Delcam Advanced Manufacturing Solutions

PowerMILL 2014 R2

World-leading 2, 3 and 5-axis CAM software

www.powermill.com

Vortex

Now Available

What's New

Powering your productivity

Delcam Advanced Manufacturing Solutions

www.delcam-ams.com

Whether you are a new or long-term user of Delcam products, you may not be aware of the complementary technologies available from Delcam that could improve your company's productivity.

Delcam Advanced Manufacturing Solutions offers a complete and diverse range of CADCAM software solutions to integrate every aspect of your production life-cycle – from designing a complex concept, to manufacturing it and inspecting the final output – providing you with unrivalled speed, flexibility, and ease-of-use throughout your product development process.

Adding to your suite of Delcam solutions minimises the impact on your daily production operations while also enabling you to enter new markets or automate your processes with our advanced functionality.

All products work independently or together as one complete manufacturing solution tailored to your specific needs.

PowerSHAPE

www.powershape.com

PowerSHAPE provides a complete environment to manipulate surface form, build from wireframe, and add solid features for prismatic parts.

- Import/export Parasolid-based systems with no translation.
- Create complex molds from solid models, complete with split surfaces.
- Powerful surface- and face-editing tools quickly repair imported data.
- Tools to perform non-feature-based edits quickly and efficiently.

PowerMILL

www.powermill.com

PowerMILL is the world's leading specialist NC CAM software for the manufacture of complex shapes, providing advanced-machining strategies to minimise machining time and maximise finish quality.

- World-leading high-speed machining strategies.
- Advanced 5-axis machining techniques.
- Support for 64-bit platforms and multi-threading.
- Innovative collision-avoidance methods.
- Powerful toolpath editing and tool-axis stabilisation.

FeatureCAM

www.featurecam.com

FeatureCAM is a unique CAM system that uses feature-based and knowledge-based technologies for automated machining, minimising programming times for mills, lathes, turn/mill, and wire machines.

- Easy to use.
- Single interface for multiple machine platforms.
- Powerful turning and milling operations from 2.5D to 5-axis.
- Multi-threading capabilities.

Delcam for SolidWorks

www.delcamforsolidworks.com

Delcam for SolidWorks is a SolidWorks Certified Gold Product that revolutionises CAM programming inside SolidWorks.

- Integrates the feature-based technology from FeatureCAM.
- Multi-threaded toolpath algorithms from PowerMILL.
- 2-axis, 3-axis, and 5-axis positional milling and drilling.
- Turning, turn/mill, and wire EDM capabilities.
- Automatic selection of cutting tools, machining strategies, and feeds and speeds.
- Exceptional toolpath-calculation speeds.
- Set-up wizards.
- Full-machine simulation.

PartMaker

www.partmaker.com

PartMaker applies a patented Visual Programming approach to automate the programming of multi-axis Swiss-type lathes and Turn-Mill Centres.

- Easier programming of turning with live tooling via the Divide-and-Conquer programming approach.
- Automatic process synchronisation.
- Vivid 3D simulation and crash detection.
- Wide array of proven post processors for Turn-Mill Centres and Swiss-type lathes.

PowerINSPECT

www.powerinspect.com

PowerINSPECT leads the way in today's inspection market. It delivers a complete CAD-based inspection solution that can accept data from all types of hardware, including manual and CNC coordinate measuring machines, portable arms, optical measuring devices, and CNC machine tools.

- Part comparison against all mainstream CAD formats.
- Support for all types of measuring devices.
- Market-leading inspection reports that are quick to create and easy to understand.
- IGES export of measured features, including digitized curves.
- Additional modules for part alignment, laser line inspection, and tube inspection.

www.artcam.com

ArtCAM is a unique application that combines the benefits of computerised design and CNC machining in a simple-to-use format to create decorative products from artwork.

- Import 3D models, clipart, and other CAD system formats.
- Add geometric shapes, weaves, and textures.
- Choose from extensive visualisation and rendering materials.
- Use the comprehensive and customisable tool database for fast toolpath-calculation.
- Powerful toolpath simulation verifies machining times and materials.
- Design in the 3D view.

Delcam also provides a range of healthcare CADCAM solutions for the dental, custom orthotic insoles, medical implants, and orthopaedic footwear industries, as well as CADCAM solutions for footwear design and manufacturing.

You can connect with Delcam in a variety of ways:

visit: www.delcam.com | watch: www.delcam.tv | learn: www.delcam.tv/lz

PowerMILL 2014 R2

What's New

Issue 1

PowerMILL

Acknowledgements

Patents

The Raceline smoothing functionality is subject to patent applications.

Patent granted: GB 2374562 Improvements Relating to Machine Tools

Patent granted: US 6,832,876 Machine Tools

Some of the functionality of the ViewMill and Simulation modules of PowerMILL is subject to patent applications.

Patent granted: GB 2 423 592 Surface Finish Prediction

The Vortex machining functionality is subject to patent applications.

Patent application: 1121277.6 Adaptive Clearance

The MachineDNA functionality is subject to patent applications.

Patent application: 1204908.6 Machine Testing

Licenses

Intelligent cursor licensed under U.S. patent numbers 5,123,087 and 5,371,845 (Ashlar Inc.)

PowerMILL 2014R2. Published on 8 January 2014

Contents

Summary of new features

PowerMILL is the leading NC CAM software specialising in the manufacture of complex shapes typically found in the toolmaking, automotive, and aerospace industries. PowerMILL 2014R2 offers all of the original features of PowerMILL 2014, but with numerous improvements. This document describes the most significant improvements.

Toolpath preparation

There is a new composite curve creator which greatly enhances 2.5D machining and flowline machining (see page 3).

There is now a separate dialog for hole creation and editing. This simplifies the **Feature** dialog as it no longer contains the hole specific options.

- There is an improved process for hole creation because it is significantly different to the creation of other features (see page 36). There is now a separate dialog for hole creation so the **Feature** dialog no longer contains the hole specific options.
- There is a new dialog to control hole editing and hole intersections (see page 44).
- There are several changes to the Feature Set and individual feature set menus (see pages 59–60).
- You can now see a history of your boundary edits. This is similar to the history of your toolpath edits (see page 64).

You can now see a history of your boundary edits (see page 64).

There are improvements to specifying the tool stock material (see page 68).

Toolpath generation

When drilling compound holes, you can now choose which hole components you want to drill (see page 71).

When creating a drilling method, you can now add an existing drilling toolpath and notes (see page 70).

Advanced simulation

The new **Dynamic machine control** mode enables you to simulate and adjust the configuration and tool axis of a 3+2 machine tool, so you can quickly and easily improve the machine tool's access to the workpiece (see page 72).

Toolpath output

There are several improvements to NC programs functions. You can now:

- Multi-select NC programs (see page 85).
- Draw all the toolpaths used in an NC program (see page 85).
- Add toolpaths to the NC program multiple times (see page 86).
- Create fixture offset more easily (see page 70).

User interface

The new **Machine Tool Position** and **Collision** dialogs replace the **Machine Information** dialog (see page 87).

The new dialogs:

- Improve the presentation of machine tool position and collision data.
- Enables easier jogging of a machine tool’s axes.

Automation

There are enhancements to the macro programming language (see page 91).

General enhancements

There is an improved process for creating and managing custom toolbars. In PowerMILL 2014R2 you can have up to thirty-two custom toolbars at any time (see page 92).

Toolpath preparation

Composite curve creator

There is a new composite curve creator which greatly enhances 2.5D machining and flowline machining.

The **Create Composite Curve** mode toolbar merges adjacent arcs, lines, curves, and surface edges into one composite curve. This offers very similar functionality to PowerSHAPE's composite curve creator.

Selecting  displays the **Create Composite Curve** mode toolbar.

 is available from:

- the **Curve Creation** toolbar on the **Curve Editor** mode toolbar

- the **Pattern** toolbar
- the **Boundary** toolbar

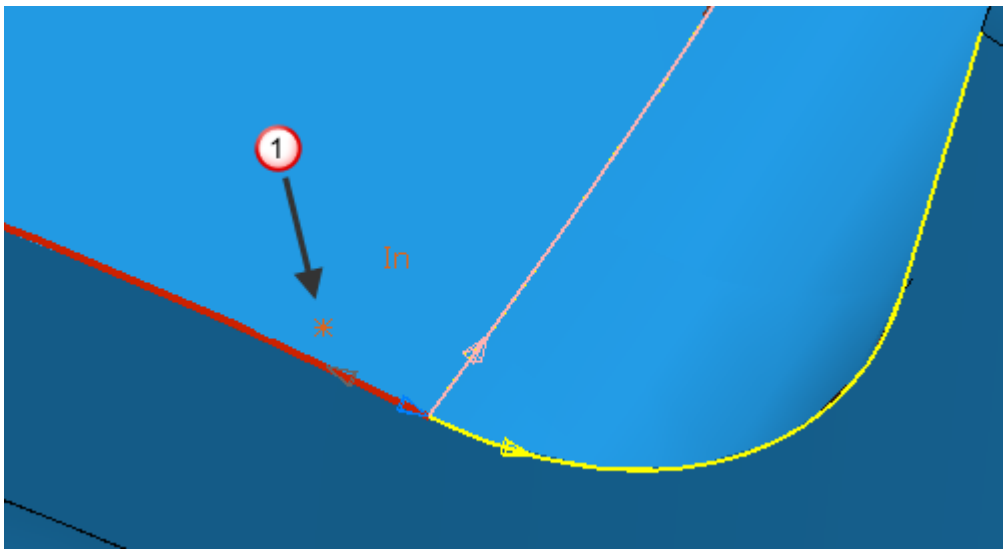
Activating a mode disables most of PowerMILL's functionality until you exit from the mode. For more information, see Mode toolbars.

You build a composite curve from existing geometry, starting by selecting a curve (see page 8) or surface (see page 10). Where you select the curve or surface determines the default direction. Then the options enable you to guide the path of the curve to create the required composite curve.

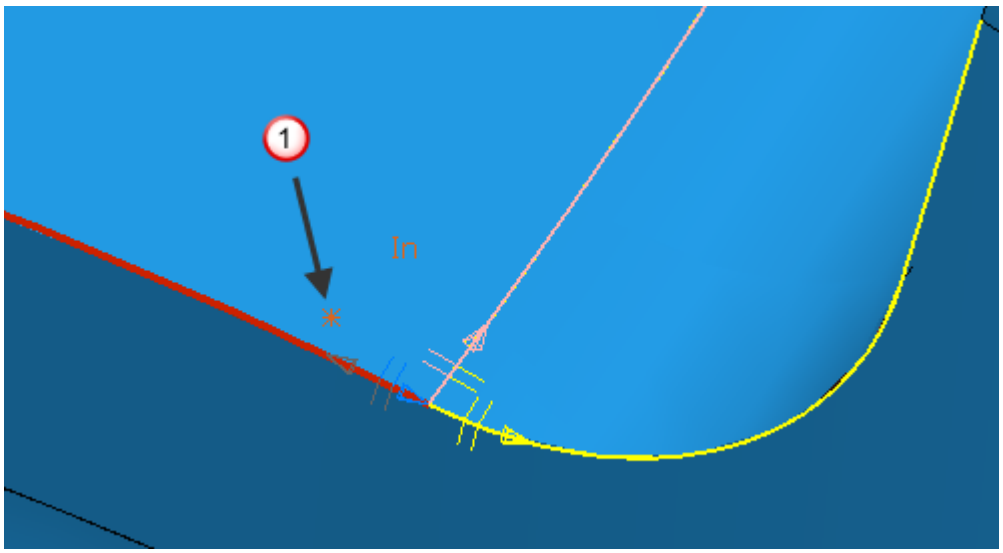
The **Create Composite Curve** toolbar works on curves (not surfaces). If a surface is in an area of interest, it selects the surface edge curve and operates on that. This edge curve is always a complete closed loop that goes all the way around the perimeter of the surface.

Surface marker — Click to display the surface markers.

deselected:



selected:



1 Surface selection point

The surface markers use the same colour scheme as the curve selection:

- Pink indicates the surface edge automatic sketching follows.
- Yellow indicates alternate surface edges.
- Blue indicates the current surface for the current curve.
- Grey indicates an alternative surface for the current curve. This is an option you are unlikely to use because it returns to the start point, but along the other surface. For more information, see Selecting curves to create a composite curve (see page 8).

Reset — Click to go back to the start and deselect all curves. It still displays the **Create Composite Curve** mode toolbar.

Marker point 1 — When sketching a composite curve, click and then click on a curve to specify the marker point. The marker points prevent the composite curve progressing beyond this point (see page 27).

If the composite curve doesn't meet this marker point, the marker point has no effect on the composite curve.

Rewind — Click to go back to either the previous branch point, or the beginning (if no branch points exist).

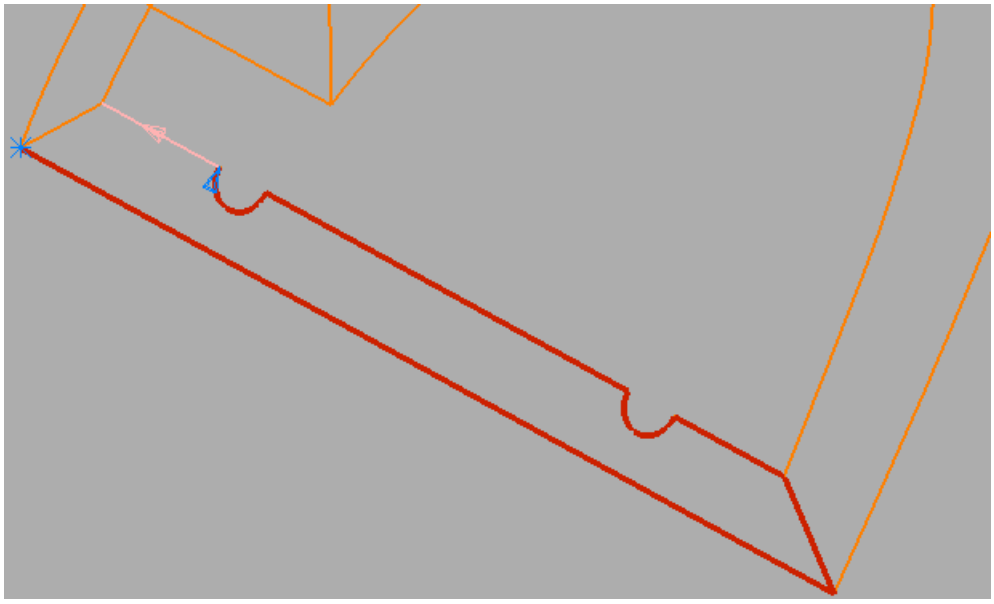
Starting with:

Clicking gives:

Backward — Click to go back to the previous curve. This takes you back to where two or more curves meet, so it may not take you back as far as the previous branch point.

Starting with:

Clicking gives:



Forward — Click to proceed to either the next branch point, or the end (if no branch points exist).

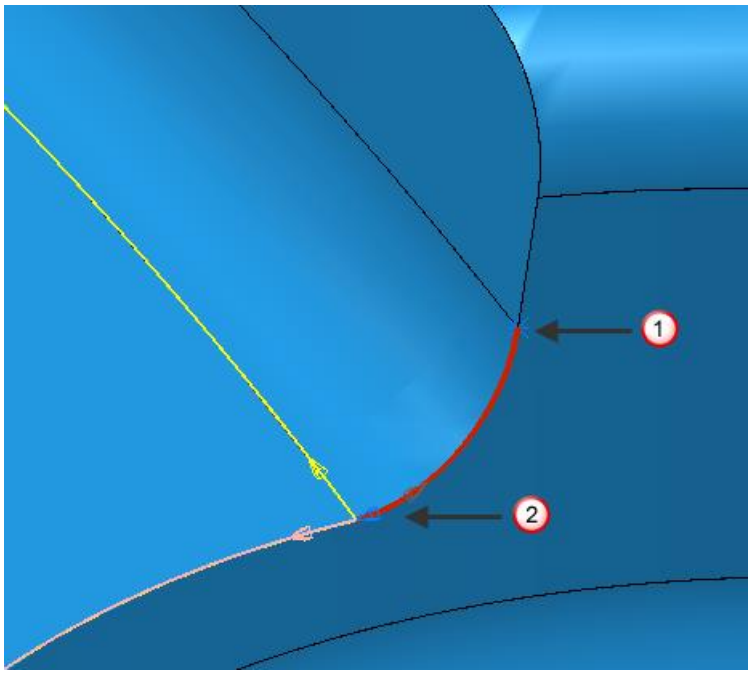
Fast Forward — Click to proceed along the automatic path as far as possible.

Marker point 2 — When sketching a composite curve, click and then click on a curve to specify the marker point. The marker points prevent the composite curve progressing beyond this point (see page 27).

If the composite curve doesn't meet this marker point, the marker point has no effect on the composite curve.

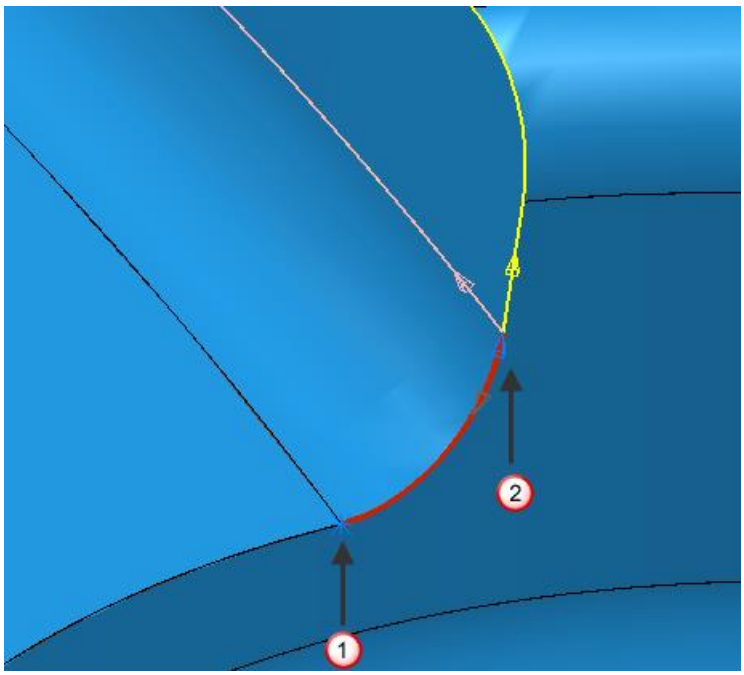
Reverse — Click to change the direction of the curve.

Starting with:



1. Start point
2. Curve direction

Clicking gives:



Save and Restart — Click to save the current selection as a composite curve. This still displays the **Create Composite Curve** mode toolbar, so you can create another composite curve in this pattern or boundary.

Direction — Hover over the button to display the **Direction** toolbar. The option you select determines the default direction at a branch point. When PowerMILL encounters a branch point, the composite curve creator displays the choices using an arrow and colour (see page 12).

Composite curve creator options — Click to raise the **Options** dialog with the **Composite curve creator** page displayed (see page 31).

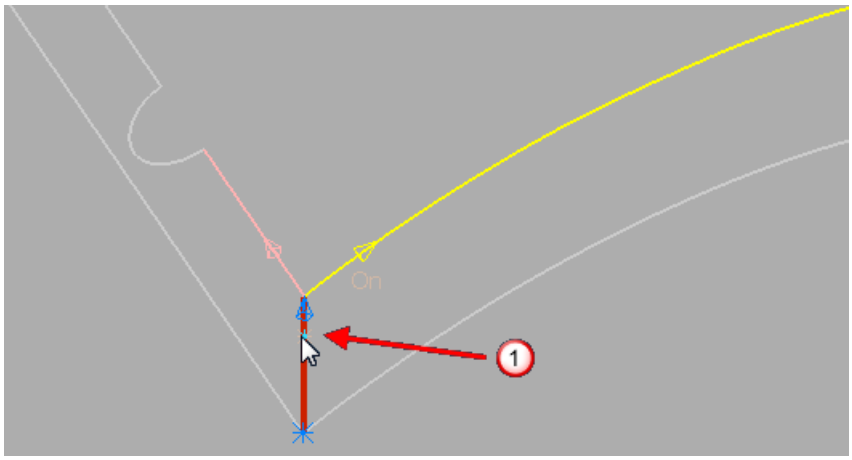
Accept changes — Click to accept the current composite curve. This returns you to the **Curve Editor** mode toolbar.

Cancel changes — Click to delete the current composite curve. This returns you to the **Curve Editor** mode toolbar.

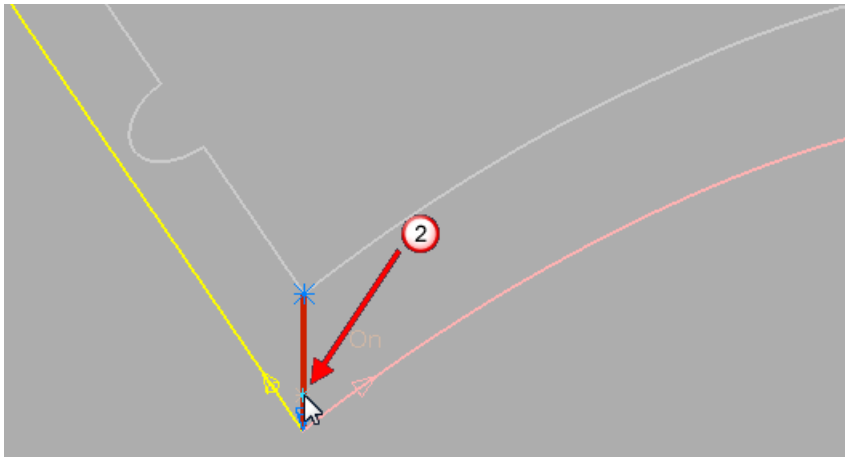
Selecting curves to create a composite curve

When the **Create Composite Curve** mode toolbar is displayed, where you select the curve affects the default direction of the composite curve.

The forward direction points to the curve-end closest to the selection point. So, selecting the curve at point 1, gives a forward direction of upwards.



Selecting the curve at point 2, gives a forward direction of downwards.



- The blue asterisk (*) indicates the start point of the composite curve.
- The blue arrow indicates the current end point of the composite curve.
- To extend or change your composite curve either click on the next curve, or use the play buttons.

  When you have created the required composite curve, click  or  to accept or cancel the curve creation.
- The red curve indicates the current extents of the composite curve.
- The pink curve indicates the direction PowerMILL follows if you click  , or  . You can also select this direction by clicking the pink curve.
- The yellow curve indicates an alternative direction. To select this direction, click the yellow curve.

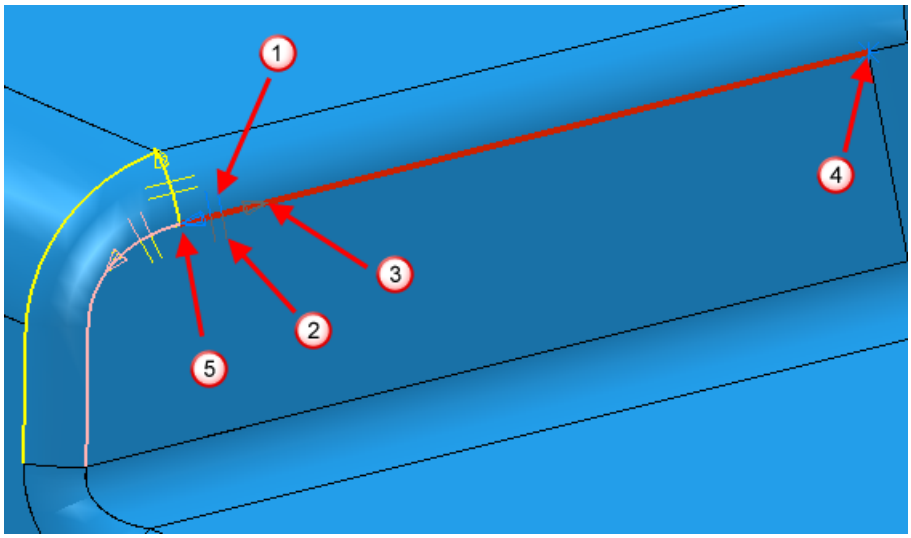
Selecting surfaces to create a composite curve

You can create a composite curve using surface edges. You don't need curves to create a composite curve.

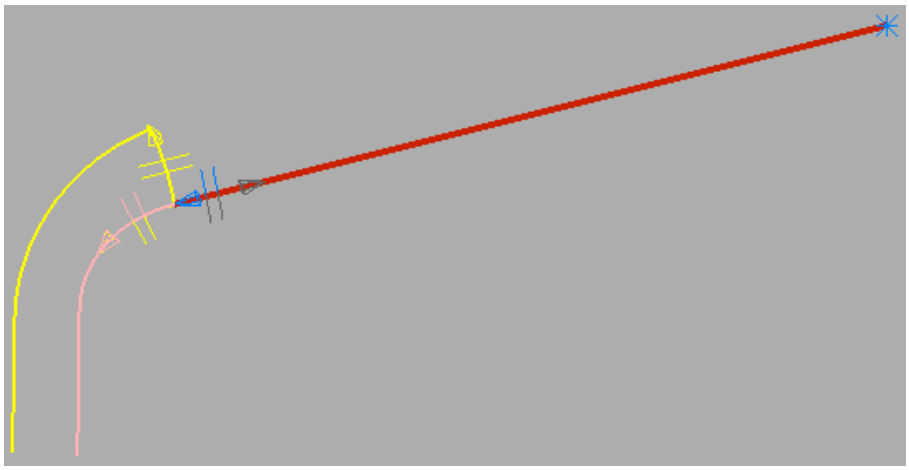
With the **Create Composite Curve** mode toolbar displayed, where you select the surface affects:

- the default direction of the composite curve.
- the surface used to create that section of the composite curve.

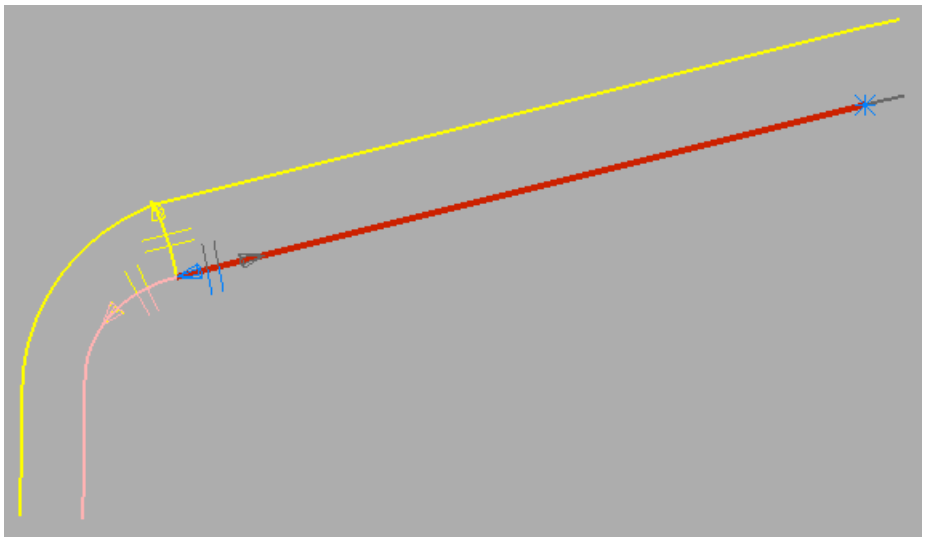
The forward direction is towards the surface corner closest to the selection point. Selecting a surface at point (1) creates the start point at the bottom right corner of the fillet surface ((4)) and a current-curve-end at the bottom left corner of that fillet ((5)). The curve used to create the composite curve is the surface edge of the fillet.



It is easier to see if you undraw the shaded model:



If you select the surface around 2, the composite curve looks the same, but the curve used to create the composite curve is the surface edge of the vertical face.



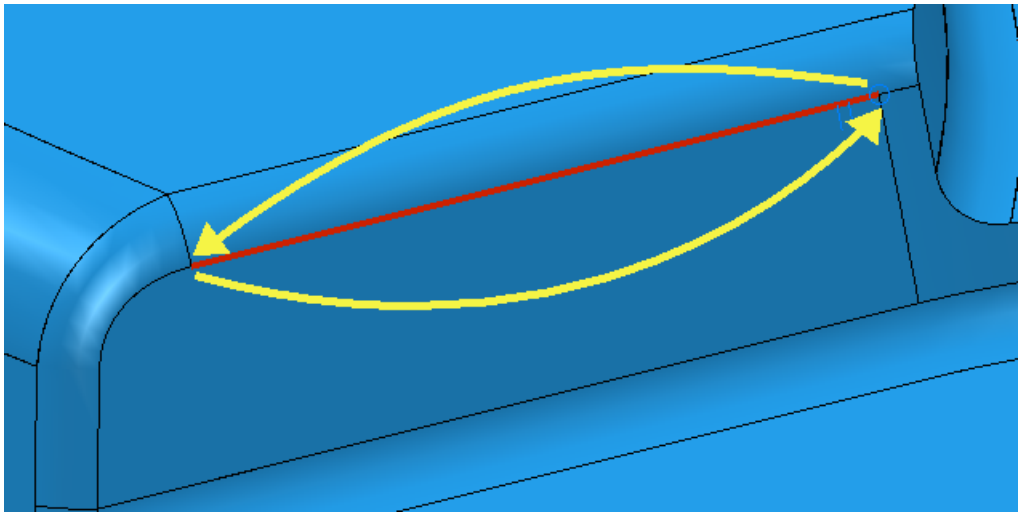
If you select the surface around 3, PowerMILL selects the surface it considers best. It does this by inspecting the p-curves of the underlying surfaces, and then selects the surface that is least likely to produce a 'faceted' curve. For instance, the edge of a cylinder produces a much better quality curve than a circle trimmed out of a plane. For example, if you select a boss or pocket in the powerdrill.dgk model, PowerMILL does not pick the trimmed plane.

- The blue asterisk (*) indicates the start point of the composite curve.
- The blue arrow indicates the current end point of the composite curve.
- To extend or change your composite curve either click on the next curve, or use the play buttons.

  When you have created the required composite curve, click  or  to accept or cancel the curve creation.
- The red curve indicates the current extents of the composite curve.
- The pink curve indicates the direction PowerMILL follows if you click  , or  . You can also select this direction by clicking the pink curve.

