

Advanced
Manufacturing
Solutions

PowerMILL 2015 R2

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

www.powermill.com



What's New

PowerMILL 2015 R2

What's New



Issue 1

PowerMILL

Copyright © 1996 - 2015 Delcam Ltd. All rights reserved.

Delcam Ltd has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of the software.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence.

Delcam Ltd grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

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.)

Contents

Summary of new features	1
Preparing toolpaths	4
Tool database improvements.....	4
Database Manager dialog	5
Edit Database Tools dialog.....	10
Generating toolpaths	12
Ribs.....	13
Rib Machining Overview.....	14
Improved collision avoidance.....	29
Drilling enhancements	30
Drilling from the stock.....	30
Automatic verification enhancements.....	31
Drilling non-planar surfaces.....	34
Enhancements to area clearance and Vortex drilling	34
Editing orientation vectors.....	36
Specify Changes tab	36
Machine axis control tab.....	37
Curve fitting toolbar.....	38
Machine Tool Simulation Issues	41
Maintaining tool speed after ramping.....	42
Aligning the tool with the toolpath	43
Limit toolpath enhancements	44
Verifying toolpaths	45
Simulating tool changes.....	46
Outputting toolpaths	47
Verifying NC programs.....	47
NC program verification dialog	48
NC program errors in the Explorer	51
Creating a new tool after toolpath verification	52
User interface	53
Customise Keyboard Shortcuts dialog.....	53
Adding expressions to custom toolbars	56

Select Concave Radii dialog	58
Editing the width of toolpath lines	60
Automating PowerMILL	61
Macro enhancements	61
Macro Debugger enhancements.....	62
Index	63

Summary of new features

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

PowerMILL 2015 R2 contains the following new features and enhancements:

Preparing toolpaths

- Tool database improvements (see page 4):
 - **Database Manager** dialog (see page 5) — Create tool tags and connection types to make searching for tools easier.
 - **Edit Database Tools** dialog (see page 10) — Assign connection-types and tags to selected tools.

Generating toolpaths

- Rib machining(see page 13):
 - **Rib Pattern** mode-toolbar (see page 22) — Sketch the centreline of ribs/channels in a mold.
 - **Rib Surfaces** mode-toolbar (see page 24) — Automatically select the walls and base surfaces of the channels associated with the sketched centreline of the ribs/channels.
- Improved collision avoidance (see page 29) — Check for collisions against the stock.
- Drilling enhancements (see page 30):
 - Drilling from the stock (see page 30) — Create drilling toolpaths that start at the top of the stock model.

- Automatic verification enhancements (see page 31) — Collision check against the stock model. You can specify the type of toolpath generated after detecting a collision.
- Drilling non-planar surfaces (see page 34) — Calculate a drilling toolpath where the top of the drill move is defined by a block surface.
- Enhancements to area clearance and Vortex drilling (see page 34) — Specify the diameter of pre-drilled holes before machining area clearance toolpaths
- Editing orientation vectors (see page 36):
 - **Specify Changes** tab (see page 36) — Edit orientation vectors in a restricted toolpath region, without modifying the tool axis.
 - **Machine axis control** tab (see page 37) — Fine-tune orientation vectors using Cartesian coordinates.
- Curve fitting toolbar (see page 38) — Convert selected curve segments to a spline. You can also choose to keep curves embedded.
- **Machine Tool Simulation Issues** dialog (see page 41) — Display a list of issues and warnings, with options for collision checking and clearance values.
- Maintaining tool speed after ramping (see page 42) into the stock.
- Aligning the tool with the toolpath (see page 43) — Align the tool axis direction with the toolpath.
- Undo toolpath limit (see page 44) — Undo the last point added to a limiting polygon.

Verifying toolpaths

- Simulating tool changes (see page 46) — Simulate the tool change cycle of a machine tool.

Outputting toolpaths

- Verifying NC programs (see page 47):
 - **NC program verification** dialog (see page 48) — Verify all of your NC programs at the same time and display issues and warnings in a new dialog.
 - NC program errors in the Explorer (see page 51) — Display any issues and warnings associated with NC programs in the Explorer.
- Creating a new tool after toolpath verification (see page 52) — Always create a non-colliding tool after a collision.

User interface

- **Customise Keyboard Shortcuts** dialog (see page 53) — Create keyboard shortcuts to perform PowerMILL commands without the use of a mouse.
- Adding expressions to custom toolbars (see page 56).
- **Select Concave Radii** dialog (see page 58) — Select concave or convex areas of the model.
- Editing the width of the toolpath (see page 60) — Edit the display width of the toolpath in pixels.

Automating PowerMILL

- Macro enhancements (see page 61) — Display entities and hide GUI items when you use mode toolbars.
- **Macro Debugger** enhancements (see page 62) — Edit variable assignments in the **Macro Debugger** dialog.

Preparing toolpaths

PowerMILL 2015 R2 contains the following changes and improvements to the preparation of toolpaths:

- Tool database improvements (see page 4):
 - **Database Manager** dialog (see page 5) — Create tool tags and connection types to make searching for tools easier.
 - **Edit Database Tools** dialog (see page 10) — Assign connection-types and tags to selected tools.

Tool database improvements

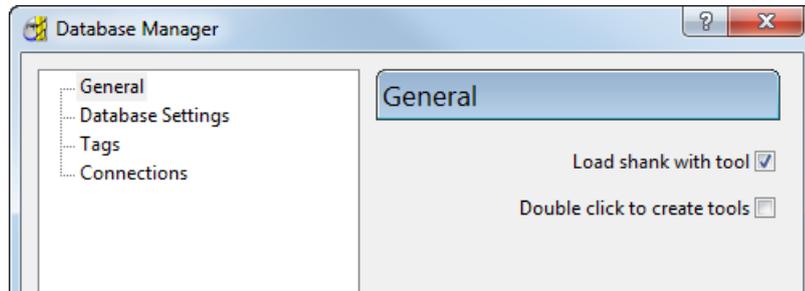
To make it easier to search the Tool and Holder databases, you can create and assign tool tags and connection-types to tools and holders. You can then search for tools based on their tags and connections.

- Use the new **Database Manager** dialog to create a tool tag or connection (see page 5).
- Use the new **Edit Database Tools** dialog to assign the tool tag or connection (see page 10).

The tool **Database Manager** dialog controls all database settings. This includes the **General** and **Database** settings that were previously displayed on the **Tools > Database** page of the **Options** dialog.

Database Manager dialog

Use the tool **Database Manager** dialog to control all database settings. To display the **Database Manager** click  on the **Tool** toolbar.

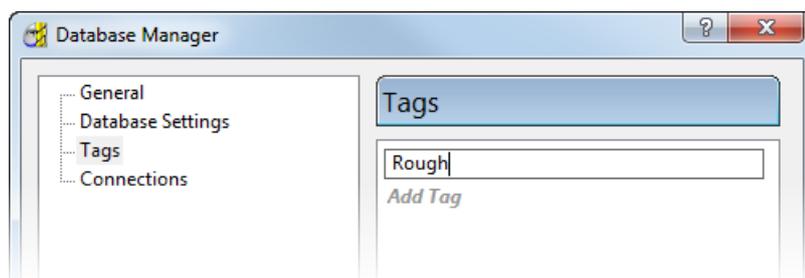


There are several pages associated with the **Database Manager** dialog:

- **General** — Settings to determine how the tool database loads and creates tools. Previously these options were available on the **Tools > Database** page of the **Options** dialog.
- **Database Settings** — Settings to determine the location of the tool database. Previously these options were available on the **Tools > Database** page of the **Options** dialog.
- **Tags** (see page 5) — Settings to create and edit a tool tag which enables you to group tools into categories according to a common feature.
- **Connections** (see page 9) — Settings to create and edit a tool holder connection-type which enables you to group tool holders by the shape or material of the join between the machine tool and tool holder.

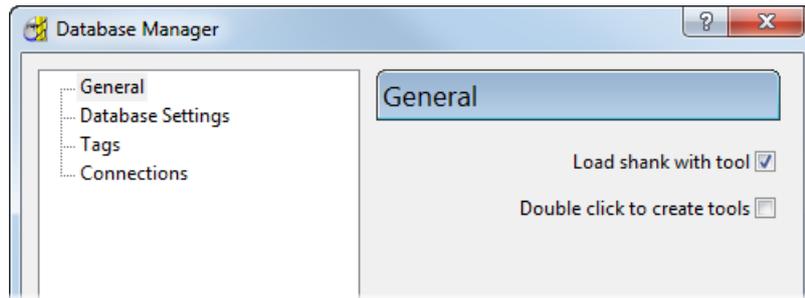
Database Manager > Tags

Use the **Tags** page to list the existing tool tags and to create, edit, and delete tags. Tool tags group tools according to a common feature, for example, strategy type. This enables you to search the tool database more easily. First add a tag to the tool database and then assign it to a tool.

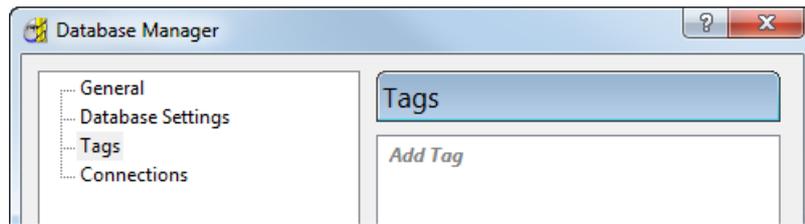


To add a tool tag to the tool database:

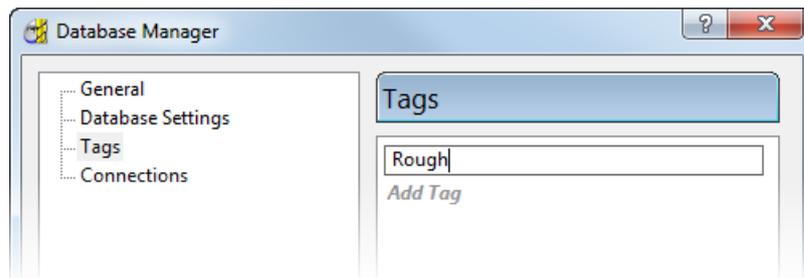
- 1 Click  on the **Tool** toolbar to display the **Database Manager** dialog.



- 2 In the tree, select **Tags** to display the **Tags** page.



- 3 Create a tag:
 - a Double-click **Add Tag**.
 - b Enter a tag name.



- c Press the **Enter** key to display the tag in the list.
- d Click **Close**.



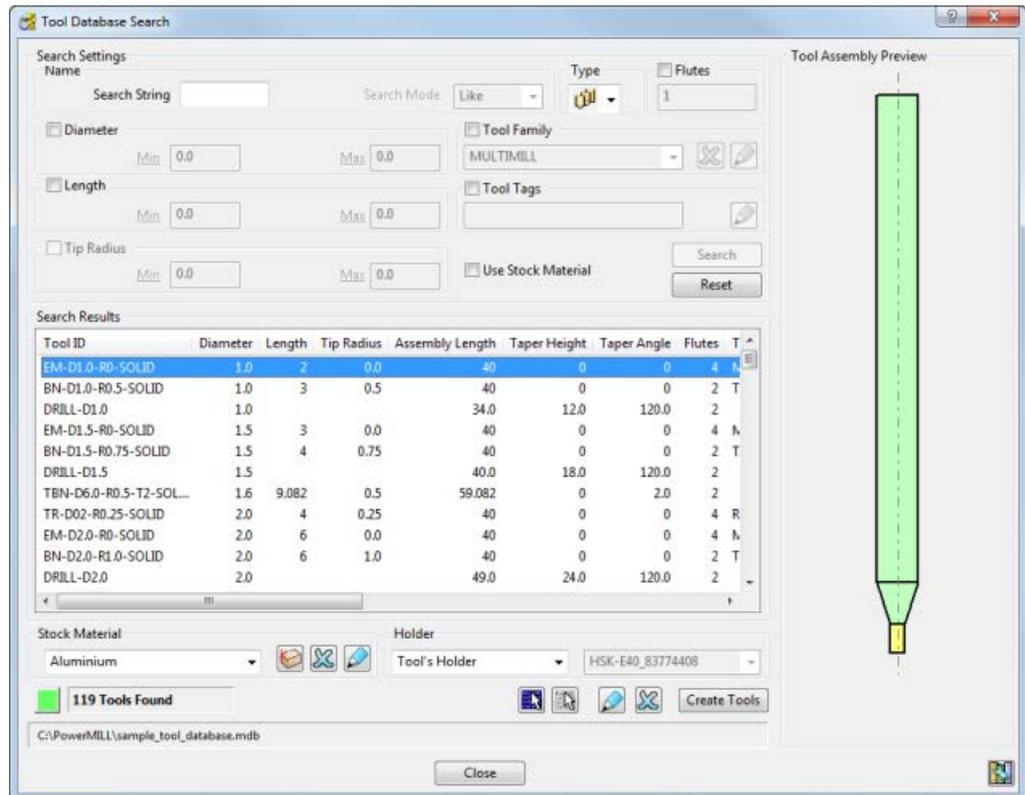
To delete a tag, on the **Tags** page of the **Database Manager** dialog, select a tag from the list and click .

Searching the tool database using tool tags

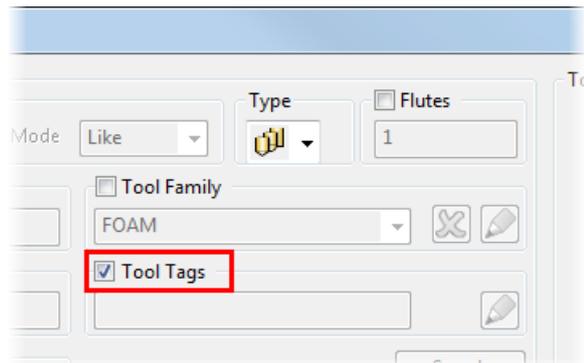
You can search for tools in the tool database using tool tags. This makes it easier to find tools.

To search for a tool tag:

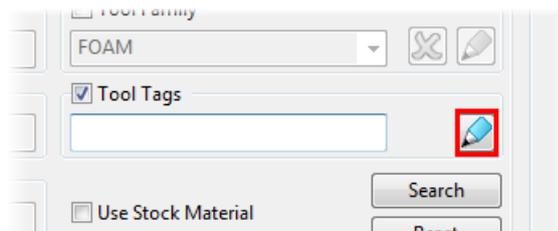
- 1 Click  on the **Tool** toolbar to display the **Tool Database Search** dialog.



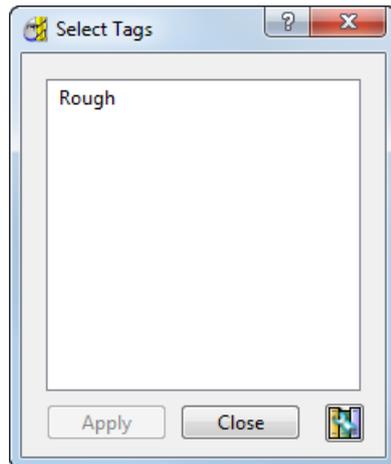
- 2 Select **Tool Tags**.



- 3 In the **Tool Tags** area, click  to display the **Select Tags** dialog.



- 4 Click a tag to select it, for example, **Rough**.



*Press and hold down the **Ctrl** key to select multiple tags.*

If you select more than one tag, all the tools with matching tags are displayed.

- 5 Click **Apply**.
- 6 Click **Search**.

Tools with a **Rough** tag are displayed in the **Search Results** area.

Tool ID	Diameter	Length	Tip Radius	Asse
Half Inch Foam Bit	12.7	76.2	6.35	
8 Inch Diamond Saw	203.2	0	0	

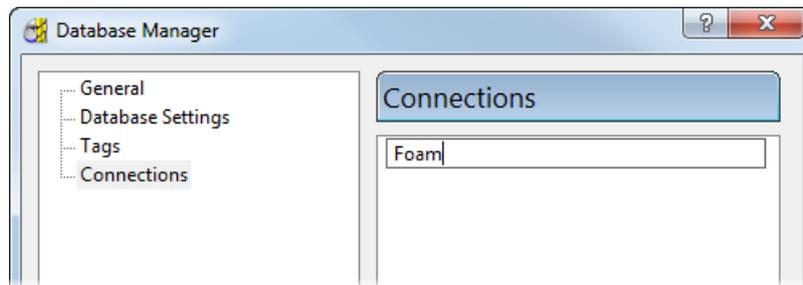


*To display all results in the Search Results area, click **Reset**.*

- 7 Click **Close**.

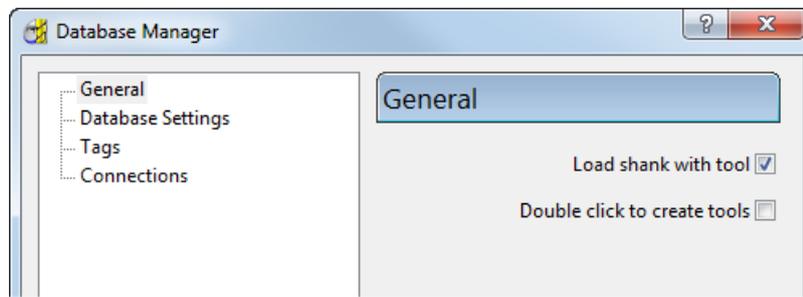
Database Manager > Connections

Use the **Connections** page to list the existing tool holder connection-types and create, edit, and delete them. Connections describe the type of join between a machine tool and a holder. For example, a connection may identify the material present at the join, or the shape of the join. This enables you to match a machine tool and a holder more easily. Connection types are added to the database and then assigned to a tool or holder.

