PowerSHAPE 2010

# What's New

## PowerSHAPE

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# What's New in PowerSHAPE 2010

The following areas have been updated in PowerSHAPE 2010:

**PowerSHAPE 2010 for all users**


Interface (see page 1)

Solid modelling with Parasolid (see page 8)

Modelling with triangles (see page 36)

Assembly modelling (see page 40)

Delcam Draft (see page 48)

Other changes (see page 51)


**PowerSHAPE 2010 for users of variants**


Delcam CopyCAD (see page 48)

Delcam Designer and Designer for Jewellery (see page 49)

Delcam Electrode (see page 50)


## User interface

The following changes have been made to the PowerSHAPE interface:


- Toolbars (see page 6)
- Dialog boxes (see page 5)
- Menus (see page 7)

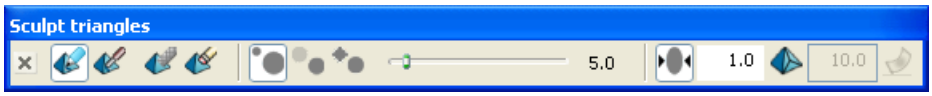
# Toolbars

Changes have been made to the following toolbars:

- Sculpt triangles (see page 2)
- Mesh editing (see page 4)
- Model fixing (see page 4)
- Solid edit (see page 4)
- Assembly toolbar (see page 5)

## Sculpt triangles

The **Sculpt triangles** toolbar has replaced the **Interactive Triangle Editing** dialog box. The toolbar is displayed when you click on the **General Edits** toolbar. By default, the toolbar is floating, but if required, the toolbar can be docked alongside existing toolbars.




1. Select one of the following operations:

   **Add material**. Use this option, in combination with the brush parameters, to add material. Using **Add materia**l does not increase the number of triangles, but lifts the existing nodes. The orientation of the model affects the way the area is built up. To undo the add action, use the **Erase** option.

   **Remove material**. Use this option, in combination with the brush parameters, to lower the triangle nodes under the brush, to make it look as though material has been carved out of the model. To undo the remove action, use the **Erase** option.

   **Smooth**. Use this option, in combination with the brush parameters, to smooth out irregularities in the triangle nodes. For each node in the smoothing area, PowerSHAPE calculates a position based on the surrounding nodes.

   **Erase**. Use this option, in combination with the **Strength**, to undo changes that you have made during the current editing session. Set **Strength** to 100% to move the node to its original position in one step. Set the option to 50% to move the node to its original position in two steps.

2. Use the following to adjust the brush:

**Brush diameter**. Click the button and use the slider (or D + mouse scroll wheel) to increase or decrease the size of the brush.

**Strength**. Click the button and use the slider (or H + mouse scroll wheel) to increase or decrease the amount each node is raised or lowered as a percentage of the brush radius.

**Brush Smoothness.** Click the button and use the slider (or M + mouse scroll wheel) to increase or decrease the smoothness of the area you are sculpting. The smoothness determines whether there is any reduction in the strength towards the edges of the brush.

A setting of 0% maintains full strength right to the edge of the brush.

A setting of 100% smooths the material across the diameter of the brush.

Use the slider to enter the value to be associated with the button you have selected.

3. Use the following to specify maximum deviation and length:

   **Limit deviation**. Click the button and enter the maximum deviation that the nodes can move.

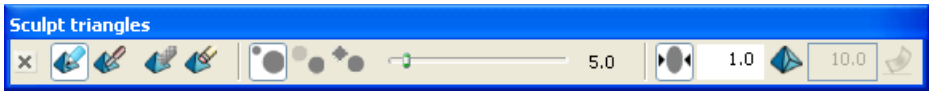
   **Refine triangles**. Click the button and enter the maximum length to be used when refining triangles.

4. Move the cursor over the model to make the required changes. When you have finished editing click × to close the toolbar.

***Show deviations*** *is under development.*

## Sculpt triangles toolbar

The **Sculpt triangles** (see page 2) toolbar has replaced the **Interactive Triangle Editing** dialog box. The toolbar is displayed when you click on the **General Edits** toolbar. By default, the toolbar is floating, but if required, the toolbar can be docked alongside existing toolbars.



## Mesh editing toolbar

A new Z-compensation tool (see page 39) has been added to the **Mesh editing** toolbar.

## Model fixing toolbar

**Repair selected solid** has replaced Make watertight on the Model Fixing toolbar.

With a solid selected, click to display one of the following:

- **Solid Doctor** (see page 15) if the solid is a Parasolid (post-version 8 solids).
- **Make Watertight** if the solid is a PowerSHAPE solid (version 8 solids).

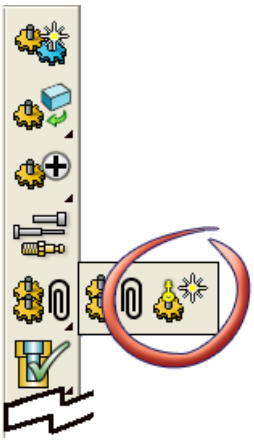
## Solid edit toolbar

Use the new **Solid edit** toolbar (see page 30) to make changes to a single solid.

The toolbar is displayed automatically when you start to edit a solid. It is also displayed by selecting **View > Toolbars > Solid Edit**.

### Assembly toolbar

**Manage attachments** (see page 42) has been added to the **Relation** flyout.



Clicking this button displays the new **Manage attachment** dialog box. This dialogue is also displayed if you select **Manage attachment** from the component popup menus.

## Dialog boxes

The following changes have been made to dialog boxes:

- The **Interactive Triangle Editing** dialog box has been replaced by the Sculpting toolbar (see page 2).
- Use the **Maximum edge length** option on the **Reduce mesh** dialog box (see page 38) to restrict the maximum edge length that should be maintained during the reduce operation.
- Use **Close model during Autofix** to close the model when fixing triangles on export (see page 39).
- Add a component that you have modelled to a custom catalog (see page 40) by selecting **Add new component** (**Manage custom catalogs** dialog box). For clarity, **Create a component** has been renamed **Copy existing component**.
- Create relations (see page 44) and Manage attachments (see page 42) dialog boxes replace the **Relations** toolbar.

## Menus

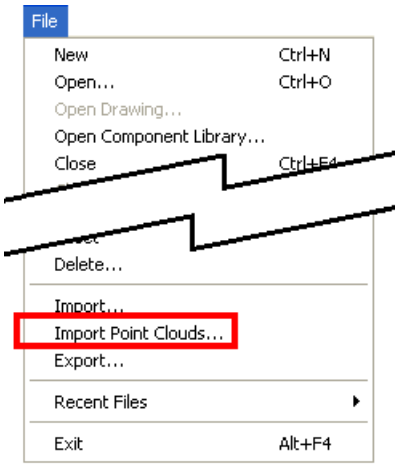
Changes have been made to the following menus:

- File (see page 6)
- Object (see page 6)

- Tools (see page 7)

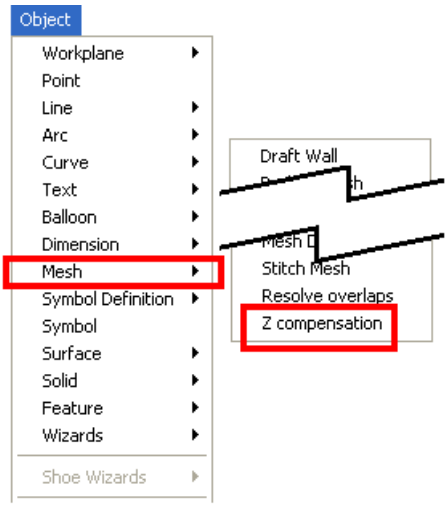
## File menu

**Import Point Clouds** (see page 48) has been added to the **File** menu.




## Object menu

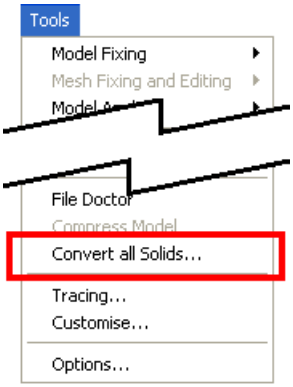
- **Z compensation** has been added to **Object > Mesh** options.



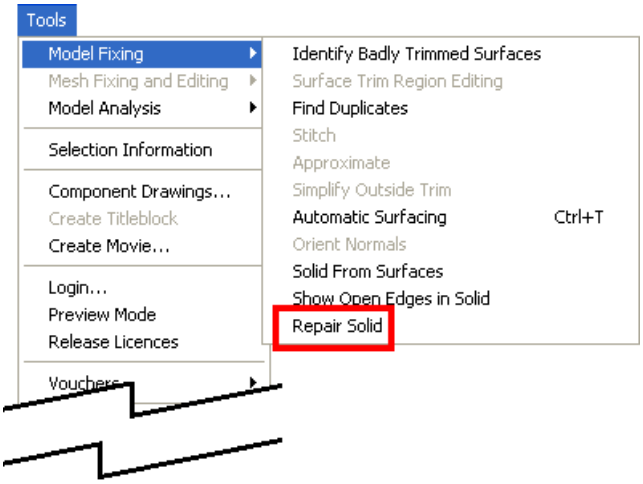
- **Transform** options have been added to **Object > Feature** options.

## Tools menu

- **Tools > Convert all solids** has been added to the menu to let you convert (see page 12) all the solids in a model to or from version 8 solids.



- **Make Watertight** has been replaced by **Tools > Model Fixing > Repair Solid**.




Select this option to display one of the following:

- **Solid Doctor** (see page 15) if the solid is a Parasolid (post-version 8 solids).
- **Make Watertight** if the solid is a PowerSHAPE solid (version 8 solids).

# Solid modelling

The following areas of solid modelling have been updated in PowerSHAPE 2010:


Solid modelling using Parasolid (see page 8)

Converting solids (see page 12)

Fixing solids using the Solid Doctor (see page 15)

Editing solids using the Solid Edit toolbar (see page 30)

Creating transform features (see page 32)

Solid variables (see page 36)


## Parasolid in PowerSHAPE 2010

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

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

The Parasolid kernel is only used with the following products:

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

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

Parasolid - the benefits (see page 9)

Parasolid - do I need to model differently? (see page 10)

Parasolid - overview of model conversion (see page 11)

Parasolid - overview of fixing models (see page 12)

## Parasolid - the benefits

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

- Faster and more robust solid modelling operations. Tests have shown that using the parasolid kernel is up to four times faster than using the PowerSHAPE kernel.
- Faster and more robust drawings.
- Faultless data conversion to and from other parasolid-based modellers. Higher quality transfer to and from other solid modellers because you can ask for parasolid data from customers instead of IGES or STEP.

