



Advanced
Manufacturing
Solutions

PowerSHAPE 2013

What's New

PowerSHAPE 2013

What's New



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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What's New in PowerSHAPE 2013

Major developments included in PowerSHAPE 2013 are:

- Enhanced Direct Modelling functionality, including **Replace Face**.

- Updated **Solid Core** functionality.

- Direct capture of point data from laser scanners.

- Interactive nesting for efficient use of materials.

User Interface

Changes have been made to the interface in the following areas:

- Menus

- Toolbars (see page 2)

- Dialogs (see page 3)

Menus

In PowerSHAPE 2013, menus have been updated in the following areas of functionality:

- General Editing** (see page 5)

- Triangle modelling - Cloud** (see page 39)

- Triangle modelling - Mesh** (see page 44)

- Solid modelling** (see page 49)

- Drafting** (see page 58)

Toolbars

In PowerSHAPE 2013, toolbars have been updated in the following areas of functionality:

General Editing

Triangle modelling (see page 43)

Solid Modelling (see page 46)

General Edits toolbar

The following changes have been made to the **General Edits** and associated toolbars:

- The **Limit mode** button (*Limit selection toolbar*) now cycles through the available modes.
- The dynamic dragging method of scaling has been re-introduced as an alternative to using the **Scale** toolbar to specify the scaling factors.
- The new **Pattern** flyout (see page 5) includes the following buttons:
 - **Create Pattern.**
 - **Nest Items** (see page 6) (new in 2013).
- Use the new flyout on the **Move** toolbar (see page 20) to define the way the movement is specified.
- There are three new options on the **Rotate** toolbar (see page 27) that let you specify the method to be used to define the rotation axis.
- **Next solution** and **Previous Solution** buttons are now available on the **Offset** toolbar (see page 33) when editing faces of a solid (*Direct Modelling*).
- A **Select** button is added to the following toolbars when editing faces:
 - Move
 - Rotate
 - Offset
 - Scale

This button lets you pick an different face while the toolbar is displayed.

Dialogs

In PowerSHAPE 2013, dialogs have been updated in the following areas of functionality:

- General Editing** (see page 5)
- Surface modelling** (see page 45)
- Solid modelling** (see page 46)
- Assembly modelling** (see page 56)
- Drafting** (see page 58)
- Render** (see page 62)
- Electrode** (see page 67)
- Toolmaker** (see page 74)
- Import / Export** (see page 79)
- Options**

Options dialogs

In addition to the changes to Data Exchange options (see page 79), the following changes have been made to **Options** dialogs:

- A new option has been added to the **Tools > Options > File > Model** dialog. Select **Use first model path as default model path** to set the default model location to the first entry in the list of model paths.
 - If the option is selected, open and save operations use the first model path as the default location. If the model path list is empty, the default location is set to the current directory.
 - If this option is deselected, the default path is specified in the **Start in** option on the PowerSHAPE shortcut
- The **Model Save As Format** options have been removed from the **Tools > Options > File > Model** dialog.
- A new option has been added to the **Tools > Options > Object > Holes** dialog.
- On the **Tools > Options > View > Shading** dialog, the options have been re-organised to include the following changes:
 - **Surfaces and Version 8 solids** and **Solids** settings can be set independently. Use the **Shading Algorithm** options to close gaps in the shading.
 - Separate settings can be applied to surfaces/version 8 solids and Parasolids.

- On the **Tools > Options > View > Views** dialog:

Wireframe Antialiasing has been renamed **Anti-aliasing**. This option is now on by default, providing smoother outlines to rotated models.

Anti-aliasing provides greatly improved graphics display, but it can impact on performance. Consequently, this functionality is disabled automatically if the graphics card has less than 1Gb of video memory.

If you wish to disable the anti-aliasing functionality, use one of the following methods:

- Make the following change to powershape.com and restart PowerSHAPE.

`enable_aa: false`

- Use the control panel of the graphics card to turn off the anti-aliasing mode of the graphics card.
- Deselect **Tools->Preferences->View->Views->Anti-aliasing**

Changing this option affects subsequently created windows; it has no effect on currently open windows.

General Editing

Changes have been made to the follow areas of General Editing functionality:

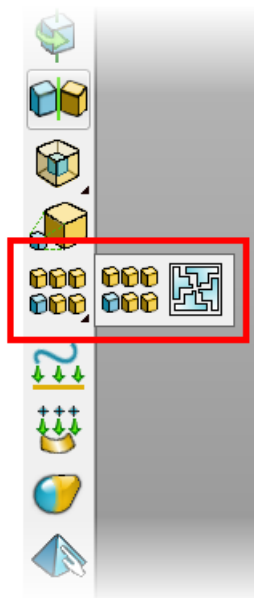
- Additional instrumentation is now added to items when using:
 - **Scale**
 - **Move**
 - **Offset**
 - **Rotate**
- When using **Best-fit Align** dialog, progression through the selection process has been improved.
- The **Pattern** button has been replaced by a new **Pattern** flyout, containing the following buttons:



Create Pattern.



Nest Items (new in PowerSHAPE 2013) displays the **Nesting** dialog (see page 6). Clicking this button is the same as selecting **Edit > General Edits > Nest**.



Nesting

Use the **Nesting** dialog to arrange PowerSHAPE objects on the principal plane of the active workplane (see page 11).

Nesting

Sheets

Select in the model window the geometry of the sheet and all the areas that must be excluded from the nesting and click the button to register them

Name	Quantity

More options >>

Parts

Select in the model window the geometry of one or several parts and register them for nesting

Name	Border	Quantity	Nested

More options >>

General options

Part distance

More options >>

Apply Close Help

Sheets

Use this section to select the geometry of the sheet that is to be included in the nesting.

Sheets

Select in the model window the geometry of the sheet and all the areas that must be excluded from the nesting and click the button to register them

Name	Quantity
Curve '5'	1

Use the following buttons to add and remove sheets from the list:



— Register the selected geometry as a sheet for nesting and add into the list.



— Remove the selected sheet from the list of sheets registered for nesting.

More options — If a sheet is selected in the list, click this button to display additional nesting options for the selected sheet.

Nesting direction: Horizontal
Starting corner: Upper left
Grain direction: None

Nesting direction — Select the required nesting direction from the drop-down list.

Starting corner — Select the starting position for the nesting from the drop-down list.

Grain direction — Select the grain direction from the drop-down list

Parts

Use this section to select the parts that are to be nested on the sheet.

Parts

Select in the model window the geometry of one or several parts and register them for nesting

Name	Border	Quantity	Nested
Curve 'figure_eig...	0	1	1

More options >>



— Add the selected geometry to the Parts list as single part. You can use this button to register multiple pieces of geometry as a single part.



— Add the selected geometry to the Parts list as multiple parts. You can use this button to register multiple pieces of geometry as multiple parts.



— Remove the selected part from the Parts list.

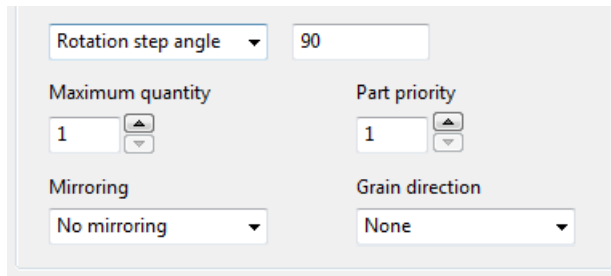
The Parts table includes the following columns:

Border — Specify the clearance around a part.

Quantity — Specify the minimum number of parts to nest.

Nested — PowerSHAPE indicates the number of parts that were nested when the last nesting operation was applied.

More options — If a part is selected in the list, click this button to display additional options for the selected part.



Rotation step angle: 90

Maximum quantity: 1

Part priority: 1

Mirroring: No mirroring

Grain direction: None

Part rotation— Select the part rotation method from the drop down list.

Rotation angle — Enter a part rotation step angle in degrees, or a list of angles separated by spaces.

Maximum quantity — Enter the maximum number of that part that can be nested. Alternatively, use the up and down arrows to set the value. Use a priority of **1** to indicate that the part has the highest priority.

Part priority — Enter the priority of the part. Alternatively, use the up and down arrows to set the value.

Mirroring — Select an option to specify if the part can be mirrored and the mirroring method to be used.

Grain direction — Select the grain direction of the part from the drop-down list

General options

Use this section to specify the general parameters that are to be used when nesting the parts.



General options

Part distance

More options >>

Part distance — Use these buttons to select how the distance between the parts is to be calculated.



— Distance between adjacent parts is the sum of the part borders.



— Distance between adjacent parts is the sum of the part borders and the border clearance.



— Distance between adjacent parts is the largest part border.



— Use this toggle button to specify if a copy of the original sheets and part geometry should be created.



— Use this toggle button to specify if the nesting boundaries of the sheets and parts should be created.

More options — Click this button to display additional general options.

Sequence parts by — Use this drop-down list to select the order in which parts with the same priority are selected for nesting.

By area — Nest parts with the biggest area first.

By perimeter — Nest parts with the biggest perimeter first.

Nesting resolution — Specify the accuracy of the nested pattern. The actual distance between the parts could be greater than the specified value, by one resolution unit. The smaller the resolution, the higher is the accuracy of the nesting, but the slower it is to calculate the layout.

Nest parts in holes of larger parts — If this option is selected, parts can be nested in the holes within larger parts.

Prefer nesting in part holes — If some of the nested parts have holes and this option is selected, PowerSHAPE will try to nest parts in the holes of the previously nested parts. If it isn't possible, parts will be nested elsewhere on the sheet.



*This option is only active when **Nest parts in holes of larger parts** is selected.*

Prefer nesting in sheet material islands — If the sheet material includes some islands and this option is selected, PowerSHAPE will try to nest parts in the holes of the sheet material islands. If it isn't possible, parts will be nested elsewhere on the sheet.



Sheet material islands are holes of locked parts; parts that are not nested but stay in the same position on the sheet.

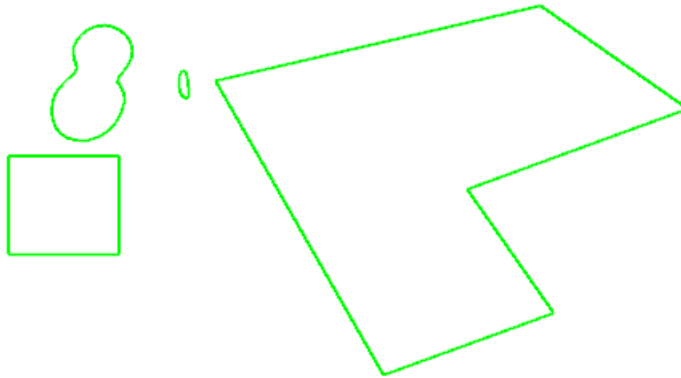
Nest filler parts up to the nesting height — A filler is a part that is included in any space after the other parts have been nested in priority order. You define a filler part by setting a maximum number of parts.



- If this option is deselected (default), the minimum number of main parts are nested and filler parts will be included as required.
- If this option is selected, the minimum number of main parts are nested and filler parts are only included up to the nesting height that has been determined by the nesting of the main parts.

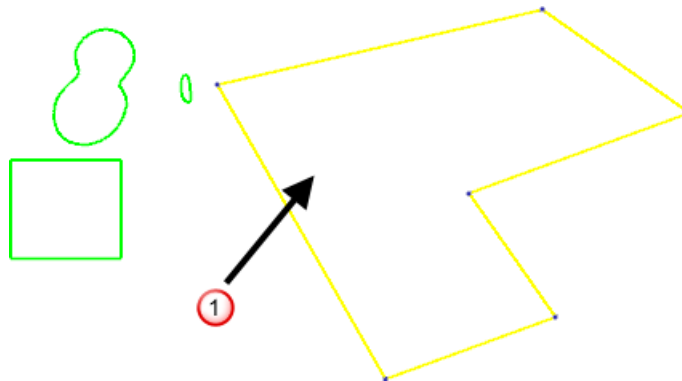
Use guillotine cuts — If this option is selected, nested parts will be aligned orthogonally so that a guillotine cut can be used to cut across the sheet (in either direction), without intersecting the nested parts.

Nesting - an example

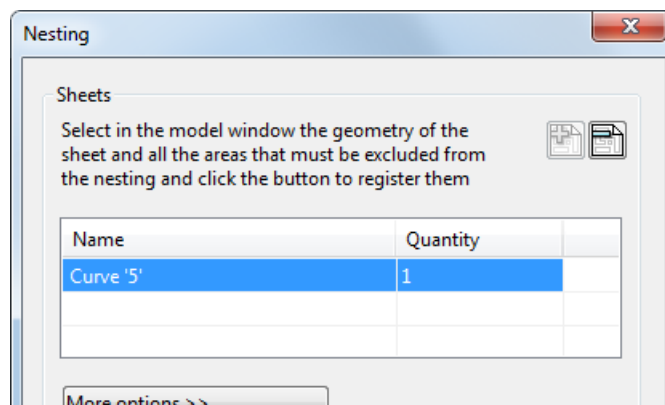
This example shows you how to nest three different parts in a sheet.



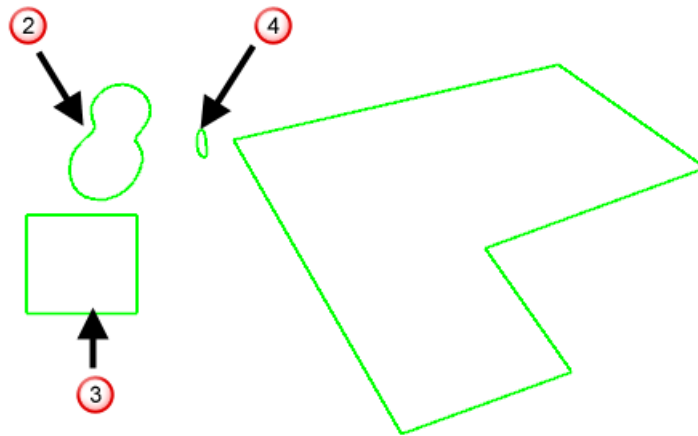
- 1 Click **Nest Items**  (*General edits toolbar*) to display the **Nesting** dialog
- 2 Select a sheet .



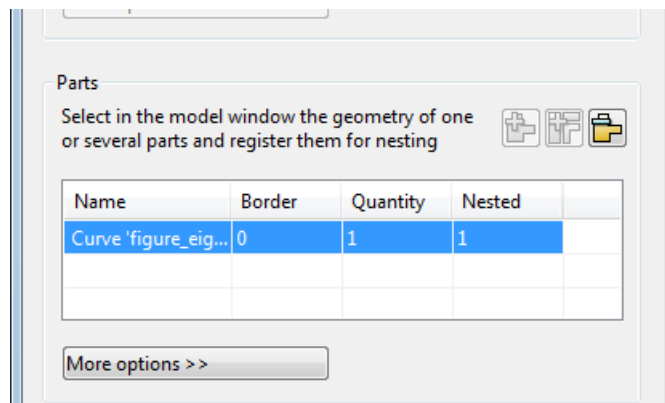
- 3 Click **Add sheet** . The sheet is added to the **Nesting** dialog



4 Select part: Curve 'figure_eight' ②.



5 Click **Add part** .



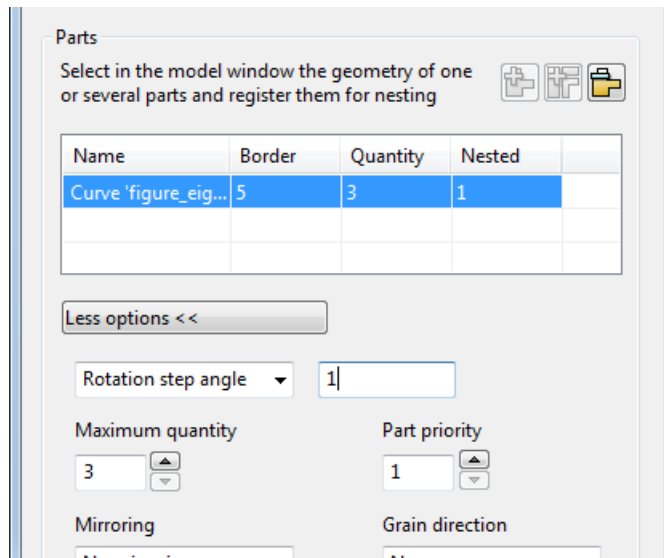
6 Set the following parameters:

- **Border** 5
- **Quantity** 3

7 Click **More options**.

8 Set the following parameter options for the part:

- **Rotation step angle** 1
- **Part Priority** 1



9 Select part: **Curve 'square'** ③.

10 Click **Add part** .

11 Set the following parameters in the parts list:

- **Border** 1
- **Quantity** 2

12 Set the following parameter options for the part:

- **Rotation step angle** 1
- **Priority** 2
- **Maximum quantity** 2

13 Select part: **Curve 'filler'** ④.

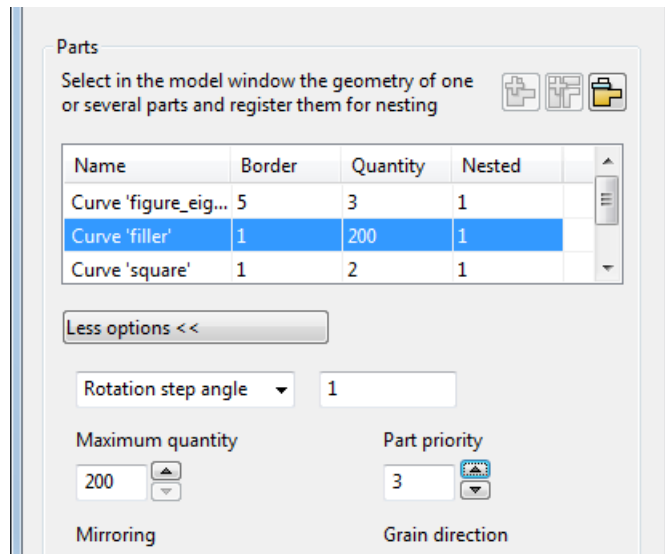
14 Click **Add part** .