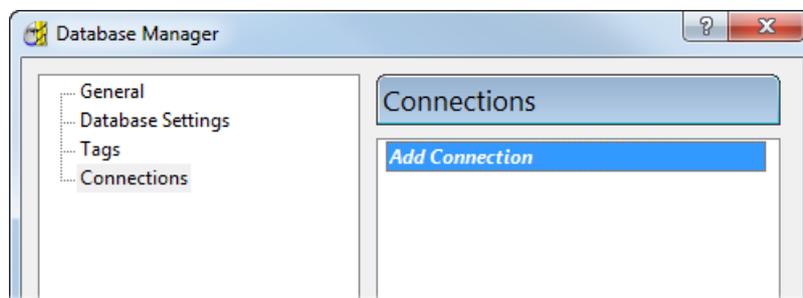


To add a connection to the database:

- 1 Click  on the **Tool** toolbar to display the **Tool Database Manager** dialog.

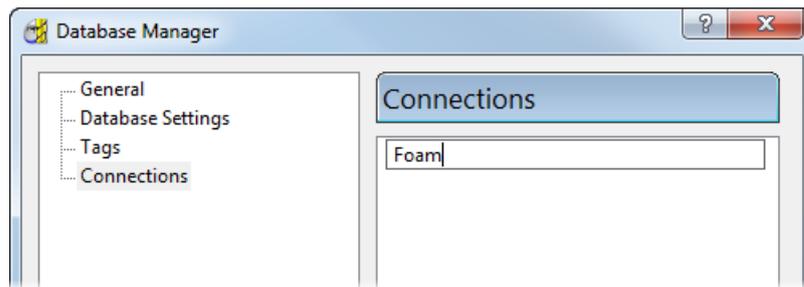


- 2 In the tree, select **Connections** to display the **Connections** page.



- 3 Create a **Connection**:
 - a Double-click **Add Connection**.

- b Enter a connection name, for example **Foam**.



- c Press the **Enter** key to display the connection in the list.

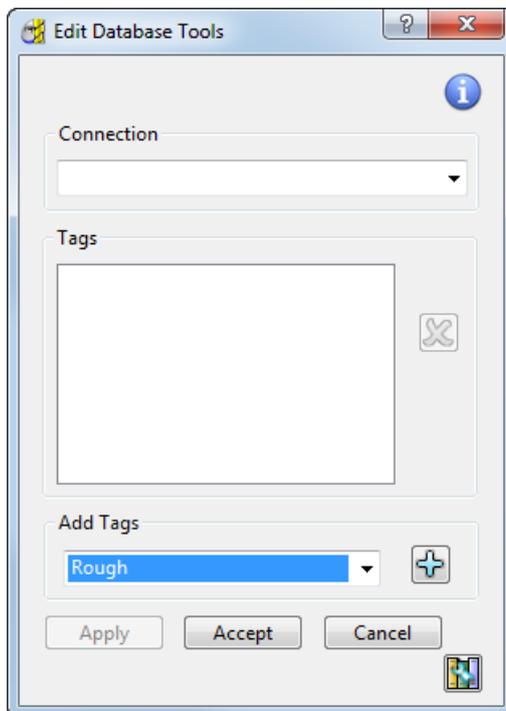
- 4 Click  to close the dialog.



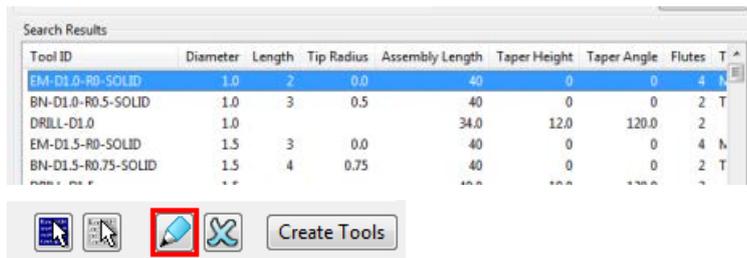
To delete a connection, on the **Connections** page of the **Database manager** dialog, select a connection from the list and click .

Edit Database Tools dialog

Use the tool **Edit Database Tools** dialog to assign connection-types and tags to the selected tools.



To display the **Edit Database Tools** dialog, select a tool from the **Search results** list in the on the **Tool Database Search** dialog, and click .



 — Hover over this button to display the tool or tools that you are editing.

Connection — Select the connection-type you want to assign to a tool or holder in the database.

Tags — Displays the tags assigned to a selected tool.

 — Click to remove a selected tag from the database.

Add Tags — Select the tags that you want to assign to a tool in the database.

 — Click to assign a selected tag to the tool. The tag is then displayed in the **Tags** list.

 — Click to display the **Database Manager** dialog (see page 5).

Generating toolpaths

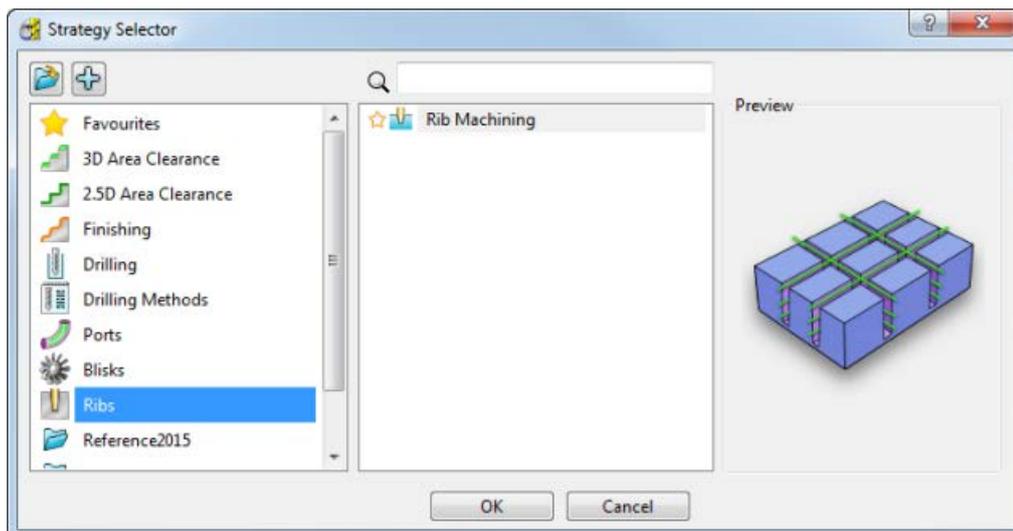
PowerMILL 2015 R2 contains the following changes and improvements to the generation of toolpaths:

- Rib machining (see page 13):
 - **Rib Pattern** mode-toolbar (see page 22) — Sketch the centreline of ribs/channels in a mold.
 - **Rib Surfaces** mode-toolbar (see page 24) — Automatically select the walls and base surfaces of the channels associated with the sketched centreline of the ribs/channels.
- Improved collision avoidance (see page 29) — Check for collisions against the stock.
- Drilling enhancements (see page 30):
 - Drilling from the stock (see page 30) — Create drilling toolpaths that start at the top of the stock model.
 - Automatic verification enhancements (see page 31) — Collision check against the stock model. You can specify the type of toolpath generated after detecting a collision.
 - Drilling non-planar surfaces (see page 34) — Calculate a drilling toolpath where the top of the drill move is defined by a block surface.
 - Enhancements to area clearance and Vortex drilling (see page 34) — Specify the diameter of pre-drilled holes before machining area clearance toolpaths
- Editing orientation vectors (see page 36):
 - **Specify Changes** tab (see page 36) — Edit orientation vectors in a restricted toolpath region, without modifying the tool axis.
 - **Machine axis control** tab (see page 37) — Fine-tune orientation vectors using Cartesian coordinates.

- Curve fitting toolbar (see page 38) — Convert selected curve segments to a spline. You can also choose to keep curves embedded.
- **Machine Tool Simulation Issues** dialog (see page 41) — Display a list of issues and warnings, with options for collision checking and clearance values.
- Maintaining tool speed after ramping (see page 42) into the stock.
- Aligning the tool with the toolpath (see page 43) — Align the tool axis direction with the toolpath.
- Undo toolpath limit (see page 44) — Undo the last point added to a limiting polygon.

Ribs

The new **Rib machining** strategy enables you to machine narrow channels in a mold (see page 14). For example, you can machine narrow grooves in a mold tool that form ribs in the resulting injection moulded part. **Rib machining** is located in the new Ribs category on the **Strategy selector** dialog.



For more information on the rib machining strategy, see:

- Rib Machining Overview (see page 14) — An introduction to rib machining.
- **Rib machining** — Machine channels in a mold which form ribs in the resulting injection-molded part.
- **Automatic verification** — Enable verification of toolpaths on creation.

- **Rib Pattern** mode-toolbar (see page 22) — Sketch the centreline of ribs/channels.
- **Rib Surfaces** mode-toolbar (see page 24) — Automatically select the walls and base surfaces of the channels associated with the sketched centreline of the ribs/channels.
- Creating a rib machining toolpath (see page 25) — Create a rib machining toolpath.

The common tabs are described in common toolpath creation controls.

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

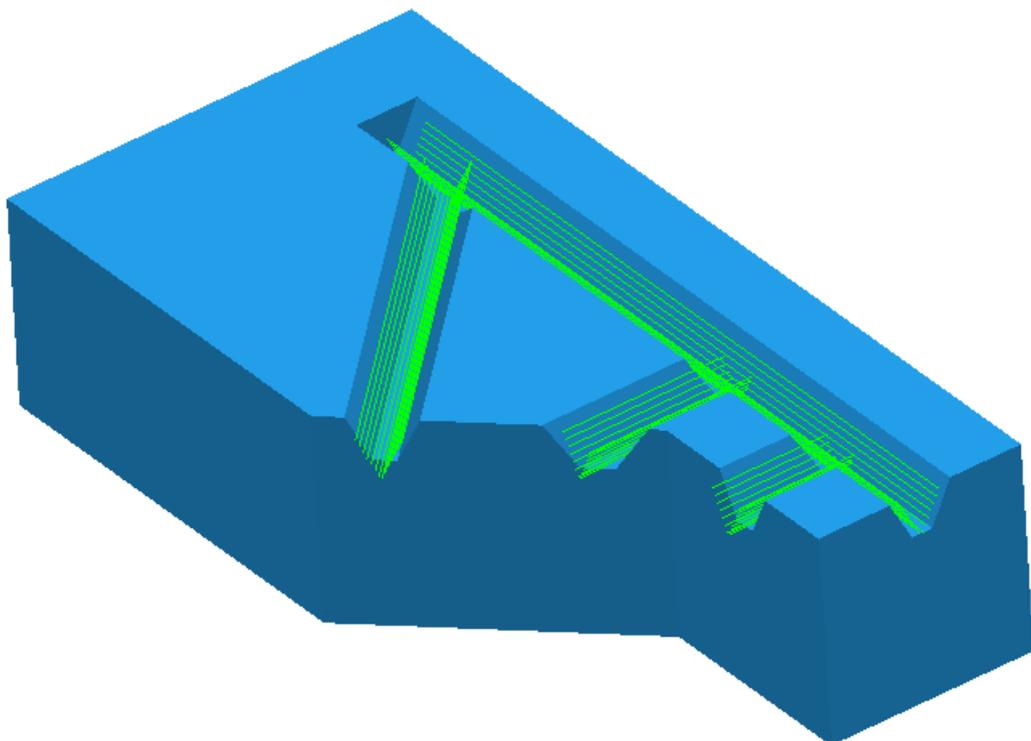
Rib Machining Overview

Use the **Rib Machining** strategy to machine channels in a mold which form ribs in the resulting injection-molded part. Typically, the channels are long, thin, and non-planar.

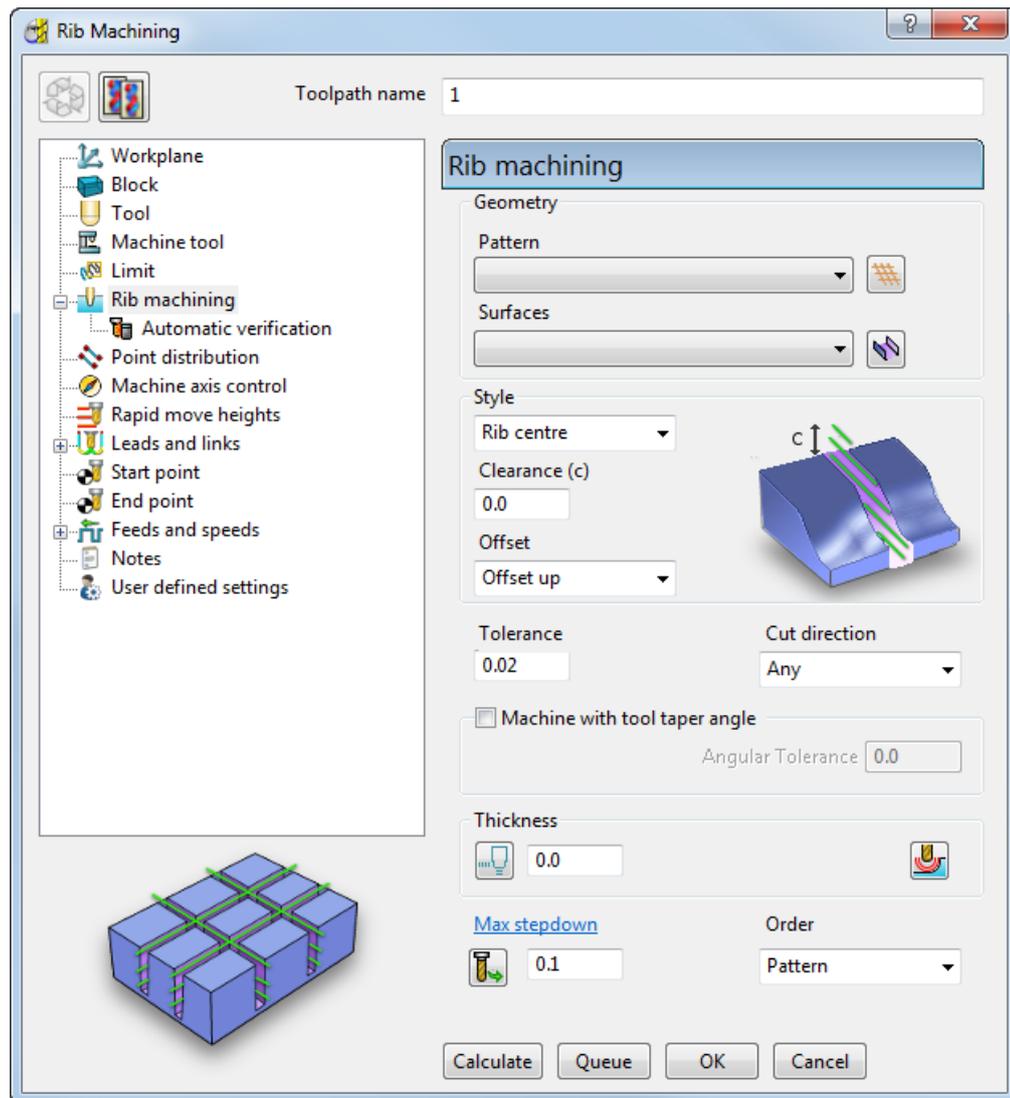
Rib machining requires:

- a pattern that is an approximate centreline of the channel.
- surfaces defining the walls and the base of the channel.
- a tool that fits inside the channel. The width of the channel must be between one and four times the diameter of the tool.

Rib machining frequently uses a form tool that reflects the exact shape of the channel.



The pages associated with the **Rib Machining** strategy are:

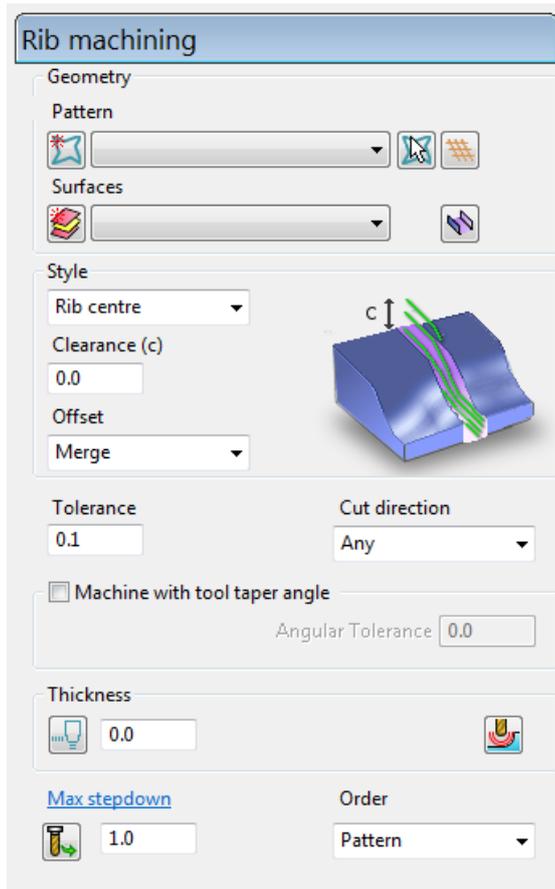


- **Rib machining** (see page 16) — The main page used to define a rib machining toolpath.
- **Automatic verification** — Settings to automatically verify the toolpath when calculating it.

