Delcam Advanced Manufacturing Solutions

PowerMILL 2012 R2

What's New

Powering your productivity

Delcam Advanced Manufacturing Solutions

www.delcam-ams.com

Whether you are a new or long-term user of Delcam products, you may not be aware of the complementary technologies available from Delcam that could improve your company's productivity.

Delcam Advanced Manufacturing Solutions offers a complete and diverse range of CADCAM software solutions to integrate every aspect of your production life-cycle – from designing a complex concept, to manufacturing it and inspecting the final output – providing you with unrivalled speed, flexibility, and ease-of-use throughout your product development process.

Adding to your suite of Delcam solutions minimises the impact on your daily production operations while also enabling you to enter new markets or automate your processes with our advanced functionality.

All products work independently or together as one complete manufacturing solution tailored to your specific needs.

PowerSHAPE

www.powershape.com

PowerSHAPE provides a complete environment to manipulate surface form, build from wireframe, and add solid features for prismatic parts.

- Import/export Parasolid-based systems with no translation.
- Create complex molds from solid models, complete with split surfaces.
- Powerful surface- and face-editing tools quickly repair imported data.
- Tools to perform non-feature-based edits quickly and efficiently.

PowerMILL

www.powermill.com

PowerMILL is the world's leading specialist NC CAM software for the manufacture of complex shapes, providing advanced-machining strategies to minimise machining time and maximise finish quality.

- World-leading high-speed machining strategies.
- Advanced 5-axis machining techniques.
- Support for 64-bit platforms and multi-threading.
- Innovative collision-avoidance methods.
- Powerful toolpath editing and tool-axis stabilisation.

FeatureCAM

www.featurecam.com

FeatureCAM is a unique CAM system that uses feature-based and knowledge-based technologies for automated machining, minimising programming times for mills, lathes, turn/mill, and wire machines.

- Easy to use.
- Single interface for multiple machine platforms.
- Powerful turning and milling operations from 2.5D to 5-axis.
- Multi-threading capabilities.

Delcam for SolidWorks

www.delcamforsolidworks.com

Delcam for SolidWorks is a SolidWorks Certified Gold Product that revolutionises CAM programming inside SolidWorks.

- Integrates the feature-based technology from FeatureCAM.
- Multi-threaded toolpath algorithms from PowerMILL.
- 2-axis, 3-axis, and 5-axis positional milling and drilling.
- Turning, turn/mill, and wire EDM capabilities.
- Automatic selection of cutting tools, machining strategies, and feeds and speeds.
- Exceptional toolpath-calculation speeds.
- Set-up wizards.
- Full-machine simulation.

PartMaker

www.partmaker.com

PartMaker applies a patented Visual Programming approach to automate the programming of multi-axis Swiss-type lathes and Turn-Mill Centres.

- Easier programming of turning with live tooling via the Divide-and-Conquer programming approach.
- Automatic process synchronisation.
- Vivid 3D simulation and crash detection.
- Wide array of proven post processors for Turn-Mill Centres and Swiss-type lathes.

PowerINSPECT **www.powerinspect.com**

PowerINSPECT leads the way in today's inspection market. It delivers a complete CAD-based inspection solution that can accept data from all types of hardware, including manual and CNC coordinate measuring machines, portable arms, optical measuring devices, and CNC machine tools.

- Part comparison against all mainstream CAD formats.
- Support for all types of measuring devices.
- Market-leading inspection reports that are quick to create and easy to understand.
- IGES export of measured features, including digitized curves.
- Additional modules for part alignment, laser line inspection, and tube inspection.

Delcam ArtCAM **www.artcam.com**

ArtCAM is a unique application that combines the benefits of computerised design and CNC machining in a simple-to-use format to create decorative products from artwork.

- Import 3D models, clipart, and other CAD system formats.
- Add geometric shapes, weaves, and textures.
- Choose from extensive visualisation and rendering materials.
- Use the comprehensive and customisable tool database for fast toolpath-calculation.
- Powerful toolpath simulation verifies machining times and materials.
- Design in the 3D view.

Delcam also provides a range of healthcare CADCAM solutions for the dental, custom orthotic insoles, medical implants, and orthopaedic footwear industries, as well as CADCAM solutions for footwear design and manufacturing.

You can connect with Delcam in a variety of ways:

visit: www.delcam.com | watch: www.delcam.tv | learn: www.delcam.tv/lz

PowerMILL 2012 R2

What's New

Release Issue 1

PowerMILL

Copyright © 1996 - 2012 Delcam plc. All rights reserved.

Delcam plc 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 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 plc 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

Licenses

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

PowerMILL 2012 R2. Published on 30 January 2012

Contents

Summary of new features 1

Toolpath preparation....1
Toolpath generation....2
Toolpath verification....2
Toolpath output....3
User interface....3
Automation....3
General enhancements....4

Toolpath preparation 5

Transformation enhancements....5
Geometry transformations....5
Feature tolerance....6
Drill length....7
Create a workplane along the axis of a hole....7
Creating a workplane from three arc points....8
Tool assembly zoom and pan controls....10

Toolpath generation 11

Area clearance step cutting....11
Step cutting (area clearance)....14
Machine axis control....18
Polar milling....19
Rest roughing improvements....25
Flat machining improvements....26
Enhancements to move toolpath start points....27
Warning about plunge moves into stock....29
Corner clearance enhancements....30
Steep and shallow finishing enhancements....31
Spiral profile drilling....32
Direction for Parametric spiral finishing....35
Drilling thickness....35
Toolpath transform enhancements....36

Toolpath verification 38

Machine maximum stock....38

Toolpath output 41

Reconnecting machine tools 41
Machine tool opacity 43
Controlling the opacity of a machine tool and part 43
Machine tool simulation enhancements 45

User interface 47

Stock Models toolbar 47
Using the Stock Model Toolbar 49
Individual Stock Model settings 51
Individual Tool State dialog 53
Raceline smoothing 54
Customise shading colours 55
Measurer 56

Automation 57

Using plugins 57
Tools > Plugins 59
Macro programming enhancements 61
Macro debugger 61

General enhancements 64

Copying toolpath and tool data to the clipboard 64
Adding a custom entry to an entity context menu 65

Index 69

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 2012 R2 offers all of the original features of PowerMILL 2012, but with numerous improvements. The most significant improvements are described in this document.

Toolpath preparation

- There are three enhancements to transformations:
  - PowerMILL now stores transformed entities in the same folder as the initial entity (see page 5).
  - You can choose the origin for transformations (see page 5).
  - There is a new **Geometry Transform** toolbar for transforming boundaries and patterns (see page 5).
- The closure tolerance is reinstated on the **Feature** dialog (see page 6).
- You can define either the full length of the drill tool or the length without the tip of the tool (see page 7).
- A new way to create a workplane along the axis of a hole using the new **Workplane from three arc points** option (see page 7).
- You can zoom in the **Tool Assembly** view (see page 10).

Toolpath generation

- There is a new **Step cutting** option which minimises terracing when creating area clearance toolpaths with a large stepdown (see page 11). It rest machines the terraces remaining from a large area clearance toolpath with the same tool in the same toolpath.
- You can now create milling toolpaths using polar rather than Cartesian coordinates (see page 18).
- You can now control toolpath fragmentation when rest roughing from a stock model (see page 11).
- When **Flat machining**, you can chose to ignore the parts of the model containing triangles (see page 26).
- You can make multiple changes to toolpath start points at one time (see page 27).
- If a toolpath plunges into stock material PowerMILL now displays a warning message (see page 38).
- **Corner clearance** enables you to choose the angle which is considered as a corner (see page 30).
- **Steep and shallow finishing** now has the shallow options on the main page (see page 31).
- You can create spiral profile drilling toolpaths (see page 32).
- For parametric spiral toolpaths, the **Cut direction** options are now **Clockwise** and **Anticlockwise,** which more accurately reflect the machining style (see page 35).
- The thickness specified on the main **Drilling** strategy page is now different to the thickness specified on the **Automatic verification** page of the drilling strategy (see page 35).
- You can now choose whether to append all transformed toolpaths into one composite toolpath or keep them as individual toolpaths (see page 36).

Toolpath verification

When verifying a toolpath with a shank or holder collision, PowerMILL now creates an improved toolpath which removes as much material as possible and avoids collisions (see page 38).

Toolpath output

- You can reconnect the machine tool model to a project (see page 41).
- You can control the opacity of a machine tool (see page 41).
- There are some general enhancements to machine tool simulation (see page 45):
  - Robot simulation is faster and more robust.
  - You can choose between toolpath point-to-point simulation mode or a feed rate simulation mode.
  - You can now move the machine tool to its home position.

User interface

- There are several changes to the **Stock Model** user interface:
  - A new **Stock Model** toolbar makes it easier to work with **Stock Models** (see page 49).
  - Individual stock model menu options are now available in a single **Stock Model** settings dialog (see page 51).
  - Individual tool state menu options are now available in a single **Stock Model Tool State** dialog (see page 53).
  - You can now drag and drop to rearrange stock model states in the explorer.

  Rearranging stock model states invalidates stock model calculations.
- **Raceline smoothing** replaces **Smoothing allowance** on the **High speed machining** page of strategy dialogs (see page 54).
- You can now edit the colours used to denote undercut regions and minimum tool radius shading (see page 55).
- The new **Measure** tab replaces the **Circle** and **Line** tabs (see page 56).

Automation

- You can now include **Plugin** programs to provide additional functionality within PowerMILL. Plugins are written outside the main PowerMILL application but can interact with it (see page 57).

- There are several enhancements to the **Macro programming language** (see page 61).
- There is a new **Macro debugger** to test, debug, step-through and track variables in your macros (see page 61).

General enhancements

- PowerMILL now makes more effective use of memory (see page 64).
- You can export models as **.stl** files even when no machine operations have taken place (see page 64).
- You can copy toolpath and tool data into the Windows clipboard (see page 64).
- You can add a custom menu option to an individual entity context menu (see page 65).
- You can now add custom options to PowerMILL entity context menus (see page 65).

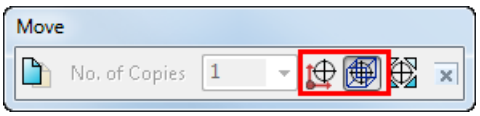
Toolpath preparation

Transformation enhancements

There are two enhancements to transformations:

- PowerMILL now stores transformed entities in the same folder as the initial entity.
- In previous versions, the default origin for transformations was the centre of the bounding box containing all the entities. Now, there is a second option of the origin of the active workplane.

  All the transform toolbars (**Move**, **Rotate**, **Mirror**, **Multiple Transform**) on curve, workplane and toolpath transforms have two additional options. For example, looking at the **Move** toolbar:

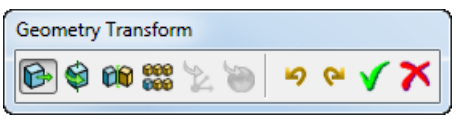


Workplane origin - when selected, the active workplane is the origin. If no workplane is active then the global coordinate system is the origin.

Bounding box origin - when selected, the origin is the centre of the bounding box containing all the entities.

Geometry transformations

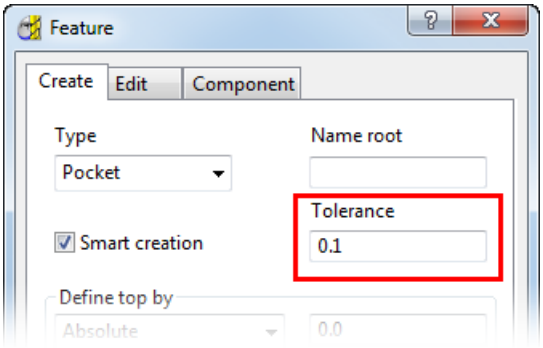
There is a new **Geometry Transform** toolbar (available from the individual boundary/pattern **Edit > Transform** menu) for transforming boundaries and patterns. This toolbar is similar to the **Toolpath Transform** and **Workplane Transform** toolbars.



*You can still transform boundaries and patterns from the **Curve Editor** toolbar (available from the individual boundary/pattern menu).*

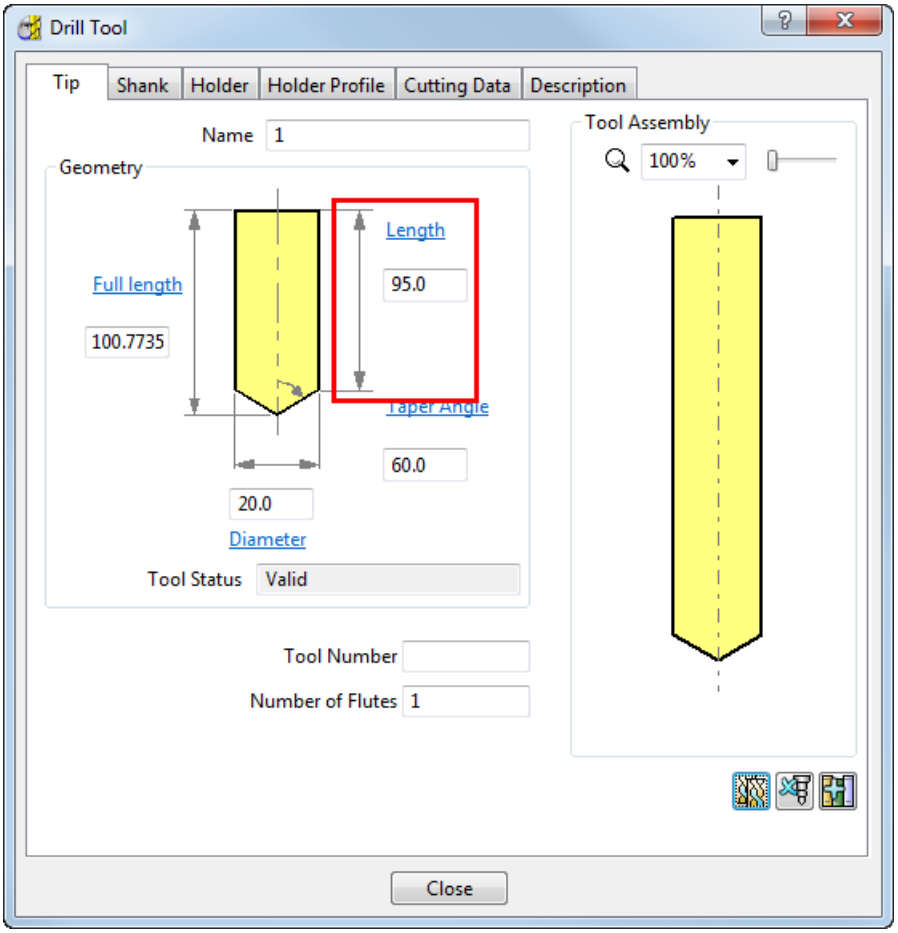
Feature tolerance

The closure tolerance is reinstated on the **Feature** dialog. In PowerMILL 2012 you could change the **Closure Tolerance** only on the **Options** dialog (available from the **Tools > Options > Tolerances > Wireframe** menu).



Drill length

The **Drill Tool** dialog now lets you define either the full length of the drill tool or the length without the tip of the tool. In previous versions, you could specify only the full length.



Create a workplane along the axis of a hole

There is a new option to create a workplane at the centre of an arc (or a circle) by selecting three points on the arc (or on the circumference of a circle). This is useful when creating a workplane at the centre of a hole with the principal editing axis aligned with the hole.

To use this option either:

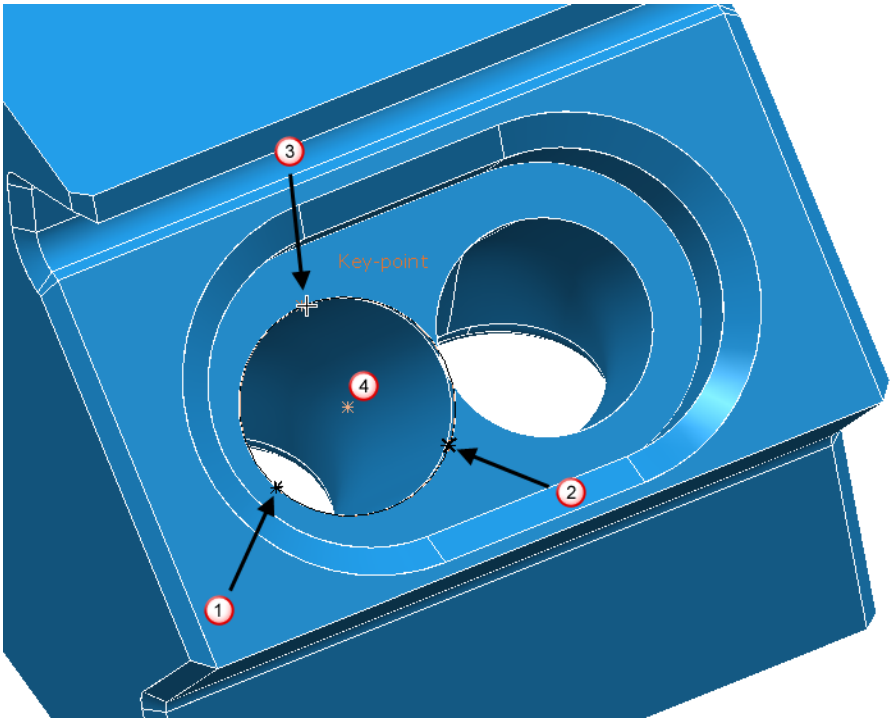
- Click the **Workplane from three arc points** button on the **Workplane** toolbar; or
- Select the **Create and orientate Workplane > Workplane from three arc points** option on the **Workplane** context menu.

Creating a workplane from three arc points

This example shows how to create a workplane at the centre of a hole, with the Z axis aligned with the hole.

1 On the **Workplane** pull-out toolbar, click the **Workplane from three Arc Points** button.

2 Click three points on the circumference of the circle that definethe hole (points 1, 2, and 3).

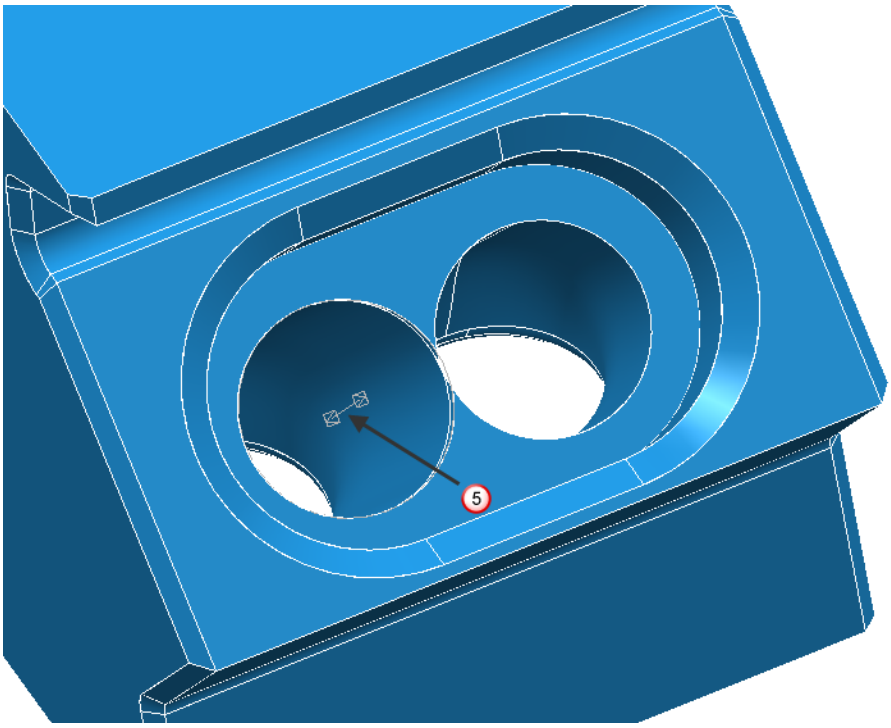


PowerMILL calculates the centre of the circle (point 4) which determines the origin of the workplane, and displays the **Confirm Origin** dialog.