- *The .psmodel* file size will be much smaller if the part has been created wholly using the parasolid kernel. If there has been any conversion from version 8 solids or to/from surfaces, this will not be the case.
- Import and export of Parasolid file formats will conserve the parasolid data with no loss of quality. At present the recommended formats are:

  Parasolid (x_t, x_b, xmt_txt, xmt_bin)

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

  SolidWorks (sldprt)

  UG NX (prt)
- Crisper, tidier graphics.

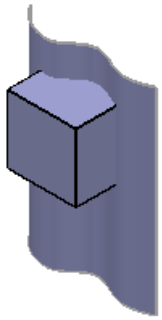
## Parasolid - do I need to model differently?

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

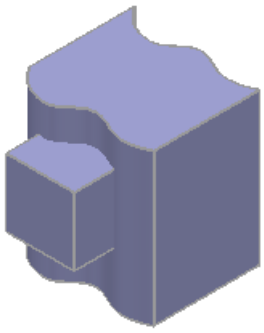
- If you want to boolean an open solid (sheet) to an active solid, and re-trim the active solid to the edges of the sheet,

  do one of the following:

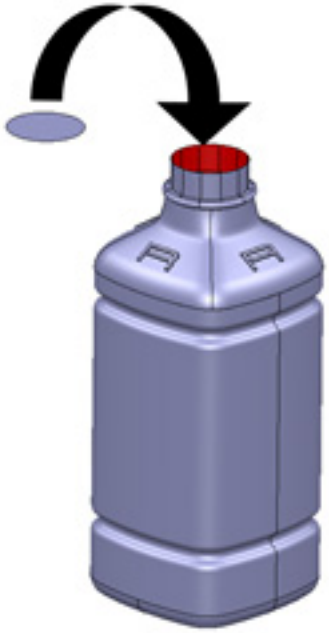
  - Extend the sheet through the active solid.



  - Form a closed solid from the sheet.



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



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

## Parasolid - overview of model conversion

Separate, detailed information is available on converting models to and from an earlier version of PowerSHAPE (see page 12). The following is a summary:

- Existing models that were created using the PowerSHAPE kernel will continue to work in PowerSHAPE 2010

- New models will use parasolid, unless you change the default by selecting the **Version 8 solids** option on the **Tools > Options > Object > Solids** dialog box.
- Parasolid and version 8 solids will co-exist in the same model.
- It is not possible to perform boolean operations between solids of different kernels. One solid must be converted before the operation is attempted.
- It is possible to convert solids in existing models to parasolid and parasolid to version 8 solids.

### Parasolid - overview of fixing models

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

- Parasolid has stricter rules on 'valid' geometry. Geometry that is valid in version 8 models is not necessarily valid in a parasolid.
- Faults in a parasolid body can be viewed and fixed using the Solid Doctor (see page 15).
- Generally, you only need to fix a fault if you are going to perform a modelling operation through the faulty area; for example, performing a boolean subtraction.

## Converting solids

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

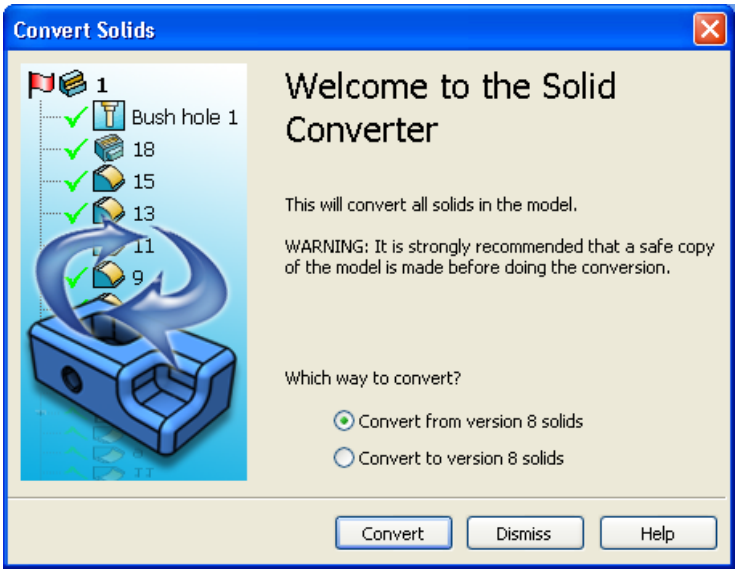
Use the following tools to convert solids to and from version 8 solids

- Use **Tools > Convert all solids** to convert all solids in a model (see page 12).
- Use the **Convert** button on the **Solid Edit** toolbar to convert a single solid (see page 14).
- Use **Edit > Convert > Convert solids to version 8 solids** and **Edit > Convert > Convert solids from version 8 solids** to convert multiple solids.

### Convert all solids

1. Click the model that contains the solids.

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




3. Select the conversion option you require.

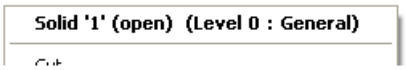
*To check the type of solid in your model, right click on a solid. If a solid is version 8,* ***Version 8*** *is displayed in the title of the solid popup menu.*

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

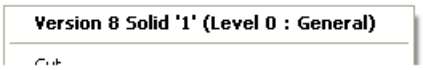
6. Click **Report** to display the **Convert Solids Report** detailing any issues with the conversion.
7. Right click on the converted solid to display the solid popup menu.

This will say:

**Solid** for post-version 8 solids.




**Version 8 Solid** if the solid is version 8.

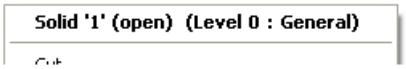



## Convert selected solids

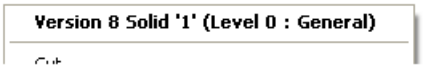
1. Click the solid to be converted to display the **Solid Edit** toolbar.
2. Click .
3. Right click on the converted solid to display the solid popup menu.

   This will say:

   **Solid** for post-version 8 solids.




   **Version 8 Solid** if the solid is version 8.



# Solid Doctor

The Solid Doctor lets you find and fix the faults in a solid. Holes, overlaps, self-intersections and bad trimming of surfaces are classed as faults. These need to be removed as they can cause further solid feature operations to fail.

## To start the solid doctor:

With a solid selected do one of the following to display the Solid Doctor dialog box:

- Select **Tools > Model Fixing > Solid Doctor**.
- Click on the **Solid Edit** toolbar.

## For further details, see:


- Using the Solid Doctor (see page 15)
- Solid Doctor dialog box (see page 24)
- List of fault options (see page 28)
- Doctor Edit toolbar (see page 29)
- Tips on using Solid Doctor (see page 29)

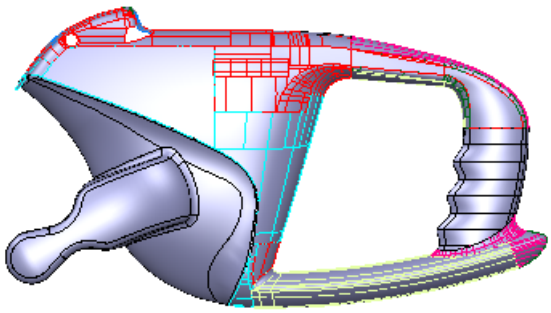

## Using the Solid Doctor

Use the **Solid Doctor** to find and fix the faults in a solid. The following are classed as faults:

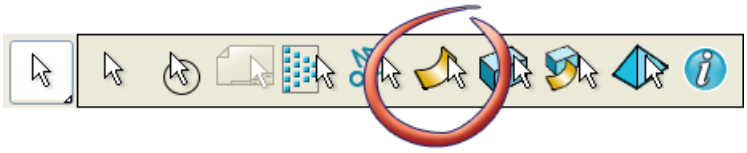
- holes
- overlaps
- self-intersections
- bad trimming of surfaces

These faults need to be removed because they may cause further solid feature operations to fail.

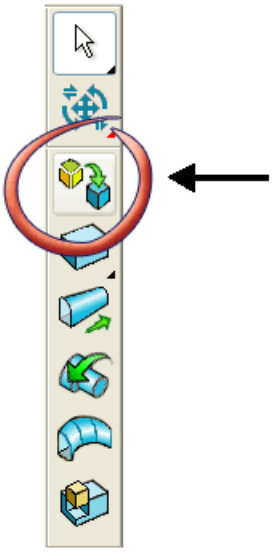
The following model is used to illustrate creating a faultless solid.



