Delcam Advanced Manufacturing Solutions

PowerMILL 2014

World leading 2, 3 and 5-axis CAM software

www.powermill.com

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

What's New

Issue 3

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 2014. Published on 16 October 2013

Contents

Summary of new features 1

Vortex and MachineDNA 1
Toolpath preparation 1
Toolpath generation 2
Toolpath verification 3
Toolpath output 3
User interface 3
Automation 3
General enhancements 4

High speed machining 5

Vortex machining 5
- Vortex 8
- High speed (Vortex) 9
- Approach (Vortex) 11
- Vortex and step cutting example 12
- Vortex machining of 2D features 16
- Analysing Vortex toolpaths 18

Creating MachineDNA 19
- Displaying MachineDNA 21

Toolpath preparation 22

Curve enhancements 22
- Insert and Edit Fillets dialog 22
- Editing a composite curve 30
- Tangent editor 34
- Repoint curve 40
- Reorder enhancements 49
- Limit curve enhancements 50

New and improved tools 51
- Barrel tool 52
- Dovetail tool 53
- Enhanced tipped disc tool 54
- Tool Assembly Preview dialog 55

Tool database enhancements 56
Stock model enhancements 58

Toolpath generation 62

Drilling enhancements 62
Drilling strategies 63
External threading 63
External Thread 65
Retraction (external thread) 66
Draft (external thread) 66
Threading (external thread) 67
Order (drilling) 70
Creating an external thread 70
Varying feed rates at drill hole intersections 73
Feed rate reduction 76
Feature > Intersection 81
Flat machining 84
Raster ordering 85
Optimised raster improvements 87
Point distribution on leads and links 88

Toolpath verification 91

Enhanced collision checking 91
Introduction to collision groups 91
Identifying components to include in collision checks 96
Creating and adding to an exclusion list 98

Toolpath output 100

NC program fixture offsets 100
Applying fixture offsets 104

User interface 108

Measuring lines 108

Automation 111

Macro programming enhancements 111
Plugin enhancements 112

General enhancements 114

Model import enhancements 116
Nc program updates 117
New Active branch in the explorer 120
Selected feature colour in the explorer 121

Index 123

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 2014 offers all of the original features of PowerMILL 2013, but with numerous improvements. This document describes the most significant improvements.

Vortex and MachineDNA

There are two new features to enhance high speed machining in PowerMILL.

- Vortex machining is an area clearance strategy which rapidly removes material from a 3D part while controlling tool load. This strategy is best suited to solid carbide tools (see page 5).
- MachineDNA determines specific characteristics of the machine tool which enables you to use the optimal settings when Vortex machining (see page 19).

Toolpath preparation

There are several new options on the **Curve Editor** mode toolbar (see page 22):

-  **Insert and edit fillets** enables you to insert and edit the fillets in a composite curve containing arcs and lines. This maintains arcs as arcs and doesn't convert them to a spline (see page 22).
-  **Tangent editor** lets you edit the tangents of a point on a Bézier curve. This enables more precise tangent editing, as previously you could only edit tangents graphically (see page 34).

-  **Repoint curve** enables you to redistribute points, or change the number of points, along a Bézier curve (see page 40).
- There are enhancements to the **Reordering curve segments** dialog (see page 49).
- You can now trim or extend both ends of a single curve segment (see page 50).

There are several enhancements to tools:

- PowerMILL now supports barrel tools. A barrel tool efficiently machines surfaces that are typically found on blades (see page 52).
- PowerMILL now supports dovetail tools. A dovetail tool enables you to efficiently machine features, such as undercuts and gears (see page 53).
- You can now have different values for the lower and upper tip radii of a tipped disc tool (see page 54).
- There is a new **Tool Assembly Preview** dialog which displays a larger image of a tool assembly. The larger image enables you to identify colliding components within the tool assembly more easily than before.

The function and usability of the tool database has been enhanced (see page 56); you can now:

- Edit and delete entities that are either associated with cutting data or tool assemblies.
- Display the **Tool Database Holder Search dialog** more easily.

There are a couple of stock model enhancements (see page 58):

- The stock model now accurately reflects machining with non-symmetrical tools, such as tipped disc tools with different values for the **Lower Tip Radius** and the **Upper Tip Radius**.
- You can now change the block in a stock model, provided there are no calculated tool or toolpath states.

Toolpath generation

There are several drilling enhancements:

- Each **Drilling Cycle type** now has its own strategy (see page 63).
- There is a new drilling strategy of **External threading** (see page 63).
- You can now create a thread milling toolpath in a tapered hole (see page 63).

- You can now vary the feed rate and spindle speed when two holes intersect (see page 73).

Flat machining on triangle models now produces better results and is faster (see page 84).

You can now create a raster toolpath without any grouping of toolpath segments (see page 85).

There are improvements to the optimised raster algorithm (see page 87).

You now have the option to specify point distribution on the non-cutting moves as well as the cutting moves of a toolpath (see page 88).

Toolpath verification

PowerMILL now supports more sophisticated .mtd files. This means you can add to and modify your machine tool's .mtd file to improve collision checking (see page 91).

Toolpath output

PowerMILL now supports fixture offsets. A fixture offset is a datum coordinate that you specify and then apply to a copy of a toolpath (see page 100).

User interface

PowerMILL 2014 reintroduces the **Line** tab on the **Measurer** dialog (see page 108).

Automation

- The macro programming language has been enhanced.
- A new horizontal plugin window has been added to PowerMILL. Now PowerMILL features both a horizontal and a vertical plugin window (see page 112).

  The horizontal layout of the new plugin window provides developers with a greater amount of flexibility when designing plugins.

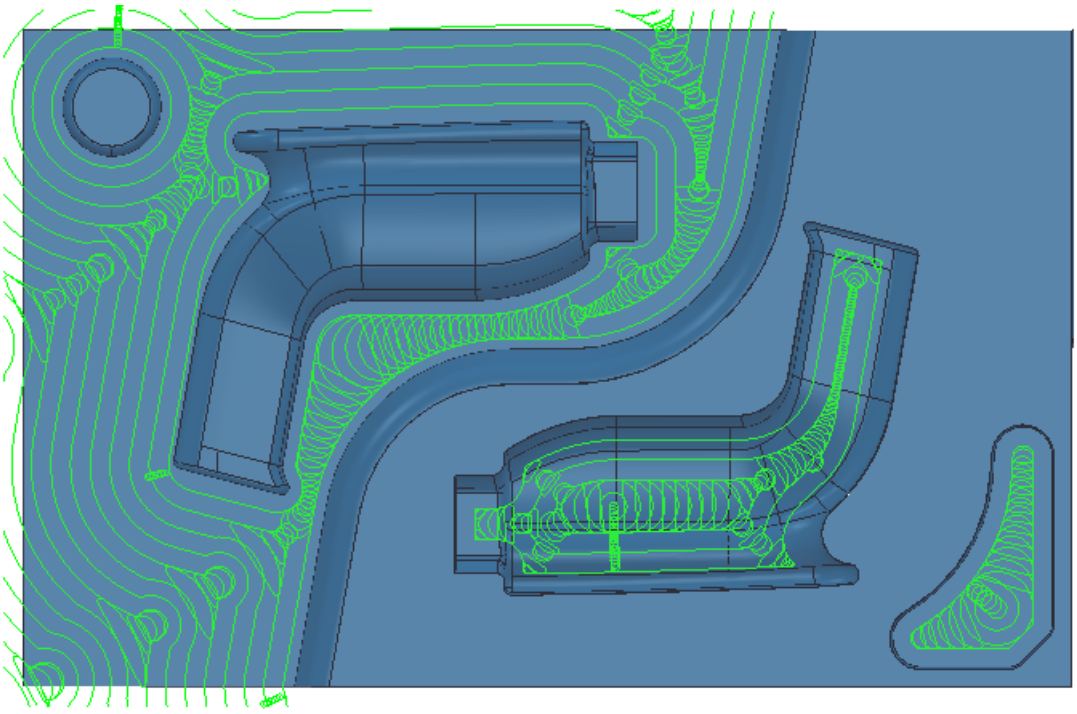
General enhancements

- If a model contains multiple solids, you can now place all the surfaces associated with a solid in a set which has the name of the original solid (see page 116).
- You can choose whether existing toolpaths and boundaries should ignore newly imported models or not (see page 116). Normally you are unlikely to want to ignore newly imported models, except when the models are clamping models.
- The functions of the **NC Programs** branch in the explorer has been updated (see page 117).
- The new **Active** branch in the explorer displays all the entities that are active at any one time (see page 117).
- Selected features are now displayed yellow in the explorer as well as in the graphics window (see page 121).
- You can now remove entities from a group (see page 114).
- The **Status** bar can now display the **Drill Thickness** and **Curve Thickness** of the active toolpath (see page 114).
- When you include a toolpath in an NC program, the toolpath is now drawn in blue. This is to help you differentiate between toolpaths that are included in an NC program and those that are not (see page 114).
- PowerMILL now automatically populates the **Output Project** field in the **Extract Electrode** dialog with the name of the selected electrode and the same folder as the selected .trode file (see page 114).

High speed machining

Vortex machining

Vortex machining is an area clearance strategy which rapidly removes material from a 3D part while controlling tool load. Vortex is best suited to solid carbide tools and is frequently used in combination with step cutting.



Vortex is an offset-style toolpath and has these main features:

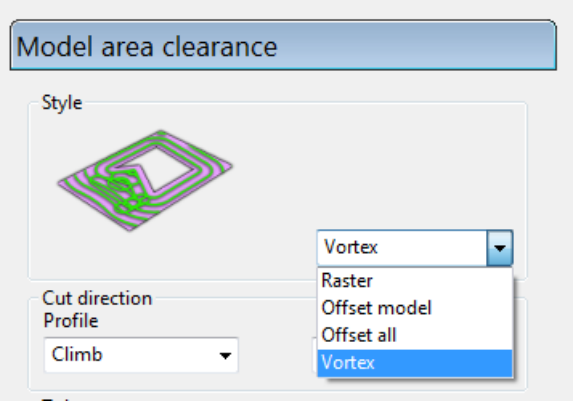
- The engagement angle never exceeds, by more than 15%, that produced by a straight line cut with a given stepover. This eliminates excessive tool load and all full-width cuts. This enables you to increase feed rates. For other area clearance strategies, the cutting values are based on the slot cutting parameters to ensure the tool can sustain full cutting engagement. As the tool approaches the maximum engagement angle, the toolpath changes to a trochoidal path to avoid tool overload.
- When used in conjunction with MachineDNA, the machine tool almost always runs at the specified feed rate. With standard area clearance toolpaths, the machine tool automatically slows down as it approaches a corner and the engagement angle increases. Vortex modifies the toolpath so the tool engagement is never exceeded and the machine tool achieves the specified feed rate. The only time the machine tool doesn't run at the specified feed rate is when the model geometry (a slot or corner) is smaller than the smallest radius that the machine can run at full speed.
- Vortex machining cuts with the side of the tool so it is designed for solid carbide tools, but you may be able to use other tools.
- As PowerMILL controls the tool engagement, you can increase the depth of cut which minimises machining time.
- Vortex machining is frequently used in combination with **Step cutting** to minimise terracing while maximising the removal rate.
- Vortex toolpaths are automatically checked for safety. If the toolpath is unsafe PowerMILL displays a warning message, otherwise it is shown as safe () (see page 18).

To maximise the benefits of Vortex machining:

- Configure the Vortex parameters to suit each machine tool. For more information, see MachineDNA (see page 19).
- Use **Step Cutting** to minimise terracing caused by the increased depth of cut.
- On the **Point Distribution** page, select an **Output type** of **Tolerance and keep arcs**. MachineDNA manages the point spacing so your machine tool can sustain the programmed feed rate.

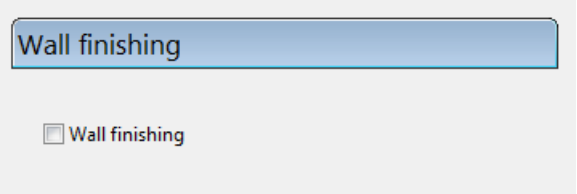
With optimum settings, Vortex machining greatly reduces machining times.

Vortex is a **Style** option on most area clearance strategies (including curve area clearance, feature set area clearance, feature set rest area clearance, model area clearance, model rest area clearance, and slice area clearance).



Selecting **Vortex**:

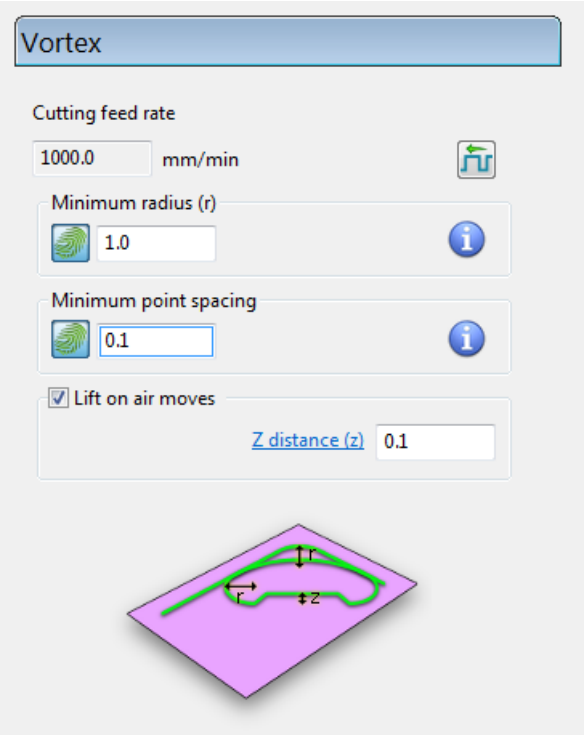
- Enables the **Vortex** page, which contains the settings required to define a Vortex toolpath (see page 8).
- Reduces the options on the **Wall finishing** page. You can select only whether to use wall finishing. If selected, this produces a final profile pass with a zero offset from the previous pass.



- Gives different options on the **High speed** page.

Vortex

Vortex creates toolpaths which machine at the specified cutting feed rate almost all of the time.



Cutting feed rate — This displays the **Cutting feed rate** specified on the **Feeds and Speeds** dialog.

The **Cutting feed rate** is a critical value to ensure the tool operates under optimum conditions. The feed rate depends on the tool and material combination. You should use the feed rate recommended, by the tool manufacturer, for side cutting (or profile cutting) as a minimum cutting feed rate. Testing has shown these recommended values to be very conservative when Vortex machining. This means, you can normally increase either the recommended feed rate or the recommended stepdown quite significantly. You can further improve machining times by using **MachineDNA** (see page 19), which cleverly determines **Minimum radius** and **Point spacing** values from the **Cutting Feed Rate** value.

For most toolpaths, entering a sharp internal corner increases the tool engagement angle and causes tool overload. To avoid tool overload you have to reduce the feed rate. For Vortex toolpaths, there is no need to reduce the feed rate, as the tool changes to a trochoidal path when entering sharp internal corners, avoiding the overload situation.

Feeds and Speeds — Click to display the **Feeds and Speeds** dialog which enables you to change the **Cutting feed rate**.

Minimum radius (r) — This is the minimum radius used in the internal trochoids. By default this is the value determined by MachineDNA. Otherwise PowerMILL uses a value suitable for a typical machine tool at the specified feed rate.

As the feed rate increases, the minimum radius increases.

It is recommended that you use the value determined by MachineDNA or PowerMILL. Only enter your own minimum radius if you have a very good reason to do so.

Minimum point spacing — This is the minimum point spacing at which the machine tool can move at the specified feed rate. By default this is the value determined by MachineDNA. Otherwise PowerMILL uses a value suitable for a typical machine tool at the specified feed rate.

If the machine tool has too many points to process, it cannot sustain the specified feed rate.

It is recommended that you use the value determined by MachineDNA or PowerMILL. Only enter your own minimum radius if you have a very good reason to do so.

Select  to calculate **Minimum radius** and **Minimum point spacing** values using MachineDNA. PowerMILL displays  to indicate it is using MachineDNA to calculate values.

If PowerMILL displays either  *you need to enable or create MachineDNA (see page 19).*

Lift on air moves — When selected, raises the height of the back of the trochoids (the portion of the toolpath which is air-cutting). When deselected the whole trochoid is at the cutting level.

Z distance (z) — Enter the lift height (the height above the cutting level).

For more information see Vortex machining (see page 5), Vortex and step cutting example (see page 12), and Vortex machining of 2D features (see page 16).

High speed (Vortex)

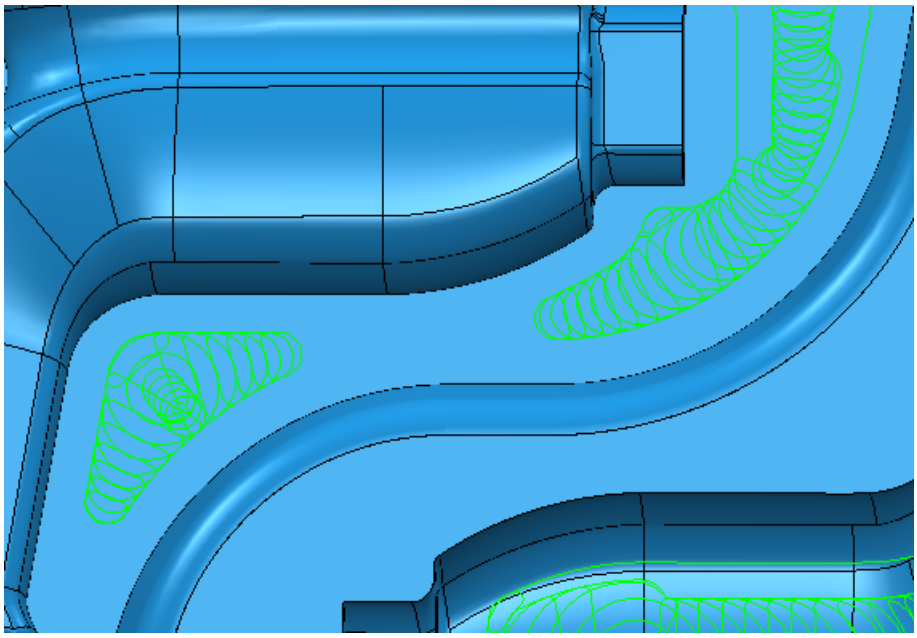
High speed controls the smoothing options to avoid sharp changes in tool direction when high speed machining. For more information on the effects of smoothing, see Tool loading.

Profile smoothing — All Vortex toolpaths use arc fitting of profiles to avoid sharp changes of direction in internal corners.

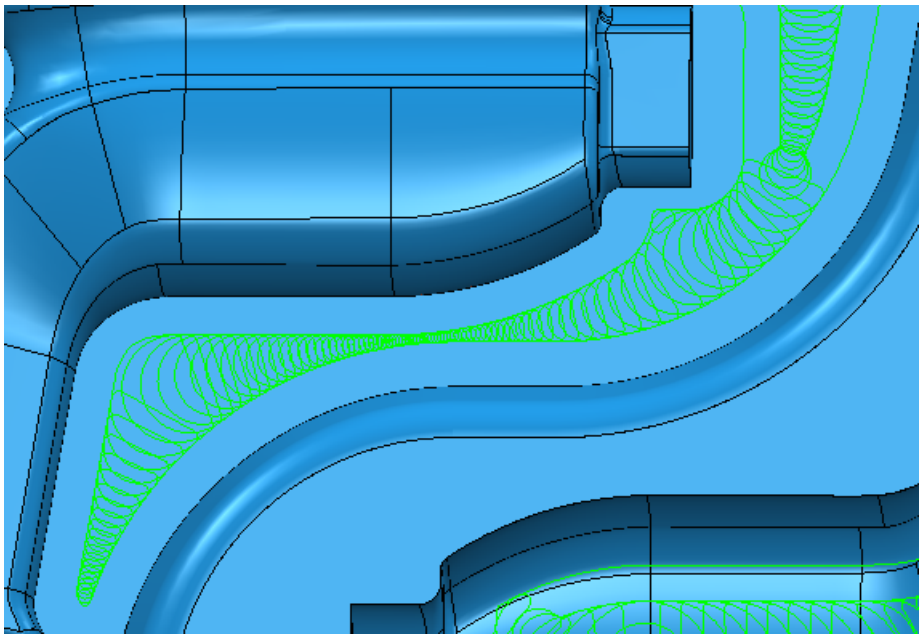
The **Radius (tool diameter units)** slider determines when to insert arcs. The radius is defined as a proportion of the tool diameter. The default value is 0.05. So, if you have a tool of diameter 20 mm (radius 10 mm) then the arc radius is 1 mm.

*The **Radius** slider has a value in the range 0.005 - 0.2*

Using a **Radius** of **0.2** produces:



Using a **Radius** of **0.05** produces:

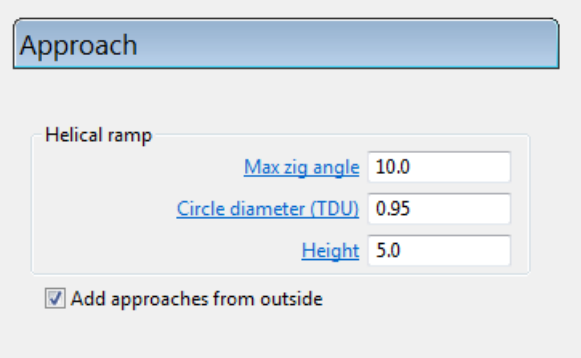


You can see that a small **Radius** value produces more trochoids.

Small trochoids are inadvisable, if possible keep the trochoids larger than 1 mm.

Approach (Vortex)

Approach specifies the helical ramp moves that are added to Vortex toolpaths to ensure the tool approaches the stock safely and does not plunge into the stock. Vortex adds these moves when the (linear/arc) approach outside moves (specified on the **Leads and Links** pages) would result in the tool plunging into the stock.



The helical ramp moves are implemented irrespective of lead settings.

Helical Ramp — Specifies the size of the helix.

Max zig angle — Enter the angle of descent formed as the tool ramps into the block.

Circle Diameter (TDU) — Enter the diameter of the circle using tool diameter units (TDUs).

***Tool diameter units** is the distance relative to the tool diameter. So, a 10 mm tool and a TDU of 2 gives an actual value of 20 mm.*

Height — Enter the height above the toolpath segment for the start of the ramp.

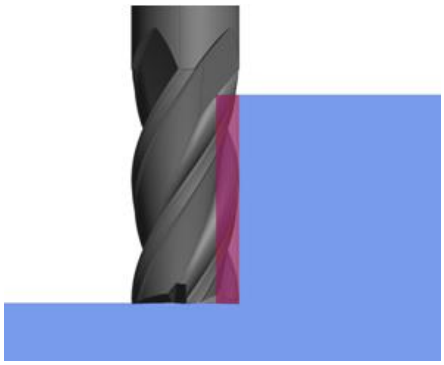
*This is measured with respect to the tool axis. For 3-axis machining, the tool axis is the Z axis. For multi-axis machining, the tool axis is specified on the **Tool Axis** dialog.*

Add approaches from outside — Select to enable level moves to approach the model from outside the block.

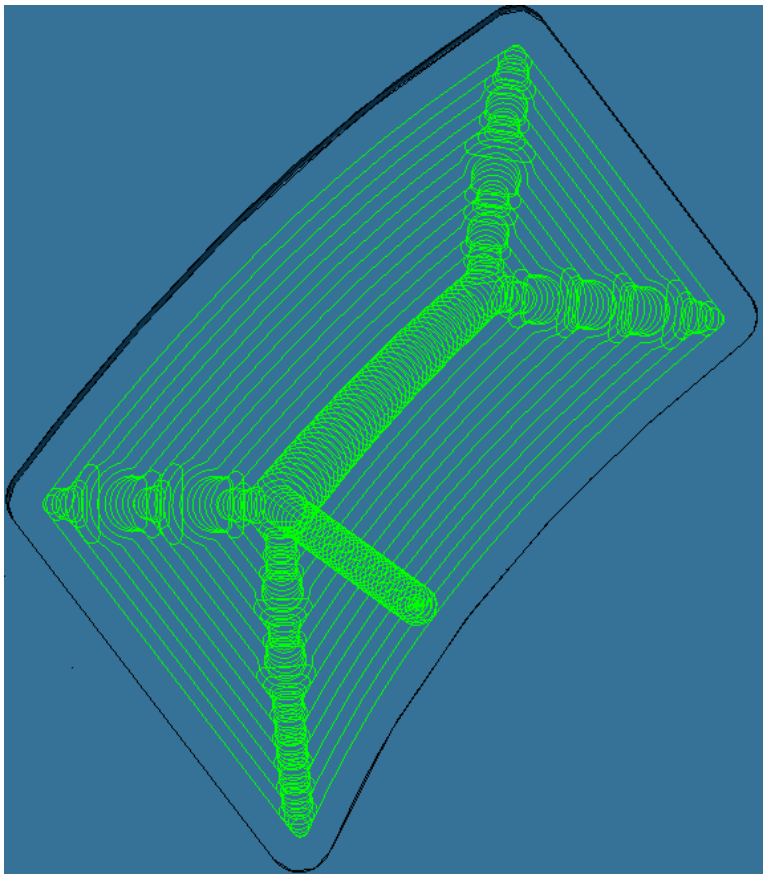
Vortex and step cutting example

This example shows you how to combine Vortex machining with step cutting to rapidly remove material.

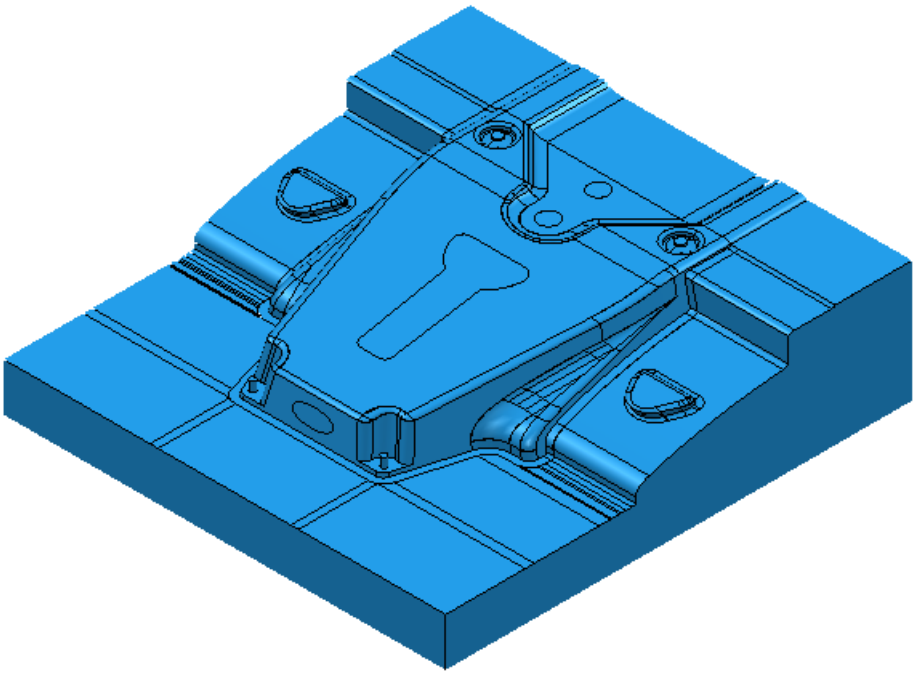
As Vortex machining cuts with the side of the tool, it is designed for to solid carbide tools, but there may be other types of tools suitable for Vortex. These tools work best when taking deep cuts with a relatively small stepover.



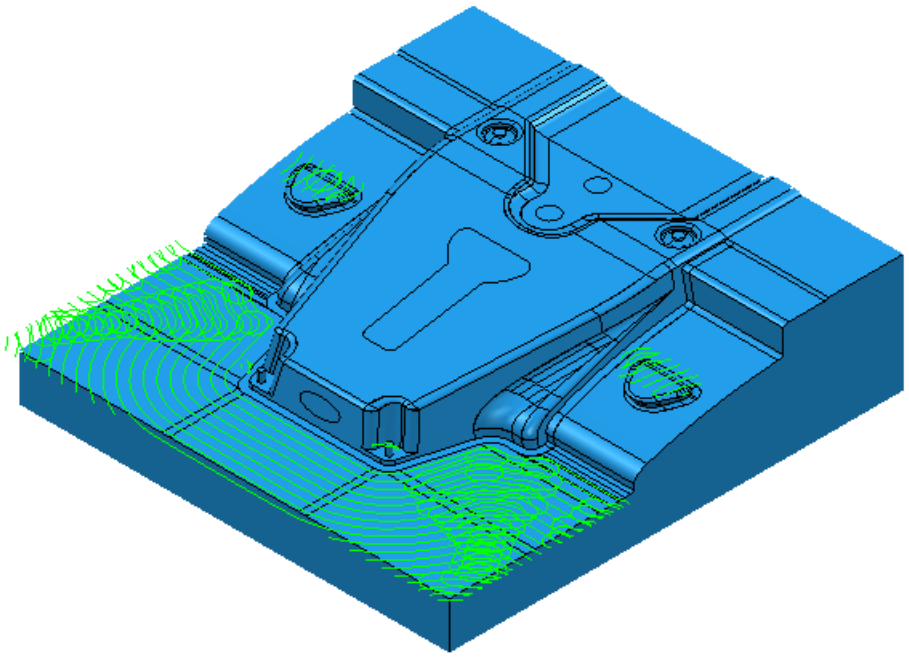
To machine effectively when taking large depths of cut, you must ensure the tool engagement angle never exceeds the specified value. This eliminates excessive tool load and all full-width cuts. PowerMILL achieves this by introducing trochoidal moves to prevent the tool from exceeding the maximum tool engagement value.



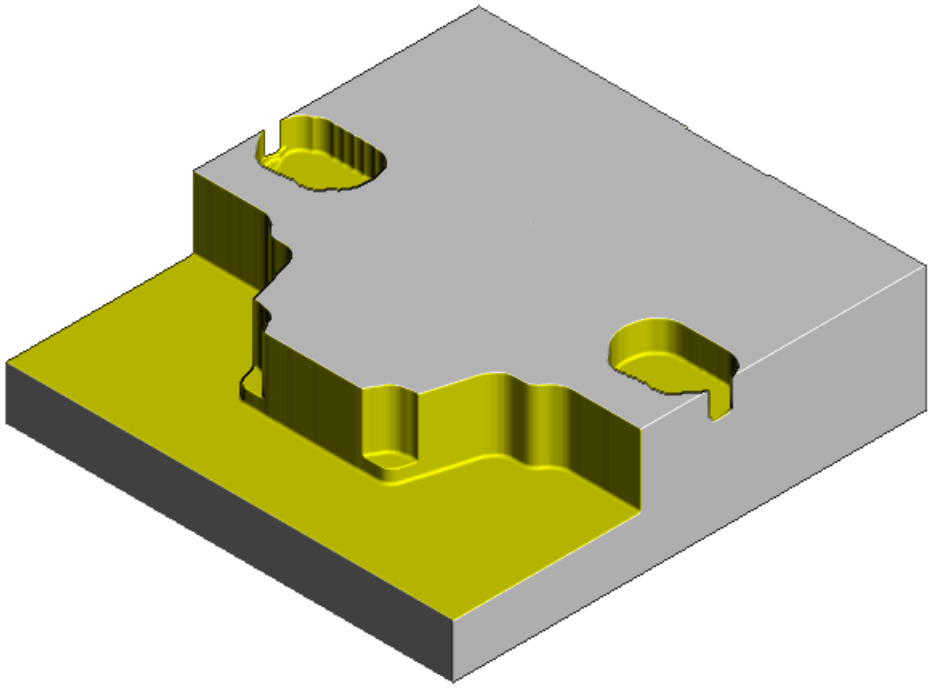
Using the Mould_Core.dgk model in the Examples folder.



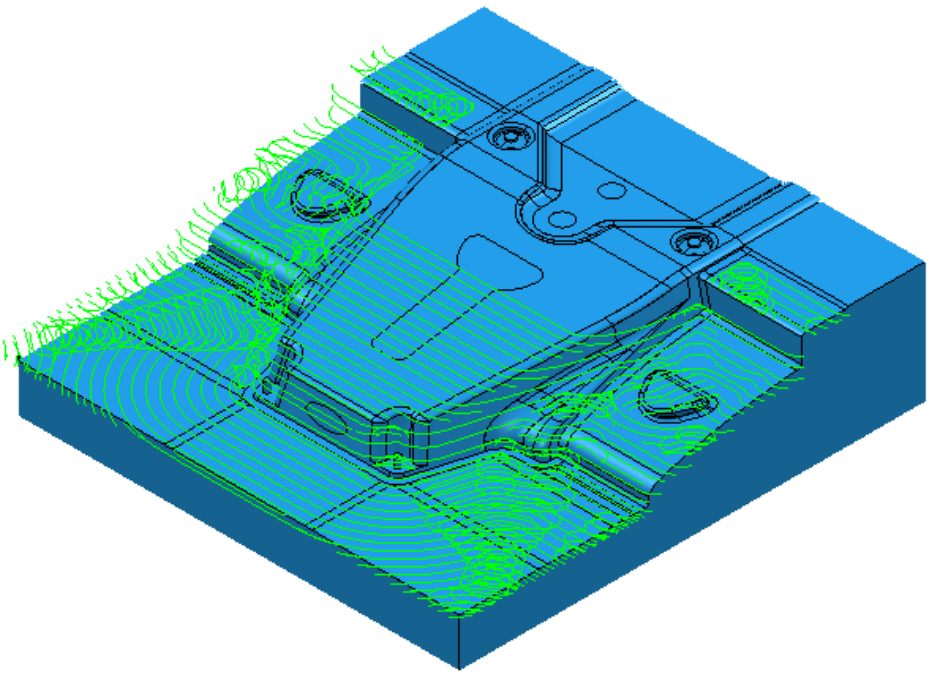
Create a Vortex toolpath without step cutting.



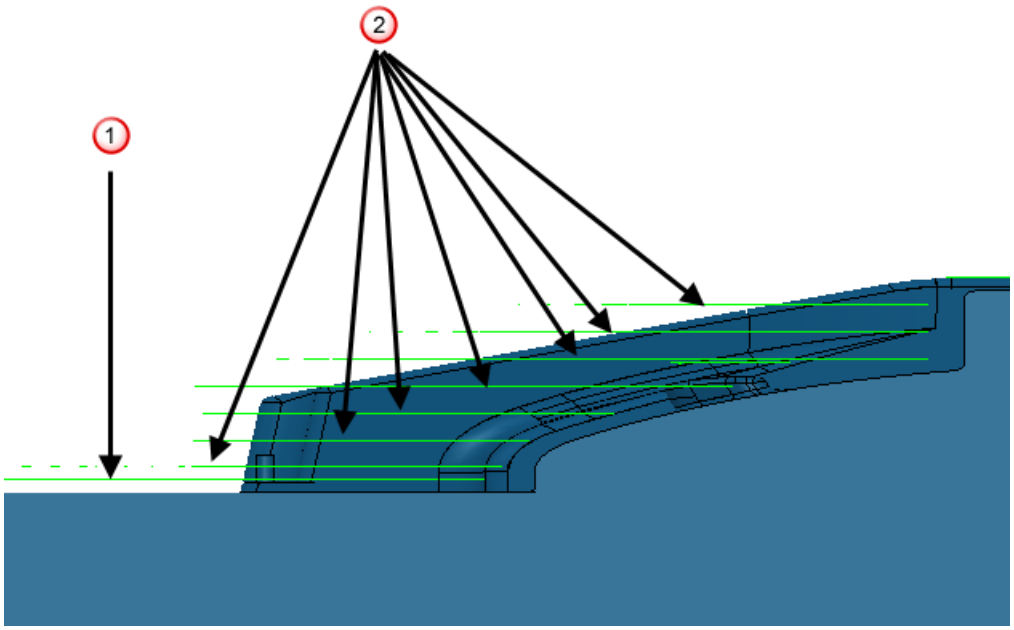
This removes vast quantities of material quickly, but leaves large terraces of unmachined stock on the part.



You can minimise the size of these terraces using the **Step cutting** options.

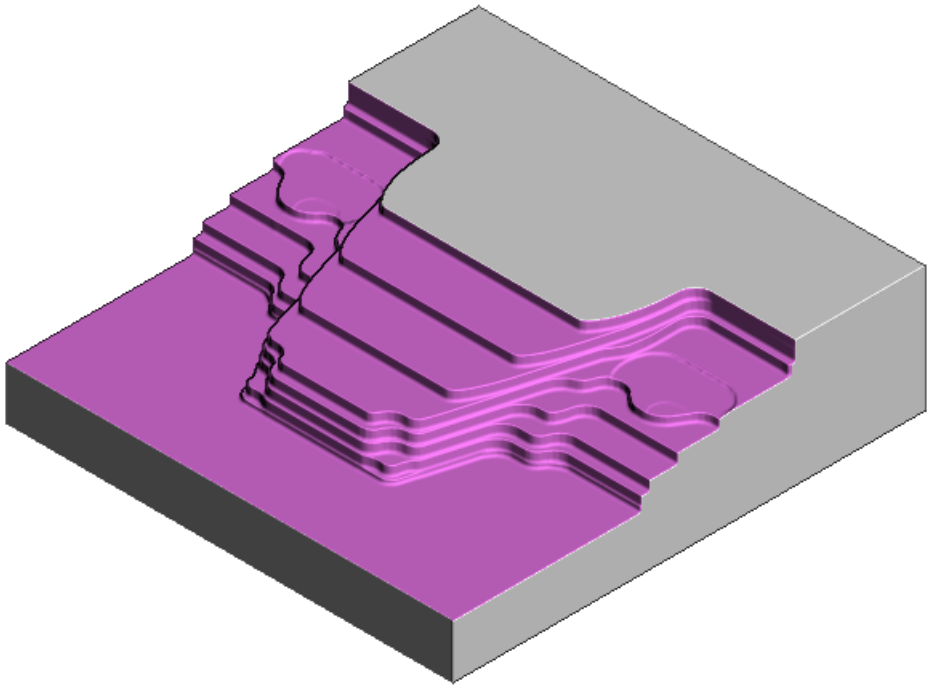


This adds extra machining slices up the part. Looking at a detail of the side view:



1. Original Vortex pass
2. Step cutting passes

It also machines more of the excess stock.

