Autodesk® PowerMill® 2017

What's New

Autodesk® PowerMill® 2017

Trademarks

Disclaimer

Contents

Summary of new features

PowerMill is the leading NC CAM software specialising in the manufacture of complex shapes typically found in the toolmaking, automotive, and aerospace industries. PowerMill 2017 offers all of the original features of PowerMill 2016, but with numerous improvements. This document describes the most significant improvements.

PowerMill 2017 contains the following new features and enhancements:

2D Machining (see page 5)

- **Separating features and holes** (see page 6) — Create, edit and organise features more easily using the new **Feature Groups** entity.
- **Feature Editor mode-toolbar** (see page 7) — Create and edit features using the new **Feature Editor** mode-toolbar.
- **Interactive feature detection** (see page 9) — Automatically create features from a model using the new **Interactive Feature Detection** dialog.
- **Open region editor** (see page 11) — Define open regions of a pocket feature using the new **Open Region Editor** mode-toolbar.
- **Feature hierarchy** (see page 13) — Analyse the parent-child relationships of nested features in a feature group using the new **2D Feature Hierarchy** dialog.
- **Enhancements to features** (see page 15) — Improvements to creating and editing features in addition to new turning and face features.
- **Enhancements to 2D machining strategies** (see page 26) — New strategies for machining improved 2D features.

Turning (see page 90)

- **Turning view** (see page 90) — Set the view mode to determine how the part is orientated by the standard views.
- **Turning workplanes** (see page 91) — Create a turning workplane where the Z axis is the rotational axis of the part.
- **Turning curves** (see page 92) — Extract a spun profile curve from a model using the new **Spun profile** dialog.
- **Turning features** (see page 96) — Create turning features from curves for turning toolpaths.
- **Turning tools** (see page 98) — Create or import turning tools.
- **Turning toolpaths** (see page 102) — Create turning toolpaths using the new turning strategies.
- **Turning simulation** (see page 104) — Simulate turning toolpaths to check for collisions.

Generating toolpaths (see page 106)

- **Automatic tool axis limits** (see page 107) — You can automatically calculate the tool axis limits for a toolpath, using the machine tool information.
- **Toolpath connections** (see page 108) — You can quickly and easily define toolpath connections, rapid move clearances, and tool safe areas using the new **Toolpath connections** dialog.
- **Area clearance offset changes** (see page 121) — There are modifications to **Offset all** and **Offset model** area-clearance strategies that reduce the number of small moves required to remove upstands.
- **Rib machining enhancements** (see page 121) — You can now generate rib machining toolpaths that cut down the centreline of a rib and along its walls.
- **External thread milling** (see page 122) — There is a new **Feature external thread milling** strategy. You can now create toolpaths with multiple start points.
- **Finishing strategies enhancements** (see page 123) — There are small enhancements to the **3D Offset Finishing** and **Steep and Shallow Finishing** strategies.

Simulating toolpaths (see page 125)

- **Simulation pausing (see page 126)** — You can now control how often and on what types of issues simulation stops.
- **Simulation playback** (see page 128) — There are new controls on the **Simulation** toolbar.

- **Simulation toolbar** — The **Simulation issues**  button is now on the **Simulation** toolbar. There is a new **Collision checking**  button on the **Simulation** toolbar. Toggle this button to turn collision checking on or off.

General enhancements **(see page 129)**

- **ViewCube** (see page 129) — Use the new ViewCube to interactively orientate the contents of the graphics window.
- **Live text creation** (see page 131) — You can now create text as wireframe, for engraving or leaving comments in a project.
- **Autodesk A360** (see page 134) — The **Tools** options includes an **Autodesk A360 option**.
- **Watertight stock models** — You can now export watertight stock models as .stl or .dmt files.
- **Save strategy parameters** — The parameters you can individually select to save with a strategy are updated to reflect the toolpath connection changes.

PowerMill rebrand

PowerMill has new branding and is now supplied under the Autodesk licensing system. Details of the license are available in the **About** dialog.

To open the dialog, select the **Help > About** menu option.

2D Machining

There are enhancements to a number of areas in PowerMill related to the 2D machining workflow and user interface. The focus of these changes is to enable you to intuitively create, edit, and machine features using 2D machining strategies.

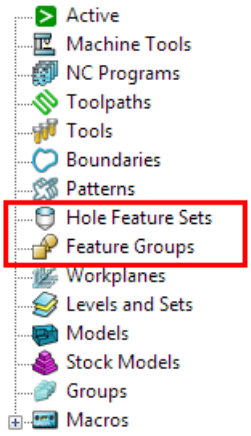
The following changes support these improvements:

- **Separating features and holes** (see page 6) — Create, edit and organise features more easily using the new **Feature Groups** entity.
- **Feature Editor mode-toolbar** (see page 7) — Create and edit features using the new **Feature Editor** mode-toolbar.
- **Interactive feature detection** (see page 9) — Automatically create features from a model using the new **Interactive Feature Detection** dialog.
- **Open region editor** (see page 11) — Define open regions of a pocket feature using the new **Open Region Editor** mode-toolbar.
- **Feature hierarchy** (see page 13) — Analyse the parent-child relationships of nested features in a feature group using the new **2D Feature Hierarchy** dialog.
- **Enhancements to features** (see page 15) — Improvements to creating and editing features in addition to new turning and face features.
- **Enhancements to 2D machining strategies** (see page 26) — New strategies for machining improved 2D features.

Separating features and holes

To support the 2D machining improvements in PowerMill 2017, features and holes are separated out of Feature Sets into their own entities. This enables you to easily create, edit and organise features on a model.

The explorer includes two new branches, **Hole Feature Sets** and **Feature Groups**.



Use the new **Feature Groups** branch to create and edit features using the new **Feature Editor** mode-toolbar (see page 7).

Use the **Hole Feature Sets** branch to create and edit holes. The workflow of this branch is identical to the old Feature Sets.

Feature editing mode-toolbar

Display the **Feature Editor** mode-toolbar by selecting **Feature Editor** from the **Feature Groups** context menu.

Use the options to create or edit features:

Tolerance — Enter a value to specify the positional tolerance for features with respect to each other. This affects how features are ordered in the hierarchy. For example, if you want to create a boss feature within a pocket feature, having a high tolerance requires the boss' lower face to lie on or very close to the pockets lower face to be recognised as the pockets child feature. A low tolerance value enables the boss feature to be recognised as a child if it is slightly misplaced.

Snap — Click to toggle between snapping using the intelligent cursor or disabling snapping.

shows snapping is disabled.

shows snapping is enabled.

The **CTRL** key temporarily disables snapping.

Select all — Click to select all features.

Select toggle — Click to deselect the selected features and select the deselected features.

Select invalid — Click to select all the invalid features in the model.

Detect features — Click to display the **Interactive Feature Detection** dialog (see page 9).

Create rectangular pocket — Click to create a rectangular pocket feature.

Create circular pocket — Click to create a circular pocket feature.

Create freeform pocket — Click to create a freeform pocket feature from a curve.

Create rectangular boss — Click to create a rectangular boss feature.

Create circular boss — Click to create a circular boss feature.

 **Create freeform boss** — Click to create a freeform boss feature from a curve.

 **Create slot** — Click to create a slot feature from a curve.

 **Create unbounded face** — Click to create an unbounded face feature.

 **Create bounded face** — Click to create a bounded face feature from a curve.

 **Create turning profile** — Click to create a turning profile feature from a curve.

 **Create freeform groove** — Click to create a freeform groove feature from a curve.

 **Create parametric groove** — Click to create a parametric groove feature.

 **Create turning face** — Click to create a turning face feature.

 **Create bore** — Click create a bore feature from a curve.

 **Colour** — Click to edit the colour of a selected feature.

 **Edit feature** — Click to edit a selected feature (see page 21).

 **Feature hierarchy** — Click to display the **2D Feature Hierarchy** dialog. (see page 13)

 **Edit profile curve** — Click to edit the profile curve of a selected feature using the **Curve Editor** mode-toolbar.

 **Edit open regions** — Click to edit the open regions of a selected pocket using the **Open Region Editor** mode-toolbar (see page 11).

 **Delete** — Click to delete a selected feature.

 **Undo** — Click to undo your changes.

 **Redo** — Click to reinstate the changes you have undone.

 **Calculator** — Click to display the **Calculator/Measure** dialog from within the **Feature Editor** mode-toolbar.

 **Accept** — Click to accept and keep all the created features. This closes the **Feature Editor** mode-toolbar and enables normal PowerMill functionality.

Cancel — Click to delete all the created features. This closes the **Feature Editor** mode-toolbar and enables normal PowerMill functionality.

Interactive feature detection

You can now easily detect and create features based on model geometry using the new **Detect Features** dialog. This is useful if you want to scan a model to detect all features with similar geometry and then create those features.

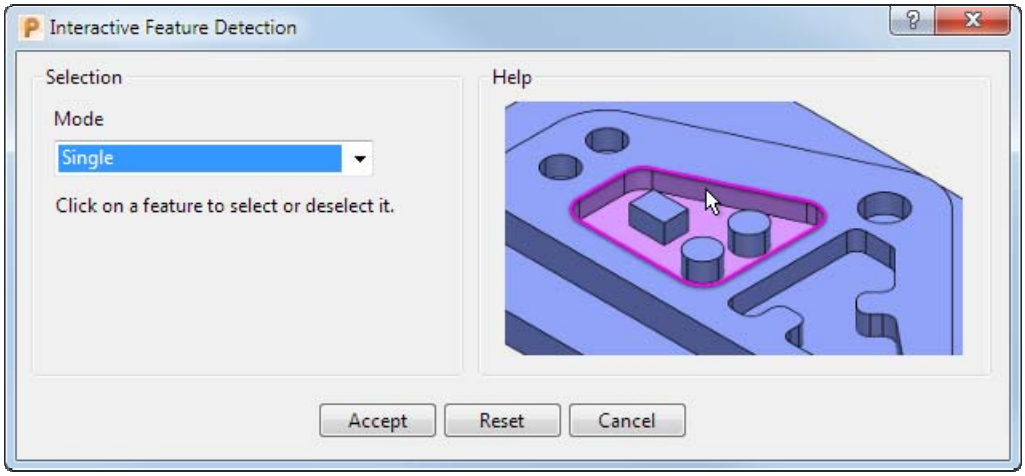
To detect and create features based on model geometry:

1. Right-click **Feature Groups** from the explorer and select **Feature Editor** from the **Feature Groups** menu.

   The **Feature Editor** mode-toolbar is displayed.

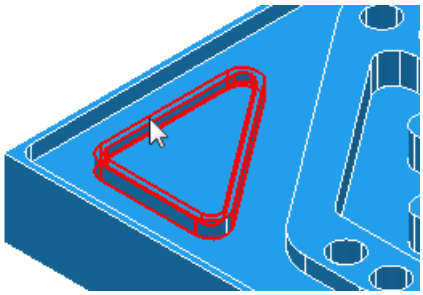
2. Click from the **Feature Editor** mode-toolbar.

   The **Interactive Feature Detection** dialog is displayed.

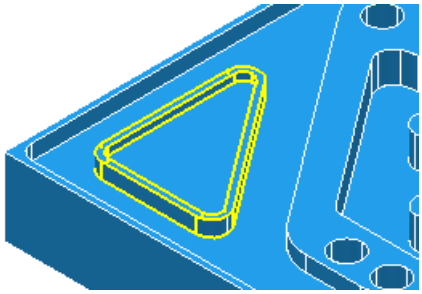


3. Select an option from the **Mode** list to specify how features are selected:
   - **Single** — Click a feature to select it.
   - **Hierarchy** — Click a feature to select it and any other features inside of it.
   - **Similar** — Click a feature to select it and any other features with similar geometry.

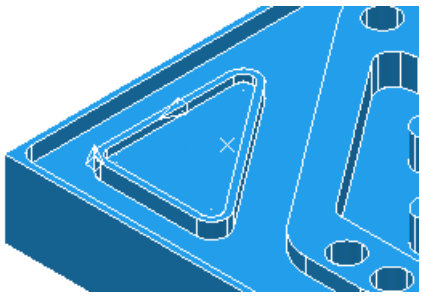
4 Hover your cursor over the model to detect the features. The feature boundary is highlighted.



5 Click to select the detected features. Click the feature again if you want to deselect it.



6 Click **Accept** to create the features and close the dialog.



The created features are added to a feature group in the explorer.

Open region editor

Use the new **Open Region Editor** mode-toolbar to define open regions on pocket features in a model.

Create open region — Click to create an open region on a pocket feature.

Delete open regions — Click to delete open regions on all selected pocket features.

Invert open regions — Click to invert open regions on all selected pocket features.

Undo — Click to undo your changes.

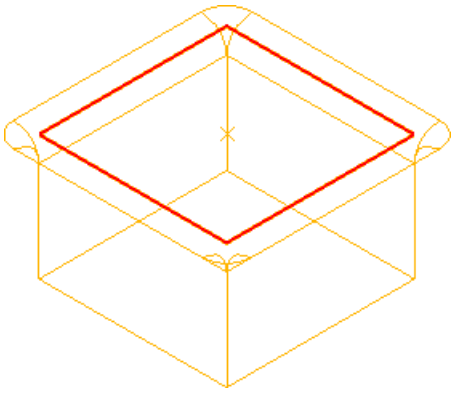
Redo — Click to reinstate the changes you have undone.

Accept — Click to accept and keep all the created open regions. This closes the **Open Region Editor** mode-toolbar and returns to the **Feature Editor** mode-toolbar.

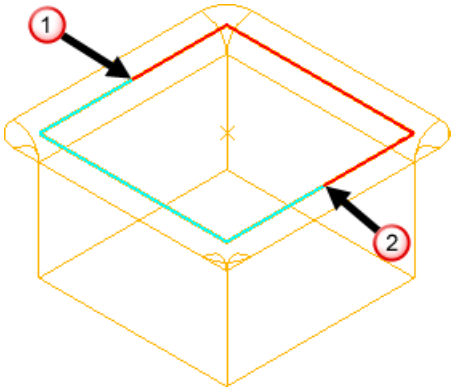
Cancel — Click to delete all the created open regions. This closes the **Open Region Editor** mode-toolbar and returns to the **Feature Editor** mode-toolbar.

To define an open region on a pocket feature:

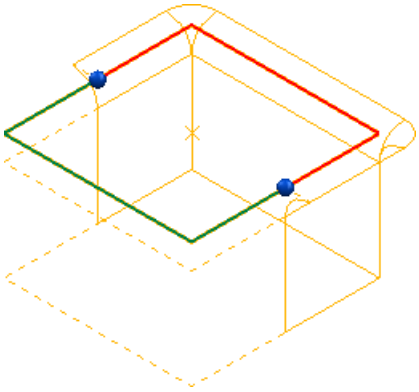
1. Click on the **Feature Editor** mode-toolbar.
   The **Open Region Editor** mode-toolbar is displayed.
2. Click on the **Open Region Editor** mode-toolbar.
3. Select a pocket feature.



4 Click on the model to specify a start ① and end point ②.

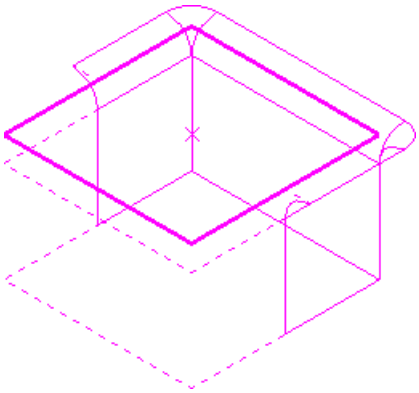


The open region is drawn and represented by a dashed line:



5 Drag the handles to redefine the start and end points if necessary.

6 Click  to accept the changes and create the open region.

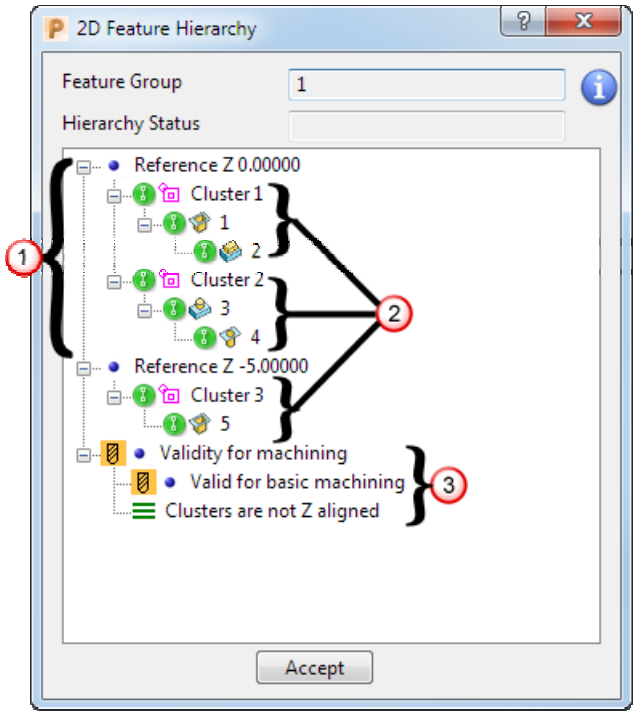


Feature hierarchy

You can now generate a hierarchy of all the features within a feature group that describes the parent-child relationships of the contained features. This enables you to quickly identify issues that prevent a feature group from being machined.

Use the **2D Feature Hierarchy** dialog to view the hierarchy, select features, select parent and child features, and check features for machining validity.

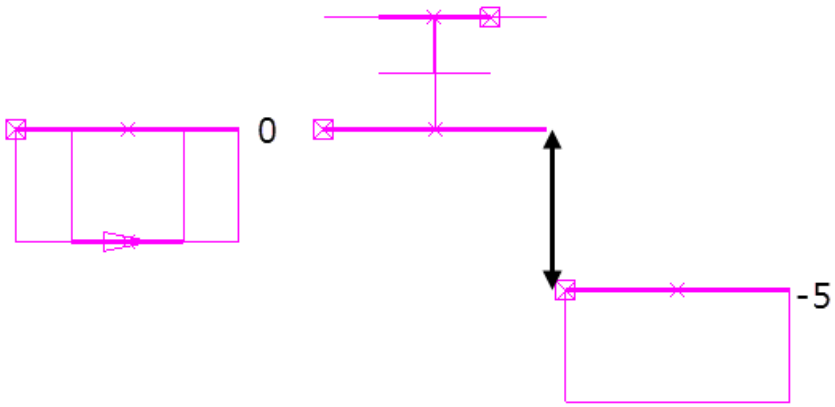
Click  on the **Feature Editor** mode-toolbar to display the **2D Feature Hierarchy** dialog:



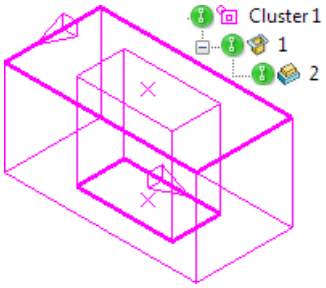
Feature Group — Displays the active feature group.

Hierarchy Status — Displays the update status of the feature hierarchy. This lets you know if the list is still being recalculated after making changes to your features. This is useful if you have a large number of features as the list can take some time to update.

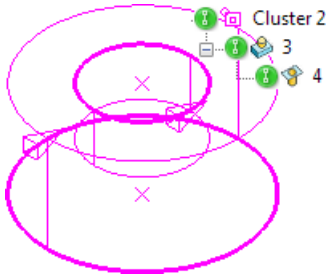
① — Displays all the feature clusters at the specified **Z** value. The **Z** value of the cluster is calculated from the parent feature, the bottom curve for bosses and the top curve for pockets. The position of the clusters along the Z-axis is defined with respect to the feature group workplane. The example below shows two feature clusters at a Z height of 0 and one feature cluster at a Z height of -5.



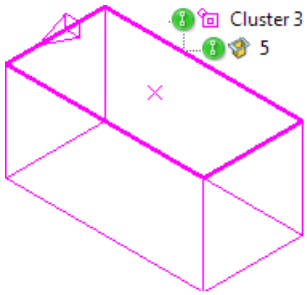
② — Displays the feature clusters and the features included within them. A cluster contains features that are related to one another by their profile curves intersecting. In this example there are three clusters:



A rectangular boss within rectangular pocket.



A circular pocket within a circular boss.



A rectangular pocket.

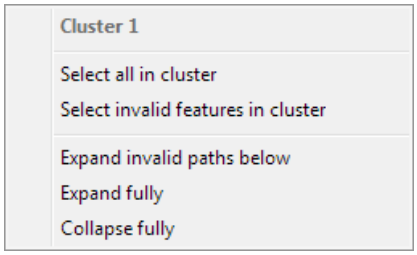
The flags adjacent to the clusters and features describe their validity:

- Valid. All geometry is logical for machining.
- Valid but contains invalid features in the tree.
- Invalid. Contains illogical geometry.

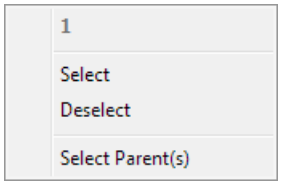
③ — Displays the feature group's validity for machining. If all clusters are valid then **Valid for basic machining** is displayed. Feature area clearance strategies require all 2D features to be aligned at the same Z height. If all the clusters are at the same Z height then **Clusters are Z aligned** is displayed.

Right-click a feature or a cluster in the **2D Feature Hierarchy** dialog to display its context menu. Use the options in the context menus to select features:

Cluster context menu



Feature context menu



Enhancements to features

There are changes and improvements to the creation and modification of features:

- **Creating features** (see page 15) — You can now directly insert rectangular and circular features, in addition to creating freeform features from curves.
- **Editing features** (see page 21) — You can now graphically edit the properties of a feature.
- **Face features** (see page 24) — There are new milling face features.

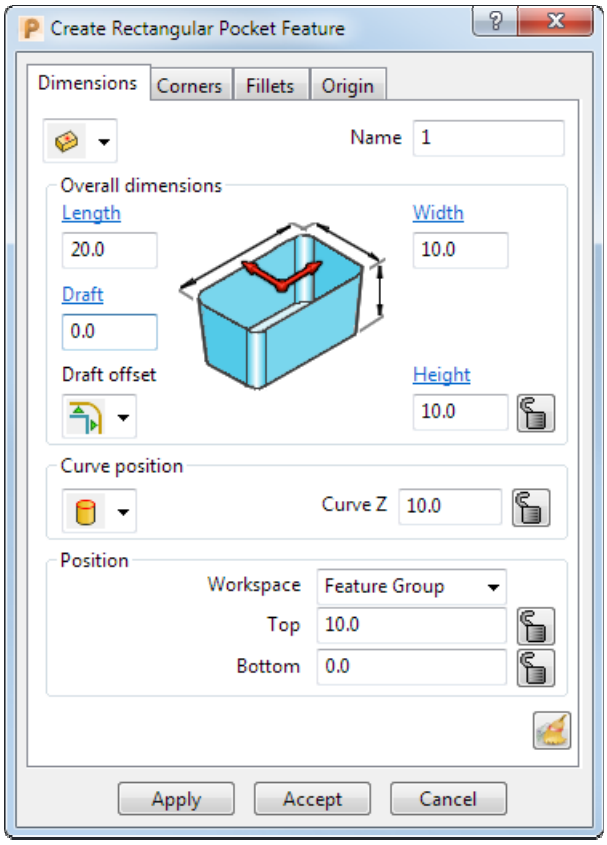
Creating features

You can now easily create features and modify their parameters using the new **Create Feature** dialog. Click a feature creation button on the **Feature Editor** mode-toolbar to display the **Create Feature** dialog.

- **Create rectangular pocket**
- **Create circular pocket**
- **Create freeform pocket**
- **Create rectangular boss**
- **Create circular boss**
- **Create freeform boss**
- **Create slot**

- **Create unbounded face**
- **Create bounded face**

The **Create Features** dialog is displayed, depending on the feature type.



The name of the dialog and the options it contains vary slightly depending on the type of feature.

Use the options on each tab to modify the parameters of your feature:

Dimensions — Use this tab to specify:

- The measurements of the feature.
- The position of the feature curve.
- The position of the feature in the workspace.
- Single creation or multiple creation mode.
- Whether draft edges are offset with sharp or round corners.

Corners — Use this tab to specify:

- The internal corner radii of the feature.

This functionality is only available for rectangular pockets and freeform pockets or bosses.

- The external corner radii of the feature.

  This functionality is only available for rectangular bosses and freeform pockets or bosses.

Fillets — Use this tab to specify:

- The top fillet radius, chamfer or sharp edge.
- The bottom fillet radius or sharp edge.
- Whether fillets and chamfers are offset with sharp or round corners.

  This functionality is only available for pockets, bosses, and slots.

Origin — Use this tab to specify:

- The location of the feature with respect to its origin.
- The position of the feature with respect to the workspace.
- The orientation of the feature in the XY plane.

Creating a rectangular pocket feature example

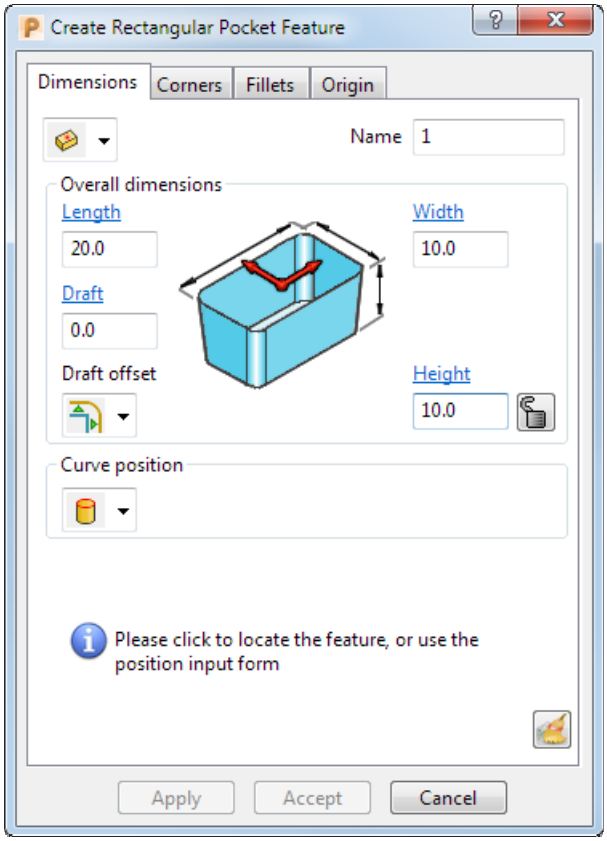
To create a rectangular pocket feature:

1. In the Explorer, right-click **Feature Groups** and select **Feature Editor**.

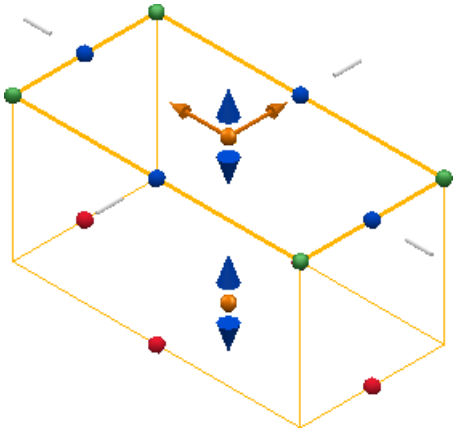
   A feature group is created and the **Feature Editor** mode-toolbar is displayed.

2. Click  **Create rectangular pocket** on the **Feature Editor** mode-toolbar.

The **Create Rectangular Pocket Feature** dialog is displayed.



3 Click in the graphics window to insert the feature.



This functionality is only available for rectangular, circular, and unbounded face features. For freeform, slot and bounded face features you must select an existing curve to create the feature from.

4 Enter a **Name** for the feature.

5 Use the options on the dialog or use the graphic handles to modify the properties of the feature.

The options and graphics handles available vary depending on the type of feature being created.

6 Click **Accept** to create the feature and close the dialog.

7 Click **Accept changes** on the **Feature Editor** mode-toolbar to save the created feature.

Creating a freeform pocket feature example

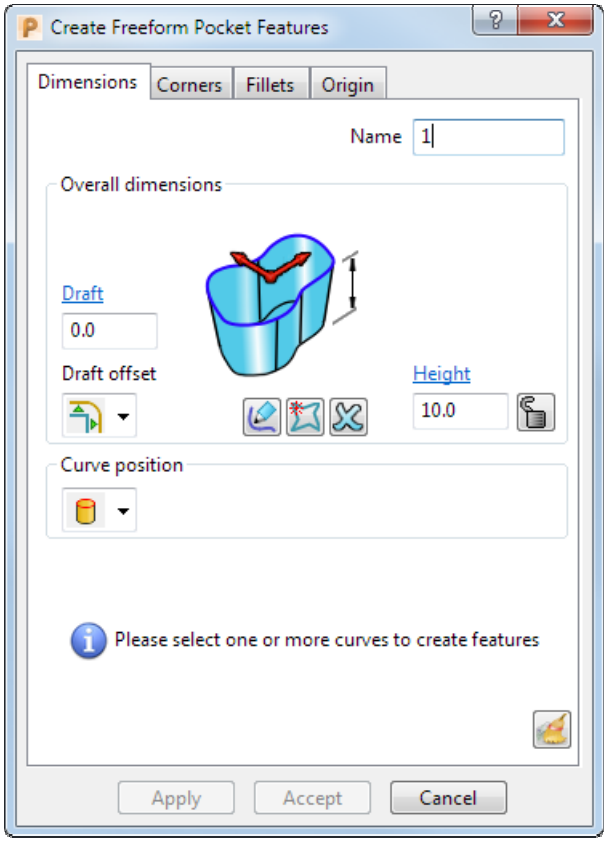
To create a freeform pocket feature:

1 In the Explorer, right-click **Feature Groups** and select **Feature Editor**.

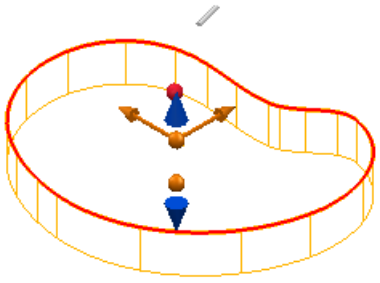
A feature group is created and the **Feature Editor** mode-toolbar is displayed.

2 Click **Create freeform pocket** on the **Feature Editor** mode-toolbar.

The **Create Freeform Pocket Features** dialog is displayed.



3 Select a curve in the graphics window to create the feature from.

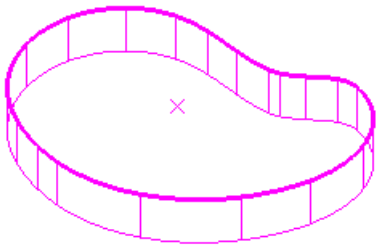


4 Enter a **Name** for the feature.

5 Use the options on the dialog or use the graphic handles to modify the properties of the feature.

The options and graphics handles available vary depending on the type of feature being created.

6 Click **Accept** to create the feature and close the dialog.

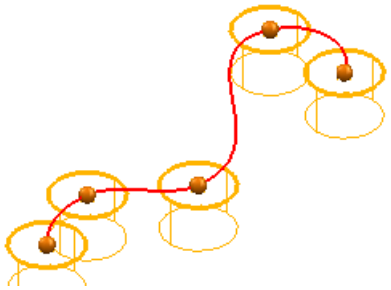


7 Click **Accept changes** on the **Feature Editor** mode-toolbar to save the created feature.

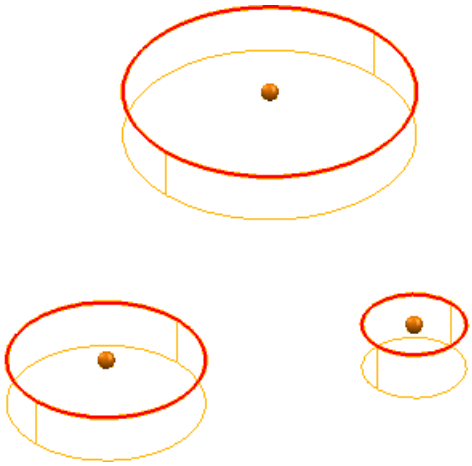
Creating multiple features example

Select an option from the **Creation mode** flyout on the **Create Features** dialog to simultaneously create multiple features:

Multiple creation — Creates a single feature at every key-point of each selected curve.



Curve creation — Detects circular or quadrilateral curves and creates an appropriate feature for each selected curve.



These options are only available for rectangular or circular pockets and bosses.

Editing features

You can now easily edit features using the new **Edit Features** dialog or the new graphic handles.

To edit a feature:

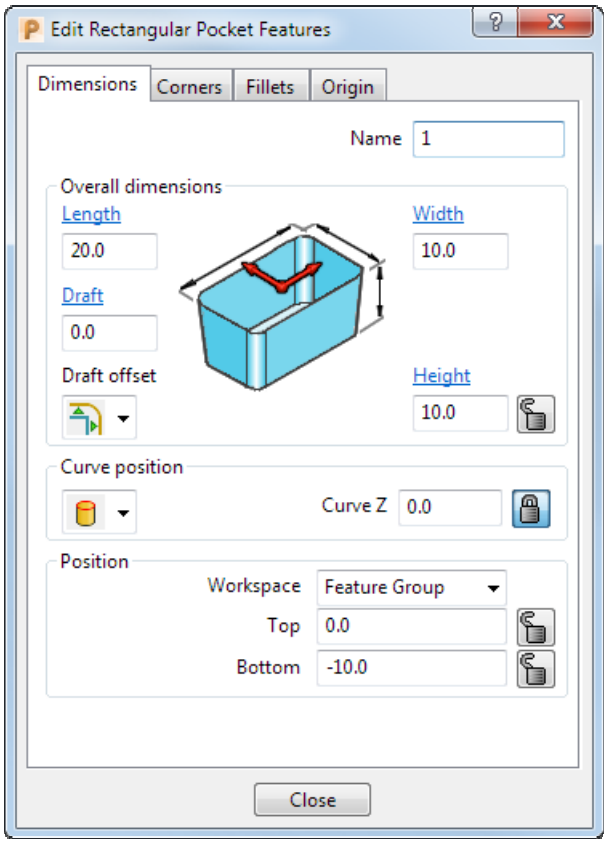
1. In the Explorer, right-click **Feature Groups** and select **Feature Editor**.

   A feature group is created and the **Feature Editor** mode-toolbar is displayed.