1. Click **Quick select all surfaces** .



2. Click **Solid** .
3. Click **Create solid from selected surfaces or meshes** .

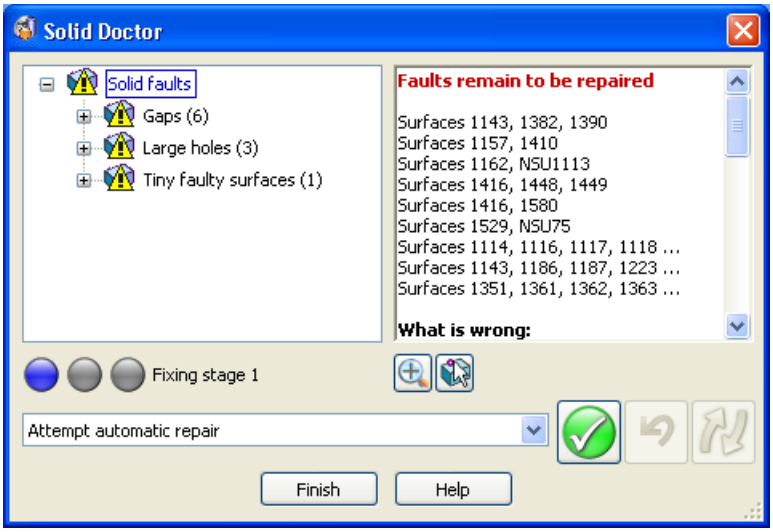


4. If you are asked if you want to fix the faults in the solid, click **Yes** to display the **Solid Doctor**.

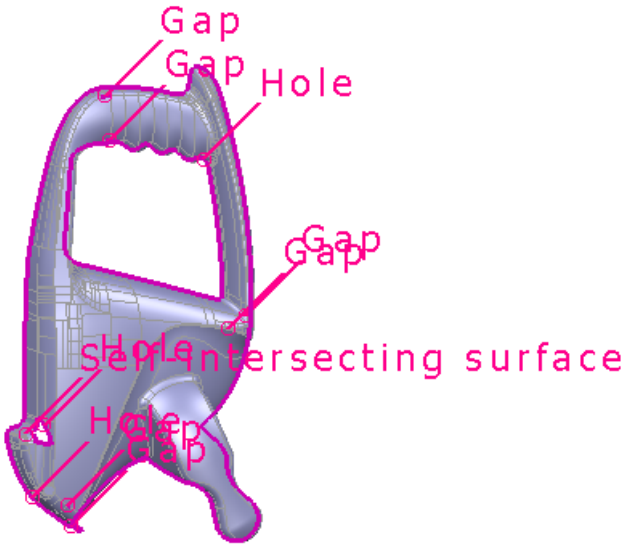
   If this message has not been displayed start **Solid Doctor** using one of the following ways:

   - Select **Tools > Model Fixing > Solid Doctor**.
   - Click on the **Solid Edit** toolbar.

The Solid Doctor dialog box is displayed.




The faults are listed in the fault tree and displayed in pink on the model.

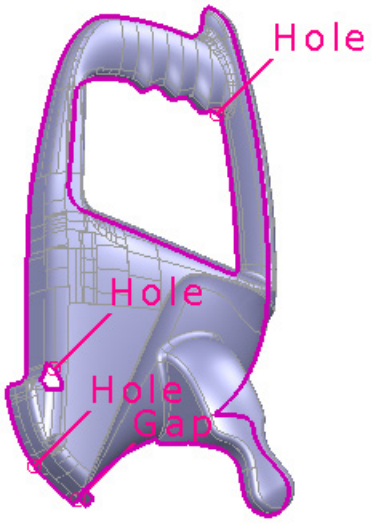



5. Ensure that **Attempt automatic repair** is selected from the **Repair options** drop down list.

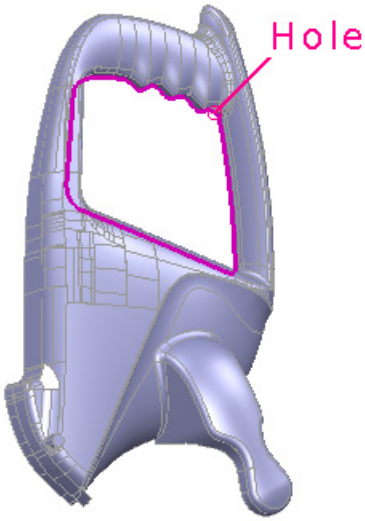
6. Click to process the repair.

   On the model, the faults that have been repaired and the corresponding fault labels turn green. **Fault has been repaired** is displayed in green at the top of the information window in the dialog box.

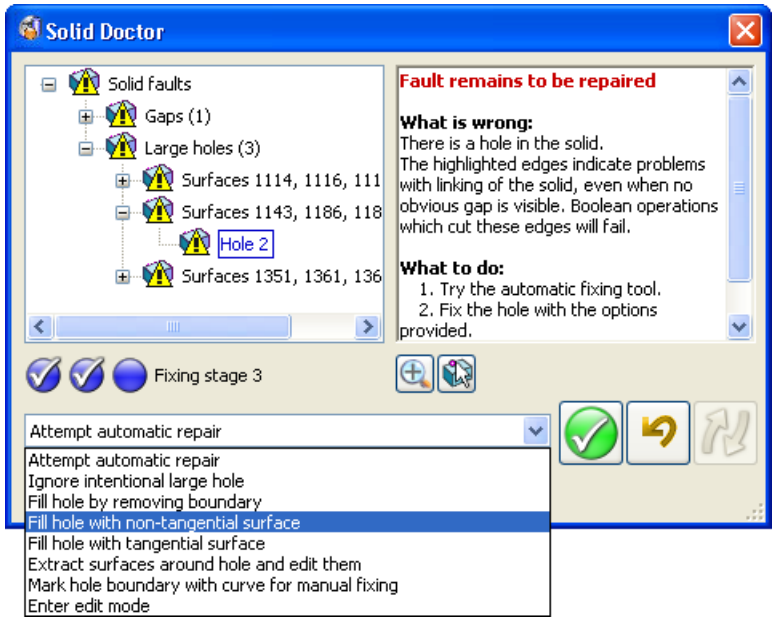
7. Click to re-check the model. Only the faults that still need fixing are displayed. Comparing this to the original shows that many of the faults have been fixed using the automatic repair feature.




8. On the model, click the hole at the top of the model to select that hole for fixing.

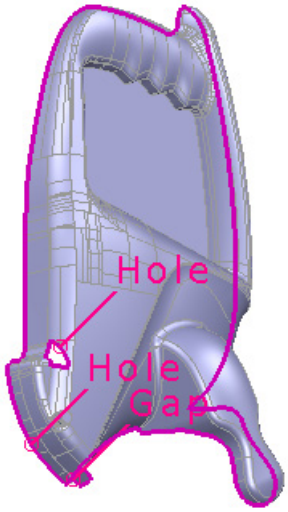


9. Select **Fill hole with non-tangential surface** from the repair option drop-down list.

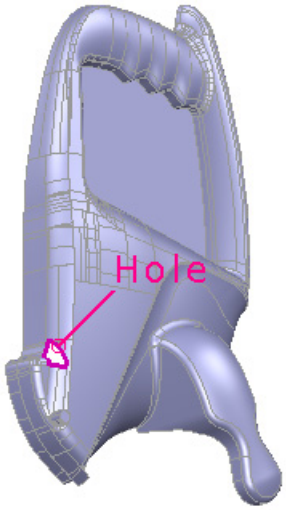



10. Click .

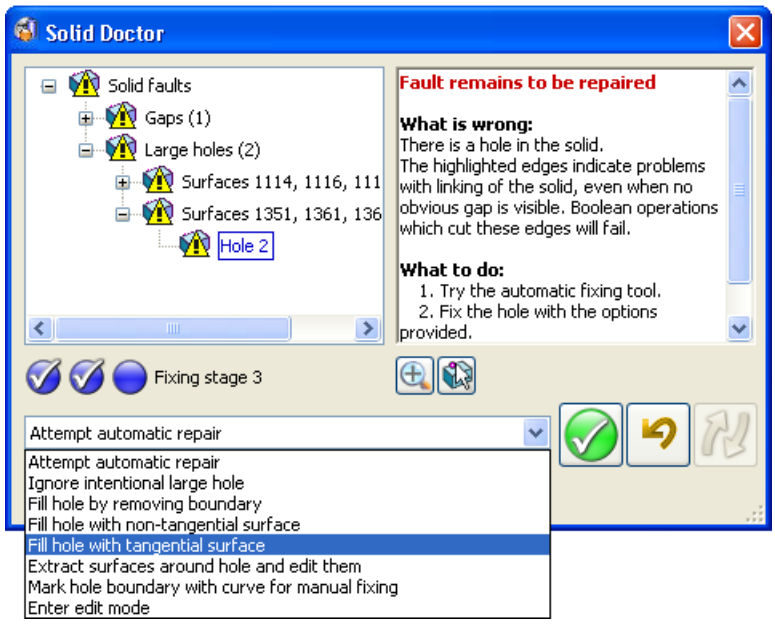
11. Click to re-check the model.



12. Click the hole on the front of the model to select that hole for fixing.



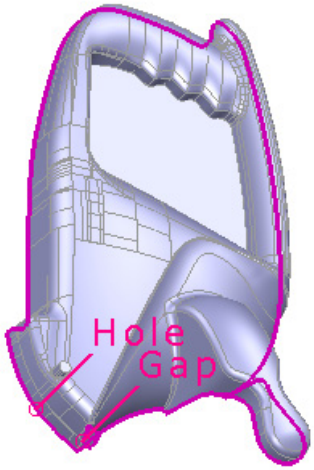

13. Select **Fill hole with tangential surface** from the repair option drop-down list.



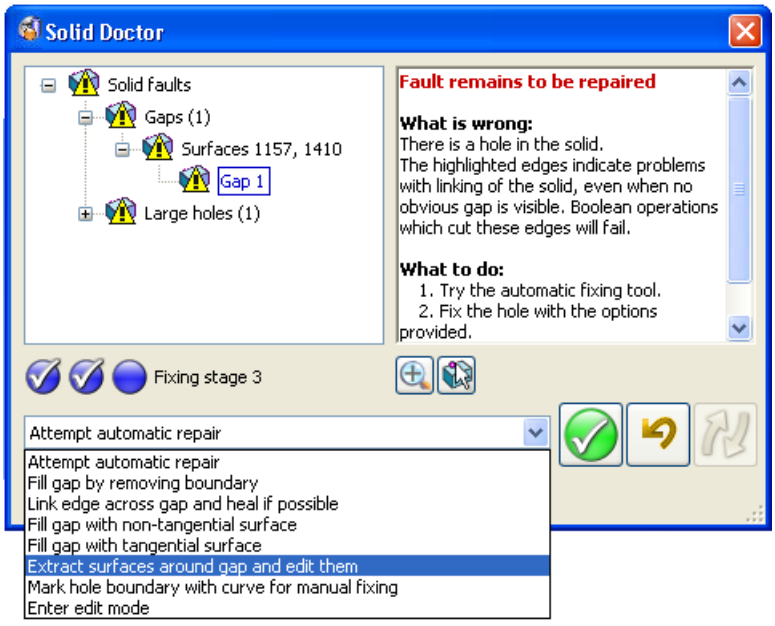

14. Click .

**Fault has been repaired** is displayed at the top of the information window in the dialog box.

15. Click to re-check the model. The model shows the remaining faults: a **Gap** and a large **Hole** round the outside. The **Gap** is in fact an overlap because it extends beyond the solid.