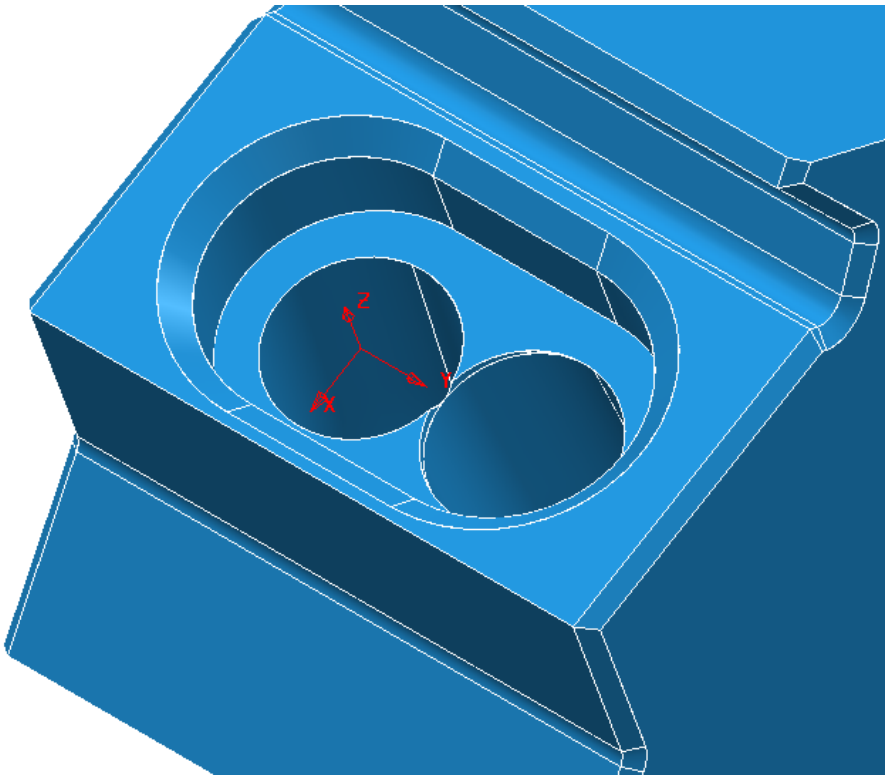
3 Click **Accept**.

You can change the location of the origin in this dialog.

4 In the graphics area, select the direction of the Z axis (5). There are two options, either into or out of the hole.



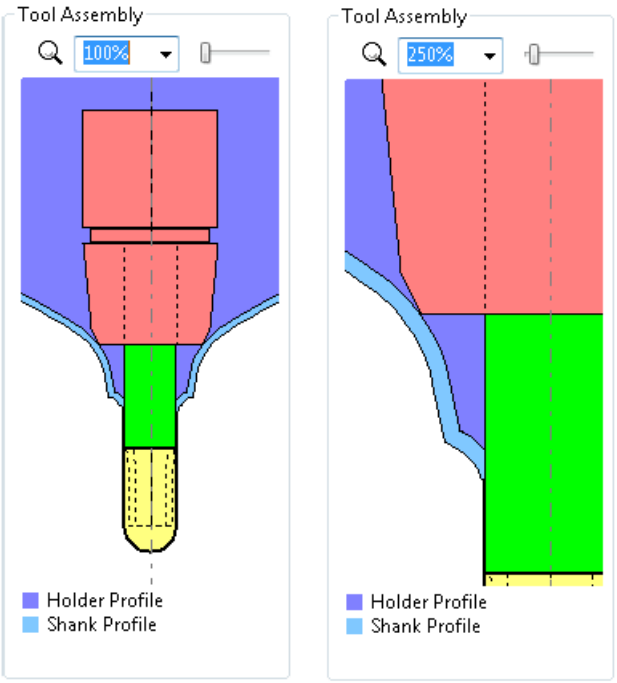
This creates the workplane.



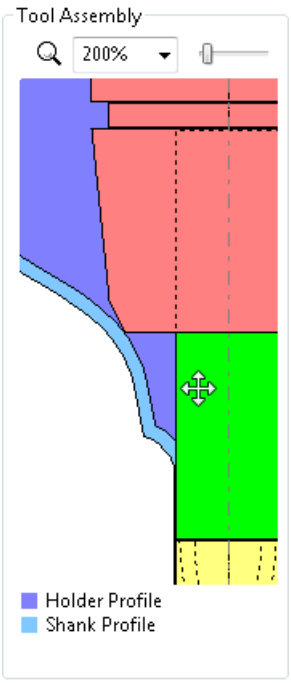
Tool assembly zoom and pan controls

You can now zoom into the **Tool Assembly** view. The zoom control in the **Tool Assembly** area of a toolpath dialog enables you to view the tool assembly in more detail.

Select the zoom level from the list, or slide the bar to select the zoom level you want.



When the cursor hovers over the tool assembly, it changes to . Click and drag the view to look at sections of the tool assembly.



Toolpath generation

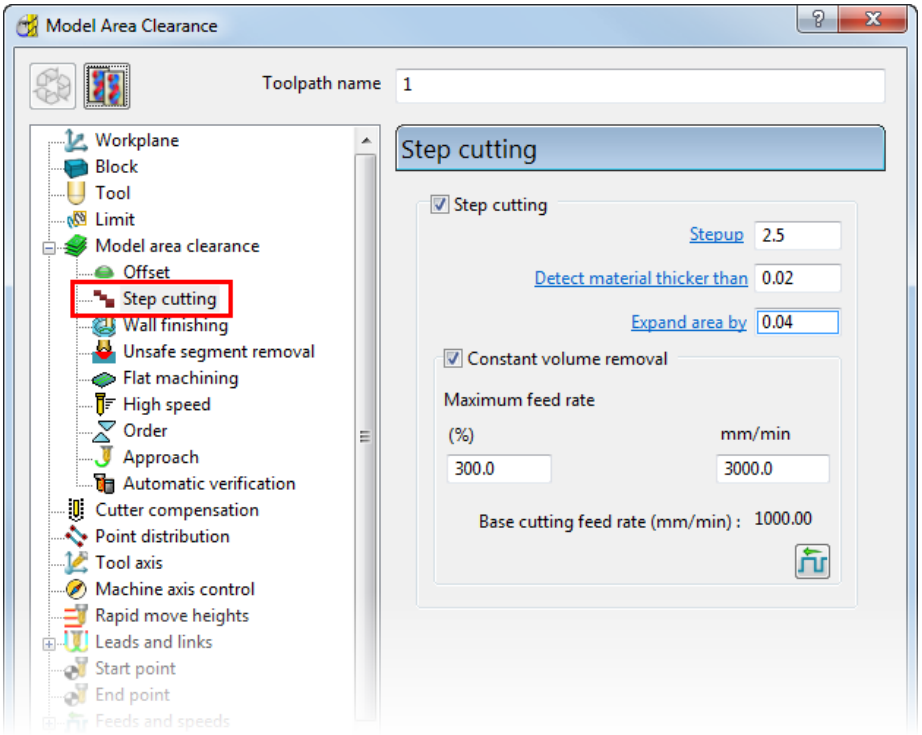
Area clearance step cutting

There is a new **Step Cutting** option which minimises terracing when creating area clearance toolpaths with a large stepdown. It rest machines the terraces remaining from a large area clearance toolpath with the same tool in the same toolpath.

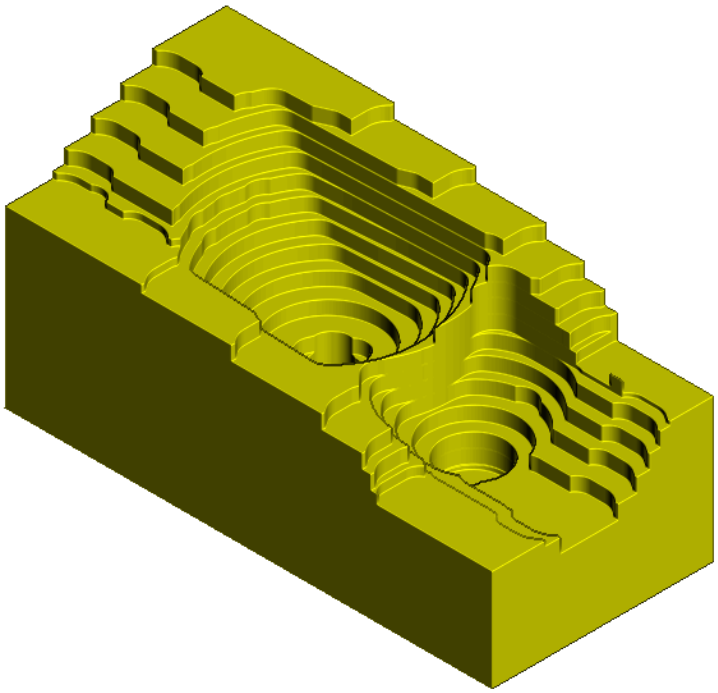
The difference between rest machining and step cutting are:

Rest machining	Step cutting
Normally uses a different (smaller) tool to the tool used in the area clearance toolpath.	Uses the same tool.
Creates a separate toolpath.	Creates one, combined toolpath.
Machines the terraces down the part.	Machines the main slices as normal (down the part) and machines the step cutting rest slices upwards from the last main slice.

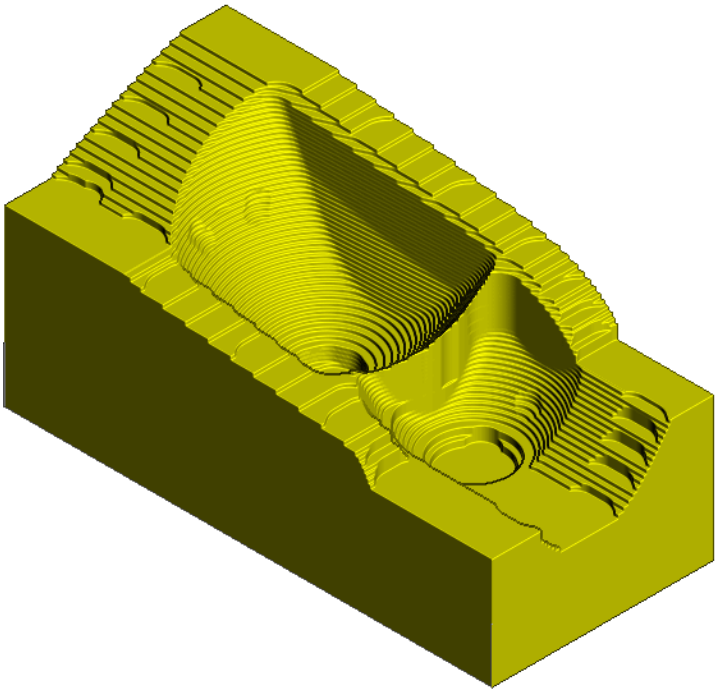
Step cutting is available for **Model Area Clearance**, **Model Profile**, **Model Rest Area Clearance**, and **Model Rest Profile** strategies.



Standard area clearance toolpath without rest machining or step cutting:



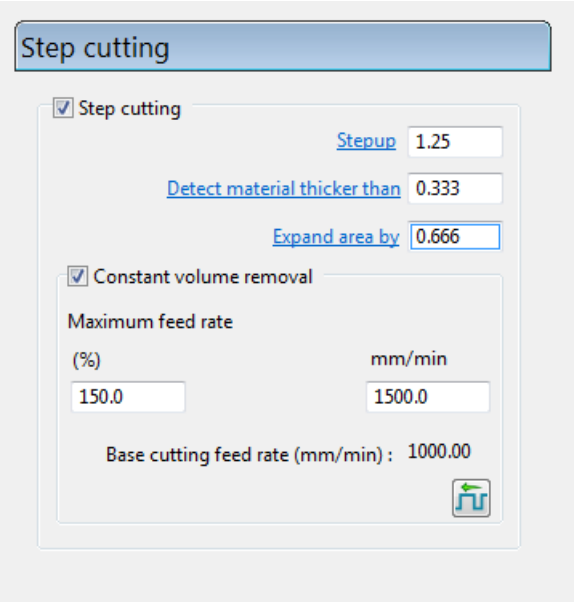
The same toolpath with step cutting:

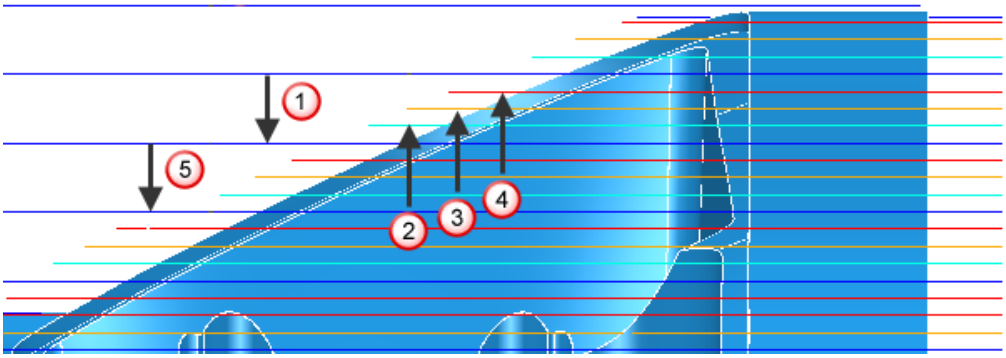


The main slices extend over the whole part, but the step-cutting slices extend only as far as they are needed to remove the terracing.

Step cutting (area clearance)

Step cutting creates intermediate toolpath slices to reduce terracing. It rest machines the terraces remaining from a large area clearance toolpath with the same tool in the same toolpath.





1 the first pass is a main slice.

2 the next pass is the lowest step-cutting slice.

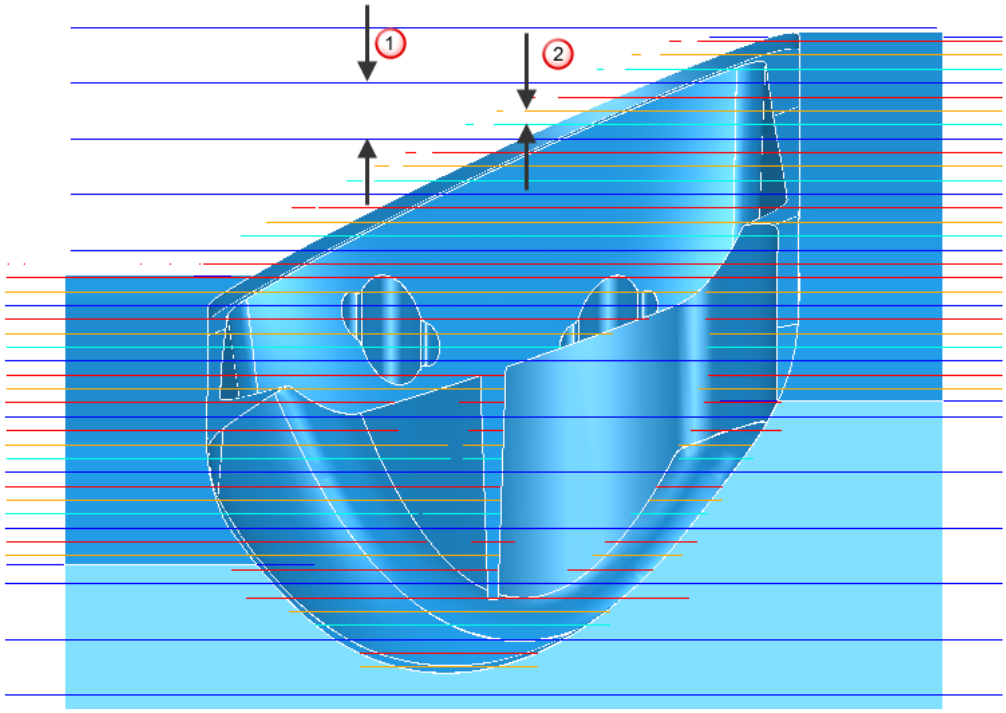
3 the next pass is the next to lowest step cutting slice and so on up the part (as shown by 4) until the previous main slice is reached.

5 the next main slice.

The main slices are cut from top to bottom as normal.

The intermediate slices are cut from bottom to top. This is possible because the depth of cut is always less than the depth of cut (or **Stepdown**) for the main slice. This is the most efficient way of cutting the part as it minimises the number of passes and maximises the depth of cut of each pass.

Stepup - the distance between the intermediate cutting levels.



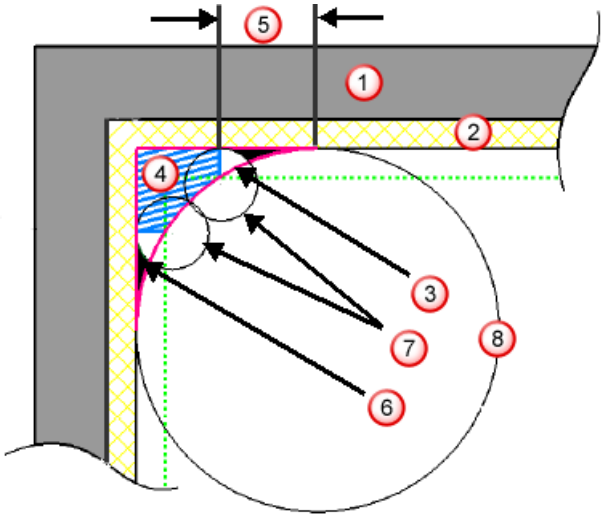
1 Stepdown of the main slice.

2 Stepup of the intermediate slice.

The main slice clears the majority of the material. The intermediate slices remove the terraces remaining after the main slice.

Detect material thicker than - PowerMILL ignores rest material thinner than the threshold specified here. This helps to avoid thin regions being rest machined, where the benefit of a second cut is negligible. These thin regions can be caused by cusps from the previous toolpath.

Expand area by - rest areas are expanded by this distance, measured along the surface. You can use it with **Detect material thicker than** to reduce the areas to be machined to the details (say, corners), and then to offset these areas slightly to ensure that all the detail (for example, on the corners) is machined.



1 - model

2 - thickness

3 - true rest material (outlined in pink)

4 - actual rest material detected (blue hatched area)

5 - amount you need to expand the area by to include all the rest material

6 - undetected material (black area)

7 - tool

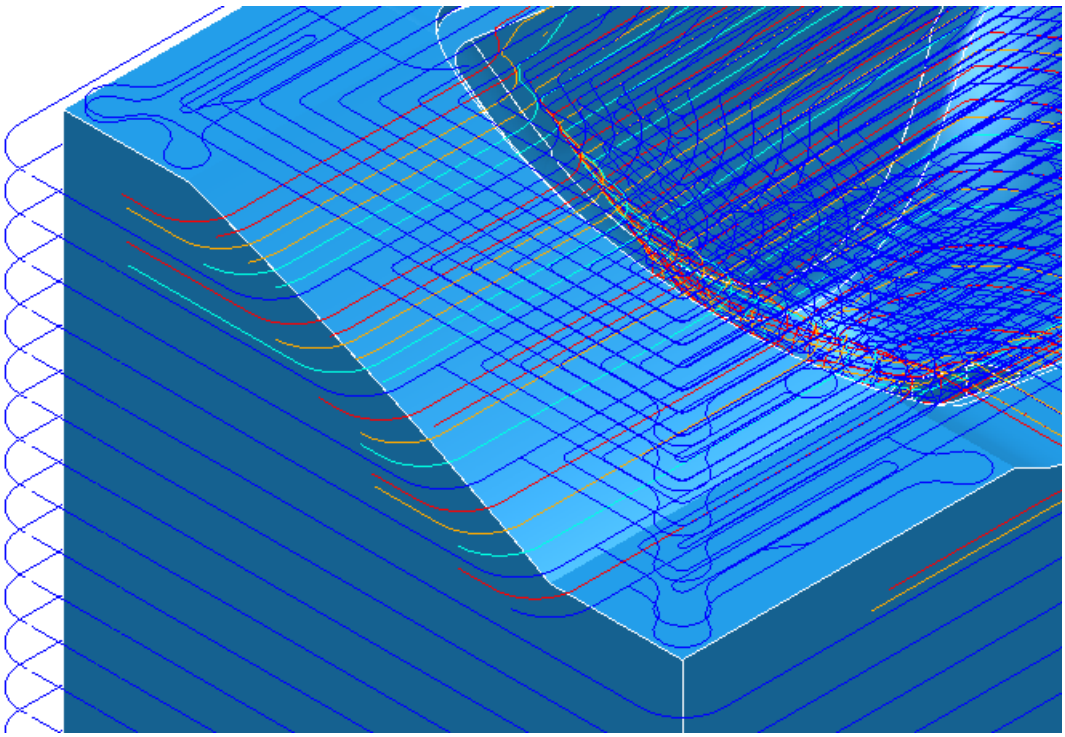
8 - reference tool

Use the **Expand area by** option to increase the rest area (the blue hatched area) and eliminate the undetected area (the black area).

Constant volume removal - increases the **Feed rate** of the intermediate slices. Because the stepdown of the intermediate slices is less than that of the main slice, you can increase the feed rate of the intermediate slices while maintaining the tool load. Each intermediate slice can have an increasing feed rate as the depth of cut gets progressively smaller.