15 Set the following parameters in the part list:

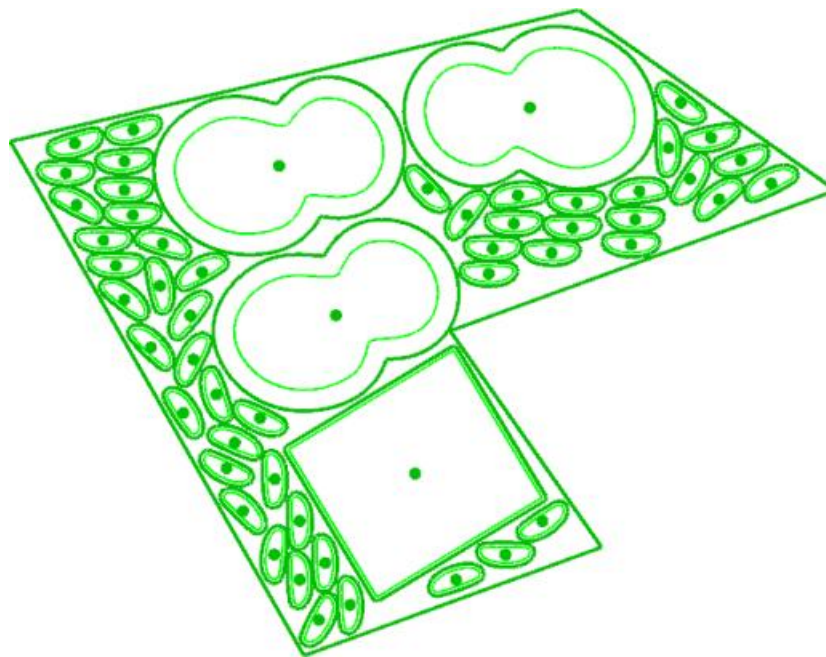
- **Border** 1
- **Quantity** 200

16 Set the following parameters options for the part:

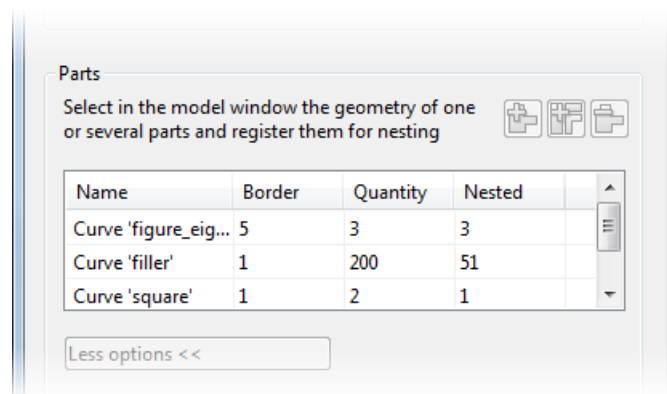
- **Rotation step angle** 1
- **Priority** 3



17 Click **Apply**. The parts will be nested according to the parameters that you have selected.



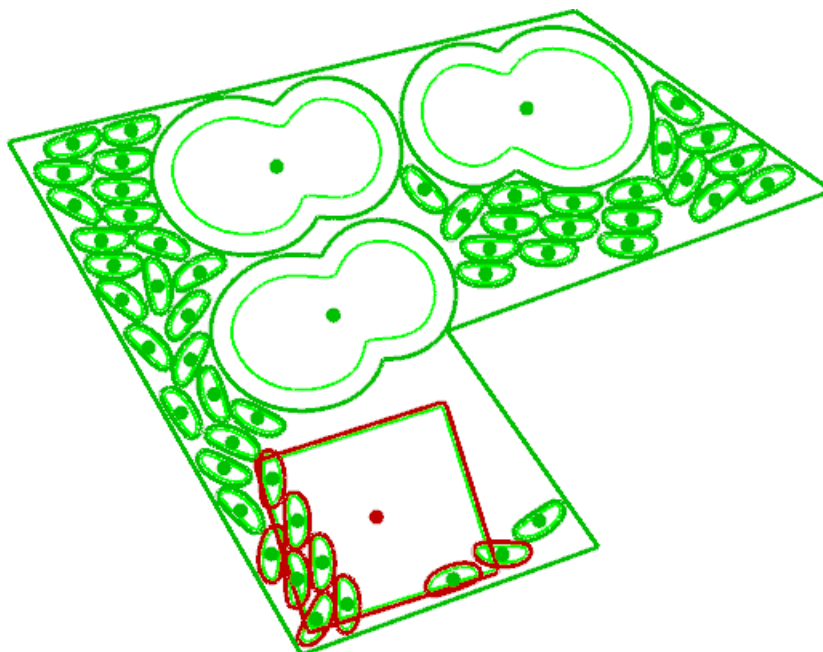
The list in the **Parts** section of the dialog is updated to show the number parts that have been nested.



This shows that:

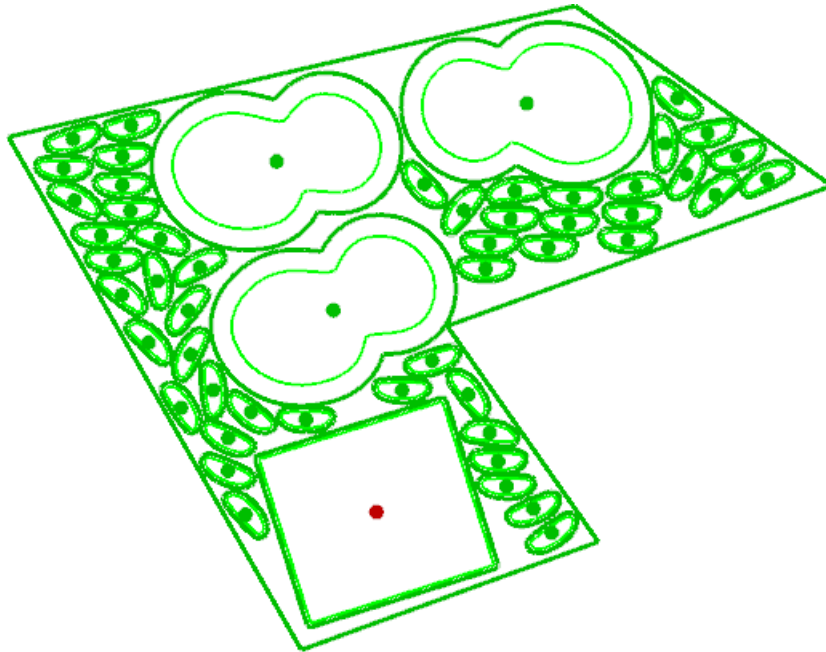
- all the required **Curve 'figure_eight'** parts (priority 1) parts have been included.
- one of the **Curve 'square'** parts (priority 2) parts have been included.
- 51 of the **Curve 'filler'** parts (priority 3) have been included.

18 Click and drag the **Curve 'square'** part. As you move the part, overlapping parts turn red.



You can also rotate a part by selecting it in the graphics window and dragging the border outline.

- 19 Click **Apply** to rearrange the parts based on the new position of the **Curve 'square'** part.



The number of **Curve 'filler'** parts that can be nested has reduced.

Parts

Select in the model window the geometry of one or several parts and register them for nesting

Name	Border	Quantity	Nested
Curve 'figure_eig...	5	3	3
Curve 'filler'	1	200	50
Curve 'square'	1	2	1

Less options <<



When you reposition an item manually, the item is locked in the new position and any subsequent nesting operations will not reposition the part. This is indicated by a red blob in the centre of the item. You can unlock a locked part by toggling the red blob to green.

Direct modelling

Direct Modelling has additional functionality in PowerSHAPE 2013.


- Replace Faces (see page 17)
 - Replacing a face - an example (see page 18)
- The following functionality has been updated in PowerSHAPE 2013 to provide additional flexibility when editing faces of a solid:
 - Move (see page 20)
 - Rotate (see page 27)
 - Offset (see page 33)
- Feature recognition (see page 34)

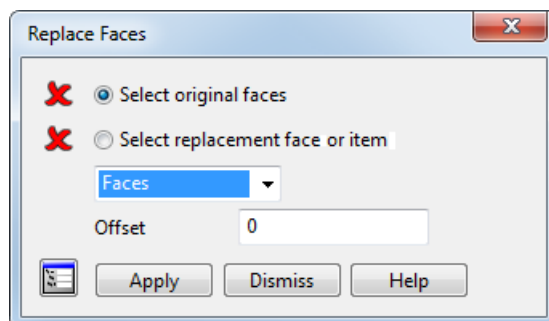
Replace Faces

Solid Replace Faces has been added to the **Solid Editing** toolbar.




This option lets you select faces of a solid and replace them with other existing faces.

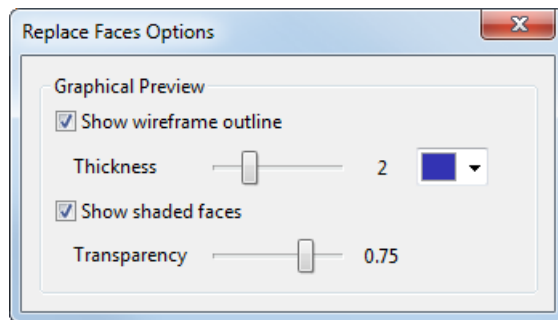
- 1 Select a solid to display the **Solid Edit** toolbar.
- 2 Click  to display the **Replace Faces** dialog.



- 3 Select the original faces. These are the faces that will be replaced. **✗** changes to **✓**.
- 4 Select the replacement face or item. **✗** changes to **✓**.

Faces is selected by default; alternatively, select **Open solid** or **Surface** from the drop-down list.

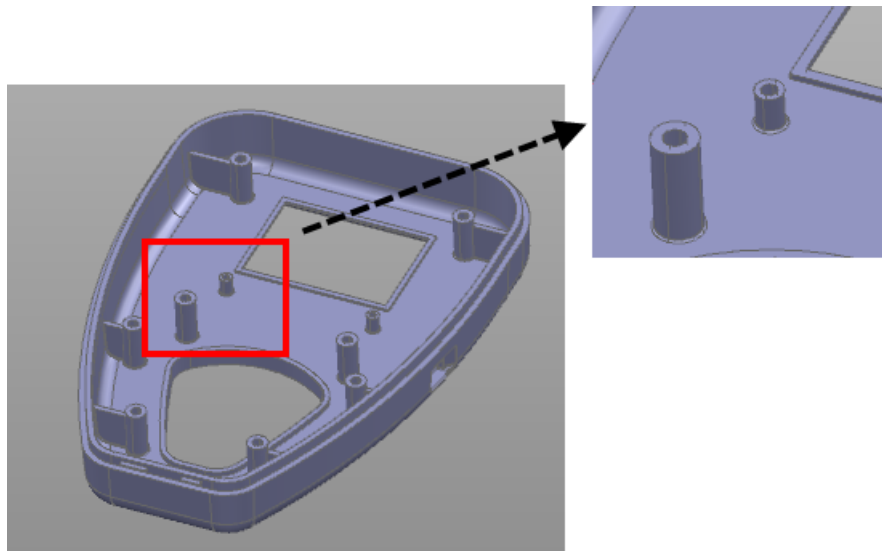
- 5 If required, enter a value for **Offset**. The offset distance is measured from the replacement entity.
- 6 If required, click  to display the **Replace Faces Options** dialog.




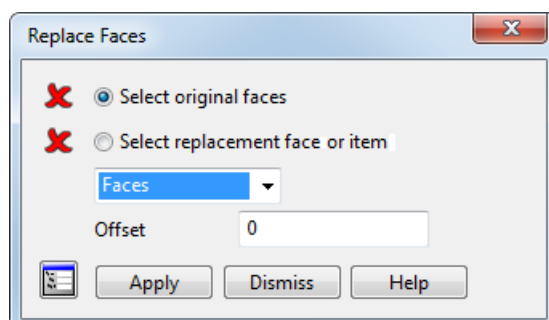
Use the options on this dialog to adjust the appearance of the graphical preview that is displayed before you click **Apply** on the **Replace Faces** dialog.

Replacing a face - an example

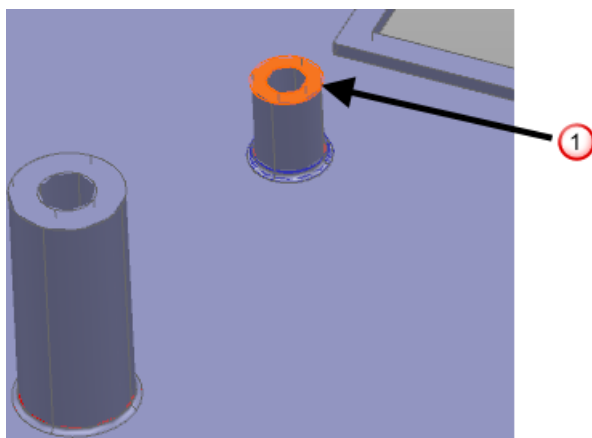
This example shows you how to replace one or more selected faces with another existing face.




- 1 Click the solid to display the **Solid Edit** toolbar.
- 2 Click  to display the **Replace Faces** dialog.

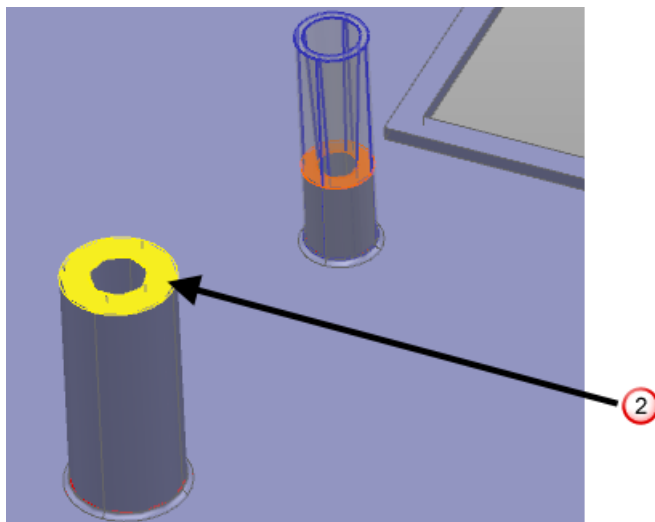


- 3 Select the original face ①. This is the face that will be replaced.

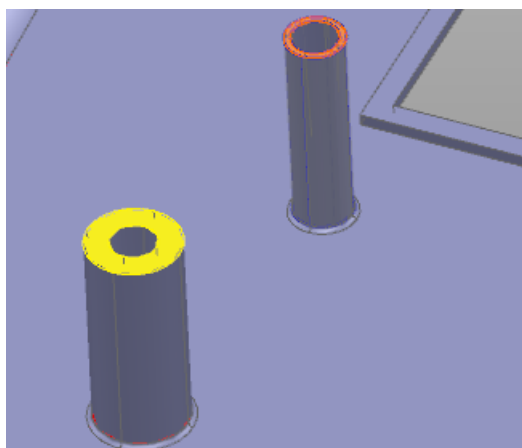


You can select the original faces before clicking  on the **Solid Edit** toolbar. In this case, there will be a ✓ next to **Select original faces**.

- 4 Select the replacement face ②. The preview of the updated face is displayed.



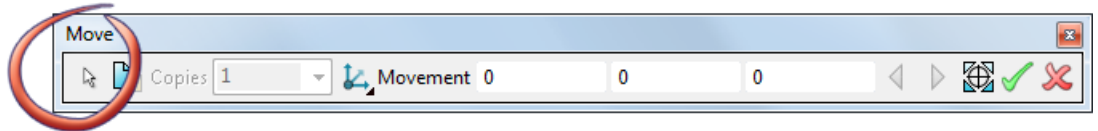
- 5 Click **Apply** to display the updated model.





Editing faces - Move

Move (*General Edits toolbar*) has been updated in PowerSHAPE 2013:

- New instrumentation has been added that helps you position the face.
- When used to edit faces, there is an additional button on the toolbar. This lets you change the face you have selected while the toolbar is displayed.

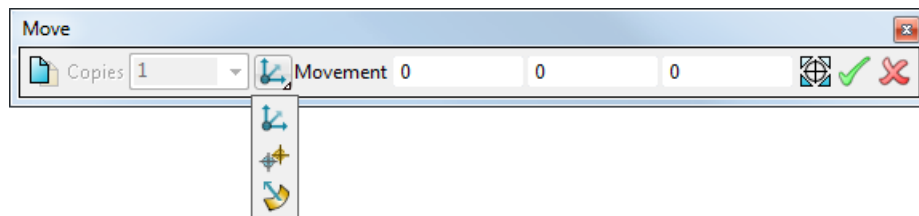


- 1 Click .
- 2 Change the face selection as required.
- 3 Click  to continue editing.






When you are in Select mode, the other buttons on the toolbar are unavailable.

- There are three additional ways to define the direction of movement:



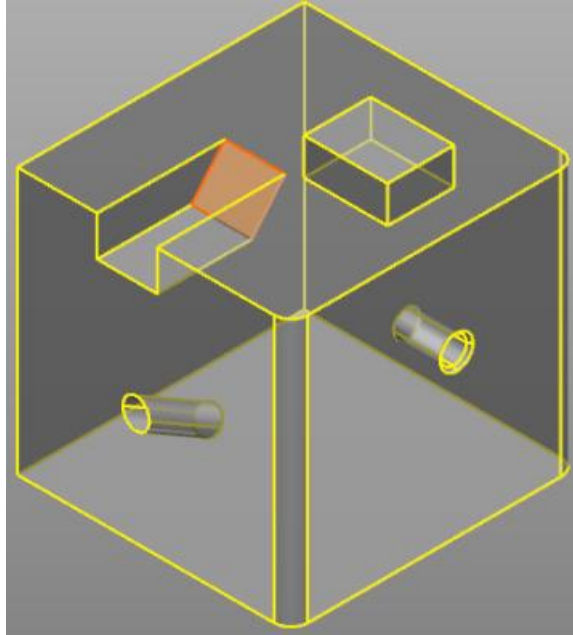
In addition to entering the coordinates to define the movement direction, you can use the following buttons on the **Move** toolbar:



-  Use a workplane axis (see page 21) .
-  Use two points (see page 22).
-  Use a normal (see page 25).

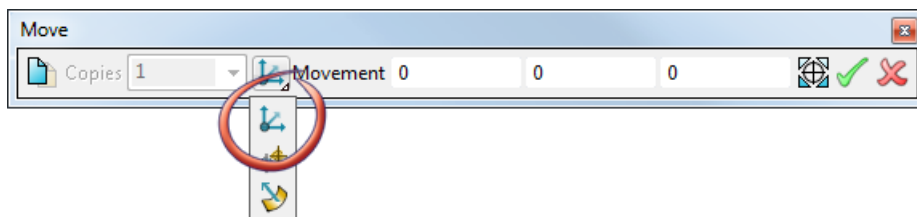
Move - using the workplane axis


Use the flyout on the **Move** toolbar to define the movement direction using the axes of the workplane. This is the default mode.

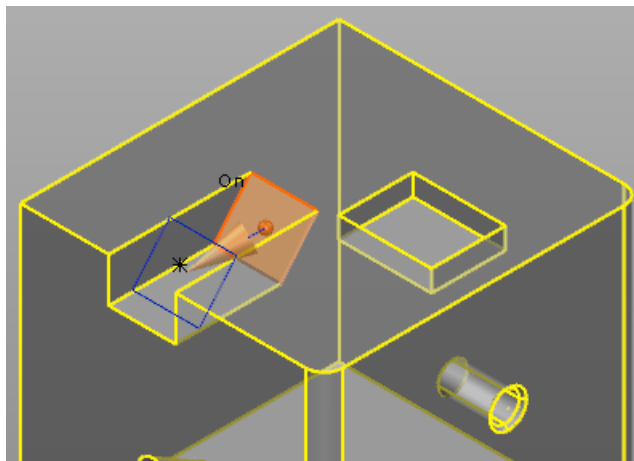
- 1 Select the face to be moved.




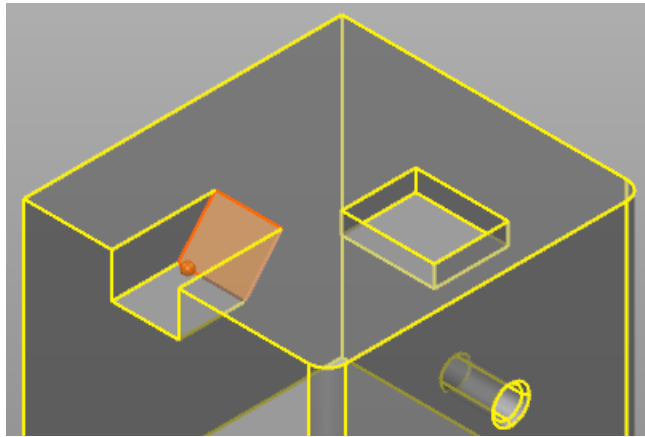
- 2 Click  (General Edits toolbar) to display the **Move** toolbar.
- 3 Click  (Move toolbar).




- 4 Either enter **Movement** values, or click a point . The preview is displayed. Drag the instrumentation as required.



- 5 Click  to move the face and update all associated geometry.

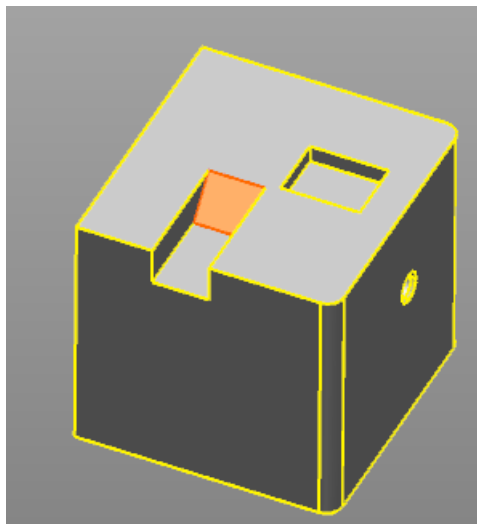




- 6 Click  to close the toolbar. The updated model is displayed.

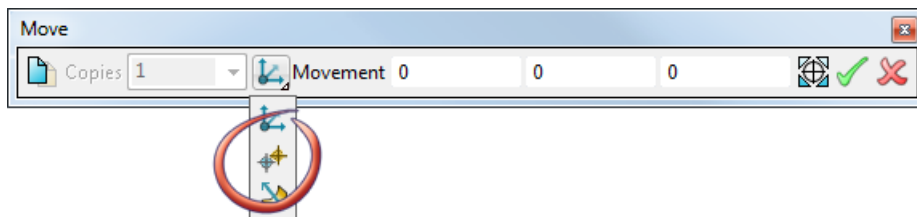
Move - using two points to define the movement direction

Use the flyout on the **Move** toolbar to define the movement direction using two points.