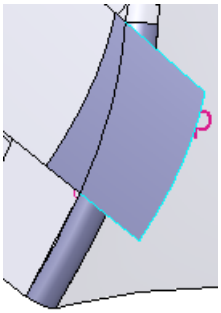



16. Select **Gaps (1)** from the tree in the dialog box.
17. Click **Extract surfaces around gap and edit them** from the **Repair Options** drop down list.

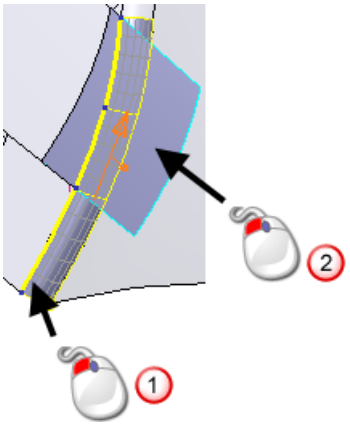



18. Click .

19. Zoom around the gap in the model window.



20. Convert the surface below to wireframe and select the composite curve ①.




21. Select **Limit selection** from **Limit** flyout on the **General edits** toolbar.

22. Select the surface ②.

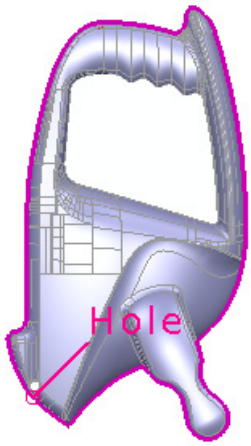
23. Click **Accept** on the Doctor Edit toolbar.

If necessary re-select the surfaces that need to be included back in the solid

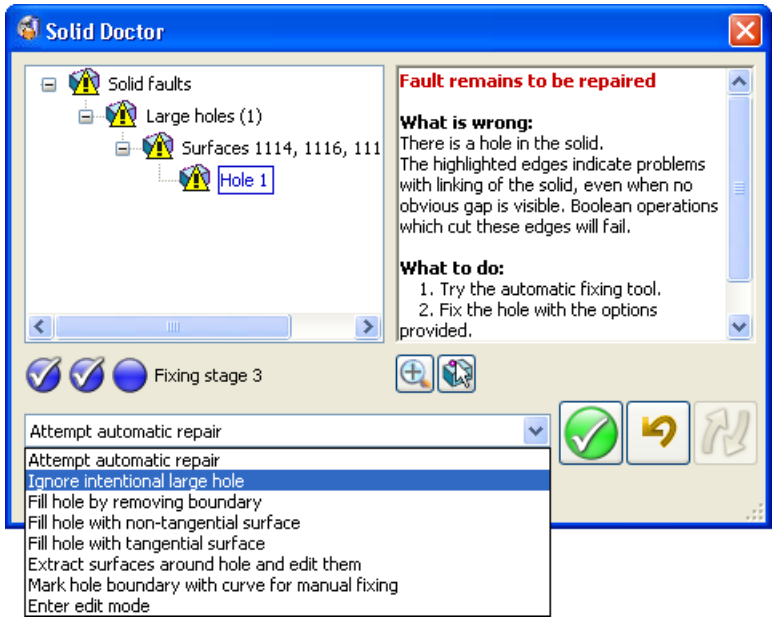
24. Click **Apply** .

25. Click to re-check the model.

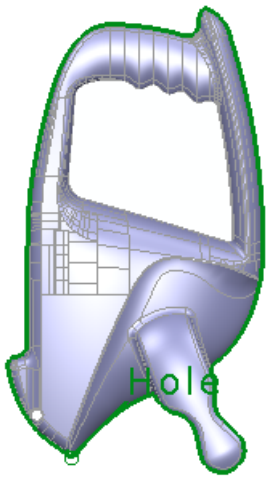
The outer edge of the solid is now marked as a hole.



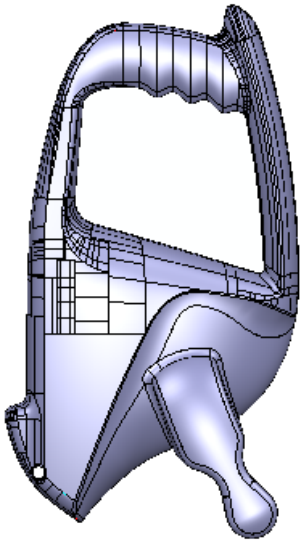

26. Select **Ignore intentional large hole** from the repair options.




27. Click . *Intentional hole in solid* is displayed at the top of the information window and the **Hole** label is displayed in green.

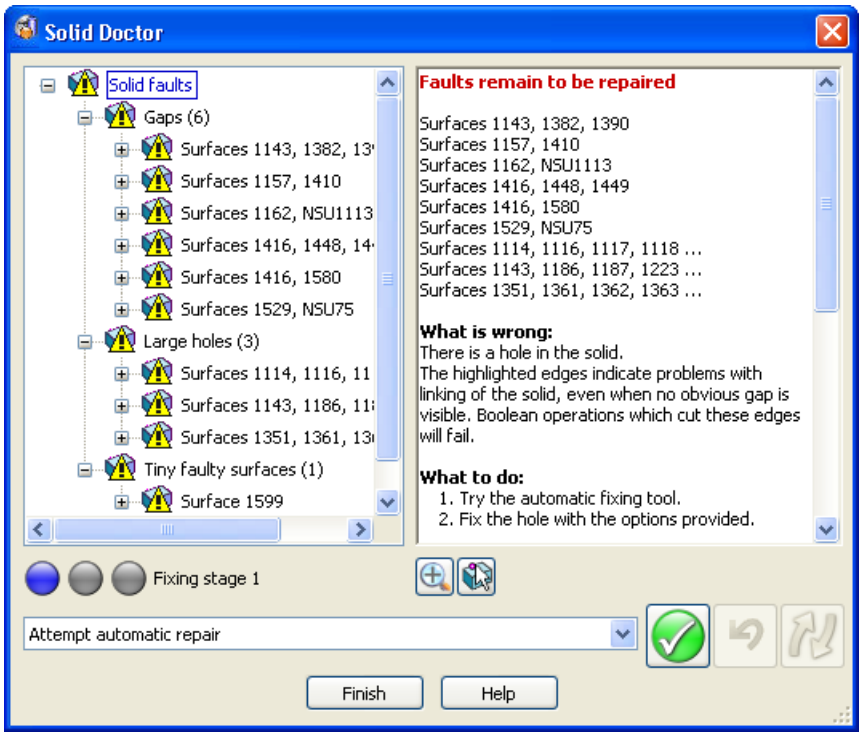


28. Click **Finish**.



## Solid Doctor dialog box

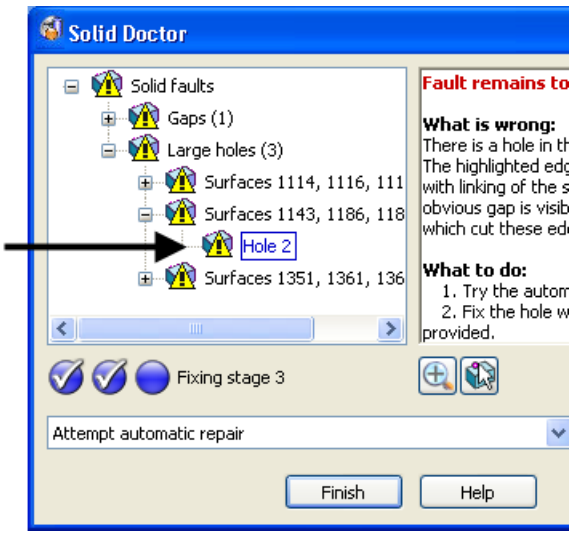
Use the dialog box to fix all the faults in a solid by removing holes and repairing the trimming of the faulty surfaces. Resize the dialog box as required, so that you can see the faults.



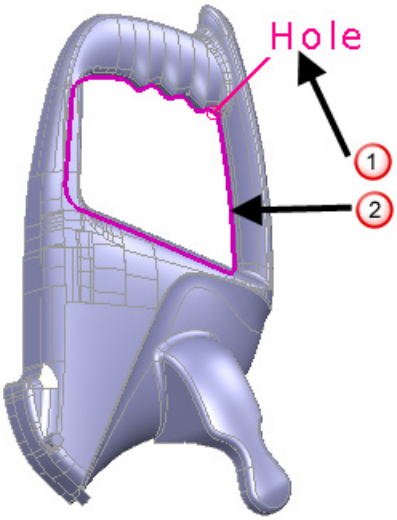
**Fault Tree** is displayed in the left panel of the dialog box and lists the faults in the solid.

Use one of the following methods to select faults in the Fault Tree or the model window:

- If you select a fault in the tree, it will be highlighted in the model.



- If you select a fault label ① or fault annotation ② in the model it will be highlighted in the tree.

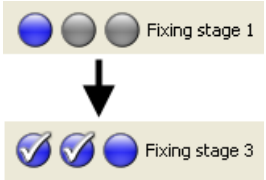



**Fault Information** for the selected fault is displayed in the right panel of the dialog box.

The following fault information is provided:

- **What is wrong:** shows a description of the fault.
- **What to do:** gives a suggested repair procedure.
- **Failed repair operations** lists the fixing operation that failed.