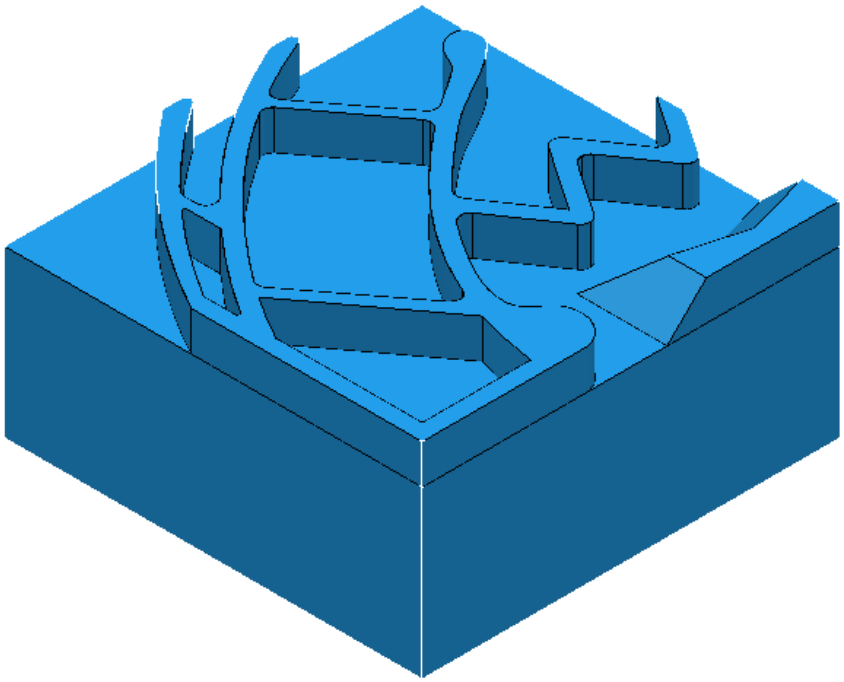


Vortex machining of 2D features

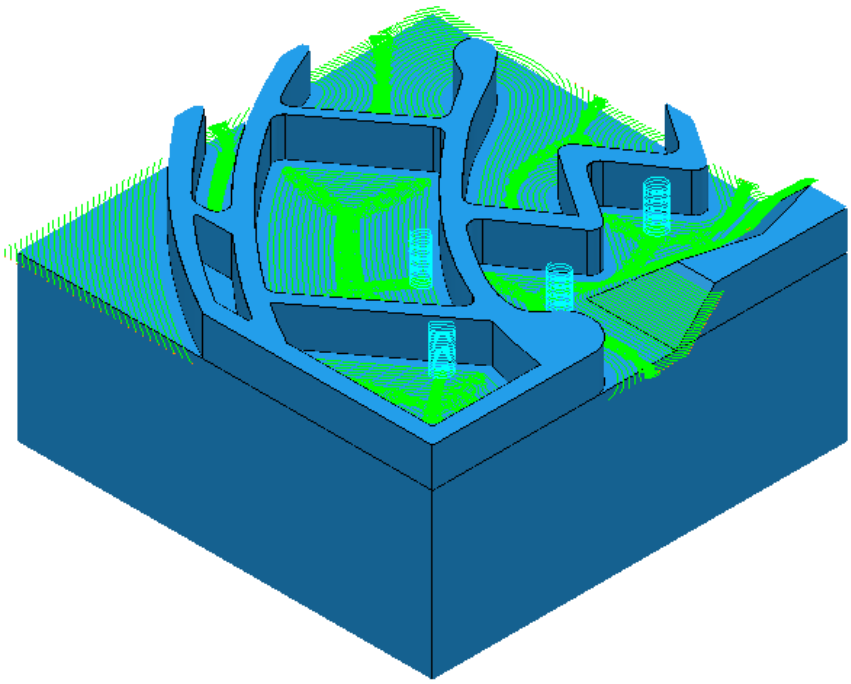
This example shows how Vortex machines pockets, channels and narrow corners.

For more information on the general principles of Vortex machining, see Vortex and step cutting example (see page 12).

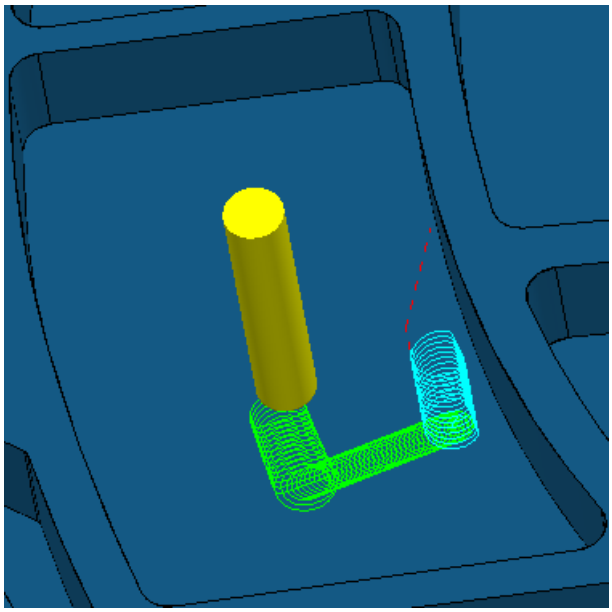
Using the 2D_Features.dgk model in the Examples folder.



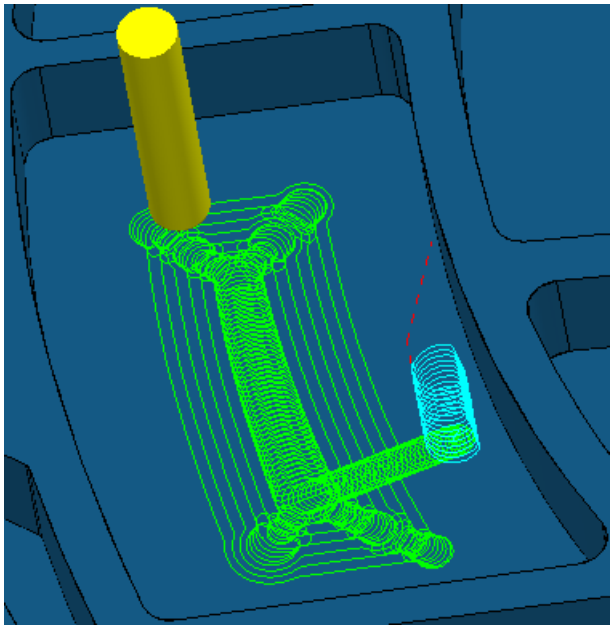
Create a Vortex toolpath:



For pockets, the tool spirals down into the pocket before using trochoidal paths over the full-width cuts.



On completion of the initial full-width cut, the trochoids are placed in the corners where the maximum tool engagement angle would otherwise be exceeded.



Analysing Vortex toolpaths

Vortex toolpaths are automatically checked for safety. Vortex automatically checks for:

- Plunges into stock.
- Excess tool engagement.
- Excess depth of cut
- Small arc movements.

If the toolpath has none of these issues it is shown as safe (), otherwise PowerMILL displays a warning message and is shown as unsafe ().

The warning message details the problems with the Vortex toolpath, for example, the minimum arc radius is too small.

To locate the problem area:

1 From the individual toolpath menu, select **Analyse**.

PowerMILL creates separate toolpaths highlighting the problem areas, in this case **1 : Small arcs_1**.

2 Analyse the problem area of the toolpath. You must make some changes to the toolpath settings to resolve the issue. You may find that changing the **Stepover** by a small amount solves the problem.

It is not advisable to edit the original toolpath by removing the problem areas as this is likely to make the problem worse.

3 Re-calculate the toolpath to check your changes have resulted in a safe toolpath.

Creating MachineDNA

The **MachineDNA Profiler** plugin profiles the unique characteristics of a machine tool. PowerMILL uses the profile data (MachineDNA) to optimise the performance of toolpaths on the machine tool to improve machining efficiency.

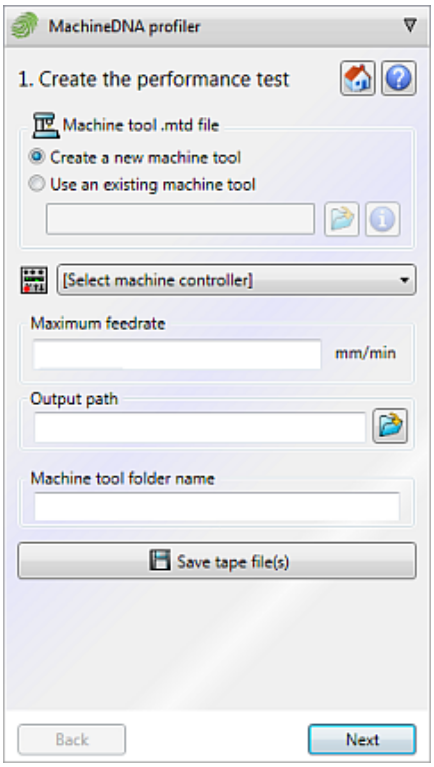
To generate the data required to create MachineDNA, carry out the MachineDNA performance test on the selected machine tool.

Vortex machining uses the test results to create 'clever' values for:

- Most efficient trochoid size.
- Optimal point distribution.

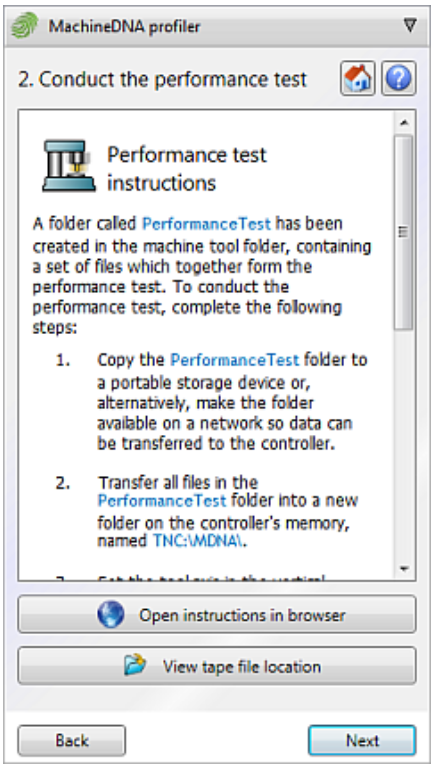
To create MachineDNA, complete the following four steps:

1 Complete the **Create the performance test** page.

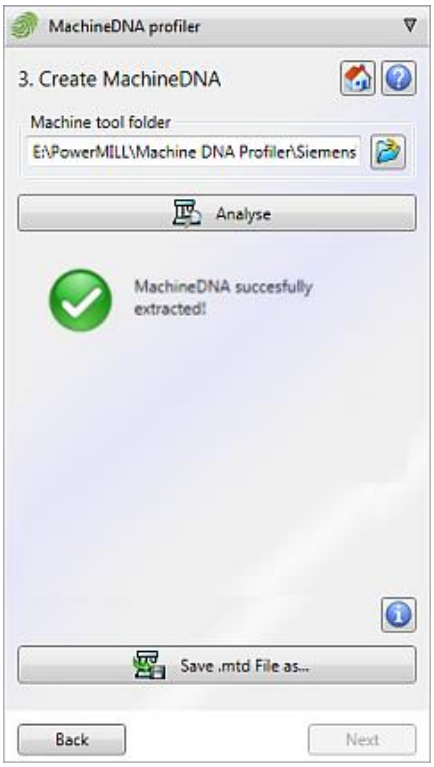


2 Carry out the performance test on your machine tool.

The **Conduct the performance test** page displays instructions to guide you through the test procedure.



3 Transfer the performance test data from the machine controller to your PC and click **Analyse** to create your machine tool's MachineDNA.



4 Save the MachineDNA .mtd file and import it into PowerMILL to begin optimising your toolpaths.

Adding MachineDNA to your PowerMILL suite

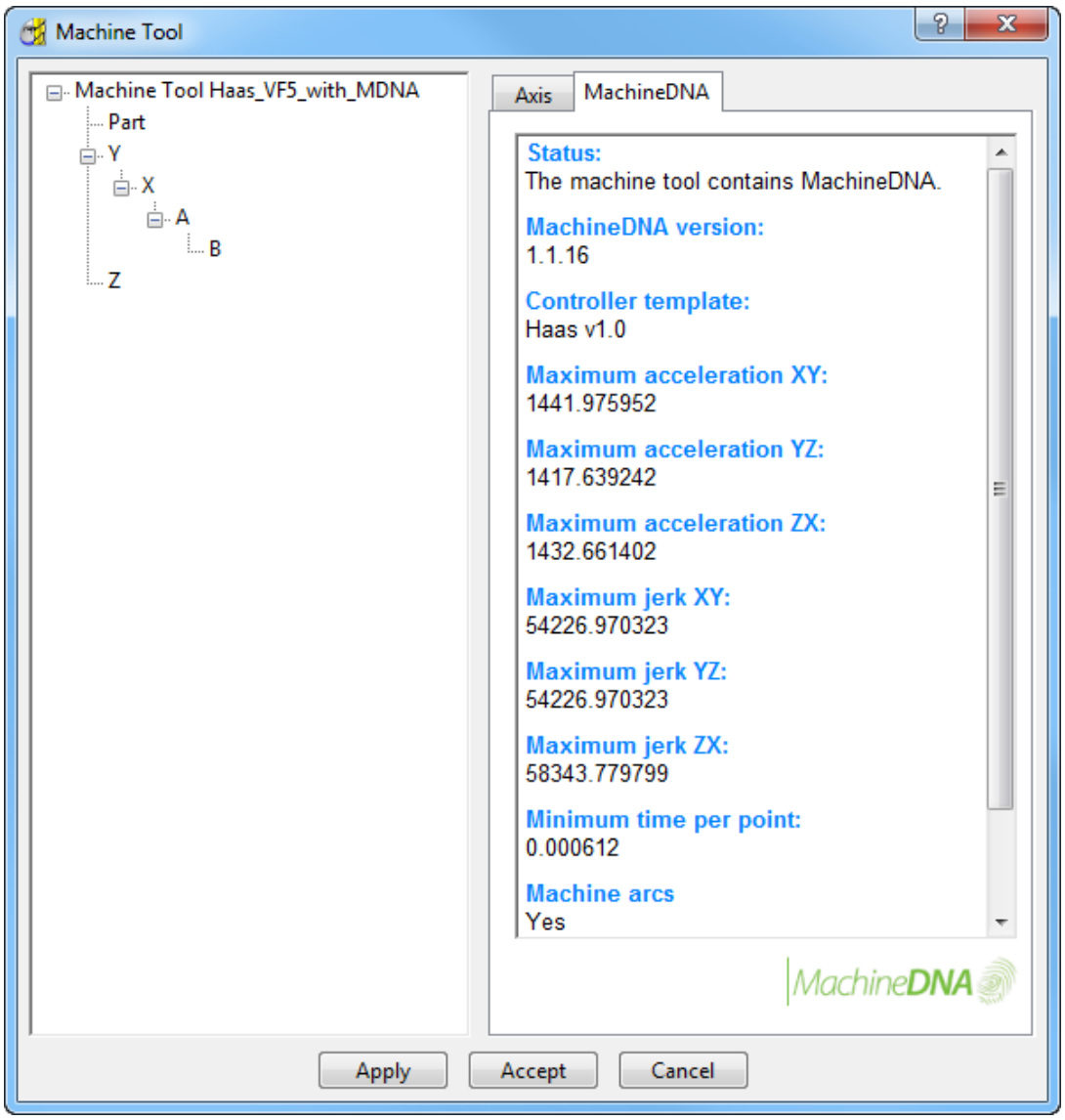
The plugin is easy to install and PowerMILL recognises it automatically.

To use the MachineDNA plugin, you must purchase a licence. The licence (including the plugin) can be purchased at any time. For more information, contact your Delcam sales agent.

Displaying MachineDNA

There is a new MachineDNA tab on the **Machine Tool** dialog that displays a machine tool's MachineDNA parameters.

If you choose MachineDNA to calculate the parameters for a Vortex toolpath, PowerMILL uses these values (see page 8).



Toolpath preparation

Curve enhancements

There are several new options on the **Curve Editor** mode toolbar:

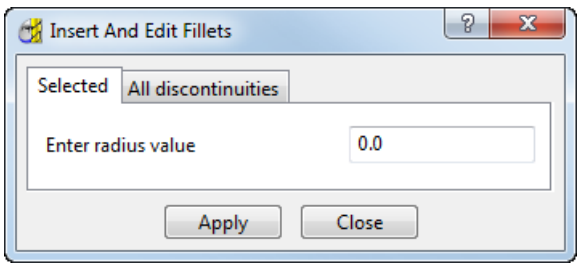
- **Insert and edit fillets** — Use to insert and edit the fillets in a composite curve containing arcs and lines. This maintains arcs as arcs and doesn't convert them to a spline (see page 22).
- **Tangent editor** — Use to edit the tangents of a point on a Bézier curve. This enables more precise tangent editing as previously you could only edit tangents graphically (see page 34).
- **Repoint curve** — Use to redistribute points, or change the number of points, along a Bézier curve (see page 40).

There are enhancements to the **Reordering curve segments** dialog (see page 49).

You can now trim or extend both ends of a single curve segment (see page 50).

Insert and Edit Fillets dialog

The **Insert and edit fillets** button on the **Curve Editor** mode toolbar inserts and edits the arcs in a composite curve containing lines and arcs.

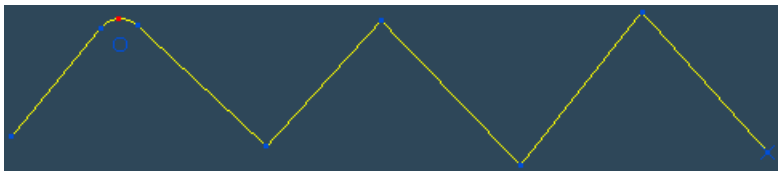


You must select a composite curve containing only arcs and lines to enable this option.

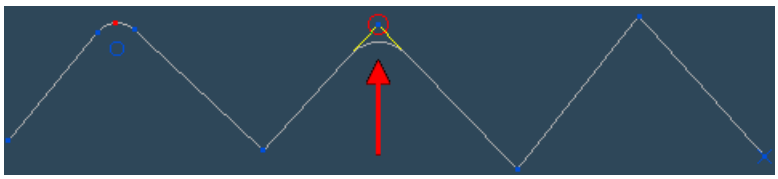
Enter Radius — Enter the radius of the fillet.

If you select a specific point or arc centre, the fillet is applied just to that point or arc.

Starting with this curve:



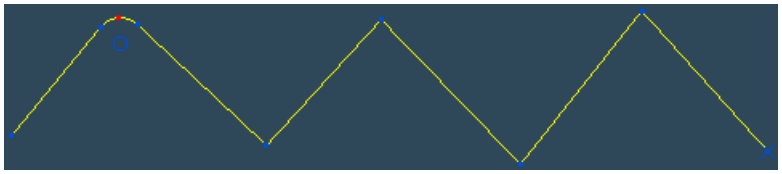
Selecting one point and entering a radius gives:



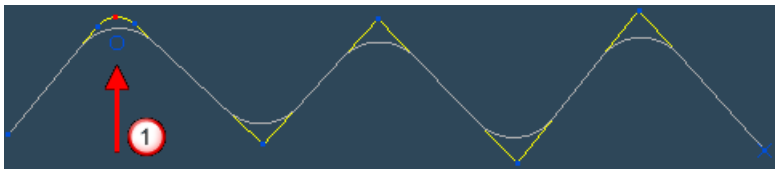
If you don't select any points or arcs, the fillet is applied to:

- Any points which are between two lines and any existing arc spans, if you select the **Selected** tab.

  Starting with this curve:

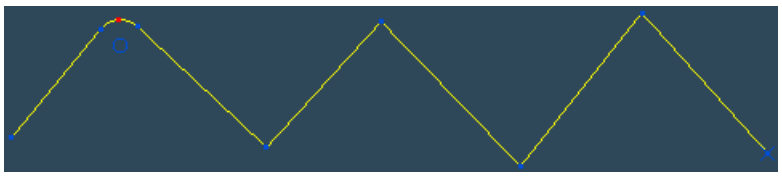
  On the **Selected** tab entering a radius produces:

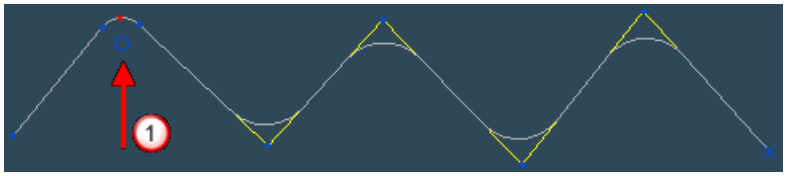
  This edits the fillet at 1 as well adding fillets at points between two lines.

- Any points which have a tangency discontinuity, if you select the **All discontinuities** tab.

  Starting with this curve:

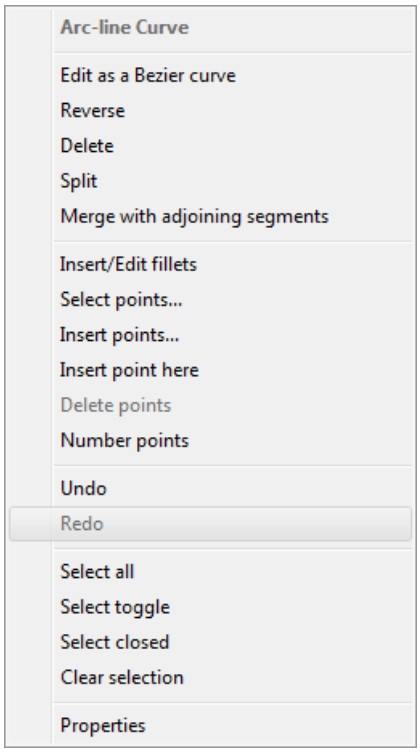
On the **All discontinuities** tab entering a radius gives:



The fillet at 1 is not changed.

Arc-line Curve menu

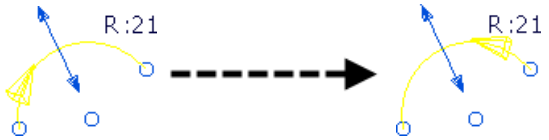
Display the **Arc-line Curve** context menu, when the **Curve editor** mode toolbar is raised, by right-clicking on a composite curve containing lines and arcs.



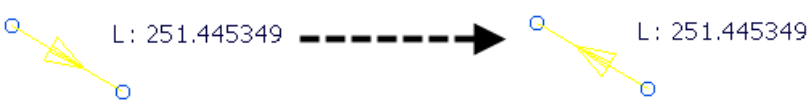
Edit as a Bézier — Converts the curve to a spline and edits it as a Bézier curve.

Reverse — Click to reverse the direction of the selected items.

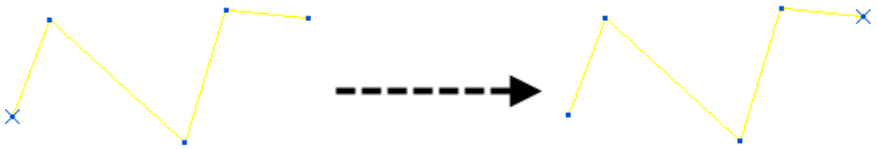
If you select an arc, it converts:



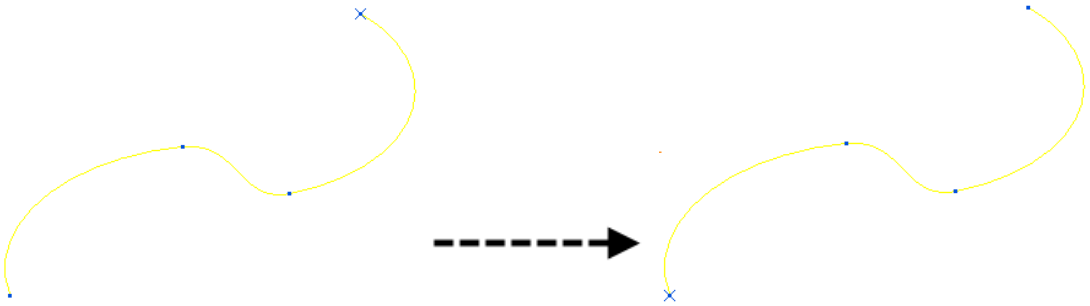
If you select a line, it converts:



If you select a continuous line there is no arrow on the line, but an X marks the end of the continuous line. It converts:



If you select a Bézier curve there is no arrow on the line but an X marks the end of the Bézier curve. It converts this:



For more information, see **Reverse selected item example**.

Delete — Click to delete the selected items. This is the same as  on the **Curve editor** mode toolbar.

Split — Click to break the selected segments into the individual curves used to create it. This is the same as  on the **Curve editor** mode toolbar.

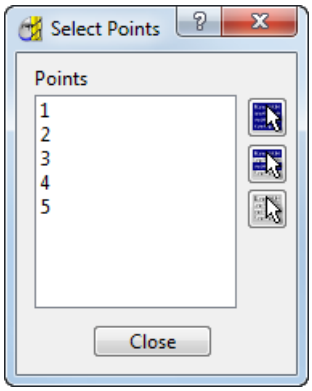
Merge with adjoining segments — Click to merge all the segments adjacent to the picked segment.

The curve editor merges segments that join the selected segment (at either end). The curve editor then merges segments that join this new segment. This continues until no more merges are possible. Where there is a choice of curves to merge, the curve with the smallest tangent angle discrepancy is chosen. This is the same as  on the **Curve editor** mode toolbar.

Insert/Edit fillets — Click to edit the fillets in a composite curve containing arcs and lines. This maintains arcs as arcs and doesn't convert them to a spline (see page 22). This is the same as  on the **Curve Editor** mode toolbar.

Select point — Click to select points which you can then edit or delete using the **Select Points** dialog. This is the same as  on the **Curve editor** mode toolbar.

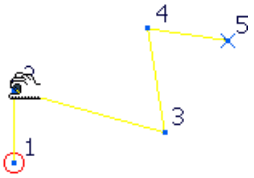
PowerMILL displays the **Select Points** dialog.



This example uses a continuous line, but the functionality works in the same way for a Bézier curve or arc-line.

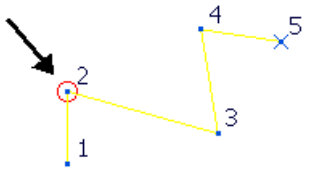
You can select a point:

- graphically; or,



- from the **Select Points** dialog.

A red circle is placed on the selected point.



You can select additional points graphically using the Shift key. You can deselect points using the Ctrl key. Shift plus a drag box selects all the points in the box and Ctrl plus a drag box deselects all the points in the box.

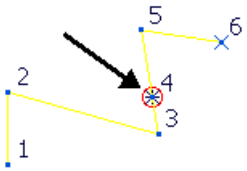
Insert point — Click to add a point between a pair of selected points. This is the same as  on the **Curve editor** mode toolbar.

Insert points here — Click to add a point where you selected the curve. This is the same as selecting  on the **Curve editor** mode toolbar and then selecting the **Through nearest point** tab on the **Insert Point into Curve** dialog.

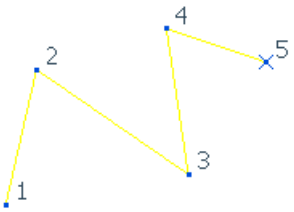
Delete points — Click to delete the selected point. This is the same as  on the **Curve editor** mode toolbar.

This example uses a continuous line, but the functionality works in the same way for a Bézier curve or arc-line.

Converts this:

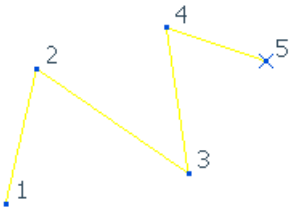


To this:



Number points — Click to number the points on the selected curve. This is the same as  on the **Curve editor** mode toolbar.

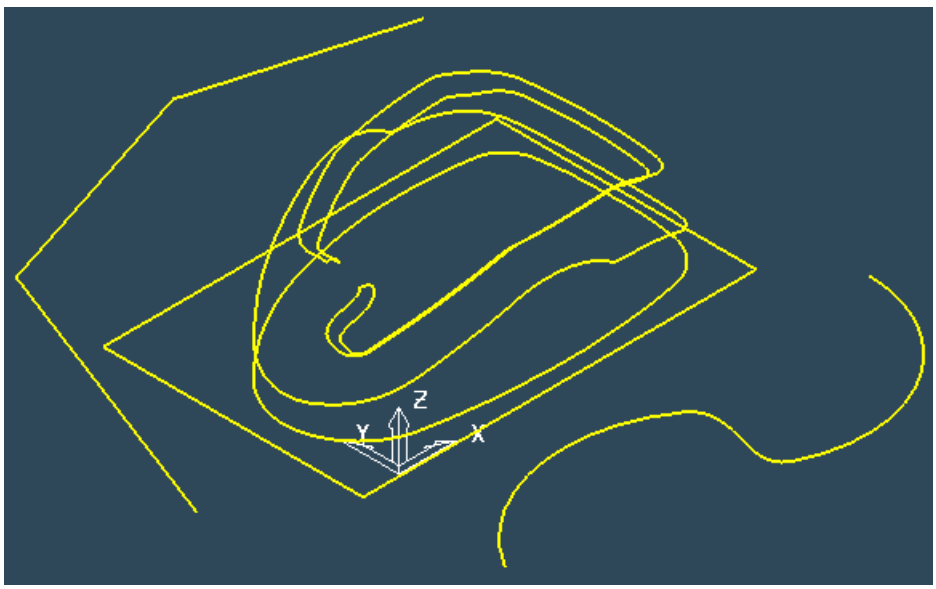
This example uses a continuous line, but the functionality works in the same way for a Bézier curve or arc-line.



Undo — Click to revert to what it was before the last change. This is the same as  on the **Curve editor** mode toolbar.

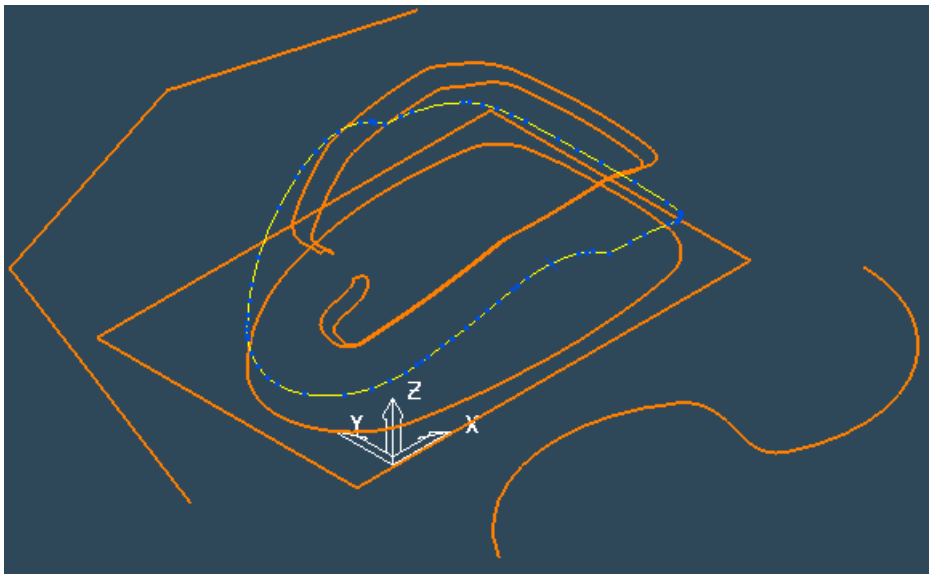
Redo — Click to reinstate the edit you have just undone. This is the same as  on the **Curve editor** mode toolbar.

Select all — Click to select all the curves. This is the same as  on the **Curve editor** mode toolbar.



Select toggle — Click to deselect the selected curves and select the deselected curves. This is the same as  on the **Curve editor** mode toolbar.

Converts this:

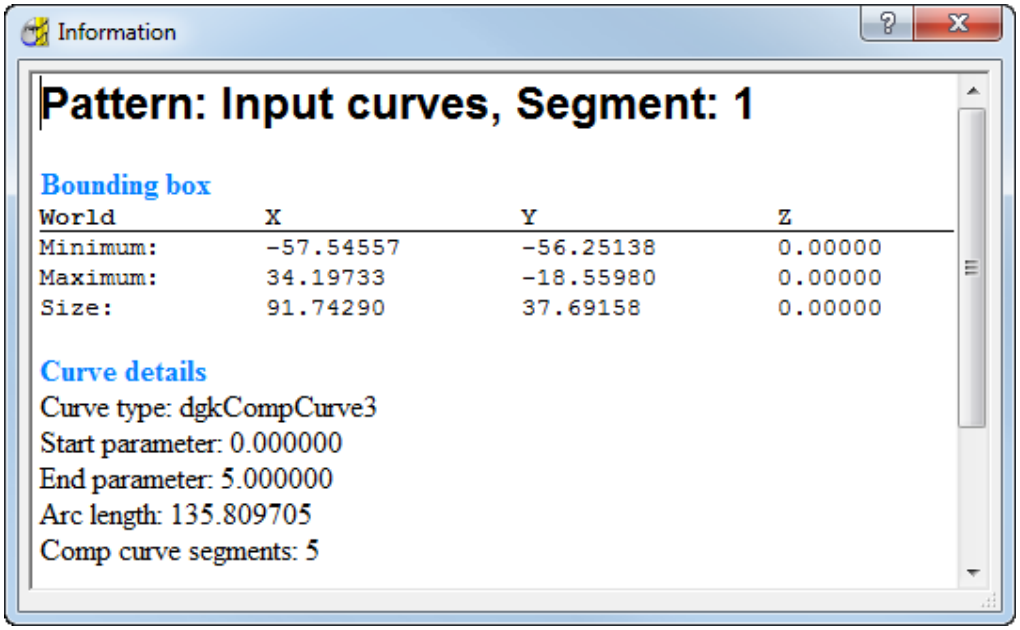


to this:

Select closed — Click to select all the closed curves and deselect the open curves. This is the same as  on the **Curve editor** mode toolbar.

Clear selection — Click to deselect the selected curves.

Properties — Click to display the extents, type, and key points of the curve.



Pattern — Displays the name of the pattern, in this case **Input curves**.

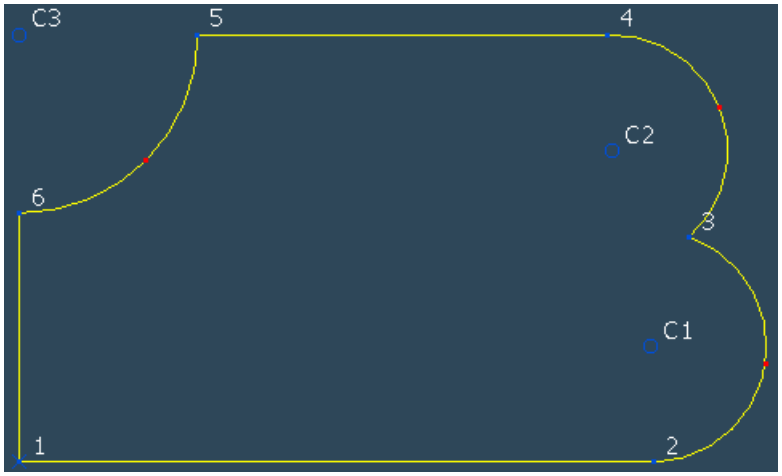
Segment — Displays the properties of this segment. In this case the first segment.

Editing a composite curve

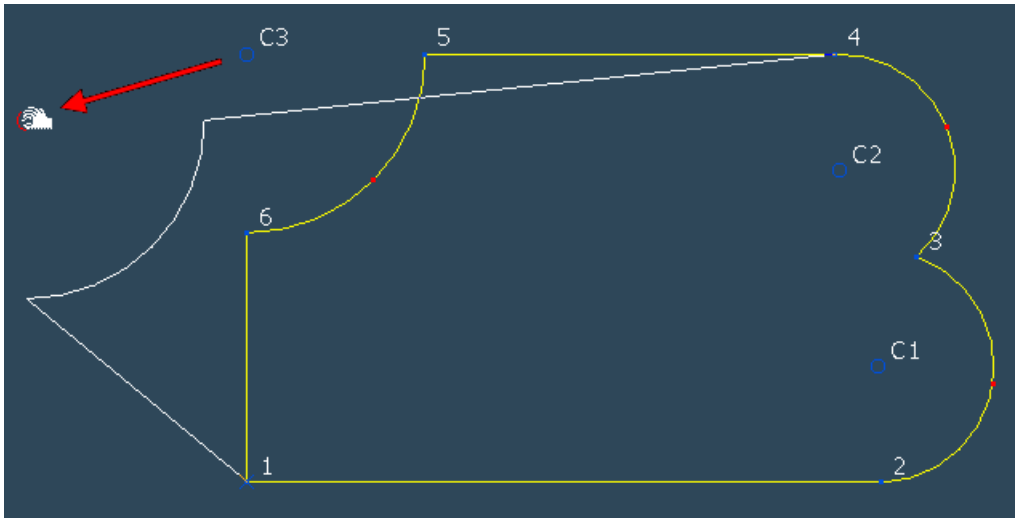
This example shows you how to edit a composite curve made up of lines and arcs. This works in a similar way to editing a line and editing an arc and assumes you know how to edit arcs and lines.

1 Select the composite curve to edit it.

If you aren't in curve editing mode, double-click the curve to edit it.

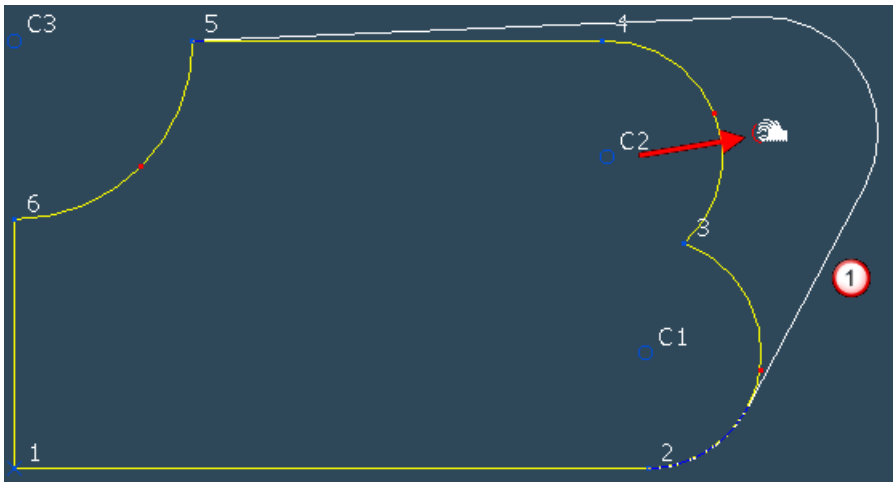


2 Click and drag point **C3**.



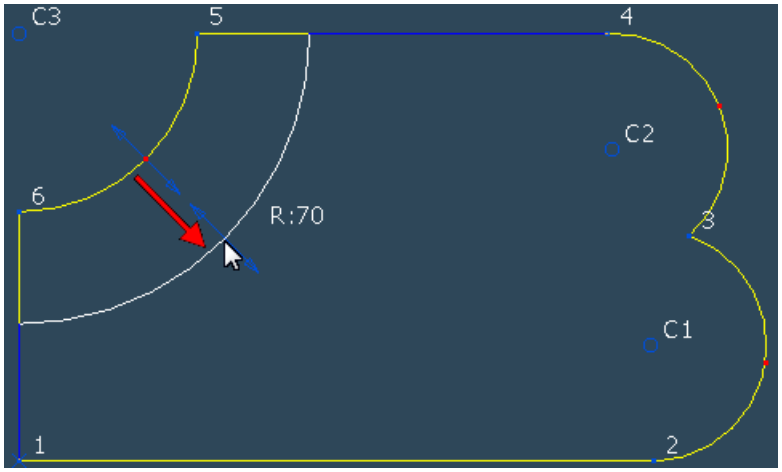
The radius of the arc remains the same. This changes the length and angle of the lines to rejoin the moved arc.