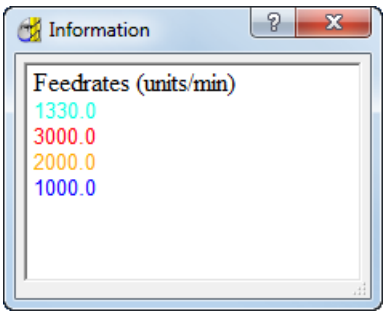
PowerMILL determines the increase in feed rate for intermediate slices based on the depth of cut. So if the maximum depth of cut for an intermediate slice is half that of a main slice, the feed rate can be twice as fast. To see the effect of the varying feed rate, select **Drawing Options > Draw Feeds** from the individual toolpath menu.

This example has a **Base cutting feed rate** of **1000** and a **Maximum feed rate** of **300%**.



The main slices are dark blue and are cut at 1000mm/min.

The first intermediate slice is cut at 1330 mm/min and the last intermediate slice is cut at 3000 mm/min.



Maximum feed rate % - the maximum allowable cutting feed rate for the intermediate slices as a percentage of the base cutting feed rate specified on the **Feeds and Speeds** dialog. This value must be larger than 100%. A value of **300** means the cutting feed rate of the intermediate slices can be up to three times faster than the base cutting feed rate.

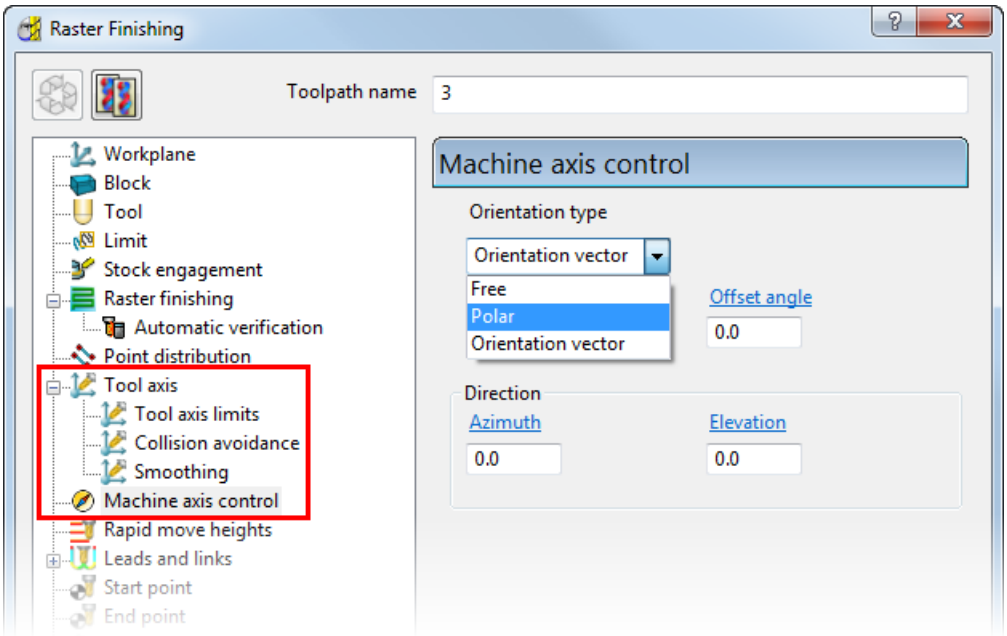
Maximum feed rate mm/min - the maximum allowable feed rate for the intermediate slices. (**Base cutting feed rate x %**).

Base cutting feed rate (mm/min) - the cutting feed rate as specified on the **Feeds and Speeds** dialog. To change this value click .

Feed rates - displays the **Feeds and Speeds** dialog.

Machine axis control

There is a new **Machine axis control** page on the strategy dialog.

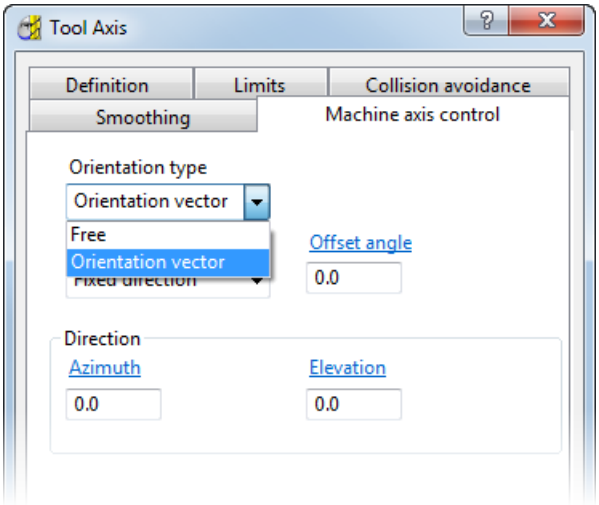


This displays the same options as those on the **Orientation vector** page in previous versions (a sub-page of **Tool axis**), but it also includes **Polar milling** (see page 19).

The **Tool axis** page no longer has an **Orientation vector** sub-page because these options are now available on the **Machine axis control** page.

The **Machine axis control** tab replaces the **Orientation vector** tab on the **Tool Axis** dialog. The tab offers the same functionality as in previous versions, because **Polar milling** is only available from the **Strategy** dialog and not from the **Tool axis** dialog.

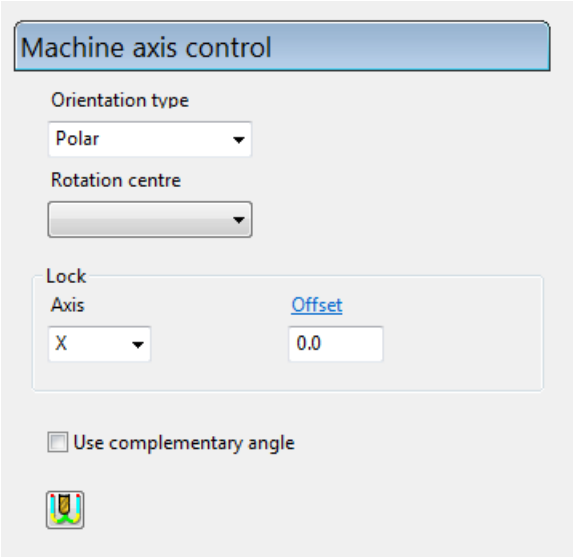
You can choose polar milling when creating and not when editing toolpaths.



Polar milling

In previous versions you could use only Cartesian coordinates to create your milling toolpaths because there was no means of specifying the table rotations to the postprocessor (the tooltip position and the tool axis are insufficient).Now you can also choose polar coordinates.

Polar is where the toolpath looks like a 3-axis toolpath (the tool axis is the Z axis), however, it uses rotations about the machine's singular axis rather than moving in X and Y.

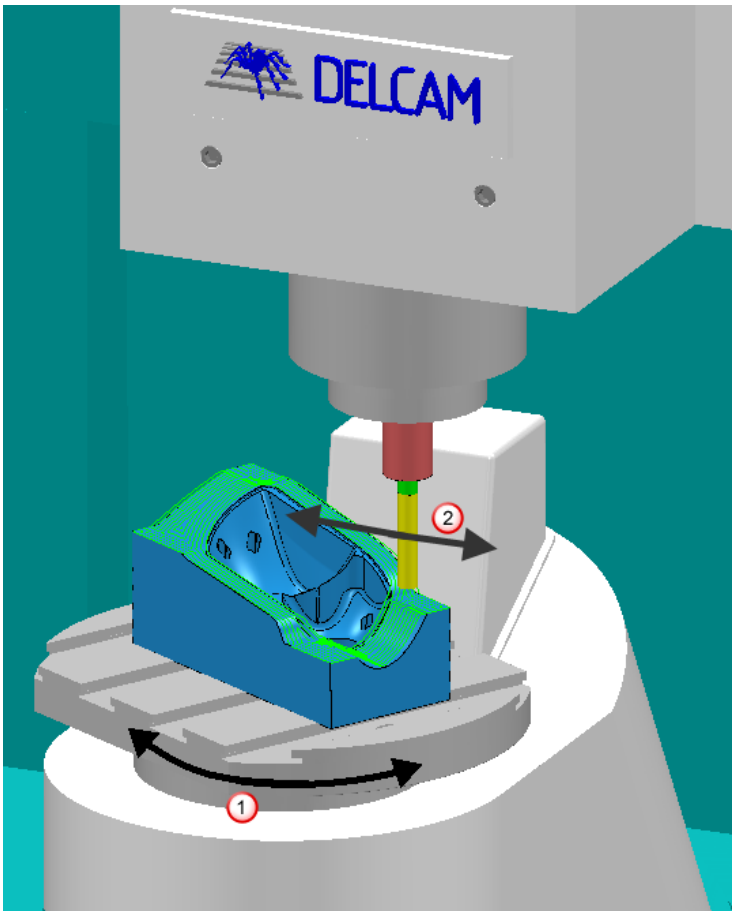


Polar milling is most useful when the machine movement in X and Y is quite limited, but the machine tool is a TABLE-TABLE machine, and is free to rotate the table about Z which overcomes the limited motion in X and Y.

Rotation centre - a workplane which defines the rotation centre for the polar milling. The Z axis must be aligned with the tool axis. Only workplanes that meet this criterion are displayed. If you don't select a workplane, PowerMILL uses the global coordinate system.

Lock - determines whether you lock the X or Y axis. One of them must be locked and is replaced with a rotary axis. Frequently, the X axis is locked and the rotary axis is the C axis.

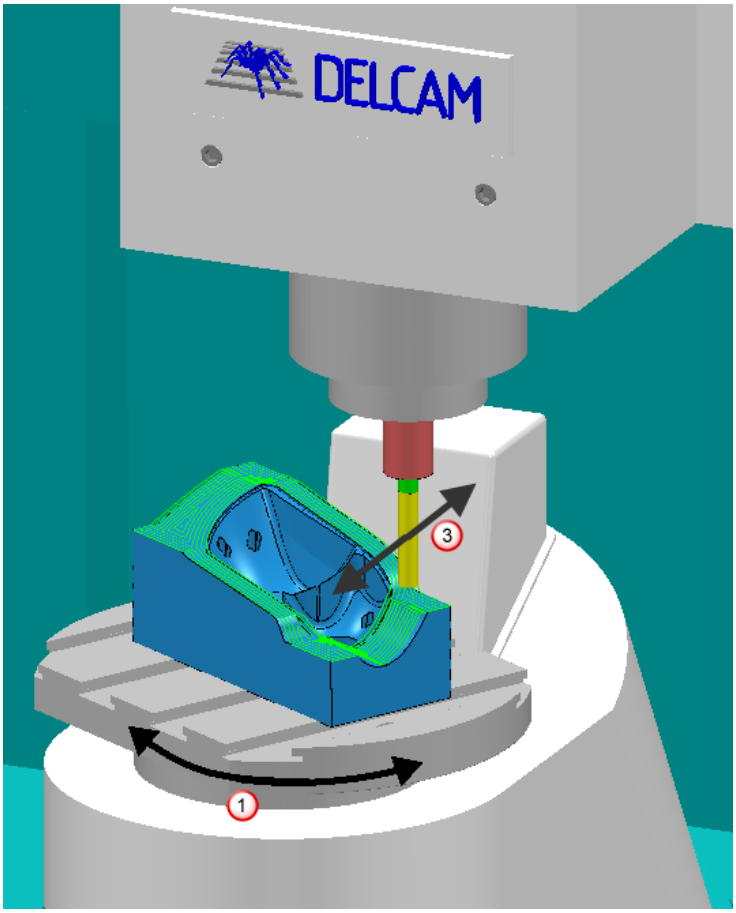
Locking the X axis allows the tool to move in Y and the table to rotate.



1 rotation about C.

2 linear movement in Y.

Locking the Y axis allows the tool to move in X and the table to rotate.

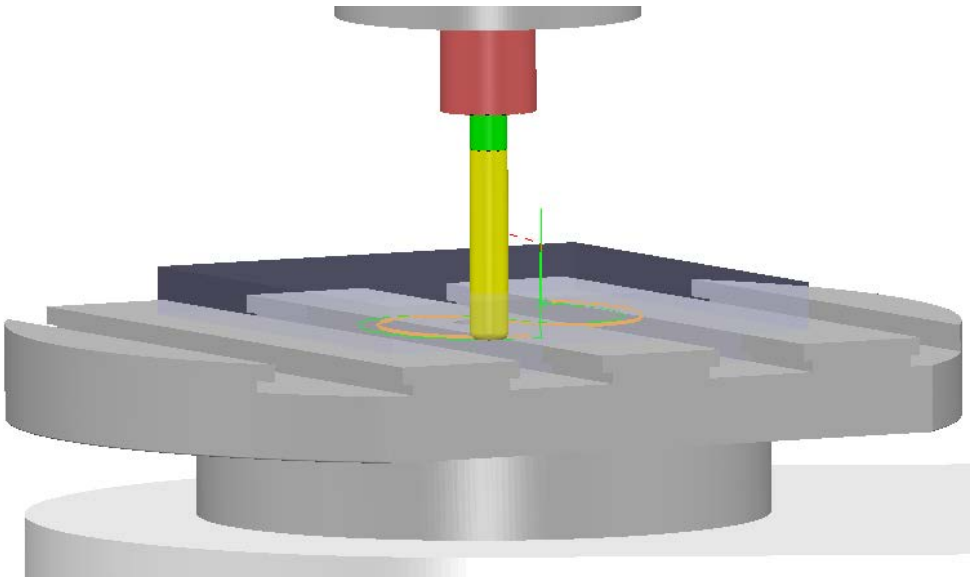


1 rotation about C.

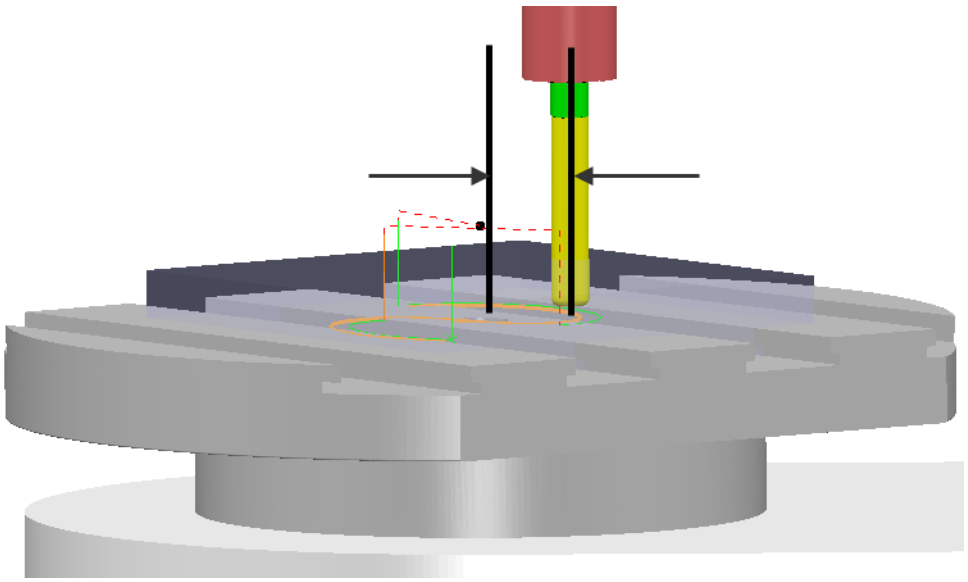
3 linear movement in X.

Offset - the offset from the workplane origin selected in **Rotation centre**. If the X axis is locked and no offset is applied, the tool is constrained to run along X 0. If an offset of 20 is applied, then the tool is constrained to run along X 20.

Offset of **0**.

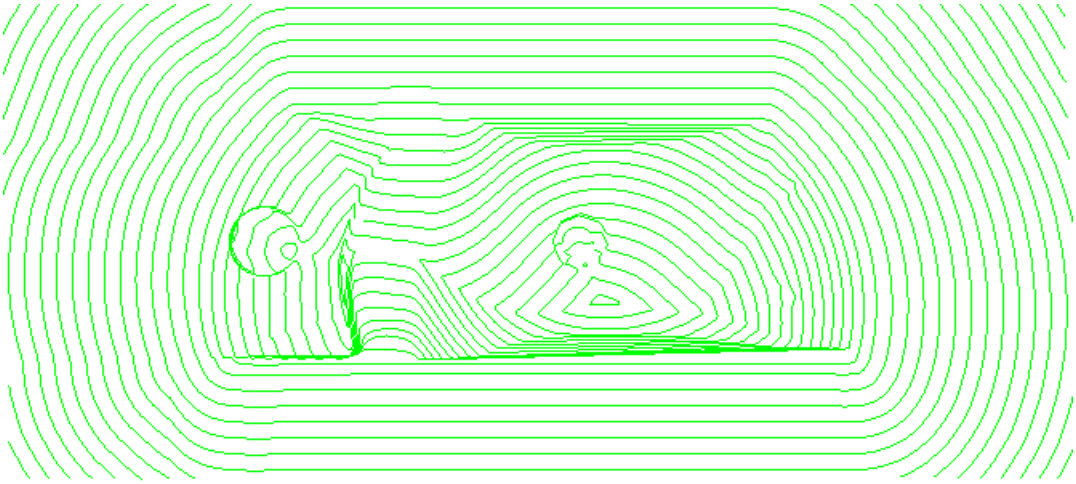


Offset of **50**.

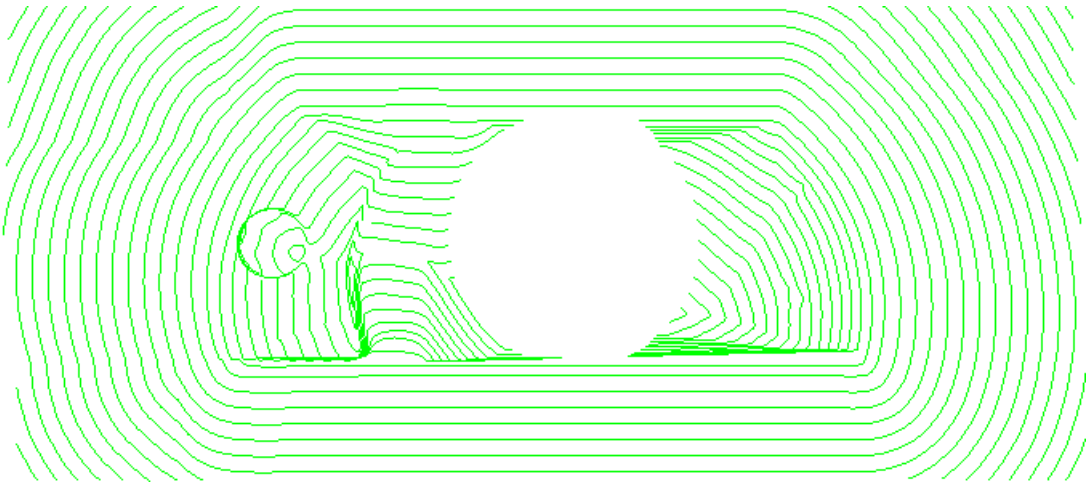


An offset gives a *null* area where no machining can take place because it is physically impossible for the tool to get inside this region. The *null* area is centred on the **Rotation centre** and has a diameter of the **Offset**. If you specify an offset, to completely machine the part, you must ensure your toolpath isn't inside the *null* area. If the toolpath does sit inside this area, a portion of the toolpath isn't created, but PowerMILL applies appropriate links.

Offset of **0**.



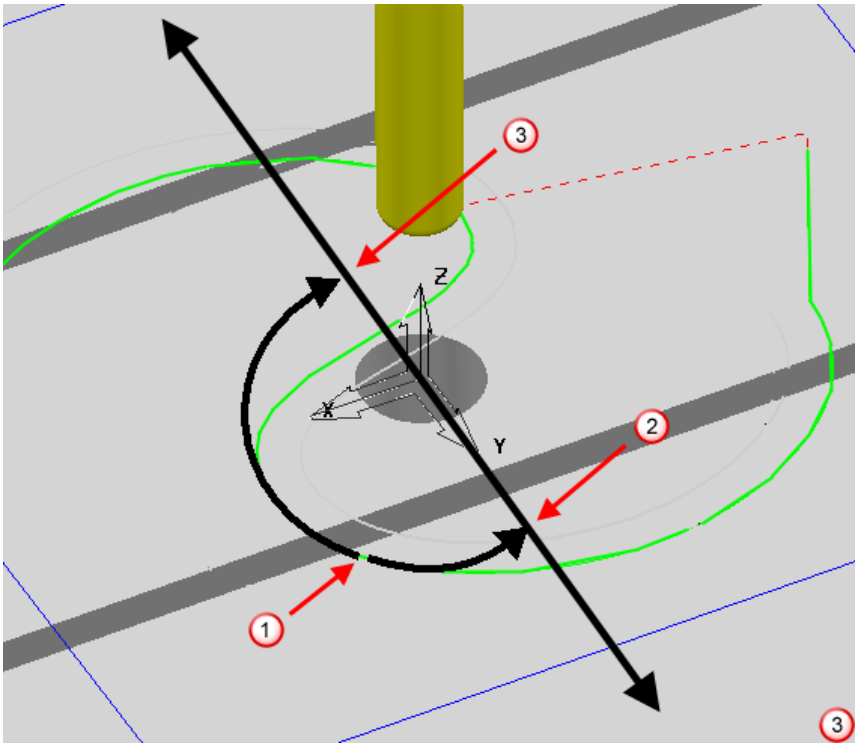
Offset of **20**.



