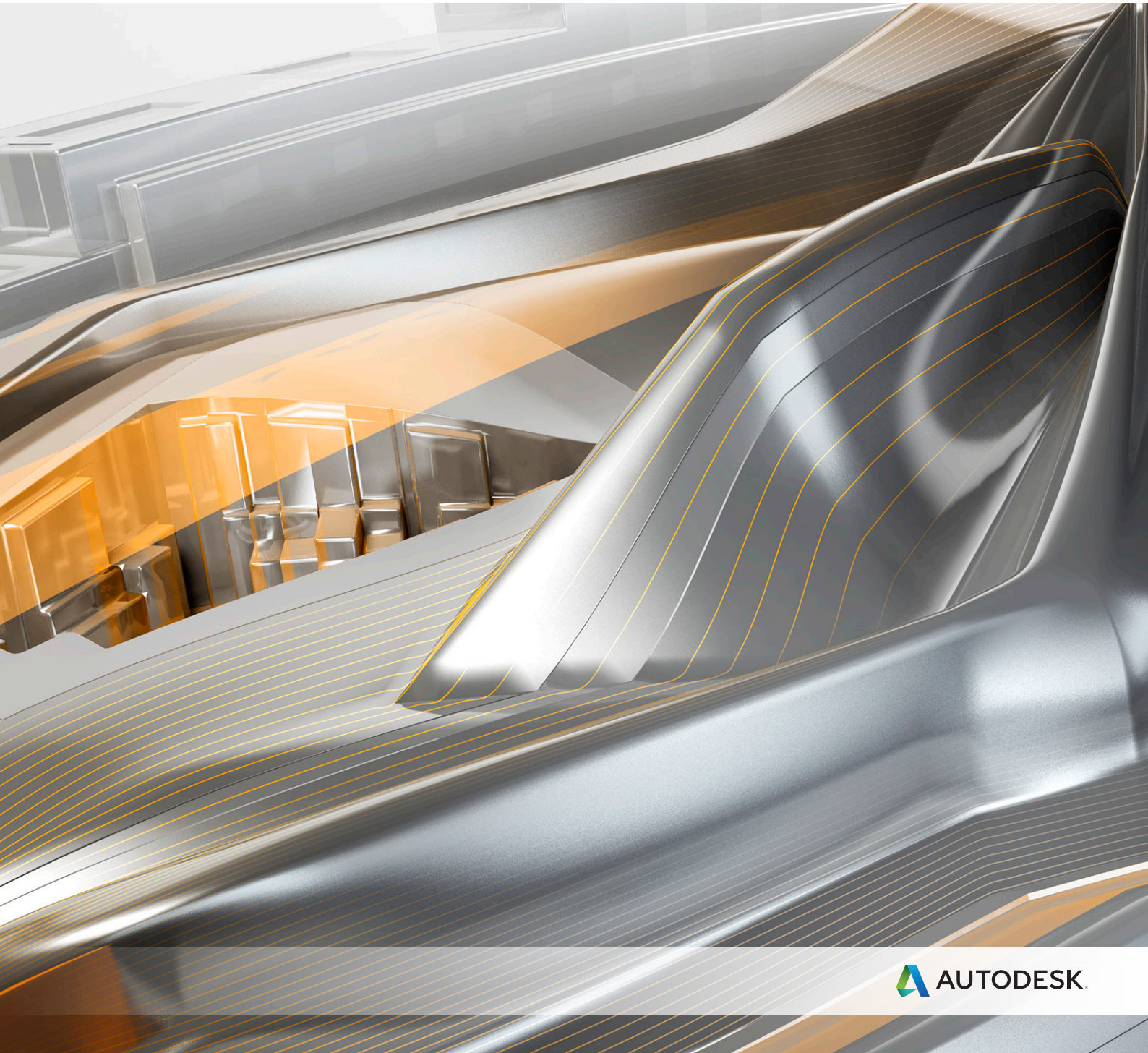
AUTODESK® **POWERMILL**®

The expert high-speed and multi-axis solution

Training Course



AUTODESK.

Copyright © Autodesk

PowerMill 2017 Five Axis Contents

Chapters	Page Number
Day 1	
3+2 Axis Machining and Drilling	1.1 - 1.36
Positional Tool Moves	2.1 - 2.12
Five Axis Tool Alignment	3.1 - 3.36
Surface Projection Finishing	4.1 - 4.16
Day 2	
Embedded Pattern Finishing	5.1 - 5.8
Five Axis Swarf Machining	6.1 - 6.26
Flowline Finishing	7.1 - 7.12
Auto Collision Avoidance	8.1 - 8.8
Tool Axis Limits	9.1 - 9.24
Tool Axis Editing	10.1 - 10.8
Day 3	
Machine Tool Simulation	11.1 - 11.14
Machine Axis Control	12.1 - 12.16
MTD files	13.1 - 13.14
Four Axis Rotary Machining	14.1 - 14.6
Port Machining	15.1 - 15.12
Blisk Machining	16.1 - 16.10
Five-Axis Tutorial	17.1 - 17.12

Copyright © Autodesk

Copyright © Autodesk

1. 3+2 Axis Machining

Introduction

3+2 Axis Machining involves indexing the head and\or bed to re-align the tool at a fixed angular, orientation, to perform machining strategies. The **3+2** alignment is actioned either by manual adjustment or as part of the CNC control.

The most common method of creating a **3+2 Axis** is by using a **Workplane** with the **Z-Axis** arranged at an appropriate ***Tool Axis*** alignment. Standard **3 Axis** strategies and their **plunge** and **retract** moves will occur relative to the **Workplane**.

3+2 Axis machining enables:-

a. All faces of a component (except the base) can be machined in one set-up, relative to a common tool setting datum.

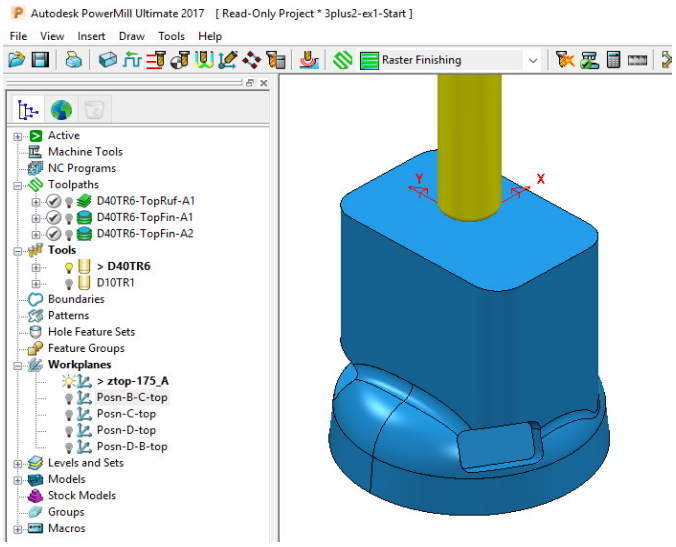
b. Access to deep sidewalls with standard length tooling.

c. Access to undercut features.

3+2 Axis - Machining Example

1. **Open** the following **Project**:

   ...\PowerMill_Data\five_axis\3plus2_as_5axis\3Plus2-ex1-Start



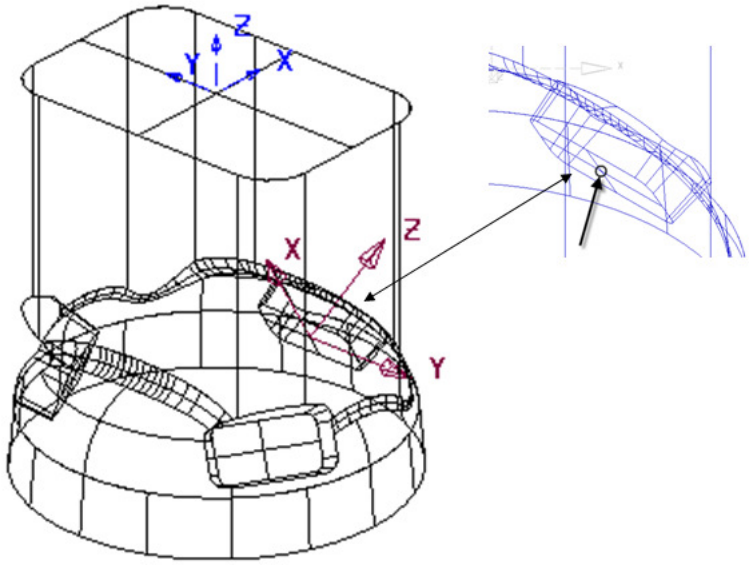
2. Save the Project as:

   ...\COURSEWORK\PowerMill-Projects\3Plus2-ex1

*A **3-Axis Roughing** strategy and two **3-Axis Finishing** strategies already exist in the imported **Project**.*

*Also, a series of **workplanes** are already included in the **Project**, with one just above the top of the model at **Ztop-175_A** at 750mm above the **Transform**. This **workplane** will be used as the main machining datum.*

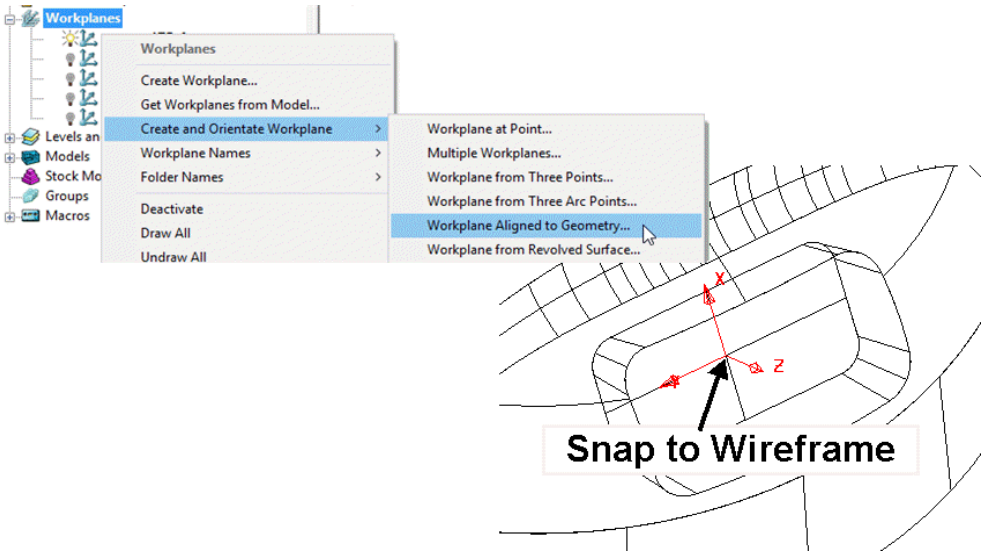
*Note the relatively high sides of the component and the orientation of the three recesses making it impossible to machine them as **3 Axis** with the tooling aligned to the vertical axis.*



The pocket located along the **X direction** will be the first to be assigned with a new **workplane** to control one of the **3+2** - **Tool Axis** alignments.

3 From the **PowerMill *explorer*** select **Workplane** > **Create and Orientate Workplane** > **Workplane Aligned to Geometry**.

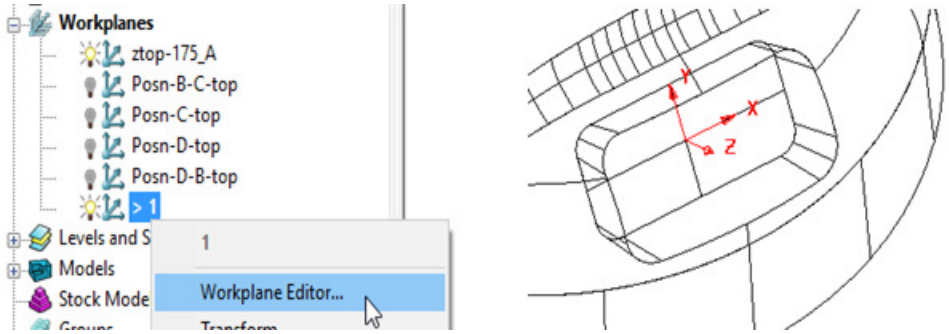
4 Using the left mouse button, snap or box the wireframe crossover at the base of the first pocket (located along **X** from **workplane *ztop-175_A***).



5 Activate the new **Workplane 1**.

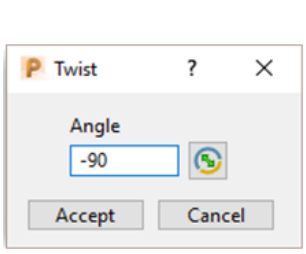
Copyright © Autodesk

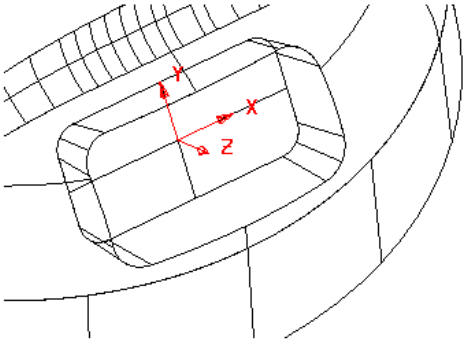
The **workplane** is automatically aligned to the wireframe with the **Z Axis** normal to the **surface**. It still requires further editing as it is advisable for the **X-Axis** to align anticlockwise as viewed from the top of the component



6 From the **PowerMill *explorer***, right-click on **Workplane 1** and from the local menu, select the **Workplane Editor** toolbar to open the following toolbar.

7 Select **Twist about Z** and in the dialog, input an **Angle** of **-90** (normal to the base of the recess).





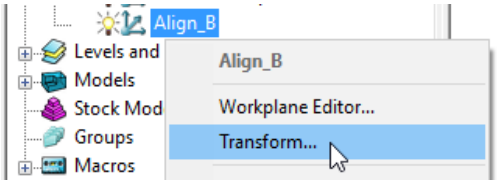
The **X-Axis** of the **workplane** should be pointing anticlockwise relative to the **Transform** (Global Datum) as viewed from the top of the component (If not already the case).

8 Select to save the changes and exit the **Workplane Editor** toolbar.

9 Rename **Workplane 1** as **Align_B**.

10 Activate the **Workplane** - **ztop-175_A**.

11 Right-click on the **Workplane, Align_B**, and from the local menu select Transform.

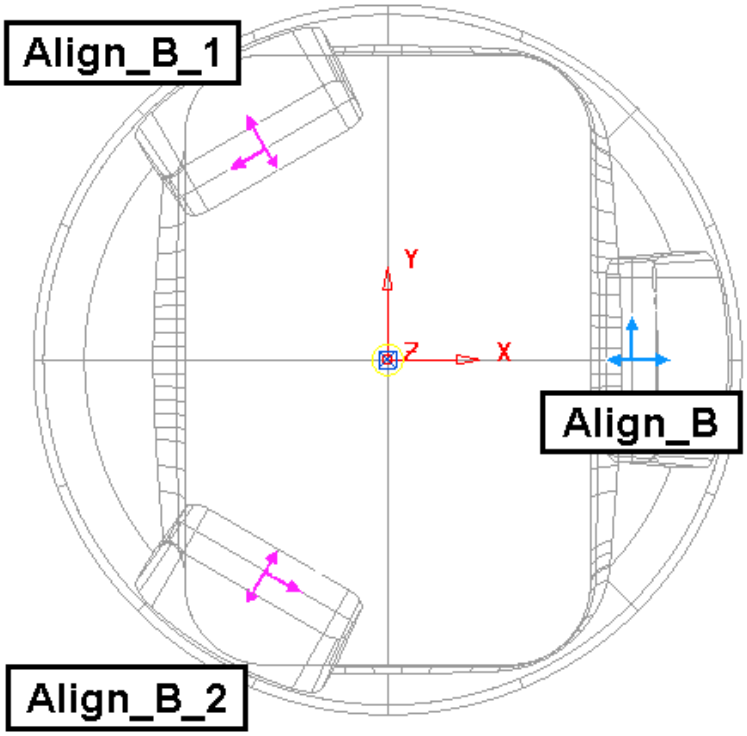


12 Select the **Rotate workplane**s option (shown arrowed below).

13 In the **Rotate** dialog, select **Keep original**.

14 Input **No. of Copies 2**, **Angle 120**, to create two new **workplanes** (Positioned normal to the base of the next pocket in an anticlockwise direction as viewed from the top).

15 Select the ***small cross*** on the right of the toolbar to exit the dialog.



16 Select at the end of the **Workplane Editor** toolbar to accept the new **Workplanes** (**Align_B_1** & **Align_B_2**).

17 Rename the 2 new **Workplanes** (in anticlockwise order) as **Align_C** and **Align_D** respectively.

The component is now ready to have the **3 Plus 2** machining strategies created, relative to the three new workplane alignments. (***Align_B***, ***Align_C***, and ***Align_D***).

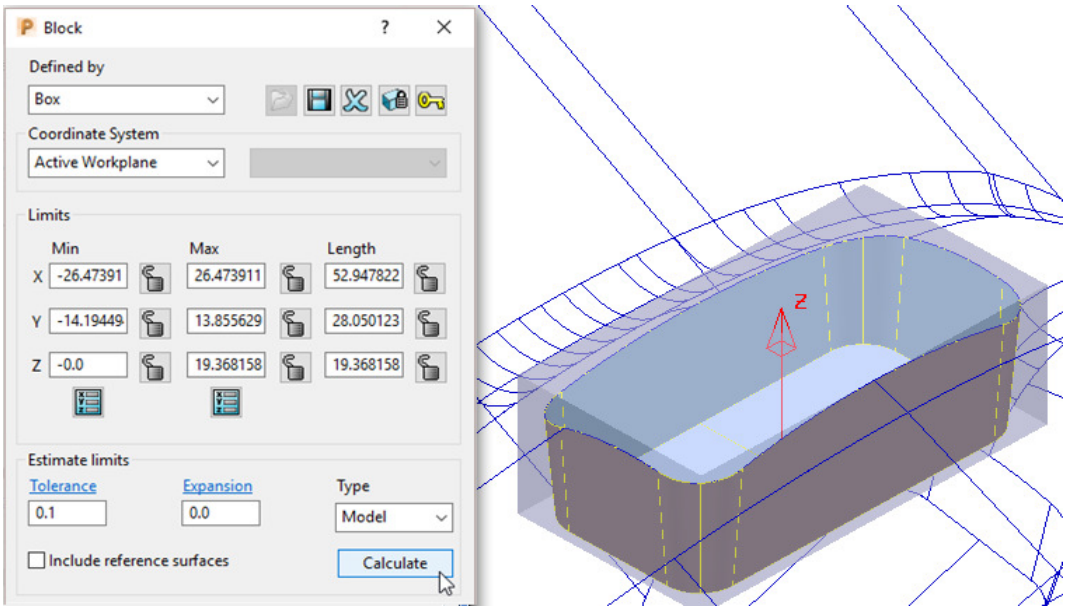
For each of the 3 Pockets a rectangular material **Block** will be created locally, relative to the aligned **3+2 workplane**. A **Model *Boundary*** will also be created around each pocket to provide even more control of the machining strategy limits.

18 Activate the **Workplane *Align_B***.

19 Select the 2 surfaces that define the pocket around **Workplane *Align_B***.

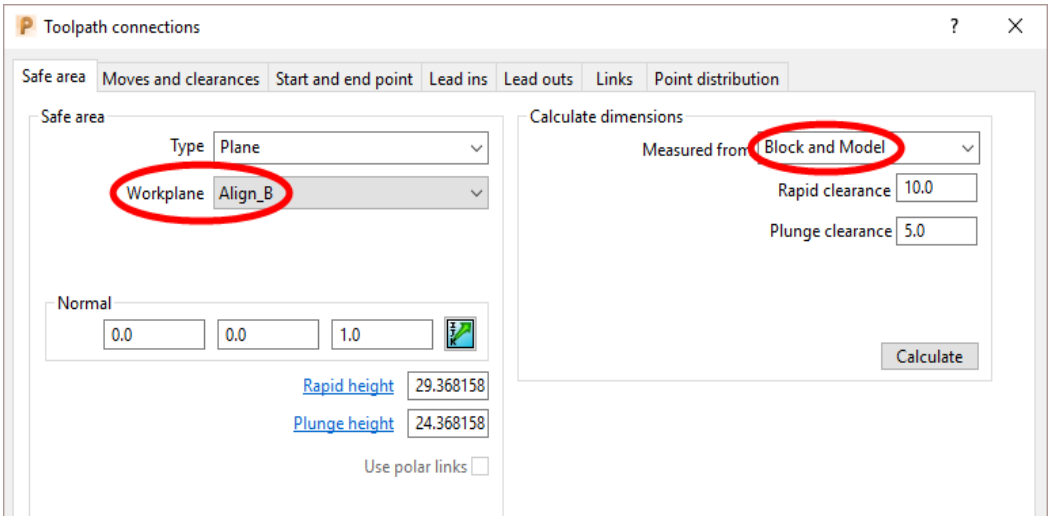
20 **Calculate** a **Block** using ***Defined by***, **Box** and ***Type***, **Model**.

Copyright © Autodesk



21 Ensure that the pocket surfaces are still selected so that the **Rapid move heights** calculation ignores the de-selected surfaces.

22 In **Rapid move heights** input the operational **Workplane** as ***Align_B*** and select **Calculate** to create values for **Safe Z** and **Start Z** aligned to the appropriate coordinate system.



*Input the **Workplane** (**Align_B**) for a **Rapid move height** plane that is parallel to the pocket base.*

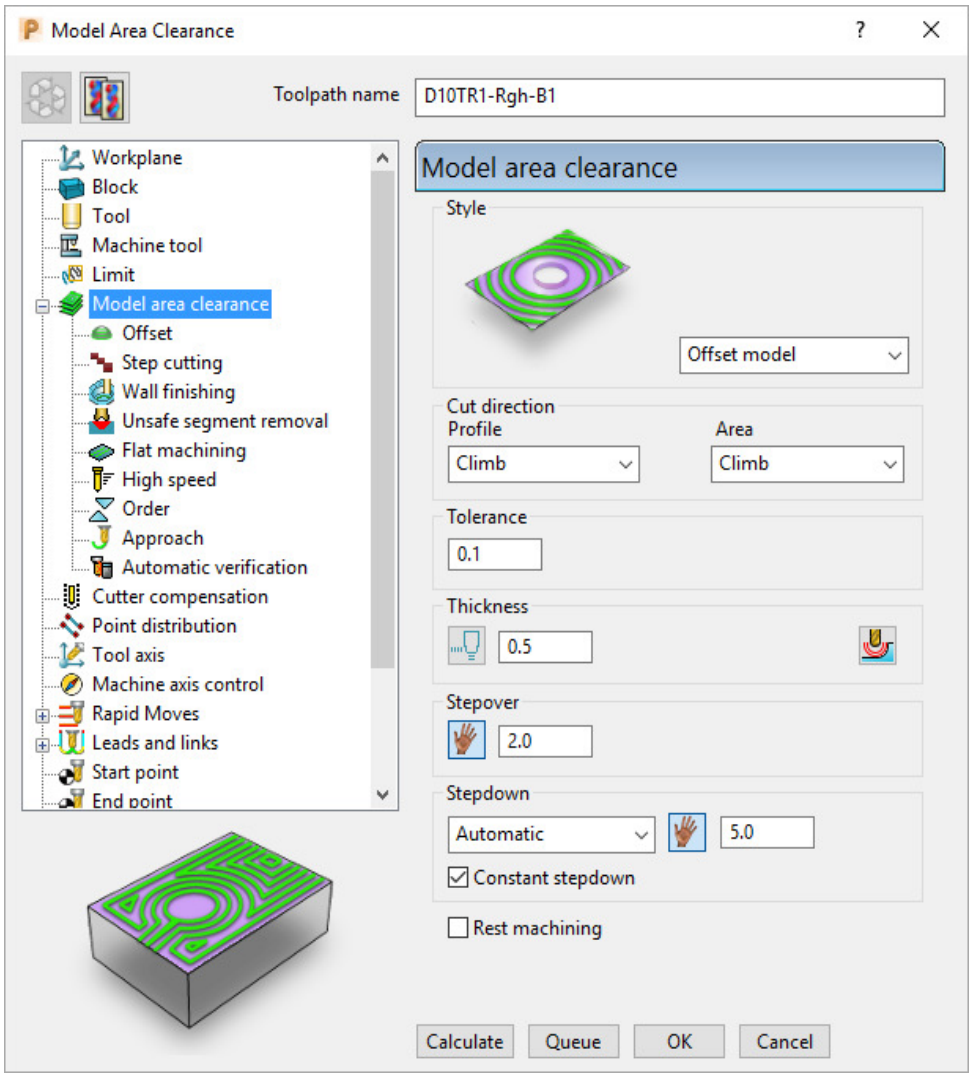
***Rapid move heights** are calculated from the maximum Z value of the **Block** and **Model** (or selected **Model** entities), whichever has the **maximum Z height**.*

23 Create a **User Defined** - **Model** - **Boundary** named **Pkt-B1** around the top edge of the pocket (surfaces still selected!) that surrounds the **Workplane** (**Align_B**).

24 **Activate** the existing **Tiprad** tool **D10TR1**.

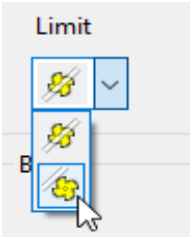
25 Set the **1st Choice**, **Lead In** to **Ramp** with **Ramp Options** as **Zig Angle 4**, **Follow Circle**, and **Diameter (TDU) 0.6**.

26 In the **Strategy Selector** dialog in **3D Area Clearance** select **Model Area Clearance** and fill in the options in the pages, exactly as shown below:



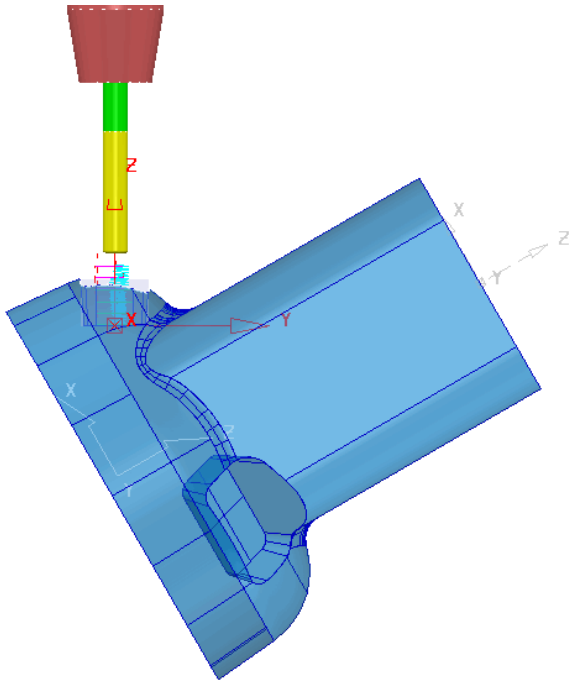
- **Toolpath name**: **D10TR1-Rgh-B1**
- **Style**: **Offset model**

27 In the **Limit** page input the **Boundary Pkt-B1** and set **Limit tool periphery to boundary**.



28 Select **Calculate** to create the toolpath and then **Cancel** the dialog.

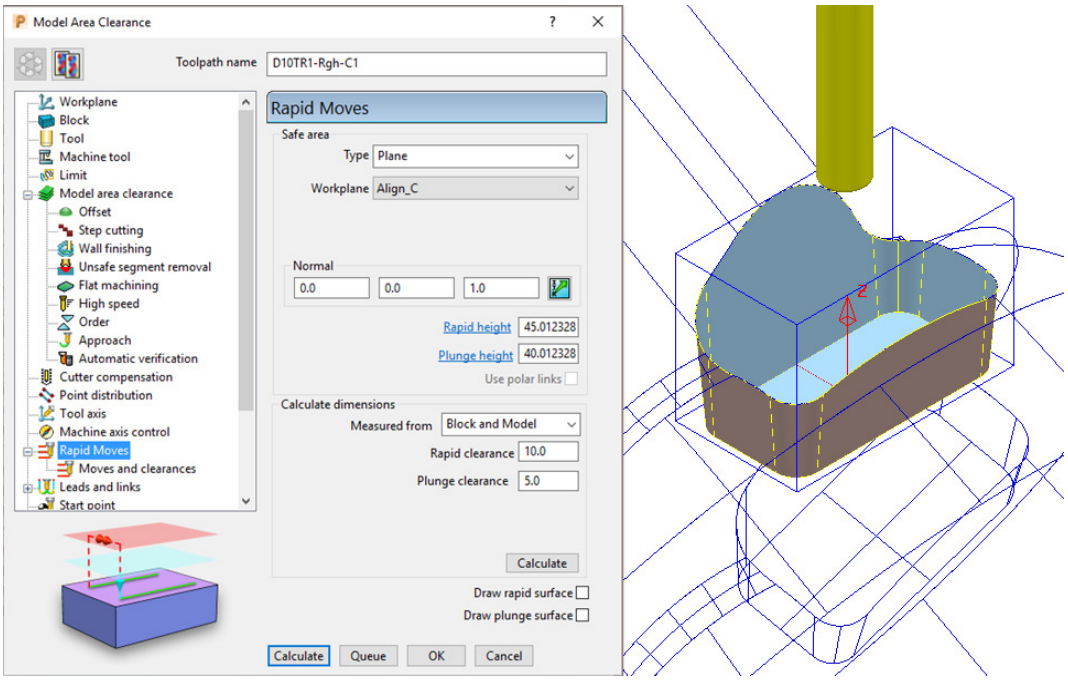
Copyright © Autodesk



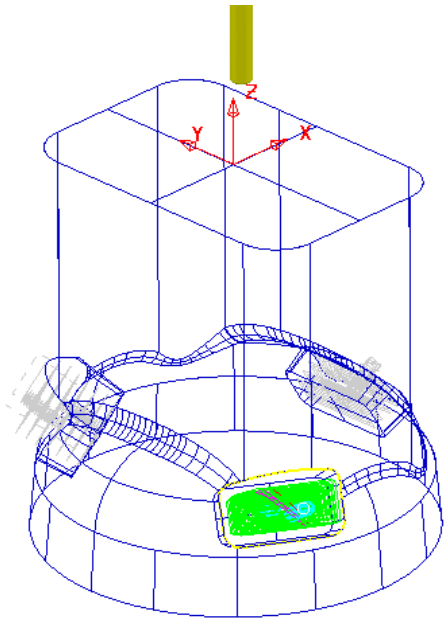
The toolpath plunge moves are aligned to the ***Active* workplane's**, Z Axis.

29 Use **Copy** to create cloned strategies for the other two recesses, adjusting the values and settings as required to create toolpaths, **D10t1-Rgh-C1** and **D10t1-Rgh-D1**.

a **Rapid Move Heights** must be defined separately for each toolpath with an appropriate **Workplane** selected in the dialog (Remember to check for the correct **Workplane** in the dialog).



30 Don't forget to create and **activate** the correct **User defined** - **Model** - **Boundaries** named **Pkt-C1** and **Pkt-D1**.

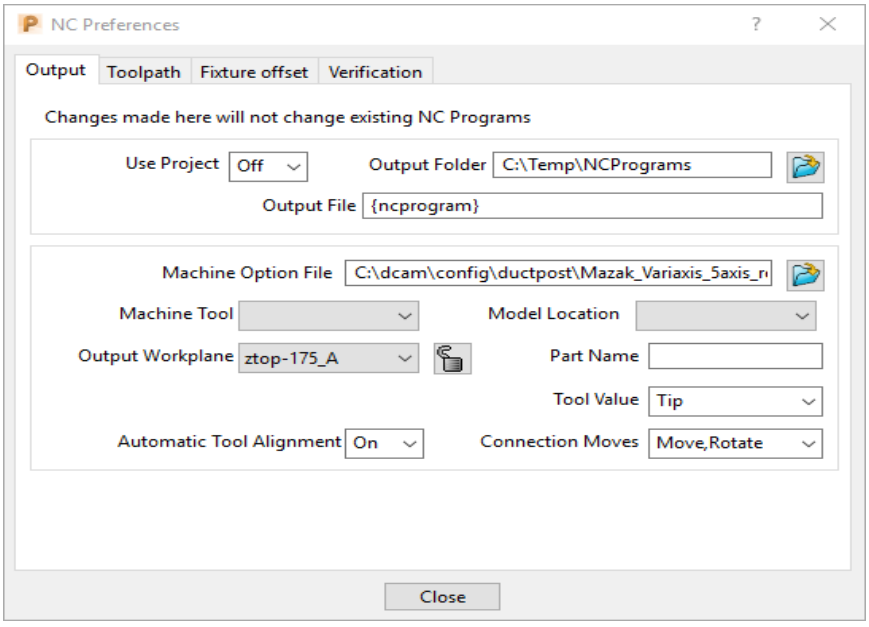


After the creation of **toolpaths** for **3 + 2 Axis** valid ncdata can only be output using a compatible, post-processor. For programs containing multi-alignment toolpaths the **NC Programs** output options create the ncdata from one datum (In this case the **Workplane** - **ztop175_A**). This option is selected in the **NC Preferences** or **NC Program Settings** dialog.

31 In the **PowerMill *explorer***, right-click on **NC Programs** and select **Preferences**.

32 From the **Machine Option File** select **Mazak_Variaxis_5axis_ref.pmoptz** as the controller.

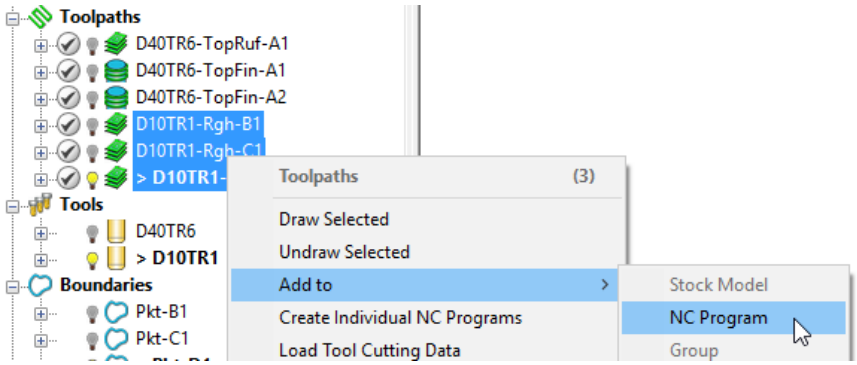
33 For the ***Output Workplane*** select **ztop-175_A**.

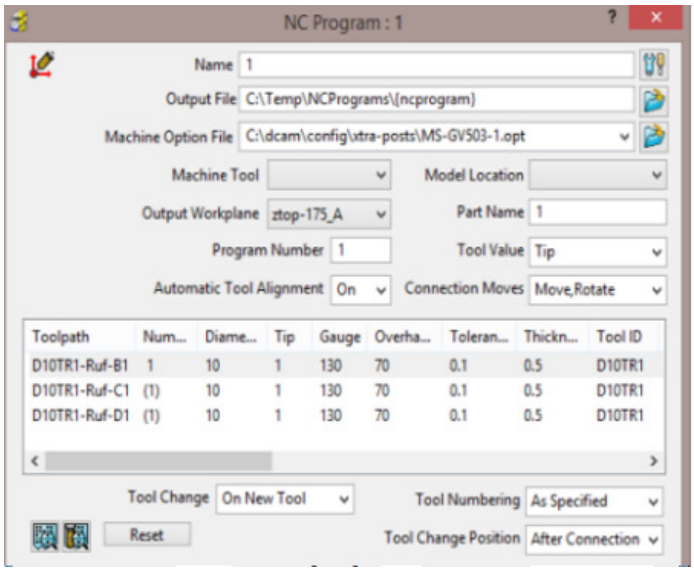


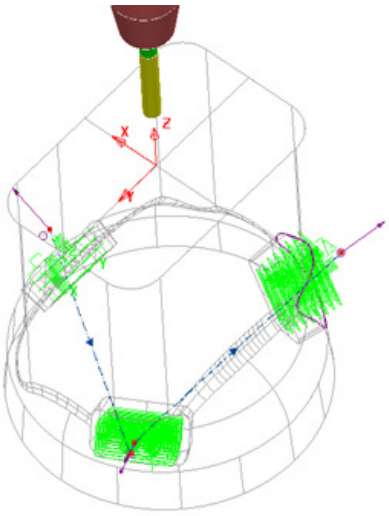
34 **Close** the dialog.

Copyright © Autodesk

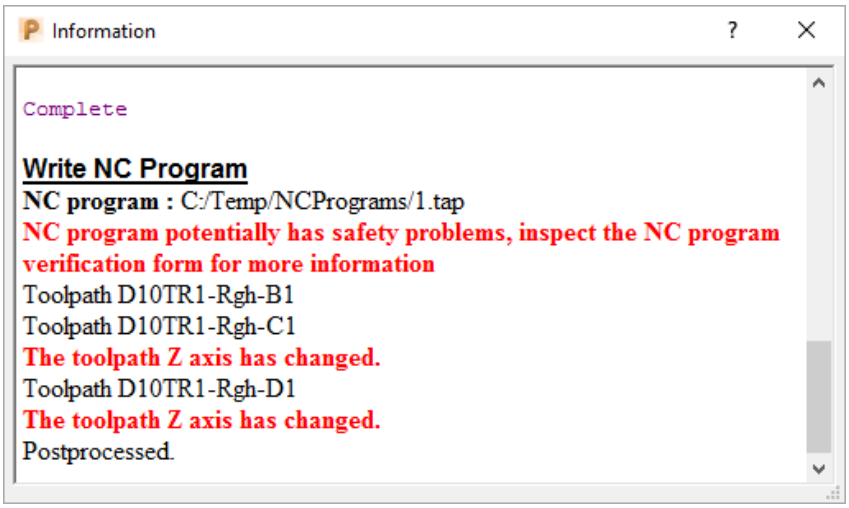
35 In the **PowerMill *explorer***, **NCPrograms**, select Create an **NC Program** and add the toolpaths, **D10TR1-Rgh-B1**, **D10TR1-Rgh-C1**, and **D10TR1-Rgh-D1**.





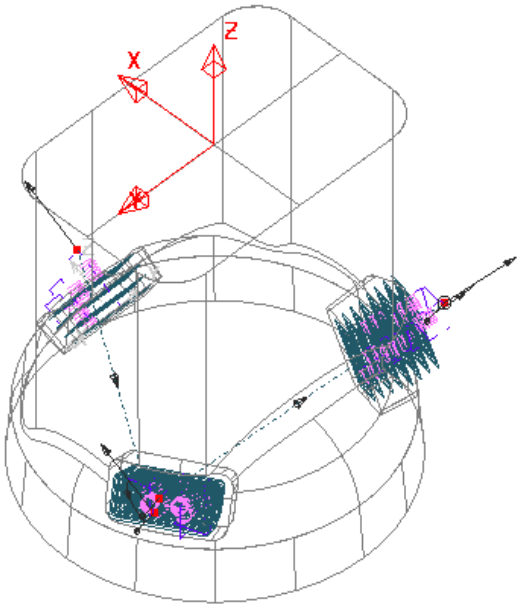


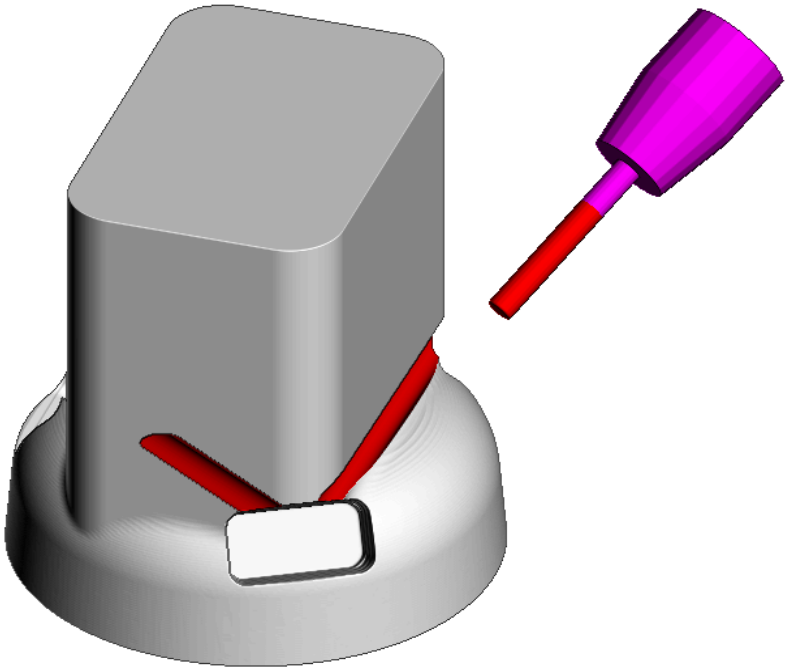
36 In the **NCProgram:1** dialog select **Write** to successfully create the ***Output File*** but note the red warning messages in the displayed **Information** dialog.



37 In the **PowerMill *explorer*** switch on the lightbulb adjacent to **NCProgram 1** to display it in the graphics area.

*The **link** moves between the three individual **3+2 toolpaths** are displayed in the **NC Program** and in this case are shown to be passing straight through the component!*





Automatic gouge protection only exists within a toolpath (Between the **Start and End Point**). After inserting the toolpaths into an **NC Program**, it is essential for the user to independently control the **Positional Moves** outside the toolpaths to eliminate any collisions with the component and fixtures.

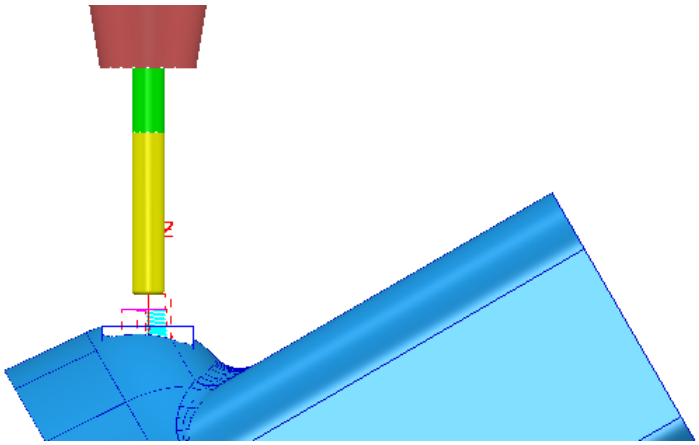
Copyright © Autodesk

Link moves controlled by Absolute Start and End points

This method of creating safe link moves between separate **3+2** toolpaths is the most universal and reliable method for all types of application and post-processor. The start and end point for each toolpath is defined with **absolute** coordinates positioned above the maximum Z height of the component.

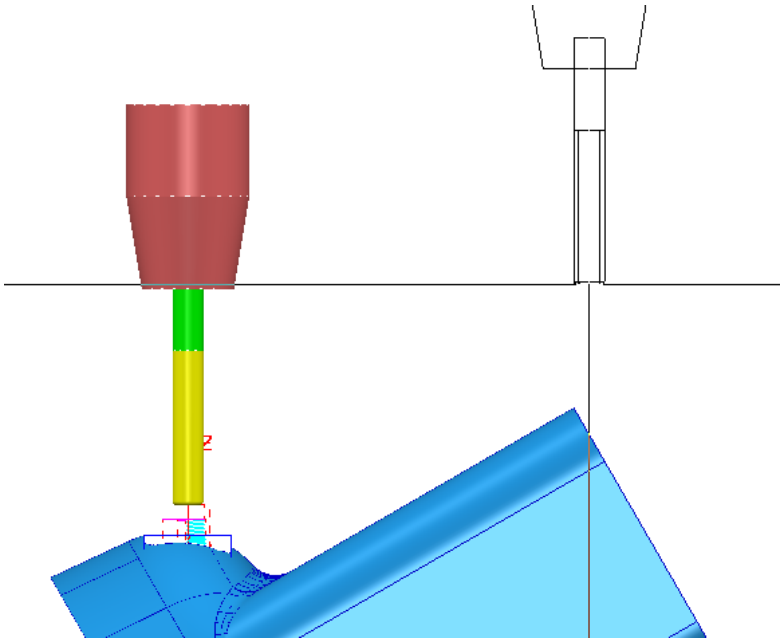
38 Activate the toolpath ***D10TR1-Rgh-B1***.

39 Select a **View from right (X)**.

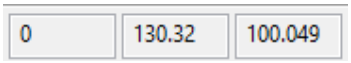


40 Switch on both **Draw - Cursor – Crosshair** (**Ctrl H**) and Tool (**Ctrl T**).

41 Move the cursor to a suitable ***Start/End*** **point** above the top of the model.



The **X**, **Y**, and **Z** coordinate values are displayed in the ***information*** **toolbar** located below the graphics area.

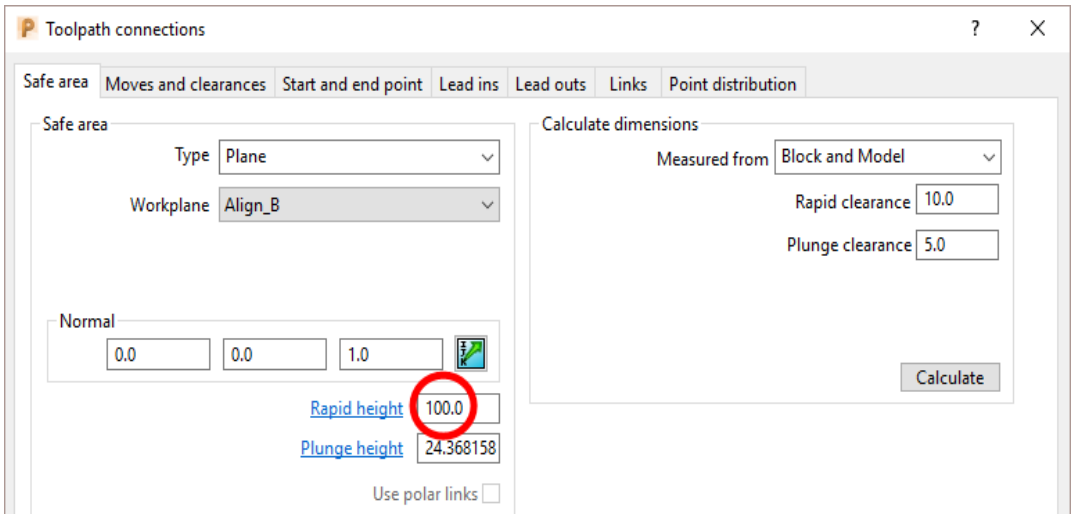


Copyright © Autodesk

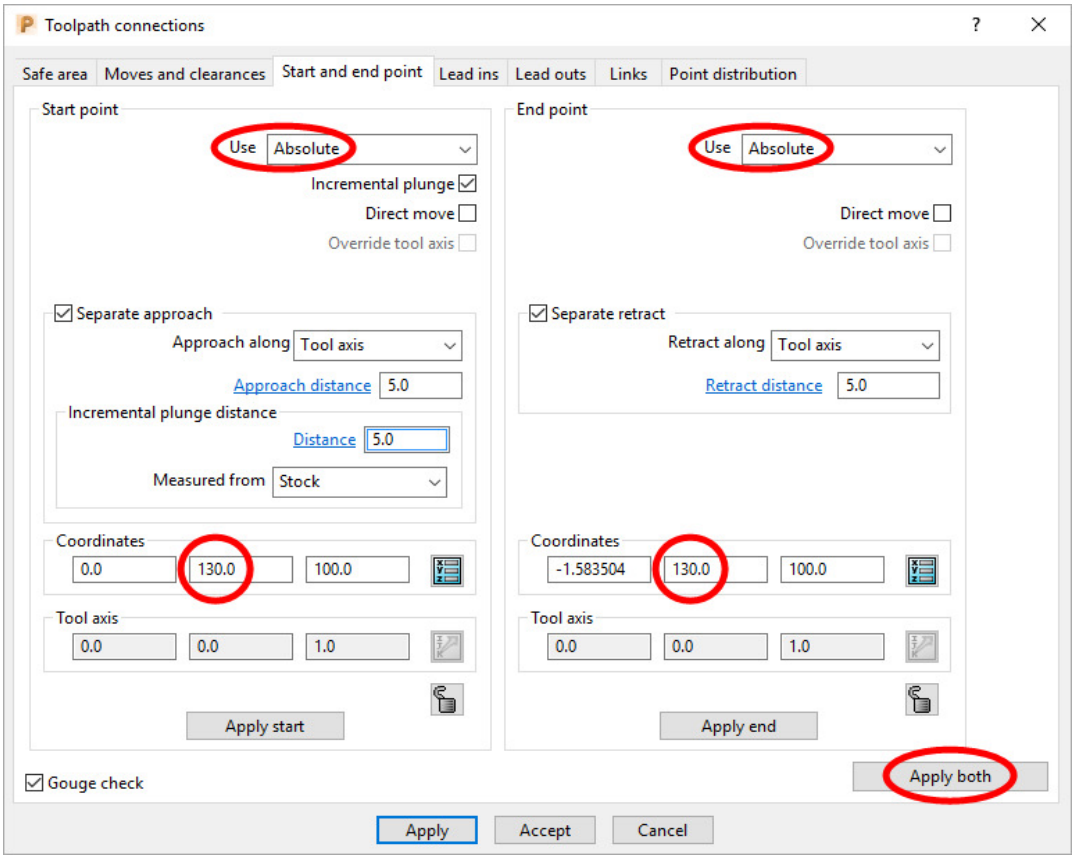
42 Make a note of the individual Y and Z values as integers eg. 130 and 100.

The **Rapid Move** - **Safe Z** value must now be matched to the **Z** value in **Start and End Point** dialog.

43 Open the **Rapid Move Heights** dialog, edit the **Safe Z** value to **100** before selecting **Apply** at the bottom of the dialog (to update the toolpath).



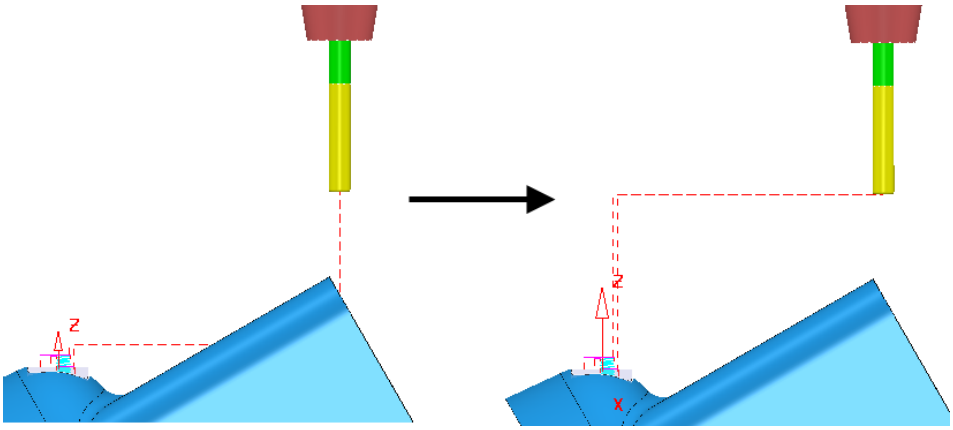
44 Open the Start and end Point dialog



45 For both the **Start point** and **End point** select **Use** - **Absolute** and input the new **Y** value (**130**).

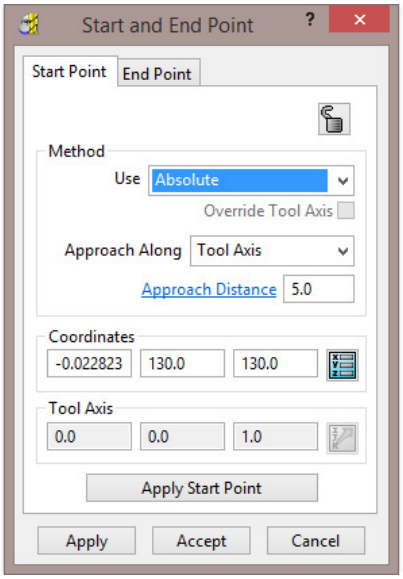
46 Select **Apply** both(to update the toolpath).

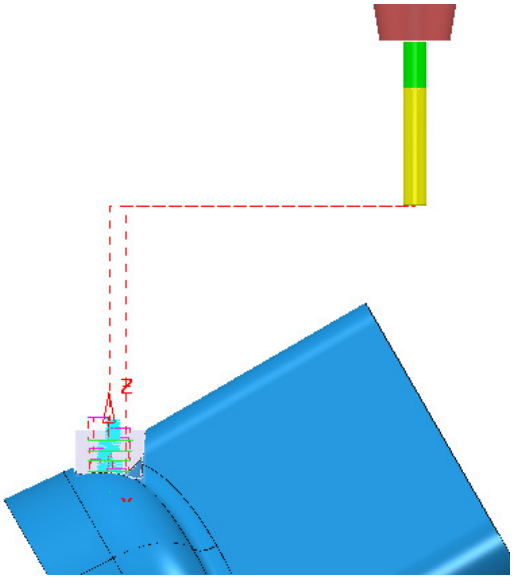
Copyright © Autodesk

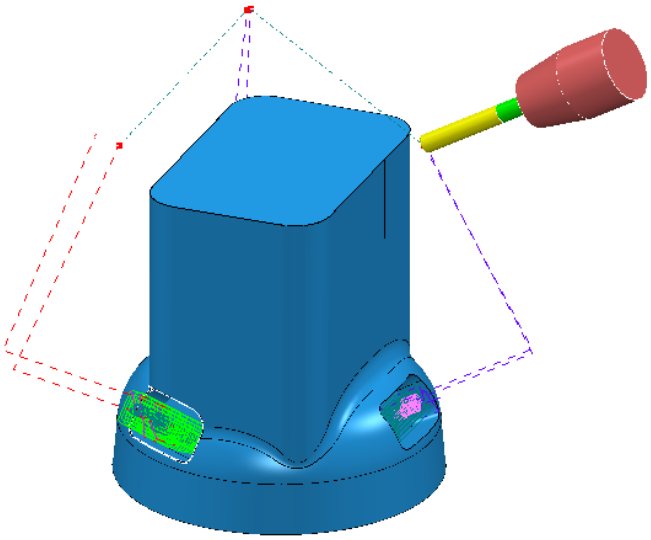


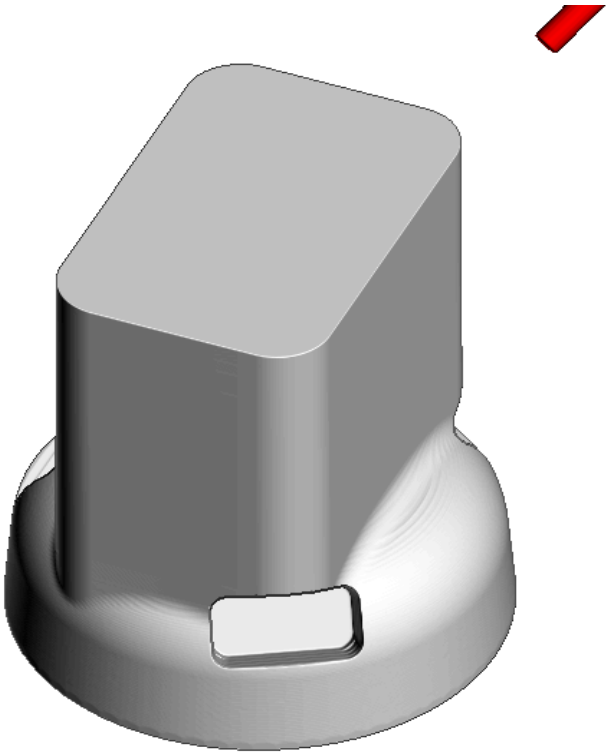
47 Repeat the above editing process on the other two **3+2** toolpaths **D10TR1-Rgh-C1** and **D10TR1-Rgh-D1** but this time use the values **130** and **120**.

*Note the new values (eg **130 120**) due to these pockets being in the corners of the main section with more of the main body to clear.*









48 Select **File** > **Save** to update the contents of the previously stored **Project**.

Copyright © Autodesk

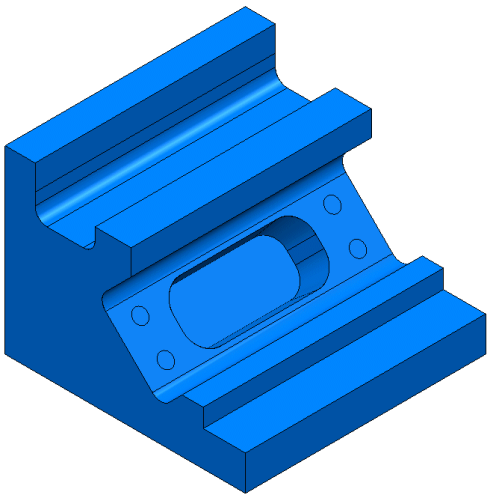
3+2 Axis – Stock Model Application

The **Stock Model** represents the un-machined material at any point in the machining process. An empty **Stock Model** is created, followed by applying the material **Block** and\or any number toolpaths to be considered in the process. The **Stock Model** is then updated by selecting **Calculate**, to display the current 'un-machined' material remaining.

1. Select **File** - **Delete All** and **Tools** - **Reset forms**.
2. **Import** the **Model**:

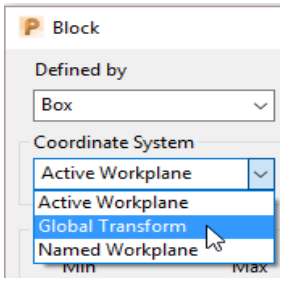
   ...\PowerMill_Data\five_axis\AnglePad\StockModelRest.dgk
3. **Save Project As**:

   ...\COURSEWORK\PowerMill_Projects\StockModel-3Plus2



The **model** includes undercut features which for a normal **3-Axis** application, would require the component to be machined in two separate set ups. However, by applying **3+2** with separate **workplanes** controlling the **tool alignments**, the whole project can be completed in one setup. During an initial **3-Axis** operation, the undercut pockets will be partially machined which provides an application for using **Stock Model** to enable the user to optimise the **3+2** machining.