The remaining pages are common toolpath creation controls.

Rib machining

Use the **Rib machining** page to create a toolpath by machining a channel (or groove) in a part.



Pattern — Select the pattern that is the approximate centreline of the channel. If you do not have a suitable pattern, click  to sketch one.

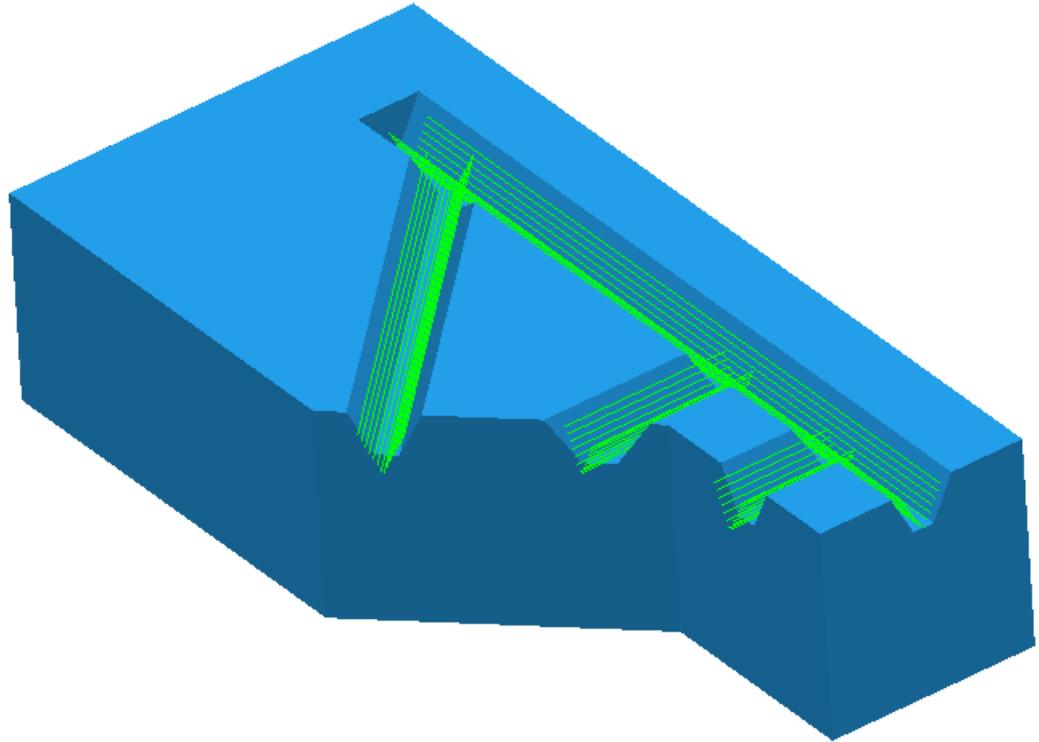
 **Rib pattern** — Click to display the **Rib Pattern** mode-toolbar which enables you to sketch the approximate centreline of the channel (see page 22).

Surfaces — Select the surfaces forming the walls and base of the channel. Alternatively, click  to select these surfaces automatically.

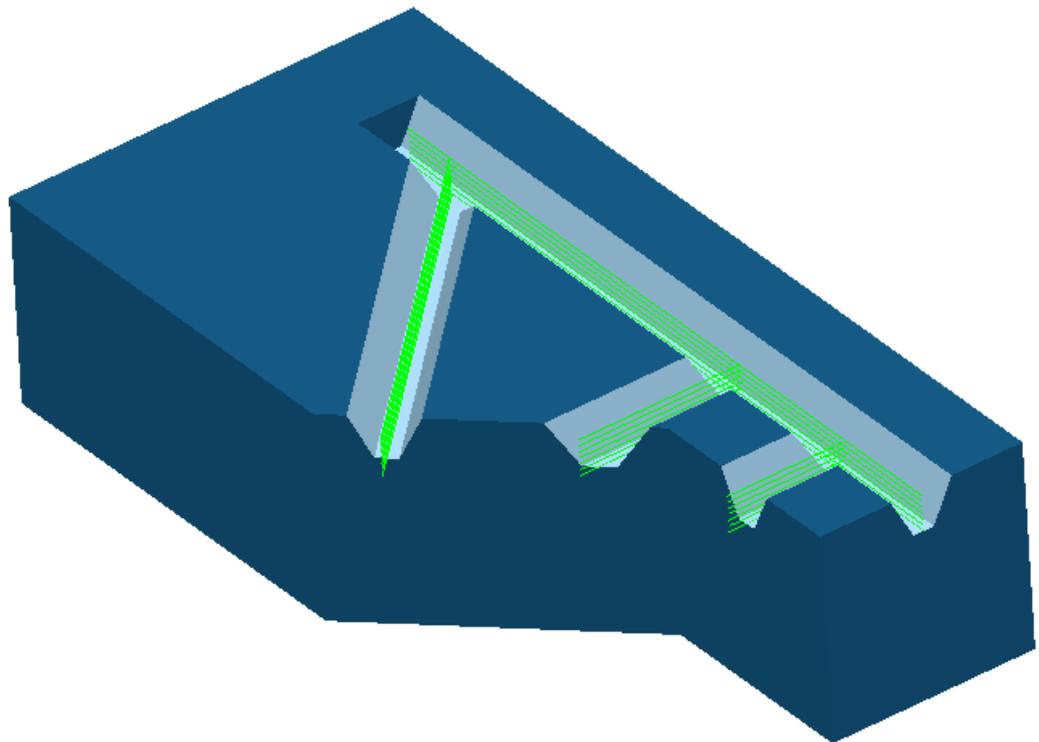
 **Rib surface** — Click to display the **Rib Surface** mode-toolbar which enables automatic selection of the wall and base surfaces of the channel (see page 24).

Style — Select whether you want to machine down both walls of the channel or down the centreline.

Selecting a **Style** of **Rib walls** creates a toolpath that machines down one side of the channel and then the other side. Use this option when the tool is smaller than the channel.



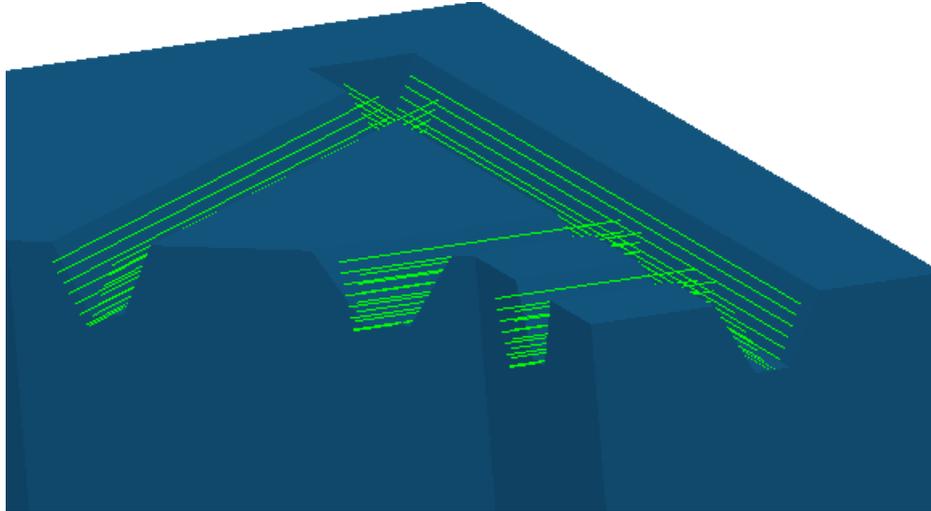
Selecting a **Style** of **Rib centre** creates a toolpath that machines down the centreline of the channel. Use this option when the tool, frequently a form tool, is the size and shape of the channel.



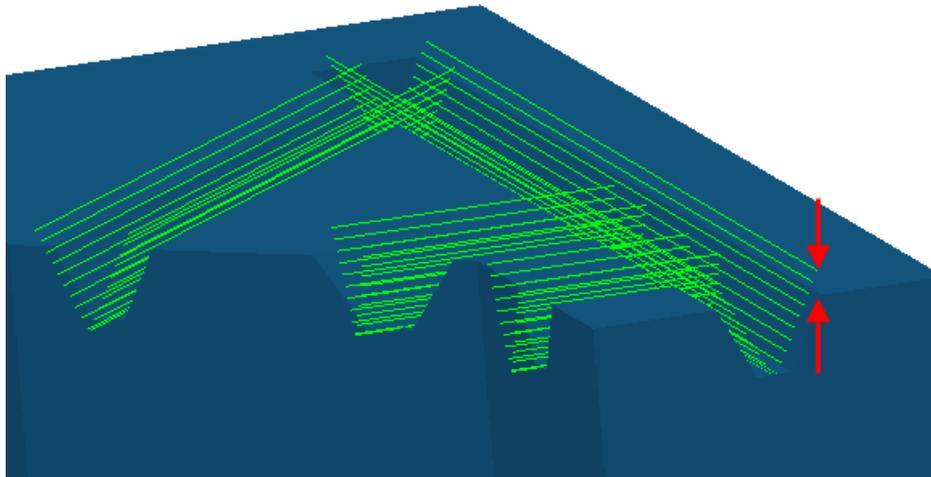
Clearance — When the top surface is not fully machined, enter the height above the top of the top of the channel where the machining starts.

For example:

A **Clearance** of **0** gives:

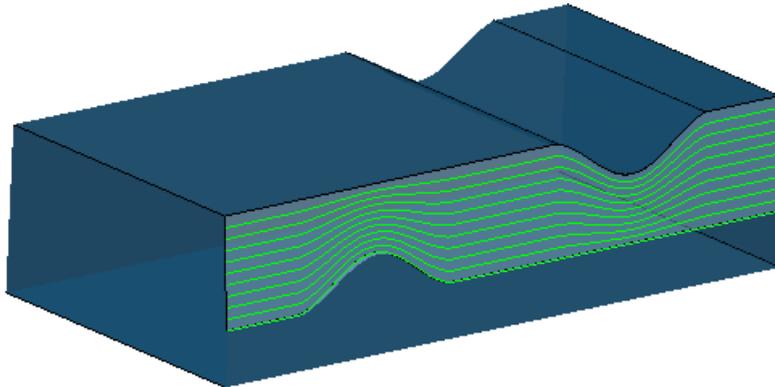


A **Clearance** of **10** gives:

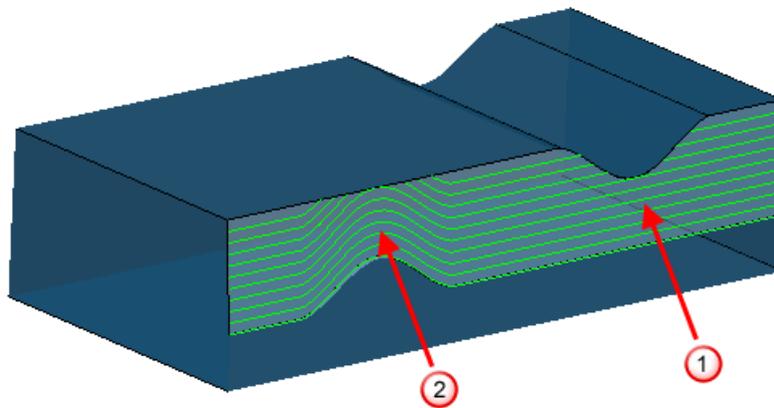


Offset — Select how to offset from the base surface to the top of the channel.

Merge — When selected, starts by offsetting the base profile and migrates the offset so that at the end it is offsetting the top profile. This minimises the number of air moves.



Offset up — When selected, offsets the base profile. This option gives a consistent depth of cut and minimises the number of downhill moves. This is important when your tool cannot make cutting moves where Z decreases.



- ① Region with no downhill moves
- ② Region with downhill moves.



*You can eliminate the downhill moves by selecting a **Cut direction of Uphill**.*

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

Cut direction — Select the milling technology.

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.

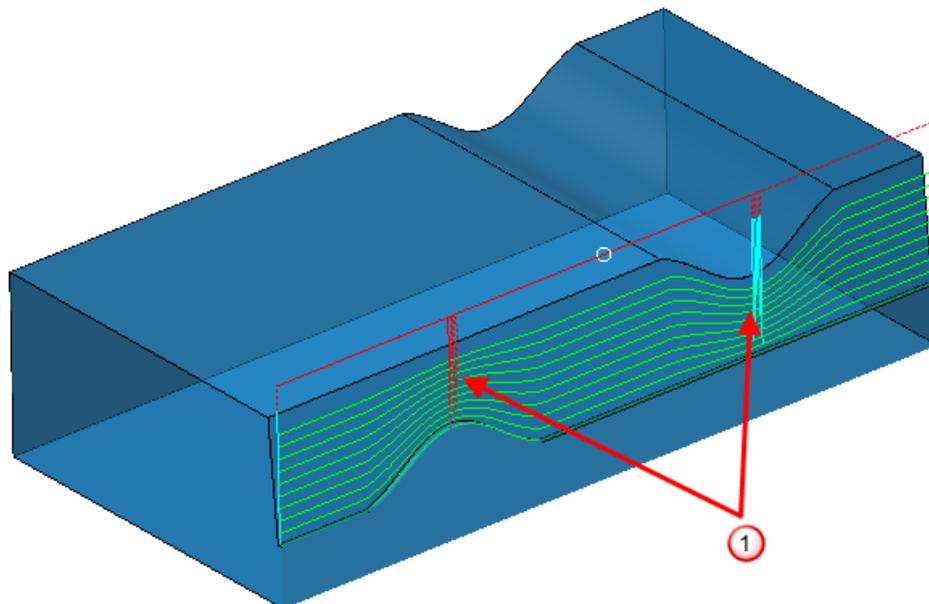
Conventional — Select to create toolpaths using only conventional or upcut milling, where possible. The tool is on the right of the machined edge when viewed in the direction of tool travel.

Any — Select to create toolpaths using both conventional and climb milling. This minimises the tool lifts and tool travel.

Pattern — Select to create a toolpath that follows the direction of the pattern. This option uses both climb and conventional milling. When you select a **Style of Rib walls**, one wall is machined using climb milling and the other using conventional milling to ensure the tool follows the direction of the pattern. With open channels, use this option to start machining from the open end of the channel to avoid plunge moves. To see the direction of the pattern, instrument it by clicking  on the **Rib Pattern** mode-toolbar.

 *Display the **Rib Pattern** mode-toolbar by clicking .*

Uphill — Select to create a toolpath that avoids cutting moves where Z decreases. This option uses both climb and conventional milling. When you select a **Style of Rib walls**, one wall is machined using climb milling and the other using conventional milling to ensure the tool never makes a decreasing Z move.



① additional toolpath lifts added to avoid cutting moves where Z decreases

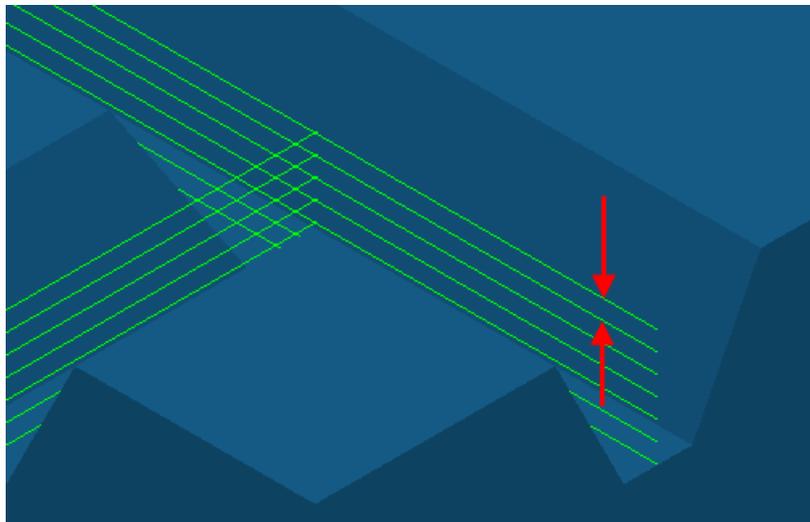
Machine with tool taper angle — When selected, the tool machines the channel using its taper angle instead of the draft angle of the channel. This is useful when the channels are not modelled accurately, do not need to be machined accurately, or you have a form tool that matches the profile of the channel.

Angular tolerance — Enter the maximum deviation from the draft angle of the channel to the tool taper angle. When this angle is exceeded, PowerMILL leaves a portion of the channel unmachined.

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.

Max stepdown — Enter the maximum distance between successive passes.



 **Copy stepdown from tool** — Click to load the axial depth of cut from the active tool's cutting data. The axial depth of cut is measured along the tool axis.

 **Copy stepover from tool** — Click to load the radial depth of cut from the active tool's cutting data. The radial depth of cut is measured normal to the tool axis.

 *If you enter the value manually, the button changes to .*

Order — Select the machining order of the selected channels.