*To avoid this problem, ensure the **Rotation centre** is at least the **Offset** distance away from the toolpath.*

Use complementary angle - determines which of the two polar milling solutions is used.

If the tool moves in the Y axis and the table rotates about the C axis, then point ① on the toolpath is machined by either rotating the table to point ② or ③. If ② is the first solution, ③ is the complementary solution.

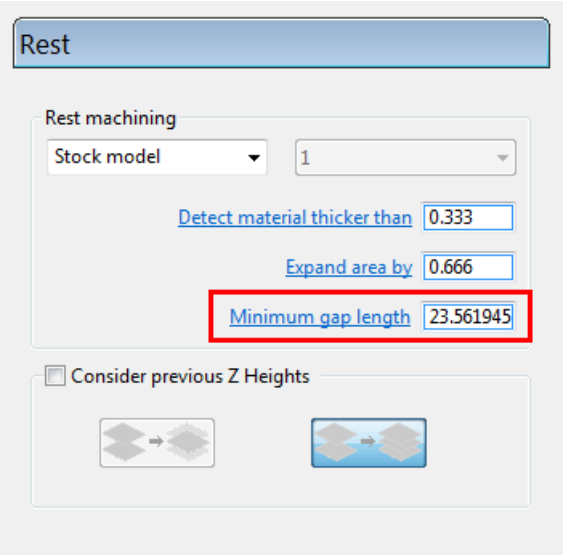


Links - displays the **Links** page of the strategy dialog, which enables you to edit the leads and links.

If you don't select **Use polar links** on the **Links** page, PowerMILL displays a warning message saying: **You have not chosen polar links**. To change them to polar links, click .

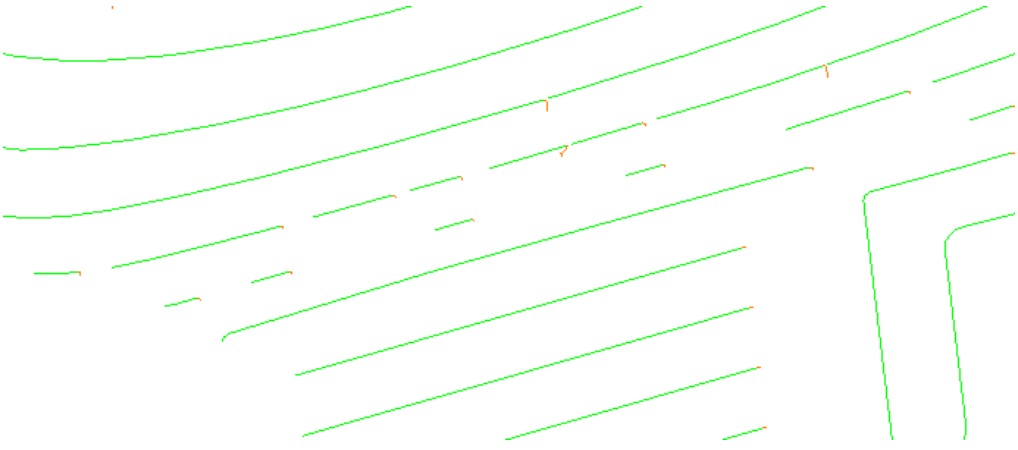
Rest roughing improvements

You can now control toolpath fragmentation (see page 11) when rest roughing from a stock model by specifying a **Minimum gap length**. This is available on the **Rest** page of any area clearance strategy.

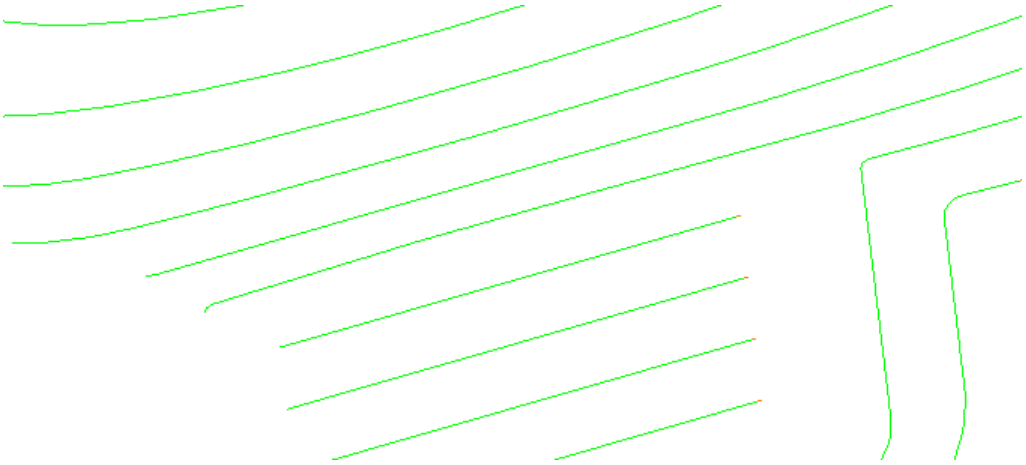


Minimum gap length controls fragmentation by replacing gaps shorter than this distance with a toolpath segment. A large value reduces fragmentation, but increases the length of the toolpath that isn't actually cutting material. A small value produces shorter toolpaths but increases the number of toolpath lifts.

A **Minimum gap length** of **0:**



A **Minimum gap length** of the default value:



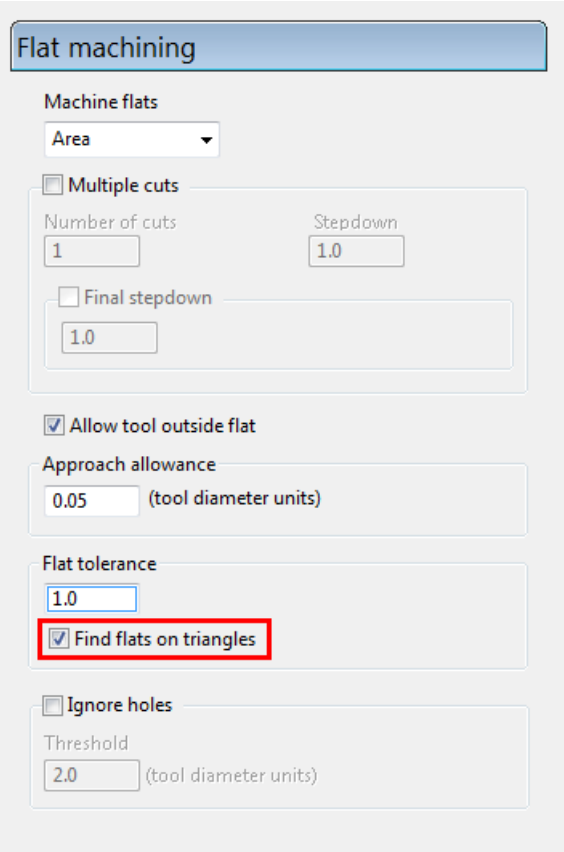
*You must have a **Rest machining reference type** of **Stock model** to enable this option.*

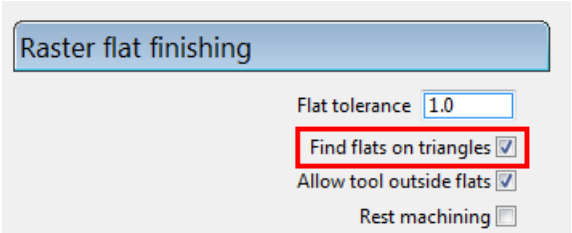
In previous versions, you had no control over this value:

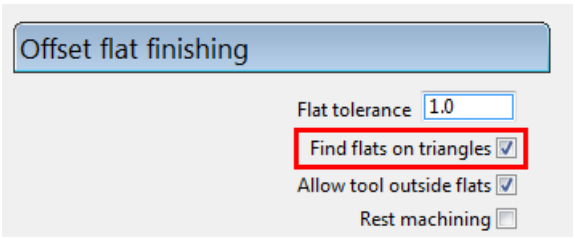
- In PowerMILL 2012, the gap length was the current default value.
- In PowerMILL 2011, the gap length was zero.

Flat machining improvements

There is an additional option of **Find flats on triangles** on the **Flat machining** page of **3D Model area clearance** strategies and on the **Offset flat** and **Raster flat finishing** strategies to specify whether to consider parts of the model containing triangles. In PowerMILL 2012, flat machining considered areas of the model which contained triangles. Now, you can choose to ignore them.



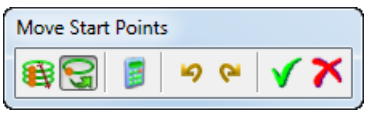




Find flats on triangles, when selected, finds flat areas on models containing triangles and surfaces. When deselected, PowerMILL doesn't consider parts of the model containing triangles.

Enhancements to move toolpath start points

You can now make multiple changes to toolpath start points simultaneously using the **Batch** option on the **Move Start Points** toolbar.



This toolbar is available from the **Move Start Points** button on the **Toolpath** toolbar.

Batch - batches toolpath start points edits and applies them when you reselect this button. You can batch edits in either or mode, but not in a mixture of both.

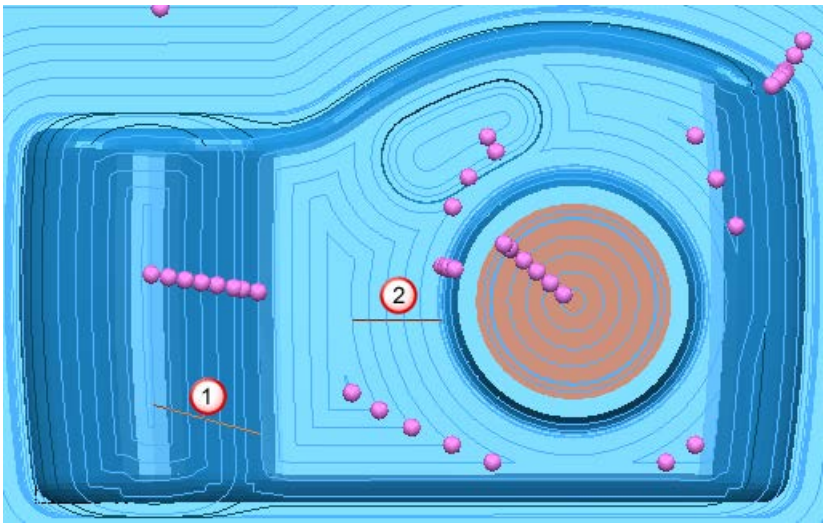
For example:

1 Click the **Batch** button.

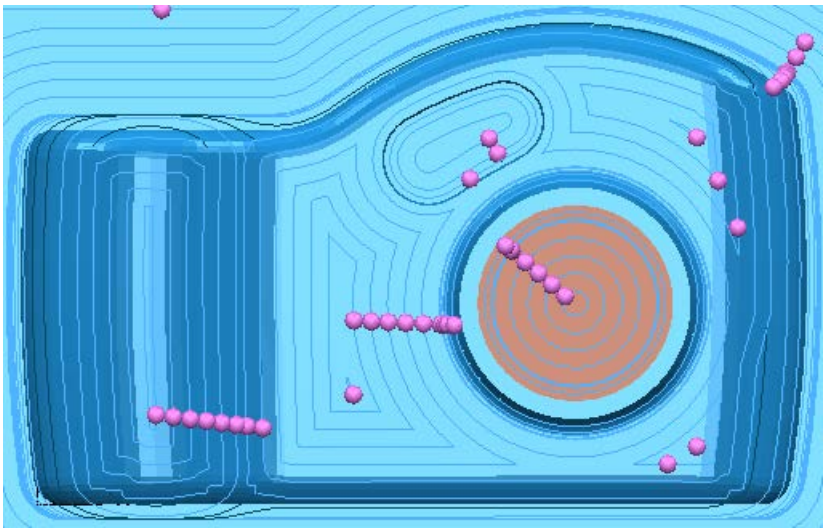
2 Click the **Move toolpath start points by Line** button.

3 Sketch a line 1.

4 Sketch another line 2.



5 Click the **Batch** button.



This moves the edited toolpath start points.

*The **Undo** button undoes one operation at a time. So you must click twice to revert to the original.*

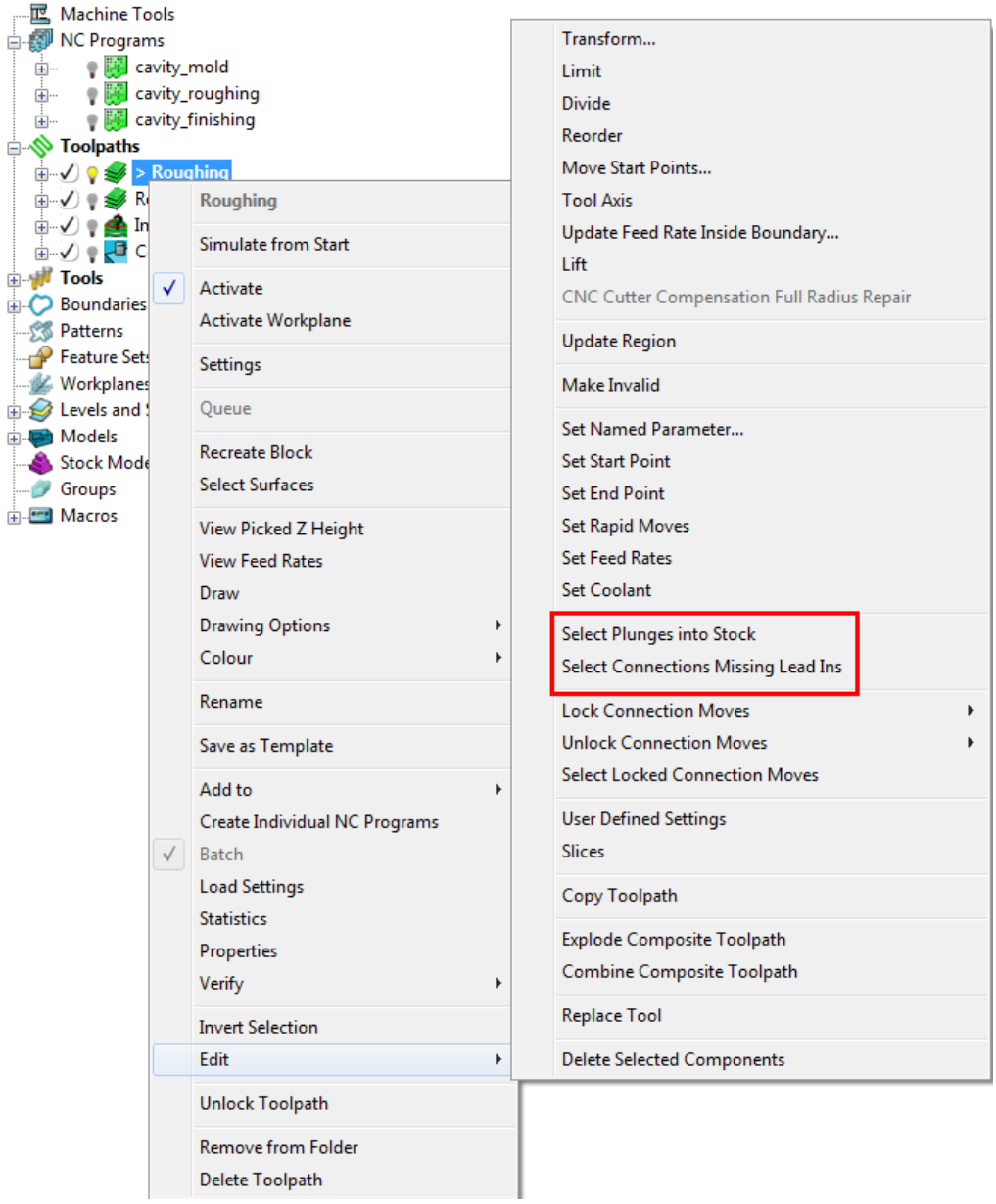
Clicking , discards all the batched edits.

Warning about plunge moves into stock

If you calculate a toolpath with a plunge move into stock material, PowerMILL displays a warning message. This reminds you to add appropriate lead in moves. In previous versions, this warning was displayed when editing a toolpath, but not on initial creation.

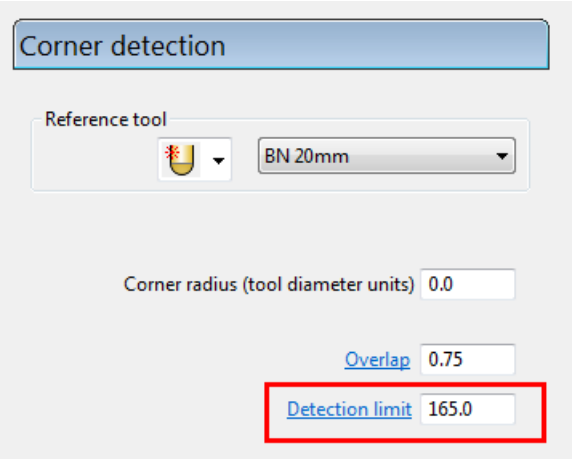
The individual toolpath **Edit** menu now enables you to check whether a toolpath has:

- any plunge moves into stock
- missing lead ins on any connection moves



Corner clearance enhancements

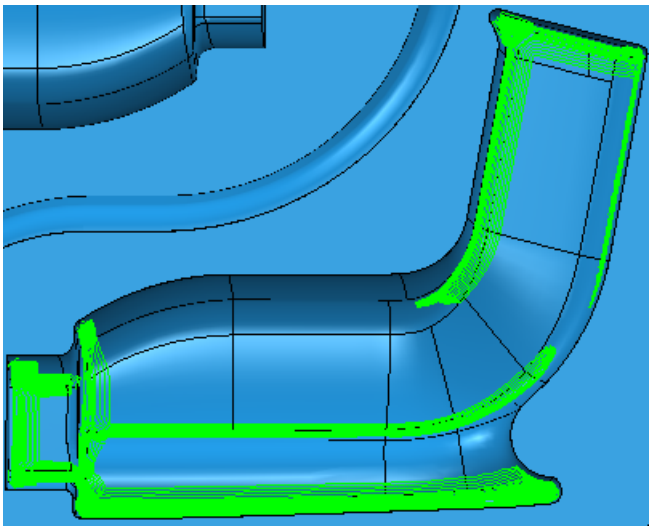
Corner clearance now enables you to specify the angle which is considered as a corner. This is available on the **Corner detection** page of the **Corner Clearance** strategy dialog.



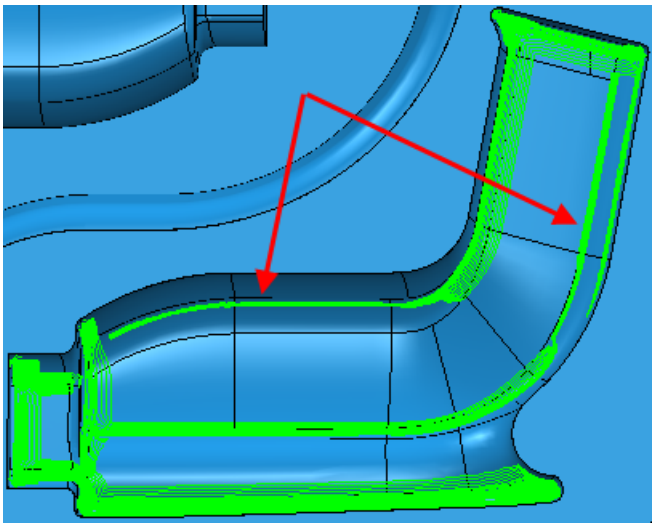
Detection limit - machines only corners sharper than this angle.

165° is the default value. This works well for most corners. However there are instances where you need to increase this value to detect the corner regions in which you are interested.

