
PowerSHAPE 2015 R2

Reference Help

Assembly modelling



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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Assembly modelling

Use the following sections to find information on assembly modelling:

Introduction to Assembly modelling (see page 3)

Component definitions (see page 22)

Power Features (see page 46)

Assemblies and Sub-assemblies (see page 79)

Components (see page 85)

Parameterised Components (see page 117)

Exploding assemblies (see page 123)

Schedules (see page 124)

Adding components using the Component Wizard (see page 126)

Assembly options (see page 138)

Introduction to Assembly modelling

Use the following sections to find information on assembly modelling:

- What is Assembly modelling? (see page 4)
- What is a component definition? (see page 4)
- What is a component? (see page 5)
- What is an assembly? (see page 5)
- What is an attachment? (see page 5)
- What is a relation? (see page 6)
- What is a sub-assembly? (see page 6)
- What is a group? (see page 6)
- What is a sub-item? (see page 6)

- Starting Assembly modelling (see page 7)
- Assembly tree (see page 9)
- Assembly modelling menus (see page 17)
- How do I find information about assembly objects? (see page 21)

What is Assembly modelling?

Assembly modelling allows you to generate complex assemblies of standard components, quickly and easily.

Assemblies are built from components. You can register solids as component definitions and then create many instances of the components in the assembly.

For example, a car can be an assembly consisting of four wheel components and one body component. Each wheel component is created from the same wheel component definition. The body component is created from the body component definition.

Generally, an assembly is:

- a set of components
- a set of relationships, defining how components fit together

Assembly modelling includes various tools that help a user to analyse the assembly that is created.

What is a component definition?

PowerSHAPE comes with several pre-defined graphical object types (such as lines, arcs, curves, surfaces and solids) that you can create to represent real objects or features of real objects.

A *component definition* is a user-defined graphical object that you can use to represent complete engineering components.

A component definition is only a mechanism for storing the geometry and not an actual instance of an object. It is created automatically when you create a component from a selected solid and stored invisibly. Once a component definition is created, you can use it to create components and build the assembly.

For example, you can create a component called *wheel*. When you create this component, a corresponding component definition will be created to contain the geometry of the wheel. To create a car assembly, you will also need a *body* component that contains the geometry of the car body. For further details, see What is an assembly? (see page 5)

What is a component?

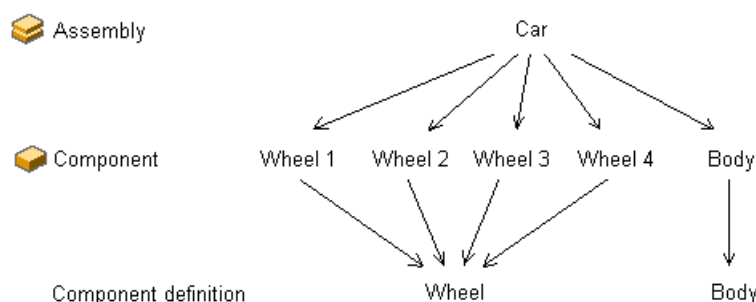
To create a visible object from component definition within assembly modelling, you must create instances of the component definitions in particular positions and orientations in your model. These instances are simply called *components*.

For our car assembly, we can create four identical wheels components from the one component definition of the wheel. The body component is created from the component definition of the body.

What is an assembly?

An assembly is a collection of components which fit together.

In the car assembly, you can create four wheel components and a body component. The assembly would also store information on how these fit together to create the car.



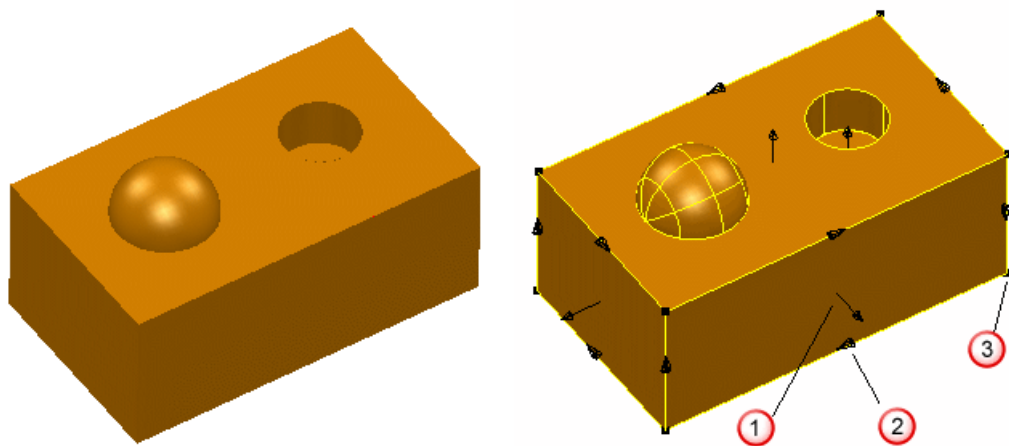
What is an attachment?

Components can be attached to other components in an assembly using *attachments*. These are planar faces, axis, straight edges or points on components where another component can be attached.

When you register a component, all the possible attachments are found automatically.

In the example below, the attachments will be generated as follows:

- ① Plane
- ② Line
- ③ Point



What is a relation?

A **relation** is a relationship between an attachment on one component and an attachment on a second component. For example, a planar attachment can be co-planar with another planar attachment, or it can be parallel by a given distance.

- Relationships that define the position of the component itself (because it is attached to another component) are called *Slave* relationships.
- Relationships that define the position of other components (because they are attached to a master component) are called *Master* relationships.

The master relationship of one component is always matched by the slave relationship on the second (related) component. The relationship will be displayed twice:

- In the **Master Relations** branch of the master component
- In the **Slave Relations** branch of the slave component

What is a sub-assembly?

A **sub-assembly** is an existing assembly that forms part of another assembly.

What is a group?

A **group** is a collection of components in an assembly, but does not define an assembly hierarchy.

What is a sub-item?

For a component, the sub-item can be the main solid, power feature solid or the workplane of the component.

For a sub-assembly, the sub-items are the components inside the sub-assembly.


For a Group, the components are the sub-items.

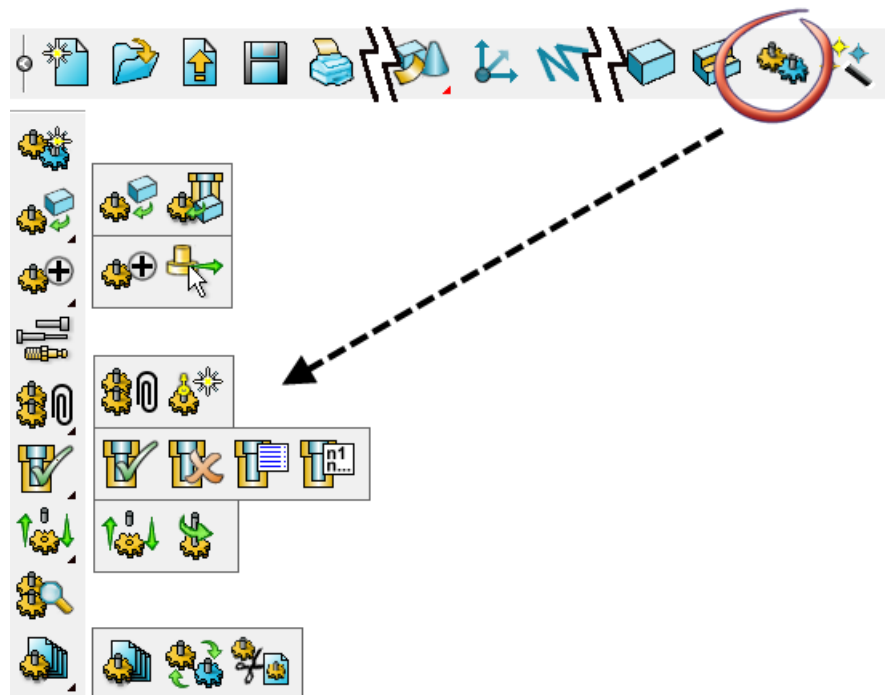
Starting Assembly modelling








You can start Assembly modelling from:










- the Main toolbar (see page 7).
- the menu (see page 8).

Using Assembly modelling from the toolbar

- 1 Ensure that you have a model displayed.
- 2 Select **Assembly**  from the main toolbar. The Assembly modelling toolbar will be displayed.



-  - Create new assembly
-  - Add component from selected solids
-  - Add component using the Intelligent Cursor
-  - Add components with advanced options
-  - Create component from selected solids with Power Features.
-  - Component wizard
-  - Add relationship using attachment

-  - Manage attachments
-  - Generate Power Features
-  - Remove Power Features
-  - Show Power Feature Summary
-  - Show advanced Power Feature options
-  - Explode assembly using distance
-  - Explode assembly using angles
-  - Check components for intersection
-  - Manage source models of imported components
-  - Refresh imported components
-  - Convert all imported components to local

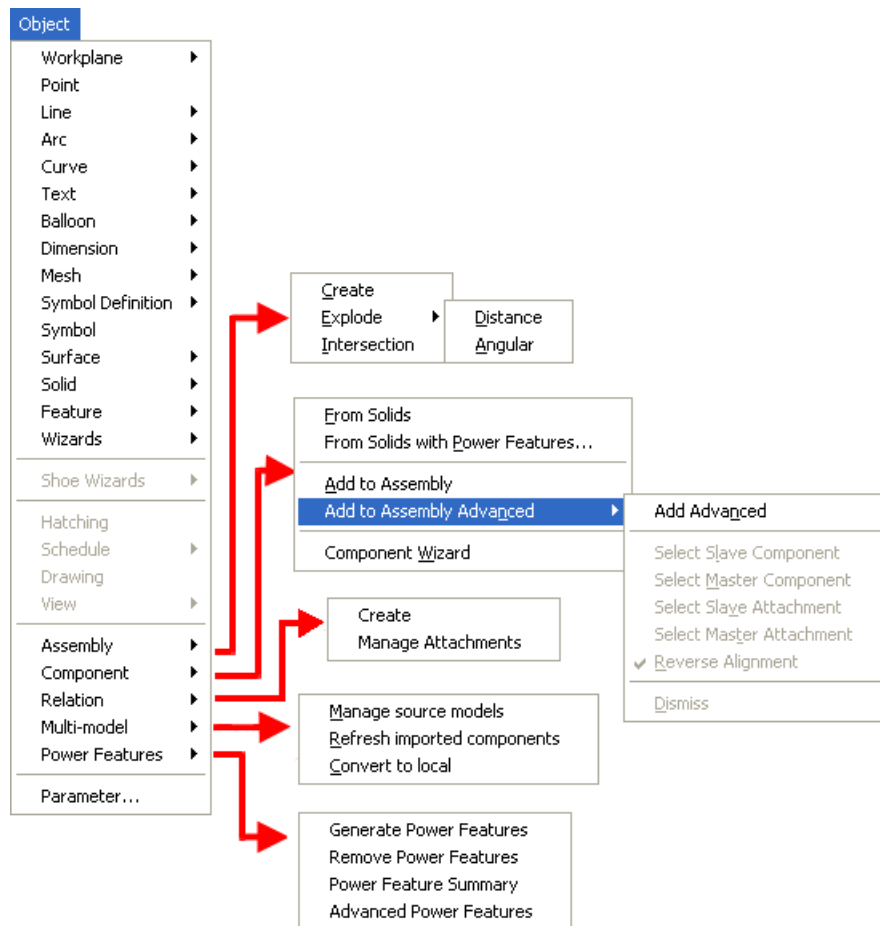
When you enter Assembly modelling, any assemblies that already exist in the model are displayed in the assembly tree and all the buttons on the Assembly toolbar are active.

- 3 Select the option you require from the toolbar.

Using Assembly modelling from the menu

- 1 Ensure that you have a model displayed.

2 Select **Object** from the main menu



3 Select the option you require from the sub-menus.

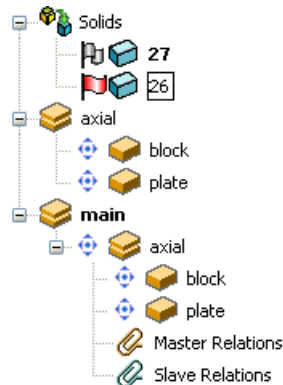
How do I exit Assembly modelling?


From the Main toolbar, select another option. The selected toolbar is displayed.


Assembly tree

The assembly tree displays all the assemblies and solids in the model.

Each assembly contains all its components and the relationships between the components. In the example, there are two assemblies and two solids (**26** and **27**) that are not part of the assembly.



 **Expand** — Click this button to expand the branches in the tree.

 **Contract** — Click this button to contract the branches in the tree.

Solids in the assembly tree

The solid and assembly trees are amalgamated to provide an integrated product. Assembly and modelling functionality are available together in the tree window.



A single solid that is not part of the assembly is displayed at the top of the tree.



If there are several solids that are not part of the assembly, they are displayed in the **Solids** branch at the top of the assembly tree.



Multiple selection in the assembly tree

You can select multiple items in the assembly tree by using **Ctrl**+click or **Shift**+click. You can select multiple items of any of the following:

- Assemblies
- Components
- Grouped Sub-assemblies
- Power Features

Icons used in the assembly tree

There are several basic icons that are used in the assembly tree:

- Solids
- Component
- Assembly and sub-assembly
- Group

Associated parameters and relationships icons are added to the basic icons (see page 13).



An assembly is not shown in the assembly tree if it is included in another assembly, unless it is the active assembly.

Basic assembly tree icons



Component name — If you select a component with the right mouse button, the following menu is displayed.

There are four different states associated with components:



Component is a source of Power Features. It may apply these features into target components.



Component is a target for Power Features. Power Features from source components have been applied into it. The Power Features are all up to date.



Component is a target for Power Features. Power Features from source components have been applied into it. Some Power Features are out of date.



The Power Feature is out of date.





Assembly name — This contains all the components and sub-assemblies in the assembly. Right-click an assembly to display a menu with appropriate options active.


main	
Delete	
Rename	
Selection Information	
Activate	
Deactivate	
Add to Assembly	
Add to Assembly Advanced	
Modify Parameters	
Modify Properties	
Convert to local	
Power Features	▶
Category Name	


If you right-click a sub-assembly, the following menu is displayed with appropriate options active.



axial (axial)	
Delete	
Rename	
Blank	
Blank Except	
Select	
Selection Information	
Activate	
Add to Assembly	
Add to Assembly Advanced	
Modify	▶
All Instances	▶
Highlight	▶
Convert	▶
Power Features	▶
Group	▶
Manage Attachments	
Category Name	
Use Style from Sub-items	
Use Material from Sub-items	
Add to Library	▶


 **Group name** — If Power Features applied in the group are out of date,  is added to icon. If you select the group with the right mouse button, the following context menu is displayed.


axial
Delete
Rename
Selection Information
Convert into Sub-assembly
Ungroup Group
Use Style from Sub-items
✓ Use Own Style
Use Material from Sub-items
✓ Use Own Material

 **Master** — This branch lists the relations in which the current component is master. Master components are highlighted in red (default colour scheme).

 **Slave** — This branch lists the relations in which the current component is slave. Slave components are highlighted in green (default colour scheme).






The details of the components in the  **Master** and  **Slave** branches are similar to that shown in the following example:













 **Coincident[0]** *bolt*- This is a **Line to Line** relation of the current component with the component called *bolt* using the **Coincident** additional relation type.

 **Relationship** — When you select a relationship in the tree, its attachments are highlighted in the assembly in the graphics window.

Summary table of assembly tree icons

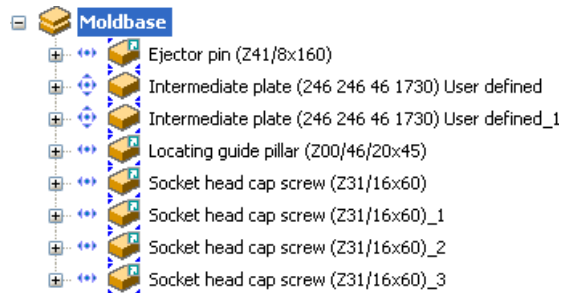
The following table shows the different Assembly modelling icons and the associated parameter and relations icons.

Components	
	Power Features folder.
	Component has up to date associated Power Feature(s). It is a target component.
	Component contains Power Feature(s). It is a source component.
	An associated Power Feature is out of date.
	Component.

	Imported component
	Power Feature is out of date
Assembly and Sub-Assembly	
	Assembly.
	Assembly from another model. Use the Refresh imported component definitions to update the component definitions.
	Sub-assembly.
	Assembly contains Power Features. It is a source component.
	Imported assembly
	Group
Parameters and relationships	
	A parameterised assembly, component or component definition, depending on the associated icon.
	The component has no relationships.
	The relationships of the component are under defined.
	There is a contradiction in relations.


Sorting the assembly tree

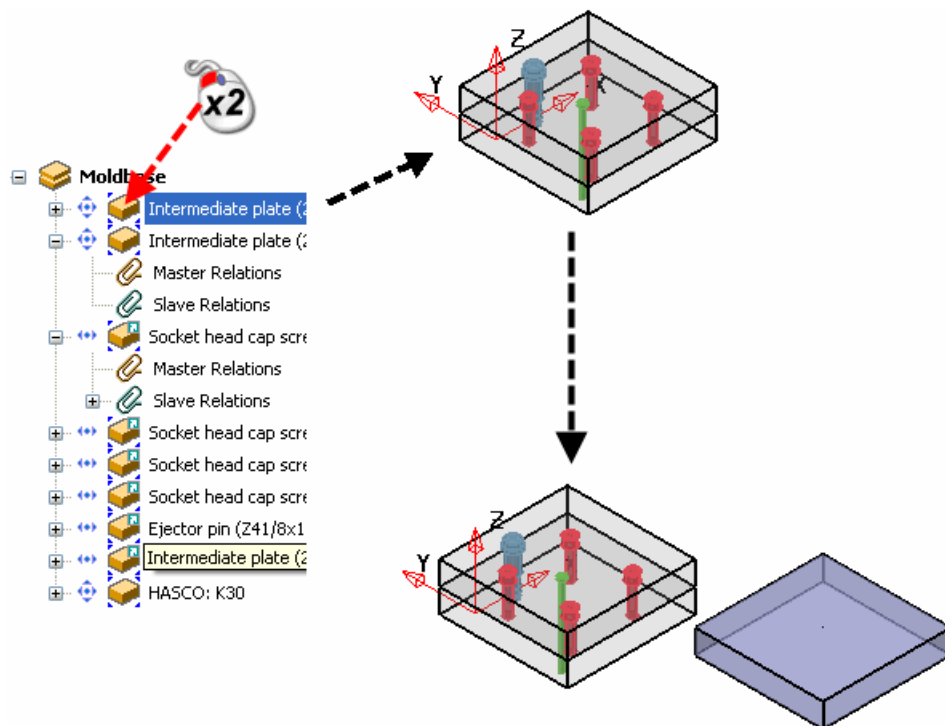
You can sort the items on the assembly tree into alphabetical order. Click the right mouse button in a blank area of the tree and select **Sort alphabetically**.



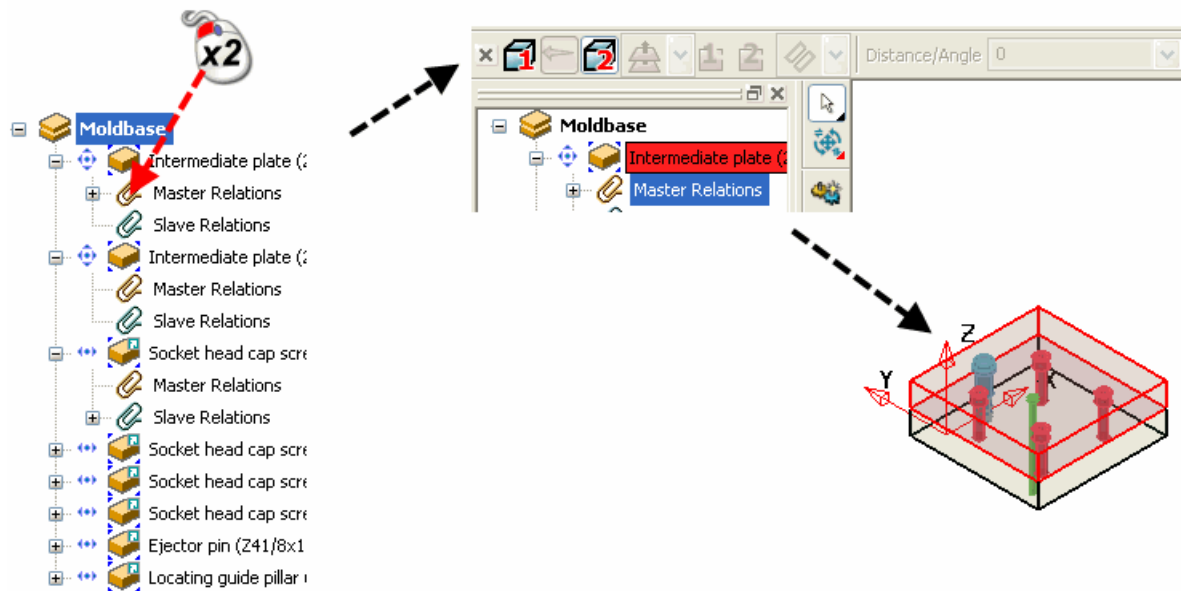
Assembly tree shortcuts

Use the following Assembly tree shortcuts to quickly add a component in free space and modify a relationship.

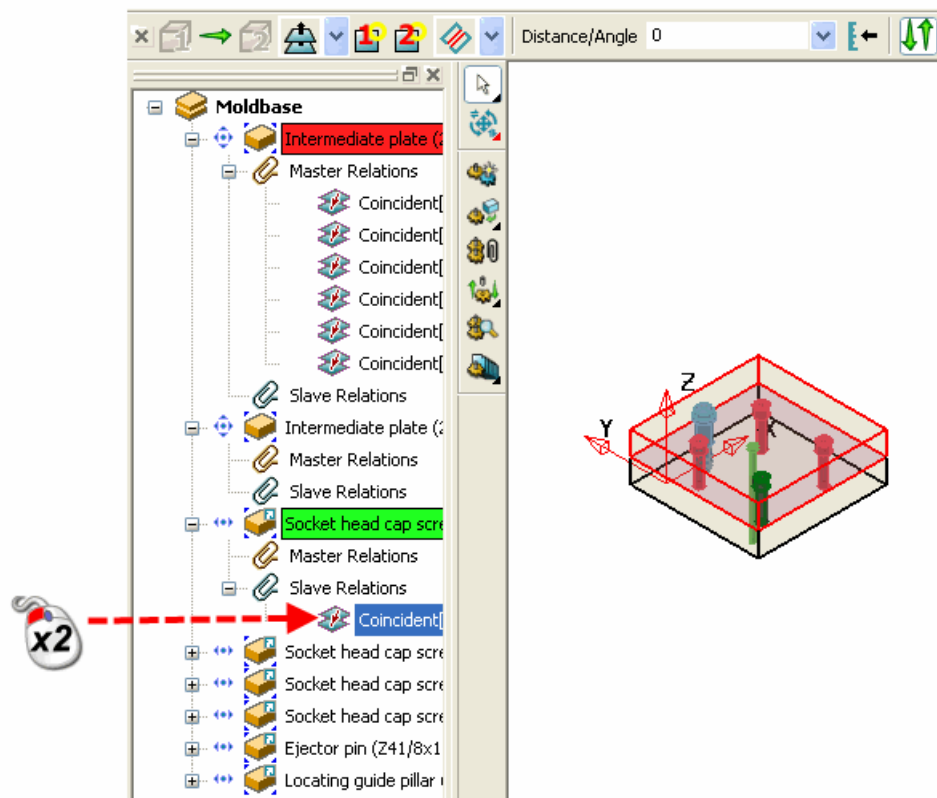
- Double-click on the component icon. The cursor changes to  and the component graphics are displayed on the edge of the model window nearest to the cursor.



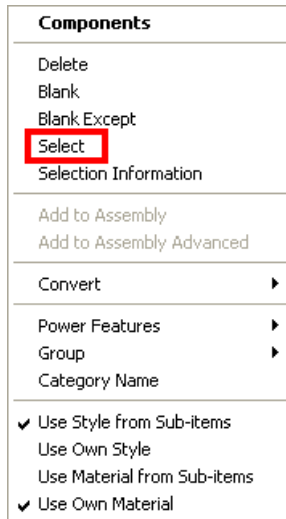
- Double-click on the relation icon to start modifying the relationship.



- Double-click on the relation branch icon of a component to start creating a relationship with the master and slave components selected.

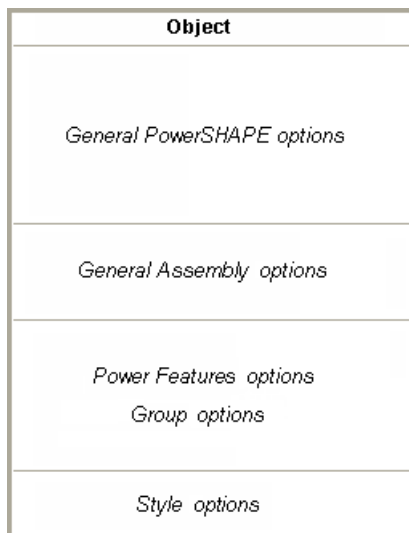


- You can use **Ctrl**+click to select multiple components in the Assembly tree. Right-click and **Select** is available on the menu. If you choose this option, the components you selected are highlighted in the model.



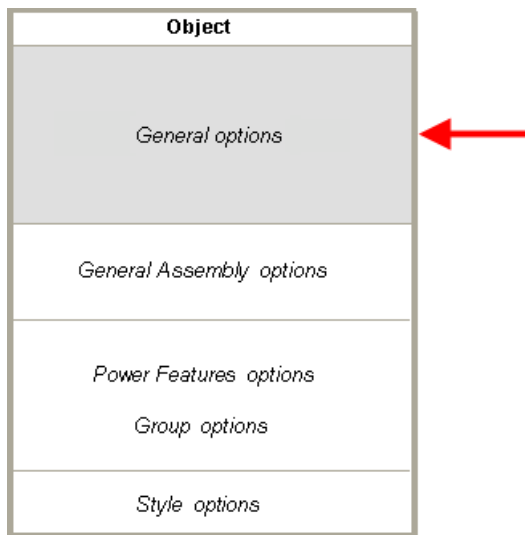
Assembly modelling menus

Related options are grouped together and are displayed in corresponding places on context menus, as follows:



The exact composition of each menu depends on the object that is selected.

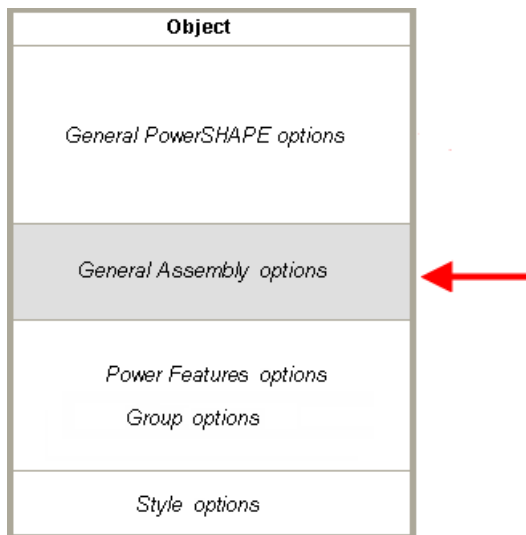
context menu: General options



Combinations of the following options are displayed in the General options section of the assembly menus:

- Cut
- Copy
- Paste
- Paste Special
- Delete
- Next selection
- Clear Selection
- Select All
- Blank
- Blank Except
- Undo
- Redo
- Selection
- Selection Information

context menus: General Assembly options

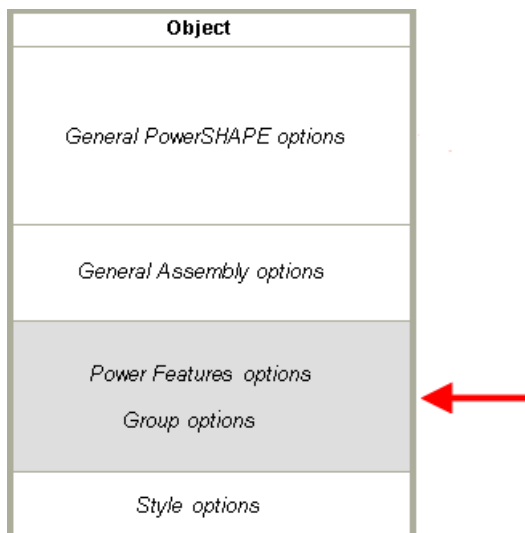


Combinations of the following options are displayed in the General Assembly options section of the assembly menus:

- Activate
- Create Component
 - Add to Assembly
 - Add to Assembly Advanced
- Modify
 - Position
 - Geometry
 - Parameters
 - Properties
- Modify Parameters
- Modify Properties
- All Instances
 - Modify Geometry
 - Modify Parameters
 - Modify Properties
- Highlight
 - All
 - Similar
- Convert
 - To Geometry
 - To Sub-assembly

- To Local
- To Components
- To Group
- Convert to Local
- Convert to Sub-assembly
- Reassign

Power Features options and Group options

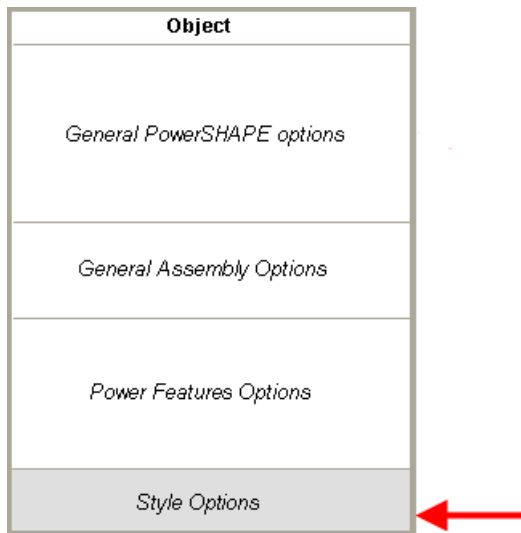


Combinations of the following options are displayed in the Power Features options section of the assembly menus:

- Power Features
 - Generate
 - Remove
 - Preserve Source
 - Preserve Target
 - Target Flag
- Group
 - Add
 - Remove
- Rename Attachment
- Drag and Drop mode
- Category Name
- Attribute
 - Power Feature Target

- Sectioned

Style options



Combinations of the following options are displayed in the Style options section of the assembly menus:

- Use Style from Sub-Items
- Use material from Sub-Items
- Use Own Style
- Use Own Material
- Thumbnail size
 - Small
 - Medium
 - Large
- Thumbnail Type
 - Wire
 - Shaded
- Transparent Background
- Update Thumbnails
- Close Parent Library

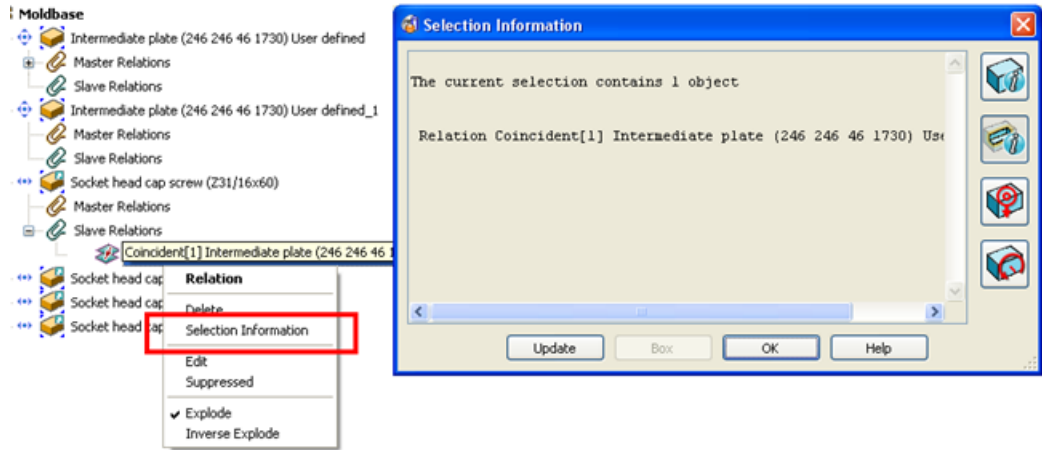
How do I find information about assembly objects?

You can find information for the assembly objects by selecting **Selection Information** on the menu of the selected item.

- 1 Right-click an assembly object in the assembly tree.

- 2 Choose **Selection Information** from the menu to display the Selection Information dialog.

The dialog contains extra information for the assembly object as well as the usual selection information, including the dimensions of the bounding box for the selected components.



Component definitions

- Overview of component definitions (see page 22)
- Working with multiple models (see page 23)

Overview of component definitions

A component definition is automatically registered when you:

- select **Create components from selected solids** from the Assembly toolbar.
- add a component by selecting **Add to Assembly** or **Add to Assembly Advanced** from the solid menu in either the assembly tree or the main graphics window.
- Select **From Selected Solids** from the **Component** options of the **Object** menu.