Pattern — Select to machine the channels in the same order as you created the pattern.

Shortest — Select to create the shortest toolpath. This minimises the air moves.

Rib Pattern mode-toolbar

Use the **Rib Pattern** mode-toolbar to sketch the centreline of ribs/channels.

The **Rib Pattern** mode-toolbar is available from the **Rib pattern**  button on the **Rib Machining** strategy dialog (see page 14).



 *Activating a mode disables most of PowerMILL's functionality until you exit from the mode. For more information, see [Mode-toolbars](#).*

 **Lines** — Hover to display the **Line** toolbar which creates new lines. You can define the lines either interactively or by entering the required coordinates. This is similar to the **Line** toolbar available on the **Curve Editor** mode-toolbar.

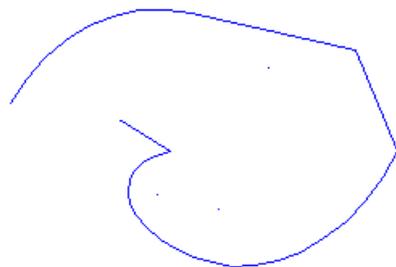


 **Continuous line** — Click to create a continuous line containing straight line segments.

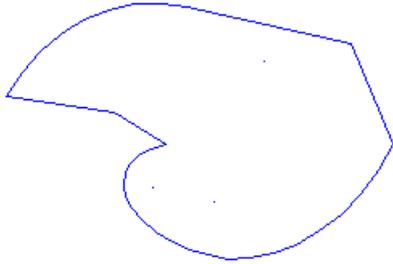


 **Close Segment** — Click to close the current segment.

It converts an open segment:



To a closed segment:



 **Start New Segment** — Click to end the current segment so the next point is the start of a new segment.

 **Finish** — Click to accept the changes and close the toolbar.

For more information, see the create lines example.

 **Single line** — Click to create a single line.

 **Delete** — Click to delete the selected items, curve segments, workplanes, or points.

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

Converts this:



to this:



 *You must select the segments you want to reverse.*

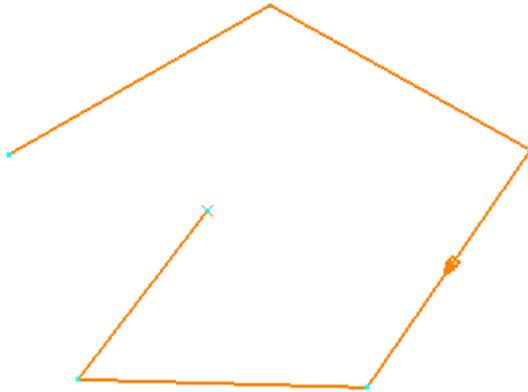
 **Reorder segments** — Click to list all the curve segments and enables you to change the order of the segments in a pattern.

 **Cut** — Hover to display the **Cut** toolbar which cuts or merges curves.



 **Curve Editor** — Click to display the **Curve Editor** mode-toolbar, which enables you to create and edit patterns. The assumption is that you first extract curves from the model, by say, extracting curves at surface boundaries, and then modify the curve to create the exact pattern you need.

 **Instrument** — All curves have a direction. If you instrument a curve, it places an arrow on each segment (pointing towards the end of the segment) and an X at the end of each segment.



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

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

 **Next** — Click to create the sketched pattern, close the **Rib Pattern** mode-toolbar and open the **Rib Surface** mode-toolbar (see page 24).

 **Accept changes** — Click to accept and keep all the created curves. This closes the **Rib Pattern** mode-toolbar and enables normal PowerMILL functionality.

 **Cancel changes** — Click to delete all the created curves. This closes the **Rib Pattern** mode-toolbar and enables normal PowerMILL functionality.

Rib Surfaces mode-toolbar

Use the **Rib Surfaces** mode-toolbar to automatically select the walls and base surfaces of the channels associated with the sketched centreline of the ribs/channels.

The **Rib Surfaces** mode-toolbar is available from the **Rib surfaces**  button on the **Rib Machining** strategy dialog (see page 14).



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

 **Find rib surfaces** — Click for PowerMILL to automatically find the rib/channel surfaces associated with the centreline pattern.



Select a pattern on the **Rib machining** dialog to enable this functionality.



Select surfaces — click to manually select the rib/channel surfaces associated with the centreline pattern.



For example, use  to select the majority of the surfaces and then use  to select or deselect the final few surfaces.



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



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



Previous — Click to close the **Rib Surfaces** mode-toolbar and return to the **Rib Pattern** mode-toolbar (see page 22).



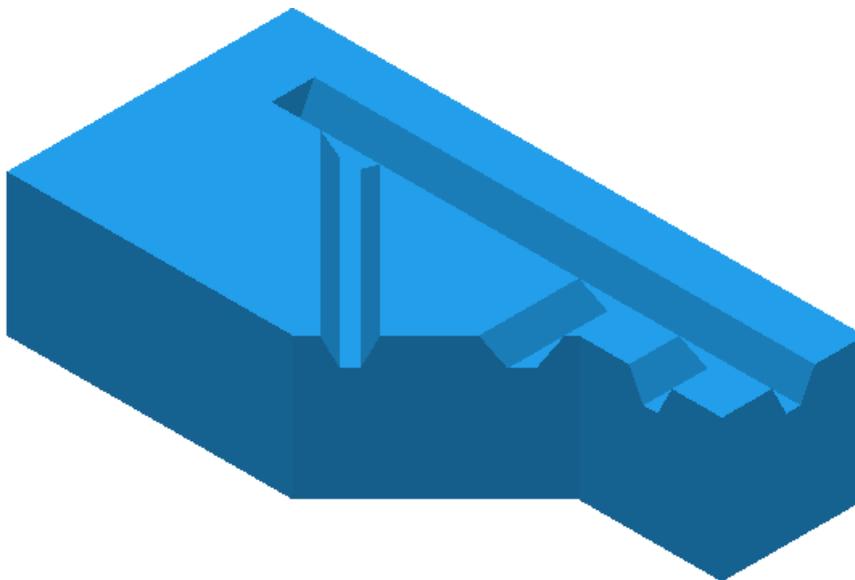
Accept changes — Click to accept and keep all the created curves. This closes the **Rib Pattern** mode-toolbar and enables normal PowerMILL functionality.



Cancel changes — Click to delete all the created curves. This closes the **Rib Pattern** mode-toolbar and enables normal PowerMILL functionality.

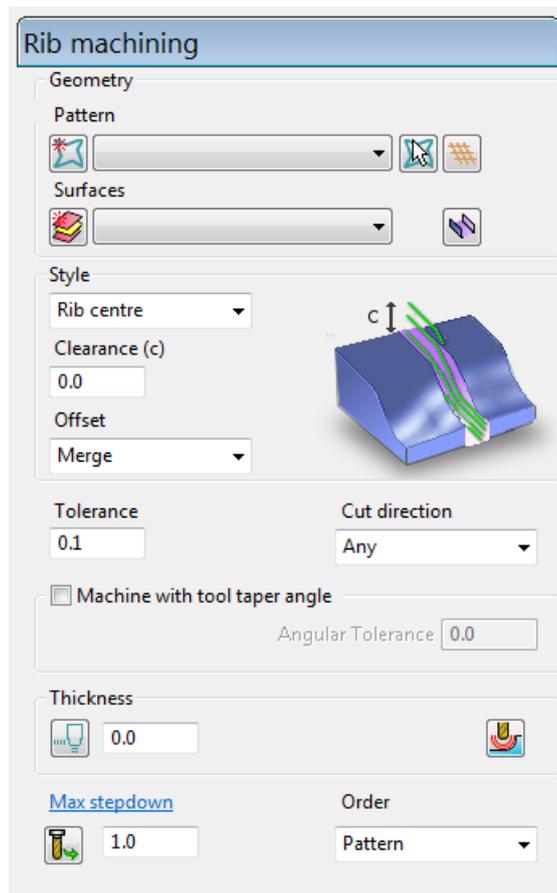
Creating a rib machining toolpath

This example shows you how to create a rib machining toolpath. This uses the [Ribs_multi.dmt](#) model in the **Examples** folder and a simple rectangular block.



In this example the ribs are far wider than in a typical ribbed model as this makes it easier to understand the process.

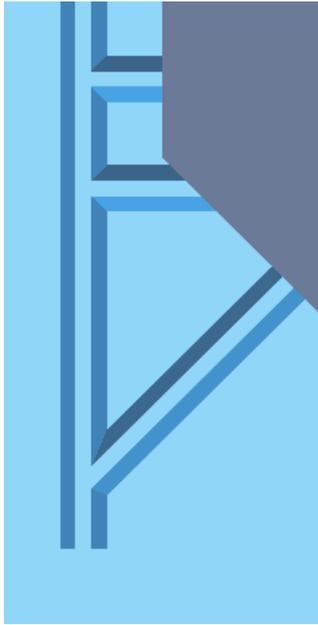
- 1 Create a tapered spherical tool with:
 - a A **Diameter** of **25**.
 - b A **Tip radius** of **7**.
 - c A **Taper angle** of **25**.
 - d A **Length** of **125**.
- 2 Click the **Toolpath Strategies**  button,
- 3 From the **Ribs** category select **Rib Machining**. This displays the **Rib Machining** dialog.



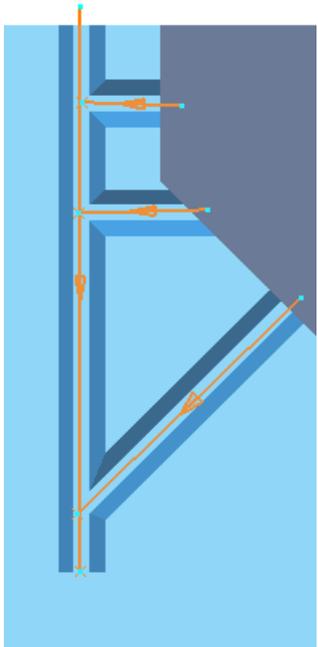
- 1 Click the **Rib pattern**  button to display the **Rib Pattern** mode-toolbar. This is a simplified version of the **Curve Editor** mode-toolbar.



This automatically changes the view to the view down Z.



- 2 Click the **Line**  button on the **Line** toolbar .
- 3 Sketch the approximate centreline of the four channels as four lines. For complete machining of each channel, the lines must extend beyond the open end of the channels. The direction of each line determines the machining direction. Because most rib machining tools break when plunging, it is best to start each line from the open end of the channel.

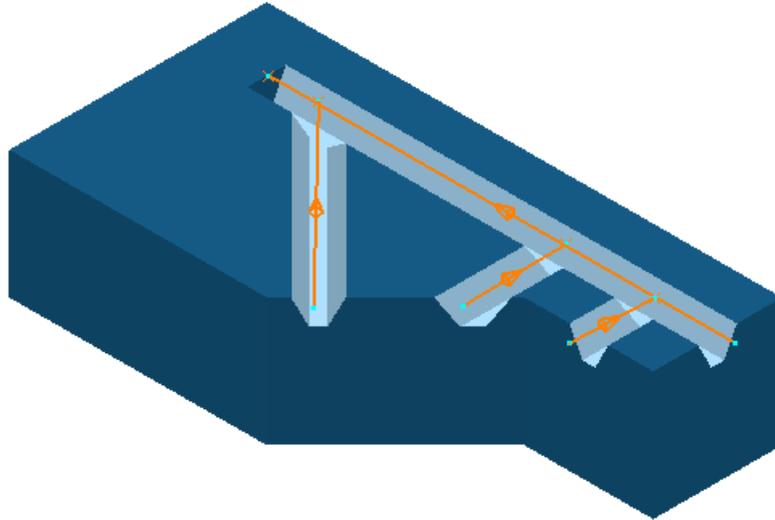


- 4 If you can't see the direction of the lines, click **Instrument** .

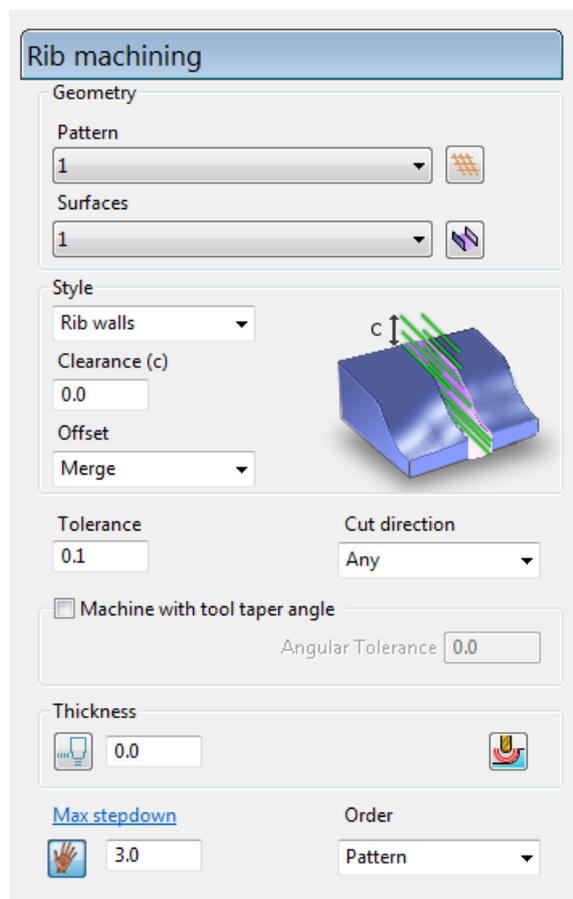
- 5 Click **Next**  to display the **Rib Surfaces** mode-toolbar.



- 6 Click the **Find rib surfaces**  button. This automatically selects all the wall and base surfaces of the channel.



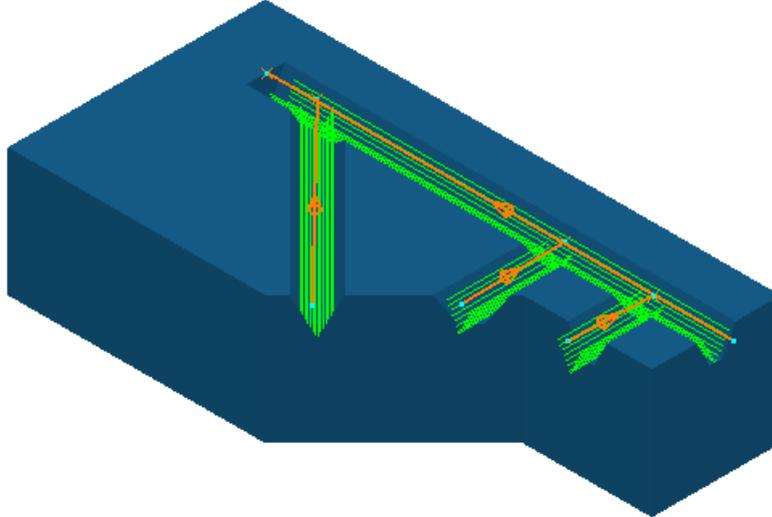
- 7 Click . This closes the **Rib Surfaces** mode-toolbar and displays the **Rib Machining** dialog.



- 8 On the **Rib Machining** dialog:

- a Select a **Style** of **Rib walls**.
- b Enter a **Max stepdown** of **3**.
- c Click **Calculate**.

This creates the rib machining toolpath.



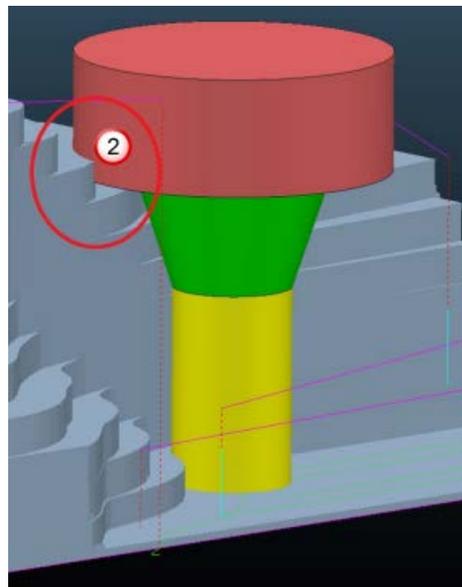
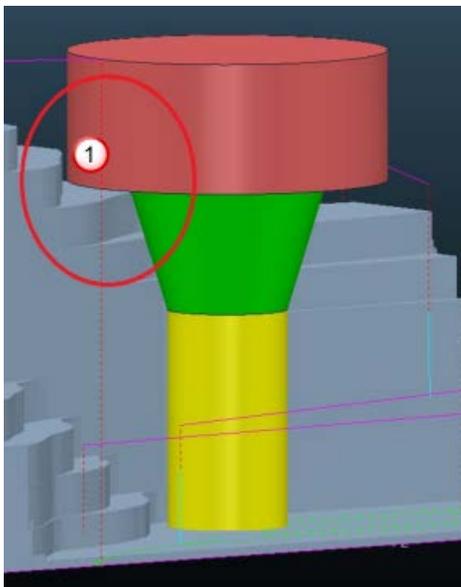
Improved collision avoidance

You can collision-check the unmachined stock of area clearance toolpaths against the shank and holder. In previous versions, you could only collision-check the shank and holder against the model, not the stock.