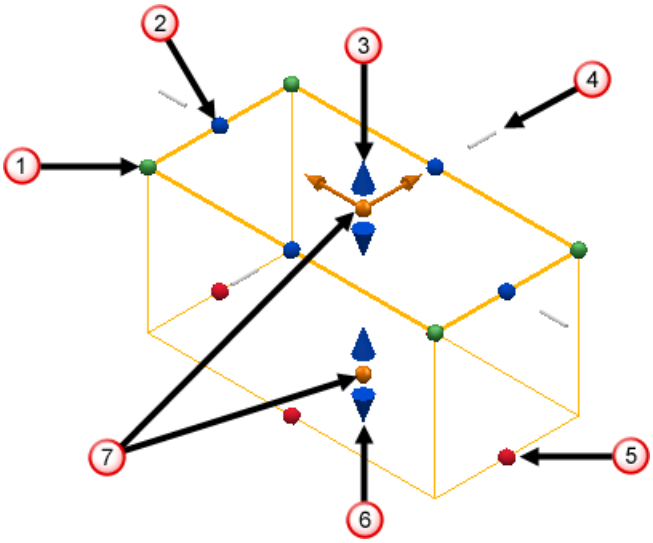
2. Select the feature you want to edit and click . Alternatively you can double click the feature.

The **Edit Features** dialog is displayed, depending on the feature type.

3 For example, selecting a rectangular pocket displays the **Edit Rectangular Pocket Features** dialog.



4 Use the options on the dialog or use the graphic handles to modify the properties of the feature.



1 — Click and drag the green spheres to edit the corner radii of the feature.

2 — Click and drag the blue spheres to edit the length of the feature.

3 — Click and drag the blue arrows to edit the top face position of the feature.

4 — Click and drag the grey bar to edit the profile curve position in the Z axis.

5 — Click and drag the red spheres to edit the draft angle of the feature.

6 — Click and drag the blue arrows to edit the bottom face position of the feature.

7— Click and drag the orange spheres to edit the position of the feature.

The options and graphics handles available vary depending on the type of feature being created.

5 Click **Close** to accept the changes and close the dialog.

6 Click **Accept changes**  on the **Feature Editor** mode-toolbar to save the edited feature.

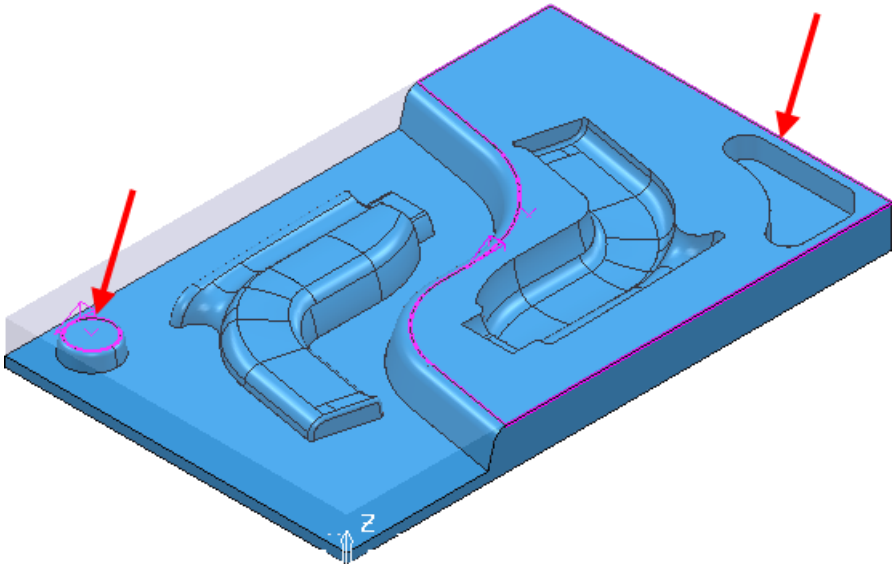
Face features

You can now create 2D face features. Faces are simple features that define a 2D plane to be machined. There are two types of face features:

Bounded

An area defined by a closed curve. All material above and within this area is machined.

The image below displays two bounded face features at different heights on the model.



To create a bounded face feature:

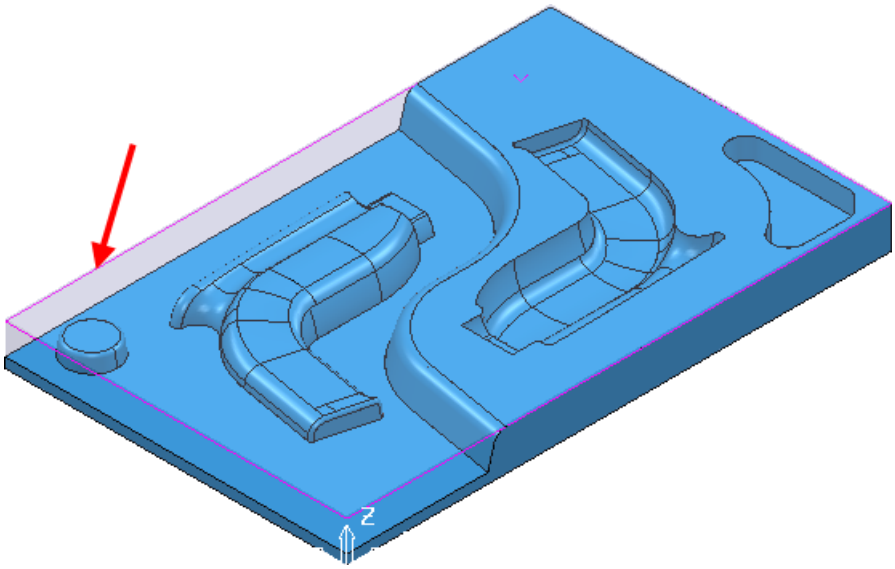
1. Click **Create bounded face** on the **Feature Editor** mode-toolbar.

   The **Create Bounded Face Features** dialog is displayed.

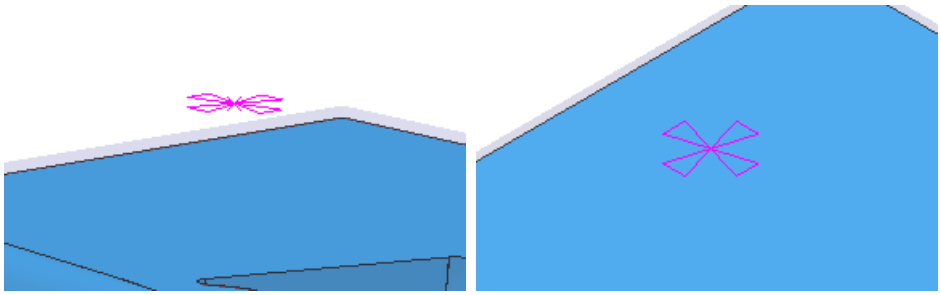
2. Select a curve on the model. If no curve exists click **Edit profile curve** to display the **Curve Editor** mode-toolbar and create a curve.
3. Click **Accept** to create the feature and close the dialog.
4. Click **Accept changes** on the **Feature Editor** toolbar to save the created feature.

Unbounded

All material above this area is machined. The size of the feature is limited by the size of the block.



If the unbounded face feature does not intersect with the block it is represented in the graphics window as follows:



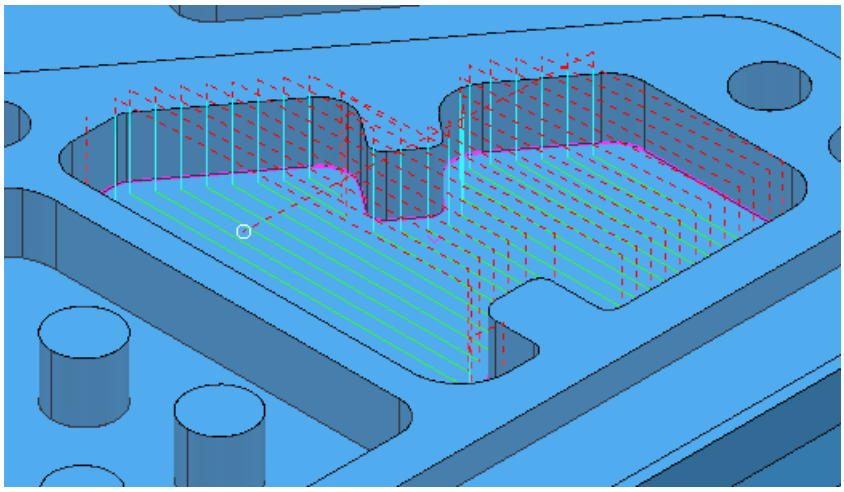
To create an unbounded face feature:

1. Click **Create unbounded face** on the **Feature Editor** mode-toolbar.

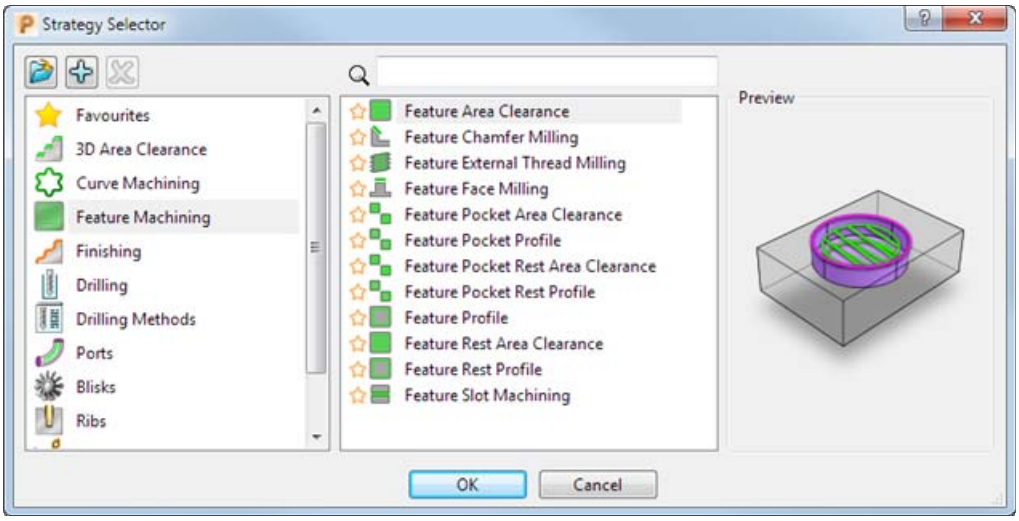
   The **Create Unbounded Face Features** dialog is displayed.
2. Click on the model in the graphics window.
3. Click and drag the graphic handles to adjust the height of the feature.
4. Click **Accept** to create the feature and close the dialog.
5. Click **Accept changes** on the **Feature Editor** toolbar to save the created feature.

2D feature machining strategies

There are new strategies you can use to make 2D feature machining toolpaths.



Use the new **Feature Machining** page of the **Strategy Selector** dialog to select a feature machining strategy.

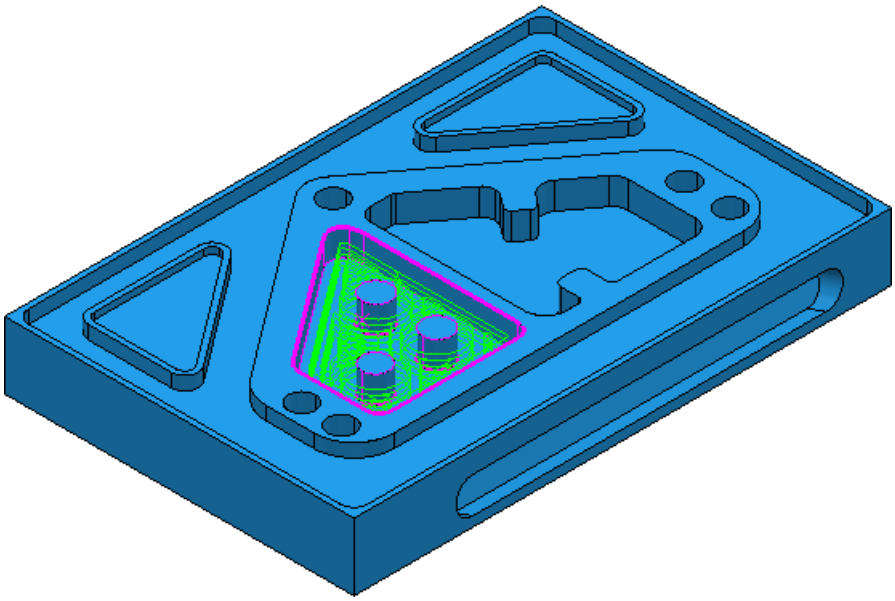


- **Feature Area Clearance (see page 28)** lets you choose between raster and offset styles to remove material from a feature.
- **Feature Chamfer Milling** (see page 32) bevels sharp corners on features using specific chamfer tools.
- **Feature External Thread Milling** (see page 40) creates an external thread on a boss feature.
- **Feature Face Milling** (see page 48) clears flat surfaces specified by bounded or unbounded face features.
- **Feature Pocket Area Clearance** (see page 53) lets you choose between raster and offset styles to remove material from pocket features. You can machine multiple pockets at different Z heights using one strategy.

- **Feature Pocket Profile** (see page 57) machines around the profiles of the pocket features.
- **Feature Pocket Rest Area Clearance** (see page 61) adds rest machining options to the **Feature Pocket Area Clearance** strategy.
- **Feature Pocket Rest Profile** (see page 66) adds rest machining options to the **Feature Pocket Profile** strategy.
- **Feature Profile** (see page 71) machines a profile around the feature at each Z height.
- **Feature Rest Area Clearance** (see page 75) adds rest machining options to the **Feature Area Clearance** strategy.
- **Feature Rest Profile** (see page 81) adds rest machining options to the **Feature Profile** strategy.
- **Feature Slot Machining** (see page 86) — machines slot features in a model.

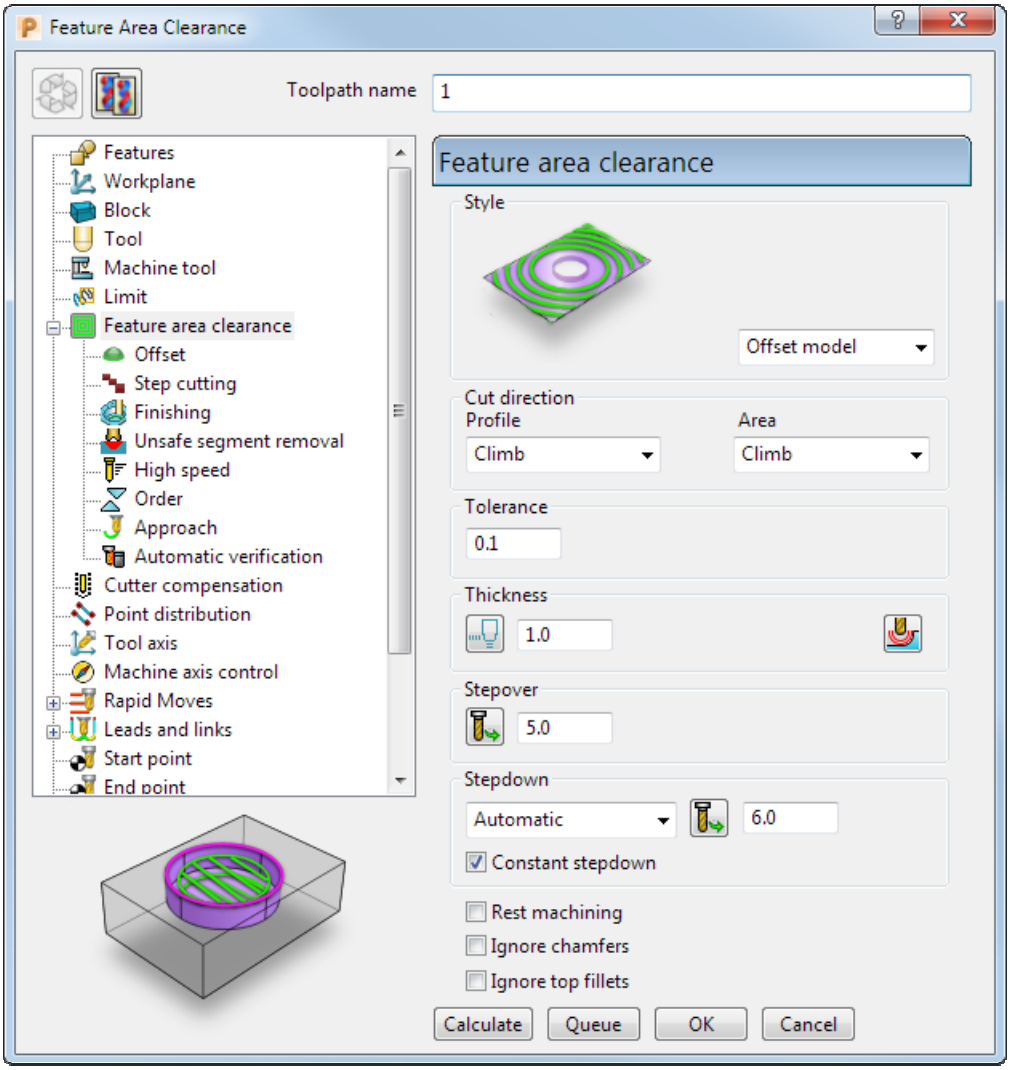
Feature Area Clearance Overview

Use the **Feature Area Clearance** strategy to rapidly remove material from a 2.5D part. Offset toolpaths work well in the bottom of pockets whereas raster toolpaths are often used on open parts. Vortex machining enables you to increase the feed rate whilst maintaining surface quality and tool life.



Feature area clearance machines the active feature group.

There are several pages associated with the **Feature Area Clearance** strategy:



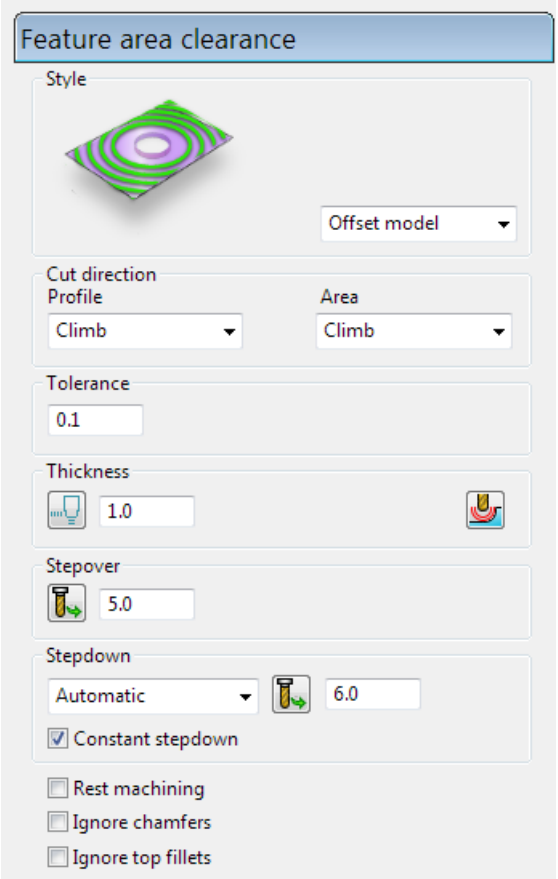
- **Feature area clearance** (see page 30) — The main page used to choose the area clearance styles and associated settings.
- **Raster** — Settings to define a raster area clearance style. This page is available when you select a **Style** of **Raster** on the main page.
- **Offset** — Settings to define offset area clearance styles. This page is available when you select a **Style** of **Offset model** or **Offset all** on the main page.
- **Vortex** — Settings to define a Vortex area clearance style. This page is available when you select a **Style** of **Vortex** on the main page.
- **Step cutting** — Settings to define in-line rest roughing. This minimises terracing when creating area clearance toolpaths with a large stepdown. This is available when you select a **Stepdown** of **Automatic**.

- **Finishing** — Settings to perform a **Final Stepover** and a **Final Stepdown** which are different from the normal stepover and stepdown.
- **Unsafe segment removal** — Settings to remove small toolpath segments.
- **High speed** — Settings for the smoothing options to avoid sharp changes in tool direction when high speed machining.
- **Order** — Settings to control the order of machining.
- **Approach** — Settings to control how the tool approaches the path.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

Feature area clearance

Use the **Feature area clearance** page to create a toolpath by slicing the features at specified Z heights and then create an offset or raster pass at each Z height.



Style — Select the raster, offset, or Vortex style to use for removing material.

Cut direction — Select a milling style for **Profile** and **Area**.

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

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

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

Stepover — Enter the distance between successive area clearance passes at a single Z height.

 **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*  *.*

Stepdown — Enter the distance between different machining levels.

 **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.

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

Constant Stepdown — When selected, all the machining levels are equispaced, and the **Stepdown** value is a maximum stepdown. When deselected, the difference between consecutive machining levels is the **Stepdown** value for all levels except the last one which is at the bottom of the block.

*This option is only available if you select a **Stepdown** of **Automatic**. If you select a **Stepdown** of **Manual**, the **Constant Stepdown** option is available on the **Area Clearance Z Heights** dialog.*

Rest machining — Select to change the strategy to **Feature Rest Area Clearance** and make the **Rest** page available with the options for rest machining. This option is not selected by default for this strategy.

Ignore chamfers — When selected the chamfers are not machined.

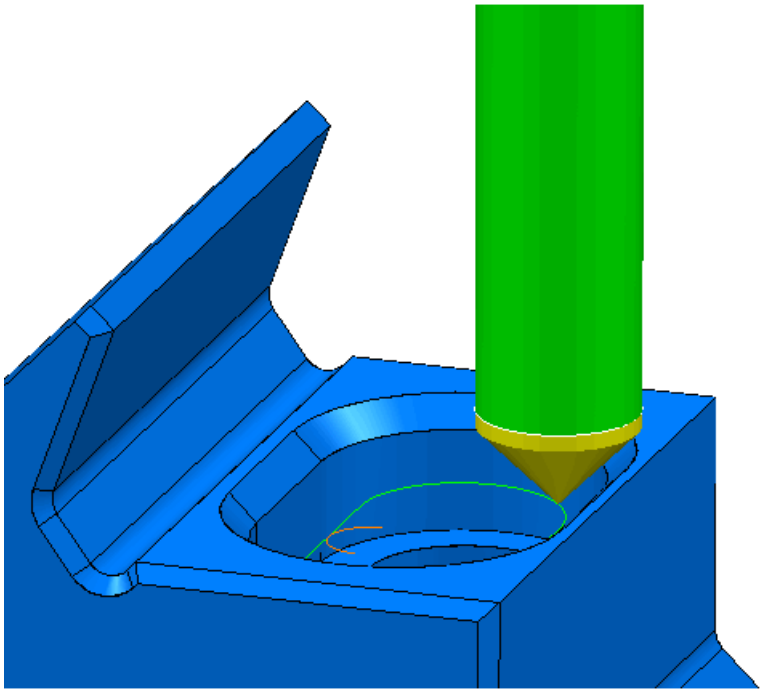
Ignore top fillets — When selected the top fillets are not machined.

Feature Chamfer Milling Overview

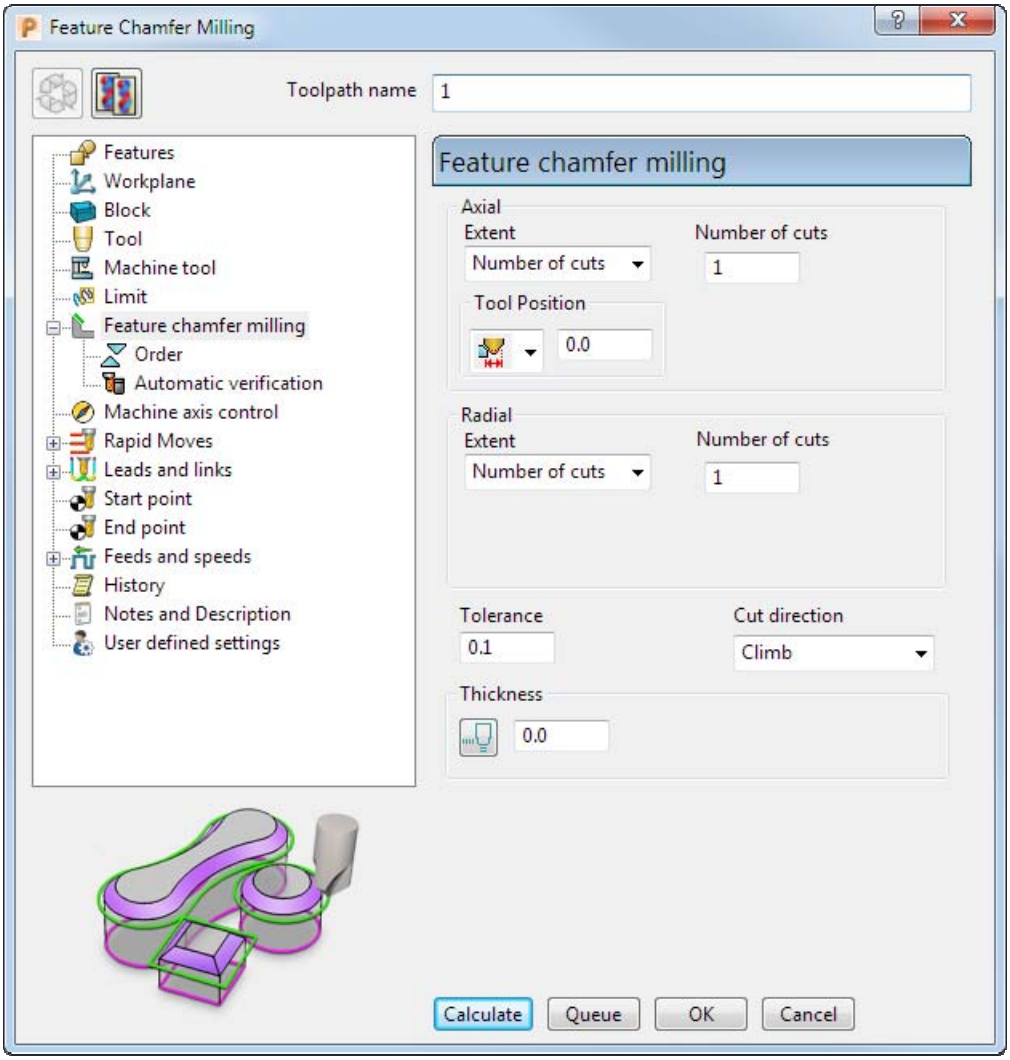
Use the **Feature Chamfer Milling** strategy to bevel sharp corners using specific chamfer tools where the chamfer information is specified by the feature geometry. Feature chamfer milling is used to break sharp outside-edges of a model after it has been completely machined. Feature chamfer milling:

- minimises hand deburring operations
- improves part assembly, by adding a taper to one or both mating edges
- is used for aesthetic and safety reasons.

In many cases feature chamfer milling is used to produce a single finishing pass as chamfer tools are generally larger than the chamfer feature.



There are several pages associated with the **Feature Chamfer Milling** strategy:

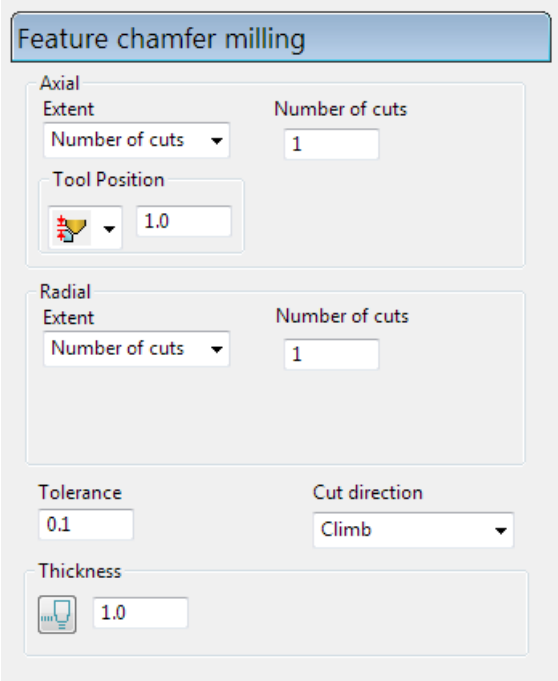


- **Feature chamfer milling** (see page 34) — The main page to specify the options for the feature chamfer milling strategy.
- **Order** — Settings to control the order of machining.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

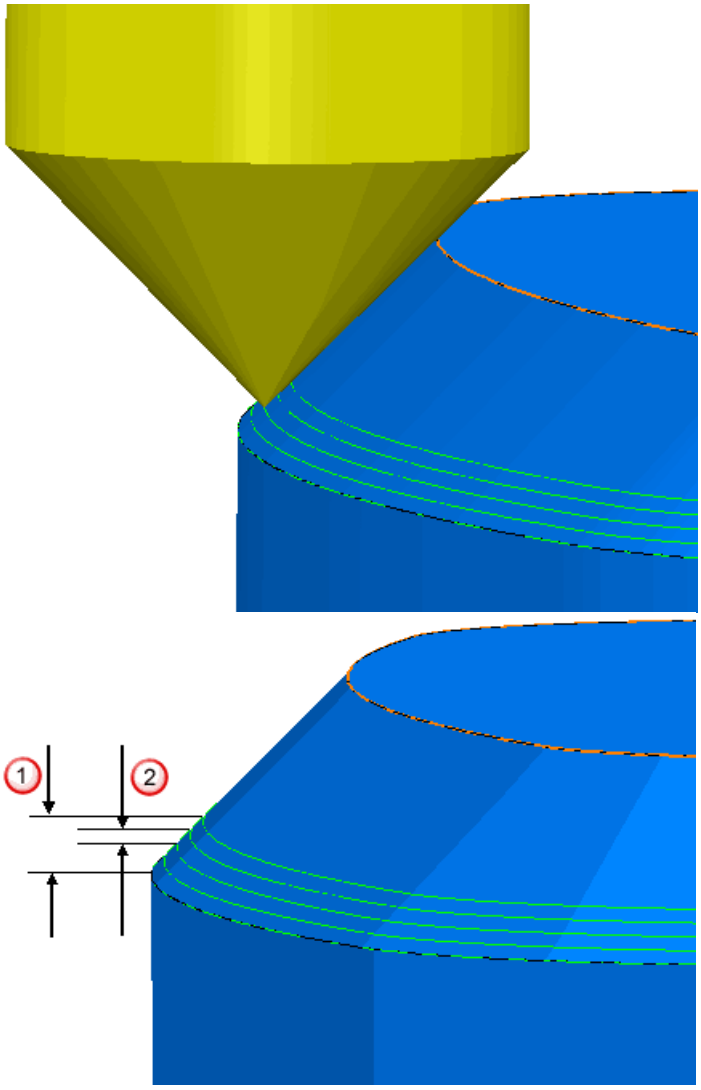
Feature chamfer milling

Use the **Feature chamfer milling** page to specify the cutting range in the axial and radial directions of the tool.



Axial

Multiple cuts are generated along the axial direction.



1 Limit — Defined by the block, the chamfer feature or the number of cuts.

2 Stepdown

Extent — Select how to calculate the limits of multiple passes.

Number of cuts — Enter a number to limit the number of passes.

Chamfer extent — Machine the total height of the chamfer.

Stock extent — Machine the distance between the tool start position and the end of the block.

Extra depth — Specify an extra distance above and below the chamfer for your toolpath range.

*This option is only available for when you select an **Extent** of **Chamfer extent**.*

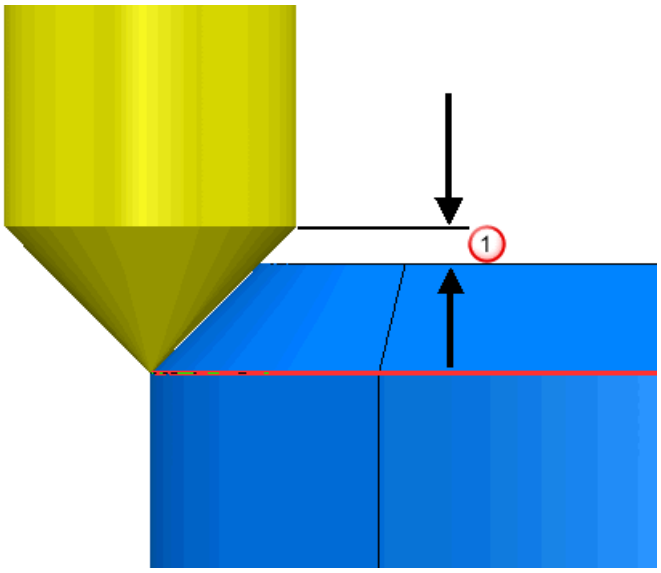
Stepdown — Enter the maximum distance between successive passes.

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.

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

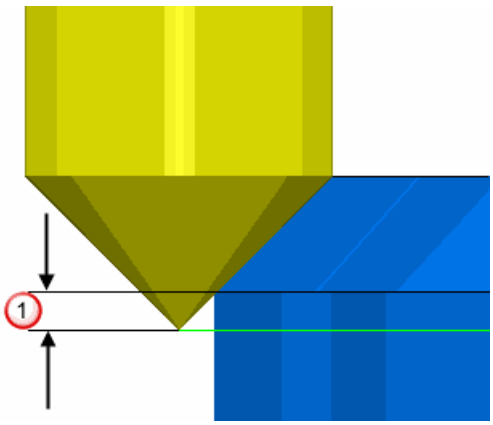
Tool position — Select the position of the tool relative to the chamfer.

Top axial depth — The distance the tool extends above the top of the chamfer.



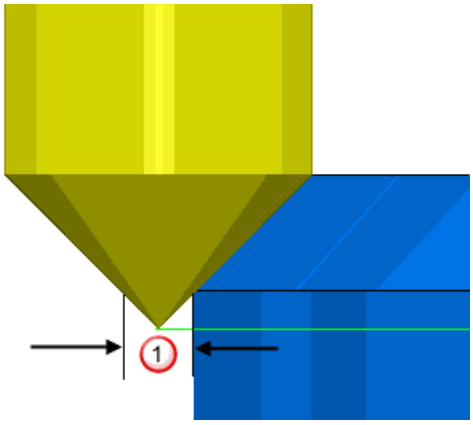
1 — Top axial depth

Bottom axial depth — The distance the tool extends below the bottom of the chamfer.