A component definition contains the geometry and attachments of the component. Attachments are automatically generated from the geometry and stored with the component definition. The **Auto attachment factor** is defined on the General page of the Assembly Options (see page 139). This factor is applied to the bounding box of the geometry of the component to create a new bounding box. Any attachments that lie within this bounding box are automatically included in the component definition.



Component definitions act like templates and are used to create instances of a component. Nearly all assembly operations are performed on component instances (components). Usually, component definitions are not visible to you, but there are two areas where you will use component definitions:


- Component Libraries (see page 31)
- Multi-model assemblies (see page 23)


Working with multiple models


This functionality lets you work with components that originate in different models. **Multi-model** groups together the functionality that handles working with components that originate in another model.

To use the Multi-model options:

- 1 Click  to display the Assembly modelling toolbar.
- 2 Click  (Assembly modelling toolbar) to display the **Multi-model** flyout.
- 3 Select one of the options from the flyout.

 **Manage source models** — Change the source model for an imported component definition. For further details, see Managing source models (see page 23).

 **Refresh imported components** — Updates the component definition to reflect any changes made to the component definition stored in the Component Library. For further details, see Manually refreshing imported component definitions (see page 26).

 **Convert to local** — Convert all imported components to local, breaking the link to the original model. For further details, see Converting an imported component definition (see page 31).

These options are also available by selecting **Multi-Model** from the **Object** menu.

Managing source models

You can change the source model for an imported component definition using the **Manage source of imported component definitions** dialog. This means that you can choose a different source model in a different location. This is particularly useful if the original source model has been renamed, deleted or moved to another location.

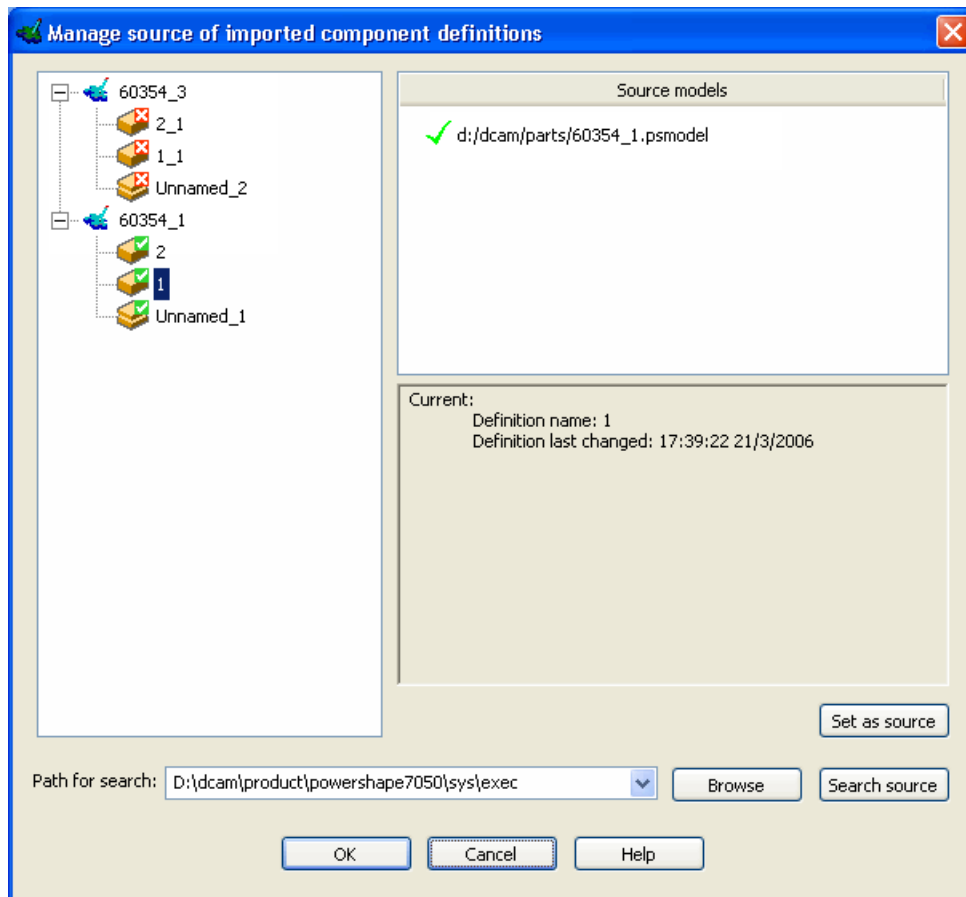
- 1 Select **Manage source models of imported components** from the **Multi-model** flyout.




- 2 This displays the **Manage source of imported component definitions** dialog. This is the equivalent of selecting **Manage source models** from the **Multi-model** option of the **Object** menu. If the model has no imported component definitions then the dialog will not be displayed.

Managing source models dialog

Use this dialog to change the source model for an imported component definition.



Source models — The tree shows the list of the source models, with their associated imported component definitions. The icons on the left represent the imported definition and those on the right represents the definition in the source model. If the link between these is broken (for instance if the component definition in the source model has been renamed or deleted), the icon is changed to indicate the problem .

Current: This displays information about

- the component definition of the item that is selected in the assembly tree
- the model selected in the **Source models** section.

Path for search — Select a path from the drop down list or enter a path in the box.

Browse — Displays the **Choose path for source models** dialog. The path that you choose in this dialog will be transferred to the **Path for search** drop down list when you click **Select**.

Search source — Starts search for source models, using the path that you have selected in **Path for search**. Any source models that are found are added to the list in the **Source models** window.

Set as source — From the list displayed in the **Source models** window, select the appropriate source model for the definition and click **Set as Source**.

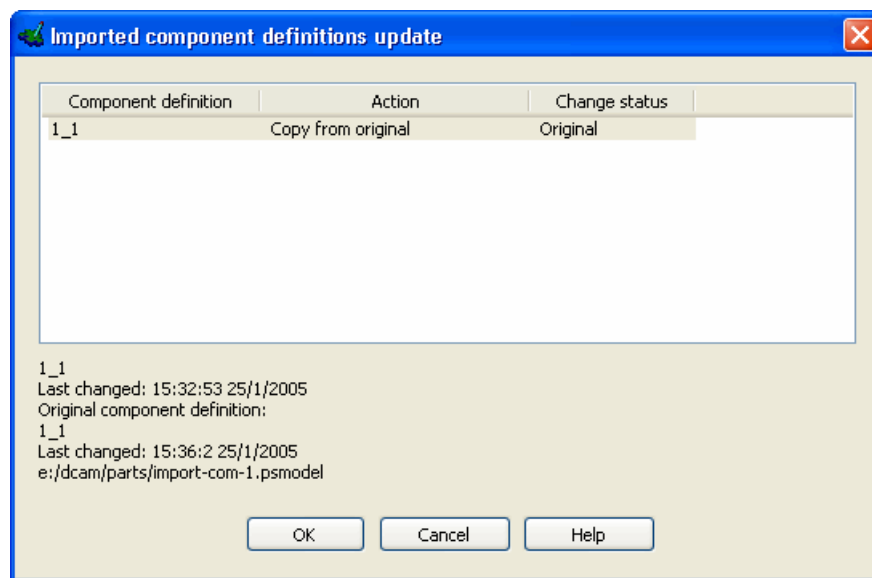
OK — The selected source model is linked to the corresponding definitions.

Imported component definitions

You can synchronise an imported component definition used in a model with the version of the component definition stored in the Component Library.



Automatic update of imported component definitions

If you change a component definition stored in the Component Library and have not manually refreshed the imported definitions, opening or saving a model containing a component based on the original component definition, will display the Imported component definitions update dialog (see page 27).



Manually refreshing imported component definitions

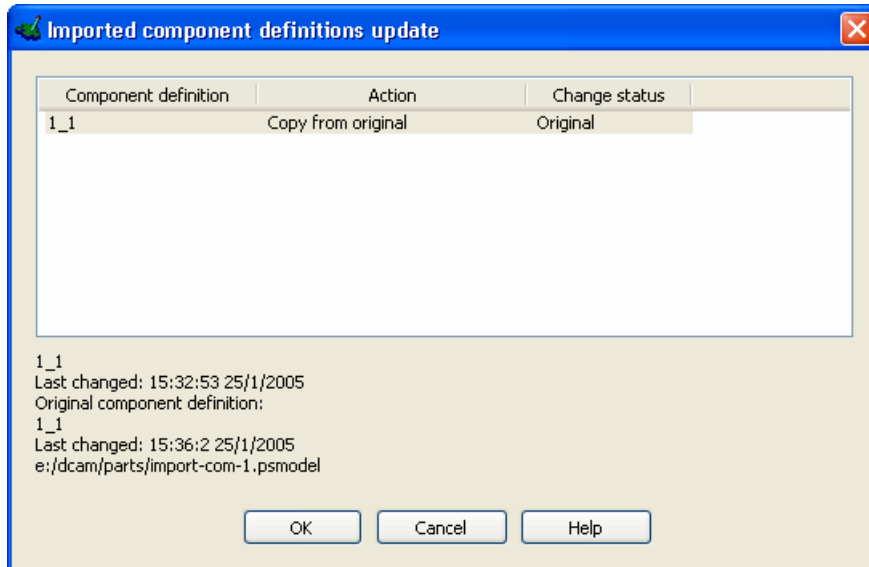
When you change a component definition held in the Component Library, you can manually refresh imported definitions. Use one of the following methods to display the Imported component definitions update dialog (see page 27):

- Select **Refresh imported component definitions**  on the **Multi-model**  flyout.
- Select **Refresh imported definitions** from the **Multi-model** option of the **Object** menu.

Refreshing the imported definition checks that:

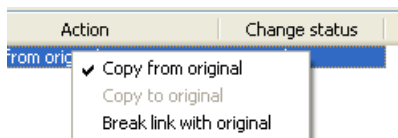
- the component definition exists.
- the component definition used in the model is the same version as the component definition in the Component Library.

Imported component definitions update dialog



Component definition — The name of imported component definition that needs to be synchronised. There will be an entry for each component definition that has been updated.

Action — Select one of the following options to define how the component definitions are synchronised.



- **Copy from original** — Replace the imported component definition with its original.
- **Copy to original** — Replace the original component definition with the imported one.
- **Break link with original** — Convert the imported component definition to local. From this point, the component based on the imported component definition will be unaffected by changes to the component definition in the Component Library.

Change status — The status of the original and imported component definitions relative to each other. This will be one of the following:

- **Original** — The original component definition has been changed.
- **Imported** — The imported definition has been changed.
- **Both** — The original and imported definitions have both been changed.


The following information about the selection is displayed below the list of component definitions:

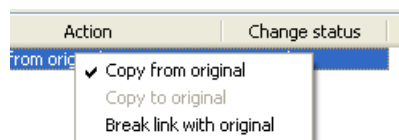
Name of current component definition
Time of the last changed to this definition
Name of original component definition
Time of last change to the original definition
Path to the model that contains original definition.

OK — Component definition is synchronised.

How do I synchronise imported component definitions?

To see how synchronising imported definitions works using a component definition that is held in a Component Library, work through the following steps:

- 1 Open a new model.
- 2 Open a Component Library.
- 3 Using a component definition from the Component Library, create a component in the model.
- 4 Using the **Open** option of the **File** menu, open the library model of the component definition that you have imported into your model.
- 5 Modify the geometry of the component definition; for example, move the hole.
- 6 Save the library model.
- 7 On your working model select Refresh imported components  from the **Multi-model** flyout. The Imported component definitions update dialog (see page 27) is displayed.
- 8 Select the option from the context menu that is displayed when you right click the **Action** heading.



- 9 Click **OK**.

The working model is updated.

Checking imported component definitions

When opening an assembly model imported component definitions are checked for the following problems:

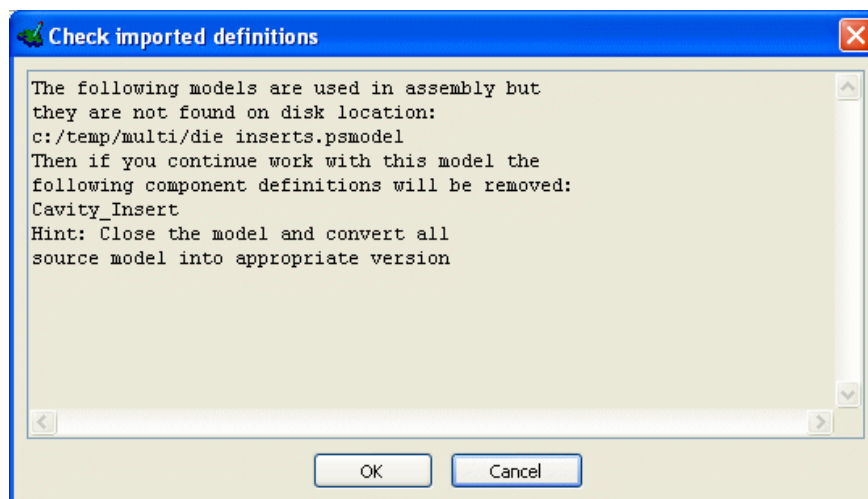
- Source model of the component definition has a version that is less than the current PowerSHAPE version. For further details, see Model has an earlier version number (see page 29)

- Source model of the component definition is not found at the expected disk location, or cannot be opened. For further details, see Component definitions are not at expected location (see page 30)

Model has an earlier version number

When you open a model that contains component definitions where the source model is a different version of PowerSHAPE, a warning box is displayed. This lists

- The models where the version number is not the current PowerSHAPE version.
- The models that are used in the assembly but cannot be found at the expected locations.

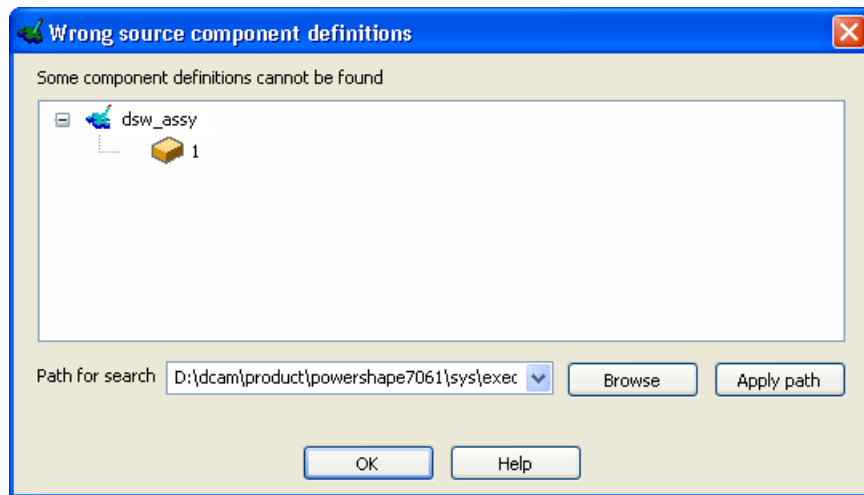


OK — Remove the component definition(s) from the current assembly.

Cancel — Close the model.

Component definitions are not at expected location

If the current model contains imported component definitions and the source model has been deleted or renamed, the **Wrong source component definitions** dialog is displayed.



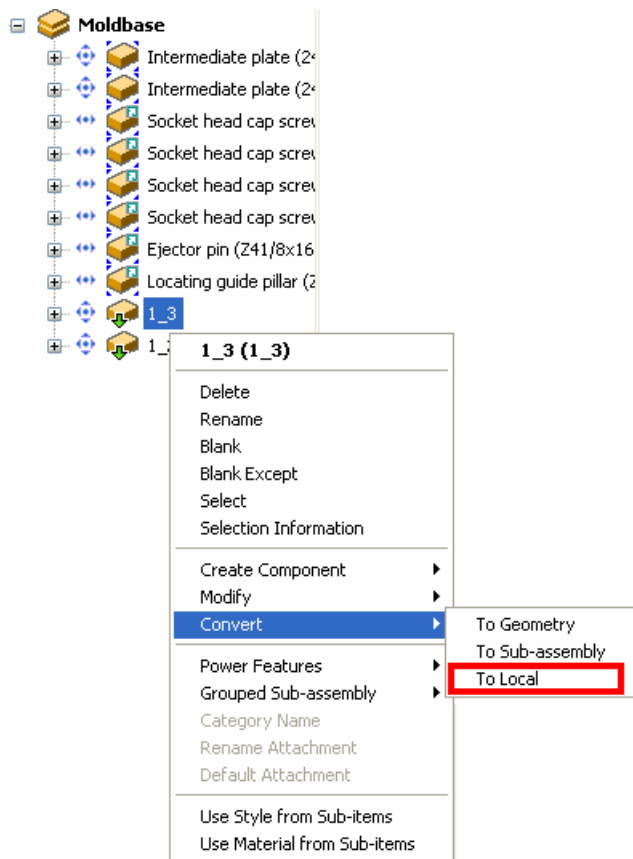
Path for search — The initial path that is searched for the model.

Browse — Use the **Browse for Folder** dialog to choose the path to the correct source model.

Apply path — The specified folder is scanned. If a required model or component definition is found in this folder, it is removed from the list in the dialog.

Converting an imported component definition

You can convert an imported component definition into a local component definition by selecting **Convert to local** from the context menu for the imported component.



In the example, the component 1_3 has been imported into the *moldbase* assembly.

Component Libraries

There are two types of Component Library:

- Component Libraries supplied with PowerSHAPE. These are component libraries containing the component definitions of generic, standard-sized components for cap screws, shoulder bolts and dowel pins.
- Component Libraries created by converting solids into components and saving the model as a component library to create a component library (see page 32). Those component definitions can be used in other assemblies as well.

There are two context menus associated with a Component Library.

- Single item context menu (see page 34) is displayed when you right-click on a library component.

- Component Library Window context menu (see page 39) is displayed when you right-click in free space in the component library window (not on a component).

Using a component library

If a component library exists, you can:

- select a Component Library (see page 32).
- display the Component Library window (see page 33).
- add components to an assembly using the Component Library (see page 41).
- add components to a Component Library (see page 42).
- copy a component to another library, category or component library window (see page 45).

Creating a Component Library

To create a component library:

- 1 Save your model, ensuring that the name is different from template names.
- 2 Select **Tools > Options** from the menu.
- 3 In the **Options** dialog, in **Assembly** options, select the **Component Libraries** page.
- 4 Select **This model is a component library**.
- 5 Save the model again.

Selecting a Component Library

- 1 Select **File > Open Component Library** from the menu.
- 2 Select the component library you want to use.

Component Libraries - A list of the Component Libraries.

Component Definitions Preview - This displays the component definitions held in the selected Component Library.

Name Filter - When the list of model names becomes very long, it can be difficult to find the model you want. Entering a filter reduces the list to only those models that match the filter pattern. For example, *MOD** will show all libraries beginning with *MOD*.

Clear - Clear the entry in **Name Filter**.

Password - If the model is protected by a password, you will need to enter the password in here before selecting Open.

Browse - Use this dialog to select and open a library *.psmodel* file.

3 Click **Open**.

Displaying the Component Library

To display the Component Library select **View > Windows > Component Library** from the menu.



Unless all the component definitions have a category name, the tab in the Component Library window is unnamed.

Each category name (see page 33) that has been used when creating the component definitions is displayed as a separate tab in the Component Library.

Thumbnail images are displayed for all the component definitions in the Component Library. The tooltip for each thumbnail displays information about the library component.

If there are more tabs than can be displayed in the window, you can scroll the tabs using the left and right mouse buttons:

- **Left click** — Left-click on a scroll arrow to scroll through the tabs by a small, fixed amount.
- **Right click** — Right-click on a scroll arrow to scroll through the tabs in larger steps. The size of the step depends on the width of the window and is approximately equal to the width of the tab area that is visible.

Holding either mouse button scrolls through the tabs until you reach the last tab in the direction you are scrolling.

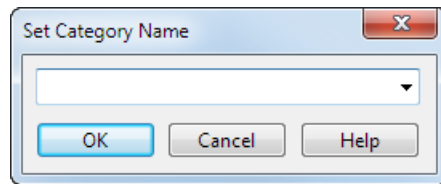
Creating a category for component definitions in a component library

Add a category name to a component definition to indicate the type of component definition. You can use the category name to filter the component definitions when adding a component.

To add a component definition into a category:

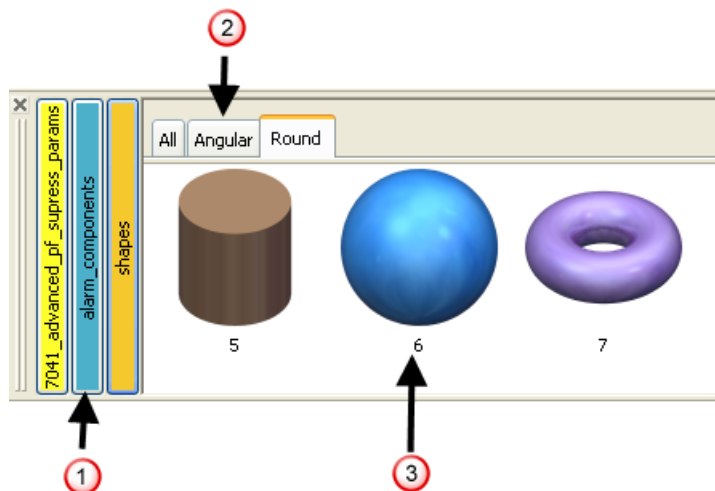
- 1 Right-click the component in the assembly tree and select **Set Category Name**.

The **Set category name** dialog is displayed.



- 2 Select a category name from the list of existing category names, or enter a new category name.
- 3 Click **OK**.

Layout of the Component Library Window



- 1 Open Component libraries
- 2 Categories
- 3 Components

The components in a component library are automatically arranged within the window so that only the vertical scroll bar is required.

The default height of the component library window is a single row. If the window is bigger than this, the height is saved when the program is closed.

Single item context menu

One of the following context menus is displayed when you use the right mouse button to select a component in the Component Library window. The popup that is displayed will depend if the selected component is in the current model.

- If the selected component is in the current model:

u_couple_1
Delete Instances
Rename
Information
Activate
Add to Assembly
Add to Assembly Advanced
Modify ▶
Highlight all Instances
Category Name
Attributes ▶
Thumbnail View Direction ▶
Rebuild Thumbnail
Add to New Library

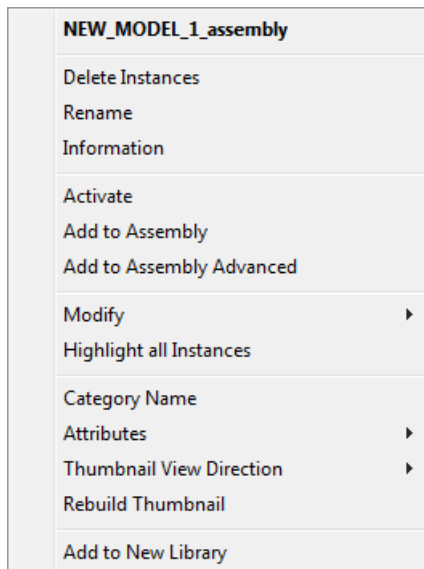
- If the selected component is in a library, but not in the current model:

Component library (parametric)
Information
Add to Assembly
Add to Assembly Advanced
Highlight all Instances
Category Name
Add to New Library
Close Parent Library

- If the selected component is in a library and in the current model:

cyl
Delete Instances
Information
Add to Assembly
Add to Assembly Advanced
Modify ▶
Highlight all Instances
Convert to Local
Category Name
Add to New Library

- If the selection is an assembly:




The options below are present on one or more of the context menus:

Information - Displays the **Selection Information** dialog.

Delete instances - Deletes the components in the model that have been created from this component. The component is also removed from the component library.

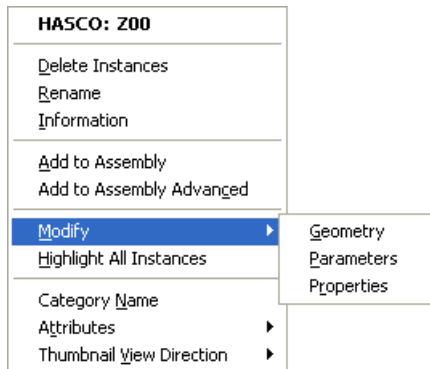
Rename - Edit the name of the selected component in the component library

Add to Assembly - Use this option to position the component using the cursor. The cursor will change shape  and the component graphics will appear on the edge of the model window nearest to the cursor. Move the cursor to the required position and click the left mouse button. You can add multiple instances of the same component. When you have added all required components, click



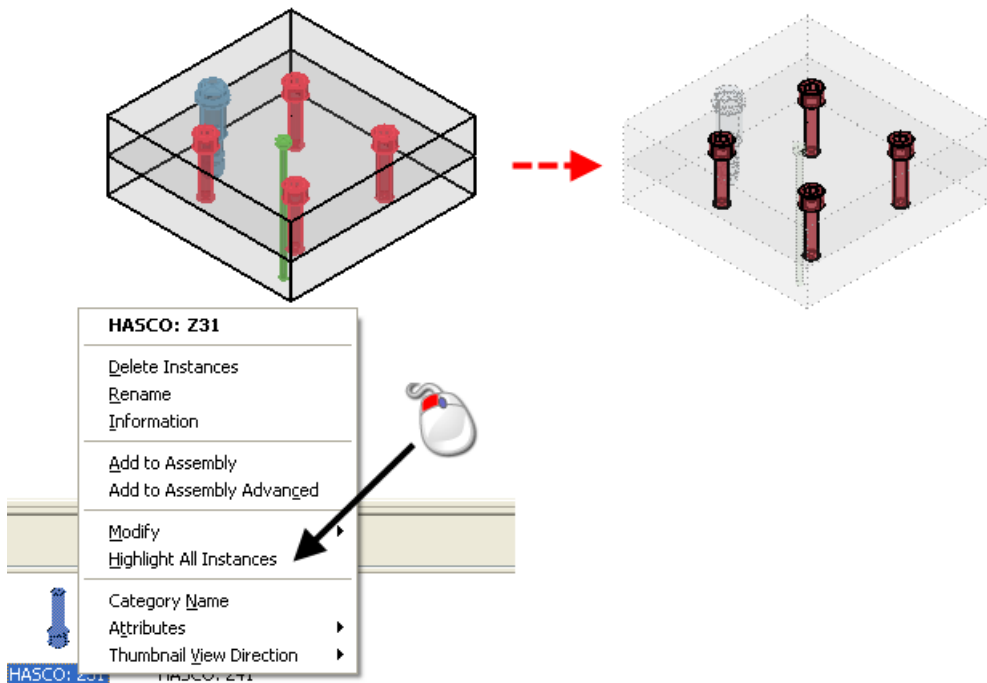
Add to Assembly Advanced - Use this option to position the component using Insert axial component toolbar (see page 105).

Modify - Use these options to modify the component in the component library. This change will affect all instances of the component in all models that use this library.



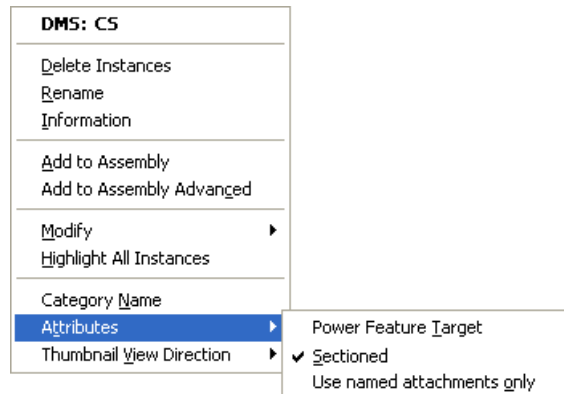
Move the mouse over the context menu options to highlight the components that will be modified.

Highlight All Instances - All instances of this component in the model are highlighted and the remainder of the model is dimmed.



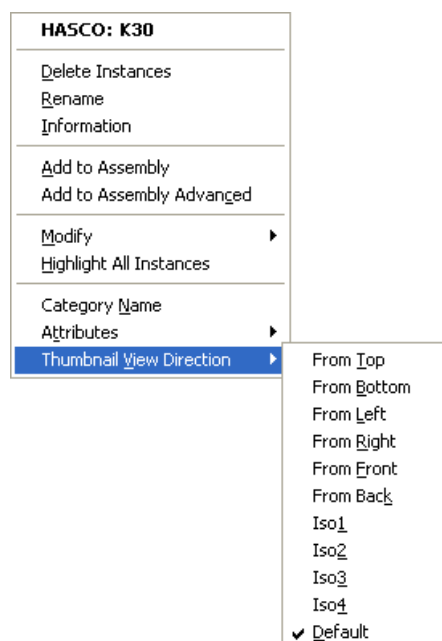
Category Name - Click on this to open the **Set Category Name** dialog. This lets you set the category name and changes the name that appears on the tab in the Component Library window.

Attributes - Set the attributes of the selected component in the component library. Any changes you make will affect all instances of the component in all models that use this library.



- **Power Feature Target** - If selected, the component will be a Power Feature target (see page 48)
- **Sectioned** - If selected, revolved components are sectioned when creating a Section View of the assembly in Delcam Drafting.
- **Use named attachment only** - If selected, the attachments that are available during positioning operations (drag and drop, creating components on cursor and creating relations using relation toolbar) are restricted. If deselected, all attachments are available.

Thumbnail View Direction - Use these options to set the view direction for the thumbnail for the selected item. The default viewing direction is defined on the **General** page of the **Assembly Options**.



Close Parent Library - Close the library that contains the selected component.


Highlighting components using the Component Library popup

Select the **Highlight All instances** option on the Component Library popup to highlight the components on the model that originate in the component library of another model.


Selecting components using the Component Library popup

Select components in the Component Library using standard Windows selection techniques:

- If you select a single component with the mouse, you can add additional components to the selection by holding down the CTRL


key and clicking a component 

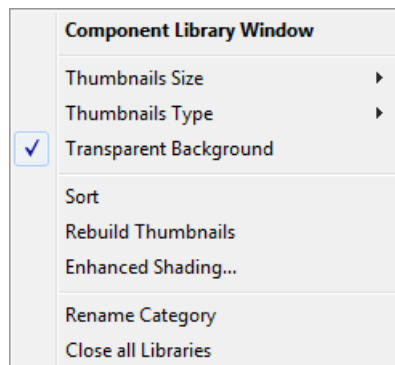
- To add a range of components to the component you have already selected, hold down the SHIFT key and click another

component 

Component Library Window context menu

Use this popup to change the appearance of the Component Library Window.

- 1 Right-click  in free space in the component library window (not on a component) to display the context menu.



- 2 Select the option you require.

Thumbnail Size — Choose the size of the thumbnails (**Small**, **Medium** or **Large**) that are displayed in the component library window. The default size is **Large**.

Thumbnail Type — You can display the thumbnails in shaded or wireframe mode. The default setting is **Shaded**.

Transparent Background — If *ON*, the background of the thumbnails in the component library is transparent. If *OFF*, the background is coloured.

Sort by — This lets you sort the thumbnails into alphabetical order.

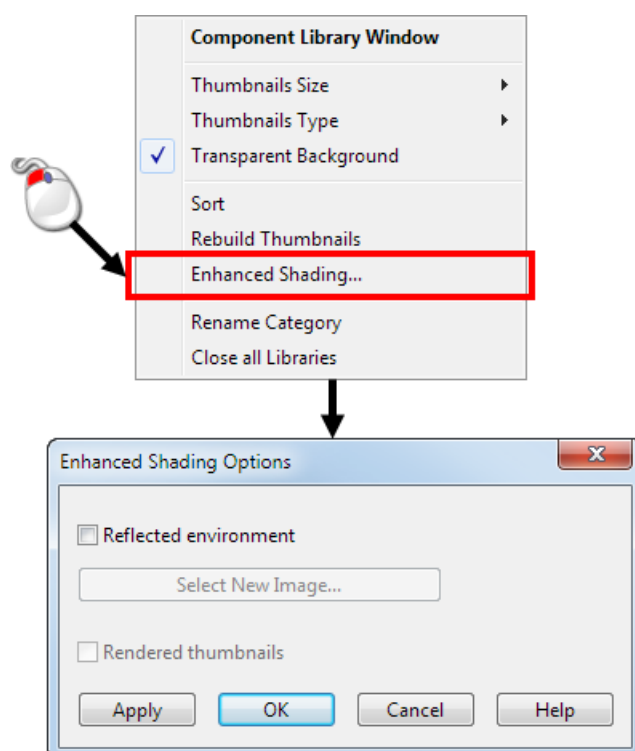
Rebuild Thumbnails — The thumbnails in a component library are automatically updated in the following circumstances:

- A component is added to the active assembly
- A component is removed from the active assembly
- Changes are made to the geometry, category name or model definitions name of the active assembly.

This means that the thumbnails of the component definitions of the active model are always up to date and do not require manual updating using the **Rebuild** option. However, changes to component definitions in the inactive libraries will not be updated automatically and will need to be updated manually.

Enhanced Shading — If selected, photo-realistic shading is added to the thumbnails in the component library window.

