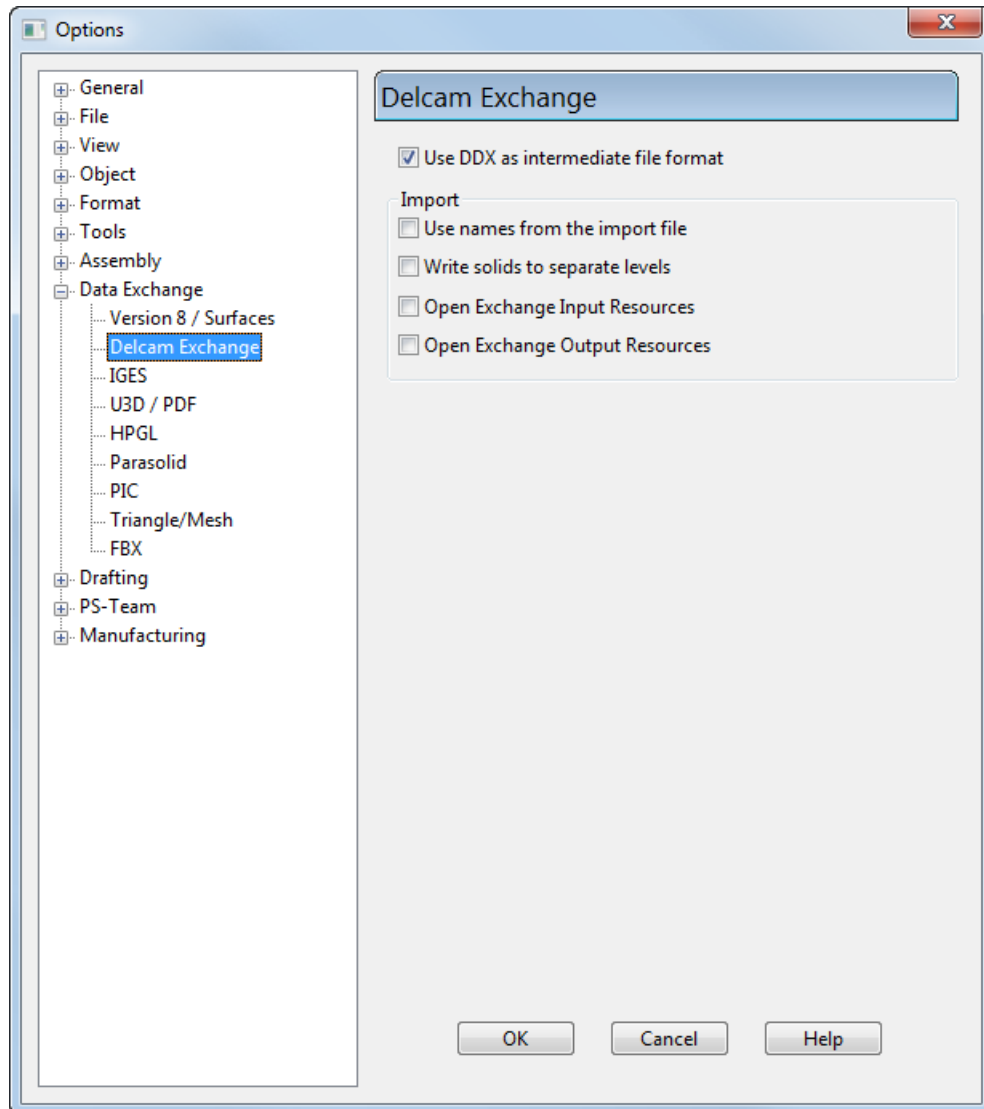
Specify the options to be used with Delcam Exchange:

Use DDX as intermediate file — Select this option to use .ddx as the intermediate file format during import and export. Deselect it to use the .dgk file format.

Use names from the import files — Select this option to use the names of the items in the imported file when importing using Delcam Exchange.

Write solids to separate levels — Select this option to place individual solids onto separate levels, when importing a file containing a number of solids, regardless of their previous level information.

Open Exchange Input Resources — Select this option to display the Delcam Exchange **Input Resources** dialog when importing.

Open Exchange Output Resources — Select this option to display the Delcam Exchange **Output Resources** dialog when importing.

IGES options

Use this page to set how data is imported from or exported to IGES files.

> *If you have selected a Target System on the File Export dialog, the export settings for the Target System are displayed automatically.*
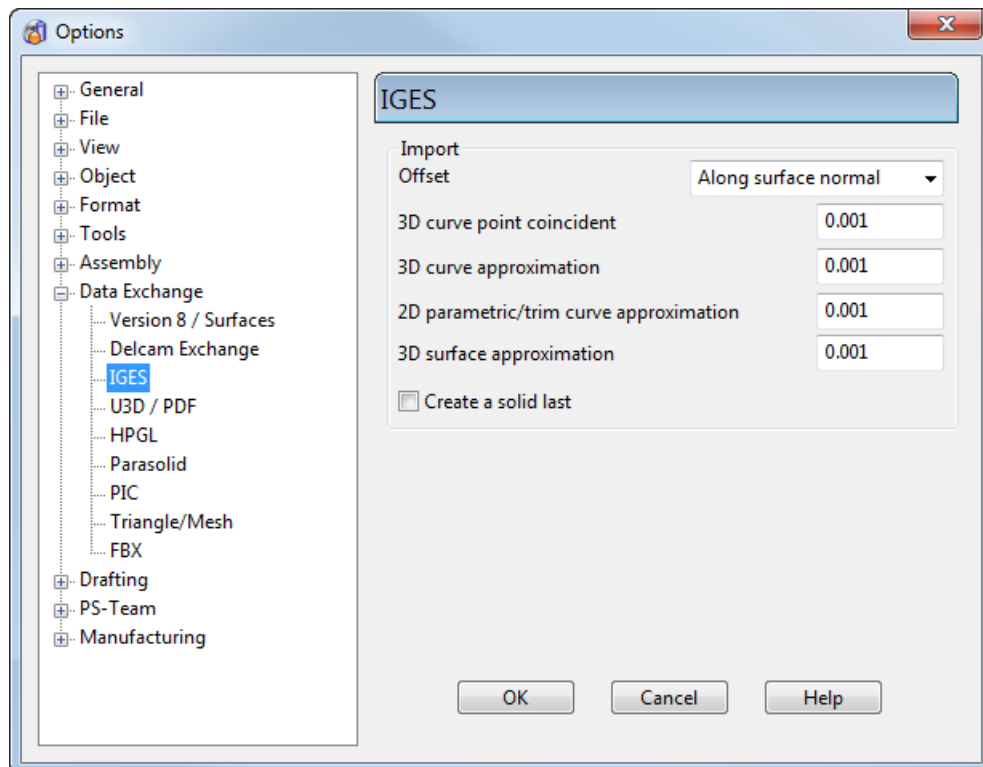
Import

The following import options are available:

Offset — Select the offset direction of the offset surface when importing IGES files. You can choose the direction to be along the surface normal or use the indicator in the IGES file.

The next five items on the dialog are tolerances used to import IGES files.

> *When you import an IGES file, these tolerances are used regardless of those in the IGES file. For example, when importing from a solid modelling program with exceptionally high internal tolerances, the file is imported using the tolerances on this page rather than the really high values saved with it.*

If you change the tolerances, and they are either too small or too large, a warning is displayed indicating the minimum or maximum recommended value you can have.

3D curve point coincident — This is the maximum distance between two 3D points at which these points occupy the same position in 3D space. If more than this tolerance, the points are said to be distinct.

3D curve approximation — Certain types of curves cannot be imported into PowerSHAPE, so they are changed to curves which can be read. This tolerance controls the maximum deviation of the approximated curve from the original curve.

2D parametric/trim curve approximation — All 2D trimming curves are converted to pcurves. This tolerance controls the maximum deviation of the created pcurve from the original trimming curve.

3D Surface approximation — Certain types of surfaces cannot be imported, therefore they are changed to surfaces which can be read. This tolerance controls how much the approximated surface differs to the original surface.

Create a solid last — Select this option to create a solid last from a .pslast file

U3D / PDF options

Use this dialog to specify the U3D/PDF options
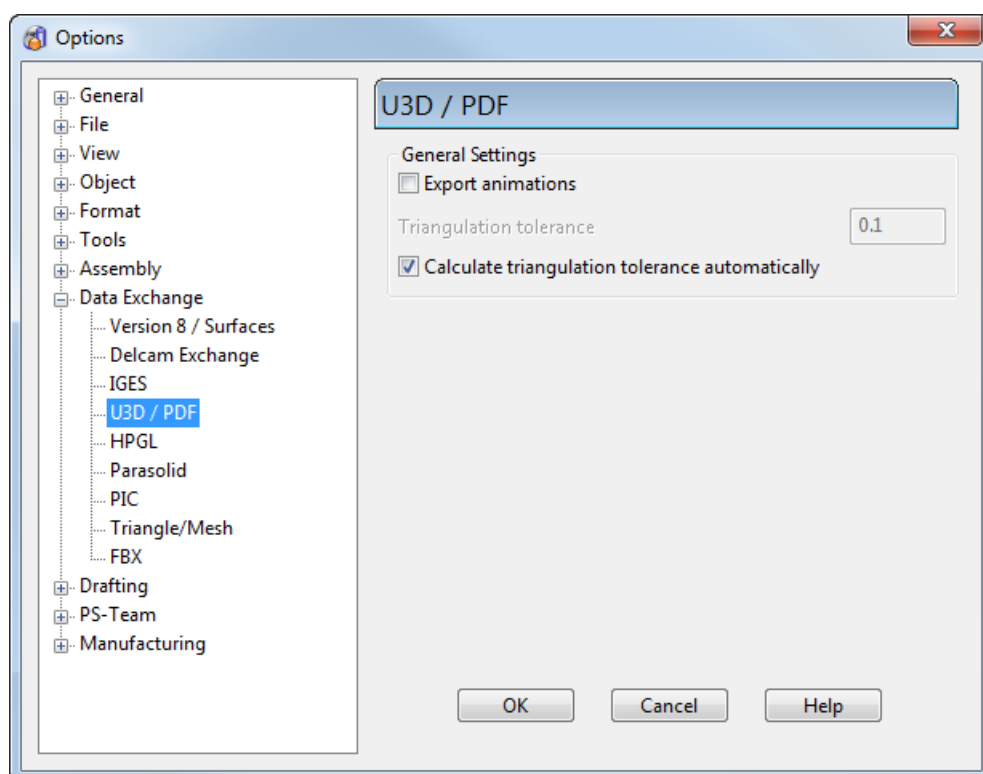
General Settings

Export animations — If selected, automatically generate animation if it is not present.

Triangulation tolerance — If available, set the tolerance used to create triangles for export.

Calculate triangulation tolerance automatically — If deselected, set the tolerance using the **Triangulation tolerance** option.

HPGL options

Use the **HPGL** options page to specify the conversion factors to be used when exporting to HP plotter format.
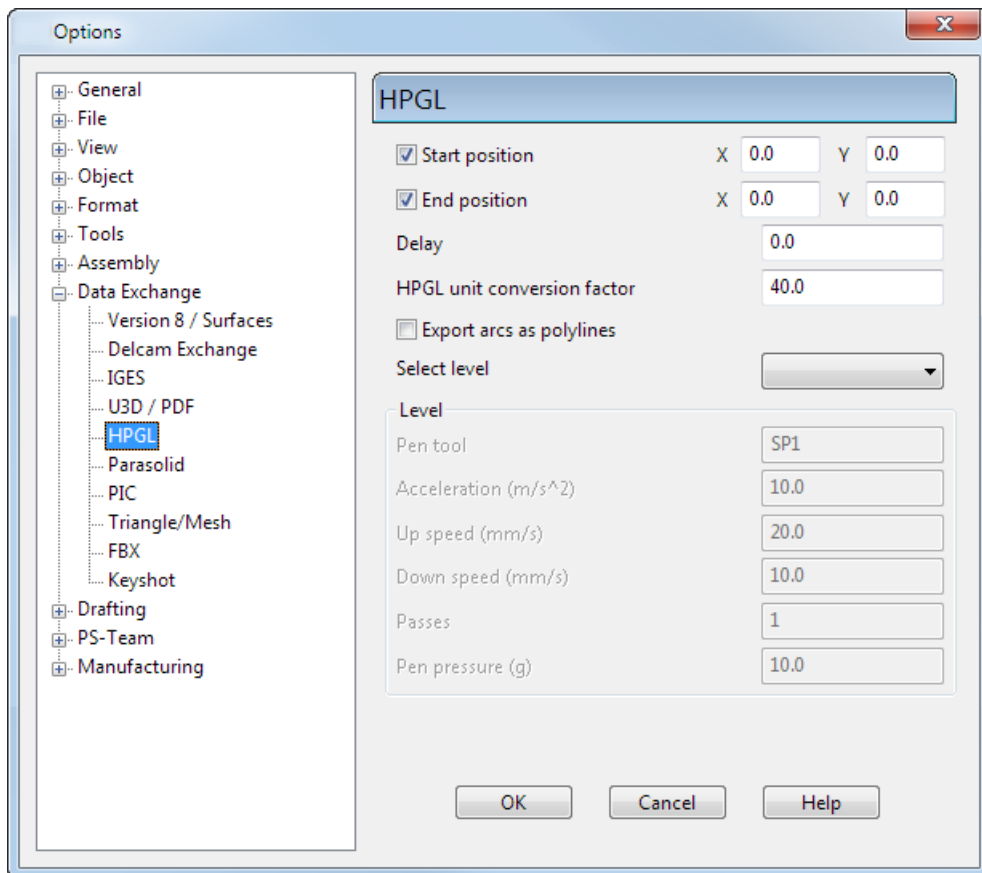
Start position — Enter the X and Y positions to define the start position of the tool.

End Position — Enter the X and Y positions to define the end position of the tool.

Delay — The delay for the tool movement.

HPGL conversion factor — Enter a value to define the conversion factor to be applied when PowerSHAPE units are converted to HPGL units.

Export arcs as polylines — If *ON*, arcs are exported as polylines. If *OFF*, arcs are exported as arcs.

Select level — Select a level from the drop down list. All the objects that you assign to a particular level are grouped together when exported.

Level

These options let you select a level and define its attributes.

- **Pen Tool** — Tool name.
- **Acceleration** — Speed of the tool.
- **Up Speed** — Up speed of the tool.
- **Down Speed** — Down speed of the tool.
- **Passes** — Enter the number of passes.
- **Pen pressure (g)** — Pressure for the tool.

Parasolid options

Use the **Parasolid** options page to specify the options to be used when importing and exporting Parasolid files.
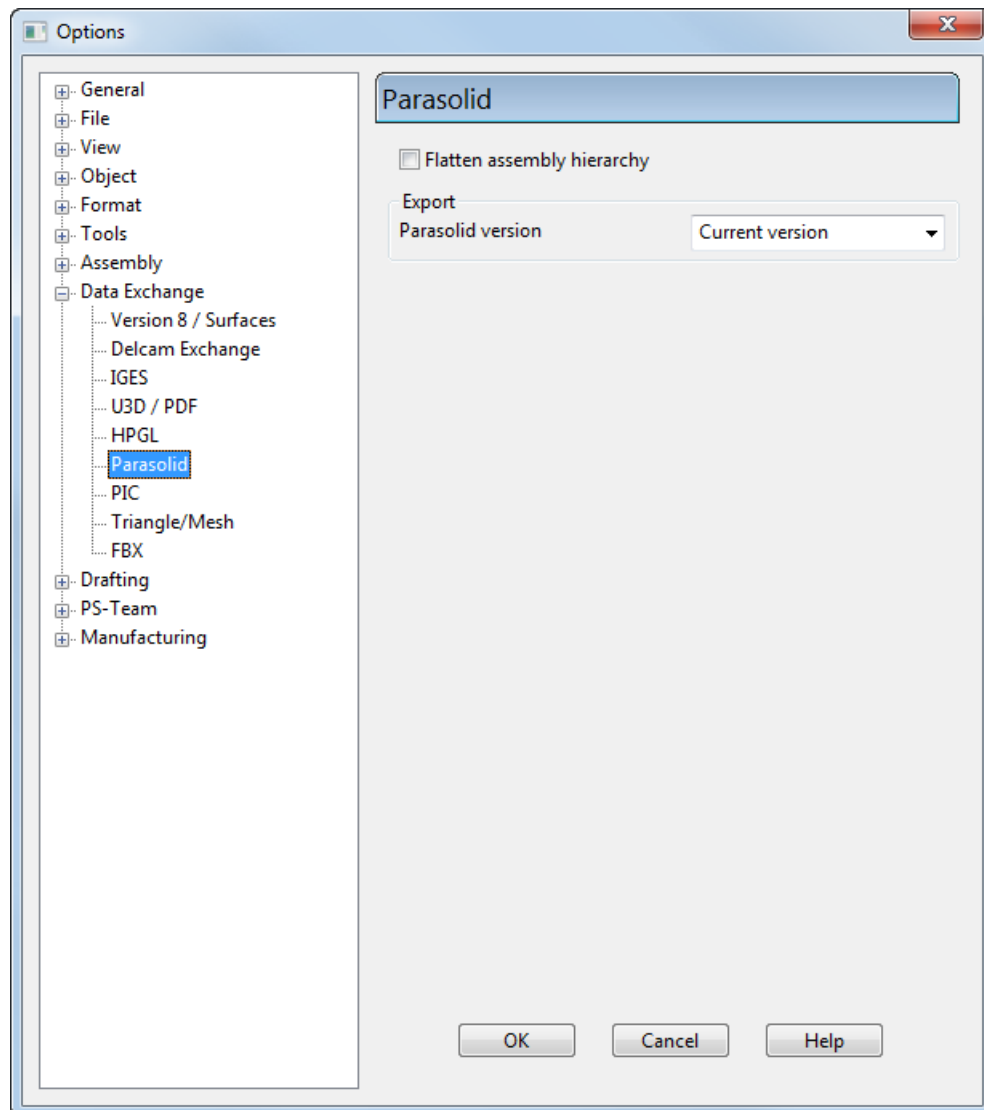
Flatten assembly hierarchy — Flatten the assembly hierarchy when exporting or importing Parasolid files.

This option should be used when exporting Parasolid files to a system that does not support assemblies. It can also be used if you wish to work with the geometry as individual solids without the assembly structure.

Parasolid version — Use the drop-down list to select the **Parasolid version** to be used when exporting. The default is **Current version**; use the drop-down list to select export to an earlier version.

PIC options

Use this dialog to define settings to be applied to **Picture** (*.pic*) files
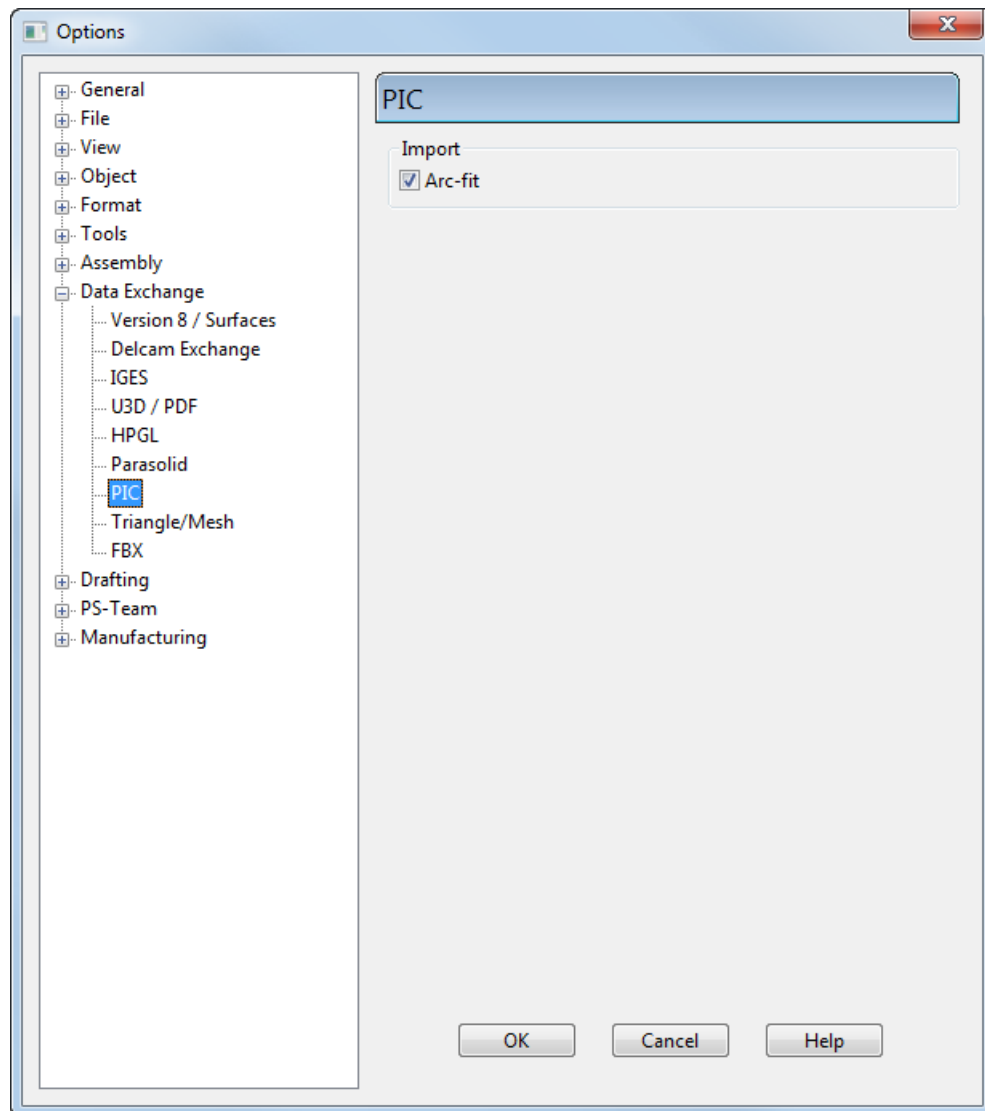
Arc-fit — If *ON*, arc-fitting will be done automatically when a *.pic* file is imported. If *OFF*, no arc fitting is carried out when importing a *.pic* file. This option is *ON* by default

Triangle/Mesh options

This sets the options for exporting STL/DMT files.