1 — Bottom axial depth

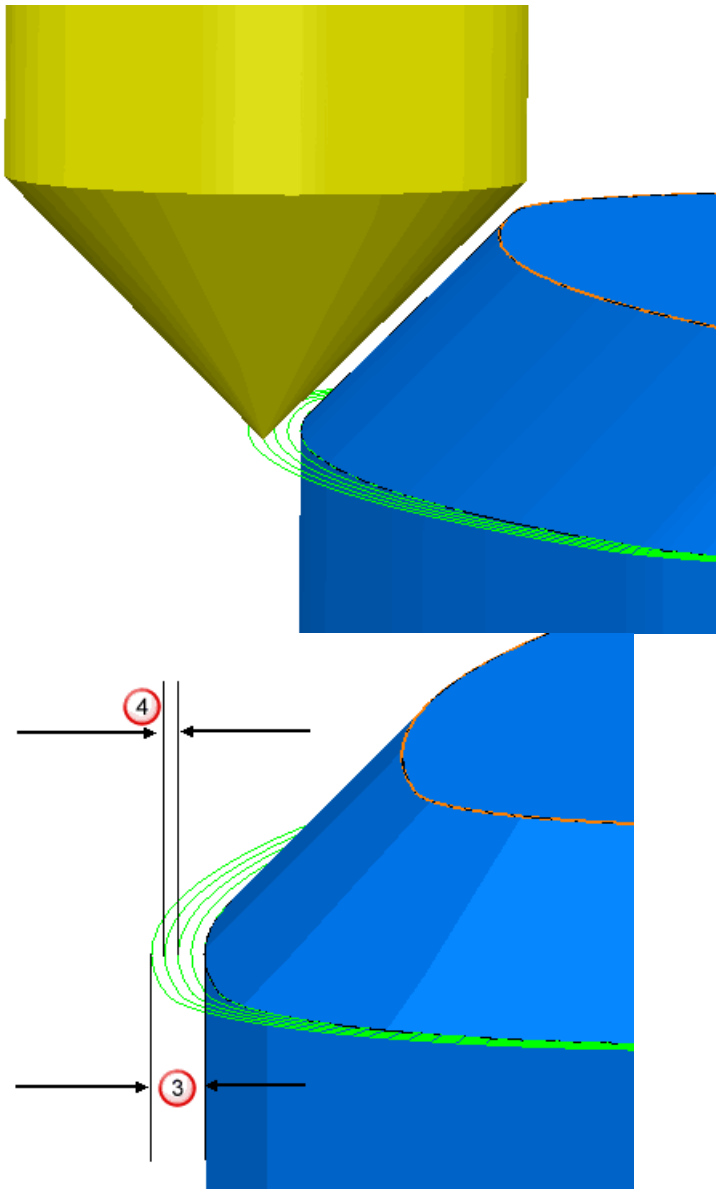
 **Gauge diameter** — The effective tool diameter at the contact point of the tool on the chamfer.



1 — Gauge diameter

Radial

Multiple cuts are generated along the radial direction.



3 Limit — Defined by the block, the chamfer feature or the number of cuts.

4 Stepover

Extent — Select how to calculate the limits of multiple passes.

Number of cuts — Enter a number to limit the number of passes.

Chamfer extent — Machine the total width of the chamfer.

Stock extent — Machine the distance between the tool start position and the end of the block.

Extra width — Specify an extra distance radially around the chamfer for your toolpath range.

*This option is only available for when you select an **Extent** of **Chamfer extent**.*

Stepover — Enter the distance between successive area clearance passes at a single Z height.

 **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  .

*These options are only available when you select an **Extent** of **Chamfer extent** or **Stock extent**.*

Other options

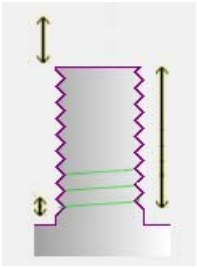
Tolerance — Enter a tolerance value to determine how accurately the toolpath follows the feature.

Cut direction — Select the milling technology.

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

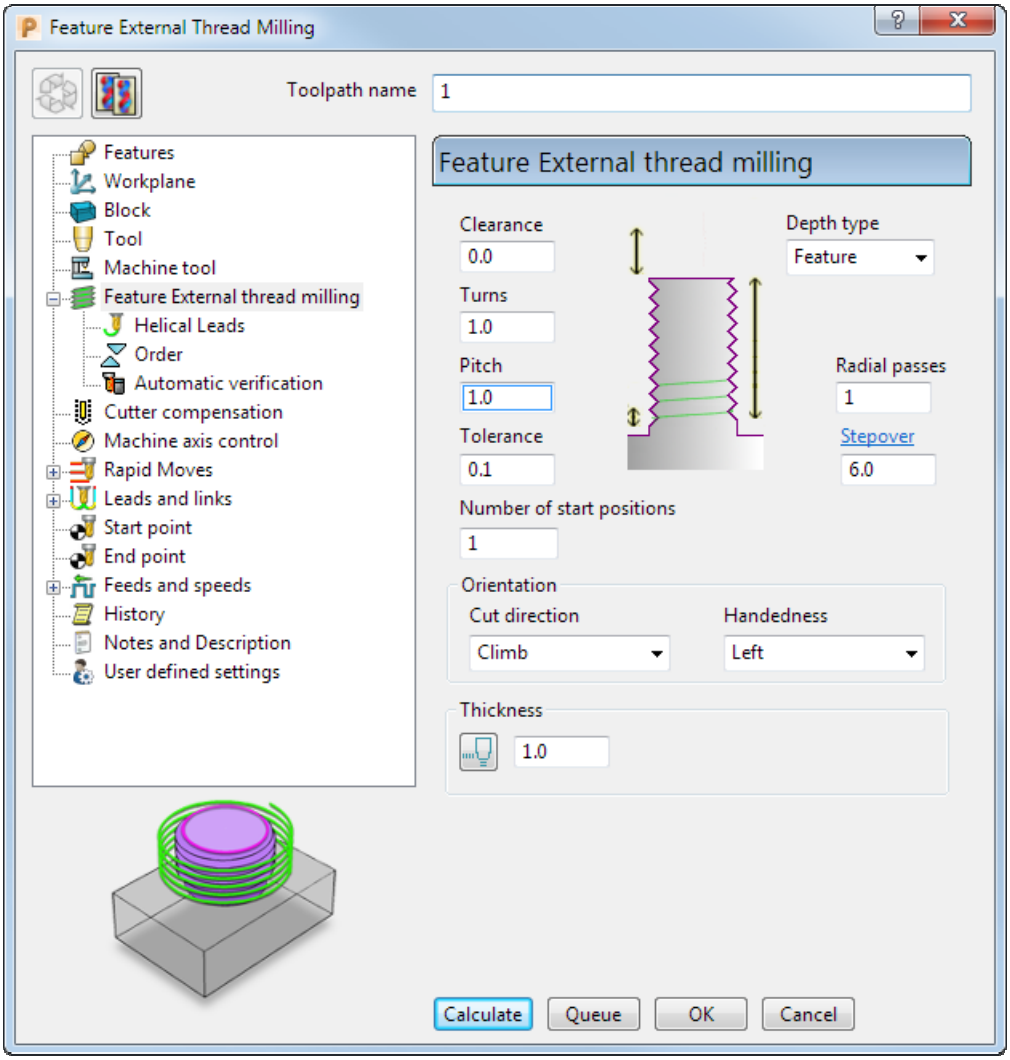
Feature External Thread Milling Overview

Use the **Feature External Thread Milling** strategy to create an external thread on a boss.



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

There are several pages associated with the **Feature external thread milling** strategy:



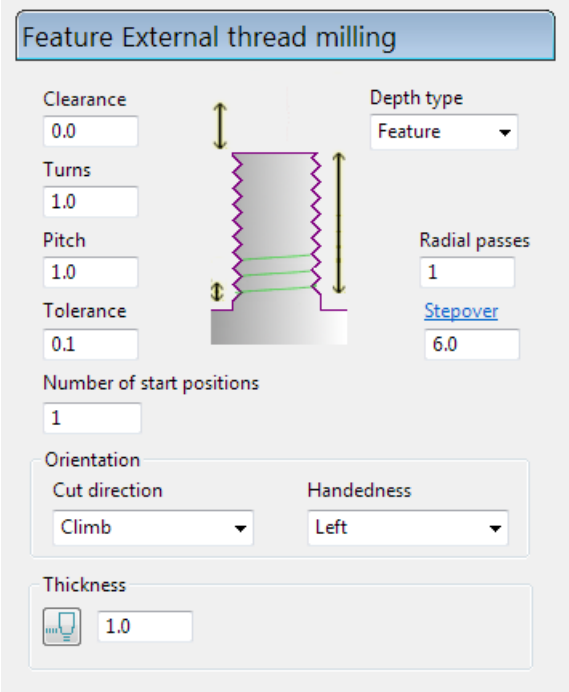
- **Feature external thread milling** (see page 42) — The main page used to define an external thread.
- **Helical leads** (see page 44) — Settings to control the lead angle and the lead radius.
- **Order** — Settings to control the order of machining.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

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

The common tabs are described in common toolpath creation controls.

Feature external thread milling

Use the **Feature external thread milling** page to create an external thread.



Depth type — Select how to determine the maximum thread depth.

Feature — The thread starts at the base of the boss. This disables the **Depth** field as PowerMill calculates this value.

User defined — The thread starts at a distance below the top of the boss. Enter this distance in the **Depth** field.

Depth — Enter the maximum thread depth. This option is only available if you select a **Depth type** of **User defined** .If you select a **Depth type** of **Feature**, PowerMill calculates this values for you.

Clearance — Enter the distance above the top of the boss. By default, this is the same as the **Incremental start Z**.

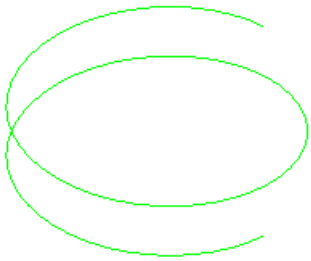
Turns — Enter the number of turns of the thread milling tool. Tall bosses may need more than one turn.

Pitch — Enter the distance from one thread groove to the next.

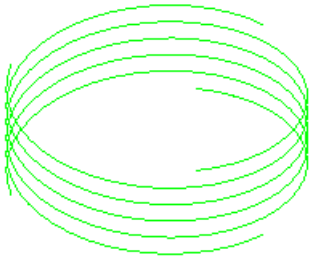
Tolerance — Enter a tolerance value to determine how accurately the toolpath follows the boss.

Number of start positions — Enter a value to create identical toolpaths that are rotated about the tool axis and spaced equidistantly. This enables you to generate intertwined threads.

Toolpath with one start position

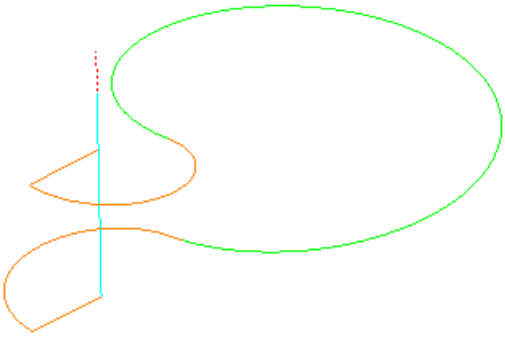


Toolpath with three start positions

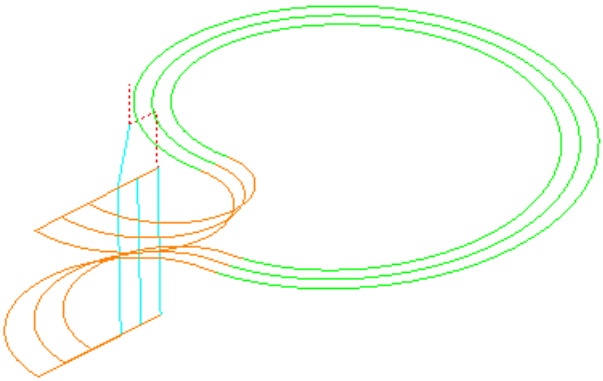


Radial passes— Enter the number of radially spaced helical toolpaths.

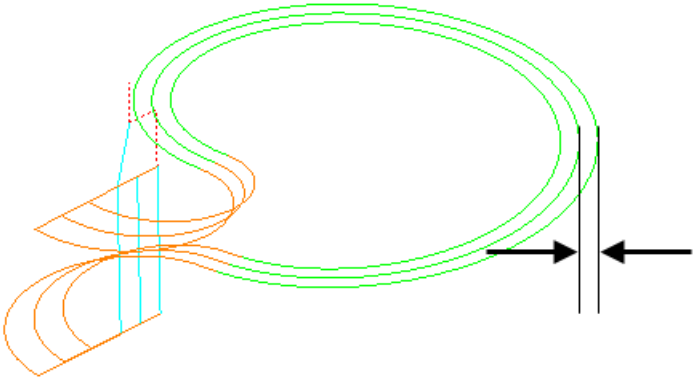
One pass:



Three passes:



Stepover — Enter the distance between successive passes.



Cut Direction — Select the milling technology. The combination of **Cut Direction** and **Handedness** determines the cut direction.

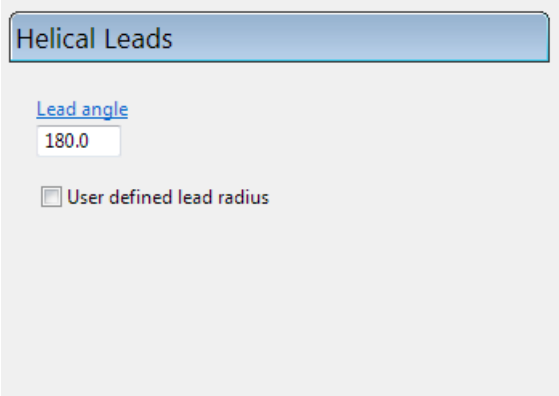
	Right hand thread	Left hand thread
Climb	Clockwise, downwards	Clockwise, upwards
Conventional	Anti-clockwise, upwards	Anti-clockwise, downwards

Handedness — Select the rotational direction of the thread as it moves in the positive Z direction.

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

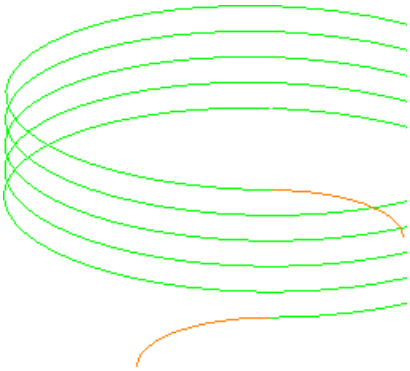
Helical leads

Use the **Helical leads** page to specify the arc angle and radius of the toolpath leads.

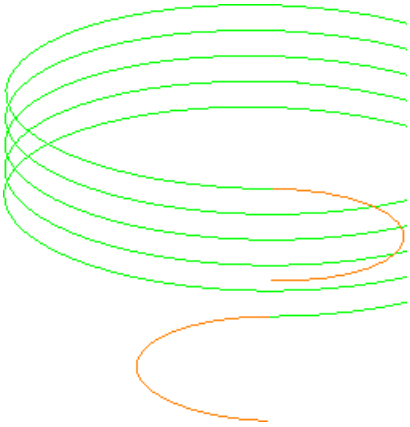


Lead angle — Enter a value to specify the arc angle of the toolpath leads. This angle must be greater than 5°.

Lead angle at 90°



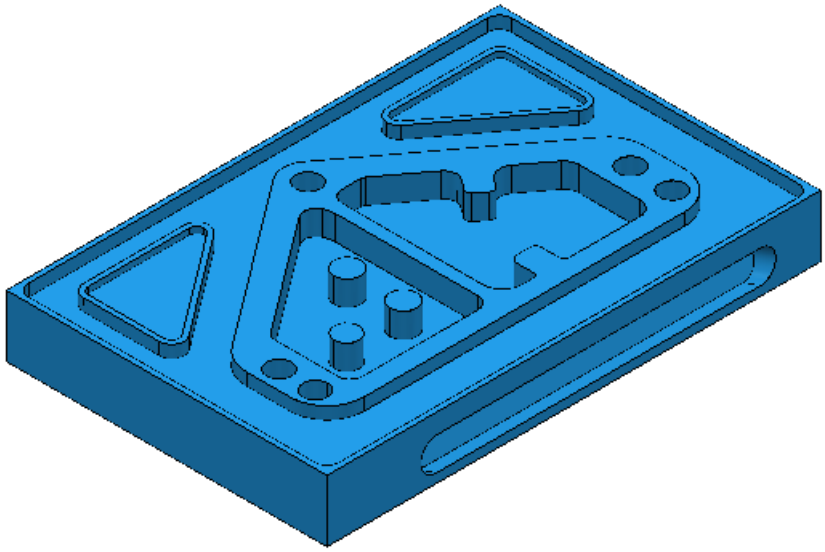
Lead angle at 180°



User defined lead radius — Select this option to manually specify the arc radius of the toolpath leads.

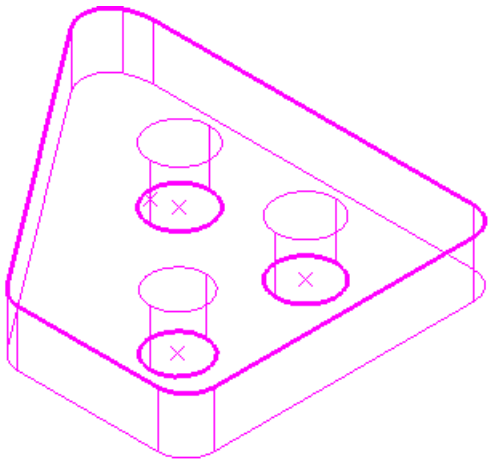
Creating an external thread

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



1 Create a feature group containing the three bosses.

You can create this as a pocket with three bosses.

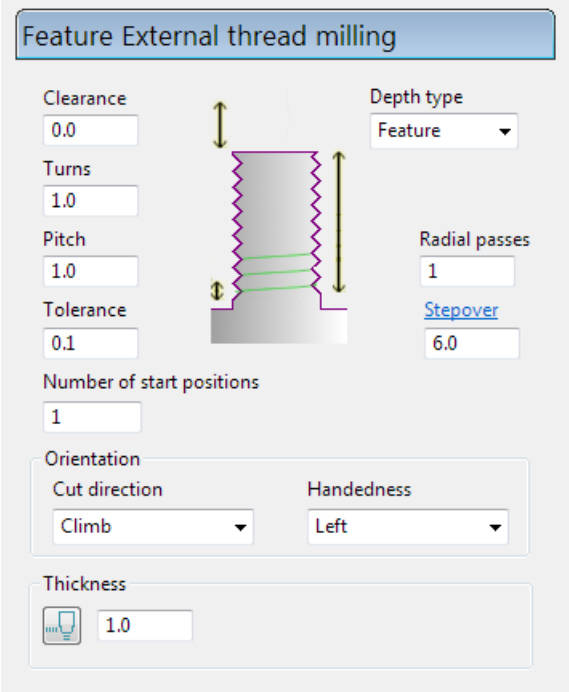


2 Calculate the block and create a thread mill tool.

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

4 Select the **Feature Machining** tab, followed by the **Feature External Thread Milling** option.

5 On the **Feature external thread milling** page:



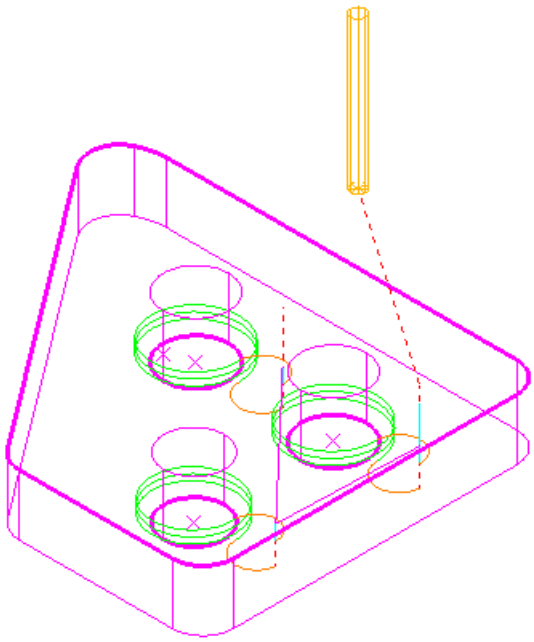
a Select a **Depth Type** of **Feature**.

b Enter a **Clearance** of **0**.

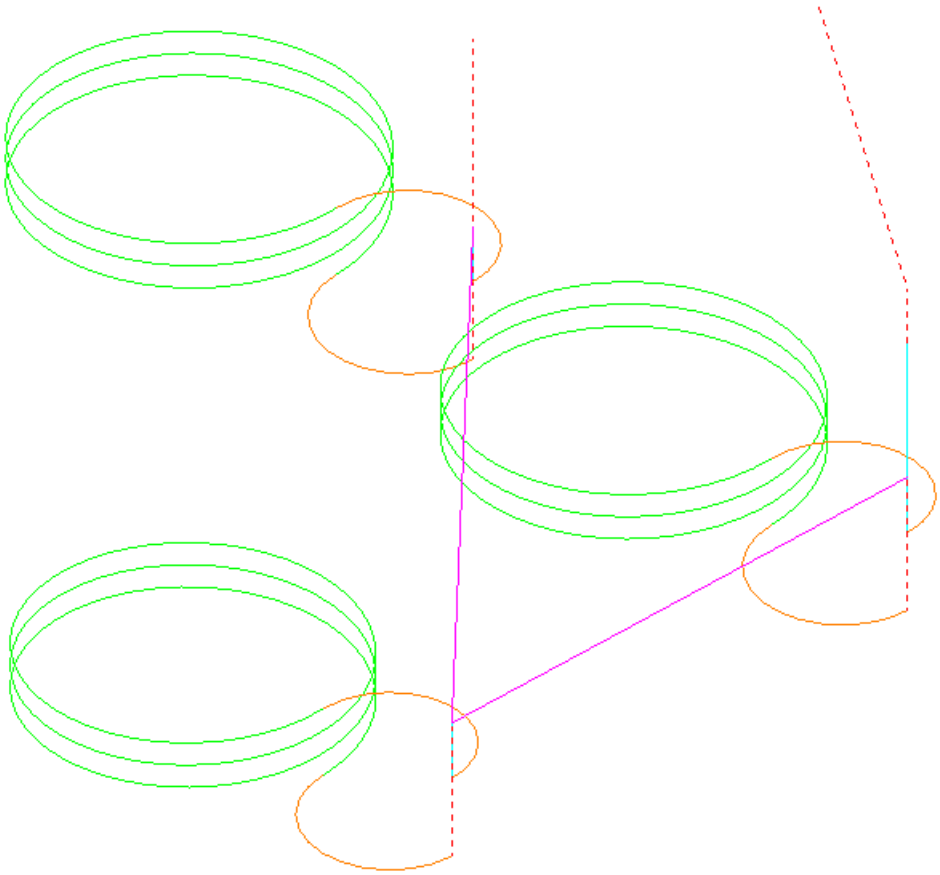
c Enter a **Turns** of **3**.

d Enter a **Pitch** of **2**.

e Click **Calculate**.



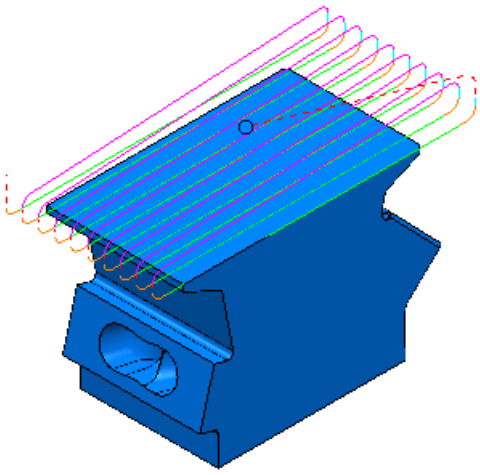
Looking in detail:



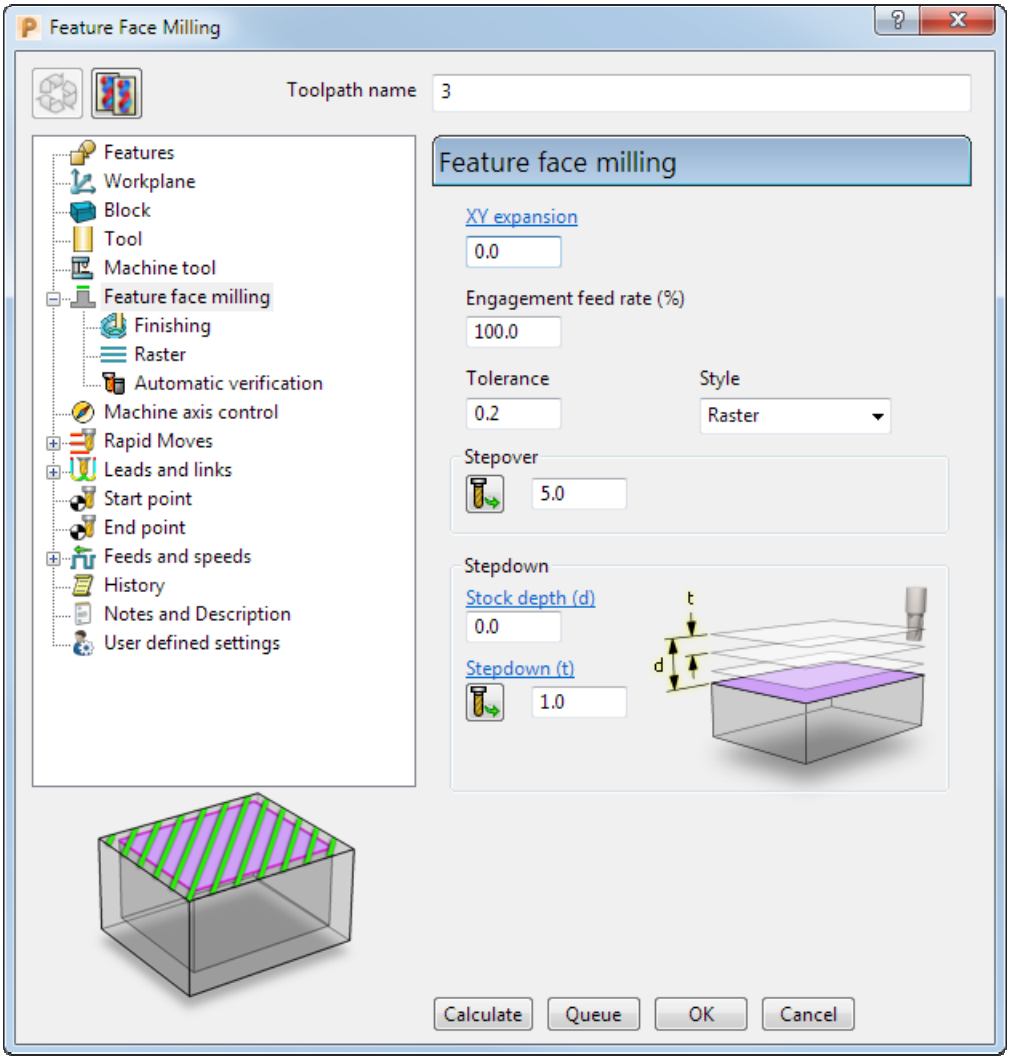
Feature Face Milling Overview

Use the **Feature Face Milling** strategy to produce flat surfaces by guiding a large cutter over the surface of the workpiece specified by bounded or unbounded face features. This strategy enables you to machine multiple face features at different Z heights.

The cutter is a Face Mill consisting of a rotating holder containing turning inserts. The cutter spindle always has an axis of rotation perpendicular to the work piece surface. The geometry of the cutter only enables relatively small depths of cut. Face milling is commonly used to machine flat surfaces on ground vehicle powertrain components, engine blocks, and transmission valve bodies.



There are several pages associated with the **Feature Face Milling** strategy:



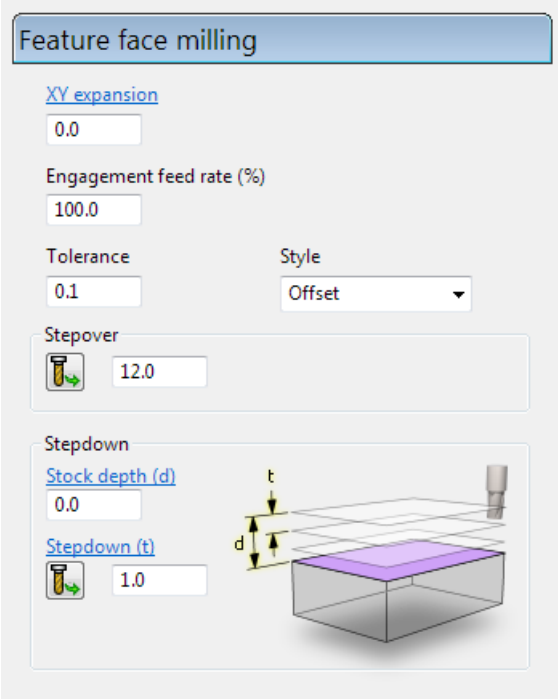
- **Feature face milling** (see page 50) — The main page used to specify a face milling toolpath.
- **Finishing** — Setting to specify a final stepdown depth of cut value.
- **Raster** — Settings to specify a raster area clearance style.
- **Offset** — Settings to specify an offset area clearance style
- **High speed** — Settings to control smoothing options to avoid sharp changes in tool direction when high speed machining. This page is only available for offset area clearance styles.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

To create a **Face Milling** toolpath you must define a block and a tool. Face milling works with any tool and uses the flat end of the tool.

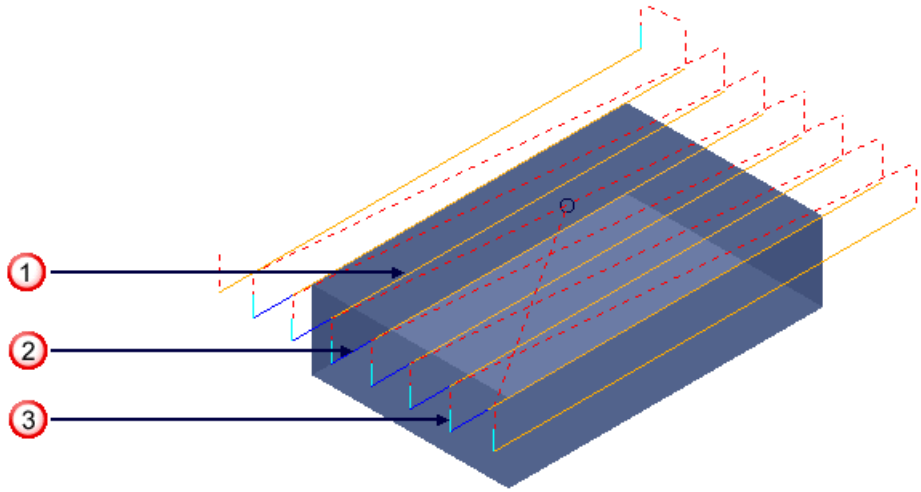
Feature face milling

Use the **Feature face milling** page to specify the feature face machining parameters.



XY expansion — Enter an allowance in the X and Y directions on the block. This is useful where the actual stock is larger than the PowerMill block as it ensures that the face milling toolpath machines all of the stock.

Engagement feed rate (%) — Enter a value to control the feed rate of the toolpath, as it approaches the block, until it fully engages with the block. This is useful to prevent tool inserts from breaking when they plunge into the model with high feed rate. By default this is set to 100% so there is no engagement feed rate reduction.



1 - Cutting feed rate

2 - Engagement feed rate

3 - Plunging feed rate

Any lead in applied to the toolpath will also acquire the engagement feed rate.

Tolerance — Enter a value to determine how closely the toolpath follows the feature.

Style — Select the **Raster** or **Offset** style to use for removing material.

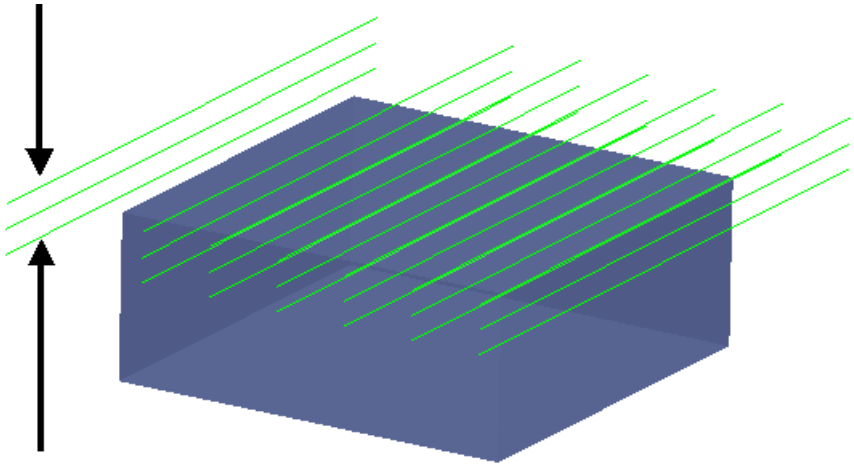
Stepover — Enter the distance between successive machining passes.

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.

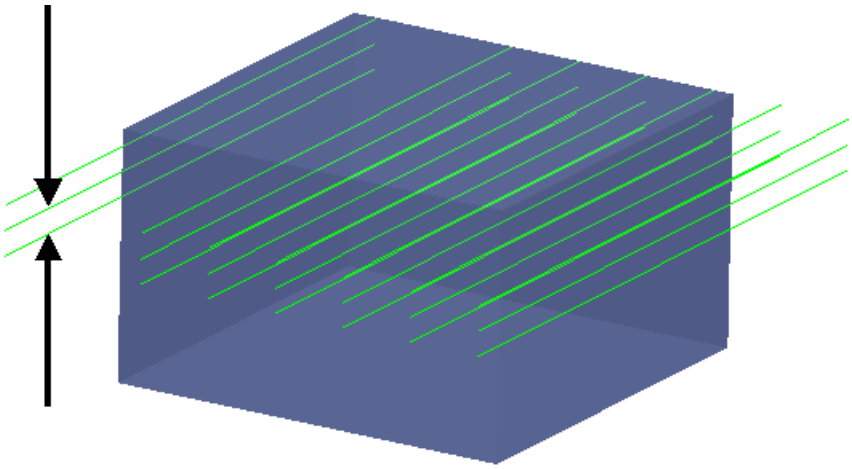
If you enter the value manually, the button changes to .

Stepdown

Stock depth — Enter the total depth of material to remove.



Stepdown — Enter the maximum distance between successive passes.

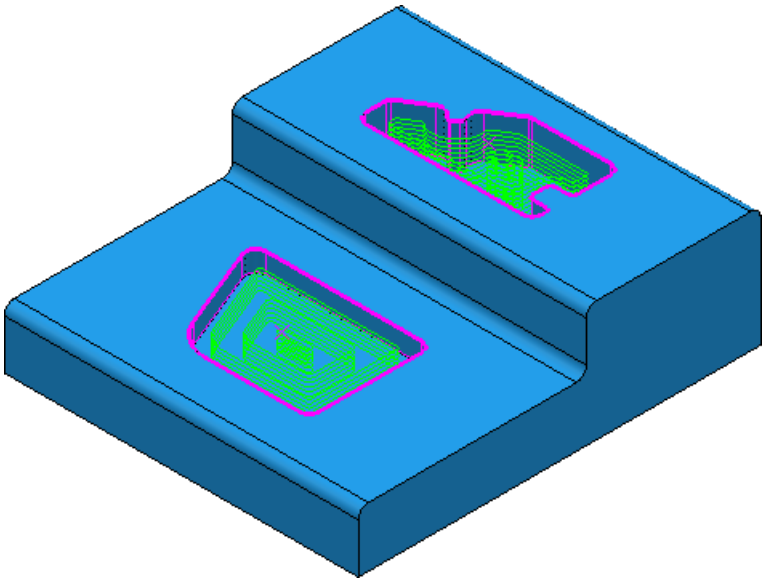


Copy stepover from tool — Click to load the radial depth of cut from the active tool's cutting data. The radial depth of cut is measured normal to the tool axis.