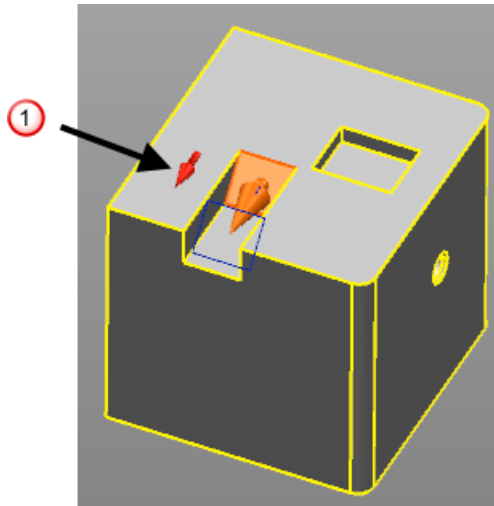
- 1 Select the face to be moved.



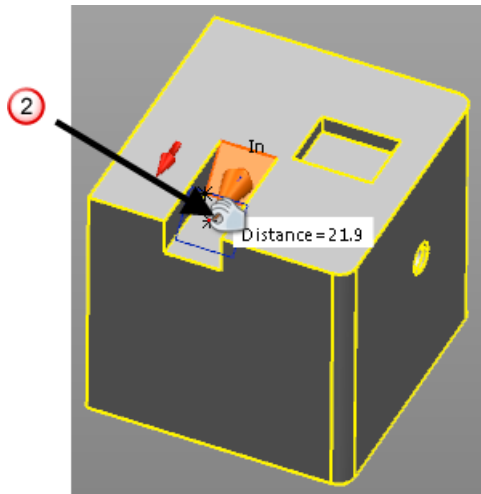
- 2 Click  (*General Edits toolbar*) to display the **Move** toolbar.
- 3 Click  (*Move toolbar*).




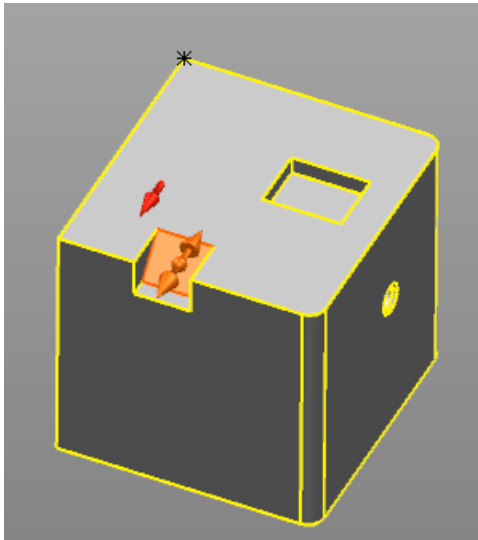
- 4 Click two points to define the move direction. The instrumentation indicates the movement direction ①.



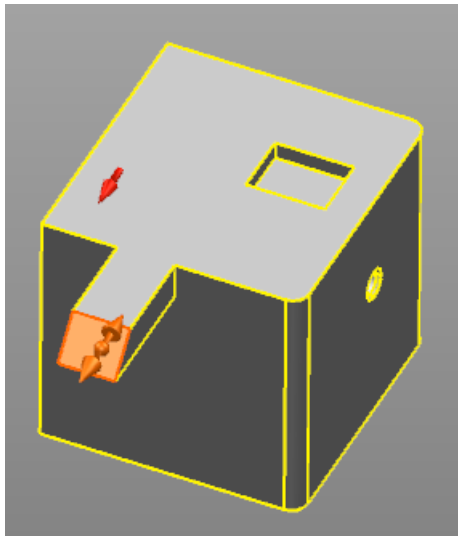
- 5 Drag the instrumentation. ②. The instrumentation can only be moved along the direction of movement. The **Distance** updates as you drag the instrumentation.




- 6 Click  to move the face and update all associated geometry.



If you move the face outside the model, the geometry will still be updated to reflect the move.

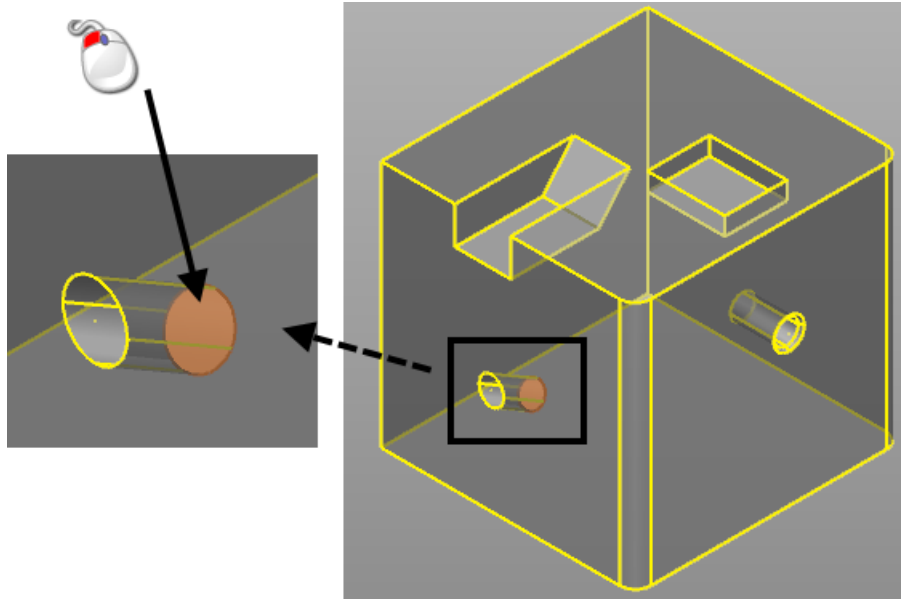




- 7 Click  to close the toolbar. The updated model is displayed.

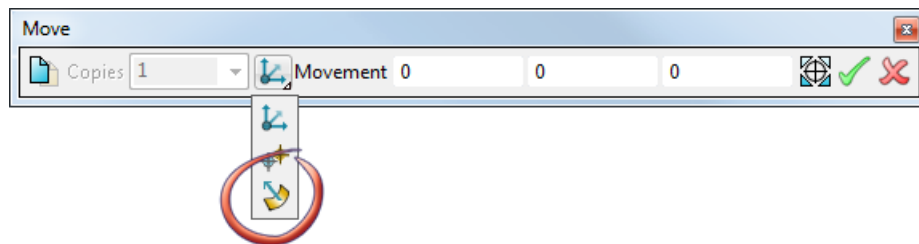
Move - using the normal of an item to define the movement

Use the flyout on the **Move** toolbar to define the movement direction using an item.

- 1 Select the face to be moved.

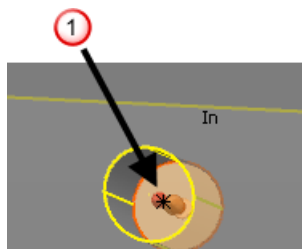


- 2 Click  (General Edits toolbar) to display the **Move** toolbar.
- 3 Click  (Move toolbar).

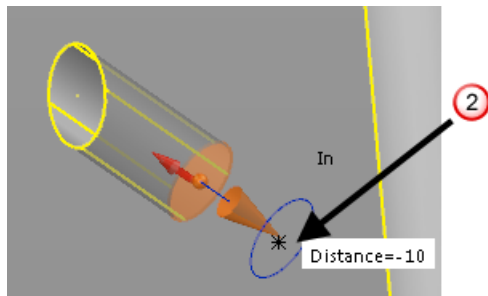


The toolbar is updated to reflect your selection.

- 4 Select an item 1; it may be necessary to rotate the model manually before you can make the selection. The move direction will be the normal to the geometry at that point. The instrumentation indicates the movement direction.

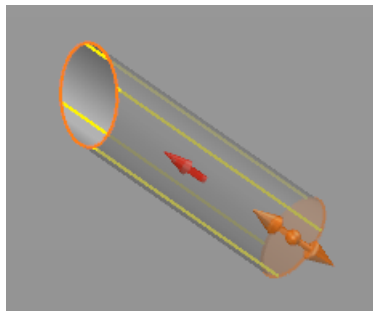


- 5 Enter a **Distance** or drag the instrumentation and click to fix the location ②.

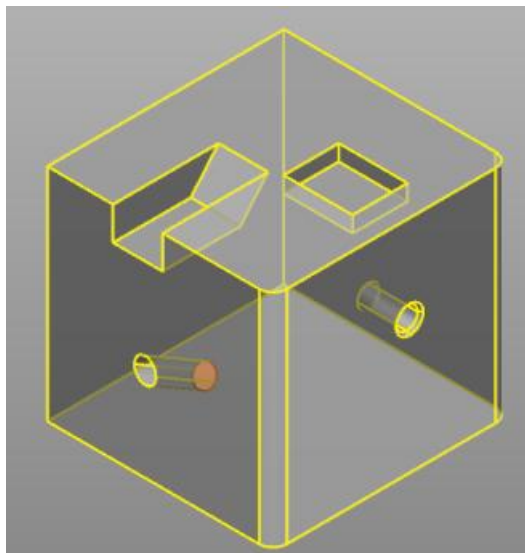


*If you drag the instrumentation, you can only be move along the direction of movement. The **Distance** tip updates as you drag the instrumentation.*

- 6 Click ☒ to move the face and update all associated geometry.



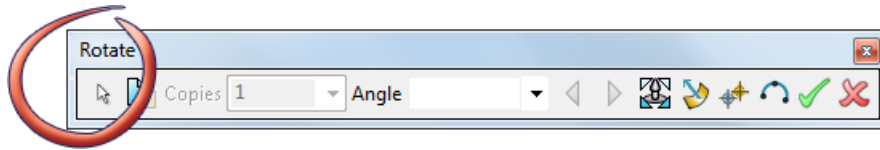
- 7 Click ☐ to close the toolbar. The updated model is displayed.





Editing faces - Rotate

Rotate (*General Edits toolbar*) has been updated:

- When used to edit faces, there is an additional button on the toolbar. This lets you change the face you have selected while the toolbar is displayed.



- 1 Click .
- 2 Change the face selection as required.
- 3 Click  to continue editing.

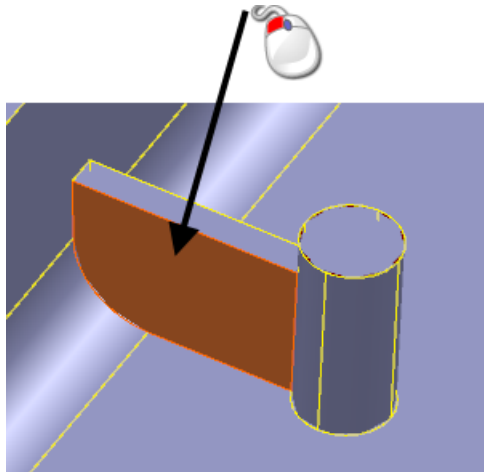


When you are in Select mode, the other buttons on the toolbar are unavailable.


- There are now three additional ways to define the rotation axis:
Using a normal as the rotation axis (see page 27)
Using two points to define the rotation axis (see page 29)
Using three points to define an arc as the rotation axis (see page 31)

Rotate - using the normal of an item as the rotation axis

- 1 Select the face to be rotated.

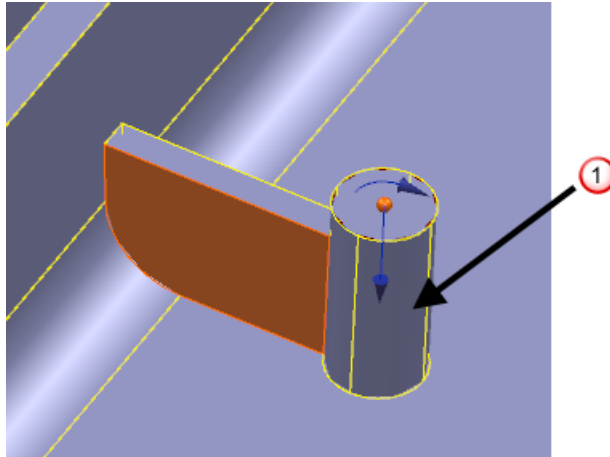


- 2 Click  (*General Edits toolbar*) to display the **Rotate** toolbar.

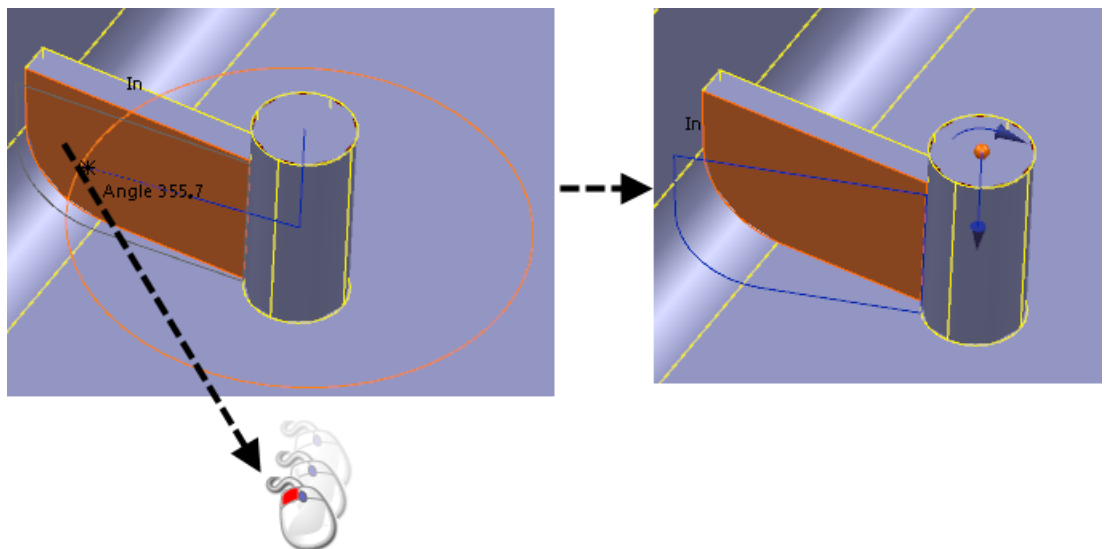
- 3 Click  to select the option to define the rotation axis using a normal. The toolbar is updated to indicate the next selection to be made.




- 4 Click on geometry **1**. The rotation axis is aligned to the selected normal.

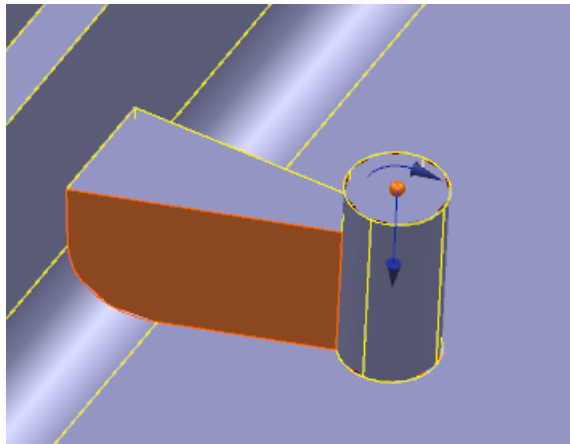



- 5 Specify the **Angle** by entering a value in the toolbar. Alternatively, click and drag the face as shown below:



- 6 Release the mouse button when the face is in the required position.

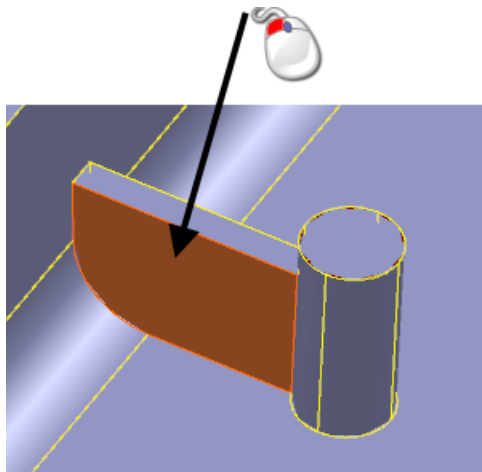
- 7 Click . The face is displayed in the new location; all associated geometry has been automatically updated.




- 8 Click  to close the toolbar.

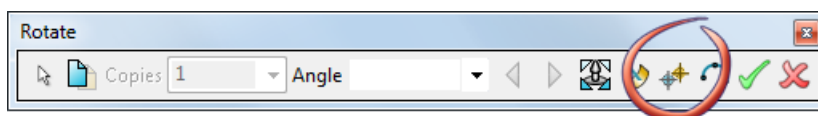
Rotate - using two points to define the rotation axis

- 1 Select the face to be rotated.



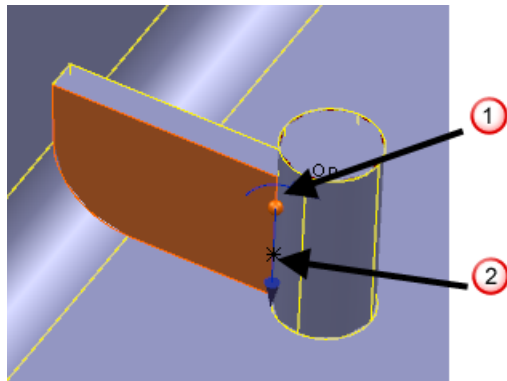
- 2 Click  (*General Edits toolbar*) to display the **Rotate** toolbar.

- 3 Click  to select the option to define the rotation axis using two points. The toolbar is updated to indicate the next selection to be made.

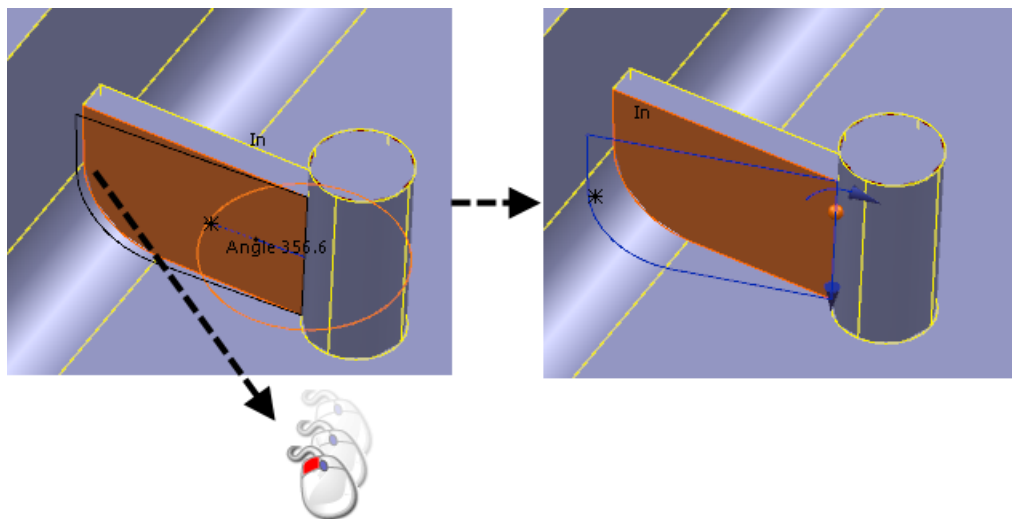



- 4 Select first point **1**. The toolbar is updated to indicate the next selection to be made.

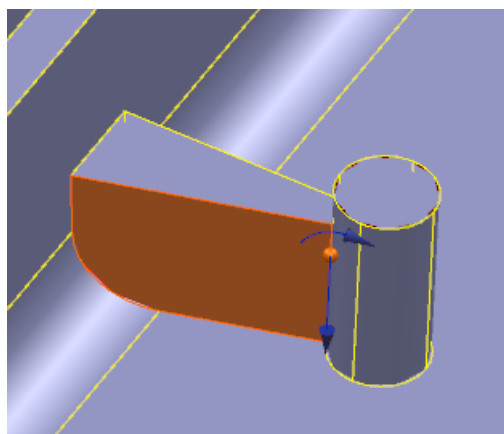
- 5 Select second point ②.