If you have selected a Target System on the File Export dialog, the export settings for the Target System are displayed automatically.

DMT options are only applicable when using the built-in local translator.
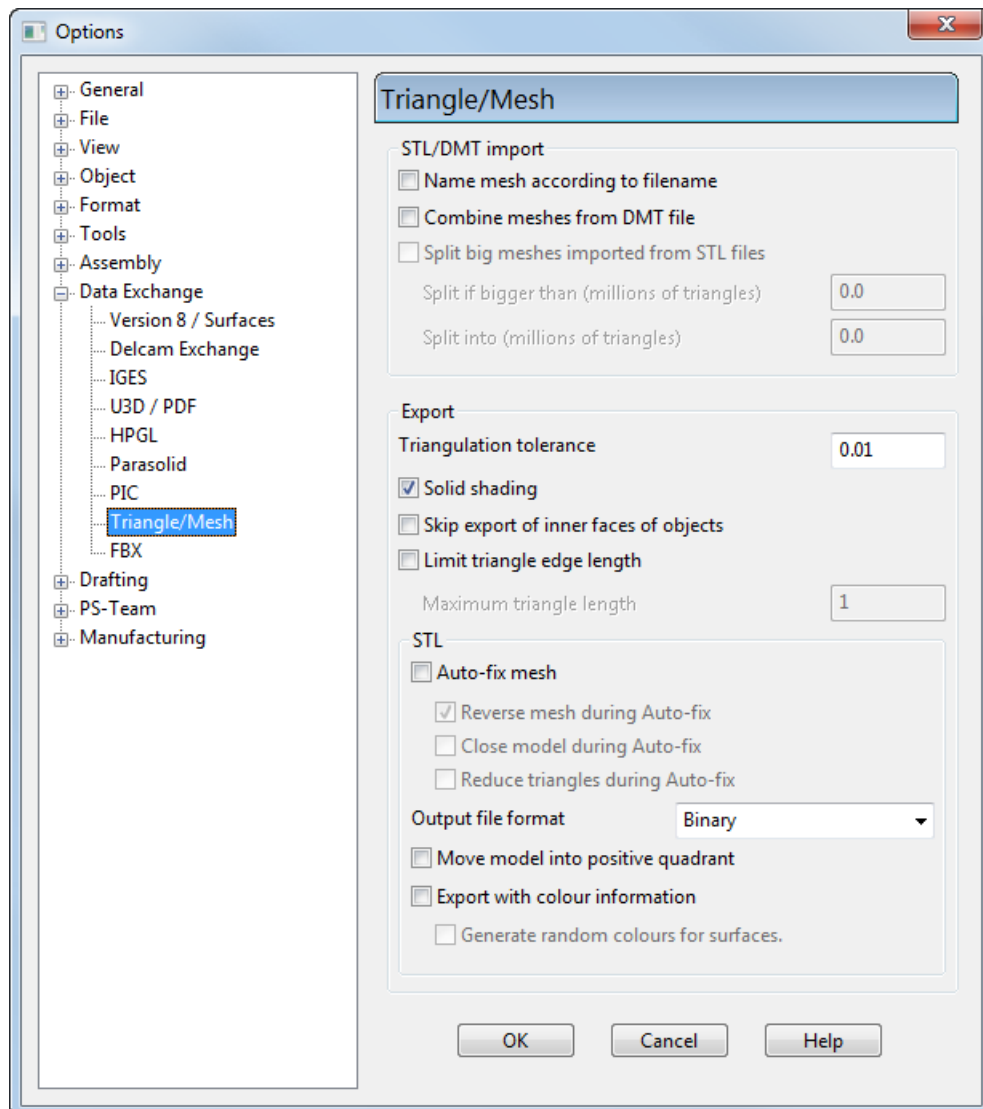
STL/DMT Import

Name mesh according to filename — Select this option to use the filename as the name for the mesh. If necessary, a numerical suffix will be added to ensure that the mesh name is unique. This option is deactivated by default.

Combine meshes from DMT file — If selected, everything in a DMT file is combined into a single mesh. This option is deselected by default.

Split big meshes imported from STL files — *This option is under development*.

Export

These options are not applicable when the user switches to Delcam Exchange.

Triangulation tolerance — This is the tolerance used for exporting triangles.

Solid shading — Uses the improved shading of watertight solids when triangulating for the export of triangles.

Skip export of inner faces of objects — Select this option to skip the inner faces of the objects to reduce the file size when they are exported to .obj.

Limit triangle edge length — If selected, the length of a triangle is limited to the given value.

Maximum triangle length — If **Limit triangle length** is selected, you can enter a value in the text box.

STL

Auto-fix mesh on STL export — This option is deselected by default. If selected, the new triangles are automatically fixed so that the mesh is closed. Select this option to display the following:

> **Reverse mesh during Auto-fix** — The exported triangles are reversed when fixing triangles on STL export.

> **Close model during Auto-fix** — The model is closed when fixing triangles on export. The option is deselected by default.

> **Reduce triangles during Auto-fix** — Reduce the number of triangles on export.

Output file format — Select the output file format from the drop-down list. Files can be exported in binary or ASCII format.

Move model into positive quadrant — If selected, all triangle coordinates are offset so that they are all positive.

Export STL with colour information — If selected, the materials applied to surfaces are preserved when the file is exported and the colours that are stored with the individual triangles are exported.

Generate random colours for surfaces — If selected, this assigns materials randomly to exported surfaces.

FBX options

Use this dialog to define settings to be applied to **Filmbox** (.*fbx*) files.
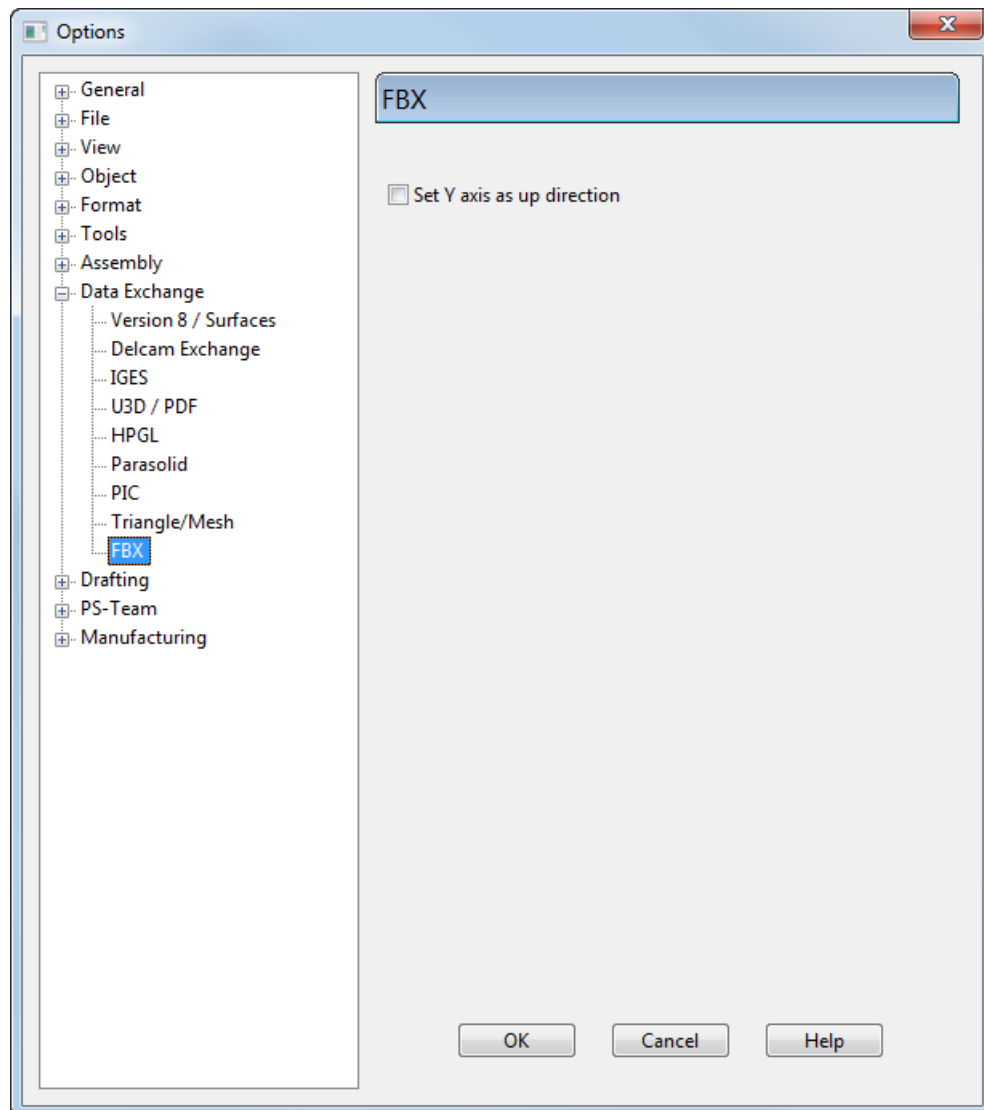
Set Y axis as up direction — Select this option to set the Y axis as the up direction for imported .*fbx* files.

Drafting options

The following options pages are grouped under the **Drafting** category:

- **Dimensions** (see page 155)
  - **Annotation**
  - **Lines**
  - **Tolerance / Dual**
- **Views** (see page 155)

- **General**
- **Section**
- **Hidden**
- **Centrelines**
- **Tapped Details**
- **Line Styles**
- **Balloon** (see page 153)
- **Bill Of Materials** (see page 153)
- **Drawings** (see page 154)
- **Hatching** (see page 154)
- **Schedule** (see page 154)
- **Text** (see page 154)

Balloon options

This is part of the Drafting functionality.

For further details, see Setting the balloons on the Options dialog.

Bill of Materials

Use this dialog to specify the **Rough Oversize Value**.

1 Select **Tools > Options > Drafting > Bill of Materials** to display the **Options** dialog.
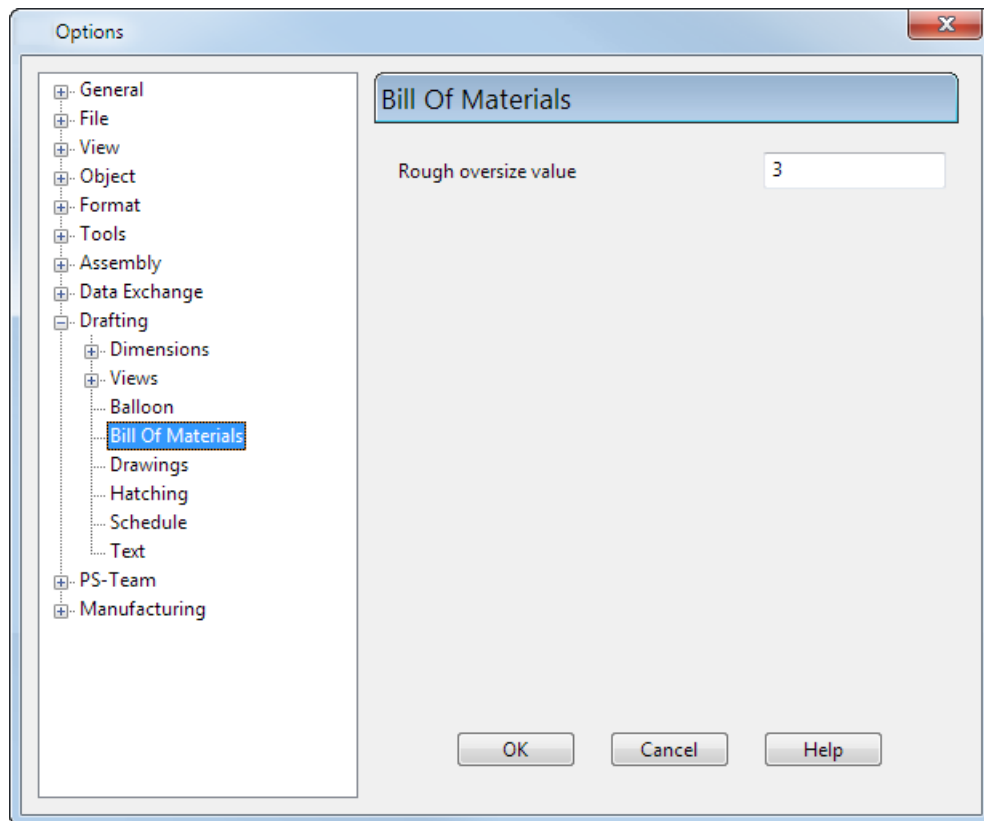
2 Enter **Rough oversize value**. This value is the additional allowance that is added to the bounding box to calculate the stock size required to make the item.

Drawings options

This is part of the Drafting functionality. For further details, see Changing the settings of a drawing.

Hatching options

This is part of the Drafting functionality. For further details, see Changing the default hatching.

Schedule options

This is part of the Drafting functionality. For further details, see Setting the schedule options on the Options dialog.

Text options

This page contains options for creating text in the Drafting module. For further details, see Changing the default settings for text.

Dimensions - Annotation/Lines/Tolerance options

This is part of the Drafting functionality. For further details, see Editing dimension defaults using the Options dialog.

Drafting Views options

This is part of the Drafting functionality. For further details, see Changing the settings of a view in Delcam Draft..

PS-Team options

Specify the PS-Team server and the user details using the PS-Team dialog.
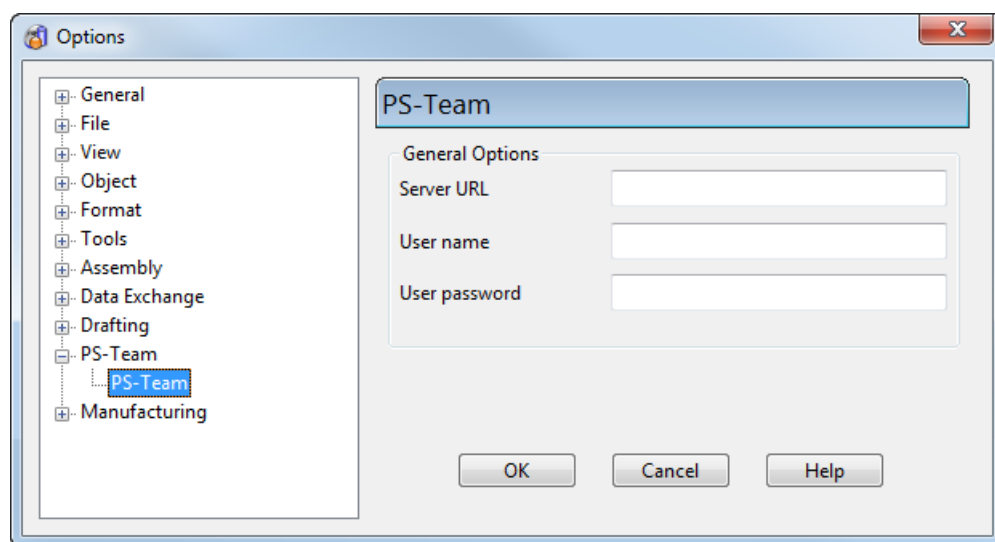
Set the following **General Options** for recording changes to models:

- **Server URL**
- **User name**
- **User password**

Manufacturing options

The following options pages are grouped under the **Manufacturing** category:

- **Electrode** (see page 156)
  - General
  - Base
  - Alignment
- **Toolmaker** (see page 156)
  - General
  - Power Feature Clearances

- **Export** (see page 156)

Electrode options

This is part of the Delcam Electrode functionality. For further details, see Changing the Delcam Electrode options.

Toolmaker options

Use these **Options** pages to select default settings to be used in Toolmaker. There are two pages of Toolmaker options:

General options

Custom Power Feature Clearances

Export options

Use this dialog to specify options for exporting solid hole features to PowerMILL as PowerMILL hole features.

- If PowerMILL and PowerSHAPE are installed on the same computer, you can export the data directly to PowerMILL by selecting **PowerMILL** from the **Module** menu.

- If PowerMILL and PowerSHAPE are not on the same computer, you can create a macro and an xml file to export hole features as PowerMILL hole features using the **Create Hole Features for all DGK** option (see below). You can then run the file or macro in PowerMILL.
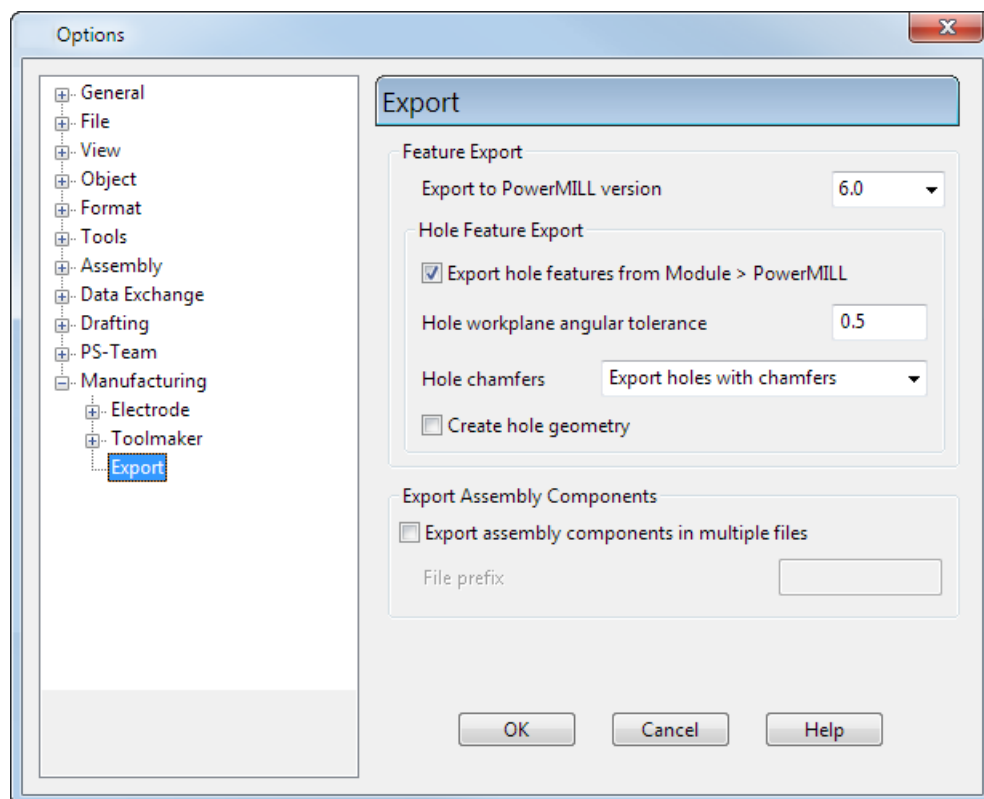
Hole Feature Export

Export hole features from Module > PowerMILL — If selected, this option exports solid hole features as PowerMILL's hole features. If deselected, the features are exported as surface geometry.

Create Hole Geometry — Export the hole as geometry as well as a feature.

Hole workplane angular yolerance — This angular tolerance determines whether two holes lie in the same plane. If the holes lie in the same plane within the angular tolerance, one workplane is exported to define their common plane. If they don't, a workplane for each hole is exported.

The macro and xml file contain the hole features as PowerMILL's hole features. If **Generate Hole Toolpaths** is *ON*, the macro and file also contains the toolpaths for the hole features.

The name of the macro and the xml file is the same name as the dgk file. The macro will have a .mac extension and the xml file will have a .xml extension. Both are created in the same directory as the dgk file.

Export to PowerMILL version — Select the version of PowerMILL you want the macro and xml file to be compatible with:

- **6.0** — This is the default setting and only exports holes.
- **2013** — This export all feature types using multiple feature sets.

  It is recommended that you use the xml file wherever possible.

Hole chamfers — Select an option from the drop-down list to specify the export of holes chamfer sections to PowerMILL.

Export Assembly Components

Export assembly components in multiple files — If selected, a single file is exported containing all the exported entities plus individual files for exported components. If deselected, only the single file is exported. This option is only valid when exporting using Delcam Exchange.

File prefix — Use this option with the multiple file export option to add a prefix to the file name being exported.

Data Exchange options

These options on the **Options** dialog, allow you to set how data is imported to or exported from PowerSHAPE.

- Version 8 / Surfaces options (see page 139)

- Delcam Exchange options (see page 143)
- DDX options
- IGES options (see page 144)
- U3D / PDF options (see page 145)
- HPGL options (see page 146)
- Parasolid options (see page 148)
- PIC options (see page 149)
- STL/DMT options (see page 149)

Exporting

- Exporting data from a model
- Export dialog
- Table of file types that can be exported

Importing

- Importing data into a model
- Table of file types that can be imported

Macro menu

The **Macro** menu allows you to create and run macros.

