PowerMILL 2016

What's New

Issue 1

PowerMILL

Acknowledgements

Patents

The Raceline smoothing functionality is subject to patent applications.

Patent granted: GB 2374562 Improvements Relating to Machine Tools

Patent granted: US 6,832,876 Machine Tools

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

Patent granted: GB 2 423 592 Surface Finish Prediction

The Vortex machining functionality is subject to patent applications.

Patent application: 1121277.6 Adaptive Clearance

The MachineDNA functionality is subject to patent applications.

Patent application: 1204908.6 Machine Testing

Licenses

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

PowerMILL 2016. Published on 15 July 2015

Contents

Summary of new features

PowerMILL is the leading NC CAM software specialising in the manufacture of complex shapes typically found in the toolmaking, automotive, and aerospace industries. PowerMILL 2016 offers all of the original features of PowerMILL 2015R2, but with numerous improvements. This document describes the most significant improvements.

PowerMILL 2016 contains the following new features and enhancements:

Preparing toolpaths

- Planar section (see page 3) — Creates planar sections that slice the model along a plane.

Generating toolpaths

- Project mirroring (see page 10) — Machines symmetric parts and maintains machining characteristics.
- Single blade finishing (see page 17) — Creates toolpaths to machine single-blade aerofoils.
- Rib machining enhancements (see page 28) — The **Offset** list contains the following new options:
  - **Floor only** — Creates toolpaths along the base of the channel.
  - **Constant Z** — Creates toolpaths with a constant Z height.
- Toolpath visualisation (see page 28) — Displays safe areas translucently.

Verifying toolpaths

- NC program verification enhancements (see page 30) — Enables you to turn off machine tool checks for connections or toolpaths.
- Automatic collision checking enhancements (see page 32) — Enables you to select what is being collision checked in **Model Area Clearance** and **Model Rest Area Clearance** strategies.

Outputting toolpaths

- Carousel tool changes (see page 34) — Simulates tool changes on a carousel or a tool-changer.
- Tool Position (see page 35) — Displays the coordinates of the tool when it is attached to the machine tool.
- General enhancements (see page 36):
  - Automatic transparency of machine tools — PowerMILL automatically makes machine tools transparent if they obstruct the view of a model during a simulation.
  - Overhang adjustment on collision — Modifies only the tool overhang after a collision.
  - Tool drawing during simulation — Toggles the drawing of a tool during simulation.

Automating PowerMILL

- Model metadata (see page 38) — **Delcam Exchange** can now extract the metadata from 3rd party CAD files and communicate them to PowerMILL during the translation process.

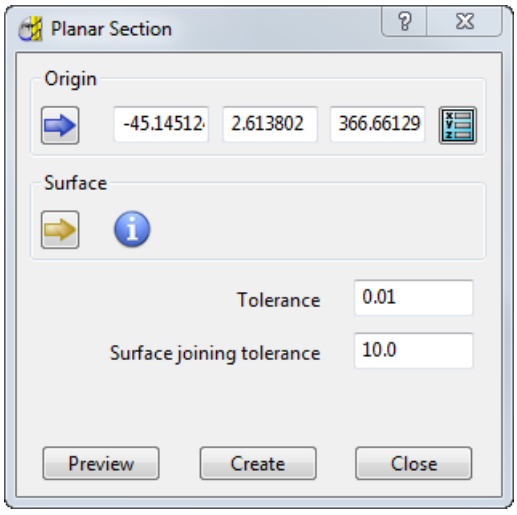
Preparing toolpaths

PowerMILL 2016 contains the following changes and improvements to the preparation of toolpaths:

- Planar section (see page 3) — Create planar sections that slice the model along a plane.

Planar Section creation

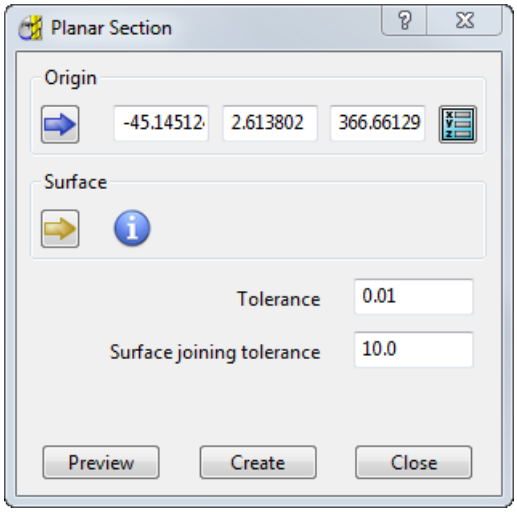
Use the **Planar Section** dialog to create curves that slice the model along a plane. For an example of planar section creation, see Creating a planar section example (page 5).



*Before you create a planar section, you must create a **Pattern** or **Boundary.***

Planar Section dialog

Use the **Planar Section** dialog to create curves that slice the model along a plane.



To display the **Planar Section** dialog:

1. In the Explorer, right-click an individual pattern or boundary.
2. Select **Curve Editor**. The **Curve Editor** mode-toolbar is displayed.
3. Click on the **Curve Creation** pull-out toolbar.

The dialog contains the following options:

Origin — Use to specify the point of origin of the planar section.

— Click to switch to position mode, which enables you to specify the origin point of the planar section.

— Enter the X, Y and Z coordinates of the origin point of the planar section.

Position — Click to display the **Position** dialog. Use this to enter coordinates of the plane origin.

Surface — Use to select a surface on the model which the curves are created on. If nothing is selected, the entire model is used.

 — Click to switch to surface selection mode.

 — Hover over this button to display information about the selected surface.

Tolerance — Enter a value to determine how accurately the toolpath follows the contours of the model.

Surface joining tolerance — Enter a tolerance value to determine if toolpaths can cross gaps between surfaces. For example, to create a continuous toolpath across a gap, enter a large **Surface joining tolerance**. To split toolpath segments between surfaces, enter a small **Surface joining tolerance**.

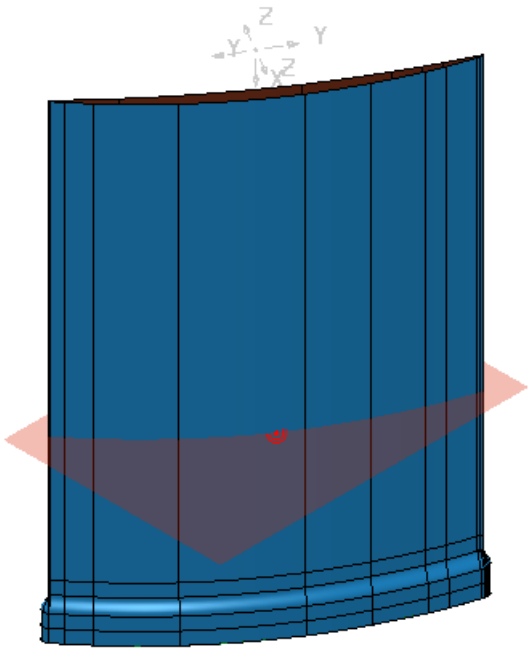
 *For an example of creating a planar section, see Creating a planar section example (page 5).*

Creating a planar section example

To create a planar section:

1. Ensure the **Intelligent Cursor** is enabled.
2. Display the **Planar Section** dialog:
   a. In the Explorer, right-click an individual pattern or boundary.
   b. Select **Curve Editor**. The **Curve Editor** mode-toolbar is displayed.
   c. Click  on the **Curve Creation**  pull-out toolbar.

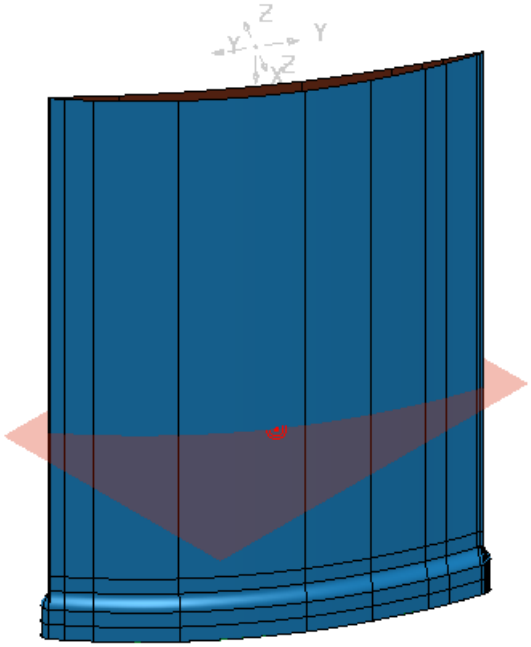
The **Planar Section** dialog is displayed. The position of the origin point is displayed on the model.



The cursor changes to .

3 Click on the model to define a point which the slicing plane passes through. The orientation is determined using the principal editing plane of the active workplane. The coordinates of the curve are displayed in the **Origin** area of the **Planar Section** dialog.