- 6 Specify the **Angle** by entering a value in the toolbar.
Alternatively, click and drag the face as shown below:



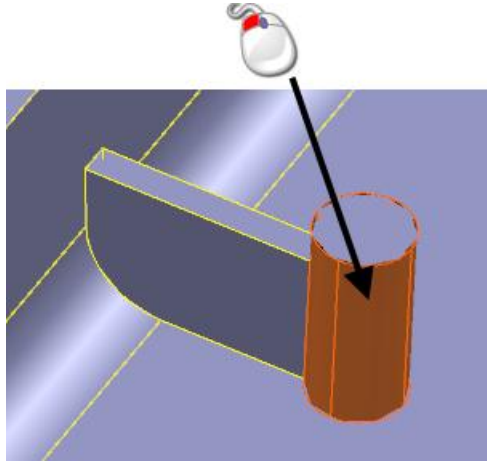
- 7 Release the mouse button when the face is in the required position.
- 8 Click . The face is displayed in the new location; all associated geometry has been automatically updated.





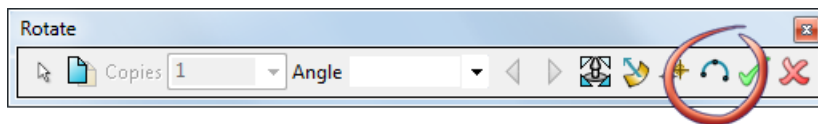
- 9 Click  to close the toolbar.

Rotate - using three points to define an arc

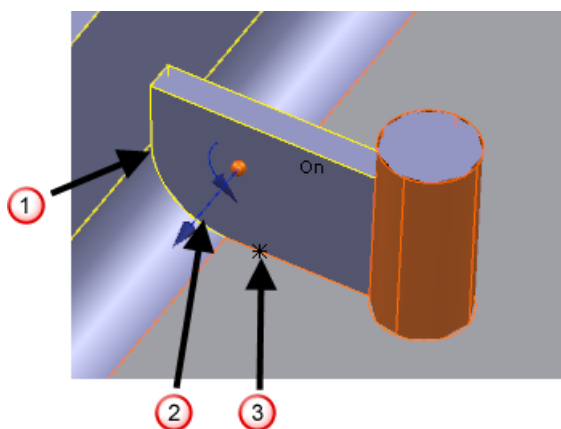
- 1 Select the face to be rotated.



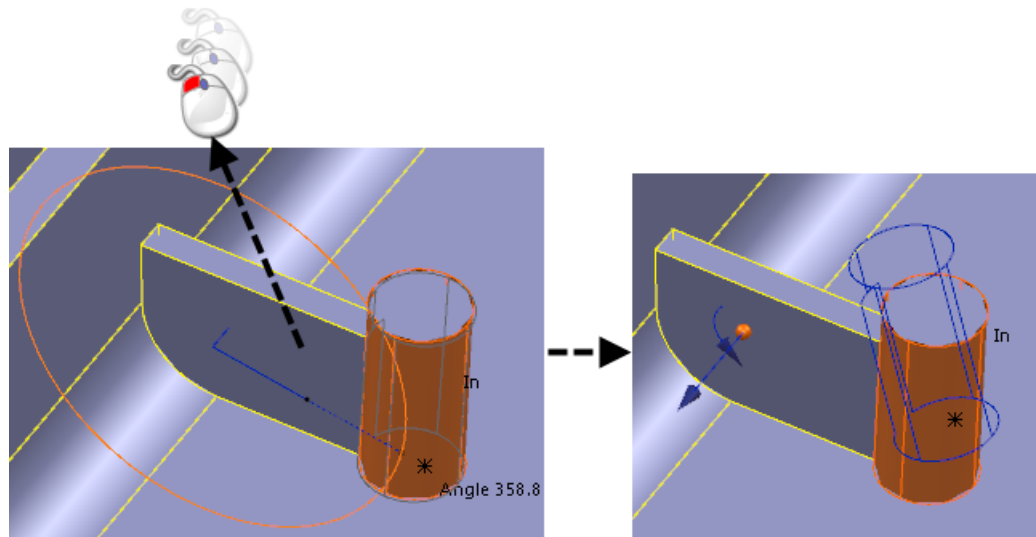
- 2 Click  (General Edits toolbar) to display the **Rotate** toolbar.
- 3 Click  to select the option to define an arc using three points.




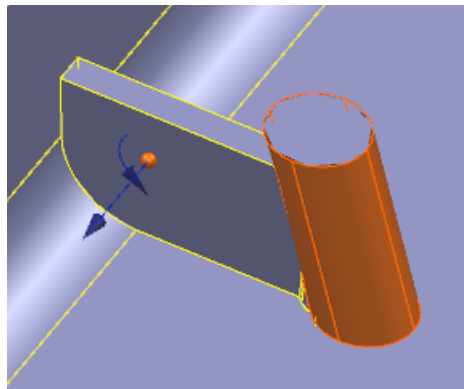
- 4 Select the first point **1**. The toolbar is updated to indicate the next selection to be made.
- 5 Select the second point **2**. The toolbar is updated to indicate the next selection to be made.
- 6 Select the third point of the arc **3**. The rotation axis is created at the centre of the arc.



- 7 Specify the **Angle** by entering a value in the toolbar.
Alternatively, click and drag the face as shown below:



- 8 Release the mouse button when the face is in the required position.
- 9 Click . The face is displayed in the new location; all associated geometry has been automatically updated.

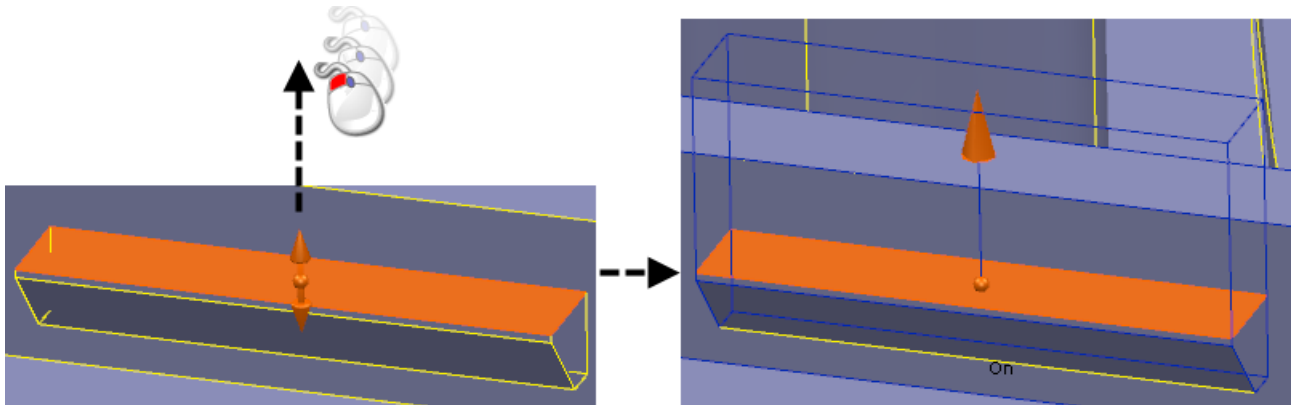


- 10 Click  to close the toolbar.

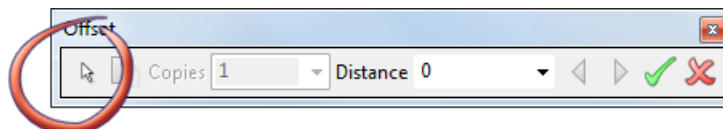
Editing faces - Offset



Offset (*General Edits toolbar*) has been updated:

- You can use the new drag handle to specify the offset distance. The **Distance** value on the toolbar is updated when you release the mouse button after dragging the handle.



- The following are added to the **Offset** toolbar when editing faces:
 - Next solution** and **Previous Solution** buttons. If there is more than one possible solution, use these buttons to select the required solution
 - Offset the selection** ✓ and **Dismiss** ✗.
- When used to edit faces, there is an additional button on the toolbar. This lets you change the face you have selected while the toolbar is displayed.



- 1 Click .
- 2 Change the face selection as required.
- 3 Click  to continue editing.



When you are in Select mode, the other buttons on the toolbar are unavailable.

Feature recognition

The following improvements have been made to feature recognition in PowerSHAPE 2013:

- Adjacent fillets are now recognised when recognising holes, cuts, bosses and pockets and protrusions.
- The behaviour of the dialog buttons for fillet recognition has been changed:
 - If you click **Apply** and no selection is made, **OK** becomes active.
 - If you click **Apply** and a selection is made, **OK** and **Cancel** become active.

Cancel will ignore the latest selection and save any recognitions you have already made.

OK will save a feature for selection.

Point clouds

There have been a number of additions to cloud functionality in PowerSHAPE 2013. The following

- Changes to toolbars, including the addition of the new **Acquire Points** toolbar (see page 36) to support direct data capture from a laser.
- Additional menu options (see page 39) have been added.
- The point cloud data file can now contain more than just point data on each line of the file. The x y and z must be the first 3 columns of data.

For example, PowerSHAPE can now read a file that contains the following line:

0.23124 1.27482 9.288374 JJJJJJ KKKKKK LLLLLL



The first three columns of data must be numeric. The remaining data can be text.

- You can now cut or copy selected points from a cloud and paste them elsewhere.
- When using Delcam Exchange 6710 or later, the speed of importing IGES files containing large clouds has been improved.





The import speed is unchanged if you are using an earlier version of Delcam Exchange.

Toolbar options (Cloud)

The following changes have been made to toolbars that are used with a cloud:

- Use the **Acquire Points** toolbar (see page 36) to capture point data from laser scanners.

- **Box Select**  has been added to the **Cloud edit** toolbar.






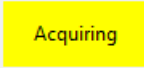
- You can now use the **Generate a mesh**  button to generate a single mesh from multiple clouds.

Acquire Points toolbar

The **Acquire Points** toolbar has been added in PowerSHAPE 2013.






Before you can start scanning your part, you must:

- 1 Click **Connect the arm, for laser heads**  on the **Arm** flyout (*Status bar*) to display the **Acquire Points** toolbar.
- 2 Click  to open the **Delcam CMM Driver Configuration** dialog (see page 37) and select a connection protocol.
- 3 Click  to connect to the selected device. Status information is updated to .
- 4 Click  to configure the device settings using manufacturer specific dialogs.
- 5 Start scanning your part. The status changes to  when you are acquiring points.

Additional tools

Use the following buttons on the toolbar as required:

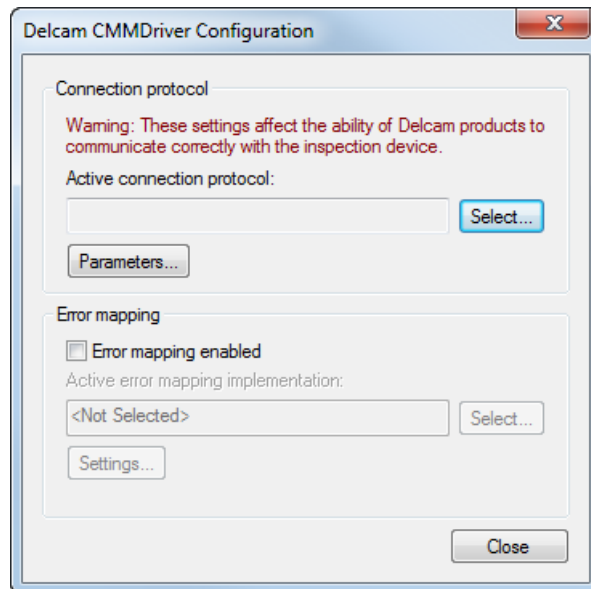
- Click  to delete the most recent sweep. This button is unavailable until you have performed at least one sweep.
- Click  to open or close the **Sweep Information** dialog (see page 38).
- Use the following buttons to specify how the clouds are displayed:
 -  — If selected, each sweep is assigned a distinct colour. You can choose the default setting for this option using the **Tools > Options > General > Arm** dialog (see page 38).



- If selected, the view is locked in the current position. You can choose the default setting for this option using the **Tools > Options > General > Arm** dialog (see page 38).

Delcam CMMDriver Configuration dialog

Use this dialog to specify the connection protocol for the device.



To select a connection protocol:

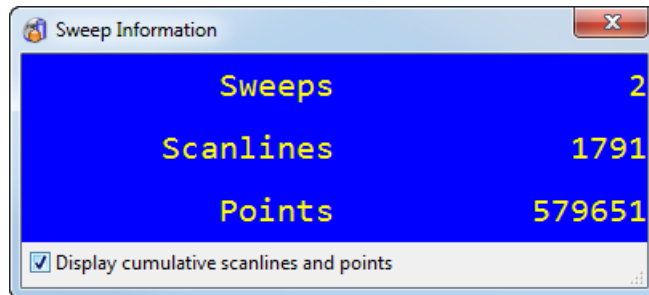
- 1 Click **Select** to open the **Select Connection Protocol** dialog.
- 2 Select a connection protocol from the list.
- 3 Click **OK**. The dialog closes and the selected protocol is displayed as the **Active connection protocol**.
- 4 If the protocol you selected requires additional parameters, click **Parameters** to open a parameters dialog and specify the settings.
- 5 Click **Close**.




Error mapping is not supported by PowerSHAPE.

Sweep Information dialog

The **Sweep Information** dialog displays the number of sweeps, scanlines and points that have been captured in the acquisition session.



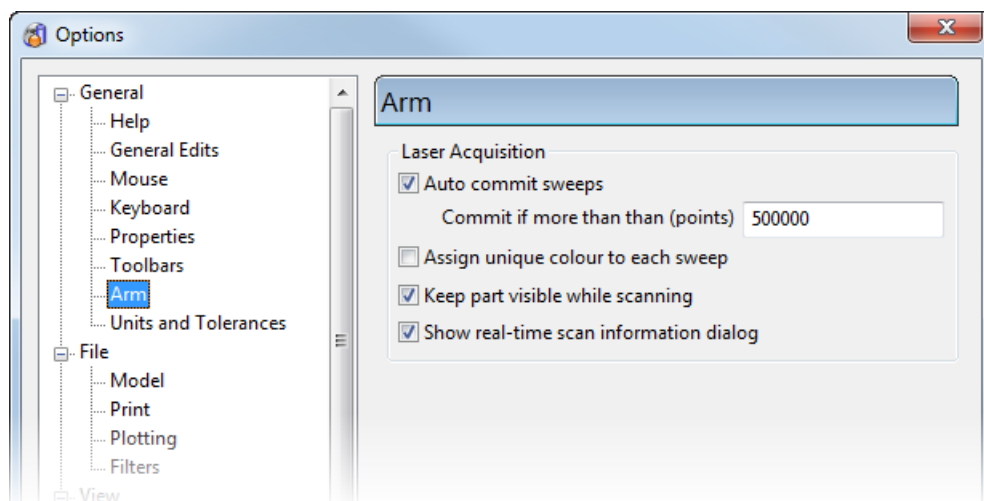
- Click  on the **Acquire Points** toolbar to display the **Sweep Information** dialog. This dialog is displayed when you open the toolbar if you have selected **Show real-time scan information dialog** on the **Tools > Options > General > Arm** dialog (see page 38).
- Deselect **Display cumulative scanlines and points** to display only the data for the current sweep.




This dialog can be resized to increase the font size, which will make it easier to view if your measuring device is located away from your machine.

Arm options


Use the options in this dialog to set some defaults for the acquisition session.



Auto commit sweeps — Sweeps are saved automatically when the number of points taken exceeds the number specified in the box. The clouds can be combined (see page 41) after all points have been acquired.

Assign unique colour to each sweep — Each sweep is assigned a unique colour. This is the same as clicking  on the **Acquire Points** toolbar.

Keep part visible while scanning — The acquired points remain visible whilst you carry out further scanning.

Show real-time scan information dialog — The **Sweep Information** dialog (see page 38) is displayed when the **Acquire Points** toolbar is displayed. This is the same as clicking  on the **Acquire Points** toolbar.

Menu options (Cloud)

There are several changes to cloud options on menus in PowerSHAPE 2013.

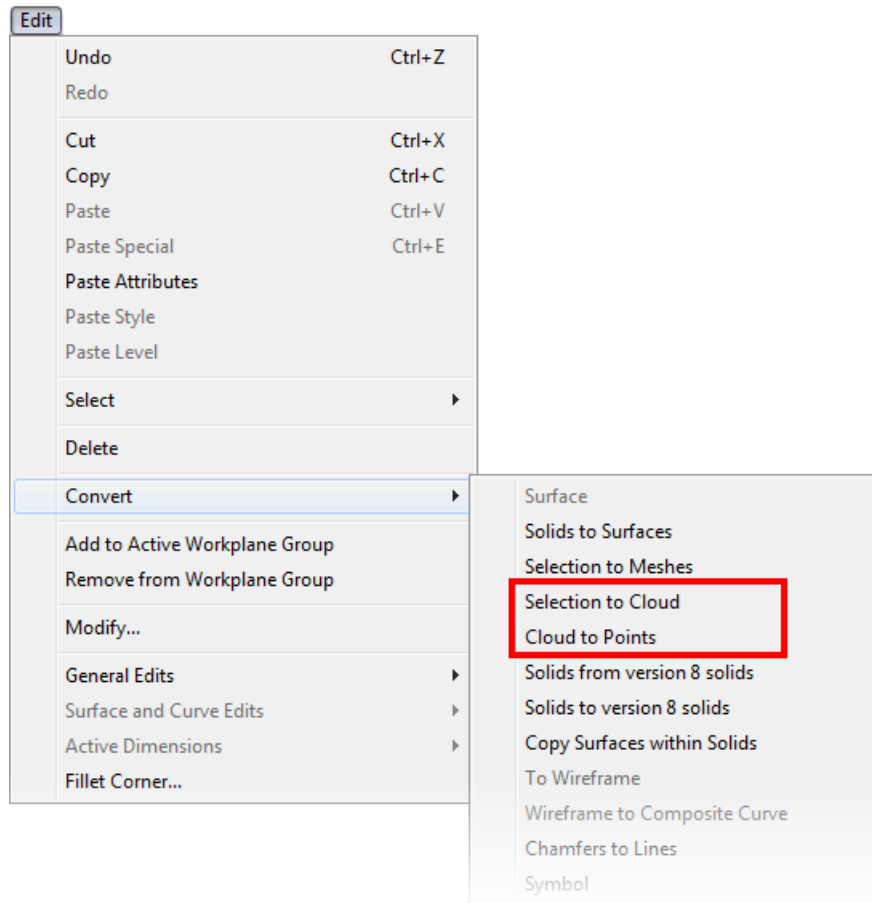
- Edit menu (see page 40)

- Object menu (see page 41)

- Cloud popup menu (see page 42)

Edit menu

The **Edit > Convert** menu has two new options:



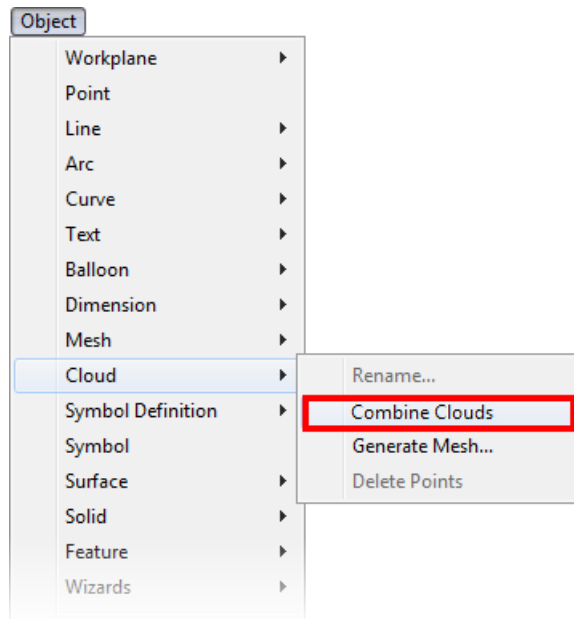
Selection to Cloud — converts a group of points or a mesh to a point cloud.

Converting multiple meshes will create multiple clouds; so, converting two meshes will create two clouds.

Cloud to Points — explodes a cloud to points.