3 Click and drag point **C2**.



If possible, when the arc being moved is adjacent to another arc, the two arcs are trimmed at their intersection point. If there is no intersection point then a tangential line is added (1).

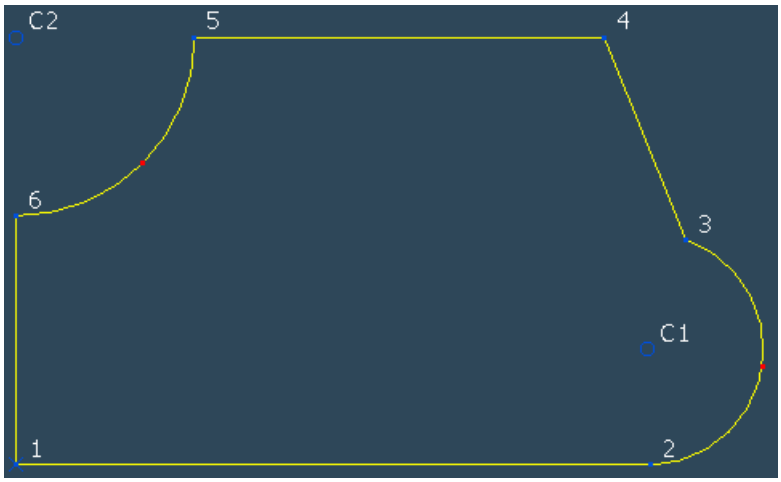
4 Click and drag the red point between point **5** and **6**.



The centre point of the arc remains unchanged. The radius of the arc changes and the lines attached to the arc are re-trimmed.

5 Select point **C2** and click  to delete the arc.

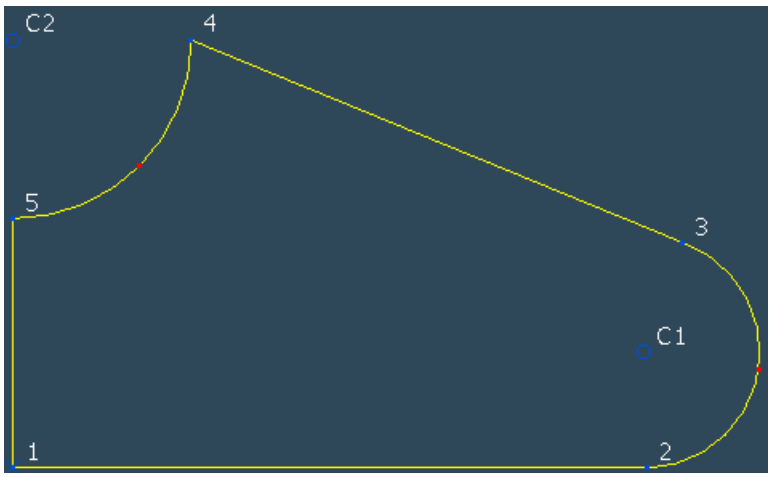
This replaces the arc with a line between points **3** and **4** and renumbers the arcs (so **C3** becomes **C2**).



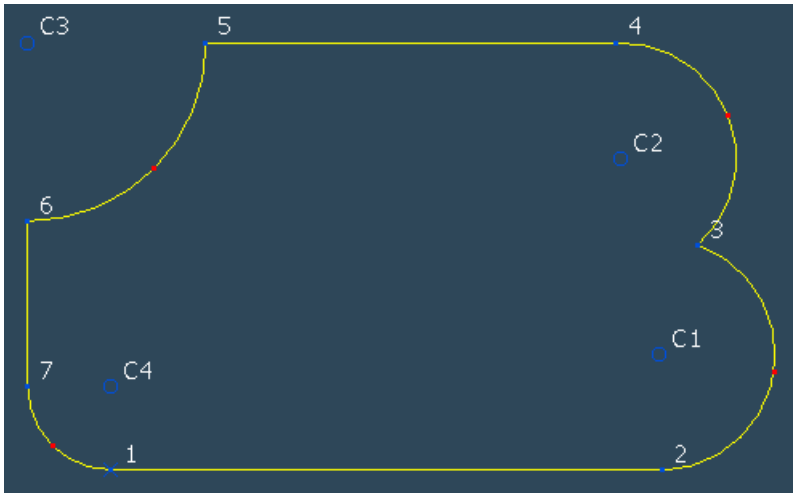
This replaces the arc with a line between points **3** and **4**.

6 Click to revert to step 4.

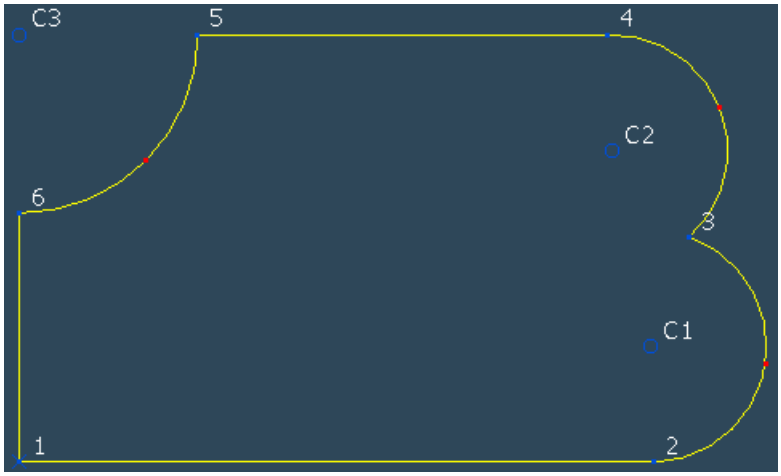
7 Select point **4** and click to delete the arc.



If you have a filleted corner:

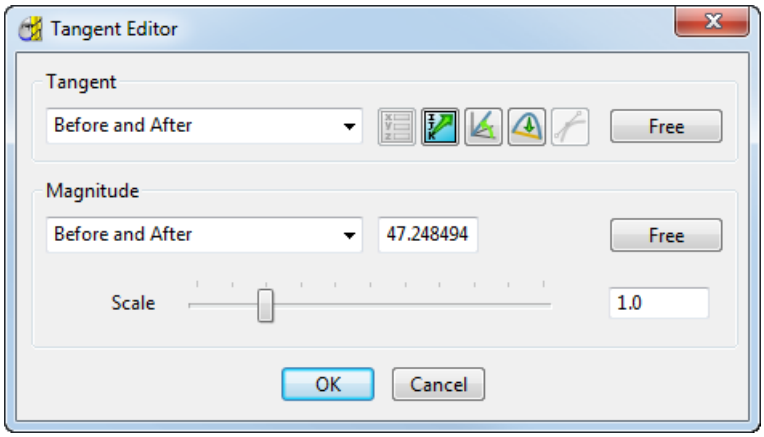


Deleting the fillet (**C4**) assumes you want to reduce the fillet radius to 0.



Tangent editor

The **Edit tangents** button on the **Curve Editor** mode toolbar edits the tangents of selected points in a Bézier curve.

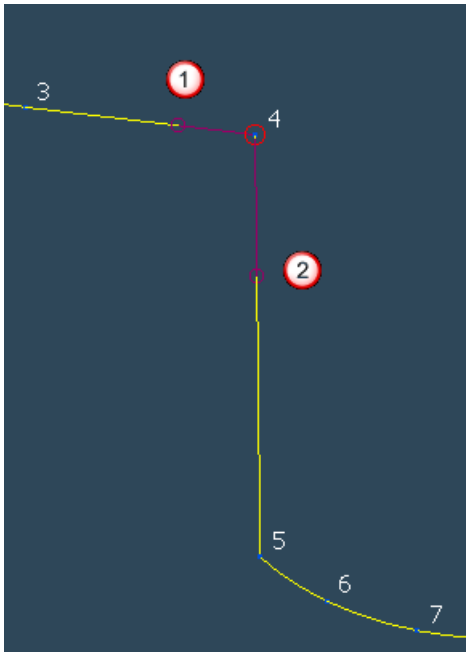


Tangent — Edits the tangent directions. Select which tangent you want to edit.

Before and after edits both tangents. This is the default option.

Before edits the tangent approaching the point.

After edits the tangent after the point.



1 Tangent before

2 Tangent after

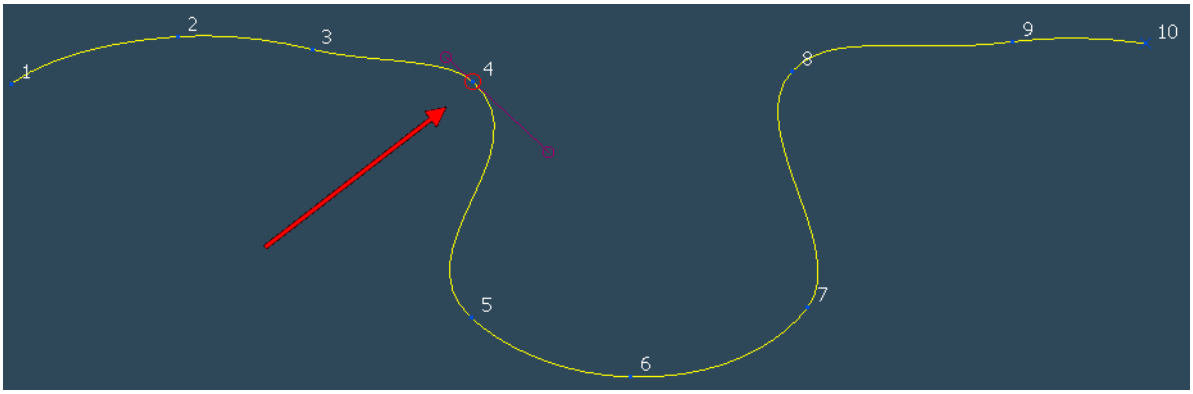
 — Click to display the **Position** dialog. Use the dialog to manually enter coordinates and locate items in the graphics window. This option isn't available if you have a **Tangent** option of **Before and after**.

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

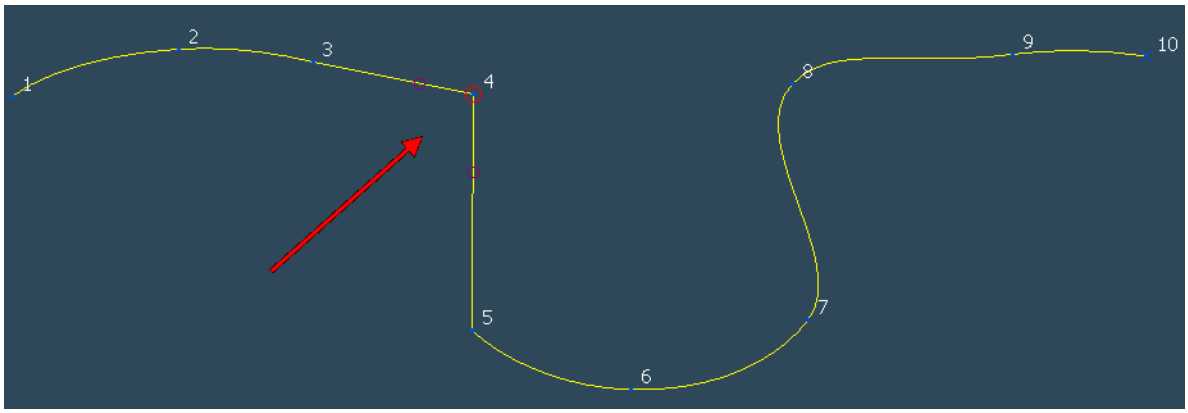
 **Azimuth/Elevation** — Click to display the **Azimuth/Elevation** dialog, which enables you to enter the azimuth and elevation angles.

 **Straighten** — Click to change the tangency direction to form a straight line span.

With the tangent **Before and After** selected, converts the tangents on point 4 from this:

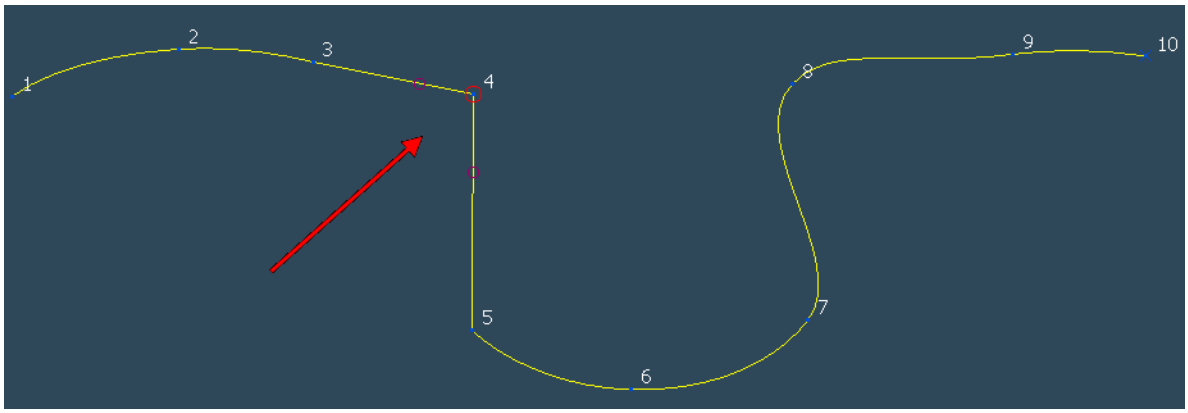


to this:

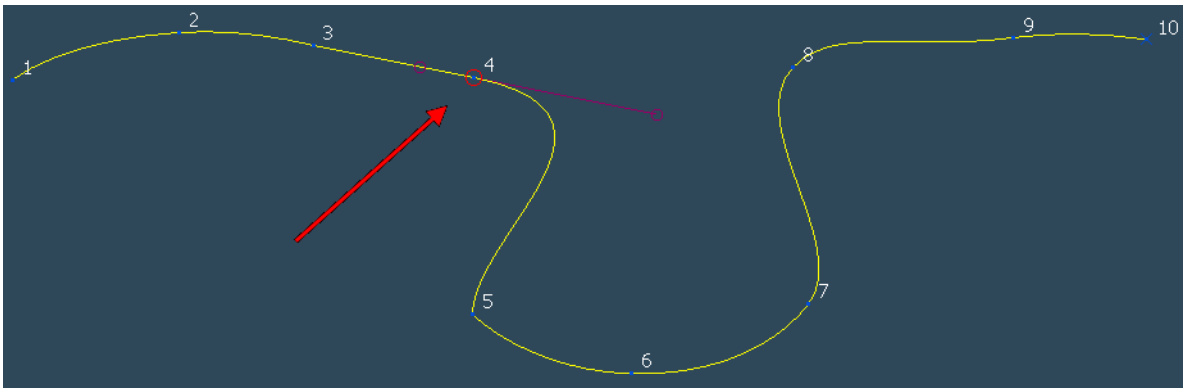


 **Align tangent** — Click to align the selected tangency direction with the other tangent at that point. This option isn't available if you have a **Tangent** option of **Before and after**.

With the tangent **After** selected, converts the tangent on point 4 from this:



to this:



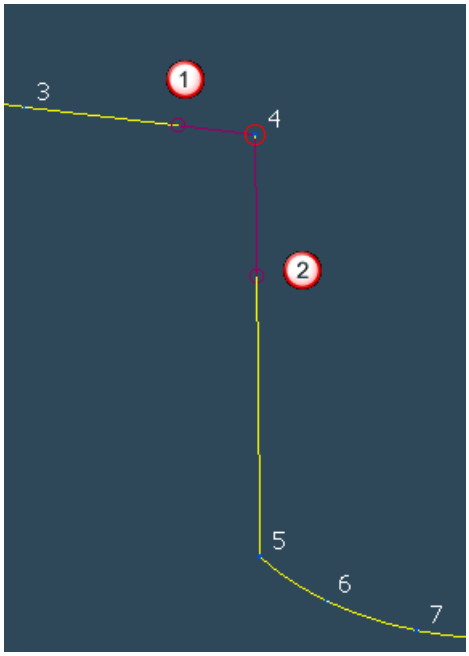
Free — Click to free the tangent direction. This maintains the point location and fits a smooth curve through the selected points.

Magnitude — Edits the tangent magnitude. Select which tangent you want to edit.

Before and after edits both tangents. This is the default option.

Before edits the tangent approaching the point.

After edits the tangent after the point.



1 Tangent before

2 Tangent after

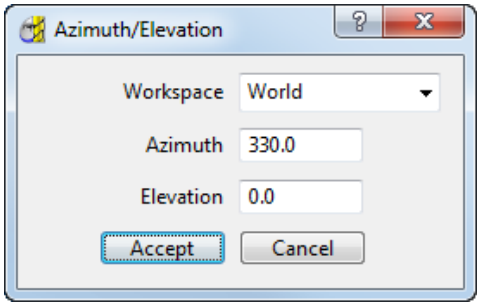
—Enter the magnitude of the tangent.

Free — Click to edit the tangent magnitudes. This maintains the point location and fits a smooth curve through the selected points.

Scale — Edit the tangent magnitudes by scaling the current value using the slider or entering a scaling factor (see page 39).

Azimuth/Elevation dialog

Clicking  on the **Tangent Editor** dialog displays the **Azimuth/Elevation** dialog (see page 34).



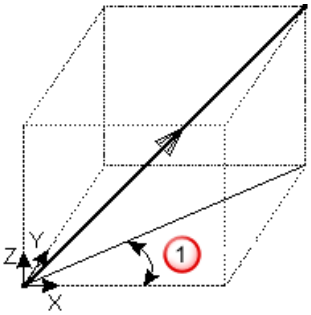
Workspace — Specify which workspace you are working in.

World specifies the point with respect to the global coordinate system.

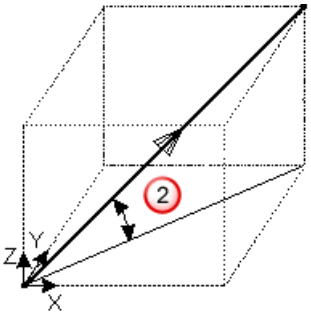
Workplane specifies the point with respect to the active workplane.

This option is only available if an active workplane exists, and, if you are editing a workplane, you are editing a non-active one.

Azimuth — Enter the angle of the line in the XY plane. The rotation is measured anticlockwise about the Z axis with 0° at the X axis.

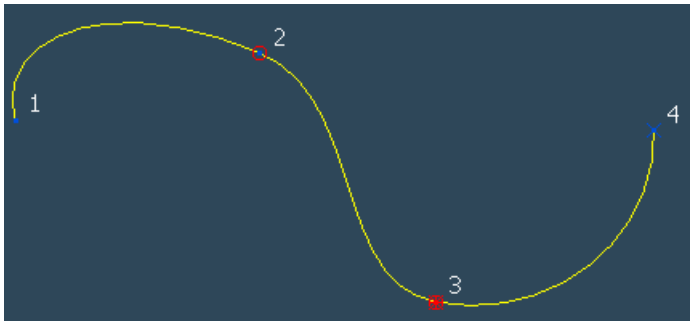


Elevation — Enter the angle of the line relative to the vertical (Z).

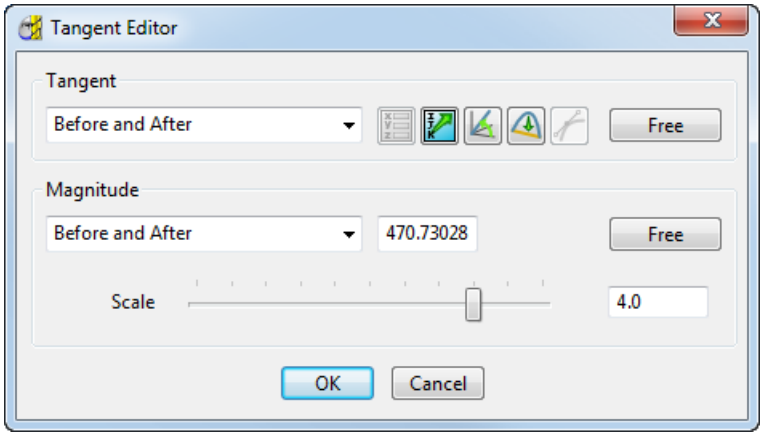


Editing the tangent magnitude

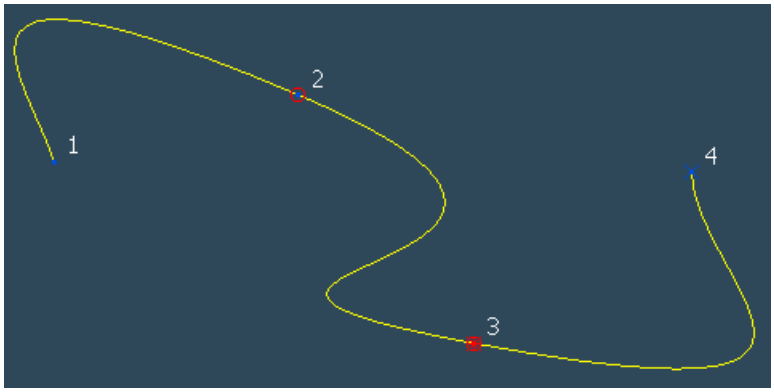
Starting with this curve:



1. Double-click on the curve.
2. Select points 2 and 3 (pressing the Shift key while selecting point 3).
3. Click  on the **Curve Editor** mode toolbar to display the **Tangent Editor** dialog.

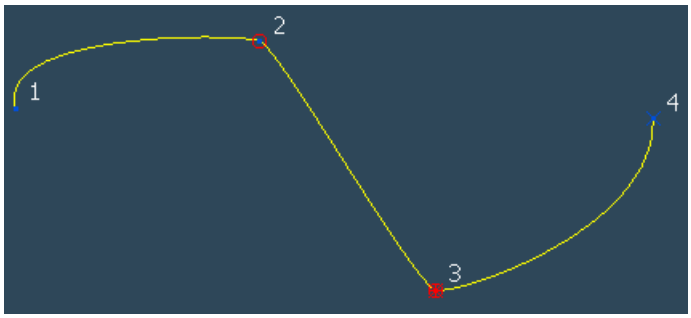


4. Enter a **Scale** of **4**.



5. Click **Cancel**.
6. Click  .

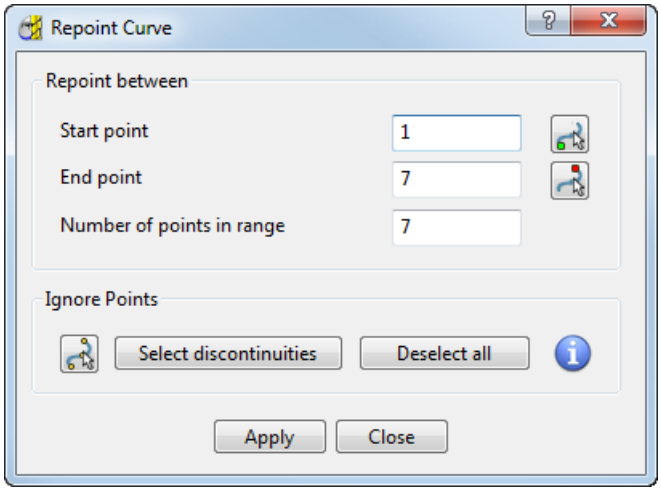
7 Enter a **Scale** of **0.2**.



8 Click **OK**.

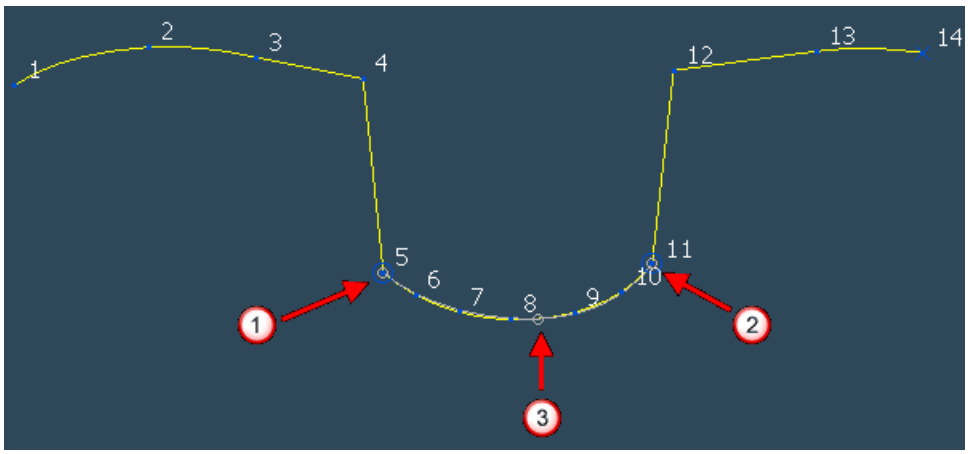
Repoint curve

The **Repoint curve** button on the **Curve Editor** mode toolbar redistributes points along a Bézier curve. The start point and end point remain in the same position and have the same tangency direction as before; the remaining points are distributed at equidistant intervals between the start and end points.



Since repointing a curve uses point numbers it is often useful to display them by clicking on ***Number Points*** *.*

Repoint between — Determines which portion of the Bézier curve you want to redistribute. For an example, see Reducing the number of points in a curve (see page 45).



1 Start point

2 End point

3 With a **Number of points in range** of **3** this replaces points 6 – 10 with one point here.

Start point — Enter the point number where you want the redistribution to start.

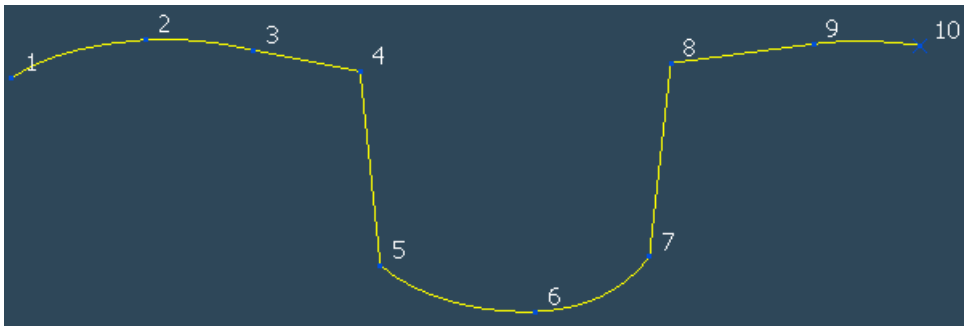
Select Start — Click to select the start point interactively. This changes the cursor to . Then click on the point where you want to start the redistribution.

End point — Enter the point number where you want the redistribution to end.

Select End — Click to select the end point interactively. This changes the cursor to . Then click on the point where you want to end the redistribution.

Number of points in range — Enter the number of points in the redistributed curve. This is inclusive of the start and end points, so the minimum value is 2. The points are evenly distributed along the curve

For example, using this curve:



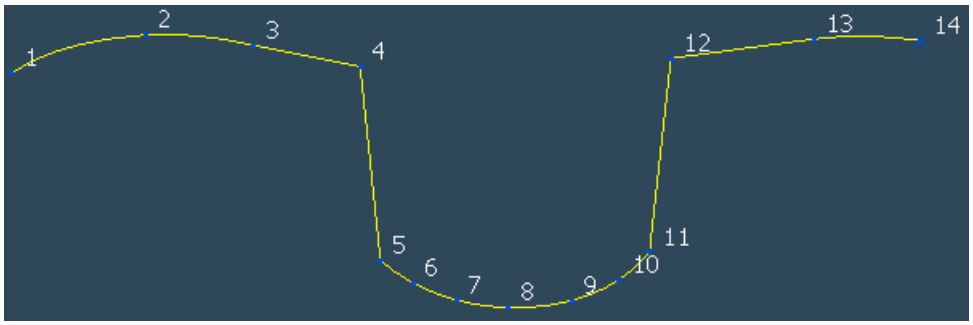
a Enter a **Start point** of 5.

b Enter an **End Point** of 7.

c Enter a **Number of points in range** of 7.

d Click **Apply**.

Produces:

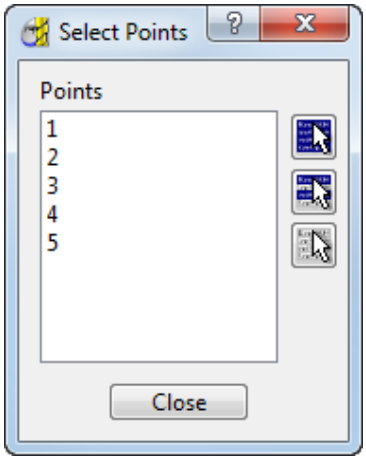


Ignore Points — When redistributing the points along a curve there may be specific points, for example, those with specific tangency directions or location, that you want to keep. The selected points remain unchanged during the repoint calculation. For an example see Ignoring points when redistributing points in a curve (see page 46).

*The **Number of points in range** excludes any points selected here. So, if you have 4 ignored points and a **Number of points in range** of 10 you end up with 14 points in the curve. The start point, end point and 4 ignored points are in the same position and have the same tangency direction as before, the 8 remaining points are distributed at equidistant intervals between the start and end points.*

Select point — Click to select points which you can then edit or delete using the **Select Points** dialog.

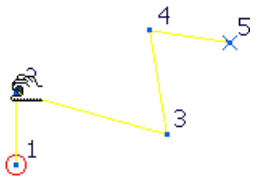
Displays the **Select Points** dialog.



This example uses a continuous line, but the functionality works in the same way for a Bézier curve or arc-line.

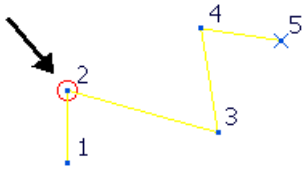
You can select a point:

- graphically; or,



- from the **Select Points** dialog.

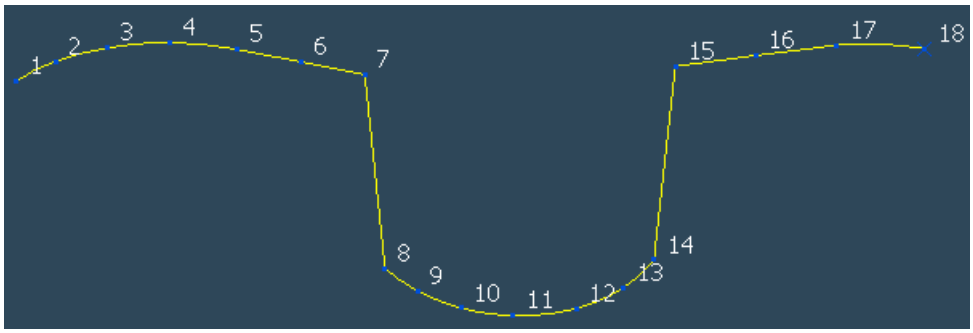
A red circle is placed on the selected point.



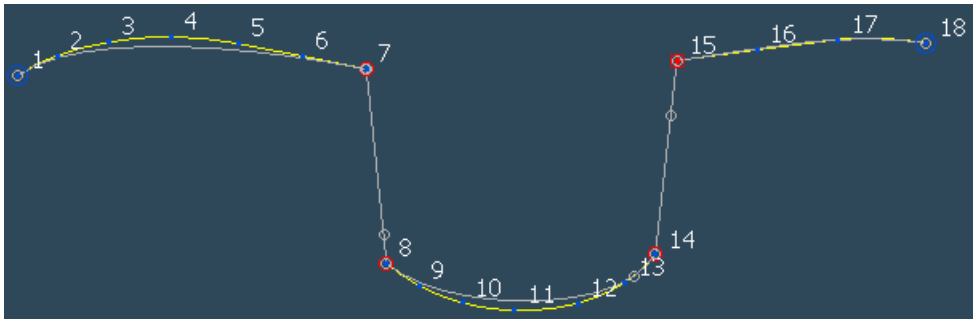
You can select additional points graphically using the Shift key. You can deselect points using the Ctrl key. Shift plus a drag box selects all the points in the box and Ctrl plus a drag box deselects all the points in the box.

Select Discontinuities — Click to select the points with tangent discontinuities (where the tangent before the point is different to the tangent after the point).

In this example:

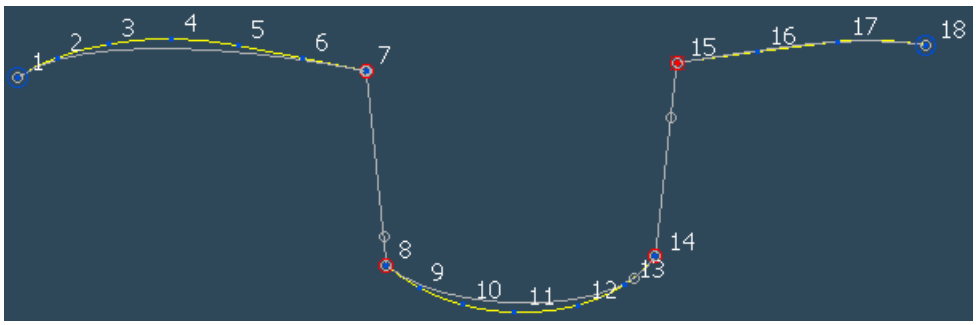


Clicking on **Select continuities** selects points 7, 8, 14, and 15.

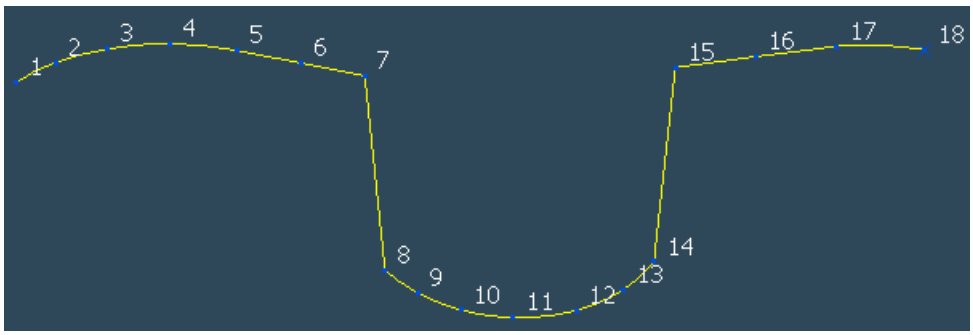


Deselect All — Click to deselect all selected points.

Clicking on **Deselect All** converts this:



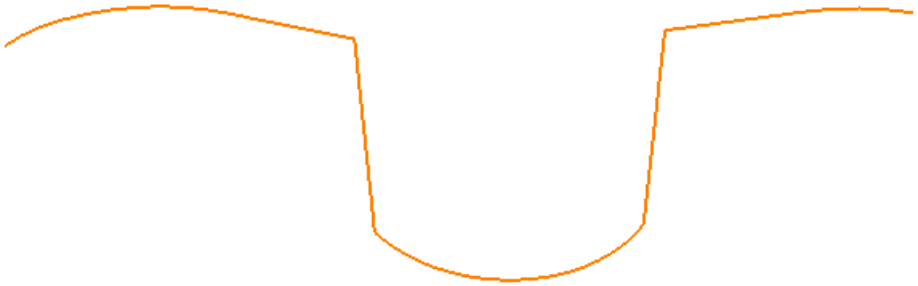
To this:



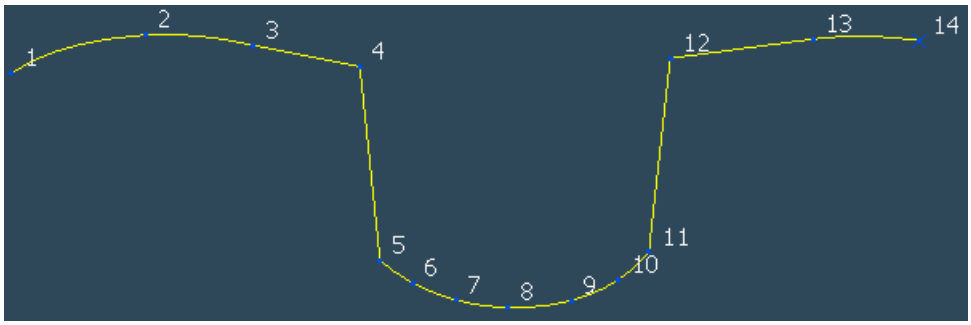
Information — Hover to give balloon help.

Reducing the number of points in a curve

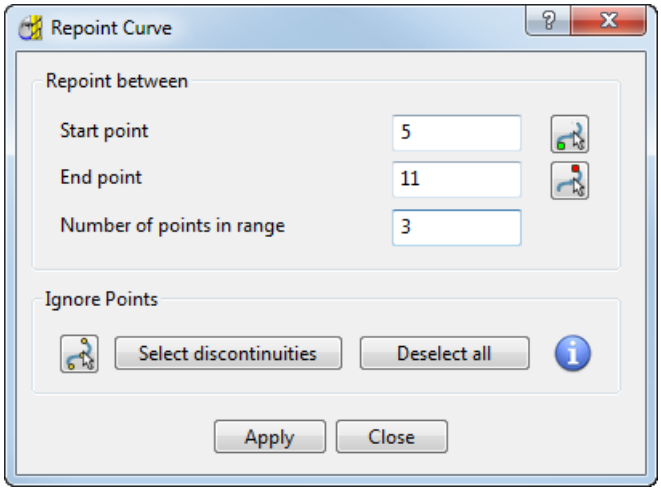
This example shows you how to reduce the number of points in a Bézier curve. Starting with this Bézier curve:



1. Double-click on the Bézier curve to open the **Curve Editor** mode toolbar.
2. Click **Number Points** to show the points in the Bézier curve.