The following table compares a **Model Area Clearance** toolpath in **PowerMILL 2015 R2** and **PowerMILL 2015**:

PowerMILL 2015 R2

PowerMILL 2015



① No collision

② Collision

To enable this function, select **Automatic collision checking** on the **Automatic verification** strategy page.

Automatic verification

Thickness 1.0

Head clearance 600.0

Automatic collision checking

Collision checking against Stock

Holder clearance 0.0

Shank clearance 0.0

Drilling enhancements

There are several enhancements to drilling. You can:

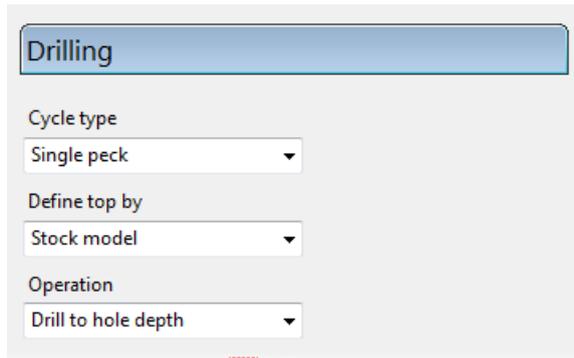
- start drilling at the top of the stock (see page 30).
- check for collisions against the stock model (see page 31).
- specify the type of toolpath generated after detecting a collision (see page 31).
- drill blocks with a non-planar surface (see page 34).
- specify **Entry moves** in Vortex strategies (see page 35).
- specify the diameter of a pre-drilled hole (see page 35).

There are also changes to the way in which drilling toolpaths are defined (see page 35).

Drilling from the stock

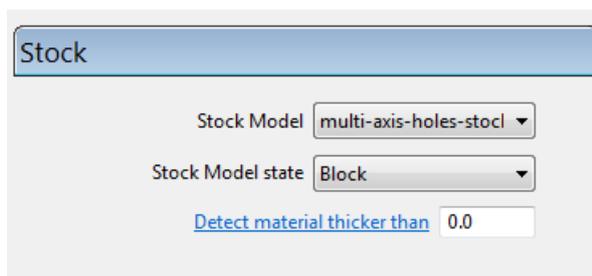
You can create drilling toolpaths that start at the top of the stock model. This enables you to drill holes from the start of the material, not the block, which minimises air moves. To create a drilling toolpath starting at the top of the stock, you must select the following options in the **Drilling** strategy dialog:

- 1 On the **Drilling** strategy page, select **Stock model** from the **Define top by** list.



The screenshot shows the 'Drilling' strategy configuration page. It features a title bar 'Drilling' and three dropdown menus: 'Cycle type' set to 'Single peck', 'Define top by' set to 'Stock model', and 'Operation' set to 'Drill to hole depth'.

- 2 On the **Stock** page:



The screenshot shows the 'Stock' configuration page. It features a title bar 'Stock' and three settings: 'Stock Model' set to 'multi-axis-holes-stocl', 'Stock Model state' set to 'Block', and a 'Detect material thicker than' field with the value '0.0'.

- a select a **Stock Model** from the list.
- b select a **Stock Model state** from the list. This is the state of the stock material at a particular point in the machining process.

If the currently active state in a stock model is a:

Block — The stock model is the initial state of the stock, before any entities are applied to it.



*Depending on how it is used, an individual **Stock Model** can also be a block.*

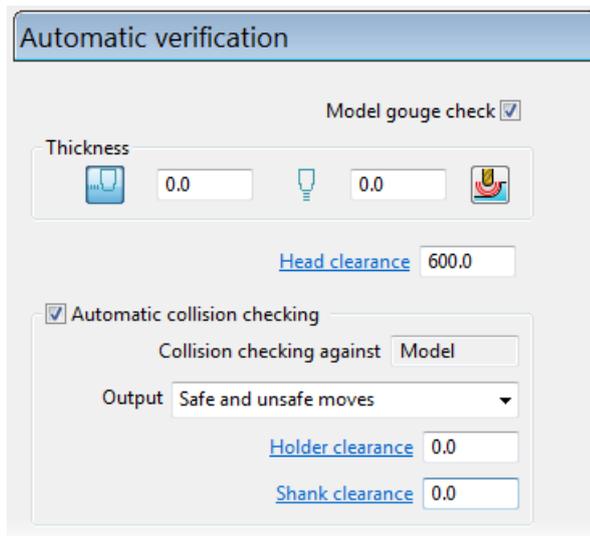
Toolpath — The stock model is the state of the stock, after applying a toolpath to the stock's previous state.

Tool — The stock model is the state of the stock after applying a dense raster toolpath (automatically created using the tool).

- a enter a value in the **Detect material thicker than** field. PowerMILL ignores rest material that is thinner than the specified threshold.

Automatic verification enhancements

There are several enhancements to the drilling **Automatic verification** page.



You can:

- collision check against the stock model.
- specify the type of toolpath generated after detecting a collision with the model, stock model or block.

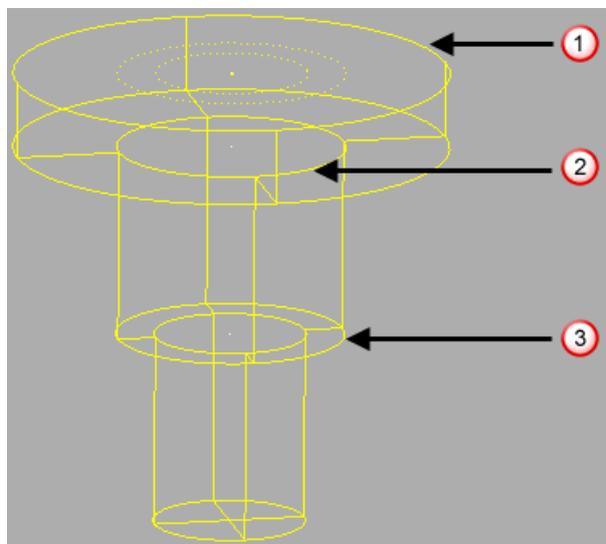
Collision checking against — Displays what is collision checked.



*The option selected in the **Define top by** list on the **Drilling** page, determines what is displayed.*

Hole top — Drilling starts at the top of the hole.

Component top — Drilling starts at the top of the component specified in the **Top component** field rather than the top of the hole. This is useful when drilling compound holes.



① Hole top, first component top

② Second component top.

③ Third component top.

Block — Drilling starts at the top of the block.

Model — Drilling starts at the top of the model.

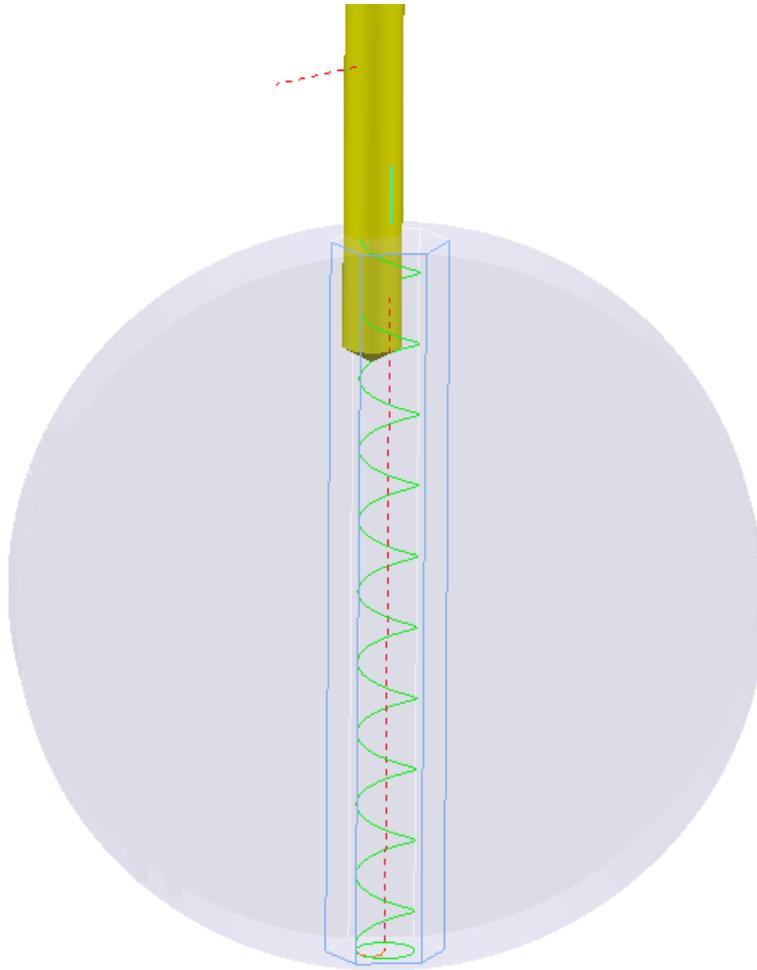
Stock model — Drilling starts at the top of the stock. This displays the **Stock** page in the **Drilling** strategy dialog.

Output — Select the type of toolpath generated on detecting a collision:

- **Safe and unsafe moves** — Generates the whole toolpath including the colliding segments.
- **Safe split moves** — Splits the toolpath into safe and unsafe moves, and only outputs the safe moves.
- **Moves in safe holes only** — Generates toolpaths only in collision-free holes.

Drilling non-planar surfaces

You can calculate a drilling toolpath where the top of the drill move is defined by the block surface, including blocks with curved surfaces. In previous versions of PowerMILL, this was restricted to planar, horizontal block surfaces.

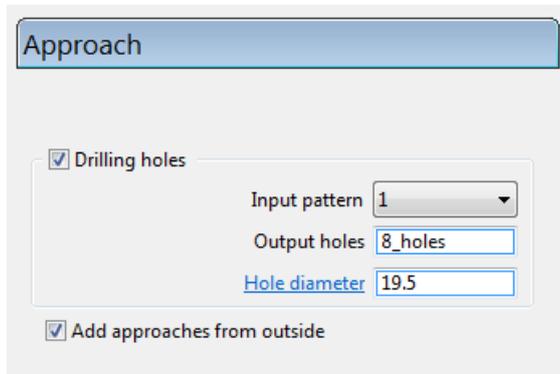


Enhancements to area clearance and Vortex drilling

There are several enhancements to area clearance drilling. In PowerMILL 2015 R2, you can specify the diameter of the hole (see page 35). There are also changes to the way that drilling toolpaths are defined (see page 35).

Specifying the diameter of a hole

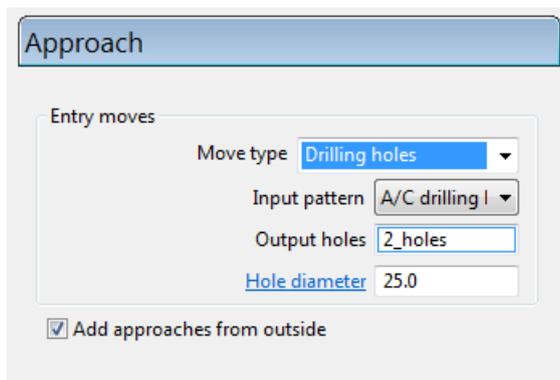
You can specify the diameter of pre-drilled holes before machining area clearance toolpaths. This enables you to access closed pockets without the need for helical ramps, saving you time. The default diameter is 150% of the tool diameter; to change this, enter a **Hole diameter** on the **Approach** page of the area clearance strategy. This generates a feature set with the required drilling positions. You must then create a drilling toolpath to machine the drilling positions.



The screenshot shows the 'Approach' dialog box. Under the 'Drilling holes' section, which is checked, there are three fields: 'Input pattern' set to '1', 'Output holes' set to '8_holes', and 'Hole diameter' set to '19.5'. There is also a checked checkbox for 'Add approaches from outside'.



You can also enter hole diameters for Vortex machining. On the **Approach** page of the **Model Area Clearance** dialog, select **Drilling holes** from the **Move type** list. Enter a value for the **Hole Diameter** that is greater than the diameter of the tool.



The screenshot shows the 'Approach' dialog box. Under the 'Entry moves' section, there are four fields: 'Move type' set to 'Drilling holes', 'Input pattern' set to 'A/C drilling I', 'Output holes' set to '2_holes', and 'Hole diameter' set to '25.0'. There is also a checked checkbox for 'Add approaches from outside'.

Changes to drilling toolpath definition

PowerMILL 2015 R2 uses feature sets to define drilling positions and hole diameters. Using feature sets enables you to generate your own drilling toolpath with the appropriate drilling cycles. PowerMILL creates the following features when you calculate an area clearance toolpath:

- patterns containing the drilling position
- features containing the drilling holes

Editing orientation vectors

You can edit orientation vectors in a restricted toolpath region, without modifying the tool axis. As the direction of the orientation vectors is defined by Cartesian components, you can fine-tune orientation vectors, which is useful when using robots.

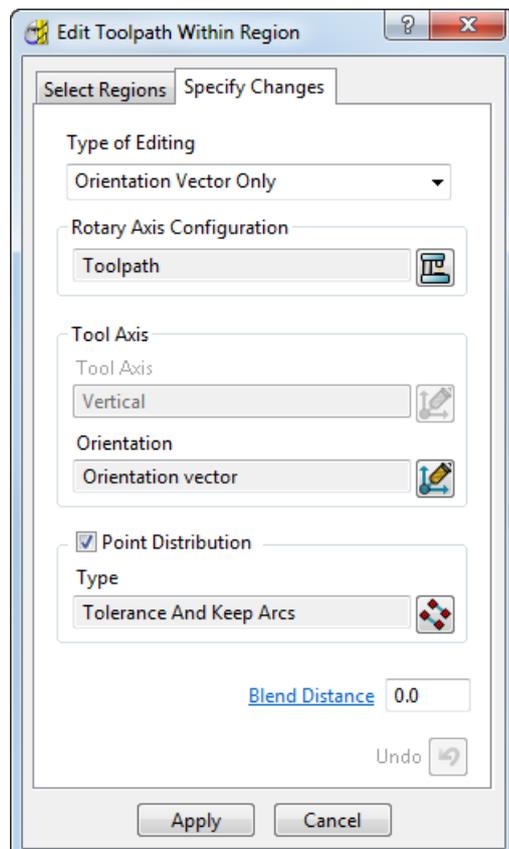
These options are available on the following tabs:

- The **Specify Changes** tab of the **Edit Toolpath Within Region** dialog (see page 36). Use this to edit only the orientation vectors.
- The **Machine axis control** tab of the **Tool Axis** dialog. Use this to define the orientation vectors in Cartesian coordinates.

Specify Changes tab

The **Specify Changes** tab (previously named **Edit Tool Axis**) of the **Edit Toolpath Within Region** dialog includes the following enhancements:

- There is a new **Orientation Vector Only** option in the **Type of Editing** list. Select this option to edit the orientation vectors without modifying the tool axis.



- There is a new **Orientation** field, which displays the **Orientation** type.

- There is a new **Orientation**  button. Click this button to display the **Tool Axis** dialog and select an **Orientation type**. Your selection is displayed in the **Orientation** field in the **Edit Toolpath Within Region** dialog.

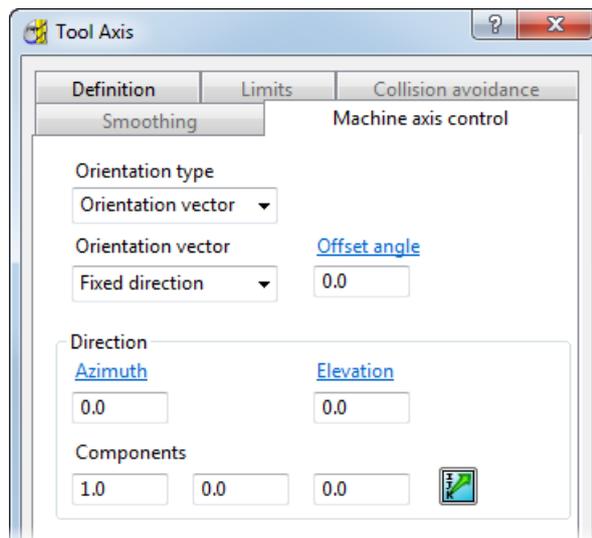


The **Edit Toolpath Within Region** dialog was previously called **Tool Axis Editing** dialog.

Machine axis control tab

The **Machine axis control** tab of the **Tool axis** dialog includes the following enhancements:

- Components** fields — Enter values in the fields to edit the machine axis direction by defining a vector using its i, j and k components.



To display the **Components** field, on the **Machine axis control** page:

- select **Orientation vector** from the **Orientation type** list.
 - select **Fixed Direction** from the in the **Orientation vector** list.
- Direction**  button — Click to display the **Direction** dialog, which provides additional options to edit orientation vectors.

Curve fitting toolbar

You can now fit a curve after it has been accepted. The new **Curve fitting** toolbar on the **Create Composite Curve** mode-toolbar enables you to convert selected curve segments to a specific type of curve, or choose to keep curves embedded.



To display the **Curve fitting** toolbar:

- hover over the **No curve fitting**  button on the **Create Composite Curve** mode-toolbar.
- select an option from the **Curve fitting** list in **Options > Curve Editor**.

