PowerSHAPE 2015 R2

Reference Help

Modelling concepts

PowerSHAPE

Acknowledgements

Patent Information

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

Morphing functionality is subject to patent application GB 2401213.

PowerSHAPE 2015 R2. Published on 03 March 2015

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Basic modelling concepts

Use the following sections to find information on basic modelling concepts:

PowerSHAPE

PowerSHAPE enables you to design complex shapes using the powerful 3D modelling tools in PowerSHAPE. It is ideally suited to preparing models for manufacture; either from scratch or by editing a part-complete model, imported from another CAD system.

PowerSHAPE supports standard Microsoft Windows™ functionality such as cut, copy and paste, drag-and-drop, and rubber-band editing of objects. You can also edit any modelling object (such as line, arc, curve) by right-clicking and using the context menus. The menus contain all the main operations available for the selected object.

Power Solution

PowerSHAPE is integrated with all the other Power Solution products. For example: PowerMILL, which prepares NC data to cut PowerSHAPE models on a CNC machine tool.

Over the years, DUCT has grown, from its roots at Cambridge University in England, into the market-leading software for the design and manufacture of molds, tools, and dies around the world.

The Power Solution products retain the power and flexibility of Delcam's DUCT5 CAD/CAM system, but with a modern Microsoft Windows™ style interface. The DUCT range of integrated CAD/CAM products have been continuously developed for over 20 years.

Ease-of-use

Ease-of-use is central to PowerSHAPE:

- You can create most objects with one or two mouse clicks.
- You can dynamically drag dimensions using an object's editing handles to create the desired size and shape.
- You can also enter discrete dimension values using dialogs.
- The Intelligent Cursor™ generates construction lines and highlights objects' snap points and tangencies as the cursor passes over an object.

Modelling for manufacture

PowerSHAPE simplifies the addition of manufacturing features such as:

- chamfers.
- fillets.
- split surfaces.
- draft surfaces.

The software is designed to handle components with complex surface geometry, making it an excellent tool for designing molds, dies, electrodes, and patterns.

Surfacing tools

PowerSHAPE offers sophisticated surfacing tools, including surface creation from curves and dynamic surface manipulation, allowing quick and easy creation and subsequent modification of the CAD surfaces.

Starting PowerSHAPE

To start PowerSHAPE:

1. From the **Start** menu select **All Programs**.

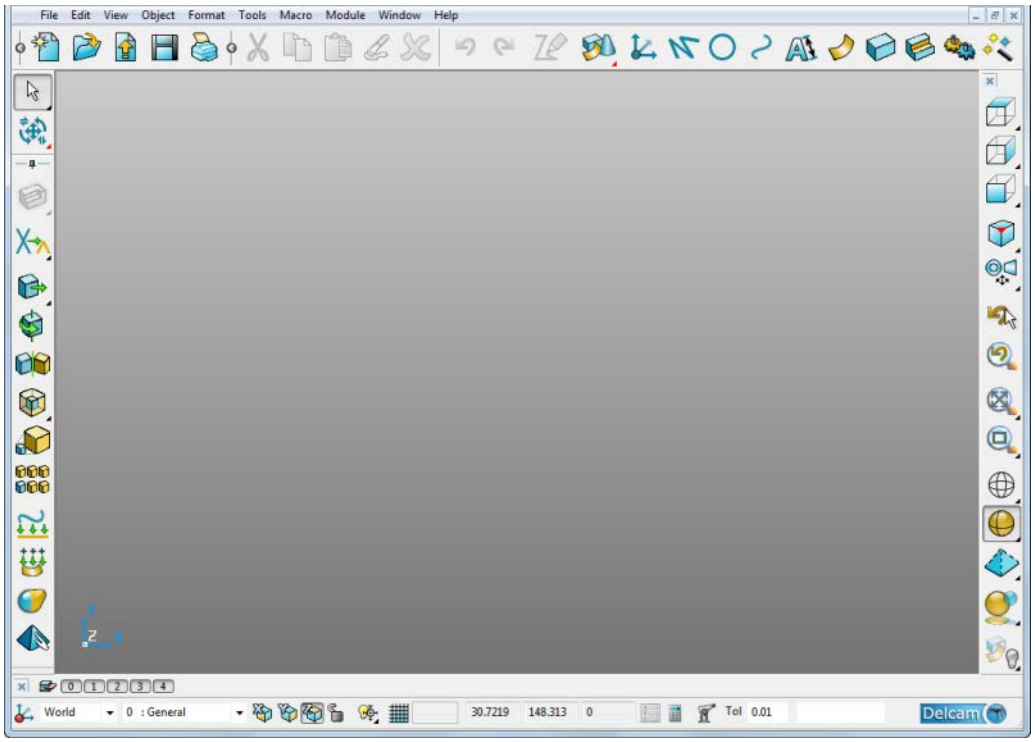
2 Select **Delcam**.

3 Select **PowerSHAPE**.

4 Select **PowerSHAPE 2015 R2**.

After a brief wait, the program starts up. A new large window appears on the screen.

This is **PowerSHAPE**.



Alternatively, you can double-click the **PowerSHAPE** icon on your desktop  to start PowerSHAPE.

For advanced start-up options see:

- Starting PowerSHAPE in different modes (see page 188).
- Table of PowerSHAPE start modes (see page 190).
- Memory allocation.
- Licensing information.

PowerSHAPE functionality

The PowerSHAPE product is available in the following forms:

1. **PowerSHAPE** (see page 8) — This is the standard product, including wireframe, surface, and solid modelling.
2. **PowerSHAPE Pro** (see page 8) includes:
   - rendering
   - triangle modelling
   - assembly modelling
   - drafting, morphing
   - reverse engineering functionality
   - Delcam's unique "Total Modelling" functionality for adding 3D artwork and textures to designs.
3. **PowerSHAPE-e** (see page 9) is a completely free version of PowerSHAPE-Pro.
4. **Delcam Designer** (see page 9) is the solid and surface modelling companion for ArtCAM JewelSmith.
5. **PowerMILL Modelling** (see page 10) provides easy-to-use surface modelling tools to prepare models for machining.
6. **PowerSHAPE Companion** provides users with functionality for creating and repairing 3D solid models.
7. **PartMaker Modeling** uses **PowerSHAPE** to design products from scratch and to preparing existing product models for manufacture

The following are also available:

1. **Delcam Toolmaker** includes
   - PowerSHAPE Pro
   - toolmaking functionality
   - assembly modelling
   - drafting functionality
2. **Delcam Electrode** gives access to the electrode wizard. A copy of PowerSHAPE, PowerSHAPE Pro, or Toolmaker is required.
3. **Delcam Drafting** allows the creation of drawings with hidden detail sections, annotations, and Bill of Materials.
4. **Delcam Crispin SoleEngineer** includes:
   - PowerSHAPE

- grading wizards
- heel wizards

*If you have not purchased a licence for any option, you can still try the functionality by running **PowerSHAPE-e**.*

PowerSHAPE

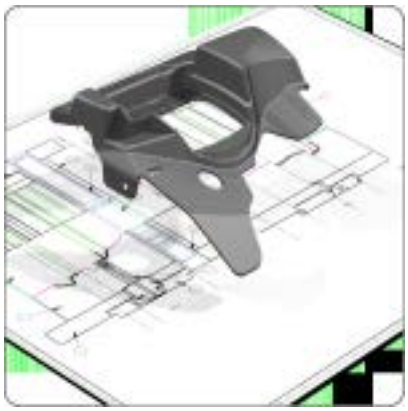
This contains the core functionality, including basic surface, solid, triangle, and wireframe modelling. It also includes generating engineering drawings using drafting functionality.

PowerSHAPE Pro

This contains:

- PowerSHAPE
- Additional triangle modelling features
- Assembly modelling to connect components together
- Rendering to creates photo-realistic images
- Morphing for whole-body edits
- Total modelling to add wrapped 3D artwork and textures to designs
- Reverse engineering features

PowerSHAPE-e



PowerSHAPE-e is a free version of PowerSHAPE-Pro that you can use to evaluate design concepts, or share models with your supply chain. It is ideal for use in education for CAD teaching or research and you can also use it to evaluate PowerSHAPE functionality before buying. You can print only small drawings (not full engineering drawings) and you cannot export files (for example for manufacture) except on a pay-per-use basis.

Delcam Designer

Delcam Designer is a companion for ArtCAM and is intended for users designing models from scratch, instead of working from imported models.

Delcam Designer does not include:

- modelling for manufacture commands (such as creation of draft curves and draft surfaces).
- flattening/unwrapping of curves and surfaces.

PowerMILL Modelling

PowerMILL Modelling provides easy-to-use surface modelling tools to prepare models for machining. Solids are not available in PowerMILL Modelling.

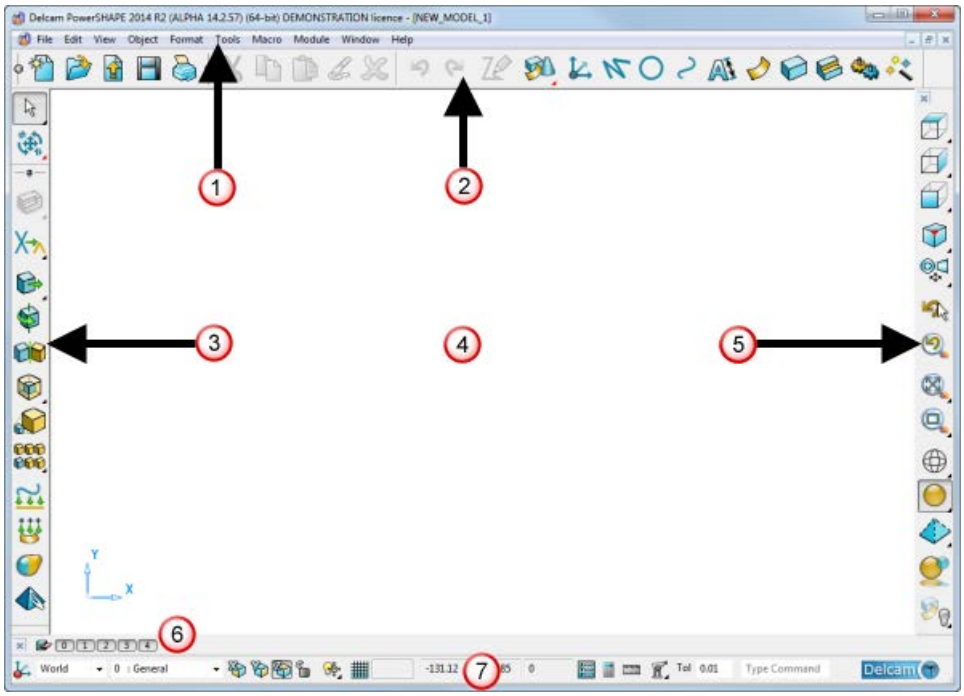
More about PowerSHAPE

You can find the latest information on **PowerSHAPE** from the Delcam website www.powershape.com.

User interface

Use the following sections to find information on the user interface:

Screen layout



PowerSHAPE uses a Windows-style interface with menus and toolbars. The areas of the screen are identified as follows:

1 Menu

2 Main toolbar

3 Sub-toolbars — When you select a button on the Main toolbar, in most cases an associated sub-toolbar is displayed.

4 Graphics area

5 Views toolbar

6 Layers toolbar

7 Status bar

Using PowerSHAPE menus

You can control most functionality in PowerSHAPE on the menus. These expand from the menu bar that is permanently displayed across the top of the software. In most cases, functionality that is available on the menus is also available from one of the toolbars.

Clicking on a menu option opens the selected menu (see page 12). This contains commands and further sub menus. Options are dimmed when not available.

An arrow ▸ icon shows that a menu option has a sub-menu.

Using the keyboard

- Navigate around the menus with the arrow keys to pre-select items.
- Press **Enter** to select an item.
- Press **Esc** to close the menus.
- Use keyboard shortcuts to open menus. Hold down the **Alt** key and press an appropriate underlined character for the menu. For example, **Alt+E** opens the **Edit** menu. Combinations of keys are indicated using the + key. For example **Alt+F+S** is the same as selecting **File > Save**.

PowerSHAPE menus

Use the following links for details of the tools and functions available from the menus.

File	Controls model file related tasks such as **Open**, **Save**, **Close**, and **Delete**.
Edit	Provides the **Undo/Redo** commands and tools for handling the clipboard such as **Cut**, **Copy**, and **Paste**. It also provides general editing tools applicable to most object types, such as **Mirror** and **Rotate**.

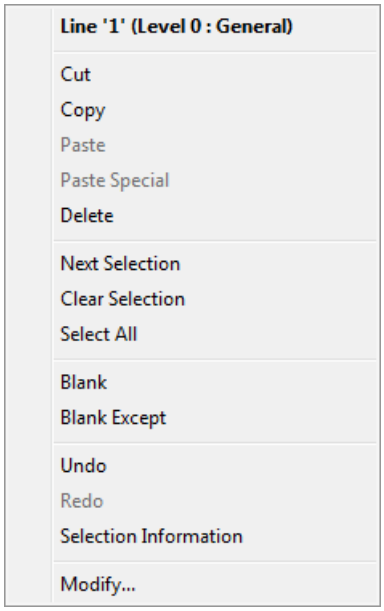
View	Controls how the models are displayed in the graphics windows, providing a range of pre-set views. Select **View Layout** to display the **View Edit** dialog, which enables you to configure the view. You can display toolbars using **View Toolbars**.
Object	Provides tools for creating all the available object types, such as **Workplane**, **Surface**, and **Line**. The menu option for each type has a submenu showing the available creation methods for each one.
Format	Provides options to change line types, colours, and to assign display levels. Use levels to reduce screen clutter on complex models.
Tools	Provides miscellaneous tools including model fixing and model analysis. You can also set your preferences on how the program operates using **Options**.
Macro	Provides the controls for creating a macro of your modelling actions. The macros can be played back using **Run**.
Module	Use this menu to view and open the modules that you can run while using PowerSHAPE.
Window	Lists all the open graphics windows and enables you to manage them.
Help	Enables you to access help and the Delcam websites, and check for updates. You can also view the software version and release details in the **About** dialog.
Context menus (see page 13)	Right-click an object to display a menu. This menu includes the commands appropriate for editing the object. Right clicking within a graphics window, but away from any objects displays a **View** menu.

Context menus

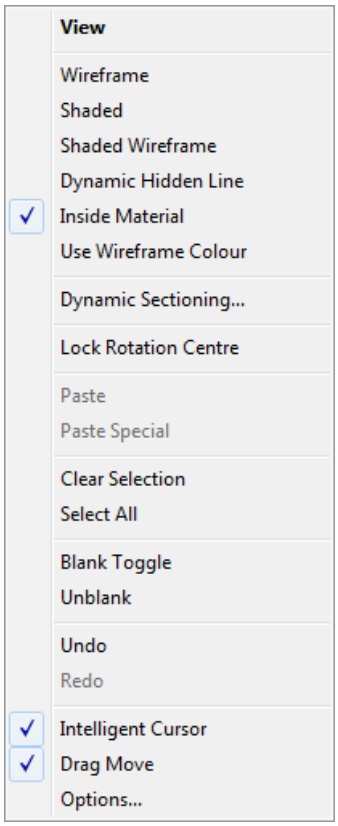
Display context menus by right-clicking in the graphics window.

The menu options displayed correspond with what you click on.

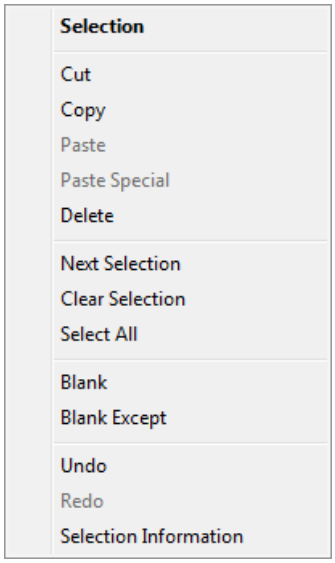
If you right-click an object, a menu displays the common edit options for that object. At the top of the menu, the type, name of the object, and the level are displayed.



Right-click in an empty area of the graphics window to display the **View** menu. This menu provides the shading options from the **View** menu and other common options.



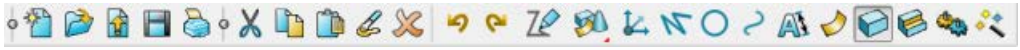
If a mixture of objects is selected and you right-click one of these selected objects, the **Selection** menu is displayed. This menu provides some common edit commands.



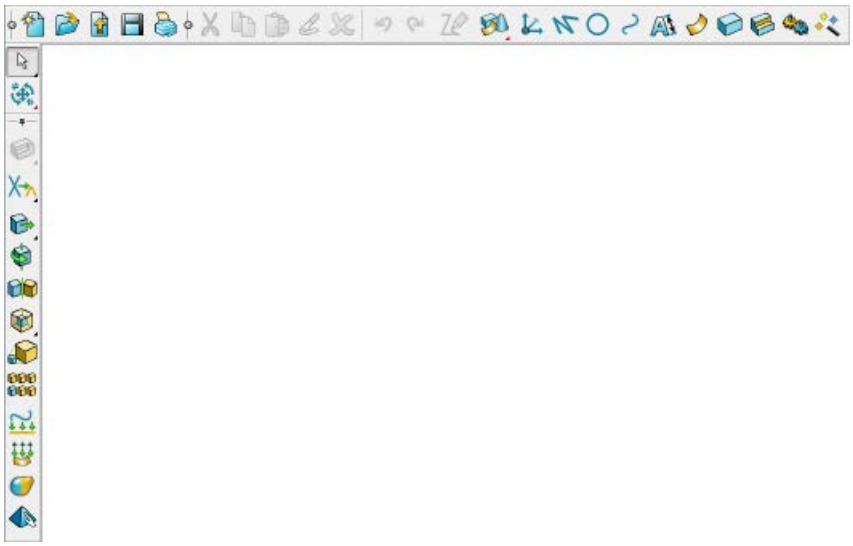
Toolbars overview

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 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**
- **Model Analysis**

- **Model Fixing**
- **Object Creation**

You can:

- Display or hide some toolbars according to the functions in use. For example the **Surface Edit** toolbar and the **Curve Edit** toolbar, shown in the image below, are displayed automatically when corresponding objects are selected.

- Expand some toolbars using the flyout (see page 29) button to display similar items or functions grouped together.
- Pin (see page 17) 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 19) 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 page of the **Options** dialog.
- Reverse the display of the (view and editing) toolbars using the **Toolbars** options page of the **Options** dialog.
- Remove some toolbars from the screen by clicking the Close button on the toolbar.

Using the toolbars

To use a toolbar:

1. Position the cursor over a button.
2. Left-click the button to activate the function.

The buttons on toolbars can include:

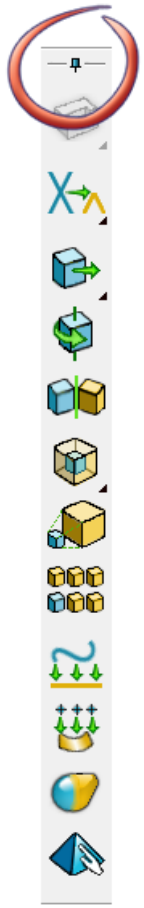
- an icon to represent the function
- a text description
- a shortcut to a Main menu item
- flyouts (see page 29) to secondary toolbars.

Some toolbars change according to the function selected, such as the **General Edit** toolbar. These have specific purposes and do not necessarily duplicate main menu options.

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.

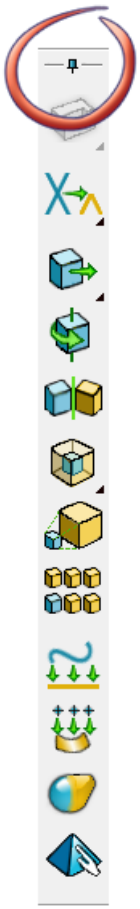
You can:

- Reposition a pinned toolbar in the graphics area (see page 18).
- Dock a pinned toolbar with another toolbar (see page 18).

Repositioning a pinned toolbar

To reposition a pinned toolbar in the graphics area:

1. Click the pin 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.

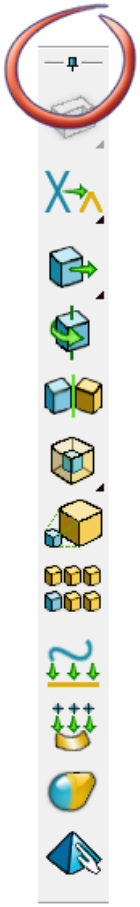
3. Click the pin again to return the toolbar automatically to its default location. The pin changes back to unpinned.

   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 21).
- You can add your own icons and you can customise buttons to be added to your toolbar using the Customise Picture dialog (see page 26).

Using your personal toolbars

Use the following sections to create and use your own toolbars:

Create a user-defined toolbar (see page 20)

Edit a user-defined toolbar (see page 23)

Display an existing user-defined toolbar (see page 23)

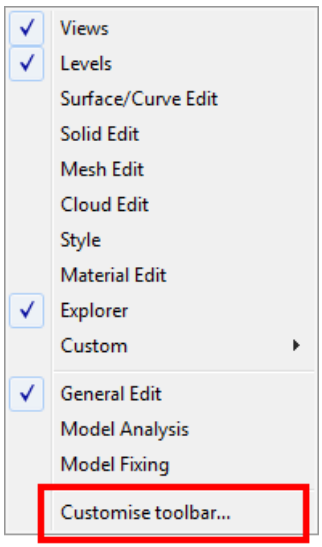
Customise buttons for user-defined toolbars (see page 23)

Sharing a user-defined toolbar and custom buttons (see page 28)

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 21) 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.**

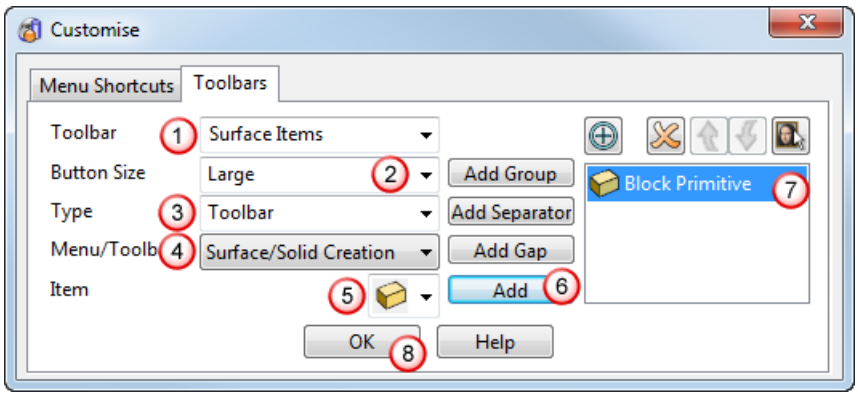
4 Select the **Menu/Toolbar** that contains the item you want to add to your toolbar, such as **Surface/Solid Creation**.

5 Select the **Item** to add, such as the **Block Primitive** button.

6 Click **Add** to add the block to your toolbar.

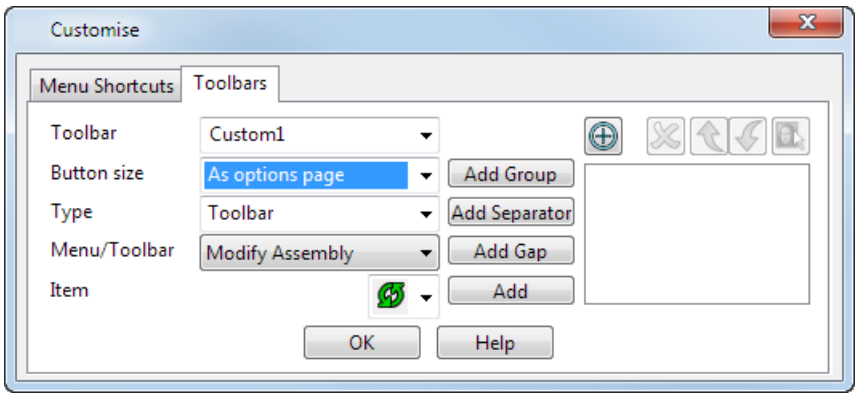
7 Added items, for example the **Block Primitive** button, are displayed in the added items list.

8 Click **OK**.



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.

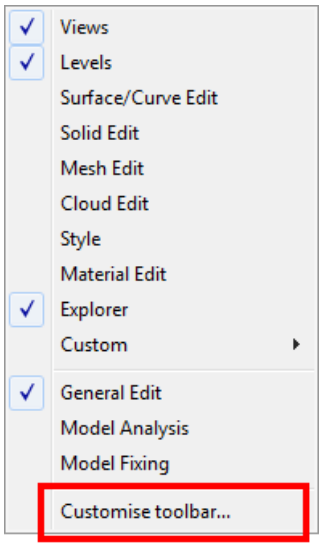
 **Move the item up the menu** — Click this button to move the selected item up the order of items on the toolbar.

 **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 25) is displayed.

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 21).
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.

Customising buttons for user-defined toolbars

When you are creating or editing a user-defined toolbar, you can edit and customise buttons. You can:

- customise an existing button on a user-defined toolbar (see page 24).
- create a new button to be added to the toolbar (see page 24).
- select a new button on the **Customise Button** dialog (see page 25).
- use the bitmap editor on the **Customise Picture** dialog (see page 26).

Customising a button on a user-defined toolbar

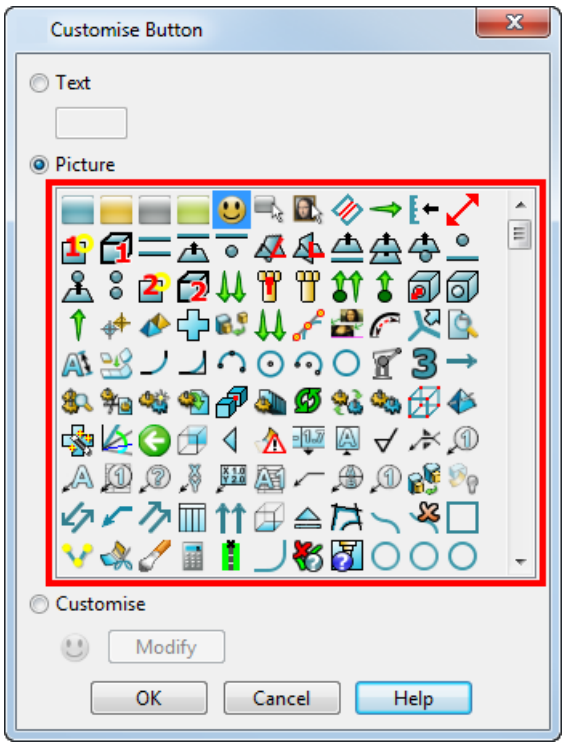
To customise a button on an existing user-defined toolbar:

1. On the **Customise** dialog (see page 21), select the button you want to customise in the *added* list.
2. Click the **Change Button** button to display the **Customise Button** dialog (see page 25).
3. Select **Customise**. The **Modify** option becomes available and a small thumbnail of the selected button is displayed.
4. Click **Modify**.
5. Use the bitmap editing functions on the **Customise Picture** dialog (see page 26) to edit the bitmap picture.

Creating a new button for a user-defined toolbar

To create a new button to be added to a user-defined toolbar:

1. Display the **Customise Button** dialog (see page 25).
2. Select the **Picture** option.
3. Select a picture or blank background.



4. Select the **Customise** option. The **Modify** option activates and a small thumbnail of the selected picture is displayed.
5. Click **Modify**.

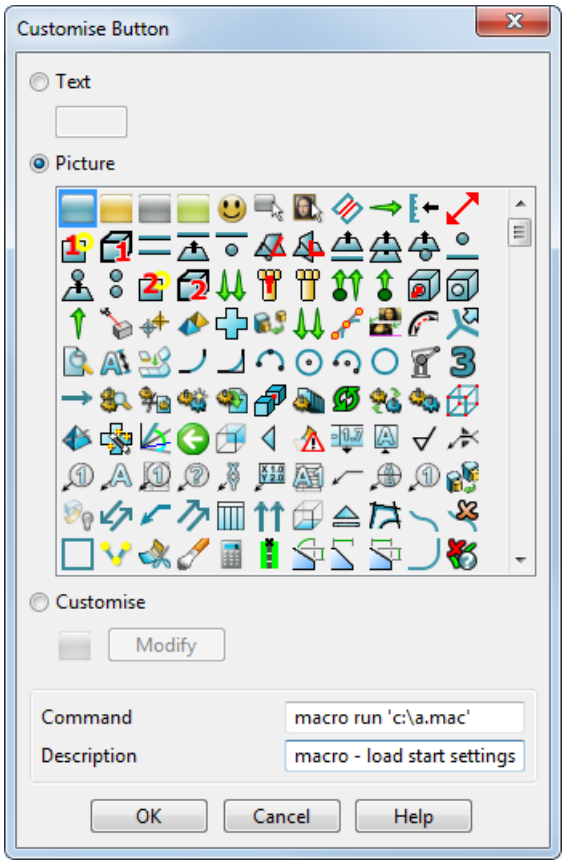
   The **Customise Picture** dialog (see page 26) is displayed.

6 Use the bitmap editing functions to edit or customise your selected button.

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 (see page 26)
- 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 (see page 26) to customise the picture before adding it to a user-defined toolbar.

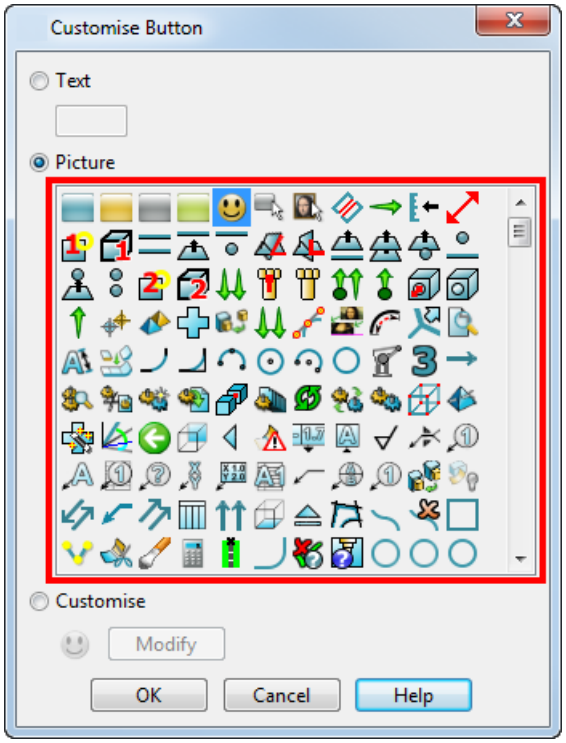
Command — The command is displayed and you can edit it.

Description — The description for the command is displayed and you can edit it.

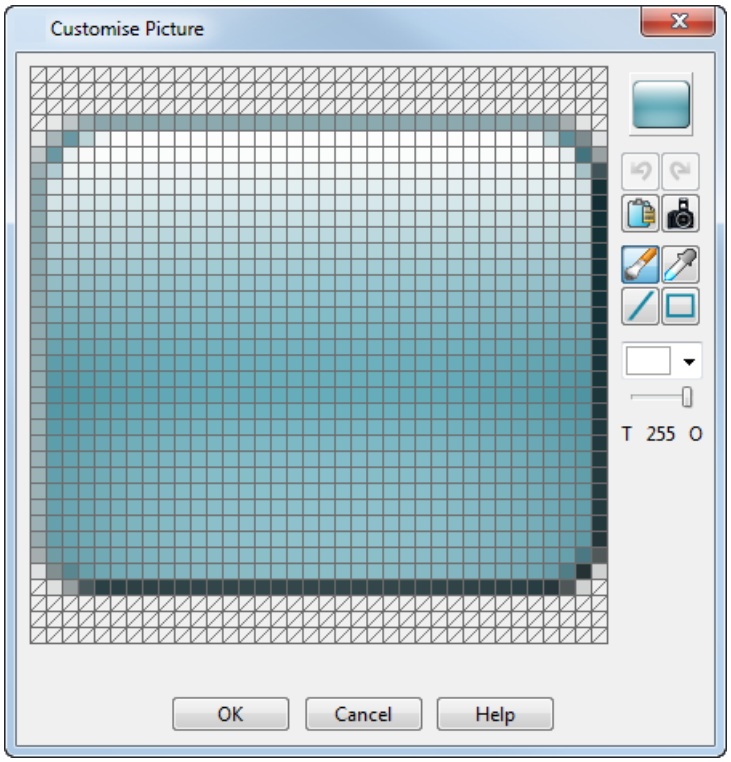
Customise Picture dialog

Use the bitmap editing functions on this dialog to customise the button selected on the **Customise Button** dialog (see page 25). You can:

- customise a button on an existing user-defined toolbar (see page 24).
- create a new button (see page 24) by selecting a picture or background button from those provided.



The selected button is displayed as a bitmap on a grid showing each pixel, ready for editing.



At the top right of the dialog there is a preview of the button, which is updated as you make changes.

Undo — Undo the last command.

Redo — Reverse the last **Undo** command.

Paste — Click this button to paste the clipboard contents. You can copy an image from an image editor (for example MS-Paint), copy the image to the clipboard and then paste it this button.

Thumbnail — Click this button to use a thumbnail of the current model.

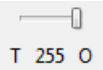
Brush — Select this option to change the colour of the pixels that you drag your mouse over to the selected **Colour** and **Transparency**.

Dropper — Select this option and click on the bitmap image to match the **Colour** setting to an existing colour.

Line — Select this tool to draw a straight line with your cursor. Set the **Colour** and **Transparency** of the line.

Box — Use this tool to draw a box. Set the **Colour** and **Transparency** of the box.

Colour — Use the colour chart to select a colour.



Transparency — Drag the slider between **T** (transparent) and **O** (opaque).

OK — Click this button to save the edited image.

Cancel — Click this button to close the dialog without saving your changes.

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.*

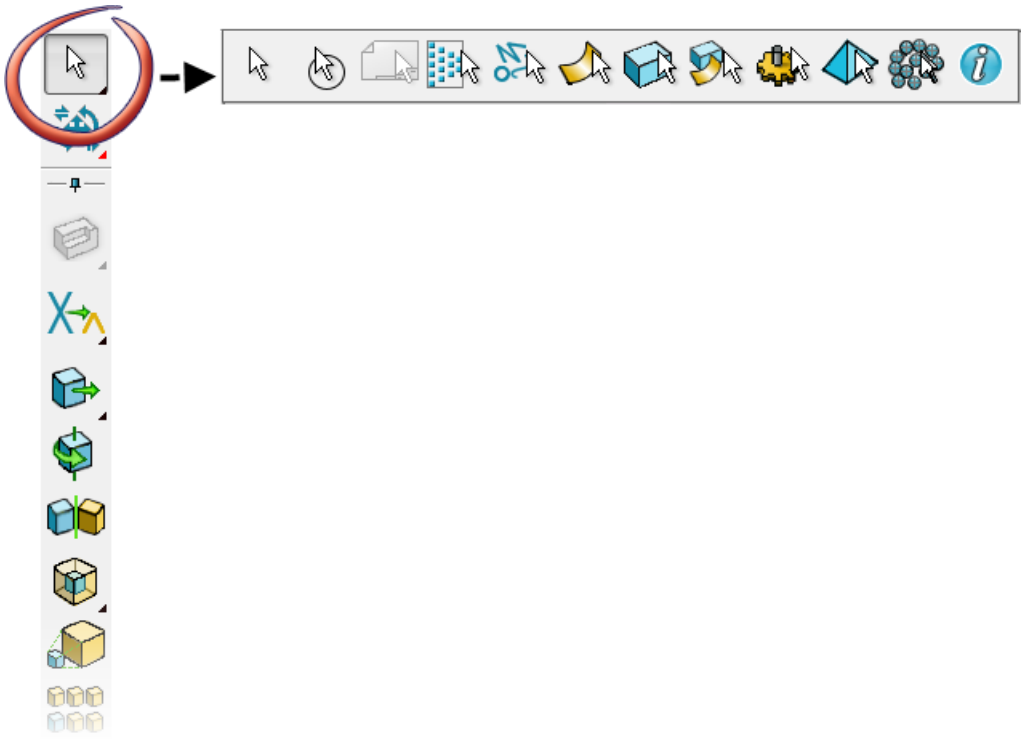
2. Select the required files. They include:

   - 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.

Flyouts overview

A flyout toolbar is an extended part of a toolbar (see page 15).



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.

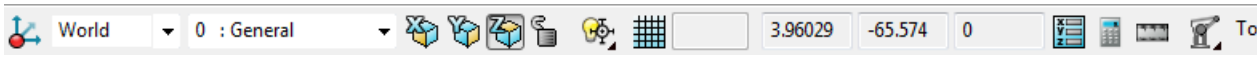
- Black icon — Flyouts with a black icon group similar buttons, for example the **Offset** flyout. Right-click on the flyout button to show the flyout toolbar and click the button you want. The most recently used button from the flyout toolbar is displayed as the flyout button.

You can display a flyout toolbar in one of these ways:

- Hover your cursor over the flyout button. The flyout toolbar is hidden when you move your cursor away.
- Right-click on the flyout button. The flyout toolbar is displayed until you click.

Status bar overview

The Status bar is displayed at the bottom of the screen.



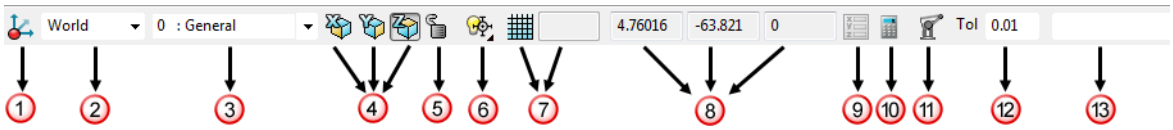
Use the Status bar to:

- Set the default drawing plane (see page 111).
- Set point-locking (see page 137).
- Display the Intelligent Cursor flyout (see page 56).
- Display the Grid (see page 136).
- Enter point co-ordinates (see page 116).
- Display the Position dialog (see page 119).
- Display the Calculator (see page 140).
- Display the Measure dialog (see page 160).
- Connect/Disconnect the Arm (see page 225).

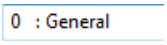
For further details, see Using the status bar (see page 30)

Using the status bar

The **Status bar** is fixed at the bottom of the screen.



Key	Object	Name	Description
1		**Temporary workplane**	Click this button to create or remove a temporary workplane. For further details see, Creating a temporary workplane in the Wireframe modelling manual.

2		**Workplane**	Use this list to activate and de-activate workplanes. You can also use it to change the name of the selected workplane. For further details, see Workplane list.
3		**Current Level**	This displays the level of the selected item or the level you are working on. Use this to change the level of the selected items. Only named levels that appear in the **Level** dialog are displayed in this list. For further details, see **View > Toolbars > Levels**
4		**Principal plane**	Select a button to set the principal plane of the current workspace. Each button displays the label of the axis that is normal to the principal plane. For example, the button with label **Z** represents the **XY** principal plane.
5		**Point lock**	Toggle this button to enable or disable point-locking when entering positions using the mouse. For further details, see Using point-locking (see page 137).
6		**Intelligent Cursor**	Right-click this flyout button to display the Intelligent Cursor flyout toolbar (see page 56).
7		**Grid on/off**	Toggle this button to enable or disable the grid. The cursor snaps to the grid lines Grid when positioning objects.

7		**Grid scale**	View and edit the scale of the grid (see page 136) when it is enabled.
8		**Coordinates**	The X, Y, and Z coordinates of the cursor are displayed respectively.
9		**Position**	When you are creating objects, click this button to display the **Position** dialog (see page 119), where there are various tools for entering positions.
10		**Calculator**	Click this button to open the **Calculator** (see page 139).
11		**Connect/Disconnect the Arm**	Use this flyout to connect or disconnect an arm (see page 225). To connect an arm you need an appropriate licence file and drivers installed on your computer.
12		**Tol**	The general tolerance variable is displayed. You can edit the value.
13		**Command input**	Use this to: ▪ enter data (see page 32). ▪ view previously entered data (see page 33). ▪ display the Command window (see page 33).

Using the Status bar to enter data

There are two ways to enter data.

- Click in the **Command input** box. Enter the data and press **Enter**.

- Move the cursor into the graphics window and enter the data. The data displays in the **Command input** box. Press **Enter** to enter the data.

Using the Status bar to view previously entered data

Click in the **Data entry box**.

1. Use the *up* and *down* arrow keys on the keyboard to display the previously entered data.
   - The *up* arrow key displays previously entered data upwards from the last entry.
   - The *down* arrow key displays previously entered data downwards from the first entry.
2. If you want to enter any of the data again, press **Enter** when it is displayed in the **Data entry box**.

Using the Status bar to display the command window

Move the cursor into the **Data entry box** and double-click in the box.

The **Command** window is displayed.

Dialog overview

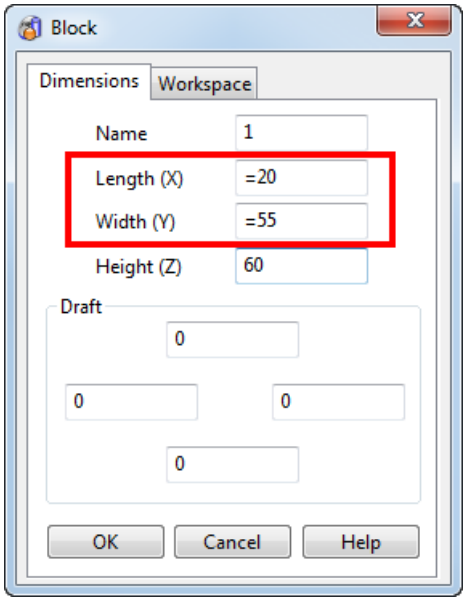
- A dialog is a movable box that is displayed when you need to provide information to the software. Dialogs may contain:
  - data entry boxes
  - expandable lists
  - buttons
- Most operations in a dialog are controlled by using the mouse or by using the menus. Some options and functions display other dialogs. For example, when you open a model a dialog is displayed for you to select a model from a list.
- Options that are followed by ..., display dialogs for you to make choices and enter details.
- Click **OK** to use the new information and close the box.

Text boxes and data boxes

Use these boxes to enter text or numbers. Click in the box, keeping the mouse cursor inside the box, then type the characters. The data is entered when you press *Enter*.

A value that is defined with a parameter or expression is indicated by the use of = in addition to the value.

For example, if you enter **Length** as **a=20** and **Width** values as **a+35**, the length and width values will be displayed in the dialog as **=20** and **=55** respectively.



Drop-down lists

Enter your data in one of the following ways:

- Click in the box and type in your data.
- Click the arrow  to expand the box showing any previous entries. You can then click on the one you want.

Option menus

These are displayed as drop-down lists that expand to show a menu list.

Click the arrow  to expand the list. Click on an option in the list to select it. The menu collapses to display the selected option.

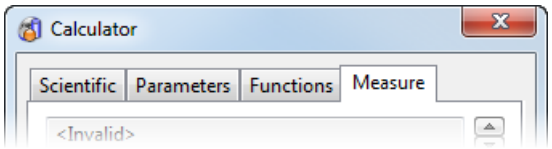
Value box

Increase or decrease the value by clicking the up or down arrow. You can also type a new value into the box.

Tabs

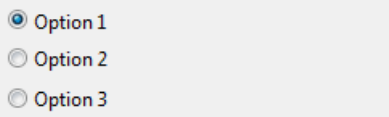
Tabs are used in dialogs to show a different work page for each tab.

The example below shows the tabs on the **Calculator** dialog.

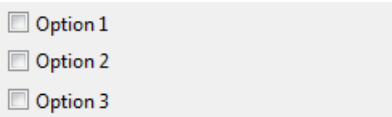


Option selection buttons

There are two types of option selection buttons.



These buttons are small and placed next to their options. When one button is clicked, a dot appears in the button and the other previously clicked button pops out. Only one option button can be *ON* at any time and one option button must always be selected. It is not possible to have them all *OFF*.



These options buttons are fully independent. A group of these can be any combination of *ON* and *OFF*. Click a button to set *ON* and click again to set *OFF*.

Buttons

Each button displays a unique image or text to represent a command. When clicked, the button appears to press-in and the associated command is activated. When the command is not available, the icon image or text appears grey and the button will not press-in when clicked.

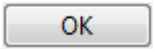
Apply button

Click to make the changes required, but to leave the dialog displayed.

When a dialog is displayed, you can click the right-mouse-button in the graphics area as a shortcut for **Apply** or **Accept**.

__Apply__ has precedence over __Accept__, so when a dialog has both __Apply__ and __Accept__ buttons available, clicking the right mouse button is the same as clicking __Apply__.

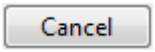
OK button



Click to make the changes required and close the dialog.

Pressing __Enter__ is the same as clicking __OK__ on a dialog.

Cancel button



This button closes the dialog, discarding any edits carried out on the model while it was displayed.

Using the mouse

The mouse is the main device for controlling the software. You can click buttons or objects with or without keyboard modifier keys (**Alt**, **Shift**, or **Ctrl**) to alter the way you can create, select, move, edit, or view models on the screen. The following terms are used in the documentation:

- **Click** — Press and release the left mouse button (see page 39).
- **Double-click** — Rapidly click the left mouse button twice.
- **Drag** — Hold down the left mouse button and move the mouse while holding the button down.
- **Move** — Move the mouse without pressing any of the mouse buttons.

When other mouse buttons are to be used the required button is specified; for example

- **Middle-click**
- **Middle-drag**
- **Right-click**

The mouse controls such as *double-click speed* and *hit radius* are set to default values, but can be specified on the **Mouse** page of the **Options** dialog.

The cursor that is displayed when you use the mouse indicates the operation that you are doing.

Context-sensitive cursors

The cursor that is displayed when you use the mouse reflects the operation that is being carried out. Context-sensitive cursors are available in the following areas:

Context sensitive cursors — Annotation toolbar

 — Add text along a curve.

 — Add text (horizontally or vertically).

Context sensitive cursors — Curve toolbar

 — Stop curve.

 — Close curve.

 — Sketch curve.

 — Select start and end point (composite curve).

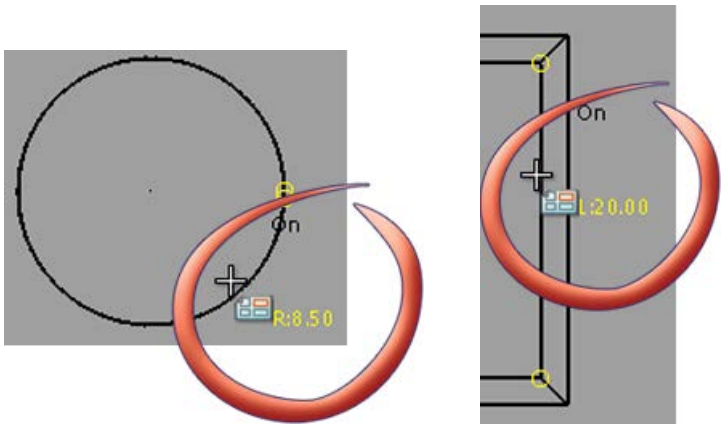
Context sensitive cursors — Drafting

 — Activate view.

 — View selection.

 — Drawing selection.

Arc and line lengths are displayed on the cursor when creating dimensions.



Context sensitive cursors — General edits

— Limit cut.

— Limit point.

— Move/copy/stretch (also used for drag-move).

— Rotate items.

— Mirror items.

— Mirror items across a user defined plane.

— Mirror items in a wireframe line.

— Offset items.

— Scale items.

Context sensitive cursors — View toolbar

— Zoom to box.

— Zoom.

— Rotate.

— Pan.

Context sensitive cursors — Other

— Warning/error.

 — Cursor selection/edge-pick hand.

 — Add component.

 — Principal plane-lock toggled.

 — Action not allowed.

 — Action not allowed, plane.

 — Action not allowed, arc.

 — Paste attributes.

 — Selection filter.

Using the left mouse button

This is the main mouse button for selecting menu options, clicking buttons on toolbars and dialogs, and for editing model objects. It has different effects according to where the cursor is positioned.

You can use the left mouse button:

- over a menu (see page 39)
- over a non-active graphics window (see page 40)
- over an active graphics window (see page 40)
- on an object (see page 44)
- in a dialog (see page 41)
- after a Cut or Copy (see page 48).

Over a menu

	**Single Click** highlights and selects menu options.
	**Double Click** has no particular effect over menus. PowerSHAPE assumes you intended to single click.

	**Drag** by holding down the left mouse button and moving the mouse to move the cursor. You can: ▪ Drag the cursor over objects in the graphics window to highlight them for selection. ▪ Edit and create dimensions by dragging them to the required position. ▪ Edit objects by dragging their handles.
	**Move** the cursor over the different options, highlighting the ones that are available. You can: ▪ Move the cursor over menus to display submenus where they are available. ▪ Move the cursor over toolbars to display flyouts where they are available. ▪ Activate construction lines when the intelligent cursor is active. ▪ Move the cursor along construction lines.

Over a non-active graphics window

Single-click in a non-active graphics window activates the window. The title of the active window is highlighted.

Over an active graphics window

Single Click in an active graphics window usually signifies an attempt to select an object or to enter a position. Click away from any object to clear the current selection.

When entering positions, click away from any object to input a position on the principal plane of the active workplane.

Double-click has no effect unless you are pointing at an object.

Drag (hold down the **Left** mouse button while dragging) to draw a rubber band box. Objects within the box are selected, indicated by their yellow highlight colour and handles.

Note:

- Any other objects already selected and not within the box become deselected.
- If the box contains no objects, the current selection is cleared.
- When more than one object is selected, the graphical handles are not displayed.

Shift-Drag (hold down the **Shift** key while dragging) draws a rubber band box. Objects within the box are added to the current selection.

Ctrl-Drag (hold down the **Ctrl** key while dragging) draws a rubber band box. Objects within the box are added to the current selection or deselected if they are already selected.

Ctrl-Shift-Drag (hold down both the **Ctrl** and **Shift** keys while dragging) draws a rubber band box. Objects within the box are deselected from the current selection.

