
PowerSHAPE 2015 R2

Reference Help

Delcam Draft



PowerSHAPE

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Patent Information

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

Morphing functionality is subject to patent application GB 2401213.

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Delcam Draft

Use the following sections to find information on Delcam Draft:

Drafting (see page 3)

Drawings (see page 6)

Notes (see page 38)

Views (see page 42)

Annotation (see page 116)

Dimensions (see page 189)

Hatching (see page 242)

Balloons (see page 147)

Schedules (see page 269)

Embedded Bill of Materials (see page 304)

Symbols (see page 260)

Drafting - an overview

The following sections give an overview of the drafting functionality:

Introduction to Delcam Draft (see page 3)

Start Delcam Draft (see page 4)

Exit Delcam Draft (see page 6)

What is the Delcam Draft?

Delcam Draft within PowerSHAPE provides tools to create a 2D representation (an engineering drawing) of your model.

There are two ways you can create a drawing of your model:

- From scratch using wireframe.

- Using a 3D model created in PowerSHAPE.

You can fully annotate your drawing using tools such as hatching, dimensioning, and text.



Create a solid of your model where possible before drafting from it. This improves the quality and speed of hidden-line views. It also improves the accuracy of dimensioning.

Entering Drafting mode

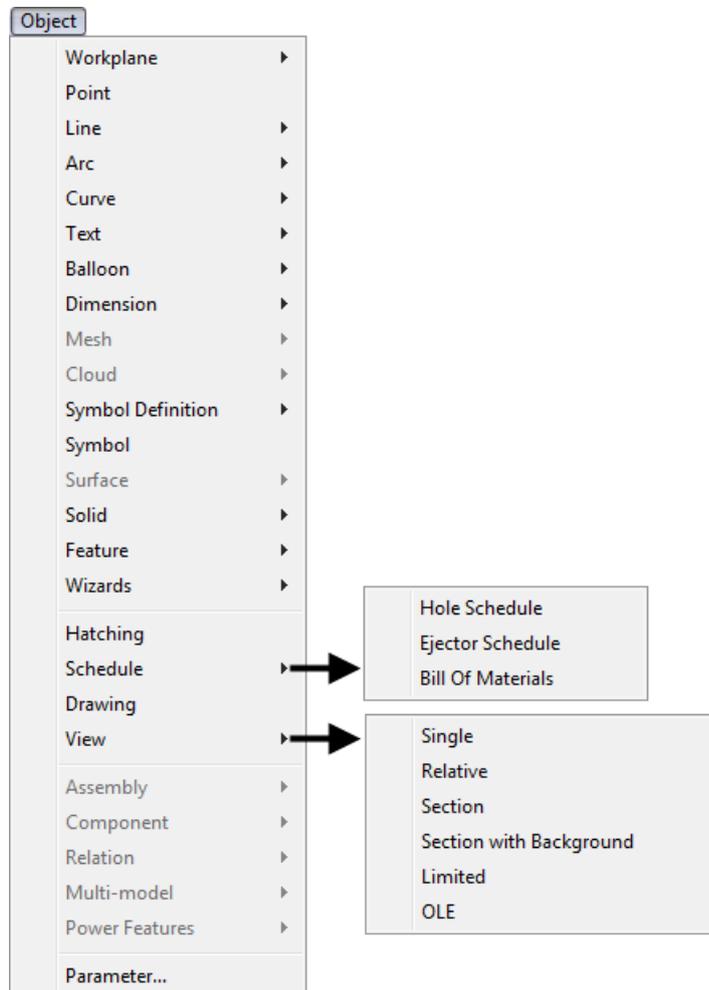
You can start Delcam Draft in the following ways:

Using the menus

- Select **Module > Drafting** from the menu. The drafting toolbar is displayed.



- You can select drafting options from the **Object** menu.





Assembly modelling functionality is available from Drafting mode.

Using the toolbar

- 1 Click  (Main toolbar).



This is the same as hovering over  and selecting  from the flyout..



The **Drawing Selector** dialog is displayed and the drafting toolbar is displayed. For further details, see Creating a new drawing (see page 8).



If a drawing is already open when you enter Drafting mode, the last accessed drawing is displayed in the Open Drawing dialog.



When you switch between Modelling mode and Drafting mode, the correct mode is automatically enabled. For example if you select a drawing, Drafting mode is switched on automatically.

The **Drafting** toolbar contains these buttons:



Drawing (see page 8)



View (see page 42)



Workplane



Line



Arc



Curve



Annotation (see page 116)



Hatching (see page 243)



Symbol (see page 260)



Schedule (see page 269)

Exiting Drafting mode



Click  to exit Drafting mode and display the **Main** toolbar.

Drawings

Use the following sections to create and edit drawings in PowerSHAPE:

Introduction to drawings, including examples (see page 6)

Create a drawing (see page 8)

Create a title block on a drawing (see page 13)

Create objects on a drawing (see page 14)

Editing a drawing (see page 15)

Drawing templates (see page 19)

Creating Component Drawings (see page 28)

What is a drawing?

A drawing is equivalent to a piece of paper. A model can contain a number of drawings. You can use drawings to:

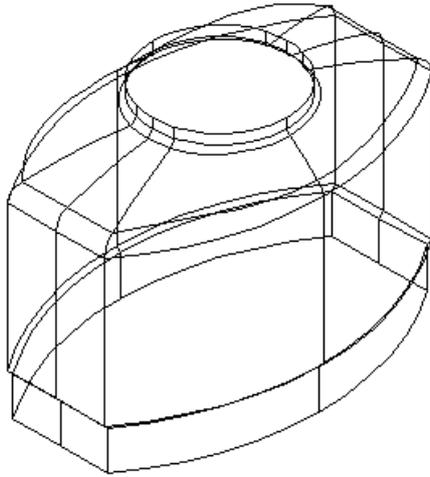
- enter and annotate 2D geometry.
- create views (see page 42) of a 3D model and annotate the model directly.



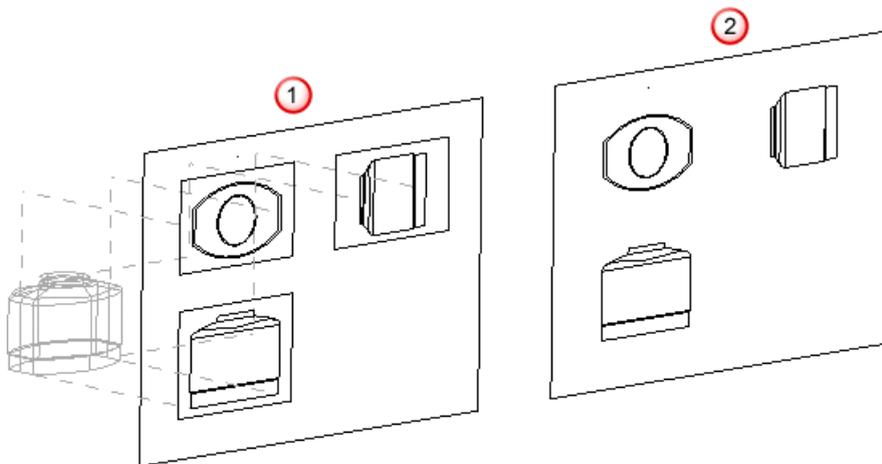
Create a solid of your model where possible before drafting from it. This improves the quality and speed of hidden line views. It also improves the accuracy of dimensioning.

Drawing examples

You can create a drawing of the following example model in two ways.



- Create the model in PowerSHAPE and create views on a drawing of the different projections of the model ①.
- Create objects directly onto the drawing ②.



Objects in 3D models and drawings

Objects can exist in two places:

- In the **3D** model.
- On the **drawing**.

Objects created in the **3D** model are displayed:

- in the model window.
- in views on drawings.

If you create an object on a drawing, you can see it only on that drawing. These objects may partially or wholly overlay a view, as if they are drawn on the glass of the *window* through which the model is viewed.

Creating a drawing

The following sections provide information on creating drawings:

- Creating a new drawing (see page 8)
- Changing the settings of a drawing (see page 9)
- Opening existing drawings (see page 10).
- Converting wireframe drawings (see page 13)

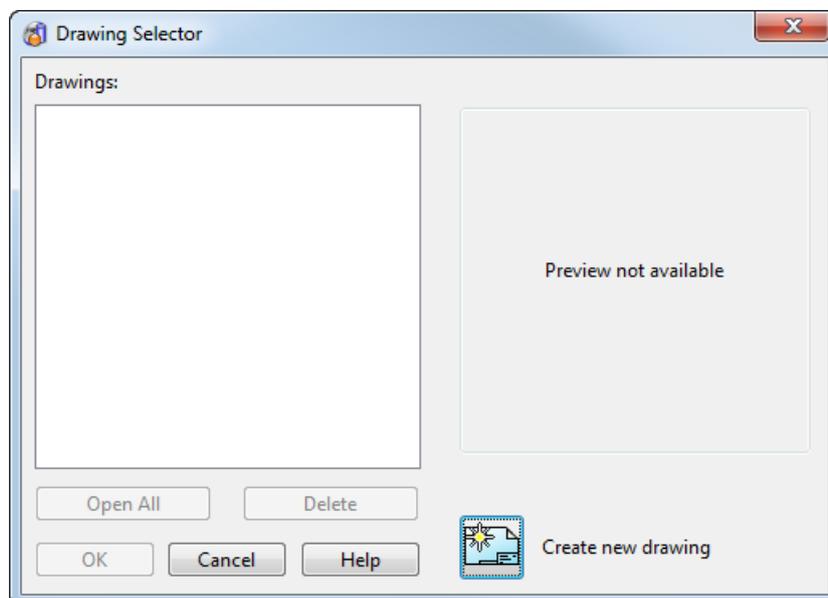
Creating a new drawing

To create a new drawing:

- 1 Click  **Modelling Mode** (*Main toolbar*), to display the **Drawing Selector** dialog.



- 2 Click **Create new drawing** .



- 3 A new drawing is displayed in the graphics window. Each drawing sheet is automatically named with an **S** followed by the drawing number.
- 4 A single view of the displayed model is created automatically and is attached to the cursor. Click in the drawing to position the single view.

You can now create either 2D geometry (see page 14) on the drawing or views (see page 42) of a 3D model.

You can change the default settings of the drawing (see page 9) using the **Options** dialog.

You can edit the drawing (see page 15).

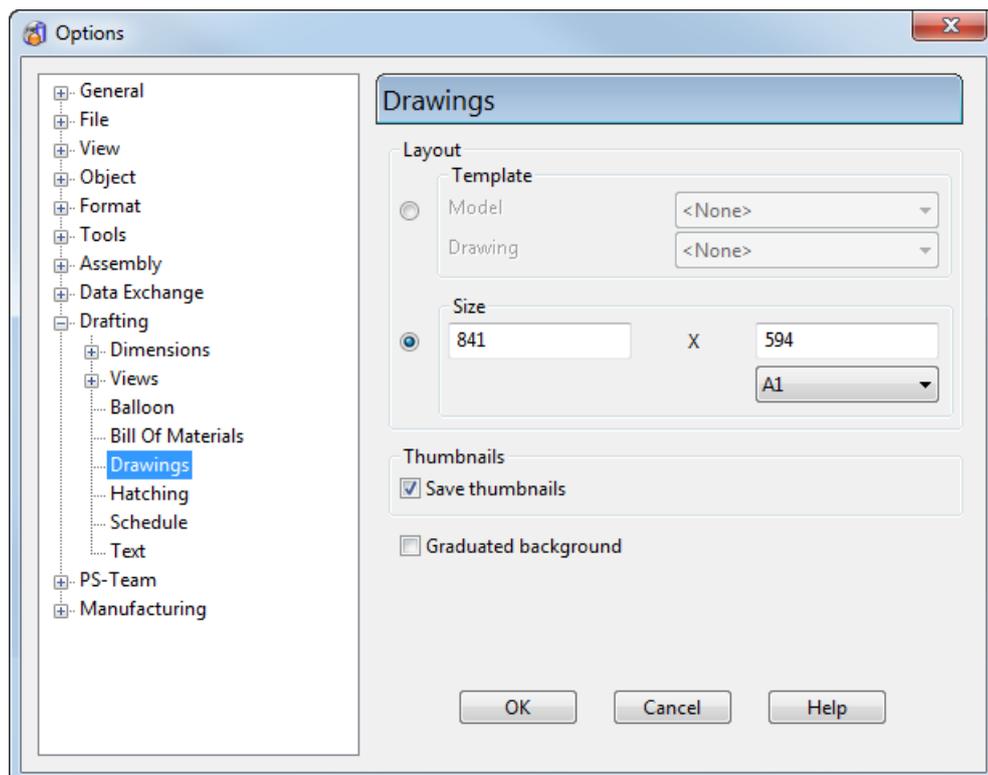
Changing the settings of a drawing

To set or change the default drawing settings:

- 1 Select **Tools > Options > Drafting > Drawings** to display the **Drawings** (see page 9) page in the **Options** dialog.
- 2 Specify the settings for the drawing.

Drawings options

Use the **Drawings** page of the **Options** dialog to specify settings for drawings.



Template — You can use a drawing from another model as a template. The model must have the word 'template' in its name.

To use a template:

- 1 In the **Layout** area, select the **Template** option.
- 2 Select a **Model** from the list.

The **Drawing** list displays drawings contained in the selected model.

- 3 Select a **Drawing** from the list.
- 4 When you create a new drawing, it takes its size and initial contents from the selected template drawing. For example, the template drawing normally contains a border frame and perhaps a company logo.

Size — The default dimensions of the drawing are displayed. To edit the dimensions of the drawing, select the option to the left and edit the values. You can also select a standard paper size from the list.

Save thumbnails — Select this option to store the thumbnails from the drawing in the model.

Graduated background — Select this option to apply any graduated background to new drawing windows. Deselect it to use the mid-colour of the graduated background in new drawing windows.

Set the graduated background colours using the **View Edit** dialog. For further details, see View > Layout.

OK — Confirm the settings and close the dialog.

Opening existing drawings

To open an existing drawing from Modelling mode:

- 1 Click  **Modelling Mode** (*Main toolbar*).

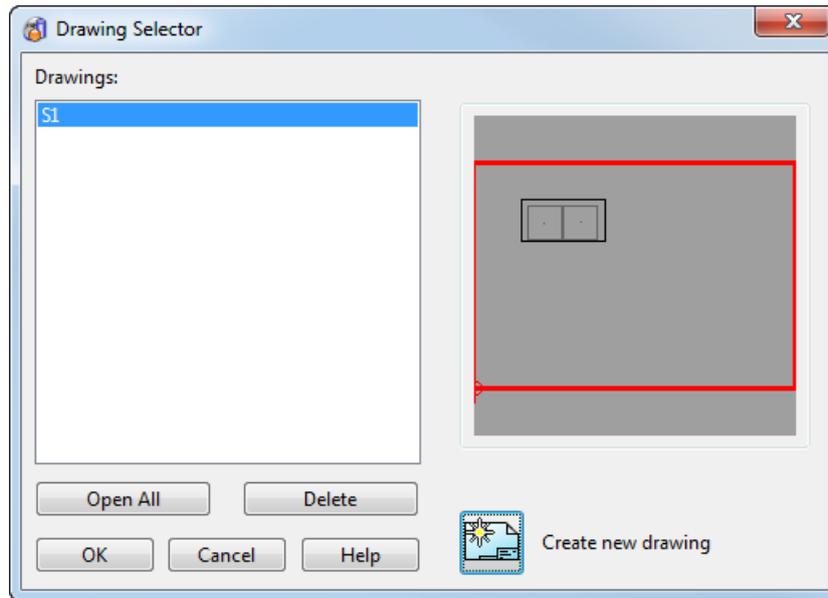


The **Drawing Selector** dialog is displayed.



If a drawing is already open when you enter Drafting mode, the last-accessed drawing is displayed.

- 2 Select the drawing you wish to open from the list of drawings displayed in the **Drawings** section on the dialog. Alternatively click the **Open All** button to select all the drawings listed.

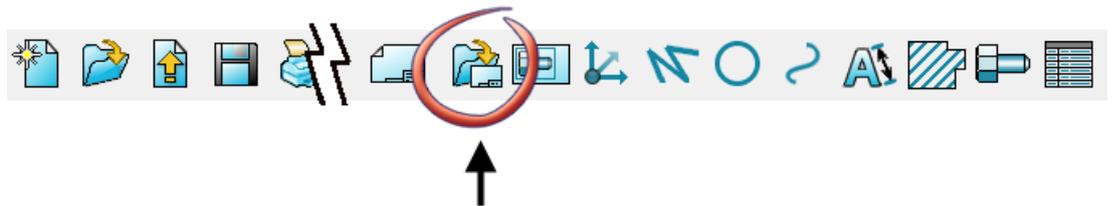


- 3 Click **OK**.

The selected drawings are displayed in the graphics window.

To open an existing drawing from Drafting mode:

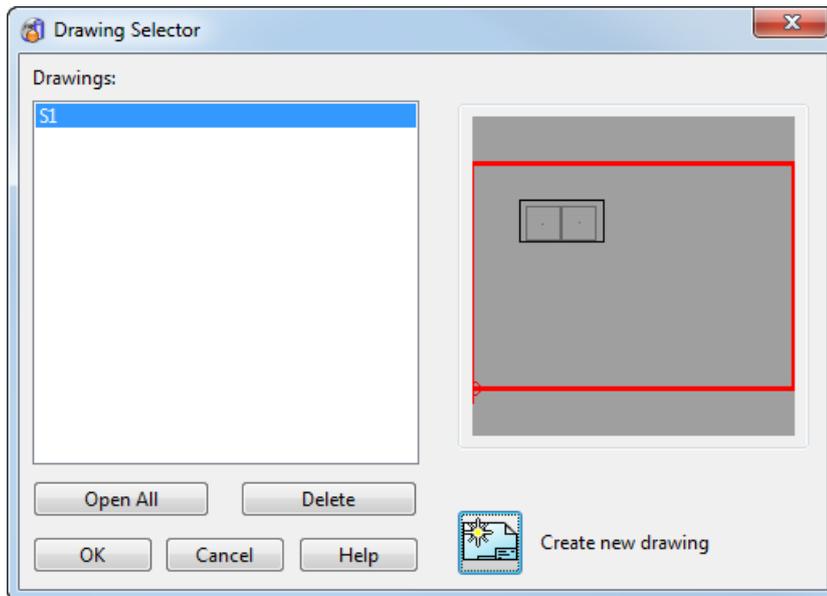
- 1 Click **Open Drawing**  on the **Drafting** toolbar.



The **Drawing Selector** dialog is displayed.

Viewing thumbnails

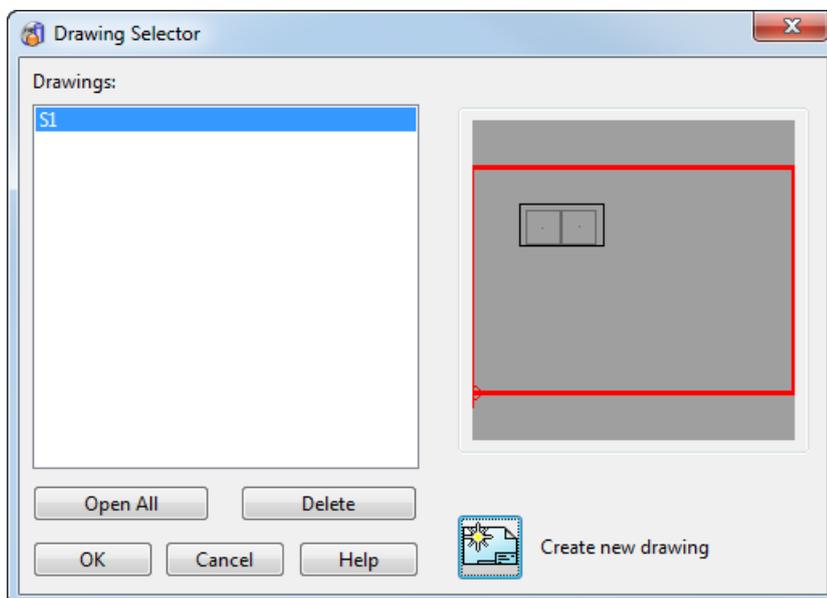
You can view thumbnail images of existing drawings in the **Drawing Selector** dialog. Drawing thumbnails are stored with the model.



- 1 Select a single drawing from the list of drawings displayed on the dialog.
A thumbnail image of the drawing is displayed on the right.
- 2 Click **OK** to open the drawing.

Drawing Selector dialog

Use this dialog to open existing drawings, or click  to create a new drawing. You can view thumbnail images of existing drawings when you select a single drawing from the list.



Drawings — Click a drawing in the list to select it. Select a single drawing to display a thumbnail image in the preview window on the dialog.

Open All — Click to select all the drawings listed.

Delete — Select a single drawing or multiple drawings from the **Drawings** list and click **Delete** to delete the selection. You can also

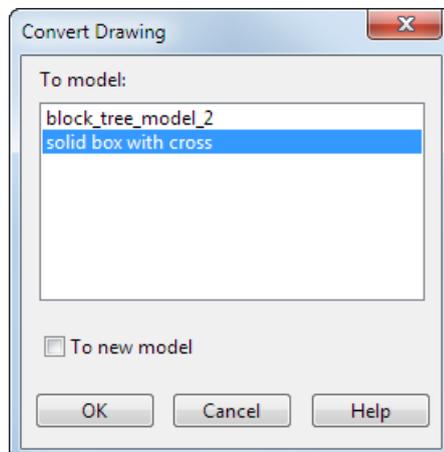
use **Delete**  on the **Main** toolbar to delete the selection.

Create new drawing  — Click to create a new drawing.

OK — Click to open the selected drawing.

Convert Drawing dialog

Use this dialog to export a 2D wireframe representation of a drawing to a view or another drawing. The drawing can be converted to an open model or to a new model.



To model — If there is one model open, this model is displayed in the dialog and is selected. The drawing is copied to this model by default. Select the **To new model** option to copy the drawing to a new model. If there is more than one model open, these are listed in the dialog. The first in the list is selected by default, but you can select any model from the list.

To new model — Select this option to copy the drawing to a new model. This option is selected automatically if there are no open models listed in the dialog.

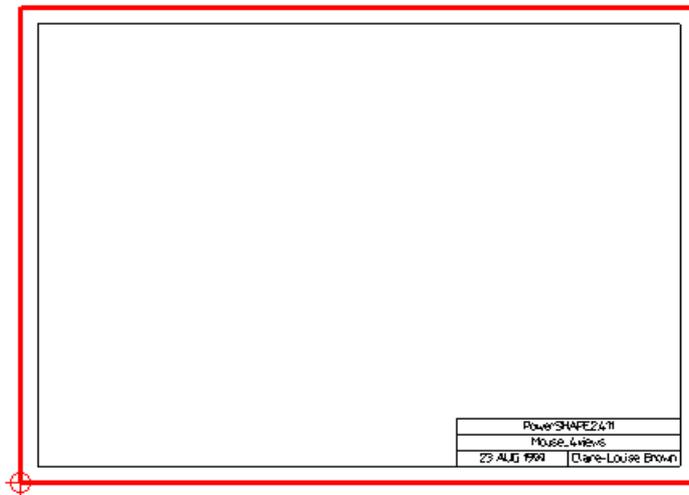
OK — Create a drawing converted to 2D geometry representation in the specified model.

Creating a title block on a drawing

A title block is a block of text containing:

- PowerSHAPE and its version number
- Name of the sheet
- Date
- Name of the author.

When you create a title block, a border is added around the drawing and the text.



To add a title block:

- 1 Ensure the drawing window is open (see page 41) and is the current window.
- 2 Select **Tools > Create Titleblock** from the menu.

You can edit the text (see page 126).

You can update the title block to the current details by creating the title block again.

How do I create objects on a drawing?

You can create objects on the drawing in exactly the same way as you would in a model.



The workspace of the drawing is only 2D; so if you create a 3D object such as a surface on a drawing, its view is constrained to the XY plane. Refer to the chapters on individual objects for further information on how to create them.

The origin of the global workspace is in the bottom left corner of the drawing and is marked by a circle with a cross as shown below. You can edit its position using the **Edit Drawing** dialog. See Edit Drawing dialog (see page 17) for further details on how to use this dialog.



Editing a drawing

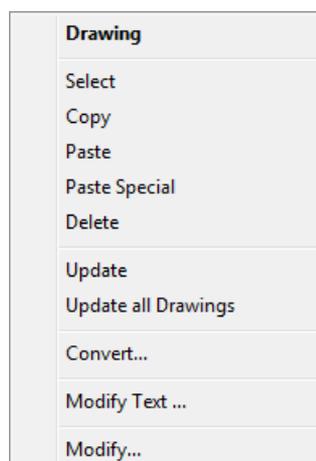
Use one of the following methods to edit a drawing:

- Double-click on the drawing to display the Edit Drawing dialog (see page 17).
- Right-click in an empty space in the drawing to display the context menu. Click **Select** from the context menu to display the Edit Drawing dialog (see page 17).



If you try to delete model geometry in a drawing view, the geometry is blanked from the view rather than being deleted from the model.

Edit Drawing context menu



Select — Select the drawing.

Copy — Copy the drawing.

Paste — Paste a drawing that you copied.

Paste Special — Attaches the copied geometry to the cursor and does not paste until you click the position you want.

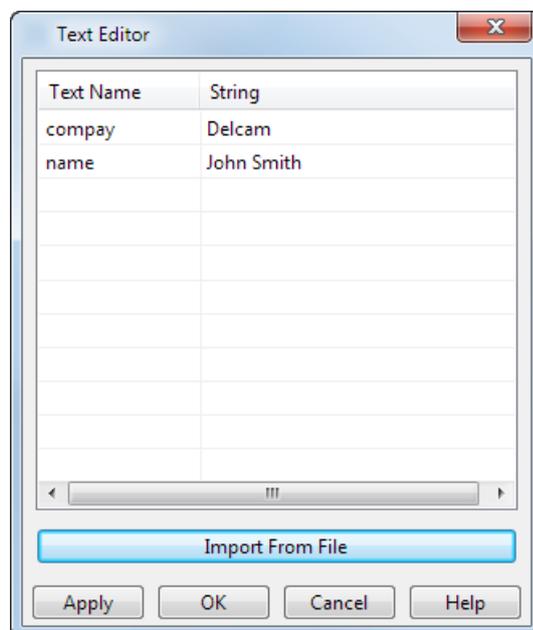
Delete — Delete the drawing.

Convert... — Display the **Convert Drawing** dialog (see page 13). This option exports a 2D wireframe representation of a drawing to a view or another model. The drawing can be converted to an open model or to a new model. Using this option is the preferred alternative to converting views on drawings within a model. Specify **Convert** settings on the **Drafting > Views > General** page (see page 83) of the **Options** dialog.

Update — Update the out-of-date views on the drawing.

Update all Drawings — Update all out-of-date views in the model.

Modify Text — Select this option to display the **Text Editor** dialog. This contains the text names and strings included on the drawing.



Use this dialog to change the strings on the drawing in one of the following ways:

- Enter the text into the **String** column of the dialog. Click **Apply** to make the substitution.
- Use the **Import from file** button to import a **.csv** file that contains substitution text strings. The format of the file is *Text Name,String* as follows:

Name_1,Substitution_1

Name_2,Substitution_2

Name_n,Substitution_n

Modify — Display the **Edit Drawing** dialog (see page 17). You can display the Edit Drawing dialog by selecting **Edit > Modify**.



Closing a drawing does not delete it from the model.

Edit Drawing dialog

Use this dialog to edit drawings.

Name — Enter the name of the drawing. Change the name of the drawing to something more meaningful. To use a drawing as a template, the name of its model must contain the word **template**.

Sheet Number — Use this option to differentiate between two drawings with the same **Name**. Drawings have a unique drawing name in the format *Name (Sheet Number)*; for example, *Fred (1)..... Fred (n)*. If the drawing sheet numbers are not consecutive after you change a sheet number, renumber the sheets automatically by selecting **Yes** when asked if you wish to renumber.

Description — Enter a description of the drawing and its contents.

Update Views — Click this button to update the contents of all the hidden and section views in the drawing.

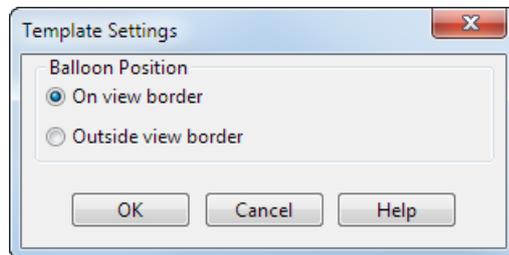
Flatten Geometry — Click this button to flatten 3D wireframe objects onto the drawing (the Z coordinate of the objects becomes zero).

Template — You can use a drawing from another model as a template.



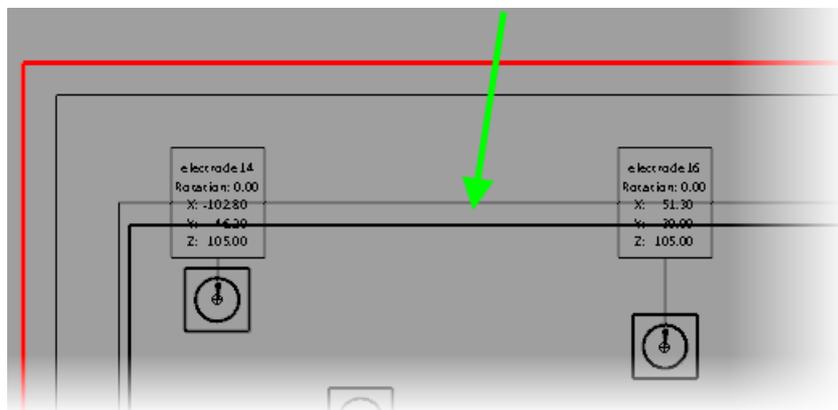
*This button is available only when you are editing a drawing in a template model. The model must have the word **template** in its name, to use a drawing as a template.*

- 1 Click the **Template Settings** button to display the **Template Settings** dialog.

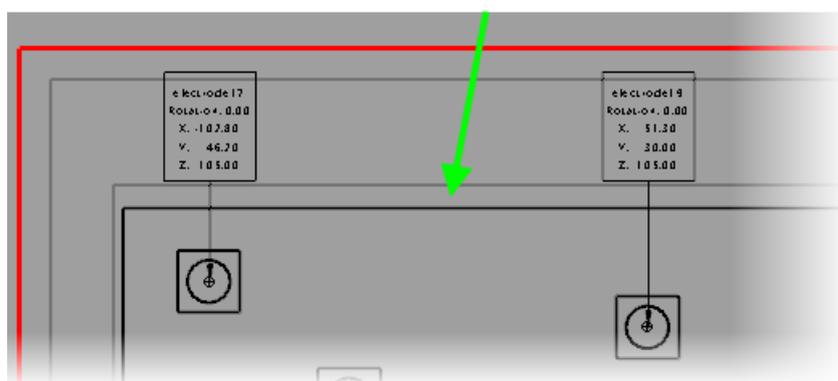


- 2 Use this dialog to specify where balloons are positioned. Balloons can be positioned in the following places:

- **On view border** (default). The **view border** is indicated by the green arrow on the drawings included below.



- **Outside view border**



The option you choose affects the position of the balloons in all drawings that you create using the drawing template.



When you are creating model drawings, you select the template to be used from the **Model** list in the **Edit Drawing** dialog.

Size — The default dimensions of the drawing. To edit the size of the drawing, select the **Size** option. You can edit the **Width** and **Height** of the drawing or select a standard paper size from the list.

Origin — Edit the origin of the global workspace of the drawing. By default, the origin is in the bottom-left corner of the drawing and is marked by a circle with a cross:



To specify the coordinates:

- Enter **X**, **Y**, **Z** coordinates.
-  **Position** — Click this button to display the **Position** dialog to use position-entry tools

OK — Make the changes to the drawing and close the dialog.

Cancel — Discard any changes and close dialog.

Drawing templates

Drawing templates are used to set up standard drawing layouts, including title blocks and views.

Setting up drawing templates (see page 19)

Using drawing templates (see page 20)

Setting up component drawing templates (see page 21)

String variables for drawing templates (see page 22)

Setting up drawing templates

Set drawing templates in the following ways:

- Using the default drawing creation method from the Drawing Options dialog.
- From the Edit Drawing dialog (see page 17).

Drawing templates can contain views. If the drawing does not already contain views, views are copied from the template drawing when the template is set.

The created views:

- display the geometry currently displayed in the model window.
- have the same attributes as the views in the template model. For example, shaded or hidden views.

The view settings can be used to control the way in which the *scale* is set in the created views. For further details, see Edit View dialog (see page 104).

You can set views to hidden only if they contain surfaces or solids.



To set up a hidden view in a drawing template you must create solid or surface geometry in the view before setting the view as hidden.

Any geometry displayed in a template drawing view is ignored when a drawing template is set. The geometry is simply an aid in setting up the view.

Any balloons created from a template drawing point to the relative position of the created drawing. Balloons that point to specific coordinates in a template drawing follow the workplane coordinates of the created drawing. When the active workplane is defined on the model, balloons point to the coordinates determined by that workplane.

Levels in a view are copied from the views set up in the drawing template. We recommend that all the view levels in drawing templates are set to *ON*. For further details, see Edit View dialog (see page 104).



If view levels are set OFF, the geometry on those levels will not be displayed in the created views.

Using drawing templates

To use a template:

- 1 Select the **Template** option on the Edit Drawing dialog. (see page 17)
- 2 Use the **Model** list to select the model you want.

To use a drawing as a template, the model must have the word 'template' in its name.

- 3 Use the **Drawing** list to select a drawing contained in that model.
- 4 Click **OK**.

Setting up component drawing templates



This is specialised functionality for component drawings used in Delcam Toolmaker.

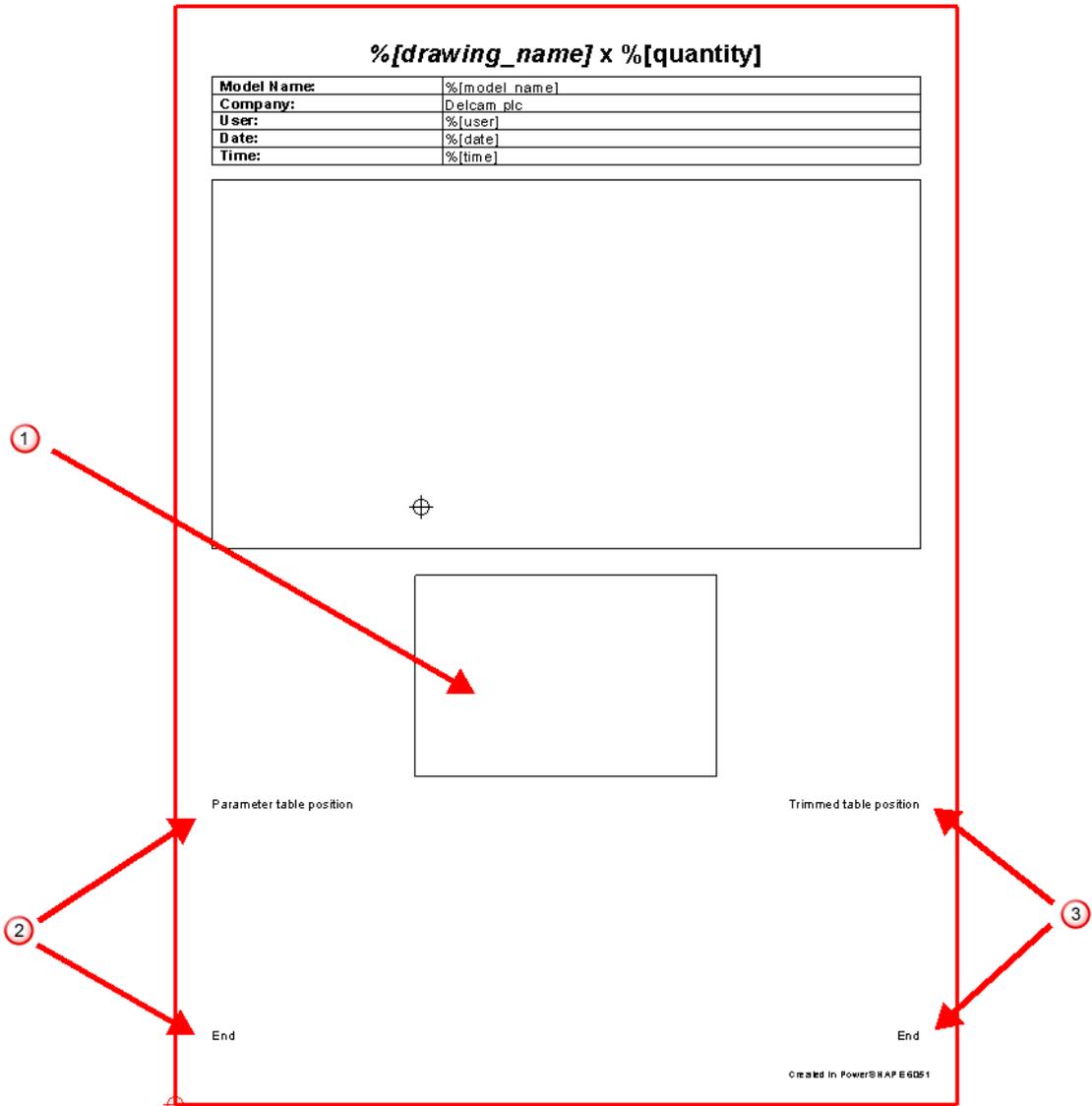
You can make the following changes to the drawing templates:

- If the template has a view called **Bitmap View** then the component drawing is placed at the position of *Bitmap View*, indicated by ① in the image below.

You can create multiple views by adding a suffix to the name of the view. For example *Bitmap View_1*, *Bitmap View_2*. The template above provides for a single view, so in this case, *Bitmap View* is placed on the drawing. *Bitmap View_1* and *Bitmap View_2* are placed on a new drawing.

- If a text exists that starts *Parameter table position* then the parameter table is placed at that position. In the template shown below, this is indicated by ②. The maximum height of the table can be given by having *End* as the final part of that text item.
- For components with a trimmed length parameter, a table is produced of the trimmed lengths if a text exists that starts *Trimmed table position*. In the template above, this is indicated by ③ in the image shown below.

- If a template drawing exists that has the same name as the component, that drawing is used. For example, if a drawing called *Ejector plate* exists in the template model then all ejector plates use that drawing. There can be several related drawings (for example, *Ejector plate_1*, *Ejector plate_2*), in which case the drawing that gives the best view, in terms of scaling, is displayed.



Text substitutions for drawing templates

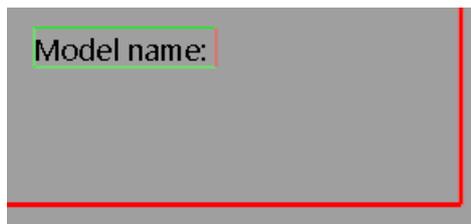
Use the **Text Substitution** dialog to manage, add, and edit drawing template properties. You can

- add general properties to drawing templates (see page 23). Properties such as, `%[author]`, `%[date]` are substituted with the relevant text when the drawing is created.
- add electrode specific properties (see page 24), for example, `%[electrode_name]`, `%[roughing]`, `%[finishing]`.

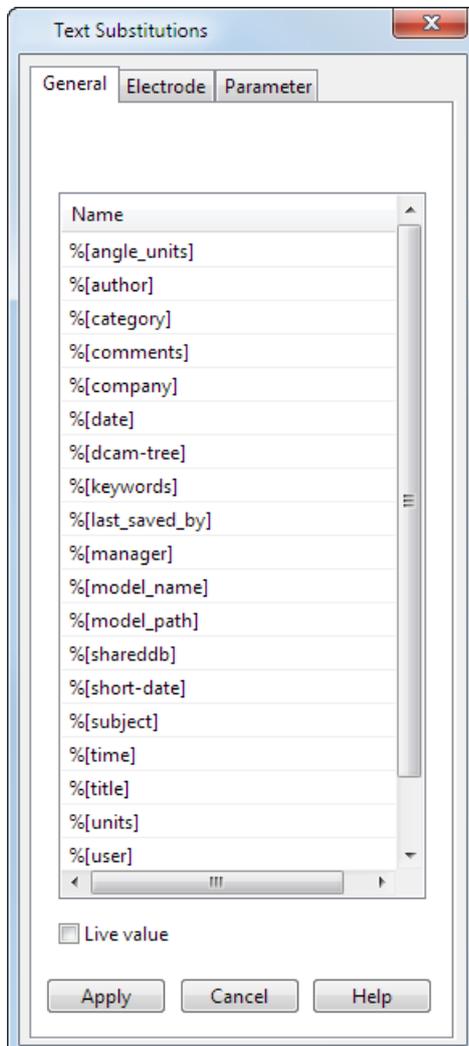
- add fixed parameters (see page 26), these are user-defined parameters.

Adding a General property to a drawing template

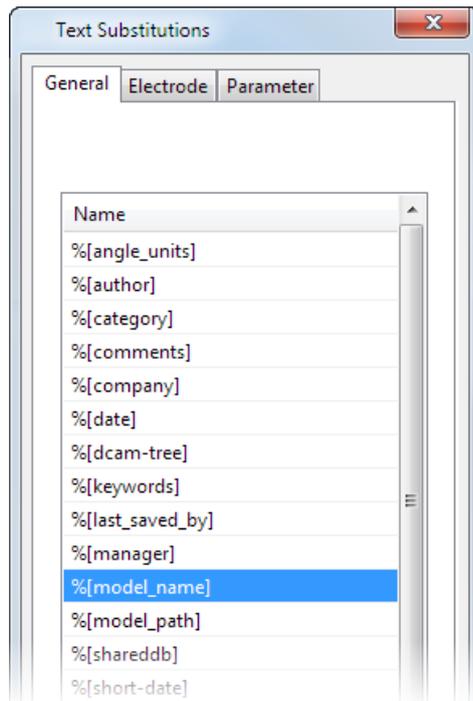
- 1 Click  to display the annotation toolbar.
- 2 Click  and click a position on the drawing where the text will be positioned.
- 3 Enter a label, if required.



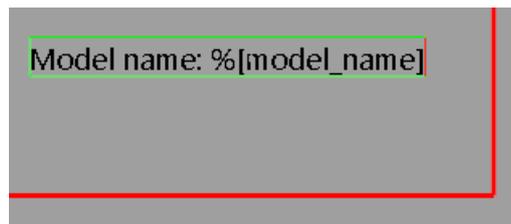
- 4 Click  (*Annotation toolbar*) to display the **Text Substitutions** dialog.



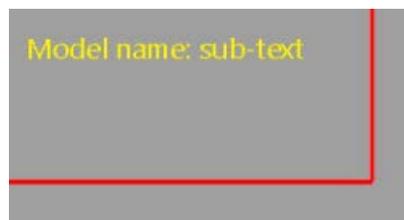
- 5 Select the required substitution string.



- 6 Select **Live value** option to re-evaluate the substitution whenever the text is drawn.
- 7 Click **Apply**.



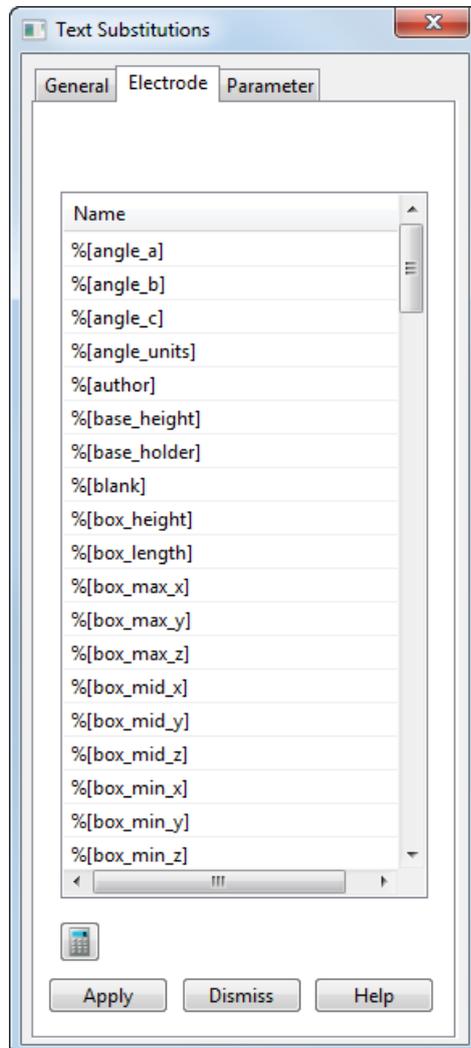
- 8 Click **Cancel** and click the drawing to display the substituted text. In this example, the name of the model was **sub-text**.



Adding an electrode substitution string to a drawing template

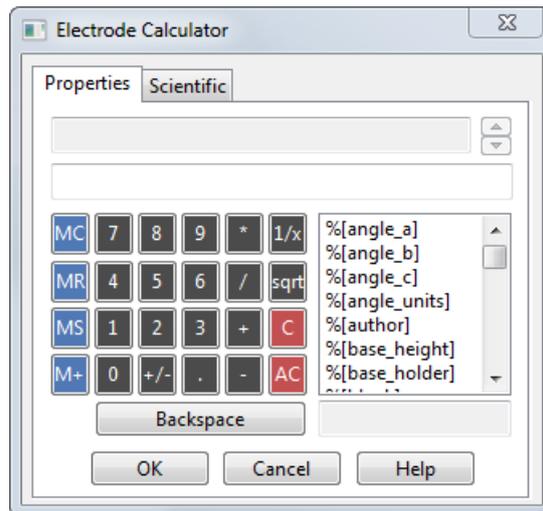
- 1 Click  to display the annotation toolbar.
- 2 Click  and click a position on the drawing where the text will be positioned.

- 3 Enter a label, if required.
- 4 Click  (*Annotation toolbar*) to display the **Text Substitutions** dialog.
- 5 Select the **Electrode** tab.



- 6 Use one of the following methods to enter the required substitution string:
 - Select the required substitution string.

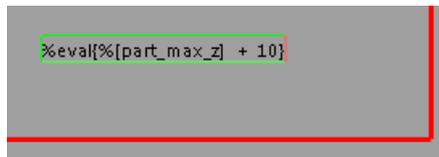
- Click  to enter an expression using the **Electrode Calculator** dialog



Use the options on the dialog to enter an expression. For example, the following expression gives a safe height of 10 above the part:

```
%part_max_z + 10
```

- Click **Apply**.

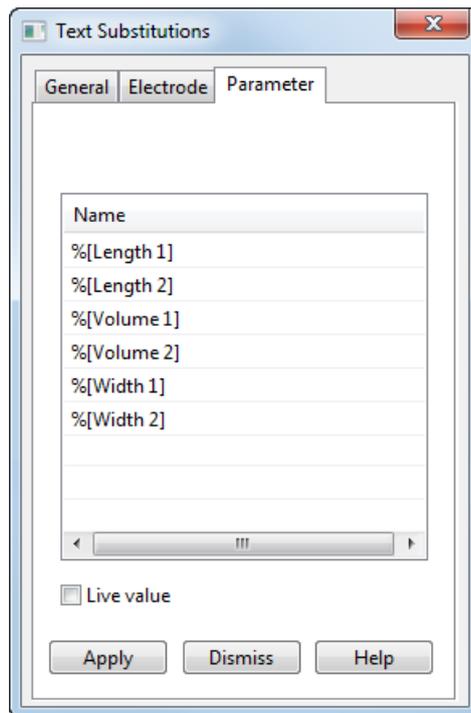


- Click **Dismiss** and click the drawing to display the substituted text

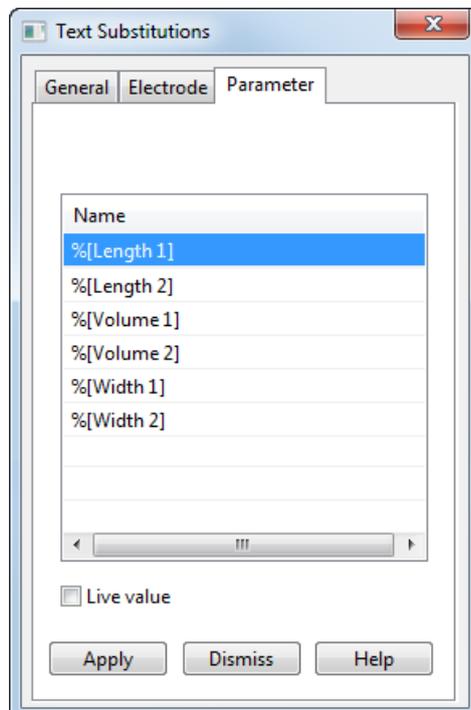
Adding a fixed parameter substitution string to a drawing template

- Click  to display the annotation toolbar.
- Click  and click a position on the drawing where the text will be positioned.
Enter a label, if required.
- Click  (*Annotation toolbar*) to display the **Text Substitutions** dialog.

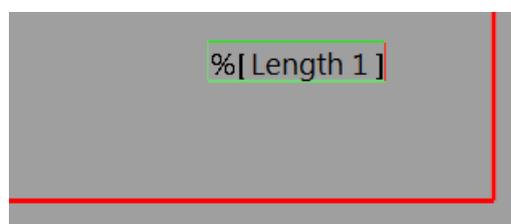
- 4 Select the **Parameter** tab.



- 5 Select the required substitution string.



- 6 Click **Apply**.



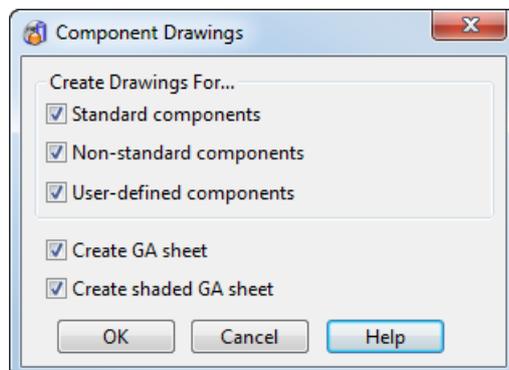
- 7 Click **Dismiss** and click the drawing to display the substituted text.

Creating Component Drawings

You can create component drawings for standard and non-standard components.

To create a Component drawing:

- 1 Select **Drafting** from the **Module** menu.
- 2 Select **Toolmaker** from the **Module** menu.
- 3 Select the model or portion of the model. The drawing that is produced will be based upon the selection that you have made at this point. If you only select part of the assembly, your drawing will be based on the selection. If you do not select a particular part of the assembly, the component drawing will include all the components.
- 4 Select **Component Drawing** from the **Tools** menu. The **Component Drawings** dialog is displayed.



Standard components — Create a component drawing for all standard components. These are components contained in the catalogues that are available.

Non-standard components — Create a component drawing of all non-standard components. These are standard catalog components that you have modified in some way (for example a bolt where you have increased the length or diameter)

User-defined components — Create a component drawing for user-defined components. These are components that you have created from a solid.

Create GA sheet — If *ON*, the GA sheet for your selection is created. This will include an embedded Bill of Materials. You can define the columns of the data that are displayed by modifying the component drawing template. For further details, see Defining the BOM parameters on the GA sheet (see page 31).

If *ON* and the component drawing template contains drawings prefixed with *ga_sheet*, the appropriate drawings will be created.

Create shaded GA sheet — If *ON*, the shaded GA sheet for your selection is created. If *ON* and the component drawing template contains drawings prefixed with *shaded_ga_sheet*, the appropriate drawings will be created.

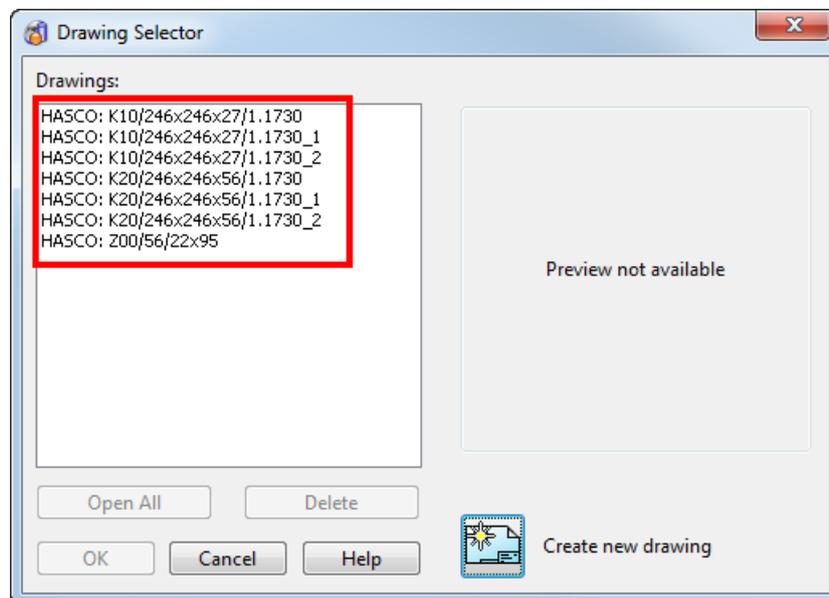
OK — The drawing is created, based on your selection. The appropriate component template is automatically selected for each type of component to allow your component drawing to be closest to 1:1 scale.

An information dialog is displayed that shows the number of component drawings that are created.

Viewing the Component Drawings

To view the individual component drawings:

- 1 Select **Open Drawing** from the **File** menu.
- 2 Select the required component drawing from the list.



Customising the Component Drawings

To change the appearance and contents of the component drawings you need to edit the component drawing template. The component drawing template must be in your *shareddb* before it can be modified.

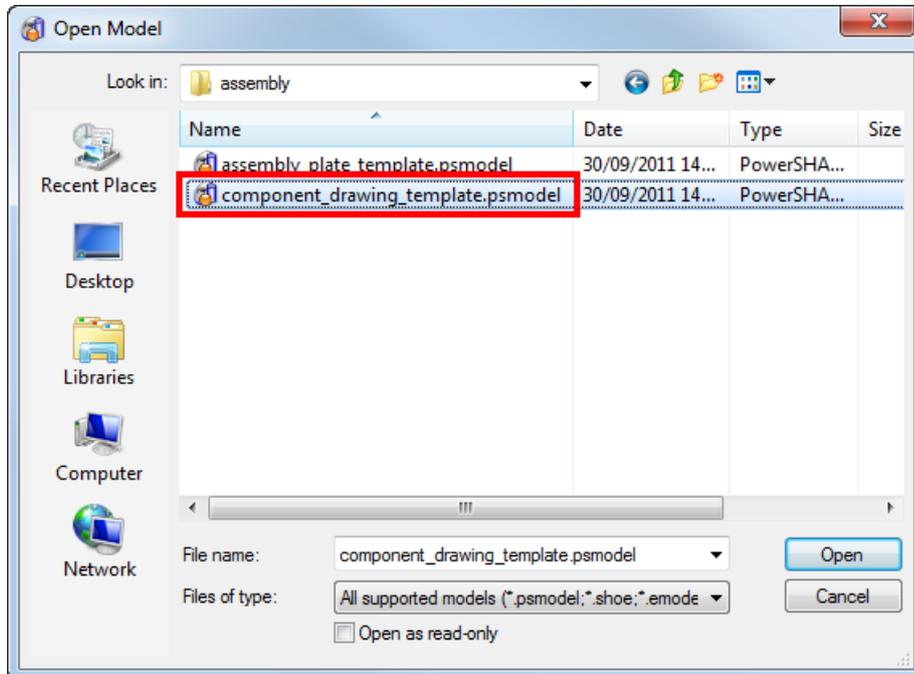
The component drawing template can be added to your *shareddb* by:

- Creating a component drawing with the supplied component template.

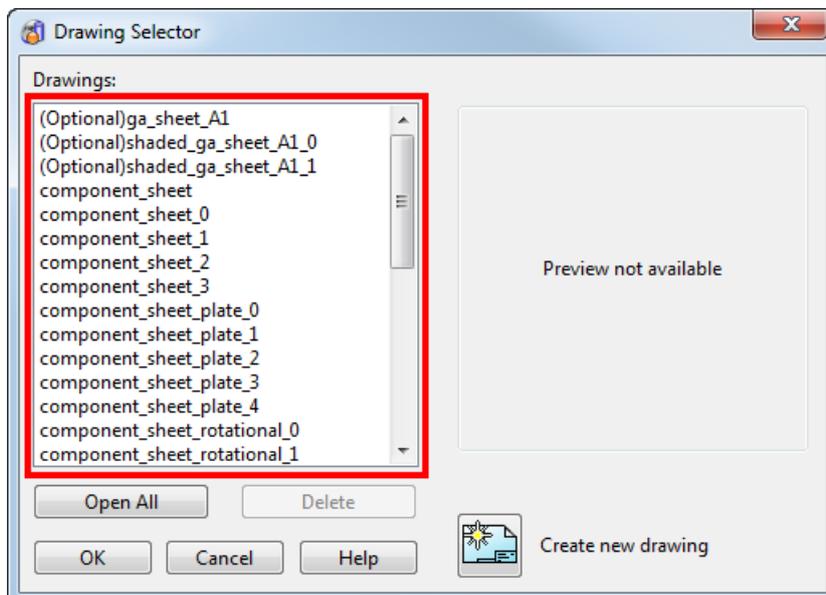
- Importing the template from the *C:\Program Files\Delcam\powershapeXXXX\file\assembly*, where PowerSHAPE is installed on drive C of your computer.

Once the template is in your *shareddb*, you can modify the template as follows:

- 1 Select **Open** from the **File** menu. From the list, select component drawing template file and click **Open**.



- 2 Select **Open Drawing** from the **File** menu. From the list select the template you wish to modify.



The following templates are included. By default, X indicates the size and is in the range 0 to 4 to reflect A0 to A4 sizes.

component_sheet_plate_X - plate components.

component_sheet_rotational_X - rotational components.

component_sheet_userdefined_X - user-defined components.

component_sheet_X - other types of components.

- 3 Use the techniques that are used to change Drawing Templates to customise the views, hole schedules, and table positions in the templates as follows:

- If text exists that starts **Parameter table position** then the parameter table will be placed at that position. The maximum height of the table can be given by having **End** as the final part of that text item.
- For components with a trimmed length parameter, a table will be produced of the trimmed lengths if a text exists that starts **Trimmed table position**.
- If the component drawing template contains drawings prefixed with the following,

ga_sheet

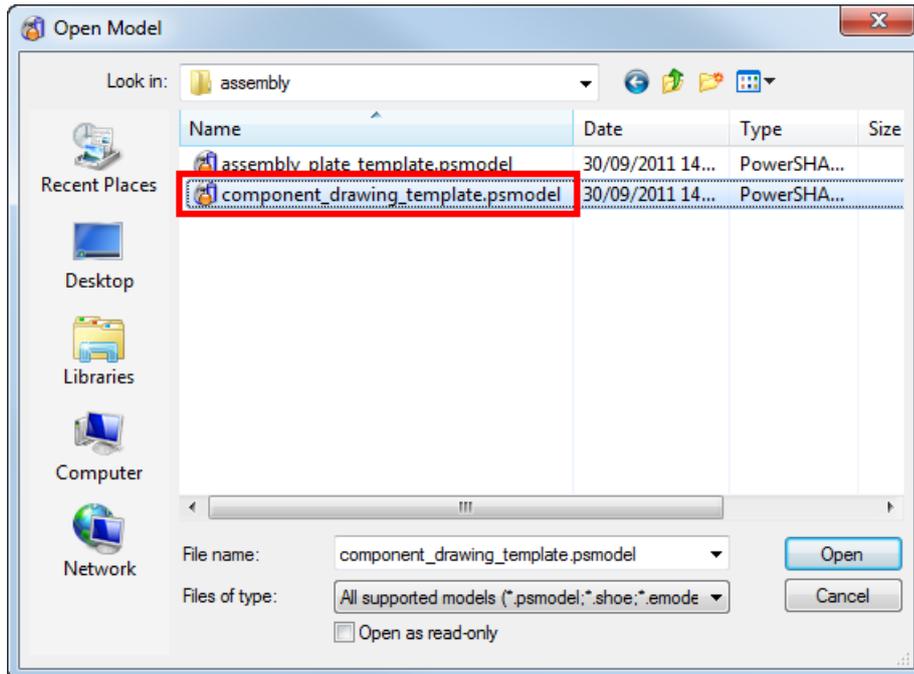
shaded_ga_sheet

the appropriate drawings will be created if the **Create GA Sheet** or **Create shaded GA Sheet** option on the **Component Drawing** dialog is selected.

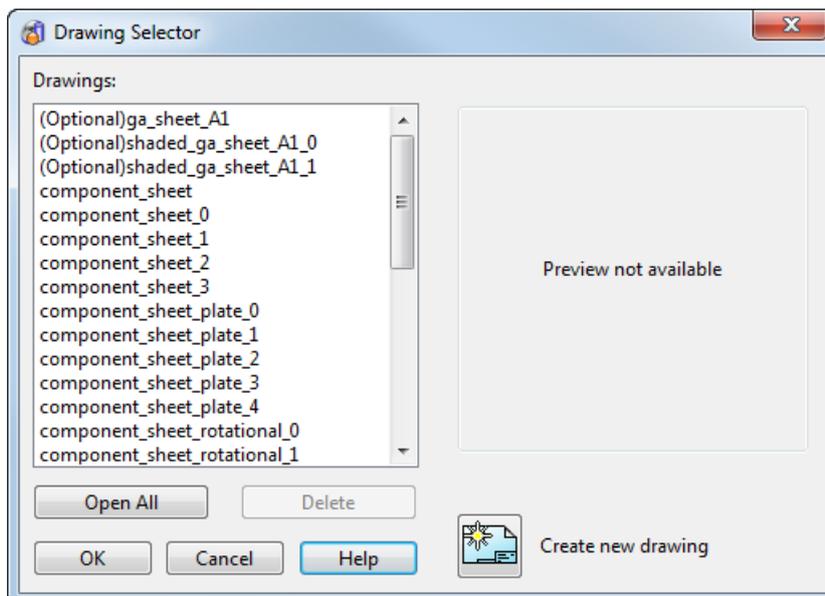
Defining the BOM parameters on the GA sheet

You can define the columns of data to be included in the embedded Bill of Material on the GA sheet by specifying the parameters on the Component Drawing template.

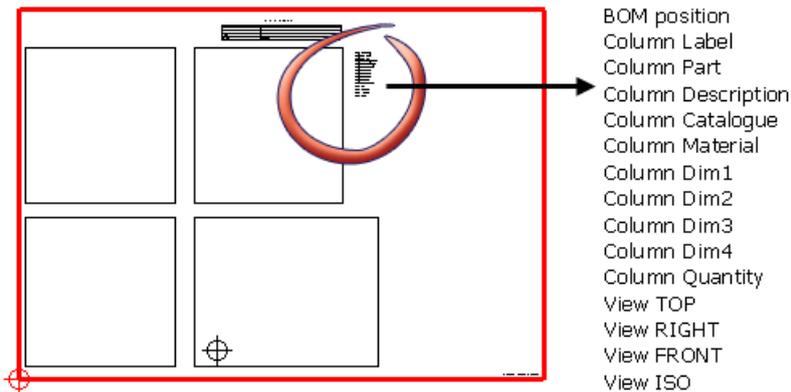
- 1 Select **Open** from the **File** menu. From the list, select the component drawing template file and click **Open**.



- 2 Select **Open Drawing** from the **File** menu. From the list select the template you wish to modify and click **OK**.



3 Select **ga_sheet**.



If text exists that starts **BOM position** then the parameter table will be placed at that position. The maximum height of the table can be given by having **End** as the final part of that text item.

- 4 Select the **BOM position** list by double-clicking or clicking with the right mouse button and selecting **Modify** from the context menu. You can now change the content of the embedded Bill of Materials by changing the individual entries in the list. The example shows a column with the label *Description*. You could modify this text to *Desc* by changing the parameter to *Column Desc*.

Additional BOM parameters for use in Component Drawings

The following BOM parameters can be added to Component Drawings:

Menu	String
Label	Label
Item	Item
Part	Part
Description	Description
Catalogue	Catalogue
Material	Material
Diameter	Dim1
Length	Dim2
Width	Dim3
Height	Dim4

Rough Diameter	RoughDim1
Rough Length	RoughDim2
Rough Width	RoughDim3
Rough Height	RoughDim4
Volume	Volume
Weight	Weight
Quantity	Quantity
Level	Level
Level Name	LevelName
Comments	Comments
Drawing	LabelDrawing
View	LabelView

Adding extra columns to a schedule of holes on Component Drawings

Find the string to add to the template in one of the following ways:

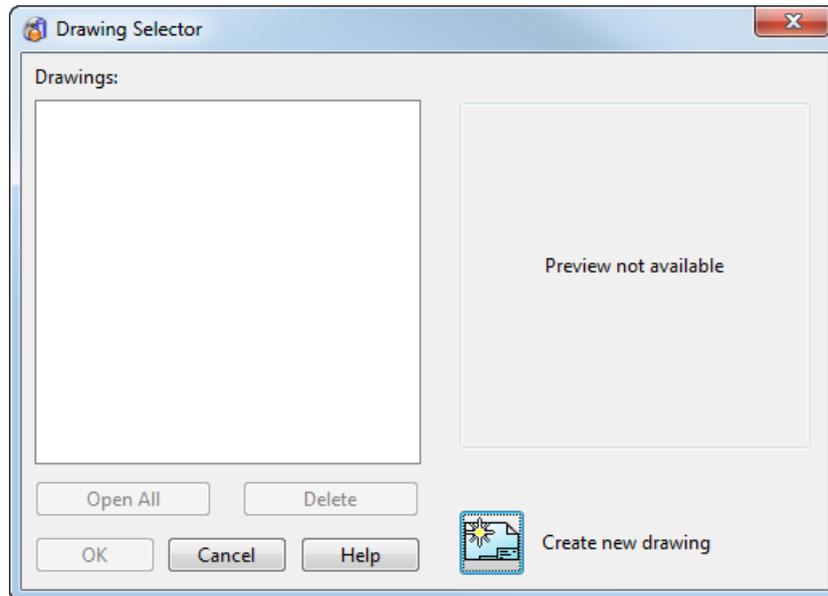
- Identify the string to be added using the lists in Additional schedule parameters for use in Component Drawings (see page 37).
- Record a macro as described below.

Recording a macro to add a column

If you want to add a column to display the **Thread Top Diameter** of holes:

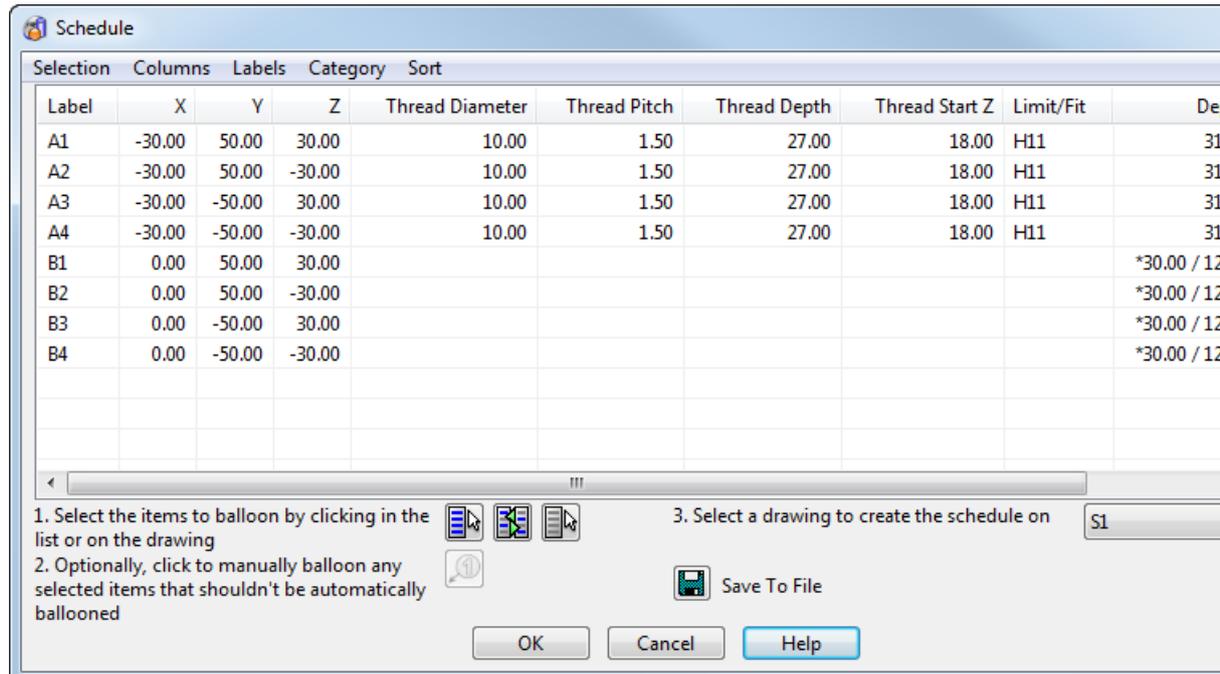
- 1 With a model selected, enter **Drafting** mode .

- 2 Click  **Create new drawing** on the dialog to display a new drawing in the graphics window.



- 3 Create an object on the drawing and make it active.
- 4 Select **Macro > Record** from the **Tools** menu. This displays **Select a file to record to** dialog.
- 5 Enter an appropriate filename for your macro.
- 6 Click **Schedule**  (*Drafting toolbar*), to display the **Schedule** toolbar

- 7 Click **Hole schedule**  (*Schedule toolbar*) to display the **Hole schedule** dialog.



- 8 Click **Column > Thread Details > Thread Top Diameter** to select the option that you require from the sub-menu.
- 9 Click **OK**.
- 10 Select **Macro > Record** from the **Tools** menu to stop recording the macro.
- 11 Open the macro file with a text editor. The commands will look like this:

Create Schedule Schedule

SCHEDCOLUMNS ON THREADTOPDIAMETER

CANCEL

The item that you are interested in is **THREADTOPDIAMETER**

This needs to be added to the **Component Drawing** template.

- 12 Open *component_drawing_template.psmodel* as described in Customising the Component Drawings (see page 29).
- 13 Using the method described in Defining the BOM parameters on the GA sheet (see page 31), open the required drawing templates. Schedules appear in the drawing templates:
- component_sheet_plate_X* - plate components.
- component_sheet_userdefined_X* - user-defined components.

- 14 For each drawing template that is to display the additional column, double-click the Schedule list and add the following into the list:

COLUMN THREADTOPDIAMETER

The next time you create component drawings, you should see the new columns.

Additional schedule parameters for use in Component Drawings

The following schedule parameters can be added to Component Drawings:

Menu	String
Label	Label
X	TopX
Y	TopY
Z	TopZ
Bottom X	BottomX
Bottom Y	BottomY
Bottom Z	BottomZ
Rotation X	RotationX
Rotation Y	RotationY
Direction	Direction
Thread Diameter	ThreadDiameter
Thread Top Diameter	ThreadTopDiameter
Thread Base Diameter	ThreadBaseDiameter
Thread Pitch	ThreadPitch
Through Thread	ThroughThread
Thread Depth	ThreadDepth
Thread Start Z	ThreadStartZ
Thread	ThreadName
Limit/Fit	LimitFit
Tolerance	Tolerance
Max Tol	MaxTol

MinTol	Min Tol
2nd Max Tol	MaxTol2
2nd Min Tol	MinTol2
Through Hole	ThroughHole
Depth	Depth
1st Depth	FirstDepth
2nd Depth	SecondDepth
3rd Depth	ThirdDepth
Diameter	Diameter
FirstDiameter	1st Diameter
SecondDiameter	2nd Diameter
3rd Diameter	ThirdDiameter
Chamfer Size	ChamferSize
FirstChamferSize	1st Chamfer Size
2nd Chamfer Size	SecondChamferSize
3rd Chamfer Size	ThirdChamferSize
Chamfer Angle	ChamferAngle
1st Chamfer Angle	FirstChamferAngle
2nd Chamfer Angle	SecondChamferAngle
3rd Chamfer Angle	ThirdChamferAngle
Hole Category	HoleCategory
Hole Use	HoleUse
Name	Name
Predrilled	Pre-drilled
Level	Level
LevelName	Level Name
Comments	Comments

Notes

Use the following sections to create and edit Notes drawings in PowerSHAPE.

Introduction to Notes drawing (see page 39)

Create a Notes drawing (see page 39)

Update a Notes drawing (see page 40)

Close a Notes drawing (see page 41)

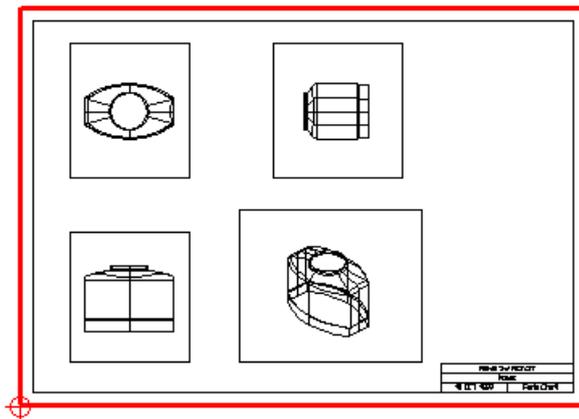
Open a Notes drawing (see page 41)

Edit a Notes drawing (see page 41)

Customising a Notes drawing (see page 42)

What is a Notes drawing?

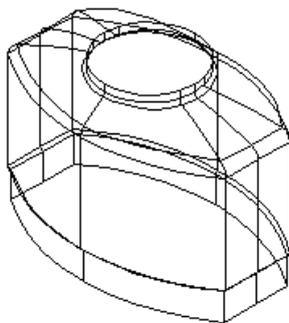
A **Notes** drawing is one with pre-defined views and a title block.



You can annotate this drawing using dimensions and text.