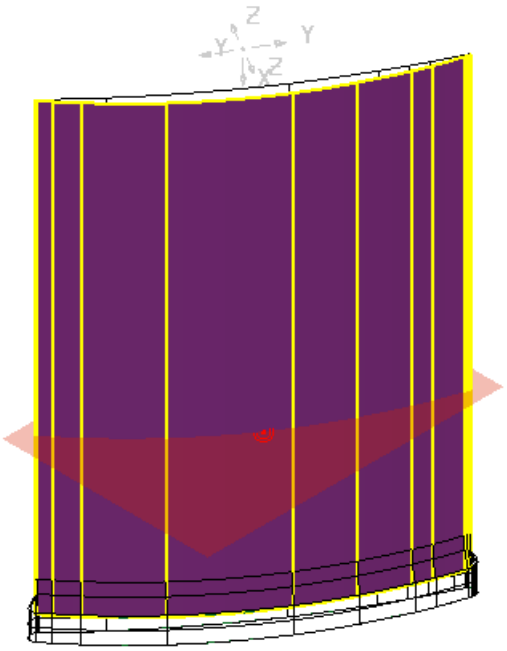
*The **Intelligent Cursor** must be enabled for the origin selection to work through the cursor.*



4 To select a surface:

a Click to switch to surface selection mode. The cursor changes to .

b Select a surface on the model. The selected surface turns purple.

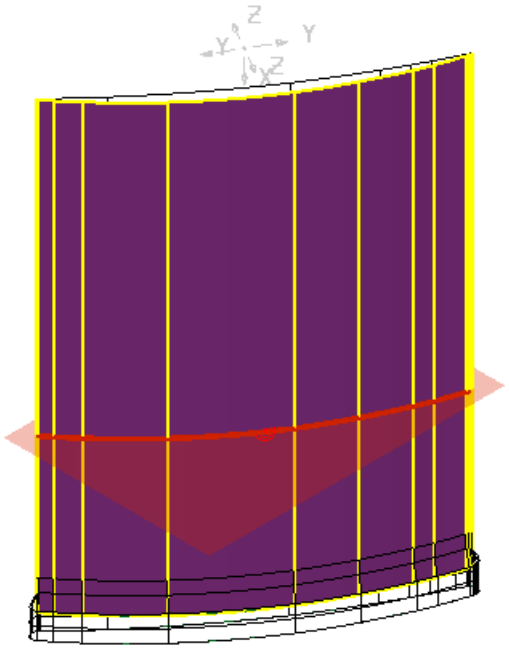


5 Enter:

a A **Tolerance of** 0.01.

b A **Surface Joining Tolerance** of 10.0.

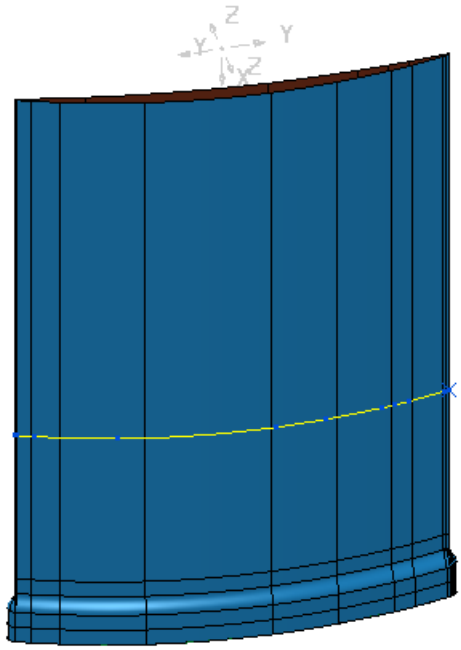
6 Click **Preview**. A preview of the curve is displayed as a thick, red line on the model.



7 Click **Create**. The segments are added to the pattern.

When the plane is previewed or created, an information balloon appears in the dialog. This lists the number of open and closed curves generated.

8 Click **Close** to close the dialog. The curve is displayed on the model.



Generating toolpaths

PowerMILL 2016 contains the following changes and improvements to the generation of toolpaths:

- Project mirroring (see page 10) — Machines symmetric parts and maintains machining characteristics.
- Single blade finishing (see page 17) — Creates toolpaths to machine single-blade aerofoils.
- Rib machining enhancements (see page 28) — The **Offset** list contains the following new options:
  - **Floor only** — Creates toolpaths along the base of the channel.
  - **Constant Z** — Creates toolpaths with a constant Z height.
- Toolpath visualisation (see page 28) — Displays safe areas translucently.

Project mirroring overview

You can mirror a project in PowerMILL. This enables you to machine symmetric parts and maintain the machining characteristics. PowerMILL mirrors all of the geometric entities, except for workplanes that are used for NCProgram or ModelLocation output. In many cases, PowerMILL geometrically mirrors the toolpaths and boundaries if the **Optimise** setting is used. This option preserves post-calculation edits. Otherwise, PowerMILL may need to recalculate entities that could not be transformed. PowerMILL warns you if any edits are not preserved, and places all toolpaths that need recalculating into a group.

*You need to manually recalculate any pre-drill holes created for **Area clearance approach** moves, these toolpaths are placed in a group.*

After you have calculated a toolpath, you may want to edit the model entities. PowerMILL can mark edited toolpaths as changed, depending on your selection from the **Machining mode** list. If your surfaces are in a Thickness Set where the **Machining mode** is set to **Machine** or **Collision**, the parameter `NonIgnoredModelState` marks the toolpath as changed. If you select **Ignore** from the **Machining mode** list, the parameter is not marked as changed. To remove a changed state from a toolpath, you can invalidate or recalculate it. The state is checked during project mirroring. Toolpaths marked as changed are placed in a group for you to manually recalculate: recalculation of toolpaths and their dependants does not occur automatically.

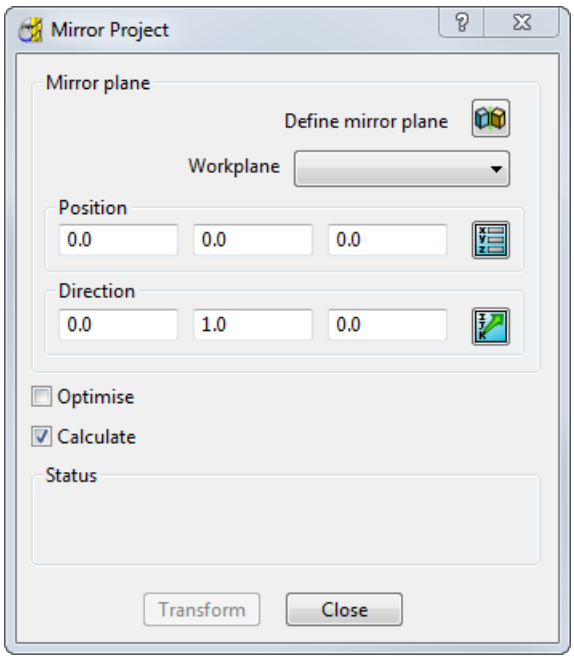
As PowerMILL may have to recalculate the **Area clearance** toolpaths, use **Thickness sets** to specify surfaces to machine or ignore when you create or remove capping surfaces for individual toolpaths.

Use Tools > Options > Import > Model > Calculated toolpaths and boundaries ignore new models to recalculate existing toolpaths without having to manually change the thickness sets.

Options controlling project mirroring are found in the **Mirror project** (see page 11) dialog.

Mirror project dialog

To display the **Mirror Project** dialog, click  on the **Main** toolbar or select Tools > Mirror Project.



The dialog contains the following options:

Define mirror plane — Click to define the mirror plane interactively. This displays the **Define Mirror Plane** mode-toolbar (see page 12).

Workplane — Select the workplane along which the project is mirrored. This activates the selected workplane.

Position — Enter the X, Y, and Z coordinates of the mirror plane.

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

Direction — Enter the component of the mirror plane along the X, Y and Z axes.

— Click to display the **Direction** dialog. This provides additional options to edit orientation vectors.

Optimise — Select to mirror the toolpath geometry, including the strategy type and editing history, instead of recalculating it.

Calculate — Select to automatically recalculate the toolpaths and boundaries.

Transform — Click to perform the transformation. The **Status** area of the dialog displays the progress of the transformation.

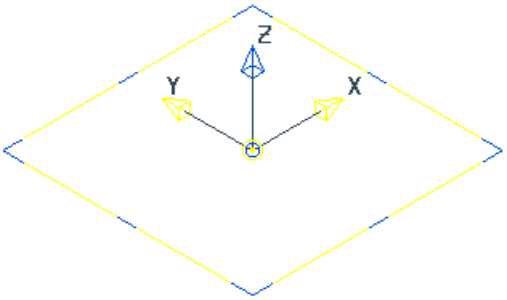
For an example of how to mirror a project, see Mirroring a project example (page 14).

