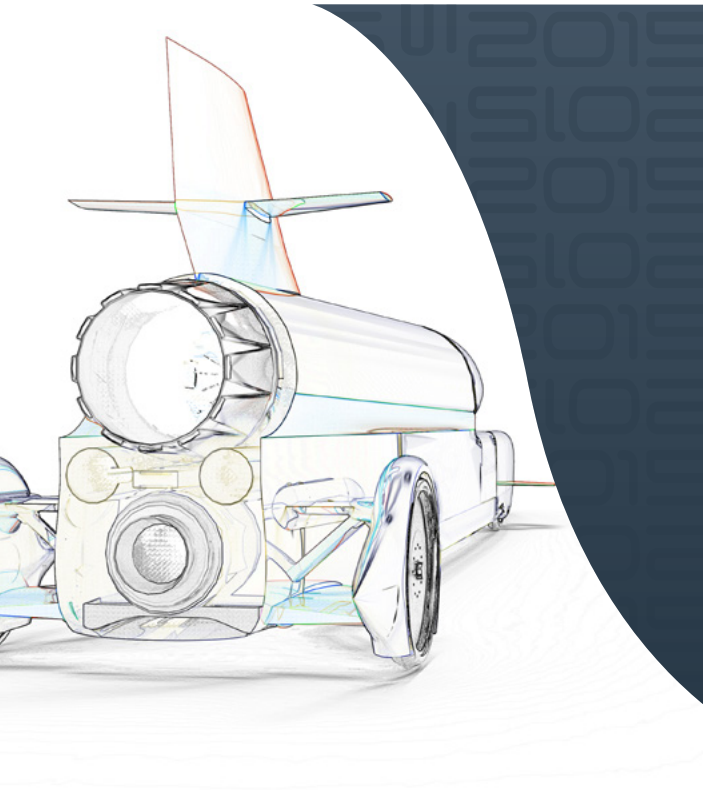


Delcam Advanced Manufacturing Solutions

PowerMILL 2015

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

www.powermill.com

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 2015

What's New

Issue 2

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 2015. Published on 15 July 2014

Contents

Summary of new features 1

Product licences 1
Preparing toolpaths 1
Generating toolpaths 1
Verifying toolpaths 2
Outputting toolpaths 3
General enhancements 3
Automating PowerMILL 3

Product licences 4

Preparing toolpaths 5

Curve editor enhancements 5
Ellipse dialog 7
Spiral dialog 10
Helix dialog 16
Opening a closed curve at a specific location 22
Open and Close curves 25
Limit Cut enhancements 26
Converting partial holes to (complete) holes 27

Generating toolpaths 29

Working with toolpath strategies 29
Saving a toolpath as a custom strategy 30
Raster finishing improvements 32
Approaching flats from outside of stock 34
Reducing the machining time of Vortex toolpaths 36
Fewer lift moves in roughing toolpaths 38
Improved tool axis and point distribution 38
Reducing undesirable movements of rotary axes 39
Updated Machine tool page 40
Editing a toolpath's tool axis and point distribution 43
Helical drilling 45

Verifying toolpaths 46

Enhanced collision checking 46
Checking for machine tool clearance 47
Improvements to NC program simulation 48

Model location workplane 49
Model location and output workplanes 49

Outputting toolpaths 50

Enhanced setup sheets 50
Using parameters to create and edit setup sheets 51
New parameters and functions 52

General enhancements 58

Dynamic machine control updates 58
Button and tooltip updates 59

Automating PowerMILL 60

Macros 60

Index 61

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 2015 offers all of the original features of PowerMILL 2014 R2, but with numerous improvements. This document describes the most significant improvements.

Product licences

- There are changes to product licences (see page 4).

Preparing toolpaths

- The **Curve Editor** mode toolbar includes several new updates and improved functions (see page 5).
- You can convert partial holes to (complete) holes more easily (see page 27).

Generating toolpaths

- Working with toolpath strategies is easier (see page 29):

  The **Strategy Selector** dialog has a redesigned user-interface.

  You can add your favourite strategies to the **Favourites** folder with a single click.

  Saving a toolpath as a custom strategy is easier.
- When using raster machining, you can let PowerMILL determine the most appropriate angle for each region (see page 32).
- There are improvements to Vortex toolpaths:

You can reduce machining time by increasing the feed rate of non-cutting moves and replacing long non-cutting moves with faster lift moves (see page 36).

Vortex toolpaths can approach flat areas from outside stock instead of always ramping into it (see page 34).

- PowerMILL can create roughing toolpaths with significantly fewer lift moves (see page 38).
- There are a number of enhancements to tool axis calculations and the distribution of toolpath points:

  PowerMILL can create multi-axis toolpaths with significantly fewer undesirable movements of a machine tool's rotary axes (see page 39).

  The **Machine tool** page of a strategy dialog includes several new options that enable you to select the most suitable coordinate system PowerMILL uses for calculating point redistribution, tool axis smoothing, and a fixed tool-axis angle (see page 40).

  There are several updates to the **Tool Axis Editing** dialog, which includes the **Machine tool**  button that displays the new **Machine tool** dialog. The new dialog enables you to specify the most suitable coordinate system for editing the tool axis and point distribution of a calculated toolpath (see page 43).
- There is a new **Lead** page for Helical drilling (see page 45).

Verifying toolpaths

- There are several improvements to the **Machine Tool Collisions** dialog. This includes a function that enables you to specify a clearance value around the machine tool which PowerMILL takes into consideration when checking for collisions (see page 46).
- There are two new options on the **NC program** dialog that enable you to specify the machine tool which the NC program is sent to, and the position of the model on the machine tool table. As well as PowerMILL saving this information in the NC program, PowerMILL uses this information simulating the NC program (see page 48).

Outputting toolpaths

- You can use PowerMILL parameters to produce more sophisticated setup sheets faster and more easily (see page 51).

General enhancements

- There is a new **Tool** tab on the **Dynamic machine control** mode dialog (see page 58).
- The **Dynamic machine control** mode provides more feedback when you jog a machine tool axis beyond its limits (see page 58).
- There is new **View boundary edits**  button on the **Boundary** toolbar, and on boundary settings dialog (see page 59).
- The **View toolpath edits** button on the **Toolpath** toolbar has a new button design (see page 59).

Automating PowerMILL

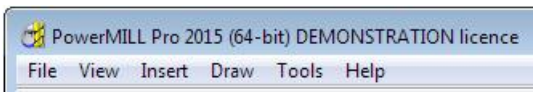
- There are a number of enhancements to the macro programming language (see page 60).

Product licences

The type of licence you are using to run PowerMILL is now displayed in the title bar of the main window. It can be one of the following types:

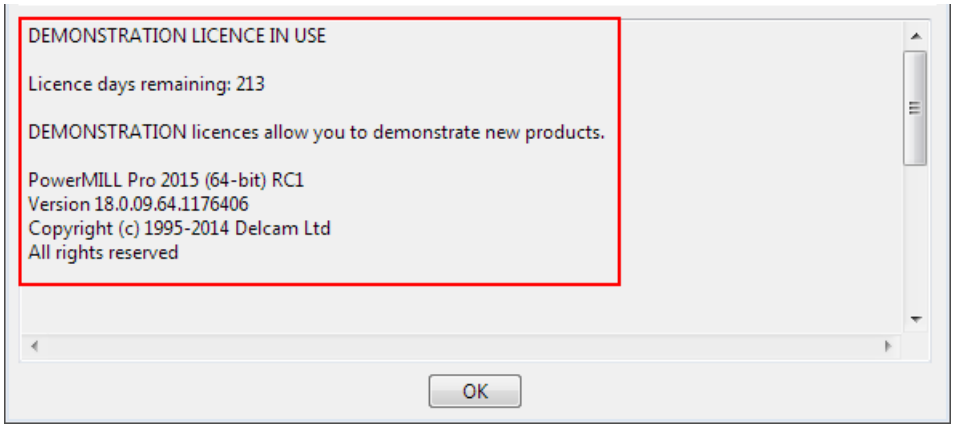
- ACADEMIC licence
- DEMONSTRATION licence
- FULL licence
- TIME-LIMITED licence

For example:



In addition, when fewer than 31 days remain of your licence, the number of days left is shown.

More details of the licence are also displayed in the **About** dialog, including, where applicable, the number of days until it expires. For example:



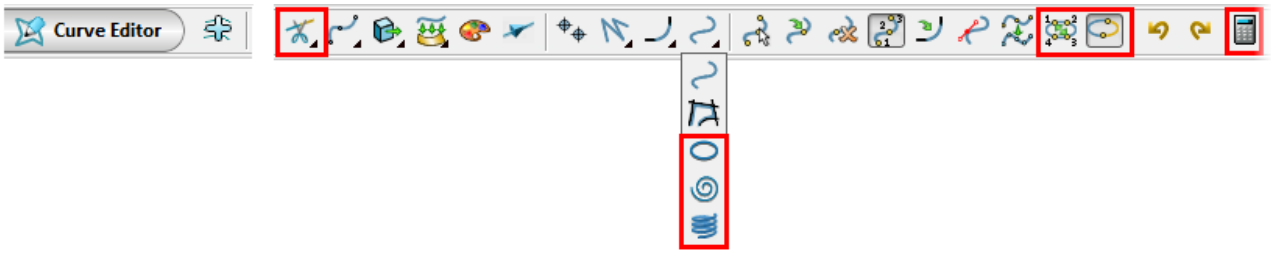
To open the dialog, select the **Help > About** menu option.

The details shown in the dialog depend on the type of licence. For more information about your licence, see 'Delcam Ltd Software Licence and Terms of Supply' in the online help or contact your sales representative.

Preparing toolpaths

Curve editor enhancements

There are several new buttons on the **Curve Editor** mode toolbar and enhancements to existing buttons.



New buttons

Ellipse — Click to create an ellipse by specifying its centre, major axis length and minor axis length (see page 7).

Spiral — Click to create a spiral by specifying the start radius, end radius and number of turns (see page 10).

Helix — Click to create a helix by specifying the height, pitch, number of turns, and radii (see page 16).

You cannot double-click an ellipse, spiral or helix to edit them.

Renumber curve points — Click to make the currently selected point in a curve the first point. Use this in conjunction with the **Curve open** button to choose where to open a closed curve (see page 22).

You must have a closed curve and a selected point to enable this option.

Curve Open — The button is toggle: Click to change an open curve to a closed curve. Click to change a closed curve to an open curve (see page 25).

Calculator — Click to display the **Calculator/Measure** dialog from within the **Curve Editor** mode toolbar. This works in exactly the same way as the **Calculator** and **Measure** buttons on the **Main** toolbar. In previous versions of PowerMILL, you couldn't access this functionality from within the curve editor.

Enhancements to existing buttons

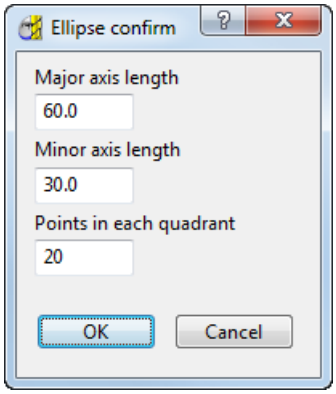
The **Cut item** now enables you divide a curve into a number of equal pieces (see page 26).

The existing functionality of cutting a curve at a specific point is unchanged.

Ellipse dialog

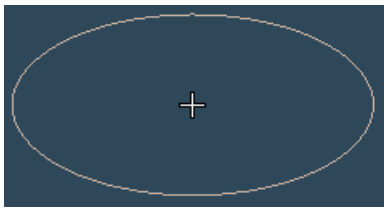
Use the **Ellipse** dialog to create an ellipse.

To display the **Ellipse** dialog:



1 Display the **Curve Editor** mode toolbar.

2 On the **Curve Creation** pull-out toolbar, click **Ellipse** , which attaches an ellipse to the cursor.



3 Locate the centre of the ellipse (either graphically or entering the coordinates).

Major axis length — Enter the major axis length by entering a value or selecting it graphically.

Minor axis length — Enter the minor axis length by entering a value or selecting it graphically.

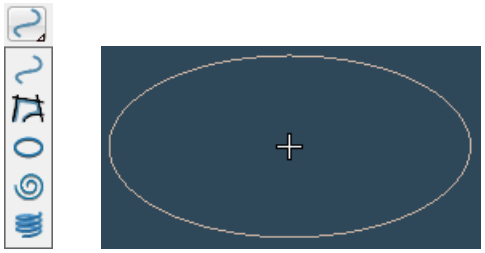
Points in each quadrant — Enter the number of points in each quadrant.

For more information, see ‘Creating an ellipse example’ (see page 8).

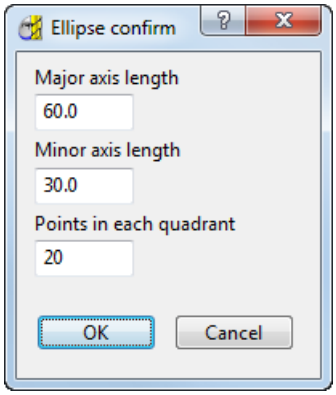
Creating an ellipse example

This example shows how to create an ellipse using the **Curve Editor** mode toolbar.

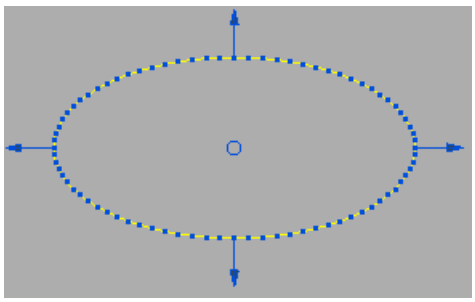
1 On the **Curve Creation** pull-out toolbar, click **Ellipse**  , which attaches an ellipse to the cursor.



2 Click to locate the centre of the ellipse, or enter the coordinates in  on the **Status** bar to display the **Ellipse** dialog.



PowerMILL also draws an ellipse.



The ellipse is yellow and has:

- blue handles that enable you to edit the **Major axis length** and **Minor axes length** graphically.
- blue points showing the number and location of the points in each quadrant.
- a blue circle at the centre that enables you to edit the location of the ellipse graphically.

3 Enter suitable values, for example:

a **Major axis length** of 20.

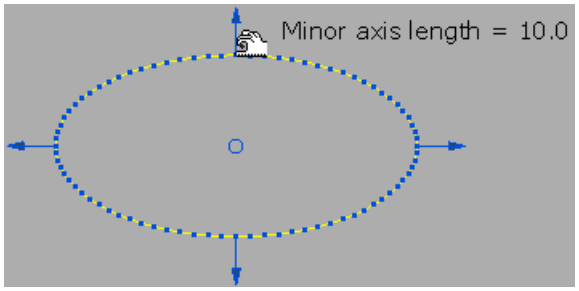
b **Minor axis length** of 10.

c **Points on each segment** of 20.

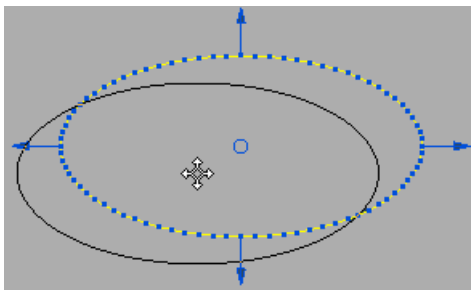
d Click **OK** to create the ellipse.

*Clicking **OK** creates a spline curve that you cannot edit.*

Alternatively, you can create the ellipse graphically. For example, if you select the minor-axis-length handle (the cursor changes to) and drag it to a new position, this changes the length.