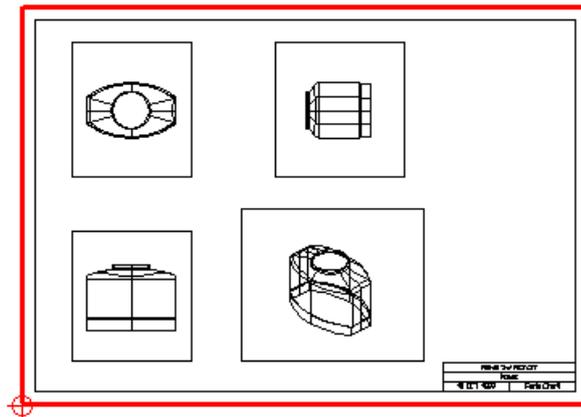
Creating a Notes drawing

To create a **Notes** drawing open the model you want to create a drawing for, as shown in the example below.



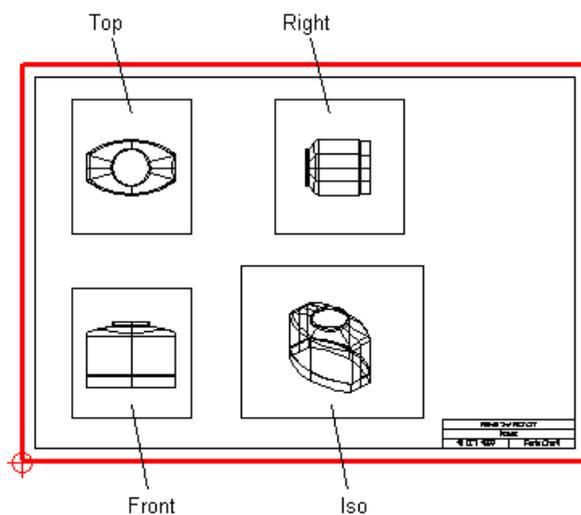
From the **Window** menu, select **Notes**.

If a **Notes** drawing does not exist, this creates a drawing containing 4 views of the objects currently displayed in your model.



If a **Notes** drawing exists, then the existing **Notes** drawing is displayed. For further details, see Opening an existing Notes drawing (see page 41) and Updating an open Notes drawing (see page 40).

The directions of the views in the **Notes** drawing are given below.



The contents of the views are automatically scaled to fit the view boundaries.

Only one **Notes** drawing can be created per model.

You can store the layout of a **Notes** drawing to use in all your future models. For further details, see Customising a Notes drawing (see page 42).

Updating an open Notes drawing

If you have a Notes drawing open in your model when you update the model, the Notes drawing is updated too. To open the Notes drawing in your model :

- 1 Select **Notes** from the **Window** menu.

By default, all new objects created in the model are automatically added to the views. The contents of the view are automatically rescaled to fit the view boundaries.

- 2 You can change settings for any one of the views by turning off the **Add Items** and **Auto Rescale** options on the **Edit View** dialog. For further details, see Editing a view (see page 100).

Closing a Notes drawing

You can close the **Notes** drawing by selecting **Close** from the **Window** menu.

The **Notes** drawing and any objects created on it are stored with the model.

To open a stored **Notes** drawing, see Opening an existing Notes drawing (see page 41).

Opening an existing Notes drawing

To open a stored **Notes** drawing, simply select **Notes** from the **Window** menu.

By default,

- all new objects created in the model are automatically added to the views.
- the contents of the view are automatically rescaled to fit the view boundaries.

For any one of the views, you can change the above behaviour by turning off the **Add Items** and **Auto Rescale** options on the **Edit View** dialog. For further details, see Editing a view (see page 100).

Editing a Notes drawing

Notes drawings can be edited like a drawing. For further details, see Editing a drawing (see page 15).

The views on the **Notes** drawing are edited like other drawing views. For further details, see Editing a view (see page 100).

If you change the name of the **Notes** drawing, then a new **Notes** drawing is created next time you select **Notes** from the **Window** menu.

Customising a Notes drawing

You can modify the layout of a **Notes** drawing to your requirements and store this to use in all your future models.

To create a customised Notes drawing:

- 1 Create a **Notes** drawing in the template model, NEW_MODEL_MASTER.
- 2 Edit the **Notes** drawing to your requirements.

You can add objects to the **Notes** drawing in the template model.

When you create a **Notes** drawing in future models, the **Notes** drawing from the template model will be used.

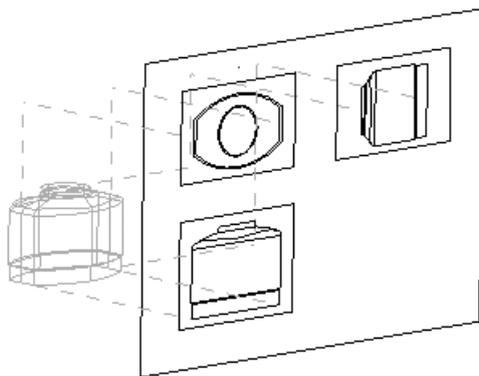
For further details, see Using drawing templates (see page 20).

Views

Different views of a model are created by looking at a model from a different direction. You can have many views on a drawing, each showing a different projection of the model.

You can also edit a model through a view on a drawing.

The drawing below shows views of different projections of a model.



Use the following sections for more details on **Views**:

- View creation (see page 43)
- Changing the settings of a view (see page 82)
- Activating a view (see page 93)
- Creating new objects in a view (see page 93)
- Updating a view (see page 94)
- Graphically editing views (see page 95)
- Editing a view (see page 100)



You can improve the quality and speed of hidden line views if you create a solid of your model where possible before drafting from it. This will also improve the accuracy of dimensioning. For further details, see [Creating a solid](#).

View creation

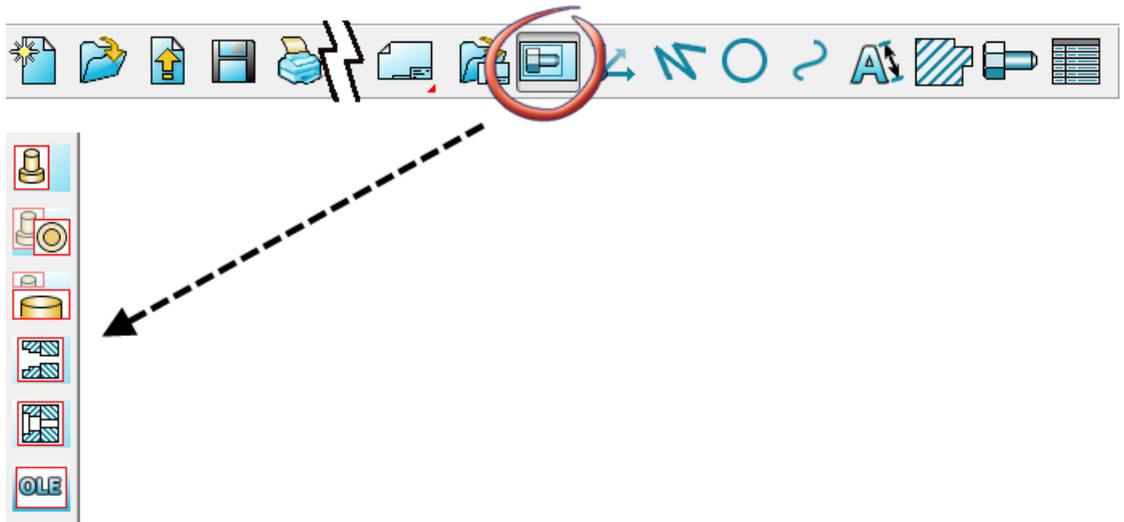
- 1 Click  (Main toolbar) to display the **Drafting** toolbar.



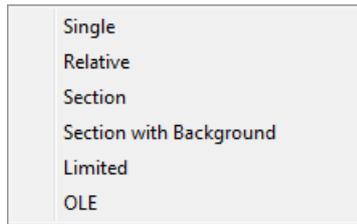
This is the same as hovering over  and selecting  from the flyout. For further details, see [Flyout overview](#).



The **Drawing Selector** dialog is displayed where you select an existing drawing or create a new drawing. The **Drafting** toolbar is also displayed.



This is the same as selecting **Object > View**, to display the **View creation** options.



Each option in the **View creation** menu is described below:



Single view (see page 44)



Relative view (see page 48)



Section view (see page 52)



Section with background view (see page 64)



Limited view (see page 78)



OLE view (see page 80)



You can improve the quality and speed of hidden line views if you create a solid of your model where possible before drafting from it. This will also improve the accuracy of dimensioning. For further details, see [Creating a primitive solid in the Solid Modelling manual](#).

Creating a single view

You can create a single view of either the whole model or of selected objects from the model.

- 1 Click  (Main toolbar) to display the **Drafting** toolbar.



This is the same as hovering over  and selecting  from the flyout. For further details, see [Flyout overview](#).

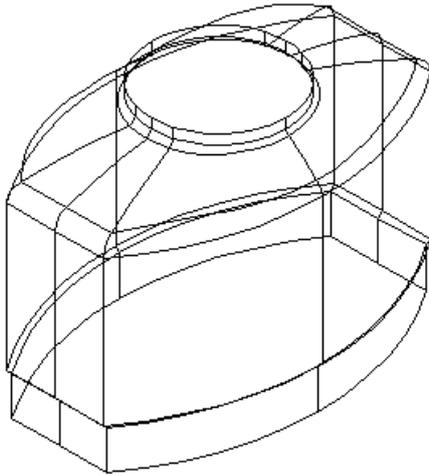


The **Drawing Selector** dialog is displayed where you select an existing drawing or create a new drawing. The **Drafting** toolbar is also displayed.

- 2 Select the required drawing window from the **Windows** menu.

If you do not have a drawing already open, you need to create a drawing or open an existing drawing. For further details, see *Creating a new drawing* (see page 8) and *Opening existing drawings* (see page 10)

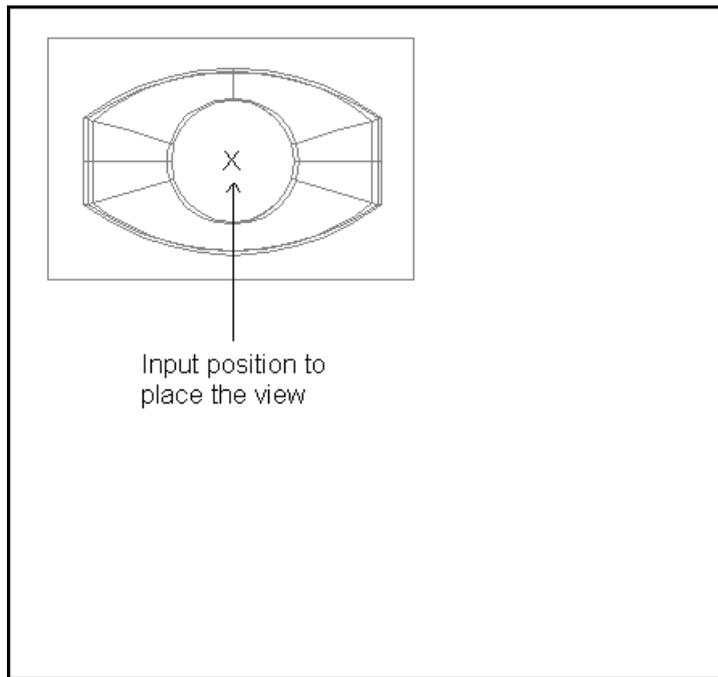
To create a drawing of the model below:



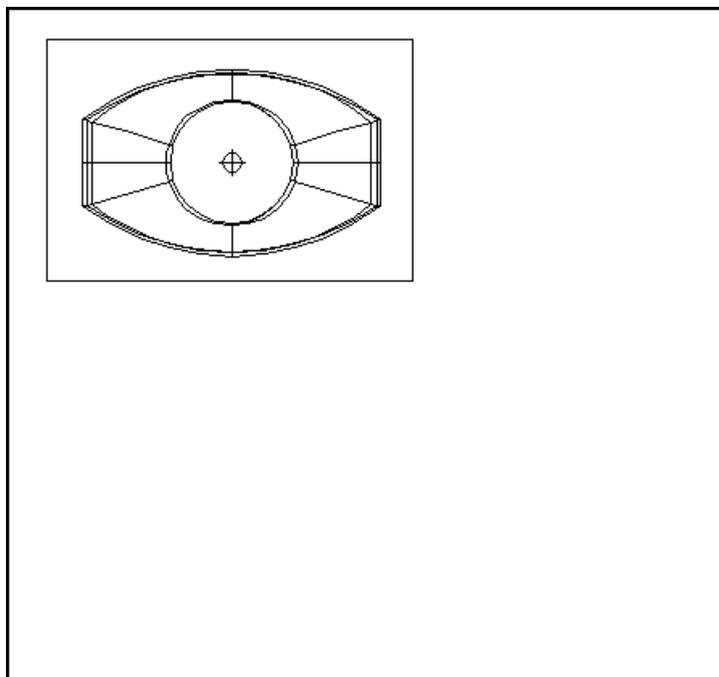
- 1 Click  (*Drafting toolbar*) to display the view creation tools.
- 2 Click  (*View creation menu*).

As you move the cursor around the drawing window, the view is attached to the cursor and is ready to be placed on the drawing.

- 3 Click on the drawing where you want to place the view.



When you input the position, the view is drawn on the drawing.

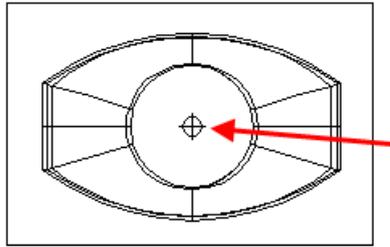


Blanked objects in the model are not drawn in the view.



By default, the view is looking down the top of the world space (that is the -Z axis). You can change this by editing the view. For further details, see [Editing a view](#) (see page 100).

In our view, you can see a mark made of a circle and two lines that cross. This marks the origin of the view. For further details, see Edit View dialog (see page 104).



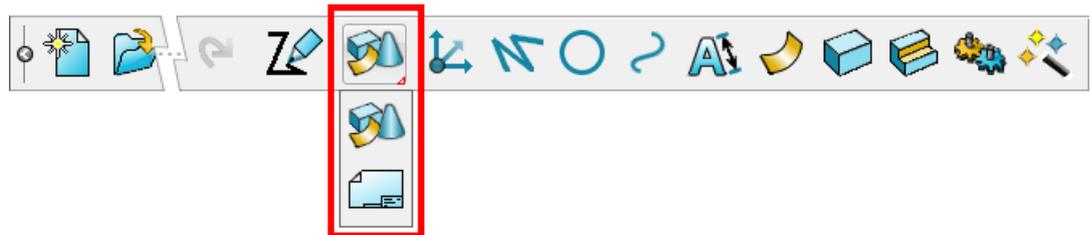
When you create a single view, you can create a relative view to it. The **Relative view** option is automatically selected in the **View creation** menu, ready for you to create relative views. For further details, see Creating a relative view (see page 48).

Creating a single view of selected objects

To create a single view of selected objects:

- 1 Click  (*Main toolbar*) to display the **Drafting** toolbar.

 This is the same as hovering over  and selecting  from the flyout. For further details, see *Flyout overview*.



The **Drawing Selector** dialog is displayed where you select an existing drawing or create a new drawing. The **Drafting** toolbar is also displayed.

- 2 If you do not have a drawing already open, you need to create a drawing or open an existing drawing. For further details, see Creating a new drawing (see page 8) and Opening existing drawings (see page 10)
- 3 From your model window, select the objects you want to display in the view. For further details, see Selecting objects.
- 4 Click  (*Drafting toolbar*) to display the view creation tools.
- 5 Click  (*View creation menu*).
- 6 From the **Window** menu, select the drawing window.

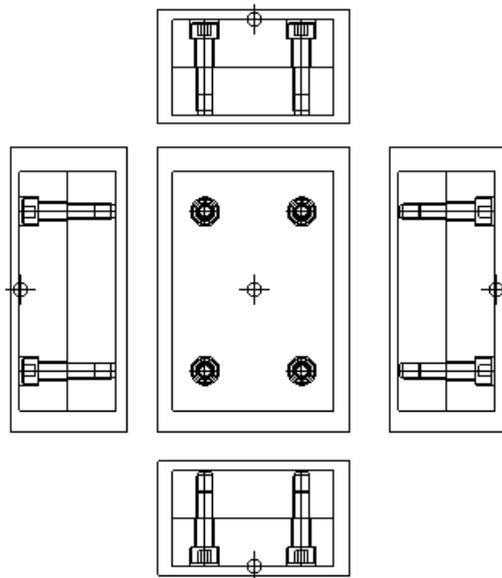
As you move the cursor around the drawing window, the view is attached to the cursor and is ready to be placed on the drawing.

- 7 Click on the drawing where you want to place the view.

Creating a relative view

A relative view is a projection of the model from the side of an existing view.

The image below shows four relative views to the view of the top of the model (using third angle projections).



To create a relative view:

- 1 Click  (Main toolbar) to display the **Drafting** toolbar.

 This is the same as hovering over  and selecting  from the flyout. For further details, see *Flyout overview*.



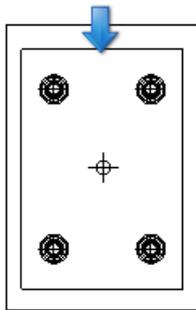
The **Drawing Selector** dialog is displayed where you select an existing drawing or create a new drawing. The **Drafting** toolbar is also displayed.

- 2 Make sure you have a drawing with at least one view on it.

If you do not have a drawing already open, you need to create a drawing or open an existing drawing. For further details, see *Creating a new drawing* (see page 8) and *Opening existing drawings* (see page 10)

If you have no views on your drawing, create a single view. For further details, see *Creating a single view* (see page 44).

- 3 Click  (*View creation menu*).
- 4 Hover the mouse over a view on your drawing. The cursor changes to an arrow to indicate the direction of the new view in relation to the existing view.



As you move the mouse around the view, the arrow changes direction for the other relative views.

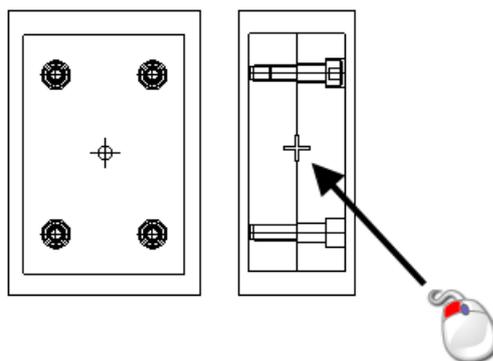
You can create an **Isometric view** by moving the mouse over a corner of the view.

- 5 When the arrow changes to the direction you want, click the mouse. The new relative view appears on the specified side, ready for you to position.

The side the view appears depends on whether the Third angle option (see page 50) on the Views Options dialog > General page (see page 83) is selected.

- 6 Click on the drawing to position and create the new view.

The new view usually is created next to the edge of the view where the projection is taken.

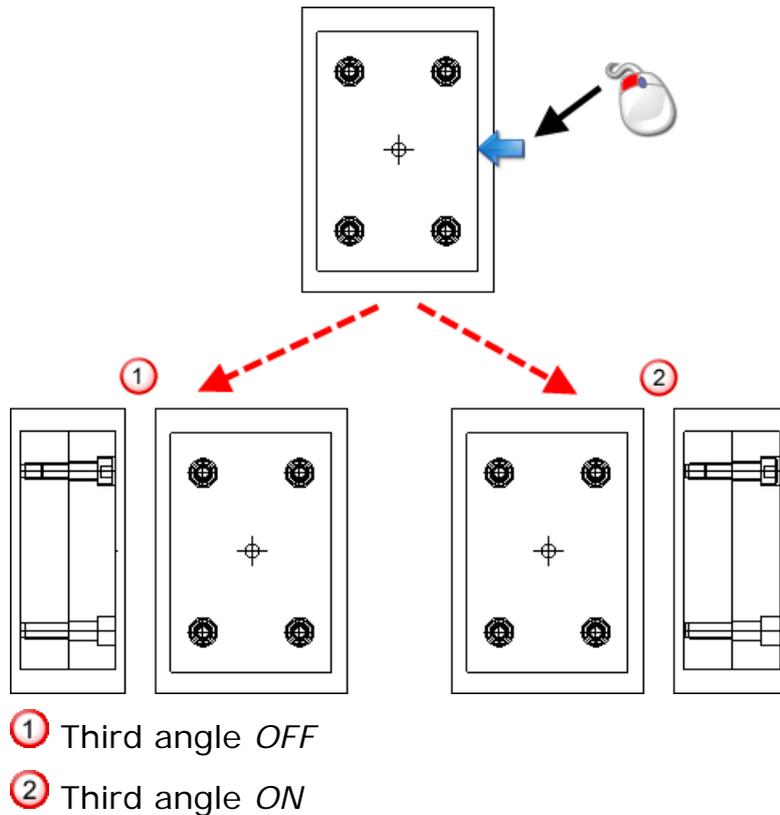


Third angle option

The side the view appears if the **Third angle** option on the Tools > Options > Drafting > Views > General (see page 83) page is selected.

If **Third angle** is *OFF*, projections are in the first angle. The default setting for **Third angle** is *ON*.

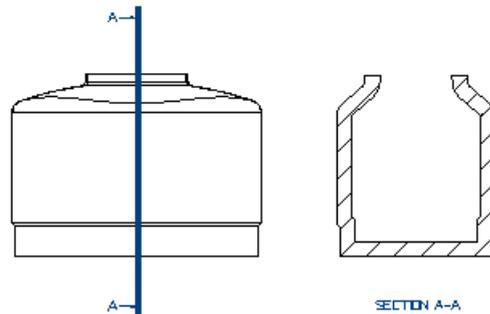
If **Third angle** is *ON*, the relative view appears on the side nearest to where you clicked the mouse. If **Third angle** is *OFF*, it appears on the opposite side.



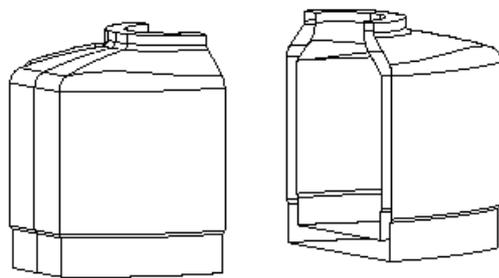
Section views

You can create a section view to show a cross section view of the model. This option does not show background detail. For further details, see Creating a section view with background detail (see page 64).

Specify the settings for section views, and annotation displayed on section views, on the Tools > Options > Drafting > Views > Section (see page 85) page. In the diagram below, the view on the right is a section view of that on the left.



The ink bottle (on the left) has been cut along the vertical line to give two pieces, one on either side of the line.



The edges of the pieces along the cut are used to define the sectional view. Arrows are added to the line to define the viewing direction.

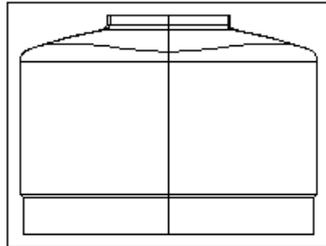
A line, which cuts a part to define a section view, is called a section line. For further details, see Creating a section line using the Section Line toolbar (see page 56).

You can have more than one sectional view on a drawing. To identify which section line belongs to a sectional view, a letter is placed at the top and bottom of the line and the sectional view has underneath it the word SECTION followed by the letter.

A warning cross is displayed across section and hidden views when they need to be updated because there has been a change to the model. The drawing of crosses is on by default and can be turned off by typing the following in the command window.

view_options mark_updated off

To create a section view of the following view on a drawing:



The following section view creation methods are available.

- Creating a section view (see page 52).
- Creating a section view with background detail. (see page 64)
- Creating Angular and Stepped section views. (see page 69)
- Creating Scrap Sections (see page 73)

Creating a section view

- 1 Click  (Main toolbar) to display the **Drafting** toolbar.

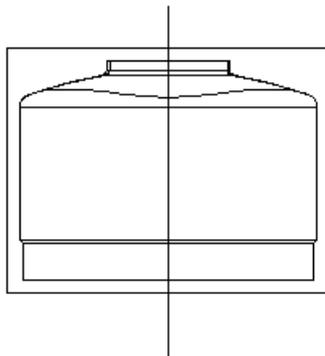
 This is the same as hovering over  and selecting  from the flyout. For further details, see *Flyout overview*.



The **Drawing Selector** dialog is displayed where you select an existing drawing or create a new drawing. The **Drafting** toolbar is also displayed.

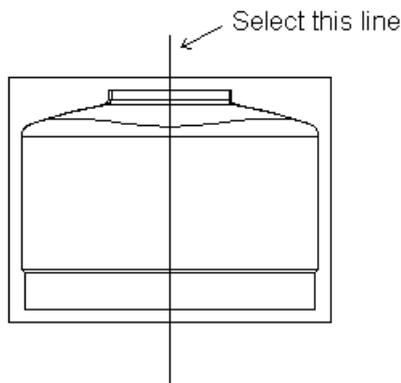
- 2 Make sure you have a drawing with at least one view on it.
If you do not have a drawing already open, you need to create a drawing or open an existing drawing. For further details, see *Creating a new drawing* (see page 8) and *Opening existing drawings* (see page 10)
If you have no views on your drawing, create a single view. For further details, see *Creating a single view* (see page 44).
- 3 Use one of the following methods to create a line through the view where you want the section:

- Create a section line automatically by using the Section Line toolbar (see page 56). For further details, see Creating a section line using the Section Line toolbar (see page 56).
- Draw a single line through the view. For further details, see Creating a line in the Wireframe modelling manual, and continue to follow the steps described below.
- Create a composite curve from a series of orthogonal lines to define a Stepped section (see page 69). For further details, see Selecting objects in the Modelling Concepts manual, and continue to follow the steps described below.



The line or composite curve can be shorter than the view it lies in. This creates a section view of the portion of the view where the line or composite curve lies.

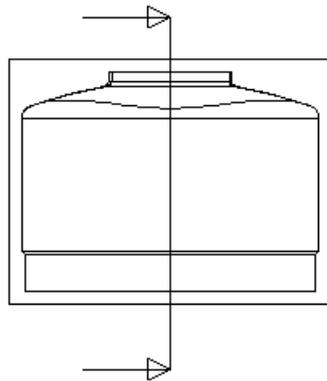
- 4 Select the line or composite curve.



- 5 Click **View**  (*View creation menu*).

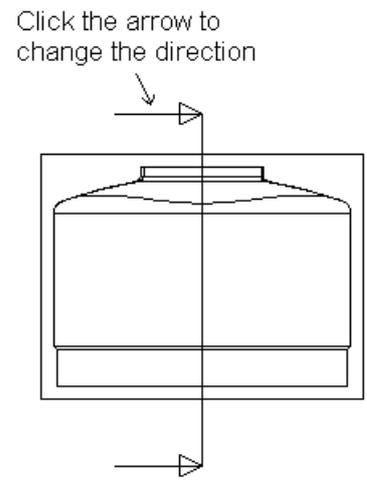
- 6 Click **Section View with Background**  (*View creation menu*).

Arrows are drawn on the line to define the viewing direction.

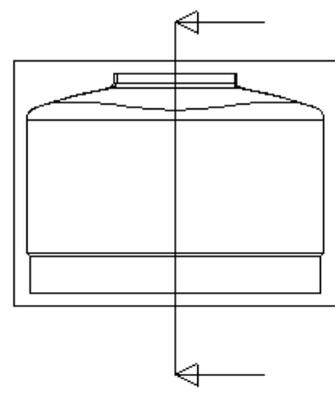


The outline of the new view moves with the cursor.

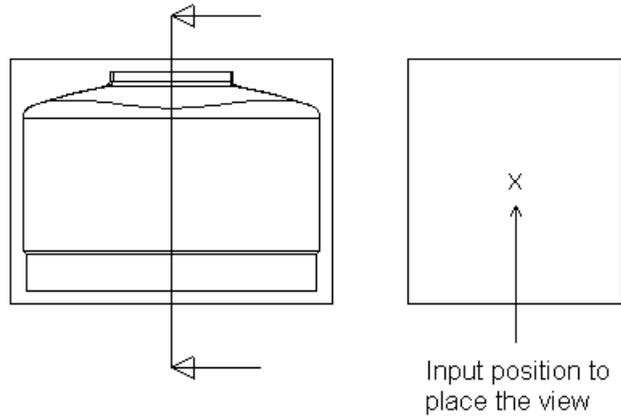
- 7 Click one of the arrows if you wish to change the viewing direction.



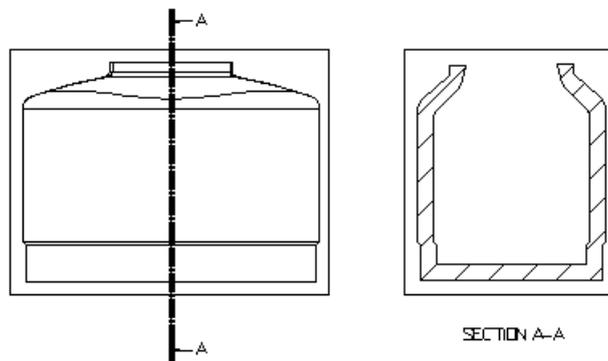
The arrows change direction to indicate that the viewing direction has changed.



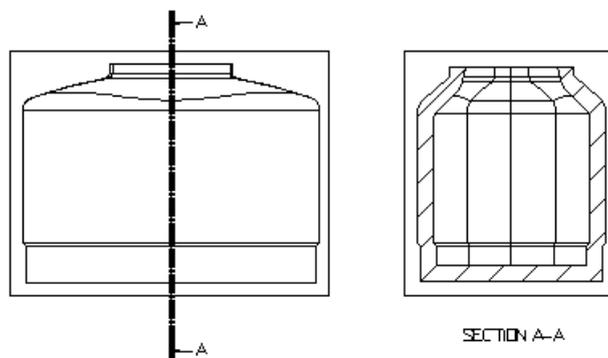
- 8 Click on the drawing to place the view. Alternatively, you can select a different drawing to place the section view. For further details, see Positioning section views on a different drawing (see page 69).



The section view is drawn.



If you selected the section with background view, the background detail is drawn too. For further details, see Creating a section view with background detail. (see page 64)



If you select either of the **Hide** options from the **View** context menu, or **Hidden** on the Edit View dialog (see page 104), background detail is hidden in section views. For further details, see Editing a view (see page 100).

Creating a section line using the Section Line toolbar

Use the **Section Line** toolbar for creating section lines through features and points in drawing views. When you accept the section line, a section view is created ready for you to put in position on your drawing.

Use the following sections to explore how to create section lines for section views.

- Section Line toolbar (see page 56)
- Creating a section line through features (see page 57)
- Creating a section line through points (see page 58)
- Creating a stepped section line using the Section Line toolbar (see page 59)
- Editing a section line. (see page 61)



It is still possible to create a section line manually, by drawing a single line through the view, or by creating a composite curve from a series of orthogonal lines. For further details, see [Creating a section view](#) (see page 52).

Section Line toolbar

- 1 Click **Object > View > Section**.
- 2 Click **OK** on the **Information** dialog to display the **Section Line** toolbar.



- 3 Select the **View** to section using the drop-down list on the **Section Line** toolbar.
- 4 Use **Set feature recognition**  to choose the mode for creating a section line (see page 56).

This option has two modes:



for creating a section line through features (see page 56)

In feature mode, solid features are highlighted as you move the cursor over them in the view. Select a feature to create a section line through the centre of the feature. If there is no feature under the cursor, the section line is created through the point.



for creating a section line through points (see page 56)

In point mode, features are not recognised and section lines are created through the points entered.



You can combine features and points in your section lines.

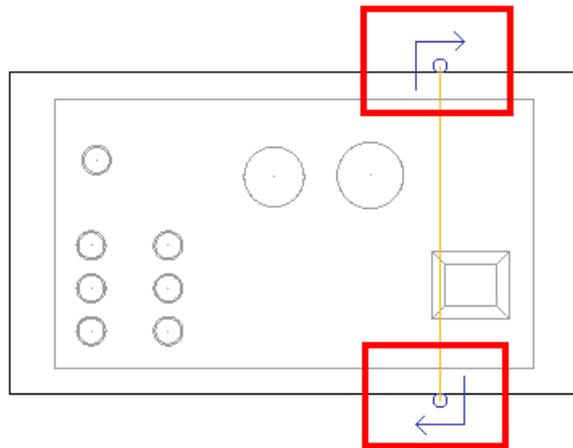
- When you have created a section line, the other options become available on the toolbar.



The following buttons are also displayed on the **Section Line** toolbar:



Toggle direction of the section line. This is the same as clicking on the view's instrumentation to change the direction of the section line direction from horizontal to vertical, or from vertical to horizontal.



✓ **Accept changes and finish** — A section view is created ready for you to place in position on the drawing.

✗ **Cancel all changes and finish.**

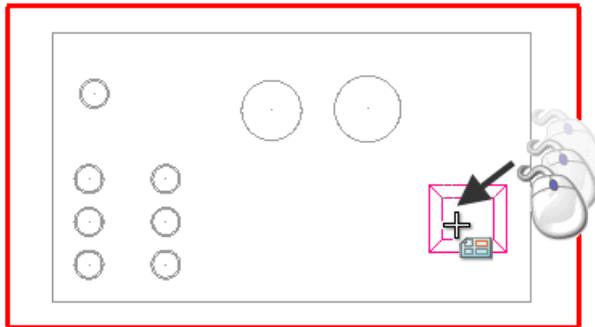
Creating a section line through features

- Click **Object > View > Section**.
- Click **OK** on the **Information** dialog to display the **Section Line** toolbar.

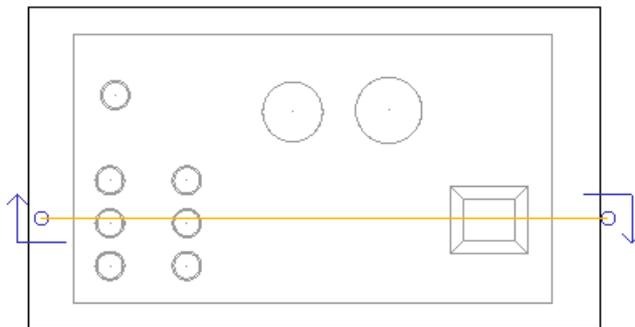


- Click **Set feature recognition**  to change the section line creation mode to **Features**.

- 4 Move the cursor over the view. The features are highlighted automatically.



- 5 Click on any feature in the view and a section line is drawn automatically through the centre of the feature.



- 6 Click  to accept the changes.

Section view creation is started automatically, ready for you to place the section view in position on the drawing.

Creating a section line through points

- 1 Click **Object > View > Section**.
- 2 Click **OK** on the **Information** dialog to display the **Section Line** toolbar.

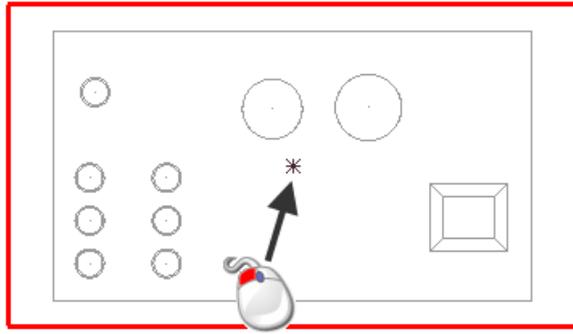


- 3 Click **Set feature recognition**  to change the section line creation mode to **Point**.

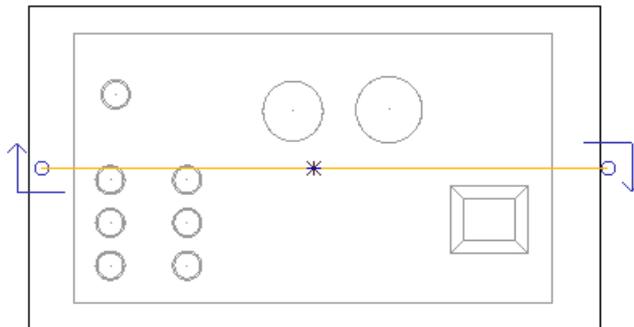


Point mode is useful for avoiding clashes between section features.

- 4 Click in the view to create a point for the section line.



A section line is drawn automatically through the point.



- 5 Click **Accept changes and finish** ✓.

Section view creation is started automatically, ready for you to place the section view in position on the drawing.

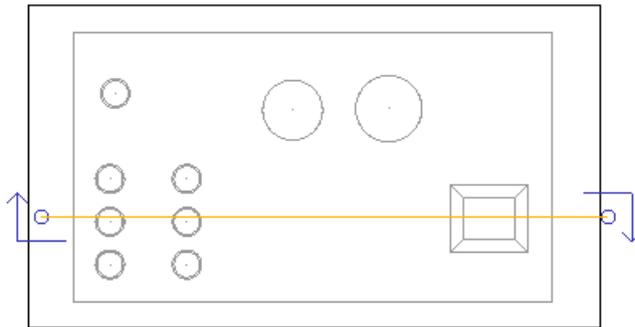
Creating a stepped section line using the Section Line toolbar

- 1 Click **Object > View > Section**.
- 2 Click **OK** on the **Information** dialog to display the **Section Line** toolbar.

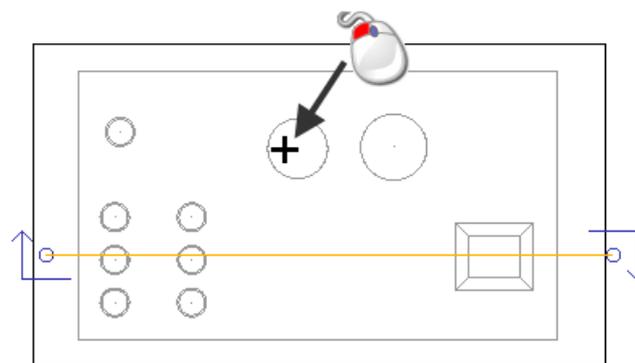


- 3 Use **Set feature recognition**  to choose the mode for creating a section line (see page 56). You can create a stepped section line to include features, points or a combination of both.

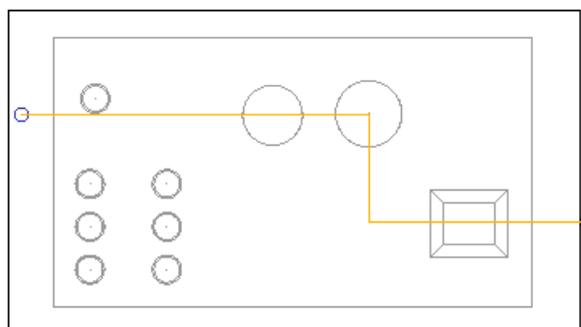
- 4 Click to create a point or click on a feature in the view to create a section line automatically, as described in Creating a section line (see page 56).



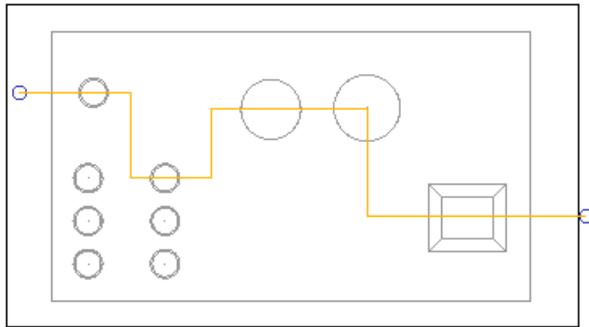
- 5 Click on another feature or create another point for the stepped section line.



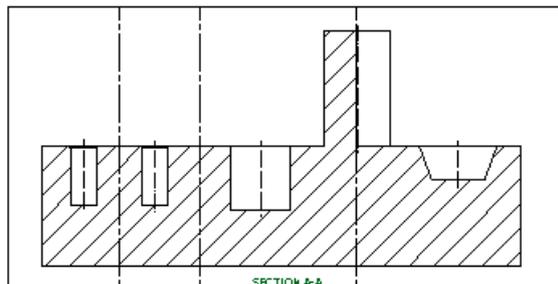
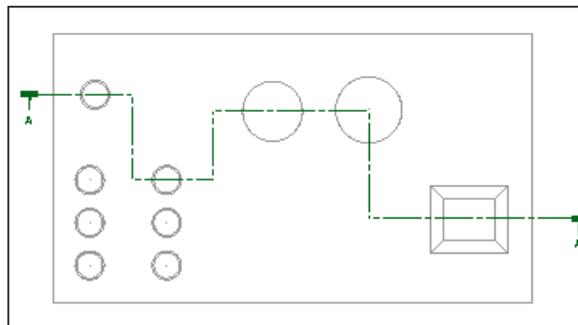
The section line is drawn and steps automatically through each feature or point selected.



- 6 You can continue to click on other features or points to create a stepped section line automatically.



- 7 Click **Accept changes and finish** ✓. Section view creation is started automatically, ready for you to place the section view in position on the drawing; shown in the example below:

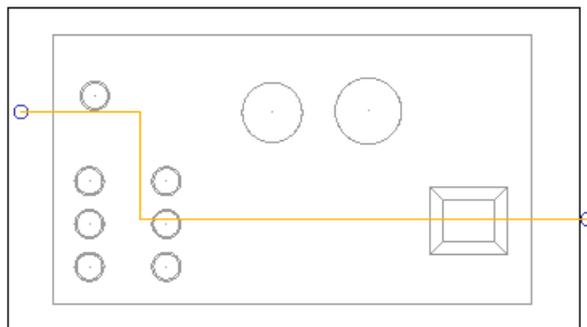
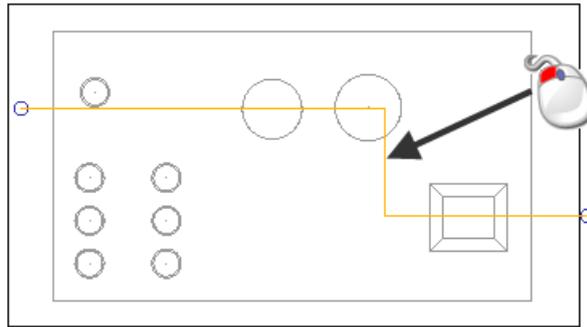


Editing a section line

To change a section line before a section line is accepted:

- 1 Click on the section line.

- 2 Drag the line to a different position.



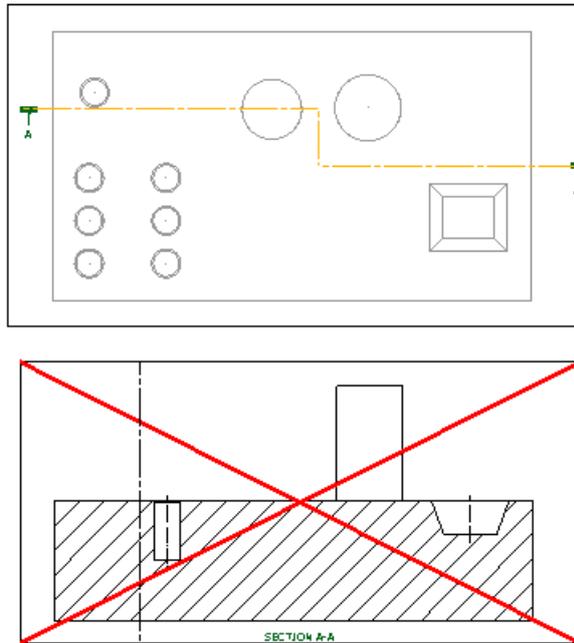
To changing a section line after a section line is accepted and a section view has been created:

- 1 Double-click on the section line to display the **Section Line** toolbar.
- 2 Drag the line to the new position.

The section view displays a red cross to show that the view is out of date because the section line has been changed.



To update the section view, click on the view using the right mouse button and select *Update*. This is the same as selecting *Update* on the *Edit View* dialog.



Using undo:

Use **Undo** to step back during section line creation.



Redo is not currently available

Deleting a section line

1 Click on a section line to select it

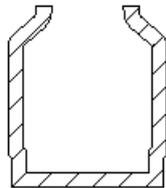
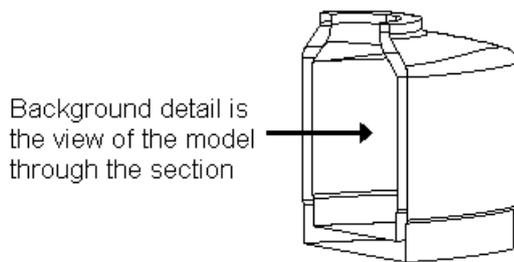
2 Click **Delete**

When you delete a section line, a **Query** dialog is displayed to ask you to proceed with deleting the section view. If you delete the section view, the section line is also deleted.

Section views with background detail

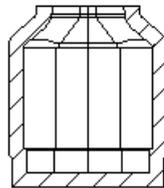
You can create a section view that displays the background detail of a section.

The background detail is the view of the model through the section.



SECTION A-A

Without background detail



SECTION A-A

With background detail

See Also:

Section views (see page 50)

Creating a section view (see page 52)

Creating a section view with background detail (see page 64)

Creating a section view with background detail

To create a section view with background detail:

- 1 Click  (Main toolbar) to display the **Drafting** toolbar.

 This is the same as hovering over  and selecting  from the flyout. For further details, see *Flyout overview*.



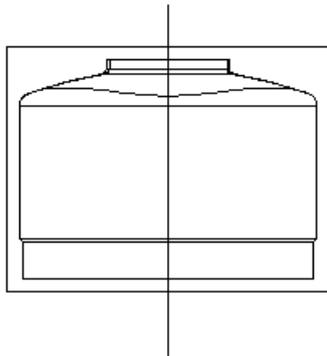
The **Drawing Selector** dialog is displayed where you select an existing drawing or create a new drawing. The **Drafting** toolbar is also displayed.

- 2 Make sure you have a drawing with at least one view on it.

If you do not have a drawing already open, you need to create a drawing or open an existing drawing. For further details, see Creating a new drawing (see page 8) and Opening existing drawings (see page 10)

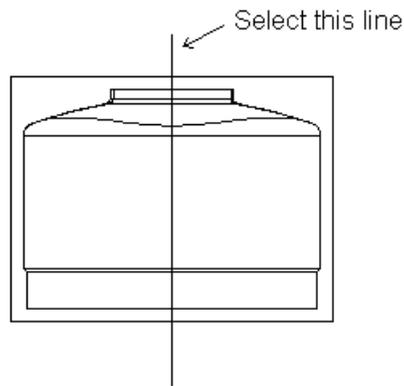
If you have no views on your drawing, create a single view. For further details, see Creating a single view (see page 44).

- 3 Use one of the following methods to create a line through the view where you want the section:
 - Create a section line automatically by using the Section Line toolbar (see page 56). For further details, see Creating a section line using the Section Line toolbar (see page 56).
 - Draw a single line through the view. For further details, see Creating a line in the Wireframe modelling manual, and continue to follow the steps described below.
 - Create a composite curve from a series of orthogonal lines to define a Stepped section (see page 69). For further details, see Selecting objects in the Modelling Concepts manual, and continue to follow the steps described below.
- 4 Select the line or composite curve on the section view.
- 5 Click **View**  (*View creation menu*).
- 6 Click **Section View with Background**  (*View creation menu*).



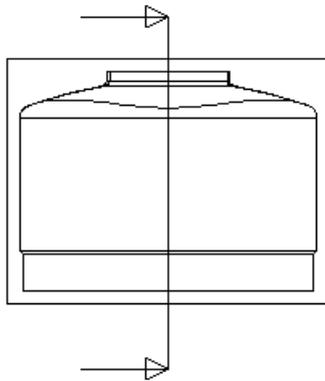
The line or composite curve can be shorter than the view it lies in. This creates a section view of the portion of the view where the line or composite curve lies.

- 1 Select the line or composite curve.



- 2 Click **View**  (*View creation menu*).

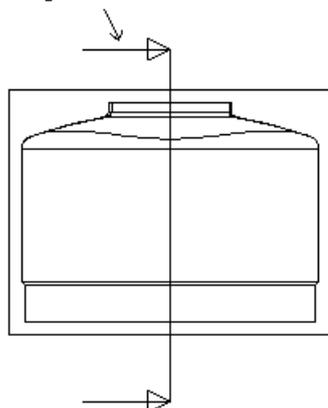
- 3 Click **Section View with Background**  (*View creation menu*).
Arrows are drawn on the line to define the viewing direction.



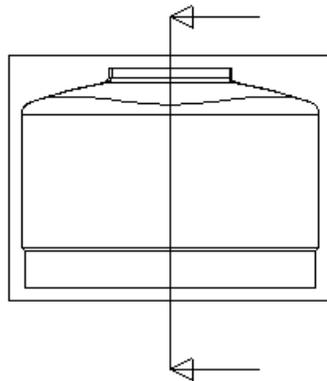
The outline of the new view moves with the cursor.

- 4 Click one of the arrows if you wish to change the viewing direction.

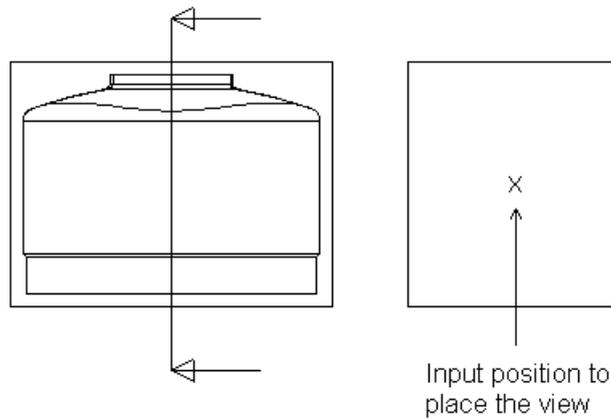
Click the arrow to change the direction



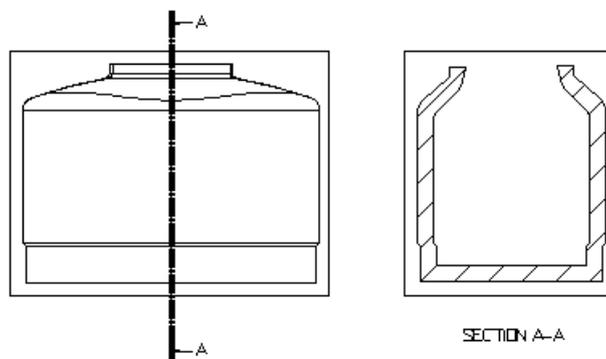
The arrows change direction to indicate that the viewing direction has changed.



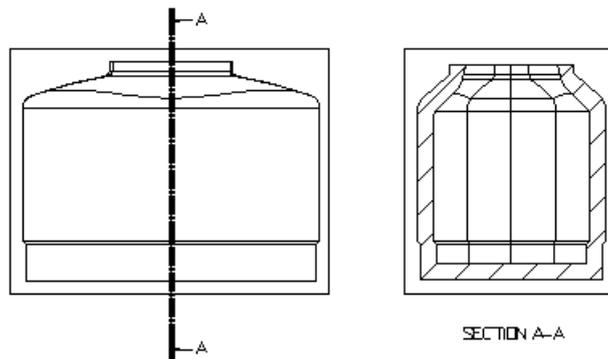
- 5 Click on the drawing to place the view. Alternatively, you can select a different drawing to place the section view. For further details, see Positioning section views on a different drawing (see page 69).



The section view is drawn.



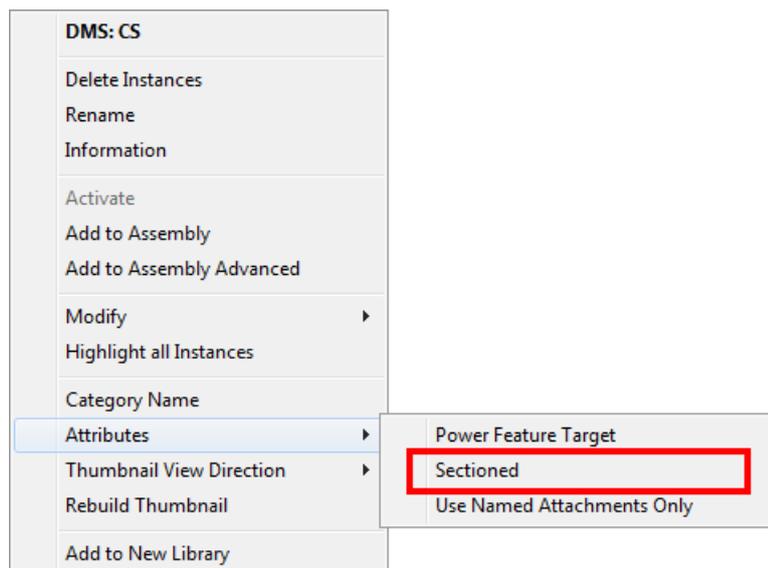
If you selected the section with background view, the background detail is drawn too. For further details, see Creating a section view with background detail. (see page 64)



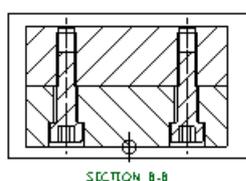
If you select either of the **Hide** options from the **View** context menu, or **Hidden** on the Edit View dialog (see page 104), background detail is hidden in section views. For further details, see Editing a view (see page 100).

Creating a section view of an assembly with revolved components

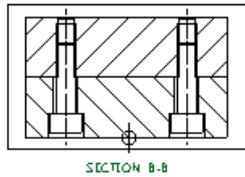
Use the **Sectioned** option on the component definition context menu to avoid sectioning a revolved component (for example a bolt), when creating a **Section View** of the assembly in Delcam Drafting.



Sectioned — If *ON*, revolved components are hatched when a **Section View** is created.



If *OFF*, revolved components are not hatched when a **Section View** is created.



Positioning section views on a different drawing

When creating section views from a model, you can position and fix the section view on another drawing. To do this:

- 1 Create a section view.
- 2 Select the required open drawing from the **Window** menu.
- 3 Position and click to fix the section view on the different drawing.

The section label is generated for the sectioned view drawing.

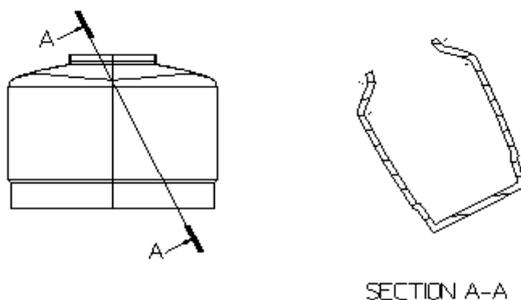
Angular and Stepped Section Views

You can create angular, stepped section and angular stepped section views.

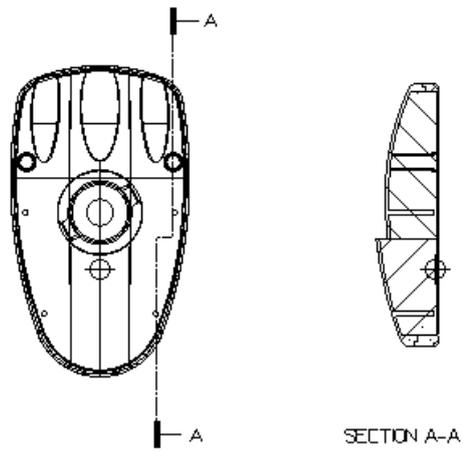
For further details, see:

- Creating angular section views (see page 71).
- Creating a section view. (see page 52)

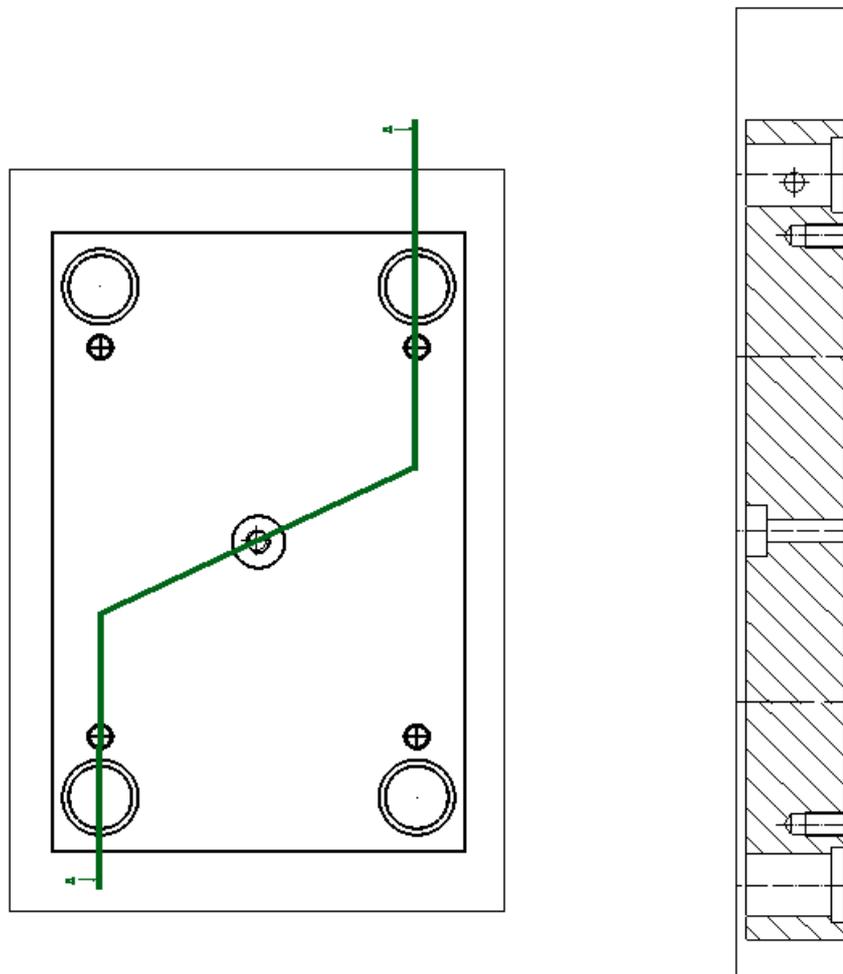
Angular section view



Stepped section view

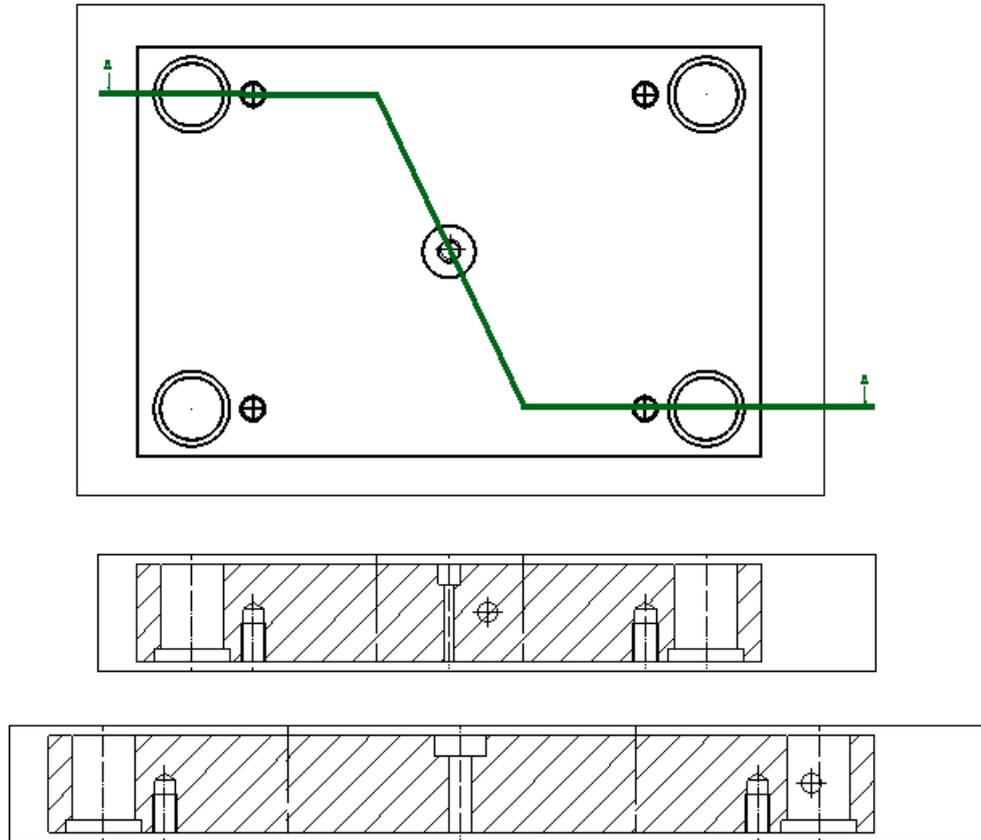


Angular stepped section view



Creating angular section views

You can create two different angular section views.



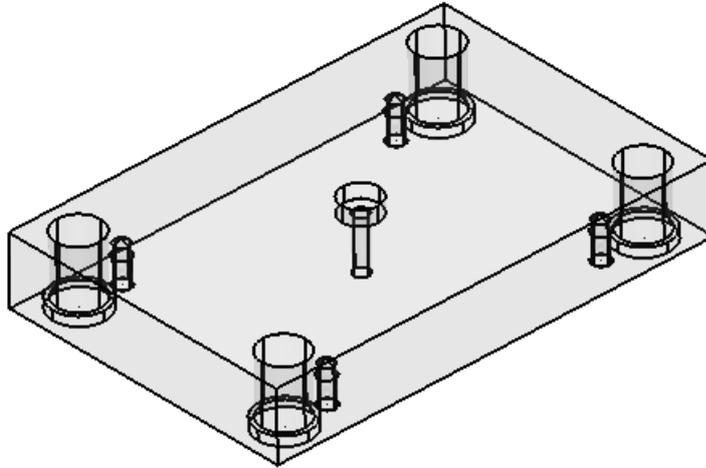
Align Angular Sections with Principal Axis — This functionality produces a section that is the same size as the original because it is viewed in a single direction. Using this option produces the top section in the above example.

Align Angular Stepped Sections — This functionality is used for accurate dimensioning. This option rotates the view in order to display the stepped section in a flattened state to allow a stepped section to be measured. Using this option produces the bottom section in the above example.

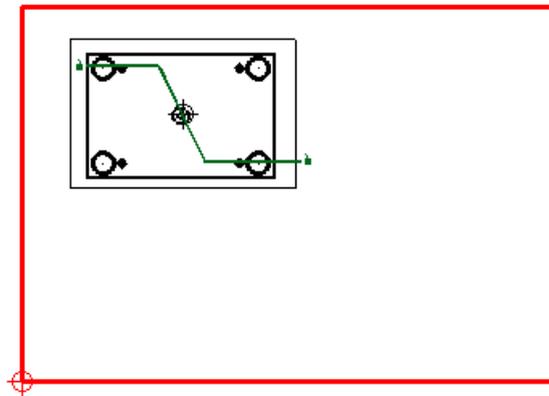
For further details, see Angular section view alignment option (see page 72).

Angular section view alignment options

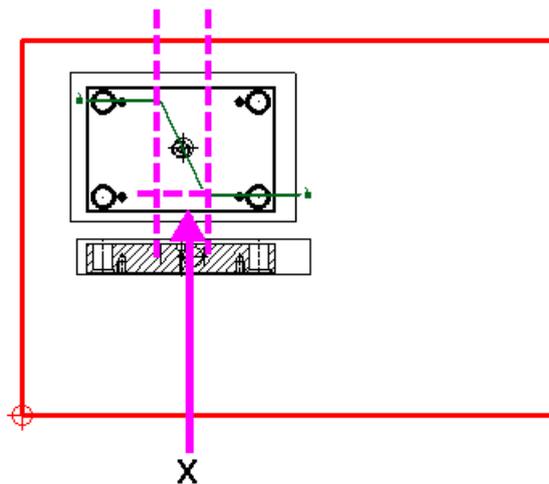
The two angular section view alignment options are used in a similar way, but produce very different results.



Create a composite curves from a series of lines to define a stepped section.

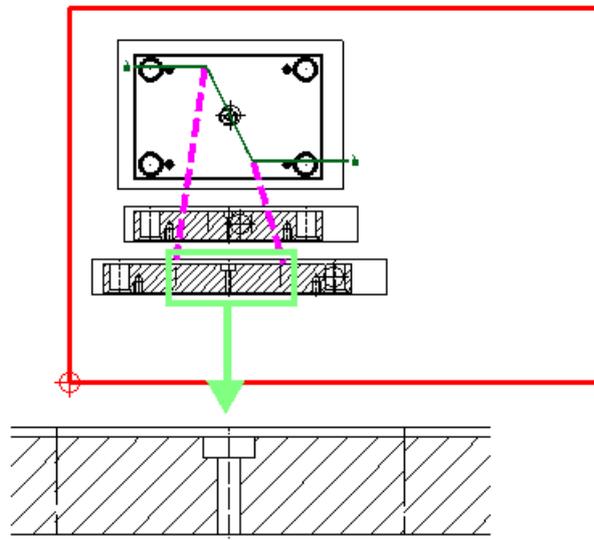


With **Align Angular Sections with Principal Axis** set to *ON*, create a section as shown below.



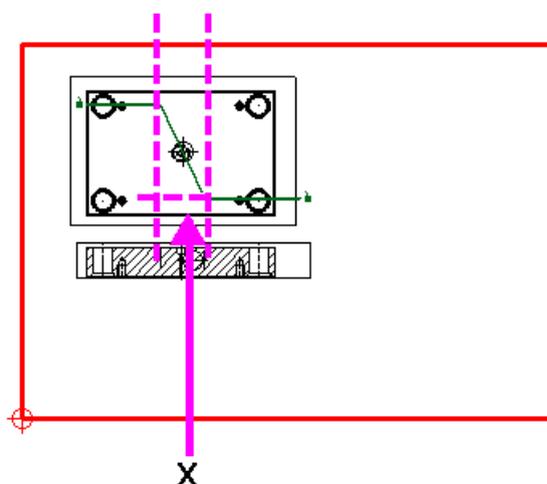
Notice that the stepped section is projected to produce a section that is the same size as the original view. This projection cannot be used for accurate measuring.

Set **Align Angular Sections with Principal Axis** to *OFF* and set **Align Angular Stepped Sections** to *ON*. Create another section below the first one.



The highlighted section above shows how the stepped section has been flattened out to give a true representation of the section of the hole. The flattened section can now be measured accurately.

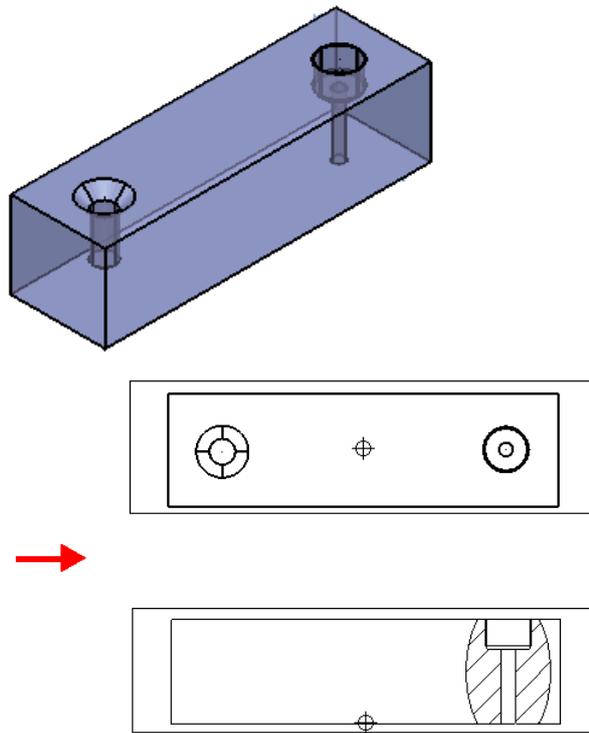
If a stepped section is less than 1mm in the view direction, the section line is invalid. So, in the example below, if **X** is less than 1mm, the section line is invalid and an error message is displayed.



Creating Scrap Sections

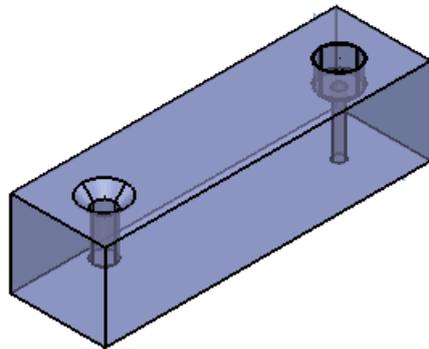
A scrap section lets you see the section of an individual hole on a hidden view. You can also create scrap sections from hole patterns.

Scrap sections cannot be created in a view that is non-orthogonal to the model. The **Scrap Sections** option on the context menu is greyed out when it is not a viable option.

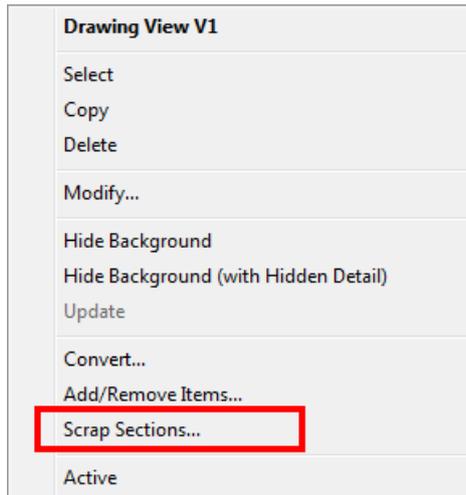


To create a Scrap Section:

- 1 Create a model similar to the one below. Create a corresponding drawing. Add a view and a relative view to the drawing.



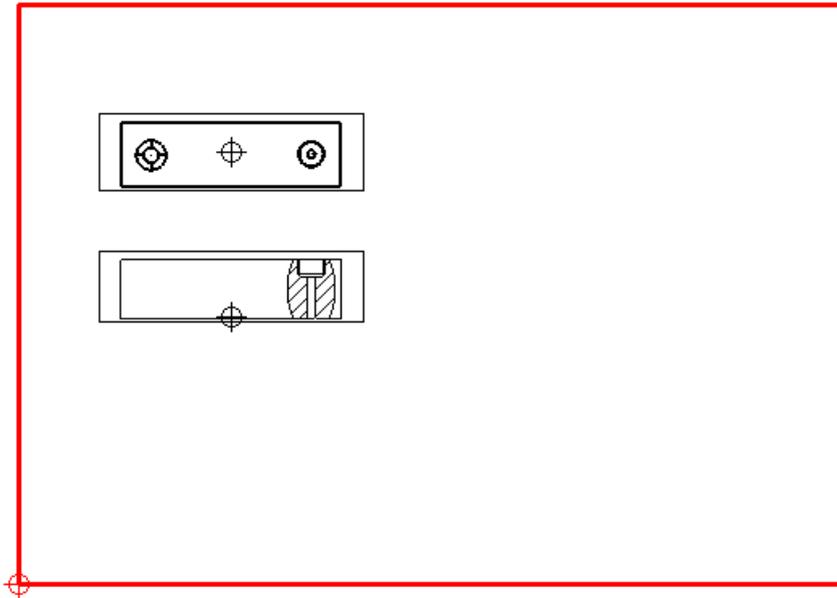
- 2 Right click the relative view to display the **Drawing View** context menu.



Scrap sections cannot be created in a view that is non-orthogonal to the model. The scrap section option on the context menu is greyed out when it is not a viable option.

- 3 Select **Scrap Sections** from the menu to display the **Scrap Sections** dialog (see page 76).
- 4 Select one hole feature and click **Add Feature**. The selected feature is added to the **Scrap Section Features** list. Add additional features to the **Scrap Section Features** list as required.
- 5 Click **OK** to mark the hole features to be displayed as a scrap section when the view is **Hidden**.
- 6 Right click the relative view to display the **Drawing View** context menu. Select **Hide**.

- 7 The view will display any hole features that were added to the **Scrap Section Features** list. All other features are hidden.

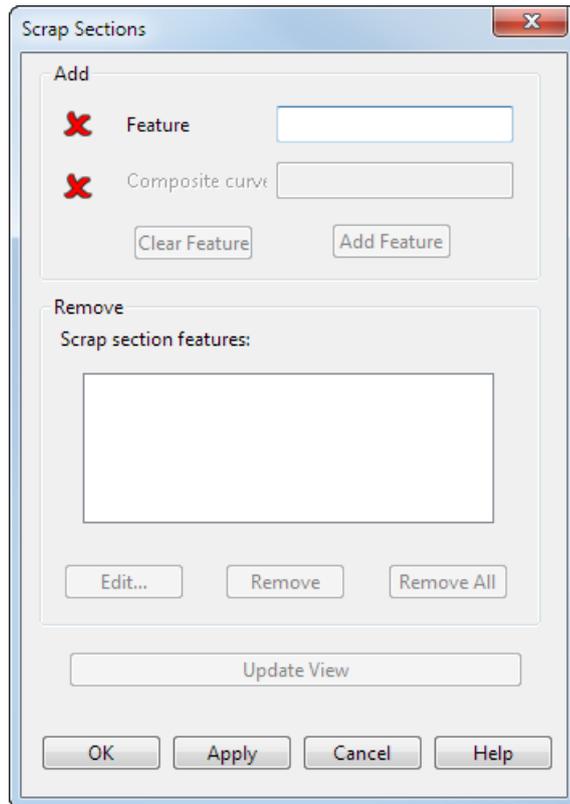


Scrap Sections dialog

Use this dialog to add, edit and remove scrap sections from the view.

You can also create scrap sections from hole patterns.

Scrap sections cannot be created in a view that is non-orthogonal to the model. The scrap section option on the context menu is greyed out when it is not a viable option.



Feature — Select a hole feature on the drawing or enter the name of the feature. The **X** changes to **✓** when you select a valid hole feature.

Composite curve — When you select a **Feature**, the corresponding **Composite curve** is automatically selected. The **X** changes to **✓** when you select a valid hole feature.

Clear Feature — Clear the **Feature** and **Compcurve** boxes.

Add Feature — Add the selected feature to the **Scrap Section Features** list.

Edit — If you select this option, you can edit the scrap section composite curve. The curve must remain planar and closed. Select **Finish** to complete the editing.



When you select Edit, this is replaced by Finish.

Remove — Remove the selected scrap section from the **Scrap Section Features** list.