4. **Calculate** a **Block** using **Defined by** – **Box**, **Type** - **Model**, and the **Coordinate System** set to **Global Transform**.



5. **Accept** the dialog.

By creating and locking the material Block to the Global Transform (world co-ordinate system), the orientation and position will remain unchanged if a Workplane is made Active.

6 Create a Dia 16 - tip radius 3 tool named **D16TR3**.

7 Create a Dia 12 - tip radius 1 tool named **D12TR1**.

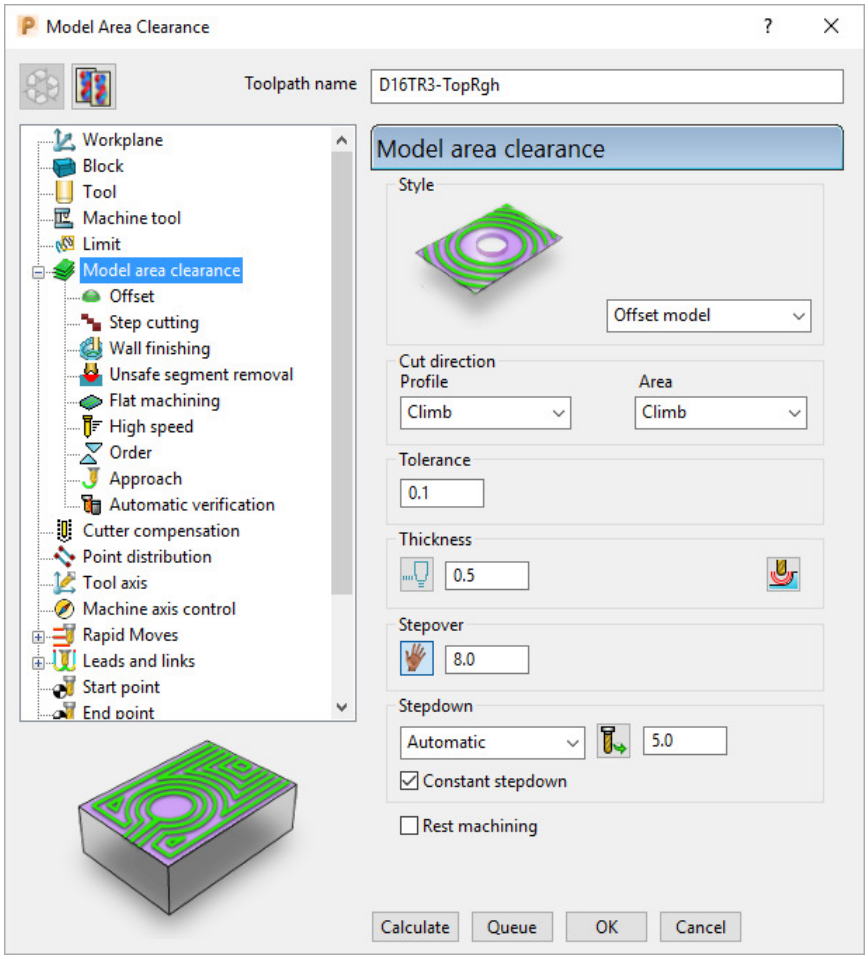
8 Activate the tool **D16TR3**.

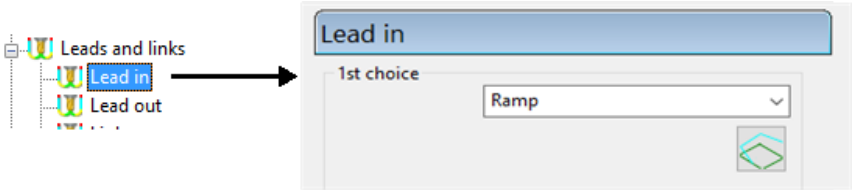
9 Activate **Workplane 1**.

10 In the **Rapid Move Heights** dialog select **Calculate**.

11 Use the default settings for both the **Start point** and **End point**.

12 Select the **Strategy Selector** and from **3D Area Clearance** select **Model Area Clearance** filling in the options in the pages, exactly as shown below:

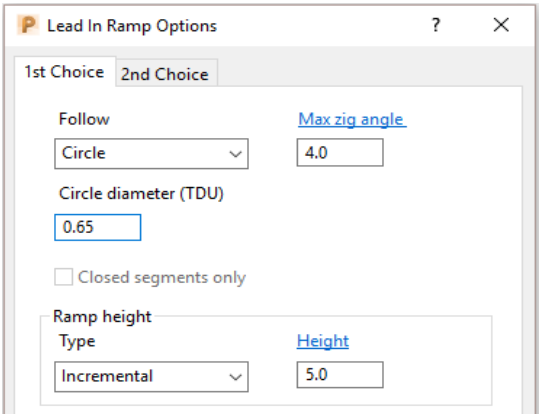




13 In the strategy ***explorer***, select the **Lead in** option and in the new page select ***1st choice*** **Ramp** followed by **Ramp options**.

Copyright © Autodesk

14 In the **Ramp Options** dialog, Input **Max zig angle 4** using a **Circle** with a diameter of **0.65** (**TDU**).



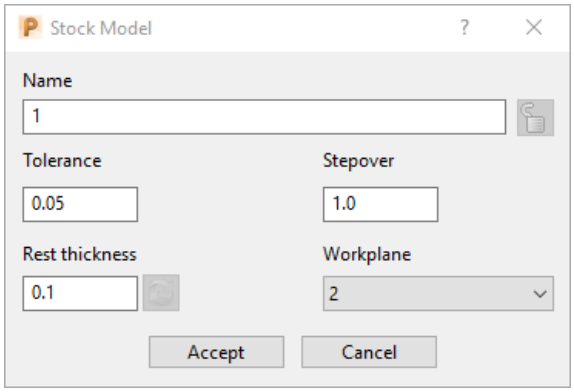
15 **Accept** both the **Ramp Options** and **Leads and Links** dialogs.

16 Back in the **main** strategy dialog, select **Calculate** to create the toolpath and then **Close**.

17 Select an **ISO 1** view.

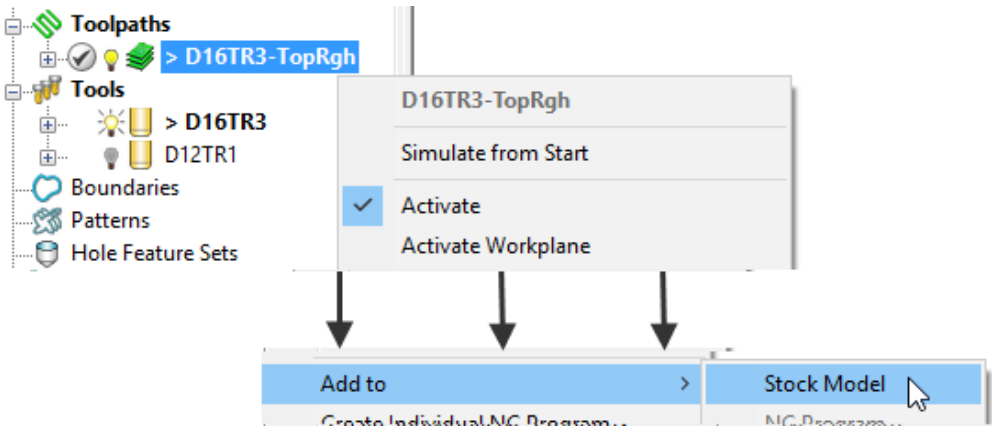
18 Activate **Workplane 2** (The **Block** remains in situ!).

19 In the **PowerMill *explorer***, right-click on **Stock Models** and from the local menu, select **Create Stock Model** to open the following dialog.



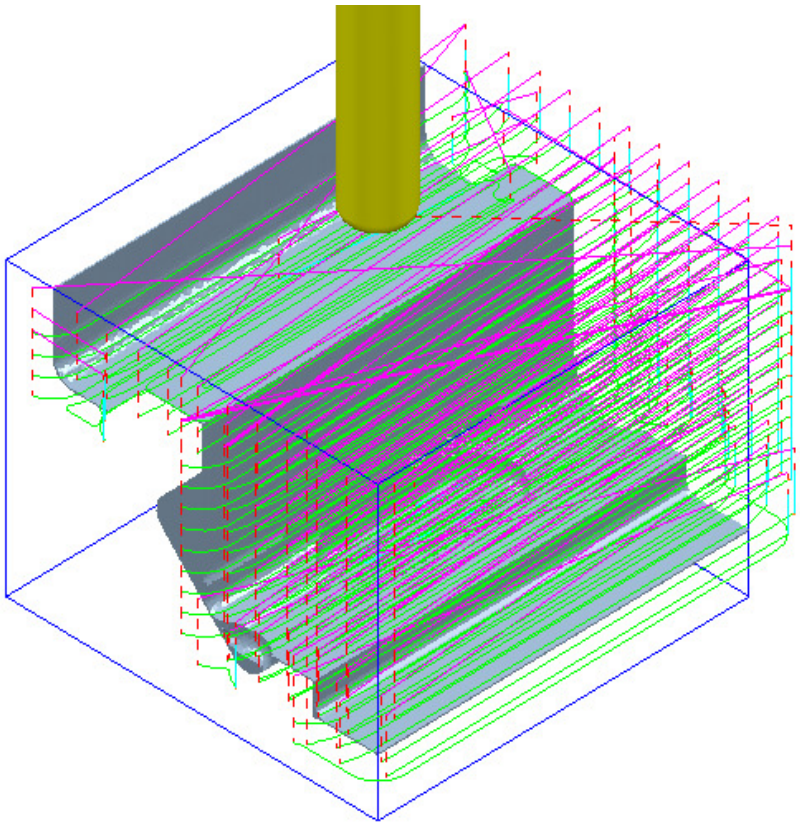
20 Input the above values and **Accept** the dialog.

21 In the **PowerMill *explorer***, right-click over the **toolpath *D16TR3-TopRgh*** and from the local menu select, **Add to** - **Stock Model**.



22 In the **PowerMill *explorer***, right-click over the new **Stock Model 1** and from the local menu, select **Calculate**.

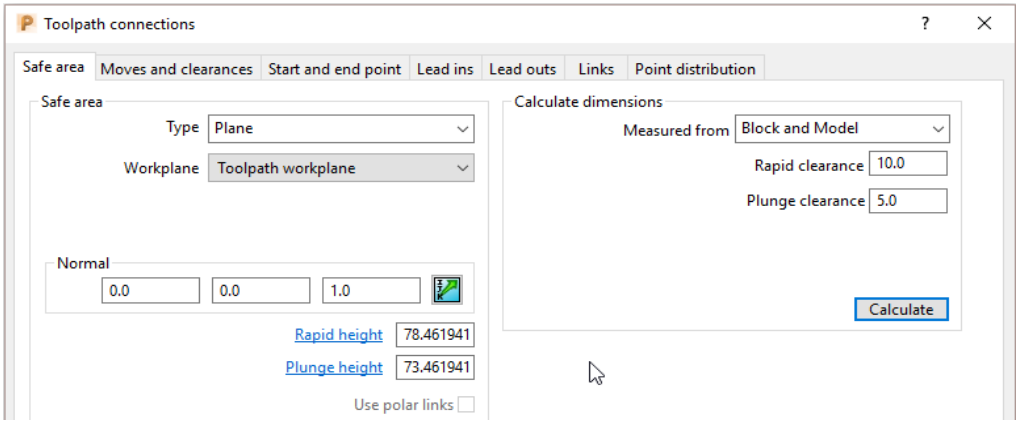
23 With the local **Stock Model** menu still open, select, **Drawing Options** > **Shaded** and **Show Rest Material**.



The **3-Axis Roughing** operation has removed all accessible material leaving a **0.5 thickness** on the component form. This is clearly visible on the displayed **Stock Model**.

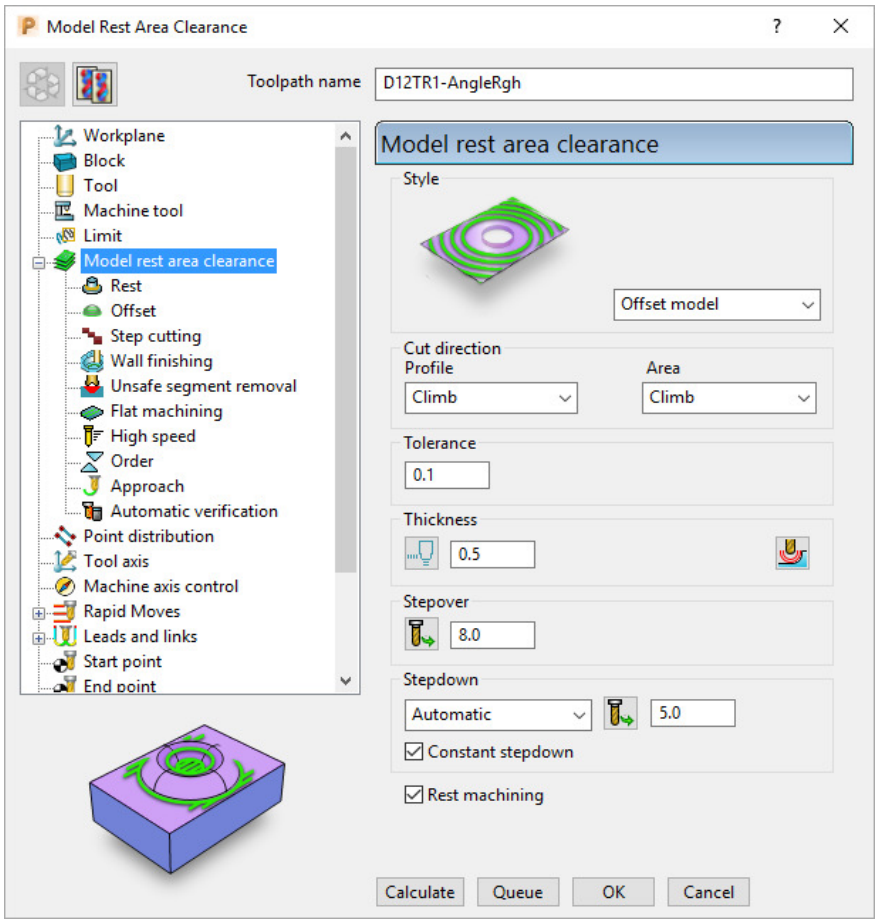
24 Activate the tool **D12TR1**.

25 From the **main** toolbar select **Rapid Move Heights** and input the correct **Workplane** definition **2** or **Toolpath Workplane** into the dialog before selecting **Calculate**, followed by **Accept**.



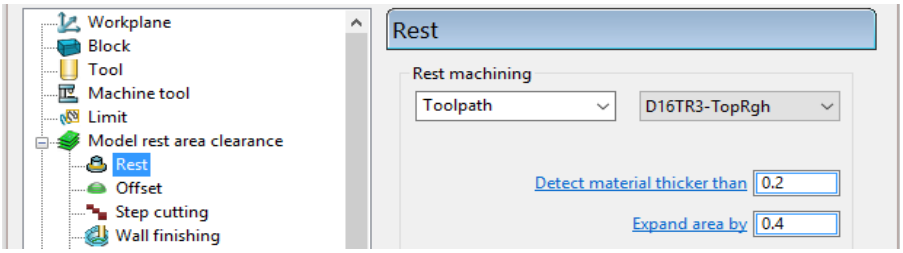
Copyright © Autodesk

26 Select the **Strategy Selector** and from **3D Area Clearance** select **Model Rest Area Clearance** and input settings exactly as shown in the following illustration:

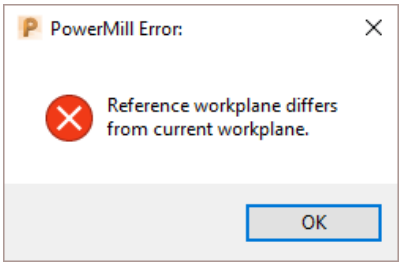


27 In the local explorer select the **Rest** page and input **Rest Machining** - **Toolpath** - **D16t3-TopRgh**.

28 Select **Calculate** to create the toolpath.



a *The strategy fails to calculate and the following **PowerMill Error** message box appears!*

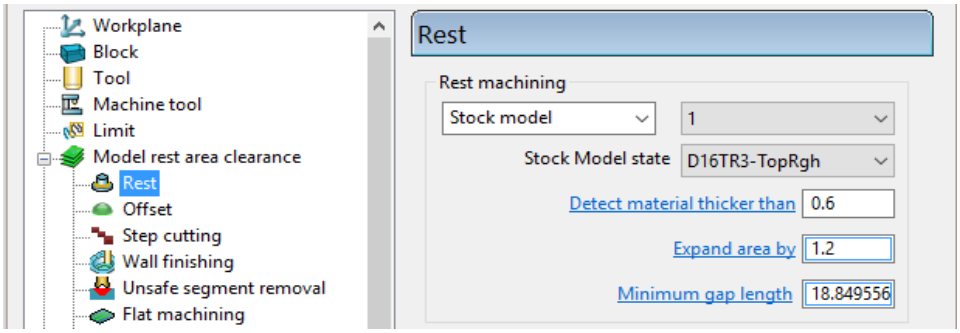


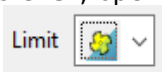
b *It is not possible to apply **Area Clearance** - **Rest machining** using a reference toolpath that has been generated relative to a different **Workplane** alignment.*

c *It is however possible to use a **Stock Model** as the reference item for **rest machining**.*

29 Select OK to close the ***PowerMill Error*** message and return to the strategy (that is still in Recycle mode).

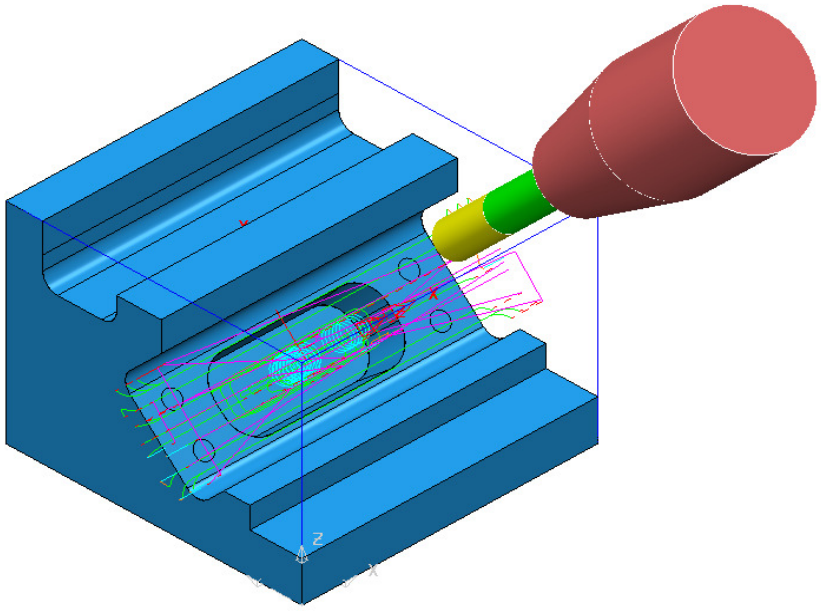
30 Set the option Rest machining as **Stock model** using **1**, and input ***Detect material thicker than*** **0.6** as shown in the following illustration.



31 In the local strategy ***explorer***, open the **Limit** page and select **Limit tool centre to block edge** 

.

32 Select **Calculate** to create the toolpath and then **Close** the dialog.



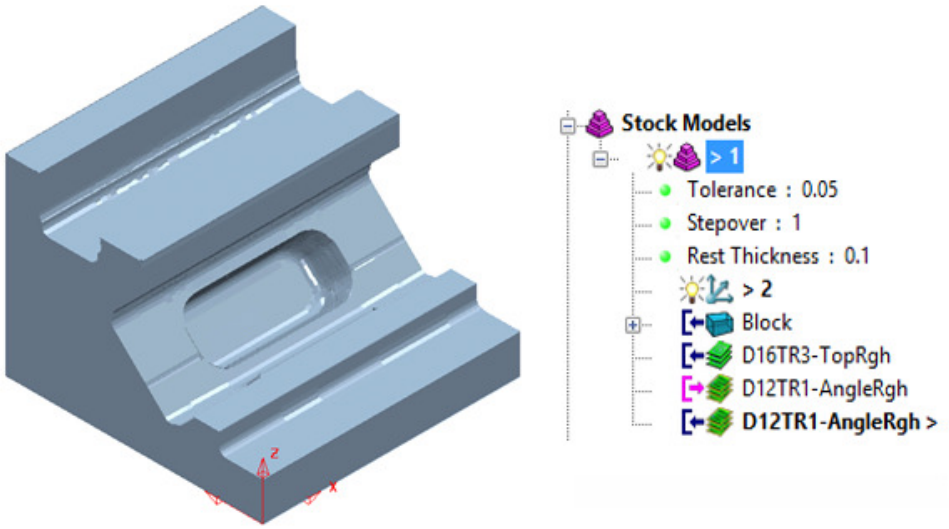
The modified **Rest Roughing** toolpath now successfully operates within the defined **Stock Model** volume (as shown above left).

The material removed by the toolpath, although referenced, is not physically included as part of the **Stock Model** at this stage.

33 In the **PowerMill explorer**, right-click on the **Toolpath *D12TR1-AngleRgh*** and from the local menu select, **Add to** - **Stock Model**.

Copyright © Autodesk

34 In the **PowerMill explorer**, right-click over the new **Stock Model 1** and from the local menu, select **Calculate**.



The **Stock Model** now displays the remaining material after both the **3-Axis Roughing** and **3+2 Roughing** operations.

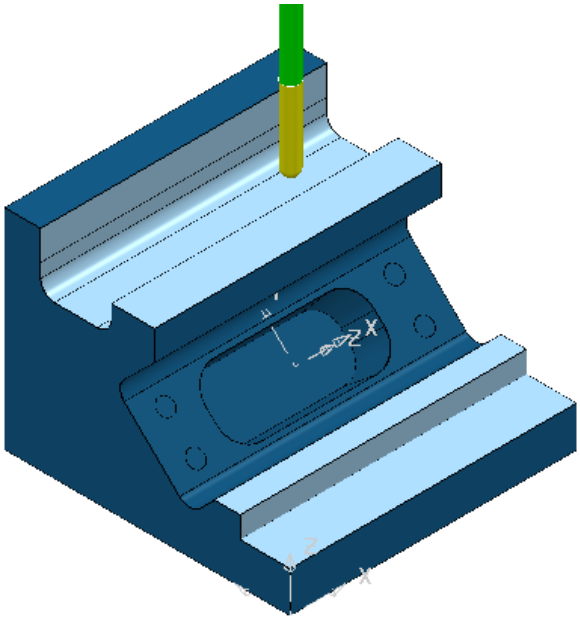
*Unlike **Area Clearance**, **Finishing** strategies cannot be directly **rest** machined to a **Stock Model**. However it is possible to create and apply **Stock Model Rest Boundaries** where required, providing suitable **rest** limits for subsequent finishing operations.*

35 Activate the toolpath **D16TR3-TopRgh** to reinstate the settings used.

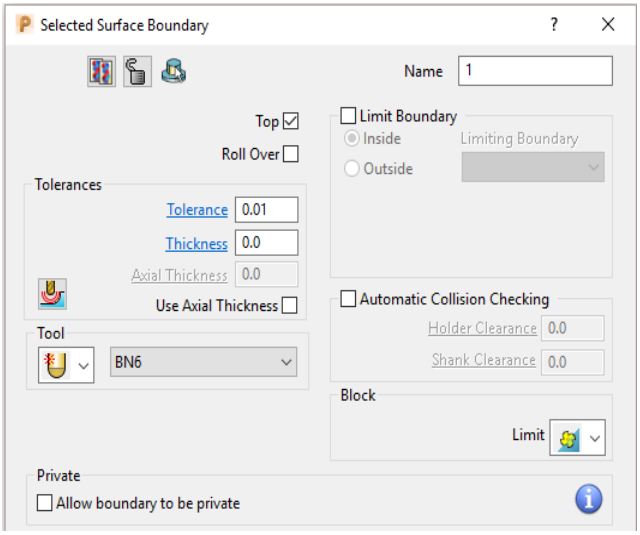
36 Create a Dia 6 Ball Nosed Tool named **BN6**.

37 Un-draw the **Stock Model** (click the light bulb off).

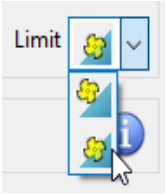
38 Select the Surfaces (shown shaded below) required for initial finish machining relative to **Workplane 1**.



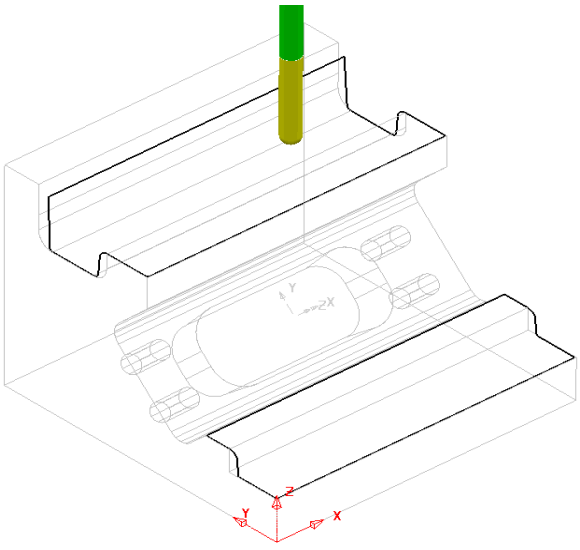
39 In the **PowerMill *explorer***, right-click over **Boundaries** and select **Create Boundary** followed by **Selected Surface** to open the following dialog.



40 Input data in the **Selected Surface Boundary** dialog exactly (as shown above) with the box named **Top** ticked and the **Limit** option set to **Limit tool centre to block edge**.



41 Select **Apply** and when processed, **Cancel** the dialog.

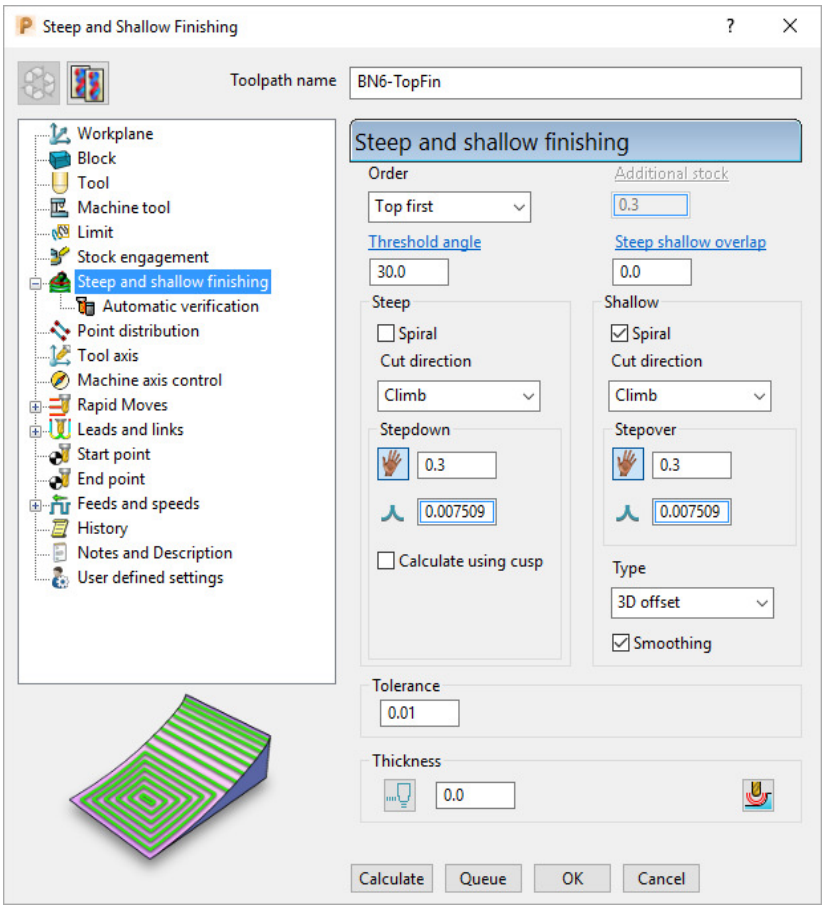


When created, the new **Boundary** will be **Active** and by default included as the **limits** (keep inside) in the following finishing strategy.

42 Select the **Strategy Selector** and from the **Finishing** dialog select the **Steep and Shallow** finishing option.

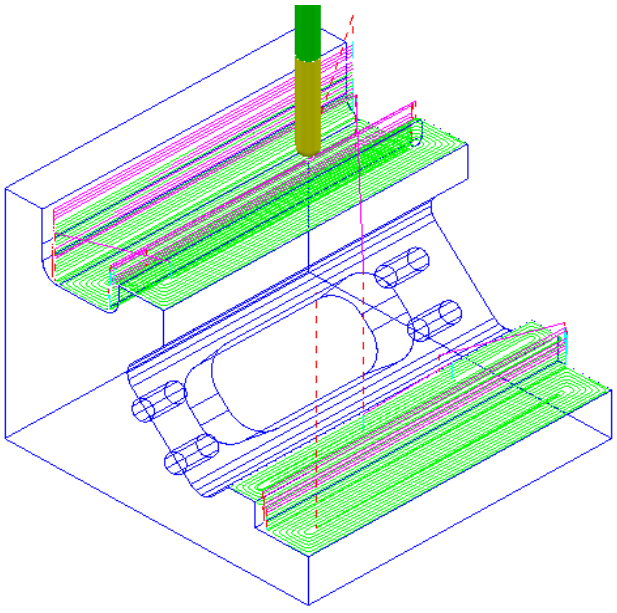
43 Enter the name **BN6-TopFin** along with the remaining values and settings exactly as shown below before selecting **Apply**.

Copyright © Autodesk



44 Set both **Lead In** and **Lead Out** to **Surface normal arc** (**Angle 90**, **Radius 3**).

45 Select **Calculate** to create the toolpath and then **Close** the dialog.



All of the features accessible from the top have now been finish machined. This finishing strategy will be added to the **Stock Model** ready for a **Stock Model Rest Boundary** to be created and applied to a **3+2** finishing strategy along **Workplane 2**.

46 In the **PowerMill *explorer***, right-click on the **Toolpath BN6-TopFin** and from the local menu select, **Add to** - **Stock Model**.

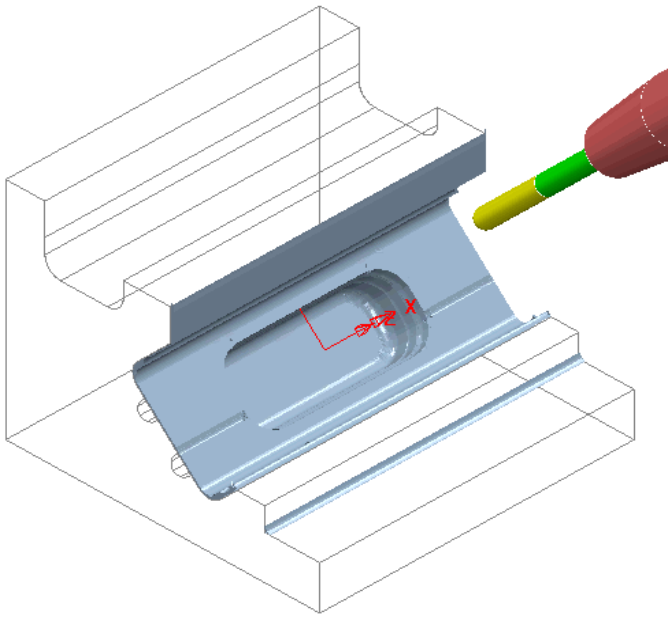
47 In the **PowerMill *explorer***, right-click over the new **Stock Model 1** and from the local menu, select **Calculate**.

48 With the local **Stock Model** menu still open, select **Show Rest Material**, followed by **Drawing Options** - **Shaded**, and finally **Calculate**.

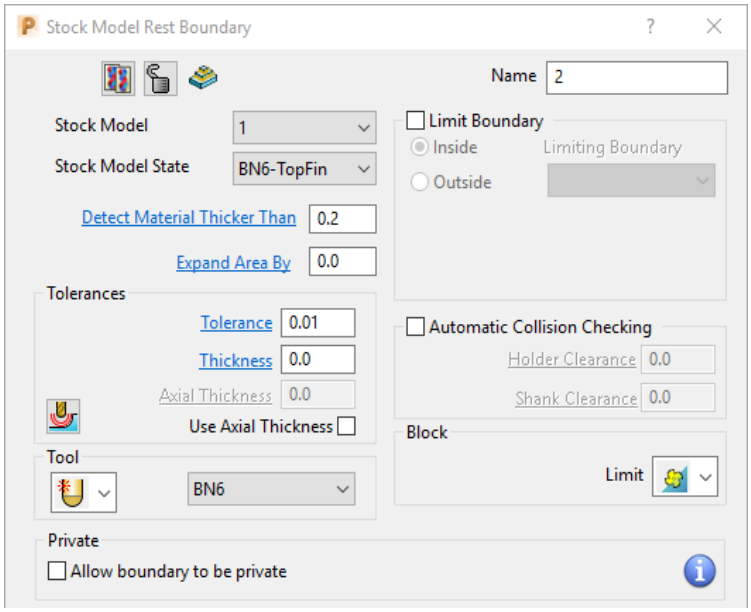
49 Select an **ISO 1**.

50 **Activate Workplane 2**.

51 From the **main** toolbar select **Rapid Move Heights** and input the correct **workplane** definition **2** or **Toolpath Workplane** into the dialog before selecting **Calculate**.

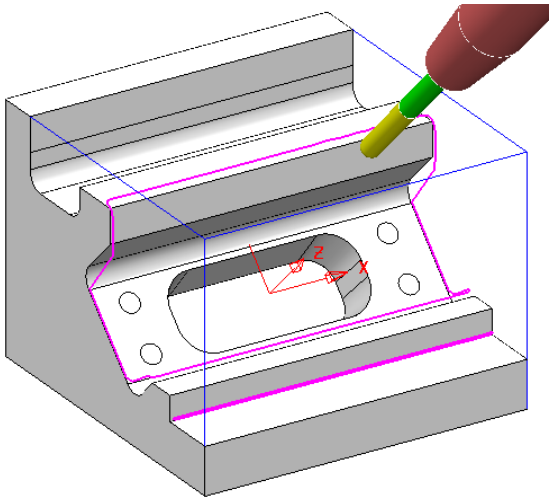


52 In the **PowerMill *explorer***, right-click **Boundaries** and select **Create Boundary** followed by **Stock Model Rest** to open the following dialog.



Copyright © Autodesk

53 Input data in the **Stock Model Rest Boundary** dialog exactly as shown above.



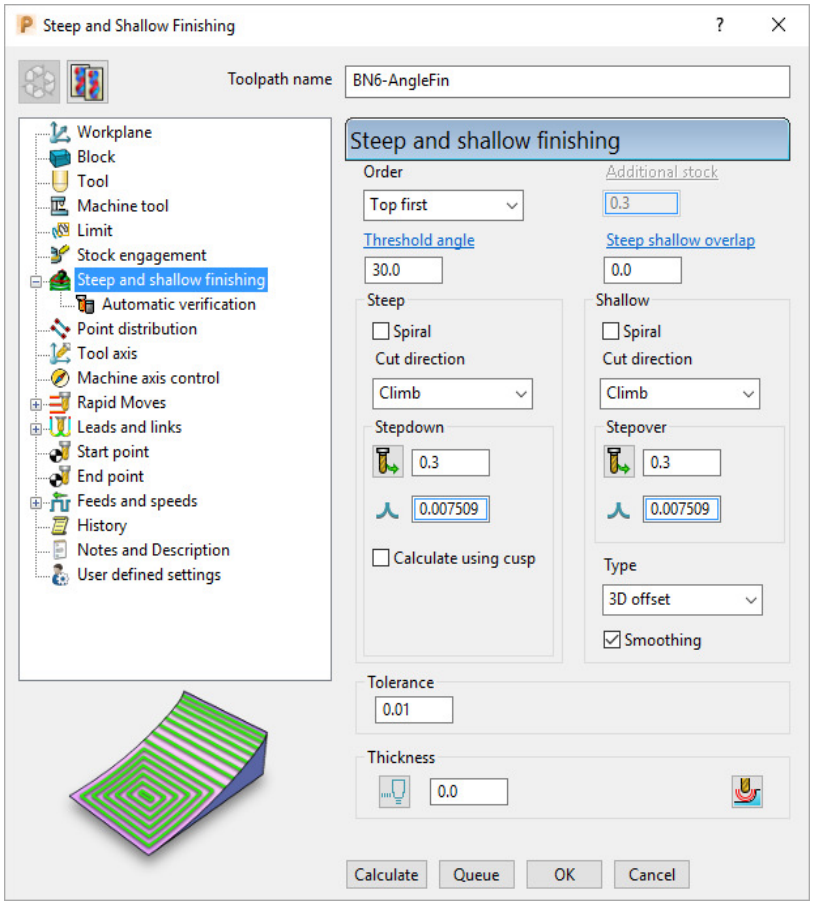
54 **Apply** and when processed, **Cancel**.

55 Select and delete the narrow segment at the base of the lower vertical wall.

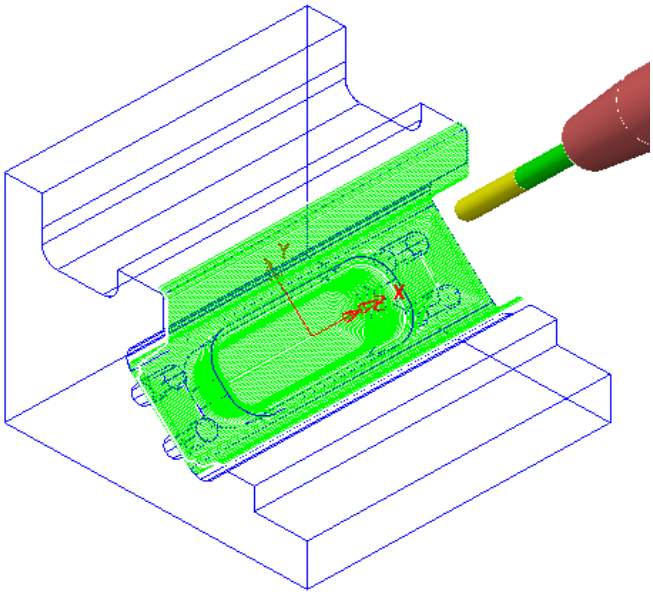
56 Ensure that new **Boundary 2** is **Active**.

57 Select the **Strategy Selector** and from the **Finishing** dialog, select the **Steep and Shallow** finishing option.

58 Enter the name **BN6-AngFin** along with the remaining values and settings exactly as shown below.



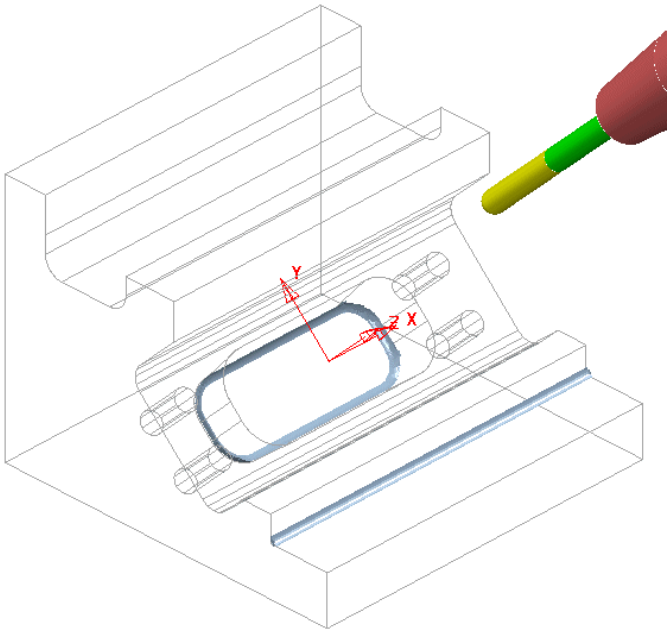
59 Select **Calculate** to create the toolpath and then **Close** the dialog.



The resultant **toolpath** will be added to the **Stock Model** mainly to confirm whether machining is now complete.

60 In the **PowerMill *explorer***, right-click on the **Toolpath *BN6-AngleFin*** and from the local menu select, **Add to** - **Stock Model**.

61 In the **PowerMill *explorer***, right-click over the new **Stock Model 1** and from the local menu, select **Calculate**.

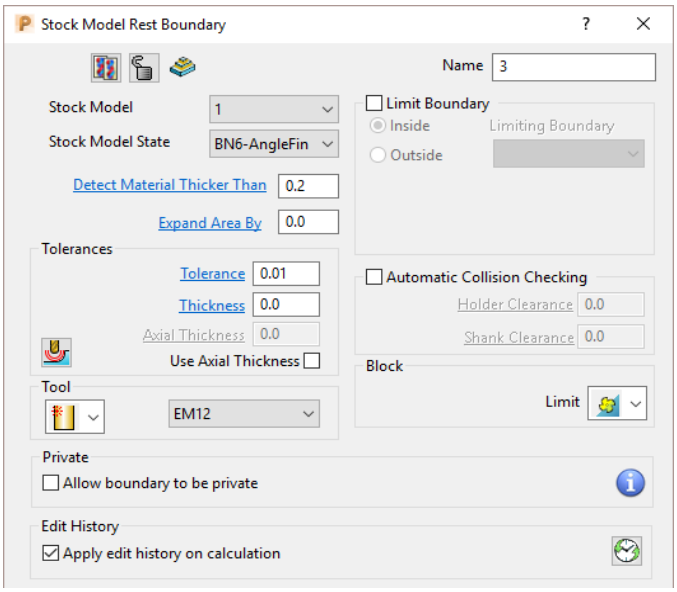


There are now 2 areas remaining as part of the **Stock Model**. As both include sharp internal corners they will need to be finished off using an End Mill.

62 Create a Dia 12 End Mill named **EM12**.

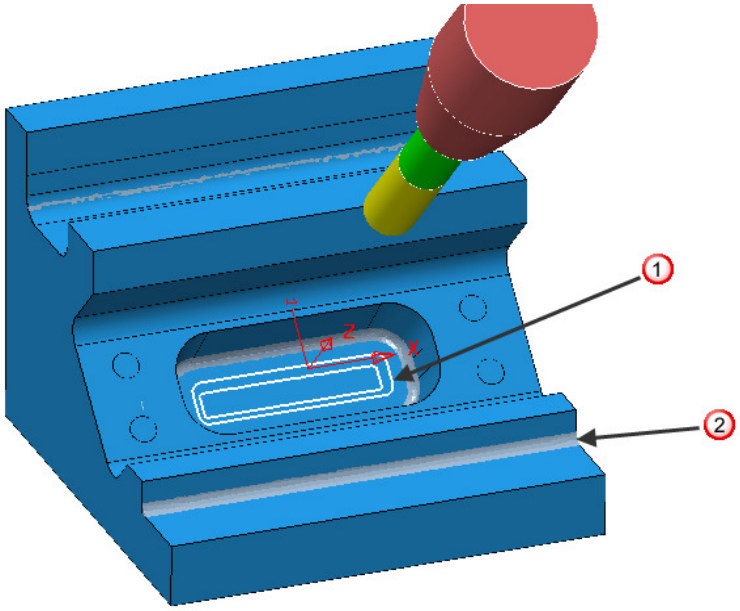
63 In the **PowerMill *explorer***, right-click over **Boundaries** and select **Create Boundary** followed by **Stock Model Rest** to open the following dialog.

Copyright © Autodesk



64 Input data in the **Stock Model Rest Boundary** dialog exactly as shown above.

65 **Apply** and when processed, **Cancel** the dialog.



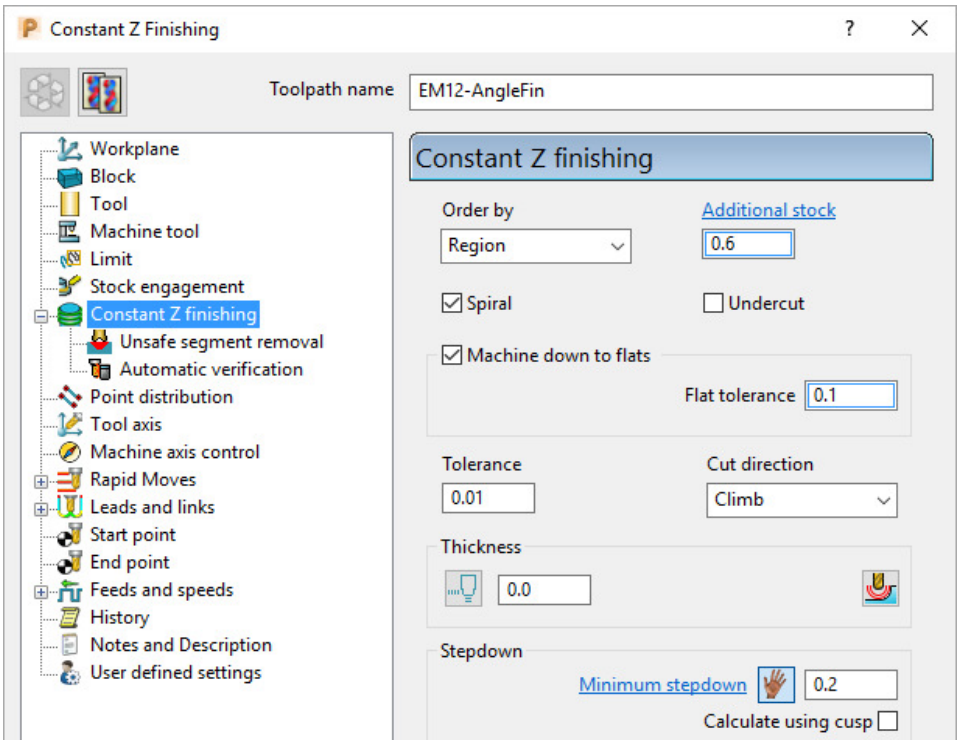
1 A new **Stock Model Rest *Boundary*** has appeared where the remaining material is accessible to the **EM12** tool.

2 This area is not accessible to the **EM12** tool, and as a result **Boundary** segments will not be created.

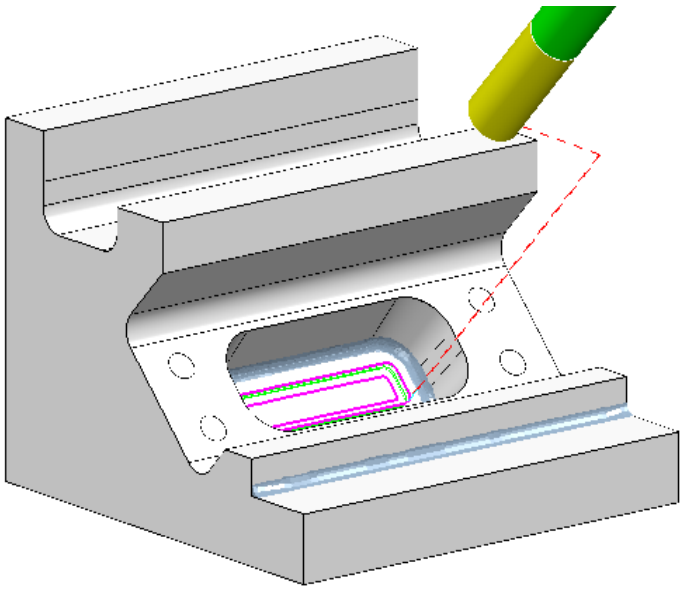
66 Ensure that the new **Boundary** (**3**) is Active.

67 Select **Toolpath Strategies** and from the **Finishing** dialog, select the **Constant Z** option.

68 Enter the name **EM12-AngleFin** along with the remaining values and settings exactly as shown on the following page.



69 Select **Calculate** and then **Close** the dialog.



The angled pocket is now fully machined and the latest toolpath will now be added to the **Stock Model**.

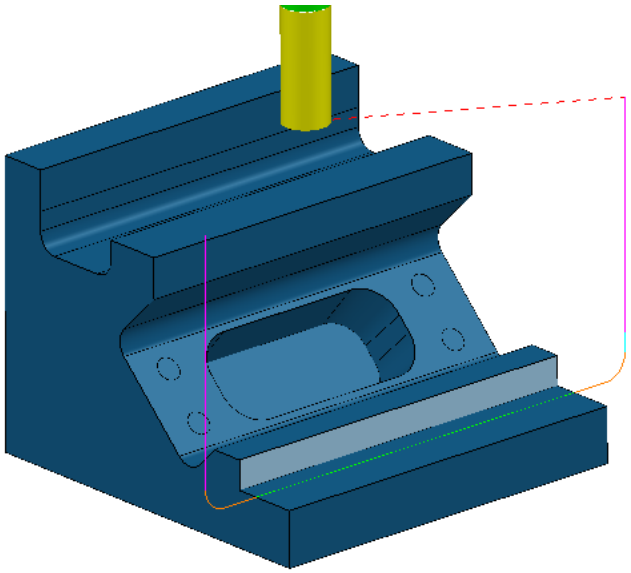
70 In the **PowerMill *explorer***, right-click on the toolpath **EM12-AngleFin** and from the local menu select, **Add to** - **Stock Model**.

71 In the **PowerMill *explorer***, right-click over the new **Stock Model 1** and from the local menu, select **Calculate**.

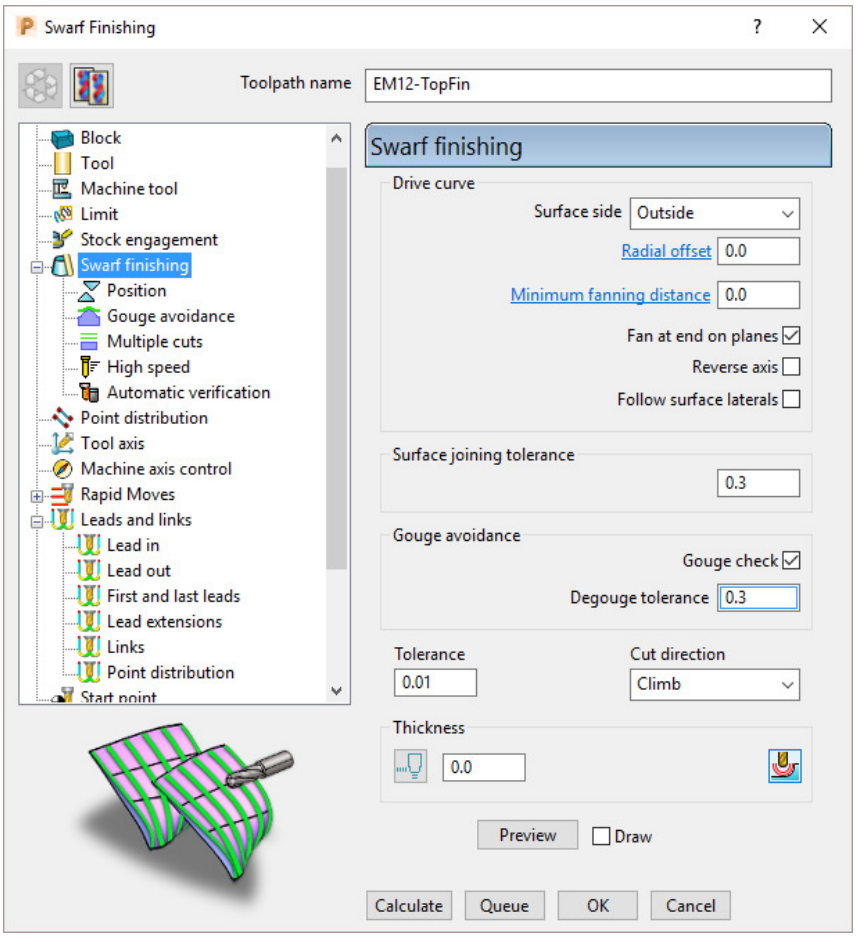
72 Activate the toolpath **D16TR3-TopRgh** to reinstate the settings used.

73 De-activate the toolpath **D16TR3-TopRgh** and activate the **EM12** tool.

Copyright © Autodesk

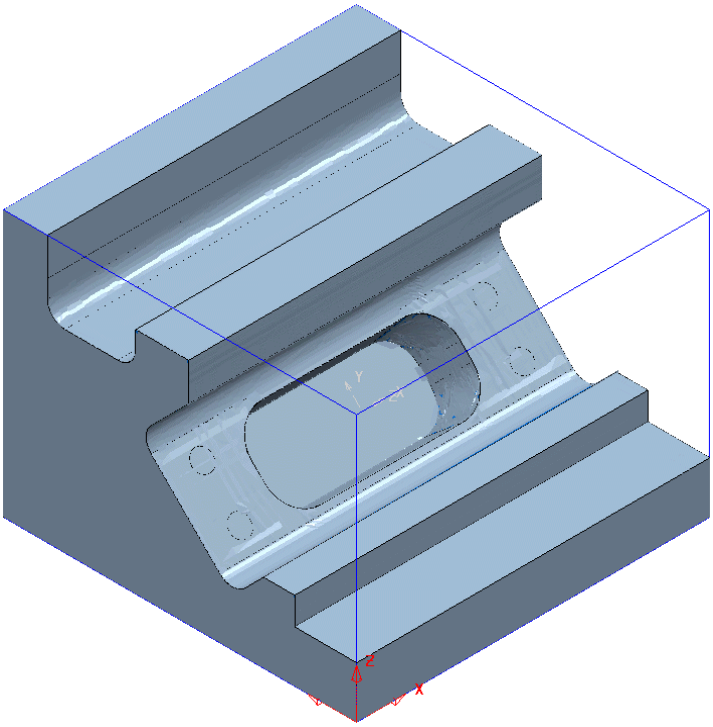


74 Create a **Swarf Finishing** strategy named **EM12-TopFin** on the selected, vertical surface (shown shaded above).



75 Set both **Lead In** and **Lead Out** to **Vertical arc** with **Distance 6**, **Angle 90**, and **Radius 4**.

76 **Add** the new toolpath **EM12-TopFin** to the **Stock Model** and **Calculate** to visually confirm that all excess material has now been removed.



*The **Stock Model** is only visible when the **Show Rest Material** option is unticked.*

77 Select **File** > **Save** to update the contents of the previously stored **Project**.

Copyright © Autodesk

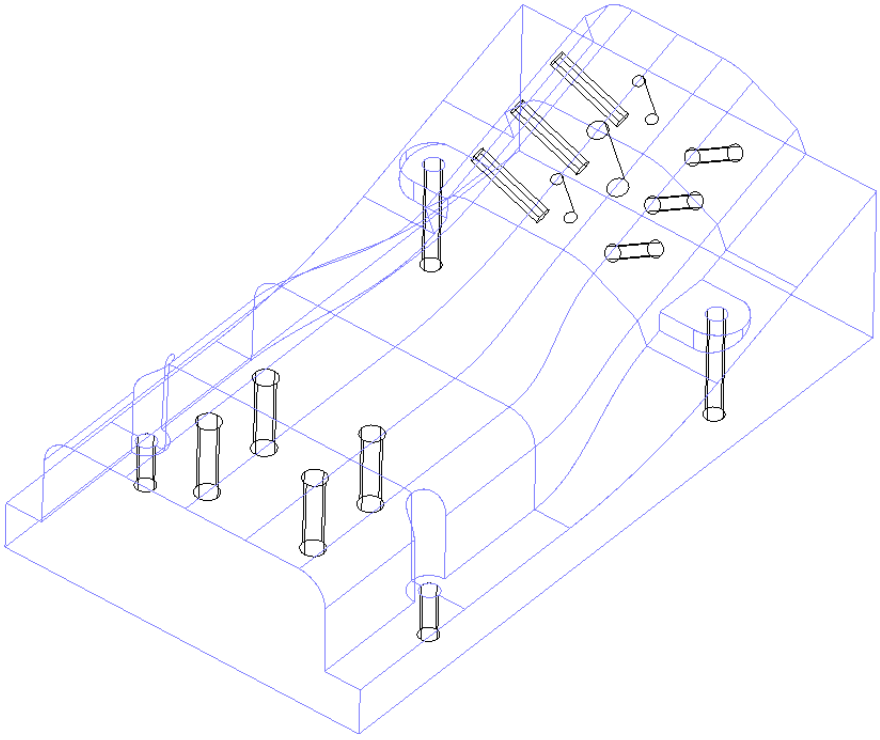
3+2 Axis - Drilling Example

The **PowerMill** - **Drilling** options operate on **Hole Features** and not directly on the **Model**. This enables drilling to take place without the need to modify or trim back the existing surface data. When the **Multi-Axis** option is ticked the **Hole Features** at different alignments will be created to a common workplane.

1 Select **File** - **Delete All** and **Tools** - **Reset Forms**.

2 **Import** the **Model**:

...\PowerMill_Data\five_axis\drill_5axis\drill5ax_ex1.dgk



3 Select **File** - **Save Project As**:

...\COURSEWORK\PowerMill-Projects\MultiAxisDrill-ex1

The first stage is to create a **Hole Feature Set** into which all cylindrical surfaces in the selected model are inserted as **Hole** features.

