Delcam Advanced Manufacturing Solutions

PowerMILL 2012

What's New

Powering your productivity

Delcam Advanced Manufacturing Solutions

Whether you are new to Delcam products or an experienced user, you may not be aware of the other complementary technologies available from Delcam that could improve your company's productivity. Delcam offers a complete and diverse range of advanced CADCAM software solutions to help every aspect of your manufacturing process from design concept to manufactured part or tool, providing you with unrivaled speed, flexibility, and ease-of-use. Adding to your suite of Delcam solutions minimizes 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 minimize machining time and maximize 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 stabilization.

FeatureCAM

www.featurecam.com

FeatureCAM is the unique CAM system that uses feature-based and knowledge-based technologies for automated machining, minimizing 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 revolutionizes 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 and Turn/Mill 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 computerized 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 visualization and rendering materials.
- Use the comprehensive and customizable 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 dental restorations, custom orthotic insoles, medical implants, orthopaedic footwear, and maxillofacial prosthetics.

You can connect with Delcam in a variety of ways:

Delcam	**www.delcam.com**
Delcam TV	**www.delcam.tv/LZ**
Learning Zones	**www.delcam.tv/LZ**

PowerMILL 2012

What's New

Release Issue 2

PowerMILL

Acknowledgements

Patents

The Raceline smoothing functionality is subject to patent applications.

Patent granted: GB 2374562 Improvements Relating to Machine Tools

Patent granted: US 6,832,876 Machine Tools

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

Patent granted: GB 2 423 592 Surface Finish Prediction

Licenses

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

PowerMILL 2012. Published on 22 August 2011

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Summary of new features

PowerMILL is the leading specialist NC CAM software for manufacturing complex shapes typically found in the toolmaking, automotive, and aerospace industries. PowerMILL 2012 offers all of the original features of PowerMILL 2011, but with numerous improvements. The most significant improvements are described in this document.

Toolpath preparation

There are additions to the **Curve Editor** toolbar (see page 3):

- You can create Bezier curves.
- You can change a continuous line into a Bezier curve.

The tool holder profile enables you to have different clearances for the holder and shank (see page 9).

There is a new **Thread Milling** tool (see page 12).

The **Point Distribution** dialog allows you to manage the tool axis angular change (see page 14).

The **Workplane Editor** toolbar has a new  **Mirror** option (see page 15).

There is a new **Workplane Transform** toolbar (see page 15).

You can select all the surfaces that any segments of the pattern are embedded in (see page 15).

You can now choose which plane you want when creating a Best fit plane (see page 18).

Toolpath generation

The **Strategy** dialog now contains all the items need to create a toolpath (see page 19).

There is a new **Flowline Finishing** strategy (see page 23). This is a 5-axis multi-surface strategy which creates a toolpath between two curves, called drive curves, by smoothly blending between them. This strategy is typically used on complex fillets or gently curved surfaces.

There is a new **Parametric Spiral Finishing** strategy (see page 47). This creates a spiral toolpath between a central curve and a check surface.

There are significant improvements to the way **Blisk Machining** handles the tool axis (see page 58).

Improvements to the toolpath statistics and NC program statistics (see page 60).

The **Spiral** option on the **Constant Z** toolpaths now allows you to spiral down onto flats.

User interface

There is a new **Measure** toolbar which provides tools to interactively populate numeric fields in dialogs with points, coordinates, distances, angles, radii, or unit vectors (see page 61).

The **Position**  dialog has two additional tabs which allow you to locate the centre point of a circle and define a position between two connected points (see page 68).

The **Direction** dialog  has additional options to align the direction vector with the view, geometry, tool, or line (see page 72).

You can now customise column views in the **NC program** and **Tool Database** dialogs (see page 74).

You can now display the centre points of arcs (see page 76).

You can now select visible reversed components from the **Models** and individual model context menus (see page 77).

Machine tools can now be directly imported and manipulated inside the explorer (see page 78).

General enhancements

You can now write your own macros exploiting PowerMILL parameters (see page 86).

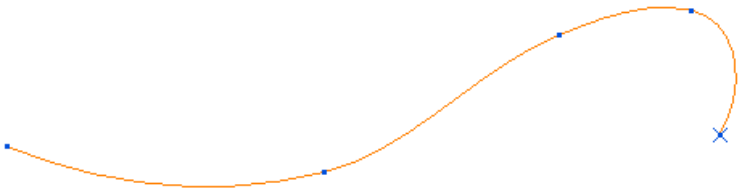
Toolpath preparation

Curve editor improvements

There are several improvements to the **Curve Editor** toolbar. These reduce the need to use PowerSHAPE or other third party CAD systems to generate complex wireframe geometry for machining.

- Editing a continuous line now uses the Bezier curve editor. This enables you to turn a continuous line into a Bezier curve on editing (see page 4).
- You can now create **Bezier curves** using the  button.

This creates a curve by joining user-specified points.

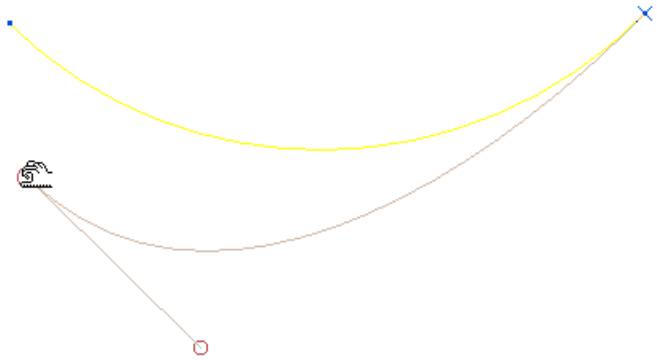


Bezier curves are parametric cubic curves. The portion of the curve between points is a span. Each span is defined by a start and end point, start and end tangent, and start and end magnitude. On creation, there is tangency continuity between spans.

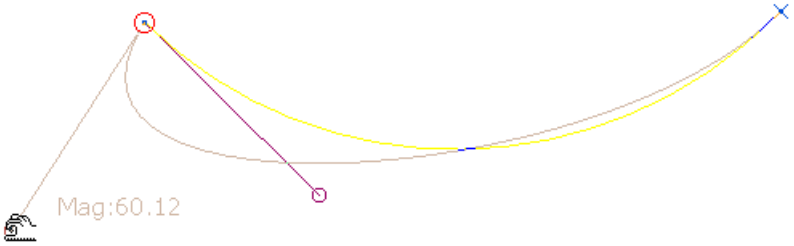
Selecting the curve and then a point on the curve displays the point, tangent, and magnitude.

Editing the curve enables you to:

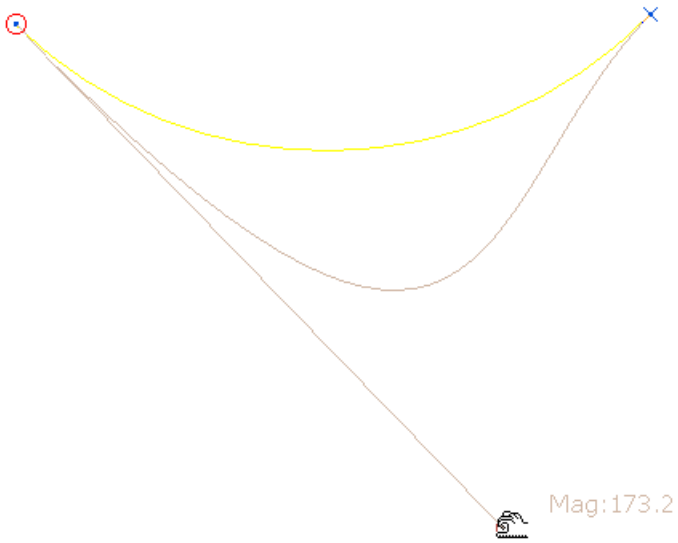
Move the start/end point:



Change the tangent direction:



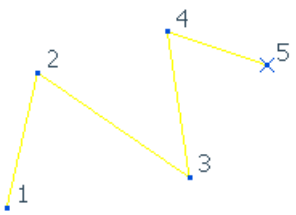
Change the tangent magnitude:



For more information, see Creating curves example (see page 5) and Editing a Bezier curve (see page 6).

Editing continuous lines example

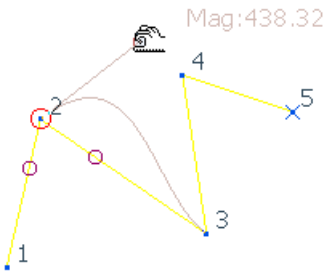
1 Select a continuous line to edit it.



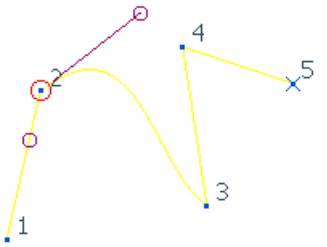
If you aren't in curve editing mode, double click the continuous line to edit it.

2 Select one of the points. The continuous line is yellow, has a red handle at the selected point and two dull red lines with circles at the end, indicating the tangent direction and magnitude.

3 Select one of the tangent handles (the cursor changes to). Dragging it to a new position changes the continuous line to a Bezier curve.



Releasing the mouse button creates a Bezier curve with one curved span and the remaining spans staying as straight lines:



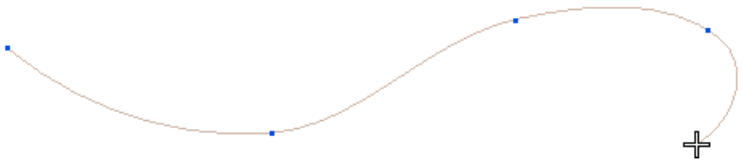
For more information on editing tangent handle, see Editing a Bezier curve (see page 6).

Creating Bezier curves example

This example shows you how to create curves.

1 Click the **Bezier curve** button from the **Curve Editor** toolbar.

2 Either, sketch the curve in the graphics area, or use the coordinates area of the status bar to describe the curve.



The blue points show the points used to create the curve.

3 Double click the final point, press the **Esc** key, or click the first point to exit the curve creation mode.

Clicking the first point creates a closed curve.

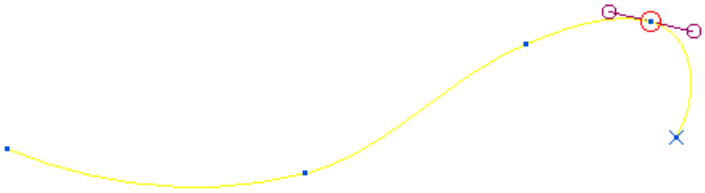
Editing a Bezier curve

This example shows you how to edit a Bezier curve.

1 Select a curve to edit it. The curve turns yellow and the points blue.

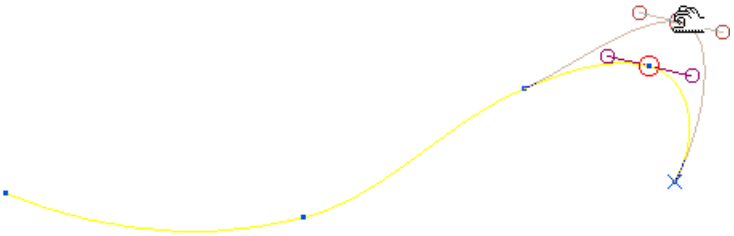
If you aren't in curve editing mode, you won't be able to see the points. In this case, double click the curve to edit it.

2 Select a point on the curve to edit it.

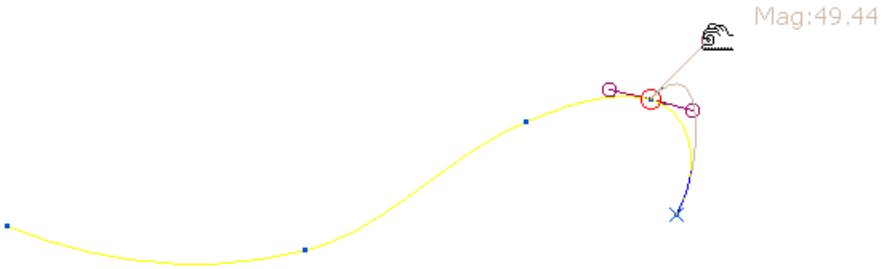


The curve is yellow, has a red handle at the selected point and two dull red lines with circles at the end, indicating the tangent direction and magnitude.

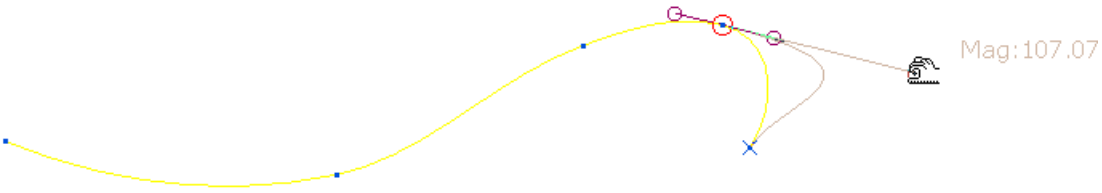
- Selecting the point handle (the cursor changes to) and dragging it to a new position changes the point location and the curve shape.



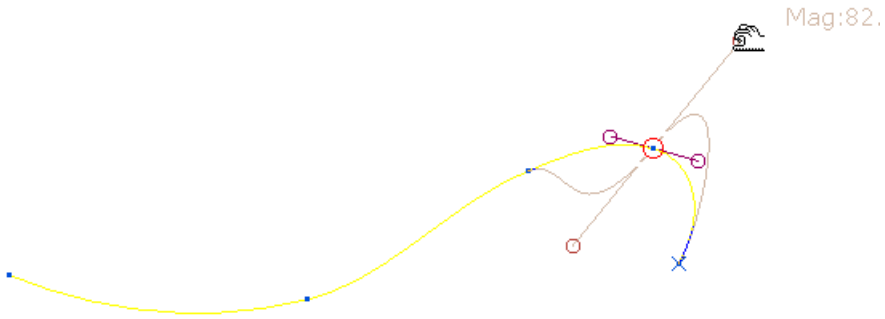
- Selecting one of the tangent handles (the cursor changes to) and dragging it to a new position changes the tangency direction and magnitude of that span.



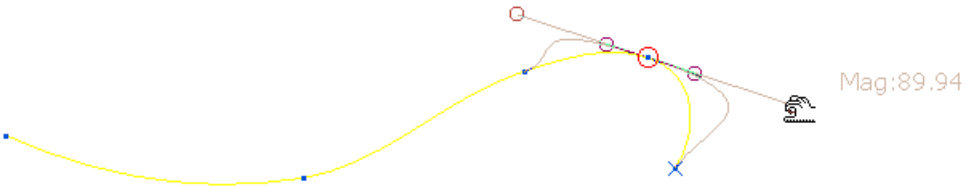
- Using the **Shift** key whilst selecting one of the tangent handles and dragging it to a new position changes the tangency magnitude of a span (but not the direction).



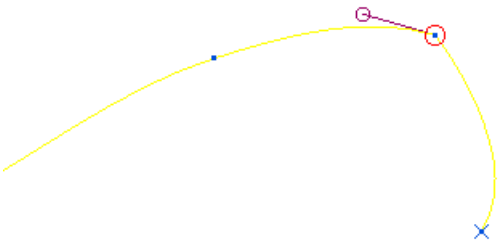
- Using the **Alt** key whilst selecting one of the tangent handles and dragging it to a new position changes the tangency direction and magnitude of both spans.



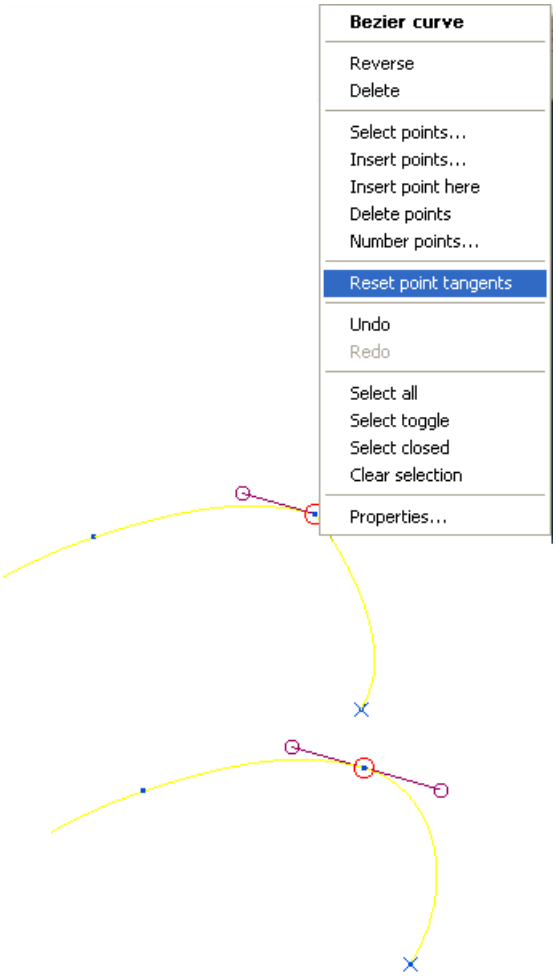
- Using the **Alt + Shift** key whilst selecting one of the tangent handles and dragging it to a new position changes the tangency magnitude of both spans (but not the direction).



- If you edit the tangency magnitude to zero, it is no longer visible.



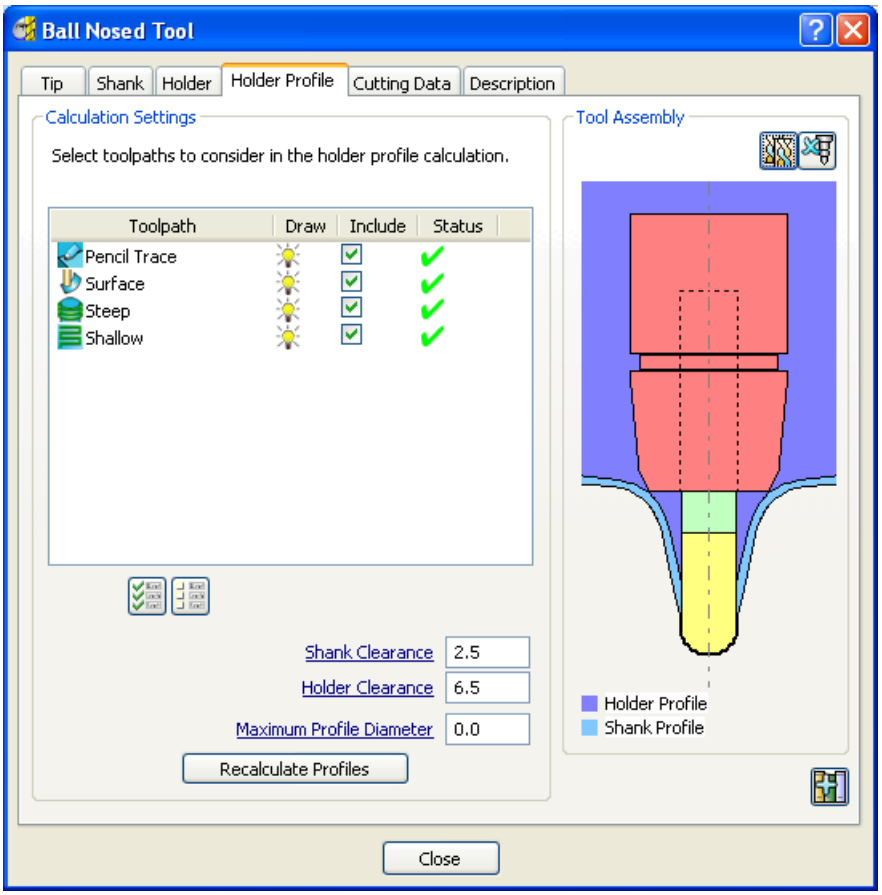
- Restore a zero magnitude tangency by selecting **Reset point tangents** on the curve context menu.



All the point options , , , and work in the same way as for continuous lines.

Tool improvements

There are two enhancements to the **Holder Profile**:



- You can now have different clearances for the holder and shank (see page 9).
- The **Recalculate Profiles** button replaces the **Refresh Profiles** button. The functionality is unchanged, it just has a more meaningful name.

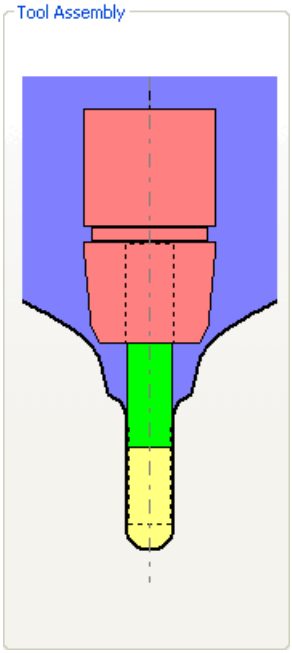
A new **Thread Milling** tool has been added to the PowerMILL tool collection (see page 12).

There are two new tool parameters **tool.Overhang** and **tool.HolderName**.

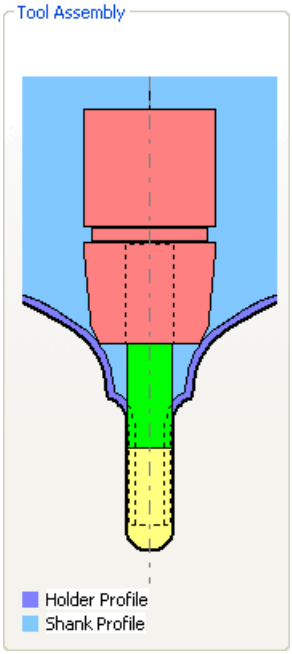
Shank and holder clearance

On the **Holder Profile** tab, you will always see the **Holder Profile** and the **Shank Profile**.

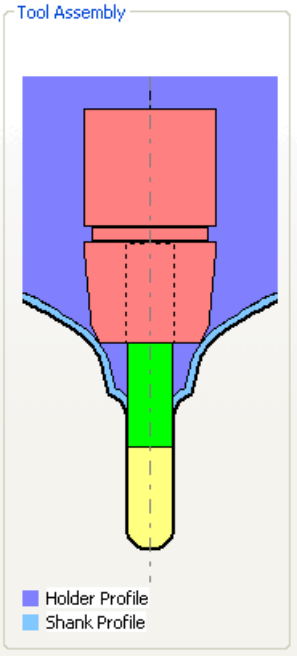
- If the **Holder Clearance** and the **Shank Clearance** are the same, you will see one profile:



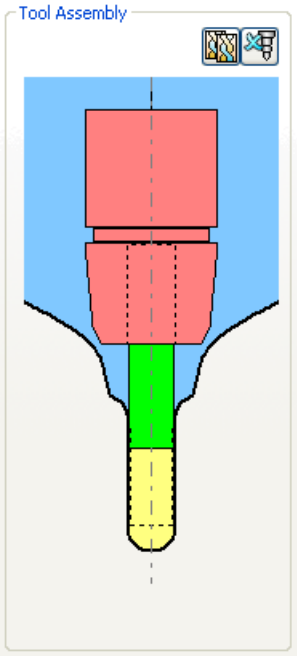
- If the **Holder Clearance** is smaller than the **Shank Clearance**, you will see two profiles:



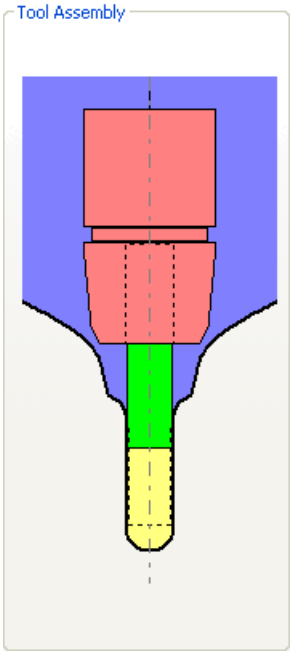
- If the **Shank Clearance** is smaller than the **Holder Clearance**, you will see two profiles:



- If you just want to see the **Shank Profile** select the **Shank** tab.