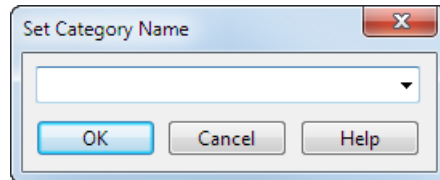
- a** Select **Enhanced shading** from the Component Library Window context menu to display the **Enhanced Shading Options** dialog.



- b** Use the options to specify the enhanced shading for the components in the active model.

- c Click **Apply** to apply the options directly from the dialog. Alternatively, apply the options later in the session by selecting **Rebuild Thumbnails (Component Library Window context menu and Component context menu in the Component Library)**

Rename Category — Select this option to display the **Set Category Name** dialog and enter the new category name.



Close all Libraries — Close all the open component libraries.

Adding components to an assembly using the Component Library

Add components from a component library to an assembly using one of the following methods:

- Double-clicking the component (see page 41)
- Using the context menu (see page 42)



You can use any component definition from any component library in another assembly.

(a) Select the appropriate tab for the component library you want to use.

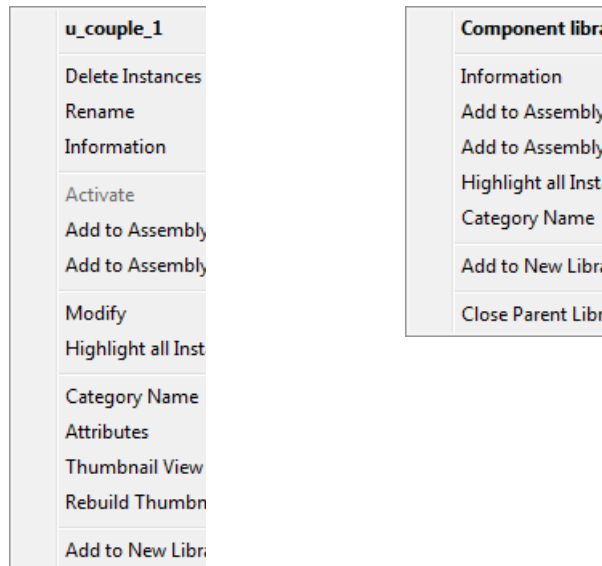
(b) Double click the definition of the component. The component will be added to your assembly.

Adding components from a library using double-click


- 1 Ensure that the Component Library window is open, displaying thumbnail images of the components in the Component Library.
- 2 Double click the thumbnail image of the definition of the component and it is positioned on the cursor using the default attachment. If you don't want to use the default attachment, try one of the following:
 - Hold down **CTRL** to flip the component orientation.
 - Press the up and down arrow keys to cycle through other attachments.

Adding components from a library using the context menu

- 1 Open the appropriate Component Library by selecting the **Open Component Library...** from the **File** menu.
- 2 Using the right mouse button, select a component in the Component Library to be added to the assembly. The popup that is displayed will depend if the selected component is in the current model.



- 3 Select one of the following to add the component to the assembly:

Add to Assembly - Use this option to position the component using the cursor. The cursor will change shape  and the component graphics will appear on the edge of the model window nearest to the cursor. Move the cursor to the required position and click the left mouse button. You can add multiple instances of the same component. When you have added all

required components, click .

Add to Assembly Advanced - Use this option to position the component using Insert axial component toolbar (see page 105).

Adding components to a component library

Supplement a component library by:

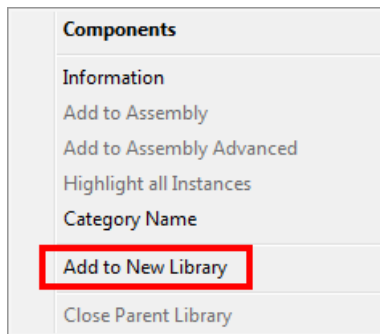
- adding components from the current component library to a new library (see page 43).
- adding a component from the current model to a new component library (see page 43).

- adding a component from the current model to the current component library (see page 45).

Adding components from the current component library to a new library



- 1 Right-click one or more selected objects in the Component Library window to display the **Component library** context menu.
- 2 Select **Add to New Library** to display **Select Library** dialog.



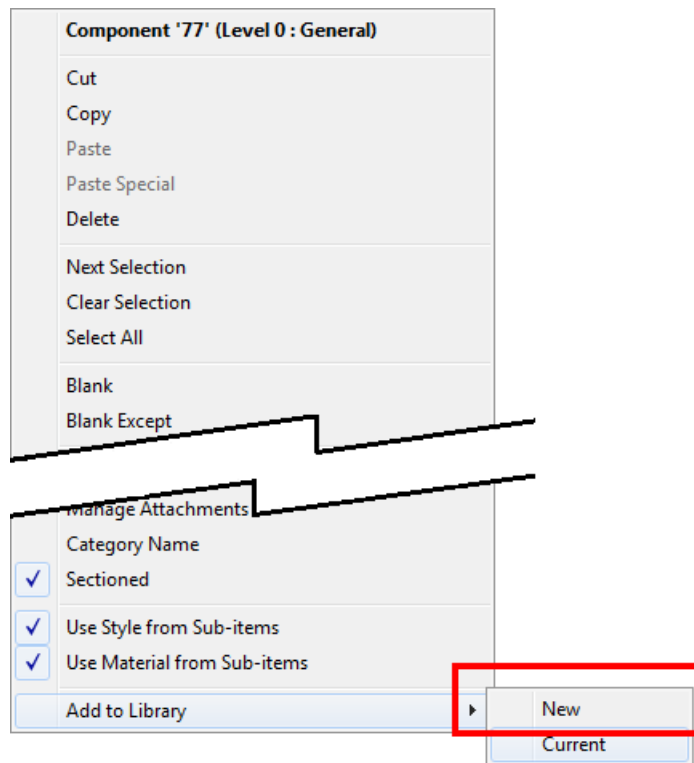
- 3 Enter the name of a new or existing library file.

Adding a component from the current model to a new component library




- 1 Right click one or more objects in the graphics window to display the **Component library** context menu.

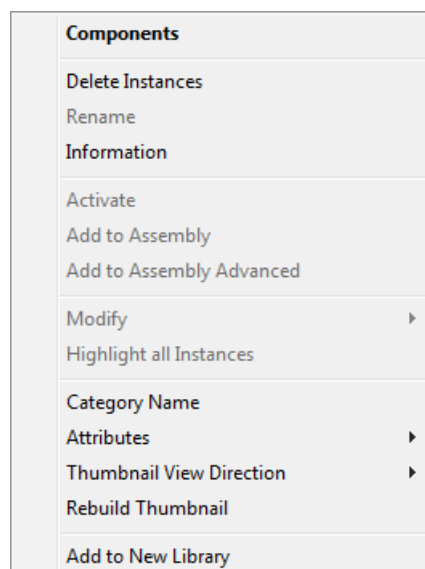
- 2 Select **Add to New Library > New** to display **Select Library** dialog.



- 3 Enter the name of a new or existing library file.


Adding a component from the current model to a new component library from the library window

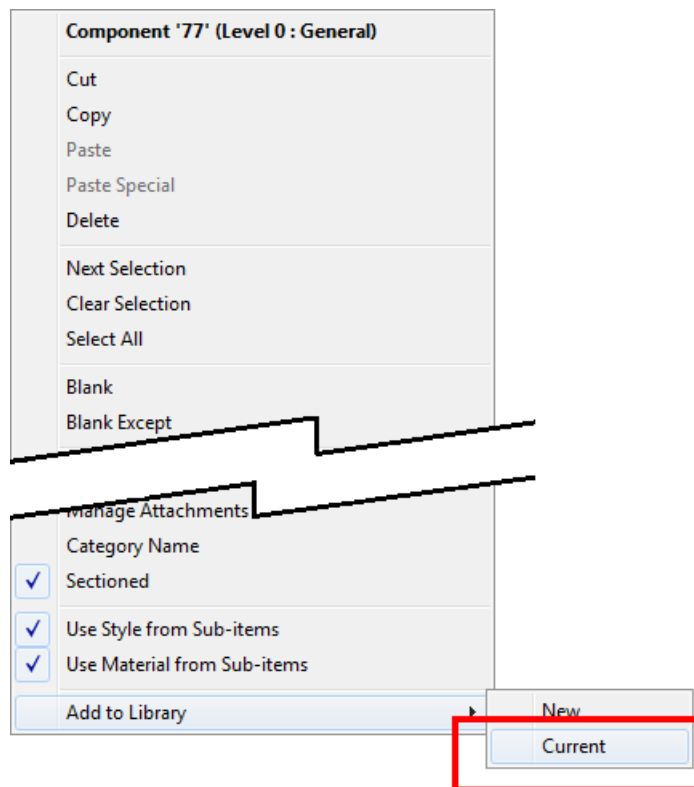
- 1 Right-click  one or more selected objects in the Component Library window to display the **Component library** context menu.



- 2 Select **Add to New Library** to display **Select Library** dialog.


Adding a component from the current model to the current component library

- 1 Right-click  one or more objects in the graphics window to display the **Component library** context menu.
- 2 Select **Add to New Library > Current** to display **Select Library** dialog.

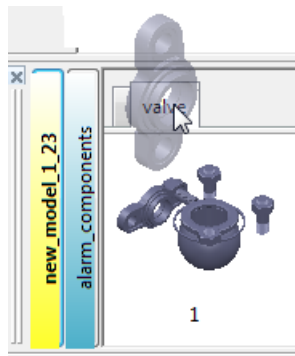


- 3 The objects are copied into the current component library.

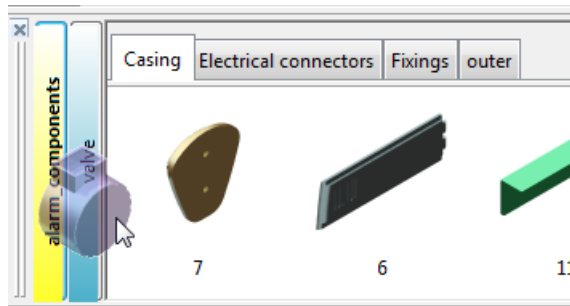
Copying a component to another library, category or component library window

- 1 Drag  the component so that the component is hovering over a vertical library tab or a category tab. The component library window that is displayed will reflect the position of the cursor.

- Adding a component to a category:



- Adding a component to a library:



- 2 When the correct library is displayed, release the mouse button.
- 3 The component is added to the library.

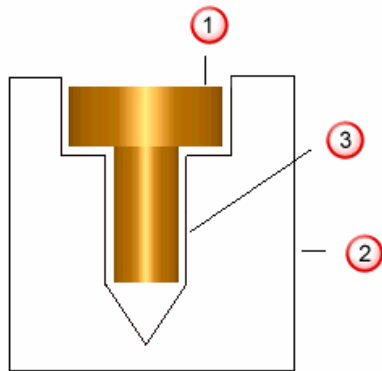
Power Features

What is a Power Feature?

A **Power Feature Solid** is a special solid for storing features associated with the component (**Main Solid**). There can be more than one Main Solid, but only one Power Feature Solid.

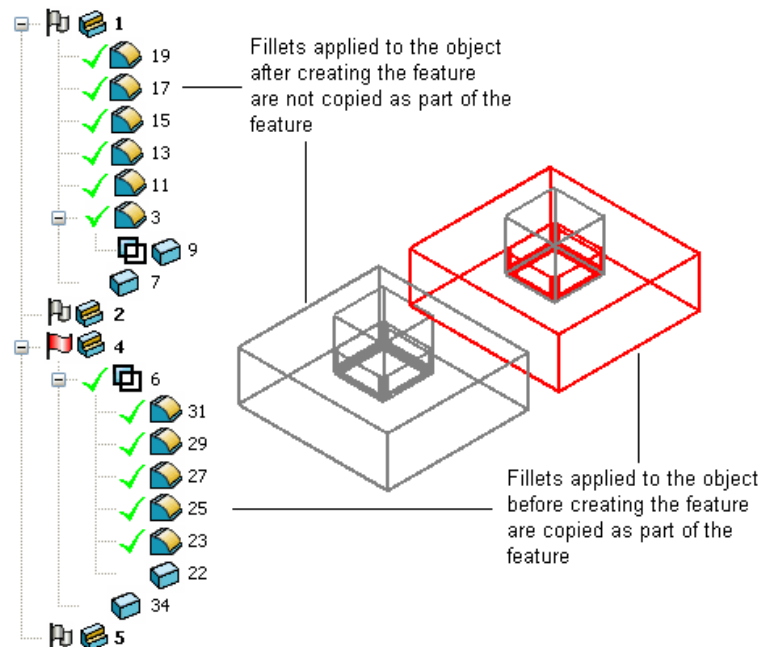
- ① - Main solid.
- ② - Power Feature solid.

③ - The Power Feature is used to create fit/clearance for the main solid.



A Power Feature is generated from a component. It is a second solid and is hidden unless the component or component definition is modified.

The Power Feature contains features that are copied to any solid that intersects with the component when **Generate Power Features** is selected. Not all features may be defined as Power Features.



Features that can be copied include hole, boolean, cut/boss and pocket/protrusion. When Power Features are applied, the name of a copied feature is based on the original name. For example,

Power Feature name - *HASCO Z41 fit hole*

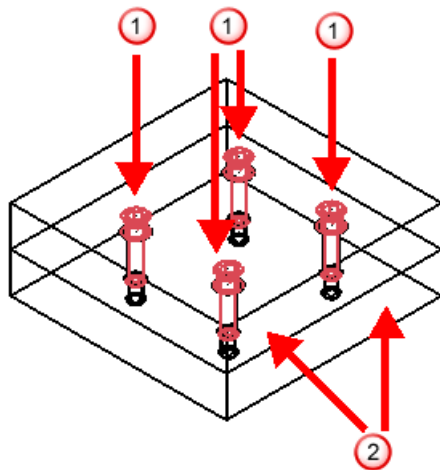
Name of copied feature - *HASCO Z41 fit hole_1*

Features that cannot be copied directly include fillets, chamfers and hollows. However, these features can be copied if they are applied to a feature that can be copied, such as a boolean

What are source and target components?


A **source component** is a component that has a Power Feature Solid registered in its component definition. It will then apply these Power Features into any intersecting target components (for example a plate). The components marked with ① on the model below are **source components**.

A **target component** is a component that has intersections with a source component. The components marked with ② on the model below are **target components**.

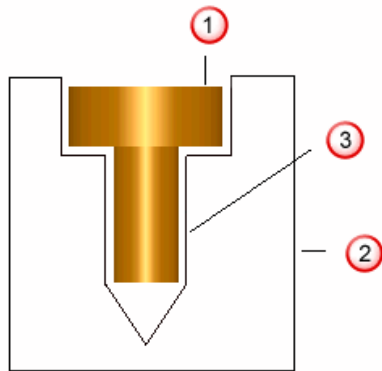


Creating a Power Feature

You register a **Main Solid** and **Power Feature Solid** as follows:

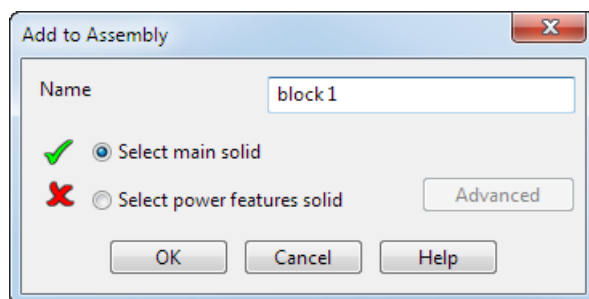
- 1 Create the solids to be used in the assembly
- 2 Select **Assembly**  from the main toolbar. The Assembly modelling toolbar will be displayed.
- 3 Select a solid with the right mouse button to display the solid context menu.
- 4 Select **Add to Assembly Advanced** to display the Add to Assembly dialog (see page 49).
- 5 Use the dialog to register components, selecting the appropriate geometry for **Main solid** and **Power Feature solid**.
 - ① - Main solid.
 - ② - Power Feature solid.

③ - The Power Feature is used to create fit/clearance for the main solid.



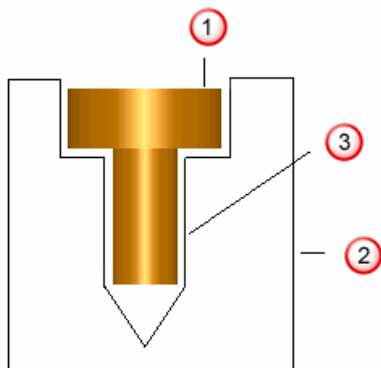
Add to Assembly dialog

This dialog is used to create a component from a solid and automatically register the associated component definition. It is displayed when you select **Object > From Solids with Power Features**.



Name — The component is given a default name that is the name of the Main Solid. You can rename this if necessary.

Select main solid — Select the component ①



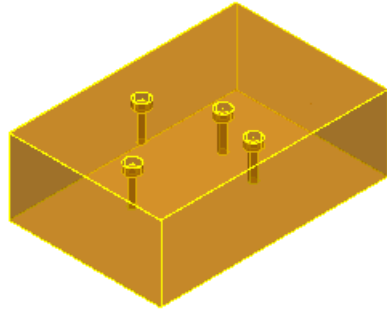
Select power features solid — Select the solid that stores the hole features associated with the Main Solid ②. The Power Feature is used to create fit/clearance ③ for the main solid.

Advanced — Displays the **Set Advanced property for Power Features** dialog.

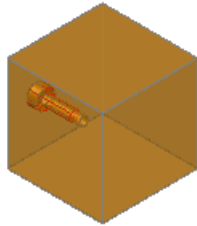
OK — Registers the component definition, removes the dialog from the screen and adds the component to the Assembly.

Using a Power Feature

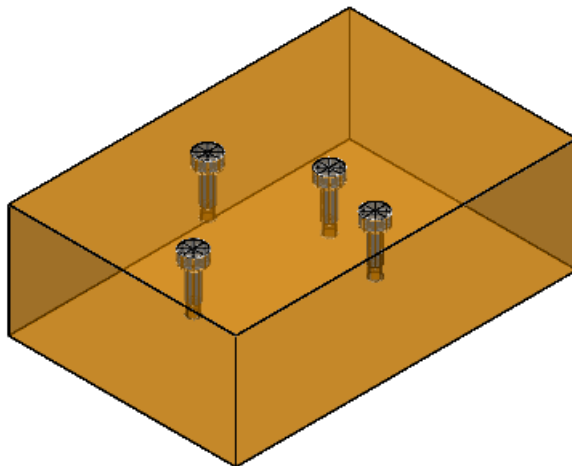
- 1 Create a single component like the one below.



- 2 Create a component for the Main and Power Feature solids



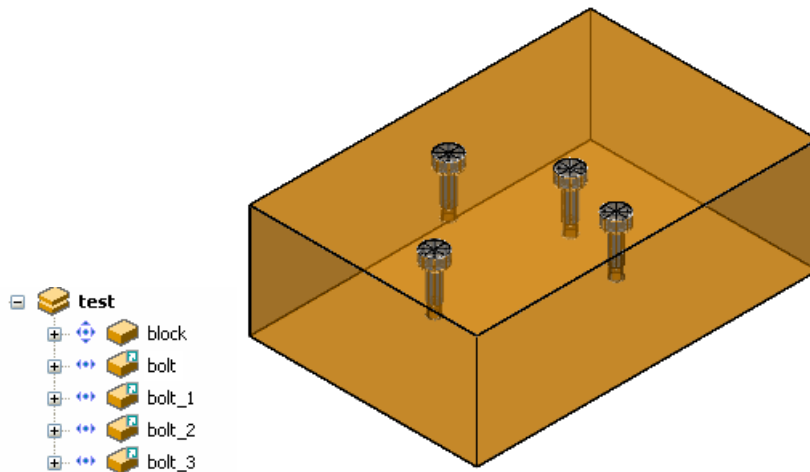
- 3 Create an assembly consisting of the single component and several components that use the Power Feature component.



Modifying a component definition for a Power Feature

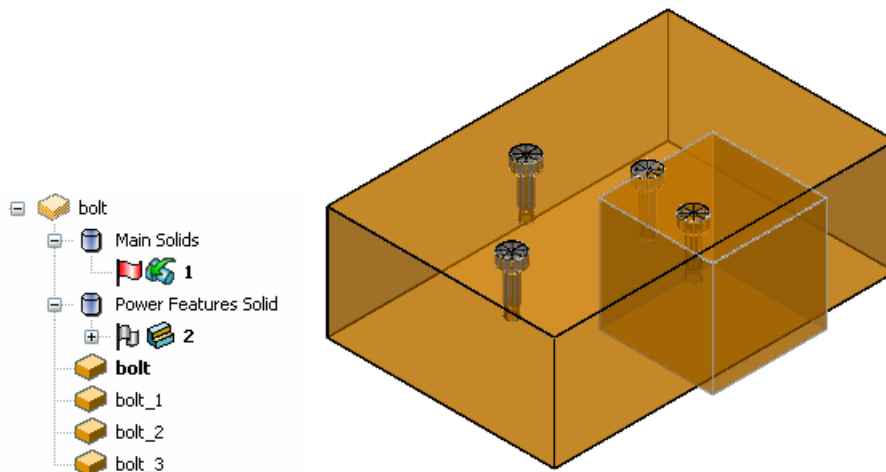
The model below shows a simple assembly called *test*.


- The *bolt* component has been created with a main solid and a Power Feature.
- The *block* is a single component.



You can modify the component definition for the component using

- assembly tree (see page 51)
- component library window (see page 52)

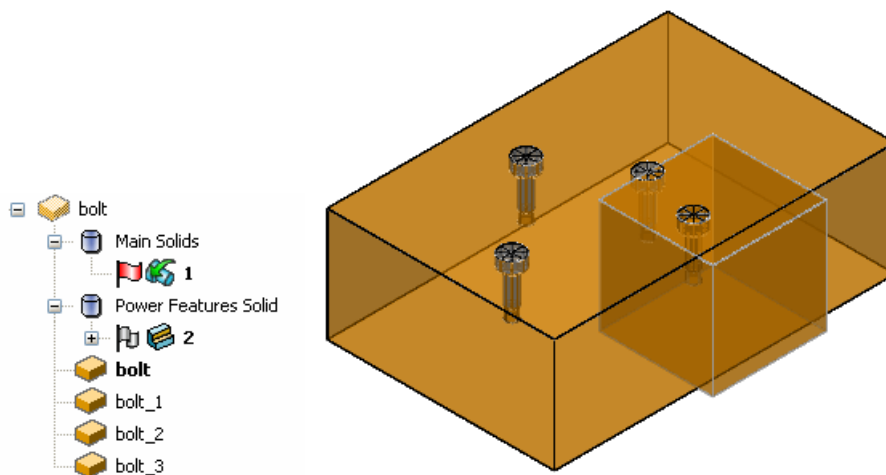



- 1 Make the required changes to the geometry
- 2 Select the  on the **Modify** toolbar. The solid converts back into the component

Modifying the power feature using the assembly tree

- 1 Right click the *bolt* component in the assembly tree to display the *bolt* popup.
- 2 Using the popup, select **Modify Geometry** from the **All Instances** options.

The geometry of the component definition is visible on the screen and the tree of the component definition is displayed.




- 3 Make the required changes to the geometry and select the  on the **Modify** toolbar. The solid converts back into the component

Modifying the power feature using the component library

- 1 Ensure that the component library window is displayed. If not, select **Component Library** from the **Windows** options of the **View** menu.
- 2 Right click the component definition for *bolt* in the component library window to display the *bolt* popup.
- 3 Using the popup, select **Geometry** from the **Modify** options.

The geometry of the component definition is visible on the screen and the tree of the component definition is displayed.



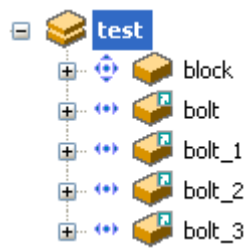
- 4 Make the required changes to the geometry and select the  on the **Modify** toolbar. The solid converts back into the component definition and is removed from the screen.

All objects are visible again and the components that use the modified component definition are updated.

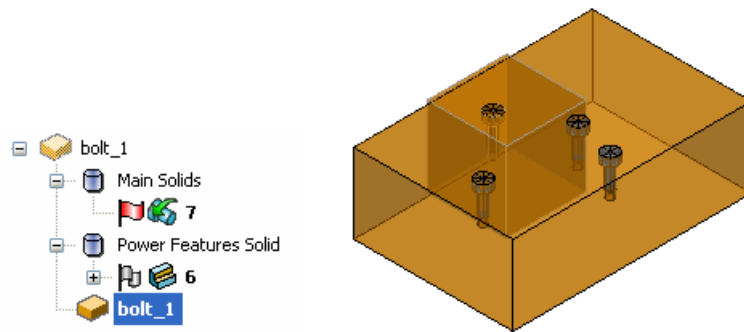
Modifying a single component in the assembly


To modify the component called *bolt_1*,

- 1 Right click on *bolt_1* in the assembly tree.



- 2 Select **Geometry** from the **Modify** options on the popup. The geometry of the component is visible on the screen and the assembly tree shows the power features and main solid.



- 3 Make the required changes to the geometry. A message is displayed if you try to modify the geometry of a Power Feature Solid to include a feature that cannot be copied to the component.
- 4 Select the  on the **Modify** toolbar. The power feature becomes hidden once more.

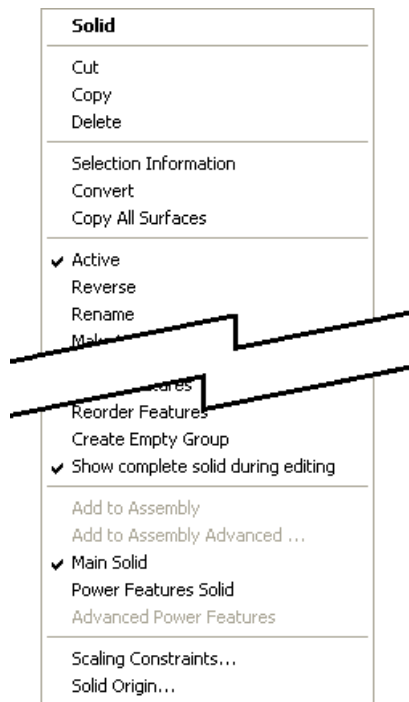


*If you are using PS-Moldmaker components, you should modify the geometry by selecting **Parameters** from the Modify options, rather than **Geometry**.*

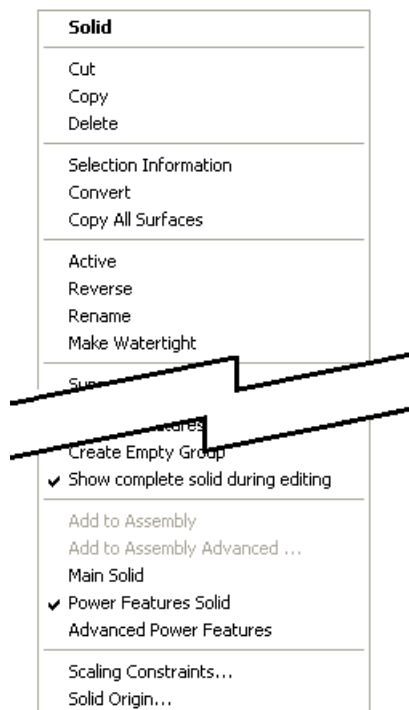
Adding and removing components and component definitions

When you **Modify** a component or component definition, you can use the solid context menu, from either the graphics window or the assembly tree, to

- Add and remove a Main Solid (there can be more than one Main Solid)



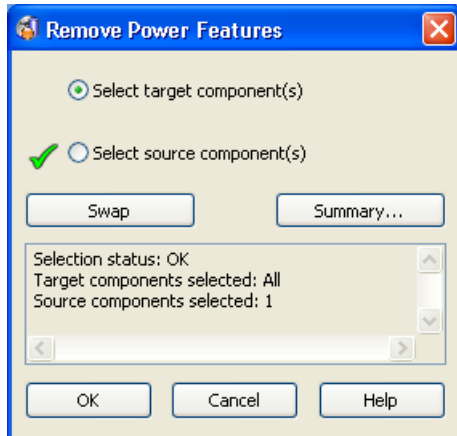
- Add and remove the Power Features Solid (there can only be one Power Feature Solid)




If you delete the component that is the origin of a Power Feature, the Power Feature is deleted.


Remove Power Features dialog

This dialog is used to remove Power Features from the selected component. It is displayed if **Show Generate Power Features** dialog is selected on the **General** page of the **Options** dialog.



Select target component(s) - If you displayed the dialog by selecting a target component (for example a plate), this option will have a  and you will only need to select a source.

If you didn't select a source component and therefore have only target components selected, all the components of the active assembly will become source components. In this case, clicking **OK** will remove the Power Feature for every instance of the target component(s) intersecting with a source component.

Select source component(s) - If you displayed the dialog by selecting a source component (for example a screw), this option will have a  and you will only need to select a target.

If you don't select a target component and therefore have only source components selected, all the components of the active assembly will be target components. In this case, clicking **OK** will remove the Power Feature for every instance of the source component intersecting with another component.

If you select a source component that does not have Power Features defined:

- an error message is displayed in the dialog.
- the component is deselected.
- the **Swap** button becomes inactive.

If source and target components have been selected the power features in the selected targets will only be removed if they have been applied from the selected source components. This means that exactly what has been generated for a given selection in generate mode will be removed.

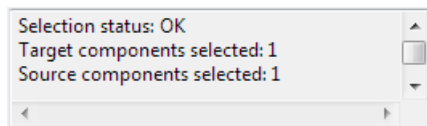
Swap - This option is only active if all selected target components also define Power Features. Selecting this option will have the following effect:

Existing target components will become source components.

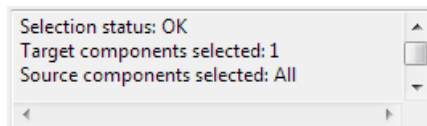
Existing source components will become target components.

Summary - This displays the **Power Features Summary** dialog so that you can see in more detail how the Power Features will be applied.

Selection information - This panel displays information on the number of target and source components that have been selected. The following information panel shows that one source and one target component have been selected.



If a source component was selected, but no target component, the message shows that all the components will be considered as targets.



OK - Remove the Power Features where the target components intersect with the source components.

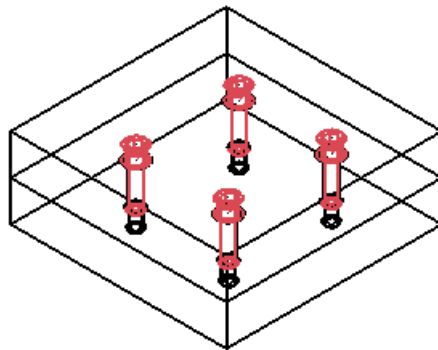
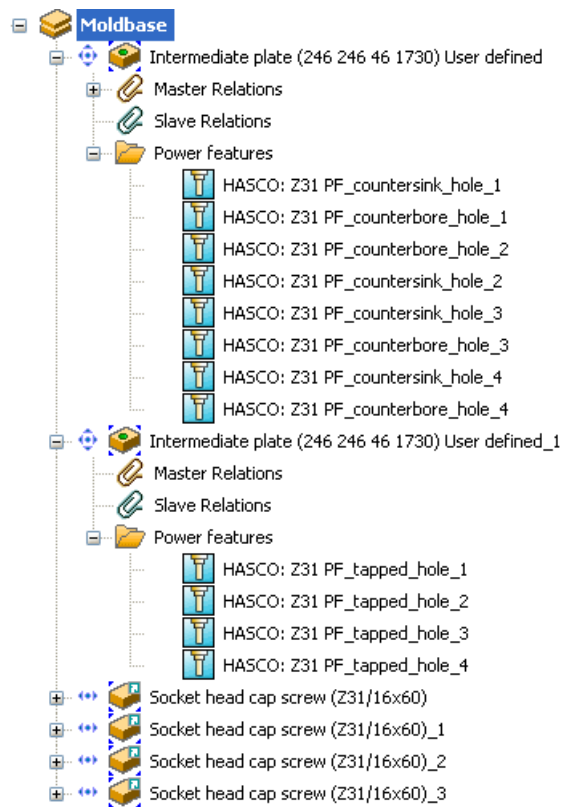
Cancel - Close the dialog and cancel any changes you have made.

Using the Power Features Summary

You can specify how Power Features are applied by using the Power Features Summary. Work through the following example to see how you can use the Power Features Summary dialog to control the intersection of the target components with the source components.

- 1 Create a plain moldbase and use **Component Wizard** to place some screws. The example below uses four *Hasco Z31* screws.
- 2 From the assembly context menu, select **Generate** from the **Power Features** options

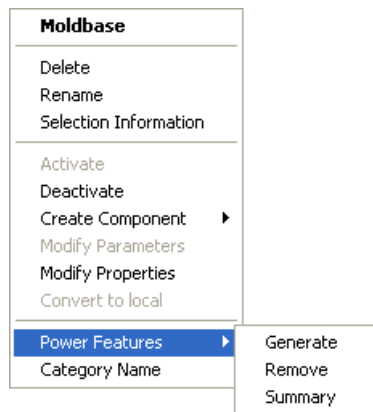
The model and the assembly tree will be updated.