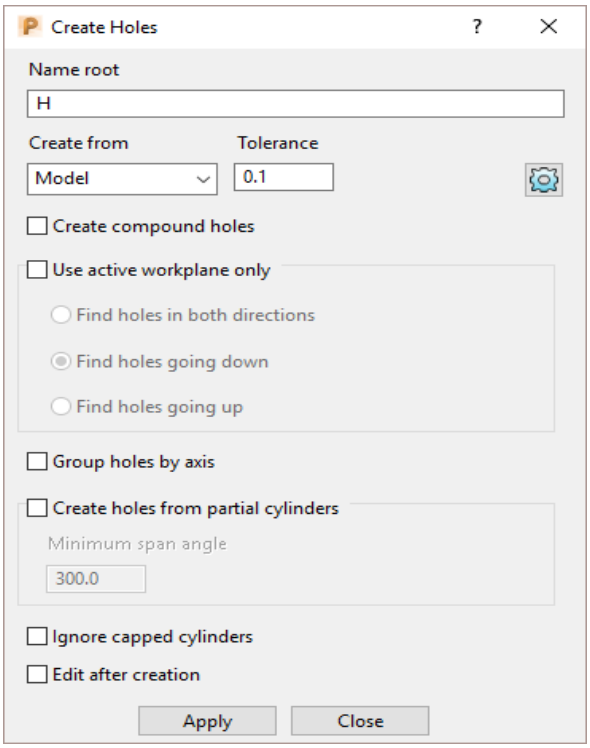
Any cylindrical surfaces within the selection will automatically be recognised as a **Hole Feature**. It is important that no **Block** exists to ensure that the **Hole Features** are arranged with the top at the **maximum Z height**.

The existence of a Block will affect the orientation of individual Hole Features in that they are more likely to be created to an unsuitable alignment.

4 Use the left mouse to select all of the **surfaces**.

5 In the **PowerMill *explorer***, right mouse click on **Hole Feature Sets**, and from the local menu select **Create Holes**.

6 Enter data into the **Create Holes** form exactly as shown.



The **Group holes by axis** option must be un-ticked for a **Multi Axis Hole** > **Feature Set** to be created (All of the ***Hole Features*** including those at different orientations will be input into a single **Feature Set**).

7 **Apply** and **Close** the dialog.

*All cylindrical surfaces within the selection will automatically be recognised as **Multiaxis Hole Features**.*

8 Undraw the model to view the newly created Features.

All 17 **Holes** in the **Feature Set** are correctly aligned, with the cross at the base representing the bottom of the hole.

9 Create a material **Block** using **Defined by** - **Box** and **Type** - **Model**.

10 In the **Rapid Move Heights** dialog, select **Calculate**.

11 Create a Dia 3 Drill named **Drill3** of Length 40.

12 Add a **Shank** component Upper\Lower Dia 3 Length 40.

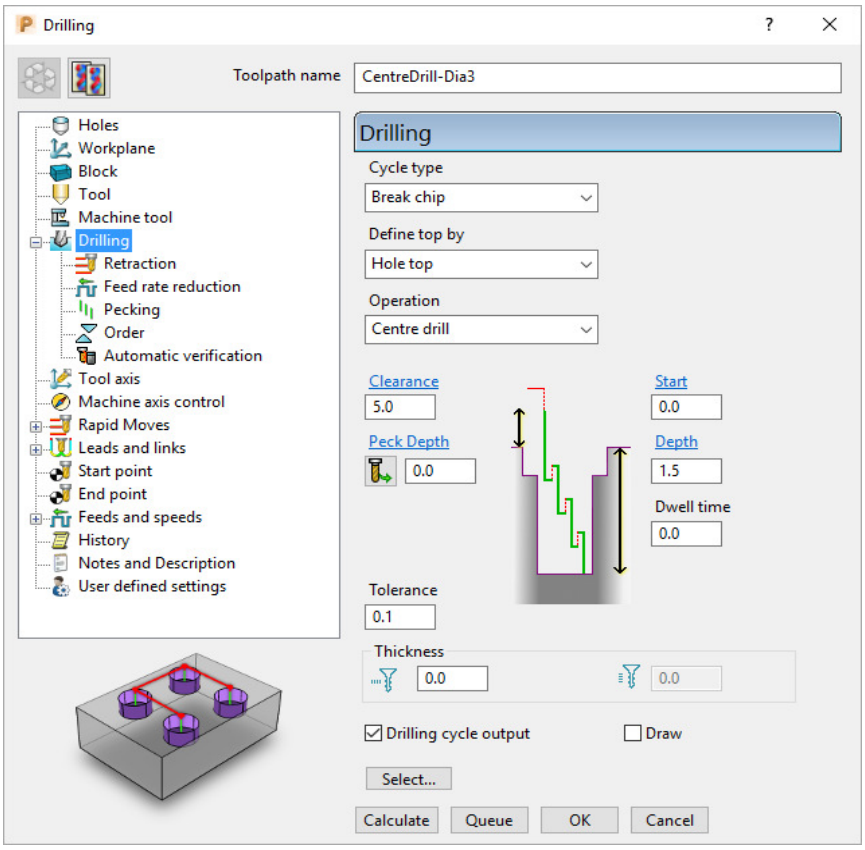
13 Add a **Holder** component Upper Dia 30 Lower Dia 30 Length 30 Overhang 60.

14 Select Toolpath Strategies and in the Strategy Selector dialog, select **Drilling**.

15 In the Drilling dialog, select the option **Drilling**.

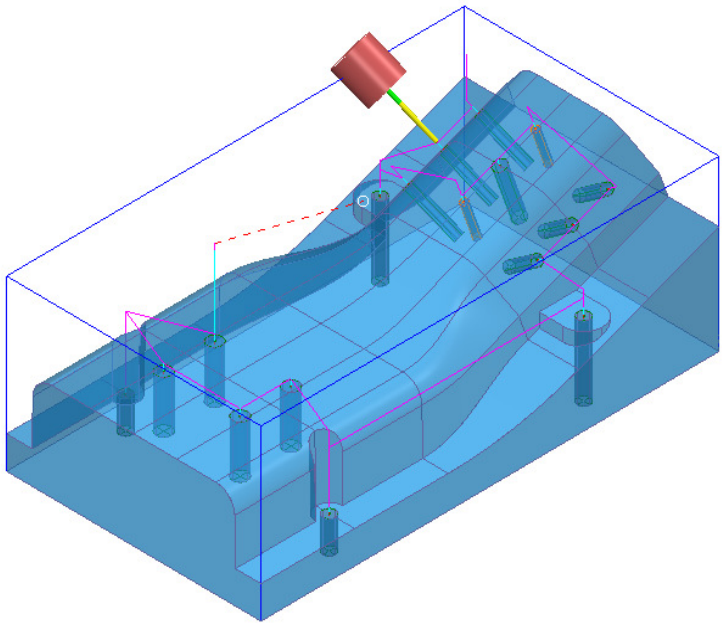
16 Rename the toolpath **CentreDrill-Dia3**.

Copyright © Autodesk



17 Back in the main **Drilling** page select **Calculate** to create the ***Centre Drilling*** toolpath, and then **Cancel** the dialog.

18 **Simulate** the toolpath.



The **Multiaxis** option has created all of the **Hole Features** aligned at different elevations in a single **Feature Set**.

The two **Dia 6 Hole Features** are to be drilled to depth with a **Dia 5 Drill** and then **Tapped**. To allow for the inevitable point angle at the bottom of the blind holes the **Tapping** operation needs to stop short of the full depth by applying a suitable **Axial Thickness** value.

19 Create a Dia 5 Drill named **DRILL5** of Length 60

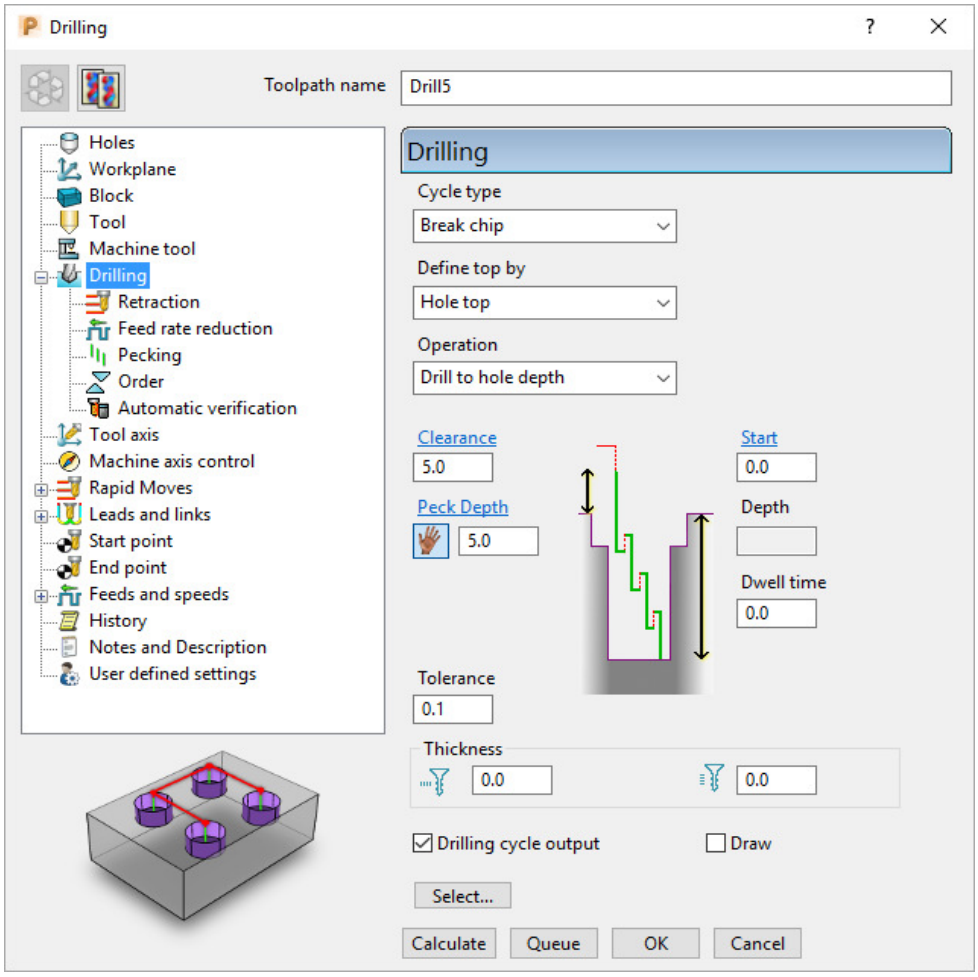
20 Add a **Shank**, Upper Dia 5, Lower Dia 5, Length 30.

21 Add a **Holder**, Upper Dia 30, Lower Dia 30, Length 30, Overhang 50.

22 Select **Toolpath Strategies** and in the **Strategy Selector** dialog select **Drilling**.

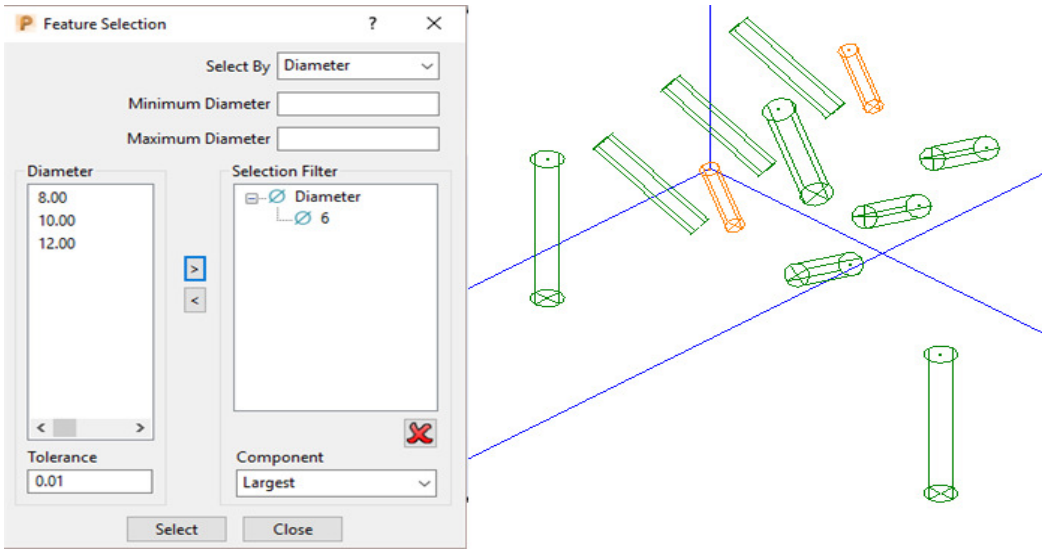
23 In the **Drilling** dialog select the option **Break Chip**.

24 Rename the toolpath as **Drill5**.



25 Select the two Dia 6 **Hole Features** (shown coloured orange below).

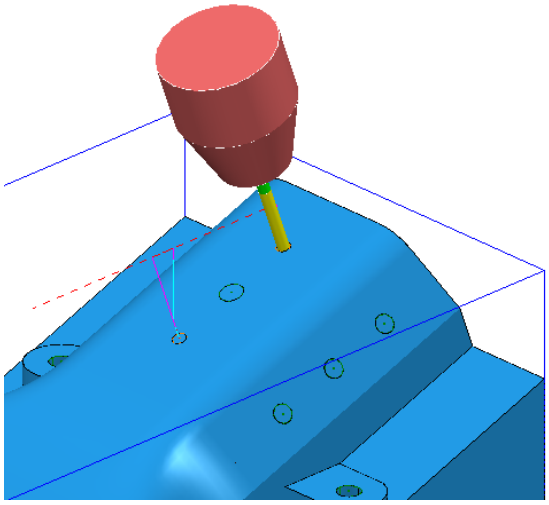
Copyright © Autodesk



26 **Close** the **Feature** Selection dialog.

27 Select **Calculate** to create the **Drilling** toolpath, then **Cancel** the dialog.

28 **Simulate** the toolpath.



The two ***Dia 6 Holes*** have now been drilled to full depth using a ***Dia 5 Drill***.

They will now be **Tapped** to a depth of **5** (***Axial thickness***) from the base of each ***Dia 6 Hole*** using a ***Dia 6 Tap*** with a **1mm *Pitch***.

29 Create a *Dia 6 Tap* (Named **M6Coarse-TAP**) with *Pitch* 1 and *Length* 25.

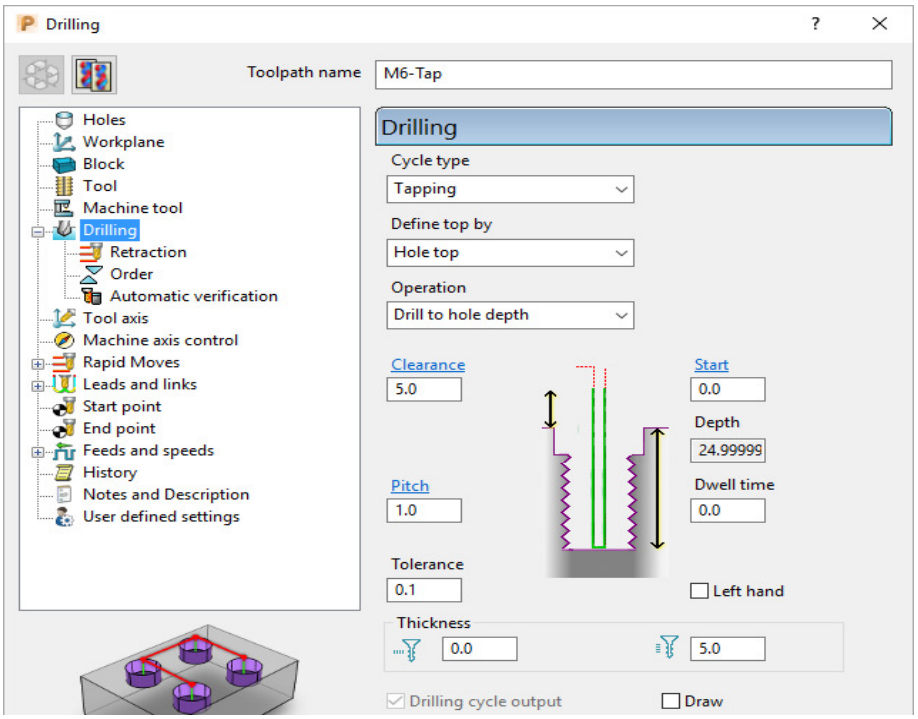
30 Add a **Shank**, Upper Dia 4, Lower Dia 4, Length 30.

31 Add a **Holder**, Upper Dia 30, Lower Dia 30, Length 30, Overhang 50

32 Select **Toolpath Strategies** and in the **Strategy Selector** dialog select **Drilling**.

33 In the **Drilling** dialog select the option **Tap**.

34 Rename the toolpath as **M6-Tap**.



35 Select the two Dia 6 **Hole Features**.

36 In the **Drilling** dialog select the option **Tapping**.

37 Rename the toolpath as **M6-Tap**.

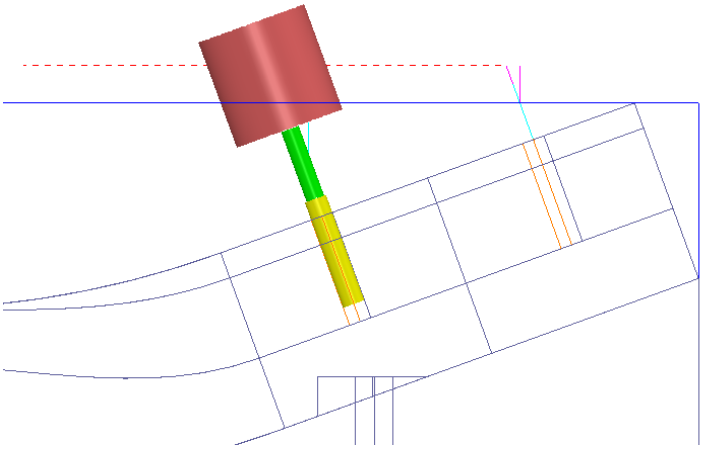
38 Set **Cycle Type** - **Tapping**, **Operation** - **Drill to hole Depth**, and **Pitch 1**.

39 Input an **Axial Thickness** value of **5**.

40 Select the two Dia 6 **Hole Features**.

41 Select **Calculate** to create the toolpath and then **Close** the dialog.

42 Select a **View from front (-Y)**.



The ***selected holes*** have been **Tapped** to a distance of 5mm short of the full depth.

43 From the **main** pulldown menus, select **File** > **Save** to update the contents of the stored **Project**.

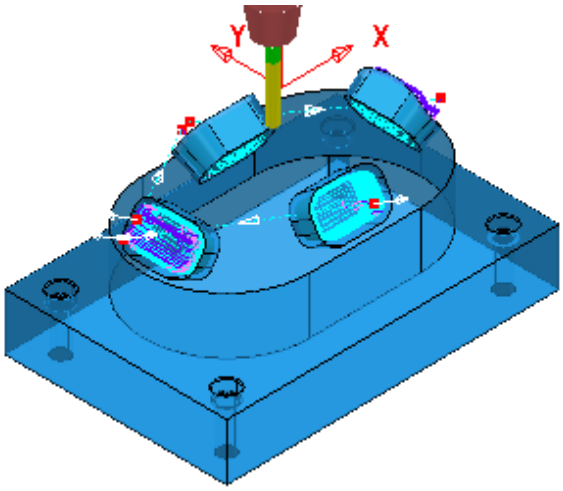
Copyright © Autodesk

2. Positional Tool Moves

Positional Tool Moves – Essential Information

When performing multi-axis, **Positional Tool Moves**, it is the user's responsibility to pay careful attention and take appropriate action to prevent collisions.

- All individual **toolpaths** should be created such that the **Start** and **End Points** are clear of any undercut features in the CAD model.
- **PowerMill** does not gouge check **positional** moves before and after an individual toolpath. These ***Pale Blue dashed*** connection moves are only visible if the **NCProgram** is drawn (as shown below).



- Neither, **PowerMill**, or a **Postprocessor** will be aware of any obstructions within the confines of a machine tool.
- The **Positional** moves displayed in a **PowerMill** toolpath **simulation** will not necessarily match all moves performed by the actual machine tool.
- For a more accurate simulation a **Machine Tool Model** along with an associated **mtd** file can be created. It must be stressed that a ***machine tool simulation*** is essentially a ***visual animation*** and **not** an ***ncprogram***, ***verification***.

Three suggested methods in **PowerMill** to create **Positional Tool Moves** include:

a. Use of **Absolute** coordinates in the toolpath **Start** and **End Point** dialog.

b. The insertion of a ***Drive Curve*** - **Pattern Finishing** strategy suitably positioned in 3D space.

c. The insertion of strategically placed **Workplanes** into an **NC Program**.

Copyright © Autodesk

Postprocessor option file – Safe Z Positional Moves

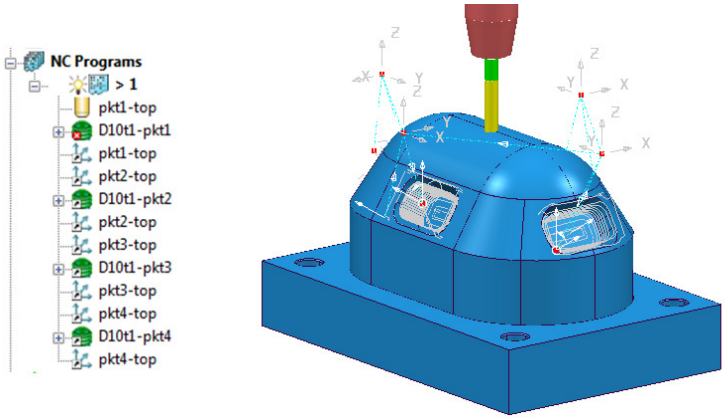
There are two different ways that a specific **Postprocessor Option file** can be created to deal with positional moves (Method 1 is the default).

a Method 1 – The **Postprocessor Option file** automatically outputs moves from/to the **Machine Tool**, ***maximum Z height*** (***Home***) at the start and end of each **Toolpath**. Note! In this case any **Workplanes** inserted into the **NC Program** (Item **c** on the previous page) are completely ignored.

The name for a typical **Postprocessor Option file** using Method 1 includes **ZMAX** at the end of the name:- **DMG_DMU10p_H530_ZMAX.pmopt**

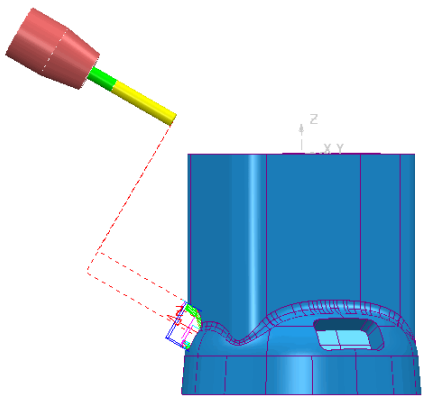
b Method 2 – The **Postprocessor Option file** does not automatically output Machine Tool **Safe Z** moves. All additional, safe positional, moves must be added by the user in **PowerMill** (In this case a network of **workplanes** can be used as shown in the illustration below).

The name for a typical **Postprocessor Option file** using Method 2 will not include **ZMAX** at the end of the name:- **DMG_DMU10p_H530.pmopt**



Tool Move with Start and End Point

Positional Tool Moves can be controlled in the **Start and End Point** dialog by use of **Absolute** (along with specific XYZ coordinates).



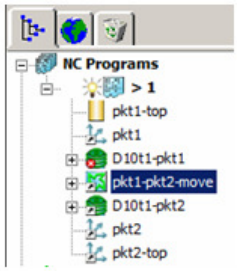
Start and End Points input as **Absolute** coordinates, above the component where it is safe to re-align the tool and/or perform rapid XY moves.

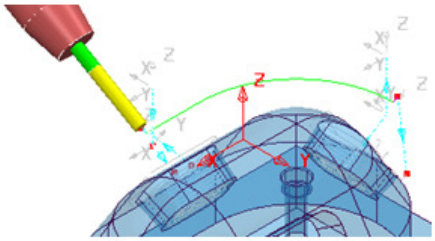
Copyright © Autodesk

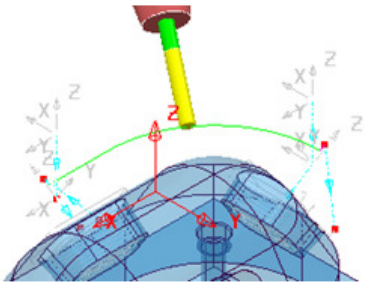
Tool Moves using a Pattern Finishing in 3D Space

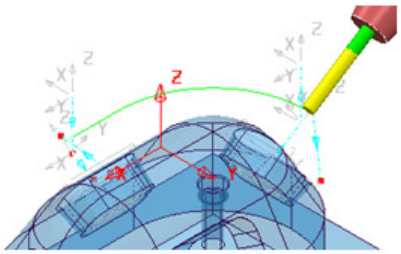
Positional Tool Moves can be controlled by running the tool along a **Pattern Finishing** strategy used as a ***Drive Curve*** in 3D space.

*A **lean** angle can be applied to maintain an angular **tool alignment** during transit.*





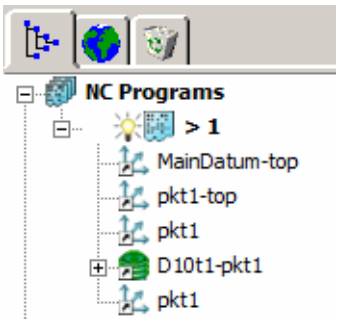




Tool Move via Workplanes in an NC Program

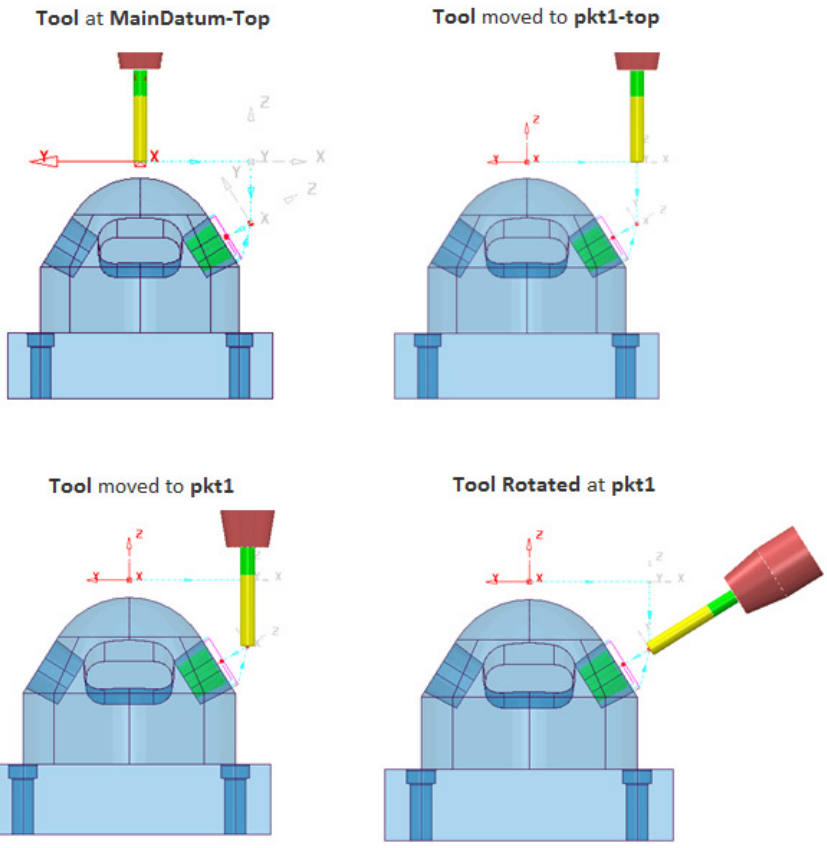
Positional Tool Moves can be controlled by strategically placed **Workplanes** inserted between the **Toolpaths** in the **NC Program** list. A **Workplane** in the **ncprogram** list can also be registered as a **Toolchange Point** if required.

*This method is only applicable to non-**ZMAX post-processor** applications.*



While a **tool** moves to a **Workplane**, it will simultaneously re-align to align to the **Workplane - Z Axis** (**Simultaneous** is the **NC Preferences** - ***default*** setting).

The following 4 illustrations show the tool movements to 3 workplanes including a rotational move prior to performing the machining strategy.

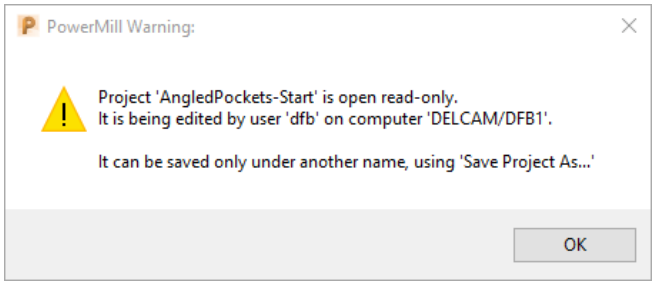


Example

An existing **Project** will be opened that contains 4 separate **3 Plus 2**, finishing toolpaths ready to be added to an **NC Program**. Once included in the **NC Program**, appropriate positional moves must be added to prevent the tool from passing through the component form while moving between toolpaths.

1 Select **File** - **Open Project**:-

....\PowerMill_Data\FiveAxis\PositionalMoves\AngledPockets-Start

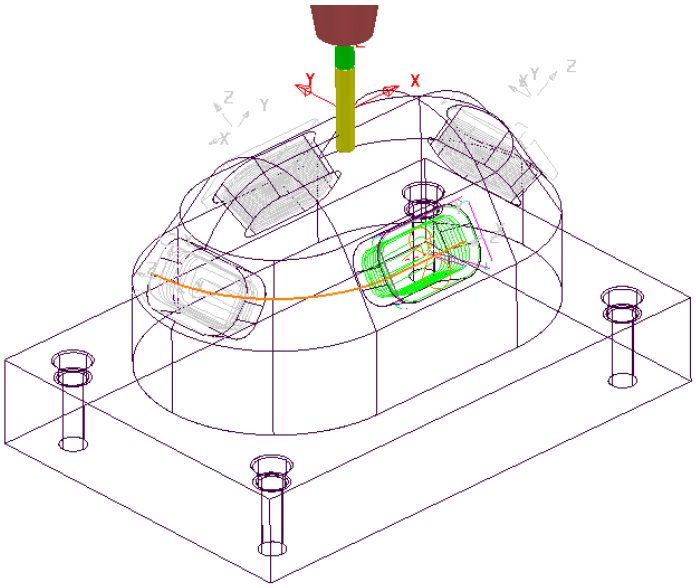


2 Select **File** - **Save Project As**:-

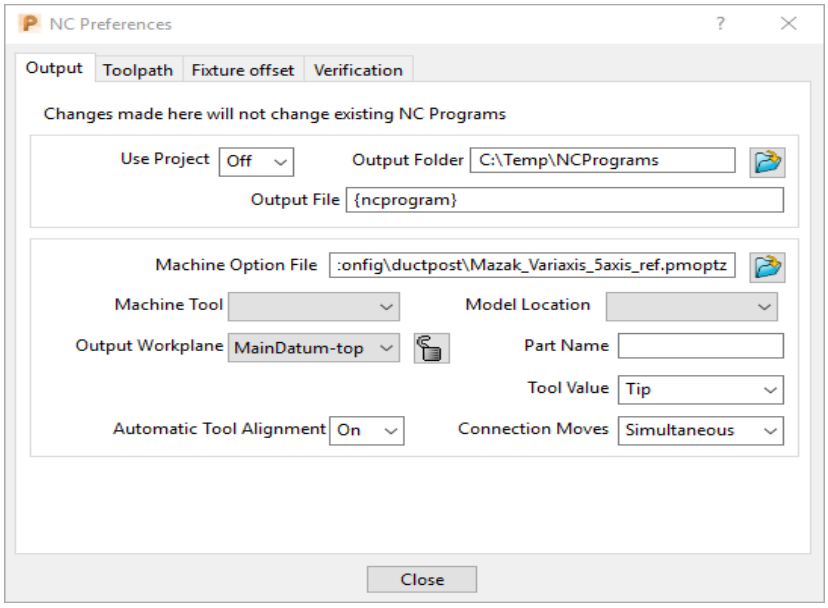
....\COURSEWORK\PowerMill_Projects\AngledPockets

Copyright © Autodesk

Connection moves using a Safe Z calculated to a 3-Axis Workplane



3 In the **PowerMill *explorer***, right-click on **NC Programs** and from the local menu, select **Preferences**.



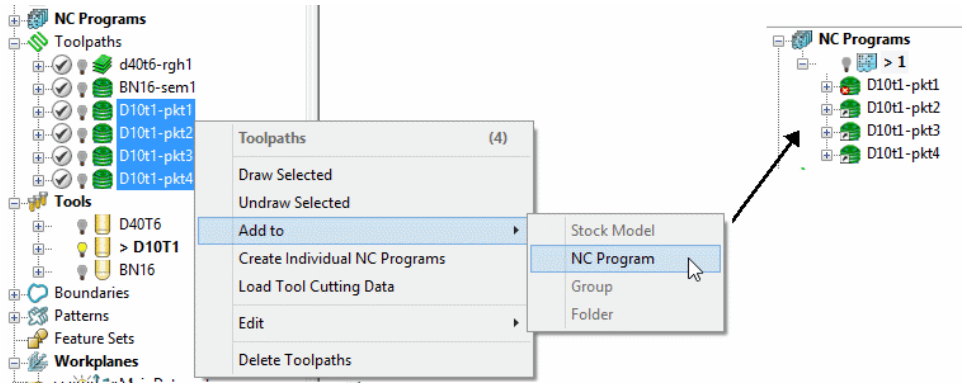
4 Select a suitable ***Five Axis***, Post-Processor ***Option File***:

C:\dcam\config\xtra-posts\ Mazak_Variaxis_5axis_ref.pmoptz

5 Select **Close** (The **NC Preferences** settings will be updated).

6 Create a new **NC Program** (It will be **Active** on creation)**.**

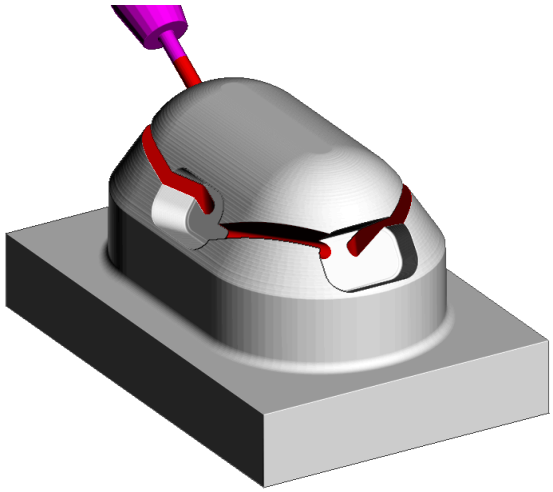
7 Select the four ***3plus2*** **Toolpaths** (***D10t1-pkt….***) then right-click over them and in the local menu select **Add to** > **NC Progam**.



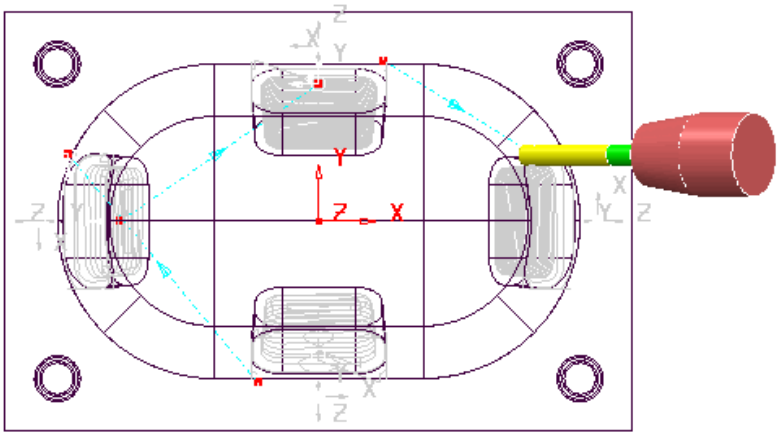
8 Select an **ISO 4** view.

9 Run a **ViewMILL** simulation on the first two, 3 Axis toolpaths; **d40t6-rgh1**, **BN16-sem1**, followed by the **NC Program** containing the four 3plus2 toolpaths.

*On closer inspection it will be observed that the **3plus2** - **Positional Tool Moves** have ploughed through the component between the individual machining strategies.*



*If the **Model** is displayed as **wireframe** then the displayed **NC Program** will also include the **link** moves (Pale Blue dashed lines) between the 4 stored **Toolpaths**.*

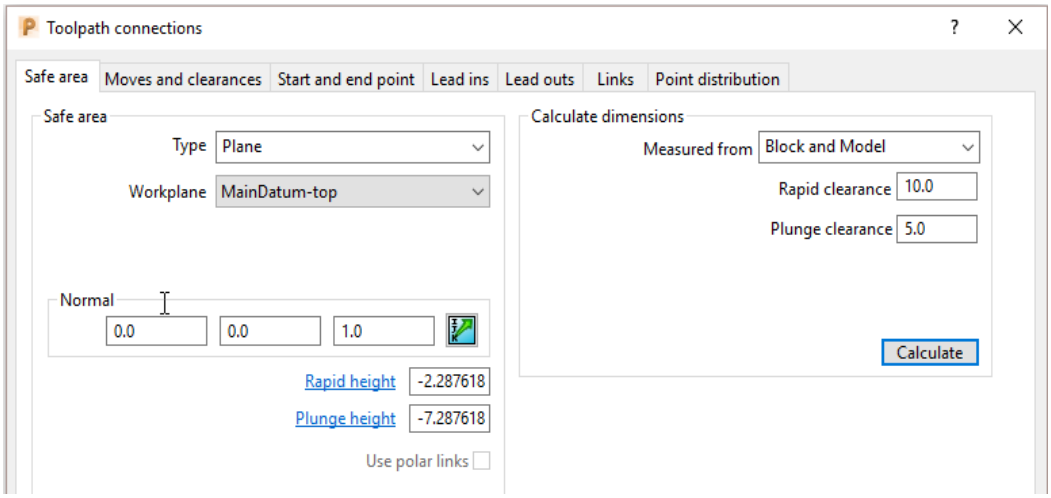


Copyright © Autodesk

10 **Activate** the **workplane MainDatum-top**.

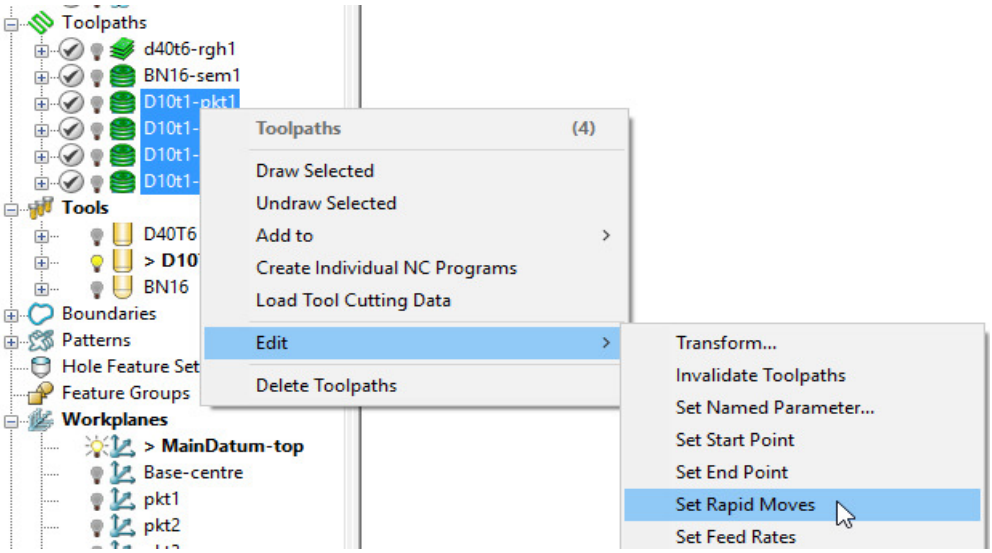
11 **Calculate** a **Block** using ***Defined by*** **Box** to the **Model** dimensions.

12 Open the **Rapid Move Heights** dialog.



13 Input ***Workplane*** **MainDatum-top** and **Calculate** before selecting **Accept**.

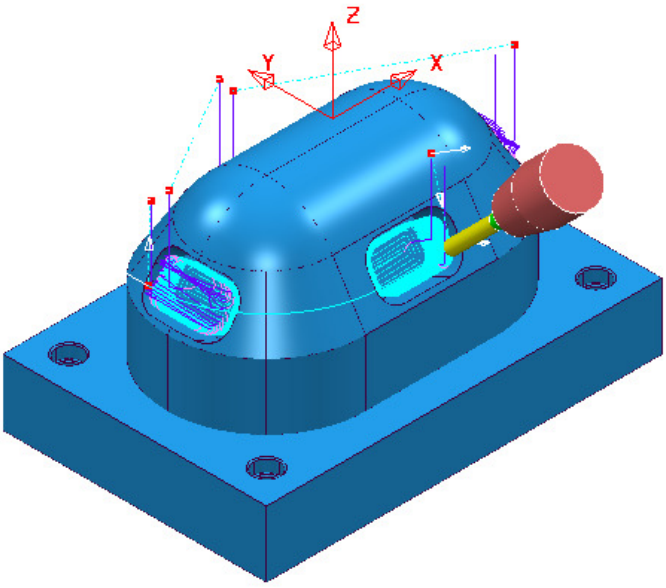
14 In the **PowerMill explorer** use the **Shift** key to multi-select the four, 3+2 **toolpaths**.



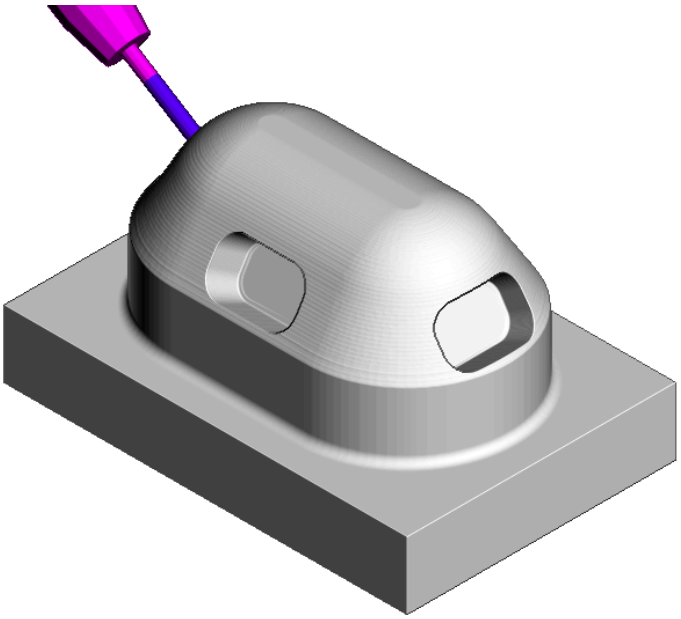
15 Right mouse click anywhere on the four selected **toolpaths**, and from the local menu select **Edit** – **Set Rapid Moves**.

Copyright © Autodesk

The **Tool** will now perform rapid moves above the top of the **model**, relative to **workplane MainDatum-top**.



16 Run a **ViewMILL** simulation on the first two, 3 Axis toolpaths; **D40t6-rgh1**, **Bn16-sem1**, followed by the **NC Program** containing the four 3plus2 toolpaths.

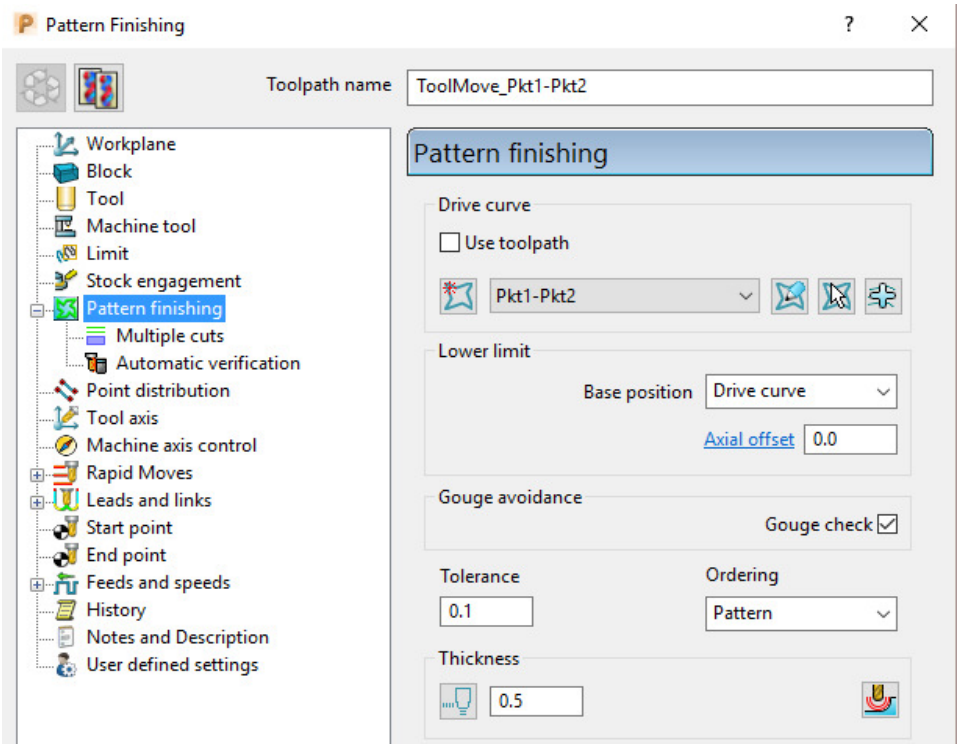


The toolpath connection moves no longer collide with the model.

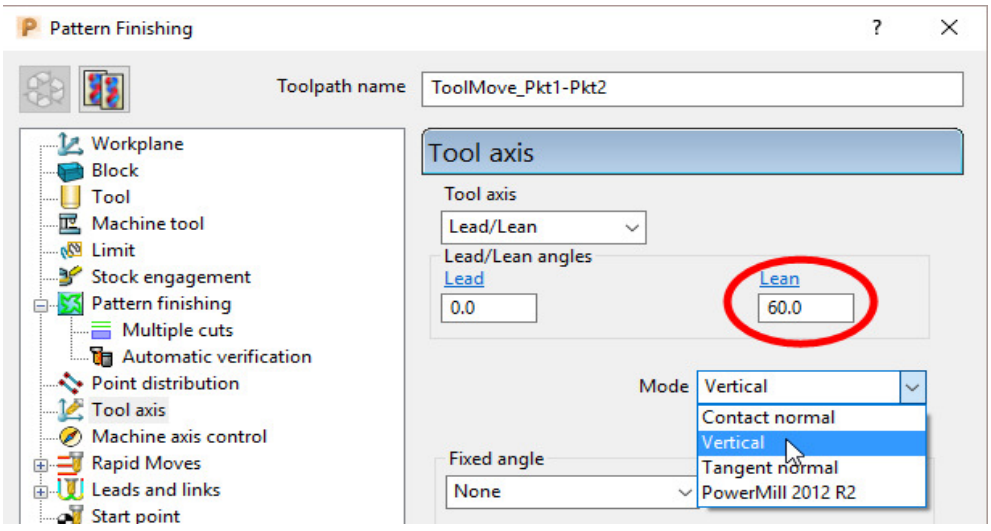
Copyright © Autodesk

Connection moves using (Drive Curve) Pattern toolpaths

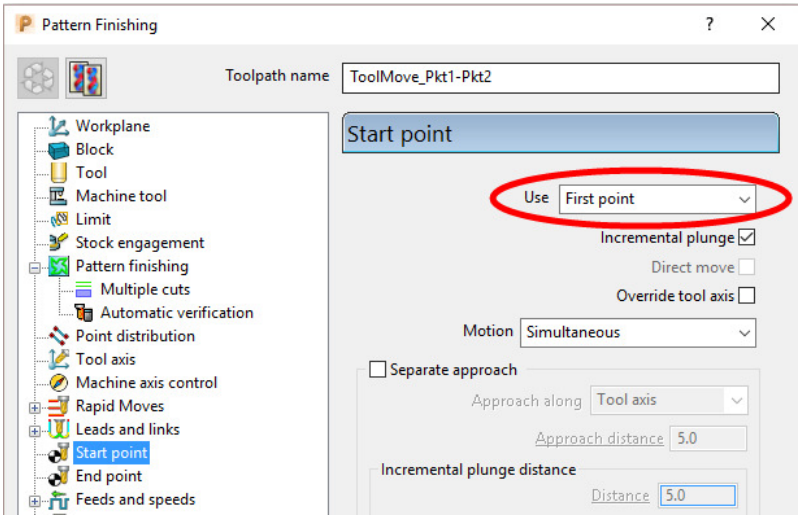
17 From the **Strategy Selector** - **Finishing** select the **Pattern** finishing strategy and enter data exactly as shown below.



18 Select the **Tool Axis** page, and input a ***Tool axis*** **Lead/Lean** with ***Mode*** **Vertical** selected and a Lean value of **60**.



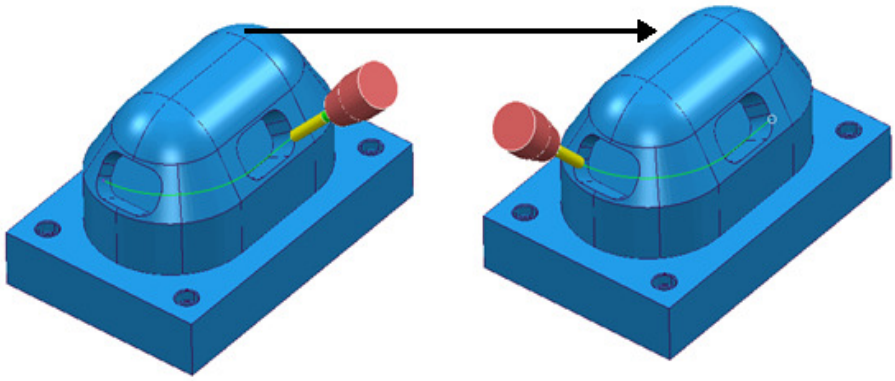
19 Select the **Start point** page, and ***Use* First point**.



20 Select the **End point** page and ***Use* Last Point**.

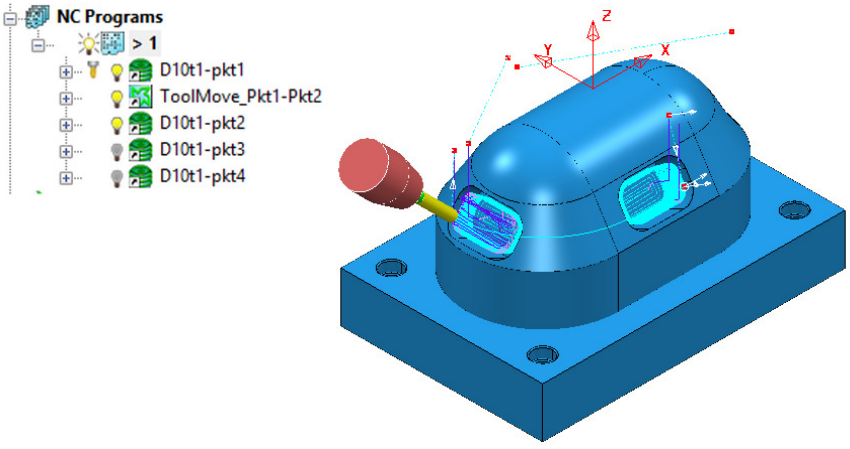
21 In **Leads and links** set both **Lead in** and **Lead out** to **None**.

22 **Calculate** the **Pattern** finishing toolpath.



The **Tool** 'winds' along the **Pattern Finishing** strategy creating a collision safe, transition around the component.

23 Insert the **Pattern Finishing** strategy (**ToolMove-Pkt1-Pkt2**) between the toolpaths, **D10t1-pkt1** and **D10t1-pkt2**.



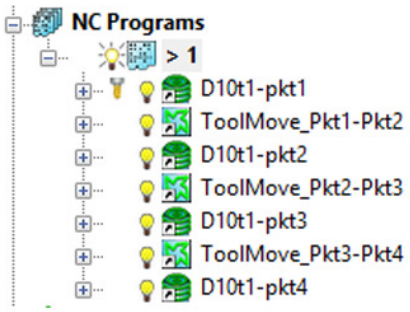
Copyright © Autodesk

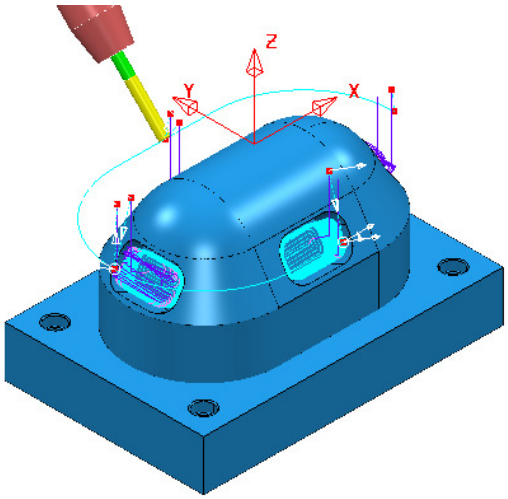
Exercise

1 Create 2 more suitably positioned, tool move **Pattern Finishing** toolpaths named **ToolMove_Pkt2-Pkt3** and **ToolMove_Pkt3-Pkt4**.

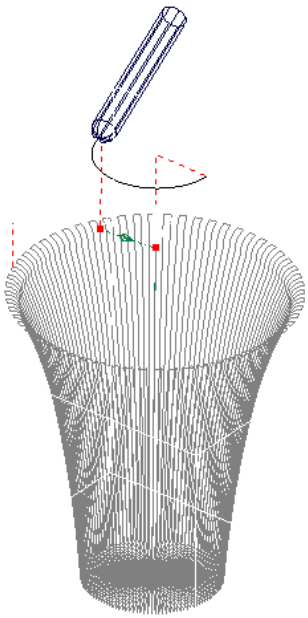
*Check the **direction** of each new **Pattern Strategy** and if necessary, reverse.*

2 Insert the new **Pattern Finishing** toolpaths into the **NC Program**.





Tool Axis 'Rewind' Move



*A useful application for using a **Pattern** as a **Positional Tool Move** is in cases where a **rotary axis limit** has been reached. A circular, **Pattern Finishing** strategy can be used to '**Rewind**' the **tool** in 3D space, back to the start of its rotary travel limits.*

Copyright © Autodesk

3. Five Axis Tool Alignment

Introduction

For **5-Axis** applications where the machine tool head and\or table, are required to perform both ***linear*** and ***rotational*** axis moves simultaneously, **PowerMill** provides a range of appropriate **Tool Axis Alignments** and **Machining Strategies**.

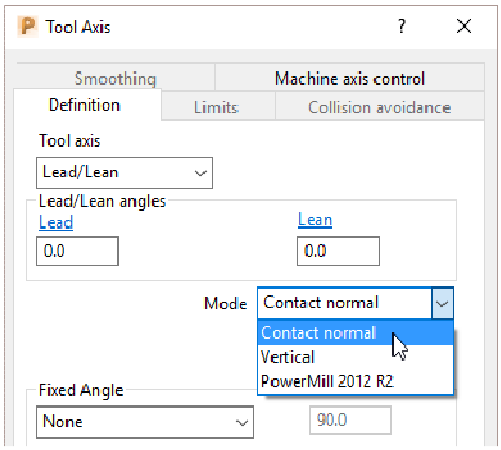
Five Axis Tool Alignment and Machining Options

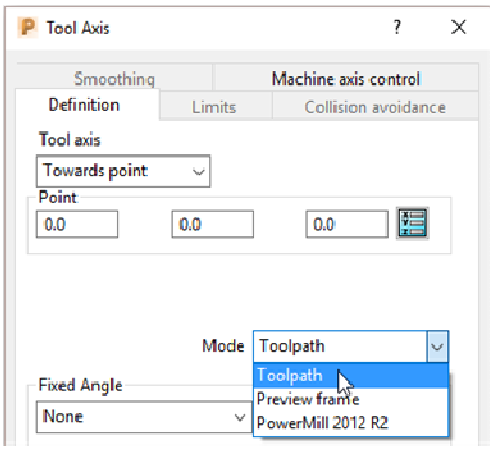
By default the **Tool Axis** alignment in **PowerMill** is set to **Vertical** for **3-Axis** applications and other options will only be available to users with a multiaxis licence.

Some strategies only support multiaxis Tool Axis alignments if used with Ballnose or Spherical tools.

The **Tool Axis Direction** dialog is accessed via the **Tool Axis** icon located in the **Main** toolbar or directly from supported **Machining Strategy** forms.

The dialog provides a list of basic **Tool Axis** alignment options which are further controlled by a ***Strategy*** dependent selection of **Mode** options.





Tool Axis alignment - Mode options

a **Contact Normal** - This measures a **Lead** angle relative to the toolpath contact normal.

b **Vertical** - This measures a **Lead** angle relative to the ***Z Axis***.

c **PowerMill 2012 R2** - Reverts back to this version that did not offer a choice of **Mode** options (The actual **Mode** options used is strategy specific).

d **Preview frame normal** - This measures a **Lead** angle relative to the projection direction to or from the ***normal*** to the ***Projection Strategy***, **Preview** pattern.

e **Toolpath** - For a basic **Tool Axis** of **From** or **Towards** the actual alignment is relative to the ***model surface*** instead of the **Preview** pattern.

f **Preview frame** - For a basic **Tool Axis** of **From** or **Towards** the actual alignment is relative to the ***Projection Strategy***, **Preview** pattern.

g **Reference tool axis** – This measures the **Lead** angle form the tool axis used in a reference toolpath used as the **Drive** curve in a **Pattern Finishing** strategy.

h **Tangent Normal** - This measures the **Lead** angle relative to the perpendicular to the tangent direction of a **Pattern** used as the **Drive** curve in a **Pattern Finishing** strategy.

i **Undercut angle** – This measures the **Lead** angle from the projection direction corresponding to the specified undercut angle in a **Parametric Spiral** strategy.