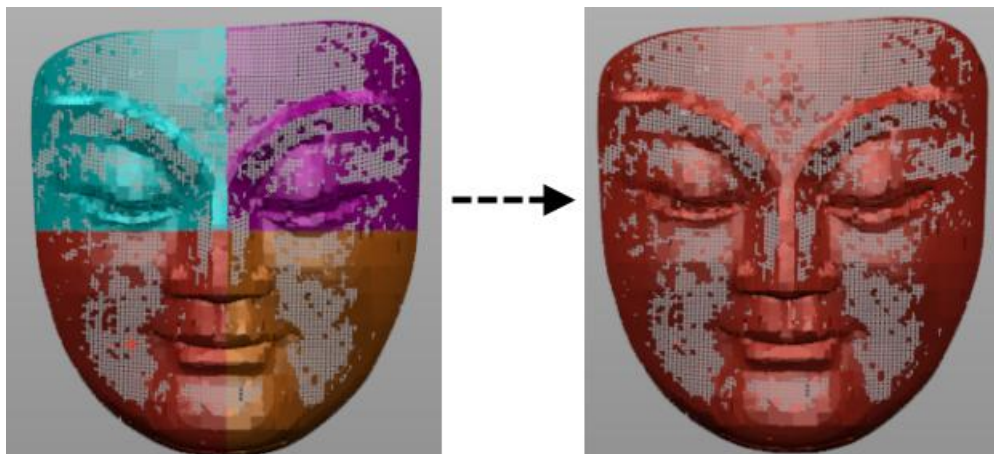
Object menu

Combine Clouds has been added to the **Object > Cloud** menu. (see page 41)



Combine clouds - an example

Use the following steps to combine four clouds into a single cloud:

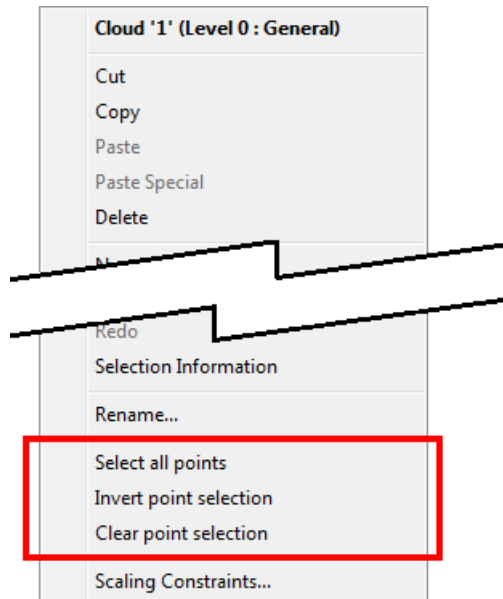


- 1 Select the clouds using a box selection. Each cloud will have a yellow selection marker.
- 2 Select **Object > Cloud > Combine** to combine the four clouds into a single cloud.

Cloud popup menu

The **Cloud** popup menu has been updated as follows:

- **Restore Selection** has been removed from the **Cloud** popup menu.
- Three new options have been added to the **Cloud** popup menu:



Select all — selects all points.

Invert selection — inverts the point selection.

Clear selection — clears the point selection.

Triangle modelling



The functionality that is available reflects the type of licence you have.

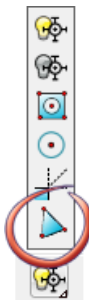
There are a number of additions and updates to triangle modelling in PowerSHAPE 2013:

- Updates to Mesh toolbars (see page 43).
- Updates to Mesh menus (see page 44).
- When generating a mesh from a point cloud, you can cancel the generation by pressing **Esc**. Alternatively, you can also use the Interrupt button on the status bar (next to the progress bar).
- You can generate a single mesh from multiple clouds using the **Generate Mesh** button (*Cloud toolbar*) (see page 35).
- When creating triangles from Parasolids, they are now generated using the 'improved' quality setting. This means that the resulting mesh, [.stl](#) or [.dmt](#) file will have fewer faults.
- The **Morph Triangle** dialogs have been renamed as follows:
Morph Triangles by Planar Regions has been renamed **Morph Triangles using Planes**
Morph Triangles by Polygon Regions has been renamed **Morph Triangles using Curves**. In addition, the order of items on the dialog has been updated to improve usability.


Toolbar options (Mesh)

The following changes have been made to toolbars that are used with a mesh:

- A new **Snap to mesh nodes** button has been added to the **Intelligent Cursor** flyout.

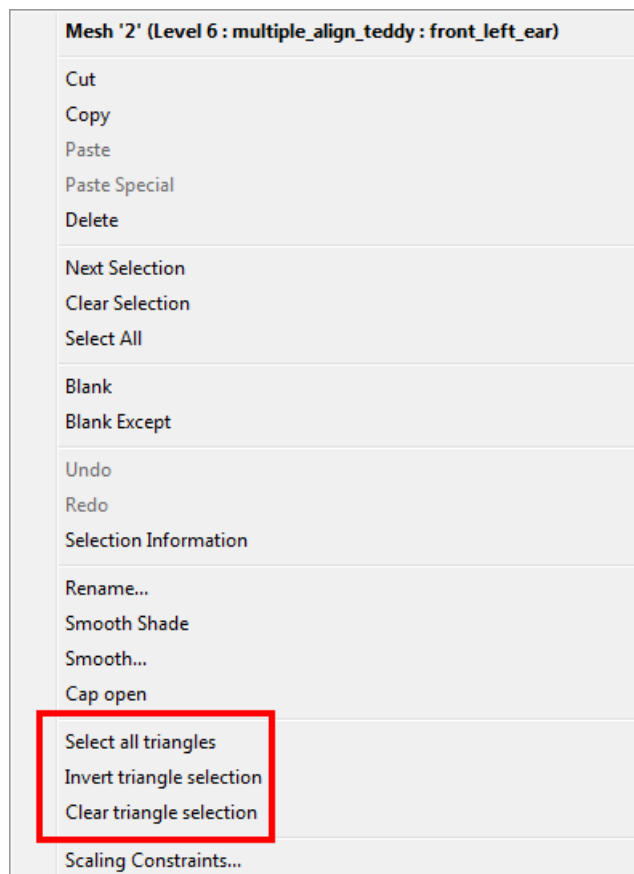


When this button is selected, the cursor will snap to the nearest node instead of the exact point on the mesh.

- You can now use the **Comparison Analysis** toolbar to export errors between a curve and a mesh to an ASCII file. The format for the errors is:
[Point on a curve], [Point position (x,y,z)], [Error unit vector(x,y,z)], [Error value]
- Points can now be projected onto a mesh using the **Project Point**  button (*General Edits toolbar*).

Menu options (Mesh)

The following options have been added to the **Mesh** popup menu:



Select all — selects all triangles.

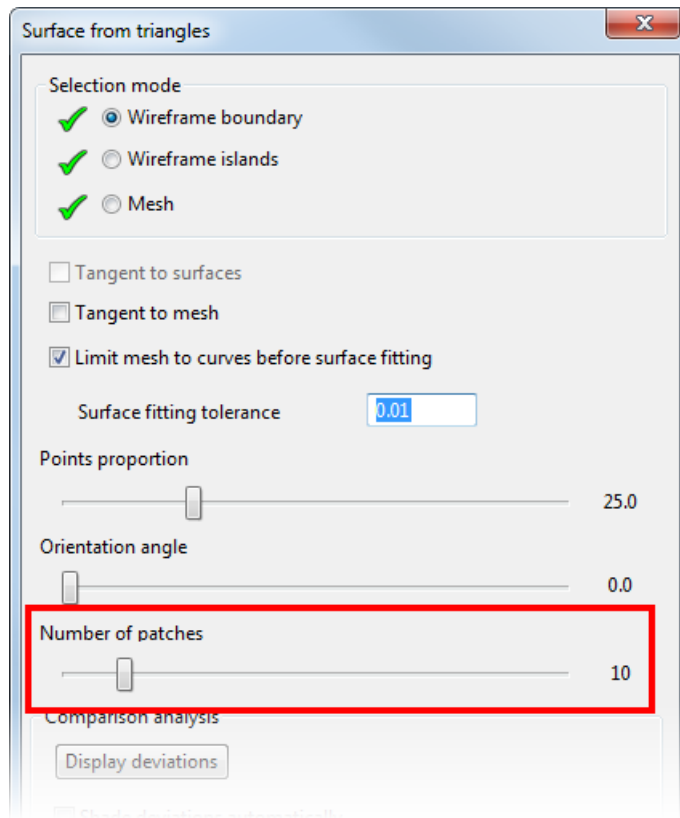
Invert selection — inverts the triangle selection.

Clear selection — clears the triangle selection.

Surface modelling

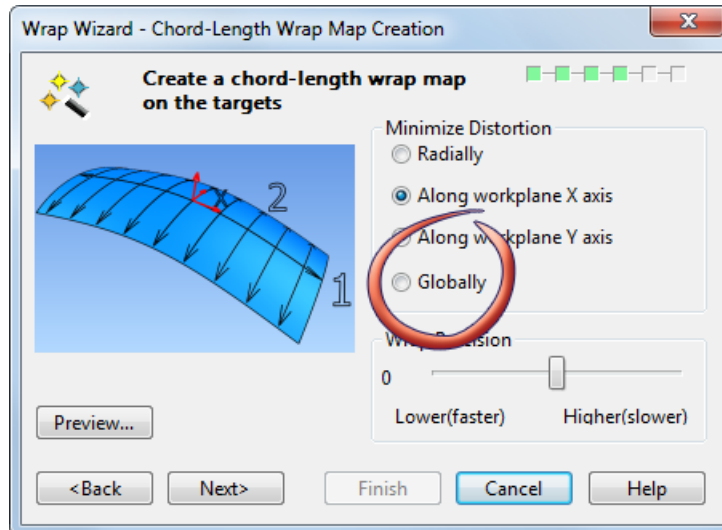
Changes have been made to the following dialog used in surface modelling:

- A new option has been added to the **Surface from Triangles** dialog in **Smart Surfer**:



Use the new **Number of patches** slider to adjust the number of patches. Increasing the number of patches will improve the fit of the surface to the triangles.

- A new method has been added to the **Chord Length Wrap Map Creation** page of the **Wrapping Wizard**:



Globally minimises the distortion equally over the whole surface.

Solid modelling

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

- Fillet recognition (see page 34)
- Solid tree (see page 49)
- Holes (see page 50)
- Solid Edit toolbar (see page 50)

Other changes to Solid Modelling

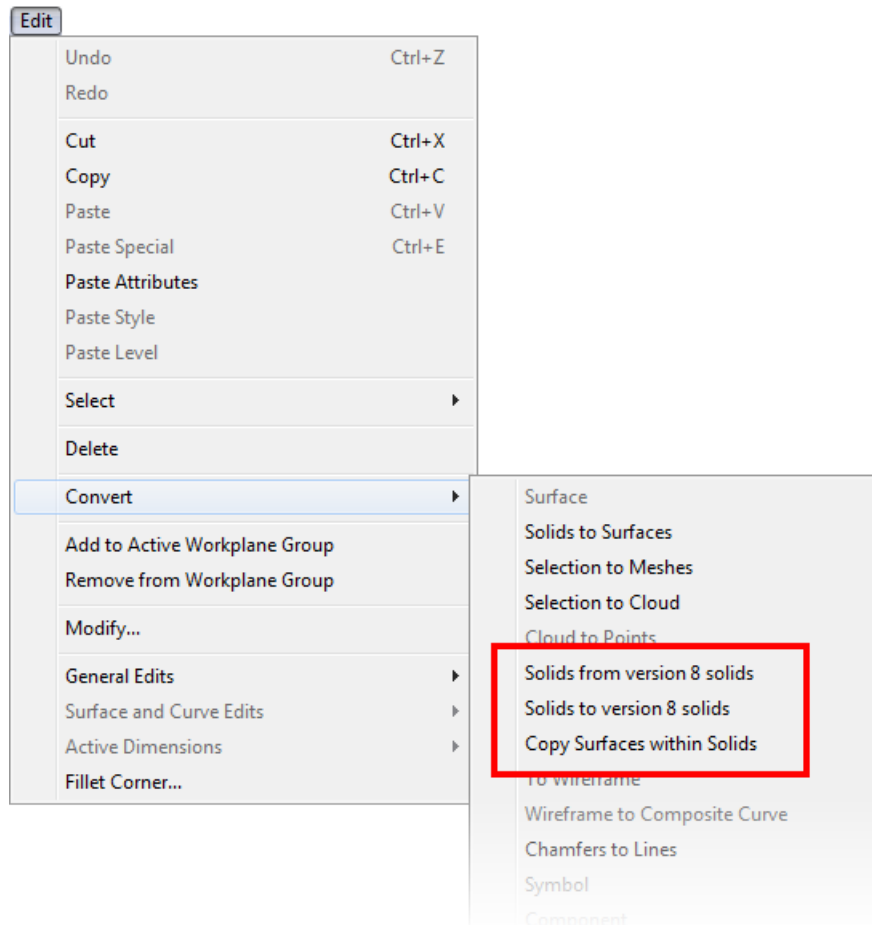
- In addition to holes, pockets, protrusions, cuts and bosses can now be exported to PowerMILL.



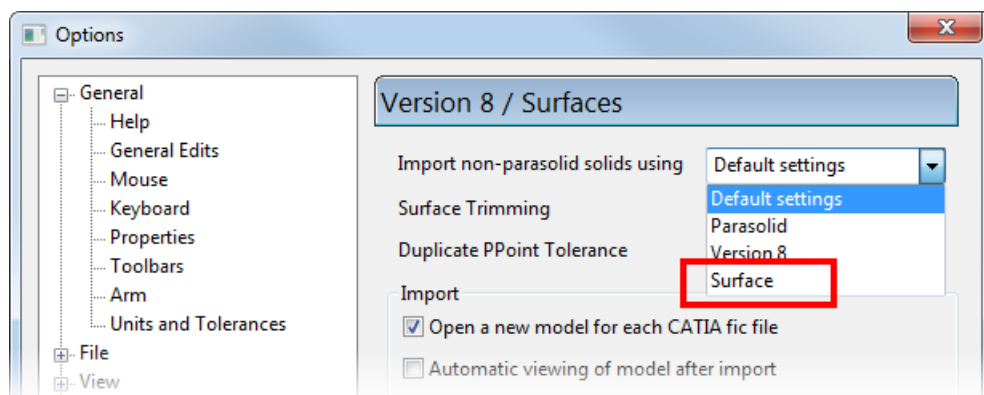
Fillet and chamfer information at the bottom and top of a feature is also exported, but is not currently used by PowerMILL.

- The **Suppress** button on the Fillet dialog now maintains its selected/deselected state when you click **Apply**.
- Updated graphics have been added to the following dialogs:
 - Hole creation.
 - Pocket /protrusion creation.
- When creating a pocket or protrusion, the values you enter in the **Pocket/Protrusion** dialog are now remembered when you click **Apply** or **OK**. This means that the next time you create a protrusion or pocket, these values will be the default values.

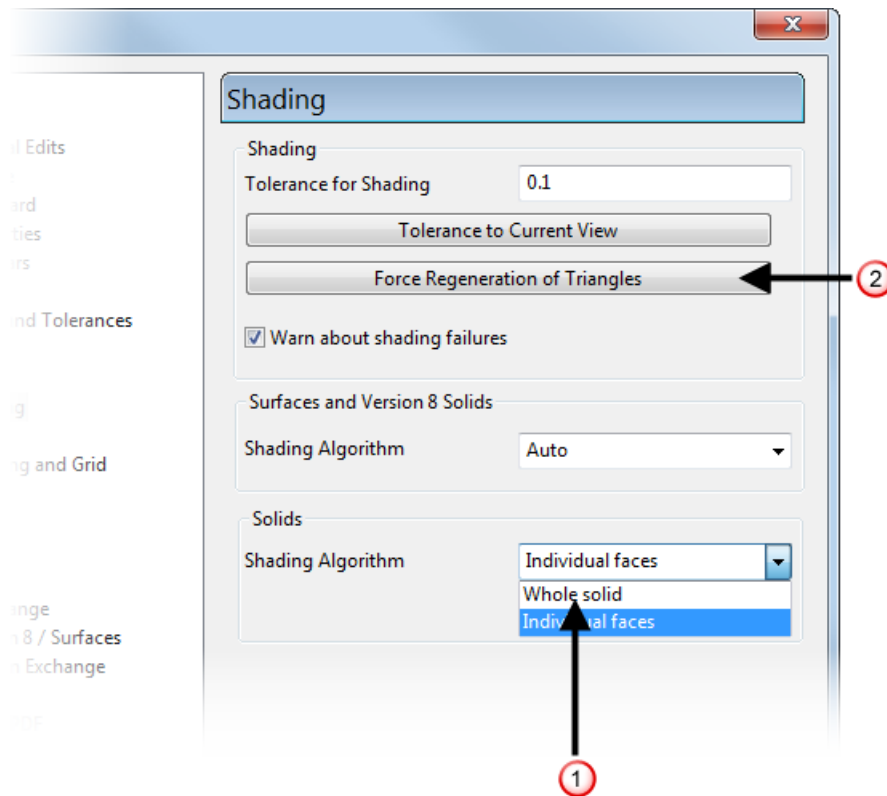
- The options on the **Edit > Convert** menu have been renamed as follows:



- A new **Surface** option has been added to the **Import non-parasolid solids** using list on the **Tools > Options > Data Exchange > Version8 / Surfaces** dialog. This lets you control the format that is used when non-Parasolid data is imported into PowerSHAPE.



- Use the **Shading Algorithm** options on the **Tools > Options > View > Shading** dialog to close gaps in the shading.



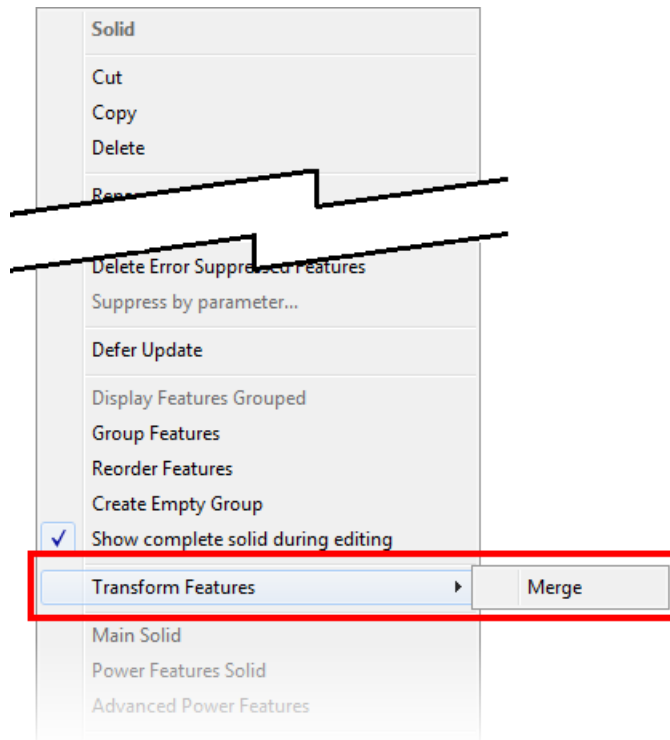
To close gaps in the shading of a Parasolid model:

- ① Select **Whole Face** from the **Shading Algorithm** (*Solids*) options.
- ② Click **Force Regeneration of Triangles**.

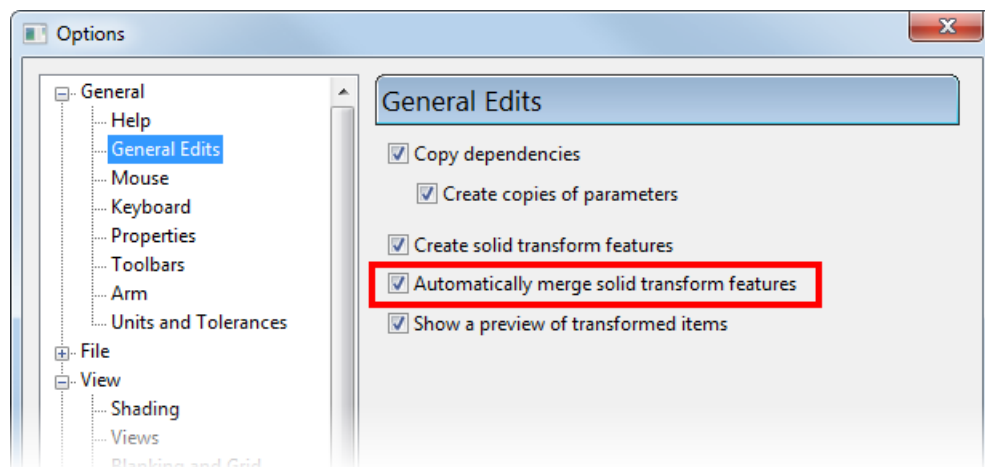
Solid tree

The following **Merge** options have been added:

- The **Transform Features > Merge** option has been added to the solid tree popup menu. This lets you merge adjacent transform features on demand.