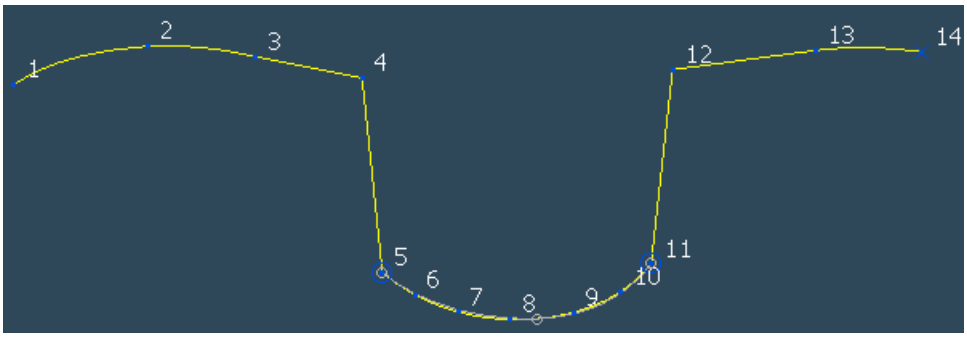


3. To reduce the number of points in the portion of the curve at the bottom of the trough, click to display the **Repoint curve** dialog.



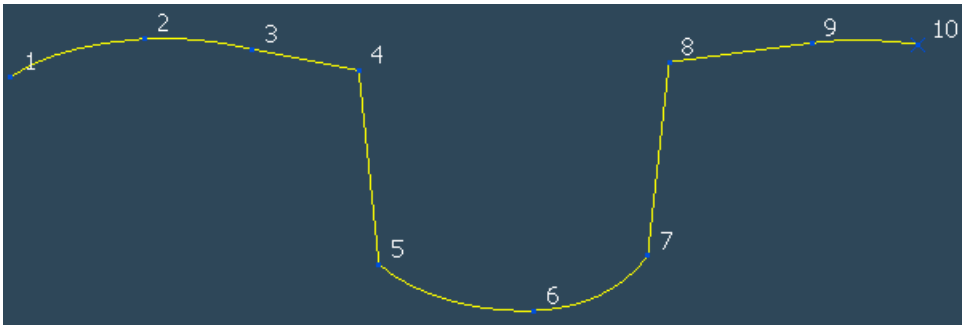
4. On the **Repoint Curve** dialog:
   a. Enter a **Start point** of 5.
   b. Enter an **End Point** of 11.
   c. Enter a **Number of points in range** of 3.

d Press the tab key to indicate the effect of the repoint calculation.



e Click **Apply**.

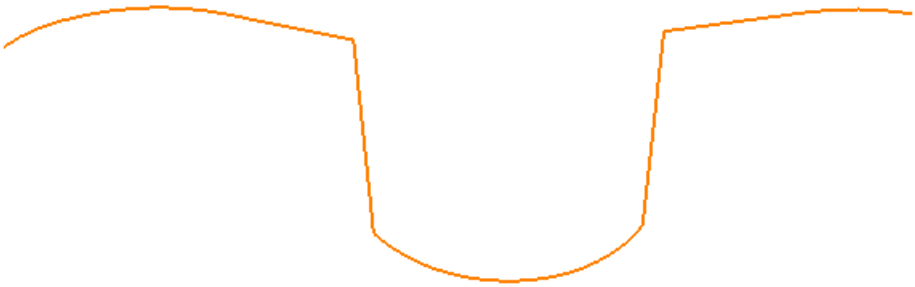
f Click **Cancel**.



5 On the **Curve Editor** mode toolbar click  to accept the changes.

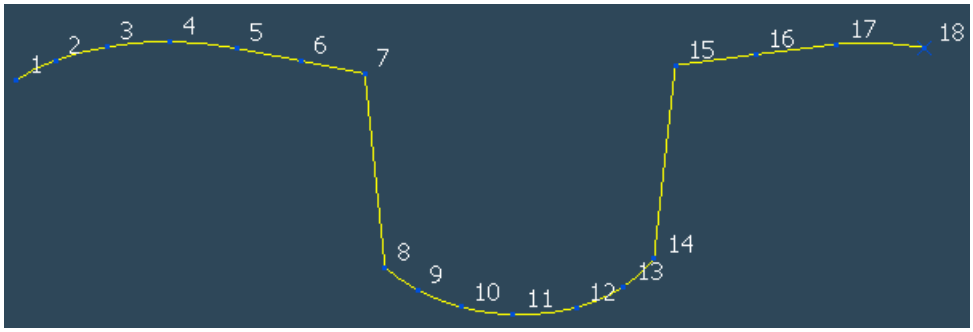
Ignoring points when redistributing points in a curve

This example shows how to reduce the number of points in a Bézier curve. Starting with this Bézier curve:

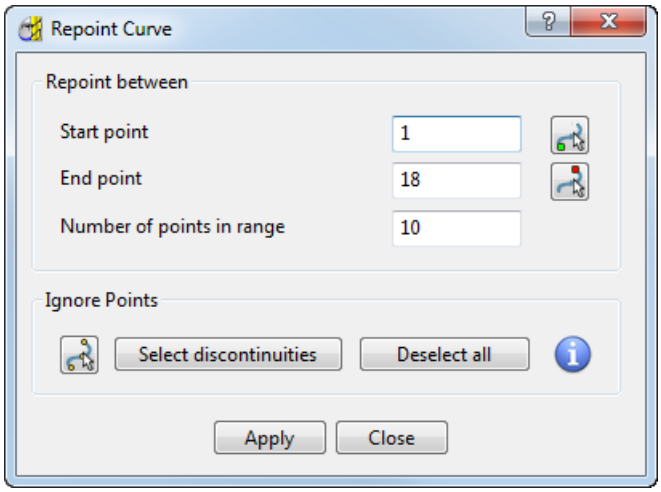


1 Double-click on the Bézier curve to open the **Curve Editor** mode toolbar.

2 Click **Number Points** to show the points in the Bézier curve.

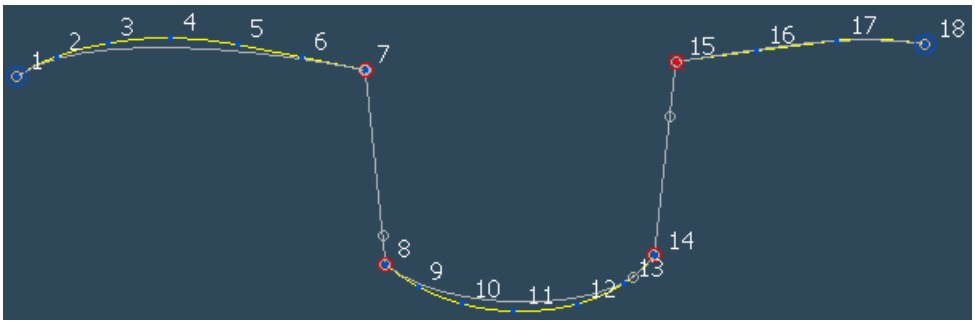


3 To reduce the number of points in the curve while still keeping points 7, 8, 14, or 15, click to display the **Repoint curve** dialog.



4 On the **Repoint Curve** dialog:

a Click **Select Discontinuities**.



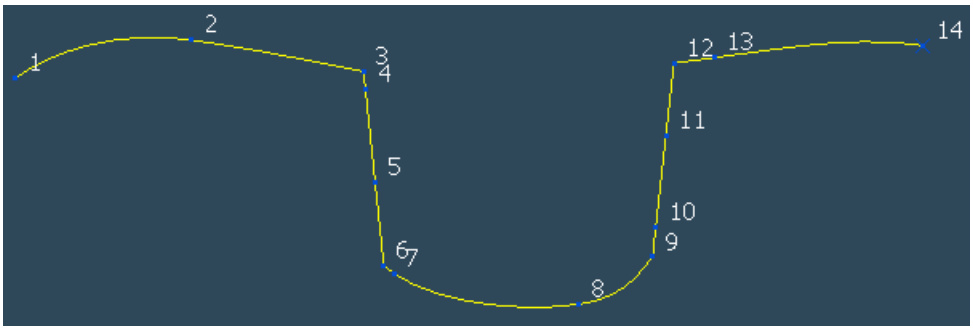
*To select additional points press the Shift key while selecting the next point, or click to use the **Point selection** dialog.*

b Enter a **Start point** of **1**.

c Enter an **End Point** of **18**.

d Enter a **Number of points in range** of **10**.

e Click **Accept**.

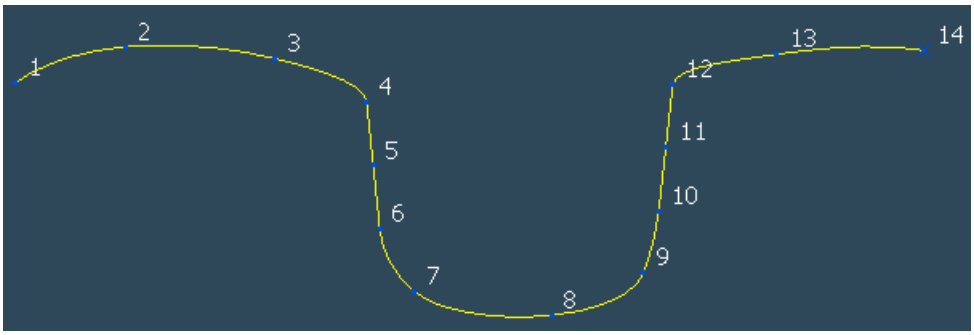


The original points 7, 8, 14, or 15 have become points 3, 6, 9, and 12 but still have the same location and tangency direction.

*Although you entered a **Number of points in range** of **10**, the actual number of points is 14 as there are 4 ignored points.*

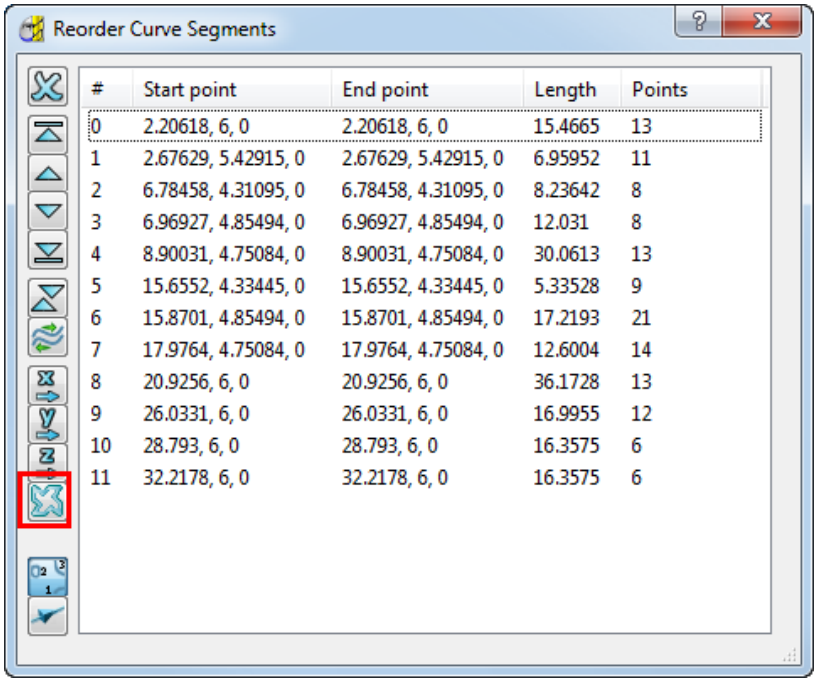
5 On the **Curve Editor** mode toolbar, click to accept the changes.

If you don't ignore the points at the tangent discontinuities, reducing the number of points in the curve produces:



Reorder enhancements

There is an additional option on the **Reordering curve segments** dialog (available from  on the **Curve Editor** mode toolbar).



 **Shortest path** sorts the curve segments to create the shortest path overall. This is particularly useful for text.

Converts this:

to this:

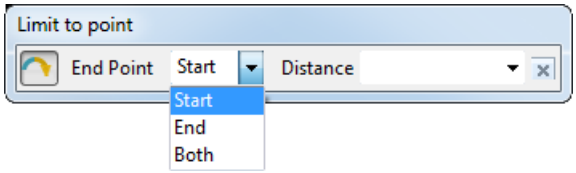
*You can reorder the segments by selecting them in the list area of the dialog and dragging them to the new location in the list. The selection supports all the standard selection options (for example **CTRL + select**, or **Shift + select** to multi-select segments).*

The drag and drop reordering is also available for toolpath segments.

Limit curve enhancements

You can now trim or extend both ends of a single curve segment. Previously, you could only trim/extend one end at a time.

This is available by clicking **Limit to point**  on the **Curve Editor** mode toolbar to display the **Limit to Point** toolbar.



You can now select an **End Point** of **Start**, **End** or **Both**.

The **Distance** field now lists the previous values.

New and improved tools

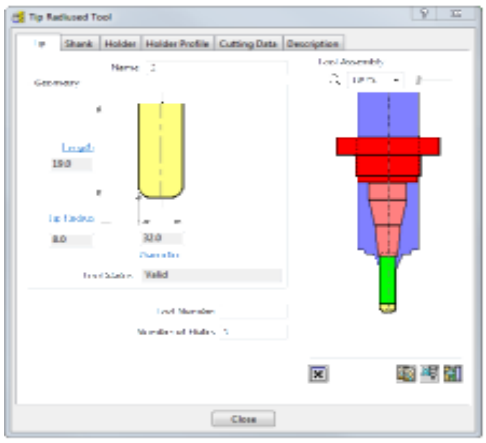
PowerMILL now features several new and enhanced tools, as well as a new **Tool Assembly Preview** dialog.

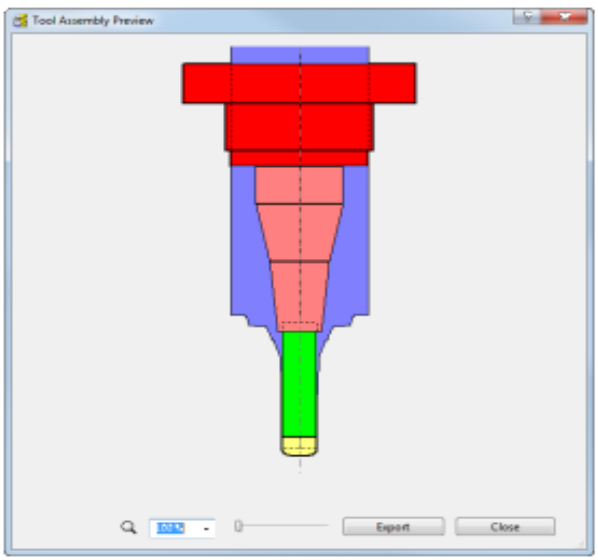
New barrel tool — Use to efficiently machine surfaces usually found on blades (see page 52).

New dovetail tool — Use to efficiently machine features such as undercuts and gears (see page 53).

Enhanced tipped disc tool — You can now enter different values for the lower and upper tip radii of a tipped disc tool (see page 54).

Tool Assembly Preview dialog — The new dialog enables you to check a tool assembly for collisions more easily than before (see page 55).



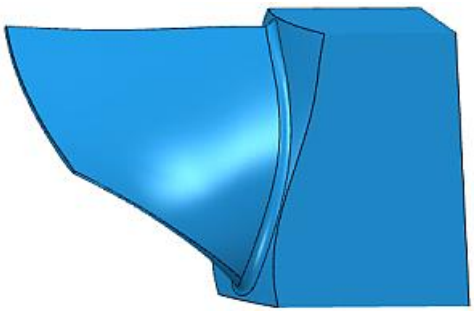


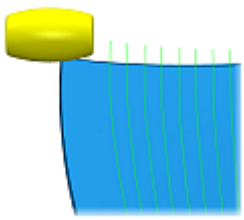
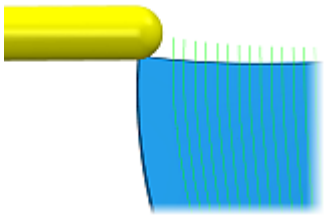
Barrel tool

PowerMILL now supports barrel tools.

A barrel tool efficiently machines the type of surfaces usually found on blades. The geometry of the tool enables you to achieve a small cusp height while using a relatively large stepover. To achieve the same cusp height with an alternative tool, you would need to reduce the stepover significantly.

The table shows the benefits of using a barrel tool to machine this blade.



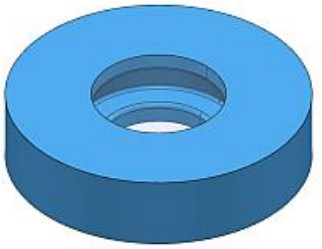
	Barrel tool	Ball nosed tool
Diameter	25.0	25.0
Cusp height	0.125	0.125
Stepover	7.74	5.0
Tool axis	Lead angle: 0.0° Lean angle: -85.0°	Lead angle: 0.0° Lean angle: -85.0°
Barrel radius	60.0	NA
		

Dovetail tool

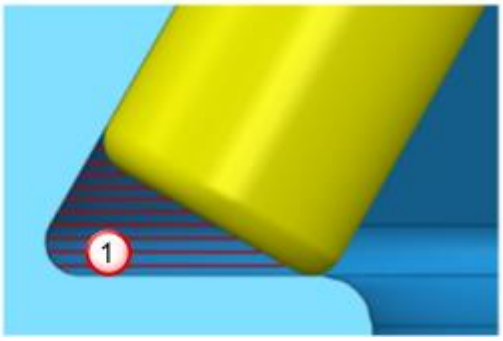
PowerMILL now supports dovetail tools.

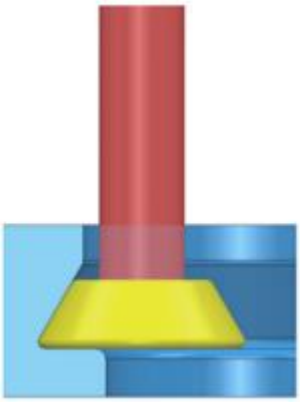
A dovetail tool efficiently machines features, such as undercuts and gears, that would otherwise require a series of more complex toolpaths involving multiple tool changes.

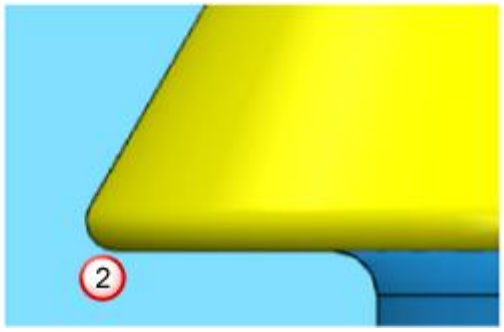
This example explains the advantages of using a dovetail tool to machine an undercut within an annulus.



In previous versions of PowerMILL, machining the conical surface of the undercut required a complex solution because a tip radiused tool, for instance, could not access the acute angle of the corner 1. After using a tipped radiused tool, you would need to use a small ball nosed tool or a tipped disc tool to sharpen the corner and remove all the material.



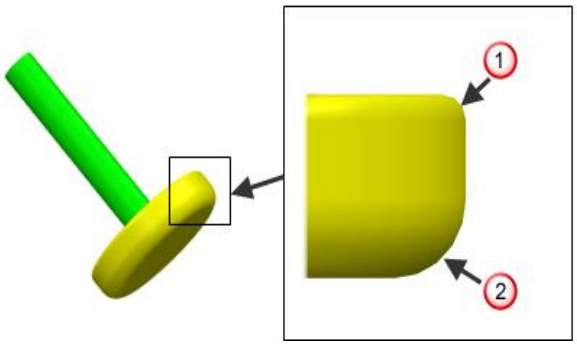




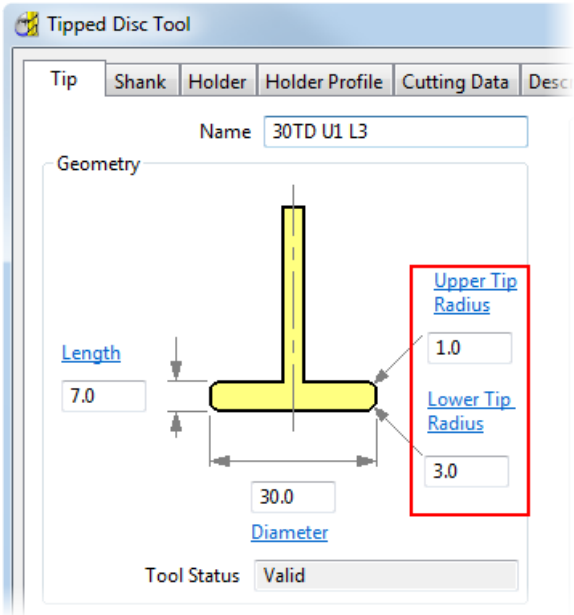
In PowerMILL 2014, the dovetail tool enables you to machine the same undercut more easily. Because the conical surface of the tool enables swarf machining, after performing area clearance, you can use the dovetail tool to form the undercut, fillet, and the horizontal surface of the base in one pass (2). Using the dovetail tool is also beneficial because it lets you use a larger tool and a better tool axis.

Enhanced tipped disc tool

PowerMILL now enables you to enter different values for the upper (1) and lower (2) tip radii of a tipped disc tool. In previous versions of PowerMILL, you could only specify a single radius value, which was applied to both the upper and lower tips of the tool.

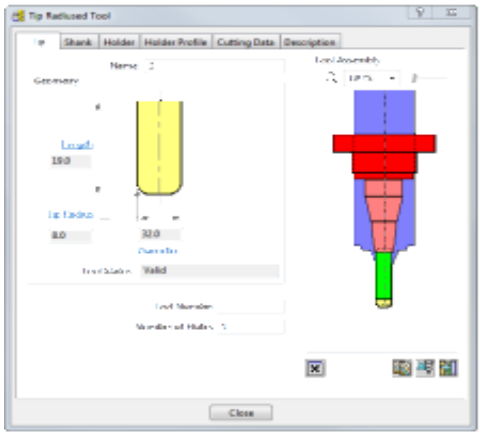


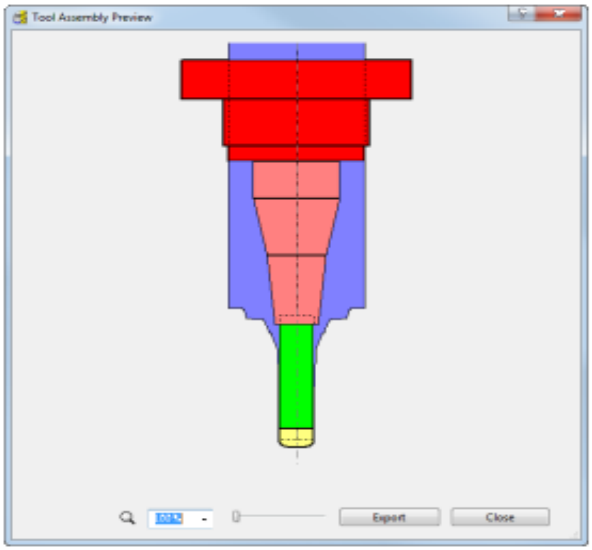
The **Tipped Disc Tool** dialog now has an **Upper tip radius** field and a **Lower tip radius** field.



Tool Assembly Preview dialog

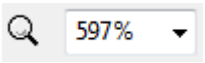
A new **Tool Assembly Preview** dialog displays a large image of the tool assembly, so it is easier to identify any components within the tool assembly that may collide.



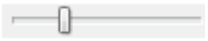


The following options are available:

To display the **Tool Assembly Preview** dialog, select  on the **Tool** dialog's **Tip**, **Shank**, **Holder**, **Holder Profile**, or **Description** tab.



— Select a value from the list or enter a value to zoom in or out of the tool assembly. You can then use the mouse to pan the image.



— Use the slide bar to zoom in or out of the tool assembly. You can then use the mouse to pan the image.

Export — Click to display the **Export Tool Assembly Preview** dialog. This enables you to save the current view of the tool assembly as a graphics file.

If you print the image of the tool assembly at a scale of 1:1, you can use it as a guide when grinding down your tool holder to avoid collisions.

For information on calculating the maximum allowable size of a tool assembly, see the Holder profile tab section.

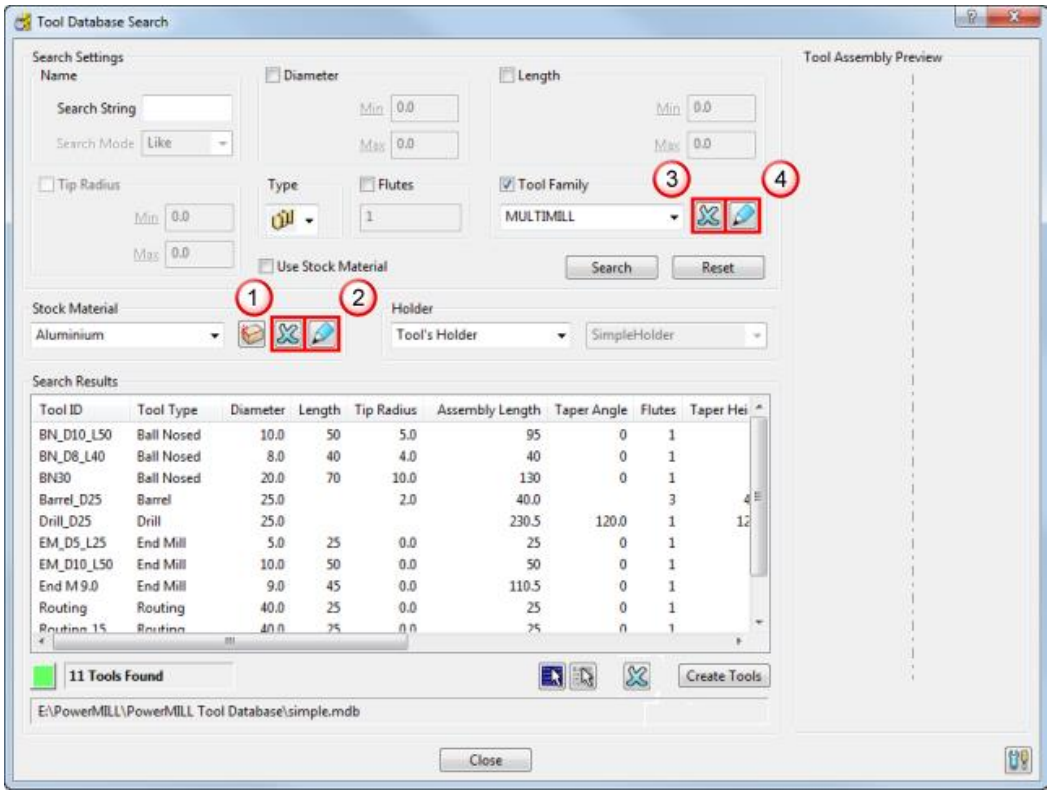
Tool database enhancements

There are three new buttons and two existing buttons with improved functions on the **Tool Database Search** dialog and the **Tool Database Holder Search** dialog. These buttons let you edit and delete entities in the tool database that are either associated with cutting data or tool assemblies.

The **Search For Holder** button is now also available on the **Tool** toolbar.

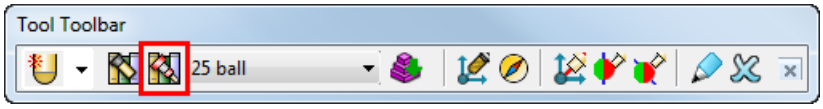
Tool Database Search dialog

1.  **Delete Stock Material**
2.  **Rename Stock Material**
3.  **Delete Tool Family**
4.  **Rename Tool Family**



Tool toolbar

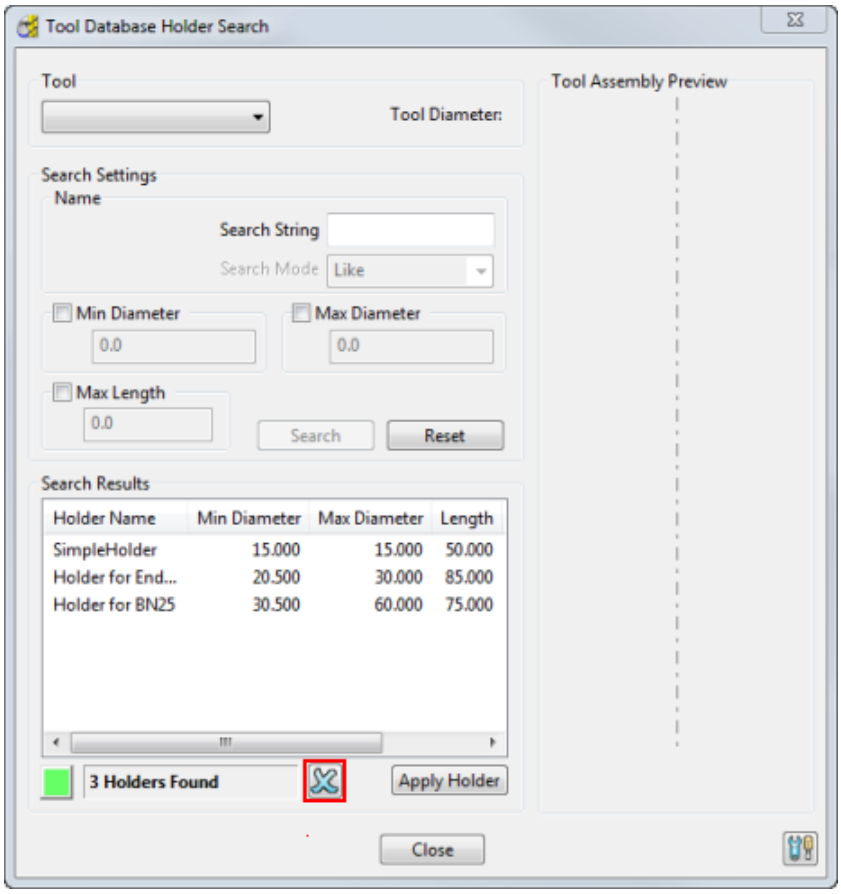
The **Search For Holder** button is now available on the **Tool** toolbar so you can display the **Tool Database Holder Search** dialog more easily. Previously, the button was only available on the **Holder** tab of the **Tool** dialog.



Tool Database Holder Search dialog

The improved function of the **Delete Select Holder** button now enables you to delete a holder even if it is used in a tool assembly.

*If you delete a holder used by a tool, the holder is removed from the tool's **Tool** dialog.*



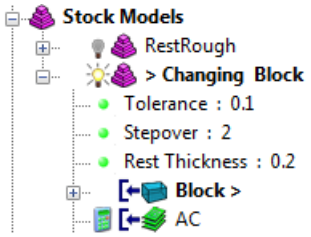
Stock model enhancements

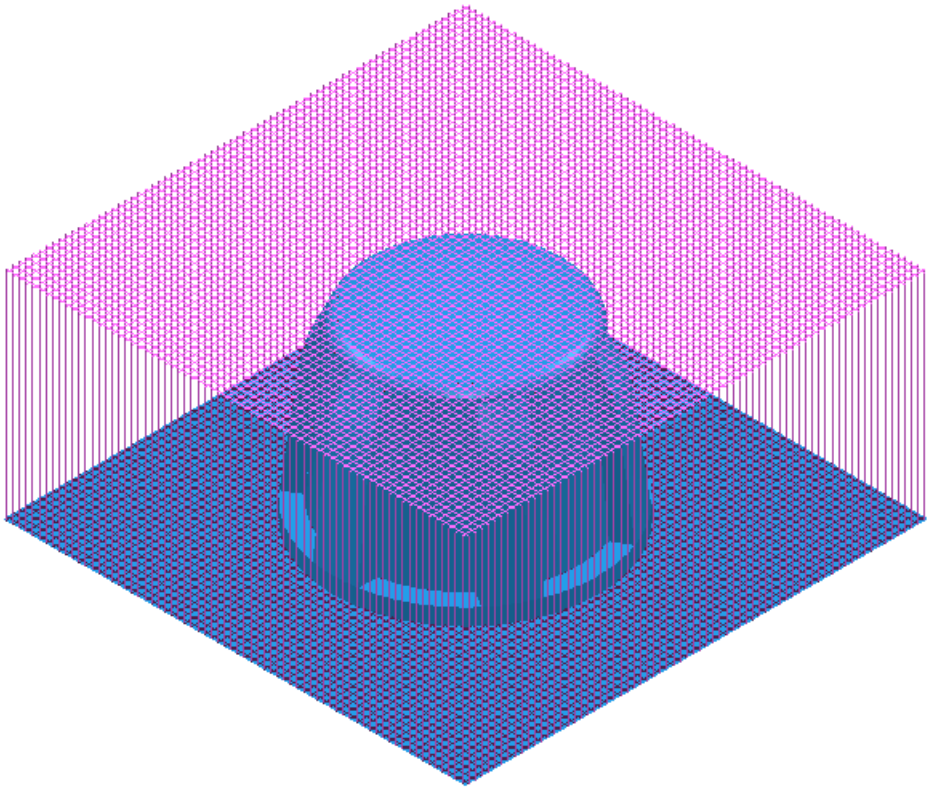
There are a couple of stock model enhancements:

- The stock model now accurately reflects machining with non-symmetrical tools, such as tipped disc tools with different values for the **Lower Tip Radius** and the **Upper Tip Radius**.
- You can now change the block in a stock model provided there are no calculated tool or toolpath states.
- You can now create folders in the **Stock Models** branch of the explorer (see page 114).

This example shows you how to change the block in a stock model. It uses the radknob.dmt model in the **Examples** folder.

1. Create a block and a toolpath.
2. Create a stock model and apply the toolpath to the stock model.
3. Right-click **Block** in the stock model and click **Calculate**.





4 On the **Main** toolbar click  to display the **Block** dialog.

a Enter a **Y Length** of **80**.

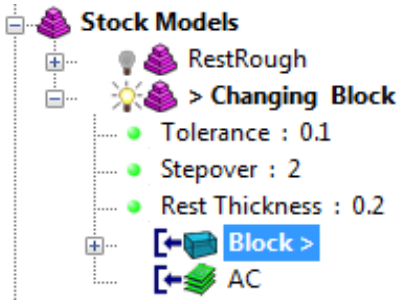
b Click **Accept**.

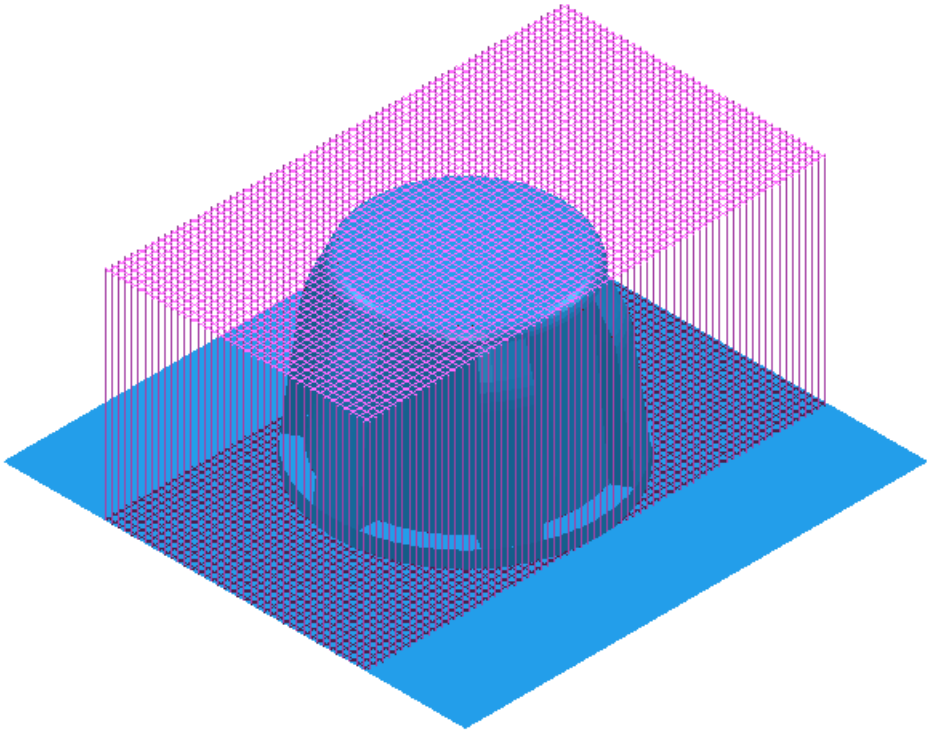
5 On the **Stock model** toolbar, click  to apply the new block to the stock model. This removes the block calculation and updates the stock model with the new size of the block.

*If you can't apply the new block then check there are no calculated tool or toolpath states in your stock model. If there are, select the **Remove Calculation** option from the individual tool or toolpath state menu and then apply the block.*

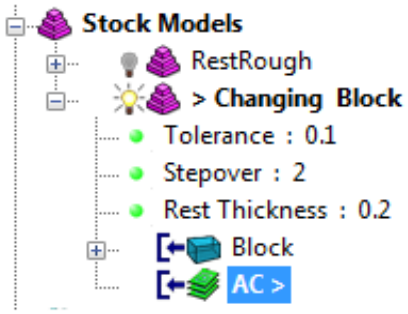
6 Click  to calculate the stock model.

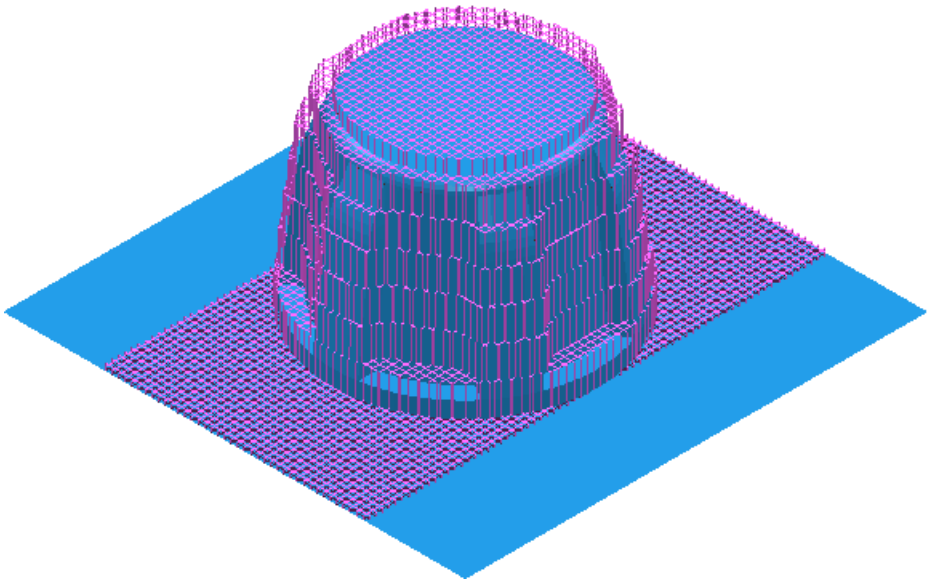
With the block state active:





With the area clearance toolpath state active:





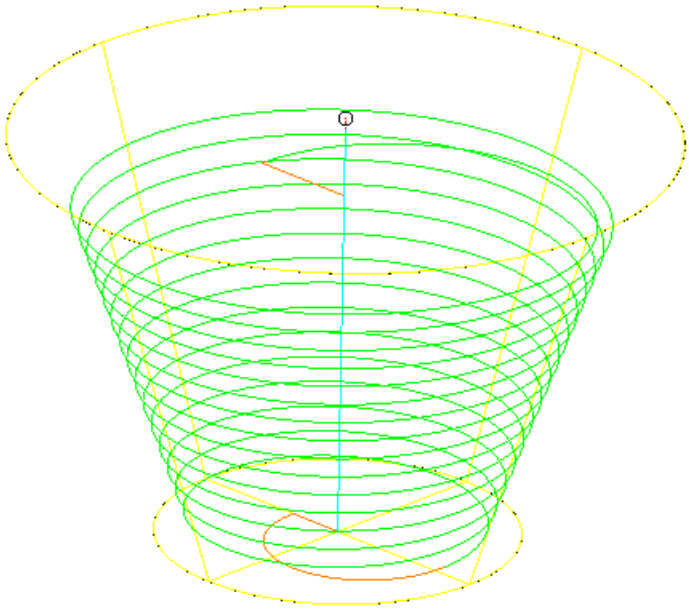
Toolpath generation

Drilling enhancements

There are several drilling enhancements:

- Each **Drilling Cycle type** now has its own strategy (see page 63).
- There is a new drilling strategy of **External threading** (see page 63).
- You can now create a thread milling toolpath in a tapered hole.