Remove All — Remove all the scrap sections from the **Scrap Section Features** list.

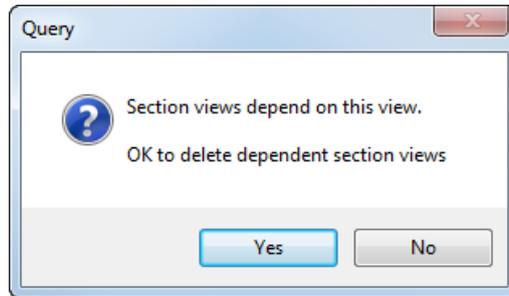
Update View — Update the view with any changes that have been made.

OK — Mark the hole features to be displayed as a scrap section when the view is **Hidden**

Cancel — Any features that are already in the **Scrap Section Features** list are retained and the dialog is removed.

Deleting section views

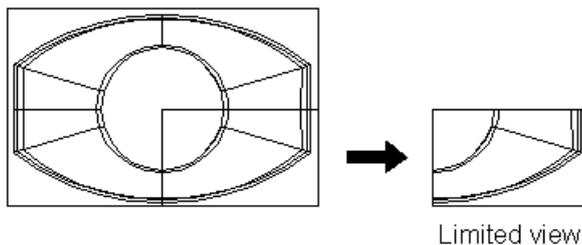
When you try to delete a drawing or a view where section views have been created, a **Query** dialog is displayed to warn you of section view dependencies.



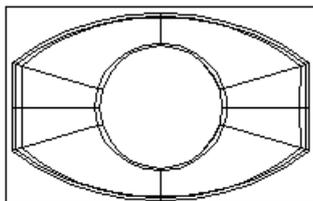
Click **Yes** to delete both the section view and the view or drawing it is dependent on.

Creating a limited view

You can create a view (known as a limited view) of a portion of an existing view.



To create a limited view of the drawing view shown below:

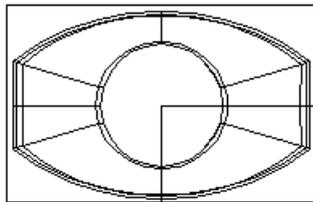


- 1 Display the **Drafting** toolbar.
- 2 Click  (*Main toolbar*). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Make sure you have a drawing with at least one view on it.

If you have no views on your drawing, create a single view. See Creating a single view (see page 44) for further details.

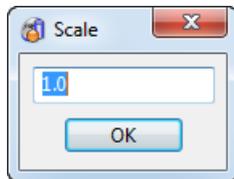
If you do not have a drawing already open, you need to create a drawing or open an existing drawing. For further details, see Creating a new drawing (see page 8) and Opening existing drawings (see page 10)

- 4 Click  (*View creation menu*).
- 5 Use the left mouse button to drag a box in an existing view to select the portion you want in the limited view.



Drag a box in an existing view

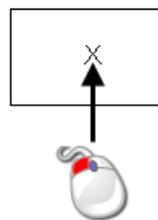
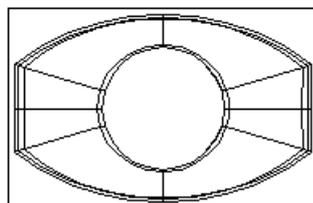
Once you selected part of the view, a dialog is displayed for you to enter a value to set the scale of the model in the limited view.



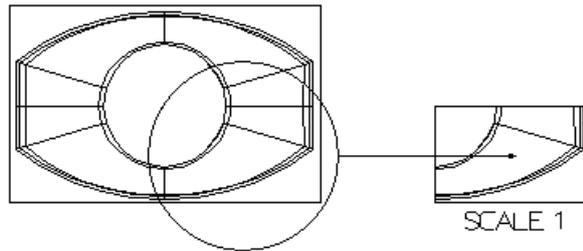
- 6 Change the value in the dialog, if you want.
- 7 Click **OK**.

The outline of the new view is attached to the cursor.

- 8 Click a position to place the new view.



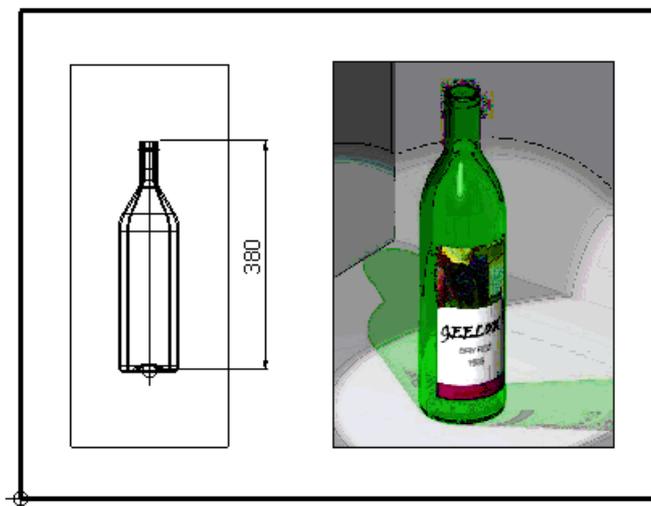
The new view is added to the drawing with markings indicating where it originally came from.



The scale of the limited view is displayed below the view. This updates if you change the scale of the view. For further details, see Edit View dialog (see page 104).

Creating an OLE view

An OLE view contains an OLE object such as a Word document. In the example below, a bitmap is inserted on the right hand side as an OLE view.

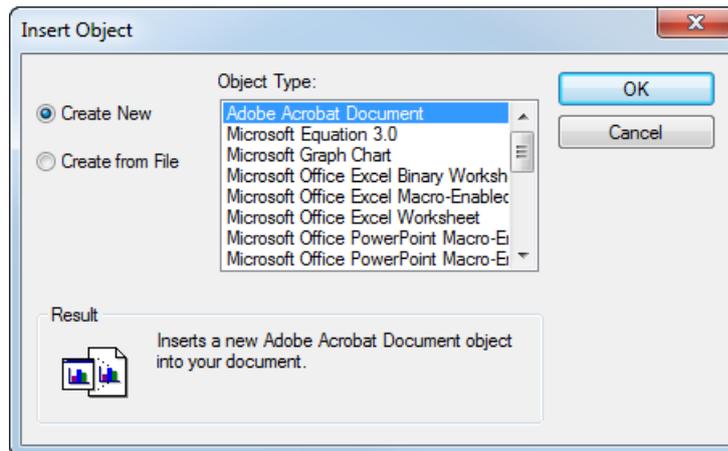


- 1 Display the **Drafting** toolbar.

Click  (*Main toolbar*). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.

- 2 If you do not have a drawing already open, you need to create a drawing or open an existing drawing. For further details, see Creating a new drawing (see page 8) and Opening existing drawings (see page 10)

- 3 Click  (View Creation menu).

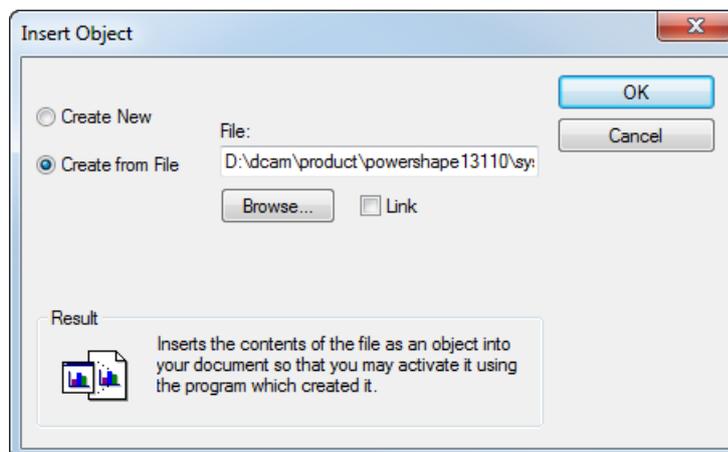


- 4 Use the dialog to select the object you want to insert.

The **Object Type** list contains all the type of objects you can insert into the drawing.

To create a new object, select an object type from the list.

To create from an existing file, click the **Create from File** option and the dialog changes to the following.



Type in the path name of the file or use the **Browse** button.

 You can only insert the type of files given in the **Object Type** list. This list is displayed when the **Create New** option is selected.

 If you select the **Link** option, the OLE view displays the external file. If you make any changes to the external file, the view is updated too. Similarly, if you edit the OLE object in the view within PowerSHAPE, the external file also updates.

If you don't select the **Link** option, a copy of the file is embedded in the view. Any changes to the external file do not change the view.

5 Click **OK**.

6 Click on the drawing where you want to place the OLE view.

If you are creating a new object, the object is added to the drawing and opened in its application, ready for you to insert data. When you save and close the file, you can see the contents of the object in the view.

If the object is created from a file, it is drawn in the view, showing its contents.

To edit the OLE object in the view, select the view and double click it to open the file in its application.

Changing the settings of a view

1 Select **Tools > Options > Drafting > Views** branch. There are the following pages of options:

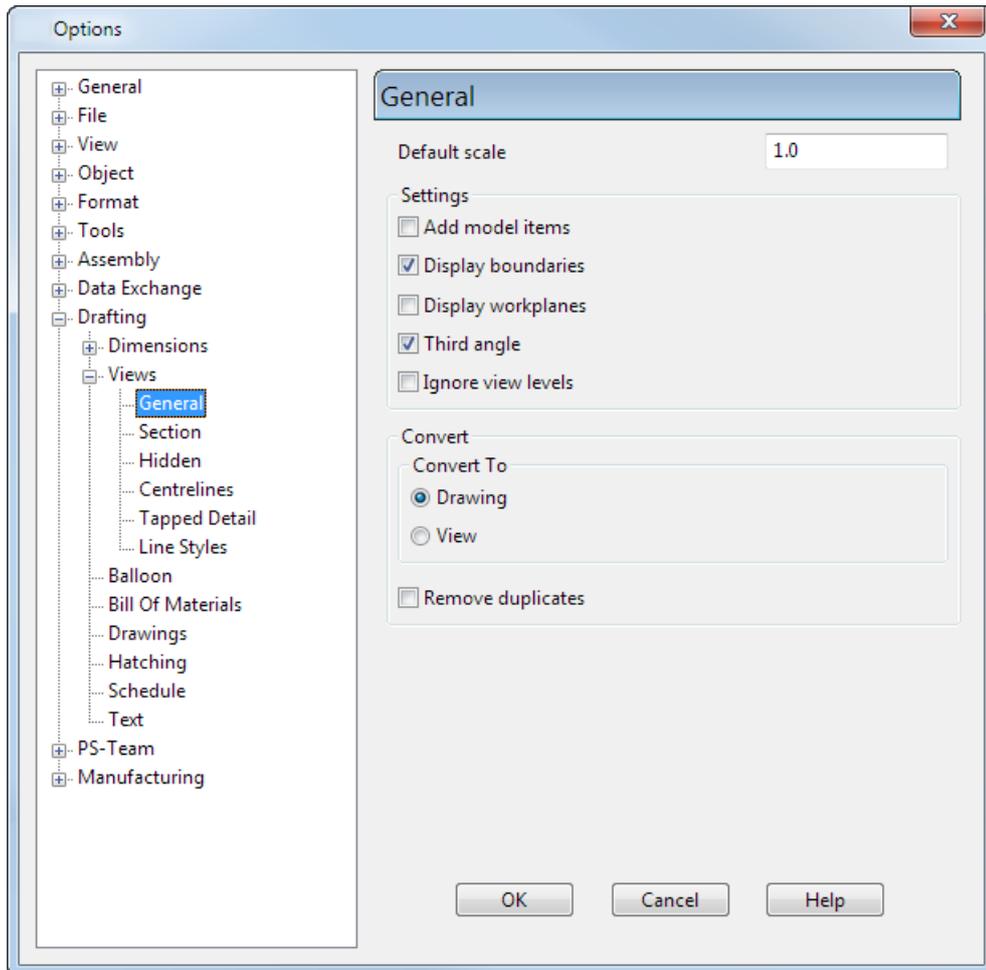
- General (see page 83)
- Section (see page 85)
- Hidden (see page 87)
- Centrelines (see page 88)
- Styles (see page 92)



The drawing view levels are now independent of the global level setting. This means that levels can be displayed in views when the global levels are set off.

Views Options dialog - General page

Use this dialog to specify **General** default view settings.



Default scale — This sets the scale of the model in the view.

Add model items — When you create an object in a view or a model, you can use the **Add model items** option to decide where else that object is displayed. The table below shows where objects are displayed when the **Add model items** option is on or off.

Add model items option	Create object in	Where else the object is displayed
ON	Model window	In all views of all drawings
OFF	Model window	Nowhere else
ON	Active view	In all views in the current drawing and in the model, but not in any views of other drawings
OFF	Active view	Nowhere else

Display boundaries — When on, the drawing and view boundaries are displayed in the drawing window. The boundary of an active view will be displayed if the option is *OFF*.

Display workplanes — Select this option to display any workplanes created in drawing views. The default is *OFF*. If the option is *OFF* a dot is drawn at the origin of the workplane in the view so that you can snap to the origin.

Third angle — When *ON*, a relative view is created using the third angle projection, otherwise the first angle is used.

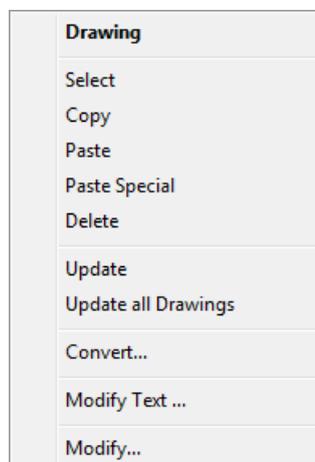
Ignore view levels — Select this option to display the content of all levels in the drawing view. Default is *OFF*.



Delcam Electrode users should update the levels in each view separately (from version 7.1).

Convert

Convert to: This sets the default option for the **Convert** command on the **Drawing** context menu.



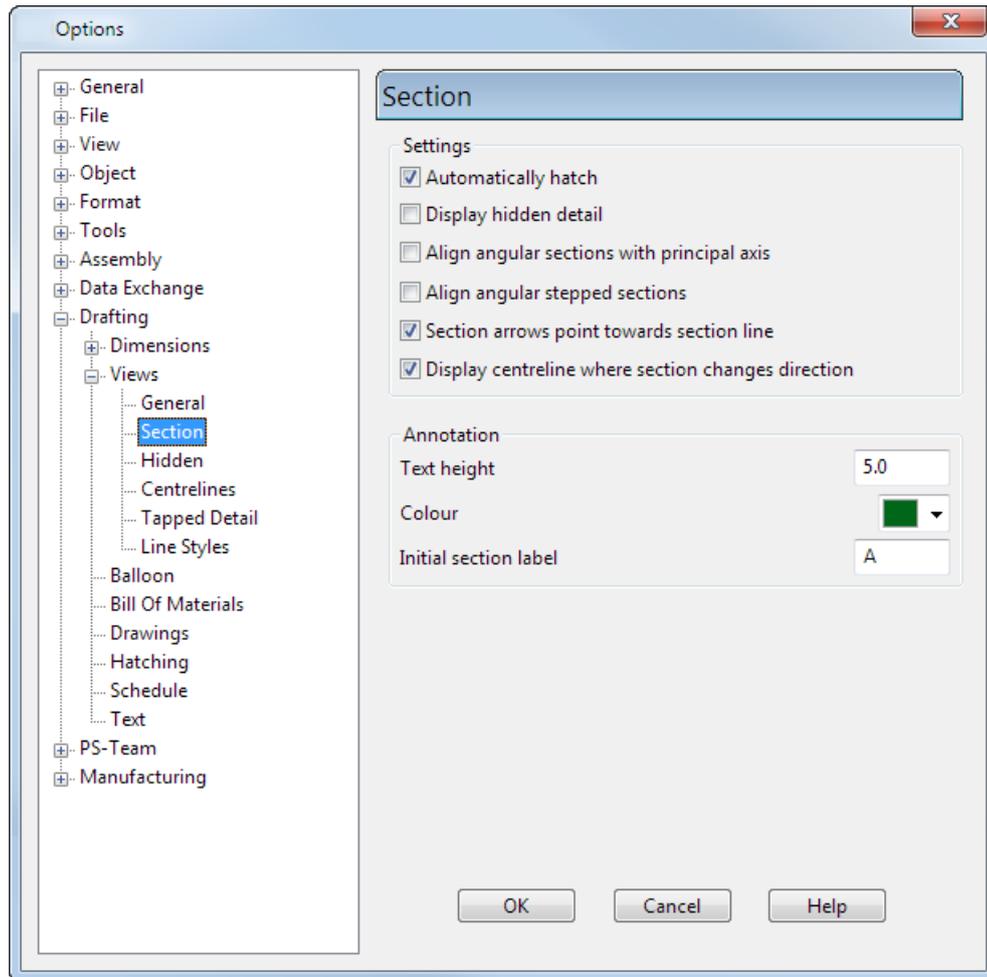
The **Convert** option exports a 2D wireframe representation of a drawing to a **View** or another **Drawing**. For further details, see *Editing a Drawing* (see page 15).

- **Drawing** — Select to convert a drawing to another drawing when the **Convert...** option is selected on the **Drawing** context menu.
- **View** — Select to convert a drawing to a view when the **Convert...** option is selected on the **Drawing** context menu.

Remove duplicates — This removes any duplicates when the view is converted. This also defines the default setting for removing duplicates using the **Remove duplicates** option available on the **Convert View** dialog (see page 102).

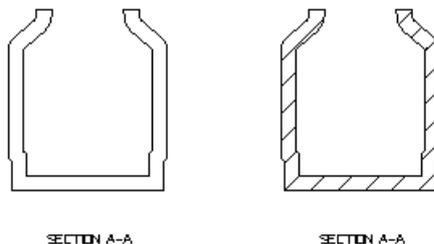
Views Options dialog - Section page

Use this dialog to specify default **Section** view settings.



Settings

Automatically hatch — When *ON*, section views are automatically hatched.

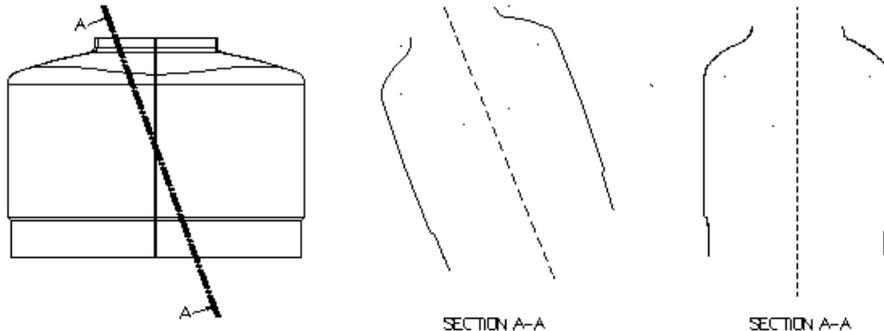


Display hidden detail — If *ON*, hidden lines in section views will be displayed.

Align angular sections with principal axis — If *ON*, an angular section view is aligned to the nearest principal axis. If *OFF*, the angular section is aligned with the section line. In the figure below, the middle view is a section view with **Align angular stepped sections** turned *OFF*. If you create the section view with **Align angular sections** *ON*, the view on the right is created. For further details, see *Creating angular section views* (see page 71).



The dashed lines are added to the drawing to show how the section view is aligned.



Align angular stepped sections - If *ON*, this option rotates the view in order to display the stepped section in a flattened state to allow a stepped section to be measured. This functionality is used for accurate dimensioning. If *OFF*, the view displays the stepped section. The default setting is *OFF*.

See *Angular section views* (see page 71) for more information on creating angular section views.

Section arrows point towards section line — If *ON*, the section arrow points to the section line. If *OFF*, the section arrow points from the section line.

Display centreline where section changes direction — If selected, the centreline is displayed where the section changes direction.

Annotation

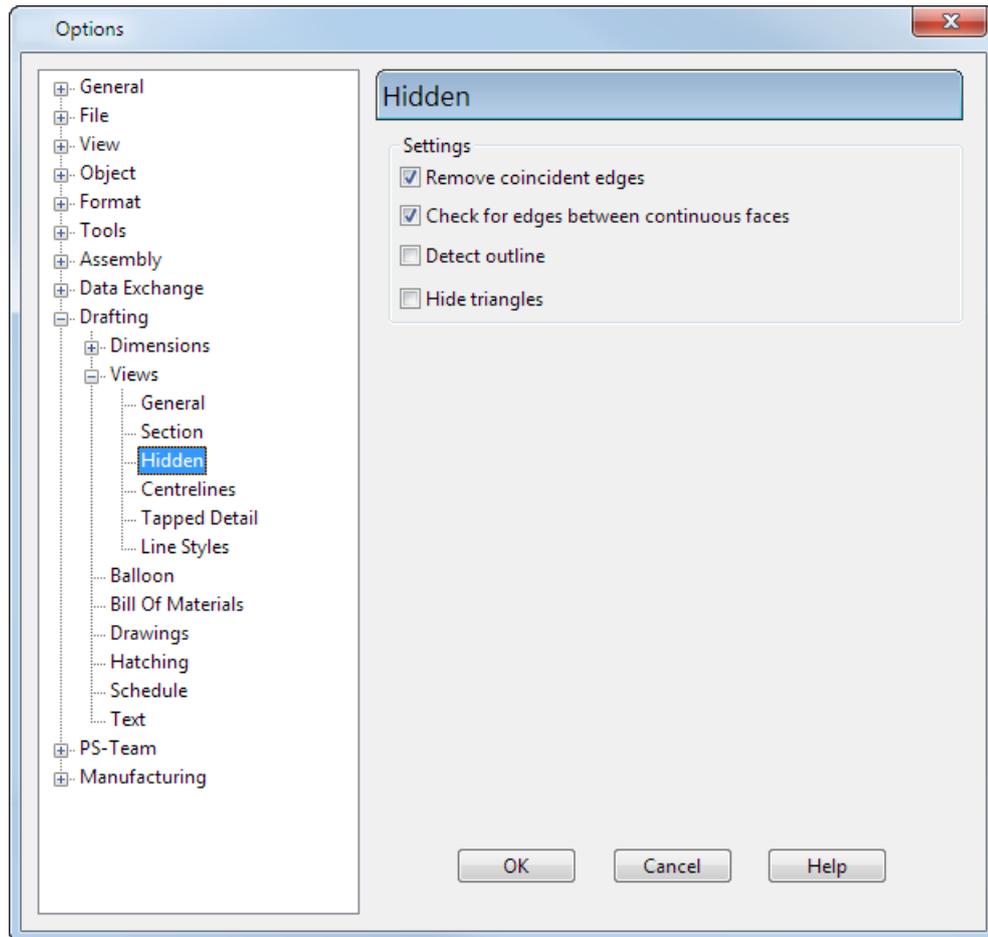
Text height — Define the height for the section annotation text.

Colour — Select a colour of the section text from the drop-down list.

Initial section label — Enter an initial label for an incremental label series that is displayed with each new section created. You can use a maximum of 32 characters for the label which should end with an alpha/numerical combination that can increment, for example *Filter D2*.

Views Options dialog - Hidden page

Use this dialog to specify default **Hidden** view settings.



When you use **Hide** on the Edit View dialog (see page 104) to perform hidden line removal, you can set the following two options.

Remove coincident edges — Remove coincident edges in the hidden view. It can be time consuming to remove coincident edges in large models. The benefit of removing the edges is that when you plot the drawing, the extra edges are not drawn. If this option is off, the plot may contain double edges where edges overlap.

Check for edges between continuous faces — If *ON*, this option:

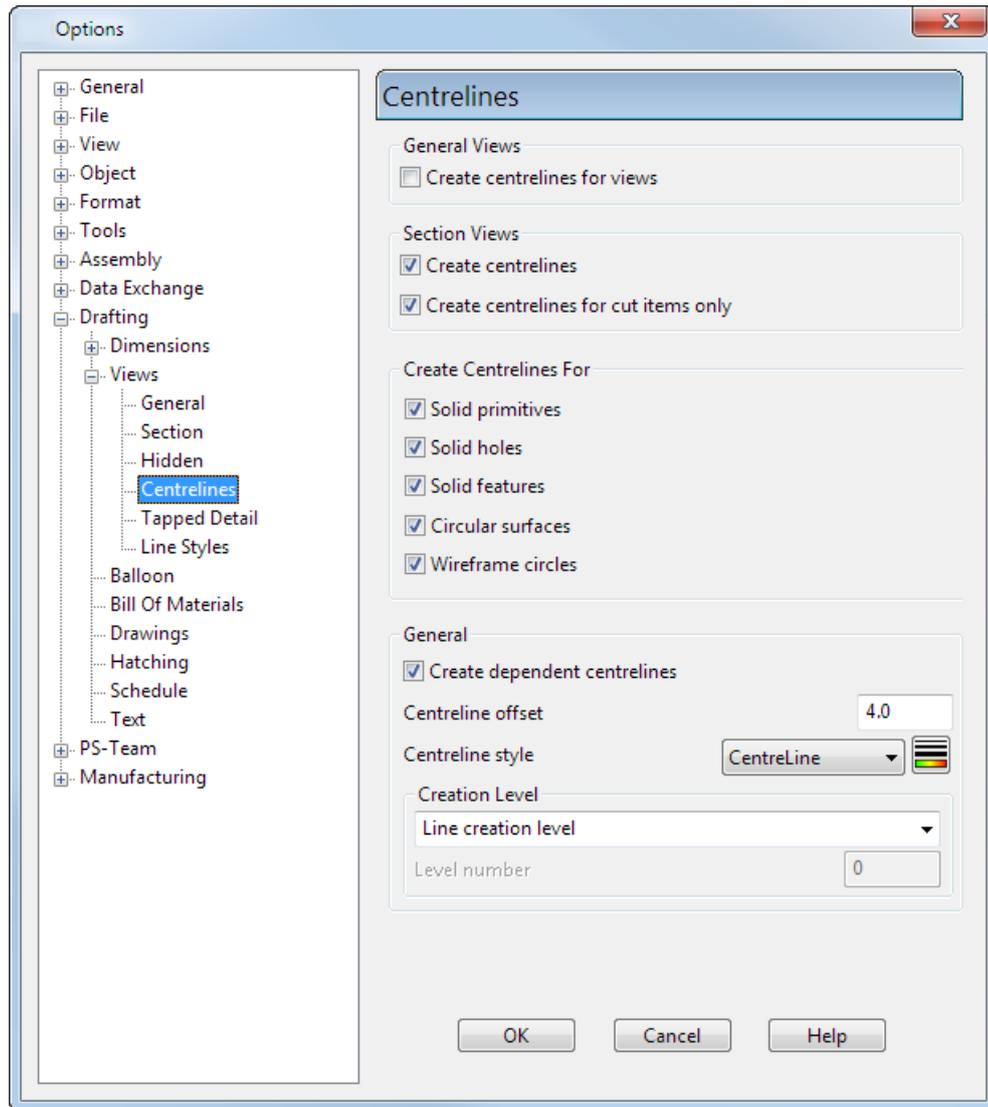
- Removes edges between continuous planar faces.
- Checks for edges between non-planar continuous faces. It flags any found as being internal. The style of the internal lines is set in the **Internal** drop-down list.

Detect outline — Select this option to detect the outline of a model in a hidden view. This locates the outline edges so a style can be applied. If *ON*, the edge is checked to see if it is an outline.

Hide triangles — Select this option to hide triangles in a hidden view.

Views Options dialog - Centrelines page

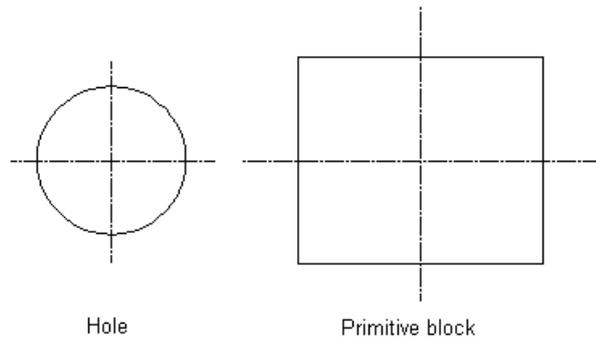
Use this dialog to specify default **Centrelines** view settings.



General Views

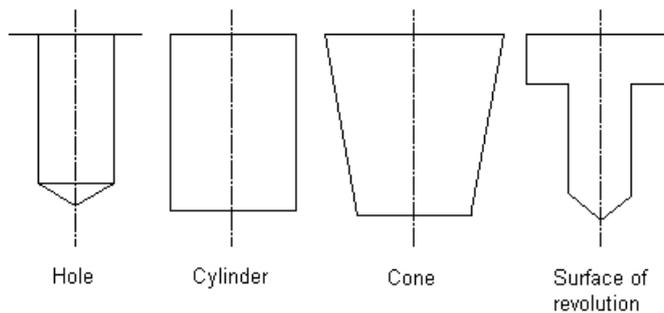
Create centre lines for views — When selected, centre lines are automatically created through certain solid features and primitives in all views except section views. How the centre line detail is drawn depends on the feature and the direction of the view.

Two lines which cross are drawn if either a hole is viewed from the top or a primitive block is in the view.



Centre lines are displayed on blocks only in plan view (looking down the Z axis).

A line is drawn along the centre if a hole, cylinder or cone is viewed from the side. A line is drawn along the axis of revolution for surfaces of revolution.



Section Views

Create centre lines — When selected, centre lines are automatically created through solid features and primitives in all section views. The centre lines drawn are only drawn if the section plane cuts the primitive or the feature. See **Create Centre Lines for Views** for details on how the centre lines are drawn for each feature.

Create centre lines for cut items only — If selected, the centre lines are only created for cut items in the section views, otherwise centre lines are created for all rotated objects.

Create Centrelines For

Use these options to control the creation of centrelines on:

- Solid primitives**
- Solid holes**
- Solid features**
- Circular surfaces**
- Wireframe circles**

General

Create dependent centrelines — Select this option to create dependent centrelines. The option is selected by default.

Centreline offset — Enter the required centreline offset value to change the amount the centreline extends past the object. The default centreline offset is **4.0mm**. The value is stored in the model.

Centreline style — Select a style for centrelines from the drop-down list.

 **Set Styles** — Click to display the **Style toolbar** where you can customise a style for the centreline.



Style changes are made the current style and are applied to the current selection.

Creation Level

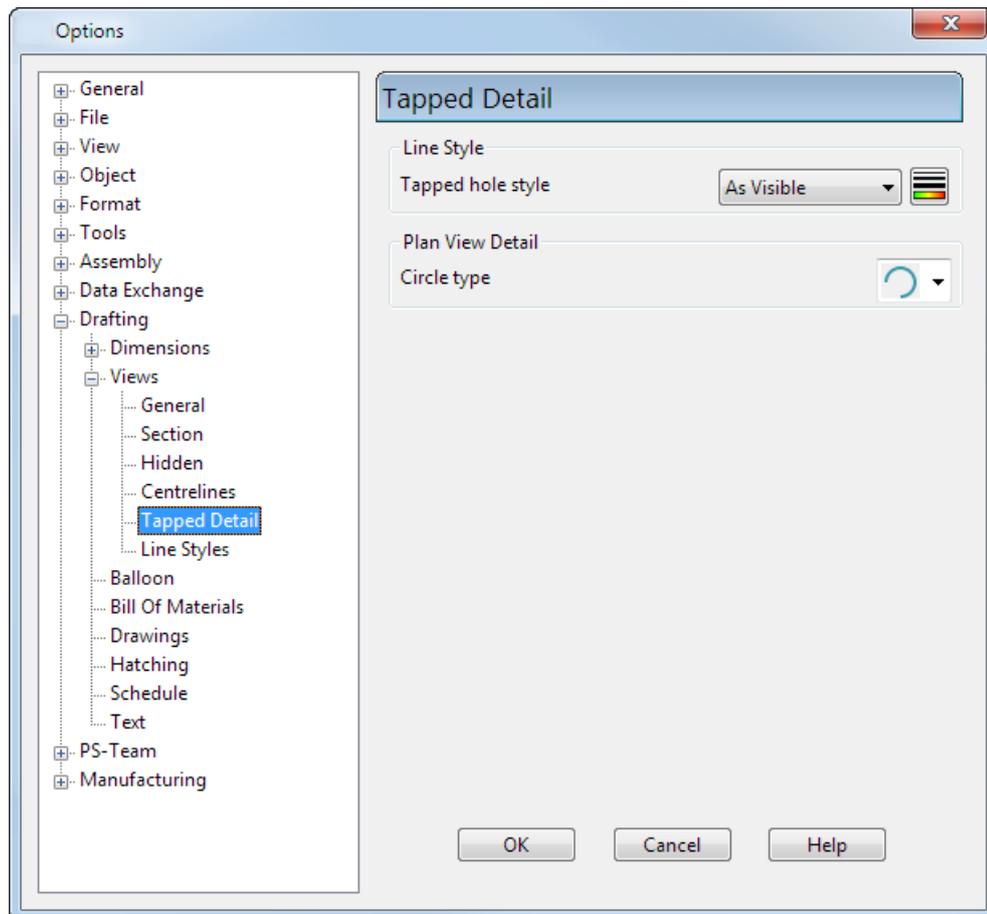
Select one of the following options from the drop down list:

- **Line creation level** — Creates a centreline on the current line creation level.
- **Level number** — Enter the level required in the **Level Number** entry box. This creates a centreline on the specified level.
- **Level of the associated object** — Creates a centreline on the level of the associated item. This option is available only when **Create Dependent Centrelines** option is selected.

Level number — This option is available when the **Level Number** option is selected from the **Creation Level** drop-down list. Enter the specific level number required for centreline creation.

Views Options dialog - Tapped Detail

Use this option page to specify the line style and plan view detail of tapped holes.



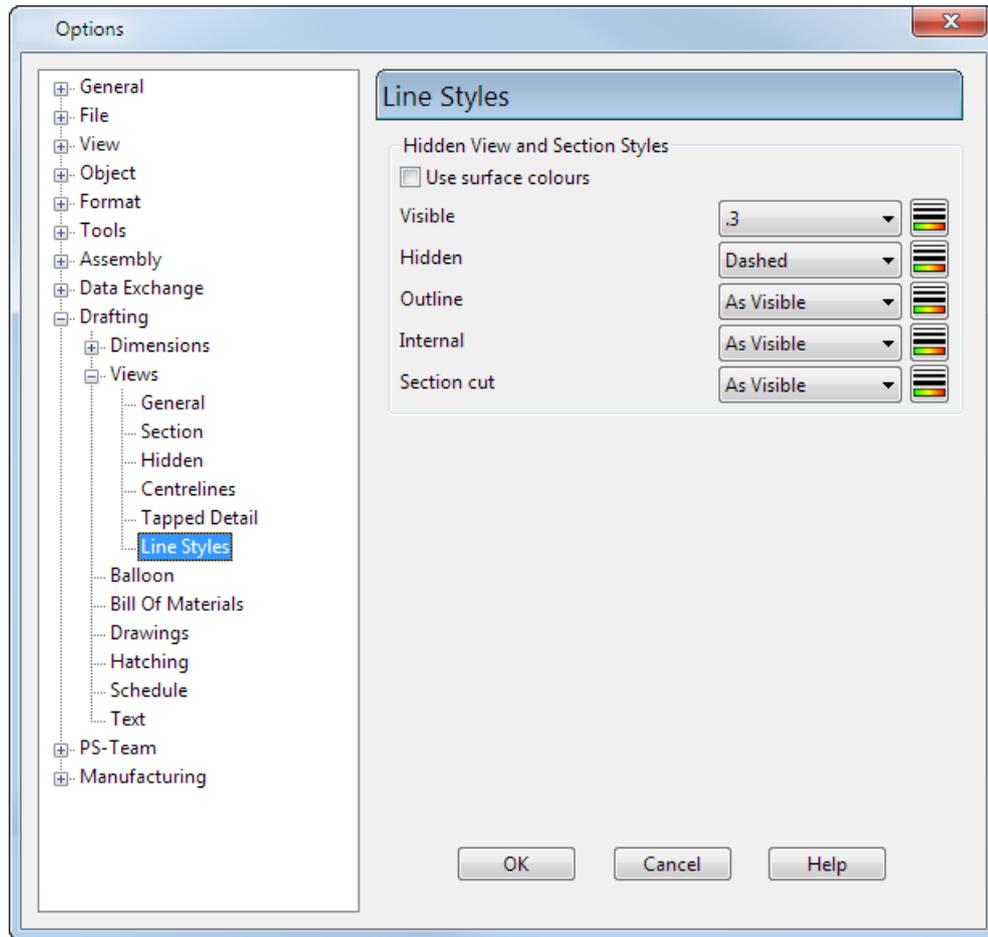
Tapped hole style — Use the drop-down list to select the style for tapped holes.

Alternatively, create a customised hole style by clicking  to display the **Style** toolbar.

Circle type — Select partial circle  or full circle  from the drop-down list. The selection is used to draw the tap detail in plan view.

Views Options dialog - Line Styles page

Use this dialog to specify default line styles.



Use surface colours (Hidden View) — The surfaces in the hidden view uses the same colour as the wireframe of the original surface.

Visible — Use the drop-down list to select the style of the visible lines in the hidden view.

Hidden — Use the drop-down list to select the style of the hidden lines in the hidden view.

Outline — Select a style to be applied to the outside edges detected in a hidden view. For the selected style to be applied, the **Detect outline** option on the **Hidden Views** page of the **Options** dialog must be selected.

Internal — Use the drop-down list to select the style of the internal edges in the hidden view. An internal edge is an edge that lies between faces that are tangent continuous.

Section Cut — Use the drop-down list to select the style for the geometry that lies on the section plane.



Set style — Click to display the **Style** toolbar, where you can customise your own style each of the **Styles** options.



Style changes are made the current style and are applied to the current selection.

Activating a view

When a view is active you can:

- Create and edit objects in it.
- Select objects and key points in the active view.

See *Creating objects in a view* (see page 93) for further details.

To activate a view:

- 1 Right-click in the view.
- 2 From the context menu, select **Active**.

The view displays a thick red border around it to show it is active.

You can also use the *Edit View dialog* (see page 104) to activate a view. See *Editing a view* (see page 100) for further details.



If you delete an object from an active view, it is also deleted from the model and any view that contains it.

Deactivating a view

Right-click the view to display the context menu and select **Active**.

Creating objects in a view

- 1 Activate the view. For further details, see *Activating a view* (see page 93).
- 2 Create objects.

The objects are created in the 3D model and not on the drawing sheet. To edit these objects later on, the view must be active. You can also edit these objects in a model window.

If the **Add model items** option is *ON* in the **View** page of the **Options** dialog, then the object is only displayed in the active view. If the option is *OFF*, then the object is displayed in the model and all the other views on the drawing. For further details, see *Changing the settings of a view* (see page 82).

You can use objects in the model. For example, snap to key points of objects in the model and create composite curves using the model.

Updating a view

Objects, in the model and not in the view, can be displayed in the view. You can also remove objects from a view and not the model.

To add all objects to a view

- 1 Activate the view. For further details, see *Activating a view* (see page 93).
- 2 From the **View** menu, select **Unblank**.

All objects, that are in the model and not in the view, are now displayed and selected in the view.

To add only some of the objects in a view

If you want to unblank only a few of the blanked objects, you can use the fact that unblanked objects are automatically selected.

- 1 Activate the view. For further details, see *Activating a view* (see page 93).
- 2 From the **View** menu, select **Unblank**.
All objects, in the model and not in the view, are displayed and selected in the view.
- 3 Be careful not to clear the selection.
- 4 While all the unblanked objects are highlighted as selected, de-select the objects you want to leave unblanked (using **Ctrl-Shift**-left mouse click).
- 5 From the **View** menu, select **Blank Selected** (to blank all the objects you want to leave blanked).

To add only some of the objects in a cluttered view

If the view is cluttered, and selecting objects to blank is difficult, you may find it more convenient to use **Blank Toggle**.

- 1 Activate the view. For further details, see *Activating a view* (see page 93).
- 2 From the **View** menu, select **Blank Toggle**. This blanks all unblanked objects and unblanks all blanked objects.
- 3 De-select the objects you want to eventually leave unblanked (using **Ctrl-Shift**-left mouse click).

- 4 From the **View** menu, select **Blank Except** to blank the de-selected objects.
- 5 From the **View** menu, select **Blank Toggle** again.



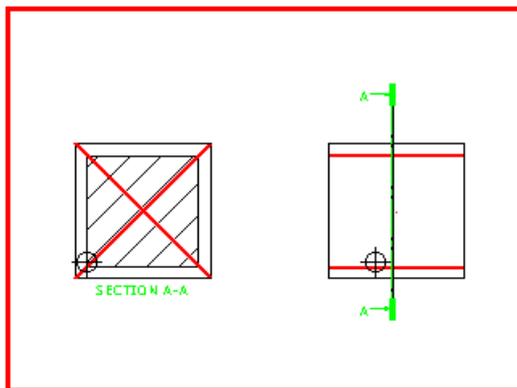
Blanking items in model view does not cause the items to be blanked in drawing views.

To remove objects from a view

- 1 Activate the view. For further details, see *Activating a view* (see page 93).
- 2 Select the objects in the view you wish to remove.
- 3 From the **View** menu, select **Blank Selected**.
The objects are removed from the view.

Out of date views

If you make changes to the model, a red cross is displayed on corresponding views to indicate the view is out of date. This is useful on more complex models.



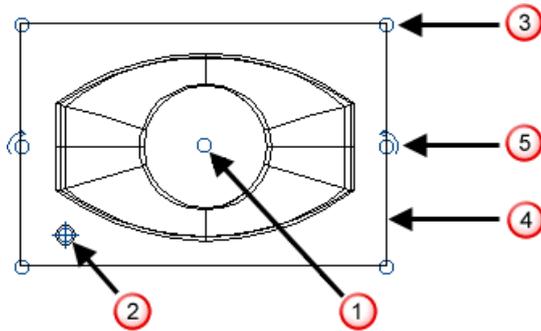
Graphically editing views

Select the view to display its graphical handles.

The cursor changes to a hand when you move the mouse over view instrumentation.



To select the view, right-click the view and click **Select** from the pop up menu.

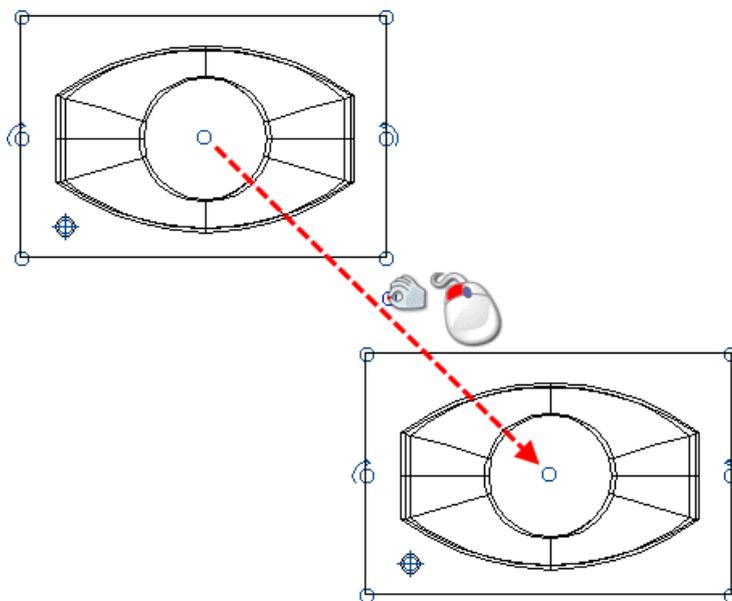


There are five types of handles:

- Centre (see page 96) ①
- Origin (see page 96) ②
- Corner (see page 97) ③
- Side (see page 98) ④
- Rotate (see page 99) ⑤

Centre handle

The centre handle is used to position the view. Grab the handle and move the view to its new position.



Origin handle

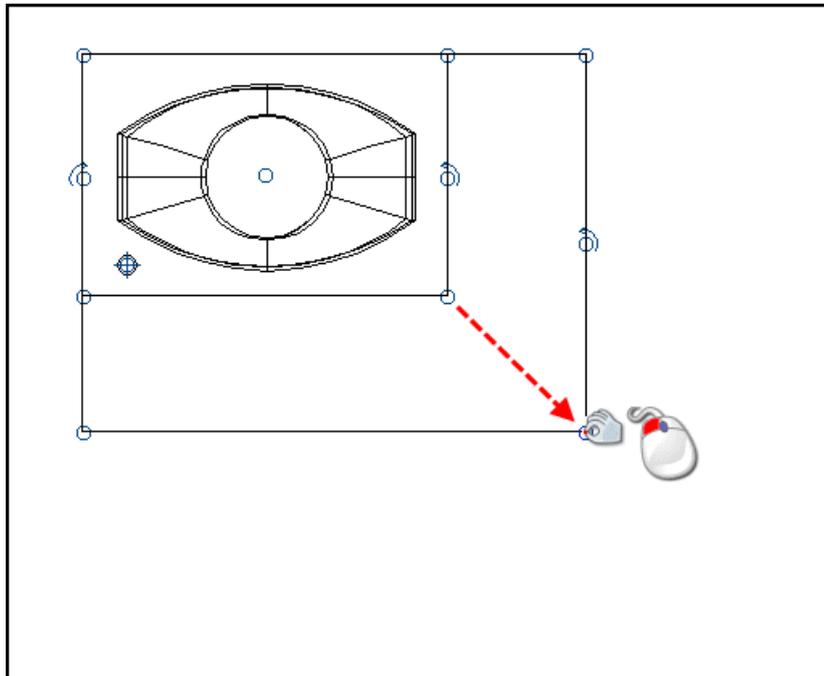
The origin handle, like the centre handle, is used to position the view. Grab the handle and move the view to its new position.

Unlike the centre handle, this handle uses the intelligent cursor to enter positions. For example, it displays construction lines and labels as you pass it over key points.

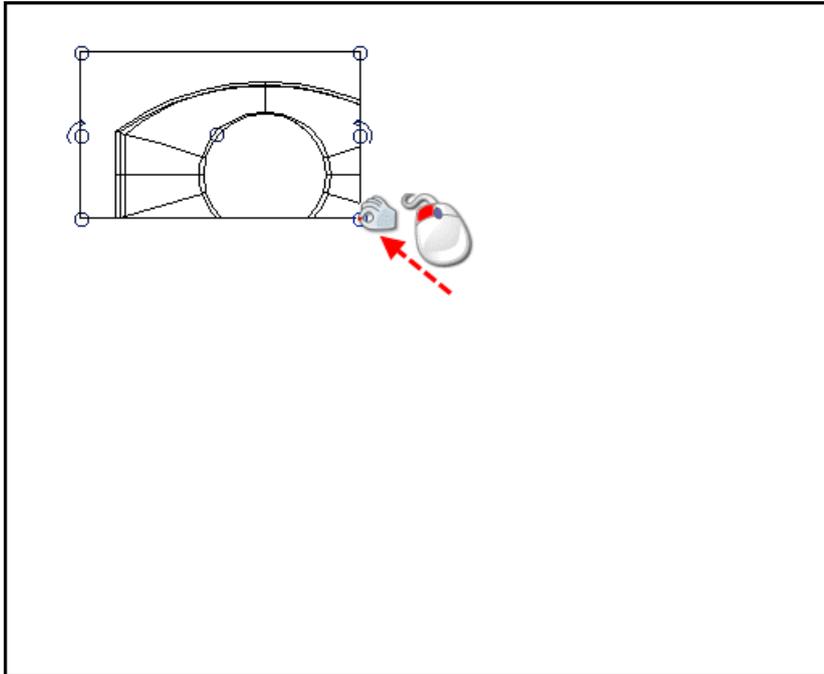
For further details, see Using the Intelligent Cursor to enter positions.

Corner handle

Each corner handle is used to resize the view. When you move the handle, the two sides attached to the handle resize.

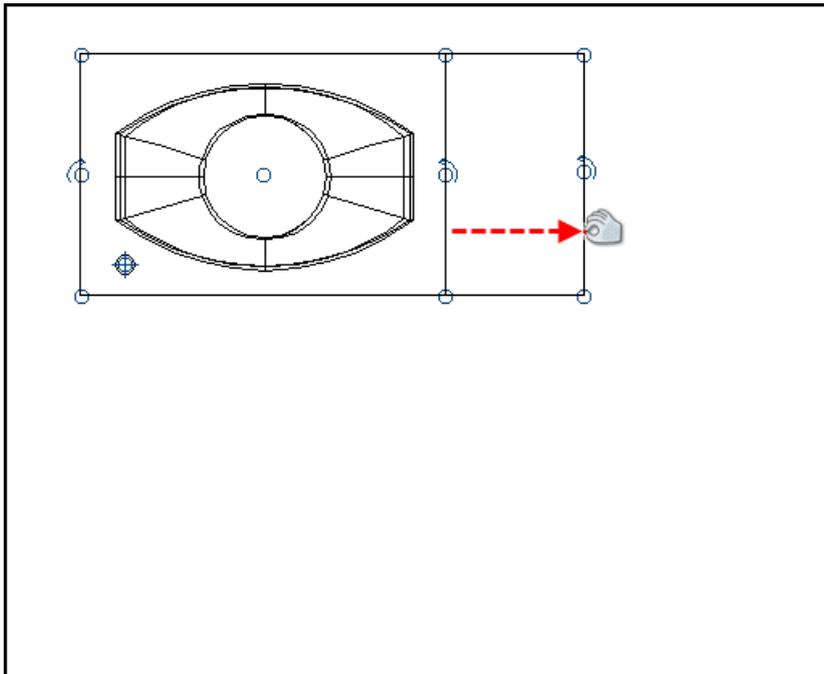


When you resize the view, you see more or less of the model. The image below shows what you see if you shrink a view. The view of the model is restricted to the boundary of the view.



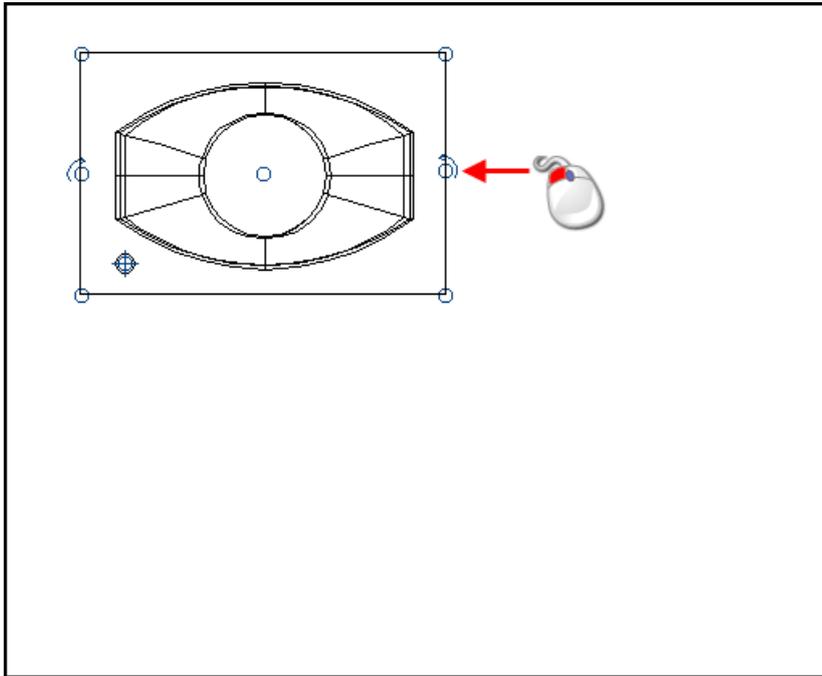
Side handle

Each side handle also resizes the view. This time only the side of the view with the handle is resized.

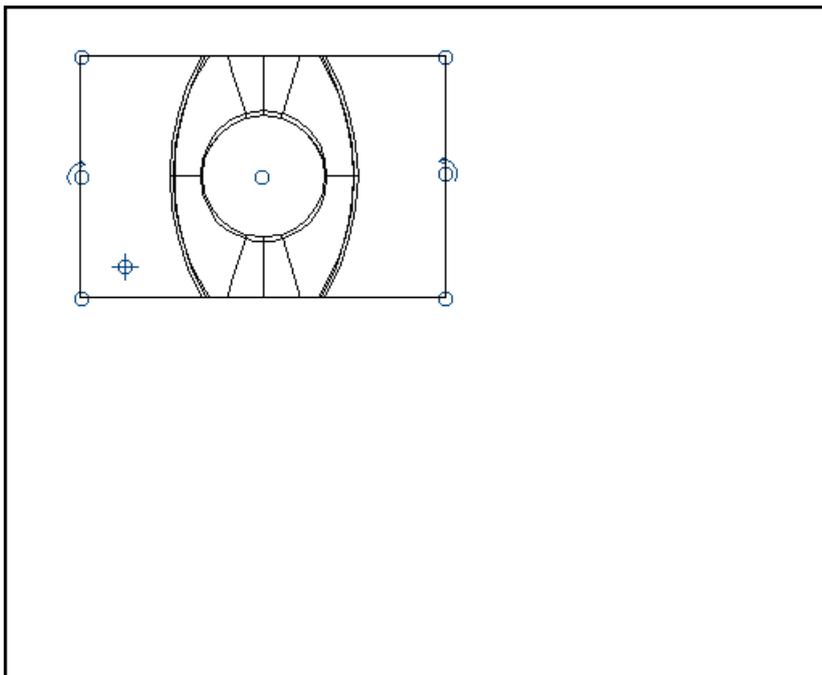


Rotate handle

Click the right rotate handle to rotate the view 90° anti-clockwise and the left rotate handle to rotate it 90° clockwise.



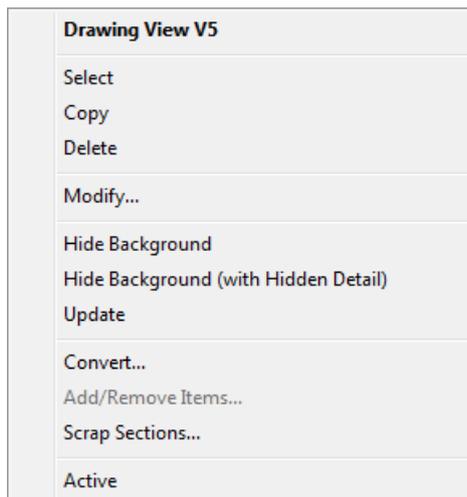
The model rotates within the view.



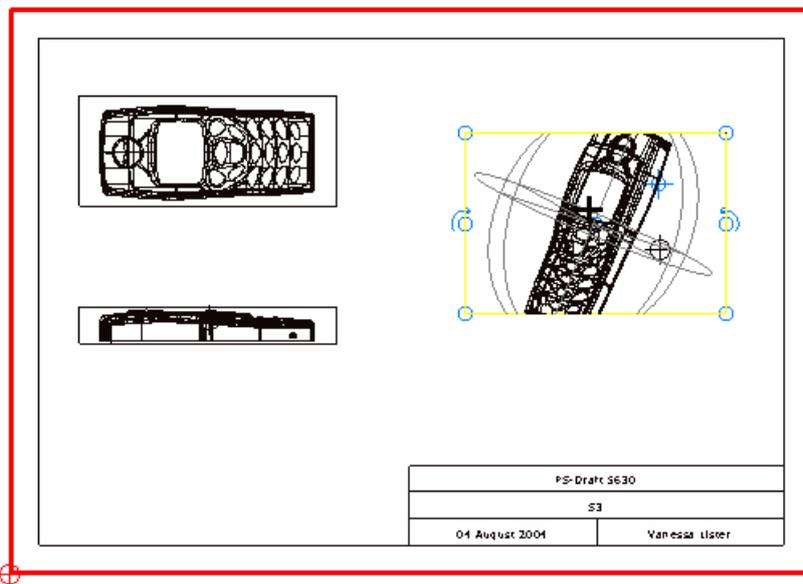
Manipulating the model in a view

You can manipulate your model in a view by turning and rotating it. To manipulate the model in a view:

- 1 Right-click the view to display the edit options.



- 2 Select **Modify** to display the Edit View dialog (see page 104).
- 3 Move the **Edit View** dialog out of the way of the view, if necessary.
- 4 Press and hold down the middle mouse button on the view and use the mouse to rotate the model in model space.



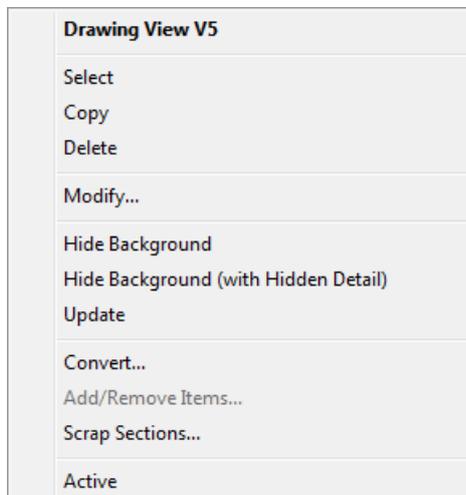
When you rotate a view in a drawing, the border of the view also rotates with the model to maintain the appropriate orientation with the view. For example, if you rotate a view from horizontal to vertical, the border also rotates to vertical eliminating the need to resize the borders.

- 5 Click **OK** on the **Edit View** dialog to save the changes.

Editing a view

There are various ways to edit a view.

- Select the view (see page 103).
Use the graphic handles (see page 95) to edit the view. The cursor changes to a hand when you move the mouse over view instrumentation.
- Select an editing option from the **Edit** menu:
 - **Edit > Modify** displays the Edit View dialog (see page 104).
 - **Edit > Convert > To Wireframe** displays the Convert View to Wireframe dialog (see page 102), which is used to convert the view into wireframe.
- Double-click the view to display the Edit View dialog. (see page 104)
- Right-click the view to display the edit options available for views. You can be in normal selection mode to right-click the view. When you select an option, the selection mode changes to drawing.



Select — Select the current view.

Copy — Copy the current view.

Delete — Delete the current view.

Modify — Display the Edit View dialog. (see page 104) When you select **Modify**, the ability to manipulate (see page 99) the model in the view is activated.

Hide background — Remove hidden lines from the view. Hidden lines are lines that are not seen on the real-life model from the current view. Background detail is also hidden in section views. If there are no items to hide, an error message is displayed. This option is an alternative to setting the view to **Hidden** from the **Edit View** dialog (see page 104).

Hide Background (with Hidden Detail) — This option does the same as **Hide**, except that hidden lines are displayed as dashed lines.

Convert... — Display the **Convert View to Wireframe** dialog (see page 102) to convert a view to wireframe.

Add/Remove Items... — Display the Add/Remove Items dialog (see page 109) to add selected items to the view, or to remove selected items from the view.

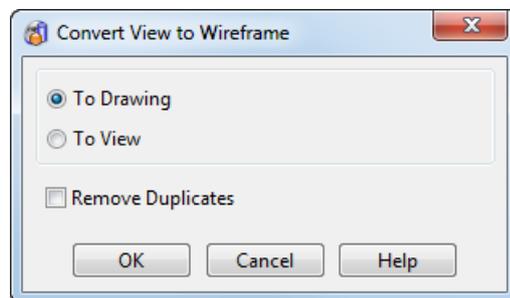
Active — Activate or de-activate the current view.

Convert — Displays the **Convert View to Wireframe** dialog (see page 102).

Convert view to wireframe dialog

This dialog is used to convert the contents of the selected view into wireframe objects (lines, arcs and curves). For further details, see Editing a view (see page 100).

- 1 Right-click the view.
- 2 Select **Convert...** to display the **Convert View to Wireframe** dialog.



To Drawing — This option converts the contents of the view into wireframe objects on the drawing.

To View — This option also converts the contents of the view into wireframe objects, but the objects remain in the 3D model and not on the drawing sheet. To edit the wireframe objects, you must activate the view first. For further details, see *Activating a view* (see page 93).

Remove Duplicates — Select or deselect to control how duplicates are removed when a view is converted. The default setting for this option defined on the Views Options dialog > General page (see page 83)

OK — Convert the view to wireframe and close the dialog.

Cancel — Ignore the command and close the dialog.

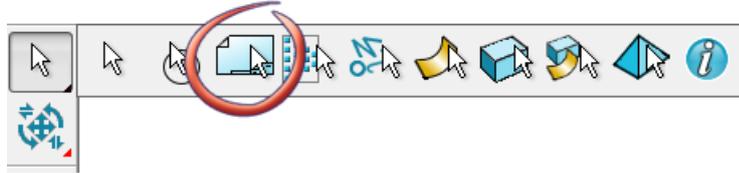


To prevent overlapping centreline patterns obscuring each other, dependent centreline geometry is merged before a view is converted.

Selecting a view

Select a view in one of the following ways:

- Click  and click  (*Selection flyout*).



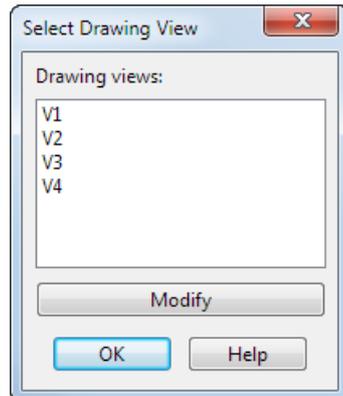
The Select Drawing View dialog (see page 103) is displayed. Select the view.

- Right-click the view and click **Select** from the context menu.
- When selecting multiple drawing views, you can use box selection to select the required views.

To deselect drawing views from the total selection, use **Ctrl+Shift** and click on the view.

Select Drawing View dialog

Use this dialog to select a view and display the Edit View dialog (see page 104).



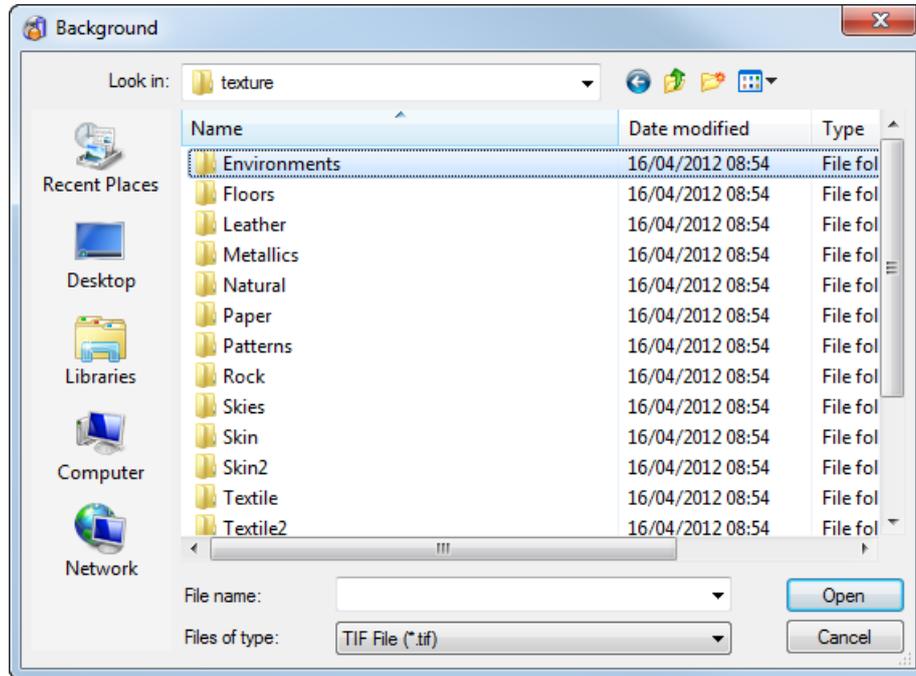
Drawing views — This contains a list of names of views in the drawing. Click a name to select the view.

Modify — Display the **Edit View** dialog to edit the selected view. For further details, see Edit View dialog (see page 104).

OK — Select the view and close the dialog.

Background dialog

Use this dialog to locate and select a background image to be displayed in a drawing view. It is displayed when you select  on the View Background dialog (see page 110).

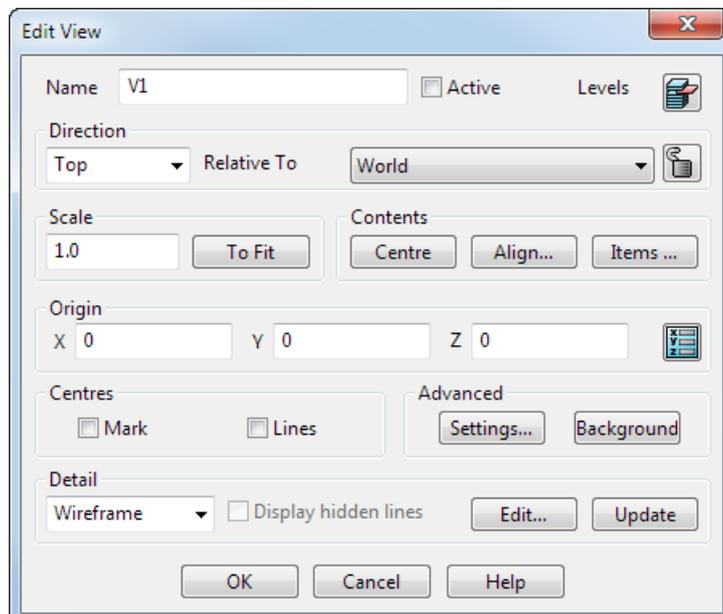


Look in — Use the drop-down list to locate the image you want to be displayed as a background in a drawing view. The view can be one of the following file types: PDF, PNG, TIF, BMP.

Open — Open the selected file.

Edit View dialog

Use this dialog to edit the selected view.

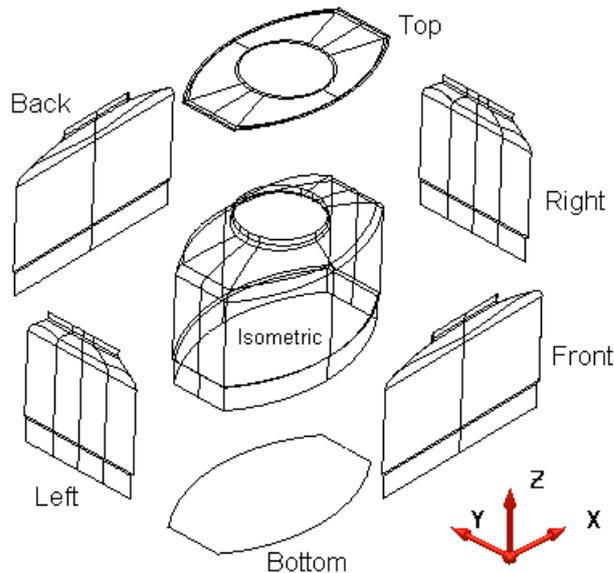


Name — Displays the name of the view. You can change it to something more meaningful.

Active — Activate the selected view (see page 93).

 — Click this button to display the Level dialog. You use this dialog to turn levels *ON/OFF* in the view.

Direction — Use this option to specify the direction of the view of the model relative to the workspace defined by **Relative To**. You can change the direction of the view. Each direction is shown below.



Relative To — This defines the workspace used by the **Origin** and the **Direction**. You can change this to **World** or any workplane defined on the model.

 — Click this button to maintain the position of balloons in relation to the view. When a view is locked :

- the direction and workplane cannot be changed on the dialog
- the view cannot be dynamically rotated.

Scale — This changes the scale at which objects are displayed in the view. If you enter a value to scale the drawing view when a background image is used, the background image will also scale.

To Fit — Scale the contents of the view to fit the view boundaries.

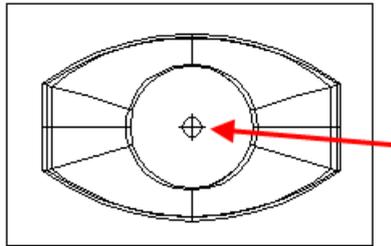
Centre — Move the contents of the view to the centre of the view.

Align — This allows you to align the selected view with another view with respect to the world or view workspace. See *Aligning two views* (see page 115) for further details.

Items — This displays the Add/Remove Items dialog (see page 109) to add or remove items from views.

Origin — Use this option to specify the origin of the view relative to the workspace defined by **Relative To**. By default, the view origin is placed at the position of the origin of the current model. You can edit the origin so that positions are entered relative to the origin when the view is active.

The origin of the view is marked by a circle and two lines which cross.



The origin is also used to align views. For further details, see [Aligning two views](#) (see page 115).

If you cannot see the view origin when you create a view, the origin of the world workspace lies outside the boundary of your view and so does the view origin.

To change the position of the view origin

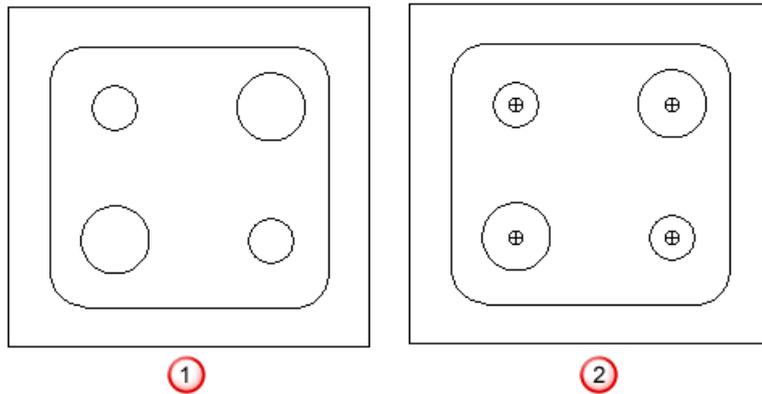
Enter the **X Y Z** coordinates or click the **Position**  button to open the **Position** dialog where you can use position entry tools.

To enter a new position for the view

- 1 Click  to display the Position dialog
- 2 Click a position for the view while the **Position** dialog is displayed.
- 3 Click **OK** on the **Position** dialog.

Centres — Use this option to specify whether to mark the centres of cylindrical surfaces.

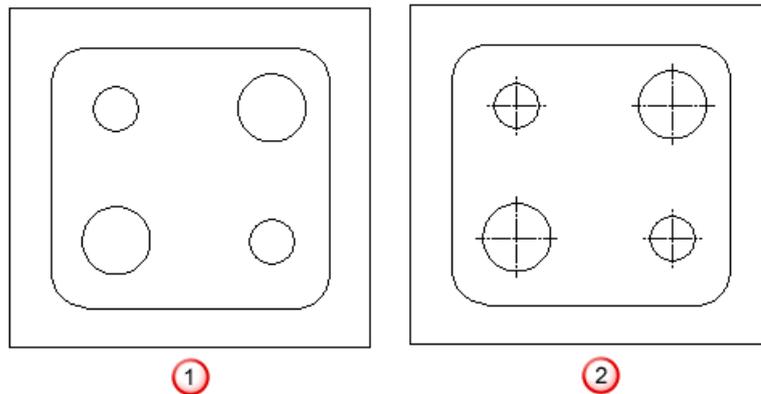
- **Mark** — If selected, this option marks the centres of cylindrical surfaces. If you move the cylindrical surfaces, the centre marks do not move.



① Mark *OFF*.

② Mark *ON*.

- **Lines** — If selected, centre lines are drawn on the following:
 - cylindrical surfaces
 - circles
 - solid cut features
 - solid boss features
 - solid bulge features

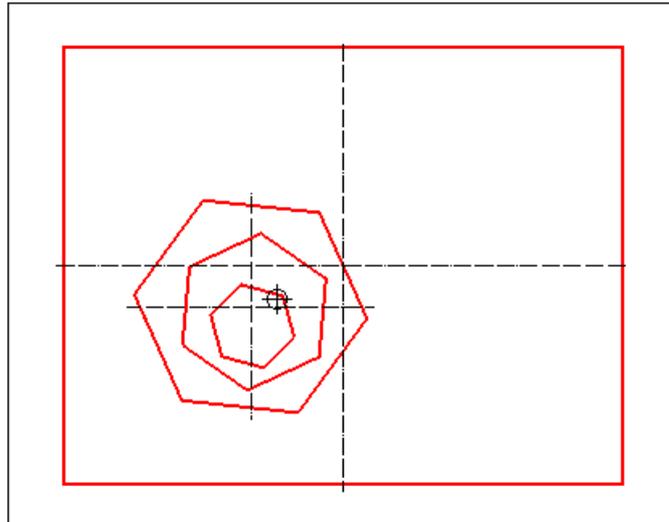


① Lines *OFF*.

② Lines *ON*.

If you move the cylindrical surfaces, the centre lines do not move.

When you mark a centreline on a solid in a drawing view using the **Lines** option, the centreline becomes dependent on the solid. Therefore, if the solid is resized, the centreline adjusts accordingly.



To prevent overlapping centreline patterns obscuring each other, dependent centreline geometry is merged before a view is converted.

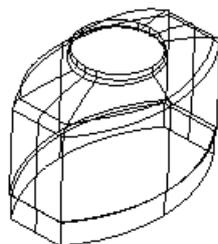
Background — Display the **View Background** dialog. Use this dialog to:

- select a background for the view
- change the magnification of the image
- display specific pages of PDF files.

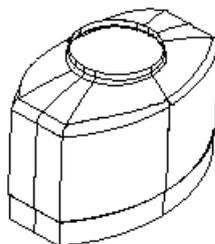
For further details, see View Flags dialog (see page 111).

Detail selector — This determines how the objects are displayed in the view. The menu contains a selection of hidden, wireframe, shaded and transparent view options.

Hidden — The following three options are only available if the view is a hidden line view

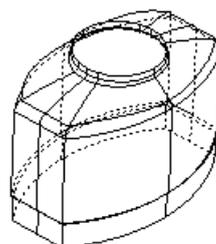


Wireframe view



Hidden view

Display Hidden
Lines off



Hidden view

Display Hidden
Lines on

Display hidden lines — If this option is *OFF*, then the hidden lines are not drawn. If *ON*, they are drawn using dashed lines.

Hidden lines are lines that are not seen on the real-life model from the current view.

Arc centre points of solid features and centre points for hole patterns are drawn in hidden views. If there are no items to hide, an error message is displayed.

