



Advanced  
Manufacturing  
Solutions

# PowerSHAPE 2013 R2

What's New



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PowerSHAPE 2013 R2

# 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 R2

Major changes included in PowerSHAPE 2013 R2 are:

Option to create a **Rewind Position** in the solid tree (see page 18).

Enhancements to **Solid Split** (see page 28).

Addition of a **Healing** page to the **Solid Doctor** (see page 16).

Updates to **Electrode Design Wizard** to enhance export options (see page 38).

Additions to the **Cloud editing** toolbar (see page 8).

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## User interface

Changes have been made to the interface and general editing (see page 4) in PowerSHAPE 2013 R2.

### Menus

The following menus have been updated in PowerSHAPE 2013 R2:

- **Edit Surface** has been removed from the **Solid** popup menu.
- New options have been added to the **Edit** menu and the **Solid tree** popup menus (see page 22).
- **Display Length** and **Display Angle** have been added to the **Dimension** popup menu when adding angular dimensions to an arc (see page 36).

### Toolbars

The following toolbars have been updated in PowerSHAPE 2013 R2:

- There are additional buttons on the **Cloud** toolbar (see page 8).
- **Thicken mesh** has been added to the **Mesh modelling** toolbar (see page 10).
- **Create an outline curve** has been added to the **Curve** toolbar (see page 12).
- Additional features are available when using the **General Editing** toolbar (see page 4).

## Dialogs

- A number of dialogs have been updated (see page 2).
- When a dialog is displayed, you can now click the right-mouse-button in the graphics area as a shortcut for **Apply** or **Accept**.



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

## Dialogs

The following dialogs have been added or updated in PowerSHAPE 2013 R2:



- The **Position** dialog now remembers the last tab that was used. You can now select composite curves when you are using the following pages of the **Position** dialog:

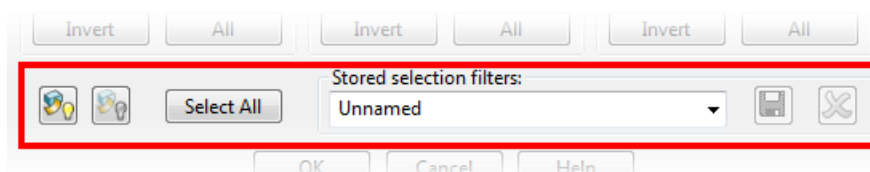
**Key point**

**Along**

**Normal**

**Intersect**

- The **Select Objects by Filter** dialog has been updated:
  - Use **Blank**  to blank items according to the filter you have set. Use **Unblank**  to unblank previously blanked items.



- Use **Type**, **Style** and **Level** to specify a filter, enter a filter name to replace **Unnamed** and click **Save selection filter** button. The filter is added to the **Stored selection filters** drop-down list.

- Use the selection filter to specify the type of item you want to snap to.
- You can now enter creation mode with the selection filter set.
- The **Create Outline Curve** dialog is displayed by clicking **Create an outline curve** (*Curve toolbar*) (see page 12).
- **Geometric Tolerance** dialog can be displayed by double clicking on the tolerance balloon
- The **Export Options** page of the **Electrode Design Wizard** has been revamped (see page 40).
- The following **Toolmaker** dialogs have been updated (see page 41):
  - **Cooling Wizard**
  - **Mold Base Wizard**
  - **Power Feature Trimming**
  - **Component Wizard**

### Solid modelling

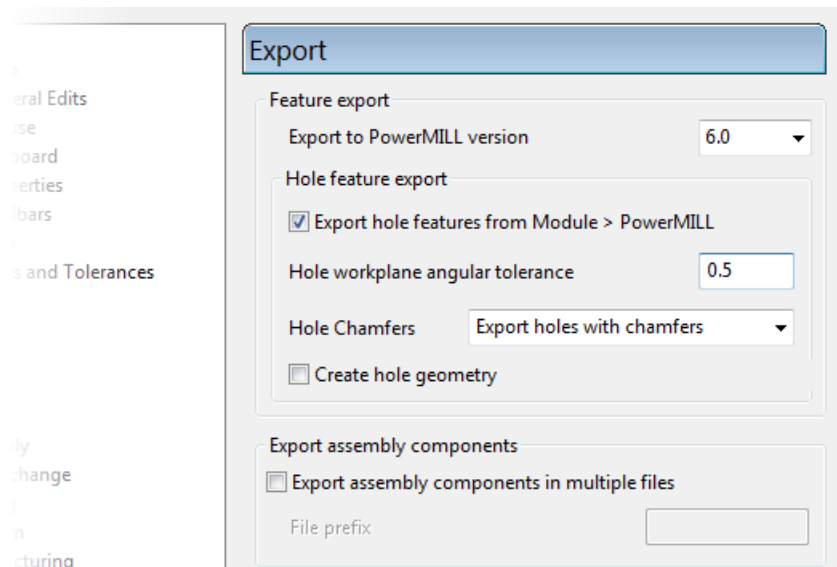
- **Solid Doctor Heal** dialog (see page 17) has been added to the **Solid Doctor**.
- A **Standard Dimensions** section has been added to the **Counterbored** option of the **Hole** dialog.
- There is a new **Solid Split** dialog (see page 28).
- **Recognise Solid Fillet** dialog has been updated.

### Options

The following **Options** dialogs have been updated:

- **STL/DMT** options page has been renamed **Triangle/Mesh**.
- Labels on the **Delcam Exchange** page have been updated.
- Labels on the **IGES** page have been updated and the **Export** section has been removed.
- New options have been added to **Tools > Options > Object > Solids** dialog (see page 25).
- There is a new **Font paths** section on the **Tool > Options > Text** dialog (see page 35).

- **Manufacturing > Export** includes updated options.



The **PowerMILL version** drop-down has been renamed to **Export to PowerMILL version** and now includes:

- **6.0** — This is the default setting and only exports holes.
- **2013** — This export all feature types using multiple feature sets.

The following option has been renamed:

- **Export hole features** to **Export hole features from Module > PowerMILL**

The following options have been removed from the dialog:

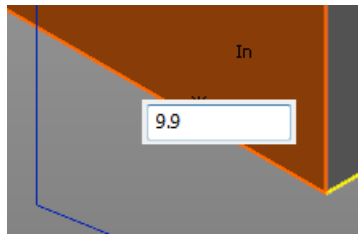
- **Create hole features for all DGK.** This pre-dates the XML export
- **Generate hole toolpaths** which used to work with the old macro export

## General Editing

The following changes have been made to General Editing in PowerSHAPE 2013 R2:

- Enter the exact value of a dimension in the box that is displayed when you have dragged instrumentation as follows:  
**Move** — modify the distance value after dragging the distance arrow along a direction.

**Rotate** — modify the angle value after dragging the selected item.



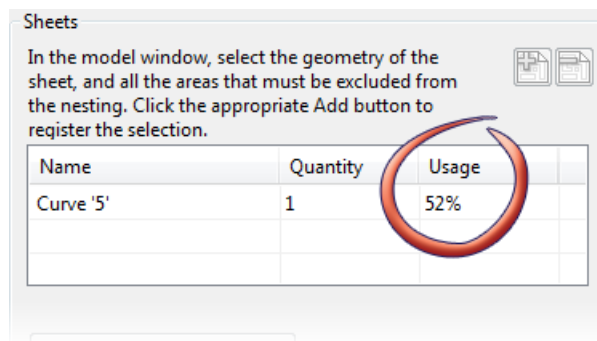
*If you don't wish to enter a value, you can ignore the box; it will disappear when you continue with your work.*

- When switching between **Move**, **Rotate** and **Scale** operations, the origin point is now preserved. Previously, the origin was reset to a default position.

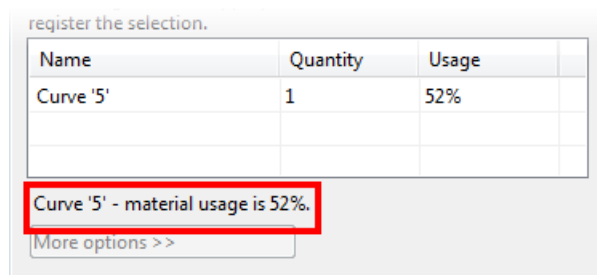
## Nesting

Nesting has been updated to include the following:

- A workplane can be registered with the part for nesting. This is useful if you want to machine the multiple parts, because the workplanes can be used to copy the toolpaths.
- Material usage statistics for each sheet. The **Usage** column on the **Nesting** dialog displays the area occupied by parts nested on all the instances of the sheet.



As you move the mouse cursor over a sheet instance in the current model, the area usage for a particular sheet instance is shown below the list of sheets. This is useful when you have several sheets on a model.



- New **Border** and **Quantity** options have been added in the **Parts** section of the dialog.

Parts

In the model window, select the geometry of one or more parts. Click the appropriate Add button to register the selection.

Name	Border	Quantity	Nested
Curve 'filler'	1	200	48
Curve 'square'	1	2	2

Less options <<

Border

1

Quantity

2

Rotation step angle ▼ 2

Maximum quantity Part priority

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## Direct modelling

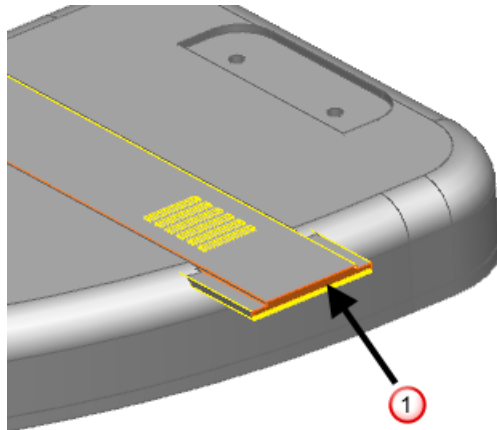
Changes have been made to the following areas of Direct Modelling functionality in PowerSHAPE 2013 R2.

- Replace Face (see page 7)
- Feature recognition - Fillet (see page 8)

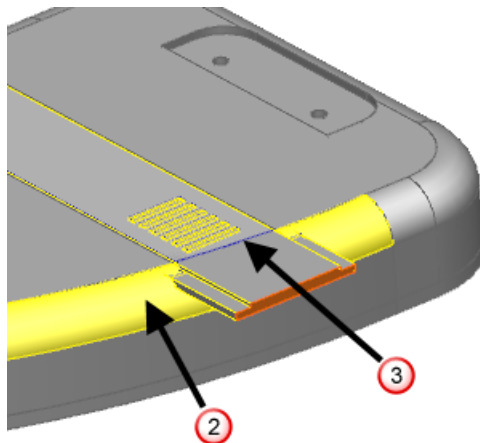
### Replace Face

You can now pick a face from a different solid to use as the replacement item; previously, you could only pick a face from the same solid.

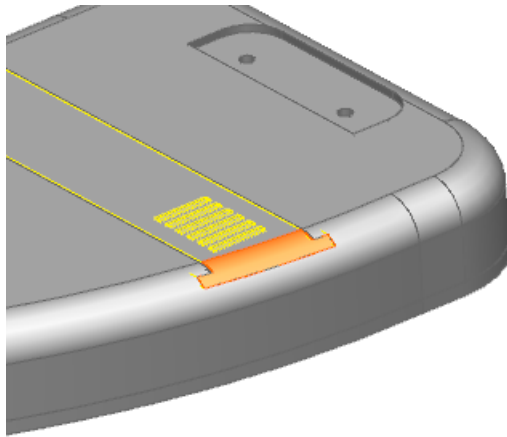
- 1 Click the solid to display the **Solid Edit** toolbar.
- 2 Click  to display the **Replace Faces** dialog.
- 3 Select the original face ①. This is the face that will be replaced.



- 4 Select the replacement face ②. The preview of the updated face is displayed ③.

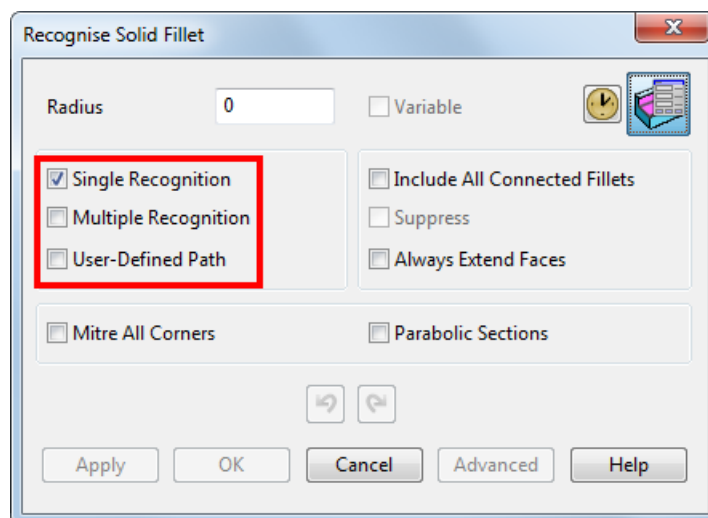


- 5 Click **Apply** to display the updated model.



## Feature recognition - Fillet

The following changes have been made to the **Recognise Solid Fillet** dialog in PowerSHAPE 2013 R2:




- **Interactive Recognition** has been renamed **Single Recognition**.
- **Automatic Recognition** has been replaced by **Multiple Recognition**. This lets you recognise all fillets of the same radius.  
When you select **Multiple Recognition**, **Include All Connected Fillets** changes to **Include Concave And Convex**. If this option is then selected, both convex and concave fillets will be included.
- **User-Defined Path** has replaced the **Manual Fix-Up Mode**. Select this option if you want to select the fillet path when fillets branch.