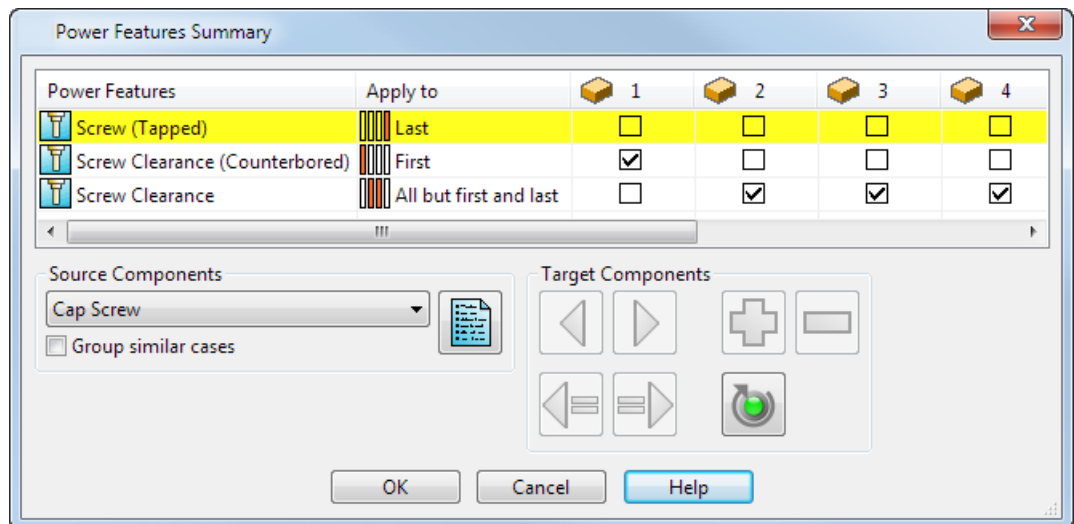
- 3 From the assembly toolbar, click **Show Power Features Summary**



or from assembly context menu, select **Power Features - Summary**.



The Power Features Summary dialog (see page 59) is displayed.

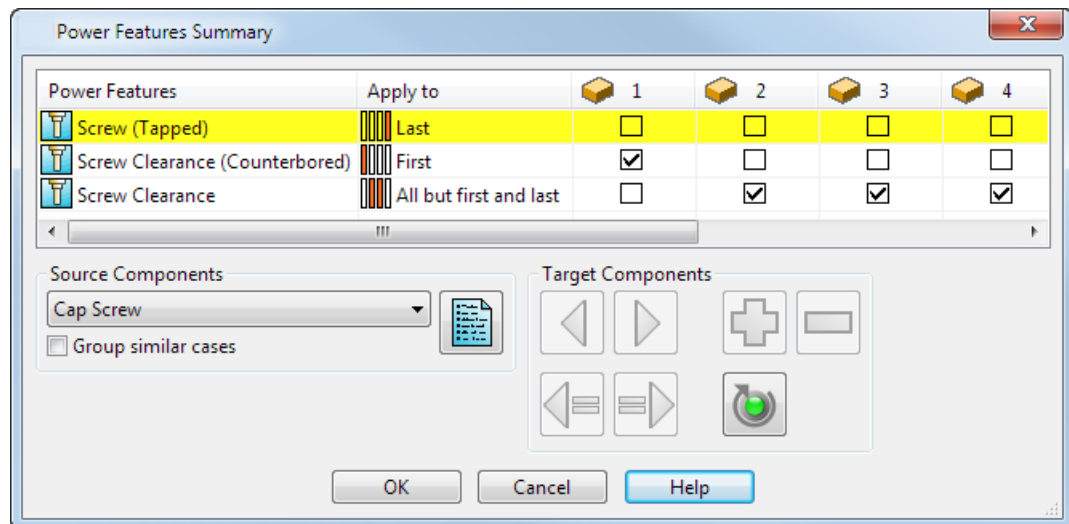


- 4 Select a **Source Component** from the drop-down list and use the options on the dialog to make any changes.
- 5 Click **OK** to save the Power Feature options that you selected/deselected. The options are applied and the Power Features are updated. The dialog is closed.

If the Power Features Summary dialog was opened by selecting the **Summary** button on the Generate Power Features dialog (see page 71), this dialog will also be closed.

Power Features Summary dialog

Use this dialog to choose how to apply Power Features.



The default size of the dialog is changed to reflect the number of Power Features and target components. You can increase the size of the dialog, but you cannot make it smaller than the default size that is initially displayed.

Power Features — The Power Feature solid of the source component. The tooltip displays the name of the Power Feature solid of the source component. This column displays:

- Feature icon
- Hole use (for example, Ejector pin clearance) for holes



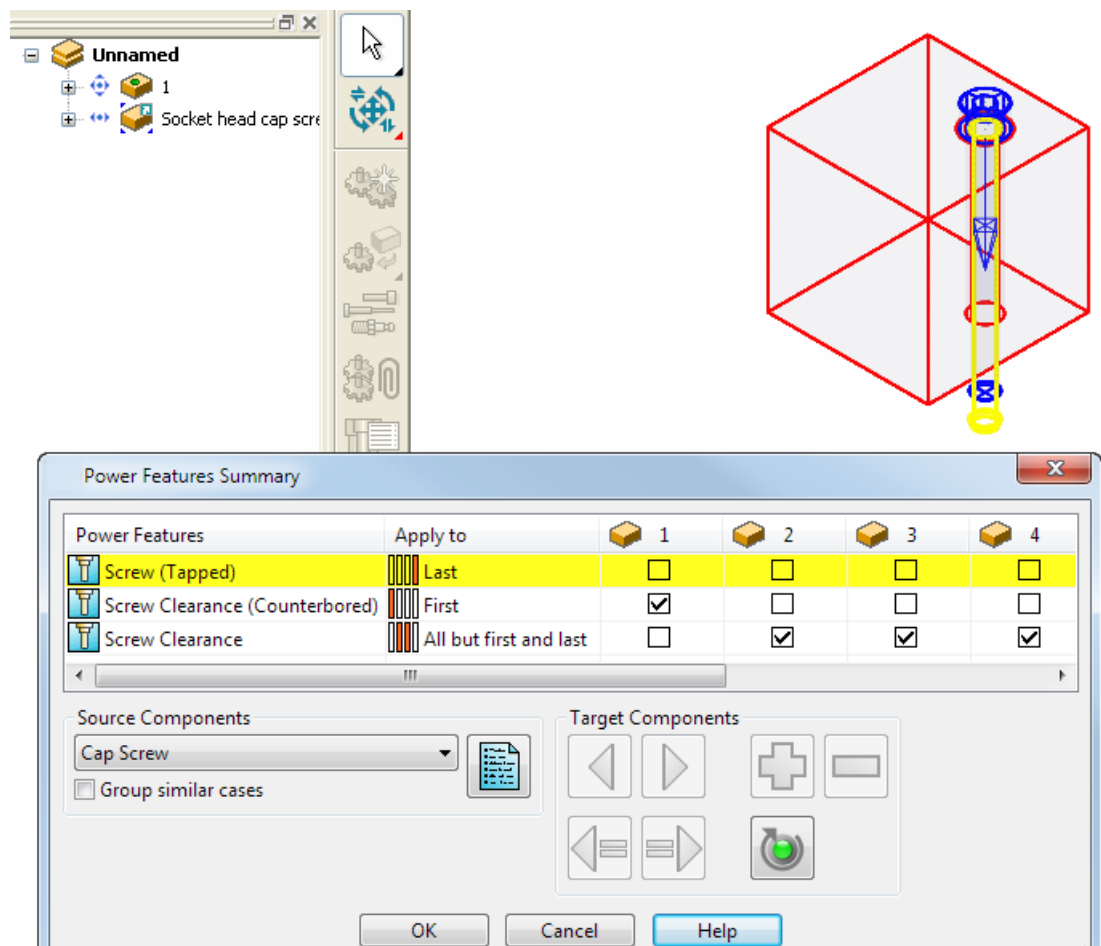
Hover over the hole use to see the hole name is displayed on the tooltip

- Feature name for features other than holes.

Selecting a Power Feature from the list:

- Highlights the row in the dialog.
- Adds instrumentation to the Power Feature in the graphics window.
- Highlights the target component in the graphics window.












For further details, see **Highlighting Power Features**.






Apply to — Each Power Feature stores a rule defining how it should be applied. These rules are based on the order in which the source component (for example, a screw) intersects with target components (for example, plates). So, a screw will apply a counterbored clearance hole to the **First** plate it intersects with.

Each source component has a direction vector used to order these intersections (displayed as a blue arrow on the **Power Features Summary** dialog). You can modify this direction using the **Modify Direction** button on the **Set Advanced Property for Power Features** dialog. The **First** intersection is the nearest point on the source component along this direction, the **Last** intersection is the furthest point.

For a given set of intersections between the source and target components, this column shows how the Power Features would be applied. These options allow you to be more specific about whether or not a Power Feature is copied to a particular target component, based on the intersection order of the target components. The options that may appear in the column are explained in the following table.

Apply to	Action
 All	Copy feature onto all components.
 All but first	Copy feature onto all components except the first one.
 All but first and last	Copy on all target component except first and last
 All but last	Copy feature onto all components except the last one.
 All but number	Copy feature onto all components except the <i>n</i> th one. For example, if Number is 3, the feature will be copied onto all components except the 3rd.
 First	Copy feature onto the first component only.
 First and last	Copy feature onto the first and the last component.
 First and second	Copy feature onto the first and the second component only.
 Last	Copy feature onto the last component only.
 Last and last but one	Copy feature onto the last and the last but one component only.
 Number	Copy feature onto the <i>n</i> th component only. For example if Number is 3, the feature will be copied onto the 3rd component only.

 1 — Each target component that intersects with the source component is shown as a column on the Power Features summary dialog. The order of the target component columns is determined by the order in which they intersect with the source component. You can change the order that the target components intersect with the

selected source components by using the **Target Components**   buttons.

Selecting a target column,

- Highlights the column in the summary
- Highlights the target in the main graphics window
- Adds instrumentation to all selected Power Features for that target

Selecting an option in the target column,

- Highlights the cell in the summary

- Add instrumentation to the Power Feature in the graphics window
- Highlights the target in the main graphics window
- Toggles whether the feature is applied (☒) or not (☐)

Source Components — Select a source component from the drop down list. If there are 4 bolts, there will be four options in the drop-down list. Once you have chosen a source component, you can select the options for each Power Feature in the table for that component.



— All the option boxes are set back to the default as indicated by the source and target components in the assembly. If you have reordered the target components in the dialog, the rules will be applied in this order.

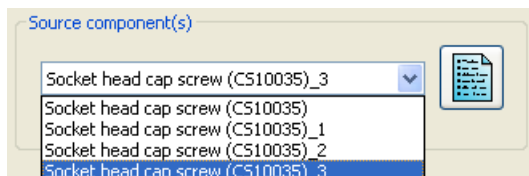


*If the **Power Features Summary** dialog is activated from the **Generate Power Features** dialog, only the selected source components will be shown in the **Select source component** dialog.*

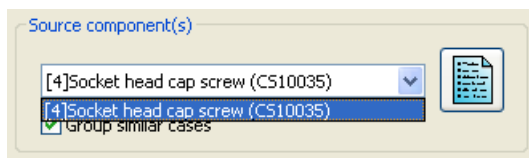
Group similar cases — If *ON*, similar source components are grouped together. Multiple components can then be modified by changing just one.

The example shows four identical components.

- 1 Open **Power Feature Summary** dialog. The **Source Components** drop down list contains all four components

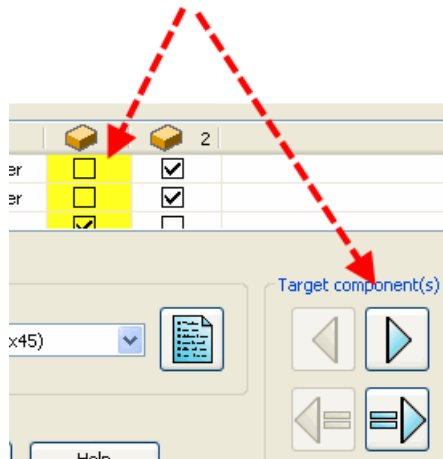


- 2 Set **Group similar cases** to *ON*. The **Source Component** drop down list now contains a single item.



- 3 Turn *OFF* the **Countersunk** option in the **Power Feature Summary** dialog. The change is applied to all the components in the group.

Target Components — The appropriate buttons in this section are highlighted when a target intersection column is selected.



Change the order that the target components intersect with the selected source components by using the following buttons.



— The intersections will be re-ordered and the current selection will be moved earlier in the order. On the dialog, the current target will be moved one column to the left.



— The intersections will be re-ordered and the current selection will be moved later in the order. On the dialog, the current target will be moved one column to the right. Clicking this button sets the target intersection to be the same as the previous intersection.



— Add the selected component as a target. The target is always added as the last intersection. You can change the intersection order of this component and define how the Power Features are applied. This button is disabled if the selected component is already a target.



— Remove the selected target. The component will be removed from the intersection order and Power Features will not be applied to this component.

You can make the target intersection the same as the previous (or next) intersection by using the following buttons:



— Sets the target intersection to be the same as the previous intersection. The following example has three target intersection columns and illustrates how this option works:

1 Select column 2

	1		2		3
<input type="checkbox"/>		<input type="checkbox"/>		<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>		<input type="checkbox"/>		<input type="checkbox"/>	
<input checked="" type="checkbox"/>		<input type="checkbox"/>		<input type="checkbox"/>	
<input type="checkbox"/>		<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	

2



Click . The target intersection columns are updated; there are now two first intersections.

	1		1		2
<input type="checkbox"/>		<input type="checkbox"/>		<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		<input type="checkbox"/>	
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		<input type="checkbox"/>	
<input type="checkbox"/>		<input type="checkbox"/>		<input checked="" type="checkbox"/>	



— Sets the later target intersection to be the same as the current intersection. The following example has three target intersection columns and illustrates how this option works:

1 Select column 2

	1		2		3
<input type="checkbox"/>		<input type="checkbox"/>		<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>		<input type="checkbox"/>		<input type="checkbox"/>	
<input checked="" type="checkbox"/>		<input type="checkbox"/>		<input type="checkbox"/>	
<input type="checkbox"/>		<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	

2



Click . The target intersection columns are updated; there are now two second intersections

	1		2		2
<input type="checkbox"/>		<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>		<input type="checkbox"/>		<input type="checkbox"/>	
<input checked="" type="checkbox"/>		<input type="checkbox"/>		<input type="checkbox"/>	
<input type="checkbox"/>		<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	

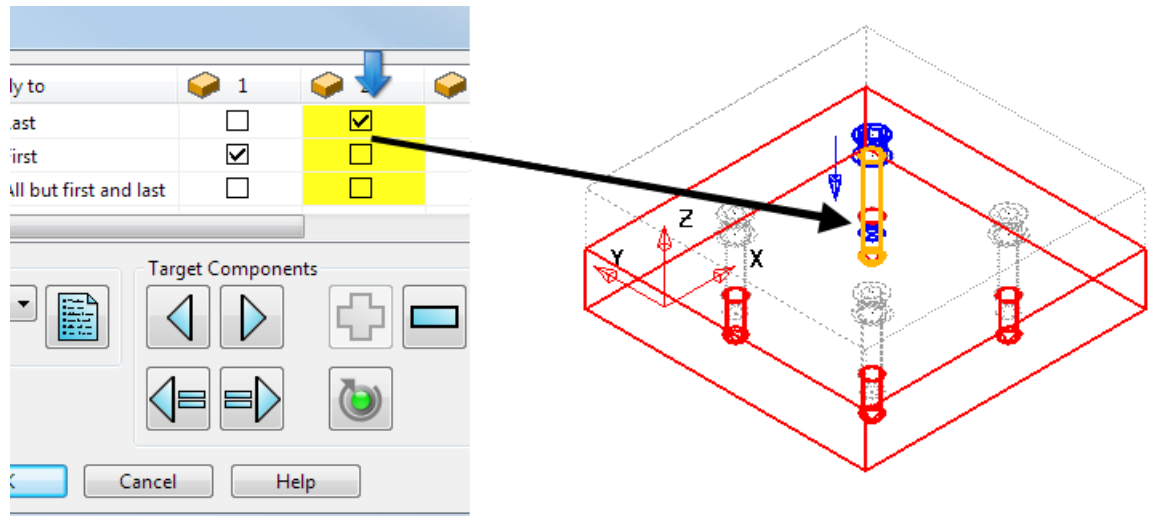


— Resets the target column order to the default and applies the default intersection rules.

OK — Save the Power Feature options that you selected/deselected. The options are applied and the Power Features are updated. The dialog is closed. If the Power Features Summary dialog was opened by selecting the **Summary** button on the Generate Power Features dialog (see page 71), this dialog will also be closed.

Highlighting of components

You can click on a row on column header in the **Power Feature Summary** dialog to highlight the corresponding item on the model. The cursor changes when hovering over an item that can be highlighted.



Clicking on a row, column or cell will highlight the relevant components and features in the main graphics window as follows:

Blue — Source component. This is a component that applies Power Features; for example, a screw.

Yellow — Power Feature(s).

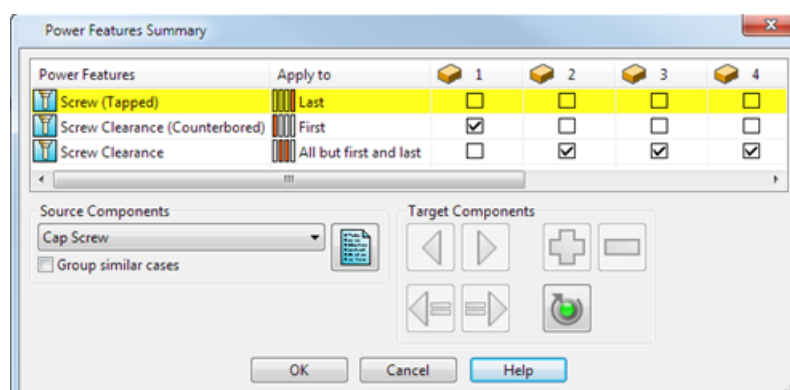
Red — Target component(s). These are components that Power Features are applied into; for example, plates.

In the following example, the first row has been selected in the dialog. This represents the tapped hole Power Feature. The Power Feature rule shows that this feature is **Applied to the Last** intersection only. The highlighting on the model reflects this:

Blue — Screw ①

Red — Last plate ②

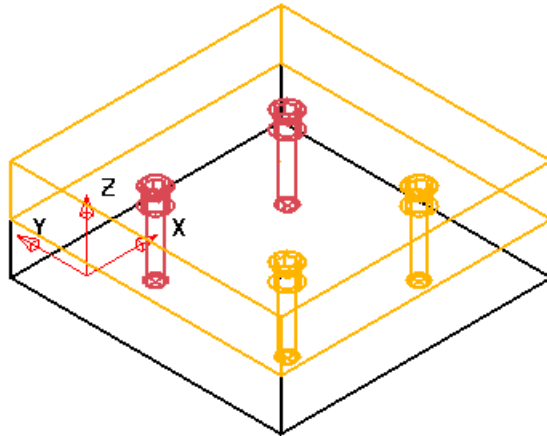
Yellow — Tapped hole feature ③



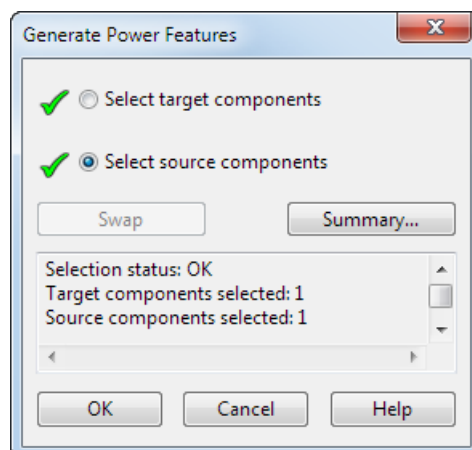
Identifying selected target components in the Power Feature Summary

You can use the **Power Feature Summary** dialog from the **Generate Power Features** dialog to confirm that the Power Features will only be generated for a selection.

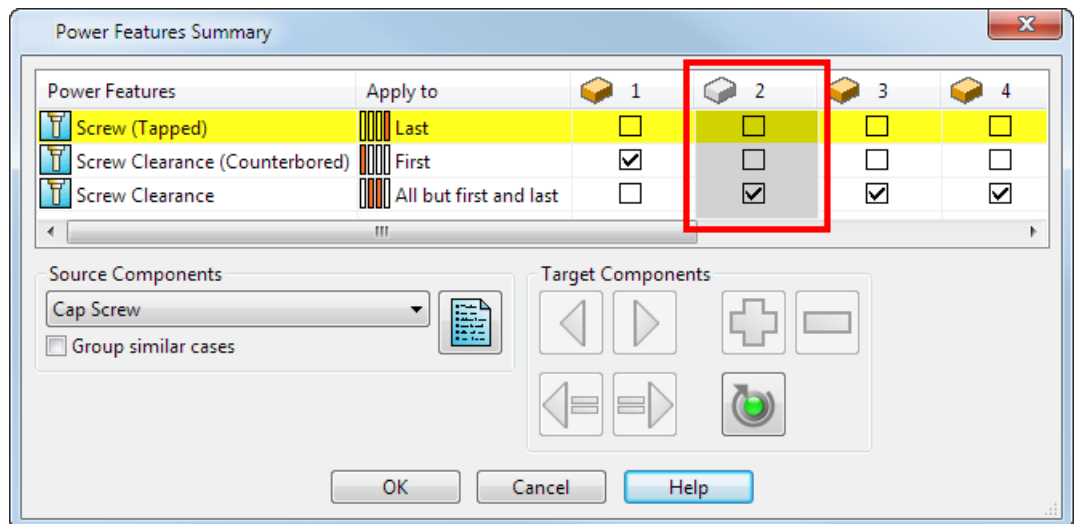
- 1 Select your source and target components.



- 2 Click the **Generate Power Feature**  button on the **Assembly** toolbar to display the Generate Power Features dialog. (see page 71)





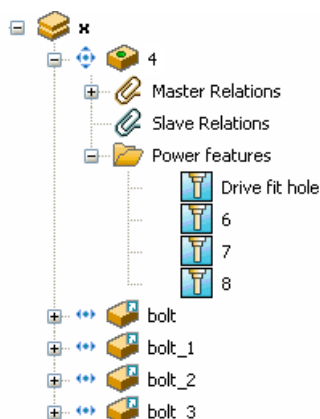
- 3 Select **Summary** to display the **Power Features Summary** dialog. Any unavailable intersections will be greyed out in the dialog.



Updating the Power Features in your model


If you add or remove a component from your model, you need to use **Generate** or **Remove** from the **Power Features** options to update the Power Features in the model.

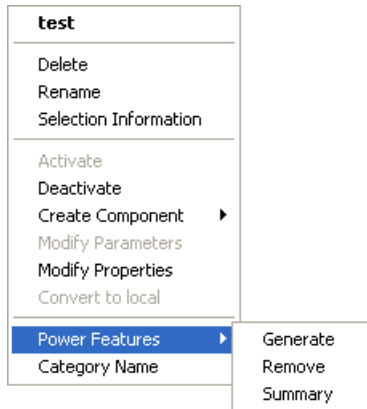
If you change a Power Feature and try to save the model, a message warns you that the definition contains features that are out of date. If you continue with the save operation, the  icon in the Power Features branch of the assembly tree will indicate the out of date Power Feature. The component that contains the out of date Power Feature is indicated by the  icon in the assembly tree.



Updating Power Features - Adding a component

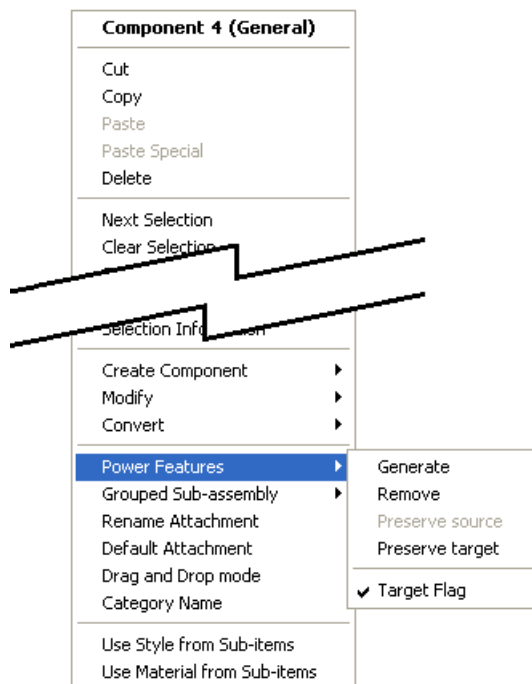
You can do this in one of the following ways:

- Click  (Assembly toolbar). If components are selected, the Generate Power Features dialog (see page 71) is displayed. If no components are selected, Power Features are generated for the active assembly.
- Right click the assembly in the assembly tree and select **Generate** from the **Power Features** options on the context menu.



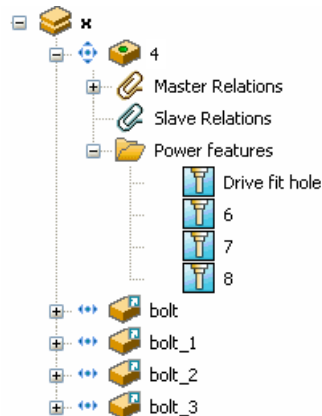
This will update existing, out of date Power Features and will also generate Power Features for any new components for the entire assembly.

- Right click the component in the graphics window and select **Generate** from the **Power Features** options on the context menu.




This method can be used to generate or update Power Features for a sub-set of components. By default, the Generate Power Features dialog (see page 71) will be displayed.

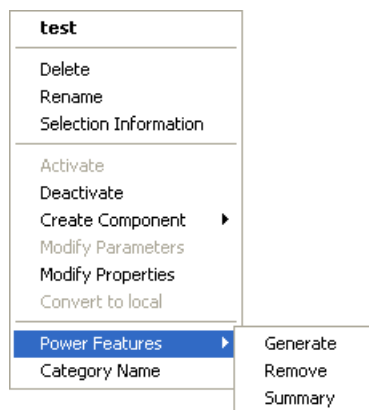
When you select **Generate Power Features**, a new branch is added to the assembly tree. The status of the Power Feature is indicated by the icon that is displayed next to feature.



Updating Power Features - Removing a component

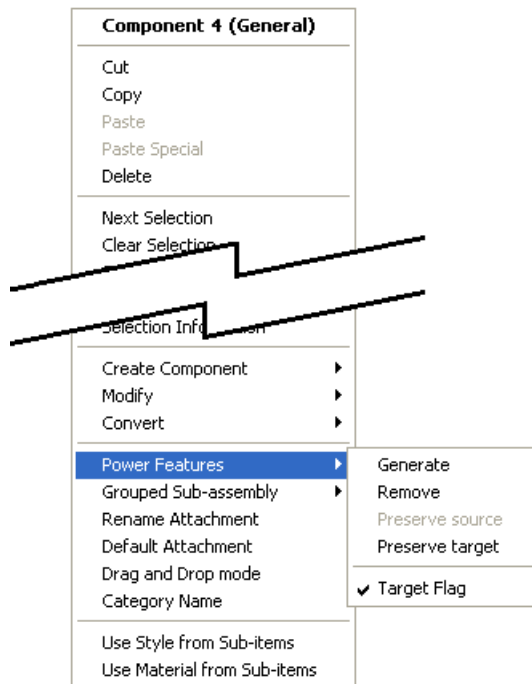
You can do this in one of the following :

- Click  (Assembly toolbar). If components are selected, the Remove Power Features dialog (see page 55) is displayed. If no components are selected, Power Features are removed from the active assembly.
- Right click the assembly in the assembly tree and select **Remove Power Features** from the context menu.



This will remove the Power Features for the whole assembly

- Right click the component in the graphics window and select **Remove Power Features** from the context menu.

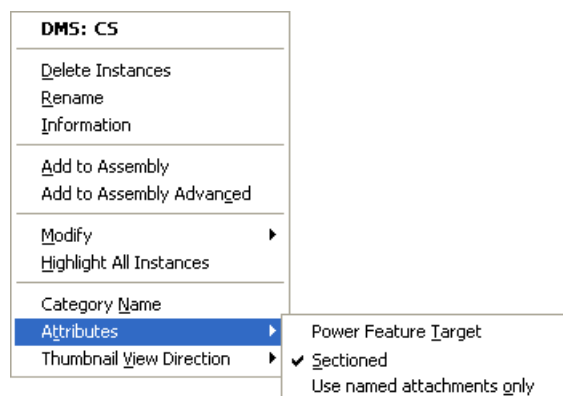


This will remove any Power Features in the selected target component(s).

Excluding components when generating Power Features

Use the **Power Feature Target** option to exclude components when generating Power Features. This acts like a template for the component definition. So, if this option is changed, any components that are subsequently created will inherit the new setting.

- 1 Open the appropriate Component Library.
- 2 Right click the component definition in the component library window, to display the component definition popup.





- 3 To exclude components using this component definition when generating Power Features, select **Power Feature Target** from the **Attributes** option of the component definition popup.

The default setting is *ON*. If you set Power Feature target *OFF*, you need to regenerate the Power Features to exclude the components from the assembly

Controlling the selection of target and source components

You can control the selection of target and source components before you generate the Power Features using the Generate Power Features dialog (see page 71). To display this,

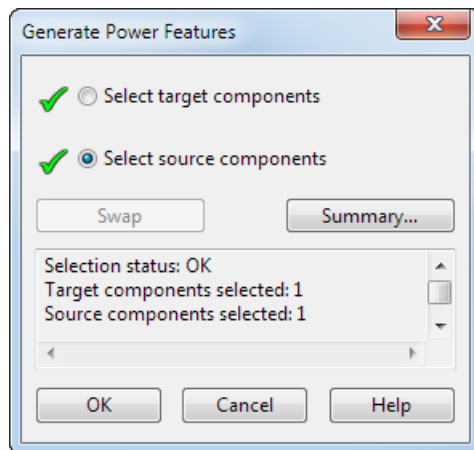
- Select a component on the model. Either click  (Assembly toolbar) or choose **Generate** from the **Power Features** options on the component popup.
- Select an assembly in the assembly tree. Either click  (Assembly toolbar) or choose **Generate** from the **Power Features** options on the assembly popup.


When **Power Features > Generate** is selected from the component popup, the dialog is only displayed if **Show Generate Power Features dialog** is selected on the **Tools > Options > Assembly > General** dialog. The dialog is not normally displayed if the option is deselected. In this case, the component selection will be filtered automatically to identify which component is the target and which is the source component. However, the dialog will be displayed if a selected component could be either a source or a target.

Generate Power Features dialog


This dialog controls the selection of target and source components.

When **Power Features > Generate** is selected from the component popup, the dialog is only displayed if **Show Generate Power Features dialog** is selected on the **Tools > Options > Assembly > General** dialog. The dialog is not normally displayed if the option is deselected. In this case, the component selection will be filtered automatically to identify which component is the target and which is the source component. However, the dialog will be displayed if a selected component could be either a source or a target.



Select target component(s) — If you display the dialog by selecting a target component (for example a plate), this option will have a  and you will only need to select a source.

If you don't select a source component and therefore have only target components selected, all the components of the active assembly will become source components. In this case, clicking **OK** will generate Power Features for every instance of the target component(s) intersecting with a source component.

Select source component(s) — If you display the dialog by selecting a source component (for example a screw), this option will have a  and you will only need to select a target.

- If you don't select a target component and therefore have only source components selected, all the components of the active assembly will be target components. In this case, clicking **OK** will generate Power Features for every instance of the source component intersecting with another component.
- If you select a source component that does not have Power Features defined, an error message is displayed in the dialog, the component is deselected and the **Swap** button becomes inactive.

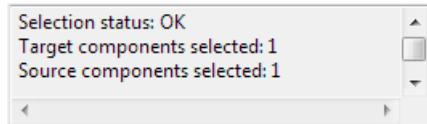
Swap — This option is only active if all selected target components also define Power Features. Selecting this option will have the following effect:

- Existing target components will become source components.

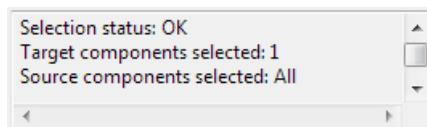
- Existing source components will become target components.

Summary — This displays the **Power Features Summary** dialog so that you can see in more detail how the Power Features will be applied.

Selection information — This panel displays information on the number of target and source components that have been selected. The following information panel shows that one source and one target component have been selected.



If a source component was selected, but no target component, the message shows that all the components will be considered as targets.



OK — Generate the Power Features where the target components intersect with the source components.

Cancel — Close the dialog and to cancel any changes you have made.