You can select:

 **No curve fitting** — Click to keep curves embedded.

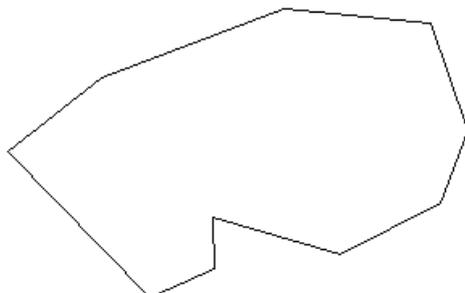
 **Spline selected** — Click to convert the selected segments into a spline. This displays the **Spline Fit** dialog.



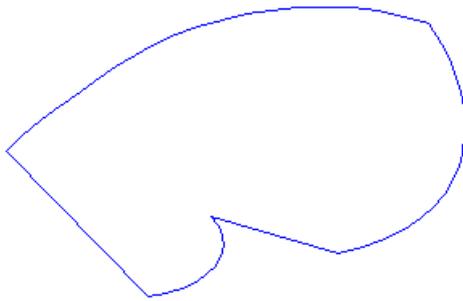
Tolerance — Enter the maximum distance between the original and fitted curves. The value specified in the **Curve Fitting Tolerances** page of the **Options** dialog, accessed from the **Curve editor options**

 button is available as an option on the drop-down list.

Spline selected converts this selected curve:



To this:



Arc Fit selected — Click to arc fit selected segments. This smooths curves. This displays the **Arc Fit** dialog.

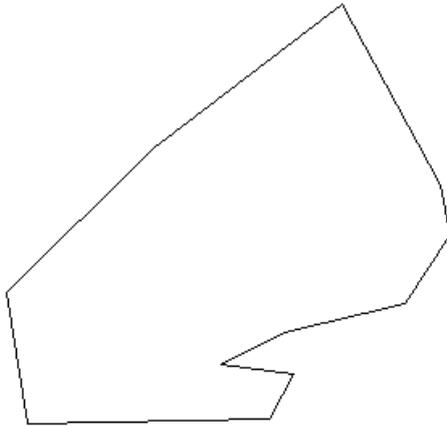


Tolerance — Enter the maximum distance between the original and fitted curves. The value specified in the **Curve Fitting Tolerances** page of the **Options** dialog, accessed from the **Curve editor options**

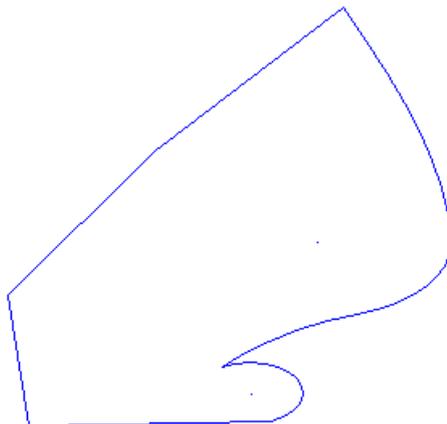


button is available as an option on the drop-down list.

Arc fit selected converts this selected curve:



To this:



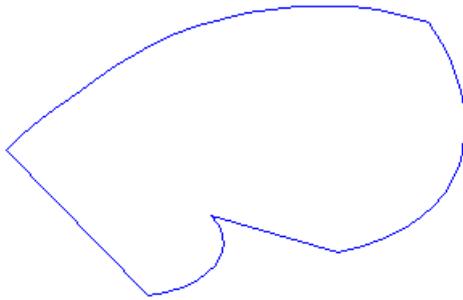
 **Polygonise selected** — Click to convert the selected segments to a series of straight lines. This displays the **Polygonise** dialog.



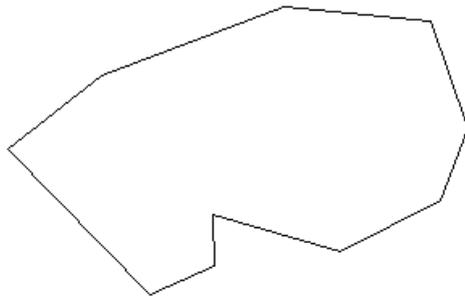
Tolerance — Enter the maximum distance between the original and fitted curves. The value specified in the **Curve Fitting Tolerances** page of the **Options** dialog, accessed from the **Curve editor options**

 button is available as an option on the drop-down list.

Polygonise selected converts this selected curve:

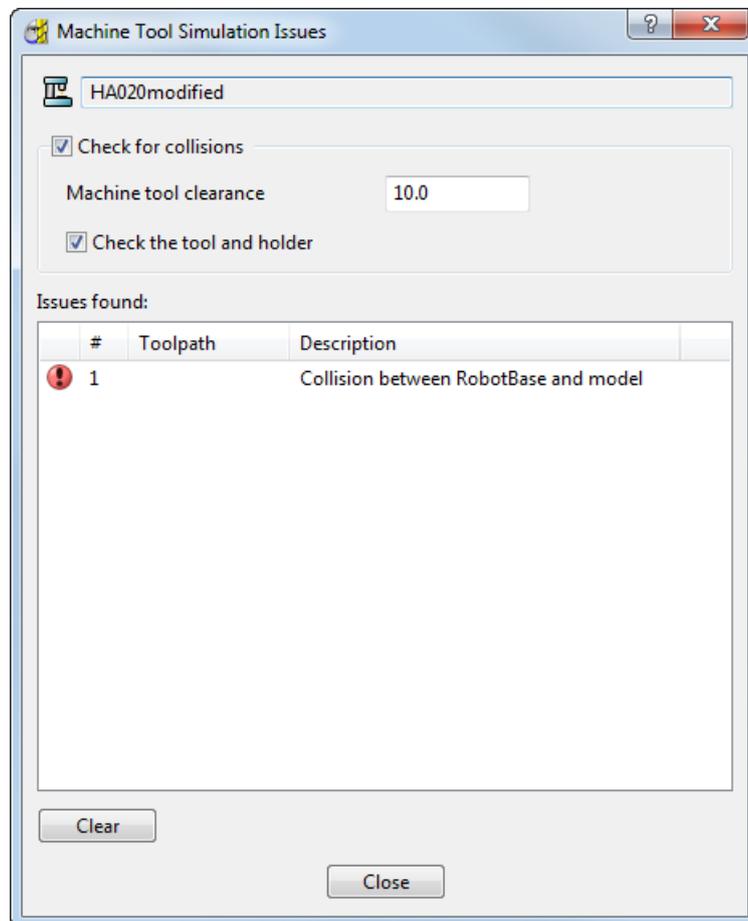


To this:



Machine Tool Simulation Issues

There is a new machine **Tool Simulation Issues** dialog, which replaces the **Machine Tool Collisions** dialog. Use this dialog to enable machine tool collision checking, specify a clearance value around the machine tool to identify 'near misses', and review any collisions or 'near misses' that occur when simulating and jogging the machine tool.



To display the dialog:

- From the **Machine Tool** toolbar, click .
- On the **Machine Tool Position** dialog, click .
- In the Explorer, right-click a **machine tool** and select **Issues**.

 HA020modified — Displays the name of the machine tool.

Check for Collisions — Click to enable collision checking.

Machine tool clearance — Enter a clearance value around the machine tool that PowerMILL takes into consideration when checking for collisions. When a machine tool axis-component and the part are within the specified clearance value, the axis-component is highlighted yellow.



PowerMILL checks for collisions and 'near misses' at fixed points. Although these points are distributed frequently, it is possible that PowerMILL may not identify small collisions that occur in between two points.

Machine tool collision-checking does not check the tool assembly for collisions. To check the tool assembly, use the **Toolpath Verification** function.

Check the tool and tool holder — Click to enable collision checking of the tool and tool holder.



If selected, the cutting part of the tool is checked against the machine, and the holder is checked against the machine and the model.

Issues found — Displays a chronological list of issues and warnings. Click on an issue in the list to move the active machine tool to the position where the issue occurs.

Clear — Click to clear the table of entries.

Maintaining tool speed after ramping

There is a new option on the **Lead feed rates** strategy page which enables you to maintain the feed rate after ramping into the stock. You can also specify the distance that this feed rate is maintained.

The screenshot shows the 'Lead feed rates' strategy page with three sections: Ramp lead in, Lead in, and Lead out. Each section has input fields for percentage and mm/min, a calculator icon, and a 'Cutting feed rate' label. The 'Ramp lead in' section includes a 'Maintain ramp feed rate' checkbox and a 'Distance' field.

Section	Percentage (%)	mm/min	Calculator	Additional Info
Ramp lead in	100.0	500.0	Yes	Plunge feed rate: 500.00 mm/min <input checked="" type="checkbox"/> Maintain ramp feed rate Distance: 0.0
Lead in	100.0	1000.0	Yes	Cutting feed rate: 1000.00 mm/min
Lead out	100.0	1000.0	Yes	Cutting feed rate: 1000.00 mm/min

To maintain the ramp feed rate, in the **Ramp lead in** area of the **Lead feed rates** strategy page, select **Maintain ramp feed rate**. Enter the distance for which the tool speed is maintained.

Aligning the tool with the toolpath

There is a new **Align tool with toolpath**  button on the **Dynamic machine control** mode-toolbar which enables you to align the tool with the toolpath. To enter the mode:

- Click **Dynamic machine control**  button on the:
 - **Tool** toolbar
 - **Toolpath** toolbar
 - **Machine tool** toolbar
- In the Explorer, right-click a tool and click **Dynamic Machine Control**.
- In the graphics area, right-click a tool and click **Dynamic Machine Control**.



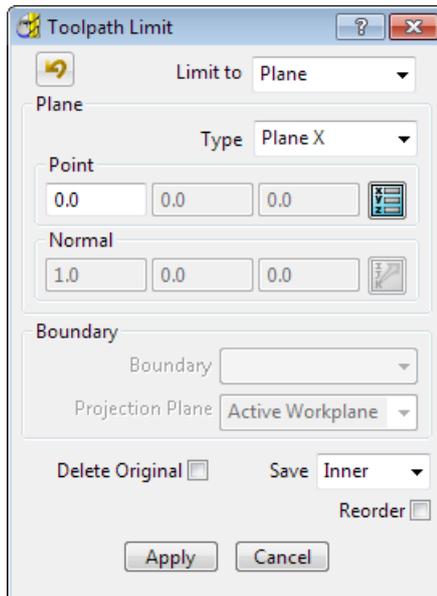
The mode is available for all tools. All of the mode functions are available when you use the following tools: ball nosed, tip radiused, end mill, tapered spherical and tapered tip. When you use other tools, such as a drill or barrel tool, the mode functions are limited.

Activating a mode disables most of PowerMILL's functionality until you exit from the mode. For more information, see [Mode-toolbars](#).

Click **Align tool with toolpath**  to align the tool axis direction with the toolpath. If the tool has a contact point, it is respected. However, it is not necessary to have the tool attached to the toolpath to realign it.

Limit toolpath enhancements

There is a new undo  button on the **Toolpath Limit** dialog which enables you to undo the last point added to a limiting polygon.



To display the **Toolpath limit** dialog:

- on the **Toolpath** toolbar, select ; or
- from the individual Toolpath context menu, select **Edit > Limit**.

To create a polygon:

- 1 Select **Polygon** from the **Limit** to list.
- 2 Sketch a polygon with any number of sides, using the mouse.
- 3 Click **Apply**.

To undo the last point added to the limiting polygon, click .

Verifying toolpaths

PowerMILL 2015 R2 contains the following changes and improvements to the verification of toolpaths:

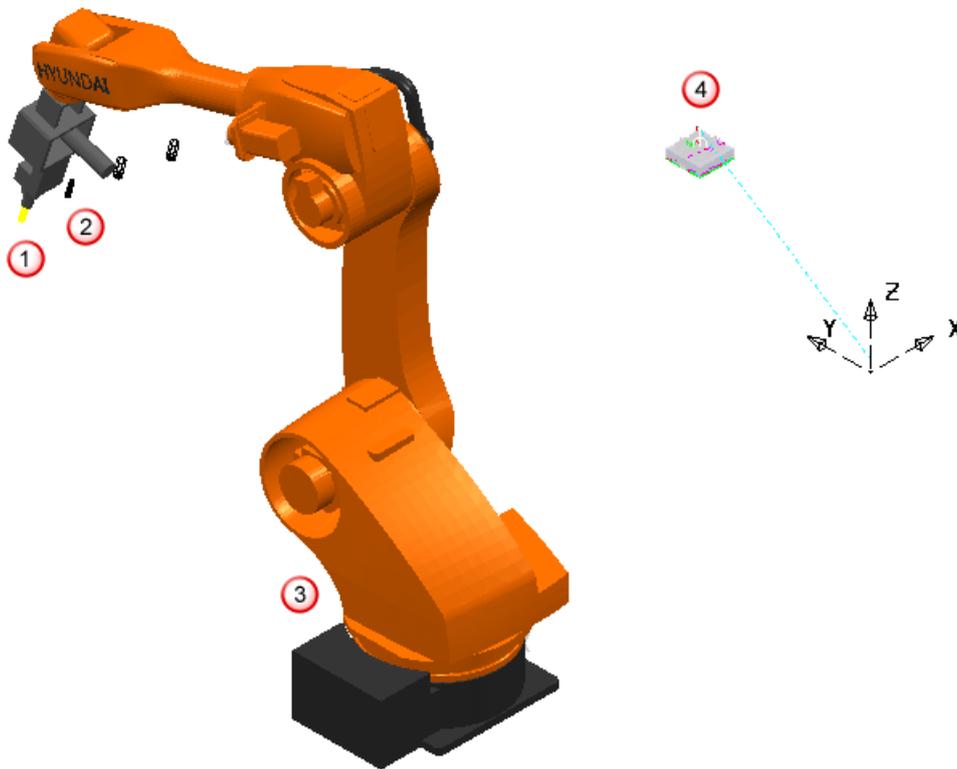
- Simulating tool changes (see page 46) — Simulate the tool change cycle of a machine tool.

Simulating tool changes

PowerMILL can simulate the tool change cycle of a machine tool. This enables you to identify issues with tool changes, such as collisions. To display 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. If you do not load an **.mtd** file, the machine tool does not load. You can still simulate toolpaths as in previous versions of PowerMILL.



The tool numbers in each toolpath determine the order of simulation.



- ① First tool change
- ② Second tool change
- ③ Robot
- ④ Model

Outputting toolpaths

PowerMILL 2015 R2 contains the following changes and improvements to the outputting of toolpaths:

- Verifying NC programs (see page 47):
 - **NC program verification** dialog (see page 48) — Verify all of your NC programs at the same time and display issues and warnings in a new dialog.
 - NC program errors in the Explorer (see page 51) — Display any issues and warnings associated with NC programs in the Explorer.
- Creating a new tool after toolpath verification (see page 52) — Always create a non-colliding tool after a collision.

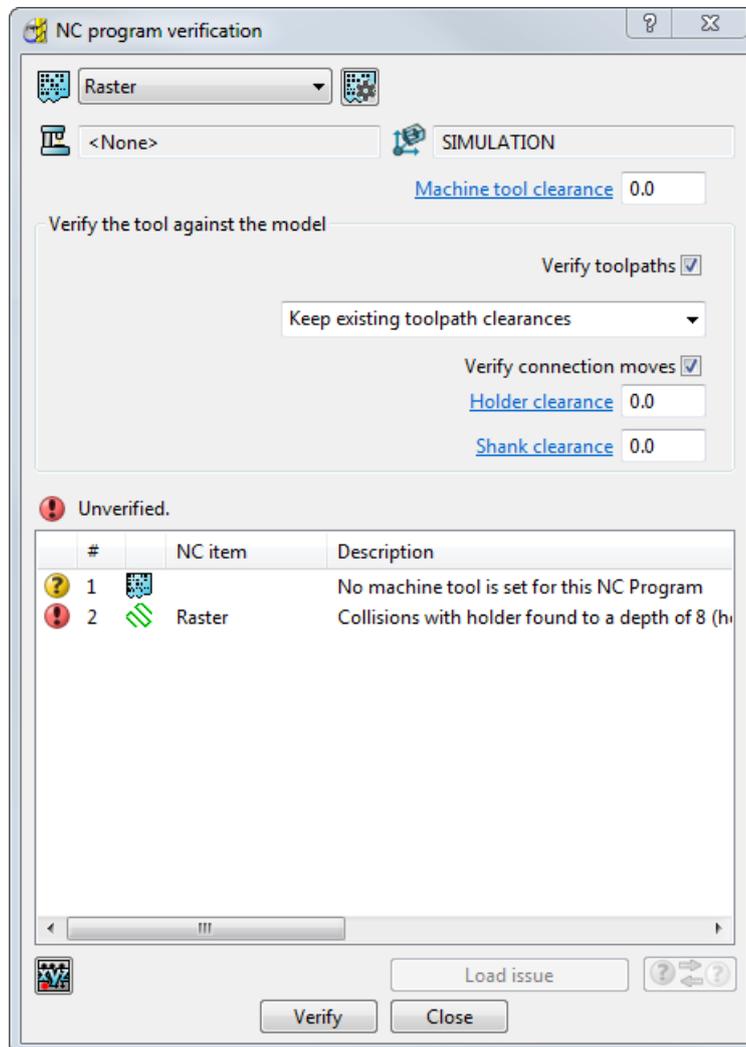
Verifying NC programs

The Explorer includes a new option that enables you to verify all of your NC programs at the same time. Any errors and collisions are displayed:

- in the new **NC program verification** dialog (see page 48). Use this to view a list issues for each NC program.
- in the PowerMILL Explorer.