Define Mirror Plane mode-toolbar

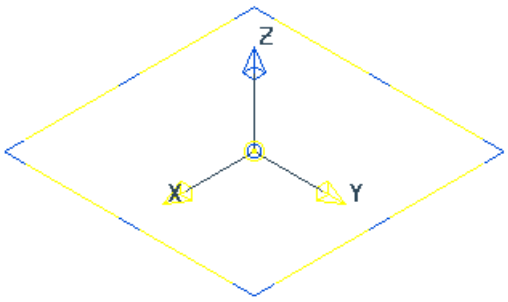
Use the **Define Mirror Plane** mode-toolbar to define the workplane of the project you want to mirror.

When you mirror a workplane, the Y axis is flipped to ensure the creation of a mathematically correct axis.

Mirroring in YZ converts this:



to this:



The **Define Mirror Plane** mode-toolbar contains the following options:

Align plane to XY plane — Align mirror plane to XY in preparation for mirroring in Z.

Align plane to YZ plane — Align mirror plane to YZ in preparation for mirroring in X.

Align plane to ZX plane — Align mirror plane to ZX in preparation for mirroring in Y

Define plane using line —Define the line by pressing the left mouse button to give a start point and release it to give an end point.

Workplane origin — When selected, uses the active workplane as the origin. If no workplane is active then uses the global coordinate system as the origin.

Bounding box origin — When selected, the origin is the centre of the bounding box containing all the entities.

Move origin — When selected, you can move the origin graphically by dragging or by entering coordinates using , , or and in the **Status** bar.

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

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

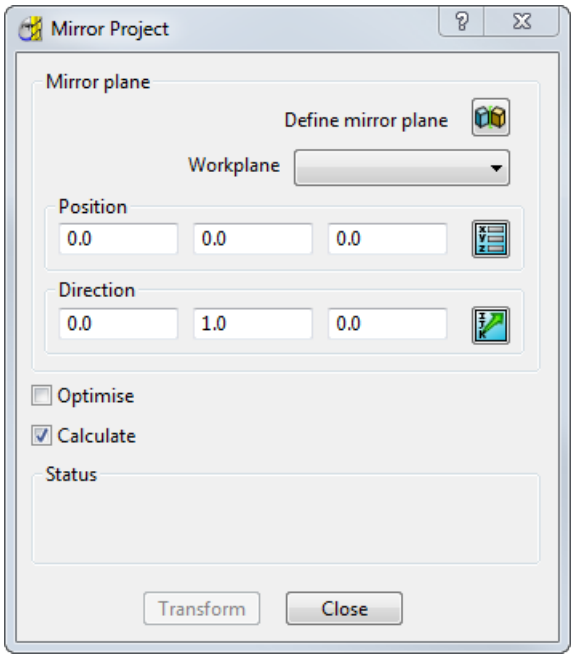
Accept changes — Click to accept and keep all the created curves.

Cancel changes — Click to delete all the created curves.

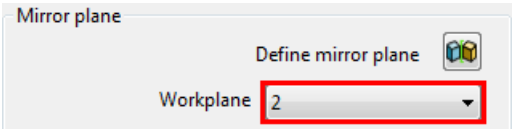
Mirroring a project example

To mirror a project:

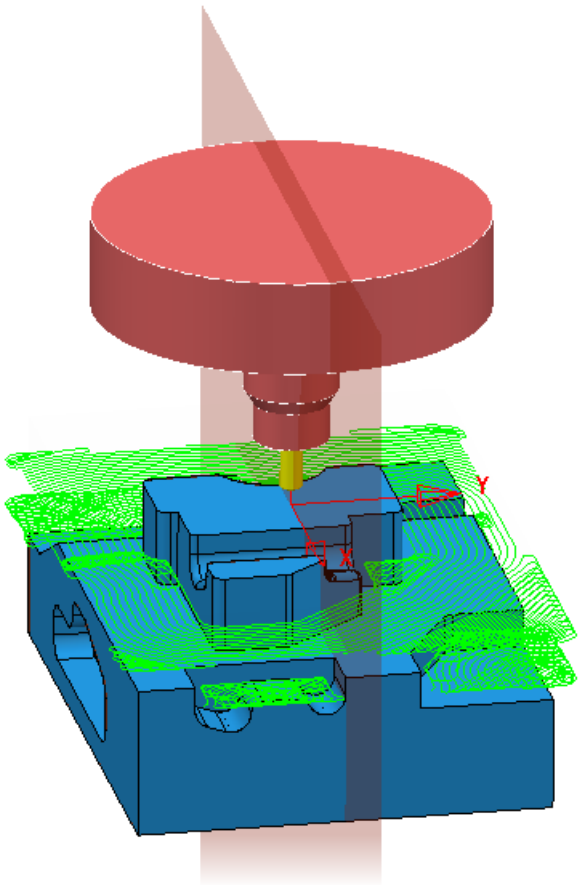
1 Click on the **Main** toolbar or select Tools > Mirror Project to display the **Mirror Project** dialog.



2 Select a workplane from the list. This represents the coordinate system that you want to mirror the project along. In this example, select **2** from the **Workplane** list.



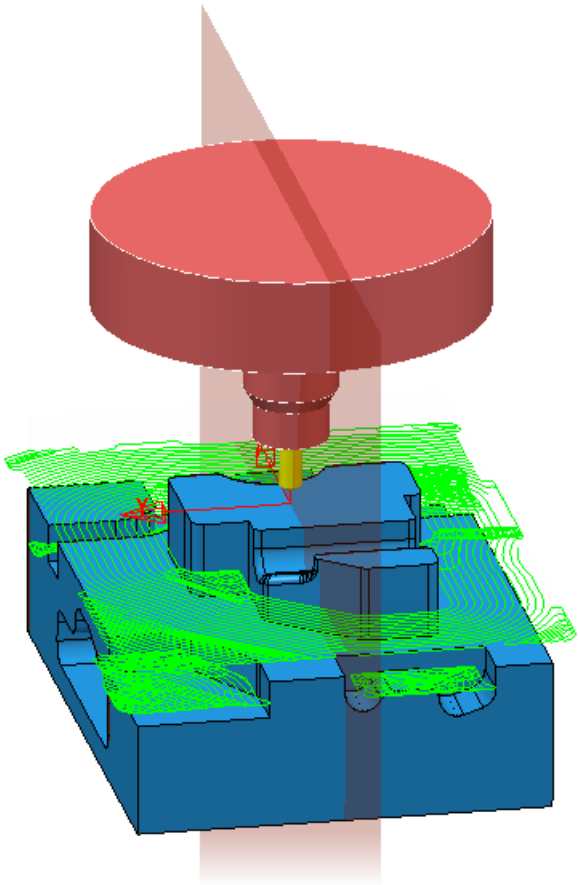
This displays the workplane on the model.



3 Select **Calculate** to automatically recalculate toolpaths and boundaries.

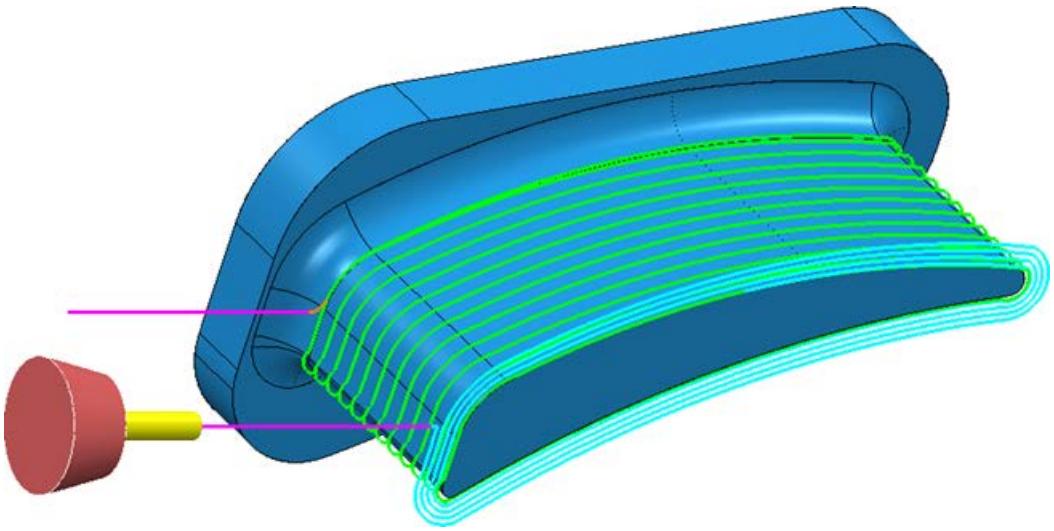
4 Click **Transform**. The **Status** area of the dialog displays the progress of the transformation.