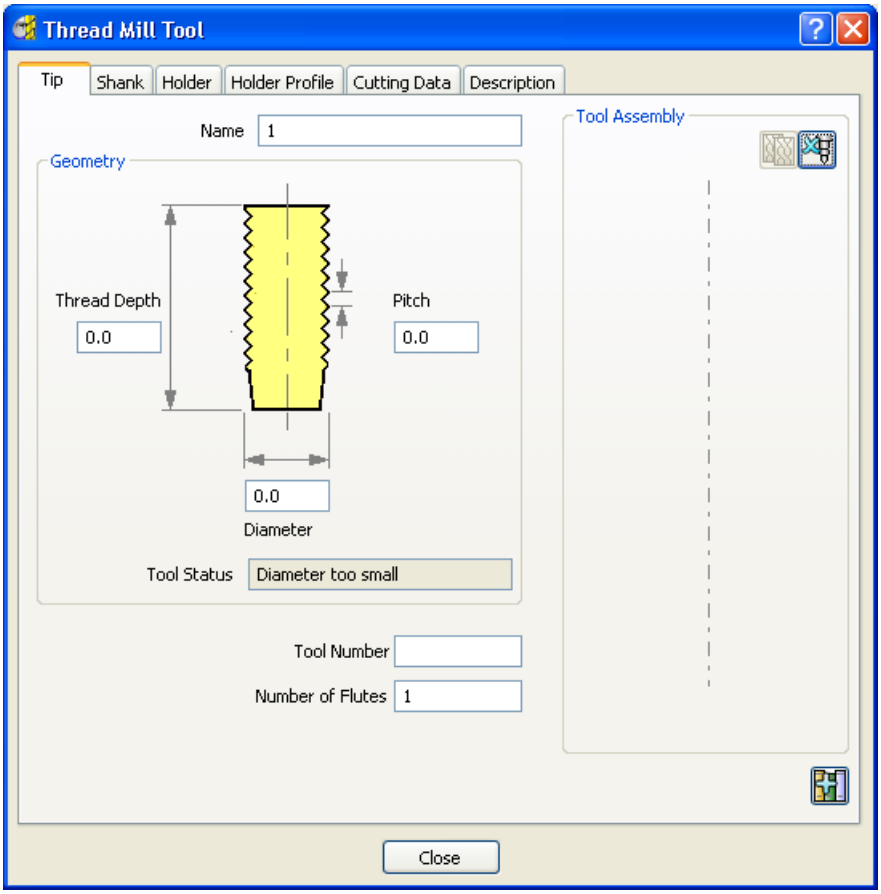
- If you just want to see the **Holder Profile** select the **Holder** tab.



Thread Milling tool

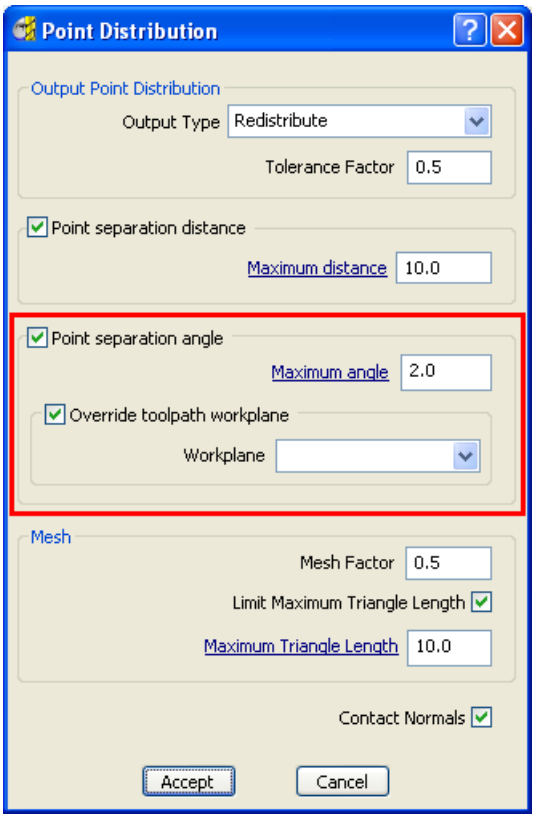
A new **Thread Milling** tool is included in the PowerMILL tool collection.

This tool is very similar to the **Tapping** tool, but with added **Pitch** controls.



Point distribution enhancements

The **Point Distribution** dialog now allows you to manage the tool axis angular change. This enables you to smooth and slow the motion of the machine tool when undergoing sharp angular changes.



Maximum angular separation - maximum angle between consecutive toolpath points. This is particularly useful when approaching the gimbal lock position of your machine tool where very small angular changes lead to massive changes in the azimuth angle. Entering an angle here slows the machine tool down by adding many more points.

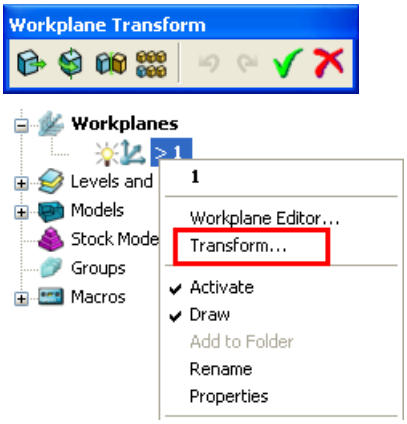
*These options are only available if you select an **Output type** of **Redistribute**.*

Override toolpath workplane - use a different workplane to the workplane used to generate the toolpath to define elevation and azimuth.

Workplane - the workplane used when calculating the angle. If no workplane is selected, the global coordinate system is used.

Workplane enhancements

The individual workplane context menu has an additional option of **Transform** which displays the **Workplane Transform** toolbar.



Workplane transformations allows you to move, rotate, or mirror a workplane. You can easily create complex rectangular and circular workplane arrays. PowerMILL displays a preview of the transformation which gives a quick visual check before transforming the workplane.

Move - transforms the workplane by the specified coordinates.

Rotate - rotates the workplane around the origin of the selected workplane by the selected angle. The **Twist** option rotates the workplane about itself. This option allows you to rotate the workplane about another workplane. This is also available on the **Workplane Editor** toolbar.

Mirror - mirrors the workplane along one of its principal planes or along an arbitrary mirror line. When mirroring a workplane, the Y axis is always flipped to ensure the creation of a mathematically correct axis. This is also available on the **Workplane Editor** toolbar.

Multiple transform - an easier method of performing multiple **Moves** or **Rotations** .

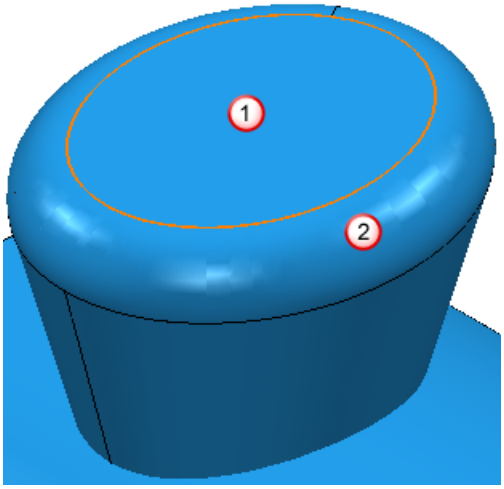
These options work in a very similar way to the same options on the **Curve Editor** toolbar.

Pattern enhancements

There is a new option of **Select Surfaces** on the individual **Pattern > Edit** menu. This selects the surfaces that any segments of the pattern are embedded in.

You can use the **Pattern > Edit > Select Surfaces** and **Pattern > Edit > Embed** options to re-embed a pattern when:

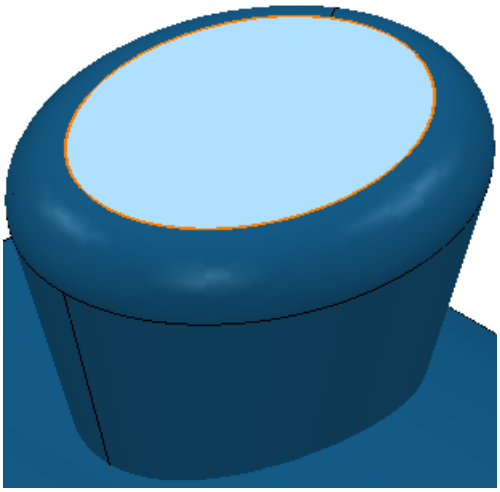
- The original surfaces are no longer in the model.
- The pattern is embedded onto the wrong surface (this can happen when the pattern lies at the junction of two different surfaces or where two surfaces are close together).



In this case, the embedded pattern could be embedded in surface 1 or 2.

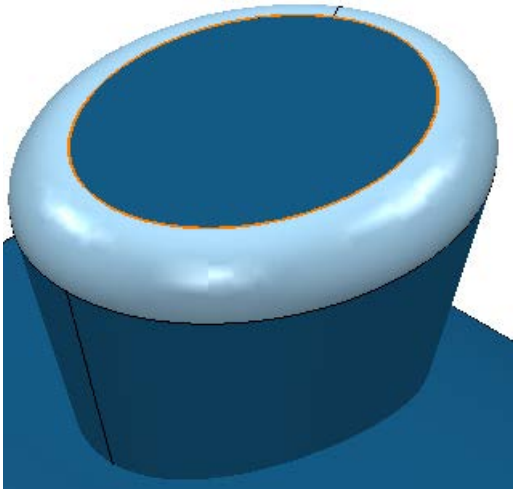
This example shows you which surface the pattern is embedded into and how to change the embedded surface.

1 From the individual pattern context menu, select **Edit > Select Surfaces**.

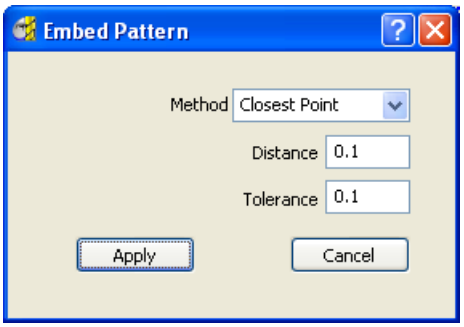


You can see that the pattern is embedded onto surface 1.

2 Select surface (2).



3 From the individual pattern context menu, select **Edit > Embed...**. This displays the **Embed Pattern** dialog.



a Select a **Method** of **Closest Point**.

b Click **Apply**.

This creates a new pattern which is embedded onto surface (2).

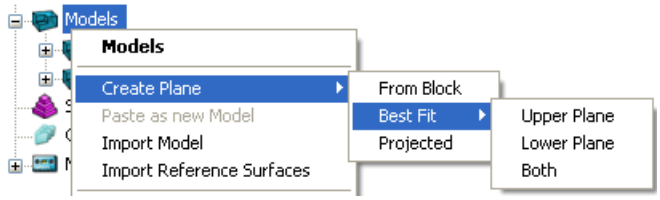
To check this new pattern is embedded onto surface (2):

1 Deselect the surface by clicking the background.

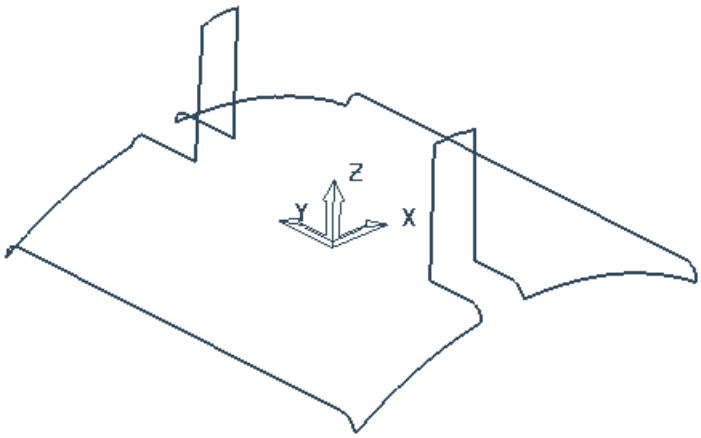
2 From the individual pattern context menu, select **Edit > Select Surfaces**.

Best fit plane

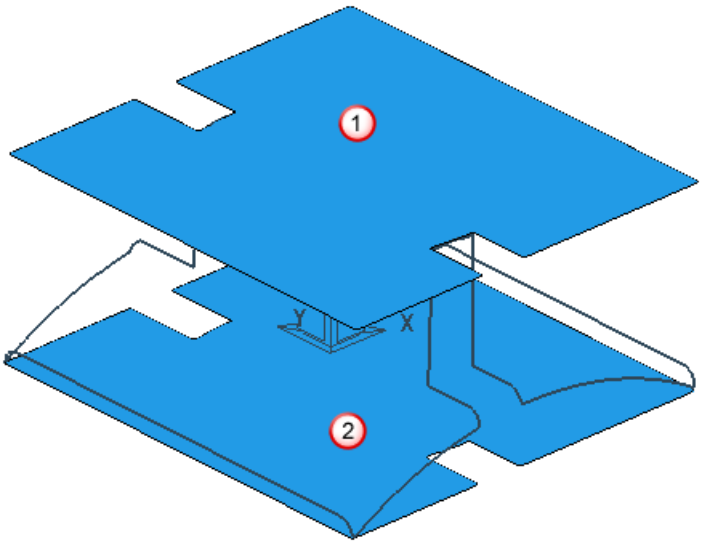
When creating planes from a boundary using the **Models > Create Plane > Best Fit** context menu, you now have the option as to whether you want to create the upper, lower or both planes. In previous versions **Both** was the only option.



This boundary:



Produces these best fit planes:



1 - Upper plane

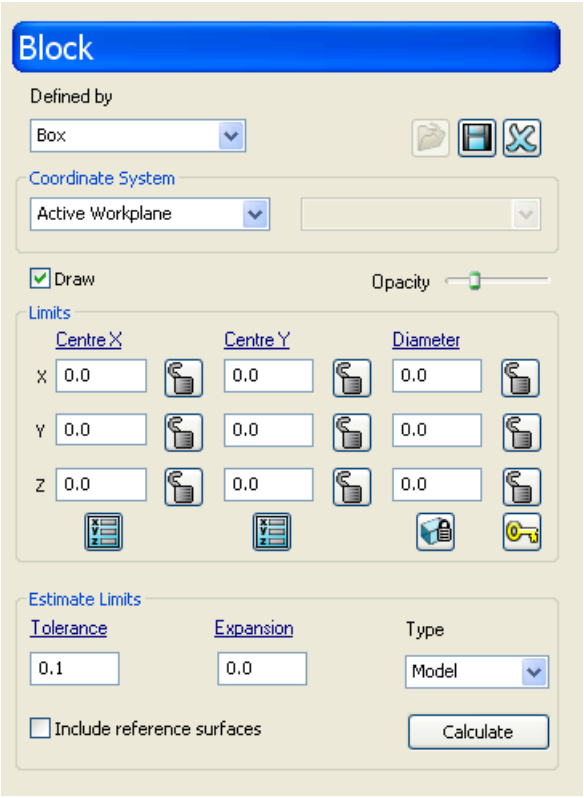
2 - Lower plane

Toolpath generation

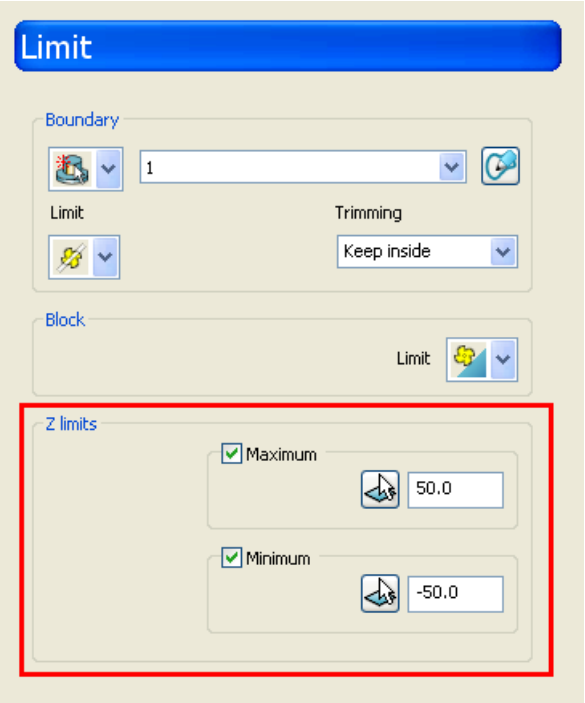
Machining strategy enhancements

Strategy dialogs now contain all items needed to create a toolpath. So you can now create a toolpath by working your way through the pages of a strategy dialog without having to raise any other dialogs such as **Block**, **Tool**, and so on.

The **Block** page now displays the standard **Block** dialog rather than just summarising the block shape, size and orientation, and having a link to the **Block** dialog.



The **Limit** page now allows you to specify a Z range for the cutting moves. These limits work within the defined block.



Maximum - enables you to select a maximum Z height. Cutting moves will always be below this value.

Z Height  **-** select the Z height graphically.

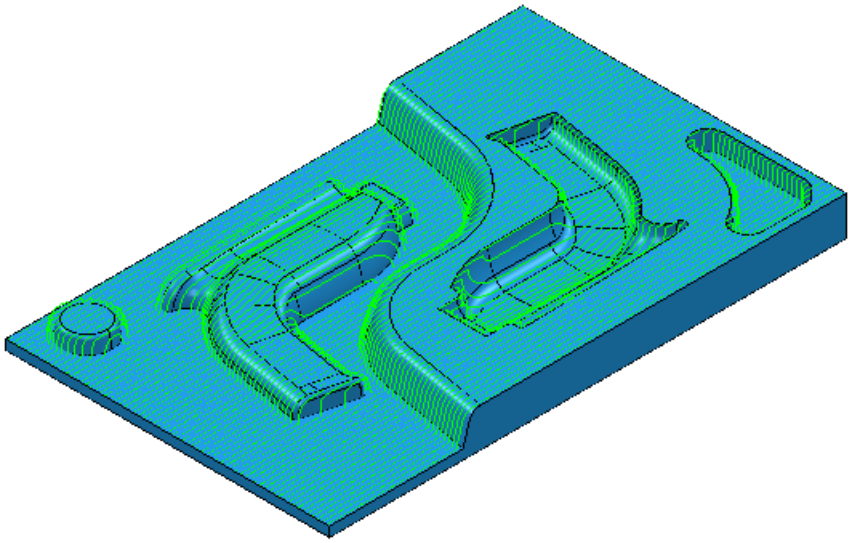
 - enter the Z height.

Minimum - enables you to select a minimum Z height. Cutting moves will always be above this value.

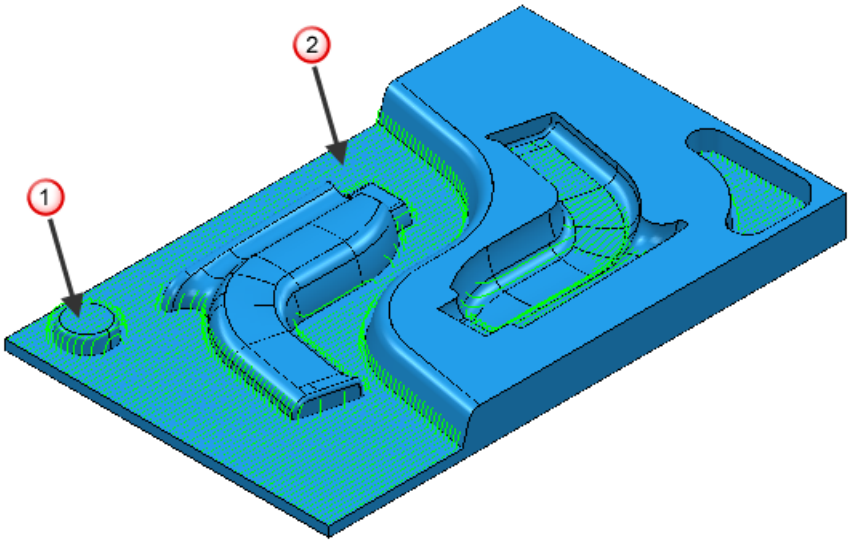
Z Height - select the Z height graphically.

- enter the Z height.

An unlimited toolpath:



A Z limited toolpath



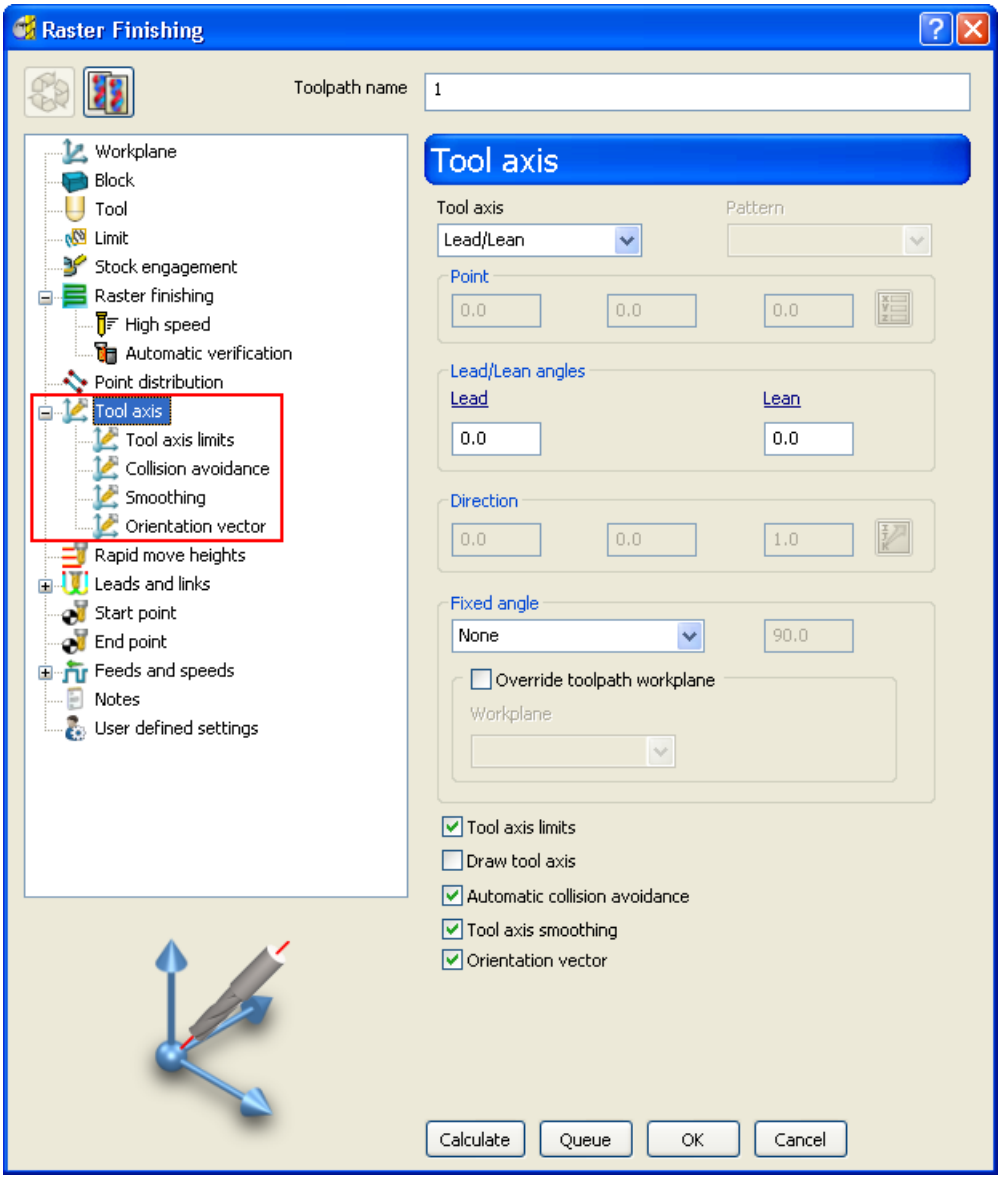
In this case, the limits are selected from the model.