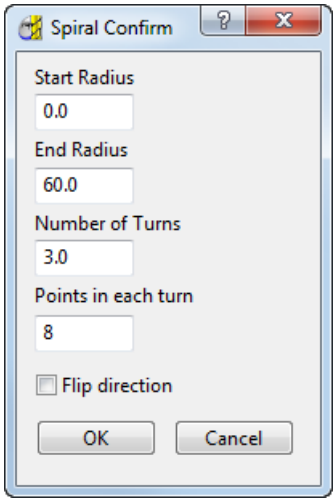
You can move the ellipse by selecting the centre handle (the cursor changes to) and dragging it to a new position.



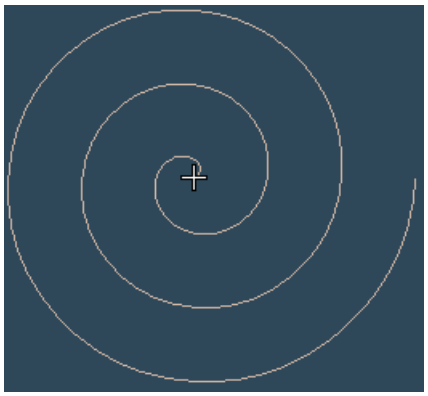
Spiral dialog

Use the **Spiral** dialog to create a spiral.

To display the **Spiral** dialog:



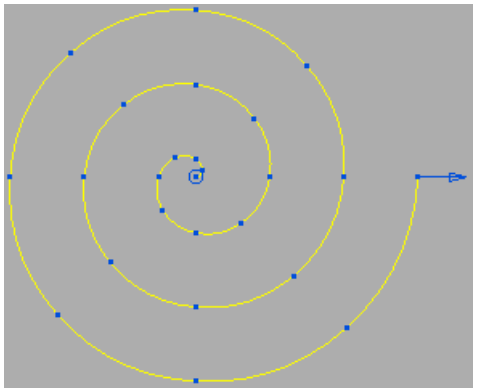
1. Display the **Curve Editor** mode toolbar.
2. On the **Curve Creation** pull-out toolbar, click **Spiral**  , which attaches a spiral to the cursor.



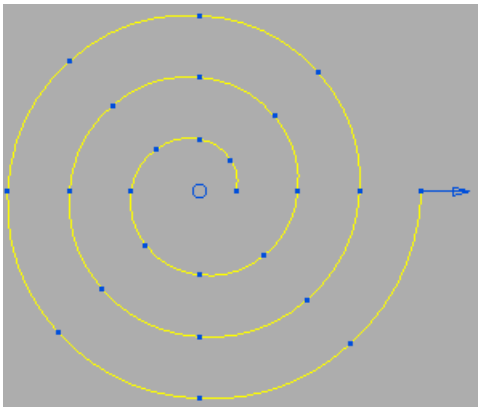
3. Locate the centre of the spiral (either graphically or entering the coordinates).

Start radius — Enter the radius at the start of the spiral by either entering a value or selecting it graphically.

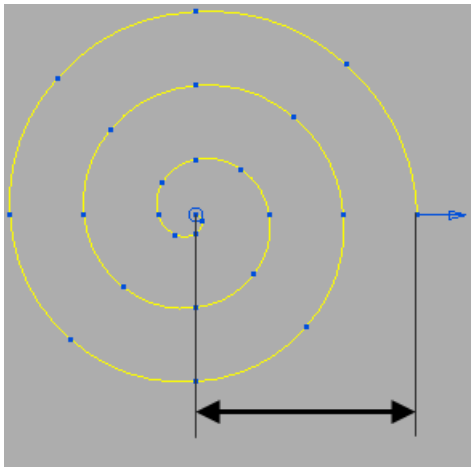
Start radius of **0**



Start radius of **10**:

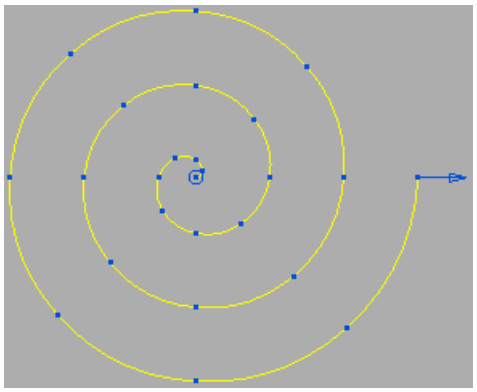


End radius — Enter the radius at the end of the spiral by either entering a value or selecting it graphically.

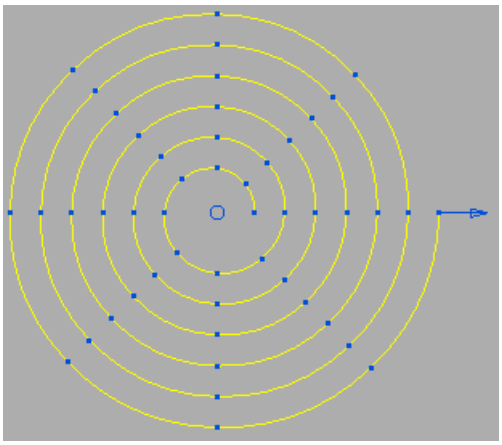


Turns — Enter the number of revolutions in the spiral.

Number of turns of 3:



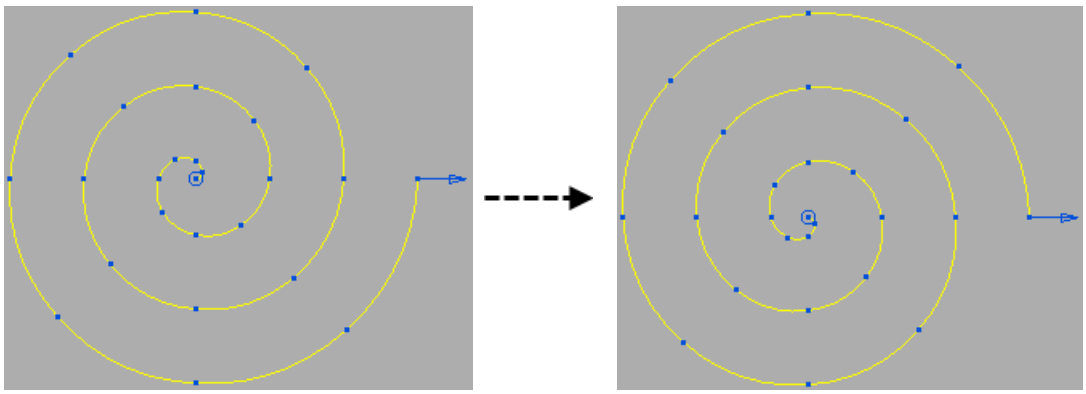
Number of turns of 6:



Points in each turn — Enter the number of points in each revolution.

Flip direction — Select to change from an anticlockwise to a clockwise spiral.

Selecting **Flip direction** converts:

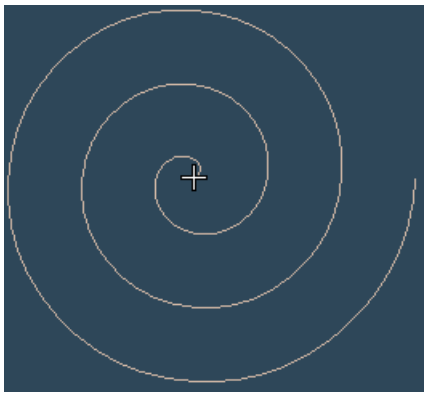


For more information, see 'Creating a spiral example' (see page 13).

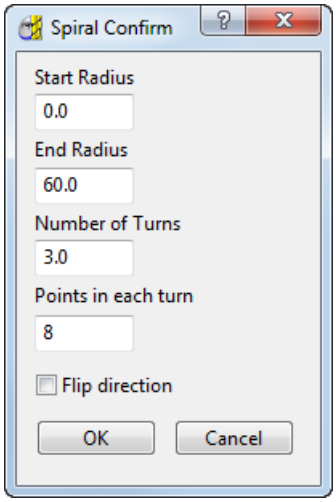
Creating a spiral example

This example shows how to create a spiral using the **Curve Editor** mode toolbar.

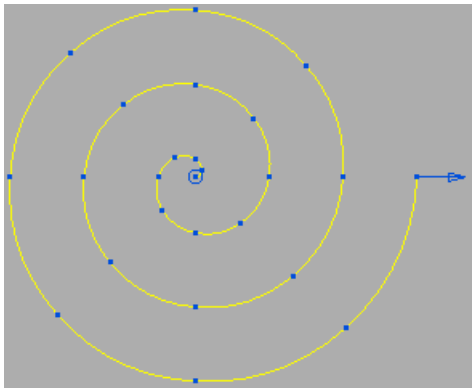
1 On the **Curve Creation** pull-out toolbar  , click **Spiral**  , which attaches a spiral to the cursor.



2 Click to locate the centre of the spiral, or enter the coordinates in  on the **Status** bar, to display the **Spiral** dialog.



PowerMILL also draws a spiral.



The spiral is yellow and has:

- a blue handle that enables you to edit the **End radius** graphically.
- blue points showing the number and location of the points in each turn.
- a blue circle at its centre that enables you to edit the location of the spiral graphically.

3 Enter suitable values, for example:

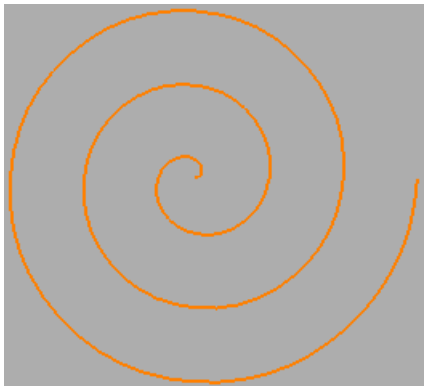
a **Start radius** of **0**.

b **End radius** of **60**.

c **Number of turns** of **3**.

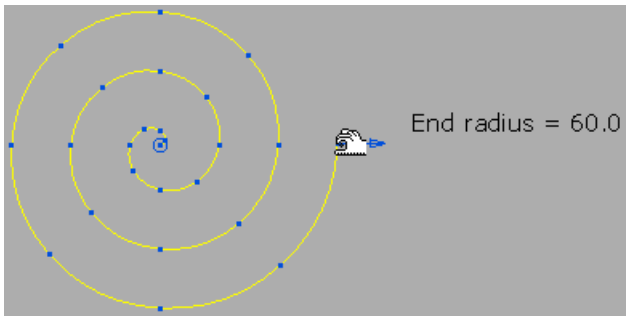
d **Points in each turn** of **8**.

e Click **OK**.

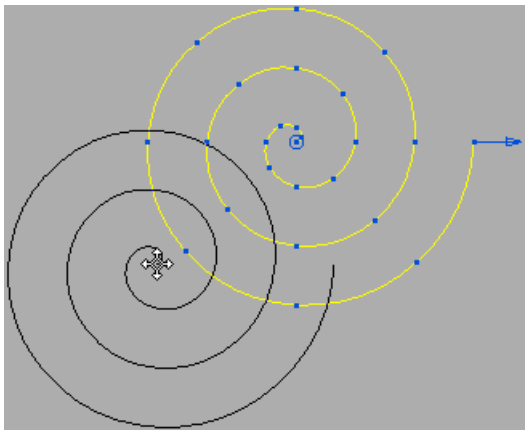


*Clicking **OK** creates a spline curve that you cannot edit.*

Alternatively, you can create the spiral graphically. For example, if you select the end radius-handle (the cursor changes to) and drag it to a new position, the end radius changes.



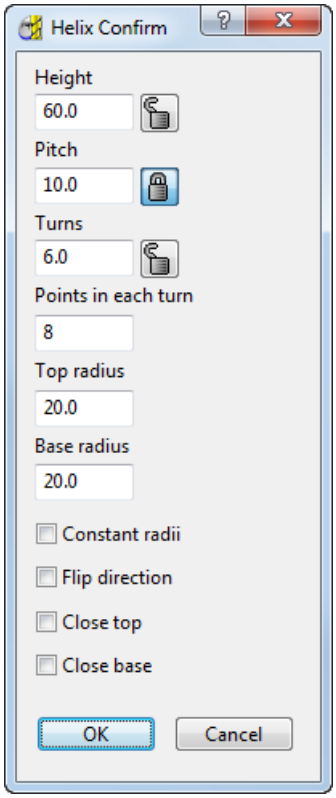
You can move the spiral by selecting the centre handle (the cursor changes to) and dragging it to a new position.



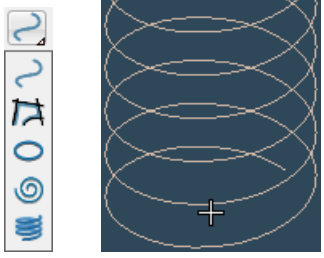
Helix dialog

Use the **Helix** dialog to create a helix.

To display the **Helix** dialog:

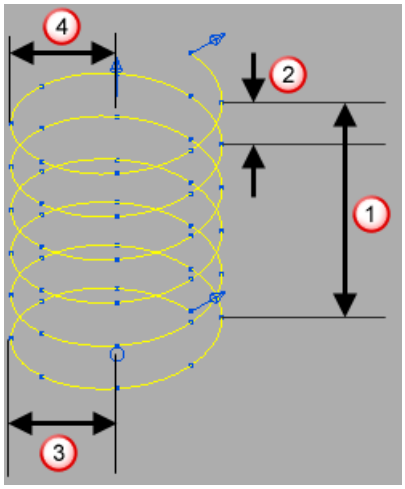


1. Display the **Curve Editor** mode toolbar.
2. On the **Curve Creation** pull-out toolbar, click  , which attaches a helix to the cursor.



3. Locate the centre of the base of the helix (either graphically or entering the coordinates).

Height — Enter the overall height of the helix.



1. Height
2. Pitch
3. Base radius
4. Top radius

*In this case, the **Base radius** is the same as the **Top radius**.*

Pitch — Enter the distance between successive turns.

Turns — Enter the number of revolutions in the helix.

Height, **Pitch**, and **Turns** are inter-related. You can edit any two while locking the third.

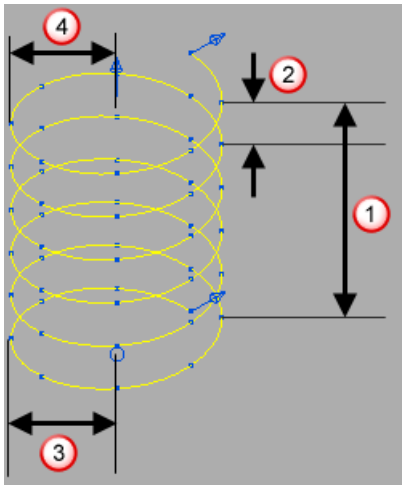
Unlocked — When displayed, you can edit the value. When unlocked, changing other values may change unlocked values.

Locked — When displayed, you cannot edit the value. When locked, changing other values doesn't change locked values.

*Click button to toggle between **Locked** and **Unlocked**.*

Points in each turn — Enter the number of points in each revolution.

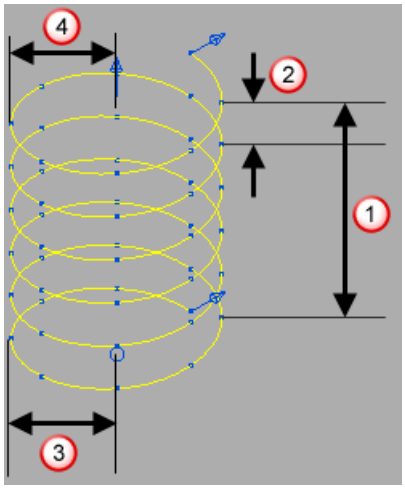
Top radius — Enter the radius at the top of the helix.