Shift-Alt-Drag (hold down the **Shift** and **Alt** keys while dragging) rotates the view around the X axis.

In a dialog

Single click to select options and press buttons.

On slider controls, clicks on the track cause the slider to step towards the cursor.

Double-click is not effective on all dialogs.

Double-click in a text box to select all the text. If you then type new text, the selected text is replaced by the new text.

Double-click on an item in a list to select the item and accept the dialog. For example, double-clicking a file name in the **Open Model** dialog, accepts the dialog and opens the model.

Shift-click (press and hold the **Shift** key while clicking) to select multiple items in a list. All items between the last item selected and the current item clicked become selected.

Drag slider controls to move the slider button.

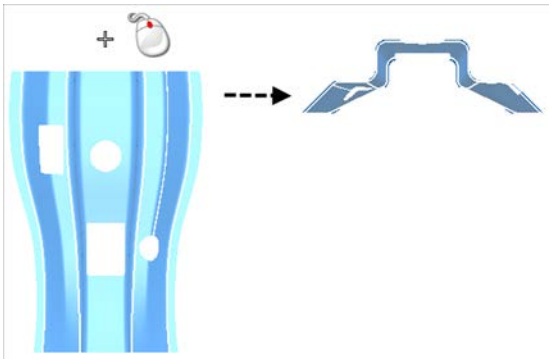
Using the middle mouse button

Use this mouse button with keyboard “modifier” keys to control the view of your model. It is not used within the dialog and does not alter a model’s design.

If you have a two button mouse, operations using the middle mouse button are available using the other two buttons. For further details, see Right-hand mouse button (see page 46) and Left and right mouse buttons (see page 48).

Middle-click to **Rotate** the view, hold down the middle button and move the mouse. The model turns as if the cursor is pushing and pulling on the outer surface of a ball and a ball graphic appears over the centre of rotation to show what is happening.

Middle-click to change **Orientation**. Position the cursor to the side of the model and click the middle mouse button. The model will be rotated so that you are viewing the model from the position of the cursor.

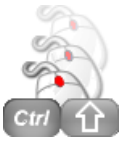


Shift+Middle-click to **Pan** a particular point to the centre of the view, hold down the **Shift** key and click the point in the model you want to appear in the middle of the view.

Ctrl+Middle-click to **Pan** a particular point to the centre of the view, hold down the **Control** key and click the point in the model you want to appear in the middle of the view.

Shift+Middle-click+drag to **Pan** the view, hold down the **Shift** key and drag the middle button. The view follows the mouse movements.

Ctrl+Middle-click+drag to **Zoom** the view, hold down the **Control** key and drag the middle button. Moving up the screen magnifies the view. Moving down the screen reduces the view



Ctrl+Shift+Middle-click+Drag to **Zoom to a box,** hold down the **Shift** and **Ctrl** keys and drag the middle button. A rubber-band box appears. When you release the mouse button, the view zooms to the box size.

Shift+Alt+Middle-click+drag (holding down both the **Shift** and **Alt** keys while dragging) to rotate the view around the Y axis.

On an object

When selected, objects are displayed with additional edit handles. For example:

- Lines have key points at each end.
- Curves have key points on arms extending from the selected control point on the curve.
- Arcs have key points at the centre and at the start and end of the arc sector.
- Workplanes have a “control rectangle” and origin.

If you drag a key point, only the aspect of the object controlled by the point will move.

To move the whole object, be careful to click on the object itself, but away from any of its *key points*.

Single-click over an unselected object to select it. The object turns yellow with its key points shown as graphical “handles”. The selected object’s key point closest to the cursor when selected becomes its anchor point.

Any other objects already selected become deselected.

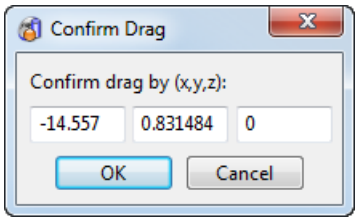
If the Intelligent Cursor is *ON*, the cursor changes as you move over objects and current selections.

Double-click to select the object, but also to send the command to display a dialog or toolbar allowing you to edit the object’s properties. For further details, see Modifying an object in the Menus and Toolbars manual.

When entering a series of points (for example, for a series of lines or a curve), double click to finish the series. A subsequent single click starts a new series.

Drag a selected object to move it. While the button is pressed, the object follows the mouse. When you release the button, the **Confirm Drag** dialog is displayed for you to confirm the **x**, **y**, **z** coordinates for the move. Although the object appears to continue to move with the cursor, the coordinates are fixed where you released the mouse button.

Click **OK** to confirm the coordinate

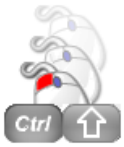


The move is relative to the active workplane.

Shift+click (hold down the **Shift** key while clicking) to add objects to the current selection.

Ctrl+click (hold down the **Ctrl** key while clicking) to add objects to the current selection or deselect those already selected.

Alt+click (hold down the **Alt** key while clicking) to create composite curves. For further details, see Creating a composite curve by tracing in the Wireframe modelling manual.

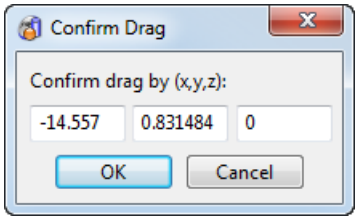


Ctrl+Shift+click (hold down both the **Ctrl** and **Shift** keys while clicking) to deselect objects from the current selection.

Ctrl+Drag (hold down the **Ctrl** key while dragging) to copy the selected objects. The copy is dropped when you release the mouse button.

This copy does not overwrite data already on the Cut/Copy clipboard. The **Confirm Drag** dialog is displayed where you confirm the **x**, **y**, **z** coordinates for the move.

The move is relative to the active workplane.



Using the right mouse button

You can use the right mouse button:

- Over an object (see page 47)
- In a text box (see page 47)
- Over an active graphics window (see page 47)
- When a dialog is displayed, you can click the right-mouse button in the graphics area as a shortcut for **Apply** or **Accept**. If **Apply** and **Accept** are unavailable, clciking the tigh mousse button will close the dialog without making any changes.

***Apply** has precedence over **Accept**, so when a dialog has both **Apply** and **Accept** buttons available, clicking the right mouse button is the same as clicking **Apply**.*

Click with this mouse button to display context menus and the calculator. You can also use this button to pan, zoom and drag the view.

Over an object

Single right click in the active model window to display context menus. These contain various options including **Cut, Copy, Paste** and **Delete**. Click over different object types or a blank space to display options on the context menu that are relevant to the object type. For further details, see context menus (see page 13).

In a text box

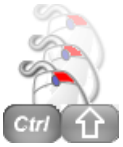
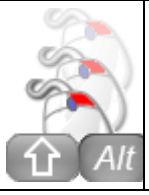
Single right-click in a numeric text box to display the **calculator**. The calculation results are displayed automatically in the text box when you press the *Enter* key or move the mouse outside the numeric text box.

In a text box, this displays a context menu with options **Undo, Cut, Copy, Paste, Delete** and **Select All**. You can use these options on the data in the box.

Shift+right-click in a numeric text box to display a context menu with options **Undo**, **Cut**, **Copy**, **Paste**, **Delete**, and **Select All**. You can use these options on the data in the box.

Over an active graphics window

	**Shift+right-click** to **Pan** a particular point to the centre of the view, hold down the **Shift** key and click the point in the model you want to appear in the middle of the view.
	**Ctrl+right-click** to **Pan** a particular point to the centre of the view, hold down the **Control** key and click the point in the model you want to appear in the middle of the view.
	**Shift+right-click+Drag** to **Pan** the view, hold down the **Shift** key and drag the right button. The view follows the mouse movements.

	**Ctrl+right-click+Drag** to **Zoom** the view, hold down the **Ctrl** key and drag the right button. Moving up the screen magnifies the view. Moving down the screen reduces the view
	**Alt+right-click-Drag** to **Zoom to a box,** hold down the **Alt** key and drag the right button. A rubber-band box appears. When you release the mouse button, the view zooms to the box size.
	**Ctrl+Shift+right-click+Drag** (hold down both the **Ctrl** and **Shift** keys while dragging the right mouse) to rotate the view. The model turns as if the cursor is pushing and pulling on the outer surface of a ball and a ball graphic appears over the centre of rotation to show what is happening.
	**Shift+Alt+right-click+Drag** (holding down the **Shift** and **Alt** keys while dragging) to rotate the view around the Z axis.

After a Cut or Copy operation

Ctrl+Alt+Click (hold down the **Ctrl** and **Alt** keys while clicking) to enter an anchor point for **Paste Special**.

Left and right mouse buttons

Shift+Alt+Drag (hold down the **Shift** and **Alt** keys while dragging) to rotate the view around the Y axis.

Left-click+right-click to rotate the view, hold down both the left and right buttons and move the mouse. The model turns as if the cursor is pushing and pulling on the outer surface of a ball and a ball graphic appears over the centre of rotation to show what is happening.

Using the keyboard

The keyboard is usually used to enter information into a dialog. This might be when you want to create or edit an object to a specific size and position.

Certain "modifier" keys are also used with the mouse to change the way the mouse behaves. For further details, see Using the mouse (see page 36).

There are also a number of keyboard shortcuts that allow you to do a variety of operations such as open and close models. For further details, see Shortcuts to menu options.

Graphics area

PowerSHAPE runs within its own window containing:

- Menus (see page 12)
- Toolbars (see page 15)
- Status bar (see page 30)
- Graphics area (see page 49)

For further details, see Screen layout (see page 11) and The coordinate system (see page 111)

The graphics area may be:

- a single modelling window that fills the graphics area.
- several modelling windows that overlay each other in the graphics area. These windows can be placed anywhere on the screen, not necessarily within the graphics area. For further details, see Window menu overview.

  Each modelling window displays views of a single model. You can open as many modelling windows for each model as you like, but closing a model’s last window also closes the model.

Modelling windows in the graphics area

Use the Window menu to control the content displayed in the graphics area:

- Create new modelling windows.
- Swap between modelling windows.
- Make a different modelling window active. To do this click on the window or select it from the menu. Only one modelling window is active at any time and this is indicated by the window's highlighted title.

- Change the display of modelling windows.

Use the View menu to:

- Change the view shown in modelling windows.
- Increase the visible modelling area by selecting Full Screen or Full Screen Preview.

Displaying modelling windows

- Use Single window mode (see page 50) to display a single working modelling window that fills the whole area.
- Use Multi-window mode (see page 51) to display several overlapping modelling windows.
- Display several tiled model windows that are tiled vertically or tiled horizontally.

Closing modelling windows

Click  at the top corner of the window.

Resizing modelling windows

Windows have resize handles at the corners and borders. Drag them to change the window size. For more information, see your operating system documentation.

Full screen mode

Use the following typed commands to make PowerSHAPE stretch to fill the screen:

GUI FULLSCREEN ON/OFF removes the taskbar and puts menu off the top of the screen.

GUI PREVIEWSCREEN ON/OFF removes the taskbar, puts menu off the top of the screen.and removes all toolbars.

The menus can still be accessed off the top of the screen.

Single window mode

Click **Windows fix/float**  to fill the graphics area (see page 49) with the active modelling window. To display a different modelling window, select it from the list on the **Window** menu.

For further details, see Window menu overview.

Multi-window mode

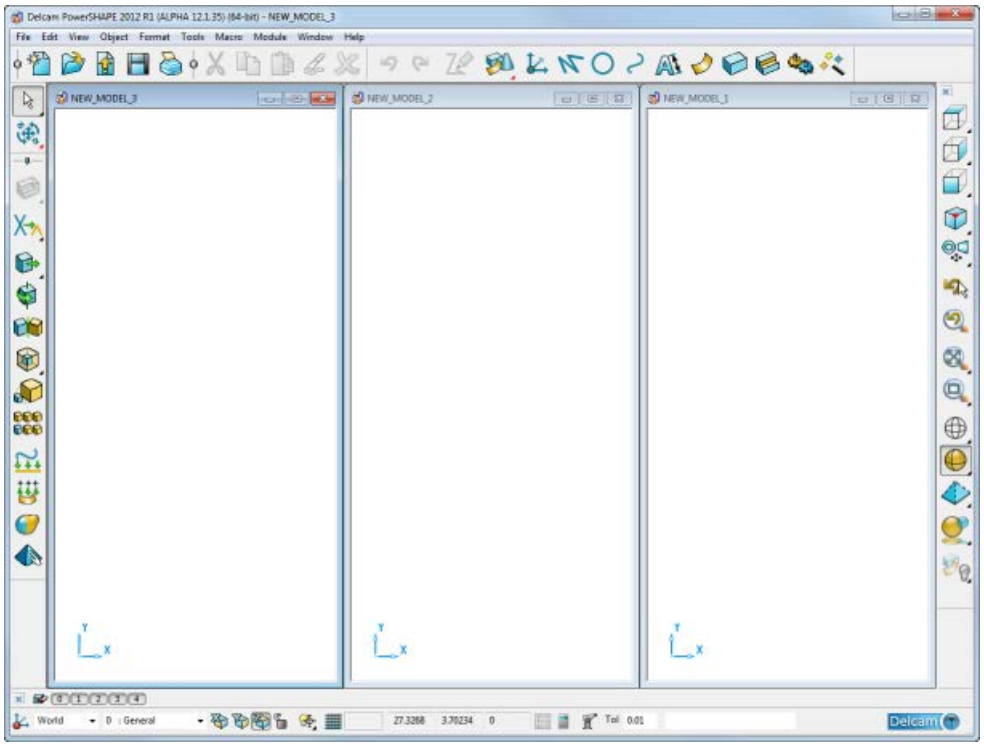
When the modelling window fills the graphics area, click **Windows fix/float** to display each window in their own frames. These modelling windows are moveable and resizable. Each has a banner across the top showing the window name.

Click on the banner (or the border) of a window to bring the window to the front of other windows. Drag the banner to move the window around the screen.

For further details, see Window menu overview in the Menus and Toolbars manual

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.

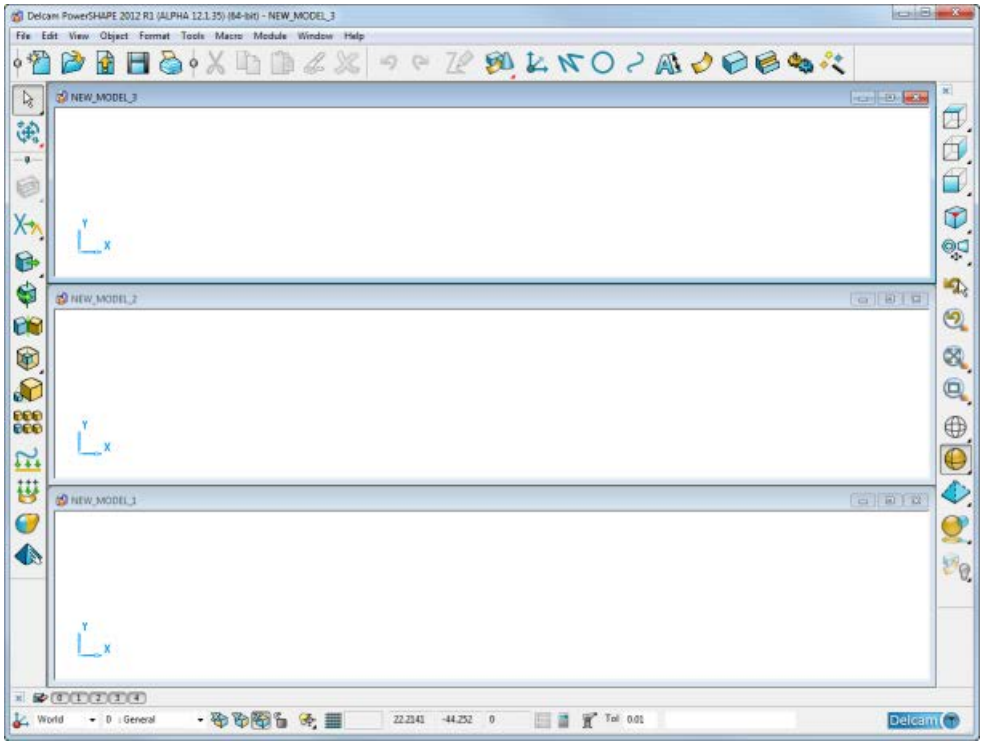


You can resize and move tiled modelling windows.

Click the **Maximise** button on a tiled window to revert to a single window display.

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.

Intelligent Cursor

Use the following sections for information on using the Intelligent Cursor:

What is the Intelligent Cursor?

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

- **Point input** — It activates construction lines and labels for dragging, snapping and for point creation.
- **Selection** — It highlights the object under the cursor ready for selection. For further details, see Selection using the Intelligent Cursor (see page 60)

  The cursor icon changes to assist you when selecting and editing objects. For further details, see Intelligent Cursor construction labels (see page 59)

Using the Intelligent Cursor

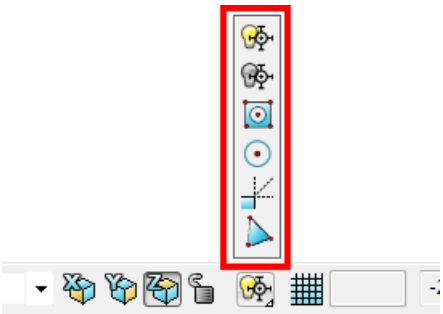
The Intelligent Cursor is enabled by default. You can enable and disable the Intelligent Cursor in one of these ways:

- Turning the Intelligent Cursor off or on (see page 53)
- Context menu (see page 54)
- The Mouse page of the Options dialog (see page 54)

Turning the Intelligent cursor on or off

The Intelligent Cursor is enabled by default. You can enable and disable the Intelligent Cursor using the options on the Intelligent Cursor flyout (see page 56) on the status bar.

1. Right-click the Intelligent Cursor button on the status bar to display the **Intelligent Cursor** flyout.



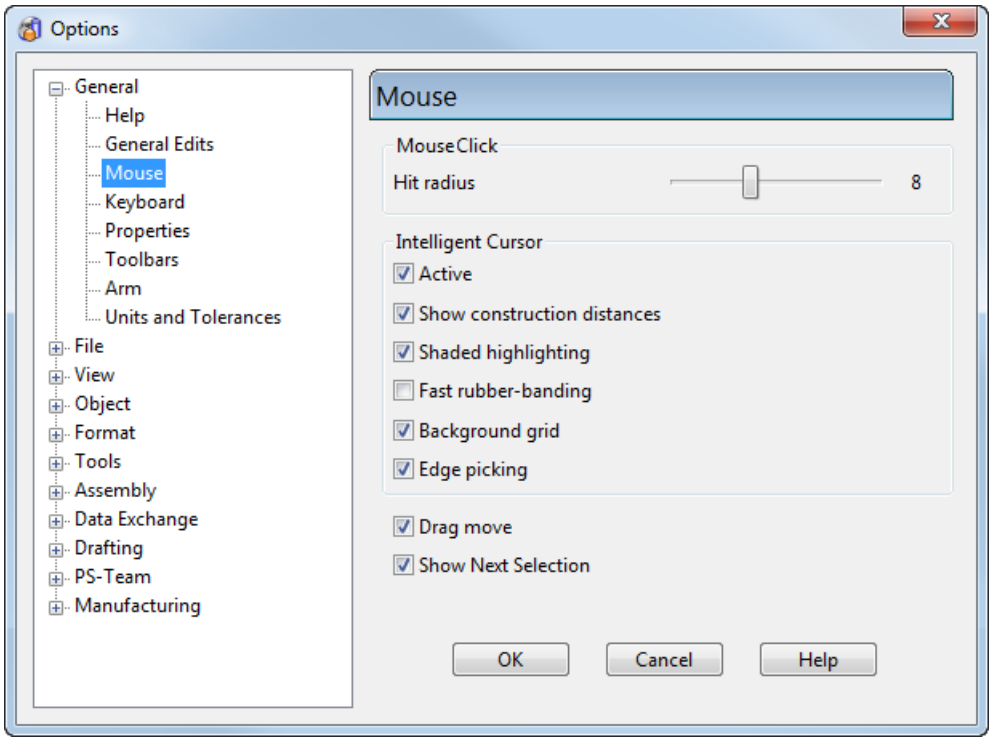
2. Click the **Intelligent Cursor Off** button to disable the Intelligent Cursor.
3. Click the **Intelligent Cursor On** button to enable the Intelligent Cursor.

View context menu

1. In modelling mode, right-click in an empty space in the graphics window to display the View menu (see page 13).
2. Deselect the **Intelligent Cursor** option to disable it.
3. Select it again to enable the Intelligent Cursor.

Mouse page of the Options dialog

Use the **Active** option to toggle the **Intelligent Cursor**. This option is selected by default.



The **Intelligent Cursor** displays in the graphics window:

- various cursor types
- construction lines
- construction labels
- construction distances

When you click **OK**, PowerSHAPE uses the new Intelligent Cursor settings. For further details on Intelligent Cursor options, see Mouse options.

Construction lines

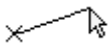
When you hover the Intelligent Cursor over an important key point (one that displays any of the construction labels (see page 59)), construction lines are created from that key point. You can now drag the cursor along a construction line and see the appropriate labels display. Clicking causes the position to snap to that point.

The following example draws a box:

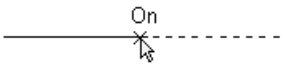
1 On the Main toolbar, select the **Line** button.

2 On the Line toolbar, select the **Continuous** button.

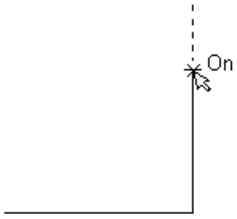
3 Click to start a line.



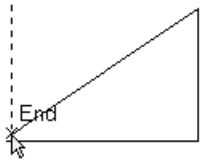
Drag along a construction line and click.



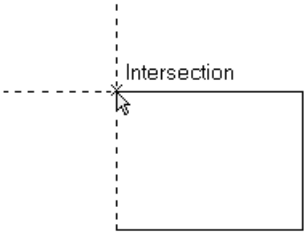
4 Drag along a 90° construction line and click. The line snaps to give a square intersection.



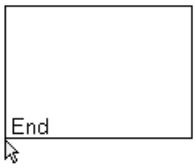
5 Drag the cursor down to the original start point and hover. The cursor displays **End** and the construction lines display.



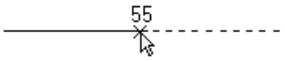
6 Drag along the vertical construction line until **Intersection** is shown.



7 Snap to the start and finish with one more click when **End** is displayed.



If the Intelligent Cursor Gridding is enabled, the label **On** in the above diagrams is replaced by a distance from the last point selected. For example:



This provides a convenient way to enter accurate dimensions.

*To disable construction lines temporarily, press and hold the **Shift** key.*

Turning construction lines on or off

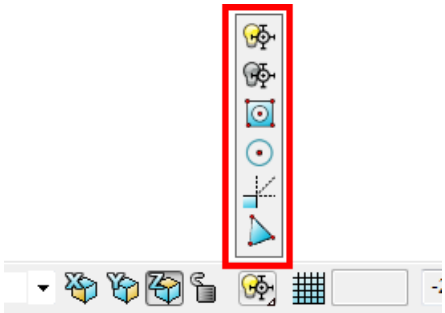
The Intelligent Cursor's construction lines are enabled by default. To disable construction lines, click the Construction Lines button from the Intelligent Cursor flyout (see page 56) on the Status bar. The construction lines are not displayed until they are turned back on. All functionality is still available.

Intelligent Cursor flyout

1 Click (*Status bar*) using the right mouse button to display the **Intelligent Cursor** flyout.

2 Select the Intelligent Cursor option you require.

The Intelligent Cursor options are for point entry only (not selection).



Intelligent Cursor on (default) (see page 53).

Intelligent Cursor off (see page 53).

 Snap to keypoints (see page 57).

 Snap to centre keypoints (see page 57).

 Construction lines on/off (see page 56).

 Click this button to snap the cursor to the nearest node instead of the exact point on the mesh.

3 Click  (*Status bar*) to return to the default flyout setting when you have finished using the Intelligent Cursor options.

Snap to keypoints

Use **Snap to keypoints** to detect only keypoints in your model when entering points. Anything that is not a keypoint is ignored by the functionality in use. On wireframe items the cursor will snap to the nearest keypoint.

When this option is selected, the cursor changes to  as a reminder that point entry is restricted to keypoints.

To activate **Snap to keypoints**:

1 Click  (*Status bar*) using the right mouse button to display the Intelligent Cursor flyout (see page 56).

2 Click  **Snap to keypoints**.

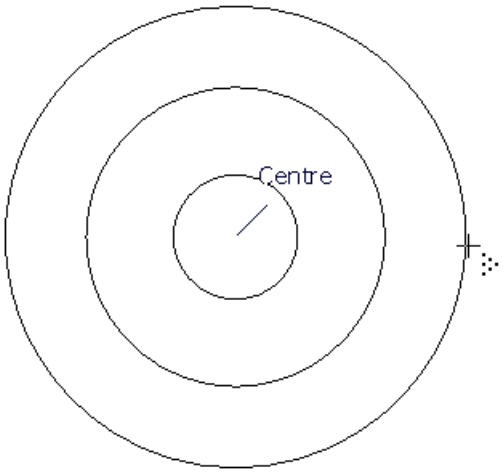
Snap to centre keypoints

Snap to centre keypoints is a restrictive option that filters out all point input except circle and hole centres. This is particularly useful when dimensioning a large number of holes.

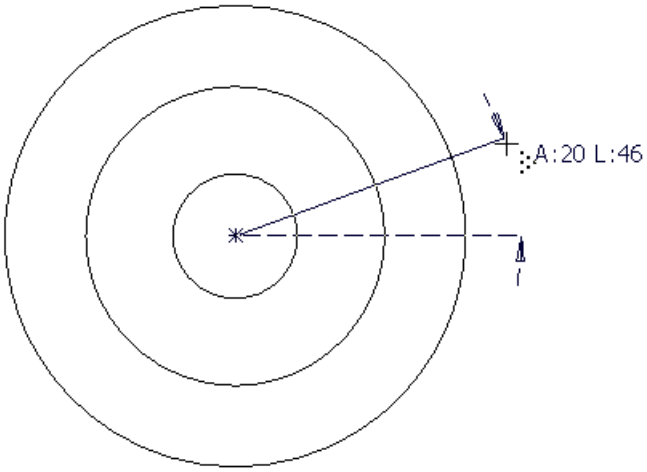
When the option is selected, the cursor changes to  as a reminder that point entry is restricted to centre keypoints.

Any point on the geometry of circles will snap to the centre of that arc. For example,

1 Hover the cursor over an arc



2 Click to snap the centre of that arc.



It is not necessary to select the centre of the arc directly.

This is a useful technique when working with overlapping or concentric circles where the centre is not easily identifiable.

To activate **Snap to centre keypoints:**

1 Click (status bar) using the right mouse button to display the Intelligent Cursor flyout (see page 57).

2 Click to activate **Snap to centre keypoints**.

Entering positions using the Intelligent Cursor

The Intelligent Cursor shows additional information to help you to define the position you want when entering positions.

When you move the cursor over an object, construction labels (see page 59) are displayed by the Intelligent Cursor. These construction labels:

- are shown and hidden as you move the cursor over them.
- indicate what happens if you click while the label is displayed.
- make snapping with the cursor more accurate. The labels display when key points are within snap range of key positions in the model. You can click when a label (see page 59) is displayed to snap to the cursor at the labelled position.

The Intelligent Cursor (see page 53) is enabled by default.

Intelligent Cursor construction labels

The labels you may see displayed by the Intelligent Cursor include:

- **End** — Shows that the position will snap to the end point of a line, arc, curve, or profile.
- **Mid-point** — Shows that the position will snap to a point mid way between the start and end points of a line or arc.
- **Centre** — Shows that the position will snap to the centre of an arc or circle, or hole feature. The name of the feature is also displayed.
- **Key-point** — Shows the keypoints on curves and surfaces where they can be manipulated. The position will snap to the nearest keypoint on the object under the cursor (for example, a surface patch corner or a curve key point). This label can also show the origin of a workplane.
- **On** — Shows that the cursor is on a line and the position will snap to a point that lies exactly on the object, but not at one of its key points (for example, end, corner, centre points).

If the **Background Grid** option is *ON*, a number representing a distance along the object from the last key point visited by the cursor is displayed. The values are rounded to the nearest sensible value. Zoom out to see coarser increments. Zoom in to see finer increments.

- **In**
  Shows that the position will snap to a point on the inside of a surface patch or a solid face (not one of the keypoints or positions along a curve).
- **Intersection**
  Shows where two objects cross and that the position will snap to the intersection between the two objects under the cursor.

- **Tangent**
  Displays when a tangent to an arc can be made. The position will be calculated so that the line or arc being created is tangential to the line or arc under the cursor. The position will not necessarily be under the cursor. This works whether you are placing a line to an arc, an arc to a line or an arc to an arc.
- **End** sign 

  This displays when clicking will complete a curve or continuous series of lines.
- **Closing a curve** 

  This displays when clicking will create a closed curve.

*We strongly recommend that you normally work with the **Construction Level** switched OFF. For further details, see Format > Levels.*

Selection using the Intelligent Cursor

When the Intelligent Cursor is enabled, objects are highlighted as you move the cursor over them. This shows which objects can be selected if you click at that point. For further details, see Turning the Intelligent cursor on or off (see page 53).

When several objects are under the cursor and one of them is selected, the Intelligent Cursor highlights the next object from the database at that position. A click now deselects the selected object, selects the highlighted object, and highlights the next one. This enables you to select the correct object from the group.

You can also right-click to display the object's menu and click **Next Selection** to select the next object.

The Intelligent Cursor option is displayed on the context menu. Select or Deselect to toggle the Intelligent Cursor on and off. Selecting this option is the same as selecting the Intelligent Cursor **Active** option from the Mouse options page of the **Options** dialog.

Using a locked axis with the Intelligent Cursor

The Intelligent Cursor can be an effective tool when used with a locked axis.

- Intelligent Cursor modification keys (see page 61)
- Creation using a locked axis (see page 61)
- Dragging on a locked axis (see page 62)
- Moving points on a locked axis (see page 62)

- Snapping to points using a locked axis (see page 63)

Intelligent Cursor modification keys

The following keys on the keyboard can be used to modify the use of the Intelligent Cursor.

X — Press and hold the **X** key on the keyboard to lock the X axis whilst in any construction mode, dragging objects and moving points.

Y — Press and hold the **Y** key on the keyboard to lock the Y axis whilst in any construction mode, dragging objects and moving points.

Z — Press and hold the **Z** key on the keyboard to lock the Z axis whilst in any construction mode, dragging objects and moving points.

N — Press and hold the **N** key on the keyboard to move points locked on the axis of the normal of the surface.

Creation using a locked axis

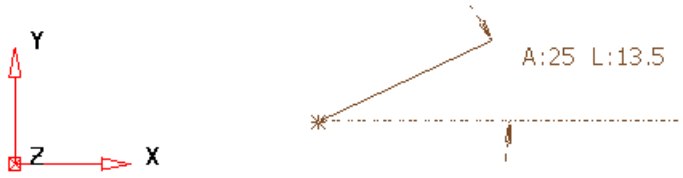
As an alternative to using construction lines during line creation, you can use the **X**, **Y**, and **Z** keyboard modifier keys to lock the corresponding orthogonal axis. This can be useful for additional precision in complex models to control the direction of item construction.

When you are creating in free space, the cursor grids the distance along the locked axis as you move the mouse.

1. Select **Line**  .
2. Enter a point.
3. Press and hold down the required **X**, **Y** or **Z** key to lock the **X**, **Y**, or **Z** axis.

   You may need to change the plane you are working in to lock the required axis. For example, to lock the Z axis you need to be entering points in the XZ or YZ plane (not XY).
4. Move the cursor to create the line to the required length.

The example below shows the construction line displaying along the X axis, whilst the mouse cursor is above. The potential line creation result is displayed at an angle of 25°.



The example below shows a similar mouse position above the x axis, but the X key has been held down to force the line creation to be locked onto the X axis, despite the cursor position located above the X axis.

Dragging on a locked axis

You can drag existing objects along a locked axis, by pressing and holding the corresponding keyboard modifier key X, Y, or Z when you start to drag. The intelligent cursor remembers where the drag started and locks that axis.

All objects in the selection are dragged along the corresponding locked axis, allowing multiple items to be moved together.

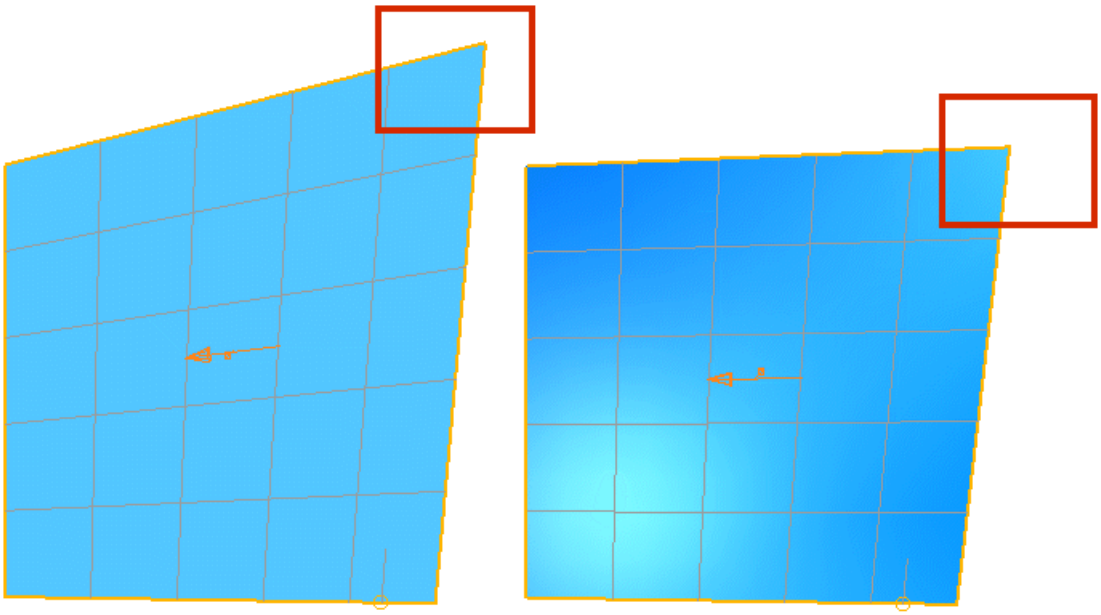
Moving points on a locked axis

When moving points on a power surface, you can use the **N** keyboard key to lock the axis to guarantee the point is being moved in a perpendicular direction from the surface. The locked axis is the original point's surface normal.

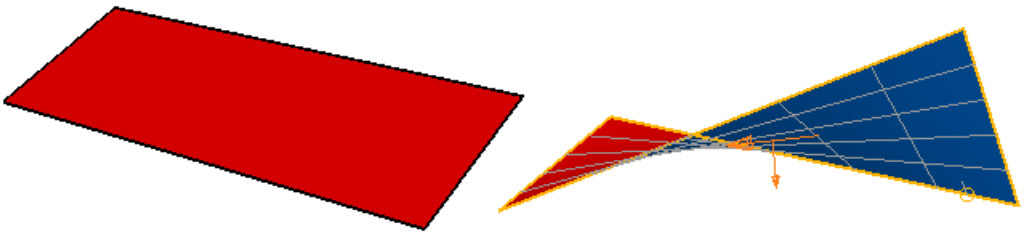
When you use the **N** key, the drag point is locked on the axis, normal to the surface from where you picked the original point.

This does not work if you are using a view that looks down the surface normal. Change views to see the point moving perpendicular to the surface.

In these examples, both corner points on the surfaces have been dragged to change their shape from a standard plane. The **N** keyboard key was used for the image on the right to move the surface in a perpendicular direction to the surface normal.



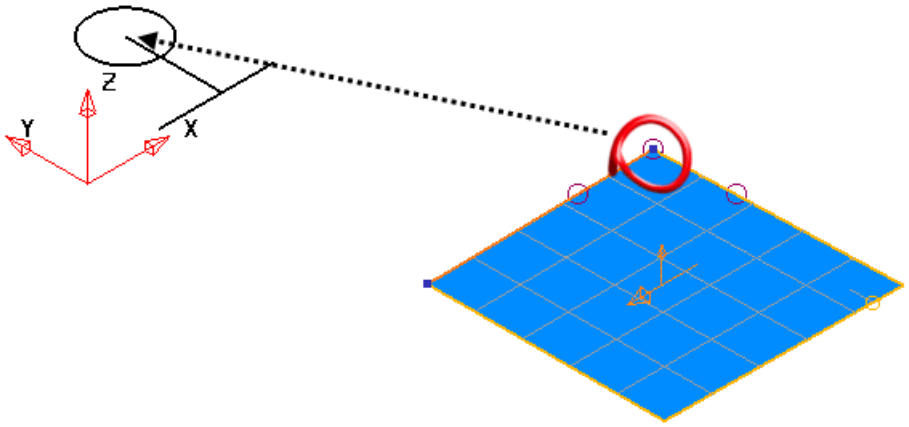
The effect is not truly visible until the surfaces are rotated, seen in the following rotated examples.



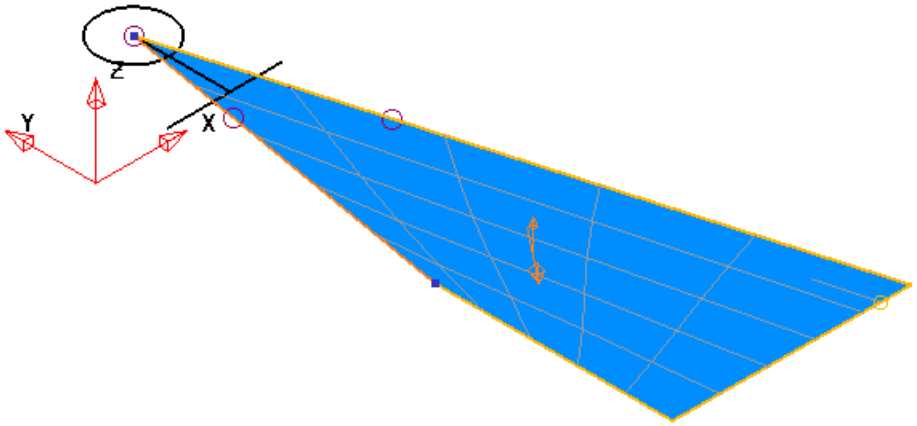
Snapping to points using a locked axis

When pressing and holding one of the keyboard keys **X**, **Y**, or **Z** to lock the axis during line construction, drag selected objects or move points, you can still move the cursor to a nearby point to snap to it. The axis remains locked as the intelligent cursor projects the resulting point from the snapped point to the locked axis.

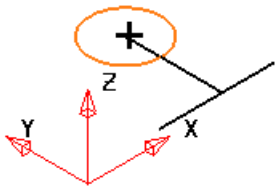
The example below shows a plane with a selected corner, ready to snap to the centre of the circle in the direction shown by the arrow.

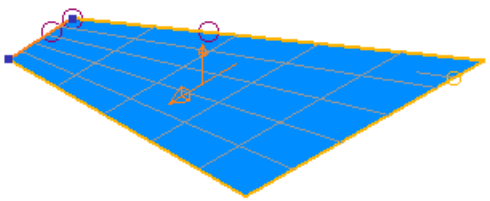


The example below shows the corner point dragged and snapped to the circle centre.



The example below shows the corner point dragged to the circle centre, but using the **X** key to lock the X axis.





Editing objects using the Intelligent Cursor

When you start to drag an object, a group of objects or an editing handle of an object, the cursor feeds back information about points you can snap to.

The Intelligent Cursor:

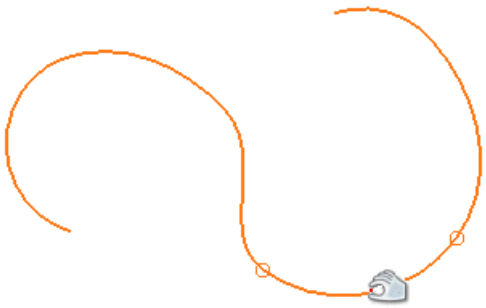
- finds intersections between wireframe items and a surface or solid.
- snaps to the projected intersection of wireframe items and surfaces.
- snaps bezier curves to triangles in a symbol imported as a solid. For further details, see **STL/DMT** options in the Menus and Toolbars manual.
- snaps to triangles in symbols (not applicable to components).

For further details, see Using the Intelligent Cursor to enter positions (see page 58) and Selection using the Intelligent Cursor (see page 60).

Object highlighting

When the intelligent cursor is enabled, objects that you can select are highlighted when you move the cursor over them.

The two keypoints nearest the cursor on a curve are highlighted and are displayed as small circles when you move the cursor over them. This helps you find them more easily and quickly. The keypoint markers display in both creation and edit modes.



For further details, see Selection using the Intelligent Cursor (see page 60).

Models

A model is a representation of a “real life” (usually engineering) component, built on the computer. You build models from a set of geometric objects (lines, arcs, curves, and surfaces) which together represent the components.

You can view a model from any angle and shade it to look very similar to the real component. You can also model other properties such as weight and volume.

Creating a model

1. Select **File > New** from the menu.

   This creates a new model file and opens a new graphics window. The new window automatically becomes the active window. The window title shows the model name (in this case New_Model) and a sequential number.

2. The first time you save this model, the **Save Model As** dialog is displayed, which prompts you to enter a name for your model. For further details, see Saving a model with a new name (see page 66).

Saving a model

When you want to keep your work,

1. Select **File > Save**.

   If a model was originally opened with write access, it is automatically saved, overwriting the stored version.

2. For a new or read-only model, you must use **File > Save As** to save the model with a new name. (see page 66)

Saving a model with a new name

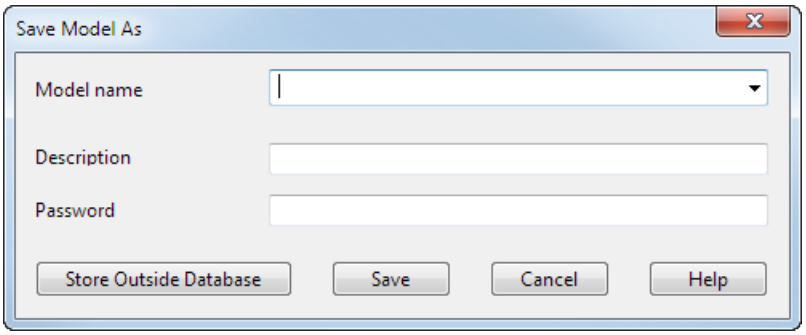
When you want to keep your work but don’t want to overwrite an existing model, you can save the model using a new name. The location of the model you save is defined in the Tools > Options > File > Model dialog. As well as the path to the model, the **Always save and open from outside the database** option defines if the model is saved inside or outside the database.

1. Select **File > Save As** from the menu.

2 The dialog that is displayed depends on the setting of the **Always save and open from outside the database** option:

- When you first use the program, the **Always save and open from outside the database** option is selected by default and you automatically use the standard Windows **Save As** dialog to save models outside the database.
- To save models in the shareddb database, select **Tools > Options > File > Model** and deselect the **Always save and open from outside the database** option.

  Selecting **Save As** from the **File** menu now displays the Save Model As dialog. (see page 192)



3 Use the dialog to save your model using a new name.

*If your models are single files and stored outside the database, you can display the **Save As** dialog instead of the **Save Model As** dialog. To do this, select the **Always save and open from outside the database** option.*

Checking for corruption

To spot any corruption as soon as possible after it has occurred, the File Doctor is run whenever you save a model. For further details, see Tools > Model Fixing.

If File Doctor finds any problems, you are asked to save the model under a new name or run File Doctor in fix mode to correct the problem.

To stop running File Doctor on every save, deselect the **Check Model Before Save** option on the **Model** page of the **Options** dialog.

A set of options is available from the **Model** page of the **Options** dialog. You can select which ones to use when using File Doctor. For further details, see Model options.

Closing a model

1. If you have more than one model open, make active the model you want to close by clicking in its graphics window or selecting the model from the **Window** menu.
2. Select **File > Close** from the menu.

   The active graphics window closes.
3. You will be prompted to save changes you have made to the model:
   - Click **Yes** to save the changes to the original file name.
   - Click **No** to close the file and lose any changes that you have not saved.
4. The model is checked to see if the data can be compressed. If the program decides that compression is possible, a dialog is displayed:
   - Click **Yes** to compress the model data and close the file.
   - Click **No** to close without compressing the model data.

To save an edited file to a new filename you need to use File > Save As (see page 66).

Other ways to close a model

A model is also closed if you do one of the following:

- Close the last open graphics window of the model (by selecting **Close** from the **Window** menu).
- Exit the program.
- Using the **File > Close and Zip** command.

In all cases, you are prompted to save any unsaved edits.

Opening a model

Each model you open is displayed in a separate graphics window on the screen. You can also show each model in a number of windows and can have up to 20 models open at the same time.

You can open a model and select its read/write status.

1. Select **File > Open** from the menu.

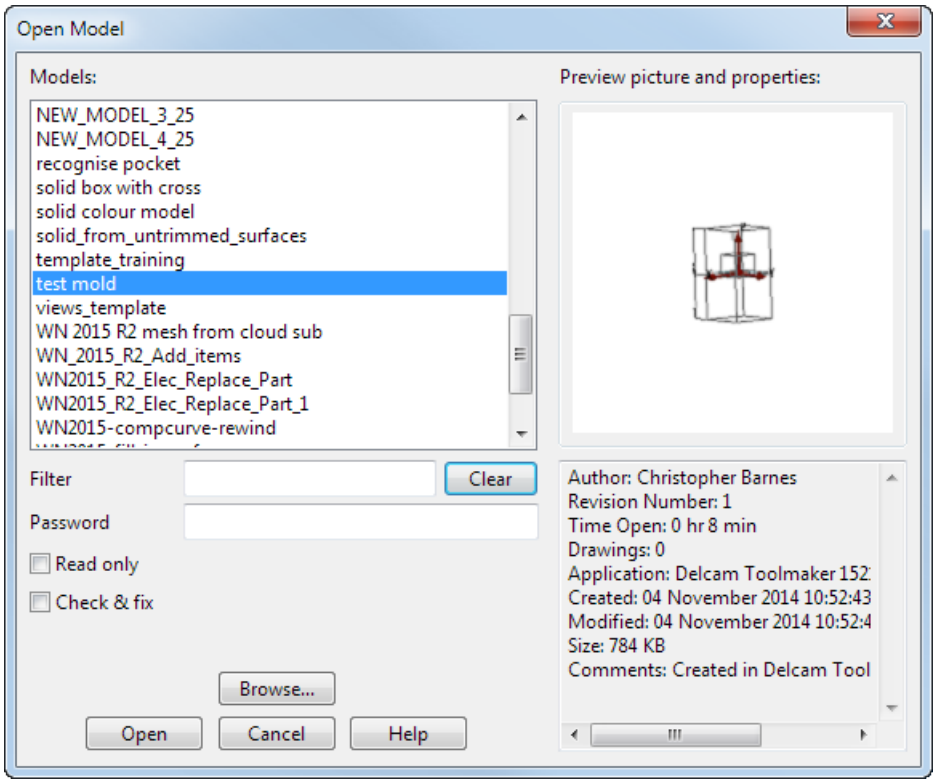
   The **Open Model** dialog is displayed
2. Use the dialog to open a model.

   For further details, see Open Model dialog (see page 69).

If your models are single files and stored outside the database, you can always display the standard Windows **Open Model** dialog. To do this, turn on the **Always save and open from outside the database** option on the **Model** page of the **Options** dialog. For further details, see Model options.

Open Model dialog

Use this dialog to open models.



Models — Select a model name from the list.

If you select any item, you can use the keyboard arrow keys ⇧⇩ to move to the next model.

*Models of the form NEW_MODEL_<number> are created when **File > New** is selected. These only appear in the **Models** list if you have system administrator security.*

Filter — When the list of model names becomes very long, it can be difficult to find the model you want. Entering a filter reduces the list to only those models which match the filter pattern, for example, entering a filter of **.vl* will list only those models which have the extension *.vl*.

We suggest that you use a designer's initials for model name extensions. For example, **mymodel.vl.**

The filter is not case sensitive. Entering a filter of **.VL* will list models that have the extension *.VL* as well as those with the extension *.vl*. For further details, see Filter options (*Menus and Toolbars*).

Clear — This clears the text in the **Filter** box.

Password — If the model is protected by a password, enter it in the text box.

Read only — If you want to open the model, but prevent it from being overwritten, click the **Read only** box until it shows a tick.

If you decide to save any changes, you will need to save the model under a new name using **File > Save As**. For further details, see Saving a model with a new name (see page 66).

Check & fix — When ON, File Doctor examines the selected model and corrects any errors before opening the model. For further details, see Tools > Model Fixing > File doctor (*Menus and Toolbars*).

Preview picture and properties — If a thumbnail of the model has been saved, it is displayed in the top window. Information about the model appears in the bottom window, including file dates, size and description.

Browse — This opens models stored as single files and not in the database. Click the **Browse** button to display the standard Windows **Open Model** dialog. Use this dialog to select and open a model.

Models stored in the directory format and not in the database must be imported.

Search — This allows you to create a filter based on the fields used for temporary and vault models. It is only available if you have a naming_convention.mac file in your shareddb folder.

The naming_convention.mac file allows you to save vault and temporary models. For further details, see Defining your naming scheme (Managing your PowerSHAPE system).

Click **Search** to display the Search for files dialog.

Recover — This button only appears if you select a model that was not closed properly, for example, if there was a power failure while working on the model. **Recover** opens the model and restores it back to its state before the failure.

Open — The selected model is opened. If the model you are opening is shaded, you can open the model in wireframe mode by pressing the  button next to the progress bar on the status bar.

Cancel — Removes the dialog from the screen without opening a model.