5 The mirrored project is displayed in the graphics window.



Single Blade Finishing Overview

Use the **Single Blade Finishing** strategy to create toolpaths to machine single blade aerofoils.



The pages associated with **Single Blade Finishing** are:

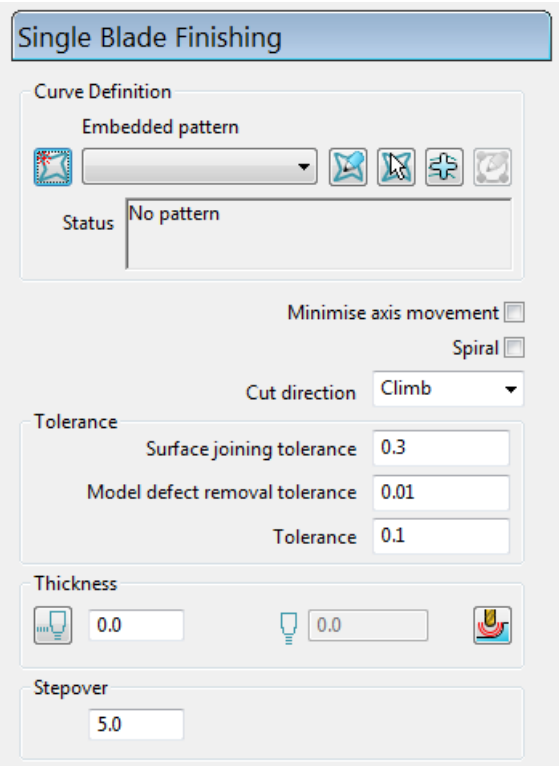
- **Single Blade Finishing** (see page 17) — The main page used to create toolpaths to machine single blade aerofoils.
- **Approach** (see page 23) — Settings to control the height and stepdown of the ramp.

The remaining pages are common toolpath creation controls.

Single blade finishing

Use the **Single Blade Finishing** strategy to create toolpaths to machine single-blade aerofoils.

To display the **Single Blade Finishing** strategy page, select **Single Blade Finishing** from the **Blisk** page of the **Strategy Selector** dialog. The page contains the following options:



Curve definition specifies the curves bounding the area you want to machine and adds the machining properties to those curves. The pattern must contain two or more curve segments.

Create pattern — Click to create a new empty pattern. You can then collect wireframe curves and add them to the empty pattern using the **Collect curves** button.

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

Curve Editor— Click to display the **Curve Editor** for the selected pattern. If no pattern is selected, PowerMILL creates a new pattern. Use the **Curve Editor** to create planar sections that are used to define single blade finishing toolpaths.

Select picked pattern — Click to select a pattern by picking in the graphics window, rather than by name in the **Select pattern** list.

Clicking  displays the **Pick Tool Pattern** mode-toolbar.

Select a pattern in the graphics window to close the **Pick Tool Pattern** mode-toolbar and display the pattern in the **Selected Pattern** field.

 **Collect curves** — Click to copy the selected curves into the pattern.Use this option if you want to collect wireframe curves.

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:

1. Individual surface boundary curves.
2. Boundary curves around a selection of surfaces.
3. Model wireframe geometry.
4. Existing pattern or boundary segments.
5. Click  to display the **Pattern Collect**  mode-toolbar. For more information, see the collecting curves example.

 **Select curves** — Displays the **Single Blade Curve Definition** mode-toolbar (see page 24). This enables you to make changes or setup the curve definition.

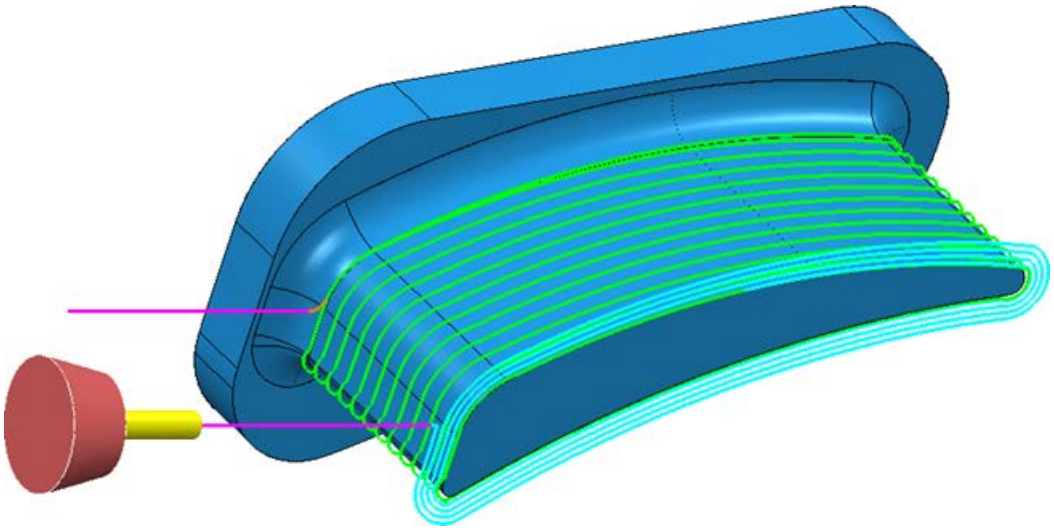
Status — Displays information about the pattern you have selected.

- **No pattern** — No pattern is selected.
- **Not defined** — The pattern contains more than 3 segments.
- **Valid** — The pattern contains 2 closed curves (and optionally 1 open curve).
- **Invalid** — The pattern does not contain any segments or does not contain 2 closed segments.

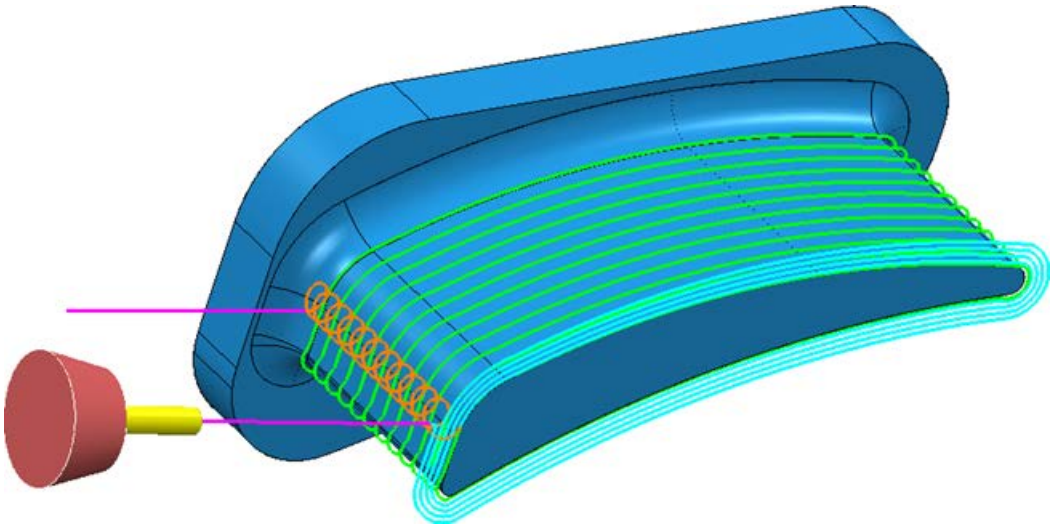
Minimise Axis Movement — Select to follow the surface normal less accurately on curved parts of the model. This creates a smoother and faster machine motion. The cutting tool rolls around the curved parts of the blade, for example, the leading and trailing edges. Select this option for roughing or semi-finishing toolpaths where cycle time is more important than surface finish.

Spiral — Select to enable the toolpath to spiral around the aerofoil. Spiral toolpaths are typically more efficient than non-spiral toolpaths as they do not contain lead-in or lead-out moves. Spiral toolpaths also avoid shock-loading the cutting tool as it leads into and out of stock. This creates a consistent surface finish and improves the cutting life of the tool.

Toolpath with **Spiral** selected:



Toolpath with **Spiral** deselected:



Cut direction — Select a milling style.

Climb — Select to create toolpaths using only climb milling, where possible. The tool is on the left of the machined edge when viewed in the direction of tool travel.

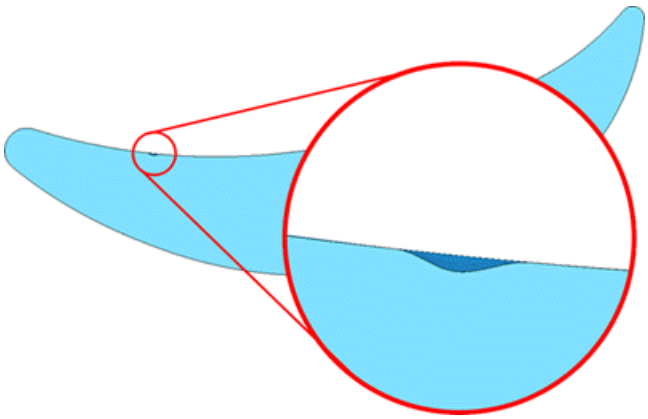
Conventional — Select to create toolpaths using only conventional or upcut milling, where possible. The tool is on the right of the machined edge when viewed in the direction of tool travel.