1. Height
2. Pitch
3. Base radius
4. Top radius

*In this case, the **Base radius** is the same as the **Top radius**.*

Base radius — Enter the radius at the bottom of the helix.

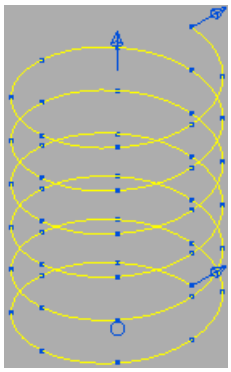


1. Height
2. Pitch
3. Base radius
4. Top radius

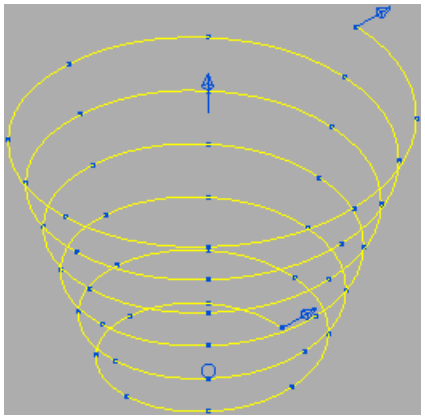
*In this case, the **Base radius** is the same as the **Top radius**.*

Constant radius — When selected creates a straight helix. When deselected enables you to create a tapered helix.

Straight helix

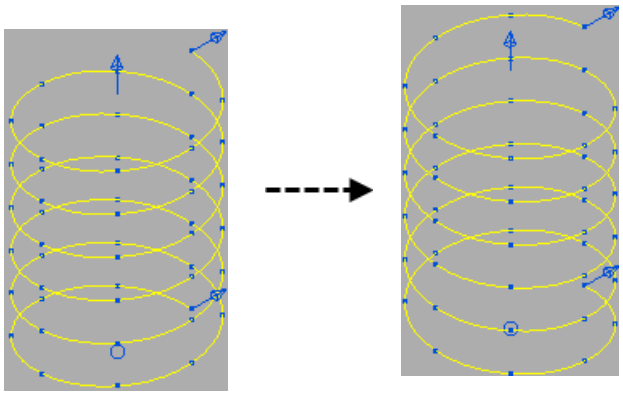


Tapered helix



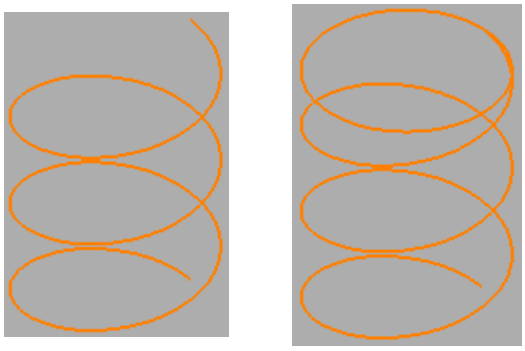
Flip direction — Select to change from an anticlockwise to a clockwise helix.

Selecting **Flip direction** converts:



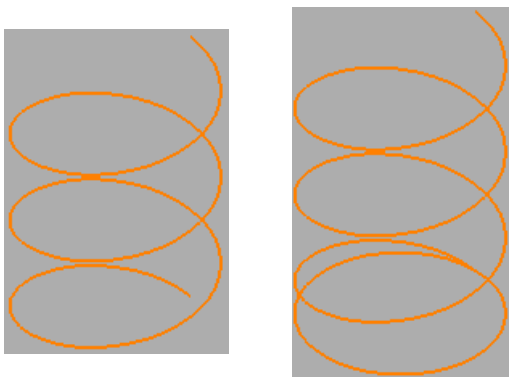
Close top — Select to end the helix with a circle at its top.

Deselected Selected



Close base — Select to end the helix with a circle at its base.

Deselected Selected

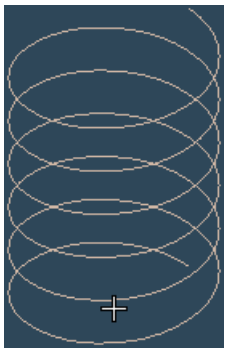


For more information, see ‘Creating a helix example’ (see page 20).

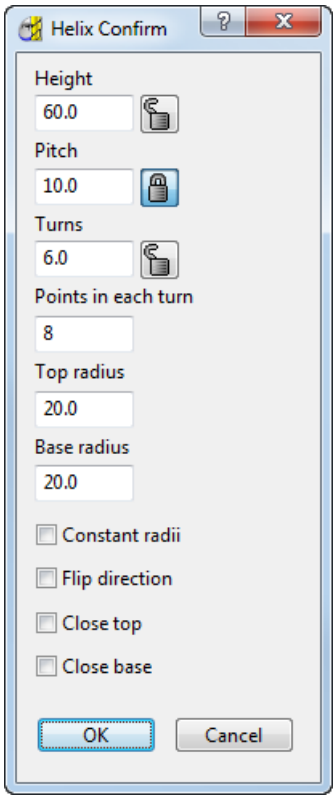
Creating a helix example

This example shows how to create a helix using the **Curve Editor** mode toolbar.

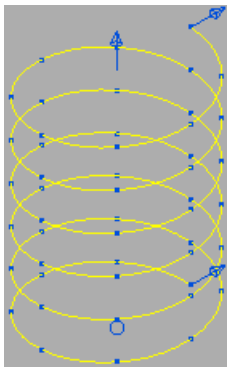
1 On the **Curve Creation** pull-out toolbar  , click **Helix**  , which attaches a helix to the cursor.



2 Click to locate the helix at the centre of its base, or enter the coordinates in  on the **Status** bar to display the **Helix** dialog.



PowerMILL also draws a helix.



The helix is yellow and has:

- blue handles that enable you to edit the **Top radius**, **Bottom radius** and **Height** graphically.
- blue points showing the number and location of the points in each turn.
- a blue circle at the centre of its base that enables you to edit the location of the helix graphically.

3 Enter suitable values, for example:

a **Height** of **60**.

b **Pitch** of **10**.

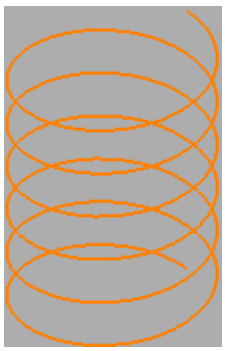
c **Number of turns** of **6**.

d **Points in each turn** of **8**.

e **Top radius** of **20**.

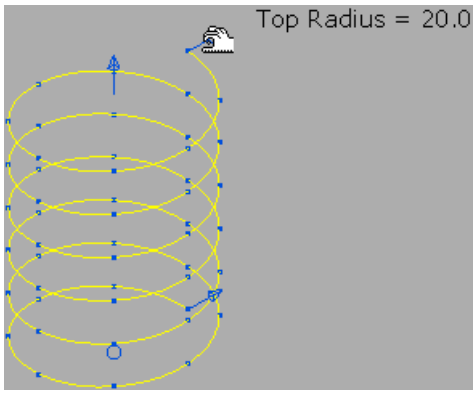
f **Base radius** of **20**.

g Click **OK**.

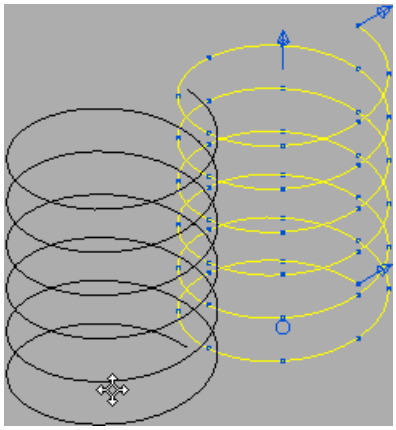


*Clicking **OK** creates a spline curve that you cannot edit.*

Alternatively, you can create the helix graphically. For example, if you select the top-radius handle (the cursor changes to) and drag it to a new position, the top radius changes.

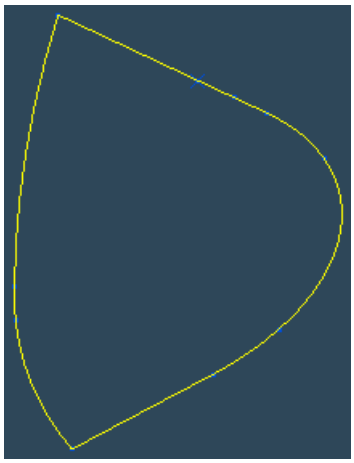


You can move the helix by selecting the centre handle (the cursor changes to) and dragging it to a new position.

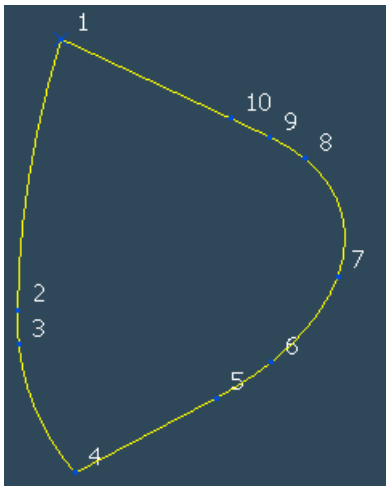


Opening a closed curve at a specific location

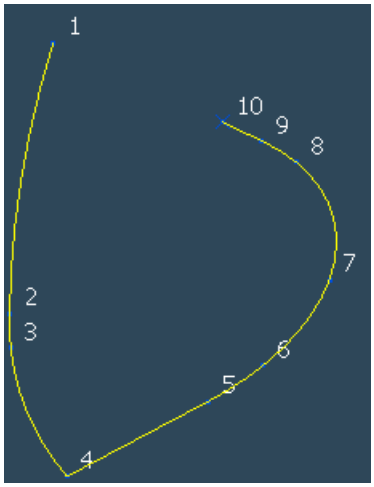
This example shows how to open a closed curve at a specific location. Starting with this curve:



1 Click **Number points** on the **Curve Editor** mode toolbar. This numbers all the points on the curve.



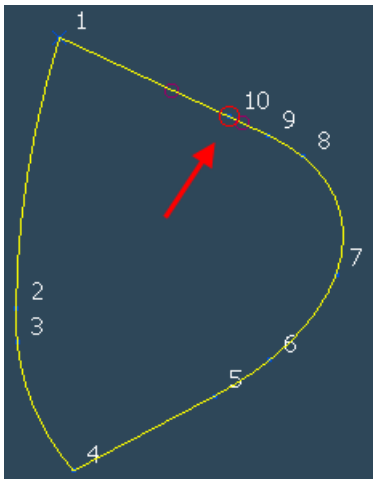
2 Click **Curve closed** on the **Curve Editor** mode toolbar.



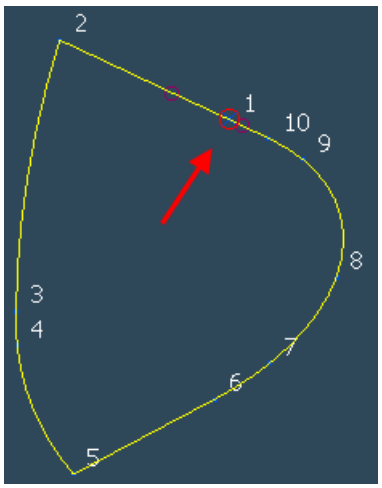
PowerMILL opens the curve by deleting the segment between the first and last point. In this case, between points 1 and 10.

3 Click **Undo** on the **Curve Editor** mode toolbar to display the closed curve in step 1.

4 Select point 10.

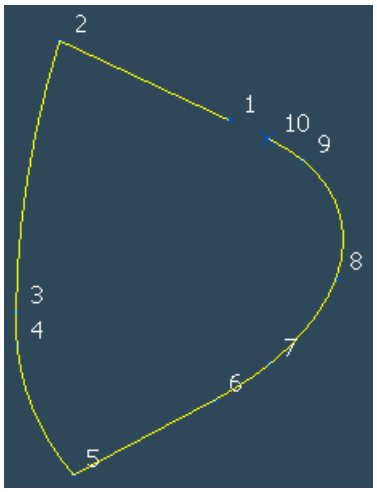


5 Click **Renumber curve points** on the **Curve Editor** mode toolbar.



This makes the selected point **point 1**.

6 Click **Curve closed** on the **Curve Editor** mode toolbar.



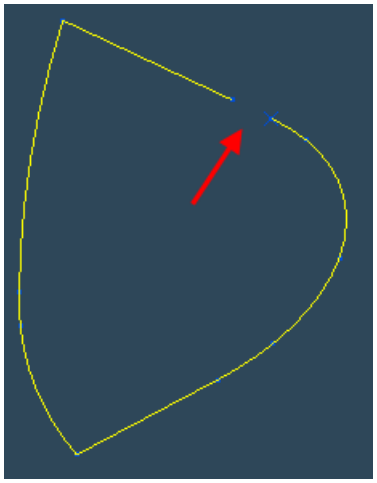
This still opens the curve between point 1 and 10 but, because the points have been renumbered, the open segment is in a different location.

Open and Close curves

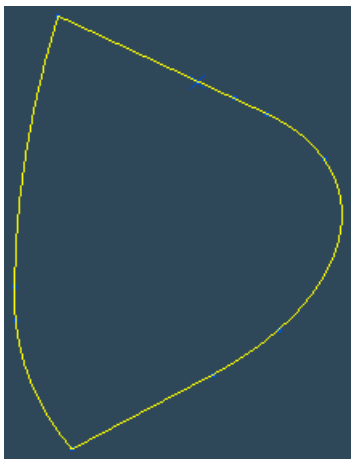
Use the function to open or close a curve.

Open curves

Starting with an open curve

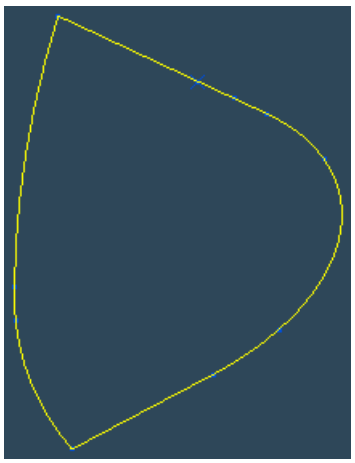


Clicking  converts it to a closed curve.

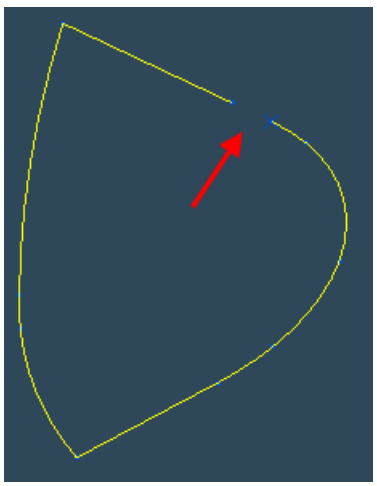


Close curves

Starting with a closed curve



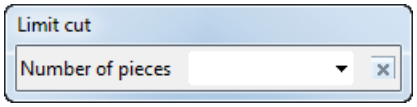
Clicking  converts it to an open curve.



When converting a closed curve to an open curve, PowerMILL removes the segment between the first and last point. For more information, see 'Opening a closed curve at a specific place' (see page 22).