**Fixing stage** graphics show the progress on fixing faults.



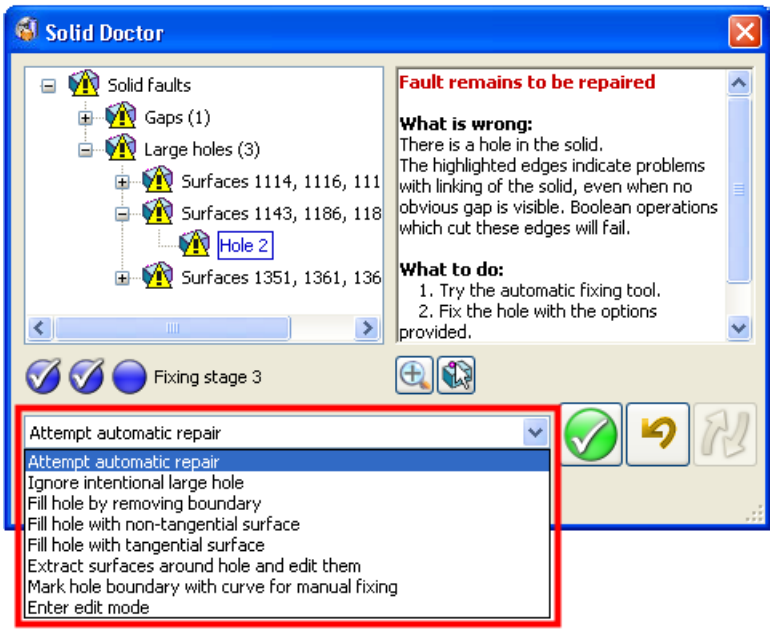

- Click to enlarge the view of the selected fault. Zoom out to view the whole solid.

- Choose how you want to display the faults in the model window.

displays the selected fault in the model window.

displays all the faults in the model window.

**Repair Options** - Select a repair option from the list of available repair operations. The default option is **Attempt automatic repair**. For further details see List of fault options (see page 28). Depending on the repair option, the Doctor Edit toolbar (see page 29) is displayed.

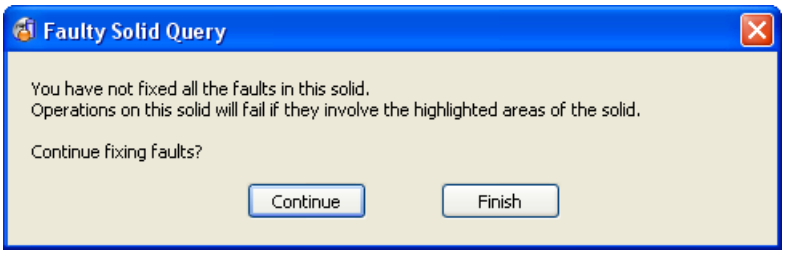



- Click to apply the repair option to the selected faults.

- Click to undo the last repair operation.

- Click to re-check the solid for faults.

**Finish** - Click to close the dialog box and re-check for faults. If faults are found the **Faulty Solid Query** dialog box is displayed.




**Continue** - Click to open the **Solid Doctor** dialog box to continue fixing faults.

**Finish** - Click to accept the changes made by the Solid Doctor.

### *List of fault options*

This lists the repair options displayed on the Solid Doctor dialog box. The repair options relevant to the selected fault are displayed in the **Repair Options** drop down list.

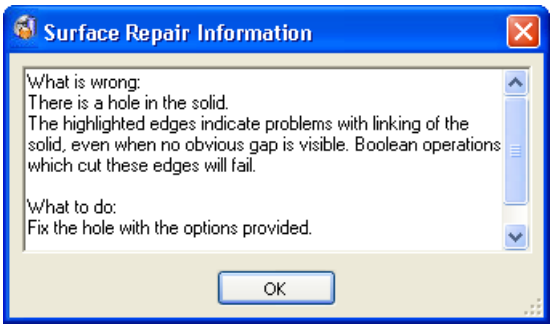
| Repair option | Action |
|---|---|
| ▪ Attempt automatic repair | Automatically fixes the fault. This combines other standard listed options to try to repair the solid. |
| ▪ Extract surfaces and edit them<br>▪ Extract surfaces around gap and edit them<br>▪ Extract surfaces around hole and edit them<br>▪ Extract surfaces at edge and edit them | Removes surfaces of the fault. Enters surface edit mode to fix the surfaces. |
| ▪ Delete surfaces | Deletes the faulty surface. |
| ▪ Delete selected surfaces | Deletes highlighted surfaces |
| ▪ Fix surface | Enters surface edit mode when a surface is selected. |
| ▪ Add selected surfaces to solid | Adds the selected surfaces and/or solids to the main solid to fill the open hole. |
| ▪ Fill gap with tangential surface<br>▪ Fill gap with non-tangential surface<br>▪ Fill hole with tangential surface<br>▪ Fill hole with non-tangential surface | Creates a fill-in surface for the selected hole in the solid. |
| ▪ Fill gap by removing boundary<br>▪ Fill hole by removing boundary | Extends the surfaces adjacent to the hole to try to fill the hole. |
| ▪ Link edge across gap and heal if possible | Heals the open edge. This may increase the solid tolerance. |
| ▪ Ignore intentional large hole | Marks the selected large hole as intentional. |

| | |
|---|---|
| ▪ Mark problem area and leave faulty surface in solid<br>▪ Mark hole boundary with curve for manual fixing | Creates composite curves around the faulty surfaces and holes. |
| ▪ Global fault with unknown fix<br>▪ Select individual fault | Not currently available. For future development. |
| ▪ Enter edit mode | Enters surface edit mode without changing the solid. |
| ▪ Fault repaired | Displays if the fault has been repaired. |

## Doctor Edit toolbar

This toolbar is displayed when you use the surface creation and editing tools to fix the surface before adding it back into the solid. The necessary curve and surface creation and editing tools are displayed.

- Click to display the **Surface Repair Information** dialog box, that displays details of suggested repair operations.




- Click to accept the surface edit changes. If a surface is selected, **Solid Doctor** tries to add the surface back into the solid.

- Click to cancel any surface edit changes.

## Tips on using Solid Doctor

It is useful to remember the following points when using Solid Doctor:

- When a fixing problems, look at the problem first and ask yourself how you would fix that problem using surfaces. Then make/repair the surfaces that way and sew them back into the solid.
- Run automatic fixing as the first step. Select the top entry in the fault tree and click It won't fix all the problems, but it will usually drastically reduce the number of faults that need fixing using other options.
- Use often to re-check the solid.
- Only selected surfaces are sewn into the solid when you leave Doctor Edit mode. You will be prompted if you forget to select a surface to be used.
- Gaps can be overlaps.
- If there are any tiny gaps between surfaces, consider removing the surfaces that are adjacent to the fault. Use one of the hole filling options to fix the fault.
- Use **Shift** and **CTRL** to add and remove surfaces from the selection.

## Using the Solid Edit toolbar

The **Solid Edit** toolbar is displayed automatically when you start to edit a solid. It is also displayed by selecting **View > Toolbars > Solid Edit**. Functionality that is accessed from the toolbar has been removed from the solid popup menu, solid tree popup menu and **Object > Solid** menu.

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

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

shows that the selected solid is active. Click the button to make the solid inactive .

Click this button and then select the individual face of a solid.

Click this button and click on a face in the solid. All the faces of the feature that the face belongs to will be selected.

Click this button and click on a face in the solid to select all the faces in the convex region of the face.

Click this button and click on a face in the solid to select all the faces in the concave region of the face.

Click this button and select one or more faces to create surface copies. This replaces **Copy Surface** on the solid popup menu.

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

(Only available for post-version 8 solids) Click this button to display the **Sew** dialog box. Select one or more surfaces or solids and click **OK**. This option replaces **Object > Solid > Sew**.

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

- **Solid Doctor** (see page 15) if the solid is a Parasolid (post-version 8 solids).
- **Make Watertight** if the solid is a PowerSHAPE solid (version 8 solids).

Click this button to convert to/from version 8 solid (see page 12).

Click this button to convert the solid to surfaces. This replaces **Convert** on the solid popup menu.

Click this button to create a mesh from the selected solid. This replaces **Convert to mesh** on the solid popup menu.

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

# Solid transform feature

New functionality has been added to PowerSHAPE 2010 to create solid transform features on solids so that its values can be changed, or the transform suppressed, unsuppressed or deleted. This means that the information about changes of position, rotation, scaling or mirroring are stored with the solid.

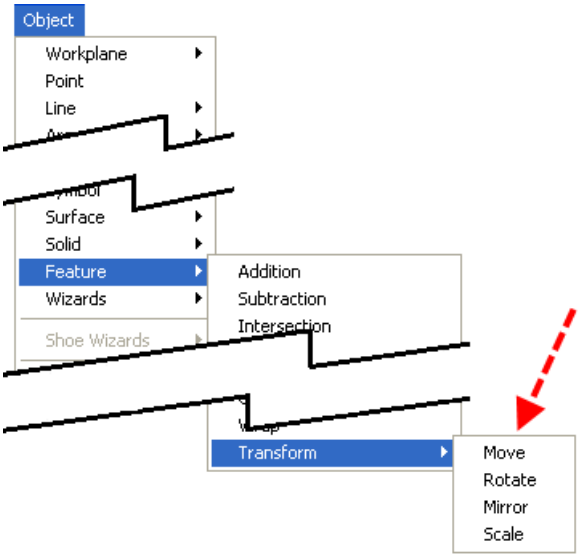
## Creating transform features

Transform features can be created by:

- Selecting one or more solids and selecting **Move**, **Rotate**or **Mirror** from the **General Edits** toolbar when the option **Tools > Options > General Edits > Create solid transform features** is selected.

  *Scale transform feature will* not *be created if you select Scale from the General Edits toolbar, even if the* ***Tools > Options >General Edits > Create solid transform features*** *is selected.*

- Selecting one of the options on the **Object > Feature > Transform** menu. These options will create a transform feature on the active solid and let you enter the parameters and show a preview of the transformed solid.




The following specific dialog boxes are displayed when using this method to create the transform feature:

Move feature (see page 34)

Rotate feature (see page 34)

Mirror feature (see page 35)

Scale feature (see page 35)

## Editing a transform feature

1. Use one of the following methods to display the relevant transform feature dialog box.
   - Double click the transform feature in the solid tree.
   - Select **Modify** from the feature's popup menu.

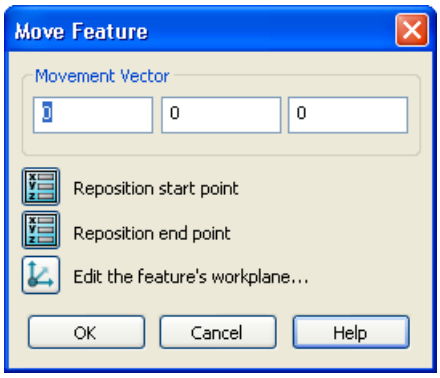
   The transform feature dialog boxes will be displayed irrespective of how the transform feature was created.
2. Use the dialog box to modify the feature.
3. Click **OK** to make the changes.