1 - Z maximum

2 - Z minimum

You can Z limit any machining strategy, except for drilling strategies where the depth is controlled by the **Operation**.

The **Tool Axis** page now displays the standard **Tool Axis** dialog rather than just summarising the tool axis and having a link to the **Tool Axis** dialog.

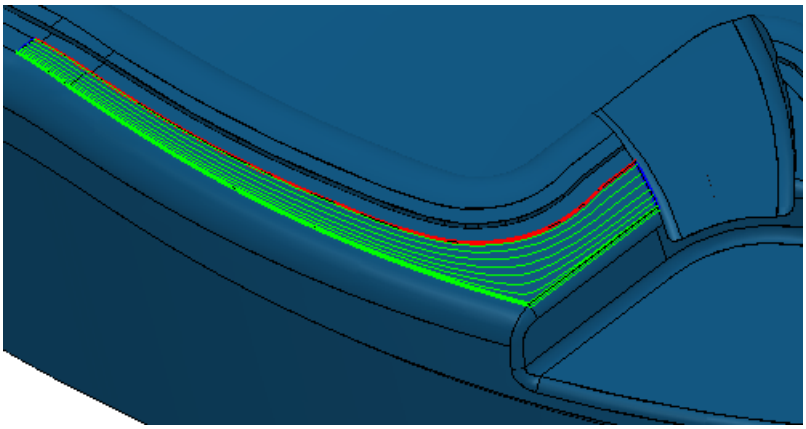


*The **Tool axis limits**, **Collision avoidance**, **Smoothing**, and **Orientation vector** pages are only available if you select the option at the bottom of the main **Tool axis** page.*

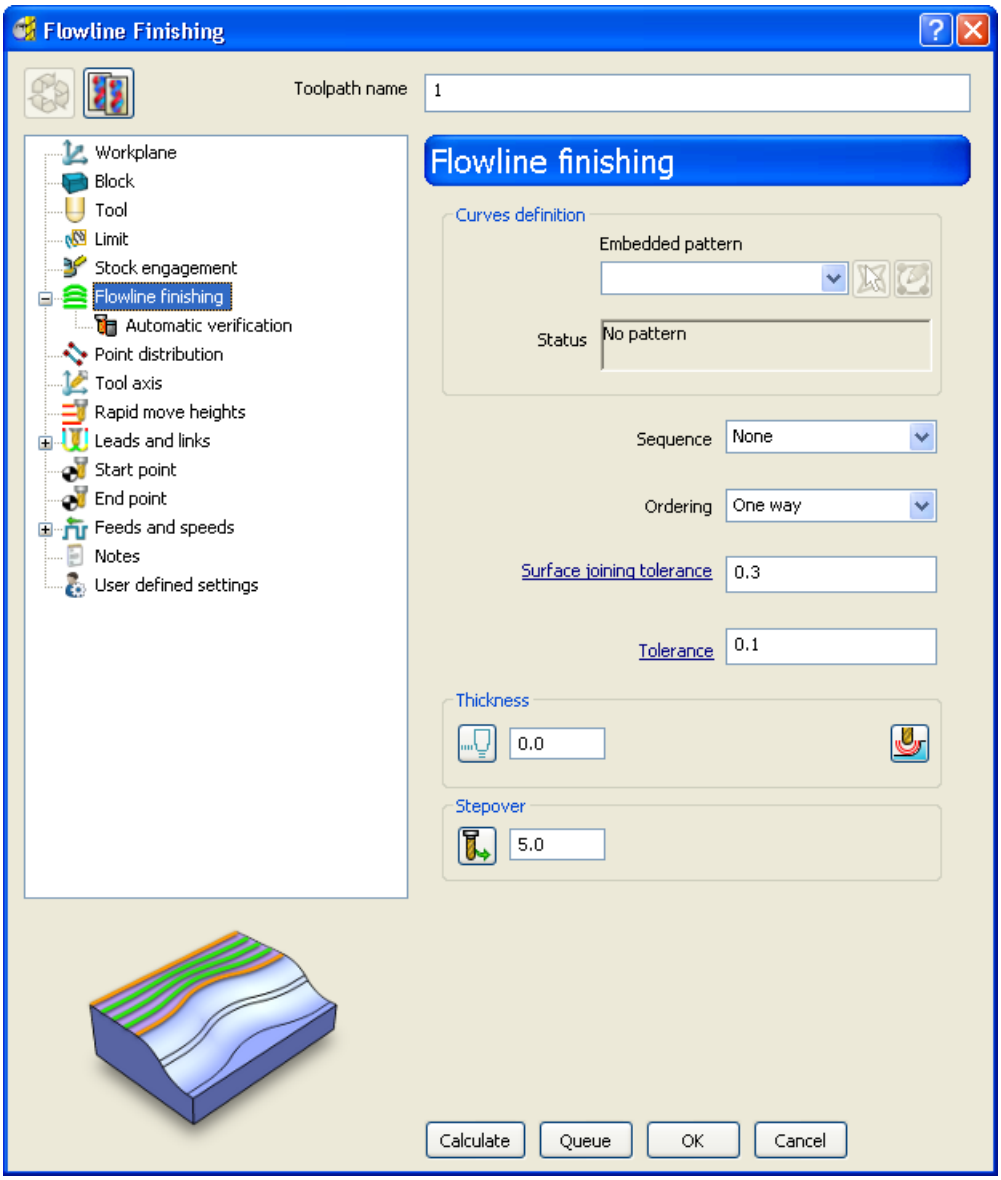
Flowline Finishing - Overview

Flowline Finishing is a 5-axis, multi-surface strategy, which creates a toolpath between two curves, called drive curves, by smoothly blending between them. Additional limiting curves may be necessary to fully define the machining area. The curves must lie on a surface (or surfaces).

This strategy enables you to machine multiple surfaces independently of their surface parameterisation. It is typically used on complex fillets or gently curved surfaces.



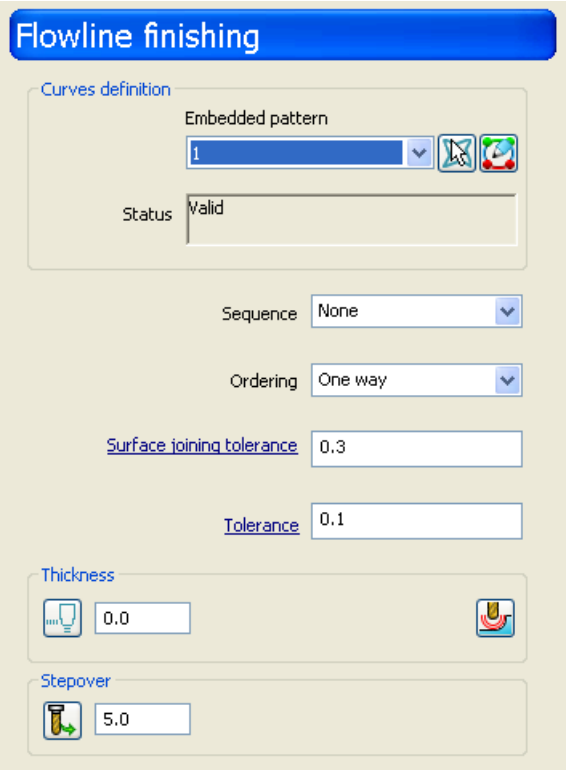
The pages associated with **Flowline Finishing** strategy are:



- **Flowline finishing -** the main page used to define a flowline toolpath.
- **Automatic verification -** enables automatic verification of toolpaths on creation.

Flowline finishing

Flowline finishing creates a toolpath from an embedded pattern. The pattern must contain several segments, typically two drive curves and two limit curves.



Curve definition - specifies the curves bounding the area you want to machine and adds the machining properties to those curves.

Selected pattern - displays the selected pattern. If no pattern is displayed, or <None> is selected, then no pattern is selected. The list contains a list of all available patterns. Since flowline toolpaths are created from embedded patterns, only the embedded patterns are listed here.

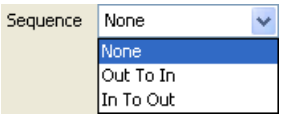
Select picked pattern - selects and activates a pattern by picking in the graphics window, rather than by name in the **Select pattern** list.

Curve definition (see page 28) **-** specifies:

- the drive curves,
- the limit curves,
- any intermediate curves,
- the machining strategy.

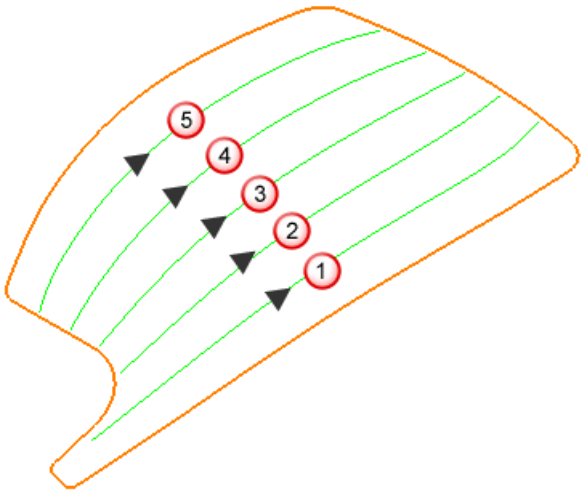
Status - summarises the current status of the selected curves, and is updated automatically as curves are selected. You need to reselect curves if **Valid** isn't displayed.

Sequence - allows you to alter the sequence of toolpath segments.



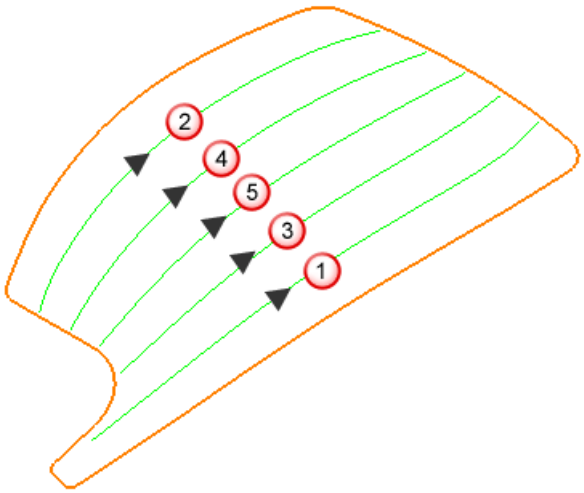
None - the raster type movement is maintained.

With an **Ordering** of **One way**:



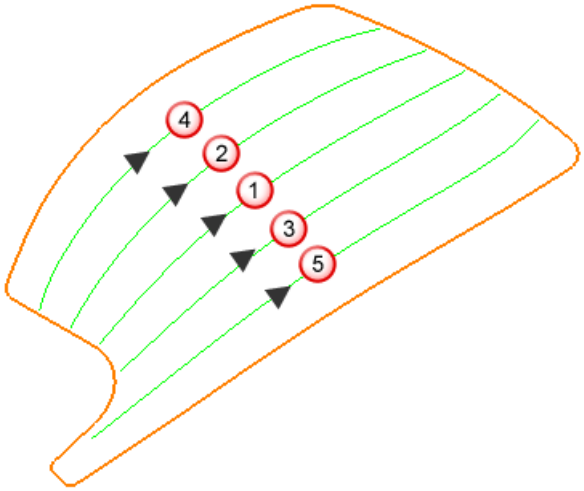
Out to In - the toolpaths are transformed from a raster type movement to a constant Z or spiral type movement.

With an **Ordering** of **One way**:



In to Out - the toolpaths are transformed from a raster type movement to a constant Z or spiral type movement.

With an **Ordering** of **One way**:



Ordering - determines how each successive toolpath segment is linked.

One way - the tool can only cut in one direction.

Two way - the tool can cut in both directions.

For more information, see Raster ordering.

Surface joining tolerance - disassociates the machining tolerance from the tolerance used to define what is a gap between surfaces. If the gap between surfaces is larger than the machining tolerance, PowerMILL creates two toolpath segments. To ensure one continuous toolpath across a gap, use a larger **Surface joining tolerance**.

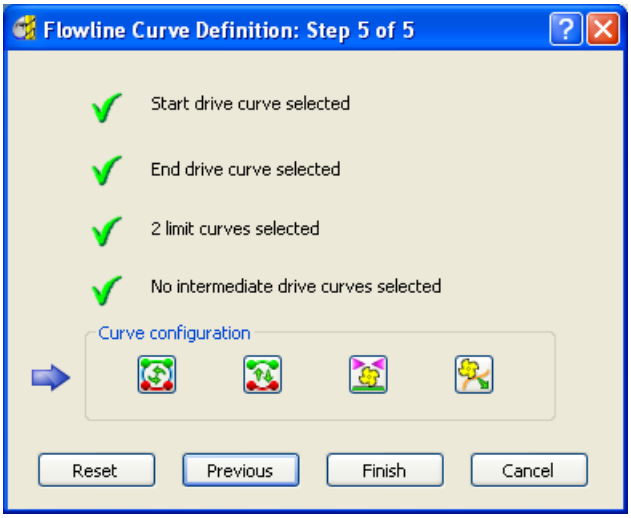
Tolerance - determines how accurately the toolpath follows the contours defined by the model.

Thickness - specifies the amount of material to be left on the part, within tolerance.

*To avoid too much material being removed, **Thickness** should be greater than **Tolerance**.*

Stepover - the distance between successive machining passes.

Curve definition



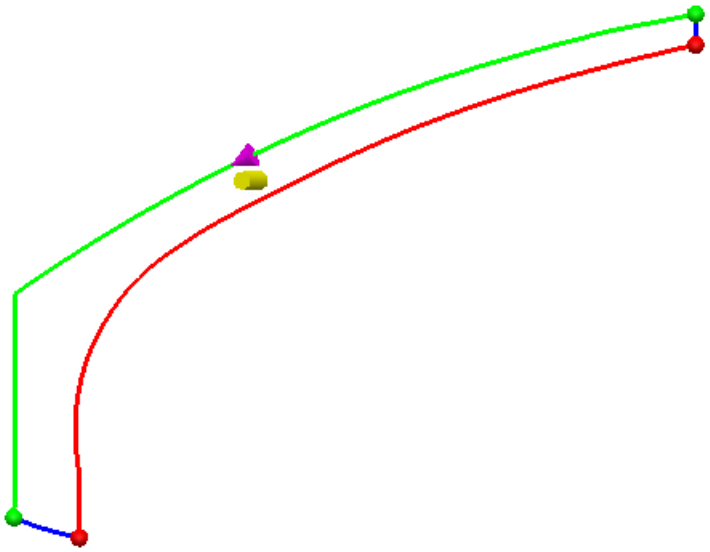
PowerMILL will try to configure the curves. So step 5 is the most likely step to be displayed.

- If the configuration is correct, click **Finish**.
- If the configuration is incorrect, then amend it using the **Curve configuration** buttons.
- If incorrect curves are selected, click **Previous** to select the correct curves.

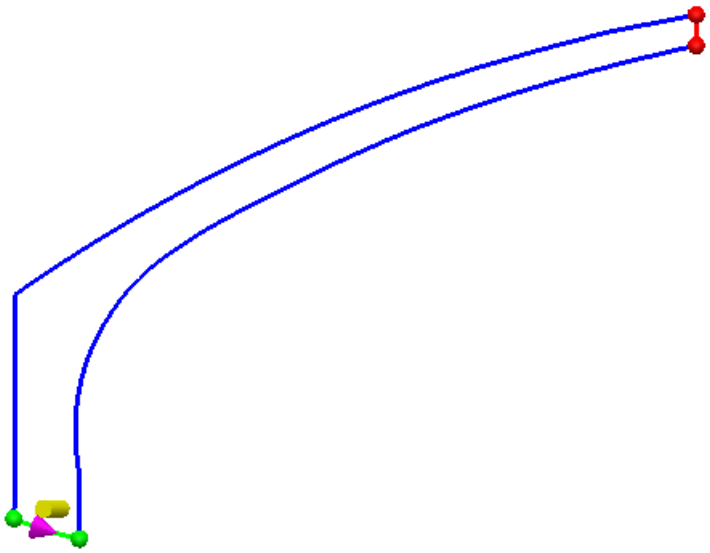
For more information, see the Flowline finishing example (see page 32).

Cycle the drive and limit curves - cycles through the different combinations of drive and limit curves.

Converts this:



to this:



Key:

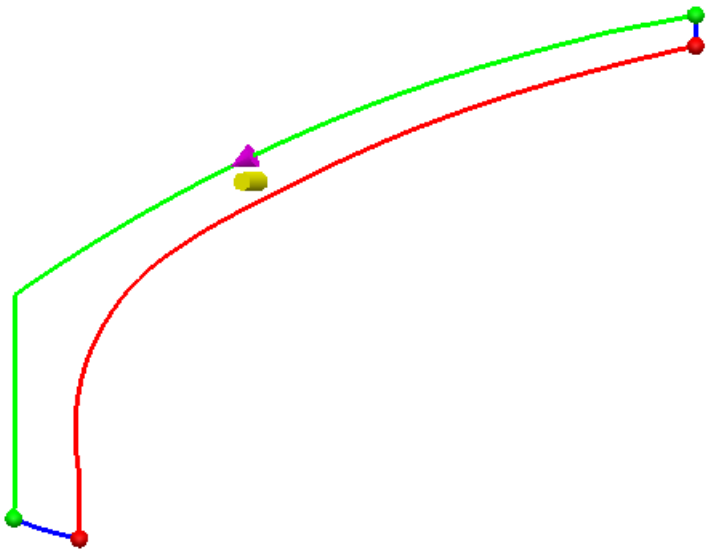
- the start drive curve.

-the end drive curve.

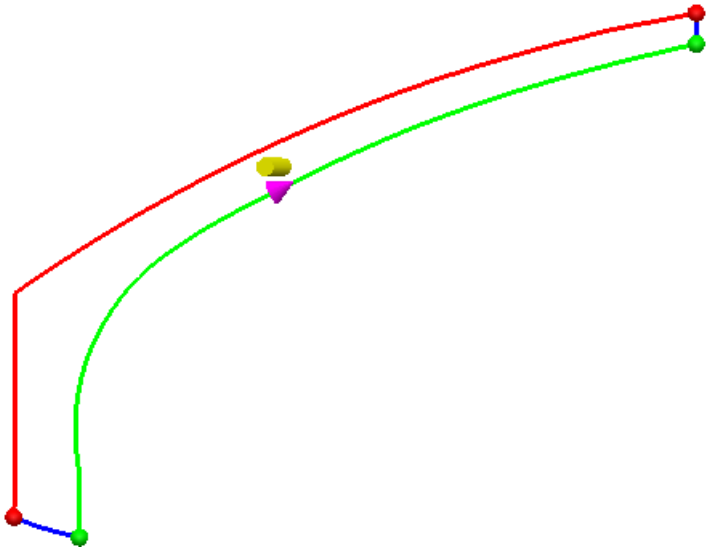
- the limit curves.

Reverse drive curves - swaps the start and end drive curve.

Converts this:

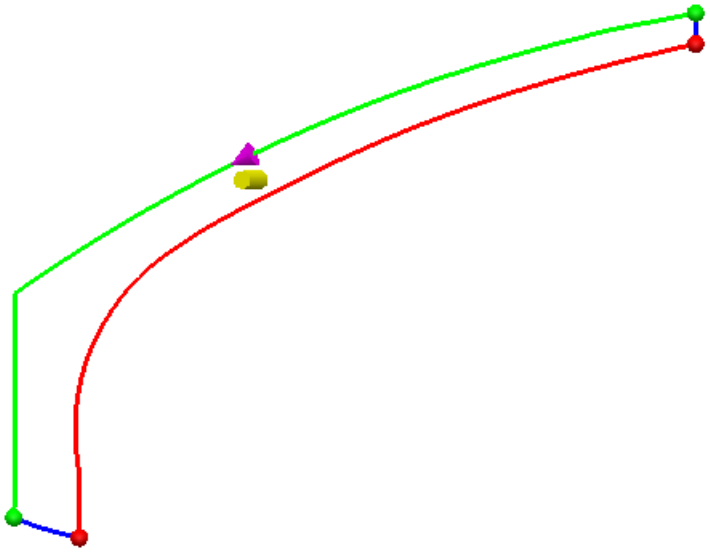


to this:

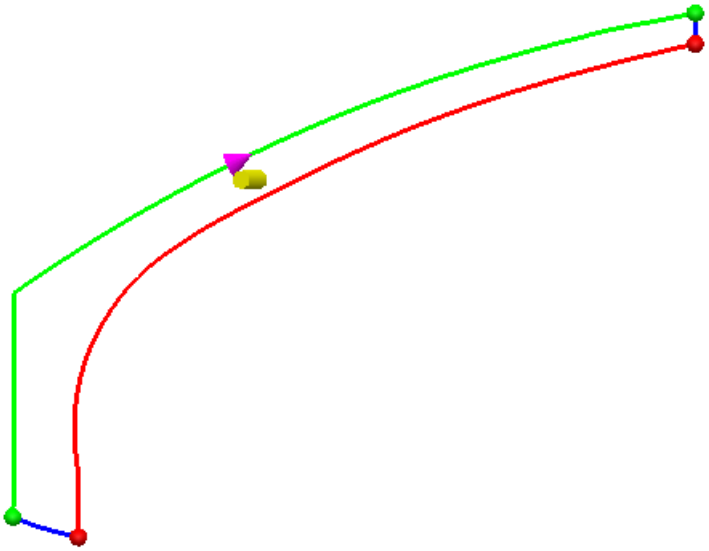


 **Change cut direction -** changes the cut direction from conventional to climb milling or from climb to conventional milling.