Limit Cut enhancements

When you select the **Cut item** button from the **Segments** pull-out toolbar  , PowerMILL now displays the **Limit cut** toolbar. This enables you to choose whether to cut a curve at a user-defined point or to cut a curve into a specified number of segments of equal length. Previously, you could cut a curve only at a user-defined point.



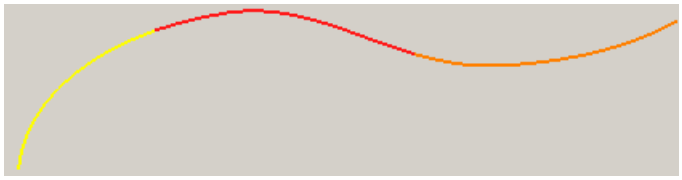
Number of pieces — Enter the number of pieces, of equal-length, you want to cut the curve into. If you do not enter a number, PowerMILL cuts the curve where you select it.

For example, starting with this curve:

1 Select the curve.

The curve is yellow when selected.

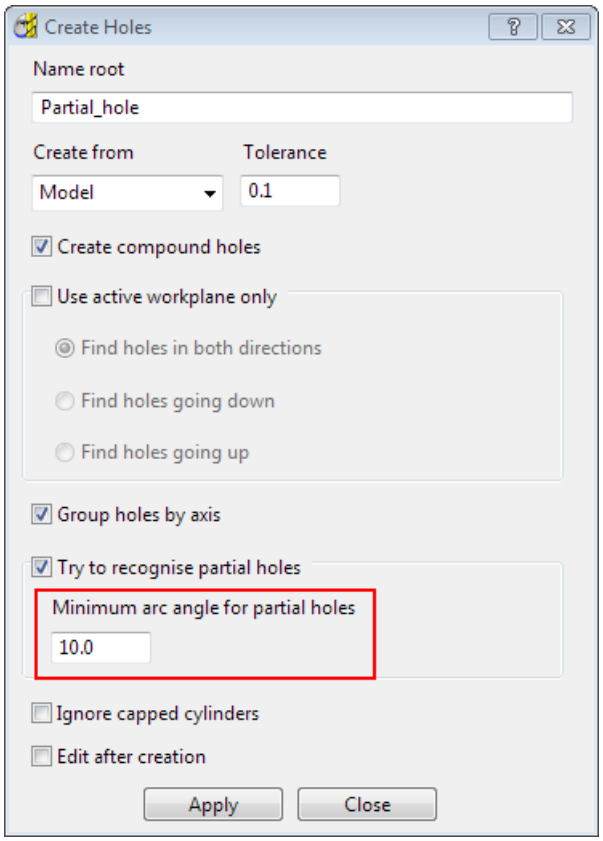
2 Enter a **Number of pieces** of 3. This cuts the curve into three segments of equal length.



For more information on cutting a curve at a specific point, see ‘Cutting a curve at a user-defined point’.

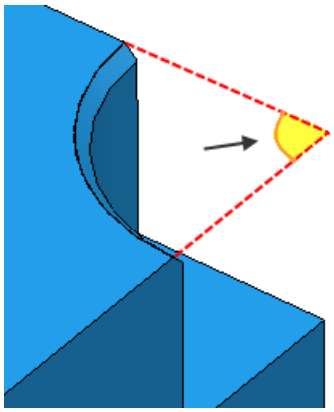
Converting partial holes to (complete) holes

On the **Create Holes** dialog and on a strategy dialog's **Holes** page, there is a new **Minimum arc angle for partial holes** field that enables you to convert partial holes into (complete) holes more easily.

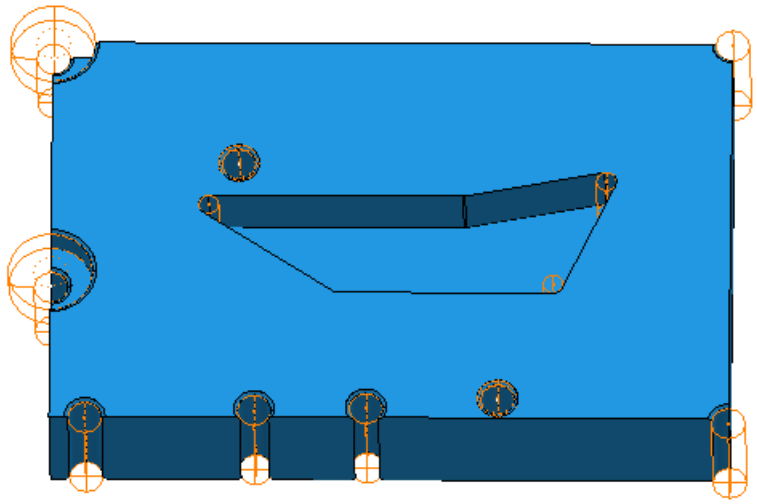


Minimum arc angle for partial holes — Enter a value to specify which partial holes PowerMILL converts to holes. To be converted, partial holes must have an arc angle, or central angle, greater than or equal to the specified arc angle. Partial holes with an arc angle less than the specified value remain as partial holes.

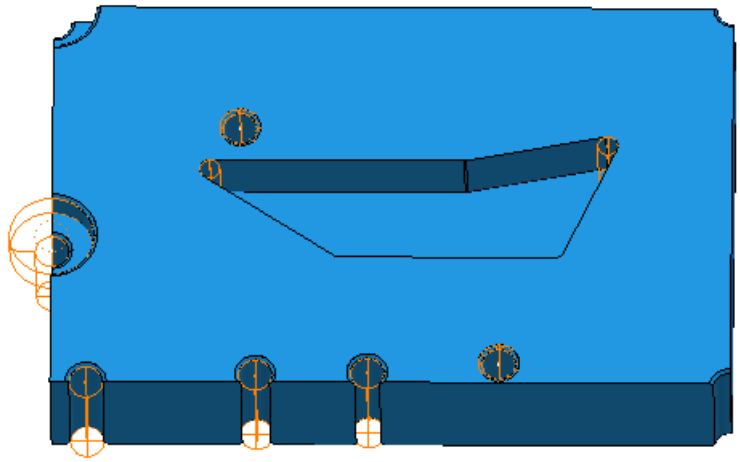
An arc angle, or central angle, is the angle produced by an arc's radii intersecting at the centre of its circle.



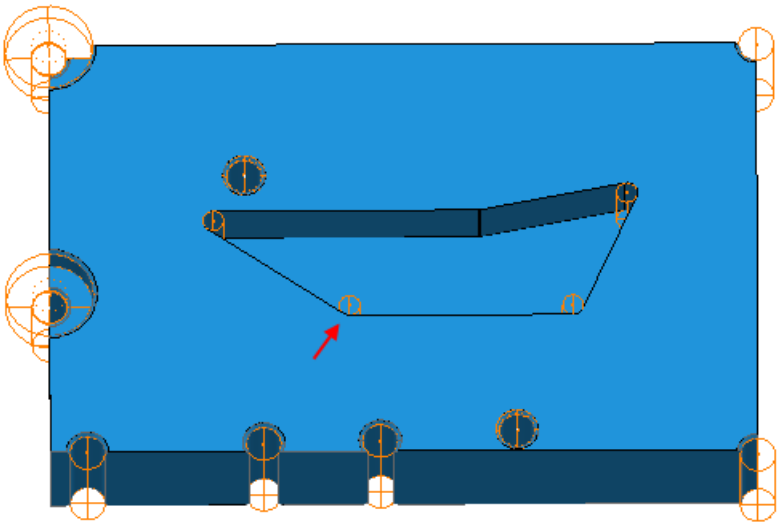
By specifying an arc angle of **20**, PowerMILL converts all the partial holes to (complete) holes:



By specifying an arc angle of **100**, PowerMILL converts eight of the twelve partial holes to (complete) holes:



To identify the shallow angle, you need to specify an arc angle of **20** and a **Tolerance** of **0.01.**



Generating toolpaths

Working with toolpath strategies

There are a number of improvements that make it easier to work with toolpath strategies.

Strategy Selector dialog

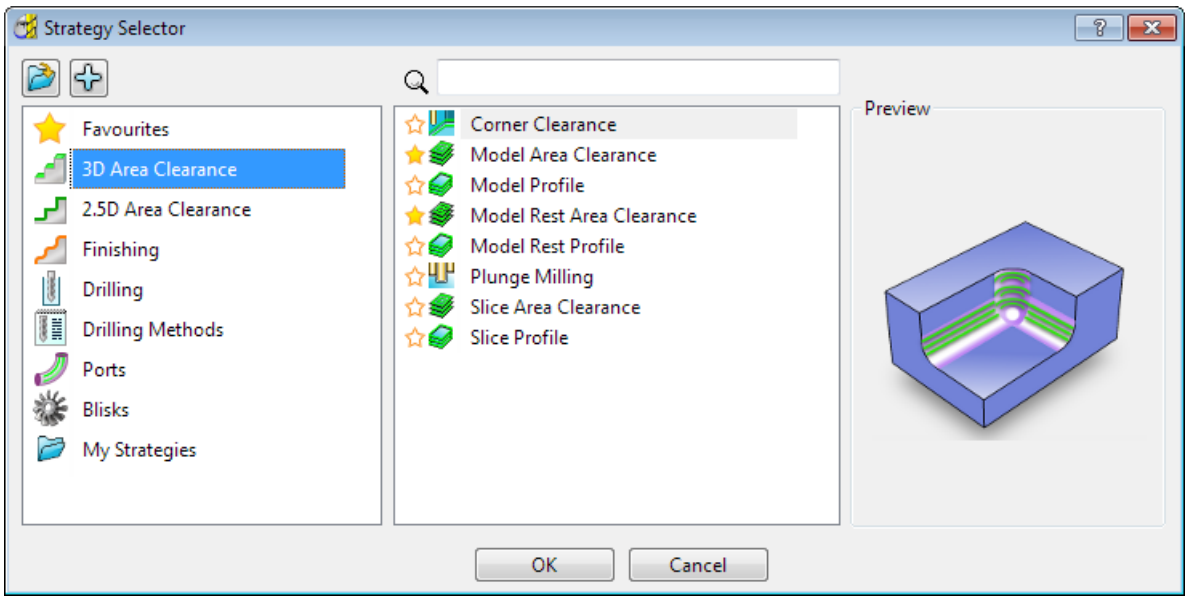
The dialog has a new layout so you can find strategies more easily. The dialog also includes two new functions that enable you to:

- add a favourite strategy to the **Favourites** category by selecting the star ★ next to the strategy. To remove the strategy from the **Favourites** category, deselect ☆ the star.

  *PowerMILL also displays your favourite strategies in the Create Toolpath list on the **Main** toolbar.*

- add new categories to the **Strategy Selector** dialog to organise your custom toolpath strategies by clicking the new **Add category** button.

  For more information on using the **PowerMILL Paths** dialog when saving a toolpath as a custom strategy, see 'Saving a toolpath as a custom strategy' (see page 30).



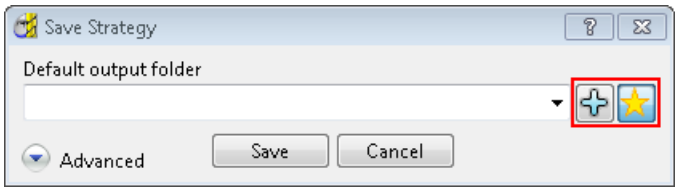
Saving a custom strategy

Saving a toolpath as a custom strategy is easier than in previous versions of PowerMILL. In addition:

- There are two new buttons on the **Save Strategy** dialog:

  Add path — Click to create a new category in which to save the toolpath strategy.

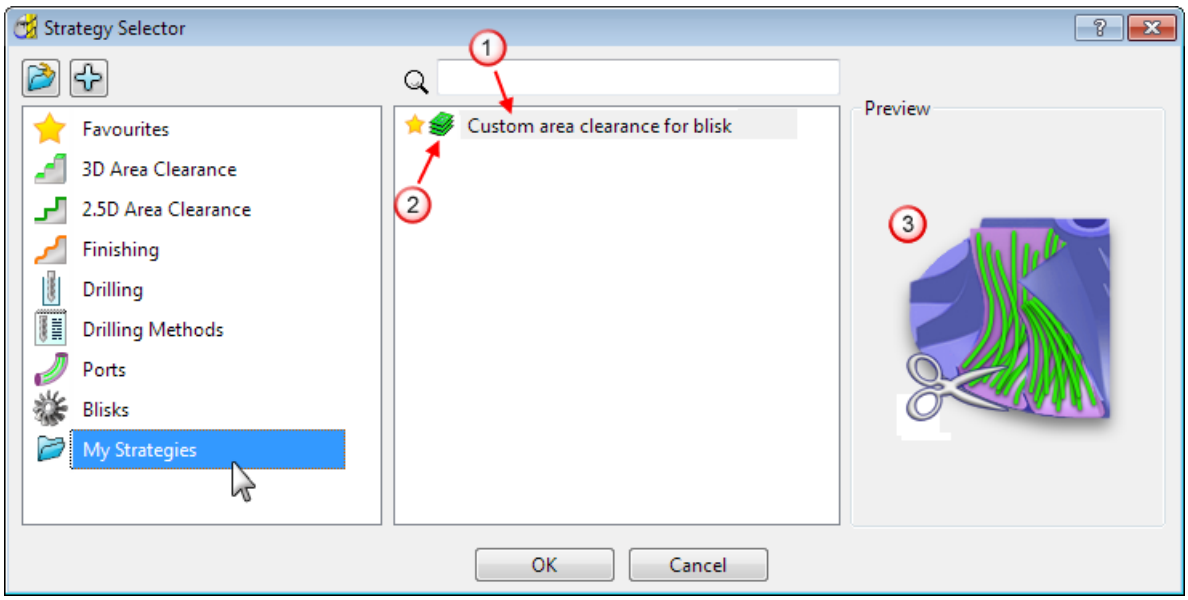
  Add to favourites — Select to add the toolpath strategy to the **Favourites** category on the **Strategy Selector** dialog and the Create Toolpath list on the **Main** toolbar.



- The **Save as Template** option on the individual Toolpath context menu is now called **Save as Strategy**.

Saving a toolpath as a custom strategy

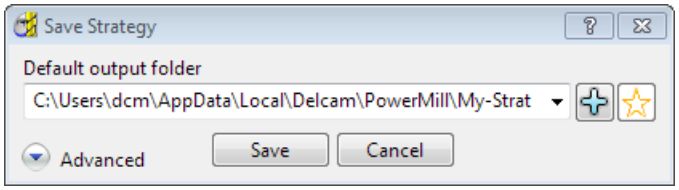
You can save toolpaths that include commonly used parameters as a custom strategy. You can then use the strategy (1) to create new toolpaths with the same parameters. After saving the custom strategy, you can also add a strategy icon (2) and a preview graphic (3).



To save a toolpath as a custom strategy:

1 In the Explorer, right-click a toolpath and select **Save as Strategy**.

This displays the **Save Strategy** dialog.



2 Specify which category to save the strategy.

By default, PowerMILL saves the strategy to the **My Strategies** category (C:\Users\xxx\AppData\Local\Delcam\PowerMILL\My Strategies).

To save the strategy in a different category:

a Click  .

This displays the **PowerMILL Paths** dialog.

b Click  and specify a new path.

c Click **OK**.

If you create a new category with the same name as an existing category, PowerMILL merges the content of the two categories.

d Click **Close** on the **PowerMILL Paths** dialog.

3 Choose to save all, or only a selection, of the toolpath parameters. By default, PowerMILL saves all toolpath parameters.

To save a selection of parameters:

a Click **Advanced** and select **Save selected parameters only**.

b Select the parameters you want to save.