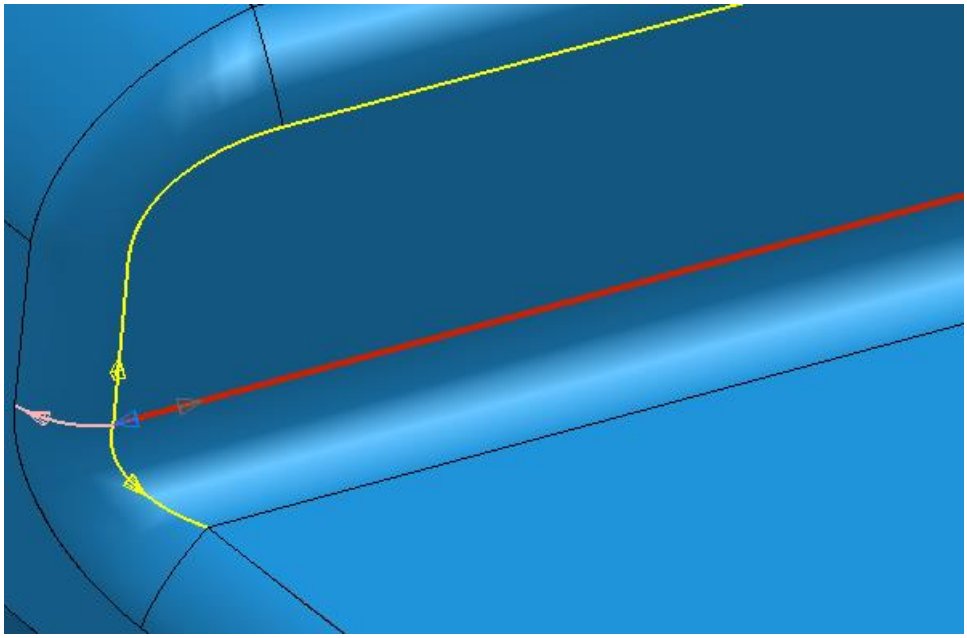
- The yellow curve and surface markers indicate alternative directions and alternative surfaces that PowerMILL uses to create the next section of the composite curve. To select a yellow option, click on the appropriate yellow curve or surface marker.
- The grey arrow and surface indicate a possible, though unlikely, solution. Clicking it enables you to create a composite curve which has a start point at the bottom right corner of the fillet surface and goes along to the bottom left corner of the fillet and then returns to the start point by following the vertical surface edge.



Composite Curve Direction toolbar

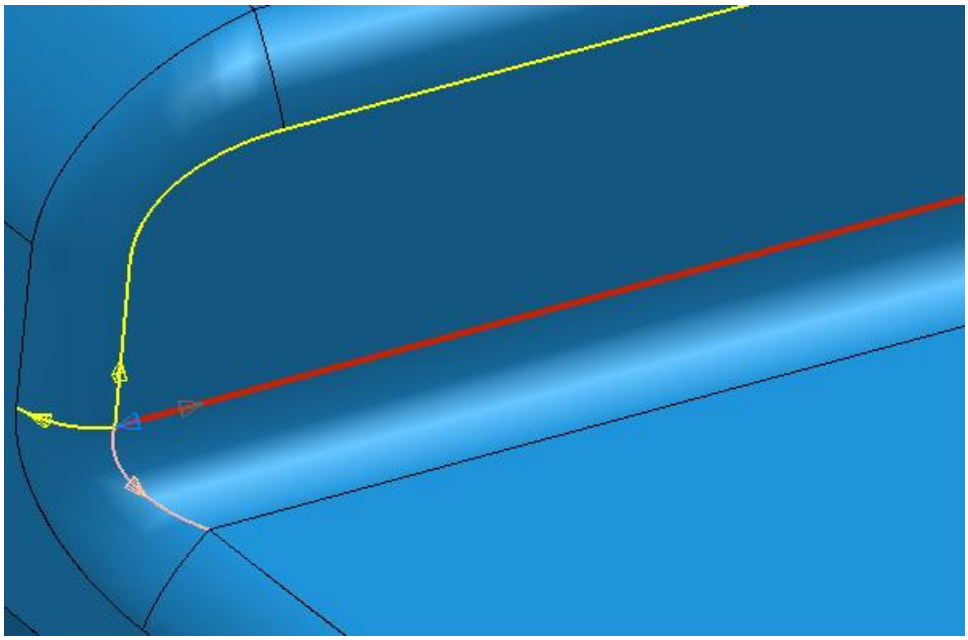
Hovering over **Direction**  on the **Create Composite Curve** toolbar displays the **Direction** toolbar. Use the toolbar to determine the default direction at a branch point.

Clicking **Straight** gives:

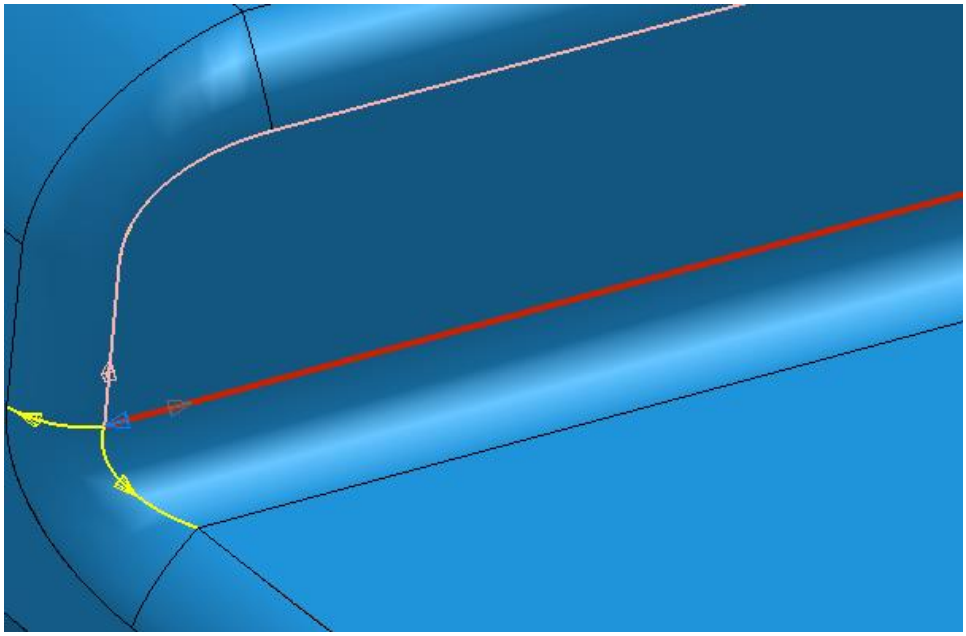


Pink indicates the direction that automatic sketching follows. Yellow indicates an alternative.

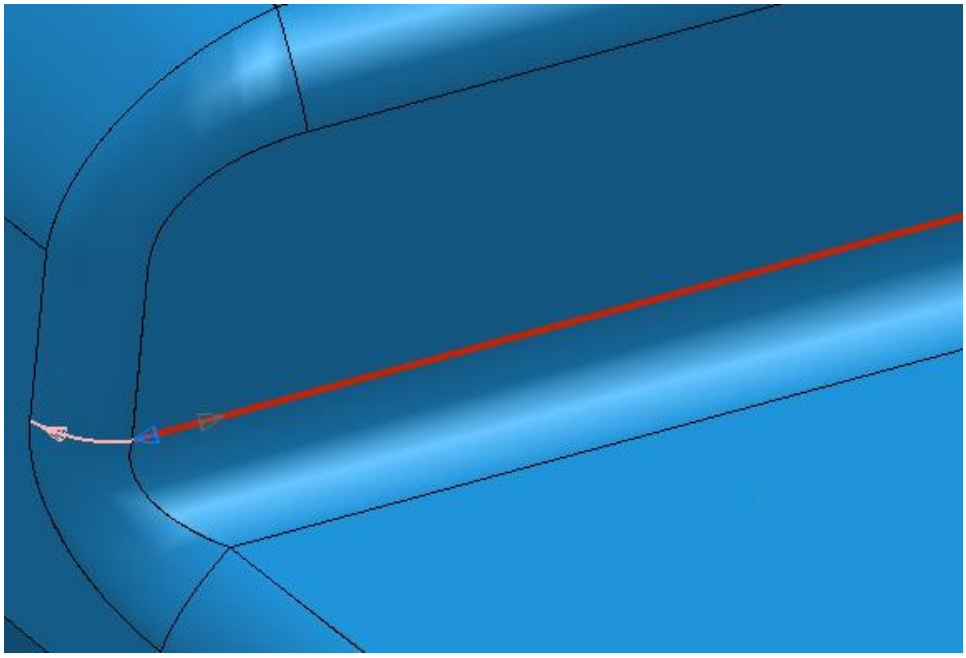
Clicking **Left** gives:



Clicking **Right** gives:

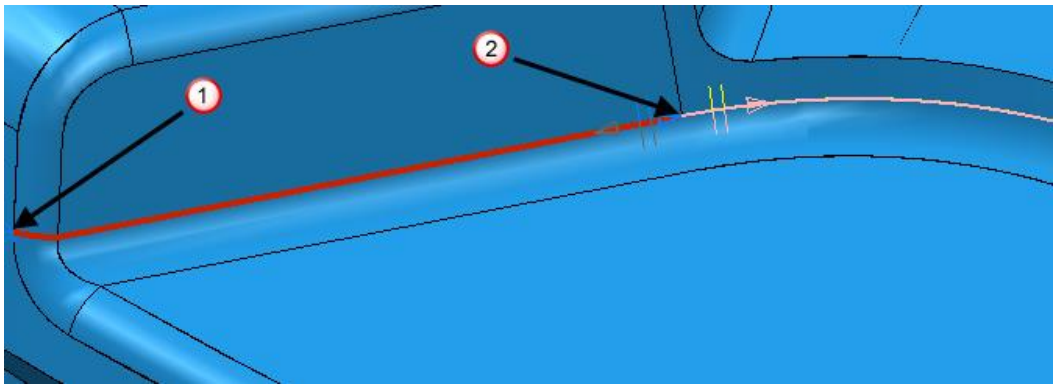


Clicking **Stop at discontinuity** gives:



This is the same as **Straight** provided the angle at the branch point is less than the discontinuity angle specified on the **Composite curve creator** page of the **Options** dialog. However, it stops if the angle at the branch point is greater than the discontinuity angle specified (see page 31).

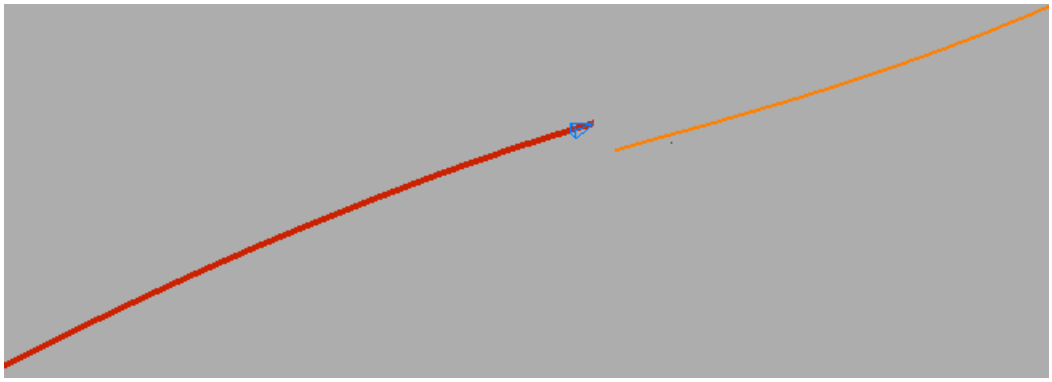
Clicking on the pink curve gives:



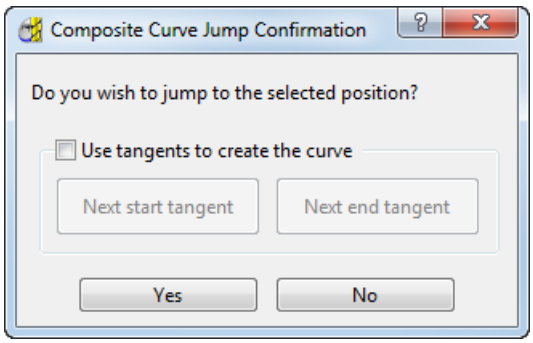
In this case, the composite curve extends as far as point 1. Because the angle at the branch point is greater than the discontinuity angle, the composite curve creator extends the curve from the other end (point 2).

Composite Curve Jump Confirmation dialog

When creating a composite curve, PowerMILL stops when it reaches a gap in the geometry. Click on the curve you want to join.

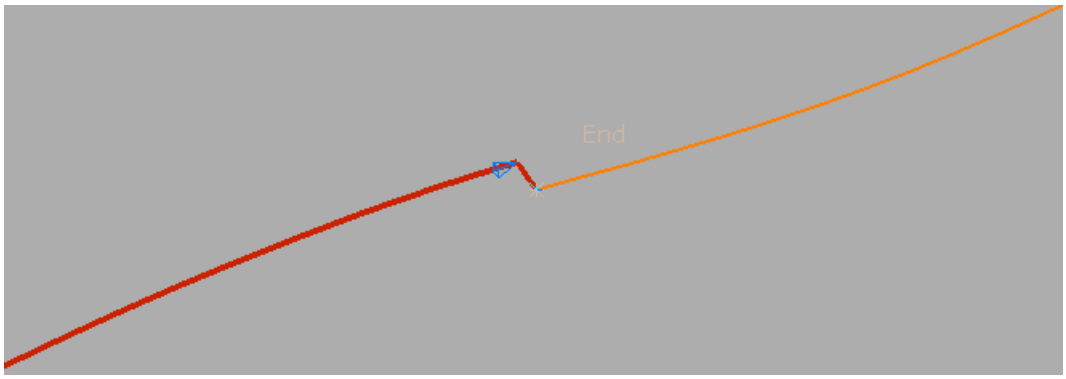


Clicking on the orange curve displays the **Composite Curve Jump Confirmation** dialog.

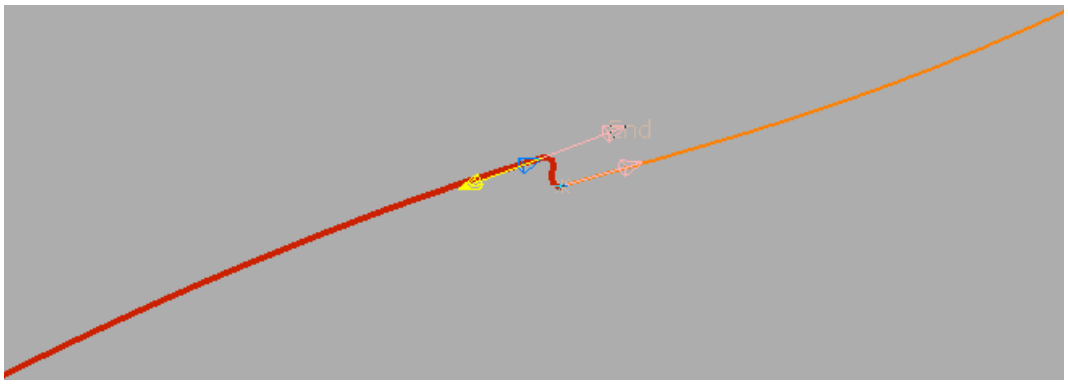


Use tangents to create the curve — When deselected, joins the curves with a straight line. When selected, joins the curves with a tangential curve.

Use tangents to create the curve deselected:



Use tangents to create the curve selected:



Next start tangent — Click to cycle through the various tangent options.

Next end tangent — Click to cycle through the various tangent options.

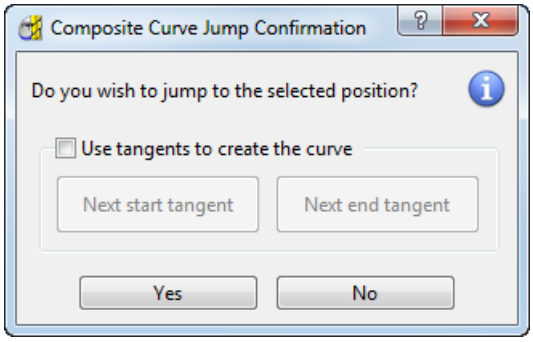
Yes — Click to accept the join and return to the **Create Composite Curve** toolbar (see page 12).

No — Click to reject the join and return to the **Create Composite Curve** toolbar (see page 12).

For more information, see Creating a composite curve from 'gappy' wireframe geometry (see page 24).

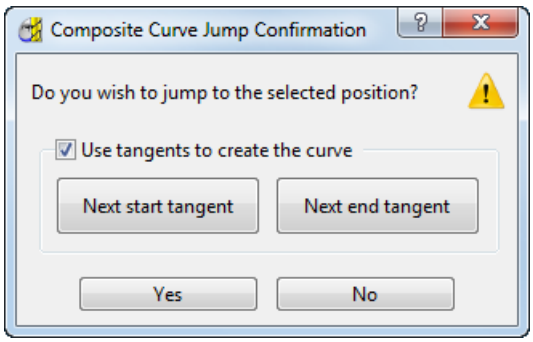
If you are creating a composite curve by selecting surfaces rather than existing curves, and you want to bridge a gap, the **Composite Curve Jump Confirmation** dialog has an extra field.

Use tangents to create the curve deselected:



— When displayed, the composite curve remains an embedded curve. Hover over to display more information.

Use tangents to create the curve selected:



— When displayed, the composite curve is no longer an embedded curve. Hover over to display more information .

If you create a composite curve by selecting surfaces or curves extracted from a surface, the resulting composite curve is an embedded pattern.

Creating a composite curve from surface geometry

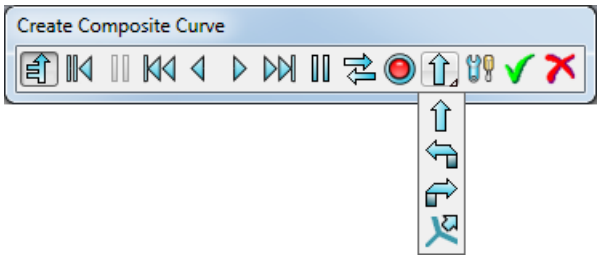
This example shows how to create a composite curve from surface geometry. The result is a composite curve which is also an embedded pattern.

This uses the 5axis_with_holes.dgk model in the **Examples** folder.

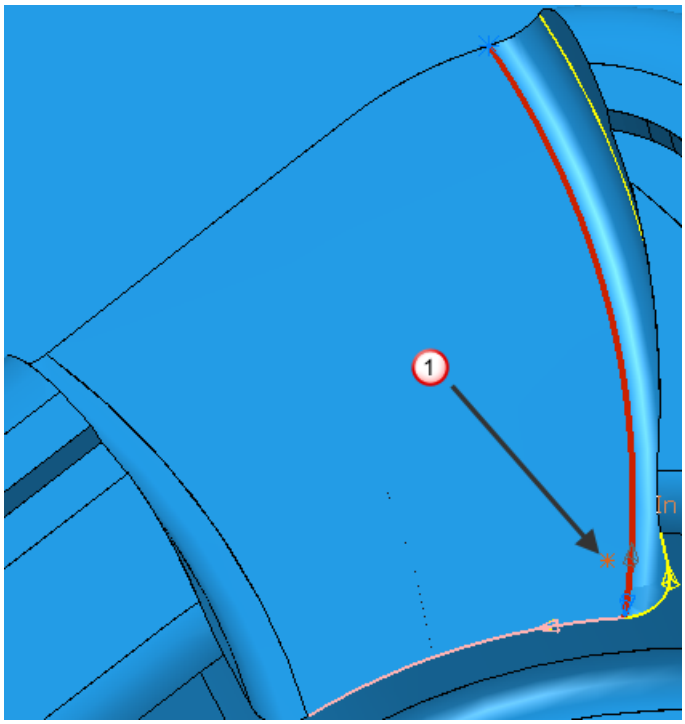
1. Click on the **Pattern** toolbar to create a new pattern.
2. Click on the **Pattern** toolbar. This displays the **Curve Editor** mode toolbar.
3. Click on the **Curve Creation** toolbar.

This displays the **Create Composite Curve** mode toolbar.

4 Click  on the **Direction** toolbar.



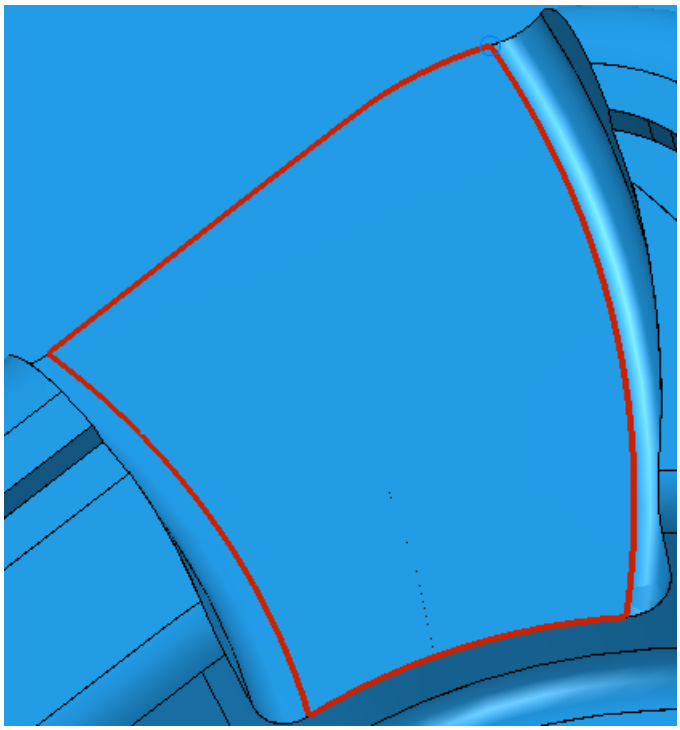
5 Click on the surface near 1.



This selects the right edge of the surface.

- Red indicates the selected surface curve.
- Pink indicates the surface that automatic sketching follows.
- Yellow indicates an alternative surface.
- Grey indicates an alternative surface for the current curve. This is an option you are unlikely to use because it returns to the start point, but along the other surface. For more information, see Selecting curves to create a composite curve (see page 8).

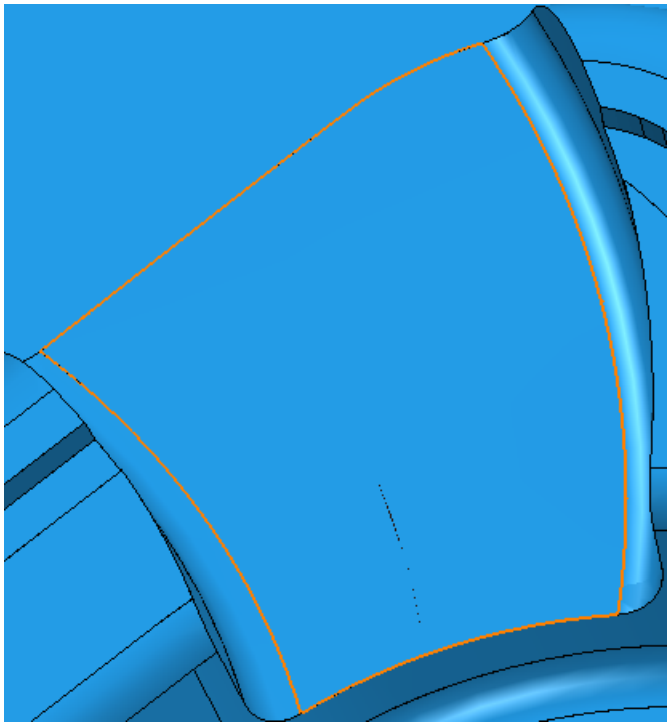
6 Click on the **Create Composite Curve** mode toolbar to create the automatic path.



*Use the **Esc** key or click (next to the progress indicator) if you want to stop the composite curve creation.*

7 Click on the **Create Composite Curve** mode toolbar.

8 Click on the **Curve editor** mode toolbar. This creates an embedded pattern which follows the surface edges.

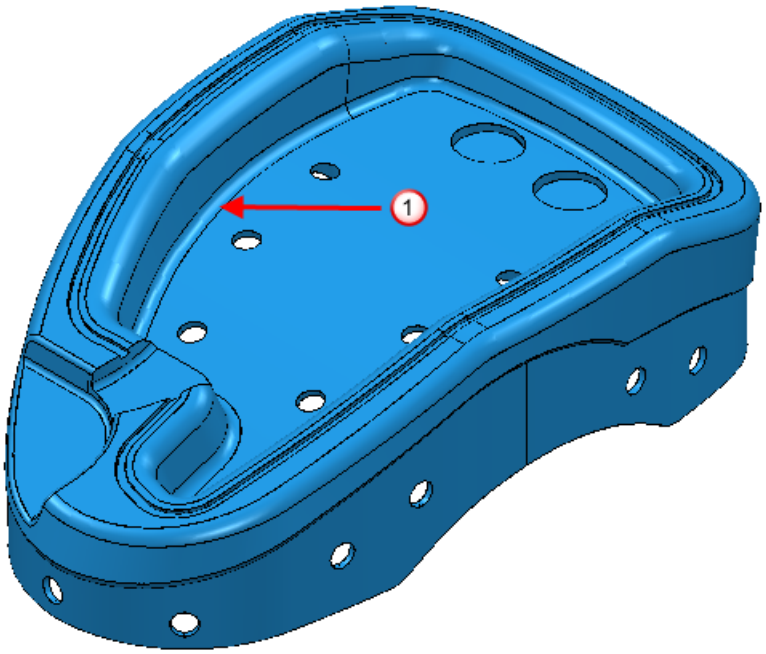


Automatic creation of a composite curve

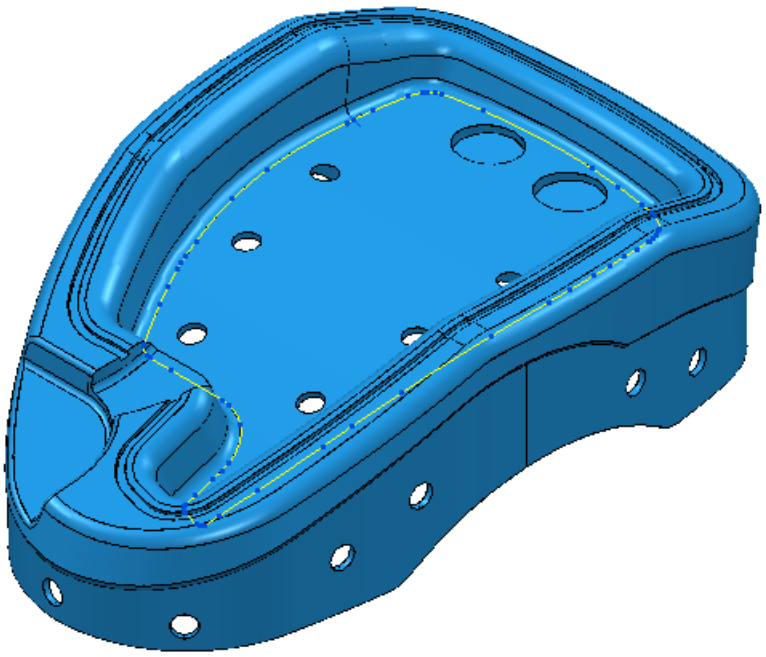
This example shows how to create a composite curve automatically by clicking a surface edge.

It uses the 5axis_with_holes.dgk model in the **Examples** folder.

1. Click on the **Pattern** toolbar to create a new pattern.
2. Click on the **Pattern** toolbar. This displays the **Curve Editor** mode toolbar.
3. Use the **Alt** key while selecting the surface edge near 1.

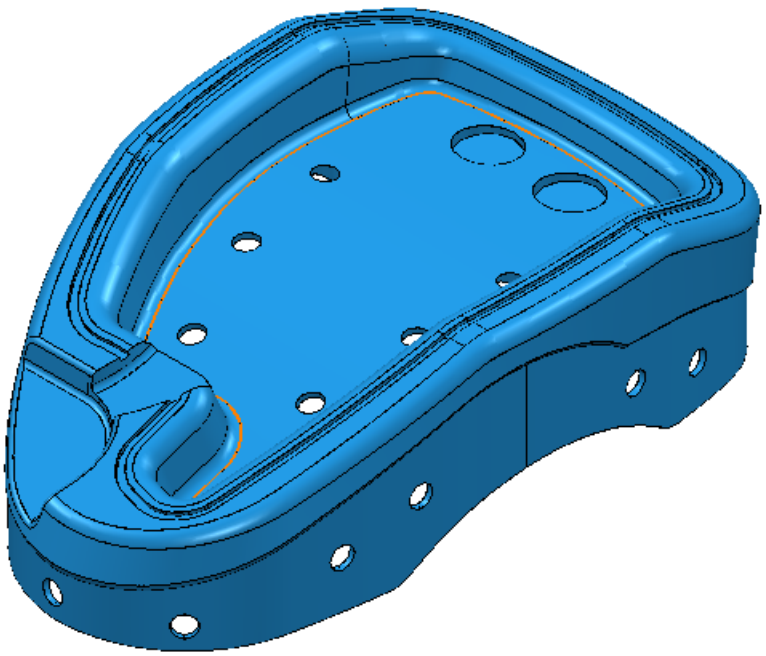


PowerMILL creates a composite curve using the default options. The resulting curve is highlighted.



*Use the **Esc** key or click (next to the progress indicator) if you want to stop the composite curve creation.*

4 Click on the **Curve editor** mode toolbar. This creates an embedded pattern of the resulting curve.

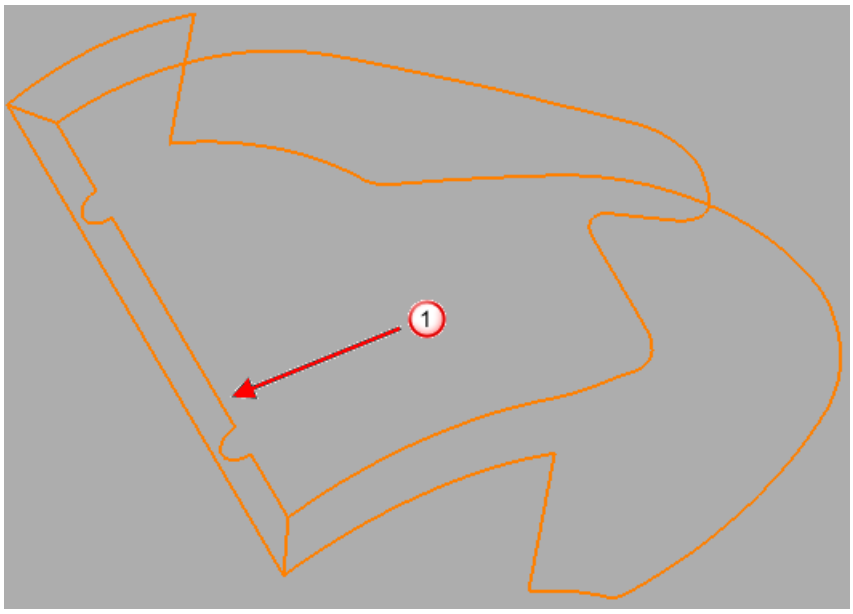


Creating a composite curve from wireframe geometry

This example shows how to create a composite curve from wireframe geometry.

This uses an edited boundary generated from the 5axis_with_holes.dgk model in the **Examples** folder.