Starting up PowerSHAPE and opening a model

When you start up PowerSHAPE at the command prompt, you can open an existing model at the same time by typing:

powershape model_name

For further details, see Other ways of opening psmodels in PowerSHAPE (see page 71).

For the above commands to work, you must have FILE OPEN IFNONE in your login macro.

Other ways of opening psmodels in PowerSHAPE

As well as using the **File Open** command to open models in the single file format (.psmodel), you can also:

- Double-click a .psmodel file. If the program is not running, it will start up and then open the model.
- Right-click the .psmodel file in Windows Explorer and select **Open**. You can open a selection of models in this way.
- Drag and drop one or more .psmodel files into the program window.
- Double-click a PowerSHAPE.exe shortcut.
- Double-click a .psmodel shortcut.
- In a web browser, click on a URL to a .psmodel; for example, file://pshape/models/bottle_example.psmodel. You can either open the model immediately or download it and open it later.

Recently opened models

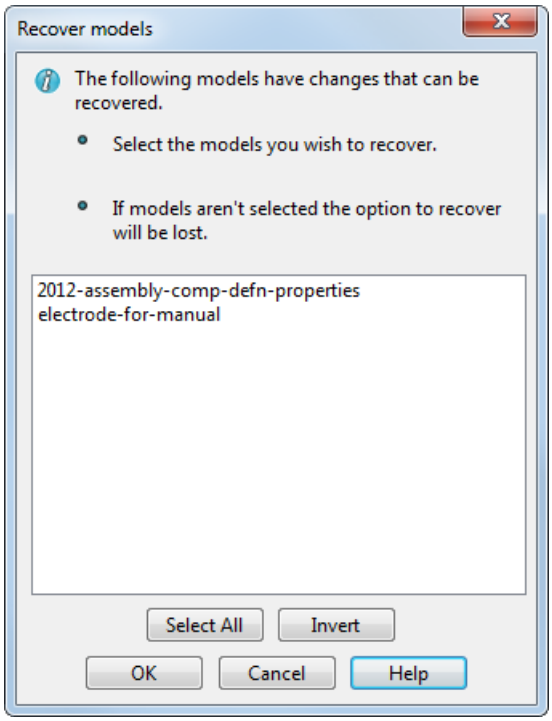
Select the **Recent Files** option at the bottom of the **File** menu to see the last four models listed. The *read* or *write* status when they were opened is also shown.

To open a recently opened model, select the **File** menu and click the model name you want. It will open with the same read or write status as last time.

If the model is not listed or you want a different read/write status, use the **File > Open** option. For further details, see Opening a model (see page 68)

Recovering a model

If your PowerSHAPE session crashes, it may be possible to recover the model you were working on. When you restart PowerSHAPE, after a crash, the following dialog is displayed:



1. Select the models that you wish to recover. You can use **Select All** and **Invert** to select multiple models.
2. Click one of the following:
   - **OK** to recover all selected models in the list.
   - **Cancel** to cancel the recovery process. All recoverable changes will be lost.
3. Following a model recovery, the recovered model must be saved to allow recovery from a second crash.

   If your model does not contain any post-version 8 solid data (Parasolid), the models will be recovered to the command before the crash.

   If your model does contain post-version 8 solid data (Parasolid), models can only be recovered to the state at which the last autosave occurred.

   The time between autosaves is defined using the **Time between autosaves** option on the **Tools>Options>File>Model** dialog.

Closing and opening zipped models

Files can be compressed to make them take less disk space. This is known as *zipping* a file. By zipping a model, you can save about 60% of disk space, but it takes longer to open as it needs time to unzip.

To close and zip a model,

1. Save the model if you haven't already done so.
2. Select **File > Close and Zip** from the menu.

   The model file is stored in a zip file and the original file of the model is deleted.

To open a zipped model,

1. Select **File** > **Open** from the menu.

   The **Open Model** dialog is displayed.
2. Select the model using the dialog.
3. If the model is stored outside the model database, click the **Browse** button to display the **Open** dialog. Use this dialog to select and open the zipped model. You may need to select **Zip** from the **Files of type** menu.
4. Click **Open**.

   A copy of the model is extracted from the zip file and stored in the same folder as the zip file. While the model is open, any changes are saved to the copy.

   *Zipped directory models are opened using **File Import**.*

When you finish working on the model, you can close it using:

- **File > Close** — This saves the copy you are working on and deletes the zip file.
- **File > Close and Zip** — This adds the copy to the zip file and then deletes the copy.

Changing the view of a model

Each window can contain:

- a single view of the model.
- two views of the model splitting the model window vertically.
- four views of the model splitting the model window vertically and horizontally.

Each of these views can be from any direction, at any scale, and can be parallel, isometric or perspective projections. Surfaces can be drawn as wireframe or shaded and objects can be blanked from specific views to reduce clutter.

To change the view of a model:

- Use the **Window** menu to create, delete and select windows.
- Use the **View** menu commands to alter the view of the model in the active window.

Select **View > Layout** to display the **View Edit** dialog which allows you to set up customised view arrangements. For further details, see View Edit dialog (*Menus and Toolbars*).

Select **View > Toolbars > Views** to display the **Views** toolbar, which provides many options to change the view. For further details, see View > Toolbars > Views (*Menus and Toolbars*).

- Use the middle mouse button (used with certain keyboard modifier keys) to dynamically alter the view by rotating, zooming and panning. For further details, see Middle mouse button (see page 42).
- **Quick Rotation** allows rotation of the selected object when you hold down the **R** on the keyboard and scroll the mouse-wheel. The object is rotated about its principal axis by **1** degree for each notch on the mouse-wheel.

Resetting a model

You can delete all the changes you have made to a model since you last saved it.

***Reset** permanently removes all unsaved edits from the active model. A warning is displayed, but once you click Yes, it cannot be undone.*

1. From the **File** menu, select **Reset**. A warning is displayed.
2. Click **Yes** to lose the edits. The model is reset.

   Click **No** to keep the edits. The reset command is ignored.

Changing the password of a model

You can change the password of a model as follows:

1. Open the model with read-write permissions.
2. In a graphics window of the model, click in an empty space to make sure nothing is selected.

3 From the **File** menu, select **Properties** to display the File Properties dialog

4 Use the dialog to edit the details and password of the model.

Finding out what a model contains

You can find out how many objects are stored, details of current levels and levels of objects in the selected model. You can edit details and information about a model and you can change the password of a model.

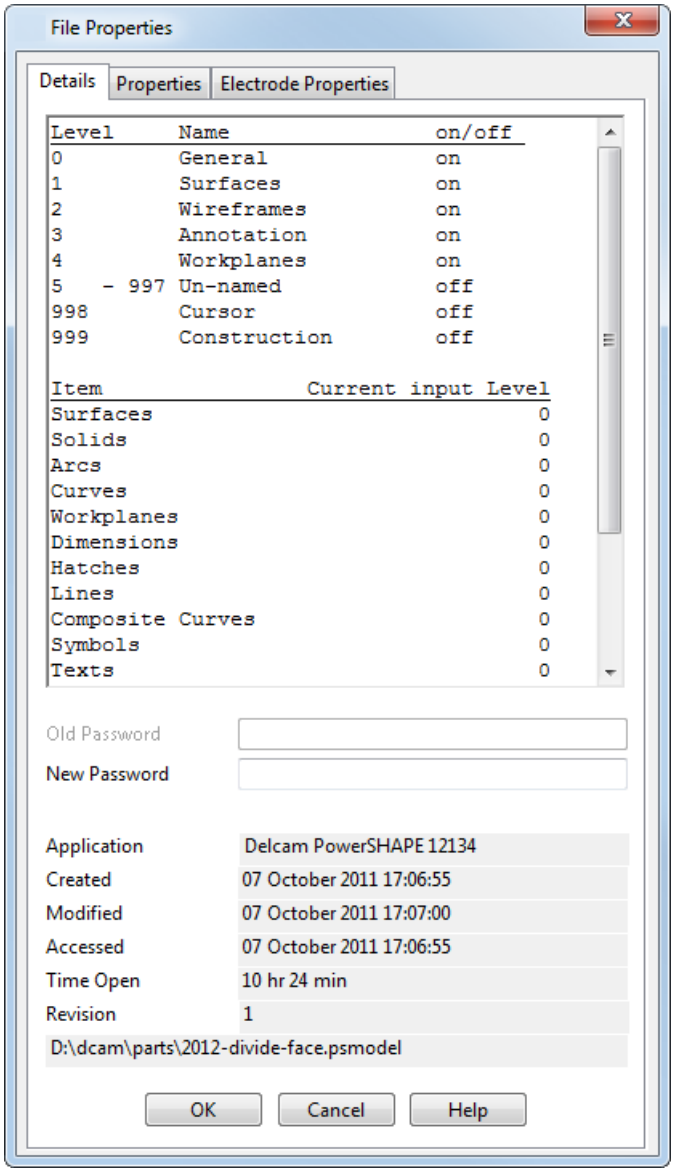
From the **File** menu, click **Properties** to display the **File Properties** dialog.

This contains the following tabs:

- **Details**
- **Properties** (see page 77)
- **Custom** (see page 78)

File Properties dialog - Details

The **Details** page of the **File Properties** dialog contains information about levels and objects for the current model in PowerSHAPE.



The **Details** page tells you the following:

- Which levels are turned on and off.
- The current creation levels for objects.
- The number of objects of a particular type in the model.
- The levels on which particular types of objects lie.
- Totals for objects if they lie on more than one level.

Some of the information is automatically generated by the program. This includes:

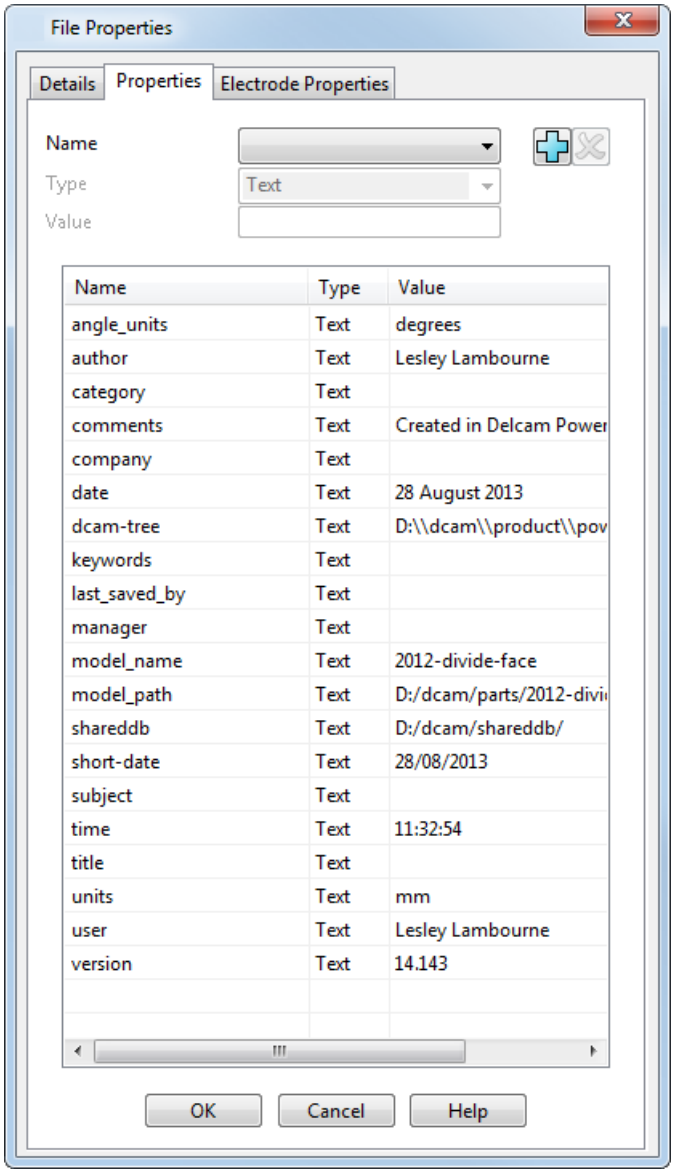
- Details about when the model was created, last modified and last accessed. The length of time that the model has been open is also included.
- Path of the model.

OK - This removes the dialog from the screen.

*To print the information that is displayed, right click in the dialog and select **Print** from the popup.*
*To save the information that is displayed, right click in the dialog and select **Save To File**.*

File properties dialog - Properties

The **Properties** page of the File Properties dialog displays information about the model.



Name - Enter the name of the custom property and click **Enter**. Other parameters now become available and the name is added to the custom property list.

 -Click to add a property.

 - Highlight a name in the list and click this button to remove a name from the list.

Type - The default property type is *Text*, but you can also choose *Date*, *Number*, *Yes or No* from the drop down list. The custom property list is updated to reflect your selection.

Value - Enter an appropriate value. The custom property list is updated accordingly.

OK - When you have entered all your custom properties, click **OK**.

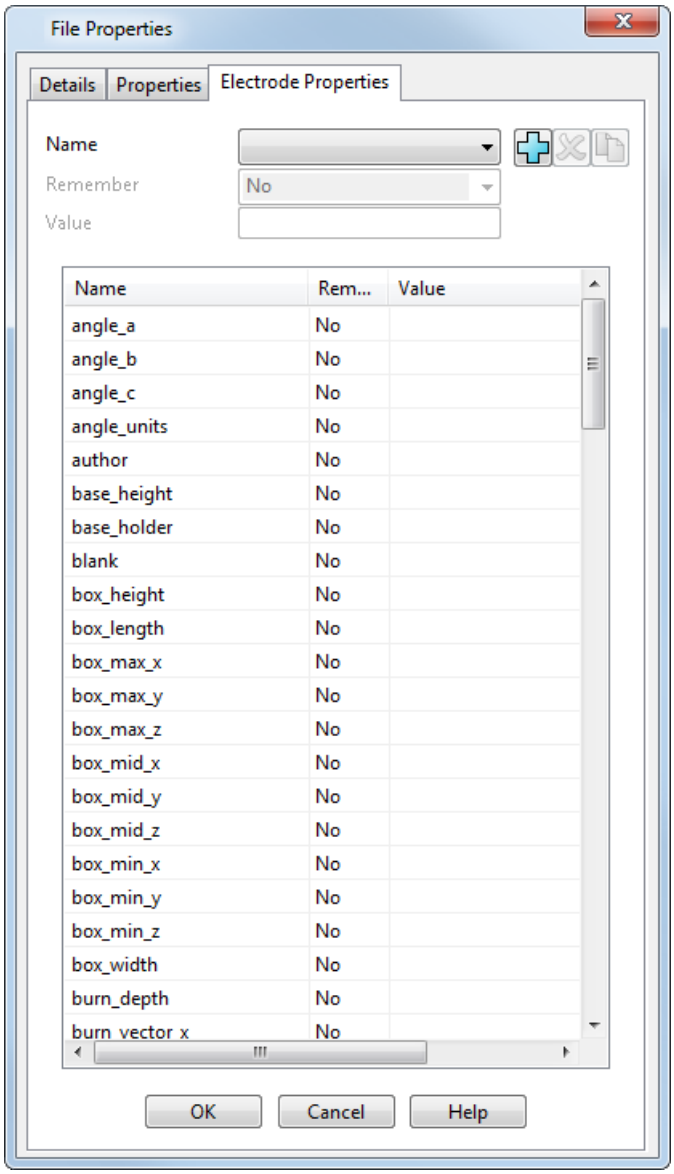
You can enter information in this dialog so that it can be included in text on your model or drawing. In addition, you can also use some information that is automatically generated by the program when you save a model.

File properties dialog - Electrode Properties

Use **Electrode Properties** to create setup strings that can be used to add custom electrode text to a drawing.

You can enter information in this dialog so that it can be incorporated into text on your electrode model or drawing. In addition, you can also use some information that is automatically generated by the program when you save a model.

Information in this dialog is used when adding an electrode substitution string to a drawing template.



Name - Enter the name of the property and click **Enter**. Other parameters now become available and the name is added to the property list.

 - Click to add a property.

 - Highlight a name in the list and click this button to remove a name from the list.

 - Create a copy of the substitution string. on the Windows clipboard so it is ready to paste elsewhere.

Remember - Select **Yes** or **No** from the drop down list. If you select **Yes**, then the value for that property is automatically entered in the **Additional Electrode Details** dialog The custom property list is updated to reflect your selection.

Value - Enter an appropriate value. The property list is updated accordingly.

OK - When you have entered all your properties, click **OK**.

Deleting a model

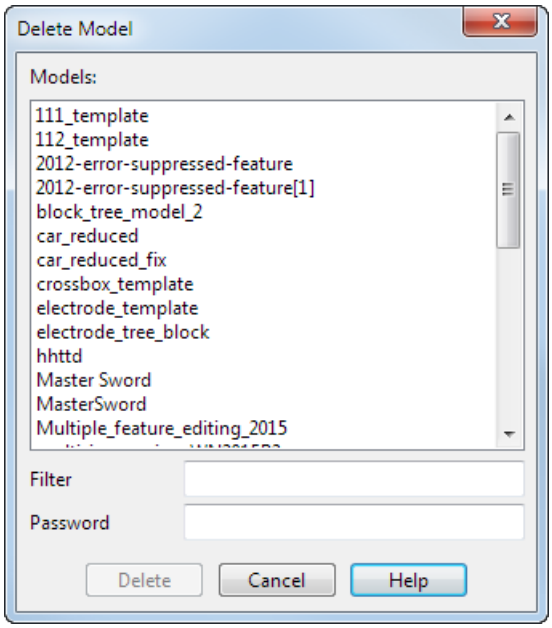
You can delete unwanted models that are stored in the database. Models stored outside the database can be deleted like any other file using Windows Explorer.

1. From the **File** menu, select **Delete** to display the Delete Model dialog (see page 80).
2. Use the dialog to delete the model.

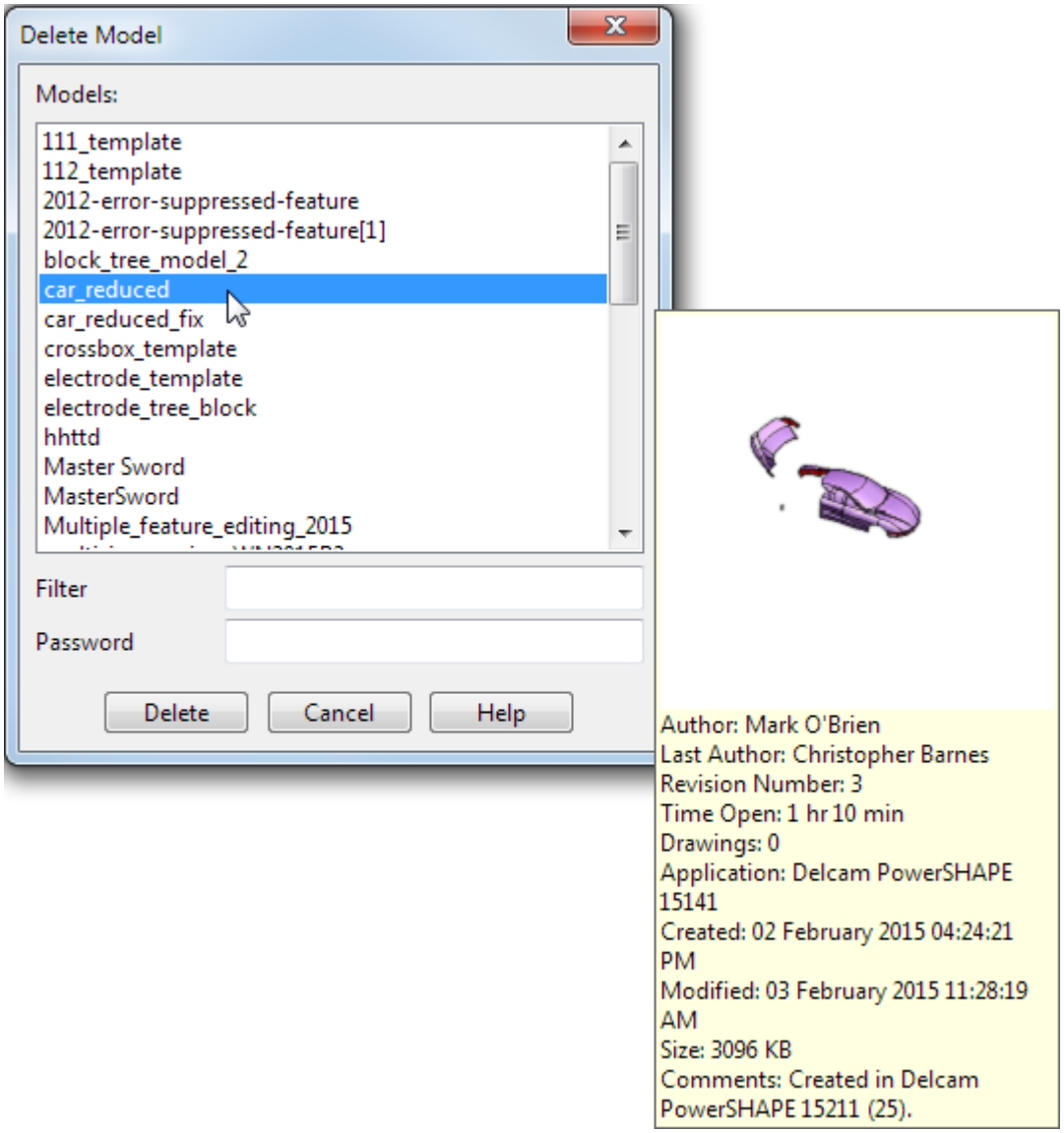
***WARNING:** Deleted models are permanently removed and cannot be recovered.*

Delete Model dialog

Use this dialog to delete models.



Models — Click on a listed model name (to select it) and the item is highlighted. If you select any item, you can use the keyboard arrow keys ⇧⇩ to move the highlight and scroll the list. The graphical tooltip displays a thumbnail of the model and file information.



*Models of the form NEW_MODEL_<number> are created when you select **File > New**. These are displayed in the Models list only if you have system administrator security.*

Select multiple listed model names to be deleted using the standard *SHIFT-Click*  or *CTRL-Click* 

technique to select them. Move the cursor over the selection to see the individual thumbnails.

Filter — When the list of model names becomes very long, it can be difficult to find the model you want. Entering a filter reduces the list to only those models which match the filter pattern, for example, entering a filter of **.vl* will list only those models which have the extension *.vl*.

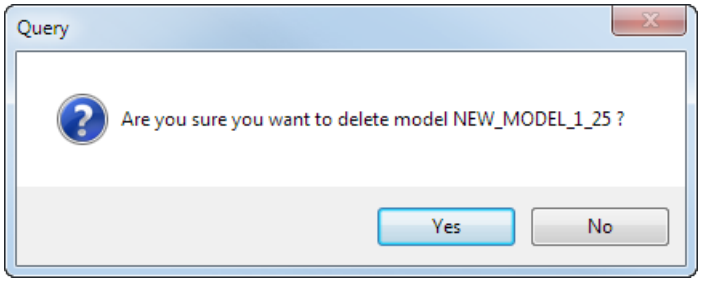
The filter is not case sensitive. Entering a filter of *.*vl* will list models that have the extension *.VL* as well as those with the extension *.vl*.

For further details, see Filter options (*Menus and Toolbars*).

We suggest that you use a designer's initials for model name extensions. For example, `mymodel.vnl`.

Password — If the model is protected by a password, enter it in the text box.

Delete — When you click this button, a confirmation message is displayed:



Click **Yes** to remove the model from the database and return to the **Delete Model** dialog. If the **Use Recycle Bin** option is selected on the **Model** page of the **Options** dialog, the model is sent to the Recycle Bin (providing the bin is not full). Otherwise, the model is permanently removed from the system.

Click **No** to do nothing and return to the dialog.

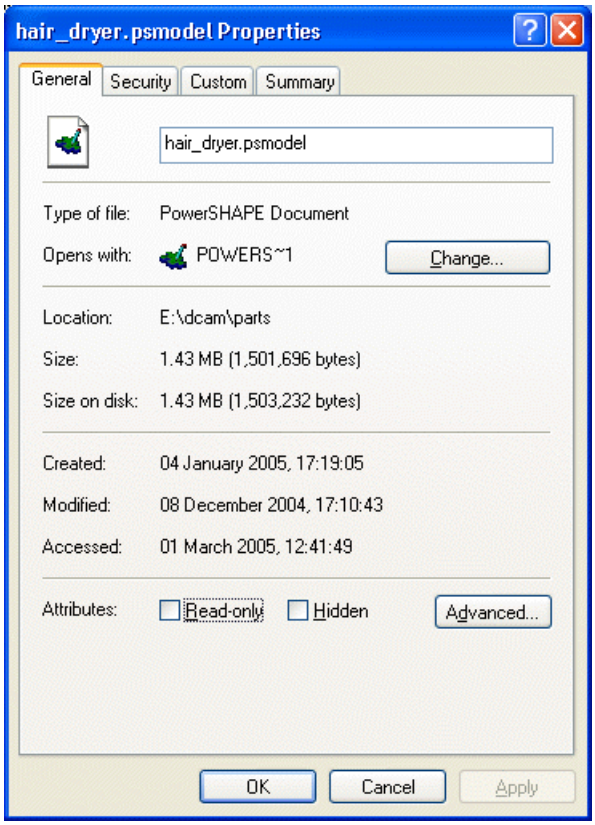
Close — When you have finished deleting models, click **Close** to remove the **Delete Model** dialog from the screen.

Finding out information about a model

You can find certain information about a model, without opening it.

1 In Windows Explorer, select a psmodel.

2 From the **File** menu, select **Properties** to display the File Properties dialog for the model.



The different tabs display information that is stored whenever a model is saved as a psmodel. For example:

- Date of creation
- Application name
- The name of the model and its description
- Current user as the Author (on the **Summary** page)
- Version of PowerSHAPE in the **Comments** section (on the **Summary** page)

When opening a psmodel, the following will be read from the Windows Explorer's **File > Properties** dialog and stored by PowerSHAPE:

- title
- subject
- author
- keywords
- comments
- last saved by
- revision number

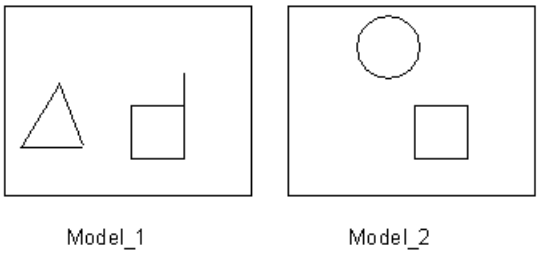
- manager
- company information

This information can be used when creating text.

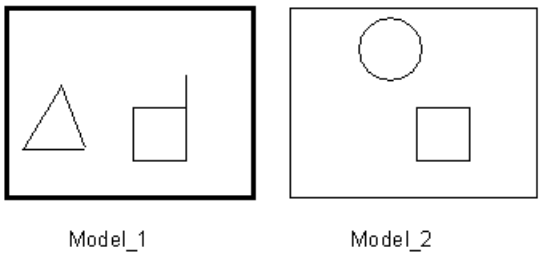
Comparing two models

You may have two models of the same component and want to see what the differences are between the two models.

1. Open the two models you wish to compare.
2. Suppose we have the following two models.



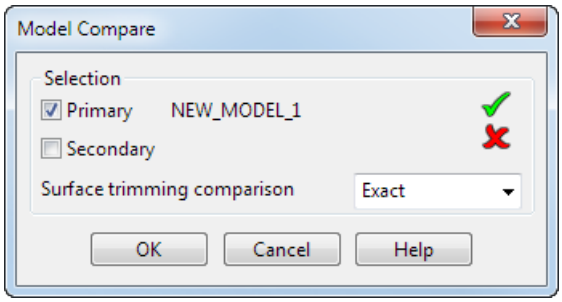
3. Click in the graphics window of the model in which you want to display the differences. This makes the graphics window active.
4. In our example, we made Model_1 the active model.



5. Select **Model Analysis**  on the **General edits**  flyout.
6. Click **Model Compare** on the **Model Analysis** toolbar.
7. Use the Model Compare dialog (see page 85) to complete the comparison.

Model Compare dialog

Use this dialog to compare models.



Use the following options to control the comparison:

Primary — Select the model in which the differences are to be shown. When the dialog is first displayed the **Primary** selection is the model of the active graphics window. You can change the primary selection by selecting the **Primary** option and clicking another graphics window. The ✔icon indicates that a primary model is selected.

Secondary — Select the model to compare against the **Primary** selection in the comparison To select the second model click the **Secondary** option and click in the graphics window of the model you want to compare. The ✘ changes to a ✔.

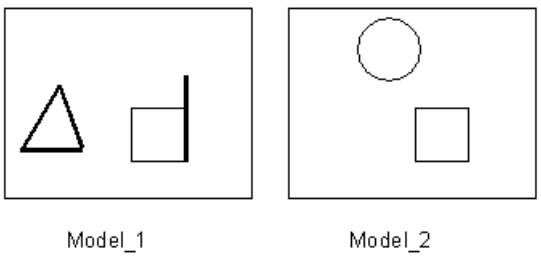
Surface trimming comparison — You can choose how to compare surfaces with trim boundaries and p-curves. There are three options:

Exact — This option finds surfaces with exactly the same pcurves and trim boundaries.

Boundaries — This option finds surfaces with exactly the same trim boundaries. Any p-curve not part of a trim boundary is ignored in the comparison.

None — This option treats two surfaces as the same if the untrimmed surfaces are the same, even if they have different trimmed regions.

OK — Compares the two models. All objects, which are in the primary model and not in the secondary model, are highlighted as selected in the primary model.



Objects that are in the secondary model but not in the primary model are not shown. To show these objects, simply carry out the above steps with the secondary model as the primary model and the primary model as the secondary.

***Warning**: Models are compared on the basis of positional information (XYZ coordinates) only. Differences, for example, in tangent direction, colour and level, are not picked up. This is done for reasons of speed. This command can be a useful tool to aid comparison of models, but you must use it with care and check manually for small differences that the command may not pick up.*

Finding duplicate objects in a model

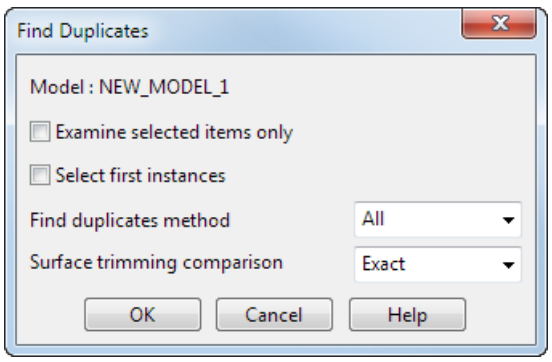
Sometimes, you may have a copy of an object on top of itself. Use this command to highlight any duplicate objects in the model.

1. Click in the graphics window of the model in which you wish to find any duplicate objects. This makes the graphics window active.
2. Select objects if you want to find duplicates in the current selection.
3. Click **Model Fixing** on the **General edits** flyout.
4. Click on the Model Fixing toolbar

   The Find Duplicates dialog (see page 86) is displayed.
5. Use the dialog to help you check if you have any duplicate objects.

Find Duplicates dialog

Use this dialog to find duplicate objects within a model.



Model — This displays the name of the model to find duplicates objects in. By default, it is the active model. You can choose another open model by either clicking in its graphics window or selecting it from the **Window** menu to make it active.

Examine selected items only — Select this option to find duplicates only in the objects which are currently selected. Otherwise, the whole model is checked.

Select first instances — If on, any instance of the object before the last is considered to be a duplicate. If off, any instance of the object after the first is considered to be a duplicated.

Find Duplicates Method — Duplicate surfaces are selected depending on the two options below.

All — This selects duplicate surfaces depending on the **Surface Trimming Comparison** option.

Untrimmed — If all duplicate surfaces are untrimmed, this option selects all the duplicates. If some of the duplicate surfaces are trimmed, it selects all of those duplicates which are untrimmed.

In some imported models, each surface is supplied untrimmed and trimmed. This command allows you to select all the untrimmed surfaces.

Surface Trimming Comparison — You can choose how to compare surfaces with trim boundaries and p-curves. There are three options:

Exact — This option finds surfaces with exactly the same pcurves and trim boundaries.

Boundaries — This option finds surfaces with exactly the same trim boundaries. Any p-curve not part of a trim boundary is ignored in the comparison.

None — This option treats two surfaces as the same if the untrimmed surfaces are the same, even if they have different trimmed regions.

OK— Closes the dialog. All duplicated objects are highlighted as selected and an information box is displayed showing how many items are duplicates.

For each set of duplicate objects, one is not selected. This allows you to operate on all duplicates within a model. For example, you can now delete them all by clicking ***Delete***  .

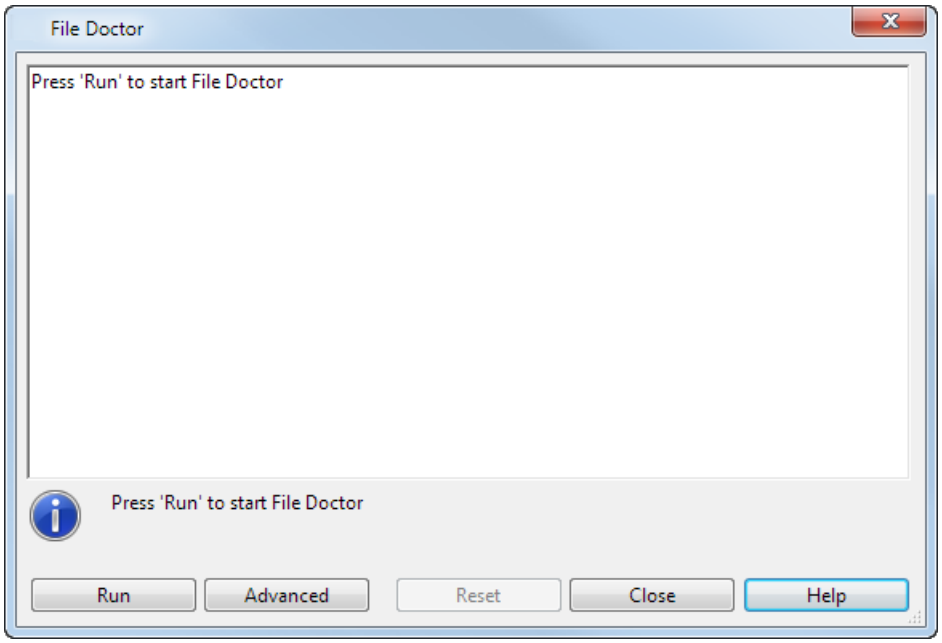
***Warning**: Duplicates are detected on the basis of positional information (XYZ coordinates) only. Objects may be selected as duplicates even if they differ in level, colour, tangent direction, or other non-positional ways.*

Tools > File doctor

This option provides tools to check and repair model files and the shared database. **File Doctor** is automatically opened if there have been errors in the session and you try to save an model using **Save As** or **Save**. This gives you the opportunity to correct errors and will help prevent you working with a model that is already corrupt.

***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.*

1. From the **Tools** menu, select **File Doctor** to display the **File Doctor** dialog.

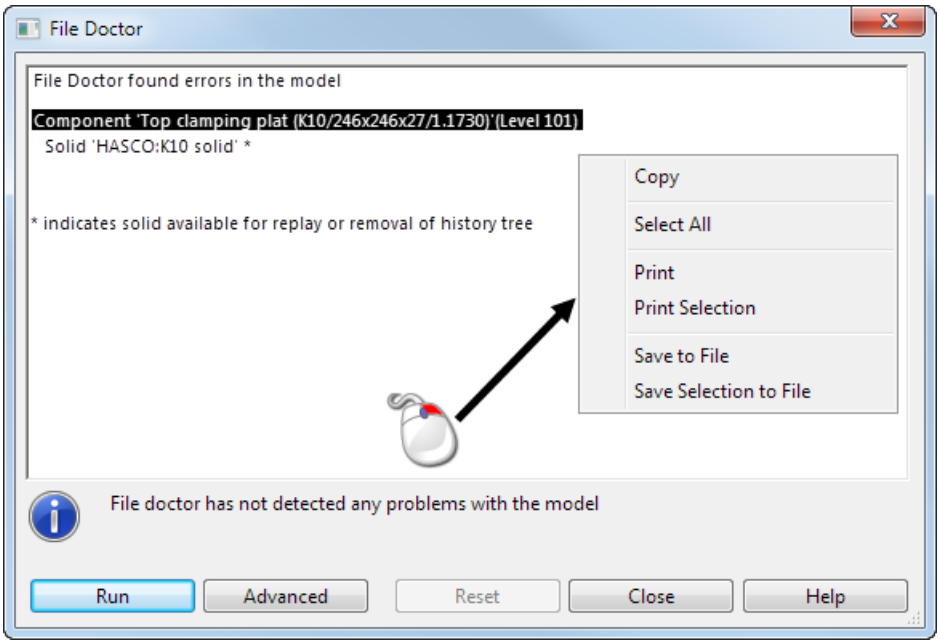


2. Use the File Doctor dialog (see page 91) or File Doctor (Advanced Mode) dialog (see page 93) to check the model.

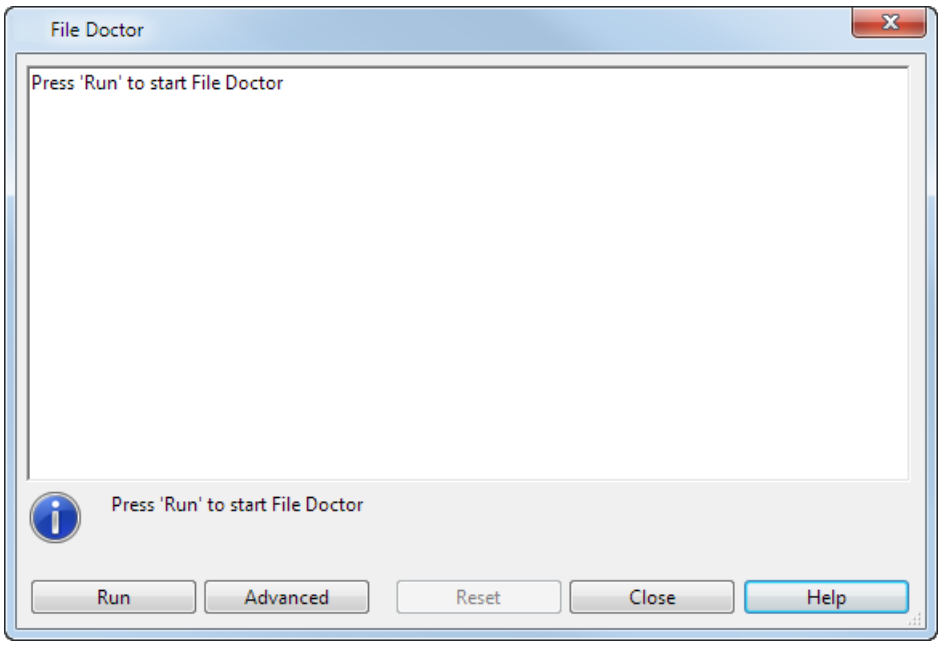
When a model is recovered a copy of the original is kept. The settings that control the action and the location of the copy can be changed by editing powershape.con. The default settings keep a copy of the original in the temp directory.

Using the File Doctor - step by step

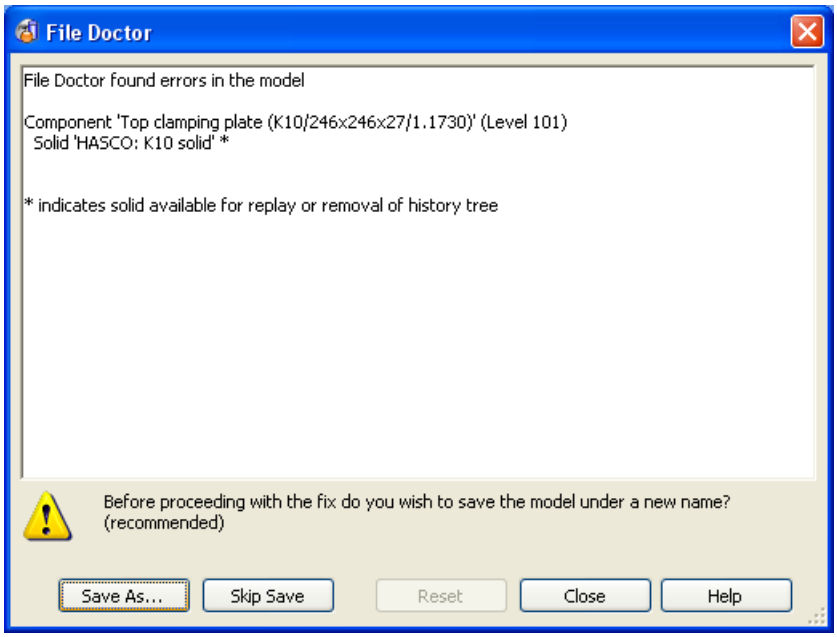
If you position the mouse in the information window of the File Doctor dialog and click the right mouse button, the context menu that is displayed includes printing and saving options.



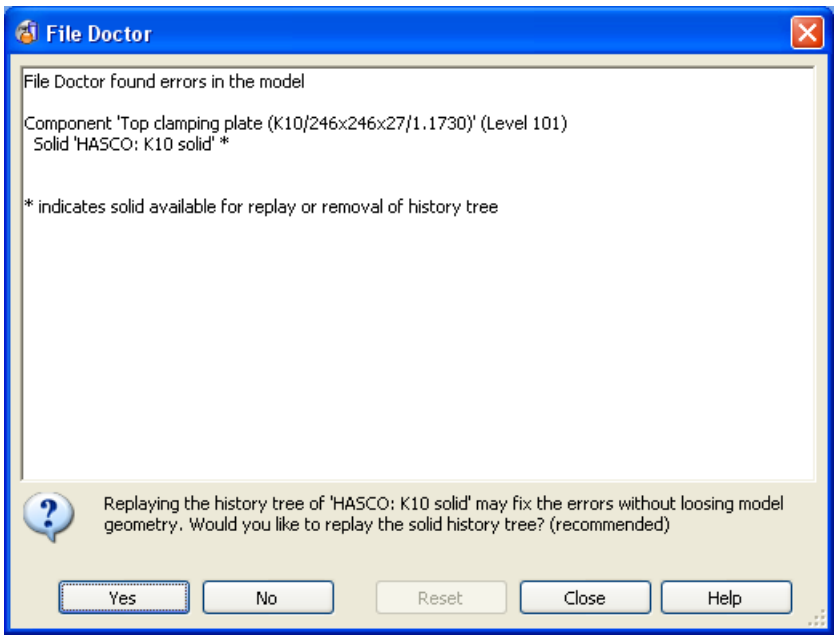
1. Ensure that the corrupt model is open. We recommend that this is the only model open.
2. Select **Tools - File Doctor** to display the standard File Doctor dialog (see page 91).



3 Click **Run**. The File Doctor tries to identify the problem and uses the dialog to tell you what is wrong with the model.



4 Click **Save As**. This is the recommended selection so that you have a ‘safe’ copy.



5 The dialog that is displayed includes a suggestion of a fix that could be applied.

The type of fix offered by the File Doctor depends on the errors that are detected in the model. If possible, File Doctor will offer a fix that does not delete model geometry, such as replaying or removing the solid history tree.

Whilst replaying a solid history tree can fix some problems, it is possible that certain errors could cause PowerSHAPE to hang or crash whilst replaying. If this should happen,

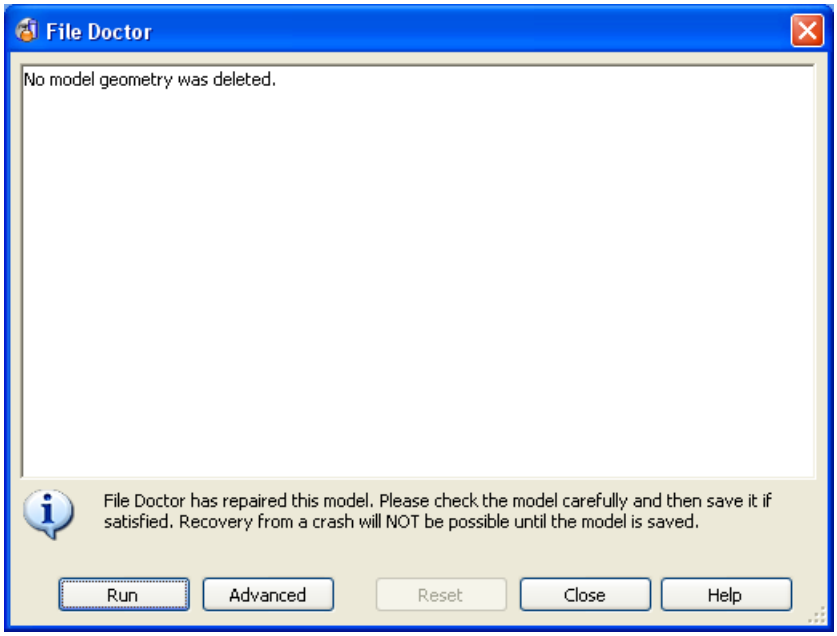
1. Restart PowerSHAPE.

2. Reload the model that was saved before the fix (step 4).

3. Run File Doctor and when offered to replay the solid(s), choose No.

4. You are now offered the choice of keeping the surface geometry (but losing the solid history tree) or keeping as much of the history tree as possible (but losing some features from the tree).

6 At the end of the process, File Doctor will tell you the geometry (if any) that had to be deleted to fix the model.



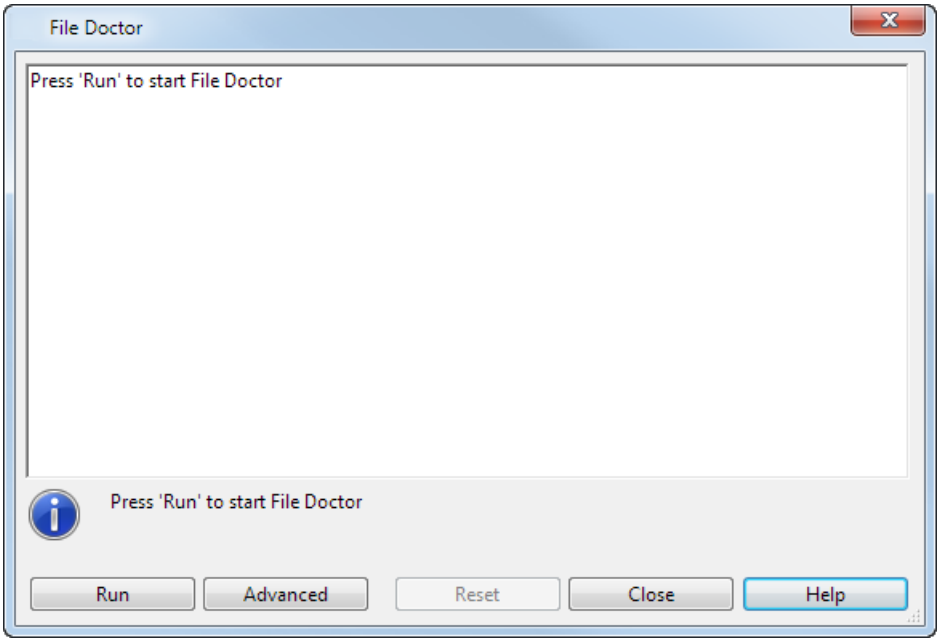
7 Click **Close** to exit File Doctor.

8 Check the model carefully and save it if you are satisfied with the fix.

Recovery from a crash is not possible until you have saved the fixed model.

File Doctor dialog

The standard version of the **File Doctor** dialog displays problems that are found with the model and suggested fixes.



The initial layout of buttons is shown. The buttons that are initially shown as **Run** and **Advanced** will change depending on the stage of the process.

The following buttons may be displayed at the bottom of the dialog:

Run - Start File Doctor check on the current model.

Advanced - Switch to advanced version of the File Doctor and display the File Doctor (Advanced Mode) dialog (see page 93). This mode gives full control over the checks and fixes that are run by the File Doctor. *Only experienced users should use this option*.

Save As - Save the model under a new name before proceeding with the File Doctor fixes.

Skip Save - Skip saving the model under a new name and move onto the fixing stage of the File Doctor.

Yes - Apply the possible fix.

No - Skip the fix.

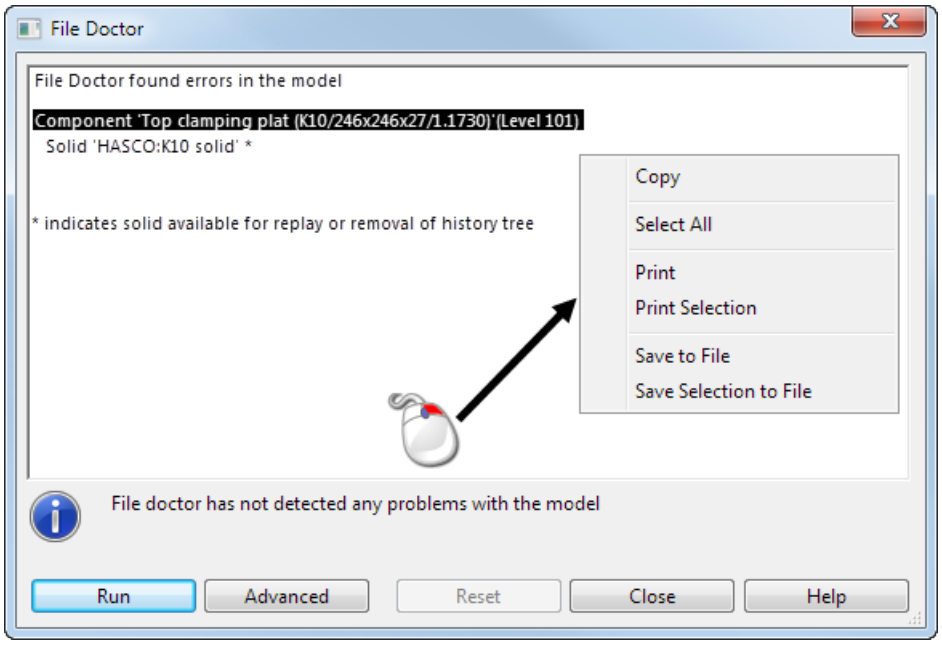
Keep geometry - Fix a solid by maintaining surface geometry but removing the solid feature tree.