Tolerance — Options to specify how accurately the toolpath follows the contours of the model.

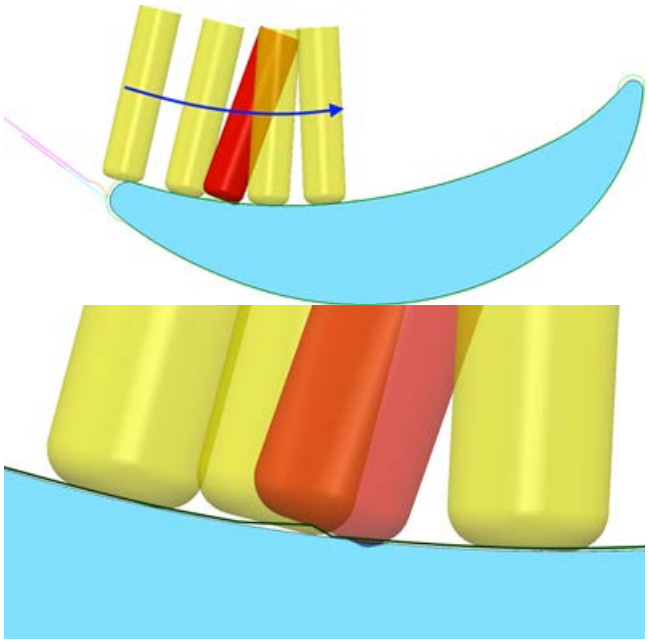
Surface joining tolerance — Enter a tolerance value to determine if toolpaths can cross gaps between surfaces. For example, to create a continuous toolpath across a gap, enter a large **Surface joining tolerance**. To split toolpath segments between surfaces, enter a small **Surface joining tolerance**.

Model Defect Removal Tolerance — Use this option to smooth toolpaths and ignore small defects in your CAD model. Enter the maximum defect thickness that is ignored by the toolpath. **Model Defect Removal Tolerance** improves the machining quality, as the tool does not dwell on defects below a specified size.

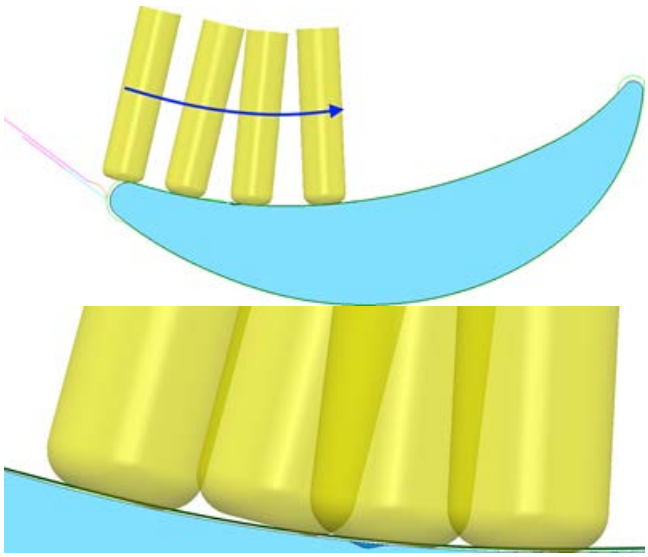
This aerofoil contains a defect that is 0.25 mm deep:



If **Model defect tolerance** is set to **0.01** mm (default) PowerMILL generates a toolpaths where the tool tip rolls into the defect. The tool axis wobbles, which results in an unacceptable machine motion:



If you enter a **Model defect tolerance** that is greater than the defect size (for example, **0.5** mm), PowerMILL ignores the defect and produces a smooth toolpath:



Tolerance — Enter a value to determine how accurately the toolpath follows the contours of the model.

Thickness — Enter the amount of material to be left on the part.

Component thickness — Click to display the **Component thickness** dialog, which enables you to specify the thicknesses of the different surfaces.

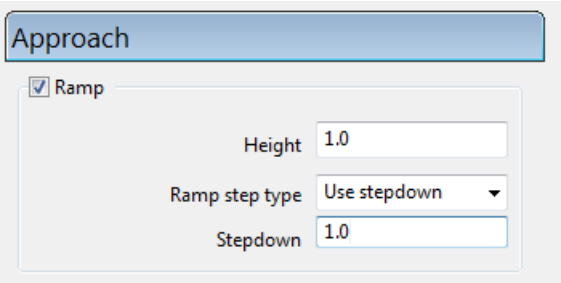
Stepover — Enter the distance between successive machining passes.

*You can edit the ramp options on the **Approach** page of the **Single Blade Finishing** strategy (see page 23).*

Single Blade Finishing Approach page

Use the **Ramp** option on the **Approach** page to engage the cutting tool into the stock in a controlled manner. The **Ramp** option is a 5-axis extension of the main toolpath and preserves lead and lean tool axis. As the **Ramp** move is created as part of the main toolpath calculation, it cannot be edited without re-calculating the toolpath. This is in contrast to the functionality of other **Ramp** options within PowerMILL, for example, in the **Leads and Links** dialog.

Ramp options are displayed on the **Approach** page of the **Single Blade Finishing** strategy.



Ramp — Select to enable you to edit the ramp options.

Height — Enter the height above the toolpath segment at which the ramp starts. This is measured with respect to the tool axis.

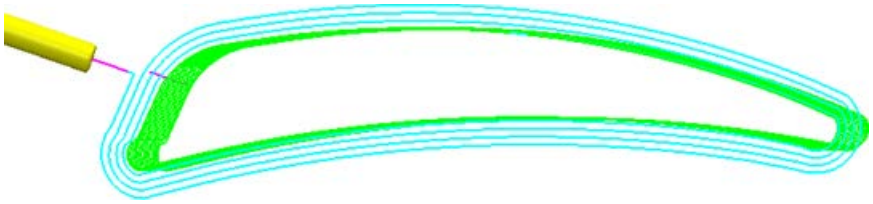
Ramp step type — Select **Use stepdown** or **Use angle**.

If you select **Use stepdown**, enter a value in the **Stepdown** field.

If you select **Use angle**, enter a value in the **Angle** field or click **Angle** to display the **Measure** mode-toolbar.

Stepdown — Enter the distance between different machining levels. This option is only available if you select a **Ramp step type** of **Use stepdown**.

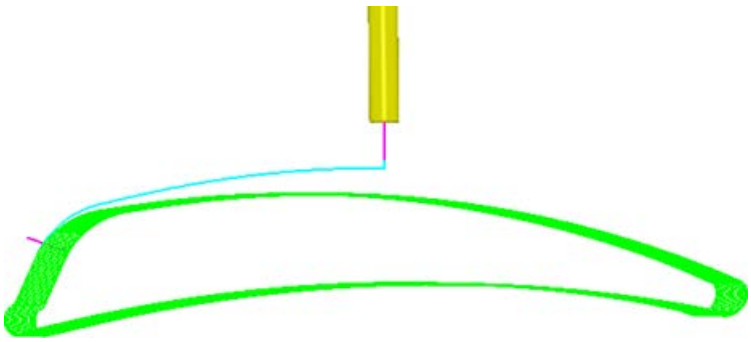
For example, if you enter a **Height** of 20 and a **Stepdown** of 5:



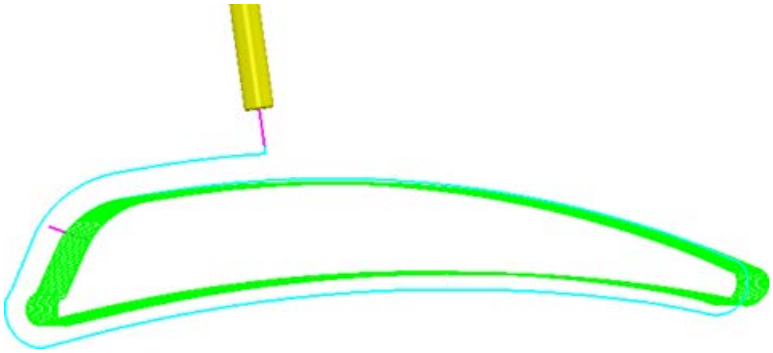
If you enter a **Height** of 20 and a **Stepdown** of 20:

Angle — Enter the angle at which the ramp enters the material or click **Angle** to display the **Measure** mode-toolbar. This option is only available if you select a **Ramp step type** of **Angle**.