- Adjacent transform features are merged automatically if **Tools > Option > General Edits > Automatically merge transform features** is selected (default).



These options are useful if you do several **Move** operations on a solid in succession, resulting in several **Move** transforms in the tree. The **Merge** options will merge the multiple entries into a single entry.

Holes

- A new option has been added to the **Tools > Options > Object > Holes** dialog:
Recognise tapped holes is selected by default. Deselect this option to prevent holes from being recognised as tapped.
- If changing a value on the **Hole** dialog causes the definition of the hole to be invalid, the status bar help for the greyed out **Accept** and **Apply** buttons now indicates that the values are incorrect.

Solid Edit toolbar

The **Solid Edit** toolbar has been updated to include the following new buttons:



Continuous Lasso lets you select faces of the solid by drawing a lasso around the required faces. After making the selection, move the mouse over the selected faces to highlight the single faces.

- 1 Select a solid to display the **Solid Edit** toolbar.

- 2 Click .


- 3 Click and drag the mouse around the faces that you wish to select.



Discrete Lasso lets you select faces of the solid by drawing a lasso around the required faces. After making the selection, move the mouse over the selected faces to highlight the single faces.

- 1 Select a solid to display the **Solid Edit** toolbar.

- 2 Click .

- 3 Click the mouse to define the shape of the lasso that will select the required faces. To complete the lasso, click the mouse on the initial point. The cursor changes to  to show that you can complete the lasso.



Solid Replace Faces has been added to the toolbar (see page 17). This option lets you select faces of a solid and replace them with other existing faces or surfaces.

Solid Core

PowerSHAPE 2013 includes changes to **Solid Core** functionality. A new flyout on the **Solid** toolbar has two buttons:



Create a **Solid Core** using wireframe (see page 51)



Create a **Solid Core** from selection (see page 52)

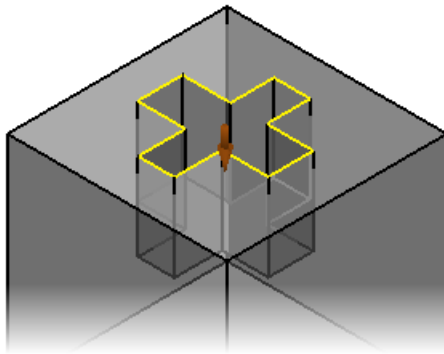
Solid Core using wireframe

Existing **Solid Core** functionality has the following additions in PowerSHAPE 2013:

- You can now use solid core functionality with:
 - a composite curve and a selected solid.
 - a composite curve and the active solid.

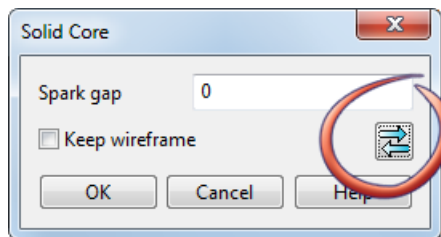
Earlier versions of PowerSHAPE required the solid to be the active solid.

- A graphical arrow is now displayed on the model to show the extrusion direction.



Reverse the extrusion direction in one of the following ways:

- Click the **Flip** button on the dialog.



- Click the arrow to reverse the extrusion direction.

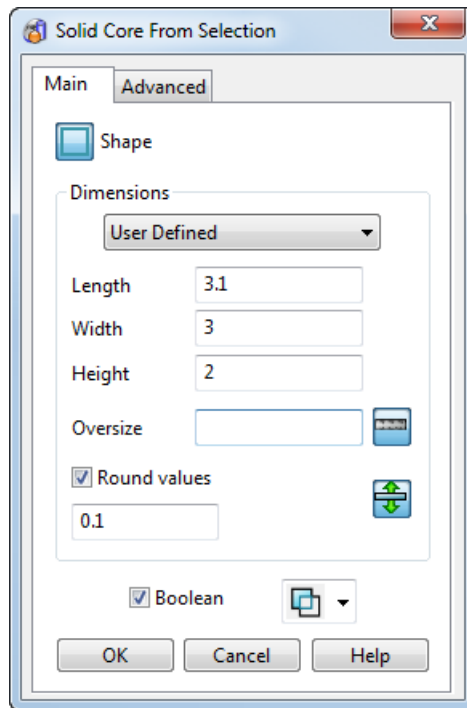
Solid Core from selection

A new **Solid Core from selection** button has been added to the **Solid Core** flyout.

To create a solid core from selected faces:

- 1 Select the faces.



- 2 Click  to display the **Solid Core From Selection** dialog.



- 3 Use the options on the **Main** tab to specify the main settings for the core.
- 4 Use the options on the **Advanced** tab to specify additional settings.

Main tab


Use the settings on this tab to specify the main settings for the core.

- 1 Click the **Shape** button to select one of the following shapes:
-  If you select this option, you can define the length, width and height of the core.
 -  If you select this option you can define the diameter and the height of the core.
- 2 Use the options in the **Dimensions** section to specify the dimensions of the core.
- Choose **User Defined** (default setting) to use set the dimensions of the core to the tightest values enclosing the selected items.

Alternatively, to use previously defined shapes, select an option from the drop-down list. The options on this list are defined in [blanks.csv](#).




Information on blanks.csv is included in the Electrode section of the PowerSHAPE on-line help.


- Enter an **Oversize** value to increase the dimensions by the amount you enter. Use  to toggle between specifying **Oversize** as a measurement or a percentage.

Entering an oversize value of **0** reverts the dimensions back to the tightest values enclosing the selected items. This is the same as choosing **User Defined** in the drop-down list

- Select **Round values** to round the dimension values.

For example, if the length of the core is **10.367** and **Round values** is selected and a rounding value of **0.5** is entered, the length will be rounded to **10.5**.

- You can define how use of the graphical drag handles is applied. If  is displayed, the opposite side of the core will also be updated when the handle is dragged.

If  is displayed, the opposite side of the core is not updated.

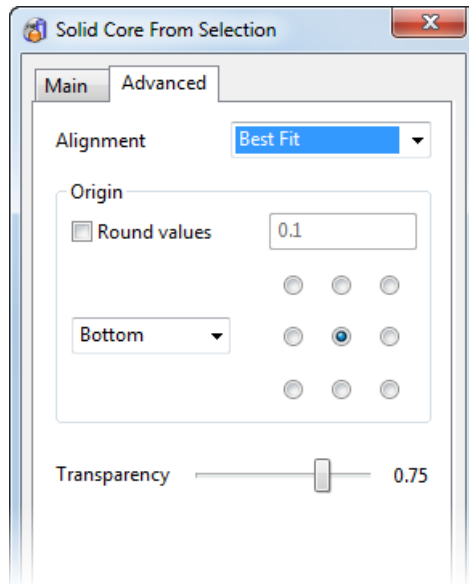
- 3 Use the **Boolean** option to control whether or not the core should be Boolean subtracted or Boolean intersected with the selected faces.
 - If deselected, the operation will create a solid core as a primitive solid.
 - If selected, the operation will create a solid with a Boolean feature.



The Boolean section does not appear if the selected items are not faces of a solid.

Advanced tab

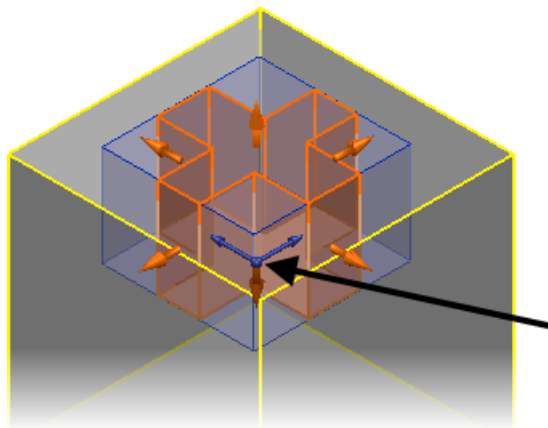
Use the options on the **Advanced** tab to specify additional settings.



- 1 Select **Best Fit** or **Workplane** from the **Alignment** drop-down list.

If you select **Best Fit**, the Z-axis of the core is aligned with the Z-axis of the currently active workplane, but it is rotated to produce the tightest bounding box around the selected items.

- 2 Use the **Origin** options to specify the origin position of the primitive solid:



- **Round values** — Select this option to round the values and enter the rounding factor.
- Use the grid to position the origin of block.
- Use the drop-down list to specify the location of the origin:
 - Bottom** — The origin will be at the minimum Z coordinate of the solid.
 - Top** — The origin will be at the maximum Z coordinate of the solid.

- 3 Use the **Transparency** slider to specify the transparency of the faces of the graphical preview of the core.



To avoid the need to re-enter the value, the rounding value that you enter is remembered for your PowerSHAPE session.

Assembly modelling

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

- Import of assemblies from STEP has been improved.

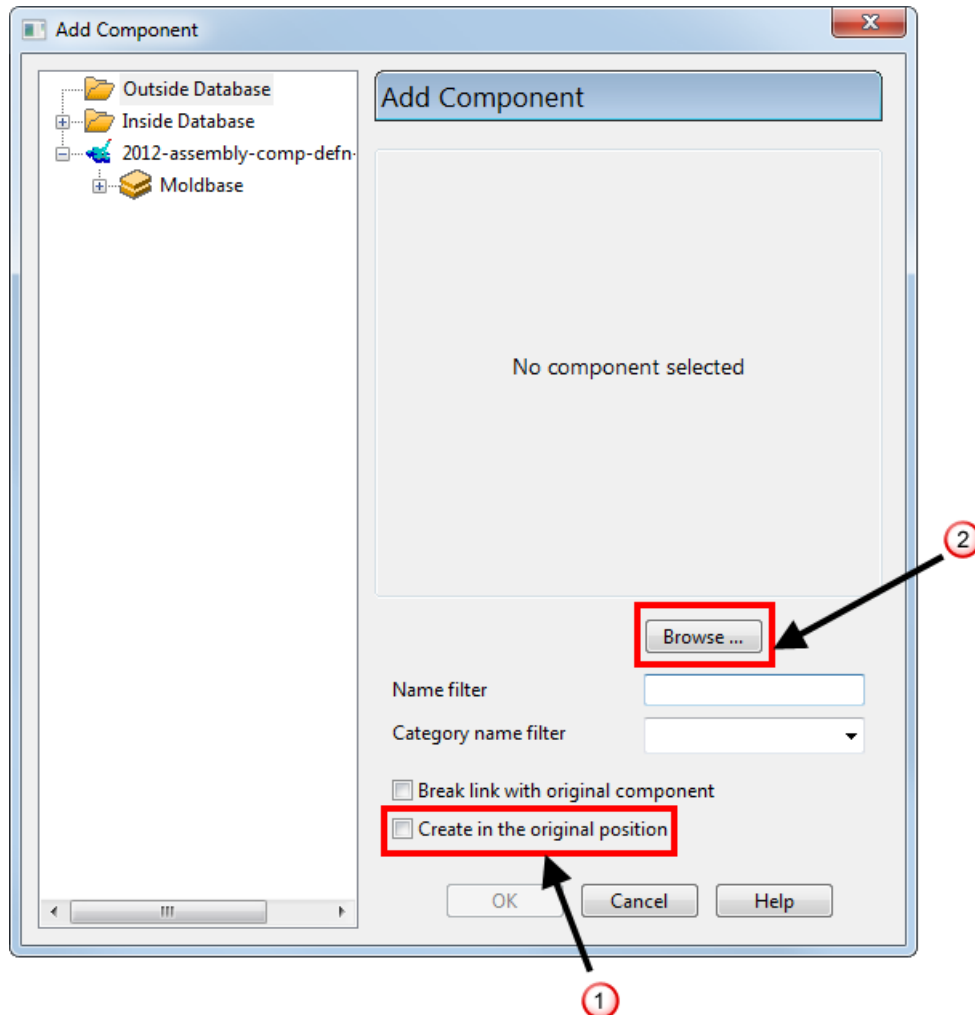
Create Relations dialog

The **Create Relations** dialog has been updated as follows:

- You can now undo each relation separately when using the **Create Relation** dialog.
- When you have selected the first attachment, the **Select second attachment** option is made live automatically.

Add Component dialog

The **Add Component** dialog has been updated as follows:



- The **Create in the original position** option has been added to the dialog ①.

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.

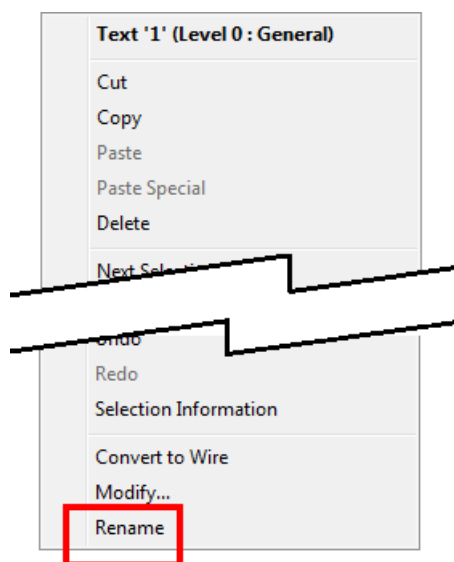
Create in the original position is useful for component positioning in a multi-model assembly. If the option is selected, the new component is inserted in the same position as in its original model.

- The position of the **Browse** button has been updated ②. Use this button to locate components to add to the model. These components must be in inactive unopened models.

Delcam Draft

Changes have been made to following areas of Delcam Draft:

- Edit Drawing dialog (see page 58)
- Edit View dialog (see page 60)
- Drawing popup menu (see page 60)
- Options dialogs (see page 61)
- Tangent construction lines from straight edges of faces are now supported.
- **Rename** has been added to the **Text** popup menu.

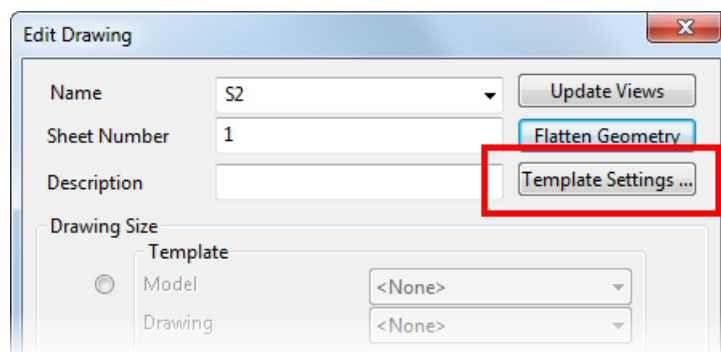


Edit Drawing dialog

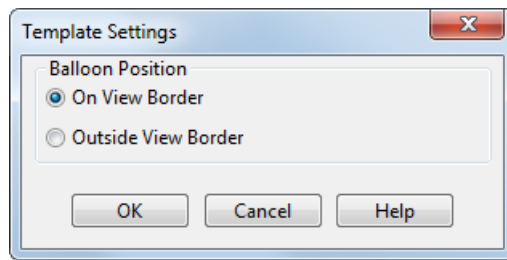
A new **Template Settings** button has been added to the **Edit Drawing** dialog.



*The **Template Settings** button is only available when you are editing a drawing in a template model. This model must have **template** in the model name.*

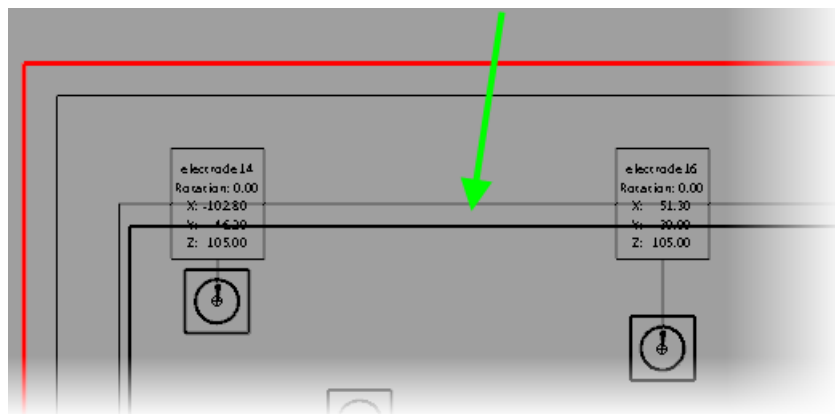


- 1 Click **Template Settings** to displays the **Template Settings** dialog.

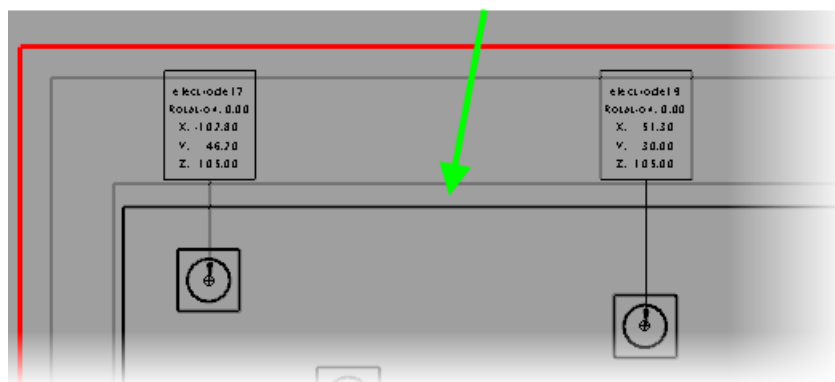


- 2 Use this dialog to specify where balloons are positioned. Balloons can be positioned in the following places:

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



- **Outside View Border**



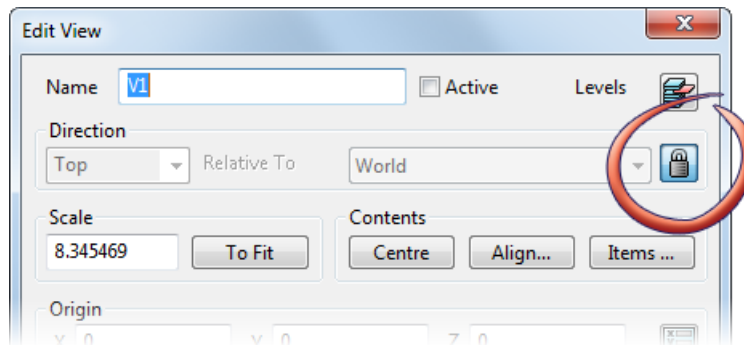
The option you choose affects the position of the balloons in all drawings that you create using the drawing template.



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

Edit View dialog

A lock has been added to the **Direction** option on the **Edit View** dialog.



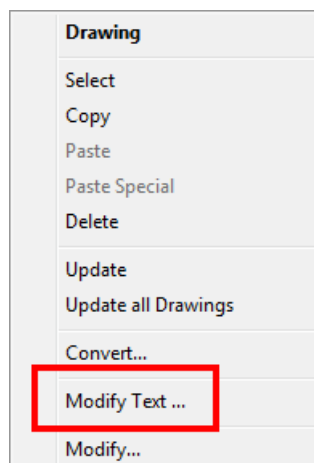
To maintain the position of balloons in relation to the view, click the lock button.

When a view is locked:

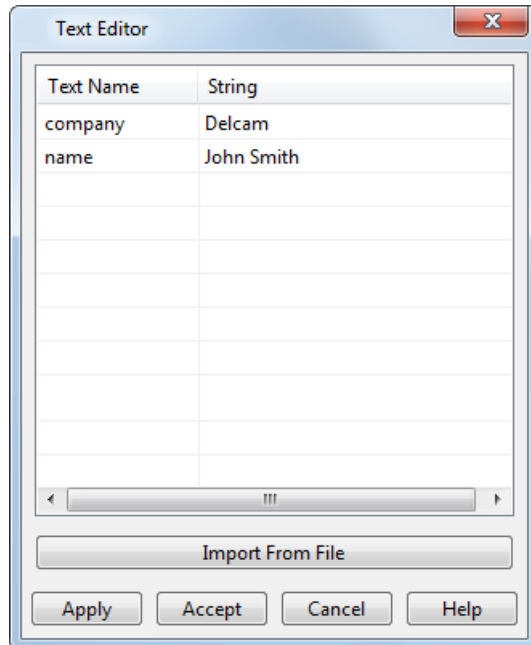
- the direction and workplane cannot be changed on the dialog
- the view cannot be dynamically rotated.

Drawing popup menu

Modify Text has been added to the **Drawing** popup menu.