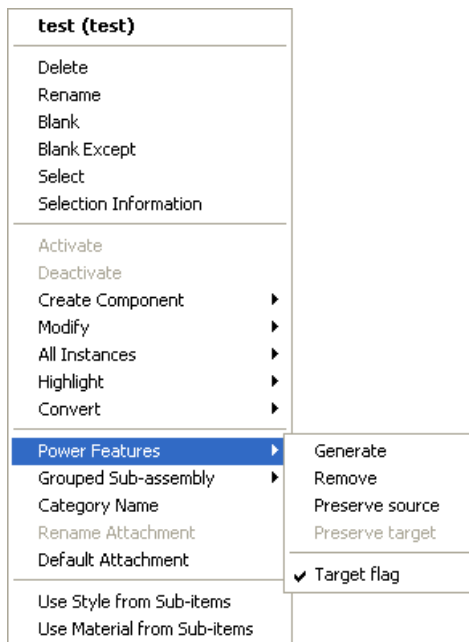
If you wish the Power Features to be generated automatically, do one of the following:

- Select **Generate Power Features** from **Power Features** sub-menu on the assembly popup in the assembly tree.
- Select **Tools > Option > Assembly > General** and set the **Show generate power features dialog** option to **OFF**. This will suppress the display of the **Generate Power Features** dialog.

Preserving Power Features

There are two **Preserve** options on the context menus for assemblies and components. They let you define if Power Features that have been applied to target component(s) are preserved when the source component(s) are deleted.

The following popup shows the options displayed on a component popup in the assembly tree.



Preserve PFs source - If you select a source component and this option, any Power Features that the source component has applied to target component(s) will be preserved if the source component is deleted.

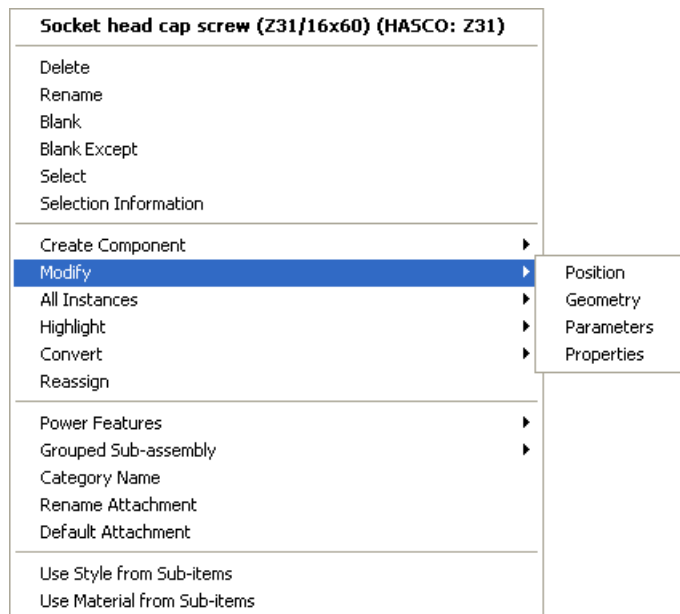
Preserve PFs target - If you select a target component and this option, any Power Features that source component(s) have applied to the target component will be preserved if the source component(s) are deleted.


Setting advanced properties for Power Features

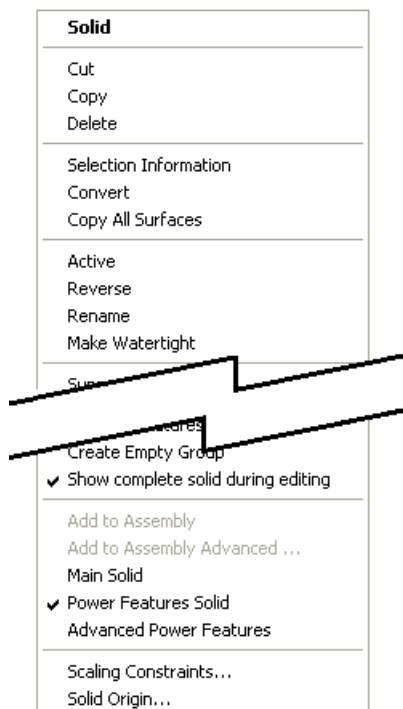
You can change the intersection rules associated with a Power Feature when modifying the geometry of a Power Feature solid.

- 1 Select the required component in the assembly tree.

- 2 Right click the selected component in the assembly tree to display the component popup



- 3 Select **Geometry** from the **Modify** options on the component popup. The component definition tree displays the **Power Features Solid** and **Main Solids** branches.
- 4 Click  to expand the **Power Features Solids** branch.
- 5 Right click the power feature solid to display the solid popup.




- 6 Select **Advanced Power Features** to display the Set advanced property for Power Features dialog (see page 76).
- 7 Use the dialog to change the intersection rules associated with a Power Feature.

8 Click **OK**.



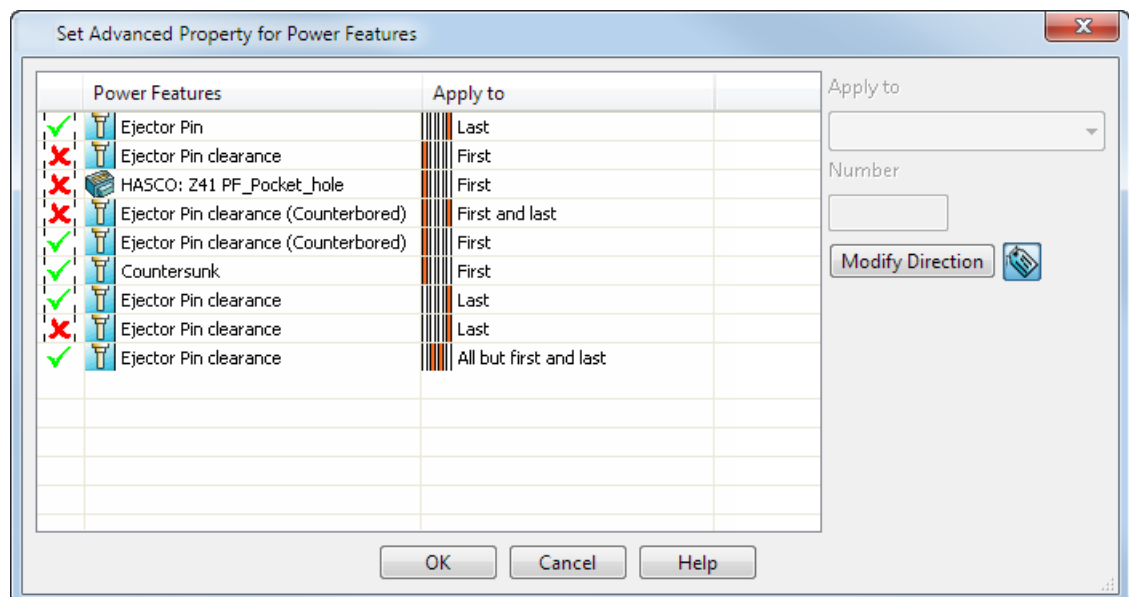
If you have selected a single Power Feature source



component, click  to display the Set advanced property for Power Features dialog (see page 76). This lets you define how the Power Feature is applied and modify the direction of the Power Feature. If you have not selected a single Power Feature source component, an error message is displayed.

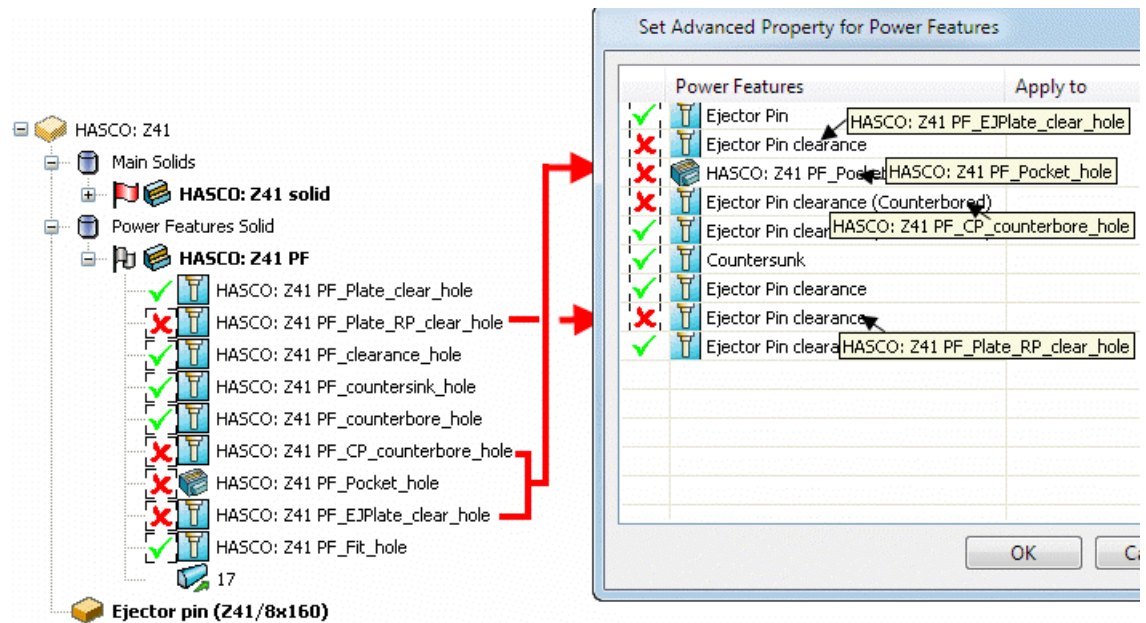
Set advanced property for Power Features dialog

Use this dialog to change the intersection rules associated with a Power Feature.

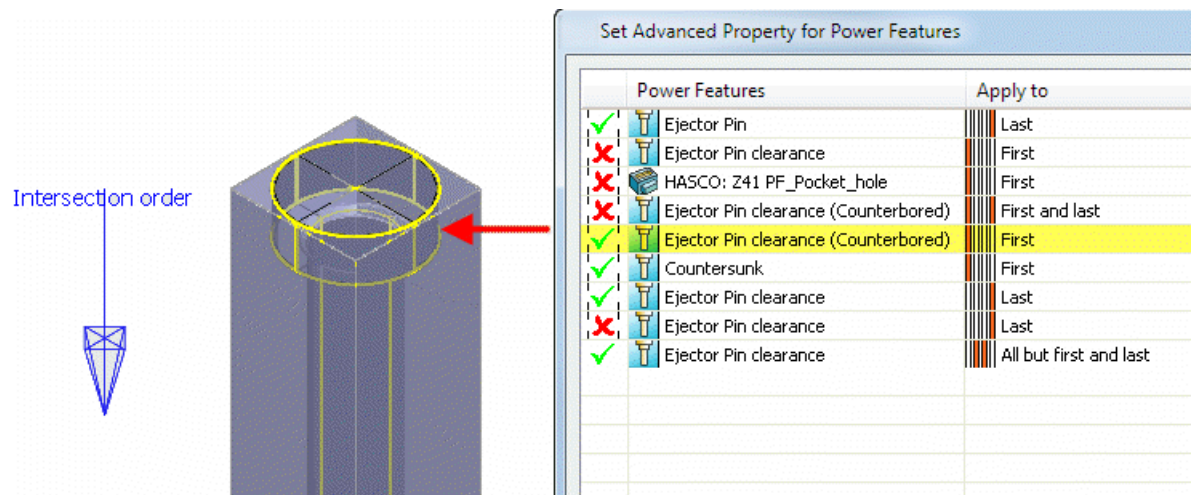


Power Features — This column contains a list of the Power Features for a source component. If you want to change the intersection rules associated with a Power Feature, select the Power Feature from the list.

Power Features that have been suppressed using the **Suppress by parameter** option are displayed as shown below.














An unsuppressed Power Feature that is selected in the list in the dialog is highlighted on the assembly.



Apply to — Each Power Feature stores a rule defining how it should be applied. These rules are based on the order in which the source component (for example, a screw) intersects with target components (for example, plates). So, a screw will apply a counterbored clearance hole to the **First** plate it intersects with.

Each source component has a direction vector used to order these intersections (displayed as a blue arrow on the **Power Features Summary** dialog). You can modify this direction using the **Modify Direction** button on the **Set Advanced Property for Power Features** dialog. The **First** intersection is the nearest point on the source component along this direction, the **Last** intersection is the furthest point.

For a given set of intersections between the source and target components, this column shows how the Power Features would be applied. These options allow you to be more specific about whether or not a Power Feature is copied to a particular target component, based on the intersection order of the target components. The options that may appear in the column are explained in the following table.

Apply to	Action
 All	Copy feature onto all components.
 All but first	Copy feature onto all components except the first one.
 All but first and last	Copy on all target component except first and last
 All but last	Copy feature onto all components except the last one.
 All but number	Copy feature onto all components except the <i>n</i> th one. For example, if Number is 3, the feature will be copied onto all components except the 3rd.
 First	Copy feature onto the first component only.
 First and last	Copy feature onto the first and the last component.
 First and second	Copy feature onto the first and the second component only.
 Last	Copy feature onto the last component only.
 Last and last but one	Copy feature onto the last and the last but one component only.
 Number	Copy feature onto the <i>n</i> th component only. For example if Number is 3, the feature will be copied onto the 3rd component only.

You can change the option that is shown in the Apply to column as follows:

- 1 Click on the row that you wish to modify.
- 2 Select an option from the **Apply to** drop down list. If you select **Number** or **All but number** from the drop down list, the **Number** option will become active.

Number — Enter the number of the component that will be used by the **Number** or **All but number** rules.



Modify Direction — Modify the direction of the Power Feature using the Workplane dialog.

 — Toggles the **Intersection order** label *ON* and *OFF*

Assemblies and Sub-assemblies

Creating a new assembly

Before you can create components, you should create an assembly to hold the components.

- 1 Click  to display the Assembly modelling toolbar.
- 2 Click **Create new empty assembly**  to display the **Create New Assembly** dialog. This is the same as selecting the **Object** menu, followed by **Assembly** and then **Create**.
- 3 Use Create New Assembly dialog (see page 79) to create a new assembly.



*If an assembly does not already exist, it is created automatically when you select **Create Component** from the assembly tree or the component toolbar
It is created with default name **NNNNNN_assembly***

*where **NNNNNN** is the name of model.*

*For example, **NEW_MODEL_1_11_assembly** is the name of the assembly that is created from the model called **NEW_MODEL_1_11**.*

Create New Assembly dialog

This lets you create a new assembly.



The dialog box titled "Create New Assembly" has a blue title bar with a close button. It contains three input fields: "Name of new assembly" with a dropdown menu showing "Unnamed", "Description" with a text box containing "none", and "Category name" with a dropdown menu. Below these fields is a checkbox labeled "Create in a new model" which is unchecked. At the bottom are three buttons: "OK", "Cancel", and "Help".

Name of new assembly - You can enter the name of the new assembly in the text box. The drop down list contains the names of existing assemblies.

Description - A description of the assembly may be optionally given.

Category name - Select a category name from the drop down list of existing category names, or enter a new category name in the text box.

Create in a new model - If ON, the assembly is created in a new model when you click **OK**.

OK - Creates a new assembly and removes the dialog from the screen. The new assembly appears in the assembly tree (see page 9). You can now add components (see page 85) to the assembly.

Editing an existing assembly

As soon as you enter Assembly modelling, all the existing assemblies in the model are displayed in the assembly tree (see page 9).

1 Before you can make changes to an assembly, you must make it the active assembly (see page 80). Only one assembly can be active at a time.

2 Add a component or edit existing components. For further details, see:

Creating components (see page 85)


Adding an axial component (see page 90)

Adding relationships between components (see page 91)

Editing relationships (see page 106)

Editing a component in the assembly (see page 109)

Activating an assembly

You can only edit an active assembly. To make any assembly active, right click its **Assembly** icon  and select **Activate** from the pop up menu.

The components of the newly active assembly are now visible.

Converting components into a sub-assembly

You can create a sub-assembly from selected components by using the Edit menu or the context menus in either the assembly tree or the graphics window. Selected components can be:

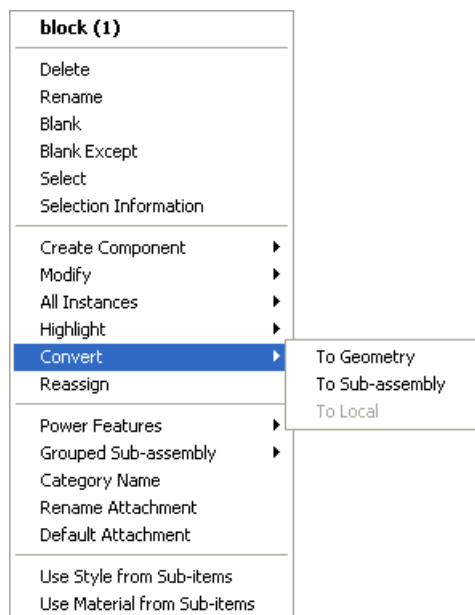
- One or more components.
- One or more sub-assemblies.
- A mixture of components and sub-assemblies.

When converting a group of selected components into a sub-assembly, relationships will, wherever possible, be preserved as follows:

- Relations between the selected components will be preserved.
- Relations will be preserved in cases where the master component is a selected component and the slave component is not a selected component.
- To avoid possible dependency loops, relations will be removed where the slave component is one of the selected components and the master is not a selected component.

Creating a sub-assembly using the assembly tree

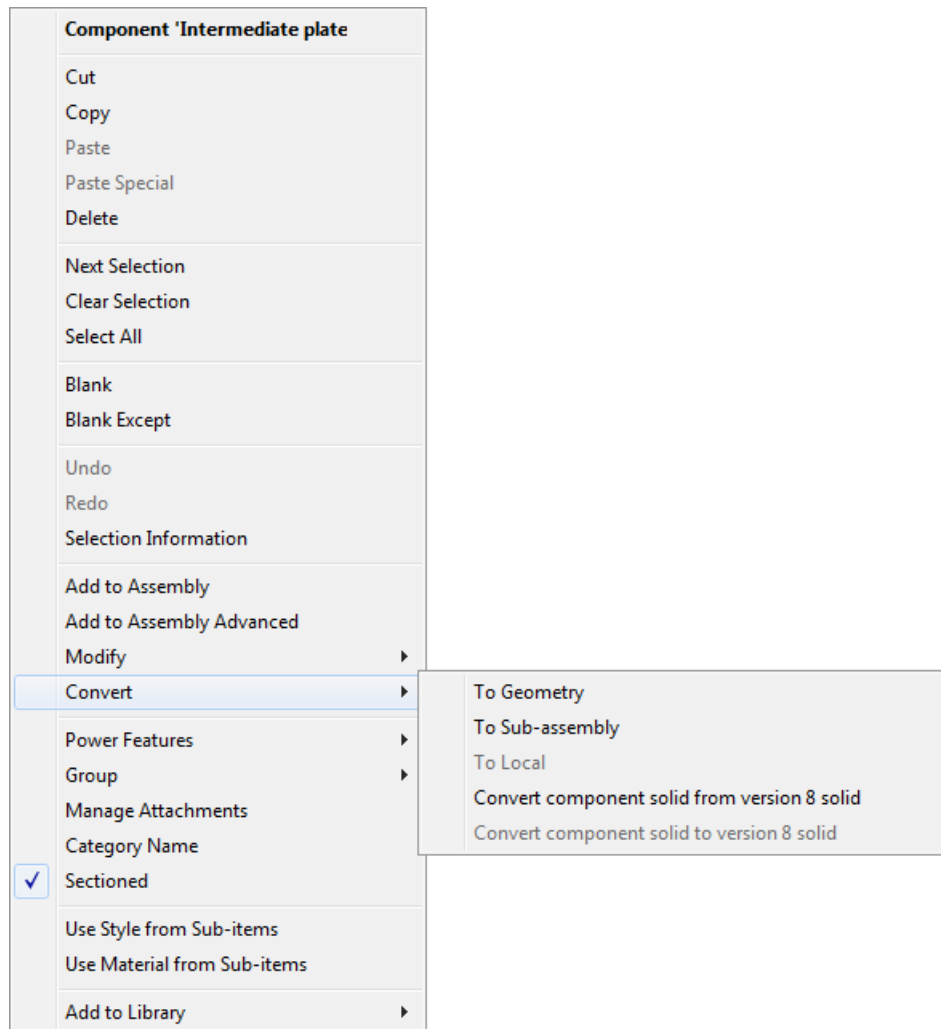
- 1 Using the assembly tree, select the components to be included in the sub-assembly.
- 2 Right click the selection to display the context menu.



- 3 Select Convert.
- 4 Select **To Sub-assembly** from the **Convert** options

Creating a sub-assembly using the graphics window

- 1 Using the graphics window, select the components to be included in the sub-assembly.
- 2 Right click the selection to display the context menu.



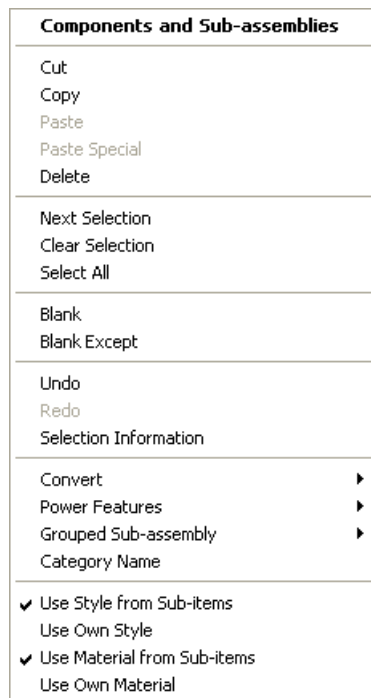
- 3 Select **Convert > To Sub-assembly** from the options

Using Sub-assemblies and Components

You can apply assembly operations to components and sub-assemblies using a context menu.

- 1 Select your components and/or sub-assemblies

- 2 Click the right mouse button over the selection to display the context menu.



How do I make a copy of a component or sub-assembly?

You can copy a component or a sub-assembly using the pop up menu in the graphics window.

- 1 Right click over the component or sub-assembly you wish to copy. Select **Copy** from the pop up menu. The component or sub-assembly will be highlighted.
- 2 Right click and select **Paste**. A copy of the component or sub-assembly will be added to the model.

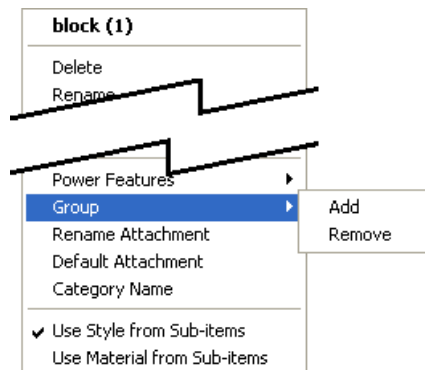
You can also make a copy of a component or sub-assembly by selecting the component and using the **Edit** menu on the toolbar.

Grouping and ungrouping assembly components

You can add an assembly component to a Group and remove an assembly component from a Group using options on the context menu in the assembly tree and the graphics window.

Using the assembly tree menu

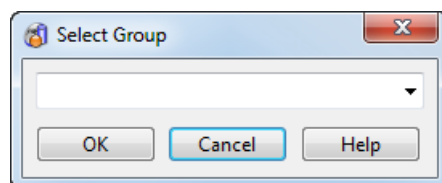
When you right click on an assembly component in the assembly tree, the following context menu is displayed.



Add - Choosing this option will display the **Select Group** dialog.

Remove - Choosing this option will remove the selected component from the group. This option is only active if the selected component has been previously added to a group.

Select Group dialog



- 1 Select an existing Group name from the drop down list.
- 2 Click **OK**. The selected component will be added to Group you selected.

A new item will be added to the assembly tree.

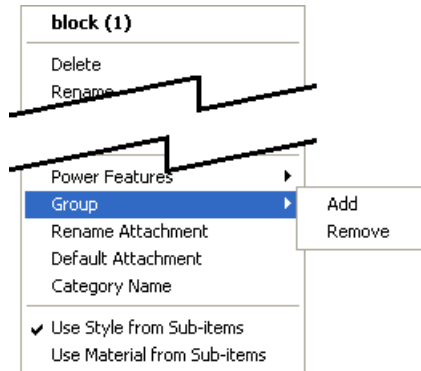


You can also enter a new **Group** name. If this name is not unique within the assembly, the name will be automatically made unique.

Ungrouping a Group

You can ungroup a **Group** as follows:

- 1 In the assembly tree, right click the group to display the context menu.



- 2 Select **Group**
- 3 Select **Remove**. All components are ungrouped from the Group and the Group is removed from the assembly tree.

Renaming a Group

- 1 Select the appropriate **Group** with the right mouse button to display the **Group** popup
- 2 Select **Rename** from the options on the popup
- 3 Enter the new name and click **OK**.

Converting a sub-assembly into a Group




You can convert a sub-assembly into a Group using the assembly tree context menu.


Converting a Group into a sub-assembly


You can convert a Group into a sub-assembly using the assembly tree context menu.


Components


Creating components using the toolbar or menu

- 1 Ensure that you have a model displayed.
- 2 Select **Assembly**  from the main toolbar. The Assembly modelling toolbar will be displayed.
- 3 Select one of the following from the **Create from solids**  flyout or **Add component**  flyout

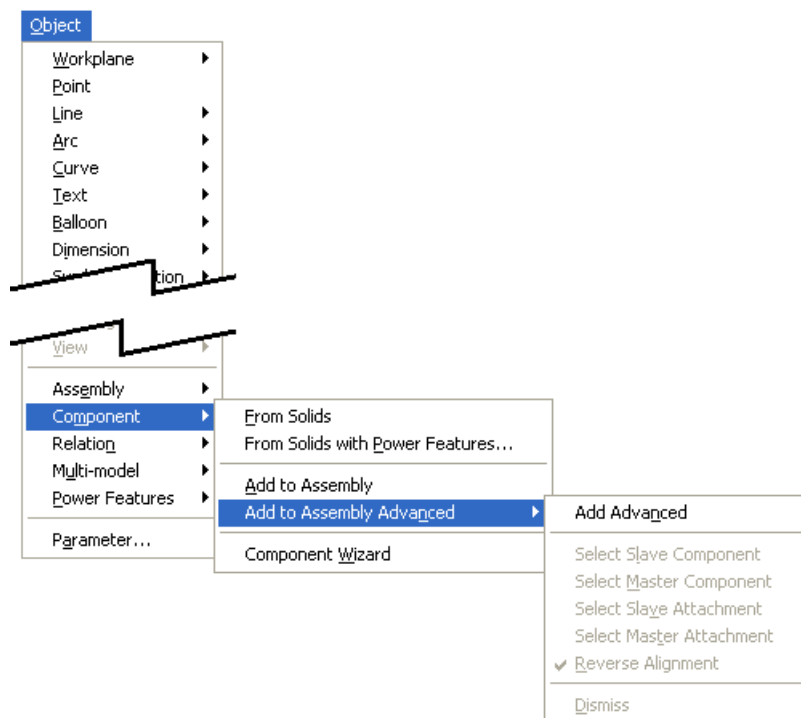
 - Add a component from selected solids (see page 86) (Create from solids flyout)

 - Creating components from selected solids with Power Features (see page 87) (Create from solids flyout).

 - Adding a component using the Intelligent Cursor (Add component flyout)



 - Adding a component with advanced options (see page 90) (Add component flyout)

You can also create components using the **Component** options of the **Object** menu.



Creating components from selected solids



You can create components in the active assembly from selected solids and registers their component definitions in one step.

- 1 Click  to display the Assembly modelling toolbar.
- 2 Make sure you have an active assembly.
- 3 Select some solids.
- 4 Click  (Component from solid flyout).

The selected solids become components and remain in their original positions. Component definitions are automatically created for the components. The names of the components and their definitions are the same as their original solid.

Creating components from selected solids with Power Features



This lets you define a new component from selected solids with Power Features. This is the same as the option **Add to assembly with Power Features** that is available from the solids context menu.

- 1 Click  to display the Assembly modelling toolbar.
- 2 Make sure you have an active assembly.
- 3 Select some solids with Power Features.
- 4 Click  (Component from solid flyout).

The selected solids become components and remain in their original positions. Component definitions are automatically created for the components. The names of the components and their definitions are the same as their original solid.

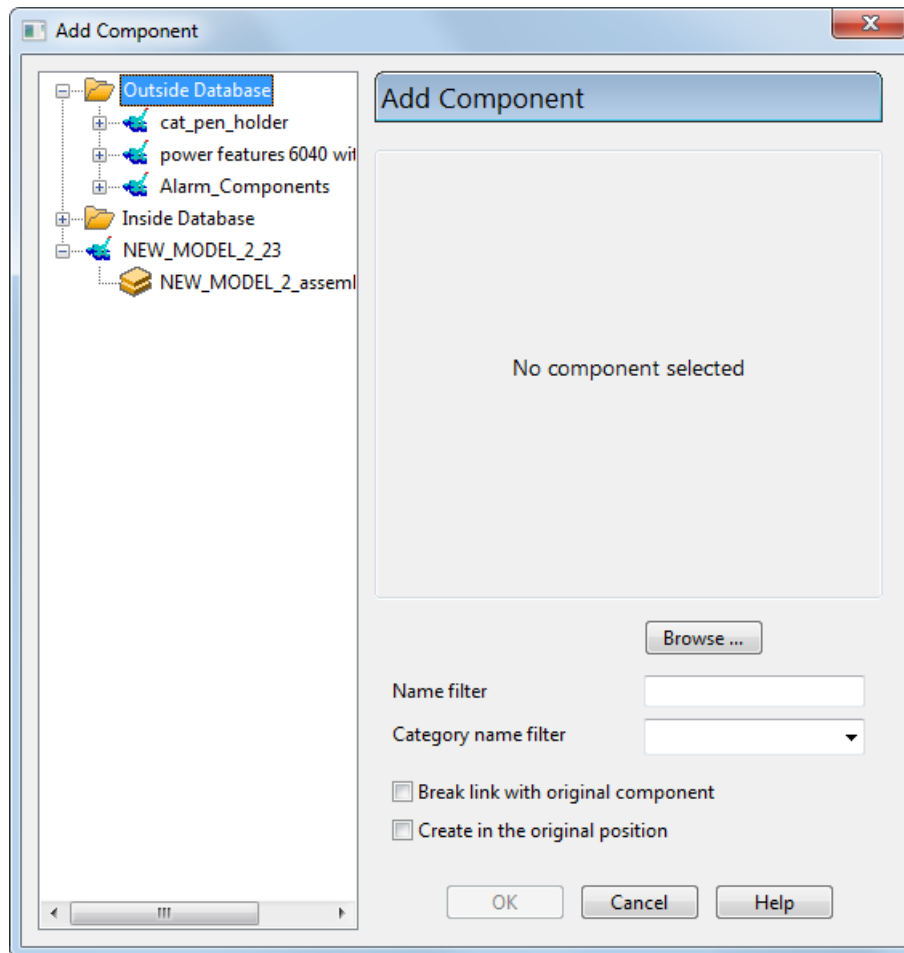
Adding components

You can add components using the Intelligent Cursor.

- 1 Click  to display the Assembly modelling toolbar.
- 2 Click  (Add component flyout). The **Add Component** dialog is displayed.
- 3 Use the Add Component dialog (see page 88) to select a component to add to the model.
- 4 Click **Apply**. The component is attached to the cursor. You can position the component and it is automatically aligned to the geometry beneath the cursor.

Add Component dialog

Use this dialog to add the selected component to the assembly. The tree in the left-hand pane of the dialog box reflects the assembly models you have open and the current working model.



Use the tree to select a component or assembly from the current model.

Browse — Use this button to locate components to add to the model. These components must be in inactive unopened models.

Name filter — Enter a string to filter the list of models that appears in the **Inside Database** and **Outside database** folders.


To display models that have a name starting with *SHOE*:


- 1 Ensure that **Inside Database** and **Outside Database** are expanded
- 2 Enter **SHOE*** in **Model name filter**.
- 3 Press *Enter*.

Category name filter — You can use this drop down list to filter component definitions by category name. The drop down box contains all the category names found in the active model.

When you have selected a category name, the tree browser will show only those definitions where the category name matches the selected filter.

Break link with original component — Use this option to retain or break the link with the original component definition when you add a component from the Component Library or another model.

If deselected, the component definition is added to the assembly tree, but a copy of the component definition is not stored with the assembly. The 'linked' component definition is indicated by the  in the assembly tree.


If selected, a copy of the component definition is created and stored with the assembly. The local component definition is indicated by the standard component definition  icon in the assembly tree and it can then be used in the same way as any other component definition. If the component definition is modified in the originating assembly or in the Component Library, use the Refresh imported

component definitions  button to update the component definition in the current assembly.



*When the dialog is displayed, the initial status of this option is controlled by the **Create local copy of component definition** on the **General** page of **Assembly options**. If you change the status of the option within the **Add Component** dialog, the new status only applies for the component you are currently adding.*

Create in the original position — If this option is selected, the component is located into the position where the geometry was registered. The option is only active when you add a component

using **Add component on Intelligent Cursor**  button.




***Create in the original position** is useful for component positioning in a multi-model assembly. If the option is selected, the new component is inserted in the same position as in its original model.*

Apply — Adds the component to the assembly. The component is added outside the bounding box of the existing components in the assembly. If the component is the first one added to the assembly, it is in the same position as its original solid.