## Tips when using transform features

- When using the General Edits toolbar, transform features will not be created on solids that do not have history trees, whatever the setting on **Tools > Options > General Edits**.

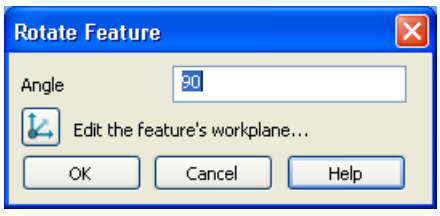
- If you use **Copy original**, the original won't have a feature associated with it, but the copies will.
- Features below a transform feature in the solid tree are not transformed. When modifying an earlier feature it will appear in its original state.
- Transform features can be suppressed, deleted and reordered. If the transform is suppressed or deleted, the solid will revert to its original state (before transformation)

## Move feature




1. **Movement Vector** defines the distance to be moved along each axis. These axes refer to the feature's workplane.
2. To define the translation by clicking on the model, use to display the **Position** dialog box to **Reposition start point** and **Reposition end point**.
3. Click to edit the feature's workplane using the **Workplane** dialog box. The feature's workplane acts as a reference point for the transform.
4. Click **OK** to create the transform feature. The move feature is added to the feature tree.

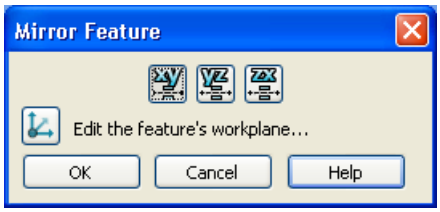
## Rotate feature




1. Enter an angle of rotation as measured around the z axis of the feature's workplane. A preview of the rotated solid is displayed with the existing position.

2. Click to edit the feature's workplane using the **Workplane** dialog box. The feature's workplane acts as a reference point for the transform.
3. Click **OK** to create the transform feature. The rotate feature is added to the feature tree.

## Mirror feature

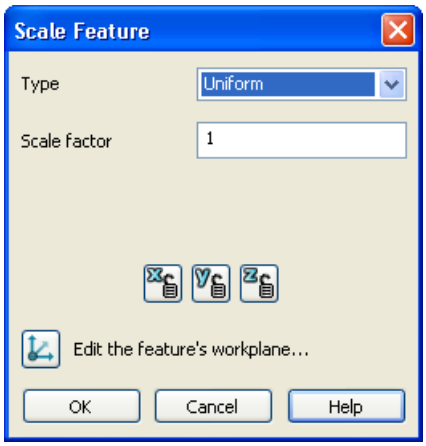



1. Use to change the plane in which the mirror transform takes place.
2. Click to edit the feature's workplane using the **Workplane** dialog box. The feature's workplane acts as a reference point for the transform.
3. Click **OK** to create the transform feature. The mirror feature is added to the feature tree.

## Scale feature

A scale feature can only be created if you select **Object > Feature > Transform > Scale** to display the **Scale Feature** dialog box. It will *not* be created if you select from the General Edits toolbar, even if the **Tools > Options >General Edits > Create solid transform features** is selected.

Use this dialog box to specify the scale feature.



1. Use the drop down list to select the **Type** of scale to be used (**Uniform**, **Non-uniform** or **Projected volume**).
2. Enter a single **Scale** factor for **Uniform** scaling. For **Non-uniform** scaling enter scale factors for X, Y and Z. For scaling **Projected volume**, enter values for **Original volume** and **Desired volume**.
3. Use to lock the scale factor to 1 for a particular axis. The X, Y and Z scale factors refer to the axes of the feature's workplane.
4. Click to edit the feature's workplane using the **Workplane** dialog box. The feature's workplane acts as a reference point for the transform.
5. Click **OK** to create the transform feature. The scale feature is added to the feature tree.

*The results that are achieved using transform features may not necessarily be as expected. This is because functionality such as* ***Scaling Constraints*** *is not applied. Therefore, if it is essential that the original behaviour is ensured, deselect the* ***Create solid transform feature*** *option on the* ***Tools > Options > General edits*** *dialog box.*

## Solid variables

The following new solid variables have been added:

solid.active - returns the name of the currently active solid.

solid[N].parasolid - returns *1* if the solid is a parasolid, else *0*.

solid[N].v8 - returns *1* if the solid is a version 8 solid, else *0*.

solid[N].surface[M].id - returns the id number of the Mth surface of solid N, or the representation number if a parasolid solid.

solid[N].surface[M].name - returns the name of the Mth surface of solid N. This is the same as the existing command solid[N].surface[M].

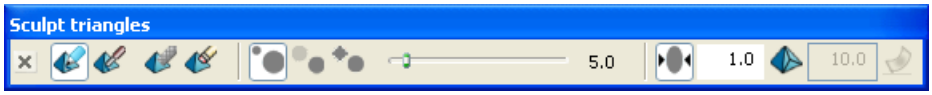
# Modelling with triangles

Additions have been made to the following areas of triangle modelling in PowerSHAPE 2010.

Sculpting triangles (see page 2)
Reduce mesh dialog box (see page 38)
Mesh editing toolbar (see page 39)
Importing point data (see page 48) (Delcam CopyCAD only)
Options (see page 39)

## Sculpt triangles

The **Sculpt triangles** toolbar has replaced the **Interactive Triangle Editing** dialog box. The toolbar is displayed when you click on the **General Edits** toolbar. By default, the toolbar is floating, but if required, the toolbar can be docked alongside existing toolbars.




1. Select one of the following operations:

   **Add material**. Use this option, in combination with the brush parameters, to add material. Using **Add materia**l does not increase the number of triangles, but lifts the existing nodes. The orientation of the model affects the way the area is built up. To undo the add action, use the **Erase** option.

   **Remove material**. Use this option, in combination with the brush parameters, to lower the triangle nodes under the brush, to make it look as though material has been carved out of the model. To undo the remove action, use the **Erase** option.

   **Smooth**. Use this option, in combination with the brush parameters, to smooth out irregularities in the triangle nodes. For each node in the smoothing area, PowerSHAPE calculates a position based on the surrounding nodes.

   **Erase**. Use this option, in combination with the **Strength**, to undo changes that you have made during the current editing session. Set **Strength** to 100% to move the node to its original position in one step. Set the option to 50% to move the node to its original position in two steps.

2. Use the following to adjust the brush:

   **Brush diameter**. Click the button and use the slider (or D + mouse scroll wheel) to increase or decrease the size of the brush.

**Strength**. Click the button and use the slider (or H + mouse scroll wheel) to increase or decrease the amount each node is raised or lowered as a percentage of the brush radius.

**Brush Smoothness.** Click the button and use the slider (or M + mouse scroll wheel) to increase or decrease the smoothness of the area you are sculpting. The smoothness determines whether there is any reduction in the strength towards the edges of the brush.

A setting of 0% maintains full strength right to the edge of the brush.

A setting of 100% smooths the material across the diameter of the brush.

Use the slider to enter the value to be associated with the button you have selected.

3. Use the following to specify maximum deviation and length:

   **Limit deviation**. Click the button and enter the maximum deviation that the nodes can move.

   **Refine triangles**. Click the button and enter the maximum length to be used when refining triangles.

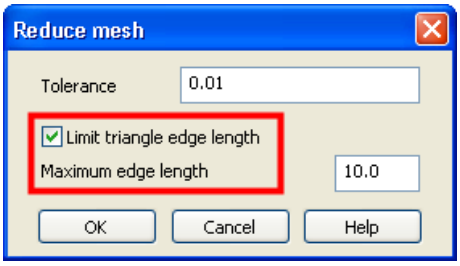
4. Move the cursor over the model to make the required changes. When you have finished editing click × to close the toolbar.

***Show deviations*** *is under development.*

## Reduce mesh dialog box

Use the **Maximum edge length** option on the **Reduce mesh** dialog box to define the maximum edge length that should be maintained during the reduce operation.

1. Select **Limit triangle edge length** to display the option.

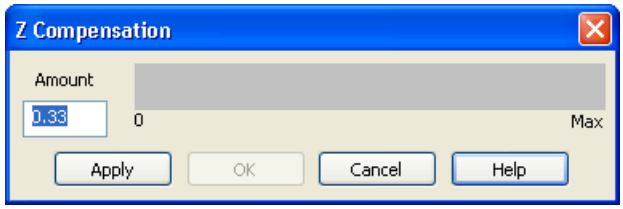



2. Enter the maximum length of the triangle edge.

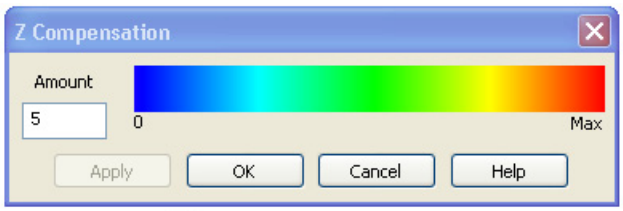
## Z-compensation tool

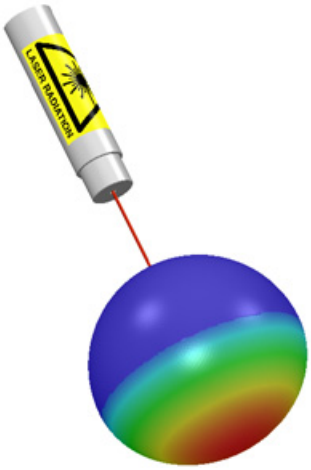
Use the **Z-compensation** tool to cancel the overcure effect when preparing meshes for manufacture on stereolithography machines.

1. Create a solid sphere.
2. Select (Solid editing toolbar) to convert the solid to triangles.
3. Select the mesh to display the **Mesh editing** toolbar.
4. Click to display the **Z compensation** dialog box.