Keep history - Fix a solid by removing features with errors from the feature tree.

Reset - Reload the model that was saved before the File Doctor fix. This option is only available if the model was saved before the fix.

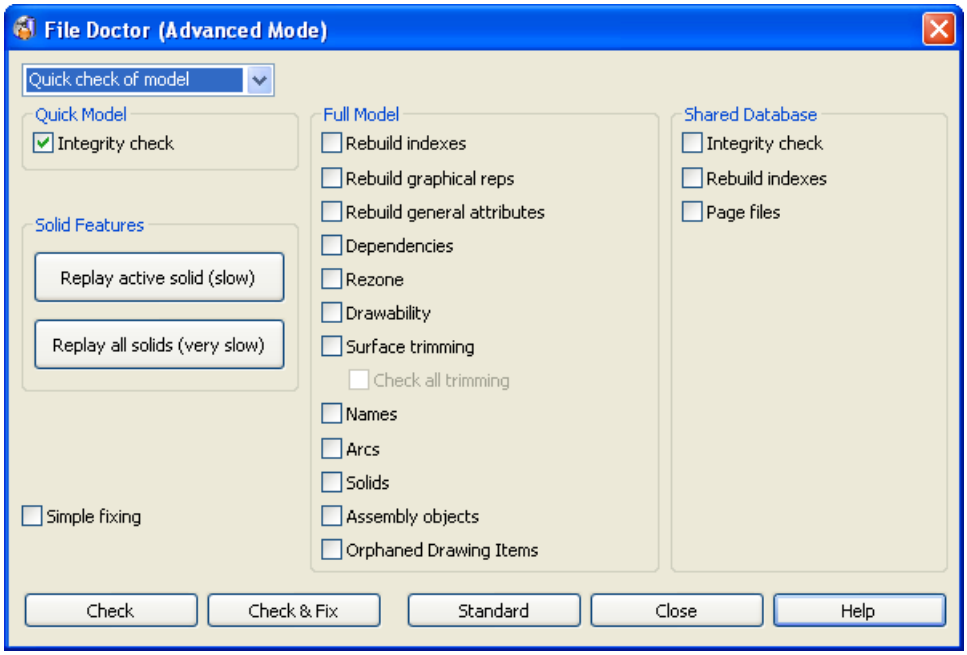
Close - Finish File Doctor.

If you position the mouse in the information window of the File Doctor dialog and click the right mouse button, the context menu that is displayed includes printing and saving options.



File Doctor (Advanced Mode) dialog

File doctor allows you to recover damaged models which would otherwise be unusable. If you would prefer to use the standard **File Doctor** dialog, type **Standard** in the command input box on the **Status** bar.



- Options from this menu automatically turn on the checks on the dialog. Use this menu to select one of the following:

Quick Check of Model - This selects only the **Integrity check** in the **Quick Model** section.

Full Check of Model - This selects the **Integrity check** and all the checks in the **Full Model** section.

Check Shared Database - This selects the **Integrity check** and all the checks in the **Shared Database** section.

User-defined - This allows you to select any check. It is also selected if you change the checks after selecting one of the three options above.

Quick Model - This section contains the following check:

Integrity check - This checks if any objects within the model refer to missing objects and explicit indexes.

If you select **Check**, an information window is displayed showing a list of errors. You will be alerted to any missing indexes.

If you select **Check & Fix**, the objects containing errors are deleted from the model. File Doctor displays an information window, showing the list of faults (if any) within the model that have been cured and any missing indexes that have been created. However, under some circumstances (such as where many objects refers to a missing object), automatic fixing can cause considerable damage. We therefore recommend that **Check & Fix** is used only when you have made a backup copy of the model.

Replay active solid (slow) and **Replay all solids (very slow)** allow you replay the solid feature trees.

Full Model - This section contains the following checks:

- **Rebuild indexes** - All models contain index files. This option clears the model indexes and builds new ones, but only when **Check & Fix** is clicked. There is no **Check** option for indexes.

- **Rebuild Graphical Reps** - ensures that each object geometry definition has a matching and valid graphical representation. The option is only available under the **Check & Fix** option. To explain what it does:
  Each object in PowerSHAPE (such as lines and arcs) has two main components. These are the **geometry definition** and the **graphical representation**. The latter describes how the object is currently displayed in the various views on the screen. It can sometimes become corrupted (e.g. missing or incorrect) while the geometry definition is correct. Typical problems are that the object displays incorrectly or not at all or is not selectable.
- **Rebuild General Attributes** - If you choose **Check**, you are told if there are any errors in the styles of the objects. If you select **Check & Fix**, the styles with errors are replaced with a solid, foreground colour and width 0.3 style.
- **Dependencies** - If you choose **Check**, you are told if there are any errors in surface and vector dependencies. Dependencies are relationships between geometric objects. For example, a solid block or cylinder has a relationship between the main surface and the two capping surfaces. Surface primitives have dependencies between the surface and the set of parameters (for example, length and diameter) that define them.

  If you select **Check & Fix**, File Doctor deletes any bad dependencies.

  *We recommend you first run **Rebuild Indexes** from this section as database table indexes are often corrupt when surface dependencies are corrupt.*

  *With **Check & Fix**, editing dependencies of some objects may alter the dependencies of other objects. We advise you to run **Check & Fix** repeatedly until no further dependency errors are reported.*
- **Rezone** - The model is divided into zones. When an object is picked, only the zones beneath the cursor are searched, which considerably speeds-up the picking operation. If an object becomes incorrectly zoned (as can happen occasionally), it displays correctly, but cannot be picked. This option regenerates these zones, curing most of the problems associated with unpickable objects. It is only available with **Check & Fix**.
- **Check Drawability** - PowerSHAPE should be able to draw (on the screen) every object within a model. This option checks each object and deletes any that cannot be drawn. This option also checks for items that are undisplayable. If the File Doctor is run in **Check & Fix** mode, the undisplayable items are deleted.

Drawability in this context has no connection with "drawing" as used in Drafting.

- **Check Surface Trimming** - If you choose **Check**, you are told if there are any trimming problems on the surfaces. If you select **Check & Fix**, then PowerSHAPE attempts to repair the surfaces. If some boundaries can't be repaired, they are deleted and you are told which ones are deleted.

  Select the **Check all trimming option** on the **Advanced File Doctor** dialog to ignore any previous surface trimming checks and re-check and fix every surface.
- **Check Names** - If you choose **Check**, you are told if objects have duplicate or invalid names. If you select **Check & Fix**, PowerSHAPE renames objects with duplicate or invalid names.
- **Check Arcs** - A valid arc has a start, end, mid and centre point and each point has a different ID and is owned by the arc. If you choose **Check**, you are told if any arc in the model is invalid. If you select **Check & Fix**, PowerSHAPE deletes invalid arcs.
- **Check Solids** - This checks solids and their feature trees for corruption. If you select **Check & Fix**, the corruption is repaired. If the corruption is caused by one of the features in the solid tree, PowerSHAPE tries to fix this feature. If this is not possible, the feature tree for the solid is deleted.
- **Check Assembly Objects** - This checks components and component definitions for corruption. If you select **Check & Fix**, the corruption is repaired. If this is not possible, data that cannot be repaired will be deleted.
- **Check Orphaned Drawing Items** - Some model items that are displayed in a drawing view but not displayed in model windows, such as centrelines, can be orphaned if the drawing view is deleted. This option checks for and removes any orphaned drawing items.

Shared Database - This section contains the following checks:

- **Integrity check -** This does a full integrity check/fix on the shared database.
- **Rebuild indexes -** This deletes and rebuilds the shared database indexes. These can sometimes become corrupted. For example, a hardware or software problem (such as insufficient disk space) occurring while creating, saving or deleting a model can cause damage. Affected models disappear from the menu list (as if deleted) or can appear twice. Regenerating usually solves these problems. The option is only available with **Check & Fix**.

- **Page Files -** rebuilds the page files, ensuring all are correct. It is available only with **Check & Fix**. To explain the purpose of the files:
  The database tables keep a temporary copy of their current state in page files, from which data is paged in and out from core memory. These files can be damaged by a hardware or software problem occurring during a database update. The damage will affect all Delcam programs which use that database.

*We recommend that all users stop working with Delcam products before any option from the **Shared Database** section is used.*

Simple Fixing - If *ON*, simple fixes will be carried out when running File Doctor checks. Any corrupted data will be deleted. If *OFF,* the File Doctor will attempt to work round the corrupted data. For example, it will attempt to remove a single corrupt feature from the solid tree instead of removing the entire tree. Using **Simple Fixing** is more reliable, but it may delete a lot more of the model data.

Check - This checks and displays a report of any faults found by the **File Doctor** routine(s).

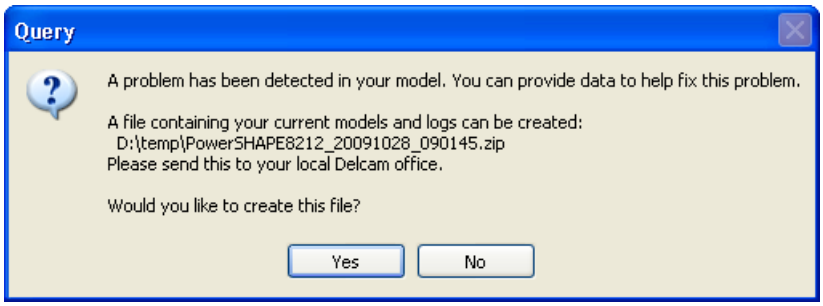
*The **Check** only option is more useful to Delcam Technical Support staff than to regular users. You may find the errors reported rather unintelligible.*

Check & Fix - This checks and repairs the data within the model.

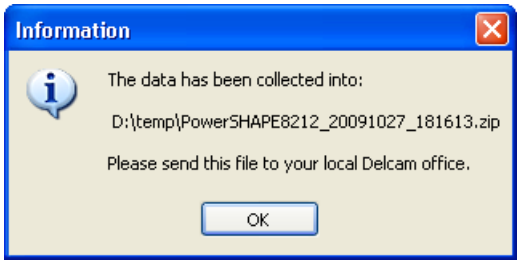
***WARNING:** All errors found by **File Doctor** are potentially serious and could cause the program to crash or malfunction (perhaps causing further corruption). You should always remove the errors with **Check & Fix** before continuing. In many cases, it works by deleting faulty objects. If this deletes large portions of your model, contact Delcam Technical Support, who may be able to retrieve some of the lost data. However, do **NOT** continue working on a corrupt model or database.*

Submitting model information

When a database error is detected a dialog is displayed with a request to submit your model and macros to Delcam.



1 Click **Yes** to create the zip file. It is recommended that you create as requested.



2 Click **OK** to complete the operation.

3 Send the file to your local Delcam supplier.

Note that the date and the time are included in the filename.

Importing and Exporting a model

Use the following sections to import and export data:

Exporting data from a model (see page 98)

Importing data into a model (see page 104)

Exporting data from a model

You can export models to formats other than PowerSHAPE.

1. Select the model or the specific objects you want to export.
2. Select **File** > **Export** from the menu to display the first page of the **Export** Wizard.
3. Use the **Export** Wizard to select the target system and settings for exporting files. Alternatively, click **Finish** to export the data using the following defaults:
   - export selected items, or the entire model if nothing is selected.
   - if a workplane is active, export the geometry to active, otherwise, export to World.
   - if a drawing window is active, export the converted drawings.

*The default for .dgk is to export to World. You can use the **Export** Wizard to export .dgk to the active workplane.*

For advanced modelling concepts, see:

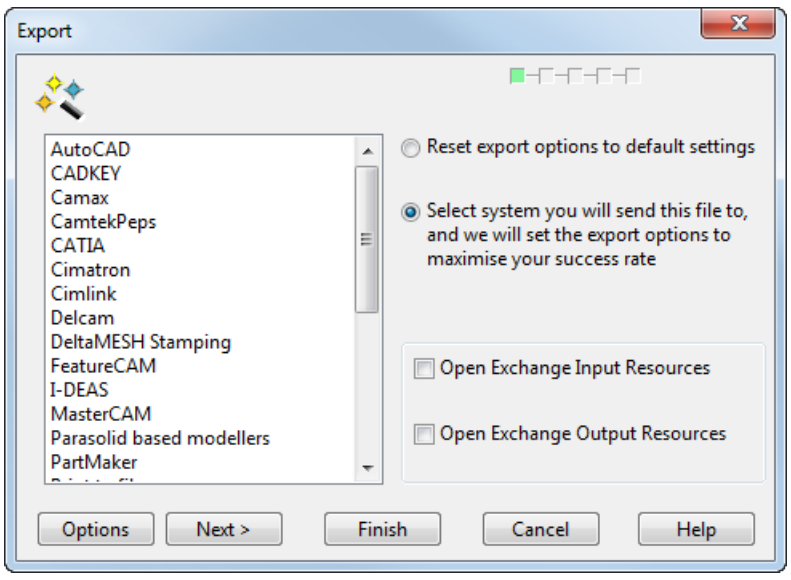
- Using Delcam Exchange to translate data (see page 197)
- Exporting data without a licence (see page 204)
- Exporting materials (see page 205)

Using the Export Wizard

Use the **Export** Wizard to:

- export selected model items, visible items or all items.
- export relative to an active workplane or World.
- export drawings.

1. Select **File > Export** to display the **Export** Wizard.



2. Select the target system for export from the list.

3 Click **Next** to display the **Export File** dialog.

The file name of your model is automatically displayed as the filename. Change the **File name** as required and select the file type from the **Save as type** list.

4 Click **Save**. The subsequent pages that are displayed depends on your export:

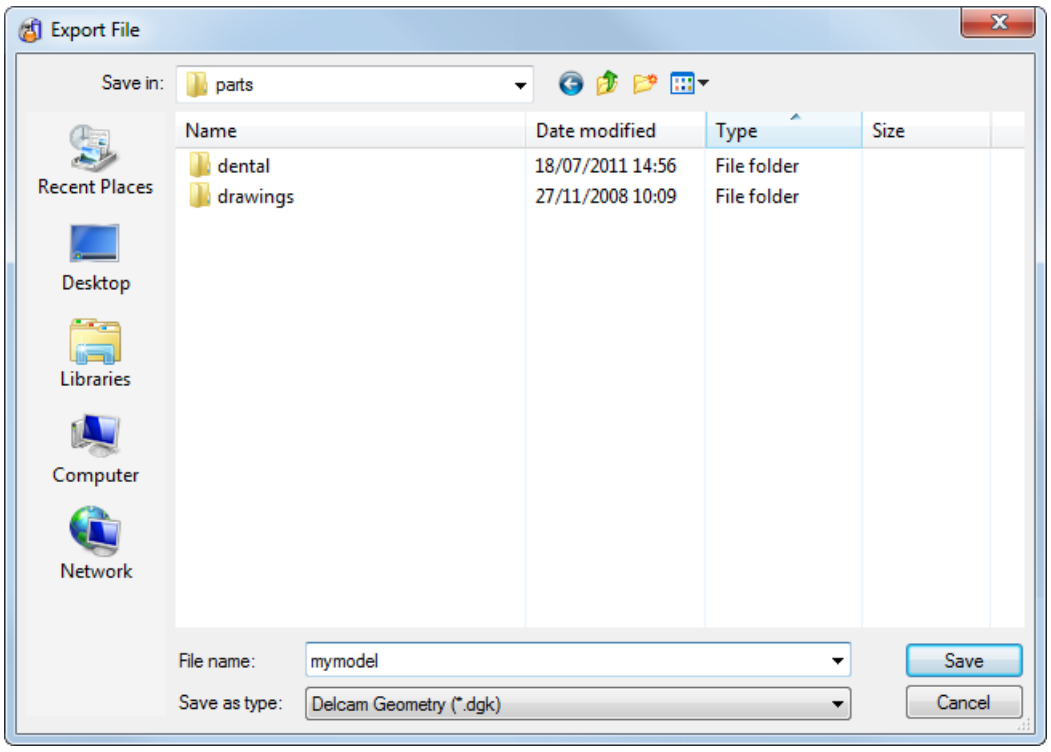
- if you are exporting a model that includes at least one drawing, page 2 is displayed (see page 101).
- if you are exporting a model that does not include drawings, page 3 is displayed (see page 102).
- if your model includes workplanes, page 4 is displayed (see page 103).
- if you are exporting a model that includes mutiple drawings, page 5 is displayed (see page 103).

5 Select the appropriate options on the export pages that are displayed.

6 Click **Finish** to complete the export.

Export File dialog

Use this dialog to export files in the format selected on the Export dialog.



1 Select the correct **Save in** folder. If necessary, use the following to locate the required folder:

- Click  to go up one level in the folder structure.
- Click  to create a new folder.
- Click  to display a menu containing options on how to display the files in the dialog.

2 Enter the **File name**. If you enter a file name with no extension, the **Save as type** extension is automatically added.

If you enter a filename with a valid export extension, the **Save as type** extension type overrides the type entered in the file's name.

3 Select the required export file type from the **Save as type** list (see page 198).

4 Click one of the following:

- **Save** to export data as specified.
- **Cancel** to remove the dialog from the screen without exporting any files.

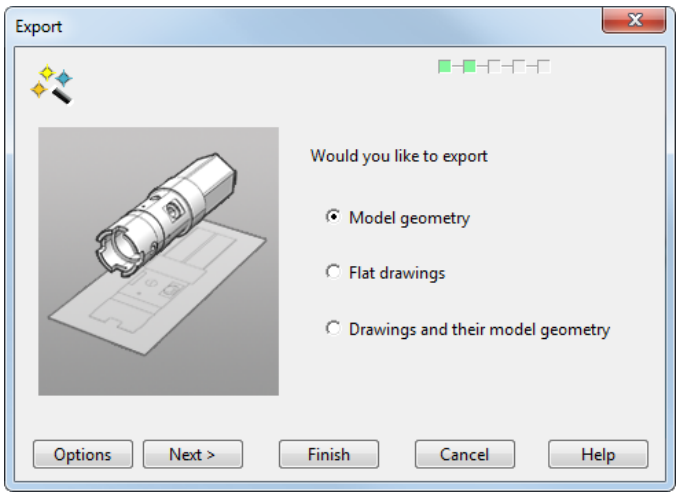
You can use either the PowerSHAPE internal translators or Delcam Exchange to export data (see page 197).

You cannot export badly trimmed surfaces. PowerSHAPE tells you if badly trimmed surfaces are detected on export and gives you two options:

- Export the selection with the offending surfaces untrimmed.
- Abandon the export. This allows you to fix the surfaces using the trim region editing tools.

Page 2 - Exporting a model that includes drawings

Use this page to export a model that includes drawings.



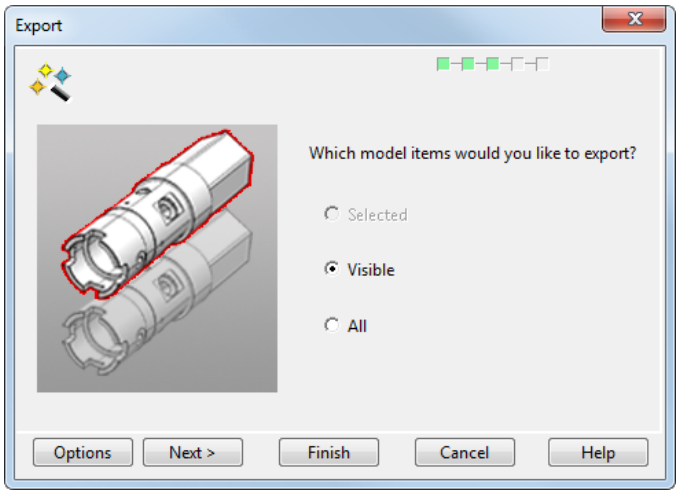
1 Select one of the options to specify the items to be exported:

- model geometry only
- drawings
- drawings and associated model geometry

2 Click **Next** or **Finish**.

Page 3 - Exporting a model

Use this page to specify the model items that you want to export.



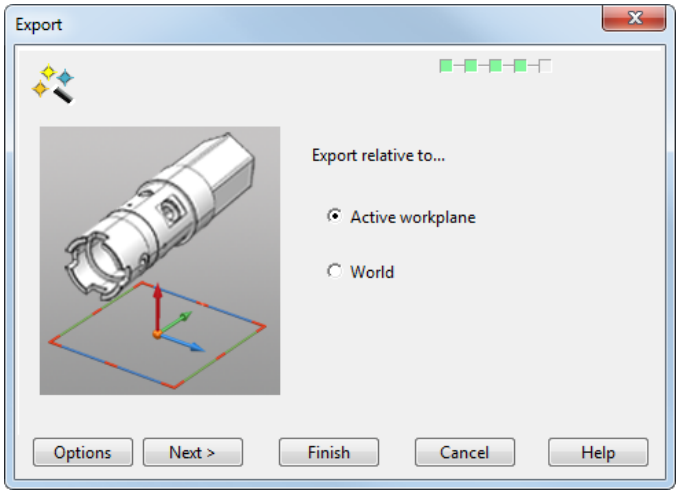
1 Select one of the options to specify the items to be exported:

- If you pre-selected some items, then **Selected** is the default option.
- **Visible** exports items that are visible on the screen; if an item is blanked it is not exported. This option is useful for exporting the workplanes with a model.
- **All** is the default option and is automatically selected if there are no pre-selected items.

2 Click **Next** or **Finish**.

Page 4 - Exporting workplanes

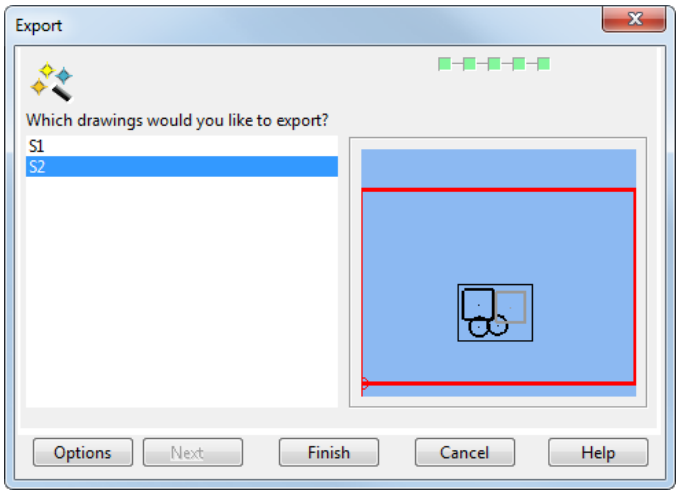
Use this page to specify the workplane to be used at export.



1. Select the option to be used when exporting:
   - Export relative to **Active workplane**.
   - Export relative to **World**.
2. Click **Next** or **Finish**.

Page 5 - Selecting a drawing to export

Use this page to select the drawings to be exported. This page is displayed only if there are multiple drawings associated with the model.



1. Select the drawings to be exported.
2. Click **Finish**.

Importing data into a model

When you import a model, it is automatically opened as a new model. However if you import any other file, it is opened in the current model window.

When you import a file, the model is zoomed to full, so the geometry is visible. This is particularly useful if the items are not close to the origin.

You can use one of the following methods to import data:

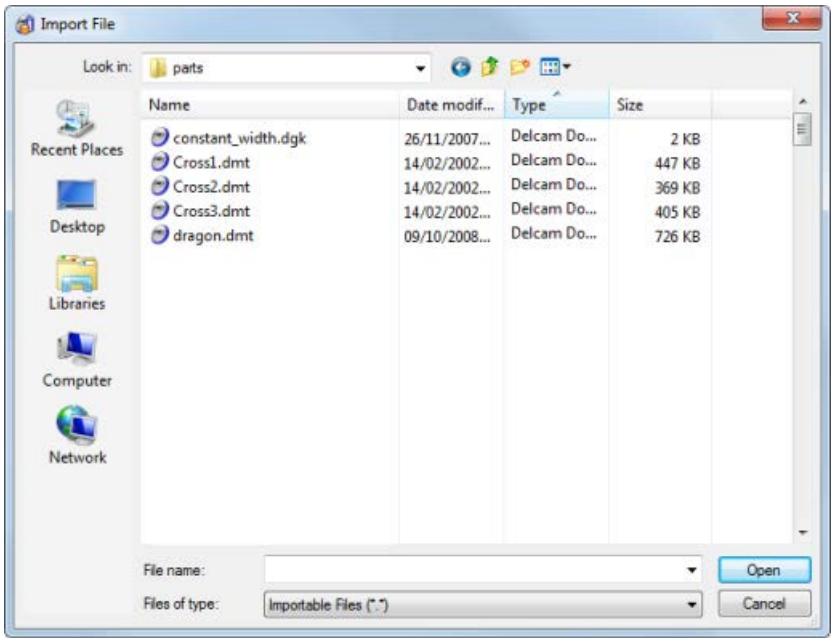
- Select **File > Import**.
- Click Import (see page 104) on the **Main** toolbar.
- Use the Advanced Import options (see page 208) to set up the data.
- Drag and drop files into the graphics area to import them.

For advanced modelling concepts, see:

- Advanced Import options
- Importing data without a licence (see page 216)
- Importing Materials (see page 218)

Using the Import option

1. Select **File > Import**.
2. Browse to the folder containing the file you want to import.



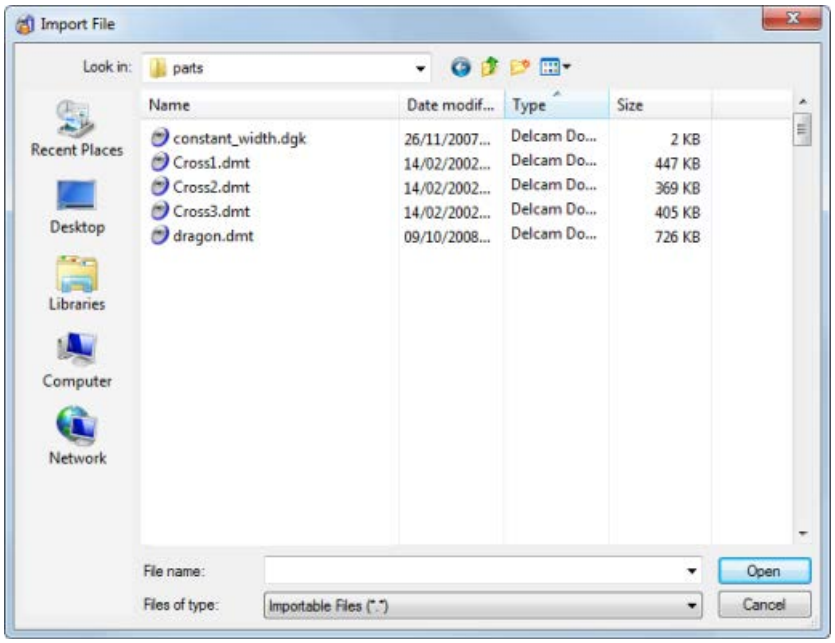
3. Click **Open**.

   The file is imported and opened.

For further details, see Importing data into a model (see page 104).

Import dialog

Use this dialog to select files to import.



1. Select the correct **Look in** folder. If necessary, use the following to locate the required folder:
   - Click  to go up one level in the folder structure.
   - Click  to create a new folder.
   - Click  to display a menu containing options on how to display the files in the dialog.
2. Enter the **File name**.
3. Select the required import file type from the **Files of type** list (see page 209).

   This displays the filter pattern which filter the file names of the current directory. By default, the pattern is *.* which displays all files. This selector lists the default filters for the file types that can be imported. You can select a filter from the list. When a filter is selected, files satisfying that filter are displayed.

   Files of type *is used only to help find files more easily. It does not determine the type of file to be imported. When a file is imported, the program looks at the contents of the file to determine its type and not its name. For example, if* ***Files of type*** *is set to *.pic, you can still import an IGES file by typing in the name in the* ***File name*** *text box. If an IGES file is called file.pic, this filter will help find it, but the file will still be imported as an IGES file.*

When importing IGES files, the program uses its own default tolerance values. You can change the defaults on the **Options** dialog before importing. The new values affect only the current model. Remember, large tolerances produce poor quality surfaces and small tolerances make the application run slowly. You may need to experiment to find the best balance for your models.

4 Click one of the following:

- **Open** to import data as specified.

  When you import certain files, a log file is generated containing details about the import. This log file is added to either the directory containing the imported file or the temp directory.

  If you import a model that has the same name as one already in the model database, its name is appended with [n] where n is an integer. The integer starts at 1 and increases as other models with the same name are imported. When such a model is imported, you need to open it using the Open Model dialog. (see page 69)

  Generally, models are automatically opened in the program when they are imported.

- **Cancel** to remove the dialog from the screen without importing any files.

*If you import files into a new model, the name of the last imported file (minus its extension) is automatically displayed as the **Model Name** in the **Save Model As** dialog when you save the model. For example, if the name of the last file imported is handle.igs, the **Model Name** is handle in the **Save Model As** dialog. This name is also displayed in the banner of the model's window, next to the new model's name. For example,* NEW_MODEL_1(handle).

You can use either PowerSHAPE's internal translators or Delcam Exchange to import data. For further details, see Using Delcam Exchange to translate data (see page 197).

What is PowerSHAPE-e?

PowerSHAPE-e is a free version of PowerSHAPE that enables you to design and share your design with others.

When you are happy with your design, you can export it to another application using the voucher system.

Before installing and using PowerSHAPE-e, please check the hardware and operating-system requirements on the Delcam website (http://www.delcam.com/hardware-updates).

Exporting a model from PowerSHAPE-e

To export a model from PowerSHAPE-e:

1. Create your model.
2. Save your model.
3. Export your model.

   The **Voucher Wizard** (see page 108) is displayed.

When you have purchased a voucher, you can do any number of exports of that model; if you change your model, you must buy a new voucher.

Importing an e-model to PowerSHAPE

To import an existing e-model into PowerSHAPE:

1. Select **File > Open** from the menu.

   The **Open Model** dialog is displayed.
2. Select the file you want to open and click the **Open** button.

   The **Voucher Wizard** (see page 108) is displayed.

Voucher Wizard

Select **Tools > Voucher > Manage Account** to display the **Voucher Wizard** to manage your account and make a deposit into your account.

Use the pages of the **Voucher Wizard** to manage your account:

Registration (see page 108)

Login (see page 109)

Account Deposit (see page 109)

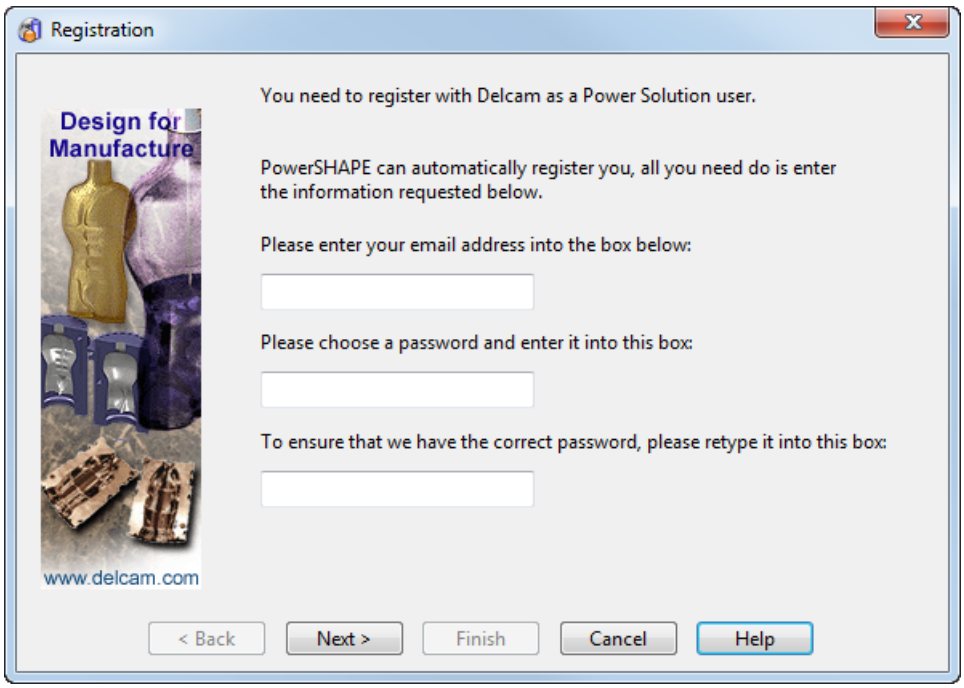
Address (see page 110)

Card Details (see page 111)

Deposit Receipt (see page 111)

Voucher Wizard - Registration

The first time you use the Voucher Wizard the **Registration** page is displayed.

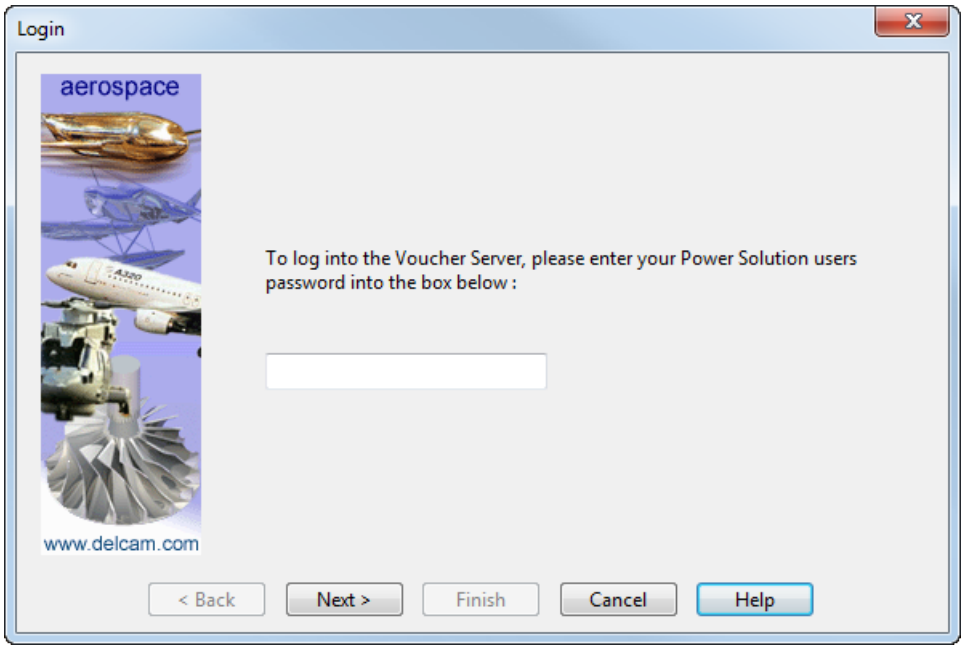


1 Enter the details requested. When you have entered information, the **Next** button becomes available.

2 Click **Next** to display the **Login** page (see page 109).

Login

To see your remaining credit, or purchase additional vouchers, you must connect to the Voucher Server.

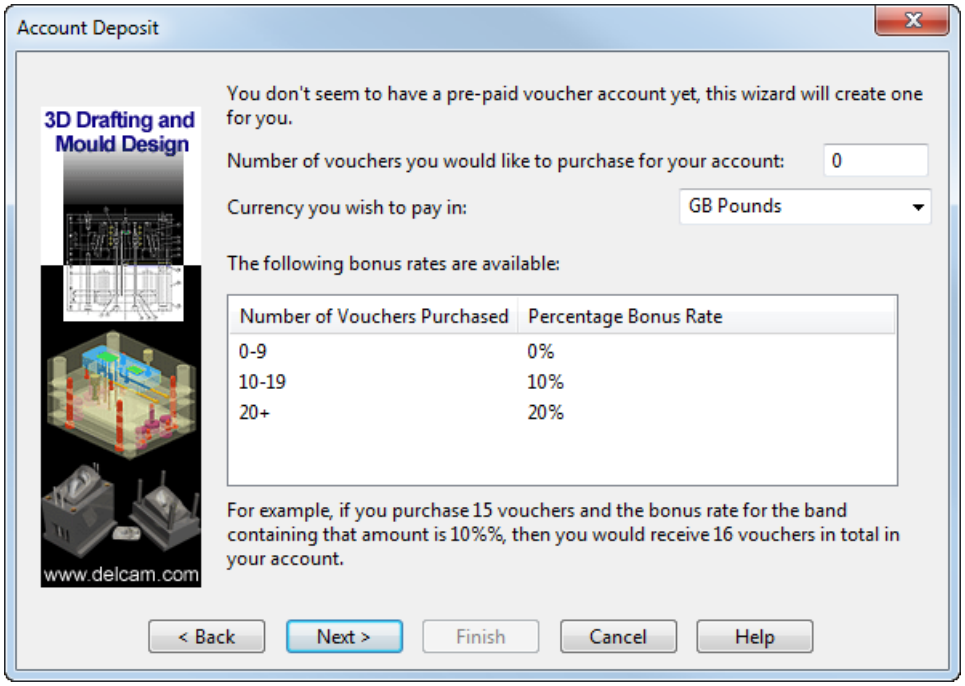


1 Enter your password.

2 Click **Next** to display the **Account Deposit** page (see page 109).

Account Deposit

Use this page to buy additional vouchers. The table helps you calculate the bonus credit you will acquire (if any), based on the amount of your deposit.

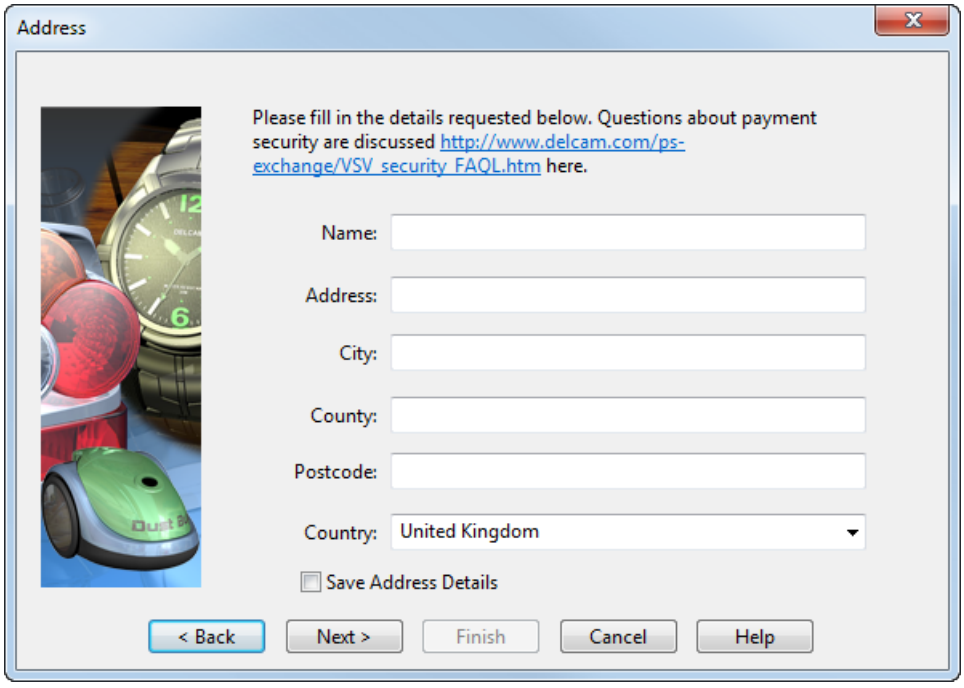


1 Enter the amount of credit you wish to purchase and select the currency.

2 Click **Next** to display the **Address** page (see page 110).

Address

Use this page to enter your address.

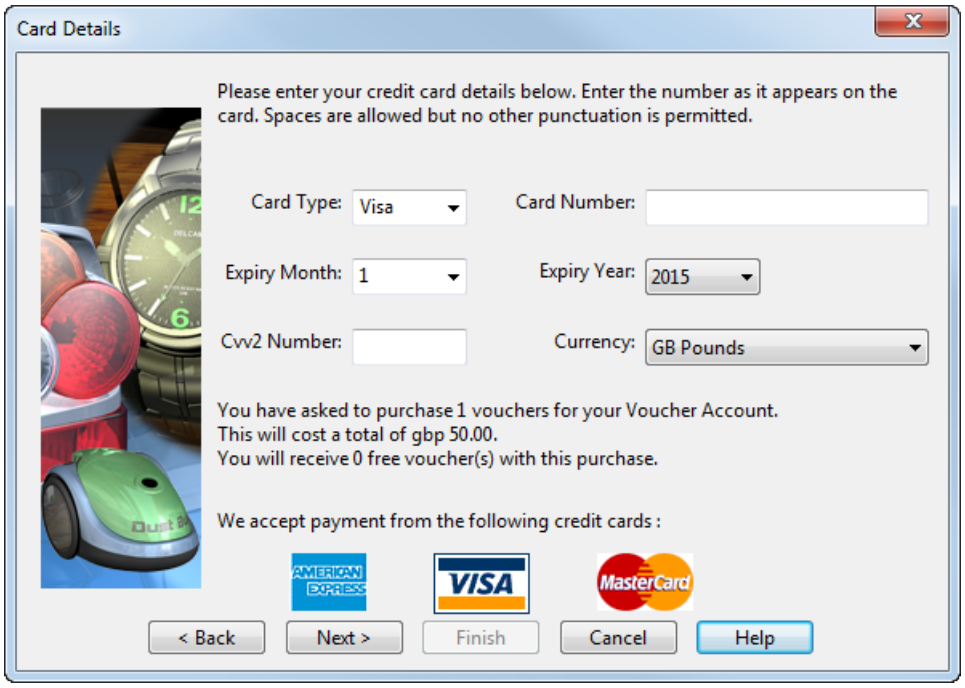


1 Enter your name and address details in the dialog.

2 Click **Next** to display the **Card Details** page (see page 111).

Card Details

Use this page to enter your card details.



1 Enter your card details. The amount of money you asked to deposit in your account on page 2 of the wizard is displayed below the **Currency** list.

2 Click **Next** to display the **Deposit Receipt** page (see page 111).

Deposit Receipt

A confirmation is displayed. This shows the amount of credit you have purchased, your balance, and order reference number.

You can print a copy of the information on the page. This is your receipt.

The coordinate system

The coordinate system defines the position of every object within the global workspace. The coordinate system contains the three axes **X**, **Y**, and **Z** which meet in the centre of the workspace, at a position called the origin.

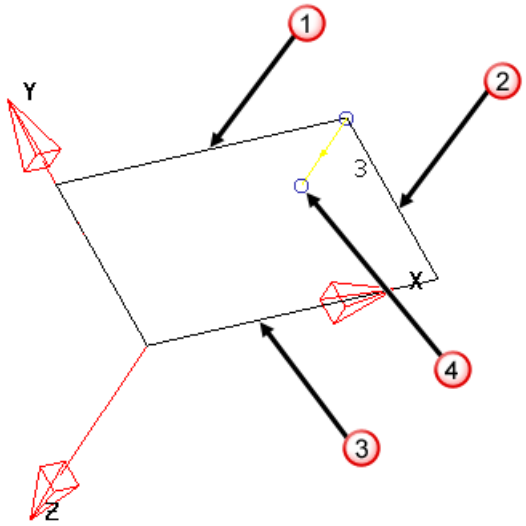
All models are created within the global workspace. Every position in a model is defined as three coordinates. For example, a position of 9,5,3 means 9 along the **X** axis, 5 along the **Y** axis and 3 along the **Z** axis. These coordinates are shown in the image below:

① **X** = 9

② **Y** = 5

③ **Z** = 3

④ **Position** 9,5,3

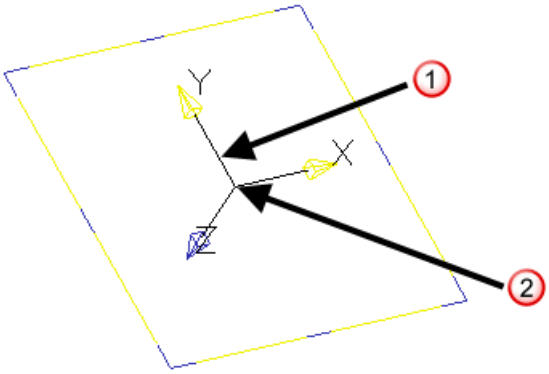


You create a model while working entirely within the global workspace, selecting the default drawing plane (also known as the principal plane) and entering coordinate values when necessary.

However, models can contain many angled faces and it is often easier to specify coordinates relative to a face rather than to the model as a whole. You can do this using workplanes (see page 112), shown in the image below:

① Dimensional axes **X**,**Y**,**Z**

② Origin



For further details, see Creating a workplane in the Wireframe modelling manual.

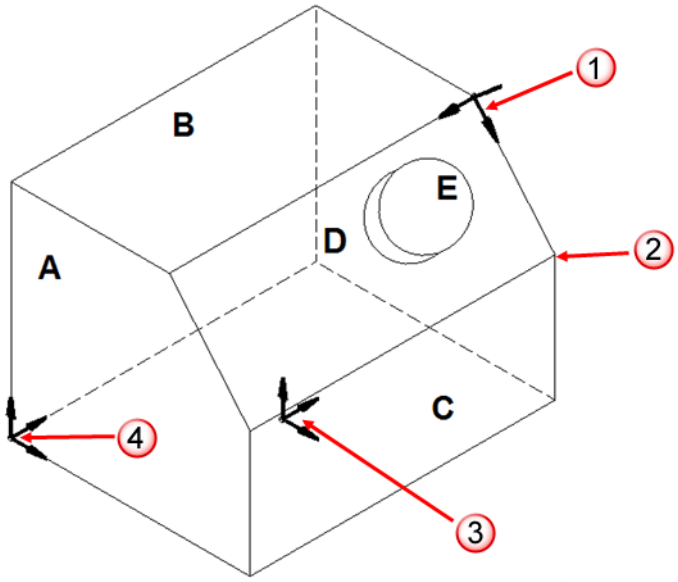
Workplanes and the coordinate system

A workplane is a graphical object defining a *local coordinate system*.

You can:

- have as many workplanes as you like.
- place workplanes anywhere in the model.
- rotate the axes to align conveniently with features of the model.

The image below shows example of a *local coordinate system*:



1 Local coordinate system (Workplane). Convenient for:

- Face **D**.
- Angled feature **E**.

2 Local coordinates 20,0,0. Global coordinates 437,62.8,28.7

3 Global workspace coordinate system for faces A, B, C.

4 Local coordinate system (Workplane). Ideal for faces A,B,C

Only one workplane can be active at a time, which is known as the *active workplane*. It is shown in red in the default colour scheme. Inactive workplanes are grey.

Any specified coordinates apply to this workplane, not to the global workspace. PowerSHAPE converts the local values to global values automatically and invisibly.

The three axes of a workspace listed below define three planes. These are known as the principal planes (see page 114).

- the XY plane lies through the X and Y axes
- the ZX plane lies through the X and Z axes
- the YZ plane lies through the Y and Z axes

Principal planes and the coordinate system

The **Principal plane** buttons  on the Status bar (see page 30) define which plane is the principal plane of the workspace. You can see this graphically as the axis of the rotation tracker updates in the modelling window. When clicking the mouse to enter positions in an empty space, the positions lie on the principal plane.

Each button displays the label of the axis which is normal to the principal plane.

Entering positions

To create an object, you need to specify its position in the workspace. Entering positions is one of the most common and important operations use the following methods for entering positions:

Entering positions using the cursor (see page 114)

Entering positions using the Intelligent Cursor (see page 58)

Entering positions using the status bar (see page 116)

Entering positions using dialogs (see page 118)

You can also create point objects to mark locations in your model.

Entering positions using the cursor

As the cursor moves across the principal plane (see page 114) (for example XY of the active workplane or global workspace), its coordinates with respect to the active workplane are shown in the status bar (see page 30). To enter a new position, move the cursor and click.

If the cursor is clicked within a certain radius of an existing object, the cursor snaps to the closest *key point* that exists on the object or snaps to a point of *intersection* between objects. This radius is known as the *hit radius*.

Entering positions using the Intelligent Cursor

The Intelligent Cursor shows additional information to help you to define the position you want when entering positions.

When you move the cursor over an object, construction labels (see page 59) are displayed by the Intelligent Cursor. These construction labels:

- are shown and hidden as you move the cursor over them.

- indicate what happens if you click while the label is displayed.
- make snapping with the cursor more accurate. The labels display when key points are within snap range of key positions in the model. You can click when a label (see page 59) is displayed to snap to the cursor at the labelled position.

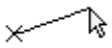
The Intelligent Cursor (see page 53) is enabled by default.

Construction lines

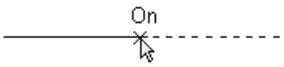
When you hover the Intelligent Cursor over an important key point (one that displays any of the construction labels (see page 59)), construction lines are created from that key point. You can now drag the cursor along a construction line and see the appropriate labels display. Clicking causes the position to snap to that point.

The following example draws a box:

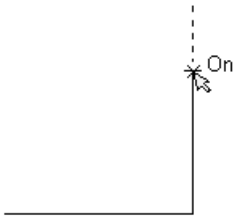
1. On the Main toolbar, select the **Line** button.
2. On the Line toolbar, select the **Continuous** button.
3. Click to start a line.

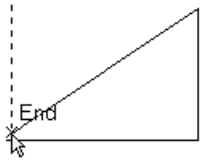
   Drag along a construction line and click.

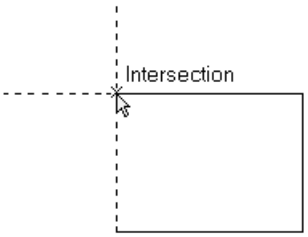
4. Drag along a 90° construction line and click. The line snaps to give a square intersection.

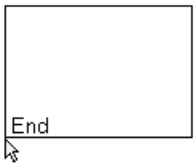
5. Drag the cursor down to the original start point and hover. The cursor displays **End** and the construction lines display.