---

## Cloud modelling


The following have been added to cloud modelling functionality:

- Use **Edit > Convert > Selection to Cloud** to convert:
  - all selected lines, arcs, curves and composite curves to a single point cloud. The cloud has no normal or colour information.
  - all points in the selection to a single cloud. This cloud is separate from the cloud that is created by converting lines, arcs, curves and composite curves.
  - all meshes in the selection to individual point clouds; a cloud retains the normal and colour information from the mesh.
- Use **Restore previous selection**  (*Views toolbar*) to restore the previous sub-selection made on a cloud.

### Cloud toolbar

The following changes have been made to the buttons on the **Cloud** toolbar:





- **Box selection** of points in a cloud is now only enabled when the





**Box selection** button is selected .

You can drag the cloud when **Box selection** is deselected



- The following changes have been made to the buttons on the **Cloud** toolbar:

Old	Function	New
	<b>Generate mesh</b> from the selected cloud	
	<b>Extract selection</b> has been added to the toolbar. Clicking this button extracts the currently selected points from the current cloud and creates a new cloud from those points.	
	<b>Combine clouds</b> has been added to the toolbar. Clicking this button is the same as selecting <b>Object &gt; Cloud &gt; Combine</b> to combine the selected clouds.	

	<p><b>Colour clouds uniquely</b> has been added to the toolbar. Use this button to apply a different colour to all the selected clouds.</p> <p>The colour that is applied to a particular cloud is linked to the cloud number; whenever <b>Colour clouds uniquely</b> is selected, the clouds will always use the same colour.</p>	
	<p><b>Reset to default colour</b> has been added to the toolbar. This applies the default colour (red) to every selected cloud.</p>	
	<p><b>Delete selected points</b> has been updated and re-positioned on the toolbar.</p>	


## Mesh modelling

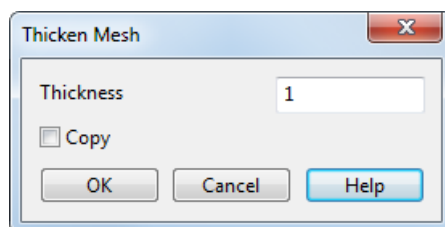
Mesh modelling has been updated in PowerSHAPE 2013 R2.

Mesh modelling toolbar

**Thicken mesh** has been added to the **Mesh modelling** toolbar.



- 1 Click  to display the **Thicken Mesh** dialog.



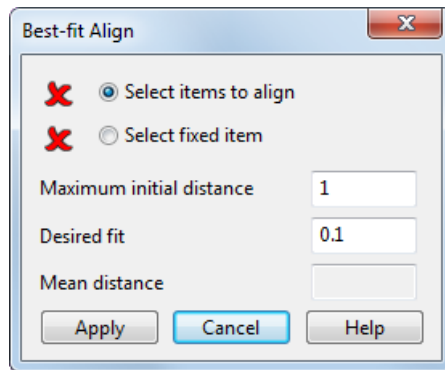
- 2 Enter a **Thickness** to set the thickness required for the mesh.
- 3 Select **Copy** to thicken a copy of the mesh and keep the original unchanged.
- 4 Click **OK** to thicken the mesh and close the dialog.


### Best-fit align dialog

Changes have been made to the **Best-fit align** dialog:

- Labels have updated.

- Reference Item drop-down list has been removed.



You can now select multiple items before clicking  (*General Edit toolbar*).

- One item is selected automatically to be the fixed item; you can change this selection if required.
- The items to be aligned are all highlighted in the primary colour; the fixed item is highlighted in the secondary colour (purple in the 2013 R2 default colour scheme)

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## Wireframe modelling

The following functionality has been added in PowerSHAPE 2013 R2:

- Create an outline curve. (see page 12)
- Create a custom chamfer (see page 13).
- There are two new variables for finding the apparent and elevation angles of a line (these are the same values as shown on the line editing form).

Each of these commands returns a REAL value (the angle in current units - degrees or radians).

`LINE [xxx] .APPARENT`

returns the apparent angle of the line using the current working plane of the currently active workspace

`LINE [xxx] .ELEVATION`

returns the angle of elevation that the line makes using the current principal plane of the currently active workspace.

You can optionally specify which principal plane to use:

`LINE [xxx] .APPARENT.XY`

`LINE [xxx] .APPARENT.YZ`

`LINE [xxx] .APPARENT.ZX`

`LINE [xxx] .ELEVATION.XY`

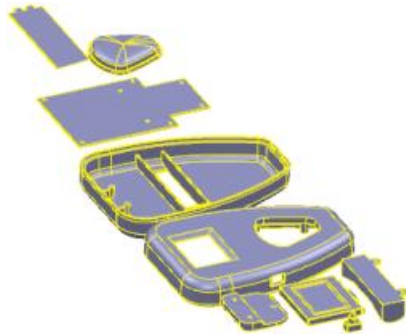
`LINE [xxx] .ELEVATION.YZ`


`LINE [xxx] .ELEVATION.ZX`

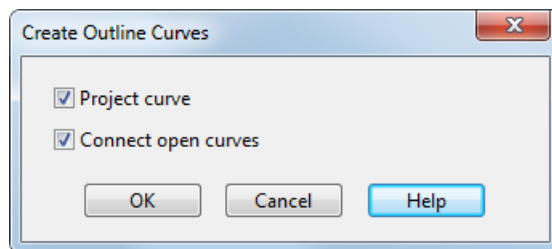
## Creating an outline curve

Create a composite curve outline around the selected surfaces, solids and components.

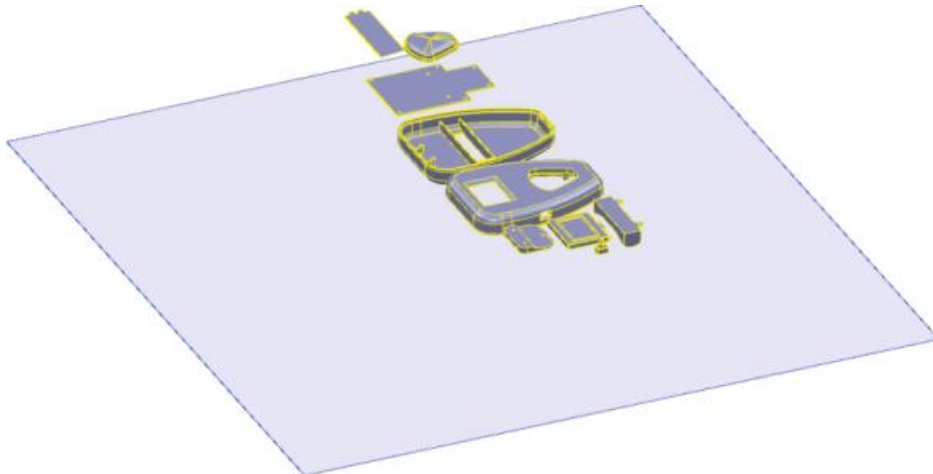
- 1 Select the objects that will be used to create the outline curve.



- 2 Click  (*Curve toolbar*) to display the **Create Outline Curves** dialog.

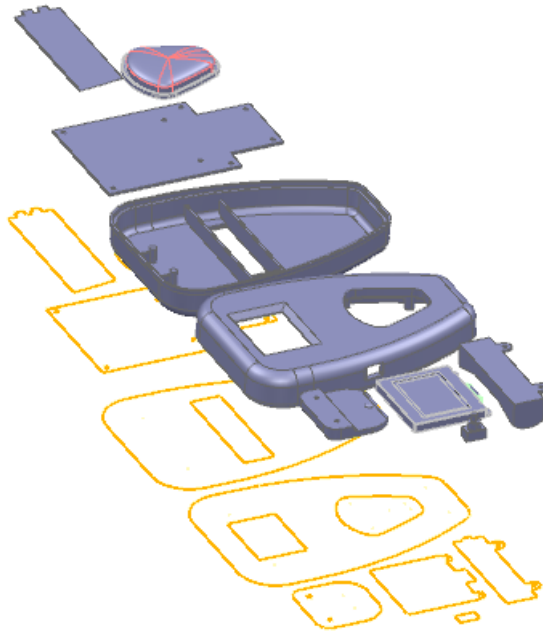


The projection plane is displayed aligned normal to the principal plane.



- 3 Use  to change the principal plane.

- 4 Click **OK** to create outline curves for the selected objects.





If **Project curve** is selected (default), the outline curves will be drawn on the projection plane, as shown.

If **Project curve** is deselected, the outline curves are created in the same position as the objects. Blank the objects to see the outline curves.

## Creating a custom chamfer

Create a custom chamfer between two connected lines:

- 1 Draw two connected lines.
- 2 Click .
- 3 Enter values in the command line. For example, the following specifies a line length of 50 and an angle of 30:  

```
l 50  
a 30
```
- 4  the two lines to create the chamfer.

---

## Surface modelling

The following have been added to surface modelling in PowerSHAPE 2013 R2:

- There are new variables to calculate the surface tangent vector at any (T, U) value. Previously it could only be calculated at lateral or longitudinal points.

```
surface[entity_name].evaluate(T; U).udirb
```

```
surface[entity_name].evaluate(T; U).udira
```

Tangent vector U, direction before/after (around lateral) of the specified (T,U) point on the surface

```
surface[entity_name].evaluate(T; U).tdirb
```

```
surface[entity_name].evaluate(T; U).tdira
```

Tangent vector T, direction before/after (along longitudinal) of the specified (T,U) point on the surface

- The following new variables have been included in PowerSHAPE 2013 R2:

```
surface[entity_name].evaluate(T; U)
```

Coordinates of the specified (T,U) point on the surface

```
surface[entity_name].evaluate(T; U).position
```

Coordinates of the specified (T,U) point on the surface

```
surface[entity_name].evaluate(T; U).normal
```

Normal direction of the specified (T,U) point on the surface

```
surface[entity_name].evaluate(T; U).draft_angle
```

Draft angle of the surface at specified (T,U) point

```
surface[entity_name].evaluate(T; U).curvature.min
```

Minimum curvature of the surface at specified (T,U) point

```
surface[entity_name].evaluate(T; U).curvature.max
```

Maximum curvature of the surface at specified (T,U) point

- A new arc tangent variable has been added:

```
atan2(arg1;arg2)
```

This is useful for finding the azimuth and elevation for a unit vector [i; j; k]

```
let azimuth = atan2(j; i)
```

```
let elevation = asin(k)
```

---

## Solid modelling

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

- Primitive solids (see page 15)
- Solid Doctor (see page 16)
- Solid tree (see page 17)
- Solid features (see page 27)

- Solid macro commands (see page 34)

### Other changes to solid modelling

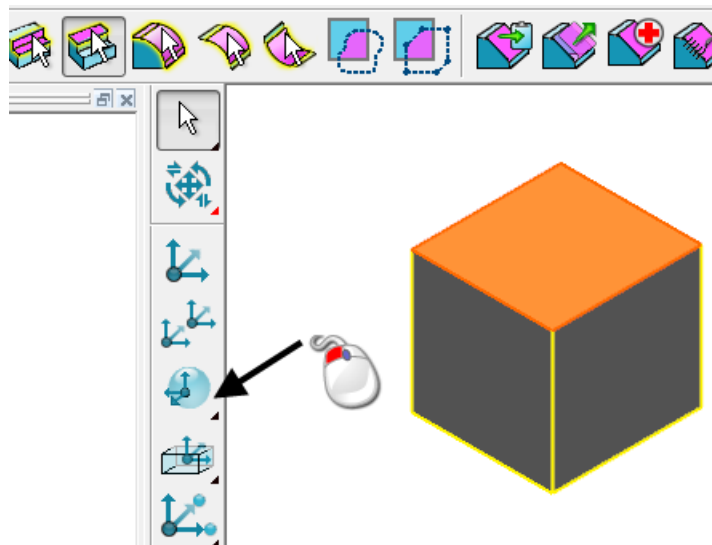
- **Edit Surface** has been removed from the solid popup menu. Use Direct Modelling functionality to edit and replace faces of a solid.
- The options on the **Tools > Options > Object > Solids** dialog (see page 25) have been updated in PowerSHAPE 2013 R2.

## Primitive solids

The following enhancements have been made to solid modelling with primitives:

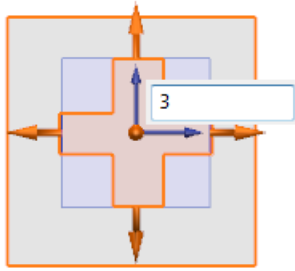
- Select one or more faces on a selected primitive solid by clicking one of the face selection buttons (*Solid Edit toolbar*) and selecting the required faces.
- You can create a workplane in the centre of a face of a selected solid.

Click **Create a single workplane aligned to geometry** (*Workplane toolbar*) to create a workplane in the centre of the selected face.



- When the **Solid Core from Selection** dialog is displayed, you can change the face selection using the **Solid Edit** toolbar selection buttons and **Restore selection** (*Views toolbar*).

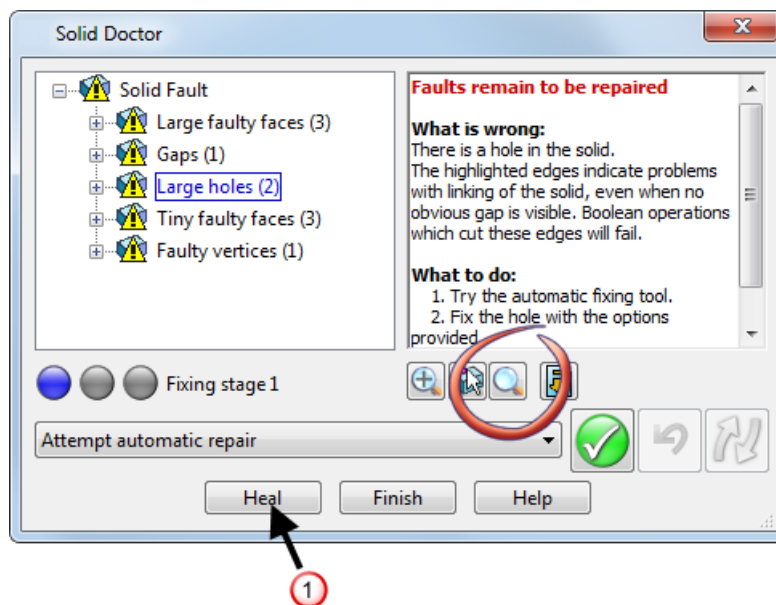
- Enter an exact value in the box that is displayed when you have dragged instrumentation when creating a **Solid core from selection**.