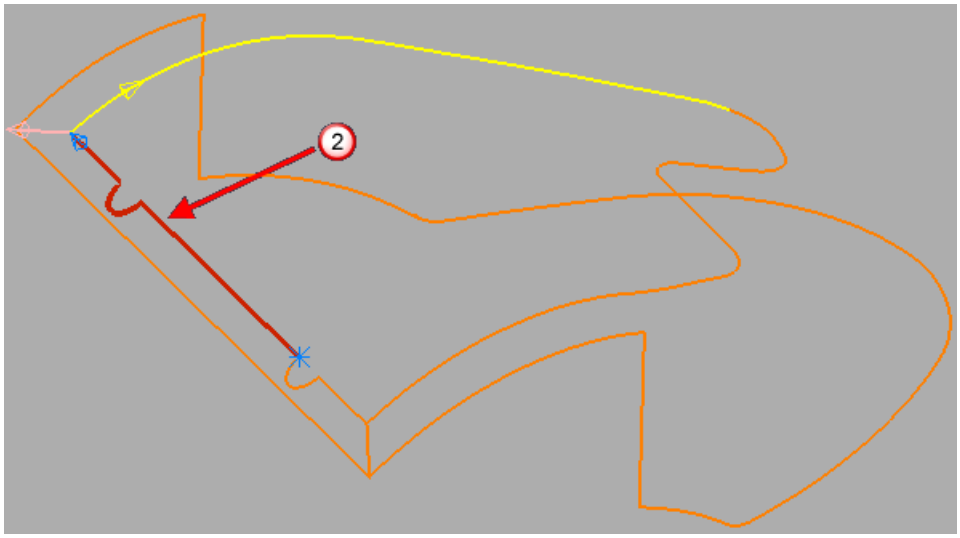
1 Double-click on the curve, near point 1, to raise the **Curve Editor** mode toolbar.



2 Click on the **Curve Creation** toolbar.

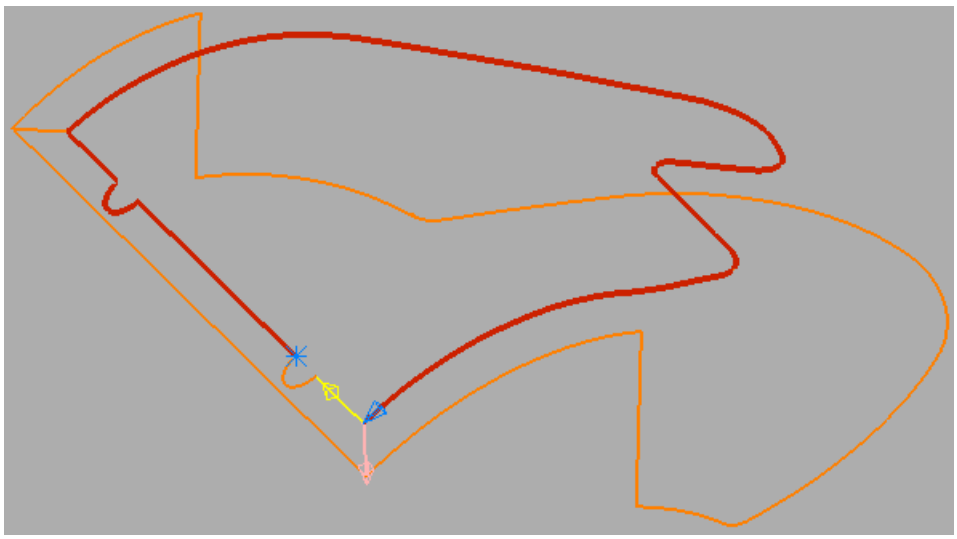
This displays the **Create Composite Curve** mode toolbar.

3 Click the curve near point 2.

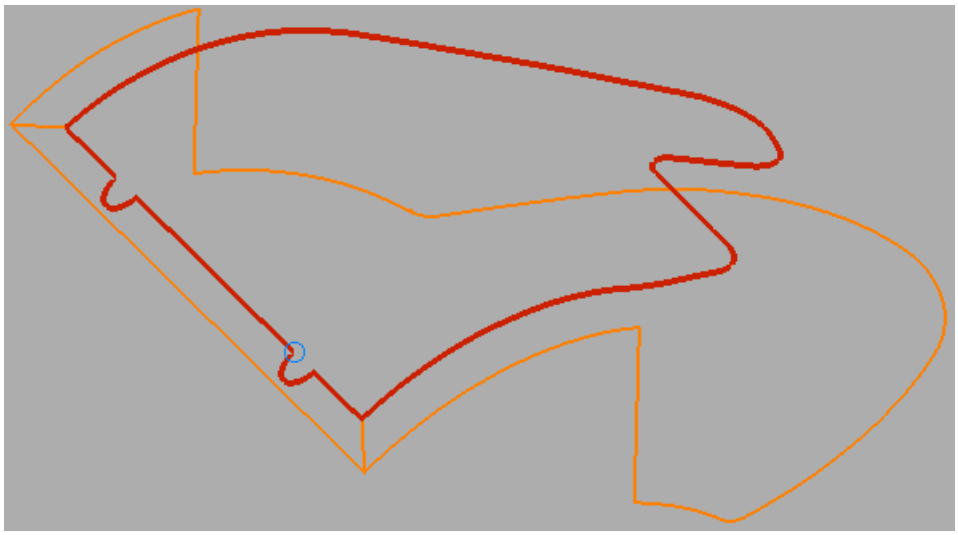


- Red indicates the selected curve.
- Pink indicates the curve that automatic sketching follows.
- Yellow indicates an alternative route.

4 Click the yellow curve.



5 Click the yellow curve.



6 Click on the **Create Composite Curve** mode toolbar.

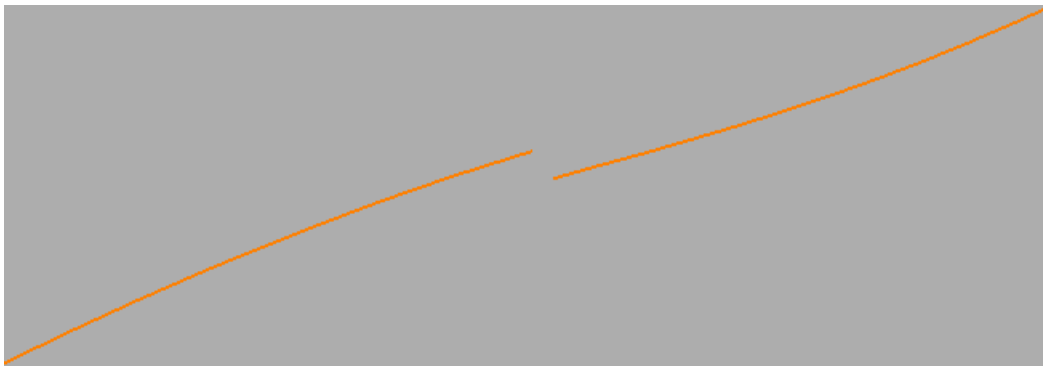
7 Click on the **Curve editor** mode toolbar.

*If, before step 3, you selected a **Direction** of , you can click on the **Create Composite Curve** mode toolbar to create the automatic path.*

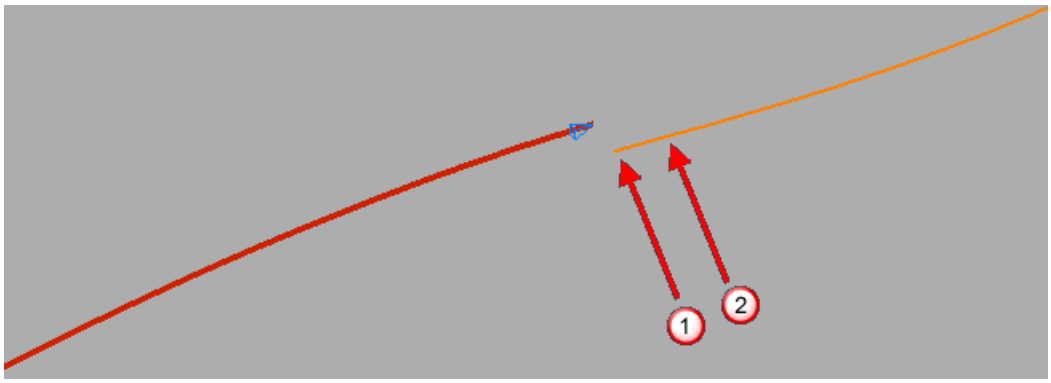
Creating a composite curve from 'gappy' wireframe geometry

When creating a composite curve you have the ability to 'jump' gaps in the geometry. For example:

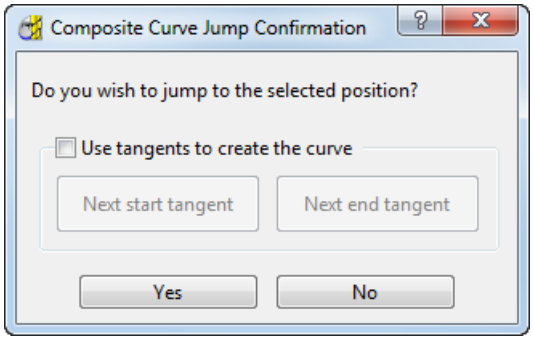
1 Starting with a curve with a gap between neighbouring segments.



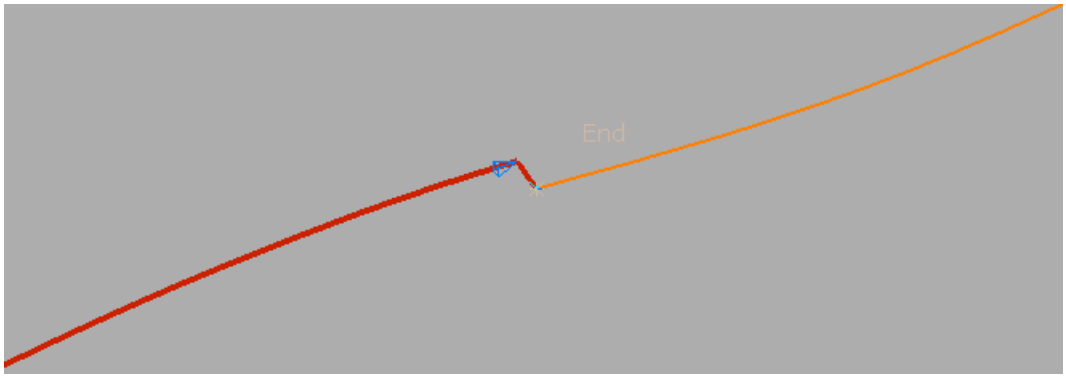
2 When creating a composite curve, PowerMILL automatically stops when it reaches the gap.



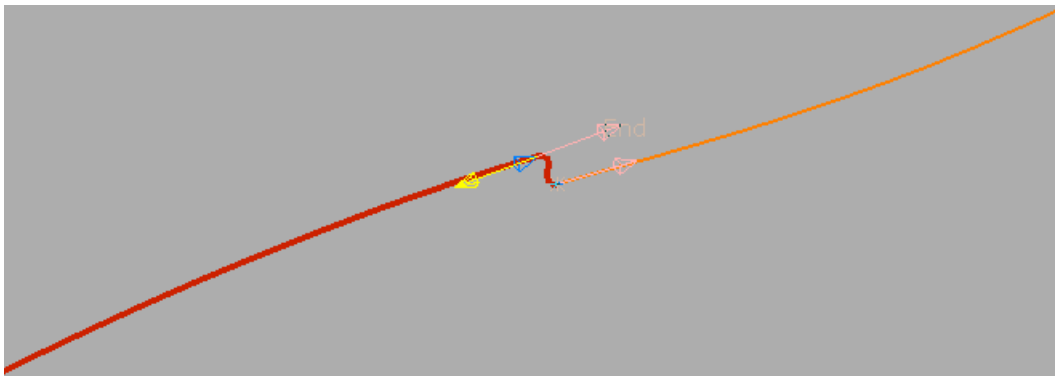
3 Click the end of the orange curve, around 1. This displays the **Composite Curve Jump Confirmation** dialog.



Deselecting the **Use tangents to create the curve** option joins the curve with a straight line.



Selecting the **Use tangents to create the curve** option joins the curves with a tangential curve.

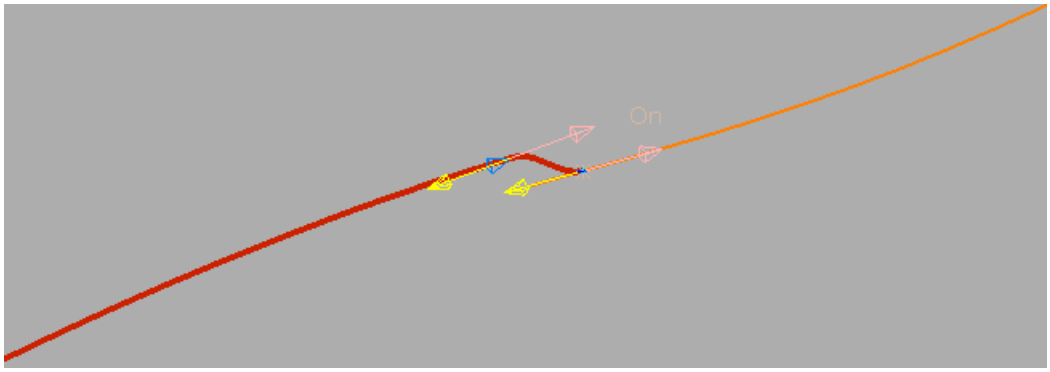


4 Click **Yes** on the **Composite Curve Jump Confirmation** dialog.

5 Click on the **Create Composite Curve** mode toolbar.

6 Click on the **Curve editor** mode toolbar.

You don't have to select the end of a curve you can select any point along it. For example, clicking around (2) in step 2 gives:

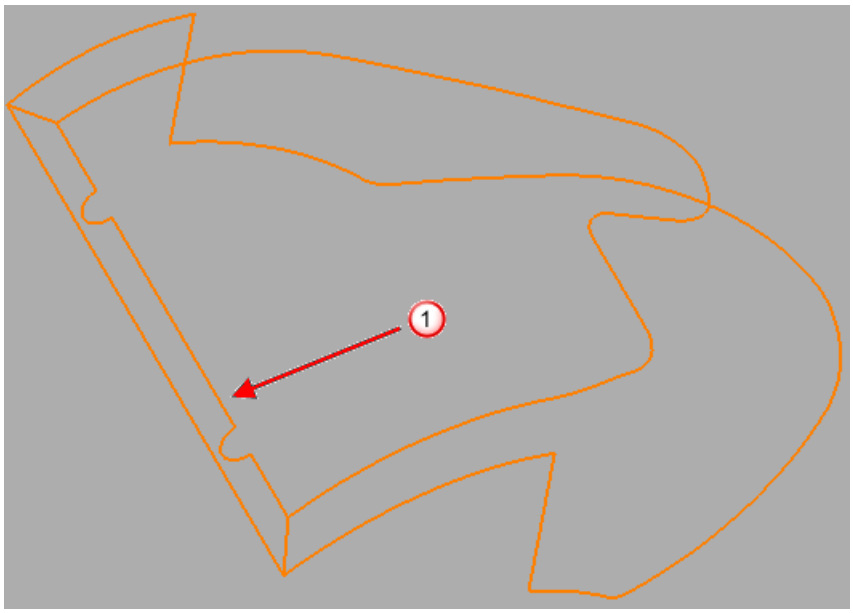


Creating a composite curve using marker points

This example shows how to create a composite curve limited by marker points. You can have up to two marker points on any composite curve.

This uses an edited boundary generated from the 5axis_with_holes.dgk model in the **Examples** folder.

1 Double-click on the curve, near point 1, to raise the **Curve Editor** mode toolbar.

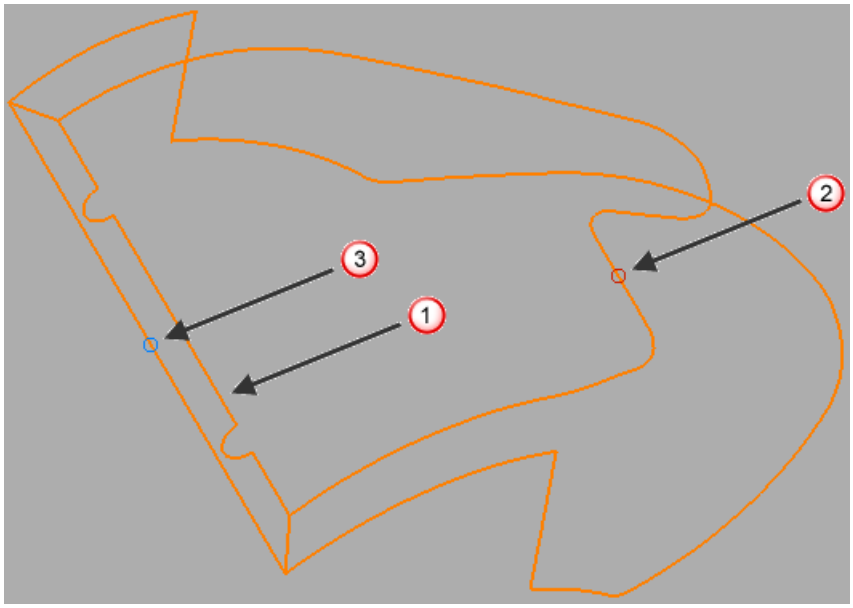


2 Click  on the **Curve Creation** toolbar.

This displays the **Create Composite Curve** mode toolbar.

3 Click the first **Marker Point**  button.

4 Click the curve near point 2. This places a red circle at the selection point.



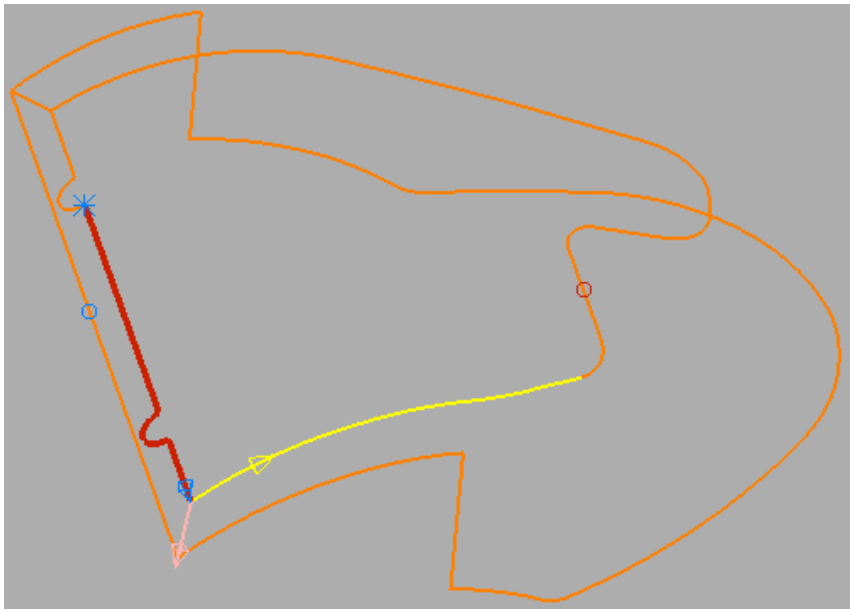
*To remove the marker point, click the first **Marker Point** button and then click the marker.*

*If the circle is blue, you have selected the second **Marker Point** button, not the first.*

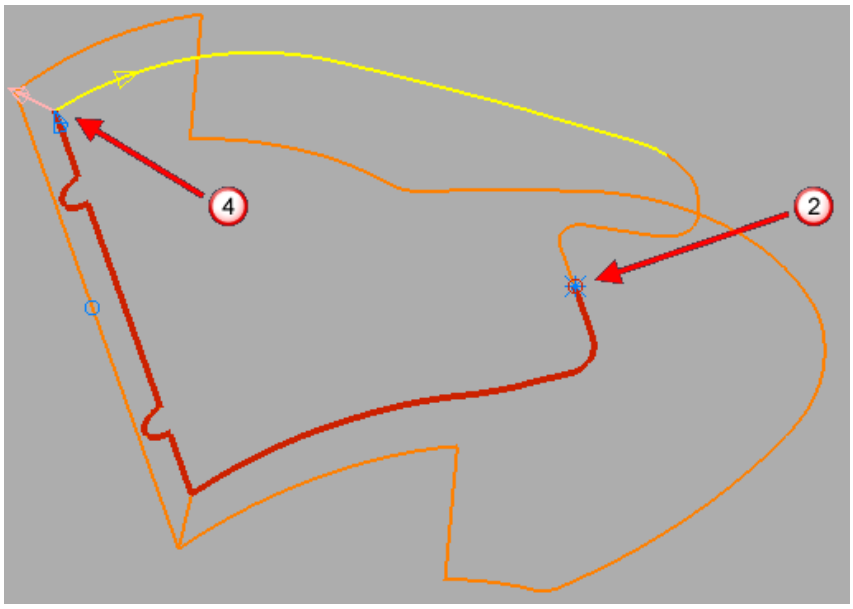
5 Click the second **Marker Point** button.

6 Click the curve near point 3. This places a blue circle at the selection point.

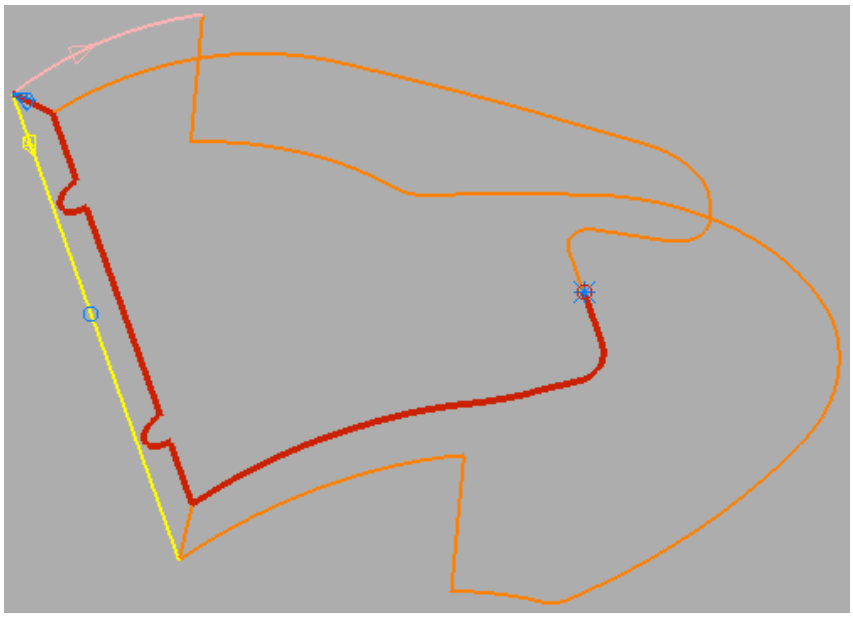
7 Click the curve near point 1.



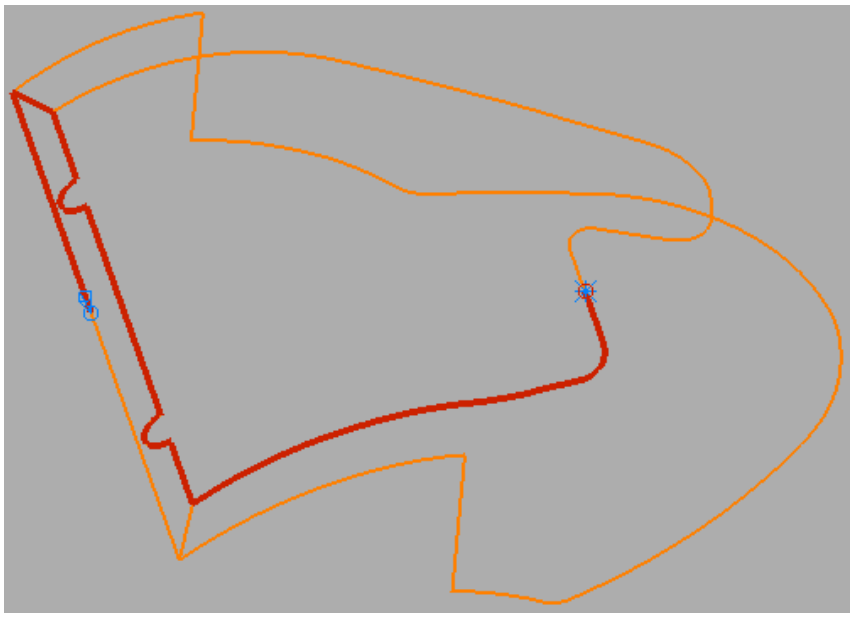
8 Click the yellow curve. The composite curve creator goes as far as the first marker point, 2. It then reverses and continues to the next branch point, 4.



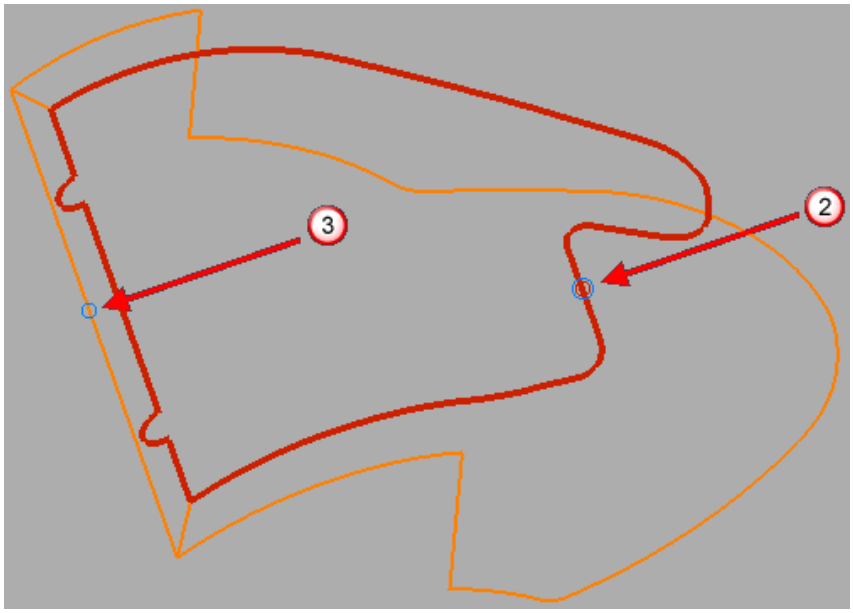
9 Click the pink curve.



10 Click the yellow curve. The composite curve creator goes as far as the second marker point and stops.



PowerMILL ignores the marker point if you choose a route that doesn't meet the marker point.



(2) The first marker point shows the start and end point of the closed composite curve.

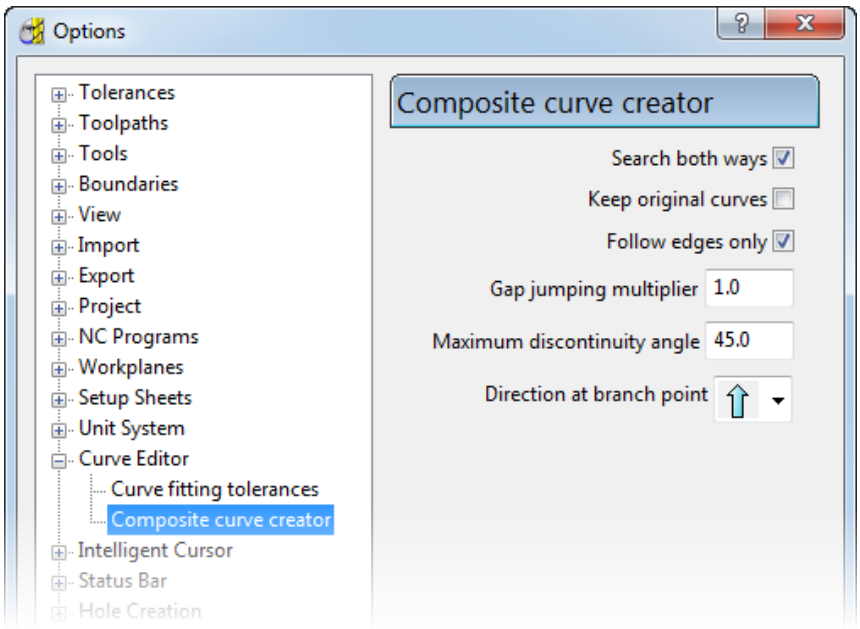
(3) The second marker point, which is ignored.

11 Click on the **Create Composite Curve** mode toolbar.

12 Click on the **Curve editor** mode toolbar.

Composite curve creator options

The **Create Composite Curve** mode toolbar has various default options which you can specify on the **Composite curve creator** page of the **Options** dialog. This is available from **Tools > Options > Curve Editor > Composite curve creator** menu.

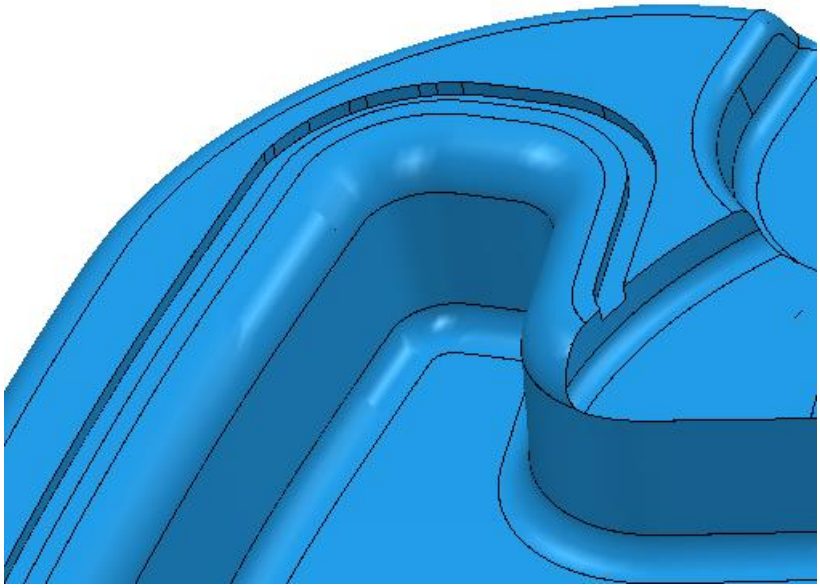


Search both ways — When selected, the **Composite Curve Creator** searches both ends of the curve to find adjacent segments. When deselected, the composite curve creator searches only the current curve to find adjacent segments.

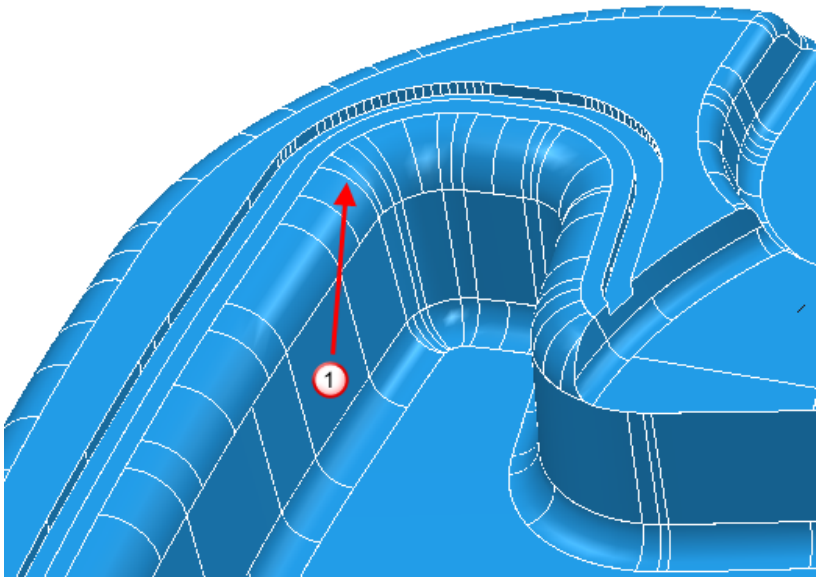
Keep original curves — When selected, keeps the individual curves used to create the composite curve as well as the composite curve. This prevents the removal of the original curves from the pattern/boundary.