For example, if you enter a **Height** of 20 and an **Angle** of 5:



If you enter a **Height** of 20 and an **Angle** of 1:



Single Blade Curve Definition mode-toolbar

Use the new **Single Blade Curve Definition** mode-toolbar to:

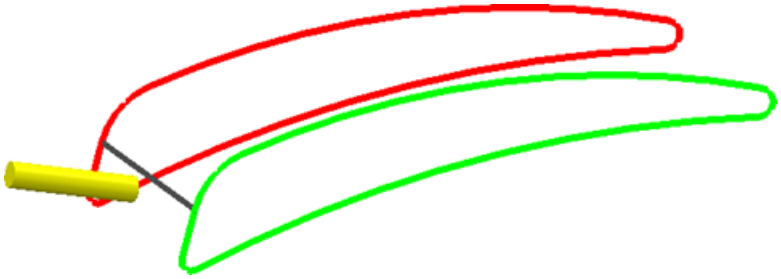
- specify the curves bounding the area you want to machine.
- add machining properties to the curves you specify.

Use the **Single Blade Curve Definition** mode-toolbar to select a start curve, an end curve, intermediate curves and to specify the machining properties. To display the **Single Blade Curve Definition** mode-toolbar, click **Select curves** on the **Single blade finishing** strategy page (see page 17).

The mode-toolbar contains the following options:

Select start curve — Select a closed curve where you want the machining to start.

Select end curve — Select a closed curve where you want the machining to end.



Select seam curve — Select an open curve that joins the start and end curves. This curve defines the start point of each toolpath segment and must connect to the start and end curves.

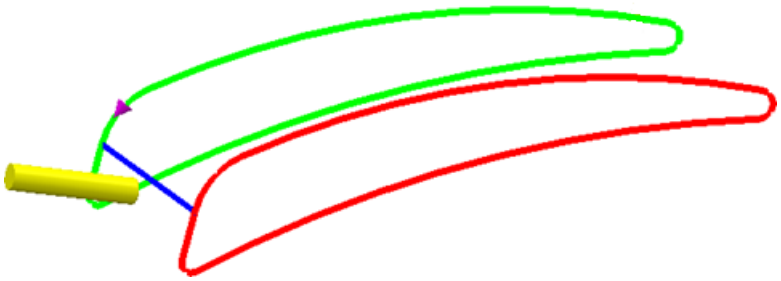
To minimise the likelihood of leaving a dwell mark on the machined component, the seam curve should be located by the leading or trailing edge of the blade.

Select starting point — If there is no seam curve to select, click **Select starting point** to define the starting point of the curve.

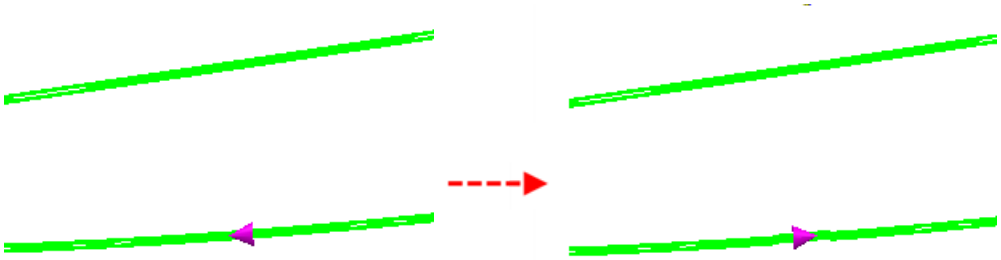
In order to select a starting point, you must have a valid tool for the strategy, with a diameter greater than zero.

Reverse drive curves —Swaps the start and end drive curve.

The drive curve is reversed:

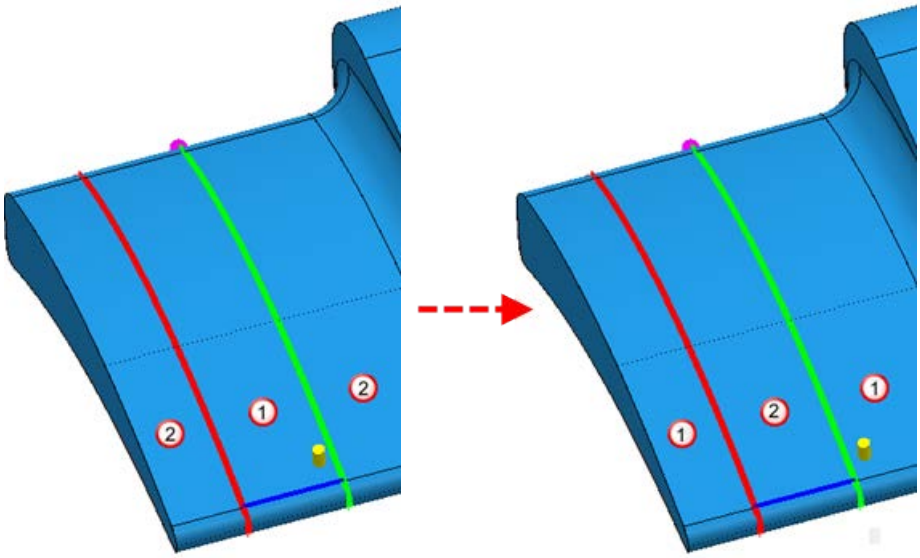


Change cut direction —Changes the cut direction from conventional to climb milling or from climb to conventional milling.



Reverse side to machine —Toggles the side of the curve that is machined. For example, click to toggle the creation of toolpaths from the area between the two curves, to the area outside the two curves. A tool is displayed to indicate which side of the curves is machined.

Click **Reverse side to machine** to toggle the machining side.



① Area in which toolpaths are generated.

② Area in which toolpaths are not generated.

The tool is displayed in the machining area.

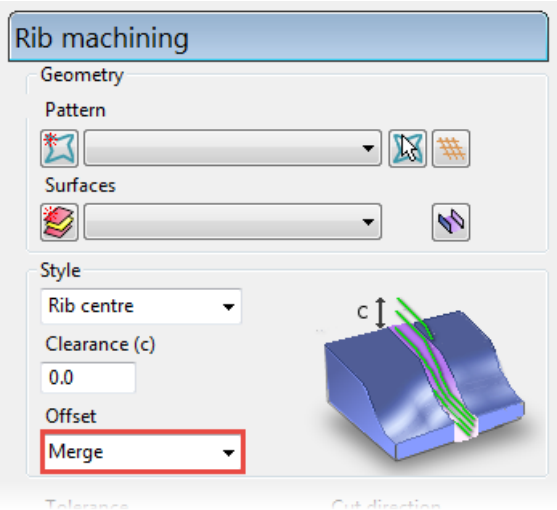
Curve definition status — Hover to display curve definition information.

Accept Changes — Click to accept and keep the changes.

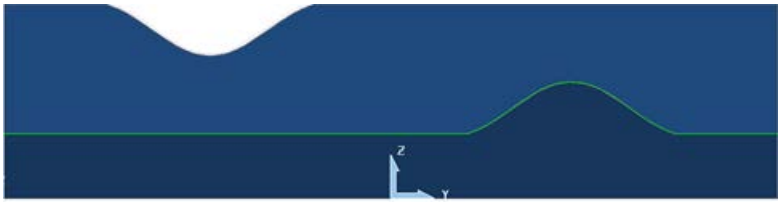
Cancel Changes — Click to delete the changes.

Rib machining enhancements

There are enhancements to the **Rib machining** strategy page. The **Offset** list contains the following new options:



- **Floor only** — Select to create a toolpath along the base of the channel.



- **Constant Z** — Select to create toolpaths with a constant Z height.

PowerMILL reorders **Constant Z** toolpaths so that segments within the same pocket are cut together. This decreases the length of connecting moves and makes the toolpath more efficient.

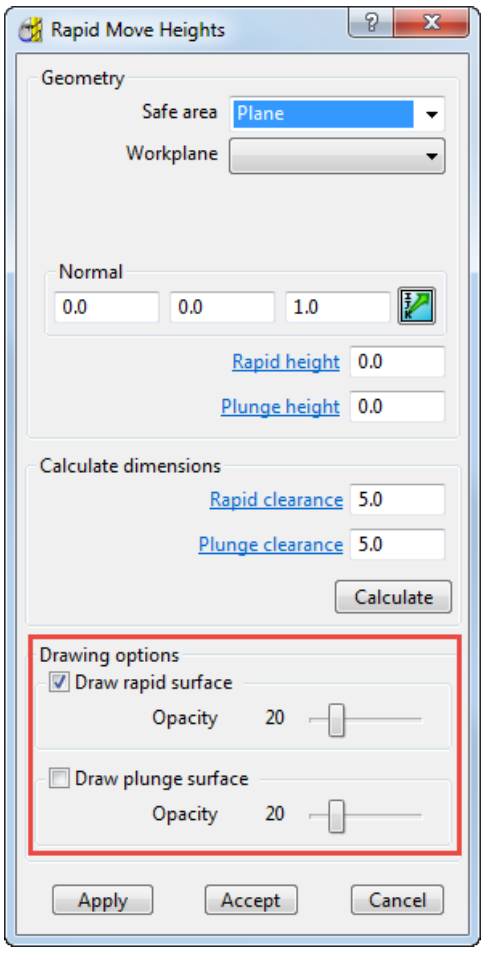
Toolpath visualisation

Toolpath visualisation has the following enhancements:

- **Rapid move heights** dialog and strategy page — There is a new **Drawing options** area with options to **Draw rapid surface**, **Draw plunge surface** and adjust the opacity of both surfaces.
- **Toolpath** toolbar — There are two new buttons on the **Toolpath** toolbar, which enable you to **Draw Rapid Surface** and **Draw Plunge Surface** .