Converts this:

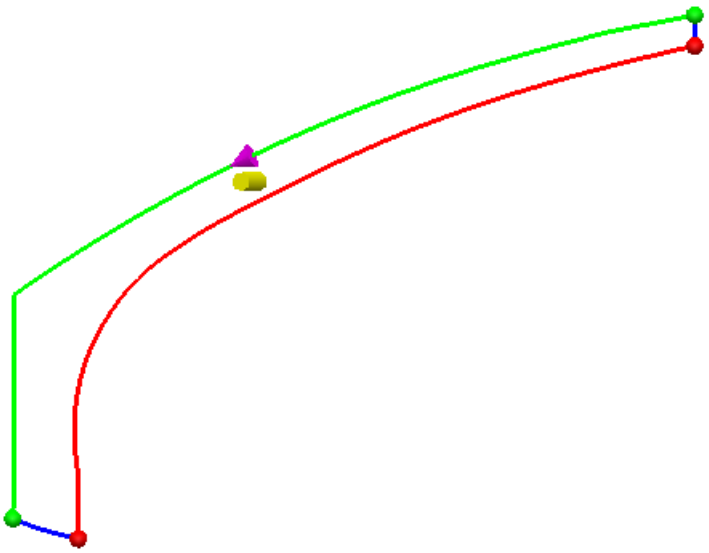


to this:

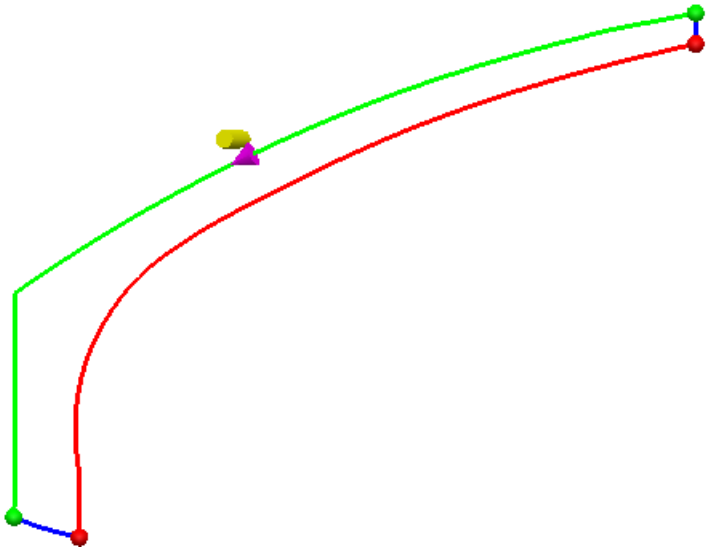


 **Reverse side to machine -** reverses the side of the curve which is machined.

Converts this:

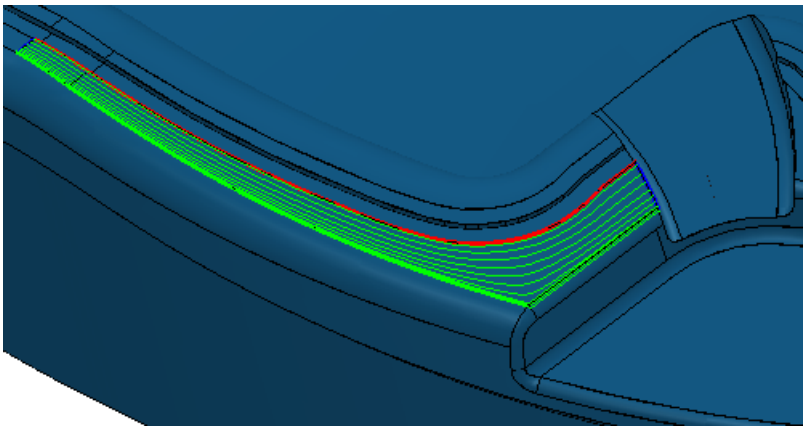


to this:

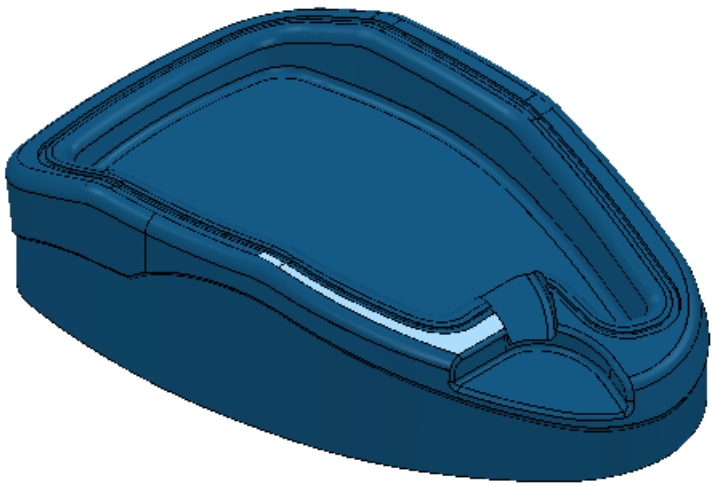


Flowline finishing example

This example shows you how to create a flowline toolpath. It uses the **5axisModel.dgk** in the **Examples** folder with a block and a 5 mm ball nosed tool.

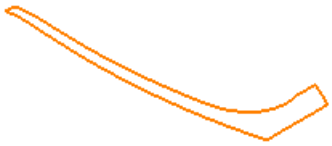


1 Select the two highlighted surfaces.



2 On the **Pattern** toolbar, click the **Create pattern** button .

3 On the **Pattern** toolbar, click the **Insert model into active pattern** button .

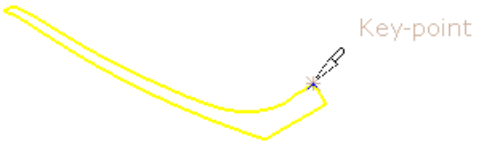


This creates an embedded pattern at the edge of the selected surfaces.

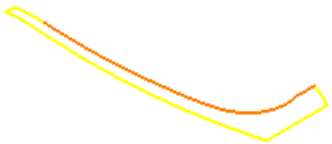
4 The pattern currently contains one curve and needs to be broken into four curves. On the **Pattern** toolbar, click the **Curve editor** button . This displays the **Curve editor** toolbar.

5 On the **Curve editor** toolbar, select the **Cut item** button from the **Cut** toolbar  .

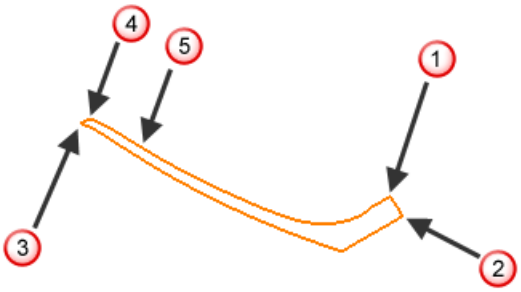
6 Select the curve where you want to cut it. The cursor changes to a knife and the curve turns yellow.



7 Click the curve again to actually cut it and see two distinct curves.

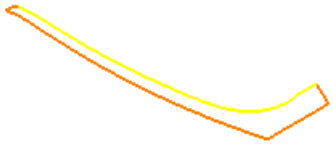


This breaks the curve at 1.



8 Repeat steps 7 and 8 to break the curve at 2, 3, and 4.

9 The pattern is still split at 5. To merge the two segments, select the two segments.



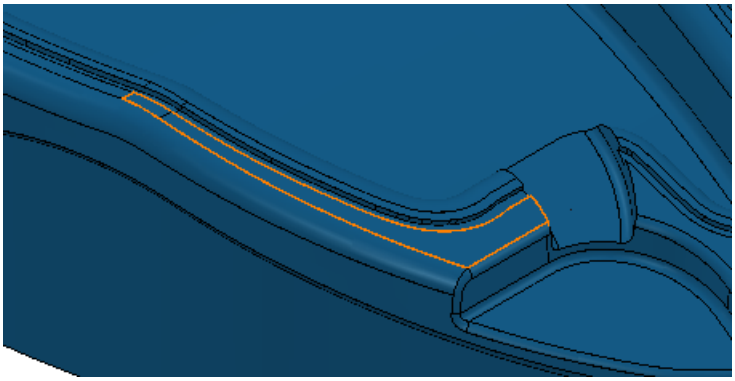
10 Select the **Merge selected segments** button from the **Cut** toolbar  .

A PowerMILL information dialog states:

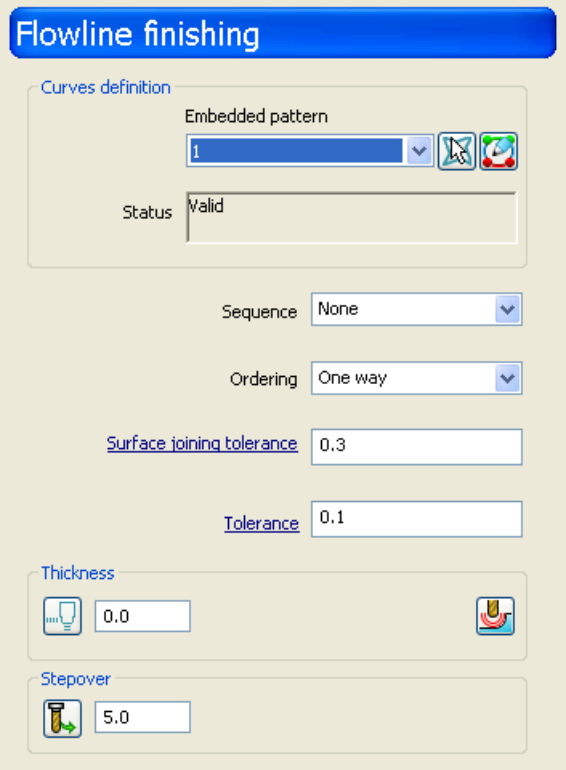
Geometry reduced from 5 to 4 pieces. 0 Closed, 4 Open. Reduced selected curves from 2 to 1.

11 Click **OK** to close the **PowerMILL information** dialog.

12 On the **Curve editor** toolbar click to accept the changes.

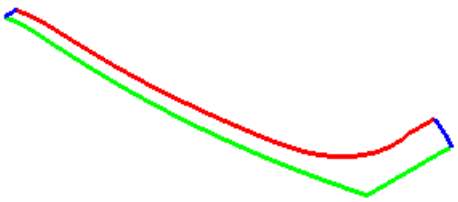


13 From the **Strategy Selector**, select the **Finishing** tab and select **Flowline finishing** strategy.



a Select the embedded pattern.

The **Status** says **Valid** and the pattern changes colour. If this isn't the case, see Defining the flowline curves (see page 37).



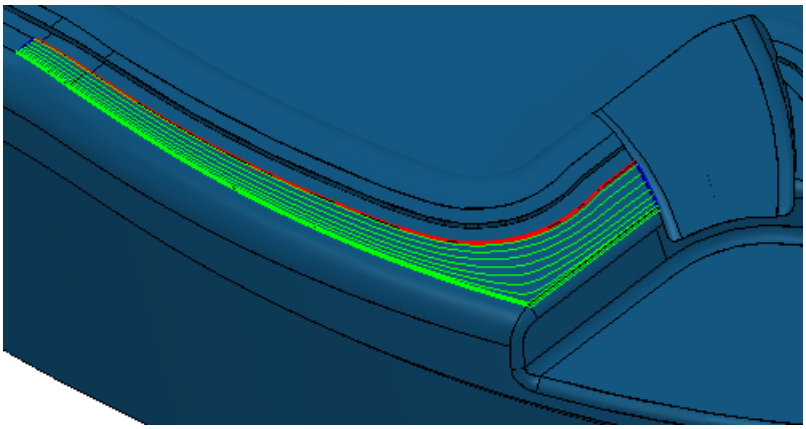
The green curve is the start drive curve.

The red curve is the end drive curve.

The blue curves are the limit curves.

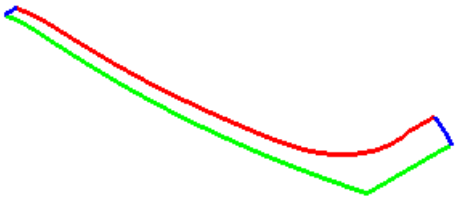
b Enter a **Stepover** of **3**.

c Click **Calculate**.



Curve definition for flowline toolpaths

In many cases, when you select an embedded pattern for flowline finishing toolpaths, PowerMILL automatically creates the correct curve definition.



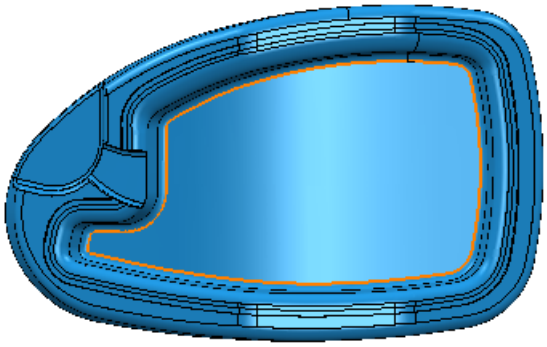
However, there are times when either:

- The curve definition isn't what you want.
  - Automatic curve definition isn't possible (see page 37).
  - You may want to re-order the limit and drive curves, or add in extra drive curves.
  - The curves may contain more surfaces than you expect.
- The curves in the embedded pattern aren't valid. This may be because:
  - The pattern isn't embedded.
  - The embedded pattern doesn't contain the correct number of curves.
  - The curves in the embedded pattern are self intersecting.
  - The curves in the embedded pattern don't meet within the **Closure tolerance**.

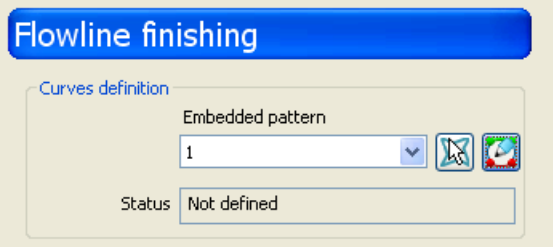
In many instances, you require four curves to create a flowline toolpath, but not always. There is no correct number of curves required to create a flowline toolpath (see page 39). There must be at least two.

Defining the flowline curves

If PowerMILL can't work out the drive or limit curves then you can select them. This example uses the **5axisModel.dgk** in the **Examples** folder and assumes you have created an embedded pattern.



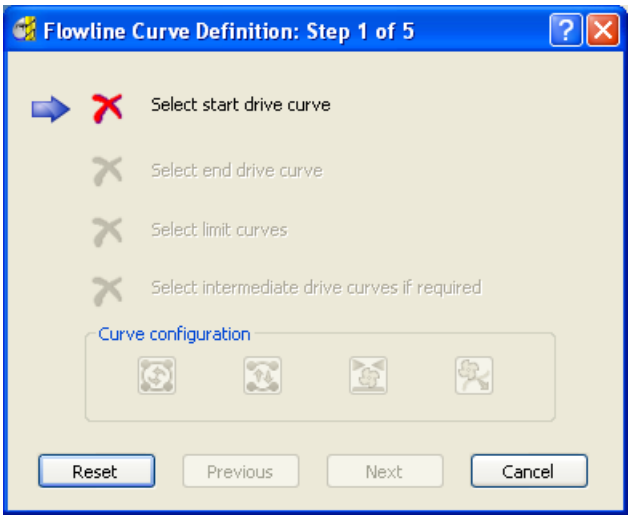
1 From the **Strategy Selector**, select the **Finishing** tab and the **Flowline finishing** option.



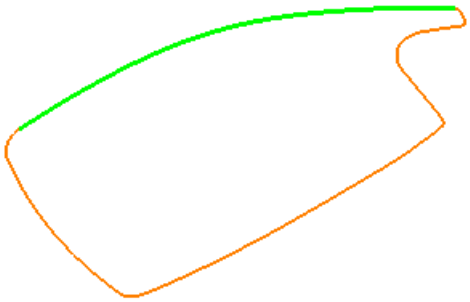
2 Select the embedded pattern.

The **Status** says **Not defined**.

3 Click on the **Curve definition** button . This displays the **Flowline Curve Definition** dialog.

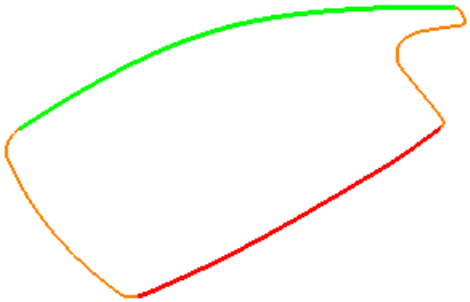


4 Select the start drive curve.



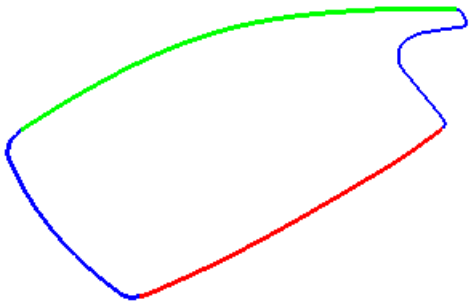
The start drive curve is green.

5 Select the end drive curve.



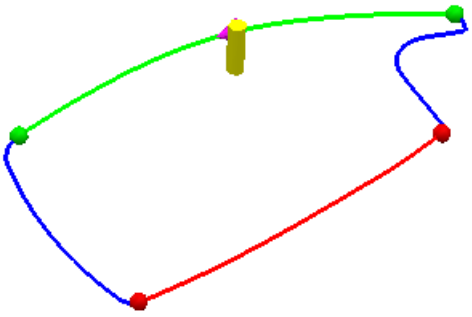
The end drive curve is red.

6 Select a limit curve.



The limit curve is blue.

7 Select the second limit curve (use the **Shift** key whilst selecting it).



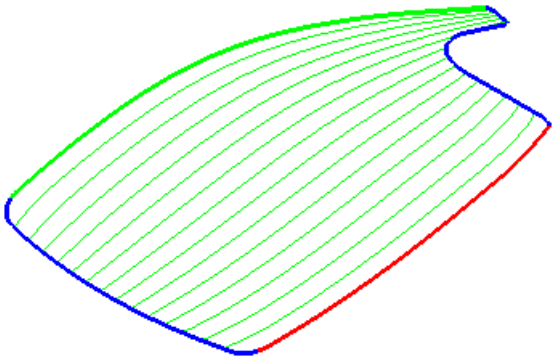
The second limit curve is blue and the cut direction and machining area are also defined.

8 Click **Finish**. This closes the dialog and returns you to the **Flowline Finishing** dialog.

9 On the **Flowline finishing** page:

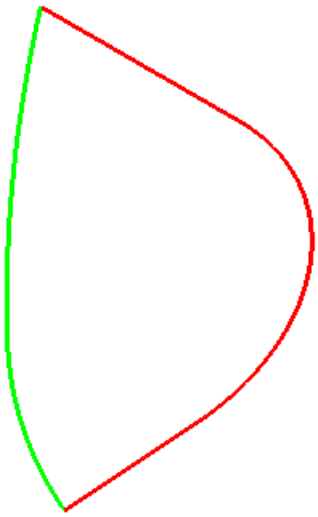
a Enter a **Stepover** of **10**.

b Click **Calculate**.

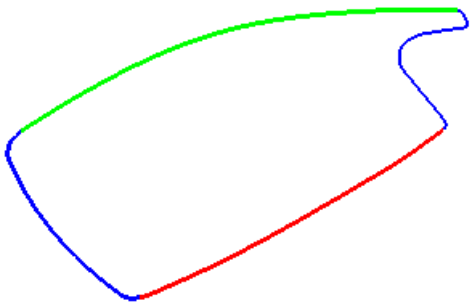


Correct number of curves for flowline toolpaths

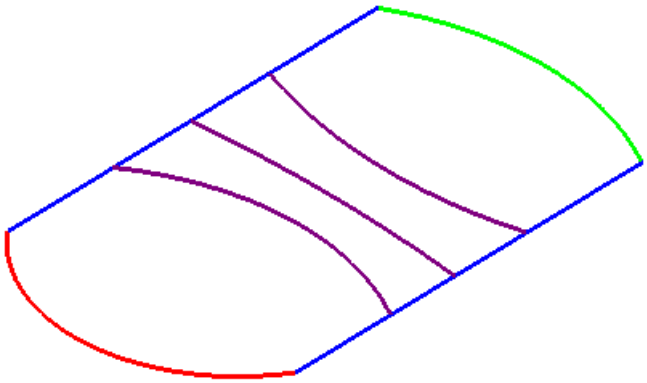
For flowline toolpaths, you must have a start drive curve and an end drive curve.



You frequently have one or two limit curves.

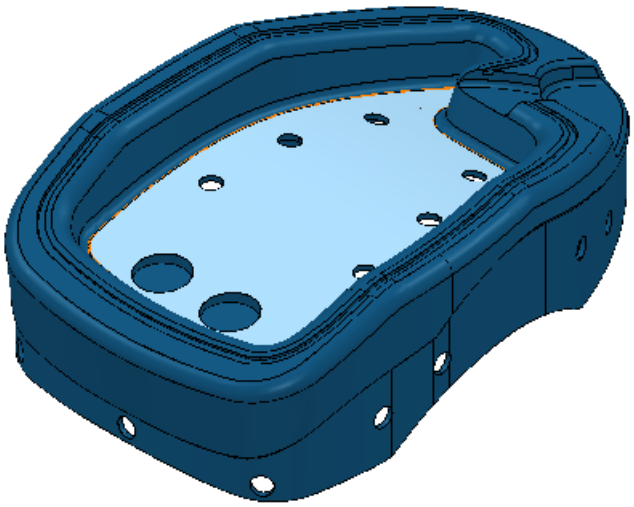


Occasionally you may have one, or more, intermediate drive curves.



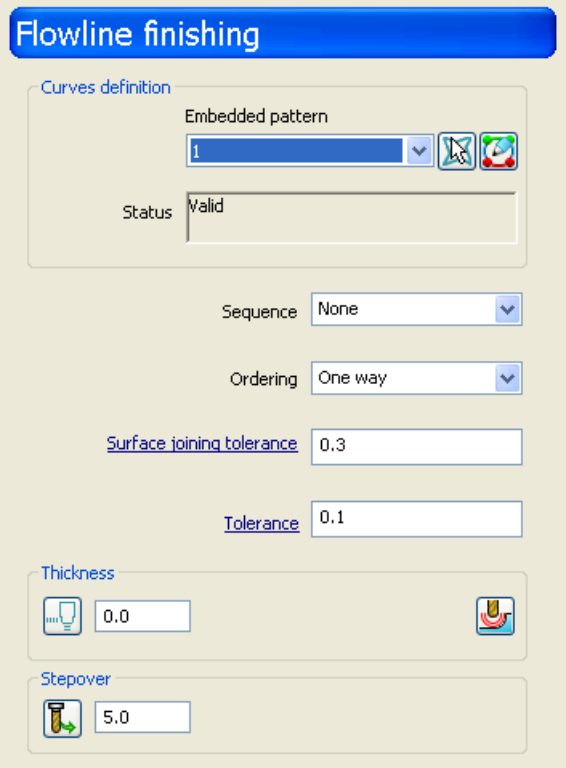
Flowline toolpaths and models with holes

When creating flowline toolpaths the area inside the embedded pattern is machined. However, if you have holes in the model, this area may not be what you expect. This example uses the **5axis_with_holes.dgk** model in the **Examples** folder with a block and a 5 mm ball nosed tool.