Follow edges only — When selected, follows surface edges. When deselected, and the wireframe is displayed, enables you to select internal constant parameter lines within a surface as well as the surface edges. This can be useful for generating the drive curve for flowline machining.

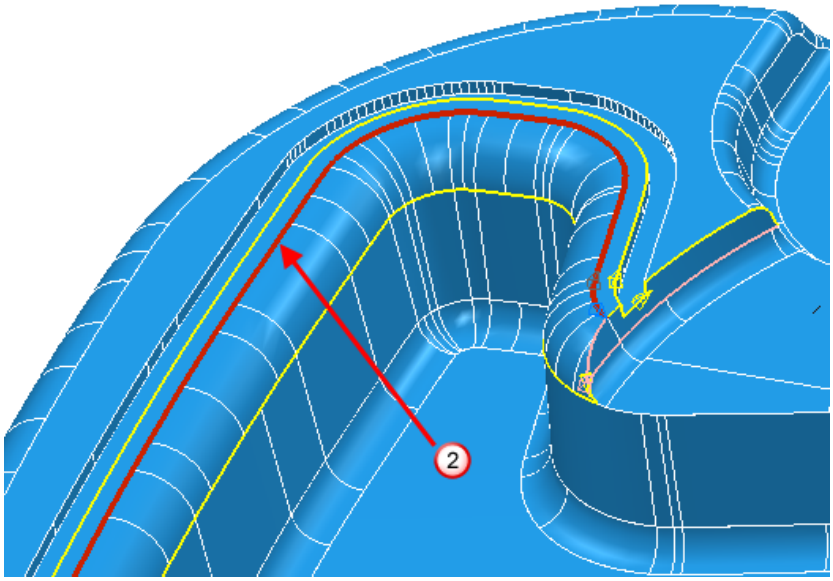
Starting with this model:



If you select the surface around 1.

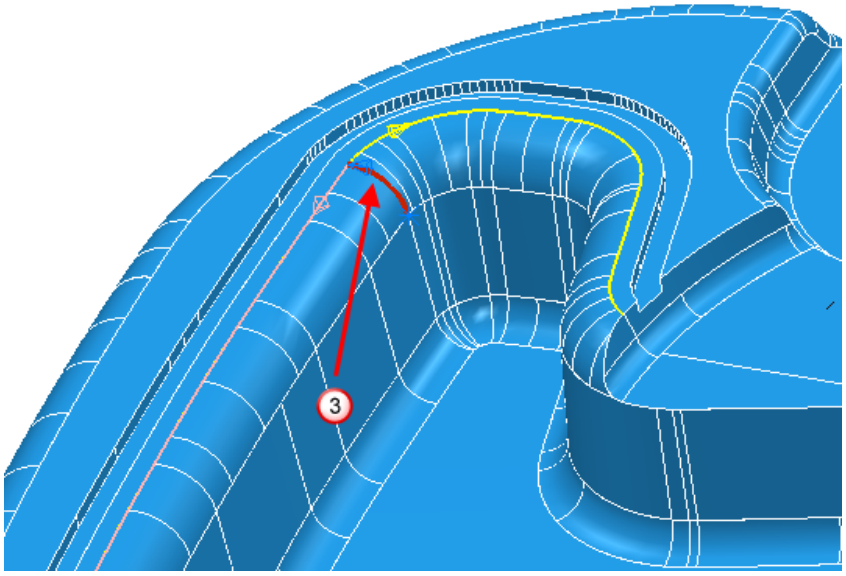


With **Follow edges only** selected:



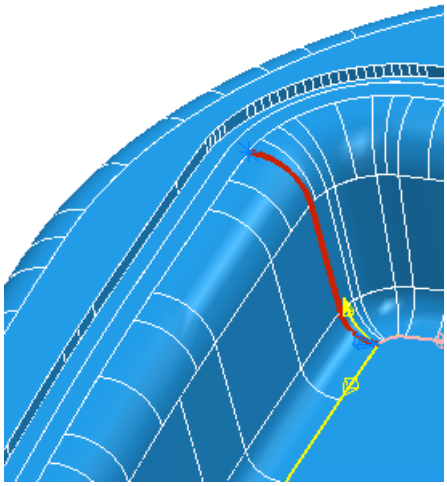
You can select surface edges (2), but not surface internals.

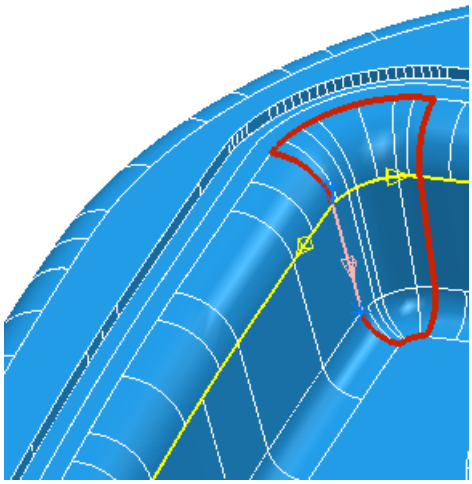
With **Follow edges only** deselected, and the wireframe displayed:



You can select surface edges or surface internals (3).

With **Follow edges only** deselected, you can create curves such as:



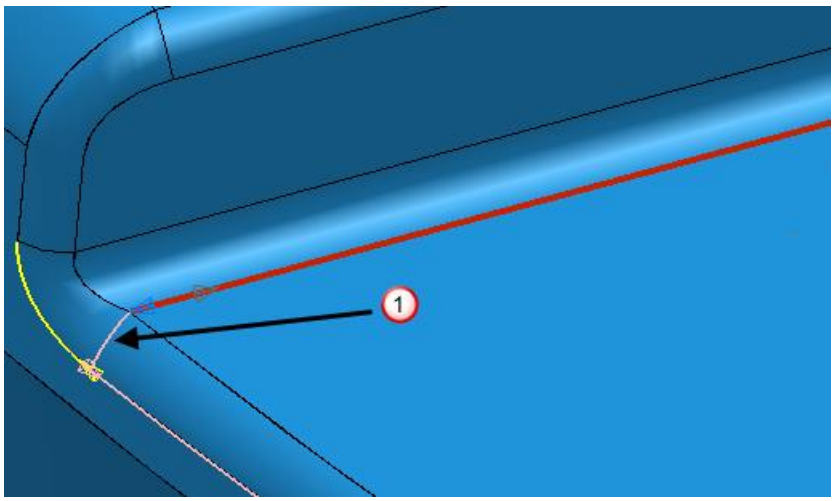


Gap jumping multiplier — Enter a value to determine if PowerMILL can automatically jump the gap. This value is multiplied by the **Closure tolerance** to give an actual value. If the gap is larger than this value the **Composite Curve Jump Confirmation** dialog is displayed. If the gap is smaller than this value PowerMILL works as if there is no gap (see page 15).

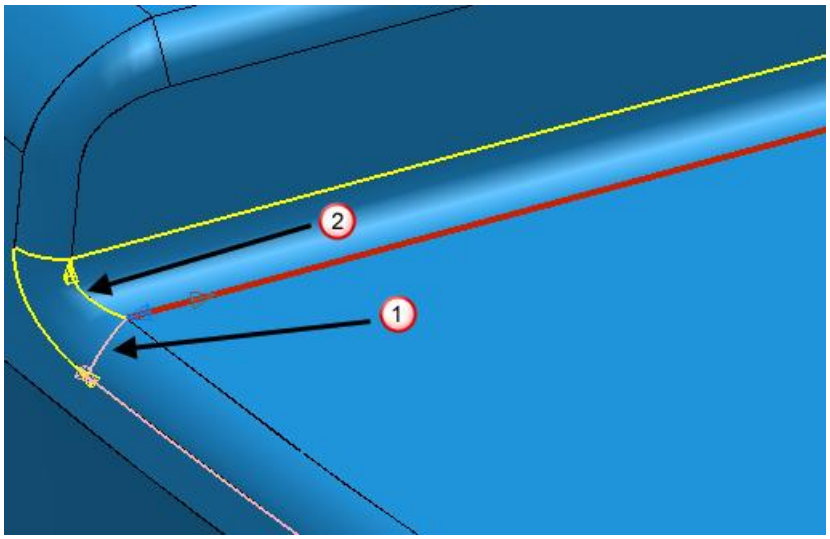
Maximum discontinuity angle — Enter the maximum angle, at a branch point, which is still considered as straight.

Changing the **Maximum discontinuity angle** effects the options presented when you select a direction of  .

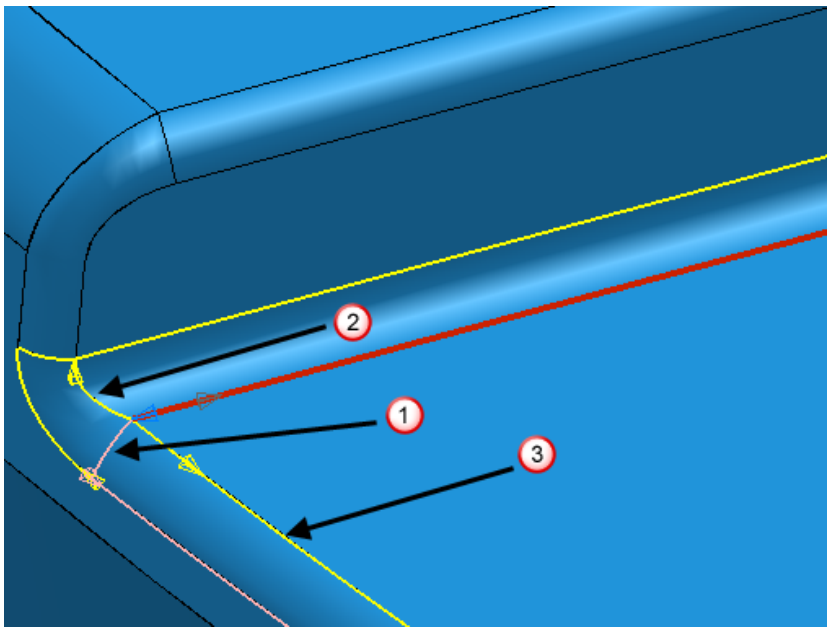
If you enter a small **Maximum discontinuity angle** the options are almost straight, in this case, (1).



If you increase the **Maximum discontinuity angle** the number of options increase, in this case, 1 and 2.



Further increasing the **Maximum discontinuity angle** adds more options, in this case, 1, 2, and 3.



Direction at branch point — Select the default direction displayed on the **Create Composite Curve** mode toolbar (see page 12).

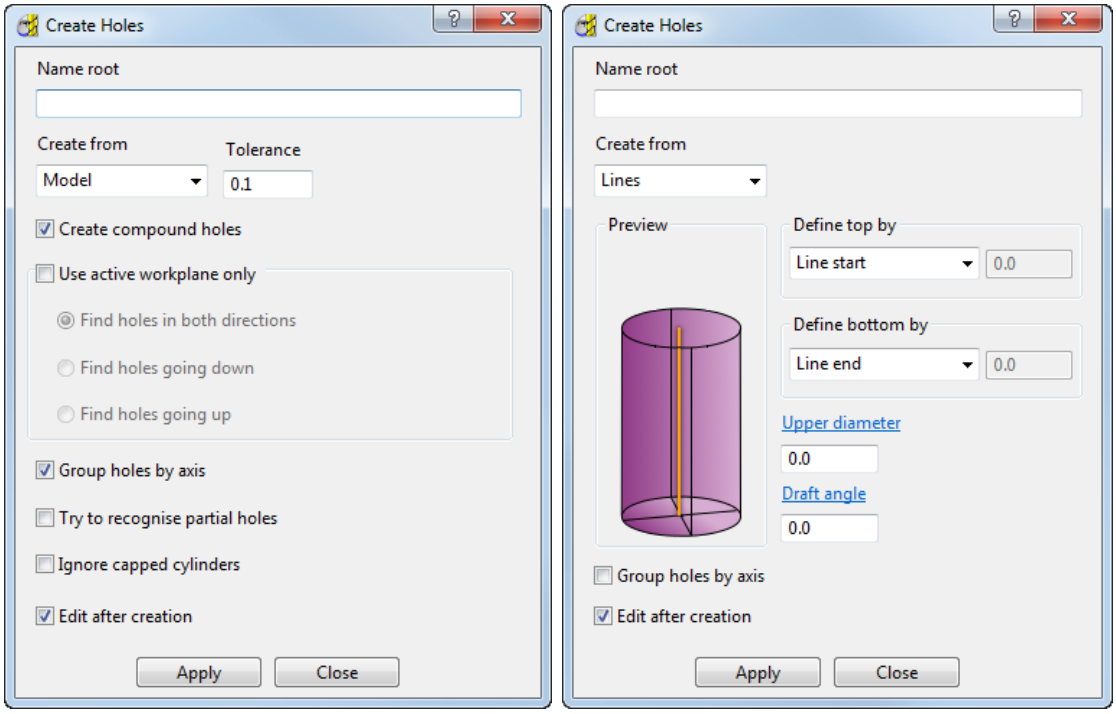
Improved hole creation and editing

There are several enhancements to hole creation and editing:

- There is an improved process for hole creation as it is significantly different to the creation of other features. There is now a separate dialog for hole creation so the **Feature** dialog no longer contains the hole specific options (see page 36).
- There is a new dialog to control hole editing, and hole intersections (see page 44).
- There are several changes to the Feature Set menu and individual feature set menu (see page 60).
- There is a new page on the **Options** dialog to specify the hole creation options (see page 63).

Create Holes dialog

Use the **Create Holes** dialog to create holes.



Display the **Create Holes** dialog by either:

- selecting **Create Holes** on the **Feature Sets** menu; or
- selecting **Create Holes** on the individual feature sets menu.

Name root — Enter a name for the hole. Each hole has the root name followed by a number. If there is no root name then the hole just has a number.

Create from — Select how to recognise holes.

Points — Select to create holes from points in the pattern. These points define the centre of the hole.

Circles — Select to create holes from circles in the model.

Model — Select to create holes from holes in the model.

Pairs — Select to create holes from pairs of circles in the model.

Curves — Select to create holes from any curve in the model that can flatten into a circle in the given workplane.

Lines — Select to create holes from lines in the pattern. These lines define the top, bottom, and axis of a hole.

Plunges — Select to create holes from the plunge moves of the active toolpath.

Normals — Select to create holes from contact-normal moves of the active toolpath.

*If you select a **Create from** of **Model**, the dialog options are different. For more information see **Create Hole > From Model dialog** (see page 40).*

Define top by — These options determine the height of the top of the hole.

Absolute — Select to define the top as an absolute Z height.

Height from Bottom — Select to define the top as a distance from the bottom of the feature/hole.

Maximum Curve Z — Select to define the top as the maximum Z height of the selected curve, location ①. All features use a curve (bits of model or pattern) as the input data. These curves have a minimum and a maximum Z value, which you can use for feature definition. This is useful for hole definition where you have a set of Z positions that represent the top or bottom of the hole.

Minimum Curve Z — Select to define the top as the minimum Z height of the selected curve, location ②.

① Maximum Z
② Minimum Z

Curve centre — Select to define the top as halfway between the top and bottom of the curve (midway between 1 and 2).

Top of Block — Select to define the top as the height of the top of the block.

Line Start — Select to define the top as the start of the line.

*This option is available only if you selecte a **Use of Lines**.*

10.0 — Enter the height of the top of the feature.

Define bottom by — These options determine the height of the bottom of the hole.

Absolute — Select to define the bottom as an absolute Z height.

Depth from Top — Select to define the bottom as a distance from the top of the feature/hole.

Maximum Curve Z — Select to define the bottom as the maximum Z height of the selected curve, location 1. All features use a curve (bits of model or pattern) as the input data. These curves have a minimum and a maximum Z value, which you can use for feature definition. This is useful for hole definition where you have a set of Z positions that represent the top or bottom of the hole.

Minimum Curve Z — Select to define the bottom as the minimum Z height of the selected curve, location 2.

1 Maximum Z
2 Minimum Z

Curve centre — Select to define the bottom as half way between the top and bottom of the curve. Midway between 1 and 2.

Bottom of Block — Select to define the bottom as the height of the bottom of the block.

Line End — Select to define the bottom as the end of the line.

*This option is available only if you selecte a **Use of Lines**.*

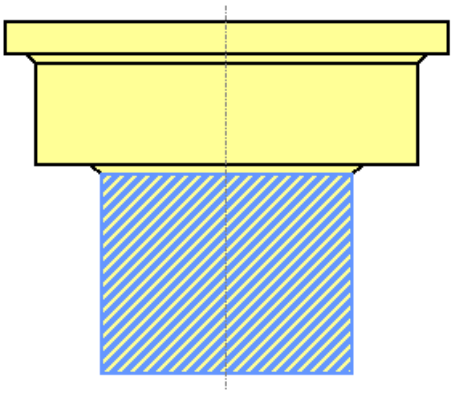
10.0 — Enter the height of the bottom of the feature.

Upper diameter — Enter the upper diameter of the hole.

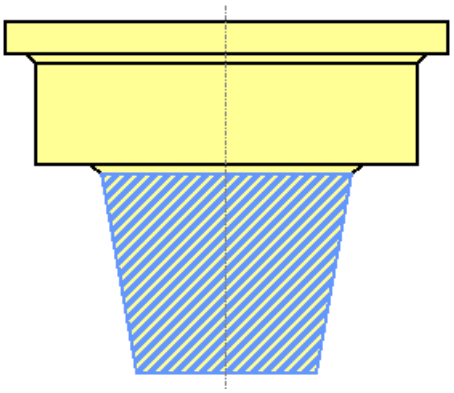
*The **Upper diameter** is the same as the **Lower diameter** unless the hole has a **Draft angle**.*

Draft angle — Enter the draft angle of the hole.

Draft angle of **0**:

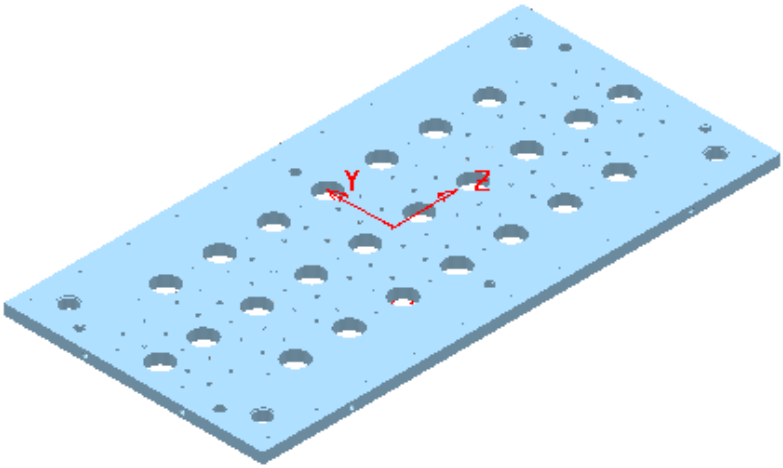


Draft angle of **10**:

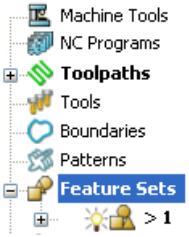


Group holes by axis — When selected, sorts the holes into feature sets by workplane. In this case, your machine tool must have3+2-axis drilling capability. When deselected, places all the holes in one feature set. In this case, your machine tool must have multi-axis drilling capability.

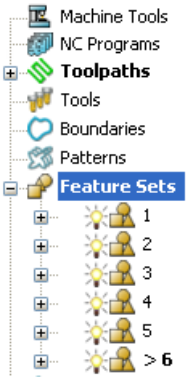
This example uses the RetainerPlate.dgk model. It has an active workplane, and the whole model is selected:



Deselecting **Group holes by axis** gives:



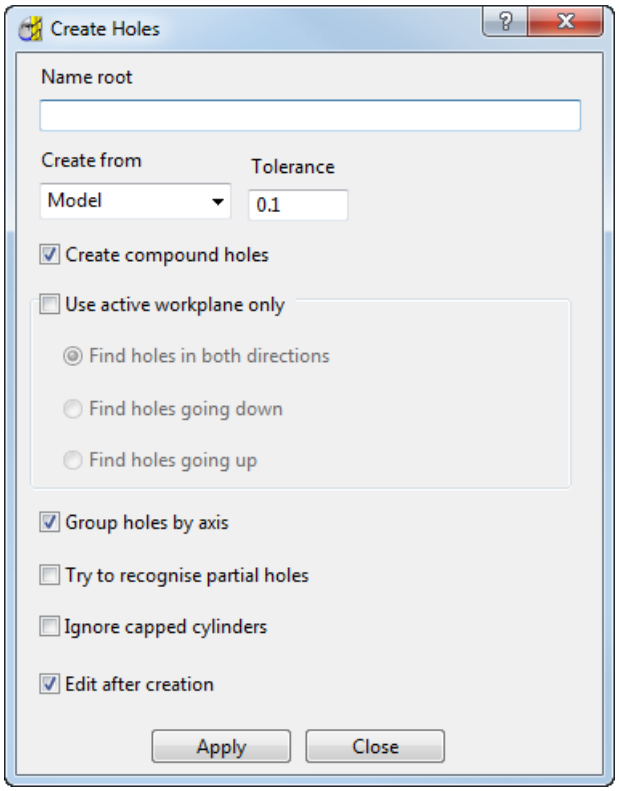
Selecting **Group holes by axis** gives:



Edit after creation — When selected, clicking **Apply** creates the holes and then displays the **Edit Hole** dialog (see page 44).

Create Holes from model dialog

Use the **Create Holes** dialog with a **Create from** of **Model** to create holes from holes in the model.



To display this dialog, select **Create Holes** on the **Feature Sets** menu.

Name root — Enter a name for the hole. Each hole has the root name followed by a number. If there is no root name the hole just has a number.

Create from — Select how to recognise holes.

Points — Select to create holes from points in the pattern. These points define the centre of the hole.

Circles — Select to create holes from circles in the model.

Model — Select to create holes from holes in the model.

Pairs — Select to create holes from pairs of circles in the model.

Curves — Select to create holes from any curve in the model that can flatten into a circle in the given workplane.

Lines — Select to create holes from lines in the pattern. These lines define the top, bottom, and axis of a hole.

Plunges — Select to create holes from the plunge moves of the active toolpath.

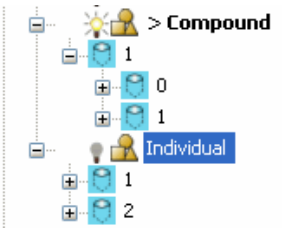
Normals — Select to create holes from contact-normal moves of the active toolpath.

Tolerance — Enter the tolerance used to create holes.

Create compound holes — When selected, creates one compound hole containing several components. When deselected, creates several individual holes (superimposed on each other).

A compound hole is one hole containing several components; in this case, a hole named **1** with components **0** and **1**.

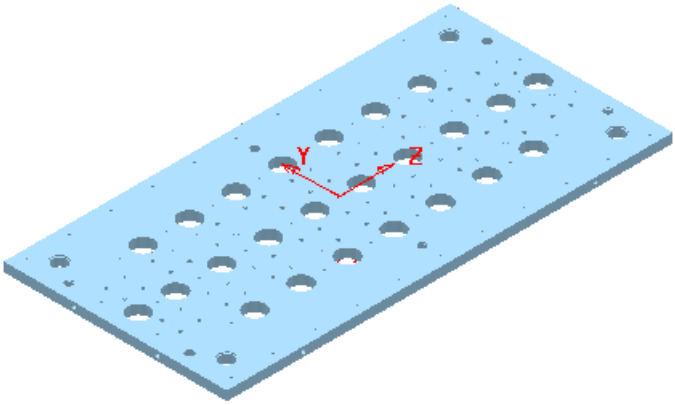
If you create them as individual holes, there are two holes named **1** and **2**.



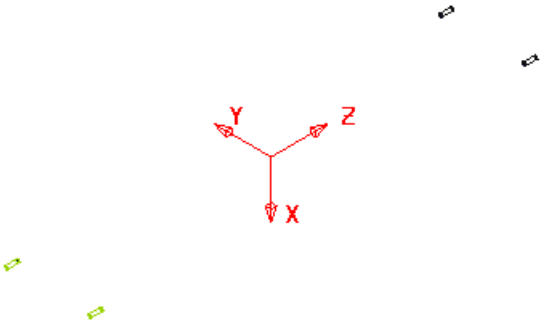
For more information, see **Compound Holes**.

Use active workplane only — Select to create only the holes that have a Z axis that aligns with the Z axis of the active workplane.

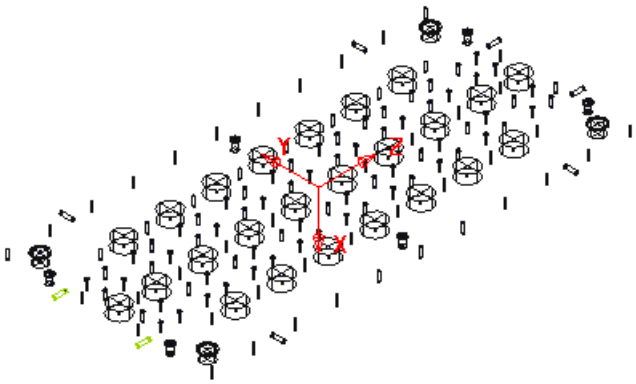
This example uses the RetainerPlate.dgk model. It has an active workplane, and the whole model is selected:



Selecting the **Use active workplane only** gives:



Deselecting **Use active workplane only** gives:



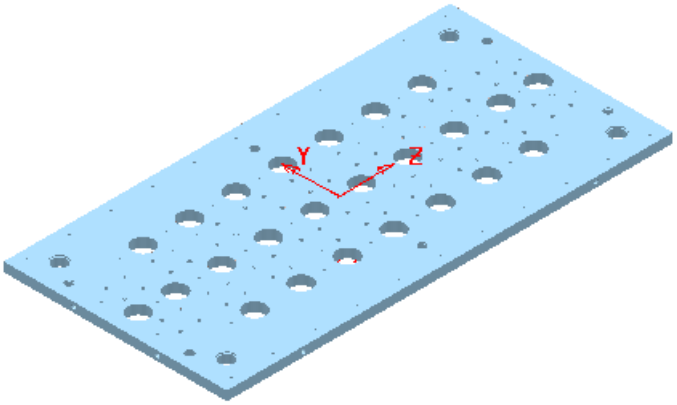
Find holes in both directions — When **Group by hole axis** is selected, places holes from both directions into a single multi-axis feature set. When **Group by hole axis** is deselected, splits the holes into two feature sets (one for up and one for down).

Find holes going up — Places the holes going up into a single feature set.

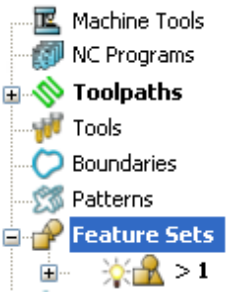
Find holes going down — Places holes going down into a single feature set.

Group holes by axis — When selected, sorts the holes into feature sets by workplane. In this case, your machine tool must have3+2-axis drilling capability. When deselected, places all the holes in one feature set. In this case, your machine tool must have multi-axis drilling capability.

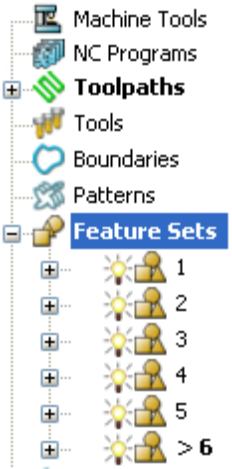
This example uses the RetainerPlate.dgk model. It has an active workplane, and the whole model is selected:



Deselecting **Group holes by axis** gives:



Selecting **Group holes by axis** gives:



Try to recognise partial holes — Select to enable the creation of holes from a model that has poorly, or incompletely defined, holes. For example, creating a hole from a pair of arcs.

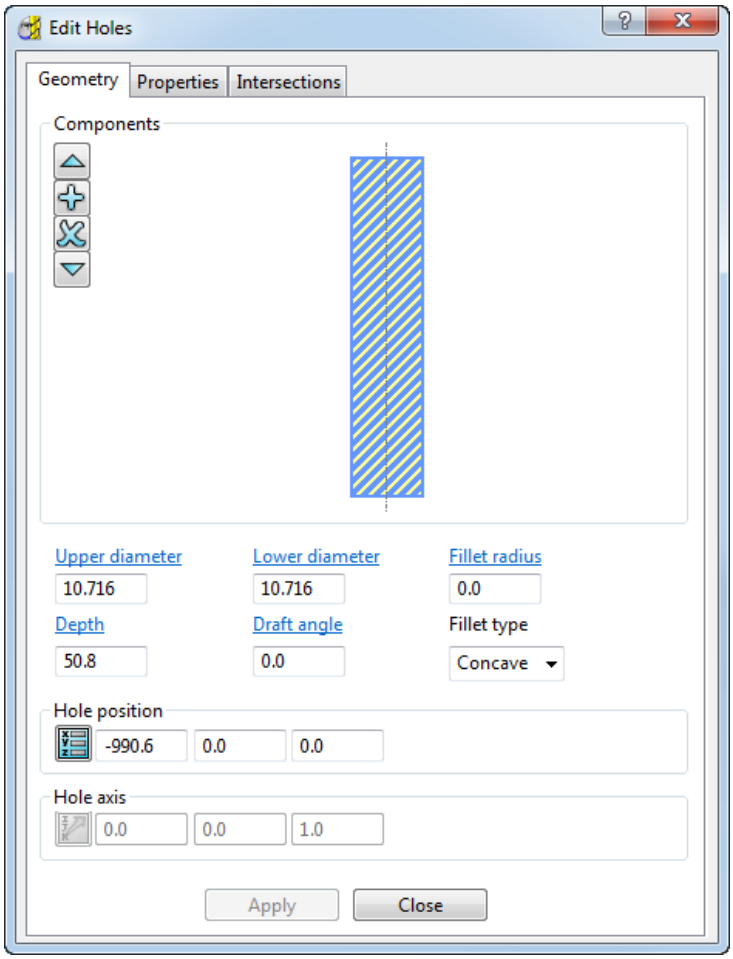
Use this option as a last resort because it may find many unwanted (or unexpected) holes.

Ignore holes with no open ends — When selected, creates holes which have at least one open end, and ignores holes with two closed ends.

Edit after creation — When selected, clicking **Apply** creates the holes and then displays the **Edit Hole** dialog (see page 44).

Edit Holes dialog

Use the **Edit Holes** dialog to edit the geometry, properties or hole intersections.