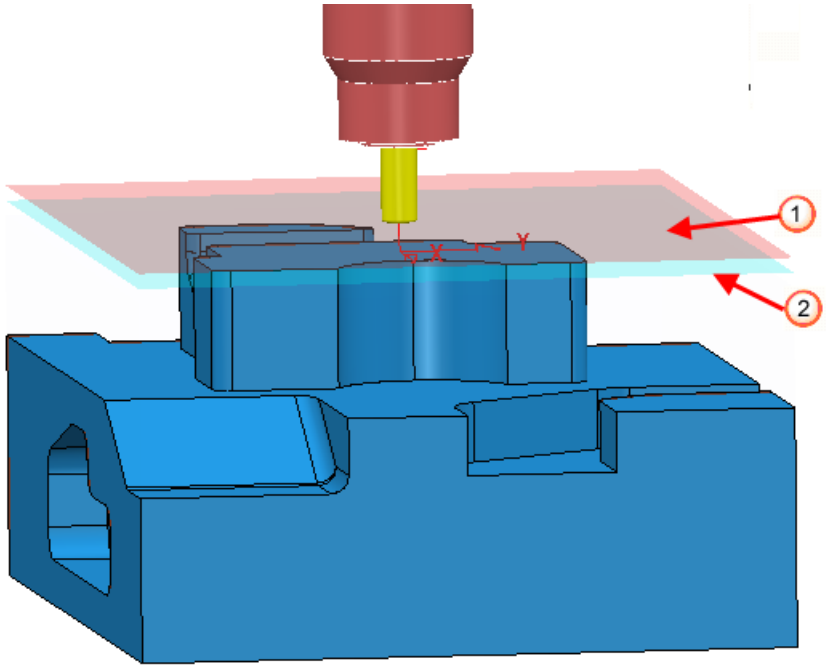
Drawing options

The new options on the **Rapid move heights** dialog and the strategy pages are:



Draw rapid surface — Select to draw the surface of the safe area. Use the slider to set the level of opacity of the rapid surface.

Draw plunge surface — Select to draw the surface beyond which the plunge moves are made. This is the region of space beyond which approach moves are made at the plunge rate. Use the slider to set the level of opacity of this surface.



1. Rapid surface drawn, with an opacity of **25**.
2. Plunge surface drawn, with an opacity of **25.**

*For a box-shaped surface area, the plunge surface is the same as the rapid surface. Therefore, **Draw plunge surface** is unavailable for box-shaped surface areas.*

Verifying toolpaths

PowerMILL 2016 contains the following changes and improvements to the outputting of toolpaths:

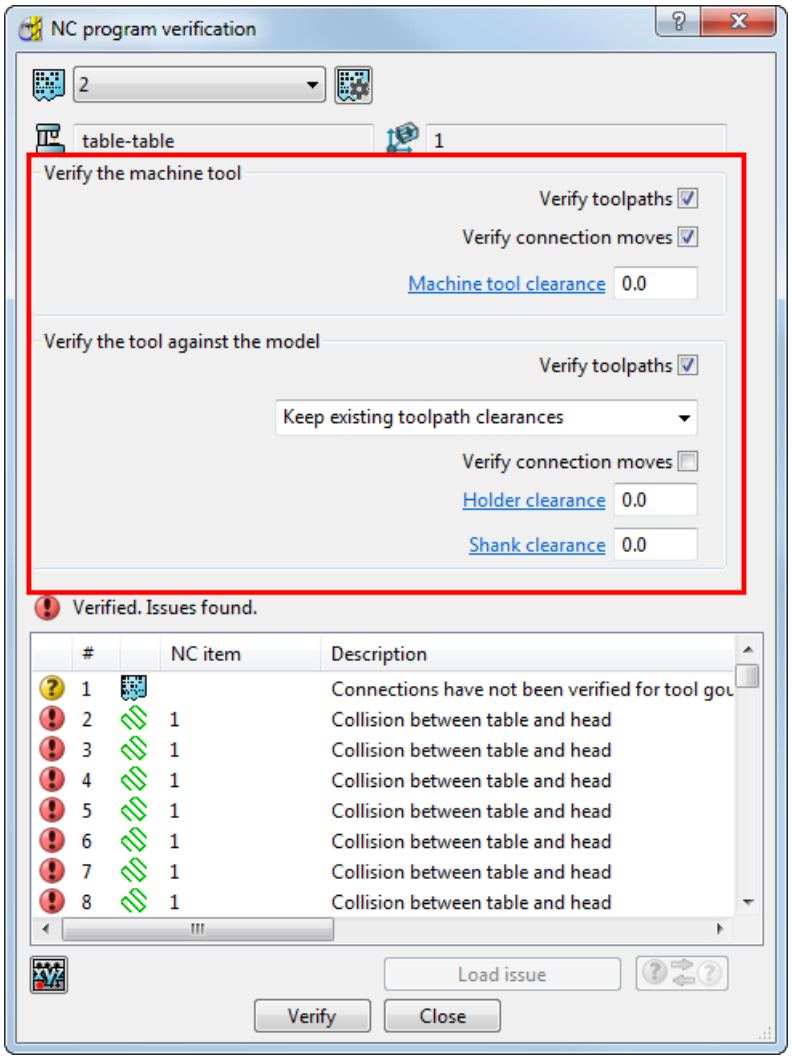
- NC program verification enhancements (see page 30) — Enables you to turn off machine tool checks for connections or toolpaths.
- Automatic collision checking enhancements (see page 32) — Enables you to select what is being collision checked in **Model Area Clearance** and **Model Rest Area Clearance** strategies.

NC program verification enhancements

Enhancements to the verification of NC programs enable you to turn off machine tool checks for connections or toolpaths. This is in addition to previous options that enable you to control shank and holder collisions for connections or toolpaths.

Enhancements are found on the **NC program verification** dialog and the **Verification** tab of the **NC Preferences** dialog. To display the **NC program verification** dialog, select **Verify all** from the **NC Programs** context menu. Alternatively, to verify a single NC program, from the individual NC program context menu, select **Verify**.

The enhancements to the NC program verification dialog are as follows:



Verify the machine tool

This area contains options which enable you to collision check the machine tool against itself and other components:

Verify toolpaths — Select to check the toolpaths for collisions involving the machine tool.

Verify connection moves — Select to check the connections for collisions involving the machine tool.

Machine tool clearance — Enter the machine tool clearance you want to use for verification. Entering a value clears the verification for the NC program.

Verify the tool against the model

This area contains options which enable you to collision check the shank and holder against the model. This does not affect checks with the machine tool.

Verify toolpaths — Select to check for collisions between the tool, the holder and the model.

Increase toolpath clearances — Select to check for collisions in all toolpaths.

Keep existing toolpath clearances — Select to check for collisions in all toolpaths that are not verified.

Verify connection moves — Select to include connection moves in the verification.

Holder Clearance — Enter the minimum allowable distance between the model and the tool holder profile.

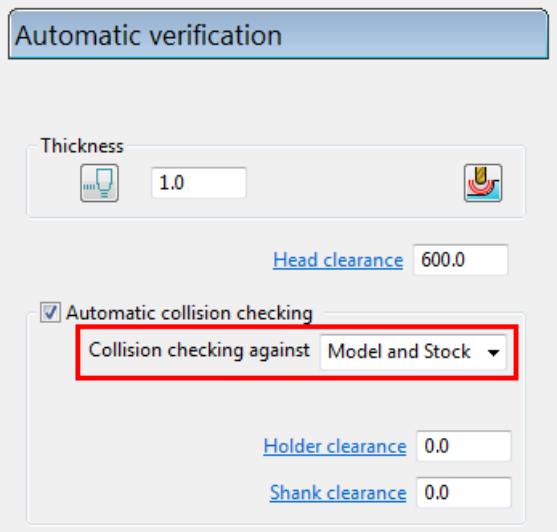
Shank Clearance — Enter the minimum allowable distance between the model and the tool shank profile.