- 1 Click **Modify Text** to display the **Text Editor** dialog. This contains the text names and strings included on the drawing.



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

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

Name_1,Substitution_1

Name_2,Substitution_2

Name_n,Substitution_n

Options dialog

The following changes have been made to the **Options** dialogs:

- Options have been added to the **Tools > Options > Drafting > Views > Centrelines** dialog to control the creation of centrelines. The options are selected by default.
- **Remove Trailing Zeros** has been added to the **Tools > Options > Drafting > Dimensions > Annotation** dialog.


If **Remove Trailing Zeros** is selected, any trailing zeros, after the decimal point, are removed from the dimension annotation. The option is deselected by default.

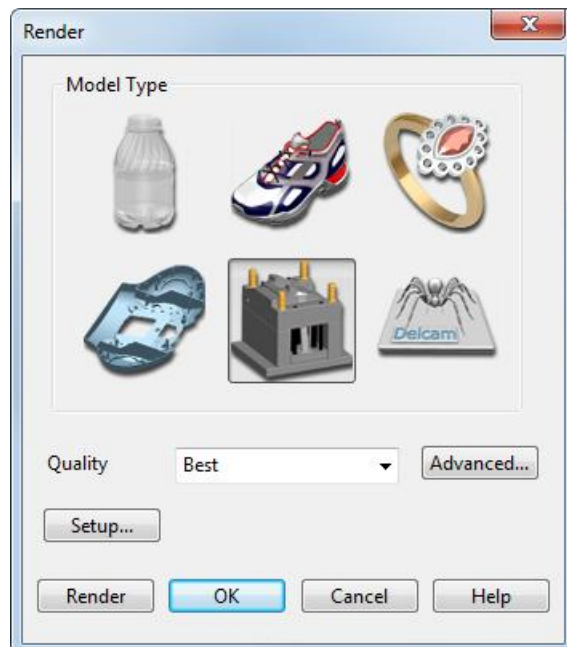
Delcam Render

Changes have been made to the following areas of Delcam Render in PowerSHAPE 2013:

- The **Render** window now opens inside PowerSHAPE. Previously, render functionality opened a separate window. Use the **Window** menu or **Ctrl+Tab** to swap between the model and render windows.
- Rendering now allows multi-threading; so you can start one render and leave it running in the background whilst you start another render and/or continue modelling in PowerSHAPE.
- The rendering dialogs have been revamped in PowerSHAPE 2013. For completeness, the following dialogs are described:
 - **Render** (see page 62)
 - **Advanced Render Settings** (see page 63)
 - **Render Setup**

Render dialog

- 1 Click  (Views toolbar) to display the **Render** dialog.



- 2 Click one of the following buttons to select the type of model you are rendering:



Bottle



Shoe



Jewellery



Plastic



Metallic



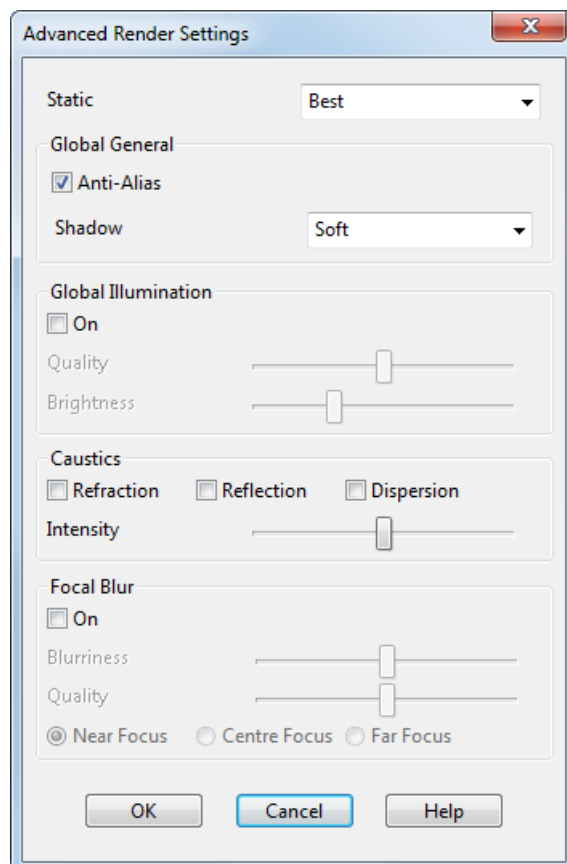
Generic/Custom

For each model type, use the **Quality** list to select the render quality level. The model type and render quality you select controls the default settings for rendering.

- 3 Click **Advanced** to display the **Advanced Render Settings** dialog (see page 63).
- 4 Click **Setup** to display the **Render Setup** dialog.

Advanced Render Settings

Use this dialog to adjust the controls for rendering of your model.



- 1 Specify the image quality by selecting one of the following **Static** options:
 - **Preview**
 - **Medium**
 - **Best** (default)
- 2 Select the **Global General** options as required:
 - **Anti-alias**.
 - Specify **Shadow** as **Off**, **Hard** or **Soft** (default).
- 3 Select **Global Illumination** to light your scene realistically using ambient, diffuse lighting. Use the sliders to adjust the **Quality** and **Brightness** of the illumination.



The higher the quality selected, the slower the rendering speed.

- 4 Select the light focusing options to be used:
 - **Refraction** focuses light through a transparent object, such as glass, water or clear plastic.
 - **Reflection** focuses light reflected off a mirrored surface, such as polished metal or jewellery.
 - Use **Dispersion** in the following ways:

Select **Dispersion** as well as the **Refraction** or **Reflection** options to create an optical effect that simulates light splitting through glass, such as a prism or gem stones.

Select **Dispersion** without selecting the **Refraction** or **Reflection** options to show the dispersion effect in glass objects.
 - Use the **Intensity** slider to adjust the brightness and smoothness of the focussed light spots.



*You must select one of the **Shadow** options from the dialog for the **Refraction** and **Reflection** options to render successfully.*

- 5 Select **Focal Blur** to create depth of field around one of the following focal points:
 - **Near Focus**
 - **Centre Focus**
 - **Far Focus**

Use the sliders to adjust the **Blurriness** and **Quality** of the focal blur:

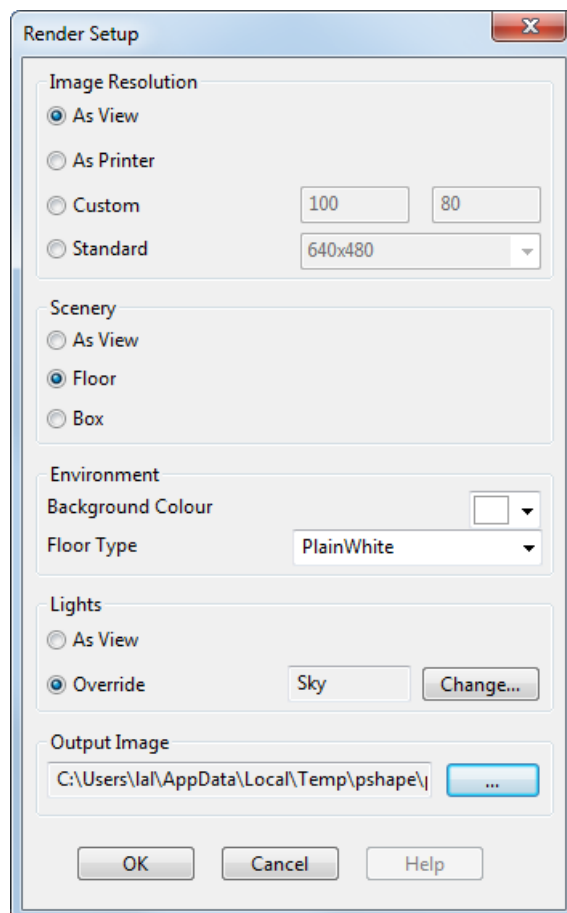
Blurriness defines distance from the focal point where the focus drops off.

Quality of render impacts on the rendering speed. Low quality gives a grainy result but has a quicker rendering speed. Higher quality gives a smoother result but has a slower rendering speed.




- 6 Click **OK** to accept the changes to the settings.

Render Setup dialog

Use this dialog to specify the settings to be used when rendering a model.



- 1 Specify the **Image resolution** to be used. Select one of the options to define the number of pixels used in the rendered image:
 - Select **As View** for the image to use the same resolution as the view.
 - Select **Printer** for the image to have same aspect ratio as the selected paper size in the selected printer minus a border all the way round.

- Select **Custom** option to define your own resolution.
 - Select **Standard** to use commonly used resolutions such as VGA (640 x 480) and full HDTV (1920 x 1080).
- 2 Select one of the following **Scenery** options for your rendered model:
- As View**
- Floor**
- Box**
- 3 Specify the following **Environment** elements for your rendered model:
- a Select a **Background Colour** from the colour palette.
-  *If the **Enhanced Shading** or the **Reflected Environment** options are used, these settings override any background colour selected.*
- a Select a **Floor Type** pattern for your rendered model from the options in the list. The floor patterns have a fixed scale in millimetres so you may need to scale your model to achieve realistic proportions.
- 4 Specify the **Lights** to be used:
- Select **As View** to use the same light settings as the modelling view.
 - Select **Override** and click **Change** to display the **Light Studio Selector** dialog. Use this dialog to select type of lights you want to use.
-  *By default, the lights used are Standard. These are fine for modelling but they do not make a very interesting image. However, lights that make interesting images, tend not to be suitable for modelling. By default, the Lights option overrides the model view's lights with a set called SpotsFill.*
- 5 Specify the path for the **Output Image**. Use the default path that is displayed or click  to select the required location.
- 6 Click **OK** to accept the changes to the setup.

Delcam Electrode

Delcam Electrode now incorporates:

- Design in PowerSHAPE. This uses **Electrode Wizard - Design** (formerly Electrode Wizard).
- Machining in PowerMILL.
- Inspection in PowerINSPECT.

All manufacturing data is transferred seamlessly between the modes using the [.trode](#) file format.

Changes have been made to the following in the 2013 of Electrode.

Electrode Wizard - Design (see page 67)

Frames (see page 70)

Electrode Wizard - Export Options page (see page 71)

Options (see page 73)

Electrode Wizard - Design

The following changes have been made to **Electrode Design Wizard** in PowerSHAPE 2013:

- Graphic images in the **Electrode Design Wizard** have been updated.
- The burn region, base and clearance on pre-modelled solids are identified and the burn-region shading is added. These shadings, along with values for burn height and base height are exported in the [.trode](#) file and read by the **Electrode Machining Wizard** in PowerMILL.

The solid is no longer checked to ensure that it is watertight when the **Electrode Design Wizard** is started.

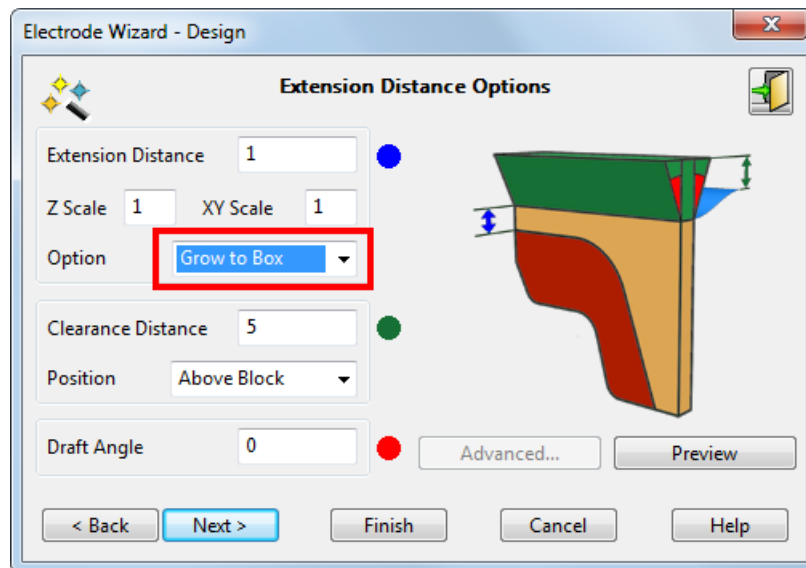
- Collision checking between pre-modelled electrodes and the part has been speeded up and the accuracy improved. Checking is now controlled by a new option on the **Options > Electrode > General** page (see page 73).
- A new **Grow to box** option (see page 68) has been added to the **Options** list on the **Extension Distance Options** page.
- Use the **Edit Drawing** dialog to specify the positioning of electrode balloons (see page 58). By default, automatically positioned electrode balloons will no longer overlap the view border.
- The view direction of electrode template drawing is now locked. (see page 60) This is to maintain the position of the balloons in relation to the view.

If you have modified the electrode template model you will need to lock the views in your model to take advantage of this improvement:

- 1 Open the model.
- 2 Display the drawings.
- 3 Lock the views.

Grow to box

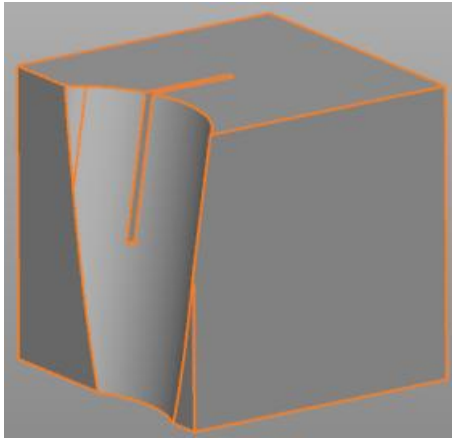
Grow to box has been added to the **Options** list on the **Extension Distance Options** page.



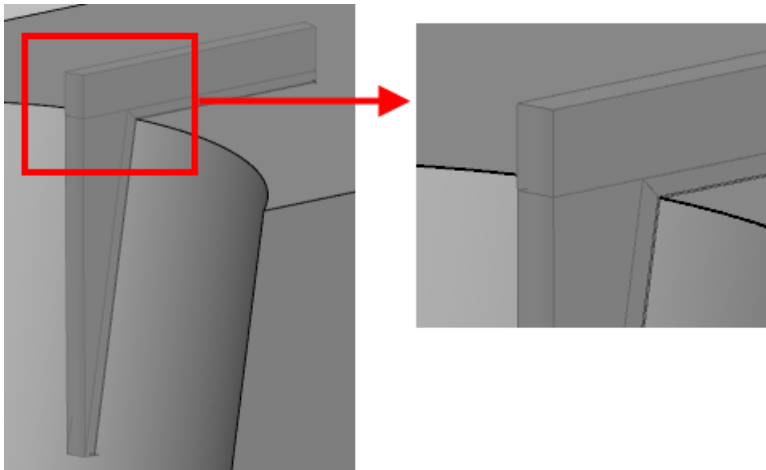
Grow to box uses an expanded bounding-box of the burn region as a target to limit the extension. The surfaces will generally be extended further than the extension distance until they intersect with this box. This produces a square, planar side to open-ended ribs.

Example

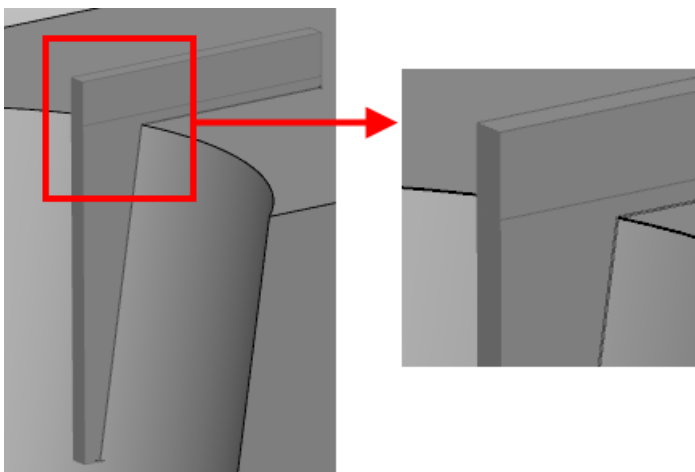
This example uses the same model used to produce two different electrodes.



The normal extension methods do not give the desired result because the angle on the edge of the rib produces an angled face in the clearance region where the shape is not critical.



The grow to box solution gives a neater electrode.

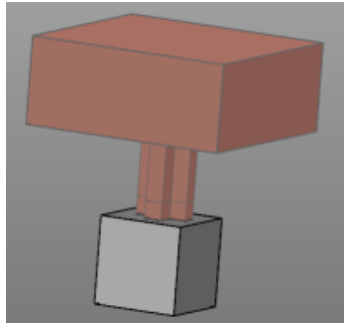


*The greater the curvature of the edge of the rib, the greater the benefit of using **Grow to box**.*

Frames

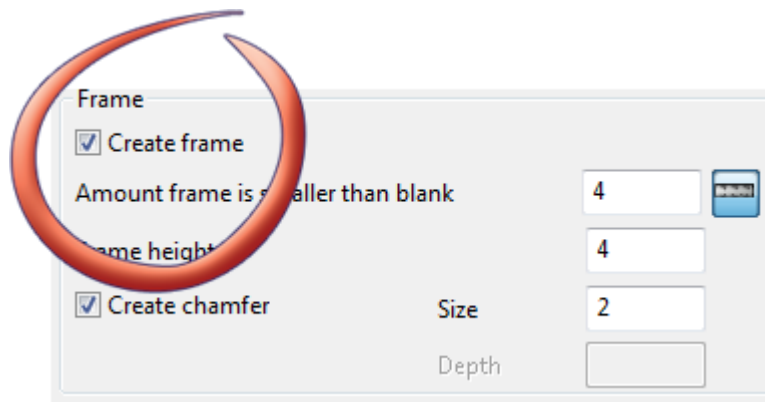
PowerSHAPE 2013 includes support for electrode frames. An electrode frame is an offset section of the electrode base that is machined accurately to allow pre-setting of the electrode.


The following electrode was created without a frame:

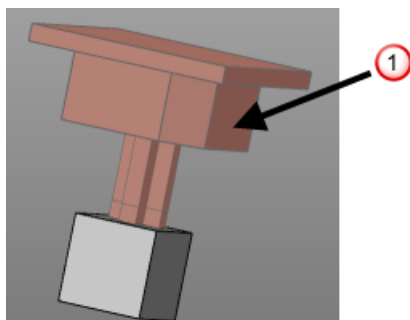


To create a similar electrode with a frame:

- 1 Select **Create frame** on the **Tool > Options > Manufacturing > Electrode > Base** dialog.



- 2 Click **OK**.
- 3 Click  to start the Electrode Wizard.
- 4 Use the **Electrode Wizard** to create an electrode that includes a frame ①.



Defining electrode frames using the blanks.csv file

You can define electrode frames by adding fields to the [blanks.csv](#) file that specifies the default blanks. These new fields define:

- frame length.
- frame width.
- frame height.
- chamfer size (optional).

For example:

0,blank_with_frame,12,,30,Copper,,,8,6,4,3

In this blank definition, the last four arguments specify the frame:

Length: 8

Width: 6

Height: 4

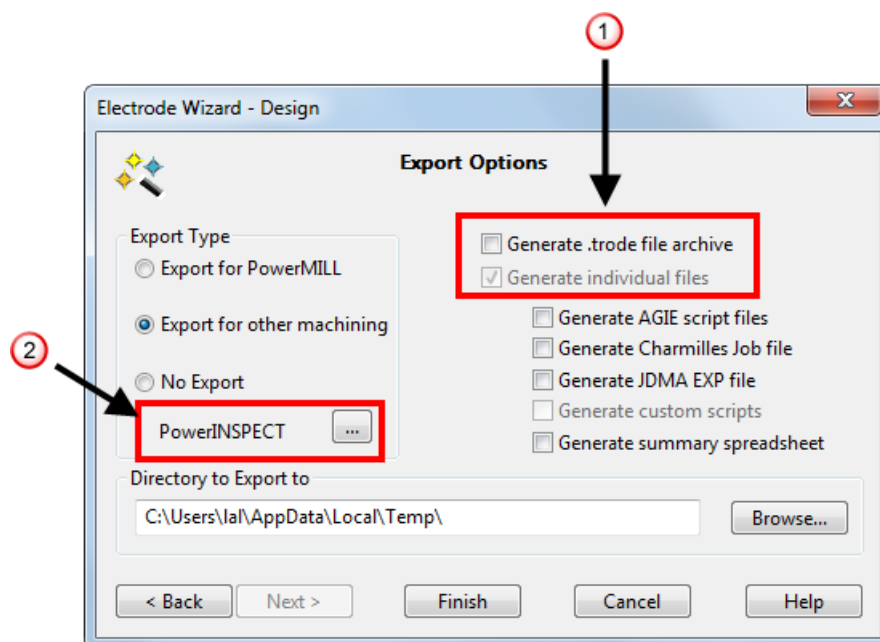
Chamfer size (CS): 3



Further details on the creating blanks definitions is included in the [blanks.csv](#). This can be edited with any text editor.