Display the **Edit Holes** dialog by either:

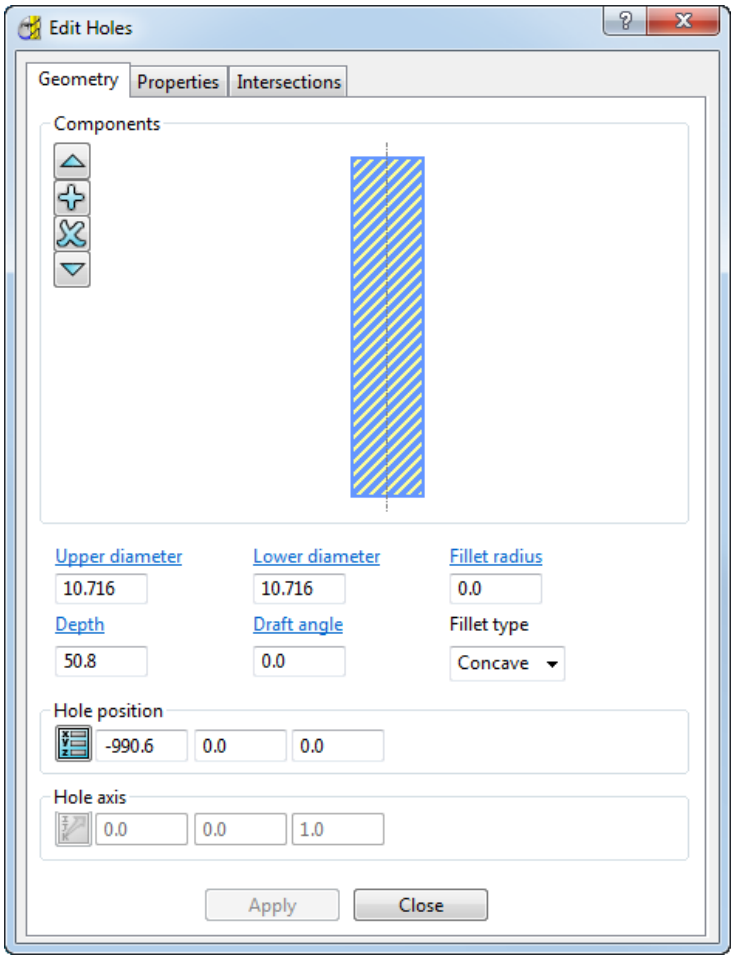
- selecting **Edit Holes** on the **Feature Sets** menu; or
- selecting the **Edit after creation** option on the **Create Hole** dialog (see page 36).

There are three tabs associated with this dialog:

- Geometry — Select to edit the size of the hole, or hole component (see page 45).
- Properties — Select to edit the hole metadata, for example name and colour (see page 57).
- Intersection — Select to delete hole intersections (see page 58).

Edit Holes > Geometry

The **Geometry** tab on the **Edit Hole** dialog edits a component of a hole (see page 44).



Move Up — Click to move the selected component up one position.

Add component — Click to add a component. The new component is the same size as the selected component. You can then edit its size in the lower half of the dialog.

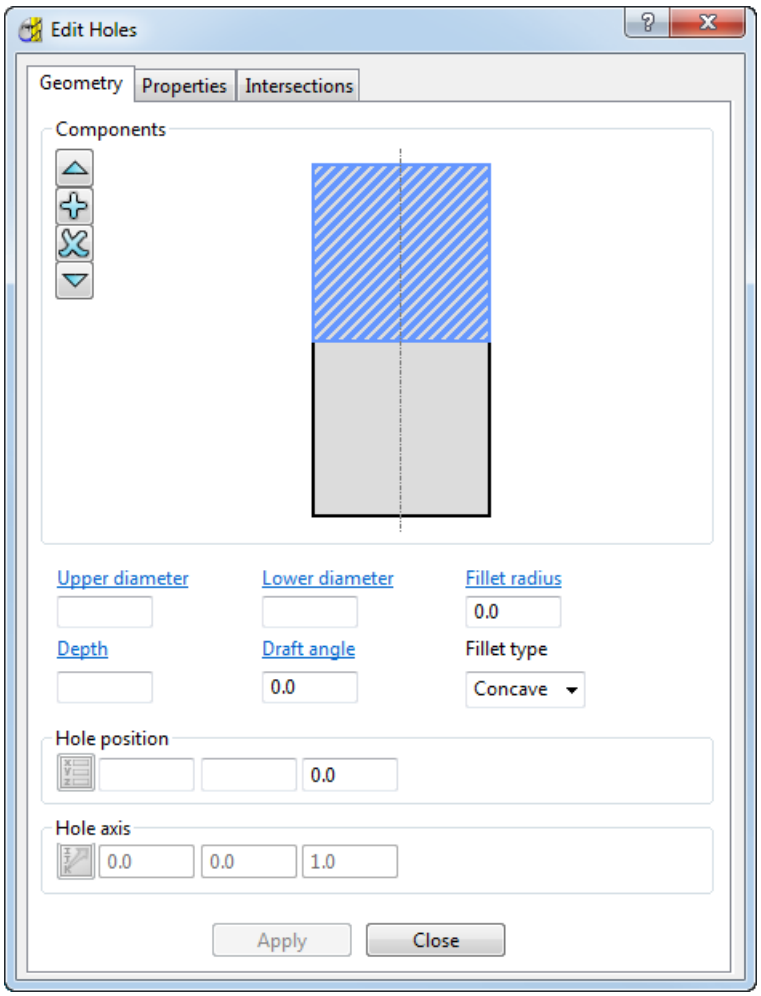
Delete Selected — Click to delete the selected component.

Move Down — Click to move the selected component down one position.

The graphic displays the hole you are editing. For compound holes, the currently editable component is cross-hatched blue. To edit a different component, select it in the graphics pane.

You can use the keyboard arrows (↑ ↓) to select the relevant component which can be useful when selecting small components.

If you select multiple holes that are the same size and have the same components, you can edit them all together. If you select multiple holes that differ, the differing components are grey. However, you can still move them and edit the values in the lower part of the dialog.



Where the selected holes have the same value, the value is displayed (for example **Draft angle**).

Where the selected holes have differing values, the field is empty (for example **Upper diameter**).

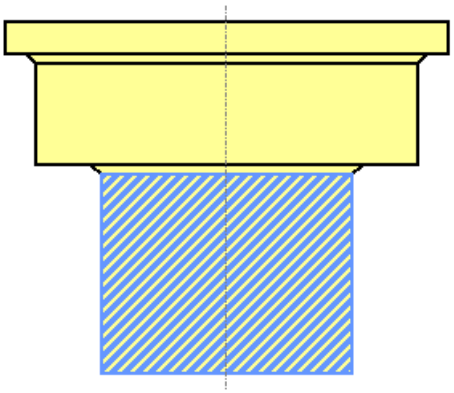
Upper diameter — Enter the diameter at the top of the current (cross-hatched component. This must be greater than or equal to the **Lower Diameter**.

Depth — Enter the vertical height of the component.

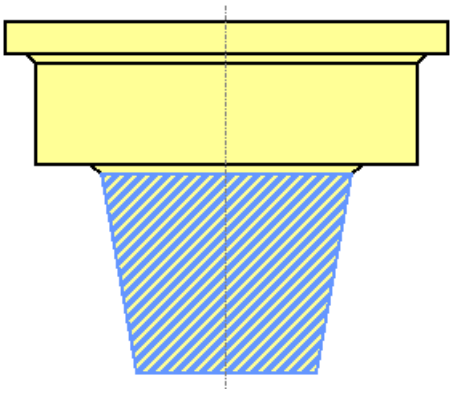
Lower diameter — Enter the diameter at the bottom of the current (cross-hatched) component. This must be less than or equal to the **Upper Diameter**.

Draft angle — Enter the draft angle of the hole.

Draft angle of **0**:

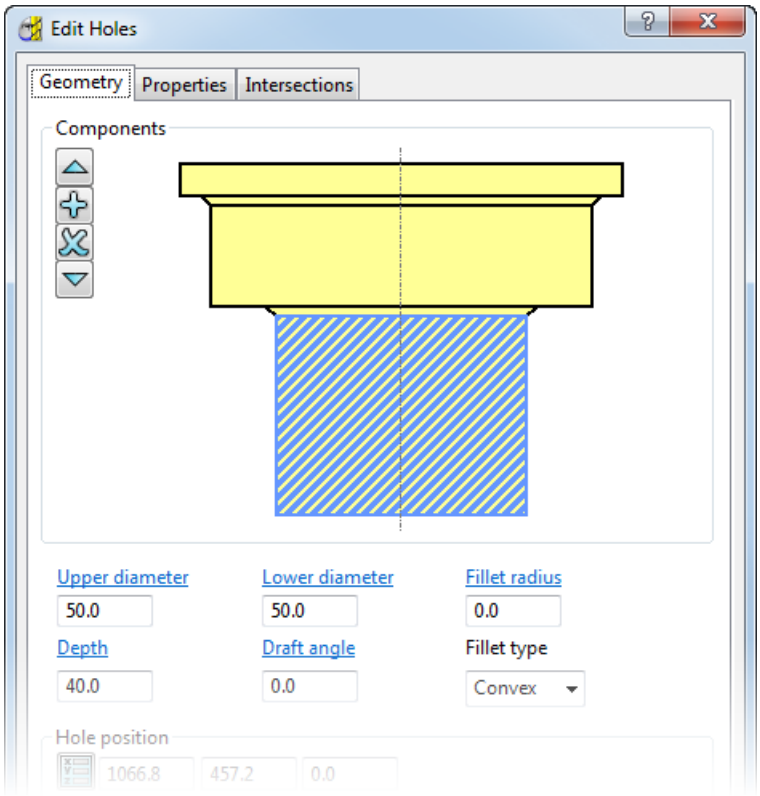


Draft angle of **10**:

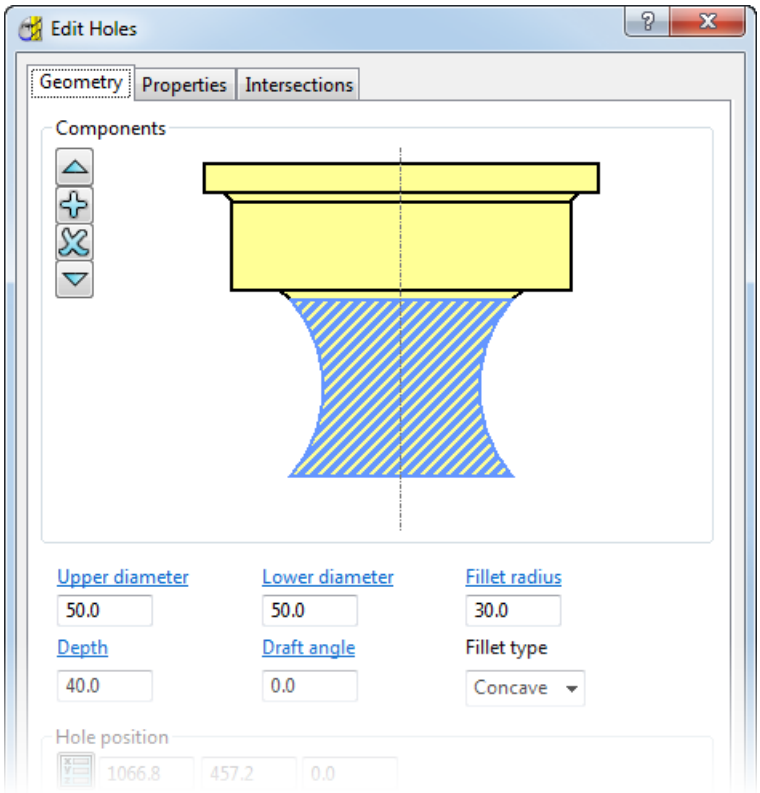


Fillet radius — Enter the radius.

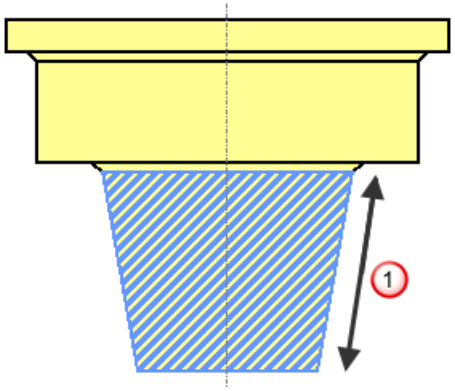
A **Fillet radius** of **0** gives a standard hole component with no fillet:



Entering a **Fillet radius** gives:



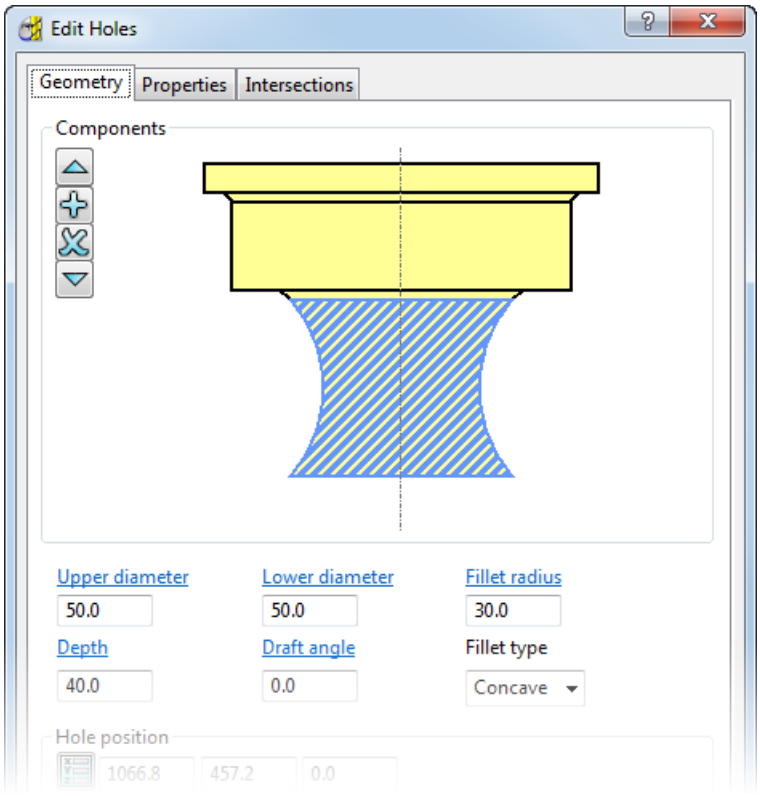
*The minimum fillet radius has a diameter equal to the length (1) of the hole component. The length of the component is the same as the **Depth** for a hole with no draft angle.*



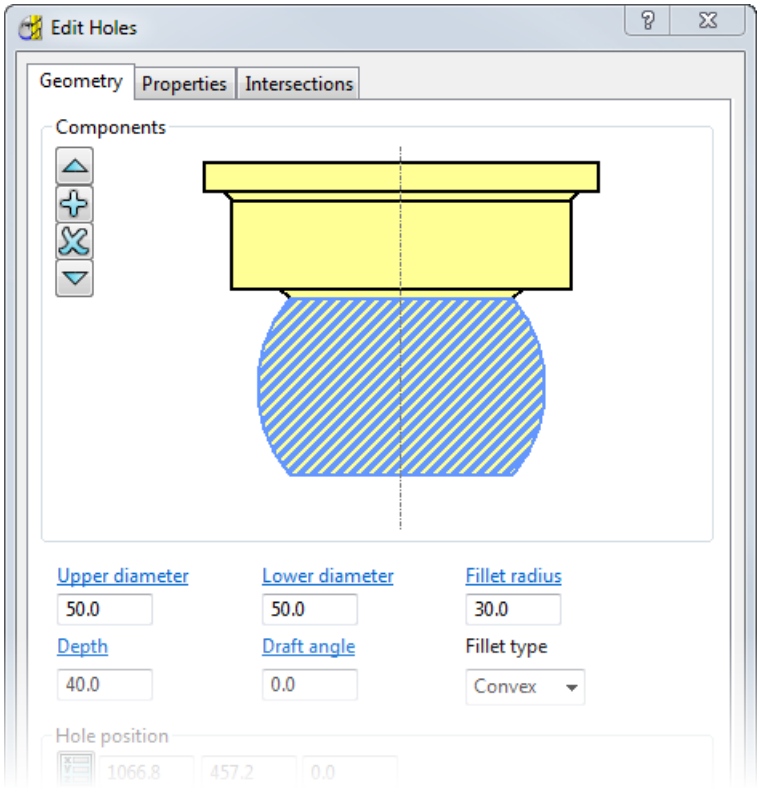
The diameter of the fillet must be at least as large as the length of the component.

Fillet type — Select either a **Concave** or **Convex** fillet.

Fillet type of **Concave**:



Fillet type of **Convex**:



Hole position — Enter the x, y, z coordinates of the centre of the hole.

 — Click to display the **Position** dialog. Use the dialog to manually enter coordinates and locate items in the graphics window.

Hole axis — Enter the orientation of the Z axis of the hole.

 — Click to display the **Direction** dialog, which enables you to edit the direction of an item.

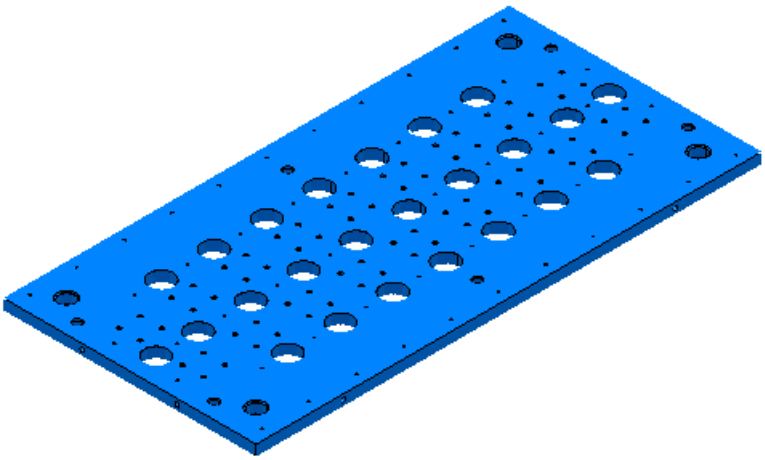
 — When displayed, hover over  to see more information.

 *If you edit any of these fields you must click **Apply** for them to take effect.*

For an example on using this dialog, see Editing automatically identified holes (see page 50).

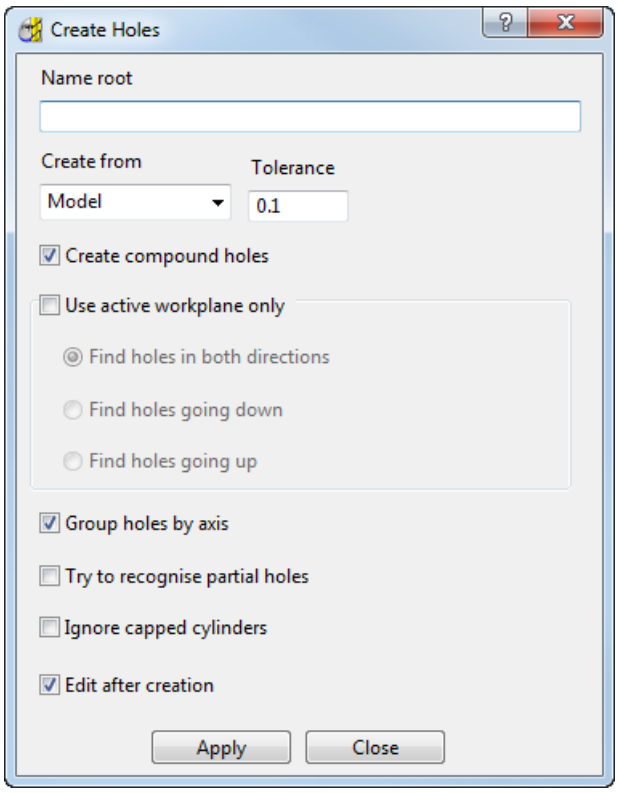
Editing automatically identified holes

This example shows how to edit holes generated automatically from the **Create Holes** option. It uses the RetainerPlate.dgk model in the **Examples** folder.



1. Select part of the model, including some of the holes.
2. From the **Feature Sets** menu, select **Create Holes**.

   This displays the **Create Holes** dialog.



3 On the **Create Holes** dialog:

a Select a **Create from** of **Model**.

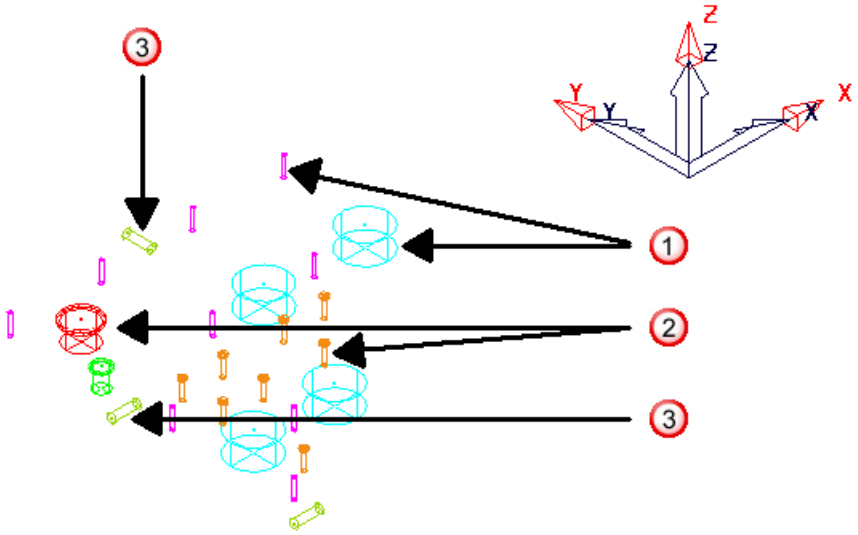
b Select **Create compound holes**.

c Deselect **Group holes by axis**.

d Select **Edit after creation**.

e Click **Apply**.

PowerMILL recognises the selected holes.

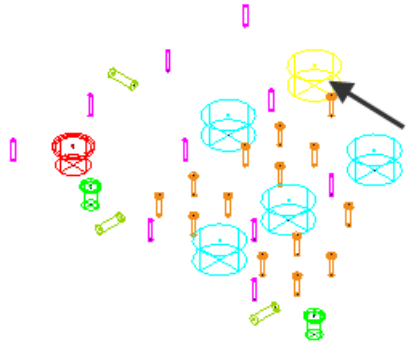


1 Standard holes with the Z axis of the hole aligned with the workplane Z axis.

2 Compound holes with the Z axis of the hole aligned with the workplane Z axis.

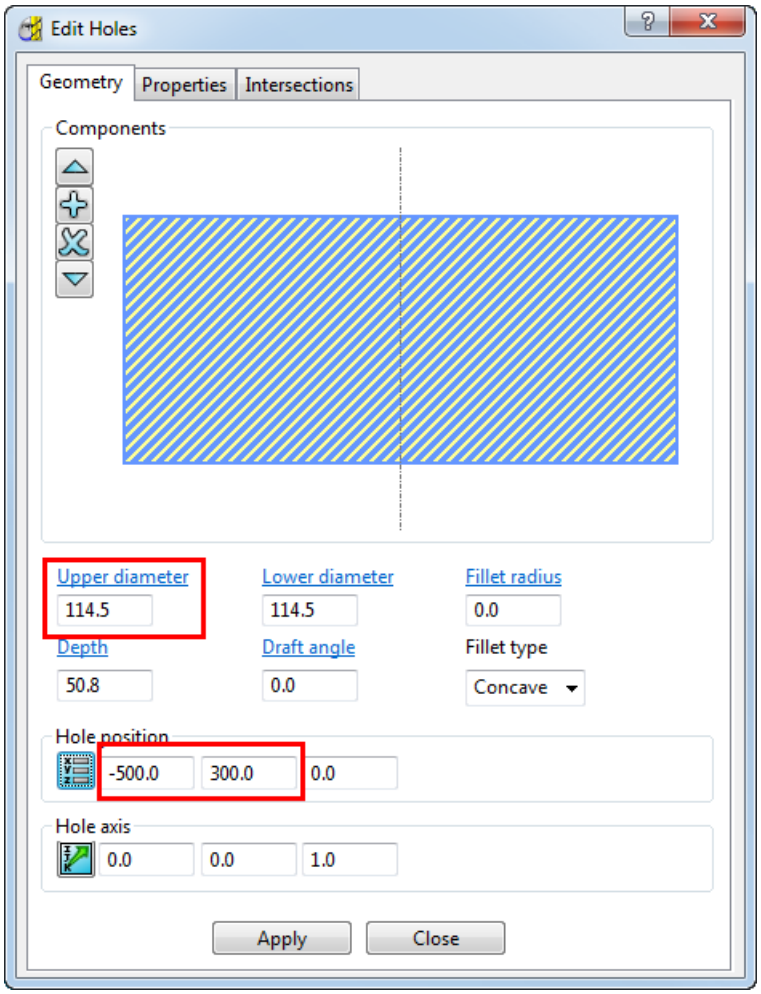
3 Standard holes with the Z axis of the hole not aligned with the workplane Z axis.

4 Select the hole you want to edit.



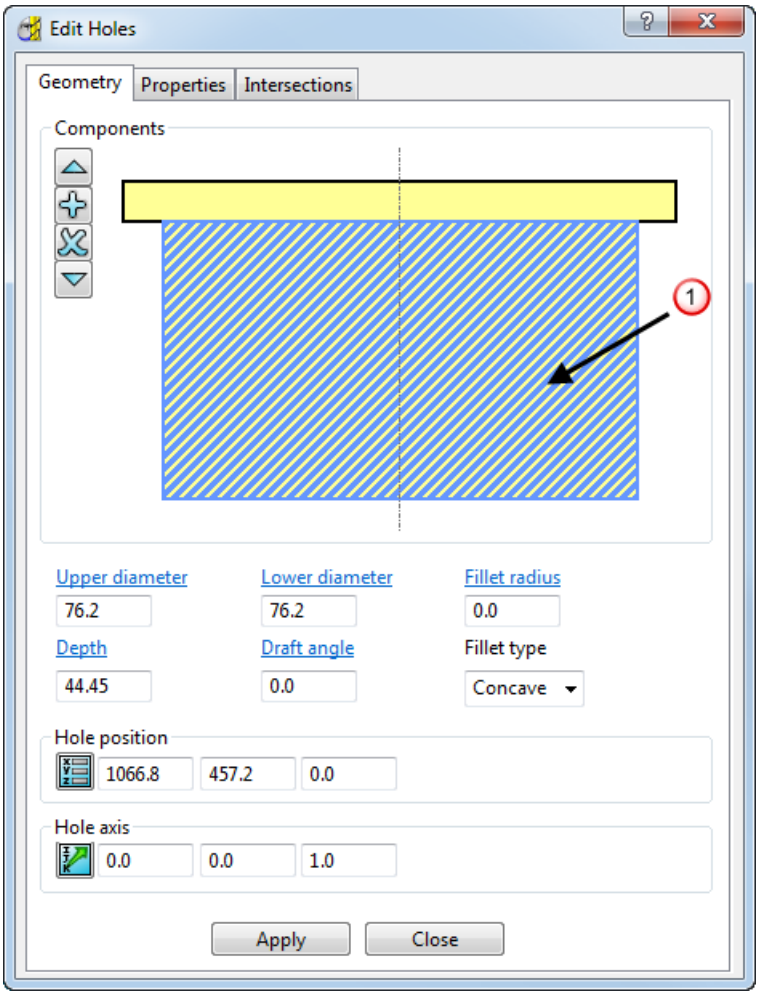
You can also edit holes graphically.

5 On the **Edit Holes** dialog, edit the required fields, in this case, the **Upper Diameter**, and **X** and **Y** coordinates.



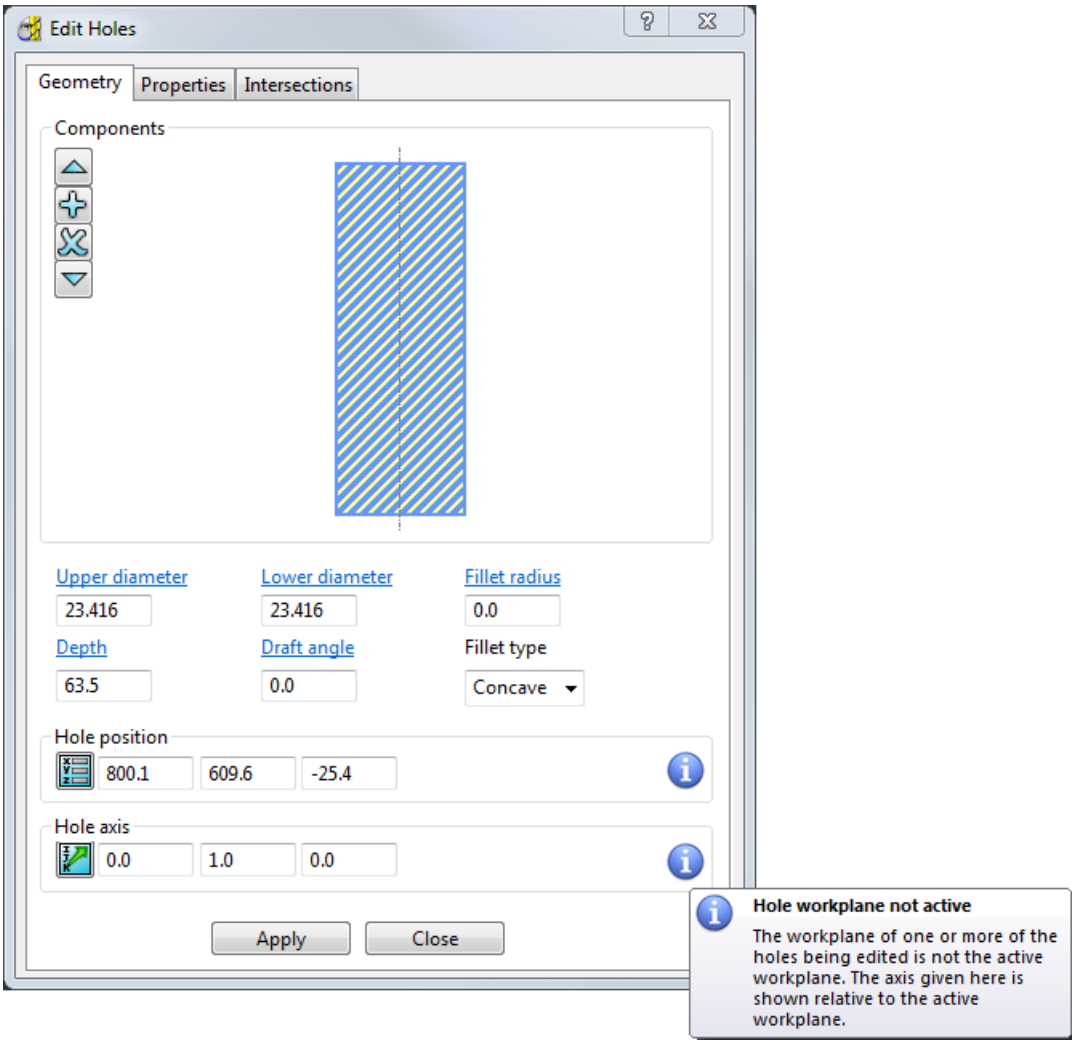
*Altering the **Upper Diameter** automatically updates the **Lower Diameter**.*

If you edit a compound hole, select the relevant component in the graphical area of the dialog (the selected component is cross-hatched), and then edit the appropriate values.