If you are adding a parameterised component (see page 118) to the assembly, the **Parameter List Editor** dialog will be displayed. You can use this to edit the values of the parameters. The dialog remains on the screen, so that you can add more components.

OK — Removes the dialog from the screen.




If you click **Undo**  after adding components using the **Add component** dialog, all the components that you added using the dialog will be deleted.

Adding a component with advanced options

- 1 Click  to display the Assembly modelling toolbar.
- 2 Click  (Add component flyout). The **Add Component** dialog is displayed.
- 3 Use the Add Component dialog (see page 88) to select a component to add to the model. The component to be added to the assembly must have an axial attachment.

Adding a component using the assembly tree or component library

You can add an existing component to an assembly using the assembly tree or the component library

- 1 Double click on a component icon in the assembly tree or the thumbnail in the component library
- 2 The cursor will change shape  and the component graphics will appear on the edge of the model window nearest to the cursor. Position the components using the cursor.

Creating components using the assembly tree

You can create a component from a solid using the assembly tree.

- 1 Right click the solid in the assembly tree.
- 2 Select **Add to Assembly** from the context menu.

If an assembly does not already exist, it is created automatically when you select **Add to Assembly** from the context menu or

Create components from selected solids  (Add components flyout)



The new assembly is created with default name
NNNNNN_assembly

where **NNNNNN** is the name of model.

For example, **NEW_MODEL_1_11_assembly** is the name of the assembly *that is created from the model called NEW_MODEL_1_11*

Adding relationships between components

You can add relationships to components using the **Create Relations** dialog.

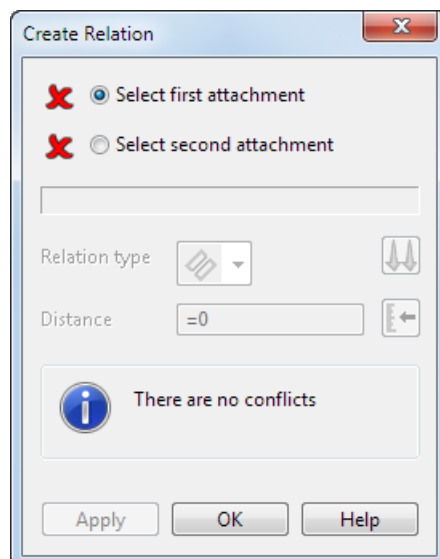
- 1 Make sure you have at least two components on the screen.
- 2 Click  to display the Assembly modelling toolbar.
- 3 Click **Add relation using attachments**  on the **Relations** flyout to display the **Create Relations** dialog.
- 4 Use the Create Relations dialog (see page 91) to create relationships between two components in the assembly. The new relationship is added to the assembly tree. For further details, see What is the assembly tree? (see page 9)

Creating relations using attachments

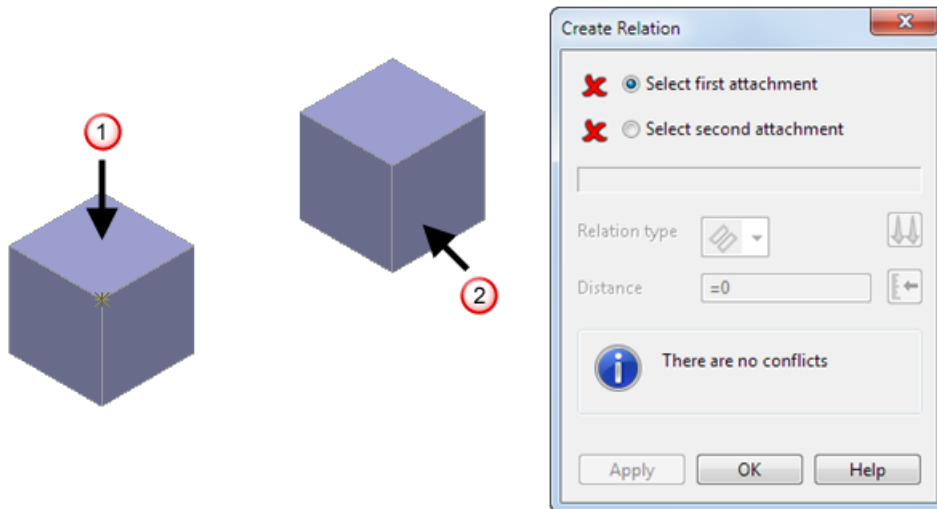
Use the **Create Relations** dialog to create relations between components. **Create Relations** replaces the **Relations** toolbar.





- 1 Click  from the **Relations** flyout on the **Assembly** toolbar.

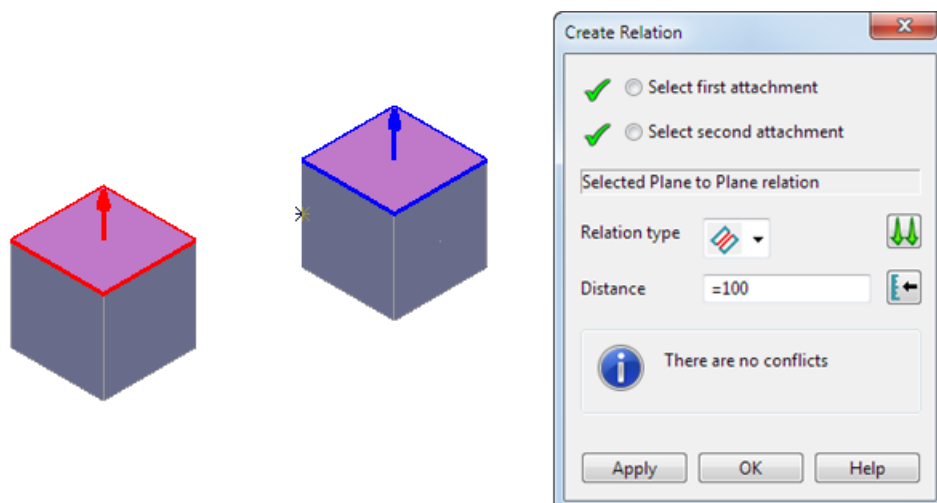
The **Create Relation** dialog is displayed.





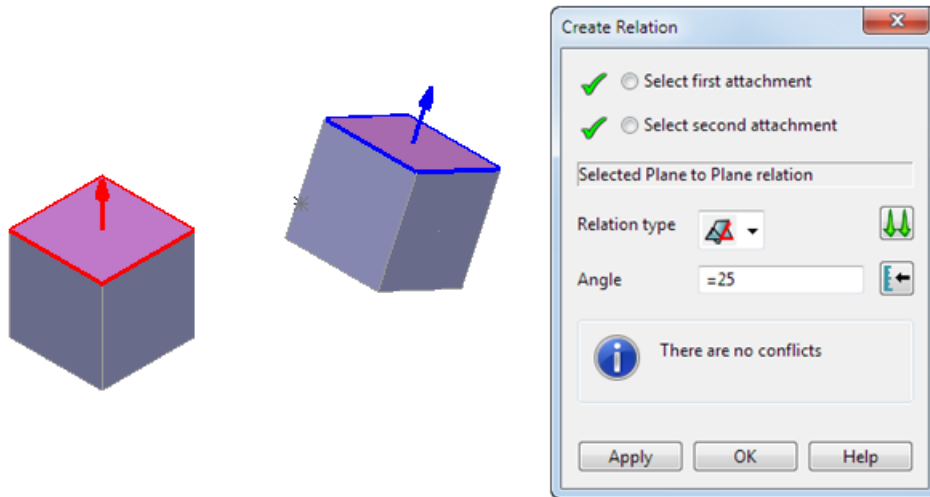
The following simple example illustrates the options on the dialog:




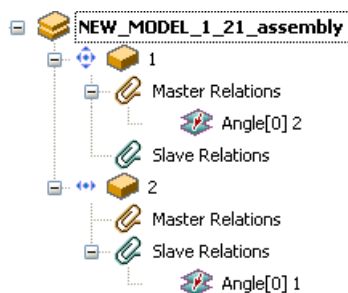
- 2 Select the first attachment ①. This is the master attachment. The cross  icon changes to a tick  icon..
- 3 Select the second attachment ②. This is the slave attachment. The model and the dialog are updated as follows:
 - A **Coincident Plane to Plane relation** is created between the two components. *Selected Plane to Plane relation* is displayed in the dialog and  is displayed as the **Relation type**.
 - The slave component ② is rotated to reflect the relationship you have defined.
 - The information panel is updated to indicate if there is a conflict.
- 4 Enter a **Distance** of **100** or click  and use the slider in the **Angle/Distance Scaler** dialog (see page 97) to specify the distance. The model will be updated.



- 5 Click  to display **Relation type** options.
- 6 Select  to change the **Relation type** to **Angular**.
- 7 Enter **25** as the **Angle**.



- 8 Click  to **Anti-align** (see page 97) so that the slave attachment lies in the opposite direction to the master attachment.
- 9 Click **Apply** to create the relation you have just defined. The dialog is still displayed to allow you to specify additional relations in a more complicated model.
- 10 Click **OK** to finish creating relations. The relationship that you have created is added to the assembly tree.






Relations are added to the tree when the second attachment is selected. If you realise you have made an incorrect selection before you click Apply, do one of the following:

- *Close the dialog, delete the relation from the assembly tree and then create the relation using the correct selection.*
- *With the dialog still displayed, use Select first attachment and Select second attachment to modify the relation.*

Selecting a Relation type

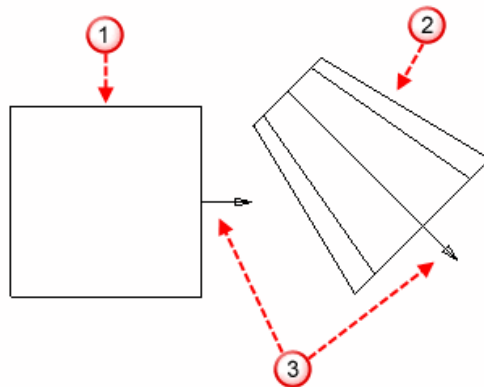
The **Relation type** determines the relationship between the attachments on the two components in the assembly.

There are three options:

- **Coincident** 
- **Angular/Parallel** 
- **Perpendicular** 

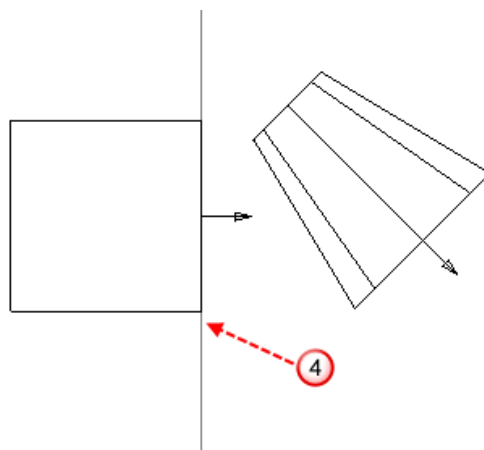
In the 2D example below,


- ① is the master component
- ② is the slave component.
- ③ shows the selected plane attachments.



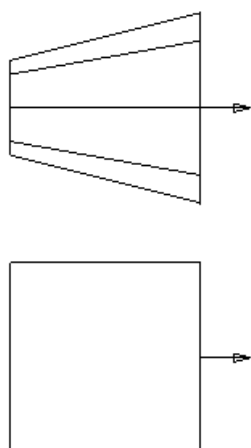
The slave component is positioned using the attachment of the master component.

In the example, the master has a plane attachment. The plane of the attachment is used to position the slave component. The line representing the plane is indicated by ④



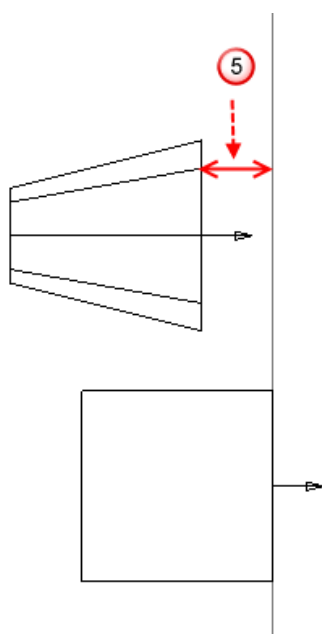
Coincident  - The slave is positioned so that its attachment is coincident with the attachment of the master.


In the example, the slave is positioned so that its plane attachment lies in the same plane as that of the master.



You can position the slave such that its attachment is at a distance from the attachment of the master. Enter a **Distance** or use the Angle/distance scaler (see page 97) to define a distance.

In the example, the slave is positioned so that its plane attachment is at a distance from the plane attachment of the master. The distance is indicated by 5.



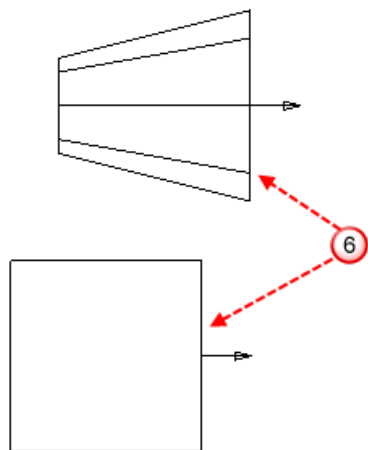
Angular/Parallel  - The slave is positioned such that its attachment is at an angle to the attachment of the master. If the angle is 0, the slave is positioned so that its attachment is parallel to the attachment of the master. Enter an Angle or use the Angle/distance scaler (see page 97) to define an angle.


In the example, the slave is positioned so that its plane attachment is parallel to the plane attachment of the master. The position of the two parallel planes is indicated by ⑥.

Angular/Parallel relation type may be set for the following types of relations:

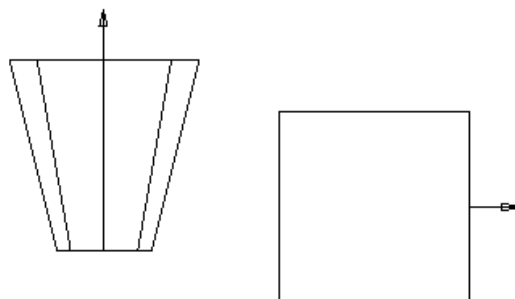
- plane to plane
- plane to line
- line to plane
- line to line

The actual distance between the planes is undefined at this stage and needs to be defined by another relation type.



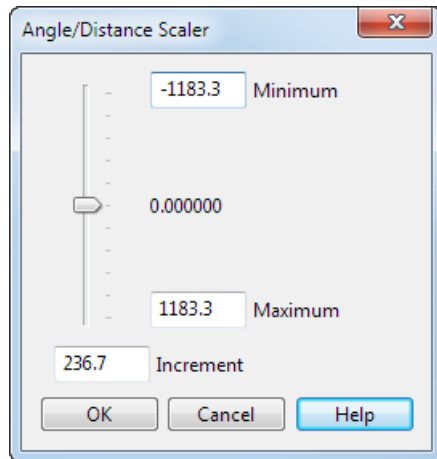
Perpendicular  - The slave is positioned such that its attachment is perpendicular to the attachment of the master.

In the example, the slave is positioned so that its plane attachment is perpendicular to the plane attachment of the master.



Angle/distance scaler

Use the **Angular/Distance Scaler** dialog to dynamically change the angle or distance of the relationship between the two components.




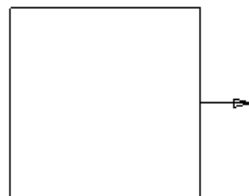
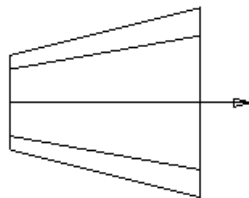
Adjust the following to change the angle or distance between the two components:


- Move the slider to change the angle or distance between the two components. The scale on the slider increases or decreases using the value in the **Increment** box.
- Enter a different **Increment** value to change the increment of the scale.
- Enter alternative **Maximum** and **Minimum** values for the slider.

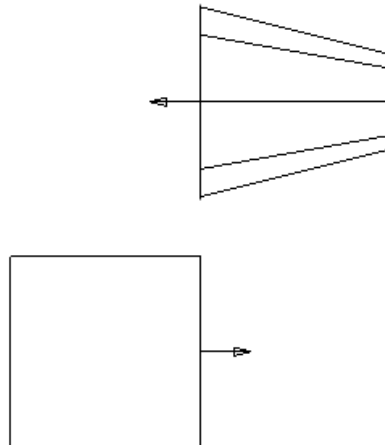
Orientation

This determines how the two attachments are aligned. There are two options:

- **Align**  - The attachment of the slave lies in the same direction as the master's.



- **Anti-align**  - The attachment of the slave lies in the opposite direction as the master's.

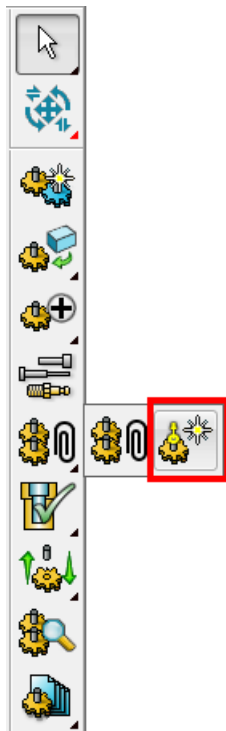


Managing attachments

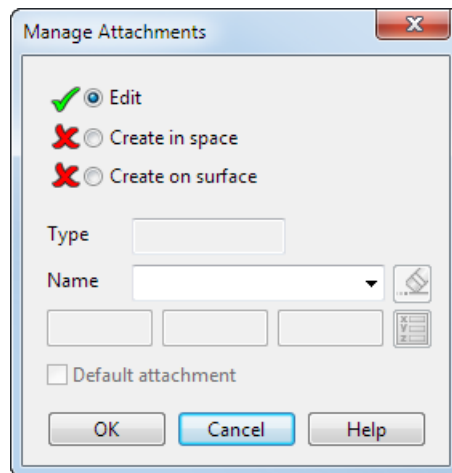
Use the **Manage Attachments** dialog to:

- edit existing attachments.
- create new attachments.

With a component selected, click  from the **Relation** flyout on the **Assembly** toolbar. Alternatively, select **Manage Attachments** from the component context menus.



The **Manage Attachments** dialog is displayed.



Editing existing attachments

Edit is selected by default. This lets you:

- rename an attachment by selecting the existing name from the drop-down list and enter the new name.
- set the selected attachment as the **Default attachment**.

Creating attachments

Use the **Manage Attachments** dialog to create attachments in one of the following ways:

- Creating an attachment in space is useful when you are trying to position a component and there is no other geometry to attach to.



Click **Create in space** and click the required position. The attachment is created at the specified position. The attachment position will not update if the component geometry is modified.

- Creating an attachment on surface is useful when you are trying to position a component and a default attachment of plane, point or line isn't available. **Create on surface** lets you attach to a specified position.

To create an attachment on a surface, click **Create on surface** and click anywhere on the surface. The attachment is linked to the underlying surface parameters and will update if the geometry is modified.

When you have created the attachment, the **Edit** option is selected on the dialog. Use the following options to modify the attachment:

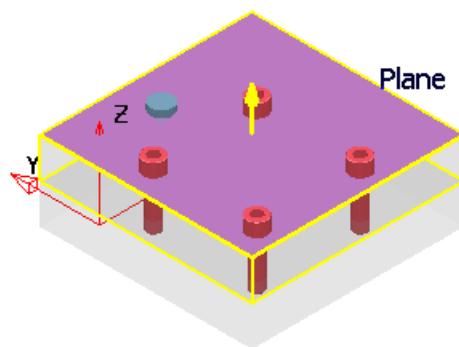
- By default, the new attachment is named *Unnamed*. Enter a new name as required.

- Edit the **X**, **Y** and **Z** coordinates or click  to enter the coordinates using the **Position** dialog.
- Click  to remove the attachment that you have just created.
- Click **Default attachment** to make the new attachment the default.

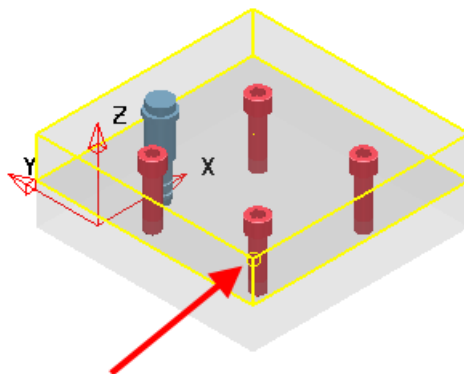
Displaying the attachment points

Add relationships to components by dragging the components. The attachment points are displayed by moving the mouse over the model as follows:

- 1 Select the component you want to be slave component
- 2 Move the mouse over the component to highlight the plane/axis attachments.



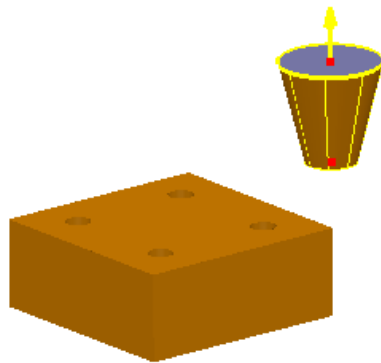
- 3 To use a point attachment, click the component again and move the mouse over the component to highlight the point attachments.



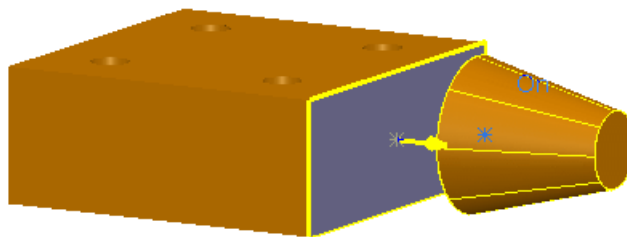
Adding relationships using drag and drop

Add relationships to components by dragging the components and using the attachment points to create a relationship between two components.

The example below uses plane/axis attachments. The face is coloured blue and a large yellow arrow indicates the attachment.

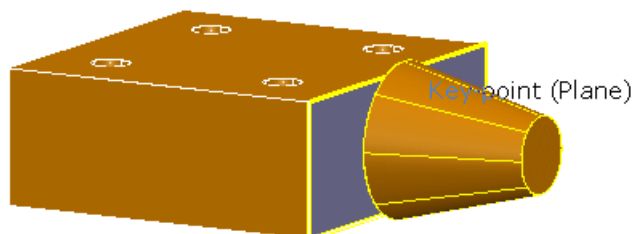


- 1 Drag this attachment towards another component. This shows the attachment on the second component (master component)



The face is coloured blue and a large yellow arrow indicates the attachment. This confirms the correct attachment is under the cursor.

- 2 Release the first attachment onto an attachment on the second component.




The first component is now related to the second component. If you release the mouse away from the second object, no relationship is created and the slave component will return to its original orientation.

The new relationship is added to the assembly tree.

For further details, see What is the assembly tree? (see page 9)

Displaying the properties of a component in the model

To display the properties of a component in your model,

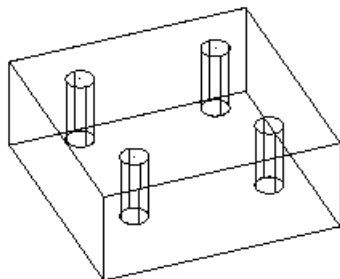
- 1 Select **View > Windows > Component Library** to display the Component Library for your model.
- 2 Right click the component in the Component Library to display the Component popup.
- 3 Select **Modify - Properties** from the component popup.
- 4 Ensure that you have at least one property defined for the component.
- 5 Click on the text creation  flyout and select the direction of the text. The properties of the component are attached to the cursor. The format of the text is defined on the Options - Assembly - General dialog.
- 6 Click near the component to display the text at that location.



Inserting axial components into holes

Use this option to insert axial components into holes. Axial components are ones such as ejector pins, bolts and screws. The command looks for plane and axis attachments on the new component. Typically, this would be the centre of a screw head or ejector pin.

A slave component is inserted onto a master component using plane and axis attachments on the components.

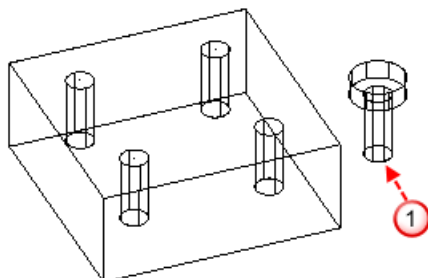
- 1 Make sure you have at least one component on the screen. The following component will be used to demonstrate how to insert axial components.




- 2 Click  to display the Assembly modelling toolbar.
- 3 Click  (Add component flyout). The **Add Component** dialog is displayed.
- 4 Use the Add Component dialog (see page 88) to select a component to add to the model. The component to be added to the assembly must have an axial attachment.

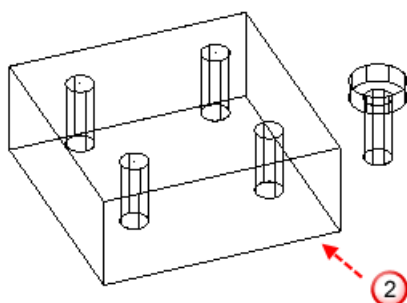
The selected component is added outside the bounding box of the existing components in the assembly. The component is added as a *slave* and is coloured green (using the default colour scheme). The Insert axial component toolbar (see page 105) is displayed.


- 5 Select the component ① as the *slave*.



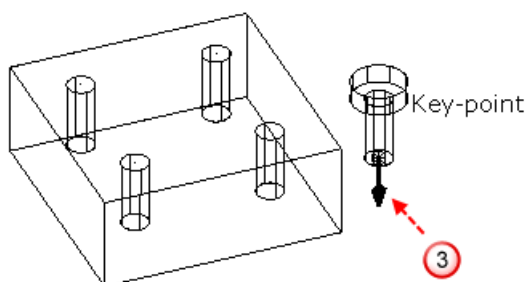
You can change the slave component if you want. To do this, click  on the **Insert axial component** toolbar to display the **Add Component** dialog. Use the dialog to select another component.

- 6 Select an existing component ② as a *master*.



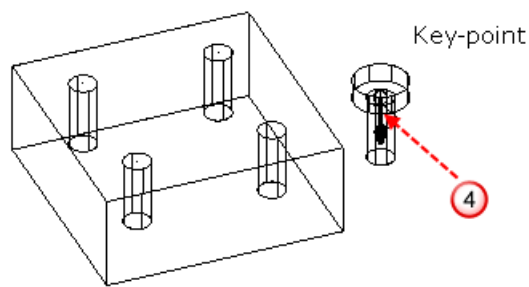
You can change the master component if you want. To do this, click  on the **Insert axial component** toolbar and then select another component.


- 7 Hover the mouse over the slave component to see its valid attachments such as ③ below.





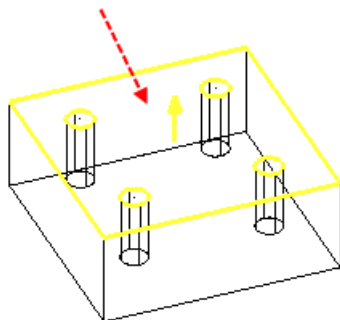
Only combined attachments are visible. These are attachments where plane and axis attachments meet.

- 8 Click the attachment  on the slave component.




To change selected slave attachment, click  on the **Insert axial component** toolbar and then select another attachment on the slave.




- 9 Use the **Distance** box to set a distance between the master and the slave components.
- 10 Use  or  to reverse the current alignment of the attachment on the slave component with the master component.
- 11 Hover the mouse over the master component to see the valid attachments.




By default, two types of attachments are visible:

- **plane**
- **combined.** Combined attachments are usually where holes exist in the master component.

You can use  on the **Insert axial component** toolbar to only highlight the selected type of attachment on the master component. This menu contains the following options.

-  Highlight either combined or plane attachments
-  Only highlight combined attachments
-  Only highlight plane attachments

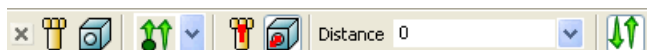
- 12** To add the slave component, click an attachment on the master component. The technique you use is slightly different depending on whether you click a plane attachment (see page 105) or a combined attachment (see page 106).

To change selected master attachment,, click  on the **Insert axial component** toolbar and then select another attachment on the master.

- 13** To exit insert axial component mode, click **Select** .

Insert axial component toolbar

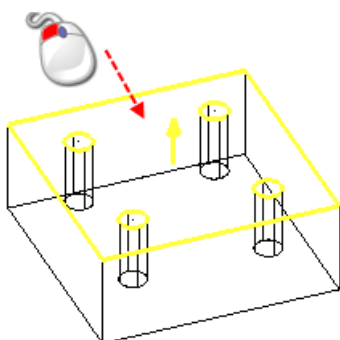
You can use this toolbar to change your component and attachment selections while you insert axial components.



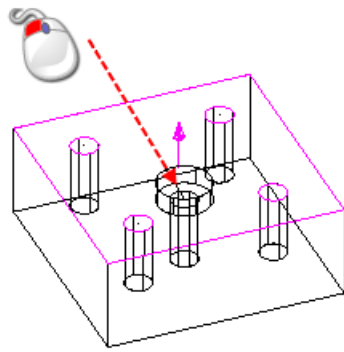
- The toolbar will automatically select its tools from left to right. As you select the required objects, the next tool is automatically clicked, ready for you to make the next selection.
- You do not need to select buttons on the toolbar unless you want to change any of its options.
- If you change any option, the toolbar continues from that option.

Plane attachment

- 1** Click a plane attachment to select the plane in which the slave component will be attached.



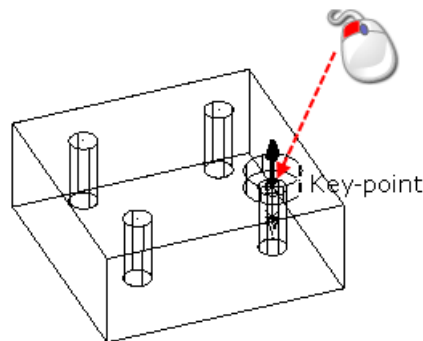
- 2 Click a position on the plane to position the slave component.



This creates a plane to plane relationship between the slave component and the master component.

Combined attachment

- 1 Click a combined attachment on the master component to fix the slave component onto attachment.



The slave component will have a line-to-line and a plane-to plane relationship with the master component.

Editing relationships using the Relation context menu

Use the **Relation** context menu to edit or delete relationships.

In the assembly tree, right click one of the following to display the **Relation** context menu.

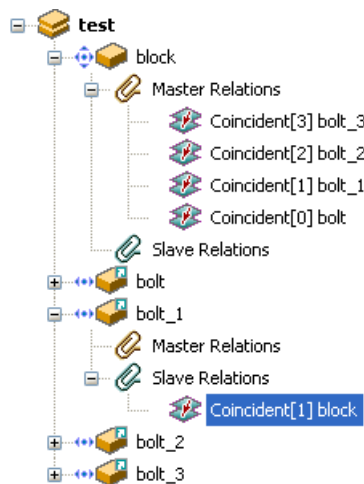
- **Master Relations**
- **Slave Relations**
- the relation (for example, **Coincident**, **Perpendicular**)

The options displayed on the context menu reflect the item you select.

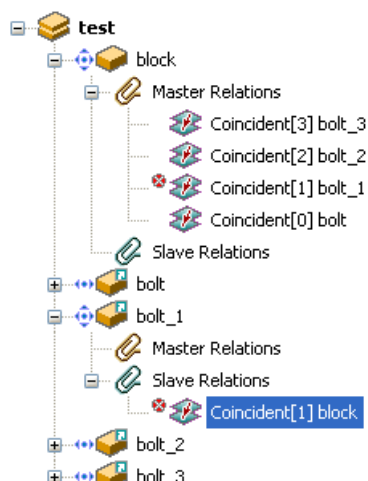
Relation
Delete
Selection Information
Edit
Suppressed

The **Relation** popup will display a combination of the following options:

- **Create** - Select this option to display the Create relations dialog (see page 91).
- **Delete** - This deletes the relationship.
- **Selection Information** - This displays the **Selection Information** dialog that gives details of the selected objects and relationships.
- **Edit** - This displays the Relationships toolbar. You can use this to edit the master and slave relationships.
- **Suppressed** - If selected, this removes the relation between the two components.

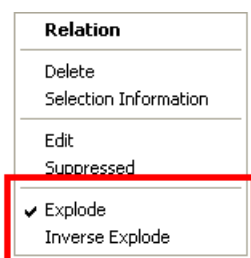


✖ appears next to the relation to indicate that it is suppressed. ✖ is also placed next to its corresponding relation.

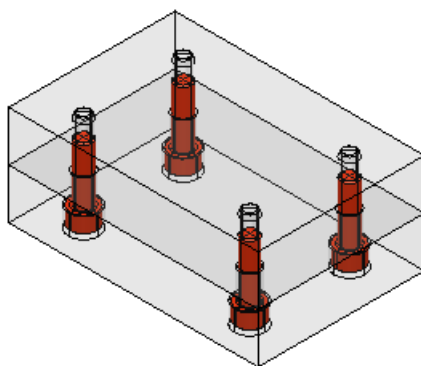


To add the relation back again, right click the relation again in the assembly tree and select **Suppressed** from the pop up menu.