Lead/Lean - Example1

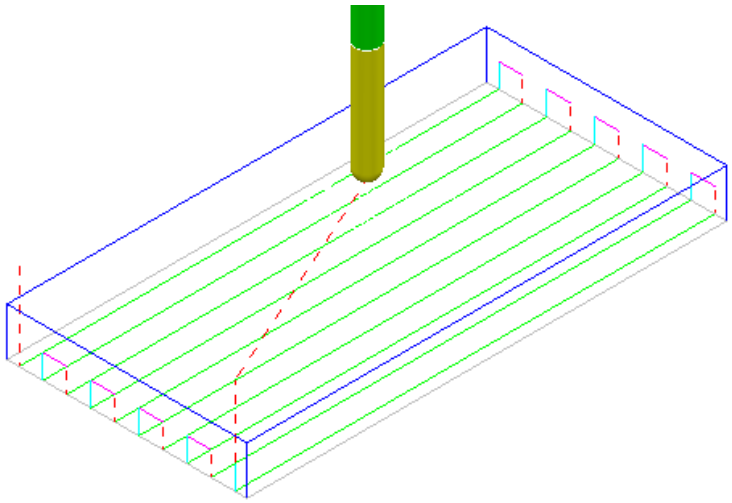
Lead allows the tool to be aligned to a specified angle ***along*** the ***toolpath direction*** and **Lean** a specified angle ***across*** the ***toolpath direction***. If both angles are zero the **tool** will be aligned along the direction that a **toolpath** is projected onto the model during creation. For **Pattern** finishing this will always be vertical and for **Projection Finishing** it will vary depending on the defined strategy projection options.

1 From the **main** pull down menus select **File** - **Delete All** and **Tools** - **Reset Forms**.

Copyright © Autodesk

2 **Open** the (Read Only) **Project**:-

….\Five_Axis\ToolAlignments\PlaneAlignments_Start



3 **Save Project As**:-

….\COURSEWORK\PowerMill_Projects\PlaneAlignments_Demo

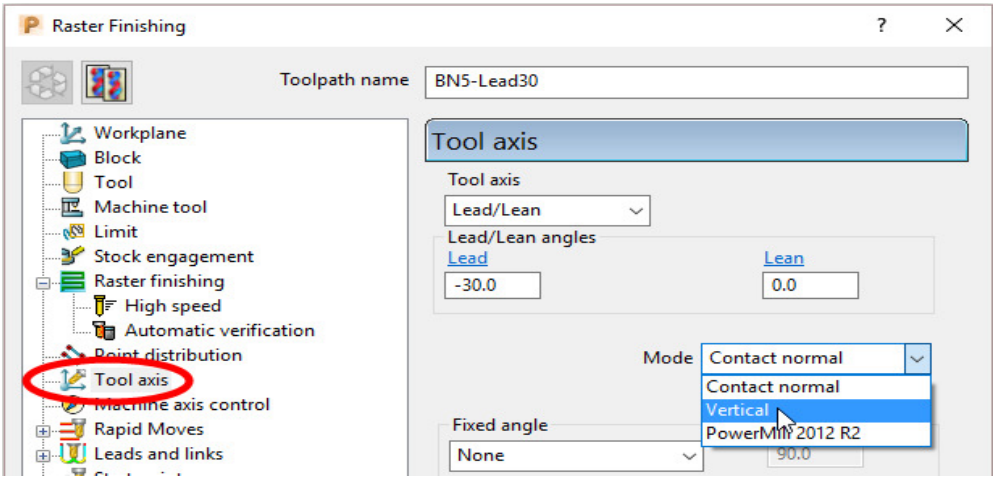
4 **Activate** and then **Simulate** the toolpath **BN5-Vertical**.

The above **Raster** toolpath is machined with the tool aligned to the **vertical** axis.

5 Right-click the **Toolpath - *Raster Vertical*** in the **PowerMill *explorer*** and select **Settings** to open the original **Raster Finishing** dialog.

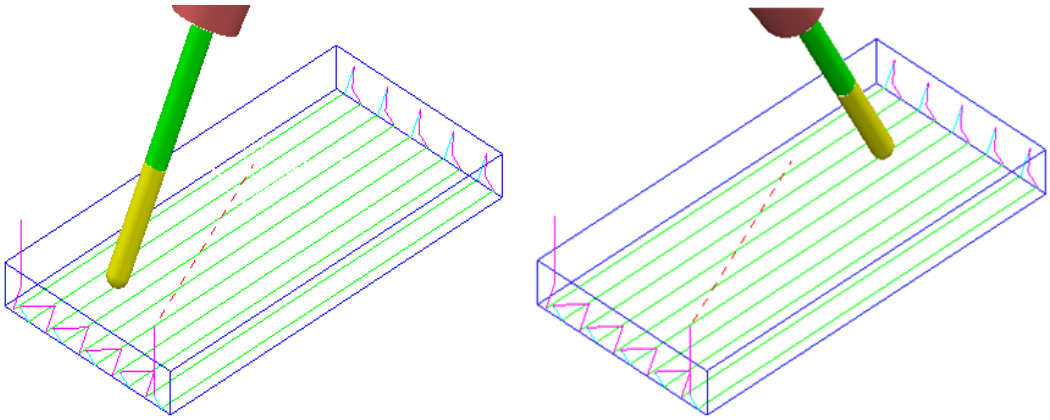
6 **Make a Copy** of the **Toolpath** and rename it as **BN5-Lead30**.

7 Select the **Tool axis** page, with the ***Tool axis*** as **Lead\Lean**, with **Lead -30**, and use ***Mode*** **Vertical**.



8 **Calculate** the **Toolpath** and then **Close** the dialog.

9 **Simulate** the **Toolpath** (**BN5-Lead30**).



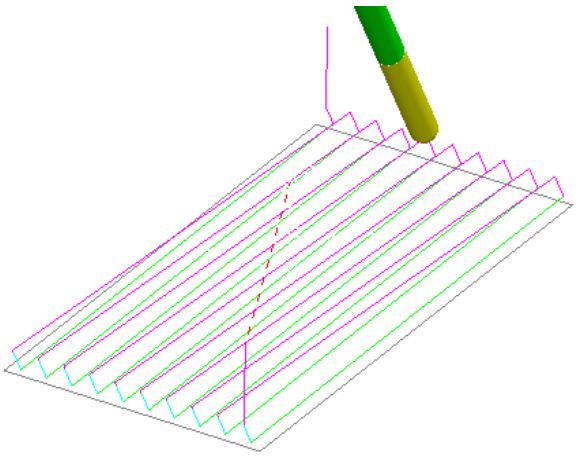
A **Raster** toolpath has been created with the **Tool axis** direction set to **Lead** at **-30** degrees relative to the ***vertical axis*** (**Z**).

*With the **Two Way** option, the **Tool Axis** direction will switch on opposing passes. This is due to the **Lead** angle being dependant on the tool track direction.*

10 In the **PowerMill *explorer***, right-click on the toolpath **BN5-Lead30** and select **Settings** to open the **Raster Finishing** dialog.

11 Select ***Re-cycle*** and change ***Style*** to **One Way**.

12 **Calculate** the **Toolpath** and then **Close** the dialog.



*With the **Style** set to **One Way** the **Tool Axis** direction remains fixed.*

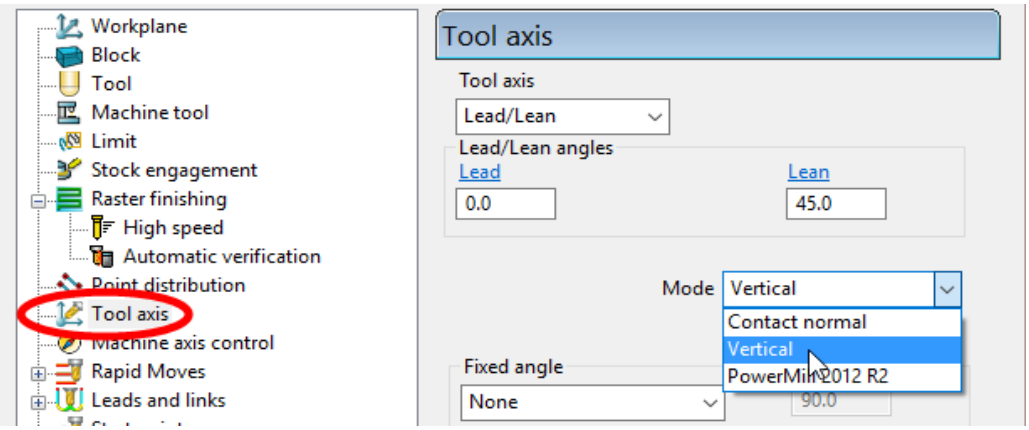
13 Right-click the toolpath **BN5-Lead30** in the **PowerMill *explorer*** and then select **Settings** to open the toolpath dialog.

14 Select **Make a Copy** of the **Toolpath** and rename it as **BN5-Lean45**.

15 Select the **Tool Axis** icon to open the **Tool Axis Direction** dialog.

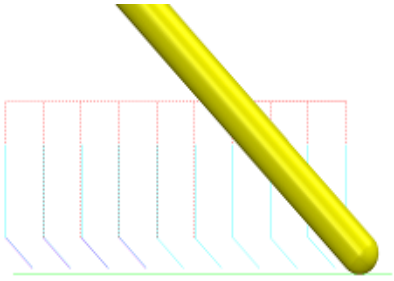
Copyright © Autodesk

16 In the **Tool Axis** page, change the **Lead angle** to **0**, and input **Lean 45**.



17 **Calculate** the **Toolpath** and then select **Cancel** to close the dialog.

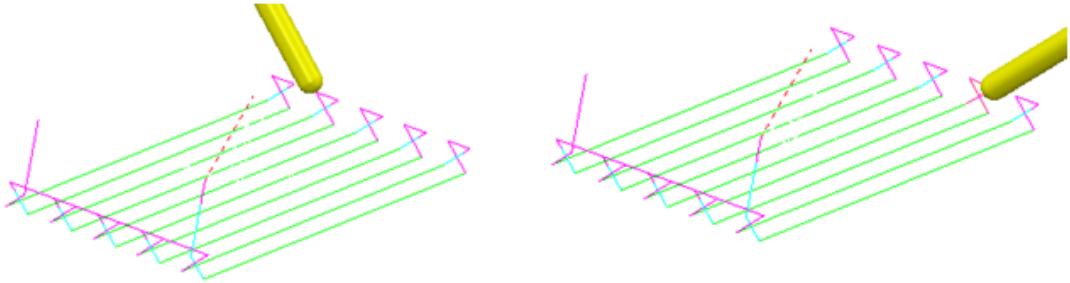
18 **Simulate** the **Toolpath**.



View from left -X

A **Raster** toolpath has been created with the **Tool Axis Direction** set to **Lean 45** degrees across the **toolpath**.

If a **Two Way** had been directly applied in the strategy dialog, the tool axis would lean in the opposite direction across alternate tool tracks.

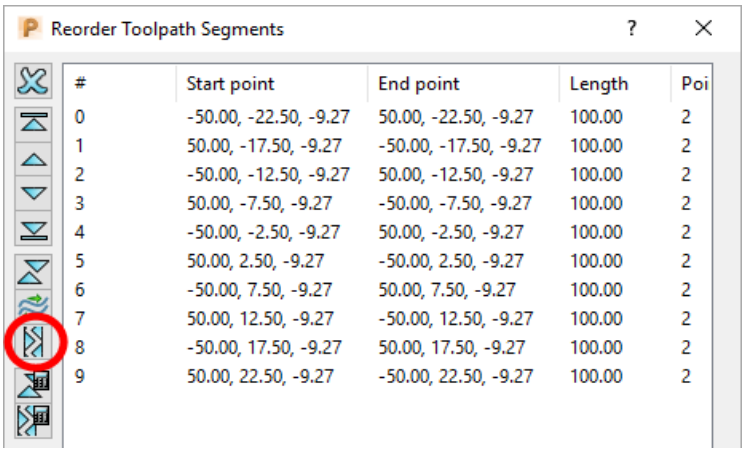


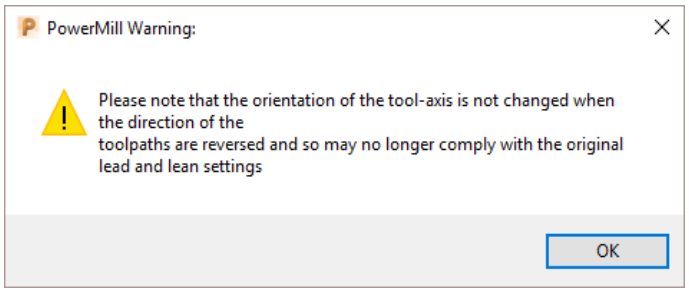
It is possible to create a **Two Way** strategy with a constant **Lean** direction by retrospectively editing a ***one way*** - **Toolpath**.

19 *Right-click* on the (***One Way***) toolpath **BN5-Lean45** and from the local menu, select **Edit** > **Reorder** to open the **Reorder Toolpath Segments** dialog.

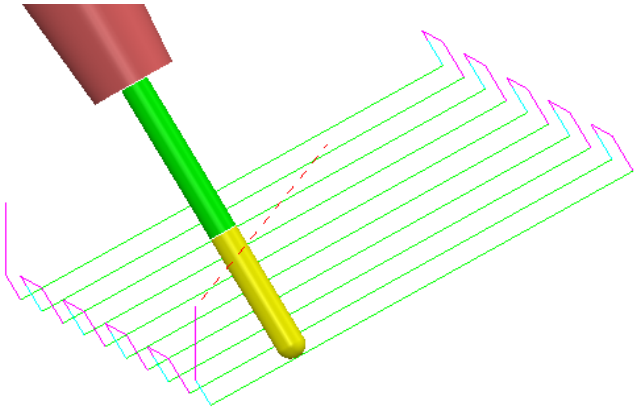
20 Select the **Alternate directions** option (All tool tracks will be edited).

Copyright © Autodesk





21 Select **OK** to close the above **PowerMill Warning** dialog.



22 Save **Project** to update:-

….\COURSEWORK\PowerMill_Projects\PlaneAlignments_Demo

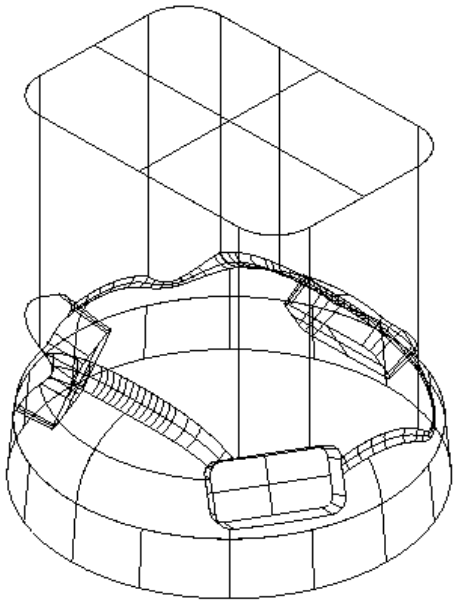
Copyright © Autodesk

Lead/Lean - Example2

23 Select **File** - **Delete all** and **Tools** - **Reset Forms**.

24 **Open** the **Project** (saved earlier during Chapter 1) from:

....\COURSEWORK\PowerMill-Projects\3Plus2-EX1



25 Define a **Dia 15mm Ball Nose** cutter named **BN15**.

26 **Activate** the **Workplane** - **ztop175_A**.

27 Create a material **Block** defined by **Box** to the ***Max\Min*** **Limits** of the **Model** and then ***edit*** the following values as shown below:

Xmin -95 Xmax -52.5 Ymin -50 Ymax 50

28 Open the **Rapid Move Heights** dialog and select **Calculate**, then **Accept**.

29 In the **Start and End Point** dialog in ***Start Point*** set **First Point Safe** and in ***End Point*** set **Last Point Safe**.

30 Select **Leads\Links** and in the **Toolpath connections** dialog set:-

Moves and Clearances:

Axial clearance **15**, ***Incremental plunge distance*** **5**

Lead In\Out: **Vertical Arc**: ***Angle*** **90, *Radius* 6**

Links: **1st choice\2nd choice**: ***Skim*** and ***Default:*** **Incremental**

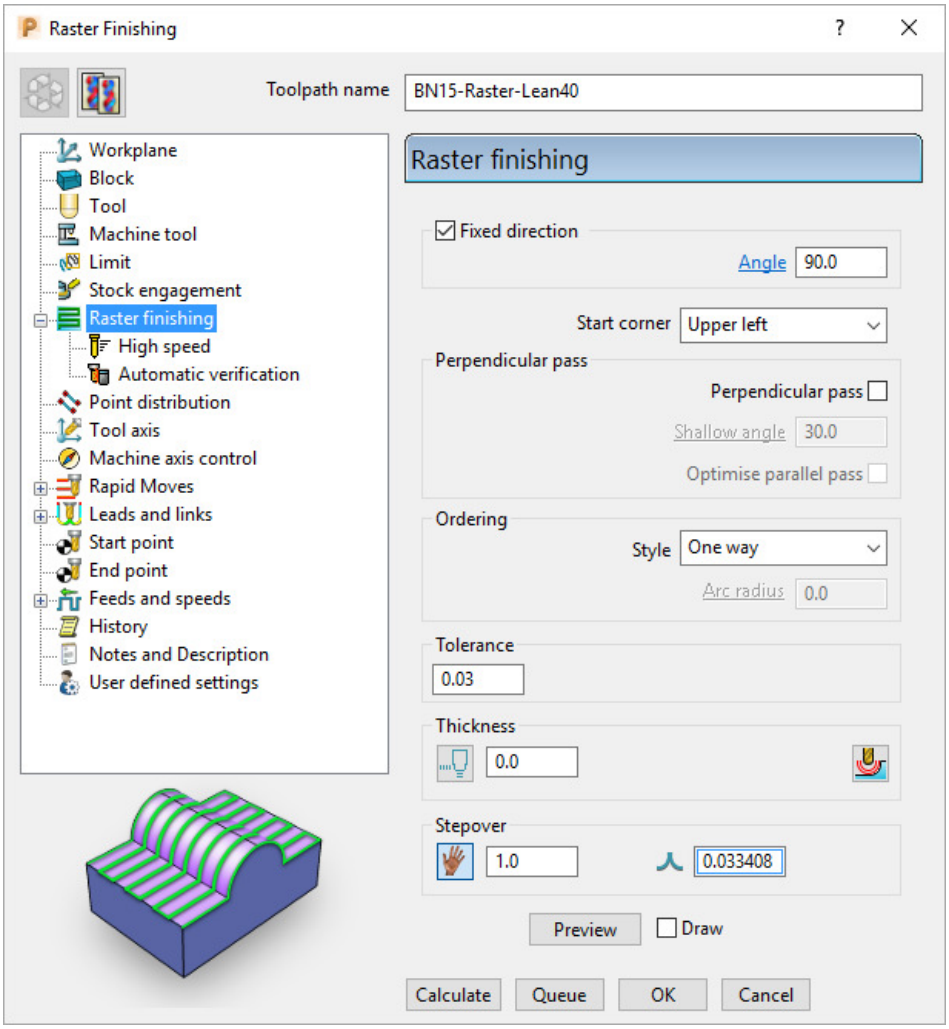
31 Ensure that no **Boundary** is **Active**.

An area of the component will be machined using **Raster Finishing**. The **Tool Axis** will be aligned to a fixed ***Lean angle*** of **40** degrees relative to the **Vertical** axis.

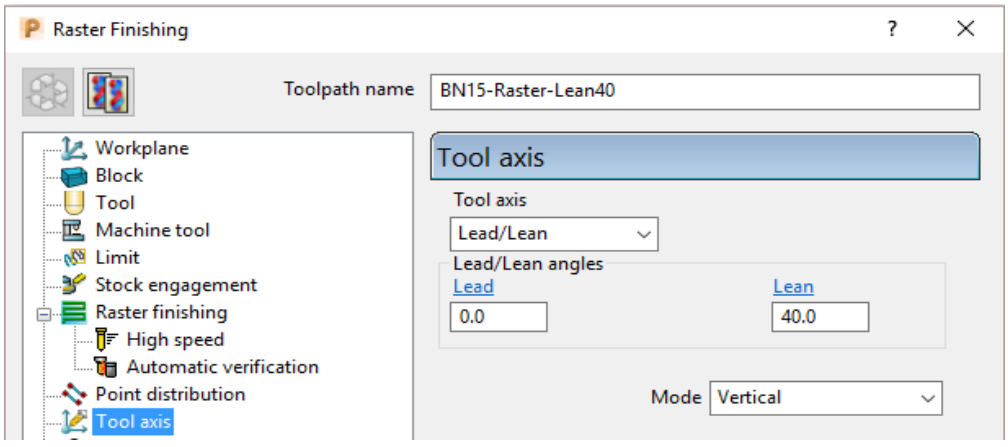
Copyright © Autodesk

32 Select the **Toolpath Strategies** icon and in the **Strategy Selector** dialog, select the **Finishing** tab.

33 Select the **Raster Finishing** dialog and enter data exactly as shown below:

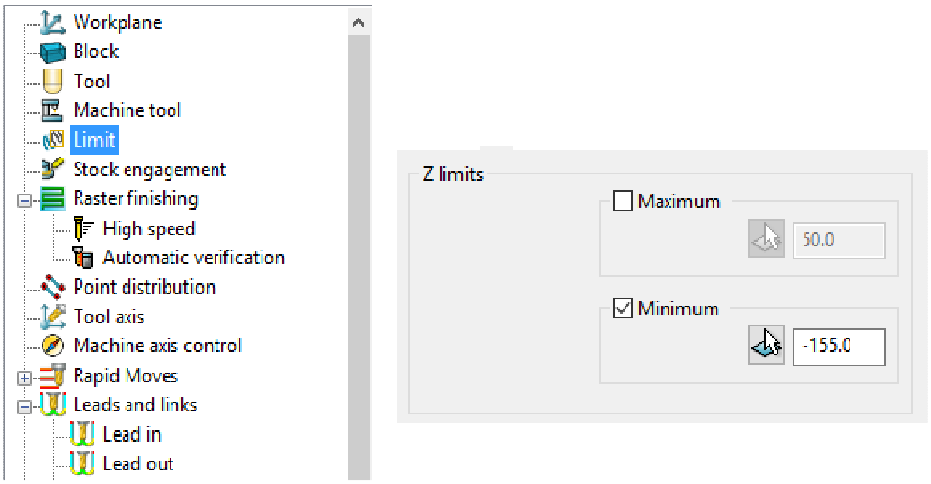


34 Select the **Tool axis** page and enter data into the dialog exactly as shown in the following illustration.

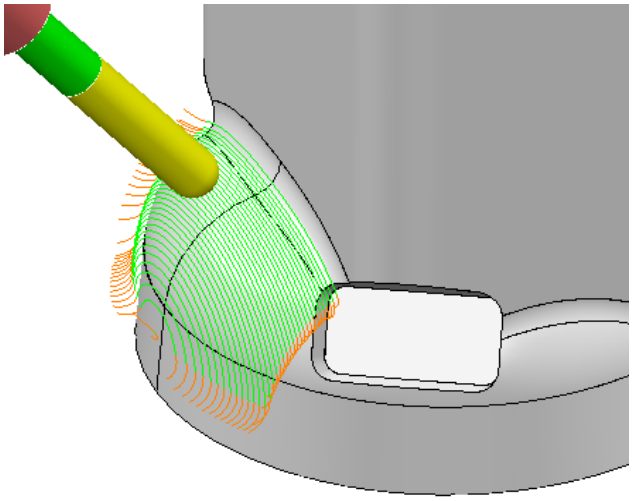


Copyright © Autodesk

35 In the **Limit** page, **Z limits**, tick **Minimum** and input the **value -155**.



36 **Calculate** the **Toolpath** and then **Cancel** to close the dialog.



The resultant toolpath starts at the lower corner and progresses towards the centre using a climb milling action (**One Way**). If it is required for the strategy to be **Two Way** then it can only be retrospectively changed using **Edit** > **Reorder** > **Alternate Direction**.

To compare the effect of applying a different machining strategy, the same area of the component will be machined using **Projection Plane Finishing**. This time the **Tool Axis** will be aligned using a **Lean** of **0** degrees relative to the ***projection direction*** of the strategy.

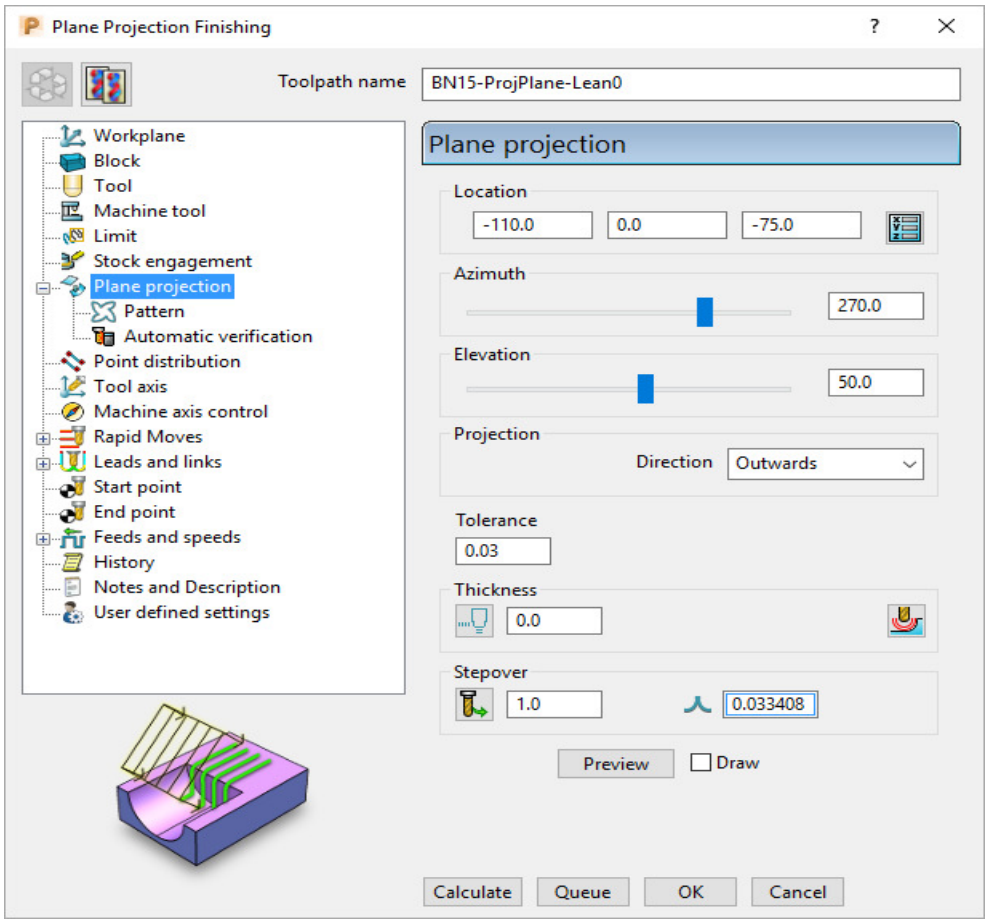
37 **Calculate** a **Block**, ***Defined by*** - **Box** using ***Type*** - **Model**.

38 Select the **Toolpath Strategies** icon and in the ***Strategy Selector*** dialog select the **Finishing** tab.

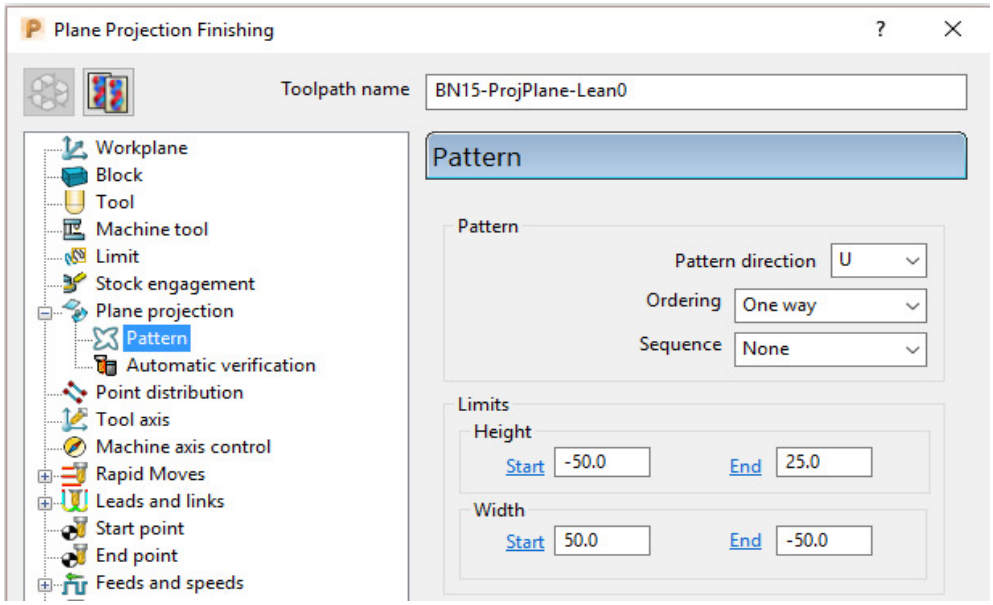
39 In the **Finishing** options select **Projection Plane Finishing**.

40 Enter data into the following 3 pages of the Plane Projection Finishing dialog, exactly as shown below:

Copyright © Autodesk

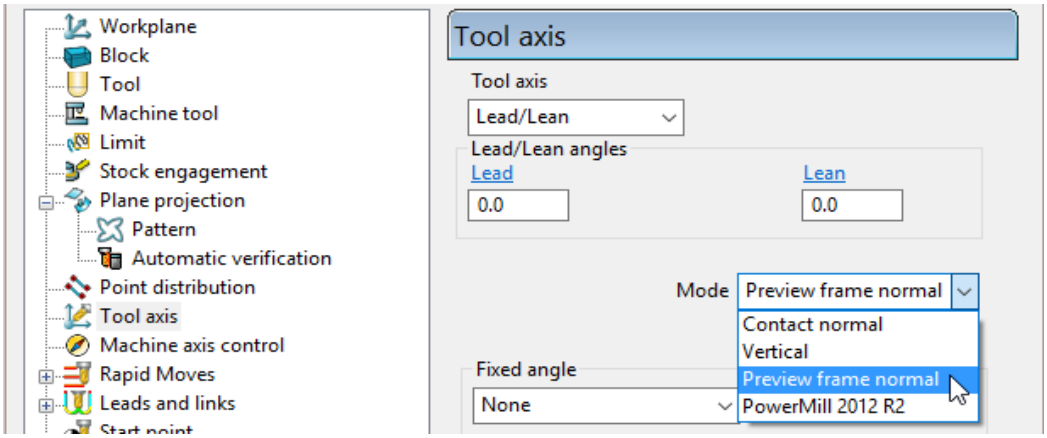


- Set **Azimuth** to **270**.
- Set **Elevation** to **50**.



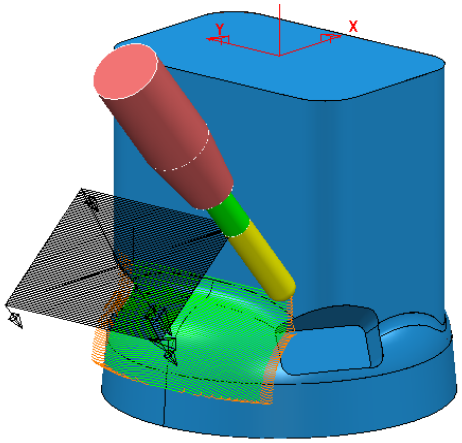
41 Set **Pattern** direction to **U**.

Copyright © Autodesk



42 **Calculate** the **toolpath** and then **Close** the dialog.

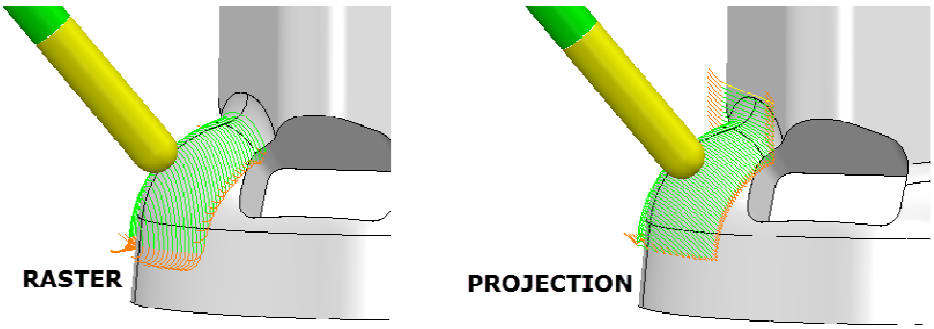
43 **Simulate** the **toolpath** and observe the associated tool alignment.



*The resultant toolpath starts at the lower corner and progresses towards the centre with a **Lead in** and **Lean out** both set to **0** creating tool alignment relative to the projection direction. Due to **Lead** and **Lean** being **0** a directly calculated **Two way** strategy is a feasible option.*

44 Select a view along the **Y Axis** and then **Simulate** both toolpaths in turn to visually, compare both strategies.

*The tool alignment is the same for both toolpaths due to a **Lean** value of **40** being applied to the **Raster** strategy.*

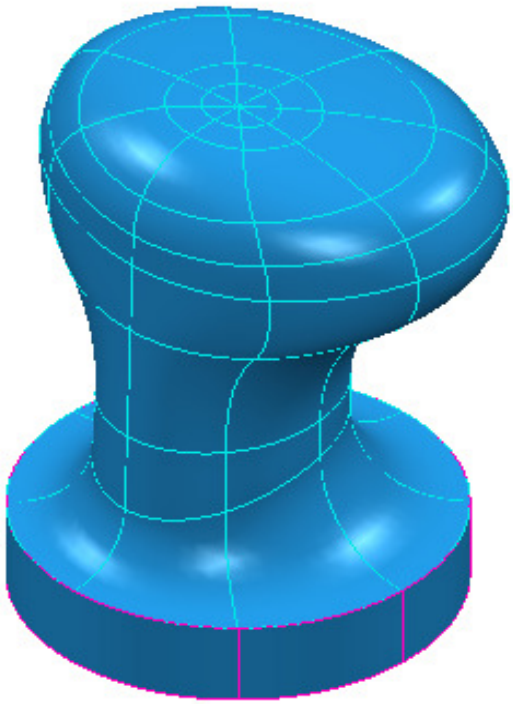


45 Select **File** - **Save Project**.

Lead/Lean - Example 3

1. Select **File** - **Delete all** and **Tools** - **Reset forms**.
2. **Import** the **model**:

   ...\PowerMill_Data\five_axis\joint_5axismc\joint5axis.dgk



3. **Save Project As**:-

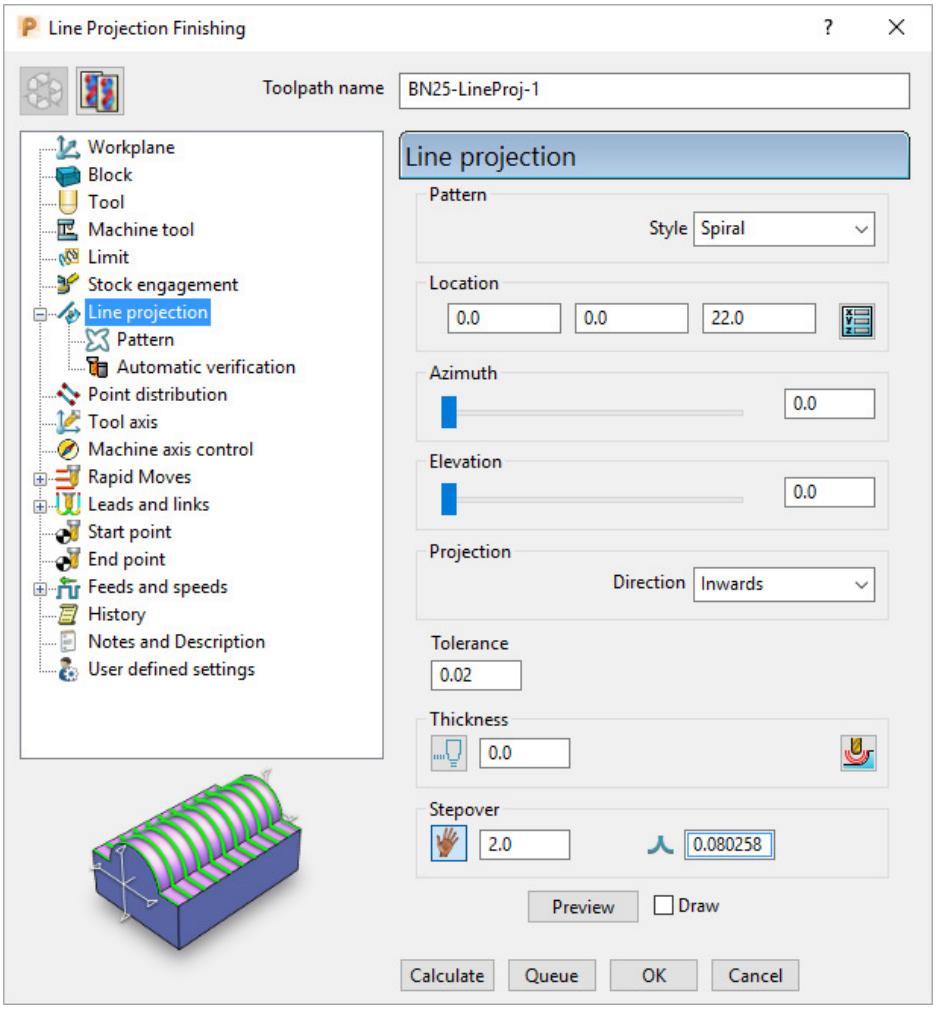
   ...\COURSEWORK\PowerMill_Projects\Joint-EX1

4. Create the material **Block** using ***Defined by*** - **Box** to the **Model** dimensions and then ***expand*** it by **15** in **X** and **Y** only.
5. Create a **Dia 25 Ball Nosed** cutter (**BN25**).
6. Select **Calculate** in the **Rapid Move Heights** dialog.
7. Use the **Start and End Point** default settings.
8. Select **Leads and Links** and input the following settings:
   - In ***Moves and clearances***: ***Tick*** the box **Automatically extend** with ***Maximum length*** **50**, input ***Incremental plunge distance*** **10**, and ***Axial clearance*** **45**.
   - In **Start and end point**: ***Tick*** the box **Incremental plunge**.
   - For ***Links*** set both ***1st choice*** and ***2nd choice*** to **Skim**.

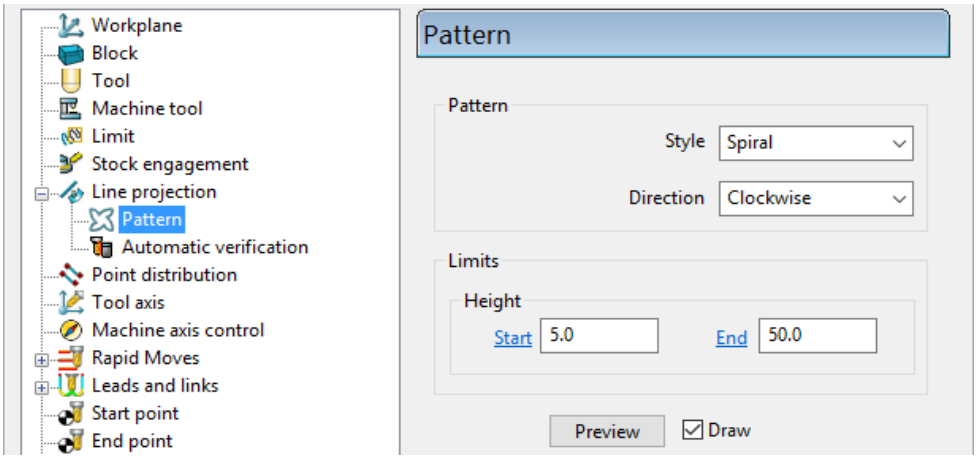
Copyright © Autodesk

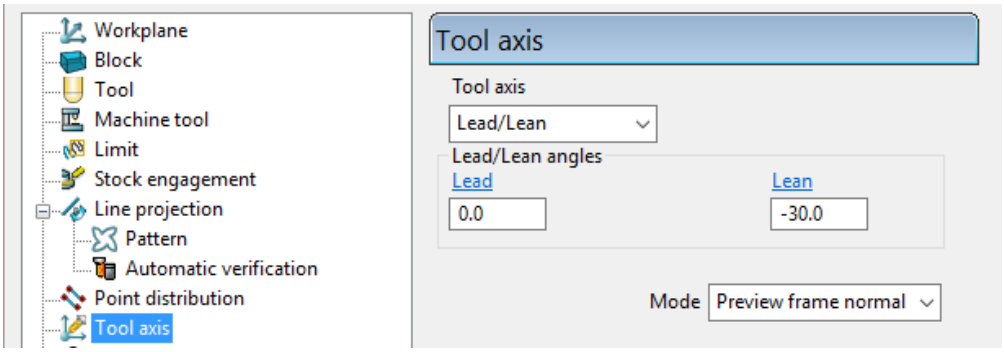
9 Select the **Toolpath Strategies** icon and in the **New Strategies** dialog, select the **Finishing** tab.

10 Select **Projection Line Finishing** and enter data exactly as shown in the following 3 pages of the dialog.

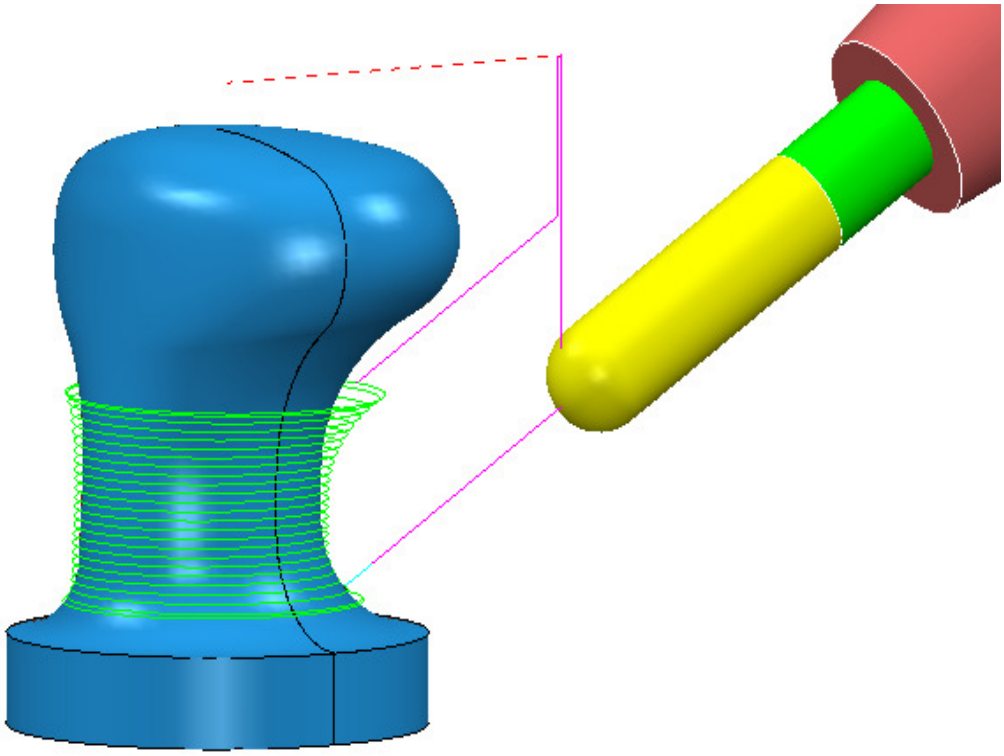


- Set **Azimuth** to **0**
- Set **Elevation** to **0**





11 **Calculate** the **Toolpath** and then **Cancel** to close the dialog.



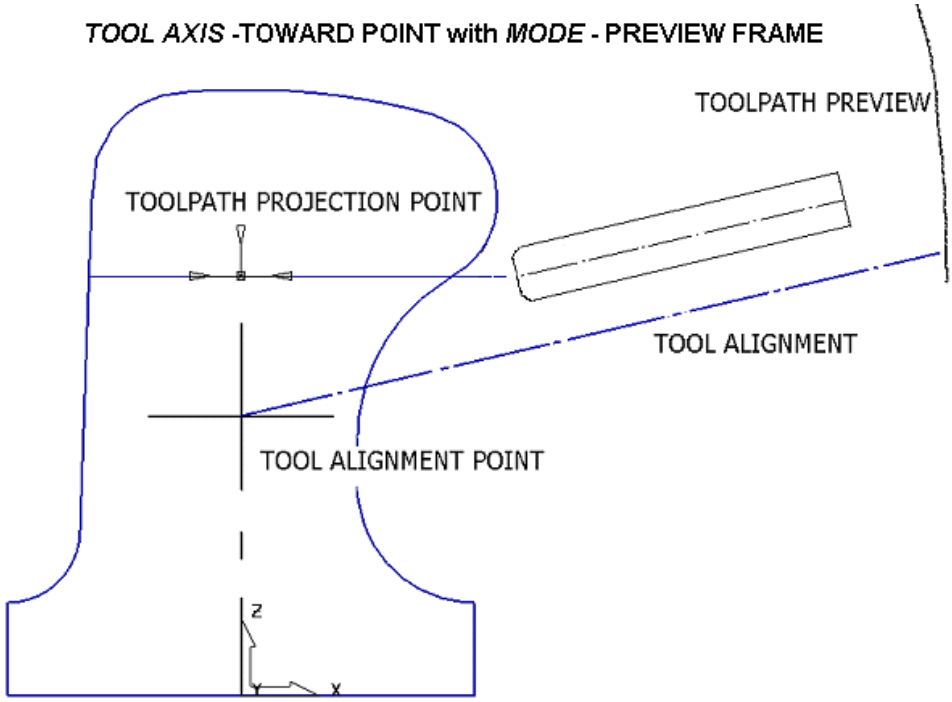
The next section continues with the machining of the upper part of this component.

12 **Save** the **Project** (but do not close it down).

Copyright © Autodesk

Toward \ From Point

These options allow the **Tool Axis** alignment to be based on a user-defined point during the generation of toolpaths (In this case **Towards Point**). If **Mode** - **Preview frame normal** is used then the actual alignment is taken relative to the **Preview** pattern for the strategy and not the final toolpath.



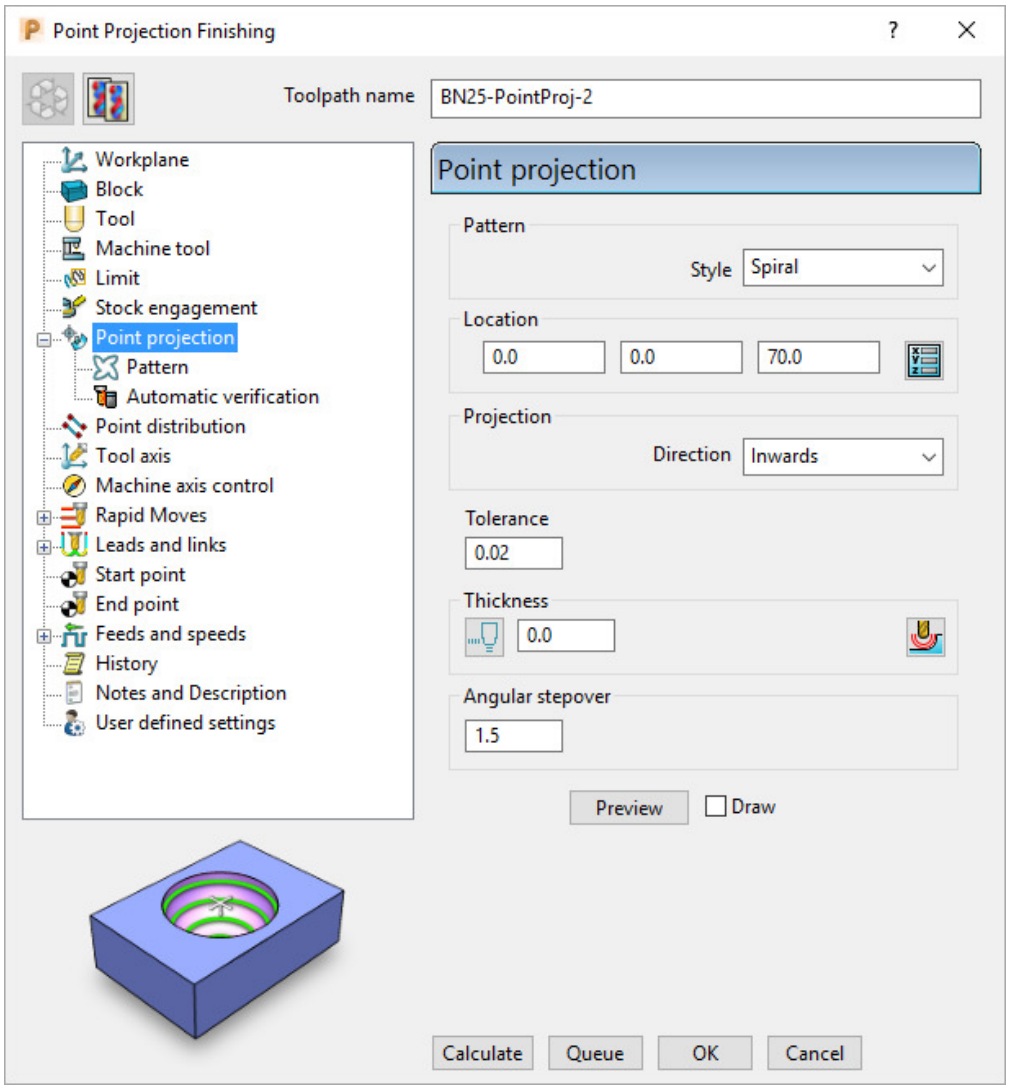
13 Set **Leads/Links** as follows:

- In ***Moves and clearances***: ***Tick*** the box **Automatically extend** with ***Maximum length*** **50**, input ***Incremental plunge distance*** **10**, and ***Axial clearance*** **45**.
- **Lead In/Out**: **Surface normal arc**, ***Angle*** **90** ***Radius*** **6**
- **Lead In/Out Extensions**: ***Extended Move***, **Distance 30**
- **Links**: **Skim**

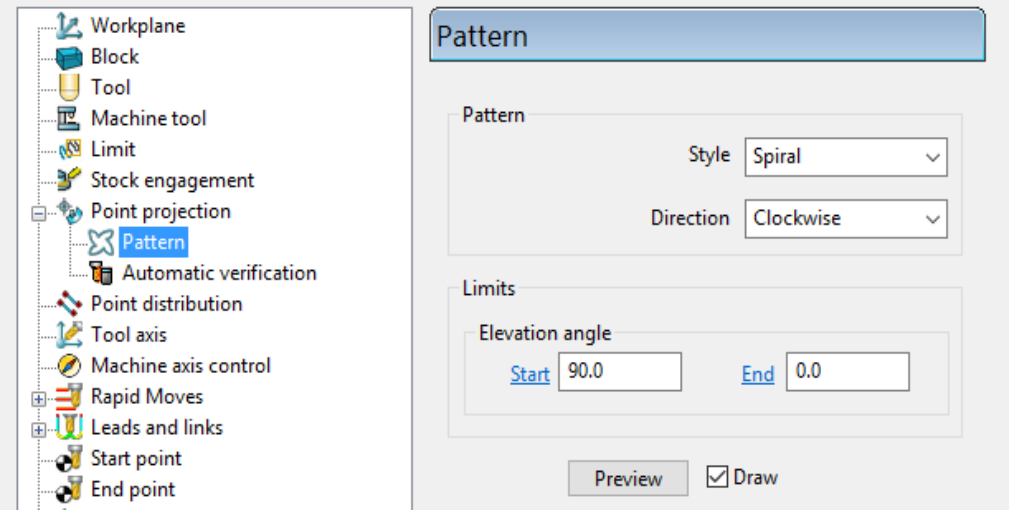
14 Select the **Toolpath Strategies** icon and select the **Finishing** tab located in the new **Strategy Selector** dialog.

15 Select the **Projection Point Finishing** option.

16 In the **Point Projection Finishing** dialog main page, enter data exactly as shown in the following illustration.



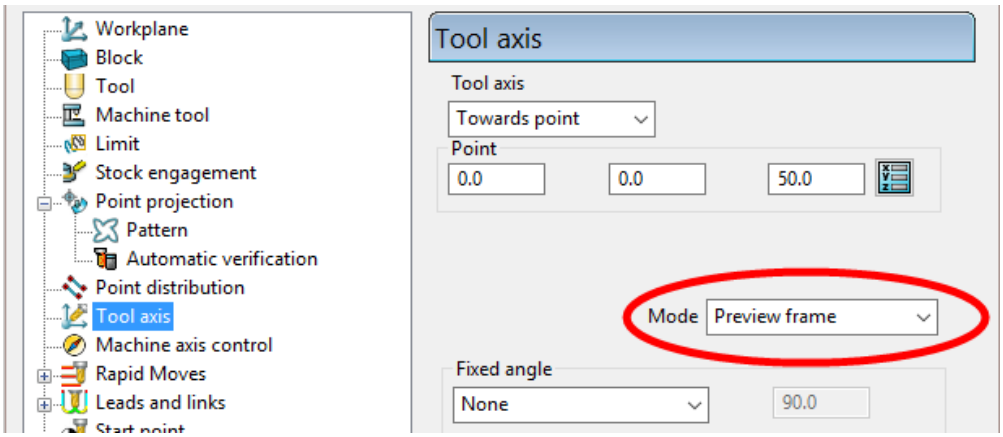
17 Select the **Pattern** page and enter data into the dialog exactly as shown in the following illustration.



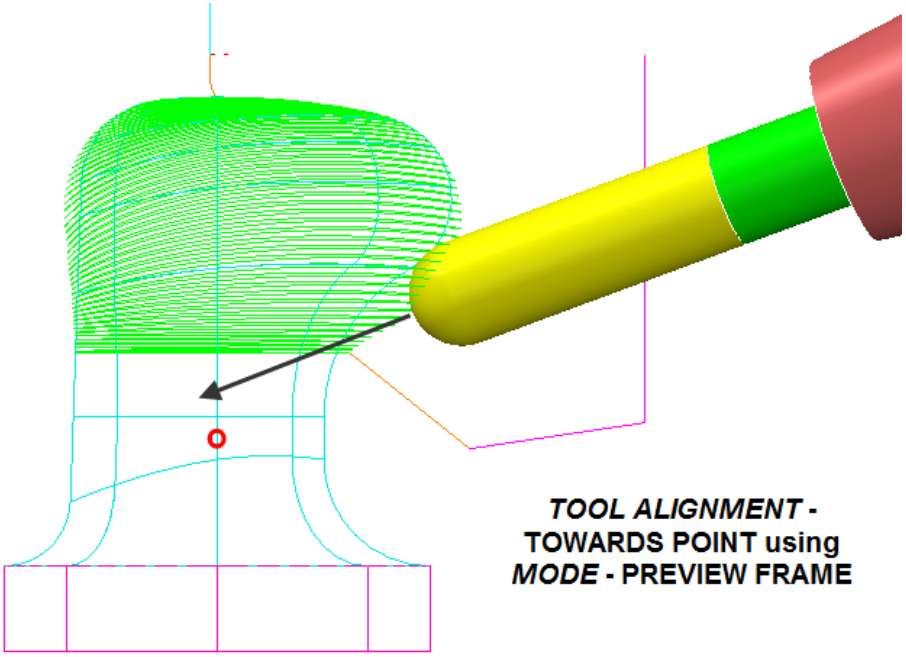
18 Input **Elevation Angle** data as:- **Start 90** and **End 0**.

Copyright © Autodesk

19 Select the **Tool Axis** page and enter data into the dialog exactly as shown in the following illustration.



20 **Calculate** the **Toolpath** but <u>do not</u> close the dialog.

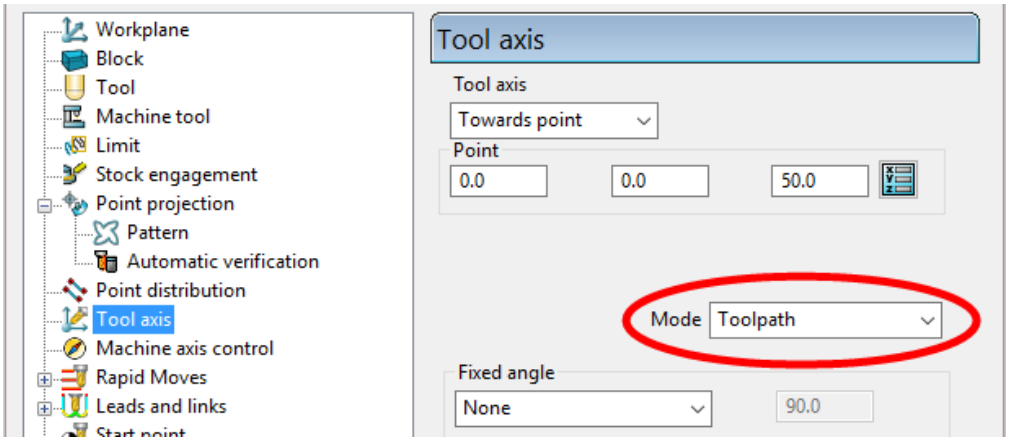


*The **Tool Alignment** point is approximately **20mm** below the **Projection Finishing** focal point to ensure that the spindle axis remains at an elevation angle to the machine tool table during machining hence avoiding collision.*

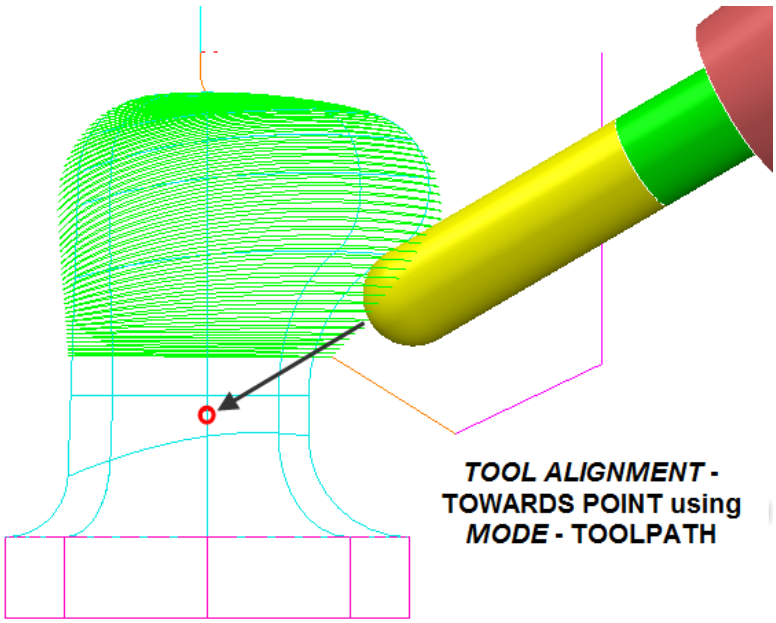
Mode – Preview Plane:- The actual **Tool** alignment occurs on the **Projection Point Finishing - Preview** pattern, after which the tool is then projected on to the model (towards the **strategy - focal point**).