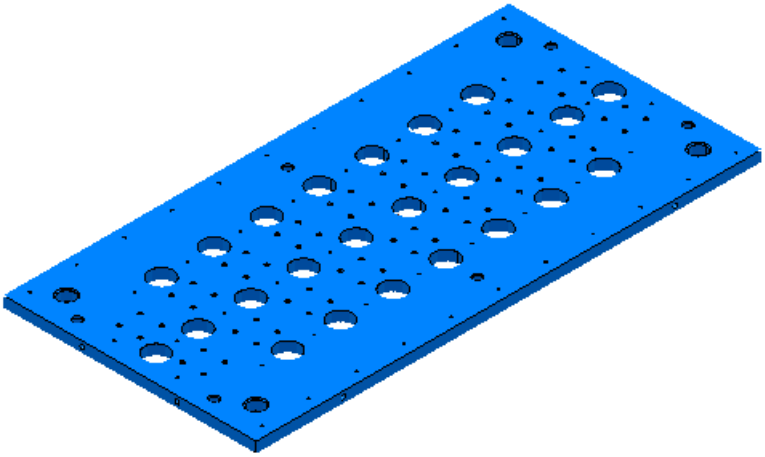
1 the selected component.

When you select a hole where the Z axis of the hole is not aligned with the workplane Z axis, the ***Hole position*** *coordinates are shown relative to the active workplane rather than the workplane of the hole. In this case* *is added to the dialog. Hovering over* *displays more information.*



Creating and editing compound holes

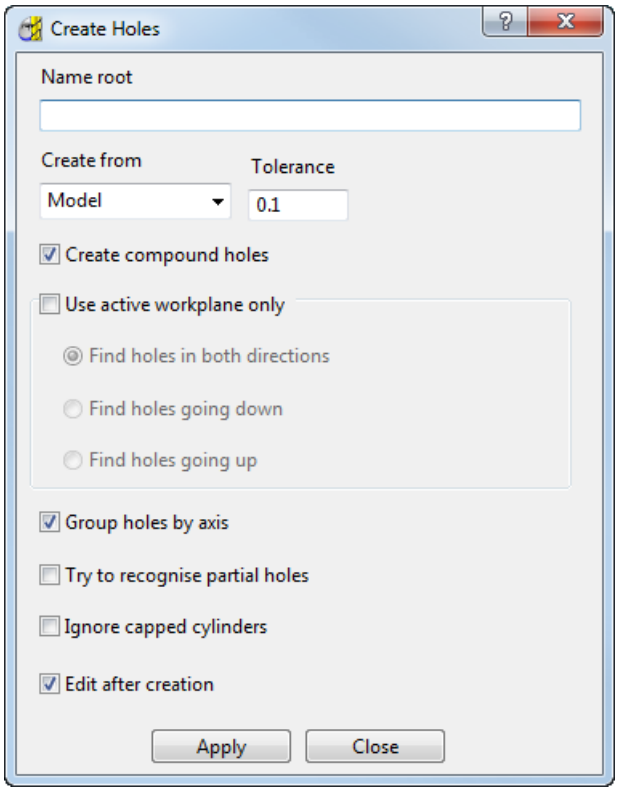
This example shows how to edit holes generated automatically from the **Create Holes** option. It uses the RetainerPlate.dgk model in the **Examples** folder.



1 Select part of the model, including some of the holes.

2 From the **Feature Sets** menu, select **Create Holes**.

This displays the **Create Holes** dialog.



3 On the **Create Holes** dialog:

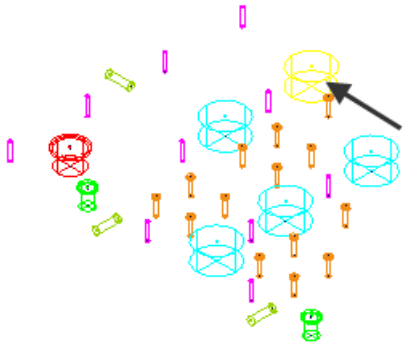
- **a** Select a **Create from** of **Model**.
- **b** Select **Create compound holes**.
- **c** Deselect **Group holes by axis**.

d Select **Edit after creation**.

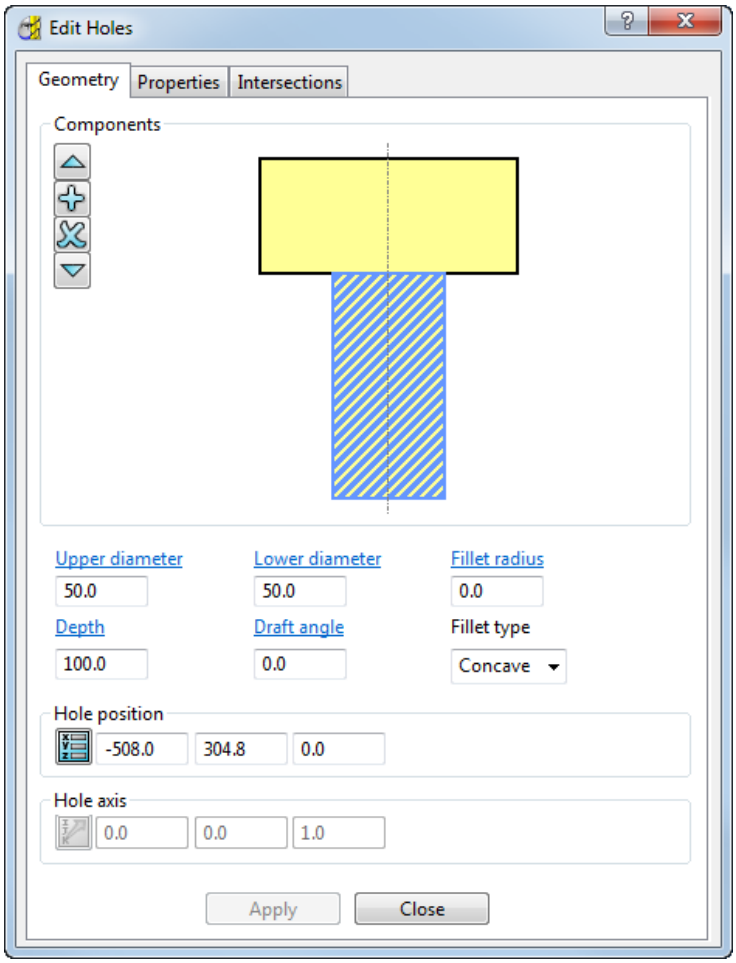
e Click **Apply**.

PowerMILL recognises the selected holes.

4 Select the hole you want to edit.



5 On the **Edit Holes** dialog:



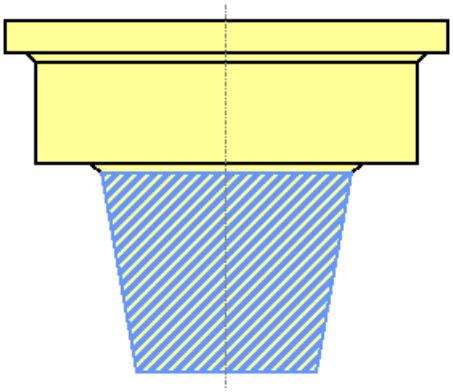
a Click  to add a second component.

b Select the lower component.

c Enter an **Upper diameter** of **50**.

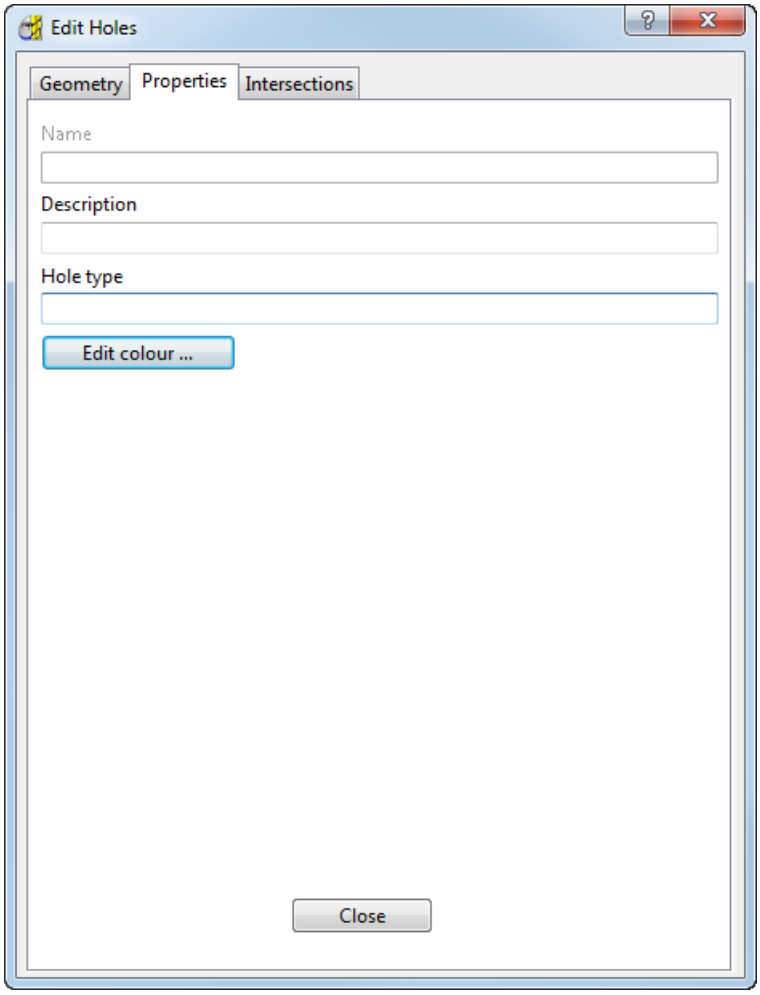
d Enter a **Depth** of **100**.

You can use this dialog to build up a more complex compound hole, such as:



Edit Holes > Properties

Use the **Properties** tab on the **Edit Hole** dialog to edit the hole metadata, for example, name and colour (see page 44).



Name — Enter the name of the active hole.

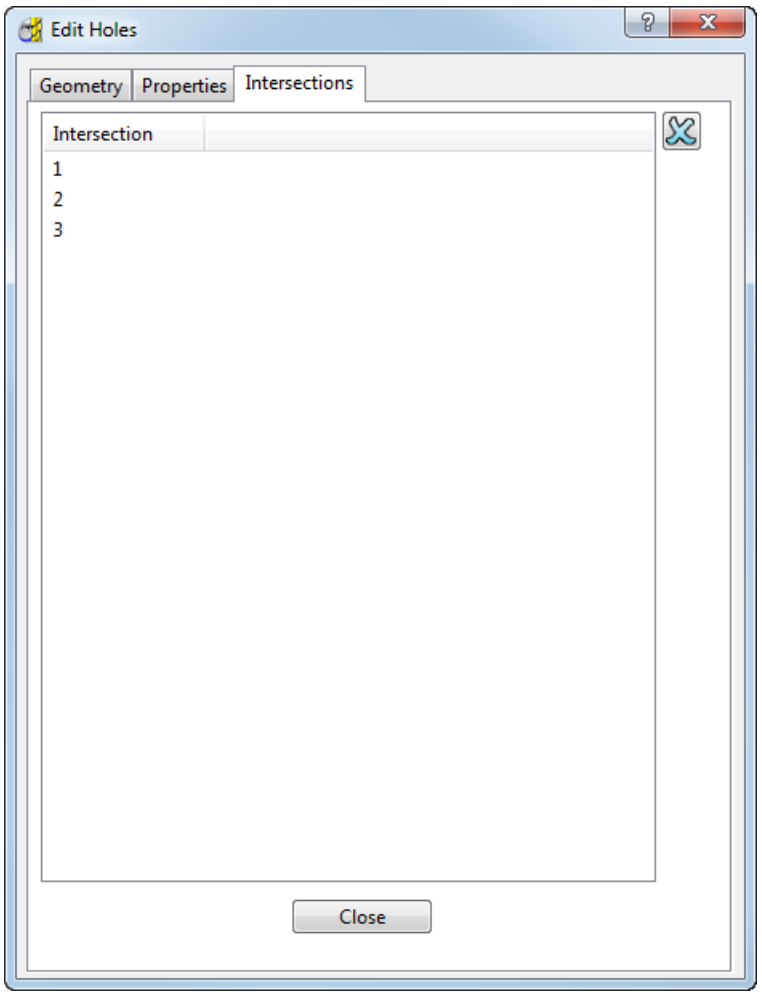
Description — Enter a description. You can use this description as a selection criteria when selecting holes or for drilling methods (you can give a set of holes a description then state that the method only applies to holes with that description). For more information, see Select Features.

Hole Type — Enter a description. You can use this description as a selection criteria when selecting holes or for drilling methods (you can give a set of holes a description then state that the method only applies to holes with that description). For more information, see Select Features.

Edit Colour — Click to display the standard Windows **Color** dialog. Select the relevant colour and click **OK** to close the dialog. Use to change the colour of the selected feature or hole.

Edit Holes > Intersection

The **Intersections** tab on the **Edit Holes** dialog enables you to delete hole intersections (see page 44).



*To use this you must first identify the hole intersections (from the **Feature Sets** context menu, select **Find Hole Intersections**).*

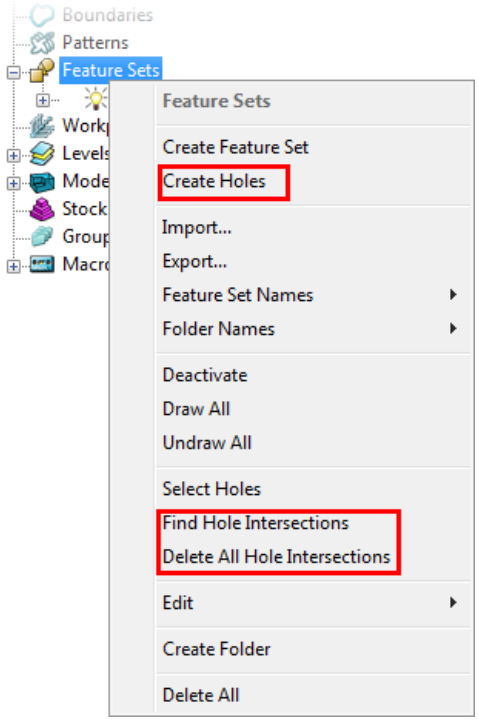
The left side of the dialog displays a list of the intersections of the selected hole.

Delete — Click to delete the intersections selected in the left side of the dialog.

For more information, see Reducing feed rates at hole intersections or Deleting a hole intersection.

Enhancements to the Feature Set menu

There are several changes to the **Feature Set** menu.



The **Feature Set Editor** option is now on the **Edit** menu.

There are several new options:

Create Holes this displays the settings used to create the hole. Hole creation now uses a different dialog to feature creation (see page 36). This combines the **Create Feature Sets** option (with a **Type** of **Hole**) and the **Recognise Holes in Model** option.

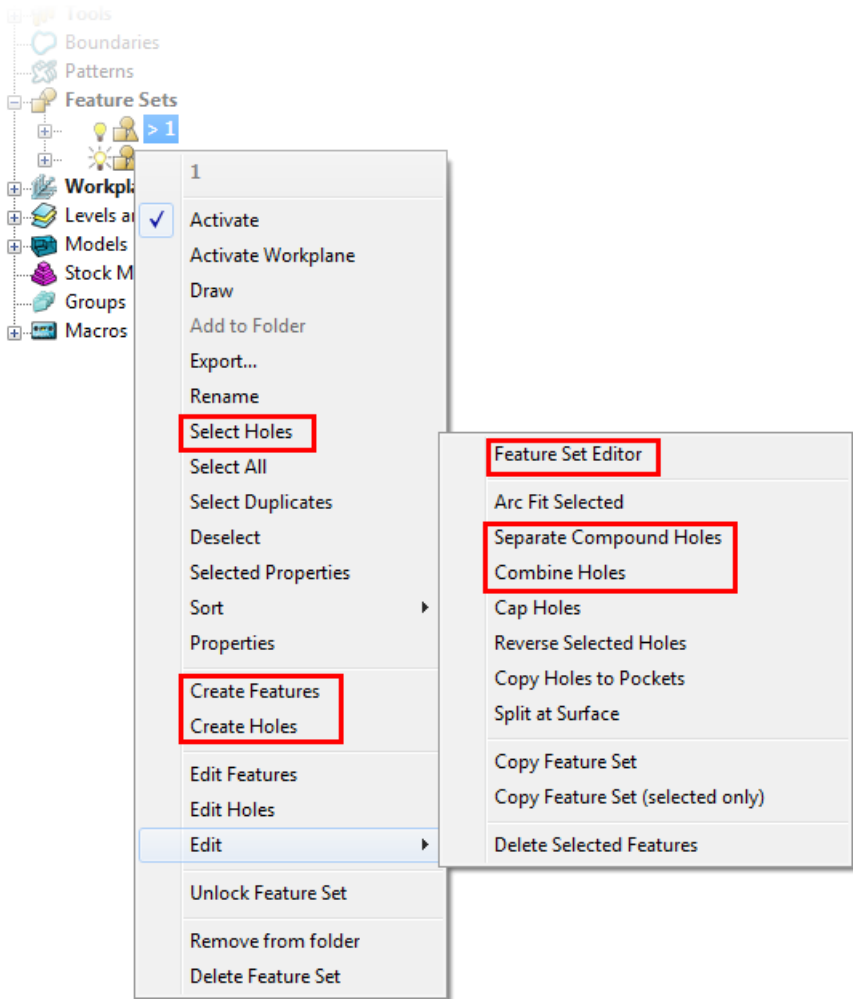
Find Hole Intersections and **Delete All Hole Intersections** are lower in the menu than in previous versions.

Select Hole replaces **Select Feature**. It is now called **Select Hole** because you can use it only to select holes and not other features.

The **Preferences** option is no longer available because these options are now available on the **Create Feature Set** or **Create Holes** options.

Enhancements to the individual feature set menu

There are several changes to the individual feature set menu.



The **Feature Set Editor** option is now on the **Edit** menu.

There are several new options:

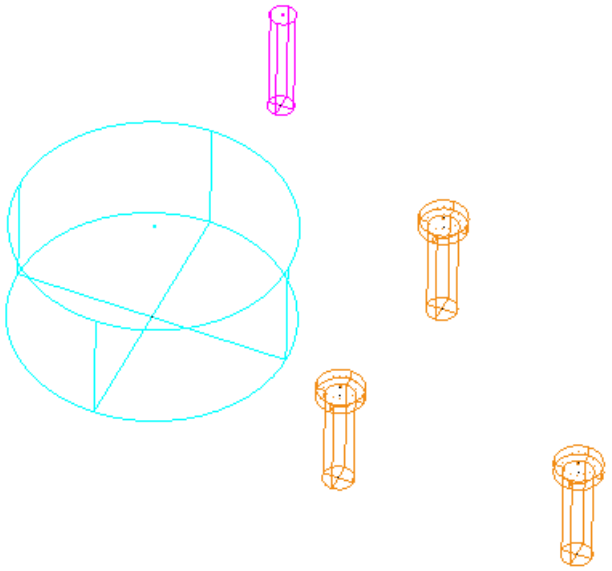
Select Hole was called **Select Feature**. It is now called **Select Hole** because you can use it only to select holes and not other features.

Create Features replaces the **Settings** option in previous versions. This displays the settings used to create the feature.

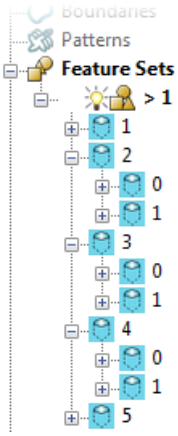
Create Holes displays the settings used to create the hole (see page 36).

Separate Compound Holes separates all compound holes into single holes. This is on the **Edit** menu.

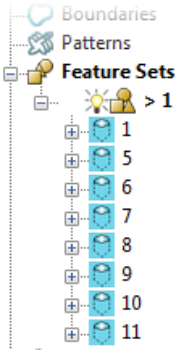
In this example there are two standard holes and three compound holes. The compound holes are orange.



Looking at the feature set you can see the holes:



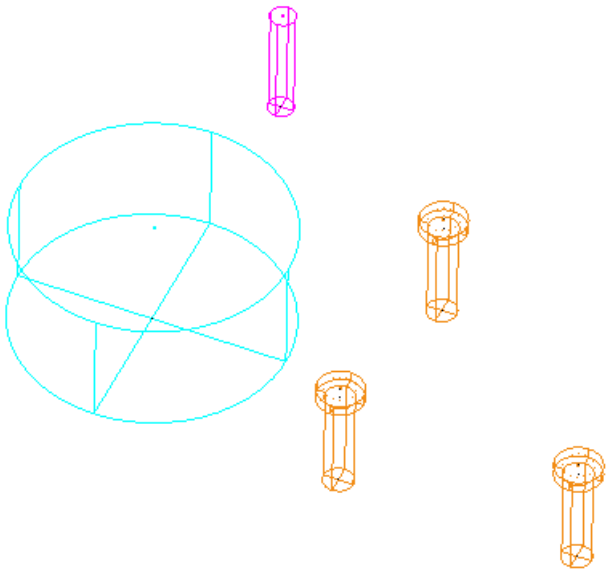
Selecting **Separate Compound Holes** converts the compound holes to their individual hole components.



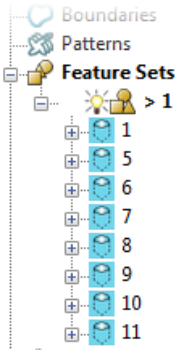
If you select one (or more) compound holes then PowerMILL separates only the selected holes.

Combine Holes combines single holes into compound holes. This is on the **Edit** menu.

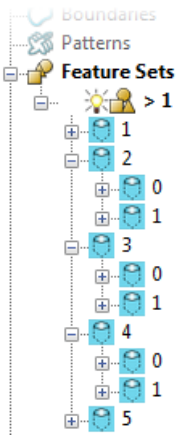
In this example there are eight holes. Starting with these holes.



Looking at the feature set, you can see the holes:



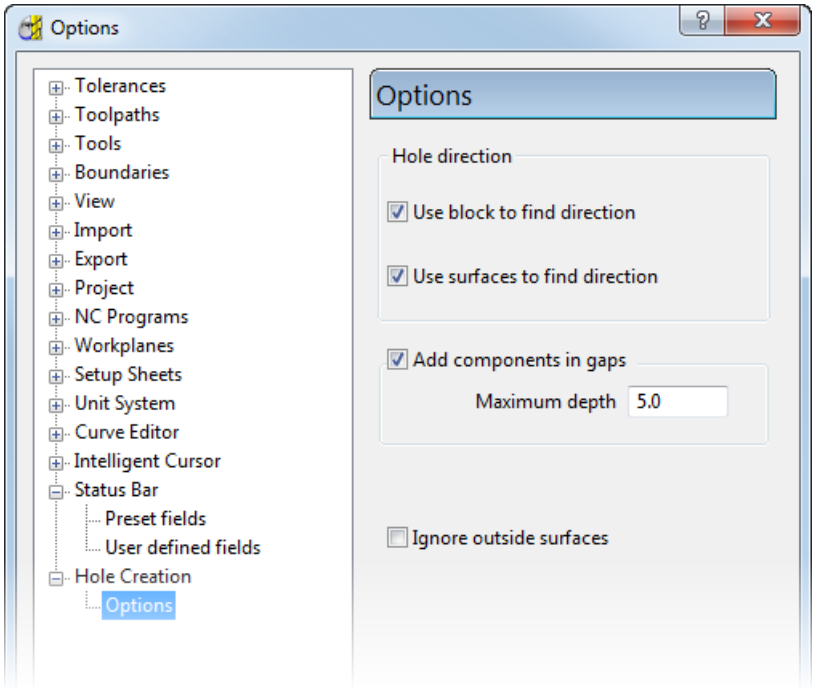
Selecting **Combine Holes** converts the individual hole components, which share a Z axis, to a compound hole.



Hole creation options

There is a new page on the **Options** dialog of **Hole Creation > Options**.

The **Hole Creation Options** determine the default options used to create a hole.



Use block to find direction — Select to use the block to determine the direction of holes. When selected, holes point towards the block centre.

Use surfaces to find direction — Select to use the surfaces to determine the direction of holes.

Add components in gaps — Select to add extra hole components to fill in vertical gaps between hole components in compound holes.

Maximum depth — Enter the maximum depth of a hole component you can add when selecting **Add components in gaps**. No additional components are added if a compound hole contains a gap larger than this value.

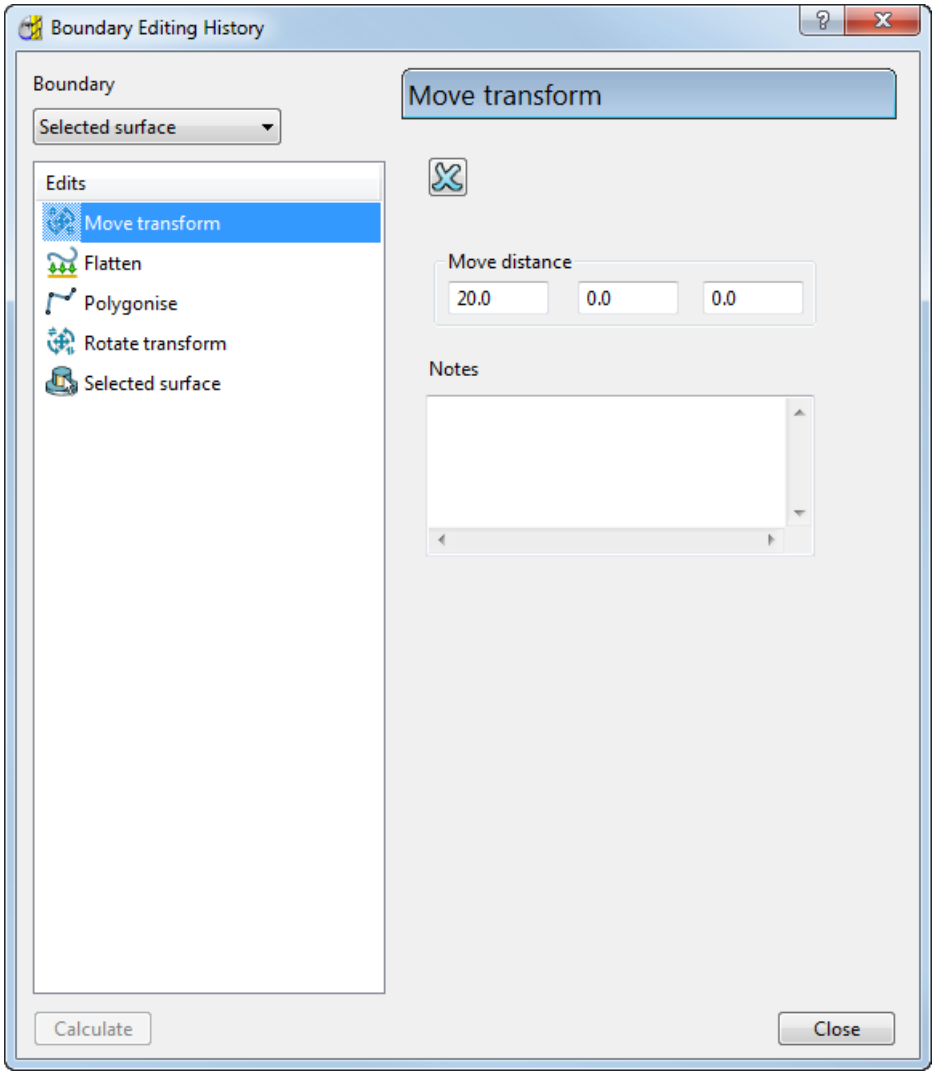
Ignore outside surfaces — Select to use the inner surface to create a hole, this is useful where you have a hole inside a boss.When deselected, PowerMILL creates two holes: one for the inside and one for the outside, this is useful for tapped holes where both surfaces are modelled.

*This option is disabled when **Use surfaces to find direction** is deselected.*

Boundary editing history

You can now see a history of your boundary edits. This is similar to the history of your toolpath edits.

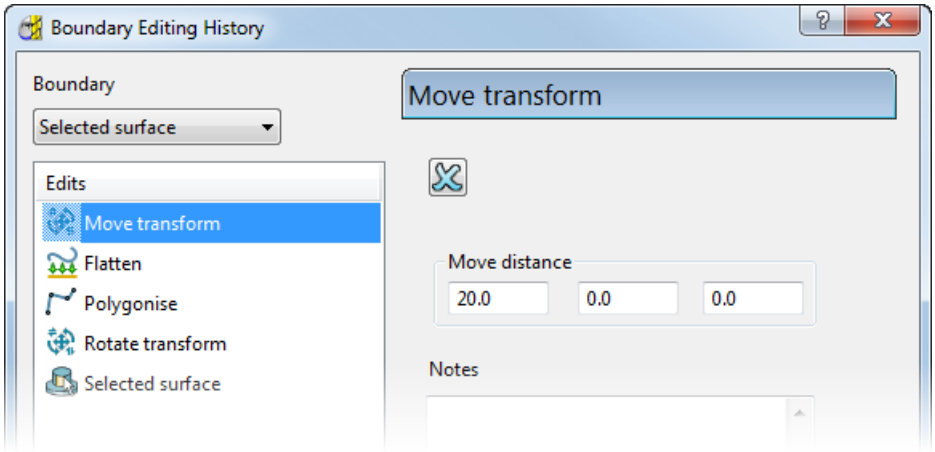
Use the **Boundary Editing History** dialog to show a history of all the edits on a boundary. It also enables you to edit some of them.



The **Boundary Editing History** dialog is available from the individual boundary **Edit > Boundary Editing History** menu or by clicking on the **Boundary** toolbar.

Boundary — Select the boundary to review and re-edit the boundary's previous edits.

Edits — The pane displays a list of all the edits performed on the boundary. The most recent edit is at the top of the list and the original boundary is at the bottom of the list. The right side of the dialog displays information on the selected edit.



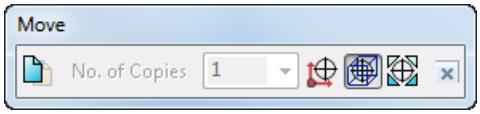
In this case, the edit moved the boundary by 20 mm in X and you can edit these values.

If you make more than one transform before accepting the result, PowerMILL shows a complex transform rather than the separate individual transforms.

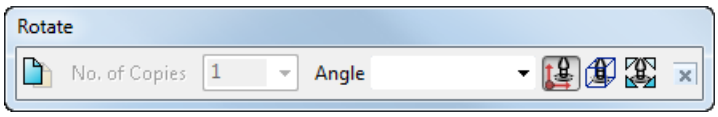
For example:

1. From the individual boundary menu, select **Edit > Transform**. This displays the **Boundary Transform** mode toolbar.