- **Explode** and **Inverse Explode** let you include or omit the current relation from an explode operation and switch direction of exploding.




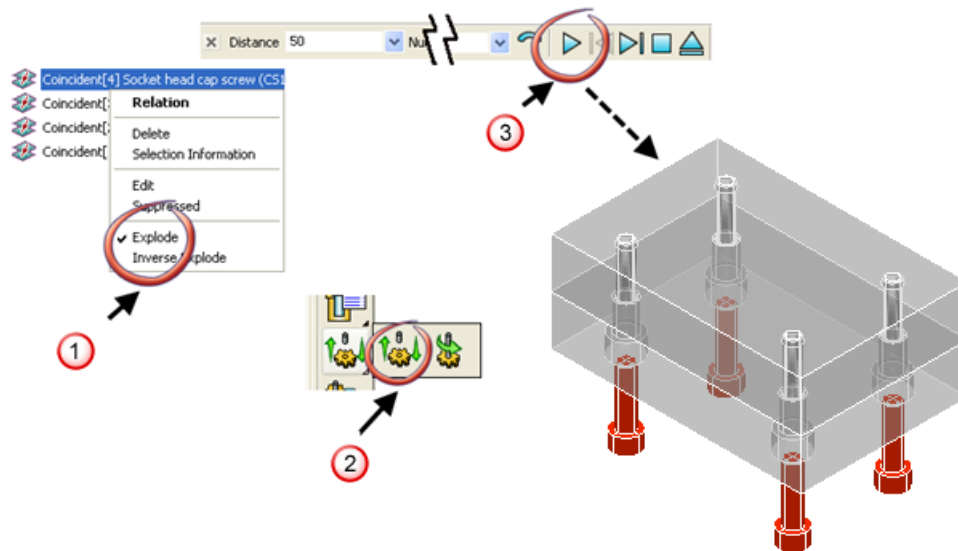
The model below can be used to show how these options work.





Explode - Switches the simulation functionality *ON* and *OFF*.

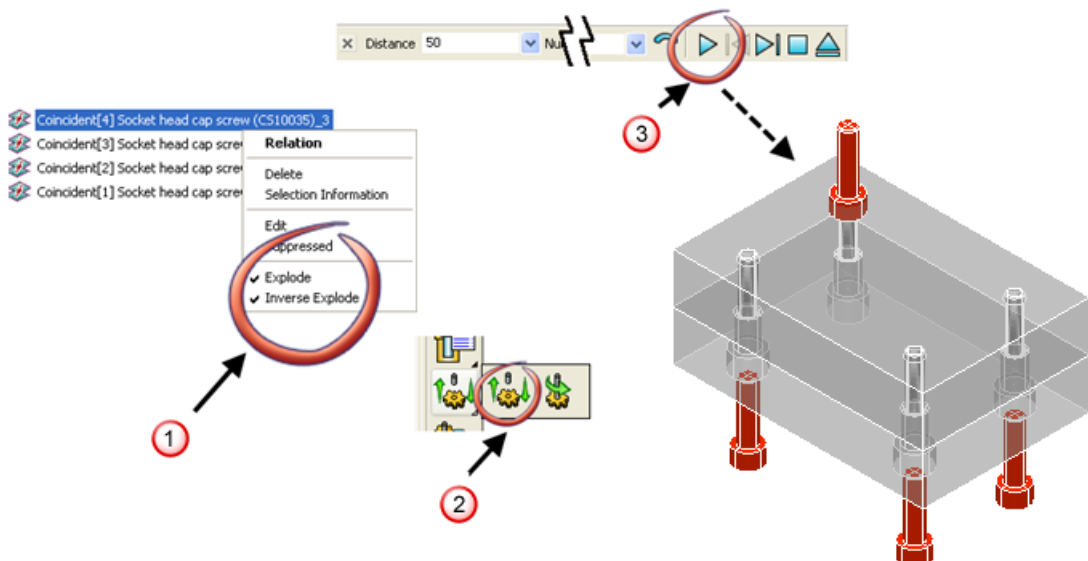
- 1 Select **Explode** from the **Relation** popup. ①
- 2 Select  (assembly toolbar) to display the **Explode** toolbar. ②

- 3 Select  from the **Explode** toolbar, to run the explode operation. ③



Inverse Explode - Inverse Explode is only available when **Explode** is **ON**. If **Inverse Explode** is set to **ON**, the component will explode in the opposite direction to the default.

- 1 Select **Explode** and **Inverse Explode** from the **Relation** popup. ①
- 2 Select  (assembly toolbar) to display the **Explode** toolbar. ②
- 3 Select  from the **Explode** toolbar, to run the explode operation. ③

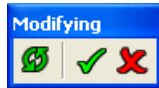





Editing a component in the assembly

You can modify the geometry of a component using standard modelling commands.

- 1 Right click the component either in the graphics window or the assembly tree and select **Modify** from the pop up menu to display the options.
- 2 Select **Geometry** from the **Modify** options. The component is converted back into a solid and you are in modelling mode, ready to modify the component.

The **Modify** toolbar is displayed.



- 3 Modify the solid as appropriate. When you modify all instances of a component, you will only see the changes in the current component.
- 4 Click  to preview the changes in all instances.
If you don't want to store the modified solid as a component, click  on the **Modify** toolbar.
- 5 Once you have finished modifying the solid, click  on the **Modify** toolbar.

The solid converts back into a component.



*1 - When you **Modify** a component, you can add or remove a Main Solid or a Power Feature Solid using the context menu of the solid in either the graphics window or the assembly tree.
2 - If the component you are editing is parameterised, you can change the value of any parameter that is independent. For further details, see How do I edit a parameterised component? (see page 119).*

Repositioning a component

You can reposition any component in an active assembly in one of the following ways:

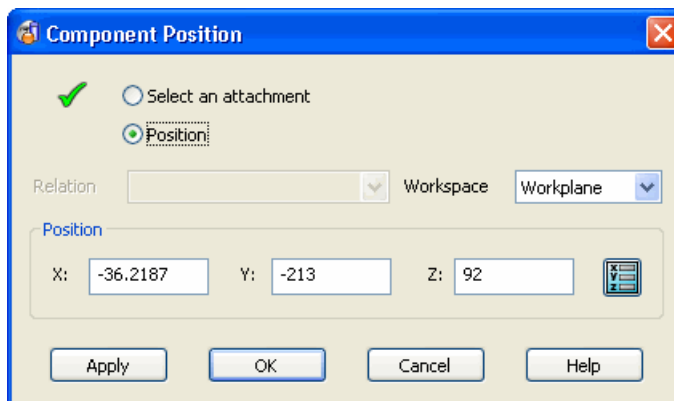
- Selecting the attachment of a component and moving the component.
- Right click the component and select **Position** from the **Modify** options on the pop up menu. Use the Component Position dialog (see page 111) to reposition the component.
- Double click the component. Use the Component Position dialog (see page 111) to reposition the component.

- Right-click the component in the assembly tree and select **Position** from the **Modify** options on the pop up menu. Use the Component Position dialog (see page 111) to reposition the component.
- The 'drag and drop' technique can be used to reposition a component.
 - 1 Move the cursor over the component to highlight an attachment
 - 2 Drag the component to the required position.

As you move over the model, the component will align itself to other geometry it is dragged over. Dropping the component on another component will automatically create a relation.
- You can use **Quick rotate** (hold down **R** and rotate the cursor wheel) whilst dragging a component. The component will revolve around the attachment axis.

Component Position dialog

Use this to move a component within an assembly.



Select an attachment - To reposition a component relative to one of its attachments, select this option and then select an attachment on the component.

The selected attachment will be used as an origin for repositioning. The **X** changes to **✓** to indicate an attachment is selected.

Position - Select this option to edit the position.

Relation - This menu is only available if all of the following conditions are true:

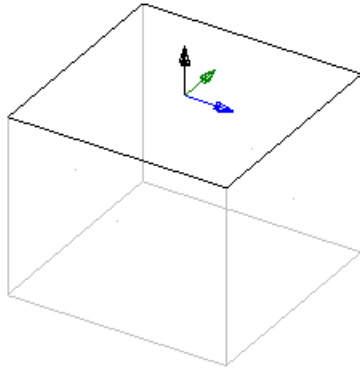
- No attachment is selected.
- The component is a slave.
- The component has either planar-to-planar or planar-to-point relationships, where the planar attachment is on a master component. These relationships appear in the **Relation** menu.

When you select a relationship, the attachments on the component and its master component become highlighted.

You can use this dialog to reposition the slave component on the planar attachment of the master component.


Workspace - Select a workspace.

- **Relative** - (Only available if a relation is selected). The coordinates are measured from the origin of the plane. This origin is marked as shown below.



- **World** - The coordinates are measured relative to the world workspace.
- **Workplane** - The coordinates are measured from the origin of the active workplane.

Position - Enter new coordinates using one of the following methods:

- Click a new position on the model.
- Enter new **XYZ** coordinates directly in the dialog.
- Click  to display the Position dialog to use position entry tools.

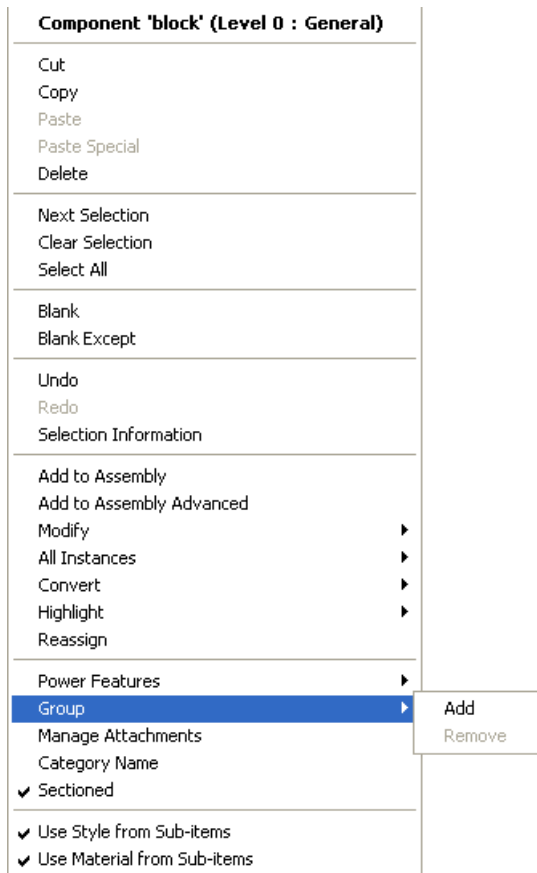
Apply - Apply the changes. The dialog will still be displayed, so you can reposition another component.

OK - Accepts the position and closes the dialog. If any relation conflicts with the new position, it will be deleted.

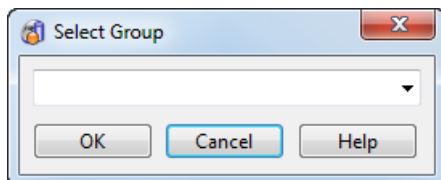
Cancel - Cancels the changes and closes the dialog.

Grouping components

When you right click an assembly component in the graphics window, the following context menu is displayed.



Add - Select this option to add one or more selected components to a Group, using the **Select Grouped Sub-assembly** dialog.



Select an existing Group name from the drop down list and click **OK**. The component will be added to Group you selected.

A new item will be added to the assembly tree.





You can also enter a new Group name. If this name is not unique within the assembly, the name will be automatically made unique.

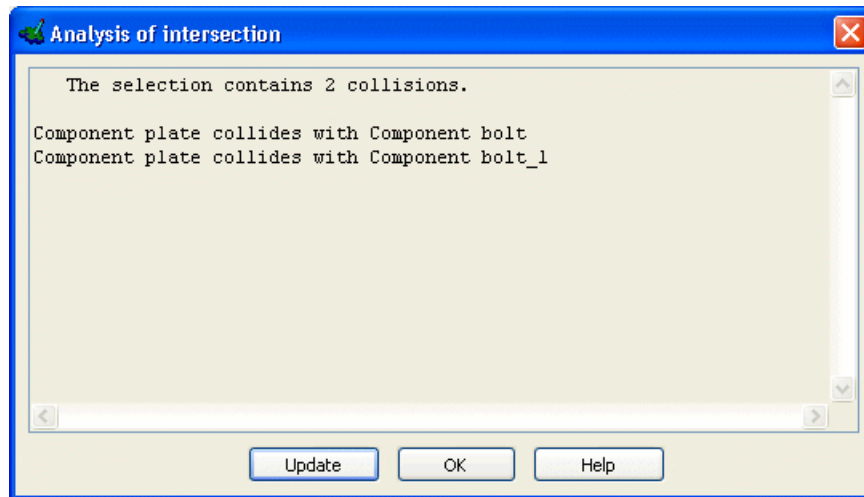
Remove - Select this option to remove the selected component from the Group. This option is only active if the selected component has been previously added to a Group.

How do I find out if components intersect?

You can check whether components intersection in an assembly.

- 1 Select the components.

- 2 Click  to display the Assembly modelling toolbar.
- 3 Click  (assembly modelling toolbar). The **Analysis of intersection** dialog is displayed.



This dialog reports any collisions between the selected components.

- 4 If you want to check other components, select them and click **Update**.
- 5 Click **OK** to remove the dialog from the screen.

Converting a component into a solid

You can convert a component into a solid using the context menu

- 1 Right click the component you wish to convert, either in the assembly tree or in the graphic window
- 2 Select **Convert** from the context menu.
- 3 Select **To Geometry** from the options. Any relationships created using the component will be deleted.

You can also convert a component into a solid using the **Edit** menu:

- 1 Select the component.
- 2 Select the **Edit** menu, followed by **Convert** and then **Component**.

Converting a sub-assembly into its components

You can convert a sub-assembly into its components, whilst keeping the relationships between the components in the sub-assembly.

- 1 Right click the sub-assembly in the assembly tree.
- 2 Select **Convert** from the context menu.
- 3 Select **To Components** from the options

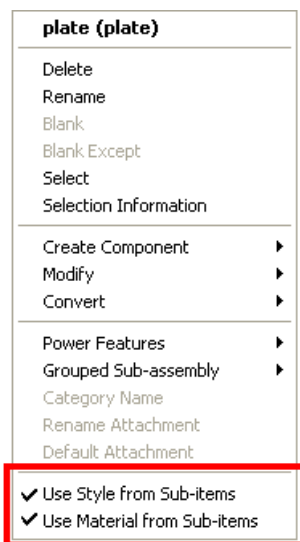
You can also convert a sub-assembly into its components using the **Edit** menu:

- 1 Select the sub-assembly.
- 2 Select the **Edit** menu, followed by **Convert** and then **SubAssembly**.

Setting the material on a single component

You can set the material and style of a single component or a single component in a Group, using the Assembly Tree.

- 1 In the assembly tree, right click a single component or a single component in a Group.
- 2 Select one of the following options from the context menu.



Use Style from Sub-items - If *ON*, the component uses the style attributes of sub-items. If *OFF*, it uses the creation style of the component.

Use Material from Sub-items - If *ON*, the component uses the material attributes of the sub-items. If *OFF*, it uses the creation material of the component.

So, if you reset the material on a component, it uses the same material as the component definition. If you reset the material on a sub-assembly, its components will use their own materials.

You can also set the material on multiple components (see page 116) using the popup in the assembly tree.

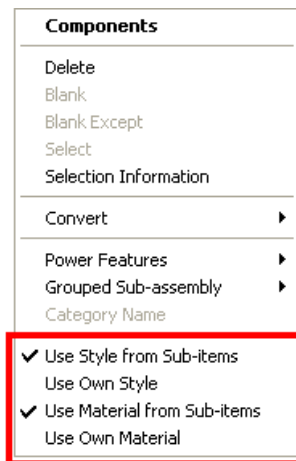
For further details, see What is material? in the Delcam Render manual.

Setting the material on multiple components

You can set the material and style of a selection containing more than one component using the options on the context menu in the assembly tree. These options allow you to set or reset the style and material for many components in a single operation.

The selection can be a

- Group
- Multiple selection of components
- Multiple selection of Groups.



Use Style from Sub-items - The selection uses the style attributes of sub-items.

Use Own Style - The selection uses the creation styles of the selection.

Use Material from Sub-items - The selection uses the material attributes of the selection.

Use Own Material - The selection uses the creation material of the selections.

Renaming an attachment

You can give a name to a selected attachment. This can be useful for automation, when writing macros or Visual Basic.

- 1 Right click the component in the graphics window and select **Manage Attachment** to display the **Manage Attachment** dialog (see page 98).
- 2 Select the attachment from the dropdown list.
- 3 Enter the name for the attachment in the text box and click **OK** to remove the dialog from the screen.

Parameterised Components



Introduction

You can create a parameterised component from a solid that has dimensions defined by parameters. This allows you to create components with different dimensions from the same solid.

Assembly Parameters

Parameters can be shared between solids when they are added to an assembly.

To see how local and shared parameters work within an assembly, work through the following steps:


- 1 Create a block with height **h=100**
- 2 Create a cylinder with height **h**.
- 3 Select the block and click **Create components from selected solids** .
- 4 Select the cylinder and click **Create components from selected solids** .




It is important to add the block and the cylinder using separate operations.

- 5 Modify the height of the block component. The height of the cylinder component should remain unchanged.

This is because, by default, components have local parameters. Although both solids were defined using the parameter **h**, once they are added to the assembly as components, the parameters are independent and **h** can have different values. This is useful because assemblies often have multiple instances of the same object. For example, you would not necessarily want all the screws to have the same length.

- 6 Create a block with height **h=100**
- 7 Create a cylinder with height **h**
- 8 Select both block and cylinder and click the **Create components from selected solids** .



A screenshot of the 'Moldbase' context menu in SolidWorks. The menu is open, showing various options. The 'Modify Parameters' option is highlighted with a red rectangular box. Other visible options include 'Delete', 'Rename', 'Selection Information', 'Activate', 'Deactivate', 'Add to Assembly', 'Add to Assembly Advanced', 'Modify Properties', 'Convert to local', 'Power Features', and 'Category Name'.



- 1 Create a parameterised solid
- 2 Right click the solid to display the solid context menu.
- 3 Select **Add to Assembly** from the context menu. The parameterised solid is converted to an component and added to the assembly.



Modifying parameters

Changing the parameters for each component

- You can have many components in an assembly that are based on a single component definition, but with different values for their independent parameters.
- When you change the value of a parameter of a component, it preserves any relationships with other components.
- You can change the value of any independent parameter. If this parameter is used by another parameter in that component, the dependent parameter will also be updated. The dependent parameter is not displayed in the Parameter List Editor dialog (see page 121).

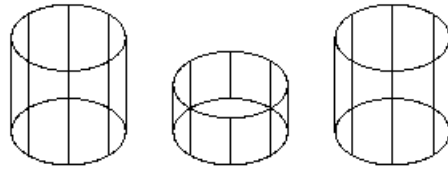
Modify parameters with the same name on multiple toolmaker components

- Components with the same name that are from the same catalog are modified using the Component Wizard. This lets you apply the same parameter values to the selected components.
- If selected components are mix of Toolmaker components and general components, the Parameter List Editor dialog is displayed. Enter a new value to modify a parameter.
- If all selected components are Toolmaker components, but are from different catalogs or have different names, the Parameter List Editor dialog is displayed. Enter a new value to modify a parameter.

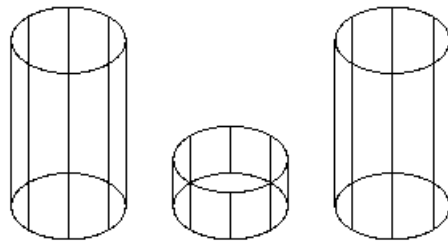
Editing a parameterised component

- 1 Right click the component in the browser tree.
- 2 Choose one of the following options to display the Parameters List Editor dialog (see page 121).
 - Select **Parameter** from the **Modify** options of the component context menu. This modifies the parameters of parameterised component that are selected in the assembly tree.
 - Double-click on a parameterised component in the model.
 - Select **Modify Parameters** from the **All Instances** options of the component context menu. This modifies the parameters of components based on the same definition that use exactly the same parameter values.

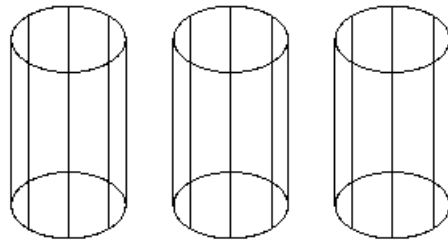
The following example shows how this option can be used. All the components were created with the same radius, but the middle one has a different height.



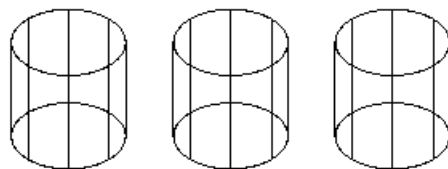
If we change the height of the component on the left using the **All Instances** option, the two identical components change shape.



If you change the height of the middle to match the other two, all three components will become identical.

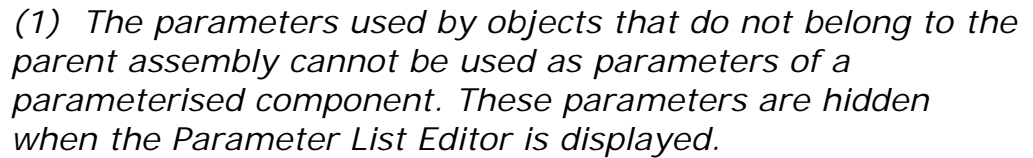


So if you edit any parameter on one of the three objects using the **All Instances** option, all three will change shape.




- 3 Use the Parameter List Editor dialog (see page 121) to change the value of the parameters.

You can also use this dialog to change the dimensions of the component as you add it to the assembly. For further details, see Adding a parameterised component to an assembly (see page 118)



- Typing an expression into the **Distance/Angle** box on the Relationship toolbar.
- Right clicking on the **Distance/Angle** text box to raise the calculator. You can enter an expression or a value in the calculator. When you press **Accept** on the calculator, the value (or the value of the expression) will be shown in the **Distance/Angle** text box.

Parameterised relations are marked by  in the tree browser.

When a parameter is applied to a relation, the current assembly becomes parameterised.

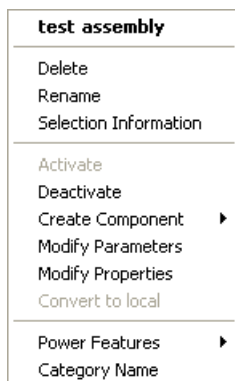
A parameterised assembly is marked by  in the tree browser.

Editing a parameterised assembly

You can modify the parameters used in relations and parameterised components in an assembly.

To edit the parameters used in an assembly

- 1 Right click the assembly in the browser tree to display the assembly context menu.

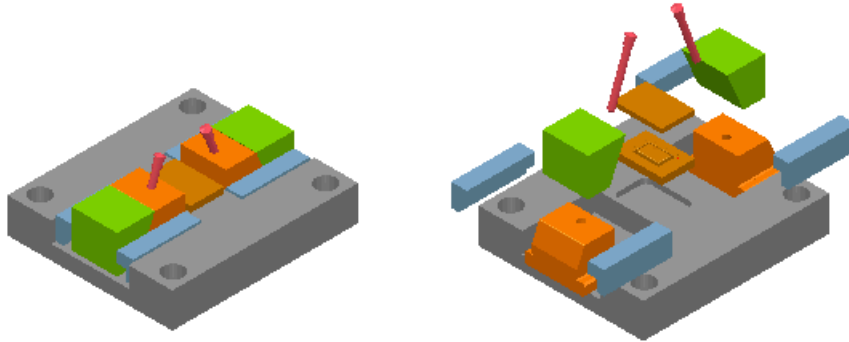


- 2 Choose **Modify Parameters** to display the Parameter List Editor dialog (see page 121)
- 3 Click on **Apply**. The updated assembly will be displayed.
- 4 Click on **OK** to remove the dialog from the screen.




Exploding assemblies

Examining an assembly

The **Explode** toolbar lets you create an exploded view of an assembly to show how it is put together.



You can explode an assembly using the distance or angle parameter from the relationships of the components.

- 1 Make sure you have an active assembly. Some of its components must contain either a distance or angle relation.
- 2 Click **Explode**  to display the flyout.
- 3 Select one of the two options to create an exploded view of the active assembly:
 -  Explode the assembly using a distance parameter. The view will be exploded along the axis in which the distance was defined when creating the relationship between the components.
 -  Explode the assembly using an angle parameter. The view will be exploded around the axis in which the angle was defined when creating the relationship between the components.
- 4 Select the options on the **Explode** toolbar (see page 123).

Explode toolbar

This toolbar is displayed when an option is selected from the **Explode** flyout.



The items on the toolbar are grouped as follows:

The following let you enter parameters for the explode operation:

Distance - Enter a distance of exploding of the active assembly, or select a previously used distance from the pull down list.
(Note - This option is only available if you are exploding the assembly using a distance parameter.)



Angle – Enter an angle of exploding for the active assembly, or select a previously used angle from the pull down list. (Note - This option is only available if you are exploding the assembly using an angle parameter.)

Number of steps - Enter the number of steps for exploding, or select a previously used number from the drop down list.

Step time - Enter a time between steps during exploding, or select a previously used time from the drop down list.

The following buttons that let you control the explode operation:



If this is *ON* , the assembly will be exploded and re-assembled in a continuous loop. To exit from the cycle, click the  button on the processing bar.



Start exploding the active assembly.



Previous step of the exploding operation.



Next step of the exploding operation.



Stop the exploding operation. The original, unexploded assembly will be displayed.



Abandon exploding and exit.

Schedules

Creating a schedule for components and assemblies (see page 124)

Creating hole schedules for assemblies (see page 125)

Creating a schedule for components and assemblies

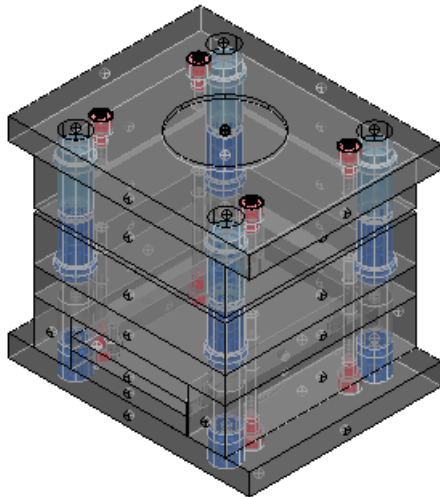
You can create schedules from assemblies. A schedule for an assembly is a bill of material. This is a table of components/sub-assemblies used by an assembly.

You can also create schedules from components. Since components are part of an assembly, a schedule from components is similar to the one for assemblies.

For further details, see What is a schedule? in Delcam Draft..

Creating hole schedules for assemblies

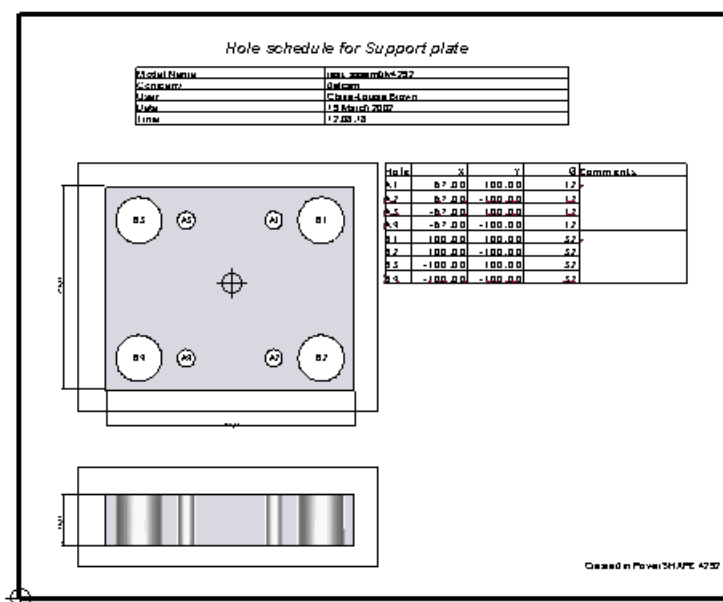
You can automatically create detail drawings of hole schedules from an assembly. You can create hole schedules for a mold base design like the one below.



- 1 Select **Component Drawings** from the **Tools** menu. The Component Drawing dialog is displayed.
- 2 Set the options in the dialog as required
- 3 Click **OK**

A drawing is created for each plate in the assembly and contains the following:

- Two views of the plate: top and side
- Dimensions of the plate (for molds only)
- A hole schedule and labels



You can customise the drawings using the template model called *component_drawing_template.psmodel*.

Change the template for the drawings

The drawings are created from template drawing in the *component_drawing_template* model in the *dcam/parts* folder. The template drawing is called *component_drawing_template*.

You can modify the template drawing to your own requirements.

During installation, the *component_drawing_template* model is put in the following folder:

[C:/dcam/product/powershape152XX/file/assembly](#)

When you create hole schedules for an assembly for the first time, a copy of this model is automatically imported into the *dcam/parts* folder.

Using the Component Wizard

For information on adding components using the Component Wizard, see:

Adding components to an assembly (see page 126)

Selecting a component (see page 126)

Defining a component using the Component Wizard dialog (see page 128)

Component Colour scheme (see page 137)

Adding components to an assembly

You can add components from standard catalogs to your assembly.


Product specific catalogs are supported, so different component libraries are available with, for example, Delcam Designer, Delcam Toolmaker and Delcam Crispin SoleDesign.

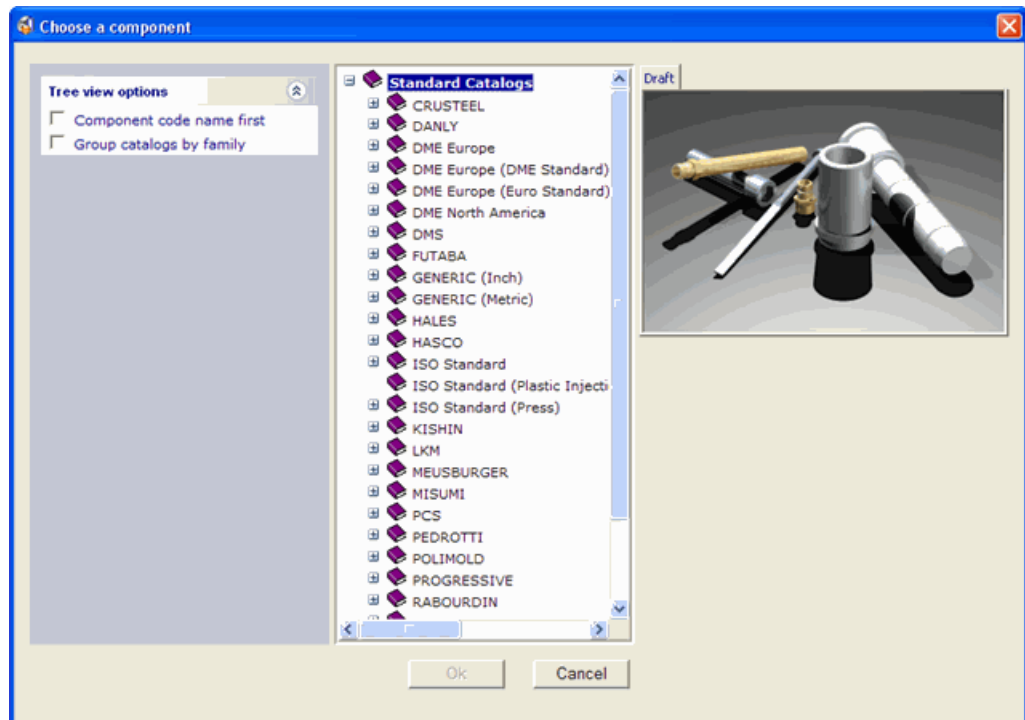
To create a component in the assembly,

- 1 Select a component from the catalog (see page 126).
- 2 Define the dimensions and material of the component (see page 128).

Selecting a component

A component is selected from the catalog as follows:

- 1 Click  (Assembly toolbar) to display the **Choose a component** dialog



- 2 Use the options to select a component.

Tree view options — Controls the way that the catalogs are displayed in the tree.

- **Component code name first** — If selected, the components are listed using their catalog part numbers.
- **Group catalogs by family** — If deselected, the catalogs for an individual manufacturer are listed separately. If selected, catalogs are grouped by manufacturer. So,

DME-North America

DME Europe

DME Europe (DME Standard)

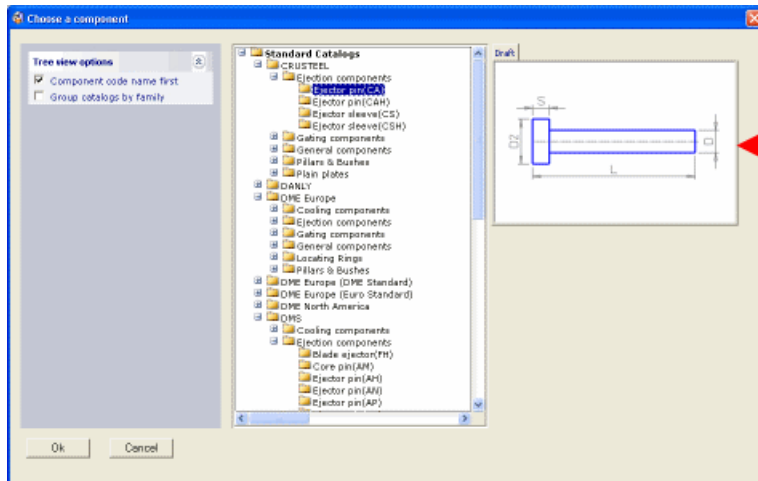
DME Europe (Euro Standard)

will all appear under **DME** in the tree.