21 In the **Point Projection** dialog select ***Re-cycle*** and then open the **Tool Axis** page.

22 Change the ***Tool axis*** **Mode** to **Toolpath**.



23 **Calculate** the **Toolpath** and then **Cancel** to close the dialog.



Mode – Toolpath:- The **Tool** alignment is applied directly on to the finished toolpath (towards the **strategy focal point**).

24 Select **File** - **Save Project**.

Copyright © Autodesk

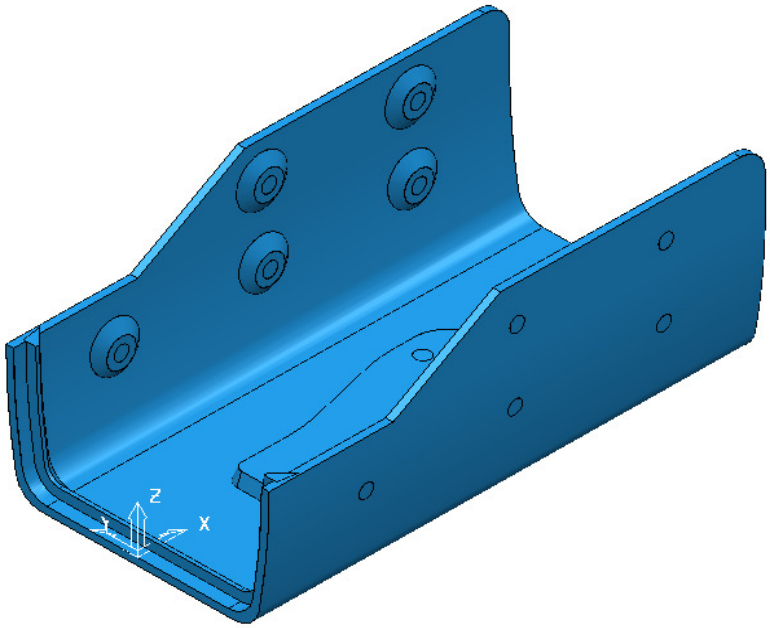
Toward\From Line

These options allow **tool axis alignment** to be ***towards*** or ***from*** a user-defined line, specified by a ***Vector*** direction through a suitably positioned **XYZ** coordinate.

1 Select **File** - **Delete all** and **Tools** - **Reset forms**.

2 **Import** the **Model**:-

...\PowerMill_Data\five_axis\Casing\from-line-model.dgk



3 **Save Project As**:

...\COURSEWORK\PowerMill_Projects\CameraCasing

4 Create a **Dia 12 Ball Nose** cutter named **BN12** of **Length 55** with:-

- **Shank** of ***Dia*** **12** - ***Length*** **40**.
- **1st Holder** - ***Lower Dia*** **25** - ***Upper Dia*** **40** - ***Length*** **40**.
- **2nd Holder** - ***Upper/Lower Dia*** **40** - ***Length*** **60** - ***Overhang*** **90**.

5 Define the **Block** (use **Box**) to the **Model** limits.

6 Select **Calculate** in the **Rapid Move Heights** dialog.

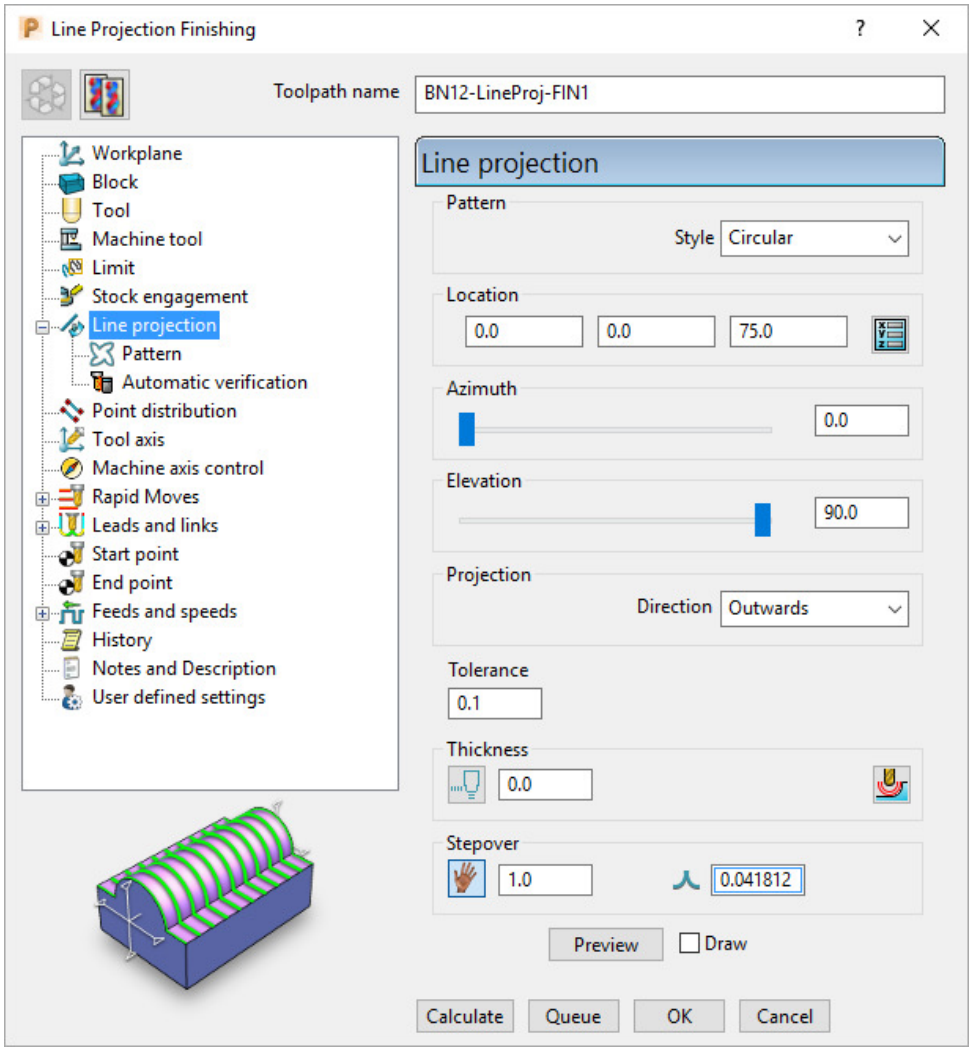
7 Set **Leads\Links** as follows:

- **Z heights**: ***Skim*** **5**, ***Plunge*** **5**
- **Lead In/Out**: ***1st Choice*** – **Surface normal arc**, ***Angle*** **45**, ***Radius*** **5.0**
- **Links**: ***1st choice*** – **Circular Arc**, Tick ***Apply constraints*** with **Distance < 3**, and ***2nd choice*** – **Skim**.

8 Select the **Toolpath Strategies** icon and in the new **Strategy Selector** dialog, select the **Finishing** tab.

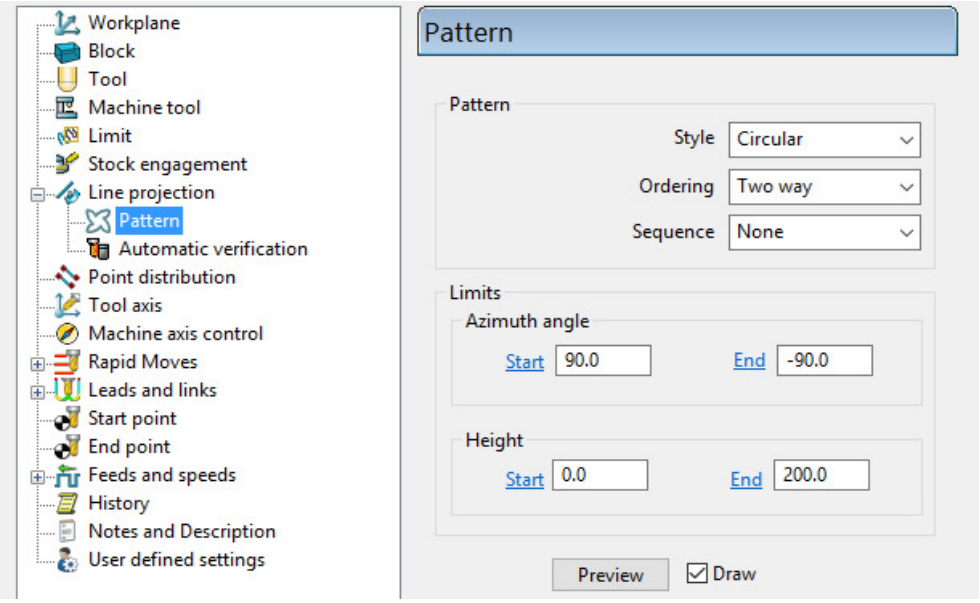
9 Select the **Projection Line Finishing** option.

10 In the **Line Projection Finishing, Pattern**, and **Tool Axis** pages enter data exactly as shown in the following illustrations.



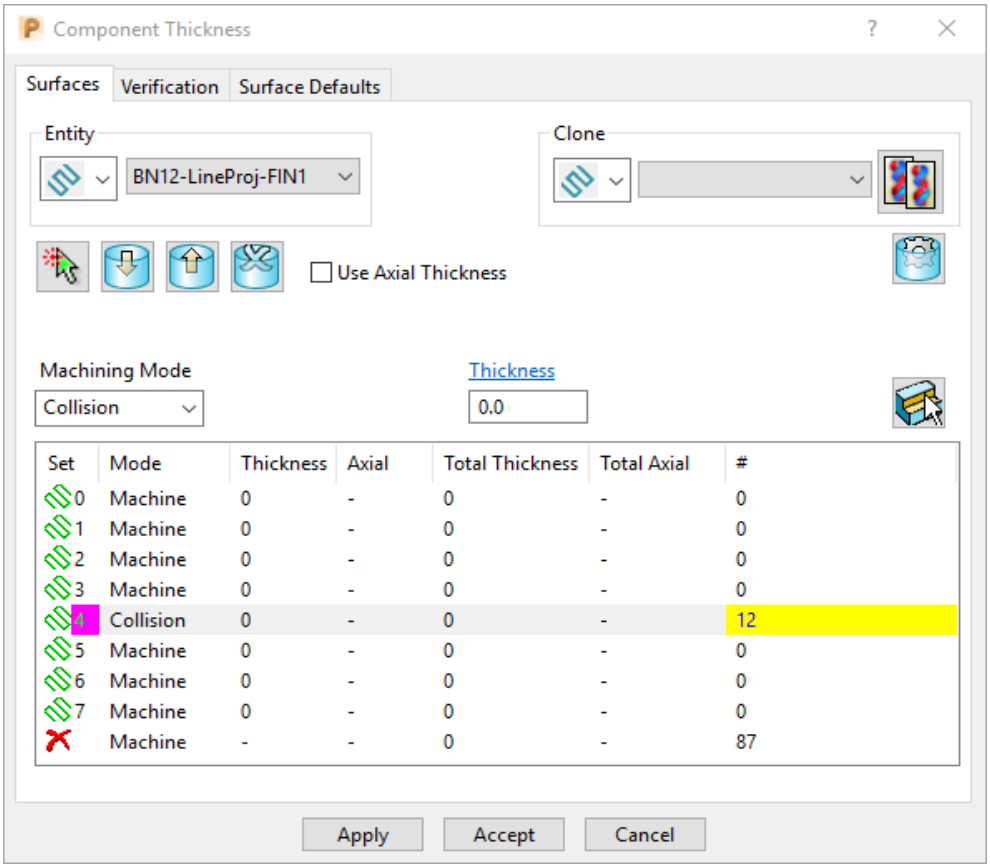
- Set ***Style*** to **Circular**
- Input ***Location*** as **0 0 75**
- Set **Azimuth** to **0**
- Set **Elevation** to **90**
- Set ***Direction*** **Outwards**
- Use ***Tolerance*** **0.01**
- Use ***Stepover*** **1**

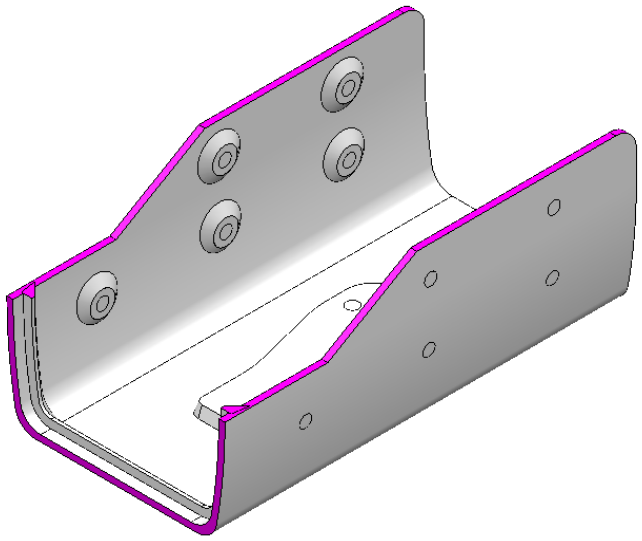
Copyright © Autodesk



11 Back in the main **Line projection** page, select **Component Thickness** .

12 Select any one of the 8 rows in the **Component Thickness** dialog.



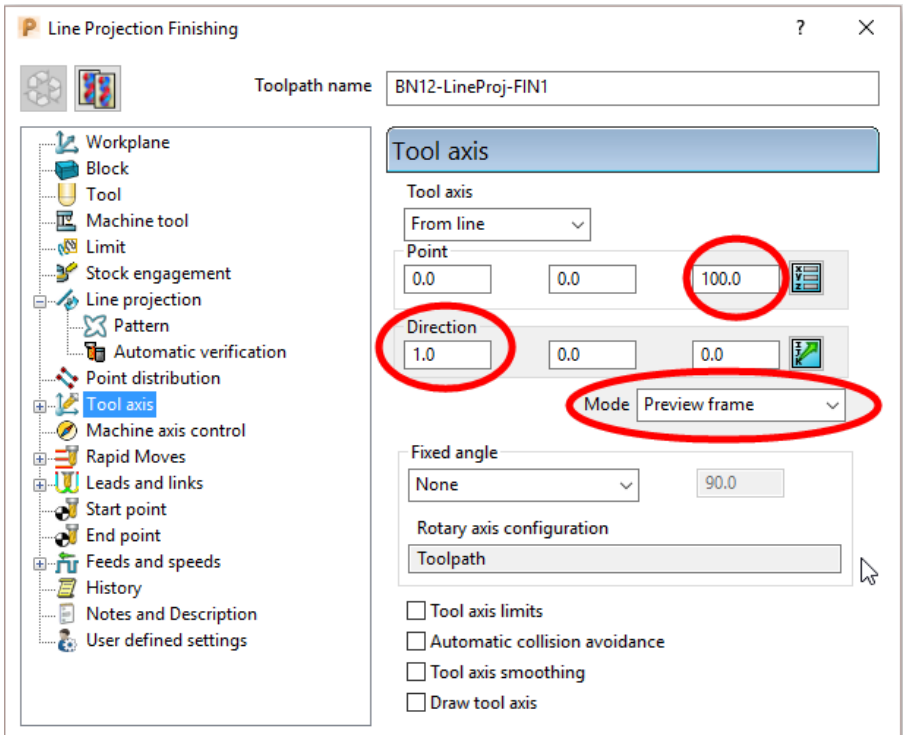


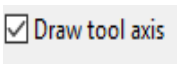
13 Select all of the edge **surfaces** on the **Model** and acquire them to the selected row in the **Component Thickness** dialog.

14 Set the **Machining Mode** to **Collision**.

15 **Apply** and **Accept** the **Component Thickness** dialog.

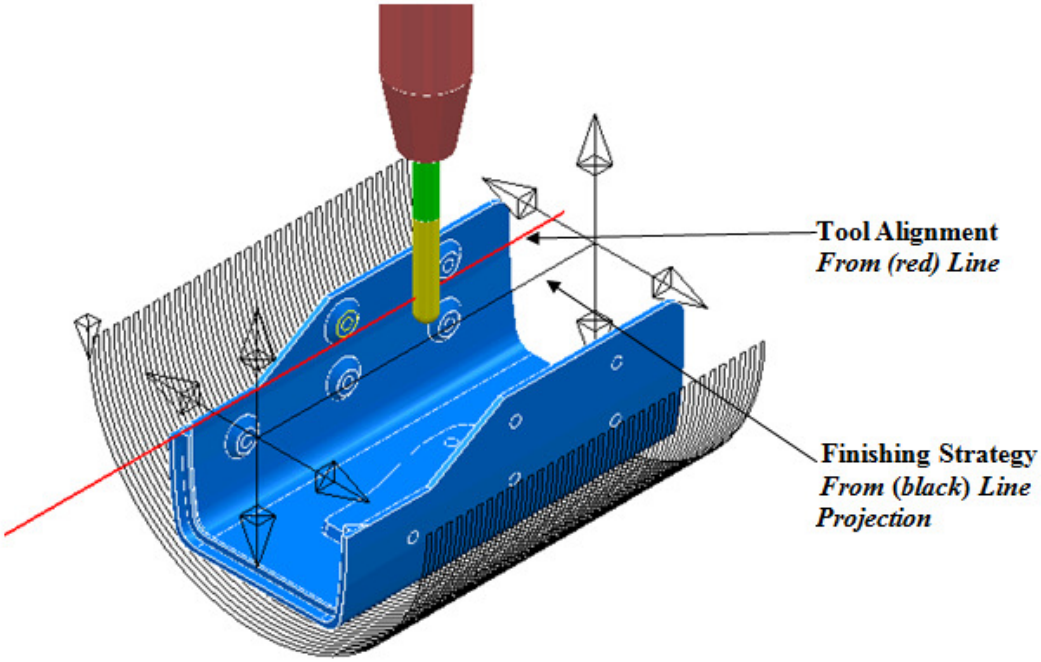
16 Back in the **Line Projection** dialog select the **Tool Axis** page and enter data exactly as shown below.



17 Tick the **Draw Tool Axis** box 

to display the red, **Tool Alignment** line relative to the **Model** (as shown in the following image).

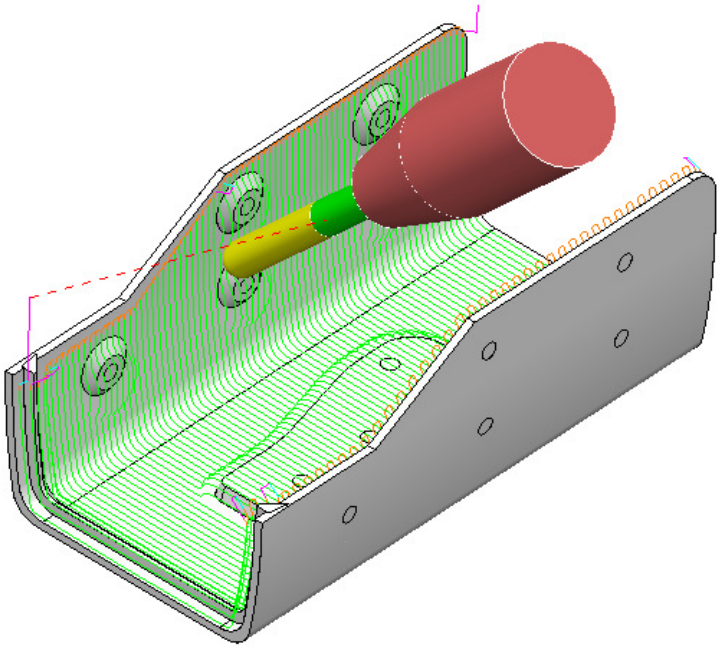
Copyright © Autodesk



18 Click the **Preview** tab to view the strategy before selecting **Calculate** to create the **Toolpath**.

19 Close the **Line Projection Finishing** dialog.

The end result (with a larger stepover) is as shown in the following illustration:



The use of **Tool Axis – From line** with **Mode – Preview frame** has ensured that the whole of the internal form, including the undercut areas, is fully accessible. Note that in this case it gives a better result than if **Mode – Toolpath** had been used.

20 **Save** the **Project**.

Toward\From Curve

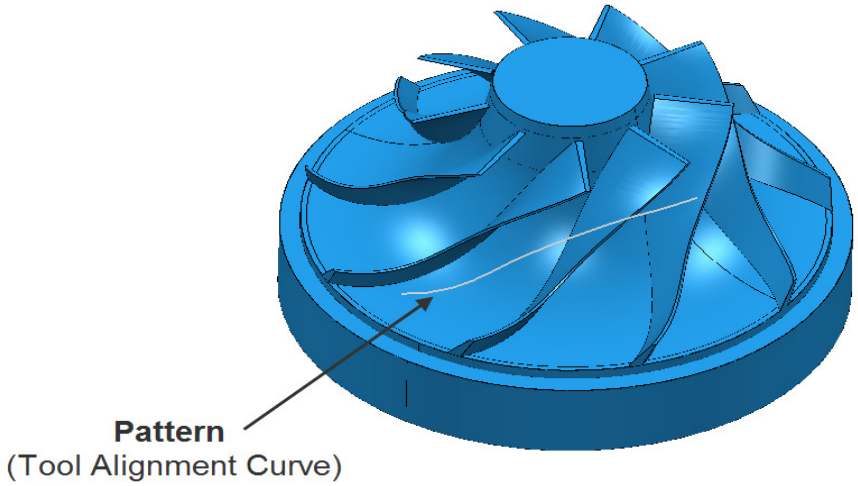
These options allow the ***tool axis alignment*** to be through a user-defined curve (**Pattern**).

The following example will use the **Surface Finishing** strategy. This is possible due to each of the two different blade areas to be machined being ***smooth***, ***single surfaces*** with the edges aligned to the ***surface curves*** (The resultant tool tracks will be aligned to these curves).

1. Select **File** - **Delete all** and **Tools** - **Reset forms**.
2. **Import** the **Model**:

   ....\PowerMill_Data\five_axis\Impeller\ impeller+Curve.dgk
3. **Save Project as**:

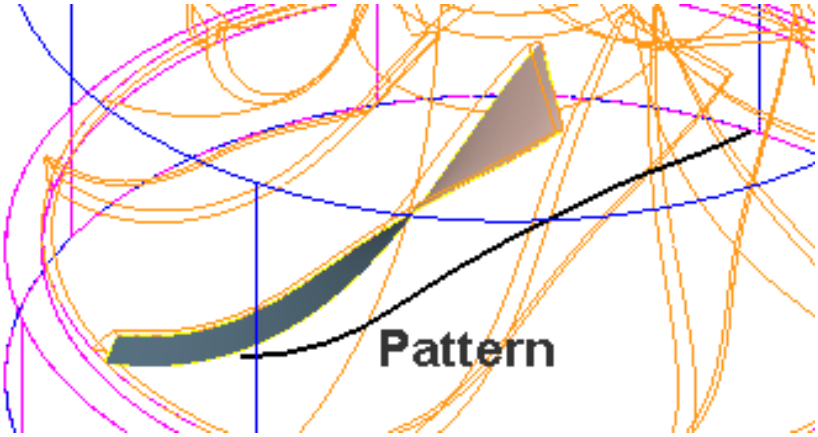
   ....\COURSEWORK\PowerMill_Projects\ Impeller+Curve-EX1
4. Create an empty **Pattern** and rename it as **Align2Curve**.
5. In the graphics area, select the **wireframe**, ***alignment curve*** that has been imported with the **Model** (Ensure that both the **Plain Shade** and **Wireframe** views are switched on!).
6. Right mouse click on the **Pattern** (***Align2Curve***) and from the local menu, select **Insert** > **Model** to copy the selected ***alignment curve*** as a **Pattern** segment.



7. Create a **Dia 3 Ball Nosed** cutter (named **BN3-LR**) of **Length 35** with the following **Shank** and **Holder** dimensions:
   - **Shank** - ***Upper\Lower*** **Dia 3** - ***Length*** **25**
   - **Holder 1** - ***Upper*** **Dia 15** - ***Lower*** **Dia 10** - ***Length*** **50**
   - **Holder 2** - ***Upper\Lower*** **Dia 15** - ***Length*** **35**
   - **Overhang 50**
8. Create a **Block** defined by **Cylinder** to the **Model** dimensions.
9. Set **Links** as **1st choice/2nd choice** to **Skim** and ***Default*** to **Incremental**.
10. In the **Rapid Move Heights** dialog select **Calculate**.

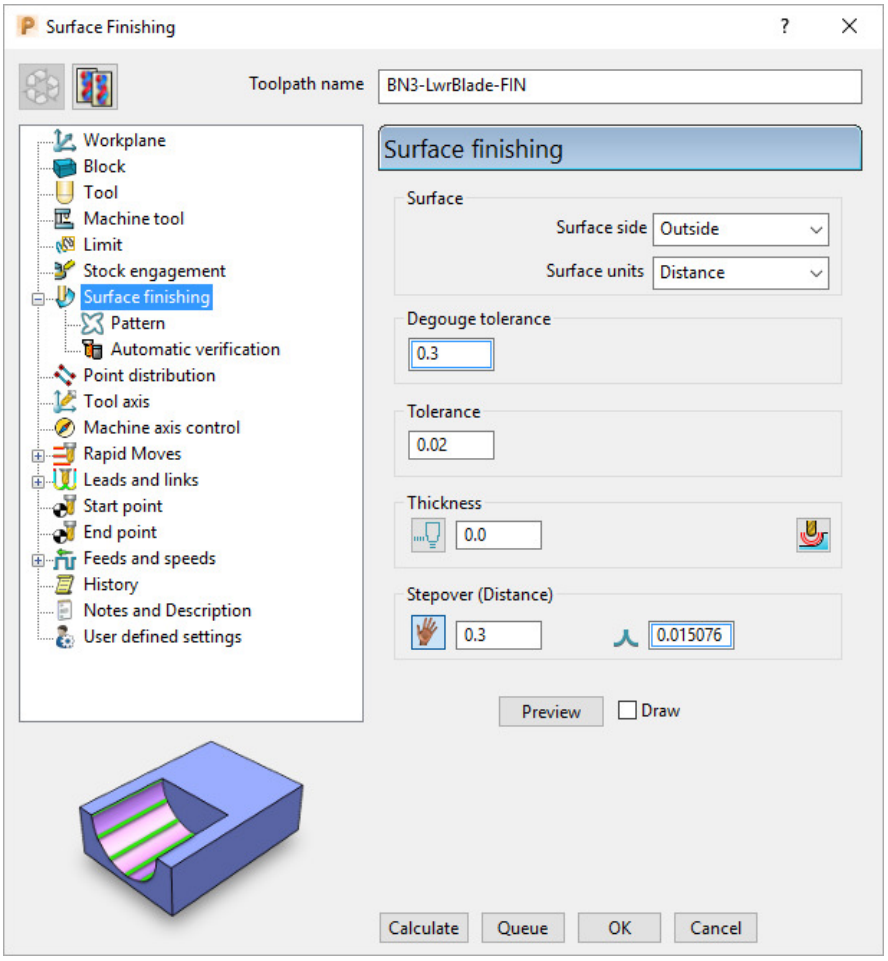
Copyright © Autodesk

11 Select the underside face of the blade (nearest to the **Pattern**) to be used in the first **Surface Finishing** strategy.

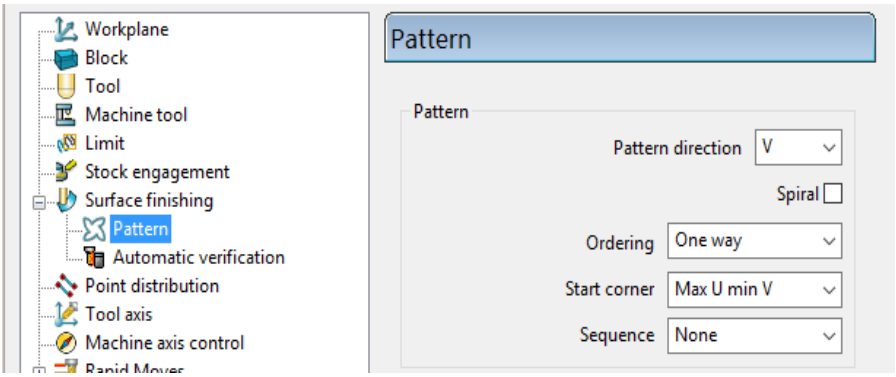


12 Select the **Toolpath Strategies** icon and in the **Strategy Selector** dialog select the **Finishing** tab.

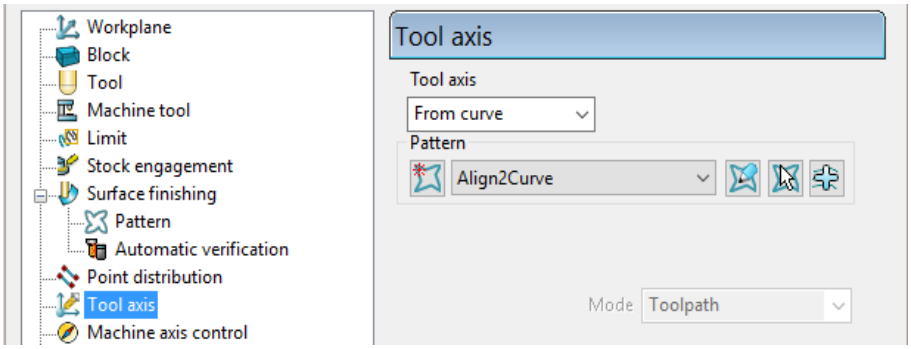
13 Select **Surface Finishing** and fill in the pages of the dialog exactly as shown in the following illustration.



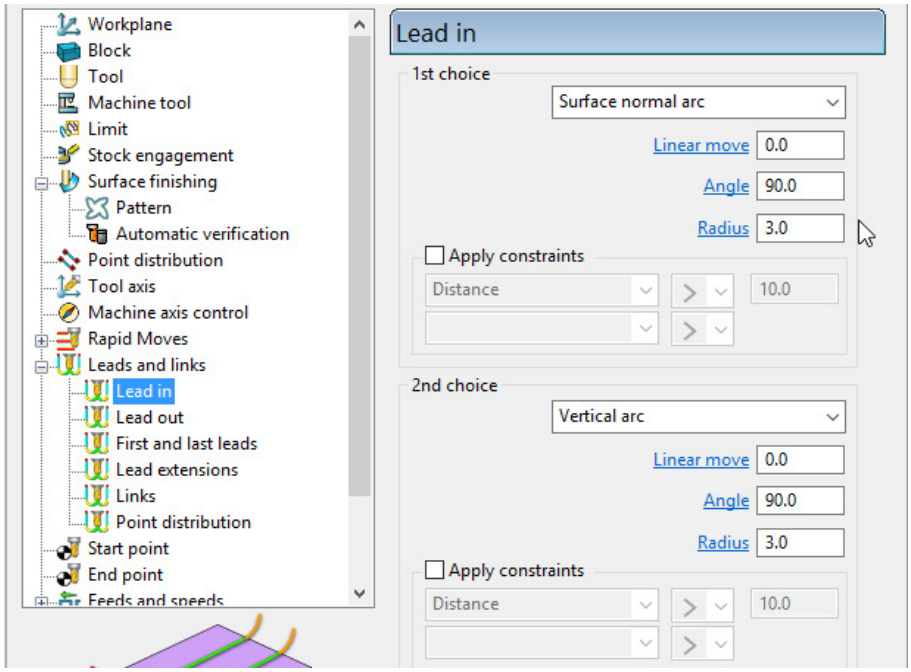
- ***Surface side*** - **Outside**



- ***Pattern direction*** - **V**
- ***Ordering*** - **One way**
- ***Start corner*** - **Max U min V**



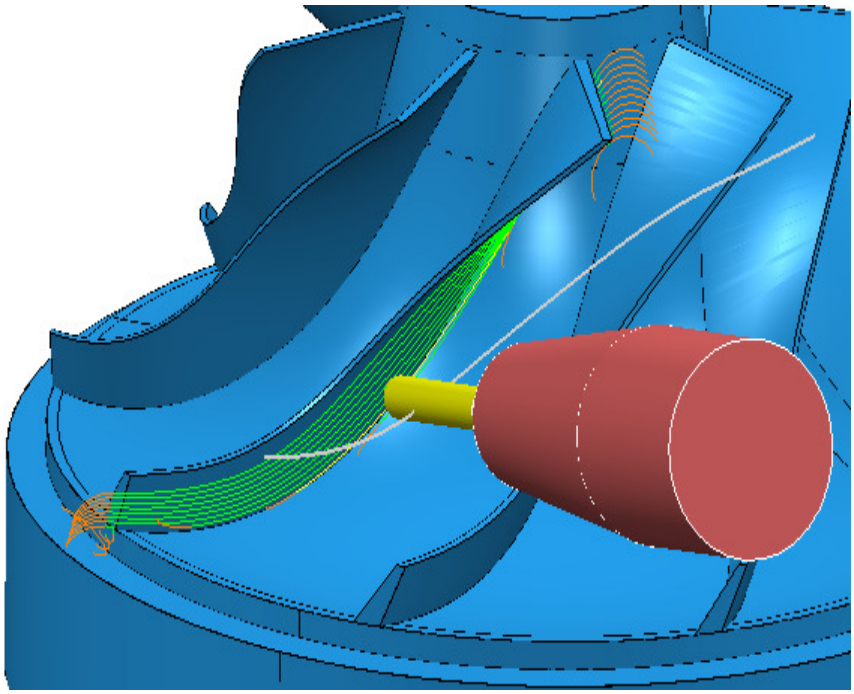
14 For both **Lead in** and **Lead out** use the following for ***1st*** and ***2nd*** **choice**.



15 Click the **Preview** tab and view the strategy before selecting **Calculate** to create the **Toolpath**.

16 Close the **Surface Finishing** dialog.

Copyright © Autodesk

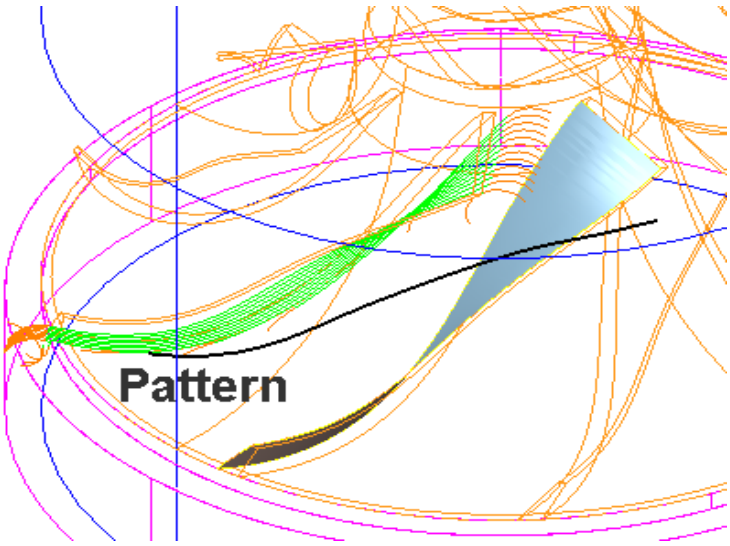


The **Tool Axis** is always aligned through the specified **Pattern** (Curve) while performing the **Surface Finishing** strategy.

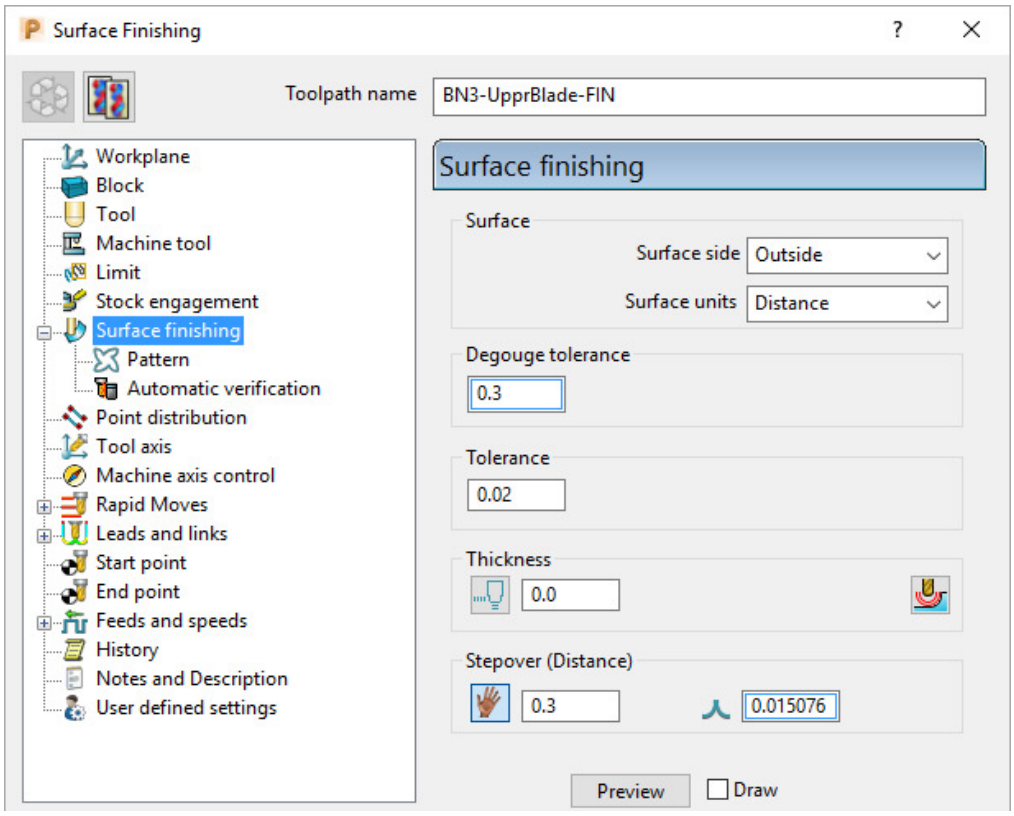
17 Right-click on the existing **Surface Finishing** strategy and from the local menu, select **Settings** to re-open the dialog.

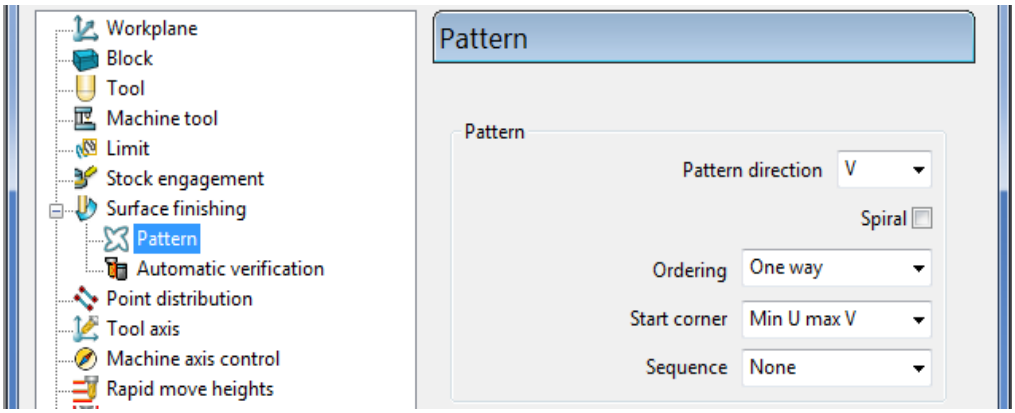
18 In the dialog, select **Make a copy of the Toolpath** .

19 Select the upper side, blade surface (nearest to the **Pattern)** to be used as the **Reference Surface** in the 2nd **Surface Finishing** strategy.



20 Open the **Surface Finishing** and **Tool Axis** dialogs, entering the data exactly as shown on the next page and select **Calculate**.



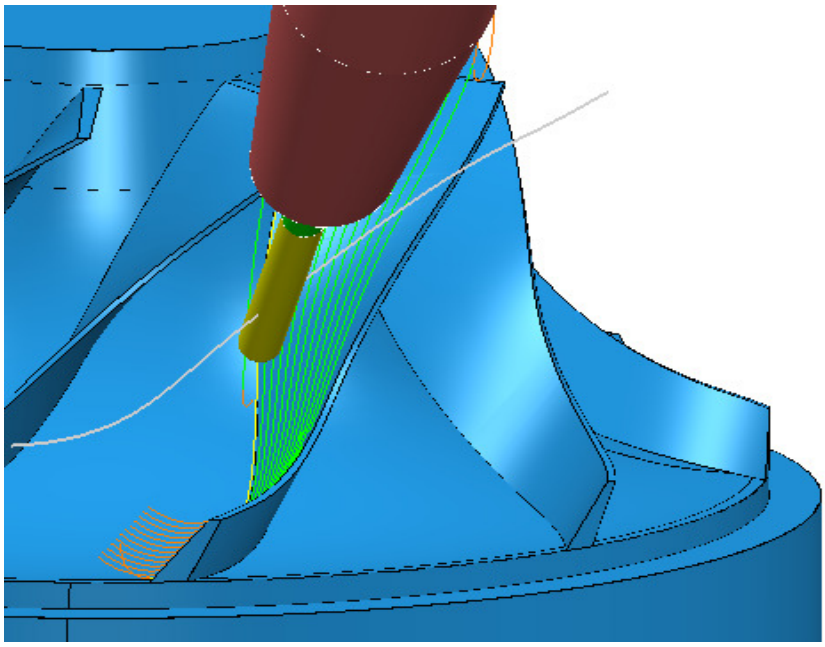


- ***Pattern direction*** **- V**
- ***Start corner*** - **Min U max V**

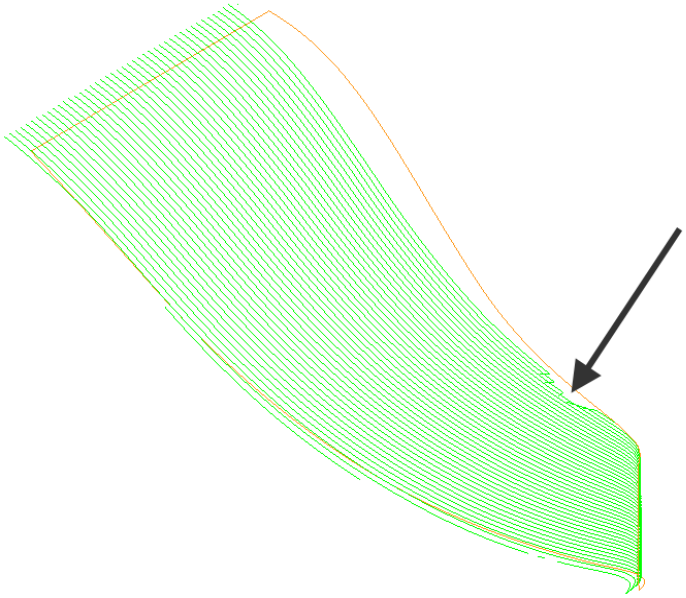
21 Click the **Preview** tab to view the strategy before selecting **Calculate** to create the **Toolpath**.

22 **Close** the **Surface Finishing** dialog.

Copyright © Autodesk



The ***Tool Axis*** is always aligned through the specified **Pattern** (Tool Alignment Curve) while performing the chosen machining strategy (**Surface Finishing**).



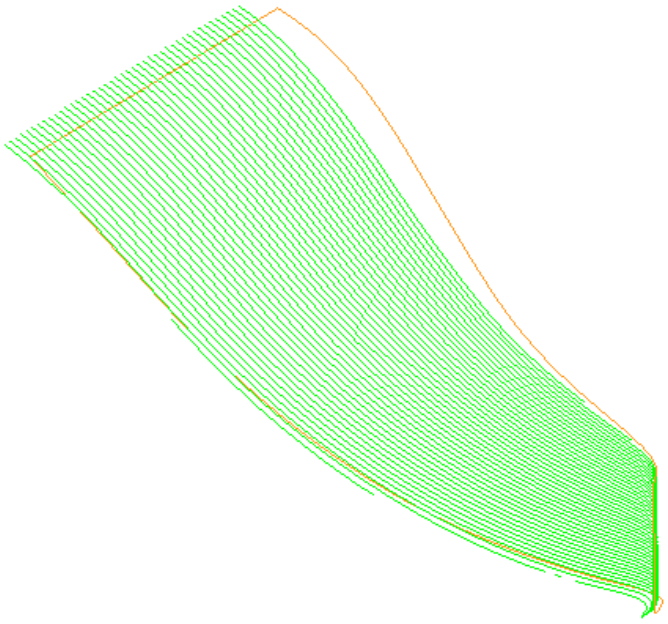
A small area on the latest toolpath is fragmenting along the top edge as arrowed above. This is caused by one of the other blades getting in the way of the projection process at this point. In this instance an easy fix can be applied by using a copy of the existing **Pattern** and moving it upwards by **1.5**.

23 Create a copy of **Pattern Align2Curve** which will automatically be named as **Align2Curve-1**.

24 Move the new **Pattern** upwards by **1.5mm**.

25 Recycle the toolpath **BN3-UprBlade-FIN** with the **Tool axis** – **from curve** options using **Pattern Align2Curve-1**.

Copyright © Autodesk



The toolpath fragmentation on the upper edge has now been fixed.

26 **Activate** and **Verify** each **toolpath** in turn, checking both the **Gouge** and **Collision** status (Use a **Shank Clearance** of **1** and a **Holder Clearance** of **2**).

*If all is safe the **Verify** process will result in a **Blue tick box** adjacent to each toolpath in the **PowerMill explorer**.*

27 **Save** the **Project**.

Copyright © Autodesk

Fixed Direction

This allows the **Tool Axis** to be set to a fixed angle, specified as a user defined, **Vector**. In this case it is applied to the finishing of an angled recess on the supplied **Model**.

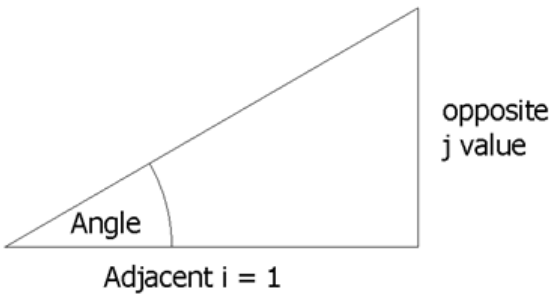
Angle to Vector conversion table

Fixed Direction *Tool Alignment* – Vectors\Angular equivalents

The table represents angular directions and vectors on the XY plane (Z=0).

ANGLE		**VECTORS**	
(Degrees)	(I	J	K)
0	1	0.0000	0
5	1	0.0875	0
10	1	0.1760	0
15	1	0.2680	0
20	1	0.3640	0
25	1	0.4660	0
30	1	0.5770	0
35	1	0.7000	0
40	1	0.8390	0
45	1	1.0000	0
50	1	1.1920	0
55	1	1.4280	0
60	1	1.7320	0
65	1	2.1450	0
70	1	2.7470	0
75	1	3.7320	0
80	1	5.6710	0
85	1	11.4300	0
90	0	1.0000	0

$$\text{Tan (Angle)} = \frac{\text{(opposite)}}{1}$$

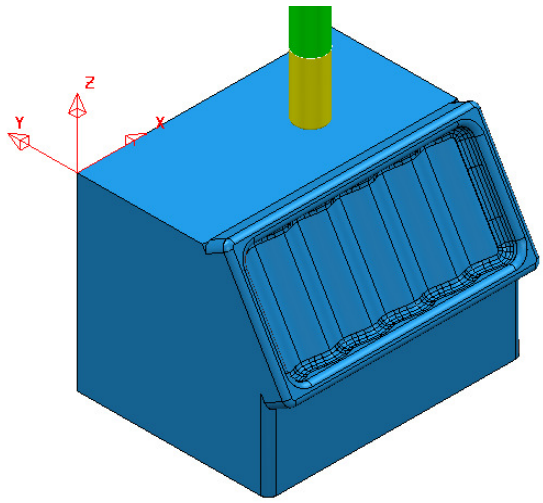


The ***Tool Alignment*** is set to a **Fixed Direction** relative to the currently active **Workplane** by inputting suitable values as an ***I J K vector*** along the **Tool Axis** (towards the spindle). Although the correct values for a ***vector*** may require the user to exercise their trigonometry skills, this method does provide full flexibility for defining compound angles.

1 From the **main** pull down menus, select **File** - **Delete All** and **Tools** - **Reset Forms**.

2 **Open t**he (read only) **Project**:

...\PowerMill_Data\five_axis\AngledFrame\ToolAxisFixed1-Start



3 **Save Project As**:

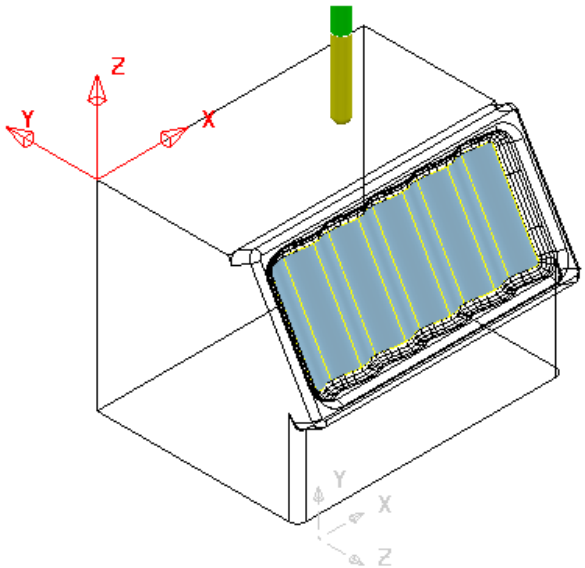
...\COURSEWORK\PowerMill_Projects\ToolAxisFixed-EX1

4 Activate and then *de-activate* the Toolpath **D12t1-ruf-TOP**.

This will restore the settings (**Workplane**, **Rapid Move Heights**, etc) ready to apply a finishing strategy on the corrugated angled form.

5 Activate the **BN6** tool.

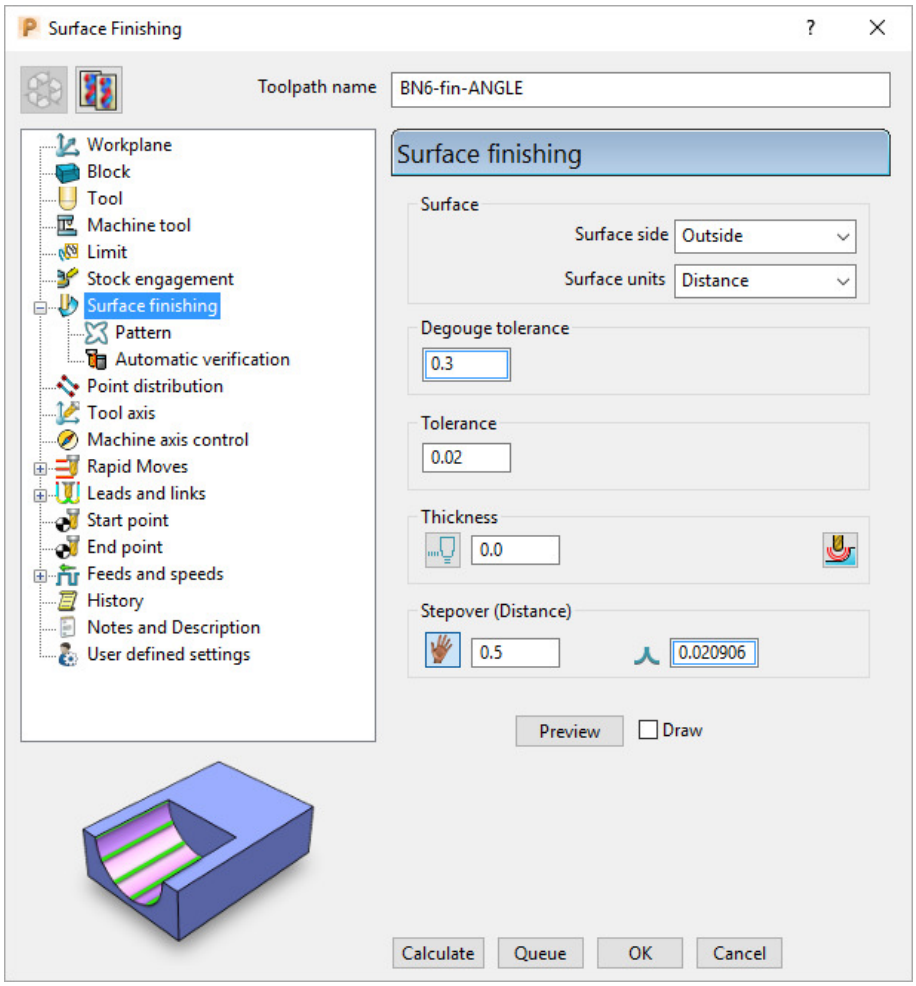
6 Select the undulating **Surface** on the base of the angled recess.



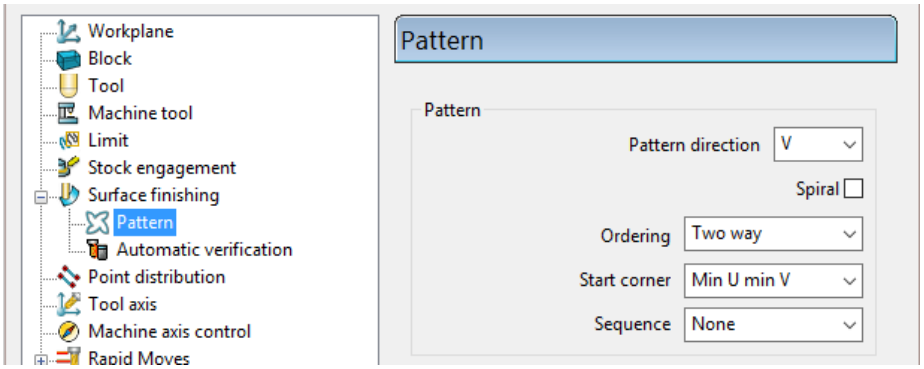
Copyright © Autodesk

7 Select the **Toolpath Strategies** icon and in the **Strategy Selector** dialog select the **Finishing** tab.

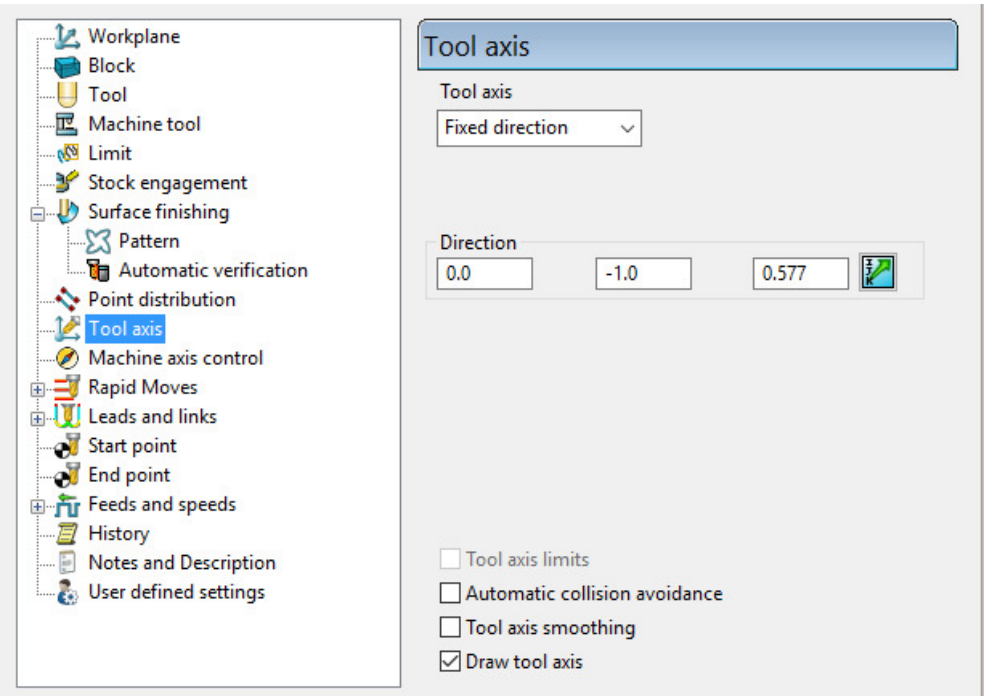
8 Select the **Surface Finishing** option and fill in the pages of the dialog exactly as shown in the following illustrations.