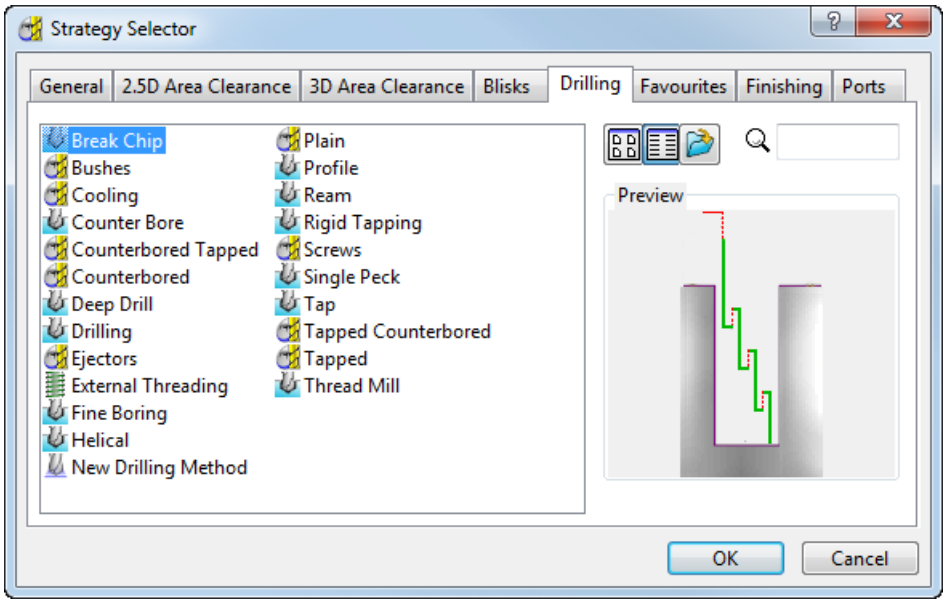
  Thread milling with a **Draft Angle** of 20° produces:



- You can now vary the feed rate and spindle speed when two holes intersect (see page 73).
- You can now select any component in a compound hole. In previous versions of PowerMILL you could select the first five components, or the last component. This is useful when automating drilling using methods.

Drilling strategies

Each drilling **Cycle type** now has its own strategy.



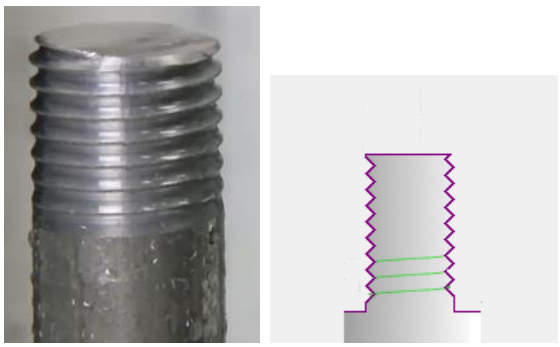
Selecting an option from the list displays the **Drilling** dialog with the option selected as the **Cycle type**.

*You can still change the drilling strategy by selecting a different **Cycle type** on the main drilling page.*

External threading

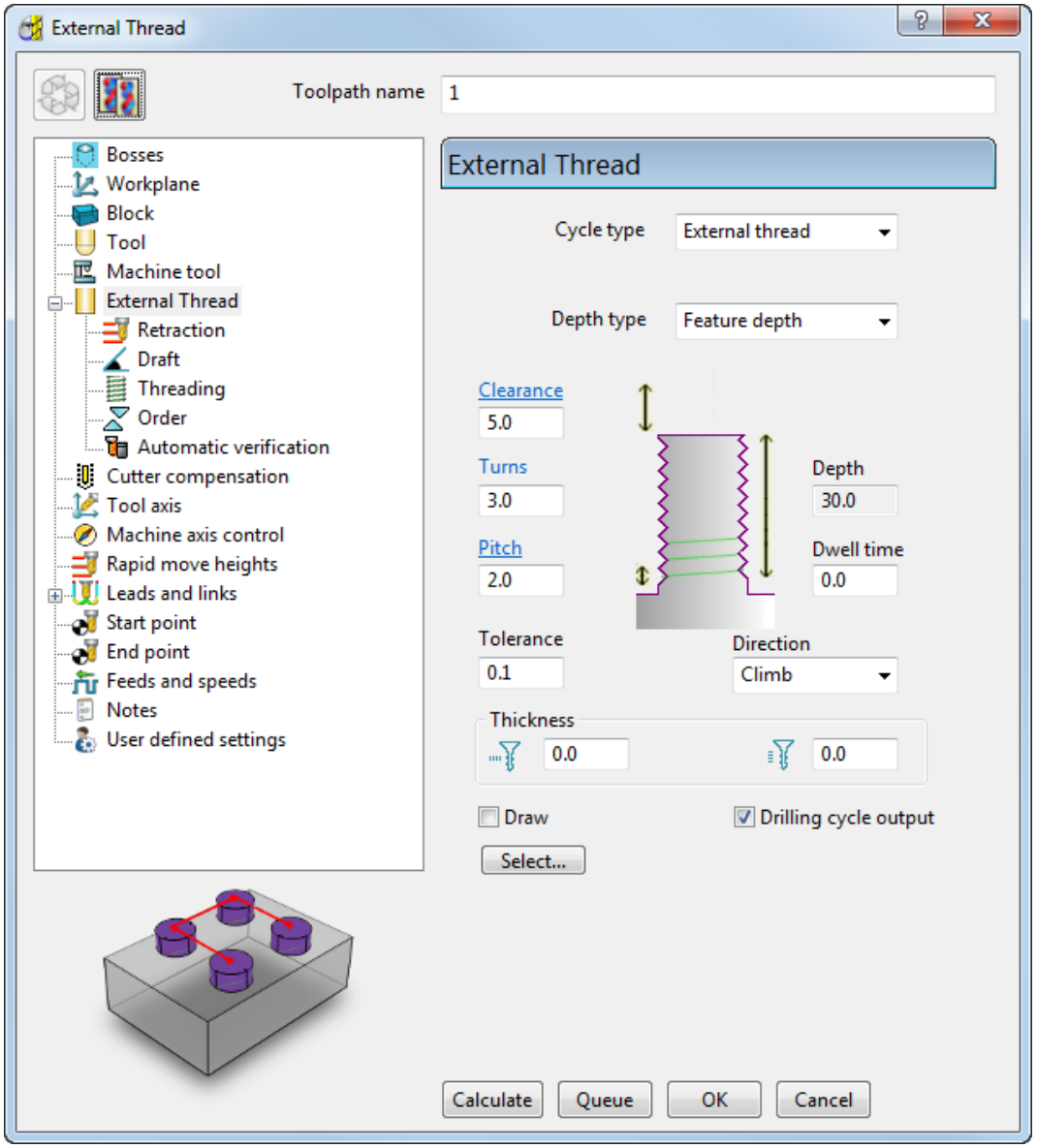
There is a new strategy on the drilling tab of the **Strategy selector** of **External thread**. This works in a very similar way to **Thread milling** on the **Drilling** dialog.

External thread creates an external thread on a boss.



To create an external thread you must use a thread mill tool.

There are several pages associated with the **External thread** strategy:



- **External Thread** — The main page used to define an external thread.
- **Retraction** — Settings to control how far the thread tool retracts between pecks (see page 66).
- **Draft** — Settings to create a tapered boss (see page 66).
- **Threading** — Settings for thread creation (see page 67).
- **Order** — Settings to control the order of machining (see page 70).
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

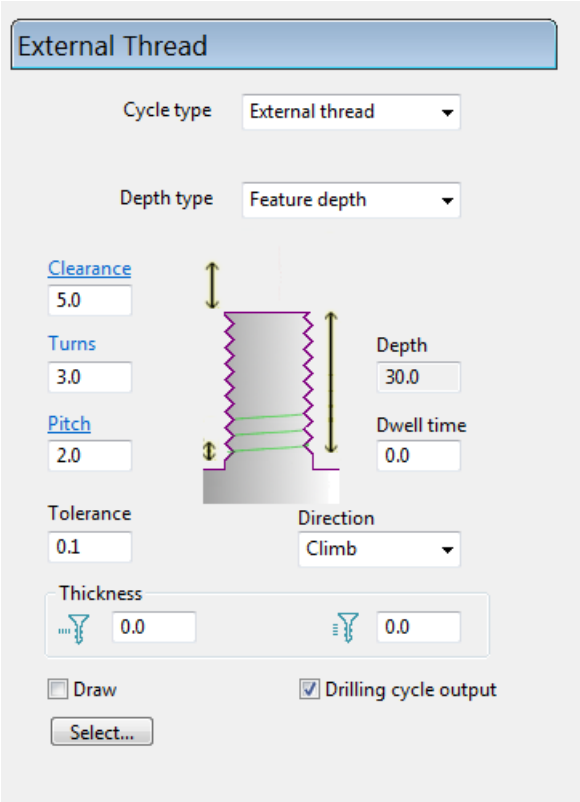
For more information on the **Strategy Selector** dialog, see Toolpath Strategies.

The common tabs are described in common toolpath creation controls.

For more information see Creating an external thread (see page 70).

External Thread

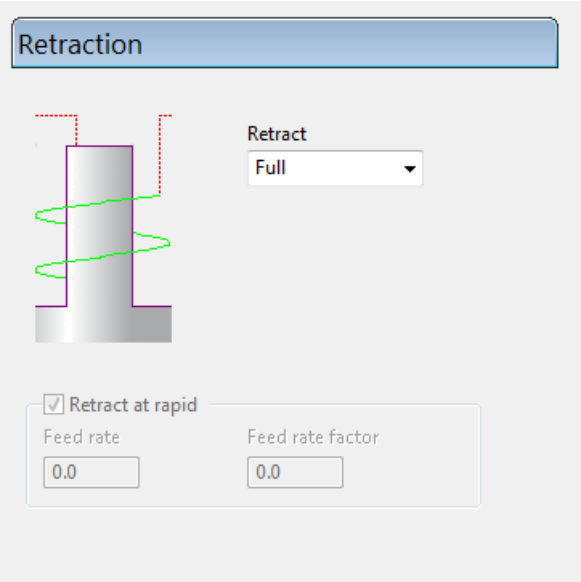
The **External Thread** page controls how to create an external thread.



This is the same as the standard **Drilling** page.

Retraction (external thread)

Retraction controls how far and at what speed the drill retracts up the hole between pecks.



Retract — Select how far the tool retracts between features.

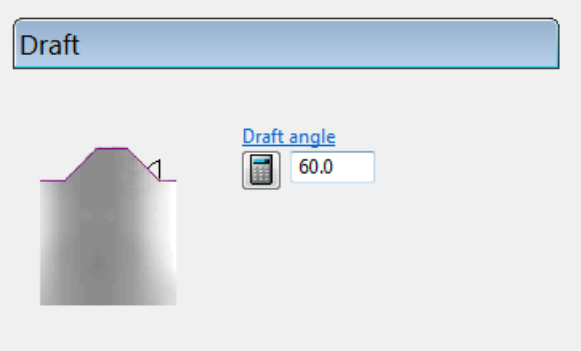
Full — The tool retracts up to the **Safe area** between features.

Partial — The tool retracts up to the **Clear plane** between features.

The remainder of the dialog is not available for this strategy.

Draft (external thread)

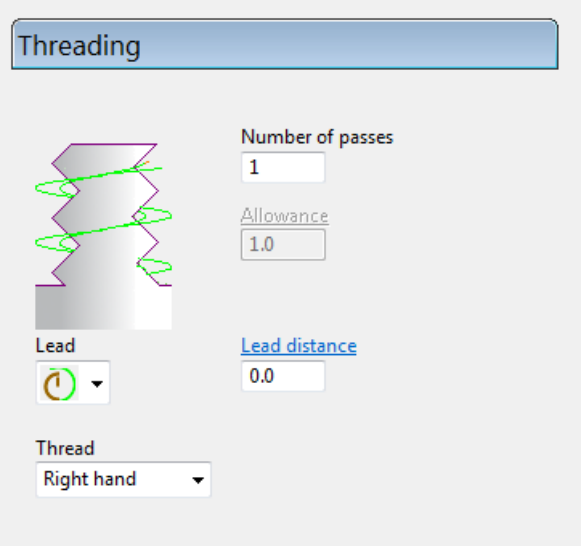
The **Draft** page enables you to add a draft angle to a boss.



Draft angle — Enter the **Draft angle**.

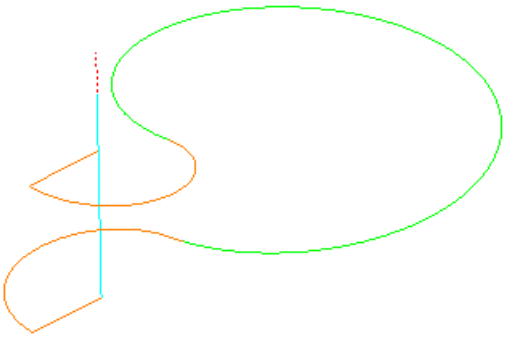
Threading (external thread)

The **Threading** page controls how to create an external thread.

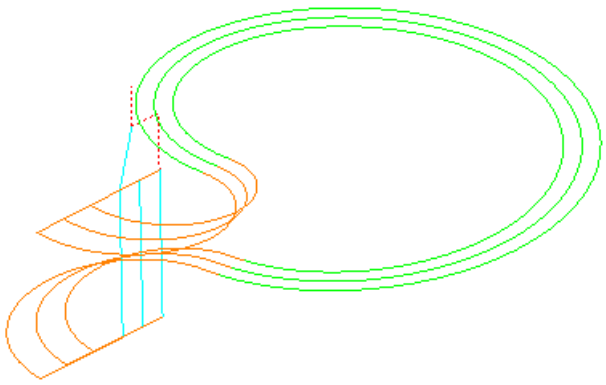


Number of passes — Enter the number of cuts.

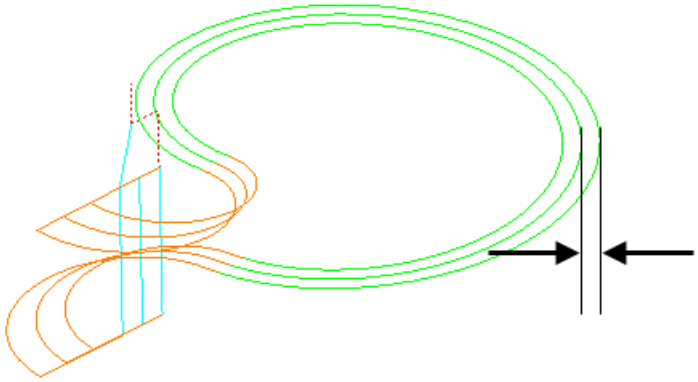
One pass:



Three passes:



Allowance — Enter the distance between successive passes.



Lead — Select the type of lead. The **Lead in** is a straight line in the plane normal to the hole axis followed by a helix, using the thread pitch. The **Lead out** is the reverse of this. All the options have the same straight line portion, but varying helical lead length.

 — 180° helical lead.

 — 90° helical lead.

 — 45° helical lead.

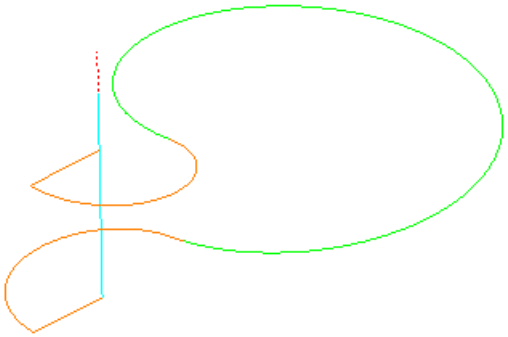
On the right, the table shows the effects of the differing leads on an external thread toolpath for an upwards, right hand thread.

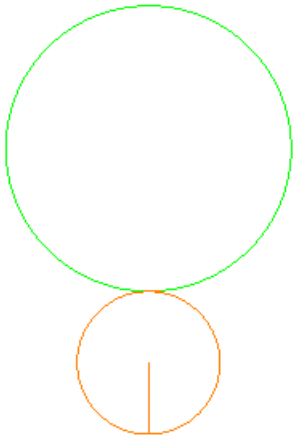
Thread — Select whether you want a **Right hand** or **Left hand** thread. The **Thread** combines with the **Direction** (on the main drilling page) to give a cut direction.

	Right hand thread	Left hand thread
Climb	Clockwise, downwards	Clockwise, upwards
Conventional	Anti-clockwise, upwards	Anti-clockwise, downwards

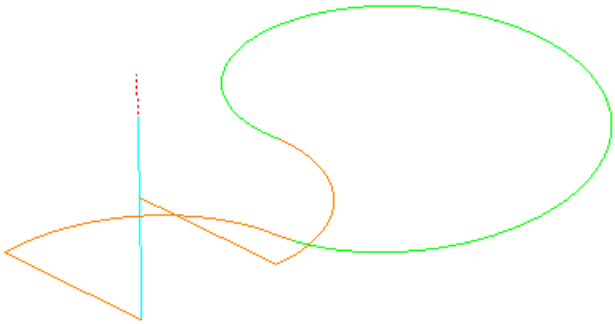
ISO View **View down Z**

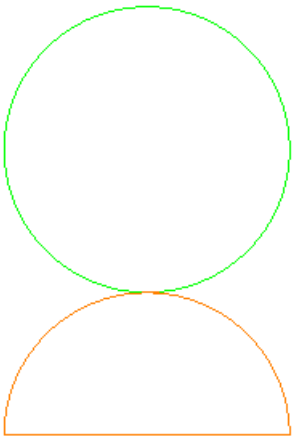
180°



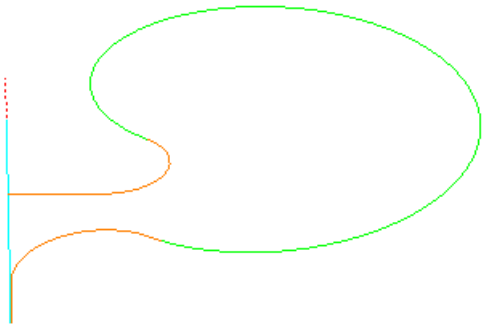


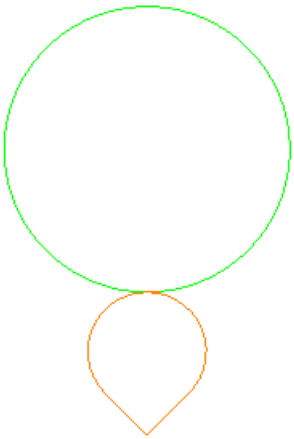
90°





45°





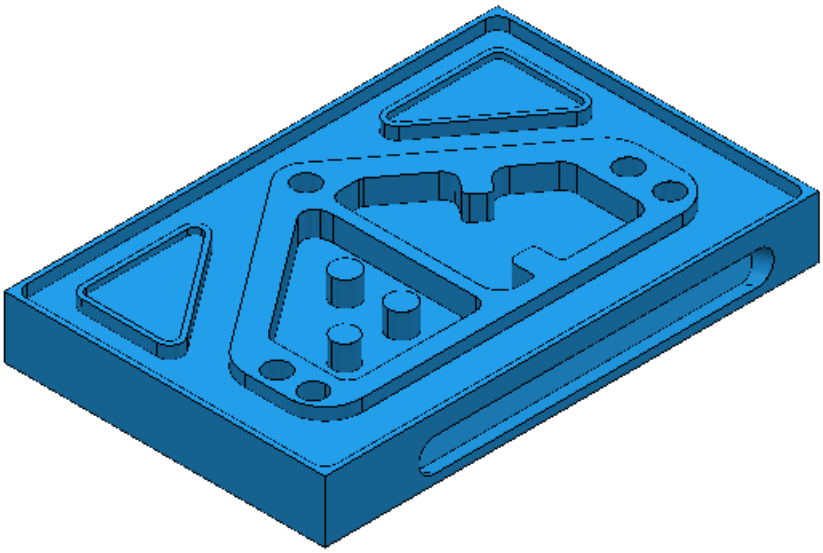
Order (drilling)

Sorting  — Select the drilling order of the selected holes.

Unless you select the **Explorer order**  , the start point is always the hole nearest to the tool start point.

Creating an external thread

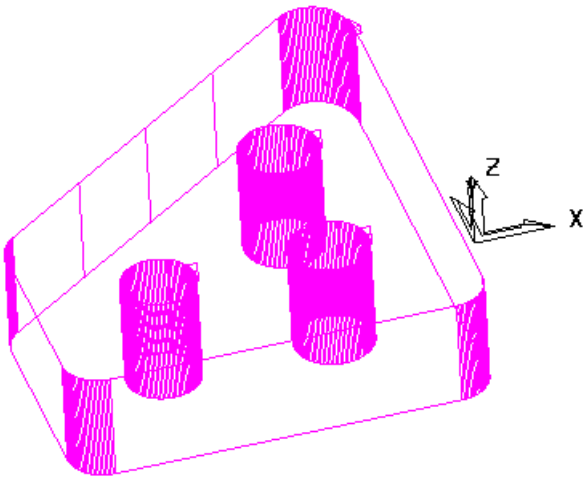
This example shows you how to create an external thread on a boss. It uses the 2DExample.dgk model in the Examples folder.



1 Create a feature set containing the three bosses.

 *You can create this as a pocket with three bosses.*

For more information on how to create the bosses see Smart Creation.

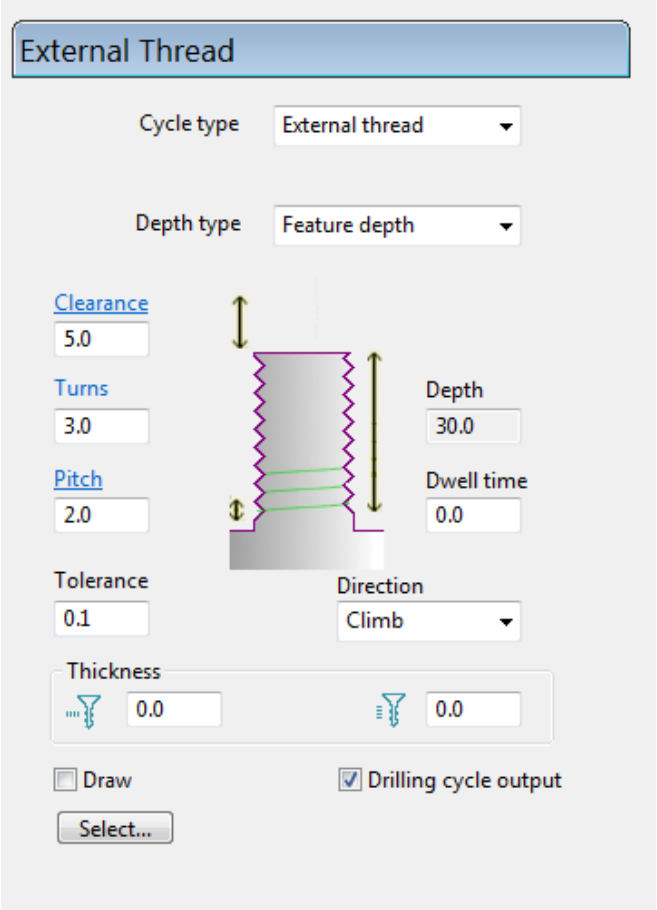


2 Calculate the block and create a thread mill tool.

3 On the **Main** toolbar, select the **Toolpath strategies** button.

4 Select the **Drilling** tab, followed by the **External Thread** option.

5 On the **External Thread** page:



a Select a **Depth Type** of **Feature depth**.

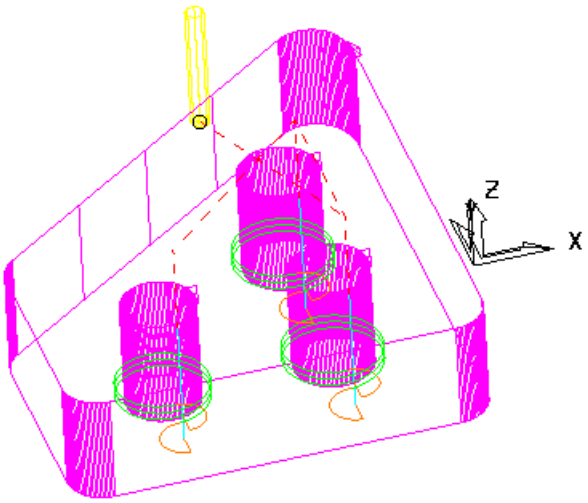
b Enter a **Clearance** of 5.

c Enter a **Turns** of 3.

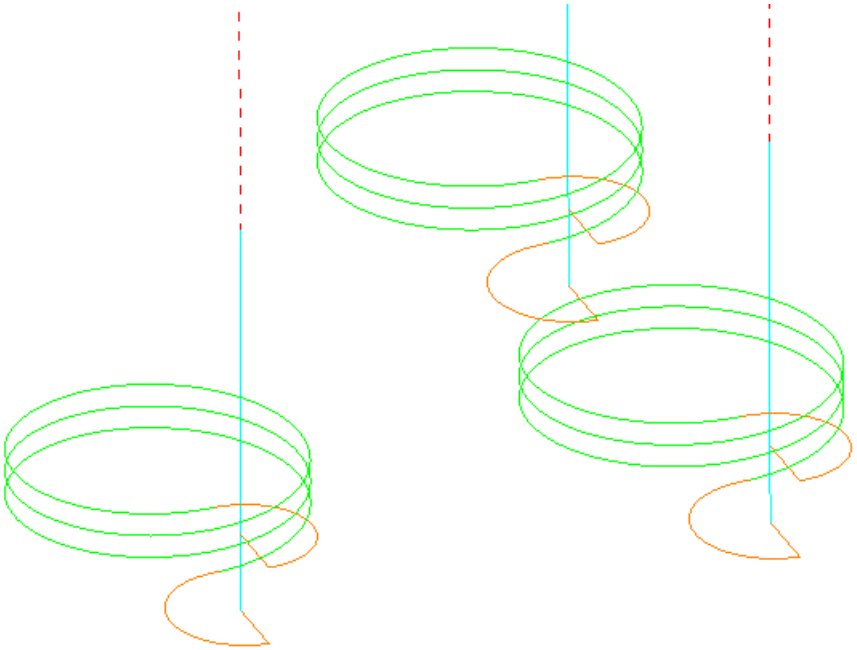
d Enter a **Pitch** of 2.

e Click **Calculate**.

*The **Cycle type** is automatically set to **External thread**.*



Looking in detail:



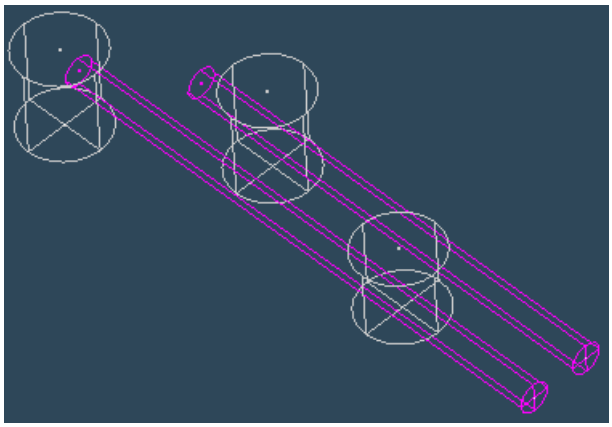
Varying feed rates at drill hole intersections

You now have the ability to vary the feed rate and spindle speed when two holes intersect. To do this you must:

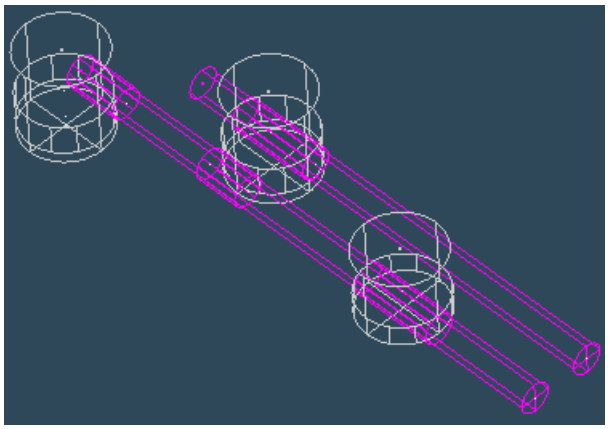
- Identify the hole intersections.
- Specify the feed rate and spindle speed reductions.

This example shows you how to find the hole intersections and reduce the feed rate and spindle speed at these intersections.

Starting with a model containing two feature sets, one containing three holes (black), the other containing two holes (magenta).

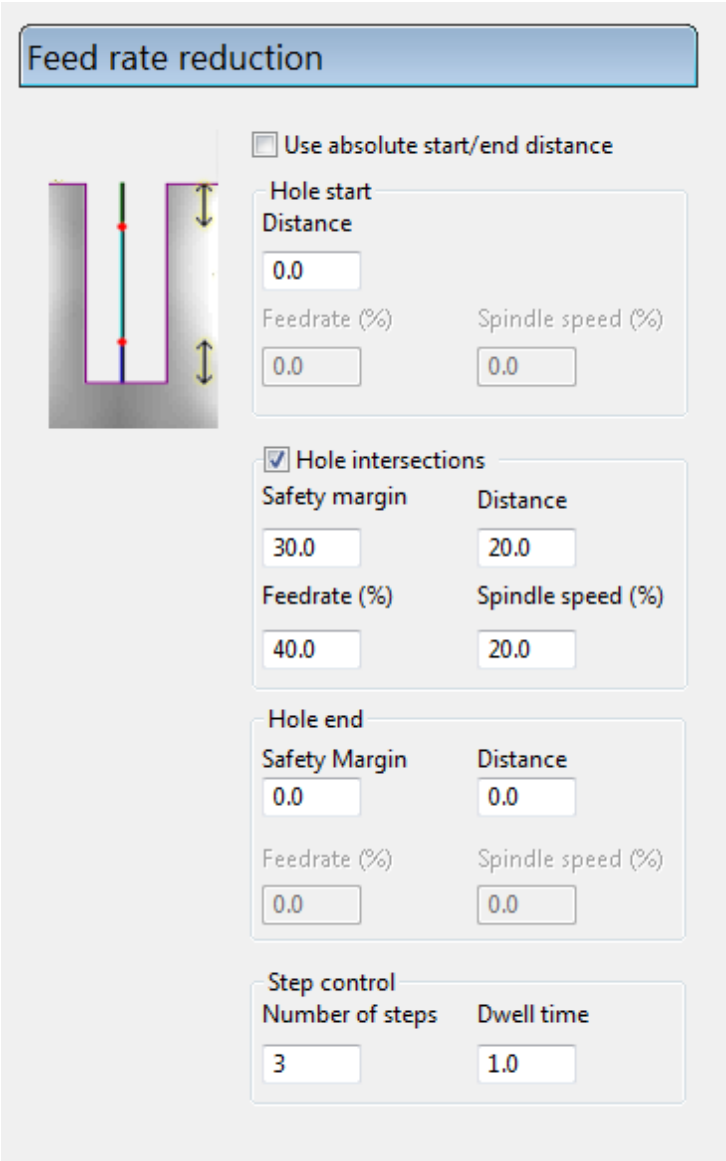


1. From the **Feature Sets** context menu, select **Find hole intersections**. This creates intersections wherever the two sets of holes intersect fully or partially.



2. Create a block and drill tool.
3. Display the **Single Peck Drilling** dialog.

4 Select the **Feed rate reduction** page:



a Select **Hole intersections**.

b Enter a **Safety margin** of 30 in the **Hole intersections** area.

c Enter a **Distance** of 20 in the **Hole intersections** area.

d Enter a **Feed rate** of 40 in the **Hole intersections** area.

e Enter a **Spindle speed** of 20 in the **Hole intersections** area.

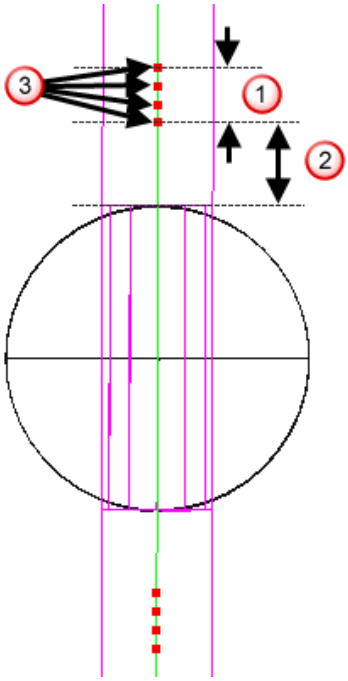
f Enter a **Number of steps** of 3.

g Enter a **Dwell time** of 1.

h Click **Calculate**.

This creates a drilling toolpath which slows down at the hole intersections.

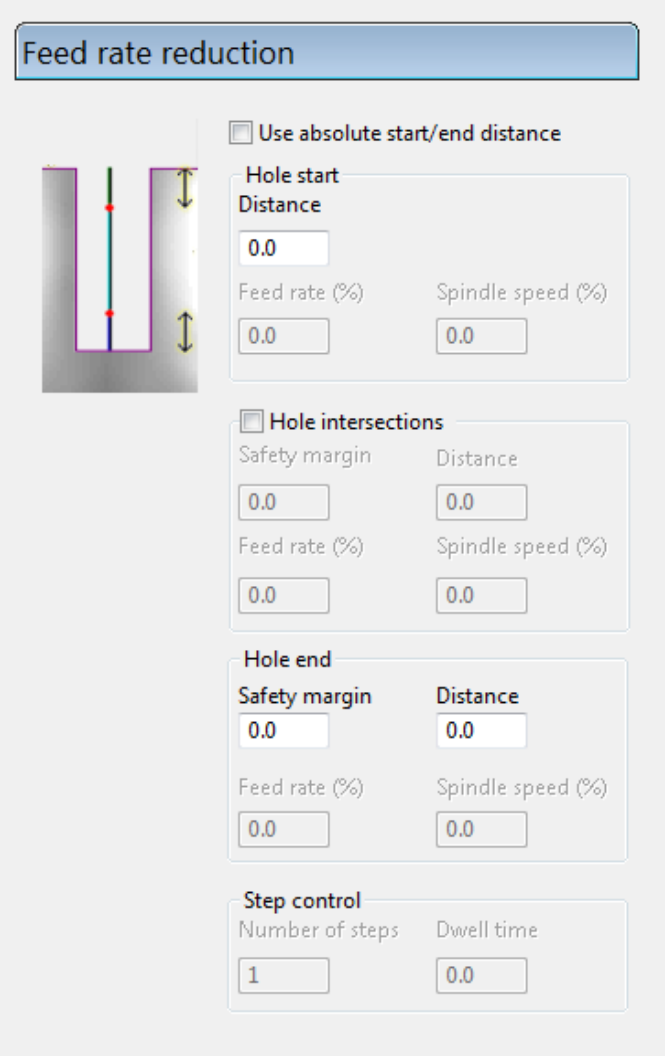
5 On the **Toolpath** toolbar click  to show each step.



1. Distance
2. Safety margin
3. Steps

Feed rate reduction

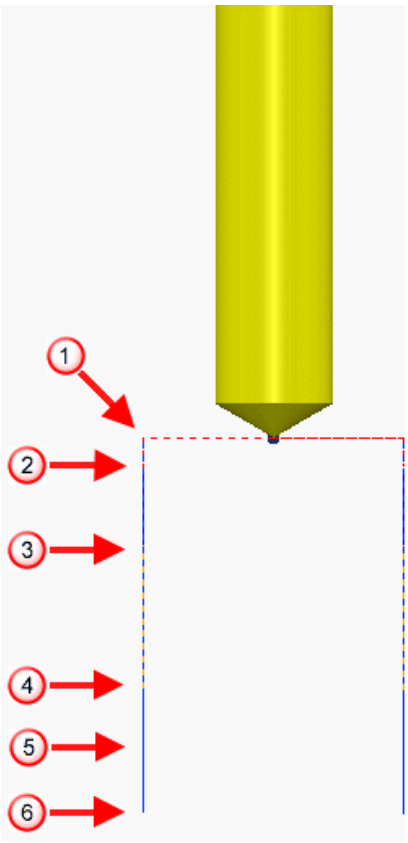
Feed rate reduction reduces the feed rate towards the top, or bottom or intersection of a hole. This enables the tool to slow down at the bottom of a hole, or move on slowly, and then speed up (or even "slow, quick, slow" for the situation where there are two holes, one above the other with air in between). If the machine tool has a drilling cycle which supports this option, set it up in the option file (otherwise, on the **NC Program** dialog select a **Driling Cycle Output** of **off**).



Use absolute start/end distances — Select to enter the **Hole start > Safety margin** and **Hole end > Safety margin** and **Hole end > Distance** as an absolute distance rather than a factor. When deselected a value of 0.2 reduces the feed rate and spindle speed for 20% of the hole length. When selected a value of 0.2 reduces the feed rate and spindle speed for 0.2 mm.

Factors must have a value between 0 and 1.

Hole start determines how to reduce the feed rate and spindle speed at the top of a hole.