Detection limit - 165°



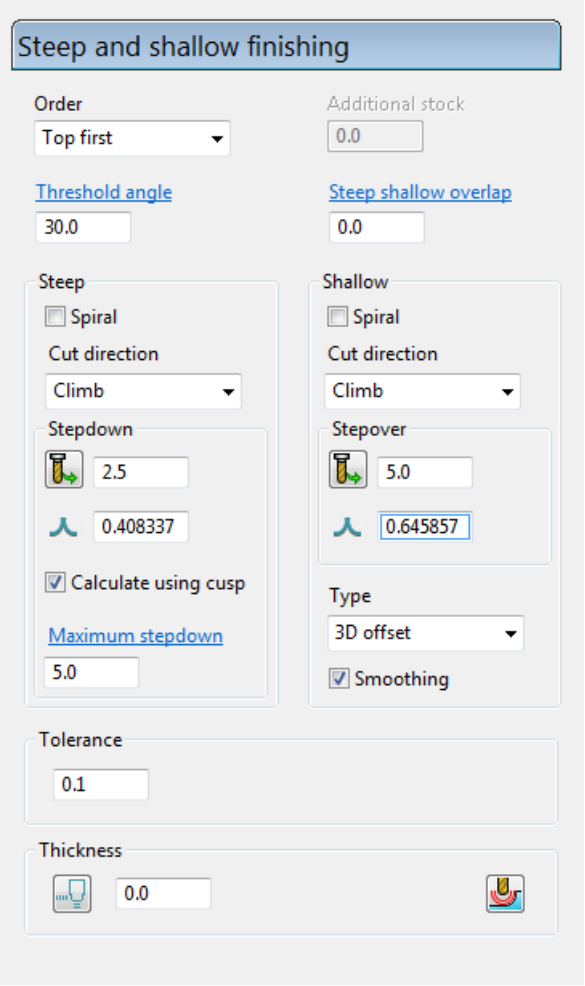
Detection limit - 170°



Steep and shallow finishing enhancements

Steep and shallow finishing now has shallow options on the main page,instead of on a separate page. This new main page also enables you to have a different **Stepdown** for machining steep portions to the **Stepover** used for machining shallow regions.

You can now specify the **Stepdown** using the new **Calculate using cusp** option. When selected, the **Cusp height** determines the stepdown, with a maximum stepdown determined by the **Maximum stepdown** value, which prevents an excessive stepdown on vertical walls. When deselected, the stepdown between successive Z heights has a constant value equal to the **Minimum Stepdown**.



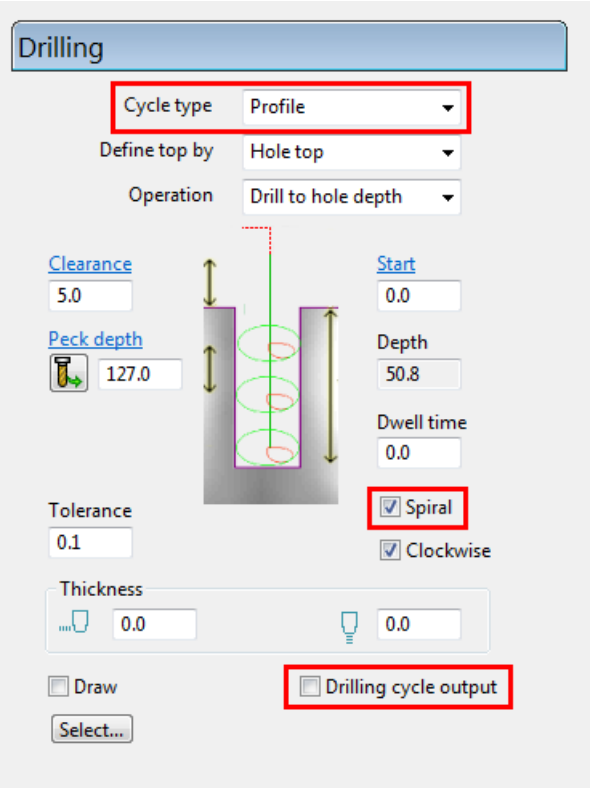
The other options on the dialog remain unchanged, but are in a different location.

Spiral profile drilling

You can now create spiral profile drilling toolpaths. This option is available from the **Drilling** strategy dialog when you:

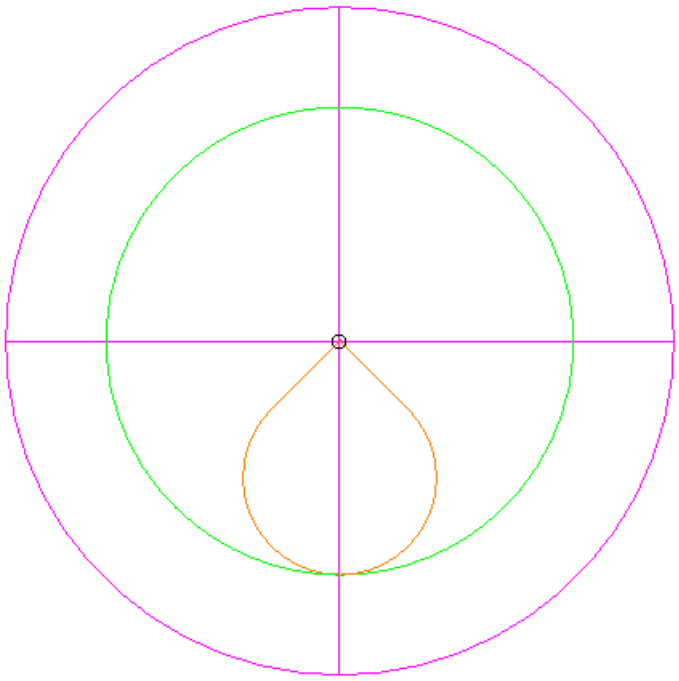
1. Select a **Cycle type** of **Profile**.
2. Deselect **Drilling cycle output**.

3 Select **Spiral**.

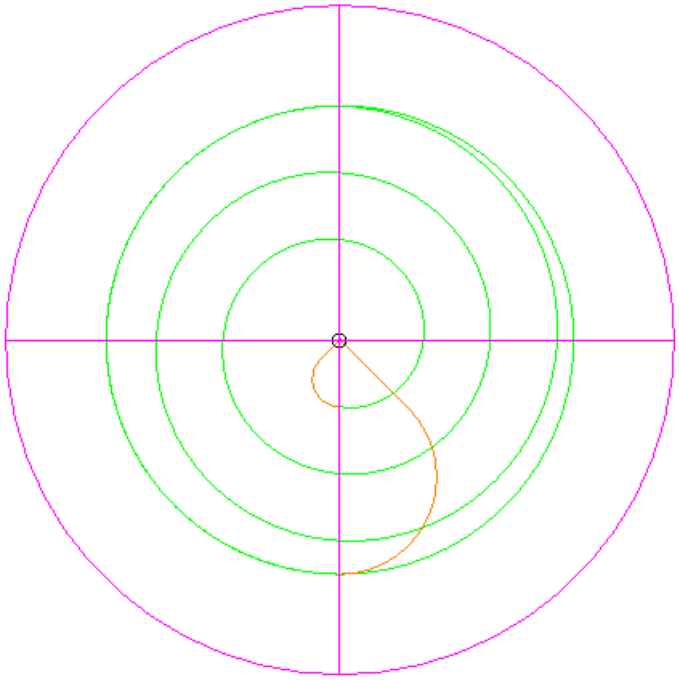


Spiral creates a spiral toolpath which moves from the centre of each hole with a "tear drop" lead and spirals out using the stepover defined on the **Profile** page.

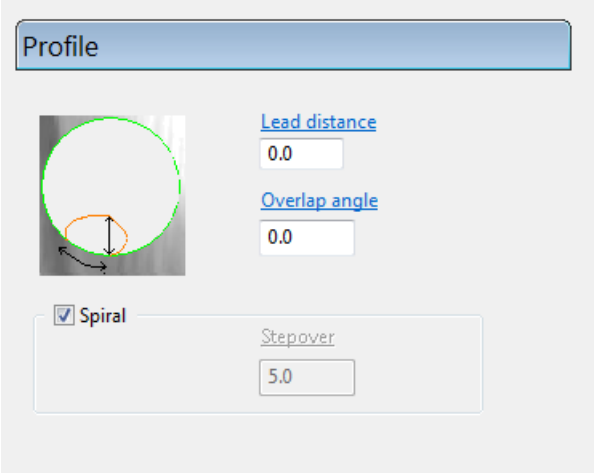
Profile drilling toolpath with **Spiral** deselected:



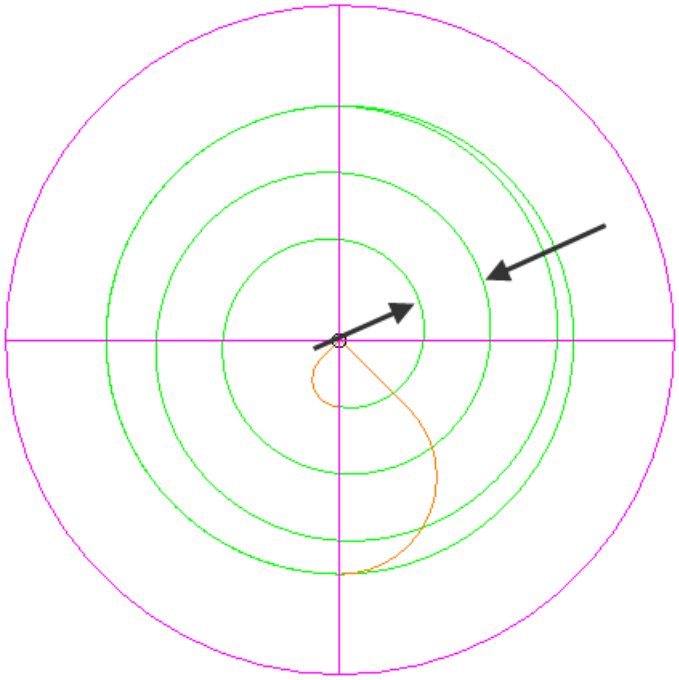
Profile drilling toolpath with **Spiral** selected:



If you select **Spiral** on the main page, then the **Profile** page displays the spiral **Stepover** rather than the **Final profile pass** options.



Spiral stepover is the distance between adjacent passes:



Direction for Parametric spiral finishing

The milling strategy used for parametric spiral toolpaths more accurately reflects the machining style. In PowerMILL 2012 R2 the **Direction** field has two options:

Clockwise - produces a conventional milling toolpath when machining a pocket and a climb milling toolpath when machining a boss.

Anticlockwise - produces a climb milling toolpath when machining a pocket and a conventional milling toolpath when machining a boss.

In PowerMILL 2012, the **Cut direction** field had the two options, **Climb** and **Conventional**. These options produced the expected results when machining a pocket, but not when machining a boss.

Drilling thickness

The thickness specified on the main **Drilling** strategy page is now different to the thickness specified on the **Automatic verification** page of the drilling strategy.

- The thickness specified on the main **Drilling** page is used for toolpath generation. This is the thickness used for toolpath generation and enables you to drill a hole deeper than the hole feature by creating a drilling toolpath with a negative axial thickness. This is useful for through holes.
- The thickness specified on the **Automatic verification** page is used for collision checking. This ensures there is no shank or holder collisions at the top of the hole by collision checking the toolpath to a positive thicknesses, specified on the **Automatic Verification** page.

If you enter a different toolpath thickness to verification thickness PowerMILL displays a warning message:

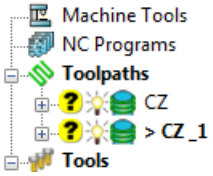
The verification thickness is different from the drilling thickness.

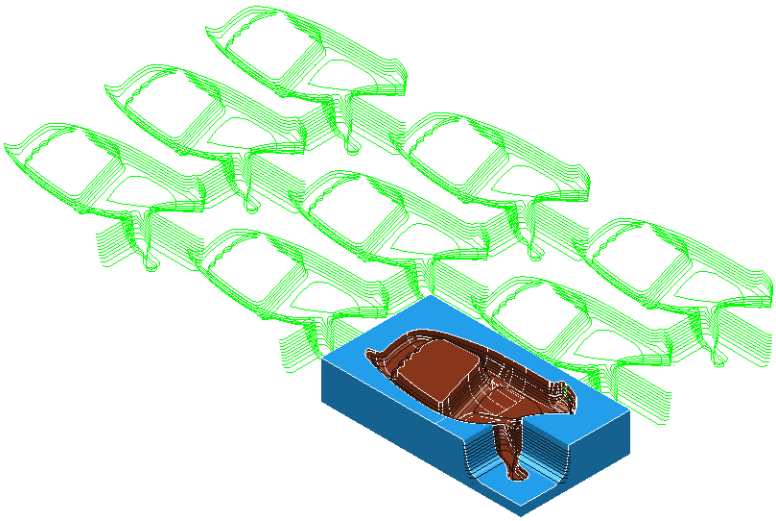
Toolpath transform enhancements

You can now choose whether to append all transformed toolpaths into one composite toolpath or keep them as individual toolpaths. In previous versions they were created as individual toolpaths. This new **Toolpath append** option is available on the **Toolpath Transform** toolbar.

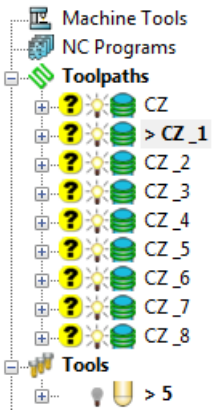
For example, creating a multiple transform on a constant Z toolpath called **CZ**.

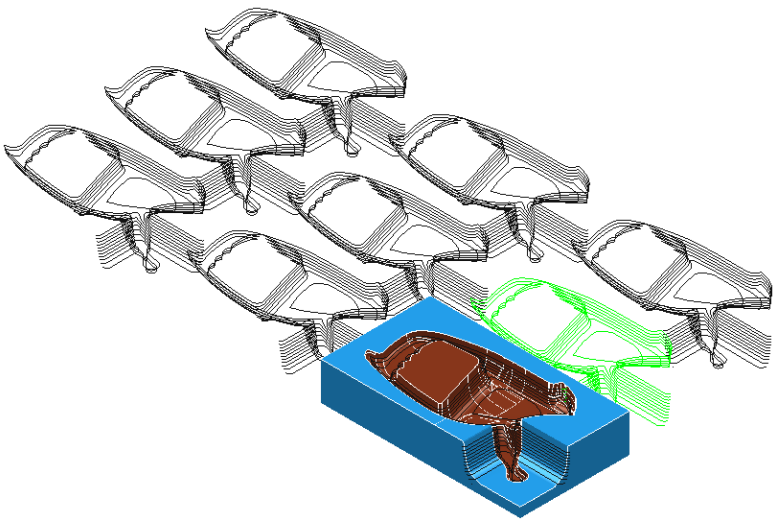
Selecting creates one new toolpath:





Deselecting creates eight new toolpaths:



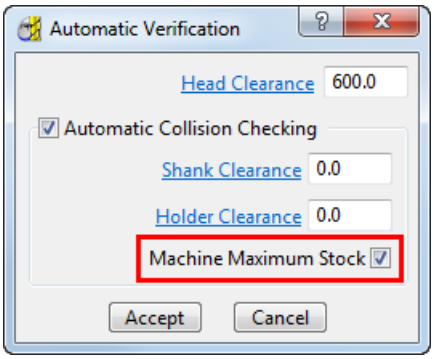


Toolpath verification

Machine maximum stock

When using automatic verification on a toolpath with a shank or holder collision, PowerMILL can now create an improved toolpath which removes as much material as possible and avoid collisions.

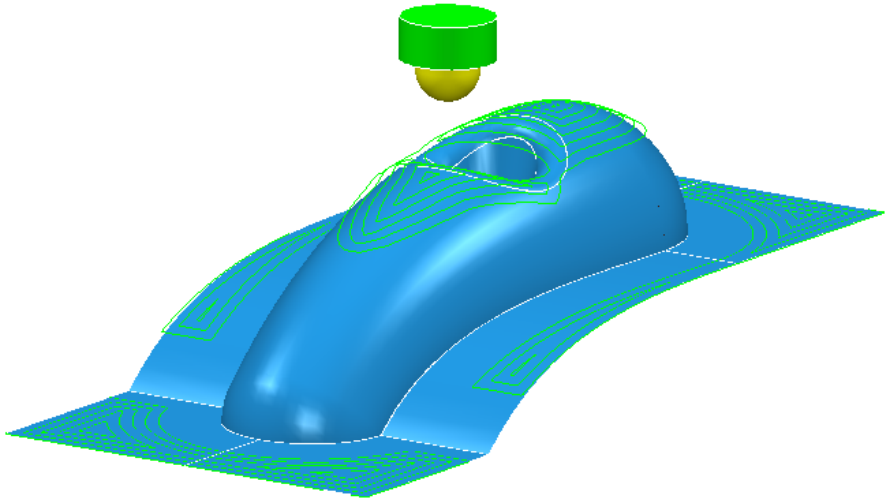
This option is available on the **Automatic Verification** dialog.



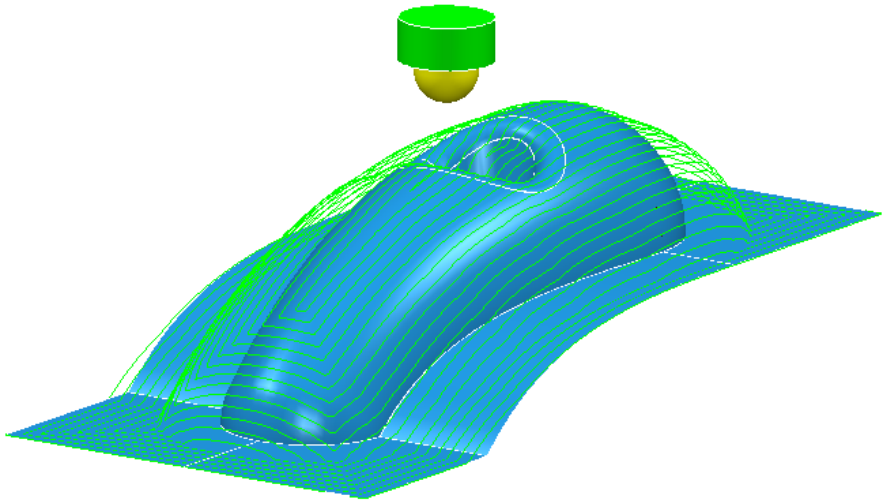
When a shank or holder collision occurs, **Machine maximum stock** adjusts the tool tip position to avoid collisions whilst removing as much material as possible.

These examples use the **cowling.dgk** model available in the examples folder and show a 3D offset toolpath and a tool with a wide shank.

Machine maximum stock deselected, PowerMILL removes the toolpath segments where the shank or holder collides with the part.

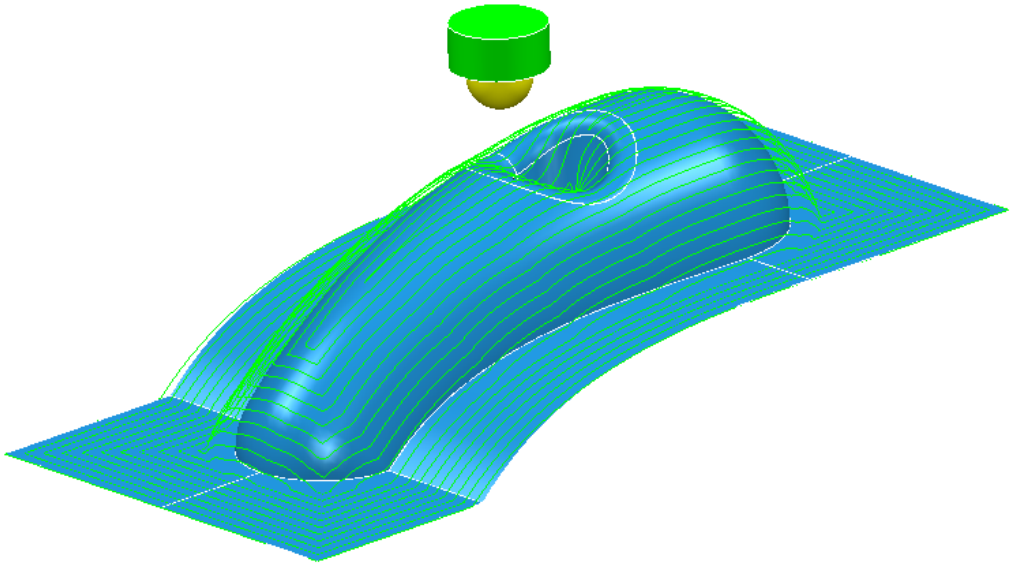


Machine maximum stock selected, PowerMILL moves the toolpath segments away from the part to avoid the shank or holder collision.