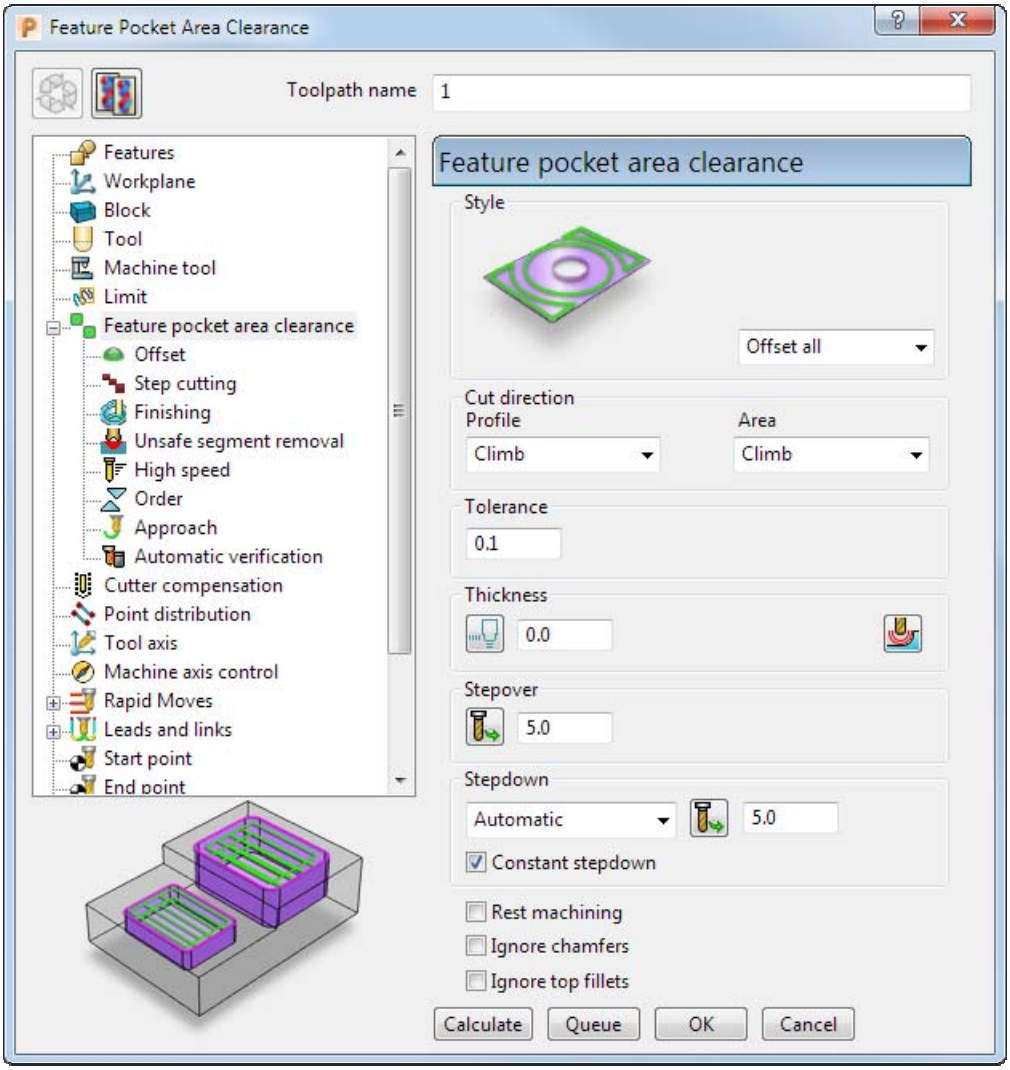
If you enter the value manually, the button changes to .

Feature Pocket Area Clearance Overview

Use the **Feature Pocket Area Clearance** strategy to efficiently remove large volumes of material for individual pocket features using one strategy. This strategy enables you to machine multiple pockets at different Z levels.



There are several pages associated with the **Feature Pocket Area Clearance** strategy:



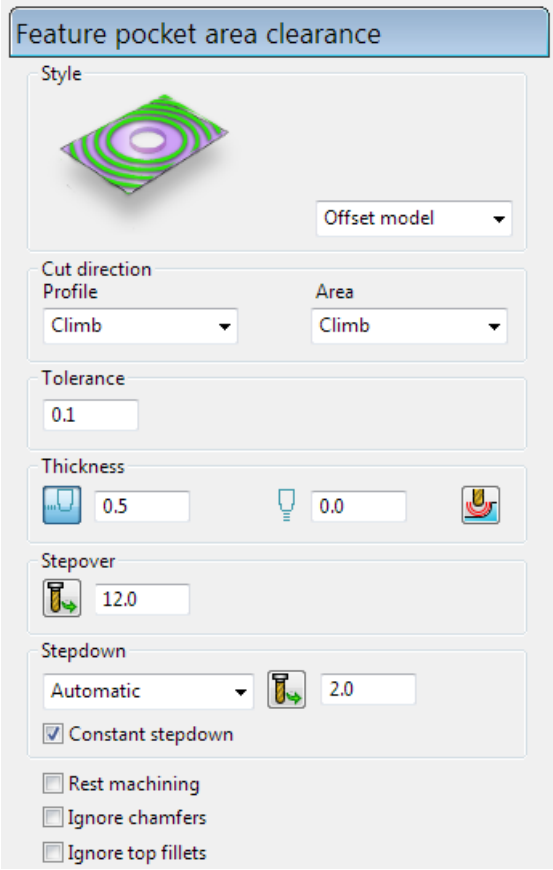
- **Feature pocket area clearance** (see page 55) — The main page used to specify a feature pocket area clearance toolpath.
- **Raster** — Settings to define a raster area clearance style. This page is available when you select a **Style** of **Raster** on the main page.
- **Offset** — Settings to define offset area clearance styles. This page is available when you select a **Style** of **Offset model** or **Offset all** on the main page.
- **Vortex** — Settings to define a Vortex area clearance style. This page is available when you select a **Style** of **Vortex** on the main page.
- **Step cutting** — Settings to define in-line rest roughing. This minimises terracing when creating area clearance toolpaths with a large stepdown. This is available when you select a **Stepdown** of **Automatic**.

- **Finishing** — Settings to perform a **Final Stepover** and a **Final Stepdown** which are different from the normal stepover and stepdown.
- **Unsafe segment removal** — Settings to remove small toolpath segments.
- **High speed** — Settings to control smoothing options to avoid sharp changes in tool direction when high speed machining. This page is only available for offset area clearance styles.
- **Order** — Settings to control the order of machining.
- **Approach** — Settings to control how the tool approaches the path.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

Feature pocket area clearance

Use the **Feature pocket area clearance** page to create a toolpath by slicing the pocket features at specified Z heights and then creates an offset or raster pass at each Z height. You can machine multiple pockets at different Z heights using one strategy.



Style — Select the raster, offset, or Vortex style to use for removing material.

Cut direction — Select a milling style for **Profile** and **Area**.

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

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

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

Stepover — Enter the distance between successive area clearance passes at a single Z height.

 **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*  *.*

Stepdown — Enter the distance between different machining levels.

 **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.

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

Constant Stepdown — When selected, all the machining levels are equispaced, and the **Stepdown** value is a maximum stepdown. When deselected, the difference between consecutive machining levels is the **Stepdown** value for all levels except the last one which is at the bottom of the block.

This option is only available if you select a **Stepdown** of **Automatic**. If you select a **Stepdown** of **Manual**, the **Constant Stepdown** option is available on the **Area Clearance Z Heights** dialog.

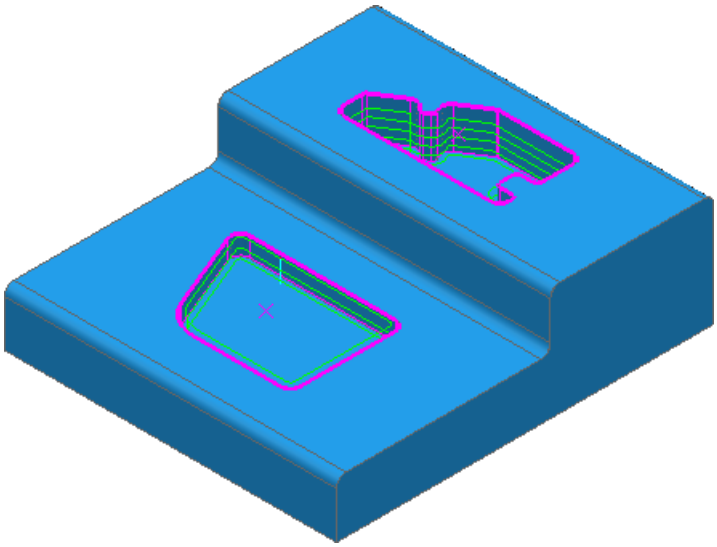
Rest machining — Select to change the strategy to **Feature pocket rest area clearance** and make the **Rest** page available with the options for rest machining. This option is not selected by default for this strategy.

Ignore chamfers — When selected the chamfers are not machined.

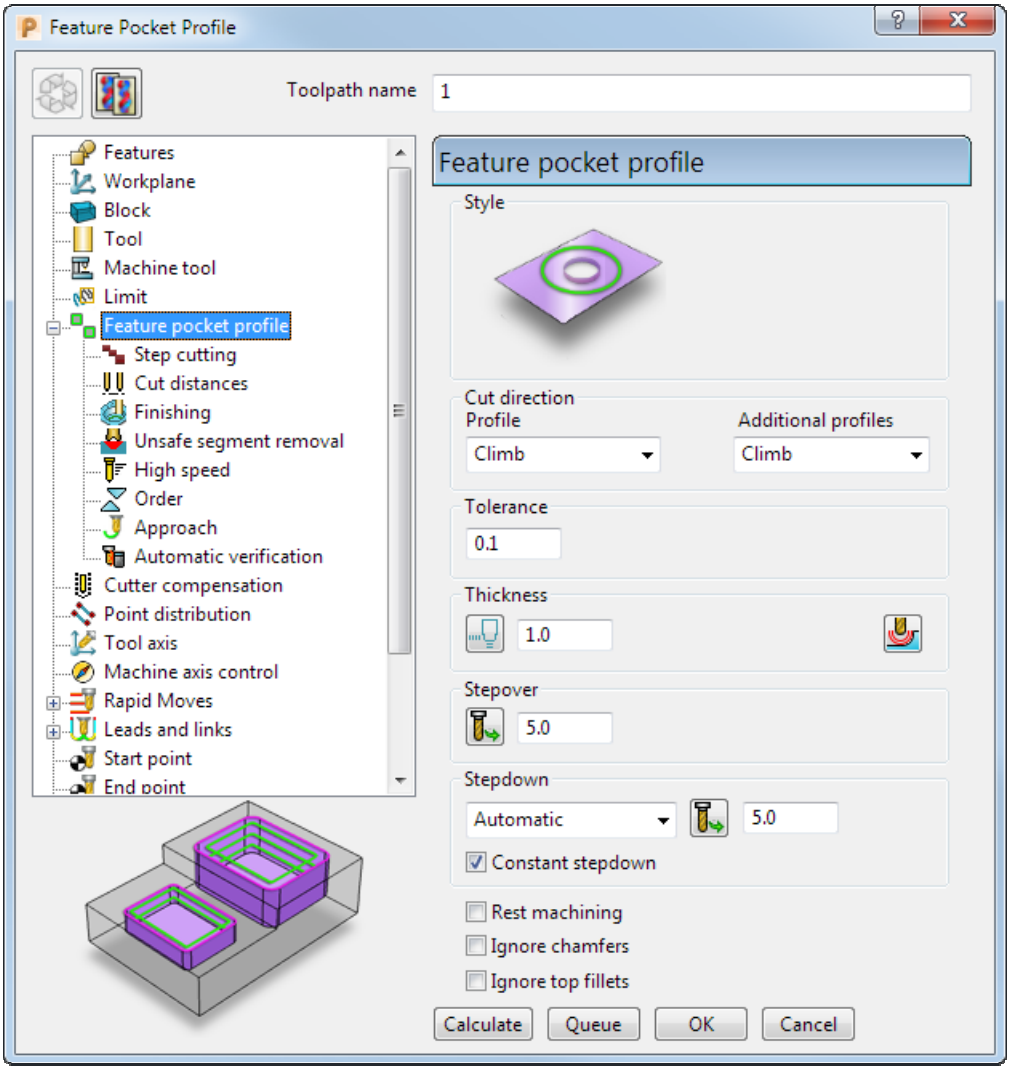
Ignore top fillets — When selected the top fillets are not machined.

Feature Pocket Profile Overview

Use the **Feature Pocket Profile** strategy to create a simple toolpath that slices the pocket features at specified Z heights and then machines the pocket features profiles at each Z height. This strategy enables you to machine multiple pockets at different Z levels.



There are several pages associated with the **Feature Pocket Profile** strategy:



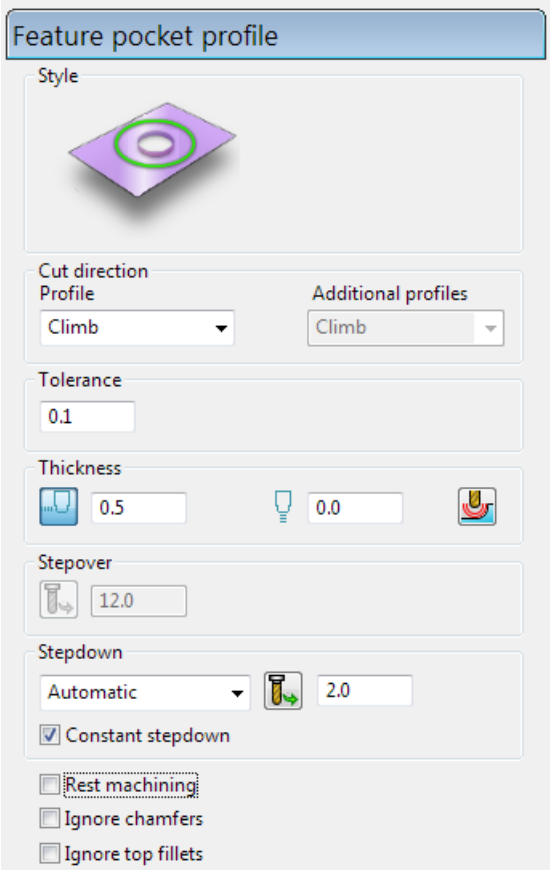
- **Feature pocket profile** (see page 59) — The main page used to specify a feature pocket profile strategy
- **Step cutting** — Settings to define in-line rest roughing. This minimises terracing when creating area clearance toolpaths with a large stepdown. This is available when you select a **Stepdown** of **Automatic**.
- **Cut distances** — Settings to control the number of profile cuts.
- **Finishing** — Settings to perform a **Final Stepover** and a **Final Stepdown** which are different from the normal stepover and stepdown.
- **Unsafe segment removal** — Settings to remove small toolpath segments.
- **High speed** — Settings to control smoothing options to avoid sharp changes in tool direction when high speed machining. This page is only available for offset area clearance styles.

- **Order** — Settings to control the order of machining.
- **Approach** — Settings to control how the tool approaches the path.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

Feature pocket profile

Use the **Feature pocket profile** page to create a toolpath by slicing the pocket features at specified Z heights and then creates a profile pass at each Z height. You can machine multiple pockets at different Z heights using one strategy. The Z heights are defined from the previous area clearance toolpath and are used to eliminate large terraces.



Cut direction — Select the milling technology. When you have several profile passes you can have a different cut direction for the final profile pass.

Profile — Select the cut direction of the final profiling pass.

Additional profiles — Select the cut direction of all passes except the final profiling pass.

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

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

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

Stepover — Enter the distance between successive machining passes.

 **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*  *.*

Stepdown — Enter the distance between different machining levels.

 **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.

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

Constant Stepdown — When selected, all the machining levels are equispaced, and the **Stepdown** value is a maximum stepdown. When deselected, the difference between consecutive machining levels is the **Stepdown** value for all levels except the last one which is at the bottom of the block.

*This option is only available if you select a **Stepdown** of **Automatic**. If you select a **Stepdown** of **Manual**, the **Constant Stepdown** option is available on the **Area Clearance Z Heights** dialog.*

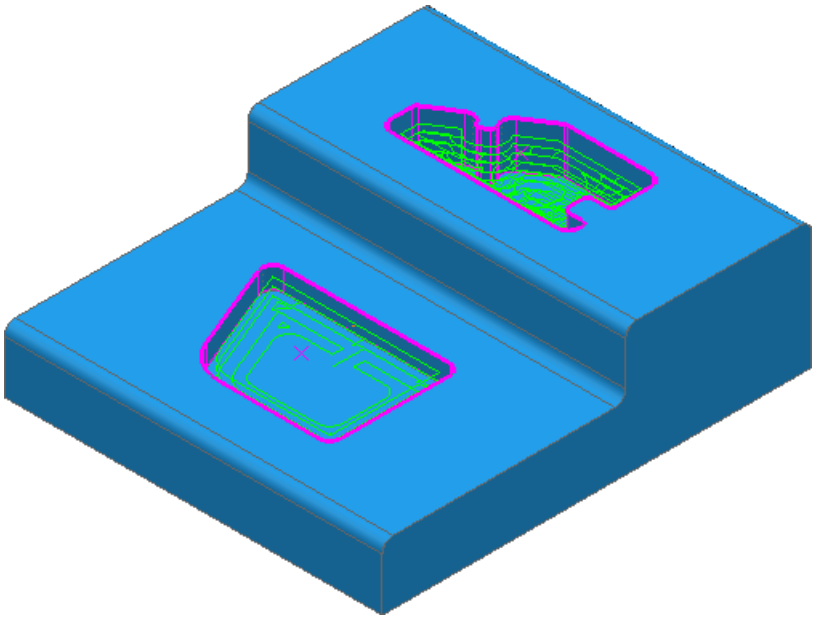
Rest machining — Select to change the strategy to **Feature pocket rest area clearance** and make the **Rest** page available with the options for rest machining. This option is not selected by default for this strategy.

Ignore chamfers — When selected the chamfers are not machined.

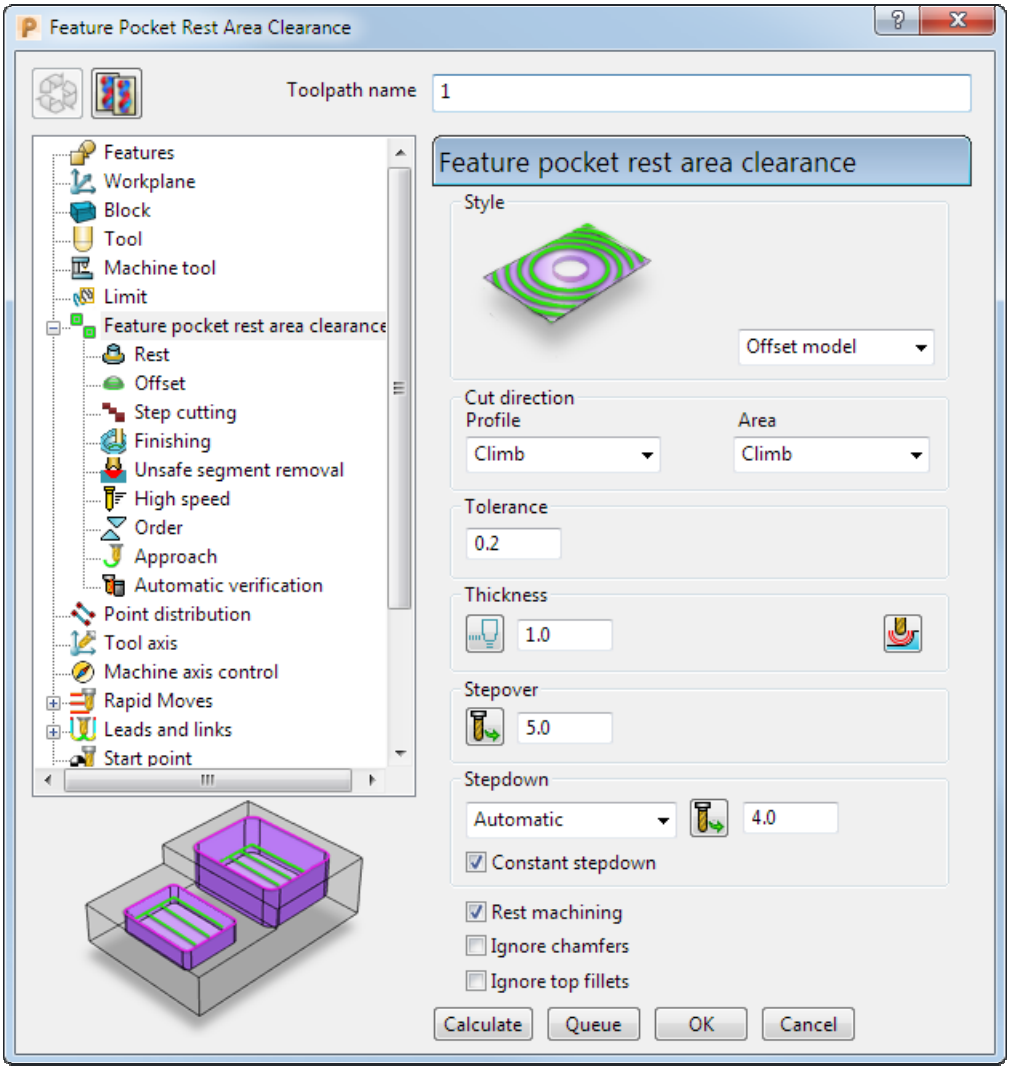
Ignore top fillets — When selected the top fillets are not machined.

Feature Pocket Rest Area Clearance Overview

Use the **Feature Pocket Rest Area Clearance** strategy after a pocket area clearance strategy to rough areas of the pocket features using a small tool, that a large tool could not reach. This strategy enables you to machine multiple pockets at different Z levels.



There are several pages associated with the **Feature Pocket Rest Area Clearance** strategy:



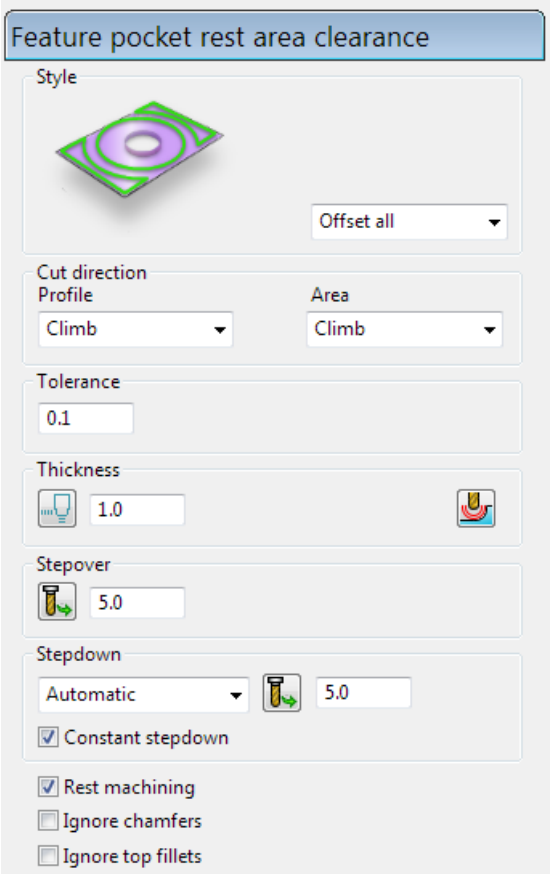
- **Feature pocket rest area clearance** (see page 64) — The main page used to specify a feature pocket rest area clearance toolpath.
- **Rest** — Settings to define rest machining.
- **Raster** — Settings to define a raster area clearance style. This page is available when you select a **Style** of **Raster** on the main page.
- **Offset** — Settings to define offset area clearance styles. This page is available when you select a **Style** of **Offset model** or **Offset all** on the main page.
- **Vortex** — Settings to define a Vortex area clearance style. This page is available when you select a **Style** of **Vortex** on the main page.

- **Step cutting** — Settings to define in-line rest roughing. This minimises terracing when creating area clearance toolpaths with a large stepdown. This is available when you select a **Stepdown** of **Automatic**.
- **Finishing** — Settings to perform a **Final Stepover** and a **Final Stepdown** which are different from the normal stepover and stepdown.
- **Unsafe segment removal** — Settings to remove small toolpath segments.
- **High speed** — Settings to control smoothing options to avoid sharp changes in tool direction when high speed machining. This page is only available for offset area clearance styles.
- **Order** — Settings to control the order of machining.
- **Approach** — Settings to control how the tool approaches the path.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

Feature pocket rest area clearance

Use the **Feature pocket rest area clearance** page to create a toolpath by slicing the pocket features at specified Z heights and then creates an offset or raster pass at each Z height. You can machine multiple pockets at different Z heights using one strategy.



Style — Select the raster, offset, or Vortex style to use for removing material.

Cut direction — Select a milling style for **Profile** and **Area**.

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

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

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

Stepover — Enter the distance between successive area clearance passes at a single Z height.

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*  .

Stepdown — Enter the distance between different machining levels.

 **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.

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

Constant Stepdown — When selected, all the machining levels are equispaced, and the **Stepdown** value is a maximum stepdown. When deselected, the difference between consecutive machining levels is the **Stepdown** value for all levels except the last one which is at the bottom of the block.

This option is only available if you select a **Stepdown** of **Automatic**. If you select a **Stepdown** of **Manual**, the **Constant Stepdown** option is available on the **Area Clearance Z Heights** dialog.

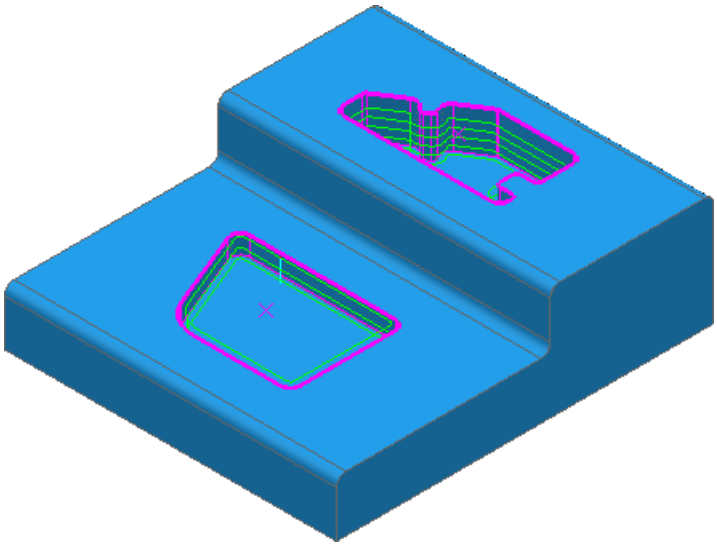
Rest machining — Select to enable the **Rest** page which contains the options for rest machining. If deselected, the strategy switches to the **Feature Pocket Area Clearance** strategy. This option is selected by default in this strategy.

Ignore chamfers — When selected the chamfers are not machined.

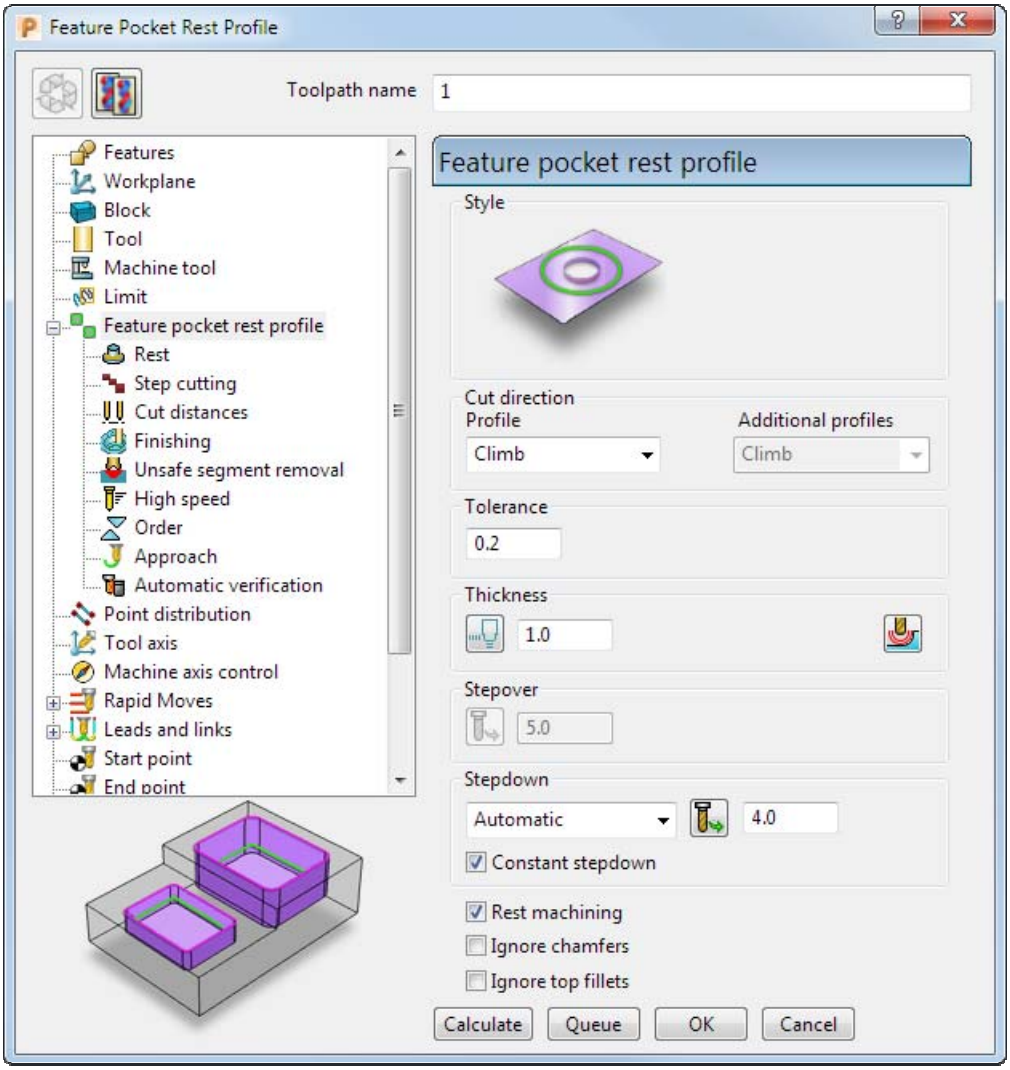
Ignore top fillets — When selected the top fillets are not machined.

Feature Pocket Rest Profile Overview

Use the **Feature Pocket Rest Profile** strategy that creates a toolpath around the pocket profile after the pocket has been roughed using an area clearance strategy. This strategy enables you to machine multiple pockets at different Z levels.



There are several pages associated with the **Feature Pocket Rest Profile** strategy:



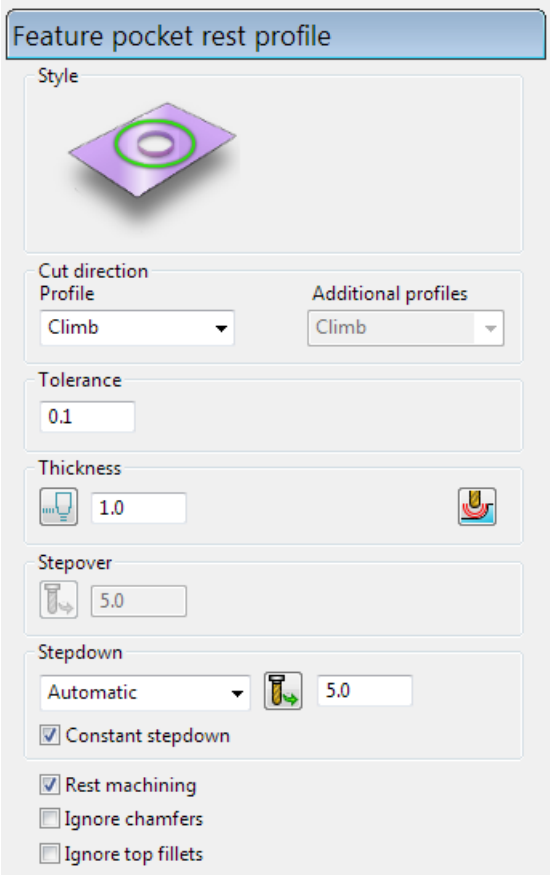
- **Feature pocket rest profile** (see page 68) — The main page used to specify a feature pocket rest profile toolpath.
- **Rest** — Settings to define rest machining.
- **Step cutting** — Settings to define in-line rest roughing. This minimises terracing when creating area clearance toolpaths with a large stepdown. This is available when you select a **Stepdown** of **Automatic**.
- **Cut distances** — Settings to control the number of profile cuts.
- **Finishing** — Settings to perform a **Final Stepover** and a **Final Stepdown** which are different from the normal stepover and stepdown.
- **Unsafe segment removal** — Settings to remove small toolpath segments.
- **High speed** — Settings for the smoothing options to avoid sharp changes in tool direction when high speed machining.

- **Order** — Settings to control the order of machining.
- **Approach** — Settings to control how the tool approaches the path.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

Feature pocket rest profile

Use the **Feature pocket rest profile** page to create a toolpath by slicing the pocket features at specified Z heights and then creates a profile pass at each Z height. You can machine multiple pockets at different Z heights using one strategy. The Z heights are defined from the previous area clearance toolpath and are used to eliminate large terraces.



Cut direction — Select the milling technology. When you have several profile passes you can have a different cut direction for the final profile pass.

Profile — Select the cut direction of the final profiling pass.

Additional profiles — Select the cut direction of all passes except the final profiling pass.

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

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

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

Stepover — Enter the distance between successive machining passes.