*If you don't wish to enter a value, you can ignore the box; it will disappear when you continue with your work.*

## Solid Doctor

The following changes have been made to the **Solid Doctor**:



- Click  to suspend zooming and rotating when you select a fault on the fault list. The button changes to  to indicate that the view is not updated when new faults are selected.
- Click **Heal** 1 on the **Solid Doctor** dialog to display the **Solid Doctor Healing** dialog (see page 17).



*Large vertex faults are now only displayed on the fault page of the Solid Doctor if they are very large, involve three or more surfaces and lie on a faulty surface.*

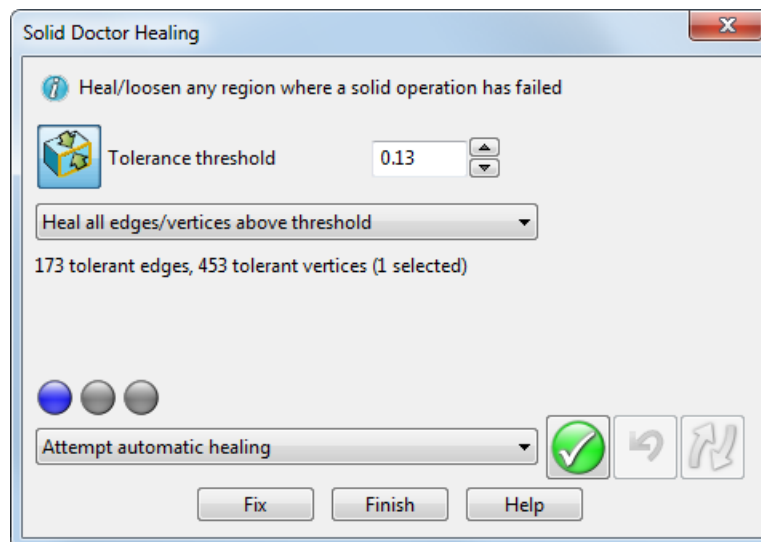
## Solid Doctor Healing dialog



Use this dialog to heal or relax a region where a solid operation has failed.


Edge tolerances can be healed (*tightened*) or relaxed (*loosened*). If a solid operation is failing, try loosening the edge tolerances in that area.

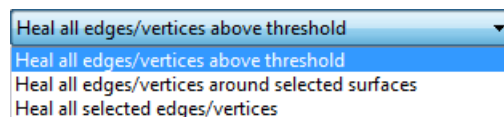



*Healing and relaxing is optional. Normally it is only necessary to heal or relax locally if a solid operation fails and the solid has no faults.*

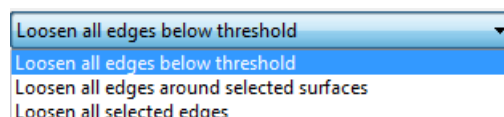


Use  and  to toggle the threshold that you are using.

When  is displayed, select one of the options from the drop-down list to heal a region.



When  is displayed, select one of the options from the drop-down list to loosen a region.



## Solid tree

Changes and additions have been made to several areas of solid tree functionality:

Setting a Rewind Position (see page 18)

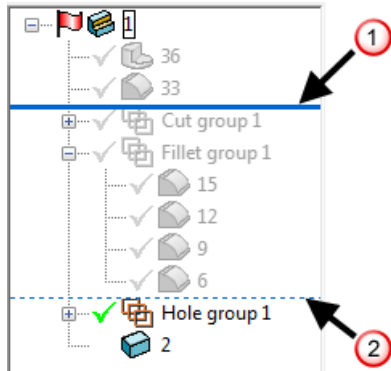
Popup menus (see page 22)

Transforms - Dissolve and Merge (see page 24)

Options (see page 25)

## Setting a Rewind Position

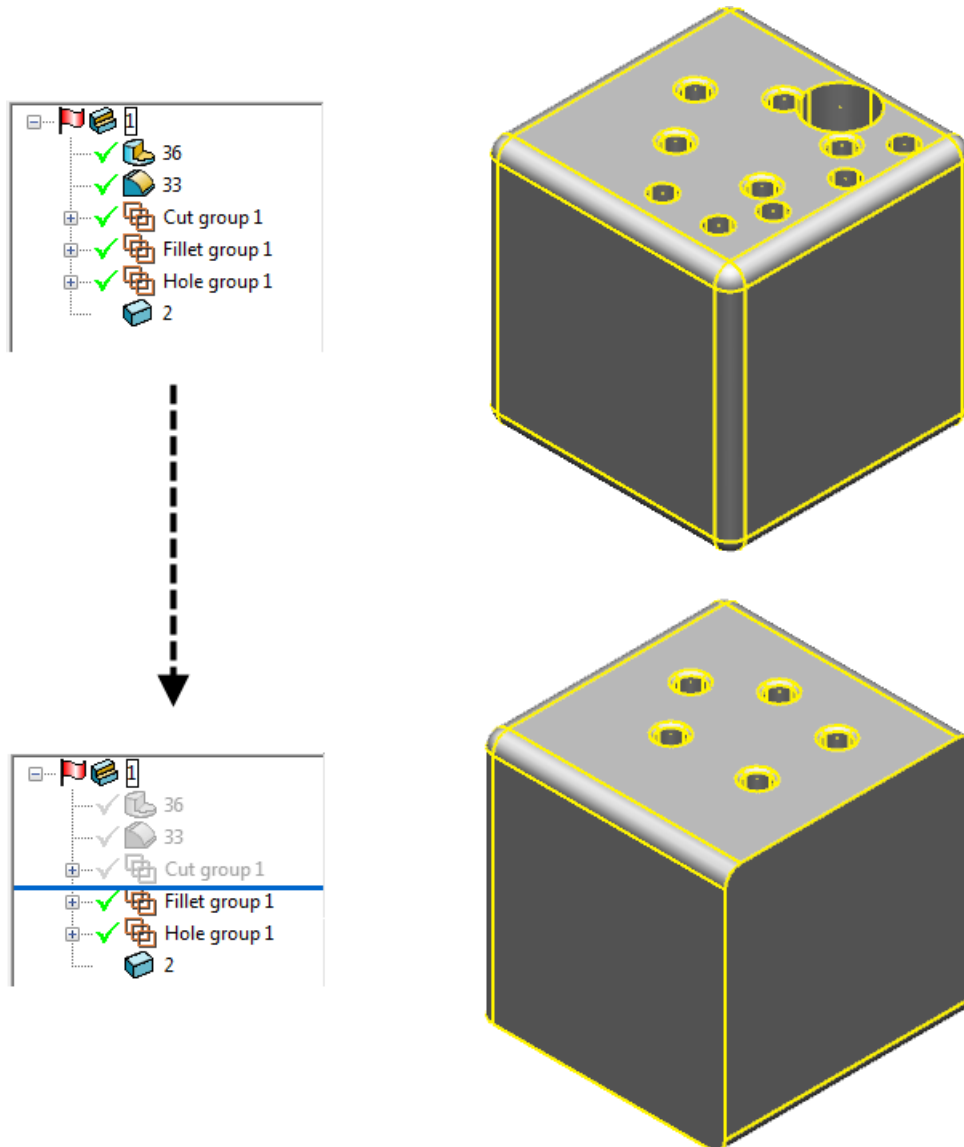
You can now set a **Rewind Position** in the solid tree.



The rewind position appears as a horizontal bar on the solid tree **1**.



This bar can be dragged up and down the solid tree using **2**. A dashed line **2** is displayed to show the potential new position of the bar during the drag operation.



The **Rewind Position** lets you:

- create a new feature anywhere in the tree (see page 20). If you create a new feature, it will appear at the rewind position. This means you do not have to create it at the top of the tree and then re-order it to where you want it.
- preserve features that would be automatically removed (for example, Direct Modelling operations). Only features that are below the rewind position are replaced. They will be replaced by a single base feature, without history.

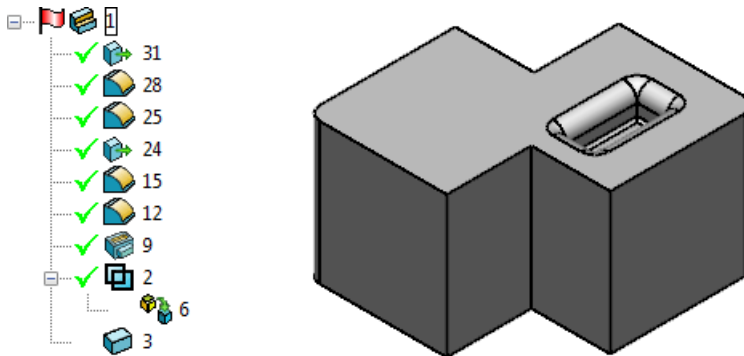
- export solids in an earlier state; for example, prior to a part revision.


Features that are rewound appear greyed-out in the solid tree. They are unavailable for selecting, or editing.

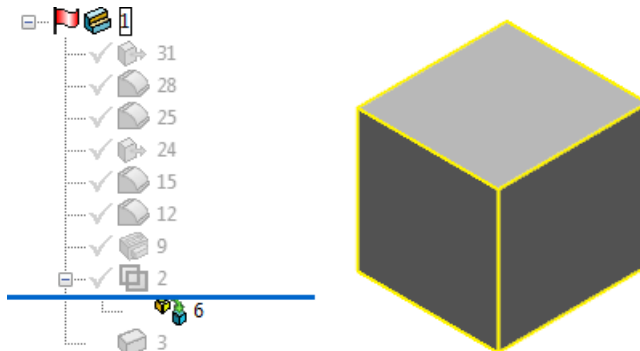
### ***Creating a feature at a specified place in the tree***


Use **Rewind Position** to create a new feature anywhere in the tree.

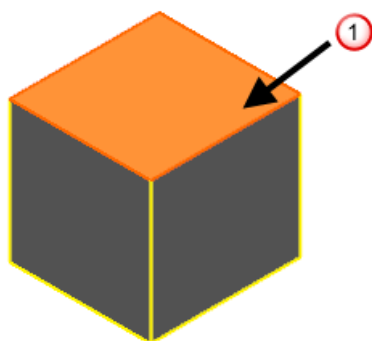
This example reduces the height of the general solid and creates a hole feature on the general solid.



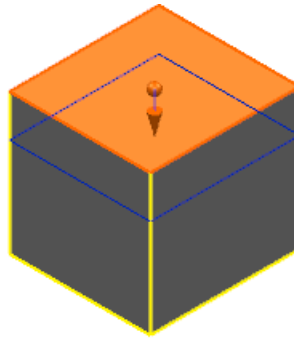
- 1 Right-click 
- 2 Click **Rewind to here** to suppress the features of the solid and leave the general solid displayed.





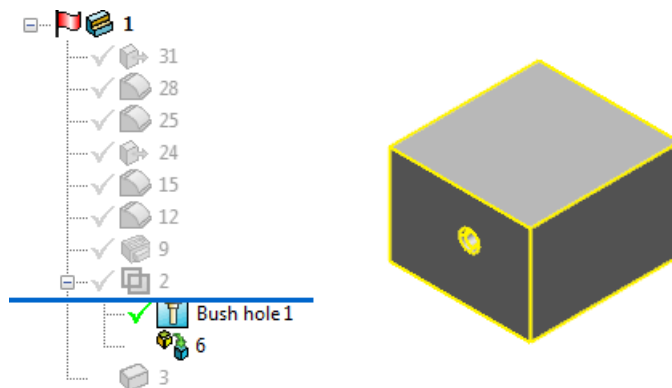
- 3 Click  (*Solid edit toolbar*) and select the individual face ①.



- 4 Move the face by entering **-15** in the Z value on the **Move** toolbar.



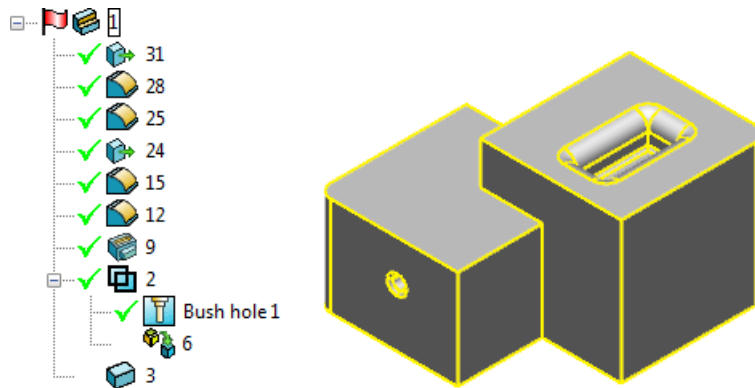
- 5 Click **Apply** .
- 6 Click **Dismiss** . The solid still has a history tree.
- 7 Add a hole feature to the solid. The hole is added to the tree in the sub-branch above the general solid feature.



- 8 Move the cursor over the **Rewind** bar so that the cursor changes to a double-headed arrow.