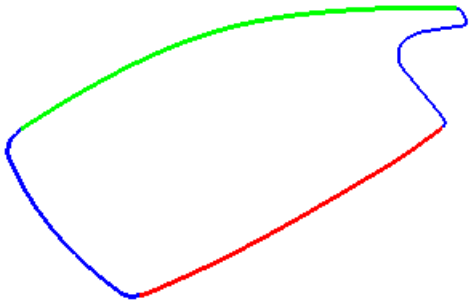
To create a flowline toolpath on the selected surface:

1 From the **Strategy Selector**, select the **Finishing** tab and select **Flowline finishing** strategy.

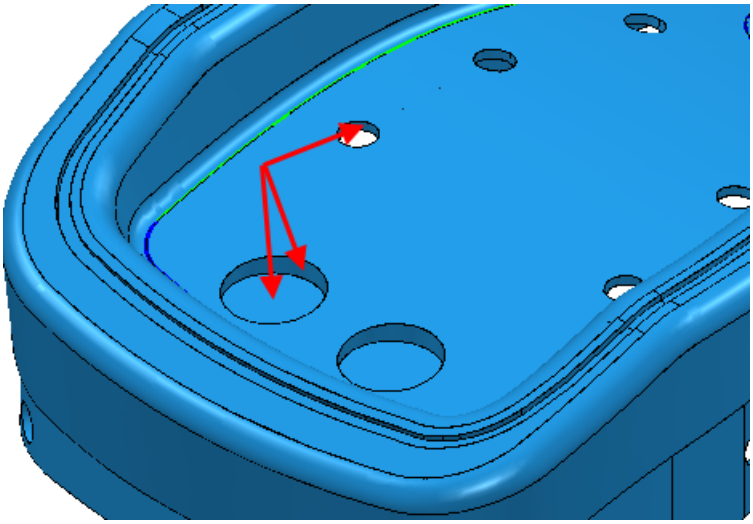


a Select the embedded pattern.

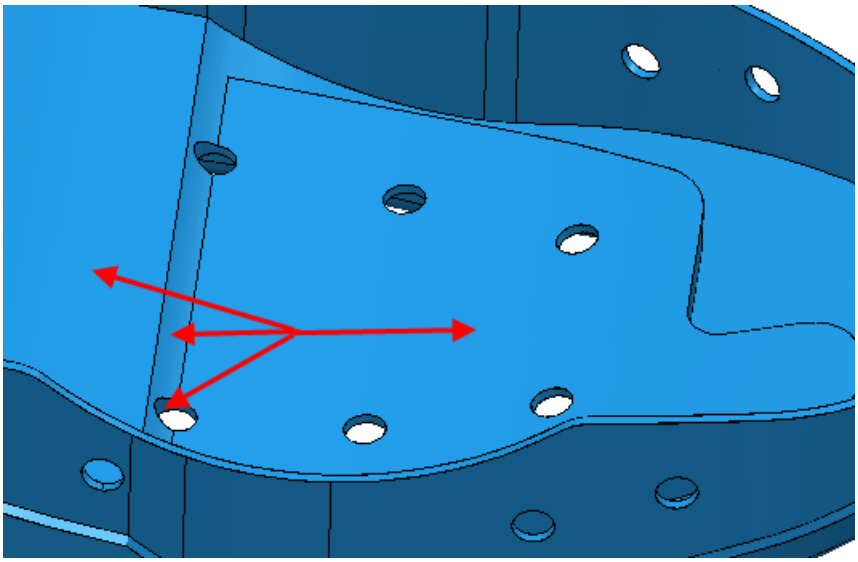
b The **Status** says **Valid** and the pattern changes colour.



However, if you now try and **Calculate** this toolpath you won't get your required toolpath. PowerMILL creates a toolpath on all surfaces inside this pattern. So the surfaces in the holes are also considered.



Since these surfaces have other surfaces next to them, they are also considered.

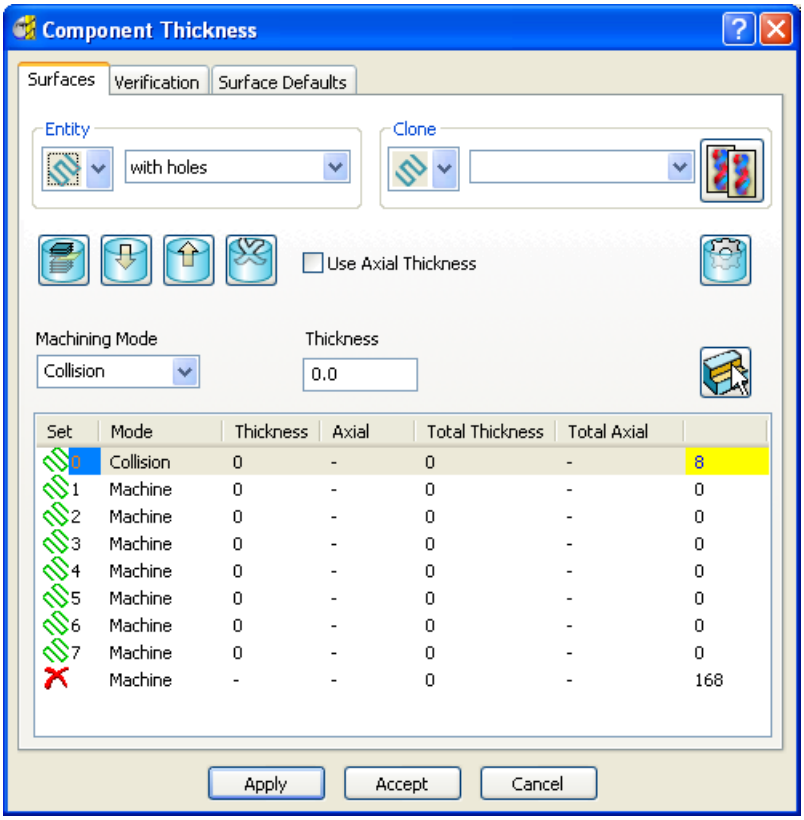


And so on, over the whole model.

To avoid this problem, the cylindrical surfaces need to be ignored when machining.

1. Create a batch flowline toolpath by defining the curves and then click **OK**.
2. Select all the cylindrical surfaces.

3 From the **Thickness Preferences** button  select the **Surfaces** tab, and select the top thickness set.



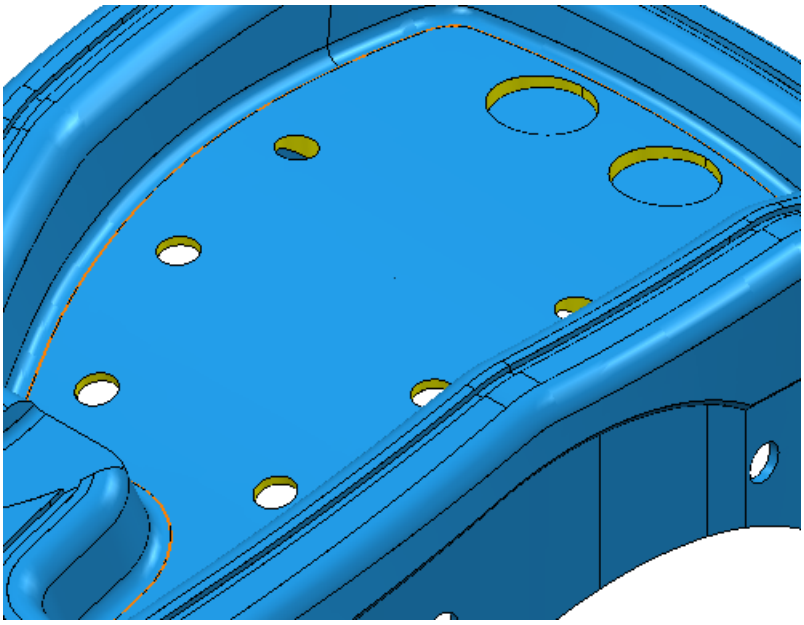
a Select a **Machining Mode** of **Collision**.

b Click **Acquire Components**  .

c Click **Apply**.

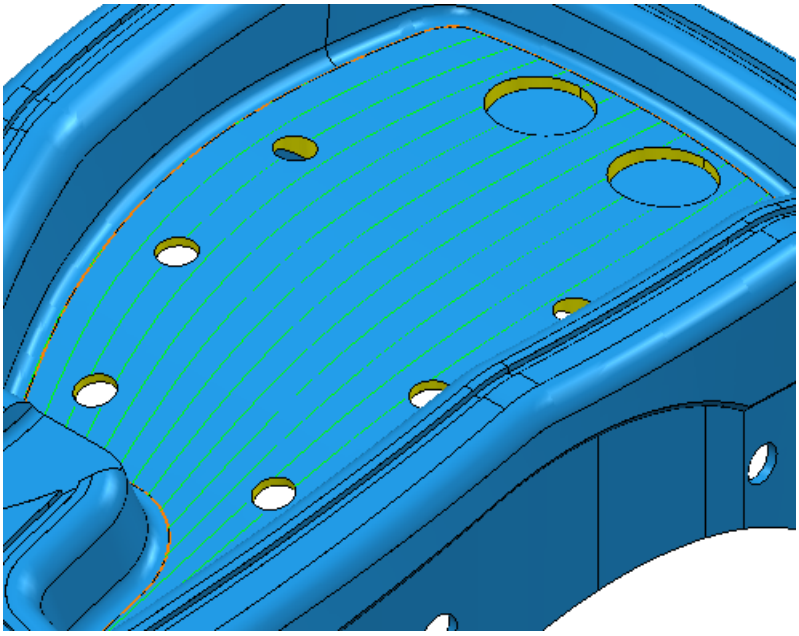
d Click **Accept**.

4 Select **Machining mode** shade  on the **Shading** toolbar  on the **View** toolbar. The components that have a **Machining Mode** of **Collision** are yellow.



5 From the individual toolpath context menu, select **Settings**.

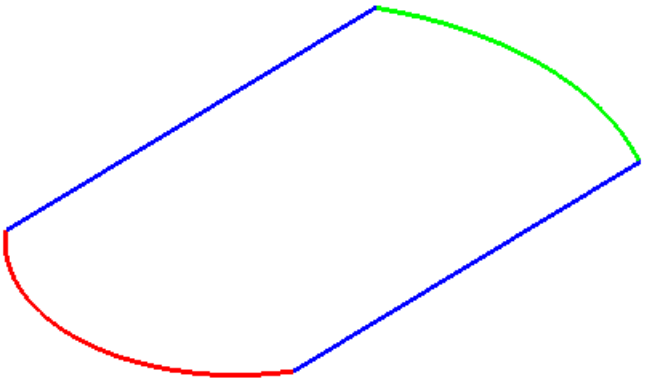
6 Click **Calculate**.



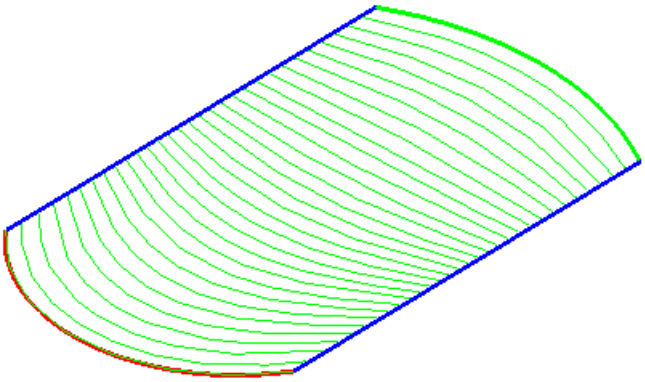
Intermediate curves in Flowline toolpaths

Intermediate drive curves are used to control the location of the toolpath segments. The machining area is unaffected. This is best shown by example.

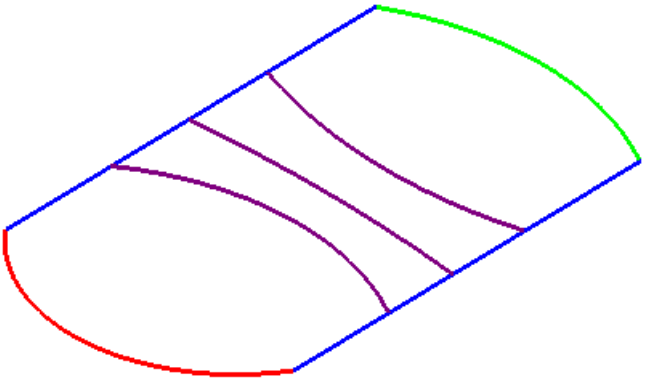
This example has two drive curves and two limit curves, but no intermediate drive curves:



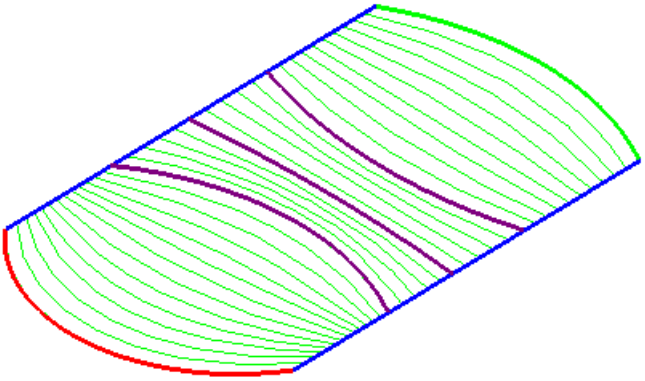
and generates this toolpath:



This example has two drive curves, two limit curves, and three intermediate drive curves:

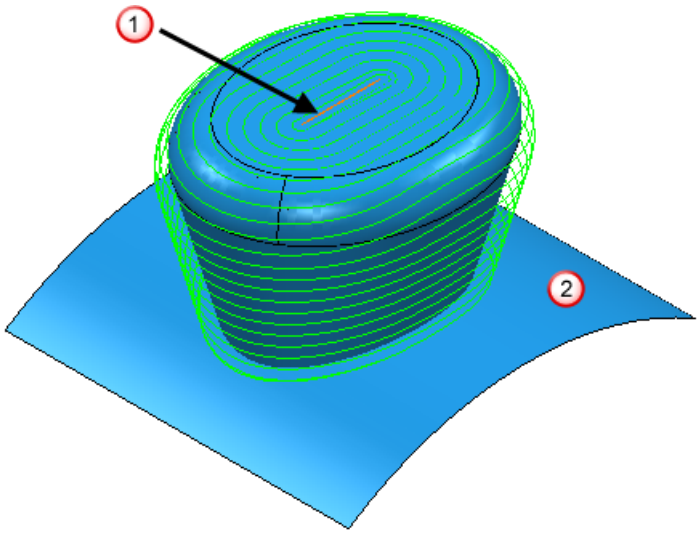


and generates this toolpath:



Parametric Spiral Finishing - Overview

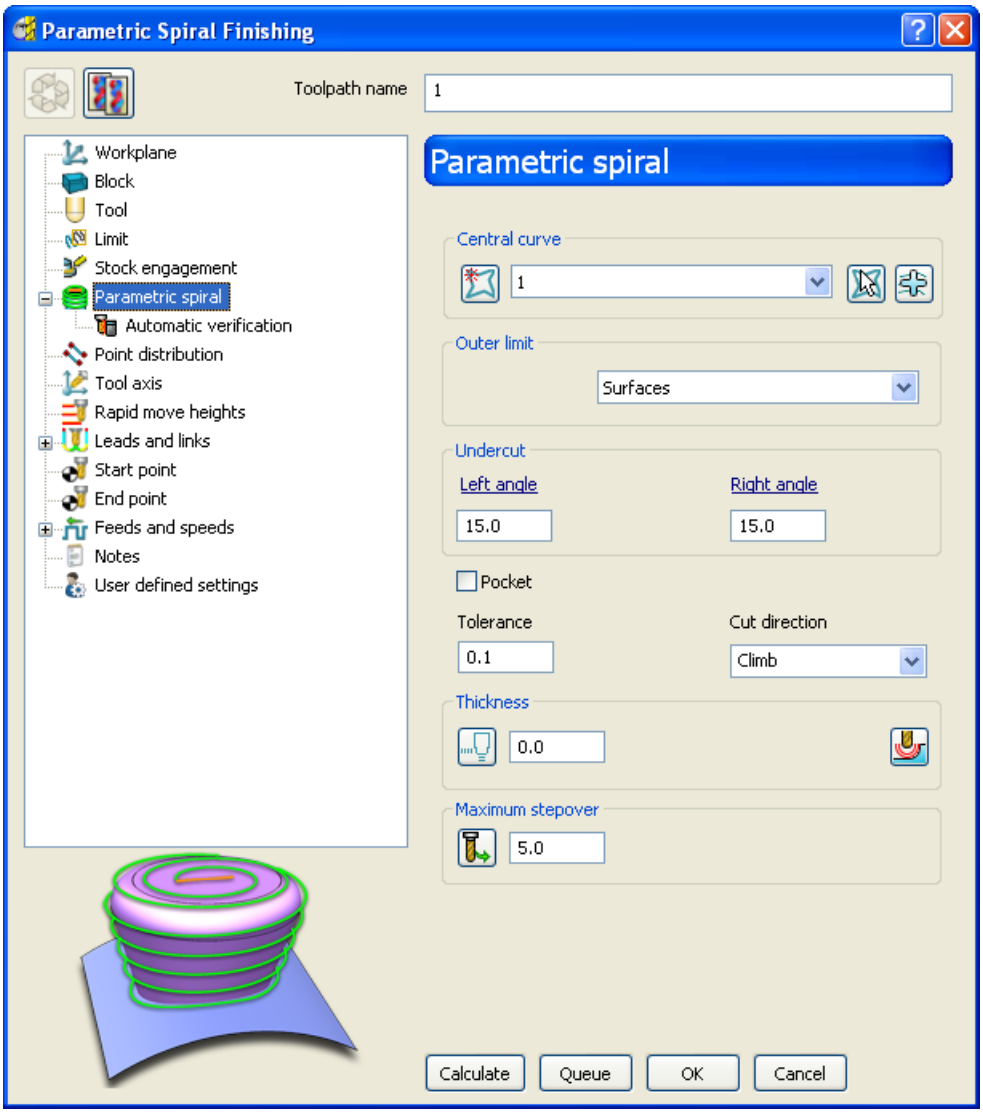
Parametric Spiral Finishing creates a spiral toolpath between a central curve and a check surface. It machines some undercuts.



1 - central curve

2 - check surface

The pages associated with **Parametric Spiral Finishing** are:



- **Parametric Spiral -** the main page to define a parametric spiral finishing toolpath.
- **Automatic verification -** enables automatic verification of toolpaths on creation.

The remaining pages are common toolpath creation controls.

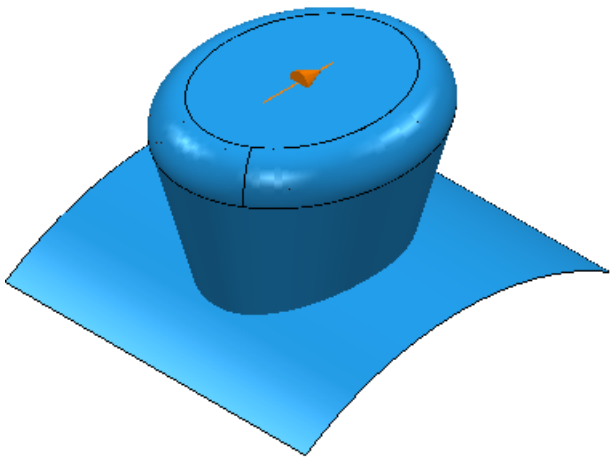
Parametric spiral finishing

Parametric spiral finishing creates a spiral toolpath between a pattern and a surface.

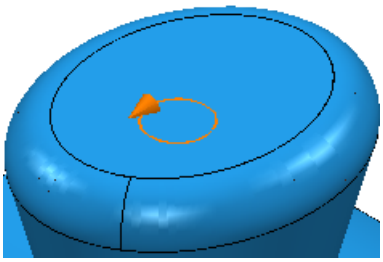
Central curve - a curve or point which determines the start of the toolpath when machining a boss, or the end of the toolpath when machining a pocket. The curve must be a pattern containing one segment. You can use a closed or open pattern.

When you select a central curve, it is instrumented so you can see the orientation of the spiral curve.

An instrumented open curve:



An instrumented closed curve:



If the pattern isn't on the model, then it is projected onto the model and this projection is used as the central curve.

Create pattern - creates a new empty pattern.

Selected pattern - displays the selected pattern. If no pattern is displayed, or <None> is selected, then no pattern is selected. The list contains a list of all available patterns.

Select picked pattern - selects and activates a pattern by picking in the graphics window, rather than by name in the **Select pattern** list.

Collect curves - copies the selected curves into the pattern.

This provides a fast, powerful means of extracting curve geometry from a surface model and copying it into the active pattern/boundary. You can insert:

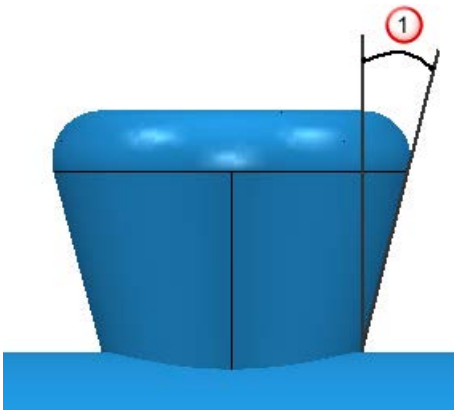
- Individual surface boundary curves.
- Boundary curves around a selection of surfaces.
- Model wireframe geometry.
- Existing pattern or boundary segments.

For more information, see the collecting curves example.

Outer Limit - the surfaces defining the outer extents of the toolpath. The limiting surfaces must be in a level or set. If you don't select a check surface PowerMILL uses the block to limit the toolpath.

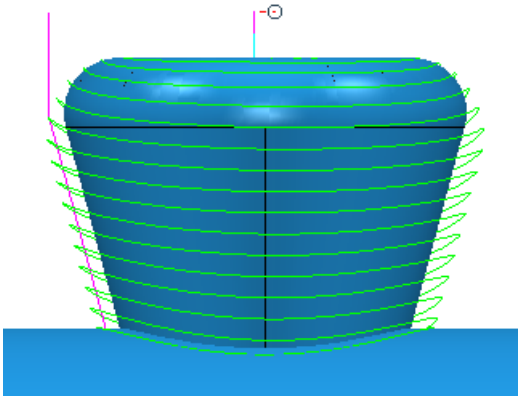
*If there are gaps in the **Outer Limit**, then the **Parametric spiral finishing** strategy tries to machine the model below the gaps. This may introduce imperfections in the toolpath.*

Undercut - the undercut is defined with respect to the **Central curve**. You can have a different angle on each side of the curve. The curve direction is displayed when you select a **Central curve**. If the slope of the part (measured from the vertical) is less than the **Undercut angles** you can machine the part.

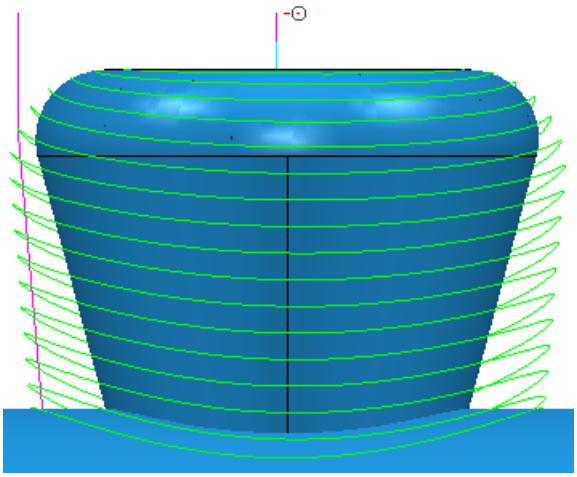


The **Undercut angle** is measured from the vertical.