 **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*  .

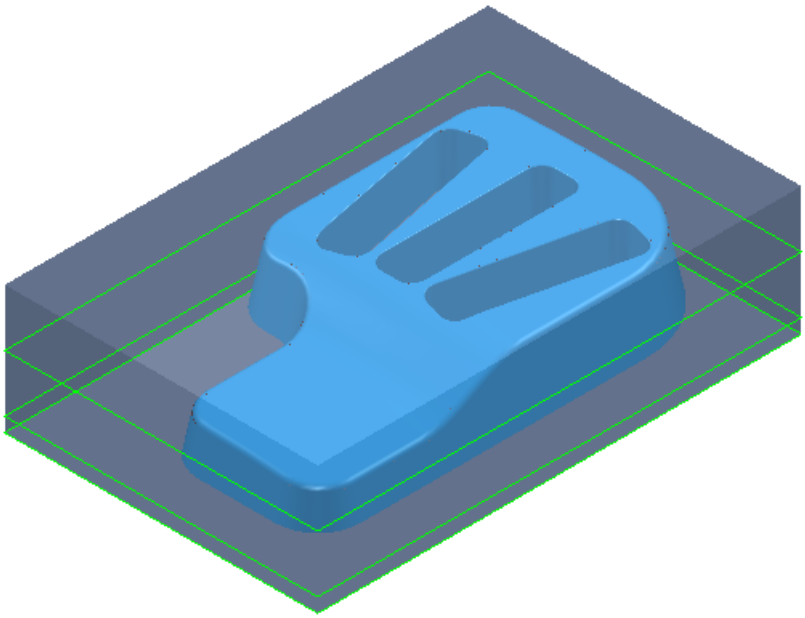
Stepdown — Enter the distance between different machining levels.

 **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.

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

Constant Stepdown — When selected, all the machining levels are equispaced, and the **Stepdown** value is a maximum stepdown. When deselected, the difference between consecutive machining levels is the **Stepdown** value for all levels except the last one which is at the bottom of the block.

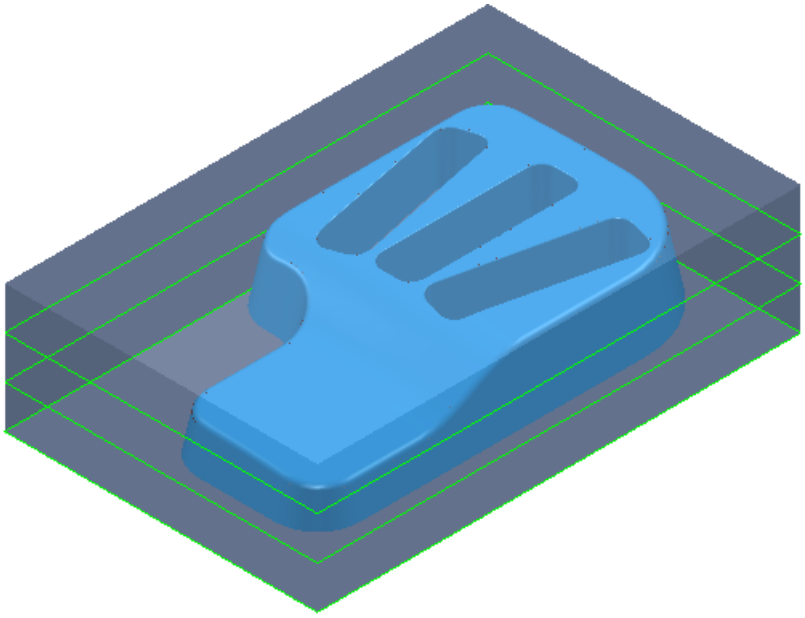
Constant Stepdown deselected:



With a **Stepdown** of **20**, the Z heights are at 15, -5, and -10.

The stepdown is the amount specified for all levels (in this case **20**) except for the last one, which is at the bottom of the block (in this case a **Stepdown** of **5**).

Constant Stepdown selected:



With a **Stepdown** of **20**, the Z heights are at 20, 5, and-10. This gives an effective stepdown of 15.

The stepdown is the same between all levels but is not necessarily the amount specified. In this case, PowerMill uses a **Stepdown** of **15** rather than **20**.

*This option is only available if you select a **Stepdown** of **Automatic**. If you select a **Stepdown** of **Manual**, the **Constant Stepdown** option is available on the **Area Clearance Z Heights** dialog.*

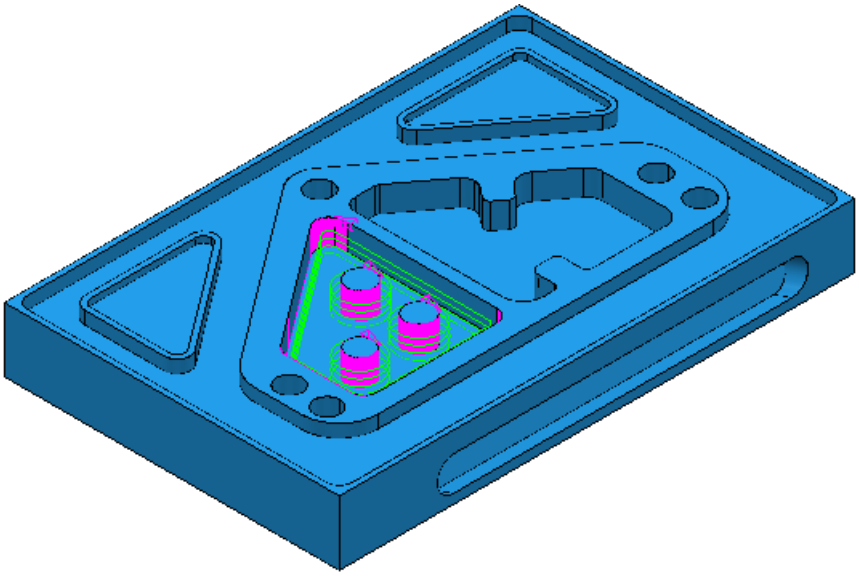
Rest machining — Select to enable the **Rest** page which contains the options for rest machining. If deselected, the strategy switches to the **Feature Pocket Profile** strategy. This option is selected by default in this strategy.

Ignore chamfers — When selected the chamfers are not machined.

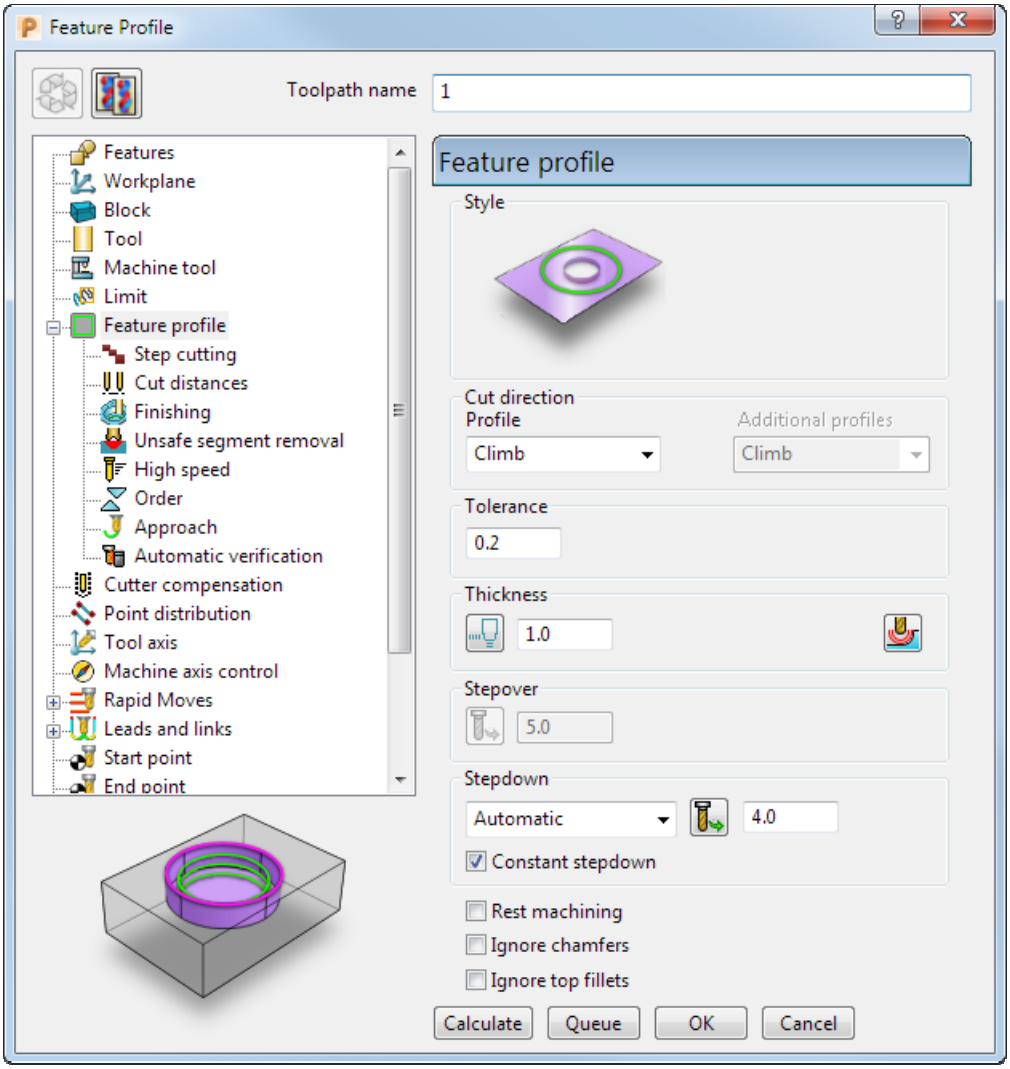
Ignore top fillets — When selected the top fillets are not machined.

Feature Profile Overview

Use the **Feature Profile** strategy to create a toolpath by slicing the feature set at specified Z heights and then machines the feature profiles at each Z height.



There are several pages associated with the **Feature Profile** strategy:



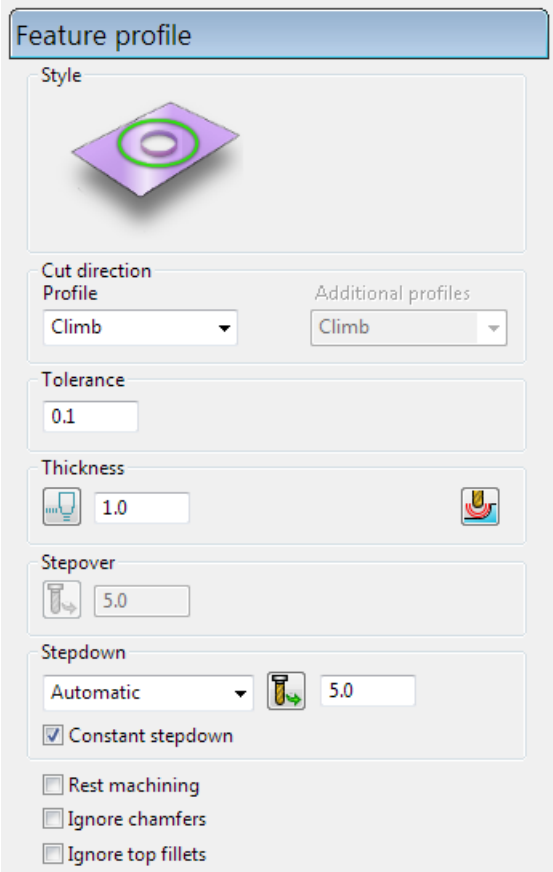
- **Feature profile** (see page 73) — The main page used to machine a feature using a profile strategy.
- **Step cutting** — Settings to define in-line rest roughing. This minimises terracing when creating area clearance toolpaths with a large stepdown. This is available when you select a **Stepdown** of **Automatic**.
- **Cut distances** — Settings to control the number of profile cuts.
- **Finishing** — Settings to perform a **Final Stepover** and a **Final Stepdown** which are different from the normal stepover and stepdown.
- **Unsafe segment removal** — Settings to remove small toolpath segments.
- **High speed** — Settings to control smoothing options to avoid sharp changes in tool direction when high speed machining. This page is only available for offset area clearance styles.
- **Order** — Settings to control the order of machining.

- **Approach** — Settings to control how the tool approaches the path.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

Feature profile

Use the **Feature profile** page to create a toolpath by slicing the features at specified Z heights and then machines the feature profiles at each Z height.



Cut direction — Select the milling technology. When you have several profile passes you can have a different cut direction for the final profile pass.

Profile — Select the cut direction of the final profiling pass.

Additional profiles — Select the cut direction of all passes except the final profiling pass.

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

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

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

Stepover — Enter the distance between successive machining passes.

 **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*  *.*

Stepdown — Enter the distance between different machining levels.

 **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.

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

Constant Stepdown — When selected, all the machining levels are equispaced, and the **Stepdown** value is a maximum stepdown. When deselected, the difference between consecutive machining levels is the **Stepdown** value for all levels except the last one which is at the bottom of the block.

*This option is only available if you select a **Stepdown** of **Automatic**. If you select a **Stepdown** of **Manual**, the **Constant Stepdown** option is available on the **Area Clearance Z Heights** dialog.*

Rest machining — Select to change the strategy to **Feature Rest Profile** and make the **Rest** page available with additional options for rest machining. This option is not selected by default in this strategy.

Ignore chamfers — When selected the chamfers are not machined.

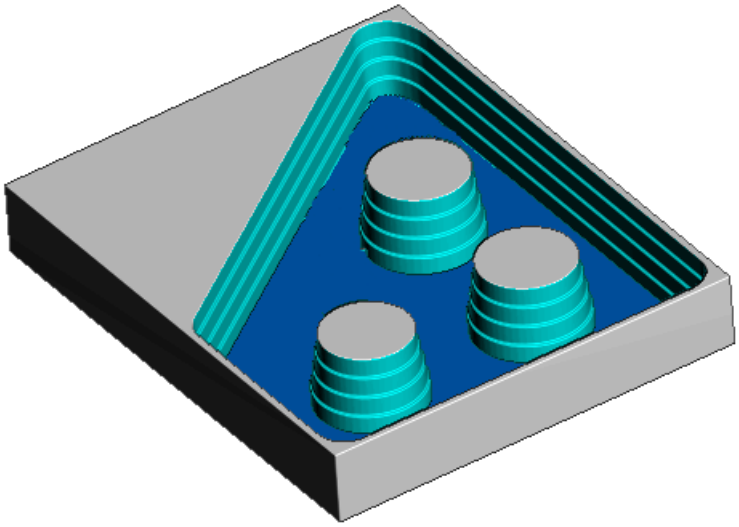
Ignore top fillets — When selected the top fillets are not machined.

Feature Rest Area Clearance Overview

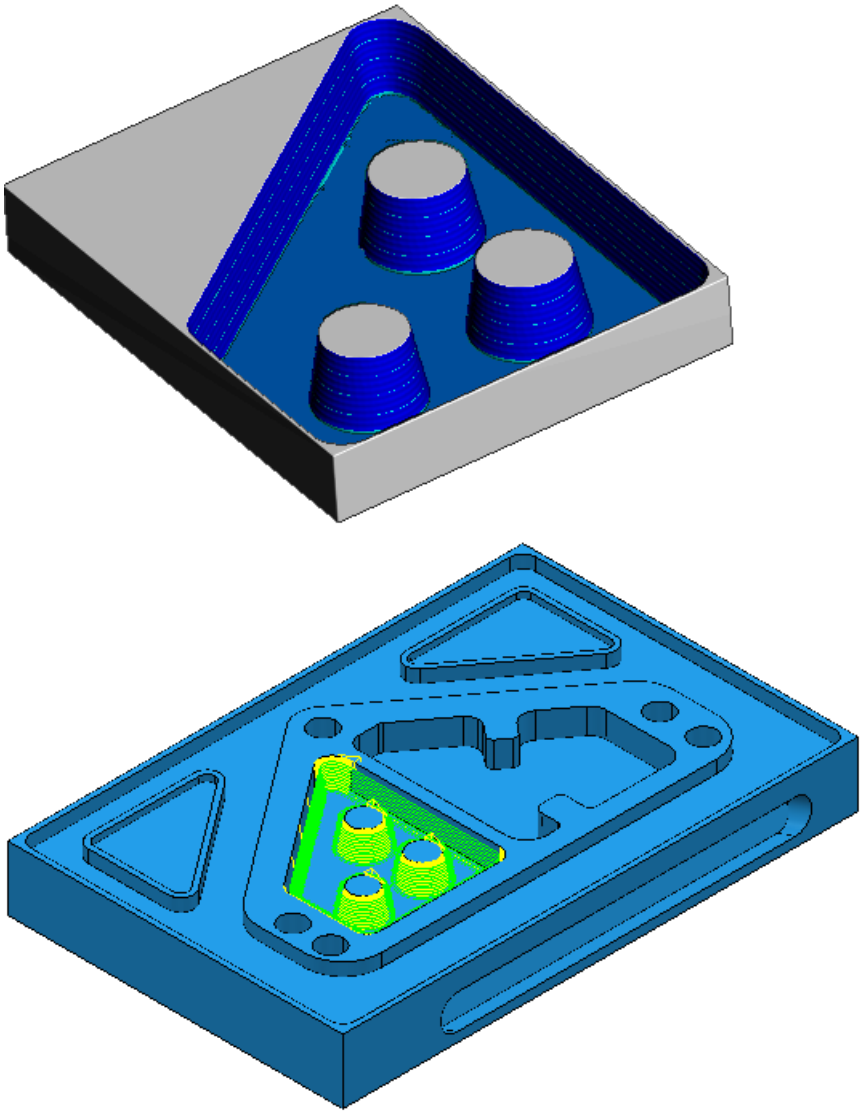
Use the **Feature Rest Area Clearance** strategy to eliminate large terraces. Area clearance strategies carry out efficient volume removal with a large tool and then rest area clearance strategies use a smaller tool to rough areas of the feature set that the large tool could not reach, such as pockets and corners.

This is easier to see looking at a ViewMill simulation.

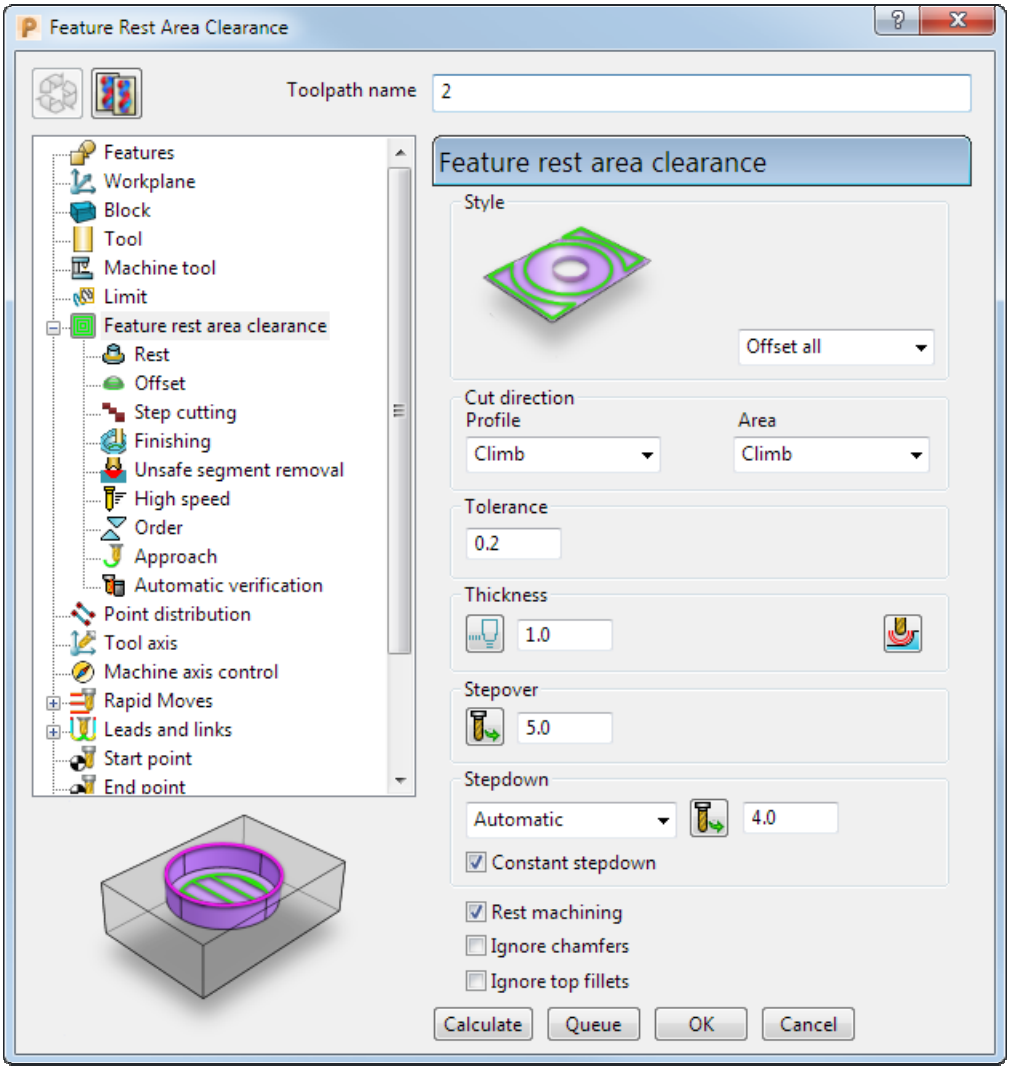
If you start with an area clearance toolpath:



A feature rest area clearance toolpath, based on this area clearance toolpath, gives:



There are several pages associated with the **Feature Rest Area Clearance** strategy:



- **Feature rest area clearance** (see page 79) — The main page used to choose the area clearance styles and associated settings.
- **Rest** — Settings to define rest machining.
- **Raster** — Settings to define a raster area clearance style. This page is available when you select a **Style** of **Raster** on the main page.
- **Offset** — Settings to define offset area clearance styles. This page is available when you select a **Style** of **Offset model** or **Offset all** on the main page.
- **Vortex** — Settings to define a Vortex area clearance style. This page is available when you select a **Style** of **Vortex** on the main page.

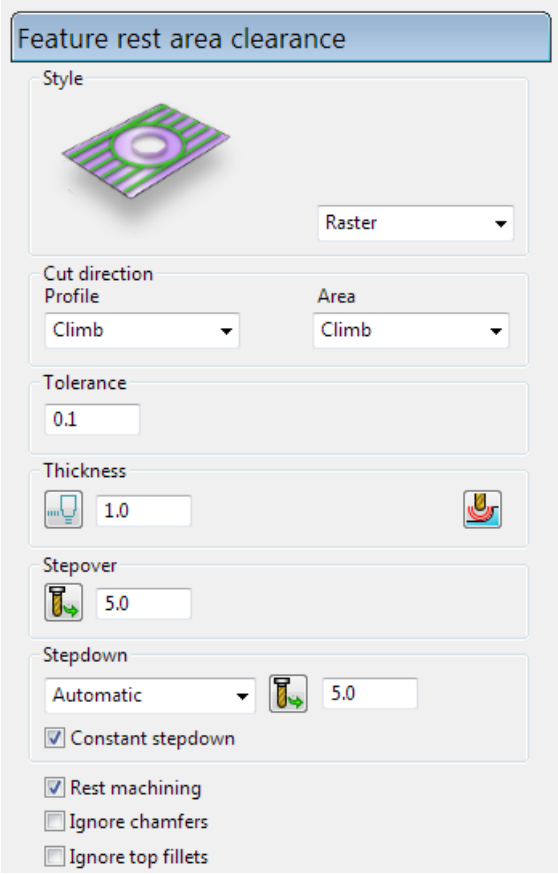
- **Step cutting** — Settings to define in-line rest roughing. This minimises terracing when creating area clearance toolpaths with a large stepdown. This is available when you select a **Stepdown** of **Automatic**.
- **Finishing** — Settings to perform a **Final Stepover** and a **Final Stepdown** which are different from the normal stepover and stepdown.
- **Unsafe segment removal** — Settings to remove small toolpath segments.
- **High speed** — Settings to control smoothing options to avoid sharp changes in tool direction when high speed machining. This page is only available for offset area clearance styles.
- **Order** — Settings to control the order of machining.
- **Approach** — Settings to control how the tool approaches the path.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

Cutter compensation is not available for rest machining strategies.

The remaining pages are common toolpath creation controls.

Feature rest area clearance

Use the **Feature rest area clearance** page to create a toolpath by slicing the feature set at specified Z heights and then creates an offset or raster pass at each Z height. The Z heights are defined from the previous area clearance toolpath and are used to eliminate large terraces.



Style — Select the raster, offset, or Vortex style to use for removing material.

Cut direction — Select a milling style for **Profile** and **Area**.

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

Thickness — Enter the amount of material to be left on the stock within tolerance.

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

Stepover — Enter the distance between successive machining passes.

 **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*  *.*

Stepdown — Enter the distance between different machining levels.

 **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.

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

Constant Stepdown — When selected, all the machining levels are equispaced, and the **Stepdown** value is a maximum stepdown. When deselected, the difference between consecutive machining levels is the **Stepdown** value for all levels except the last one which is at the bottom of the block.

 *This option is only available if you select a **Stepdown** of **Automatic**. If you select a **Stepdown** of **Manual**, the **Constant Stepdown** option is available on the **Area Clearance Z Heights** dialog.*

Rest machining — Select to enable the **Rest** page which contains the options for rest machining. If deselected, the strategy switches to the **Feature Area Clearance** strategy. This option is selected by default in this strategy.

Ignore chamfers — When selected the chamfers are not machined.

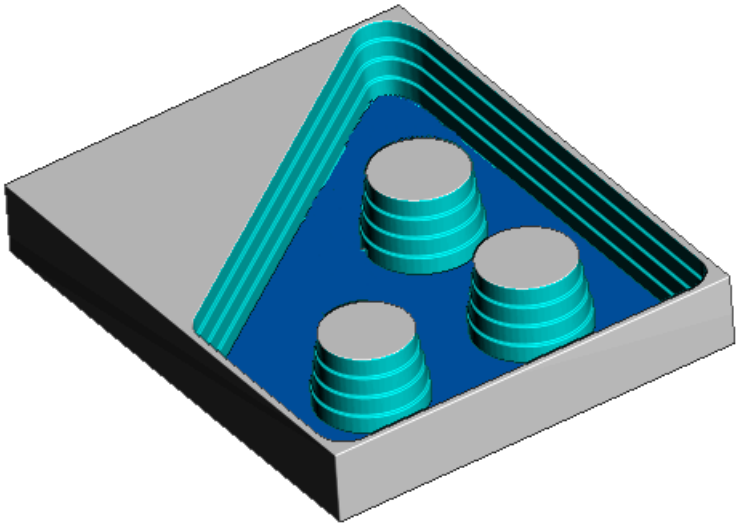
Ignore top fillets — When selected the top fillets are not machined.

Feature Rest Profile Overview

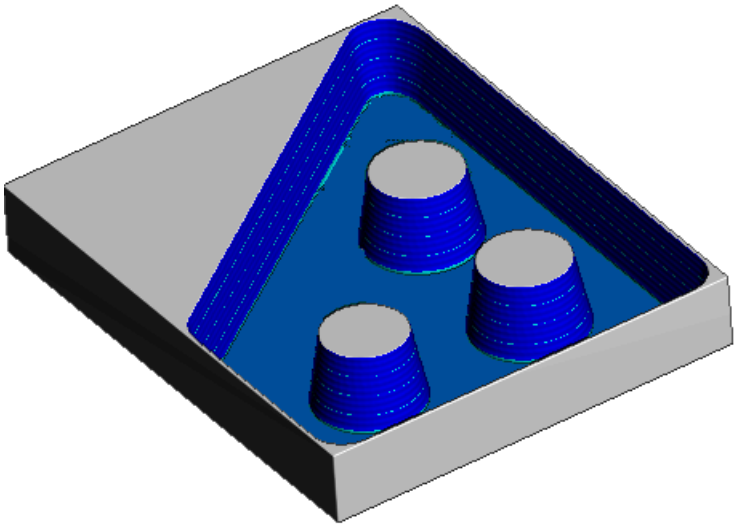
Use the **Feature Rest Profile** strategy to eliminate large terraces. Area clearance strategies carry out efficient volume removal with a large tool and then rest area clearance strategies use a smaller tool to rough areas of the feature set profile that the large tool could not reach, such as pockets and corners.

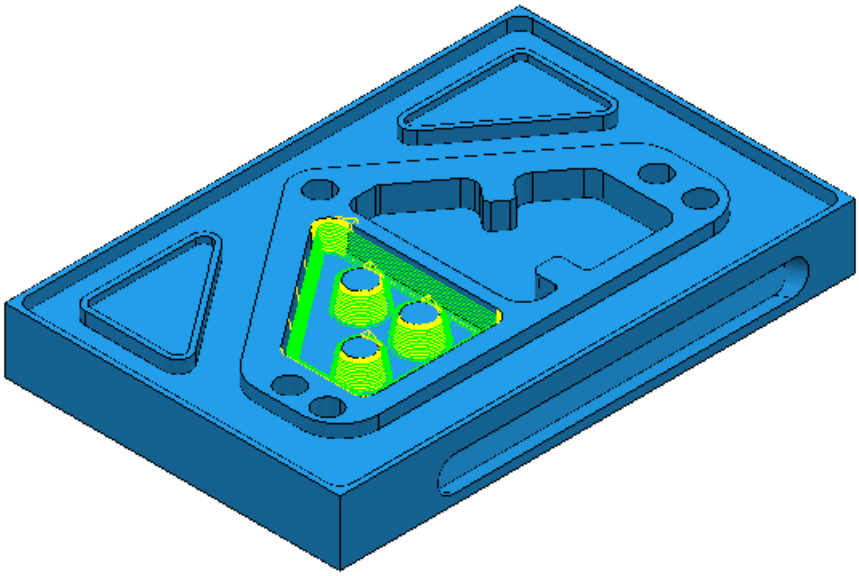
This is easier to see looking at a ViewMill simulation.

If you start with an area clearance toolpath:

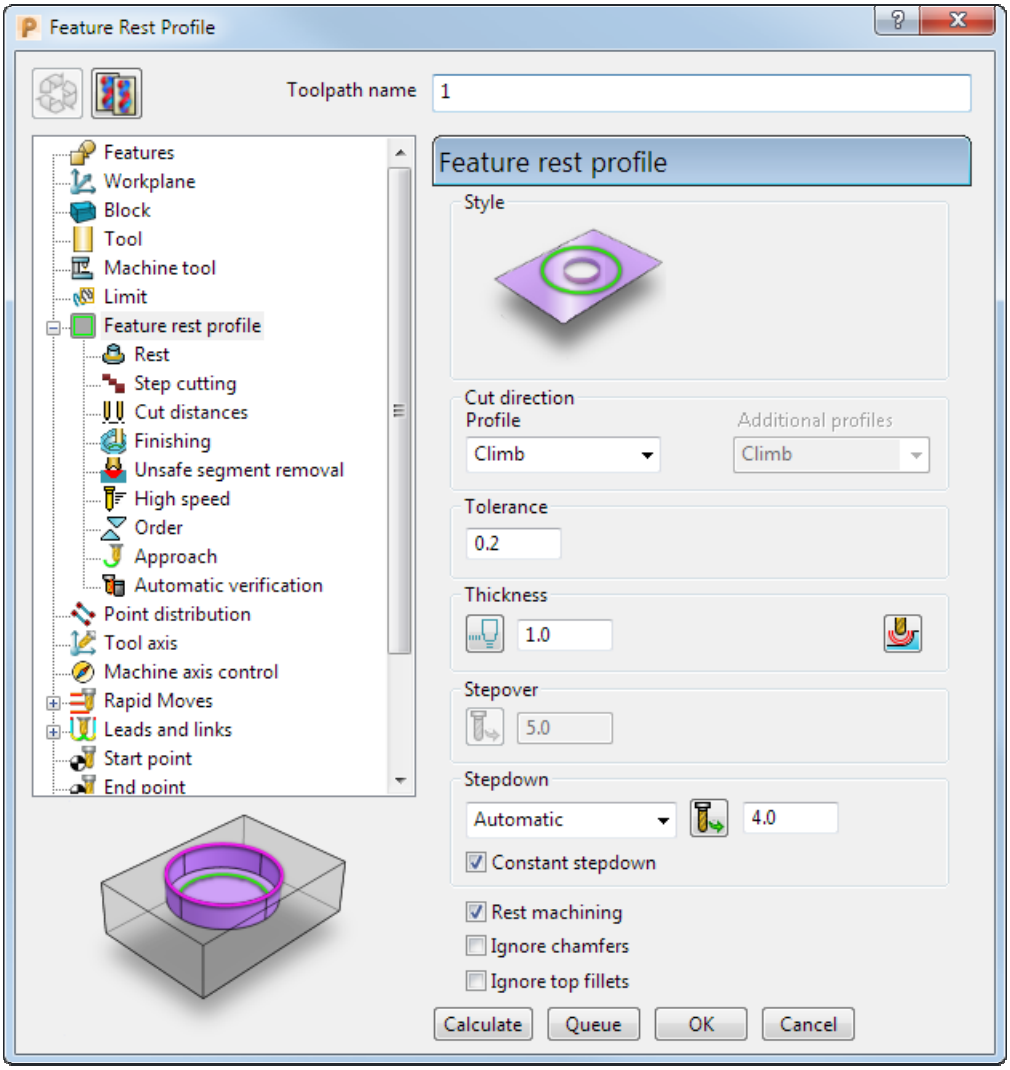


A feature rest area clearance toolpath, based on this area clearance toolpath, gives:





There are several pages associated with the **Feature Rest Profile** strategy:



- **Feature rest profile** (see page 84) — The main page which contain settings to machine a feature using a rest profile strategy.
- **Rest** — Settings to define rest machining.
- **Step cutting** — Settings to define in-line rest roughing. This minimises terracing when creating area clearance toolpaths with a large stepdown. This is available when you select a **Stepdown** of **Automatic**.
- **Cut distances** — Settings to control the number of profile cuts.
- **Finishing** — Settings to perform a **Final Stepover** and a **Final Stepdown** which are different from the normal stepover and stepdown.
- **Unsafe segment removal** — Settings to remove small toolpath segments.

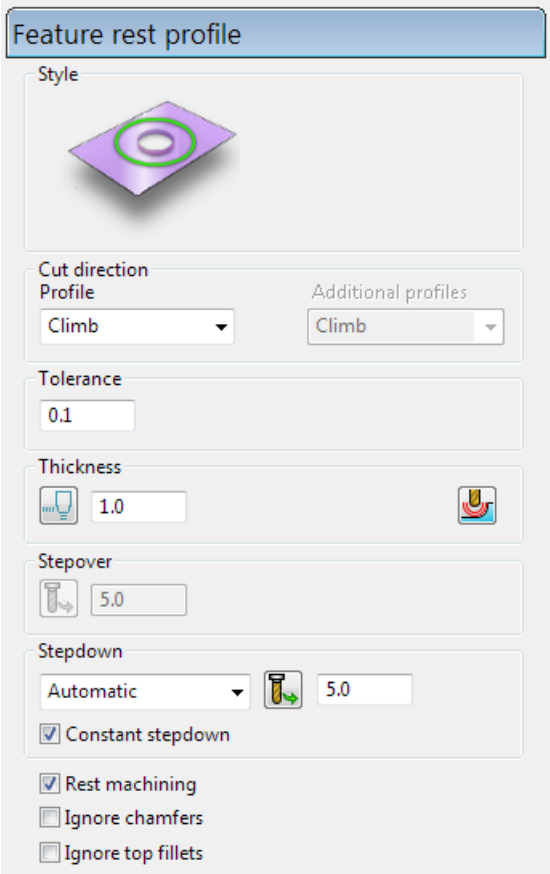
- **High speed** — Settings to control smoothing options to avoid sharp changes in tool direction when high speed machining. This page is only available for offset area clearance styles.
- **Order** — Settings to control the order of machining.
- **Approach** — Settings to control how the tool approaches the path.
- **Automatic verification** — Settings to automatically verify toolpaths on creation.