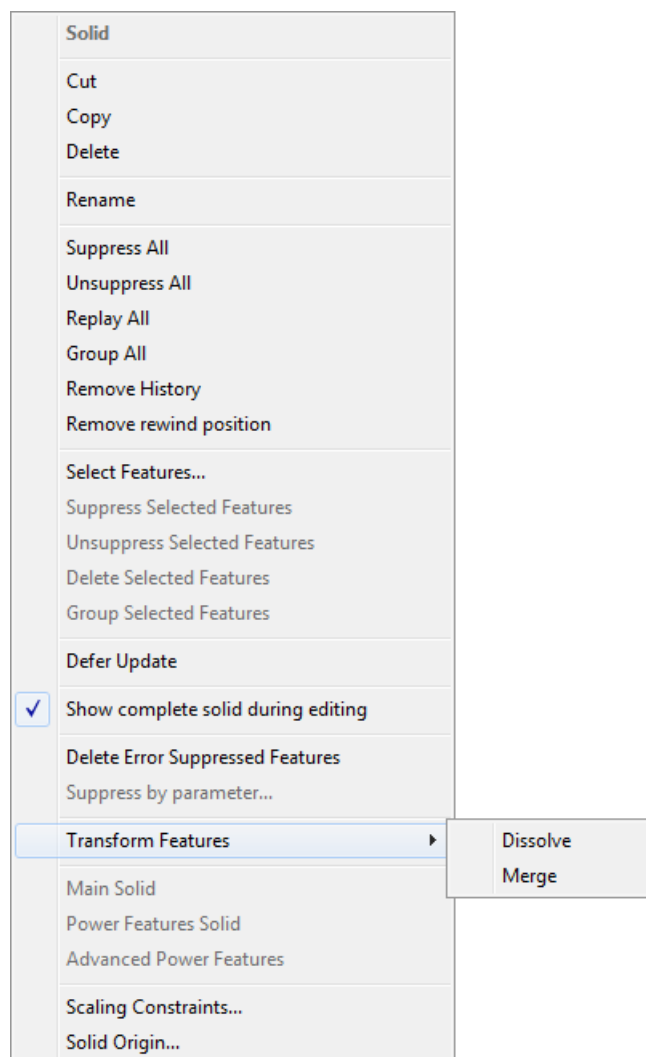


- 9 Right-click and select **Remove rewind position** from the popup menu.



## Popup menus

The options on the **Solid** tree popup menu have been reordered.



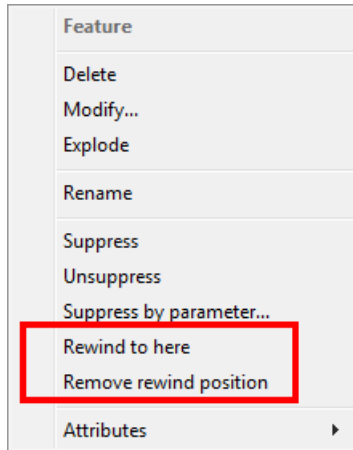
## Rewind functionality

The solid tree popup menus have been updated to reflect rewind functionality:

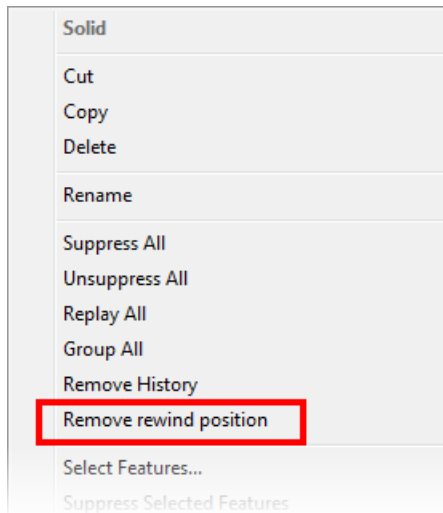
- The **Feature** menu now includes:

**Rewind to here** sets the rewind position immediately above the feature you clicked.

**Remove rewind position** removes the rewind position. All the greyed-out rewound features are applied to the solid.

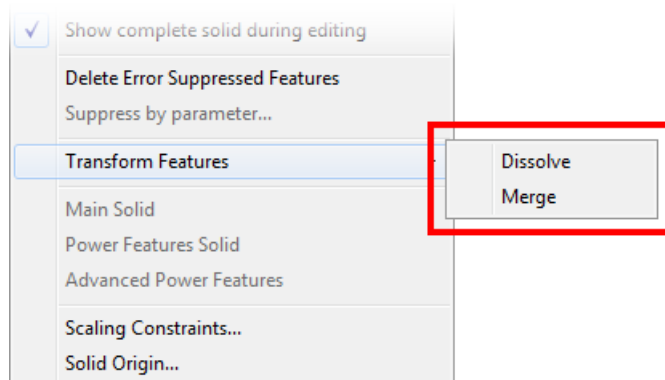


- The **Solid** menu now includes **Remove rewind position**.

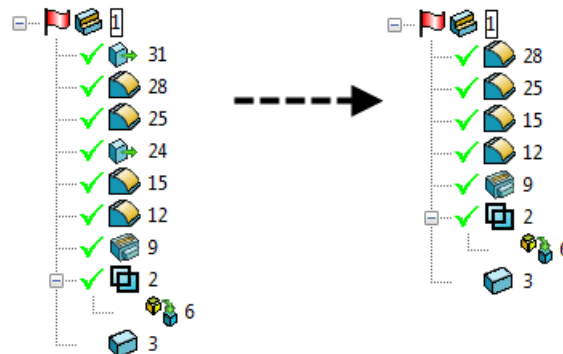


## Transforms - Dissolve and Merge

Use the **Dissolve** and **Merge** options on the **Transform** section of the solid tree popup menu to rationalise the solid tree:

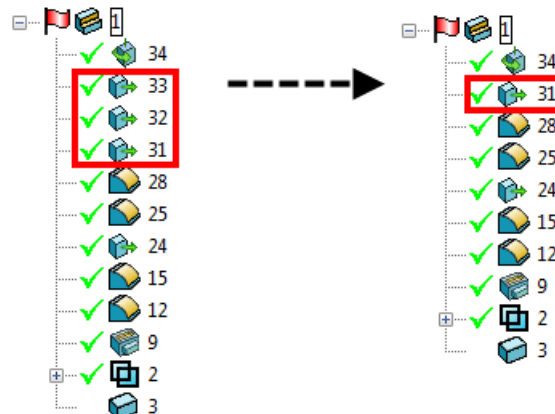


- Select **Dissolve** to remove the transform features from the solid tree. An **Information** dialog is displayed to tell you how many transforms have been removed.



The solid is left in its transformed position; this requires the solid tree to be automatically rebuilt.

- Select **Merge** to merge together any adjacent transform features of the same type. For example, if you move a solid three times, selecting **Merge** will merge the three transform features into one.





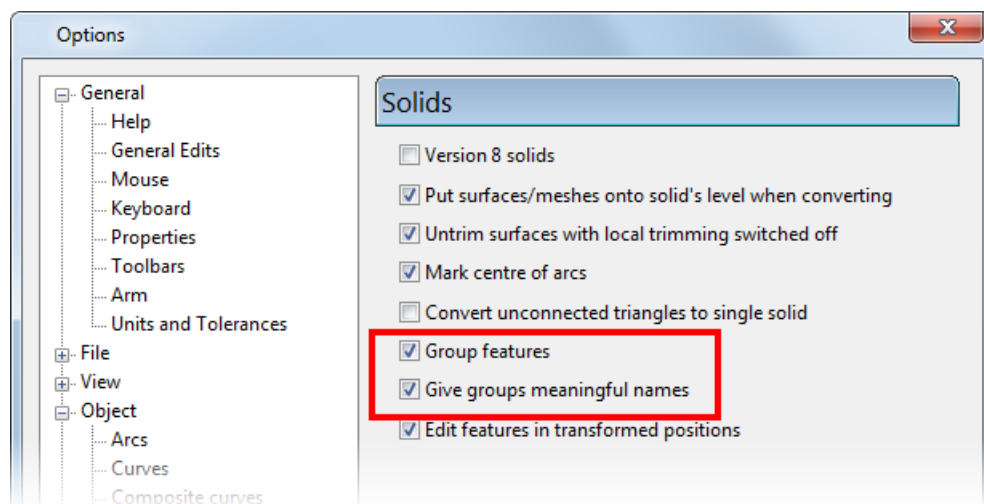
*If the Automatically merge solid transform features option is set on the Tools > Options > General > General Edits dialog, the transform features will be automatically merged as they are created.*

## Options dialog

The options dialog has been updated to include new options for grouping features and editing features.

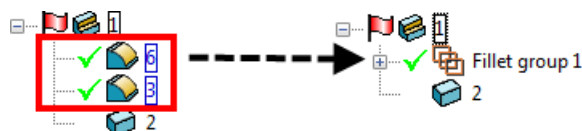
### Grouping features

Select the new **Group features** option on the **Tools > Options > Object > Solids** dialog to create groups from all sequentially created new features of the same type. This option is selected by default.

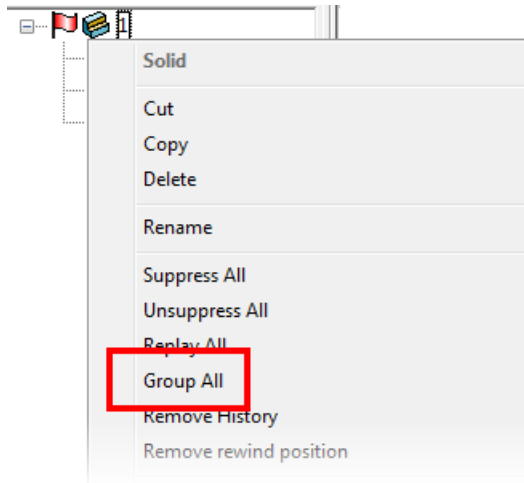


When the **Group features** option is selected, a group of features will be created when you:

- sequentially create five or more features of the same type.
- select individual features in the tree and select **Group All** from the **Solid** tree popup menu.

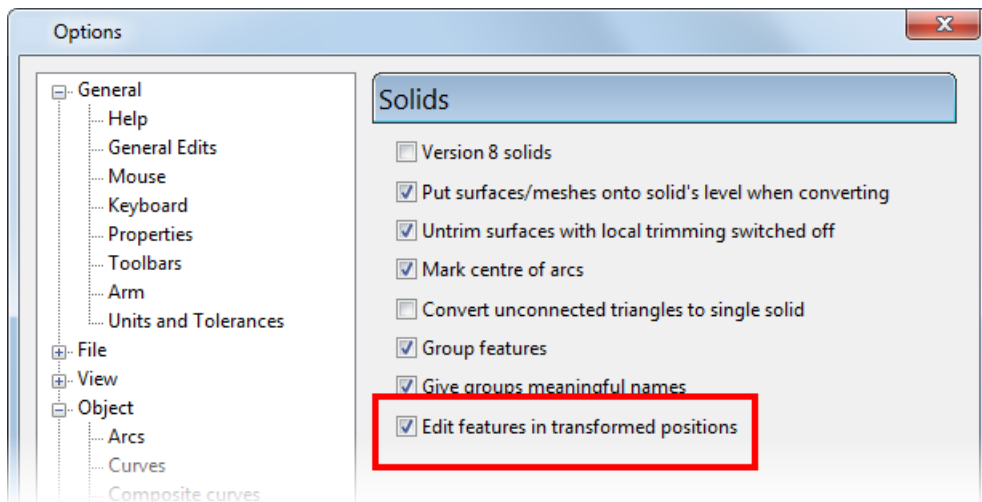


- select individual features on the model and select **Group All** from the **Solid** tree popup menu.

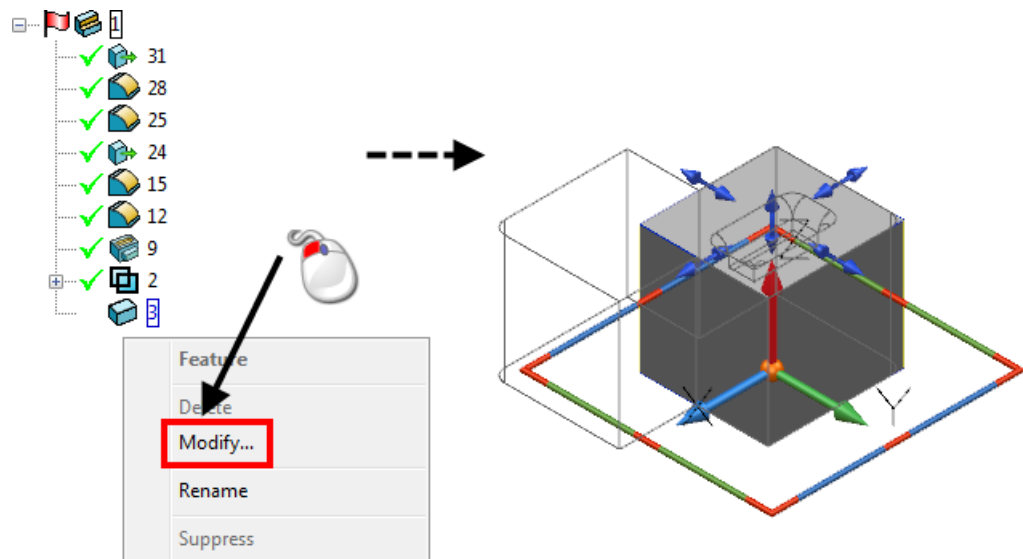


### Editing features

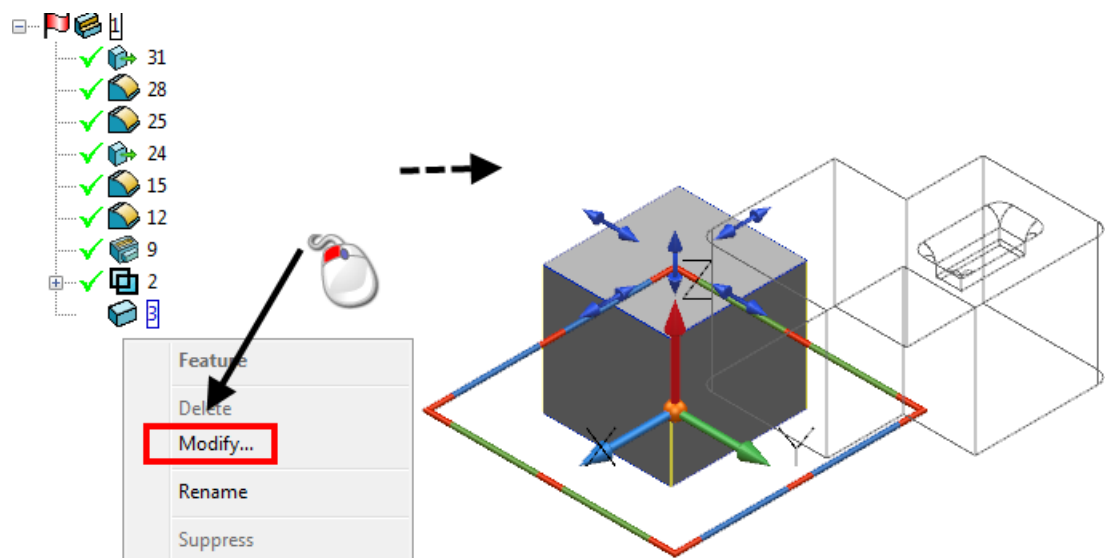
Use the new **Edit features in transformed positions** option on the **Tools > Options > Object > Solids** dialog to control how solid features appear while they are being edited. This option is selected by default.