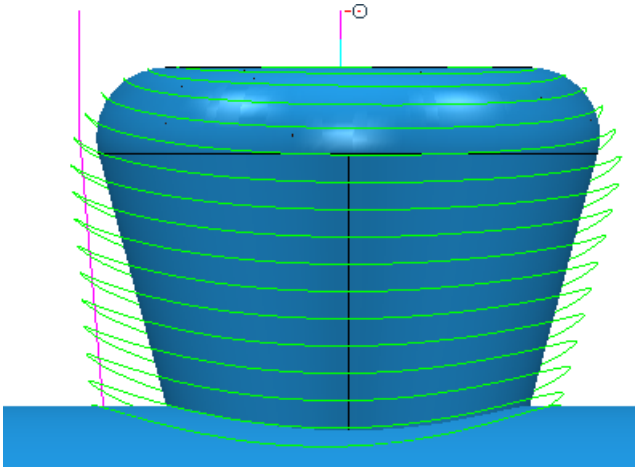
If the angle is the same as, or larger than 1 the complete profile is machined:



If the angle is the less than 1 the complete profile isn't machined:



If you have a different **Left angle** to **Right angle** then the toolpath follows the profile differently on either side of the **Central curve**.



This option is more useful when you are machining an asymmetric part.

Angle - the maximum allowable undercut angle. This option is available when you select a point or a closed curve as your **Central curve**.

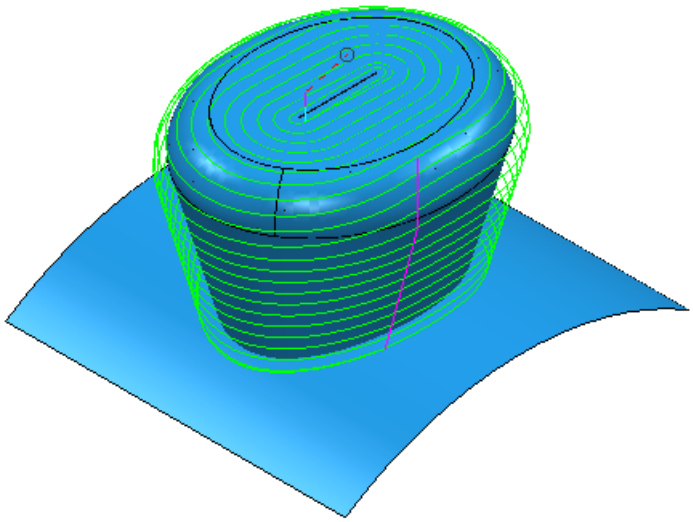
Left angle - the maximum allowable undercut angle to the left of the **Central curve**. This option is available when you select an open curve as your **Central curve**.

Right angle - the maximum allowable undercut angle to the right of the **Central curve**. This option is available when you select an open curve as your **Central curve**.

*By default, this strategy uses a **Tool axis** of **Automatic** which enables you to machine undercuts within the limits specified. If you must have a vertical tool axis, use a tipped disc tool to machine undercuts.*

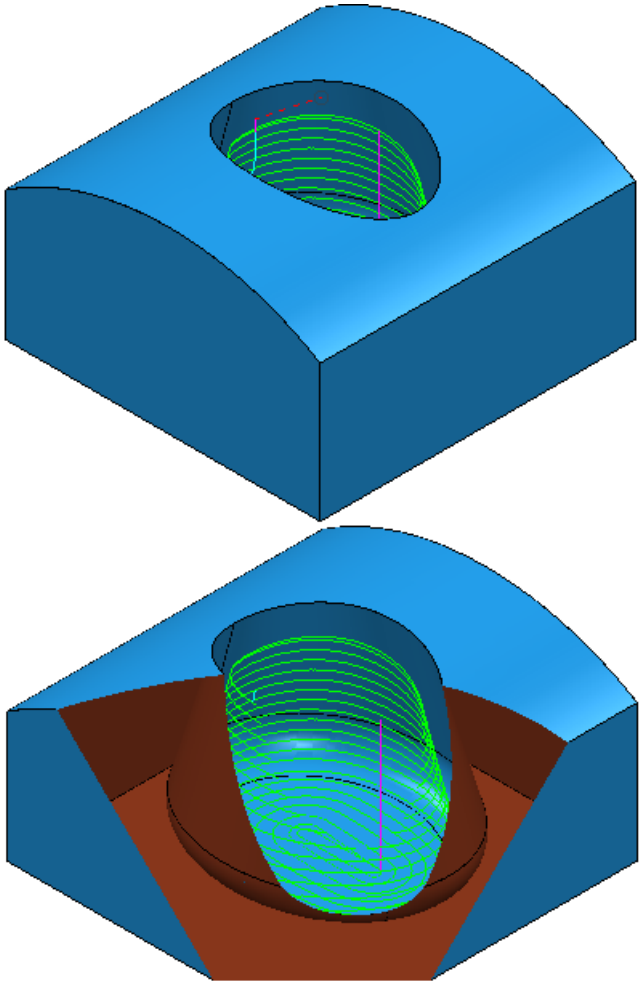
Pocket - machines a pocket when selected and a boss when deselected. This is deselected by default.

Pocket - deselected:



The toolpath is generated from the curve to the surface.

Pocket - selected:



The toolpath is generated from the surface to the curve.

*A (near) vertical **Outer Limit** is likely to produce a better pocket toolpath than a (near) horizontal limit.*

Tolerance - determines how accurately the toolpath follows the contours defined by the model.

Cut direction - determines the milling strategy.

Thickness - specifies the amount of material to be left on the part, within tolerance.

- displays the **Component thickness** dialog, which allows you to specify the thicknesses of the different surfaces.

Maximum stepover - the maximum distance between successive machining passes.

Set stepover from tool **-** loads the radial depth of cut from the active tool cutting data. The radial depth of cut is measured normal to the tool axis.

Edited - the value is entered by you (or another user).Click to change this to the automatically calculated value.

Stepover - the distance between successive machining passes.

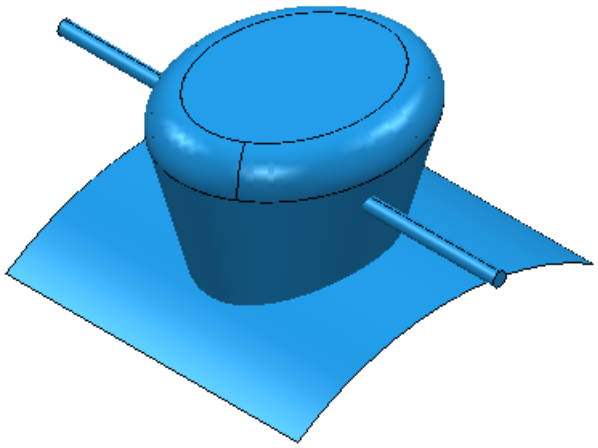
*If you enter a **Stepover** value then changes to .*

Example of parametric spiral finishing

This examples shows you how to create parametric spiral finishing toolpath. This uses:

- The **Boss.dgk** and **Pins.dgk** model in the **Examples** folder.
- The **CentralCurve.dgk** pattern in the **Examples > Pattern** folder.
- A rectangular **Block**.

- A 10 mm ball nosed tool.



The toolpath must machine the boss and avoid the pins.

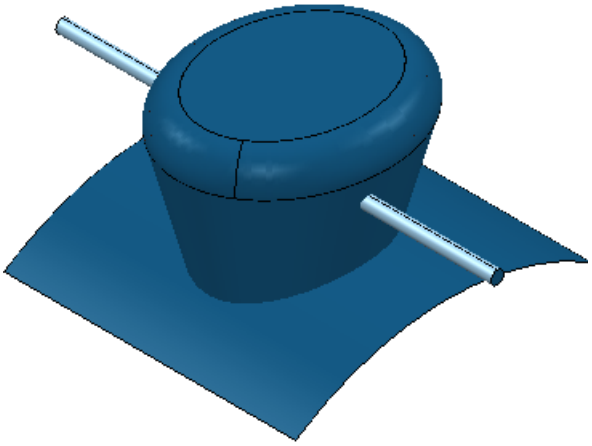
In **Parametric Spiral Finishing** toolpaths, surfaces with a **Machining Mode** of **Collision** behave differently to other strategies.

In parametric spiral toolpaths:

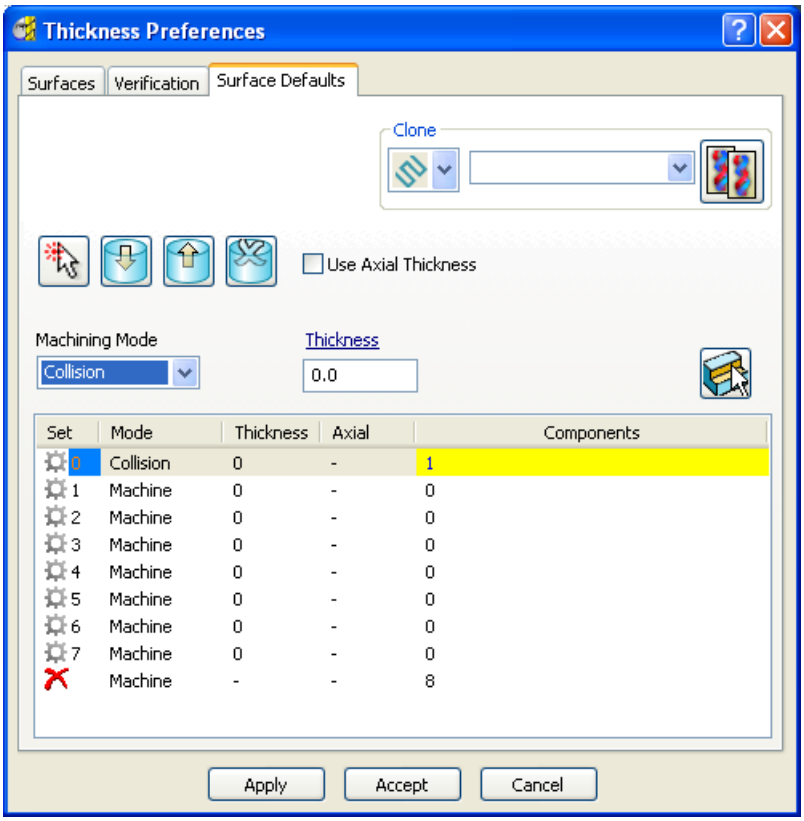
1. A spiral pattern is created by ignoring the surfaces set to **Collision**.
2. The pattern is lifted above the surfaces set to **Collision** along the projection direction of the tool.

This enables fast creation of the required toolpath.

1. To avoid machining the pins, select the pin.



2 Click  to display the **Component Thickness** dialog.



a Select the **Surface Defaults** tab.

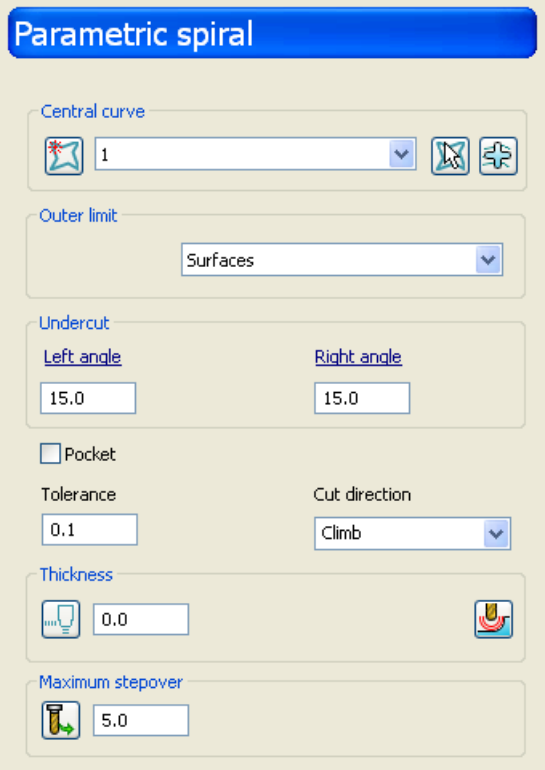
b Select the top set in the list of sets in the bottom of the dialog.

c Click  **Acquire Components** to place the pins in this set. The number of selected **Components** changes to **1**.

d Select a **Machining Mode** of **Collision**.

e Click **Accept**.

3 From the **Strategy Selector**, select the **Finishing** tab and select **Parametric Spiral Finishing** strategy.



a Select a **Central curve** of the pattern you created from **CentralCurve.dgk**.

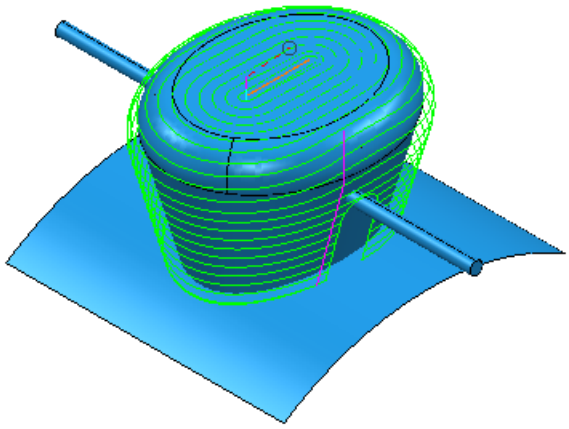
b Select an **Outer limit** of **Surfaces**.

c Enter a **Left angle** of 15.

d Enter a **Right angle** of 15.

e Enter a **Maximum stepover** of 5.

f Click **Calculate**.

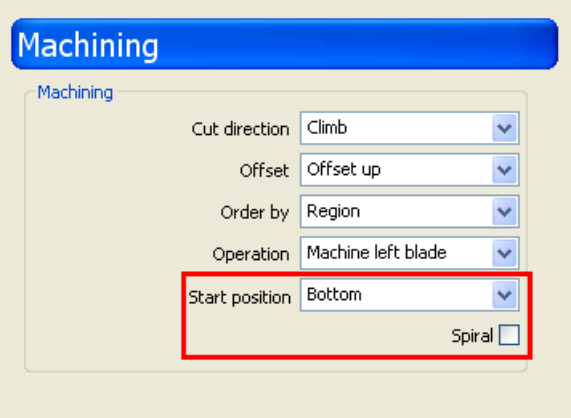


Blisk machining

There are significant improvements to the way blisk machining handles the tool axis. This gives:

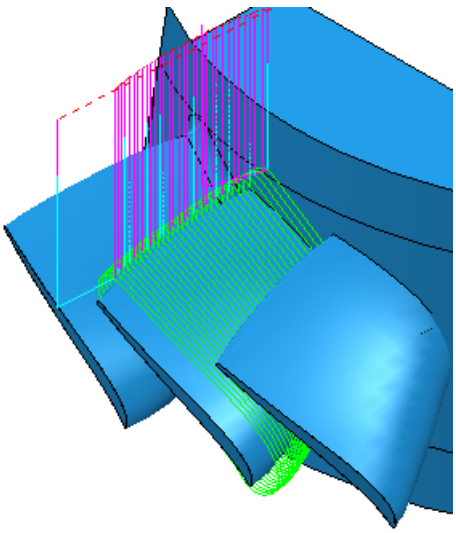
- Improved ability to find a safe tool axis. This enables you to fully machine more parts.
- Reduction of changes in tool axis between passes. This reduces marks on the part.

Blade machining has two new options on the **Machining** page:

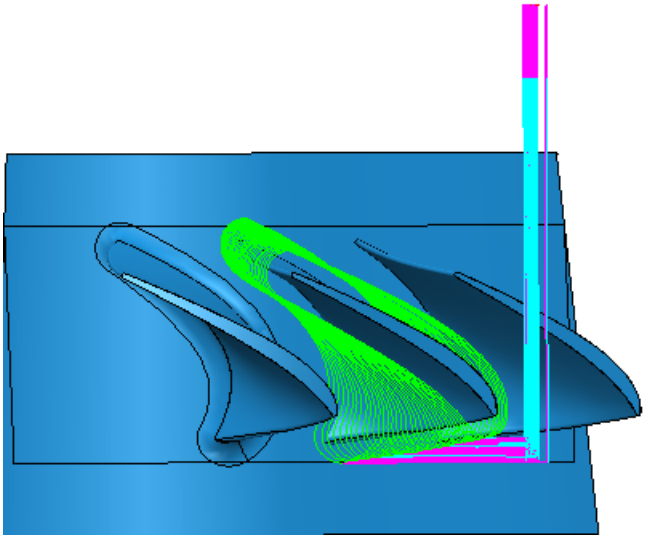


Start position - defines whether the machining starts at the top or the bottom of the blade.

Start position - Top



Start position - Bottom



***Start position** is not available if you have an **Operation** of **Machine all faces**.*

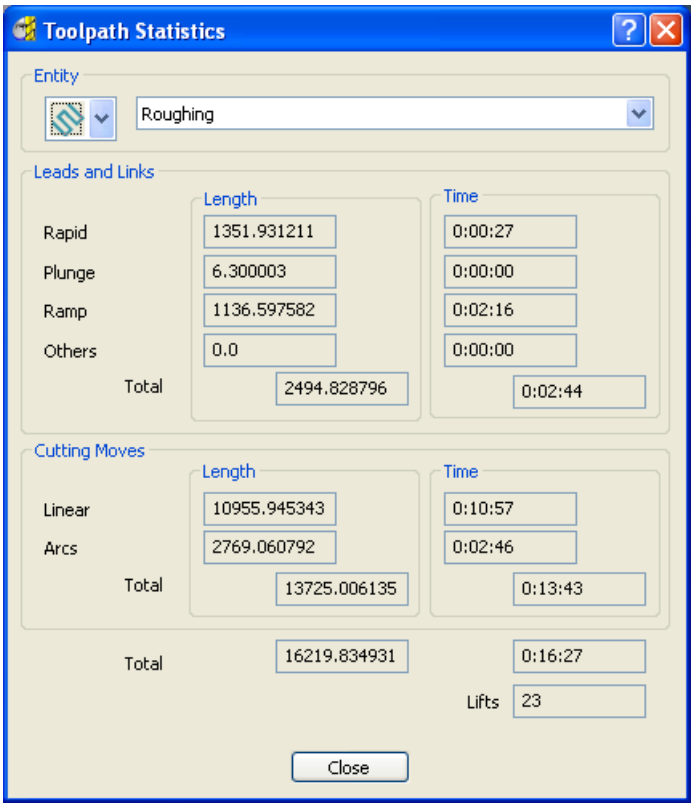
Spiral - enables spiral toolpath between consecutive closed passes around a blade.

***Spiral** works best if you use an **Offset** of **Merge**. If you select **Offset up** or **Offset down** the toolpath may contain open passes which can't form part of the spiral.*

***Spiral** is not available if you have an **Operation** of **Machine all faces**.*

Statistics

The toolpath statistics and NC program statistics are improved.



- You can now view the statistics of any toolpath or NC program. Previously, you could only see the statistics of the active toolpath/NC program.
- The dialog is updated whenever a toolpath/NC program is updated or another toolpath/NC program is selected. In previous versions, you had to click **Estimate** to update the dialog with the new values.
- Toolpath and NC program statistics now have the parameters **Statisitics.CuttingMoves** and **Statisitics.LeadsandLinks**. You can use these statistics parameters in expressions.

User interface

Measuring interactively

The **Measure** toolbar provides tools to interactively populate numeric fields in dialogs with points, coordinates, distances, angles, radii, or unit vectors.

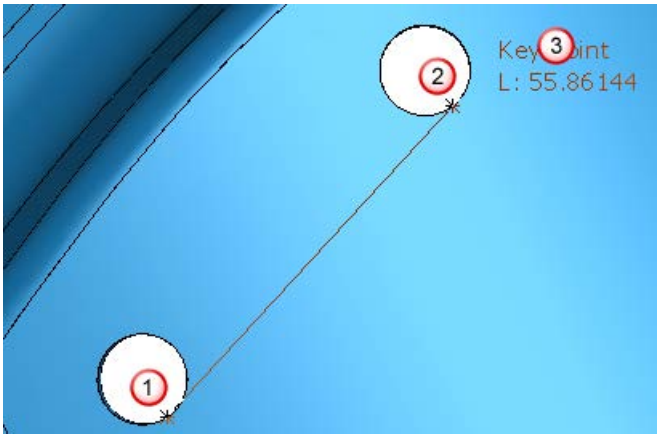
Click a field with a hyperlink in PowerMILL to display the **Measure** toolbar and perform interactive measurements in the graphics window (see page 66).

*You can also right-click in a numeric field and select **Measure** to display the **Measure** toolbar.*

The measuring tool selected by default is based on the value being measured. For example, if you are measuring **Head clearance**, the **Measure the distance between two points** tool is selected.

Similarly, if you are measuring the **Angular threshold**, the **Measure a minor angle** tool is selected. The cursor shows additional information to help you define the points you want.

 **Measure the distance between two points -** select two points to measure the distance between them.

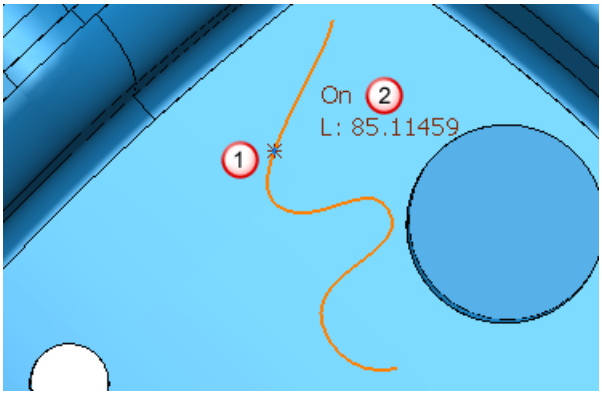


1 - First point.

2 - Second point.

3 - Length between the first and second points.

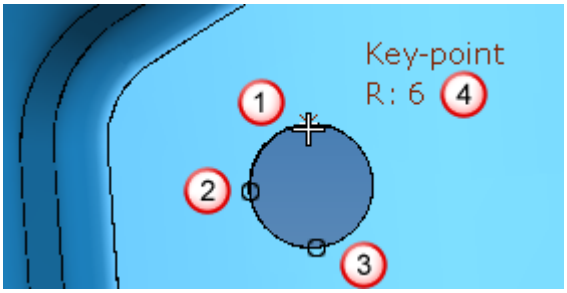
 **Measure a length -** select a curve to measure its arc length. A curve can be a pattern segment, a boundary segment, or a wireframe curve within a model.



1 - Selected curve.

2 - Length of the selected curve.

 **Measure a radius -** select an arc or circle to determine its radius.

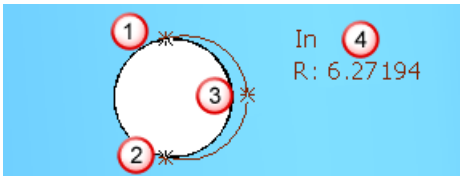


1 - Selected point.

2 and 3 - Reference points to indicate the extents of the circle.

4 - Measured radius.

Measure a radius from three points - select three points to determine the radius of the arc or circle that passes through the points.