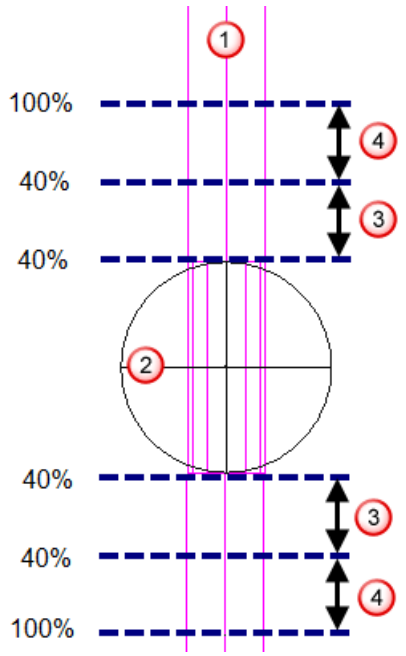
1. Safe Z.
2. Top of the hole.
3. Feed rate reaches 100%.
2 - 3 **Hole start > Distance**.
4. Feed rate starts to decrease again.
4 - 5 **Hole end > Distance**.
5. Feed rate reaches reduced value.
5 - 6 **Hole end > Safety margin**.
6. Bottom of the hole.

Distance — Enter the distance/factor below the top of the hole over which the start feed rate and start spindle speed are reduced. At the top of the hole the feed rate and spindle speed are reduced by the percentage specified in the **Feed rate (%)** and **Spindle speed (%)**. At a distance of **Distance** below the top of the hole the feed rate and spindle speed are increased to the value specified in the **Feeds and Speeds** dialog (available from  on the **Main** toolbar). Entering a value here enables the **Feed rate (%)** and **Spindle speed (%)** fields.

Feed rate(%) — Enter the percentage reduction of the feed rate required at the top of a hole. A value of 0.25 reduces the feed rate to 25% of the feed rate specified on the **Feeds and Speeds** dialog.

Spindle speed (%) — Enter the percentage reduction of the spindle speed required at the top of a hole. A value of 0.25 reduces the spindle speed to 25% of the spindle speed specified on the **Feeds and Speeds** dialog.

Hole intersection — Select to enable feed rate and spindle speed reductions at hole intersections. This slows down the feed rate and spindle speed just above a hole intersection and then speeds it up again after the hole intersection.



1. Hole you are drilling.
2. Intersecting hole.
3. **Safety margin**.
4. **Distance**.

The percentage figures show the feed rate if you have a **Feed rate (%)** of **40**.

*To reduce feed rates and spindle speeds at hole intersections you must locate these intersections by selecting **Find hole intersections** on the **Feature Sets** context menu.*

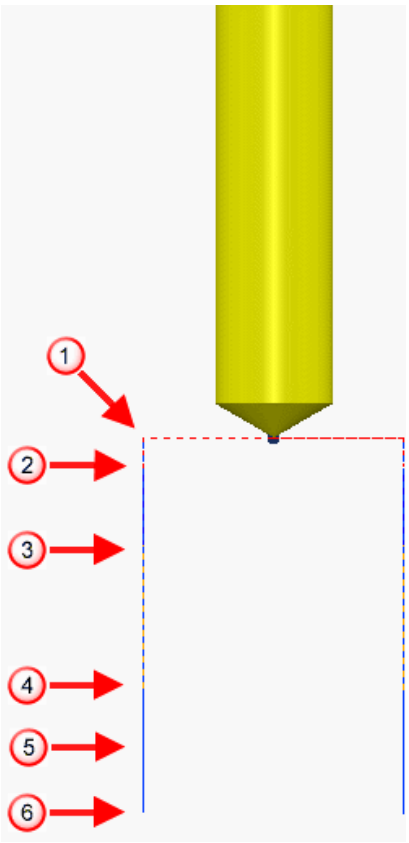
Safety Margin — Enter the distance above the hole intersection at which the start feed rate and spindle speed are reduced to the percentage specified in **Feed rate (%)** and **Spindle speed (%)**. Entering a value enables the **Feed rate (%)** and **Spindle speed (%)** fields. This also specifies the distances below the hole intersection where the feed rate and spindle speed are increased from the reduced rate to full speed.

Distance — Enter the distance over which the feed rate and spindle speed are reduced.

Feed rate(%) — Enter the percentage reduction of the feed rate required at the hole intersections.

Spindle speed (%) — Enter the percentage reduction of the spindle speed required at the hole intersections.

Hole end determines how to reduce the feed rate and spindle speed at the bottom of a hole.



1. Safe Z.
2. Top of the hole.
3. Feed rate reaches 100%.

2 - 3 **Hole start > Distance**.

4. Feed rate starts to decrease again.

4 - 5 **Hole end > Distance**.

5. Feed rate reaches reduced value.

5 - 6 **Hole end > Safety margin**.

6. Bottom of the hole.

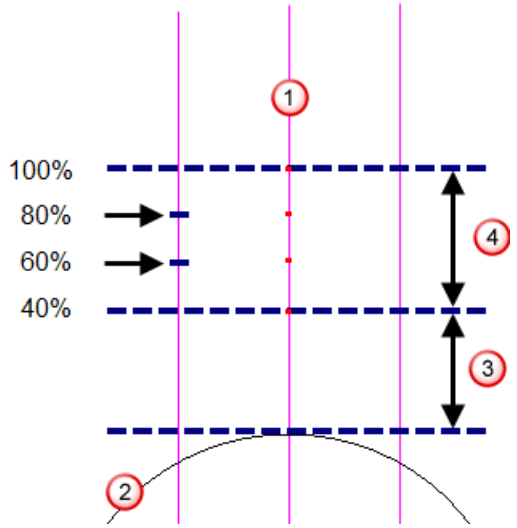
Safety Margin — Enter the distance/factor above the bottom of the hole at which the start feed rate and spindle speed are reduced to the percentage specified in **Feed rate (%)** and **Spindle speed (%)**. Entering a value enables the **Feed rate (%)** and **Spindle speed (%)** fields.

Distance — Enter the distance/factor over which the feed rate and spindle speed are reduced.

Feed rate(%) — Enter the percentage reduction of the feed rate required at the bottom of the hole.

Spindle speed (%) — Enter the percentage reduction of the spindle speed required at the bottom of the hole.

Step control determines how to reduce the feed rate over the **Distance**. The feed rate decreases in fixed increments and not linearly.



1. Hole you are drilling.
2. Intersecting hole.
3. **Safety margin**.
4. **Distance**.

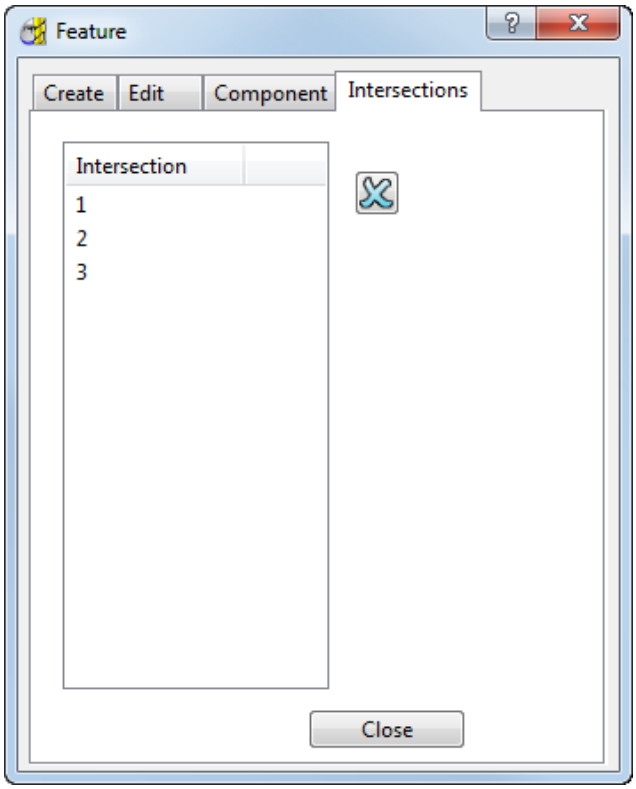
The percentage figures show the feed rate if you have a **Feed rate (%)** of **40** and a **Number of steps** of **3**.

Number of steps — Enter the number of steps required to reduce the feed rate.

Dwell time — Enter the dwell time at each step.

Feature > Intersection

The **Intersections** tab on the **Feature** dialog enables you to delete hole intersections.



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

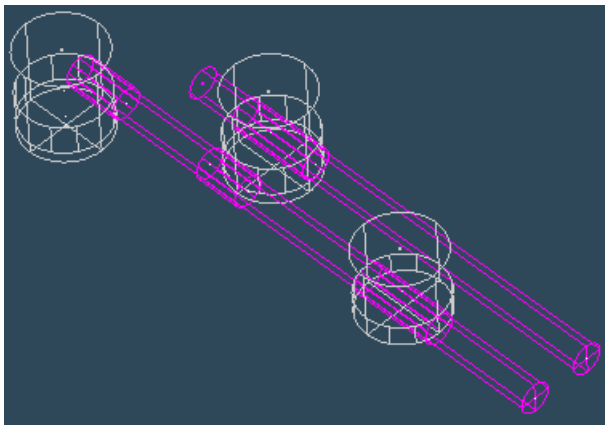
The left side of the dialog displays a list of the intersections of the selected hole.

Delete — Click to delete the intersections selected in the left side of the dialog (see page 82).

For more information see Varying the feed rate at hole intersections or Deleting a hole intersection (see page 82).

Deleting a hole intersection

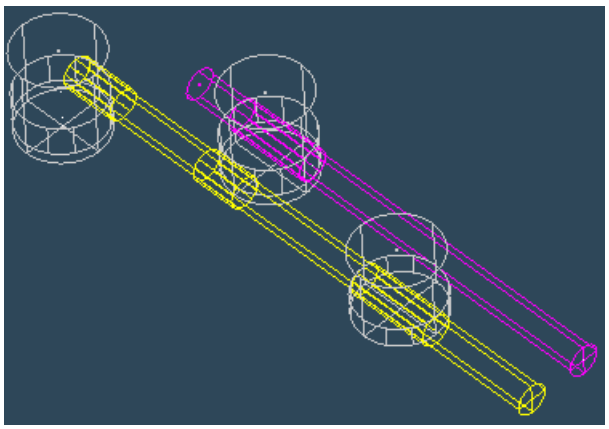
This example shows you how to delete an intersection in a hole. Starting with this model:



The holes are in two feature sets: the three white holes in one feature set and the two magenta holes in another (currently active) feature set.

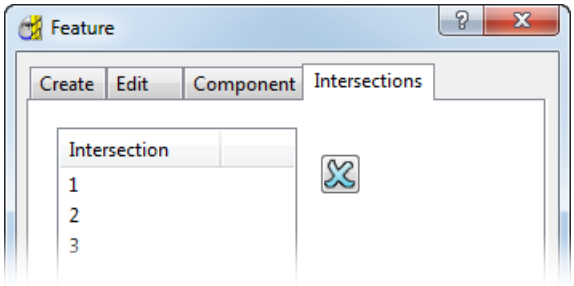
*To create the intersections from the **Feature Sets** context menu, select **Find hole intersections**.*

1 Select the hole with three intersections.



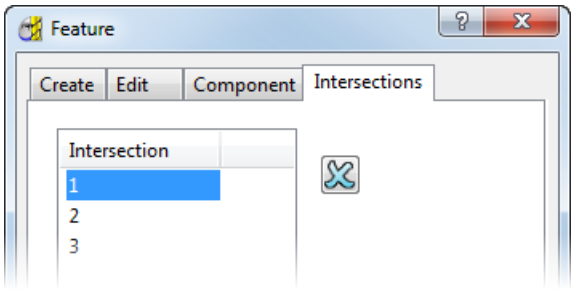
2 From the individual feature set menu select **Settings** to display the **Feature** dialog.

3 Click the **Intersections** tab.

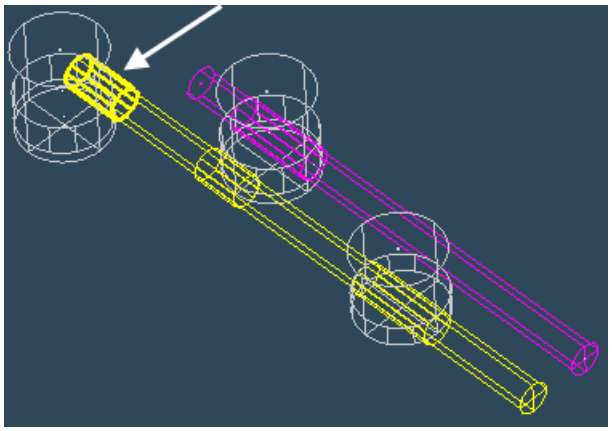


This shows there are three intersections with the selected hole.

4 Select intersection **1** in the left side of the dialog.

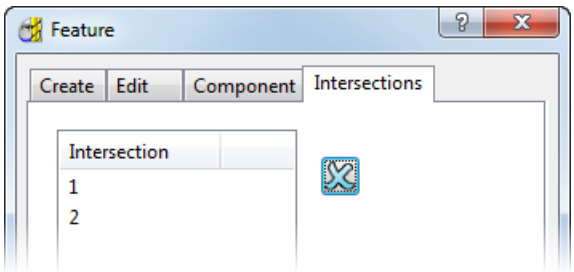


This displays the selected intersection with a thicker outline:

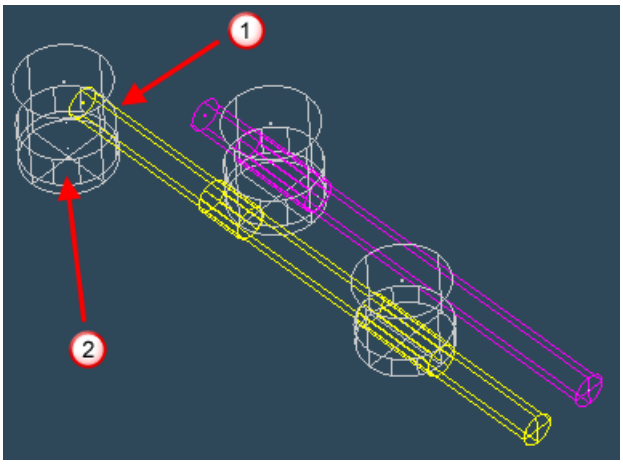


5 Click **Delete** to delete the intersection.

This deletes intersection **1** and renames the remaining intersections from **2** and **3** to **1** and **2**.



This updates the feature set to remove one intersection:



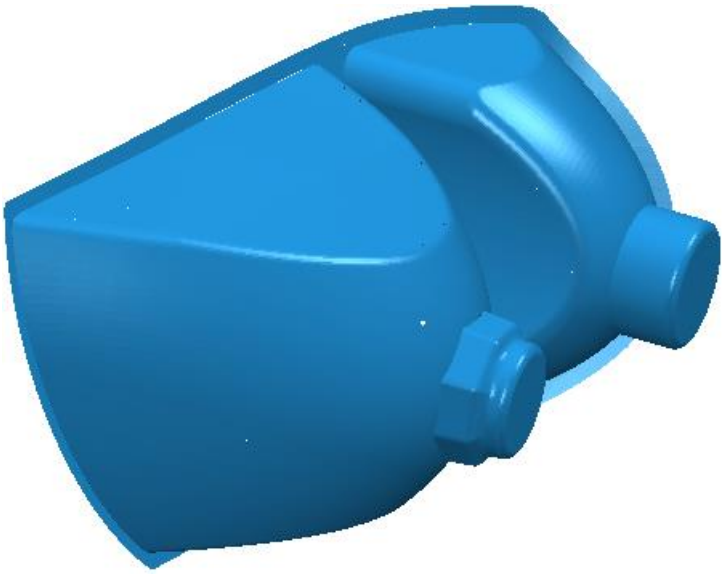
1 The deleted intersection.

2 The intersection with the other hole remains.

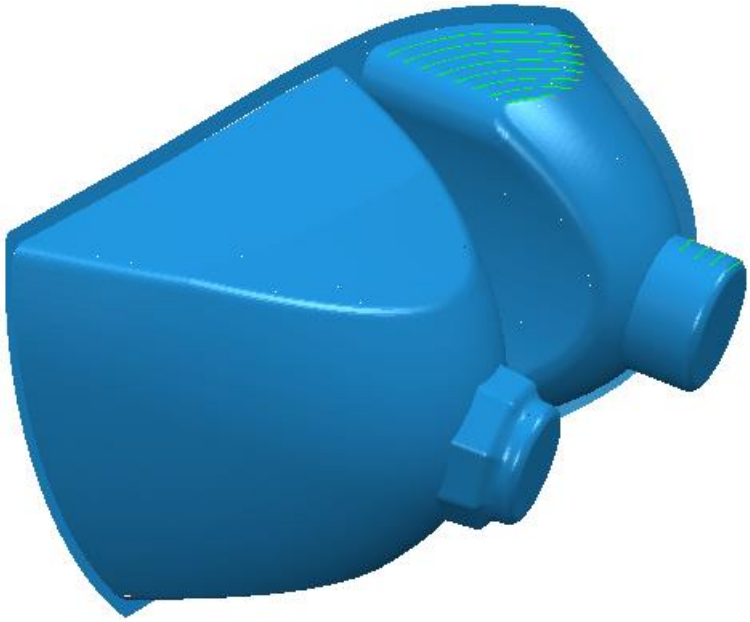
Flat machining

Flat machining on triangle models is now faster and produces better results, which are as good as flat machining on surface models.

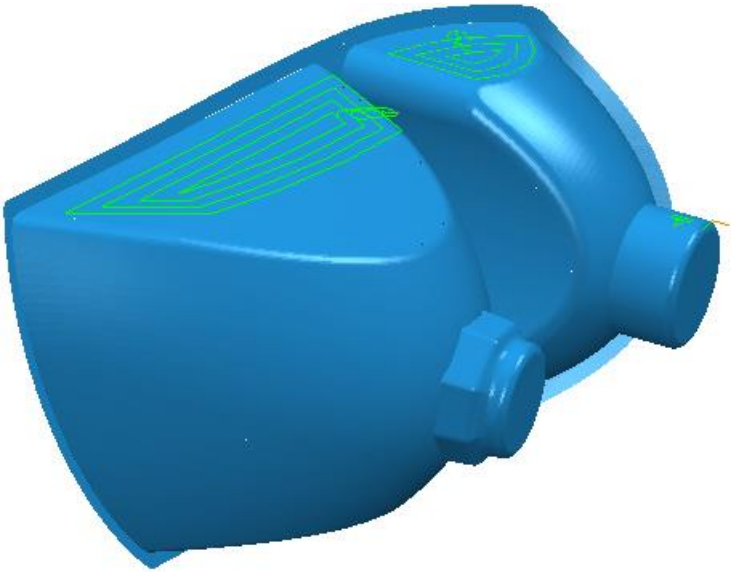
Looking at this headlamp example:



In PowerMILL 2013 an offset flat machining toolpath produces:

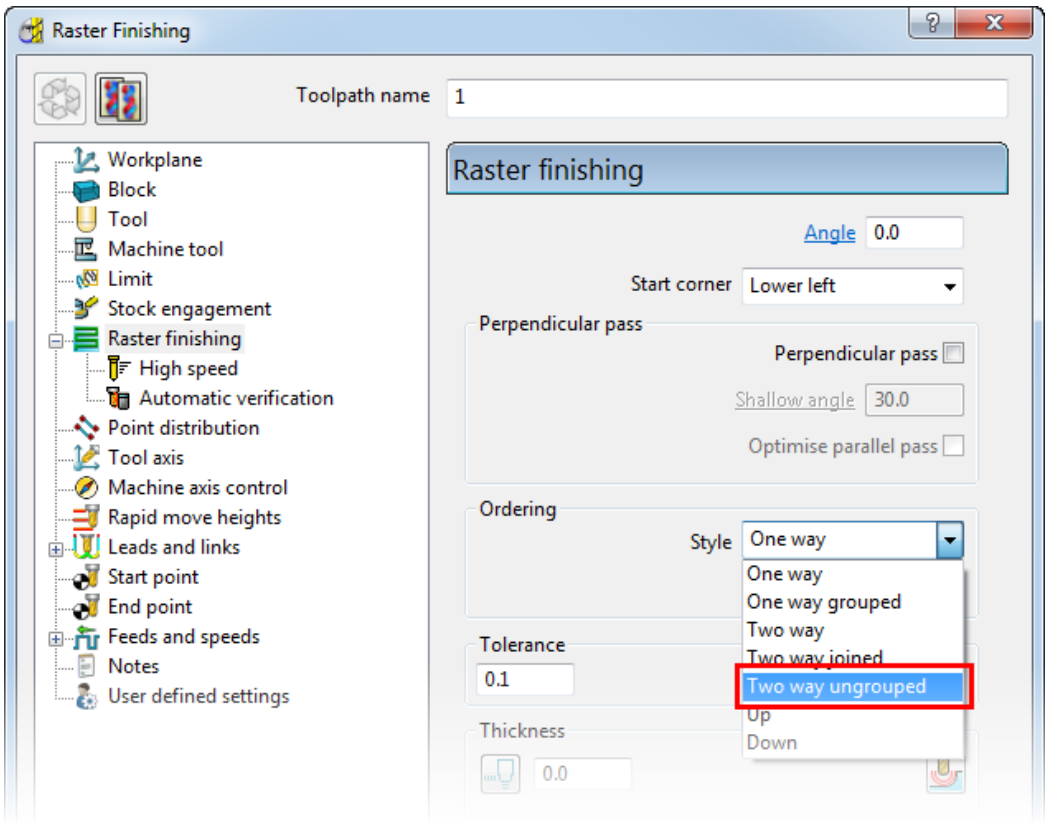


In PowerMILL 2014 an offset flat machining toolpath produces:



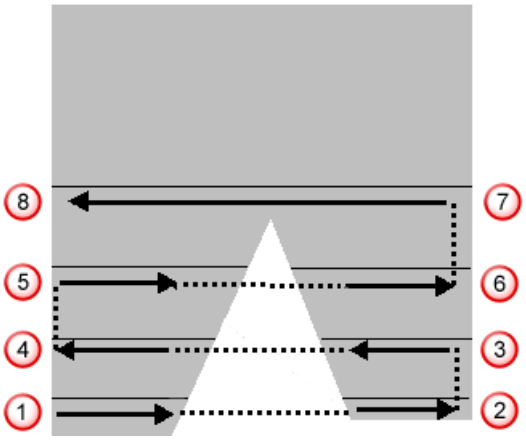
Raster ordering

There is a new ordering option when creating a raster toolpath of **Two way ungrouped**. This enables you to create a raster toolpath without any grouping of toolpath segments. This option is available for two-way toolpaths.



In this example the numbered lines represent toolpath segments.

Two way ungrouped — The tool can cut in two directions:

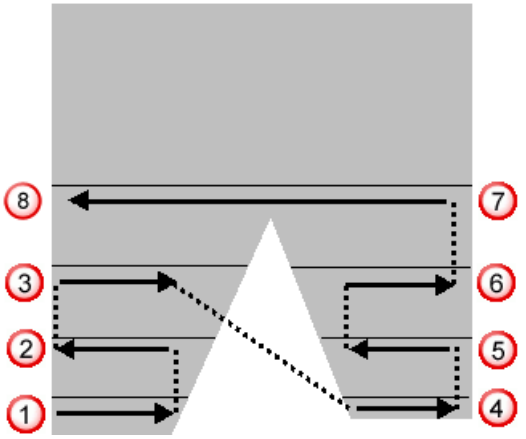


Two way and **Two way joined** — The tool can cut in two directions and PowerMILL chooses the shortest distance between each toolpath segment.

Two way — The toolpath segments are joined with lead and link moves.

Two way joined — The toolpath segments are joined with cutting moves.

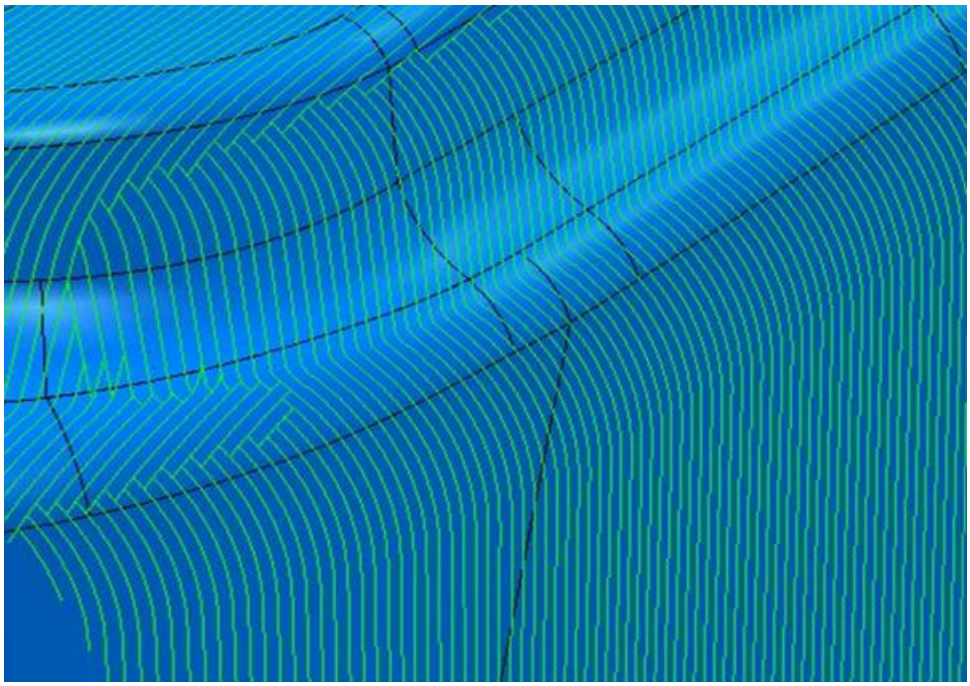
The order is the same for both options; the difference is whether the joining moves are cutting moves or not:



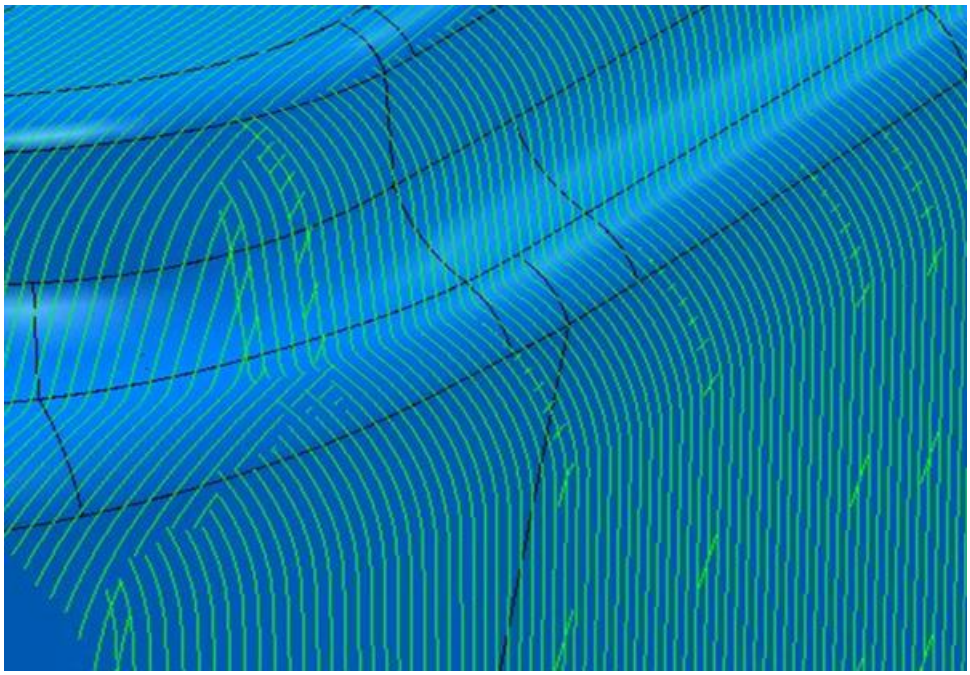
Optimised raster improvements

There are improvements to the optimised raster algorithm. The toolpaths:

- Are less fragmented.
- Have no overlapping segments.



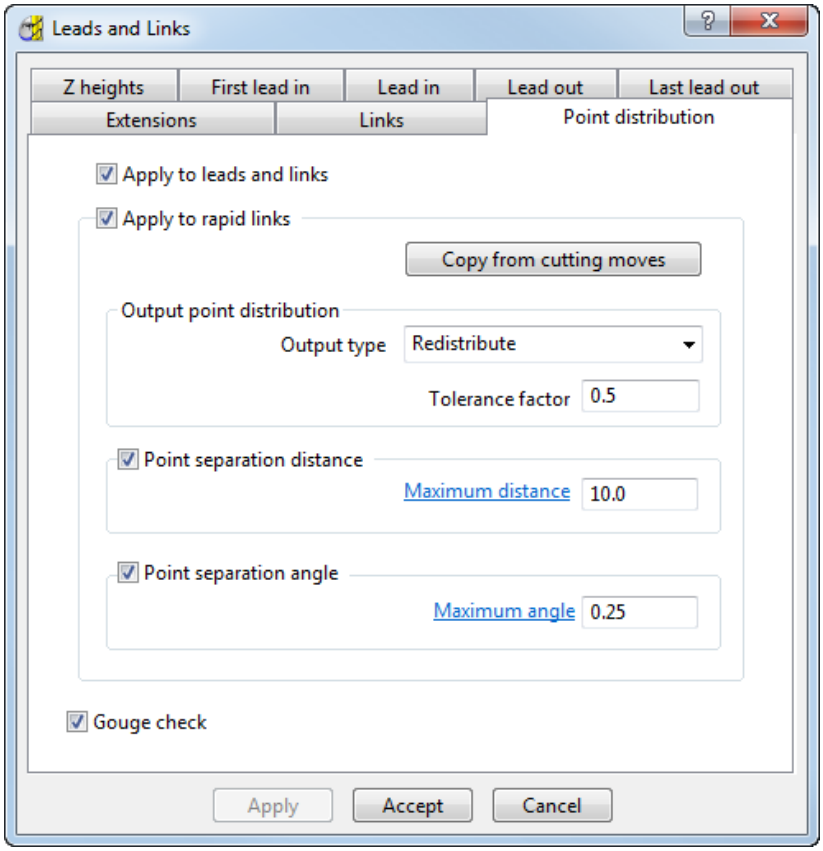
Whereas previous versions produced:



Point distribution on leads and links

You now have the option to specify point distribution on the non-cutting moves as well as the cutting moves of a toolpath.

The **Point distribution** tab on the **Leads and links** dialog enables you to control the point distribution along the leads and links. This is similar to the **Point distribution** options available for the toolpath cutting moves.



Apply to leads and links — Select to use the point distribution options used by the toolpath cutting moves for all the lead moves and the feed rate link moves. You specify the point distribution of the cutting moves on either the **Point Distribution** dialog (available from  on the **Main** toolbar) or the **Point distribution** page of a strategy dialog.

Apply to rapid links— Select to specify the point distribution required for all the moves within a rapid link move. The rapid link moves are: **Safe**, **Incremental**, and **Skim**. The point distribution is applied to the plunge moves as well as the rapid moves within the rapid link.

Copy from cutting moves — Click to copy the point distribution options used by the toolpath cutting moves to the rapid link moves

Output point distribution controls the point distribution of the rapid link moves.

Output type — Select the type of point distribution in the rapid link moves.

Tolerance and keep arcs — Select to automatically remove unnecessary points in the rapid moves while maintaining tolerance. The points are not equispaced, as unnecessary points are deleted.

Some of the points may be blue: these indicate the arc centres. The red points either side of the blue ones are the arc ends.

Tolerance and replace arcs — Select to replace arcs with straight line segments and to remove unnecessary points in the rapid moves while maintaining tolerance. This is similar to **Tolerance and Keep Arcs** except that all arcs are replaced by straight line segments (polylines). This option is suitable for machine tools which don't handle arcs well.

Redistribute — Select to allow the insertion of new points. This ensures a constant distance between points, only inserting extra points if they are necessary to keep tolerance. This can be especially useful when selecting the **Point separation distance**, or **Point separation angle** options. **Redistribute** may increase toolpath creation time but reduce time on the machine tool. This option is suitable for machine tools that can handle large numbers of equispaced points.

Fit arcs — Select to produce rapid moves with arcs inserted wherever possible. This option is suitable for machine tools which handle arcs well, but is available only for 3-axis toolpaths.

For diagrams of these options applied to cutting moves see **Point distribution > Output type**.

Tolerance factor — Enter a value to determine how many points are removed. This must have a value between **0** (removes a minimum number of points) and **1** (removes a maximum number of points while maintaining tolerance, so the rapid link moves contain the minimum number of points).

Point separation distance — Select to enter the maximum distance between rapid link move points.

Maximum distance — Enter the maximum distance between consecutive rapid link move points.

*These options are not available if you select an **Output type** of **Fit arcs**.*

Point separation angle — Select to enter the maximum change in angle between rapid link move points.

Maximum angle — Enter the maximum angle between consecutive rapid link move points. This is particularly useful when approaching the gimbal lock position of your machine tool, where very small angular changes can lead to massive changes in the azimuth and elevation angles. To ensure the maximum angle is not exceeded, PowerMILL adds additional toolpath points, which in turn slows down the machine tool.

*These options are available only if you select an **Output type** of **Redistribute**.*

Gouge check — Select this option for PowerMILL to gouge-check all leads and links.

Toolpath verification

Enhanced collision checking

PowerMILL now supports more sophisticated .mtd files. This means you can add to and modify your machine tool's .mtd file to improve collision checking. Within the .mtd file, you can now:

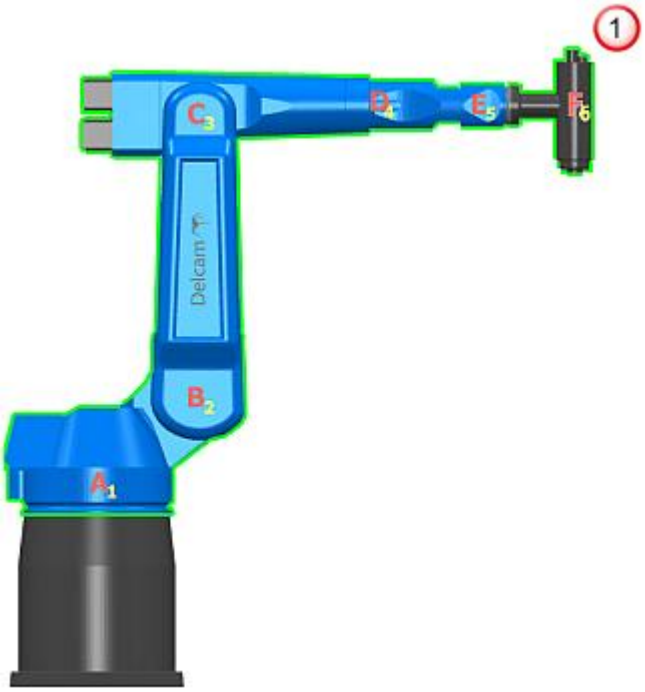
- split an existing collision group into multiple collision groups, so PowerMILL can check for collisions between each group (see page 91).
- identify static components in the machine cell, such as walls and tables, so PowerMILL can check for collisions between the static components and the machine tool (see page 96).
- create and add components to an exclusion list to reduce calculation time or to prevent PowerMILL reporting false collisions caused by inaccurate modelling of the machine tool (see page 98).

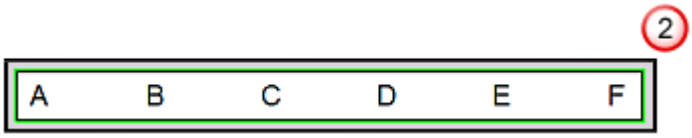
Introduction to collision groups

You can now split an existing collision group, within a kinematic chain, into multiple collision groups, so PowerMILL can check for collisions between each group.

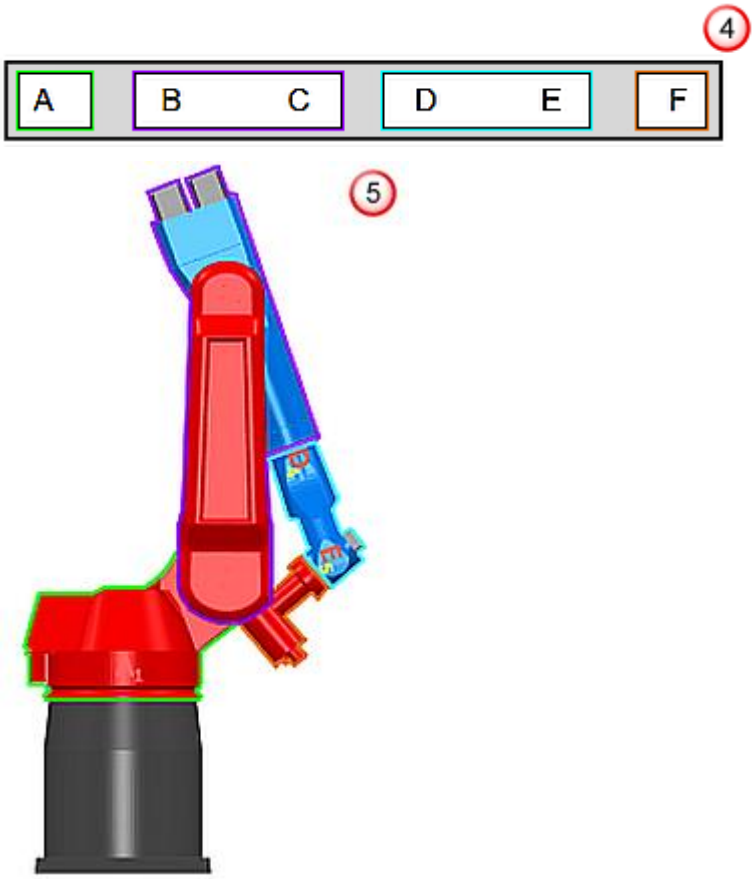
The kinematic chain is the section in an .mtd file that contains the information that defines the behavior of the axis-components (the moving parts) of a machine tool or, in this case, a robot.

In previous versions of PowerMILL, the axis-components were grouped in one collision group. This meant for the following robot (1), with its axis components in one collision group (2), PowerMILL did not recognise this orientation of the robot (like many other orientations) as a collision between A, B and F axis components (3).





Because PowerMILL 2014 recognises kinematic chains that contain multiple collision groups, you can now check for collisions between the axis components. With the collision group split into multiple collision groups (4), PowerMILL recognises the same orientation of the robot as a collision (5).



The single collision group is split into four collision groups because the components in each group have the potential to collide with the components in the other collision groups.

It is not necessary to split B and C nor D and E into smaller groups because B cannot collide with C and D cannot collide with E. Also, it is best practice to create as few collision groups as possible because each collision group increases the time PowerMILL takes to check for collisions.

PowerMILL does not check for collisions between adjacent axis components, because it often registers a collision between the two components due to inaccurate modelling of the joint between them, which is, in reality, safe. This rule also applies to axis components that are adjacent but are in different collision groups.