Macro	
Run...	Ctrl+M
Step...	
Step All...	
Continue	
Until	
Abandon	
Record...	
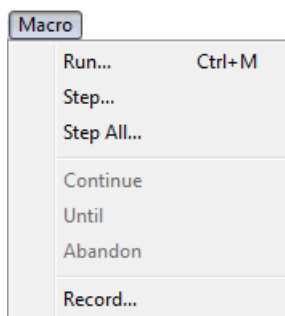
Macro

Macros allow you to create automatic routines for repetitive operations. For example, you might want to loop through all the surfaces in a model and change the colours.

Macros are usually created by recording a sequence of operations. The resulting file can then be edited using a text editor. The macro language is covered in Customising PowerSHAPE.

Use the Macro options (see page 137) page of the **Options** dialog to set the mouse options to be used when recording macros.

For further details see:

Macro > Run (see page 159)

Macro > Step (see page 159)

Macro > Step All (see page 160)

Macro > Continue (see page 161)

Macro > Until (see page 161)

Macro > Abandon (see page 162)

Macro > Record (see page 162)

Macro > Run

Select this option to run a previously recorded macro.

1 Select **Macro > Run** to display the **Select A Macro To Run** dialog.
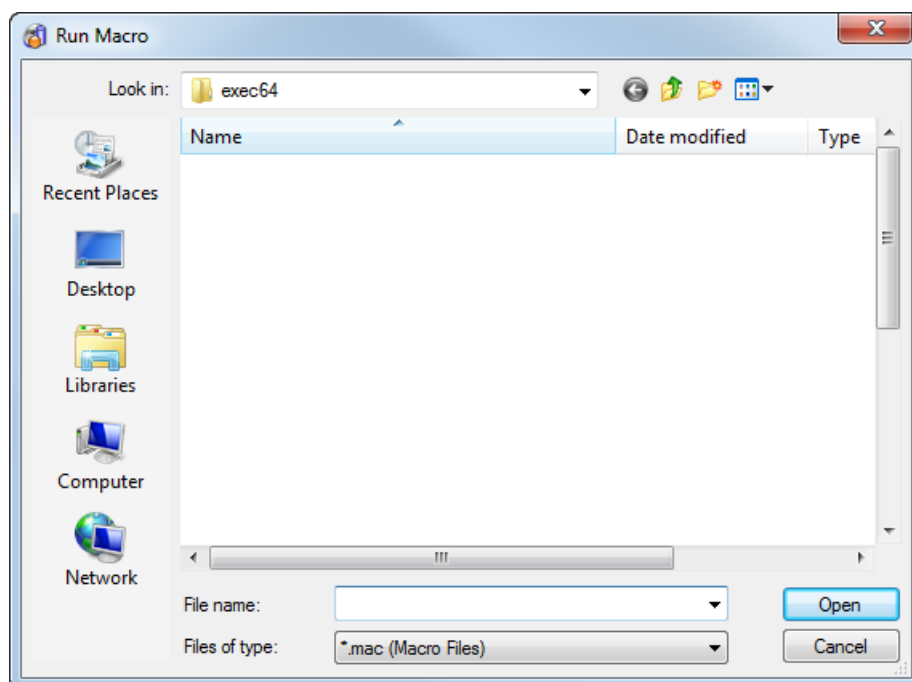
2 Select the macro you want to run. Its name appears in the **File name** text box.

3 Click on **Open** to run the macro.

For further details, see **Macros** in the Customising PowerSHAPE manual.

Macro > Step

Select this option to run a macro one step at a time. Any macros which are nested will be run.

Step runs the macro one step at a time, however if there is a macro nested within the macro you are stepping, the nested macro is run. You are then returned back to the macro you are stepping through. A nested macro is one which is initialised within a macro that is already running.

1 From **Macro** menu, select **Step** to display the **Step Through Macro** dialog.
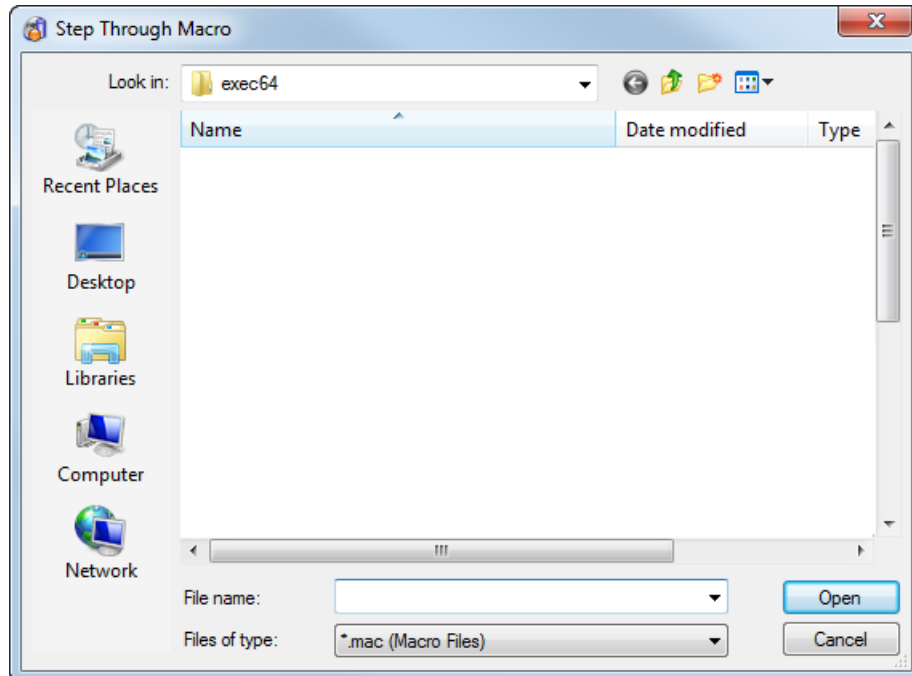
2 Select the macro to step through.

3 Click **Open** to begin stepping through the macro.

> *Macros which are nested within the macro you are stepping are run rather than stepped. If you want to step nested macros, see* **Macro > Step All**

4 Press **Enter** each time you want to carry out another step.

5 To stop stepping through the macro, select **Macro/Abandon**.

For further details, see Running a macro one command at a time in the Customising PowerSHAPE manual.

Macro > Step All

Select this option to run the selected macro and any nested macros to be run one line at a time.

1 From the **Macro** menu, select **Step All** to display the **Step Through Macro** dialog.
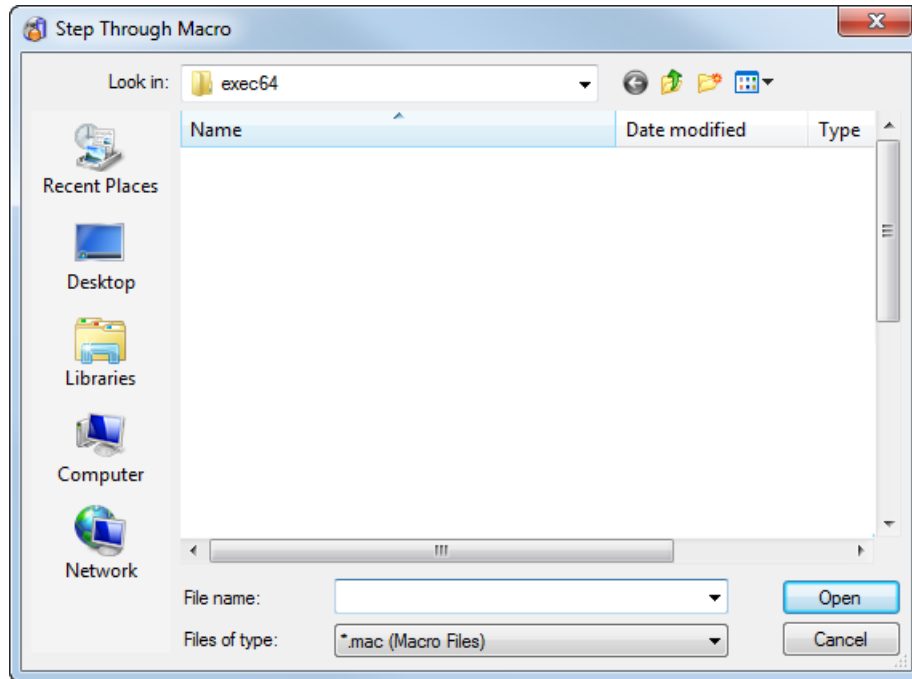
2 Select the macro to step through.

3 Click **Open** to begin stepping through the macro.

> *Macros that are nested within the macro you are stepping through, are also stepped through.*

4 Press **Enter** each time you want to carry out another step.

5 To stop stepping through any of the macros, select **Macro/Abandon**.

For further details, see Running a macro one command at a time in the Customising PowerSHAPE manual.

Macro > Continue

Use this option to run the remainder of a stepping macro.

1 Select **Macro > Continue** to run the macro. This will run the remainder of the macro rather than step through it.

Macro > Until

The **Until** command is only available when the macro is stepping.

When issued, the command prompts for a regular expression. It will then run the macro up to and including the first line that contains the regular expression. It will then revert to stepping.

The regular expressions used are based on Perl and are caseless, so if the macro is

if ($error == 1) {
print e1
} elseif ($error == 2) {
print e2
}

the following expressions would all match the last but one line.

print e2

p.*2

PRINT E2

The advantage of this command is that it makes it easier to step through long macros or macros that include modal dialogs. Other techniques to do this require editing the macro.

Macro > Abandon

Use this option to stop a running or stepping macro.

1 From the **Macro** menu, select **Abandon** to stop the macro.

Abandon will only be active if you are currently running or stepping a macro.

Macro > Record

Select this option to record a macro.

1 Select **Macro > Record** to display the Record Macro dialog.

2 In the **File name** text box, enter the name of the file you want to record to. If you enter the name of an existing file, it is overwritten with the new commands.

3 Click **Save** to begin recording the macro.

4 Now work through the set of commands you want to record.

5 To stop recording the macro, select **Macro > Record**. You can use any text editor to view and edit a macro.

For further details on the macro language, see the Customising PowerSHAPE manual.

Module menu

The **Module** menu contains a list of modules you can run whilst using the program.
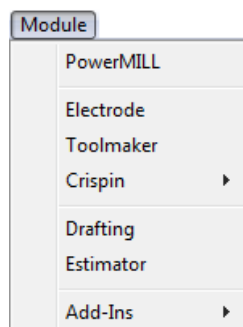
Module > PowerMILL

This starts up PowerMILL.

Module > Electrode

This starts the Electrode wizard.

For further details, see Starting Delcam Electrode from the Module menu.

Module > Toolmaker

This starts up Delcam Toolmaker.

Module > Crispin

Use this option to select one of the following Delcam Crispin shoe products.

SoleEngineer
LastMaker
Engineer
ShoeDesign
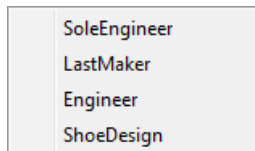
SoleEngineer

Select this option to:

- Model shoe soles and grade shoes and heels using Sole Engineering.

- Create a shoe last by selecting a series of points and surfaces from a model of a shoe using Last Engineering.

- Edit an existing last using the Last Editor.

Additional options

The following are available as a separate cost option:

LastMaker

Select this option to:

- Define lasts, by importing scans of physical lasts or importing digital lasts from other products

- Edit lasts to generate new last designs. Editing options include:

  - Parametric changes (for example stick length and girth).

  - Local and global shaping.

  - Replacement of one toe shape by another.

Engineer

Select this option to create and edit 2D patterns for every shoe piece.

- Changes to the patterns will update all associated lines automatically (for example mirrors and margins).
- Template 'edge types' (including margins, notches and marker lines) can be applied in a single automated operation.
- Once the patterns have been developed for the main size, the graded patterns for all other sizes in a range can be generated.

ShoeDesign

Select this option to design a shoe in three dimensions from a 3D digital last. Curves are sketched onto the last and thickened leather/fabric panels are generated, with realistic textures. The result is a photorealistic 3D image of the shoe. The image can be used for:

- Design review and approval.
- Automatically generating flattened 2D patterns that are ready for the pattern engineering process.

Module > Drafting

This starts up or exits Delcam Draft. It is the same as selecting

Drafting mode from the mode flyout on the **Main** or **Drafting** toolbars.

When Delcam Draft starts, the main toolbar changes from **Modelling** tools to **Drafting** tools.

For further details, see Entering Drafting mode.

If you do not have a licence to run Delcam Draft, you can run an evaluation copy:

1 From the **Tools** menu, select **Evaluation Mode**.

2 From the **Module** menu, select **Drafting**.

Module > Estimator

This starts up or exits Delcam Estimator. When Delcam Estimator starts, the Estimator toolbars are displayed.

If Delcam Draft is not installed, then this is the same as

*selecting **Drafting mode*** *on the **Main** or **Estimator** toolbars.*

Module > Add-ins

This allows you to add macros, commands and applications to the Add-in menu.

For further details, see Introduction to creating OLE applications in the Customising PowerSHAPE for your application manual

To add a macro or a command to the Add-in menu:

1 Select **Manager** from the **Add-in** options of the **Module** menu.
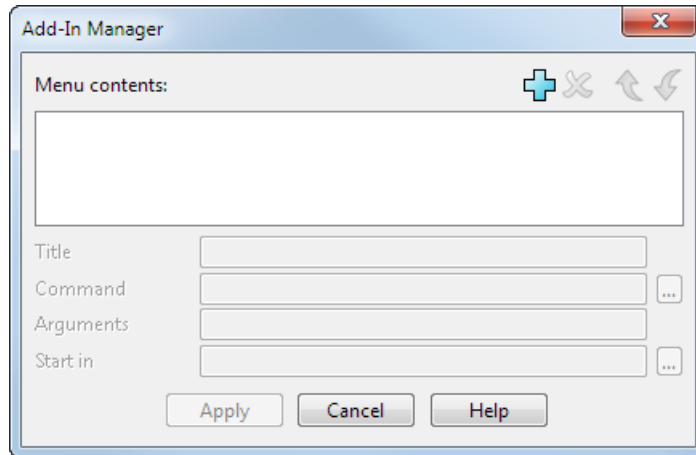
2 Click on the dialog to add a new item.

3 Make sure the new item is selected.

4 Give the item a suitable name in the **Title** box. This name will be used in the **Add-in** menu.

5 If you are adding a macro, put its path in the **Command** box.

6 If you want to add program commands, leave the **Command** box empty and put the commands in the **Arguments** box.

7 Click **Apply** to add the item to the **Add-in** menu.

Window menu

Use the **Window** menu to help you manage the windows. It controls how they are displayed and which is the active window.
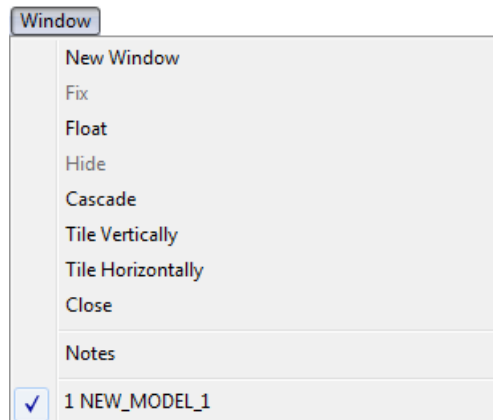
Up to 20 different models can be open at the same time and an unlimited number of graphics windows for each model. The only constraints are the power and capacity of your computer and your own ability to manage a large number of open windows. For further details, see Using the graphics area in the Modelling Concepts manual.

Window > New Window

Use this option to open a new window for the active model. It becomes the active window and initially shows the default top view of the whole model. You can change the views in the usual ways.

Window > Fix

Fix expands the active window to fill the working area. If set to a large size, this can give a larger working area without manually re-sizing the active window.

*If you set the program window to a small size, **Fix** can reduce the available working space.*

For further details, see Single window mode.

Window > Float

Float displays the active view in a window, at the default size. This window can be re-positioned and resized as required. All the currently open model windows are given their own working window. If **Float** is selected, the windows overlap in a cascade style.

For further details, see Multi-window mode.

Window > Hide

Hide reduces the need to open and close graphics windows by hiding the active window, without actually closing it. You can display the hidden window by selecting it from the list of windows on the **Window** menu.

*If only one window is open, the **Hide** option is unavailable because you must have at least one window visible at any time.*

Window > Cascade

This layers all open windows so that the title bar of each window is visible.
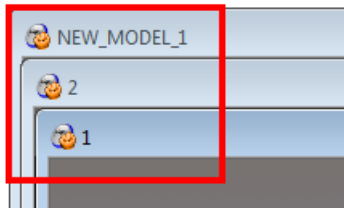
Click on the **Maximise** button on a *Cascaded* window to revert to a single window display.
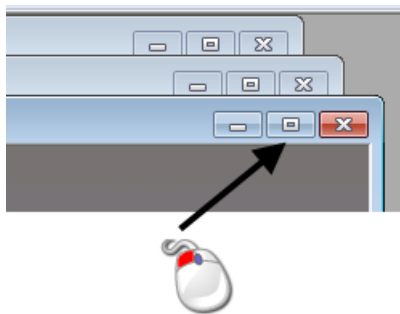
Window > Tile Vertically

This displays open model windows vertically so they are all visible within the graphics area. The sizes of each window depends on the number of windows open.
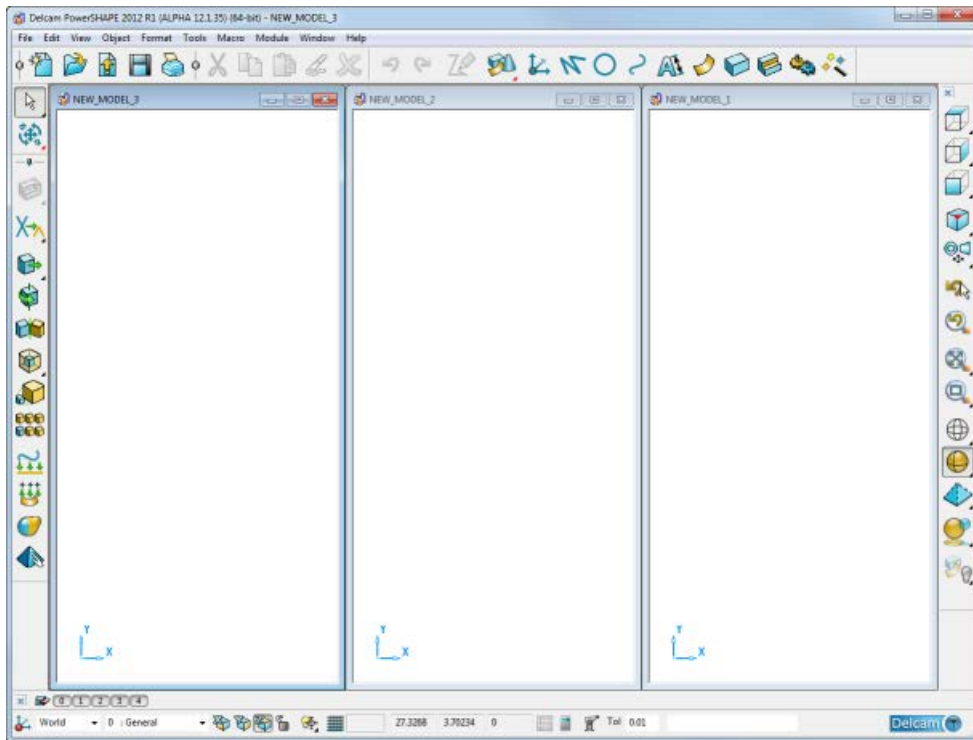
You can resize and move tiled modelling windows.

Click the **Maximise** ▭ button on a tiled window to revert to a single window display.

Window > Tile Horizontally

This displays open modelling windows horizontally so they are all visible within the graphics area. Their individual sizes depend on the number of windows open.
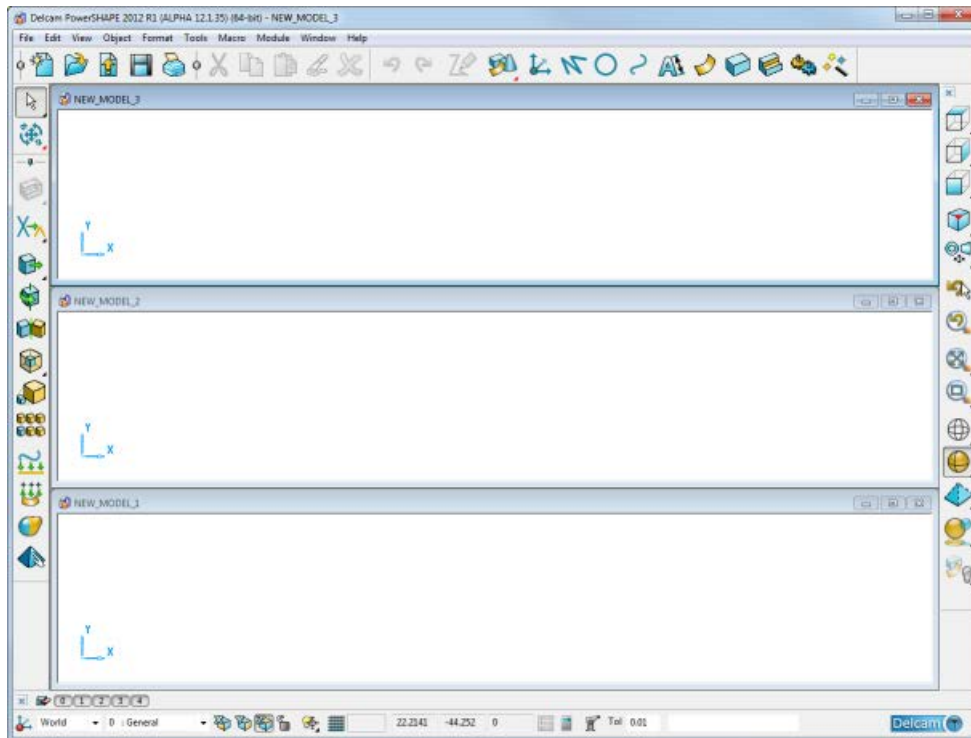
You can resize and move tiled modelling windows.