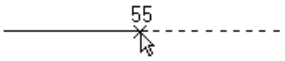
6 Drag along the vertical construction line until **Intersection** is shown.



7 Snap to the start and finish with one more click when **End** is displayed.



If the Intelligent Cursor Gridding is enabled, the label **On** in the above diagrams is replaced by a distance from the last point selected. For example:



This provides a convenient way to enter accurate dimensions.

*To disable construction lines temporarily, press and hold the **Shift** key.*

Entering positions using the status bar

You can enter Cartesian **(see page 116)** and polar **(see page 117)** positions in the **Data entry box** on the Status Bar (see page 30).

You can also type the coordinates in the graphics window and they are entered automatically into the **Data entry box** on the status bar. When you press the **Enter** key, coordinates are entered into PowerSHAPE.

For more complex constructions click the **Position** button on the Status Bar, to display the **Position** dialog (see page 119).

Entering Cartesian positions

The syntax for entering Cartesian positions (using the status bar) is as follows:

[workspace] X [Y [Z]] [units]

Square brackets **[ ]** show elements that are optional.

workspace specifies the origin from which to measure the coordinates.

You can use the following options:

- **world** = world workspace. You can abbreviate *world* to **w**.
- **absolute** or @ = absolute workspace. This uses the origin of the current workspace. You can abbreviate *absolute* to **abs**.
- **relative** = relative workspace. This uses the last point entered as the origin. You can abbreviate *relative* to **re**.

X [Y [Z]] - These are the *X Y* and *Z* coordinates and are entered in this order by default. If the second and third values are not given, they are treated as zeros.

The order of the *X Y* and *Z* coordinates is determined by the **Local point input** option on the **Workplane** page of the **Options** dialog. For further details, see Workplane options (*Menus and Toolbars*).

units - You can enter the type of units, for example, **mm** or **inches**. Default units are used if none are specified.

Examples of Cartesian coordinates are given below.

20 30 40

w 15

@ 25 60.8 mm

abs 50 mm

re -30.7 0 90

You can also enter polar positions. For further details, see Entering polar positions (see page 117).

Entering polar positions

The syntax for entering polar positions (using the status bar) is as follows:

> angle [angle_units] length [length_units]

Square brackets **[ ]** show elements which are optional.

> indicates that polar coordinates are being entered.

angle is the value of the angle.

angle_units is the unit type, for example, `degrees`.

length is the value of the length.

length_units is the unit type, for example, `mm`.

Examples of polar coordinates are given below.

> 45 30

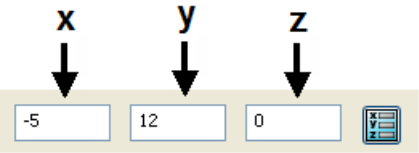
> 45 degrees 30 mm

> 30 2.5 inches

You can also enter Cartesian positions (see page 116).

Entering positions using dialogs

Where a dialog expects a position to be entered, separate text boxes for the X, Y, and Z axes are provided. Click the **Position** button to display the **Position** dialog (see page 119) for more complex point entry tools.



To enter a position as three coordinate values directly into a dialog:

1. Click in the **X** axis box to select it and enter the value
2. Press the **tab** key to select the **Y** axis box, or click in the box.
3. Enter the value and repeat to enter the value for the **Z** axis box.
4. Click **OK** on the dialog to store the values.

Position dialog overview

When it is not sufficient to enter positions using the cursor, the status bar, or dialogs, use the tools on the Position dialog (see page 119) to assist with more complex constructions. For example, you may want to specify a point at a particular proportion along an object.

the values for the X, Y, Z axes

Display the Position dialog (see page 119) using one of the following methods:

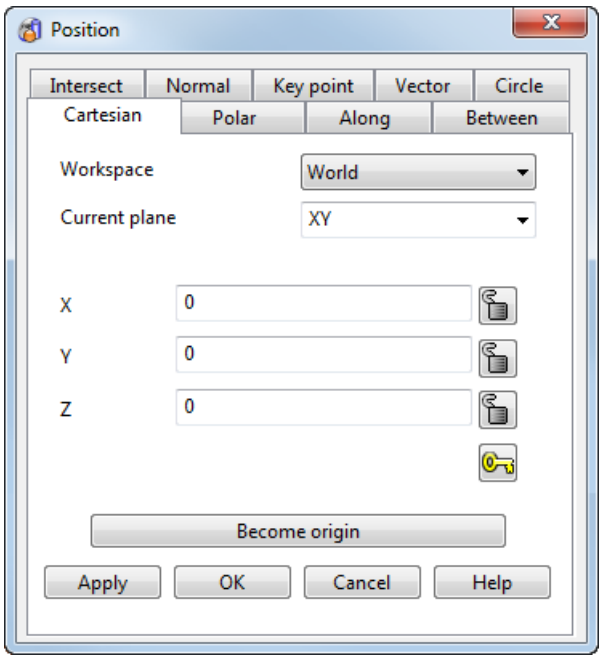
- Click **Position** on the status bar (see page 30)
- Click on a dialog.

For further details, see Entering positions using dialogs (see page 118).

*The **Position** button is active only when positions are required by the program. For example, when in object creation mode or when editing objects.*

Position dialog

Use the tools on the **Position** dialog to specify positions and constructions (see page 114).



Click a tab on the dialog to display the relevant page of position entry tools:

Cartesian (see page 119)

Polar (see page 121)

Along (see page 123)

Between (see page 125)

Intersect (see page 126)

Normal (see page 128)

KeyPoint (see page 130)

Vector (see page 132)

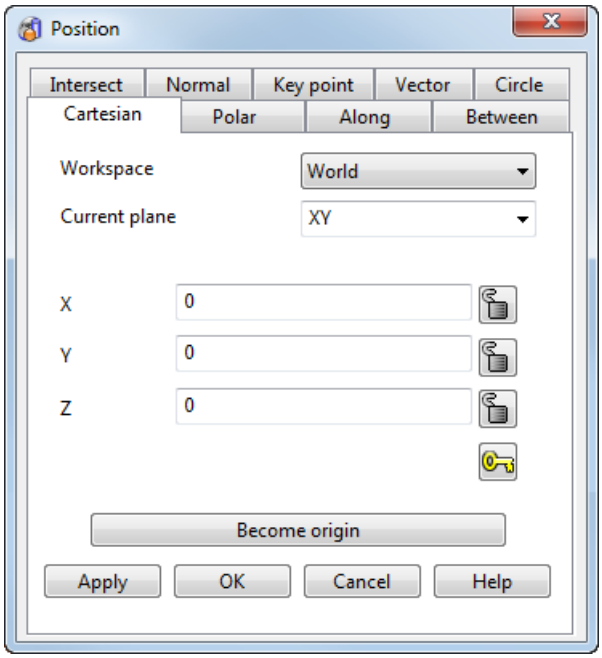
Circle (see page 134)

Cartesian tab

Use the **Cartesian** tab of the **Position** dialog (see page 119) to enter positions in the separate entry boxes provided for each axis. You can lock each individually by clicking  .

Usually you can enter a value in a box without affecting the other values. However if you have locked some components on the **Polar** tab (see page 121), then the values of the other fields may change to comply with the constraints created by the locks.

If you click a position in the graphics window, its Cartesian coordinates are displayed in the dialog.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Current plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

Locked and **Unlocked** — Click to toggle between **Locked** and **Unlocked**. Locking a value ensures that it remains fixed regardless of what happens to other components. The lock remains ON until you click again to unlock it. When an editing box is affected by a lock, it is dimmed to show that it is unavailable.

Unlock all — Click this button to unlock all of the values above.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

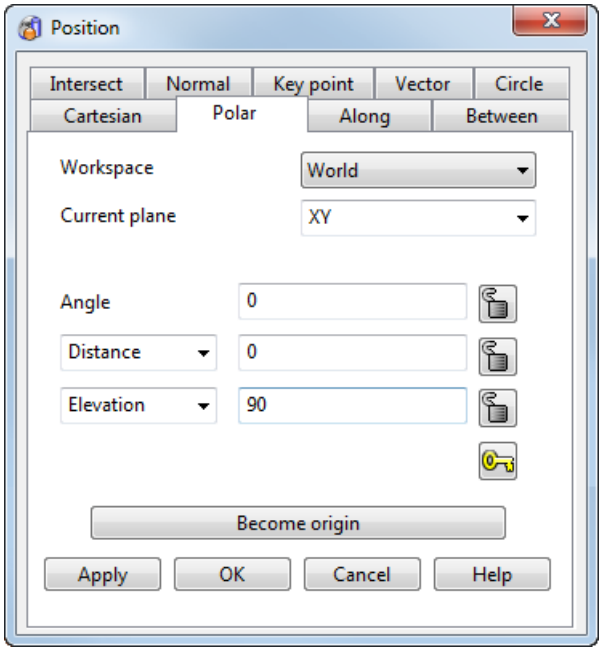
OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Polar tab

The **Polar** tab of the **Position** dialog (see page 119) is affected by the **Current plane** setting, which determines how the coordinates are calculated from the **Angle** and **Elevation/Height** fields.

Click a position in the graphics window to enter **Polar** coordinates into the dialog.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Current plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

Angle — Enter the polar angle from the origin in the currently selected plane in the selected workspace.

Distance — Select one of the following options from the drop-down list:

- **Distance** — Enter the distance from the workplane origin.
- **X**, **Y**, or **Z** — Enter a distance on the axis on the current active plane.

Elevation — Select one of the following options from the drop-down list:

- **Elevation** — Enter the angle of elevation above the active plane from the origin of the workspace.
- **Height** — Enter the distance perpendicular to the selected plane. For example, the **Z** component when the **XY** plane is selected.

 **Locked** and  **Unlocked** — Click to toggle between **Locked** and **Unlocked**. Locking a value ensures that it remains fixed regardless of what happens to other components. The lock remains ON until you click again to unlock it. When an editing box is affected by a lock, it is dimmed to show that it is unavailable.

 **Unlock all** — Click this button to unlock all of the values above.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

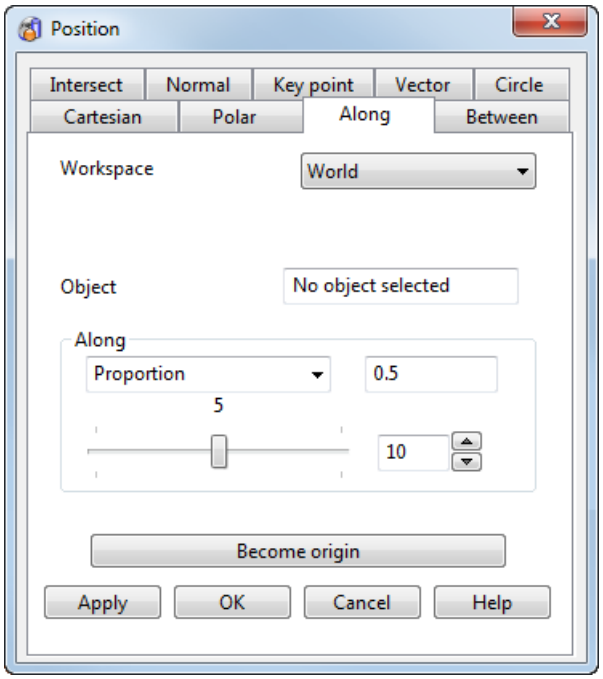
OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Along tab

Use the **Along** tab of the **Position** dialog (see page 119) to create a position at a chosen parameter along a selected object.

*You can select composite curves when you are using the **Along** tab of the **Position** dialog:*



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Object — Select an object using the mouse. When an object is selected, its name is displayed in the text box.

Select one of the following options and enter a value, or use the slider:

Proportion — Enter a measurement for the proportional distance from the start of the object. For example, enter **0.5** for half way along or enter **0.25** for a quarter (or 25%) of the way along from the start of the object.

Distance — Enter a measurement from the start of the object.

The **slider** indicates the portion from the start of the object. By default, the slider has 10 divisions. You can change the number of graduations on the slider by using  . Use the arrows to select a value or type a value into the box.

 **Locked** and  **Unlocked** — Click to toggle between **Locked** and **Unlocked**. Locking a value ensures that it remains fixed regardless of what happens to other components. The lock remains ON until you click again to unlock it. When an editing box is affected by a lock, it is dimmed to show that it is unavailable.

 **Unlock all** — Click this button to unlock all of the values above.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

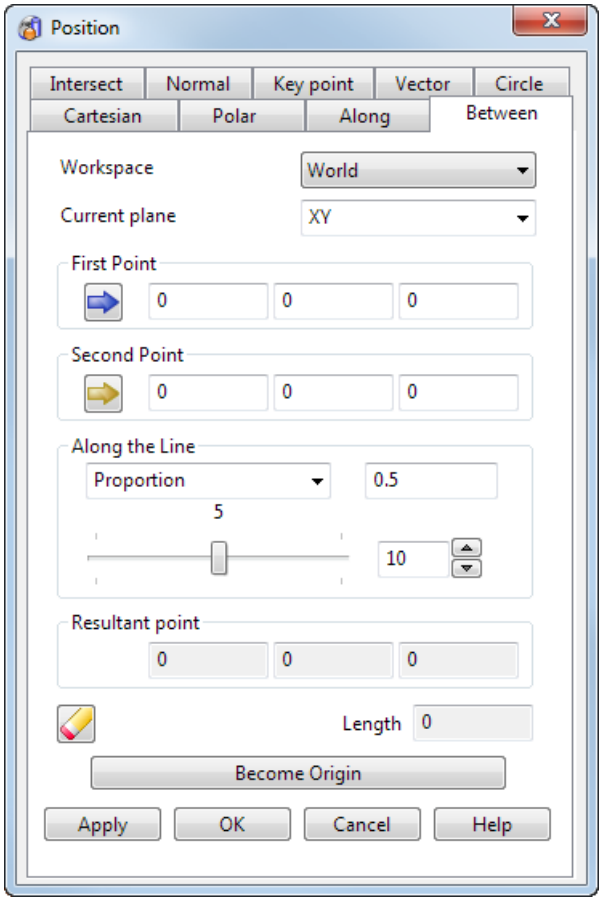
Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Between tab

Use the **Between** tab of the **Position** dialog (see page 119) in the same way as the **Along** tab (see page 123), but instead of using points at the ends of an object, select the start and end points which need not be on the same object. This allows accurate placement of a position between two connected or unconnected positions.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Current plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

Click two positions in the model to enter the values for the following options:

First Point — This displays the coordinates of the first click.

Second Point — This displays the coordinates of the second click.

Select one of the following options and enter a value, or use the slider:

Proportion — Enter a measurement for the proportional distance from the start of the object. For example, enter **0.5** for half way along or enter **0.25** for a quarter (or 25%) of the way along from the start of the object.

Distance — Enter a measurement from the start of the object.

The **slider** indicates the portion from the start of the object. By default, the slider has 10 divisions. You can change the number of graduations on the slider by using  . Use the arrows to select a value or type a value into the box.

Resultant point — This displays the coordinates of the *between* value as defined by the dialog.

 **Reset** — Click this button to reset the values on the page.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

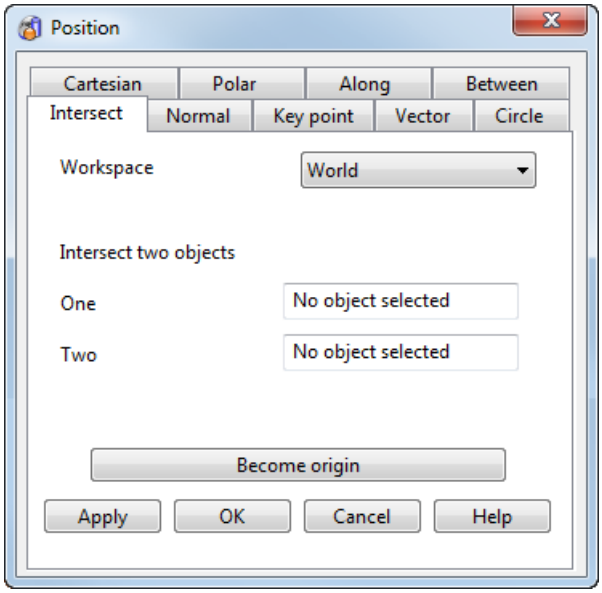
OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Intersect tab

Use the **Intersect** tab of the **Position** dialog (see page 119) to find the position where two objects intersect.

*You can select composite curves when you are using the **Intersect** page of the **Position** dialog:*



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Select two objects in the model to find the intersection. The names of the selected objects are displayed in the following options:

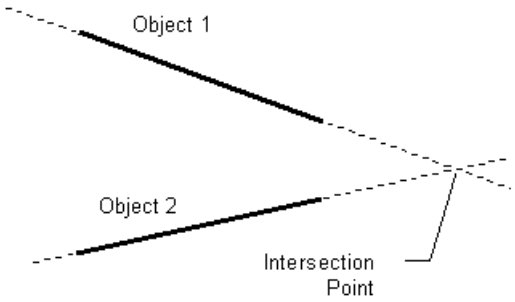
One — The name of one of the objects selected is displayed.

Two — The name of the other object selected is displayed.

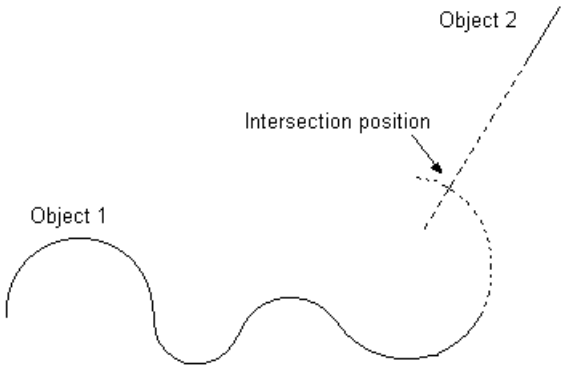
In order to find intersections, PowerSHAPE does the following:

- If the position does not lie on either object, the objects are extended.
- Lines are treated as if they are of infinite length.

The intersection point of two lines that don't cross is shown below.



- Arcs are treated as full circles.
- If there are two possible intersections, the intersection position is the one closest to the last mouse click.
- Curves are extended by maintaining the curvature at the end point.



Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

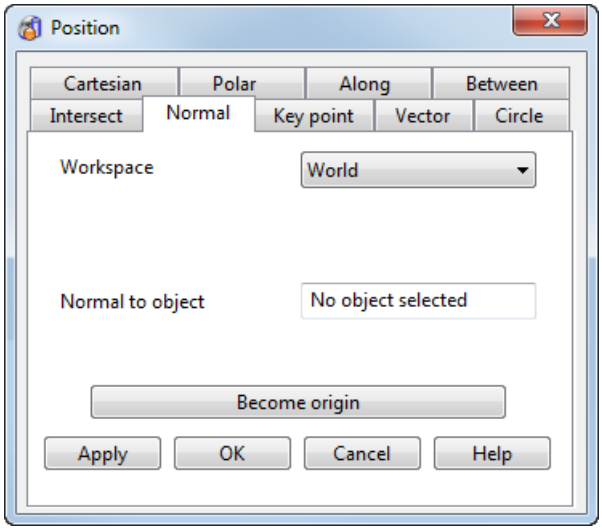
OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Normal tab

Use the normal tab of the **Position** dialog (see page 119) to enter a position which is perpendicular to an object from the origin of the workspace displayed on the dialog.

*You can select composite curves when you are using the **Normal** tab of the **Position** dialog:*



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Normal to object — Click the object you want the position to be *normal* to. The name of the object is displayed in the box and the cursor moves to the position *normal* to the object.

To allow positions to be found that do not lie on the object, PowerSHAPE does the following:

- Lines are treated as if they are of infinite length
- Arcs are treated as full circles.

An error dialog is displayed if a position cannot be found normal to the selected object.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

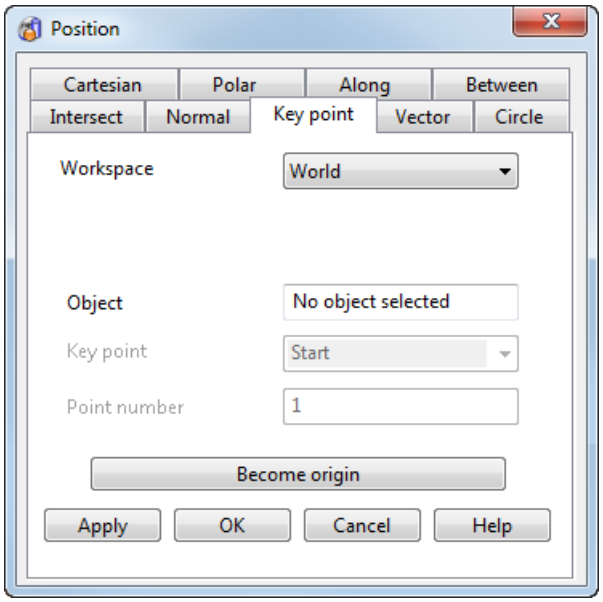
OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Key point tab

Use the **Key point** tab of the **Position** dialog (see page 119) to select a position at a specific key point on the selected object. This is useful where more than one object has key points at the same position and (for example) you want to snap to a hidden key point on a specific object. Key points are a quick and convenient method for selecting specific objects.

*You can select composite curves when you are using the **Key point** page of the **Position** dialog:*



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Object — Select an object. The name of the object is displayed.

Key point — This determines which point on the object is selected. Select from the following options:

Centre — This applies to the centre point of arcs and circles.

Start and **End** — This can apply to any object as they all have start and end positions.

Point Number — This is intended for use with curves that can have many positions along their length. The **Point Number** box activates where you enter a number.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

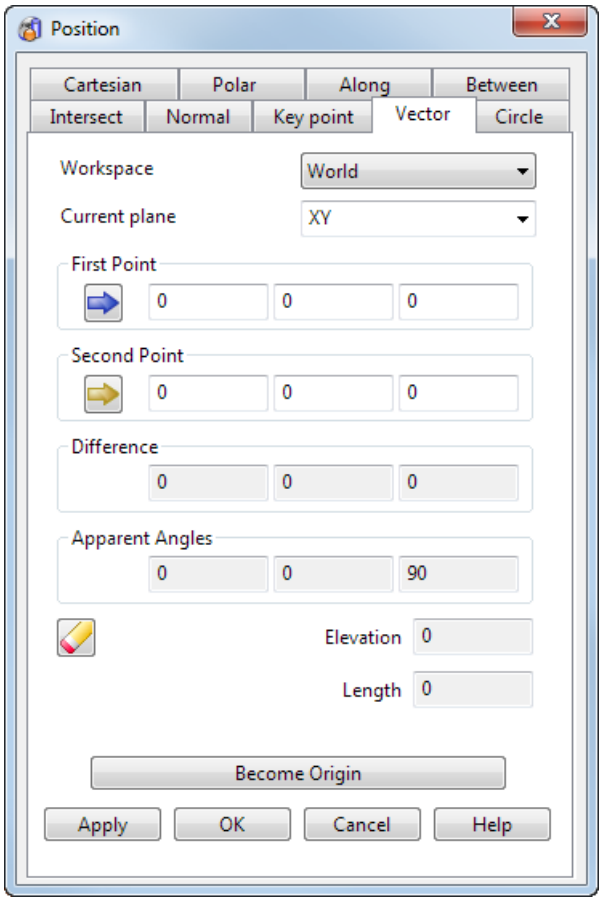
Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Vector tab

Use the **Vector** tab of the **Position** dialog (see page 119) to measure a vector between two points and use it to define a position. The offsets, angles, length, and elevation between the two points are displayed.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Current plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

You must already have a starting position from which the new position is entered.

First Point — Enter the first point of the vector or click on a point on the model to enter its coordinates. This point is displayed in pink on the screen (using the default colour scheme).

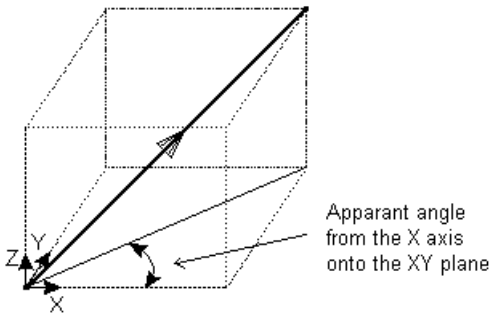
Second Point — Enter the second point of the vector or click on a point on the model to enter its coordinates. This point is displayed in red on the screen (using the default colour scheme).

When you click **OK** or **Apply**, a new position is entered by going along the vector from the start position.

Information from the two positions of the vector is displayed automatically in the boxes for the following options:

Difference — These are the differences in the X, Y, and Z values of the first and second points of the vector.

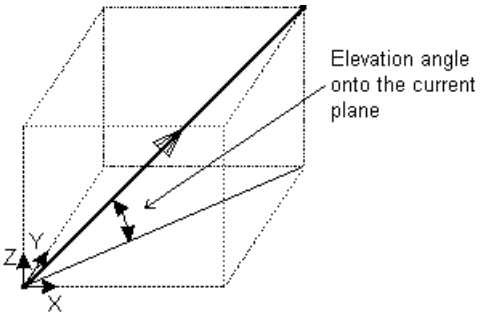
Apparent Angles — The *apparent angle* is measured between the projection of the vector onto a plane and the selected axis. If the axis is X, the plane is XY. For Y the plane is YZ and for Z it is ZX. An example is shown below of the apparent angle for the X axis:



Reset — Click this button to reset the values on the page.

Length — This is the true length (in 3D) between the points.

Elevation — This is the elevation angle of the new vector from the current plane.



Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

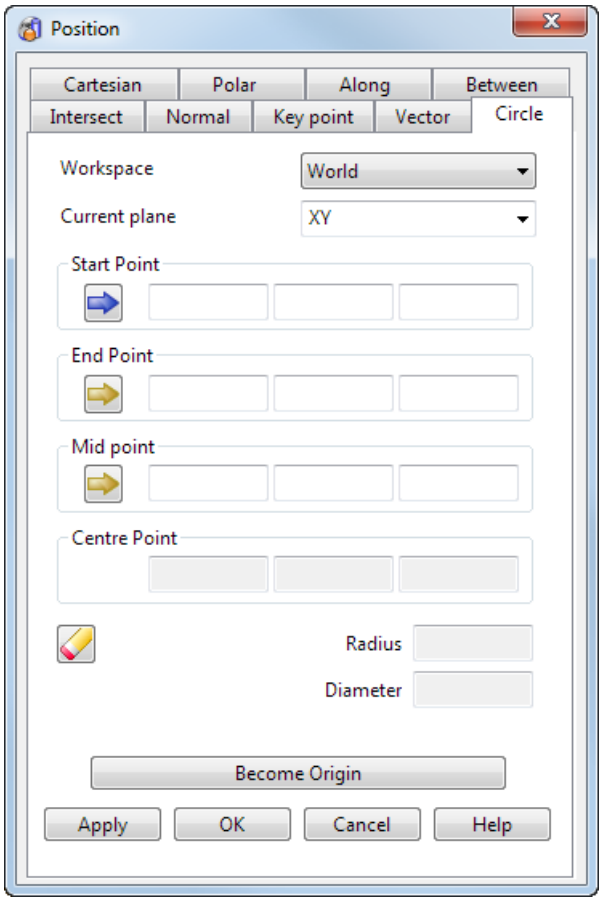
Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Circle tab

Use the **Circle** tab of the **Position** dialog (see page 119) to measure the dimensions of an arc using three points.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.

If any items are locked, the locks are discarded when you select a different workspace.

Current plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

StartPpoint — Enter the start point of the arc or click on a point on the model to enter its coordinates. This point is displayed in pink on the screen (using the default colour scheme).

End Point — Enter the end point of the arc or click on a point on the model to enter its coordinates. This point is displayed in pink on the screen (using the default colour scheme).

Mid point — Enter the mid point of the arc or click on a point on the model to enter its coordinates. This point is displayed in pink on the screen (using the default colour scheme).

Centre Point — The coordinates of the centre point of the arc are displayed.

 **Reset** — Click this button to reset the values on the page.

Radius — The radius of the arc is displayed.

Diameter — The diameter of the arc is displayed.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

*When you have opened the **Position** dialog in creation mode, clicking **OK** places the cursor in the circle centre.*

Entering angles

Enter an angle using degrees, minutes, and seconds in any dialog that expects an angle. The format is one of the following:

> **d;m;s**

>**-d;m;s** (for negative angle)

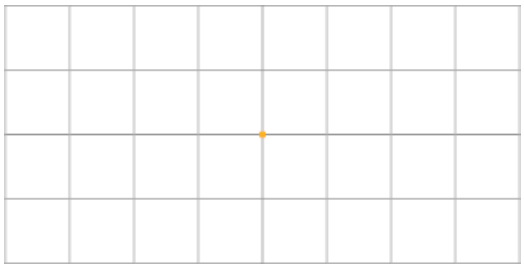
>**-(d;m;s)** (for negative angle)

where **d**, **m**, and **s** are numbers.

*This syntax does not work for the **Calculator** dialog, or for the calculator that is embedded in the **Parameter** dialog.*

The grid

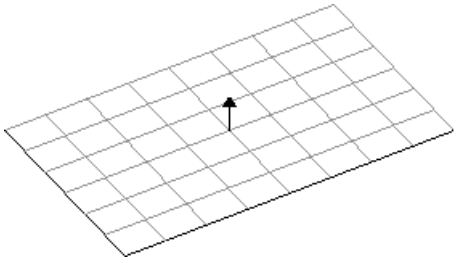
A grid is an array of points joined by lines. Use the grid as a guide to quickly enter positions by snapping to its points.



Using the grid

To use the grid turn the grid ON (see page 137).

The grid is drawn on the principal plane. If you change the principal plane, the grid automatically redraws on the new plane. The principal plane is displayed on the grid. Also displayed is the arrow at the origin of the current workspace. This shows the axis *normal* to the principal plane.



You can:

- Snap to the intersections of grid lines as if they are key points.

View the grid as either ruled or dotted lines by changing the Grid type **option on the** View **page of the** Options **dialogs**

The scale of the current grid is displayed in the Status bar (see page 30). The scale can be locked (see page 30) in the status bar to fix the scale. The scale is locked by default to a value of 10. You can set your own grid scale by typing your value into the grid scale text box in the Status bar (see page 30). The grid scale is locked automatically to that value.

To zoom in and out on a locked grid scale:

- Click **Zoom in**  to increase the scale.
- Click **Zoom out**  to decrease the scale.

To unlock the scale, deselect the **user defined scale** option on the **Blanking and Grid** page of the **Options** dialog. For further details, see Blanking and Grid in the Menus and Toolbars manual.

Turning the grid On and Off

To turn ON the grid

From the Status bar (see page 30), click **Grid on/off**  to draw the grid on the screen.

To turn OFF the grid

From the Status bar (see page 30), click **Grid on/off**  to remove the grid from the screen.

Using point-locking

When point-locking is ON, every position you enter using the mouse is locked onto the principal plane. For example, if the principal plane is *XY*, every point you enter using the mouse will have a *Z* coordinate of zero.

You can:

- Turn point-locking on (see page 137)
- Turn point-locking off (see page 138)

Turning point-locking on

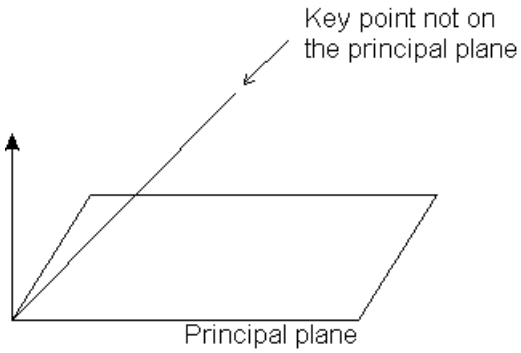
From the **Status bar**, click the **Point lock** button  . (It changes to a shut lock ).

Point-locking affects only points entered using the mouse.

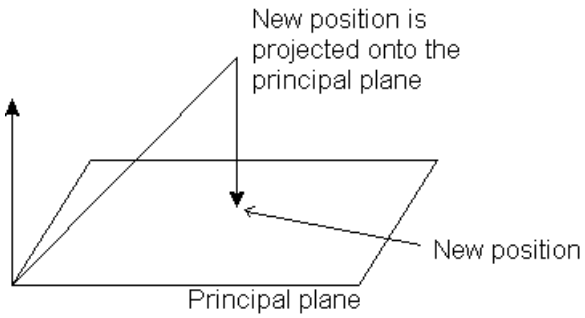
To enter a position that is not on the principal plane, you can:

- Use the Position dialog (see page 119).
- Type in the coordinates.

If you snap to a key point which does not lie on the principal plane (as shown below),



then the new position is projected onto the principal plane.



When you hover the intelligent cursor (see page 58) over such a key point, it changes to show that clicking the key point will project any new position onto the principal plane.

Turning point-locking off

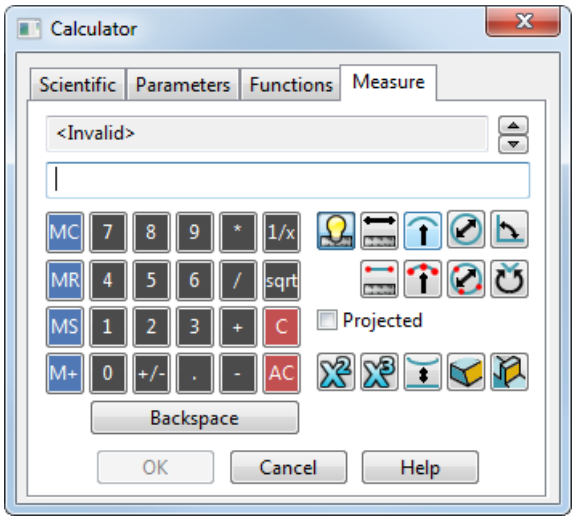
From the Status bar, click the **Point lock** button  . (It changes to an open lock ).

Calculator and measuring

Use the following sections to find information on the calculator and measuring:

What is the calculator?

The calculator enables you to make calculations while working on a model.



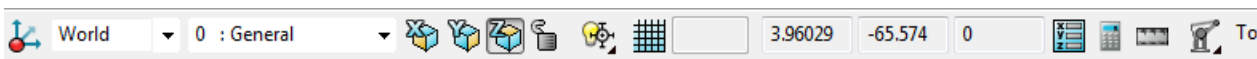
You can use the calculator in the following ways:

- Operate it in the same way as any pocket calculator.
- Enter complex expressions.
- Measure objects in the model.
- Interact data with dialogs, automatically entering the result into the selected boxes.

For further details, see Using the calculator (see page 139).

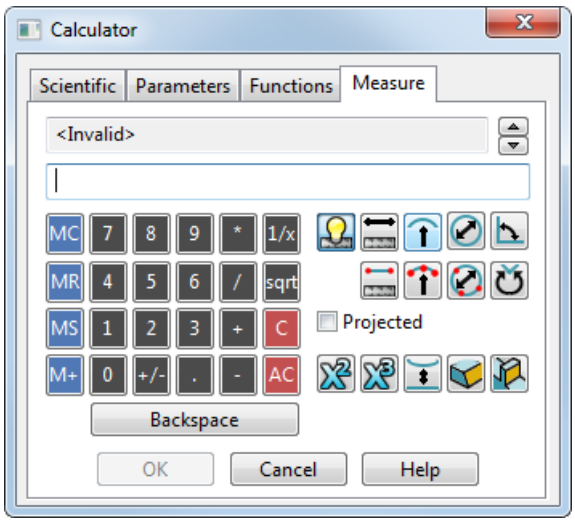
Using the calculator

From the **Status Bar**, click the **Calculator** button.



The Calculator dialog (see page 140) is displayed.

The calculator operates in the same way as any pocket calculator. You can click buttons to enter numbers and commands or you can use the keyboard numeric keypad.



To use the calculator from within a dialog:

1 Right-click in a box (used to enter numbers).

2 Use the calculator as required.

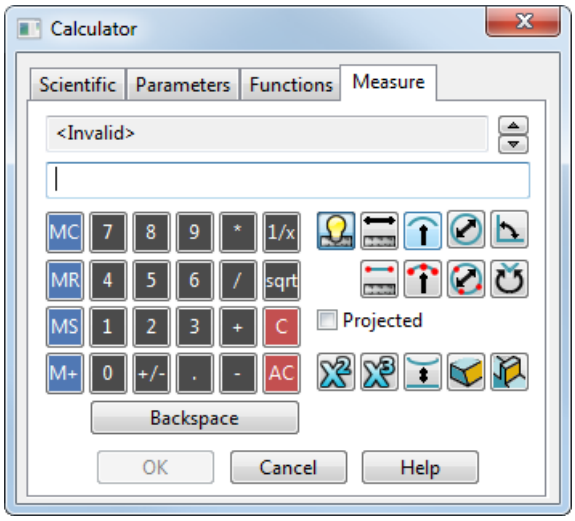
3 Click **OK**.

The results are automatically added to the box.

Calculator dialog

Use the calculator to make calculations while working on a model.

For further details, see Using the calculator (see page 139) and Using the calculator to measure (see page 146).



At the top of the dialog are tabs that provide the calculator with additional features for you to use to build expressions. You can switch between tabs as you calculate an expression.

- **Scientific** (see page 143)
- **Parameters** (see page 144)
- **Functions** (see page 145)
- **Measure** (see page 146)

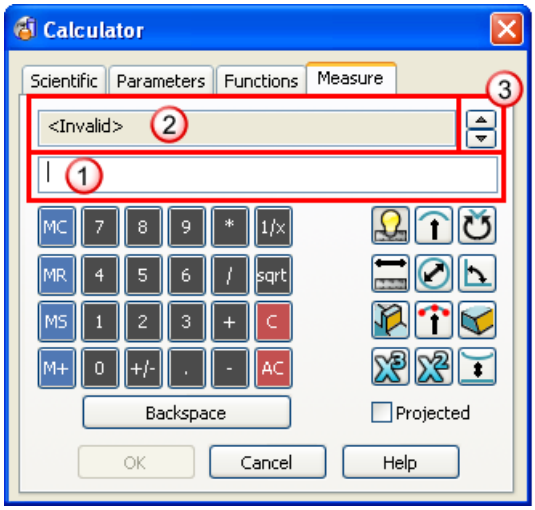
The calculator has two boxes:

1 Displays the expression.

2 Displays the results of the calculations.

3 Controls the number of decimal places shown on the result.

- - Increases the number of decimal places shown on the result.
- - Decreases the number of decimal places shown on the result.



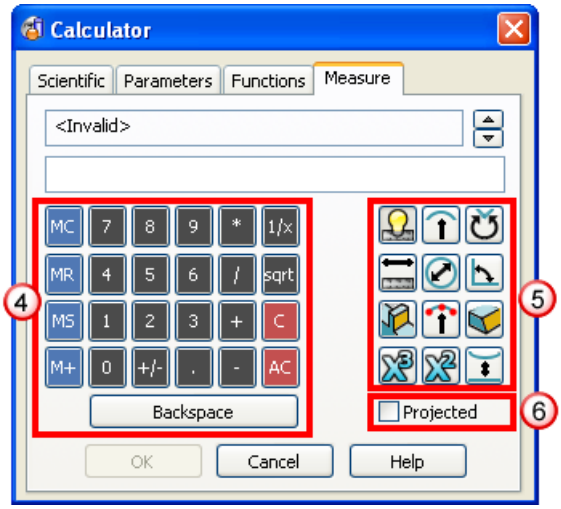
Use the keyboard or the numeric keypad to enter an expression for calculation. As you enter expressions, the results in the expressions box are updated.

The calculator also contains:

4 The numeric keypad. This is used in the same way as any other calculator to add, subtract, divide and multiply numbers. Memory function keys aid calculations.

5 Operations for the selected tab. The operations displayed correspond with the selected tab.

6 The **Projected** option. If On, the measurement is projected onto the current plane. Use this option with the measurement options. For further details, see Using the Projected option (see page 158).



Other keypad functions available are:

MC - Clears expressions stored in the memory.

MR - Retrieves a value from memory, for example after using **M+** to see the result.

MS - Stores the current expression into the memory.

M+ - Adds the current expression to the value already in memory.

The calculator remembers the memory value when it is closed and reopened.

1/x - Divides one by the current expression.

Sqrt - This calculates the square root of the current expression.

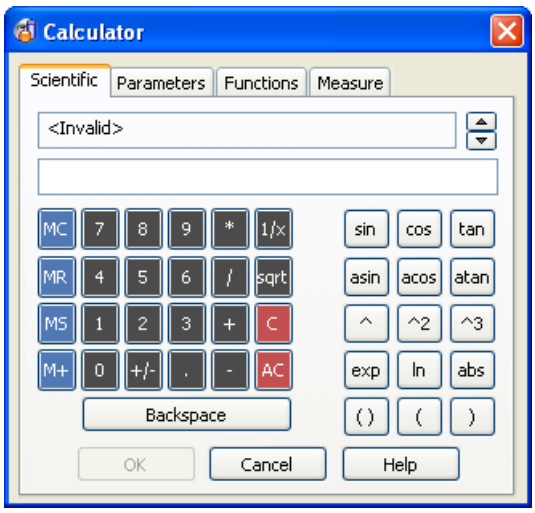
C - Removes the current expression.

AC - Deletes any expression stored in the memory and removes the current expression.

Backspace - Removes the character to the left of the cursor.

Scientific tab

Use the **Scientific** tab of the **Calculator** (see page 140) to calculate expressions using the scientific functions.



When you select a function, the current expression is enclosed in brackets and the function is performed on the value of the current expression.

sin - This calculates the sine of the current expression.

cos - This calculates the cosine of the current expression.

tan - This calculates the tan of the current expression.

asin - This calculates the angle whose sine is equal to value of the current expression.

acos - This calculates the angle whose cosine is equal to value of the current expression.

atan - This calculates the angle whose tangent is equal to value of the current expression.

^ - This is the "to the power of" sign.

For example, if the current value of the expression is **2** and you select **^** then enter the value **3**, the following calculation is performed:

(2)^3=2*2*2=8

^2 - This calculates the expression to the power of 2.

^3 - This calculates the expression to the power of 3.

exp - This calculates the exponential value of the current expression with respect to *e*, the base of the natural logarithms.

For example, if the current value of the expression is *2* and you select **exp**, the following calculation is performed:

exp(2) = e^2 = 7.389056

ln - This calculates the natural logarithm of the current expression.

For example, if the current value of the expression is *7.389056* and you select **ln**, the following calculation is performed:

ln(7.389056) = logarithm(7.389056) = 2

abs - This calculates the absolute value of the current expression. In other words, removes the minus sign.

For example, if the value of the expression is *-56.98* and you select **abs**, the new value is *56.98*.

( ) - Adds brackets around the current expression.

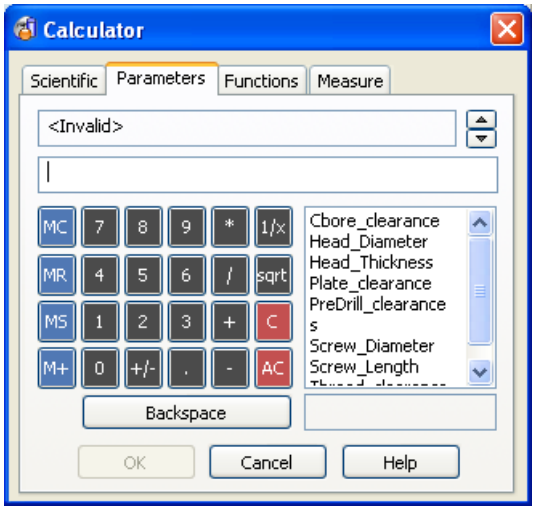
(- Adds a single left bracket to the current expression.

) - Adds a single right bracket to the current expression.

Parameters tab

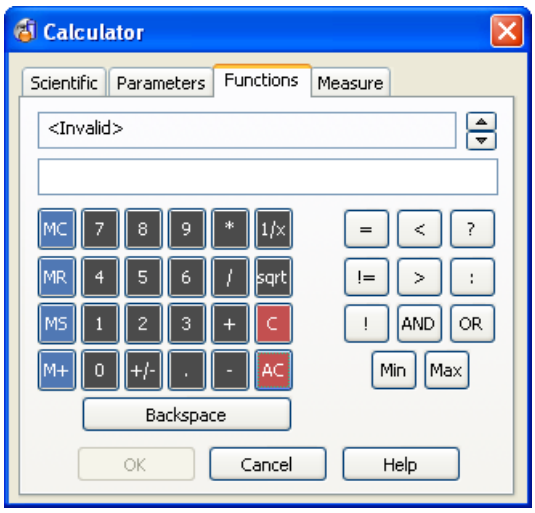
Use the **Parameters** tab of the **Calculator** (see page 140) to specify a parameter.

A parameter is a user-defined variable that is used to store numbers and expressions. For further details, see Using a parameter (see page 185).



Functions tab

Use the **Functions** tab of the **Calculator** (see page 140) to enter expressions using functions.



The result of a logical expression is 0 or 1, where 0 is false and 1 is true. You can combine buttons to give more operators.

Each logical operator is discussed below.

A == B
outputs 1 if A equals B and 0 otherwise

A != B
outputs 1 if A does not equal B and 0 otherwise

A < B
outputs 1 if A is less than B and 0 otherwise

A <= B
outputs 1 if A is less or equal to B and 0 otherwise

A > B
outputs 1 if A is greater than B and 0 otherwise

A >= B
outputs 1 if A is greater or equal to B and 0 otherwise

A AND B
outputs 1 if A and B are true and 0 otherwise

A OR B
outputs 1 if either A or B is true and 0 otherwise

! A
outputs 1 if A is false and 0 if true. This is known as the **not** operator.

test ? result_true : result_false
if test is true then output result_true otherwise output result_false.

Example 1

a>=b ? a+b : a-b

This outputs a+b if a>=b and a-b if a<b.

Example 2

a= (b>20)?10:((b>30)?20:30)

This example uses a nested expression to determine the value of a, based on the value of b

if b > 20 then

a = 10

else if (b > 30) then

a = 20

else

a = 30

Min - This determines the minimum value of a list of values in the expression.

The list of values in the expression must be of the form:

A1; A2; … ; AN

Max - This determines the maximum value of a list of values in the expression.

The list of values in the expression must be of the form:

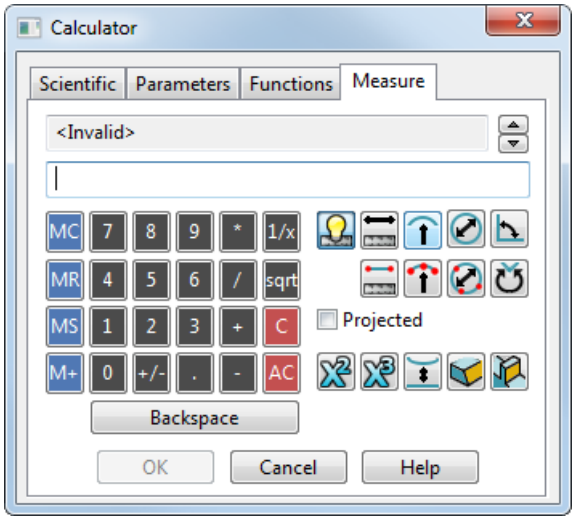
A1; A2; … ; AN

Using the calculator to measure

Use the **Measure** tab of the **Calculator** (see page 140) to create expressions using the measurement options. The measuring options are displayed on the right of the Calculator.

Straight line segments of curves are treated as lines.

For further details, see Using the calculator (see page 139).



Auto measure (see page 147)

Measure length (see page 148)

Measure length using two points (see page 149)

Measure radius (see page 149)

Measure radius using 3 points (see page 156)

Measure diameter (see page 149)

Measure diameter using three points (see page 150)

Measure minor angle (see page 150)

Measure major angle (see page 150)

Projected - Using the Projected option (see page 158)

Measure area (see page 155)

Measure volume (see page 155)

Measure minimum distance (see page 157)

Measure draft angles (see page 157)

Measure wall thickness (see page 156)

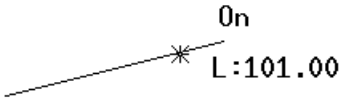
Measuring automatically

Use this option to intelligently measure lines, arcs, and the radius of curvature on arcs, curves, and surface curves.

1. Select **Model Analysis** on the **General edits** flyout.
2. Click **Calculator** (model analysis toolbar)
3. Click the **Auto-measure** button.
4. Hover the mouse over an object.

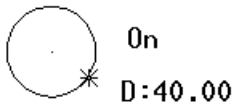
   If the object is a line or an arc, the intelligent cursor displays the intelligent measurement of that object. The intelligent measurement of lines and arcs are:

   - Line - length



   - Arc - radius

   - Full arc (circle) - diameter



   If a span of a curve or a surface curve has constant curvature, the radius of curvature is displayed when you move the cursor over the span.

   To enter that value into the calculator, click the object. The value is entered into the expressions box.

   If no intelligent measurement is associated with an object, then no value is displayed.

   When no value is displayed and you click a position on the screen, the calculator waits for you to enter a second position. As you move the cursor, a rubber-banded line displays from the first position to the cursor position. The length of the rubber-banded line is also displayed. Once you enter the second position, the shortest distance between the two points is entered into the expressions box.

Measuring the length

This option measures:

- the length of a line
- the span of an arc
- the length of a curve or a composite curve
- the distance between two points

To measure the length,

1. Select **Model Analysis**  on the **General edits**  flyout.
2. Click **Calculator**  (model analysis toolbar)
3. Click the **Measure Length**  button.
4. Either select the object or click two points to enter the value into the expressions box.

Measuring the radius

Use this to measure the radius of any arc or circle, and the curvature on curves and surface curves.

1. Select **Model Analysis**  on the **General edits**  flyout.
2. Click **Calculator**  (model analysis toolbar)
3. Click the **Measure Radius**  button.
4. If you move the cursor over an arc, its radius is displayed. If a span of a curve or a surface curve has constant curvature, the radius of curvature is displayed when you move the cursor over the span.
5. Click the object to enter the value into the expressions box.