1 - First selected point.

2 - Second selected point.

3 - Third selected point.

4 - Measured radius.

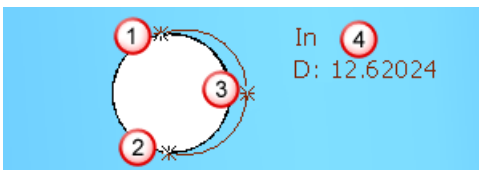
Measure a diameter - select an arc or circle to measure its diameter.

1 - Selected point.

2 and 3 - Reference points to indicate the extent of the circle.

4 - Measured diameter.

Measure a diameter from three points - select three points to measure the diameter of the circle that passes through the points.



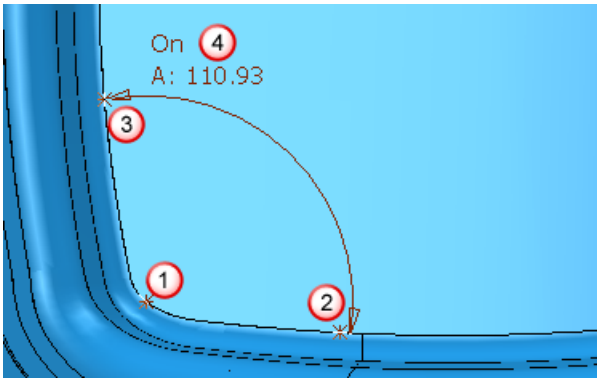
1 - First selected point.

2 - Second selected point.

3 - Third selected point.

4 - Measured diameter.

Measure a minor angle - select three points to measure an angle of less than 180°. When measuring, first select the centre point and then select the next two points.



When measuring, first select the centre point and then select the next two points.

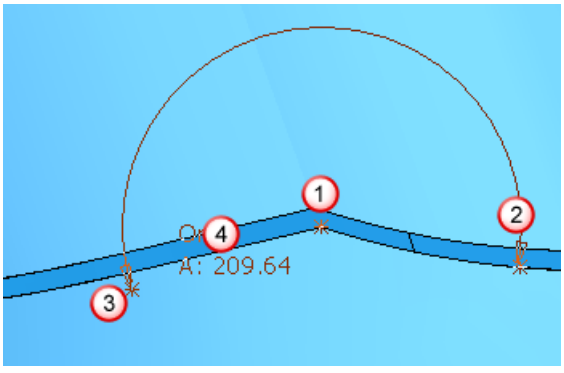
1 - Centre point.

2 - Second selected point.

3 - Third selected point.

4 - Measured minor angle.

Measure a major angle - select three points to measure an angle of more than 180°. When measuring, first select the centre point and then select the next two points.



When measuring, first select the centre point and then select the next two points.

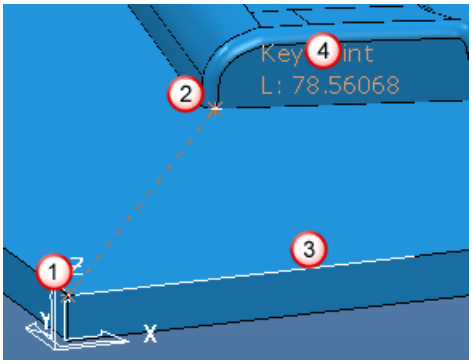
1 - Centre point.

2 - Second selected point.

3 - Third selected point.

4 - Measured major angle.

 **X component -** measures the X component of the line.



1 - First selected point.

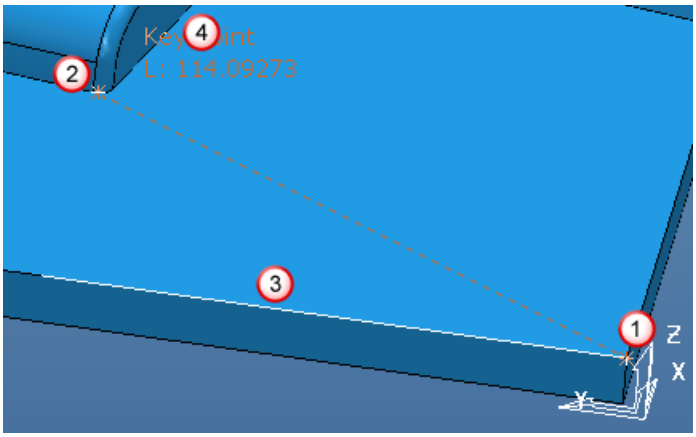
2 - Second selected point.

3 - Guide line for the X component.

4 - Length of the line.

The X component is the length of the guide line (3).

 **Y component -** measure the Y component of the line.



① - First selected point.

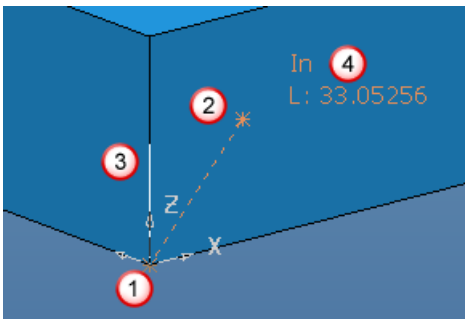
② - Second selected point.

③ - Guide line for the Y component.

④ - Length of the line.

The Y component is the length of the guide line (③).

Z component - measure the Z component of the line.



① - First selected point.

② - Second selected point.

③ - Guide line for the Z component.

④ - Length of the line.

The Z component is the length of the guide line (③).

Project - measure the value after projecting onto the principal , , editing planes.

- **Cancel -** cancels the measuring mode. The **Measure** toolbar is closed, and normal PowerMILL functionality is restored.

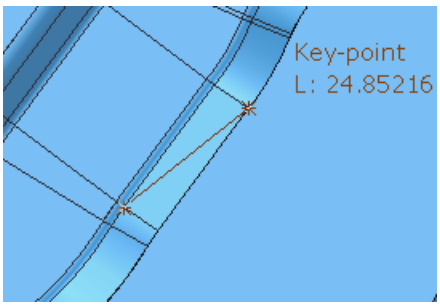
Using the Measure toolbar

To calculate the **Final stepover** as the length between two points on the model using the **Measure** toolbar:

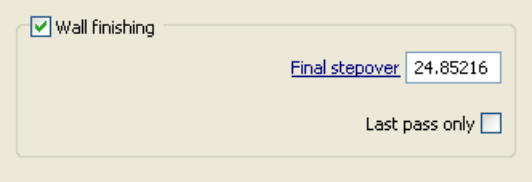
1 Click the **Final stepover** hyperlink.

2 On the **Measure** toolbar which appears, click **Measure the distance between two points**  .

3 Select two key-points on the model to calculate the distance between them. The cursor displays additional information to help you with your selection.

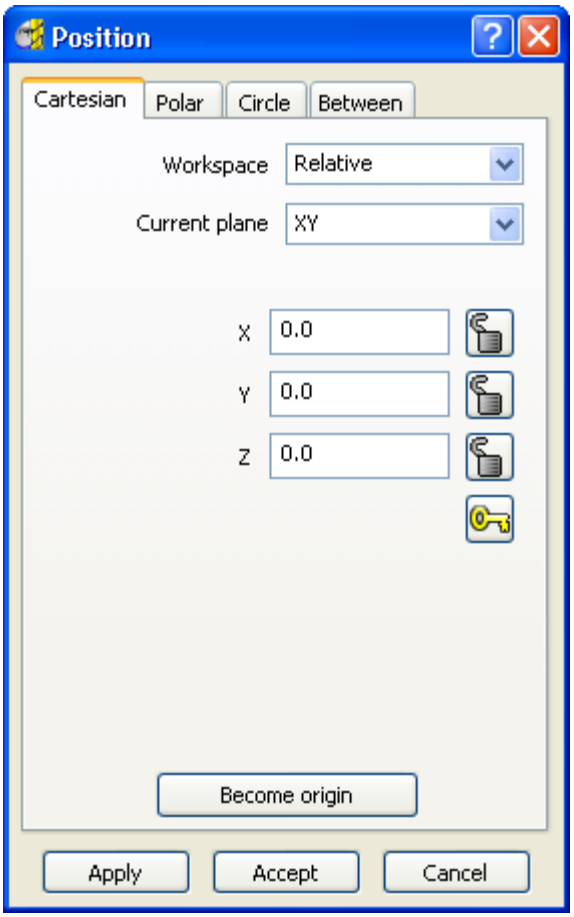


Selecting the second key-point closes the **Measure** toolbar and the measured length is entered into the **Final stepover** field.



Additional options to locate items

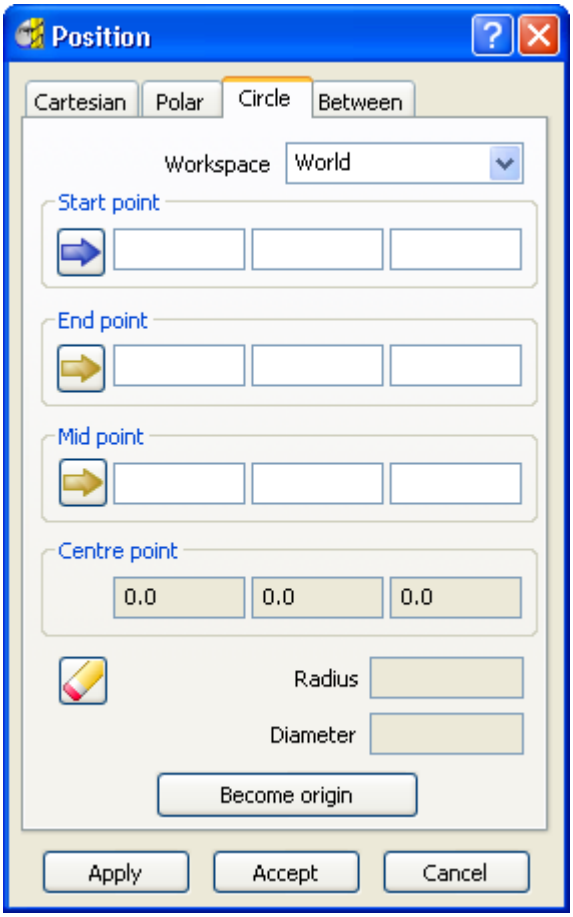
The **Position** dialog has two additional tabs which allow you to locate the centre point of a circle and define a position between two connected points. This is in addition to the **Cartesian** and **Polar** tabs.



- **Circle** (see page 69) - calculates the centre of a circle from a start, end, and mid point.
- **Between** (see page 71) - calculates a new point between two existing points.

Position - Circle

The **Circle** tab of the **Position** dialog allows you to construct a circle and calculate the centre point.



Workspace - determines 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.

Relative - specifies the point relative to the local origin.

indicates the current point, indicates the next available point. PowerMILL cycles through the points as you make selections in the graphics window. You can click to define a particular point.

Start point - select a start point in the graphics window to define its X, Y, Z coordinates.

End point - select an end point in the graphics window to define its X, Y, Z coordinates.

Mid point - select a mid point in the graphics window to define its X, Y, Z coordinates.

You can also manually enter the X, Y, Z coordinates to define a point.

Centre point - the centre point of the defined circle. The centre point will always respect any locks that are set on the **Cartesian** or **Polar** tabs. For example, if Z is locked when defining a circle, the values of X and Y coordinates will correspond to the defined circle, but the Z coordinate value will remain unchanged.

Radius - radius of the defined circle.

Diameter - diameter of the defined circle.

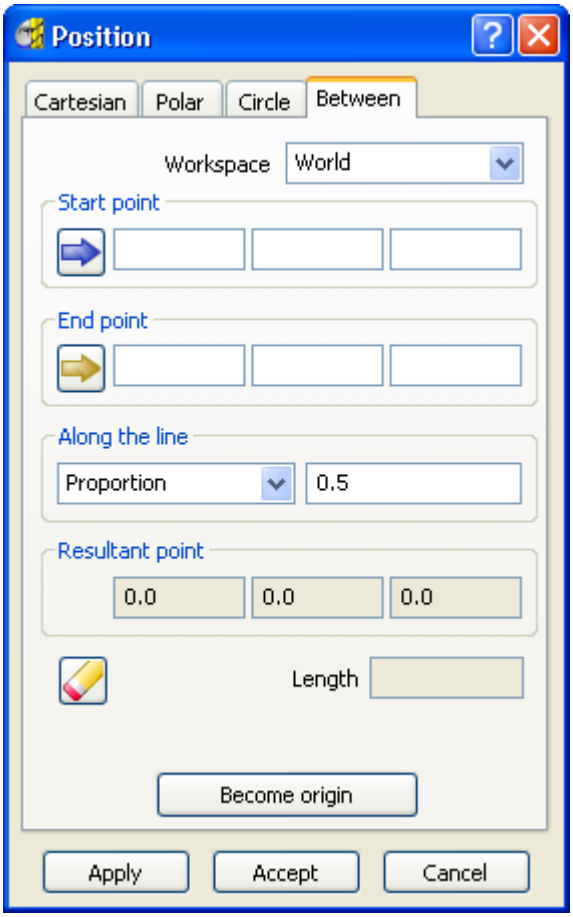
The **Centre point**, **Radius**, and **Diameter** of the circle are automatically calculated from the **Start**, **End**, and **Mid** points.

 **Reset all** - resets the **Start**, **End**, and **Mid** points.

Become origin - the point defined becomes the item origin. Any relative coordinates are then measured from this origin.

Position - Between

The **Between** tab of the **Position** dialog allows you to construct a line from two points and calculate any point along its length.



Workspace - determines 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.

Relative - specifies the point relative to the local origin.

indicates the current point, indicates the next available point. PowerMILL cycles through the points as you make selections in the graphics window. You can click to define a particular point.

Start point - select a start point in the graphics window to define its X, Y, Z coordinates.

End point - select an end point in the graphics window to define its X, Y, Z coordinates.

You can also manually enter the X, Y, Z coordinates to define a point.

Along the line - how you specify the measurement.

Proportion - enter a fraction for the proportional distance from the start of the object. For example, 0.25 indicates quarter of the way between the points.

Distance - enter a distance from the start of the point.

Resultant point - displays the coordinates of the point you create. The resultant point always respect any locks that are set on the **Cartesian** or **Polar** tabs. For example, if Z is locked when defining a position, the values of X and Y coordinates correspond to the defined position, but the Z coordinate value remains unchanged.

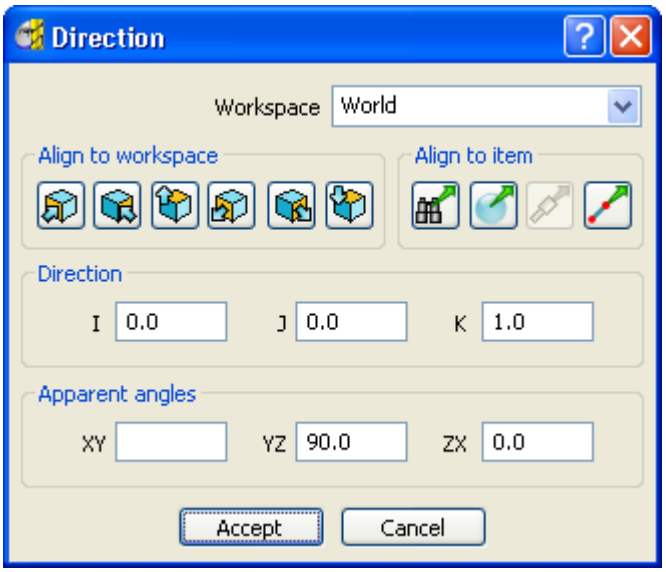
Length - displays the length of the resulting line.

Reset all - resets the **Start** and **End** points.

Become origin - the point defined becomes the item origin. Any relative coordinates are then measured from this origin.

Additional options to align the direction vector

The **Direction** dialog has additional options to align the direction vector.

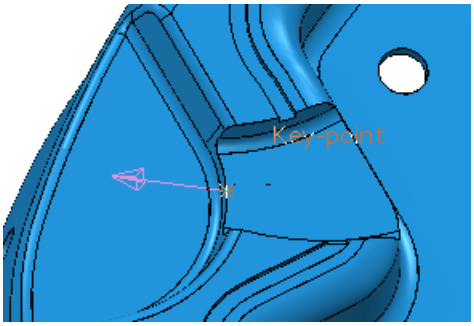


Align to item - aligns the direction vector with the view, geometry, tool, or line.

 **Align with view -** aligns the direction vector with the current graphics view.

 **Align with geometry -** aligns the direction vector so that it is normal to the surface at that point.

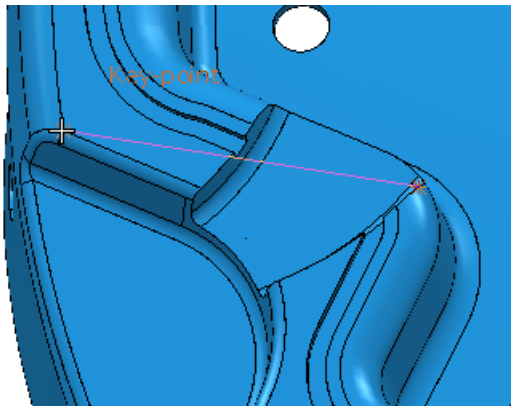
The direction vector is aligned so that it is normal to the surface at that point.



 **Align with tool -** aligns the direction vector so that it is aligned with the active tool axis.

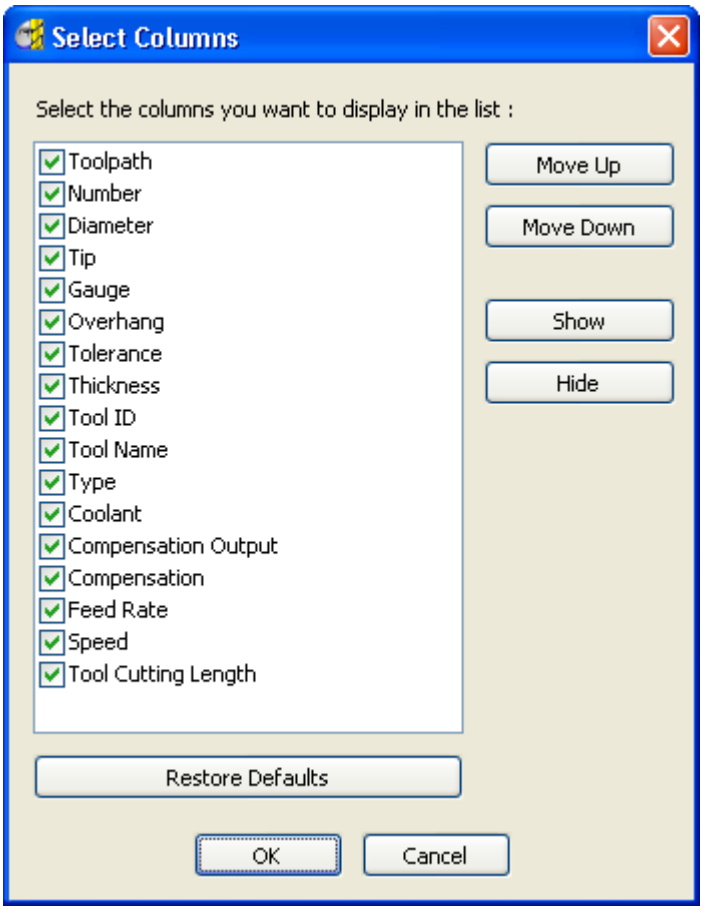
 **Align with a line -** select two points in the graphics window to define a line and align the direction vector.

The direction vector is aligned with the line defined by two points.



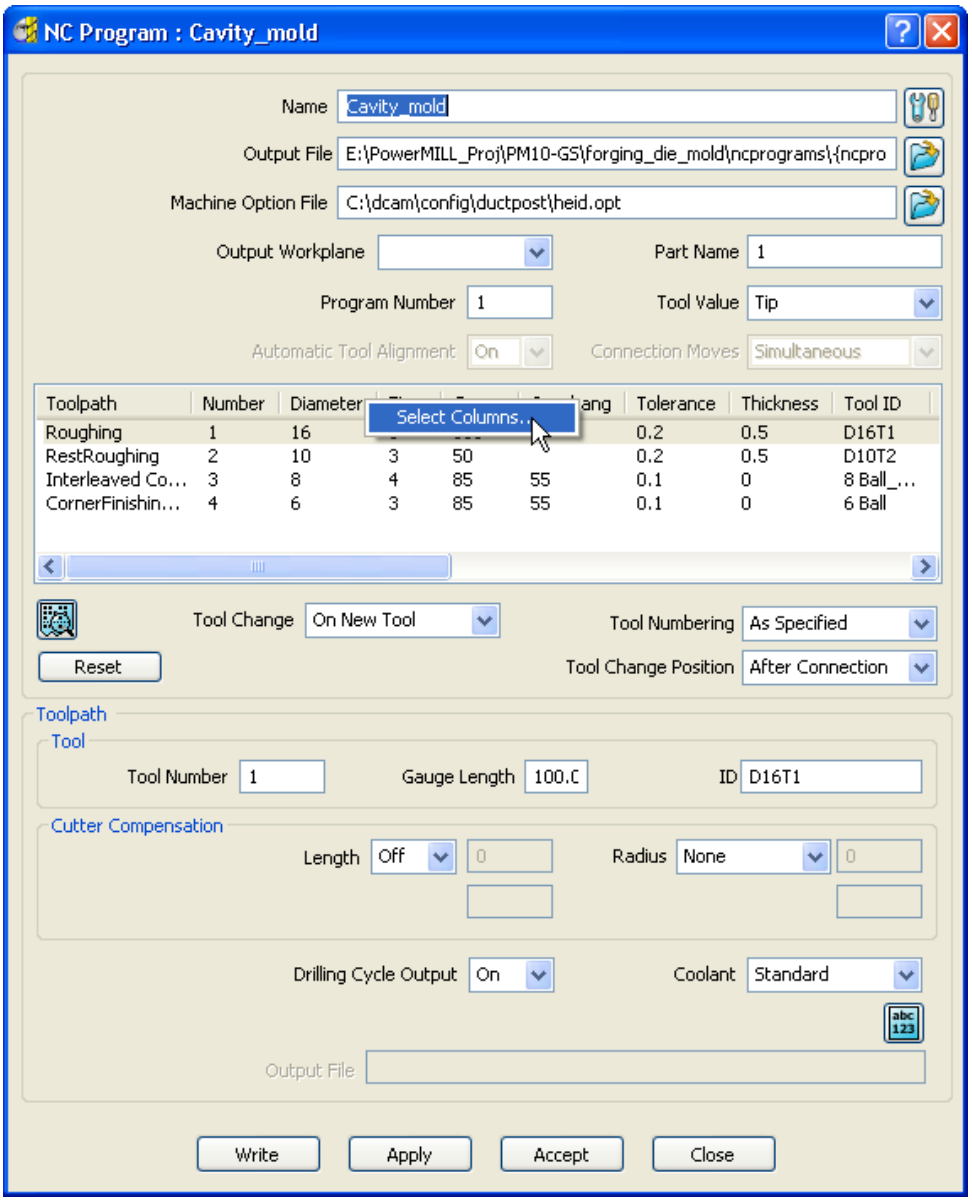
Customising Column Views

A new **Select Columns** dialog is available to customise column views that control and prioritise the amount of information displayed in the **NC Program** and **Tool Database** dialogs.