Click the **Maximise** button on a tiled window to revert to a single window display.

Window > Close

This closes the active window. If it is the last open window for the model, the model will also be closed. This is the same as selecting **File > Close**.

- If there are any unsaved changes, a warning dialog is displayed.
- If you have not made any changes, the model is closed immediately.

Window > Notes

This creates a drawing with predefined views and a title block.

For further details, see What is a Notes drawing?

Window > Individual graphics windows

All open windows are listed at the bottom of the **Window** menu. The tick next to the name of a window indicates the current active window.

- To select another window, select the **Window** menu and click the name of the window you want to make active.

- Use **ALT W +** *number* as a shortcut to access a particular modelling window. For example:

  ALT W 1 will display window **1**

  ALT W 2 will display window **2**

  ALT W 0 will display window **10**

Help menu

The **Help** menu displays the tools related to the on-line help system.
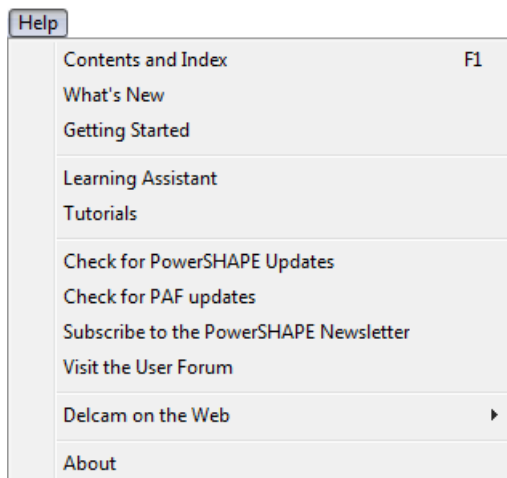
Contents and Index (see page 172)

What's New (see page 172)

Getting Started (see page 172)

Learning Assistant (see page 172)

Tutorials (see page 174)

PowerSHAPE updates (see page 174)

PAF updates (see page 174)

PowerSHAPE newsletter (see page 174)

User Forum (see page 174)

Delcam on the Web (see page 175)

About (see page 176)

Help > Contents and Index

Select this option to display the contents and index of the help file. The right hand frame displays the welcome page and provides links to further information about the program and the on-line tutorials.

Help > What's New

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

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

- If full documentation is installed, the What's New will appear in a new, floating window.

Help > Getting Started

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

Help > Learning Assistant

This displays and removes the **Learning Assistant** window.

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

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

To create a single line using the Learning Assistant:

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

2   Click the **Lines** button. This will display a new list of options.
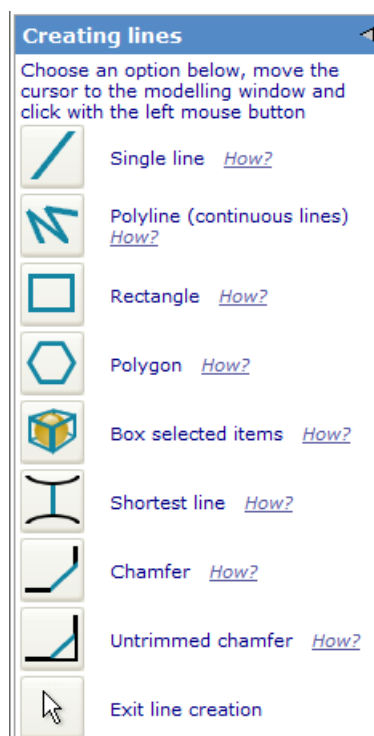
3 Click **How?** to see details about creating a line.

4 Click the **Single line** button and move the cursor into the graphics window to create the line.

Help > Tutorials

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

Help > Check for PowerSHAPE updates

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

Help > Check for PAF updates

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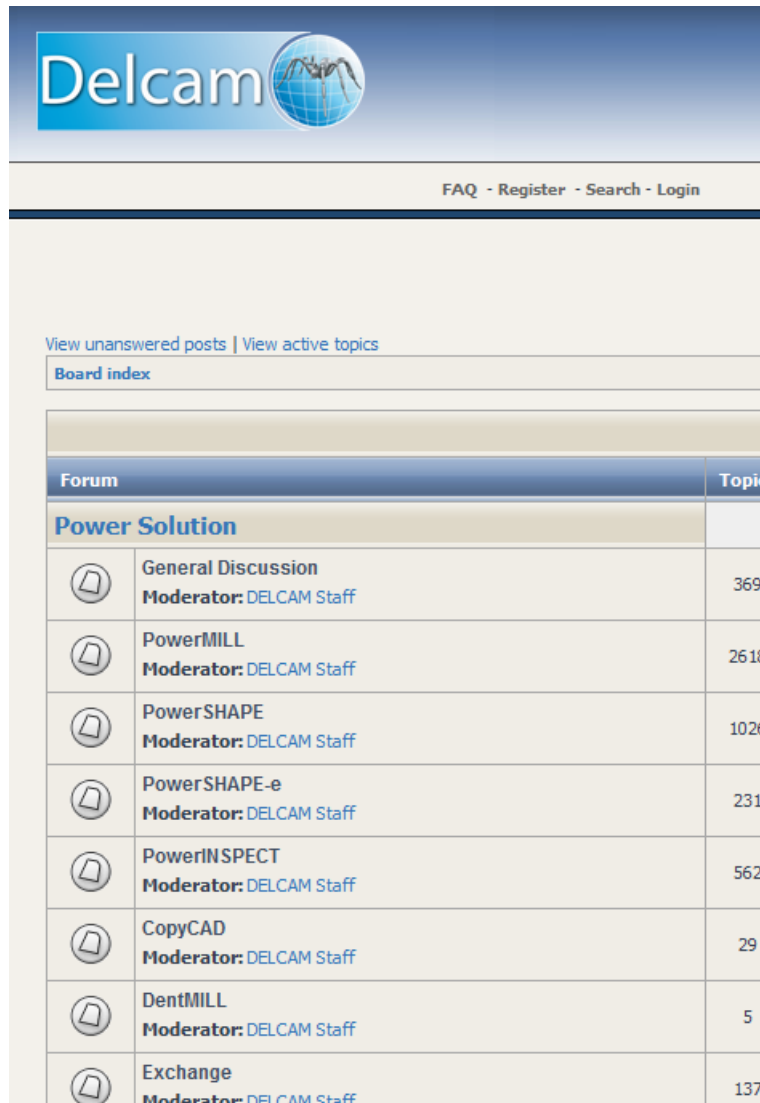
Help > Subscribe to the PowerSHAPE newsletter

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

Help > Visit the User Forum

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

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

Help > Delcam on the Web

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

PowerSHAPE Home Page — Select this option to display the product home page www.powershape.com/powershape/powershape.htm (http://www.powershape.com/powershape/powershape.htm). This includes news and information about the software.

Delcam Home Page — Select this option to display the Delcam home page www.delcam.com (http://www.delcam.com). This includes and information on all Delcam products.

Help > About

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

The dialog also provides:

- licence and user information.

- the language set on your PC and that used by PowerSHAPE.

- patent information.

Click **OK** to close the dialog.

Shortcuts

Use the following sections to find information on shortcuts:

Shortcuts to menu options (see page 176)

Shortcuts to frequent commands (see page 177)

Customising menu shortcuts (see page 180)

Mouse shortcuts (see page 181)

Shortcuts to menu options

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

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

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

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

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

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

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

4 Press the character underlined in your required option.

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

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

- Once the **Alt** key has activated the menus, you can navigate using the arrow keys ⇦⇧⇩⇨.

- To accept a menu, press **Enter**.

- To do nothing and close a menu, press **Esc.**

Shortcuts to frequent commands

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

Menu option	Shortcut*
File > New	Ctrl + n
File > Open	Ctrl + o
File > Close	Ctrl + F4
File > Save	Ctrl + s
File > Print	Ctrl + p
File > Exit	Alt + F4
Edit > Undo	Ctrl + z
Edit > Cut	Ctrl + x
Edit > Copy	Ctrl + c
Edit > Paste	Ctrl + v
Edit > Paste Special	Ctrl + e
Edit > Select > Clear selection	Ctrl + d
Edit > Select > Select All	Ctrl + a
View > Single (of model) > Top	Ctrl + keypad 5
View > Single (of model) > Bottom	Ctrl + keypad 0
View > Single (of model) > Left	Ctrl + keypad 4

Menu option	Shortcut*
View > Single (of model) > Right	Ctrl + keypad 6
View > Single (of model) > Front	Ctrl + keypad 2
View > Single (of model) > Back	Ctrl + keypad 8
View > Single (of model) > Iso 1	Ctrl + keypad 1
View > Single (of model) > Iso 2	Ctrl + keypad 3
View > Single (of model) > Iso 3	Ctrl + keypad 9
View > Single (of model) > Iso 4	Ctrl + keypad 7
View > Single (about selection) > Top	Alt + keypad 5
View > Single (about selection) > Bottom	Alt + keypad 0
View > Single (about selection) > Left	Alt + keypad 4
View > Single (about selection) > Right	Alt + keypad 6
View > Single (about selection) > Front	Alt + keypad 2
View > Single (about selection) > Back	Alt + keypad 8
View > Single (about selection) > Iso 1	Alt + keypad 1
View > Single (about selection) > Iso 2	Alt + keypad 3
View > Single (about selection) > Iso 3	Alt + keypad 9
View > Single (about selection) > Iso 4	Alt + keypad 7
View > Shading > Wireframe	F2
View > Shading > Shaded	F3

Menu option	Shortcut*
View > Shading > Transparent Shaded	F4
View > Shading > Transparent Shaded Wire	F10
View > Shading > Shaded Wire	F11
View > Shading > Dynamic Hidden Line	F12
View > Look From > Last View	F5
View > Zoom Full	F6
Tools > Model Analysis > Smoothness Shading	F7
Tools > Model Analysis > Undercut Shading	F8
Tools > Model Analysis > Minimum Radius Shading	F9
View > Blank > Blank Selected	Ctrl + j
View > Blank > Blank Except	Ctrl + k
View > Blank > Blank Toggle	Ctrl + y
View > Unblank	Ctrl + l
View > Refresh	Ctrl + r
Object > Surface > Smart Surfacer	Ctrl + t
Object > Surface > Fillet	Ctrl + f
Object > Curve > Composite Curve	Ctrl + h
Macro > Run	Ctrl + m
Swap individual graphics windows in the Window menu	Ctrl + F6
Help > Contents	F1

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

Other shortcuts

Operation	Shortcut*
Bold text	Ctrl + b
Italic text	Ctrl + i
Underline text	Ctrl + u
Diameter character in text	Ctrl + Alt + d
Degrees character in text	Ctrl + Alt + s
Plus/minus character in text	Ctrl + Alt + p

Operation	Shortcut*
Interrupt current operation	Esc
Exiting a creation mode	Esc
Halt program (after accepting confirmation dialog)	Esc - 3 times within 1 second

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

Customising menu shortcuts

You can create your own shortcuts for menu options.

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

2 Use the **Menu Shortcuts** tab of the **Customise** dialog to set up your own shortcuts.
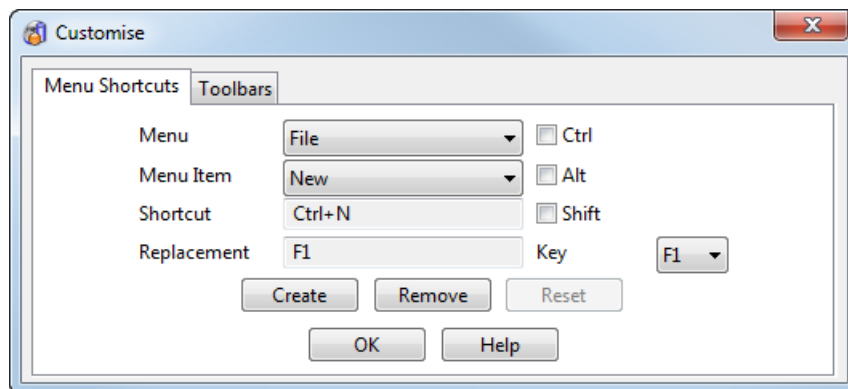
Menu — Select a menu from the list of the menus that appear at the top of the screen. .

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

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

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

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

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

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

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

Reset — Reset all the shortcuts back to the original shortcuts.

OK — Close the dialog.

Mouse shortcuts

The toolbars contain buttons, labelled with icons to indicate the tool function. In most cases, these are a quick way of implementing a menu option. Simply place the cursor over a button and click the left mouse button.

Greyed-out buttons cannot be used with the current selection. The equivalent menu options will also be unavailable.

The mouse is often used with keyboard "modifier" keys to increase the range of options. These controls are central to using PowerSHAPE and therefore are not considered as shortcuts.

For further details, see Using the mouse.

Toolbars and flyouts

For further general information on toolbars and flyouts, select one of the following:

- Toolbars overview (see page 182)
- Flyout overview (see page 192)

Toolbars overview

Toolbars are areas of the screen that contain buttons. Click on buttons using the mouse to activate the functionality. In most cases, toolbar buttons provide shortcuts to options on the menus.

The **Main** toolbar (see page 196) is always visible and is displayed across the top.
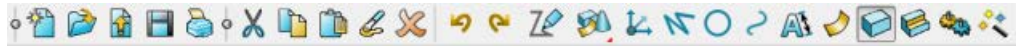
The **General Edits** toolbar is displayed down the side by default.
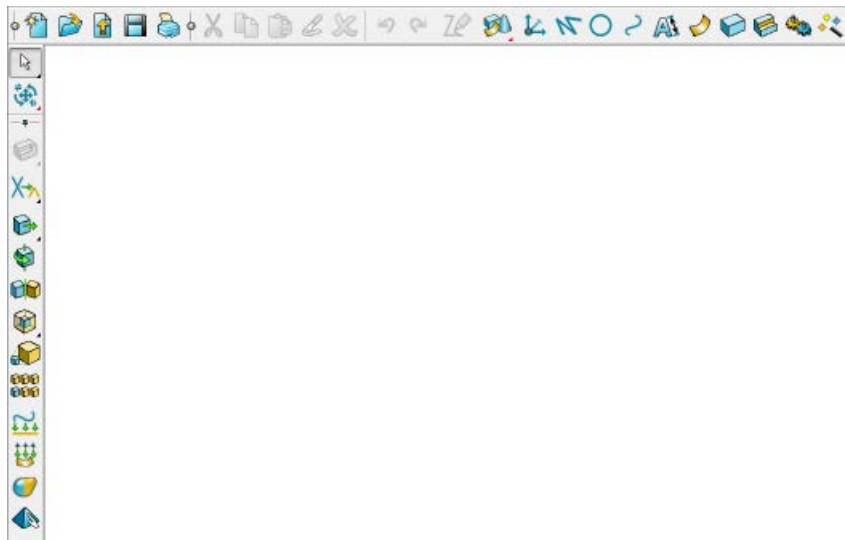
This is also where the following toolbars are displayed when you select a corresponding object, menu item, or button.

- **General Edits** (see page 204)
- **Model Analysis** (see page 234)
- **Model Fixing** (see page 233)
- **Object Creation**

You can:

- Display or hide some toolbars according to the functions in use. For example the **Surface Edit** toolbar (see page 202) and the **Curve Edit** toolbar (see page 202), shown in the image below, are displayed automatically when corresponding objects are selected.

- Expand some toolbars using the flyout button to display similar items or functions grouped together.

- Pin (see page 183) some toolbars anywhere on the screen, or dock them onto another toolbar using their *pins* to keep them visible when you display another toolbar.

- Create your own toolbars (see page 186) to contain your favourite buttons, menu items, or functions.

- Display a toolbar as follows:

  1 Right-click on an empty space on any toolbar to display a menu of available toolbars.

  2 Select the toolbar that you wish to display from the menu.

- Display large or small buttons on the toolbars using the **Toolbars** options (see page 86) page of the **Options** dialog.

- Reverse the display of the (view and editing) toolbars using the **Toolbars** options (see page 86) page of the **Options** dialog.

- Remove some toolbars from the screen by clicking the Close ✖ button on the toolbar.

Pinned toolbars

A pinned toolbar is always visible. You can move and reposition it anywhere on the screen, or dock it with another toolbar.

Toolbars that are in their default location have the unpinned ⚊📌⚊ icon.

You can pin the following toolbars:

- **General Edit**
- **Model Analysis**
- **Model Fixing**

Using pinned toolbars

If you have not used pinned toolbars before, pinned toolbars remain in their default location in the software.

If you have already pinned a toolbar, it displays at its last pinned position.

You can:

- Reposition a pinned toolbar in the graphics area (see page 184).
- Dock a pinned toolbar with another toolbar (see page 185).

Repositioning a pinned toolbar

To reposition a pinned toolbar in the graphics area:

1 Click the pin ⊡ᵖ⊐ on the toolbar you wish to move, for example the **General Edits** toolbar. The whole pin is displayed only when the toolbar is in its default position.

The pin icon changes to pinned ⬜•⬜, the toolbar undocks and displays in the graphics area ready to be repositioned.

2   Click on the title bar of the toolbar and drag it to the required position in the graphics area.

3   Click the pin again to return the toolbar automatically to its default location. The pin changes back to unpinned ⬜📌⬜.

The next time you click the pin on the toolbar, the pinned toolbar is displayed automatically in the last pinned position.

Docking a pinned toolbar

To dock a pinned toolbar with another toolbar:

1   Click the pin ⬜📌⬜ button on the toolbar you wish to move, for example the **General Edits** toolbar. The whole pin is displayed only when the toolbar is in its default position.

The pin icon changes to pinned ⬜•⬜, the toolbar undocks and displays in the graphics area ready to be repositioned.

2   Click on the title bar of the toolbar and drag it over the edge of another toolbar. It snaps automatically into the toolbar area. You can also dock a toolbar at the edge of the screen.

3 Click the pin again to return the toolbar automatically to its default location. The pin changes back to the unpinned ▬🖈▬ icon.

The next time you click the pin on the toolbar, the pinned toolbar is displayed automatically in the last pinned position.

User-defined toolbars

A user-defined toolbar is a toolbar that you create to contain your favourite buttons, menu items, or functions.

▪ Your customised toolbar can contain both menu and toolbar items as well as command lines.

▪ You can send your user-defined toolbar to other users.

▪ You can group or separate options on your toolbar using the **Customise** dialog (see page 189).

▪ You can add your own icons and you can customise buttons to be added to your toolbar using the Customise Picture dialog.

Using your personal toolbars

Use the following sections to create and use your own toolbars:

Create a user-defined toolbar (see page 186)

Edit a user-defined toolbar (see page 188)

Display an existing user-defined toolbar (see page 188)

Customise buttons for user-defined toolbars

Sharing a user-defined toolbar and custom buttons (see page 191)

Creating a toolbar

To create a toolbar:

1 Right-click on an empty section of any toolbar and select **Customise toolbar** from the menu.
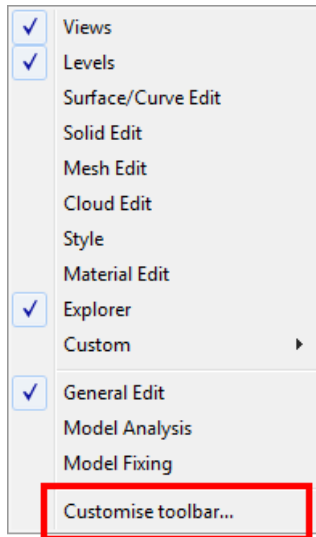
The **Customise** dialog (see page 189) is displayed.

2 On the **Toolbars** tab define your toolbar and select the items you want to add.

3 Click **Add** to add each new item to the toolbar.

4 Use the buttons on the dialog to order and organise the toolbar items.

The example below describes adding a block primitive from the surface creation toolbar to a new custom toolbar.

① Select a toolbar and enter a new name, such as **Surface items**.

② Select the **Button Size** size from:

- **Large**
- **Small**
- **As Options Page**

③ For the **Type**, select **Toolbar.**

④ Select the **Menu/Toolbar** that contains the item you want to add to your toolbar, such as **Surface/Solid Creation**.

⑤ Select the **Item** to add, such as the **Block Primitive** button.

⑥ Click **Add** to add the block to your toolbar.

⑦ Added items, for example the **Block Primitive** button, are displayed in the added items list.