Measuring the diameter

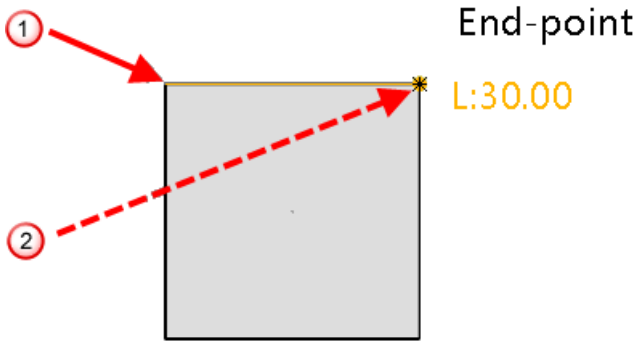
Use this to measure the diameter of an arc.

1. Select **Model Analysis**  on the **General edits**  flyout.
2. Click **Calculator**  (model analysis toolbar)
3. Click the **Measure Diameter**  button.
4. If you hover the mouse over an arc, its diameter is displayed.
5. Click the arc to enter the value into the expressions box.

Measuring the length using two points

1. Click first point  .

2 Hover over the second point ② to display the length. Click the point to display the length value in the dialog.

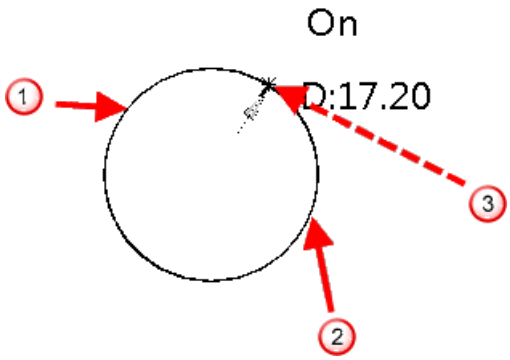


Measuring the diameter using three points

1 Click first point ①.

2 Click second point ②.

3 Hover over the third point ③ to display the diameter. Click the point to display the diameter value in the dialog.



Measuring angles

1 Select **Model Analysis** on the **General edits** flyout.

2 Click **Calculator** (model analysis toolbar).

3 Click the **Measure Minor Angle** button to measure minor angles or click the **Measure Major Angle** button to measure major angles.

4 Select from the following items:

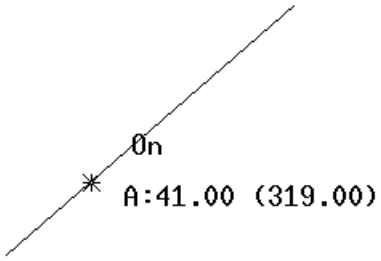
- One line (see page 151)
- Two lines (see page 151)
- Arcs (see page 152)
- One line and one position (see page 153)

- Three positions (see page 154)

Measuring angles - one line

1 Hover the cursor over the line.

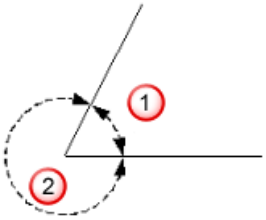
The minor and major angles are both displayed, where the type not required is in brackets.



2 Double-click the line in exactly the same position to enter the value into the expressions box on the calculator.

Measuring angles - two lines

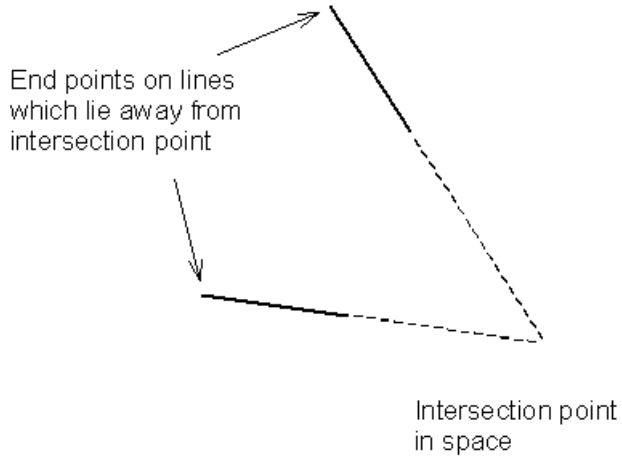
The minor and major angles are measured between the two lines as shown below.



1 Minor angle
2 Major angle

Select one line and then select the other line. The angle between the two lines is entered into the expressions box on the calculator.

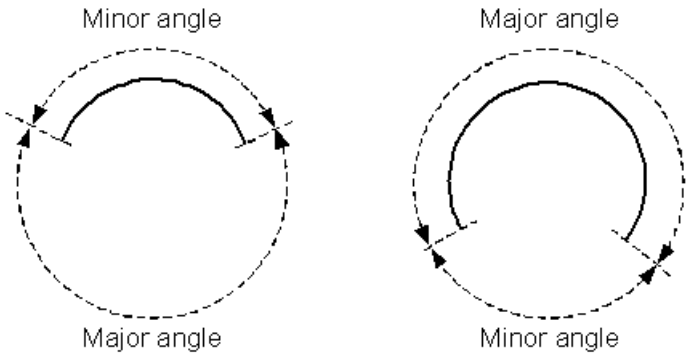
In the example below, there are two lines that do not lie on the same XY, YZ, or ZX plane, but intersect in space. The angle is measured on the plane that is defined by the intersection point and the end points of the lines that lay away from the intersection point.



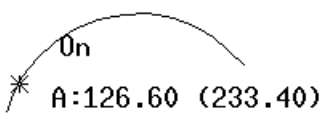
If you select two lines that do not intersect in space, no angle exists between two such lines. The position selected on the second line is treated as a position and the angle is calculated as described when a line and a position are selected.

Measuring angles - arcs

The minor and major angles are the spans of the sector of the arc and the missing sector of the arc, shown in the example image below:



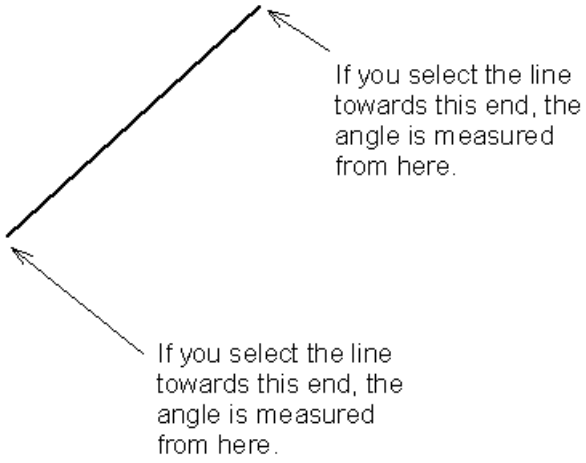
When you hover the mouse over an arc, both the minor and major angles are displayed, where the type not required is in brackets.



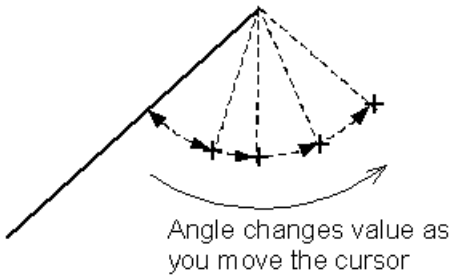
Measuring angles - one line and one position

1 Select a line.

Depending on which end of the line is selected, you will see a rubber-banded angle from the line to intelligent cursor.



If the top end of the line is selected, the angle is measured from the top part of the line. As you move the mouse cursor, the rubber-banded angle moves too.



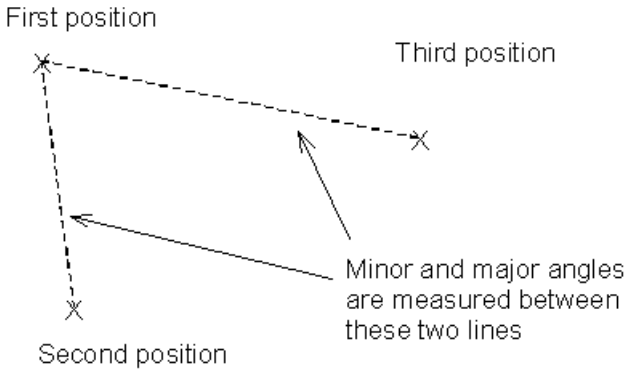
Use this rubber-banded angle as a guide to determine the angle you want to measure.

2 Enter the position. The angle is entered into the expressions box.

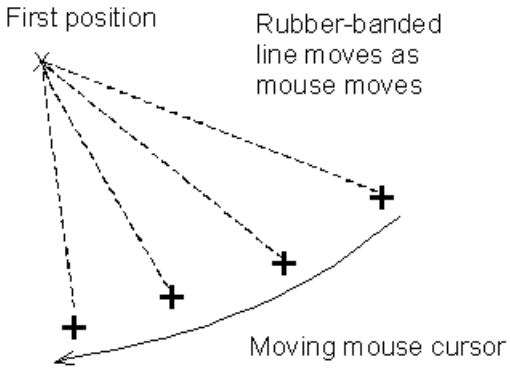
The angle is calculated in the plane made by the two end points of the line and the position.

Measuring angles - three positions

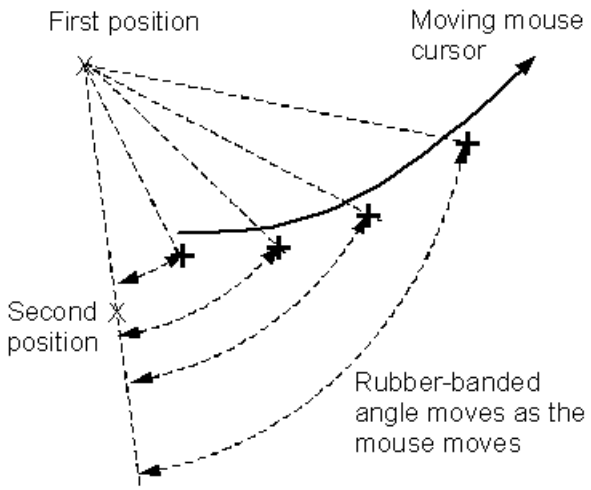
The three positions define two lines, shown in the image below. The minor and major angles are measured between these two lines.



1 Enter the first position, to display a rubber-banded line from the position to the mouse cursor.



2 Use the rubber-banded line to enter the second position. A rubber-banded angle is displayed, shown in the image below.



3 Choose the third position. The angle is displayed in the expressions box.

The angle is calculated in the plane made by the two end points of the line and the position.

Measuring the volume

Use this to measure the volume of surfaces, solids, or triangulated symbols.

1 Select **Model Analysis** on the **General edits** flyout.

2 Click **Calculator** (model analysis toolbar)

3 Click the **Measure Volume** button.

4 Select the surface or solid.

Measuring the area

Use this to measure the area of a composite curve, surface, solid, or triangulated symbols.

1 Select **Model Analysis** on the **General edits** flyout.

2 Click **Calculator** (model analysis toolbar)

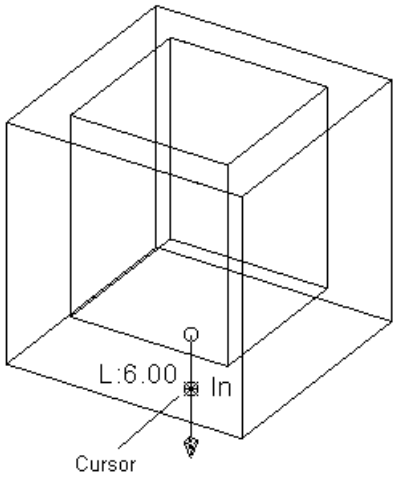
3 Click the **Measure Area** button.

4 Select either a composite curve or a surface or a solid.

Measuring wall thickness

This option measures the wall thickness of a solid.

1. Select **Model Analysis** on the **General edits** flyout.
2. Click **Calculator** (model analysis toolbar).
3. Click the **Measure Wall Thickness** button.
4. Move the cursor over the solid to display the wall thickness (see below).
5. Click the solid to enter the value into the expressions box.



Measuring the radius using 3 points

Use this to measure the radius of an arc using three points. It is ideal for measuring the radius of curvature on curves and surface curves.

1. Select **Model Analysis** on the **General edits** flyout.
2. Click **Calculator** (model analysis toolbar).
3. Click the **Measure Radius From 3 Points** button.
4. Click the first position to define the start point of an arc.

5. Click the second position to define the end point of an arc.

6 Click the third position to define a position on the circumference of the arc.

The value is entered into the expressions box.

Measuring draft angles

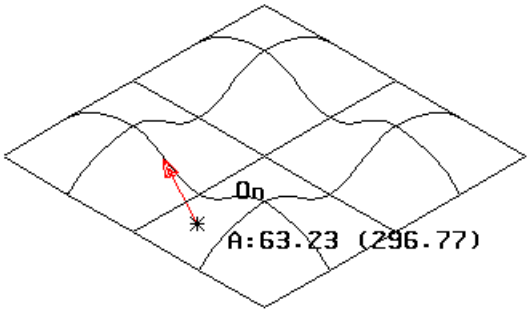
Use this to measure the draft angle of surface and solid points.

1 Select **Model Analysis** on the **General edits** flyout.

2 Click **Calculator** (model analysis toolbar).

3 Click the **Measure Draft** button.

4 Hover the mouse over a position on a surface or solid. The minor draft angle is displayed with the major draft angle in brackets. An arrow representing the surface normal of the position is also drawn.



By default, only draft angles of positions on the surface curves are measured. You can measure the draft angle of positions within a surface patch by holding down the **Shift** key. The draft angle is measured from the principle plane to the normal of the point on the surface under the cursor.

5 Click the position on the surface to enter the draft angle into the expressions box.

Measuring the minimum distance

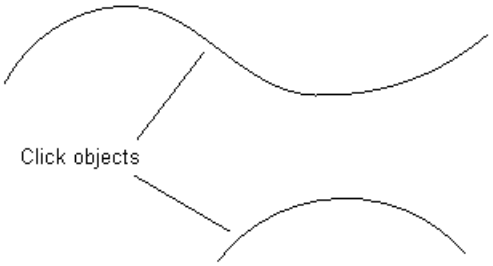
Use this to measure the minimum distance between two objects.

1 Select **Model Analysis** on the **General edits** flyout.

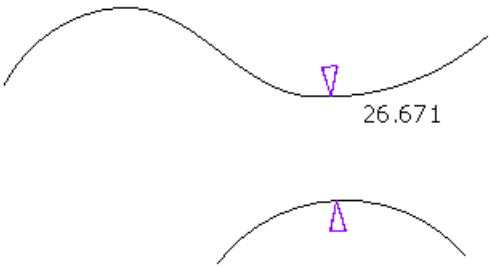
2 Click **Calculator** (model analysis toolbar).

3 Click the **Minimum Distance** button.

4 Click two objects to enter the value into the expressions box.



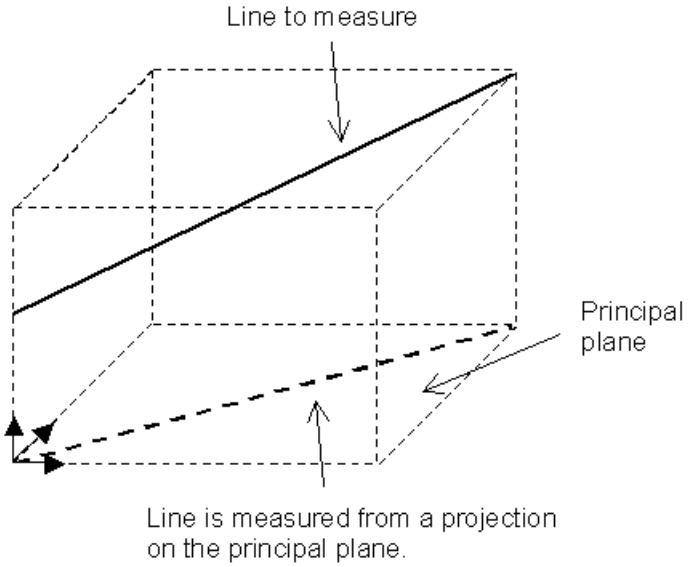
Arrows are drawn on the objects to show where the minimum distance is measured.



Using the Projected option

Use the **Projected** option on the **Calculator** dialog (see page 140) to take measurements from projections on the principal plane.

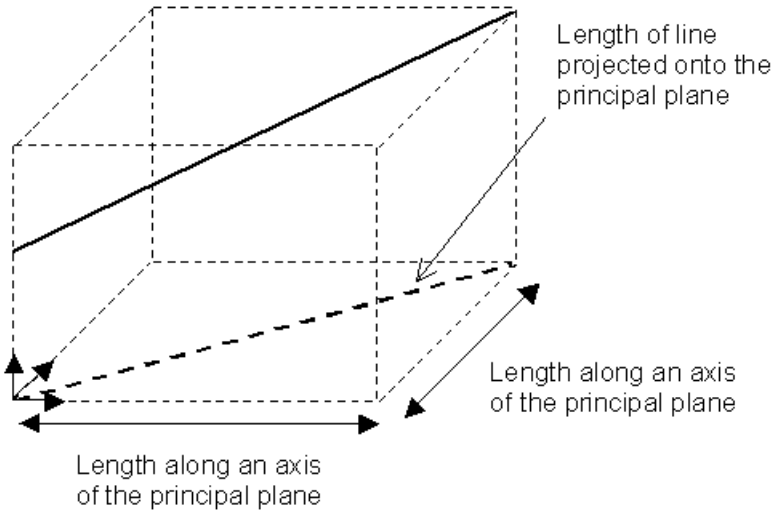
A line is measured from a projection on the principal plane, shown in the image below.



You can measure three values from the projection:

- The length

- The length along either of the axes of the principal plane.



To measure the length of the projection,

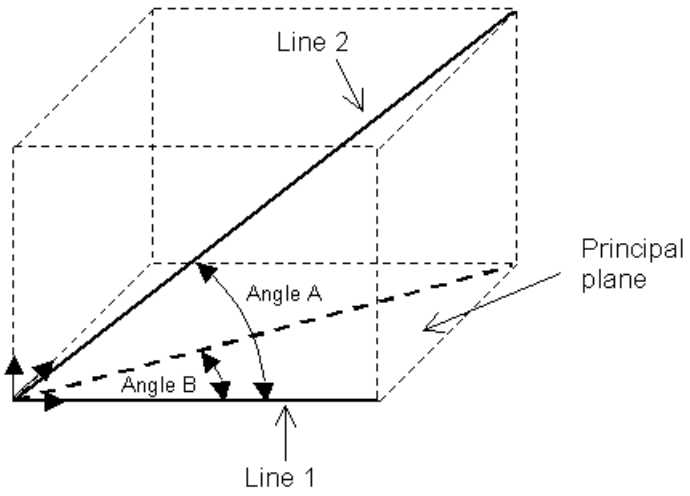
1. Hover the mouse over the line until a value displays.
2. Double click the line to enter the value into the expressions box.

To measure the length along the axis,

1. Click the line once.
2. Move the mouse in the direction parallel to the axis you want. As you move the mouse, a value displays.
3. Click the mouse to enter the value into the expressions box.

To measure angles of lines and positions that are projected onto the principal plane,

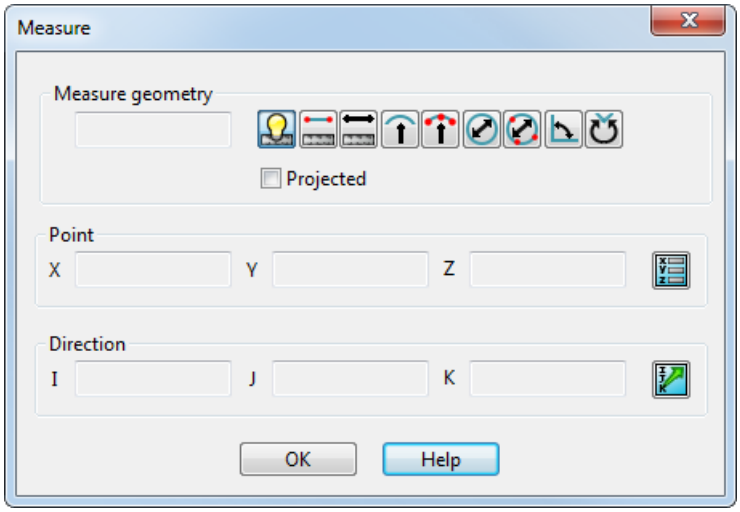
- Deselect the **Projected** option *(OFF)* to measure *Angle A* between *Line 1* and *Line 2*
- Select the **Projected** option *(ON)* to measure *Angle B* between *Line 1* and *Line 2*.



You cannot calculate projections of arcs. Therefore, if you measure an arc, its value is the same regardless of whether ***Projected*** *is on or off.*

Measure dialog

Use the **Measure** dialog to calculate points, coordinates, distances, angles, radii, or unit vectors.

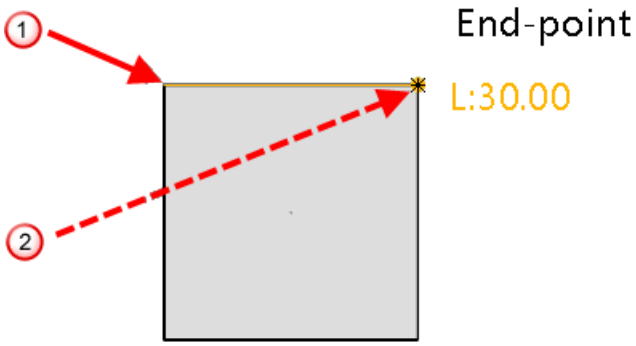


Auto-measure (see page 147)

Measure length between two points

1 Click first point 1.

2 Hover over the second point 2 to display the length. Click the point to display the length value in the dialog.



Measure length (see page 148)

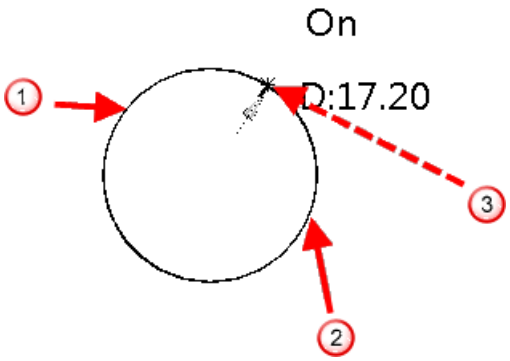
Measure radius (see page 149)

Measure Radius From Three Points (see page 156)

Measure Diameter (see page 149)

Measure Diameter from Three Points

1. Click first point ①.
2. Click second point ②.
3. Hover over the third point ③ to display the diameter. Click the point to display the diameter value in the dialog.



Objects

Use the following sections to find information on objects:

Creating an object

Creating an object to your specifications is a two-step creation process:

1. Create an initial object in the model. At this stage, the object may not be exactly the shape you want.
2. Edit the object until it is exactly the shape you want.

To create an object:

1. Select the object type and creation method from the **Object** menu or from the **Main** toolbar.
2. Enter one or more positions and/or selections as necessary.
3. Drag the edit handles at the object's key points to resize it.

As the cursor can operate only in two dimensions, all the positions it creates fall on the active plane. If the point you want is not on that plane, you will need to use the Status Bar (see page 30) to change the current active plane and/or enter a specific position. For more complex positions, you can use the **Position** dialog (see page 119).

In some infrequent cases, you may need to provide more information about how the object is to be modified. This is achieved by **Edit - Modify** which provides appropriate tools for each object type. It is displayed by double clicking the object or by selecting **Modify** from the **Edit** menu.

For further details, see Edit menu overview (*Menus and Toolbars*).

Editing an object

To edit object(s), first select the objects you want to change. If you select only one object, graphical handles are displayed that are specific to that type of object. If you select more than one object, the handles are not displayed.

Delete the object

- Press the keyboard **Delete** key
- Click  on the main toolbar or
- Select **Delete** from the **Edit menu**.

This works on any number of objects.

Drag move an object

This works with any number of objects.

1. Select an object by positioning the cursor over it, but away from any of its edit handles or key points.
2. Press and hold the left-hand mouse button. The object is picked by its key point closest to your click position and dragged as you move the mouse.
3. Release the mouse button to drop the object.

Drag copy an object

- Press and hold the **Ctrl** key while drag moving an object. A copy of the original is moved to the new position.

Cut or Copy an object

1. Click  or  from the **Main toolbar** or select **Cut** or **Copy** from the **Edit menu**.
2. Click **Paste**  into another position, another model window or another model.

Reshape an object

Drag edit handle(s) to alter the size and/or shape of an object.

- Lines - For further details, see Graphically moving a line (Wireframe modelling).
- Arcs - For further details, see Graphically editing arcs (Wireframe modelling).
- Curves - For further details, see Graphically editing curves (Wireframe modelling).
- Surfaces - For further details, see Editing the power surface (Surface modelling manual).
- Solids - For further details, see Editing a solid (Solid modelling)..
- Workplanes - For further details, see Graphically editing workplanes (Wireframe modelling).

Trim or **transform** an object using the editing tools:

- Limit - For further details, see Edit - General Edits - Limit Selection (*Menus and Toolbars*).

- Rotate - For further details, see Rotating by dragging (*Menus and Toolbars*).
- Mirror -For further details, see Edit - General Edits - Mirror (*Menus and Toolbars*).
- Move/Copy - For further details, see Edit - General Edits - Move (*Menus and Toolbars*).
- Scale - For further details, see Edit - General Edits - Scale (*Menus and Toolbars*).

Modify the parameters of an object:

- Select **Modify** from the **Edit** menu or double click the object. This displays an object-specific dialog or toolbar.

Editing objects using the Intelligent Cursor

When you start to drag an object, a group of objects or an editing handle of an object, the cursor feeds back information about points you can snap to.

The Intelligent Cursor:

- finds intersections between wireframe items and a surface or solid.
- snaps to the projected intersection of wireframe items and surfaces.
- snaps bezier curves to triangles in a symbol imported as a solid. For further details, see **STL/DMT** options in the Menus and Toolbars manual.
- snaps to triangles in symbols (not applicable to components).

For further details, see Using the Intelligent Cursor to enter positions (see page 58) and Selection using the Intelligent Cursor (see page 60).

Object dialogs and toolbars

Double click on an object to display an object related dialog or a toolbar. The options displayed correspond with the object type.

For further details, see:

- Editing an arc
- Editing a curve
- Editing a line
- Editing a workplane
- Editing a Power Suface

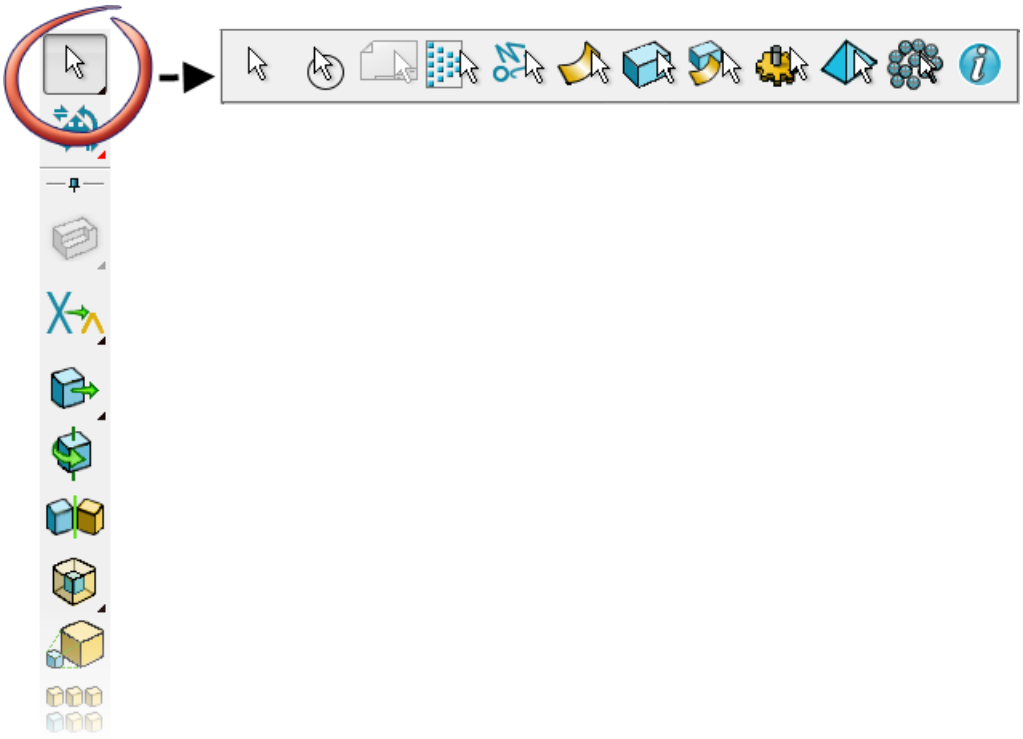
- Editing a primitive surface

Selecting objects

If you want to do anything to an object, you must select it first. This is a basic requirement for editing objects.

Some creation and construction commands allow selection of objects, but most selection can be done by using the selection options on the **Select** flyout.

1 Click the **Select**  flyout button to display the **Select** flyout toolbar.



The **Select** flyout toolbar contains the following selection options:

 **Selection information** — Move the cursor over the **Select** flyout button to display selection information as a Tooltip

 **Partial box** (see page 167)

 **Whole box** (see page 168)

 **Drawing view**

 **Selection filter** (see page 169)

 **All wireframes (see page 171)**

 **All surfaces (see page 172)**

 **All solids (see page 171)**

 **All surfaces and solids** (see page 172)

 **All components and sub-assemblies** (see page 172)

 **All meshes** (see page 172)

 **All clouds (see page 172)**

 **Selection information** (see page 172)

The current selection is highlighted. Single selected objects have their key points visible as graphical handles. When more than one object is selected, the key point handles are not displayed, but the objects remain highlighted.

Selecting single objects

Click on an object in the graphics window to select it. A selected object is highlighted with its key points visible as graphical handles.

You can click:

- on an object (see page 44).
- over an object (see page 47).

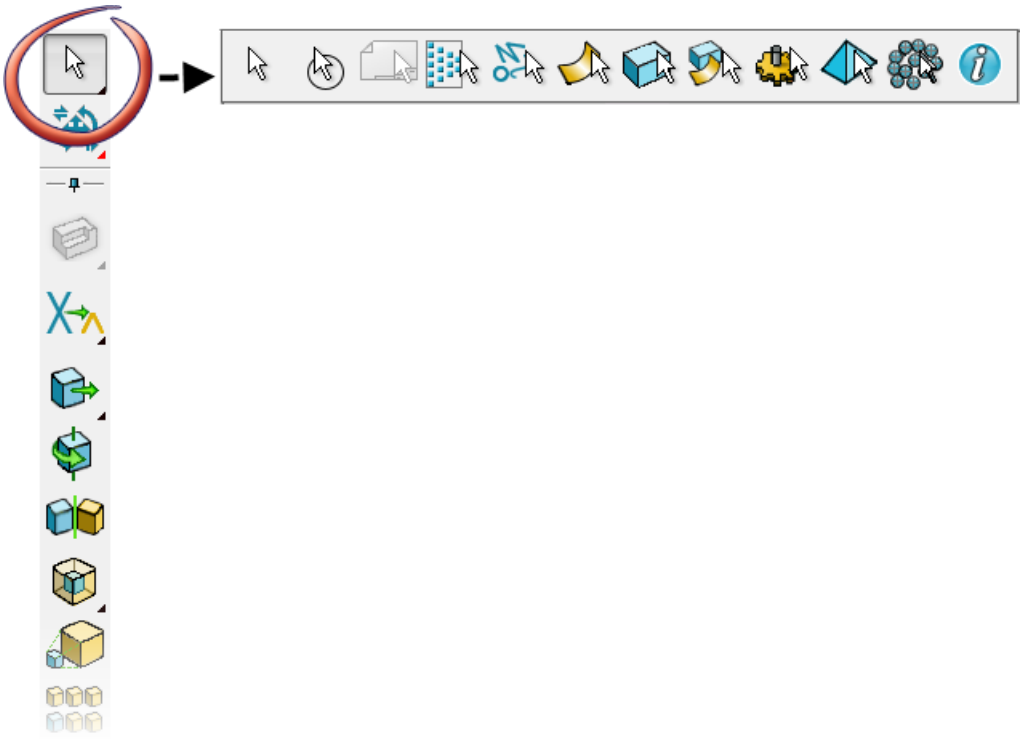
Selecting multiple objects

Select multiple objects in one of the following ways:

- Use the left mouse button to click on the object and it will highlight with its key points visible as graphical handles. Certain **modifier** keys are also used with the mouse to change the way the mouse behaves. For example, hold down  and click on another object. This object is also highlighted and added to the selection. When more than one object is selected, the key point handles are not displayed, but the objects remain highlighted.

  Hold down  and click on objects to add a multiple number of items to the selection. For further details, see Using the mouse (see page 36).

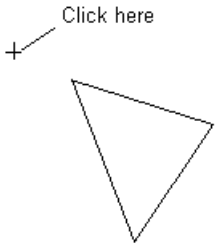
- Click the **Select** flyout  to display the selection and selection information options. You can use specific options on this flyout to select multiple objects. For further details, see Selecting objects (see page 165)



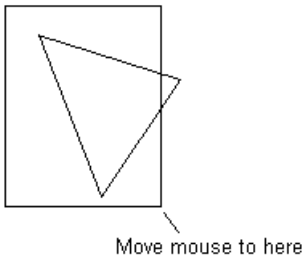
Selecting objects - Partial box

Partial box  (*Selection flyout*) is the default selection option (see page 165) when the program is started.

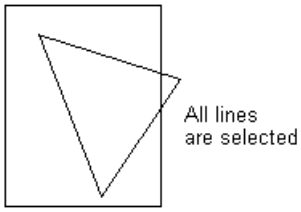
1 Using the left mouse button, click a point on the screen where you want the box to start.



2 Keeping the mouse button pressed, move the mouse to a second position on the screen. This creates a rectangular "rubber band" on the screen.



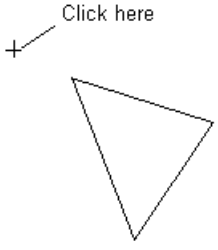
3 Release the mouse to select all of the objects that are partially or completely enclosed in the box. In the example shown below, all the lines are selected as they are partially or completely enclosed by the box.



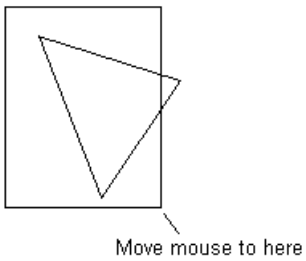
Using whole box selection

1 Select the **Whole Box**  button (*Selection flyout*). For further details, see Selecting objects (see page 165)

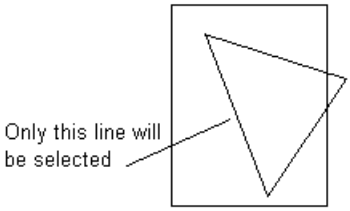
2 Using the left mouse button, click a position on the screen where you want the box to start.



3 Keeping the mouse button pressed, move the mouse to a second position on the screen. This creates a rectangular "rubber band" on the screen.



4 Release the mouse to select all of the objects that are completely enclosed in the box. In the example shown below, only the line completely enclosed by the box is selected.



Adding or removing objects

To add or remove objects from a selection, press and hold the **Ctrl** key while clicking or dragging the mouse to:

- deselect objects that are currently selected.
- select objects that are currently deselected.

Selection filter

Complex models can become cluttered with objects making it difficult to select the ones you want. The selection filter helps you to choose the type of object you want to select. You can also select styles of objects and levels on which objects are kept.

Use the selection filter to specify the type of item you want to snap to.

You can enter creation mode with the selection filter set.

Using the selection filter

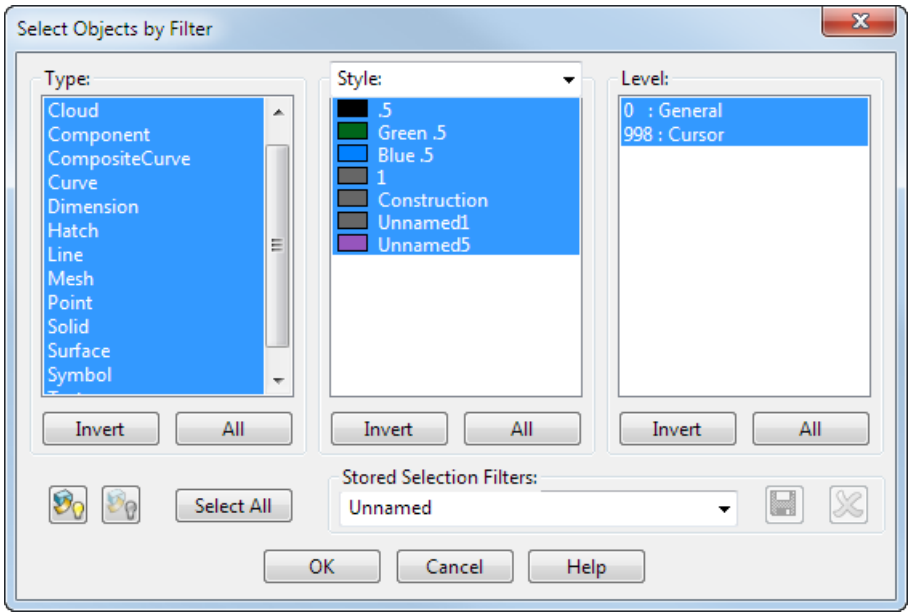
1 Click the **Selection Filter**  button *(Selection flyout)*.

2 Use the **Selection Filter** dialog (see page 170) to filter the selections.

Selection Filter dialog

You can use this dialog in one of the following ways:

- Select an object on the screen and the dialog displays its **Type**, **Style**, and **Level**.
- Choose the **Type** and **Style** of the objects to select and click **Select All**. All the objects that are of the correct type and style are selected.



Type — This shows a list of the different objects that can be created. By default each type is selected. Clicking on an item selects or deselects it. Items that are selected are highlighted. When you have selected items from the list, then only those items can be selected on the screen.

Style / Pattern / Colours / Widths / Materials — Select one of the following from the drop-down list. The list of styles, patterns, colours, width or materials used in the model is displayed in the centre panel of the dialog. Clicking on an item selects or deselects it. Items that are selected are highlighted. When you have selected items from the list, only those items can be chosen on the screen. For example, you can filter the selection of objects by colour, width or material.

Level — Each level that is used is listed here. The level name is displayed alongside the level number. By default, each level is selected. Clicking on an item selects or deselects it. Items that are selected are highlighted. When you have selected items from the list, only those items can be chosen on the screen.

Invert — This inverts the current selection, that is, those which are selected become unselected and those which are unselected, selected.

All — This selects all items in the list. Each item will be highlighted to show that it is selected.

Select All — When you click on this, all items on the screen, which match the selections you made on the dialog, will be selected.

 — Click this button to blank items according to the filter you have set.

 — Click this button to unblank previously blanked items.

Stored Selection Filters - You can save a selection filter that you have specified:

1. Use **Type**, **Style** and **Level** to specify a filter.
2. Enter a filter name to replace **Unnamed**
3. Click  to save the selection filter. The filter is added to the drop-down list.

To delete a stored selection filter:

1. Select the filter from the **Stored Selection Filters** drop-down list.
2. Click 

OK — This saves any changes and closes the dialog. You can only select those items you specified on the dialog.

Cancel — This closes the dialog and discards any changes made to it.

To return to normal selection after using a filtered selection

1. Open the **Filter** dialog again.
2. Make sure that all options in the three panels of the dialog are selected.
3. Click **OK**.

Selecting all wireframe objects

Click the **All Wireframe**  button (*Selection flyout*) to select all the wireframe objects in a model.

Selecting all solids

Click the **All Solids**  button (*Selection flyout*) to select all the solids in a model.

Selecting all surfaces

Click the **All Surfaces**  button (*Selection flyout*) to select all the surfaces in a model.

Selecting all surfaces and solids

Click the **All Surfaces and Solids**  button (*Selection flyout*) to select all the solids and surfaces in a model.

Selecting all components and sub-assemblies

Click the **All Components**  button (*Selection flyout*) to select all components and instances of sub-assemblies.

Selecting all meshes

Click the **All Meshes**  button (*Selection flyout*) to select all the meshes in a model.

Selecting all clouds

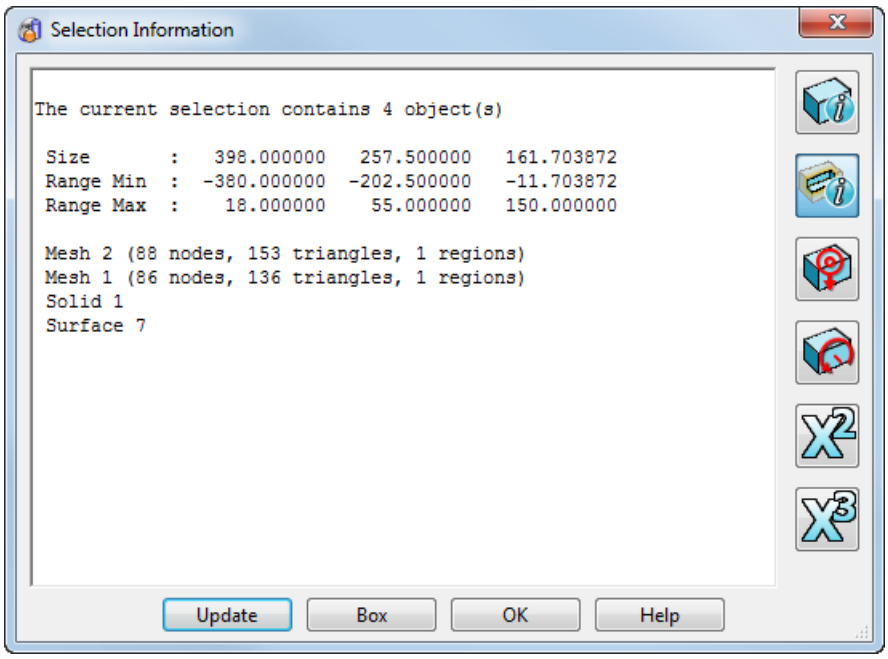
Click the **All Clouds**  button (*Selection flyout*) to select all clouds.

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.

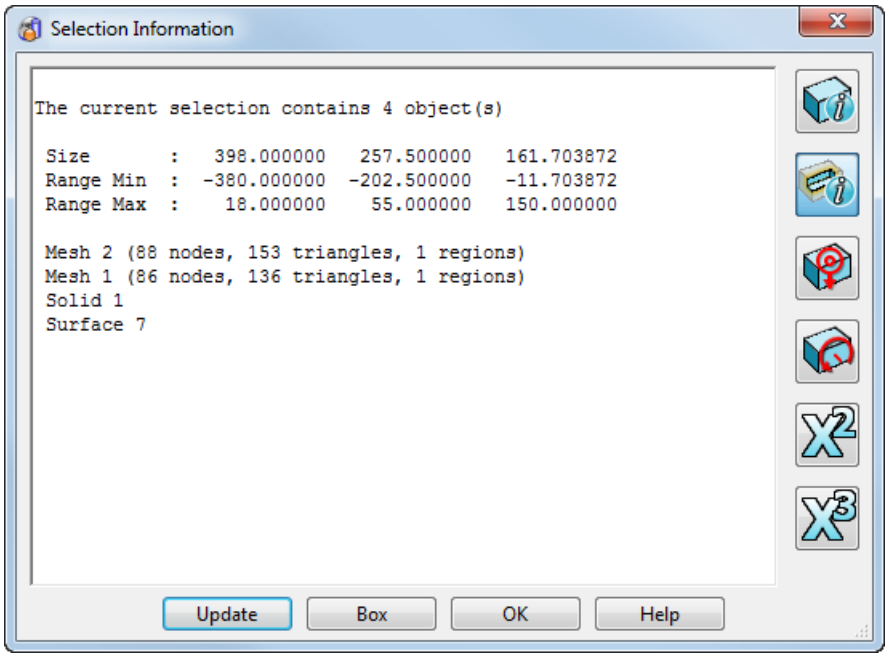
1. Select one or more objects.
2. Click  (*Selection flyout*) or select **Tools > Selection Information**.

If a single object is selected, then information about that object is displayed in the Selection Information dialog (see page 173). 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.

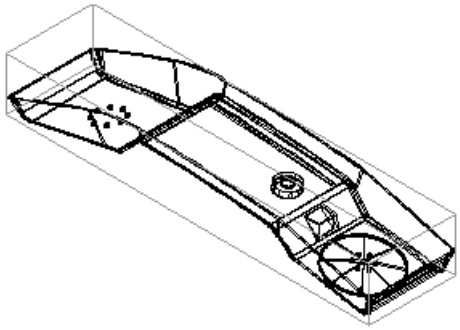
 Toggle moment of inertia.

 Toggle the display of the area of the selection.

 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.

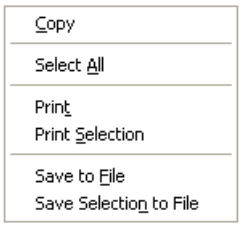


OK — Close the dialog. If a bounding box has been drawn, this is removed.

 *Use the Selection information context menu (see page 175) to copy, print or save the information displayed in the dialog.*

Selection information menu

Right-click on the Selection information dialog (see page 173) to display a menu providing copying, printing, and saving functions.



Deselecting objects

To deselect all objects, either:

- Click in a space away from any objects, or
- Select **Edit > Select > Clear Selection** from the menu.

Selecting coincident objects

When several objects are beneath the cursor and clicking selects the wrong one, click again without moving the mouse and the program selects the next object in the database at that position. Repeat until the one you want is selected.

Selection using the Intelligent Cursor

When the Intelligent Cursor is enabled, objects are highlighted as you move the cursor over them. This shows which objects can be selected if you click at that point. For further details, see Turning the Intelligent cursor on or off (see page 53).

When several objects are under the cursor and one of them is selected, the Intelligent Cursor highlights the next object from the database at that position. A click now deselects the selected object, selects the highlighted object, and highlights the next one. This enables you to select the correct object from the group.

You can also right-click to display the object's menu and click **Next Selection** to select the next object.

The Intelligent Cursor option is displayed on the context menu. Select or Deselect to toggle the Intelligent Cursor on and off. Selecting this option is the same as selecting the Intelligent Cursor **Active** option from the Mouse options page of the **Options** dialog.

Object key points

All objects have a number of *key points* which define the object's position and size.

These are:

- **Lines** - at each end and the mid-point.
- **Arcs** - at each end of the sector, the centre point, and the mid-point.
- **Curves** - at each point where the curve can change direction.
- **Surfaces** - at each corner of the surface patch (the key points on the associated curves behave as for curves).
- **Workplanes** - at the origin and end of each axis pointer.

Selection anchor point

When the object is selected, the program chooses one of its *key points* as the object's *anchor point* for the current selection. The next selection may (or may not) use a different *key point*.

The anchor point is used by the program to decide where to position the object, for example during all types of moves and copies. The rules for which *key point* is chosen to be the anchor are as follows:

- When a single object is selected, the anchor point is the object's *key point* which was nearest the cursor when you selected the object.
- When objects are selected using a rubber box, the anchor point is at the centre of the smallest box capable of containing the objects. This applies whether the rubber box contains one or many objects.
- If objects are *shift selected* individually, the anchor point for the group is the *key point* nearest the cursor on the last object selected.

When you want a specific anchor point for a group,

1. Select a group of objects, using the rubber-box.
2. Press the **Control** key and deselect the objects.
3. Press the **Control** key and reselect an object.

   The anchor point for the group selection is now on this object rather than at the centre of the rubber box.

Selecting an object by name

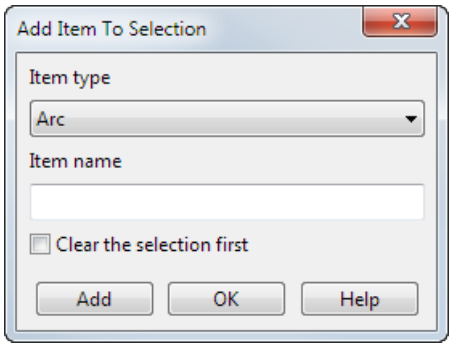
To select an object by its name:

1. Select **Edit > Select > Select by Name** from the menu.

   The **Add Item To Selection** dialog (see page 177) is displayed.
2. Use the dialog to select an object by its name.

Add Item To Selection dialog

Use this dialog to select an object by its name.



Item type — Select the type of the object you want to select.

Item name — Enter the name of the object.

Clear the selection first — Select this option to deselect all currently selected objects.

Add — This adds the selected object to the current selection. If the object is blank, it unblanks when selected.

The dialog remains on the screen so that you can add more objects to the selection.

Parameters

Use the following sections to find information on parameters:

What is a parameter?

A parameter is a user-defined variable, which is used to store numbers and expressions.

For example, parameters *pi* and *radius* are defined as follows:

pi = 3.141592654

radius = 56

A third parameter *circumference* can be defined using parameters pi and radius:

circumference = 2*pi*radius

When you update the *radius*, *circumference* is automatically updated.