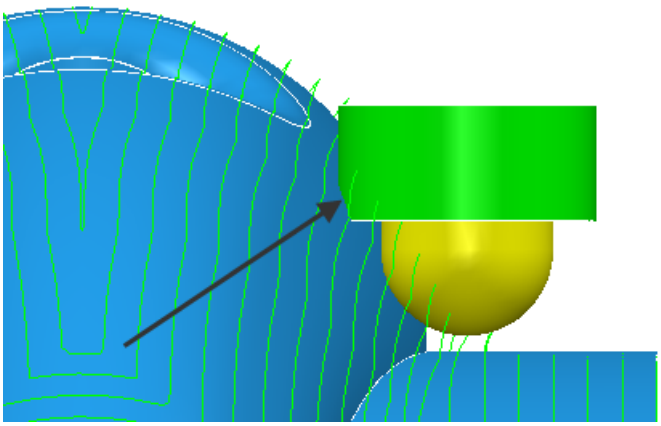


This leaves stock on the part, but minimises the amount.

If you deselect **Automatic Collision Checking**, the tool shank isn't collision checked.



In this case, the shank collides with the part.



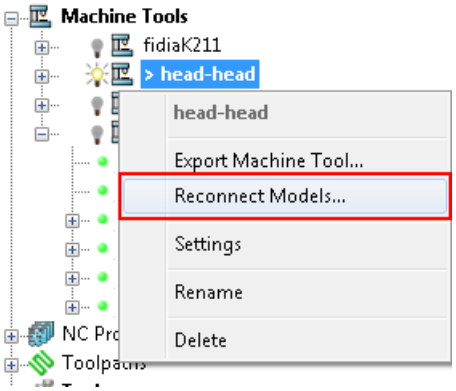
When using the **Machine maximum stock** option:

- The cutter doesn't need to touch the part, it is as close as possible to the part, and the holder and shank are collision-safe.
- When applied to a constant Z toolpath, the toolpath is moved so it remains a constant Z toolpath.
- When applied to a raster toolpath, the toolpath is moved up Z so the segment remains a typical raster segment.

Toolpath output

Reconnecting machine tools

The **Reconnect Models** option, available from the individual machine tool context menu, enables you to reconnect **.mtd** files to the machine tool.

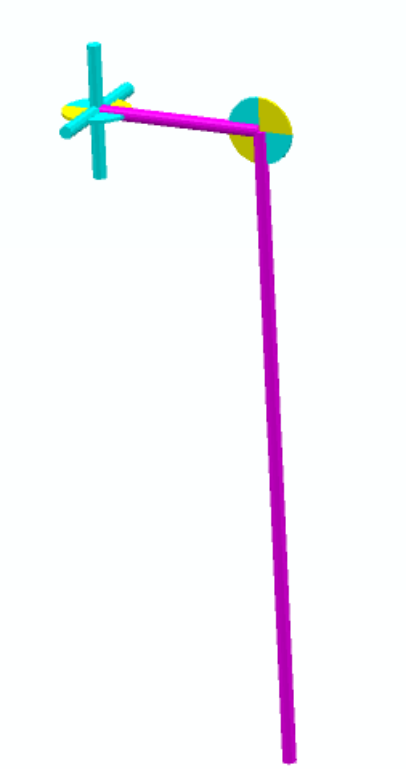


To minimise the size of a PowerMILL project and make it easier to transfer, the machine tool data is stored in a project, but the machine tool model files are not.

If the project is moved to another computer or the machine tool model files are moved to a different location, the **Reconnect Models** option allows you to select and reconnect the machine tool model files.

If a machine tool is disconnected, the tool axes are displayed as lines. This is different for each machine tool.

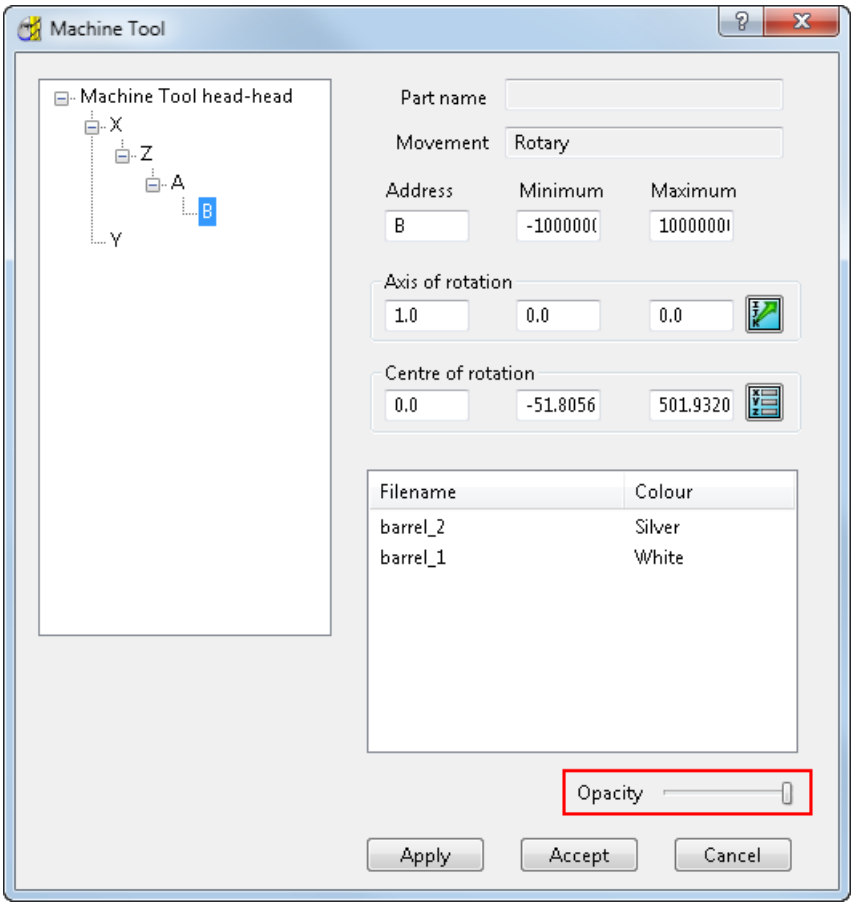
For example, when the machine tool **Head-Head** (available in the PowerMILL Examples/MachineData folder) is disconnected from its model files, it is displayed as:



Reselect the **.mtd** file to display the complete machine tool.

Machine tool opacity

You can now control the translucency of a machine tool or an individual machine part using the **Opacity** control on a machine tool dialog. This enables you to view machine tool components in more detail.



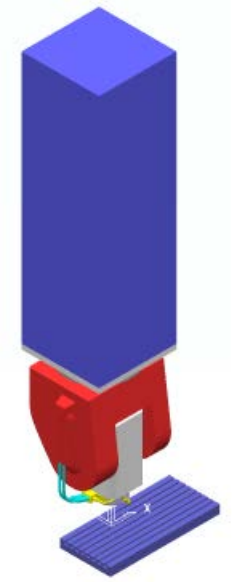
Select a machine or machine part and move the slider to the left to increase the translucency; move the slider to the right to decrease the translucency (see page 43).

Controlling the opacity of a machine tool and part

This example shows you how to change the opacity of a machine tool.

1 From the **Machine Tools** context menu, select **Import Machine Tool**.

2 Browse to the PowerMILL examples folder, select and import **fidiaK211.mtd**.

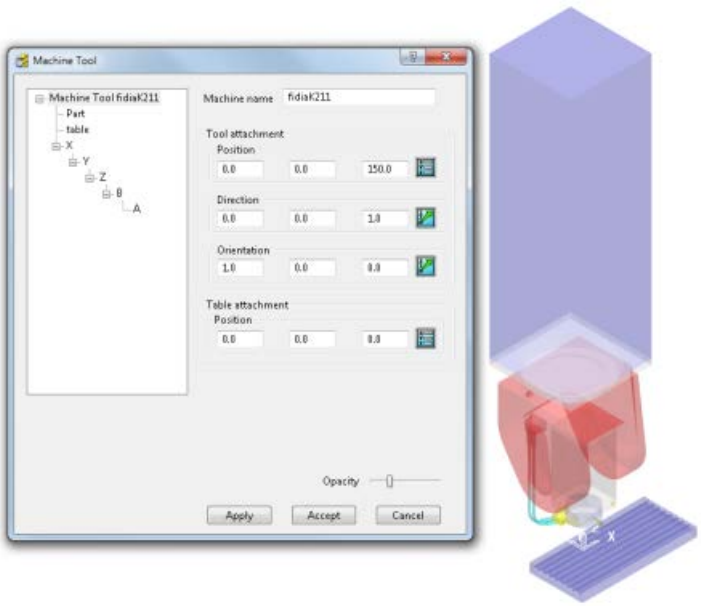


3 From the **fidiaK211** context menu, select **Settings**. This displays the **Machine Tool** settings dialog.

4 To increase the translucency of the complete machine tool:

a In the **Machine Tool** settings dialog, select **Machine Tool fidiaK211**.

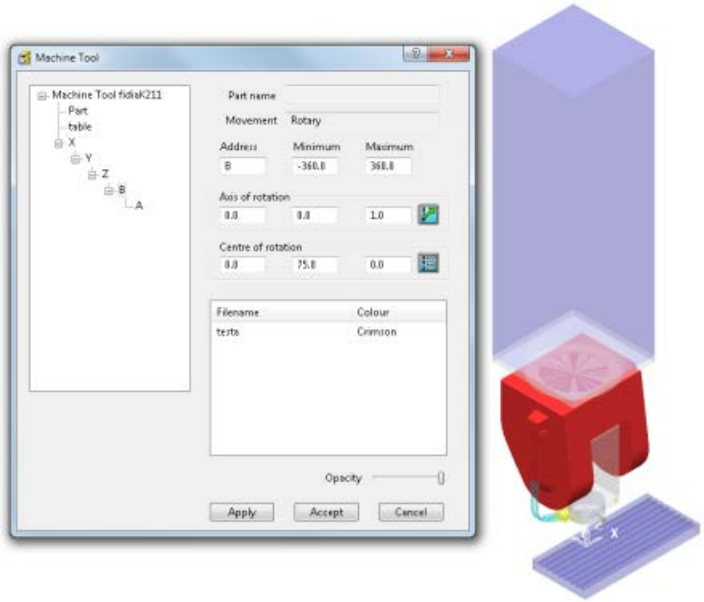
b Move the **Opacity** slider to the left.



5 To decrease the translucency of an individual machine part.

a In the **Machine Tool** settings dialog, select the individual machine part, in this example, machine part **B**.

b Move the **Opacity** slider to the right.



Machine tool simulation enhancements

Robot simulation

Improvements to the kinematic solver make robot simulation more robust and up to two hundred times faster. This enables much better toolpath simulation when also displaying specific (robot) machine tools.

Simulation mode

When simulating toolpaths, you can now choose between **Feed rate simulation** or **Point-to-point simulation** modes using the simulation mode button on the **Simulation** toolbar. In previous versions, **Point-to-point simulation** was the only option.

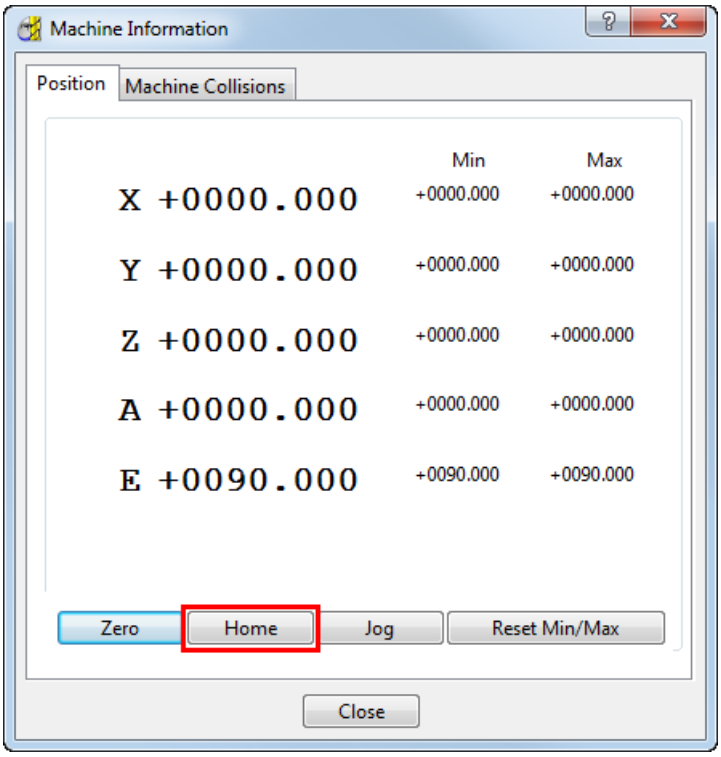
*The mode button displayed on the **Simulation toolbar** is either or depending on the previously selected simulation mode.*

Feed rate simulation - toolpath is simulated based on feed rates. The simulation speed, managed using the speed control, is a multiple of the feed rate.

Point-to-point simulation - toolpath is simulated from point-to-point. The simulation speed, managed using the speed control, controls the number of toolpath points covered in a second.

Home position

There is an additional button of **Home** on the **Position** tab of the **Machine Information** dialog (available from on the **Machine Tool** toolbar). This moves the machine tool to its home position.



User interface

Stock Models toolbar

The **Stock Models** toolbar enables you to create and edit **Stock Models**. **Stock Models** are an alternative method to using reference toolpaths when rest machining.

To display the **Stock Models** toolbar, either:

- From the **Stock Models** context menu, select **Toolbar**; or
- Select **View > Toolbar > Stock Models** from the menu.

Create a Stock Model - creates a new empty stock model and displays the **Stock Model** settings dialog where you can set up parameters for the stock model.

- selects a stock model and activates it. This list displays the available **Stock Models** in a project.

- selects a stock model state in the active stock model and activates it. This list displays the available stock model states in the selected stock model.

Stock Model Settings - displays the **Stock Model** settings dialog where you can edit the currently active stock model parameters.

Queue Stock Model - adds the selected **Stock Model** to the calculation queue.

Calculate Stock Model - calculates the selected **Stock Model**.

Copy Stock Model - creates a copy of the **Stock Model**. This creates a new stock model with a single block state.

Import - displays the **Import Stock Model** dialog which allows you to import **Stock Models**.

Use the dialog to browse and select ***.stkmd** files (**Stock Model** files). This loads a new **Stock Model** entity with a single block state.

Export - exports the stock model. Select this option to display the **Save Stock Model** dialog. Enter a **Stock Model** name, browse to the save directory and click **Save**.

This saves the current active state of a stock model into a new stock model as a single state.

Apply Block - creates a stock model of the block. All block types are supported (arbitrary, boundary, or rectangular).

Apply Active Toolpath - applies the active toolpath after the active stock model state. If no state is active, the toolpath is added after the last stock model state and is made the active state.

Apply Active Tool - applies the active tool after the active stock model state. If no state is active, the tool is added after the last stock model state and is made the active state.

Remove Active Stock Model State - removes the active stock model state from the stock model. This invalidates all subsequent stock model states.

Draw Stock Model - draws the stock model as a wireframe or shaded stock model.

Draw Wireframe - draws the stock model as a wireframe.

Draw Shaded - draws a shaded stock model.

Show Material - displays the rest, removed, and all stock material.

Show All Material - displays all stock material.

Show Rest Material - displays material left over after the stock model state operation.

Show Removed Material - displays material removed after the stock model state operation.

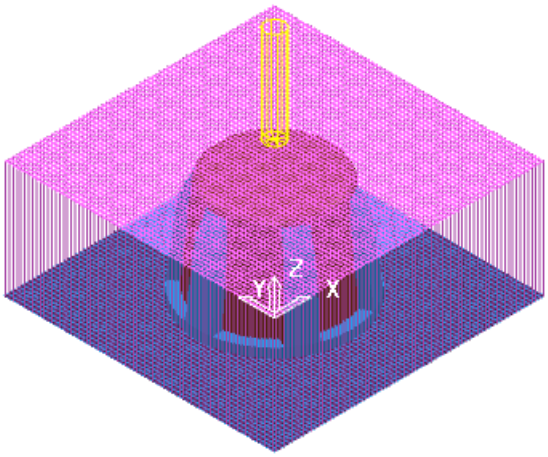
Delete Stock Model - deletes the active stock model.

Using the Stock Model Toolbar

This example shows how to create and manipulate stock models using the **Stock Model** toolbar. This uses the **radknob.dmt** model available in the PowerMILL examples folder.

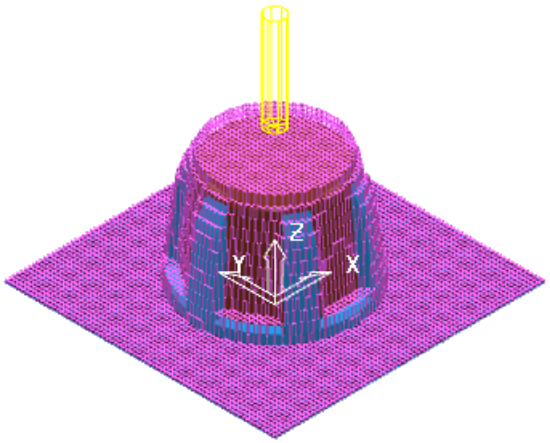
Creating a basic stock model

1. Calculate a **Block** and using a 15 mm **End Mill** tool:
   a. Create a **Model Area Clearance** toolpath with default values.
   b. Create a **Constant Z Finishing** toolpath with default values.
2. From the **Stock Models** context menu, select **Toolbar** to display the **Stock Model** toolbar.
3. Click to create a new stock model. This displays the **Stock Model** dialog.
   a. Change the name to **Roughing**.
   b. Click **Accept**.
4. Click to apply the block to the stock model.
5. Click to calculate the block state of stock model.



6. With the **Model Area Clearance** strategy activated, click to **Apply** the active toolpath to the stock model.

7 Click to **Calculate** the toolpath state of stock model.



Using a stock model as a block state

When you create a copy of a stock model, the currently active state in the stock model is used as the block state in the copy.

For example, if you create a copy of **Stock Model 1** with an active toolpath state, the copy created, **Stock Model 1_1**, contains a single block state which represents the stock from the active toolpath state in the primary stock model.

1 With the toolpath state active in the primary stock model, click to create a copy of the **Roughing** stock model.

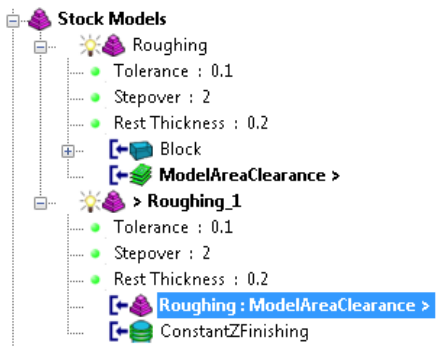
   This creates a stock model named **Roughing_1** with the toolpath state as the block.

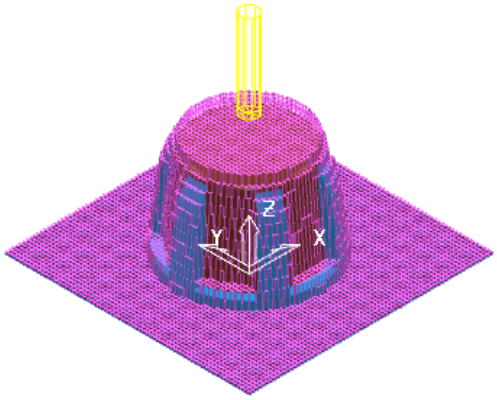
2 Activate the **Constant Z Finishing** strategy you created earlier.

3 On the **Stock Model** toolbar, click followed by to **Apply** and **Calculate** the toolpath state of the stock model.

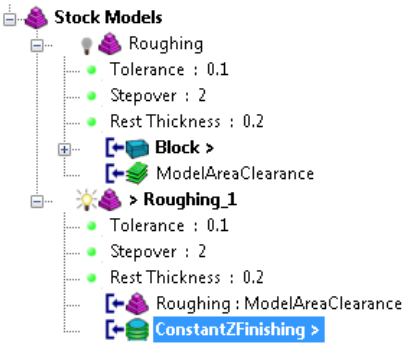
   The stock model **Roughing_1** now contains two states:

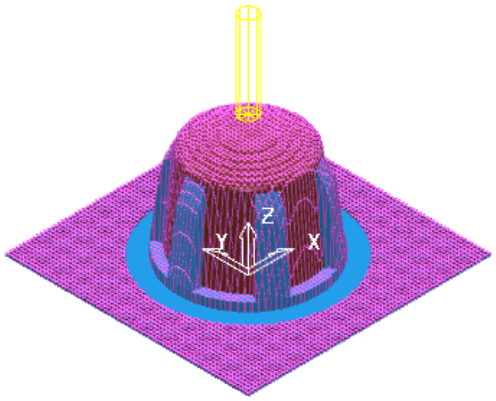
- The first state represents the block state, which is the **ModelAreaClearance** toolpath state inherited from the primary stock model **Roughing**.