To customise column views in the **NC Program** or **Tool Database** dialogs:

1 Right-click the column heading and choose **Select columns**.



2 In the **Select Columns** dialog:

- Select the check box of each item that you want to view.
- Deselect the check box of each item that you do not want to view.

*You can also use the **Show** or **Hide** buttons to select or deselect the column items.*

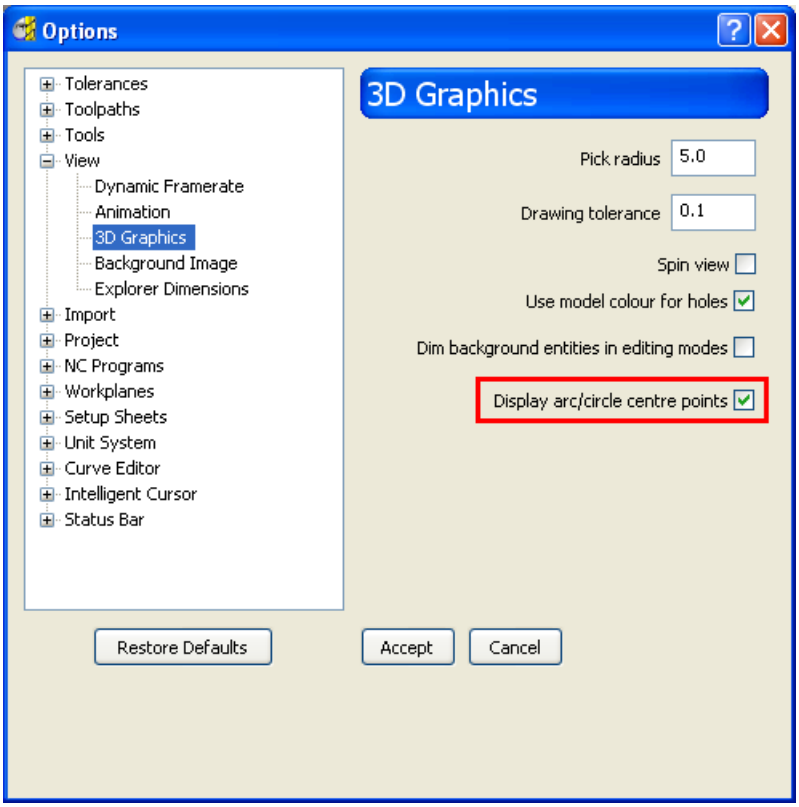
3 To reorder the items in the columns, select the item, and then click **Move Up** or **Move Down.**

4 Click **OK**.

To clear any of your customisations, click **Restore Defaults**.

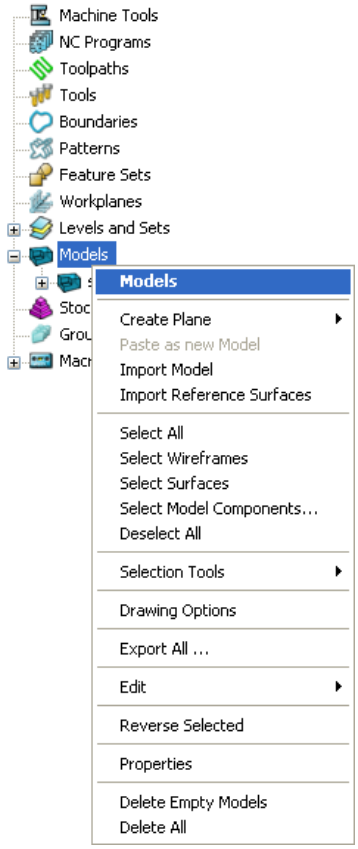
Displaying the centre point of arcs

There is a new option on the **Tools > Options > View > 3D Graphics** menu of **Display arc/circle centre points**. This displays the centre point of every arc and circle in a pattern, boundary, or model.



Select Visible Reversed Components

You can now select visible reversed components easily using the **Select Visible Reversed Components** option. This option is available from the **Models** and individual model context menus in explorer.



Selection Tools - contains additional selection tools.

Select Duplicate Components - selects duplicate components. If you then use the **Delete Selected** menu option, then the duplicates are deleted leaving a model with no duplicates.

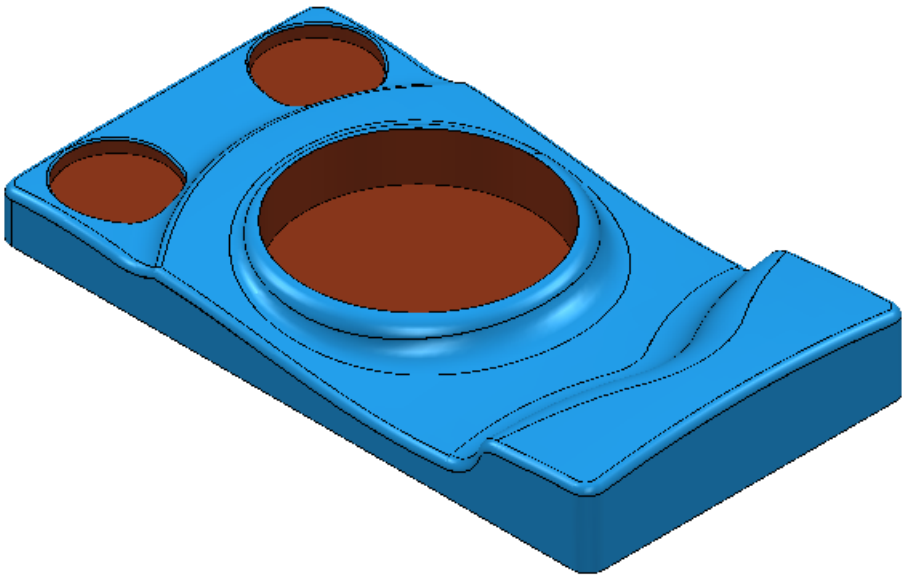
Deselect Duplicate Components - deselects duplicate components in the current selection. If you select the components that you want, but think that there may be duplicates, you can use the **Deselect Duplicate Components** command to ensure that no duplicate components are selected. The duplicate components are not deleted, but only removed from the selection. This will then allow you to create your toolpath (such as Swarf Machining) without any problems.

Select Visible Reversed Components - selects all components which have visible reversed surfaces in the current view orientation.

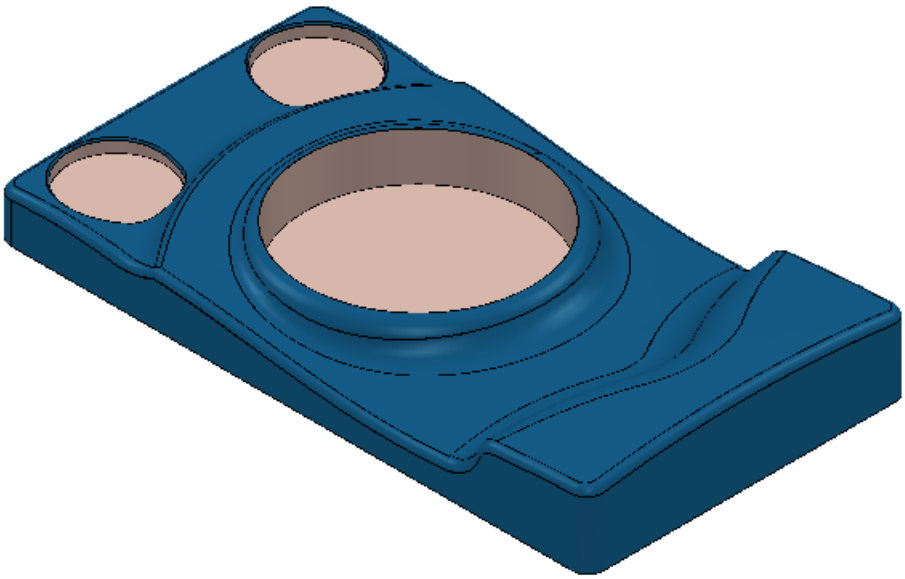
The **Select Visible Reversed Components** option selects the reversed components in the current view orientation.

For example:

Reversed components not selected appears as:



Using the option, the selected reversed components appear as:



*This can be used with **Reverse Selected** (on the model context menu) to reverse all the incorrectly oriented surfaces.*

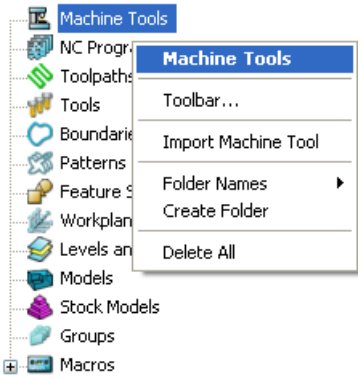
The quality of the selection is dependent on the quality of the model. When using this option, if there are gaps in surfaces which reveal inside surfaces, then those surfaces will be selected as well.

Machine Tools

You can now directly import and manipulate machine tools from the PowerMILL explorer.

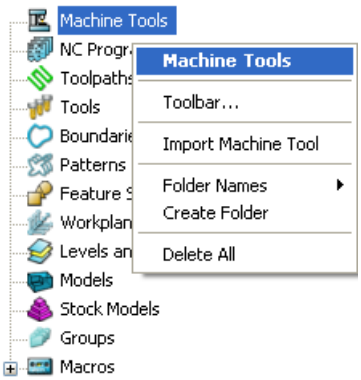
Projects can directly contain many machine tools. They are shown under the **Machine Tool** branch in explorer.

The context menu for the **Machine Tool** entity allows you to import machine tools and modify machine tool settings.



Machine tools context menu

In explorer, right-click **Machine Tools** to display the context menu.



Machine Tools - the name of the context menu.

Toolbar - raises the toolpath **Simulation Toolbar**.

Import Machine Tool - displays the **Import Machine Tool** dialog, which allows you to import **.mtd** files (see page 80).

Create Folder - creates a folder called **Folder1**. You can then rename this folder in the normal way, and then drag and drop entities into the folder. For more information see Folders.

Delete All - deletes all machine tools.

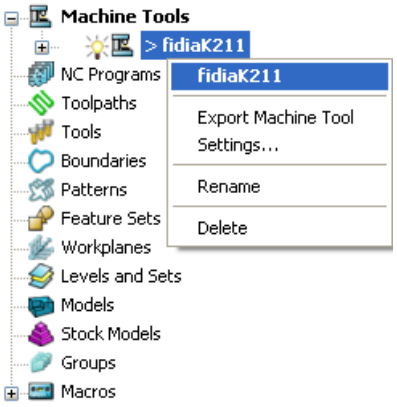
Importing a machine tool

To import a machine tool into a project:

1. In explorer, right-click **Machine Tools**, and select **Import Machine Tool**.
2. In the **Import Machine Tool** dialog, select the **.mtd** file that you want to import and click **Open**.

Individual Machine Tools context menu

The individual **Machine Tools** context menu is raised by right-clicking on a specific machine tool in the explorer.



fidiaK211 - the name of this menu, and the selected machine tool.

Export Machine Tool - select to export machine tool settings into an **.mtd** file (see page 85).

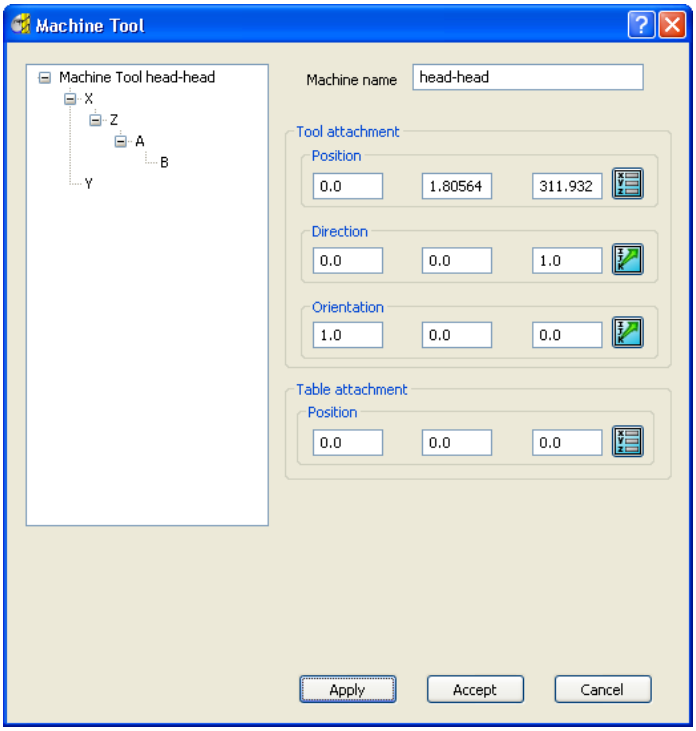
Settings - displays the **Machine tool** settings dialog (see page 81).

Rename - allows you to rename the machine tool.

Delete - deletes the machine tool.

Settings - Machine Tool

To view the imported machine tool settings, in explorer, right-click the imported machine tool and select **Settings** to display the **Machine tool** dialog (see page 80).



MachineTool - the machine tool imported into PowerMILL.

X, Z, A, B, Y - the parts of the machine tool named according to their axes.

Machine name - the name of the machine tool as it appears in the explorer.

Tool attachment - the position where the milling tool is attached to the machine.

Position - enter the position of the tool attachment. - displays the **Position** dialog which lets you manually enter coordinates and locate items in the graphics window.

Direction - enter the direction of the tool attachment. - displays the **Direction** dialog which lets you edit the direction of an item.

Orientation - orientation of the tool attachment. - displays the **Direction** dialog which lets you edit the direction of an item.

Table attachment - the position where the model to be machined is attached to the table.

Position - enter the position of the table attachment.  - displays the **Position** dialog which lets you manually enter coordinates and locate items in the graphics window.

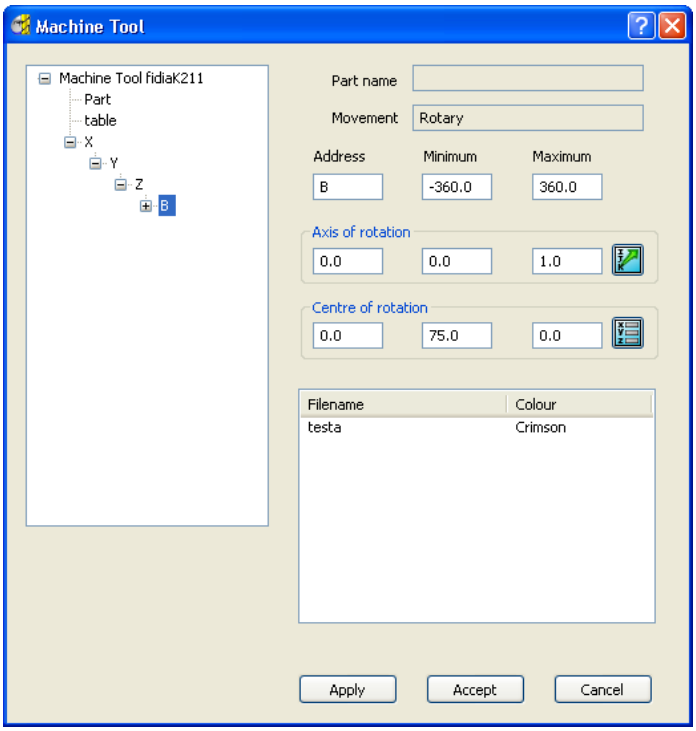
Accept - accepts the values on the dialog and closes the dialog.

Cancel - closes the dialog without updating the values.

Settings - Machine Part

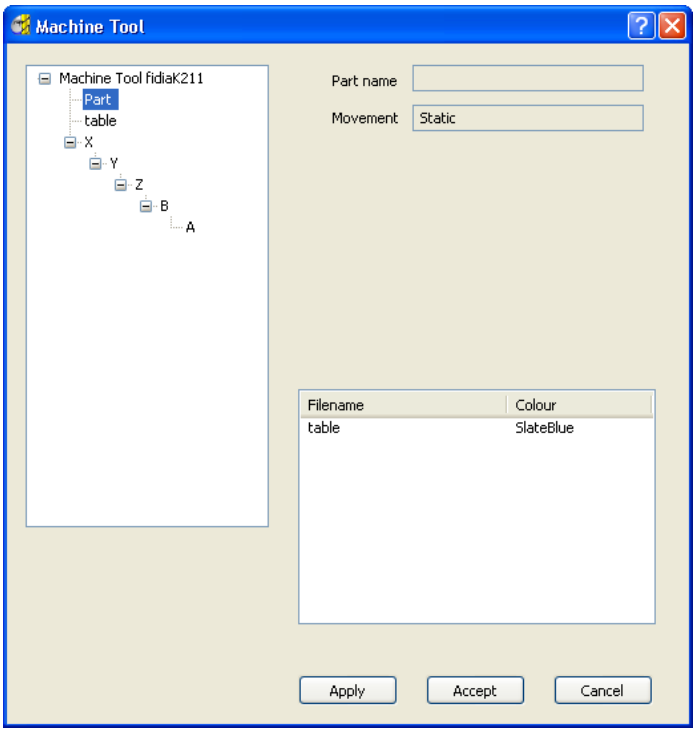
Machine parts are individual components of a machine tool, named according to their axes.

To view the properties of a machine part, select the machine part in the **Machine Tool** dialog.



Machine parts are classified according to the type of movement they perform. They are:

- **Static -** machine parts which do not move.



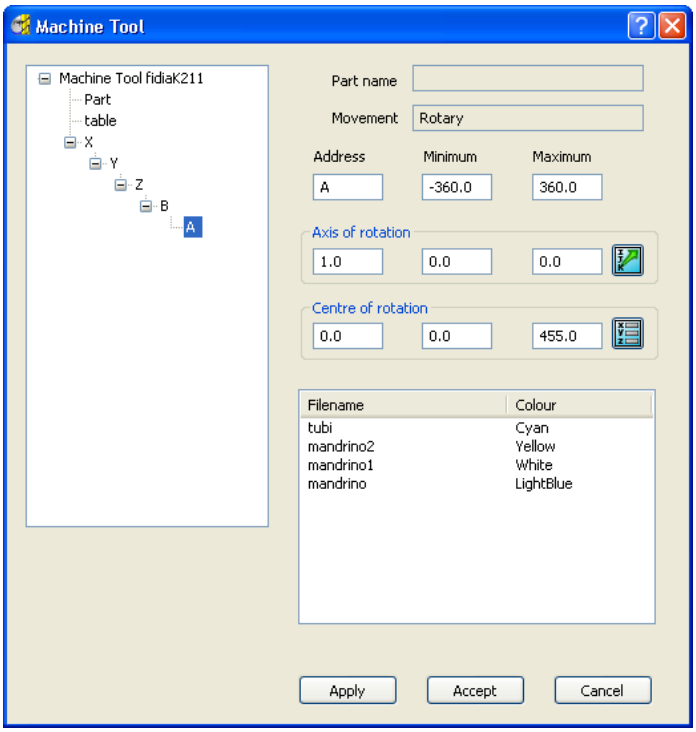
Part name - the name of the machine part as in the imported machine file.

Movement - Static - the type of machine part movement.

Filename - the machine part files that are referenced by the imported machine file.

Colour - the colour of the machine parts which are referenced by their respective machine part files.

- **Rotary -** machine parts with a rotary movement.



Part name - the name of the machine part as in the imported machine file.

Movement - Rotary - the type of machine part movement.

Address - the machine part axis name.

Minimum - the smallest possible angular limits of the axis.

Maximum - the largest possible angular limits of the axis.

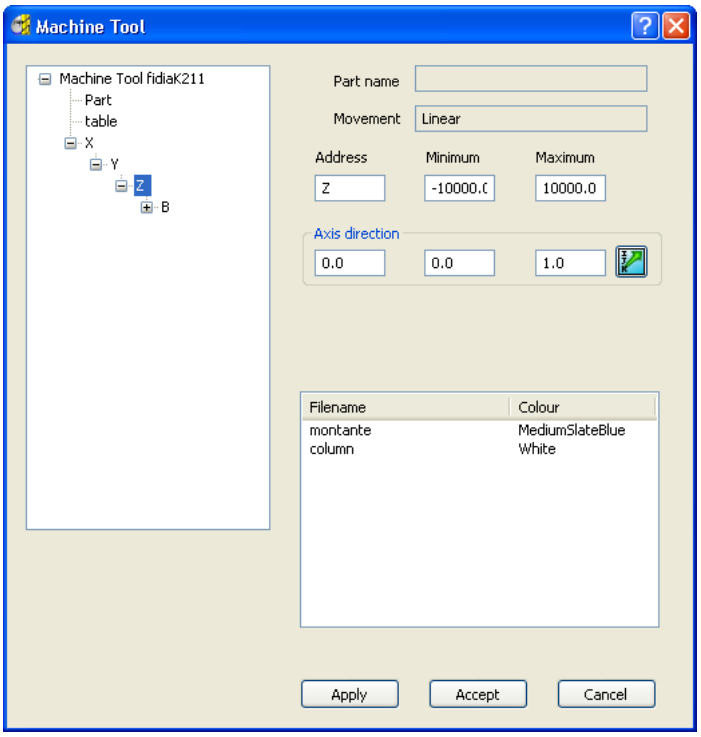
Axis of rotation - the machine part axis of rotation.  - displays the **Direction** dialog which lets you edit the direction of an item.

Centre of rotation - the machine part centre of rotation.  - displays the **Position** dialog which lets you manually enter coordinates and locate items in the graphics window.

Filename - the machine part files that are referenced by the imported machine file.

Colour - the colour of the machine parts which are referenced by their respective machine part files.

- **Linear -** machine parts with a linear movement.



Part name - the name for the machine part as in the imported machine file.

Movement - Linear - the type of machine movement.

Address - the machine part axis name.

Minimum - the smallest possible limits of the axis.

Maximum - the largest possible limits of the axis.

Axis direction - the machine part axis direction.  **-** displays the **Direction** dialog which lets you edit the direction of an item.

Filename - the machine part files that are referenced by the imported machine file.

Colour - the colour of the machine parts which are referenced by their respective machine part files.

Exporting a machine tool

To export a machine tool from a project:

1. In explorer, under **Machine Tools**, right-click the machine tool to be exported, and select **Export Machine Tool**.
2. In the **Export Machine Tool** dialog, enter a name for the machine tool **.mtd** file, select a location, and click **Save**.

General enhancements

Programming language

There is a new PowerMILL macro programming language which allows you to:

- Exploit PowerMILL parameters.
- Construct expressions.
- Use a range of relational and logical operators.
- Evaluate expressions.
- Assign values to variables and parameters by using assignments.

This is a very powerful tool which enables you to automate PowerMILL.

There is a separate Macro Programming Guide available as a PDF for download and is included in the main reference help.

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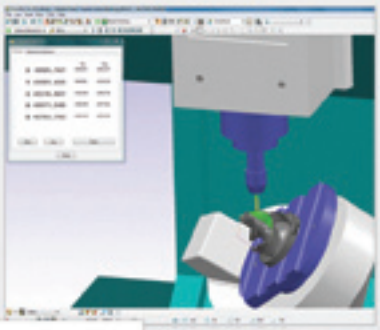
PowerMILL 2012

PowerMILL 2012



new features

Delcam TV

www.delcam.tv

PowerMILL Learning Zone

www.delcam.tv/lz

PowerMILL Website

www.powermill.com

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