NC program verification dialog

Use the **NC program verification** dialog to verify your NC programs. To display the **NC program verification** dialog, from the **NC Programs context menu**, select **Verify all**. Alternatively, to verify a single NC program, from the individual NC program context menu, select **Verify**.



 Raster — Select an NC program to verify.

 — Click to display the **NC program** dialog. Use this to view and edit the detail of the NC program.

 table-table — Displays the name of the machine tool.

 SIMULATION — Displays the name of the workplane.

Machine tool clearance — Enter the minimum distance between the model and the machine tool. If you want to measure this, click **Machine tool clearance** to display the **Measure** mode toolbar.

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

Increase toolpath clearances — Select to verify toolpaths that have been previously verified with lower clearance values.

Keep existing toolpath clearances — Select not to verify toolpaths that have been previously verified with lower clearance values.



*If you select **Keep existing toolpath clearances**, warnings are still displayed in the Verification list.*

Verify connection moves — Select to include connection moves in the verification. This applies to the tool and the model.

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.*

Safety status indicator — Displays an icon which identifies the most severe issue found in verification.



— Collisions found



— Other issues, for example, unverified toolpaths.



— Gouge checked



— Gouge, and holder checked



— Gouge, holder, and machine tool checked

Verification list — Displays a chronological list of issues and warnings.

Issues are added to the list when:

- there is a collision between the tool (or the shank, or holder) and the model.
- there is a machine tool collision.
- the machine tool cannot reach a location.
- the machine tool exceeds its limits.

Warnings are added to the list when:

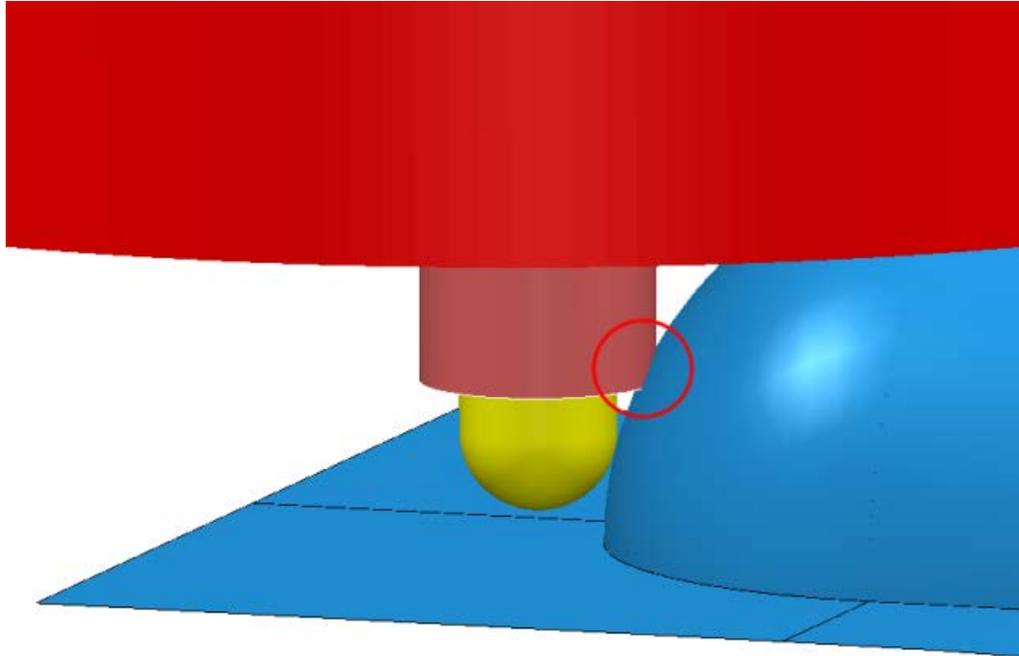
- a toolpath is not verified.
- a toolpath could not be verified.
- a toolpath is verified, but the NC program has higher clearance values.

- a connection is not verified.
- the machine tool overlaps with the clearance of another component.



— Click to display the **Machine Tool Position** dialog.

Load Issue — Select an error from the verification list and click **Load issue** to display the error on the model.



— Click to ignore the selected warnings. In the verification list, the selected warning changes from  to .

Verify — Click to refresh the verification of the NC program. This is useful when you edit the options in this dialog.

Close — Click to close the dialog.

Clearing verification

You can clear verification for the machine tool list and the machine tool status for:

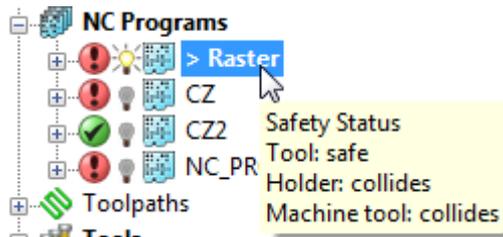
- all NC Programs — From the **NC programs** context menu, select **Clear all NC Program verification**.
- a single NC program — From the individual **NC program** context menu, select **Clear Verification**.



This does not change the verification status of the toolpath or the model.

NC program errors in the Explorer

The Explorer displays the issues and warnings associated with NC programs. Hover over an NC program to display the **Safety Status**.



The icons display the most severe issue in the NC program:

Issues are added to the list when:

- there is a collision between the tool (or the shank, or holder) and the model.
- there is a machine tool collision.
- the machine tool cannot reach a location.
- the machine tool exceeds its limits.

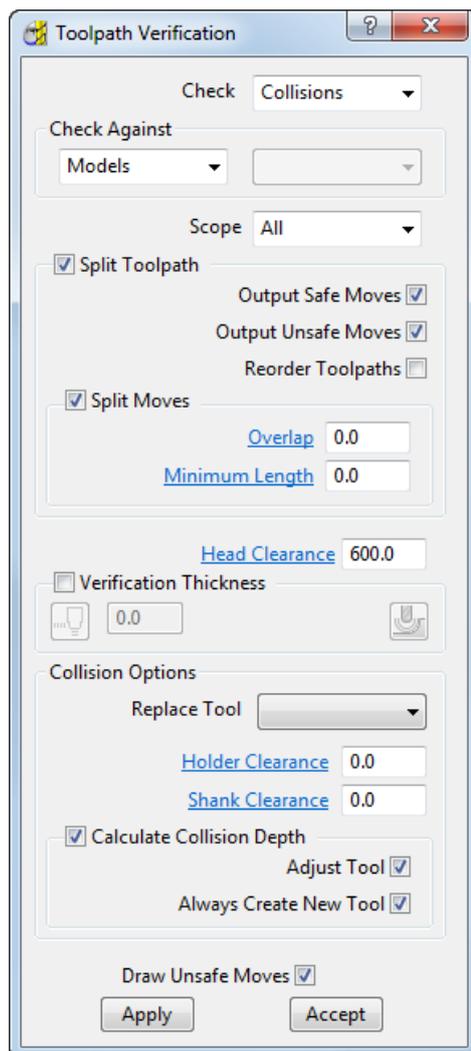
Warnings are added to the list when:

- a toolpath is not verified.
- a toolpath could not be verified.
- a toolpath is verified, but the NC program has higher clearance values.
- a connection is not verified.
- the machine tool overlaps with the clearance of another component.

- ❗ — Collisions found
- ❓ — Other issues, for example, unverified toolpaths.
- ✅ — Gouge checked
- ✅ — Gouge, and holder checked
- ✅ — Gouge, holder, and machine tool checked

Creating a new tool after toolpath verification

There is an **Always Create New Tool** option in the **Collision Options** area of the **Toolpath Verification** dialog. Select this option to always create a non-colliding tool after a collision has been detected. When **Always Create New Tool** is selected, PowerMILL does not search for a safe tool in the **Tools** list as it did in previous versions, it automatically creates a new tool with safe properties (overhang, cutting length, collision depth). To display the **Toolpath Verification** dialog, select **Verify > Toolpaths** from the **Toolpath** context menu.



User interface

PowerMILL 2015 R2 contains the following changes and improvements to the user interface:

- **Customise Keyboard Shortcuts** dialog (see page 53) — Create keyboard shortcuts to perform PowerMILL commands without the use of a mouse.
- Adding expressions to custom toolbars (see page 56).
- **Select Concave Radii** dialog (see page 58) — Select concave or convex areas of the model.
- Editing the width of the toolpath (see page 60) — Edit the display width of the toolpath in pixels.

Customise Keyboard Shortcuts dialog

You can use keyboard shortcuts to perform PowerMILL commands without the use of a mouse. Use a combination of keys on the keyboard to perform tasks that are usually accessed on toolbars, menu bars, or using macros. Keyboard shortcuts comprise either:

- a function key (F2–F12).



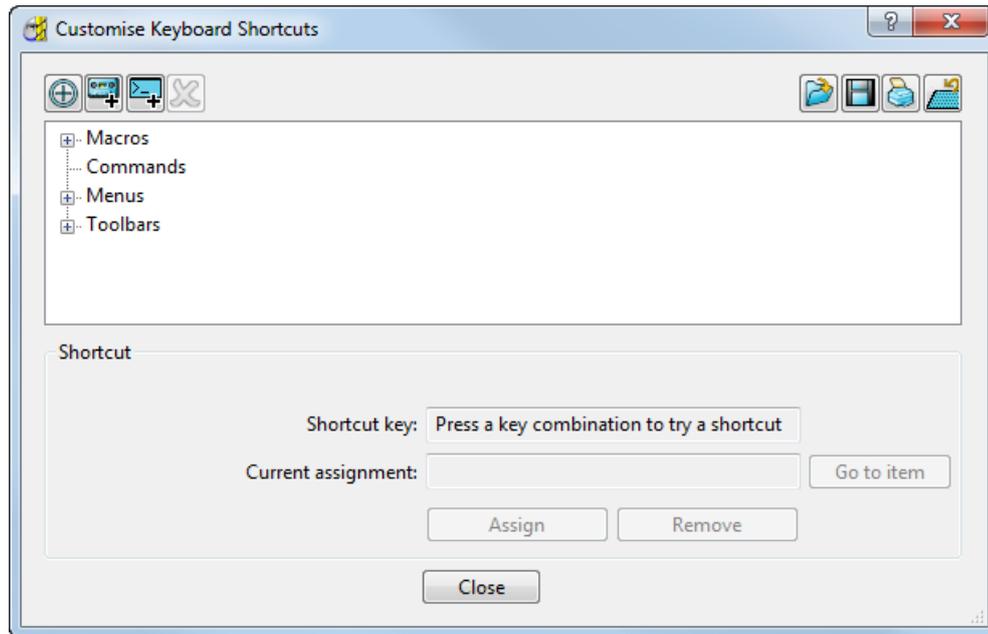
*You cannot use F1 as this is used to display the **Help** window.*

- a control key (Ctrl, Alt, or a combination of both), and one of the following keys:
 - alphanumeric keys— a-z, 0–9, punctuation keys and symbol keys.
 - navigation keys — for example, End, Page Up and Insert.



You can also use the Shift key in combination with any of the above keys.

Use the new **Customise Keyboard Shortcuts** dialog to assign keyboard shortcuts to commands. To display this dialog, select **Customise Keyboard Shortcuts** from the **Tools** menu.



 **Control selector** — Click and drag over a toolbar button. The button is selected and displayed in the menu list, and in the **Toolbar** field.

 *While dragging, the mouse changes to \oplus and places a box around the selected button.*

 **Add macro shortcut** — Click to add a new macro to the menu list and display the **Macro** field.

 **Add command shortcut** — Click to add a new command item to the menu list and display the **Command** field.

 **Delete** — Click to remove a selected shortcut.

 **Import shortcuts** — Click to display the **Select keyboard configuration file to import** dialog. This enables you import an existing keyboard configuration.

 **Export shortcuts** — Click to display the **Export keyboard configuration to a file** dialog. This enables you to save your keyboard configuration.

 **Print shortcuts** — Click to display a list of your keyboard shortcuts in the PowerMILL HTML browser and display the **Print Preview** window.

 **Reset** — Click to restore the default keyboard settings.

Menu list — expand the:

- **Macros** branch to display the file path, with a file selector.
- **Commands** branch to display your commands.
- **Menus** branch to display a list of PowerMILL menus that you can assign a keyboard shortcut to.
- **Toolbars** branch to display a list of PowerMILL toolbars that you can assign a keyboard shortcut to.

The **Shortcut** field displays the file path of the item you selected.



The name of the shortcut field changes, depending on whether a macro, command, menu or toolbar item is selected.

Shortcut key — Displays the keyboard shortcut assigned to the selected item in the menu list. To create a new keyboard shortcut, press a combination of approved keyboard keys and click **Assign**.

Current assignment — If a keyboard shortcut entered in the **Shortcut key** field is already assigned to an item in the menu list, this field displays the current assignment of that keyboard shortcut.

Go to item — If a shortcut is displayed in the field, click to locate the item in the menu list.

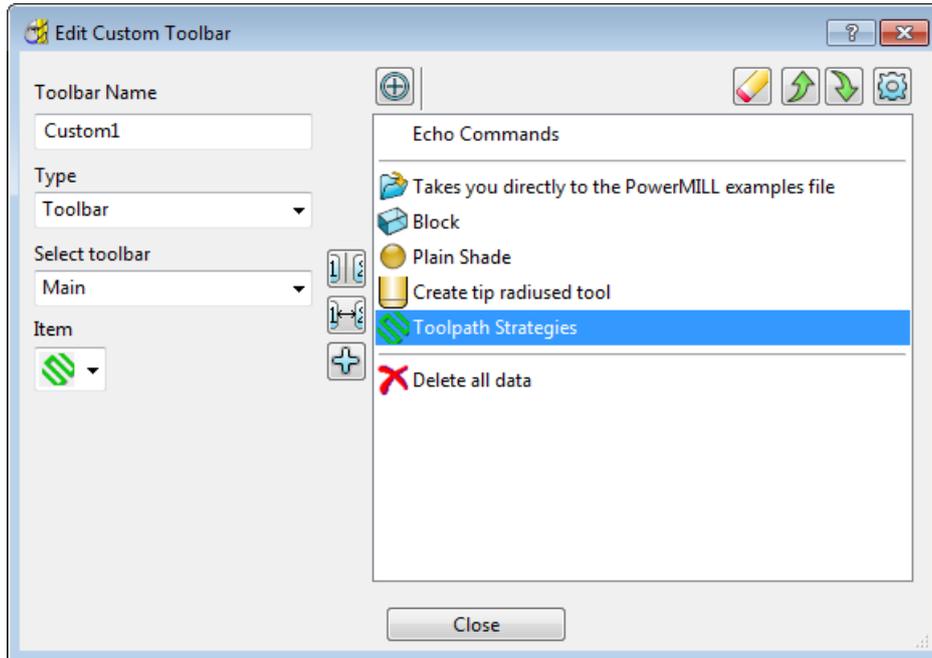
Assign — Click to assign a shortcut to an item.

Remove — Click to remove a shortcut from an item.

Close — Click to close the dialog.

Adding expressions to custom toolbars

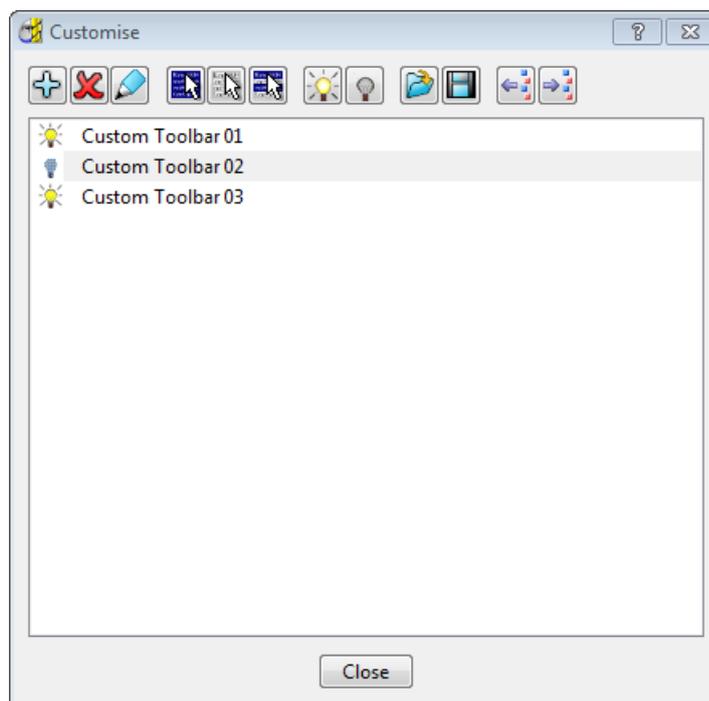
There is a new **Expressions** option in the **Edit Custom Toolbar** dialog that enables you to add expressions to custom toolbars.



The toolbar automatically updates when you change an expression.

To display the **Edit Custom Toolbar** dialog:

- 1 Select **View > Toolbar > Custom> Customise**.