- When **Edit features in transformed positions** is selected, the feature is edited in its transformed position. So, if a Boolean block has been moved, the block would be edited in the current position.



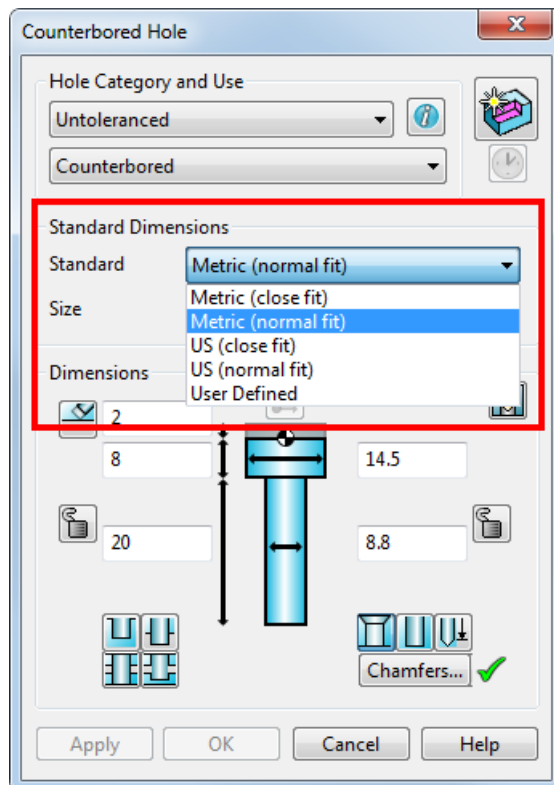
- When **Edit features in transformed positions** is deselected, the feature is edited in its position before the transform was applied. So, if a Boolean block has been moved, the block would be edited in its original position, before the block was moved.



## Solid features

The following changes have been made to solid feature functionality:

- A **Standard Dimensions** section has been added to the **Counterbored** option of the **Hole** dialog. This lets you set the counterbored hole dimensions to match standards that fit screw sizes easily.



- Splitting a solid has been updated (see page 28).
- You can now create non-equal distance solid chamfers (see page 32).

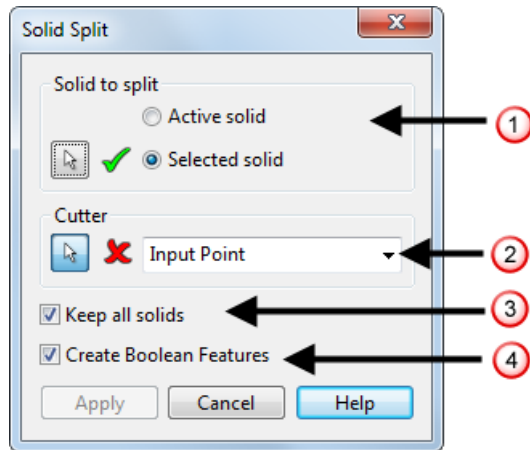
## Splitting a solid

Splitting a solid functionality has been updated in PowerSHAPE 2013 R2.

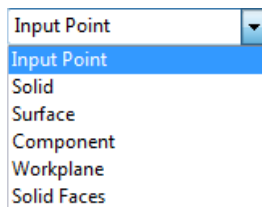


*If you select **Solid split** without having a suitable cutter item selected or without having an active solid to work on, the new **Solid Split** dialog is displayed.*


*If you have pre-selected a cutter, the dialog will not be displayed and the solid will be split according to default settings.*



- 1 Select the solid to be split ①.
- 2 Select the cutter type from the drop-down list ②.



*You can split a solid using multiple faces (see page 29).*

- 3 Click  and select the cutter.
- 4 Select **Keep all solids** to retain all the new solids created by the split operation ③.
- 5 Select **Create Boolean features** to create Boolean features ④.
- 6 Click **Apply** to split the solid.

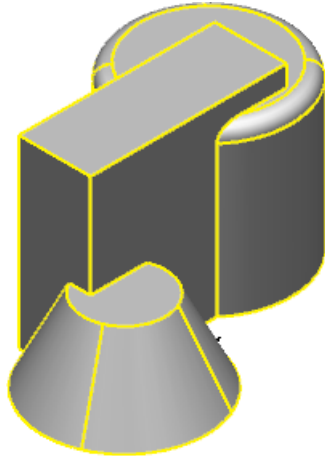
### ***Splitting a solid using multiple faces***

You can now split a solid using multiple selected faces.




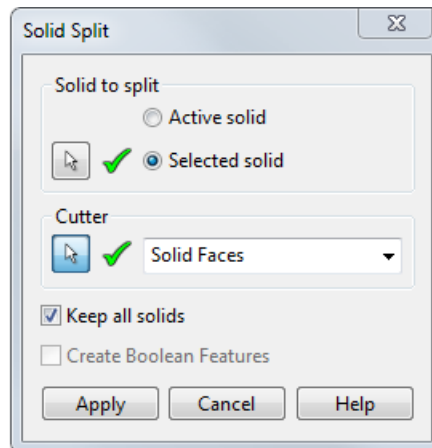
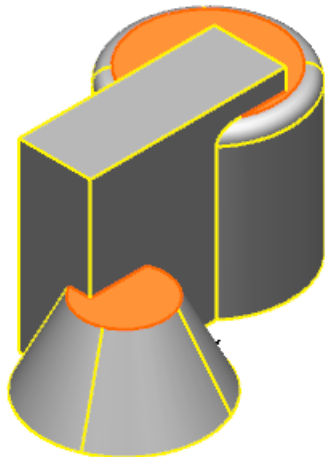
*This example does not use a pre-selected cutter.*

- 1 Select the solid to be split.

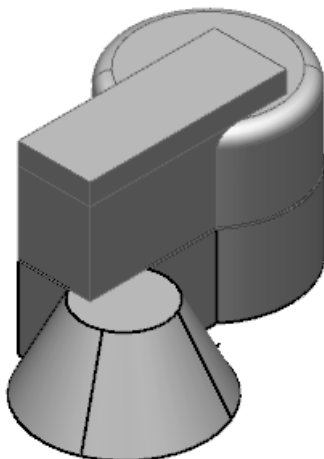


- 2 Select **Solid Faces** from the **Cutter** drop-down list.

- 3  to select the faces to be used as cutters.

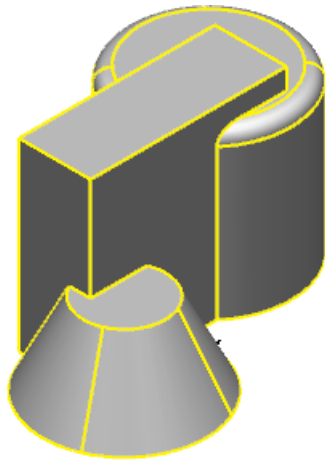


- 4 Click **Apply** to split the solid.

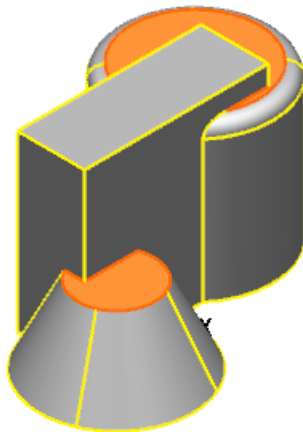



## ***Splitting a solid using a pre-selected cutter***

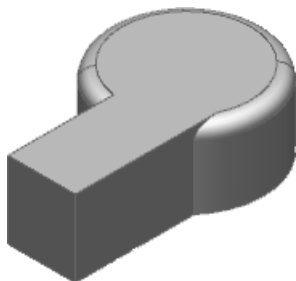
You can now split a solid using multiple selected faces as the cutter. When multiple faces are selected, the single solid is split and no extra solids are created



- 1  to select the faces to be used as cutters.

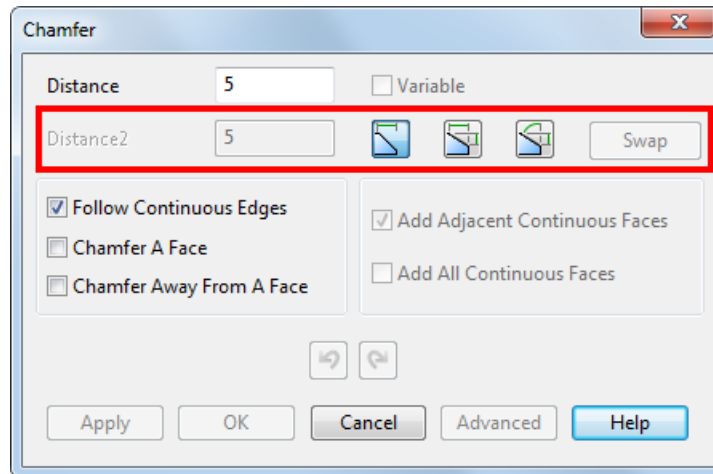


- 2 Click  (*Solid Feature toolbar*) to split the solid.







## Chamfers


You can now create non-equal distance solid chamfers, that have either two different distances, or a distance and a user-defined angle.

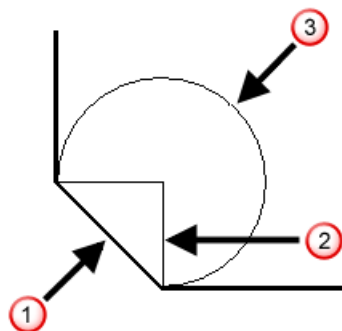


Use the following buttons to select the way the chamfer will be defined:

-  Create a chamfer using a single distance (see page 32).
-  Create a chamfer using two distances (see page 33).
-  Create a chamfer using a distance and an angle (see page 33).
-  Swap the effect of unequal distances.

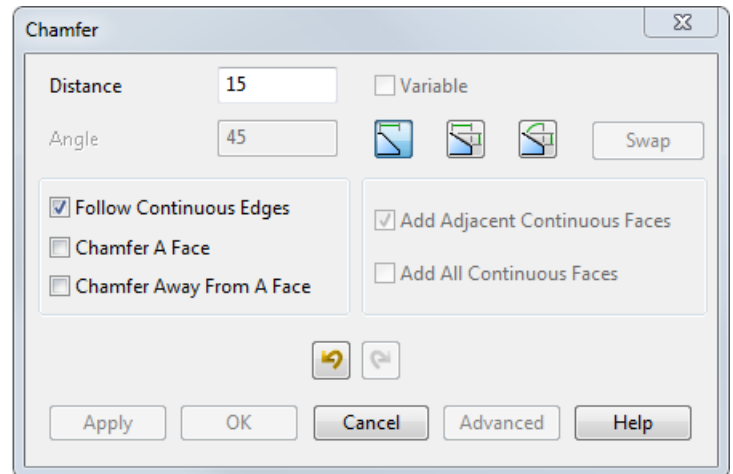
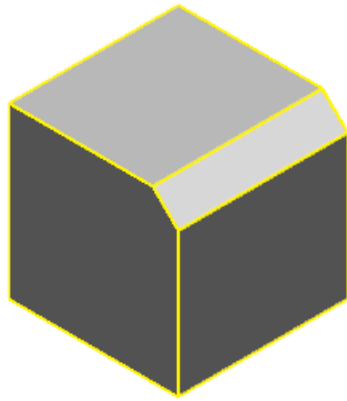
### *Creating a chamfer using a single distance*

- 1 Click .
- 2 Enter the **Distance** to be used to create the chamfer as shown below.