- The second state represents **Constant Z Finishing** toolpath state applied to the stock model.

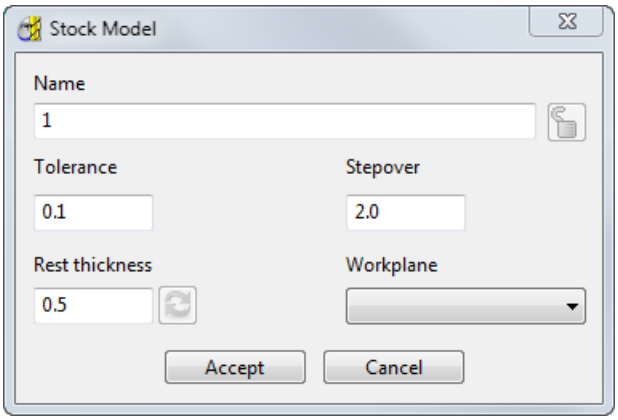




Individual Stock Model settings

Individual **Stock Model** settings are now available from the **Stock Model** settings dialog. Access the dialog from the **Settings** option on the individual stock model context menu.

The **Stock Model** dialog defines the parameters of the stock model.



To display the **Stock Model** dialog:

- From the **Stock Models** context menu, select **Create Stock Model**; or
- From the individual stock models context menu, select **Settings**; or
- From the **Stock Models** toolbar, select .

Name - the name of the stock model.

- click to unlock the stock model and enable editing.

*A **Stock Model** is automatically locked when it is used by another entity.*

Tolerance - determines the tolerance of a slice when a new stock model is created. The stock model is stored as a set of closed slices of the solid stock in X and Y. You cannot change this value after the stock model state has been calculated.

Stepover - the step between slices. You cannot change this value after the stock model state has been calculated.

Rest thickness - determines the minimum thickness displayed when using the **Drawing Options > Show Rest Material** option. The stock model ignores rest material thinner than this threshold. It is similar to the **Detect material thicker than** option on the **Boundary** dialog.

Refresh - click to refresh rest material. You can refresh rest material only if the **Drawing Options > Show Rest Material** option is enabled and you have modified the **Rest thickness**.

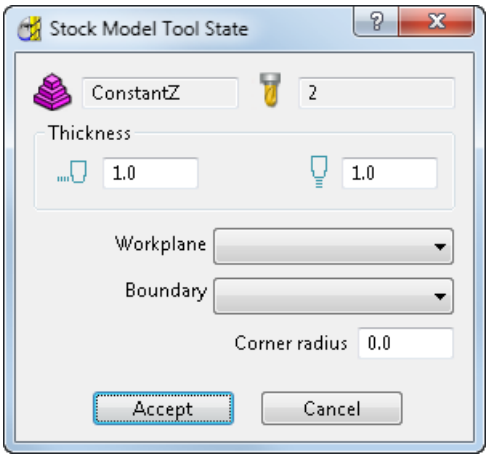
Workplane - select a workplane from the list to define the stock model. You cannot set or change the workplane after the stock model has been calculated.

Accept - accepts the values in the dialog.

Cancel - cancels the dialog without making any changes.

Individual Tool State dialog

Individual stock model tool state settings are now available from the **Stock Model Tool State** settings dialog. Access the dialog from the **Settings** option on the individual stock model tool state context menu.



ConstantZ - the name of the stock model.

2 - the name of the tool used to create the stock model tool state.

Radial thickness - the radial thickness used to define the stock model tool state.

Axial thickness - the axial thickness used to define the stock model tool state.

Corner radius - the corner radius used to define the stock model tool state.

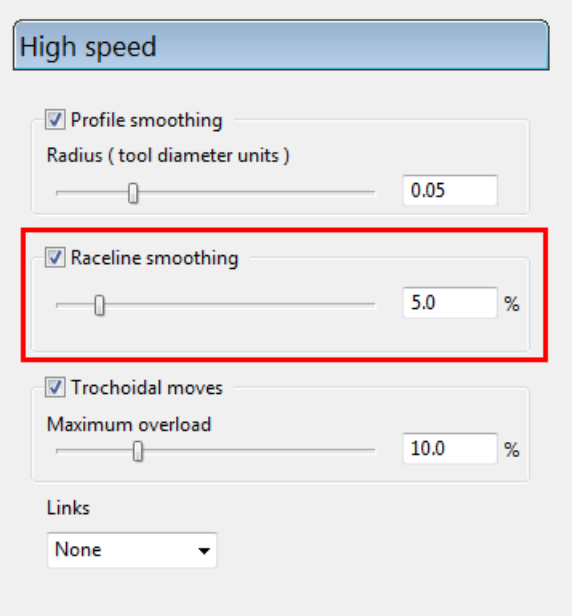
Workplane - lists available workplanes in the project. Select a workplane to use it to define the stock model tool state. You cannot set or change the workplane after the stock model state has been calculated.

Boundary - lists available boundaries in the project. Select a boundary to use it to define the stock model tool state. You cannot set or change the boundary after the stock model state has been calculated.

*You cannot change or modify the **Radial thickness**, **Axial thickness**, **Corner radius**, **Workplane**, and **Boundary** values for a calculated stock model state.*

Raceline smoothing

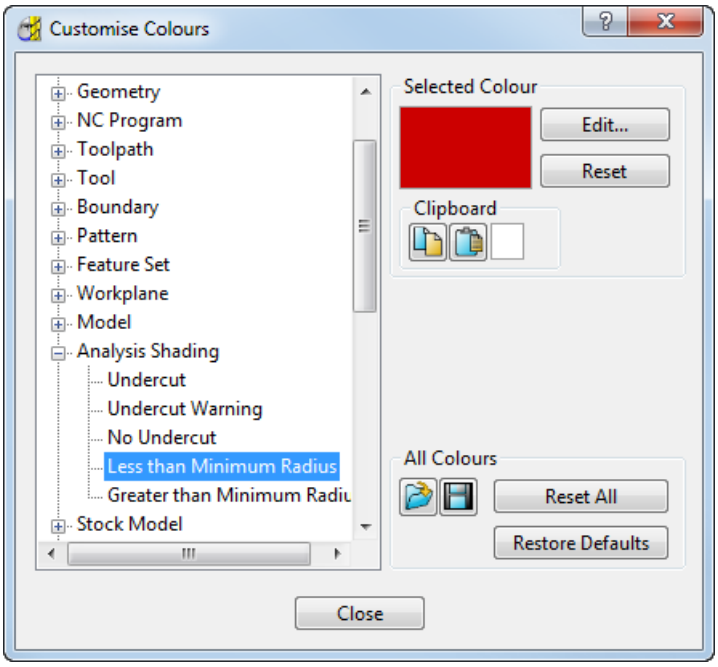
Raceline smoothing replaces **Smoothing allowance** on the **High speed machining** page of strategy dialogs.



The functionality remains unchanged, the name change makes it clearer which option provides arc fitting and which option provides raceline smoothing.

Customise shading colours

You can now edit the colours used to denote undercut regions and minimum tool radius shading. This is available from the **Tools > Customise Colours > Analysis Shading** menu.



The **Undercut**, **Undercut Warning**, and **No Undercut** colours are used when you select **Draft Angle shade** on the **Shading** toolbar.

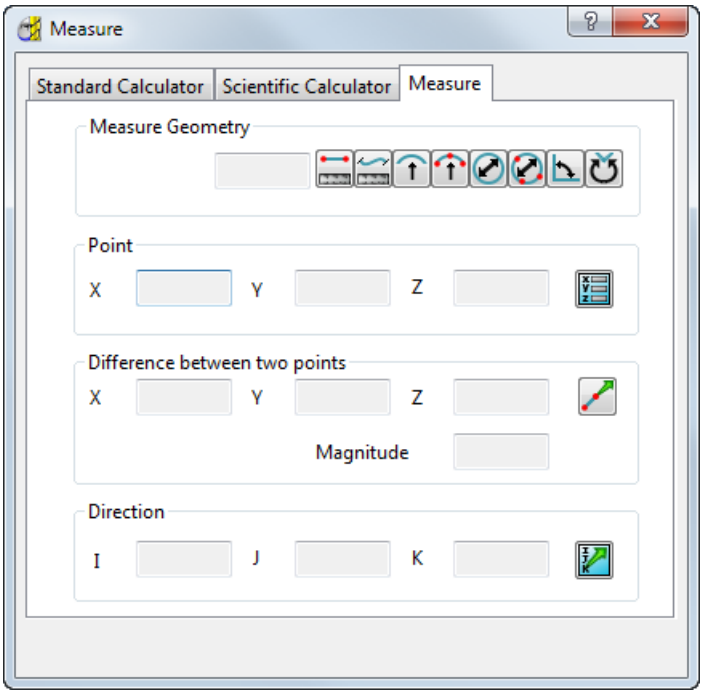
The **Less than Minimum Radius** and **Greater than Minimum Radius** colours are used when you select **Minimum Radius shade** on the **Shading** toolbar.

*The shading toolbar is available from the **View** toolbar.*

The default values remain unchanged.

Measurer

A new **Measure** tab has been introduced in PowerMILL 2012 R2.



- Use the **Measure Geometry** section to calculate points, coordinates, distances, angles, radii, or unit vectors.
- Click to display the **Position** dialog to locate items in the graphics window.
- Click to select two points in the graphics window to define a line and measure the difference in X (and Y and Z).
- Click to display the **Direction** dialog to view the direction of an item.

*Click inside a PowerMILL field and select **Measure** from the context menu to access the **Measurer**.*

This replaces the **Circle** and **Line** tabs.

Automation

Using plugins

You can now include programs which are written outside the main PowerMILL application but can interact with it. Sample plugins (such as **Create text**) are included with PowerMILL but others are specific to your setup.

The **Create text** plugin allows you to create text in a variety of fonts, sizes, and styles as a pattern or toolpath without having to use a CAD package.

Installing a plugin

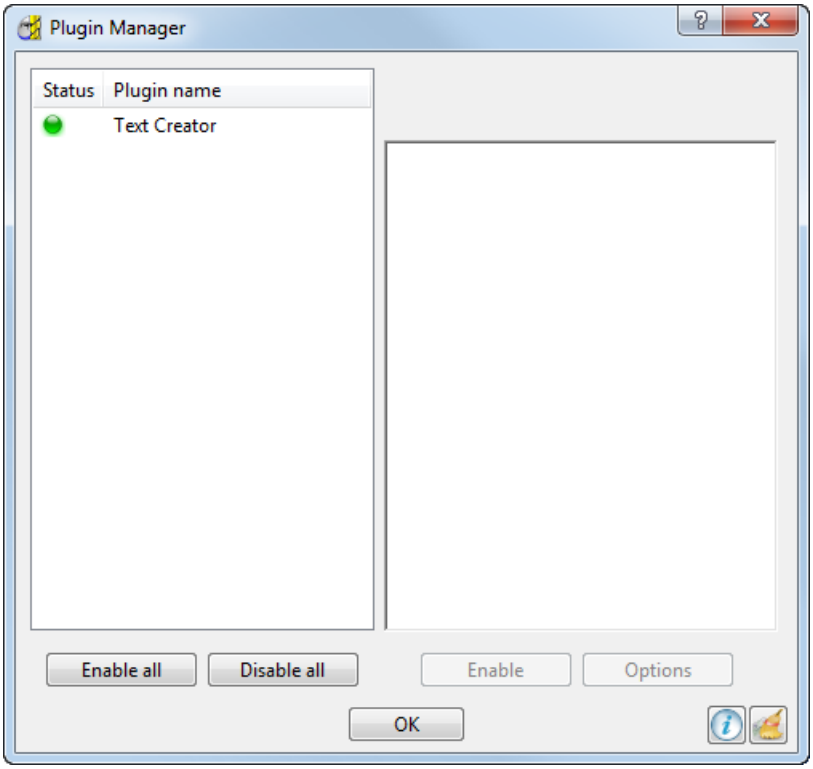
PowerMILL plugins are saved as ***.msi** files and are installed on a specific PC for all users of the PC. To install the **Create text** plugin:

1. In Windows explorer, right-click **OutlineFontInstaller.msi** and select **Install**.
2. Follow the installation wizard.
3. Start PowerMILL.

   It may take longer to start PowerMILL when installed plugins are loaded.

   PowerMILL starts with the **Create text** plugin in a pane within the main window.
4. If the **Create text** pane isn't visible, select **View > Toolbar > Plugins**.
5. You can control the default width of the plugin pane from **Tools > Options > View > Explorer Dimensions**.

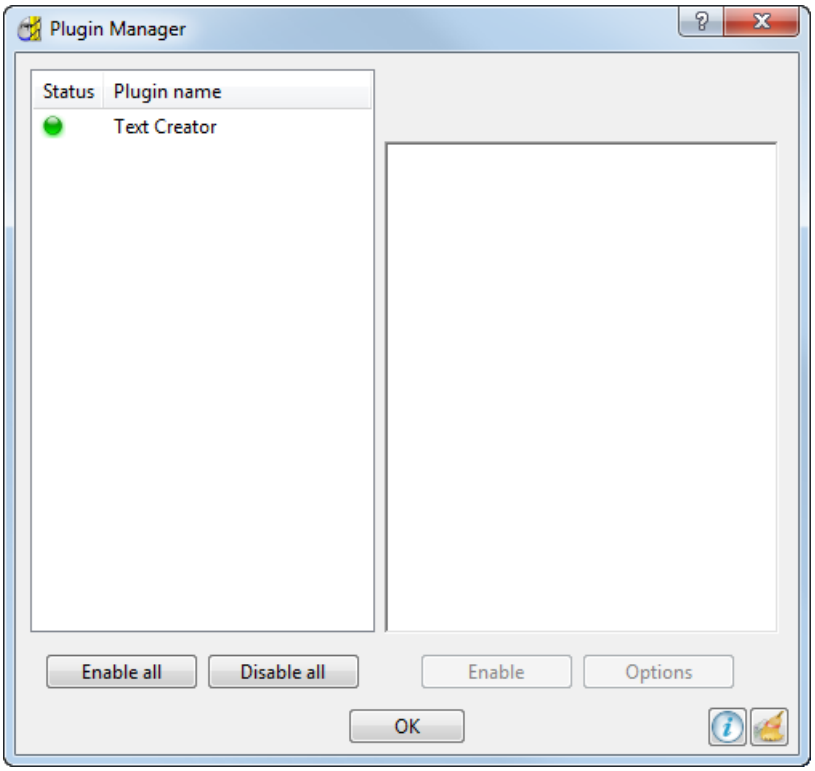
6 Control which plugins are enabled (and so available) from the **Plugin Manager** dialog available from the **Tools > Plugins** menu (see page 59).



You must have administrator rights to install a plugin.

Tools > Plugins

The **Plugin Manager** shows which plugins are installed on your PC and which are enabled.



The right side of the dialog displays all the available plugins. The left side displays information on the selected plugin.

Status - shows which plugins are available.

- available and enabled.

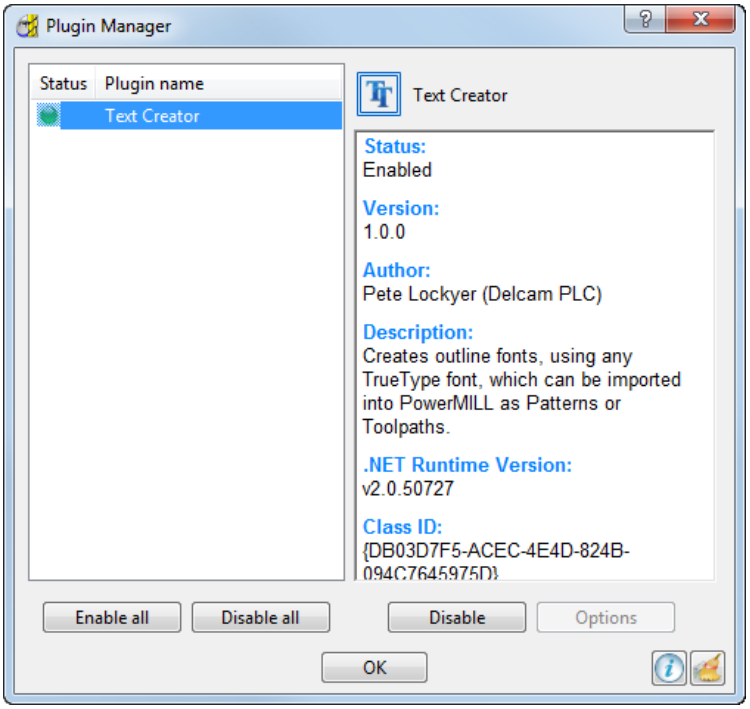
- available but disabled.

- requires a newer version of PowerMILL to run. PowerMILL can load the plugin and display its details.

- not available. PowerMILL can't load the plugin, so it can't display any details about it. This happens if the plugin is corrupt, or hasn't been registered correctly.

Plugin name - shows a list of all the plugins installed on this PC.

Plugin summary - the right-hand pane shows a summary of the selected plugin.



Enable all - enables all the plugins installed on this PC.

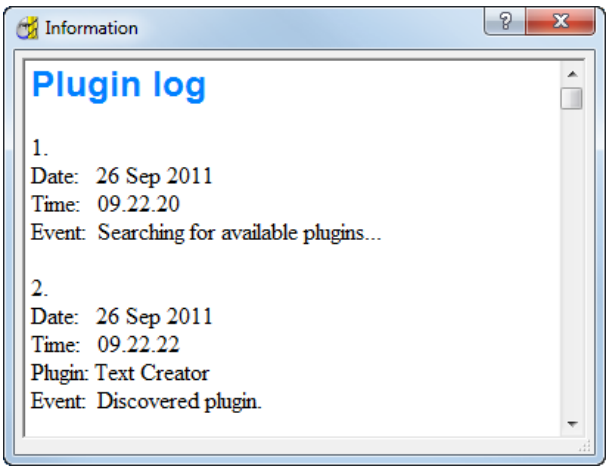
Disable all - disables all the plugins installed on this PC.

Enable/Disable - either enables the disabled plugin, or disables the enabled plugin.

This button refers to the plugin selected in the left-hand pane.

Options - displays the **Options** dialog of the plugin. If the plugin doesn't have an **Options** dialog this button is unavailable.

Information - displays the plugin log.



Clean up log - removes all items in the plugin log.

For more information, see Using Plugins (see page 57).

Macro programming enhancements

There is a new **Debug** option on the **Macros** menu which opens the **Macro Debugger** (see page 61). This replaces the **Step Through Macro** option available in previous versions.

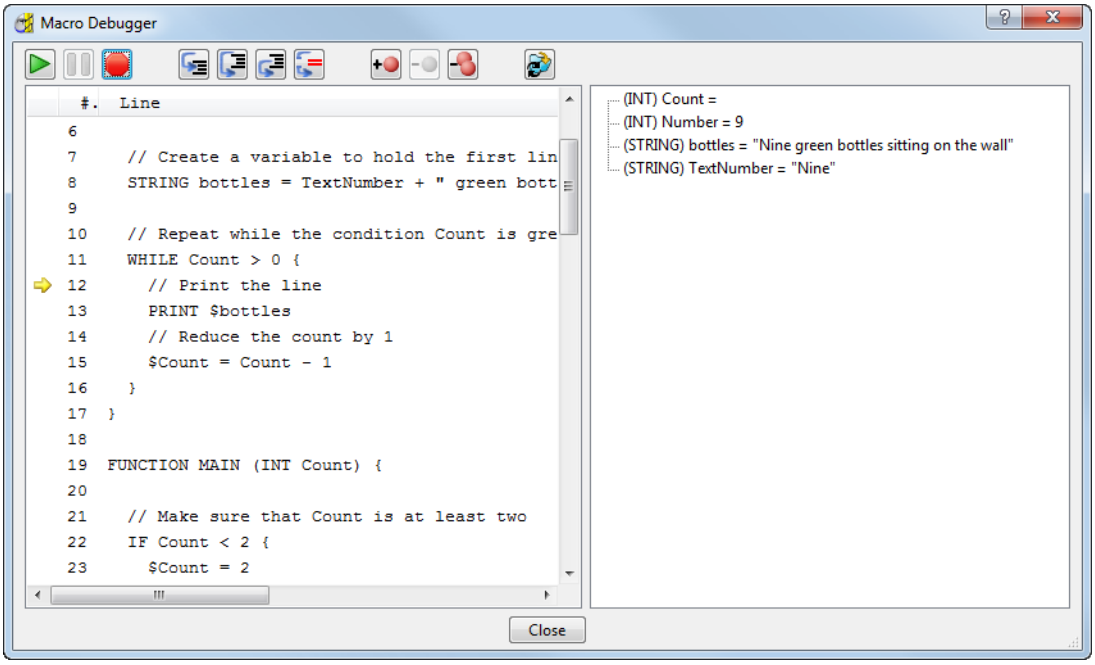
The macro programming language now includes:

- Lists (as well as arrays).
- Vector and point variables.
- An option to terminate the current macro.
- Some constants.
- Entity-based functions.
- A built-in evaluation function.
- The ability to return the pathname of the current project.