Cutter compensation is not available for rest machining strategies.

The remaining pages are common toolpath creation controls.

Feature rest profile

Use the **Feature rest profile** page to create a toolpath by slicing the feature set at specified Z heights and then creates profile pass at each Z height. The Z heights are defined from the previous area clearance toolpath and are used to eliminate large terraces.



Cut direction — Select the milling technology. When you have several profile passes you can have a different cut direction for the final profile pass.

Profile — Select the cut direction of the final profiling pass.

Additional profiles — Select the cut direction of all passes except the final profiling pass.

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

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

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

Stepover — Enter the distance between successive machining passes.

 **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*  *.*

Stepdown — Enter the distance between different machining levels.

 **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.

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

Constant Stepdown — When selected, all the machining levels are equispaced, and the **Stepdown** value is a maximum stepdown. When deselected, the difference between consecutive machining levels is the **Stepdown** value for all levels except the last one which is at the bottom of the block.

 *This option is only available if you select a **Stepdown** of **Automatic**. If you select a **Stepdown** of **Manual**, the **Constant Stepdown** option is available on the **Area Clearance Z Heights** dialog.*

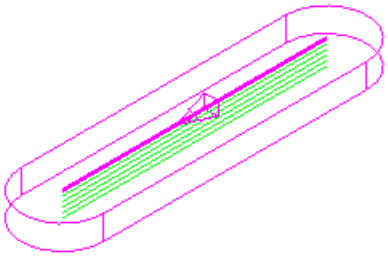
Rest machining — Select to enable the **Rest** page which contains the options for rest machining. If deselected, the strategy switches to **Feature Profile** strategy. This option is selected by default in this strategy.

Ignore chamfers — When selected the chamfers are not machined.

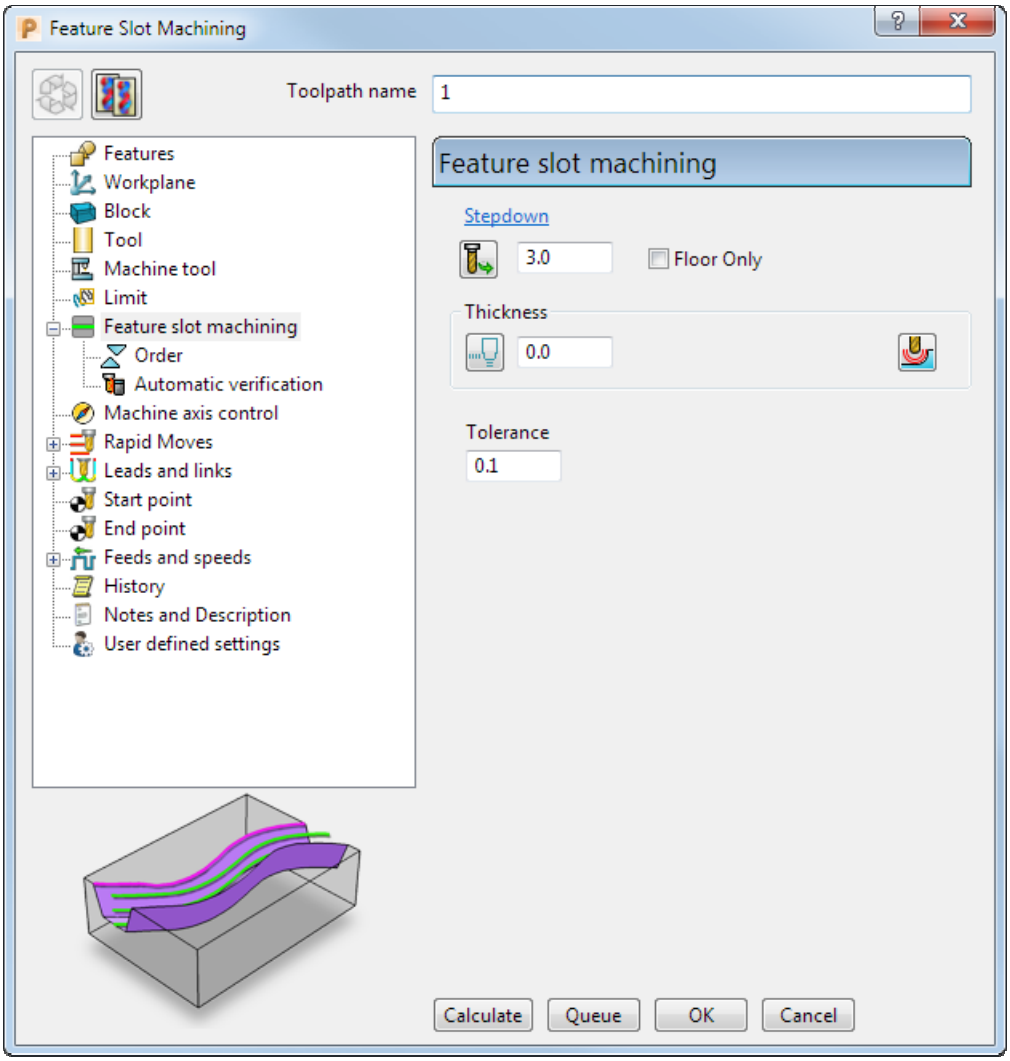
Ignore top fillets — When selected the top fillets are not machined.

Feature Slot Machining Overview

Use the **Feature slot machining** strategy to machine all slot features in a feature group.



There are several pages associated with the **Feature Slot Machining** strategy:



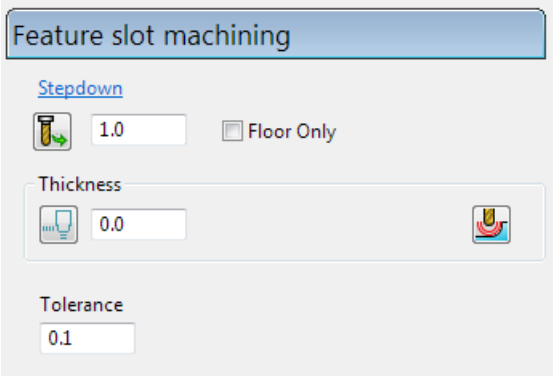
- **Feature slot machining** (see page 87) — The main page used to specify a feature slot machining toolpath.
- **Order** — Settings to control the order of machining.

- **Automatic verification** — Settings to automatically verify toolpaths on creation.

The remaining pages are common toolpath creation controls.

Feature slot machining

Use the **Feature slot machining** page to create a toolpath that machines slot features in a feature group.



Stepdown — Enter the distance between different machining levels.

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.

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

Floor only — Select to create one toolpath along the base of the slot channel.

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

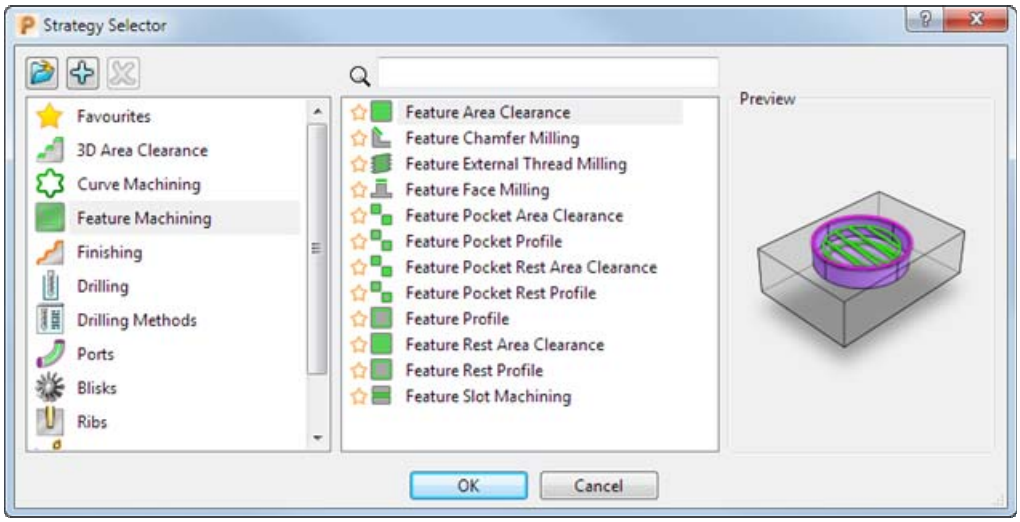
Component thickness — Click to display the **Component thickness** dialog, which enables you to specify the thicknesses of the different surfaces.

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

Creating a 2D feature strategy example

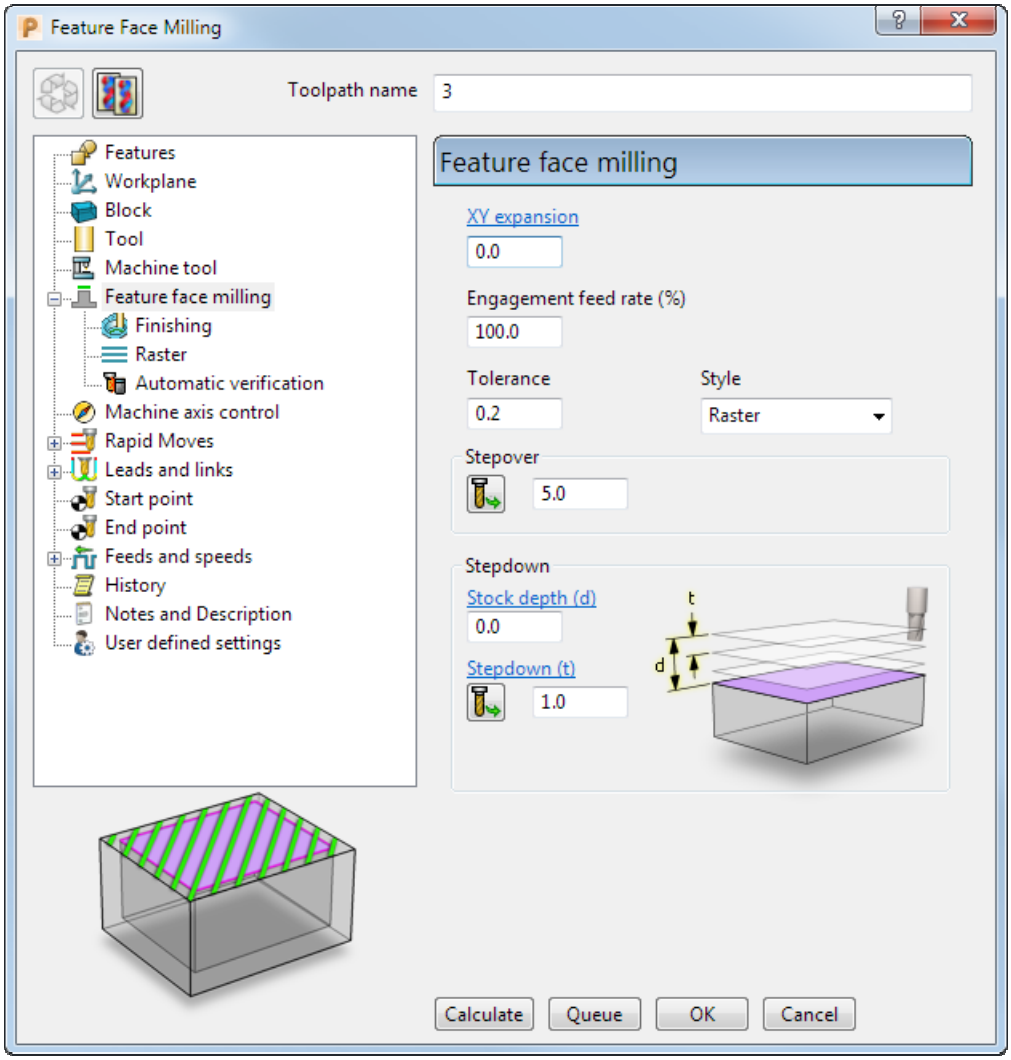
To create a 2D feature machining toolpath:

1. In the **Main** toolbar, click **Toolpath strategies**  , or right-click **Toolpaths** in the Explorer and select **Create toolpath**.
2. Select the new **Feature Machining** page in the **Strategy Selector** dialog.



3. Select a new feature strategy and click **OK** to display the strategy dialog.

For example, selecting **Feature Face Milling** displays the **Feature face milling** page of the **Feature Face Milling** strategy dialog.



4 On the **Features** page, select the **Feature group** that contains the feature you want to machine.

5 Click the features on the model you want to machine. Use **Ctrl + Click** on a feature to remove it from your selection.

Alternatively use the buttons on the **Features** page to select the features:

- **Select all** — Click to select all features.
- **Select toggle** — Click to deselect the selected features and select the deselected features.
- **Deselect all** — Click to deselect all features.

6 Click **Calculate** to calculate the toolpath.

7 Click **OK** to close the dialog.

Turning

You can now create toolpaths for turned parts in PowerMill.

You can program a turning part without a model using features, or you can extract information from a model.

To program a turning part:

1. Set the View mode to turning (see page 90).
2. Create (and activate) a Workplane (see page 91).
3. Create the turning curves (see page 92).
4. Create the turning features (see page 96).
5. Create or import turning tools (see page 98).
6. Create the turning toolpaths (see page 102).
7. Simulate the turning toolpaths (see page 104).

View mode

Set the view mode to determine how the part is orientated by the standard views.

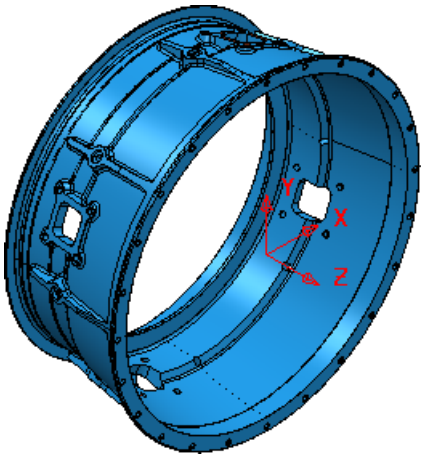
Use the new buttons on the **Viewing** toolbar to set the view mode:

- **Turning view** — Select this option to use the turning standard views, where the Z axis is horizontal in the graphics window. For example, the top view displays the ZX plane.
- **Milling view** — Select this option to use the milling standard views, where the Z axis is vertical in the graphics window. For example, the top view displays the XY plane.

Alternatively, use the new **View > View Mode** menu options.

Creating turning Workplanes

Create a turning Workplane where the Z axis is the rotational axis of the part.



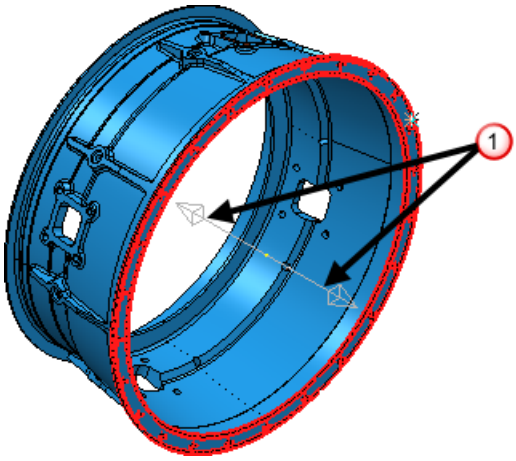
To create a turning Workplane:

1. Click the **Turning view** button on the **Viewing** toolbar to enter the turning view mode.
2. In the **Information** toolbar, select **Use the XY face of the workplane**.

   This sets the Principal editing plane to be looking down the Z axis.
3. Use the new **Create Workplane from Revolved Surface** mode:
   - In the Explorer, right-click **Workplanes** and select **Create and Orientate Workplane > Workplane from Revolved Surface**.
   - In the **Information** toolbar, click **Workplane from revolved surface**.

   The **Create Workplane from Revolved Surface** mode-toolbar is displayed.

4 Select a rotated surface in the graphics window to locate the Workplane origin at its centre.

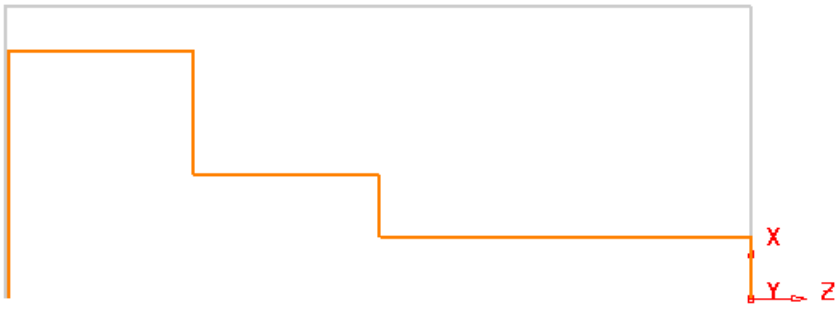


5 Select the Z axis direction (1) in the graphics window.

The **Create Workplane from Revolved Surface** mode-toolbar is closed and the workplane is created.

Creating curves for turning

You can draw curves to program the turning part, or you can extract them from the model.



To create the curves required for turning:

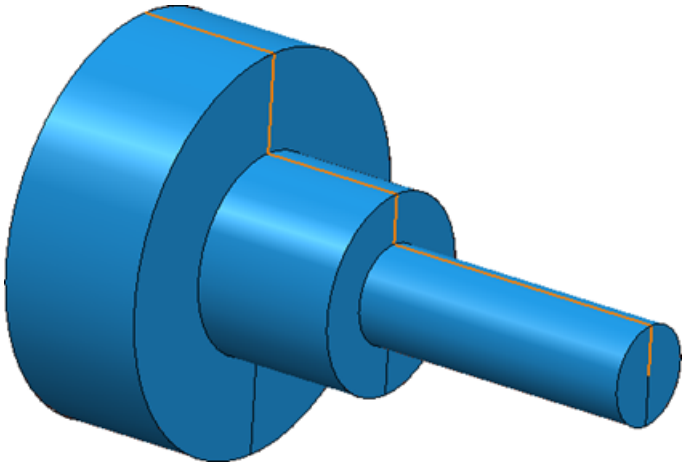
1 Create a Pattern curve to define the shape of the block.

This is not required for using a simple cylindrical block.

2 Create Pattern curves to define the shape of the turning features.

For example, you can create one curve for outer profiling and one for interior boring. You do not need separate curves for rough and finish operations.

If you are using a model, you can extract the feature curves from the model.

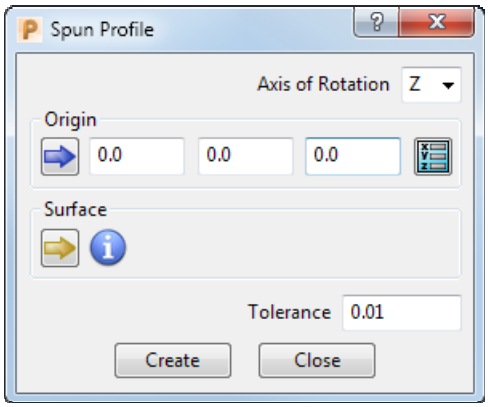


To extract a profile curve from a model:

a In the Explorer, right-click **Patterns** and select **Curve Editor**.

b In the **Curve Editor** mode-toolbar, click the new **Create spun profile** button from the **Curves** pull-down menu.

c The **Spun Profile** dialog is displayed.



d Select an option from the **Axis of rotation** list to specify which axis of the active workplane to revolve around.

e To change the position of the rotation origin, click **Switch to position mode** in the **Origin** area.

f Enter the coordinates of the rotational axis origin, or select it in the graphics window.

g Under **Surface**, click **Surface selection mode** .

h Select surfaces in the graphics window from which you want to create curves, or select nothing to use all available surfaces.

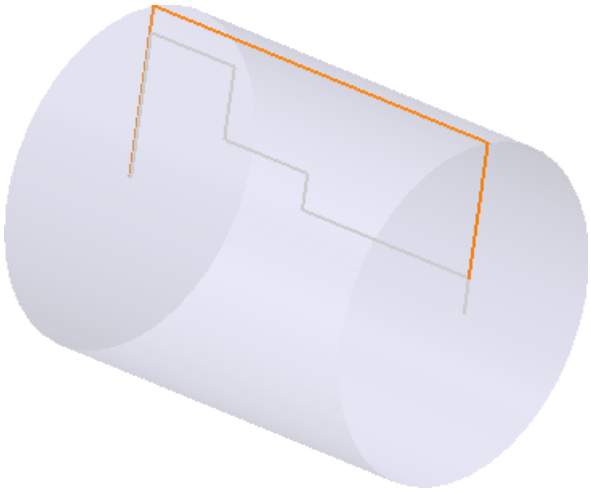
Move your cursor over the icon to display information about which surfaces are selected.

i Click **Create** to create the curve and close the dialog.

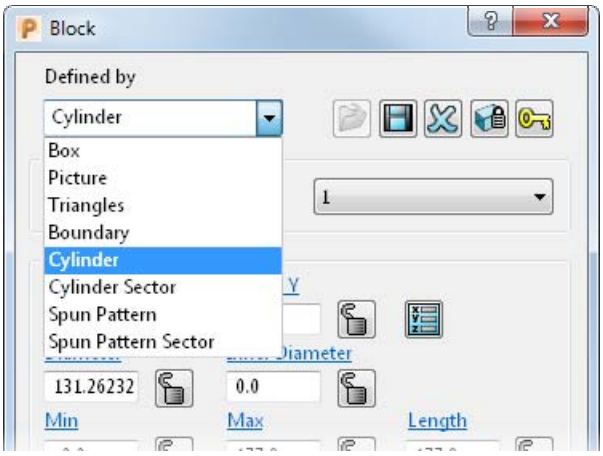
If this does not work correctly, ensure the correct principal editing plane is selected.

Defining the block for turning

Set up the block so it is rotationally symmetrical about the Z axis. You can define the block by entering the dimensions directly, by rotating a curve about the Z axis, or by calculating it from a model.



There are new and updated options in the **Block** dialog.



To define a cylindrical block by entering the dimensions:

1 Under **Defined by**, select **Cylinder**.

2 Ensure the **Coordinate System** is selected as the centre of rotation of the block.

3 Enter the **Min**, **Max,** and **Diameter** values to define the outer size of the cylinder.

4 Enter the new **Inner Diameter** value to define the block as a hollow tube.

To define the block by rotating a curve about the Z axis:

1. Activate the curve you want to rotate to form the block.
2. Under **Defined by**, select the new **Spun Pattern** option.
3. Ensure the **Coordinate System** is selected as the centre of rotation of the block.

To calculate a cylindrical block from the model size:

1. Under **Defined by**, select **Cylinder**.
2. Ensure the **Coordinate System** is selected as the centre of rotation of the block.
3. Enter an **Expansion** to leave additional material around the model.
4. In the **Type** list select **Model**.
5. Click **Calculate**.

There are new **Sector** options that you can use for sector spun parts.

- **Azimuth start** — Enter a value to specify the start of the azimuth angle range.
- **Azimuth end** — Enter a value to specify the end of the azimuth angle range.
- **Range** — Enter a value to specify the size of the azimuth angle range.

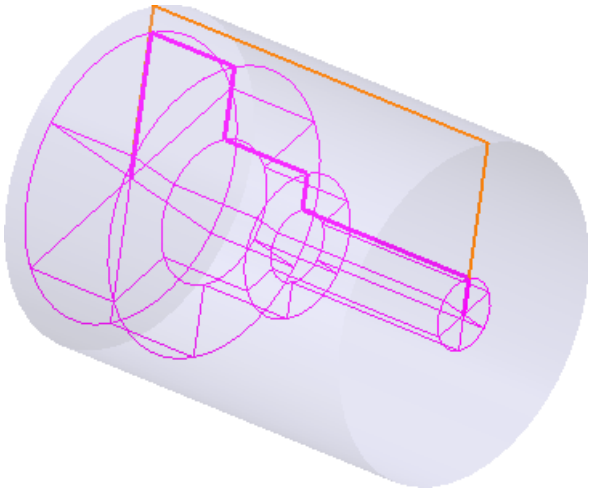
There are new and updated options in the **Block** dialog, under **Defined by**:

- **Cylinder** — This option now includes an internal diameter component (**Inner Diameter**).
- **Cylinder Sector** — Create the block as a **Cylinder** but with azimuth angles, enabling you to create a sector of a cylinder.

- **Spun Pattern** — Create the block by rotating a pattern about the Z axis of the selected coordinate system. This option requires a pattern in the XZ plane that does not self-intersect and does not cross the X axis. You can create a spun pattern using the new **Create spun profile** option in the curve editor.
- **Spun Pattern Sector** — Create the block as a **Spun Pattern** but with azimuth angles, enabling you to create a sector of a cylinder.

Creating turning features

Create turning features from curves to define the machining limits. Features are required to create turning toolpaths.



To create a turning feature:

1 In the Explorer, right-click **Feature Groups** and select **Feature Editor**.

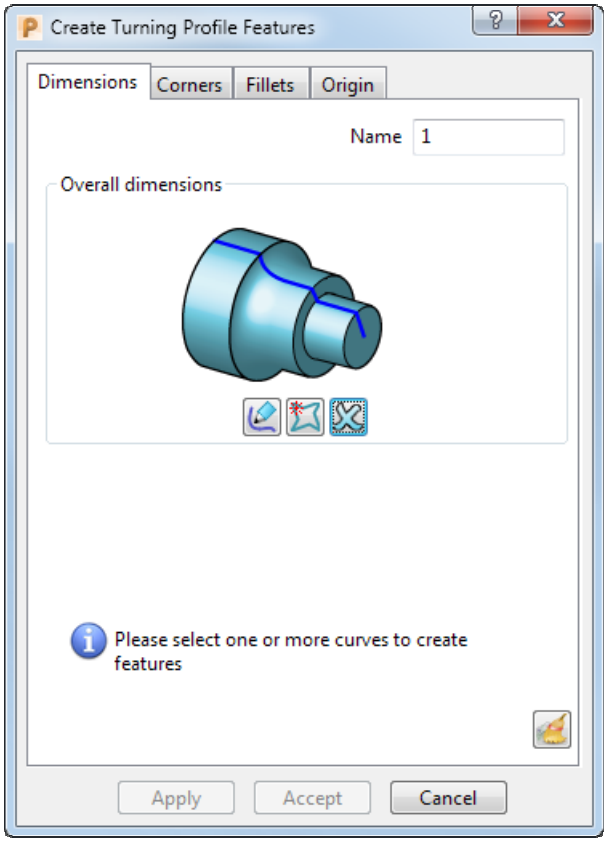
A feature group is created and the **Feature Editor** mode-toolbar is displayed.

2 Select one of the new turning feature types:

-  **Create profile**
-  **Create a freeform groove**
-  **Create a parametric groove**
-  **Create turning face feature**
-  **Create bore feature**

The **Create Features** dialog is displayed, depending on the feature type.

For example, selecting **Create profile** displays the **Create Turning Profile Features** dialog.



3 Enter a **Name** for the feature.

4 Select a curve in the graphics window.

This curve must be an open curve in the XZ plane that does not cross the Z axis.

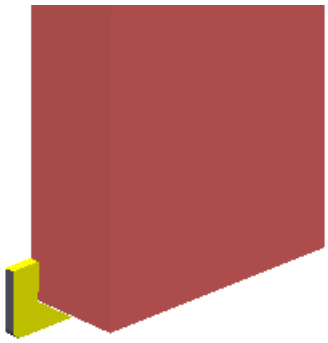
The dialog is updated with information about the selected curve.

5 Click **Accept** to create the feature and close the dialog.

6 Click **Accept changes** on the **Feature Editor** mode-toolbar to save the created feature.

Creating turning tools

There are new tool categories for turning tools.



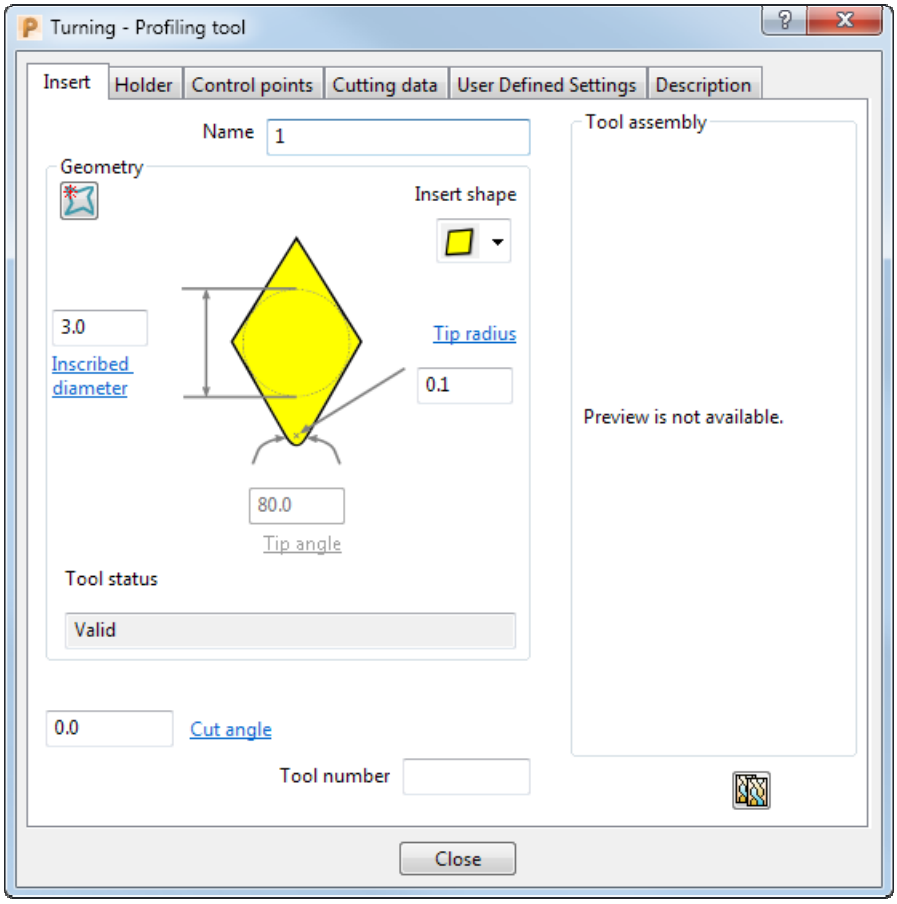
Select one of the new options on the **Tool** toolbar to display the **Tool** dialog.

- **Create a profiling turning tool**
- **Create a grooving turning tool**

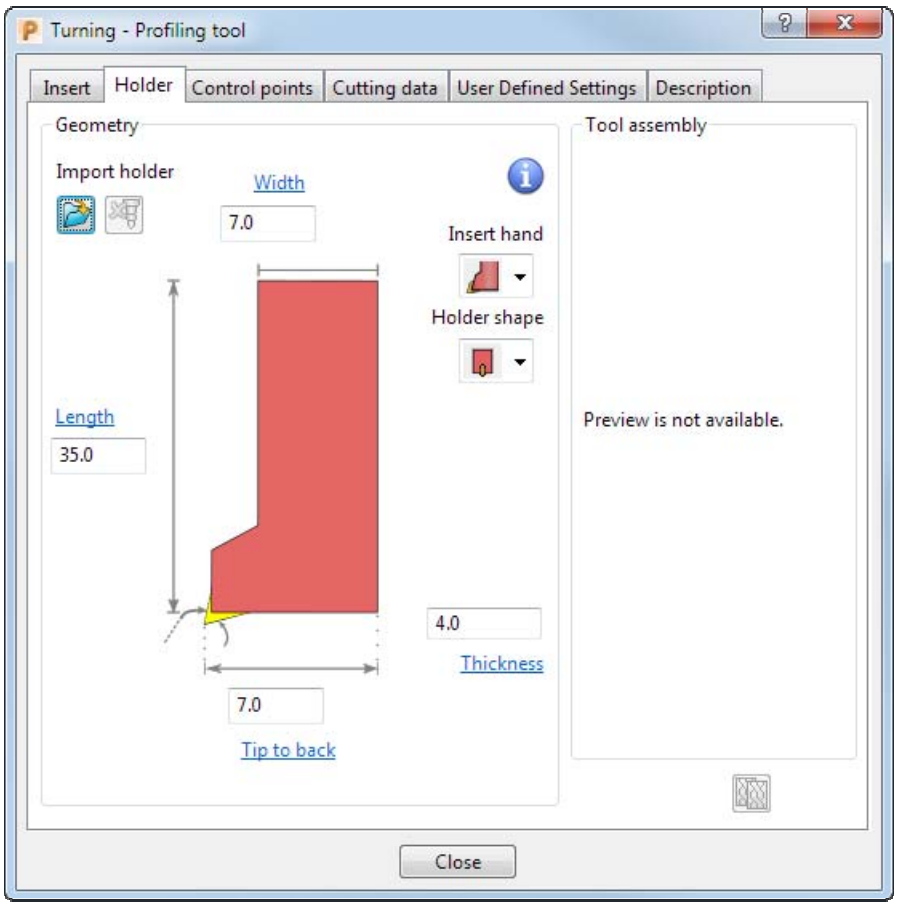
Alternatively, right-click **Tools** in the explorer and select one of the options under **Create tool > Turning**.