⑧ Click **OK**.
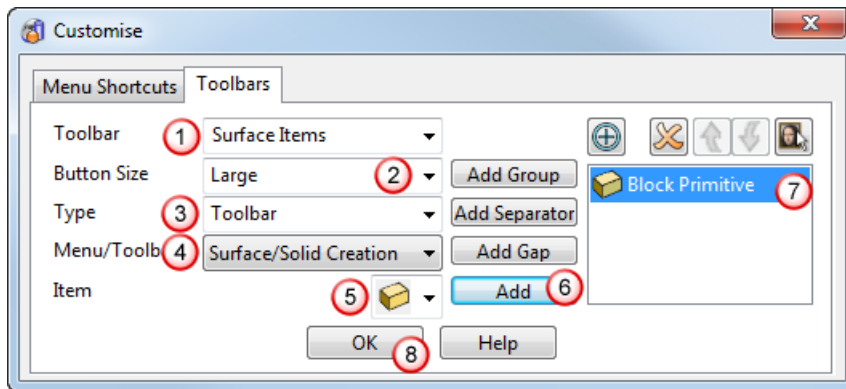
Edit a user-defined toolbar

1 Select **Customise toolbar** from the context menu, displayed when you right-click on an empty section of any toolbar.
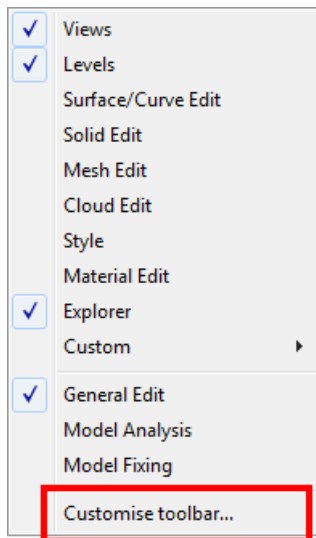
2 Select the **Toolbars** tab on the Customise dialog (see page 189).

3 Select the toolbar you want to edit from the **Toolbar** drop-down list.

4 Make your changes.

5 Click **OK**.

Displaying an existing user-defined toolbar

1 Select **View > Toolbars > Custom.**

2 Select the toolbar you want to display.

Customise dialog - Toolbars

Use this dialog to define up to four customised toolbars.
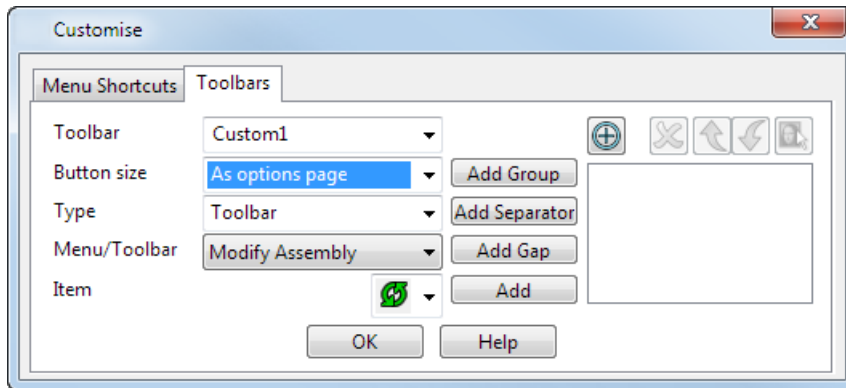
Toolbar — Enter a new name for a new toolbar or select an existing user-defined toolbar from the list. You can define up to four toolbars.

Type — Select from the following:

- **Menu** — Select this option to add a menu item to your toolbar and select the menu from the **Menu/Toolbar** list.

- **Toolbar** — Select this option to add a toolbar item to your toolbar and select the toolbar from the **Menu/Toolbar** list.

- **Command** — Select this option to enter your own **Command** and **Description** to be added to your toolbar.

Menu/Toolbar — Select the menu or toolbar that contains the option you want to add to your new toolbar.

Item — Select the menu option or toolbar button that you want to add to your new toolbar.

Add Group — Click this button to add a group marker. The group marker —•— icon is displayed in the list of added items at the right of the dialog. A group marker shows the beginning of a group of buttons, menu items, or commands. On the toolbar, the group marker is a toggle button:

Hide Group — Click this button to contract the items in the group.

Show Group — Click this button to expand the items in the group.

Add Separator — Click this buttom to add a separator marker to divide up your toolbar items. The separator marker —— icon is displayed in the list of added items at the right of the dialog.

Add Gap — Click this button to add a small space between items on the toolbar. A gap is displayed in the list of added items at the right of the dialog.

Add — Click this button to add the selected **Item** to the toolbar. The item is displayed in the list of added items at the right of the dialog.

 Drag Crosshairs — Click and hold the mouse button and drag the crosshairs over a toolbar button. The button is added automatically to custom toolbar and is displayed on the dialog.

 Remove — Click this button to remove the item selected in the added items list from your toolbar.

 Move the item up the menu — Click this button to move the selected item up the order of items on the toolbar.

 Move the item down the menu — Click this option to move it down the order of items on the toolbar.

 Change button icon — Click this button to change the icon for the button selected in the added items list. The **Customise Button** dialog (see page 190) is displayed.

Customise Button dialog

Use this dialog to:

- enter text to display as the item's button.

- select a picture to be displayed as the item's button.

- select a picture to edit or customise using the bitmap editing functionality on the **Customise Picture** dialog

- change the description of a command. This option is displayed only if you have selected **Command** from the **Type** list.
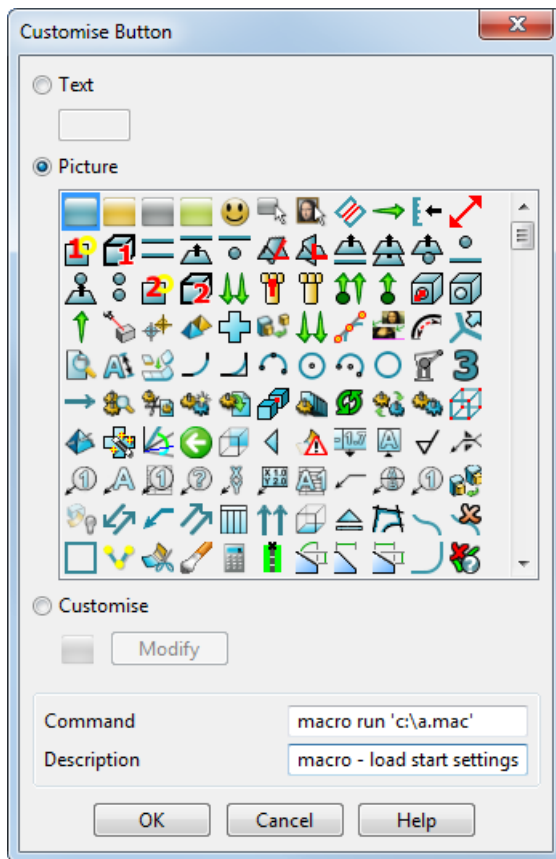
Text — Select this option and enter text to be displayed as a button.

Picture — Select this option and select an icon to be displayed as a button.

Customise — If selected, the **Modify** button is displayed.

Modify — Click this button to display the selected picture in the bitmap editor on the **Customise Picture** dialog to customise the picture before adding it to a user-defined toolbar.

Command — The command is displayed and you can edit it.

Description — The description for the command is displayed and you can edit it.

Sharing a user-defined toolbar and custom buttons

User-defined toolbars have a name in the format CustomToolbarX.

Custom buttons have a name in the format ToolbarX_PictureY.bmp.

To send a user-defined toolbar and its buttons to another user:

1 Find the user-specific files. They are stored in xxxx\PowerSHAPE where xxxx is one of the following:

- C:\Documents and Settings\[username]\Application Data (XP installation)
- C:\Users\[username]\AppData\Roaming\ (Windows 7 and Windows 8 installation)
- the location that is indicated by your **HOME** variable.

> *To find the Local config folder (PowerSHAPE), enter* ***print app.paths*** *in the command window.*

2 Select the required files. They include:
- a file that contains the toolbar, for example CustomToolbar1.
- one or more button files from the toolbar, for example Toolbar1_Picture1.bmp.

3 Email the files to another user, with instructions to copy the files into the folder containing their user-specific files.

Flyout overview

Access a flyout toolbar using a flyout button. Flyout buttons have a small triangle icon in the bottom right corner. There are two types of flyout buttons:

- Red ◢ icon — Flyouts with a red icon contain mode buttons. Right-click on the flyout button to display the flyout toolbar. Select a mode button to display the buttons for the mode on the toolbar.

  For example, if you select the **Model Fixing** mode button from the **General Edits** flyout, the toolbar displays the model-fixing buttons.

  > *You can cycle through the available modes by repeatedly clicking the flyout button.*

- Black ◢ icon — Flyouts with a black icon group similar buttons, for example the **Offset** flyout. Right-click on the flyout button to show the flyout toolbar and click the button you want. The most recently used button from the flyout toolbar is displayed as the flyout button.

Select flyout

1 Click the **Select** flyout ⬚ button to display the selection and selection information options.
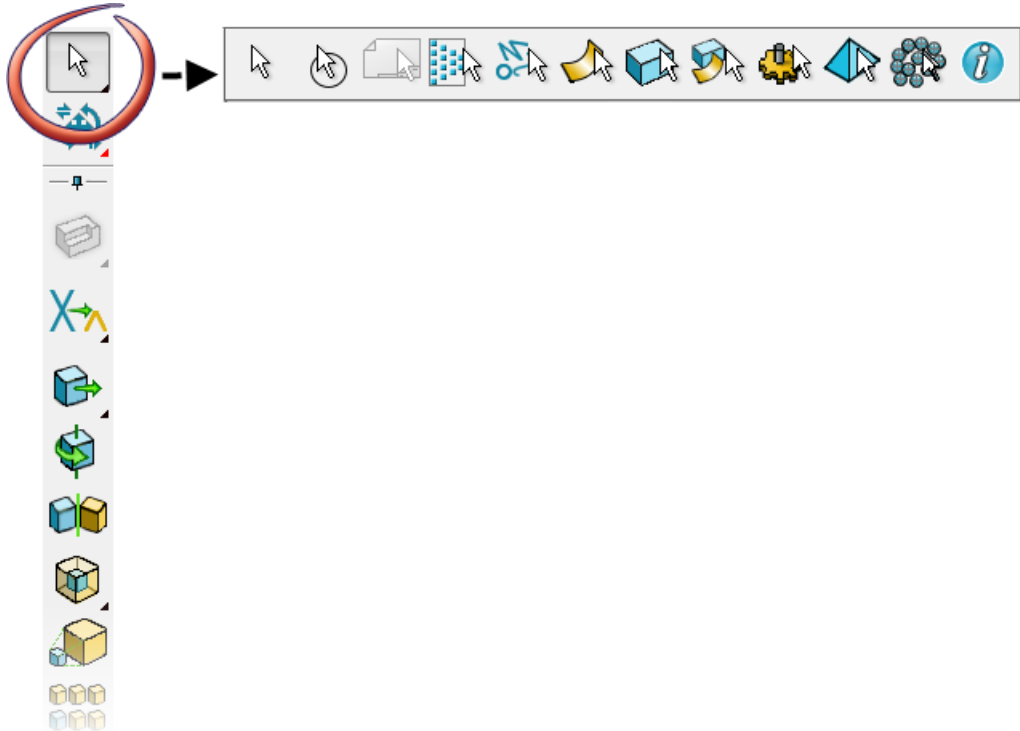
For further details, see Selecting objects.

If you move the cursor over the Select flyout button, selection information is displayed as a Tooltip

General editing flyout

1 Click the **General Edits** ⬚ button to display the **General Editing** flyout

2 Click one of the buttons on the flyout. The appropriate editing toolbar will be loaded.

General edits (see page 204)

Model fixing (see page 232)

Model analysis (see page 234)

Style toolbar

This is displayed when you do one of the following:

- Select **Style** from the **Toolbars** options of the **View** menu.
- Select **Style Toolbar** from the **Item colour** drop-down menu (*Main toolbar*).

You can use the **Style** toolbar to change:

- the style of a selected object.
- the default creation style of an object.
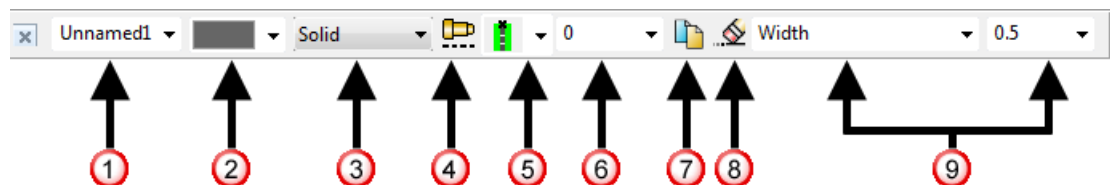
For further details, see View > Toolbars > Style (see page 1).

Annotation toolbar

This combines the following tools and options when used in Drafting mode,

① — **Text**

② — **Drawing**

③ — **Dimensions**

The **Annotation** toolbar is also available when in Modelling mode but some drafting options, such as symbols and hatching, remain unavailable outside Drafting mode.

To display the Annotation toolbar:

1 Select **Annotation** from the main toolbar.

The **Text Editor** toolbar displays automatically.
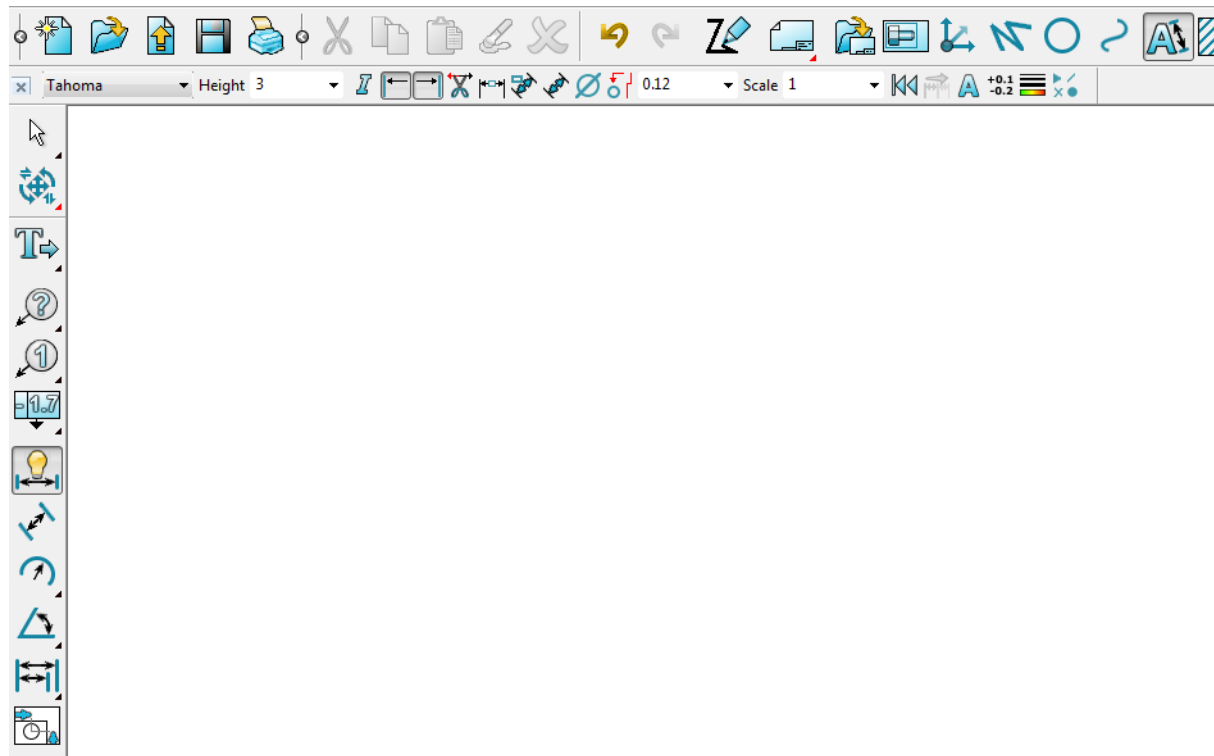
2 Select the option you require from the toolbars. If you select a dimensioning or balloon option from the **Annotation** toolbar, the text options will be supplemented with the appropriate balloon or dimension options.

For further details, see Balloon toolbar and Dimension toolbar.

Main toolbar

This toolbar is always visible and includes the following sections:

- Model and file management.
- Standard editing.
- Modelling options.
- Object creation.

The model management and standard editing sections can be

collapsed and expanded by clicking on the toggle button.

Model and file management

All the following options have corresponding commands on the **File** menu.

New (see page 8) — Click this button to open a new model.

Open (see page 8) — Click this button to open an existing model.

Import (see page 20) — Click this button to import data into the model.

Save (see page 8) — Click this button to save the active model.

Print (see page 10) — Click this button to open the **Print** dialog to print the model.

Standard editing

All the following options have corresponding commands on the **Edit** menu.

Cut (see page 23) — Click this button to cut the selected objects from the model.

Copy (see page 23) — Click this button to copy the selected objects from the model.

Paste (see page 24) — Click this button to paste cut or copied objects into the model.

Paste Attributes (see page 25) — Click this button to paste attributes from one object to another.

Delete (see page 29) — Click this button to delete the selected objects.

Modelling options

Undo — Undo the last command. This can be used with active dimensions to undo the last dimension change. When you have finished using active dimensions, Undo **(see page 22)** will undo all changes, rather than individual changes.

Redo — Reverse the last **Undo** command. Redo (see page 23) can be used with active dimensions to redo the last dimension change.

Item colour — Select the item colour using the colour palette that is displayed. When you hover over a colour selectors, the RGB value for the colour is displayed as a tooltip. The current colour for new objects is shown on the button. You can also display the style (see page 194) toolbar using this option.

Modelling/Drafting Mode — Select an option from this flyout to enter the Drafting (see page 165) mode and display the Drafting options on the toolbar. If Delcam Drafting it is not licensed or you have chosen to start PowerSHAPE in Estimator mode, you enter the **Delcam Estimator** instead.

Object creation

The following object creation options are displayed on the main toolbar. When you select one of these buttons, the corresponding toolbar is displayed.

Workplane

Line

Curve

Arc

Annotation (see page 194)

Surface

Solid

Feature

Assembly modelling (see page 199)

Wizards

Assembly toolbar

Click the **Assembly** button to display the **Assembly** toolbar.
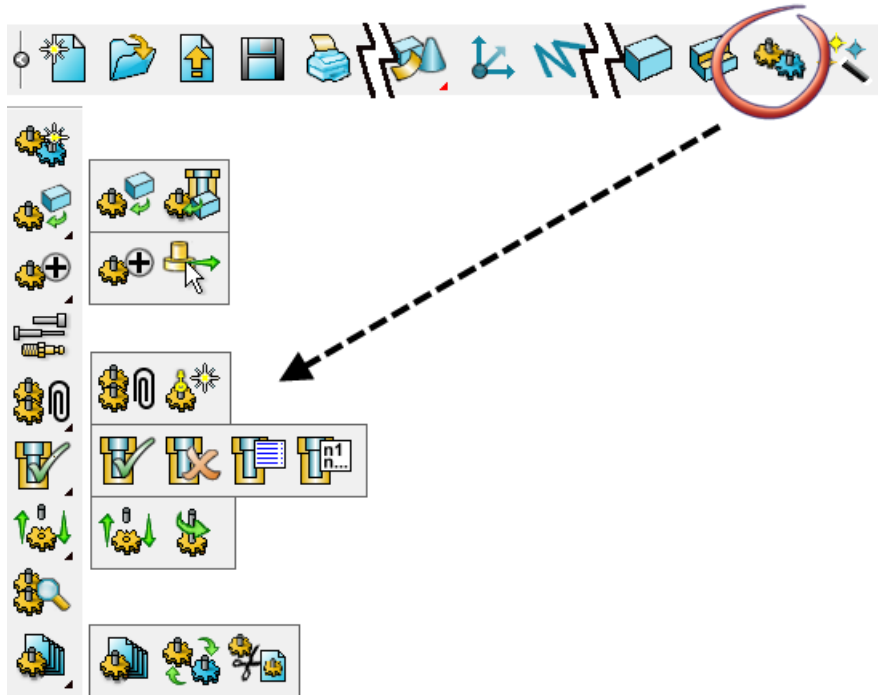
Drafting toolbar

Click the **Modelling Mode** button (*Main toolbar*).

This is the same as hovering over and selecting from the flyout. For further details, see Flyout overview (see page 192).
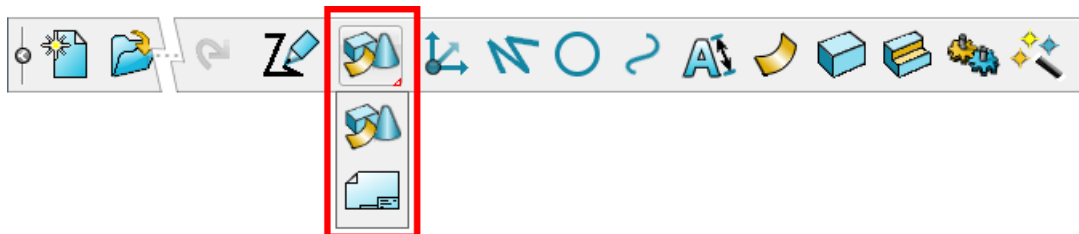
The Drawing Selector dialog and the drafting toolbar are displayed.

For further details, see Entering Drafting mode.

Views toolbar

This provides options to change the view. The toolbar below shows the options that are available from each flyout.