Manage custom catalogs — Use the options in this section to create, edit and structure catalogs. Details and examples of this functionality are included in the Custom catalogs dection of the Delcam Toolmaker manual.



— This gives the list of components in the catalog. Select a component to create from this list. When you select a component from a catalog, the image of the component is displayed in the dialog.



The image shows the dimensions you can set on the selected component. For certain components, there are tabs over the image. Selecting a different tab lets you see different images for these components. This makes it easier for you to see all the parameters of complex components.

OK — Selects the component and opens the page for that component in the browser window. This page is used to define the dimensions of the component (see page 128). Each component will have its own page in the browser window.

Cancel — Removes the dialog from the screen.

Defining a component using the Component Wizard dialog

Use the Component Wizard dialog to:

- Edit the size of a component.
 - 1 Select **Modify Parameters** from the Component context menu to display the Component Wizard dialog.
 - 2 Edit the component and select **Apply**. The model will be updated.
- Set the dimensions and material of the component.

- 1 Select dimensions and material values from the catalog.
- 2 Enter your own values, as required, to create user-defined components. These can still have some of the properties of a standard component. However, user-defined components have to be manufactured, whereas standard components can be ordered from a catalog supplier.



*You can also display the **Choose a component** dialog as follows:*

- 1. Click on the Component wizard dialog to ensure that it is active*
- 2. Hold down the **Alt** key and press **C***

There are four types of tab that are displayed in the dialog

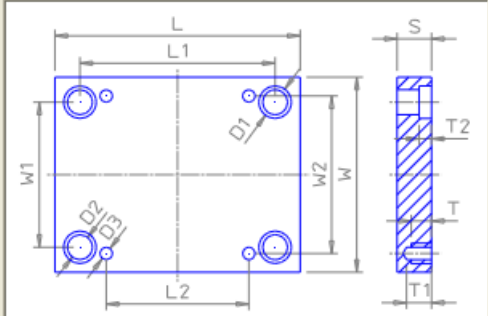
- Component Wizard - Standard tab (see page 130)
- Component Wizard - Custom tab (see page 133)
- Component Wizard - Key & Trim tab (see page 135)
- Component Wizard - Clearances tab (see page 136)

Component Wizard - Standard tab

The **Standard** tab displays the dimensions for the catalog component that you have selected.

Component Wizard

HASCO
Cavity plate (K20)



Standard Custom

Catalog reference

K20/100x130x12/1.1730

Plate width (W)	Plate length (L)	Thickness (S)
99.7	129.7	12
		17
		22
		27
		36

Material

1730
2085
2162
2311
2312

Mold width	Mold length	Pillar holes
100	130	Yes

Screw holes

Yes

Pillar holes parameters >>

Screw holes parameters >>

Function Cavity plate

☐ Position the component

Apply Dismiss Help

The exact parameters that are displayed reflect the type of component you are defining. When the program is run on a low resolution screen, the levels toolbar is automatically switched off and height of the panel is limited to leave more space for the component parameter controls.

The image in the graphic window is the component you selected using the Choose a component dialog. The image shows the labels of the dimensions on the selected component.

For certain components, there are additional tabs on the dialog. These tabs correspond to the tabs displayed in the **Choose a component** dialog. Selecting a different tab lets you see different images for these components. This makes it easier for you to see all the parameters of complex components. The different images are displayed by selecting a tab on the wizard.

Plate width (W)	Plate length (L)	Thickness (S)
99.7	129.7	12
		17
		22
		27
		36

Material
1730
2085
2162
2311
2312

Mold width	Mold length	Pillar holes
100	130	Yes

The default dimension and material options for the component that you have selected from the catalog are displayed in this section of the dialog. To see all the catalog options for the component, click

Show All

. Select the dimensions and material you require. Options to customise settings are available on the Custom tab (see page 133).

As you select a dimension from one column, the numbers of the other columns reduce to show valid combinations only.

Material - Select the International Standard number for the material to be used.

Pillar holes parameters	>>
Screw holes parameters	>>

Click on the coloured bar to reveal the fields.

Pillar holes parameters <<		
Width distance	W1	154
Length distance	L1	152
Diameter	D2	26
Counterbore diameter	D1	32
Counterbore depth	T2	6.3

To hide the fields, click the coloured bar again.


Function - The option you select specifies which component you intend to position the component on. Either select an option from the drop down list or use the default option (**Selected on model**) to pick the component in the model.

Position the component - If this option is selected and you click **Apply** on the browser page, the object is added to the assembly. You can then position it using the following technique from Assembly modelling.

- 1 Select an existing component. Valid attachments are now visible on the new component.
- 2 Click an attachment on the new component. Valid attachments on the existing component now become visible. There are two types of attachments on the existing component: plane and axis attachments.

Plane attachment - This is shown as an arrow. If you hover the mouse over this attachment, the attachment on the new component is aligned in the same plane as the existing attachment. When you click the mouse, the new component is placed at the cursor position and aligned with this plane attachment.

Axis attachment - This is shown as an arrow with a circle at the bottom. If you hover the mouse over this attachment, the attachment on the new component is fixed on the axis attachment. If you click the mouse anywhere on the screen and don't hover over another attachment, the new component is snapped onto this axis attachment.

- 3 Once a new component is added to the existing component, another one appears. Cancel the command by clicking ; the new component disappears.

The component is also registered as an assembly component. You can create further copies of this component or modify it using Assembly modelling techniques (see page 7).

Apply - This creates the component on the screen.

Component Wizard - Custom tab

Use the options on the **Custom** tab to change the standard dimensions.

The screenshot shows the 'Component Wizard' dialog box for a 'HASCO Cavity plate (K20)'. It features a technical drawing of the cavity plate with dimensions L, L1, L2, W, W1, W2, S, T, T1, T2, D1, D2, and D3. The 'Custom' tab is selected, showing input fields for Mold width (100), Mold length (130), Plate width W (99.7), Plate length L (129.7), Thickness S (12), and Material (1730). Checkboxes for 'Pillar holes' and 'Screw holes' are both checked. There are expandable sections for 'Pillar holes parameters' and 'Screw holes parameters'. At the bottom, there is a 'Function' dropdown set to 'Cavity plate', a checkbox for 'Position the component', and 'Apply', 'Dismiss', and 'Help' buttons.

Parameter	Value
Mold width	100
Mold length	130
Plate width W	99.7
Plate length L	129.7
Thickness S	12
Material	1730

☒ Pillar holes
☒ Screw holes

Function: Cavity plate
☐ Position the component

Buttons: Apply, Dismiss, Help

Use the options like **Plate width**, **Plate Length** and **Thickness** to change the dimensions. The exact parameters that are displayed in the **Custom** tab reflect the type of component you are defining.

Mold width	100
Mold length	130
Plate width W	99.7
Plate length L	129.7
Thickness S	12

Material - Select the International Standard number for the material to be used.

Pillar holes parameters	>>
Screw holes parameters	>>

Click on the coloured bar to reveal the fields.

Pillar holes parameters <<		
Width distance	W1	154
Length distance	L1	152
Diameter	D2	26
Counterbore diameter	D1	32
Counterbore depth	T2	6.3

To hide the fields, click the coloured bar again.


Function - The option you select specifies which component you intend to position the component on. Either select an option from the drop down list or use the default option (**Selected on model**) to pick the component in the model.

Position the component - If this option is selected and you click **Apply** on the browser page, the object is added to the assembly. You can then position it using the following technique from Assembly modelling.

- 1 Select an existing component. Valid attachments are now visible on the new component.
- 2 Click an attachment on the new component. Valid attachments on the existing component now become visible. There are two types of attachments on the existing component: plane and axis attachments.

Plane attachment - This is shown as an arrow. If you hover the mouse over this attachment, the attachment on the new component is aligned in the same plane as the existing attachment. When you click the mouse, the new component is placed at the cursor position and aligned with this plane attachment.

Axis attachment - This is shown as an arrow with a circle at the bottom. If you hover the mouse over this attachment, the attachment on the new component is fixed on the axis attachment. If you click the mouse anywhere on the screen and don't hover over another attachment, the new component is snapped onto this axis attachment.

- 3 Once a new component is added to the existing component, another one appears. Cancel the command by clicking ; the new component disappears.

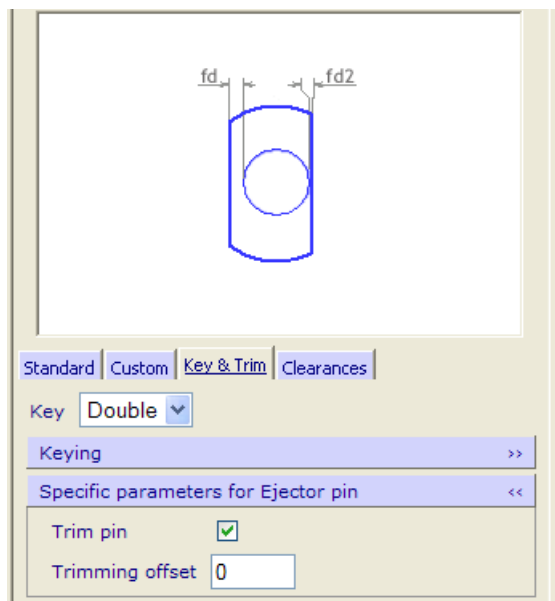
The component is also registered as an assembly component. You can create further copies of this component or modify it using Assembly modelling techniques (see page 7).

Apply - This creates the component on the screen.

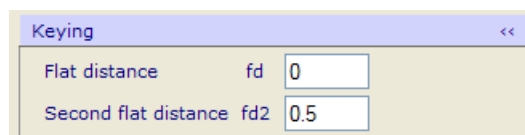
Component Wizard - Key & Trim tab

The **Key & Trim** tab displays the parameters for the single or double slot keyed pins that are available for plain, stepped, blade ejector pins and ejector sleeves. The parameters that are displayed are specific to the type of component you are creating.

The parameter labels refer to the dimensions on the graphic in the dialog.



Keying - Click the coloured bar to display the options.



Specific parameters for Ejector pin - Click the coloured bar to hide the options.

Trim pin - (Only available for ejector pins, sleeves and blades) By default, ejector pins and blades are now trimmed when you create them. Turn the option off if instant trimming is not required.

Trimming offset - (Only available if Trim pin is selected) The offset allows the pin to be trimmed so that it projects from the product surfaces. This compensates for the pressure on the pin when the mold is closed and ensures that when the mold is closed, the pin is never lower than the product surfaces. This helps create a good finish to the molded part.


Function - The option you select specifies which component you intend to position the component on. Either select an option from the drop down list or use the default option (**Selected on model**) to pick the component in the model.

Position the component - If this option is selected and you click **Apply** on the browser page, the object is added to the assembly. You can then position it using the following technique from Assembly modelling.

- 1 Select an existing component. Valid attachments are now visible on the new component.
- 2 Click an attachment on the new component. Valid attachments on the existing component now become visible. There are two types of attachments on the existing component: plane and axis attachments.

Plane attachment - This is shown as an arrow. If you hover the mouse over this attachment, the attachment on the new component is aligned in the same plane as the existing attachment. When you click the mouse, the new component is placed at the cursor position and aligned with this plane attachment.

Axis attachment - This is shown as an arrow with a circle at the bottom. If you hover the mouse over this attachment, the attachment on the new component is fixed on the axis attachment. If you click the mouse anywhere on the screen and don't hover over another attachment, the new component is snapped onto this axis attachment.

- 3 Once a new component is added to the existing component, another one appears. Cancel the command by clicking ; the new component disappears.

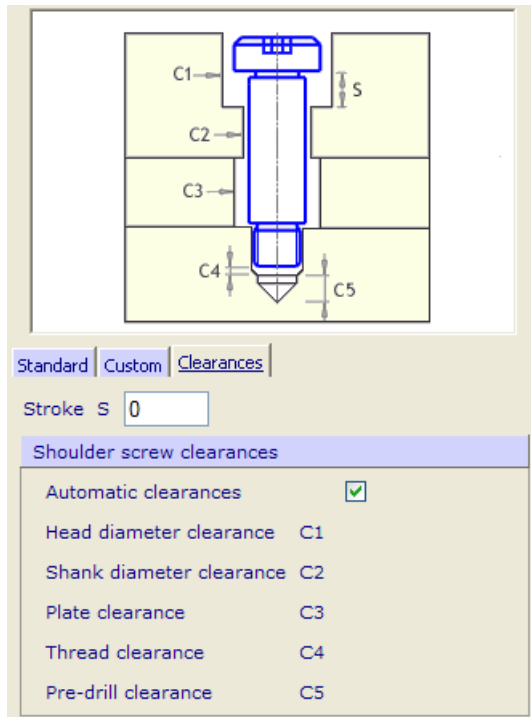
The component is also registered as an assembly component. You can create further copies of this component or modify it using Assembly modelling techniques (see page 7).

- 4 **Apply** - This creates the component on the screen.

Component Wizard - Clearances tab

The **Clearances** tab displays the parameters that are required for components such as guide pillars, ejector pins, screws, sprue bushes, support pillars and dowels. The parameters that are displayed are specific to the type of component you are creating.

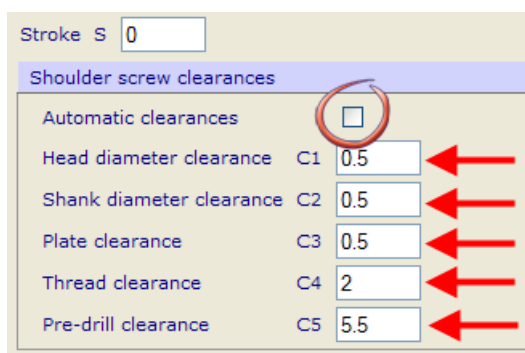
The parameter labels, for example *C1*, *C2*, *C3* and *C4*, refer to the dimensions on the graphic in the dialog.



Automatic clearances - If this option is selected, the mounting clearances **C1**, **C2** and **C3** have predefined values based on the following:

- **Body diameter and Body length** dimensions (selected on the **Standard** tab).
- **Stroke** dimension.

If **Automatic clearances** is deselected, the parameter boxes are available. Enter the clearances that you require.



Component Colour scheme

To ensure clarity when designing both the 3D model and 2D drafting, the component colour scheme uses the following rules:

- With the exception of plate and riser components, the wireframe colour of components use the colour of the component material.

- The wireframe colour of plate and riser components is set to the default colour of the components for the colour scheme.
- Plate components are assigned a material that is 90% transparent, allowing you to see clearly all the components and features inside the plates.

Assembly options

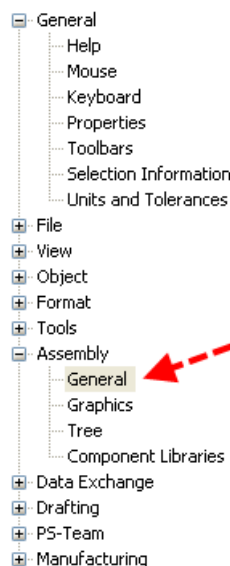
There are four pages of Assembly options:

- General options (see page 138)
- Graphics options (see page 141)
- Tree options (see page 142)
- Component Libraries options (see page 144)

General options

Use this page to define the operation of assembly modelling.

- 1 Select **Tools > Options**. This displays the **Options** dialog.
- 2 Expand the **Assembly** branch and select **General**.

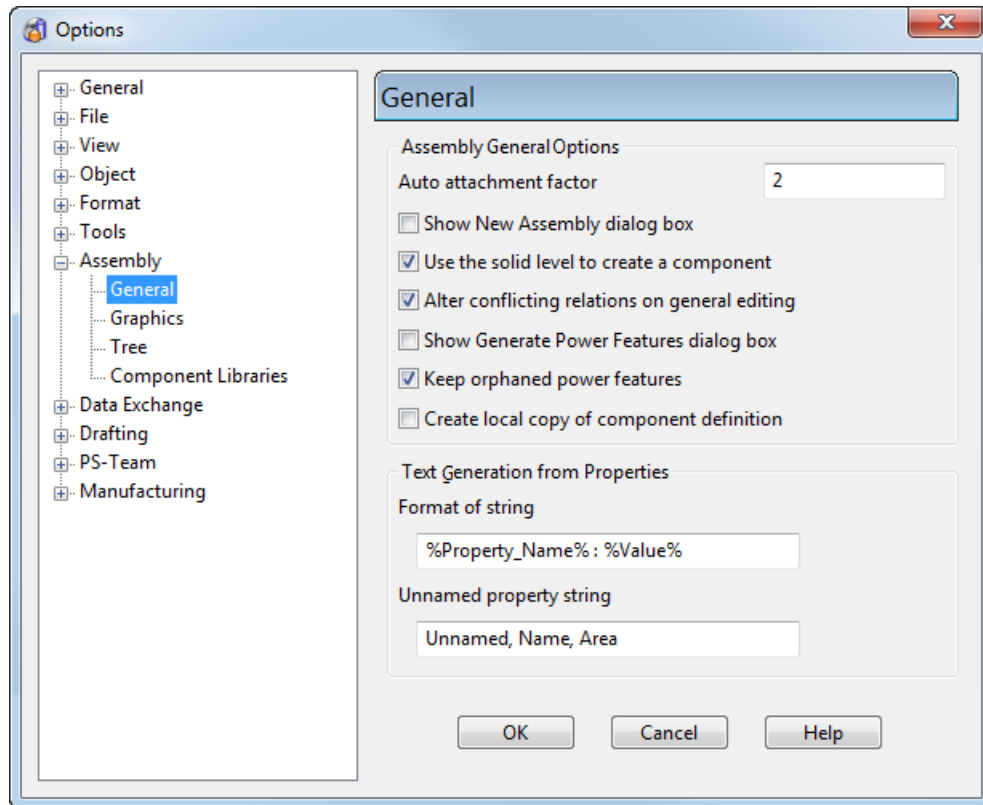


- 3 Make the required changes to the options on the dialog. For further details, see **Options > General** (see page 139).
- 4 Click **OK**.

The dialog is closed and the new settings will now be used. The current model and any new model you create will use these settings. However, when you open existing models, they will open with the settings saved with the model.

Options - General

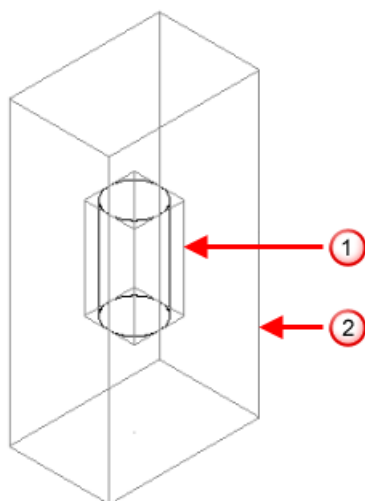
Use this page to define the operation of assembly modelling.



Assembly General Options

This group of options control the general operation of assembly modelling.

Auto attachment factor — Enter the factor by which the original bounding box of the geometry of the component grows. If an attachment is outside the new bounding box, it is not included in the component definition.



① Original bounding box of the geometry of the component definition

② Bounding box when an **Auto attachment factor** of 2 is applied.

Show New Assembly dialog box — Select this option to display the **Create New Assembly** dialog when you create a component and there is no current assembly. The **Create in new model** option in the **Create New Assembly** dialog is disabled because you can use the solid from the current model.

Deselect this option to create the new assembly automatically with a default name. **Description** and **Category Name** are set to *<none>*.

Use the solid level to create a component — Select this option to create components on the level of the source solid when creating components directly from the solid. When you modify a component, the solids of the component definition are moved onto the level of the component being modified. When converting components to solids, the solids are created on the component level.

Deselect this option to create components on the current level.

Alter conflicting relations on general editing — Select this option to alter or remove a relationship to allow a component to be repositioned. Deselect this option to keep all the relations and restrict the new position of the slave component.

Show Generate Power Features dialog box — Select this option to display the **Generate Power Features** dialog when you select **Generate Power Features** from the source or target component popup.

If deselected, the dialog is not displayed and the effect of selecting **Generate Power Features** from the component popup is one of the following:

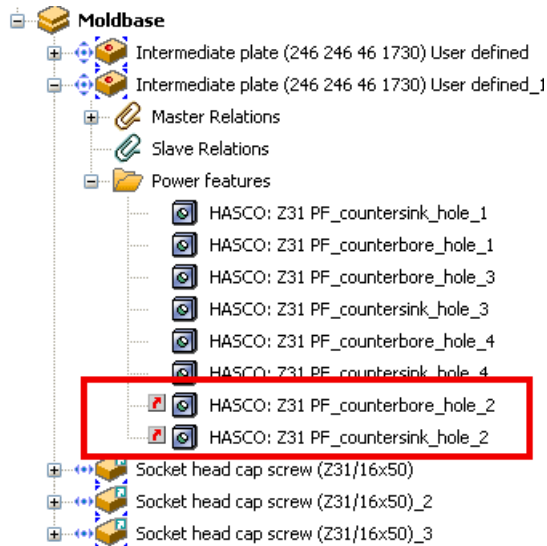
- If you select a target component, **Generate Power Features** generates Power Features for all the source components that intersect with the target.
- If you select a source component, **Generate Power Features** generates Power Features for all the target components that intersect with the source.



*This option also controls the behaviour of the **Remove Features** dialog.*

Keep orphaned power features — When you delete a component that is the only one associated with a particular power feature, the power feature is orphaned.

If selected, the orphaned power feature is marked as out of date. The orphaned power feature is removed from the tree when you select **Generate Power Features** from the assembly context menu.



If deselected, the orphaned power feature is removed from the assembly tree.

Create local copy of component definition — This controls the setting of **Create local copy of component definition** on the **Add Component** dialog.

If *ON*, the option on the **Add Component** dialog is always set *ON*.

If *OFF*, the option on the **Add Component** dialog is always set to *OFF*.

Text Generation from Properties

Use these options to customise the format of the text that is generated from the component definitions.

Format of string — Enter the format of the text that is generated from the Component properties. This text is used when displaying the Component properties on the model. The default setting is *Property Name : Value*.

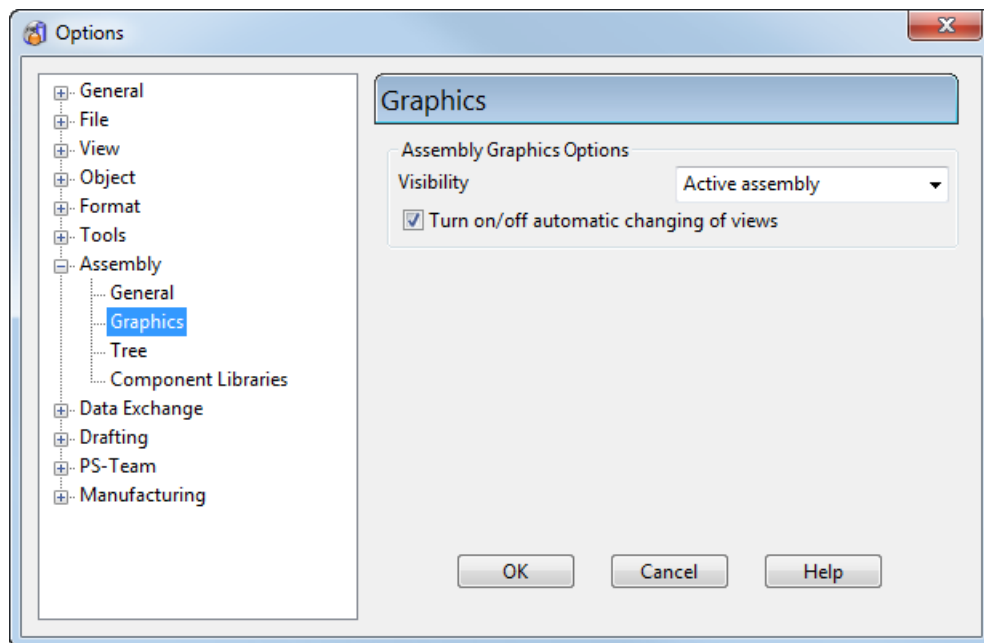
Unnamed Property String — Use this option to decide the properties that have their name displayed. For each prefix in this list, the value of the property is included, but the property name is not displayed.

Graphics options

You can select which of assemblies are visible using the **Tools** menu.

- 1 From the **Tools** menu, select **Options**. This displays the **Options** dialog.

- 2 Select **Assembly**, followed by **Graphics**.



- 3 Select the **Visibility** from the list:

Active Assembly — Only the current active assembly is visible.

Top Assemblies — Only the top assemblies (those that are not inserted into other assemblies) are visible.

All Assemblies — All Assemblies are visible.

- 4 Update the **Turn on/off automatic changing of views** option if required. Select the option to change the views automatically when a new component is created. Deselect the option to keep the views even if the component that is created is outside the view bounding box and therefore invisible.
- 5 Click **OK**.

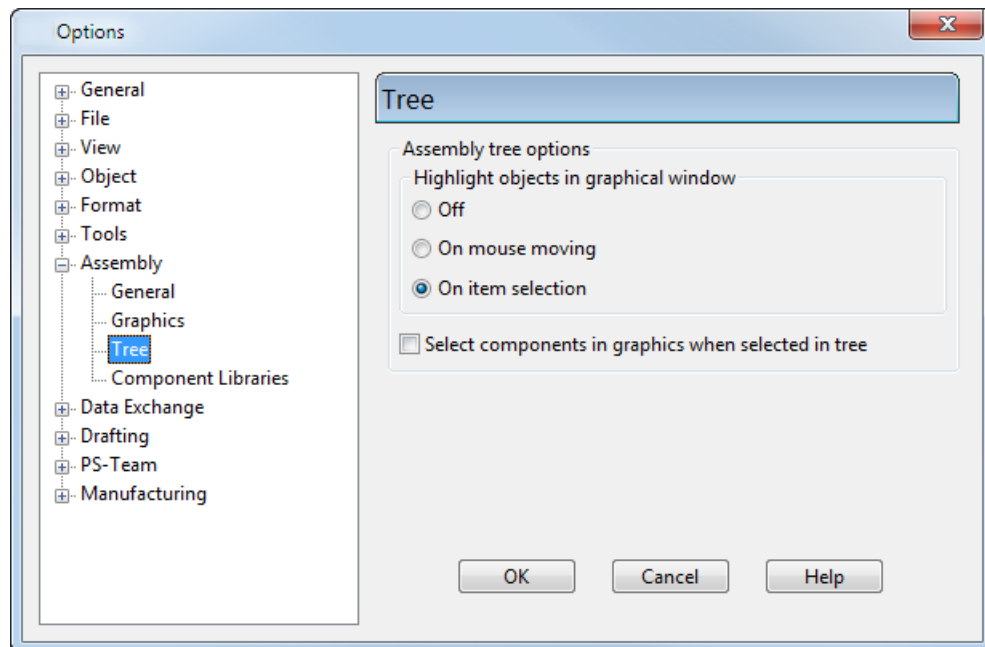
The dialog is closed and the new settings are used. The current model and any new model you create use these settings. However, when you open existing models, they open with the settings saved with the model.

Tree options

Use this page to control when objects in an assembly are highlighted.

- 1 From the **Tools** menu, select **Options**. This displays the **Options** dialog.

2 Select **Assembly**, followed by **Tree**.

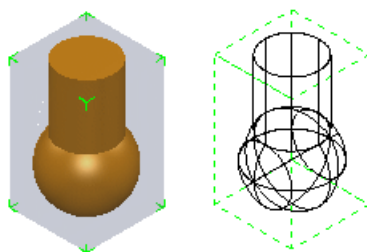


3 Select the options you require.

Off — Objects in the assembly are not highlighted by selecting an item in the assembly tree.

On mouse moving — The object under the cursor in the assembly tree is highlighted in both the tree and graphical window. The item under the cursor is drawn in a bounding box around the highlighted component.

If the assembly is displayed in shaded mode, the bounding box is drawn as a transparent shaded box. A dashed wireframe box is used if the assembly is displayed in wireframe mode.



On item selection — The object you select in the assembly tree is highlighted in the graphical window. The object is not highlighted when the cursor is moved outside the tree browser.

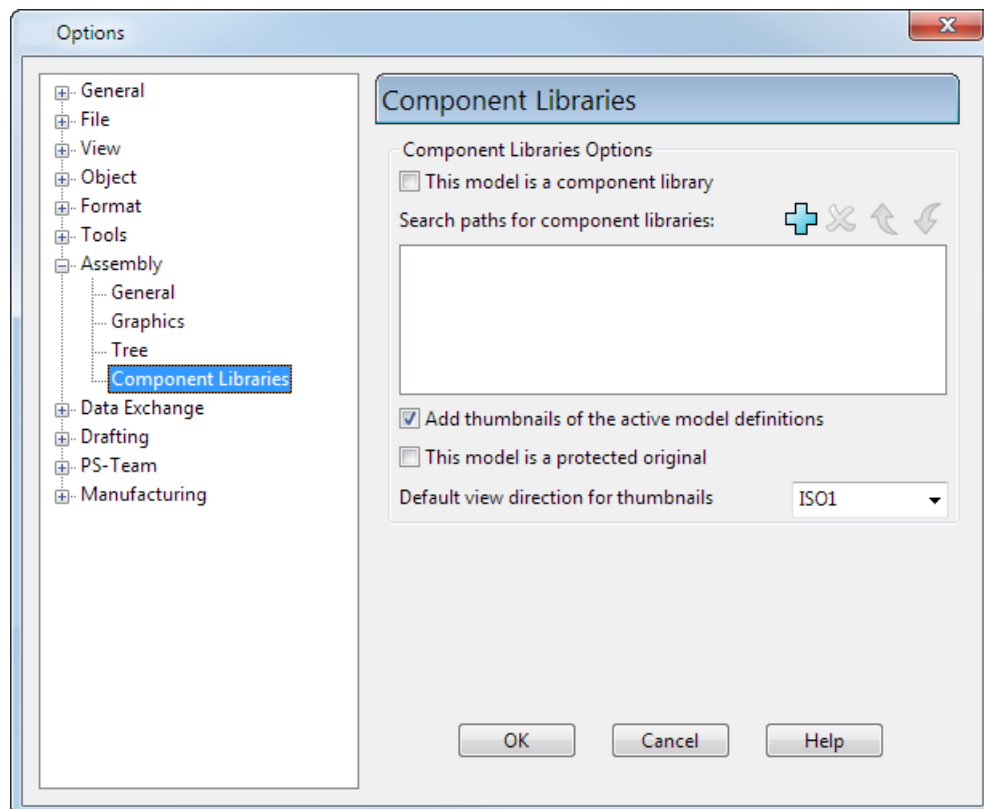
Select components in graphics when selected in tree — If ON, a component selected in the assembly tree is also selected in the model. If OFF, a component in the assembly tree is highlighted in the model. The default setting is OFF.

- 4 Click **OK**. The dialog is closed and the new settings are now used. The current model and any new model you create use these settings. However, when you open existing models, they open with the settings saved with the model.

Component Libraries options

Use these options to define how component libraries are used.

- 1 From the **Tools** menu, select **Options**. This displays the **Options** dialog.
- 2 Select **Assembly**, followed by **Component Libraries**.



This model is a component library — If *ON*, the active model is saved as components in the component library.

If you select this option,

- 1 A thumbnail image of the definition is saved in the model.
- 2 The **Query** for deleting unused component definitions is not displayed.
- 3 You can use the **Open Component Library** dialog and the **Component Library** window to create components.

For further details, see Component Libraries (see page 31).

Search paths for component libraries — This window shows the list of paths that are searched for component libraries..

The path to the component library that was used by an assembly is stored with the assembly. If, when required, the component library is no longer at the stored location, the paths that are specified in this window are searched for the required component library. When using the default settings, no search path is initially included.

You can change the search list by using the associated buttons.



— Click this button to add a new path to **Search paths for component libraries** using the **Browse For Folder** dialog. Select the path to add to the list of search paths.



— Click this button to removes selected path from the list.



— Click this button to move the selected path up in the search order.



— Click this button to move the selected path down in the search order.

To replace a selected path:

- 1 Double-click a path in the window.
- 2 Select a replacement path from the **Browse for Folder** dialog

Add thumbnails of the active model definitions — If *ON*, the thumbnails of the component definitions for the active model are added to the component library window when opening a model, updating a model or closing a model. The thumbnails of the old active model are removed from the window.

This model is a protected original — This gives more control over updating rights on the standard component libraries. If *ON*, you cannot

- update the original definitions from the dependent models.
- save the updated model to the protected original. You are given the opportunity to save the model to another name, using **Save As**.

Default view direction for thumbnails — Select a view direction from the list to set the orientation of the component definitions that are shown in the component library window.

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