A warning cross is displayed across hidden and section views when they need to be updated because there has been a change to the model. The drawing of crosses is on by default and can be turned off by typing the following in the command window:

view_options mark_updated off

Edit — This displays the Edit Hidden View dialog (see page 112), that is used to edit hidden and visible lines.

Update — Click to update the contents of a hidden or section view.



When a section view is updated, any hatch styles are maintained.

A red cross is drawn across a view when it is out of date and requires updating. For further details, see Out of date views (see page 95).

Flags — This displays the View Flags dialog (see page 111) to change various settings for the view.



We suggest you create a solid of your model where possible before drafting from it. This will improve the quality and speed of hidden line views. For further details, see Creating a solid.



Solid features in hidden views are highlighted when the cursor is moved over them.

OK — Store the changes made to the view and closes the dialog.

Cancel — Discards any changes made to the view whilst the dialog was displayed and closes the dialog.

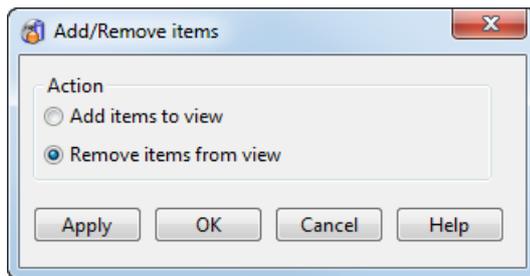
Add/Remove Items dialog

Use this dialog to add or remove items in a drawing view to control the content of a view.

To add/remove items

- 1 Select the **Items** button on the **Edit View** dialog.

- 2 Use the **Add/Remove** dialog to add or remove items.



Add items to view — Select this option to add items to the view that have previously been removed.

Remove items from view — Select items in the view then select this option to remove the items from the view. The items are flagged once they are removed so they can be added back in if required. Select the **Add items to view** option to add the items back to the view.

If the view is a hidden view or contains hidden detail, the display changes to wireframe so the items can be seen in order to be selected.

Apply — Select to apply the changes to the view.

OK — Apply the changes to the view and close the dialog.

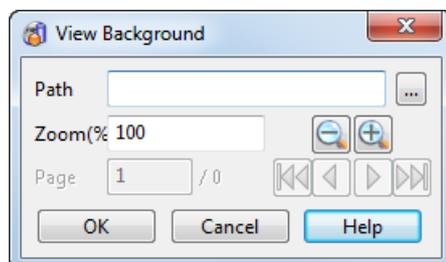


*When you click **OK**, A dialog is displayed to prompt you to select **Update** to update the view with the changes.*

- 3 Select **Update** on the **Edit View** dialog to update the view.

View Background dialog

Use this dialog to select a background to the view, change the magnification of the image, and display pages of a PDF file.



Path — Displays the path for the view filename selected in the **Background** dialog. The view can be one of the following file types: PDF, PNG, TIF, BMP.



PDF files will only work if the full version of Adobe Acrobat is installed and not the free reader.

You should ensure that the Updates Preference option for Acrobat is set to Manually.

 — Click to display the Background dialog (see page 104). You use this dialog to locate the file you require.

Zoom(%) — Enter the percentage of zoom you want the background image to be and press **Enter**.

 — Click to zoom into the background image.

 — Click to zoom out of the background image.

Page — Enter the page number of the PDF file you want to display, and press **Enter**.

 — Display the first page of the PDF file.

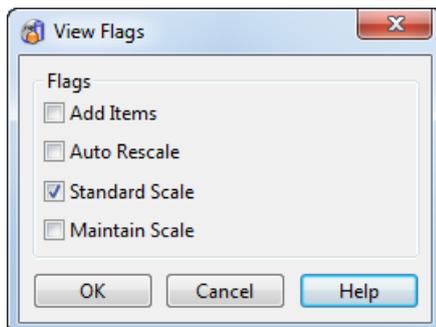
 — Display the previous page of the PDF file.

 — Display the next page of the PDF file.

 — Display the last page of the PDF file.

View Flags dialog

Use this dialog to change various settings for the view.



Add Items — If selected, all new objects created in the model are added to the view.

Auto Rescale — If selected, the contents of the view are scaled to fit the view boundaries when a closed drawing is opened.

Standard Scale — If selected, the contents of the view use a standard scale (such as 1.0 2.0 5.0) to fit the view. If deselected, the contents can use the largest scale to fit the view. This allows the contents to scale to a larger size when you click the **To Fit** button on the Edit View dialog. (see page 104)

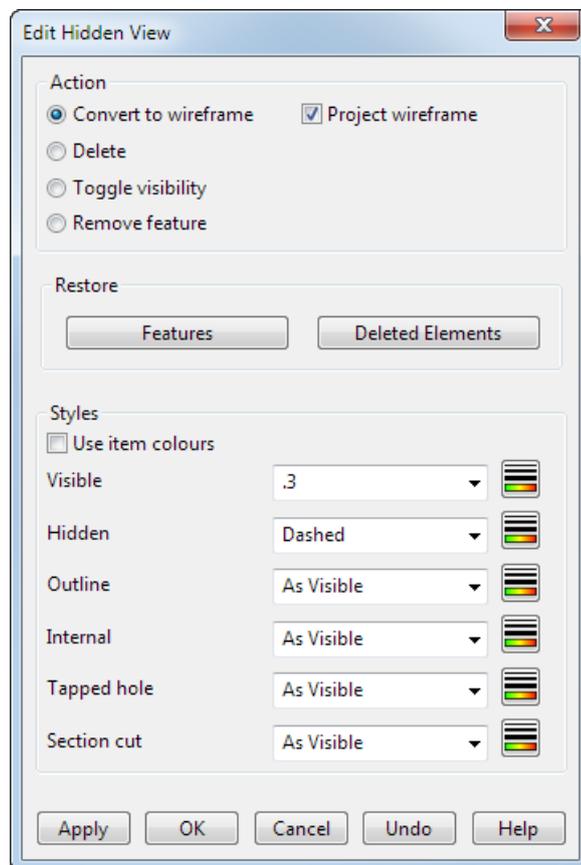
Maintain Scale — If selected, the scale of a view is maintained when using drawing templates. You specify the scale for a view in the Edit View dialog. (see page 104)

OK — Changes the settings for the view and closes the dialog.

Cancel — Discards any changes and closes the dialog.

Edit Hidden View dialog

Use this dialog to edit hidden and visible lines. You can also rotate a model dynamically in the selected view using the middle mouse button when this dialog is displayed. For further details, see [Manipulating the model in a view](#) (see page 99).



Action

This contains operations you can do on selected hidden and visible lines from the view. Lines are selected in the usual way.

Convert to wireframe — Converts the selected lines into wireframe.

Delete — Deletes the selected lines from the view.

Toggle visibility — Toggles the styles of the selected lines from hidden to visible or visible to hidden.

Remove feature — Removes solid features permanently from the selected hidden view.

Project wireframe — When you convert the selected lines to wireframe, this option projects the wireframe onto the view plane.

Restore

Features — Redisplays any features that were hidden in the view due to selecting the **Hidden** detail option on the **Edit View** dialog.

Deleted Elements — Select this option to restore items in a hidden view that have been previously deleted.

Styles

This sets the style of the visible and hidden lines.

Use item colours — If selected, the lines are the same colour as the wireframe of the item.

Visible — Select the style of the visible lines from the list.

Hidden — Select the style of the hidden lines from the list.

Outline — Select a style to be applied to outside edges detected in a hidden view. The default option is **As Visible**.

Internal — Select the style of the internal edges from the list. An internal edge is an edge that lies between faces that are tangent continuous.

Tapped hole — Select the style for tapped holes from the list.



Tapped hole detail is dependent on the hole feature.

Section cut — Select the style for section cuts from the list.



Set Styles — Click to display the style toolbar where you can customise a style for hidden views.

Apply — Applies the changes to lines in the view.

OK — Applies the changes to lines in the view and closes the dialog.

Cancel — Closes the dialog.

Undo — This undoes the changes to the view back to when you last clicked the **Apply** button.

Deleting hidden detail

Hidden detail can be deleted so that it does not reappear when the view is regenerated. When you delete hidden detail, select the **Update** button on the **Edit View** dialog (see page 104) to regenerate the view.

The deleted geometry can be restored by selecting the **Deleted Elements** button on the **Edit Hidden View** dialog (see page 112).

Editing Multiple Views

You can select multiple views and edit them simultaneously using the **Edit Multiple Views** dialog.



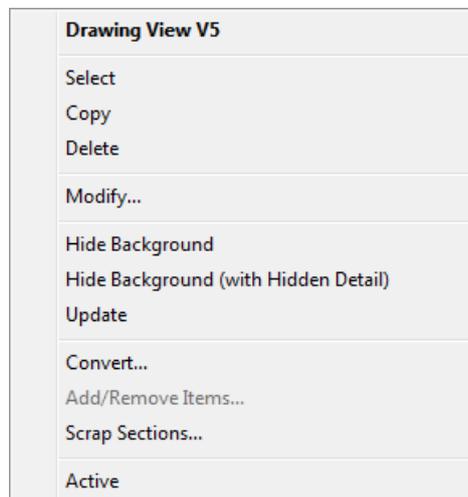
You can use box selection to select the required views. To deselect drawing views from the total selection, use **Ctrl+Shift+left-click** on the view.

You can use the following editing actions on a multiple selection of views:

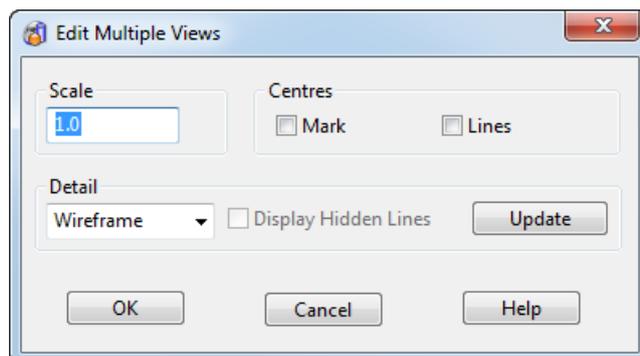
- Delete
- Convert
- Move
- Rescale
- Set centrelines and centre marks
- Set view detail
- Update the view to display the edits.

To edit a multiple selection of views

- 1 Right-click the view to display the edit options available for views.



- 2 Select **Modify** to display the **Edit Multiple Views** dialog that contains view editing options from the **Edit View** dialog.



Scale — Enter a value to change the scale that objects are displayed in the selection of views.

Centres

Mark — Select to mark the centres of cylindrical surfaces.

Lines — Select to draw centre lines on cylindrical surfaces and circles.

Detail — Select an option to determine how objects are displayed in the selected views.

Display Hidden Lines — When **Hidden** is selected from the **Detail** drop-down list, you can select this option to display hidden lines in the views.

Update — Select this option to update the display of views with any changes made.

Aligning two views

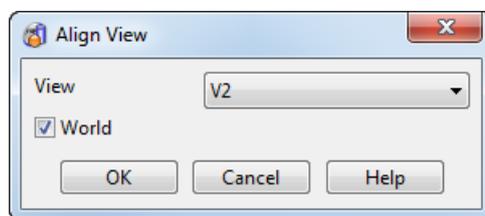
Views are aligned by aligning origins within two views. This can be used to accurately align objects within a view to their corresponding objects in another view.

This feature is useful if you import models which contain 2D drawings, particularly from **Delcam Toolmaker**.

- 1 Right-click the view and select **Modify** to display the Edit View dialog (see page 104).
- 2 From the dialog, click **Align** to display the **Align View** dialog (see page 115)

Align View dialog

Use this dialog to accurately align objects within two views on a drawing.



- 1 Select a **View** to be aligned. Alternatively, select a view on the drawing by clicking it.
- 2 Select **World** to align the world origins of the two views. Deselect this option to align the view origins.

By default, the view and world origins are in the same position. If the world origin is not suitable for aligning the views, you can move the view origins of the two views to somewhere more suitable. Use the **Edit View** dialog (see page 104) to change the origins of the two views.

If you have several views to align, move the view origin to a position common to the views.

- 3 Click **OK** on the **Align View** dialog to see the two views align.

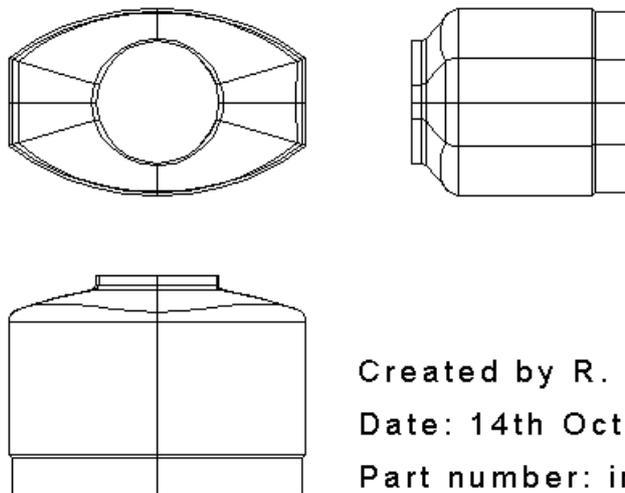
The view does not move, just the contents of view. The contents of the modified view may look blank.

- 4 If necessary, edit the borders of view to give the required view. For further details on editing the borders of the view, see Graphically editing views (see page 95).

Annotation

Annotation is text added to your drawing. Use annotation to give details about the model, such as when it was created and who created it.

Ink bottle



For further details, see:

Annotation toolbar (see page 116)

Editing text (see page 126)

Inserting strings from the File Properties (see page 145)

Annotation toolbar

The **Annotation** toolbar combines the following tools and options,

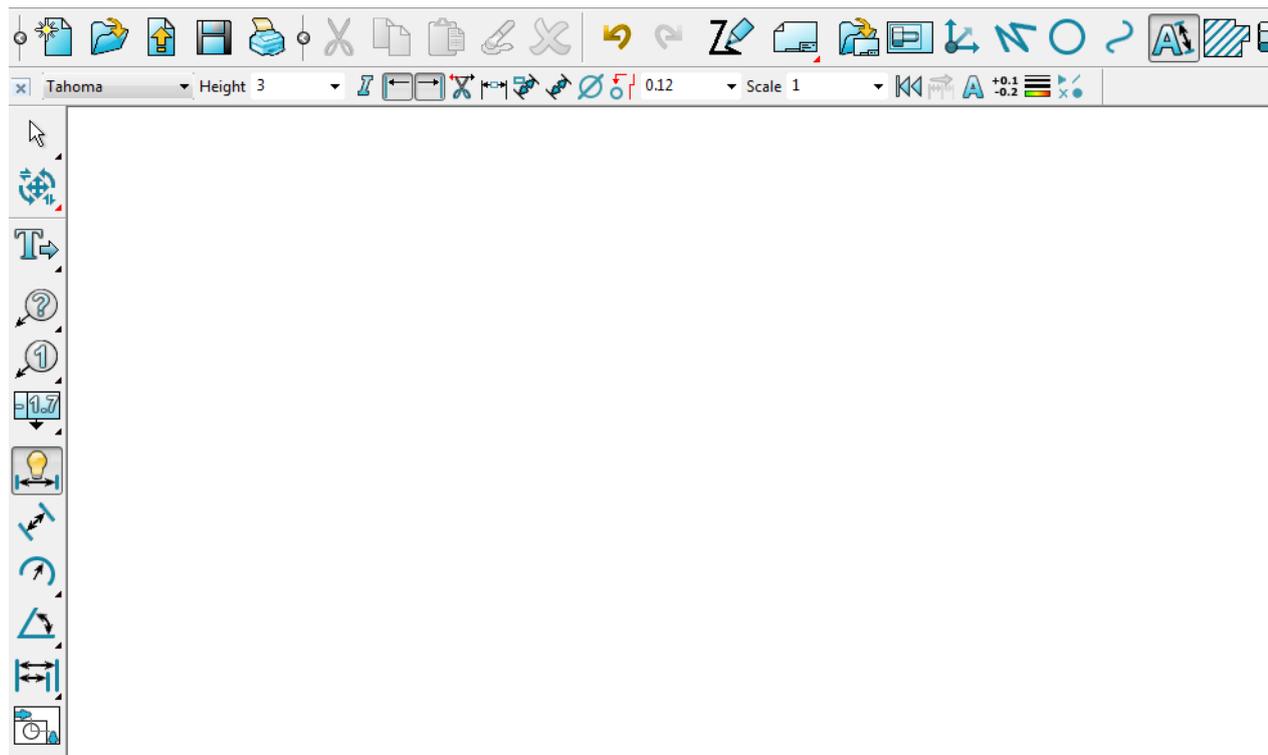
- ① Text
- ② Drawing
- ③ Dimensions



To display the **Annotation** toolbar, click the **Annotation**  button on the main toolbar.



The **Text Editor** toolbar displays automatically as the **Annotation** toolbar is displayed.



The **Annotation** toolbar is also available when in Modelling mode but some drafting options, such as symbols and hatching, remain unavailable outside Drafting mode.

Entering annotation creation mode

- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116). This is used to edit attributes of text, for example, the font. The **Text Editor** toolbar (see page 129) displays automatically when the **Annotation** toolbar is displayed.

The text creation options are shown in ① below on the **Annotation** toolbar (see page 116).



- 3 Click  (*Annotation toolbar*) to open the **Text creation** flyout and display the text options.

 Create horizontal text

 Create vertical text

 Create text on a curve

 *Text using the Duct font is created differently to text using the other fonts. For further details, see [Creating Duct text](#) (see page 122) and [Editing Duct text](#) (see page 144)*

 *Multi-byte languages such as Japanese and Korean create and edit all text fonts in the same way as the Duct font.*

Creating a block of text

- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).

- 3 Click  (*Annotation toolbar*) to display the text options.
- 4 Choose one of the options in the **Text creation** flyout. The cursor changes to show that you are in text creation mode.

You can change the default style of the text using the **Text Editor** toolbar (see page 129).

If you create vertical text, the individual characters are horizontal.

D
e
l
c
a
m

You can switch between horizontal and vertical text at any time while creating text using the **Horizontal**  button and the

Vertical  button from the **Text creation** menu. You need to

select a curve in order to use **Text along a curve** .

You can angle (see page 135) the font.

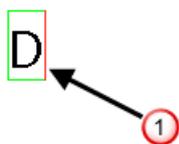
Text using the Duct font is created differently to text using the other fonts. For further details, see Creating Duct text (see page 122).

- 5 Click a position in the graphics window where you want the text to lie. This is the text position.

An insertion point appears in the shape of a bar cursor. This is the position where PowerSHAPE will create the text. If you have selected vertical text, then the insertion point is horizontal.

- 6 Type in the text.

As you enter the first character, it is displayed on the screen and a rubber-banded box starts to form around the text. The insertion point moves to the end of the character .



As you enter more characters, the rubber-banded box enlarges to enclose the new characters.

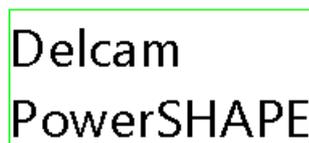


To create a new line of text, press **Enter**. The rubber-banded box extends to enclose the second line too and the insertion point moves to the start of the next line.



The bottom left corner of the rubber-banded box remains at the position you enter in Step 5.

Continue typing.



When the rubber-banded box is displayed around the text block, you are in *text insertion mode* and you can type in new text.

At any time during the creation, you can change the style of the text using the **Text Editor** toolbar (see page 129).

You can also change the default settings for text using the **Options** dialog. For further details, see Changing the default settings for text (see page 123).

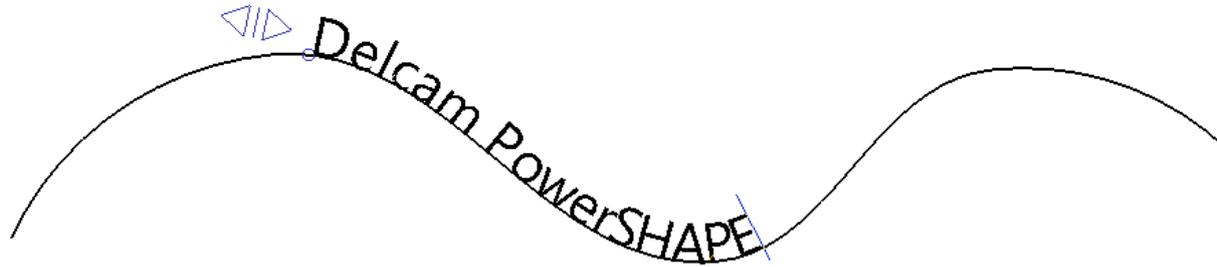
To exit text insertion mode, click outside of the text block. You can now edit the whole text block. For further details, see Editing text (see page 126). If you click again, you can start creating a new block of text.

To exit text creation mode, click **Select** .

Creating text on a curve

- 1 Click  to display the **Annotation** toolbar.
- 2 Click  (*Annotation toolbar*) to display the text options.

- 3 Click **Text along a curve**  on the text flyout. The cursor changes to .
- 4 Click a point along the curve to add text.



Text can be added to any arc or curve. When you have created the text, edit the text on the curve as required (see page 141).



DUCT text cannot be used when creating text on a curve.

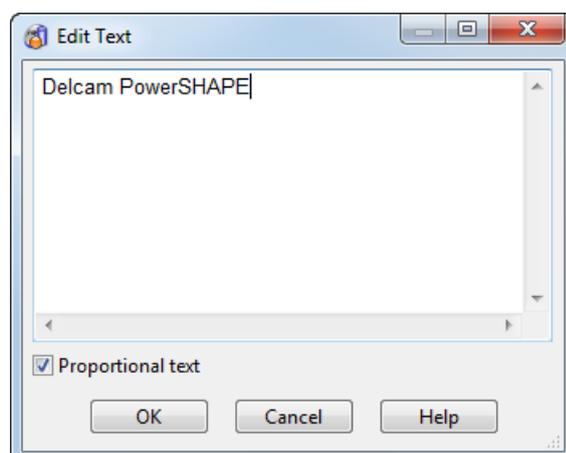
Creating DUCT text

- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Select **Duct** from the **Font** selector on the **Text Editor** toolbar.

Most tools on the *Text Editor toolbar* (see page 129) are available to use on the text.

- 4 Click a position on the screen.

This displays the **Edit Text** dialog.



- 5 Type the text in the dialog.

You can make the text fixed-width or proportionally spaced.

- 6 Click **OK** or click on the screen.

The text is drawn on the screen.

Delcam PowerSHAPE



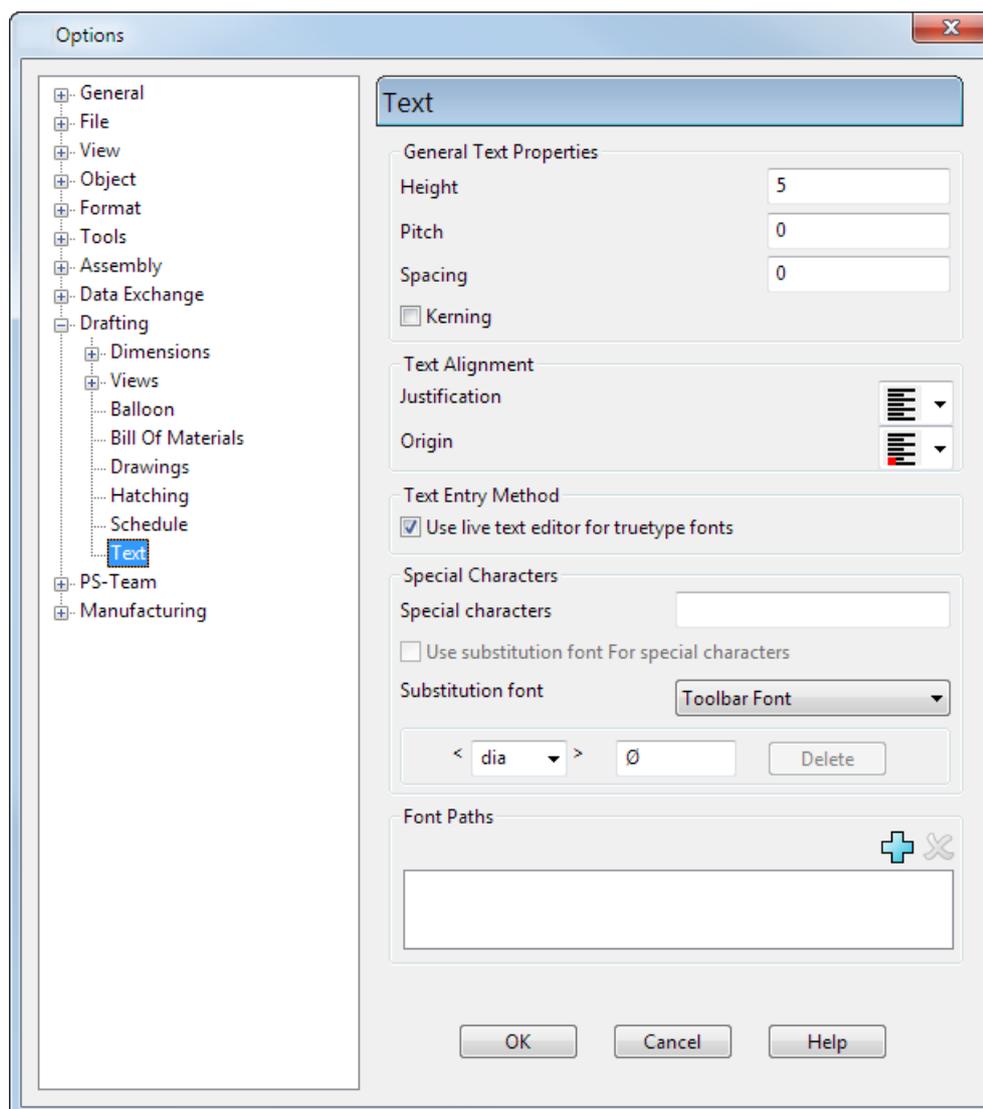
DUCT text cannot be wrapped on a curve.

Changing the default settings for text

You can change the default settings of future text using the **Options** dialog.

- 1 From the **Tools** menu, select **Options...** to display the **Options** dialog.
- 2 From the **Drafting** branch of the browser tree, select the **Text** option. This displays the **Text** page of the **Options** dialog.

3 Use this dialog to set the default text options.



General Text Properties — Use these options to modify the properties of the characters.

Height — Enter the height of the largest character.

Pitch — Enter the extra spacing between two characters. It is calculated as a multiple of the height of the text.

Spacing — Enter the spacing between two lines of text in the same block. It is also specified as a multiple of the height of the text. Use the text box to change the spacing.

Kerning — This adjusts the spacing between particular pairs of letters depending on the font design.

Text Alignment — Modify the justification and origin point of the text.

Justification — This is how the text in the block aligns with the left and right margins. Use this option menu to change the justification of the text.

Origin — This is the origin of the text block. The origin is the 'anchor point' used to position the text and can be set at any of the four corners of the text box, the mid points of the sides of the text box or the centre of the text box. Use the option menu to change the origin of the text.

Use live text editor for truetype fonts — Select this option to create and edit text using the text insertion mode. The rubber-banded box is displayed around the text to indicate you are in text insertion mode. Deselect this option to create (see page 122) and edit (see page 144) all text fonts in the same way as the Duct font using the **Edit Text** dialog.

Special characters — Enter or paste special characters (see page 136) from the **Unicode Character Map** dialog. They are stored in the model and are available in the **Character** list on the **Text Editor** toolbar (see page 129).

Use substitution font For special characters — If *ON*, the special characters in the list on the **Text** toolbar will use the font defined in the **Substitution Font** menu.

Substitution font — Select the font used by substituted strings. If you select **Toolbar Font**, the font on the toolbar is used.



— These define the substitution strings. The box enclosed in < > contains the string you type in and the box next to it contains the replacement string.

Three standard substitution strings are provided:

- **<dia>** — diameter symbol
- **<deg>** — degree symbol
- **<p/m>** — plus/minus symbol

These standard strings are also used by balloons and dimensions.

When you type **<dia>** in PowerSHAPE and then deselect the text, you can see the diameter symbol. If you edit the text, the **<dia>** string is displayed again.

To define your own strings:

- 1 Type in the string you want to type in PowerSHAPE in the box enclosed in < >. A maximum of three characters is allowed.
- 2 In the box next to it, type in the replacement string.

Do not forget to enclose the string in < > when you type the text in the graphics window.

If you use a substitution string in your model and then later change the definition of its replacement string, all the strings in your model using that substitution string will be replaced too. This also applies to the standard strings.

Delete — This deletes the substitution string displayed in the box enclosed in < >. You cannot delete the standard substitution strings.

Font Paths — If you have created new fonts, using ArtCAM font creator, use the **Font paths** section to specify the location of font files.

- 1 Click  to display the **Browse for folder** dialog.
 - 2 Choose a font directory from the tree and click **OK**.
- OK** — Click to accept the changes.

Editing text

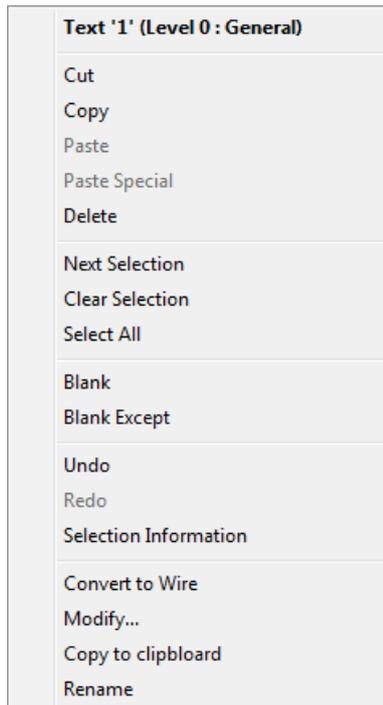
- 1 Click **Select**  to enter selection mode.
- 2 Select the text to display the **Text Editor** toolbar (see page 129). Use this toolbar to edit all the text within the text block.

When text is selected it can be dragged to a new location.

You can also use one of the following methods to edit the text:

- Select **Edit > Modify** to enter text insertion mode and display the *Text Editor toolbar* (see page 129). The rubber-banded box is displayed around the text to indicate you are in text insertion mode.
- Select **Edit > Convert > To Wireframe** to convert text to wireframe (see page 145).
- Double-click the text to enter text insertion mode and display the *Text Editor toolbar* (see page 129). The rubber-banded box is displayed around the text to indicate you are in text insertion mode.

- Right-click the text to display the edit options available for text. The name of the text and the level on which it lies is at the top of the context menu.



Convert to Wire is the only option unique to this context menu. This option converts the text to wireframe (see page 145).



You edit Duct text (see page 144) differently to text using the other fonts.

Moving the insertion point in a text block

- 1 Ensure you are in text insertion mode.

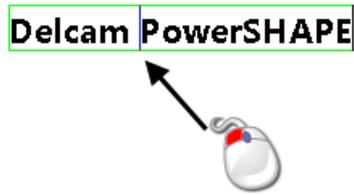
If you are not already in text insertion mode, double-click a block of text. You can see if you are in text insertion mode when the rubber-banded box is displayed around the text block.

- 2 Use the arrow keys or click the mouse in the text block to move the insertion point.

Selecting characters in text-insertion mode

- 1 Double-click the text block to enter text-insertion mode.

- 2 Left-click on the text to move the insertion point, either to the start or end of the subset of characters that you want to select.



- 3 Hold down the left mouse button on the insertion point and drag the cursor over the subset of characters. A box is drawn around the selected text.



You can select characters over several lines.

- 4 Edit the selected text using the **Text Editor toolbar** (see page 129)

Deleting a character

- 1 Double-click the text block to enter text-insertion mode.
- 2 Left-click on the text to move the insertion point in front of the character that you want to delete.
- 3 Press the **Backspace** key.



*If you use the **Delete** key, then the text in front of the insertion point is deleted.*

Cut, copy, and paste a string of characters

To cut or copy a string of characters into the Windows clipboard:

- 1 Double-click the text block to enter text-insertion mode.
- 2 *Select a string of characters (see page 127).*
- 3 From the **Edit** menu, select **Cut** or **Copy**.



*You can also use the keyboard shortcuts **Ctrl X** for cut and **Ctrl C** for copy.*

To paste the string from the Windows clipboard into any block of text:

- 1 Double-click the text block to enter text-insertion mode.
- 2 Move the cursor to where you want to insert the text.
- 3 From the **Edit** menu, select **Paste**.



You can also use the keyboard shortcut **Ctrl V** for paste.

Inserting a character

There are two insertion modes:

- Overtyping.
- Inserting.

Overtyping Mode

Each character you type replaces an existing character to the right of the insertion point.

Inserting Mode

When you enter new text, existing text is moved to the right.

- 1 Move the insertion point to where you want to insert the new text.
- 2 Press the **Insert** key.
- 3 Type in the new text.

According to the mode you selected, text is added to the block.

Text Editor toolbar

Select a block of text or enter text creation mode, to display the **Text Editor** toolbar. When creating balloons, a version of the toolbar is displayed that includes balloon options. All unrequired style items are suppressed when balloon options are displayed.



Use the toolbar to edit the style of text using the following options:

 The font used in the selected text is displayed in the font drop down list. PowerSHAPE supports a range of fonts from ArtCAM, including the wide character stick font, **(AFN) Korean_single_line**.

Height — Select the height of the largest character (see page 131).

 — Bold text (see page 131).

 — Italic text (see page 131).

 — Underline text (see page 132).

Pitch — Spacing between individual characters (see page 132).

Spacing — Spacing between lines in the selected block of text (see page 134).

Angle — Angle of the text to the horizontal (see page 135).



— Text justification.



— Text origin.



— Sets the selection filter to text (see page 135).



— Decreases the decimal places by (see page 136) 1.



— Increases the decimal places by 1 (see page 136).

Character — Add special characters (see page 136).



— Display Text substitutions (see page 139).

You can edit the style of the text when either a block of text is selected, or you are in text-insertion mode.



Most tools on the Text Editor toolbar are available to use on the Duct text. You can convert the text to PowerSHAPE text by changing the font.

If you are in PowerMILL modelling mode when you add text to a model, the text will be converted to wireframe when you return to PowerMILL.

Changing the font of the text

- 1 Select the text you wish to change.

The following text uses the **Arial** font:

Delcam PowerSHAPE

- 2 Select a **Font** from the list on the **Text Editor** toolbar. The font of the text is updated to reflect the new selection.

The drop-down list displays the scalable fonts available on your computer. On a Windows computer, these fonts are called TrueType fonts.

In this example, the font is changed to **Times New Roman**.

Delcam PowerSHAPE

It is now **Courier New**.

Delcam PowerSHAPE



You cannot use Duct text when creating text on a curve.

Changing the height of the text

- 1 Double-click the text block to enter text-insertion mode.

Delcam PowerSHAPE

- 2 Drag the mouse over the word **PowerSHAPE**.

Delcam **PowerSHAPE**

- 3 Use the **Height** list to enter or select a value. For example, change the value from 5 to 10.

Delcam **PowerSHAPE**

If you change the height of a whole block of text, the line spacing of the text block adjusts. If you change the height of just a few characters in a block of text, then the spacing of only the lines containing that text adjusts.



If you do not select any text, then any new text at the insertion point is created using the new height value.

Creating bold text

- 1 Select the text you want to change.

- 2 Click the **Bold**  button.

If your text does not change to bold, then the bold version of that font does not exist.

- 3 To change back to normal text, click the **Bold**  button again.



If you do not select any text, then any new text at the insertion point is created as bold.

Creating italic text

- 1 Select the text you want to change.

- 2 Click the **Italic**  button.

If your text does not change to italics, then the italic version of that font does not exist.

- 3 To change back to normal text, click the **Italic**  button again.



If you do not select any text, then any new text at the insertion point is created as italics.

Creating underlined text

- 1 Select the text you want to change.
- 2 Click the **Underline**  button.
- 3 To change back to normal text, simply click the **Underline**  button again.



If you do not select any text, then any new text at the insertion point is created as underlined.

Justifying text

You can choose whether the text is aligned to the left, right, or centre of the block. By default, lines of text in a block are left-justified.

To change the justification of the text:

- 1 Select the text block to display the **Text Editor** toolbar.
If you are in text insertion mode, then you do not need to do anything.
- 2 From the **Text Justification** drop-down list , select the appropriate option.

The following options are available:

-  — Left justification
-  — Centre justification
-  — Right justification

Changing the pitch

Pitch is the distance between each character in a line. It is a multiple of the character height.

- 1 Select the text you want to change.
- 2 Select or enter a new value in the **Pitch** list.

The pitch of the following text has been increased to **1** from **0**.
The characters move further apart.

Delcam PowerSHAPE



D e l c a m P o w e r S H A P E



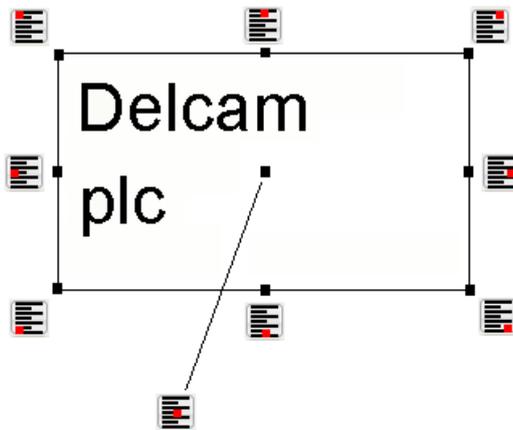
If you do not select any text, then any new text at the insertion point is created using the new pitch value.

Creating text from a different origin

Each block of text has an origin. This origin anchors the text block to the text position. The text is created relative to that anchor.

When you enter new lines of text, the text block moves such that the origin and text position always coincide.

There are nine origin positions:



Each button is described below.



— Bottom left



— Bottom centre



— Bottom right



— Centre left



— Centre



— Centre right



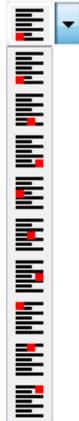
— Top left

 — Top centre

 — Top right

By default the origin is at the bottom-left of the text block. The effect of typing in new lines of text is that the text block moves up the screen.

- 1 Select a block of text. The origin of the text below is bottom left.
- 2 Choose an option from the **Origin** list.



The block of text is re-positioned to anchor the text position at the origin.

Changing the line spacing

Line spacing is the distance between two lines of text. It is a multiple of the character height.

- 1 Select the text you want to change.
- 2 Select or enter a new value in the **Spacing** list.

The spacing of the following text has been increased to **1** from **0**. The lines move further apart.

Delcam PowerSHAPE

Delcam PowerSHAPE

Delcam PowerSHAPE  Delcam PowerSHAPE

Delcam PowerSHAPE

Delcam PowerSHAPE



The line spacing adjusts to fit the character with the largest height in the text block.

Delcam PowerSHAPE

Delcam **Power**SHAPE

Delcam PowerSHAPE



If you do not select any text, then any new text is created using the new spacing value.

Creating angled text

You can create angled text. The angle is measured from the horizontal.

- 1 Select the text you want to change.
- 2 Enter or select an **Angle**.

The following text has been angled by **30**.

Delcam PowerSHAPE  Delcam PowerSHAPE



Text is drawn as an outline, but is displayed as filled when viewed parallel to the screen and of a reasonable size.

Setting the selection filter to text only

- 1 Select some text.
- 2 Press in the **Text filter**  button.

You can now select only text on the screen.

To exit the text selection mode, do one of the following:

- Click the **Text filter**  button if text is selected.
- From the selection option menu, click **Select** .
- Enter any creation mode.



When you add a schedule, the selection filter automatically sets to text only.

Changing the number of decimal places of text numbers

If the selected text in a schedule is a single number, you can either increase or decrease its number of decimal places by 1.

- 1 Select text that is a number.
- 2 Click either the **Decrease decimal places** button  or the **Increase decimal places**  button.

The number of decimal places decreases or increases by 1.



If you enter a number, it is not dependent on any geometry and so you cannot increase its number of decimal places.

Adding special characters to text

- 1 Double-click the text if you are not already in text-insertion mode.
- 2 Move the text insertion point to where you want to insert the special character.
- 3 Select a character from the **Character** list to insert into the text.
By default, the list contains diameter, degrees, and plus/minus characters.

You can insert the default special characters directly into the text using the following shortcuts.

∅	diameter	 +  + 
±	plus/minus	 +  + 
°	degrees	 +  + 

You can add new characters into the **Character** list by:

- typing the character into the **Character** list,
- pasting the character into the **Character** list (right-click in the list and select **Paste** from the menu),
- typing the Windows character value of the required character into the **Character** list.

The new character is also inserted in the text on the screen.

To store the characters in the model and add them to the **Character** list, insert the characters into the **Special Characters** box on the **Text** page (see page 123) of the **Options** dialog.

Using the Character Map dialog

You can insert special characters directly into the text on the screen. Hold down the **Alt** key and use the numeric keypad to type the Windows character value of the required character. This also inserts the character into the **Character** list.

To insert **Unicode** characters:

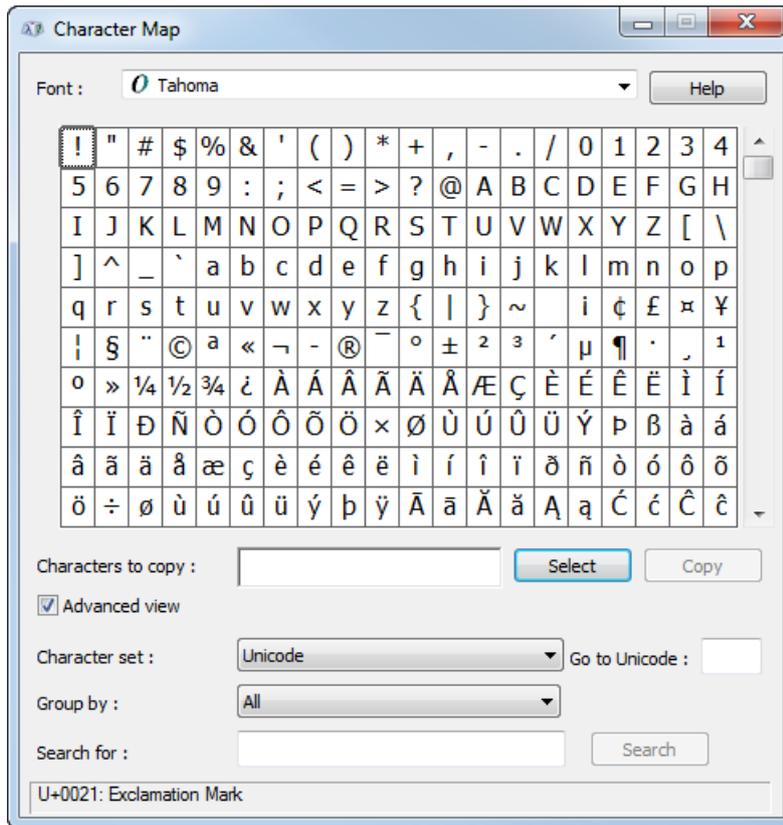
- 1 Double-click the text if you are not already in text-insertion mode.
- 2 Move the cursor to where you want to insert the special character.

In the example below, we want to insert the ÷ (divided by) character between the two words **height** and **width**.

height	width
--------	-------

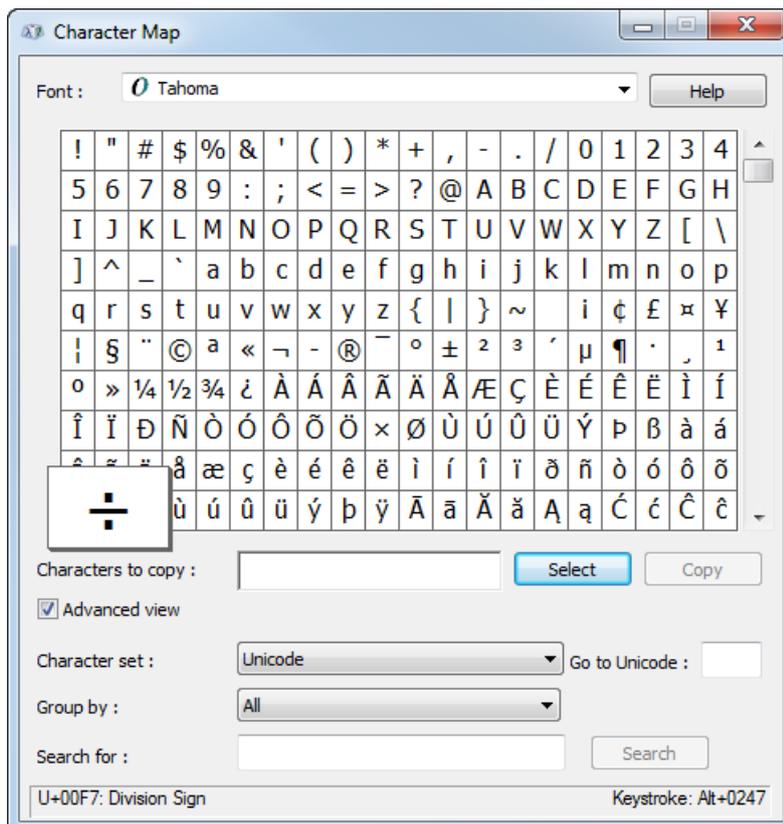
- 3 From the Windows **Start** menu, select **All Programs > Accessories > System Tools > Character Map** to display the **Character Map** dialog. The available Windows characters and their values are displayed in this dialog.
- 4 Select the **Advanced view** option to display all the options on the dialog.
- 5 From the **Character set** list, select **Unicode**.
- 6 Ensure the **Font** in the **Character Map** dialog is the same font used in PowerSHAPE.

The example below shows the PowerSHAPE default font **Tahoma**.



7 Click a character from the dialog.

The example below shows the ÷ (division) character.



In the bottom-right corner of the **Character Map** dialog, the **Keystroke** sequence is displayed, in this example **Alt+0247**.

- 8 In PowerSHAPE, hold down the **Alt** key and use the numeric keypad to type in the number code for the character. When you release the **Alt** key the character is displayed in the list and in the text.

In this example, hold down the  key and press   
.

height ÷ width

You can also type just the number directly into the **Character** list in PowerSHAPE.



Only Windows character values between 32 and 256 are valid.

Adding special characters using Copy and Paste

Another way to insert the character is by using copy and paste. To do this:

- 1 Click the character in the Windows **Character Map** dialog.
- 2 In the dialog, click the **Select** button. This displays the character in the **Characters to Copy** box.
- 3 Click the **Copy** button.
- 4 Go to PowerSHAPE and right-click in the **Character** list.
- 5 Select **Paste** from the context menu.

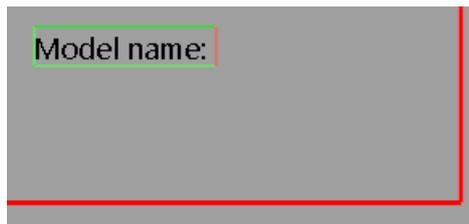


Only Windows character values between 32 and 256 are valid.

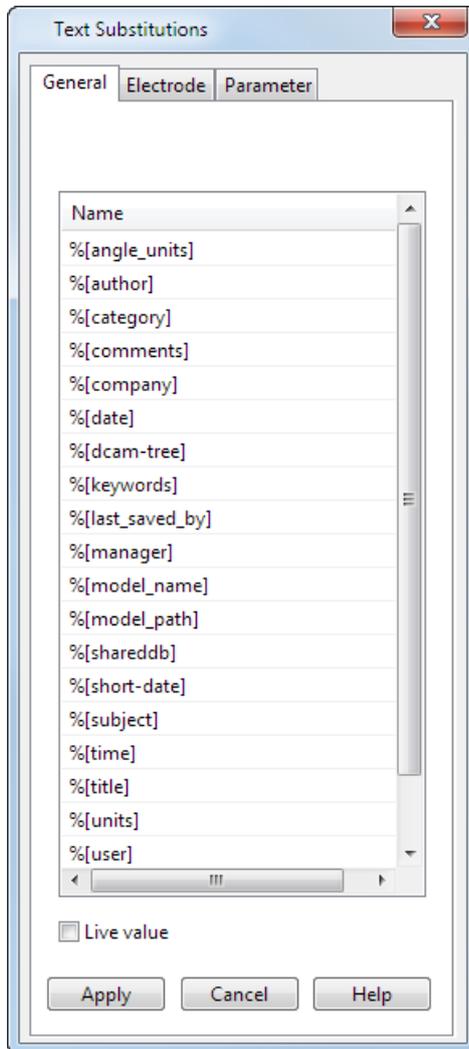
Text substitutions

- 1 Click  to display the annotation toolbar.
- 2 Click  and click a position on the drawing where the text will be positioned.

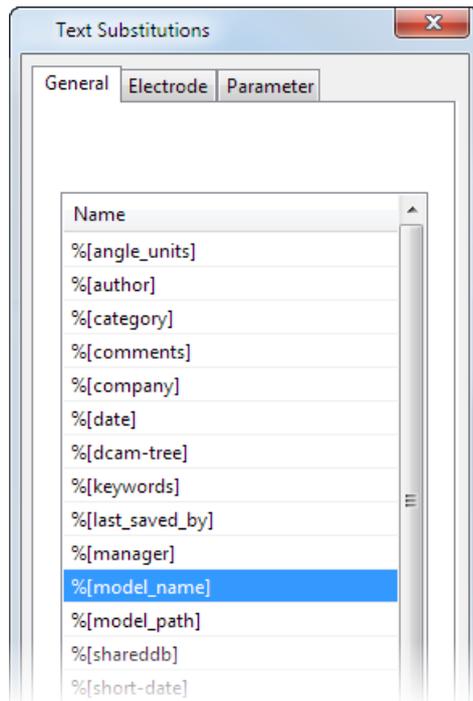
- 3 Enter a label, if required.



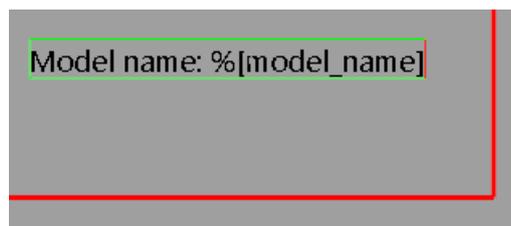
- 4 Click  (Annotation toolbar) to display the **Text Substitutions** dialog.



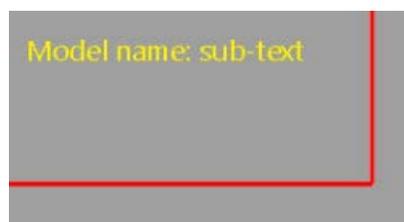
- 5 Select the required substitution string.



- 6 Select **Live value** option to re-evaluate the substitution whenever the text is drawn.
- 7 Click **Apply**.



- 8 Click **Cancel** and click the drawing to display the substituted text. In this example, the name of the model was **sub-text**.



Editing text on a curve

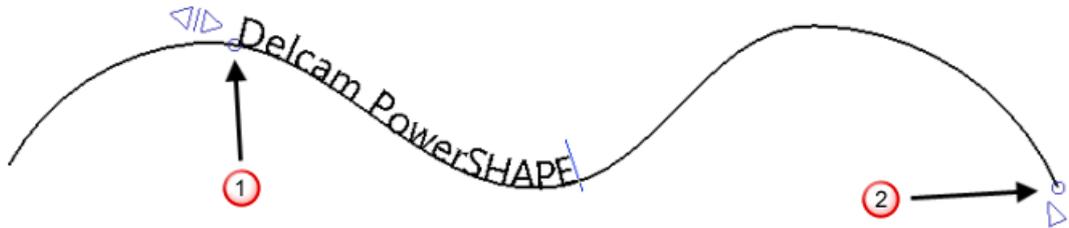
You can edit text written on a curve using the options on the **Text Editor** toolbar (see page 129).

In addition, you can:

- move the text along the curve (see page 142).
- reverse the text (see page 142).

- mirror the text (see page 143).
- change the position of the text (see page 143).

Moving text along a curve



- 1 Double-click the text to edit the text.
- 2 Move the cursor over the start point of the text **1**. The cursor changes to a hand .

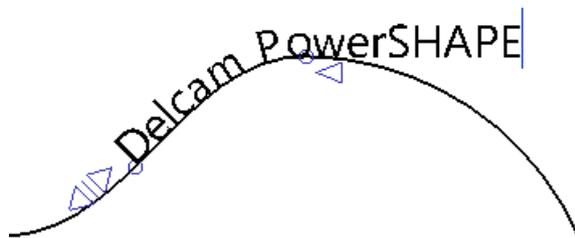


- 3 Use  to move the text. The text follows the line of the curve until it reaches the end point **2**.

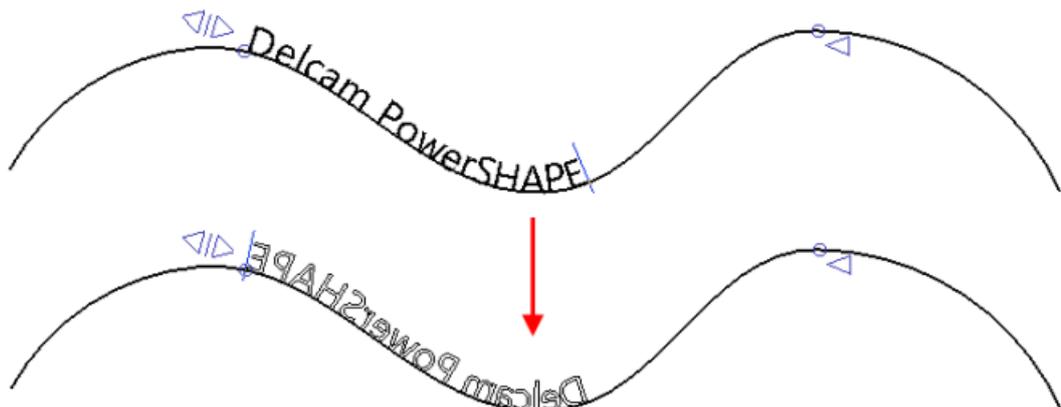
When text is created, the end point is created at the end of the curve. You can drag the end point along the curve to any point between the start point and the end of the curve.



Text that is moved beyond the end point no longer follows the line of the curve and continues on the same tangent.



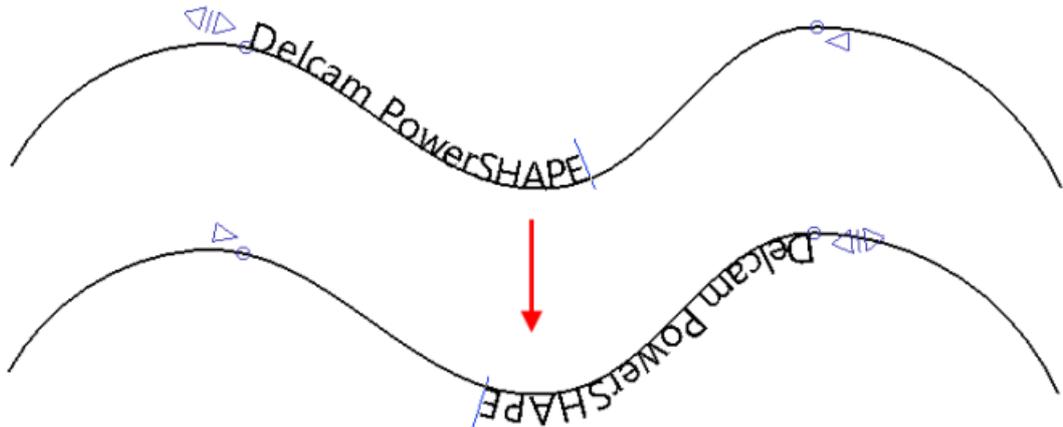
Reversing the text



- 1 Double-click the text to edit the text.
- 2 Click  to reverse the text.
- 3 Click  again to return the text to the original direction. Any edits made to the text whilst reversed are retained.

 *Reversed text can still be edited using the **Text editor** toolbar.*

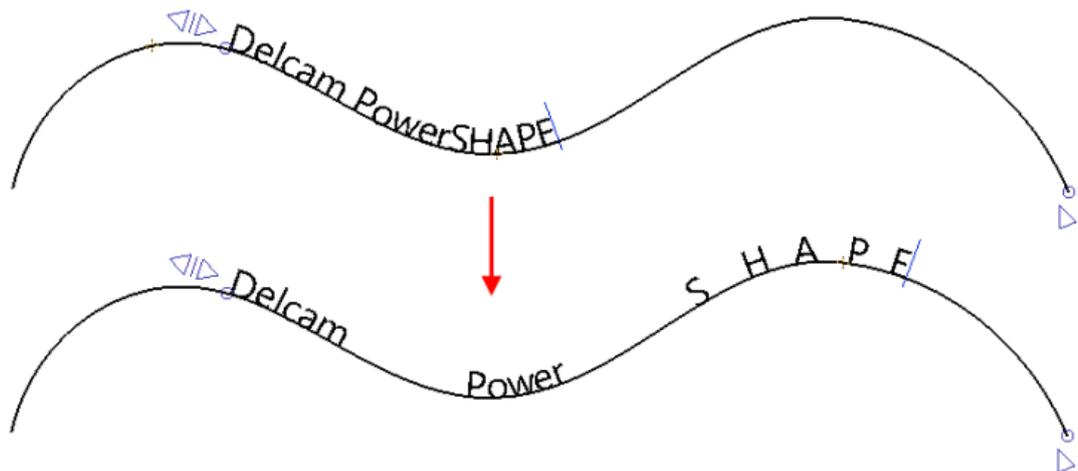
Mirroring the text



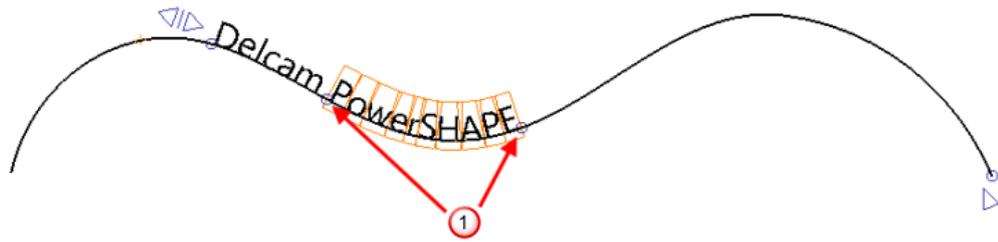
- 1 Double-click the text to edit the text.
- 2 Click  to mirror the text. When mirrored, the start and end points are reversed. These points can be moved as required.
- 3 Click  to return the text to the original direction. If you have moved the start and end points, they will not revert back.

 *Mirrored text can still be edited using the **Text editor** toolbar*

Changing the position of the text



- 1 Double-click the text to edit the text.
- 2 Click any point in the text and drag the mouse to select characters.



A start and end point is created either side of your character selection boxes **1**.

- 3 Move the cursor over the new start/end point until the cursor changes into a hand  .

- 4 Use  to move the selected characters along the curve.

 Use **Ctrl**  to move the text along the curve, maintaining the spacing between the selected text and the next lot of text.

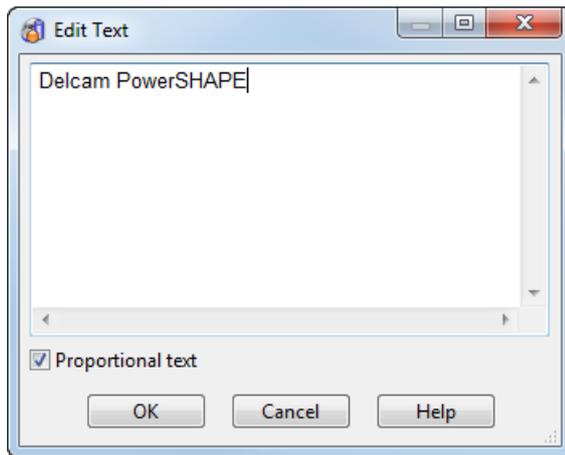
After the position has been changed, the text can still be moved/reversed/mirrored as a complete entity, retaining the changes.

Editing Duct text

- 1 Double-click the text.



The **Edit Text** dialog is displayed, showing the selected text.



- 2 Edit the text in the dialog.

You can also make the text fixed-width or proportionally spaced using the dialog.

- 3 Change the words to:

`This is text.`

- 4 Click **OK** or click in the graphics window. The text is altered.

This is text.



Most tools on the Text Editor toolbar are available to use on the Duct text. You can convert the text to PowerSHAPE text by changing the font.

Converting text to wireframe

- 1 Select the text.

Sample text

- 2 Select **Edit > Convert > To wireframe** from the menu.
- 3 The outline of the text is converted to wireframe.

Sample text

Inserting strings from the File Properties

When creating text, you can enter one of the following strings and it will be substituted:

%title

%subject

%author

%manager

%company

%keywords

%comments

%version

%date

%short-date - this is the date in locale short date format

All but the last three are read from fields in the **File > Properties** dialog in Windows Explorer. For further details, see Finding out what a model contains in Modelling Concepts..



*If a field is empty in the dialog, a - is substituted (except for **%author** where the current user is substituted).*



***%version** refers to the version number of PowerSHAPE you are using.*

- 1 Go into text-creation mode.
- 2 Type some text containing a %string.

Created by %author

- 3 Click **Select**  to go into selection mode.

The text is now substituted.

Created by Razia Ghani

In our example, the string is substituted by the name of the user who saved the original model. If the model is new, then the name of the current user appears.

If you double click the text, you will see the original string.

Updating the strings from File Properties

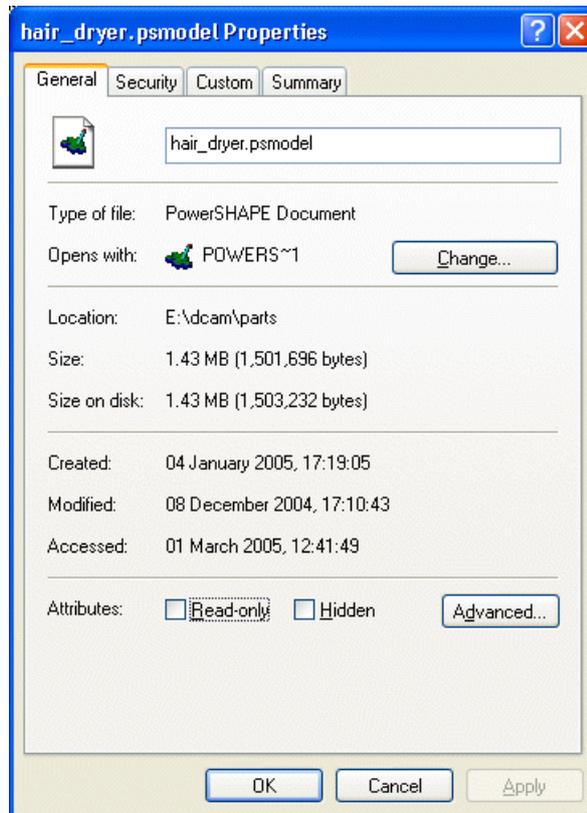
As the fields on the Windows Explorer's **File > Properties** dialog change, so do any special strings in the text.

For example,

- 1 Suppose your model contains the following string.

Created by %author

- 2 Save and close the model.
- 3 In Windows Explorer, select the *psmodel*.
- 4 From the **File** menu of Windows Explorer, select **Properties** to display the **Properties** dialog for the model.



- 5 Change the **Author** field on the **Summary** page of the **File > Properties** dialog (Windows Explorer).
- 6 Open the model again and the new name is substituted for the **%author** string.

Balloons

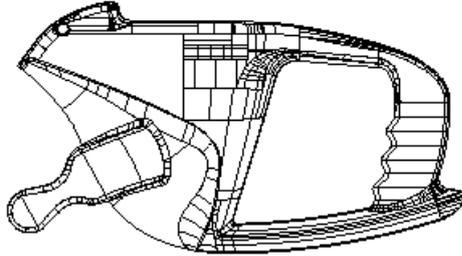
A *balloon* is a label with an arrow that points to an area on your drawing. The label can be a code number, letter, or description.

You can use balloons to:

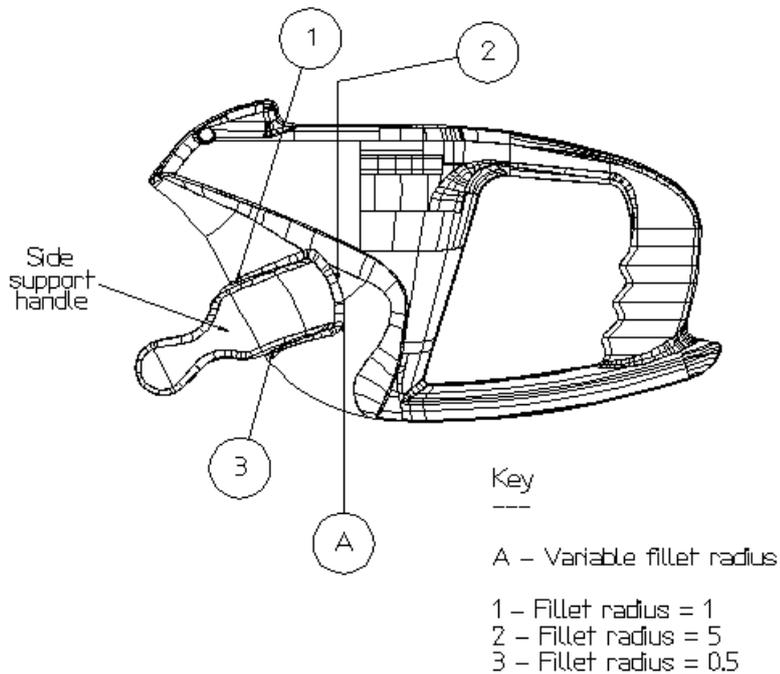
- identify parts of your model.
- describe features on your model.

- add numbered labels representing a key to information in a bill of material or similar table.

The following chainsaw handle example has a fillet surface on the support handle on the side.



You can use balloons to label the model to show the details. A key is provided in the bottom right corner explaining the code of each balloon.



You can see from this example that there are different types of balloons that you can create.

Creating a balloon

You can create the following types of balloon:

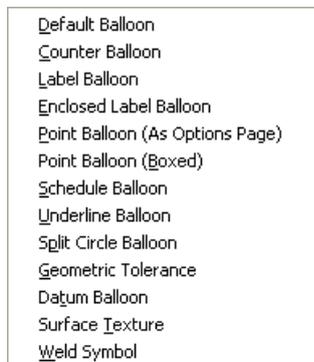
- Default (see page 156)
- Point (as Options page) (see page 158)
- Counter (see page 159)
- Label (see page 161)

- Enclosed label (see page 163)
- Point (boxed) (see page 165)
- Schedule (see page 166)
- Underline (see page 168)
- Split circle (see page 171)
- Geometric tolerance (see page 172)
- Datum (see page 176)
- Surface texture (see page 178)
- Weld symbol (see page 180)

Entering balloon-creation mode



- 1 Click  (**Main** toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 From the **Object** menu, select **Balloon** to display the **Balloon creation** options.



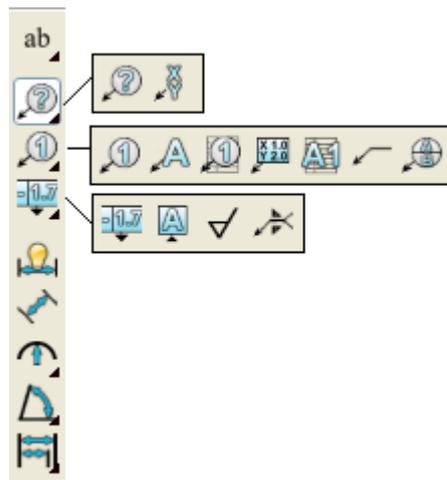
- 3 Select a balloon type from the menu to display the Annotation toolbar (see page 116).

- 4 The balloon creation options are shown in ② below on the Annotation toolbar (see page 116).



You can also see these options by selecting  from the **Drafting** toolbar.

- 5 Select the option you require from the appropriate flyout.



The following balloons are available from the annotation toolbar



Default (see page 156)



Point (as Options page) (see page 158)



Counter (see page 159)



Label (see page 161)



Enclosed Label (see page 163)



Point (Boxed) (see page 165)



Schedule (see page 166)



Underline (see page 168)



Split circle (see page 171)



Geometric Tolerance (see page 172)



Datum (see page 176)



Surface Texture (see page 178)



Weld Symbol (see page 180)

Balloon toolbar

When creating a balloon, Balloon options are added to the end of a version of the Text Editor toolbar (see page 129). Any unrequired style items are suppressed.



Counter — This formats the counter used in balloons. The rightmost character is the counter and can only be alphanumeric. This character increases by 1 unit when you create a new balloon. All the other characters are added as they are to the label.

To reset the value of the counter, change the rightmost character to the new value.

To define the counter, enter **%c** as the **Default** value on the **Balloon** page (see page 152) of the **Options** dialog.



— If selected, a leader is drawn from the balloon. If deselected,

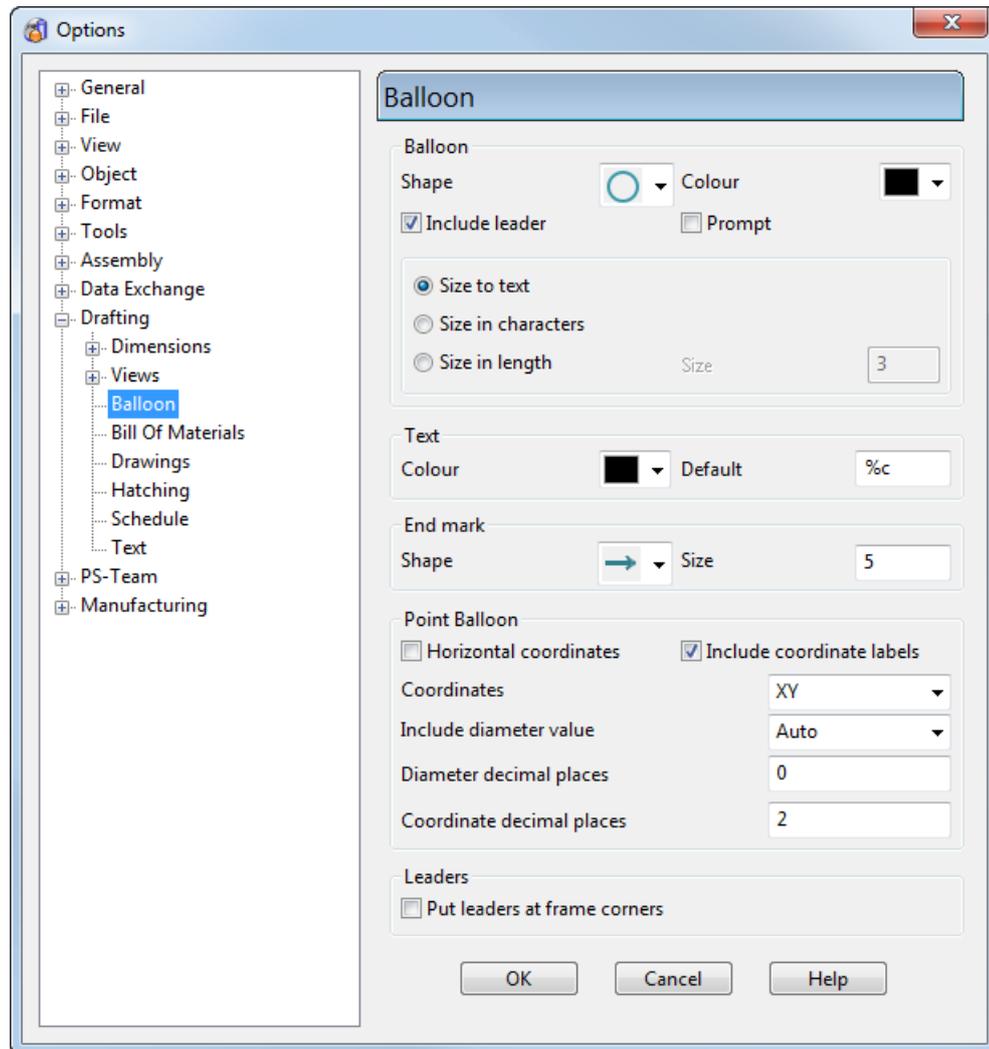


, a leader is not drawn.

After you have created a balloon, you can edit (see page 126) its label text like any other text.

Setting the default balloon Options

Use the **Balloon** page of the **Options** dialog to set the default balloon settings.



To open this page:

- 1 Select **Tools > Options** from the menu.
The **Options** dialog is displayed.
- 2 In the browser tree, select **Drafting > Balloon**.
The **Balloon** options page is displayed.

The page has these sections:

- **Balloon**
- **Text**
- **End mark**
- **Point balloon**
- **Leaders**

Balloon

Use this section to controls the graphic that encloses the text.

Shape — Select the balloon shape from the options.



None

A1 →



Underline

A1 →



Arc

A1 →



Split circle

A1 →



Box

A1 →

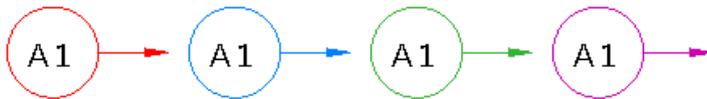


Triangle

A1 →

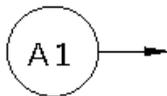
Colour — Select the colour of the balloon outline from the list.

For example:



Include leader — Select this option to use a leader.

For example:



Deselect this option if you don't want a leader:



Prompt — Select this option to display the **Balloon Confirmation** dialog (see page 162) when creating a default balloon.



When creating Counter and Label balloons, this dialog is displayed by default.

Size to text — Select this option to fit the balloon size to the text. The minimum size of the balloon is 3 characters.

Size in characters — Select this option to specify the balloon size in terms of number of characters. Enter the number of characters as the **Size** value.