- Parameters can be used to automatically update your model when their definitions are updated. For this to work, you must create your model using relational geometry and define its objects using parameters.
- Parameters can be used anywhere you need to enter numbers. Parameter names are case sensitive. So,

  FRED

  Fred

  fred

  are three different parameters
- You can create a parameter in a text box on a dialog or toolbar by typing

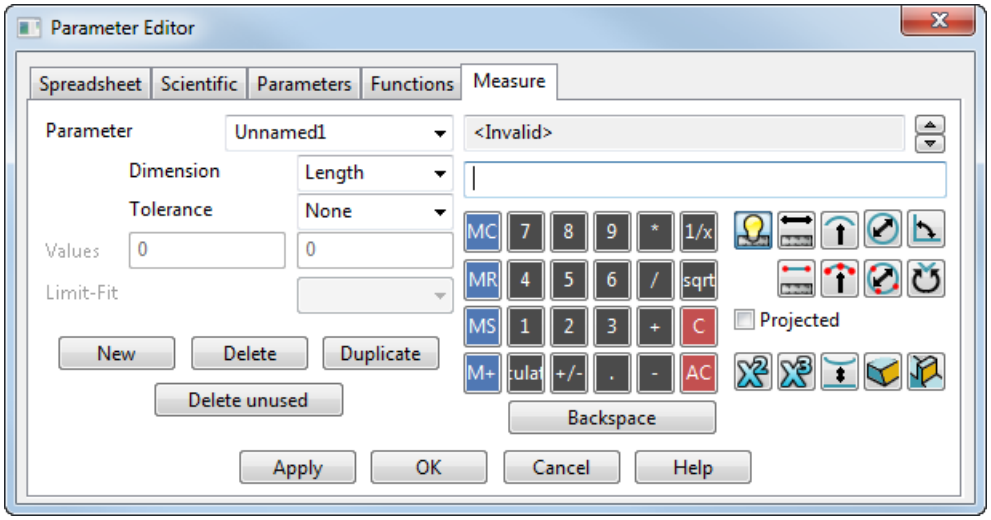
  param_name=expression

where *param_name* is the name of the parameter and *expression* is an expression representing the numeric value of the parameter.

Creating a parameter

1. Select **Object > Parameter** from the menu.

   The **Parameter Editor** dialog is displayed showing the **Measure** tab.



2. Click **New** to activate the options.
3. In the **Parameter** text box, a default name for the parameter is displayed. Change it to something more meaningful to remind you what the parameter is used for.
4. Click **Apply**. The parameter will now appear in the **Parameter** drop down list, although you have not attached a value to it at this stage.
5. Using any of the pages on the dialog, select your newly created parameter from the drop down list.
6. Enter a value and click **Apply** to assign the value to your parameter.

   You can enter the value in a number of formats. The following are all valid for a length parameter:

   8 + 1 gives a value of **9**. If you don't include the unit of measure (in this case, length), your default unit is used, as defined on the Units and tolerances options page. The unit of measure will be displayed when you subsequently select the parameter from the drop down list.

   8mm + 1in gives a value of **33.4 mm** (assuming that your length units are set to mm on the Units and tolerances options page)

When a parameter has been created it is available to be used. For further details, see Using a parameter (see page 185).

Parameter Editor dialog

The Parameter Editor dialog contains five tabs:

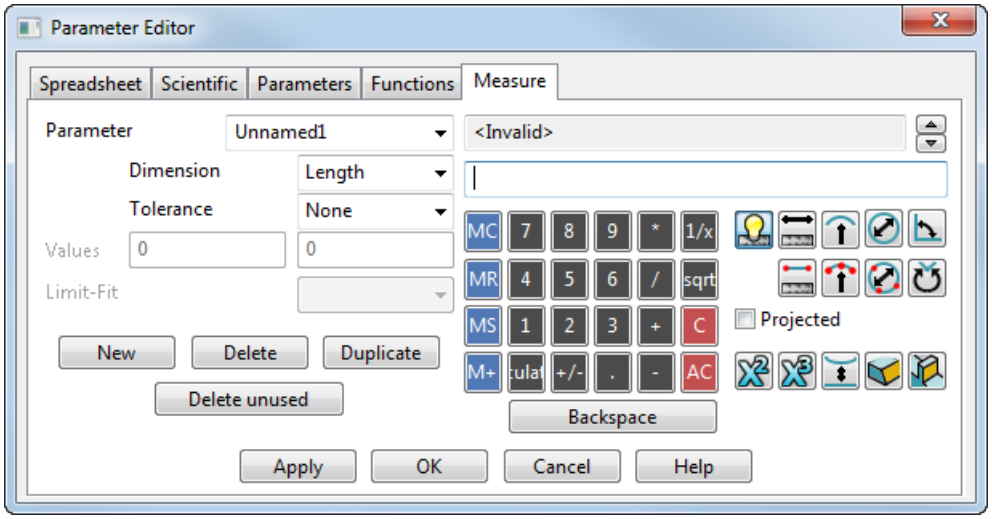
Measure (see page 182)

Functions (see page 182)

Parameters (see page 183)

Scientific (see page 183)

Spreadsheet (see page 184)



Each page on the dialog has the following features:

- the parameter creation options
- the numerical keypad
- the calculator (see page 139) functions
- the expression text box
- the result text box

Each page on the dialog has the following options.

Parameter - Enter a name for the parameter or select a name from the drop down list.

Dimension - By default, a parameter is a measurement of length. However, by using this option menu you set it to ratio, length, area, volume or angle.

Tolerance - By default, a parameter has no tolerance associated with it. The **Tolerance** drop down list contains various types of tolerance you can set:

None

-:-

+:-

-:+

+:+

Limit-Fit

Values - If you choose one of the four **Tolerance** options with the **+** and **-** symbols, use the **Values** text boxes to enter the tolerance values.

Limit-Fit - If you choose the tolerance value **Limit-Fit**, use the **Limit-Fit** drop down list to enter a limit value or select one from the list of common values.

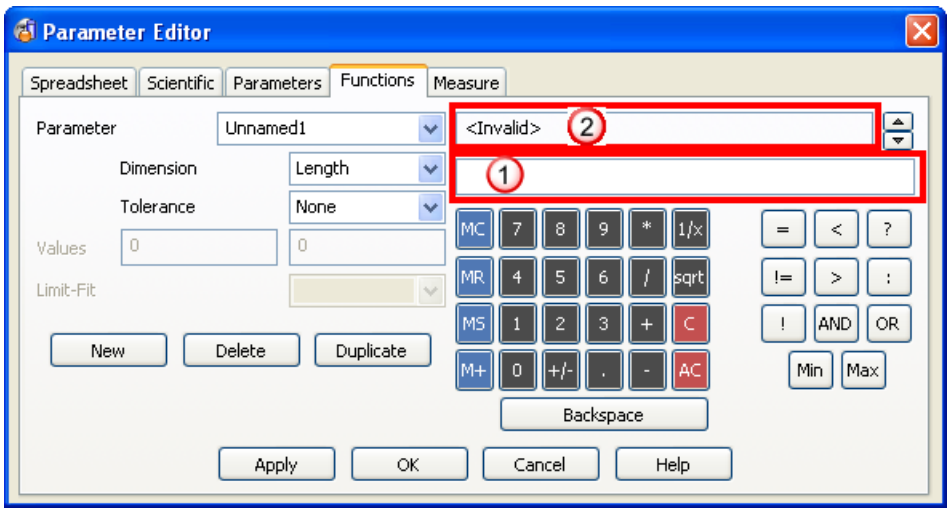
New - Select this option to create a new parameter.

Delete - To delete an existing parameter, use the **Parameter** drop down list to choose a parameter and then click the **Delete** button.

Duplicate - To create a copy of an existing parameter, use the **Parameter** drop down list to choose a parameter and then click the **Duplicate** button.

Result - (2) The measurement or calculation results are displayed here.

Expression - (1) This displays values entered by using the numerical keypad and buttons. You can also enter a calculation or a value for a parameter.



Numerical keypad - Use the keypad to enter a value for a parameter.

Backspace - Use this to move the cursor one space to the left in the **Expression** text box. Any value entered in the space to the left is deleted.

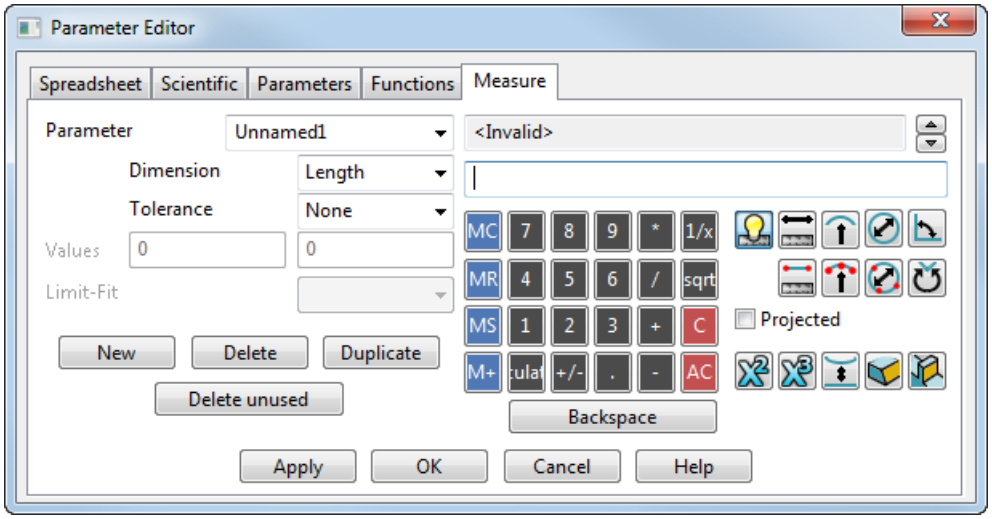
Apply - Click **Apply** to create the parameter.

OK - Click to confirm the parameters you have created or modified. The dialog is removed.

Cancel - Click to remove the dialog and deletes all the parameters created whilst the dialog was displayed. Any parameters modified are also changed back to their original values.

Parameter dialog - Measure

This page displays the calculator measurement buttons.



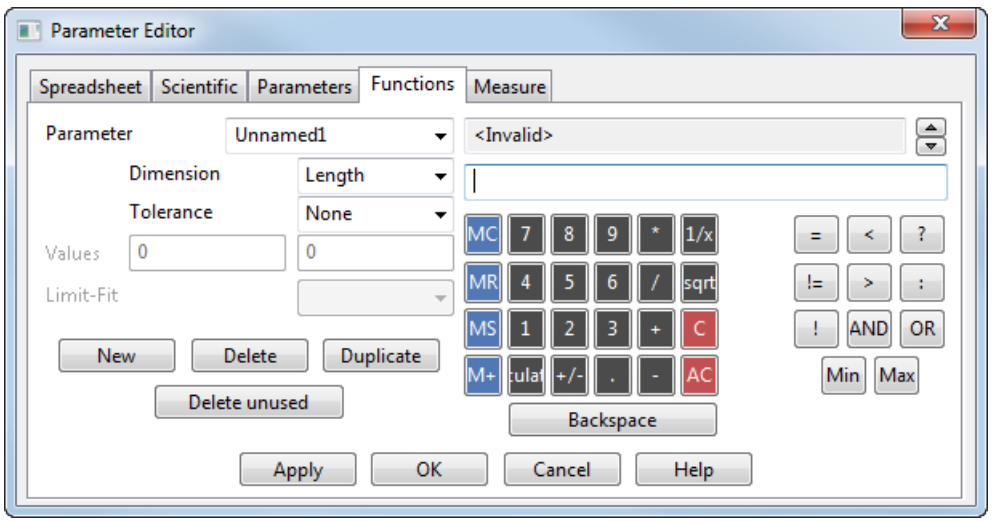
Use the measurement options in the same way as Using the calculator to measure (see page 146).

For further details, see Using the calculator (see page 139)

Projected - If *ON*, the measurement is projected onto the current plane. Use this option with the measurement buttons. For further details, see Using the Projected option (see page 158).

Parameter dialog - Functions

Use the functions on this page to create calculations and logical expressions in parameters.

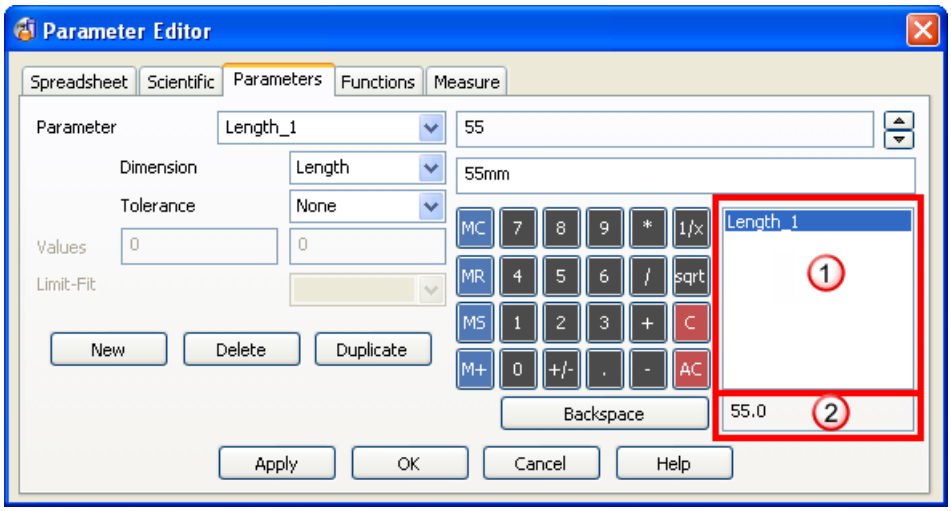


Use the functions in the same way as the calculator functions (see page 145).

For further details, see Using the calculator (see page 139).

Parameter dialog - Parameters

Use this page to create or select previously created parameters and their values.

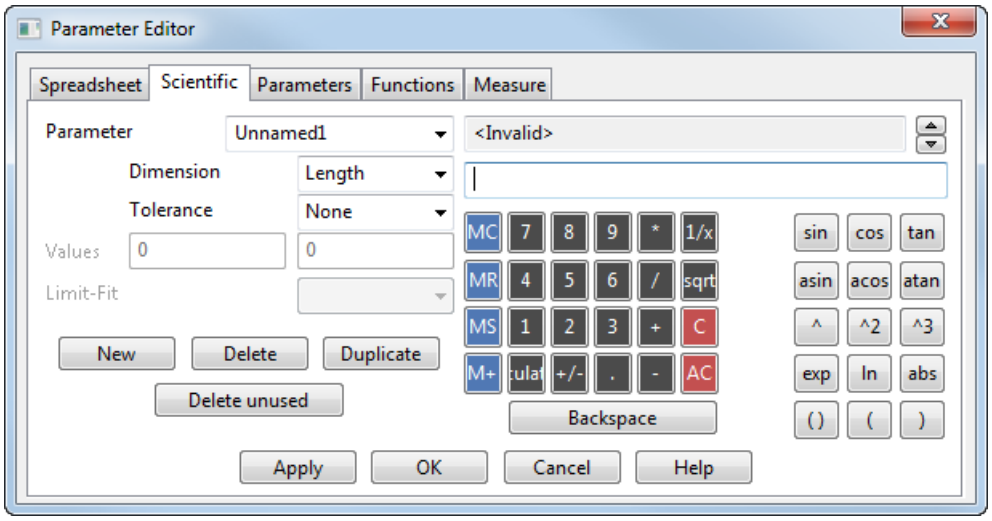


Select parameter - The list of previously created parameters is displayed in the large text box (1).

Current Value - The value for each parameter in the list is displayed in the small text box (2).

Parameter dialog - Scientific

Use this page to use scientific functions for creating calculations in parameters.



The functions are used in the same way as the calculator scientific functions (see page 143).

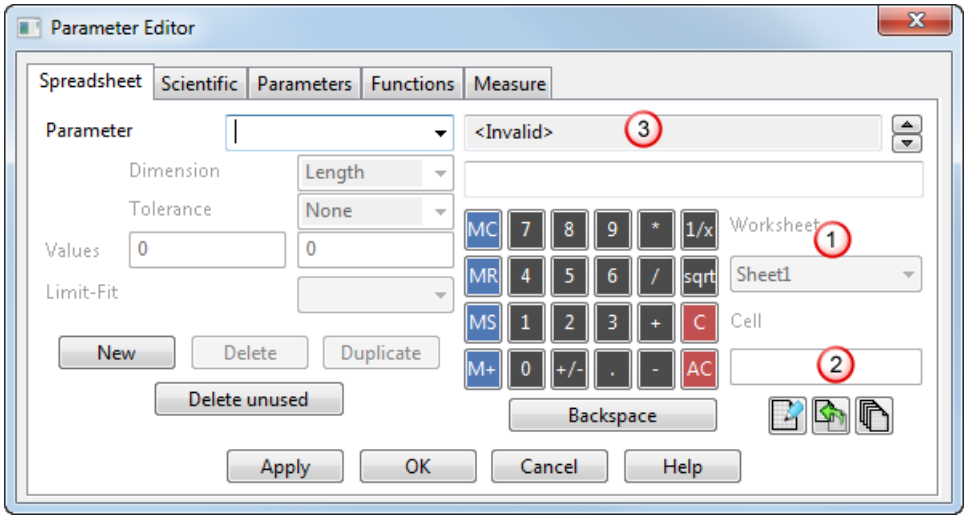
For further details, see Using the calculator (see page 139).

Parameter dialog - Spreadsheet

Use this page to display a Microsoft Excel spreadsheet where you can create and edit data that you want to store with a model. For example, you might wish to enter complex calculations for material densities or volumes.

Each new model has a new, blank spreadsheet ready for use.

You need a licence for Microsoft Excel to use the spreadsheet options.



Worksheet - 1 This becomes active when the **New** button is selected. Select the worksheet you want to work with in the spreadsheet.

Cell - 2 This becomes active when the **New** button is selected. Enter the spreadsheet cell containing the value you want to work with. For example **A1** or **D4**. The value stored in the cell is displayed in the **Expression** text box 3.

Modify spreadsheet - Click to display a Microsoft Excel spreadsheet. You can edit existing data, or enter new data, that you want to store with the model.

Synchronise parameters with spreadsheet - This synchronises PowerSHAPE with the Excel spreadsheet to update all the values when changes have been made. Select this option each time you edit the spreadsheet.

Create multiple spreadsheet parameters - Select this option to create numerous parameter names and values in a Microsoft Excel spreadsheet. This allows you to apply the parameters all at the same time.

For advanced modelling concepts, see

- Creating parameters in a spreadsheet (see page 219)
- Creating multiple parameters in a spreadsheet (see page 221).

Creating parameters by typing in values

You can create a parameter in a text box on a dialog or toolbar by typing:

param_name = value

where *param_name* is the name of the parameter and *value* is the numeric value of the parameter.

The type of parameter created (that is, length, angle, area, volume or ratio) is determined by the values used in the text box.

For further details, see Creating a parameter (see page 179).

Editing a parameter

1. From the **Object** menu, select **Parameter** to display the **Parameter Editor** dialog.
2. Use the **Parameter** drop-down list to select a parameter to edit.
3. Use the Parameter Editor dialog (see page 180) to edit the parameter definition.

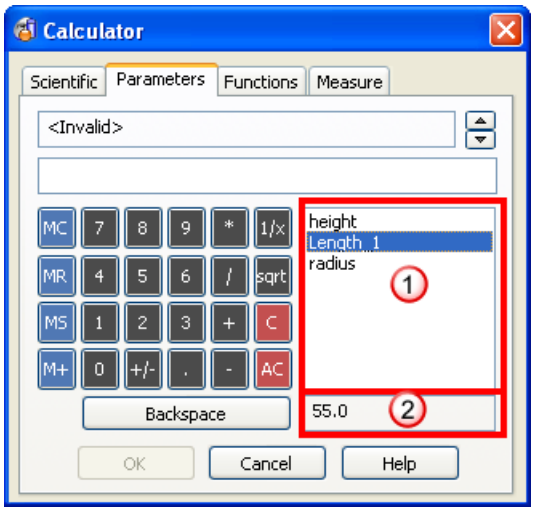
Using a parameter

Whenever you can enter a numeric expression using the calculator, you can specify a parameter instead. The current value of the parameter is used instead of entering a number. To set up the parameters ready to be used, see Creating a parameter (see page 179).

1. Display the calculator (see page 139). If you are entering a value into a dialog, right clicking where you would enter the number displays the calculator. For further details, see Calculator dialog (see page 140).
2. Use the calculator as normal. Steps 3 and 4 give instructions on putting the value of a parameter into your calculation.
3. Click the **Parameters** tab to display the calculator parameters options (see page 144).

   All the parameters that have been created are listed  .

4 Click a parameter to display its value ② below the list of parameters.



5 Double click on the parameter you want to use.

The name of the parameter is entered into the calculation indicating that the parameter value will be used in the calculation.

6 Once you have completed your calculation, click **OK**.

The resulting value is entered into the dialog or toolbar.

Finding out if a value uses parameters

You can find out if a value uses parameters in one of the following ways:

- A value that is defined with a parameter or expression is indicated by the use of = in addition to the value.
- Display the dynamic help for a box on a dialog by putting the cursor in the box. If the value uses parameters, the dynamic help will show you the expression containing the parameters in addition to the usual help message. This only works for values that are parametric. A parametric value is one that remembers the parameters used to define it. When the value of any parameter is changed, the parametric value updates automatically.

  Create multiple parameter names (see page 221) and values in a Microsoft Excel spreadsheet in order to apply them to objects all at the same time.

  The following are examples of where Parametric values exist on dialogs:

  - Solid Cut
  - Solid Boss

- Primitive Solid
- Primitive Surface
- Solid Extrusion
- Solid Fillet
- Solid Bulge
- Solid Thicken
- Solid Chamfer
- Solid Hole
- Solid Hollow
- Surface Extrusion
- Chamfer Editor
- Active Dimensions

Advanced modelling concepts

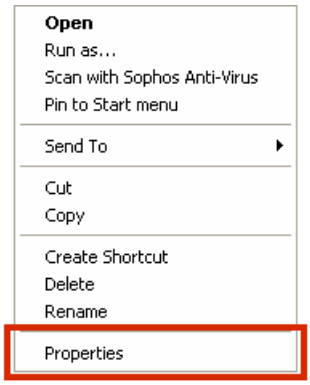
Use the following sections to find information on advanced modelling concepts:

Starting PowerSHAPE in different modes

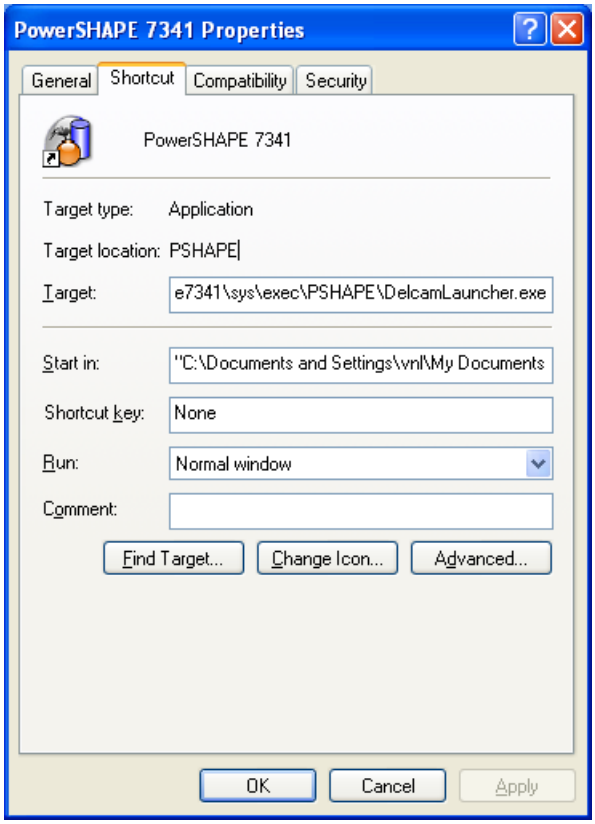
You can start PowerSHAPE in different modes.

1 Before starting the program, right click the PowerSHAPE icon  on your desktop.

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



The PowerSHAPE **Properties** dialog is displayed, showing the following **Target** path:



3 At the end of the **Target** path enter a space followed by the switch for the required start mode. The example below shows the switch **-designer**

"C:\Program Files\Delcam\ *XXXX\sys\exec*\DelcamLauncher.exe" -designer

where * is the product installed (for example PowerSHAPE Pro).

This will start PowerSHAPE in **PowerSHAPE Designer** mode.

For further details, see Table of PowerSHAPE start modes (see page 190).

The availability of start modes is subject to obtaining the relevant product licence.

To use a combination of start modes (for example **-designer -normal**) type them in the **Target** path leaving a space between them.

To view the available start modes in PowerSHAPE, use the switch **-help** in the **Target** path before starting the program. The dialog shown below is displayed:

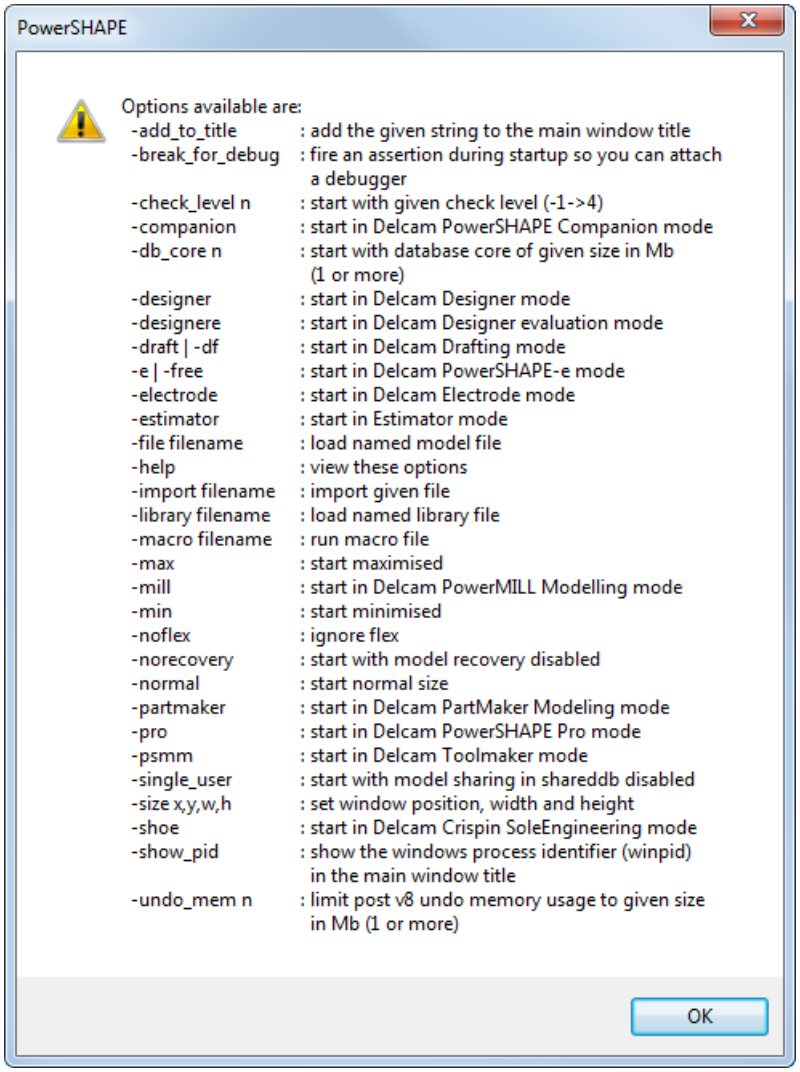


Table of PowerSHAPE start modes

The available start modes are described in the following table.

Start mode	Switch
Products	
Start in **PartMaker Modeling** mode This mode checks for the COMPANION paf and will not start if the correct licence is missing. The default tolerances for this mode are: ▪ general tolerance: 0.0001 inches ▪ shading tolerance: 0.001 inches	**-partmaker**
Start in **Designer** mode	**-designer**
Start in **Designer evaluation** mode	**-designere**
Start in **Drafting** mode	**-draft** **-df**
Start in **Electrodemaker** mode	**-electrode**
Start in **Pressmaker** mode	**-press**
Start in **PowerMILL Modelling** mode	**-mill**
Start in **PowerSHAPE-e** mode	**-e** **-free**
Start in **PowerSHAPE Pro** mode	**-pro**
Start in **Sketcher** mode	**-sketcher**

Start mode	Switch
Commands	
Start with **n check level** (-1->4)	**-check_level n**
Start with **database core of n size** in MB (1 or more)	**-db_core n**
Import given file	**-import filename**
Load named model file	**-file filename**
Load named library file	**-library filename**
Run macro file	**-macro filename**
Pass **arguments to macro** at runtime	**-macro filename arguments**
View these options	**-help**

Window sizes	
Start **normal** size	**-normal**
Start **maximised**	**-max**
Start **minimised**	**-min**
Set **window position, width and height**	**size x,y,w,h**
Set **window size** and **position**	**-normal -window_size:0,0,1280,1000**

The availability of start modes is subject to obtaining the relevant product licence.

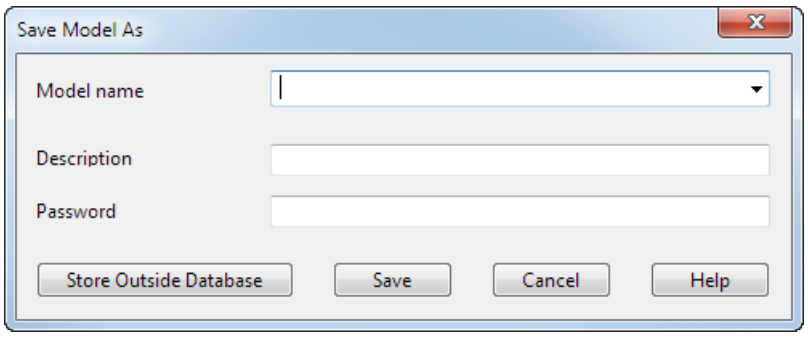
Models, advanced

For further advanced details about models, select from the following:

- Save Model As dialog (see page 192)
- Creating a template model (see page 195)
- Keeping track of changes made to a model (see page 196)

Save Model As dialog

Use this dialog to save a model using a new name.



Model name — Type a suitable name for your model. If there is already a model with the name that you enter, a warning message is displayed, to avoid you accidentally overwriting an existing model. You can choose to continue saving your new model to an existing filename, in which case the existing model will be overwritten.

If you import files into a new model, the name of the last one imported minus its extension is automatically displayed as the Model Name when you save the model. For example, if the name of the last file imported is handle.igs, the Model Name is handle.

Description — Type a suitable descriptive message. This is to help you find specific models that might have similar names.

You can leave this box blank if you do not need a description.

Password — If you want to protect the model, enter a password. However, don't forget it, because you may not be able to open the model when you need to.

Store Outside Database *— This allows you to store models anywhere on the local disk or network. Click the button to display the* ***Save As*** *dialog. Use the* ***Save As*** *dialog to select where to save the model and then click* ***Save*** *to store the model.*

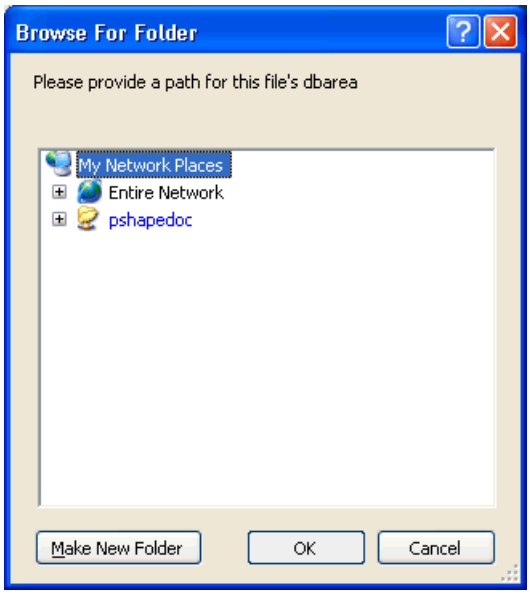
If models are stored in the database, the models are saved and organised on the local disk or network. When you want to open the model, you only need to remember the model name to find the appropriate files.

Save — This saves the model and removes the dialog from the screen. The model is saved in the default format. For further details, see Changing the default format for saving models (see page 194).

Browse for folder dialog

Use this dialog to define the folder for storing the vault model.

1. Select the folder.
2. Click **OK**.



Cancel - This removes the dialog from the screen without saving the model.

*On a new or read-only model, **File - Save** displays the Save Model As dialog (see page 192).*

Changing the default format for saving models

When you install PowerSHAPE, you can select the default format for storing models as follows:

- a single file (psmodel).
- a set of files in a folder.

In the current session of PowerSHAPE you can change the default format for storing models.

1. From the **Tools** menu, select **Options** to display the **Options** dialog.
2. Select the **Model** option.
3. In the **Model Save As Format** section, select the default method for storing models.
4. **Directory** - Stores your model using the directory structure.
5. **Single** - Stores your model as a single file.
6. Click **OK**.

*With the **File Save** command, a model is always saved using the same structure regardless of the **Model Save As Format** option.*

Saving a model with a thumbnail

By default, a new thumbnail of your model is automatically created whenever you save your model.

You can choose to save the thumbnail when you want by changing the settings as follows:

1. Open the model.
2. From the **Tools** menu, select **Options** to display the **Options** dialog.
3. Use ⊞ to expand the **File** options.
4. Click **Model** to display the **Model** page of the **Options** dialog.
5. Change the **Save Mode** setting to **Manual.**
6. Click **OK**.

To save the thumbnail,

1. Change the view of the model to the one you want to appear in your thumbnail.
2. From the **File** menu, select **Save Thumbnail**.
3. From the **File** menu, select **Save** or **Save As** to save the model and the thumbnail. The **Save mode** setting is also saved so whenever you open the model you can manually update the thumbnail.

NEW_MODEL_MASTER

A template model is a model that is used as the basis for all new models. This model is called:

NEW_MODEL_MASTER_29

If a template model exists, a copy of this model is created each time you create a new model using the **File > New** command.

Your template model can store things like:

- your company defaults for model-dependent options on the **Options** dialog, such as tolerance.
- your standard level settings and names.
- line styles.

Even standard geometry (such as symbols) can be included in this model.

You must have system administrator security to create and edit the template model.

To create a template model:

1. Make sure no-one else is using PowerSHAPE and that you have system administrator security.
2. Create a model containing all the required settings.
3. Save the model with the name **NEW_MODEL_MASTER_29** as follows:
   a. Select **Tool > Options** to display the **Options** dialog.
   b. In the dialog, select **File > Model** from the tree.
   c. Deselect **Always save and open from outside the database**.
   d. Click **OK**.
   e. Save the model.
4. Delete all existing models with names of the form *NEW_MODEL_<*>_29*

(where * is the number of the model).

For example, *NEW_MODEL_1_29*.

If **Always save and open from outside the database** is deselected and you have administration access you will be able to do this by selecting **File > Delete**. Alternatively:

a Open the **Command** window.

b Type: **admin delete model**

c Type: **NEW_MODEL_*_29**

(where * is the number of the model). For example, NEW_MODEL_1_22.

5 Select **Always save and open from outside the database** to start saving models outside the database again.

The template model can be edited as if it were any other model.

Keeping track of changes made to a model

If **Delcam PS-Team** is installed on a server and visible from your PC, you can set up a topic to keep track of the changes made to your models.

On the **PS-Team** page of the **Options** dialog, you can specify the names of the topics to use for models named explicitly or by fields. We refer to vault and temporary models as models named 'by fields' and all other models as 'explicitly'. You can also specify the server and the URL for PS-Team.

PS-Team is an Internet-based software tool, provided by Delcam, to support communication between teams of people.

To start PS-Team, select **PS-Team** from the **Application** menu to display **PS-Team** in the browser window. If the server and URL for PS-Team are not set up on the **PS-Team** page of the **Options** dialog, a dialog appears asking for them.

If the browser window is displayed, you can click:

- the **PS-Team** tab  to display the Delcam PS-Team page.
- the **History** tab  to display the task associated with the current model.

When a model is saved with a new name, a new task is automatically created in **PS-Team**. In the task for the model, you can manually add information to the task associated with a model to record any changes.

When you import and export data, the task is automatically updated giving details of the import and export files.

If you delete a model, the status of the task is changed to *complete*.

For further details, click **Help** in the PS-Team window.

Importing and exporting, advanced

PowerSHAPE uses Delcam Exchange (see page 197) to translate data files from other software in cases where the in-built translators are not used.

The following sections give details of advanced export and import methods:

Exporting data, advanced (see page 197)

Importing data, advanced (see page 208)

Using Delcam Exchange to translate data

Delcam Exchange is a stand-alone program which is linked to the program to allow you to translate certain types of data. It can also be used on its own.

The following sections show you how to use Delcam Exchange within PowerSHAPE to translate data, but not how to use Delcam Exchange stand-alone.

Use either PowerSHAPE or Delcam Exchange to translate data. For further details, see Importing files using Delcam Exchange (see page 215) and Exporting files using Delcam Exchange (see page 204).

Delcam Exchange understands the same licence options as the program's built-in translators. For example, if you have a licence for importing CATIA files, you can use this to import CATIA files using Delcam Exchange. However, if you have not purchased a licence for importing CATIA files, for example, you can still use Delcam Exchange to import the file by purchasing a 'pay-per-use' voucher from Delcam's web site.

Exporting Data - Advanced

Select Tools > Options > Data Exchange > Version 8/ Surfaces and use the **Export** options for exporting files.

For further details about exporting data, select from the following:

- Using Delcam Exchange to translate data (see page 197)

- Exporting data without a licence (see page 204)
- Exporting materials (see page 205)

Table of file types that can be exported

The following table shows the file types that can be exported using the **File > Export** option.

Delcam Geometry (*.dgk)
Delcam Geometry + Features (*.dgk)
DDX (*.ddz)
DDX + DGK (*.ddz)
Rhino 4 (*.3dm)
Rhino 5 (*.3dm)
Point Formats (*.asc)
ACIS (*.sat)
Adobe Acrobat Document (*.pdf)
CADDS 4 Double Precision (*.c4x)
CATIA5 (*.CATPart)
Delcam Machining Triangles (*.dmt)
DWG (*.dwg)
DXF (*.dxf)
HPGL (*.plt)
IGES (*.ige)
IGES (*.igs)
IGES (*.iges)
Parasolid (*.x_b)
Parasolid (*.x_t)
Parasolid (*.xmt_bin)
Parasolid (*.xmt_txt)
Picture (*.pic)
STEP (*.step)
STEP (*.stp)
Stereolithographic - STL (*.stl)
Universal 3D Sample Software (*.u3d)
VDA-FS (*.vda)
Virtual Reality Markup Language (*.wrl)
Wavefront OBJ (*.obj)
POV-Ray scene (*.pov)
Delcam Electrode Archive (*.trode)
PowerSHAPE-e Model (*.emodel)
PS11320 - PS13223 (*.psmodel Ver23)

 indicates that this file format uses Delcam Exchange as the default translator.

Delcam Geometry (*.dgk)	This is a special format that defines the product's data in a single file. If objects are selected, only symbols, points, surfaces, solids, and wireframe objects in the selection are exported. If a workplane is active, it is automatically exported. If nothing is selected, the program exports symbols, points, surfaces, solids, wireframe objects and workplanes on levels which are selected. A symbol is exported as its individual objects. Symbol triangles are exported as a *dmt* file during *dgk* export. They will have the same name as the *dgk* file but with an extension *.dmt*. Component names are preserved when exporting the components as solids, for example when exporting an assembly component as *.dgk* file. If you have a model containing solids, surfaces and triangle data, you can select all the objects and export to a *.dgk* file. A corresponding *.dgk.dmt* file will also be exported. For details on importing these exported files see **Delcam Geometry (*.dgk)** in Table of file types that can be imported (see page 209)
Delcam Geometry + Features (.dgk)	Features will be in .mac and .xml files. This works in the same way as exporting selecting the **Create Hole Features for all DGK** on the **Manufacturing > Export** options dialog.
DDX (.ddz)	Export data in the *.ddz* format. This format is used to transfer data in Sketcher and PowerMILL Modelling. Geometric tolerances and datums are included in the export file.

DDX + DGK (.ddz)	Export data in the *.ddz* format. This format is used to transfer data in Sketcher and PowerMILL Modelling. All Parasolid solids are converted to V8 solids. Geometric tolerances and datums are included in the export file
Rhino 4(*.3dm) **Rhino 5(*.3dm)** 	Export the model as a Rhino file
Point Formats (.asc)	Export point cloud data in *.asc* format
ACIS (*.sat) 	Export selected surfaces and solids to this format. If nothing is selected, all the solids and surfaces in the model are exported. Solids are exploded into surfaces when you export.
Adobe Acrobat Document (*.pdf)	Exports drawings as .pdf files.
CADDS (*.c4x) 	Define the model as a CADDS file. CADDS files of version 4 can be exported from PowerSHAPE. The entire model is exported regardless of what is selected.
CATIA5 (*.CATPart) 	Export CATIA5 files using **Delcam Exchange** for exporting surfaces.
Delcam Machining Triangles (*.dmt)	Export the selected triangles for use by other Delcam products. Set the tolerance for exporting triangles on the **Triangle/Mesh** page of the **Options** dialog. You can also set the tolerance for exporting triangles on the **Import/Export** page of the **Options** dialog.
DWG Files (*.dwg) 	Export only the drawings in the model.

DXF Files (*.dxf) 	Define the model as DXF data (AutoCAD data). To export DXF files, you can use either Delcam Exchange or the DXF internal translator.
HPGL (*.plt)	Export the file as a plot file
IGES (*.ige) **IGES (*.igs)** **IGES (*.iges)** 	Define the model according to the IGES (International Graphics Exchange Standard) specification. Most High level CAD systems can provide models in this model format. Only selected objects are exported. If nothing is selected, you are asked whether you want to export the entire model. If a drawing window is selected, only this drawing is exported. Otherwise the model and all its drawings are exported. You can set how data is exported to IGES files using the **Options** dialog. Delcam Exchange may not export drafting objects to an IGES file. If you try to do so, a message is displayed explaining the drawbacks of using Delcam Exchange. The message asks you if you want to continue to use Delcam Exchange. If you select *NO*, the command is cancelled.
Parasolid (*x_b) **Parasolid (*.x_t)** **Parasolid (*.xmt_bin)** **Parasolid (*.xmt_txt)**	Export the selected surfaces to this format. If no surfaces are selected, then all the surfaces are exported. It is possible to export to earlier versions of Parasolid by selecting the appropriate version in the **Parasolid** options dialog. For details on methods for exporting Parasolid files, see Parasolid export methods.

Picture (*.pic)	Defines the model as a wireframe using arcs, lines and curves. PowerSHAPE can write DUCT5 pictures. Only selected wireframe objects and surfaces are exported.
pslast (*.pslast)	Export a shoe last to MIND software. *This option is only displayed in footwear mode.*
psstyle (*.psstyle)	Export style lines to MIND software. *This option is only displayed in footwear mode.*
Rhino (*.3dm) 	Export the model as a Rhino file
STEP (*.step) **STEP (*.stp)** 	Export selected surfaces and solids to this format. If nothing is selected, you are asked whether you want to export the entire model.
Stereo lithographic file STL format (*.stl)	Export a triangulated representation of each selected solid or surface in STL (Stereo Lithography) format. STL files normally represent closed solids as they are primarily used for rapid prototyping and many rapid prototyping machines require a fully closed triangular mesh. PowerSHAPE attempts to produce a closed mesh from the selected model. If it fails, you are prompted to close the mesh manually or save an 'open' STL file which may not be suitable for rapid prototyping. You can set options for exporting triangles on the **Triangle/Mesh** page of the **Options** dialog. You can also set the tolerance for exporting triangles on the **Import/Export** page of the **Options** dialog.
Universal 3D Sample Software (*.u3d) 	Export 3d data and embedded u3d data.

VDA-FS Files (*.vda) 	Defines the model as a VDA-FS file. VDA-FS version 2 files can be exported directly. All VDA-FS entities are supported, with the following constraints: curves of a degree higher than 3 are approximated by cubic beziers to tolerance. Only selected surface, solid and wireframe objects are exported.
Virtual Reality Markup Language (*.wrl)	Export the selected surfaces and solids to this format. Any VRML reader can then view these. By default, the material, lighting and view data are exported with the model. To export just the view data with the model, change the **Output Mode** option to model on the **Options** dialog.
Wavefront (*.obj)	Export a text file containing polygonal faces.
POV-Ray scene (*.pov)	Export a POV-Ray scene.
Delcam Electrode Archive (*.trode)	Export *.trode* files.
PowerSHAPE-e Model (*.emodel)	Export the model as a PowerSHAPE-e model.
PowerSHAPE model version **(*.psmodel VerXX)** where XX is the model version number.	Export the model to the previous PowerSHAPE format.

Exporting files using Delcam Exchange

A list of file types that are available when exporting using Delcam Exchange are given below.

ACIS Files (*.sat)
AutoCAD Files (*.dwg;*.dxf)
CATIA5 Files (*.CATPart)
DMT Files (*.dmt)
IGES Files (*.igs; *.ige; *.iges)
Parasolid Files (*.x_t; *.xmt_txt; *.x_b; *.xmt_bin)
PDF Files (*.pdf)
DGK Files (*.dgk)
Rhino Files (*.3dm)
STEP Files (*.stp; *.step)
STL Files (*.stl)
DDX Files (*.ddz; *.ddx)
U3D Files (*.u3d)
VDAFS Files (*.vda)
All Files (*.*)

For full details on what objects are exported using Delcam Exchange, see the on-line help available within Delcam Exchange.

Exporting without a licence

If you don't have a licence to export a particular type of file, you can purchase vouchers on a pay-per-use basis in **Delcam Exchange**.

1 From the **File** menu, select **Export** (see page 98).

The program detects automatically that there is no valid licence and displays a dialog giving you the option to view the file in **Delcam Exchange**.

2 Select one of the following options:

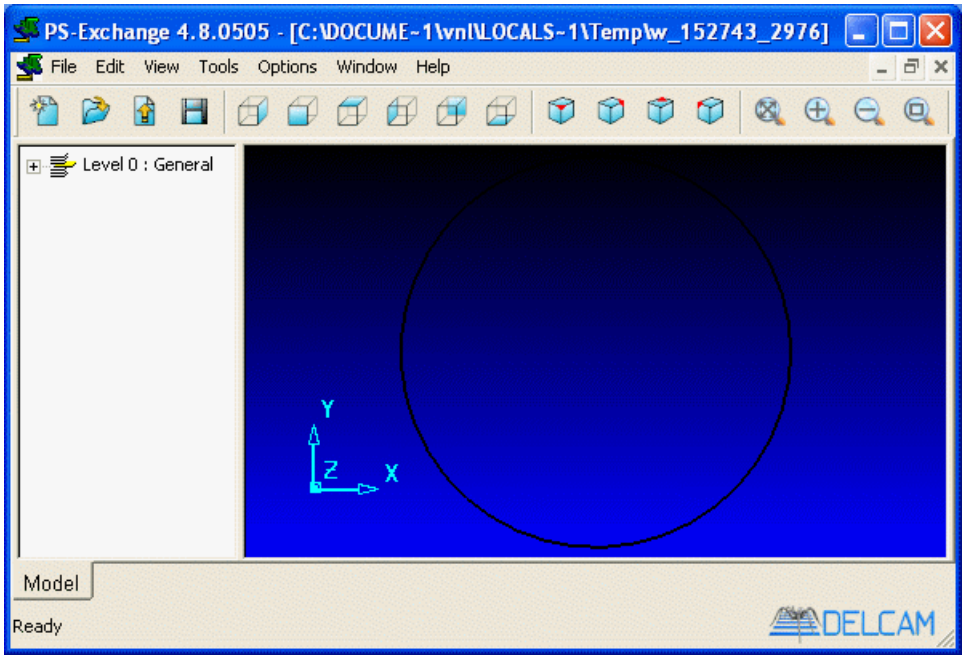
Yes — Starts **Delcam Exchange** where you can view the file and purchase a voucher to complete the export process (see page 205).

No — A dialog is displayed, detailing possible reasons for the failed export. The export may fail for opne of the following reasons:

- You have an invalid licence file.
- **Delcam Exchange** is incorrectly installed.
- **Delcam Exchange** registry entries are invalid.

Exporting without a licence using Delcam Exchange

Use this dialog to continue exporting files without a licence using **Delcam Exchange**. The file you want to export is displayed in the viewer.



1 Select **File - Export**.

2 Follow the **Delcam Exchange** file export process.

You will be prompted to purchase a voucher in order to complete the export for the selected file.

For further details, see the help provided with Delcam Exchange.

Exporting files using PowerSHAPE-e

If you want to use Delcam Exchange and don't have a licence to translate a particular type of file, you can purchase vouchers on a pay-per-use basis using the **Voucher Wizard**. For further details, see Using Vouchers (see page 108).

Exporting materials

1 Select **Format > Materials** to display the Material Selection dialog.

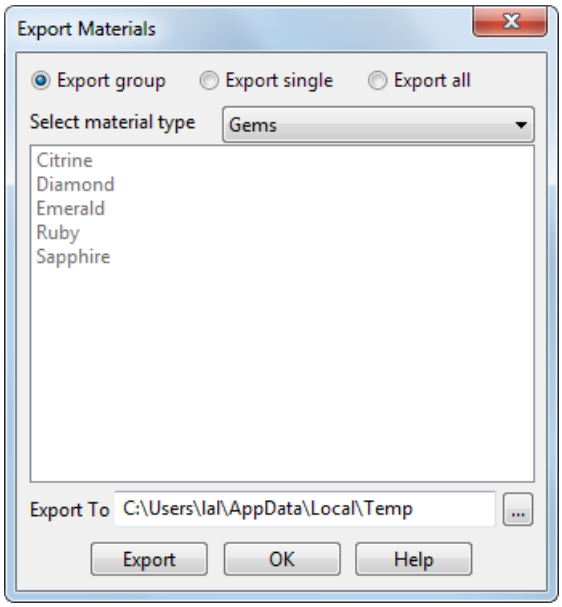
2 Select **Export**.

This option exports the following:

- A group of materials.
- A single material.
- All materials.

Export Materials dialog

Use this dialog to select materials to export.



To export a group of materails

1. Select **Export group** to export a group of materials.
2. From the **Select material type** list, select the group of materials that you want to export.
3. Enter the **Export To** path or click the Browse ... button to specify the path to the new *ExportMaterials* folder that is created to hold the materials images and the .csv file for your exported materials.
4. Click **Export**.
5. Click **OK** to export the group of materials and close the dialog.

To export a single material

1. Select **Export single Material** to export a single material.
2. From the **Select material type** list, select the group of materials that you want to export from.
3. From the list of corresponding materials displayed in the dialog, select the specific material you wish to export.
4. Enter the **Export To** path or click the Browse ... button to specify the path to the new *ExportMaterials* folder that is created to hold the materials images and the .csv file for your exported materials.
5. Click **Export**.
6. Click **OK** to export the material and close the dialog.