Export Options page (Electrode Wizard - Design)

The **Export Options** page of the **Electrode Wizard - Design** has been updated in PowerSHAPE 2013.

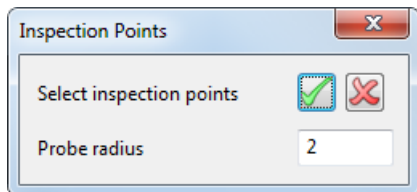


① Use the new option on the **Export Options** page to:

- generate a [.trode](#) file archive

- specify if you want to include individual files in the export directory in addition to the [.trode](#) file archive. This option is selected by default if you are generating a [.trode](#) file archive.
- export to JDMA EXP format. This format is used by Makino EDM machines

② Select inspection points and specify the radius of the inspection probe using the **Inspection Points** dialog.

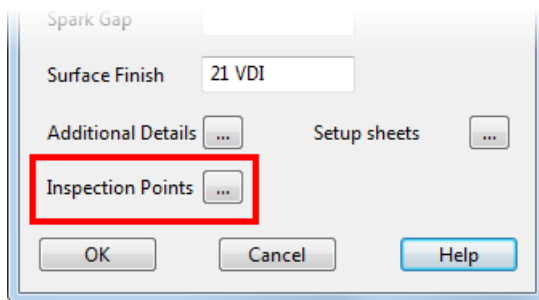


*This dialog is displayed by clicking the **PowerINSPECT** button on the **Export Options** page.*

These inspection points are exported as PowerINSPECT compatible [.pts](#) files that are included in the [.trode](#) file archive.

Editing inspection points

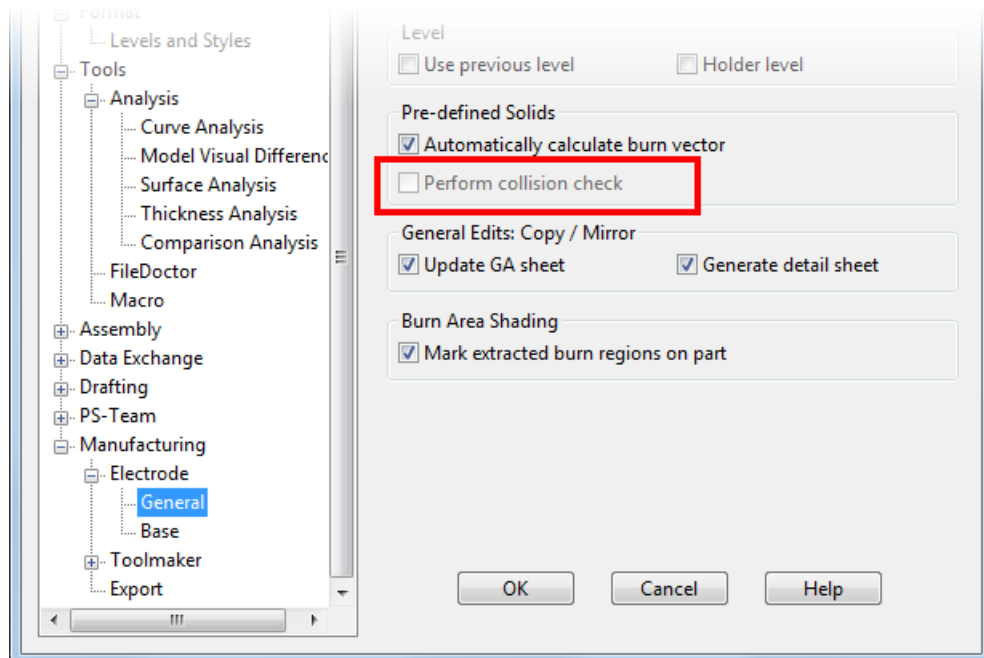
Inspection points on an existing electrode can be modified by clicking **Inspection Points** on the **Edit Electrode** dialog to display the **Inspection Points** dialog.



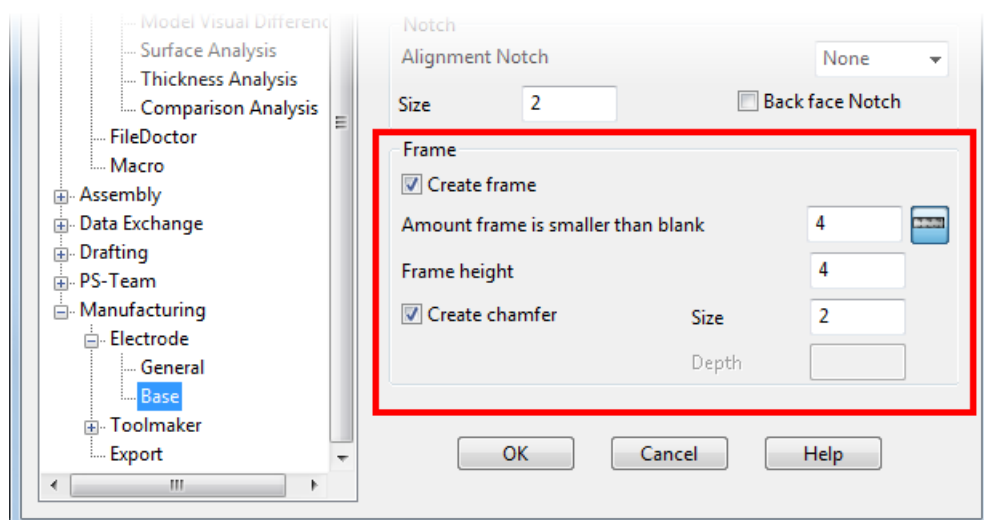
Options

The following changes have been made to the electrode **Options** dialogs:

- Use the **Perform collision check** option on the **Options > Electrode > General** page to control collision checking between pre-modelled electrodes and the part.



- Use the **Frame** section of the **Options > Manufacturing > Electrode > Base** page to define a frame to be applied to the base if parameters are not defined in the [base.csv](#) file.



If **Create frame** is selected, you can define the following:

- Frame height.
- Size and depth of a chamfer on the frame.

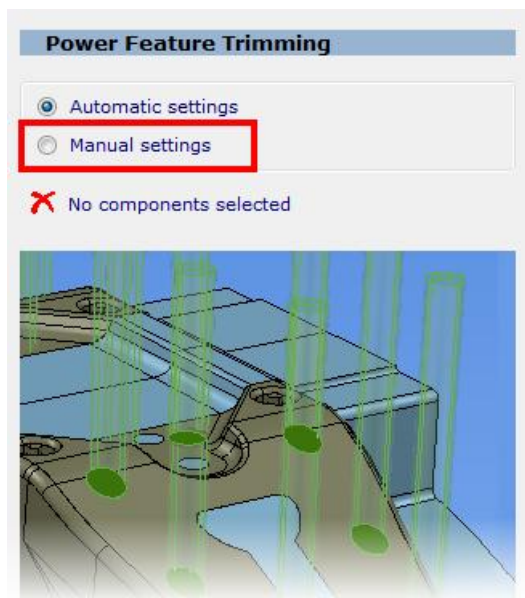
Delcam Toolmaker

Changes have been made to the following areas of Delcam Toolmaker:

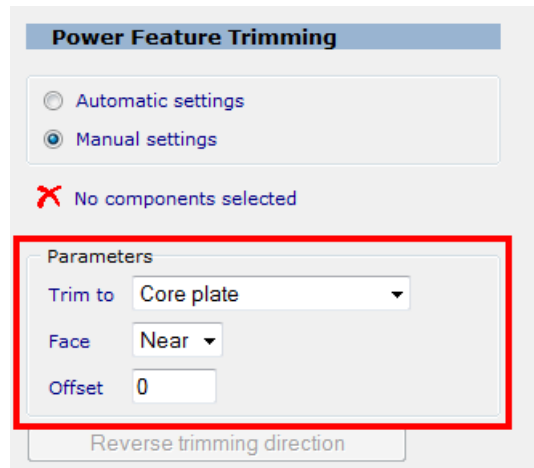
- Images have been updated on the following:
 - Cooling Wizard dialogs.
 - **Define mold locks** dialog in the Mold Lock Wizard.
- **Power Feature Trimming** has been revamped (see page 74) to provide more flexibility.
- Tabs on the **Cavity-Core Wizard** has been updated (see page 77).

Power Feature Trimming

Power Feature Trimming has been revamped to provide more flexibility. As an alternative to **Automatic Settings**, you can now use the **Manual settings** option to trim pins to the selected component, solid or surface.



If you select **Manual settings**, the **Parameters** section is displayed; use this section to specify the parameters to be used for trimming the pins.



The **Parameters** section contains two drop-down menus:

- **Trim to** defines the plate and contains these options:
 - **Core plate** (available when core pin, ejector sleeve or ejector pin components are selected)
 - **Cavity plate** (available when core pin or return pin components are selected)
 - **Selected plate**
 - **Selected solid / surfaces**
- **Face** defines the face of the plate to trim to and contains these options:
 - **Near**
 - **Far**

If you select components with different functions (for example, an ejector pin and a return pin), the **Trim to** drop-down list will contain all options (Core plate, Cavity plate, Selected plate, Selected solid /surfaces). When you click **Trim**, a dialog will ask you to confirm if you want to apply trimming to all selected pins.

Reverse trimming direction lets you change the trimming direction for all selected components in the model. The trimming direction for individual selected components is indicated by instrumentation. You can change the trim direction for a component by clicking on the instrumentation arrow.

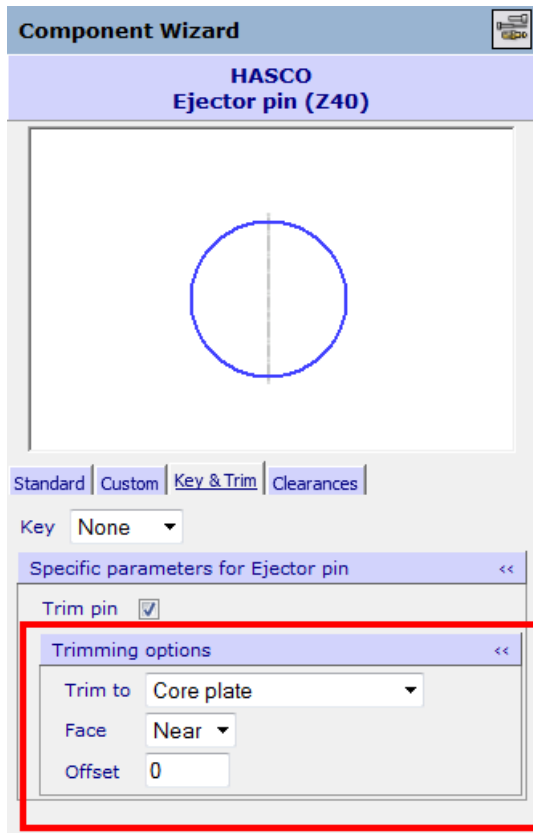
When you have specified the trim parameters, click **Trim** to apply the changes when you re-generate the Power Feature.



*If you are unhappy with the trimming operation, click **Undo** (Main toolbar) to undo the previously applied trimming operations.*

Trimming options on the Component Wizard

Manual settings from the **Pin Trimming Wizard** are also available on the **Key & Trim** tab of the **Component Wizard**.



Cavity-Core Wizard

In the Cavity Core Wizard, the **Inserts/Pockets** and **Multiple inserts** tabs have been merged into the **Inserts and pockets** tab.

Dimensions

Minimum pocket clearances

In X, MX: 30 In Y, MY: 30 In Z, MZ: 10

Store as default

[Show the clearances diagram](#)

Mold base: **Inserts and pockets** | Sides

Offsets of the pocket(s)

In X: 0 In Y: 0

☐ Through pocket(s)

Define vertical position of the inserts by:

☒ D1 and D2 ☐ D1 and G ☐ D2 and G

D1: 59.7 D2: 64.69 G: 5

Use the general edits commands to move, copy, rotate, mirror, pattern the selected inserts

Insert
Insert "1"

Update the list

☒ Define a pocket ☐ Combined pocket

Both

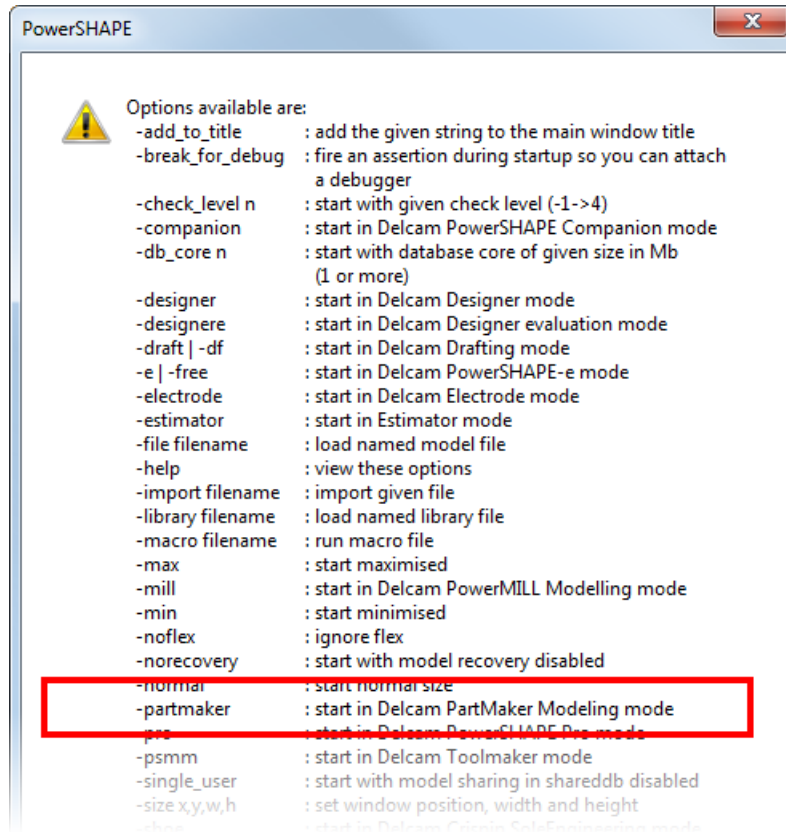
Pocket..

Other changes

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

- Import / Export (see page 79), including changes to **Options** dialogs used in data exchange.
- Macro variables for planes of primitives (see page 80)
- Macro variables to convert between drawing, view and world space (see page 81)

- **PowerSHAPE Companion** has been renamed **PartMaker Modeling**.
- **partmaker** has been added to the PowerSHAPE startup switches.

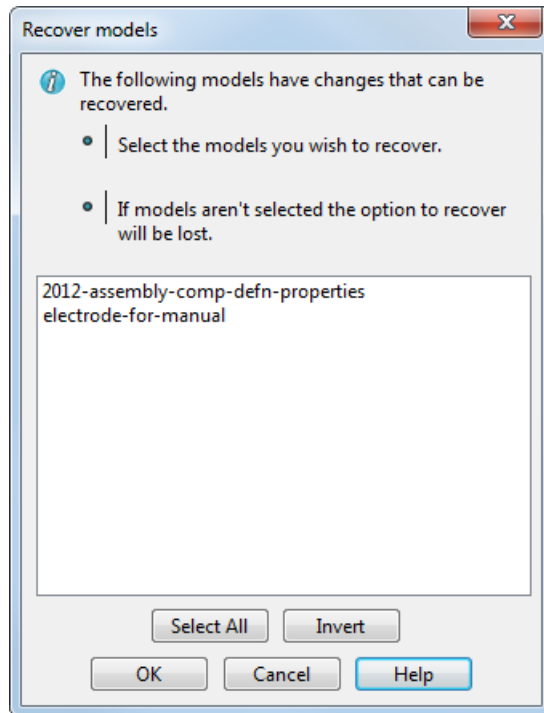


If you start the program in this mode, it will use the COMPANION paf and will not start if the correct licence is missing.

In PartMaker Modeling mode, the default tolerances will be as follows:

- general tolerance: *0.0001* inches.
- shading tolerance: *0.001* inches.

- The **Recover models** dialog lets you select the models that you wish to recover in the event that PowerSHAPE closes unexpectedly.



Import / Export

The following general changes have been made to import/export:

- **.ddz** file format is now used to transfer data in **Sketcher** and **PowerMILL Modelling**.
- When using Delcam Exchange 6710 or later, the speed of importing IGES files containing large clouds has been improved.



The import speed is unchanged if you are using an earlier version of Delcam Exchange.

Data Exchange options

The following changes have been made to the **Tools > Options > Data Exchange** dialog:

The following options have been removed from the **Tools > Options > Data Exchange** dialogs:

- **Use Exchange for IGES import** has been removed from the **Delcam Exchange** dialog
- **Use Exchange for IGES export** has been removed from the **Delcam Exchange** dialog
- **Use world workspace** has been removed from the **STL/DMT** dialog.

The following **Tools > Options > Data Exchange** dialogs have been updated:

Delcam Exchange

IGES

Parasolid

STL/DMT

The following dialogs have additional options in PowerSHAPE 2013:

- **Use DDX as intermediate file** has been added to the **Tools > Options > Data Exchange > Delcam Exchange** dialog. The option was previously on the **Tools > Options > Data Exchange > DDX** dialog, which has been removed in PowerSHAPE 2013.
- A new **Surface** option has been added to the **Import non-parasolid solids using** list on the **Tools > Options > Data Exchange > Version8 / Surfaces** dialog (see page 46).

Macro variables for planes of primitives

Use the following to enquire the details of the axes of the workplane of a primitive solid.

The following return the X, Y or Z unit axis vector of the primitive's workplane. The vector is defined in relation to the currently active workplane:

SOLID[<name>].XAXIS

SOLID[<name>].YAXIS

SOLID[<name>].ZAXIS

The following return the X, Y or Z entity of the unit axis vector of the primitive's workplane. The vector is defined in relation to the currently active workplane:

SOLID[<name>].XAXIS.X

SOLID[<name>].XAXIS.Y

SOLID[<name>].XAXIS.Z

SOLID[<name>].YAXIS.X

SOLID[<name>].YAXIS.Y

SOLID[<name>].YAXIS.Z

SOLID[<name>].ZAXIS.X

SOLID[<name>].ZAXIS.Y

SOLID[<name>].ZAXIS.Z

These commands can be used for:

- solid primitives
- surface primitives
- solid feature primitives

PRINT SOLID[1].XAXIS

PRINT SOLID[1].XAXIS.Z

PRINT SURFACE[1].YAXIS

PRINT SURFACE[1].YAXIS.Z

PRINT FEATURE[1].XAXIS

PRINT FEATURE[1].XAXIS.Y

Macro variables to convert between drawing, view and world space

Use the following variables to convert between drawing, view and world space:

DRAWING[drawing_name].VIEW[view_name].DRAWING_TO_VIEW[x ; y ; z]

DRAWING[drawing_name].VIEW[view_name].DRAWING_TO_WORLD[x ; y ; z]

DRAWING[drawing_name].VIEW[view_name].VIEW_TO_DRAWING[x ; y ; z]

DRAWING[drawing_name].VIEW[view_name].WORLD_TO_DRAWING[x ; y ; z]

You can also use X/Y/Z modifiers with these variables:

DRAWING[drawing_name].VIEW[view_name].DRAWING_TO_VIEW[x ; y ; z].X

returns the x-ordinate of the converted point.

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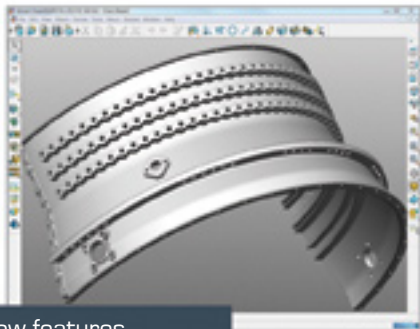
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PowerSHAPE 2013

PowerSHAPE 2013



new features

Delcam TV



www.delcam.tv

PowerSHAPE 2013 Learning Zone



www.delcam.tv/lz

PowerSHAPE Website



www.powershape.com



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