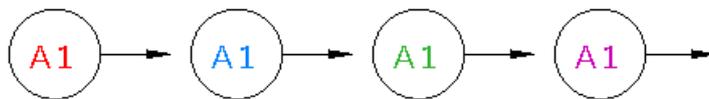
Size In length — Select this option to specify the balloon size in the current units, such as millimetres. Enter the exact size of the balloon as the **Size** value.

Size — Enter the exact width of the balloon.

Text

Colour — Select the colour of the text in the balloon.

For example:



Default — Default text for balloons. The **%c** represents the counter. When you create a balloon, %c is substituted by the text in the **Counter** box on the **Text Editor** toolbar. Any other text in the **Default** box is added as it is to the label.

You can change this text if you want. For example, if **Default** is:

Balloon number: %c

and the **Counter** on the **Text Editor** toolbar is set to **A1**, balloons are created with labels:

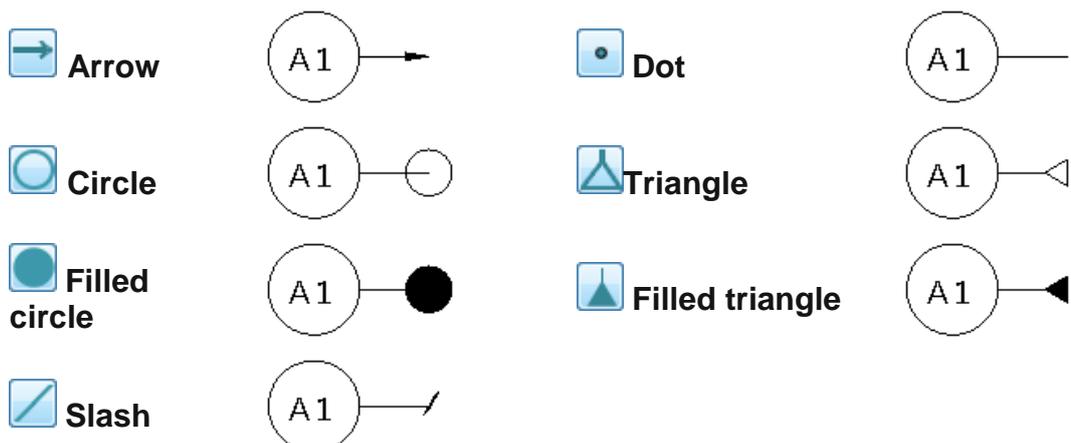
Balloon number: A1

Balloon number: A2

End mark

The end mark is the end of the pointer.

Shape — Shape of mark at the end of the pointer.



Size — Enter the width of the mark in mm.

For example:



Point Balloon

Point balloons display coordinates.

Horizontal coordinates — Select this option to position the coordinates horizontally.

Include coordinate labels — Select this option to add labels in front of coordinates in the point balloons.

Coordinates — Select the coordinates to include in point balloons, from:

- **XY** — XY coordinates are displayed.
- **XYZ** — XYZ coordinates are displayed.
- **Z** — Only the Z value is displayed.

Include diameter value — Select one of these options:

- **Auto** — The diameter is displayed when you click the balloon's pointer in the centre of a circle.
- **On** — The diameter is always displayed.



If the balloon's pointer is not at the centre of a circle, a hyphen (-) is displayed.

- **Off** — The diameter value is not displayed.

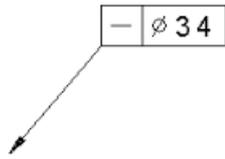
Diameter decimal places — Enter the number of decimal places to display for a diameter value in a point balloon. If the units for the model are inches, additional decimal places are displayed.

Coordinate decimal places — Enter the number of decimal places to display for coordinate values in a point balloons. If the units for the model are inches, additional decimal places are displayed.

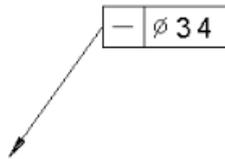
Leaders

Put leaders at frame corners — Select this option to display leaders at the corner of the frame.

For example, with **Put leaders at frame corners** selected:



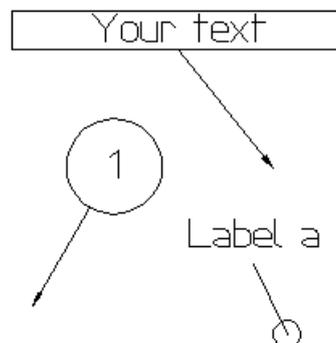
With **Put leaders at frame corners** deselected, leaders are displayed at the centre of an edge of the frame.



OK — Accept the changes.

Creating a default balloon

You can create a balloon using the settings on the **Balloon** page of the **Options** dialog. By changing the settings, you can create various balloons.



- 1 Click the **Drafting Mode**  button (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 If required, change the setting on the **Balloons** page of the **Options** dialog. For further details, see *Setting the balloons on the Options dialog* (see page 159).
- 3 Click the **Annotation**  button (drafting toolbar).
- 4 Click the **Default balloon**  flyout (annotation toolbar).
- 5 Click the **Default balloon**  button (default flyout).

The *Balloon toolbar* (see page 151) is displayed alongside the Text toolbar. The options on the Balloon toolbar are mainly used to edit text in balloon labels. You use this toolbar to set the value of the counter..

- 6 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.

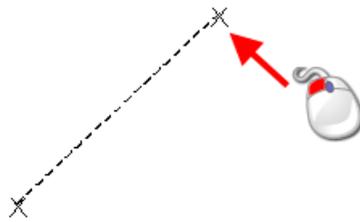


- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

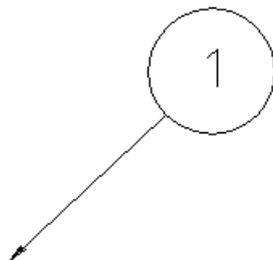
- 7 Set the position where you want the balloon, in one of these ways:

- Click the mouse on the drawing



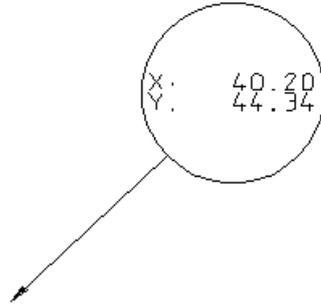
- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

The balloon is added to the drawing, for example:

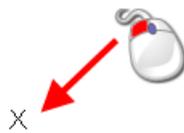


Creating a point balloon (as Options page)

A point balloon uses coordinates at its pointer as its label and the default balloon shape from the **Balloons** page of the **Options** dialog.



- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Default balloon**  flyout (annotation toolbar).
- 4 Click the **Point balloon**  button (default balloon flyout).
The **Balloon** toolbar (see page 151) is displayed. This is mainly used to edit text in balloon labels.
- 5 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.

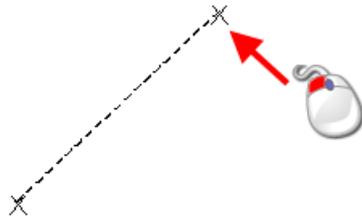


- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

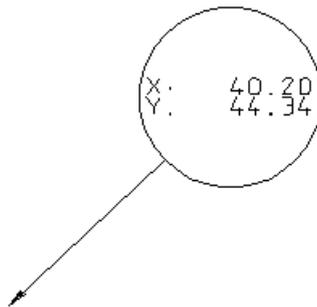
- 6 Set the position where you want the balloon, in one of these ways:

- Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

The balloon is added to the drawing.

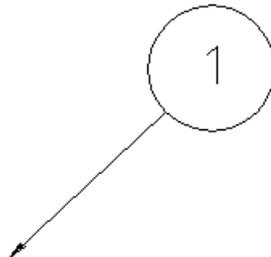


To display the XYZ coordinates or the Z coordinate, select the relevant option from the **Co-ordinates** list on the Balloon (see page 152) page of the **Options** dialog.

You can also remove the coordinate labels from the balloons by turning off the **Include coordinate labels** option on the same page. There are other options which you can set.

Creating a counter balloon

A counter balloon is enclosed in a circle and its text is as defined in the **Options** dialog.



The counter increases by 1 when you create the next balloon.

- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.

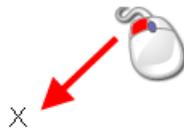
2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).

3 Click the **Counter balloon**  flyout (*annotation menu*). The **Balloon** toolbar (see page 151) is displayed, which is mainly used to edit text in balloon labels.

4 Click the **Counter balloon**  button.

5 Set the point you want to label, in one of these ways:

- Click the mouse on the drawing.

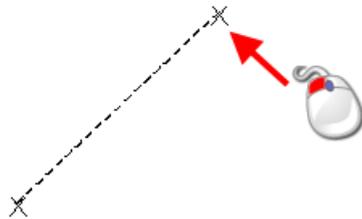


- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

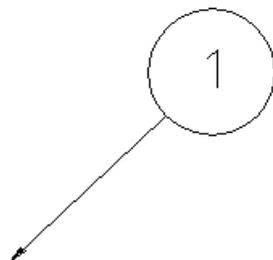
6 Set the position where you want the balloon, in one of these ways:

- Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

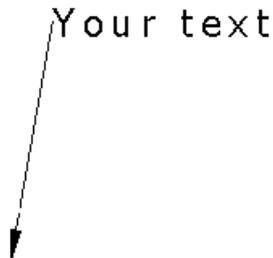
The balloon is added to the drawing.



To reset the value of the counter, see *Balloon toolbar* (see page 151).

Creating a label balloon

A label balloon is one with text not enclosed in a shape. You are prompted to specify the text of the label during creation.



- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Counter balloon**  flyout (*annotation menu*). The **Balloon** toolbar (see page 151) is displayed, which is mainly used to edit text in balloon labels.
- 4 On the flyout, click the **Label balloon**  button.
- 5 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.

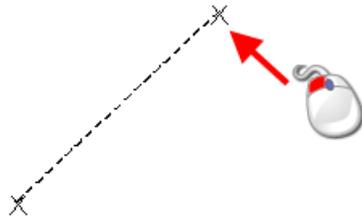


- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

- 6 Set the position where you want the balloon, in one of these ways:

- Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

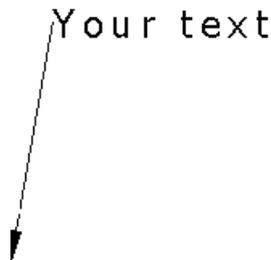
When you set the position, the **Balloon Confirmation** dialog is displayed.

- 7 Use the **Balloon Confirmation** dialog (see page 162) to enter the text you require to appear in the balloon.

In the example below, **Your text** was entered in the dialog.

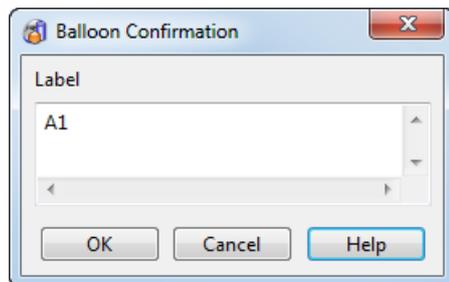
- 8 Click **OK**.

The balloon is added to the drawing.



Balloon Confirmation dialog

Use this dialog to edit the text of the balloon. This dialog is automatically displayed when creating counter and label balloons. To display this dialog when creating default balloons, select **Prompt** on the **Balloon** page (see page 152) of the **Options** dialog.



Label — Enter the balloon text.



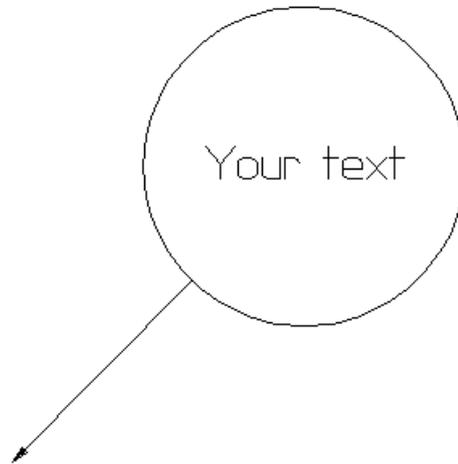
Changing this label does not increment the counter.

OK — Makes the changes to the balloon and removes the dialog from the screen.

Cancel — Cancels the current balloon creation and removes the dialog from the screen.

Creating an enclosed label balloon

An enclosed label balloon has text enclosed in a circle. You can specify the text of the label during creation.



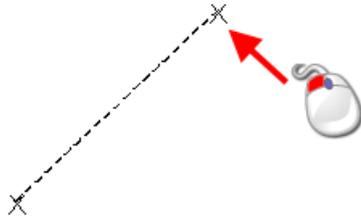
- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Counter balloon**  flyout (*annotation menu*). The **Balloon** toolbar (see page 151) is displayed, which is mainly used to edit text in balloon labels.
- 4 On the flyout, click the **Enclosed label balloon**  button.
- 5 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.



- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

- 6 Set the position where you want the balloon, in one of these ways:
 - Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

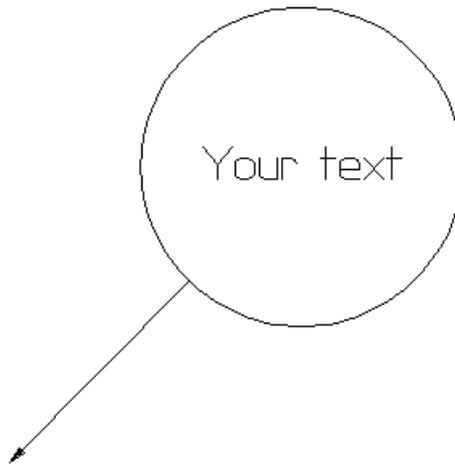
When you click the position, the **Balloon Confirmation** dialog is displayed.

- 7 Use the **Balloon Confirmation** dialog (see page 162) to enter the text you want in the balloon.

In the example below, **Your text** was entered in the dialog.

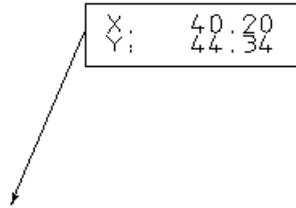
- 8 Click **OK**.

The balloon is added to the drawing.



Creating a point balloon (boxed)

A point balloon displays the coordinates from its pointer location as its label and is enclosed in a box.



- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Counter balloon**  flyout (*annotation menu*). The **Balloon** toolbar (see page 151) is displayed, which is mainly used to edit text in balloon labels.
- 4 On the flyout, click the **Point balloon (boxed)**  button.
- 5 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.

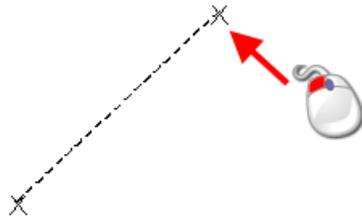


- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

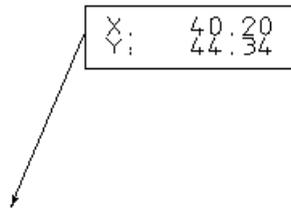
- 6 Set the position where you want the balloon, in one of these ways:

- Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

The balloon is added to the drawing.

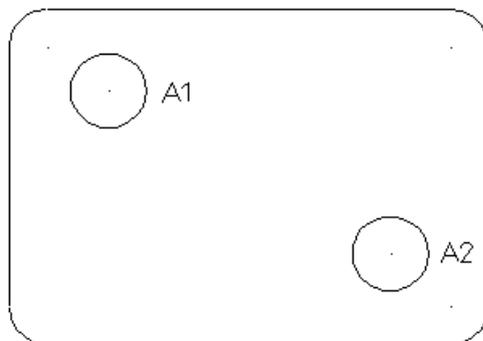


By default, only the X and Y coordinates are displayed. Alternatively, you can set the **Coordinates** option to **XYZ** or **Z** on the Balloon page (see page 152) of the **Options** dialog.

You can remove the coordinate labels from balloons by deselecting **Include coordinate labels**.

Creating a schedule balloon

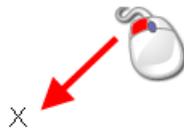
A schedule balloon marks a position to include in the schedule for balloons. In the example below, A1 and A2 are schedule balloons. They have no pointers displayed.



You can use schedule balloons as labels to mark drill holes.

- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.

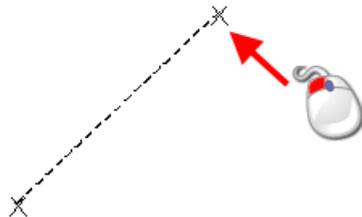
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Counter balloon**  flyout (*annotation menu*). The **Balloon** toolbar (see page 151) is displayed, which is mainly used to edit text in balloon labels.
- 4 On the flyout, click the **Schedule balloon**  button.
- 5 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.



- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

- 6 Set the position where you want the balloon, in one of these ways:
 - Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

The balloon is added to the drawing.

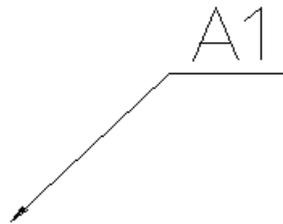
A1

This type of balloon does not have a pointer.

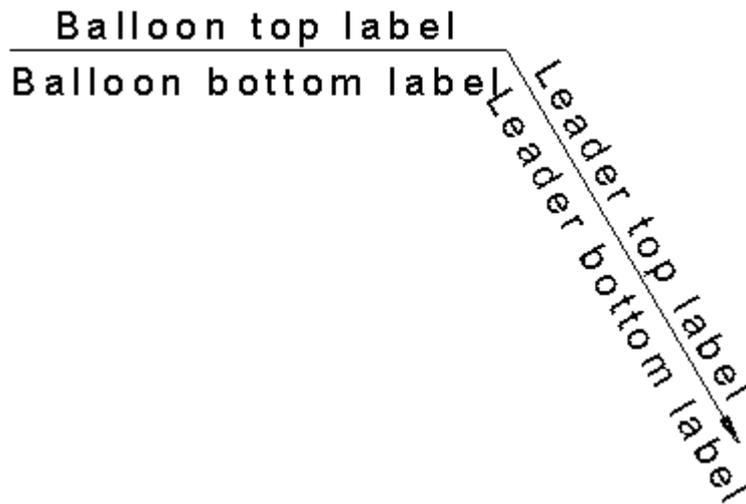
When you have created a collection of schedule balloons, you can create a schedule (see page 283) to display the labels of the balloons and their coordinates.

Creating an underline balloon

An underline balloon is one whose label is underlined and its text is as defined in the **Options** dialog.



The counter increases by 1 unit when you create the next balloon. If the **Standard** (see page 152) is set to **USDD**, you can enter text labels along the balloon and the leader.



- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Counter balloon**  flyout (*annotation menu*).
- 4 Click  (counter flyout)

The *Balloon toolbar* (see page 151) is displayed, which is mainly used to edit text in balloon labels. The value of the counter is set using this toolbar.

- 5 Set the point you want to label, in one of these ways:

- Click the mouse on the drawing.

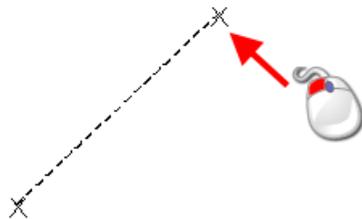


- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

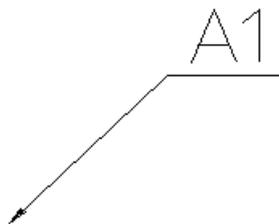
- 6 Set the position where you want the balloon, in one of these ways:

- Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

If the Standard is not USDD, the balloon is added to the drawing.

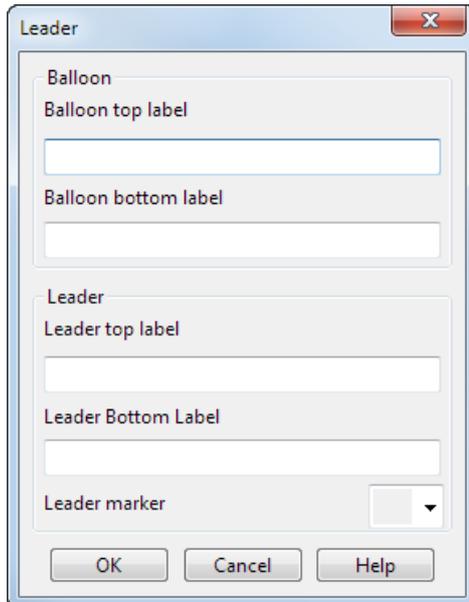


To reset the value of the counter, see Balloon toolbar (see page 151).

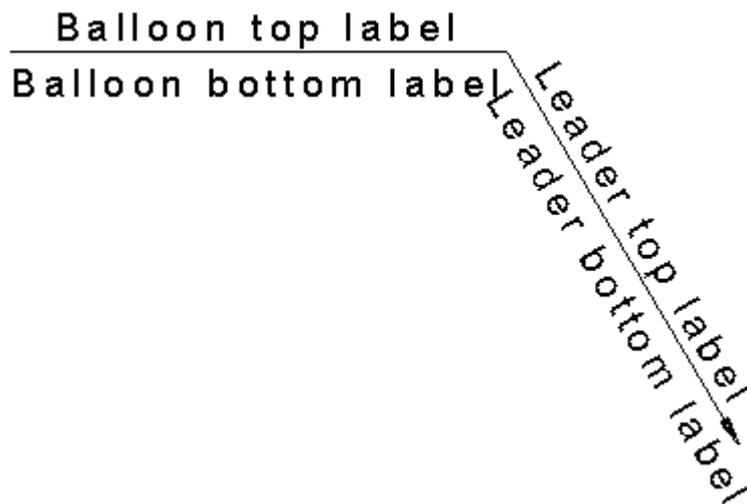
If the standard is USDD, the Leader dialog (see page 170) is displayed. Complete this dialog to add the text to the balloon.

Leader dialog

Use this dialog to add text to the balloon.

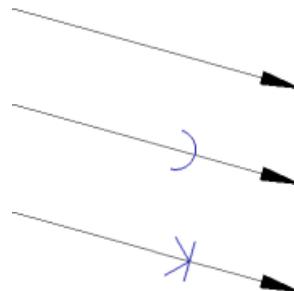


- 1 Enter the text that you want to display in the balloon.



- 2 Select the style of **Leader Marker** from:

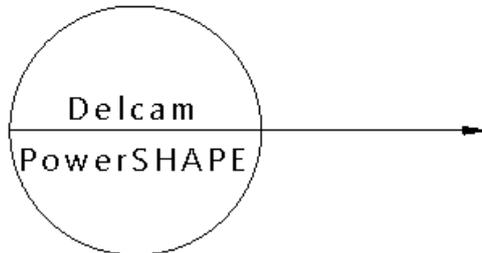
- No marker**
- Bonding marker**
- Glue marker**



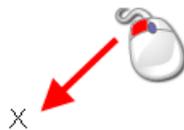
- 3 Click **OK** to create the balloon.

Creating a split circle balloon

A split circle balloon is one with text in the top and bottom halves of a circle. You are prompted to specify the text of the labels during creation.



- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Counter balloon**  flyout (*annotation menu*). The **Balloon** toolbar (see page 151) is displayed, which is mainly used to edit text in balloon labels.
- 4 Click the **Split circle balloon**  button.
- 5 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.

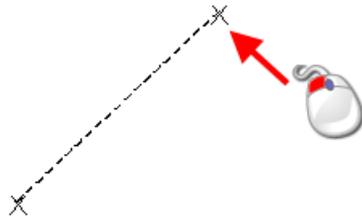


- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

- 6 Set the position where you want the balloon, in one of these ways:

- Click the mouse on the drawing



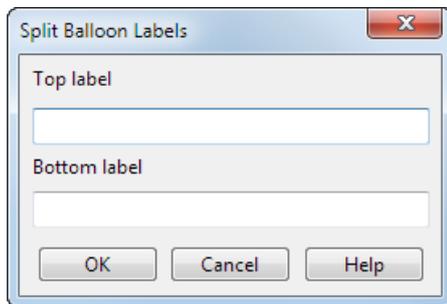
- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

When you set this position, the **Split Balloon Labels** dialog (see page 172) is displayed.

- 7 Use the dialog to enter the balloon text you want.

Split Balloon Labels dialog

Use this dialog to edit the text in split circle balloons.



Top label — Enter the text you want to appear in the top half of the balloon.

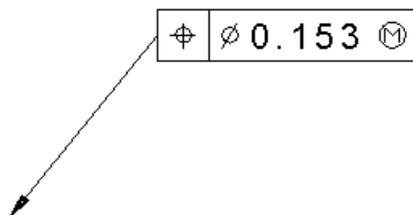
Bottom label — Enter the text you want to appear in the bottom half of the balloon.

OK — Creates the balloon and removes the dialog from the screen.

Cancel — Cancels the current balloon creation and removes the dialog from the screen.

Creating a geometric tolerance balloon

A geometric tolerance balloon enables you to show certain deviations of a feature using standard symbols, for example:



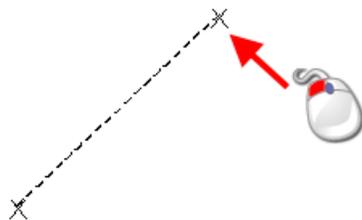
- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Geometric Tolerance**  flyout to display the **Balloon toolbar** (see page 151). The options on this toolbar are used to edit text in balloon labels.
- 4 Click  (Geometric tolerance flyout).
- 5 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.



- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

- 6 Set the position where you want the balloon, in one of these ways:
 - Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

The **Geometric Tolerance** dialog (see page 174) is displayed.

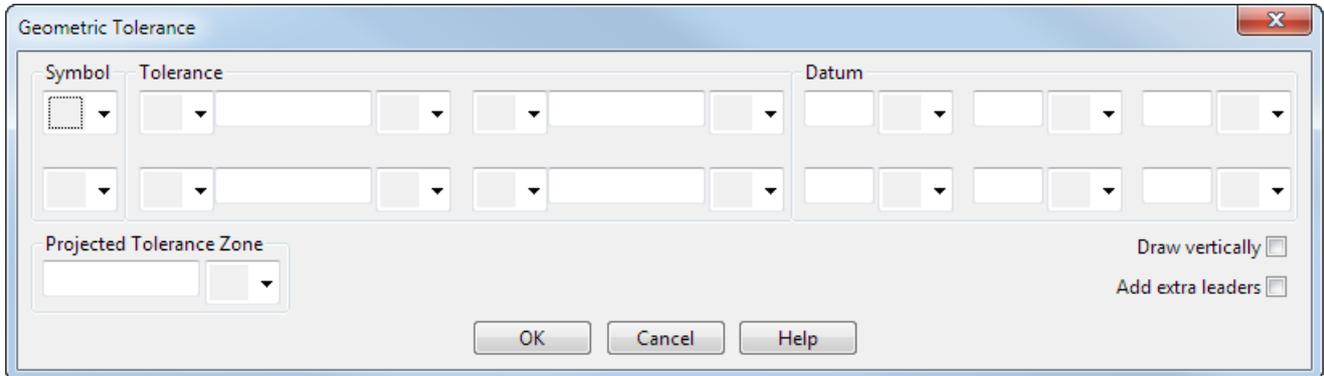
- 7 Use the **Geometric Tolerance** dialog (see page 174) to enter the symbols and values of the geometric tolerances.



There is a option called **Put leaders at frame corners** on the **Balloons** page of the **Options** dialog. If selected, the leader of the geometric tolerance balloon is at the corner of the frame. If deselected, it is centred on an edge of the frame.

Geometric Tolerance dialog

Use this dialog to define the geometric tolerances enclosed in the balloon.



Symbol — Select a tolerance symbol from the list.

- | | |
|---|--|
| <input type="checkbox"/> None | <input checked="" type="checkbox"/> Squareness |
| <input checked="" type="checkbox"/> Straightness | <input checked="" type="checkbox"/> Angularity |
| <input checked="" type="checkbox"/> Flatness | <input checked="" type="checkbox"/> Position |
| <input checked="" type="checkbox"/> Roundness | <input checked="" type="checkbox"/> Concentricity |
| <input checked="" type="checkbox"/> Cylindricity | <input checked="" type="checkbox"/> Symmetry |
| <input checked="" type="checkbox"/> Line profile | <input checked="" type="checkbox"/> Circular run-out |
| <input checked="" type="checkbox"/> Surface profile | <input checked="" type="checkbox"/> Total run-out |
| <input checked="" type="checkbox"/> Parallelism | |

Tolerance —

- Select whether you want a diameter symbol.
 - No diameter symbol
 - Diameter symbol
- Enter the tolerance value.
- Select a material condition symbol.
 - None
 - At maximum
 - At least

 Regardless of feature size

Datum — This defines the datum from which the tolerances are measured.

To have two rows of geometric tolerance in the balloon, set the second row of **Symbol**, **Tolerance**, and **Datum** options.

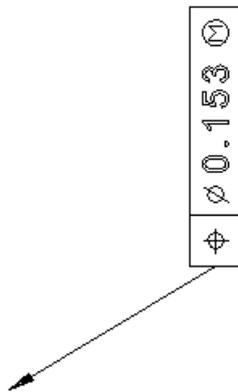
Projected Tolerance Zone —

- Enter a value.
- Select whether you want a projected tolerance zone symbol.

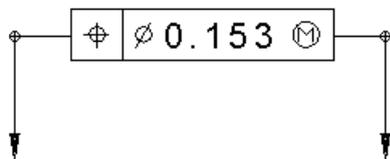
No symbol

Projected tolerance zone symbol

Draw vertically — Select this option to create the balloon vertically.



Add extra leaders — Select this option to have more than one leader. The **Add Leaders** dialog (see page 187) is displayed when you click the **OK** button.



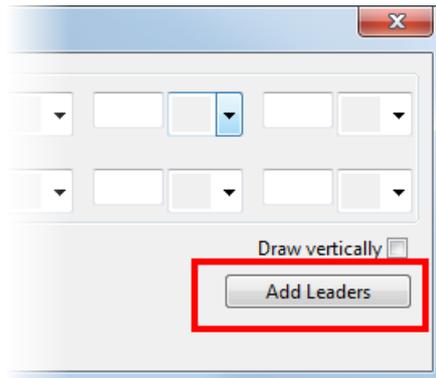
The balloon above has dog-legged pointers (see page 183).

OK — Creates the geometric tolerance balloon and removes the dialog from the screen.

Editing a geometric tolerance

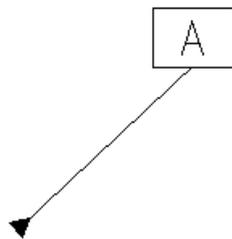
You can edit a geometric tolerance by double clicking on the tolerance balloon to display the **Geometric Tolerance** dialog.

When you are editing the geometric tolerance, the **Add Extra Leaders** option is replaced by the **Add Leaders** button. Click **Add Leaders** to display the **Add Leaders** dialog.



Creating a datum balloon

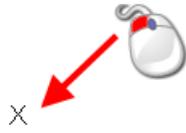
A datum balloon usually points to a datum which is referred to in a geometric tolerance.



The text in the balloon is set by the counter. The counter increases by 1 unit when you create the next balloon.

- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Geometric Tolerance**  flyout to display the **Balloon toolbar** (see page 151). The options on this toolbar are used to edit text in balloon labels.
- 4 On the flyout, click the **Datum balloon**  button.
- 5 Set the point you want to label, in one of these ways:

- Click the mouse on the drawing.

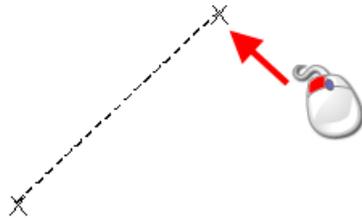


- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

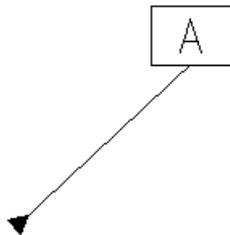
- 6 Set the position where you want the balloon, in one of these ways:

- Click the mouse on the drawing



- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

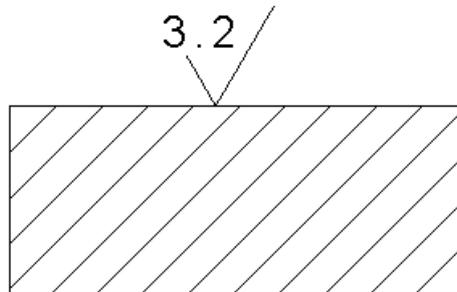
The balloon is added to the drawing.



To reset the value of the counter, see *Balloon toolbar* (see page 151).

Creating a surface texture balloon

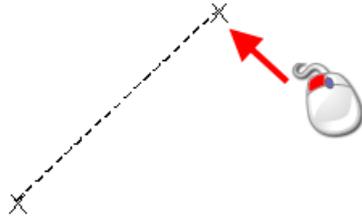
A surface texture balloon enables you to show texture on a surface using standard symbols, for example:



- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Geometric Tolerance**  flyout to display the **Balloon toolbar** (see page 151). The options on this toolbar are used to edit text in balloon labels.
- 4 In the flyout, click the **Surface texture**  button.
- 5 If you want to place the symbol directly onto the surface, ensure the **Pointer On/Off** button is deselected  on the **Balloon toolbar**. Otherwise, ensure the **Pointer On/Off** button is selected .
- 6 If you have a pointer on the symbol, click the position on the surface where you want the pointer to touch.



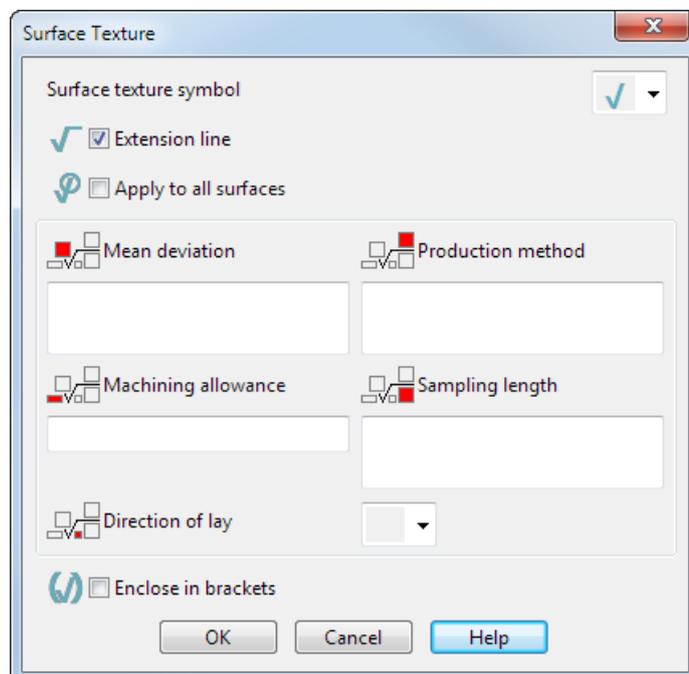
- 7 Click the position where you want the symbols of the surface texture to lay. If you don't have a pointer, this position should be clicked on the surface.



- 8 If you don't have a pointer, click another position on the surface. The symbol is drawn parallel to the line made by the two positions just entered.
- 9 When you have set the two positions, the **Surface Texture** dialog (see page 179) is displayed. Use it to enter the symbols and values of the surface texture.

Surface Texture dialog

Use this dialog to define the surface texture.



Surface texture symbol — Select a surface texture symbol from the list.

Extension line — This extends the line to fit the texture information.

Apply to all surfaces — This adds a circle to the symbol to indicate the texture is applied to all the surfaces in the model.

Fill in the following to define the surface texture:

- **Mean deviation**

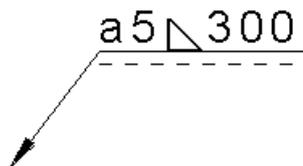
- **Production method**
- **Machining allowance**
- **Sampling length**
- **Direction of lay**

Enclosed in brackets — Select this option to encloses the surface texture balloon in brackets.

OK — Creates the surface texture balloon and removes the dialog from the screen.

Creating a weld symbol

A weld symbol enables you to represent a weld using standard symbols, for example:



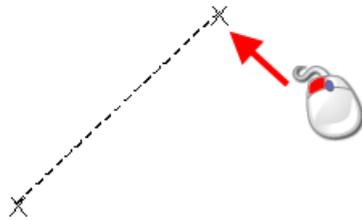
- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click the **Geometric Tolerance**  flyout to display the **Balloon toolbar** (see page 151). The options on this toolbar are used to edit text in balloon labels.
- 4 In the flyout, click the **Weld Symbol**  button.
- 5 Set the point you want to label, in one of these ways:
 - Click the mouse on the drawing.



- Enter X Y coordinates in the **Command input** field on the **Status bar**, for example **80 40**.

A rubber-banded line is attached from the position to the mouse cursor.

- 6 Set the position where you want the balloon, in one of these ways:
 - Click the mouse on the drawing

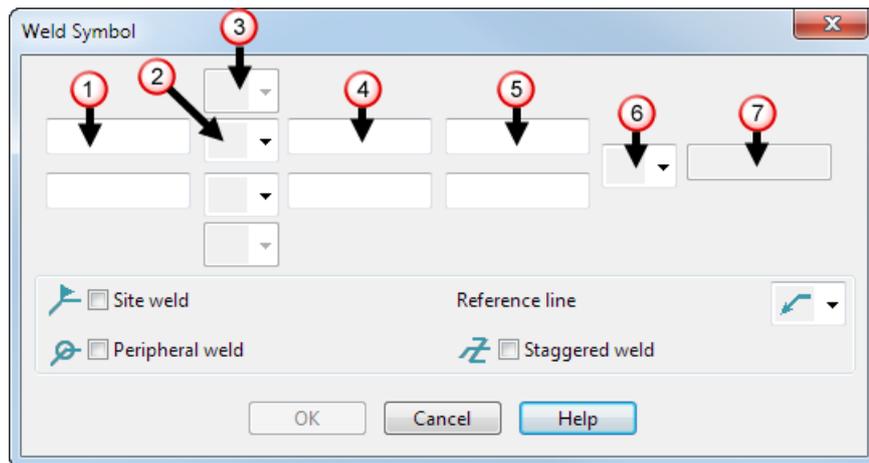


- Enter the absolute X Y coordinates in the **Command input** field on the **Status bar**, for example **120 70**.

When you set this position, the **Weld Symbol** dialog (see page 181) is displayed. Use it to enter the symbols and values of the weld symbol.

Weld Symbol dialog

Construct the weld symbol using the options on the dialog.



- ① — Enter a weld cross section dimension.
- ② — Select an elementary symbol from the list.

- | | |
|--|--------------------------------------|
| <input type="checkbox"/> None | <input type="checkbox"/> Single J |
| <input type="checkbox"/> Edgeflange | <input type="checkbox"/> Backing run |
| <input type="checkbox"/> Squarebutt | <input type="checkbox"/> Fillet |
| <input checked="" type="checkbox"/> Single V | <input type="checkbox"/> Plug |
| <input checked="" type="checkbox"/> Single Bevel | <input type="checkbox"/> Spot |
| <input type="checkbox"/> Single U | <input type="checkbox"/> Seam |

3 — Select a supplementary symbol from the list. This option is only available if you have selected an elementary symbol

- None
- Flat
- Convex
- Concave

4 — Enter the number of elements.

5 — Enter the length of the elements.

6 — Select an end mark from the list.

- None
- End mark

7 — Enter a welding process and class. This field is only available if you have selected an end mark.

Site weld — Select this option to add a site weld symbol to the balloon.

Peripheral weld — Select this option to add a peripheral weld symbol to the balloon.

Staggered weld — Select this option to add a staggered weld symbol to the balloon.

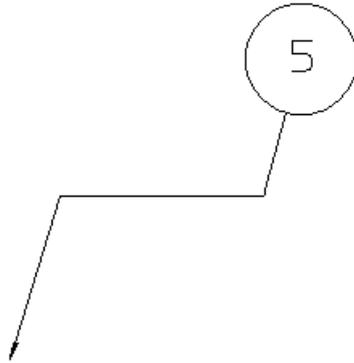
Reference line — Select a reference line option from the list:

- None
- Top
- Bottom

OK — Creates the weld symbol and removes the dialog from the screen.

Creating balloons with dog-legged pointers

You can create a balloon with a dog-legged pointer.



If you have a complex model with many curves and lines displayed, you may find it difficult to find a direct path from the object you want to point to and the label text. In this case, you may find it easier to re-direct the line of the balloon by using a dog-legged balloon.

- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 Click any balloon creation option (annotation toolbar).
- 4 Click the position where you want the pointer of the balloon to touch.



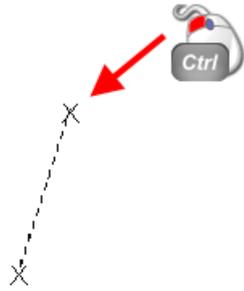
To cancel creating a balloon at any time, right-click in the graphics window. You go back to balloon creation mode, where you can select the type of balloon to create.

- 5 Hold down the **Ctrl** key and click the positions to dog-leg the pointer.

Remember not to enter the position of the balloon label.

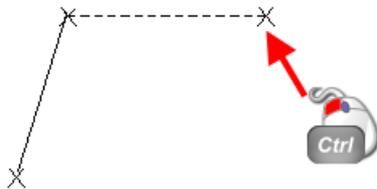
A rubber-banded line is attached from the last position entered to the mouse cursor.

In our example, we have entered a position as shown below.

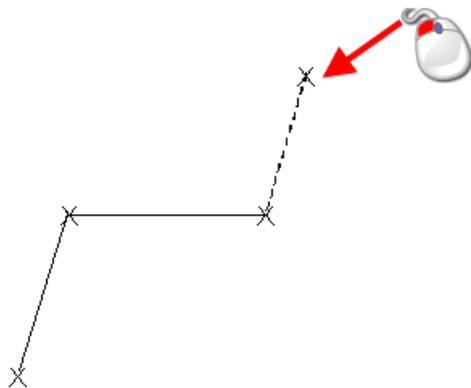


You can continue adding as many positions as you want in the dog-legged pointer.

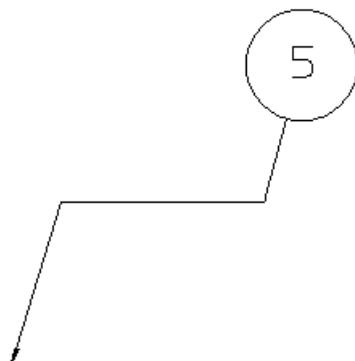
In our example, we have added another position as shown below.



- 6 Release the **Ctrl** key and enter the last position of the balloon.



The balloon is added to the drawing. (You may have to fill in the Balloon Confirmation dialog (see page 162) before the balloon is added.)

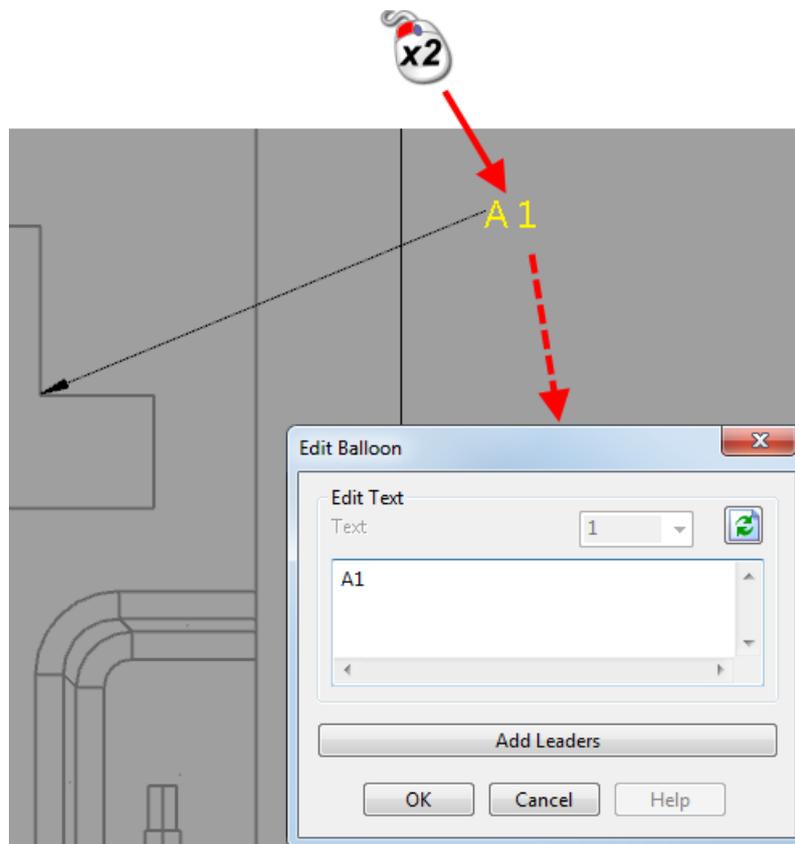


Editing a balloon

The label of a balloon is a symbol (made from individual objects) and its pointer is a line.

You can:

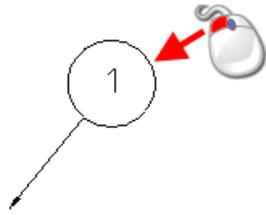
- move a balloon (see page 185).
- move a leader (see page 186).
- edit balloon text (see page 187).
- add leaders (see page 187).
- reposition a dog-legged leader (see page 189).
- double-click on a balloon to display the **Edit Balloon** dialog. Use the options on this dialog to edit the items in the balloon.



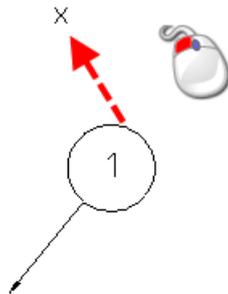
Moving a balloon

To move a balloon:

- 1 Click once on the balloon to select it.



- 2 Click and drag the balloon to a new position.
In this example, the balloon is dragged up and left.



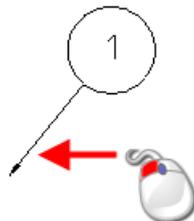
The balloon is moved to the new position. The leader is updated automatically.



Moving a leader

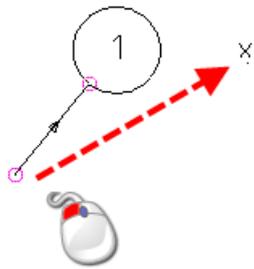
To move a balloon's leader:

- 1 Click once on the leader to select it.

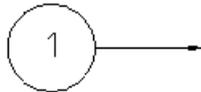


- 2 Click and drag the end of the leader to a new position.

In this example, the end is dragged to the right of the balloon.



The leader is moved to the new position:

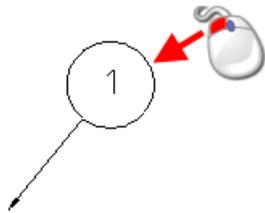


The position where the leader touches the balloon is updated automatically.

Editing balloon text

To edit balloon text:

- 1 Select the label of the balloon.

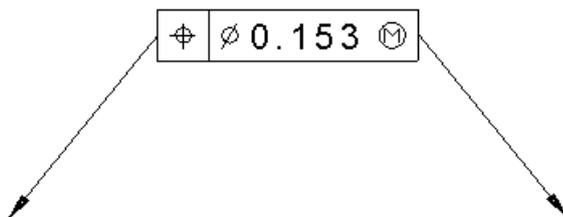


The label is a symbol.

- 2 Select **Edit > Convert > Symbol** from the menu.
- 3 Edit the text (see page 126).

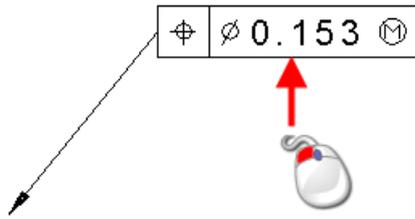
Adding leaders

When you have created a balloon, you can add more leaders to it, for example:

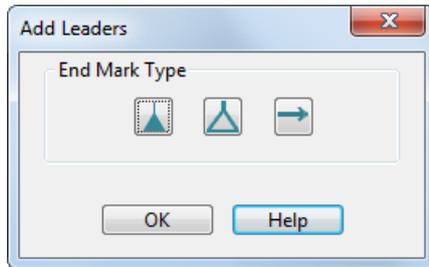


To add leaders to a balloon:

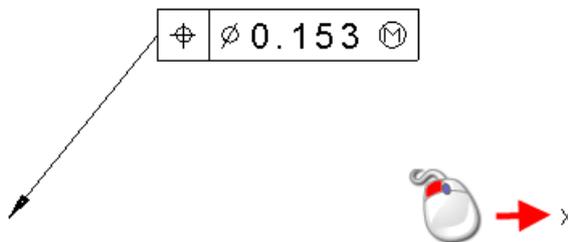
- 1 Select only the label of the balloon.



- 2 From the **Edit** menu, select **Modify** to display the **Add Leaders** dialog. This is the same as double-clicking on the label.



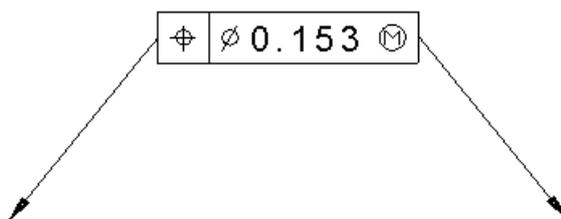
- 3 Click one of the leader heads on the dialog. Do not click **OK** yet.
- 4 Click the position where you want the pointer of the leader.



The new leader leaves the balloon from the nearest edge of the label.

You can force the leader to leave the corner of a balloon with a box label. This is done by turning on the **Put leaders at frame corner** option on the **Balloon** page of the **Options** dialog.

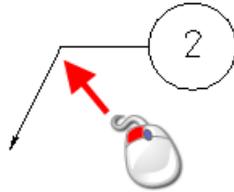
A new leader is drawn from the balloon.



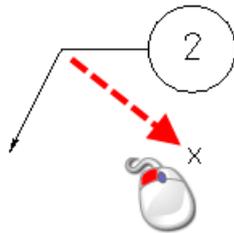
- 5 Continue adding more leaders by selecting a type of leader and then entering its position.
- 6 Click **OK** to close the dialog.

Repositioning a dog-legged pointer

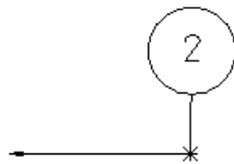
- 1 Select the point object which joins two lines of the pointer of the balloon. A point object looks like a star.



- 2 Drag the point object to a new position.



This gives the balloon shown below. The ends of two lines joining the point object move too.



Dimensions

A dimension is a measurement of a length or an angle.

- Dimensions are added to a model to:
 - check it is the correct size.
 - pass information to another engineer.
- You can specify dimensions as:
 - linear dimensions (see page 190).
 - radius and diameter dimensions (see page 191).
 - angular dimensions (see page 191).
- Dimensions consist of leaders, witness lines, marks, and annotation. These are the dimension attributes (see page 191).

Add dimensions to a drawing by:

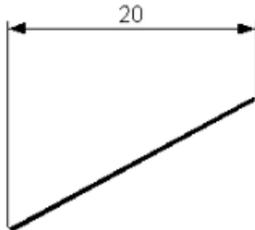
- Creating a dimension (see page 192).
- Editing a dimension (see page 218).



You can also use the measuring features on the calculator to check the model for the correct dimensions.

Linear dimensions

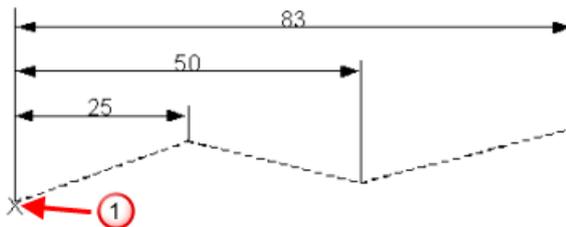
A linear dimension measures the length of a line or the distance between two points.



Linear dimensions can also be:

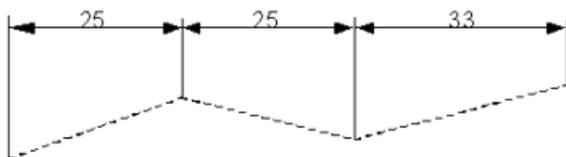
- basepoint
- chain
- ordinate.

Basepoint dimensions

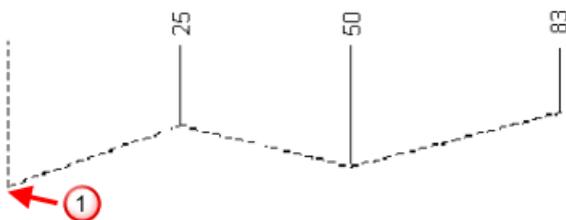


① Basepoint

Chain dimensions



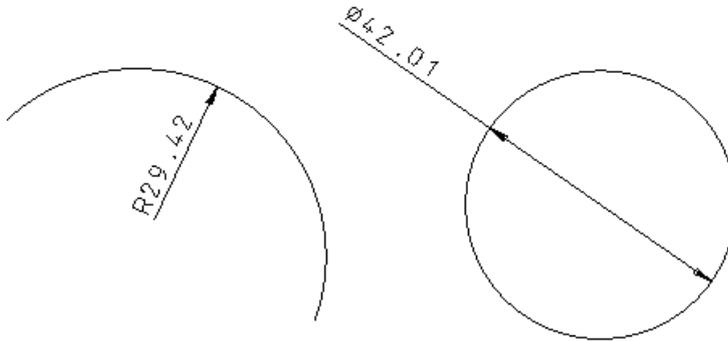
Ordinate dimensions



① Origin point

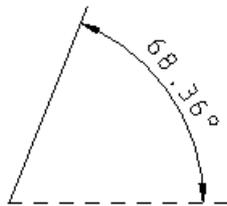
Radius and diameter dimensions

A radial dimension measures the radius or diameter of an arc or a circle.



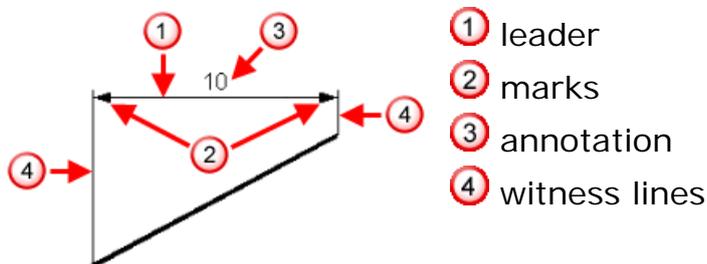
Angular dimensions

An angular dimension measures an angle.



Dimension attributes

Dimensions consist of leaders, witness lines, marks, and annotation. These are the attributes of dimensions.



Leaders are the lines or arcs that represent the dimension values. *Witness lines* are those that connect the leader to the item. *Marks* are the ends of the leader, that touch the witness lines. *Annotation* is the text associated with the dimension.

You can edit each attribute to control the appearance of a dimension on a drawing. For example, the annotation can be:

- forced to be horizontal.
- embedded into the dimension leader.

- displaced from the leader.

Creating a dimension

You can add the following types of dimension to a drawing

- Automatic (see page 194)
- Between two parallel lines (see page 199)
- Radius (see page 200)
- 3 point radius (see page 200)
- Diameter (see page 201)
- Angular (see page 202)
- Base-point (see page 205)
- Chain (see page 208)
- Ordinate (see page 211)
- Angled (see page 214)
- Dimension a view (see page 215)

Entering dimension-creation mode

- 1 Click the **Drafting Mode**  button on the **Main** toolbar. The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Click the **Annotation**  button to display the **Annotation** toolbar (see page 116).
- 3 From the **Object** menu, select **Dimension** to display the **Dimension creation** options.



You can also see these options on the **Drafting** toolbar. Click  to display the **Annotation** toolbar (see page 116) that is used to edit attributes of dimensions (for example, text).

The dimension annotation options are shown in ③ below on the **Annotation toolbar**:

③ Dimensions



Each **Dimension creation** option is described below:



Automatic dimension (see page 194)



Parallel dimension (see page 199)



Radius dimension (see page 200)



3 point radius dimension (see page 200)



Diameter dimension (see page 201)



Minor angle dimension (see page 204)



Major angle dimension (see page 204)



Base-point dimension (see page 205)



Chain dimension (see page 208)



Ordinate dimension (see page 211)



Dimension a view (see page 215)



If dimensions are created on a model rather than a drawing, they are projected onto the principal plane of the current workspace.



Straight line segments of curves are treated as lines.



We suggest you create a solid of your model where possible before drafting from it. This will improve the accuracy of dimensioning. For further details, see Creating a solid

Creating an automatic dimension

The type of object you select determines the type of dimension created. For example:

- If you select a line, a linear dimension is created.
- If you select an arc, a radius dimension is created.
- If you select a circle, a diameter dimension is created.

Use automatic dimensions to create linear dimensions between two positions by entering two positions.

If no automatic dimension is available for an object, PowerSHAPE accepts the selection position as the first position of a two-position dimension.

Selecting a line

1 Display the **Drafting** toolbar.



2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.

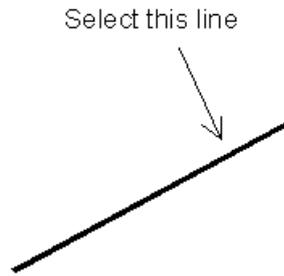


3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.



Click the **Automatic dimension**  button to display the Dimension toolbar (see page 222).

4 Select a line.



Ensure you do not click a key point (the end of the line). If you click a key point by mistake, PowerSHAPE assumes you want to create an automatic dimension between two positions and displays a rubber band line from the position to the cursor. To continue, start from step 1 again.

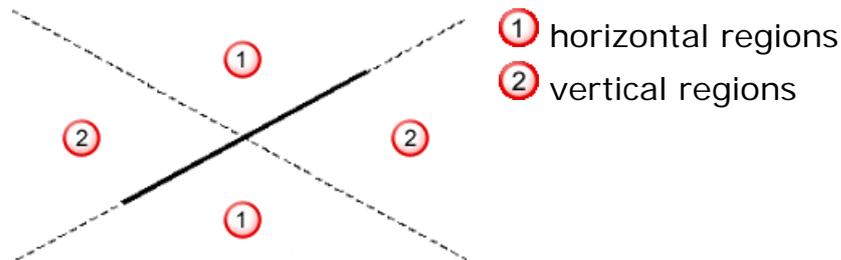


Clicking the midpoint selects the whole line segment. Midpoints are no longer treated as key points.

5 Enter the position of the text.

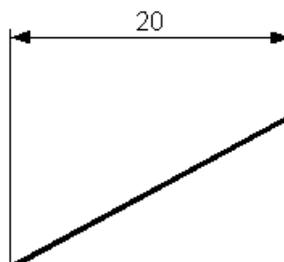
A vertical or horizontal dimension is created depending on where you input the text position. Possible horizontal and vertical dimensions are displayed as you move the mouse.

For our example, the regions of the possible horizontal and vertical dimensions are shown below.

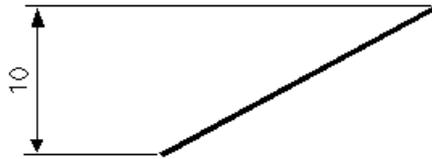


Move the cursor until the dimension you want is displayed.

You can either create a horizontal dimension as shown below,



or a vertical dimension as shown below.



If you want the true dimension of the line, click the **Parallel dimension**  button in the **Dimension** toolbar.

Selecting an arc/circle

1 Display the **Drafting** toolbar.



2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.



3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.



Click the **Automatic dimension**  button to display the Dimension toolbar (see page 222).

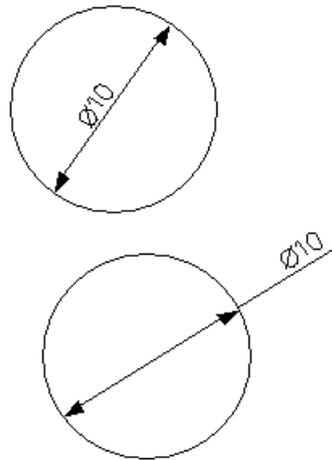
4 Select an arc or a circle.

Ensure you do not click any key points. If you click a key point by mistake, PowerSHAPE assumes you want to create an automatic dimension between two positions and displays a rubber band line from the position to the cursor. To continue, start from Step 1 again.

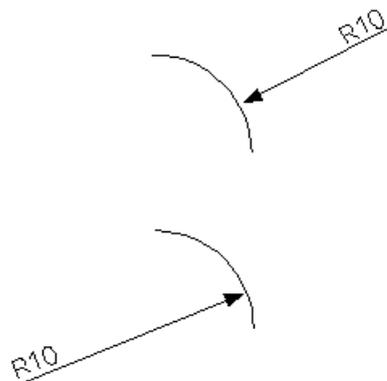
5 Input the position of the text.

If an arc is selected, the dimension of the radius is created. If a circle selected, then the dimension of the diameter is created.

For a circle, the dimension is created as shown below. The text position determines whether the text is drawn inside or outside the circle.



For an arc, the dimension is created as shown below. Again, the text position determines where the dimension text is drawn.



Clicking two positions

- 1 Display the **Drafting** toolbar.

- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.

- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.

- 4 Click the **Automatic dimension**  button to display the Dimension toolbar (see page 222).

- 5 Click two positions.

- a Click the first position:



A *rubber-band* line is attached from this position to the cursor.



- b Click the second position. The rubber-band line is no longer displayed. The dimension value is displayed and moves with the cursor.

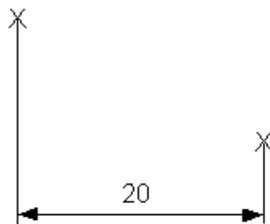


If you clicked the first position on an arc (see page 196) or a line (see page 194) and not on any of their key points, the automatic dimension for those objects is created. If this is not what you want, start again from Step 1.

- 6 Click to set the dimension position.

A vertical or horizontal dimension is created depending on where you click the dimension position. This is similar to when you select a line (see page 194).

You can either create a horizontal dimension:



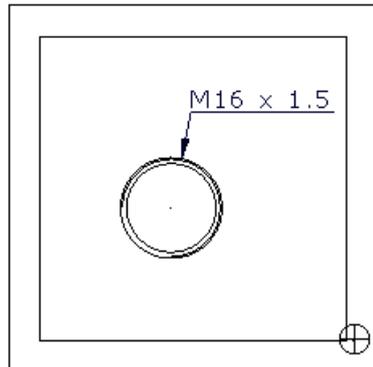
or a vertical dimension:



Dimensioning a tapped hole

If a tapped hole is dimensioned around the perimeter of the hole, the thread details are displayed. The information that is displayed depends on the standard of the tapped hole.

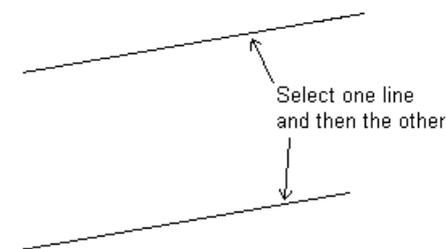
- The ISO metric standard uses *thread designation/nominal diameter/pitch/optional tolerance*. The following example shows, a 16mm IMF hole. This is dimensioned as *M16 x 1.5*, showing the diameter of the hole and the pitch of the thread.



- The American standard uses *nominal diameter/threads per inch/thread designation*. For example, *3/4 -24 UNF*.
- The British standard uses *nominal diameter/thread designation*. For example, *1/4 BSP*.

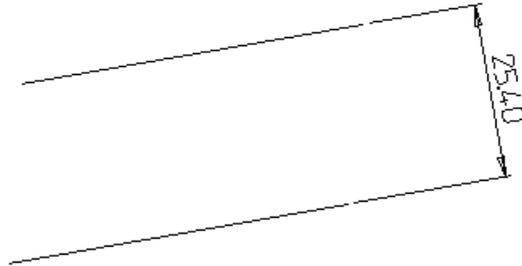
Creating a dimension between two parallel lines

- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.
- 4 Click the **Parallel dimension**  button to display the Dimension toolbar (see page 222).
- 5 Select a line.
Ensure you do not click any key points. If you click a key point by mistake, start from Step 1 again.
- 6 Select another line or click a point.



If you click a point, the dimension is measured perpendicular from the line to the point.

- 7 Input the text position.



Creating a radius dimension

This is used to create a radius dimension on a circle.

- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.
- 4 Click the **Radius**  button.
- 5 Select a circle.
- 6 Input the text position.

Creating a 3 point radius dimension

This creates a radius dimension using three points. This is ideal for dimensioning the radius of curvature on a surface or curve.

- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.
- 4 Click the **3pt Radius**  button.

- 5 Click the first position to define the start point of an arc.



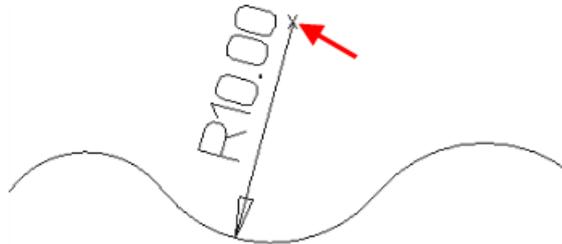
- 6 Click the second position to define the end point of an arc.



- 7 Click the third position to define a position on the circumference of the arc.

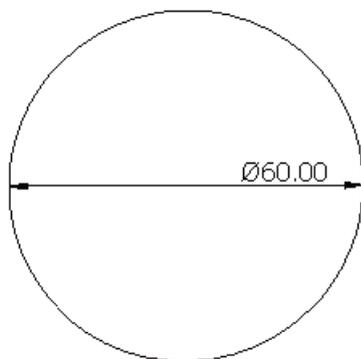


- 8 Click the text position to add the dimension.



Creating a diameter dimension

This creates diameter dimensions on circles and lines. For example, you may want to show the diameter of the top view of a cylinder as shown below.



- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.



Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.



- 3 Click the **Diameter**  button.
- 4 Select an arc or a line, or enter two positions.
- 5 Input the text position.

Creating an angular dimension

Angles are measured between two lines (see page 202).

Between any two lines, there are two angles: a minor and a major angle. The minor angle is less than or equal to 180° and the major angle is greater than or equal to 180° .



The intersection position of the two lines defines the pivot of the angle.

You can:

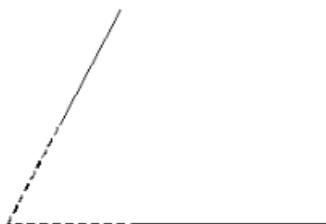
- create a minor angle dimension (see page 204).
- create a major angle dimension (see page 204).
- create an angular dimension on an arc (see page 204).

Creating an angular dimension

There are several ways of defining the two lines.

Select two lines

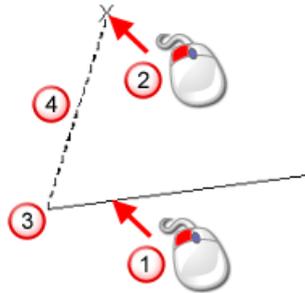
Select two lines. If the two lines intersect, PowerSHAPE finds their intersection to determine the angle (even if PowerSHAPE has to extend the lines to find their intersection).



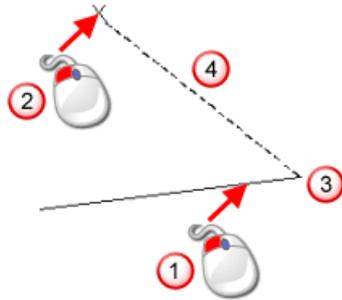
Select one line and click an end point

Select a line **1** and then click a position to set the end point of a second line **2**. The end point of the first line nearest to where you clicked to select it **3** becomes the end point of the second line **4**.

For example, selecting this line nearest its left end point results in:

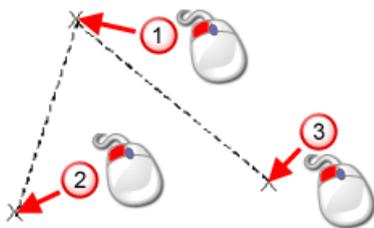


Selecting the same line nearest its right end point results in:



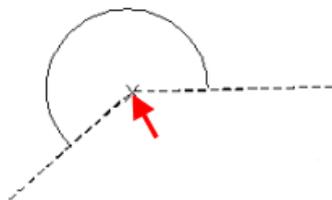
Click three end points

You can define two lines by clicking three positions. The first line is defined between click **1** and click **2**. The second line is defined between click **1** and click **3**.



Select an arc

If you select an arc, the two lines are defined from the arc centre:



Creating a minor angle dimension

- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.
- 4 Click the **Minor Angle**  button.
- 5 Define two lines (see page 202).
- 6 Click to set the text position.

Creating a major angle dimension

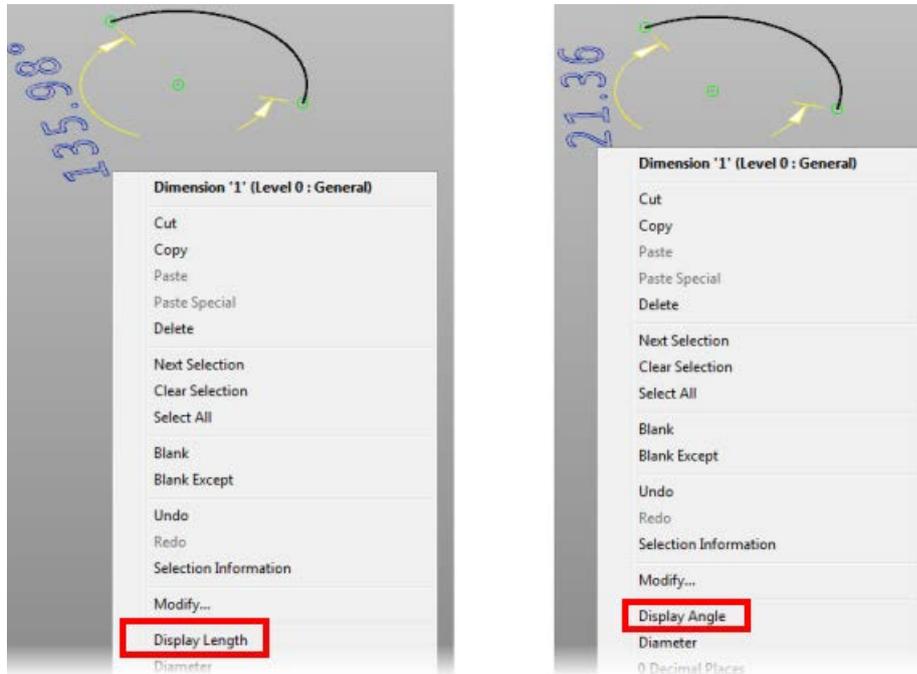
- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.
- 4 Click the **Major angle**  button. The Dimension toolbar (see page 222) is displayed. This can be used to edit attributes of the new dimensions, for example, text.
- 5 Define two lines (see page 202).
- 6 Click to set the text position.

Creating an angular dimension on an arc

When you create an angular dimension on an arc, you can toggle between displaying the length and displaying the angle.

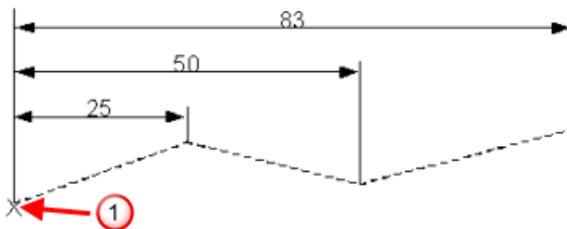
- 1 Create a three-point arc.
- 2 Click the **Minor Angle**  button.
- 3 Click the centre of the arc and then the two end points of the arc.
- 4 Move the cursor to required position and click to fix the position of the dimension information.

- 5 Place the cursor on the dimension and right-click to display the **Dimension** context menu.
- 6 Use the options **Display Length** and **Display Angle** to toggle between displaying the length and displaying the angle



Creating base-point dimensions

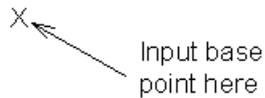
Base-point dimensions are those from a common position (the base point) to another point, for example:



① Basepoint

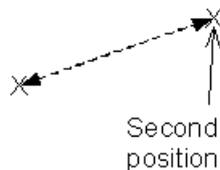
- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.
- 4 Click  (annotation toolbar).

- 5 Click on the drawing to position the base point.



- 6 Input the position you want to dimension to.

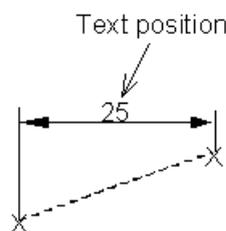
A rubber-banded line is attached from the base-point to the mouse cursor.



- 7 Input the text position.

A vertical or horizontal dimension is created depending on where you input the text position. Possible horizontal and vertical dimensions are displayed as you move the mouse.

In our example, we input the following text position.

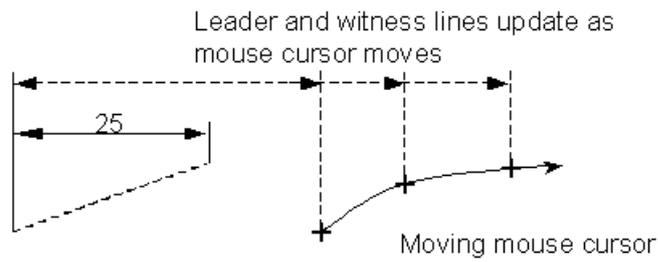


The orientation of this dimension is used by all of the other dimensions in the base-point dimension.

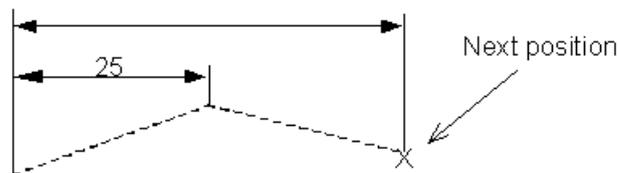
The example dimension is horizontal. This means that all the other dimensions created in this base-point set will be horizontal too.

- 8 Input a position for the next dimension.

A rubber-banded dimension is attached to the base point and the mouse cursor. As you move the mouse, the leader and witness lines of the rubber-banded dimension vary.



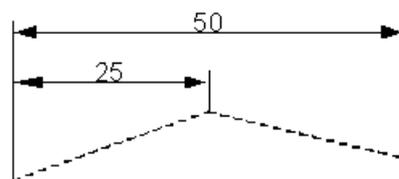
The distance of the leader from the leader of the first dimension is a fixed offset (as defined on the **Dimensions** page of the **Options** dialog).