Use the **Tool** dialog to specify the tool properties. For example, for an outer profiling tool:

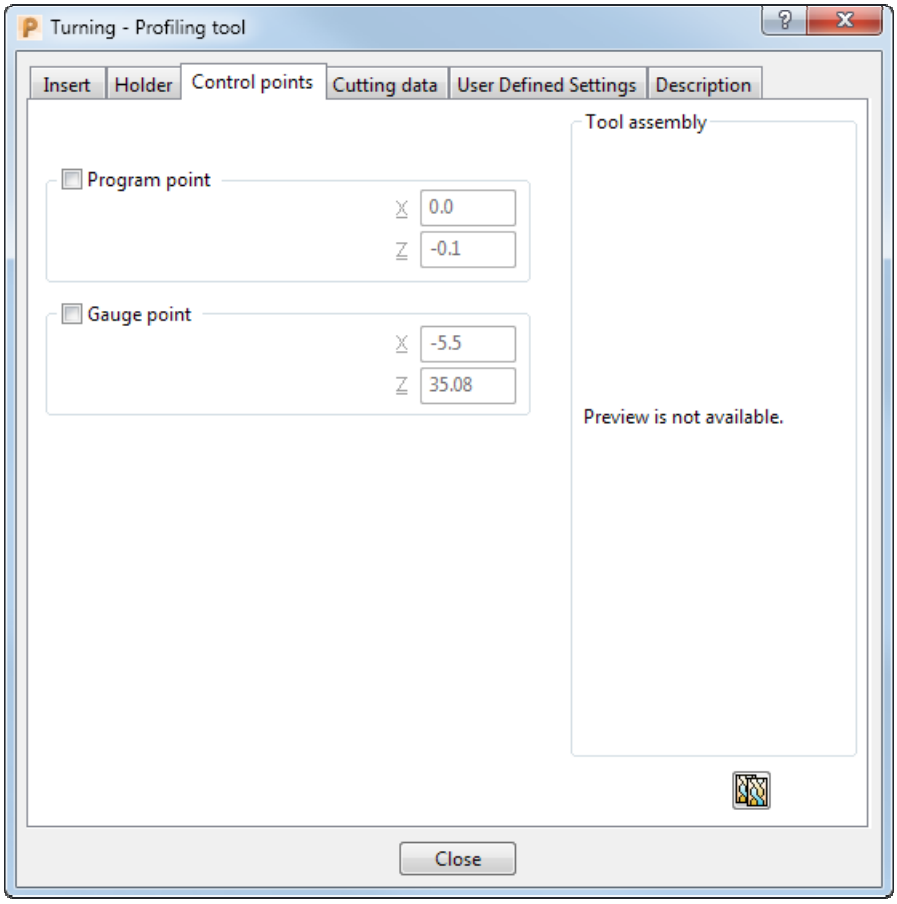
- Use the **Insert** tab to specify the shape and size of the cutting insert.



- Use the **Holder** tab to specify the shape, size and orientation of the tool holder. Alternatively, you can import a tool holder from a model.



- Use the **Control points** tab to specify the tool program point and gauge point relative to the insert tip radius centre.



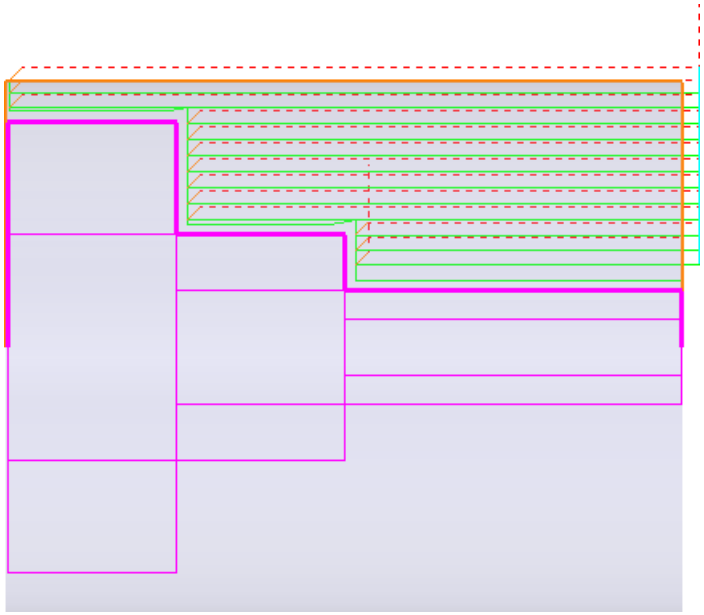
  - **Program point** — Select this option to manually specify the point position of the insert that is programmed. If this option is deselected the program point is calculated automatically from the tool geometry.
  - **Gauge point** — Select this option to manually specify the position where the tool holder attaches to the machine tool with respect to the insert tip originw . If this option is deselected, the gauge point is calculated automatically from the tool geometry.

Saving turning tools

You cannot add turning tools to the tool library, but you can save them as a template to import them into another document.

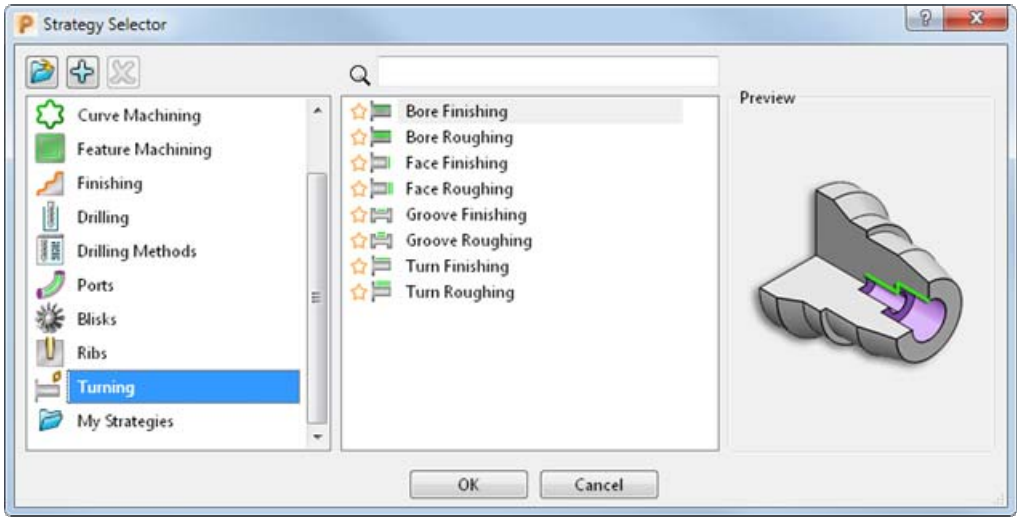
Creating turning toolpaths

There are new strategies that you can use to create turning toolpaths.



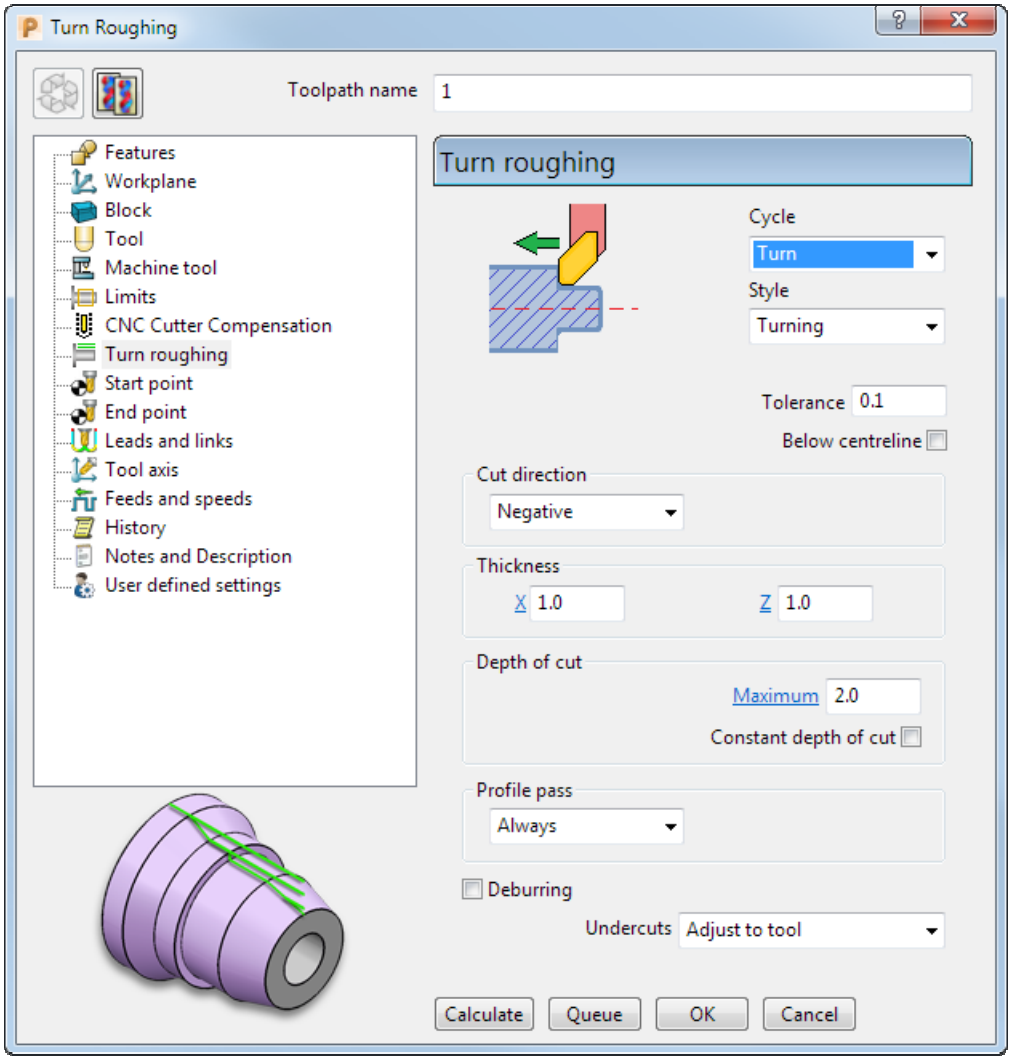
To create a turning toolpath:

1. In the **Main** toolbar, click **Toolpath strategies**  , or right-click **Toolpaths** in the Explorer and select **Create toolpath**.
2. Select the new **Turning** page in the **Strategy Selector** dialog.



3. Select a new turning strategy and click **OK** to display the strategy dialog.

For example, selecting **Turn Rouging** displays the **Turn roughing** page of the **Turn Roughing** strategy dialog.



4 On the **Features** page, select the **Feature group** that contains the feature you want to machine.

5 To select only one feature from a feature group, select **Specific features**, select a feature in the graphics window, and click **Select features** .

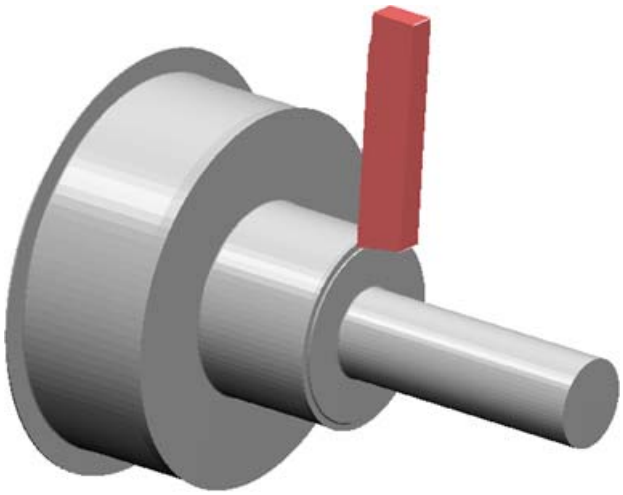
Move your cursor over the icon to display information about which features are selected.

6 Click **Calculate** to calculate the toolpath.

7 Click **OK** to close the dialog.

Simulating turning toolpaths

Simulate toolpaths to check for collisions and see the result.



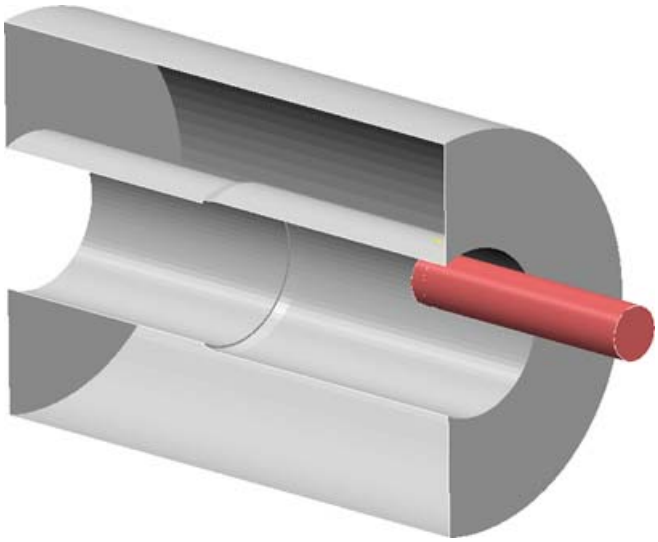
To simulate turning toolpaths:

1. In the Explorer, right-click a calculated toolpath or NC program and select **Simulate from Start**.
2. If you want to view the stock simulation, click **ViewMill on/suspend** on the **ViewMill** toolbar to enable ViewMill.

   You can simulate turning toolpaths without ViewMill.
3. In the **Simulation** toolbar, use the **Speed** control to adjust the speed of the simulation.
4. Click **Play** to start the simulation.

   *Click **Go to End** to quickly see the end result. Any collisions found are listed in the **Machine Tool Simulation Issues** dialog.*
5. When finished click **Exit** to end the simulation.

You can simulate internal boring operations by using the **View > dynamic sectioning** option.



Generating toolpaths

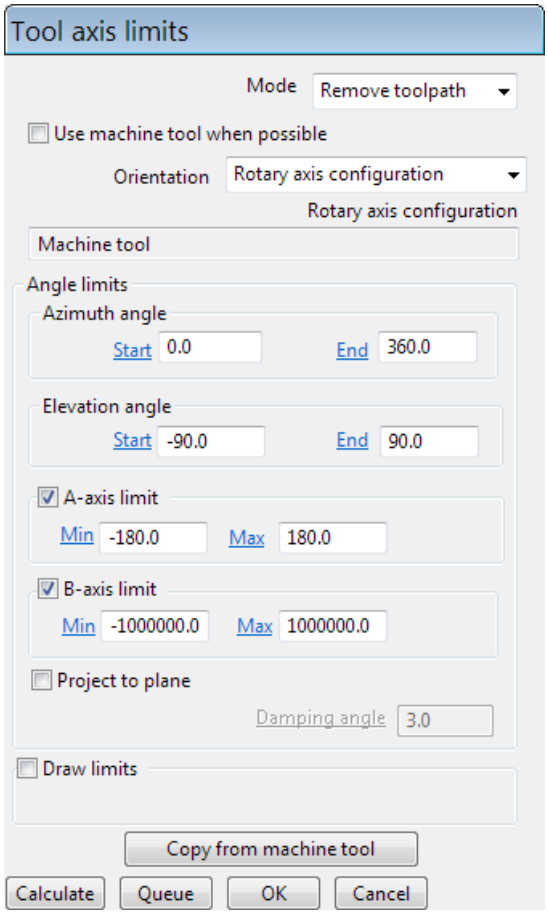
PowerMill 2017 contains the following changes and improvements to the generation of toolpaths:

- **Automatic tool axis limits** (see page 107) — You can automatically calculate the tool axis limits for a toolpath, using the machine tool information.
- **Toolpath connections** (see page 108) — You can quickly and easily define toolpath connections, rapid move clearances, and tool safe areas using the new **Toolpath connections** dialog.
- **Area clearance offset changes** (see page 121) — There are modifications to **Offset all** and **Offset model** area-clearance strategies that reduce the number of small moves required to remove upstands.
- **Rib machining enhancements** (see page 121) — You can now generate rib machining toolpaths that cut down the centreline of a rib and along its walls.
- **External thread milling** (see page 122) — There is a new **Feature external thread milling** strategy. You can now create toolpaths with multiple start points.
- **Finishing strategies enhancements** (see page 123) — There are small enhancements to the **3D Offset Finishing** and **Steep and Shallow Finishing** strategies.

Automatic tool axis limits

You can now automatically calculate the tool axis limits for a toolpath, using the machine tool information.

To enable this functionality there are enhancements to the **Tool axis limits** strategy page:



Use machine tool when possible — Deselect this option to manually calculate the tool axis limits. This option is selected by default.

Orientation — Select one of the following options from the list to define the reference frame for the tool axis limits.

- **Rotary axis configuration** — Select this option to limit the tool axes with respect to the machine tool rotary axes.
- **Manual** — Select this option to limit the tool axes with respect to a selected workplane.

A-axis limit — Select this option to apply limits to the machine tool's first rotational axis. Enter **Min** and **Max** values to define the limits.

B-axis limit — Select this option to apply limits to the machine tool's second rotational axis. Enter **Min** and **Max** values to define the limits.

*The A-axis limit and B-axis limit fields are named after the rotary axis address of the machine tool. For example if your machine tool has A and C rotary axes, the dialog displays **A-axis limit** and **C-axis limit**. The rotational axes must be orthogonal.*

Copy from machine tool — Click this button to automatically enter the minimum and maximum values of the tool axis limits using the machine tool data.

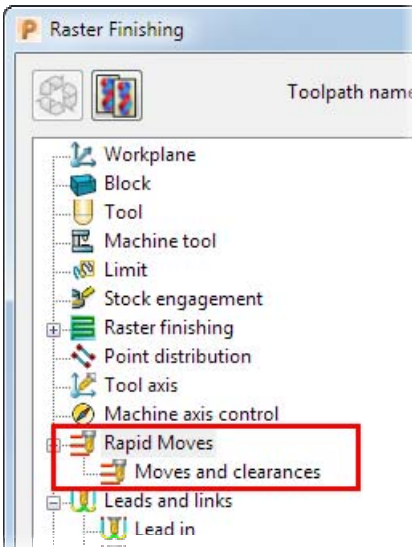
Toolpath connections

There are enhancements to the user interface and workflow associated with the generation of toolpaths that enables you to quickly and easily define toolpath connections, rapid move clearances, and tool safe areas.

To support these improvements the following dialogs have been removed and had their functionality consolidated into the new **Toolpath connections** dialog (see page 109).

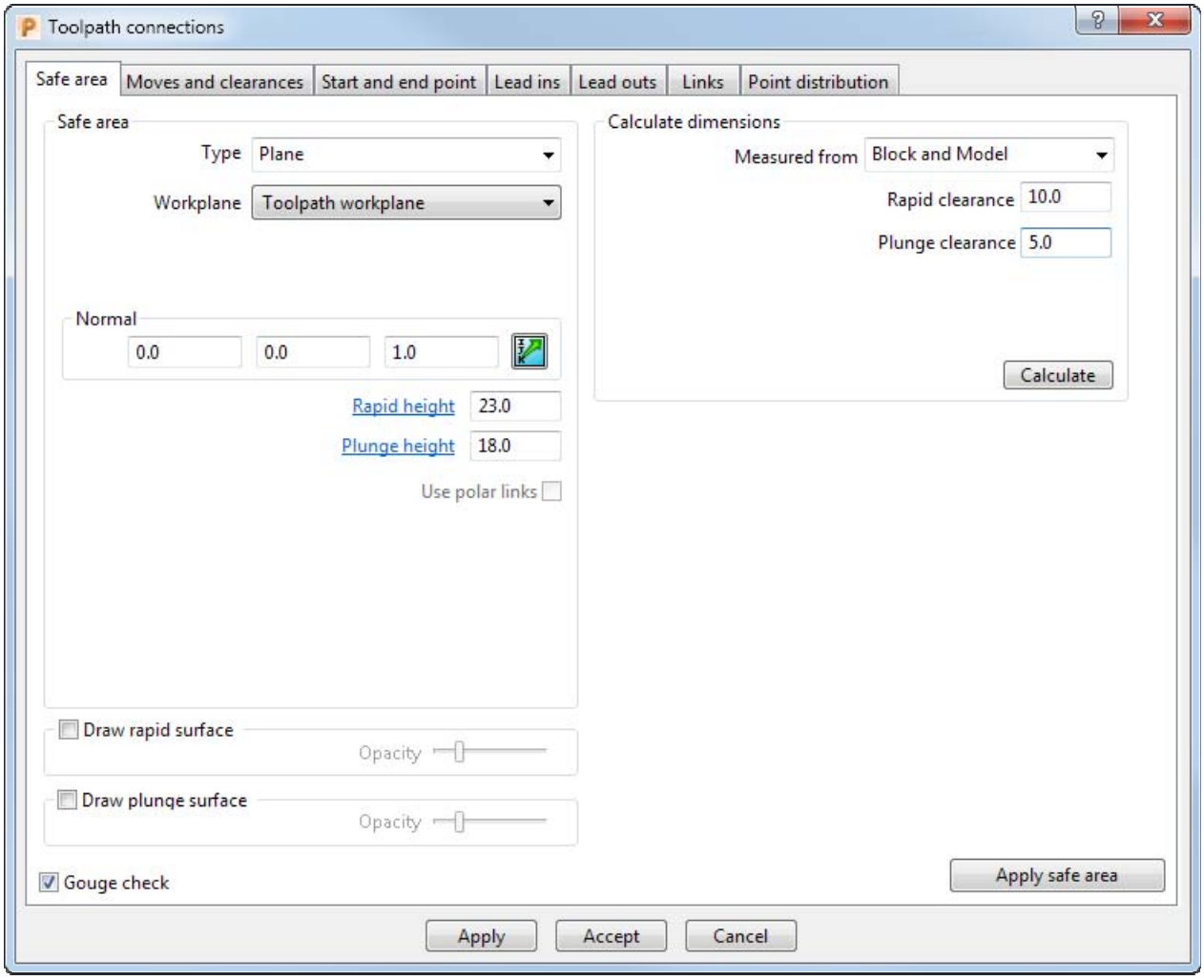
- Rapid Move Heights.
- Start and End Point.
- Leads and Links.

Additionally the **Strategy** dialog is updated with new pages reflecting the new tabs on the **Toolpath connections** dialog.



Toolpath connections dialog

Use the tabs on the **Toolpath connections** dialog to control the different properties of the toolpath connections.



Safe area — Use the options on this tab to define the size and shape of the safe area in which the tool can move at a rapid rate.

Moves and clearances — Use the options on this tab to define the limits of rapid moves and clearances.

Start and end point — Use the options on this tab to define the position and orientation of the start and end points of the toolpath.

Lead ins — Use the options on this tab to define the tool's motion before a cutting move.

Lead outs — Use the options on this tab to define the tool's motion after a cutting move.

Links — Use the options on this tab to define the link moves between cutting moves in a toolpath.

Point distribution — Use the options on this tab to define the distribution of points along the leads and links in a toolpath.

Improvements to safe areas

There are several changes across PowerMill in version 2017 that make it easier to define and use the safe areas with respect to the rapid moves in links and toolpath connections.

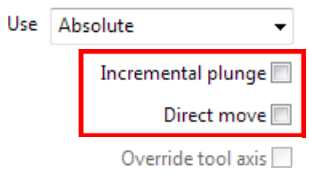
Using the options on the **Safe area** tab you can now specify:

- A length for cylindrical safe areas and whether a tool can move over the end faces of the finite cylinder (see page 111).
- Which sides of a box shaped safe area the tool is free to move over during rapid moves (see page 113).
- The reference object with respect to which the safe area dimensions are calculated (see page 115).

Using the options on the **Moves and clearances** tab you can now specify:

- The orientation of the skim plane used in planar skim links (see page 116).
- If you want to use planar skim moves for cylindrical and spherical safe areas. (see page 116)

There are two new options on the **Start and end point** tab:

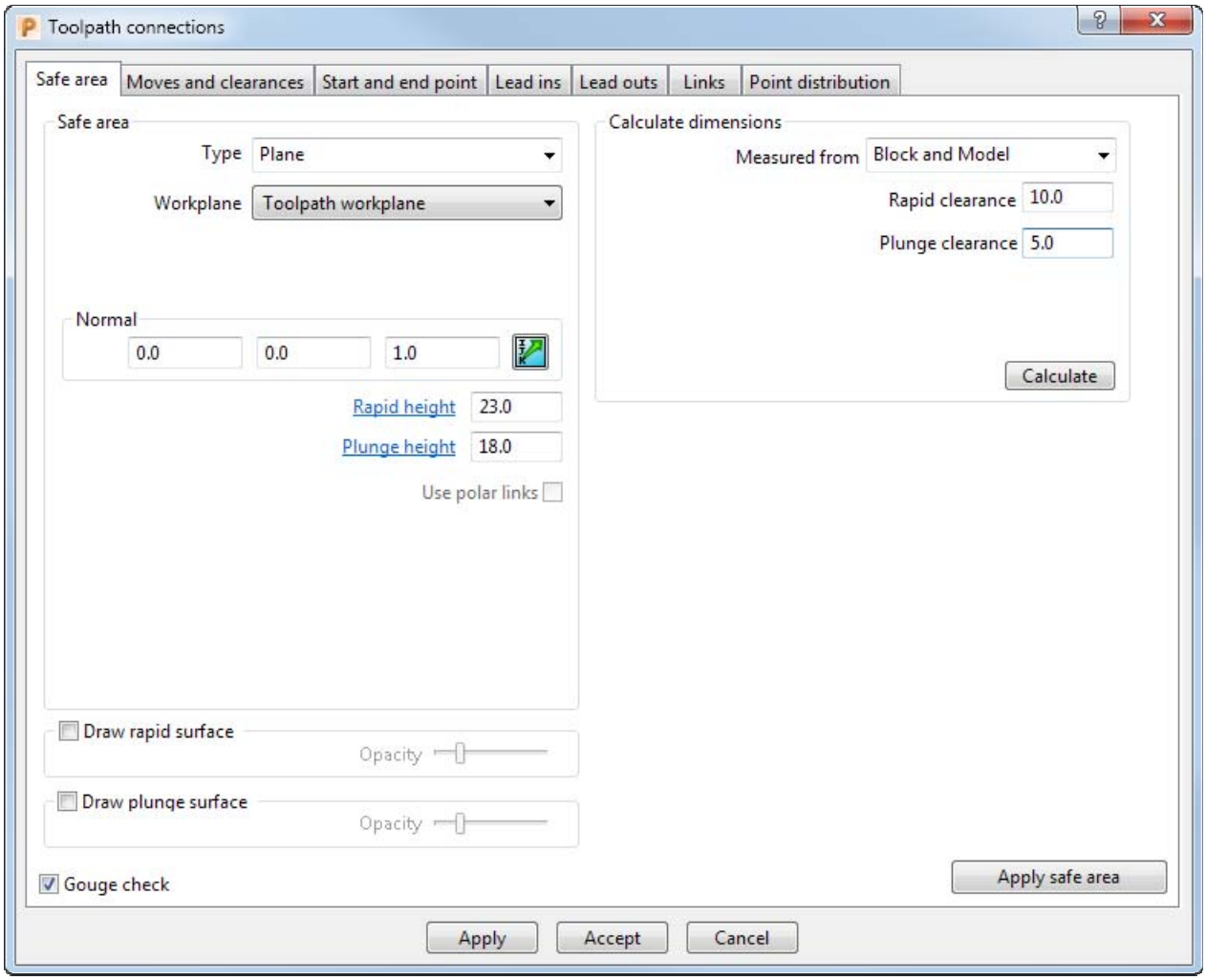


- **Incremental plunge** — Select to specify whether the plunge move at the start of a toolpath is made at an incremental distance relative to the target point. You can define the incremental plunge distance on the **Moves and clearances** tab.
- **Direct move** — Select to specify whether an absolute move from a start point or to an end point is made directly, avoiding the safe area if possible.

Changes to cylindrical safe areas

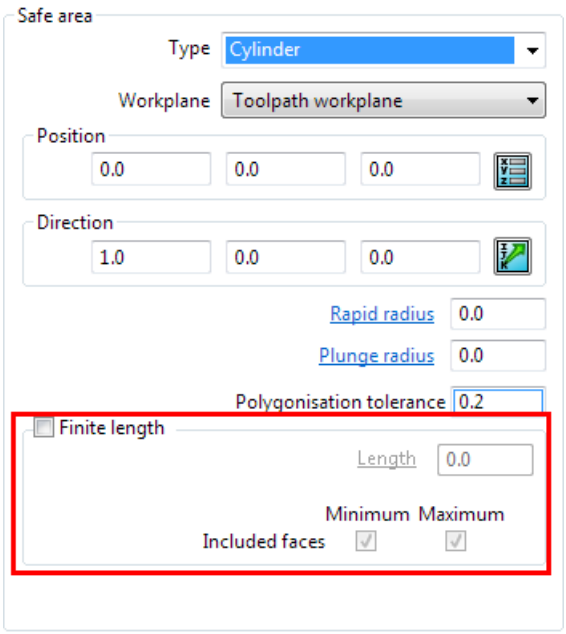
To specify the length of a cylindrical safe area and include end faces for the tool to move over:

1. Click  on the **Main** toolbar to display the **Safe area** tab on the **Toolpath connections** dialog.



2. Select **Cylinder** from the **Type** list.

The dialog is updated to display the options for a cylindrical safe area:

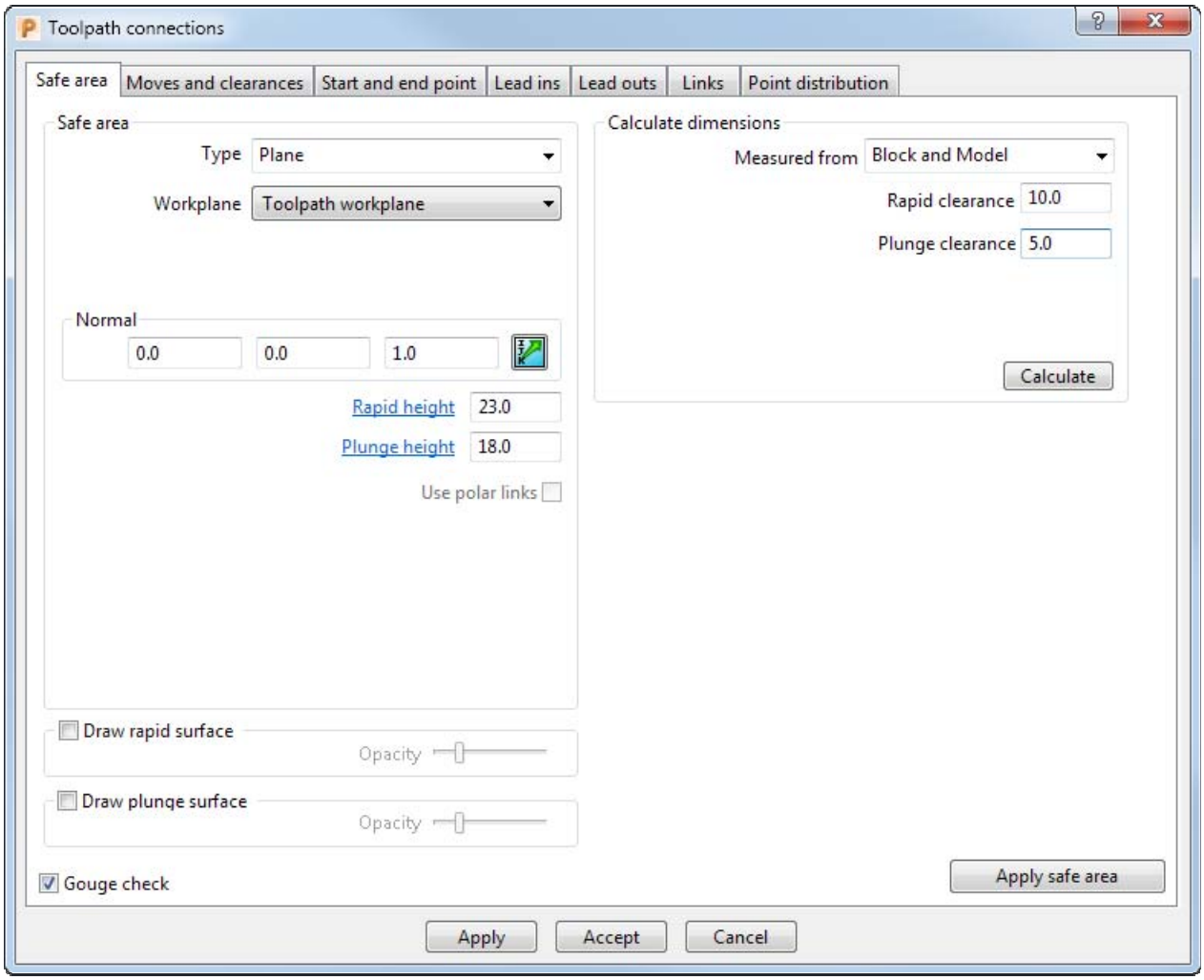


3. Select **Finite length** and enter a value in the **Length** field to specify the length of the cylindrical safe area.
4. Select or deselect the **Minimum** and **Maximum** options to specify which faces are included in the safe area:
   - **Minimum** — Allow rapid moves to cross the start face of the cylinder, defined with respect to the axial direction.
   - **Maximum** — Allow rapid moves the cross the end face of the cylinder, defined with respect to the axial direction.
5. Click **Apply safe area** to save your changes and calculate the safe area and any moves defined from it.

Changes to box safe areas

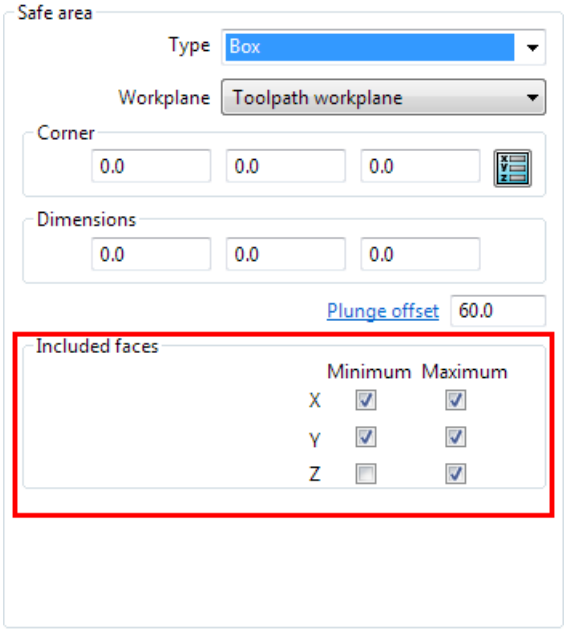
To specify which sides of a box shaped safe area the tool is free to move over

1 Click  on the **Main** toolbar to display the **Safe area** tab on the **Toolpath connections** dialog.



2 Select **Box** from the **Type** list.

The dialog is updated to display the options for a box safe area:



3 Toggle the options in the **Included faces** area to specify the sides of the box over which rapid moves are allowed. By default all faces are select except for the bottom face (-Z).