2. Click  on the **Boundary Transform** mode toolbar. This displays the **Move** toolbar.



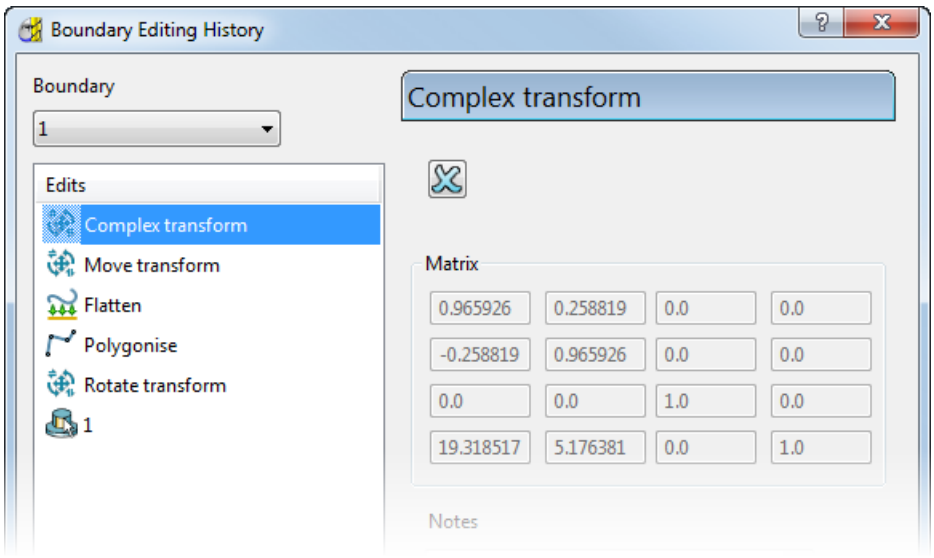
3. On the **Move** toolbar:
   a. Enter the coordinates **20 0 0** in the  field on the status toolbar.
   b. Click  to close the **Move** toolbar.
4. Click  on the **Boundary Transform** mode toolbar. This displays the **Rotate** toolbar.



5. On the **Rotate** toolbar:
   a. Enter an **Angle** of **15**.
   b. Click  to close the **Move** toolbar.

6 Click  on the **Boundary Transform** mode toolbar to accept the two transforms.

7 From the individual boundary menu, select **Edit > Boundary Editing History**. This displays the two transforms as one complex transform.



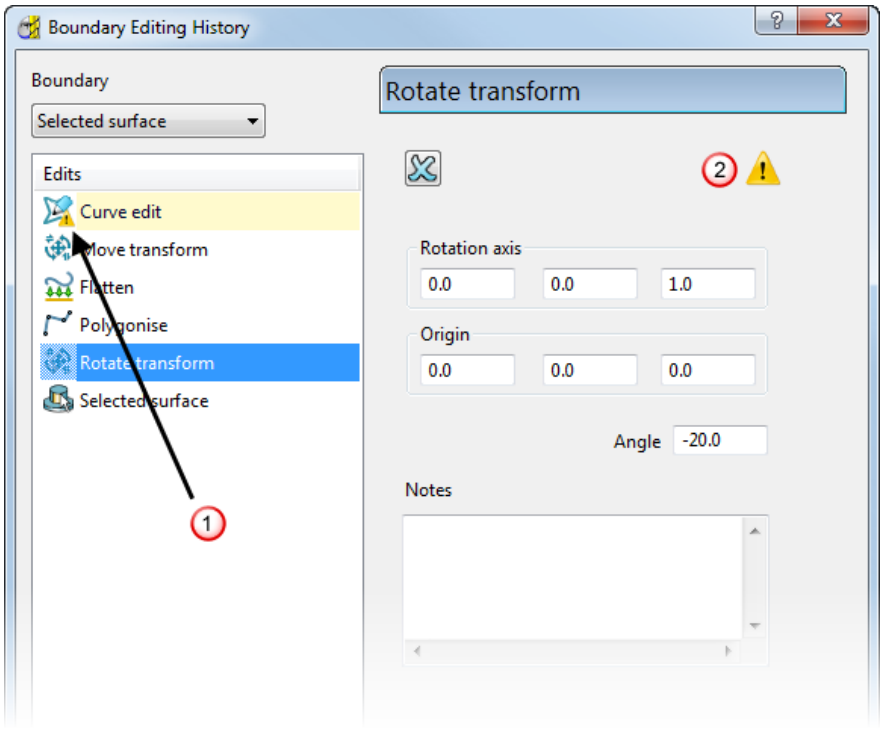
In this case, you can see the matrix that combines the move and rotation but you cannot edit these values. You can delete this transformation by clicking  .

 **Delete** — Click to delete the selected edit.

 — When displayed, the boundary contains selective edits. Hover over  to display more information.

*If your boundary edits contain selective edits you can modify the edits, but clicking on the **Calculate** button only applies the edits up to, but not including, the selective edit.*

A selective edit is an edit that requires you to select a curve segment or a point. For example, **Delete Selected Components**.

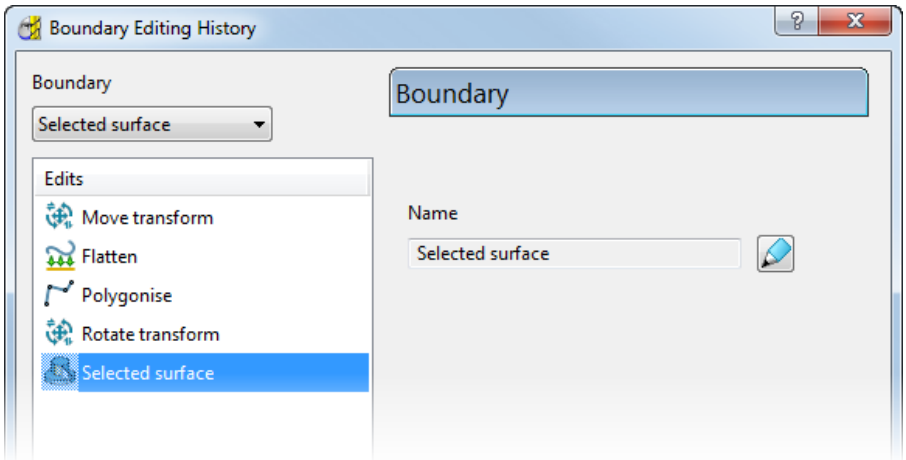


1 The selective edit.

2 ⚠ Indicates that if you edit any values, you can re-calculate your boundary, but the history is not fully re-applied.

Notes — Type any specific notes to help you understand the edit.

If you select the boundary, rather than an edit, the right side of the dialog displays the **Boundary** page.



Name — Displays the name of the boundary.

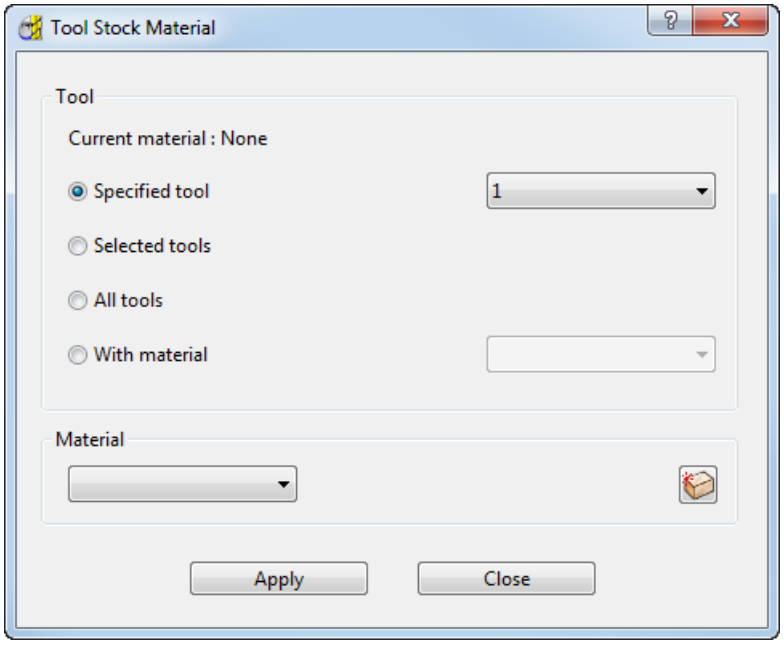
— Click to display the **Boundary** dialog. This is the same dialog that is displayed when you create or edit a boundary.

Calculate — Click to recalculate the boundary based on edits you make to this dialog.

Improvements to specifying the tool stock material

In previous versions of PowerMILL you could enter a new stock material of a tool, but you were unable to see the existing stock materials. So, it wasn't easy to use an existing material as you had to match the spelling and capitalisation correctly. Now you can select an existing material from a list, or create a new material.

Use the **Tool Stock Material** dialog to specify or change the stock material of tools.



The **Tool Stock Material** dialog is available on the **Tools** context menu, or on the individual tool context menu.

Tool — Use these options to identify the tools whose stock material you want to change.

Specified tool — Select to enter a stock material for the tool selected in the list.

— Select the required tool from the list.

Selected tools — Select to enter a stock material for all tools selected in the explorer.

All tools — Select to enter a stock material for all tools in the project.

With material — Select to enter a stock material for all tools which currently have the material specified in the list.

 — Select the required material from the list. If no material is selected, the material is applied to all tools which don't have a material.

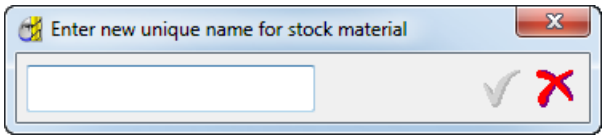
Material — Use these options to specify the required stock material.

 — Select a stock material from the list. If there are no materials listed, or the material you require isn't listed then you must create a new material using  .

 **New Stock Material** — Select to create a new stock material. This displays the **New Stock Material Name** dialog.

To create a new stock material:

1. Click  to displays the **New Stock Material Name** dialog:



2. Enter a name for the new stock material

Spaces are not allowed in stock material names.

*So, for example, **Mild Steel** is not a valid name because there is a space in the name; but the names **Mild_Steel** or **MildSteel** are valid.*

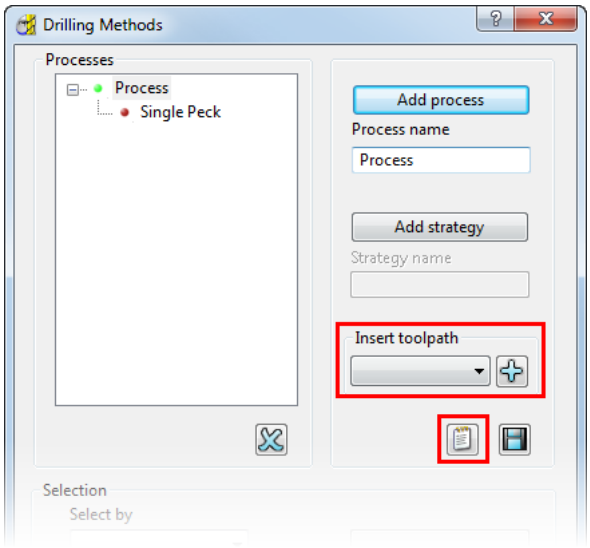
3. Click  .

Toolpath generation

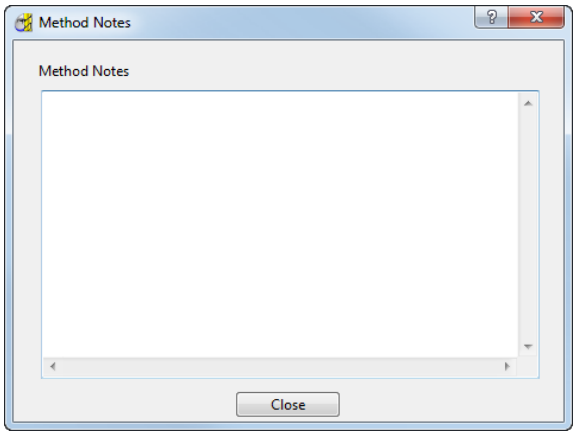
Drilling enhancements

When drilling compound holes, you can now choose which hole components you want to drill (see page 71).

When creating a drilling method you can now:



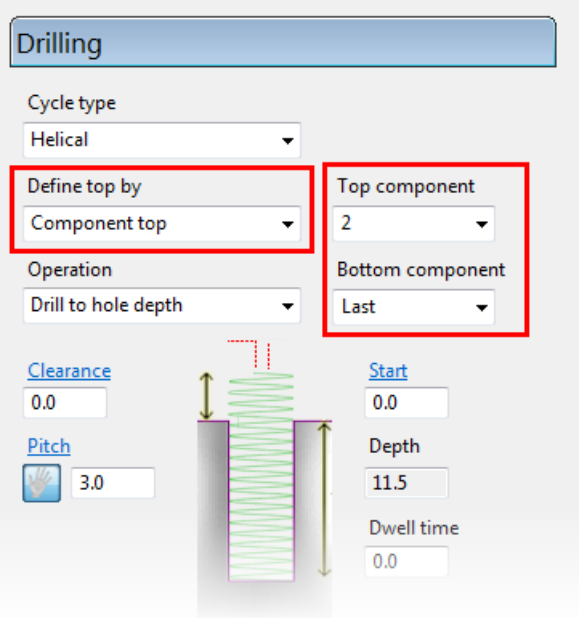
- Add a toolpath: select an existing drilling toolpath, then click  to add it to the drilling method.
- Add a note: click **Note**  to display the **Method Notes** dialog. Use the dialog to enter a note about the drilling method. The notes appear at the top of the method xml file.



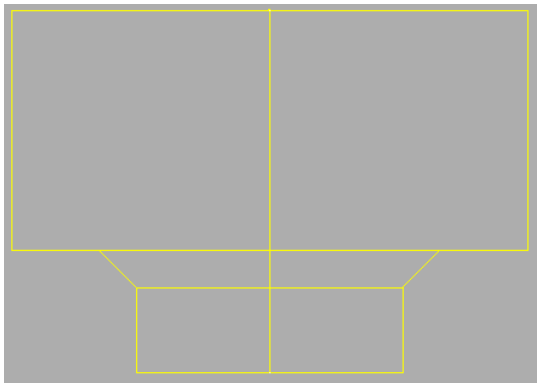
Drilling enhancements

When drilling compound holes, you can now choose which hole components you want to drill.

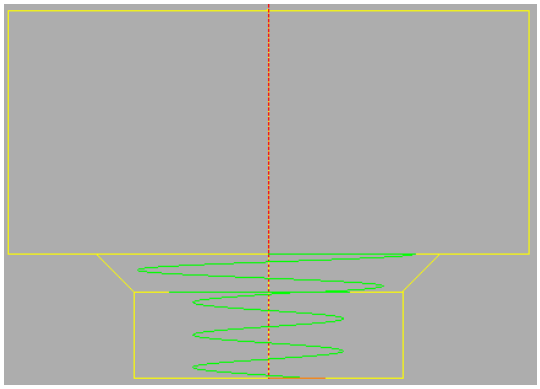
On the drilling strategy dialogs you can now select the **Bottom component**. If you select a **Define top** by of **Component top** you can also specify a **Top component**.



For example, with this compound hole:



Selecting a **Top component** of **2** and a **Bottom component** of **3** or **Last** produces a toolpath on the bottom two components.



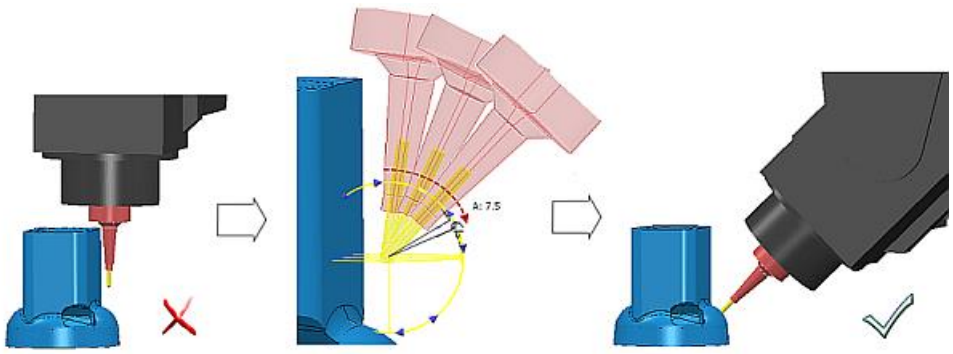
Advanced simulation

Dynamic machine control mode

The new **Dynamic machine control** mode enables you to adjust the configuration and tool axis of a 3+2 axis machine tool, so you can quickly and easily improve the machine tool's access to the workpiece.

The mode includes:

- Interactive grab-handles let you drag the machine tool into new positions (see page 74).
- The **Dynamic machine control** mode toolbar features a range of buttons that compliment the use of the grab-handles. The **Swap machine tool configuration** button  is very useful for orientating a machine tool with an asymmetrical head (see page 76).
- The **Dynamic machine control** mode dialog lets you manually specify the machine tool position and tool axis, instead of using the interactive grab-handles. The dialog also enables you to specify the increment value for rotary axes (see page 78).



In this graphic, access to the workpiece is improved by rotating the grab-handle to adjust the machine tool's configuration and tool axis.

Entering the Dynamic machine control mode

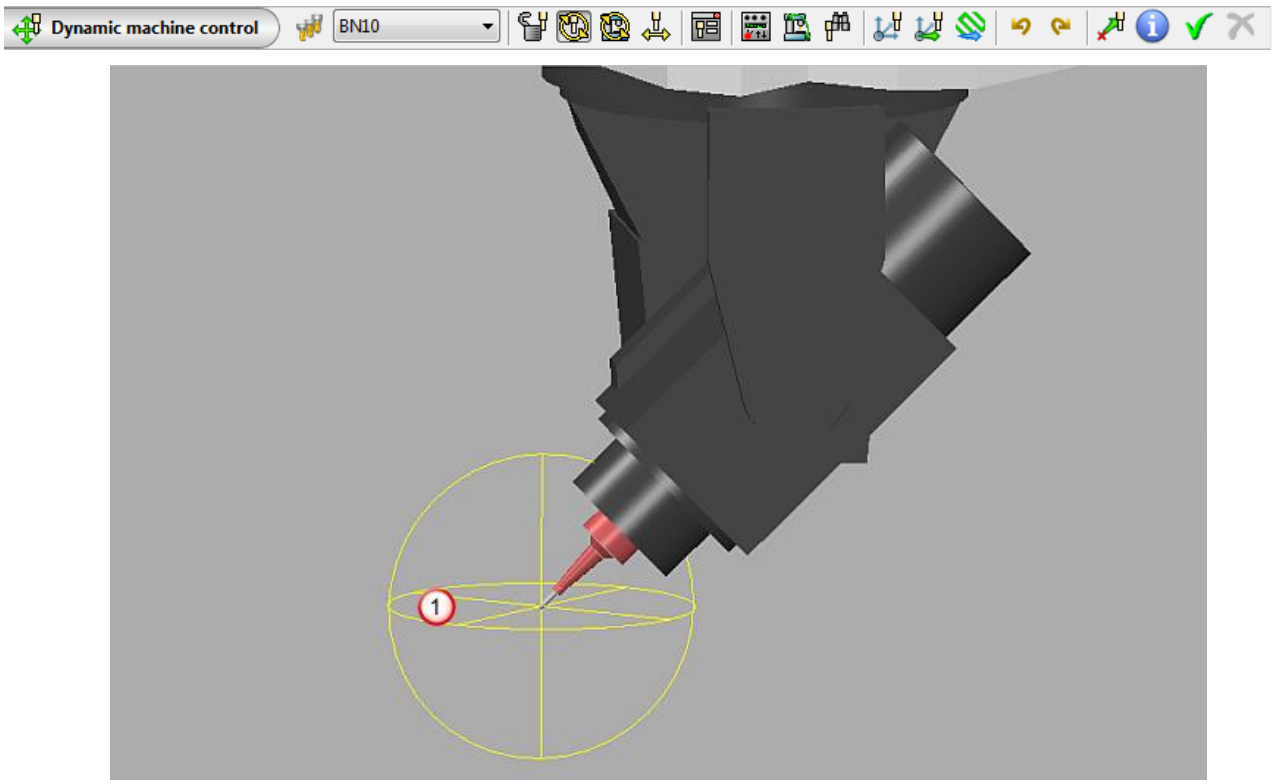
*To use the **Dynamic machine control** mode, you must have the PowerMILL Advanced Simulation licence (POWERMILL–ADVSIM).*

You can enter the mode at any time.

The mode is only available for the following tools: ball nosed, tip radiused, end mill, tapered spherical or tapered tip.

Activating a mode disables most of PowerMILL's functionality until you exit from the mode. For more information, see Mode toolbars.

To enter the mode, click on the **Tool** toolbar. Entering the mode displays grab-handles around the tool ① and the **Dynamic machine control** mode toolbar.



Repositioning the machine tool interactively

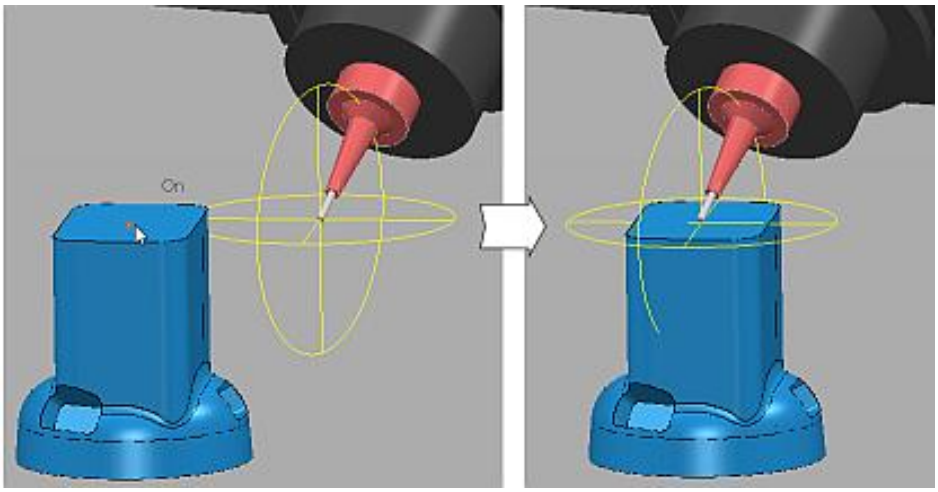
You can improve the machine tool's access to the workpiece by repositioning the machine tool with the new interactive features.

*You may find it useful to use the interactive features to reposition the machine tool approximately and then use the **Dynamic machine control** mode dialog to position the machine tool more accurately (see page 78).*

The interactive features include:

Jump to a point

Double-clicking on the model makes the machine tool jump to the selected point.

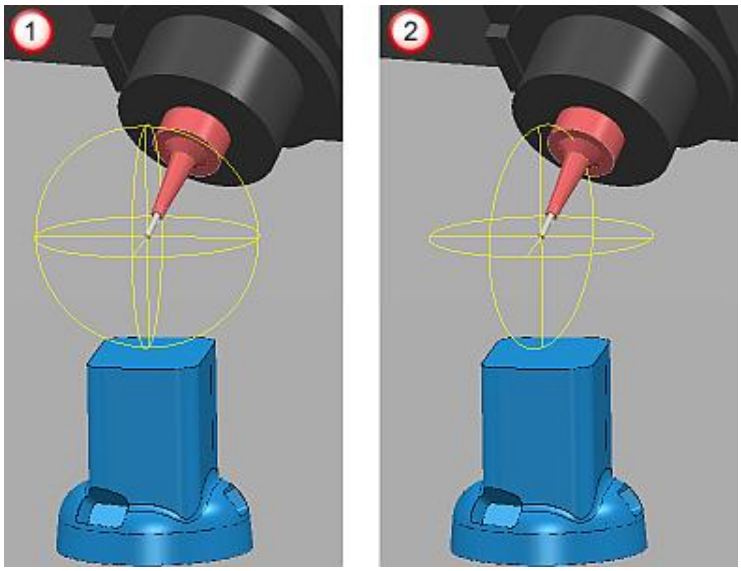


Grab-handles

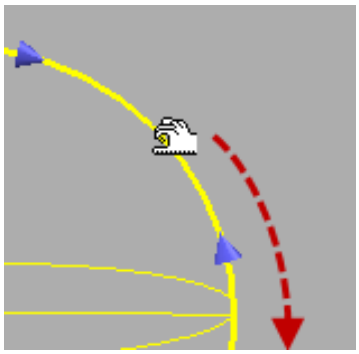
Each grab-handle is associated with an axis. By dragging a grab-handle, you move the machine tool within the associated axis. The circular grab-handles rotate the tool axis, and the linear grab-handles move the tool along a single axis. There are two sets of grab-handles that you can toggle between so you can achieve the best tool configuration and tool axis:

Clicking displays the grab-handles aligned to the active workplane (1).

Clicking displays the grab-handles aligned to the axes of the machine tool (2).

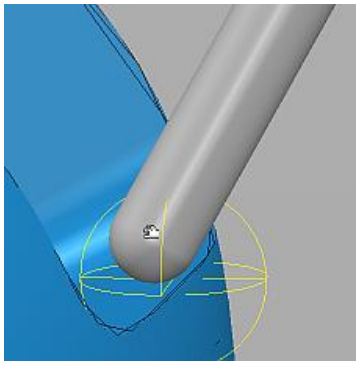


When you hover the cursor over a drag-handle, the drag-handle is highlighted. You can then drag the machine tool into a new position.



Dragging the tool

When you hover the cursor over the tool, the cursor changes to  . This enables you to drag the machine tool free-hand across the model.



Dragging the tool free-hand (or 'scanning' the model with the machine tool) is a useful technique for quickly seeing if the current tool axis is appropriate for machining the surface of interest.

Using the Dynamic machine control mode toolbar

The buttons on the **Dynamic machine control** mode toolbar compliment the use of the grab-handles. For more information about using the grab-handles, see Repositioning the machine tool interactively (see page 74).

BN4 **Select tool** — Click to select the tool you want to simulate. This enables you to simulate an alternative tool to the one used by the current toolpath. If the cutting geometry of the alternative tool is the same as the original tool, you can click **Update toolpath** to update the toolpath so it now uses the alternative tool.

Lock tool axis — Click to lock the tool axis. This limits the machine tool movement to the X, Y, and Z axes, which is useful if you want to preserve the tool axis while changing the configuration of the machine tool.

Rotate around current workplane — Click to display the grab-handles aligned to the active workplane (see page 74).

Use machine tool graphics — Click to display the grab-handles aligned to the axes of the machine tool (see page 74).

Use tool positioning graphics — Click so you can simultaneously move the machine tool in the X, Y, and Z axes of the active workplane.

Advanced settings — Click to display the **Dynamic machine control** mode dialog. The dialog enables you to specify the tool orientation (position and axis), as well as increment values to restrict the movement of the tool's rotary axes (see page 78).

Machine Tool Position — Click to display the **Machine Tool Position** dialog. The **Machine Tool Position** dialog displays the position of each machine tool axis. You can use the dialog for reference purposes and to jog the machine tool's axes by using the dialog's interactive features (see page 87).

Swap machine tool configuration — Click so PowerMILL reorientates the machine tool but without changing the tool axis. This is especially useful for moving an asymmetrical machine head out of the way.

Align with view — Click to align the tool axis direction with the direction of your current view.

Create workplane aligned with tool — Click to create a workplane aligned with the machine tool.

Create workplane at tool tip — Click to create a workplane at the tool tip.

Choose the workplane that suits your preferred working method.

Update toolpath — Click to update the toolpath with the new machine tool position.

*After updating the toolpath, you must check the toolpath for collisions using the **Toolpath verification** function.*

*The **Update Toolpath** function can be used only when you are using a tool with a spherical tool tip. To update a toolpath that uses a tool with a non-spherical tip, you have to enter the values of the new tool position in the toolpath's strategy dialog.*

Undo — Click to revert to what it was before the last change.

Redo — Click to reinstate the edit you have just undone.

Nearest valid position — If you move the machine tool beyond its physical limits, you can click so PowerMILL moves the tool to the nearest valid position.

PowerMILL displays one of these icons if there is an issue.The icon displayed depends on the severity of the issue. Hover the cursor over the icon for more information.

Accept changes — Click to accept changes and resume normal PowerMILL functions.

Close — Click to exit the mode without saving changes.

Using the Dynamic machine control mode dialog

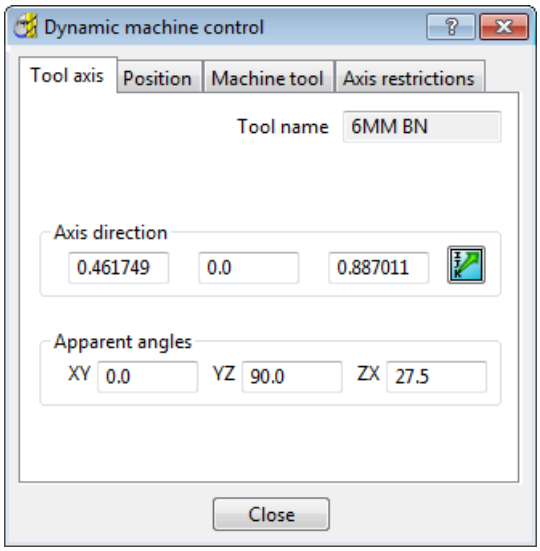
There are four tabs on the **Dynamic machine control** mode dialog:

- **Tool axis**
- **Position**
- **Machine tool**
- **Axis restrictions**

To display the new dialog, click  on the **Dynamic machine control** mode toolbar (see page 76).

Tool axis tab

The **Tool axis** tab on the **Dynamic machine control** mode dialog displays the tool axis direction as vectors and apparent angles. The values update automatically when you use the grab-handles to adjust the tool axis.



Tool name — The name of the tool.

Axis direction — Specify the tool axis as vectors.

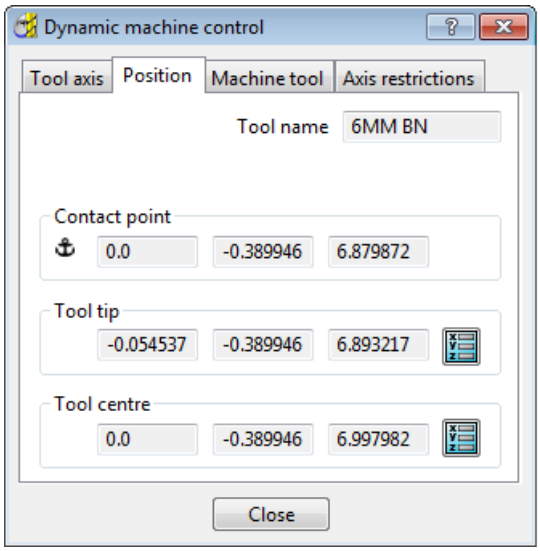
Apparent angles — Specify the tool axis as apparent angles.