5. Enter an **Amount** and press the tab key.
6. Click **Apply** to display the model with the compensation applied.



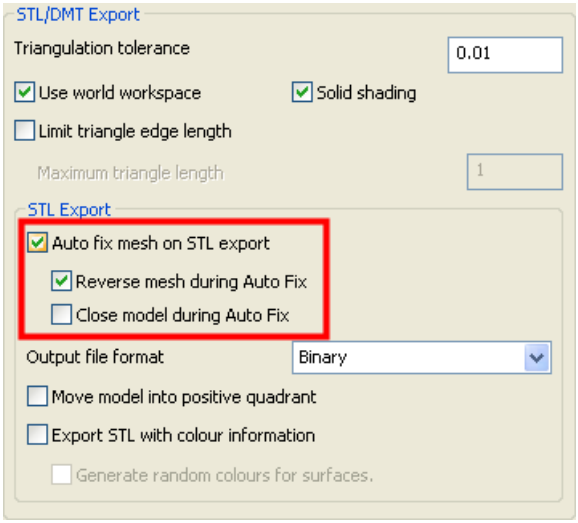

*In the example a large value has been used to exaggerate the effect.*

## Options

Use the new **Close model during Autofix** to close the model when fixing triangles on export.

1. Select **Tools > Options > Data Exchange > STL/DMT**.

2. Select **Auto fix mesh on STL export** to display the options:




**Close model during Autofix** is deselected by default.

# Assembly modelling

Changes have been made to the following areas of Assembly modelling:

Component wizard (see page 40)

Manage attachments (see page 42)

Create relations (see page 44)

Other changes (see page 47)

## Component wizard

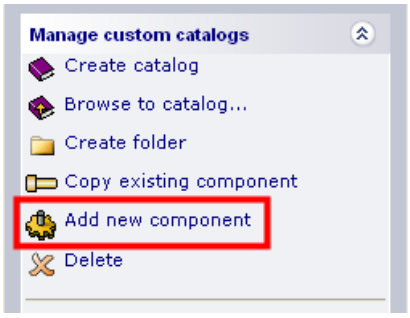
The concept of **Custom catalogs** was introduced in PowerSHAPE 8.0. It is now possible to add a user-modelled component to a custom catalog.

*For clarity,* ***Create a component*** *has been renamed* ***Copy an existing component****.*

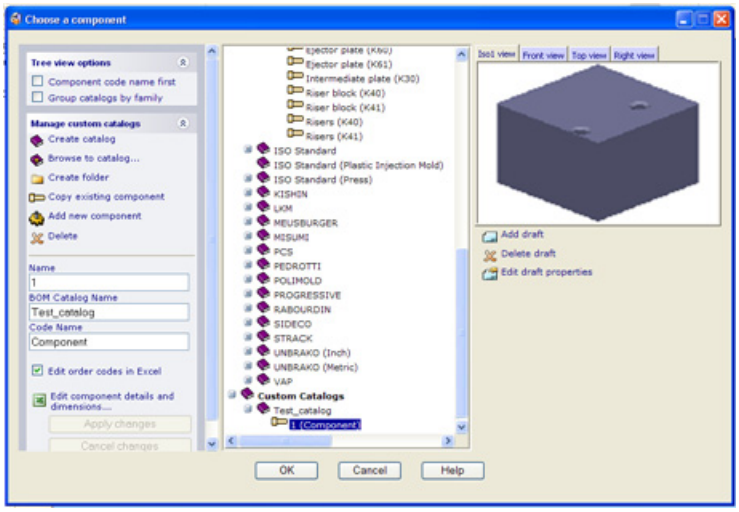
### Adding a user-modelled component:

With the source component displayed:

1. Click **Add new component**.

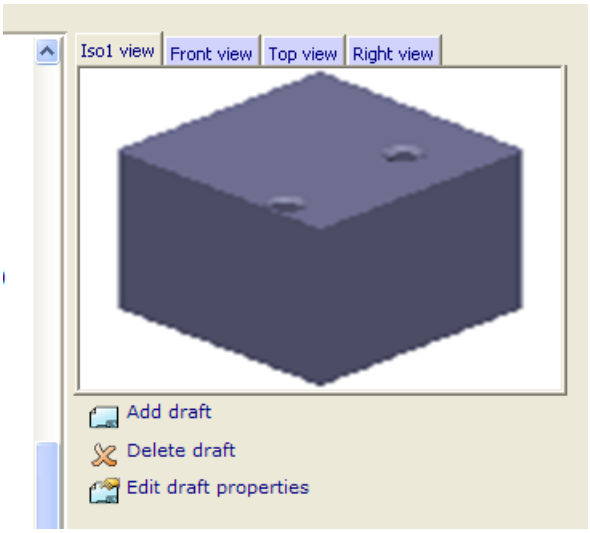



2. Select the following:
   - Source component on the model.
   - **Destination folder**.
3. Click **Use selected**.
4. Click **Add** to add the component to the custom catalog. The dialog box updates to display the component:




5. Click **Edit component details and dimensions** to display the MS Excel spreadsheet for the component.
6. Change the name and dimensions in the spreadsheet as required. Switch back to PowerSHAPE.
7. Click **Apply changes** to:
   - Update the **Custom Catalogs** tree to reflect any component name change.
   - Close the spreadsheet.

- Display the component in the graphic window on the dialog box.



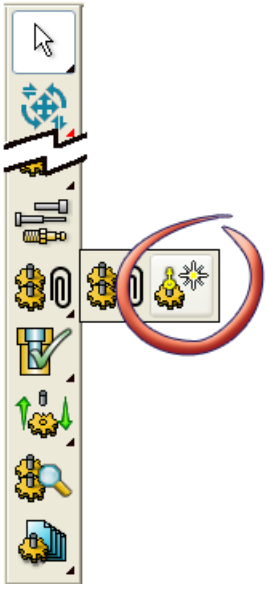

8. Use the following options as required:
   - Click to **Add draft**. Choose the required graphic from the **Select a Draft** dialog box. The selected graphic is added to a **Draft** tab.
   - To delete a tab, select the tab and click to **Delete draft**.
   - To change the title of a tab, select the tab and click **Edit draft properties.** Enter the new tab **Title** and click **OK** (just below the **Title**) to change the tab.
9. Add further user-defined components as required.
10. When all components have been added to the custom catalog, click **OK** to finish.

## Managing attachments

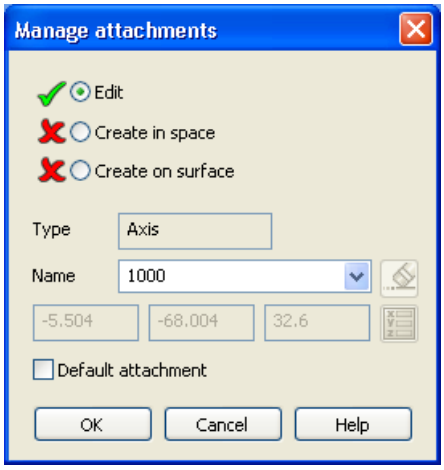
Use the new **Manage attachment** dialog box to:

- edit existing attachments.
- create new attachments.

With a component selected, click on the **Relation** flyout (Assembly toolbar). Alternatively, select **Manage attachment** from the component popup menus.



The **Manage attachment** dialog box 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 box 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 box. Use the following options to modify the attachment

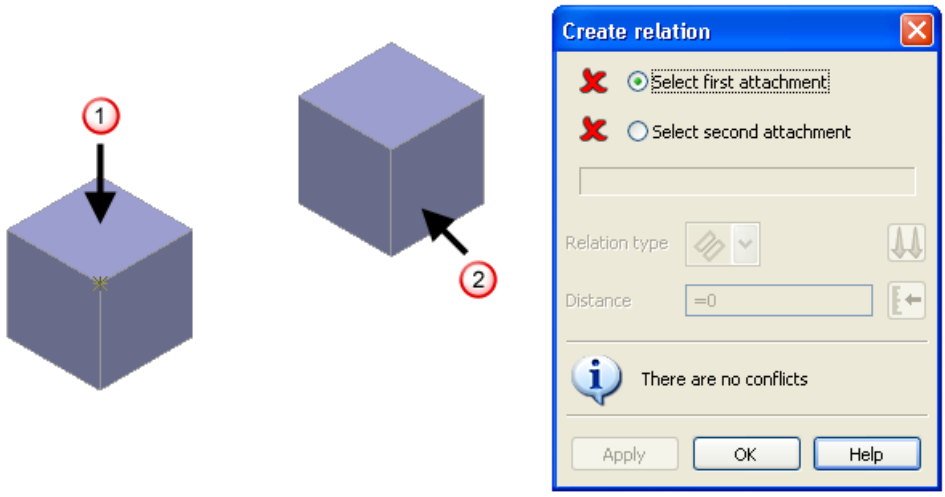
1. By default, the new attachment is named *Unnamed*. Enter a new name as required.
2. Edit the **X**, **Y** and **Z** coordinates or click to enter the coordinated using the **Position** dialog box.
3. Click to remove the attachment that you have just created.
4. Click **Default attachment** to make the new attachment the default.