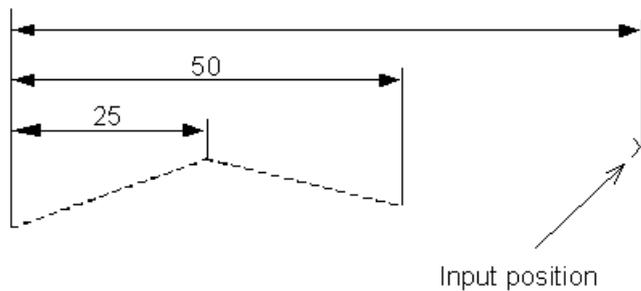
The dimension is added to the drawing. The text is automatically centred on the leader. You can deselect **Centred text** on the **Dimensions** page of the **Options** dialog.



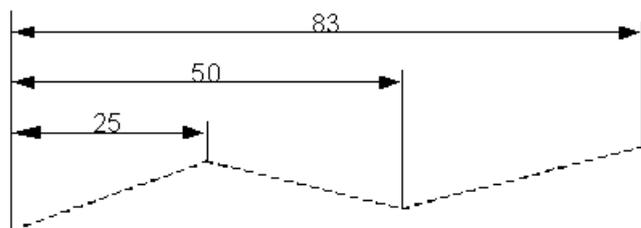
*If you deselect **Centred text**, you must click a position to place the text on the leader.*



- 9 Continue inputting positions to complete the dimensions.



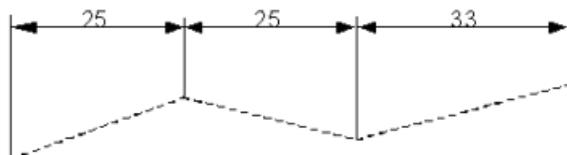
With **Centred text** selected, each new dimension requires only one mouse click.



- 10 To finish creating base-point dimensions, click **Select** .

Creating a chain of dimensions

A chain of dimensions is a group of dimensions linked together such that where one dimension finishes the next one starts. An example is shown below.

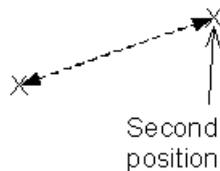


- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.
- 4 Click  (dimension creation menu).
- 5 Input the start position of the chain.

A rubber-banded line is attached from the start point to the mouse cursor.



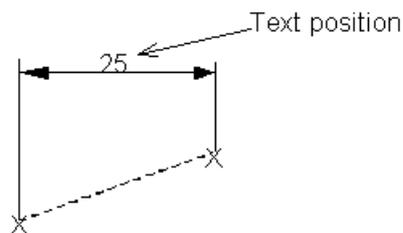
6 Input the position you want to dimension.



7 Input the text position.

A vertical or horizontal dimension is created depending on where you input the text position. Possible horizontal and vertical dimensions are displayed as you move the mouse.

In our example, we input the following text position.



The orientation of this dimension determines that of all the other dimensions in the chain dimension.

Our dimension is horizontal. This means that all the other dimensions created in this chain set will be horizontal too.

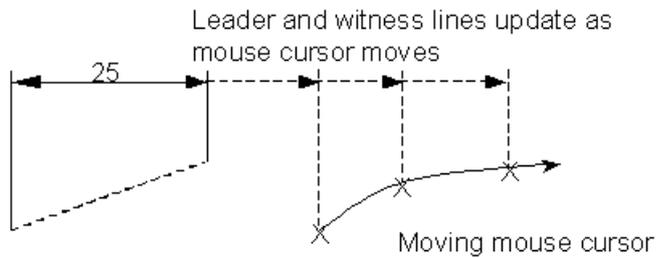
The text is automatically centred on the leader. If you do not want centred text, set the **Centred Text** option on the **Annotation** options dialog to *OFF*. For further details, see Dimension defaults - Annotation options (see page 232).



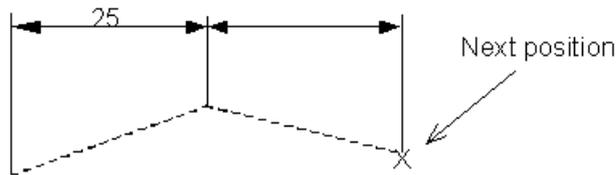
If you deselect **Centred text**, you must click a position to place the text on the leader.

8 Input the position of the next dimension.

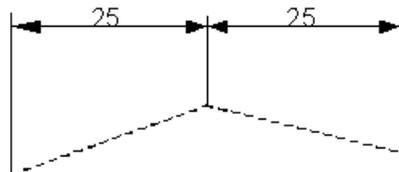
You will notice a rubber-banded dimension attached to the end of the first dimension and the mouse cursor. As you move the mouse, the leader and witness lines of the rubber-banded dimension vary.



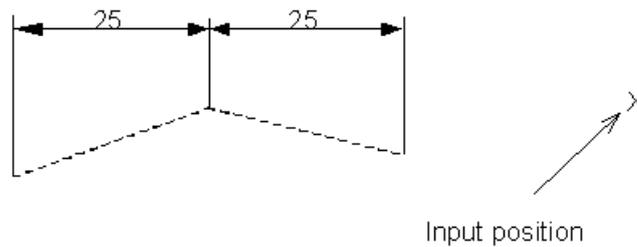
In our example, we input the following position.



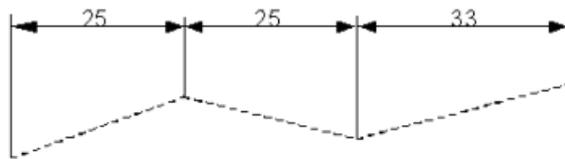
The dimension is added to the drawing. Notice that the text is automatically centred on the leader.



- 9 Continue inputting positions to complete the chain of dimensions.



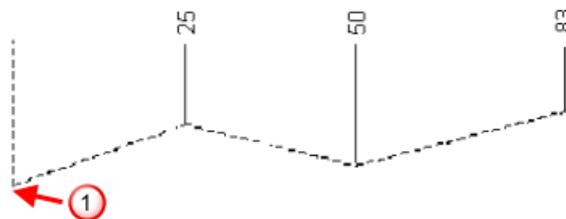
With **Centred text** selected, each new dimension requires only one mouse click.



- 10 To finish creating chain dimensions, click **Select** .

Creating ordinate dimensions

Ordinate dimensions are created along an axis of the principal plane of the current workplane. These dimensions measure distances from a common origin. An example is shown below. Notice that no leader line is drawn.

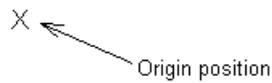


- ① Origin point

The dashed lines are drawn for illustration purposes and are not displayed when you create such a dimension.

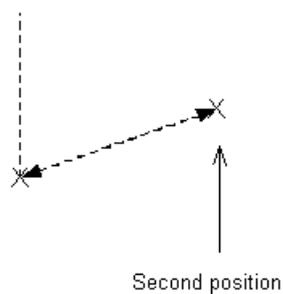
- 1 Display the **Drafting** toolbar.
- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Click  (main toolbar). The Annotation toolbar (see page 116) is displayed. This can be used to edit attributes of the new dimensions, such as text.
- 4 Click  (annotation toolbar).

- 5 Input the origin of the ordinate dimension.



- 6 Input the position you want to dimension.

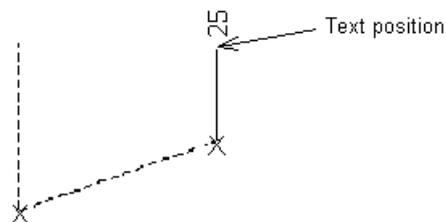
A rubber-banded line is attached from the base-point to the mouse cursor.



- 7 Input the text position.

A vertical or horizontal dimension is created depending on where you input the text position. Possible horizontal and vertical dimensions are displayed as you move the mouse.

In our example, we input the following text position.

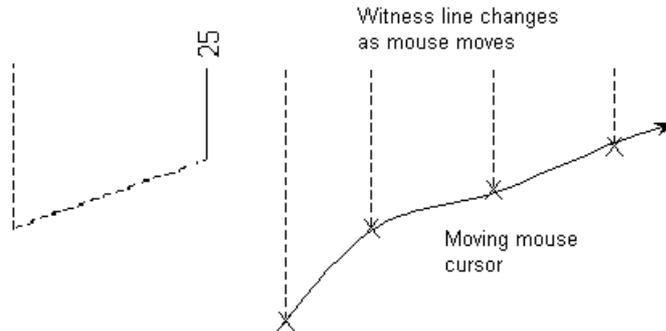


The orientation of this dimension determines that of all the other dimensions in the ordinate dimension in this series.

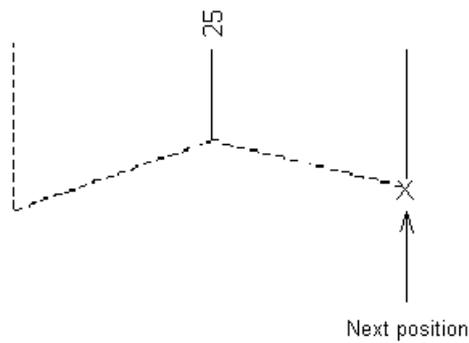
Our dimension is horizontal. This means that all the other dimensions created in this ordinate set will be horizontal too.

- 8 Input the position for the next dimension.

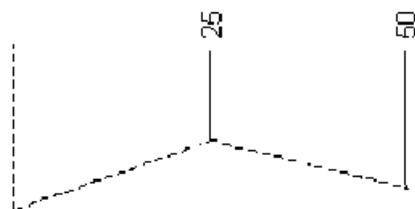
A rubber-banded witness line is attached to the mouse cursor. As you move the mouse, it varies.



One end of the witness line remains in line with the text position.



Once you input the dimension the value is displayed.



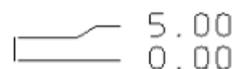
The text of the new dimension is automatically centred with that of the first dimension.

If you do not want centred text, deselect the **Centred Text** option on the **Drafting > Dimensions > Annotation** (see page 232) page of the **Options** dialog.



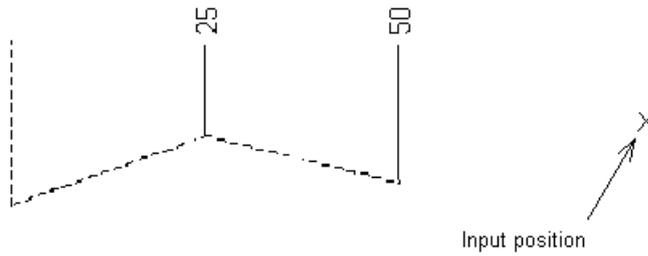
*If you deselect **Centred text**, you must click a position to place the text on the leader.*

If the text of the new dimension is too close to the text of the previous one, its leader line is automatically created with a dog leg.

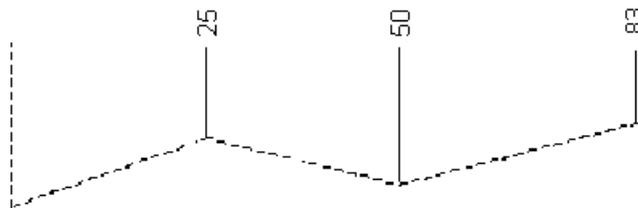


You can set the type of dog leg on the **Drafting > Dimensions > Lines** page of the **Options** dialog.

- 9 Continue inputting positions to complete the dimensions.



With **Centred text** selected, each new dimension requires only one mouse click.

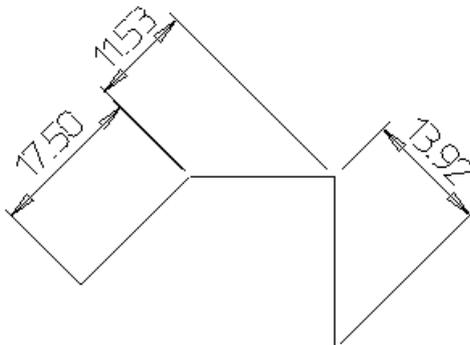


- 10 To finish creating ordinate dimensions, click the **Select** button

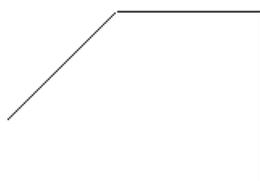


Creating angled dimensions

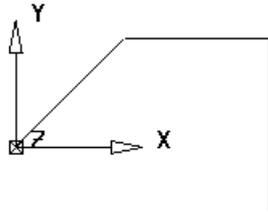
You can create dimensions projected at any angle.



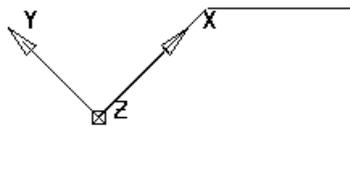
- 1 Create the geometry.



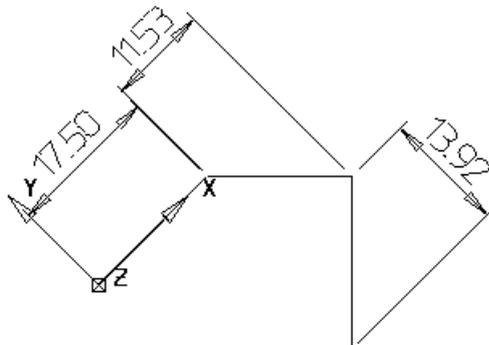
- 2 Create a workplane.



- 3 Position the workplane such that the principal plane is at the angle you wish to project the dimensions.



- 4 Create the dimensions.



Creating dimensions in a view

This is used to create dimensions automatically for a selected drawing view.



The **Dimension a view**  option is available only when a drawing is displayed.

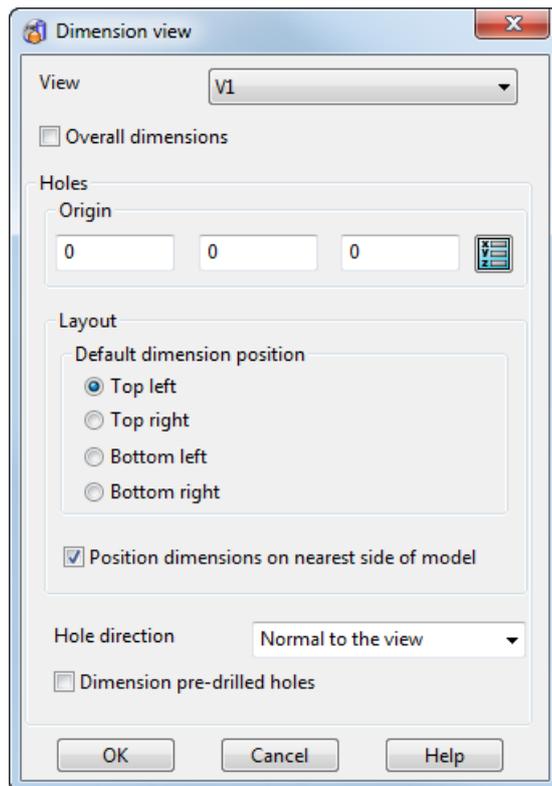
- 1 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Open an existing drawing (see page 10) or create a new drawing (see page 8).

- 3 Click  (drafting toolbar)
- 4 Click  (annotation toolbar).
- 5 Use the **Dimension view** dialog (see page 216).

Dimension view dialog

Use this dialog to:

- Create dimensions of holes (ordinate dimensions).
- Create overall item dimensions (default dimensions).



View — Select the view from the list, or click inside the required view in the drawing.

Overall dimensions — Select this option to displays the **X** and **Y** dimensions on the item in the selected view, if the dimensions do not already exist.

Holes:

Origin — The origin is set to the *view origin* by default. If you change the origin, the *view origin* updates to the new position. The origin is marked by centrelines.

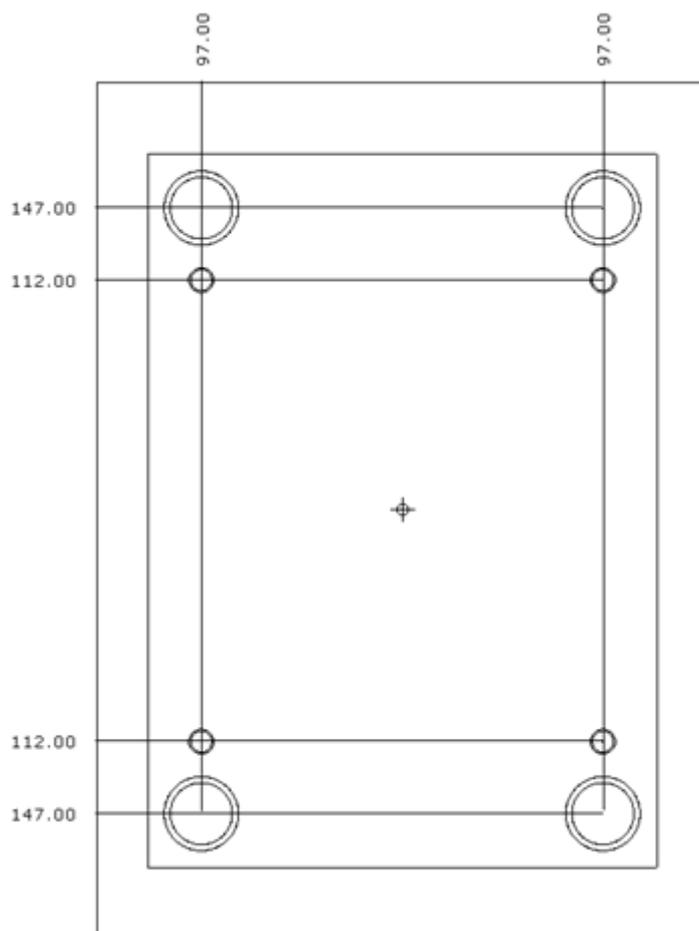
To change the origin:

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

Layout — You can choose the layout for the text of the ordinate dimensions at the following positions:

- **Top Left** (default)
- **Top Right**
- **Bottom Left**
- **Bottom Right**

The example image of a view below shows the layout of hole dimensions using the **Top Left** (default) layout option.



Position dimensions on nearest side of model — Select this option to position the dimension text on the side nearest to the hole centre. This is useful for plates with hundreds of holes, because it improves the clarity of the drawing.

Deselect the option to position all the dimensions on one side; the position of the dimension text is specified by selecting **Top Left**, **Top Right**, **Bottom Left**, or **Bottom Right**.

Hole direction — Select which holes to dimension:

Normal to the view — Select this option to include only the holes normal to the view.

At any angle — Select this option to include holes at any angle.

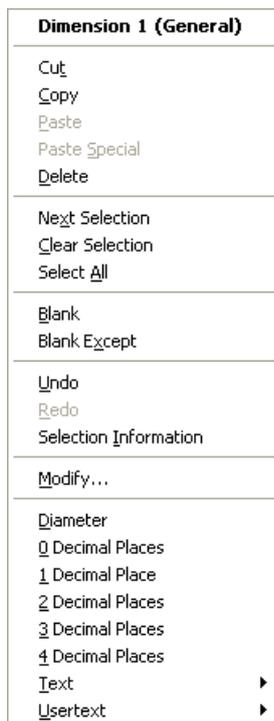
Dimension pre-drilled holes — Select this option to allow plates with pre-drilled holes to be dimensioned.

If dimensions are close together, the dimension leader lines are drawn doglegged automatically to prevent overlapping.

Editing a dimension

With a dimension selected, you can edit the dimension in the following ways:

- You can delete the dimension(s) by clicking **Delete**  or pressing the **Delete** key.
- **Edit Modify** displays the **Dimension** toolbar.
- Double-click the dimension to display the **Dimension** toolbar.
- Right-click the dimension to display the edit options and select **Modify**. Use the Dimension dialog (see page 227) to edit the dimension..
- Right-click the dimension to display the edit options available for dimensions. The name of the dimension and its level is at the top of the context menu.

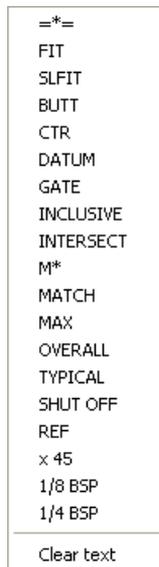


Options, unique to this context menu, are discussed below.

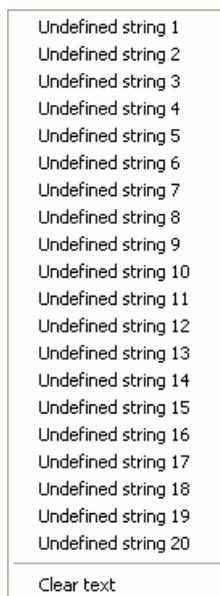
Diameter — adds a diameter symbol to the dimension.

0 - 4 Decimal Places — restricts the dimension value to the given number of decimal places.

Text — displays the context menu given below, which adds text to your dimension.

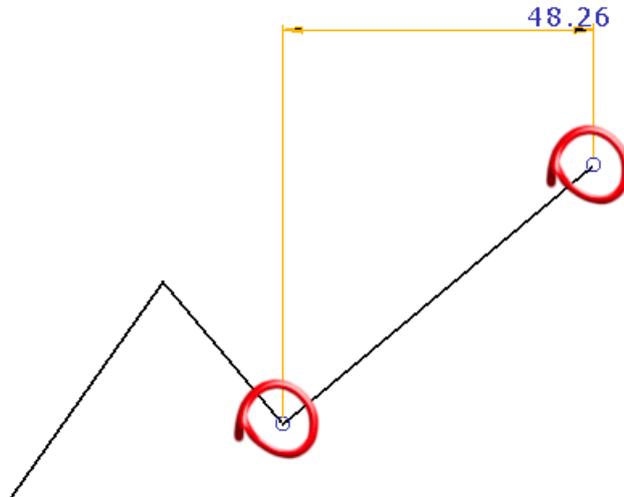


User text — displays the context menu given below. This contains a list of strings you can define to add before and after dimension values. The strings are defined on the **Annotation** page of the **Dimension** dialog.



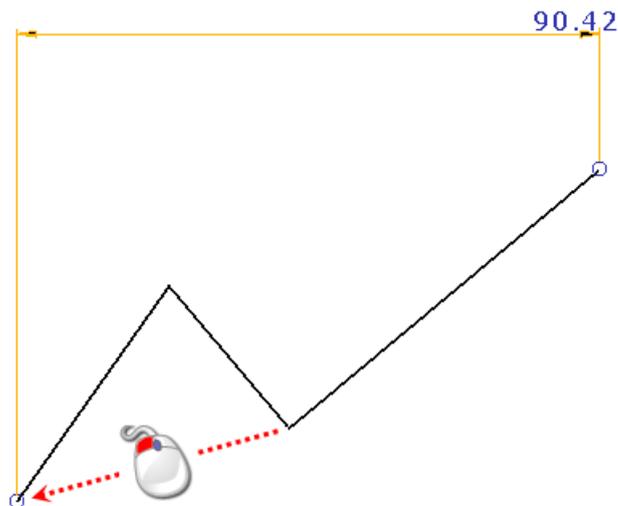
- You can update the geometry (see page 221) of an object that is dimensioned.
- You can specify settings for dimensions using the **Drafting** options on the **Options** dialog. For further details, see Editing Dimension defaults using the Options dialog (see page 232).

- To move the dimension text, drag it.
- To move the crank on the dog leg of an ordinate dimension, drag it.
- The witness lines on dimensions have instrumentation handles to let dimensions be edited.



To edit a dimension:

- 1 Select the dimension.
- 2 Click on the handle and drag to the new position.



For multi-point dimensions such as **Basepoint**, **Chain**, and **Ordinate**, you can edit the dimension when completed.

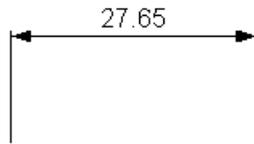


*This functionality does not apply to **Angle** dimensions.*

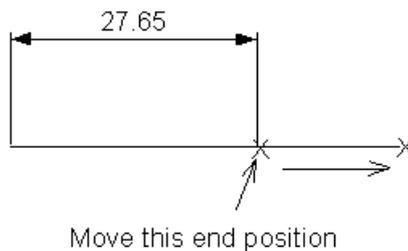
Updating the geometry of an object

When you create a dimension by clicking an object, the dimension becomes related to the object. If you edit the geometry of the object, the dimension is updated too.

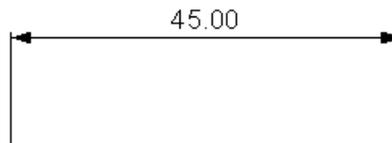
Suppose the following dimension is created by clicking the line.



When one of the end points of the line



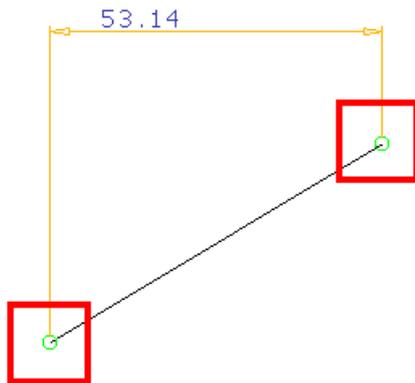
The dimension is updated too.



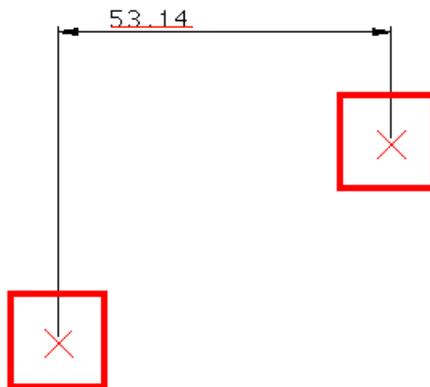
Dimension instrumentation

Red crosses are shown when dimensions need attention because a link (dependency) between the dimension and geometry is broken.

The example below shows a dimension on a diagonal line showing its links (dependencies) on the line's *ends*.



If the line is deleted, a red cross is displayed and the dimension value is underlined in red to show that a link (dependency) is broken.



To repair the dimension and remove red crosses, do the following:

- 1 Drag each cross (data point) on the dimension and snap it to the correct geometry to create a new link (new dependency).
- 2 Delete the dimension.

Dimension toolbar

This toolbar is used to edit attributes of dimensions.



If you edit an attribute of a completed dimension in either creation or selection mode, then only the attribute of that dimension changes.

If you edit an attribute whilst creating a dimension, the attribute becomes fixed and is used to create all future dimensions in the current creation session.

Dimension attributes changed using the **Dimension** toolbar override future changes made using the **Options** dialog. If you edit any attribute on the **Dimension** toolbar, it remains fixed to the dimension. So if you now change that item on the **Options** dialog, it does not change that item for the edited dimension.

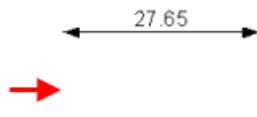
Each item on the toolbar is discussed below:

Tahoma  — Font used in the text.

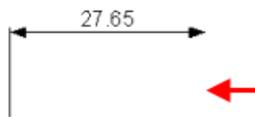
Height — Edits the height of the text.

 — When selected, the text is italic. You can also use **Ctrl + I** to turn this button on and off.

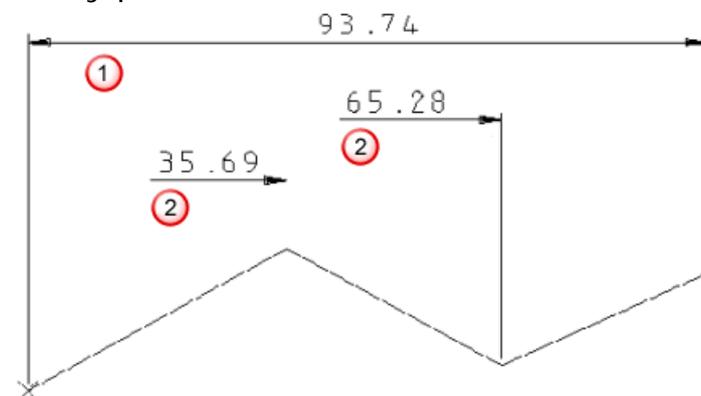
 — Deselect this button to remove the first witness line from the dimension.



 — Deselect this button to remove the second witness line from the dimension.

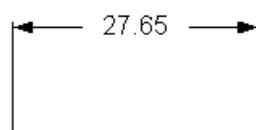


 — Trims the leader line to the position of the dimension annotation. This can be used when the end of the leader is obvious, for example, in base-point and chain dimensions. Trimming leaders is only possible for linear dimensions.



- ① **Trim leader to text** deselected — The leader line is not trimmed.
- ② **Trim leader to text** selected. The leader line is trimmed to the dimension annotation.

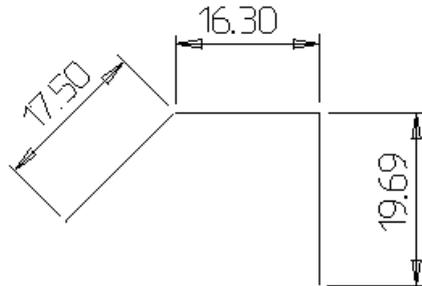
 — Embeds the text in the leader line.



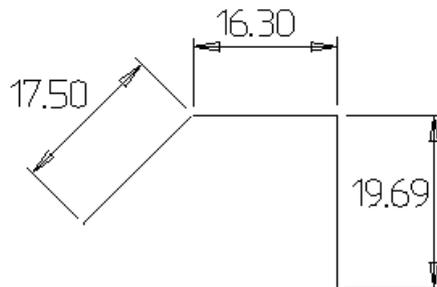


— This changes all text so that it is positioned horizontally to the principal plane.

Suppose dimensions are angled as shown below.

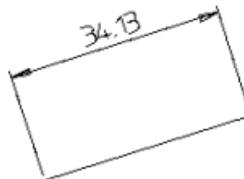
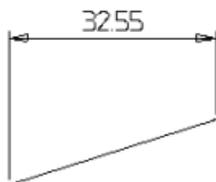


When text is changed to horizontal, the dimensions look like:



— This gives true measurements for linear dimensions. When on, a projected linear dimension changes to display the true dimension. When off, the true dimension changes back to projected.

Projected dimension **True dimension**



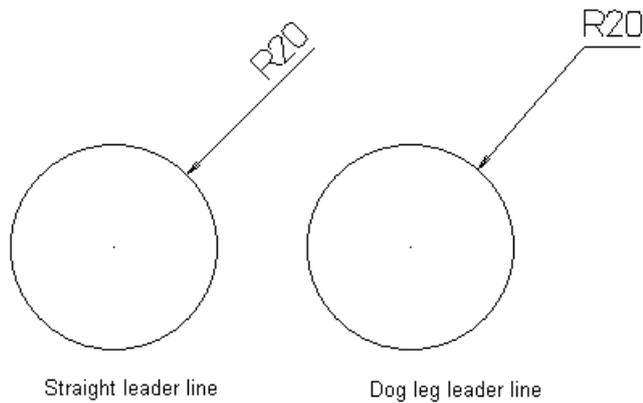
A linear dimension measures the length of a line or the distance between two points.



— When on, the diameter symbol is displayed before the dimension value. If the dimension is the radius of an arc, then the value changes to its diameter.



— This toggles the leader lines of radial and ordinate dimensions from straight to dog leg.



— This list sets the number of decimal places or denominator of the fractional part of the dimension value. You can choose an option from the following menu.

- 0
- 0.1
- 0.12
- 0.123
- 0.1234
- 1/2
- 1/4
- 1/8
- 1/16
- 1/32
- 1/64

Scale — Scales the value of the dimension. In your model, you can create objects using a different scale. To create dimensions of the 'true' values, you can enter the value of the scale in the **Scale** box.

The dimension of the following line has value 60 and scale 1.



If we change the **Scale** to 3, the dimension value becomes 20.



When you change the scale factor, a warning is displayed.

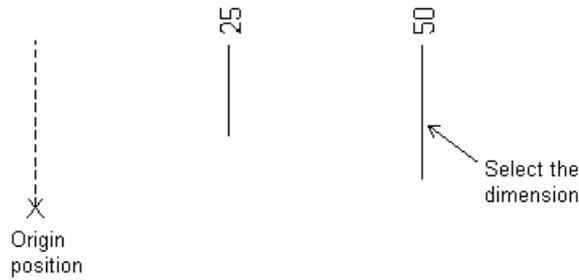
 — Resets the attributes to the settings on the **Dimension** pages on the **Options** dialog.

 — Restart creating base-point, chain and ordinate dimensions.

You can restart base-point, chain and ordinate dimensions without entering the origin or the text position again.

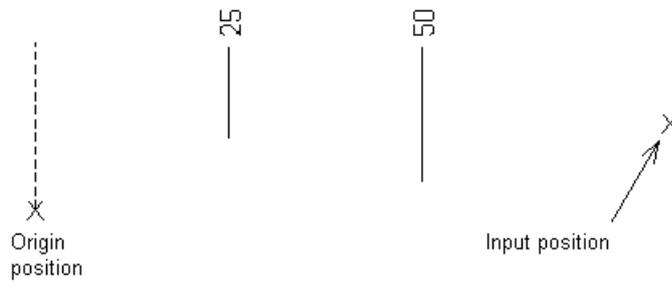
1 Select the dimension.

If you are restarting a set of base-point or chain dimensions, select the last one.

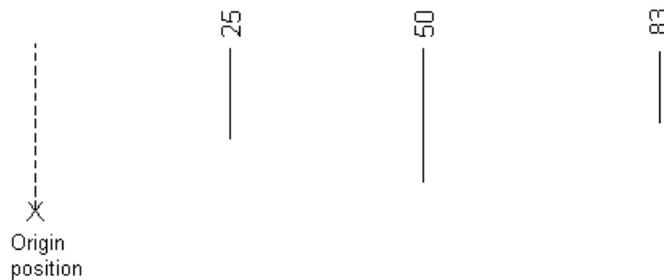


2 Select the **Restart multi-part dimension** button .

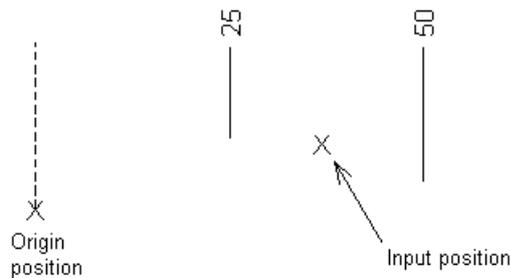
3 Click positions on the screen to continue creating new dimensions.



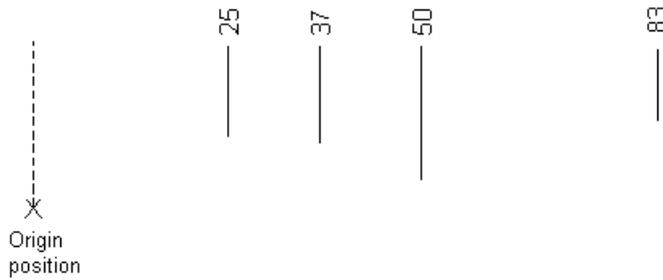
A new dimension is added.



With ordinate dimensions, you can also input positions between dimensions.



A new dimension is added between the existing dimensions.



 — Displays the Annotation page of Dimension dialog (see page 227).

 — Displays the Tolerance page of Dimension dialog (see page 229).

 — Displays the Style page of Dimension dialog (see page 231).

 — Displays the Mark page of Dimension dialog (see page 231).

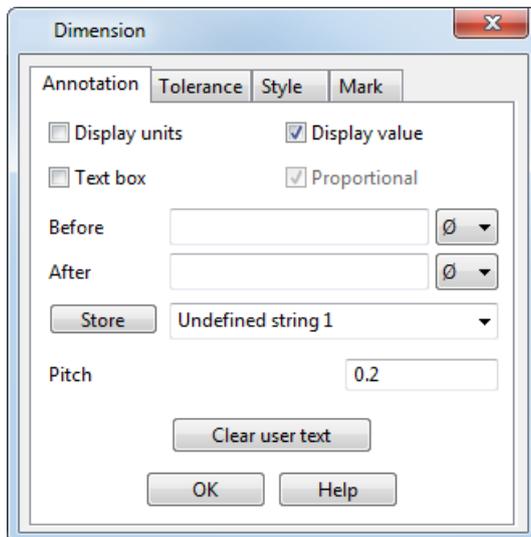
Dimension dialog

The **Dimension** dialog contains these tabs:

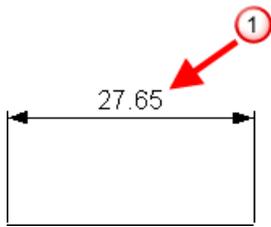
- **Annotation** (see page 227)
- **Tolerance** (see page 229)
- **Style** (see page 231)
- **Mark** (see page 231)

Annotation tab

This edits the annotation of the dimension.



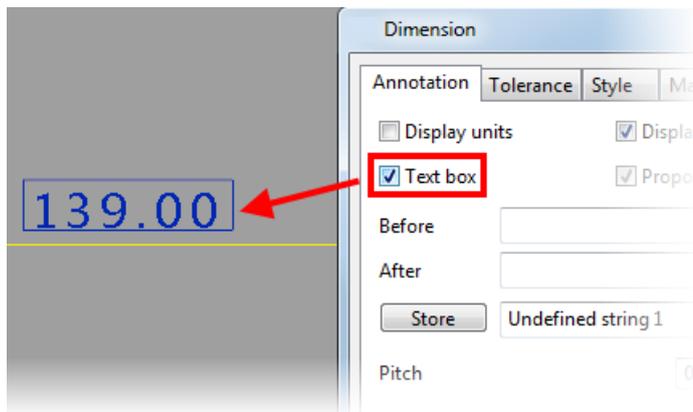
Annotation is the text of the dimension.



Display units — When on, this displays the current units after the dimension value.

Display value — This displays and removes the automatically measured dimension value. Turn on to display the dimension value and turn off to remove it.

Text box — Select this option to add a text box around the dimension.



Proportional — (Duct font only) When on, the space occupied by each character is not fixed.

Before — In the text box, enter text you want to appear before the dimension value. The combo box next to the text box contains three symbols: the diameter, the degree and the plus and minus symbols. If you select a symbol from the combo box, it is inserted into the text box.

After — In the text box, enter text you want to appear after the dimension value. The combo box next to the text box contains three symbols: the diameter, the degree and the plus and minus symbols. If you select a symbol from the combo box, it is inserted into the text box.

Store — This allows you to store **Before** and **After** strings to use in other dimensions.

- 1 Input the strings you want to store in the **Before** and **After** boxes.
- 2 Use the selector on the right of this button to select a string option.
- 3 Click the **Store** button. The string is now stored.

The number 88.88 appears in the string's option to show where the dimension value will lie in relation to the string.



*If the **Before** and **After** boxes are empty, the string in the selector becomes undefined.*

You can select the stored string later by:

- Using the selector on this dialog
- Right clicking on the dimension to display the **Dimension** pop up menu and then select **User text** followed by the required string.

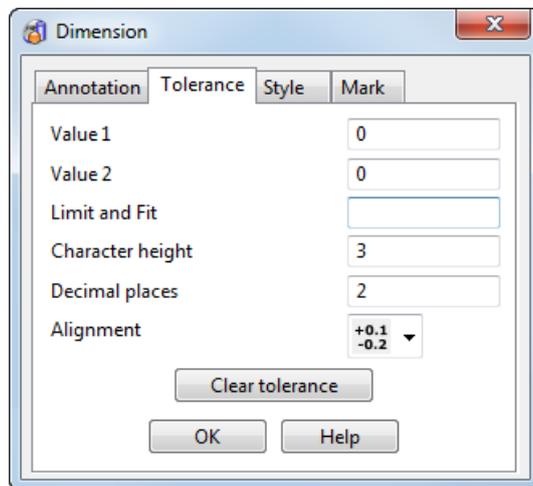
Pitch — Enter a character pitch value to change the spacing of annotation text on an individual dimension or a selection of dimensions.

Clear user text — This button clears any text in the **Before** and **After** text boxes. It also turns **Display units** off.

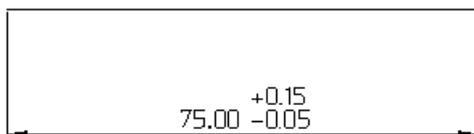
OK — Removes the dialog.

Tolerance tab

This edits the tolerance of the dimension.



Each dimension can have two tolerance values associated with it. The tolerance values show the range within which a dimension value can vary.



If you are in dimension creation mode and you change values on this dialog, the values are stored for the current session of dimension creation.

Value 1 — One value of the range.

Value 2 — Second value of the range.

Limit and Fit — Enter limit and fit codes here. The tolerances for holes and shafts are automatically added to the dimensions.

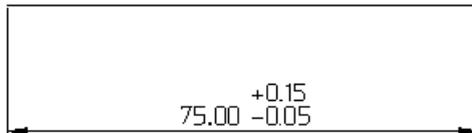
Character height — Text height of the tolerance values.

Decimal places — Number of decimal places the tolerance values can take.

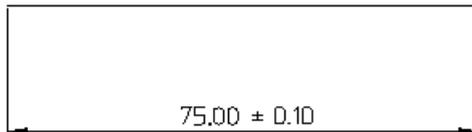
Alignment — You can edit the tolerance to display in three formats.



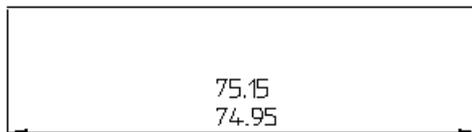
- If you select the first format  and enter **0.15** for **Value 1** and **-0.05** for **Value 2**, a dimension is displayed similar to that shown below.



- If you select the second format  and enter **0.1** for **Value 1**, a dimension is displayed similar to that shown below.



- For the third format , if you enter **Value 1** as **0.15** and **Value 2** as **-0.05**, a dimension is displayed similar to that shown below.

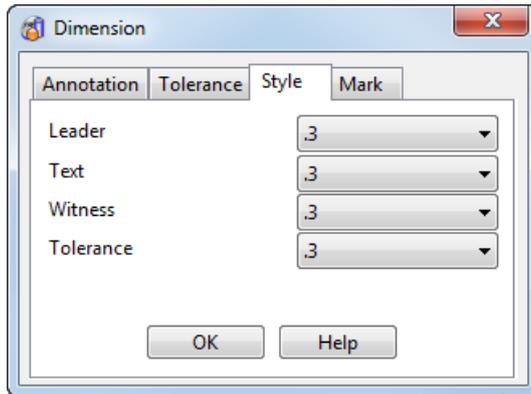


Clear tolerance — This button sets the contents of the **Value 1** and **Value 2** boxes to zero.

OK — Removes the dialog.

Style tab

This edits the style of the dimension.



Select the style of the following objects are given:

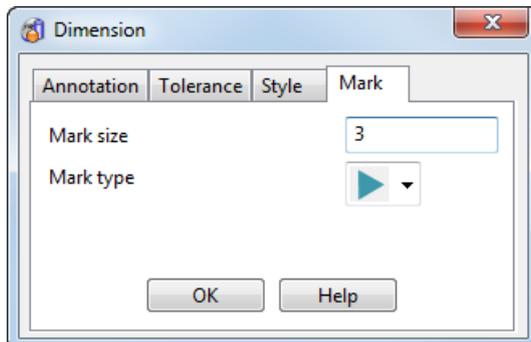
- **Leader**
- **Text**
- **Witness**
- **Tolerance**

To change the style for a particular object, select a new one from its selector.

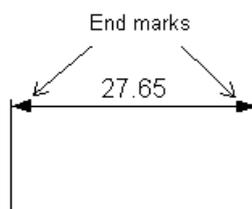
OK — Removes the dialog.

Mark tab

This edits the end marks on the leader of the dimension.



There are two end marks on the leader.



Mark size — Enter the size of the mark at the ends of the leader.

Mark type — Select the type of mark to use at the ends of the leader from the list.

OK — Removes the dialog.

Editing Dimension defaults using the Options dialog

The **Options** dialog is used to edit the default attributes of dimensions.

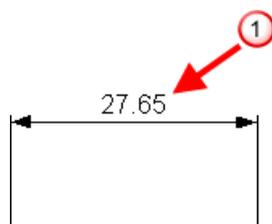
Unlike most defaults on the **Options** dialog, changes to dimension defaults can affect dimensions already created on the current model - if they were created with default ('standard') settings. This allows a drawing to be quickly changed from one set of defaults to another, as is required when changing from, for example, British (BSI) to American (ANSI) standards. If an attribute of a dimension has been edited using the **Dimension** toolbar, then any changes in the **Options** dialog to the default value of that attribute does not affect it.

To edit Dimension defaults:

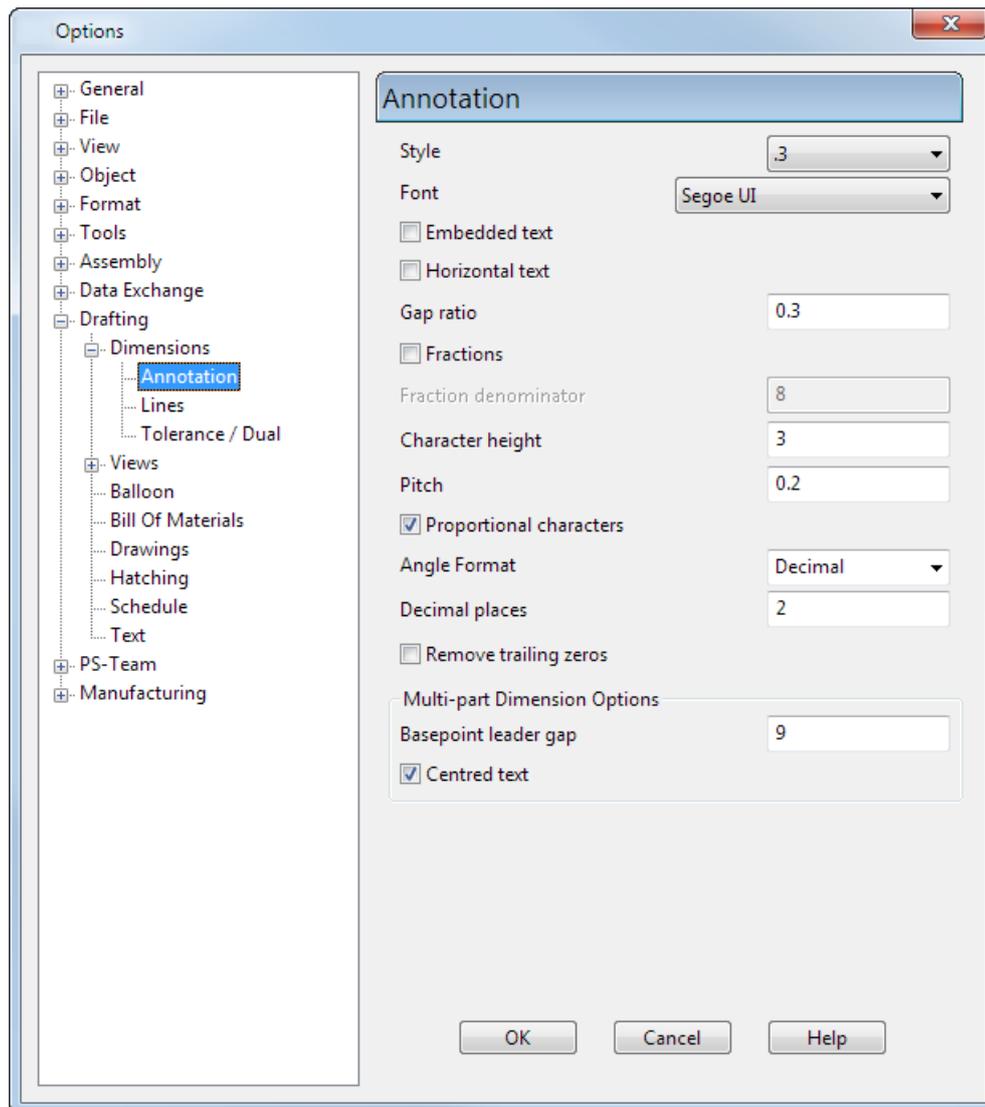
- 1 Select **Tools > Options** to display the **Options** dialog.
- 2 Select the **Drafting > Dimensions** options.
- 3 Select one of the following:
 - **Annotation** for the annotation of the dimension.
 - **Lines** for the leader and the witness lines of the dimension.
 - **Tolerance / Dual** for the tolerance of the dimension and displaying dual dimensions.

Annotation options

Annotation is the dimension text **1**.



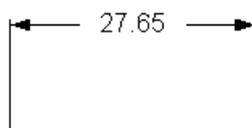
Use the dialog to change the annotation options.



Style — Select a line style from the list

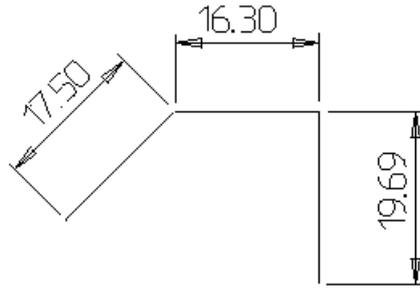
Font — Select the font to be used for the annotation.

Embedded text — Select this option to embed the text in the leader line.

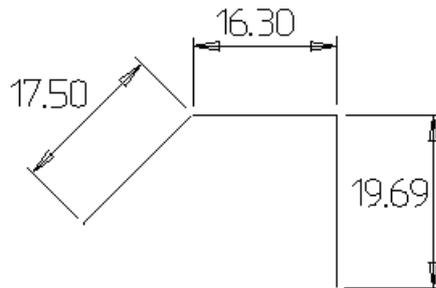


Horizontal text — Select this option to change all text so that it is positioned horizontally to the principal plane.

Suppose dimensions are angled as shown below.

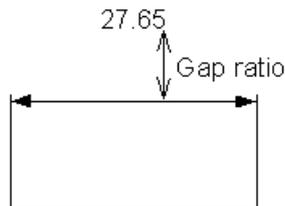


When text is changed to horizontal, the dimensions look like:

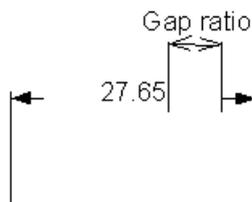


Gap ratio — Edits the gap between the leader and the text.

If **Horizontal text** is set, the gap ratio is as shown below.



If **Embedded text** is set, the gap ratio is:



Fractions — Converts the decimal part of the dimension into a fraction, by default to the closest eighth.

Fraction denominator — Sets the fraction denominator to a particular value. This is useful if you want to compare dimension values.

Character height — Edits the height of the text.

Pitch — This sets the amount of space between the characters.

Proportional characters — (*Duct font only*) When on, the space occupied by each character is not fixed.

Angle Format — Changes how an angular dimension is displayed. You can choose the **Angle Format** to display all angle dimensions as one of the following:

- Decimals
- Degrees
- Degrees - Minutes
- Degrees - Minutes - Seconds

Decimal places — Decimal places are the number of digits printed after the decimal point. By default, the numbers are printed to two decimal places.

Remove trailing zeros — If this option is selected, any trailing zeros, after the decimal point, are removed from the dimension annotation. The option is deselected by default.

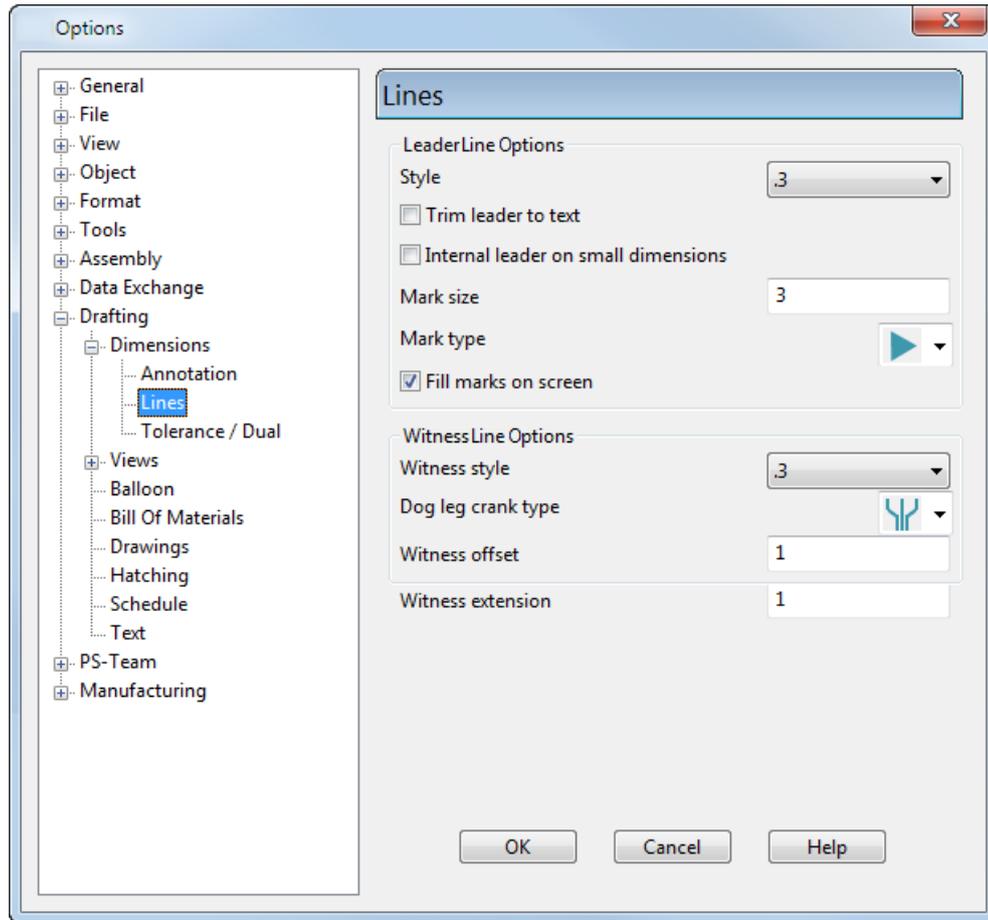
Multi-part Dimension Options — The following options are for base-point, chain and ordinate dimensions:

Basepoint leader gap — This is the distance between the leader and the text for base-point dimensions.

Centred text — When selected, the text is centred on the leader line. When deselected, you must enter a position to specify where the text lies on the leader.

Lines options

Use this dialog to change the lines options for dimensions.

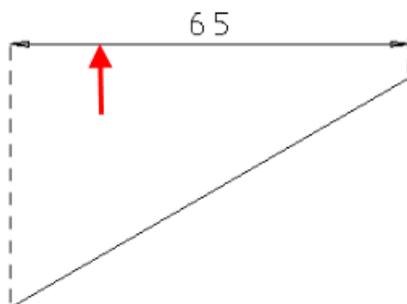


Leader Line Options

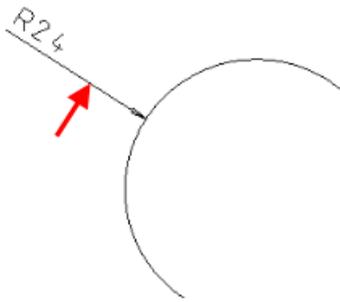
Leader lines are straight lines or arcs that represent the dimension values.

In these examples, the leader lines are indicated with a red arrow.

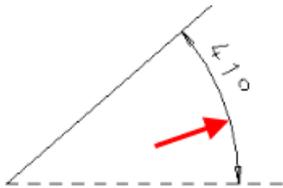
Linear leader line:



Radial leader line:



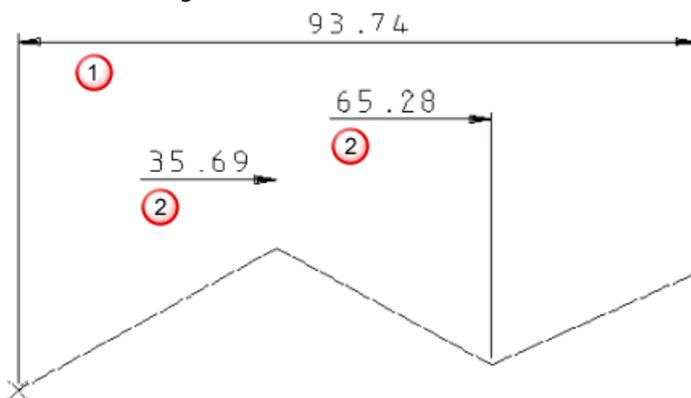
Angle leader line:



Style — Select the default style of the leader line from the list.

Trim leader to text — Select this option to trim the leader line to the position of the dimension annotation.

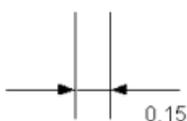
This can be used when the end of the leader is obvious, for example, in base-point and chain dimensions. Trimming leaders is possible only for linear dimensions.



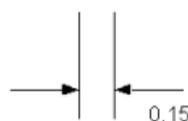
- 1 **Trim leader to text** deselected — The leader line is not trimmed.
- 2 **Trim leader to text** selected. The leader line is trimmed to the dimension annotation.

Internal leaders on small dimensions — When you have a dimension with leaders placed on either side of the dimension, as shown below, this option adds a line so that no gap exists between the arrows of the leader.

Internal leader on small dimensions selected



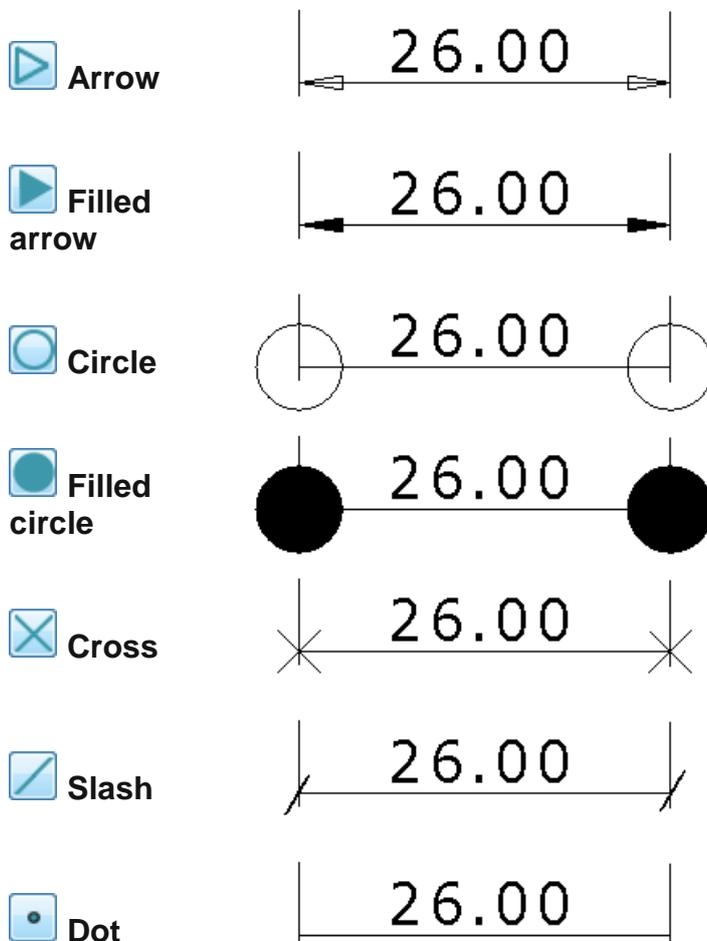
Internal leader on small dimensions deselected



Mark size — Enter the size of the mark at either end of the leader.

Mark type — Select the type of mark to display the end of the leader.

Select the mark type from:



Fill marks on screen — Select this option to shows filled dimension marks as filled on the screen.

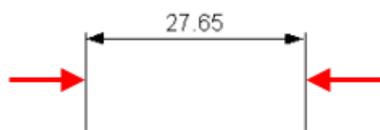


To improve graphics performance, deselect this option and show filled marks as unfilled.

Witness Line Options

Witness lines connect the leader to the object.

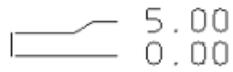
In this example, the witness lines are indicated with red arrows:



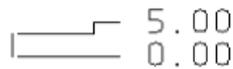
Witness style — Select the style of the witness line from the list.

Dog leg crank type — When creating ordinate dimensions, witness lines automatically use dog legs if the text does not fit. Select the type of crank on the dog leg from:

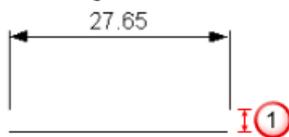
-  **Angled**



-  **Stepped**



Witness offset — Enter the offset distance of the witness line from the object.

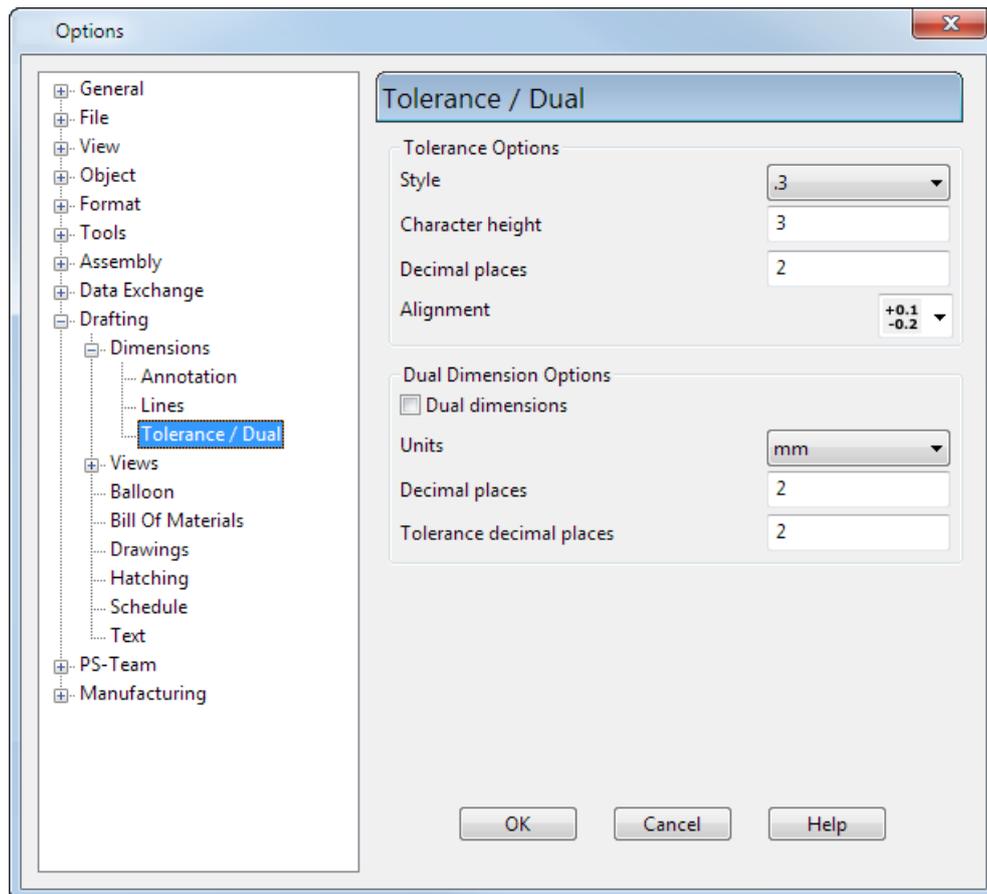


 **Witness offset** distance

Witness extension — Enter a value into the to set an extension distance in millimetres to all witness lines.

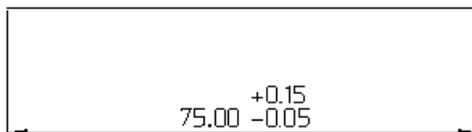
Tolerance/Dual options

Use this dialog to change the tolerance/dual options for dimensions.



Tolerance Options

Each dimension can have two tolerance values associated with it. The tolerance values show the range within which a dimension value can vary.



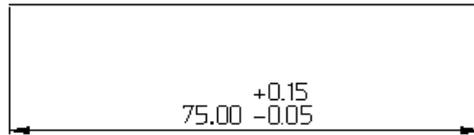
Style — Select the line style of the tolerance from the list.

Character height — Enter the height of tolerance values.

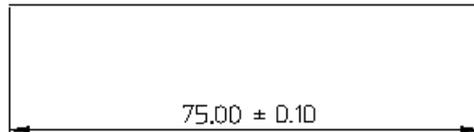
Decimal places — Enter the number of decimal places to display for tolerance values.

Alignment — Select the alignment style for tolerances from the list.

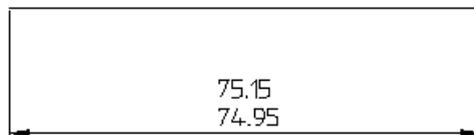
- +0.1
-0.2 **Alignment type 1** — For example, if the upper tolerance is 0.15 and the lower tolerance is -0.05, this is displayed as:



- ±0.1 **Alignment type 2** — For example, if the upper and lower tolerance are both 0.1, this is displayed as:

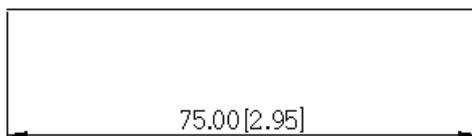


- 40.1
39.8 **Alignment type 3** — For example, if the dimension value is 75, the upper tolerance is 0.15 and the lower tolerance is -0.05, this is displayed as:



Dual Dimension Options

Dual dimensions are when the dimension values are displayed using a second set of units. The standard units are displayed without brackets and the dual dimension units in brackets.



Dual dimension — Select this option to display dual dimensions.

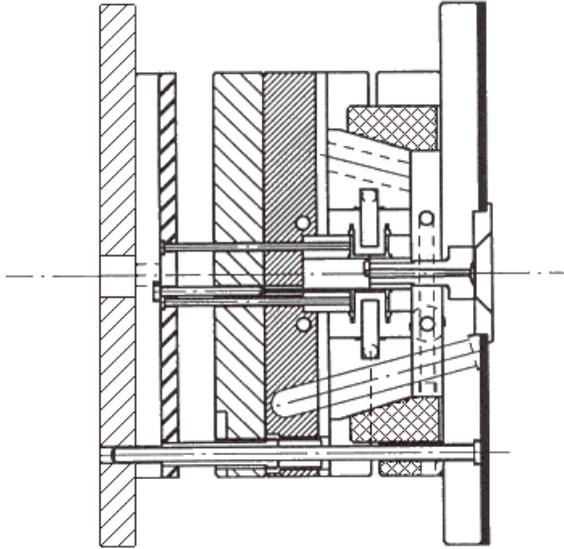
Units — Select the units for the dual dimensions.

Decimal places — Enter the number of decimal places for the dual dimensions.

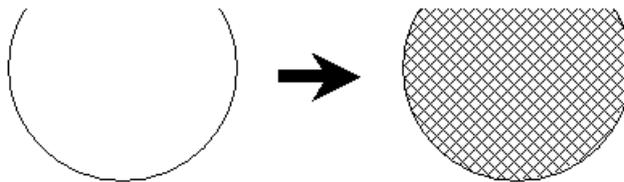
Tolerance decimal places — If tolerances are displayed, enter the number of decimal places for the tolerances of the dual dimensions.

Hatching

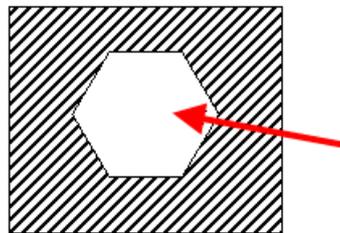
Use *hatching* to add a pattern to a selected region on a drawing. You can add different patterns to different regions.



You can hatch open regions. An imaginary line closes the ends of the boundary, for example:



You can create holes and islands within the hatch:



A drawing is usually hatched to:

- show cross-sections.
- clarify which parts of the drawing are made of the same material, or distinguish different types of materials.

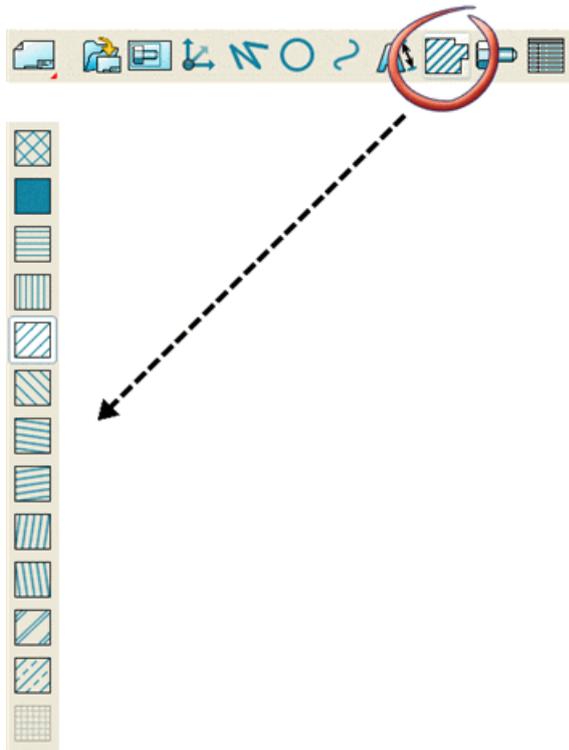
Creating hatching

- 1 Display the **Drafting** toolbar.

Click  (**Main** toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.



- 2 Click  (Drafting toolbar) to display the hatch pattern options.
- 3 Select a hatch pattern (hatch creation menu).



To use an existing user-defined hatch:

- a Select the **User Defined Hatch Pattern** button  to display the **User Defined Hatch Patterns** (see page 246) dialog.
 - b Select a pattern and click **OK**.
- 4 Select the hatch boundary (see page 244).
 - 5 Use the **Create Hatch** control panel to create the hatched region.



This control panel is similar to the one used for creating a composite curve by tracing, except the 'turn left at branch' is the default option on the **Create Hatch** control panel. This option can help you to quickly create a hatch boundary if you select the first object carefully.

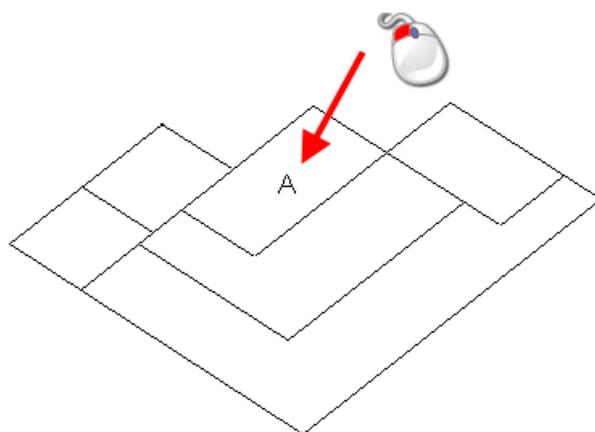


*If you select a composite curve, the **Create Hatch** control panel is not displayed and the hatch is automatically created.*

Selecting the hatch boundary

There are two ways to select the hatch boundary:

- **Selecting an area enclosed by wireframe.** Click in free space within an area enclosed by wireframe objects. Clicking as shown below will hatch region A

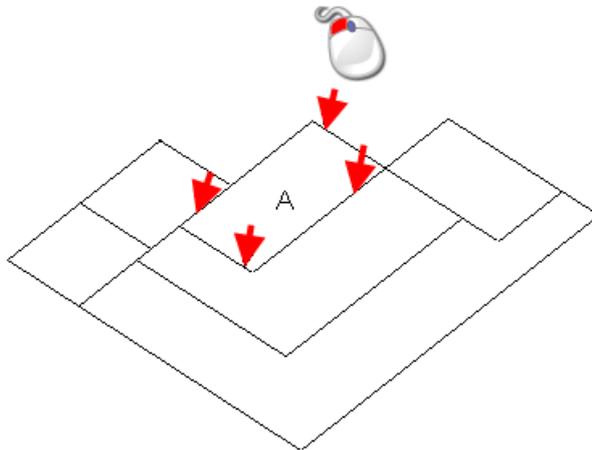


- **Using wireframe to define the boundary.** Click a wireframe object as the start object of the hatch boundary.

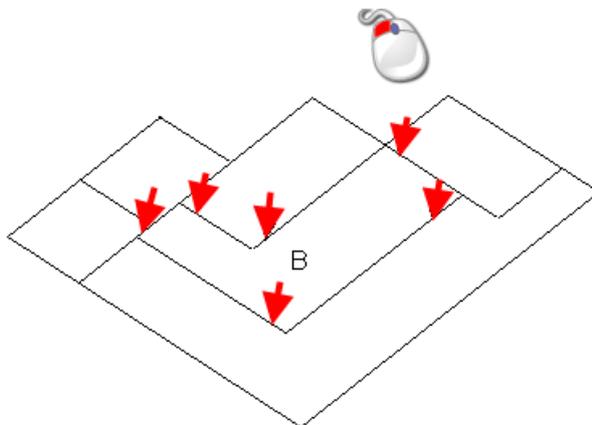
Once an object is selected, the **Create Hatch** control panel is displayed and a boundary is automatically selected.

Once you have decided where you want the hatch boundary to lie, select an object near to an intersection such that you want the boundary to turn left at that intersection.

To hatch region *A*, click the objects as shown below.



To hatch region *B*, click objects as shown below.



As you can see, in the turn left mode, one mouse click is enough to select the complete hatch boundary.

You can use the control panel to edit the hatch boundary. This control panel is used in exactly the same way as the Composite curve control panel.

When the required boundary is selected, click the **Save** button  on the control panel. The selected hatch pattern is added to the selected region.

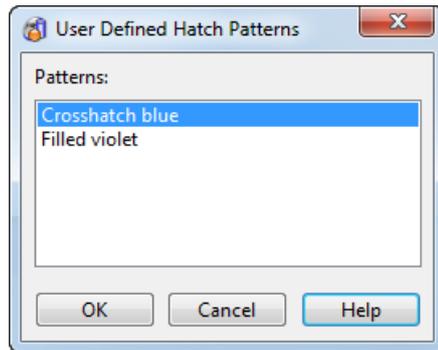
When you have created the hatch, you change the pattern by editing the hatch (see page 254).



*You can edit the default settings of each hatch pattern using the **Hatching** page (see page 249) of the **Drafting Options** dialog.*

User Defined Hatch Patterns dialog

Use this dialog to select a user defined hatch pattern.