For more information, see the Macro Programming Guide.

Macro debugger

The **Macro Debugger** enables you to test, debug, step-through, and track variable values in your macro.



When debugging a macro you tend to have a portion of code that you are interested in (either to understand how it works, or why it isn't working). To look at this portion of code you:

- Insert breakpoints at points of interest.

- Run the macro using **Execute**. This allows you to execute your macro up to the point of interest to ensure everything is in the correct state.
- Use the various **Step** functions to understand what is going on in your macro line by line.
- Run the macro using **Execute** to the next point of interest, or to the end of the macro.

Execute - runs the macro from the current position until it reaches either a **breakpoint** or the end of the macro.

shows the current position.

If you pause a macro, then **Execute** resumes the macro execution.

Pause - inserts a temporary **breakpoint** after executing the current line.

Stop - stops the macro from executing and returns to the start of the macro.

The step commands execute the current line and then wait for further debugger interaction. This enables you to see the effect of a macro statement or command and so helps your understanding of the macro. The variable values assigned at this point of the macro are displayed in the right-hand pane.

Step into - executes the current line only and then waits for further debugger interaction. If the current line is a macro function call, then the debugger pauses before executing the first line of the function. This allows you to check a function.

Step over - executes the current line only and then waits for further debugger interaction. If the current line is a macro function call, then the debugger pauses after the entire function has executed. This is useful when you know the function works, so you don't need to look at it in detail, but want the results from it.

Step out - executes the current line and all subsequent lines within the block. When the block of commands ends, PowerMILL waits for further debugger interaction.

Skip next line - skips the next line without executing the command. This is useful when you know a command will execute successfully but takes a long time to calculate, say calculating a toolpath. However, it can be dangerous to **Skip next line** because future commands may require the output from this command.

Add breakpoint - places a breakpoint at the start of the selected line. A breakpoint forces the macro to stop at this point. The variable values assigned at this point of the macro are displayed in the right-hand pane. You can insert multiple breakpoints in your macro.

For example, if you put a breakpoint at the end of a loop you can examine the variable values and see how they change with each iteration. This enables you to find out if the macro is behaving as you'd expect.

Another example is: if you put a breakpoint at the beginning of a function you can see which variables and parameters are passed to the function.

Delete breakpoint - removes a breakpoint from the selected line.

Delete all breakpoints - removes all breakpoints from the macro.

Reload macro - stops the current execution of the macro and reloads the macro file. This is useful when you have fixed a problem with your macro and want to run it again to check it.

The left pane shows the macro you are debugging.

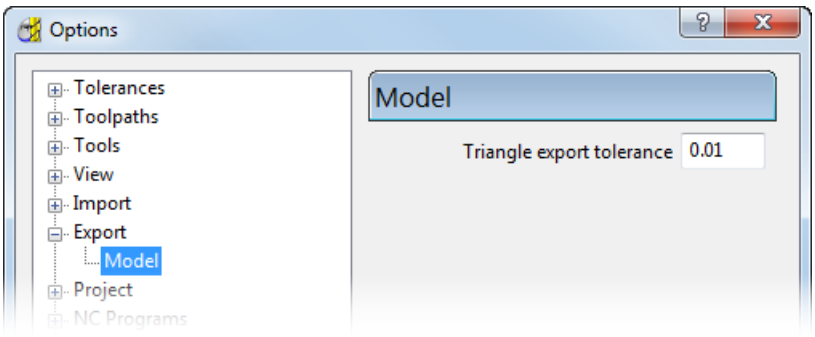
The right pane shows the variable assignment at that point in the macro.

If you want to run the debugger on a macro that takes an argument in `Main()` *you must use the command* `MACRO DEBUG mymacro.mac arguments` *where* ***mymacro*** *is the name of the macro and* ***arguments*** *is the list of arguments you want to pass to the macro you are about to debug.*

General enhancements

There are a few general enhancements:

- PowerMILL now makes more effective use of memory. This is particularly noticeable on large parts which now machine more quickly.
- Arc fitting is much improved and more reliable. Arc fitting is available from the high speed pages of area clearance, constant Z and raster strategies.
- You can now export models as **.stl** files even when no machine operations have taken place. You can determine the **Triangle export tolerance** on the **Options** dialog (available from **Tools > Options > Export > Model** menu).

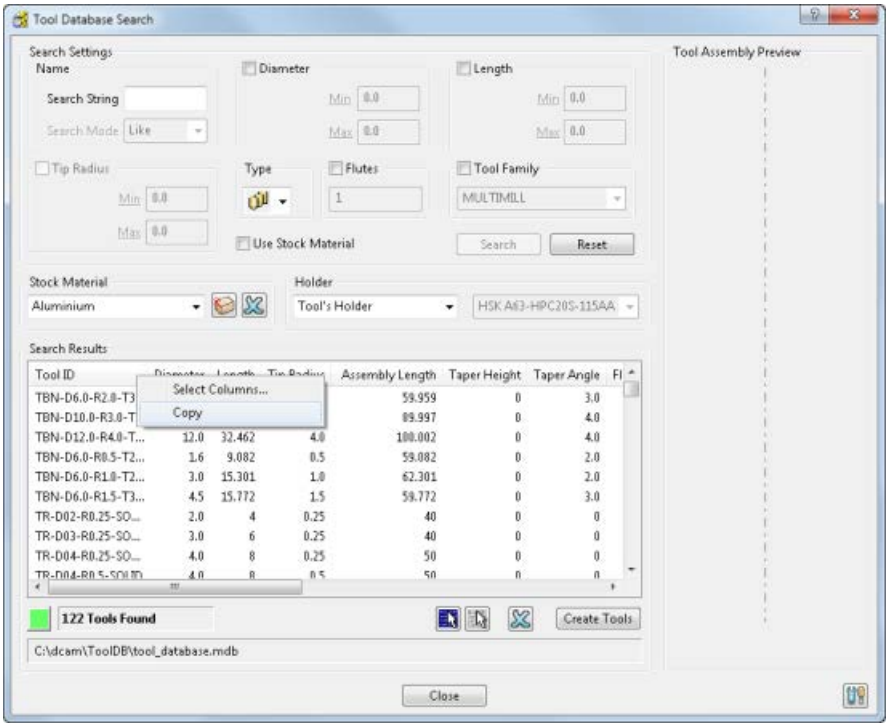


Triangle export tolerance determines the accuracy used to translate from the **dgk** format to an **stl** format.

Copying toolpath and tool data to the clipboard

You can copy toolpath and tool data from the **NC Program** and **Tool Database** dialog to the Windows clipboard. This is useful if you want to create reports using other applications.

To copy data in the **NC Program** or **Tool Database** dialogs, right-click the column heading and select **Copy**.



This copies all data from the dialog to the Windows clipboard.

*To copy specific rows, select individual rows, right-click the column heading and select **Copy**.*

For example, data copied from the tool database to a spreadsheet looks similar to:

	A	B	C	D	E	F	G	H
1	Tool ID	Diameter	Length	Tip Radius	Assembly	Taper Height	Taper Angle	Flutes
2	TBN-D6.0-R2.0-T3-SOLID	6	21.029	2	59.959	0	3	2
3	TBN-D10.0-R3.0-T4-SOLID	10	31.497	3	89.997	0	4	2
4	TBN-D12.0-R4.0-T4-SOLID	12	32.462	4	100.002	0	4	2
5	TBN-D6.0-R0.5-T2-SOLID	1.6	9.082	0.5	59.082	0	2	2

*You can also copy data from the **Toolpath List** and **Tool List** dialogs.*

Adding a custom entry to an entity context menu

You can now add custom options to PowerMILL entity context menus.

To add a menu option to an entity context menu

1 Create an XML file and name it as the entity type it represents.

The names allowed are: Boundary, Featureset, Group, Level, MachineTool, Macro, Model, NCProgram, Pattern, StockModel, Tool, Toolpath, and Workplane.

The name is not case sensitive.

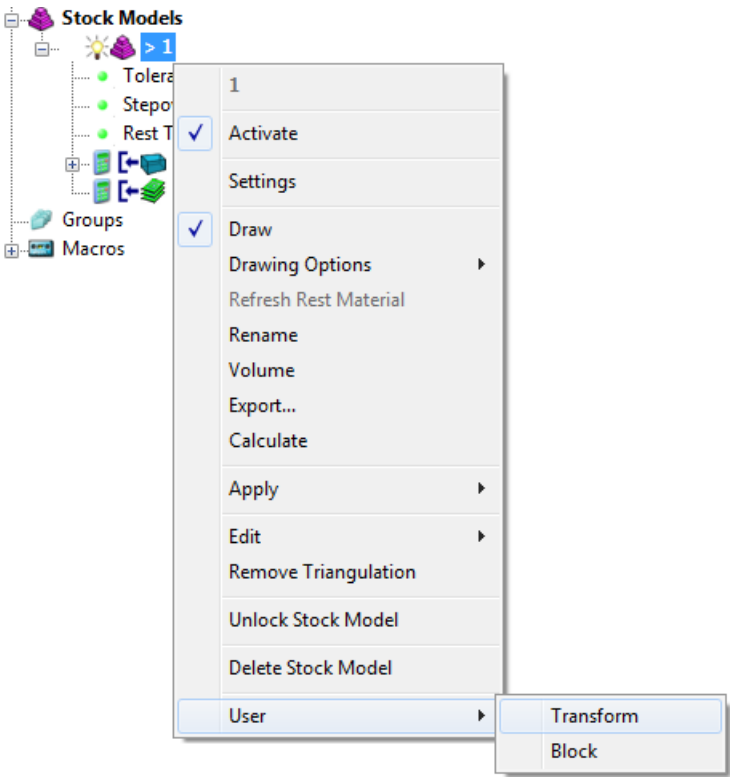
2 Place the <entity_type>.xml file in the PowerMILL Home\pmill folder.

Format of the custom entity menu option

The format of the custom addition to the PowerMILL entity menu is:

```
<?xml version="1.0" ?>
  <menupage>
    <button label= "The name of the menu option"
    command= 'SOME POWERMILL COMMAND/>'
  </menupage>
```

For example, to create this sample submenu for an individual stock model entity:



The code in the **stockmodel.xml** file is:

```
<?xml version="1.0" ?>
  <menupage>
    <button label= "Transform" command= 'FORM MODELTRANS
    EDIT MODEL "%s" INTERACTIVE'/>
    <button label="Block" command='FORM BLOCK'/>
  </menupage>
```

Where:

label - is the name of the menu item.

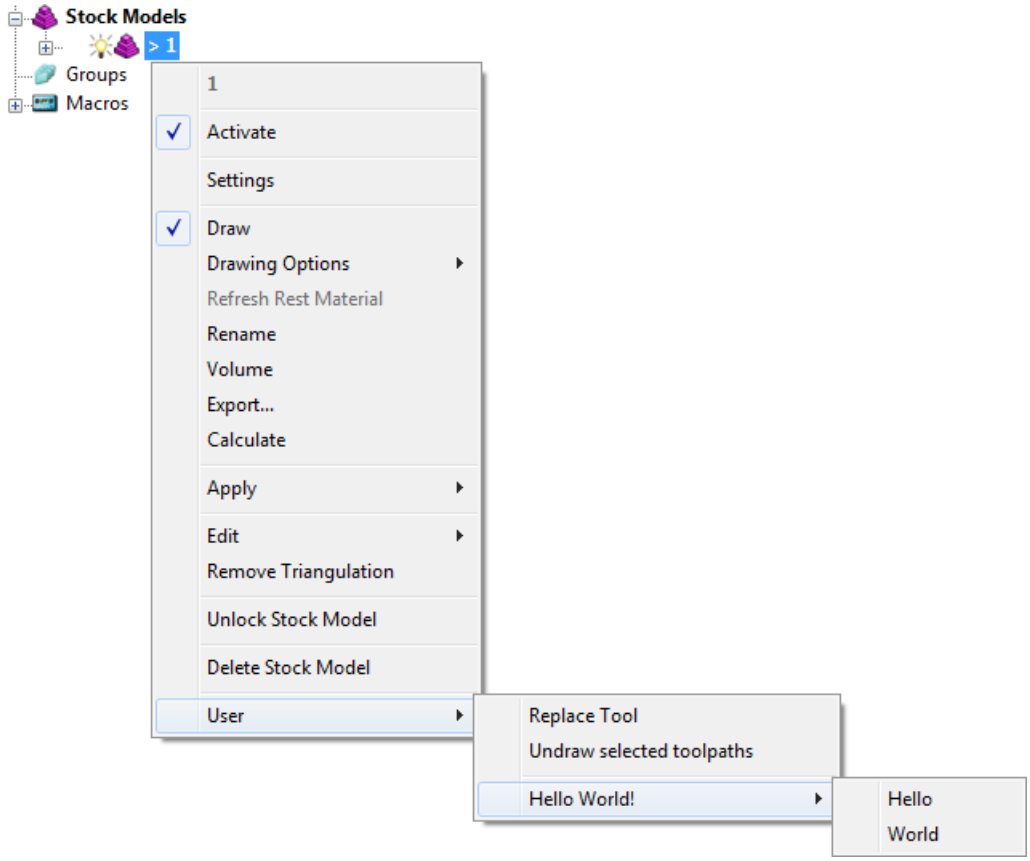
command - is the PowerMILL command.

%s - is the wildcard character which represents the individual entity name.

Adding spaces and submenus to the menu option

To add spaces to the menu, add the `<spacer/>` tag to the XML code. To add submenus, include the submenu name in a `<menupage>` tag.

For example, to create this **User** menu and submenu called "**Hello World!**" with relevant spaces:



The code in the **stockmodel.xml** file is:

```
<?xml version="1.0" ?>
<menupage>
  <button label="Replace Tool"
    command='EDIT TOOLPATH "%s" REPLACE_TOOL ;'
    multiple_selection="allowed"/>

    <button label="Undraw selected toolpaths"
    command='MACRO d:/temp/test.mac "%s"'
    multiple_selection="allowed"/>

  <spacer/>
```

```
  <menupage label="Hello World!">
    <button command='PRINT "HELLO"' label="Hello"/>
    <button command='PRINT "WORLD"' label="World"/>
  </menupage>

</menupage>
```

Enabling multiple selections for a menu option

To enable the menu option over multiple selections, add the keywords **multiple_selection="allowed"** for each **<button />** tag.

For example:

```
    <button label= "Transform"
    command= 'FORM MODELTRANS EDIT MODEL "%s"
    INTERACTIVE'
    multiple_selection="allowed"/>
```

Index

A

Appending transformed toolpaths • 36
Arc fitting • 64
Area clearance toolpaths
  Area clearance step cutting • 11
  Detection limit • 30
  Flat machining • 26
  Flats on triangles • 26
  Minimum gap length • 25
  Plunge move warning • 29
  Rest roughing fragmentation • 25
Automatic collision checking
  Machine maximum stock • 38
Automatic verification
  Machine maximum stock • 38
  Thickness - drilling • 35

B

Base cutting feed rate - step cutting • 14
Batch move toolpath start points • 27
Boundary transformations • 5
Bounding box origin • 5
Breakpoints • 61

C

Colours • 55
  Draft angle shading • 55
  Minimum radius shading • 55
  Undercut shading • 55
Complementary angle - polar milling • 19
Constant volume removal - step cutting • 14
Copying data • 64
Corner clearance
  Detection limit • 30
Custom entity menu options • 65
Customise colours • 55

D

Detect area thicker than - step cutting • 14
Detection limit • 30
Draft angle shading • 55
Drill length • 7
Drilling
  Drill length • 7
  Spiral profile drilling • 32
  Thickness - drilling • 35

E

Expand area by - step cutting • 14

F

Feature tolerance • 6
Feed rate - step cutting • 14
Finishing toolpaths
  Flat machining • 26
  Parametric spiral finishing • 35
  Steep and shallow finishing • 31
Flat machining • 26
  Flats on triangles • 26
Flats on triangles • 26

Fragmentation reduction • 25

G

Geometry Transform toolbar • 5
Geometry transformations • 5

M

Machine axis control • 18
  Polar milling • 18, 19
Machine maximum stock • 38
Machine tools • 41
  Machine tool opacity • 43
Macro debugger • 61
  Breakpoints • 61
Macro programming guide • 61
Maximum feed rate - step cutting • 14
Minimum gap length • 25
Minimum radius shading • 55
Move toolpath start points • 27

P

Parametric spiral finishing • 35
Pattern transformations • 5
Plugins • 57
Plunge move warning • 29
Polar milling • 18, 19
  Complementary angle - polar milling • 19
  Rotation centre - polar milling • 19
Profile drilling • 32
Programs
  Plugins • 57

R

Raceline smoothing • 54
Reconnecting machine tools • 41
Rest roughing fragmentation • 25
  Fragmentation reduction • 25
  Minimum gap length • 25
Rotation centre - polar milling • 19

S

Shading • 55
  Draft angle shading • 55
  Minimum radius shading • 55
  Undercut shading • 55
Simulation modes
  Feed rate simulation • 45
  Point-to-point simulation • 45
Smoothing
  Arc fitting • 64
  Raceline smoothing • 54
Spiral profile drilling • 32
Steep and shallow finishing • 31
Step cutting • 14
  Base cutting feed rate - step cutting • 14
  Constant volume removal - step cutting • 14
  Detect area thicker than - step cutting • 14
  Expand area by - step cutting • 14
  Feed rate - step cutting • 14
  Maximum feed rate - step cutting • 14
  Stepup - step cutting • 14
Stepup - step cutting • 14
STL files • 64
Stock models • 47
  Stock model settings • 51, 53
  Stock model toolbar • 47, 49

T

Thickness - drilling • 35
Tolerance
  Feature tolerance • 6
Tool
  Drill length • 7
  Tool assembly pan and zoom • 10
Tool assembly pan and zoom • 10
Tool axis
  Machine axis control • 18
  Polar milling • 18, 19
Toolbars
  Geometry Transform toolbar • 5
  Transform toolbar • 5
Toolpath start points • 27
Toolpath transformatios • 36
Toolpaths
  Appending transformed toolpaths • 36
  Area clearance step cutting • 11
  Detection limit • 30
  Flat machining • 26

Flats on triangles • 26
Minimum gap length • 25
Parametric spiral finishing • 35
Plunge move warning • 29
Rest roughing fragmentation • 25
Steep and shallow finishing • 31
Step cutting • 14
Toolpath transformatios • 36
Transform toolbar • 5
Transformations • 5
Appending transformed toolpaths • 36
Boundary transformations • 5
Bounding box origin • 5
Geometry transformations • 5
Pattern transformations • 5
Toolpath transformatios • 36
Workplane origin • 5

U

Undercut shading • 55

V

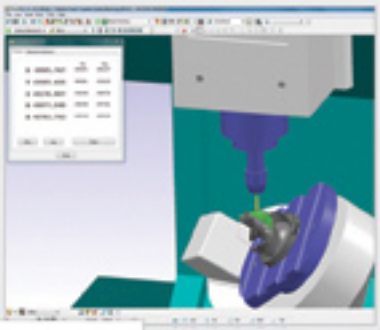
Verification
Machine maximum stock • 38

W

WN2012R2
Measurer • 56
Workplane from three arc points • 7, 8
Workplane origin • 5

PowerMILL 2012 R2

PowerMILL 2012



new features

Delcam TV

www.delcam.tv

PowerMILL Learning Zone

www.delcam.tv/lz

PowerMILL Website

www.powermill.com

THE QUEEN'S AWARDS FOR ENTERPRISE: INNOVATION 2010

THE QUEEN'S AWARDS FOR ENTERPRISE: INTERNATIONAL TRADE 2011

Powering your productivity

Delcam Headquarters | Small Heath Business Park | Birmingham | B10 0HJ | UK
+44 (0)121 766 5544 | marketing@delcam.com | www.delcam.com

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

© Copyright Delcam plc 2012. All other trademarks are the property of their respective owners.