4 If you want to add the strategy to the **Favourites** category on the **Strategy Selector** dialog, select the **Add to Favourites** button  .

5 Click **Save**.

This displays the **Save Toolpath Strategy** dialog.

6 Enter a filename and click **Save**.

This displays the strategy in the **Strategy Selector** dialog.

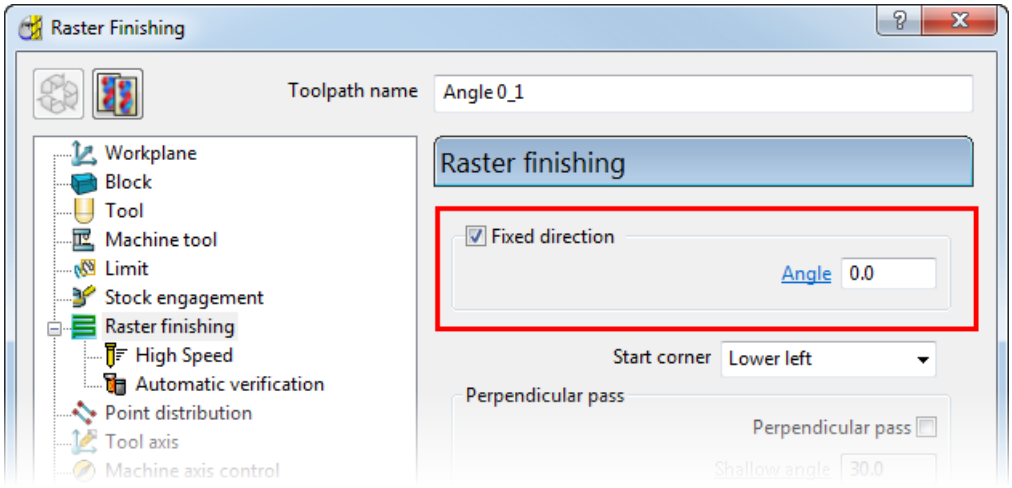
Adding a strategy icon and preview graphic

- To add a strategy icon, in a Windows Explorer window, add the icon graphic file to the folder where the strategy is saved. The graphic must:
  - be a .ico file
  - have the same filename as the strategy file; and
  - be 180 x 180 pixels (3.9 cm x 3.9 cm) or smaller.
- To add a preview graphic, in a Windows Explorer window, add the preview graphic file to the folder where the strategy is saved. The graphic must:
  - be an .png file
  - have the same filename as the strategy file; and

be 16 x 16 pixels (0.34 cm x 0.34 cm) or smaller.

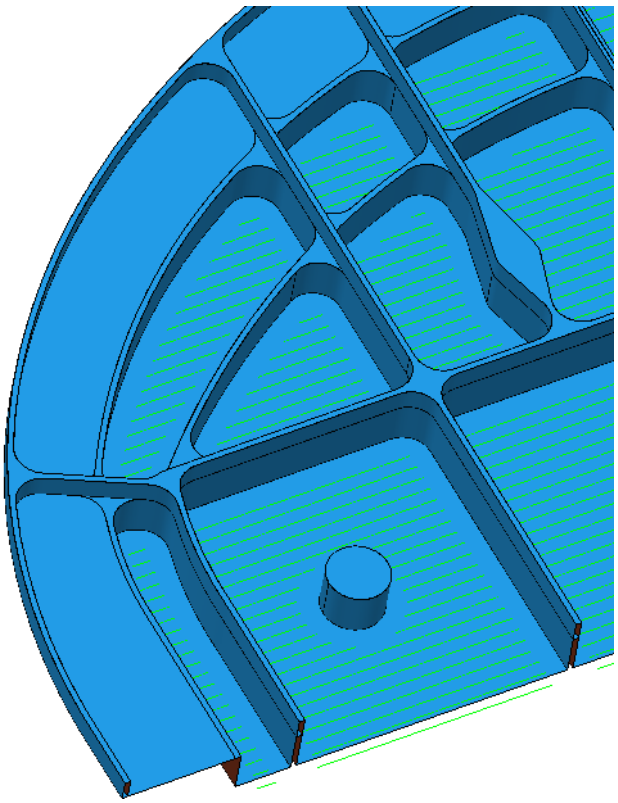
Raster finishing improvements

You now have the ability to let PowerMILL determine the most appropriate angle for each region when raster machining. In previous versions, you had to specify the angle. This provides the same functionality that already exists in steep and shallow finishing and face milling.

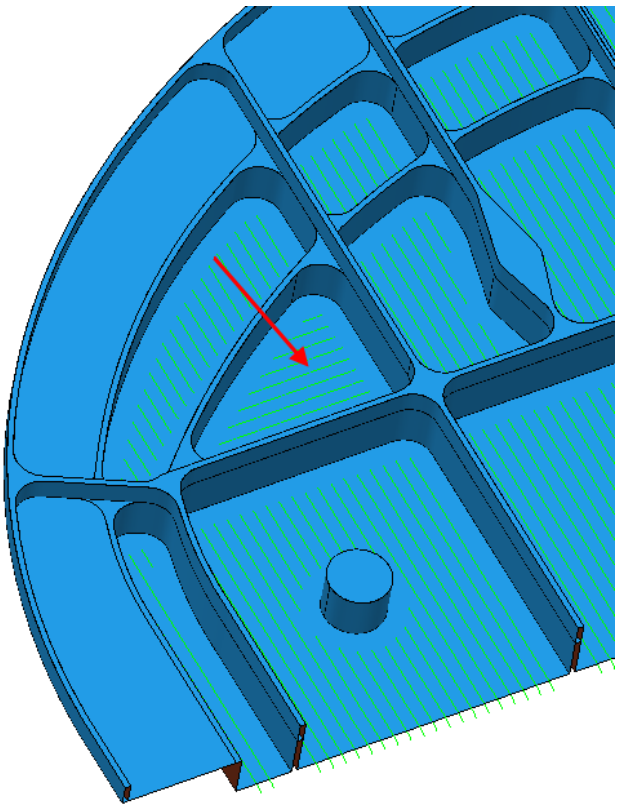


The following example uses the aero.dgk model in the **Examples** folder.

Selecting **Fixed direction** and entering an **Angle** of 0° gives:



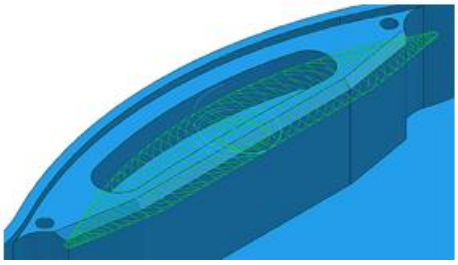
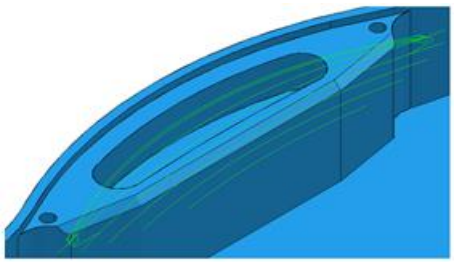
Deselecting **Fixed direction** gives:



Approaching flats from outside of stock

Vortex can approach flat areas from outside of stock instead of always ramping into it. This enables PowerMILL to generate Vortex toolpaths that are faster to machine and are compatible with a wider selection of tools.

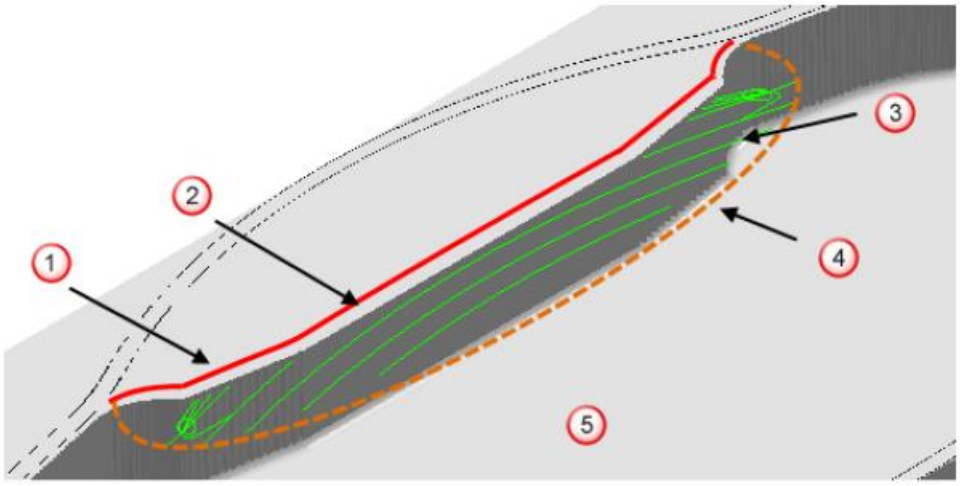
The following table compares a 2015 and a 2014 R2 Vortex toolpath machining a flat area.

PowerMILL 2014 R2	PowerMILL 2015
PowerMILL ramps	PowerMILL approaches from the outside
	
Machining time: 2.59 min Toolpath length: 2911 mm	Machining time: 1.14 min Toolpath length: 1789 mm

To approach from outside the stock, PowerMILL:

- extends a section of the flat area beyond the stock and into an area already machined; and;
- fills the extended section with cutting moves.

By extending the section to an area already machined, the machine tool can approach the flat area in open space.



1. Edge of flat area
2. Stock
3. Extended section of flat area filled with cutting moves
4. Outside edge of extended section
5. Area already machined

Criteria for toolpath to approach from outside stock

PowerMILL only extends a section of the flat area if the extended section:

- has an outside edge that the machine tool can approach.
- does not gouge the model.
- is wide enough to be profile smoothed successfully.
- can reach an area already machined within the distance of one tool diameter.

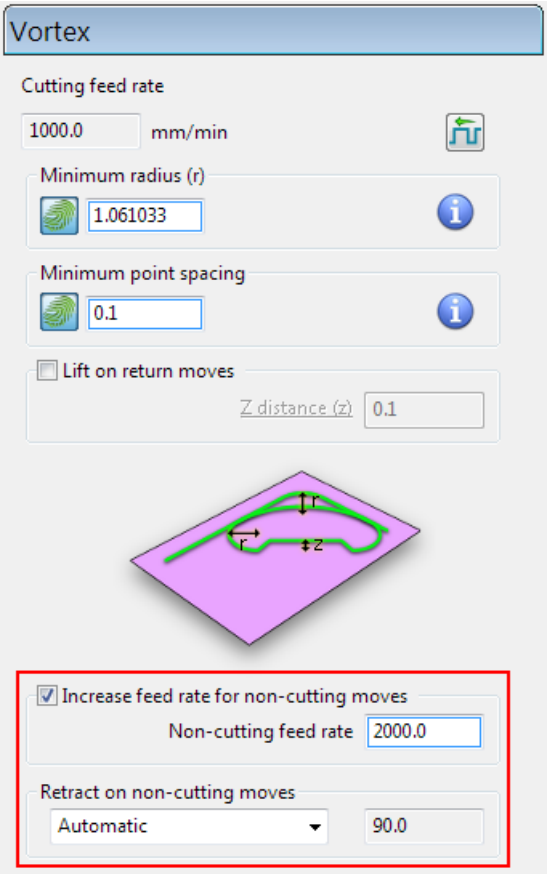
If the extended section does fails to meet the criteria, PowerMILL does not extend the flat area and instead uses a ramp move to approach the toolpath.

All area clearance toolpaths can also approach flat areas from outside of stock.

Reducing the machining time of Vortex toolpaths

The new functions on the Vortex strategy page enable you to reduce machining time by:

- increasing the feed rate of non-cutting moves; and
- replacing long non-cutting moves with more efficient lift moves.



Increase feed rate for non-cutting moves — Select to enable PowerMILL to speed up non-cutting moves.

Non-cutting feed rate — By default, PowerMILL uses a non-cutting feed rate of 2***Cutting feed rate**, but you can enter a different value in the field.

If the machine tool exhibits undesirable behaviour, such as jerking or vibrating, you can prevent this by:

- *specifying a lower non-cutting feed rate;*
- *increasing the size of the minimum radius; or*
- *increasing the minimum point spacing.*

If PowerMILL identifies any non-cutting moves that are too short for the machine tool to accelerate to the (faster) non-cutting feed rate and then decelerate back to the (slower) cutting feed rate, PowerMILL completes these non-cutting moves at a percentage of the non-cutting feed rate.

Retract on non-cutting moves — Specify whether PowerMILL replaces non-cutting moves that exceed a certain length with more time-efficient lift moves.

Never — PowerMILL keeps all non-cutting moves.

Automatic — PowerMILL replaces non-cutting moves that exceed an automatically-determined length with lift moves.

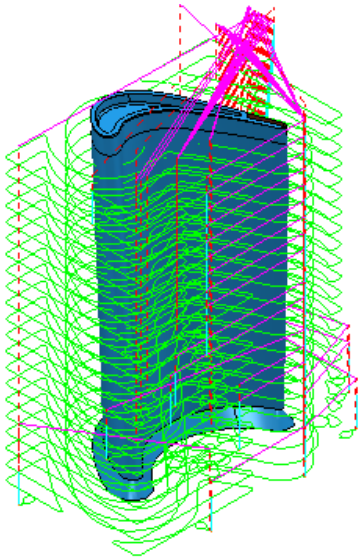
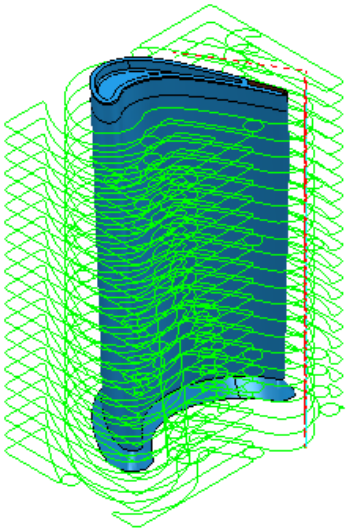
Longer than — Enter the maximum length of non-cutting moves.

*Please also note that the **Lift on air moves** option is now called **Lift on return moves**.*

Fewer lift moves in roughing toolpaths

PowerMILL can create roughing toolpaths with significantly fewer lift moves.

The following table compares a 2015 and a 2014 R2 area clearance toolpath.

PowerMILL 2014 R2	PowerMILL 2015
	
Number of lifts: 62 Machining time: 15.46 min	Number of lifts: 21 Machining time: 13.39 min

*The roughing toolpath must have a **Style** of **Offset all**, and the **Maintain cut direction** option on the **Offset** page must be deselected.*

Improved tool axis and point distribution

There are a number of enhancements to tool axis calculation and the distribution of toolpath points:

- PowerMILL can create multi-axis toolpaths with significantly fewer undesirable movements of a machine tool's rotary axes (see page 39).
- The **Machine tool** page of a strategy dialog includes several new options that enable you to select the most suitable coordinate system PowerMILL uses for calculating point redistribution, tool axis smoothing, and a fixed tool axis angle (see page 40).

- There are several updates to the **Tool Axis Editing** dialog, including the **Machine tool** button, which displays the new **Machine tool** dialog. The dialog enables you to specify the most suitable coordinate system for editing the tool axis and point distribution of a calculated toolpath (see page 43).

Reducing undesirable movements of rotary axes

When machining a multi-axis toolpath, the machine tool's rotary axes can move undesirably due to one or more toolpath points that are close to the machine tool's gimbal lock. These undesirable movements can produce surface imperfections and reduce machining efficiency.