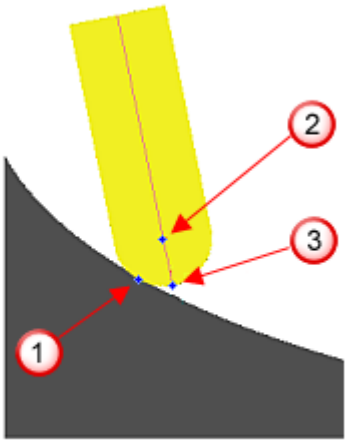
 **Direction** — Click to display the **Direction** dialog. The dialog enables you to specify the tool axis instead of using the grab-handles.

*You may find it useful to use the grab-handles to specify the tool axis approximately and then use the **Tool axis** tab to specify the tool axis more accurately.*

Position tab

The **Position** tab on the **Dynamic machine control** mode dialog displays the tool position in X, Y, and Z (relative to the active workplane). The values update automatically when you use the grab-handles to adjust the tool position.





Tool name — The name of the tool.

Contact point — Displays the contact point between the tool and the model (1).

Tool tip — Displays the position of the tool tip (2).

Tool centre — Displays the position of the tool centre (3).

*The **Tool centre** field is displayed only when you are simulating a tool with a spherical tip.*

Anchor point — The indicates which point PowerMILL is currently using as the centre of rotation when you move the machine tool:

Contact point — Used when the tool is in contact with the model.

Tool tip — Used when the tool is not in contact with the model and the simulated tool has a non-spherical tip.

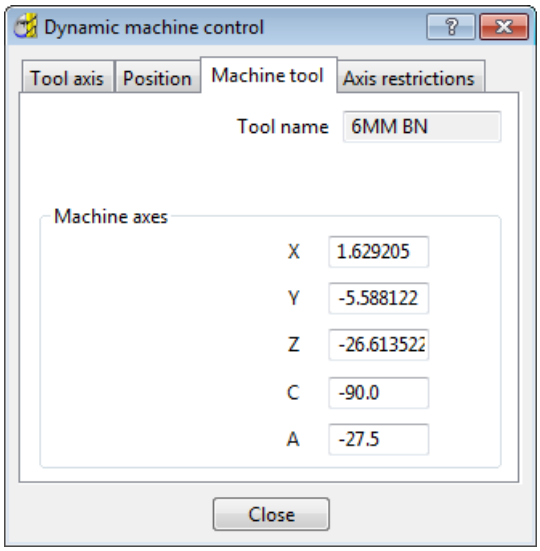
Tool centre — Used when the tool is not in contact with the model and the simulated tool has a spherical tip.

Position — Click to display the **Position** dialog. The dialog enables you to define the position of tool by specifying the position of the tip or centre of the tool.

*You may find it useful to use the grab-handles to define the tool position approximately and then use the **Position** tab to specify the tool position more accurately.*

Machine tool tab

The **Machine tool** tab on the **Dynamic machine control** mode dialog displays the position of each machine tool axis. The values update automatically when you use the grab-handles to adjust the position of the machine tool's axes.



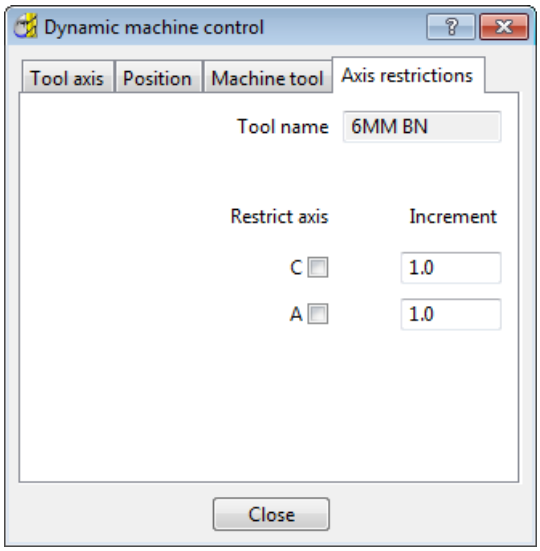
Tool name — The name of the tool.

Machine axes — Specify the position of the machine tool's axes.

*You may find it useful to use the grab-handles to reposition the machine tool's axes approximately and then use the **Machine tool** tab to specify the position of the axes more accurately.*

Axis restrictions tab

The **Axis restrictions** tab on the **Dynamic machine control** mode dialog enables you to enter the increment values of your actual machine tool's axes, so the axes of the virtual machine tool move accurately.



Tool name — The name of the tool.

Restrict axis — Select to restrict the movement of the axis by the specified increment value.

Increment value — Specify the increment value.

If you adjust an axis without an increment restriction, it is possible that the axis position may not be achievable by your actual machine tool.

If the current axis value is 83.34, for example, and you restrict the axis with an increment value of 1, when you adjust the axis, the axis value is rounded down to 83 and then increases 83 – 84 – 85 and not 83.34 – 84.34 – 85.34.

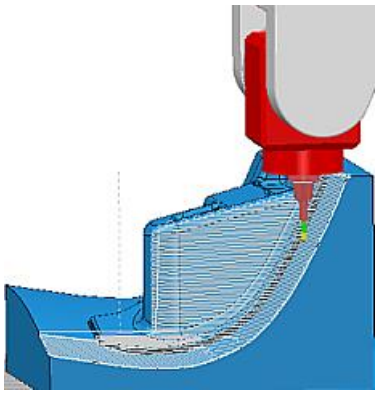
Using the Dynamic machine control example

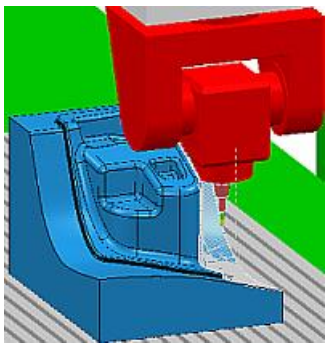
This example shows how to use the functions of the **Dynamic machine control** mode to prevent collisions between the machine tool and the workpiece.

The example uses the DynamicMachineControl_Example project in the **Examples** folder.

1. In the **Toolpaths** branch of the explorer, right-click the toolpath and select **Simulate from Start**.

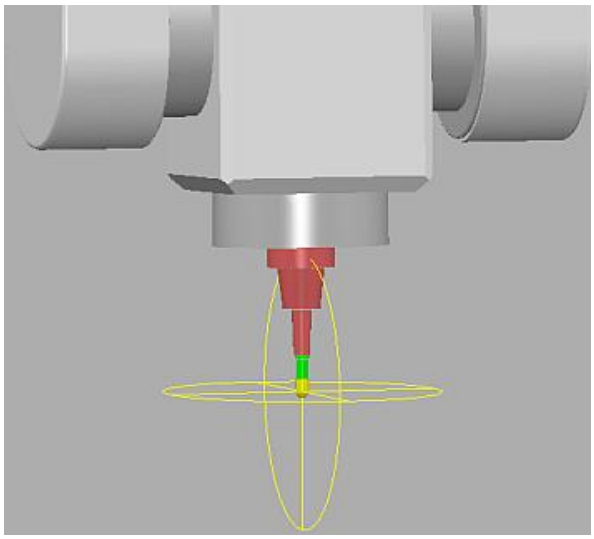
   As the machine tool machines down the surfaces of the workpiece, the head and the A axis-component of the machine tool collide with the workpiece.





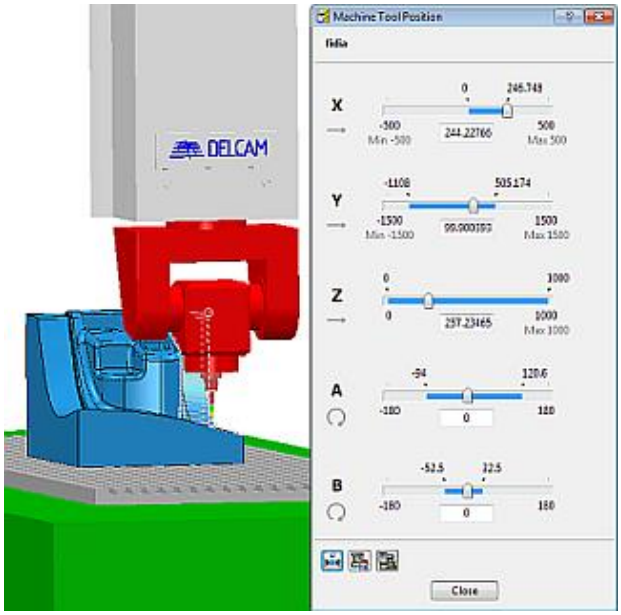
2. On the **Tool** toolbar, click  to enter the **Dynamic machine control** mode.

   This displays the **Dynamic machine control** mode toolbar and the grab-handles around the tool.



3. On the mode toolbar, click  to display the **Machine Tool Position** dialog.

The dialog displays the position of each machine tool axis. Displaying the dialog is useful because it enables you to see the movement of the axes and, in particular, if you move an axis beyond its limit.



4 Specify the increment values of the rotary axes of the actual machine tool. This ensures that any adjustments you make to the virtual machine tool's axes can be achieved by the actual machine tool.

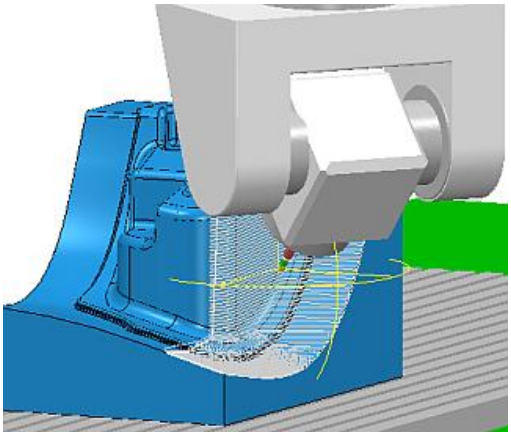
   a On the mode toolbar, click to display the **Dynamic machine control** mode dialog, and then select the **Axis restrictions** tab.

   b For both **A** and **B** axes, select the **Restrict axis** option and specify an **increment** value of 2.

5 Reconfigure the machine tool's position and tool axis.

   a On the mode toolbar, click to display the grab-handles aligned to the axes of the machine tool.

   b Use the grab-handles to reconfigure the machine tool.

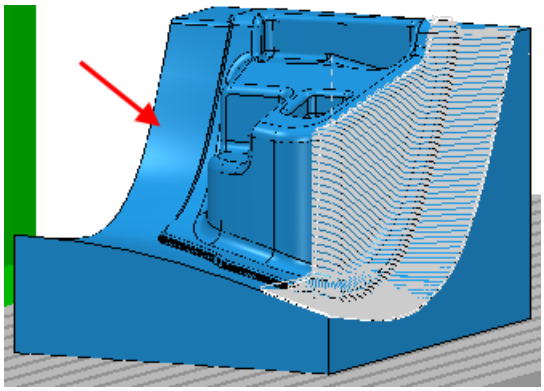


c Click to swap the machine tool's configuration. PowerMILL improves the machine tool's clearance by adjusting the configuration of the machine tool (in particular by rotating the machine head by 180 degrees) but without altering the tool axis.

d Double-click on various areas of the toolpath to make sure the new configuration and tool axis do not cause any new collisions.

6 Click to update the toolpath with the machine tool's new position and tool axis.

7 The left side of the workpiece requires a similar toolpath. Repeat step five to achieve a suitable machine tool position and tool axis.



8 Click to create a workplane aligned with the tool. You can use this workplane when you create the toolpath for the area.

9 Click to accept the changes.

Toolpath output

NC program updates

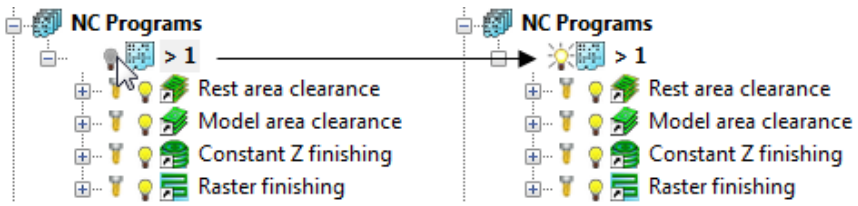
There are several updates to NC program functions:

- You can now multi-select NC programs in the explorer in the same way that you can multi-select other PowerMILL entities. In previous version you could select only one NC program at a time.
- There is a new option which enables you to draw toolpaths in the **Toolpaths** branch of the explorer that are used in an NC program (see page 85).
- You can now add toolpaths to an NC program multiple times (see page 86).
- In the **NC program** dialog, you can now create a new fixture offset by entering its name in the **Fixture offset** list.

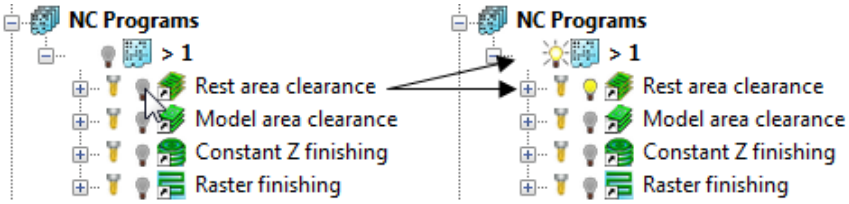
Drawing toolpaths in an NC program

The introduction of light bulbs next to toolpaths in an NC program now enables you to choose which toolpaths in an NC program you want to draw.

When you create an NC program, the toolpaths you add to it are drawn but the NC program isn't. Click the NC program's light bulb to draw the NC program and display the toolpaths.



Drawing a toolpath, which is in an undrawn NC program, automatically draws the NC program, so the toolpath is displayed.

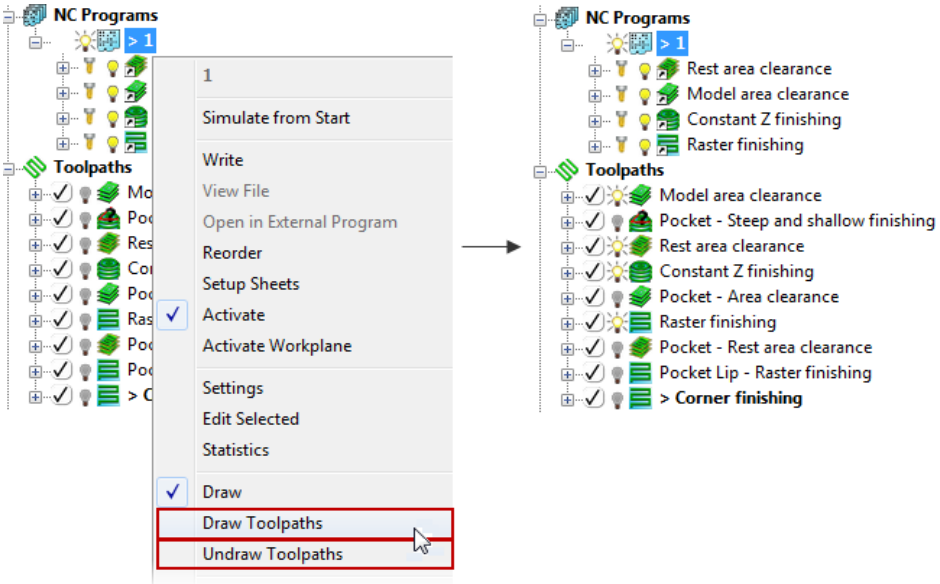


In previous versions of PowerMILL, drawing an NC program displays all the toolpaths in the NC program and, in the ***Toolpaths*** *branch, the lightbulbs next to the those toolpaths update to .*

In PowerMILL 2014R2, to display the toolpaths in the ***Toolpaths*** *branch that are included in an NC program, use the new options on the individual NC program menu:*

Draw toolpaths *— Click to draw all the toolpaths in the* ***Toolpaths*** *branch that are included in the NC program.*

Undraw toolpaths *— Click to undraw all the toolpaths in the* ***Toolpaths*** *branch that are included in the NC program.*



Duplicating toolpaths in an NC program

You can now place a toolpath into an NC program multiple times. In previous versions of PowerMILL this was not possible.

PowerMILL 2014 R2 allows toolpath duplication by default. When you write the NC program, PowerMILL displays a message to tell you that the NC program contains duplicates of a toolpath.

To disallow duplicate toolpaths, deselect the **Tools** menu **> Options > NC Programs > Allow duplicate toolpaths** option.

User interface

Positioning a machine tool

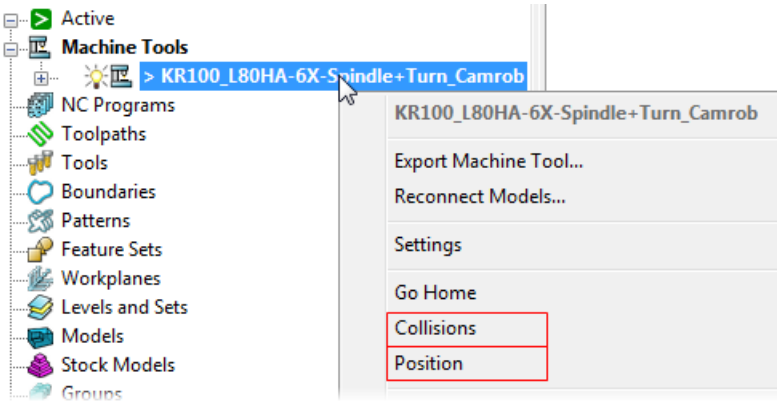
The new **Machine Tool Position** dialog replaces the **Machine Information** dialog. The new dialog features a significantly redesigned user interface that improves the presentation of the machine-tool-position data and enables you to jog a machine tool more easily (see page 88).

In previous versions of PowerMILL, the **Machine Collisions** tab displayed collision data. In PowerMILL 2014R2, collision data is displayed on the new **Machine Tool Collisions** dialog (see page 90).

To display the new dialogs:

- On the **Machine Tool** toolbar:
  - click  to display the new **Machine Tool Position** dialog.
  - click the new **Collisions** button  to display the **Machine Tool Collisions** dialog.

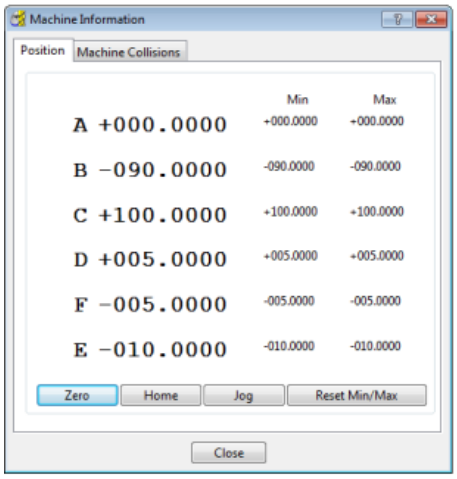
- In the explorer, right-click the active machine tool and select the new **Collisions** and **Position** options.



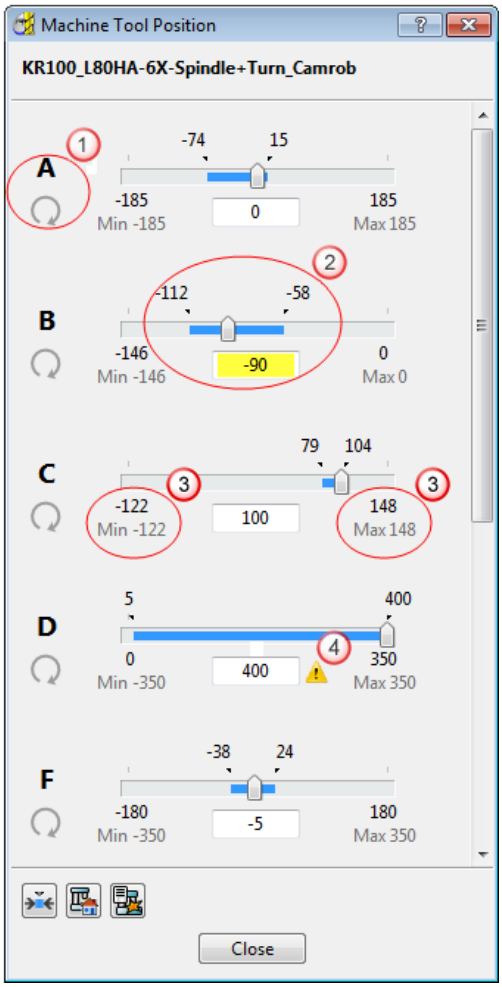
The new Machine Tool Position dialog

The new **Machine Tool Position** dialog includes the following features:

2014



2014 R2



1 The axis address (**A**) and the icon indicates the axis and the axis type.

→ Indicates a linear axis.

Indicates a rotary axis

2 Drag the thumb to jog the axis. The value **-90** is the current position of the axis. The blue line and the values **-112** and **-58** indicate the furthest points that the axis has been jogged to during the current simulation session.

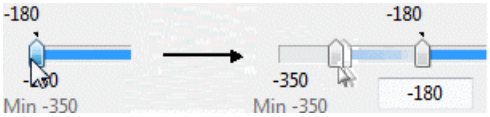
You can also jog an axis by:

- clicking the the thumb and then using the mouse scroll-wheel or arrow keys ↑ ↓.
- clicking on a point along the slider.
- entering a value in the **Position** field (highlighted yellow).

3 The **Min** and **Max** axis values are displayed at either end of the slider. The values displayed above the **Min** and **Max** values indicate the maximum range you can move the slider in one go.

If the range of the slider is not large enough to jog the axis to its **Min** or **Max** limit in one go:

- Drag the thumb to the end of the slider. PowerMILL displays the remaining range of the slider, so you can jog the axis to its **Min** or **Max** limit.



4 PowerMILL displays a warning if you enter an axis position (in the **Position** field) that is greater than the axis limit.

Reset attained axis positions — Click to clear the blue lines 2 which indicate the range of movement of each axis. It is useful to click before you jog or simulate the machine tool.

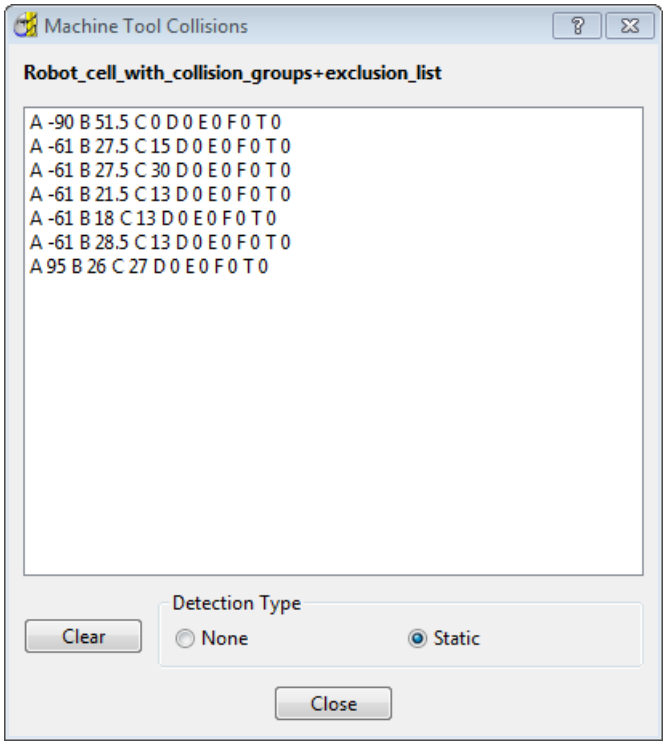
Home — Click to reset the position of the machine tool to its home position.

Collisions — Click to display the new **Machine Tool Collisions** dialog. The **Machine Tool Collisions** dialog displays any collisions that occur when you simulate or jog the machine tool.

The new Machine Tool Collisions dialog

The new **Machine Tool Collisions** dialog has the same user interface as the **Machine Collisions** tab in previous versions of PowerMILL. To display the dialog, click  on the:

- **Machine Tool Toolbar**
- **Machine Tool Position** dialog



Automation

Macro programming

The macro programming language has a number of enhancements. You can now:

- Use commands and functions to create, edit and delete files and directories.
- Extract data from lists using more succinct expressions.
- Quickly and easily determine the name of active folders.
- Automate a sequence of edits, for example to a number of template projects.
- Sort lists and arrays of scalars (numerics or strings) or objects and entities in ascending or descending order.

For more information, see the Macro Programming Guide.

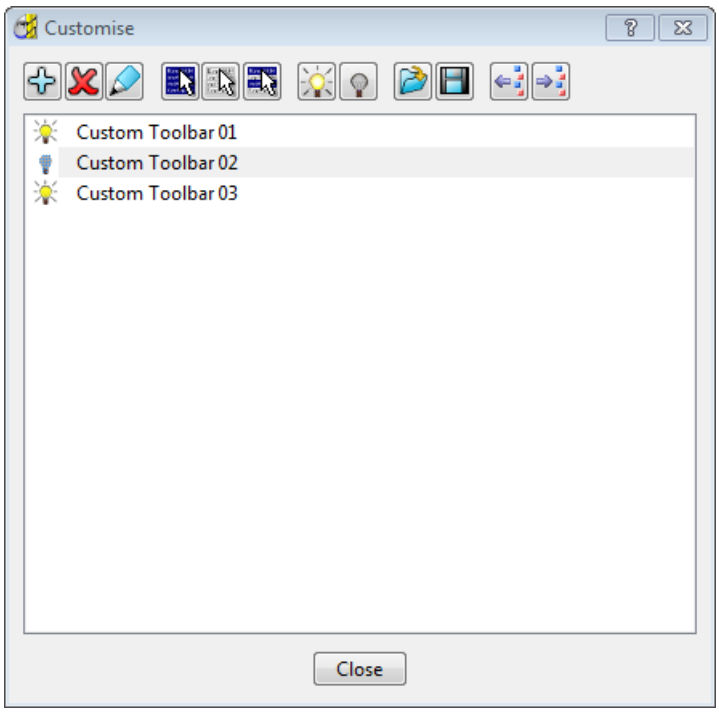
General enhancements

Creating custom toolbars

There is an improved process for creating and managing custom toolbars. In PowerMILL 2014R2 you can also have up to thirty-two custom toolbars at any time.

Creating and managing custom toolbars

To display the new **Customise** dialog, from the **View** menu, select **Toolbar** > **Custom** > **Customise**.



 **Create toolbar** — Click to create a new toolbar.

The button is unavailable if you reach the maximum number of thirty-two toolbars then.

 **Delete toolbar** — Click to permanently delete the selected toolbar(s).

 **Edit selected toolbar** — Click to display the **Edit Custom Toolbar** dialog. The dialog enables you to add, remove, and edit the buttons on the selected toolbar (see page 94).

 **Select all toolbars** — Click to select all toolbars.

 **Deselect all toolbars** — Click to deselect all toolbars.

 **Toggle toolbar selection** — Click to switch the toolbar selection so selected toolbars are deselected and deselected toolbars are selected.

 **Display selected toolbar(s)** — Click to display the selected toolbars.

 **Hide selected toolbar(s)** — Click to hide the selected toolbars that are currently displayed.

Sharing toolbars

PowerMILL 2014R2 has import and export options that enable you to share custom toolbars:

- The **Import toolbar**  and **Export toolbar**  buttons let you share toolbars between PowerMILL 2014R2 and future versions of PowerMILL. The two seats of PowerMILL can be installed on the same or different PCs.
- The **Import from older version**  and **Export to older version**  buttons are legacy functions. The functions let you share custom toolbars between PowerMILL 2014R2 and older versions of PowerMILL. To share the toolbars, both versions of PowerMILL must be installed on the same PC.

 **Import toolbar** — Click to import toolbars created by another PowerMILL user.

 **Export toolbar** — Click to export the selected toolbars so you can share them with another PowerMILL user.

 **Import from older version** — Click to import custom toolbars created in a previous version of PowerMILL.

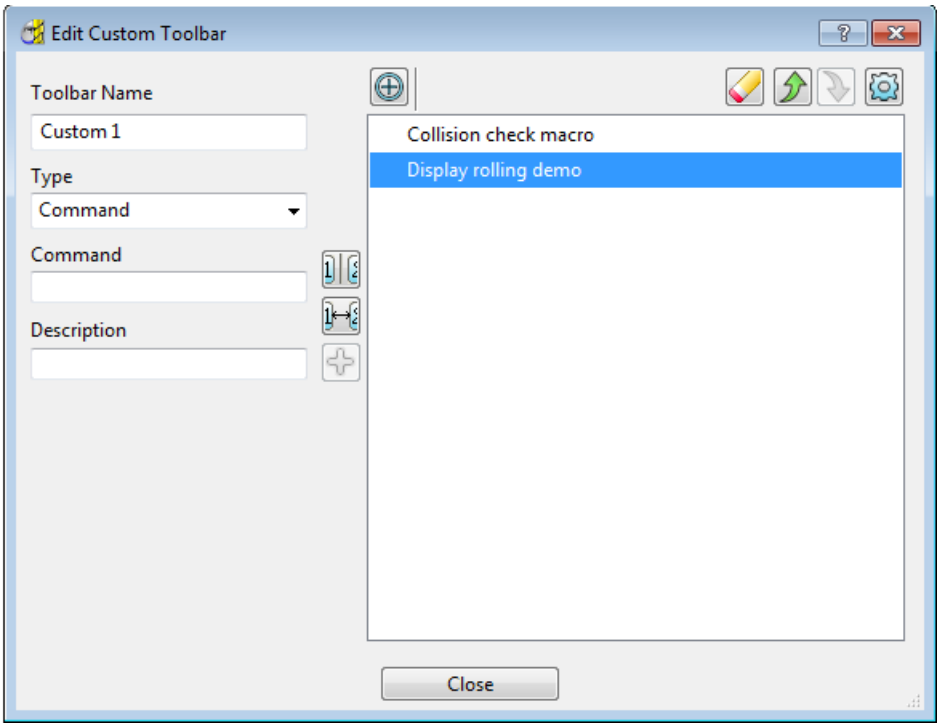
 **Export to older version** — Click to export custom toolbars to use in an older version of PowerMILL.

 *The appearance and behaviour of a toolbar does not change when you import or export it to a different version.*

 *The legacy options are temporary.*

Adding buttons to toolbars

The old **Customise** dialog features an updated design and is now called the **Edit Custom toolbar** dialog.



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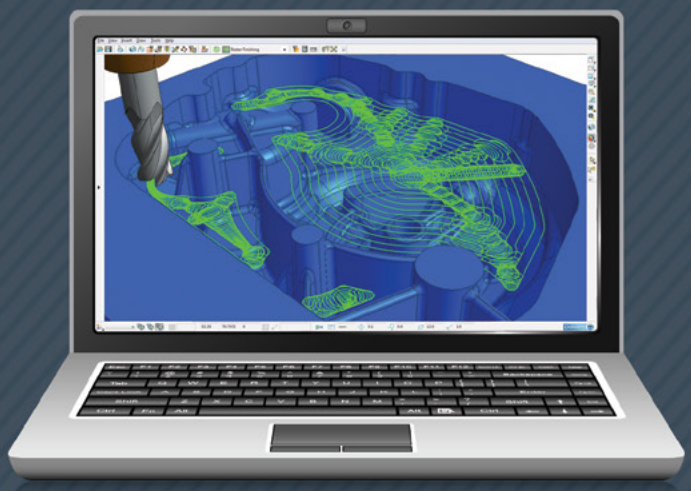

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