When model includes stencil
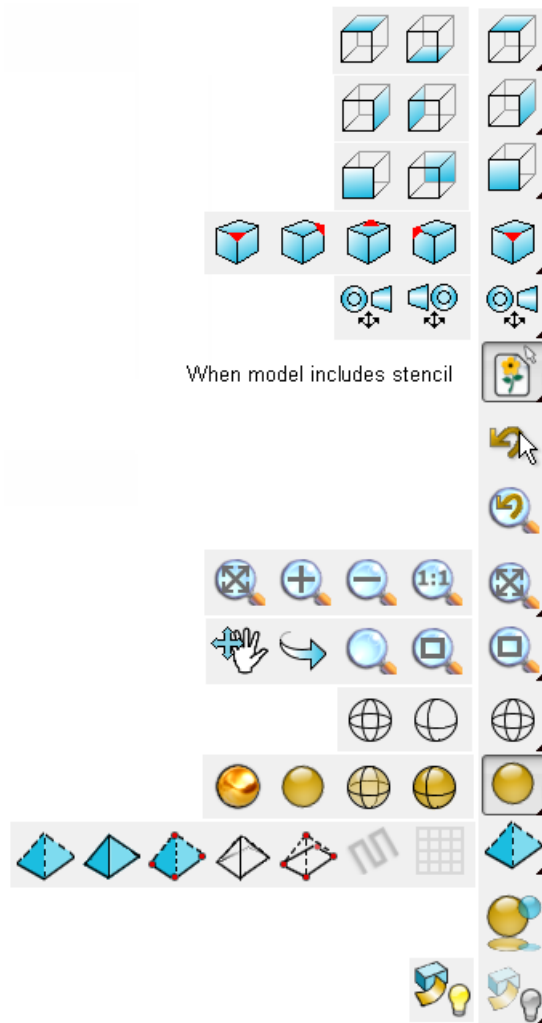
For further details, see View > Toolbars > Views (see page 142)

Cloud editing toolbar

Use the **Cloud editing** toolbar to edit the imported cloud:

Use the following options to edit the point cloud.

Use **Box select** to select cloud points by dragging a box around them.

Use **Discrete lasso** to select points by drawing a discrete lasso around them.

Use **Continuous lasso** to select points by drawing a continuous lasso around them.

Use the **Filter** button to filter the number of points in the selected cloud.

Use **Generate Mesh** to create single mesh from one or more selected clouds, or cloud points.

When generating a mesh from a point cloud, you can cancel the generation by pressing **Esc**.

Use **Generate Curve** to create a curve from a selected cloud.

Use **Extract selection** to extract the currently selected points from the current cloud and create a new cloud from those points.

Use **Combine clouds** to combine the selected clouds. It is the same as selecting **Object > Cloud > Combine** .

Use **Colour clouds uniquely** to apply a different colour to all the selected clouds.

The colour that is applied to a particular cloud is linked to the cloud number; whenever **Colour clouds uniquely** is selected, the clouds will always use the same colour.

Reset to default colour applies the default colour (red) to every selected cloud

Delete the selected points from the cloud.

Use the **Acquire Points** toolbar to acquire point data from laser scanners directly into PowerSHAPE.
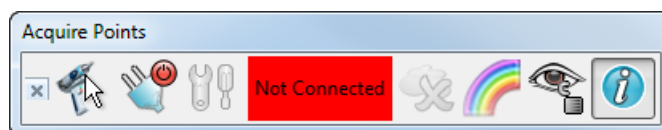
Configure — Click this button to open the **Delcam CMMDriver Configuration** dialog and select a connection protocol.

Connect — Click this button to connect to the device.

Options — Click this button to configure the device settings using manufacturer specific dialogs.

Delete Sweep — Click this button to delete the most recent sweep.

Colour Sweeps — Select this button to assign a distinct colour to each sweep.

Lock View — Select this button to lock the view in the current position.

Information — Select this button to open the **Sweep Information** dialog and display the number of sweeps, points and scanlines.

The status of the device is displayed on the toolbar:

- **Not Connected** (red)
- **Acquiring** (yellow)
- **Connected** (green)

Mesh surfacing toolbar

Use the buttons on the **Mesh surfacing** toolbar to create a surface from selected triangles.

Surface/Curve editing toolbar

This toolbar appears when you do one of the following:

- double-click a Power Surface.
- double-click a curve which is not part of a surface.

The options that are displayed reflect the object you are editing.

The **Surface edit** toolbar looks like this:

The **Curve edit** toolbar looks like this:

Mesh editing toolbar

Modify your meshes using the buttons on the **Mesh editing** toolbar.
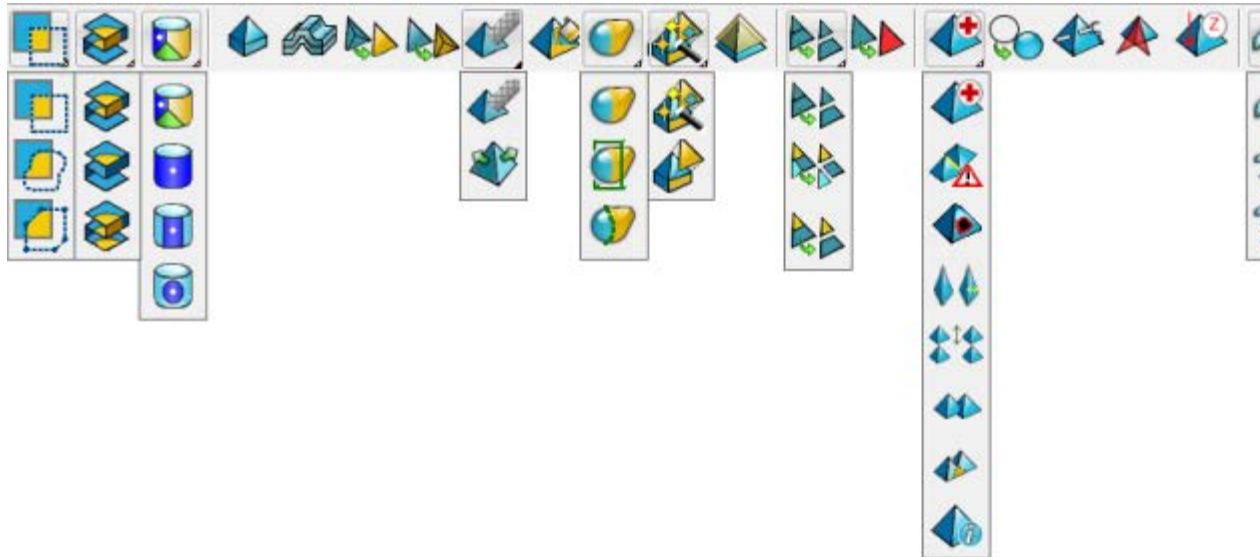
Trim region editing toolbar

The **Trim region editing** toolbar contains options for **Pcurve mode**

and **Boundary edit mode** . Click to switch between the two modes. The options that are displayed on the toolbar are determined by the type of trim editing that is selected.

For further details, see How do I trim surfaces?

Boundary editing toolbar

The **Boundary editing** toolbar enables you to edit the selected trim boundaries on a trimmed surface. The toolbar is displayed when the

Boundary edit mode option is selected on the **Trim region editing** toolbar.

Pcurve editing toolbar

The **Pcurve editing** toolbar provides tools for editing pcurves. The toolbar is displayed when the **Pcurve edit mode** option is selected on the **Trim region editing** toolbar.

General Edits toolbar

The **General Edits** toolbar is automatically displayed when the program is started.

Use the buttons on this toolbar for standard editing functions that apply to all object types. Each object type has functionality for editing that is specific to the object type.
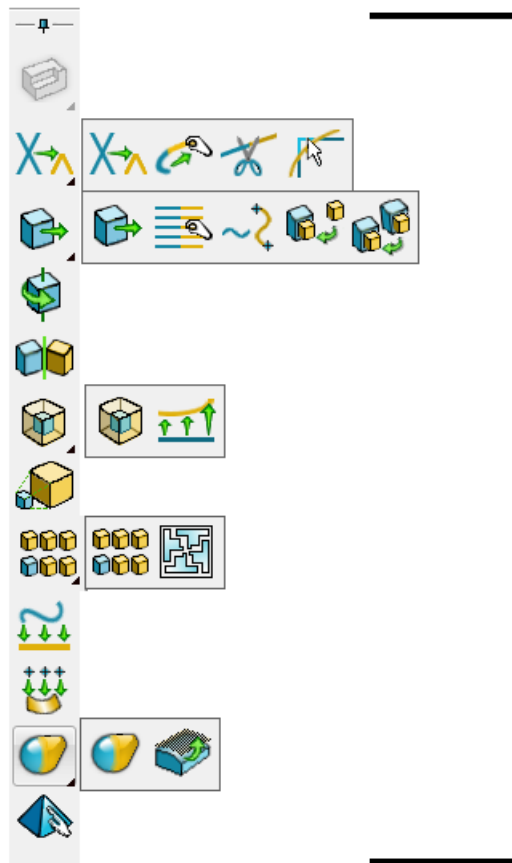
For further information on specific general editing functionality see:

Limit interactive (see page 207)

Limit point (see page 207)

Limit cut (see page 207)

Limit selection (see page 206)

Move (see page 207)

Stretch (see page 208)

Place wireframe (see page 208)

Align items (see page 208)

Best-fit-align (see page 211)

Rotate (see page 208)

Mirror (see page 213)

Offset (see page 213)

Variable offset

Scale (see page 218)

Pattern of objects (see page 219)

Nest items (see page 219)

Project (see page 231)

Project point (see page 232)

Morph objects (see page 232)

Morph to Cloud or Mesh (see page 232)

Sculpt meshes (see page 232)

Editing faces

Some **General Edit** operations can be used on faces of a solid as part of the **Direct Modelling** functionality. Examples of using the following can be found in the relevant sections of the Direct modelling manual:

- Move
- Rotate
- Mirror
- Offset

- Scale

When used to edit faces, **Next Solution** and **Previous Solution** buttons are added.
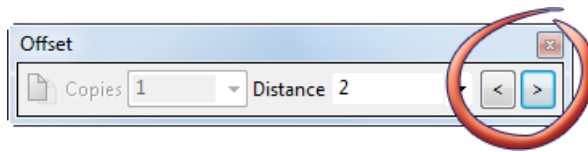
If there is more than one possible solution, use these buttons to select the required solution

Edit sub-components

Edit sub-components (see page 36) enables you to move, rotate, scale, and offset laterals and longitudinals rather than their parent surface. The type of item and sub-item that appear on the toolbar depend on your selection (wireframe, surface, or solid).

Limit selection

1 Click **Limit selection** to display the **Limit Selection** toolbar.
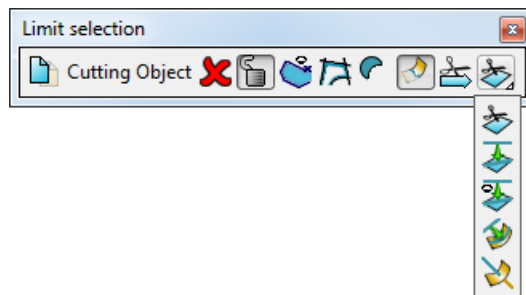
2 Use the following options on the toolbar to limit a selection of objects to a 'cutting' object:

 Keep original

 Cutting object deselected/selected.

 Cutting object unlocked/locked.

 Create and use polygon as cutter.

 Create and use composite curve as cutter.

 Keep both.

Trim both.

 Next solution.

Limit selection flyout. Repeatedly clicking on this button cycles through the following limit selection modes:

Limit selection.

Project along the axis normal to the principal plane.

Project along the axis normal to the view.

Project along surface normal.

Limit surface to a curve.

For further details, see Edit > General Edits > Limit selection. (see page 37)

Limit point

The **Limit point** (see page 52) button trims or extends objects to a point.

Limit point				
Edge/End Point	1	▼	Distance	0	▼
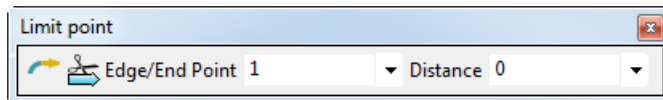
Limit cut

The **Cut object** (see page 54) button enables you to cut lines, arcs, and curves into smaller pieces, using the **Limit cut** dialog.

Limit cut	
No. of pieces	▼
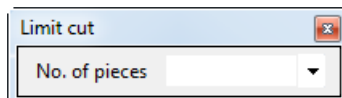
Limit interactive

The **Interactively Limit Wireframes** (see page 55) button enables you to remove portions of wireframe objects by selecting them.

Move

Click the **Move** button to display the **Move** toolbar.

Move					
Copies	1	▼	Movement 0	0	0

The toolbar is also displayed when you select **Edit > General Edits > Move** *(see page 56)*.

Use the options on the toolbar (see page 56) to move or copy an object or a group of objects by a specified offset or to a specified position. When switching between **Move**, **Rotate,** and **Scale** operations, the origin point is preserved.

Stretch

Click the **Stretch** (see page 59) button to stretch wireframe objects.

Rotate

Click the **Rotate** button to turn selected objects about the axis normal to the principal plane of the current workspace. It also enables you to create duplicate objects spaced around the rotation.
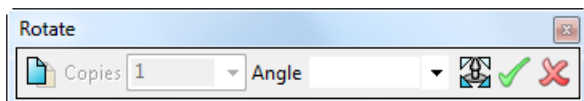
This toolbar is also displayed if you select **Edit > General Edits > Rotate** (see page 63).

When switching between **Move**, **Rotate,** and **Scale** operations, the origin point is preserved.

*You can use the **Rotate** button to edit faces as part of Direct Modelling functionality.*

Place wireframe

Use the **Place Wireframe (see page 61)** button to place a curve between two points on a single object or on separate objects.

Align items

Use the **Align items** dialog to adjust the alignment of the model.

1 On the **General Edits** toolbar, click the **Align Items** button to display the **Align Items** dialog.
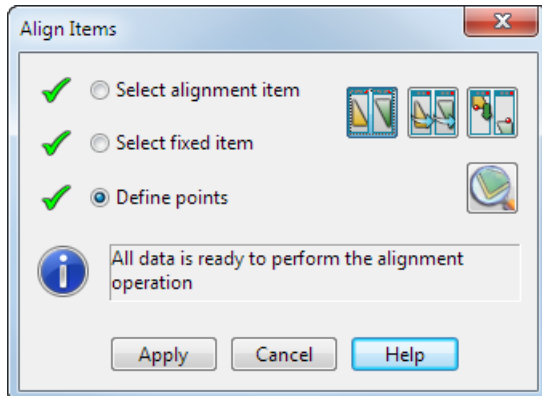
2 Select the alignment object ①. The alignment object is that object that moves.

3 Select the fixed object ②. The fixed object does not move.
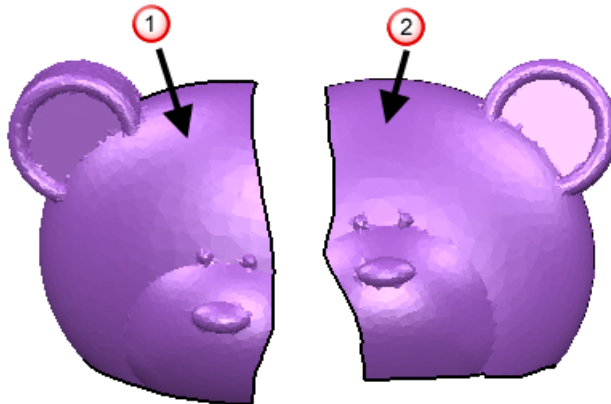
If necessary, you can split the items into separate views by clicking the **Split view** button. This button is only available when you have selected the alignment and fixed items.

Clicking the Split view button enables the following buttons on the dialog:

> **Link views** — Click this button to link the Alignment Item view and Fixed Item view together. You can pan, rotate, and zoom the views together when they are linked. This button is only available when you are using the split view.

> **Match views** — Click this button to pan and scale the alignment view to match the fixed view. This button is only available when you are using the split view.

> **Preview** — Click this button to display a preview of the alignment in a single view. This button is only available when all of the alignment points are defined.

4 Select points on the alignment and reference object to define sets of points. You must define at least three sets of points; the information panel tells you that you have sufficient points selected.

Choose points in one of the following ways:

▪ all points on alignment object followed by all points on the reference object.

▪ pairs of points, one on the alignment object and one on the reference object.

5 Adjust the point selection as appropriate. You can delete existing alignment points by clicking on them.

The cursor changes ✖ to indicate that you are hovering over a point that can be deleted.

6 Click **Apply**. The alignment object will move so that the selected points are aligned.

Continue to select objects and points to adjust the alignment as necessary.

7 Click **OK** to accept the changes and close the dialog.

Best-fit align

Use **Best-fit roughly aligned items** to align multiple items that are already roughly aligned. The example shown below has been roughly aligned using **Align items**, but the nose is not aligned correctly.
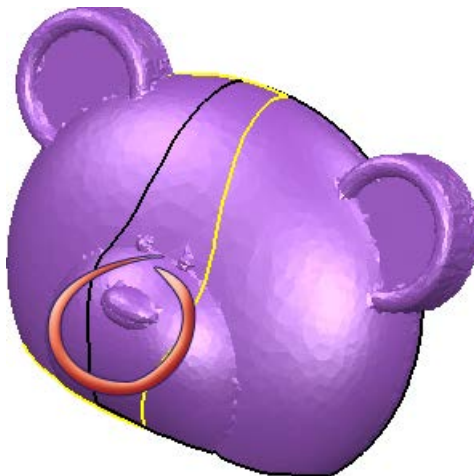
1 On the General Edits toolbar, click the **Best-fit items** button to display the **Best-fit Align** dialog.
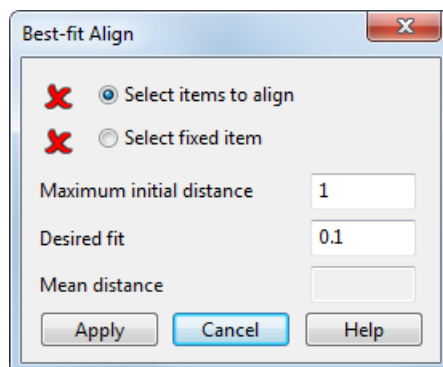
2 Select the alignment items ①. Thes are the items that move.

3 Select the fixed item ②. The fixed item does not move.
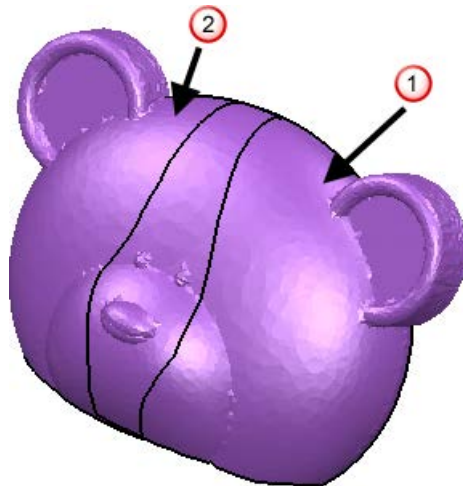
4 Click **Apply**. The alignment item moves to align with the fixed item. In this example, the alignment is satisfactory; if necessary adjust the deviation and mean error values to improve the alignment.
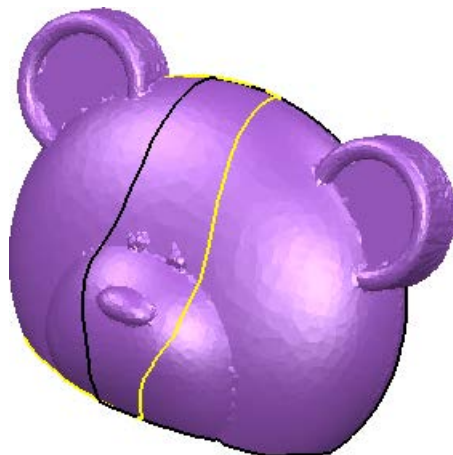
5 Click **OK** to accept the changes and close the dialog.

*You can select multiple items before clicking the **Best-fit items** button (*General Edit toolbar*).*

- *One item is selected automatically to be the fixed item; you can change this selection if required.*

- *The items to be aligned are all highlighted in the primary colour; the fixed item is highlighted in the secondary colour (purple in the 2015 R2 default colour scheme)*

Mirror

Click the **Mirror Items** button to reverse the original or make a reversed copy of the original. Alternatively select **Edit > General Edits > Mirror** (see page 69) from the menu.
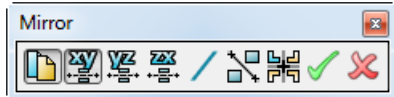
Use **Mirror** to:

- Replace the original with a reversed copy.
- Keep the original in place and position the reversed copy as required.

Mirror can be used to edit faces as part of Direct Modelling functionality.

Offset

Click the **Offset** button to move or copies objects by a set distance along the object normal. Alternatively, select **Edit > General Edits > Offset** (see page 73) from the menu.
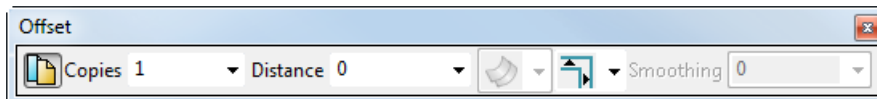
Undo and redo operations are available when the toolbar is displayed. In previous versions, clicking **Undo** closed the offset operation and removed the toolbar.

Offset can be used to edit faces as part of Direct Modelling functionality.

Variable Offset

Use the **Variable Offset** button to create variable offsets for 2D curves, composite curves, curves on surfaces, and single laterals and longitudinals.