To export all materials

1. Select **Export all** to export all materials.
2. Enter the **Export To** path or click the Browse button to specify the path to the new *ExportMaterials* folder that is created to hold the materials images and the .csv file for your exported materials.
3. Click **Export**.
4. Click **OK** to export all materials and close the dialog.

Care must be taken if you modify the .csv file. If the .csv file is damaged, the Import Materials (see page 218) function, that you use to import the images into PowerSHAPE, will fail.

Creating a .csv file for materials

In order to use the **Import Materials** functionality you must have a *.csv* file that has been created previously. The *.csv* file is created automatically when you export using the **Export Materials** option. The *.csv* file can be viewed with Microsoft Excel or other editor.

If you want to import user-created materials into PowerSHAPE you must create a *.csv* file manually.

To create or modify a .csv file you will be working with PowerSHAPE's database, and it is advisable to be familiar with the attributes of materials in the "Materials" table in the PowerSHAPE database.

1. Create and name the .csv file. There are specific names for .csv files, as follows:

   Importing a group of materials

   GroupName_MaterialGroup.csv

   Importing a single material

   MaterialName_Material.csv

   Importing all materials

   Material_all.csv

2. Enter the following lines as the top two lines in the file.

   Material File

   UserCreated

3. Make entries in the file on the following lines in the sequence:

   Material name, Group Name, Red, Green, Blue, Polish, Emission, Translucency, reflect, shader_scale

4 You must also keep image files of the corresponding materials in the same folder where the .csv file is stored. The file names must follow the example below:

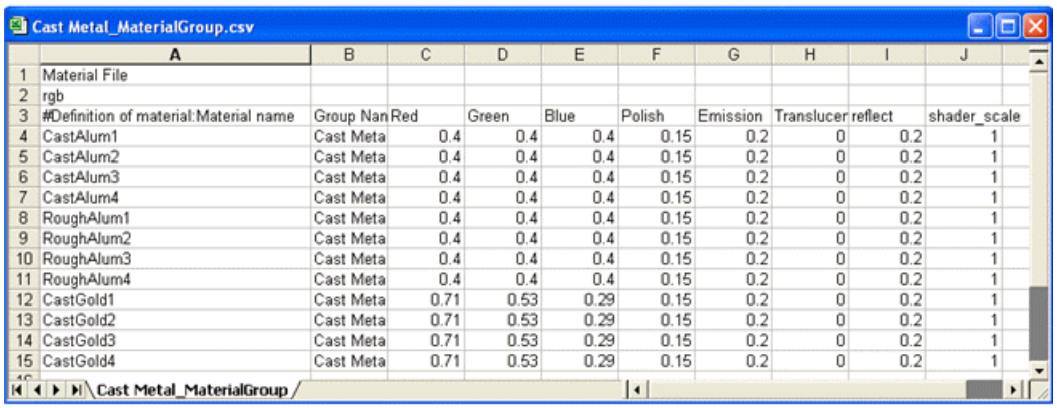
Material_Name.extension

where *extension* refers to ".jpg", ".bmp", ".tif" or ".rgb".

Viewing the csv file for materials

1 Locate the .csv file in the directory you specified as the directory to **Export to** on the **Export Materials** dialog.

2 Double click the filename to open the spreadsheet. It shows all the details for all the materials selected to be exported, shown in the example below:



Cast Metal_MaterialGroup.csv

	A	B	C	D	E	F	G	H	I	J
1	Material File									
2	rgb									
3	#Definition of material:Material name	Group Nan	Red	Green	Blue	Polish	Emission	Translucer	reflect	shader_scale
4	CastAlum1	Cast Meta	0.4	0.4	0.4	0.15	0.2	0	0.2	1
5	CastAlum2	Cast Meta	0.4	0.4	0.4	0.15	0.2	0	0.2	1
6	CastAlum3	Cast Meta	0.4	0.4	0.4	0.15	0.2	0	0.2	1
7	CastAlum4	Cast Meta	0.4	0.4	0.4	0.15	0.2	0	0.2	1
8	RoughAlum1	Cast Meta	0.4	0.4	0.4	0.15	0.2	0	0.2	1
9	RoughAlum2	Cast Meta	0.4	0.4	0.4	0.15	0.2	0	0.2	1
10	RoughAlum3	Cast Meta	0.4	0.4	0.4	0.15	0.2	0	0.2	1
11	RoughAlum4	Cast Meta	0.4	0.4	0.4	0.15	0.2	0	0.2	1
12	CastGold1	Cast Meta	0.71	0.53	0.29	0.15	0.2	0	0.2	1
13	CastGold2	Cast Meta	0.71	0.53	0.29	0.15	0.2	0	0.2	1
14	CastGold3	Cast Meta	0.71	0.53	0.29	0.15	0.2	0	0.2	1
15	CastGold4	Cast Meta	0.71	0.53	0.29	0.15	0.2	0	0.2	1

Cast Metal_MaterialGroup

Importing Data - Advanced

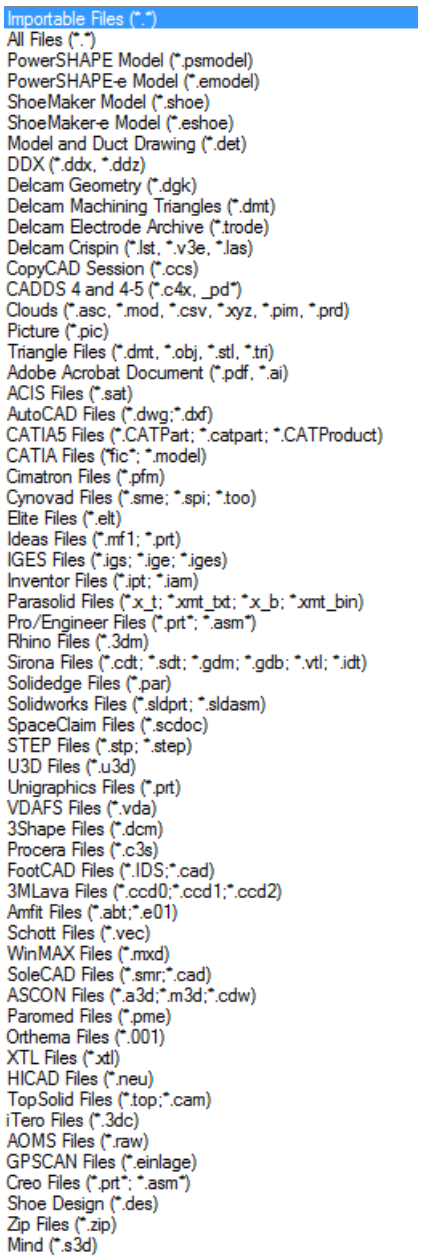
Select Tools > Options > Data Exchange > Version 8/Surfaces and use the **Import** options for importing files.

For further advanced details about importing data, select from the following:

- Starting up PowerSHAPE and importing a file (see page 216)
- Table of file types that can be imported (see page 209)
- Importing files using Delcam Exchange (see page 215)
- Importing data without a licence (see page 216)
- Importing Materials (see page 218)

Table of file types that can be imported

The file types that can be imported into PowerSHAPE are displayed in the drop-down list on the **Select a file to import** dialog.



The file types that are available to you will depend on the version of Delcam Exchange that you have installed

The file types that can be imported (using the **File > Import** option) are shown in the following table:

☆PowerSHAPE uses Delcam Exchange to import data, unless indicated in the table below.

File format	Type of file
PowerSHAPE Model (*.psmodel) ☆	This is a PowerSHAPE model using the single file format. If you have a model on removable media (tape or disk) which does not appear in your model database, it can be imported into your model database. Each model is held in a directory that contains the model files.
PowerSHAPE-e Model (*.emodel) ☆	PowerSHAPE-e model. It can be imported into PowerSHAPE on a pay-per-use basis using the Voucher Wizard.
ShoeMaker Model (*.shoe) ☆	This file format is created by Delcam Crispin ShoeMaker.
ShoeMaker-e Model (*.eshoe) ☆	This file format is created by Delcam Crispin ShoeMaker -e.
Model and DUCT Drawing (*.det) ☆	PowerSHAPE and DUCT5 model and drawing format. If you have a model that does not appear in your model database, it must be imported. Each model is held in a directory that contains the model files. The contents of the drawing are imported into a new model with the same name as the drawing.
DDX (*.ddx, *.ddz) ☆	This is data that has been exported from Delcam Exchange.
Delcam Geometry (*.dgk) ☆	This is a special format that defines the product's data in a single file. If you import a model containing solids, surfaces and triangle data that has been previously exported, it will also import the corresponding .dgk.dmt file at the same time. So, both the solids, surfaces and triangle data will be imported. The .dgk and .dgk.dmt files must be in the same folder for the import to be successful
Delcam Machining Triangles (*.dmt) ☆	A *.dmt* file is imported as a symbol. The symbol definition is also stored in the model and has the same name as the file. For further details, see **Creating a symbol definition.**

Delcam Electrode Archive (*.trode) ☆	This is a special format that defines the product's data in a single file. An electrode archive (*.trode*) file is a single file that stores a collection of one or more electrode *.dgk* files with associated metadata (*.xml*) files and setupsheet images.
Delcam Crispin (*.lst, *.v3e, *.las) ☆	Import a last of created with Delcam Crispin software.
CopyCAD session (*.ccs) ☆	All of the surfaces, curves and triangles within the file are converted to levels in the current PowerSHAPE model. Workplanes are also converted, but scanlines, points, images and views are not converted.
CADDS 4 and 4.5(_pd*), (*.c4x)	Delcam Exchange may not import drafting objects from an IGES file. If you try to do so, a message is displayed explaining the drawbacks of using Delcam Exchange. The message asks you if you want to continue to use Delcam Exchange. If you select *NO*, the PowerSHAPE translator is automatically used
Cloud Formats (*.asc, *.mod, *.csv, *.xyz) ☆	Files containing point cloud information. The formats that are supported are *.asc* - ASCII *.mod* *.csv* - ASCII text encoded as Comma Separated Values. *.xyz* - Perceptron .
Picture (*.pic) ☆	This format defines the model as a wireframe using arcs, lines and curves. PowerSHAPE can read DUCT5 pictures.

Triangle files (*.dmt, *.obj, *.stl, *.tri) ☆	A *.dmt* or *.obj* file is imported as a symbol. The symbol definition is also stored in the model and has the same name as the file. For further details, see **Creating a symbol definition**. *.obj* imports a text file containing polygonal faces. *.stl* imports the triangulated representation in STL (Stereo Lithography) format. STL files normally represent closed solids as they are primarily used for rapid prototyping and many rapid prototyping machines require a fully closed triangular mesh.
Adobe Acrobat Document (*pdf, *.ai)	Files containing text and wireframe can be imported.. PowerSHAPE Pro and Toolmaker Pro can also import curves.
ACIS (*.sat)	This format defines the model as an ACIS file
AutoCAD (*.dwg,*.dxf)	This format defines the model as DWG or DXF data (usually exported from AutoCAD)
CATIA5 (*.CATpart, *.CATProduct) **Catia (*.cat, *.exp, *fic*, *model)**	These define the model as a Catia file. The 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 the **Open a new file for each fic file** option on the **Import/Export** page of the **Options** dialog.
Cimatron (*.pfm)	This format defines the model as a CADDS file. CADDS files of version 4 and 5 can be imported into PowerSHAPE
Cynovad (*.sme, *.spi, *.too)	This file has been generated by dental software.
Elite (*.elt)	This defines the model as a Cimatron file
Ideas (*.mf1), (*.prt)	This format defines the model as an Ideas file

IGES (*.ig*)	This defines the model according to the IGES (International Graphics Exchange Standard) specification. Most high level CAD systems can provide models in this model format. You can set how data is imported from IGES files using the **Options** dialog. Blanked IGES surfaces are imported as blanked.
Inventor (*ipt)	This format defines the model as an Autodesk Inventor file
Last and Style (*.pslast, *.psstyle) ☆	Import a last and style created with MIND software *This option is only displayed in footwear mode.*
Parasolid (*.x*) ☆	This format defines the model as a UG Parasolid file
Pro/Engineer (*.asm, *.prt*)	These formats define the model as a Pro-Engineer file.
Rhino (*.3dm)	This format defines the model as a Rhino file. When importing Rhino files, all triangle data is also imported.
Sirona (*.cdt, *.sdt, *.gdm, *.gdb, *.vtl, *.idt)	Import a file from dental software inLab 3D.
Solidedge (*.par)	This format defines the model as a SolidEdge file.
Solidworks (*.sldprt, *.sldasm)	This format defines the model as a SolidWorks file.
SpaceClaim (*.scdoc)	This format defines the model as a SpaceClaim file.
STEP (*.step, *.stp)	This format defines the model as a STEP file.
U3D	This is a graphics file; U3D is the 3D element of a 3D PDF file; it can also be used as a stand-alone traingle file-format
Unigraphics (*.prt)	This format defines the model as a Unigraphics file.

VDA-FS (*.vda)	This format defines the model as a VDA-FS file. VDA-FS version 2 files can be imported directly. All VDA-FS entities are supported, with the following constraints: curves of a degree higher than 3 are approximated by cubic beziers to tolerance.
3Shape (*.dcm)	Import a file from 3shape dental software
Procera (*.c3s)	Import a file that was generated by Procera dental software.
FootCAD (*.IDS, *cad)	Import a file that was generated by IDEAS orthotic software.
3MLava (*ccd0, *.ccd1, *.ccd2)	Import a file from 3M dental software.
Amfit (*.abt, *e01)	Import a file from Amfit orthotic software
Schott (*.vec)	Import a file from Schott general purpose 3D software
WinMAX (*.mxd)	Import a file that was generated by WinMAX V3.7 MXD Files
SoleCAD (*.smr, *.cad)	Import a file that was generated by IDEAS orthotic software.
ASCON (*.a3d, *.m3d, *.cdw)	Import a file that was generated by ASCON general purpose CAD software.
Paromed (*.pme)	Import a file that was generated by Paromed orthetic software.
Orthema (*.001)	Import a file that was generated by Othema orthotic software.
XTL (*.xtl)	Import a file from dental software
HICAD (*.neu)	Import a file that was generated from the HICAD CAD system
TopSolid (*.top, *.cam)	This file was generated by the TopSolid CAD system.
iTero (*.3dc)	The file was generated by CADDent.
AOMS (*.raw)	The file is from a Sharp Shape orthotic scanner.
GPSCAN (*.einlage)	This file contains Gebiom orthotic scan data.
Creo (*.prt, *.asm)	This filewas generated by Creo CAD system

Shoe Design (*.des) ☆	This file was generated by Delcam Crispin Shoe Design.
Zip (*.zip) ☆	You can import a zip file if it contains one of the following: ▪ a single file such as *IGES* or *dgk* ▪ a single *psmodel* file ▪ a single directory model
Mind (*s3d)	Import a file from MIND shoe software.

Importing files using Delcam Exchange

A list of files you can import using Delcam Exchange are given below. We also give the benefits of using built in translators and Delcam Exchange (if any).

ACIS Files (*.sat)
AutoCAD Files (*.dwg;*.dxf)
CATIA5 Files (*.CATPart; *.catpart; *.CATProduct)
CATIA Files (*fic*; *.model)
Cimatron Files (*.pfm)
Cynovad Files (*.sme; *.spi; *.too)
DMT Files (*.dmt)
Elite Files (*.elt)
Ideas Files (*.mf1; *.prt)
IGES Files (*.igs; *.ige; *.iges)
Inventor Files (*.ipt; *.iam)
Parasolid Files (*.x_t; *.xmt_txt; *.x_b; *.xmt_bin)
Part Files (*.psmodel; doc.det)
DGK Files (*.dgk)
Pro/Engineer Files (*.prt*; *.asm*)
Rhino Files (*.3dm)
Sirona Files (*.cdt; *.sdt; *.gdm; *.gdb; *.vtl)
Solidedge Files (*.par)
Solidworks Files (*.sldprt; *.sldasm)
SpaceClaim Files (*.scdoc)
STEP Files (*.stp; *.step)
STL Files (*.stl)
DDX Files (*.ddx; *.ddz)
U3D Files (*.u3d)
Unigraphics Files (*.prt)
VDAFS Files (*.vda)
3Shape Files (*.dcm)
Procera Files (*.c3s)
FootCAD Files (*.IDS;*.cad)
3MLava Files (*.ccd0;*.ccd1;*.ccd2)
Amfit Files (*.abt;*.e01)
Schott Files (*.vec)
WinMAX Files (*.mxd)
SoleCAD Files (*.smr;*.cad)
ASCON Files (*.a3d;*.m3d;*.cdw)
Paromed Files (*.pme)
Orthema Files (*.001)
XTL Files (*.xtl)
HICAD Files (*.neu)
TopSolid Files (*.top;*.cam)
iTero Files (*.3dc)
AOMS Files (*.raw)
GPSCAN Files (*.einlage)
PDF Files (*.pdf)
All Files (*.*)

For full details on objects that are imported using Delcam Exchange, see the on-line help available within Delcam Exchange.

Starting up PowerSHAPE and importing a file

When you start up the program you can open a model and import a file at the same time by typing:

powershape -file f1 -import f2

where **f1** is the name of the model and **f2** is the path of the imported file.

You can also use the following commands:

powershape f1 f2

powerSHAPE -import f2 -file f1

To import a file into a new model, type one of the following commands:

powerSHAPE -import f2

powerSHAPE f2

For the above commands to work, you must have FILE OPEN IFNONE in your login macro. For further details, see What is the login macro? (Managing your PowerSHAPE system).

Importing without a licence

If you don't have a licence to import a particular type of file, you can purchase vouchers on a pay-per-use basis in **Delcam Exchange**.

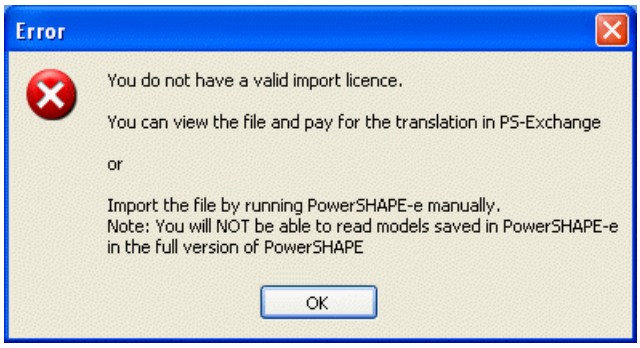
1 Select **File > Import** (see page 104).

The program detects automatically that there is no valid licence and displays a dialog giving you the option to view the file in **Delcam Exchange.**

2 Select one of the following options:

Yes — Starts **Delcam Exchange** where you can view the file and purchase a voucher to complete the import process (see page 217).

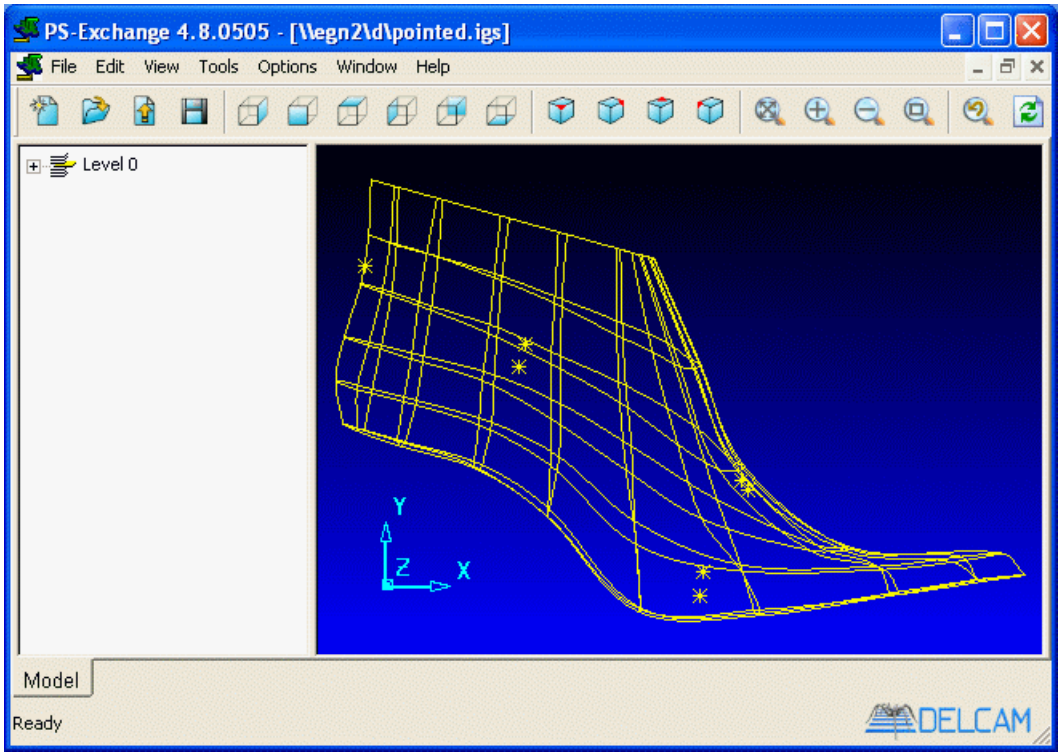
No — Exits the import process and displays a dialog



3 Click **OK** to exit. A log report is displayed.

Importing without a licence using Delcam Exchange

Use this dialog to continue the file import process using **Delcam Exchange**. The file you want to import is displayed in the viewer.



In order to import the file into PowerSHAPE you must first export it in **Delcam Exchange**. The file type you export to is automatically selected by **Delcam Exchange**.

1 In the **Delcam Exchange** viewer, select **File >Export**

The file type is already selected in the dialog.

*It is recommended that you **do not change** this file type.*

2 Click **Export** and follow the **Delcam Exchange** export process.

You will be prompted to purchase a voucher in order to complete the export for the selected file.

For further details, see the help provided with **Delcam Exchange**.

Once the file is exported in **Delcam Exchange**, you can import the file directly into PowerSHAPE, using **File >Import** (see page 104).

Importing Materials

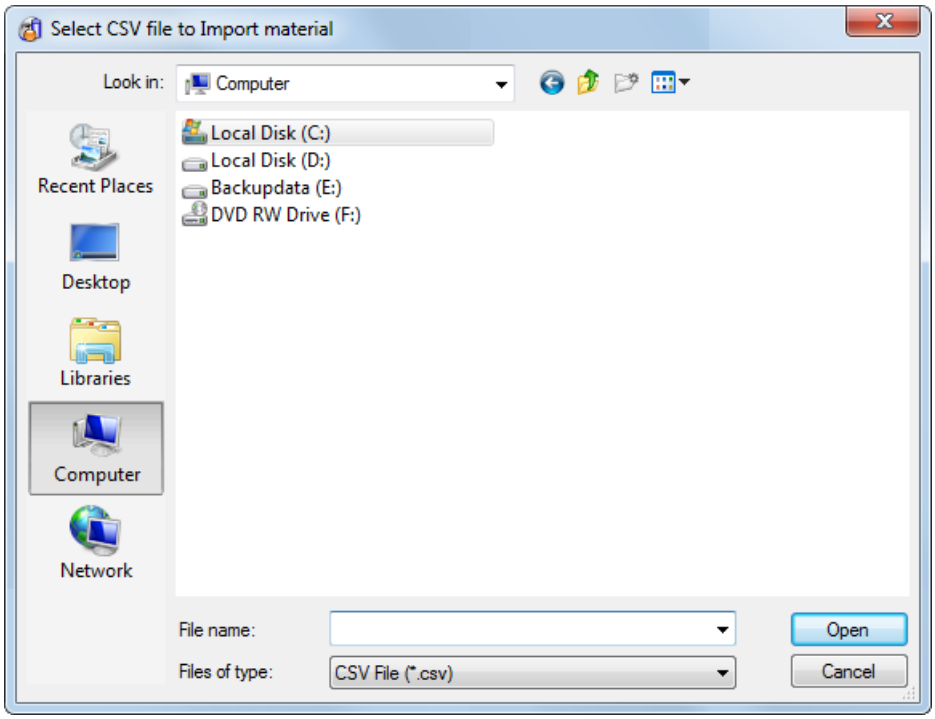
In order to import materials you must have a .csv file that has been created previously. A .csv file is created automatically when you export materials using the **Export Materials** option on the Material Selection dialog. For further details, see Exporting data from a model (see page 98)

If you want to import user-created materials you must create a .csv file manually.

1. Select **Format > Materials** to display the Material Selection dialog.
2. Select **Import**. The Select CSV file to Import material dialog (see page 218) is displayed. For further details see, Creating a csv file for materials **(see page 207)**

Select CSV file to import material dialog

Use this dialog to select CSV files to import materials.



1. Navigate to the required folder and select the valid *.csv* file containing the materials data that you wish to import into your databaserequired file.

*In order to import materials you must have a .csv file that has been created previously. A .csv file is created automatically when you export materials using the **Export Materials** option on the Material Selection dialog. If you want to import user-created materials into PowerSHAPE you must create a .csv file manually.*

2 Click **Open** to import the materials data contained in the .csv file.

You must restart the program after importing materials in order to make the materials available for use.

Parameters, advanced

For further advanced details about creating parameters, select from the following:

- Creating parameters in a spreadsheet (see page 219)
- Creating multiple parameters in a spreadsheet (see page 221)

Creating parameters in a spreadsheet

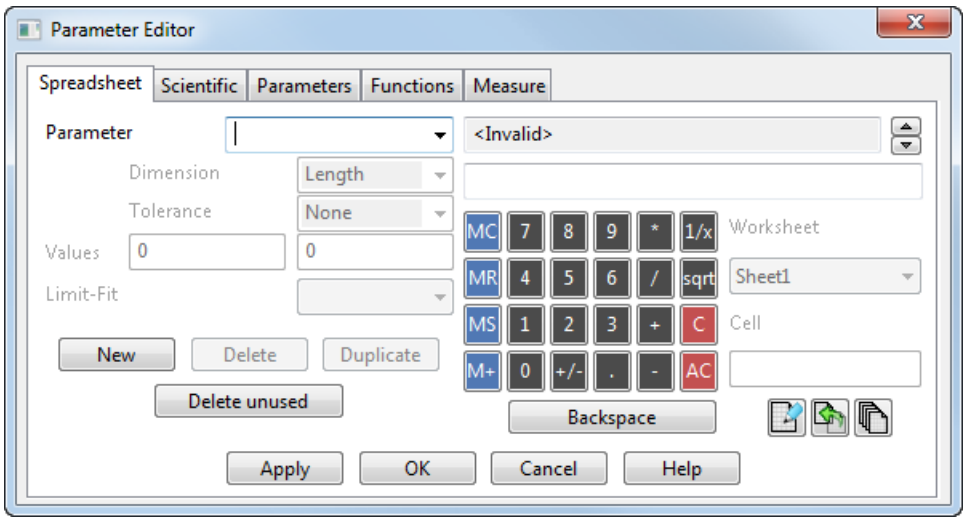
For models that require complex calculations or have specific data associated with them, these values can be created and edited in a Microsoft Excel spreadsheet that is stored with the model.

Each new model has a new, blank spreadsheet ready for use.

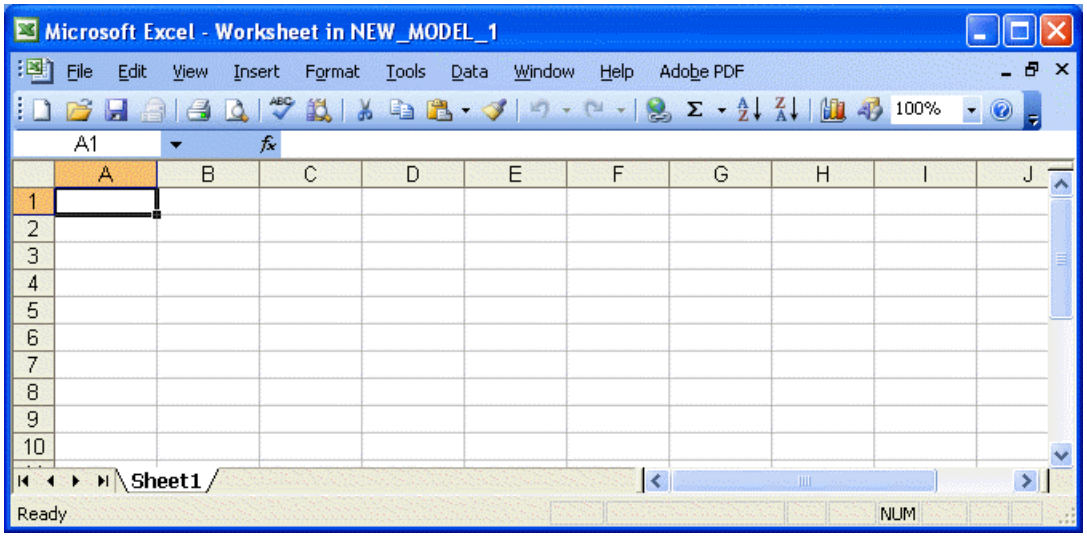
You need a licence for Microsoft Excel to use the spreadsheet options.

1 From the **Object** menu, select **Parameter** to display the **Parameter Editor** dialog.

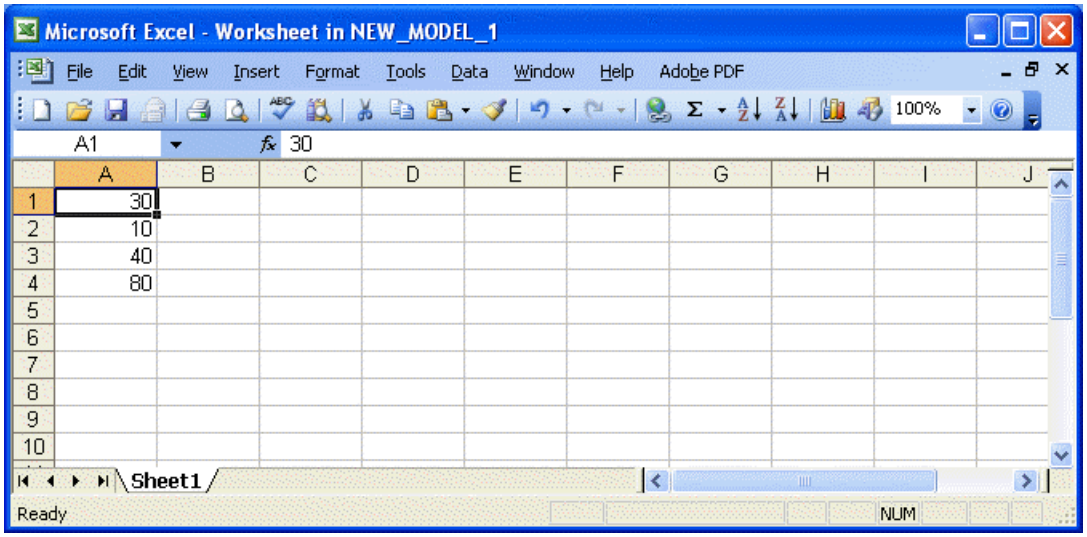
2 Select the **Spreadsheet** tab.



3 Click  to display a Microsoft Excel spreadsheet.



4 Enter values and any calculations into the spreadsheet.



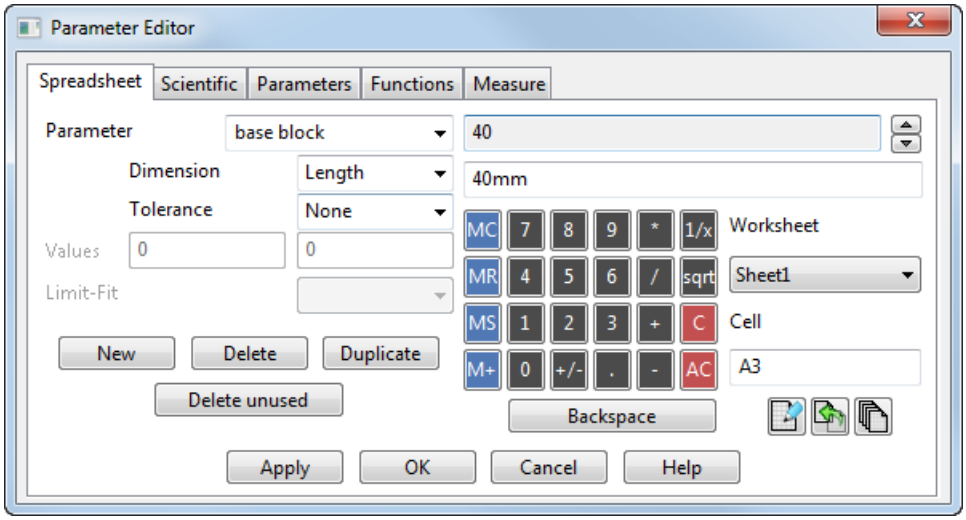
5 Select **File**, and select **Close and return to New_Model_1**, where *New_Model_1* is the name of the current model in PowerSHAPE.

6 In the **Parameter Editor** dialog, Select **New** and enter the parameter name in the **Parameter** text box.

7 In the **Cell** text box, enter the cell for the value you want to work with.

For example enter **A3**.

8 Click to display the cell's value is displayed in the **Expression** text box.



9 Click **Apply** to store the parameter and continue.

10 Click **OK** to store the parameter and exit.

Editing parameters in a spreadsheet

To edit the values in the spreadsheet:

1 Select . The spreadsheet stored with the model is displayed. You can make changes, add new values or delete values.

2 In the spreadsheet, select **File**, and select **Close and return to New_Model_1**, where *New_Model_1* is the name of the current model.

3 Select to synchronise the program with the Excel spreadsheet. The spreadsheet is updated with any changes to the values. Select this option each time you make any changes.

Use the same method to add new values.

Creating multiple parameters in a spreadsheet

You can create numerous parameter names and values in Microsoft Excel spreadsheets and apply them all at the same time.

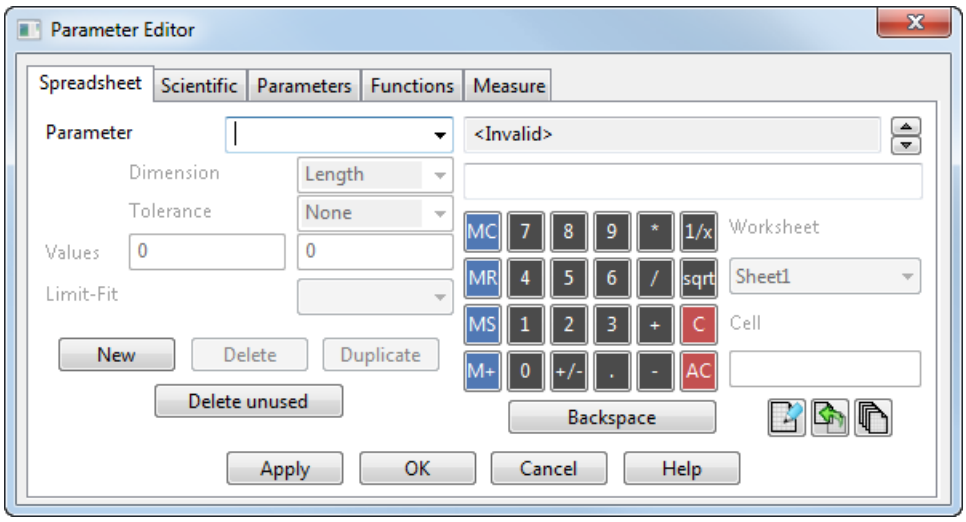
When these are created, the parameters are available to be used in a text box on dialogs. For further details, see Finding out if a value uses parameters (see page 186).

You need a licence for Microsoft Excel to use the spreadsheet options.

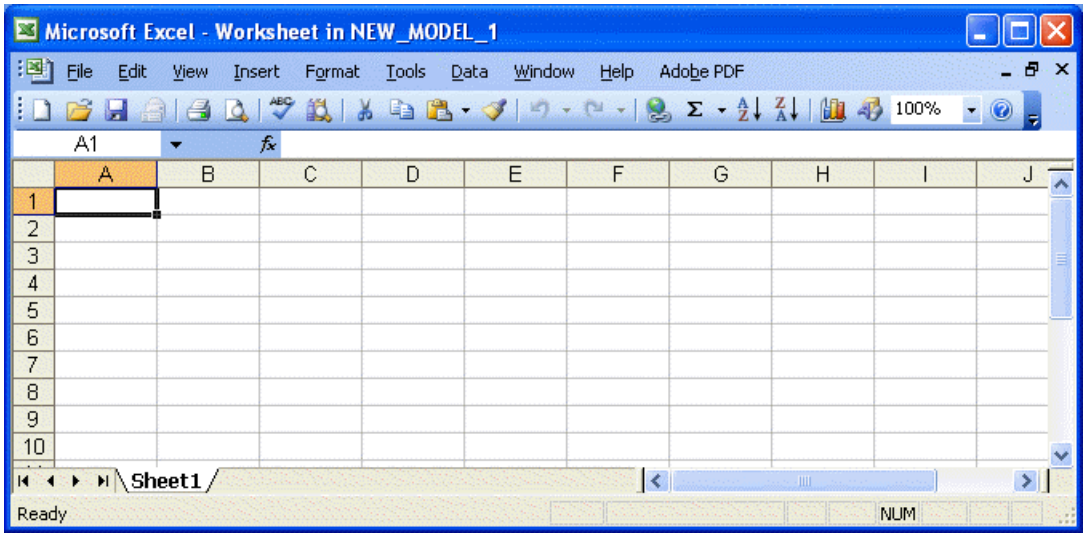
To create multiple parameters, you need to enter the data into the spreadsheet.

1 From the **Object** menu, select **Parameter** to display the **Parameter Editor** dialog.

2 Select the **Spreadsheet** page.

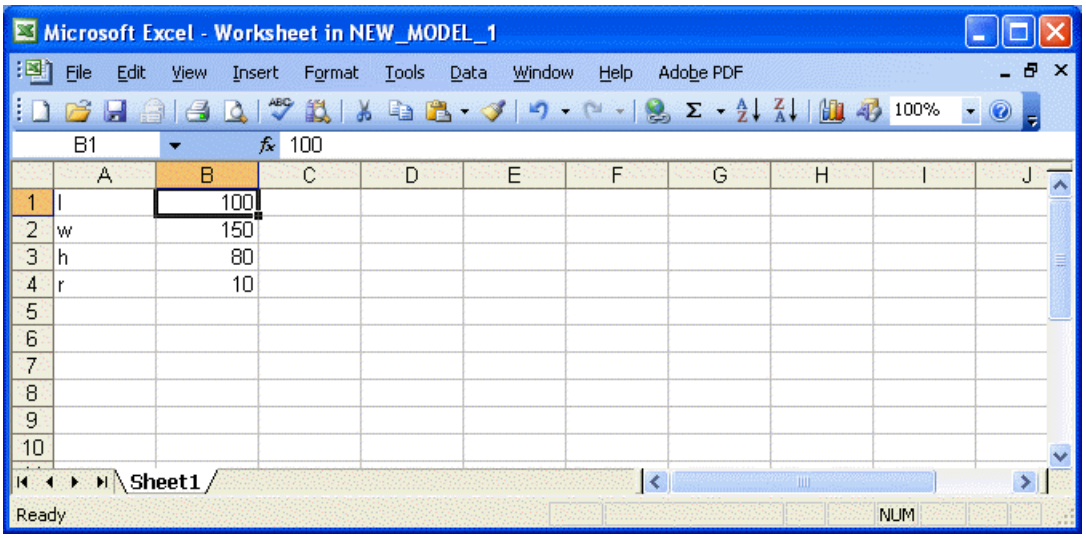


3 Select  to modify the spreadsheet. A Microsoft Excel spreadsheet is displayed.



4 Enter one column with parameter names and another column with their values.

The example below shows parameter names entered in column **A** and their values entered in column **B**.



5 Select **File**, and select **Close and return to New_Model_1**, where *New_Model_1* is the name of the current model in PowerSHAPE.

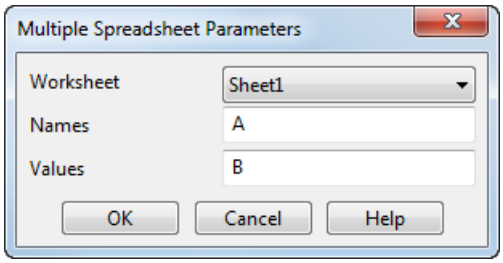
6 On the **Parameter Editor** dialog, select to create multiple spreadsheet parameters.

The Multiple Spreadsheet Parameters dialog (see page 223) is displayed.

Use this dialog to create a set of parameters with the values entered on the spreadsheet that are to be available to be selected in dialogs.

Multiple Spreadsheet Parameters dialog

Use this dialog to create a set of parameters for the values entered on the spreadsheet that are to be available to be selected in dialogs. For further details, see Creating multiple parameters in a spreadsheet (see page 221).



1 Select a **Worksheet** from the drop down list.

2 Enter the column letter from the spreadsheet where the parameter names were entered. For example **A**. For further details, see Creating multiple parameters in a spreadsheet (see page 221)

3 Enter the column letter from the spreadsheet where the parameter values were entered. For example **B**. For further details, see Creating multiple parameters in a spreadsheet (see page 221)

4 Click **OK**.

This creates a set of parameters with values that are now available to be selected in dialogs. For further details, see Finding out if a value uses parameters (see page 186)

Editing multiple parameters in a spreadsheet

Multiple parameters are entered in a Microsoft Excel spreadsheet that is stored with each model. To edit existing data in a spreadsheet,

1 From the **Object** menu, select **Parameter** to display the **Parameter Editor** dialog.

2 Select the **Spreadsheet** page.

3 Select  . A Microsoft Excel spreadsheet is displayed.

4 You can edit existing data and add new data to the spreadsheet.

5 On the spreadsheet, select **File**, and select **Close and return to New_Model_1**, where *New_Model_1* is the name of the current model in PowerSHAPE.

6 Select  on the **Spreadsheet** page of the Parameter Editor dialog (see page 184). For further details, see Creating multiple parameters in a spreadsheet (see page 221).

This synchronises PowerSHAPE with the Excel spreadsheet to update all the values when changes have been made.

Select **Synch** each time you edit the spreadsheet.

Arm

Click **Arm**  on the status bar (see page 30) and select from the following:

- Connecting the Arm (see page 225)
- Using the Arm probe (see page 226)
- Using a bounding box to define an object (see page 227)
- Creating objects using the probe (see page 227)
- Basic Paddle Operations (see page 228)

 *Arm functionality does not work with 64-bit Windows.*

Installing the Arm

PowerSHAPE currently supports the following digitising devices:

- **MicroScribe G2X**. The drivers for this are built into PowerSHAPE, so this is available without additional software.
- **WACOM tablet.** The drivers for the specific tablet need to be installed. These are provided with the tablet, or can be downloaded from the relevant web site.

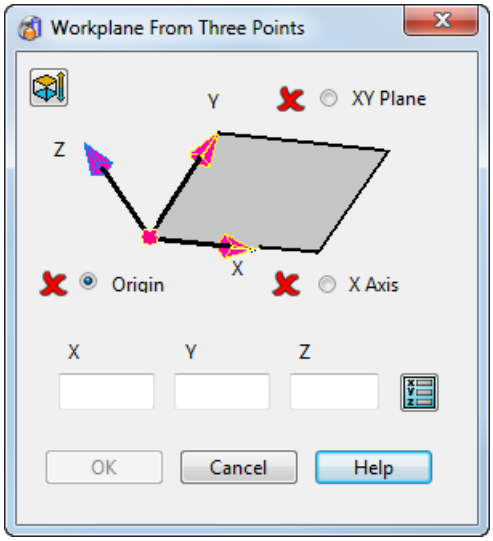
To use other devices with PowerSHAPE:

1. Install a version of PowerINSPECT.
2. Run the PowerINSPECT DRO. This will register the .dll files
3. Select **CMM** from the top toolbar followed by **Configuration**.
4. Select the connection protocol for your device. Documentation on the installation of the correct protocol for your device can be obtained from your Sales Partner.
5. Exit PowerINSPECT DRO.
6. Start PowerSHAPEand click  (*Status bar*)

Connecting the Arm

Use the **Arm**  button on the status bar to connect or disconnect the Arm. To connect to the Arm, you need to have an appropriate licence file and drivers installed on your computer.

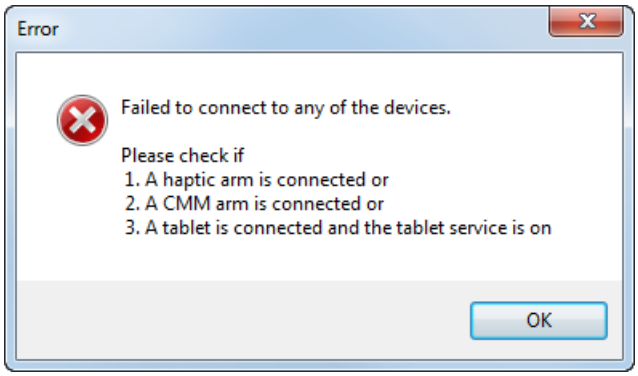
1. Select the **Arm**  button from the status bar to connect to the Arm.
2. The **Workplane from three points** dialog is displayed automatically where you can specify the three points to create a workplane.



*If you do not wish to define a workplane at this point select Cancel on the dialog. You can define a workplane later, if required, by selecting **Workplane from three points** option (Workplane menu).*

3 A spherical, red cursor is displayed to represent the Arm probe. PowerSHAPE is now ready to accept the digitised components of the probe.

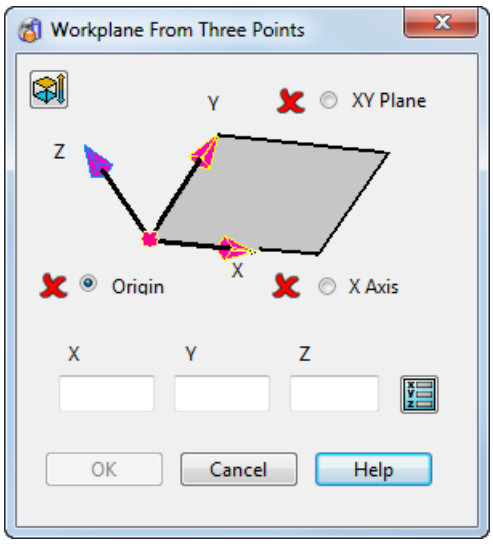
If the Arm is not connected, the following dialog is displayed.



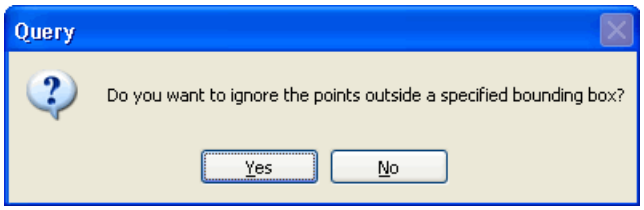
Using the Arm probe

To use the probe on the Arm:

1 Select three points to create the workplane. The corresponding **X**, **Y**, **Z** option is selected interactively on the **Workplane from three points** dialog as each point is selected.



2 Select **OK** when you have defined the workplane. The **Query** dialog is displayed.



3 Click one of the following:

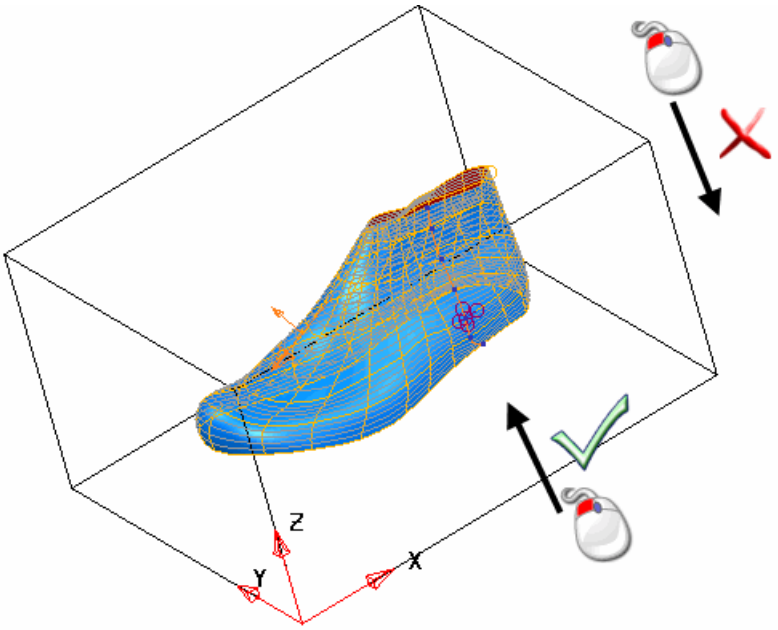
Yes to ignore the points outside the bounding box

No to create a bounding box around all the object to be defined (see page 227).

Using a bounding box to define an object

Creating a bounding box can be helpful to fix the working area in 3D space.

To define the bounding box, select two opposite corners using the probe on the Arm (see page 226).



When a bounding box is created, the Arm cursor is only displayed if it is within the bounding box. You can only define points within the bounding box. If you try to define a point that is outside the bounding box, it is not registered.

Creating objects using the probe

Create objects using one of the following techniques:

- **Digitising physical geometry**

Select a creation option, for example, line or bezier curve.

- **Digitising a surface**

  Select the **Surface Creation from Patches** option.

- **Selecting a point**

  Use the probe in the same way as you would use a mouse to select a point. The coordinates selected by the probe are in **World** workspace.

Basic Paddle Operations

Basic paddle operations are as follows:

- The small green box shows the current X, Y and Z coordinates of arm probe.
- Press the right button on the arm paddle to enter the point selected by the probe.
- Press the left button to finish entering the command.
- Press both the buttons simultaneously to cancel the current command.
- Press the left button on the arm paddle to end the current creation of a line or curve. It stays in the same application mode so that you can continue to create the next line or curve.

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