As a result, PowerMILL collision checks the axis-components like so:

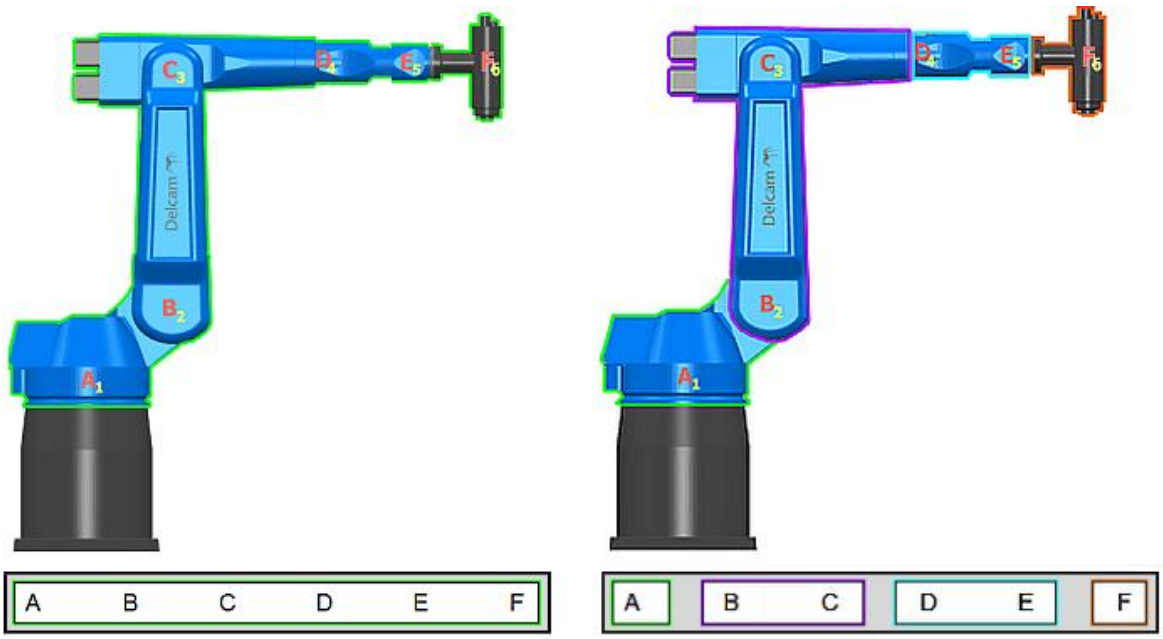
- A against C, D, E, and F.
- B against D, E, and F.

- C against A, E, and F.
- D against A, B, F.
- E against A, B, and C.
- F against A, B, C, and D.

For more information, see **Splitting a collision group** (see page 94).

Splitting a collision group

This example shows how to split the collision group in the Robot_One_CGroup.mtd file into four collision groups. The .mtd file is in the **Examples > MachineData** folder



You need to specify a NAME attribute for the A, B, D and F axis-components, as PowerMILL creates a new collision group for every axis-component with a NAME attribute.

Axis components without a NAME attribute are included in the collision group associated with the previous axis component with a NAME attribute. By not specifying NAME attributes for axis components C and E, C is included in the same collision group as B, and E with D.

Splitting the collision group into four collision groups:

1 Open the Robot_One_CGroup.mtd in an XML editor. This example uses **NotePad ++**.

The comment `<! --The arm is controlled by 6 rotary axes -->` signifies the start of the kinematic chain. Underneath the comment, the six axis-components that make up the robot are defined in their own section (referred to in XML language as a parent element).

Each axis-component section starts with the child element `<machine_part>` (1). You can identify which axis-component the section defines by looking at the axis address (2).

```
<!-- The arm is controlled by 6 rotary axes -->
  <machine_part> (1)
    <axis>
      <control_info ADDRESS="A" MIN="-185" MAX="185"
      PRIORITY="AUTO" HOME="0"/>
      <simple_rotary X="0.0" Y="0.0" Z="0.0" I="0" J="0"
      K="-1" />
```

2 Edit the `<machine_part>` element associated with the A axis by entering the attribute `NAME="Shoulder"`, so the element reads: `<machine_part NAME="Shoulder">`.

3 Edit the `<machine_part>` element associated with the B axis by entering the attribute `NAME="Upperarm"`, so the element reads: `<machine_part NAME="Upperarm">`.

4 Edit the `<machine_part>` element associated with the D axis by entering the attribute `NAME="Forearm"`, so the element reads: `<machine_part NAME="Forearm">`.

5 Edit the `<machine_part>` element associated with the F axis by entering the attribute `NAME="Head"`, so the element reads: `<machine_part NAME="Head">`.

The `<machine_part>` elements for C and E axis-components should not have NAME attributes.

6 Save the .mtd file.

By entering NAME attributes in the `<machine_part>` elements, you have split the existing collision group into four collisions groups:

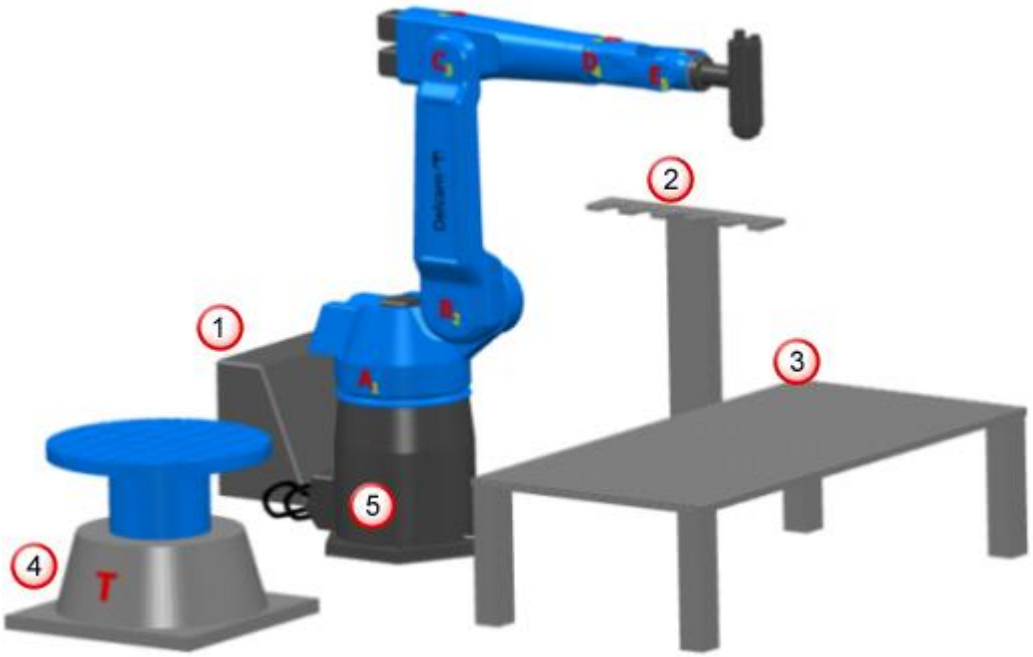
- Shoulder (containing the A axis-component)
- Upperarm (containing B and C axis-components)
- Forearm (containing D and E axis-components)
- Head (containing the F axis component)

Identifying components to include in collision checks

You can now identify static components that are in an .mtd file so PowerMILL checks them for collisions. Because PowerMILL recognises static components do not have a movement axis, PowerMILL only checks the static components for collisions with axis components (the moving components of a machine tool – and a rotary table, if present) and not with other static components.

This example shows how to add the following components so they are checked for collisions:

- Robot control 1
- Tool holder 2
- Table 3
- Base of the rotary table 4
- Base of the robot 5



PowerMILL displays static components in grey and axis components in blue.

Specify a NAME attribute for each static component; PowerMILL only collision checks components with a NAME attribute.

1. Open the Robot_cell.mtd file in an XML editor. This example uses **NotePad++**.

   Within the .mtd file, each static component is defined in its own section (referred to as parent elements in XML). Each static component is identified by a comment, for example: `<!-- The controller is static -->`, followed by the child element `<machine_part>`.

2. Edit the `<machine_part>` element associated with the controller by entering the attribute `NAME="RobotCellController"`, so the element reads: `<machine_part NAME="RobotCellController">`.

3. Edit the `<machine_part>` element associated with the tool holder by entering the attribute `NAME="RobotCellTool"`, so the element reads: `<machine_part NAME="RobotCellTool">`.

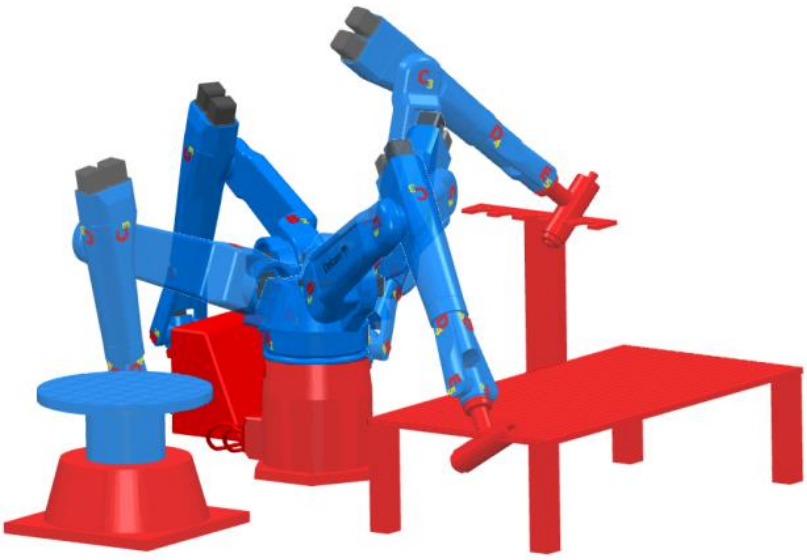
4. Edit the `<machine_part>` element associated with the table by entering the attribute `NAME="RobotCellTable"`, so the element reads: `<machine_part NAME="RobotCellTable">`.

5. Edit the `<machine_part>` element associated with the base of the rotary table by entering the attribute `NAME="RotaryTableBase"`, so the element reads: `<machine_part NAME="RotaryTableBase">`.

6. Edit the `<machine_part>` element associated with the base of the robot by entering the attribute `NAME="RobotBase"`, so the element reads: `<machine_part NAME="RobotBase">`.

7. Save the .mtd file.

Each static component is now collision checked.



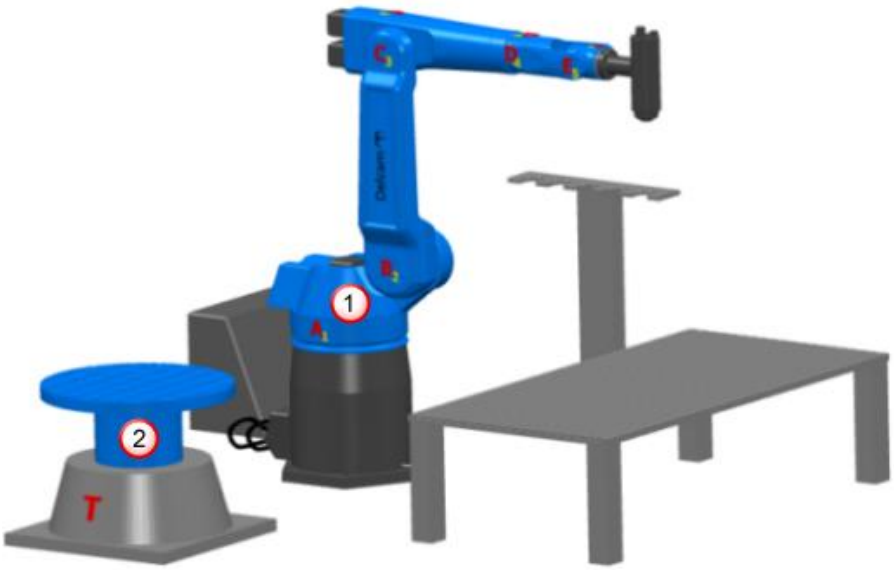
Creating and adding to an exclusion list

You can now identify static components in the .mtd file so PowerMILL checks them for collisions (see page 96). However, some of the collisions between static components and axis components that PowerMILL checks for are not necessary. For example, the tool holder and the rotary table do not need to be checked for collisions because, although the rotary table moves, the two components cannot collide.

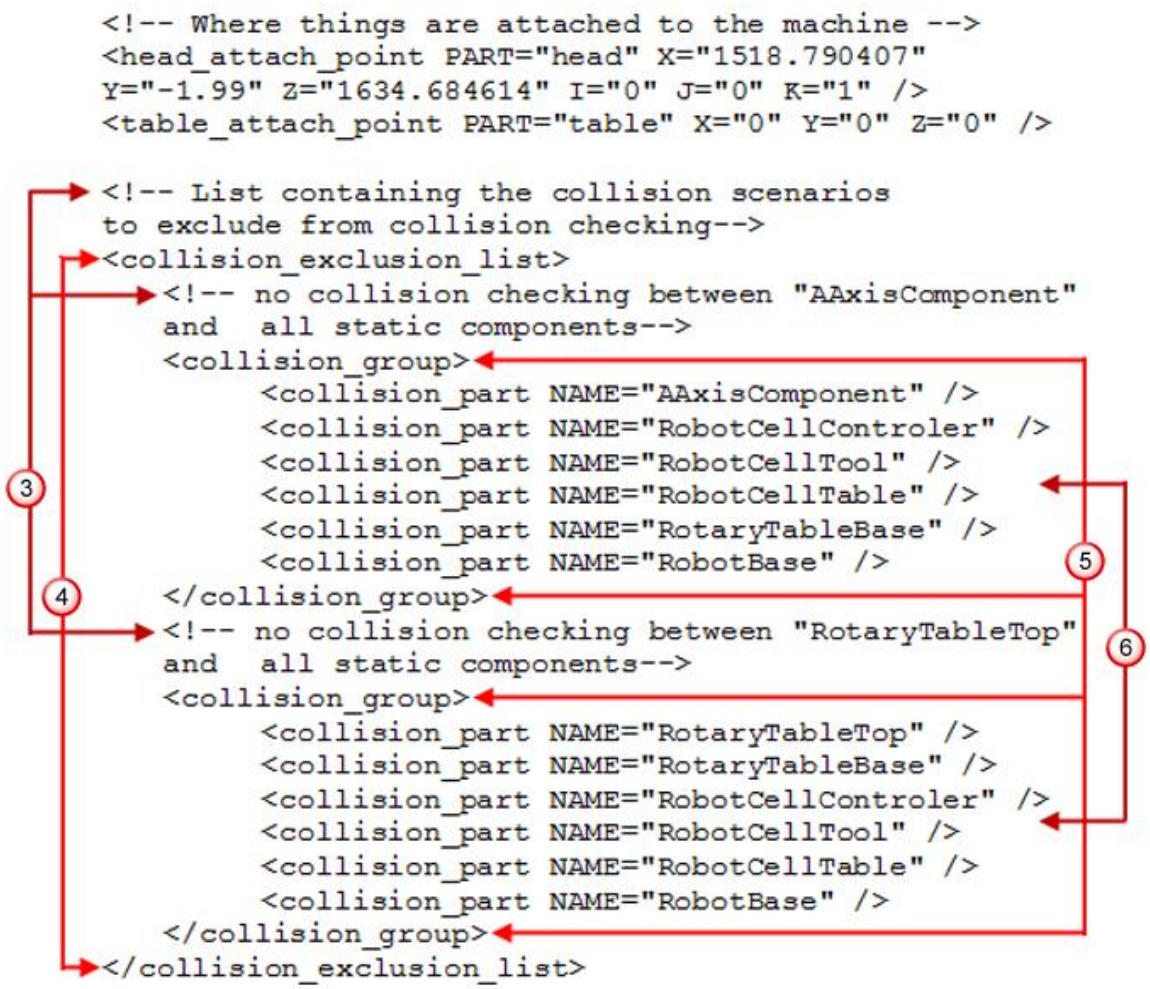
To prevent PowerMILL checking for such collisions, add the components to an exclusion list in the .mtd file. This reduces overall calculation time and can improve simulation quality.

This example shows how to create an exclusion list and add the following collision scenarios to it:

- Collision scenario 1 — Prevents PowerMILL checking for collisions between the A axis component (1) and each of the static components. (This includes the base of the robot, otherwise PowerMILL would register a collision between the two adjacent components.)
- Collision scenario 2 — Prevents PowerMILL checking for collisions between the rotary table-top (2) and each of the static components. (This includes the base of the rotary table, otherwise PowerMILL would register a collision between the two adjacent components.)



1. Open the Robot_cell.mtd file in an XML editor. This example uses **NotePad++**.
2. Underneath the parent element that specifies the attachment points, enter the following exclusion list:



3 Save the .mtd file.

Now PowerMILL does not check the A axis component or the table-top for collisions with any of the static components.

General principles when writing an exclusion list

- Use comments to provide information for yourself and other users of the file. These are ignored by PowerMILL. Each comment must start with `<!--` and end with `-->` (3).
- Start the exclusion list with `<collision_exclusion_list>` and end it with `</collision_exclusion_list>` (4).
- Start an exclusion-scenario with `<collision_group>` and end it with `</collision_group>` (5).
- Enter each component using the convention `<collision_part NAME="NameOfComponent" />` (6).

The number of components you can add to a collision scenario is unrestricted.

Toolpath output

NC program fixture offsets

PowerMILL now supports fixture offsets.

A fixture offset is a datum coordinate that you specify and then apply to a copy of a toolpath. This enables you to shift the toolpath, without needing to recalculate it, and machine it in a new position. A fixture offset also enables you to make last-minute adjustments to a toolpath's position, as you can use the machine tool controller to change the values of the fixture offset on the tape file.

*If you want to machine the same toolpath several times, use the **Move** option on the **Toolpath Transform** toolbar instead.*

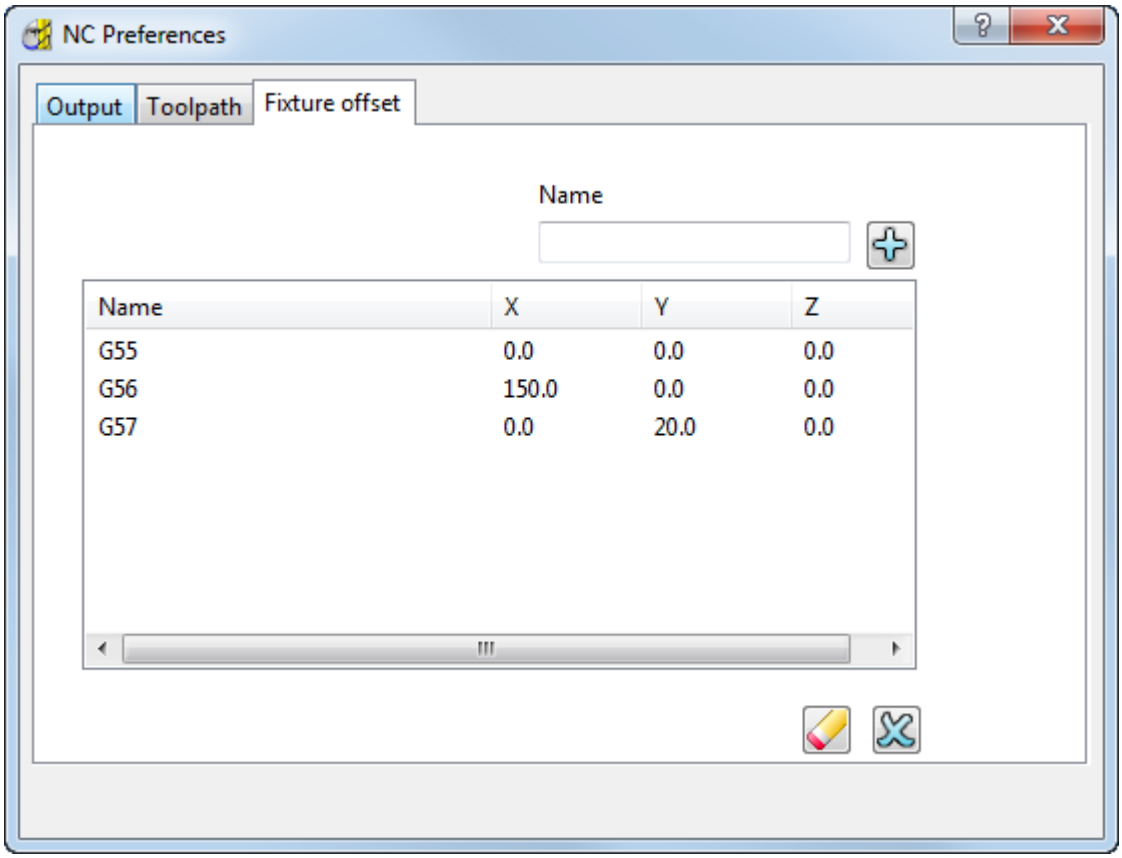
PowerMILL has two levels of **Fixture offset** tabs that have a parent-child relationship.

For an example of how to specify and apply fixture offsets, see **Using fixture offsets** (see page 104).

Parent fixture offset tab

The parent tab enables you to specify default fixture offsets that are inherited by the child **Fixture offset** tabs.

To display the parent tab, right-click **NC Programs** in the explorer and select **Preferences** > **Fixture offset** tab.



Name — Enter the name of the fixture offset.

The names you specify for the fixture offsets must be recognised by the option file that writes the NC program. For example, an option file for a Fanuc machine tool controller recognises G-codes (such as G55, G56 and G57). For advice and information on whether your option file supports this function, contact your Delcam sales agent.

Add fixture offset — Click to add the fixture offset to the **Fixture offset table**.

Fixture offset table — The table displays each fixture offset and its coordinates. To enter or change a coordinate, click in the **X**, **Y**, or **Z** field and enter the new coordinate value.

*You can leave the **X**, **Y** and **Z** fields blank because you can specify the coordinates on the machine tool controller.*

Reset fixture offset — Click to reset the coordinates of the selected fixture offset to **0**.

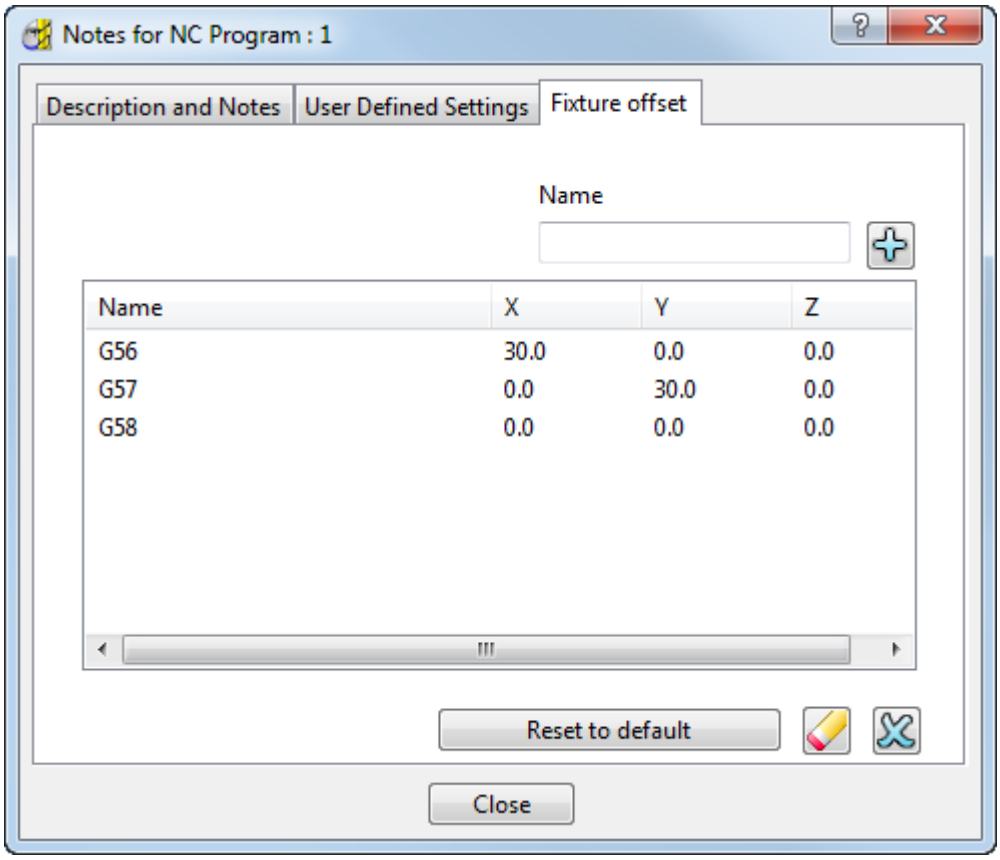
Delete fixture offset — Click to delete the selected fixture offset from the **Fixture offset table**.

Child fixture offset tab

The child tab enables you to specify the fixture offsets that you want to apply to the toolpaths in the associated NC program.

- The child tab inherits the fixture offsets specified in the parent tab.
- After you create an NC program, the child **Fixture offset** tab associated with the NC program does not continue to inherit the changes you make in the parent **Fixture offset** tab. To inherit subsequent changes, select the **Reset to default** button.
- The parent **Fixture offset** tab does not inherit changes you make in the child **Fixture offset** tab.

To display this tab, right-click an NC program in the explorer and select **Settings** > **Fixture offset list** .



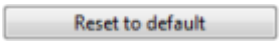
Name — Enter the name of the fixture offset.

The names you specify for the fixture offsets must be recognised by the option file that writes the NC program. For example, an option file for a Fanuc machine tool controller recognises G-codes (such as G55, G56 and G57). For advice and information on whether your option file supports this function, contact your Delcam sales agent.

 **Add fixture offset** — Click to add the fixture offset to the **Fixture offset table**.

Fixture offset table — The table displays each fixture offset and its coordinates. To enter or change a coordinate, click in the **X**, **Y**, or **Z** field and enter the new coordinate value.

 *You can leave the **X**, **Y** and **Z** fields blank because you can specify the coordinates on the machine tool controller.*



— Click so the tab inherits the default fixture offsets from the *parent* **Fixture offset** tab.

 *Changes you have made in this tab to fixture offsets that were created in the* parent ***Fixture offset*** *tab are reset, and any fixture offsets you have created in this **Fixture offset** tab are deleted.*

 **Reset fixture offset** — Click to reset the coordinates of the selected fixture offset to **0**.

 **Delete fixture offset** — Click to delete the selected fixture offset from the **Fixture offset table**.

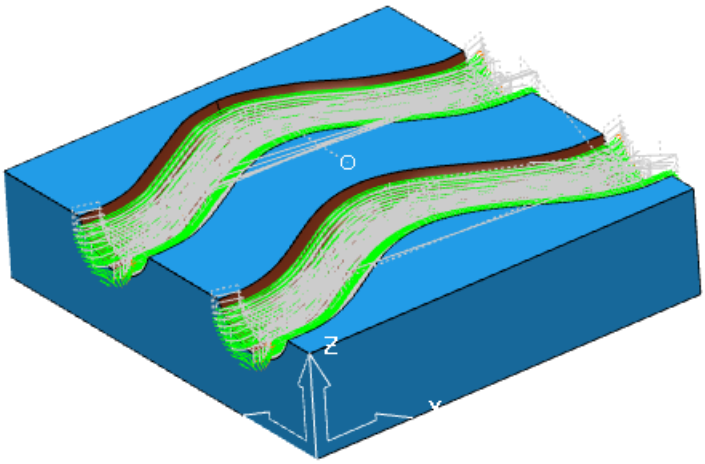
Applying fixture offsets

This example shows how to create and apply fixture offsets to toolpaths, using the Channels.dgk model in the **Examples** folder.

Create and add toolpaths to an NC program

1 Before you can apply fixture offsets, you must create the relevant toolpaths to machine the channel:

- **Roughing** — Model area clearance toolpath using a tip radiused tool with a diameter of **15**, a tip radius of **1**, and a length of **75**.
- **Rest Roughing** — Model rest area clearance toolpath using a ball nosed tool with a diameter of **15** and a length of **75**.
- **Steep and Shallow Finishing** — Steep and shallow finishing toolpath using a ball nosed tool with a diameter of **10** and a length of **50**.



2 Create four identical toolpaths:

a In the explorer, right-click the roughing toolpath and select **Settings** to display the **Model Area Clearance** dialog.

b On the dialog, click  to create a copy of the toolpath and then click **Calculate** to calculate the copy of the toolpath.

c Without closing the dialog, click  and then click **Calculate**. This creates another copy of the toolpath. Repeat again, so you have four instances of the **Roughing** toolpath.

3 Repeat step 2 to create four rest roughing toolpaths and four steep and shallow finishing toolpaths.

4 Create an NC program.

a In the explorer, right-click **NC Programs** and select **Create NC Program**.

This displays the **NC Program** dialog.

b In the **NC Program** dialog, enter the name: **Channel with fixture offsets**.

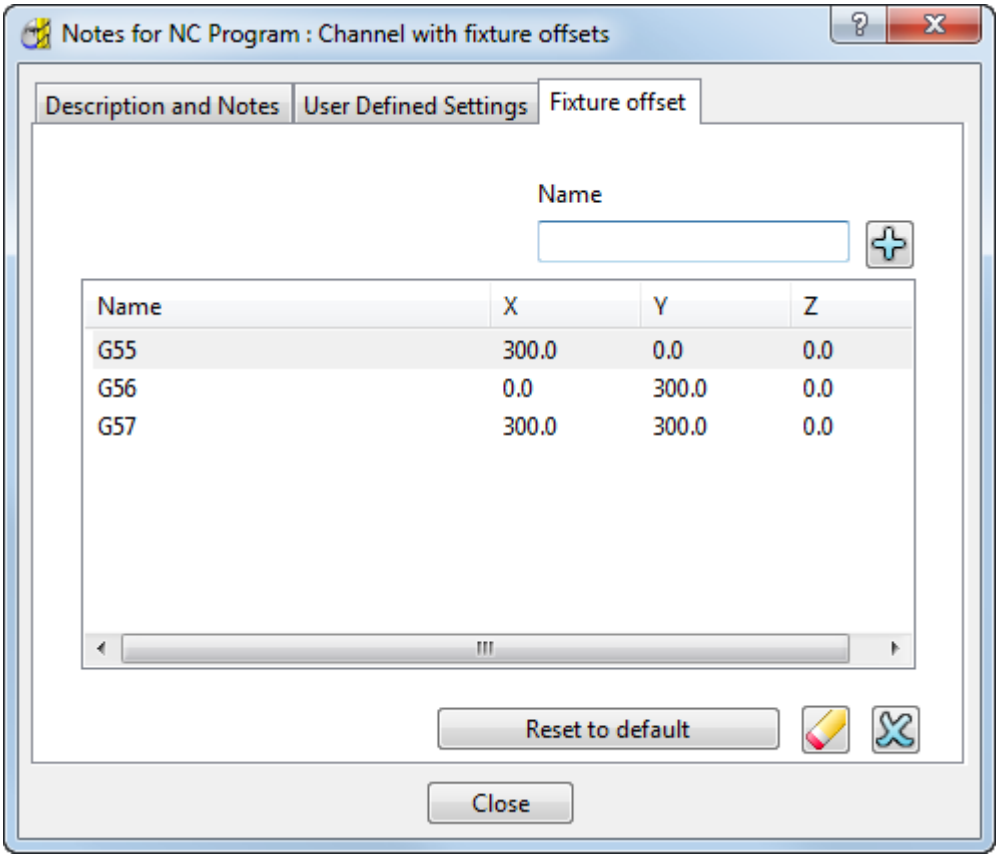
c Click **Accept**.

5 In the explorer, select all the toolpaths, right-click and select **Add to > NC Program**.

Create and apply fixture offsets

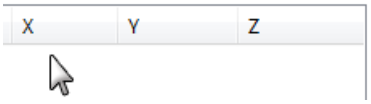
1 Create the fixture offsets:

a On the **NC Program** dialog, select  . This displays the **Fixture offset** tab.

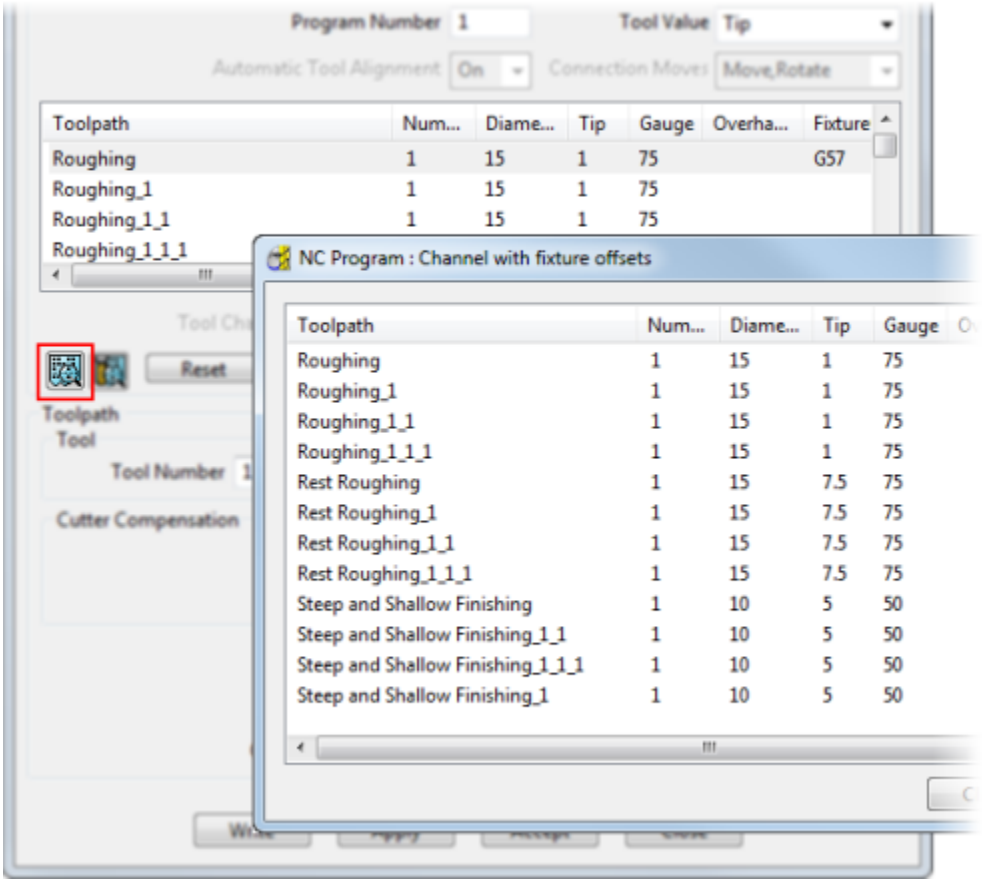


b In the **Name** field, enter **G55** and click  .

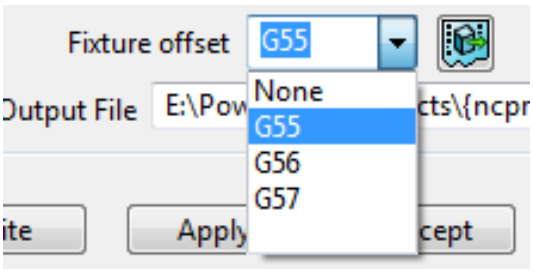
c In the **Fixture offset** table, click in the **X** field and enter **300**, then press **Enter**.



2 Repeat step 1 to create two more fixture offsets:
   - **G56** with a **Y** value of **300.**
   - **G57** with an **X** value of **300** and a **Y** value of **300.**
3 Click **Close**.
4 Apply the fixture offsets to the toolpaths in the NC program.
   a On the **NC Program** dialog, click . This displays the toolpaths in the **NC program toolpath** table in the resizable **Toolpath table** window.



   b In the **Toolpath table** window, select **Roughing_1** and then, on the **NC Program** dialog, select **G55** from the **Fixture offset** list.



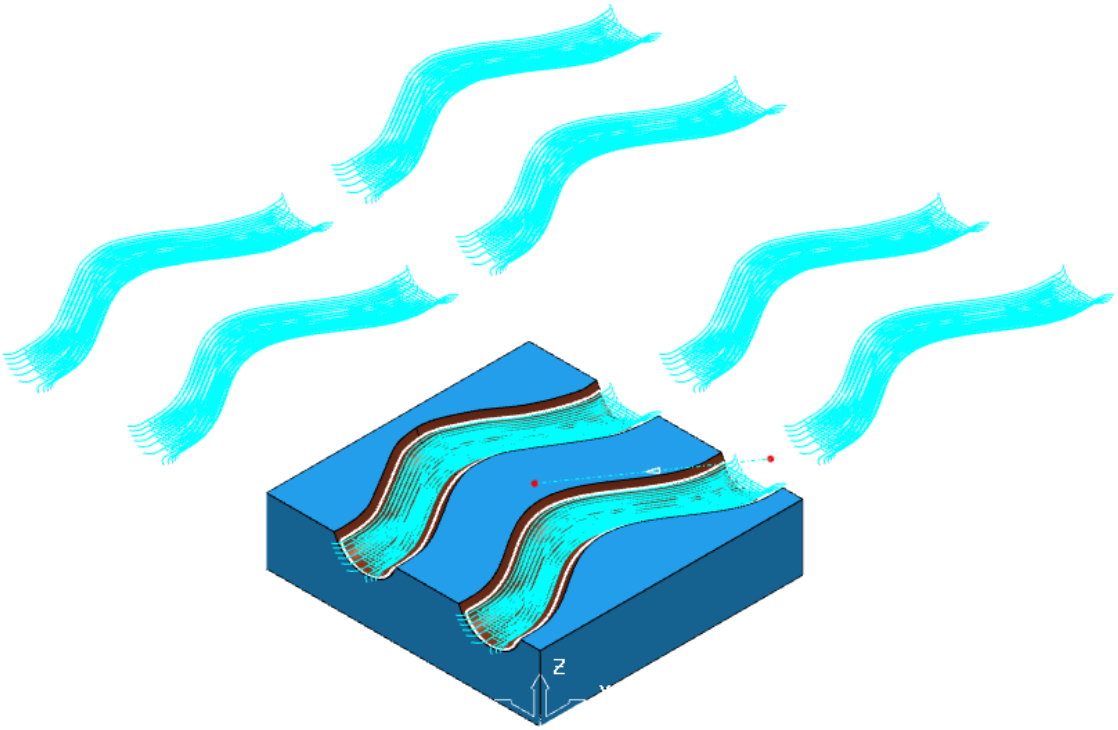
   This applies the fixture offset **G55** to the toolpath **Roughing_1**.
5 Select **Roughing_1_1** and apply the fixture offset **G56**.
6 Select **Roughing_1_1_1** and apply the fixture offset **G57**.