- ***Surface Side*** - **Outside**
- ***Stepover*** - **0.5**



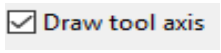
- ***Pattern direction*** - V
- ***Ordering*** - **Two way**

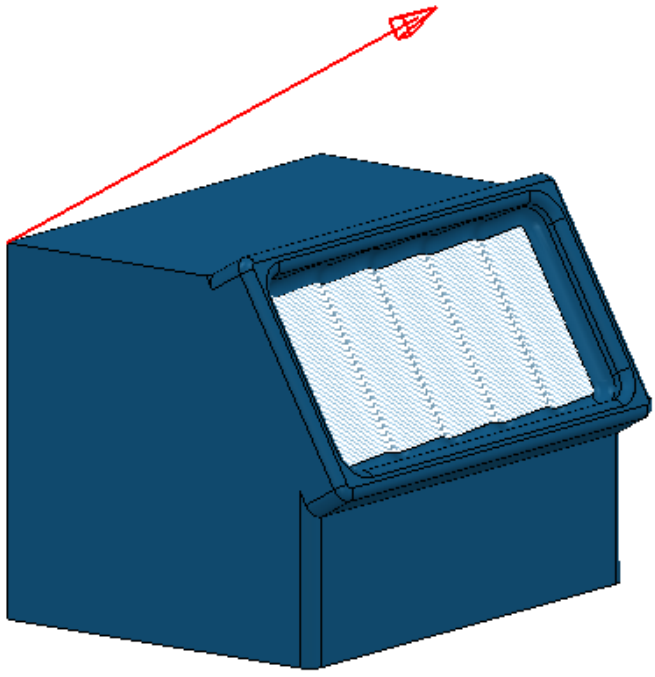


9 Set the **Tool Axis** to **Fixed direction**.

10 Input the vector ***Direction*** values: ***I*** **0** ***J*** **-1** ***K*** **0.577**

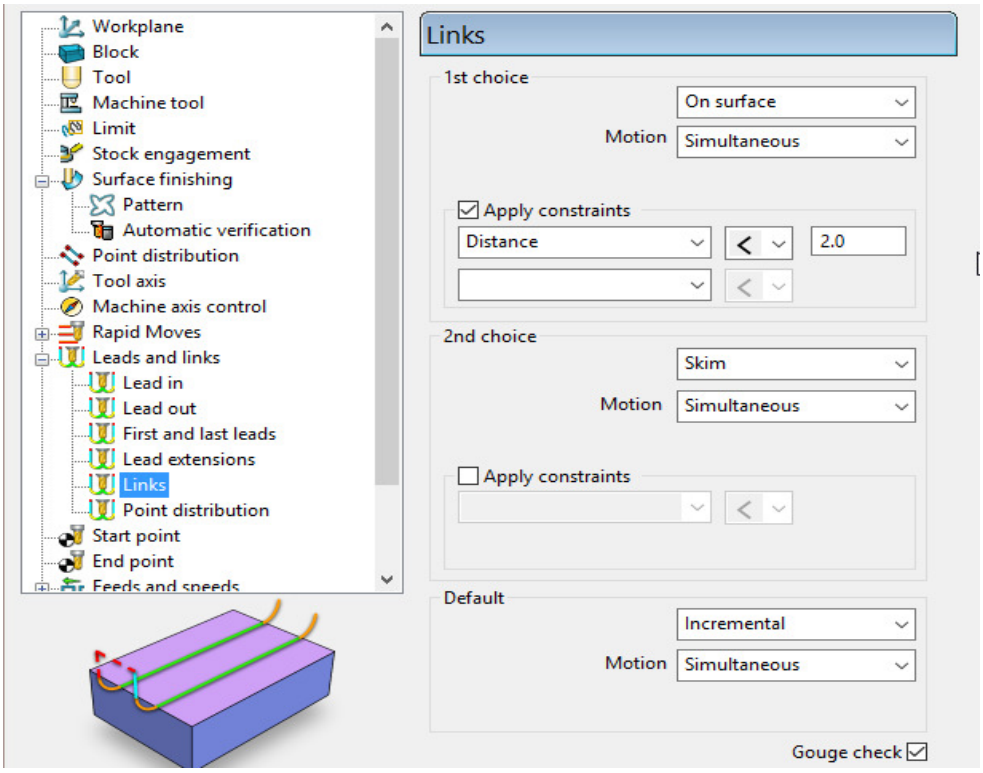
11 Tick the **Draw tool axis** box.





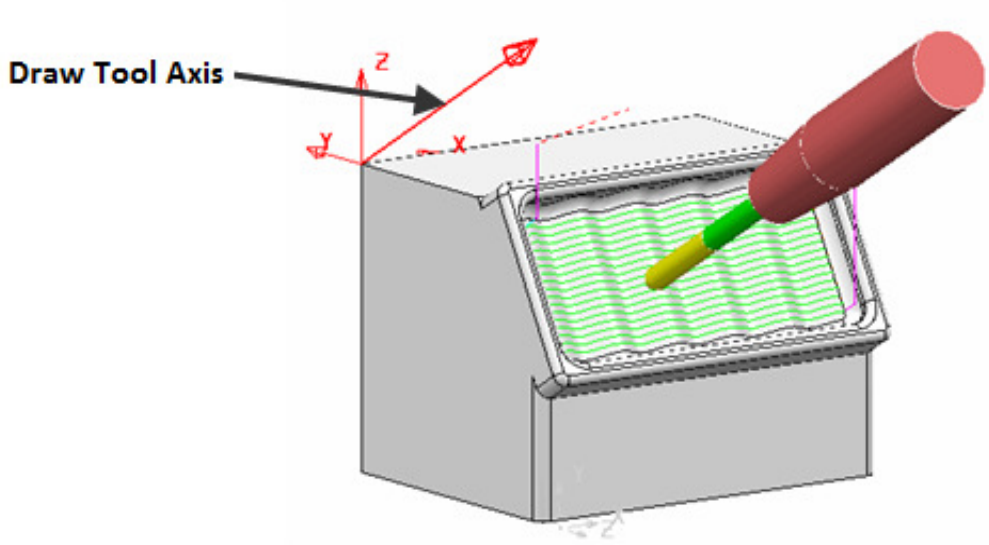
12 From **Leads and links** select the **Links** dialog and enter data exactly as shown in the following illustration.

Copyright © Autodesk



13 Click the **Preview** tab to view the strategy before selecting **Calculate** to create the **Toolpath**.

14 **Close** the **Surface Finishing** dialog.



For viewing purposes, the toolpath shown in the illustration has been created using a 2mm Stepover.

15 From the **main** pulldown menu, select **File - Save** to update the stored **Project**:

...\COURSEWORK\PowerMill_Projects\ToolAxisFixed-EX1

Copyright © Autodesk

4. Projection Surface Finishing

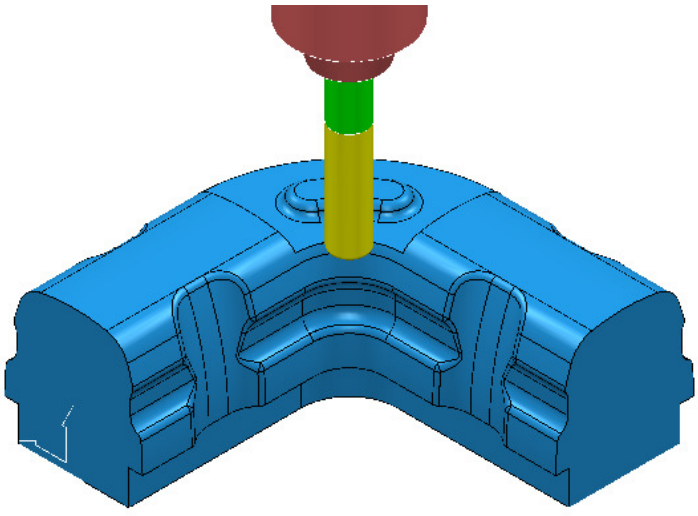
Introduction

In a **Projection Surface Finishing** the strategy is projected **Inwards** or **Outwards** along the normals of a single **Reference Surface**. For five axis applications an appropriate ***tool alignmen***t is used. The toolpath runs either across or along the natural **Reference Surface** directions (U or V) with the **Stepover** being defined by unit **Distance** or **Parametric** division between **Surface Curves**. In some cases the **Reference Surface** may form part or all, of the component to be machined.

To create a **Reference Surface** the user will require the services of a suitable **Surface Modeller**, ideally **PowerSHAPE**. For the following example the **Reference Surface** has already been created and stored as a separate ***dgk*** file to be imported as required. A **reference surface** should be as smooth and simple as possible. It should also be as close as possible to the actual ***surfaces*** that are to be machined.

1 **Open** the (Read Only) **Project**:-

...\PowerMill_Data\five_axis\CornerFixing\ProjectionSurfaceEX1-Start

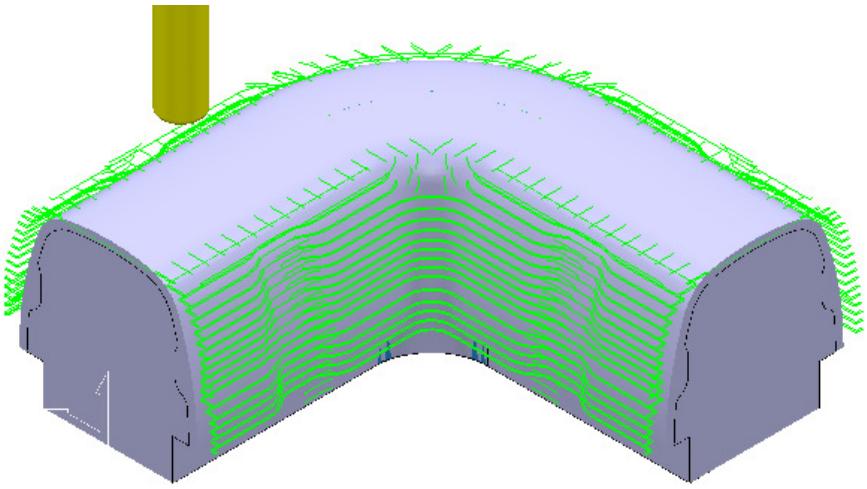


2 **Save Project As**:

....\COURSEWORK\PowerMill_Projects\ProjectionSurface-Example1

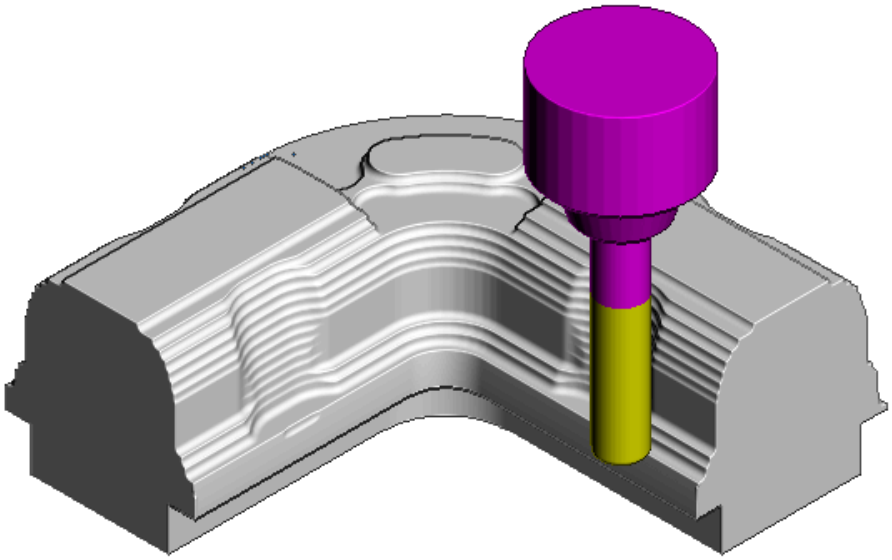
3 **Activate** the toolpath **D25TR3-RGH1** to reinstate the settings.

Copyright © Autodesk



*The **Block** used in the toolpath **D25TR3-RGH1** is defined from an imported **Triangle** model (**dmt** file) of a Casting.*

4 Run a **ViewMILL** simulation on the toolpath **D25TR3-RGH1**.



5 Save the **ViewMILL** model at this stage.

6 Return to the **PowerMill** window and temporarily **suspend ViewMILL**.

7 **Calculate** a new **Block** - ***Defined by*** **- Box** using ***Type*** – **Model**.

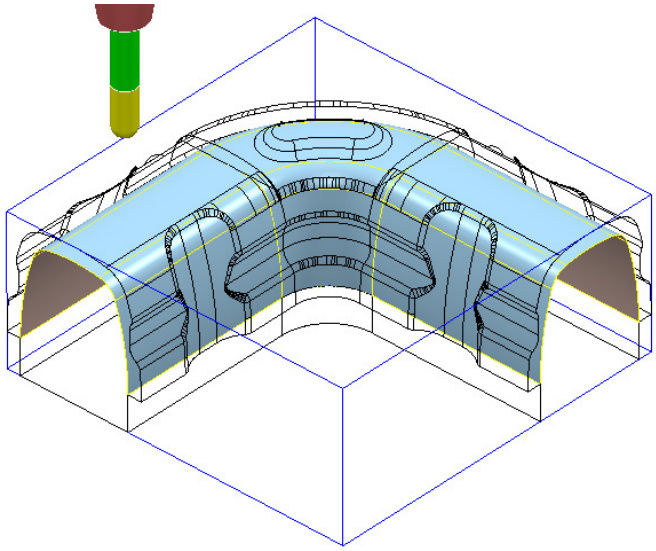
8 **Import** the **Model**:-

….\PowerMill_Data\five_axis\CornerFixing\CornerFixingRefSurf1.dgk

Copyright © Autodesk

9 In the **PowerMill *explorer***, activate the **BN16** tool.

10 In the **PowerMill *explorer*** right mouse click on the **model** named **CornerFixingRefSurf1** and from the local menu select **Select All**.

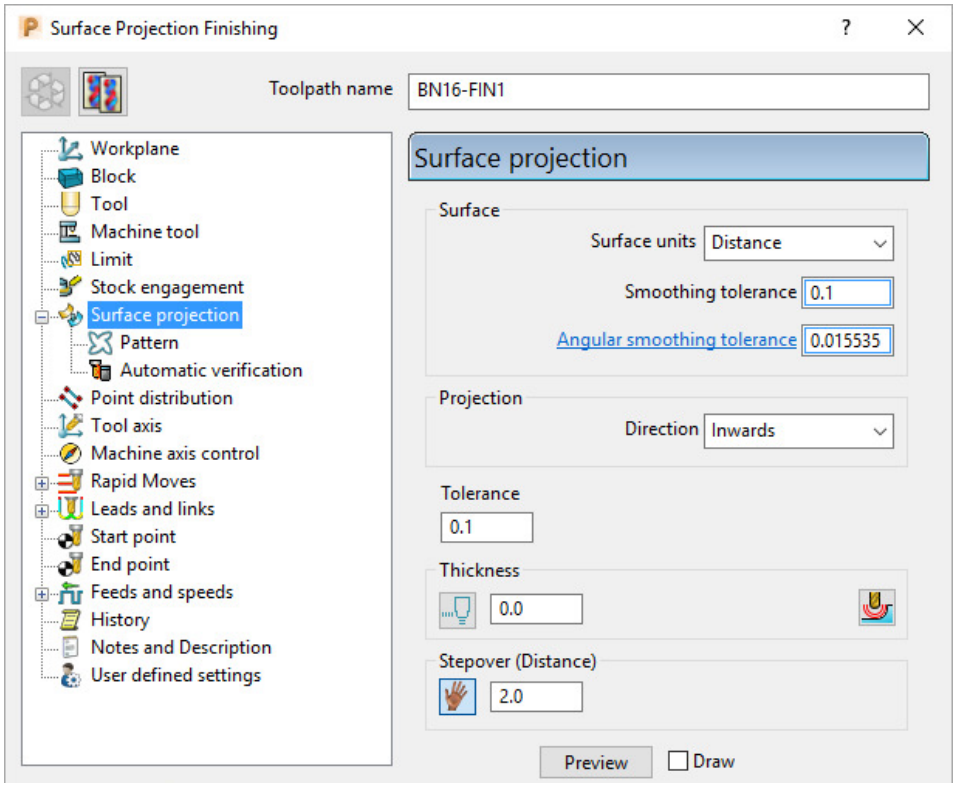


The newly ***imported*** (reference) **surface** (shown shaded above) is fully located inside the original component **model**.

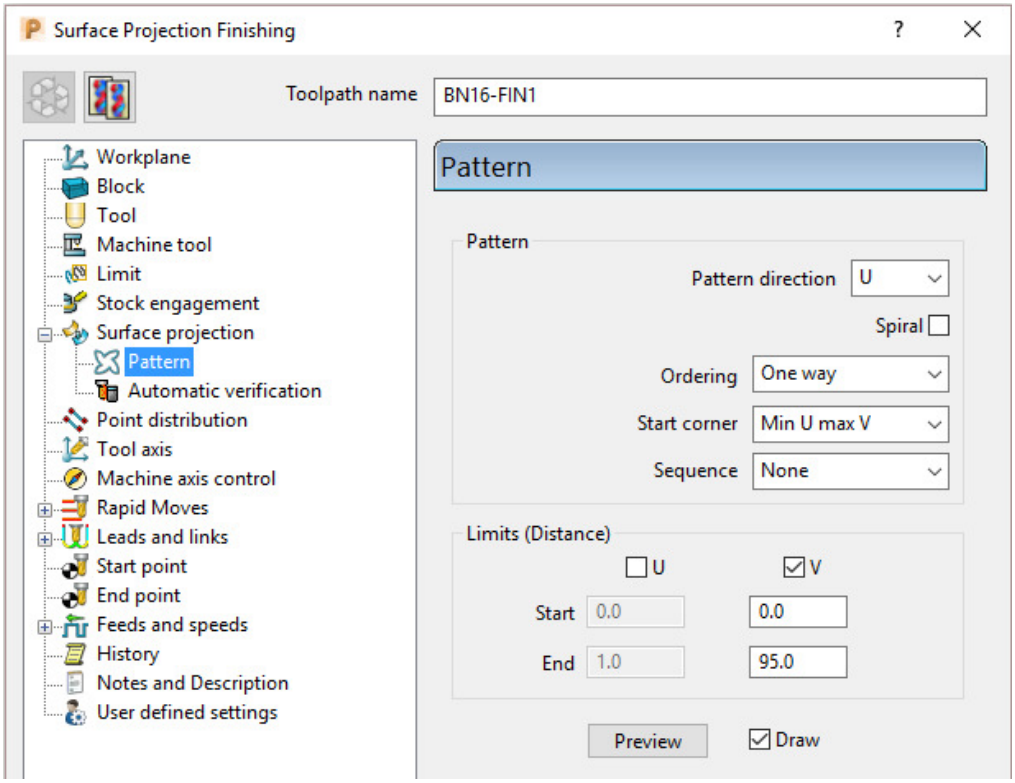
11 From the **main** toolbar select the **Toolpath Strategies** .

12 From the dialog select the **Finishing** tab followed by the **Projection Surface Finishing** option.

13 Enter the values into the ***main*** page of the **Surface Projection Finishing** exactly as shown below.

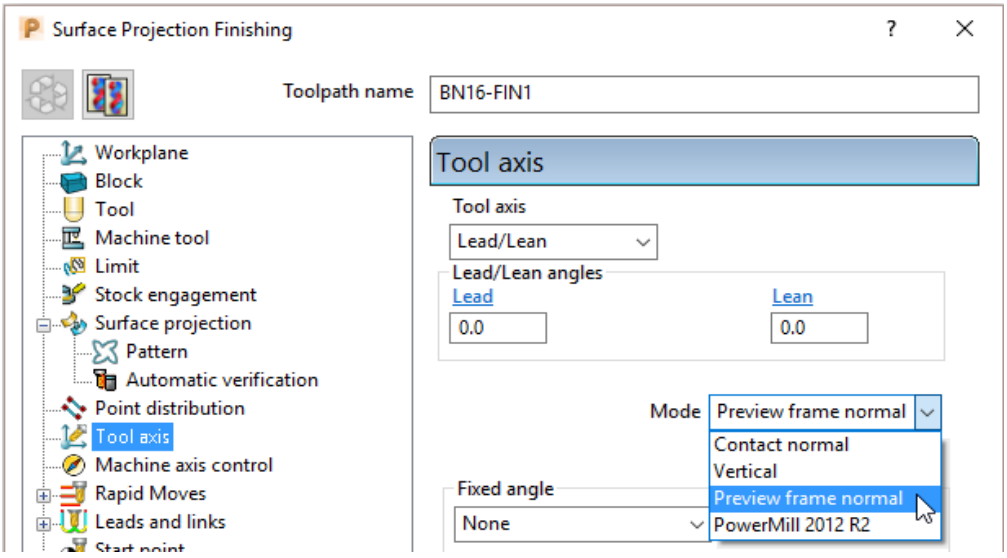


14 Enter the values into the **Pattern** page of the **Surface Projection Finishing** exactly as shown below.



- ***Pattern direction*** – **U**
- ***Start corner*** – **Min U max V**
- ***Limits (Distance)*** – **V** (ticked):- **Start 0 End 95**

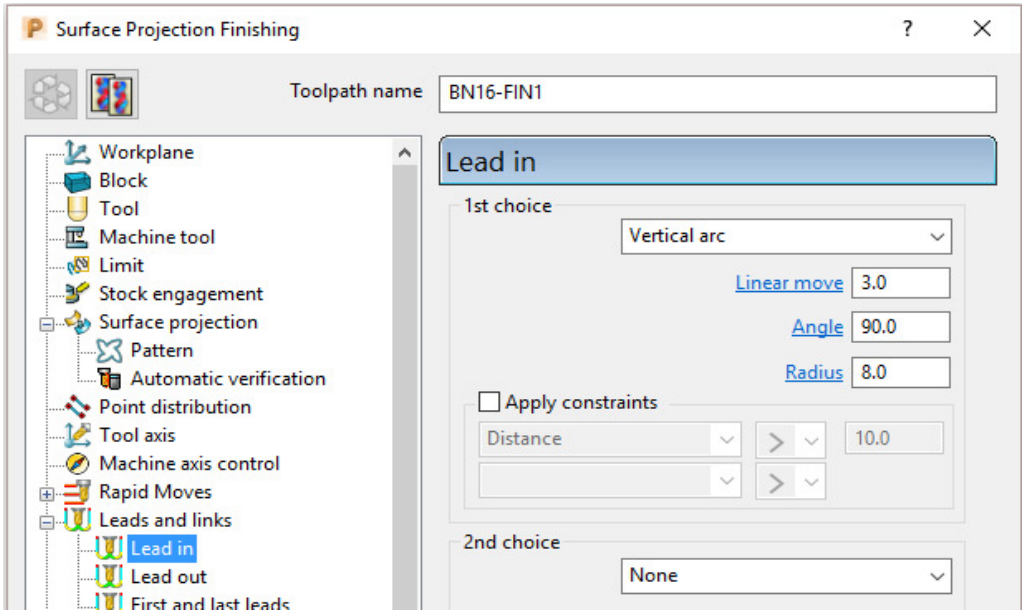
15 Enter the values into the **Tool axis** page of the **Surface Projection Finishing** exactly as shown below.



16 In the **Tool Axis** page select **Lead/Lean** with both values set to **0**.

Copyright © Autodesk

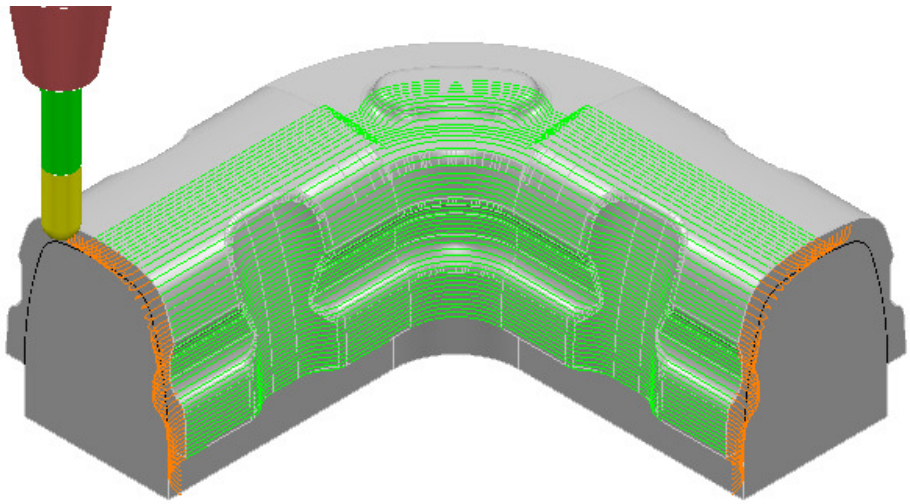
17 Enter the values into the **Tool axis** page of the **Surface Projection Finishing** exactly as shown below.



18 In the **Leads and Links** pages set:-

- Set both ***Lead in*** and ***Lead out*** – **Vertical Arc** – ***Distance*** **3**, **Angle 90**, **Radius 8**.

19 Select **Calculate** to create the **toolpath**, but do not close the dialog.



*For visual reasons a larger **Stepover** has been used on the **Toolpath** shown in the illustration*

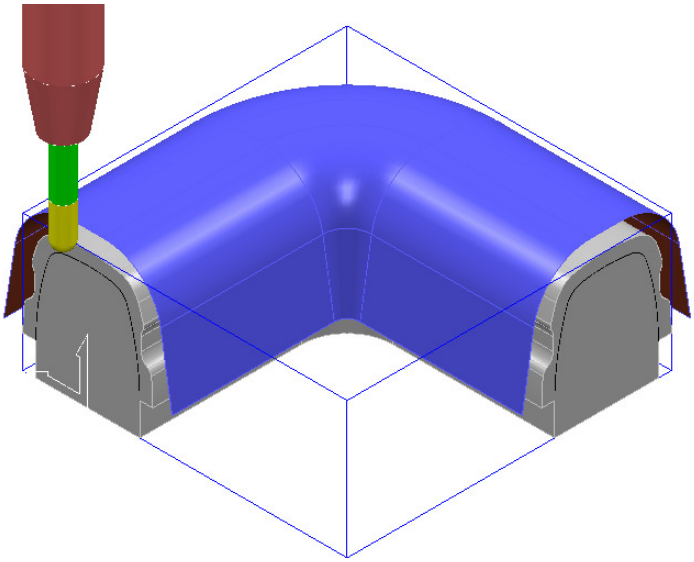
20 In the **Surface Projection** dialog select ***Make a Copy*** to re-activate the **Settings**.

It is possible for all or part, of the **Reference Surface** to exist outside of the component to be machined. In this case the option, **Import Reference Surface** will be used as it is not a part of the component model to be machined.

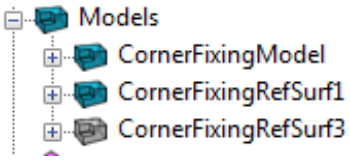
21 Right-click on **Models** in the **PowerMill *explorer*** and select **Import Reference Surfaces**.

22 In the **Reference Surface** dialog, select the **Model**:

….\PowerMill_Data\five_axis\ CornerFixing\CornerFixingRefSurf3.dgk



*The new **Reference Surface** exists outside the component to be machined. As a result of using the command **Import Reference Surfaces**, the **Machining Mode** is permanently set to **Ignore** for the **Reference Surface**.*

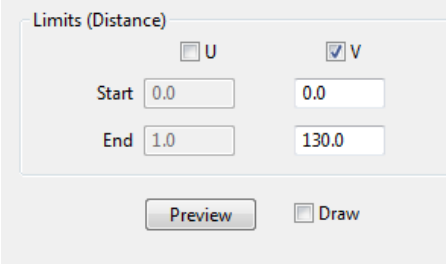


*The imported **Reference Surface** is displayed in the **PowerMill explorer** with a different colour (grey).*

23 Select the ***Reference Surface*** - **CornerFixingRefSurf3**.

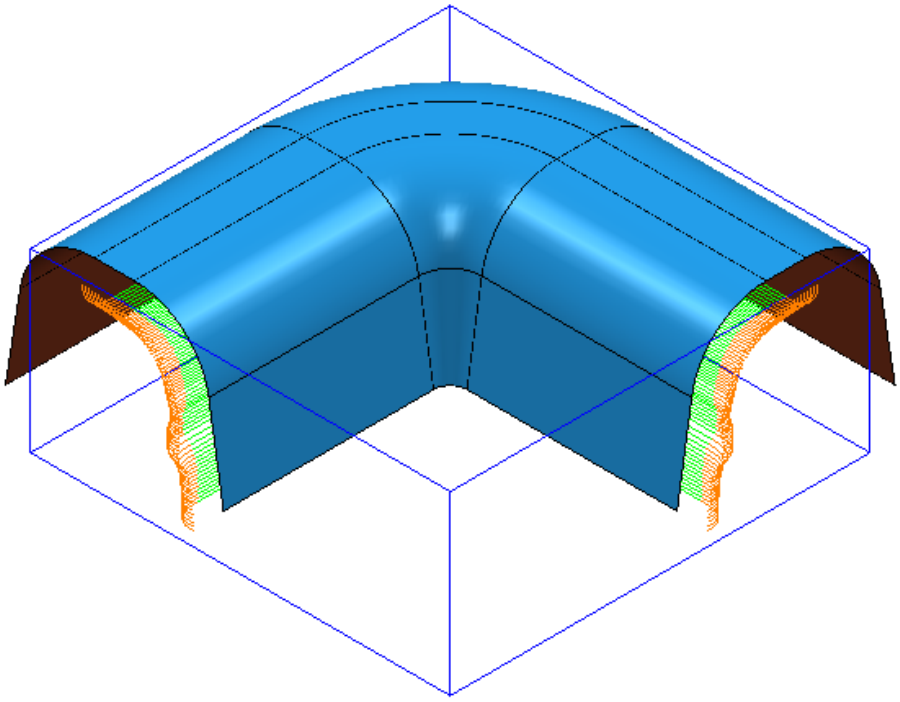
24 Rename the **Toolpath** as **BN16-FIN2**.

25 In the **Pattern** page change the ***Limits (Distance)*** **V** as shown in the image below.



26 **Calculate** the copied, **Projection Surface** Strategy.

Copyright © Autodesk

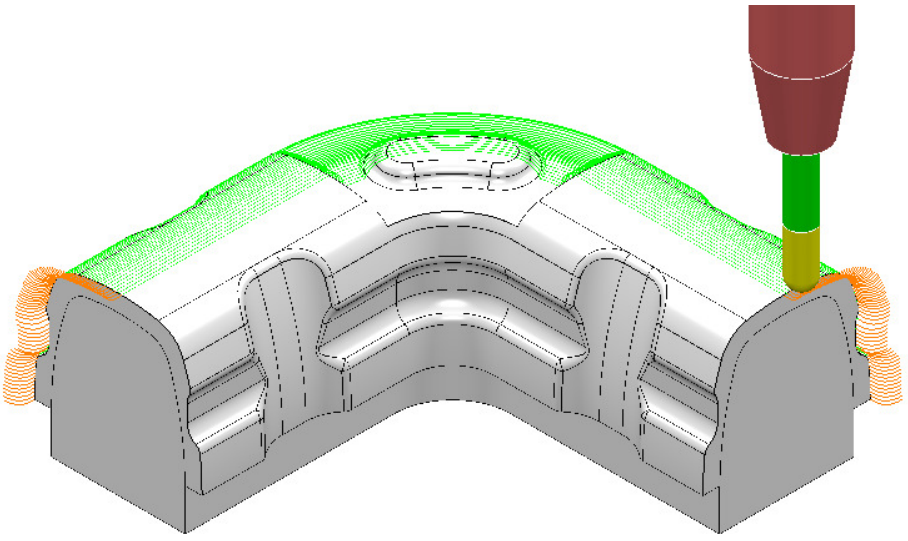


27 To update the stored **Project**, select **File > Save Project**:

....\COURSEWORK\PowerMill_Projects\ProjectionSurface-Example1

Exercise

28 Create a new **Projection Surface Finishing** (named **BN16-FIN3**) starting at the centre of the top face to complete the other side of the component model (as shown below).



29 To update the stored **Project**, select **File > Save Project**:

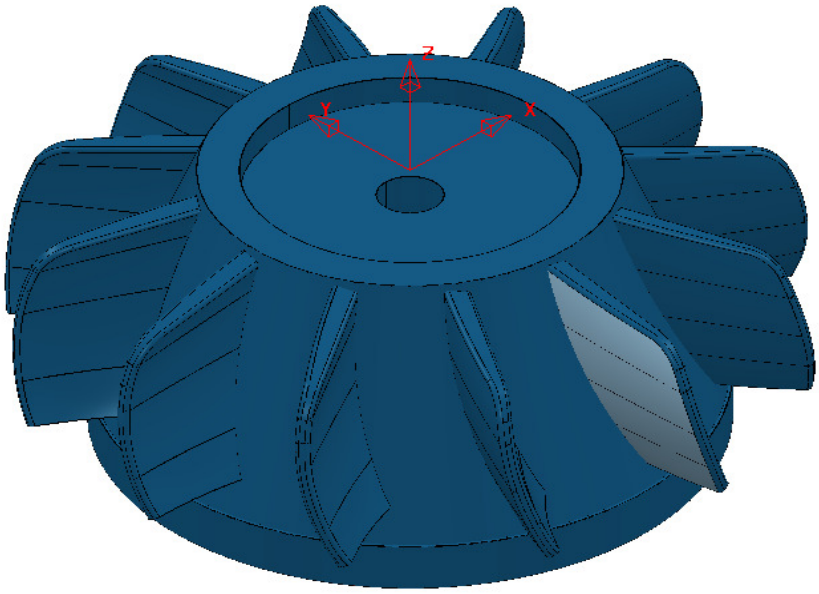
Surface Projection Range.

Depending on the application, it may be necessary to limit the infinite ***projection range*** while applying the **Surface Projection** strategy. This situation occurs where the part of the model to be machined is shielded by other surfaces that are in the projection path. This command is, at present, only available via typed input into the **PowerMill Command Window**. A more efficient way would be to store the command lines for a range of different distances in a series of **macros**, which in turn could be accessed via a **user defined menu**.

1 Select **File** - **Delete all** and **Tools** - **Reset forms**.

2 **Open** the (Read Only) **Project**:

....\PowerMill_Data\five_axis\ImpellerType2\ImpellerType2-Start



3 **Save Project As**:-

….\COURSEWORK\PowerMill_Projects\ImpellerType2-EX1

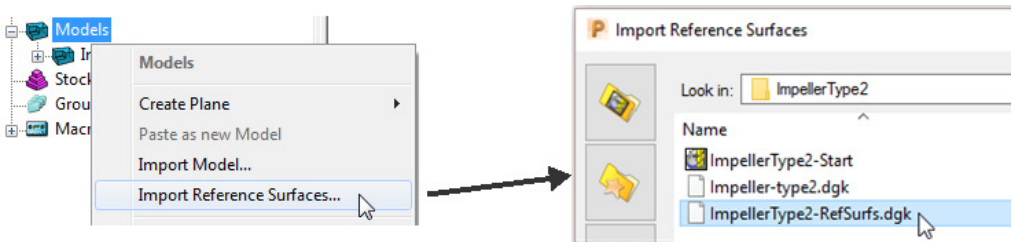
The imported **Project** already contains 3 Roughing **Toolpaths**, and a **Stock Model**.

4 Select an **ISO 1** view.

The next stage is to finish machine the 2 main faces of the blade positioned along the **–Y** direction (Shown shaded light blue in the above image). If the 2 main faces were single surfaces then it would be possible to directly apply the **Surface Machining** strategy (which is not affected by **Projection Range**). This option is not practical in this case as the main faces of the blade consist of several surfaces. As a result it is necessary to obtain 2 suitable single, reference surfaces created using CAD (ideally PowerSHAPE). These will purely be used as ***reference surfaces*** in the **Projection Surface Finishing** strategy and will not be physically machined.

Copyright © Autodesk

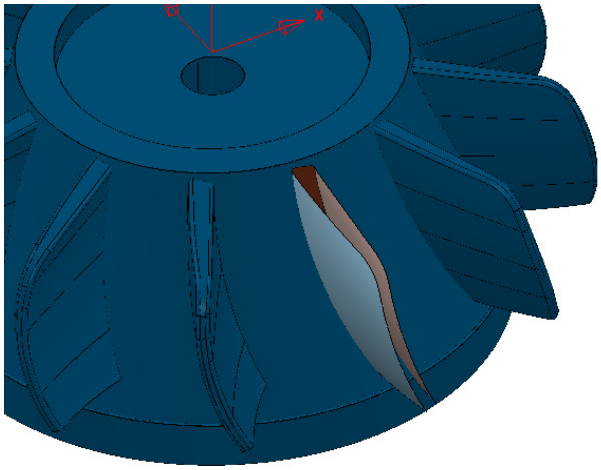
5 In the **PowerMill *explorer*** right mouse click on **Models** and from the local menu select **Import Reference Surfaces**.



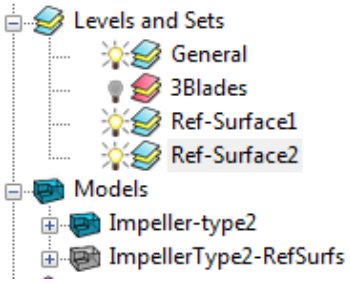
6 In the dialog browse to:-

...\PowerMill_Data\five_axis\ImpellerType2\ImpellerType2-RefSurfs.dgk

7 In the dialog select **Open** to import 2 ***reference surfaces*** *(shown below)*.



*By using this import method the 2 **reference surfaces** will always be ignored by a machining strategy during the calculation of a toolpath.*



*The **reference surfaces** are already acquired to appropriate **levels** (**Ref-Surface1** and **Ref-Surface2**) this being inherited from the CAD process.*

8 **Calculate** a **Block** using a **Cylinder** to the full **Model** dimensions.

9 **Calculate** the **Rapid Move Heights** using the default settings.

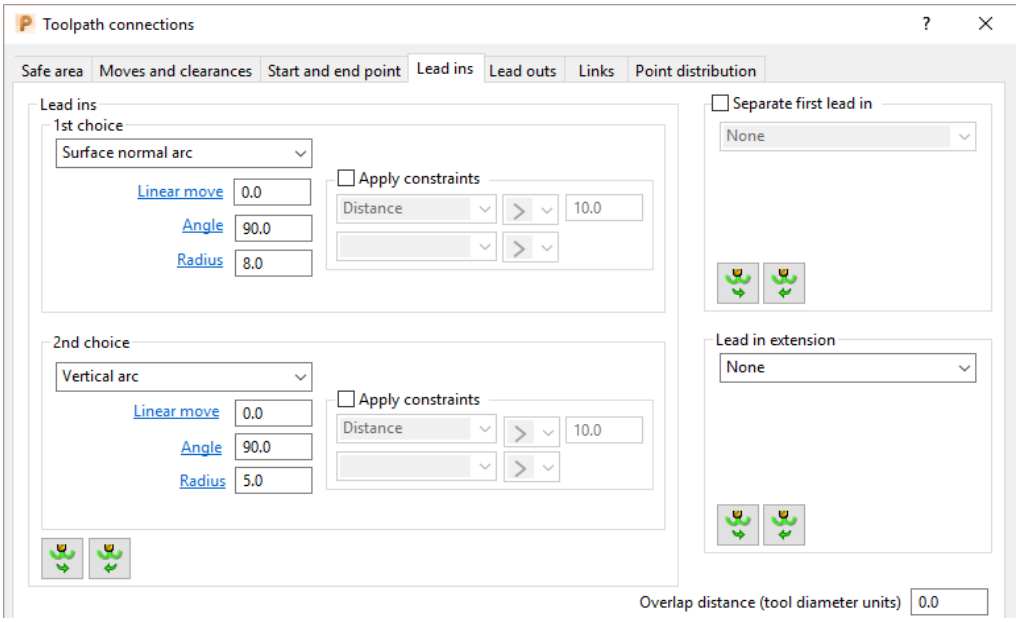
10 Switch off the **Level**, **Ref-Surface2**.

11 Select the reference **Surface** stored on the **level**, **Ref-Surface1**.

12 **Activate** the **Workplane *TopFace***.

13 From the **PowerMill *explorer***, **Activate** the **BN16** tool.

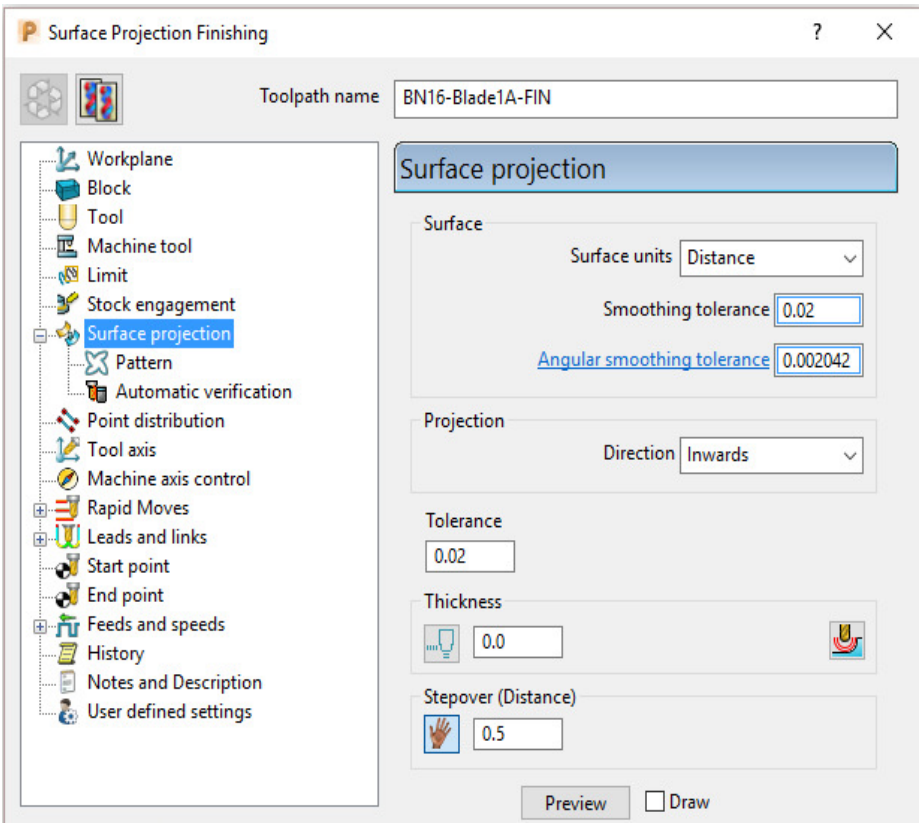
14 Set **Lead In** and **Out**, ***1st choice*** to use **Surface normal arc**.



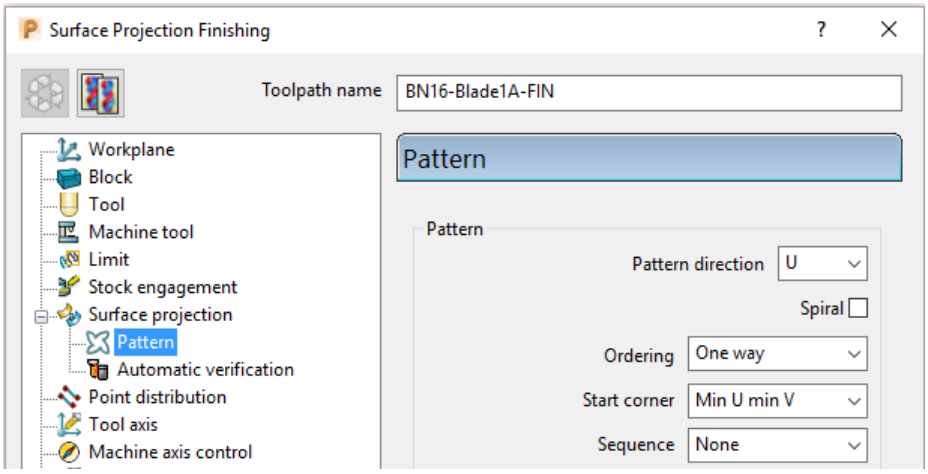
15 From the **main** toolbar select the **Toolpath Strategies** .

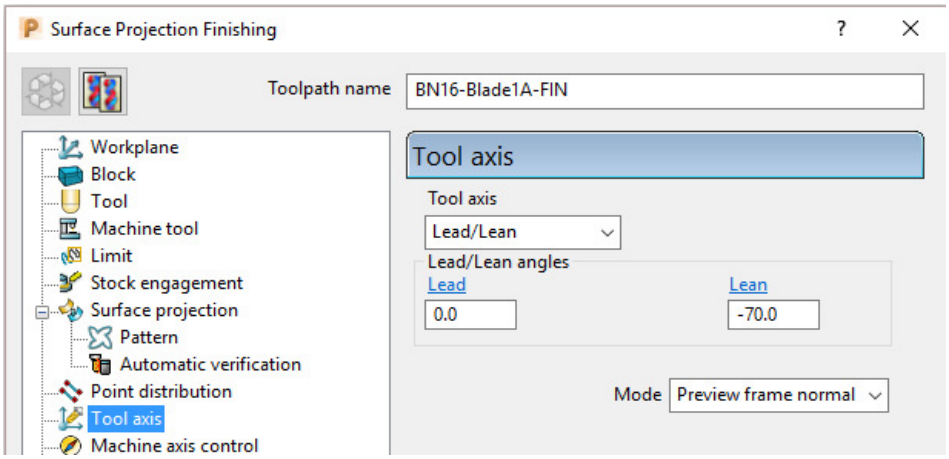
16 From the dialog select the **Finishing** tab followed by the **Projection Surface Finishing** option.

17 Enter data into the following 3 pages of the **Surface Projection** dialog exactly as shown below.

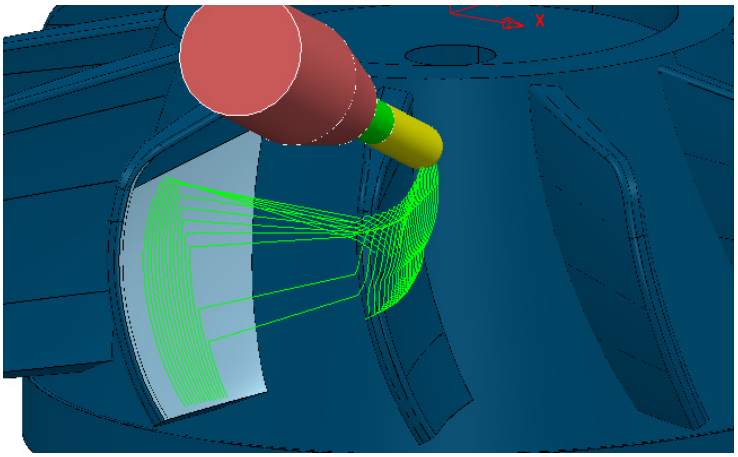


Copyright © Autodesk

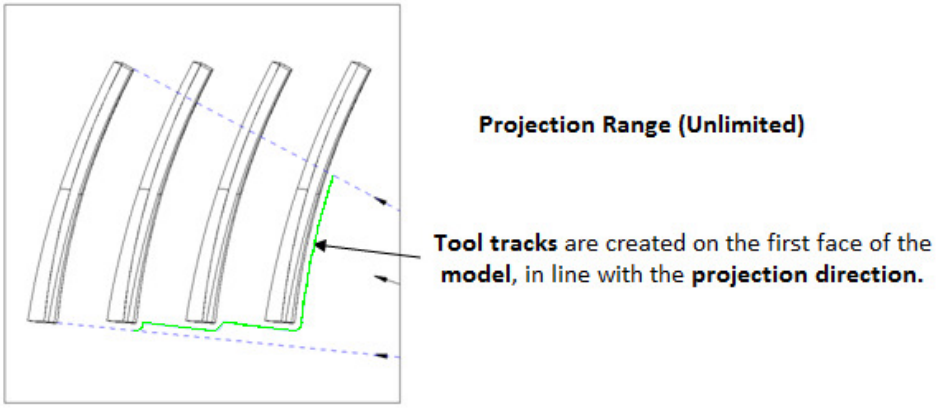




18 Select **Calculate** in the **Surface Projection Finishing** dialog to produce the following toolpath, and then **Close** the dialog.



*With the **Projection Range** set to **OFF** (default), ie. No limit, the toolpath is projected inwards towards the surface to be machined from outside the model limits. As the strategy is projected some of it will appear on neighbouring surfaces instead of those intended to be machined and create the toolpath as shown.*



This problem can be resolved by switching to the **Projection Range ON** option and assigning suitable **+** and **–** distance values.

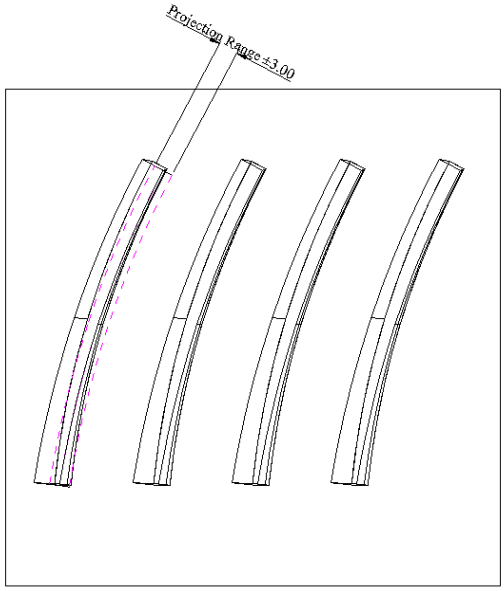
19 Right mouse click on the toolpath **BN16-Blade1A-FIN** and from the local menu select **Settings** to re-open the strategy dialog.

20 Select **Recycle** .

21 From the **main** pull down menus select: **View** > **Toolbar** > **Command**.

22 Enter the following commands into the **Command Window** that has appeared at the below of the graphics area:

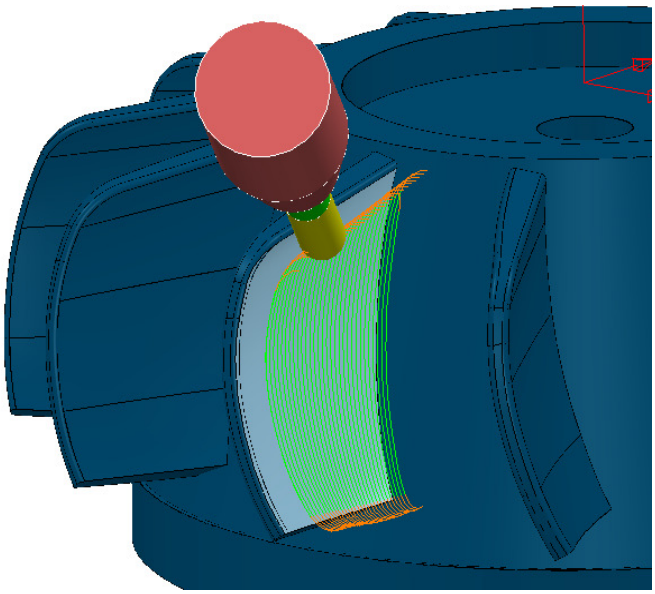
```
EDIT SURFPROJ AUTORANGE OFF
EDIT SURFPROJ RANGEMIN -3
EDIT SURFPROJ RANGEMAX 3
```



The above command input limits the **Surface Projection** Range to **+** and **- 3mm**.

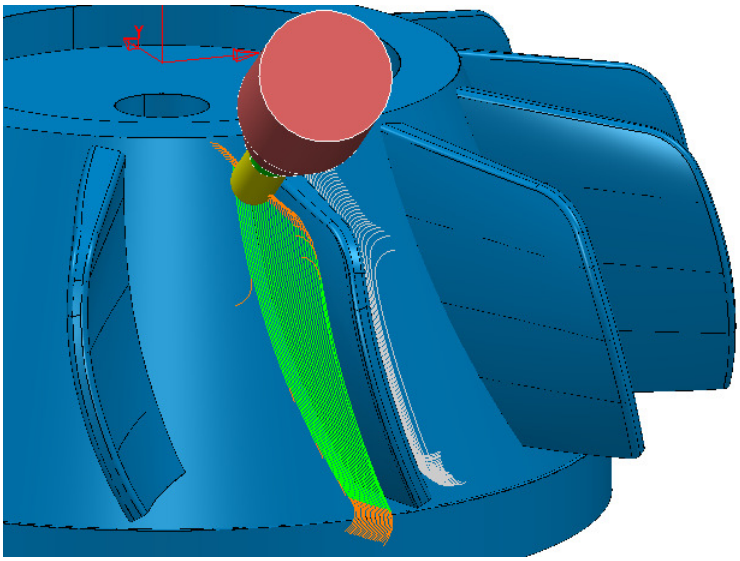
23 Select **Calculate** to create the new toolpath.

Copyright © Autodesk



24 **Simulate** the toolpath to observe the effect of the ***limited - Projection*** **Range**.

25 Create a **Copy** of this **Surface Projection** strategy on the other side of the blade (Use the reference **Surface** stored on **Level**, **Ref-Surface2**).



26 In the **Command Window** type the following to restore the default, ***infinite*** **Projection Range**:

EDIT SURFPROJ AUTORANGE ON

*This command returns to the default **infinite limit** for the **Surface Projection** distance range.*

27 select **File > Save Project** to update the stored **Project**:

...\COURSEWORK\PowerMill_Projects\Ref-Surfs-Blades

Surface Finishing and Surface Projection - Spiral

Both the **Surface Finishing** and **Surface Projection Finishing** strategies include the option to create a spiral toolpath:

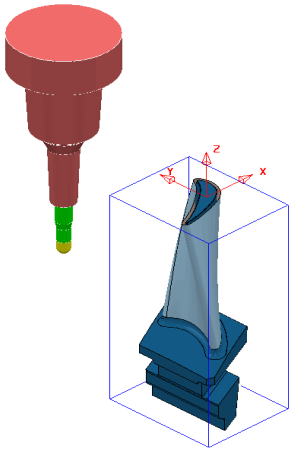
*The **stepover** is automatically adjusted so that the final cut finishes exactly on the surface edge at the end of the toolpath.*

If you select a **Surface Units** of **Distance**, then the first and last passes will match the edges of the surface. To achieve this, the intermediate passes will adjust to less than or equal to the specified **stepover**.

Projection Surface - Spiral Example

1. Select **File** - **Delete All** and **Tools** - **Reset Forms**.
2. **Open** the (read-only) **Project**:

   ....\PowerMill_Data\five_axis\ProjSurf-Blade\SurfProjSpiral-Start

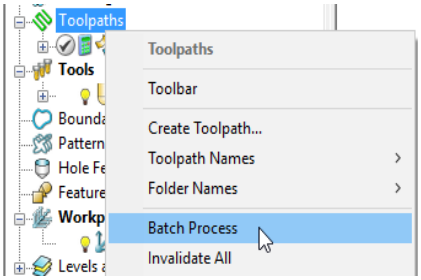


3. **Save Project As**:

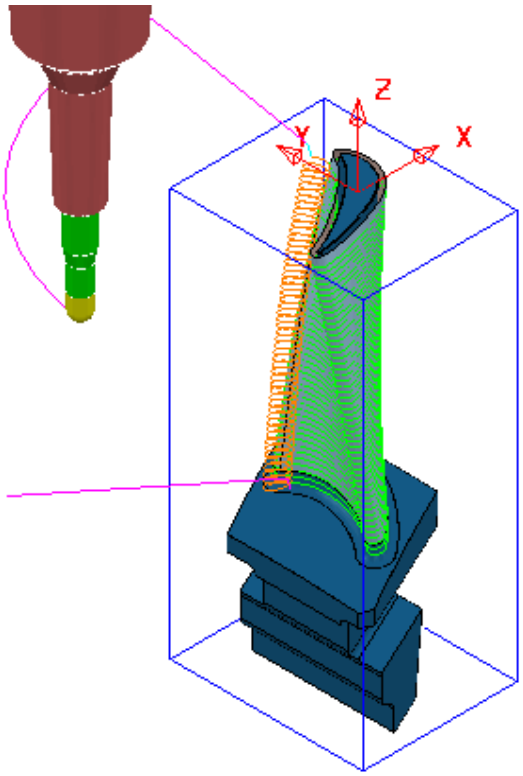
   ….\COURSEWORK\PowerMill_Projects\SurfMachine-Spiral

*The Strategy in the above **Project** is unprocessed (As indicated by the Calculator icon).*

4. In the **PowerMill *explorer***, right-click on **Toolpaths** to open the local menu and select **Batch Process** to create the toolpath.



Copyright © Autodesk

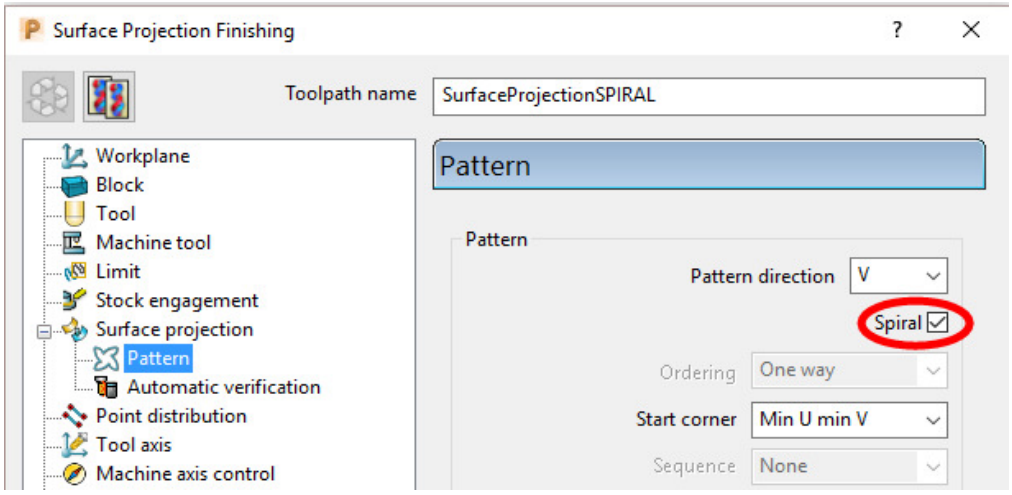


The strategy automatically reinstates the original selected **surface** used on a previous occasion. The resultant **Toolpath** contains independent, stacked tool tracks and **Surface Normal Arcs** for **Lead In**/**Out**. For this strategy it, would be both possible and more appropriate to use the **Spiral** option.