## Creating relations using attachments

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

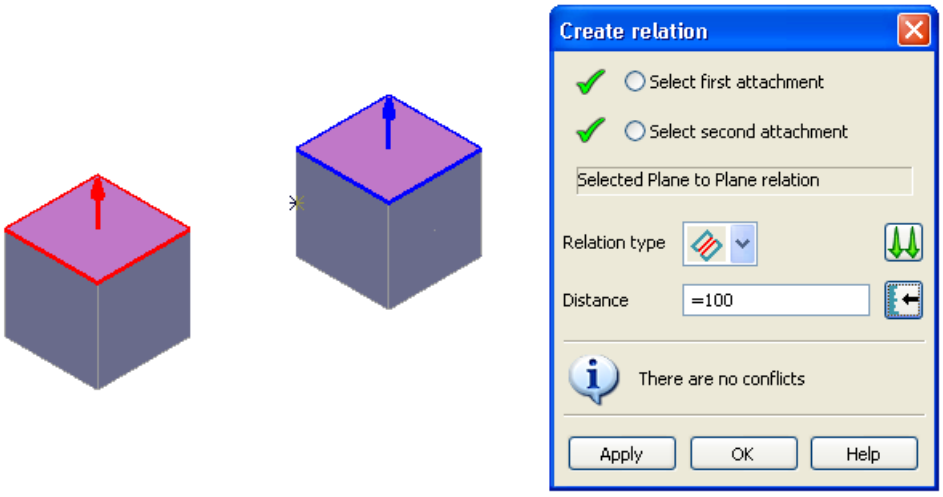
1. Click on the **Relations** flyout (Assembly toolbar)

The following simple example illustrates the options on the dialog box:

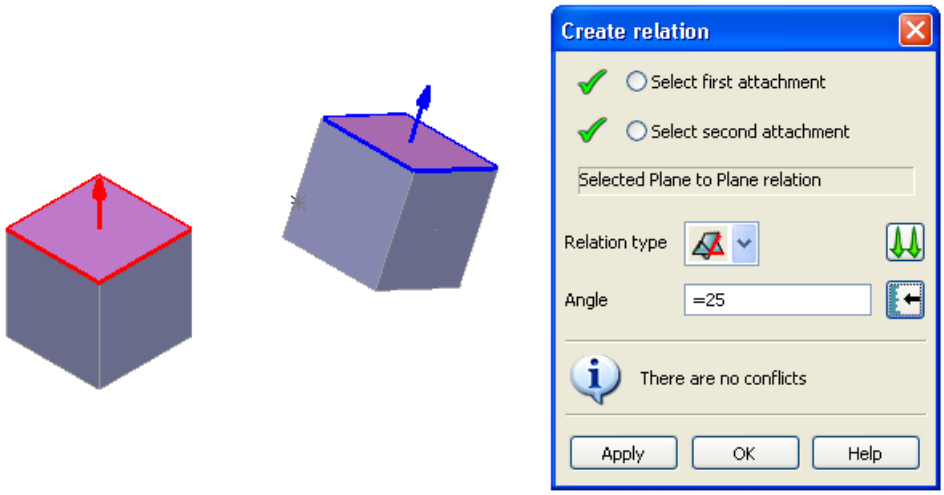



2. Select the first attachment ①. This is the master attachment. ✖ changes to ✔.
3. Select the second attachment ②. This is the slave attachment. The model and the dialog box are updated as follows:
   - A **Coincident Plane to Plane relation** is created between the two components. *Selected Plane to Plane relation* is displayed in the dialog box 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 box to specify the distance. The model will be updated.

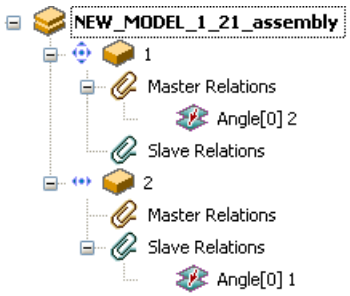



5. Click to display **Relation type** options.
6. Select to change the **Relation type** to **Angular**.
7. Enter 25 as the **Angle**.




8. Click to **Anti-align** 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 box 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 box, delete the relation from the assembly tree and then create the relation using the correct selection.*

** With the dialog box still displayed, use Select first attachment and Select second attachment to modify the relation.*

# Other changes to Assembly modelling

The following changes have been made to the general operation of Assembly modelling:

- Use the **Modify > Parameters** option on the component popup menu to modify multiple parametric components that are selected in the assembly tree.

## Interface

- **Default attachment** has been removed from the component popup menus.
- Colours and styles for highlighting attachments and components have been modified to improve clarity.

## Assembly with Parasolids

- Problems with lost attachments that may occur when converting component solids are reported in the conversion report.
- Import assemblies from Parasolid XT files.

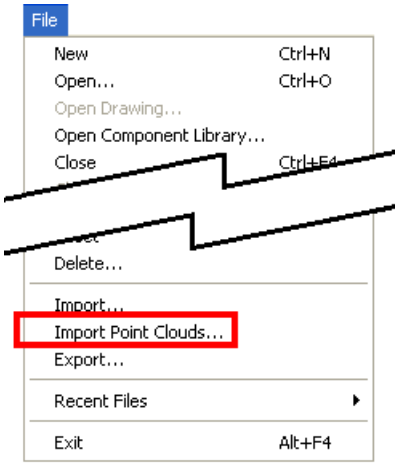
# Delcam Draft

The following change has been made to Delcam Draft:

- If the drawing sheet numbers are not consecutive after a sheet number has been changed, renumber the sheets automatically by selecting *Yes* when asked if you wish to renumber.

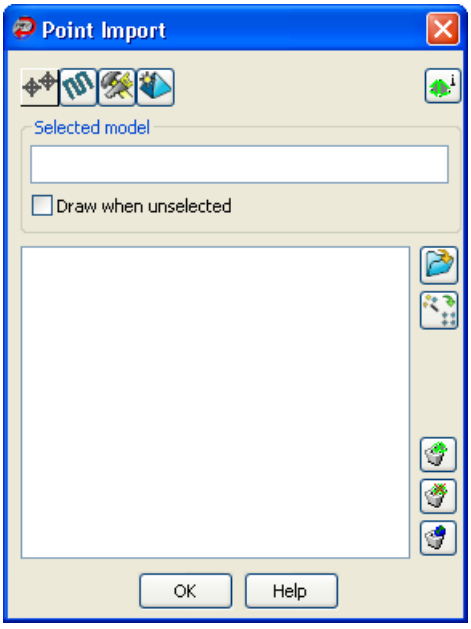
# Delcam CopyCAD Pro

**Import Point Clouds** (see page 48) has been added to the **File** menu.




## Importing point data

1. Click **File > Import Point Clouds** to display the **Point Import** dialog box.




2. Use the following buttons to import multiple points :

Import a point cloud.

Import a point cloud using the **Import Point** wizard.

3. Click to display information about the model.
4. Use the following buttons to model and edit the point data:

   Point toolbar.

   Scanline editing toolbar.

   Modelling toolbar.

   Triangle Generation toolbar.
5. Use the following buttons to delete models:

   Delete selected models

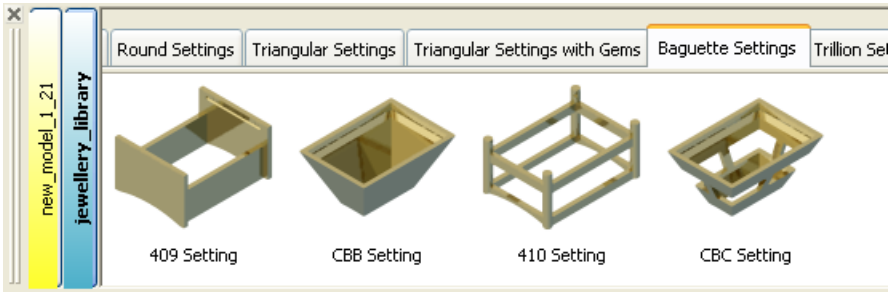
   Delete all other models

   Delete all

# Delcam Designer & Designer for Jewellery

The following changes have been included in Delcam Designer and Delcam Designer for Jewellery.

- Jewellery libraries have been added to Delcam Designer for Jewellery. They are loaded automatically when Delcam Designer for Jewellery is started and displayed in a window at the bottom of the screen. The example below shows the **Baguette settings**.




- Specific settings are loaded when using **Delcam Designer** and **Delcam Designer for Jewellery**. These ensure that:
  - the assembly tree is not displayed automatically.
  - when using 'drag and drop', the default relation is **Aligned.**

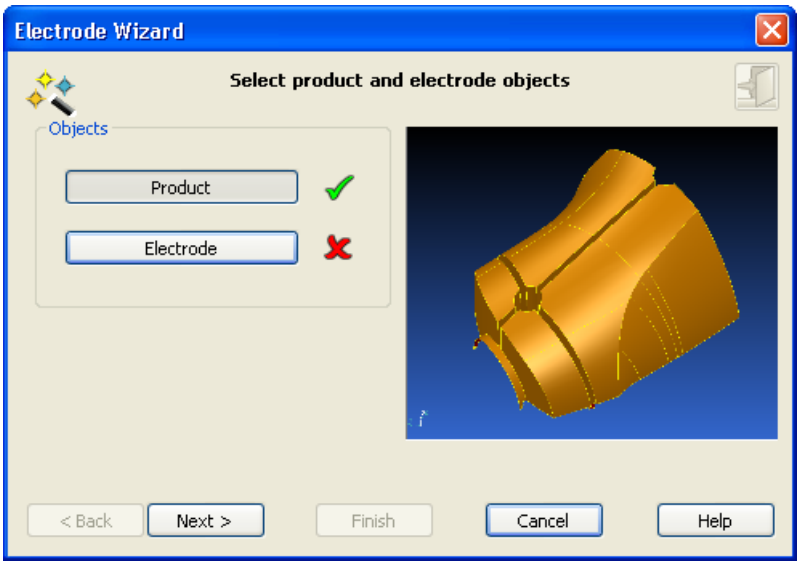
- when using 'drag and drop', the cursor snaps to components attachments only.
- base points of attachments are drawn when in shaded mode.

- When using the **Parameter List Editor** dialog box, the component parameters have a pre-defined maximum and minimum value. The edit boxes for these parameters have slightly different coloured backgrounds. If you enter a value greater than the maximum, the maximum value will be used. If you enter a value less than the minimum, the minimum value will be used.

# Delcam Electrode

Use the new selection options on the Electrode Wizard to select the surface data representing the part from which Delcam Electrode extracts the electrode.

1. Select surfaces.
2. Click from the Wizard toolbar to display the **Select product and electrode objects** page of the Electrode wizard.




3. If required, click **Electrode** and select additional electrode surfaces for shutout faces at the intersection of ribs.
4. Click **Next** to display the **Select Regions** page of the Electrode Wizard and create the electrode in the normal way.

# Other changes

The following changes have been made to the general operation of PowerSHAPE:

- PowerSHAPE releases now use a year-based title in the form PowerSHAPE 2010. This title is used in the **About** dialog box and the Main toolbar.

  For example:

  **Delcam PowerSHAPE 2010** for the first customer release after July 1st 2009

  **Delcam PowerSHAPE 2010 R2** for the second customer release after July 1st 2009

- All models opened in PowerSHAPE 2010 use model version *21*. When PowerSHAPE 2010 is installed and run for the first time, the initialisation work will open the latest NEW_MODEL_MASTER_XX you have (where XX <21) and save it as NEW_MODEL_MASTER_21 and also turn off the **Version 8 solids** option on the **Tools > Options > Object > Solids** dialog box.

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