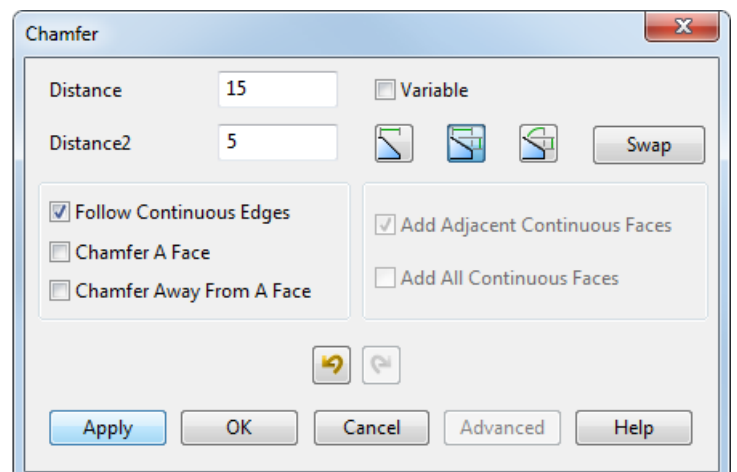
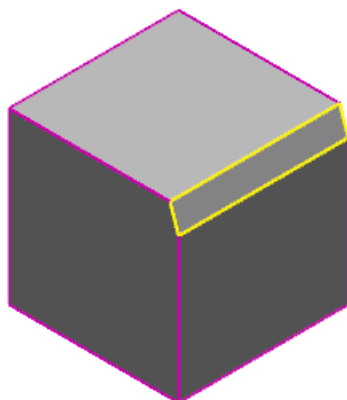
- ① Chamfer.
- ② Distance.
- ③ Arc with radius equal to distance.

- 3 Select an edge of a solid.
- 4 Click **Apply** to create a chamfer with a single distance and a radius of 45 degrees. The chamfer on the following model was created by entering a **Distance** of **15**.



### Creating a chamfer using two distances

- 1 Click 
- 2 Enter a value for **Distance**.
- 3 Enter a value for **Distance 2**.
- 4 Select an edge of a solid.
- 5 Click **Apply** to create a chamfer with two distances. The chamfer on the following model was created by entering a **Distance** of **15** and **Distance 2** of **5**.

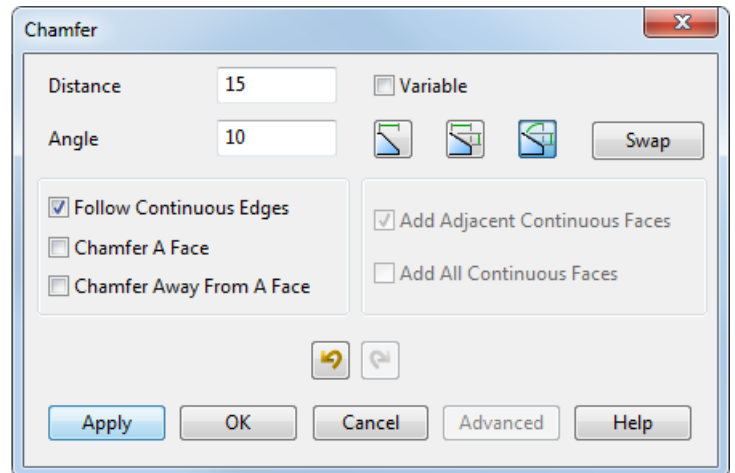
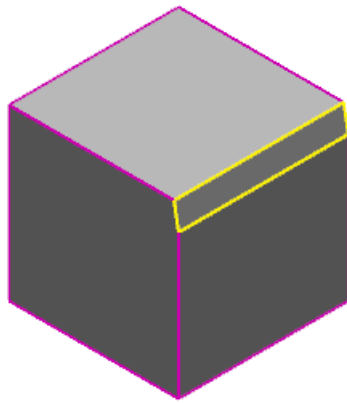


Click **Swap**  to swap the effect of unequal distances.

### Creating a chamfer using a distance and an angle

- 1 Click 

- 2 Enter a value for **Distance**.
- 3 Enter an **Angle**
- 4 Select an edge of a solid.
- 5 Click **Apply** to create a chamfer with a distance and an angle. The chamfer on the following model was created by entering a **Distance of 15** and **Angle of 10**.



Click **Swap**  to swap the effect of unequal distances.

## Solid macro commands

When in face selection mode, you can use commands to pick the faces of a selected solid.

- Use the following commands to replace the currently selected faces with named faces:

```
PICK FACE NAME <face_name>
PICK FACE <face_name>
PICK FACE REPLACE NAME <face_name>
PICK FACE NAME <face_name>
```



*This is the same as using the mouse to select the faces.*

- Use the following commands to add the named face to the current selection:

```
PICK FACE ADD NAME <face_name>
PICK FACE ADD <face_name>
```



*This is the same as holding down the SHIFT key and clicking the left mouse button.*

- Use the following commands toggle the named face into/out of the current selection:

```
PICK FACE TOGGLE NAME <face_name>
PICK FACE TOGGLE <face_name>
```



*This is the same as holding down the CTRL key and clicking the left mouse button.*

<face\_name> can be a word, string, integer or variable.. The following are all valid:

```
PICK FACE fred
PICK FACE 'fred'
PICK FACE 23
STRING face_name = 'fred'
PICK FACE $face_name
```

The commands are also available during the following operations:

- Multiple-face selection modes; if you are in convex face selection mode, several faces will be selected, spreading out from the named face.
- **Solid Draft Face**
- **Solid Replace Face**
- **Solid Divide Face**

---

## Delcam Draft

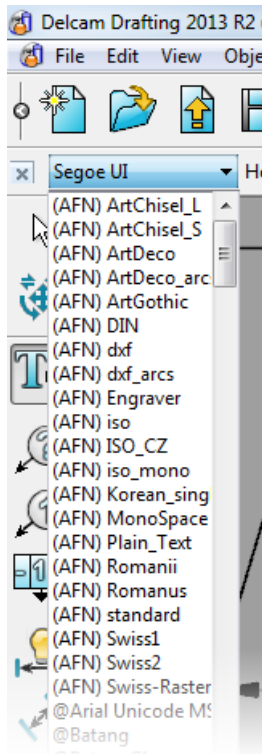
There have been improvements to following area of drafting functionality in PowerSHAPE 2013 R2:

- Using ArtCAM fonts (see page 35).
- Angular dimension on an arc (see page 36).
- Geometric tolerances (see page 37).

### Using ArtCAM fonts

- PowerSHAPE now supports a range of fonts from ArtCAM, including the following wide character stick font:

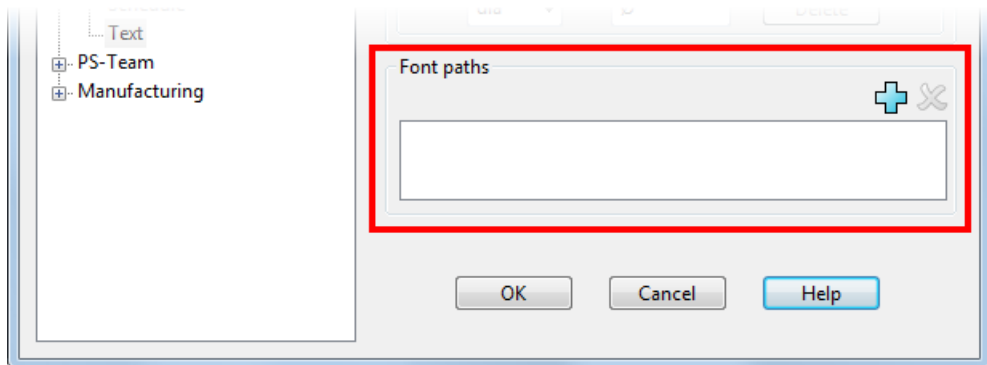
### (AFN) Korean\_single\_line




- If you have created new fonts, using ArtCAM font creator, use the new **Font paths** section on the **Tool > Options > Drafting > Text** dialog to specify the location of font files.





*This is particularly useful if you want to define new stick fonts for engraving.*

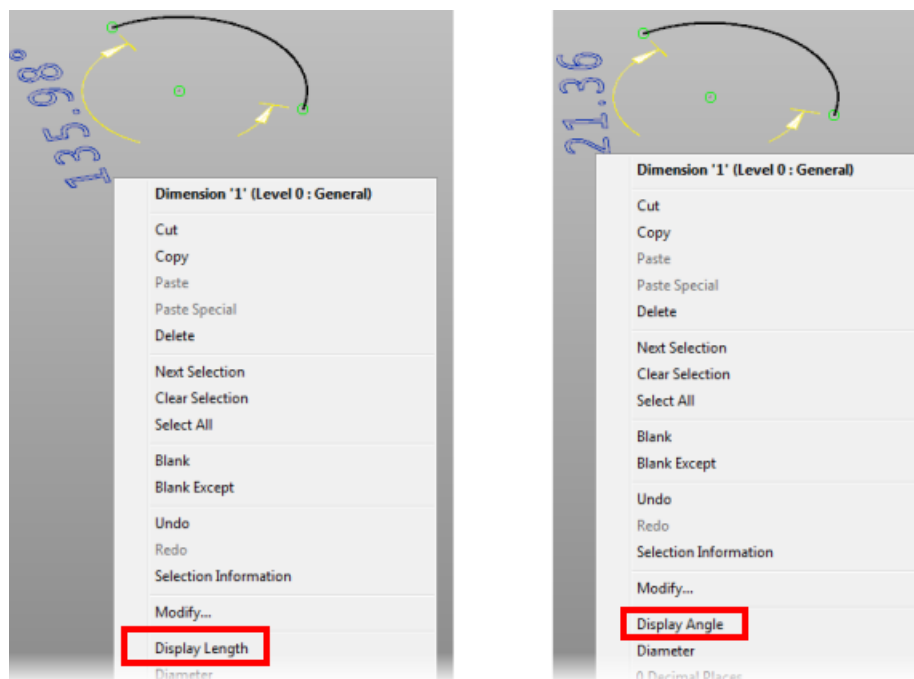


- 1 Click  to display the **Browse for folder** dialog.
- 2 Choose a font directory from the tree.
- 3 Click **OK**.

## Creating an angular dimension on an arc

When you create an angular dimension on an arc, you can toggle between displaying the length and displaying the angle.

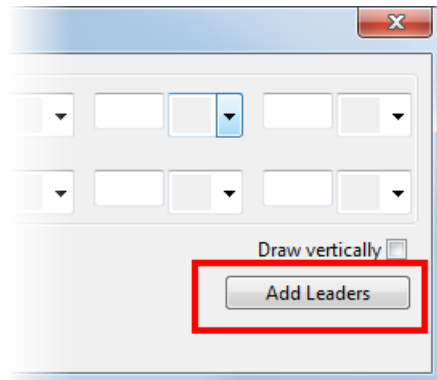
- 1 Create a three-point arc.
- 2 Click .
- 3 Click the centre of the arc and then the two end points of the arc.
- 4 Move the cursor to required position and click  to fix the position of the dimension information.
- 5 Place the cursor on the dimension and click the right mouse button to display the **Dimension** popup menu.
- 6 Use the options **Display Length** and **Display Angle** to toggle between displaying the length and displaying the angle



## Geometric tolerances

- PowerSHAPE now supports the import and export of geometric tolerances using [.ddx](#) file format.
- You can edit a geometric tolerance by double clicking on the tolerance balloon to display the **Geometric Tolerance** dialog.

When you are editing the geometric tolerance, the **Add Extra Leaders** option is replaced by the **Add Leaders** button. Click **Add Leaders** to display the **Add Leaders** dialog.



---

## Delcam Electrode

Changes have been made to several areas of **Electrode Design Wizard**:

- When a collision is detected, you can now twiddle the model and zoom to inspect the offending region before clicking **OK** on the dialog.
- It is now possible to adjust the Z value of the datum of an electrode when the datum face is selected as **Base Bottom**. When the Z value of the datum is changed from the bottom face of the electrode, the Z offset of the base and the height is adjusted so it is just the bottom face of the base that moves by the required value.
- The EROWA library has been fully updated and now contains the complete catalogue of electrode holders.
- Use the Undersize data-sets (see page 39) option to select the default data-set to be used for electrode undersizes.
- Use the changes on the Export Options (see page 40) page of the Electrode Design Wizard to set the options to be used for export of electrode data.
- Edit the [ops\\_export.con](#) file to customise export to OPS Ingersoll format (see page 41).
- Holder definition now uses a new [xml](#) format. This replaces the [.csv](#) format and lets you describe holder definitions in a logical way.

The holder definition files are located in:

.....\powershape132XX\file\electrode

The following xml files are included in PowerSHAPE 2013 R2:

PSEH\_Erowa.xml

PSEH\_Hirschmann.xml

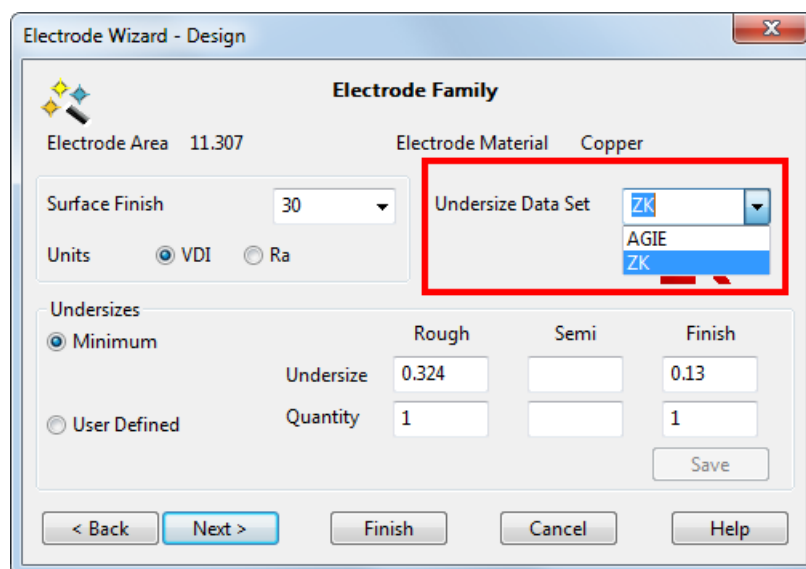
PSEH\_System\_3R.xml

You can define your own electrode holders; these must be in the format that is described in

.....\powershape132XX\file\electrode\holder\_xml\_syntax.html

## Undersize data-sets

- Use the new **Undersize Data Set** drop-down list to select the default data set to be used for electrode undersizes.