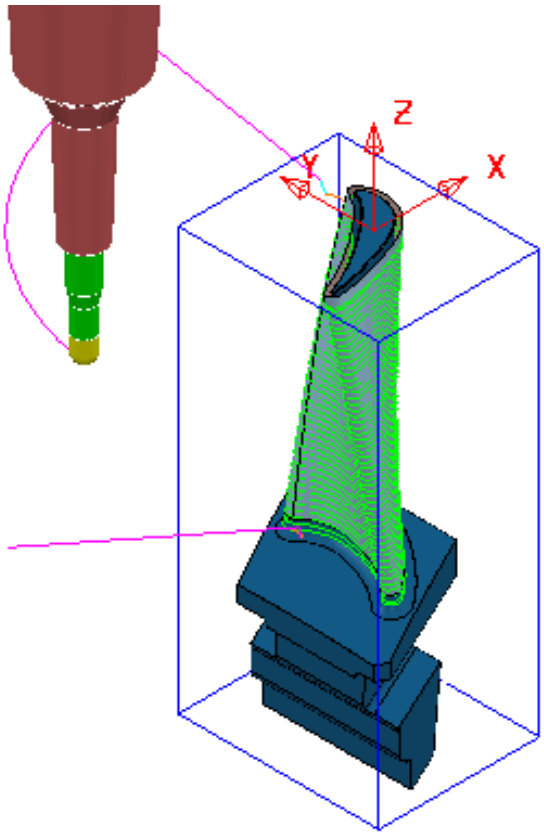
5 Right-click on the **Active** toolpath to open the local menu and select **Settings** to open the original **Surface Projection Finishing** dialog.

6 Select ***Make a Copy*** to re-activate the **Settings**.

7 Select the **Pattern** page in the **Surface Projection Finishing** dialog.



8 Rename the strategy as **SurfaceProjectionSPIRAL**, tick the **Spiral** box, and check that the ***reference surface*** is selected, before selecting **Calculate**.



This time the **Projection Surface** strategy consists of a single **Spiral** tool track, providing optimum efficiency and a smooth, consistent material removal.

9 **Close** the **dialog**.

10 Select **File** - **Save Project** to update:-

....\COURSEWORK\PowerMill_Projects\SurfMachine-Spiral

Copyright © Autodesk

5. Embedded Pattern Finishing

Introduction

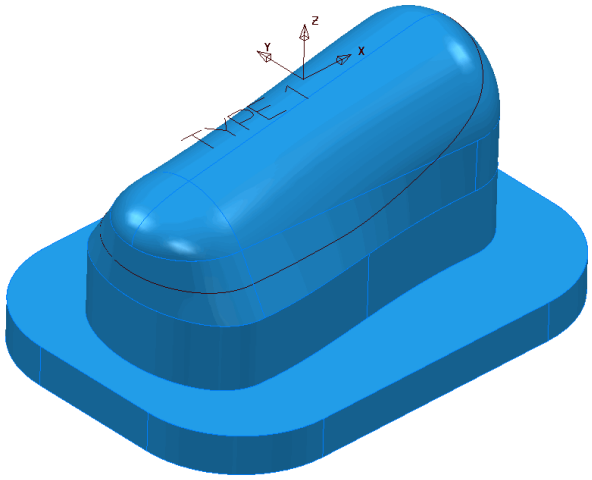
This **Five Axis** strategy not only tracks along the **Embedded Pattern** but automatically applies a **Tool Axis** alignment relative to the ***Surface Normals***.

An **Embedded Pattern** is created as a projection of an ordinary **Pattern** using the **Edit** - **Embed** option. The **Pattern** projection can be performed down the active ***Z Axis*** or towards the ***closest point*** on the **model**.

Embedded Pattern Finishing - Engraving

1 **Open** the (Read only) **Project**:

...\PowerMill_Data\five_axis\ 5axis_Embedded_Pattern\TrimPart-Start



The **model** includes some wireframe ***Text*** and a 'scribe line' **curve** running around the form.

These will each be used create separate **Patterns**. The corresponding **Embedded Patterns** can then created from the 2 new **Patterns**.

2 Select **File** > **Save Project As**:

….\COURSEWORK\PowerMill-Projects\Trim-Part

3 Activate the toolpath, **BN16-Form-FIN** to reinstate suitable settings for the **Embedded Pattern**.

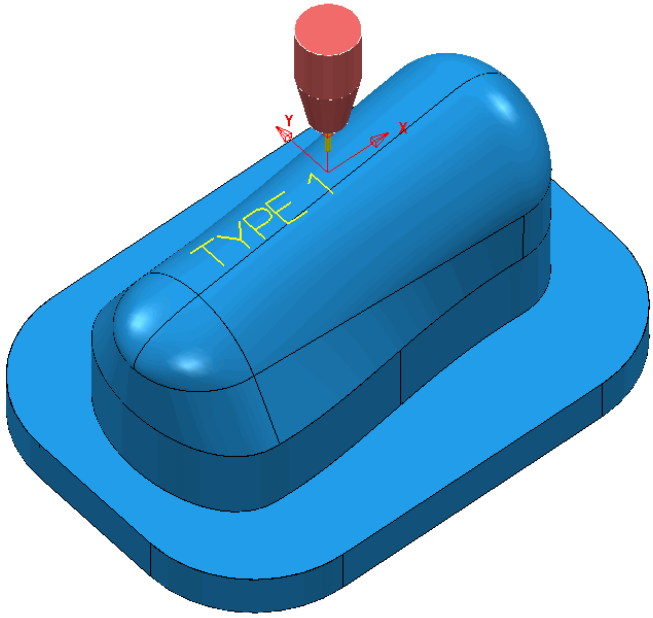
4 Activate the tool **BN2**.

5 Ensure that none of the stored **Boundaries** are *active*.

6 In the graphics area, select the wireframe part of the model named '**TYPE 1**'.

7 In the **PowerMill *explorer***, right-click on **Patterns** and from the local menu, select **Create Pattern**.

8 Right-click on the new (empty) **Pattern** and from the local menu, select **Insert - Model** to inherit the selected wireframe text, '**TYPE 1**'.

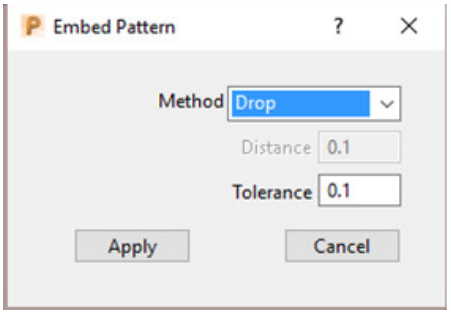


The first **Pattern** contains the text '**TYPE 1**' and is positioned above the **Model**. This **Pattern** is then used to create an **Embedded Pattern** which projects down **Z** onto the **model**. The **Embedded Pattern** will also pick up and store the **Surface Normals**.

9 Rename the **Pattern** as '**Text**'.

10 In the **PowerMill *explorer***, right-click on the **Pattern** and from the local menu select **Edit** > **Embed**.

11 From the local menu, select **Method** - **Drop** and **Apply** the dialog.



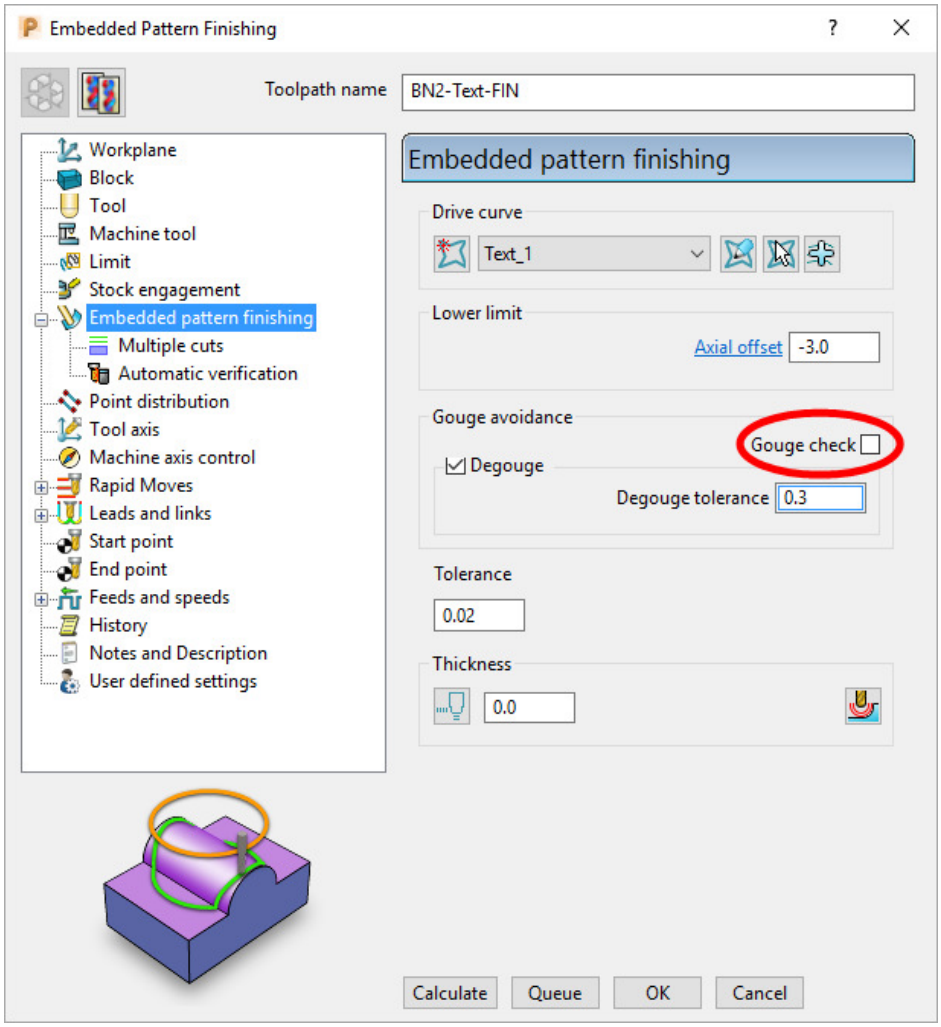
*An **Embedded Pattern** called **Text_1** is created as projected down Z and is identified by the icon* *in the explorer. The original Pattern is retained.*

Copyright © Autodesk

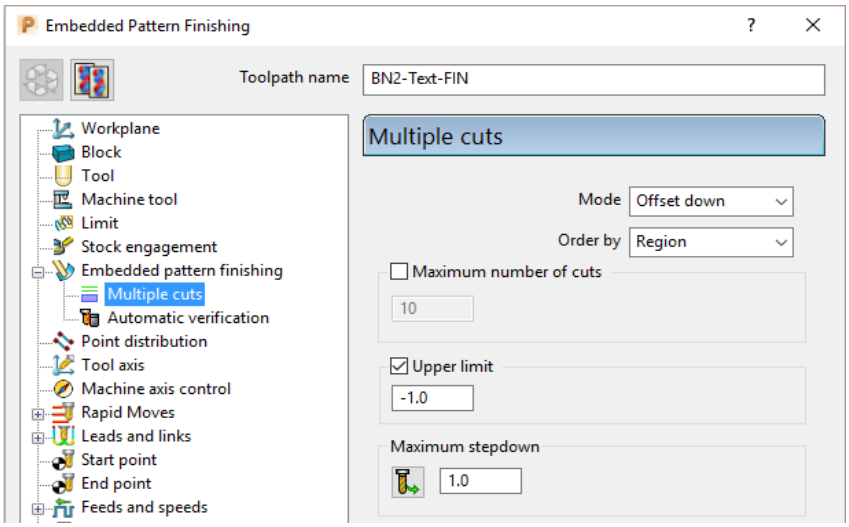
12 Select **Toolpath Strategies** and from the **Strategy Selector** dialog sect the **Finishing** tab.

13 From the **Finishing** options, select the **Embedded Pattern Finishing**.

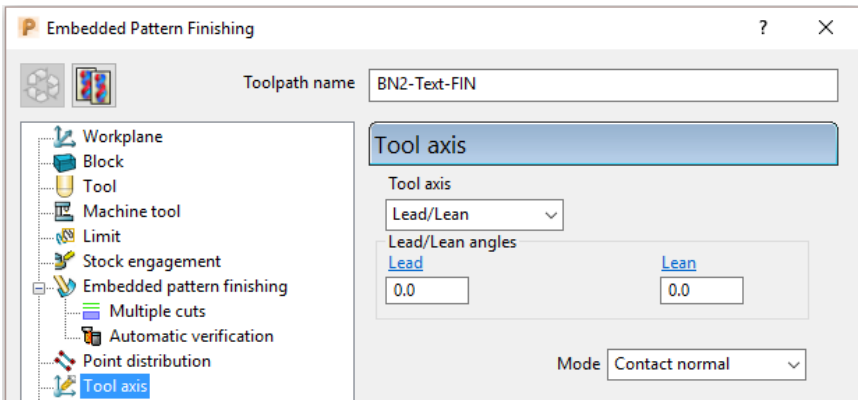
14 Enter the values into the following pages of the **Embedded Pattern Finishing** dialog exactly as shown.



- ***Drive curve*** **Text_1**
- ***Axial offset*** **-3**
- ***Untick*** **Gouge Check**



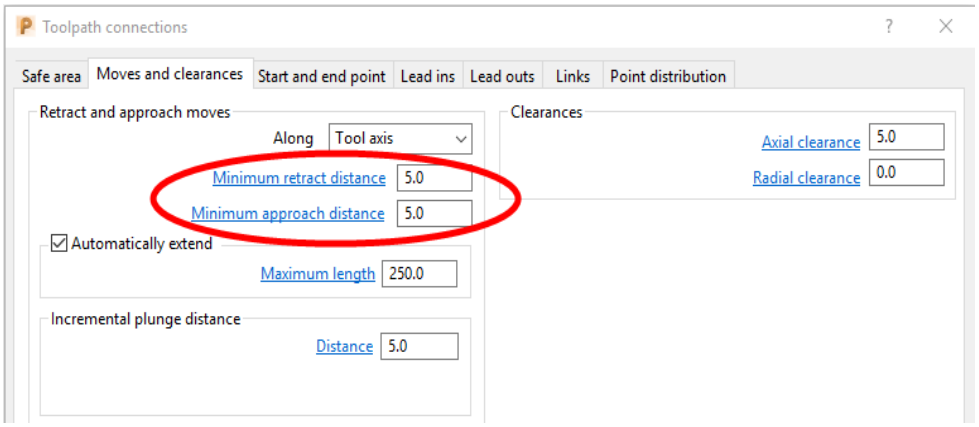
15 Tick **Upper limit** with a value of **-1**.



16 Set the **Tool Axis** to **Lead/Lean** both with **0** angles.

It is necessary to ensure that the tool plunges and retracts along the **Tool axis** during **Embedded Pattern** finishing (**Retract distance 5** and **Approach distance 5**).

17 From the **main** toolbar select **Leads and Links**, select the **Moves and clearances** page.



18 In both **Minimum retract/approach distance**, enter **5** (as shown above).

19 **Accept** the **Leads and Links** dialog.

Copyright © Autodesk

20 Back in the **main** strategy dialog select **Calculate** to create the **Toolpath**.

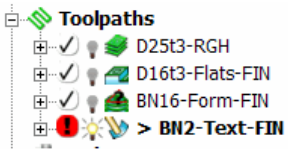
The text '**TYPE 1**' is machined to a total depth of **-3** with the tool aligned to the ***Surface Normals***.

21 Perform a **ViewMILL** Simulation on all of the **Toolpaths**.

The **Embedded Pattern** toolpath has been produced with the ***tool aligned normal*** to the **surface model**.

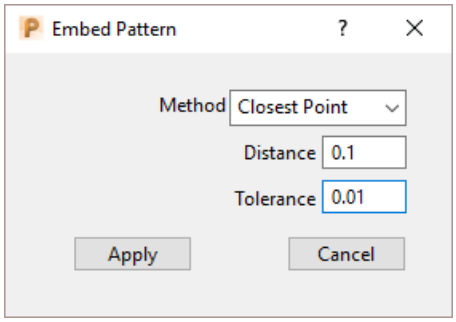
*If the toolpath is **verified** for **gouges** it will take on a **Red** 'gouge' warning. It is not possible to create a toolpath using a -ve thickness value that is higher than the tool tip radius while **Gouge check** is switched on.*

This is in technical terms a 'deliberate gouge' and will be ignored.

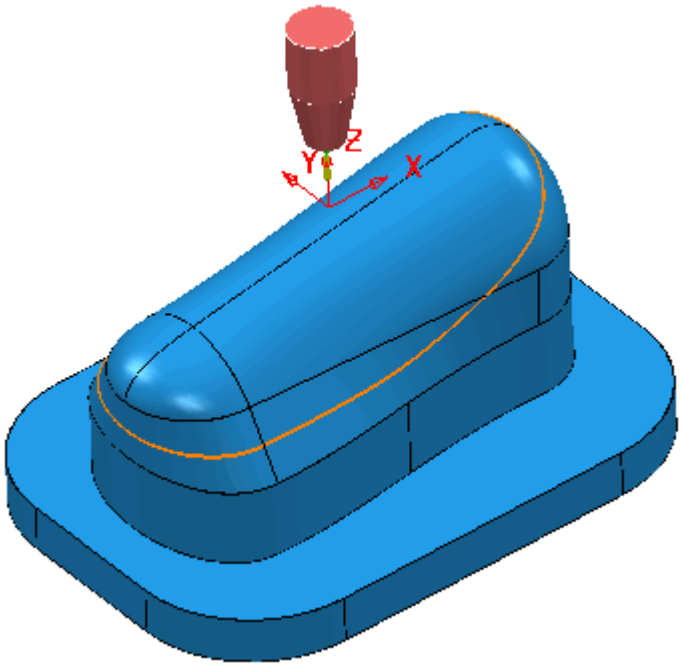


22 Create a new **Pattern** named **ScribeLine** and **Insert** the wireframe ***Curve***, running around the upper wall of the **model**.

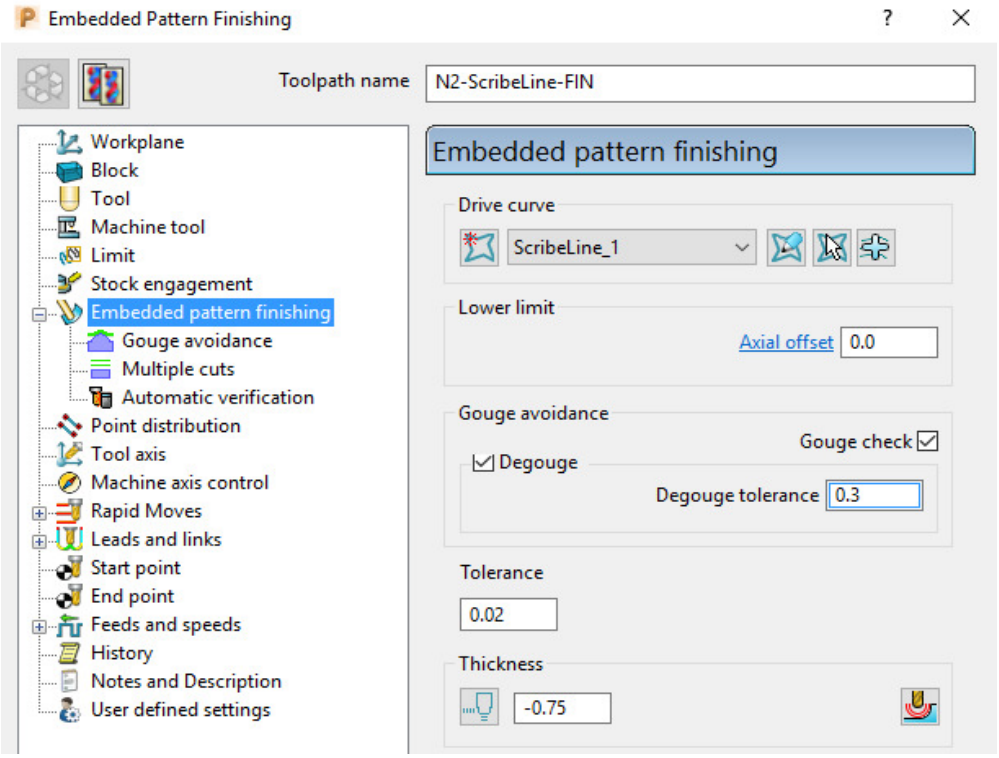
23 Create an **Embedded Pattern** from the **Pattern** (**ScribeLine***)* but this time select ***Closest Point*** in the dialog.



This time the **Embedded Pattern** results from a projection towards the nearest point on the **Model** (Normals) from the original **Pattern**.



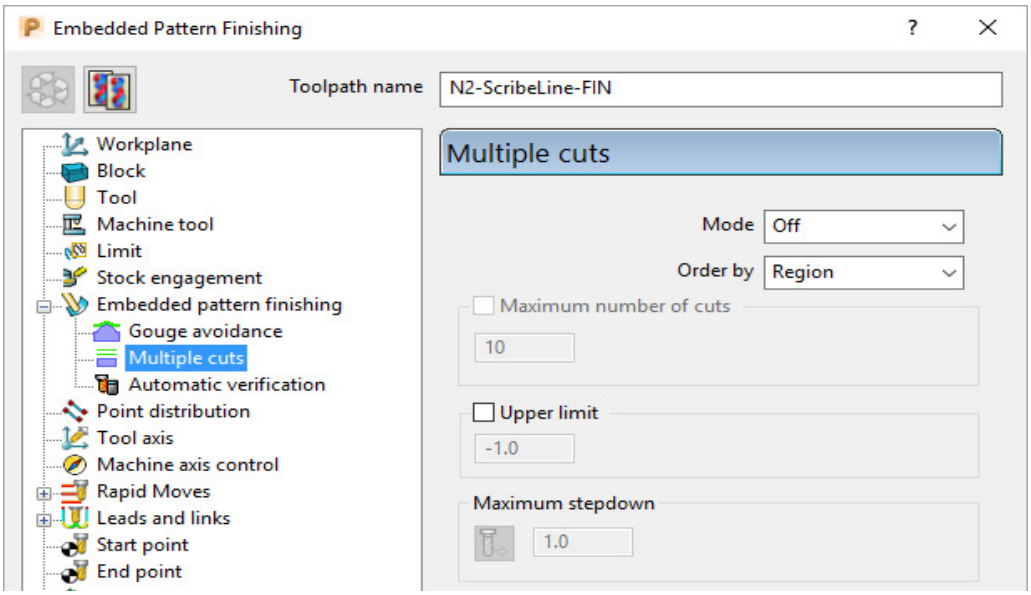
24 Open a new **Embedded Pattern** strategy named **BN2-ScribeLine-FIN** using **Drive curve** - ***ScribeLine_1***.



- ***Axial offset***: **0**
- ***Gouge check***: **Ticked**
- ***Thickness***: **-0.75**

This time the required ***Depth Of Cut*** is less than the ***Tool Tip Radius*** allowing the **Thickness** option to be used instead of **Axial offset**.

Copyright © Autodesk



- Set **Mode**: **Off**
- **Upper Limit**: **Un-ticked**

25 Select **Calculate** to create the **Embedded Pattern** toolpath.

26 Continue the **ViewMILL** Simulation on the new toolpath.

27 Select **File** > **Save Project**.

Copyright © Autodesk

6. Five Axis Swarf Machining

Introduction

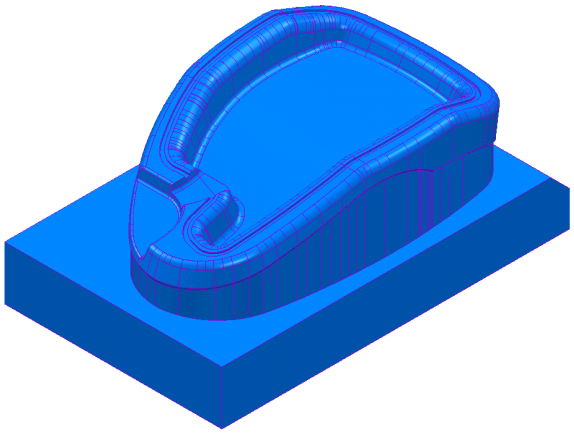
Five Axis - **Swarf Finishing** is designed to create a toolpath where the selected **surfaces** are machined with the side of the tool (the default **Tool Alignment** option is **Automatic**). A **Swarf Finishing** toolpath will only exist where the tool is able to remain in full contact with the selected **surfaces** along the whole cutting depth. In other words, the tool must always be able to find a ***Linear*** path across the **surfaces**. It is possible for the user to apply a different tool alignment (For example, **Vertical** or **Lead\Lean**) but the selected **surfaces** must still be ***Swarfable*** for machining to occur.

In cases where imported **surfaces** are supposed to be ***Swarfable*** but are not of a suitable quality, the upper and lower edges can be created as separate (wireframe) **Patterns** to be used with **Wireframe Swarf Finishing**.

Swarf Finishing - example 1

1. Select **File** > **Delete All** followed by **Tools** > **Reset Forms**.
2. **Import** the **model**:

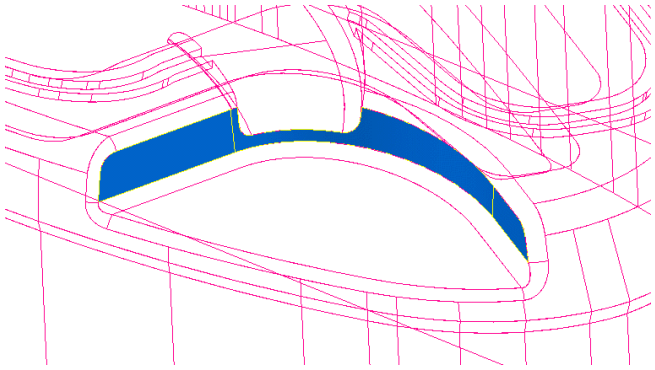
   ....\PowerMill_Data\five_axis\swarf_mc\swarf_model.dgk



3. Create a **Block** using **Type** - **Box** and **Defined by** - **Model**.
4. Create a Dia 12, tiprad 1, tool named **D12TR1**.
5. **Calculate** the **Rapid move heights**.

Copyright © Autodesk

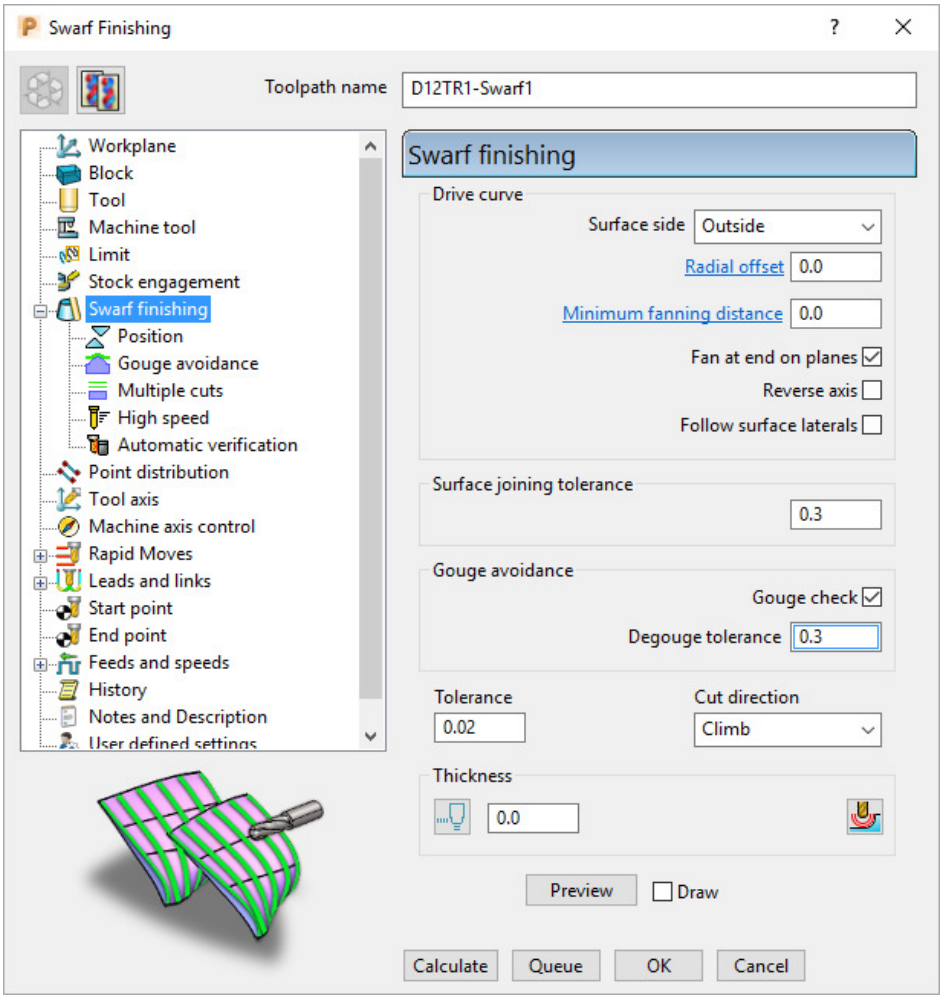
6 Change the view to **ISO 4** and select the local surfaces to be **Swarf** machined (as shown below shaded).



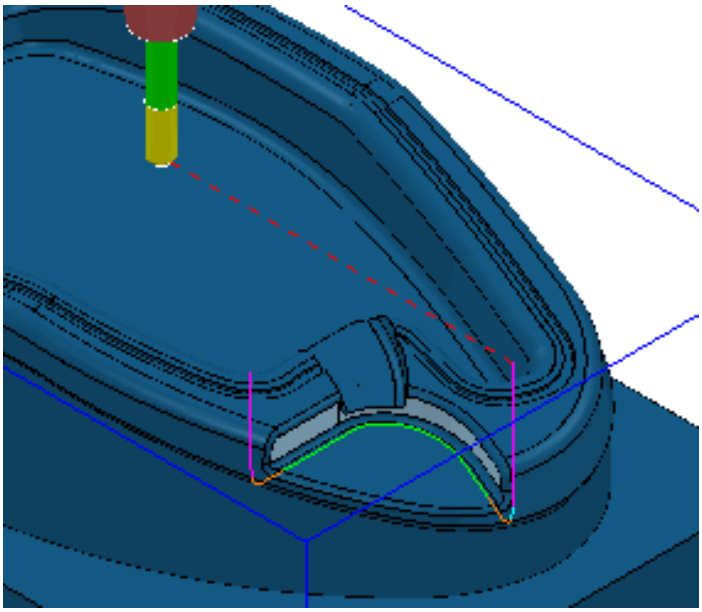
7 Select **Toolpath strategies** and from the **Finishing** dialog select the **Swarf Finishing** option.

8 Enter the values into the main page of the dialog exactly as shown below and **Calculate**.

*The default **Tool Alignment** is **Automatic** for this strategy.*



Copyright © Autodesk



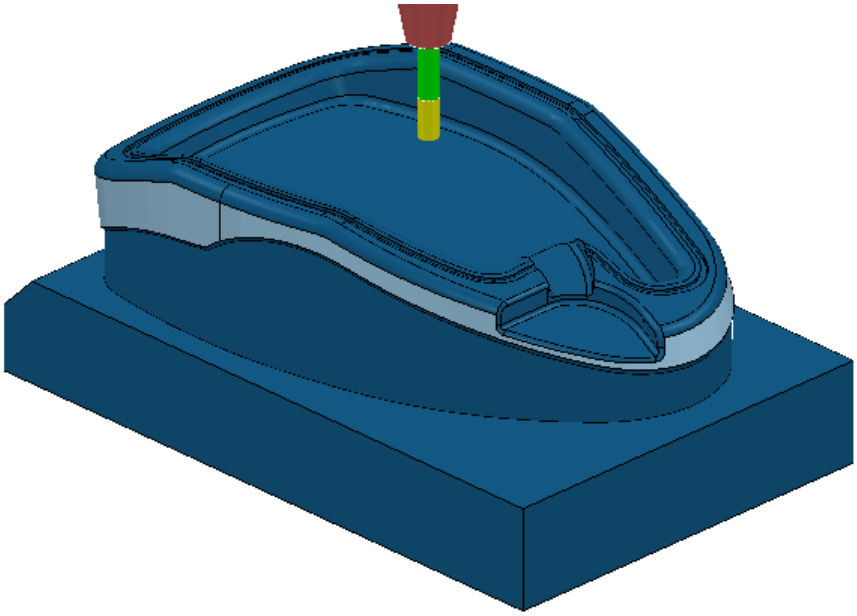
9 Open the **Leads and links** dialog via the icon and **Apply** the following settings:

Lead in/out: ***Vertical Arc*** – **Distance 10** – **Angle 90** – **Radius 5.0**

Links: ***1st/2nd choice*** – **Skim** ***Default*** – **Incremental**

10 Deselect the local surfaces, **Simulate** the resultant toolpath, and observe the constantly changing **tool alignment** as it **Swarf** machines the selected **surfaces**.

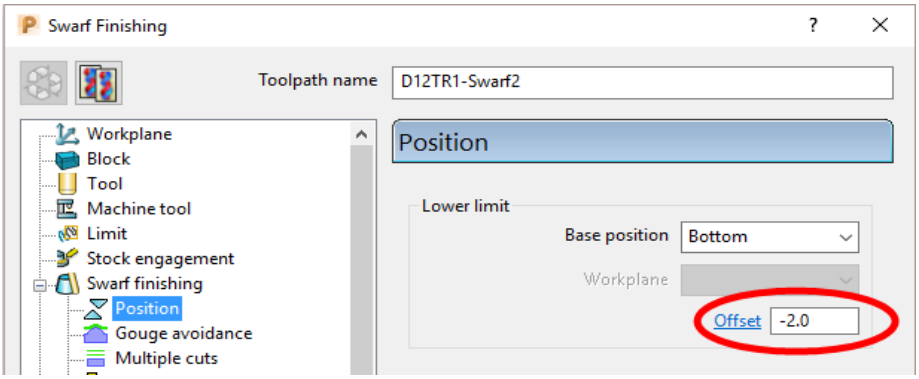
11 Select **Settings** to reopen the existing (active) **Swarf Finishing** strategy and then create a **Copy** named **D12TR1-Swarf2**.



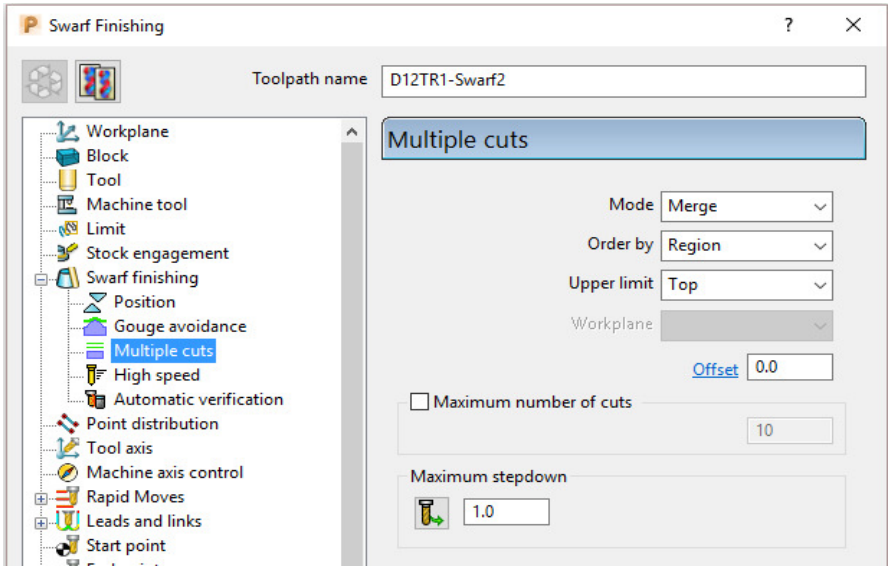
12 Select the, upper sidewall, surface (as shown above).

Copyright © Autodesk

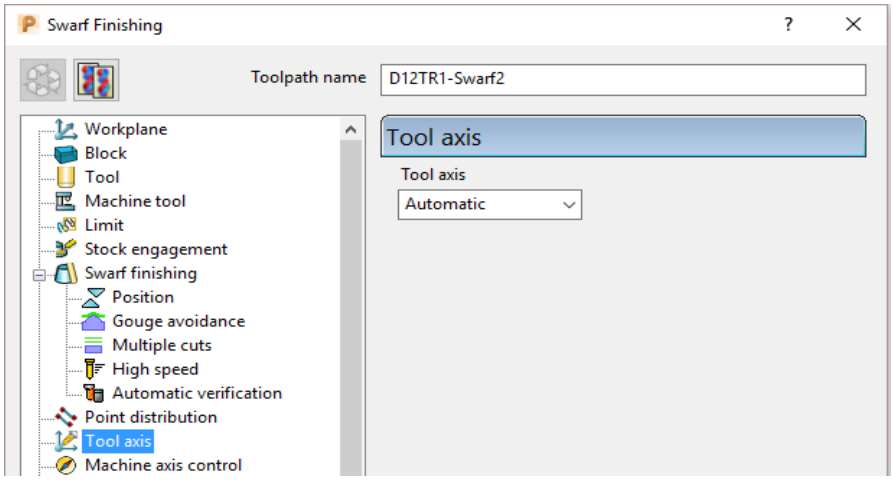
13 Select the **Position** page and set **Base position** - **Bottom** with ***Offset*** **-2**.



14 Select the **Multiple cuts** page, with the **Mode** set to **Merge** with **Maximum stepdown 1**.



15 Select the **Tool Axis** page and use the default, ***Tool axis*** - **Automatic**.



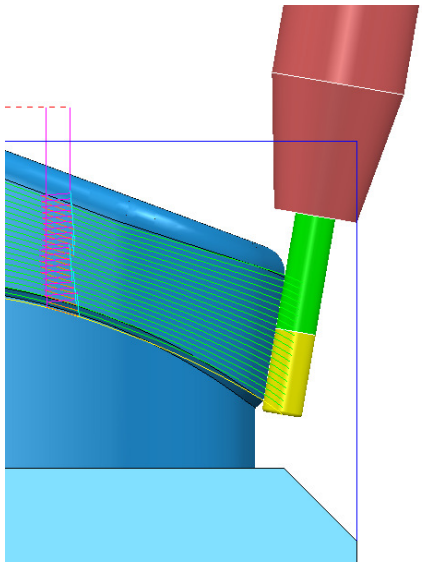
16 Select **Calculate** at the bottom of the dialog to create the toolpath.

Copyright © Autodesk

17 Open the **Leads and links** dialog via the icon and **Apply** the following settings:

Lead in/out: **Surface normal arc** – **Distance 0** – **Angle 90** – **Radius 5.0**

Links: ***1st/2nd choice*** – **Skim** ***Default*** – **Incremental**

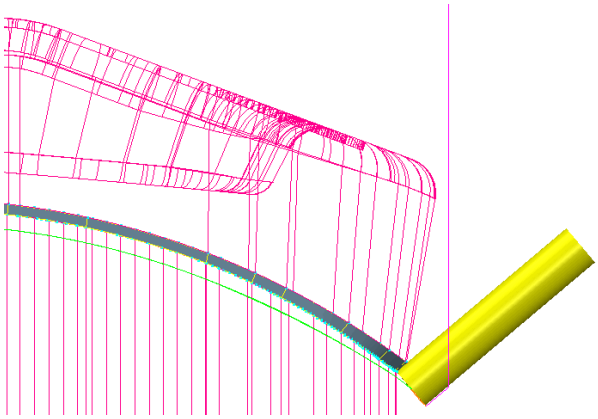


The new **Swarf Finishing** strategy steps down the selected **surfaces** while merging the **stepover** between the ***Upper*** and ***Lower contours***.

Exercise

1 Create a Dia 10 End Mill (**EM10**).

2 Create a **Copy** of the original, single pass, **Swarf Finishing** strategy to create a new toolpath aligned to the underside of the undercut recess on the outer sidewall.



3 Deselect the local surfaces, **Simulate** the resultant toolpath and observe the changing angle of the tool as it **Swarf** machines the selected surface.

4 Select **File** > **Save Project As**:

….\COURSEWORK\PowerMill-Projects\Swarf-Example

5 Select **File** > **Delete All** and **Tools** > **Reset Forms**.

Copyright © Autodesk

Swarf Finishing - Example 2

1 **Open** the **Project** (created earlier):

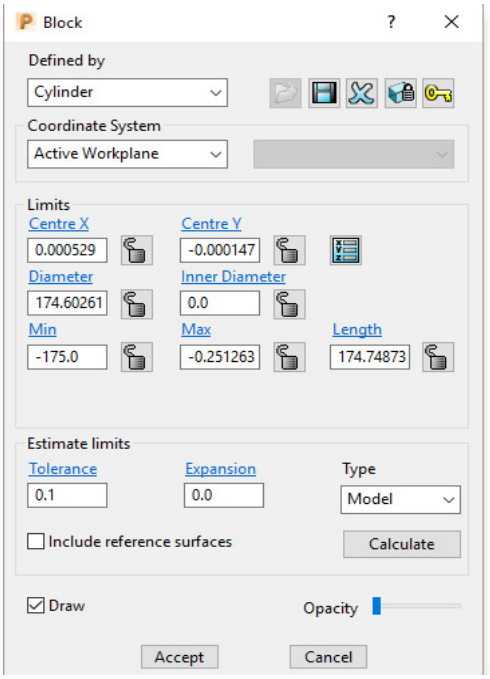
....\COURSEWORK\PowerMill-Projects\3Plus2-EX1

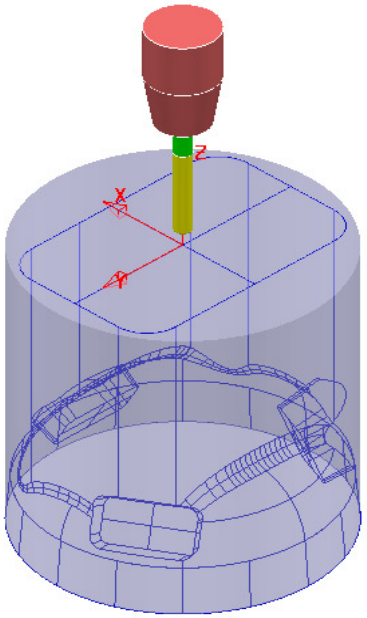
2 Activate the Dia 10 tiprad 1 tool (**D10T1**).

3 Activate the **workplane** - **ztop175_A**.

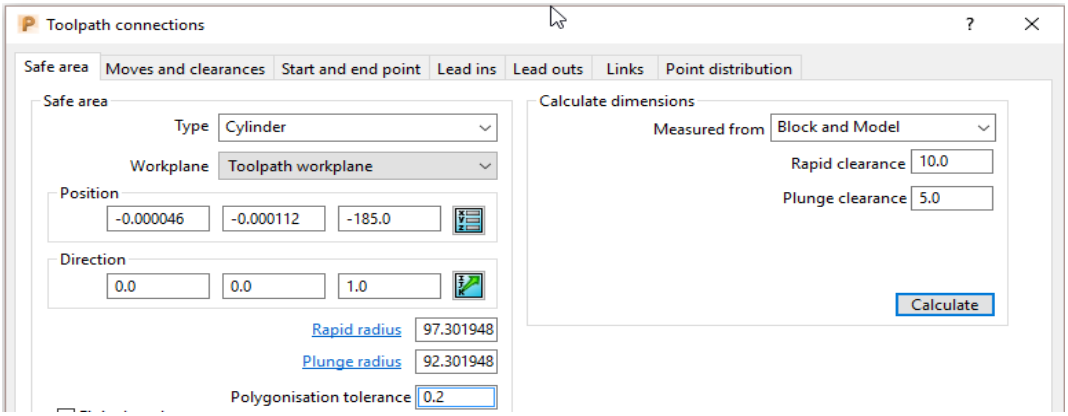
4 Make sure that no **Boundary** is **Active**.

5 Open the **Block** dialog and select the option **Defined by** - **Cylinder** before selecting **Calculate** (A cylindrical **Block** to the **Model** dimensions will be created).





6 Open the **Rapid Move Heights** dialog and set ***Safe area*** **Cylinder** and enter data into the dialog exactly as shown below, before selecting **Calculate**.



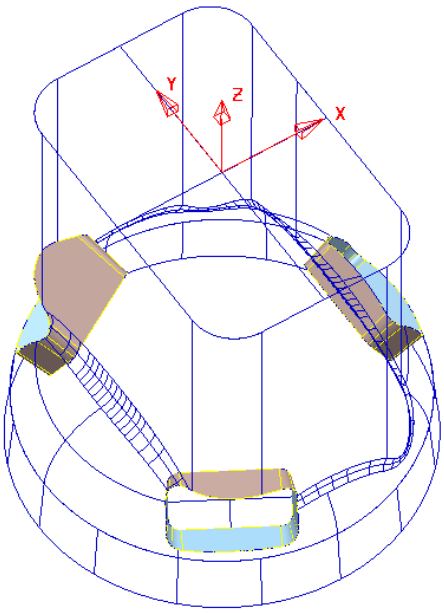
- Input the ***Workplane*** **ztop-175_A** and set a ***Direction*** (vector) **0 0 1**

The use of **Safe area - Cylinder** will produce smoother **link** moves around the cylindrical form instead of point to point, linear moves.

Copyright © Autodesk

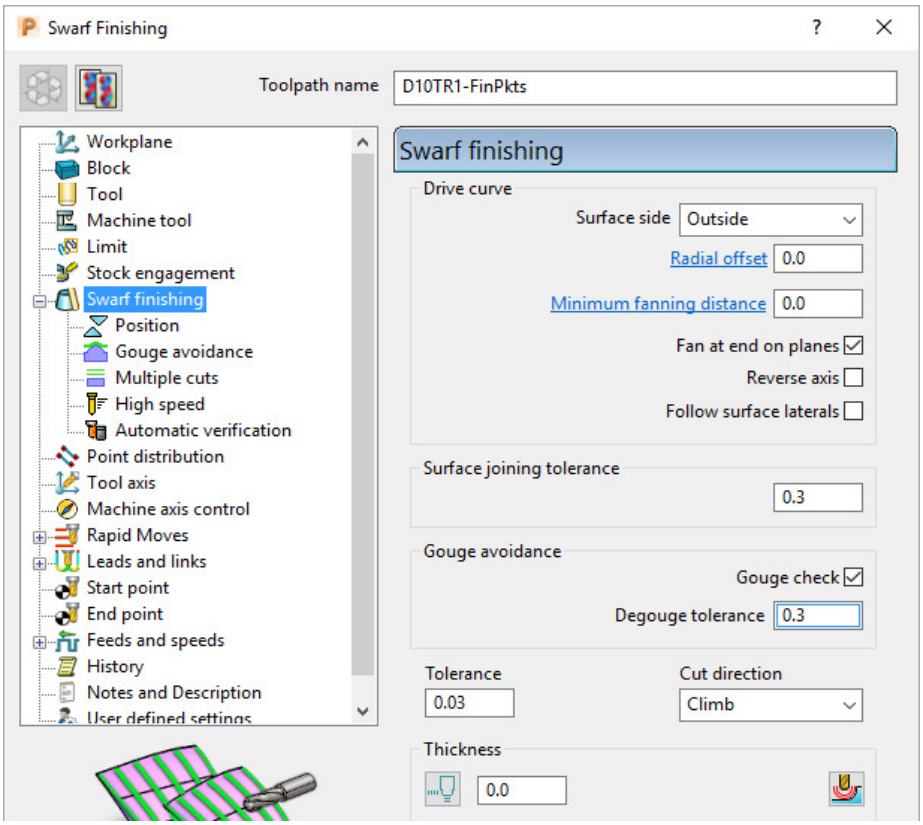
7 Set both **Lead in/out** to **Surface normal arc** with ***Angle* 90** and ***Radius* 4**.

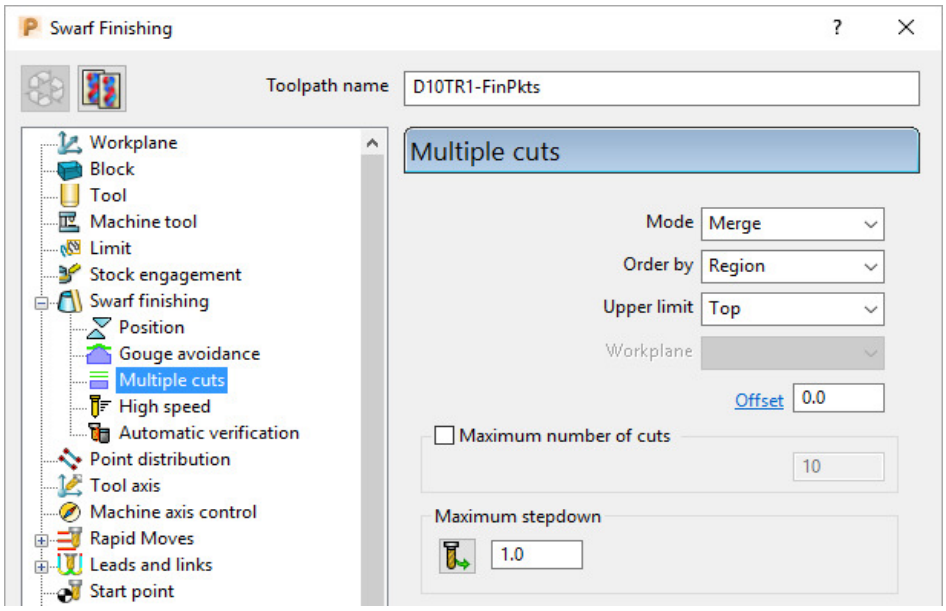
8 Select all three ***sidewall surfaces*** for the three pockets.



9 Select **Toolpath Strategies** and select the **Finishing** option located in the **Strategy Selector** dialog.

10 Enter the values into the pages of the **Swarf Finishing** dialog exactly as shown.

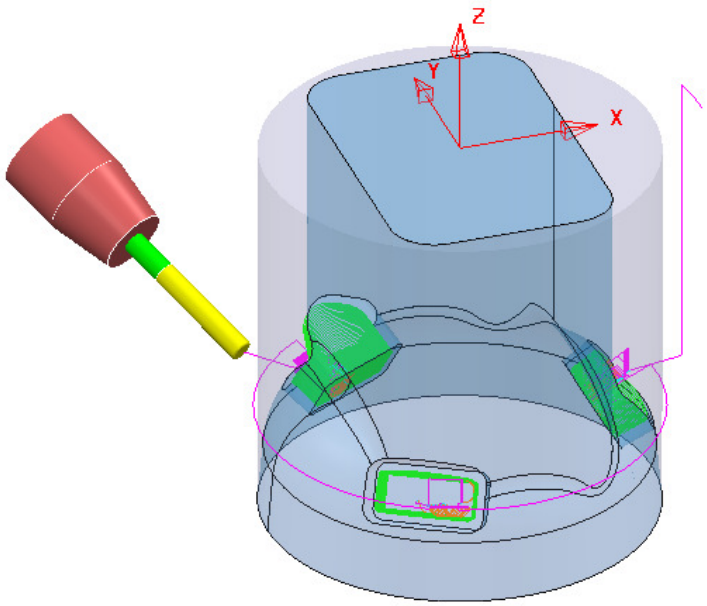




- **Mode** - **Merge** provides continuous tool tracks as variable offsets between the upper and lower edges of the swarfable, **surfaces**.
- Use: **Maximum stepdown 1.0**

*The default **Tool Axis** setting for **Swarf Finishing** is **Automatic**.*

11 Select **Calculate** to process the **Toolpath**.



Note the **link** moves follow a ***cylindrical path*** as defined in the **Rapid Move Heights** dialog. This will provide both smoother tool ***link*** and **re-orientation** moves between separate machining areas.

12 **Simulate** the toolpath (**D10TR1-FinPkts**) and observe the changing angle of the tool as it **Swarf** machines the selected surfaces.

13 Select **File** > **Save Project** to update:

….\COURSEWORK\PowerMill-Projects\3+2example

Copyright © Autodesk

Swarf Finishing - Example 3

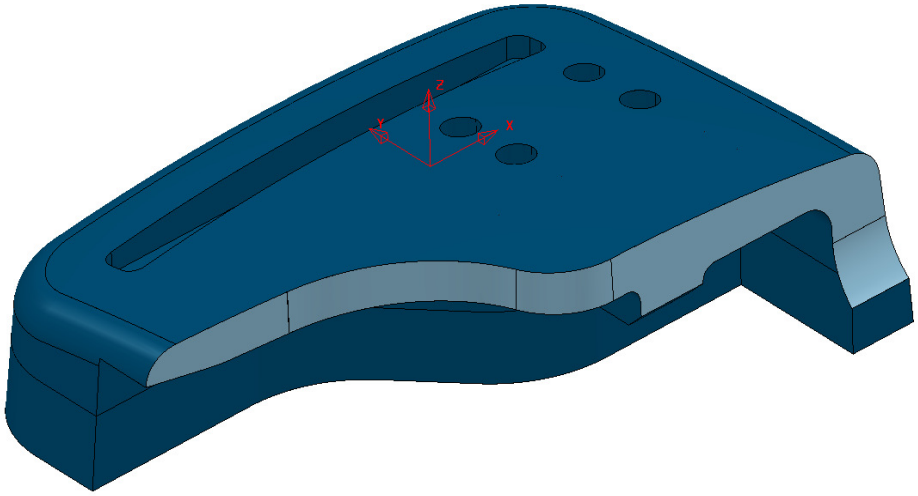
There are situations where ***swarfable surfaces*** do not provide an acceptable **tool alignment** around the component form. In these cases the only obvious solution is to use a separate CAD package (eg **PowerSHAPE**) to re-model the affected **surfaces**.

In the following example a **Swarf Finishing** strategy will be applied on 5 selected **surfaces** (As shown, shaded ***pale blue*** in the following illustration). Due to the way the **Surfaces** have been designed the resultant toolpath will not produce an acceptable tool track along the existing form. To fix this a new 'fit for purpose' ***surface model*** of the swarfable face will be ***imported*** and used to produce a much improved **Swarf Finishing** toolpath.

1 Select **File** > **Delete All** and **Tools** > **Reset forms**.

2 Select **File** > **Import Model**:

...\PowerMill_Data\five_axis\swarf_mc\SwarfSurfModel-A.dgk



3 Select **File** > **Save Project As**:

….\COURSEWORK\PowerMill-Projects\SwarfSurf-EX3

4 Activate the ***Workplane*** **TopCentre**.

5 **Calculate** a **Block**, ***Defined by*** – **Box** and using ***Type*** – **Model**.

6 Still in the **Block** dialog, **Lock** the **Z** values, input an **expansion 10**, and **Calculate** again,

7 Create a Dia 12 End Mill (named **EM12**).

8 **Calculate** the **Rapid Move Heights**.

9 Set the **Start Point** to **First Point Safe** and **End Point** to **Last Point Safe**.

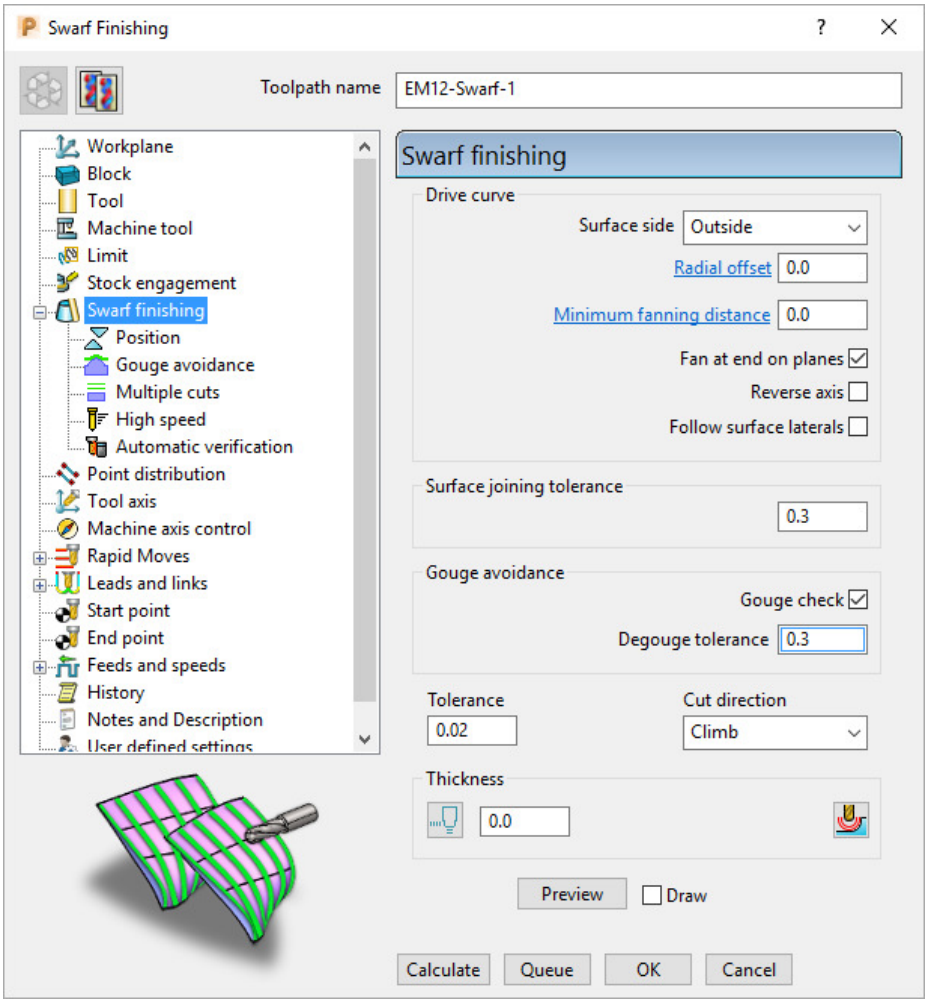
10 Set both the **Lead in** and **Lead out** to **Surface normal arc** with **Angle 90** and **radius 6**.

11 ***Select*** the 5 **Surfaces** required to be **Swarf** machined as shown ***pale blue*** in the above illustration.