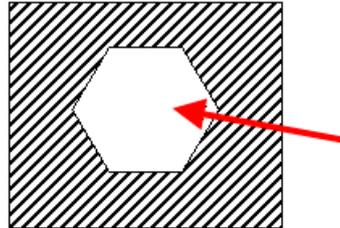


Patterns — Select the user-defined hatch pattern (see page 251) that you want to use.

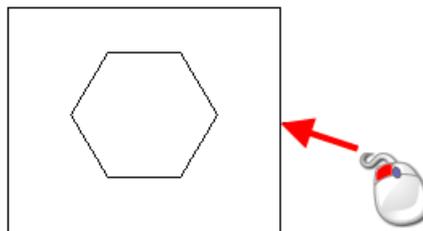
OK — Click this button to use the selected pattern and close the dialog.

Hatching an outer region

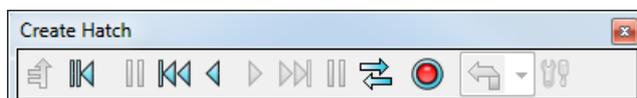
When an outer region contains smaller 'island' regions, you can hatch it without hatching the inner ones. Island regions are those which do not touch the outer one, as shown below.



- 1 Start **Hatching** and select the hatch pattern to be used. For further details, see Creating Hatching. (see page 243)
- 2 Select an object of the outer boundary.



Once an object is selected, the **Create Hatch** control panel is displayed.

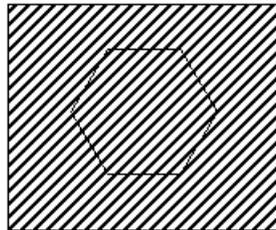


- 3 Use the control panel to select the outer hatch boundary.

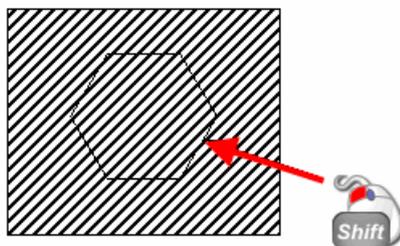
This control panel is used in exactly the same way as the **Composite curve** control panel

- 4 Once a boundary is selected, click the **Save** button  on the control panel.

The selected hatch pattern is added to the selected region.



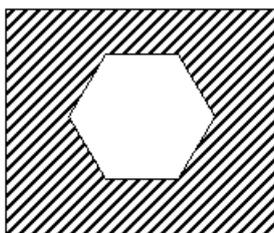
- 5 **Shift + Select** an object in the inner boundary to display the **Create Hatch** control panel. (**Shift + Select** is when you select an object whilst holding down the **Shift** key.)



- 6 Use the **Create Hatch** control panel to select the objects in the inner boundary.

- 7 Once you have selected the boundary, click the **Save** button  on the control panel.

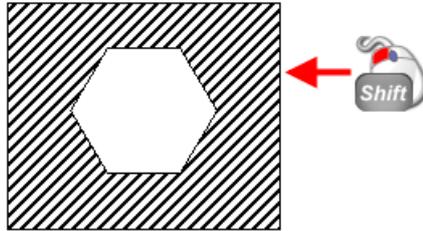
- 8 The hatch pattern of the outer region is removed from the inner the region.



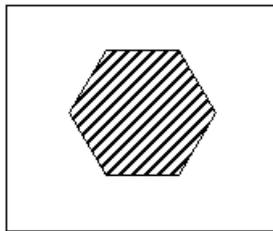
*If a composite curve is selected, the **Create Hatch** control panel is not displayed and the hatch is automatically created.*

Hatching an inner region

- 1 To toggle the hatching so that the inner area is hatched and the outer area is not hatched, **Shift + Select** the outer boundary. This displays the **Create Hatch** control panel.



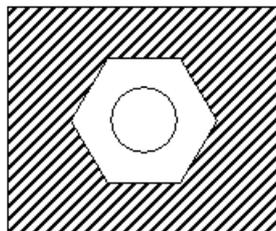
- 2 Click the **Save** button  on the control panel. The hatch pattern is toggled.



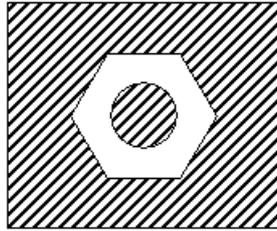
*If the outer boundary is a composite curve, the **Create Hatch** control panel is not displayed and the hatch is automatically toggled.*

Hatching an inner island

Suppose we had another island within the island as shown below.



You can hatch the island within the island to give the following.



- 1 Use **Shift + Select** to select one of the objects of innermost island. This displays the **Create Hatch** control panel.
- 2 Use the **Create Hatch** control panel to create the hatch boundary from the innermost island.
- 3 Click the **Save** button  on the control panel to create the hatch. Both areas are part of a single hatch object. If you edit one, the other is edited too.



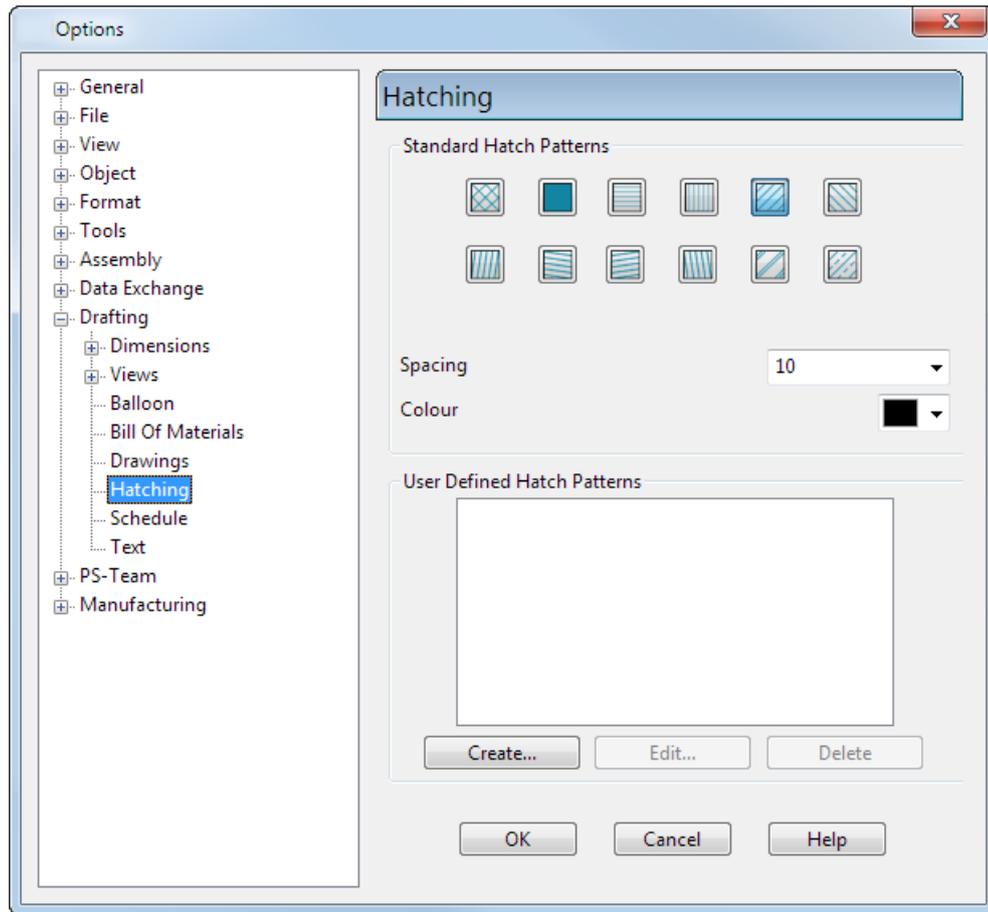
*If you select a composite curve, the **Create Hatch** control panel is not displayed and the hatch is automatically created.*

Changing the default hatching

To set the default hatching options:

- 1 Select **Tools > Options** to display the **Options** dialog.
- 2 Select the **Drafting > Hatching** option. This displays the **Hatching** page of the **Options** dialog.

Use the dialog to set the default hatch pattern.



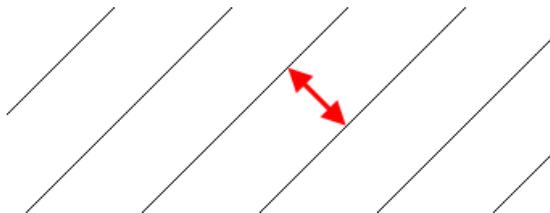
Standard Hatch Patterns

Select the default hatch pattern from the examples given. The selected pattern is highlighted in blue, for example:



For each hatch pattern, you can edit the **Spacing** and **Colour**. The spacing and colour of each hatch pattern is stored and used by the **Hatch** dialog.

Spacing — Enter the spacing between the lines in the selected hatch pattern.



Colour — Select a colour for the selected hatch pattern.

User Defined Hatch Patterns

If you do not want to use one of the **Standard Hatch Patterns**, select a *user-defined hatch pattern* (see page 251) from the list.

Create — Click this button to create a new *user-defined hatch pattern* (see page 251).

Edit — Click this button to edit the selected *user-defined hatch pattern* (see page 251).

Delete — Click this button to remove the selected pattern from the list.

OK — Click this button to save the options and close the dialog.

Creating/editing a user defined hatch pattern

You can create your hatch patterns by changing the definitions of the lines which make up the hatch.

To create/edit a user-defined hatch pattern:

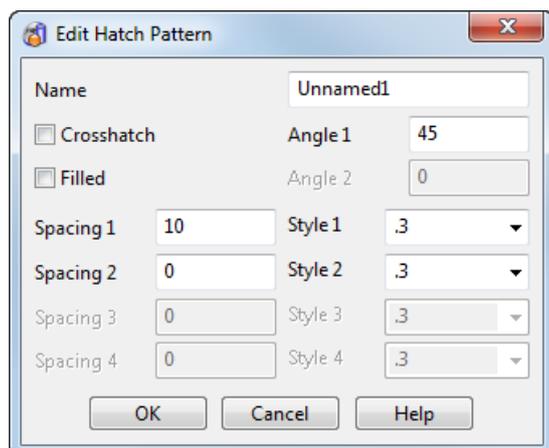
- 1 Select **Tools > Options** to display the **Options** dialog.
- 2 Select the **Drafting > Hatching** option. This displays the hatching page of the **Options** dialog.
 - **To create a new user-defined hatch pattern**, click the **Create** button to display the **Edit Hatch Pattern** dialog (see page 252).
 - **To edit an existing user-defined hatch pattern**, select a pattern from the list of **User Defined Hatch Patterns** and click **Edit** to display the **Edit Hatch Pattern** dialog (see page 252).
- 3 Use the dialog to create/edit a hatch pattern.



User defined hatch patterns are only stored in the current model. If you want to use the same hatch pattern in several models, we suggest you create the hatch in either the template model or a macro.

Edit Hatch Pattern dialog

Use this dialog to create or edit a hatch pattern.



Name — Enter the name of the hatch pattern.

Crosshatch — Select this option to create a cross hatch.

Filled — Select this option to create a filled hatch.

Angle — Enter the angle, measured from the X axis.

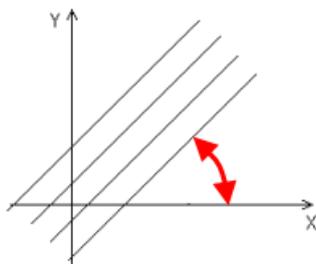
Angle 1 — Enter the angle of the hatch lines.

Angle 2 — Enter the angle of the second hatch lines.



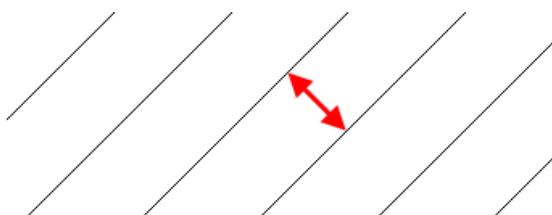
*You must select the **Crosshatch** option to access this option.*

Both **Angle 1** and **Angle 2** are measured from the X axis.

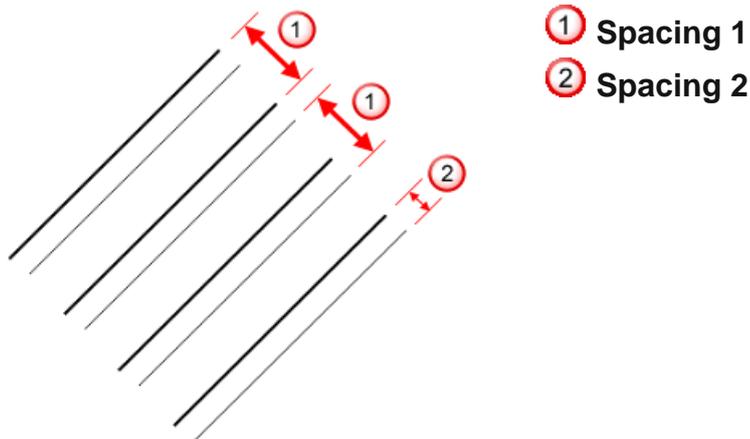


Spacing — Enter the distance between the hatch lines.

Spacing 1 — Enter the spacing between the lines in the selected hatch pattern.

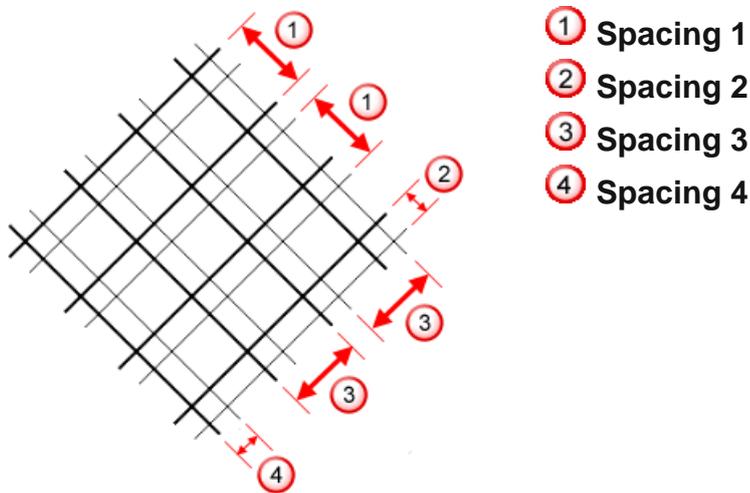


Spacing 2 — If you are using two line styles, **Spacing 1** defines the distance between lines of the same style. **Spacing 2** defines the offset of the line with **Style 1** from that of **Style 2**. This example uses two styles of line, a fine line and a thicker line:



If **Spacing 2** is 0, all lines are placed at equal distances from each other and use **Style 1**. The value of **Spacing 2** must be between 0 and the value of **Spacing 1**.

Spacing 3 and **Spacing 4** — **Spacing 3** defines the distance between lines of the same style at **Angle 2**. **Spacing 4** defines the offset of the line with **Style 3** from that of **Style 4**.



You must select the **Crosshatch** option to access the **Spacing 3** and **Spacing 4** options.

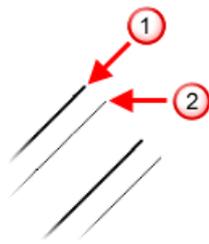


If **Spacing 2** is 0, **Style 1** is used for all **Angle 1** lines. If **Spacing 4** is 0, **Style 3** is used for all **Angle 2** lines.

Style — Select the style for each hatch line.

Style 1 — Select the style of the first line at **Angle 1**.

Style 2 — Select the style of the second line at **Angle 1**.



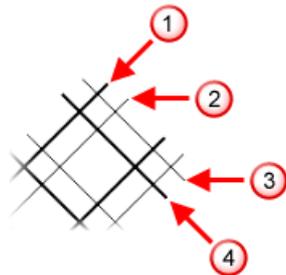
- ① Style 1
- ② Style 2



If **Spacing 2** is 0, all lines are placed at equal distances from each other and use **Style 1**.

Style 3 — Select the style of the first line at **Angle 2**.

Style 4 — Select the style of the second line at **Angle 2**.



- ① Style 1
- ② Style 2
- ③ Style 3
- ④ Style 4



If **Spacing 4** is 0, **Style 3** is used for all **Angle 2** lines.

OK — Click this button to create or edit the hatch and close the dialog.

See Creating hatching (see page 243) for details on how to add the hatch to the model.

If you have created a hatch pattern, it is available in the **User Defined Hatch Patterns** dialog (see page 246).

Editing hatching

Ensure you are in selection mode, by clicking **Select** . You can edit single and multiple hatches in the following ways:

- Select the hatch.
- Use the **Edit** menu:
 - Edit > Modify** displays the **Hatch** dialog (see page 255).
 - Edit > Convert To wireframe** — Converts the hatch lines into wireframe.
- You can edit the style of the pattern lines of the hatch using the **Style** toolbar.
- Double-click the hatch to display the **Hatch** dialog (see page 255).

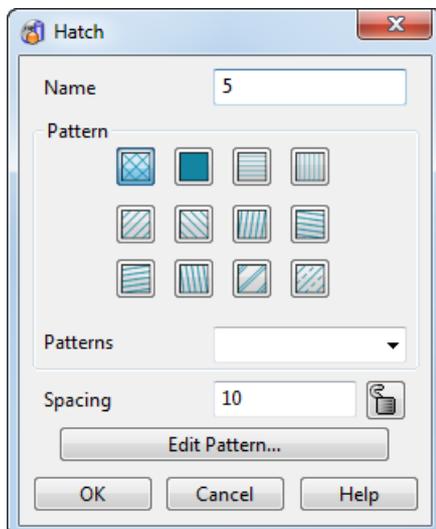
- Right-click on the hatch to display the context menu options available for hatching. The name of the hatch (for example **Hatch 1**) and its level (for example **General**) is displayed at the top of the context menu.



The five options, at the bottom of the context menu, change the pattern of the selected hatch.

Hatch dialog

Use this dialog to edit the pattern of the selected hatch and to add extra hatch boundaries when a single hatch is selected.



Name — This name of the selected hatch is displayed. You can edit the name.

Pattern

Select one of the standard hatch patterns to change the selected hatching. The selected pattern is highlighted in blue, for example:



For each of the standard hatch patterns, the colour and spacing (if not locked) match those that are set in the **Options** (see page 249) dialog.

Patterns — If you do not want to use one of the standard hatch patterns, select a *user-defined hatch pattern* (see page 251) from the list.

Spacing — This displays the default spacing (see page 249) between the lines of the selected hatch pattern.

Change this value and move the mouse outside the dialog. The hatch pattern updates. On the dialog, the button of the selected pattern is updated to show that the spacing for that pattern no longer matches the default set on the Options dialog.

If you press the button of the selected pattern again, the default spacing is applied to the hatch.

 **Lock** button — If you click this button to lock, the spacing value is fixed and all the hatch patterns use that value. If you click the **Lock** button again and select a hatch pattern, the spacing value changes to its default.

Edit Pattern — Click this button to open the **Edit Hatch Pattern** dialog (see page 252) to edit the selected hatch pattern.

OK — Saves the edits made to hatch and closes the dialog.

Cancel — Discards any edits made to hatch whilst the dialog is displayed and closes the dialog.

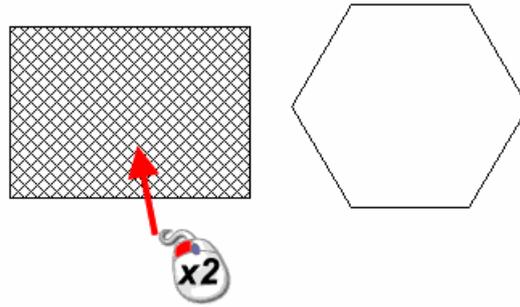
Adding extra hatch boundaries

You can edit a hatch pattern to add extra hatch boundaries to islands or completely separate areas. When this is done, all these hatch areas remain part of a single hatch object. Any edit performed on one area affects all the areas that are part of that hatching.

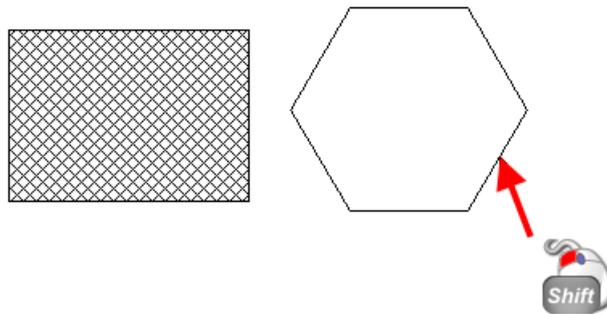
Adding separate areas

- 1 Double-click the hatch of the pattern you wish to copy. This displays the **Hatch** dialog.

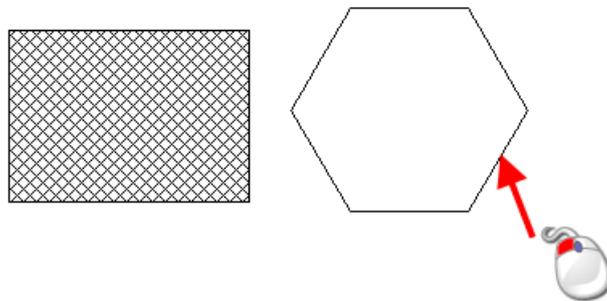
Suppose we have the following.



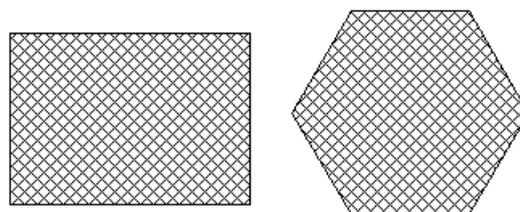
- 2 Change the pattern of the hatch, if you wish, using the **Hatch** dialog (see page 255). Do not click **OK** on the dialog.
- 3 **Shift + select** a wireframe object as the start of the new boundary. The **Create Hatch** control panel is displayed.



- 4 Use the **Create Hatch** control panel to select the boundary. See *Creating hatching* (see page 243) for details on how to use the control panel.



The new composite curve is added as an additional boundary of the original hatch and has the same pattern and spacing.



- 5 Click **OK** to save the changes.

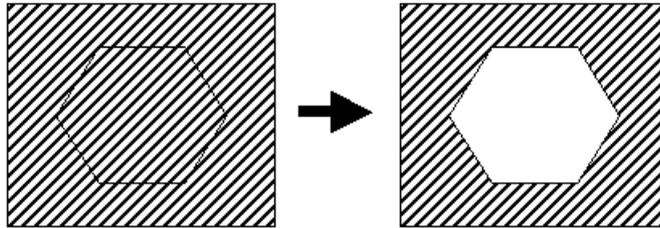


If you select a composite curve, the **Create Hatch** control panel is not displayed and the composite curve is selected as the boundary.

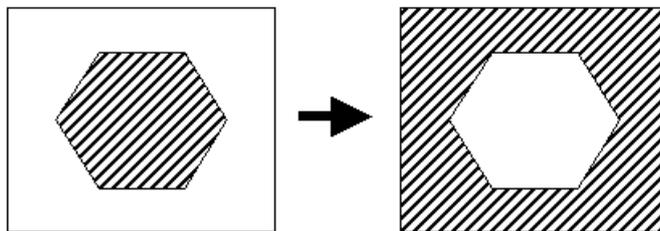
Islands

You can create islands from an existing hatch in two ways:

- Within an existing hatch



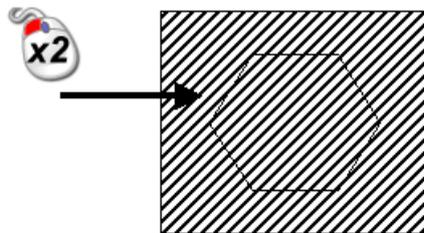
- Extend the hatch to an outer boundary and remove the existing hatch



The boundary within or enclosing the existing hatch must be a composite curve.

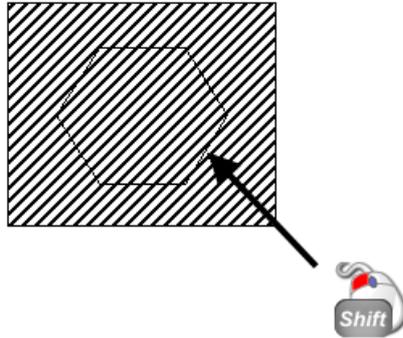
Creating an island

- 1 Double-click the hatch of the pattern as shown in the image below to display the Hatch dialog (see page 255).



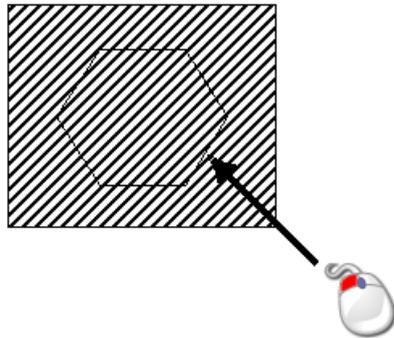
- 2 Change the pattern of the hatch, if you wish, using the **Hatch** dialog. Do not click **dialog**.

- 3 Shift + Select** a wireframe object as the start of the new boundary. (**Shift + Select** is when you select an object whilst pressing the Shift key.) This displays the **Create Hatch** control panel.

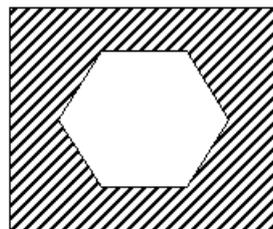


*If a composite curve is selected, then the **Create Hatch** control panel is not displayed and the composite curve is selected as the boundary.*

- 1** Use the **Create Hatch** control panel to select the boundary.
See *Creating hatching* (see page 243) for details on how to use the control panel.



In our example, the hatch is removed from the area enclosed by the composite curve.

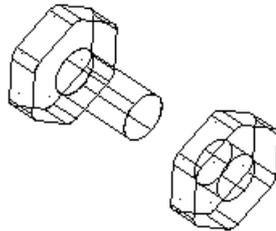


- 2** Click **OK** to save the changes.

Symbols

What is a symbol?

A symbol is a user-defined object constructed from a collection of objects. Objects used often can be stored as symbols, for example, nuts and bolts.



A symbol is created in two stages:

- 1 a symbol definition is created
- 2 a symbol is created from the symbol definition

A symbol definition is created by selecting a group of objects (for example, objects drawn to represent a bolt) and converting them into a new single object (for example, a symbol definition called 'bolt'). All the key points of the original objects are lost when these objects convert into the new object, but new key points (known as pins) can be defined to position the new object.

A symbol is created by selecting a symbol definition and positioning it on the drawing.

A single symbol definition is shared between many symbols (saving space in the model). For example, if several 'bolts' symbols are placed in a model, each symbol accesses its object data from the 'bolt' symbol definition. Each symbol only stores data about its position and orientation.

You can store symbol definitions in a library model so that you can access them from any model.

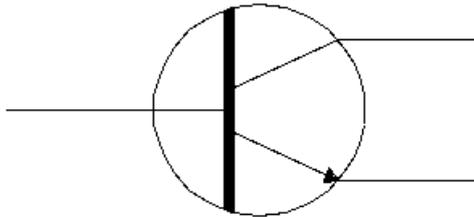
Creating a symbol definition

- 1 Make sure you are in drafting mode.



- 2 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 3 Create the objects to represent your symbol definition.

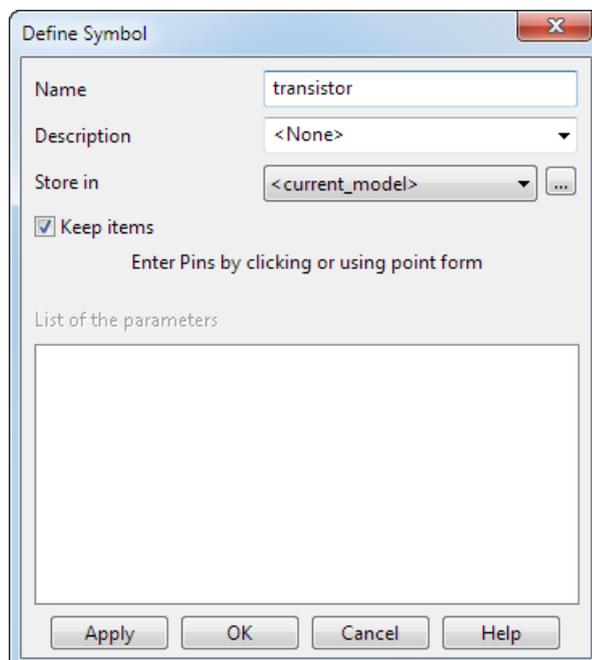
Suppose we want to create a symbol definition called *transistor* of the following objects.



- 4 Select the objects.
- 5 From the **Object** menu, select **Symbol Definition** followed by **Create** to display the **Define symbol** dialog.
- 6 Use the Define Symbol dialog (see page 261) to create the symbol definition.
- 7 Click **OK**.

Define Symbol dialog

Use this to create a symbol definition from the selected objects.



Name — The symbol definition is given a default name. Change it to **transistor**.

Description — Enter or select the description for the symbol definition. This description will be displayed in the Create symbol dialog (see page 265) to help identify the symbol definition required.

Store in — You can store the symbol definition in either the current model or a library model. For further details, see Creating a library of symbol definitions. (see page 263)

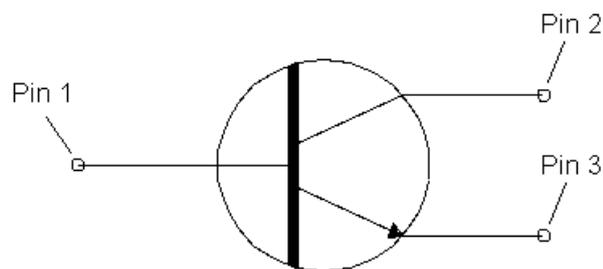
 — Click to browse for symbol libraries outside the model database. The **Open Library** dialog is displayed where you select a symbol library.

Keep items — If ON, objects representing the symbol definition are stored with it. In this case, if a symbol (created from the symbol definition) is converted (using **Edit > Convert > Symbol**), it converts into these stored individual objects. If OFF, the objects are not stored with the symbol definition. In this case, if the symbol is converted, its geometry converts into a series of lines.

Enter Pins by clicking or using point form — Pins are positions on the symbol definition. They are used to position the symbol and as key points on the symbol. You cannot accurately snap onto a symbol where no pin exists, therefore we recommend you add extra pins where other objects are most likely to connect with the symbol.

A symbol definition must have at least one pin. Enter a position to create a pin. You can either click a position on the screen or click the Position button  to display the Position dialog and use this dialog to enter a position. When you enter pins, they are marked by a circle. If you input the wrong position for a pin, simply click the pin mark to remove it.

In our example, we enter pins as shown below.



The first pin is the default pin selected when creating symbols, therefore enter the first pin as the most likely point to position the symbol.

List of the parameters — This functionality is for future development.

Apply — Creates the symbol definition and allows you to create another symbol definition. Box select the group of objects for the next symbol definition. Then fill in the dialog and enter pins.

OK — Creates the symbol definition and closes the dialog. All the selected objects are stored in the definition, undrawn and deleted from your model.

Cancel — Closes the dialog without creating a symbol definition.

Creating a library of symbol definitions

You can store symbol definitions in a library model so that you can access them from any model.

- 1 Create a model and save it with the word *library* in its name.
- 2 Use the steps in Creating a symbol definition (see page 260) to create symbol definitions in your library model.
- 3 Save the model.

When you create symbols in other models, you can now use the symbol definitions from this library model by selecting its model name from the **Model** selector in the **Create symbol** dialog. For further details, see Creating a symbol (see page 264).

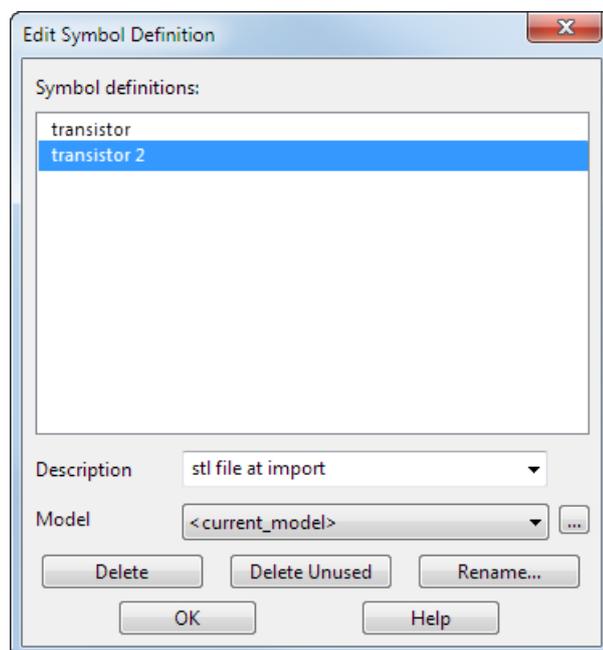
Editing a symbol definition

You can edit the name of a symbol definition and delete the symbol definition if it is not used in the selected model.

- 1 From the **Object** menu, select **Symbol definition** followed by **Edit** to display the **Edit Symbol Definition** dialog.
- 2 Use the Edit Symbol Definition dialog (see page 263) to edit symbol definition.

Edit Symbol Definition dialog

Use this to rename and delete symbol definitions.



Symbol definitions — This lists the symbol definitions in the selected model. Use this list to select symbol definitions to edit. You can select more than one symbol by clicking the names. To deselect a symbol definition, you simply click the name again.

Description — Enter or select the description for the selected definition(s).

Model — This selector contains the library models. Use it to select the model from which you want to edit a symbol definition.

To browse for symbol libraries from outside the model database, click  to display the **Open Library** dialog. Use this to select a symbol library.

Delete — This deletes the selected symbol definitions, providing they are not used in the selected model.

Delete Unused — This deletes unused symbol definitions.



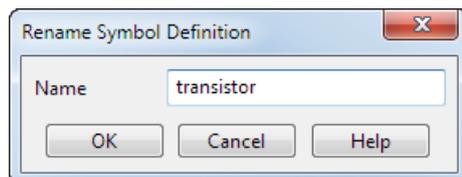
Component definitions are not affected.

Rename — This is used to rename the selected symbol definition and displays the Rename Symbol Definition dialog (see page 264).

OK — Accepts any changes and closes the dialog.

Rename Symbol Definition dialog

Use this to rename the selected symbol definition.



Name — This displays the current name of the selected symbol definition. You can change it.

OK — Accepts the new name of the symbol definition and closes the dialog.

Cancel — Closes the dialog without accepting any changes.

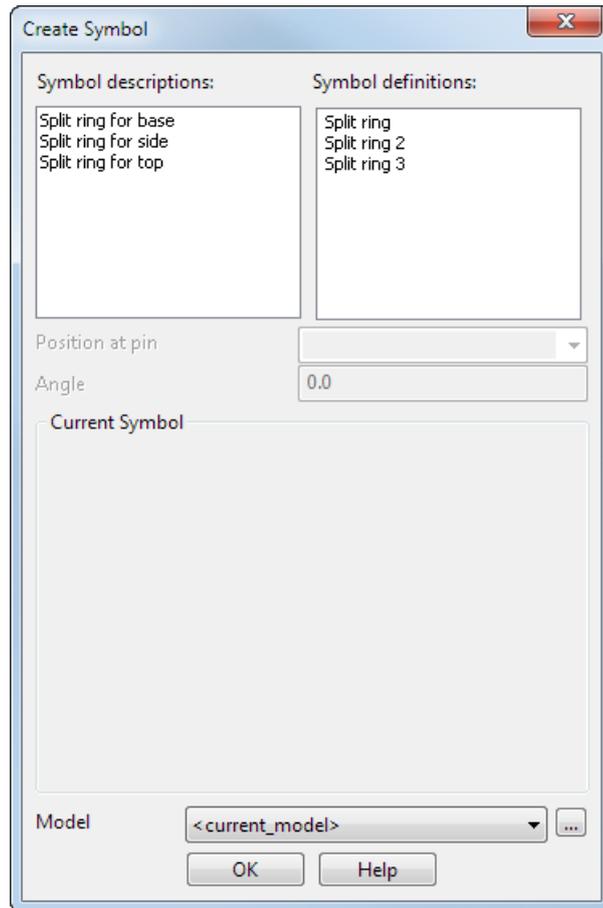
Creating a symbol

Symbols are instances of a symbol definition. Therefore, you can only create a symbol from a symbol definition which exists in either the current model or a library model.

- 1 Click  (main toolbar).
- 2 Use the Create Symbol dialog (see page 265) to create the symbol.

Create Symbol dialog

Use this to create a symbol from one of the symbol definitions.

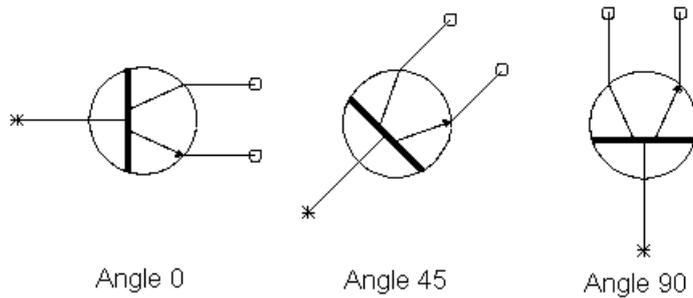


Symbol descriptions: — This displays the description entered in the **Description** text box on the **Define symbol** dialog when creating new symbol definitions.

Symbol definitions: — Use the list to select the symbol definition you want. When you select a symbol definition, it is displayed in the **Current Symbol** graphics window.

Position at pin — This contains a list of pins on the selected symbol definition. Select one to position the symbol. By default, pin 1 is selected.

Angle — This angle rotates the symbol definition to create the symbol. The rotation is about the axis normal to the principal plane. When you change the angle, the symbol definition rotates in the **Current Symbol** graphics window.



Current Symbol — This is a graphics window which displays the selected symbol definition and shows where its pins lie. The symbol definition is displayed looking down on the current principal plane.

Model — Select a library model from which to choose the symbol definition.

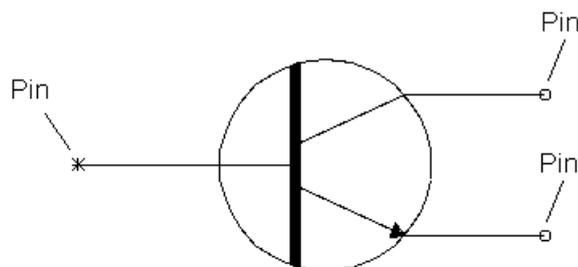
To browse for symbol libraries from outside the model database, click  to display the **Open Library** dialog. Use this to select a symbol library.

If you select a symbol definition from a library model, then its symbol definition is imported into the current model.

Once you have selected a symbol definition and its pin, enter a position to place the symbol in your model. For further details see Entering positions in Modelling Concepts..

As you move the mouse cursor in the graphics window, the symbol is attached to the cursor at the selected pin.

When you enter a position, the symbol is added to your model. On your symbol, marks are drawn at the positions of the pins.



You can continue creating more symbols whilst the dialog is displayed.

OK — Accepts changes and closes the dialog.

Editing a symbol

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

- Select the symbol.
- **Edit > Convert > Symbol** converts the selected symbol into its individual objects.
- **Edit > Modify** displays the **Symbol Editor** dialog.
- Double-click the symbol to display the Symbol Editor dialog. (see page 268)
- Right-click the symbol to display the edit options available for symbols. The name of the symbol and the level on which it lies is at the top of the context menu.



Convert - converts the selected symbol into its individual objects.

Modify - displays the **Symbol Editor** dialog.

Reverse - Symbols can be turned inside out (reversed) when their symbol definitions contain triangles. For further details see Reversing triangle symbols

Smooth - You can smooth triangle symbols in PowerSHAPE to improve the appearance of the faceted triangle mesh. You can use this option on imported files that contain triangles but do not contain normals, such as .STL, .DMT or .OBJ files.

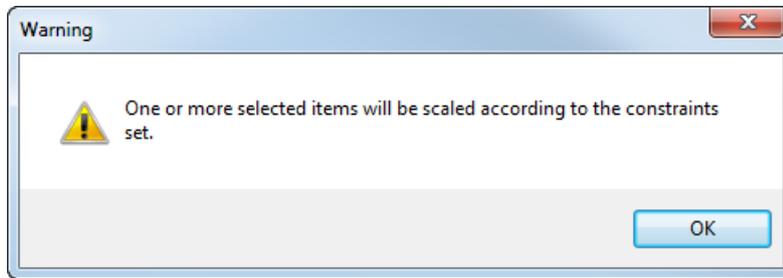
For further details see Smoothing triangle symbols

Unite - Join selected symbols to create a single symbol.

Scaling Constraints - This option displays the **Set Constraints for symbol** dialog to allow constraints to be set for the selected symbol.



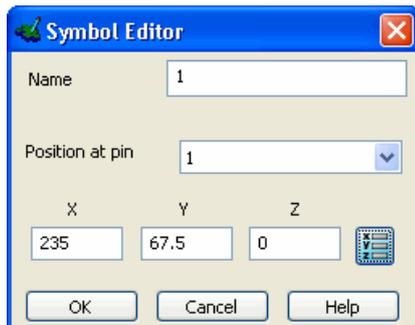
The warning message shown below is displayed when you use the **Scale** option (General edits toolbar) to scale multiple objects if any of the selected items has scaling constraints already set.



If **Keep items** is selected when its symbol definition was created, links from the object definitions to the symbol are lost. For further details, see *Creating a symbol definition* (see page 260).

Symbol Editor dialog

Use this to edit the selected symbol.



Name - Displays the name of the selected symbol. You may change the name if you wish.

Position at pin - The name of the selected pin is displayed.

X, Y and Z - These give the coordinates of the selected pin. You can change the coordinates of the selected pin to move the symbol.

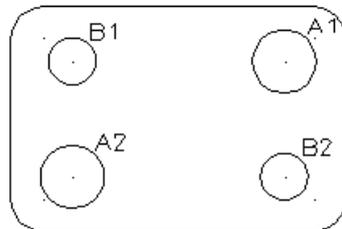
OK - Applies the changes entered in the dialog to the symbol and the dialog is removed.

Cancel - Removes the dialog without altering the symbol.

Schedules

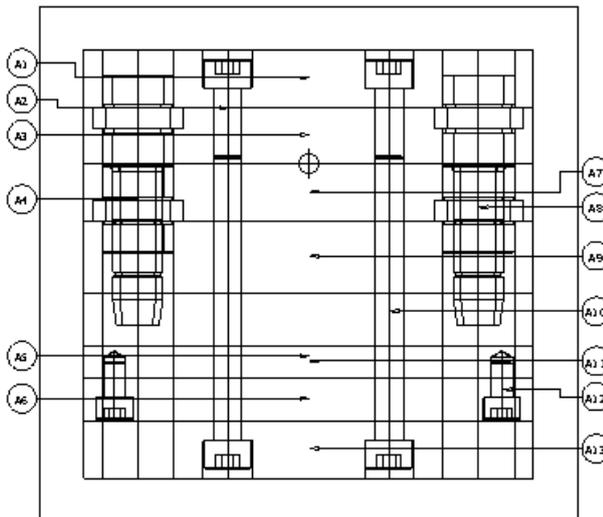
What is a schedule?

A schedule is either a table containing the coordinates from specific objects or a bill of material.



Label	X	Y	Ø	Comments
A1	144	156	20	-
A2	77	120	20	-
B1	77	156	15	-
B2	144	120	15	-

The Bill of Material (BOM) is a table of components/sub-assemblies used by an assembly.



Label	Part	Catalogue	Material	Quantity
A1	FP1212H	DMS	MS	1
A2	B20CS	DMS	MS	4
A3	B121216MS	DMS	MS	1
A4	EGP14164S	DMS	MS	4
A5	R121236	DMS	MS	1
A6	ES1212(base)	DMS	MS	1
A7	B121216MS CorePlate	DMS	MS	1
A8	EGB1416	DMS	MS	4
A9	BP1212	DMS	MS	1
A10	B80CS	DMS	MS	4
A11	ES1212(cover)	DMS	MS	1
A12	B10CS	DMS	MS	4
A13	FP1212H BottomClampingPlate	DMS	MS	1

You can create schedules from:

- assemblies
- assembly holes

- balloons
- circles
- workplanes
- symbols/components
- points
- solid holes
- cylinders

You can also append schedules to add further entries.

If **Microsoft Excel** is installed, you can create embedded schedules. (see page 270)

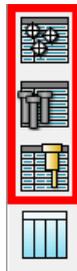
Creating an embedded schedule

- 1 Ensure that MS-Excel is installed on your computer and that **Use spreadsheet to create schedule** is selected (**Schedules** page of the **Drafting** options).
- 2 Make a drawing view active by selecting **Active** from the **Drawing View** popup window.

- 3 Click **Schedule**  (*drafting toolbar*)



- 4 Click one of the schedule options on the Schedule and BOM toolbar

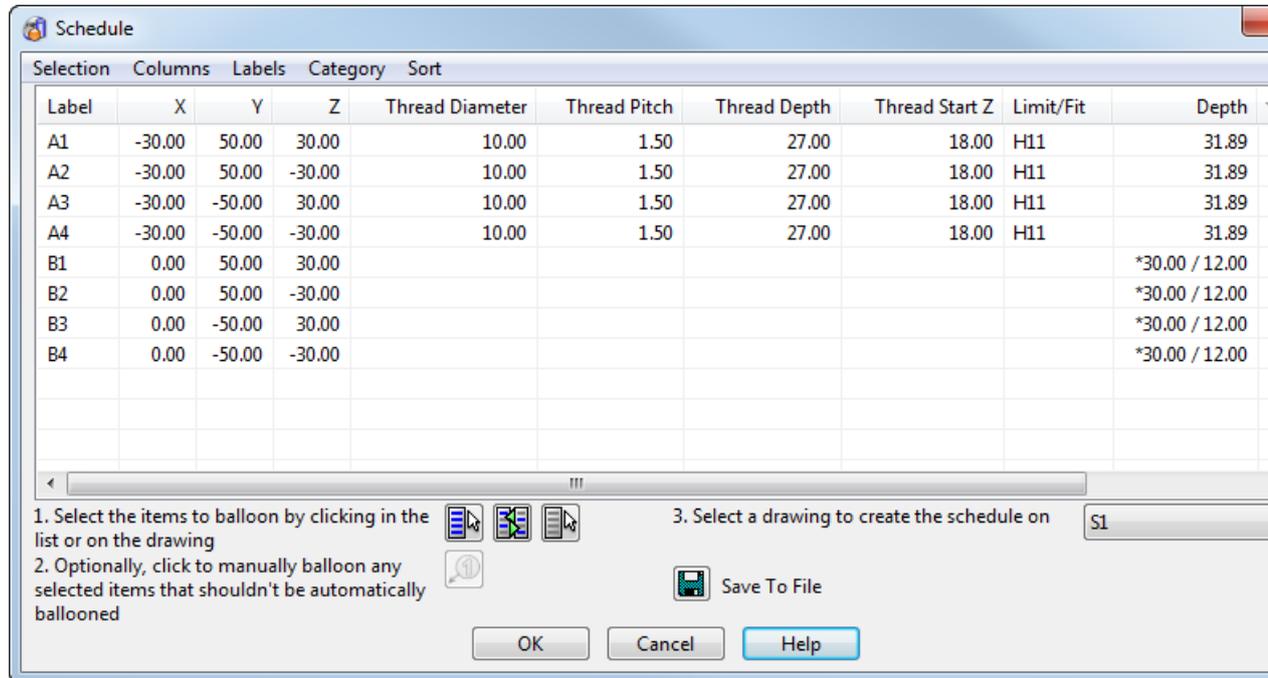


- Click  to produce a schedule for holes (see page 271).
- Click  to produce a schedule for ejector pins, core pins and return pins.
- Click  to produce an electrode summary sheet (see page 276), that includes information about the electrode.

- The **Schedule** dialog (see page 271) is displayed, containing items from the drawing view as a schedule. Manipulate the data in the **Schedule** dialog in the same way you manipulate data in the Bill of Materials dialog (see page 306).

Schedule dialog (Holes)

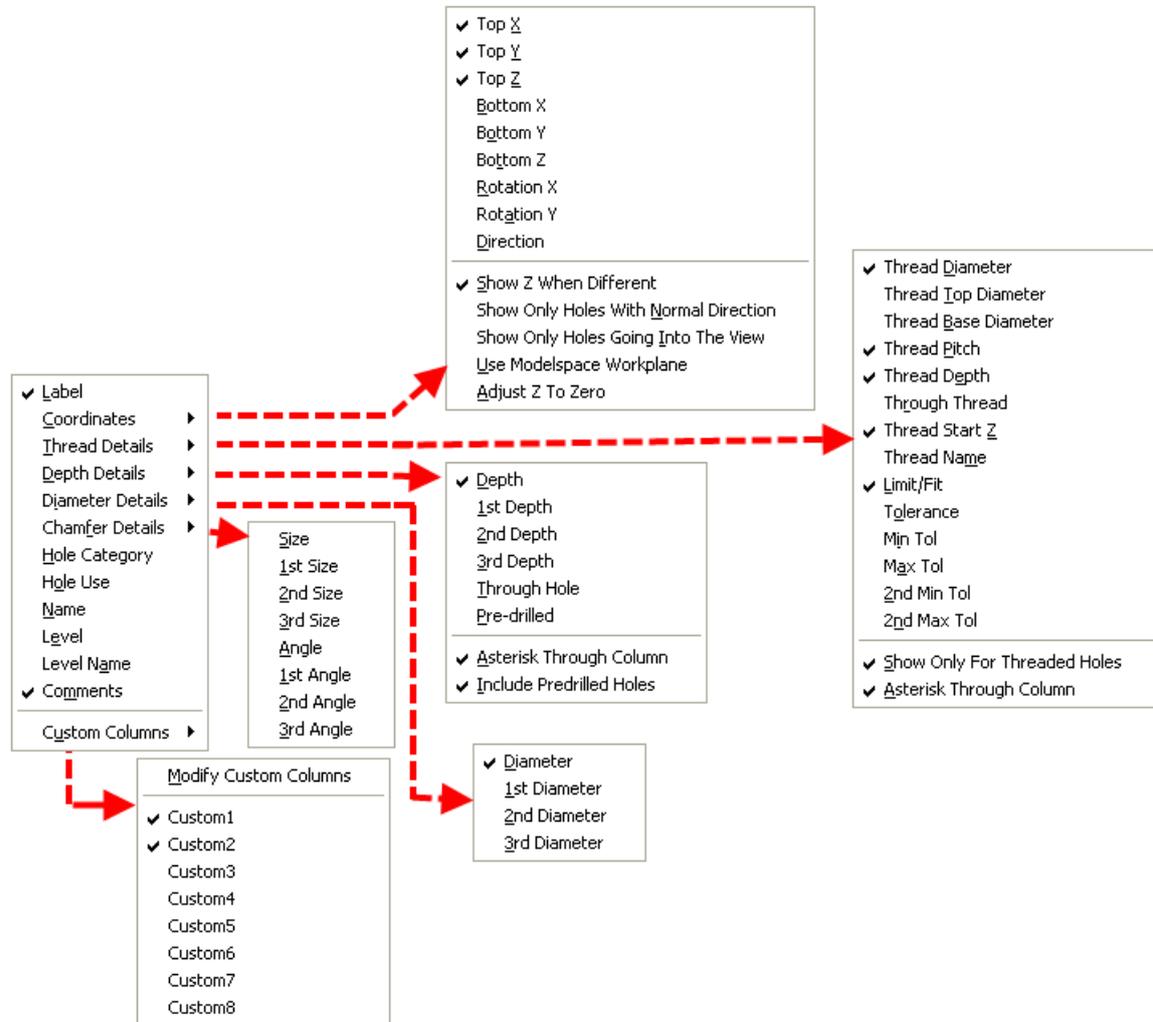
Use this dialog to specify the data that you want to be displayed on the schedule for holes.



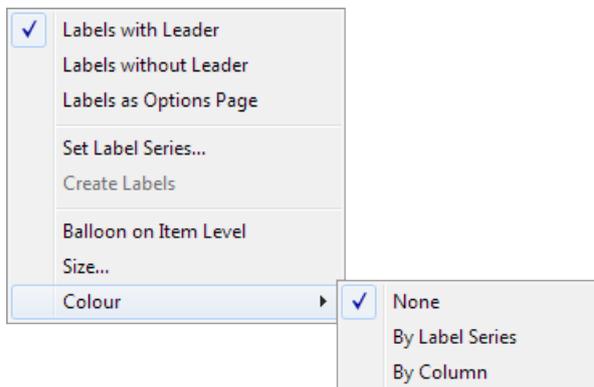
- Using the menu options on the dialog, you can select and deselect the data that is displayed. The indicates that the option is selected and the data will be displayed in the schedule.
 - Selection** — Controls the display of the data in the table.



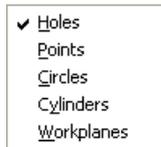
- **Columns** — Enables you define the columns of data that are displayed. Click an option to select *ON* (display) and deselect (do not display)



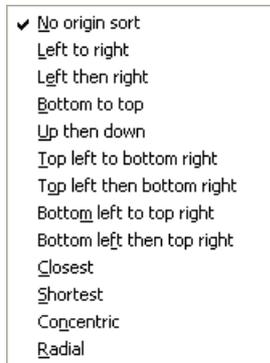
- **Labels** — Displays all the options for controlling labels and balloons. They work in the same way as labelling in Bill of Materials.



- **Category** — Select the category of item you want to include in your schedule from the following options:



- **Sort** — Use to select a sorting method from the following options:



The default sort option is **No origin sort**. Select a sorting method from the options.

You can also sort the schedule using the **Diameter** column. Click on the arrow alongside **Diameter** to reverse the sorting order.

- 2 Modify the layout of the columns as required:
 - a Click in the column header using the right mouse button.
 - b Select the option you require from the column header context menu.



Move left — Move the selected column to the left.

Move right — Move the selected column to the right.

Increase decimal places — Increase the decimal places in the column data. This option is active only if the column contains numerical data.

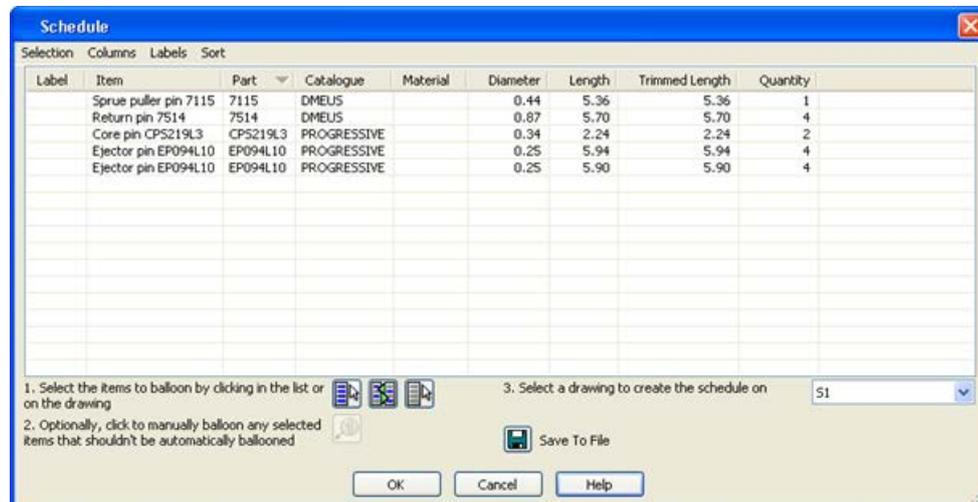
Decrease decimal places — Decrease the decimal places in the column data. This option is active only if the column contains numerical data.

- 3 If required, apply balloons and labels (see page 279) to components in drawings that are selected for the schedule.

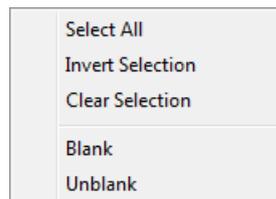
- 4 Create the schedule on the drawing, or save it to a file. Use  to toggle between displaying the data as an embedded Microsoft Excel spreadsheet or saving the data in .xlsx or .csv format.

Schedule dialog (Ejector)

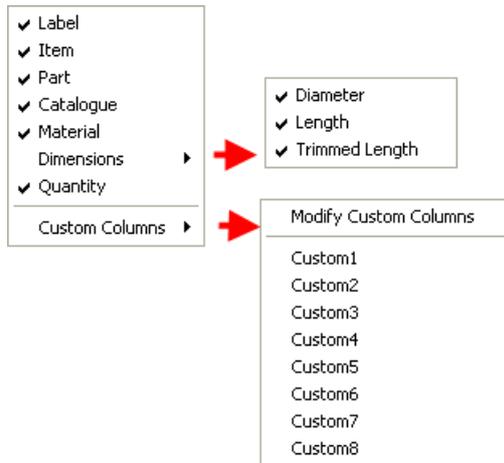
Use this dialog to specify the data that you want to be displayed on the Schedule for ejector pins, core pins and return pins.



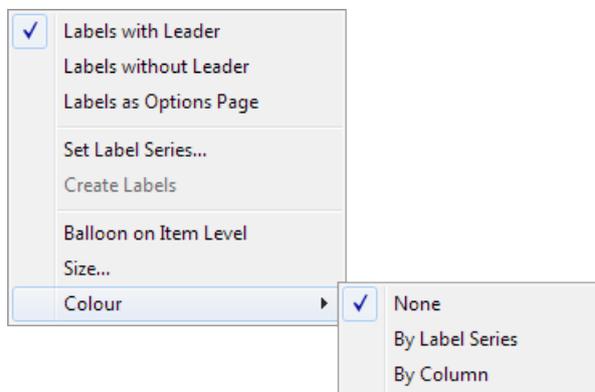
- 1 Using the menu options on the dialog, select and deselect the data that is displayed. The indicates that the option is selected and the data will be displayed in the schedule.
 - **Selection** — Controls the display of the data in the table.



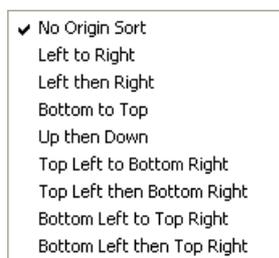
- **Columns** — Lets you define the columns of data that are displayed. Click an option to select (display) and deselect (do not display).



- **Labels** — Displays all the options for controlling labels and balloons. They work in the same way as labelling in Bill of Materials.



- **Sort** — Use to select a sorting method from the following options:



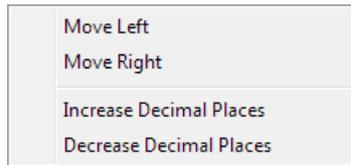
The default sort option is **No origin sort**. Select a sorting method from the options.

You can also sort the schedule using the **Diameter** column. Click on the arrow alongside **Diameter** to reverse the sorting order.

2 Modify the layout of the columns as required:

- a Click in the column header using the right mouse button.

- b Select the option you require from the column header context menu.



Move left — Move the selected column to the left.

Move Right — Move the selected column to the right.

Increase decimal places — Increase the decimal places in the column data. This option is only active if the column contains numerical data.

Decrease decimal places — Decrease the decimal places in the column data. This option is only active if the column contains numerical data.

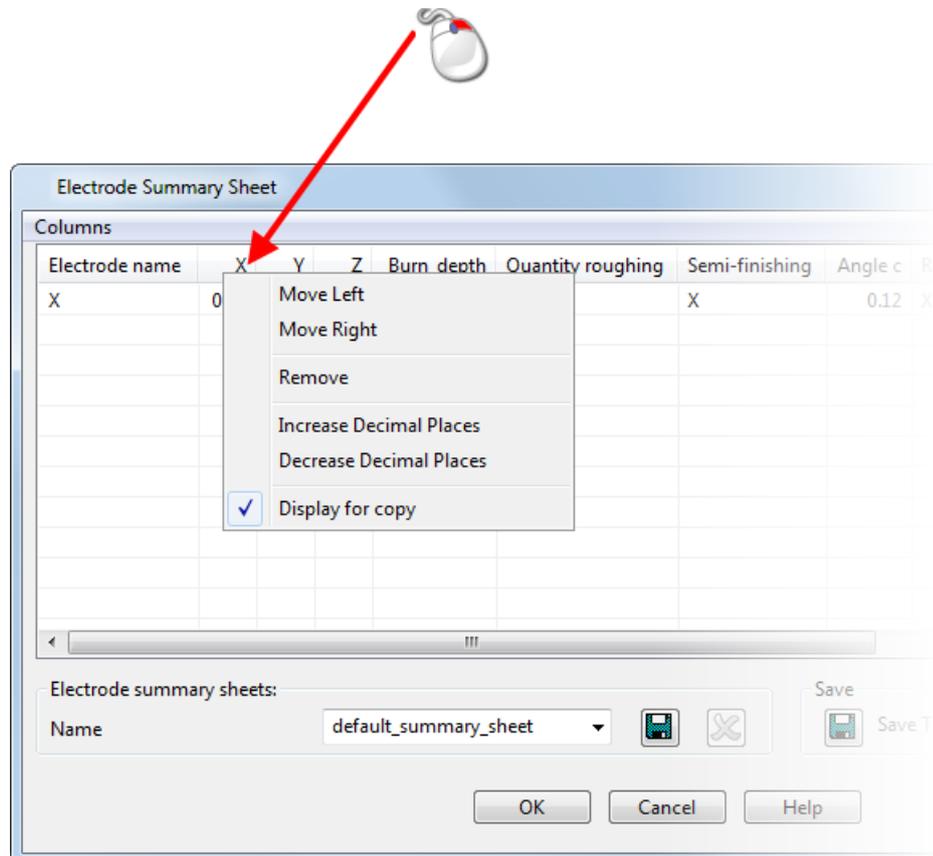
- 3 If required, applying balloons and labels (see page 279) to components in drawings that are selected for the schedule.
- 4 Create the schedule on the drawing, or save it to a file. Use  to toggle between displaying the data as an embedded Microsoft Excel spreadsheet or saving the data in [.xlsx](#) or [.csv](#) format..

Electrode Summary Sheet

You can generate an **Electrode Summary Sheet** that includes information about the electrode.

- Click the **Electrode summary sheet**  button on the **Schedule toolbar** to display information in the default layout on the **Electrode Summary** dialog.

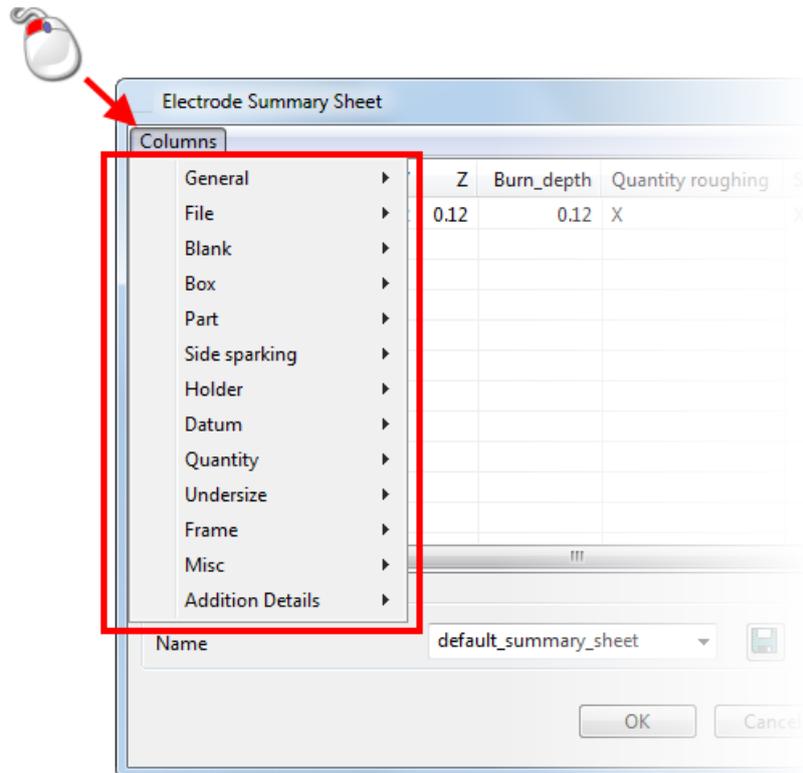
Right-click the column headings in the **Electrode Summary** dialog and use the options on the context menu to customise the layout of the summary sheet.



The context menu includes options to:

- add or remove standard columns.
- move columns left or right
- set the number of decimal places for numeric attributes
- select the attributes to display for a copy of an electrode
- The following functionality has also been added:

- Click **Columns** to display a list of columns that can be included in the sheet.



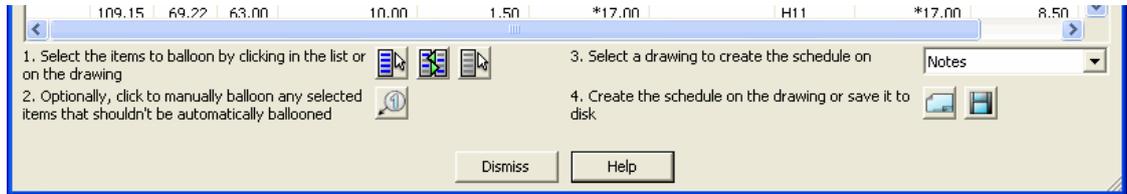
- Use an Excel template to add additional calculations or company logos.
- Dynamically change the order of the columns by dragging the column heading to the required position.
- When you click **OK**, you can select to save the summary sheet in **.csv** (Comma Delimited) or **.xlsx** (MS Excel) format.
- Electrode summary sheets created on electrode template drawings are updated when an electrode is added to an electrode drawing. Electrode copies are added under the parent electrode in the spreadsheet.



*You can also display the Electrode Summary Sheet by selecting **Modify Layout ...** from the **Worksheet** context menu on the setup sheet.*

Applying balloons and labels to schedules

The lower half of the **Schedules** dialog contains instructions and options for applying balloons and labels to items in drawings that are selected for the schedule.



- 1 Select the items to balloon by clicking one of the following buttons to define your selection, or using **Ctrl+click** to select individual items.



— Select all items in the list.

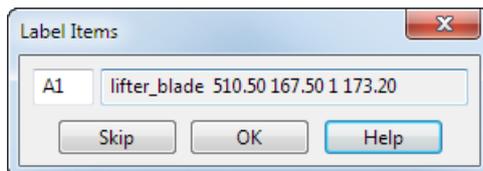


— Invert the list selection.

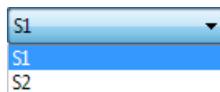


— Clear the selection in the list.

- 2  — Click to create balloons manually for selected items. The **Label Items** dialog is displayed. Use this dialog to change the name of the label series that is used when adding balloons manually.



- 3 Select the drawing where you want to create the schedule.

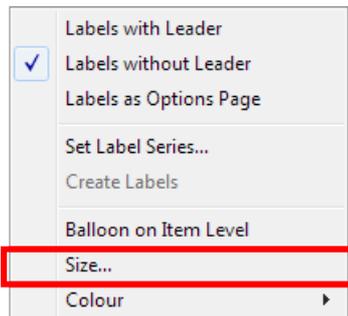


- 4 Click one of the following buttons to create the schedule .

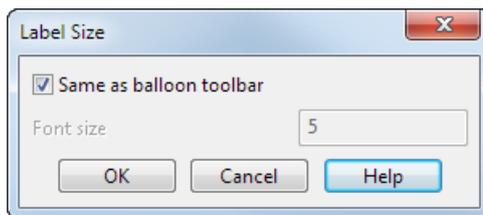
- **Create a schedule on the drawing** .
- **Save the schedule to a file** .

Changing the font size of the labels

You can define the font size of labels using the **Size** option on the **Labels** menu of the **Schedule** or **Bill of Materials** dialog.



Selecting this option displays the **Label Size** dialog.



Same as balloon toolbar — Select this option to use the label size that is defined on the **Balloon** toolbar. Deselect it to enter a new font size.

Font size — Enter the font size.



Font sizes for Schedule and Bill of Materials are independent, so you can change the font size for the Schedule and use a different size for the Bill of Materials.

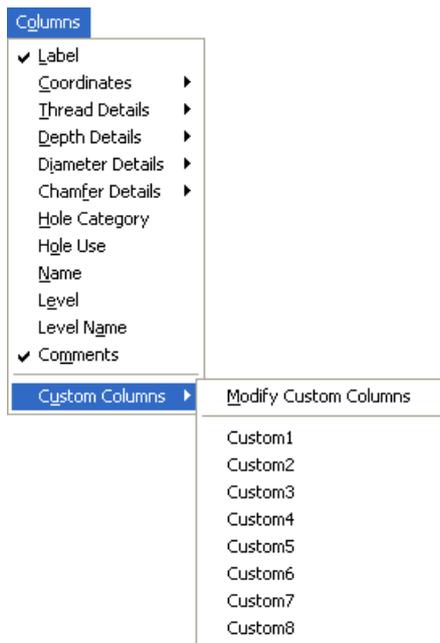
Creating customised columns of data

You can create up to 8 customised columns of data.

1 Select **Columns** menu on the dialog.



2 Select Custom columns



3 Select **Modify custom columns** to display the Custom columns dialog. (see page 313)

4 Use the dialog to create a customised column.

Updating the embedded schedule

To display the **Worksheet** context menu, right-click on the embedded schedule.



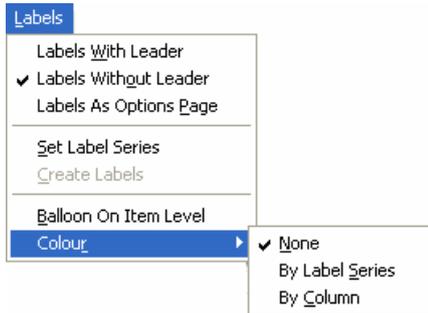
Modify — Select this option to open Microsoft Excel. Make the changes to the embedded schedule and **Save** the worksheet.

Update — Select this option to update the schedule with any changes to the items in the drawing view.

Convert — Select this option to convert the embedded schedule to text.

Adding colour to the embedded schedule

To apply colour to the embedded schedule, select **Colour** from the options on the **Label** menu of the Schedule dialog (see page 271).



None — Select this option to have no colour in the schedule.

By Label Series — Select this option to apply colours to different label types. In this example, the A1 labels are coloured green:

	F	G	H	I	O	P	Q	R	S	T	U	V
1	Label	Top X	Top Y	Top Z	Thread Diameter	Thread Top Diameter	Thread Base Diameter	Thread Pitch	Thread Depth	Through Thread	Thread Start Z	Thread Name
3	A1	-61.00	-61.00	-44.00								
4		-61.00	61.00	-17.00								
5		61.00	61.00	-17.00								
6		-61.00	-61.00	-17.00								
7		61.00	-61.00	-17.00								
8		-61.00	61.00	-44.00								
9		61.00	61.00	-44.00								
10		61.00	-61.00	-44.00								
11		-61.00	61.00	17.00								
12		61.00	61.00	17.00								
13	A1	-61.00	-61.00	17.00								
14		61.00	-61.00	17.00								
15		-61.00	-61.00	-17.00								
16		61.00	-61.00	-17.00								
17		-61.00	61.00	-17.00								
18		61.00	61.00	-17.00								
19		-103.00	78.00	28.00								
20		0.00	11.00	-17.00								
21		0.00	11.00	-109.00								
22		0.00	0.00	-6.89								
23	A1	0.00	11.00	-112.00								
24		0.00	0.00	-44.00								
25		0.00	0.00	-73.00								
26		-61.00	61.00	17.00								
27		61.00	61.00	17.00								
28		-61.00	-61.00	17.00								
29		61.00	-61.00	17.00								
30		-61.00	-61.00	90.00								
31		61.00	-61.00	-90.00								
32		-61.00	61.00	-90.00								
33		61.00	61.00	-90.00								
34		0.00	0.00	39.00								
35		0.00	0.00	-17.00								
36		0.00	11.00	-44.00								
37		0.00	11.00	-90.00								
38		0.00	11.00	-73.00								
39	A1	-61.00	-36.00	17.00	10.00	10.00		1.50	17.00	*		I.S.O. Metric Coarse 10mm
40	A1	-61.00	-36.00	-17.00	10.00	10.00		1.50	17.00	*		I.S.O. Metric Coarse 10mm
41	A1	-34.00	68.00	-90.00	4.00	4.00		0.70	7.00			I.S.O. Metric Coarse 4.0mm
42		-61.00	36.00	17.00	10.00	10.00		1.50	17.00	*		I.S.O. Metric Coarse 10mm
43		61.00	36.00	17.00	10.00	10.00		1.50	17.00	*		I.S.O. Metric Coarse 10mm
44		61.00	-36.00	17.00	10.00	10.00		1.50	17.00	*		I.S.O. Metric Coarse 10mm
45		61.00	-36.00	-17.00	10.00	10.00		1.50	17.00	*		I.S.O. Metric Coarse 10mm
46		-61.00	36.00	-17.00	10.00	10.00		1.50	17.00	*		I.S.O. Metric Coarse 10mm
47		61.00	36.00	-17.00	10.00	10.00		1.50	17.00	*		I.S.O. Metric Coarse 10mm

By Column — Click on a column heading to sort the schedule. Now select the **By Column** option to apply colour according to the sorted order. The worksheet below was sorted on the Thread Diameter column before **By Column** was selected.