7 Repeat the process to apply the fixture offsets to the rest of the toolpaths.

*Do not apply fixture offsets to the original toolpaths. Apply **G55** to toolpaths ending in _1, **G56** to toolpaths ending in _1_1, and **G57** to toolpaths ending in _1_1_1.*

8 On the **NC Program** dialog, click **Apply**.

9 To display the toolpaths in their new positions, draw ( ) the **Channel with fixture offsets** NC program.

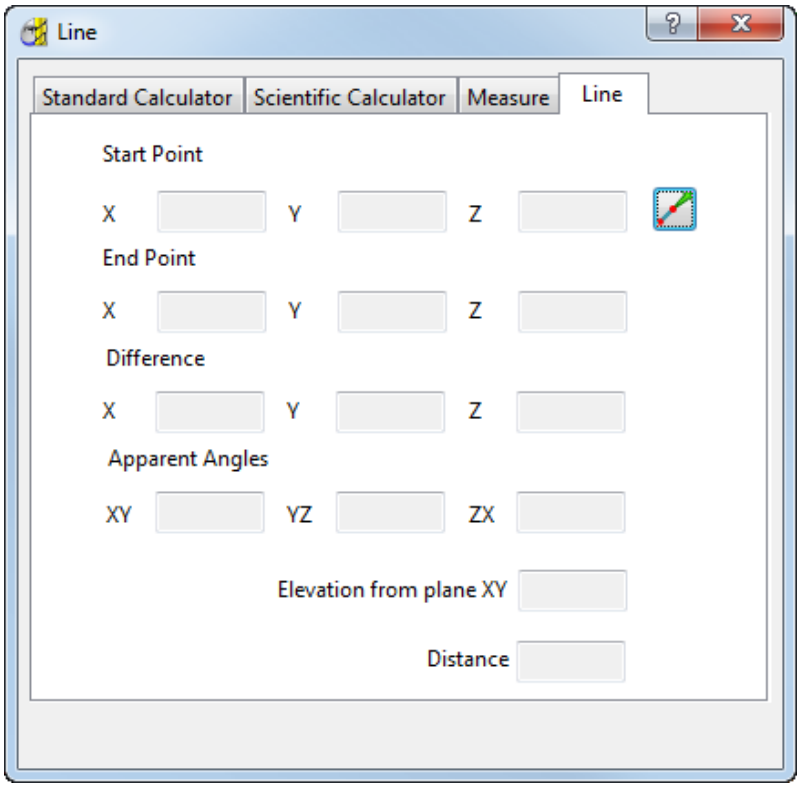


User interface

Measuring lines

PowerMILL 2014 reintroduces the **Line** tab on the **Measure** dialog. Although this functionality was available in PowerMILL 2013 it now requires fewer mouse clicks to display the distance, angle, and elevation between two points.

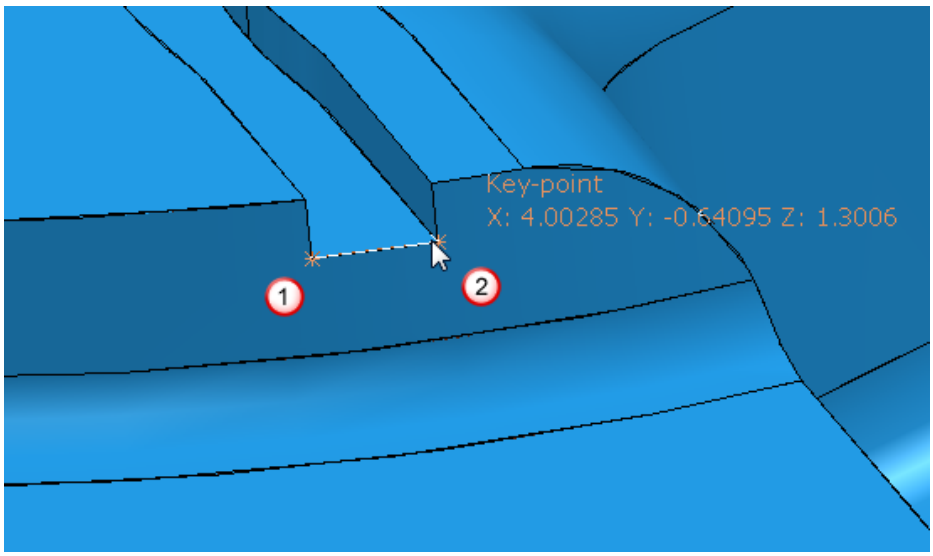
The **Measure** button displays the **Measure** tab on the **Calculator** dialog. Clicking on the **Line** tab displays the **Line** dialog.



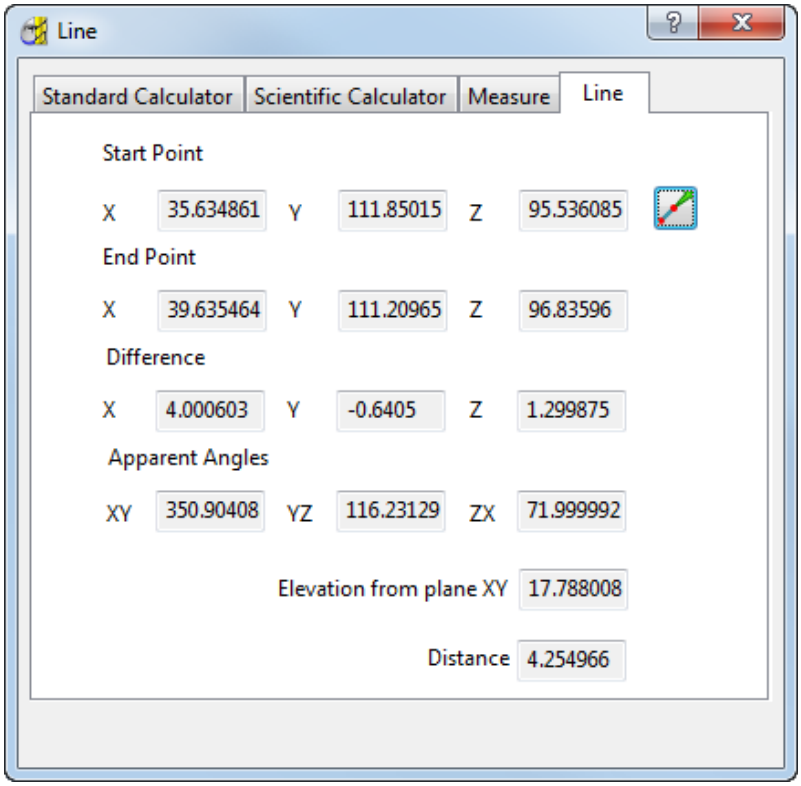
— Click to display the **Measure Difference** mode toolbar.

Click two points on the model to measure the distance and display the apparent angles.

For example:



The **Measure Difference** mode toolbar closes and updates the **Line** dialog with the distance between the two points and the apparent angles.



The other fields are calculated from these two points.

Start Point — Displays the coordinates of the first selected point.

End Point — Displays the coordinates of the second selected point.

Difference — Displays the difference in X, Y, and Z between the **Start Point** and the **End Point**.

Angle — Displays the angle in the XY, YZ, and ZX planes between the **Start Point** and the **End Point**.

Elevation — Displays the angle of the line joining the **Start Point** to the **End Point** to the principal working plane.

*Changing your principal working plane in the **Information** toolbar changes the angle measured here.*

Distance — Displays the length of a line joining the **Start Point** to the **End Point**.

Automation

Macro programming enhancements

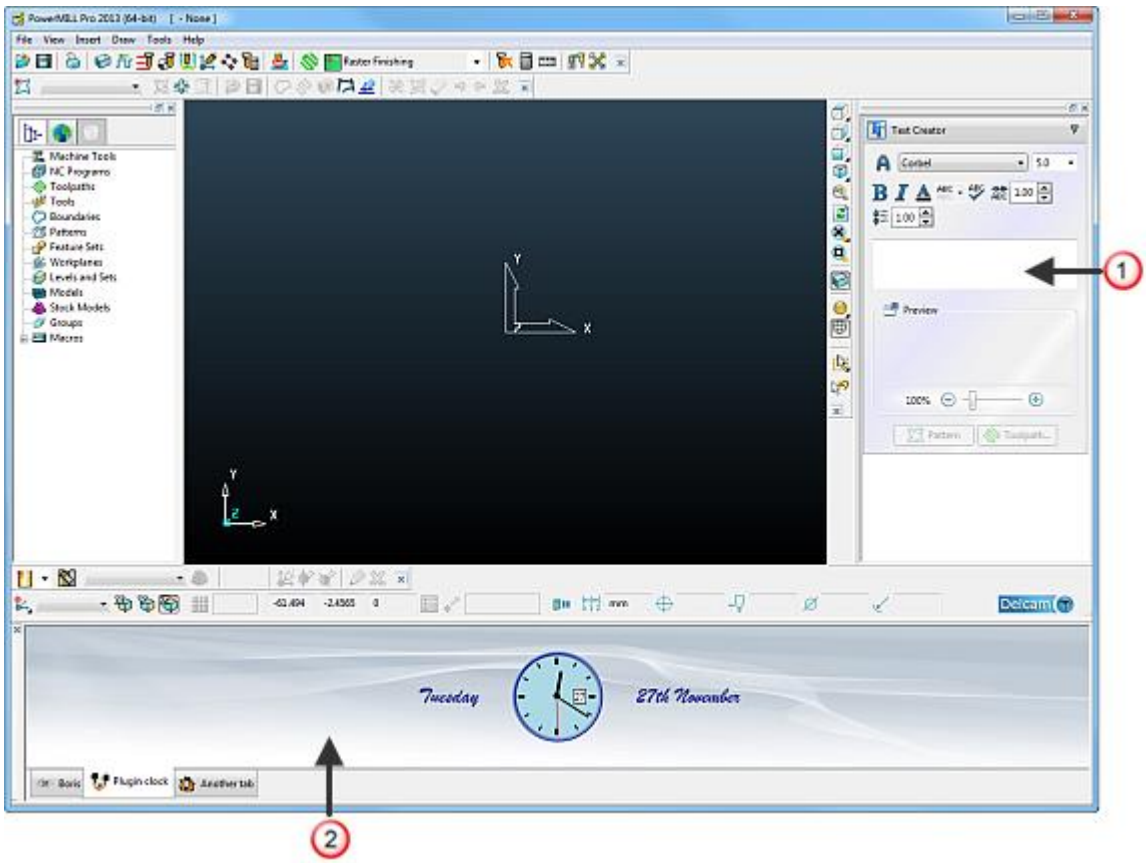
The macro programming language now lets you:

- use INPUT commands to prompt the user to select from a list of predefined options.
- create OBJECT variables to hold collections of variables.
- use date and time functions to time activities.
- use the components() function to loop over features in feature sets.
- read and write variable data to .txt files.

For more information, see the Macro Programming guide.

Plugin enhancements

A new horizontal plugin window has been added to PowerMILL. Now PowerMILL features both a horizontal and a vertical plugin window.

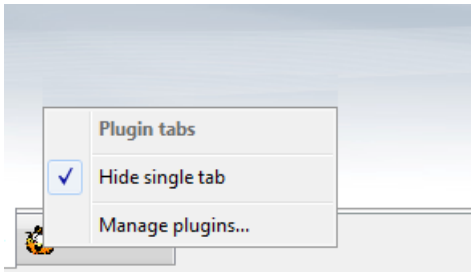


1. **Vertical plugin window**
2. **Horizontal plugin window**

The horizontal layout of the new plugin window provides developers with a greater amount of flexibility when designing plugins.

Plugins are developed to be displayed in just one of the two plugin windows. You cannot change the window in which a plugin is displayed.

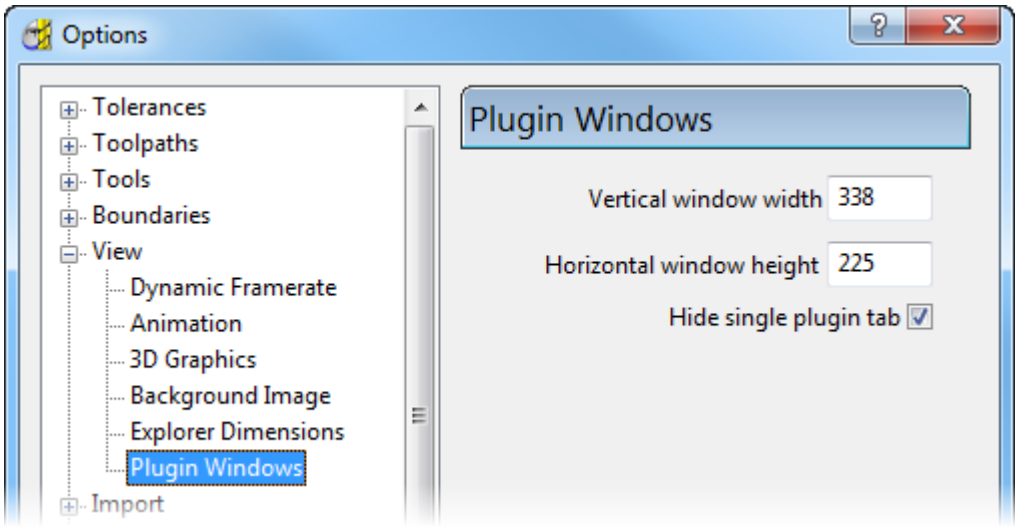
To display the horizontal plugin window's context menu, right-click on a plugin tab.



Hide single tab — Select to only display plugin tabs when there are two or more plugins enabled.

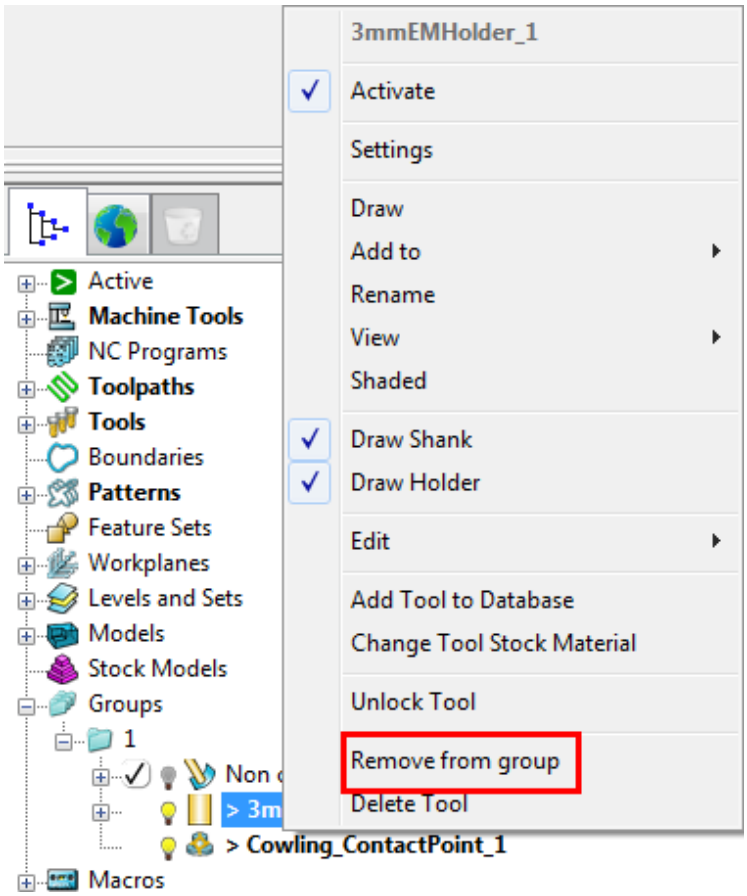
Manage plugins — Select to display the **Plugin Manager**. The **Plugin Manager** enables you to manage the plugins that are currently installed on your PC.

To specify the plugin windows' default settings, use the new **Plugin Windows** page on the **Options** dialog.

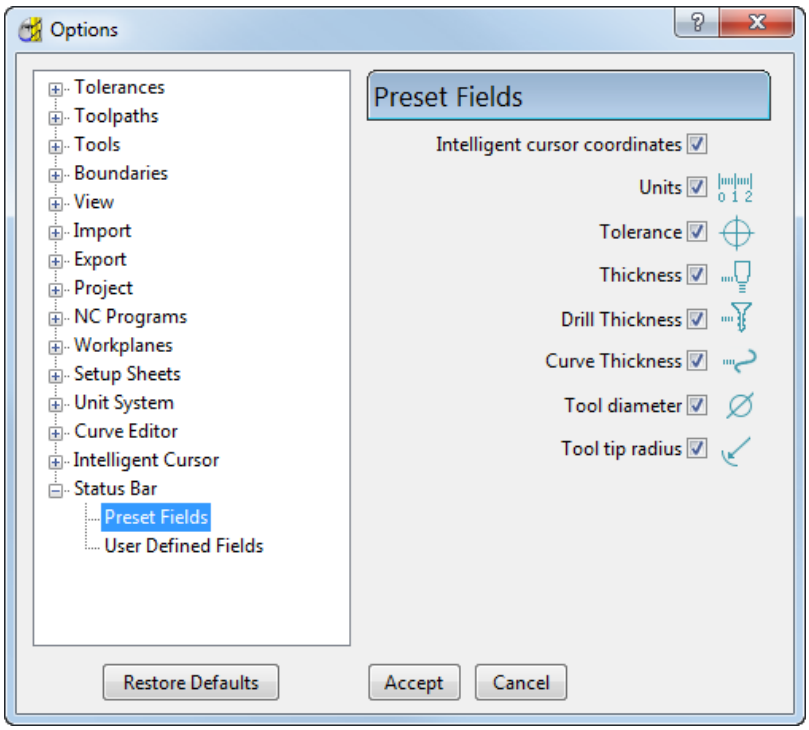


General enhancements

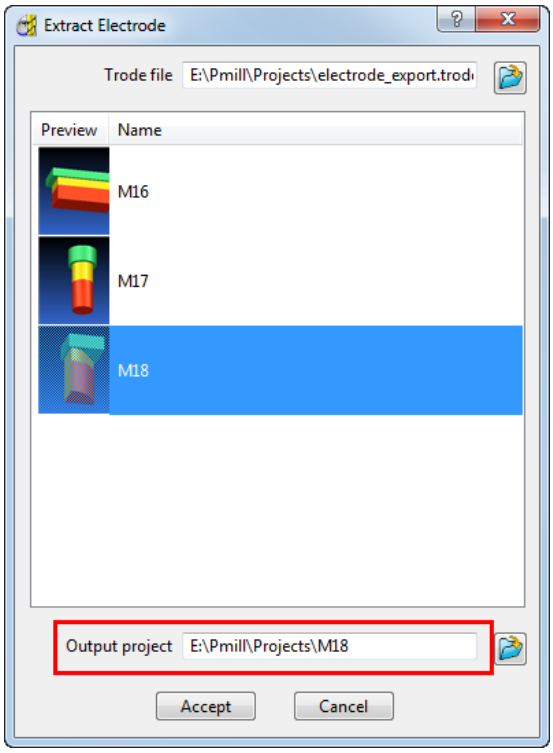
- You can now remove entities from a group by selecting the **Remove from group** menu option from the individual group entity menu. In previous versions you could remove the entity from a folder and not from the group.



- The status bar can now display the **Drill Thickness** and **Curve Thickness** of the active toolpath. To display these values, select them on the **Tools > Options > Status Bar > Preset Fields** dialog.



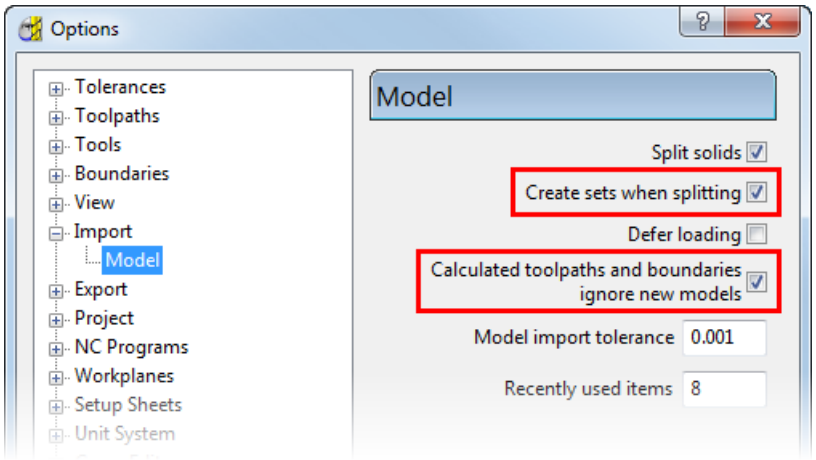
- PowerMILL now automatically populates the **Output Project** field in the **Extract Electrode** dialog with the name of the selected electrode and the same folder as the selected .trode file.



*The **Extract Electrode** dialog is available from the **File > Extract Electrode** menu.*

Model import enhancements

When importing models into PowerMILL there are two new option on the **Tools > Options > Import > Model** dialog.



Create sets when splitting — When selected, PowerMILL places all the surfaces associated with a solid in a set which has the name of the original solid. When deselected, all the surfaces are placed in a single set.

To use this option the CAD data must be sent to PowerMILL as solids and not surfaces. So, you must select the appropriate options in Delcam Exchange to import the model as a solid. By default Delcam Exchange splits solids into surfaces before importing into PowerMILL.

Calculated toolpaths and boundaries ignore new models — Select for existing toolpaths and boundaries to ignore newly imported models. This is useful when importing clamping models into a project as existing toolpaths and boundaries add the clamping model to a new thickness set with a **Machining Mode** of **Ignore**. This enables you to recalcualate existing toolpaths without having to manually change the thickness sets to accommodate the clamping model.

Nc program updates

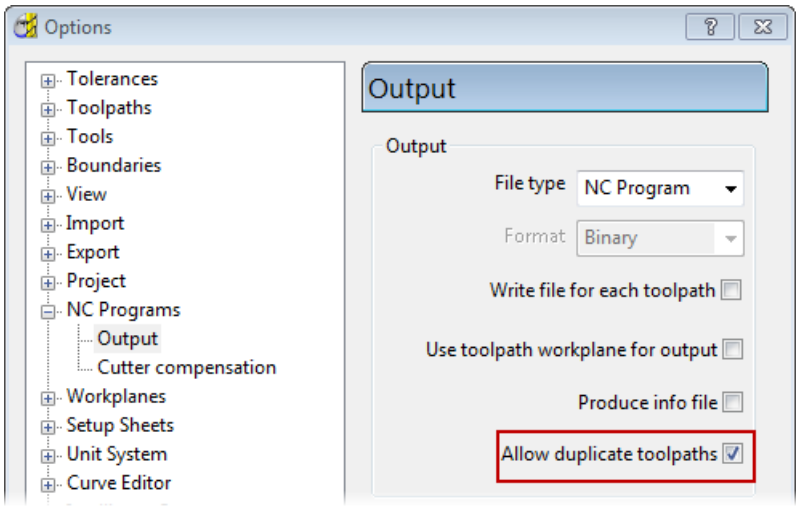
There have been a number of updates to the **NC Programs** branch in the explorer.

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.

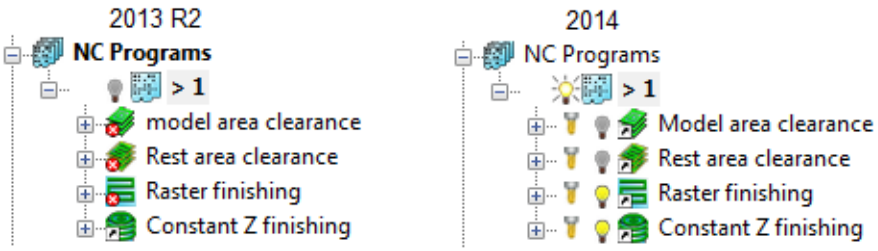
The default behavior for PowerMILL 2014 is to allow toolpath duplication, although, when you write the NC program, a message is displayed to inform you of any duplications.

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

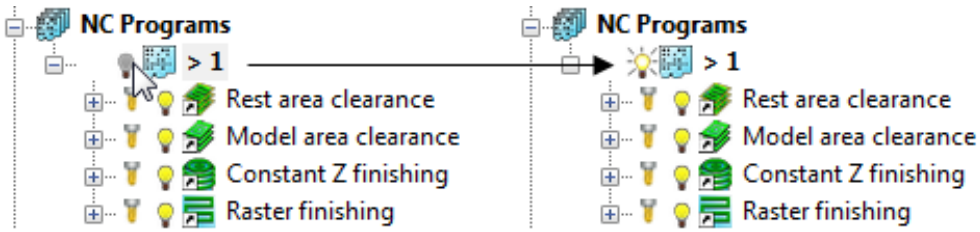


Drawing toolpaths in an NC program

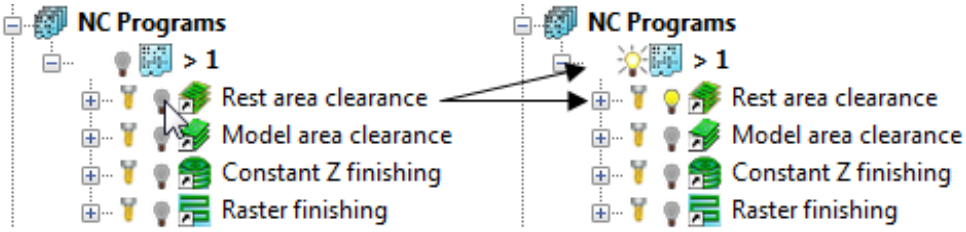
The introduction of light bulbs next to toolpaths in an NC program now enables you to choose which toolpaths in an NC program you want to draw.



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.



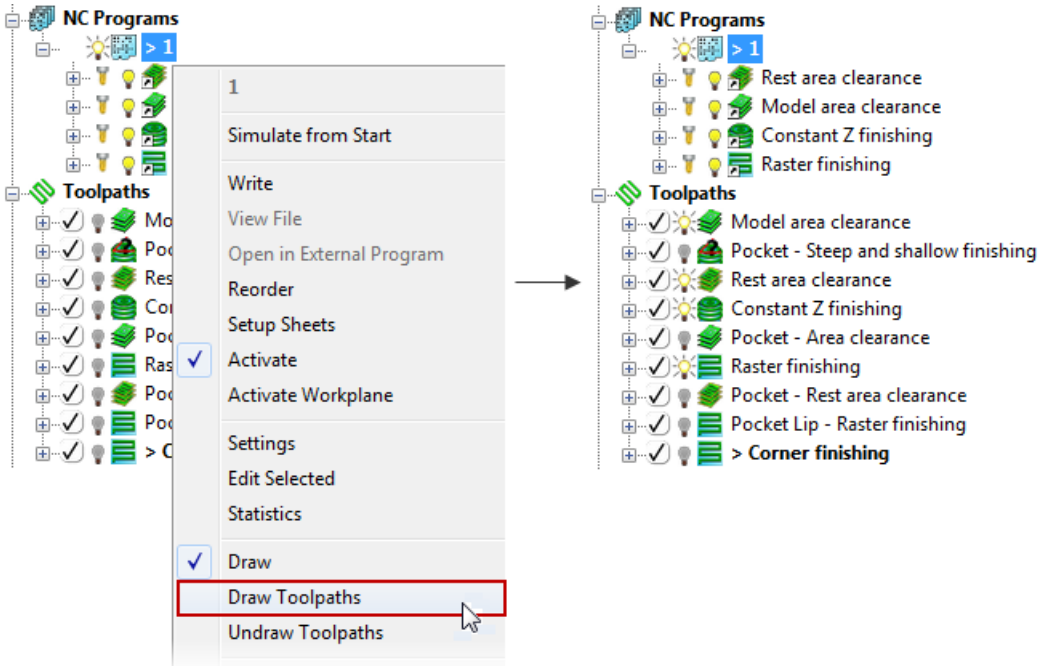
Drawing a toolpath, which is in an undrawn NC program, automatically draws the NC program, so the toolpath is displayed.



*In previous versions of PowerMILL, when you drew an NC program, the toolpaths in the NC program would be displayed and, in the **Toolpaths** branch, the lightbulbs next to the those toolpaths would update to: .*

*In PowerMILL 2014, this no longer happens by default. To draw the toolpaths included in the NC program in the **Toolpaths** branch, right-click the NC program to display the NC program's context menu and select **Draw Toolpaths**.*

*An **Undraw Toolpaths** option has also been added to the menu.*



Updated icons in the NC Programs branch

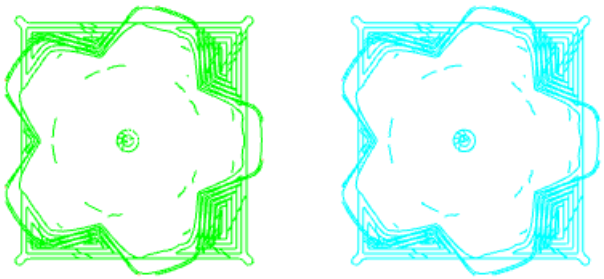
- The **NC Programs** branch of the explorer now features an updated tool change icon and a new fixture offset icon:

   — Toolpath features a tool change.

   — Toolpath features a fixture offset.

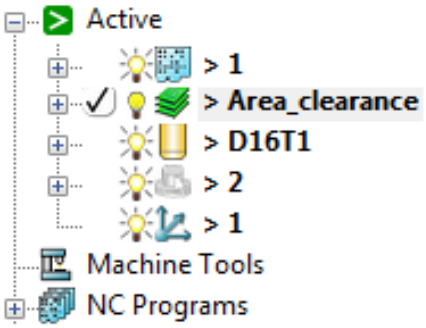
Toolpaths in an NC program now drawn in blue

- When you include a toolpath in an NC program, the toolpath is now drawn in blue. This is to help you differentiate between toolpaths that are included in an NC program and those that are not.

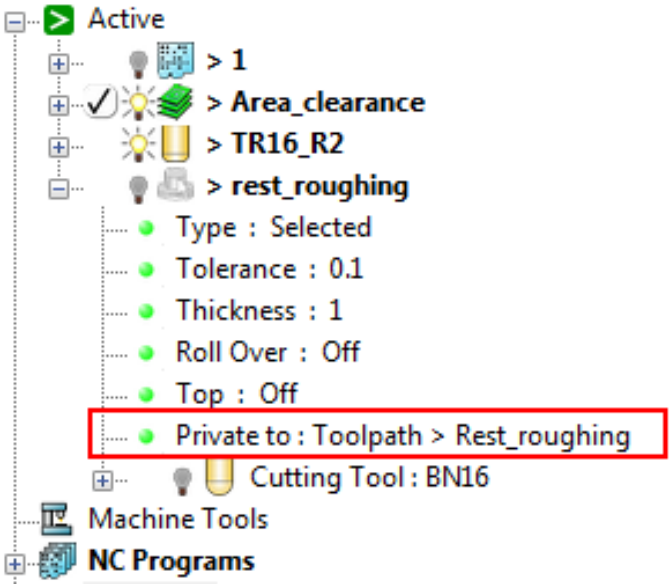


New Active branch in the explorer

There is a new **Active** branch in the explorer that displays all the currently active entities.



This is particularly useful for seeing the active boundary and its associated toolpath. In previous versions of PowerMILL, it was difficult to see the active boundary because private boundaries were displayed in the branch of their associated toolpath.



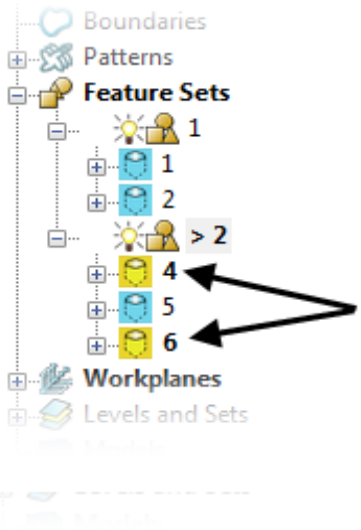
Individual active menu

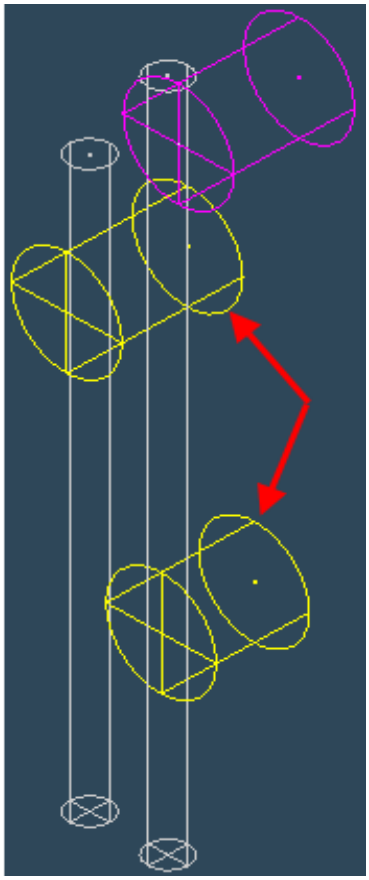
Right-click an active entity to display its context menu. For example, right-clicking the active boundary displays the individual boundary menu.

For information about the options on each context menu, see the corresponding section in the Explorer book.

Selected feature colour in the explorer

The selected features are now displayed yellow in the explorer as well as in the graphics window. This makes it much easier to locate the selected features.





Index

A

Active entity • 120
Align tangent • 34
Applying fixture offsets • 104
Arc
  Arc editing • 22
  Arc insertion • 22
Arc-line curve menu • 24
Azimuth angle • 38

B

Barrel tool • 52

C

Changing the block in a stock model • 58
Collision checking • 91
  Collision groups • 91
  Editing MTD files • 91
  Kinematic chain • 91
  Splitting collision groups • 94
Collision groups • 91
Composite curve editing • 30
Creating an external thread • 70
Curve
  Align tangent • 34
  Arc editing • 22
  Arc insertion • 22
  Arc-line curve menu • 24
  Azimuth angle • 38
  Composite curve editing • 30
  Edit arcs • 22
  Edit fillets • 22
  Editing composite curve • 30
  Elevation angle • 38
  Fillet editing • 22
  Fillet insertion • 22
  Insert arcs • 22
  Insert fillets • 22
  Limit curves • 50
  Redistribute points on a curve • 40, 46
  Reducing points in a curve • 40, 45
  Reorder curve segments • 49
  Repoint curve • 40
  Tangent align • 34
  Tangent editor • 34, 39
  Tangent magnitude • 34, 39
Curve Thickness • 115

D

Dovetail tool • 53
Draft angle - drilling • 66
Drawing toolpaths in an NC Program •117
Drill thickness • 115
Drilling • 62, 63
  Creating an external thread • 70
  Draft angle - drilling • 66
  External threading • 63, 70
  Feed rate reduction at hole intersections • 73
  Hole intersections • 73
  Reducing feed rates at hole intersections • 73
  Retract - drilling • 66
  Thread milling • 62
  Threading • 67
  Varying drilling feed rates • 73
Duplicating toolpaths in an NC program • 117

E

Edit arcs • 22
Editing composite curve • 30

Editing MTD files • 91
Electrode enhancements • 115
Elevation angle • 38
Entity
  Active entity • 120
External threading • 63, 70

F

Feature colour • 121
Feed rate reduction at hole intersections • 73
Fillet
  Fillet editing • 22
  Fillet insertion • 22
Fixture offsets • 100
  Applying fixture offsets • 104
Flat machining • 84

G

Group
  Remove from group • 114

H

Hole intersections • 73

I

Importing models • 115
Insert fillets • 22

K

Kinematic chain • 91

L

Limit curves • 50
Lines - measuring • 108

M

MachineDNA • 19
Machining
  External threading • 63, 70
  Flat machining • 84
  Optimised raster • 87
  Raster ordering • 85
  Thread milling • 62
  Varying drilling feed rates • 73
  Vortex machining • 8
Macro programming • 111
Measuring lines • 108
Model import • 116

N

NC program
  Applying fixture offsets • 104
  Drawing an NC program • 117
  Fixture offsets • 100
  NC program icons • 119
  Remove from group • 114

O

Optimised raster • 87
Ordering
  Raster ordering • 85

P

Plugin • 112
Point
  Redistribute points on a curve • 40, 46
  Reducing points in a curve • 40, 45
  Repoint curve • 40

R

Raster
  Optimised raster • 87
Raster ordering • 85
Redistribute points on a curve • 40, 46
Reducing feed rates at hole intersections • 73
Reducing points in a curve • 40, 45
Remove from group • 114
Reorder curve segments • 49
Repoint curve • 40
Retract - drilling • 66

S

Splitting collision groups • 94
Stock model • 58
  Changing the block in a stock model • 58

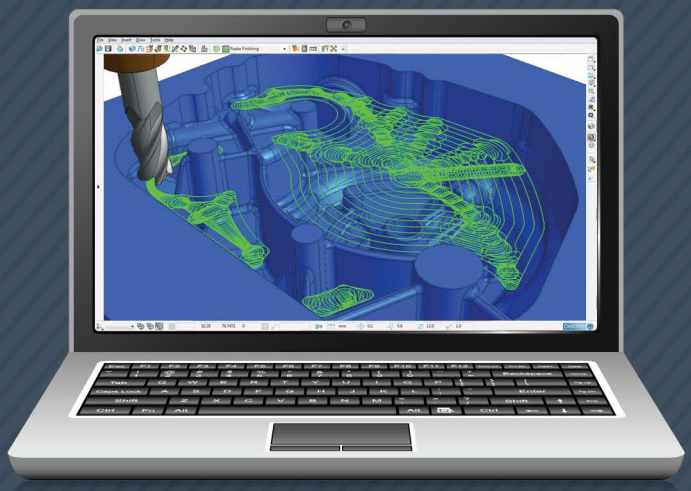
T

Tangent editor • 34, 39
Tangent magnitude • 34, 39
Thickness
  Curve Thickness • 115
  Drill thickness • 115
Thread milling • 62
Threading • 67
Tool
  Barrel tool • 52
  Dovetail tool • 53
  Tipped disc tool • 54
  Tool assembly preview • 55
Tool assembly preview • 55
Tool database enhancements • 56
Toolpath
  External threading • 63, 70
  Flat machining • 84
  Optimised raster • 87
  Raster ordering • 85
  Thread milling • 62
  Varying drilling feed rates • 73
  Vortex machining • 8

V

Varying drilling feed rates • 73
Vortex machining • 8
  Area clearance - Vortex • 8
  MachineDNA • 19
  Vortex - overview • 7
  Vortex and step cutting example • 12
  Vortex example • 16

PowerMILL 2014



www.delcam.tv

www.delcam.tv/lz

www.powermill.com

World leading 2, 3 and 5-axis CAM software

Delcam Advanced Manufacturing Solutions

Powering your productivity

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

www.delcam.com

THE QUEEN'S AWARDS FOR ENTERPRISE: INNOVATION 2010

THE QUEEN'S AWARDS FOR ENTERPRISE: INTERNATIONAL TRADE 2011

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