To use Variable Offset:

1 Select the curve.

2 Click the **General Edits** flyout to display the **General Edits** toolbar.

3 Right-click the **Offset** button to display the **Offset** flyout.

4 Click the **Variable Offset** button.

5 Use the **Variable Offset** dialog (see page 214) to create the required offsets.

Variable Offset Wireframe dialog

Use this dialog to create the required offsets.
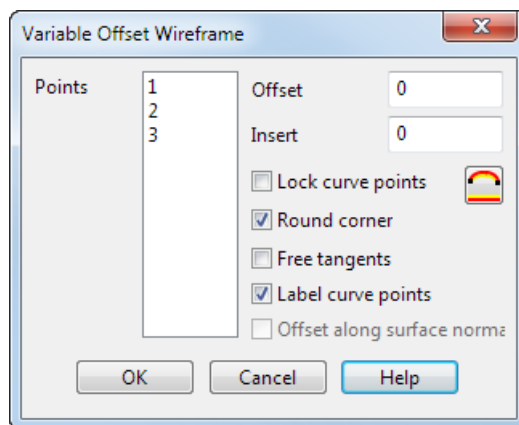
Points — All the points on the selected curve are listed. This enables you to select individual points to offset.

Offset — Enter the offset value for the points selected in the **Points** list.

Insert — This enables you to insert extra points into the offset curve. For example, if you enter **2.5**, an imaginary point is inserted halfway along the base curve between points 2 and 3. It is then mapped to the offset curve. This is the parametric value half way between points 2 and 3.

Lock curve points — When you enter a value in the **Offset** text box, the offset point is locked and the **Lock curve points** option is selected automatically. Deselect the **Lock curve points** option to unlock an offset point. This option is used with **Free the unlocked points** .

Free the unlocked points — This releases unlocked points so they offset linearly between the locked points.

In the following example, points 1 and 2 are locked and offset to a value of **5**. Point 5 is locked and offset to a value of **3**.
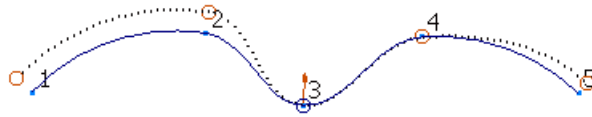
When you click the **Free the unlocked points** button, the unlocked points 3 and 4 offset linearly between the locked points, as shown below.
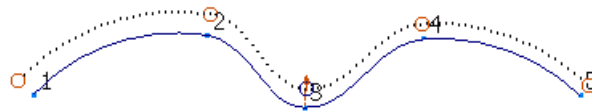
Free tangents — Select this option to free the tangent directions and its magnitudes at the offset points on the offset curve.

In the following example, point 1 on the base curve is offset to the value of **10** and point 4 is offset to the value of **50.** Selecting **Free the unlocked points** has freed points 2 and 3 to be offset linearly.

With **Free Tangents** selected, the offset curve is not straight and the tangent direction remains aligned with the base curve.
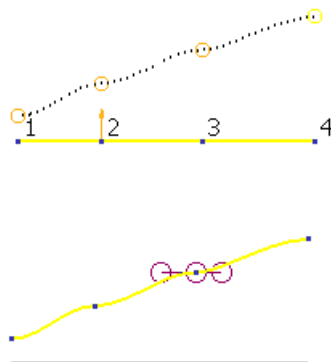
With **Free Tangents** deselected, the tangent directions and its magnitude are freed. The tangent direction now aligns with the offset curve that has straightened.
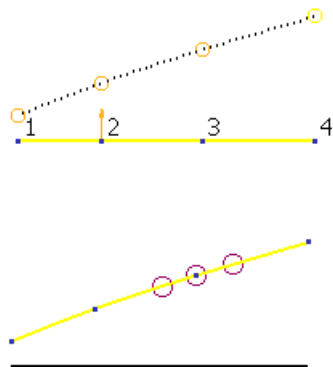
Round corner — Select this option to create a rounded corner at selected points with a discontinuity.

This is not available for offsetting laterals or longitudinals.

In the following example, an extra point is inserted automatically at the point of discontinuity, creating 4 points on the curve. This extra point is displayed in the **Points** list and can be offset in the same way as the other points.
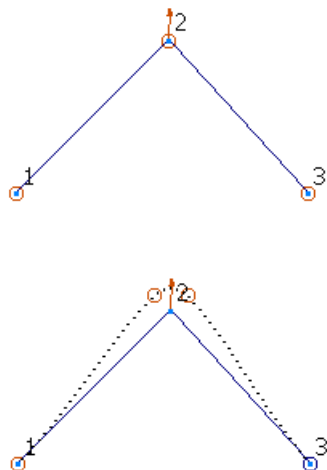
Label curve points — Select this option to display point labels.

Offset along surface normal — Select this option to offset the point in the direction of the surface normal. It is available for points that have dependencies on the surface.

To use the offset option:

1 Select the point on a surface curve.

2 Select the **Offset Along Surface Normal** option.

Enter the offset value. The point offsets in the direction of surface normal.

Offsetting points on a single surface

To offset points on a single surface:

1 Select the surface you want to offset.

2 Click the **General Edits** flyout to display the **General Edits** toolbar.

3 Right-click the **Offset** button to display the **Offset** flyout.

4 Click the **Variable Offset** button.

5 Use the options on the Variable Offset Surface dialog (see page 217) to select the four points on the outer boundary of the surface that you want to offset.
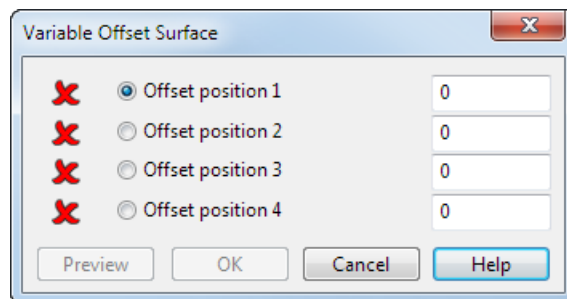
Variable Offset Surface dialog

Use the options in the dialog to select four points on the outer boundary of the surface that you want to offset. For further details, see Offsetting points on a single surface (see page 217).
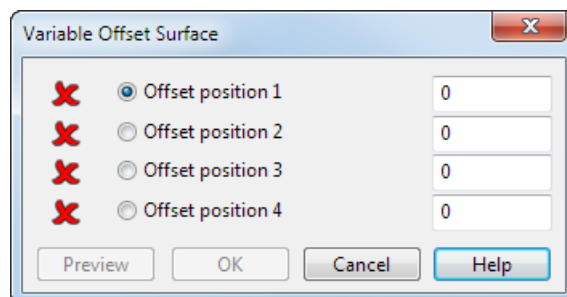
Offset position — Select each Offset Position and click on the outer boundary of the surface to position the four points, as shown in the example below. The ✖ changes to ✔.

Enter the offset distance for each point in the text box alongside each **Offset position**.
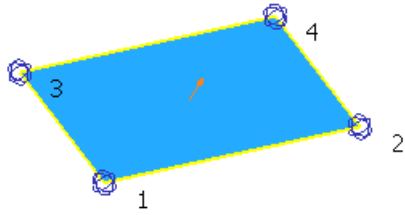
Preview — This displays the results of the offset on the surface.

OK — This creates the surface offset with the distances specified.
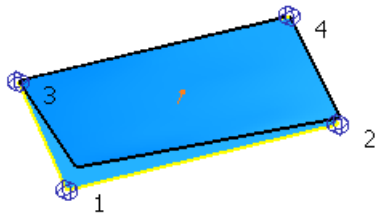
Scale

Use the options on the **Scale** toolbar to make objects larger or smaller.

Click the **Scale** button to display the **Scale** toolbar.

You can also display the toolbar by selecting Edit > General Edits > Scale (see page 78).

Use toolbar to resize objects in the following ways (see page 78):

- Using **Uniform** (see page 79) scaling.

- Using **Non-uniform** (see page 80) scaling.

- To a **Projected Volume (see page 81).**

- To an **Enclosed Volume.** Scale the volume enclosed by the selected solid and face of the active workplane. Enter the required volume in centilitres.

When switching between **Move**, **Rotate,** and **Scale** operations, the origin point is preserved.

Scale can be used to edit faces as part of Direct Modelling functionality.

Pattern of objects

Use the **Pattern of objects** (see page 86) button to create rectangular, honeycomb, and circular patterns of objects and to create patterns on wireframe items.

Nesting

Use the **Nesting** dialog to arrange PowerSHAPE objects on the principal plane of the active workplane (see page 224).
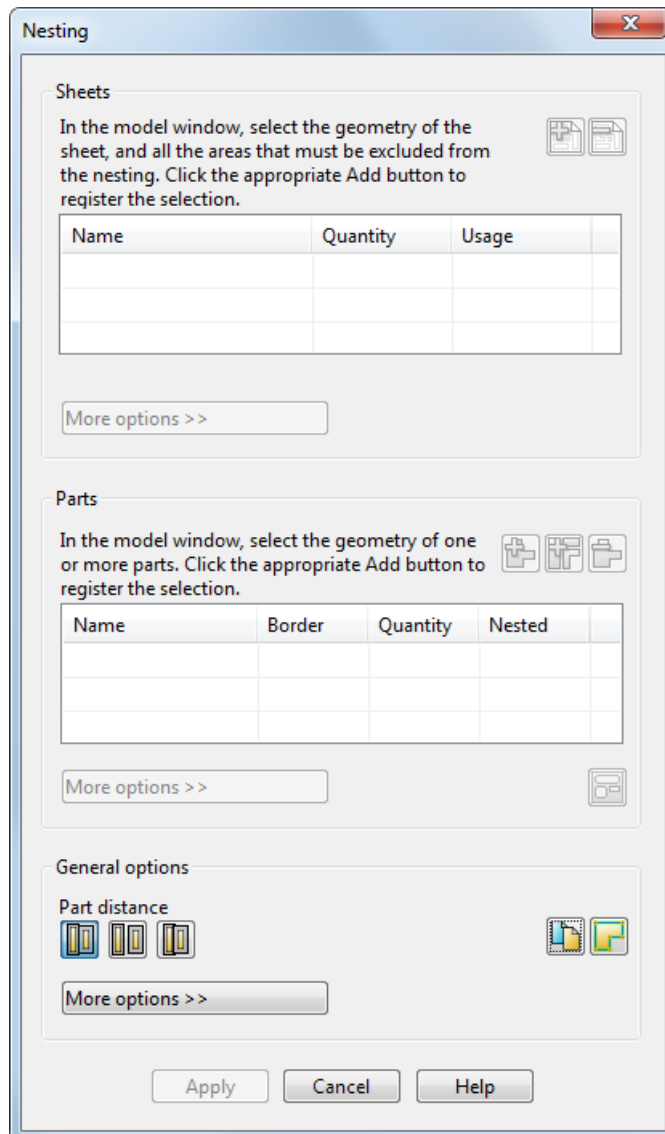
Sheets

Use this section to select the geometry of the sheet that is to be included in the nesting.
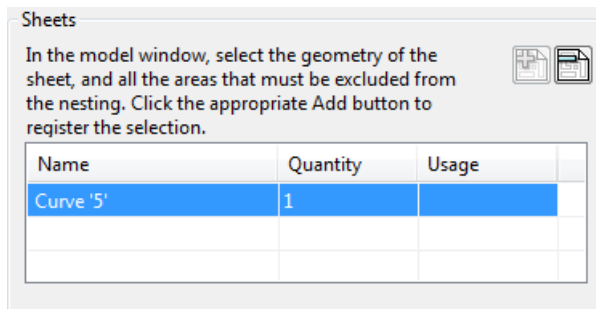
The **Usage** column on the **Nesting** dialog displays the area occupied by parts nested on all the instances of the sheet.

As you move the mouse cursor over a sheet in the current model, the area usage for a particular sheet instance is shown below the list of sheets. This is useful when you have several sheets on a model.
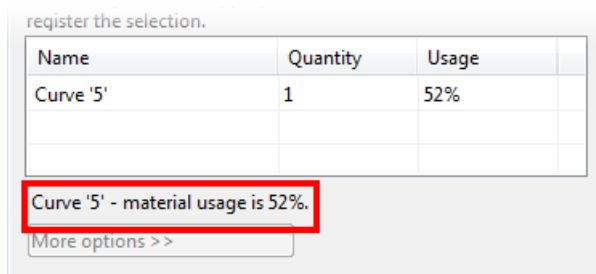
Use the following buttons to add and remove sheets from the list:

Add Sheet — Register the selected geometry as a sheet for nesting and add into the list.

Remove Sheet — Remove the selected sheet from the list of sheets registered for nesting.

More options — If a sheet is selected in the list, click this button to display additional nesting options for the selected sheet.
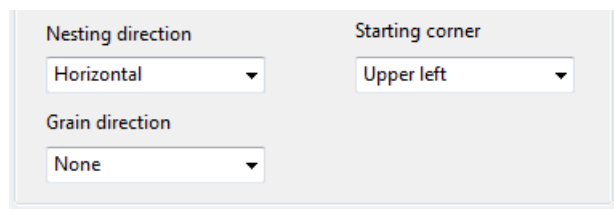
Nesting direction — Select the required nesting direction from the list.

Starting corner — Select the starting position for the nesting from the list.

Grain direction — Select the grain direction from the list

Parts

Use this section to select the parts to nest on the sheet.
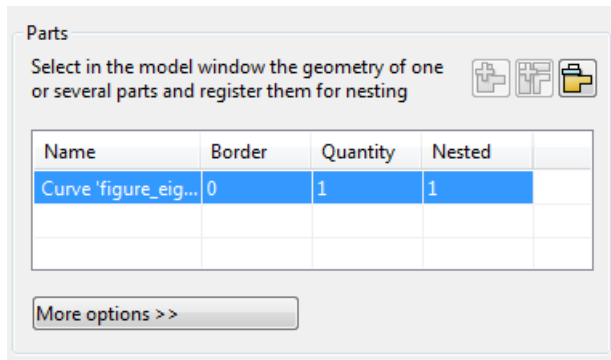
Add as single — Add the selected geometry to the **Parts** list as single part. You can use this button to register multiple pieces of geometry as a single part.

> *A workplane can be registered with the part for nesting. This is useful if you want to machine the multiple parts, because the workplanes can be used to copy the toolpaths.*

Add as separate — Add the selected geometry to the **Parts** list as multiple parts. You can use this button to register multiple pieces of geometry as multiple parts.

Remove part — Remove the selected part from the **Parts** list.

The **Parts** table includes the following columns:

Border — Specify the clearance around a part.

Quantity — Specify the minimum number of parts to nest.

Nested — PowerSHAPE indicates the number of parts that were nested when the last nesting operation was applied.

Choose boundaries — When an item has multiple boundaries (see page 230), click this button to select:

- the boundary that is used as the outline of the part for nesting purposes.
- boundaries that are simply transformed along with the part.

More options — If a part is selected in the list, click this button to display additional options for the selected part.

Border — Specify the clearance around a part.

Quantity — This is the minimum number of items of this type that the PowerSHAPE ,will attempt to nest on the sheets. When this number of items has been nested, PowerSHAPEwill attempt to nest more items, up to **Maximum quantity**.

Part rotation— Select the part rotation method from the drop down list.

Rotation angle — Enter a part rotation step angle, or a list of angles separated by spaces.

Maximum quantity — Enter the maximum number of that part that can be nested. Alternatively, use the arrows to set the value. Use a priority of **1** to indicate that the part has the highest priority.

Part priority — Enter the priority of the part. Alternatively, use the arrows to set the value.

Mirroring — Select an option to specify if the part can be mirrored and the mirroring method to be used.

Grain direction — Select the grain direction of the part from the drop-down list

General options

Use this section to specify the general parameters that are to be used when nesting the parts.

Part distance — Use these buttons to select how the distance between the parts is to be calculated.

Border to border — Distance between adjacent parts is the sum of the part borders.
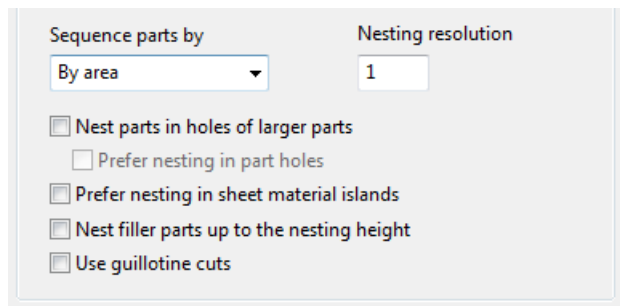
Border to clearance — Distance between adjacent parts is the sum of the part borders and the border clearance.

Overlapping — Distance between adjacent parts is the largest part border.

Create copies — Use this toggle button to specify if a copy of the original sheets and part geometry should be created.

Nesting boundaries — Use this toggle button to specify if the nesting boundaries of the sheets and parts should be created.

More options — Click this button to display additional general options.

Sequence parts by — Select the order in which parts with the same priority are ordered for nesting.

By area — Nest parts with the biggest area first.

By perimeter — Nest parts with the biggest perimeter first.

Nesting resolution — Specify the accuracy of the nested pattern. The actual distance between the parts could be greater than the specified value, by one resolution unit. The smaller the resolution, the higher is the accuracy of the nesting, but the slower it is to calculate the layout.

Nest parts in holes of larger parts — If this option is selected, parts can be nested in the holes within larger parts.

Prefer nesting in part holes — If some of the nested parts have holes and this option is selected, PowerSHAPE will try to nest parts in the holes of the previously nested parts. If it isn't possible, parts will be nested elsewhere on the sheet.

You must select **Nest parts in holes of larger parts** to access this option.

Prefer nesting in sheet material islands — If the sheet material includes some islands and this option is selected, PowerSHAPE tries to nest parts in the holes of the sheet material islands. If it isn't possible, parts are nested elsewhere on the sheet.

Sheet material islands are holes of locked parts; parts that are not nested but stay in the same position on the sheet.

Nest filler parts up to the nesting height — A filler is a part that is included in any space after the other parts have been nested in priority order. You define a filler part by setting a maximum number of parts.

- If this option is deselected (default), the minimum number of main parts are nested and filler parts are included as required.

- If this option is selected, the minimum number of main parts are nested and filler parts are included only up to the nesting height that is determined by the nesting of the main parts.

Use guillotine cuts — Select this option to align nested parts orthogonally so that a guillotine cut can be used to cut across the sheet (in either direction), without intersecting the nested parts.

Nesting — an example

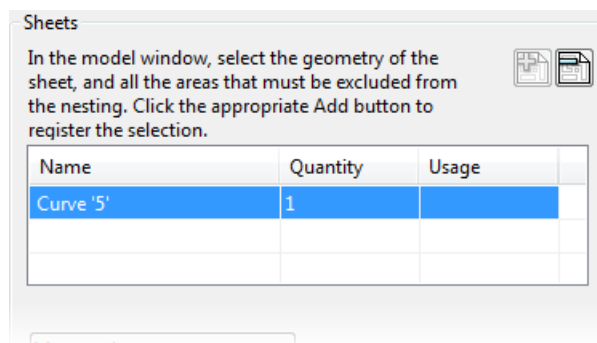This example shows you how to nest three different parts in a sheet.
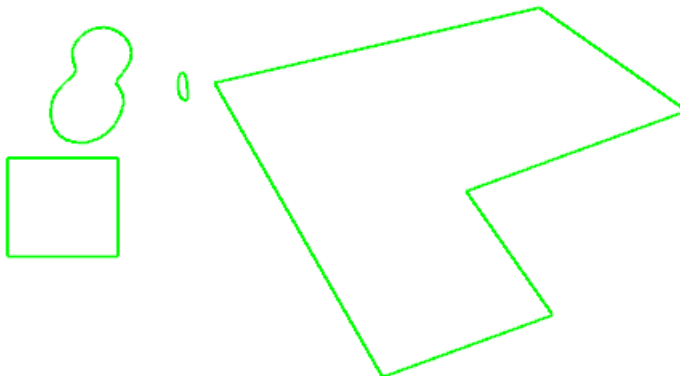
1 On the General Edits toolbar, click the **Nest Items** button to display the **Nesting** dialog
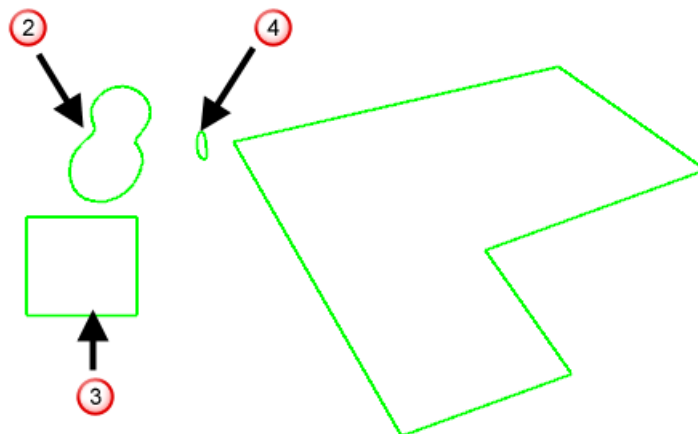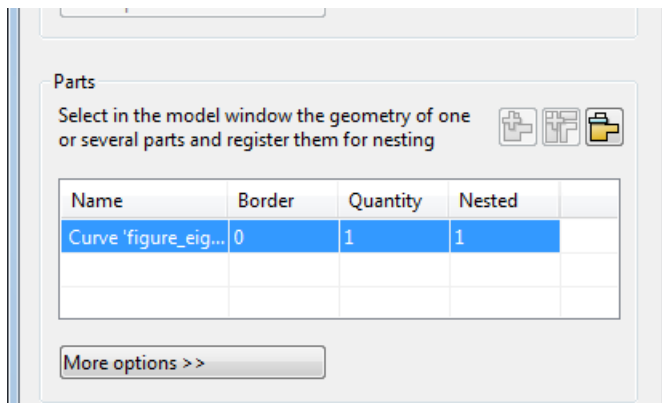
2 Select a sheet ①.
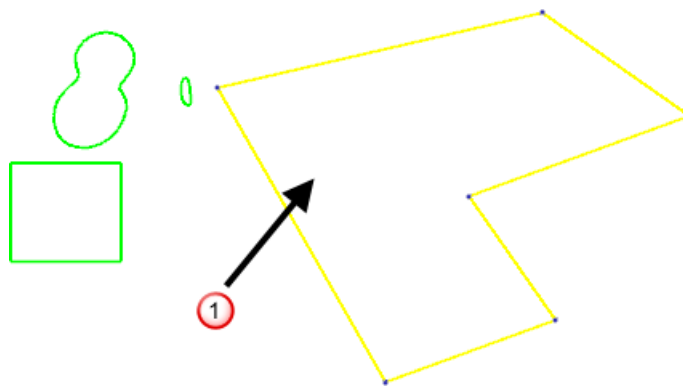
3 Click the **Add sheet** 🖼 button. The sheet is added to the **Nesting** dialog