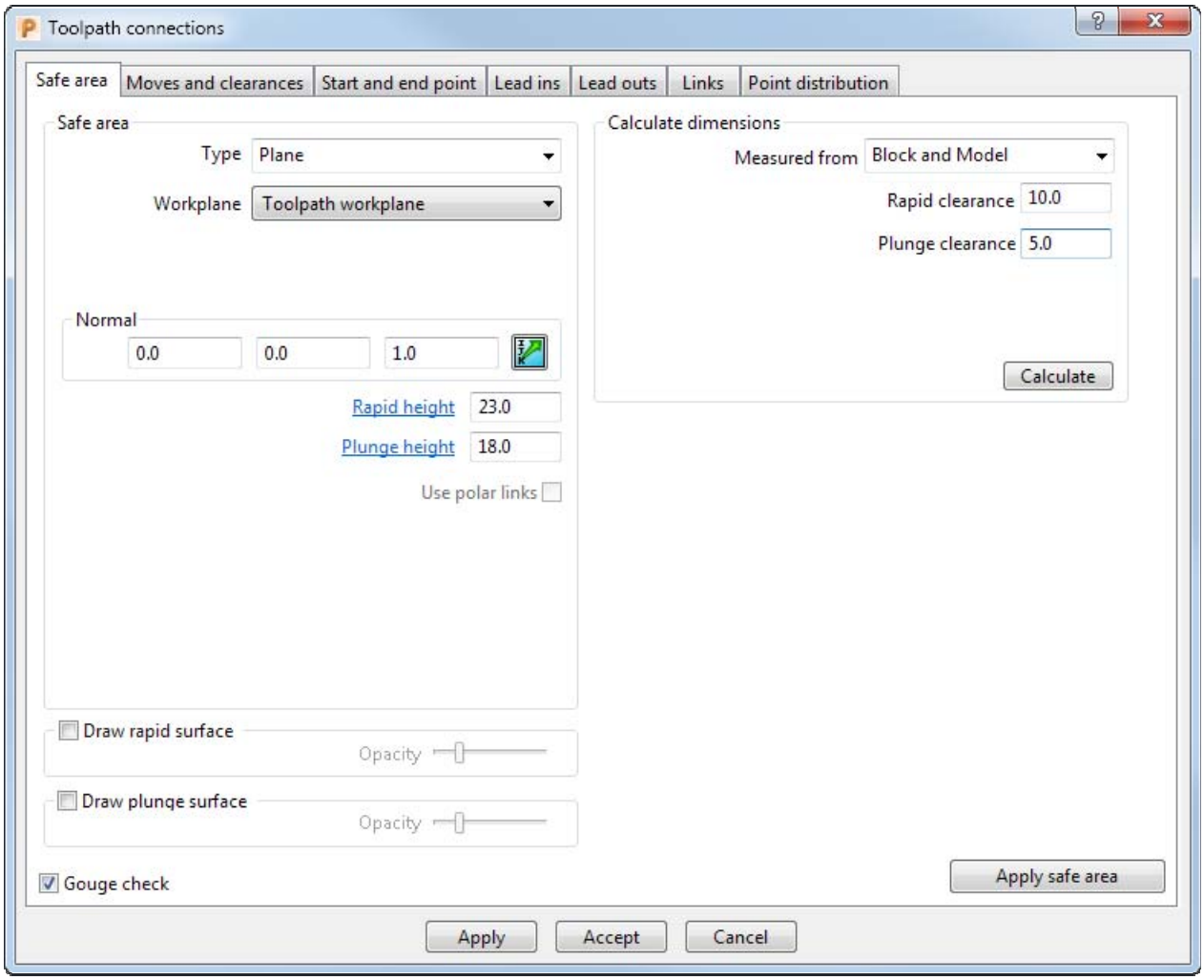
- **Minimum** — Allow rapid moves to cross the face that lies in the negative direction perpendicular to the selected axis.
- **Maximum** — Allow rapid moves to cross the face that lies in the positive direction perpendicular to the selected axis.

4 Click **Apply safe area** to save your changes and calculate the safe area and any moves defined from it.

Changes to safe area dimensions

To specify what reference object the safe area dimensions are calculated with respect to:

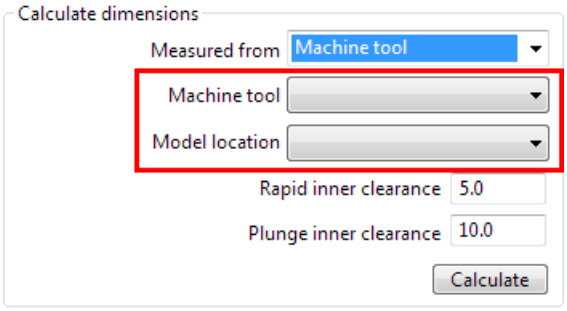
1. Click  on the **Main** toolbar to display the **Safe area** tab on the **Toolpath connections** dialog.



2. Select a reference object from the **Measured from** list:
   - **Block and Model** — The safe area is defined with respect to the box or cylindrical limits of the block and model combined. The cylindrical limits are used if the block is cylindrical and encloses the box limits of the model.
   - **Block** — The safe area is defined with respect to the box or cylindrical limits of the block.
   - **Model** — The safe area is defined with respect to the box limits of the model.

- **Machine tool** — The safe area is defined to lie within the linear travel limits of a specified machine tool. The tool axis is assumed to be aligned with the Z-axis of the **Model location** workplane.

If you select **Machine Tool** the area updates with new options:



3. Select a machine tool from the **Machine tool** list.
4. Select a workplane from the **Model location** list.
5. Click **Apply safe area** to save your changes and calculate the safe area and any moves defined from it.

Changes to planar skim moves

You can now manually specify the orientation of the skim plane when making planar skim links. This is useful if you want to create skim links that are more suitable or efficient.

To specify the orientation of the skim plane:

1. Click  on the **Main** toolbar to display the **Safe area** tab on the **Toolpath connections** dialog.
2. Click the **Moves and Clearances** tab.
3. In the **Planar skim moves** area select an orientation from the **Plane** list:
   - **Automatic** — The skim plane is normal to the Z direction of the toolpath.
   - **Interpolated** — The skim plane is normal to the vector halfway between the directions of the retract move and the reverse of the approach direction.
   - **Safe area** — The skim plane is parallel to the safe area. This option is only available for toolpaths with a planar safe area.
   - **Workplane Z** — The skim plane is normal to the Z direction of an existing workplane, selected from the **Workplane** field.
4. Click **Apply rapid moves** to save your changes and calculate the first approach, final retract and all links that have rapid moves with respect to the safe area and the new rapid move settings.

You can now use planar skim moves with non-planar safe areas.

To use planar skim moves with a non-planar safe area:

1. Click  on the **Main** toolbar to display the **Safe area** tab on the **Toolpath connections** dialog.
2. Select a non-planar safe area from the **Type** list. For example, **Cylinder**.
3. Click the **Moves and Clearances** tab.
4. In the **Planar skim moves** area select **Use with a non-planar safe area**.
5. Select an orientation from the **Plane** list.
6. Click **Apply rapid moves** to save your changes and calculate the first approach, final retract and all links that have rapid moves with respect to the safe area and the new rapid move settings.

Improvements to links

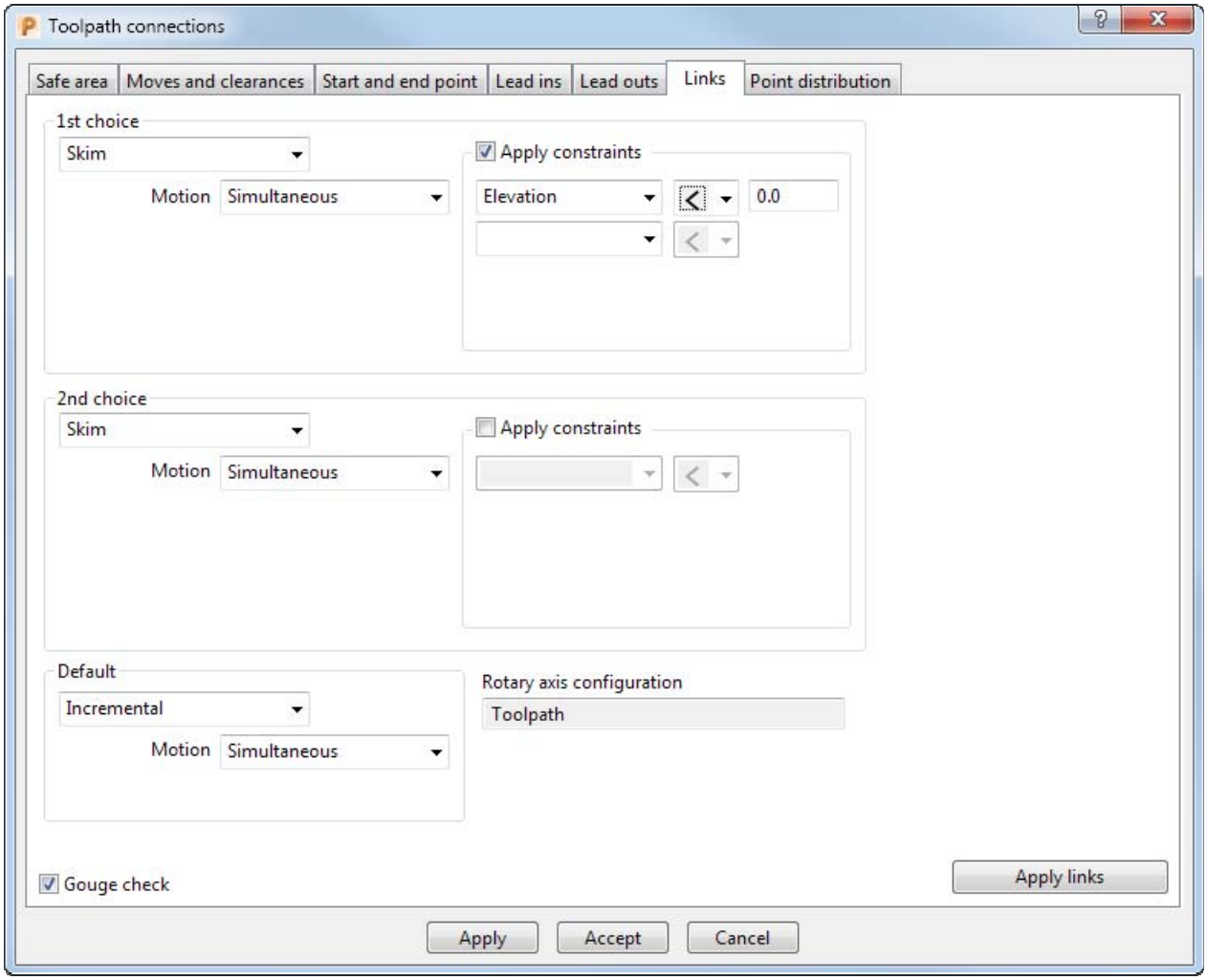
There are changes to the interface and workflow with respect to specifying how to make the link moves between the cutting moves in a toolpath. In previous versions of PowerMill you would specify a threshold value to determine whether a link was long or short, with each having their own properties

In PowerMill 2017 links are specified as a first choice link, second choice link, and a default link. You can apply constraints to the links to define the circumstances in which they are used. If the constraint criteria of the first choice link are not satisfied, or if they are satisfied but could not be applied, for example, it is unsafe, PowerMill tries to apply the second choice link. If the constraint criteria of the second choice link are not satisfied PowerMill applies the default link.

You can choose to create links with no constraints.In this instance, if it is possible, PowerMill applies the chosen link type to appropriate parts of the toolpath. If **Gouge check** is selected the **1st choice** and **2nd choice** links are only created if they are safe, otherwise the **Default** link is used.

If you choose to create links with no constraints and **Gouge check** is not selected, then the **1st choice** link will be used in all cases, regardless of safety.

Use the **Links** tab on the **Toolpath connections** dialog to specify your link options:



1st choice — Select the type of link moves from the list for your first choice. PowerMill applies this link where its constraint criteria are met.

2nd choice — Select the type of link moves from the list for your second choice. PowerMill applies this link if the constraint criteria for the **1st choice** links are not met.

Default — Select the type of link moves from the list for you default choice. PowerMill applies this link if the constraint criteria for both the **1st choice** and **2nd choice** links are not met, or if they gouge.

Motion — Select an option from the list to specify how the tool axis changes as it moves across the link. These options are only available whenever the tool axis may change along the toolpath.

Simultaneous — The tool axis is free to move along the link.

Move then rotate — The behaviour is dependent on the path:

- *Rapid surface paths* — The tool moves to the position above its target within the rapid surface, then changes the tool axis when it leaves the rapid surface.
- *Direct paths* — The tool axis is fixed until it reaches the end of the link.

Rotate then move — The behaviour is dependent on the path:

- *Rapid surface paths* — The tool axis changes when the tool reaches the rapid surface, it then moves into the next position.
- *Direct paths* — The tool axis changes at the start of the link.

Rapid surface normal — The tool axis is fixed perpendicular to the rapid surface as it moves across it. This option is only available for toolpaths that travel through the rapid surface.

Reset machine tool — The tool axis is aligned to the machine tool's inital orientation across the course of the link.

Rotation point — The tool axis is fixed as it moves through the safe area to the origin of the specified workplane. The tool axis then rotates and moves to the next link.

Apply constraints — Select this option to apply constraints to your links and then select a constraint from the list. Selecting a constraint from the list creates an additional field to apply further constraints. **1st choice** and **2nd choice** links have a maximum of 4 constraints. If more than one constraint is selected they must all be satisfied for a link to be applied.

- **Distance** — Constrain the link by its distance spanned.
- **Surface slope** — Constrain the link by the slope angle of the surface at either end of the link.
- **Angular change** — Constrain the link by the angular change of the tool axis.
- **Azimuth** — Constrain the link by the change in the azimuth angle of the tool axis.
- **Elevation** — Constrain the link by the change in the elevation angle of the tool axis.

 — Select an option from the list to define the constraint limit:

< — The constraint is satisfied if the parameter is less than the specified value.

> — The constraint is satisfied if the parameter is greater than the specified value.

Rotary axis configuration — Displays the current rotary axis configuration. This is only visible if you select constraints of **Azimuth** and **Elevation**.

Gouge check — Select this option to automatically check links for gouges. If this option is selected and a **1st choice** or **2nd choice** link gouges the part then the **Default** link is used instead.

Links example

The following example demonstrates creating arc links that are restricted to shallow regions of the model.

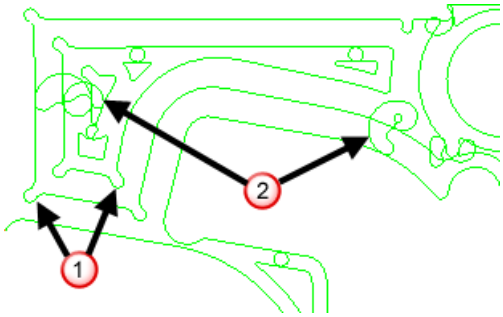
1. Click  on the **Main** toolbar to display the **Lead ins** tab of the **Toolpath connections** dialog.
2. Click the **Links** tab.
3. Select **Circular arc** from the **1st choice** list.
4. Select **Surface slope** from the constraint list.
5. Select < from the list.
6. Enter a value of **20**.
7. Click **Apply links**.

In this example circular arc links are created in areas where the surface slope at either end of the link does not reach a value greater than 20 degrees. Since no constraints were applied to the **2nd choice** link, any links that are created in areas where the surface slope is greater than 20 degrees are created as **2nd choice** links, unless they gouge, in which case they are created as **Default** links.

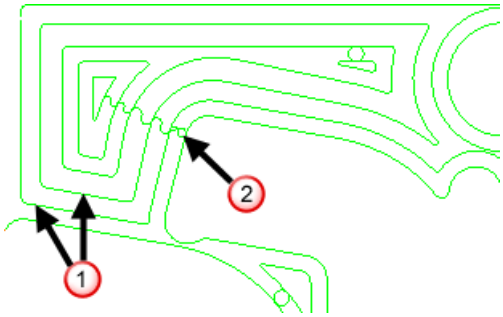
Area clearance offset changes

There are modifications to **Offset all** and **Offset model** area-clearance strategies. When you specify a stepover distance greater than the tool radius PowerMill now incrementally reduces the toolpath offset to minimise the number of small movements to remove upstands (1). Additionally, the connections between toolpaths have been optimised (2). These changes reduce the consumption of tool inserts at the cost of marginally increased cutting times.

PowerMill 2016



PowerMill 2017

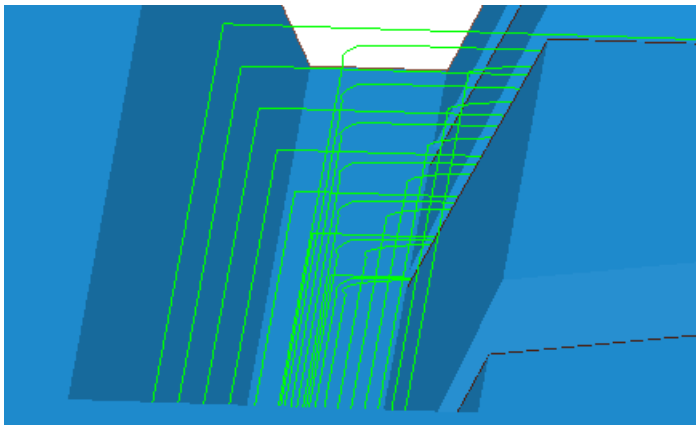


Rib machining enhancements

There are enhancements to rib machining:

- There is a new **Centre and walls** option in the **Style** list on the **Rib machining** page of the **Strategy** dialog.

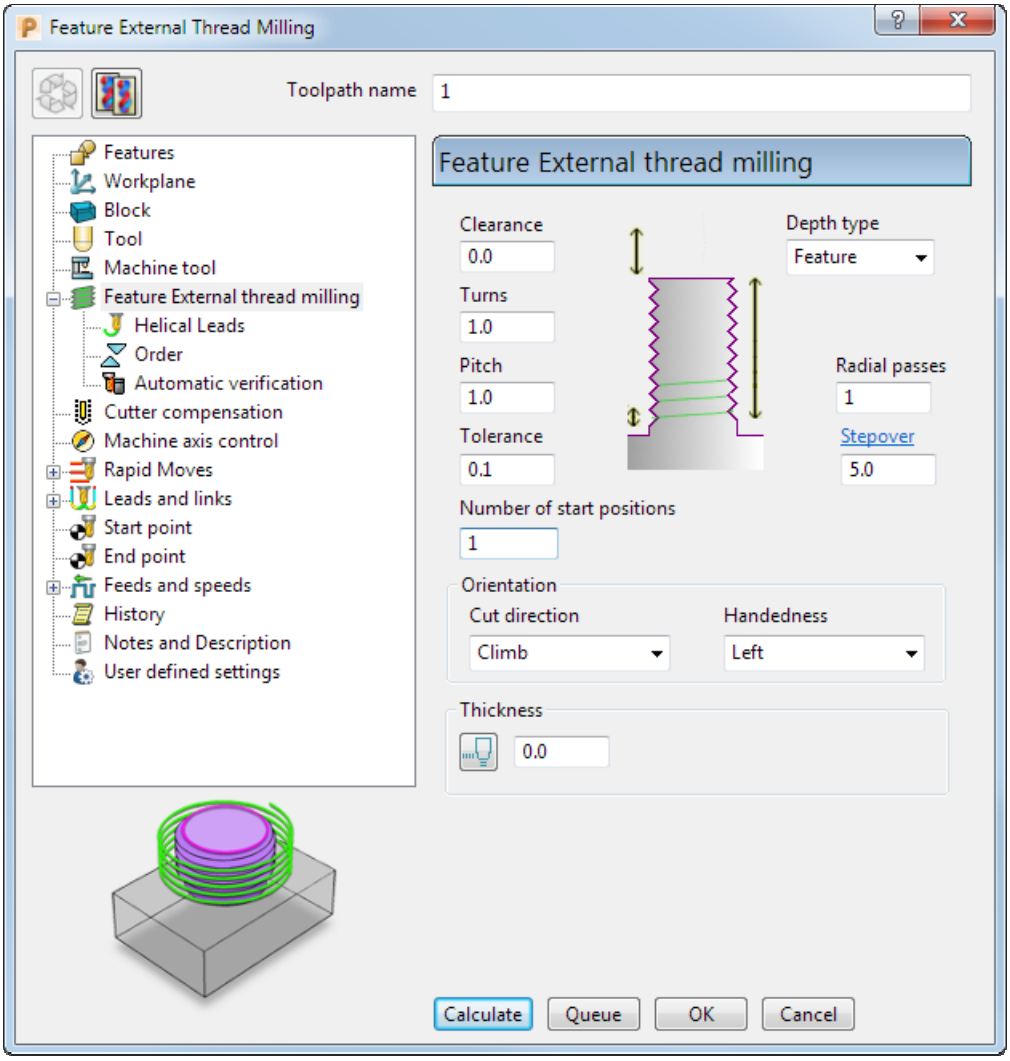
  Select this option to create a toolpath that machines down the centreline of the channel and then machines down each side of channel.



- PowerMill now correctly handles intersecting ribs at different heights for rib machining toolpaths.

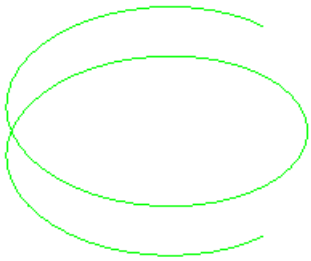
External thread milling

There is a new **Feature External Thread Milling** strategy on the **Feature Machining** page of the **Strategy Selector** dialog which replaces the old **External thread milling** strategy on the **Drilling** page.

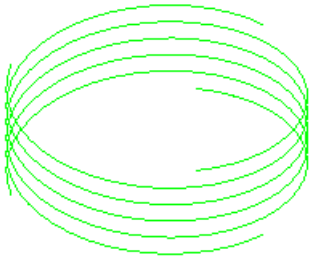


You can now create toolpaths with intertwined threads by specifying the **Number of start positions**. Enter a value to create identical toolpaths that are rotated about the tool axis and spaced equidistantly.

Toolpath with one start position



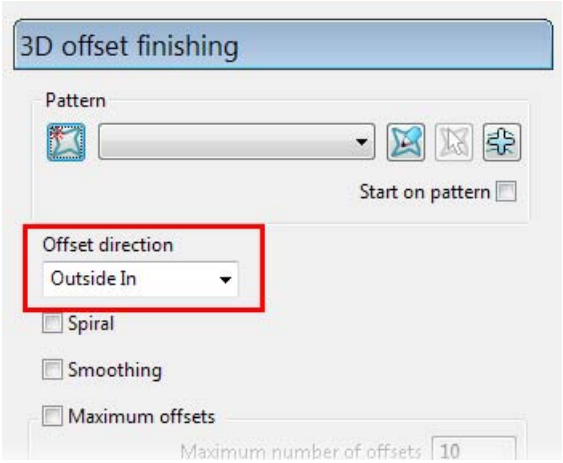
Toolpath with three start positions



Finishing strategies enhancements

3D offset finishing

You can now specify which direction the toolpath offset is calculated from.



From the **Offset direction** list, select:

- **Outside In** — Calculates the toolpath offset from the outside in.
- **Inside Out** — Calcualtes the toolpath offset from the inside out.

Steep and shallow finishing

You can now add a perpendicular pass to steep and shallow finishing strategies that use a raster style. The **Raster** sub-page of the **Steep and shallow finishing** page contains the following new options:

- **Perpendicular pass** — Select to define a second raster pass perpendicular to the first one.

- **Shallow angle** — Enter an angle to specify that the raster pass machines only the areas of the model that are steeper than this angle.
- **Optimise parallel pass —** If a raster toolpath is created with a parallel and perpendicular pass, and with a shallow angle greater than 0_, select this option to trim the parallel pass so it does not machine the areas that the perpendicular pass machines.

Simulating toolpaths

PowerMill 2017 contains the following changes and improvements to the simulation of toolpaths:

- **Simulation pausing (see page 126)** — You can now control how often and on what types of issues simulation stops.
- **Simulation playback** (see page 128) — There are new controls on the **Simulation** toolbar.
- **Simulation toolbar** — The **Simulation issues**  button is now on the **Simulation** toolbar. There is a new **Collision checking**  button on the **Simulation** toolbar. Toggle this button to turn collision checking on or off.

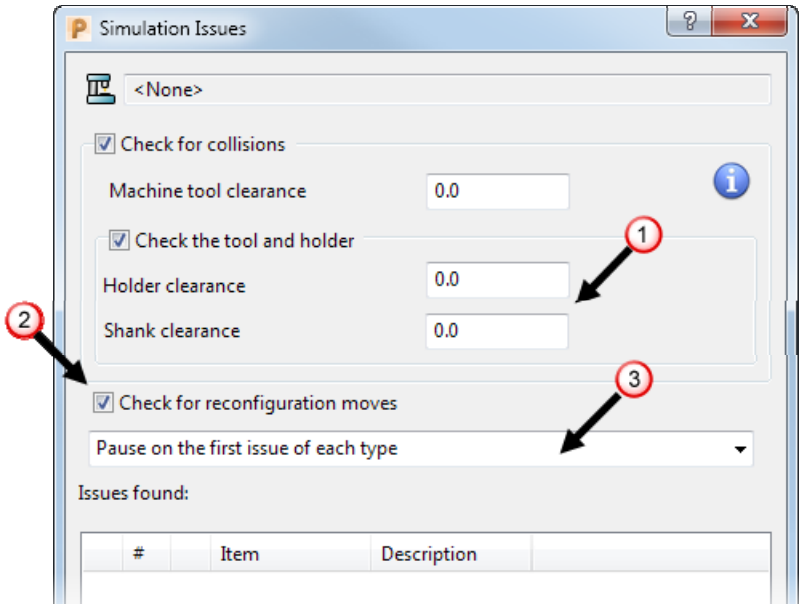
Simulation pausing

There are enhancements to the pausing of simulations.

There are new options to control:

- what type of issues simulations stop on.
- how often simulations stop on issues.

To support these improvements there are changes to the **Simulation Issues** dialog:



1 Enter values for the following:

- **Holder clearance** — Specify the minimum distance, between the tool holder and the model or machine tool, before a clearance issue is generated.
- **Shank clearance** — Specify the minimum distance, between the tool shank and the model or machine tool, before a clearance issue is generated.

2 **Check for reconfiguration moves** — Select this option to include reconfiguration moves as an issue to stop simulation. If this option is unselected then reconfiguration moves do not cause the simulation to be paused and are not added to the issues list. This option is selected by default.

3 — Select an option from the list to specify how often the simulation is paused when encountering issues:

- **Always pause on issues** — Simulation stops for all issues encountered.

- **Pause on the first issue of each type** — Simulation stops on the first issue of any type encountered. If you resume simulation it will not stop again on an issue of the same type. This option is selected by default.
- **Never pause on issues** — Simulation does not stop for any issues encountered. Encountered issues are still logged in the **Issues found** list.

Simulation playback

There are enhancements to the simulation playback controls and functionality. To support these improvements there are changes to the **Simulation** toolbar.

Select an option from the new **Item** flyout (1) to specify how the step and play buttons behave:

- Tenth of a Move — Play or step to the next tenth of a point.
- Point — Play or step to the next point.
- Five Points — Play or step five points.
- Fifty Points — Play or step fifty points.
- Component — Play or step to the next component. Components are specified as: segments, leads, links, approaches, retracts, and connections.
- NC item — Play or step to the next NC item.

Select an option from the new **Play to** flyout (2) to specify how far the simulation plays:

- **Play** — Select this option to play the simulation to the end.
- **Play item**— Select this option to play the simulation to the end of the current item. The item is defined by your selection from the **Item** flyout.

To change the simulation mode select **Tools > Options** and then select **Simulation > Simulation Mode**. The following options are available:

- **Feed rate** — Select this option to simulate at a speed relative to the feed rate.
- **Points** — Select this option to simulate at a speed relative to a constant number of toolpath points per second.
- **Distance** — Select this option to simulate at a constant speed relative to the distance moved by the tool.

General enhancements

PowerMill 2017 contains the following general improvements:

- **ViewCube** (see page 129) — Use the new ViewCube to interactively orientate the contents of the graphics window.
- **Live text creation** (see page 131) — You can now create text as wireframe, for engraving or leaving comments in a project.
- **Autodesk A360** (see page 134) — The **Tools** options includes an **Autodesk A360 option**.
- **Watertight stock models** — You can now export watertight stock models as .stl or .dmt files.
- **Save strategy parameters** — The parameters you can individually select to save with a strategy are updated to reflect the toolpath connection changes.

ViewCube

The ViewCube is a new feature in PowerMill, which enables you to change and identify the viewpoint of the Graphics window. By clicking the ViewCube's corners, faces, edges, and icons, you can use it to directly manipulate the view in the Graphics window. In addition, when you re-orient the view using a cube option, toolbar button, or keyboard shortcut, the ViewCube automatically reflects the new viewpoint.

The ViewCube is displayed in the upper-right corner of the Graphics window. Click and drag the cube to re-orient the view in any direction. Alternatively, click:

- a face to show it as an orthogonal view.

- an edge to show the adjacent faces.
- a corner to show the three adjacent faces.

In addition, when you move the cursor near the cube, the Home icon is displayed. Click the icon to show the Home view.

Manipulating orthogonal views

When you select a single-face view and move the cursor near the cube, control icons are displayed:

Click:

- to show the view of an adjacent face.
- to rotate the view clockwise through 90 degrees.
- to rotate the view counter-clockwise through 90 degrees.

Configuring the ViewCube

To control the behaviour and appearance of the ViewCube, right-click the cube and choose a menu option. Select:

- **Set current view as home and keep scale** to save the current orientation and magnification of the model as the Home view.
- **Set current view as home and scale to fit** to save the current orientation of the model as the Home view, and scale it to fit the Graphics window.
- **Options** to display and change the settings of the ViewCube.

Setting ViewCube options

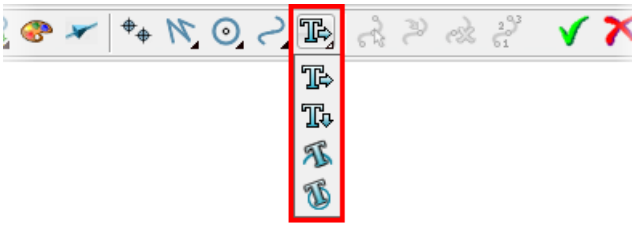
To modify the behaviour and appearance of the ViewCube, use the following options:

- **Show the ViewCube** — Select this check box to show the ViewCube in the Graphics window. Deselect the check box to hide it.
- **ViewCube size** — Select an option from the list to choose the size of the ViewCube in the Graphics window. Alternatively, select **Automatic** to resize the cube when you resize the Graphics window.
- **Keep model upright** — Select this check box to prevent the view from being inverted. If you click an edge, corner, or face that would leave the view upside down, the Graphics window switches to the selected view and then rotates to leave the view upright.

Live text creation

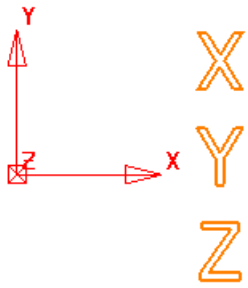
You can now create text as wireframe, for engraving or leaving comments in a project, using the new text creation tools on the **Curve Editor** mode-toolbar.

There is a new flyout on the **Curve Editor** mode-toolbar with the following buttons:



- **Horizontal text** — Click to create text in the positive horizontal direction of the principal working plane.

- **Vertical text** — Click to create text in the negative vertical direction of the principal working plane.



- **Text on a curve** — Click to create text along a curve.

- **Text on a circle** — Click to create text on a circle.

Create horizontal text example

To create horizontal text:

1. Right-click **Patterns** in the explorer and select **Curve Editor...** from the **Patterns** context menu.

   The **Curve Editor** mode-toolbar is displayed.

2. Click the **Horizontal text** button from the text creation flyout.

   The **Text** toolbar is displayed (see page 133).

3. Click in the graphics window to set the origin of the text.
4. Type your text.

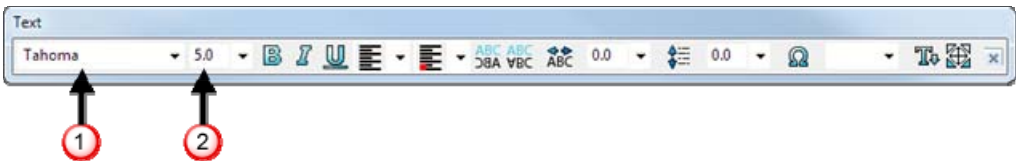
*To change the colour of the text to be visible on your background select **Tools > Customise Colours** and choose **Curve Editor > Edit Text**.*

5. Use the options on the **Text** toolbar to modify your text.

6 Click **Accept changes** on the **Curve Editor** mode-toolbar to save the created text as wireframe.

Text toolbar

Use the options on the text toolbar to modify your text:



1 — Select an option from the list to specify the text font.

Single line fonts are available when PowerMill Modelling is installed.

2 — Enter a value to specify the size of the text.

— Click to apply bold typeface to the text.

— Click to apply italic typeface to the text.

— Click to apply an underline to the text.

— Select an option from the list to specify the text justification.

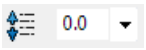
— Select an option from the list to specify the position of the text with respect to its origin. This option is only available for **Horizontal** and **Vertical** text.

— Select an option from the list to specify the position of the text with respect to the curve. This option is only available for **Text on a curve** and **Text on a circle**.

— Click to flip the text horizontally.

— Click the flip the text vertically.

0.0 — Enter a value to specify the character spacing.

 — Enter a value to specify the line spacing. This option is only available for **Horizontal** and **Vertical** text.

 — Enter an Alt code to insert special characters.

 — Click to select a curve in the graphics window. This button is only available for **Text on a curve**.

Radius — Enter a value to specify the radius of the circle. This option is only available for **Text on a circle**.

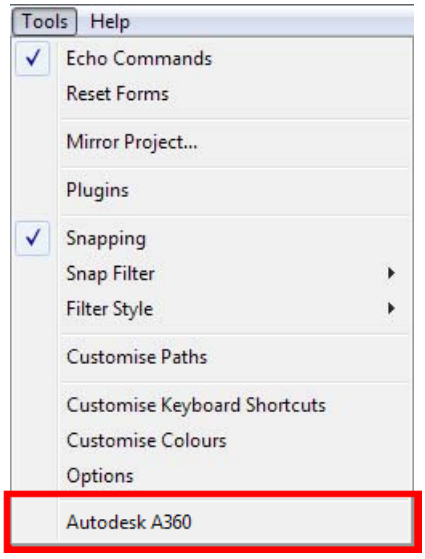
 — Click to convert horizontal text to vertical text. This option is only available for **Horizontal** and **Vertical** text.

 — Click to convert vertical text to horizontal text. This option is only available for **Horizontal** and **Vertical** text.

 — Click to graphically move the text origin in the graphics window.

Autodesk A360

The **Tools** menu includes an **Autodesk A360** option, which displays the Autodesk A360 website. Autodesk A360 is a cloud-based resource, which enables you to interact with other people and share information about projects. For example, you can use it to upload CAD model files so others can view your ideas and provide feedback.



The keyboard shortcut for this menu option is **Alt+T+A**.

Index

P

R

S

T

V

W