In PowerMILL 2015, there are a number of updates that enable PowerMILL to create multi-axis toolpaths with significantly fewer undesirable movements of a machine tool's rotary axes:

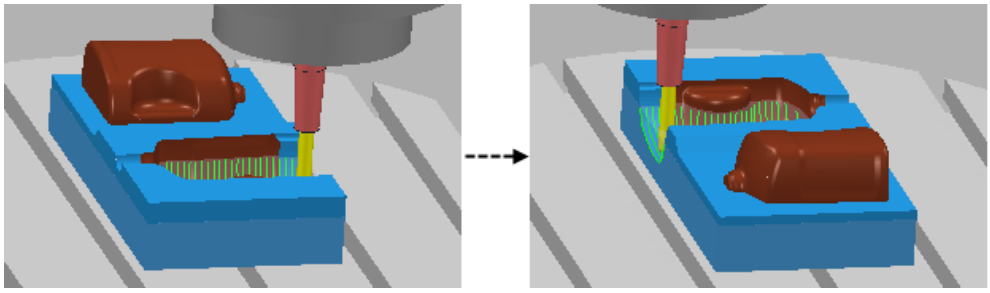
- The **Point redistribution** function is more effective at distributing toolpath points that avoid the machine tool's gimbal lock.
- There are several new options on a strategy dialog's **Machine tool** page that enable you to select the most suitable workplane PowerMILL can use for calculating point redistribution (see page 40).

These improvements also mean PowerMILL is less likely to produce clusters of toolpath points that slow down the machine tool.

What is a gimbal lock?

A gimbal lock is a specific configuration of a multi-axis machine tool that occurs when the tool axis aligns with at least one rotary axis due to the position of one or more toolpath points. This configuration results in undesirable movements because, when close to the gimbal lock, small adjustments in the tool axis can require large changes in a rotary axis.

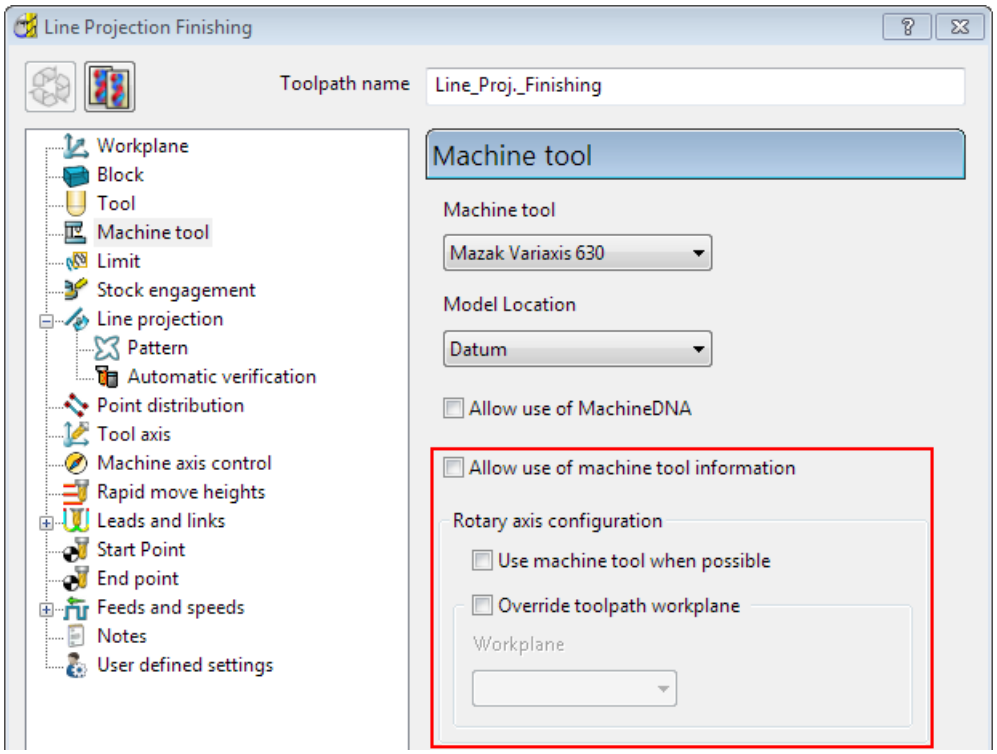
The graphic below shows the rotary axis of table-table machine tool (undesirably) rotating almost 180° in one second. This is because the machine tool machines a toolpath point at the bottom of the bottle cavity that coincides with the machine tool's gimbal lock.



Updated Machine tool page

The **Machine tool** page of a strategy dialog now includes several options that enable you to select the most suitable coordinate system PowerMILL uses for calculating point redistribution, tool axis smoothing, and a fixed tool axis angle. In previous versions of PowerMILL, you could select the toolpath workplane or a different, user-defined workplane. In PowerMILL 2015, you can also select a machine tool (.mtd file) that PowerMILL uses to calculate a suitable workplane, which can produce significantly better results.

Because the **Machine tool** page includes all of the options to configure the rotary axes, the **Tool axis** and **Point distribution** dialogs and strategy pages now include a read-only field, which displays the coordinate system selected on the **Machine tool** page, where the **Override toolpath workplane** options were displayed previously.



Allow use of machine tool information — Select to allow PowerMILL to use machine tool information to optimise toolpath calculation. As a minimum, select a machine tool from the **Machine tool** list. You can also select a workplane from the **Model location** list. If you do select a model-location workplane, it is recommended that you select the same model-location workplane as selected on the toolpath's **NC program** dialog.

Currently, PowerMILL only uses machine tool information for rotary-axis-configuration calculations. In future releases, PowerMILL will use machine tool information for more calculations.

Rotary axis configuration — The options enable you to select a suitable coordinate system PowerMILL can use for calculating the azimuth and elevation angles for redistributing points, smoothing the tool axis, and fixing the tool axis angle. If you do not select a coordinate system, PowerMILL uses the toolpath workplane.

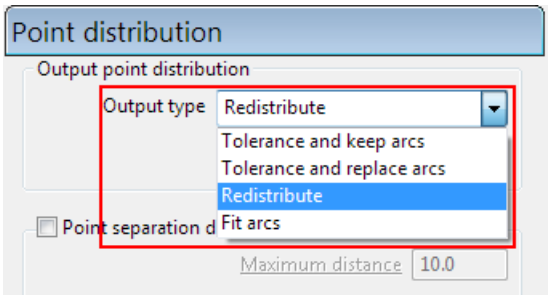
Use machine tool when possible — Select to enable PowerMILL to calculate a suitable workplane based on the machine tool (.mtd file) selected from the **Machine tool** list. You must also select **Allow use of machine tool information**.

Override toolpath workplane — Select to use a different, user-defined workplane that you select from the **Workplane** list. Using an alternative workplane can be useful if PowerMILL cannot produce good results using either the toolpath workplane or a workplane based on a machine tool's .mtd file.

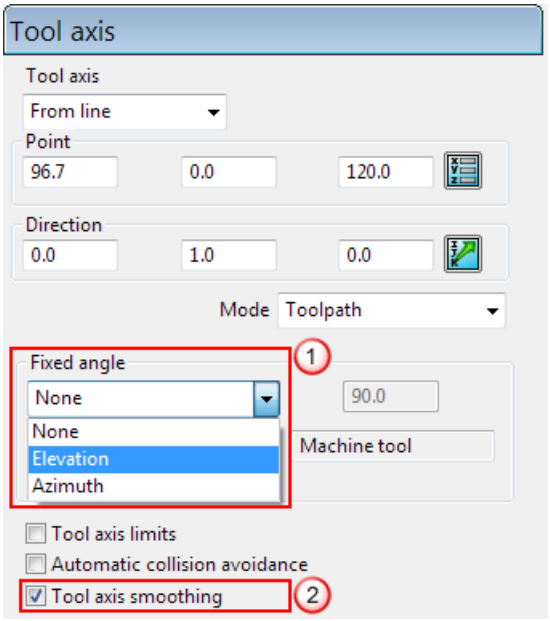
Workplane — Select a different, user-defined workplane from the list.

*For PowerMILL to use the selected workplane in rotary axis configuration calculations, you must select the relevant options on a strategy dialog's **Point redistribution** and **Tool axis** pages.*

- For point redistribution, from the **Output type** list, select **Redistribute**.



- For fixed tool axis angle, from the **Fixed angle** list, select **Elevation** or **Azimuth** (1).
- For tool axis smoothing, select **Tool axis smoothing** (2).

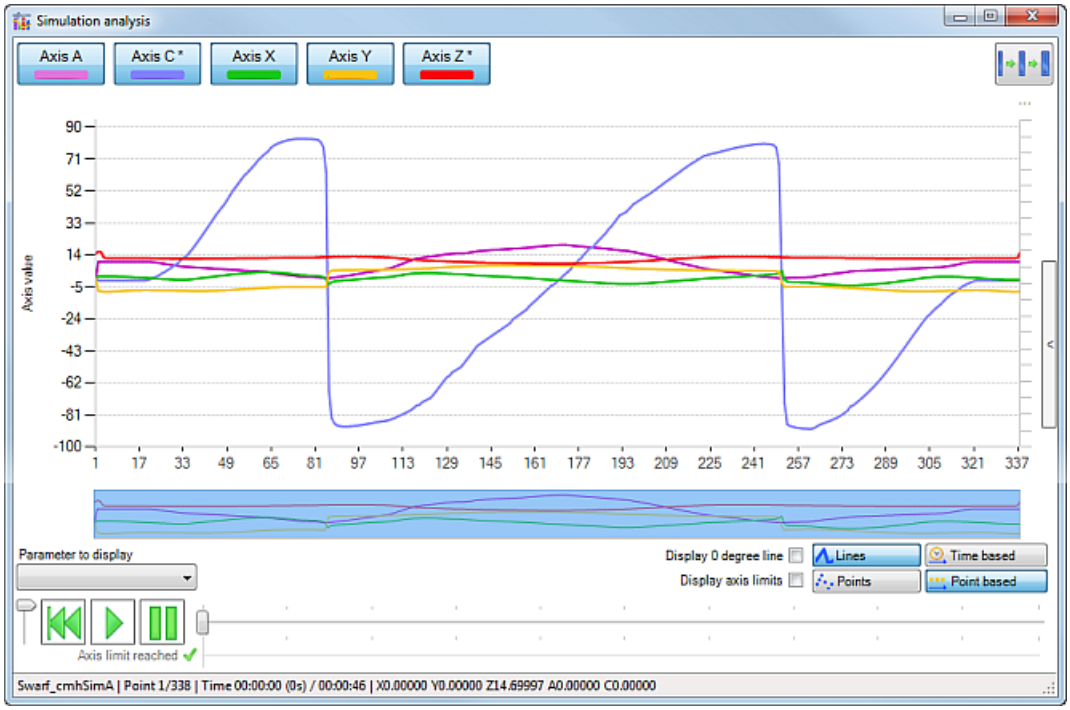


Redistributing points

When redistributing points to prevent the undesirable movement of a machine tool's rotary axes, consider the following:

- If you do not specify a coordinate system, PowerMILL assumes the position of the machine tool's gimbal lock is on the Z axis of the toolpath workplane.
- When redistributing toolpath points, PowerMILL maintains machining tolerance. Therefore, PowerMILL cannot prevent all undesirable movements of the machine tool's rotary axes. As a work around, you can move the tool axis far away from vertical, or use a vice to clamp the part at 45° to the machine table.
- You can identify undesirable movements of rotary axes visually and by using the **Machine Tool Position** dialog or the **Simulation Analysis** plugin for even more detailed feedback.

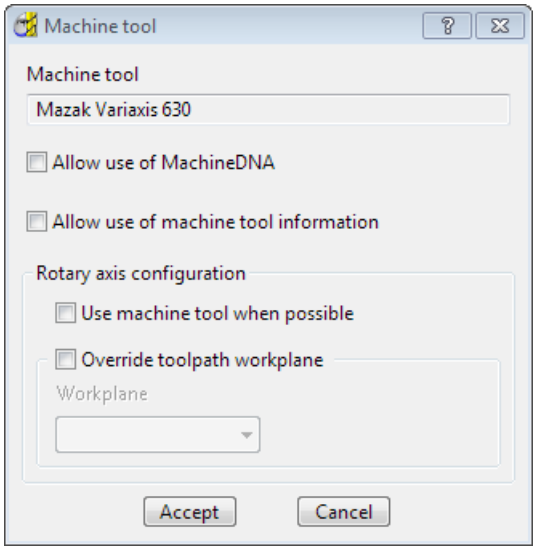
  The **Simulation Analysis** plugin is an effective tool for identifying undesirable movements of the rotary axes because it represents the movements of a machine tool's axes in a line graph. In the graphic below, the C (rotary) axis (drawn in purple) swings from approximately 85° to -85° in one second



*To use the **Simulation Analysis** plugin, you must have the PowerMILL Advanced Simulation licence (PowerMILL–ADVSIM). For information, contact your Delcam sales agent.*

Editing a toolpath's tool axis and point distribution

Use the **Machine tool** dialog to select a suitable coordinate system for editing the tool axis and point distribution of a selected area of a calculated toolpath.



To display the dialog, on the **Toolpath** toolbar, select and then from the **Edit Tool Axis** tab, select .

Machine tool — The read-only field displays the name of the machine tool specified on the strategy dialog's **Machine tool** page.

Allow use of MachineDNA — Select to enable PowerMILL to use the machine tool's MachineDNA, if available.

Allow use of machine tool information — Select to allow PowerMILL to use machine tool information to optimise the toolpath.

Rotary axis configuration — The options enable you to select a suitable coordinate system, different to the one used to initially calculate the toolpath, to use for editing the tool axis and point distribution of the selected area of the toolpath.

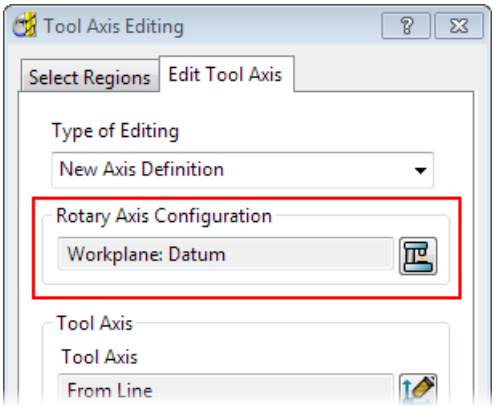
Use machine tool when possible — Select to enable PowerMILL to calculate a suitable workplane based on the machine tool (.mtd file) displayed in the **Machine tool** field. You must also select **Allow use of machine tool information**.

*If the **Machine tool** field does not display a machine tool name, then you cannot use machine tool information. To use machine tool information, recalculate the toolpath with a machine tool specified.*

Override toolpath workplane — Select to choose a workplane from the **Workplane** list.

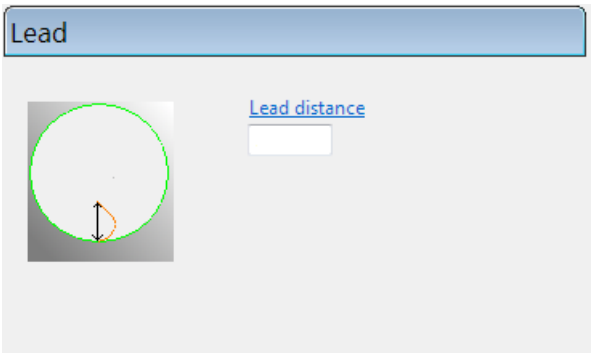
Workplane — Select a workplane from the list.

The new read-only field displays the selected coordinate system.