The logo of the currently selected option is displayed below the list.

- Use the [electrode\\_undersizes.com](http://electrode_undersizes.com) to control the undersize data sets that are available in the wizard.

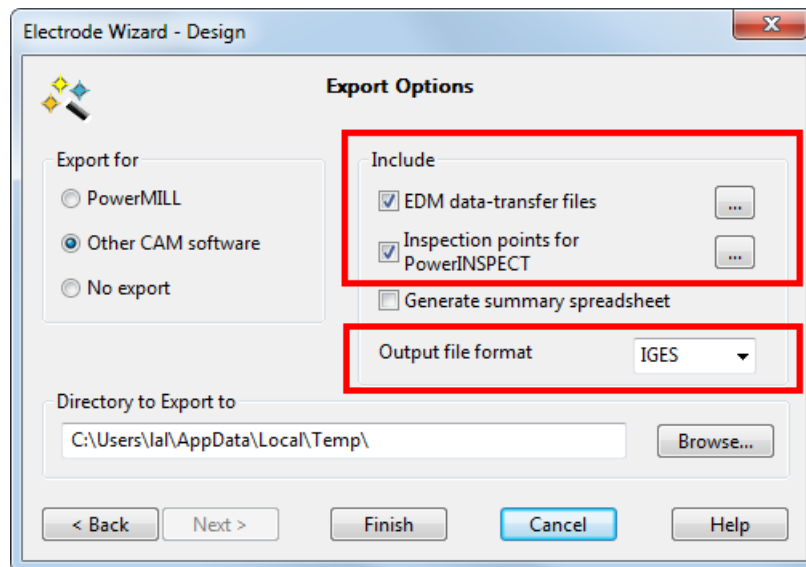
The file is installed in:

PowerSHAPE132XX/file/electrode


You can edit the configuration file to suppress competitors logos and datasets. Full instructions for customising the configuration are included in the file. By default all available datasets are included in the **Undersize Data Set** drop-down list and the logo of the selected data set will be displayed.

## Export Options

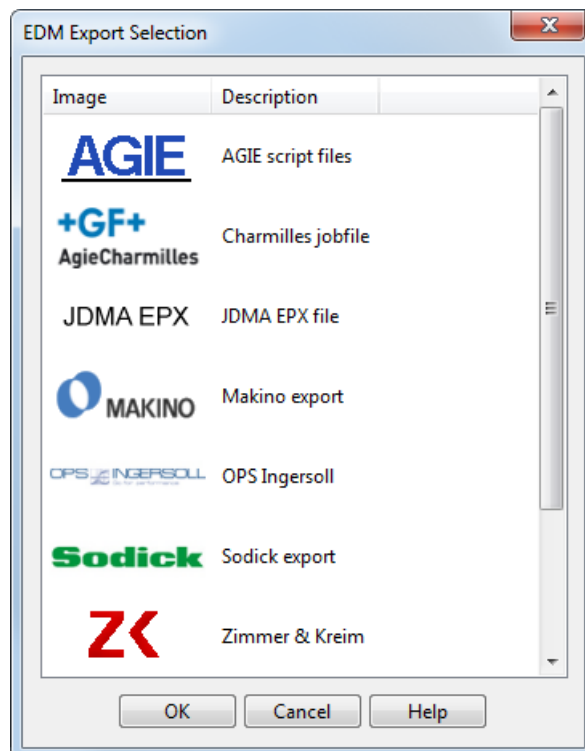
The **Export Options** page has been revamped in PowerSHAPE 2013 R2.




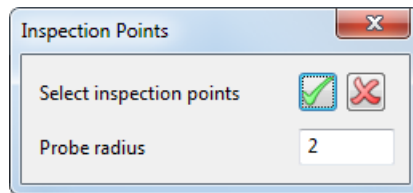
Use the following new sections to specify the export information:

- 1 Select **EDM data-transfer files** to export EDM setup files. This option is selected by default.
- 2 Click  to display the **EDM Export Selection** dialog.

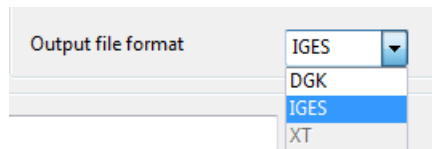
The following formats have been added to PowerSHAPE 2013 R2: **Sodick**, **Mitsubishi**, **OPS-Ingersoll** and **Zimmer & Kreim**.



- 3 Select the required option. As well as the options shown above, the dialog may include:
  - Custom script** — User-defined scripts are included using this option.
  - OPS + MultiProg** — This is the same as **OPS Ingersoll** export but will start **MultiProg** (if installed) after exporting the data.
- 4 Select **Inspection points for PowerINSPECT** to select inspection points. The option is selected by default.
- 5 Click  to display the Inspection Points dialog. Select inspection points and specify the radius of the inspection probe.



- 6 Specify the **Output file format** by selecting an option from the dropdown list.



## Export to OPS Ingersoll format

Use the [ops\\_export.con](#) file to customise the export of electrode data to the **OPS Ingersoll** format. The file is installed in:

[PowerSHAPE132XX/file/electrode](#)

PowerSHAPE will export electrodes using the cavity list notation, which groups the copies of the same electrodes and outputs their different positions as a cavity list.

The configuration file contains one option. By default this is commented out, as follows:

```
#use_cavity_lists: false
```

To export every electrode individually, edit the [.con](#) file to remove the # from the option and save the changes.

---

## Delcam Toolmaker

Changes have been made to several areas of Delcam Toolmaker:

- Mold Base Wizard (see page 42)
- Component Wizard (see page 44)

- Power Feature Trimming (see page 47)
- The following dialogs have been updated to include new graphics:
  - **Cooling Wizard.**
  - **Mold Base Wizard.**

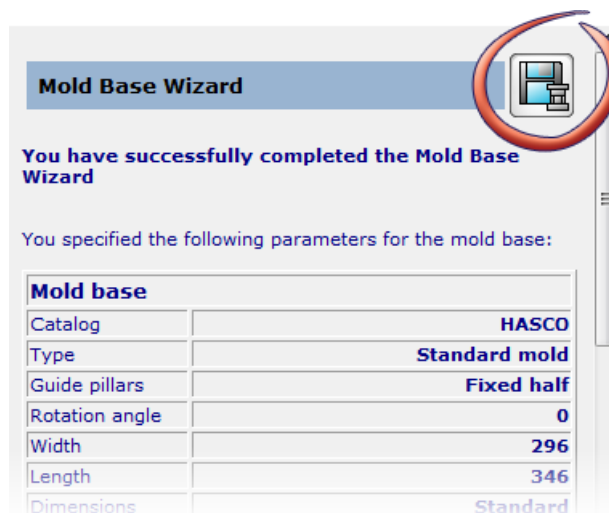
## Mold Base Wizard

**Mold Wizard** has been updated to include:

- Save the current mold parameters as a mold configuration (see page 42). This configuration will include:
  - catalog.
  - mold size.
  - mold type.
  - guide pillar placement.
  - list of plates (including all blank plates that have been added manually).
  - auxiliary components and dimensions.
- Load a saved mold configuration (see page 43). These parameters are applied to the new mold.
- Load a recently used mold configuration (see page 43). These parameters are applied to the new mold.

## Saving a mold configuration

- 1 Click  on the final page of the **Mold Base Wizard**

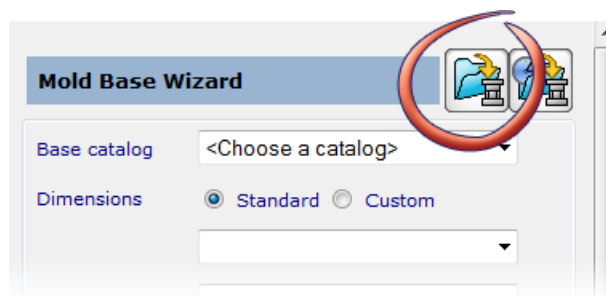


- 2 Enter a **Configuration name** and click **Save**.

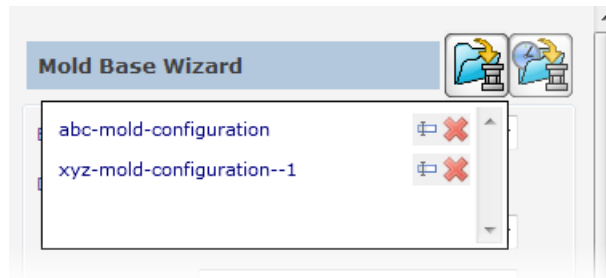




## Loading a saved mold configuration

- 1 Click .



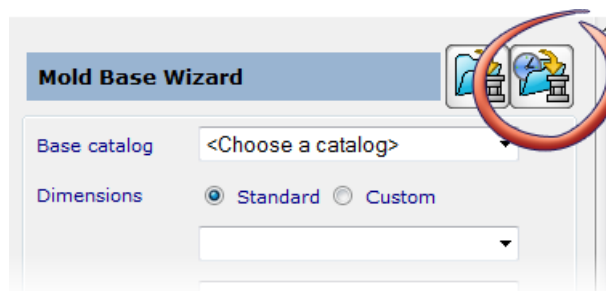
- 2 Select the name of the saved configuration from the list.



- 3 Use the following as required:
  - Click  to rename the mold configuration.
  - Click  to delete the mold configuration.

## Loading a recent configuration

- 1 Click  to load a recent mold configuration.



- 2 Select the required mold configuration from the list



## Component Wizard

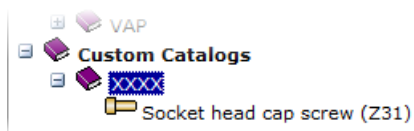
In PowerSHAPE 2012, custom component functionality has been updated; you can now change the following:

- Grouping of directions.
- Order in which dimensions and groups are displayed.

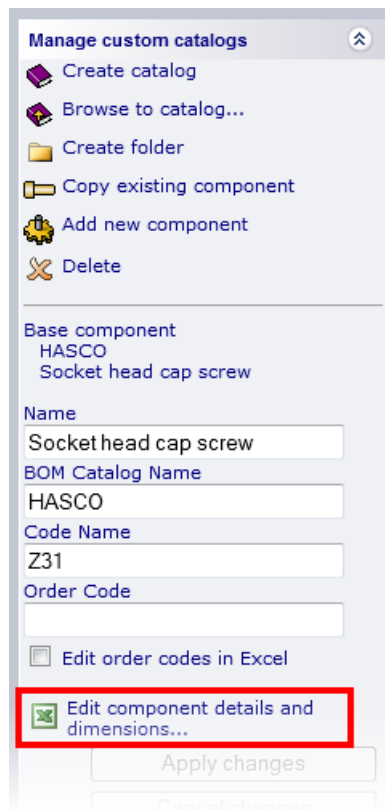
Dimensions and groups are displayed in the **Component Wizard** in the order that is shown in the Excel spreadsheet.

### Example

- 1 Create a custom catalog called **XXXX** that contains the **Hasco Socket head cap screw (Z30)**.



2 Click **Edit component details and dimensions**.



3 MS-Excel will open automatically on your task toolbar.

4 Swap to MS-Excel to display the spreadsheet that has been created automatically.

	A	B	C
1	<b>Name</b>	Socket head cap screw	
2	<b>Code name</b>	Z31	
3			
4			
5	<b>Dimensions</b>		
6	<b>Short name</b>	<b>Name</b>	<b>Group</b>
7	<i>d</i>	Diameter	
8	<i>l</i>	Length	
9	<i>dk</i>	Head diameter	More Parameters
10	<i>k</i>	Head thickness	More Parameters
11	<i>s</i>	Socket width	More Parameters

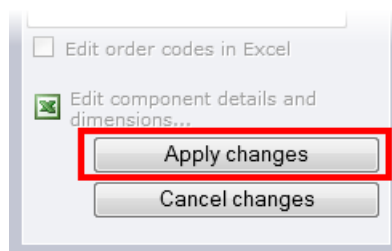
5 Enter the following in the spreadsheet:

- **Main Parameters** into the **Group** cells for **d** and **l**

- **Head Parameters** into the **Group** cells for **dk** and **k**

	A	B	C
1	<b>Name</b>	Socket head cap screw	
2	<b>Code name</b>	Z31	
3			
4			
5	<b>Dimensions</b>		
6	<b>Short name</b>	<b>Name</b>	<b>Group</b>
7	<b>d</b>	Diameter	Main Parameters
8	<b>l</b>	Length	Main Parameters
9	<b>dk</b>	Head diameter	Head Parameters
10	<b>k</b>	Head thickness	Head Parameters
11	<b>s</b>	Socket width	More Parameters

- 6 Save the changes to the spreadsheet
- 7 Swap back to PowerSHAPE and click **Apply changes**.

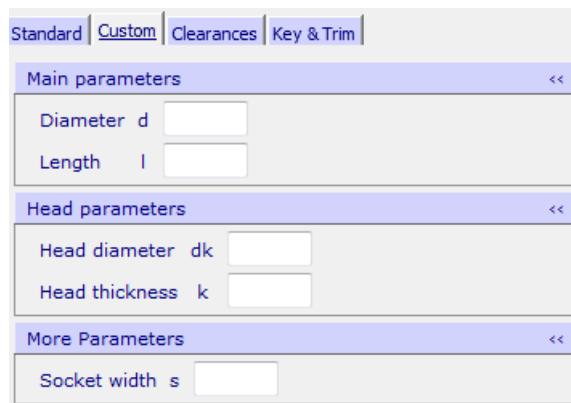


The component parameters are grouped as follows:

- Dimensions that have the same group name are combined into one group.
  - If a group with the given name does not exist, a new group is created.
  - Dimensions with no group name are included in the main group or parameters.
- 8 Double-click the component in the custom catalog tree to open the component in the **Component Wizard** window.
  - 9 Click the **Custom** tab. **Main parameters** and **Head parameters** groups are displayed.