	F	G	H	I	O	P	Q	R	S	T	U	V
1	Label	Top X	Top Y	Top Z	Thread Diameter	Thread Top Diameter	Thread Base Diameter	Thread Pitch	Thread Depth	Through Thread	Thread Start Z	Thread Name
3	A1	-34.00	68.00	-90.00	4.00	4.00	4.00	0.70	7.00			I.S.O. Metric Coarse 4.0mm
4		-34.00	68.00	-90.00	4.00	4.00	4.00	0.70	7.00			I.S.O. Metric Coarse 4.0mm
5		34.00	68.00	-90.00	4.00	4.00	4.00	0.70	7.00			I.S.O. Metric Coarse 4.0mm
6	A1	-34.00	68.00	-90.00	4.00	4.00	4.00	0.70	7.00			I.S.O. Metric Coarse 4.0mm
7		-34.00	90.00	-84.00	8.00	8.00	8.00	1.25	9.00	*		I.S.O. Metric Coarse 8mm
8		-34.00	90.00	-84.00	8.00	8.00	8.00	1.25	9.00	*		I.S.O. Metric Coarse 8mm
9		-34.00	-90.00	-64.00	8.00	8.00	8.00	1.25	9.00	*		I.S.O. Metric Coarse 8mm
10		34.00	60.00	64.00	8.00	8.00	8.00	1.25	9.00	*		I.S.O. Metric Coarse 8mm
11	A1	-61.00	-36.00	17.00	10.00	10.00	10.00	1.50	17.00	*		I.S.O. Metric Coarse 10mm
12	A1	-61.00	-36.00	-17.00	10.00	10.00	10.00	1.50	17.00	*		I.S.O. Metric Coarse 10mm
13		-61.00	-36.00	17.00	10.00	10.00	10.00	1.50	17.00	*		I.S.O. Metric Coarse 10mm
14		-61.00	-36.00	17.00	10.00	10.00	10.00	1.50	17.00	*		I.S.O. Metric Coarse 10mm
15		61.00	-36.00	17.00	10.00	10.00	10.00	1.50	17.00	*		I.S.O. Metric Coarse 10mm
16		61.00	-36.00	-17.00	10.00	10.00	10.00	1.50	17.00	*		I.S.O. Metric Coarse 10mm
17		-61.00	36.00	-17.00	10.00	10.00	10.00	1.50	17.00	*		I.S.O. Metric Coarse 10mm
18		-61.00	36.00	-17.00	10.00	10.00	10.00	1.50	17.00	*		I.S.O. Metric Coarse 10mm
19	A1	-61.00	61.00	-44.00								
20		-61.00	61.00	-17.00								
21		61.00	61.00	-17.00								
22		-61.00	-61.00	-17.00								
23		61.00	-61.00	-17.00								
24		-61.00	61.00	-44.00								
25		61.00	61.00	-44.00								
26		61.00	-61.00	-44.00								
27	A1	-61.00	-61.00	17.00								
28		-61.00	61.00	17.00								
29		61.00	61.00	17.00								
30		61.00	-61.00	17.00								
31		-61.00	-61.00	-17.00								
32		61.00	-61.00	-17.00								
33		-61.00	61.00	-17.00								
34		61.00	61.00	-17.00								
35		-103.00	78.00	28.00								
36		0.00	11.00	-17.00								
37		0.00	11.00	109.00								
38		0.00	0.00	-6.69								
39	A1	0.00	11.00	-112.00								
40		0.00	0.00	-44.00								
41		0.00	0.00	-73.00								
42		-61.00	61.00	17.00								
43		61.00	61.00	17.00								
44		-61.00	-61.00	17.00								
45		61.00	-61.00	17.00								
46		-61.00	-61.00	-90.00								

Personalising the Schedule spreadsheet

To include personalised information on a Schedule spreadsheet, such as a company logo or information, you must first set up a spreadsheet (.xls) template in your `homes/pshapeX` directory.

- 1 Create a spreadsheet (.xls) called `schedule.xls` saved to your `homes/pshapeX` directory.
- 2 In the middle of your `schedule.xls` spreadsheet, create a single cell that contains the text:

Insert data here

- 3 Add personalised details above or below the single text cell, in the positions you want the details to be displayed on the spreadsheet.

When you create a Schedule spreadsheet by saving to file, the `schedule.xls` spreadsheet is copied into the Schedule and data from the **Schedule** dialog (see page 271) is inserted at the location of this single cell.

Creating/appending a schedule

When you create or append a schedule, it is created for a specific type of object.

If you have MS-Excel installed on your machine, **Use spreadsheet to create schedule** (**Schedules** page of the **Drafting** options), is *ON* by default and you will create embedded schedules automatically. If you do not have MS-Excel installed, or you do not want to create an embedded schedule, **Use spreadsheet to create schedule** should be *OFF*.

For further details, see Creating an Embedded Schedule (see page 270).

You can either create the schedule on the screen or save it in a file.



- 1 Click  (main toolbar). The **Main** toolbar changes to the **Drafting** toolbar and displays the drafting tools.
- 2 Select the specific type of object for which you want to create a schedule.

You can choose one of the following:

- assemblies
- schedule
- balloons
- circles
- workplanes
- symbols/component
- points
- solids

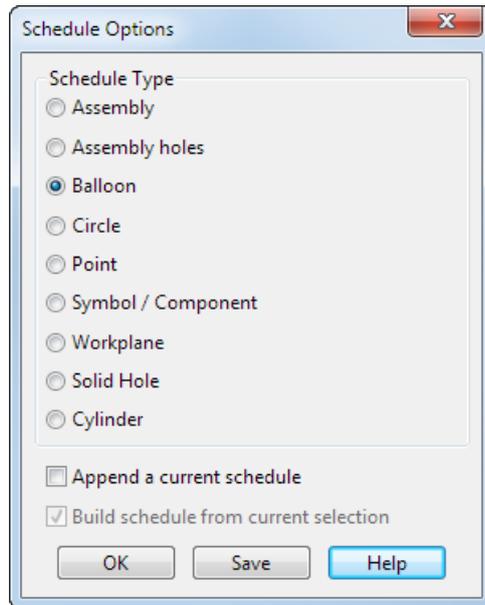
If your selection contains a mixture of objects, all the objects not required are ignored.

If you select nothing, all objects in the graphics window are treated as selected.

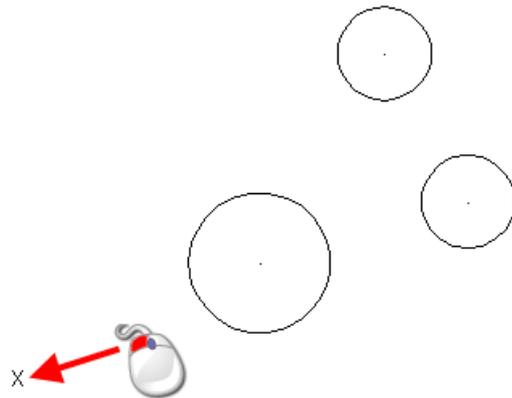
- 3 Click  (drafting toolbar) to display the **Schedule and BOM** toolbar.
- 4 Click  (schedule and BOM toolbar). The **Schedule Options** dialog is displayed. The *Balloon toolbar* (see page 151) is also displayed. This can be used to edit the text in schedules.
- 5 Use the *Schedule Options dialog* (see page 271) to select the type of schedule.

Schedule Options dialog

Use this dialog to create and append schedules using specific objects.

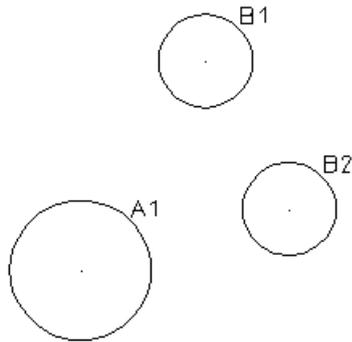


- 1 Set the options on the dialog. If you want to create a schedule on the screen, click a position.



The top left corner of the schedule is positioned where you click.

You can set the first label using the **Counter** option in the **Balloon** toolbar.



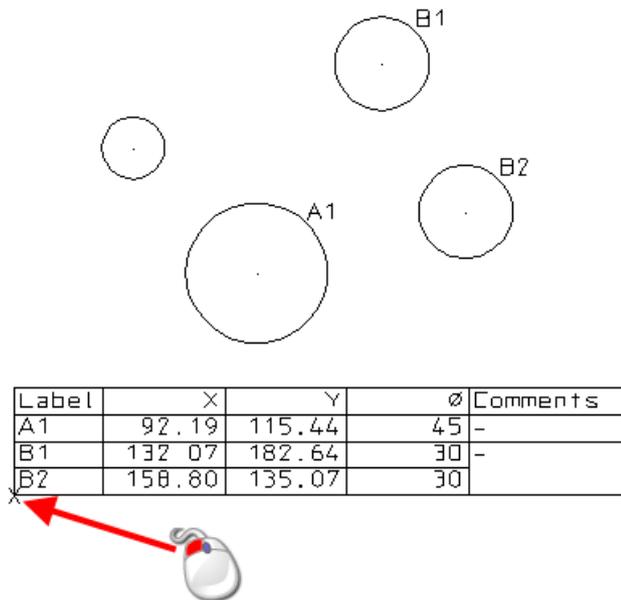
Label	X	Y	∅	Comments
A1	92.19	115.44	45	-
B1	132.07	182.64	30	-
B2	158.80	135.07	30	-

Schedule type — These are the objects for which schedules can be created.

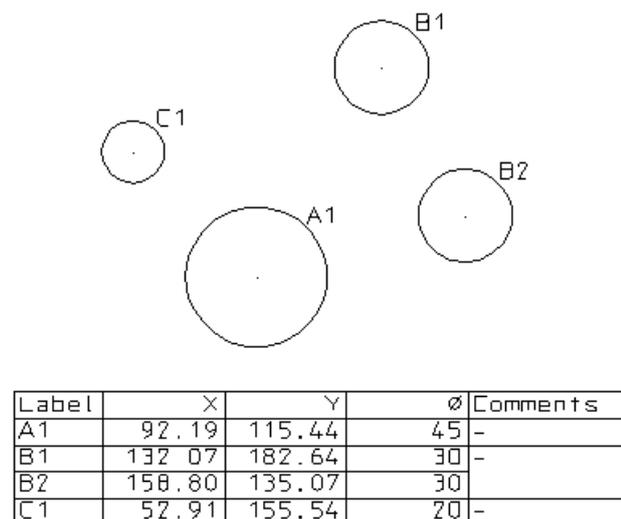
- Assembly (see page 288)
- Assembly holes (see page 289)
- Balloon (see page 147)
- Circle (see page 293)
- Point (see page 294)
- Symbol/Component (see page 295)
- Workplane (see page 296)
- Solid Hole (see page 297)
- Cylinder (see page 298)

Append a current schedule — This appends a schedule to an existing one. The appended schedule does not contain a top row with headings. It relies on the existing schedule for the top row.

Make sure you click the bottom left corner of an existing schedule to add the new rows.



The rows are appended to the schedule as shown below.



Build schedule from current selection — If *ON*, this will use the objects from the current selection. Otherwise, the objects are selected from everything in the graphics window.

OK — This removes the dialog from the screen.

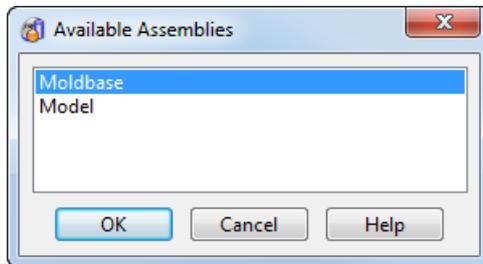
Save — Click this button to save the schedule as an .html, .csv or .txt file.

- The [.html](#) file can be viewed and printed in a browser.
- The [.csv](#) file can be read into spreadsheet program such as Excel.
- The [.txt](#) file can be read by another program such as a word processor.

Assemblies

A schedule for an assembly is a bill of material. This is a table of components/sub-assemblies used by an assembly.

If you are creating a schedule for an assembly and there are two or more assemblies in the model, you must select the assembly to be used to create the schedule, using the **Available Assemblies** dialog is displayed.

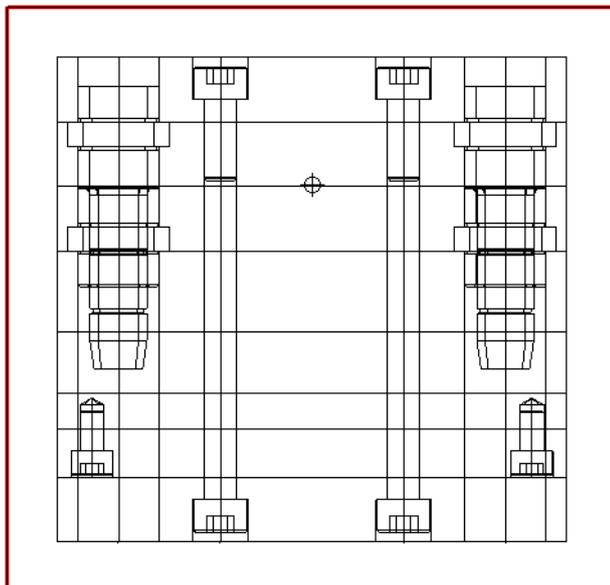


To create a schedule for one of the assemblies,

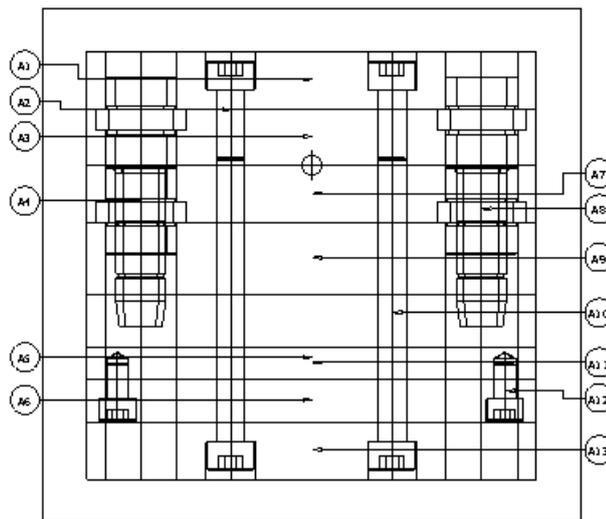
- 1 Select an assembly from the dialog and
- 2 Click **OK**.

If the assembly is in an active drawing view, the components are given labels in the schedule and balloons using those labels mark the components in the drawing view. You choose the shape of the balloons on the **Balloon** page of the **Options** dialog. For further details, see Setting the balloons on the Options dialog (see page 152).

Here is an assembly in an active view:



The following schedule is created for the assembly.



Label	Part	Catalogue	Material	Quantity
A1	FP1212M	DMS	MS	1
A2	B20CS	DMS		4
A3	R121216MS	DMS	MS	1
A4	SGP141613	DMS		4
A5	R12123E	DMS	MS	1
A6	ES1212(base)	DMS	MS	1
A7	R121216MS CorePlate	DMS	MS	1
A8	SGB1416	DMS		4
A9	BP1212	DMS	MS	1
A10	B80CS	DMS		4
A11	ES1212(cover)	DMS	MS	1
A12	B10CS	DMS		4
A13	FP1212M BottomClampingPlate	DMS	MS	1

The schedule groups all the components created from the same component definition.

The schedule can contain the following columns:

- **Label** — This label is used by the balloon to mark the component.
- **Part** — The name of the component used to define the component.
- **Catalog** — The name of the catalog if a component is selected from a catalog in **Delcam Toolmaker**. If there is no catalog name, the component is listed as user defined. If a component has been modified, its catalog name will be marked with an asterisk.
- **Material** — The material of the component.
- **Quantity** — Number of components defined from the same registered component.



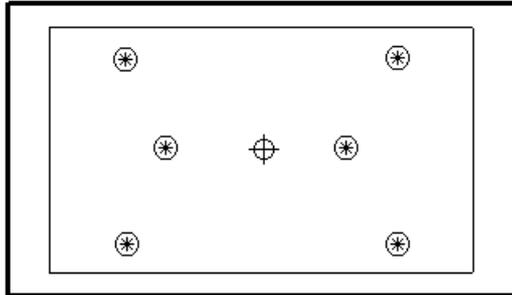
*If the schedule that is created is empty and no balloons are added to the drawing, use the context menu to check that you have made the view **Active**.*

Assembly holes

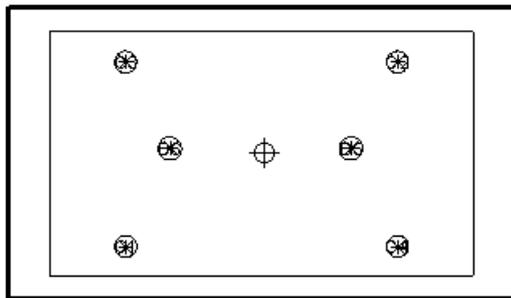
A schedule from assembly holes is a table containing details of the holes in the assembly.

To create the schedule, the assembly must be in an **Active** drawing view.

The model below is part of an assembly in an **Active** view. It is the view of an ejector plate in a mold base.



The following schedule is created for the holes in the ejector plate.



Hole	X	Y	Ø	Depth	Comments
B5	37.00	-2.00	8 / 10	34 / 9	
B6	-45.00	-2.00	8 / 10	34 / 9	
C1	-50.00	-34.00	8	9	
C2	50.00	34.00	8	9	
C3	-50.00	34.00	8	9	
C4	50.00	-34.00	8	9	

If the default **Holes To Include** setting is used from the **Schedule** page of the **Options** dialog, only holes that are normal to the view are included in the schedule.

Only the holes of the selected components are included in the schedule. If none are selected, the holes of all the components are included.

Each hole is given a label in the view. All holes with the same size diameter start with the same letter and have the same Comment column.

If holes lie in the same line and have the same diameter, they are given the same label.

Through holes are now marked with an asterisk * in the schedule.

Label	X	Y	Ø	Depth	Comments
A1	-29.33	41.08	10	20	
A2	8.00	-9.00	10	*60	
A3	-29.00	-38.00	10	20	
A4	5.36	-46.49	10	*60	
B1	-29.00	19.00	15	*60	
B2	-55.00	-31.00	15	*60	
C1	12.00	31.00	20	20	
C2	-57.00	9.00	20	20	

*Through holes

The schedule displays the labels, centres, diameters and depth of the holes. If a hole is tapered, it has two diameters.

There are options you can set for the schedule. For further details, see Setting the schedule options on the Options dialog (see page 299).

Balloons

A schedule from balloons is a table containing positions marked by pointers of balloons. It can be created from all balloon types except geometric tolerances, surface textures, weld symbols and point balloons.

If all balloons are snapped to components, a bill of materials will be produced.

For further details, see Entering balloon creation mode (see page 149).

Three schedule balloons are given in the example below.

A2

A1

B1

The following schedule is created for these balloons.

A2

A1

B1

Label	X	Y	Comments
A1	75.32	136.94	-
A2	115.41	166.24	-
B1	103.59	122.55	-

The coordinates of the pointers of the balloons are displayed in the schedule.

For schedule balloons, no pointer is visible. In this case, the pointer is the first position entered when you create the balloon.

If the pointer lies on a drawing or a model, then its coordinates are measured from the origin of the current workspace. If the pointer lies on a drawing view, then its coordinates are measured from the origin of the drawing view.

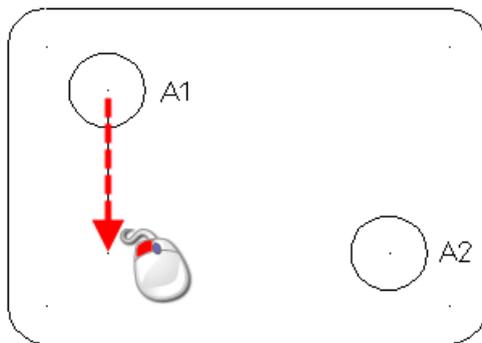
A comment column is also drawn in the schedule. Balloons A1 and A2 are from the same series. Therefore, the entries for their comments are merged.

By default, only the X and Y coordinates are displayed. To display the Z coordinate, turn on the **Include Z coordinate** option on the **Schedule** page of the **Options** dialog.

There are other options you can set. For further details, see Setting the schedule options on the Options dialog (see page 299).

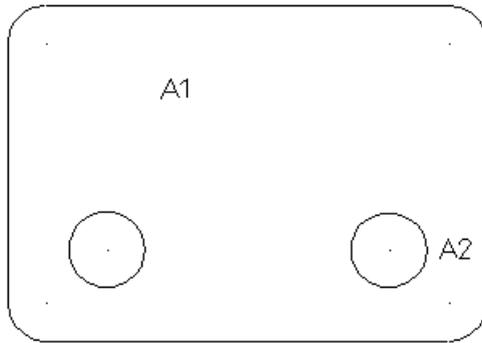
When you create a balloon by clicking its pointer on an object, say, the centre of a circle, the pointer's position and the centre of the circle become related. If you move the centre of the circle, the coordinates of the balloon update in the schedule.

In the example below, balloon A1 is related to the centre of the circle on its left. We move the circle downwards.



Label	X	Y	Comments
A1	105.03	136.91	-
A2	179.20	93.67	-

When the circle is moved, the entry in the schedule for *A1* updates too.

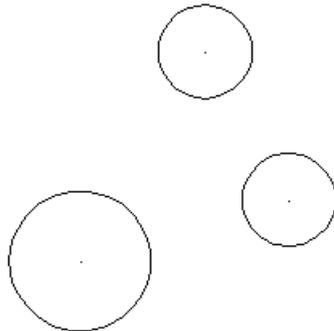


Label	X	Y	Comments
A1	105.03	93.67	-
A2	179.20	93.67	

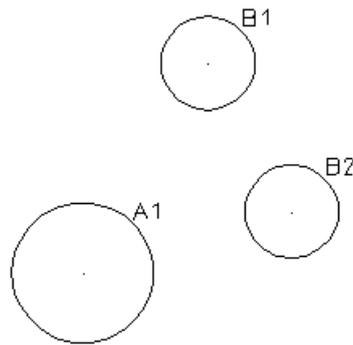
Circles

A schedule from circles is a table containing the centre positions of circles. It can also contain the diameters of the circles. For further details see [Creating a full arc \(circle\)](#)

Three circles are given below.



The following schedule is created for these circles.



Label	X	Y	Ø	Comments
A1	92.19	115.44	45	-
B1	132.07	182.64	30	-
B2	158.80	135.07	30	-

Each circle is labelled using the format AAxx where AA is any number of letters and xx is any number of digits. Circles with the same diameters have the same prefix and are put next to each other in the schedule. The label is automatically set from the **Counter** in the **Balloon** toolbar.

In our example, two circles have the diameter 30. The labels of these circles are prefixed with a B and the circles are grouped together in the schedule.

The coordinates of the centres and the diameters of the circles are displayed in the schedule. A comment column is also drawn in the schedule. The comments of circles with the same diameters are merged.

By default, only the X and Y coordinates are displayed. To display the Z coordinate, turn on the **Include Z coordinate** option on the **Schedule** page of the **Options** dialog.

For further details, see Setting the schedule options on the Options dialog (see page 299)

If you move the circle or change its diameter, the values update in the schedule.

Points

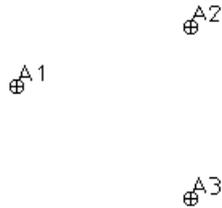
A schedule from points is a table containing the coordinates of the points.

For further details, see Creating a point.

Here are some points.



The following schedule is created for these points.



Label	X	Y	Comments
A1	86.67	145.56	-
A2	141.95	164.74	
A3	141.95	109.74	

The coordinates of the points are displayed in the schedule. A comment column is also drawn in the schedule. The rows of the comment column are merged for the selected points.

By default, only the X and Y coordinates are displayed. To display the Z coordinate, turn on the **Include Z coordinate** option on the **Schedule** page of the **Options** dialog.

There are other options you can set. For further details, see [Setting the schedule options on the Options dialog](#) (see page 299).

If you move a point, its coordinates update in the schedule.

Symbols/components

A schedule from symbols (see page 264) is a table containing the coordinates of the pins used to position the symbols.

Components are part of an assembly. A schedule from components is similar to the one for assemblies.

Here are some symbols.



The following schedule is created for these symbols.



Label	X	Y	Comments
A1	105.86	148.95	-
A2	199.36	148.95	

The coordinates of the pins used to position the symbols are displayed in the schedule. A comment column is also drawn in the schedule. The rows of the comment column are merged for the selected symbols.

By default, only the X and Y coordinates are displayed. To display the Z coordinate, turn on the **Include Z coordinate** option on the **Schedule** page of the **Options** dialog.

There are other options you can set.

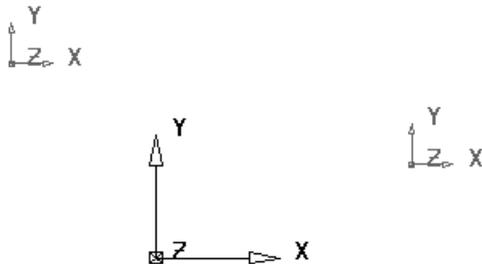
For further details, see Setting the schedule options on the Options dialog (see page 299).

If you move the symbol, the coordinates update in the schedule.

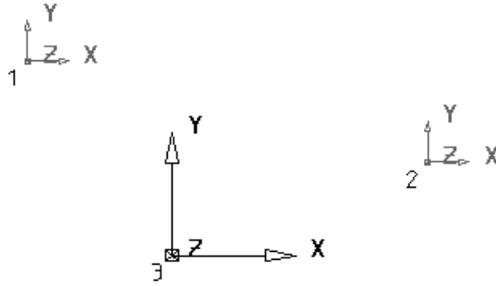
Workplanes

A schedule from workplanes is a table containing the origins of workplanes.

Three workplanes are given below.



The following schedule is created for these workplanes.



Label	X	Y	Comments
1	-46.08	62.20	-
2	81.41	29.82	-
3	0.00	0.00	-

Each workplane is labelled by its name. The coordinates of the origins of the workplanes are displayed in the schedule. The coordinates are relative to the active workplane. A comment column is also drawn in the schedule.

By default, only the X and Y coordinates are displayed. To display the Z coordinate, turn on the **Include Z coordinate** option on the **Schedule** page of the **Options** dialog.

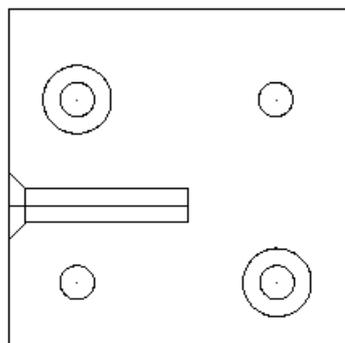
There are other options you can set. For further details, see [Setting the schedule options on the Options dialog](#) (see page 299).

If you move the workplane, the coordinate values update in the schedule.

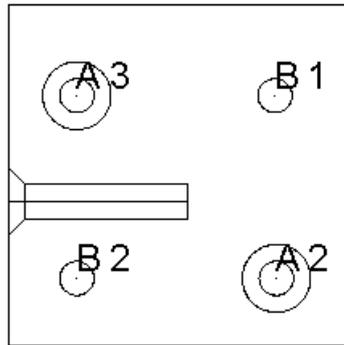
Solid holes

A schedule from holes in solids is a table containing the origin, the diameter and the depth of the holes. For further details see [Creating a solid hole](#).

A solid with holes is given below.



The following schedule is created for the holes.



Label	X	Y	Ø	Depth	Comments
A2	16.00	-19.25	6 / 12	12 / 6	
A3	-19.00	13.00	6 / 12	12 / 6	
B1	15.75	13.00	6	12	
B2	-19.00	-19.25	6	12	

The schedule groups all the holes created from the same type of hole. It displays the centres of the holes, their diameters and depth.

If the default **Holes To Include** setting is used from the **Schedule** page of the **Options** dialog, only holes which are normal to the view are in the schedule.

Through holes are now marked with an asterisk * in the schedule.

Label	X	Y	Ø	Depth	Comments
A1	-29.33	41.08	10	20	
A2	8.00	-9.00	10	*60	
A3	-29.00	-38.00	10	20	
A4	5.36	-46.49	10	*60	
B1	-29.00	19.00	15	*60	
B2	-55.00	-31.00	15	*60	
C1	12.00	31.00	20	20	
C2	-57.00	9.00	20	20	

*Through holes

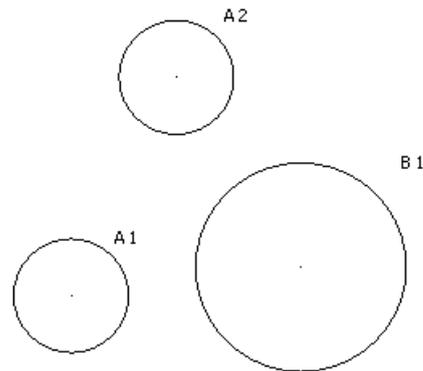
By default, only the X and Y coordinates are displayed. To display the Z coordinate, turn on the **Include Z coordinate** option on the **Schedule** page of the **Options** dialog.

There are other options you can set. For further details, see Setting the schedule options on the Options dialog (see page 299).

Cylinders

A schedule from cylinders is a table containing the centre positions, diameters and depths of cylinders.

A schedule is created for the three cylinders given below.



Label	X	Y	Ø	Depth	Comments
A1	-110.00	-5.00	60	127	
A2	-55.00	110.00	60	60	
B1	10.00	10.00	110	60	

*Through holes

Each cylinder is labelled using the format AAxx where

AA is any number of letters

xx is any number of digits.

Cylinders with the same diameters have the same prefix and are put next to each other in the schedule. The label is automatically set from the Counter in the Balloon toolbar.

In our example, two cylinders have the diameter 60. The labels of these cylinders are prefixed with an A and the cylinders are grouped together in the schedule.

The coordinates of the centres and the diameters of the cylinders are displayed in the schedule. A comment column is also drawn in the schedule. The comments of cylinders with the same diameters are merged.

By default, only the X and Y coordinates are displayed. To display the Z coordinate, turn on the **Include Z coordinate** option on the **Schedule** page of the **Options** dialog.

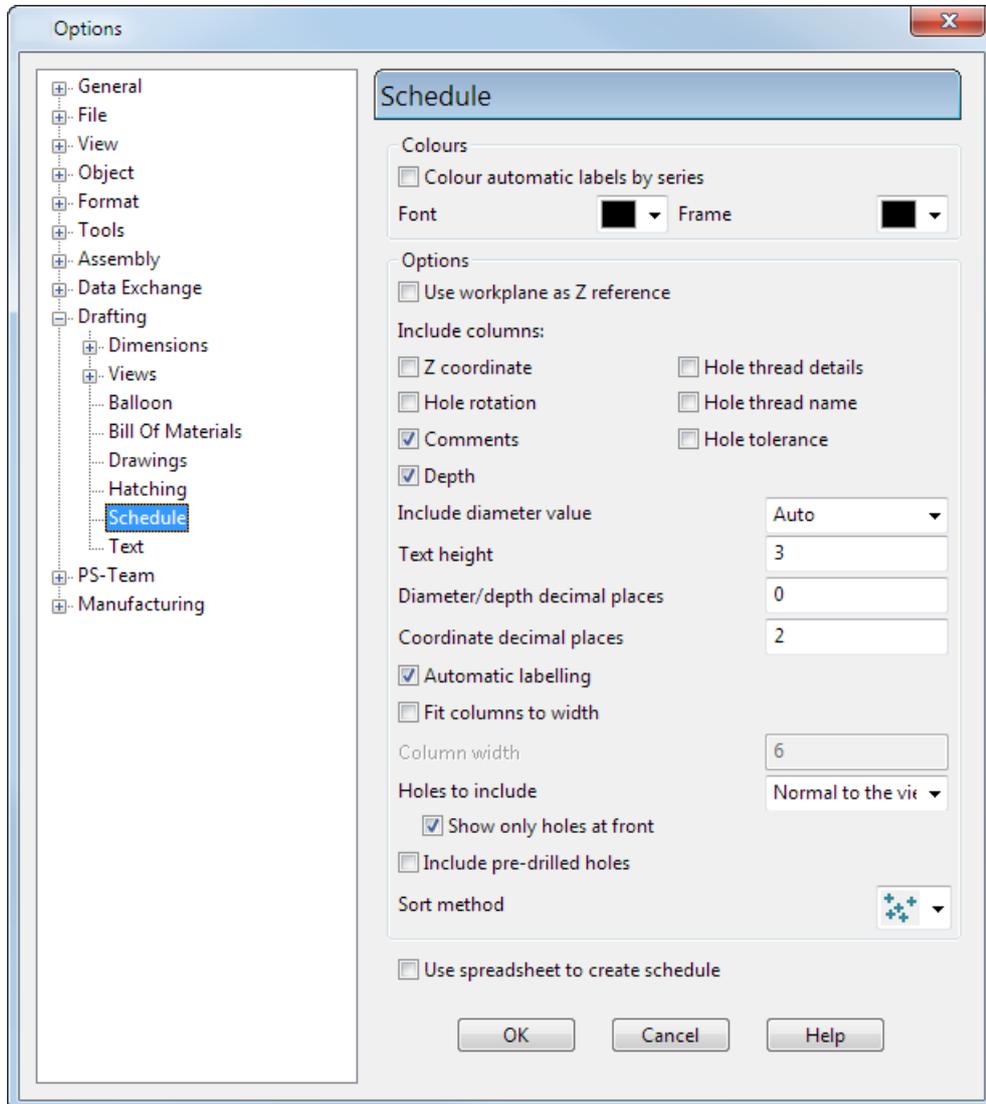
Setting the schedule options on the Options dialog

When creating and appending schedules, you can set certain aspects of their contents and layout.

- 1 From the **Tools** menu, select **Options** to display the **Options** dialog.
- 2 From the **Drafting** section of the browser tree, select **Schedule**. This displays the **Schedule** page of the **Options** dialog.

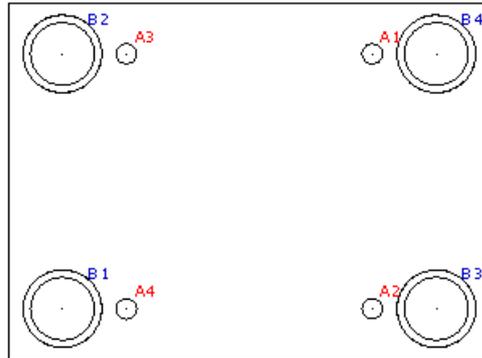


If you have Microsoft Excel installed, the **Use spreadsheet to create schedule** option at the bottom of this page is selected by default and the other schedule options are unavailable. This enables the enhanced **Schedules** and **Bill of Materials** functionality. If you do not have Microsoft Excel installed, this option is deselected. Deselect this option manually if you do not wish to use the enhanced functionality.



3 Use this dialog to set the default schedule settings.

Colour automatic labels by series — Select this option to change the colour of the labels as the series changes, for example:



Label	X	Y	∅	Comments
A1	254.41	469.39	9	
A2	254.41	315.39	9	
A3	154.41	469.39	9	
A4	154.41	315.39	9	
B1	128.41	315.39	26	
B2	128.41	469.39	26	
B3	280.41	315.39	26	
B4	280.41	469.39	26	

Font — Select the colour used by the text.

Frame — Select the colour used by the lines of the frame.

Use workplane as Z reference — Select this option to use the workplane to calculate the Z position.

Z coordinate — Select this option to include the Z coordinate in the schedule.

Hole rotation — Select this option to include columns for the X and Y rotation angle of the hole.

Comments — Select this option to include a column to add your comments when creating a schedule.

Depth — Select this option to include a column with depth when creating a schedule for assembly and solid holes.

Hole thread details — Select this option to include column for the pitch, depth, and start Z of the hole.

Hole thread name — Select this option to include a column that lists the name of the standard and size of the thread.

Hole tolerance — Select this option to include a column for the hole tolerance.

Include diameter value — Choose whether to display a column for diameter values. There are three options: **Auto**, **On**, and **Off**. **Auto** displays a diameter only if the schedule item is defined from the centre of a circle. **On** always displays a diameter column and a hyphen (-) is displayed if the schedule item is not defined from the centre of a circle. **Off** never displays a diameter column.

Text height — Enter the text height of the hole schedule text.



*This option is only available when **Use spreadsheet to create schedule** is deselected.*

Diameter/depth decimal places — Enter the number of decimal places to display for the diameter or depth value. If the units for the model are inches, additional decimal places are displayed.

Coordinate decimal places — Enter the number of decimal places to display for the coordinates. If the units for the model are inches, additional decimal places are displayed.

Automatic labelling — Select this option to display a label next to each object used in a schedule. The column heading of the object in the schedule is **Label**. If you deselect this option, no labels are displayed and the name of the object is used in the schedule. The column heading of object in the schedule is the name of object type (for example, **workplane**).

Fit columns to width — Select this option to set the widths of the X, Y, Z and diameter columns to the size specified in **Column Width**. Deselect it to enable PowerSHAPE to set the width of the columns automatically, to fit the size of the widest entry.

Column width — If you select Fit columns to width, enter the size of the width of the column. If the widest entry is larger than the column width, the width of all the columns adjust to fit that entry.

Holes To Include — Select which holes to include in a schedule:

At any angle — Include holes that are at any angle to the view.

Normal to the view — Include only holes that are normal to the view.

Whose faces are normal to the view — Include only holes whose face is normal to the view.

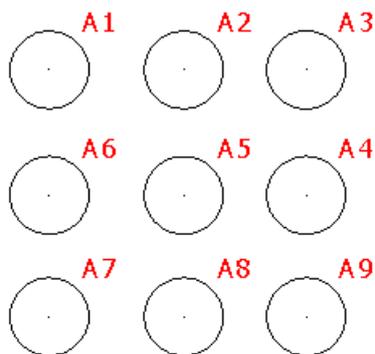
Show only holes at front — Select this option to show only holes at the front of the view.

Include pre-drilled holes — Select this option to include any pre-drilled holes in the object. Pre-drilled holes exist in some **Delcam-Toolmaker** components such as core and cavity plates.

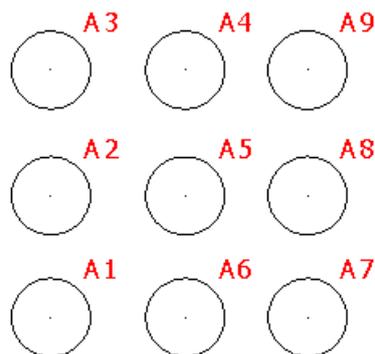
Sort method — Select how you want to label objects in the same series.

Use spreadsheet to create schedule — If you have Microsoft Excel installed, this option is selected by default and the other schedule options are unavailable. This enables the enhanced **Schedules** and **Bill of Materials** functionality. If you do not have Microsoft Excel installed, this option is deselected. Deselect this option manually if you do not wish to use the enhanced functionality.

The labels on the circles below use the **Sort Left To Right Then Back Again** method .



The same circles are shown using the **Sort Up Then Down** method .



4 Click **OK**.

Editing a schedule

A schedule is made of lines and text. These can be edited using the editing commands for lines and text. For further details, see *Editing a line* and *Editing text* (see page 126) for further details

The numbers in the schedule may depend on other objects. For example, a number representing the diameter of a circle will update if the diameter changes.

As well as appending new rows to the end of the schedule, you can append them to the middle of the schedule.

- 1 Use the **Stretch** command on the **Edit** toolbar to pull down the lower rows. For further details, see Edit - General Edits - Stretch.
- 2 Use the append command for that particular type of schedule to insert rows.

Embedded Bill of Materials

A Bill of Materials is a list of components used by an assembly and displayed in a table. The table of data can be embedded into a drawing as a Microsoft Excel spreadsheet, or saved as an OLE file.

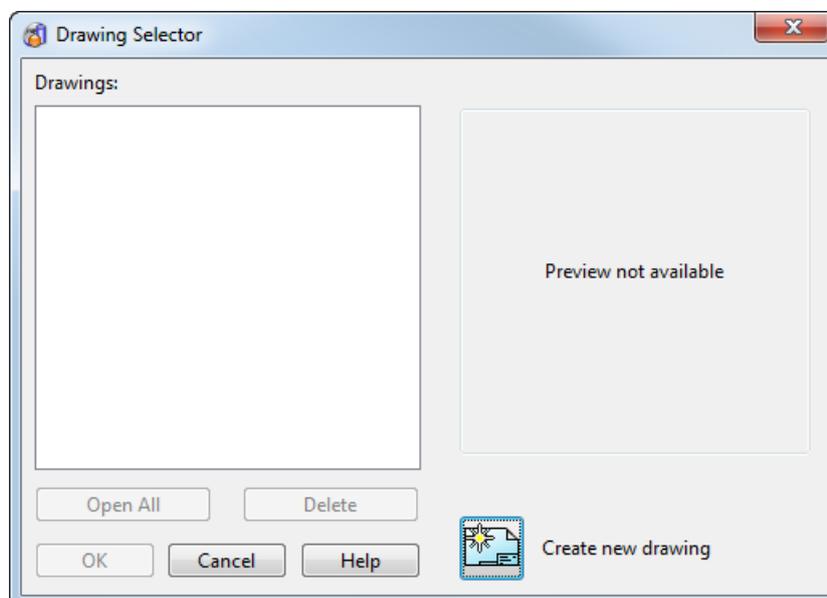


You need licences for Delcam Drafting and Microsoft Excel to use this menu option. If you do not have a licence for Microsoft Excel, you can create a Bill of Materials by

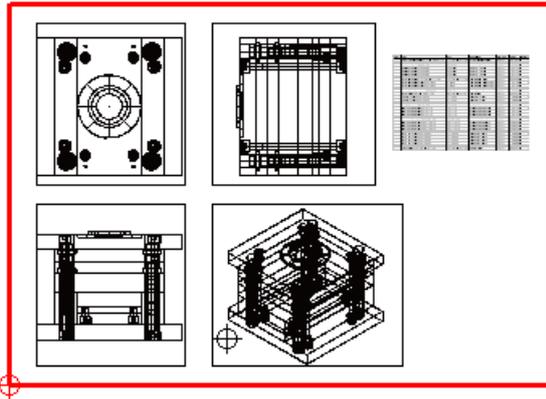
1. Selecting **Schedule** from the Object menu to display the **Schedule Options** dialog.
2. Selecting the **Assembly** option on the Schedule Options dialog.

Creating an embedded Bill of Materials

- 1 Click  in the main toolbar to enter Drafting Mode.
- 2 Open an assembly.
- 3 Click  to display the **Open Drawing** dialog.
- 4 Click **Create new drawing**  on the dialog.



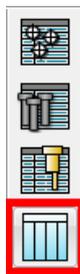
- 5 Create the views of the active assembly that you want to be labelled. The following example shows an assembly with 4 views.



- 6 Click **Schedule**  (*drafting toolbar*)



- 7 Select **Bill of Materials**  (*schedule and BOM toolbar*)

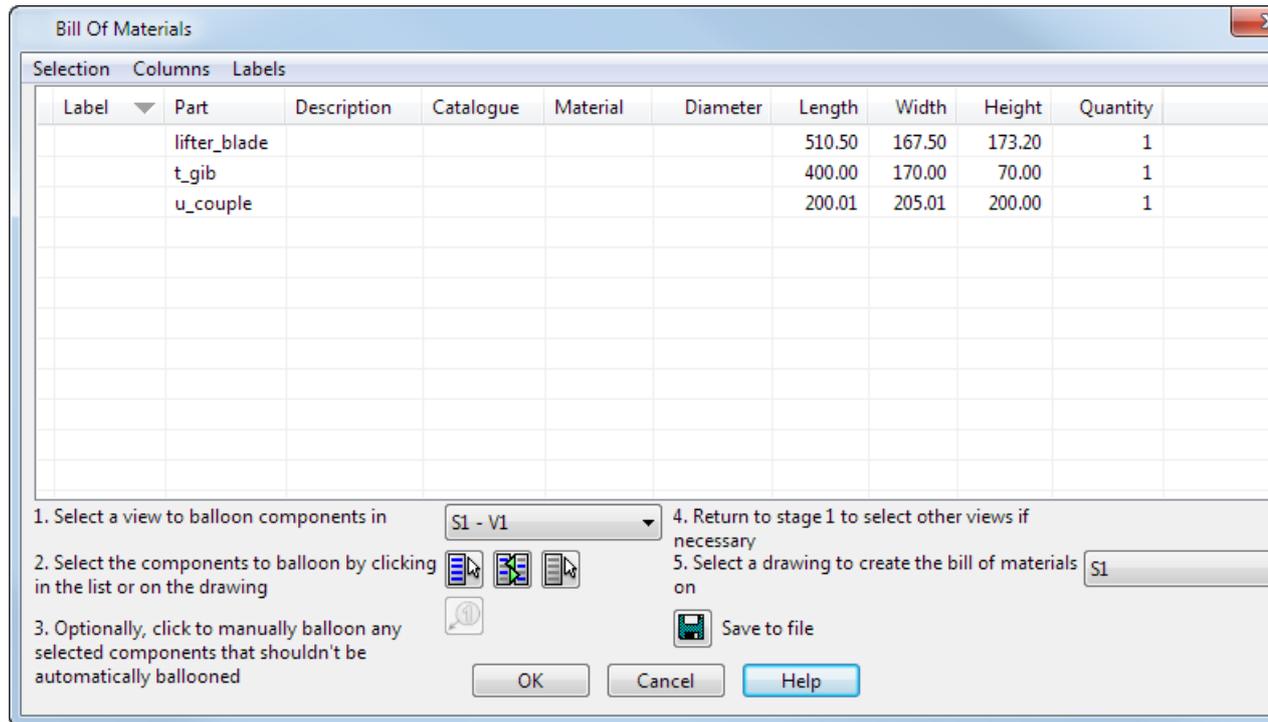


Delcam Drafting generates data from the active assembly and displays it in the **Bill of Materials** dialog. (see page 306)

- Components based on round primitives (cone, cylinder, torus, or spring) are specified using **Length x Diameter**.
 - A best-fit bounding box is used to specify sizes for components that are not based on primitives. This gives accurate sizes for components that are positioned at compound angles (for example, sliders)
 - If the part number and catalog of components are the same, but the descriptions are different, the description will be omitted in the Bill of Materials.
- 8 Use the **Bill of Materials** dialog (see page 306) to create an embedded Bill of Materials.

Bill of Materials dialog

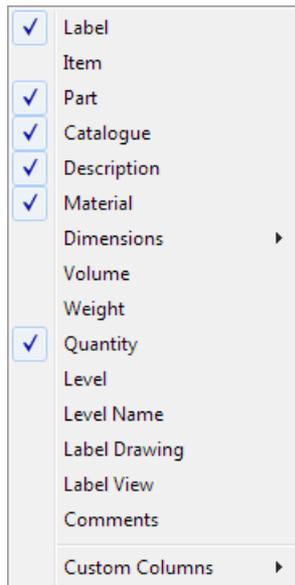
Use this dialog to specify the data that you want to be displayed as a Bill of Material:



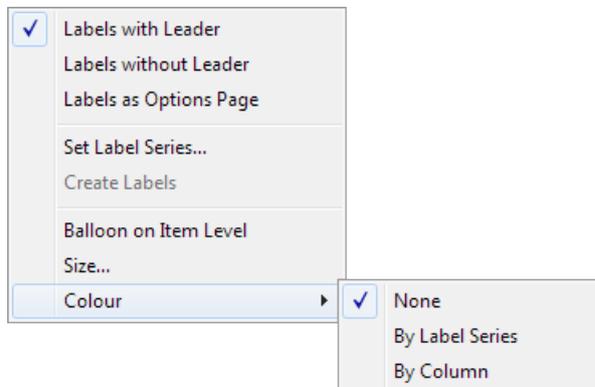
- 1 Use the menu options to define the data that is displayed in the dialog. The ✓ indicates that the option is selected and the data will be displayed in the bill of materials.
 - **Selection** controls the display of the data in the table.



- **Columns** defines the columns of data that are displayed.

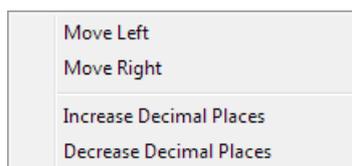


- **Labels** displays all the options for controlling labels and balloons. They work the same way as labels and balloons in schedules.



2 Modify the layout of the columns as required:

- a Click in the column header using the right mouse button.
- b Select the option you require from the column header context menu.



Move Left - Move the selected column to the left.

Move Right - Move the selected column to the right.

Increase Decimal Places - Increase the decimal places in the column data. This option is only active if the column contains numerical data.

Decrease Decimal Places - Decrease the decimal places in the column data. This option is only active if the column contains numerical data.



You can also click + drag to reposition the columns.

- 3 Modify the way items are displayed (see page 308) in the **Bill of Materials** dialog. If the items have a hierarchical structure (for example a sub-assembly), they are displayed in a summarised mode by default.
- 4 Apply balloons and labels to components in drawings that are selected for the bill of materials. For further details, see Applying label balloons and creating the Bill of Materials (see page 310).
- 5 Click  to toggle between displaying the data as an embedded Microsoft Excel spreadsheet (default) or saving the data as an OLE file.
- 6 Click **OK** to create the bill of materials on the drawing, or save it to a file.

Displaying the data in the Bill of Materials

The **Bill of Materials** dialog supports hierarchical structures such as sub-assemblies. Entries can be:

- Summarised; only top level components/sub-assemblies are listed. This is the default display mode.
- Expanded; a bold heading is added and the components are listed below, indented from the heading.
- Exploded; components in sub-assembly are listed.

In addition:

Blank can only be applied to top level entries.

Unblank only unblanks entries that you have explicitly blanked.

Changing the way the data is displayed in the BOM

Use the options on the context menu to specify the way the data is displayed in the Bill of Material table. The following will be updated to reflect the display mode that you select:

- columns that are displayed.
- quantities. The **Quantity** column is displayed in all modes.

The following example uses a slider sub-assembly to show the effect of using the different options:

- 1 The sub-assembly is currently displayed in summarised format.

Right-click the sub-assembly entry in the table to display the context menu.

Views	Label	Part	Description	Catalogue	Mate
S2-V1		Slider sub-assembly			
		Z31/4x40	Socket		

Select All
 Invert Selection
 Clear Selection
 Blank
 Unblank
 Summarise
 Expand
 Explode

- 2 Select **Expand** from the context menu. The table is displayed with the items from the sub-assembly displayed individually below the name of the sub-assembly. The individual items are indented.

Part	Width	Height	Quantity
Slider sub-assembly			1
Angle pin 1 1 8.9 100 - User defined			1
Guide strip 140 20 20.1 10.1 10 - User defined	20.10	140.00	2
Heel Block 50 50 50 100 23 0 - User defined	50.00	100.00	1
Slide Slide A 100 100 50 90 120 10 23 1 Steel User defined	120.00	100.00	1
Z31/4x40			6
Z31/4x40			3

- 3 Right click the sub-assembly heading and select **Summarise** from the context menu. The table is displayed in the summarised format.

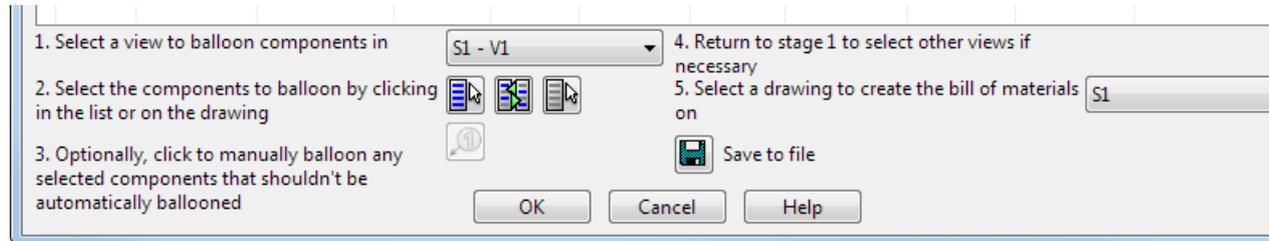
Part	Description	Width	Height	Quantity
Slider sub-assembly				1
Z31/4x40	Socket head cap screw			3

- 4 Select **Explode** to display the table with the items from the sub-assembly displayed individually.

Part	Width	Height	Quantity
Angle pin 1 1 8.9 100 - User defined			1
Guide strip 140 20 20.1 10.1 10 - User defined	20.10	140.00	2
Heel Block 50 50 50 100 23 0 - User defined	50.00	100.00	1
Slide Slide A 100 100 50 90 120 10 23 1 Steel User defined	120.00	100.00	1
Z31/4x40			9

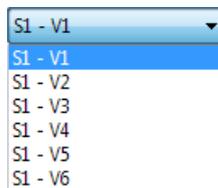
Applying label balloons and creating the Bill of Materials

The lower half of the **Bill of Materials** dialog contains instructions and options for applying balloons and labels to components in drawings that are selected for the bill of materials.

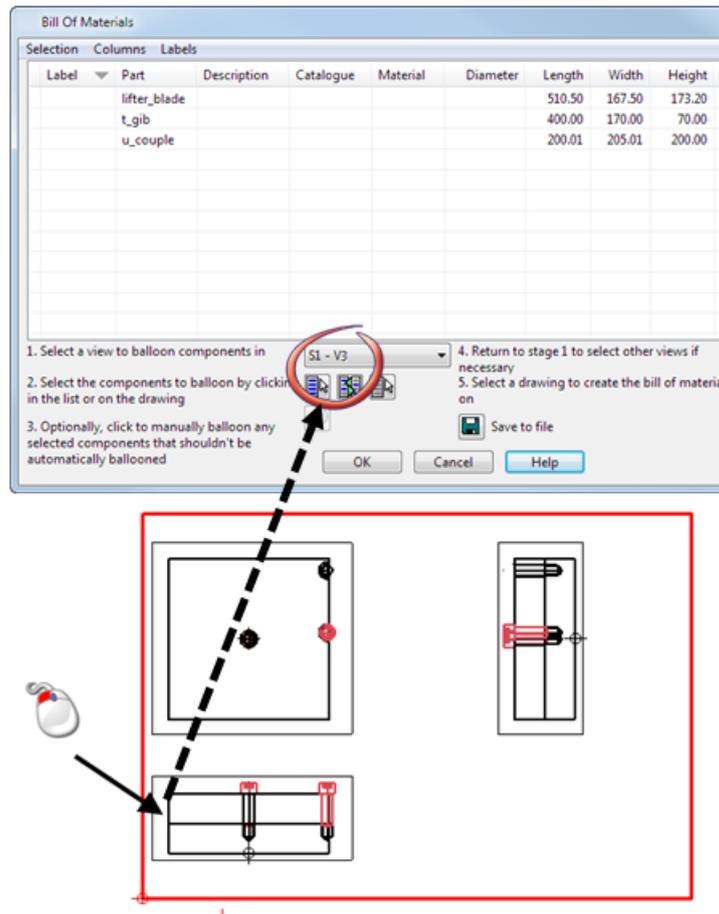


Follow the instructions and select appropriate options:

- 1 Select a view in the drawing where you want to apply component balloons. You can do this in one of the following ways:
 - Select a view from the drop-down list.



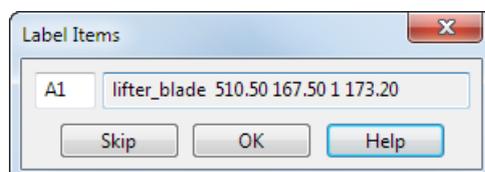
- Click a view on the drawing to choose where the balloon will be displayed. The value that is displayed in the drop-down list will be changed to match the view that you have chosen. You can then add a manual balloon to the drawing in the normal way.



- Click one of the following buttons to define the components to balloon.

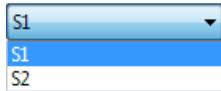
- Select all components in the list.
- Invert the list selection.
- Clear the selection in the list.

- Click to create balloons manually for selected items. The **Label Items** dialog is displayed. Use this dialog to change the name of the label series that is used when adding balloons manually.



Balloons are added to a component in a single view; the balloon will appear on the last view that you selected.

- 4 Select the drawing where you want to create the Bill of Materials.



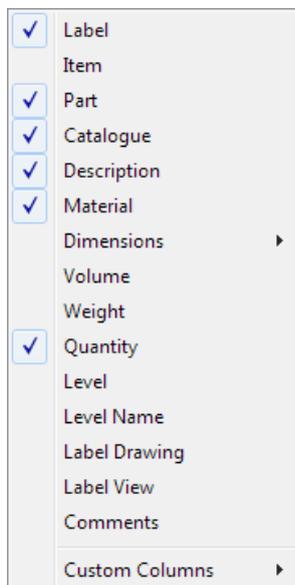
- 5 Click  or  to create the bill of materials.

- **Create a bill of materials on the drawing** .
- **Save the bill of materials to a file** .

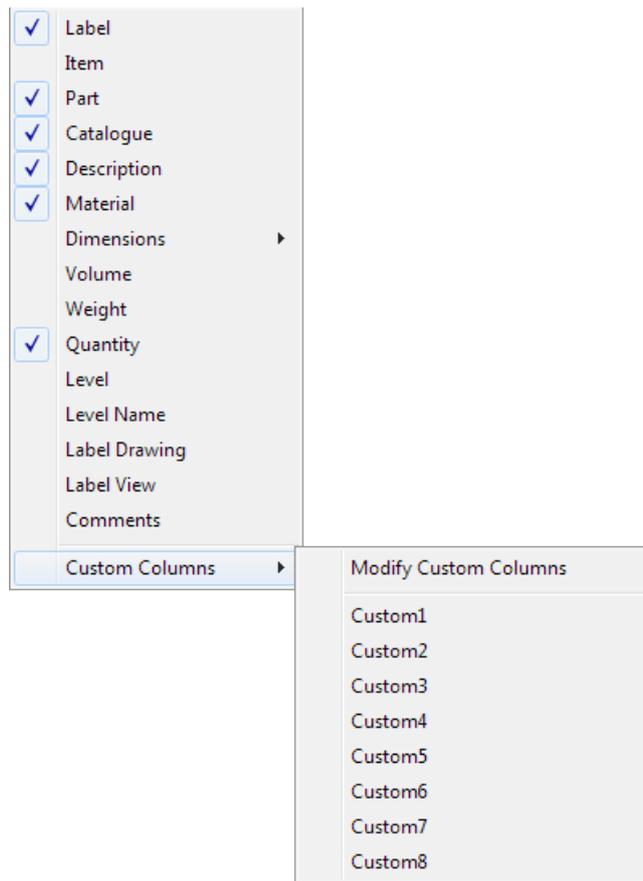
Creating customised columns of data - Bill of Materials

You can create up to 8 customised columns of data.

- 1 Select **Columns** menu on the dialog.



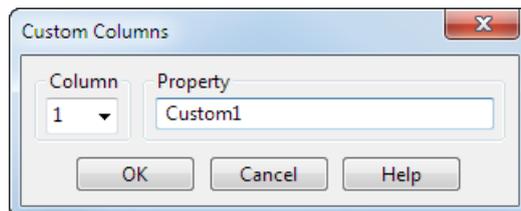
2 Select **Custom columns**



- 3 Select **Modify Custom Columns** to display the Custom Columns dialog. (see page 313)
- 4 Use the dialog to create a customised column.
- 5 Add data to the custom column (see page 314)

Custom Columns dialog

Use this dialog to create customised columns of data in the dialog.



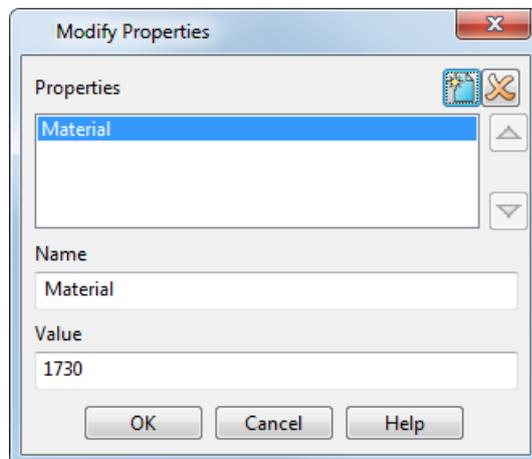
Column — Select a column from 1 - 8.

Property — Change the default name to the name of the property that contains the required data. The property name entered is also displayed as a column title. If a user defined property matches a standard column name, the value should be appended to that column. So if you have a component called *bolt* that has a user defined property called *Part*, with a value *test*, the column will be called *bolt test*.

OK — This creates the new column title and removes the dialog. The new column title is displayed in the list of custom columns that can be selected to be displayed in the dialog.

Adding data to a new Custom column

- 1 Return to the active assembly, and enter the **Assembly** application mode.
- 2 In the assembly tree, right-click on a component.
- 3 Select **Properties** from the **Modify** flyout to display the **Modify Properties** dialog.



- 4 Select **Create a New property** .
- 5 In the **Name** text box, enter the name of the newly created custom column.



Name is case-sensitive, so "Width_1" is not the same as "width_1"

- 6 In **Value**, enter the customised data.
- 7 Click **OK**

When you next create a **Bill of Materials** this information will be available to be included in the column in the **Bill of Materials** dialog.

Both component and component definition properties are displayed when the **Modify Properties** dialog is displayed for a component.

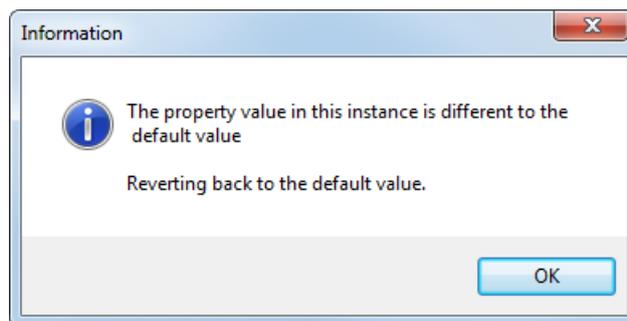
Additional information on Modify Properties dialog

Component definition properties are inherited by all instances of the component definition. Individual component instances can, optionally, overwrite the default property values set by the component definition.

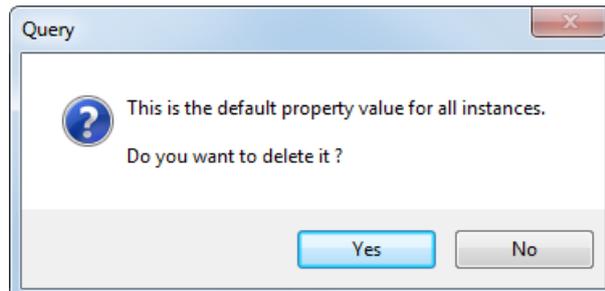
Information or **Query** dialogs are displayed if you attempt to use the **Modify Properties** dialog to delete or rename a property when this action could have unexpected consequences.

The relevant query dialog is displayed if you try to do one of the following:

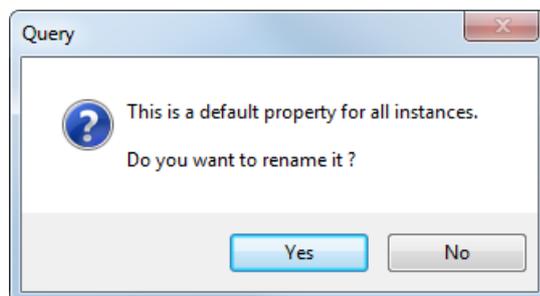
- Delete a property defined in the component and component definition.



- Delete a property defined in the component definition only:



- Rename a property defined in the component definition.



Labelling components in views

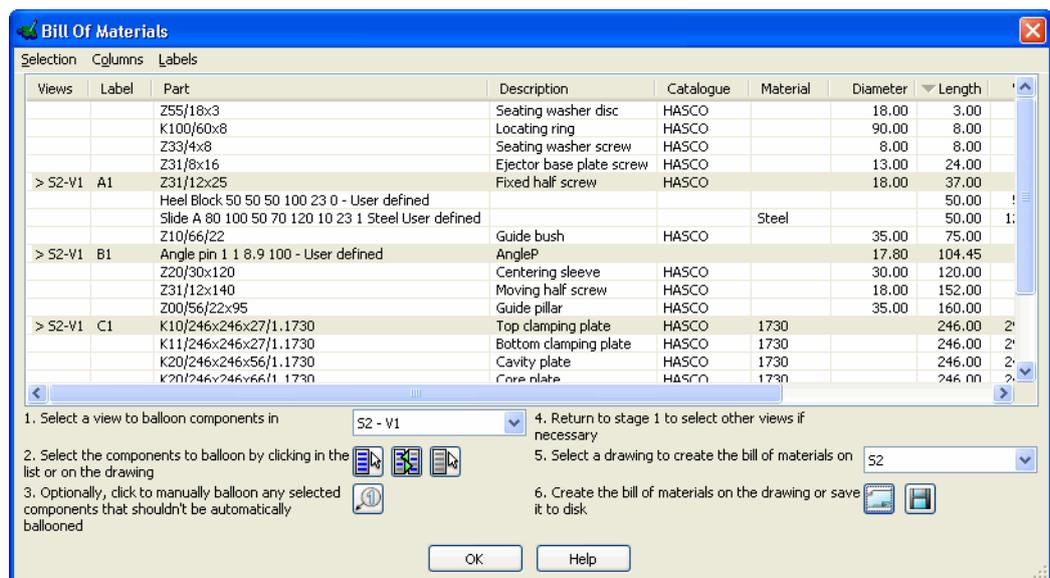
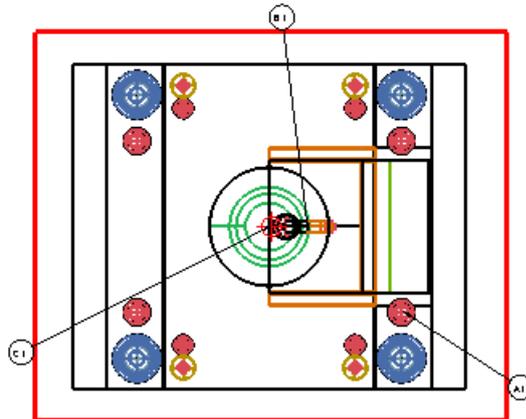
If you label all the components in one view, it may become cluttered. It can be useful to label different components in different views for clarity.

- 1 In the **Bill of Materials** dialog, select all the rows of components that you want to label
- 2 Select **Labels** on the **Bill of Materials** dialog:



- 3 Select the type of label you want to use
- 4 Click one of the following buttons to create the **Bill of Materials**: with the labels displayed:
 - **Create on Drawing** - Click on the drawing in the position you want the Bill of Materials to be displayed. The Bill of Materials is created as an OLE file that is embedded in the psmode. If a Bill of Materials is already created in a drawing this option creates an additional Bill of Materials. The existing Bill of Materials can be deleted to display only the newer version.
 - **Save To File** - This option creates the **Bill of Materials** as a Microsoft Excel spreadsheet (.xls file). The **Save As** dialog is displayed where you specify a filename and browse to the drive and folder where you want to store the Bill of Materials.
- 5 Labels are generated and displayed in the **Bill of Materials** dialog and the balloons are displayed in the chosen view.

- 6 The following example shows a labelled view of an assembly, and the labels displayed in the **Bill of Materials** dialog.



- 7 Click on a labelled component in the **Bill of Materials** dialog to see it highlight in the view. You can also click on a component in a view to see it highlight in the **Bill of Materials** dialog.



If you wish to have just one of each item displayed or labelled in the Bill of Materials dialog, sort the data by Quantity. You will notice that some rows of components do not have a quantity entered. This is because the first instance of the component displays the total quantity. If you do not wish to display these rows, you can blank them using Blank Selected from the View menu.



If data is sorted when you label the components, any identical data will have the same label series. If you wish to override this you need to relabel the components.

Labelling individual components in views

- 1 In the **Bill of Materials** dialog, click on the individual components you want to label. Hold down the **CTRL** key to select more than one component.



- 2 Click on the **Label Items** dialog (see page 306).. The **Label Items** dialog is displayed showing a description of the first item selected to be labelled.



- 3 The corresponding component is highlighted on the assembly drawing in the selected view.
- 4 Click on the highlighted component. The cursor displays a label line.
- 5 Click on the drawing to fix the balloon label. The description of the next item selected is displayed in the **Label Items** dialog.
- 6 Click in the drawing to create the label or click **Skip** if you do not want to create a label for the item displayed.

The next selected item is displayed.

The corresponding label for the component is displayed in the **Label** column in the **Bill of Materials** dialog. Any skipped items are still highlighted in the **Bill of Materials** dialog.

- 7 Click one of the following buttons to create the **Bill of Materials**: with the labels displayed:

Create on Drawing - Click on the drawing in the position you want the Bill of Materials to be displayed. The Bill of Materials is created as an OLE file that is embedded in the psmodel.



If a Bill of Materials is already created in a drawing this option creates an additional Bill of Materials. The existing Bill of Materials can be deleted to display only the newer version.

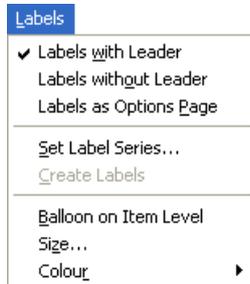
Save To File - This option creates the **Bill of Materials** as a Microsoft Excel spreadsheet (.xls file). The **Save As** dialog is displayed where you specify a filename and browse to the drive and folder where you want to store the Bill of Materials.

- 1 Labels are generated and displayed in the **Bill of Materials** dialog and the balloons are displayed in the chosen view.

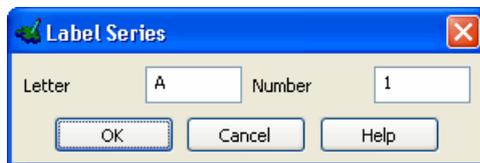
Modifying Labels

Labels are sequenced automatically in alphanumerical order, for example A1 - D8. Make changes to labelling in the **Bill of Materials** dialog as follows:

- 1 In the **Bill of Materials** dialog, select all the rows of components that you want to modify.
- 2 Select **Labels** on the **Bill of Materials** dialog:



- 3 Click **Set Label Series**. Use the **Label Series** dialog to specify the start number for the label series.



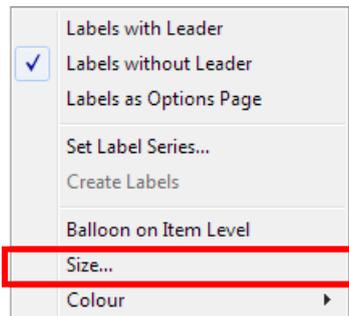
Letter - Enter the start letter for the label series. To display labels alphabetically, enter **A** and leave **Number** blank.

Number - Enter the start number for the label series. To display labels numerically, enter **1** and leave **Letter** blank.

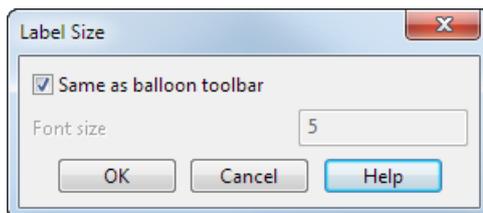
Label Series	Letter	Number
Alphabetical	A	Blank
Numerical	Blank	1
Hyphen	A-	1
Colon	A:	1

Changing the font size of the labels

You can define the font size of labels using the **Size** option on the **Labels** menu of the **Schedule** or **Bill of Materials** dialog.



Selecting this option displays the **Label Size** dialog.



Same as balloon toolbar — Select this option to use the label size that is defined on the **Balloon** toolbar. Deselect it to enter a new font size.

Font size — Enter the font size.

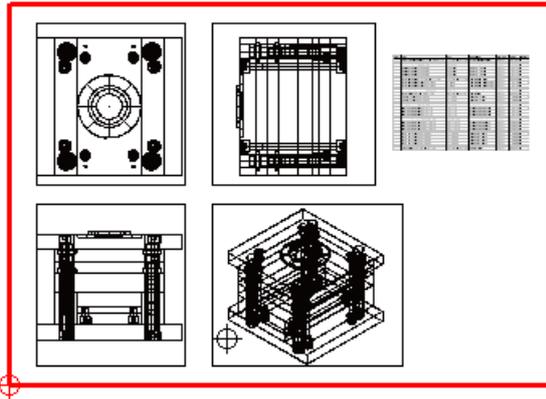


Font sizes for Schedule and Bill of Materials are independent, so you can change the font size for the Schedule and use a different size for the Bill of Materials.

Displaying the Bill of Materials as a spreadsheet

- 1 Manipulate the data in the **Bill of Materials** dialog so that the required data is displayed.
- 2 Click  on the **Bill of Materials** dialog
- 3 Click **OK**.

- 4 Click the cursor in the drawing at the position you want the spreadsheet to be displayed.



Modifying data in the embedded spreadsheet

- 1 Click on the spreadsheet in the drawing with the right mouse button.

The following context menu is displayed.



- 2 Click **Modify**.

You can also double click on the spreadsheet in the drawing to open it. The bill of materials data is displayed in a Microsoft Excel spreadsheet.

- 3 Edit the spreadsheet.

As you modify data in the spreadsheet, the changes are also displayed in the bill of materials table in the drawing.

Saving the Bill of Materials as a file

- 1 Manipulate the data in the **Bill of Materials** dialog so that the data you want to save as a file is displayed.
- 2 Click .
- 3 Click **OK** to display the **Save As** dialog.
- 4 Select **.xls** or **.csv** from the **Save as type** drop-down list.
- 5 Click **Save**.



You can embed the saved .xls file into drawings as an OLE view.

Modifying data saved as a file.

As the file was saved automatically as a .xls spreadsheet, the file can be opened and modified using Microsoft Excel.

- 1 Right click on the Bill of Materials.
- 2 Select Modify. The Bill of Materials is displayed as an MS Excel spreadsheet.
- 3 Make the changes to the spreadsheet and click  on the spreadsheet. The embedded Bill of Materials will be updated.

Personalising the BOM spreadsheet

To include personalised information on a Bill of Materials spreadsheet, (for example, a company logo), you must first set up a spreadsheet (.xls) template as a user-specific file.

- 1 Create a spreadsheet (.xls) called **bom.xls** saved to your user-specific folder.
- 2 The location of the user-specific files (for example custom toolbars and shortcuts) is structured so that these files 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\` (Vista installation)
 - the location that is indicated by your **HOME** variable.
- 3 In the middle of your **bom.xls** spreadsheet, create a single cell that contains the text **Insert data here**.



This text must not be translated.

- 4 Add personalised details above or below the single text cell, in the positions you want the details to be displayed on the spreadsheet.

When you create a Bill of Materials spreadsheet by saving to file, the **bom.xls** spreadsheet is copied into the Bill of Materials and data from the **Bill of Materials dialog** is inserted at the location of this single cell.