Helical drilling

There is a new **Lead** page for **Helical** drilling. The page enables you to specify a lead out distance to ensure the drill does not dwell at the bottom of the drilled hole.



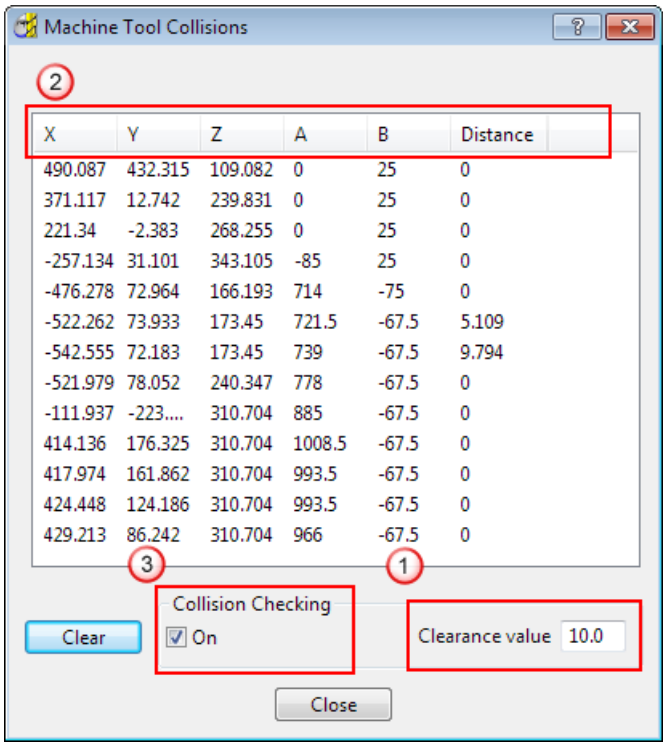
Lead distance — Enter a value to specify the distance of the lead out. The lead out move is measured from the surface inwards.

Verifying toolpaths

Enhanced collision checking

There are several enhancements to the **Machine Tool Collisions** dialog:

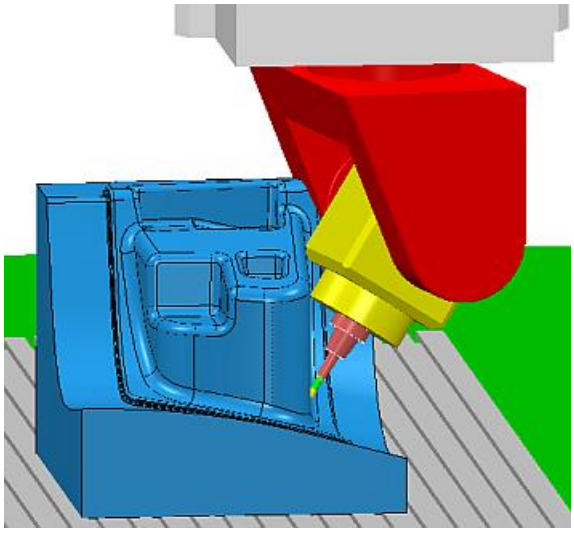
- You can specify a **Clearance value** around the machine tool that PowerMILL takes into consideration when checking for collisions (1).
- The new table design displays the coordinates of a machine tool's axis-components more clearly (2).
- It is easier to turn **Collision Checking** on and off (3).



Checking for machine tool clearance

When the distance between the part and a machine tool's axis component is less than the specified clearance value:

- The axis-component is highlighted yellow (colliding axis-components are highlighted red).



- The **Distance** column displays the distance between the axis-component and the part.

A distance of 0 means the part and the axis component collided.

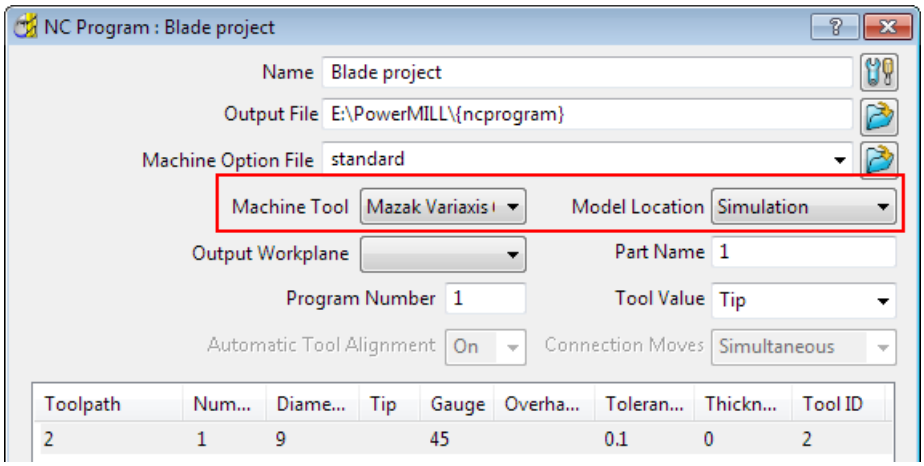
X	Y	Z	A	B	Distance
134.205	-8.644	464.931	-72	0	0
227.178	307.985	491.776	-72	0	0
475.829	3.82	110.877	-90	-60	9.111
392.475	3.82	205.923	-90	-37.5	0
392.475	3.82	205.923	-90	-37.5	0
392.475	3.82	205.923	-90	-37.5	0
392.475	3.82	205.923	-90	-37.5	0
400.933	12.811	198.883	-88	-39.5	0
312.625	-190....	198.883	-139	-39.5	4.089
399.443	12.831	198.883	-88	-39.5	0
439.28	38.541	159.646	-83.306	-49.419	9.816
423.934	38.938	176.181	-82.864	-45.448	1.627

Improvements to NC program simulation

There are two new options on the **NC program** dialog that enable you to specify:

- the machine tool you will use to machine the part; and,
- the position of the model on the machine tool table.

  PowerMILL saves the information to the NC program and uses the information when simulating the NC program.



Machine Tool — Select the machine that you will use to machine the part. PowerMILL uses the specified machine tool when you simulate the NC program.

Model Location — From the list, select the workplane to position the part for simulation. PowerMILL warns you if the model location workplane of the NC program and any of its toolpaths are different.

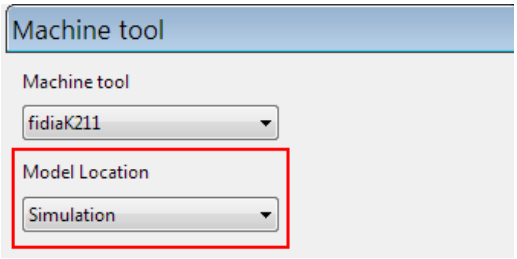
If you do not select a machine tool, PowerMILL uses the active machine to simulate the NC program.

*To simulate the NC program with a different machine tool or model location as an experiment, from the **Machine Tool** toolbar, select a different machine tool or model location, and from the Explorer, right-click the NC program and click **Simulate from Start**. These changes do not change the settings on the **NC program** dialog.*

Model location workplane

Several changes make it easier to work with model location workplanes:

- The workplane field on the **Machine tool** page of strategy dialog is now called **Model Location**. PowerMILL warns you if the model location workplane of the NC program and any of its toolpaths are different.

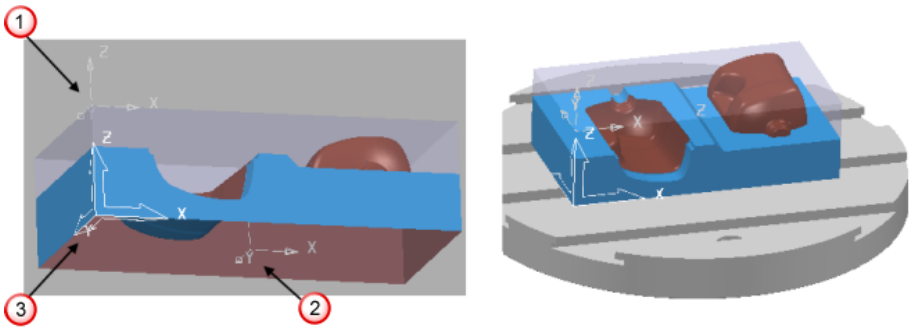


- The Model location workplane on the **Machine Tool** toolbar has a new icon. The drop-down list now automatically displays the simulated toolpath's model location workplane.

See 'Model location and output workplanes' for an explanation of the two workplanes (see page 49).

Model location and output workplanes

The following diagram explains differences between the model location workplane and the output workplane.



1 The output workplane specifies the datum position with respect to the CAD origin.

2 The model location workplane origin specifies the point on the model that lines up with the machine tool's table attachment point.

3 The CAD origin, or world origin, workplane is the workplane used when writing the CAD model in PowerSHAPE or another CAD software.

Outputting toolpaths

Enhanced setup sheets

There are several enhancements to setup sheets:

- You can use PowerMILL parameters, instead of variables, to create and edit setup sheets more easily than in previous versions of PowerMILL.

  PowerMILL still supports the previous system of setup-sheet variables, but you are recommended to use PowerMILL parameters.

- There is a new **Setup sheet summary** option on the **Help > Parameter** menu.

  Click to display the new list of setup-sheet-specific parameters. This list replaces the list of setup-sheet-specific variables that are documented in previous versions of PowerMILL **(Help > Parameter > Summary > Table 2**).

- There are a number of new functions and new parameters that display more information about NC programs and projects, including the number of toolpaths in an NC program and the cutting time of a project.

Using parameters to create and edit setup sheets

You can use PowerMILL parameters to create and edit setup sheets more easily than in previous versions of PowerMILL. The previous system of setup-sheet variables is inconsistent with PowerMILL parameters and complicates the process of creating and customising setup sheets.

Using parameters enables you to:

- Create and edit setup sheets more easily.
- Work more consistently across PowerMILL: you can use parameters for setup sheets, macros and other areas of PowerMILL, such as NC program text blocks.
- Support the following syntaxes:
  - `$par` — Use with parameters, such as `$tool`, or `${tool.Diameter}`.
  - `${expression}` — Use to populate a setup sheet with a complex value generated by an expression (an equation) that use multiple parameters, for example: `${ncprogram.statistics.cuttingmoves.times.arcs + ncprogram.statistics.cuttingmoves.times.linear}`

Dollar $ signs are not required inside curly brackets.

Finding parameters to use in setup sheets

Use the options on the **Help > Parameter** menu to find the parameters you need to create and populate setup sheets:

Reference — The manual contains detailed information on all the parameters you can use in setup sheets, macros and other areas of PowerMILL.

Summary — The list contains the same parameters as the Reference manual but with less information.

Setup sheet summary — The list only contains information about setup-sheet-specific parameters.

PowerMILL no longer lists setup sheet variables, but PowerMILL still supports setup sheets that use variables. However, you are recommended to use PowerMILL parameters for improved usability.

New parameters and functions

The new parameters and functions enable you to populate your setup sheets with:

- more information about toolpaths and tools (see page 52).
- cutting and total-machining times (see page 53).
- a pre-defined, meaningful value when a parameter is not resolved (see page 56).

For more an example of how to add these new parameters to setup sheets, see Displaying additional information on the Summary page (see page 54).

You can also use these parameters and functions in macros. For more information, see the 'Macro Programming Guide'.

Adding toolpath and tool information

To populate setup sheets with information about toolpaths and tools in an NC program, use the following functions in the relevant .html template:

- Number of toolpaths in an NC program:
  `${number_nctoolpaths(entity('ncprogram', ncprogram.name))}`
- Number of tools in an NC program:
  `${number_nctools(entity('ncprogram', ncprogram.name))}`
- Names of tools in an NC program:
  `${join(list_nctools(entity('ncprogram', ncprogram.name)), ',')}`

The function `join` separates the list of tool names with a delimiter, in this case a comma.

Adding cutting and total-machining time information

To populate setup sheets with the cutting time and total-machining time of toolpaths, NC programs and projects (in HH:MM:SS), use the new parameters `nCuttingTime` and `nTotalTime` in conjunction with the new function `time_to_string` in the relevant .html template.

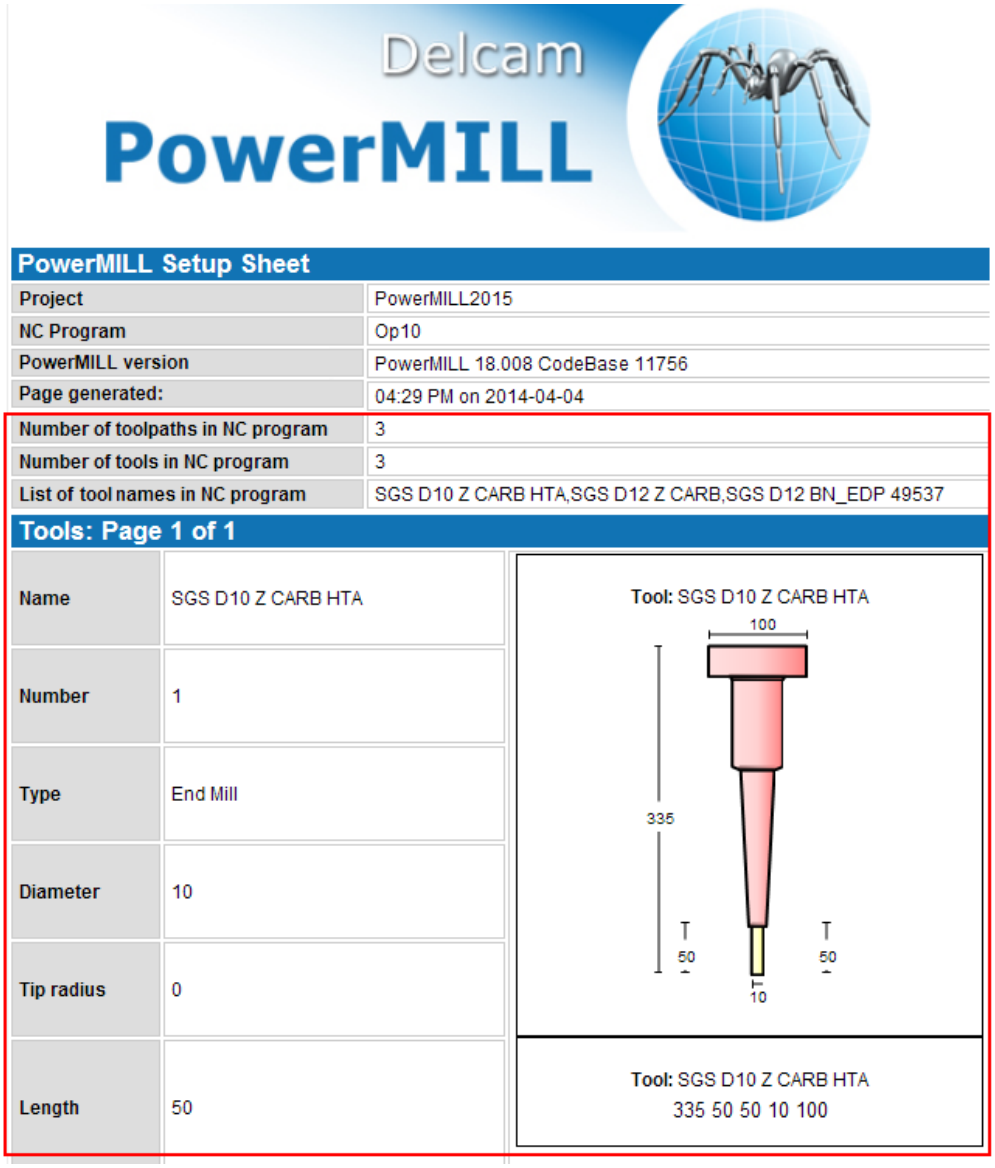
- To display the cutting and total-machining times of a toolpath, use:
  ```
  ${time_to_string(stoolpath.nCuttingTime,"M")}
  ${time_to_string(stoolpath.nTotalTime,"M")}
  ```
- To display the cutting and total-machining times of an NC program, use:
  ```
  ${time_to_string(ncprogram.Statistics.nCuttingTime,
  "M")}
  ${time_to_string(ncprogram.Statistics.nTotalTime, "M")}
  ```
- To display the cutting and total-machining times of a Project, use:
  ```
  ${time_to_string(project.nCuttingTime,"M")}
  ${time_to_string(project.nTotalTime,"M")}
  ```

`"M"` instructs the `time_to_string` function to display the retrieved value in minutes. Use `"S"` or `"H"` to display the values in seconds or hours respectively.