10 Click the groups to display the parameters.



The screenshot shows a software interface with a 'Custom' tab selected. It displays three expandable parameter groups: 'Main parameters' (containing 'Diameter d' and 'Length l'), 'Head parameters' (containing 'Head diameter dk' and 'Head thickness k'), and 'More Parameters' (containing 'Socket width s'). Each group has a double arrow icon to its right.

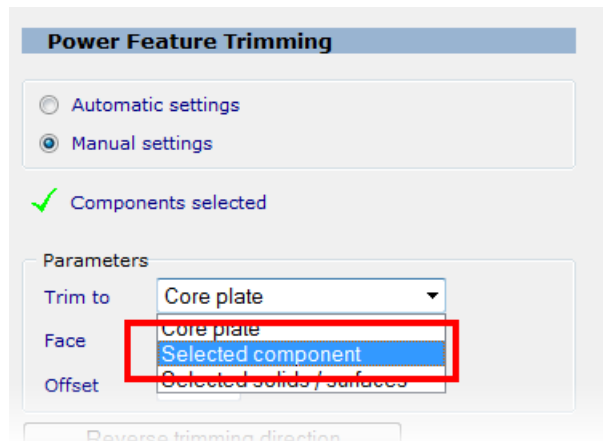


*This technique is useful for a plate; you can group parameters relating to fixing screws separately from those relating to pillar holes.*

## Power Feature Trimming

Power feature trimming has been updated in PowerSHAPE 2013 R2:

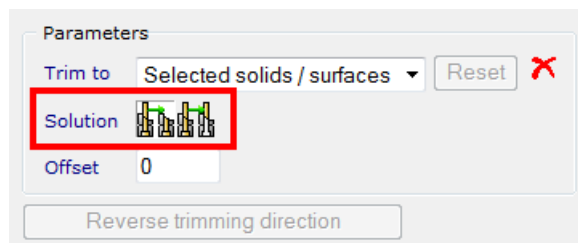
- **Selected component** has replaced **Selected plate** in the Trim to drop-down list



The screenshot shows the 'Power Feature Trimming' dialog box. It has two radio buttons: 'Automatic settings' and 'Manual settings', with 'Manual settings' selected. Below is a green checkmark and the text 'Components selected'. Under the 'Parameters' section, the 'Trim to' dropdown is set to 'Core plate'. A red rectangle highlights the dropdown list, which also shows 'Selected component' and 'Selected solids / surfaces'. The 'Face' and 'Offset' fields are also visible. At the bottom is a 'Reverse trimming direction' button.

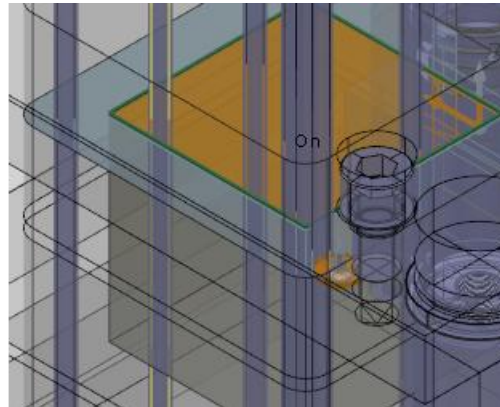
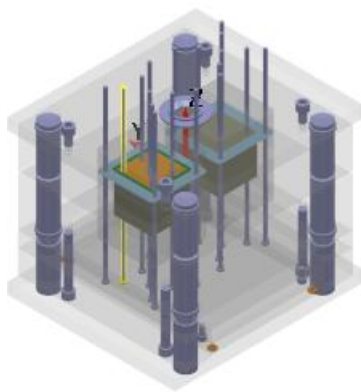
### Power Feature Trimming - choosing the correct solution

When trimming to a surface, the **Solution** buttons ensure that the correct part of the pin is left in the model.

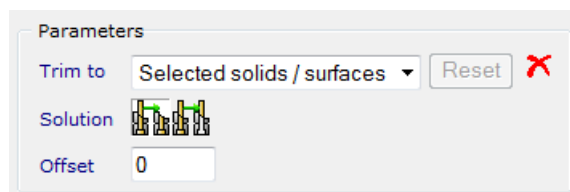



This is a close-up of the 'Solution' section of the dialog. It shows a 'Trim to' dropdown set to 'Selected solids / surfaces' with a 'Reset' button and a red 'X' icon. Below the dropdown, a red rectangle highlights four 'Solution' buttons, each with a different icon representing a different trimming outcome. The 'Offset' field is set to '0'. A 'Reverse trimming direction' button is at the bottom.

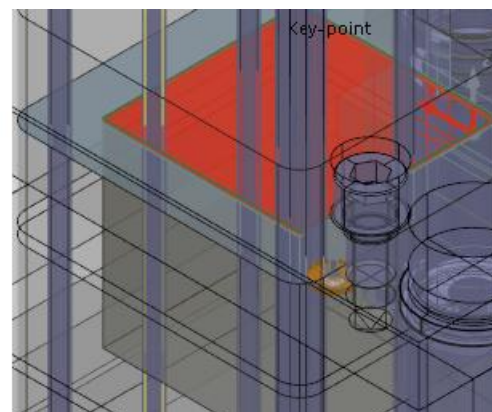
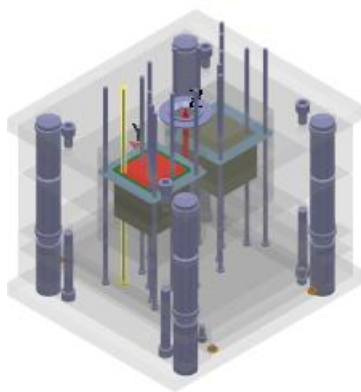
The following model includes a pin that needs to be trimmed to the top of a surface.





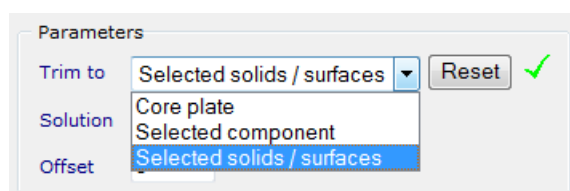
- 1 Click **Manual settings**.
- 2 Select Selected solids /surfaces from the Trim to drop-down list.



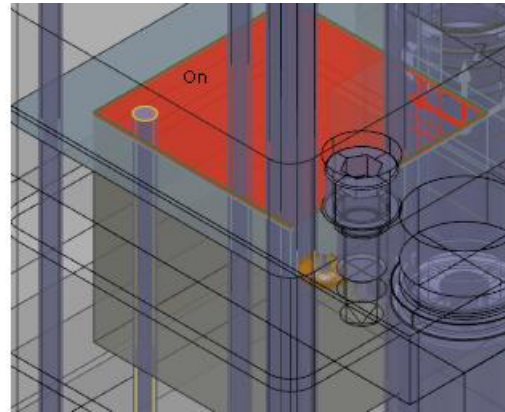
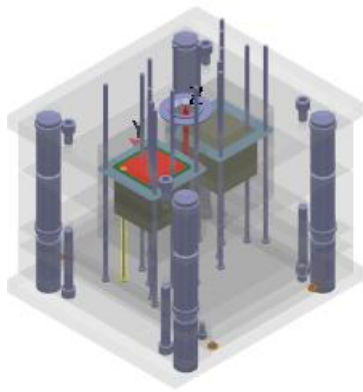
- 3 Default solution  is selected automatically.
- 4 Select a surface.



- 5 The  cross changes to  to indicate that you have selected a solid or surface.

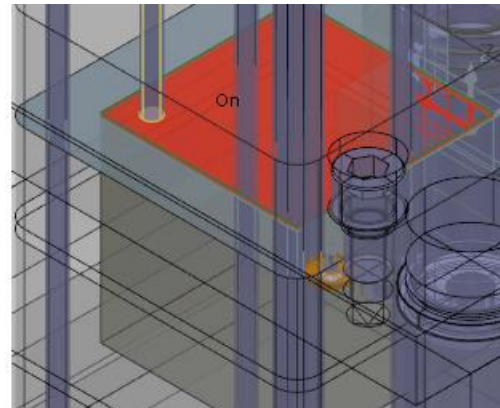
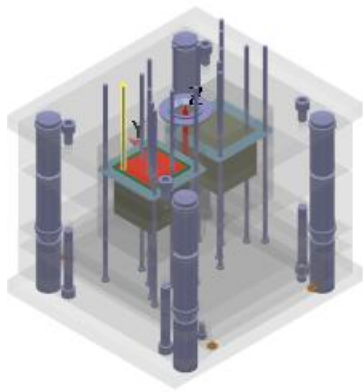


- 6 Click **Trim**. The pin is trimmed to the bottom of the surface, as shown below.



- 7 Click **Other solution** .

- 8 Click **Trim** to trim the pin to the top of the surface, as shown below.



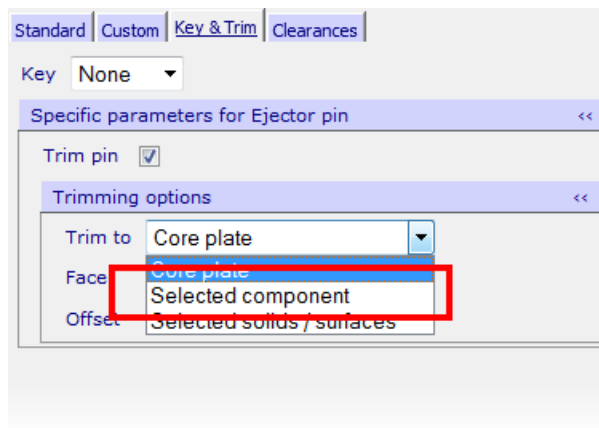
---

## Other changes

Changes have been made to the general operation of PowerSHAPE:

- Variables (see page 50)
- Import / Export (see page 51)
- The **Trim to** drop-down list on the **Key & Trim** tab of the **Component Wizard** has been updated:

**Selected component** has replaced **Selected plate**.



- PowerSHAPE now supports model sizes over 2GB by default. This is controlled by the `large_sector_size` command in the `powershape.con` file:

```
large_sector_size: true
```

This resource takes effect on new models. Alternatively, use **Save As** to save older models with a new name. Models with larger sector sizes can be opened by with older versions of PowerSHAPE.

- All models opened in PowerSHAPE 2013 R2 use model version **24**. When PowerSHAPE 2013 R2 is installed and run for the first time, the initialisation work will open the latest `NEW_MODEL_MASTER_XX` you have (where `XX < 24`) and save it as `NEW_MODEL_MASTER_24`.

## Variables

Use the following variables with PowerSHAPE 2013 R2:

- The **PRINT** command now works for expressions that evaluate strings, vectors and lists; previously, it only worked for expressions that evaluated numbers.

### Examples

```
print concatenate('abc'; 'def')  
prints the string  
abcdef
```

```
print cross([1; 2; 3]  
prints the resulting vector  
[40; -50.5; 76.23]
```

```
print atan2(-30; 40)  
prints the arctangent
```

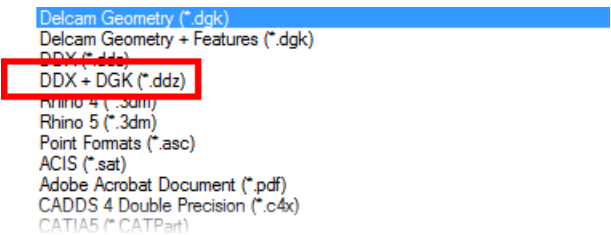
- There is a new **cross()** function that returns the cross product of two vectors. This is the vector that is perpendicular to the two vectors. For example, the cross product of the X and Y axes is the Z axis.

```
print cross([1;0;0]; [0;1;0])
returns [0;0;1]
```

## Import / Export

The following changes have been made in PowerSHAPE 2013 R2:

- When using PowerMILL Modelling, you can now import solids from other solid modellers using Parasolid.
- A new file type has been added to the list of file types that can be used when exporting:



Delcam Geometry (\*.dgl)  
 Delcam Geometry + Features (\*.dgl)  
 DDX (\*.ddx)  
**DDX + DGK (\*.ddz)**  
 Rhino 4 (\*.3dm)  
 Rhino 5 (\*.3dm)  
 Point Formats (\*.asc)  
 ACIS (\*.sat)  
 Adobe Acrobat Document (\*.pdf)  
 CADD5 4 Double Precision (\*.c4x)  
 CATIA5 (\*.CATPart)

If you select **DDX + DGK (\*.ddz)**, all Parasolid solids will be converted to v8 solids.



*This file type should be used when using PowerMILL Modelling*



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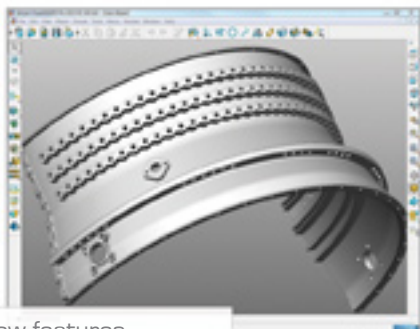
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# PowerSHAPE 2013 R2

PowerSHAPE 2013



new features

Delcam TV



[www.delcam.tv](http://www.delcam.tv)

PowerSHAPE 2013 Learning Zone



[www.delcam.tv/lz](http://www.delcam.tv/lz)

PowerSHAPE Website



[www.powershape.com](http://www.powershape.com)



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