2 Click  to create a toolbar.

3 Double-click the toolbar in the list.

To add an expression to a custom toolbar:

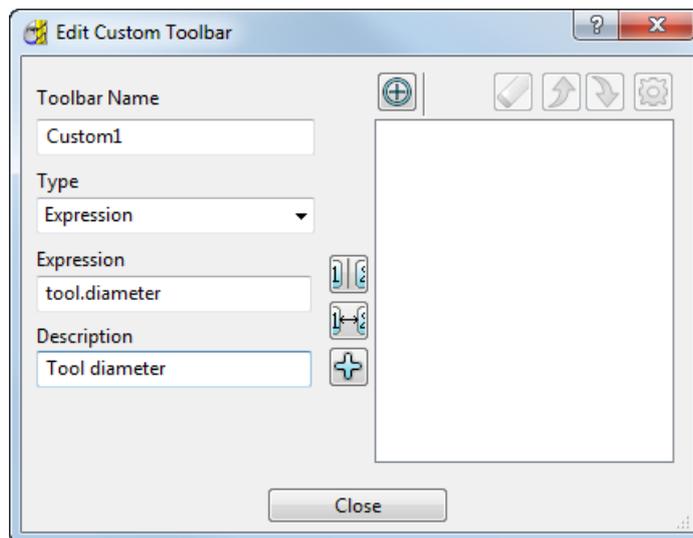
1 Select **Expression** from the **Type** list.

2 Enter a command, for example **tool.diameter**.



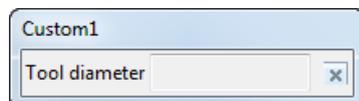
Use the command window to find the command you need.

3 Enter a description, for example **Tool diameter**.



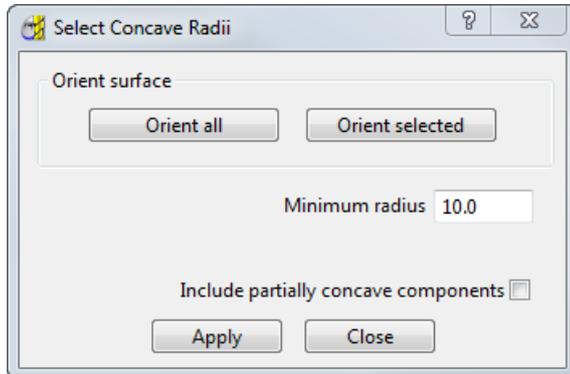
4 Click . This adds the menu option to the custom toolbar.

5 Select **View > Toolbar > Custom > Custom1** to display your custom toolbar.



Select Concave Radii dialog

Use the **Select Concave Radii** dialog to select small, concave or convex surfaces in a model. To display the dialog, right-click the model and select **Selection Tools > Select Concave Radii**.



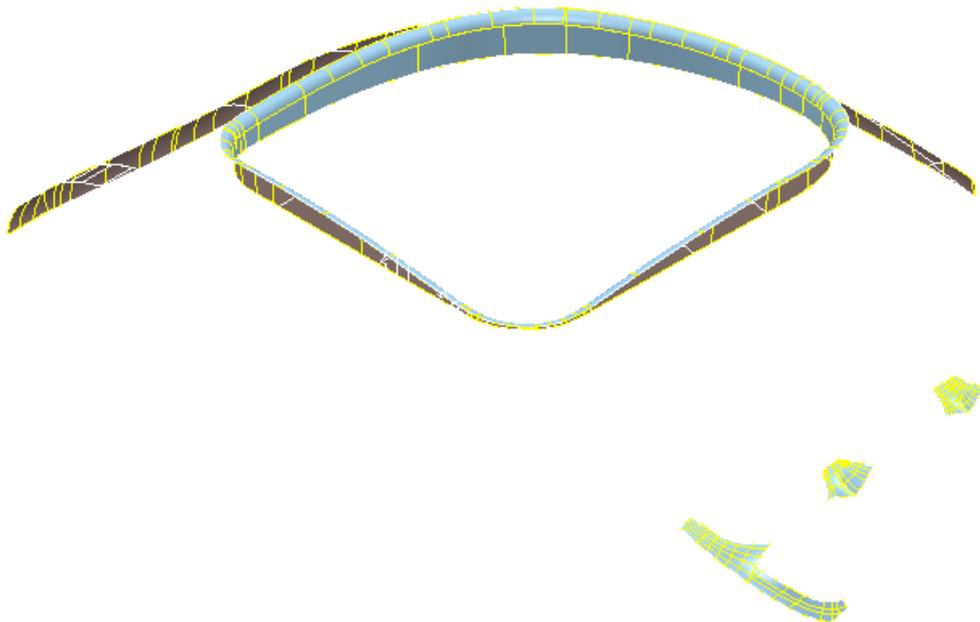
The **Orient surface** options enable you to correctly orient your model.

Orient all — Click to automatically reverse the orientation of all the surfaces which are incorrectly oriented in the current view.

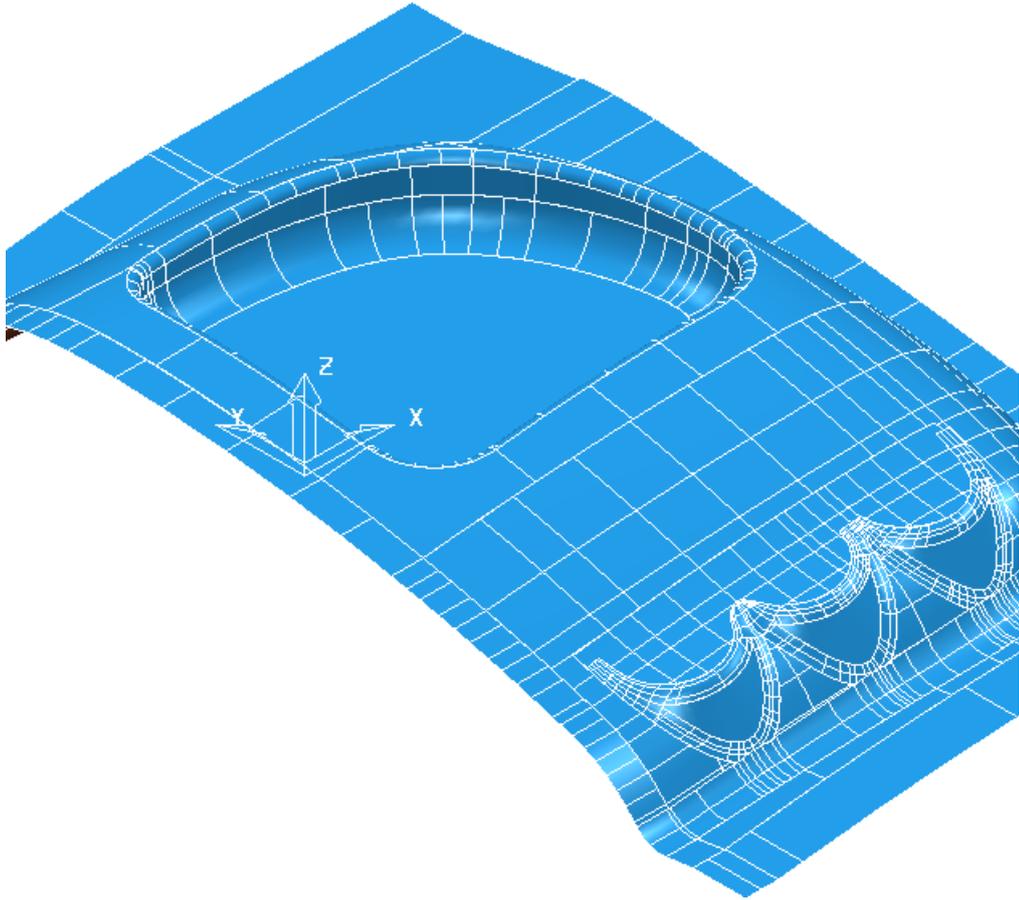
Orient selected — Click to automatically reverse the orientation of all the selected surfaces which are incorrectly oriented in the current view.

Minimum radius — Enter the minimum radius of the concave surfaces you want to select.

Include partially concave components — When selected, selects any surface which has a concave region.



When deselected, any surface which is wholly concave is selected.

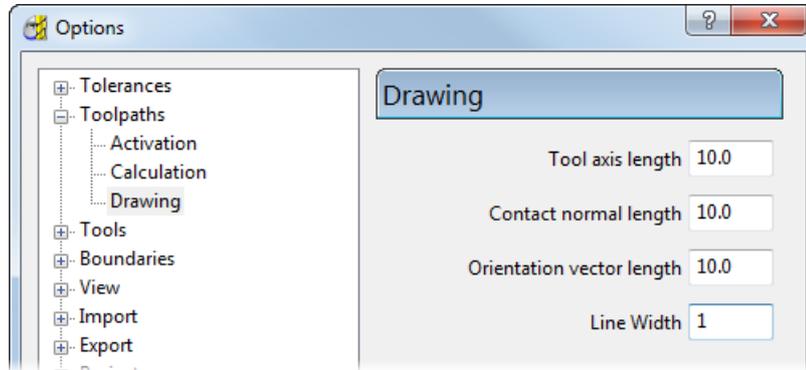


Selecting either of these options adds to your current selection. So the concave surfaces are selected, as well as any previously selected components.

You can use this dialog with the new **Minimum radius plane** list, which is located in **Tools > Options > View > 3D Graphics**. This list enables you to shade the model from the perspective of the **Workplane** (the default setting) as opposed to the **View** (the default setting in previous versions of PowerMILL).

Editing the width of toolpath lines

There is a new option on the **Drawing** page of the **Options** dialog which enables you to edit the line width of the toolpath in pixels. This enables you to increase the visibility of toolpaths when presenting on large screens.



To display the **Drawing** page, from the main menu, select **Tools > Options > Toolpath** and select the **Drawing** page. In the **Line Width** field, enter the width of the toolpath line in pixels.

Automating PowerMILL

PowerMILL 2015 R2 contains the following changes and improvements to automation:

- Macro enhancements (see page 61) — Display entities and hide GUI items when you use mode toolbars.
- **Macro Debugger** enhancements (see page 62) — Edit variable assignments in the **Macro Debugger** dialog.

Macro enhancements

PowerMILL 2015 R2 includes new parameters to improve the efficiency of user-menu macros. The parameters enable PowerMILL to:

- identify the last selected entity; and
- display the total number of selected entities and the entity PowerMILL is currently processing.

You can use the parameters to write macros that instruct PowerMILL to execute specific lines of code before and after it processes the first and last entity in a loop. Previously, PowerMILL could not identify the first and last entity so it executed the entire macro for every selected entity.

You can use PowerMILL modes, such as the **Curve editor** or the **Workplane editor**, without displaying:

- Toolbars
- Mode-toolbars
- Dialogs
- Graphics

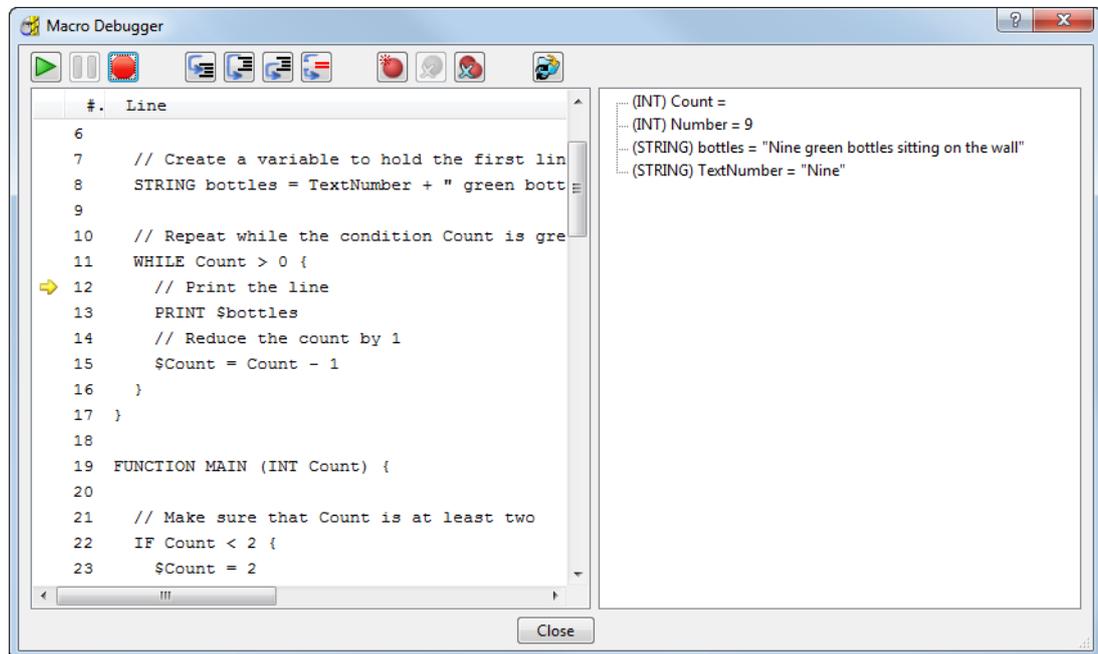
Use the NOGUI command to run the equivalent of GRAPHICS LOCK when the mode starts.



For more information, see the 'Macro Programming Guide'.

Macro Debugger enhancements

Use the **Enter parameter value** dialog to edit variable assignments in the **Macro Debugger**.



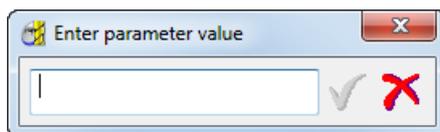
To display the **Macro Debugger** dialog, in the Explorer, right-click:

- the **Macro** menu and select a macro to debug; or
- a macro and select **Debug**.

The left pane shows the macro you are debugging. The right pane shows the variable assignment at that point in the macro.

To edit a variable assignment:

- 1 In the right pane, double click a variable assignment.
- 2 Enter a value in the **Enter parameter value** dialog.



- 3 Click  to accept the changes and close the dialog.

Index

C

- Channel machining • 14
- Collision avoidance • 29
- Customisation
 - Custom toolbars • 56
 - Customise Keyboard Shortcuts dialog • 53
 - Expressions • 56

D

- Drilling • 34
 - Approach page • 35
 - Area clearance • 34
 - Collision checking • 31
 - Definition • 35
 - Drilling from the stock • 30
 - Feature sets • 35
 - Hole diameter • 35
 - Stock • 30, 31
 - Vortex • 34

E

- Explorer • 47, 51

K

- Keyboard shortcuts • 53

N

- NC program verification • 47

- Clearing verification • 50
- Errors • 51
- NC program verification dialog • 48
- Safety status • 51

O

- Orientation vectors • 36
 - Machine axis control • 37
 - Specify changes • 36

R

- Rib machining • 13, 14

S

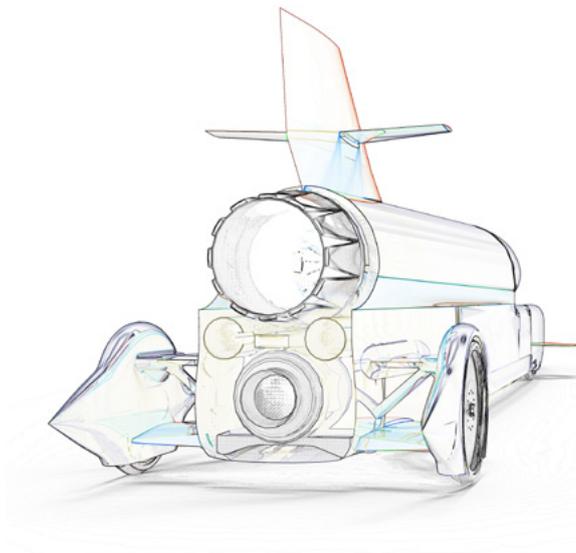
- Simulation • 46

T

- Tool changes • 46
- Tool database • 4
 - Connections • 9
 - Database manager dialog • 5
 - Edit database tools dialog • 10
 - Searching the tool database • 6
 - Tags • 5

V

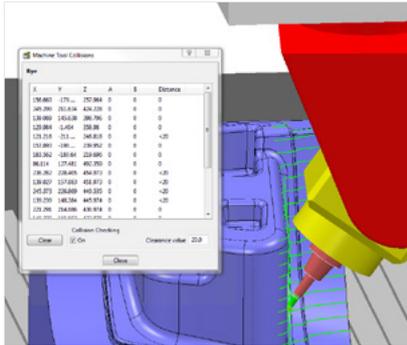
- Verification • 47



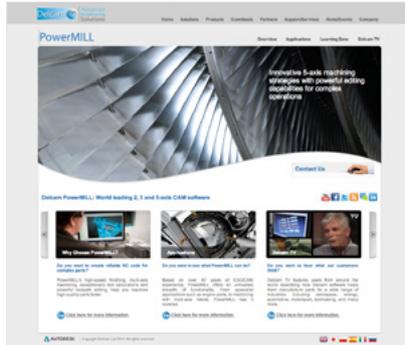
PowerMILL 2015 R2

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

www.powermill.com



PowerMILL 2015

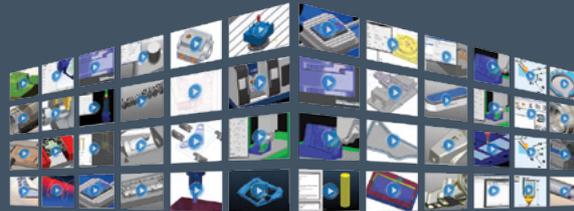


www.powermill.com



www.delcam.tv

Unlock the power
of Delcam software



LEARNING ZONE

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



Powering your productivity

Delcam

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

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