Displaying additional information on the Summary page

This example shows how to edit the Summary.html template (in the **Setup-sheets** folder) so the Summary page displays the following additional information:

- Number of toolpaths and tools in an NC Program.
- List of tool names.
- Geometry and a diagram of each tool.



1. Open the Summary.html file in an XML editor. This example uses NotePad++.
2. For the Summary page to display the number of toolpaths and tools, and a list of tool names, enter the following elements underneath the `Page Generated` element:

```
    <th>Page Generated:</th><td>{project.time} on {project.date}</td>
  </tr>
  <tr>
     <th>Number of toolpaths in NC program</th>
     <td>{number_nctoolpaths(entity('ncprogram', ncprogram.name))}</td>
    </tr>
<tr>
     <th>Number of tools in NC program</th>
     <td>{number_nctools(entity('ncprogram', ncprogram.name))}</td>
    </tr>
<tr>
     <th>List of tool names in NC program</th>
     <td>{join(list_nctools(entity('ncprogram', ncprogram.name)), ',')}</td>
     </tr>
  </tbody>

</table>
```

3 For the Summary page to display the geometry and a diagram of each tool:

a Remove the following section:

```
<!-- Toolpath summary table follows. -->
<table>

 <thead>
   <tr>
     <td colspan="4" class="title">Summary sheet ${SU_SheetNo} of ${SU_MaxSheets}</td>
   </tr>
   <tr>
     <th>Toolpath / TAP File</th>
     <th>Strategy</th>
     <th colspan="2">Tool</th>
   </tr>
 </thead>

 <tbody>
   <summaryrow>
     <tr>
       <td rowspan="5">${stoolpath.NameTag} <br /> ${stoolpath.TapFile}</td>
       <td rowspan="5">${toolpath.Strategy}</td>
       <th rowspan="1">Type</th><td rowspan="1">${tool.Type}</td>
     </tr>
     <tr>
       <th>Diameter</th><td>${tool.Diameter}</td>
     </tr>
     <tr>
       <th>Tip radius</th><td>${tool.TipRadius}</td>
     </tr>
     <tr>
       <th>Length</th><td>${tool.Length}</td>
     </tr>
     <tr>
       <th>Number</th><td>${tool.Number}</td>
     </tr>
   </summaryrow>
 </tbody>

</table>
```

b Enter the replacement section:

```
<!-- Toolpath summary table follows. -->
<table>
  <thead>
      <tr>
          <td colspan="3" class="title">Tool List</td>
      </tr>
  </thead>

  <tbody>
      <toolrow>
          <tr>
              <th rowspan="1">Name</th><td rowspan="1">${tool.identifier}</td>
              <td rowspan="6">
                  <div class="center">
                      ${setupsheets.tool.vml}
                  </div>
              </td>
          </tr>
          <tr>
              <th>Number</th><td>${tool.number}</td>
          </tr>
          <tr>
              <th>Type</th><td>${tool.type}</td>
          </tr>
          <tr>
              <th>Diameter </th><td> ${tool.diameter}</td>
          </tr>
          <tr>
              <th>Tip Radius</th><td>${tool.TipRadius}</td>
          </tr>
          <tr>
              <th>Length</th><td>${tool.length}</td>
          </tr>
      </toolrow>
  <tbody>

</table>
```

4 Save the file.

When you export the setup sheet, the Summary page now displays the number of toolpaths, tools, a list of tool names and a diagram of each tool.

Specifying meaningful values for unresolved parameters

You can now use *conditional expressions* so if PowerMILL cannot resolve a parameter, the setup-sheet field is not blank but displays a pre-defined, meaningful value. This is useful, for example, if there is an error or you forget to specify a toolpath workplane. Without a conditional expression, PowerMILL would be unable to resolve the parameter and the Workplane Name field would be blank and the cause unknown.

The following example shows how to edit the parameter `toolpath.Workplane` to include a conditional expression that displays the value '**WORLD**' if PowerMILL cannot resolve the parameter.

The basic parameter:

```
<th> Workplane Name</th><td>${toolpath.Workplane}</td>
```

Edit the parameter, as shown below:

```
<td> Workplane
Name</th><td>${select(error(toolpath.Workplane), "WORLD",
"$toolpath.Workplane.Name")}</td>
```

Toolpath			
Name	1		
Description			
Strategy	Raster		
Workplane name	WORLD		
Tolerance	0.11	Cutting Feed Rate	1000
Global thickness	0	Plunge Feed Rate	500
Radial thickness	0	Rapid Feed Rate	3000
Axial thickness		Spindle	1500
Stepover	1	Cutting Time	0:23:27
Stepdown		Total Time	0:39:55
Block	210.015 × 100.011 × 55		

How do conditional expressions work?

Conditional expressions include a question and two answers:

In this toolpath workplane example, the question `${select(error(toolpath.Workplane)` asks, 'Is there a toolpath workplane present?'

If the answer is 'No. There is no workplane.' PowerMILL prints the pre-defined value '**WORLD**'. If the answer is 'Yes. There is a workplane.' PowerMILL prints the result of the parameter string `toolpath.wokplane.name`.

It is not necessary to write '`.Name` 'in the 'question '`...(toolpath.Workplane.Name)`' because there cannot be a name if there is no workplane.

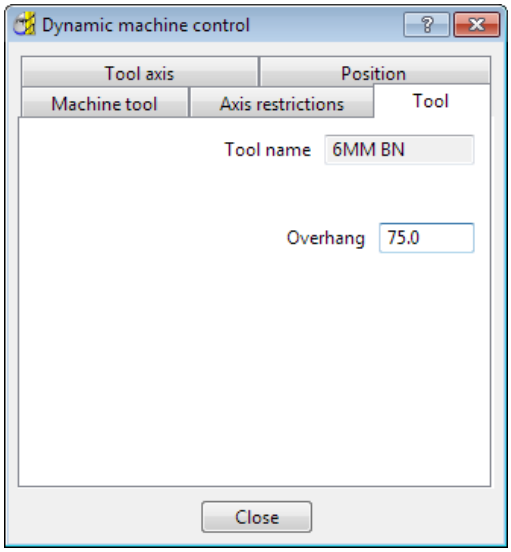
When writing your own conditional expressions, use the structure:

```
${select(error(parameter in question), "value displayed
if parameter cannot be resolved", "$parameter to be
displayed if PowerMILL can resolve parameter")}
```

General enhancements

Dynamic machine control updates

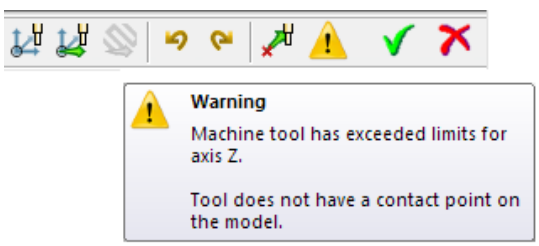
There is a new **Tool** tab on the **Dynamic machine control** mode dialog, which enables you to edit the active tool.



Tool name — The name of the tool.

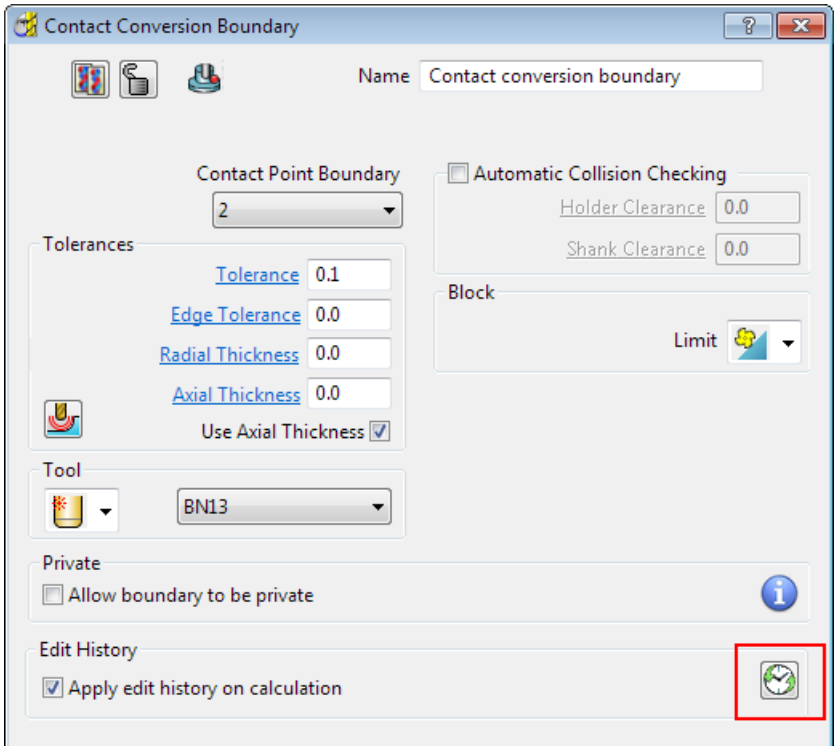
Overhang — Enter a value to edit the overhang of the tool assembly. Entering a value overrides the overhang value specified on the **Holder** tab of the **Tool** dialog.

When you use the **Dynamic machine control** mode and you jog a machine tool's axis beyond its limit, PowerMILL informs you which axis is exceeding its limit.



Button and tooltip updates

- There is new **View boundary edits** button on the **Boundary** toolbar and on a boundary settings dialog that enables you to display the **Boundary Editing History** dialog.



- The **View toolpath edits** button on the **Toolpath** toolbar has a new icon .

Automating PowerMILL

Macros

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

- Retrieve model-hierarchy elements in a list and specific elements by paths.
- Return the number of segments in a toolpath.

For more information, see the ‘Macro Programming Guide’.

Index

A

Approaching from outside stock, Vortex • 34
  Helical drilling • 45
Automatic angle mode
  Raster finishing • 32

C

Curve
  Cutting a curve • 26
  Ellipse • 8
  Helix • 20
  Open and closed curves • 22, 25
  Spiral • 13
Custom strategies • 30

D

Drilling • 45
Dynamic machine tool • 58

E

Ellipse • 8

F

Favourite a strategy • 29
Fixed direction
  Raster finishing • 32

G

General updates • 59
Gimbal lock • 39, 40

H

Helix • 20

I

Increasing efficiency
  Reducing lift moves • 38
  Vortex • 34, 36

M

Machine tool clearance • 46, 47
Machining
  Displaying machining time on a setup sheet • 52
  Raster finishing • 32
Macros • 60
Model location workplane • 49

P

Parameters
  Cutting and machining time • 53
  Displaying additional information on the Summary page • 54
  New parameters and functions • 52
  Toolpath and tool information • 52
  Unresolved parameters • 56

Updates to setup sheets • 50
Using parameters to create and edit setup sheets • 51
Product licence • 4

R

Raster
Raster finishing • 32
Raster finishing • 32
Reducing the machining time of Vortex toolpaths • 36
Rotary axis configuration • 40, 43

S

Saving a toolpath as a strategy for future reuse • 30
Setup sheets
Cutting and machining time • 53
Displaying additional information on the Summary page • 54
New parameters and functions • 52
Toolpath and tool information • 52
Unresolved parameters • 56
Updates to setup sheets • 50
Using parameters to create and edit setup sheets • 51
Simulating an NC program • 48, 49
Spiral • 13
Strategies
Approaching from outside stock, Vortex • 34
Collision data, improved presentation • 46
Drilling • 45
Favourite a strategy • 29
Raster finishing • 32
Reducing the machining time of Vortex toolpaths • 36
Roughing toolpaths • 38
Saving a toolpath as a strategy for future reuse • 30
Strategy Selector dialog • 29
Strategy Selector dialog • 29

T

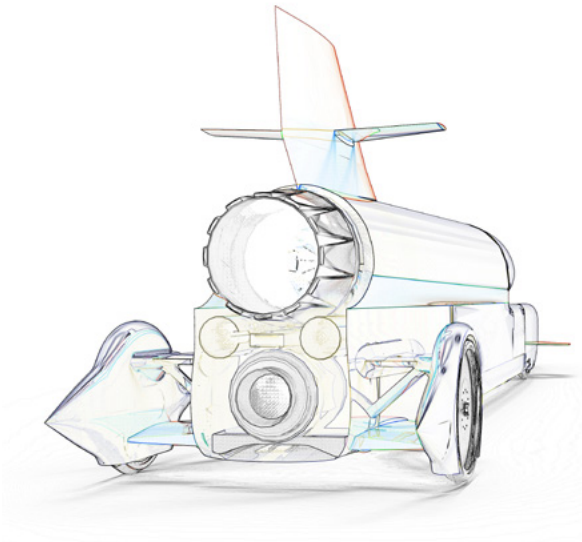
Toolpath
Approaching from outside stock, Vortex • 34
Raster finishing • 32
Reducing the machining time of Vortex toolpaths • 36

V

Vortex
Approaching from outside stock, Vortex • 34
Reducing the machining time of Vortex toolpaths • 36

W

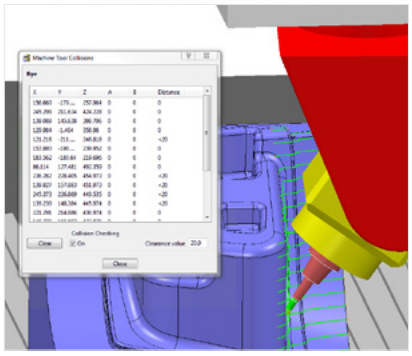
WN2015
Simulating
Simulating an NC program • 48, 49
Workplane
Gimbal lock • 39, 40
Rotary axis configuration • 40, 43
Workplane • 49



PowerMILL 2015

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

www.powermill.com

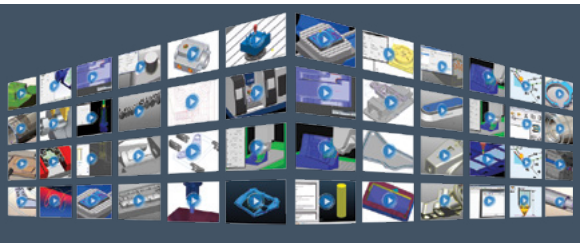


PowerMILL 2015

www.powermill.com

www.delcam.tv

Unlock the power of Delcam software



LEARNING ZONE

Visit **www.delcam.tv/lz** and see how Delcam software could benefit your business!

Delcam Advanced Manufacturing Solutions

Powering your productivity

Delcam
Small Heath Business Park, Birmingham, B10 0HJ, United Kingdom
T: +44 (0) 121 766 5544 | E: marketing@delcam.com | W: www.delcam.com

To contact your local reseller, visit **www.delcam.com/resellers**

AUTODESK.