4 Select a part: Curve 'figure_eight' ②.
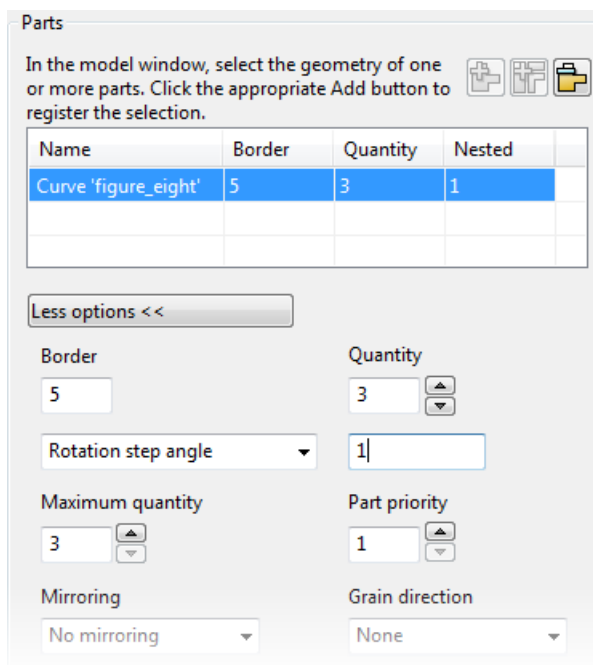
5 Click **Add part** .

6 Set the following parameters:

- **Border**: **5**

- **Quantity**: **3**

7 Click **More options**.

8 Set the following parameter options for the part:

- **Rotation step angle**: **1**

- **Part Priority**: **1**

9 Select part: Curve 'square' ③.

10 Click **Add part** .

11 Set the following parameters in the parts list:

- **Border**: **1**

- **Quantity**: **2**

12 Set the following parameter options for the part:

- **Rotation step angle**: **1**
- **Priority**: **2**
- **Maximum quantity**: **2**

13 Select part: Curve 'filler' **④**.

14 Click **Add part** 🔲.

15 Set the following parameters in the part list:

- **Border**: **1**
- **Quantity**: **200**

16 Set the following parameters options for the part:

- **Rotation step angle**: **1**
- **Priority**: **3**
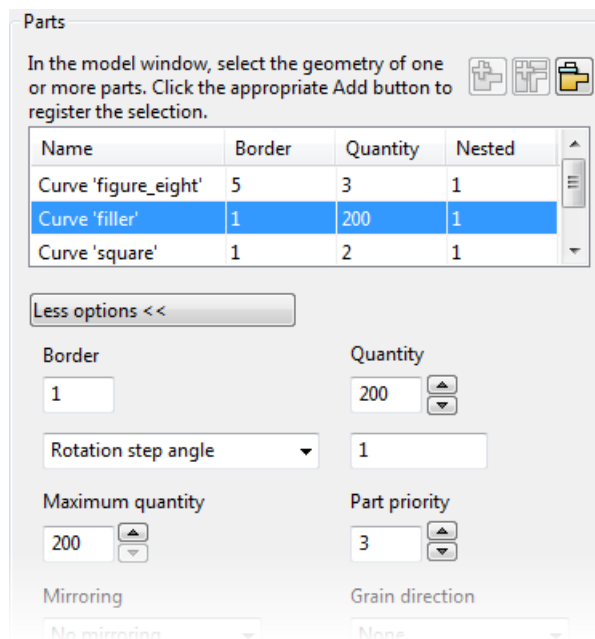
17 Click **Apply**. The parts are nested according to the parameters that you have selected.
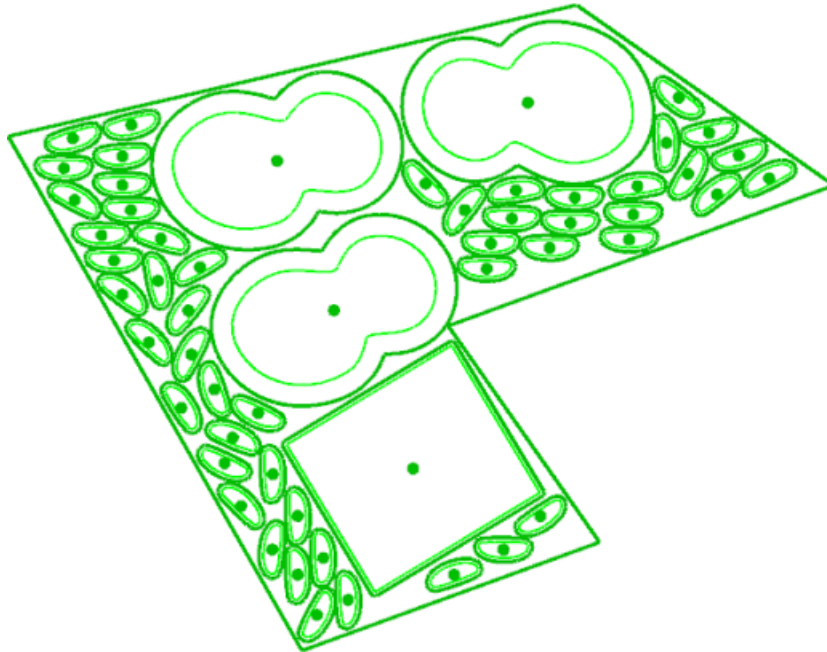
The list in the **Parts** section of the dialog is updated to show the number parts that are nested.
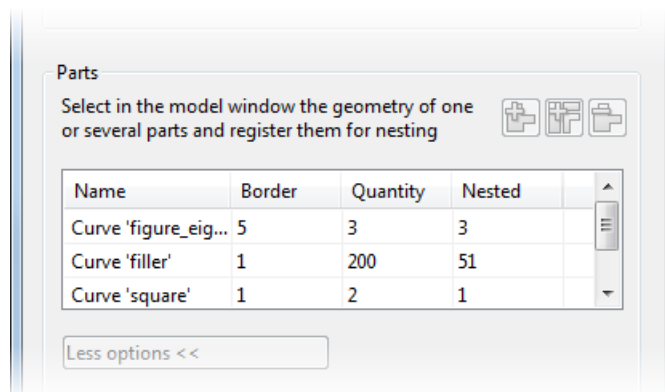

This shows that:

- all the required **Curve 'figure_eight'** parts (priority 1) parts have been included.

- one of the **Curve 'square'** parts (priority 2) parts have been included.

- 51 of the **Curve 'filler'** parts (priority 3) have been included.

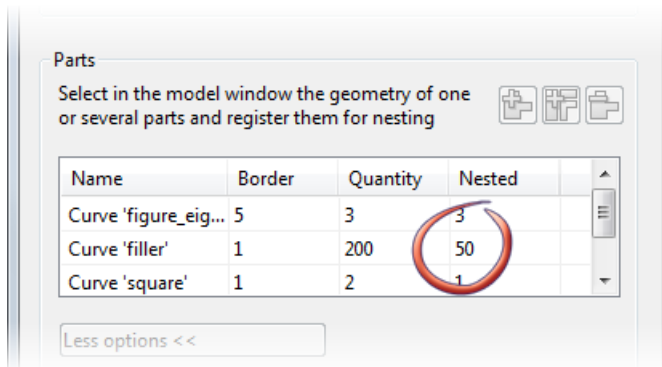**18** Click and drag the **Curve 'square'** part. As you move the part, overlapping parts turn red.
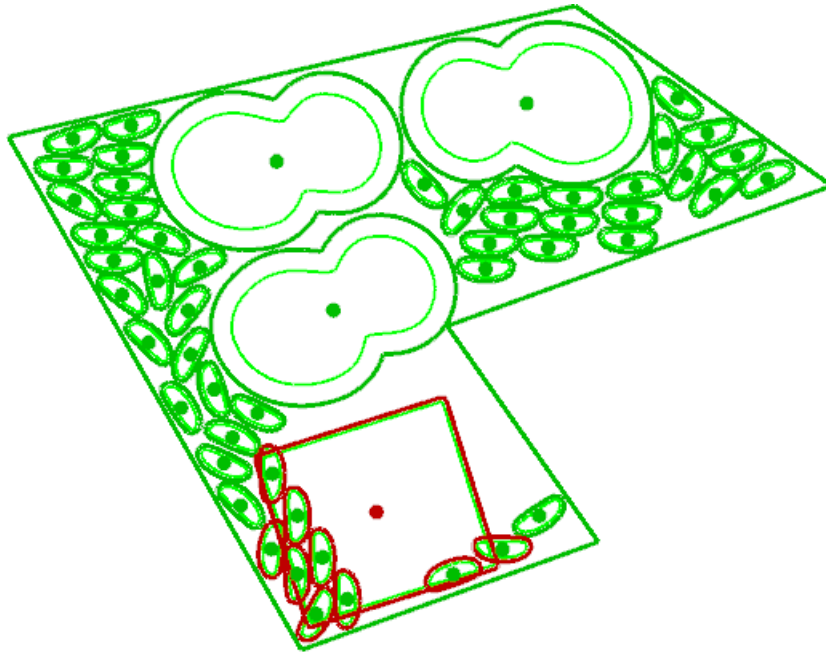
> *You can also rotate a part by selecting it in the graphics window and dragging the border outline.*

19 Click **Apply** to rearrange the parts based on the new position of the **Curve 'square'** part.
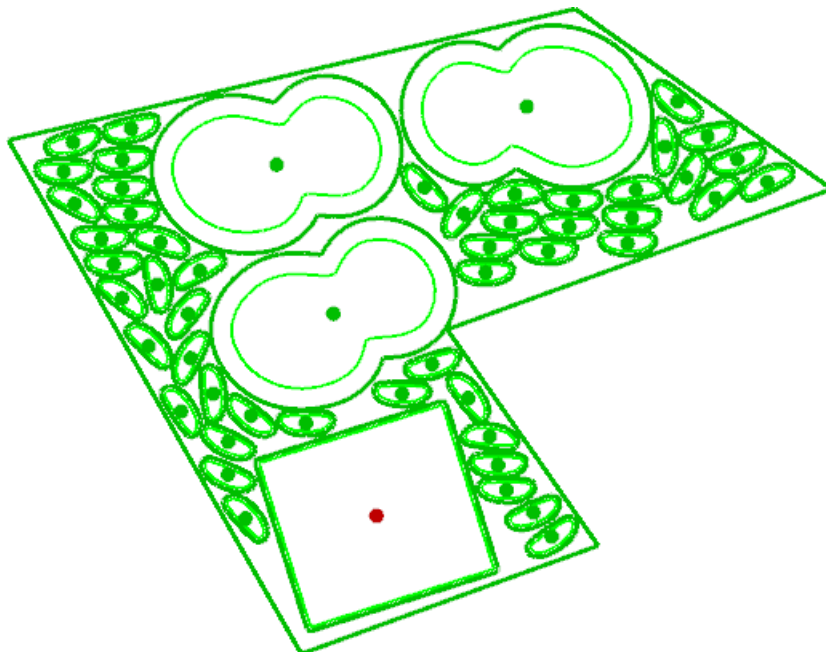
The number of **Curve 'filler'** parts that can be nested has reduced.


When you reposition an item manually, the item is locked in the new position and any subsequent nesting operations do not reposition the part. This is indicated by a red dot in the centre of the item. You can unlock a locked part by toggling the red dot to green.

Nesting boundaries — an example

The part to be nested on the sheet consists of of two concentric circles.

1 Select the sheet and click **Add sheet**.

2 Select the concentric circles and click the **Add as a single part** button.

3 Enter **10** as the **Maximum** quatity to be nested.

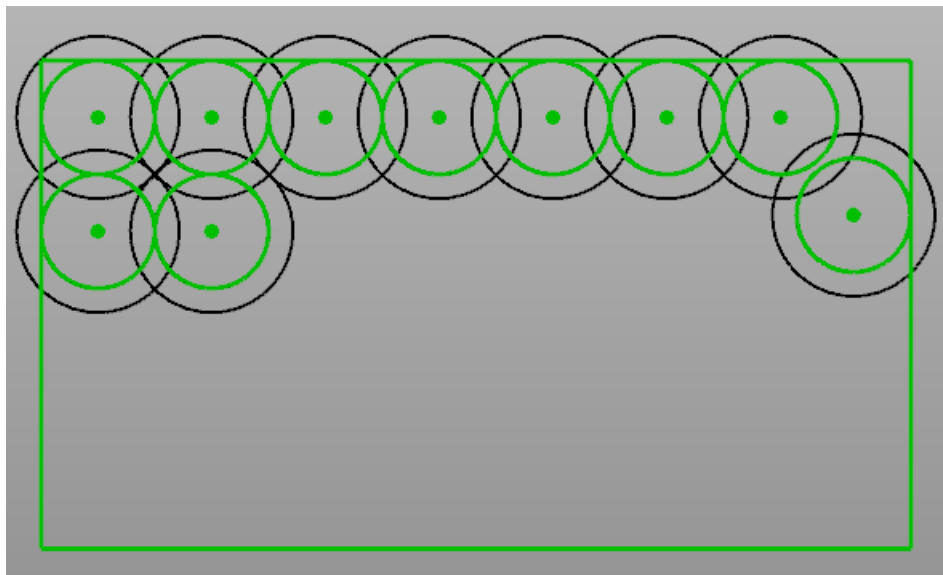**4** Click **Apply** to nest the parts.
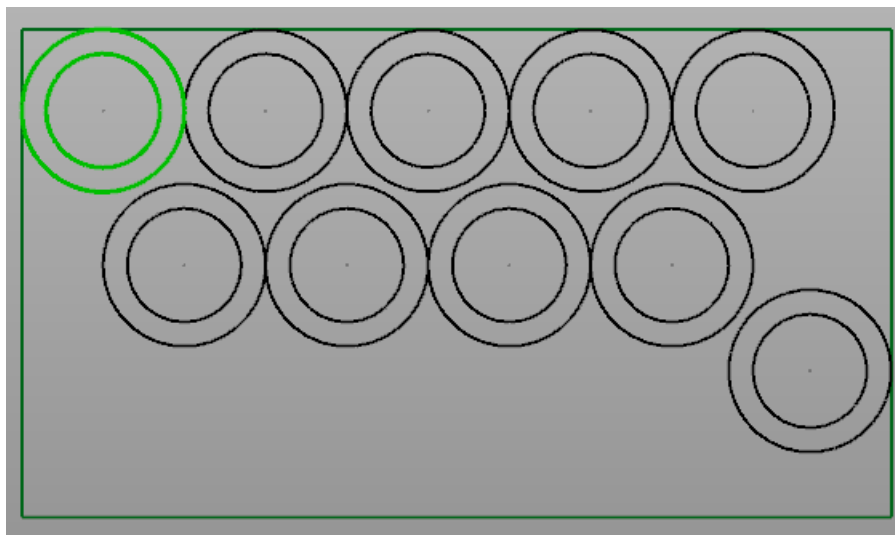
5 Click .

6 Click the outer boundary. The boundary turns red, to show that it will not be used when nesting.
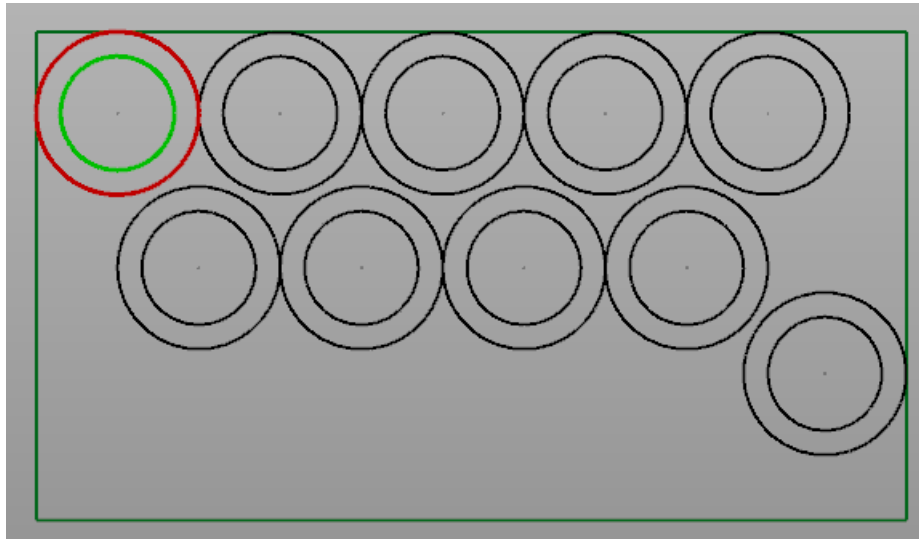
7 Click ✔ on the **Nesting** toolbar to leave nesting boundary editing mode. The options on the dialog become active.

8 Click **Apply**. The parts are re-nested, using the inner circle as the boundary.

Project

Use the **Project** (see page 103) button to project wireframe objects along an axis onto a plane at a particular point.

Project point

Use the **Project point** ![icon] button to project points along the principal axis onto a surface or a mesh (see page 104).

Morph objects

![icon] **Morph object(s)** enables you to morph the object you are editing.

For further details see Edit > General Edits > Morph object(s) (see page 105)

Morph to Cloud or Mesh

![icon] **Morph to Cloud or Mesh** enables you to morph the object you are editing with respect to a separate cloud or mesh.

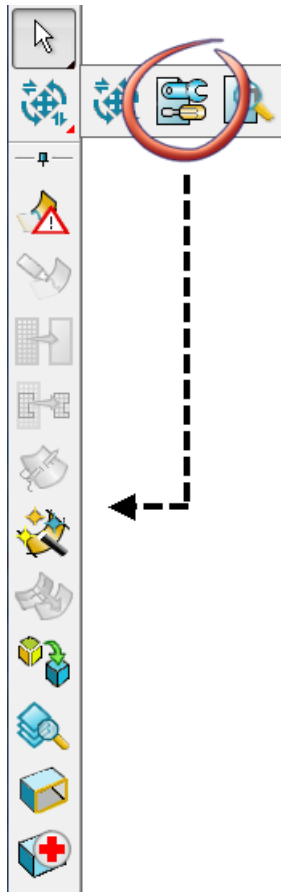For further details see Edit > General Edits > Morph to Cloud or Mesh (see page 105).

Sculpt

(Available in PowerSHAPE Pro)

Use **Sculpt** ![icon] to change the shape of meshes and surfaces. This functionality is also available by selecting **Edit > General Edits > Sculpt Meshes**.

Model fixing toolbar

Use the tools on the **Model Fixing** toolbar to fix your model.

Select from the following:

 Identify badly trimmed surfaces (see page 38)

 Surface trim region editing

 Approximate

 Simplify outside trim

 Stitch

 Smart surfacer

 Orient normals

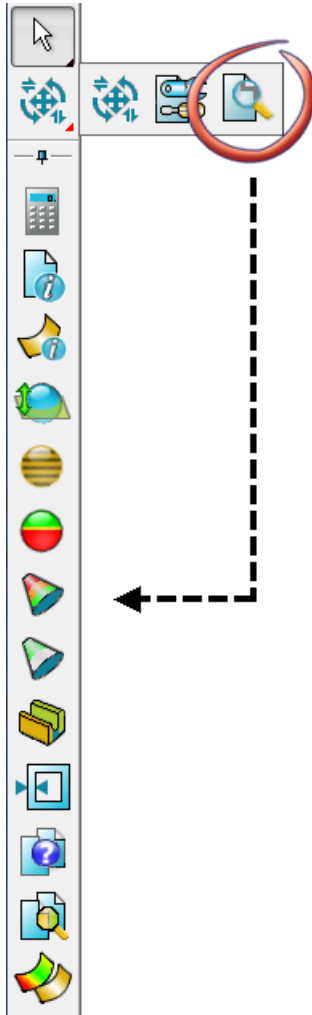 **Solid from surfaces**

 Find duplicates

Show open edges in solid

Repair solid (see page 40)

Model analysis toolbar

Use the tools on the **Model Analysis** toolbar to analyse your model.

Select from the following:

Calculator

Properties

Surface inspection

Dynamic sectioning (see page 42)

Smoothness shading (see page 45)

Undercut shading (see page 49)

Curvature shading (see page 51)

Minimum radius shading (see page 53)

Thickness shading (see page 54)

Solid wall thickness (see page 56)

Model compare

Model visual difference (see page 57)

Comparison analysis (see page 59) *(PowerSHAPE Pro licence only)*