*Click **Holder clearance** or **Shank clearance** to display the **Measure** mode toolbar.*

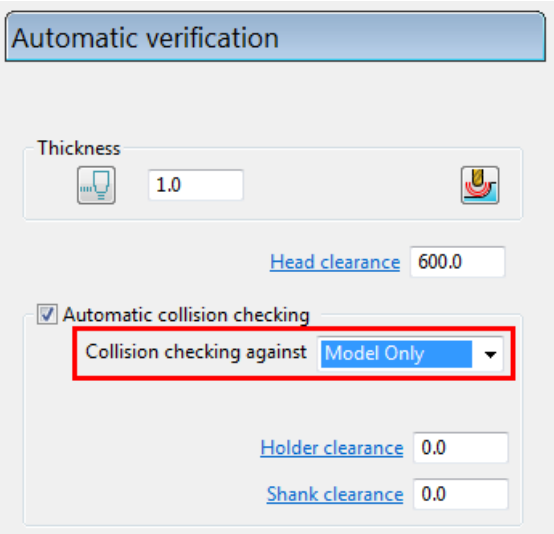
Automatic collision checking enhancements

You can now select what to collision check against, in **Model Area Clearance** or **Model Rest Area Clearance** strategies. From the **Collision checking against** list on the **Automatic verification** page, you can select to collision check:

- the **Model and Stock**.



- the **Model Only**.



The **Model Only** option is usually faster to calculate than the **Model and Stock** option. However, this option does not check for collisions between the tool holder and the stock so take care when running these toolpaths.

Outputting toolpaths

PowerMILL 2016 contains the following changes and improvements to the outputting of toolpaths:

- Carousel tool changes (see page 34) — Simulate tool changes on a carousel or a tool changer.
- Tool Position (see page 35) — Display the coordinates of the tool when it is attached to the machine tool.
- General enhancements (see page 36):
  - Automatic transparency of machine tools — PowerMILL automatically displays machine tools as transparent if they obstruct the view of a model during a simulation.
  - Overhang adjustment on collision — Modify only the tool overhang after a collision.
  - Tool drawing during simulation — Toggle the drawing of a tool during simulation.

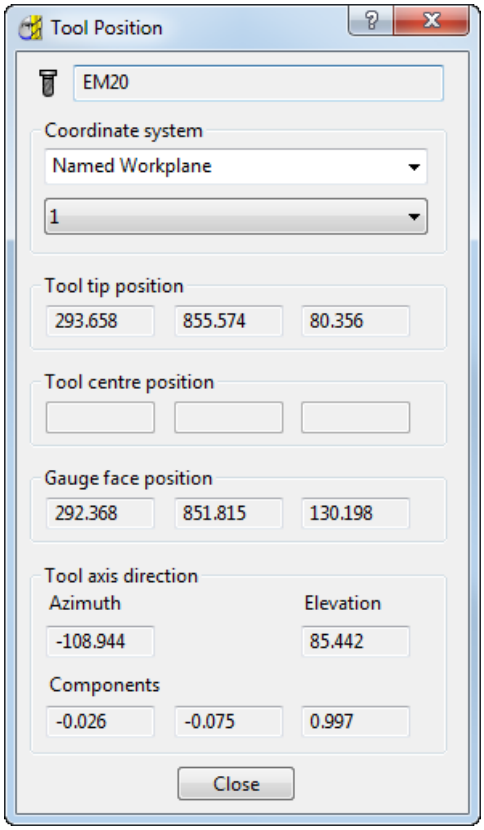
Simulating carousel tool changes

You can now simulate tool changes on a carousel or a tool-changer, where the tool moves independently from the machine tool head. This enables you to identify issues with carousel tool changes, such as collisions. To display carousel tool changes, use the **Simulation** toolbar to simulate a toolpath or NC program in the same way as in previous versions of PowerMILL.

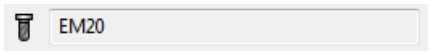
If you load an .mtd file, the machine tool is automatically displayed. Use the options on the **Simulation** toolbar to control the simulation.

Tool Position dialog

Use the **Tool Position** dialog to display the coordinates of the tool that is currently being simulated. The coordinates update as you simulate a toolpath, or edit the position of the machine tool.



To display the **Tool Position** dialog click  on the **Machine Tool** toolbar, or on the **Machine Tool Position** dialog. The dialog contains the following options:



— Displays the name of the tool attached to the machine tool.

Coordinate system — Select a coordinate system from the list:

- **Current model location** — Select to use the current workplane.
- **Active NC Program output workplane** — Select to use the output workplane specified in the active NC program.
- **Global transform** — Select to use the global coordinate system.
- **Active Toolpath model location** — Select to use the workplane of the active toolpath.
- **Active Workplane** — Select to use the active workplane.
- **Named Workplane** — Select to enable you to choose a workplane from the workplane list.

Tool tip position — Displays the **X**, **Y** and **Z** coordinates of the tool tip.

Tool centre position — Displays the **X**, **Y** and **Z** coordinates of the tool centre.

Gauge face position — Displays the **X**, **Y** and **Z** coordinates of the gauge face.

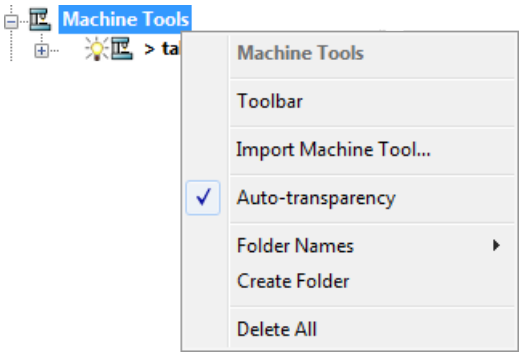
Tool axis direction — Displays the **Azimuth** and **Elevation** angles of the tool, and the **Components** of the tool (the X, Y and Z Cartesian coordinates).

General enhancements

PowerMILL 2016 contains the following general enhancements:

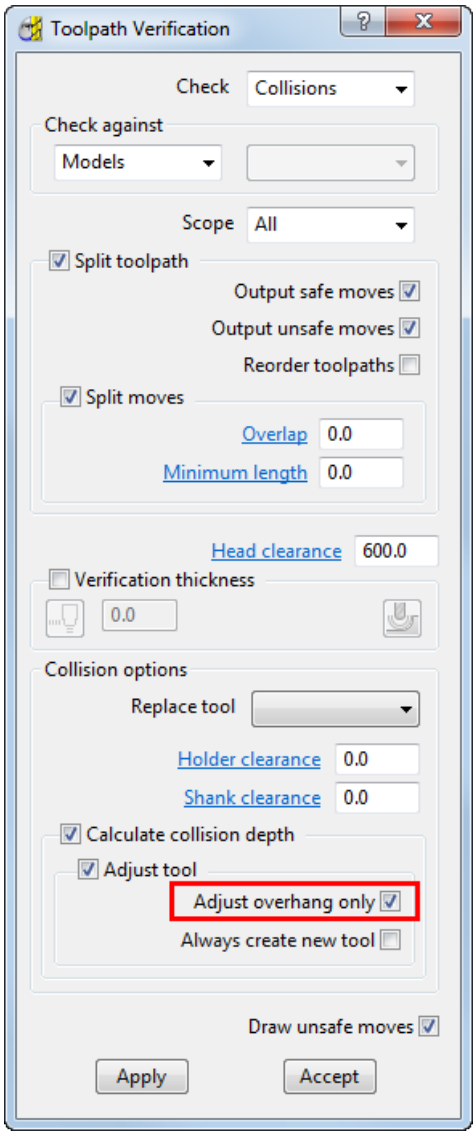
Automatic transparency of machine tools

When you rotate a machine tool, it can be difficult to see the area of interest. This is true in **Dynamic Machine Control** mode, where the machine tool rotates quickly. Machine tool parts are now translucent if they are between the user and the table attach point. The tool, however, is always opaque. To disable automatic transparency, right-click **Machine Tools** in the Explorer and deselect **Auto-transparency.**



Adjusting the overhang on collision

There is a new **Adjust overhang only** option on the **Toolpath Verification** dialog. When selected, this modifies the tool overhang value to avoid collisions. This option does not modify the length of the cutter, so shank collisions still occur. To display the **Toolpath verification** dialog, click on the **Main** toolbar.



Drawing the tool during simulation

There is a new button on the **Simulation** toolbar that controls whether a tool is drawn during simulation. always draws the tool and does not draw the tool.

Automating PowerMILL

Delcam Exchange can now extract the metadata from 3rd party CAD files and transfer them to PowerMILL during the translation process. This metadata is accessible through parameters which are associated with groups, workplanes and geometries. These are used to store information relevant to the machining of an item. This is useful if you want to write macros or create bespoke automation software.

For more information, see the MTD User Guide.

Index

T