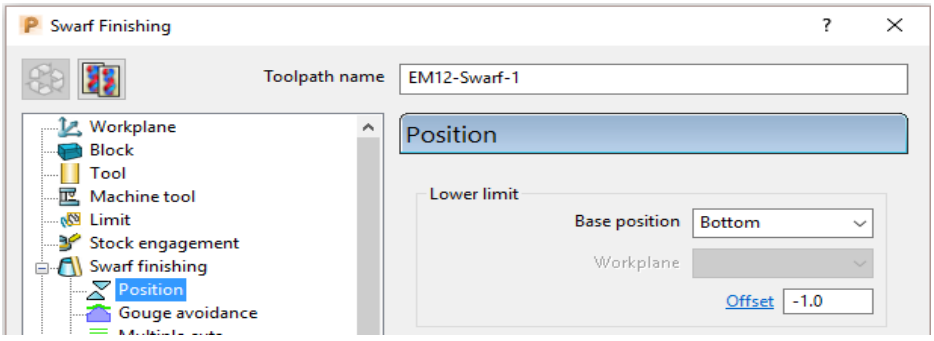
Copyright © Autodesk

12 Select **Toolpath Strategies** and select the **Finishing** option located in the **Strategy Selector** dialog.

13 Enter the values into the main page of the **Swarf Finishing** dialog exactly as shown.



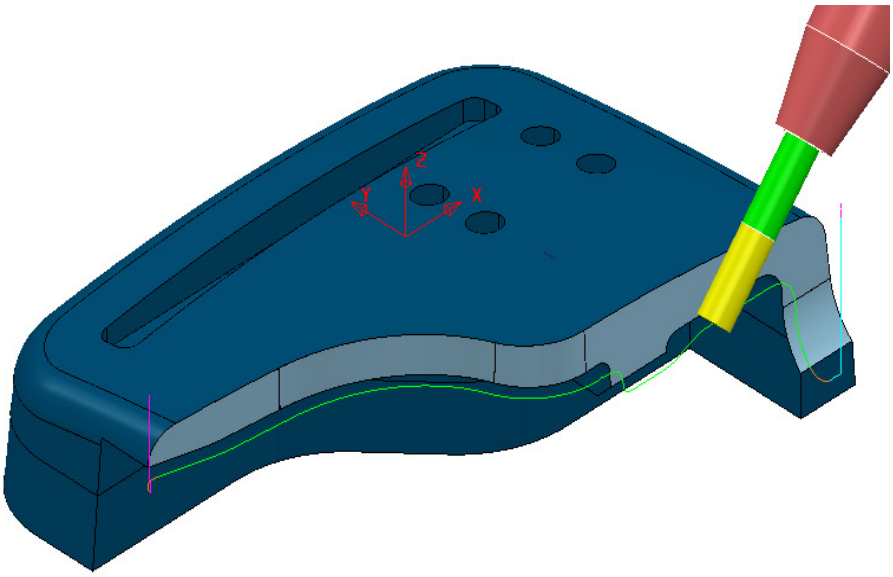
14 Select the **Position** page and input **Offset -1.0**



15 Select **Calculate** to process the toolpath.

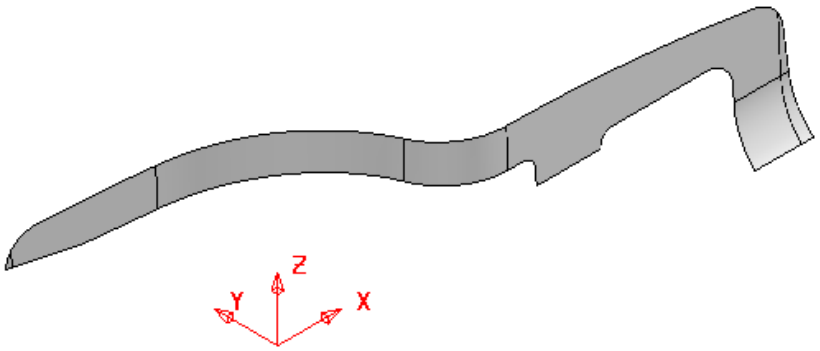
Copyright © Autodesk

16 Run a slow **Simulation** and observe the changes in tool alignment as it runs along the tool track.



The **Swarf Finishing** toolpath visually looks okay at first until it is run through a **simulation**. This reveals that the **Tool** (**EM12**) is running at an acute angle relative to the ***swarfable surfaces*** in some areas (as shown in the above illustration). Also, the cutter closely follows the lower edge, causing it to suddenly drop and lift while running around the step.

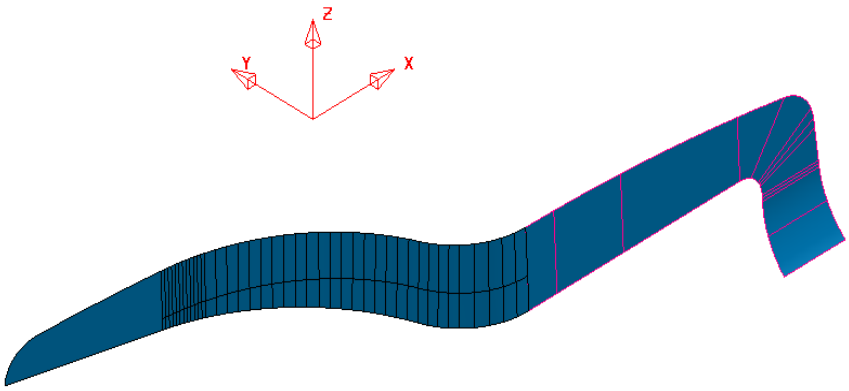
This strategy can be improved by using different **Surfaces** that are specifically designed to provide a better tool alignment during **Swarf Finishing**.



The original 5 ***reference*** **Surfaces** used in the ***toolpath*** **EM12-Swarf-1** are as shown above. These will be replaced by new **Surfaces** of exactly the same shape within the original model area, but designed specifically to work more effectively with the **Swarf Finishing** strategy.

17 Select **File** > **Import Model**:

....\PowerMill_Data\five_axis\Swarf_mc\NewSwarfSurf-B.dgk



The new **Surfaces** are designed with ***surface curves*** aligned to match the required **tool alignment**. It is also exactly the same shape as the ***5 surfaces*** used in the original **Swarf Finishing** toolpath. Compared with the original 5 surfaces, the lower edge has been extended with a smooth transition constructed along the lower edge of the form.

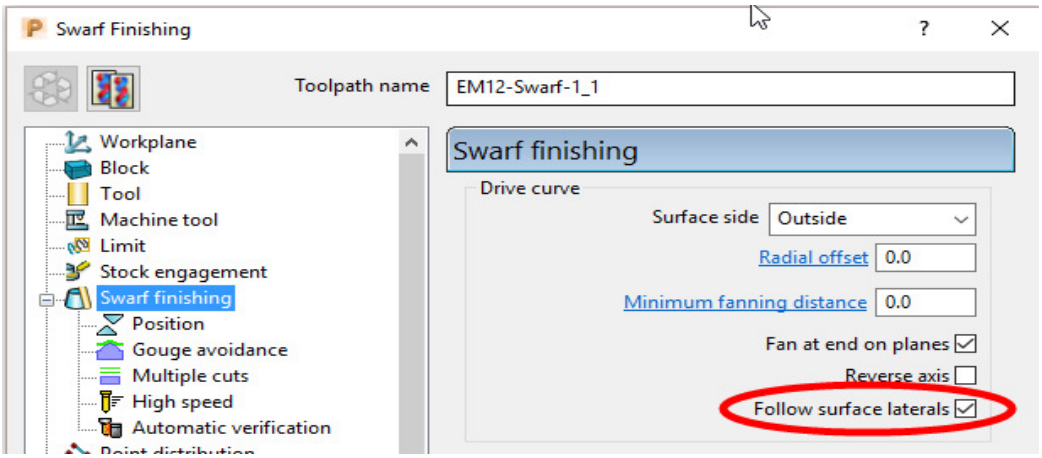
*Do Not delete the 5 surfaces used in the original **Swarf Finishing** strategy. They are still required for accurate **gouge** checking.*

18 Right mouse click on the toolpath **EM12-Swarf-1** and from the local menu select **Settings** to open the strategy dialog.

19 In the **Swarf Finishing** dialog select the option, make a **Copy** .

20 Select the new, imported reference surfaces.

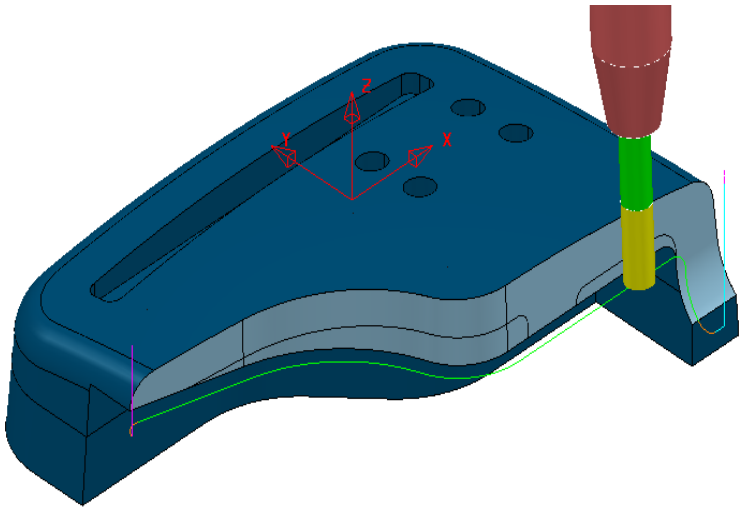
21 In the main ***Swarf finishing*** page, tick the box **Follow surface laterals** to force the tool align to the actual **surface curves**.



22 Select **Calculate** to process the new toolpath (**EM12-Swarf-1_1**).

23 Run a slow **Simulation** and observe the changes in tool alignment as it runs along the tool track.

Copyright © Autodesk



This time the **Tool** is more suitably aligned to the **swarfable surface** as it runs along the **Toolpath** (**EM12-Swarf-1_1**).

24 Select **File** > **Save Project** to update:

….\COURSEWORK\PowerMill-Projects\SwarfSurf-EX3

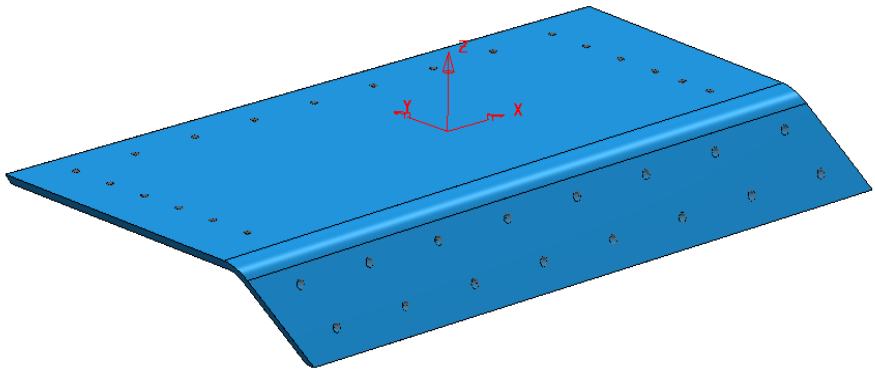
Swarf Finishing - Example 4

In the following example a **Swarf Finishing** strategy will be applied to trim the edges of an angled plate. Due to the way the original edge **Surfaces** have been designed the tool is unable to maintain a suitable alignment to the model form. To fix this a different set of swarfable edges will be used instead to produce a much improved **Swarf Finishing** toolpath. These surfaces have been created using **PowerSHAPE**.

1 Select **File** > **Delete All** and **Tools** > **Reset forms**.

2 Select **File** > **Open Project**:-

....\PowerMill_Data\five_axis\swarf_mc\AnglePlate4-Swarf-Start



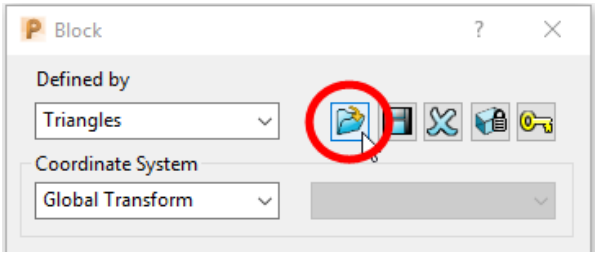
3 Select **File** > **Save Project As**:

….\COURSEWORK\PowerMill-Projects\ AnglePlate4-Swarf-EX1

4 Activate the existing drilling Toolpath **Drill20-Thro**.

5 From the **Main** toolbar select **Block**.

6 In the **Block** dialog select ***Defined by*** **Triangles** with the ***Coordinate System*** set to **Global Transform**.



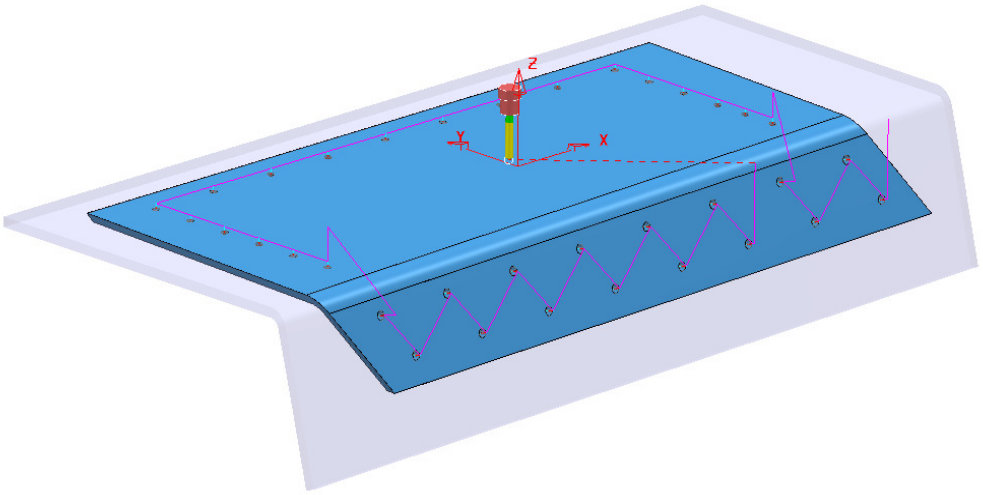
7 Select the **Load block from file** option.

8 In the ***Open Block from Triangles*** dialog and select the file:-

....\PowerMill_Data\five_axis\swarf_mc\AnglePlate4-Material.dmt

Copyright © Autodesk

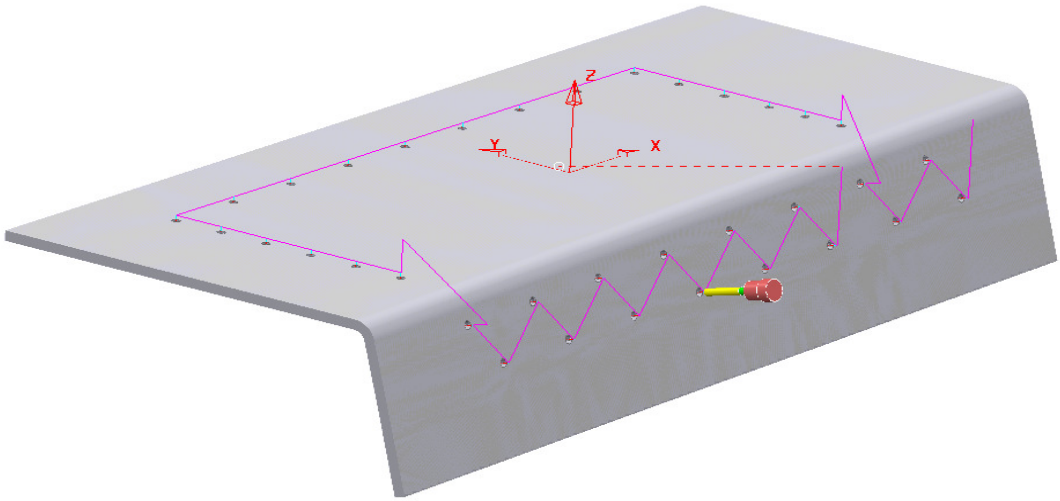
9 Select **Accept** to close the dialog.



10 Right mouse click on the toolpath **Drill20** and from the local menu select **Simulate from Start**.

11 Ensure that the **ViewMILL** toolbar is displayed and click the **Red** button (will change to **Green**) to connect it to the simulation.

12 Run a **ViewMILL** simulation on the 2 existing drilling toolpaths **Drill20-Thro** and **EM16-Dia24_Helical**.



13 Save the **ViewMILL** session at this stage, put it on suspend , and select the No image option.

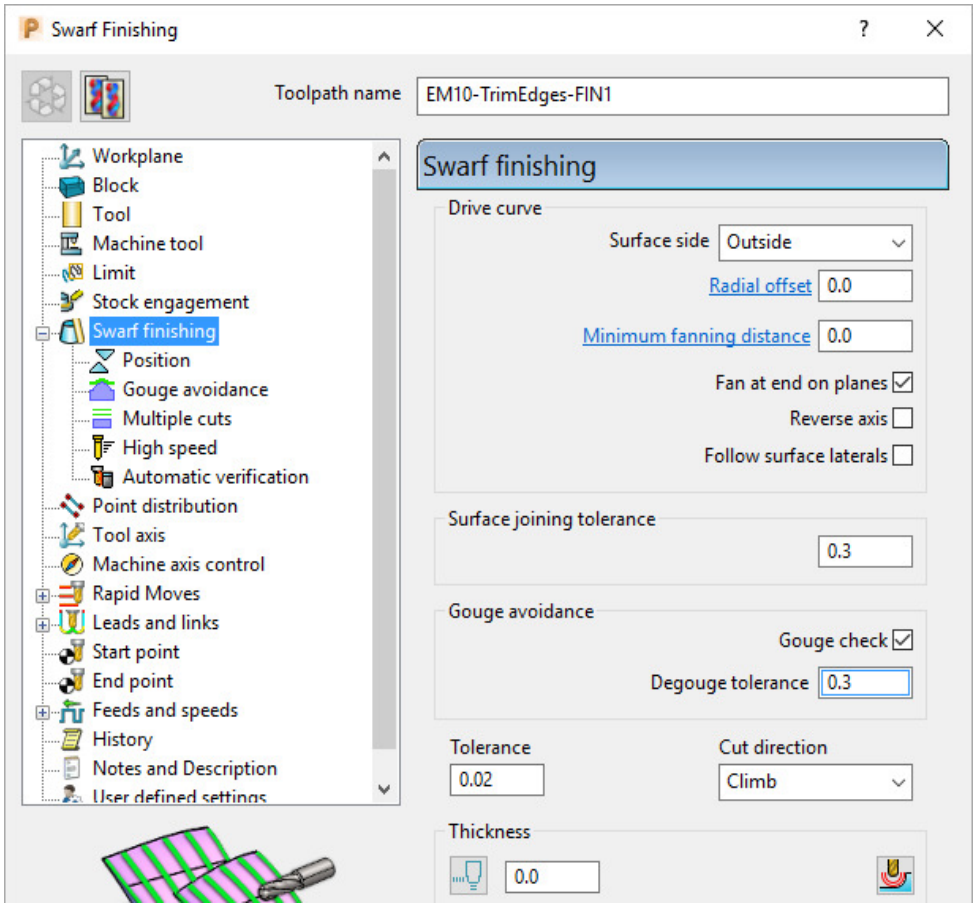
14 Activate the tool ***EM10***.

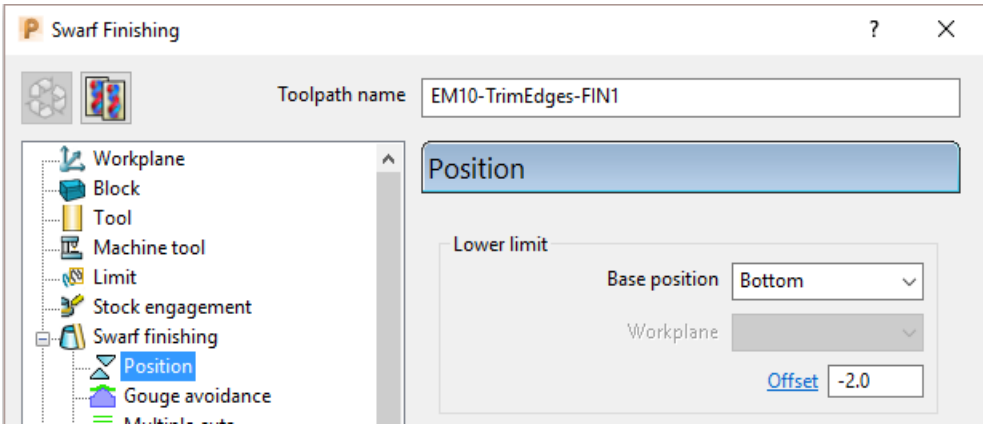
15 In the **Leads and Links** dialog set both **Lead in** and **Lead out** to **Vertical arc** with **Angle 90** and **Radius 5**.

16 In the **PowerMill *explorer***, **Levels and Sets** folder, right mouse click on the ***Set* OriginalEdges** and from the local menu click on **Select Surfaces**.

17 From **Toolpath Strategies** , select the **Finishing** option located in the **Strategy Selector** dialog.

18 Select **Swarf fininshing** and enter the values into the pages of the dialog exactly as shown below.

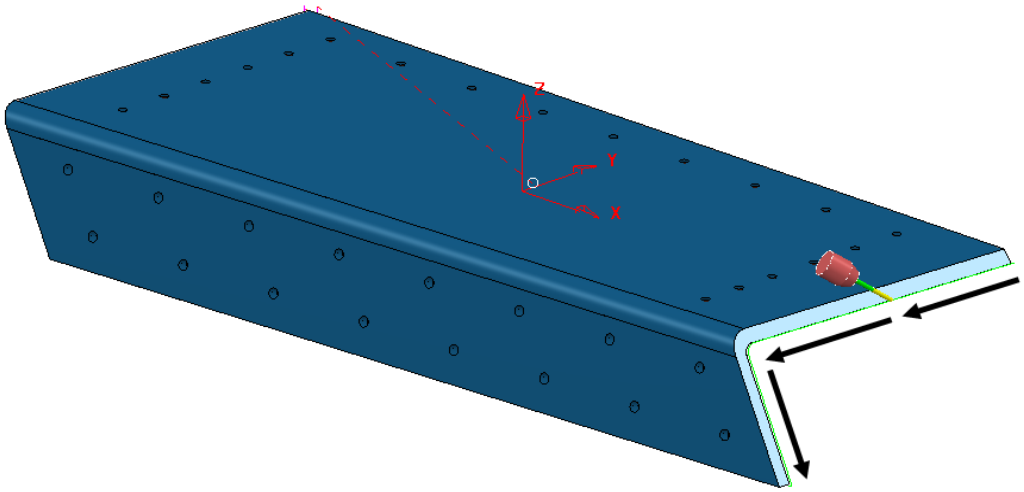




19 Select **Calculate** to process the toolpath and then **Close** the dialog.

Copyright © Autodesk

20 Run a toolpath **simulation** and observe the alignment of the tool as it tracks along the selected surfaces.

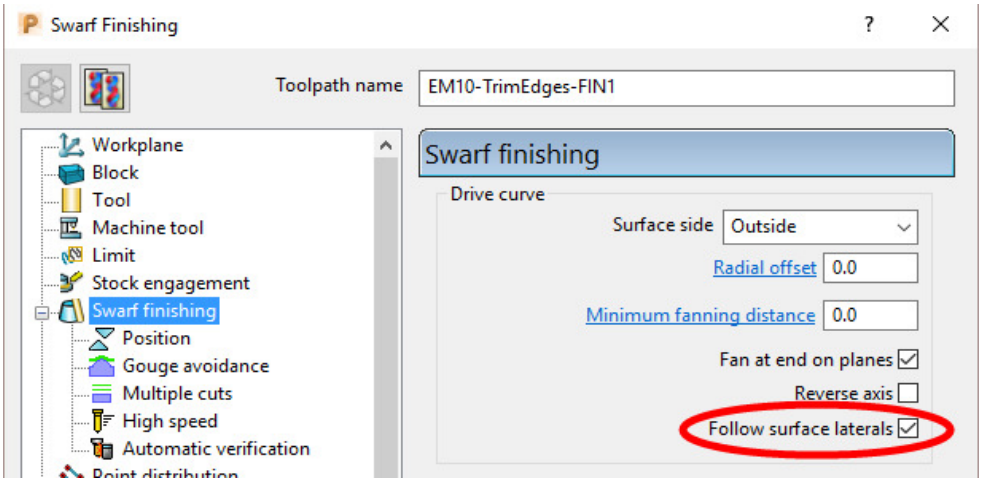


Due to the component edges being planer the alignment of the **EM12** tool is not compelled to remain ***Normal*** to the main ***Angle Plate*** surfaces. In the above image the tool has remained cranked over after tracking around a curve in the edge form. There are too many choices for the cutter to use a linear alignment across the swarfable surfaces.

21 Right mouse click on the toolpath and from the local menu select **Settings**.

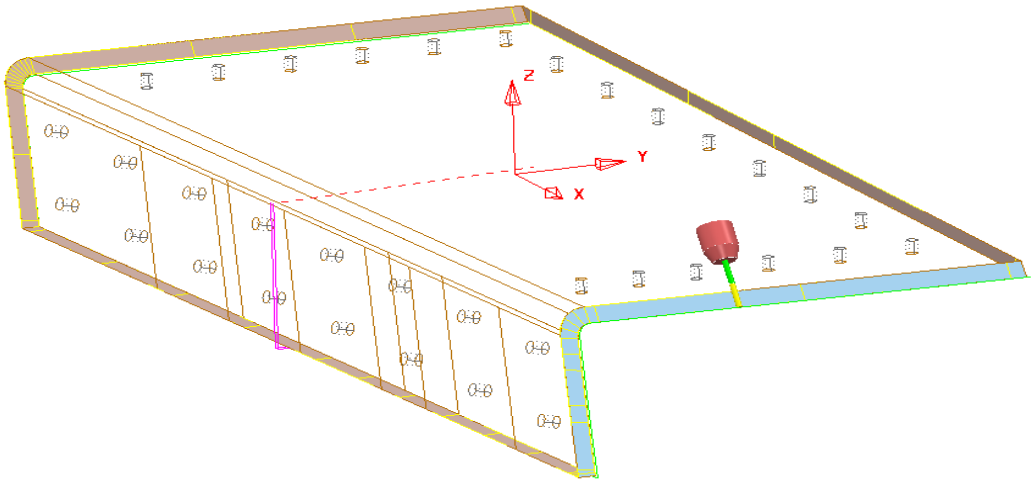
22 Select the option to **Recycle** the toolpath.

23 In the **Swarf finishing** page tick the box Follow surface laterals.



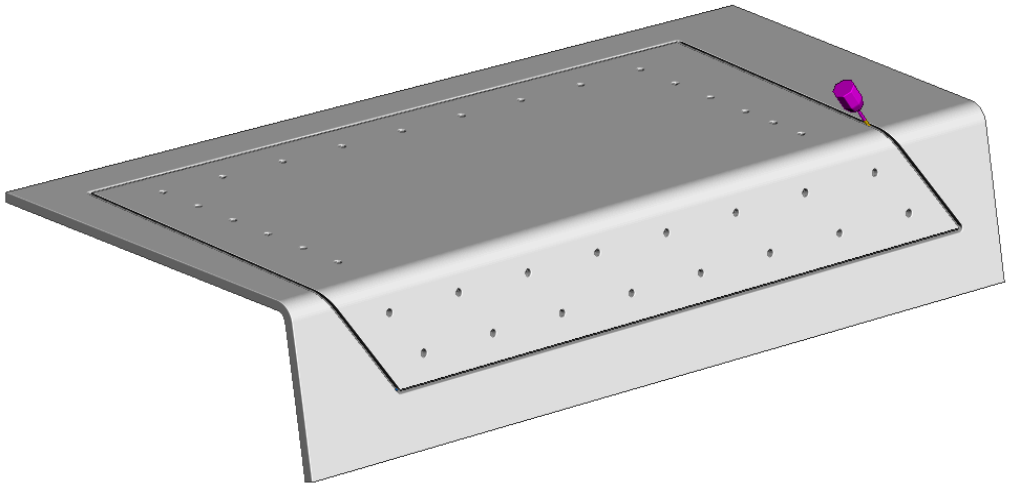
24 In the **PowerMill *explorer***, **Levels and Sets** folder, right mouse click on the ***Level*** **NewEdges** and from the local menu click on **Select Surfaces**.

25 **Calculate** the updated **Swarf finishing** strategy.



This time the tool is aligned to the ***laterals*** of the replacement swarfable surfaces. These have been re-designed using **PowerSHAPE** surface modelling to produce the above tool alignment.

26 Reconnect the existing **ViewMILL** session and simulate the latest **Swarf finishing** strategy.



27 Select **File** - **Save Project**.

Copyright © Autodesk

Wireframe Swarf Finishing

In some cases imported **surfaces** to be used in **Swarf Machining** could be of a poor quality. This would typically include such problems as ***mis-matching edges*** and/or ***un-intentional curves*** across the **surfaces**. These problems will undoubtedly result in part or complete failure to create a **Swarf Finishing** strategy.

One solution is to re-model the affected **Surfaces** (assuming the user has access to a suitable CAD system (eg **PowerSHAPE**)).

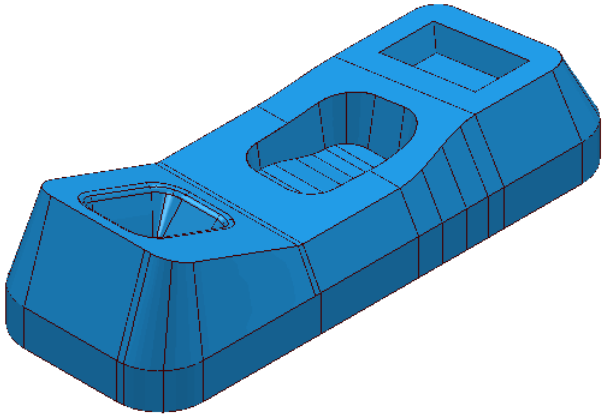
The other alternative is to create 2 separate **Patterns** that define both edges running along the area to be ***swarf machined***. The user can then use the **Wireframe Swarf Finishing** strategy.

The **model** used in the following example contains a pocket in which part of the sidewall is ***un-swarfable*** due to being slightly convex between the top and bottom edges.

1 Select **File** > **Delete All** and **Tools** > **Reset Forms**.

2 **Import** the **Model**:

….\PowerMill_Data\five_axis\swarf_mc\Wfrm-Swarf.dgk



3 Select **File** > **Save Project As**:

....\COURSEWORK\PowerMill-Projects\Wframe-Swarf

4 Activate the **Workplane** named **Datum**.

5 **Calculate** a **Block** using ***Defined by*** - **Box** and **Type** - **Model**.

6 Create a *Dia* 5 *End Mill* (**EM5**) using the following dimensions:

Tool ***Dia*** **5** ***Length*** **35**

Shank ***Lower Dia*** **5** ***Upper Dia*** **5** ***Length*** **25**

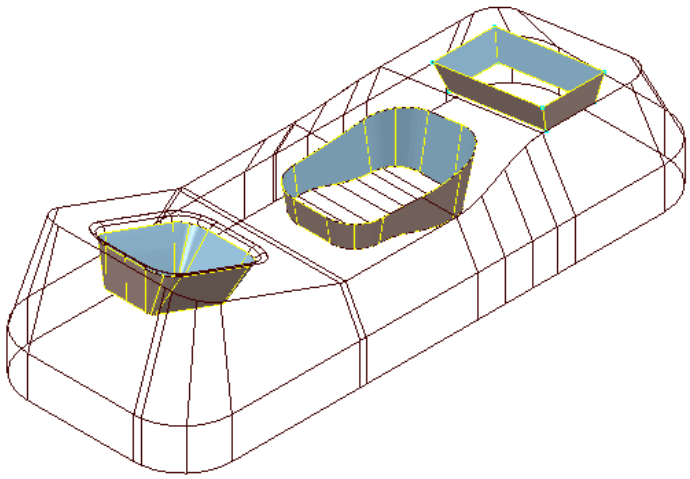
Holder-1 ***Lower Dia*** **15** ***Upper Dia*** **25** ***Length*** **15**

Holder-2 ***Lower Dia 25*** ***Upper Dia*** **25** ***Length*** **15**

Overhang 50

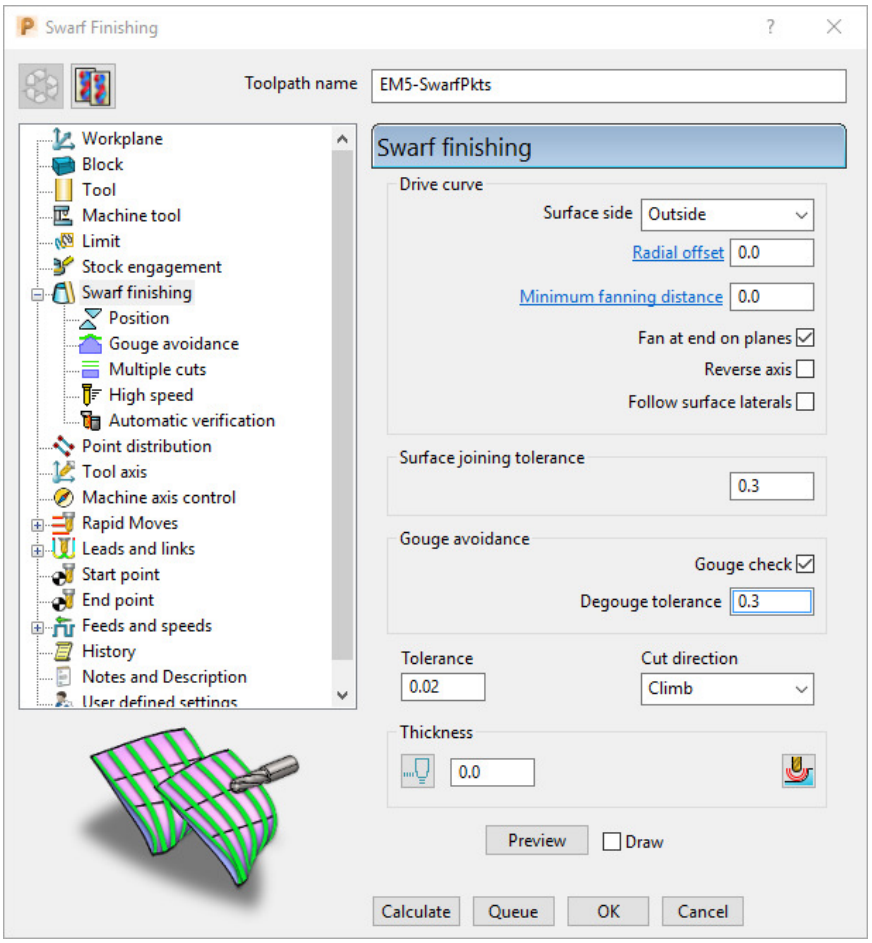
7 **Calculate** the **Rapid Move Heights**.

8 Select the ***sidewall surfaces*** on each of the 3 pockets in the component.

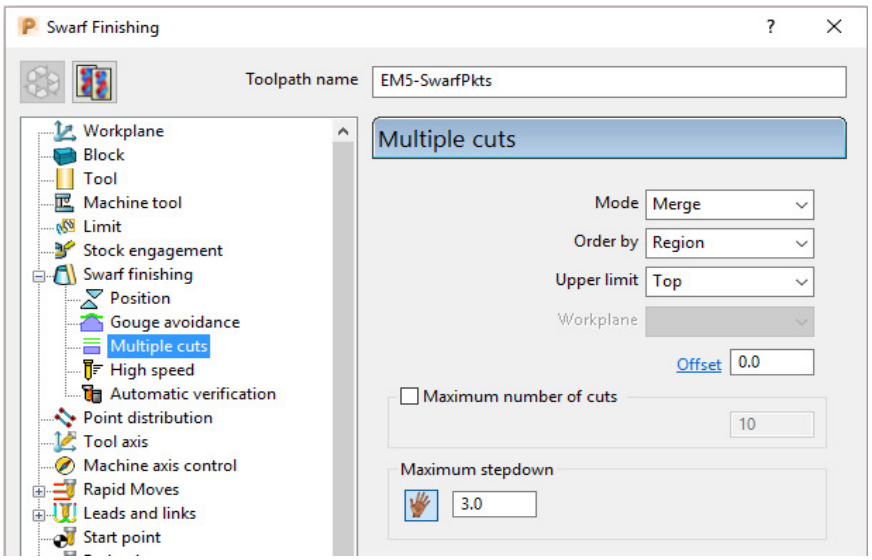


9 Select **Toolpath strategies** and from the **Finishing** tab select the **Swarf Finishing** option.

10 Enter the values into the 2 pages of the **Swarf Finishing** dialog exactly as shown.

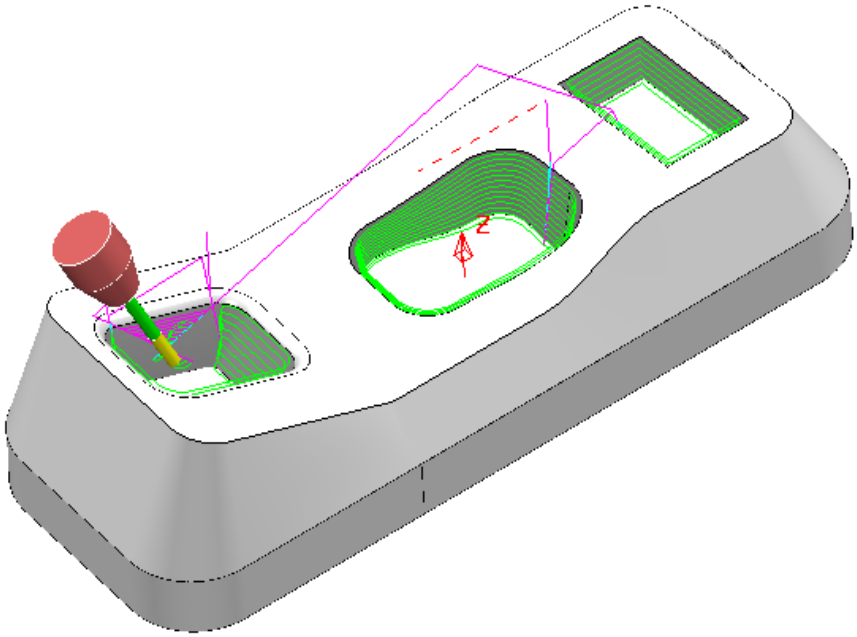


Copyright © Autodesk

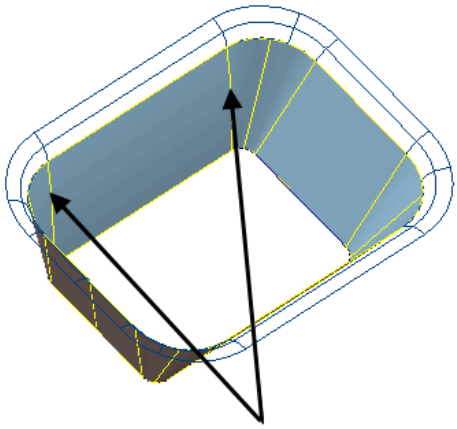


- ***Mode*** – **Merge** and ***Max Stepdown -3.0***

11 **Calculate** the toolpath.



The **Swarf Finishing** strategy has failed to machine one of the ***selected surfaces*** due to part of the sidewall being un-swarfable.



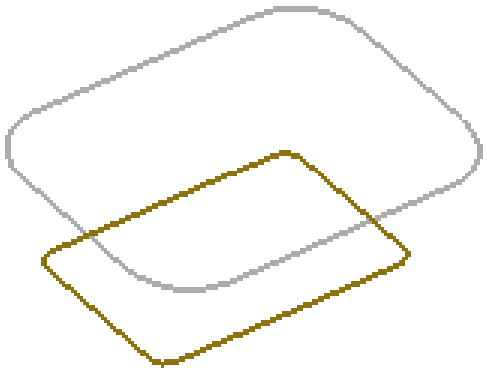
On closer inspection it is evident that some of the curves linking across the ***sidewall surface*** are convex and as a result make this area un-swarfable.

This problem will be fixed by creating separate **Patterns** along the upper and lower edges of the ***sidewall surface*** of the affected pocket. The 2 **Patterns** will then be used in a **Wireframe Swarf Finishing** strategy.

12 Select the **Surface** defining the angled sidewall of the left hand pocket (shown above).

13 Create a **Pattern** named **Upper** and right-click over it in the **PowerMill *explorer*** selecting **Insert > Model** from the local menu.

14 Select the lower **Pattern** segment.



15 Right-click on the ***Pattern*** named **Upper** and from the local menu, select the option **Edit - Copy Pattern (selected only)**.

16 ***Rename*** the new, copied **Pattern** as **Lower**.

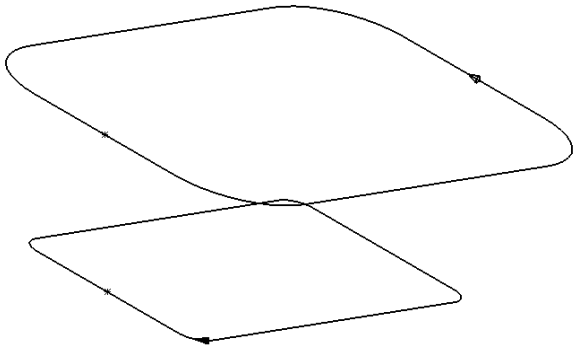
17 Select and **Delete** the ***lower segment*** from the **Pattern** named **Upper**.

For Wireframe Swarf Finishing to function correctly the two Patterns must travel in the same direction.

*It is not compulsory to align the **start points** on the upper and lower **Patterns**. It is the **first point** on the lower **Pattern** that defines the start position of the resultant toolpath.*

Copyright © Autodesk

18 To identify whether the two **Patterns** are suitable for use, right click each one in turn and select **Instrument** to display the start point and direction.



As can be clearly seen the two segments travel in opposite directions creating the need for one to be reversed.

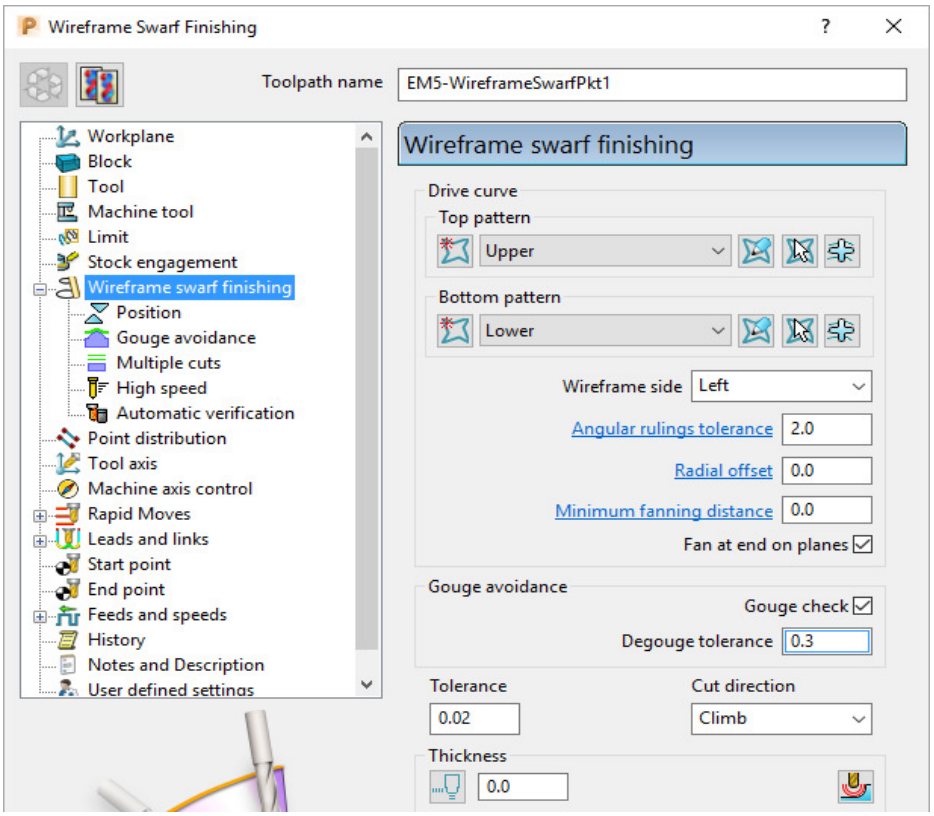
*In this case, to create a **Climb** milling strategy, the lower **segment** must be reversed.*

19 Select the lower **Pattern** segment and right mouse click on it to open the local menu.

20 From the local menu, select **Edit** > **Reverse Selected**.

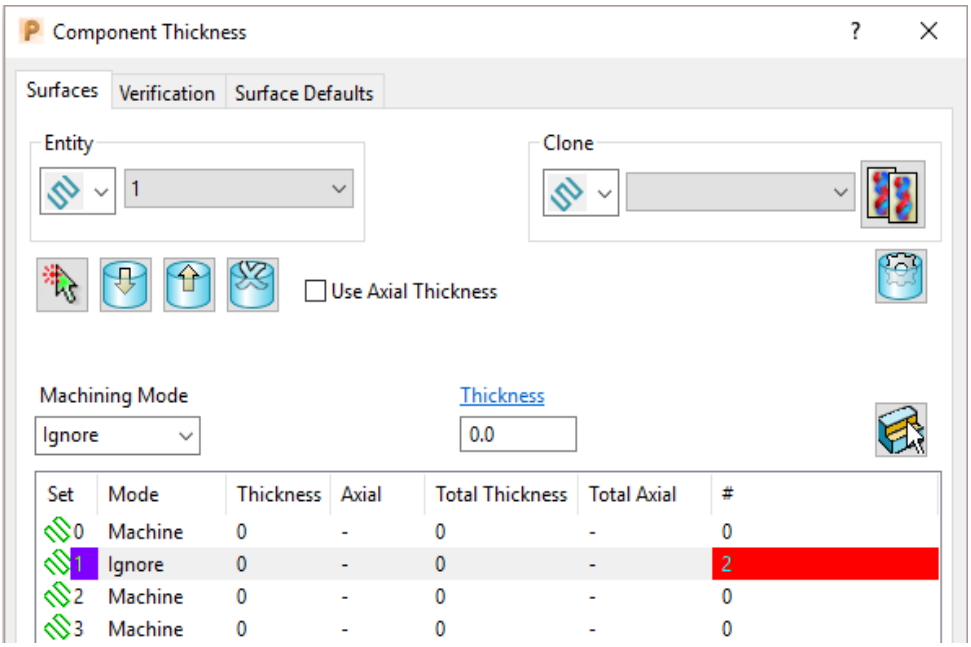
21 Select the **Toolpath Strategies** and from the **Strategy Selector** > **Finishing** options, select **Wireframe Swarf Finishing**.

22 Enter the values into the dialog exactly as shown below:



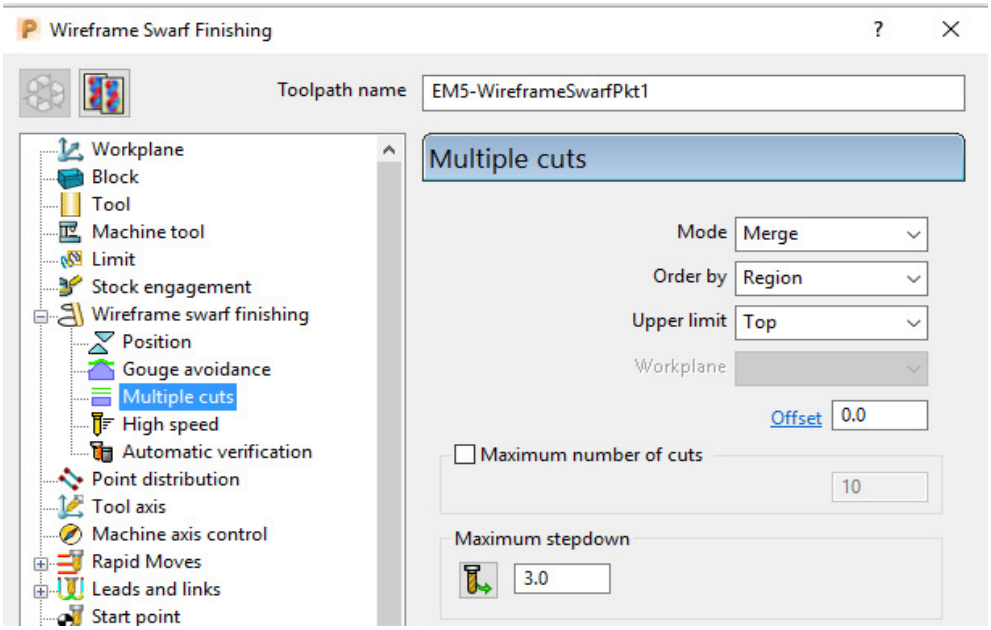
23 Select **Component Thickness** to open the following dialog.

24 Select both the sidewall and fillet surface (flanked by the **Patterns**) and acquire them to a selected row (with the **Machining Mode** set to **Ignore**) in the **Component Thickness** dialog.

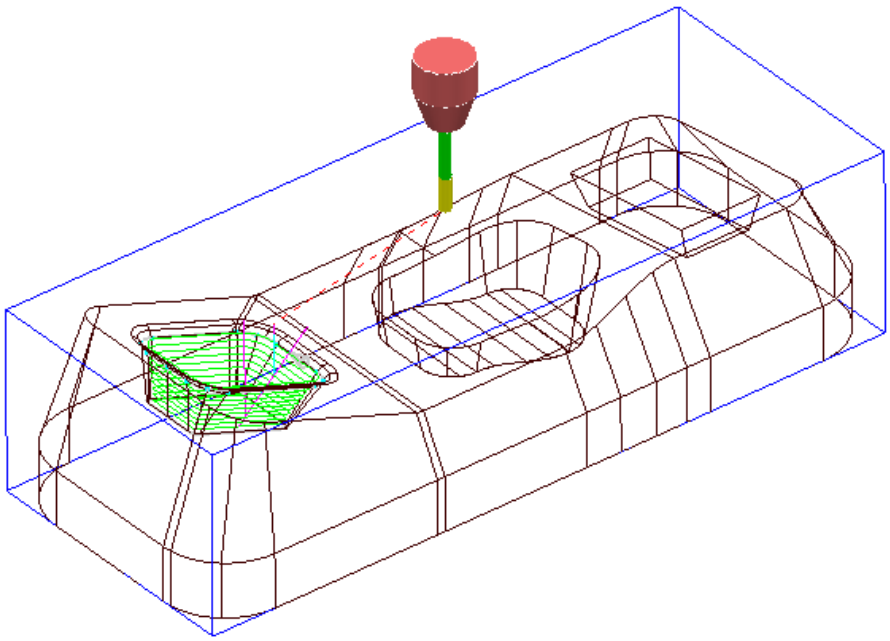


25 **Apply** the **Component Thickness** dialog.

Copyright © Autodesk



26 **Calculate** the **Wireframe Swarf Finishing** strategy.



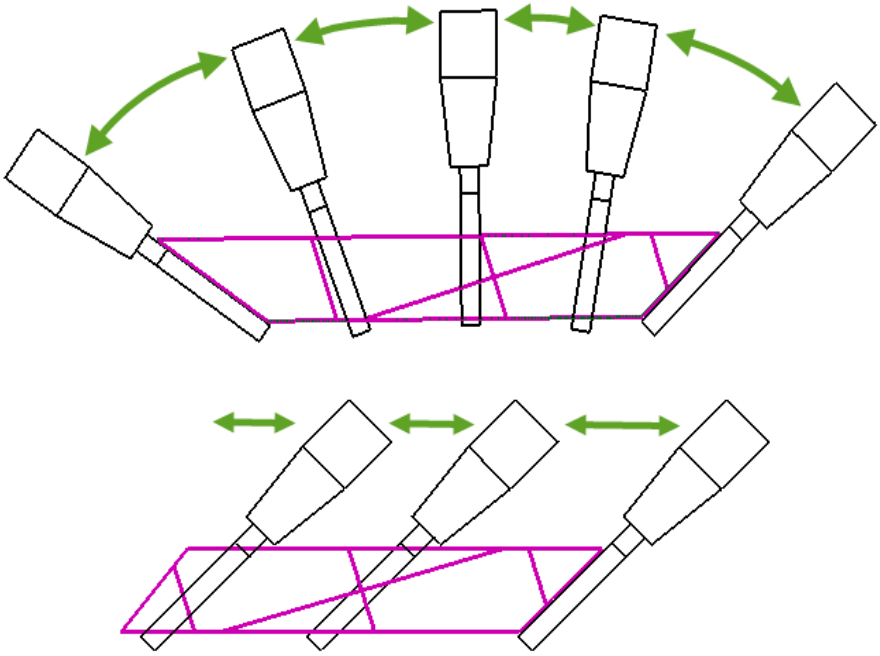
The new **Wireframe swarf finishing** strategy has successfully been applied between the ***Upper*** and ***Lower*** **Patterns**.

27 Select **File** > **Save Project** to update the **Project**:

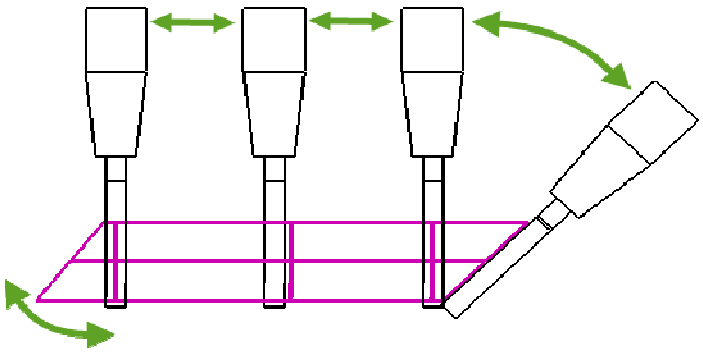
….\COURSEWORK\PowerMill-Projects\Wframe-Swarf

Swarfable Surface Design

When designing a component certain factors will dictate how successfully a **Swarf finishing** strategy can be applied to what appear to be visibly suitable **surfaces**. The fundamental rule is that **Swarf finishing** will only occur where the side of the tool can be in full contact from one side of a surface to the other (Linear). This works fine if the swarfable surfaces are twisting and turning and the tool is able to find a specific linear alignment at all times. If however a **surface** is totally planer then the linear tool alignment can occur at any angle across the swarfable edges.



A ***tick box*** option exists in the strategy to force the tool to **Follow surface laterals** (natural surface curves). For this to work it is likely that the existing planer swarfable surfaces will need to be re-modelled. This task has to be done using a suitable CAD software such as **PowerSHAPE**.



The above planer **surface** has been designed with the **laterals** aligned to match the required tool axis. The initial angular move is the required transition to or from the end of another selected swarfable **surface**.

Copyright © Autodesk

7. Flowline Finishing

Flowline Finishing

Flowline Finishing is designed to machine multiple surfaces. The strategy uses 2 ***drive curves*** and either 1 (closed form) or 2 (open form) ***end curves*** to define a virtual surface area. Intermediate drive curves can be added to further control the flowline shape. This strategy provides evenly spaced tool passes, irrespective of the underlying geometry.

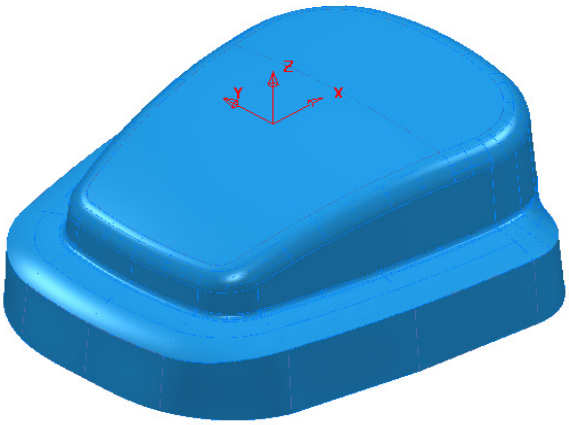
Advantages over existing toolpaths include:

- Multiple surfaces can be machined.
- Machining of undercut regions is possible.
- The toolpath passes are not determined by the surface parameterisation.
- The region being machined is not forced to be an entire surface or parametric box.
- The stepover is defined in 3D, giving total control over the maximum value.

Flowline Finishing Example

1. Select **File** > **Delete All** followed by **Tools** > **Reset Forms**.
2. **Open** the **Project**:

   ....\PowerMill_Data\five_axis\Flowline\Flowline_EX4-Start



3. Select **File** > **Save Project As**:

   ….\COURSEWORK\PowerMill-Projects\Flowline-EX4

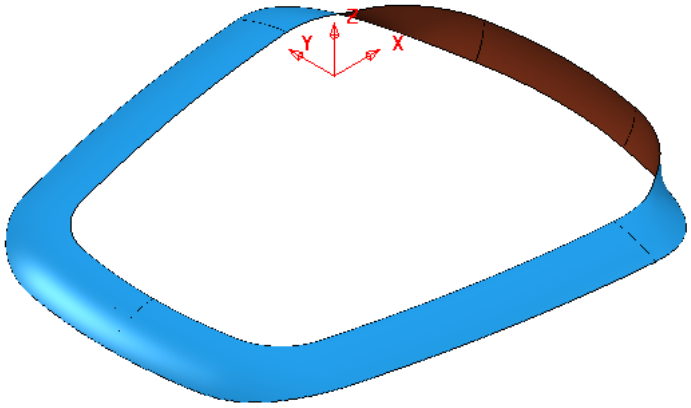
4 Activate the ***Workplane*** **MC-Datum**.

5 Activate the **BN12** tool.

6 Calculate a **Block** Using ***Defined by*** **Box** and ***Type*** **Model**.

7 Calculate the **Rapid Move Heights** using the default settings.

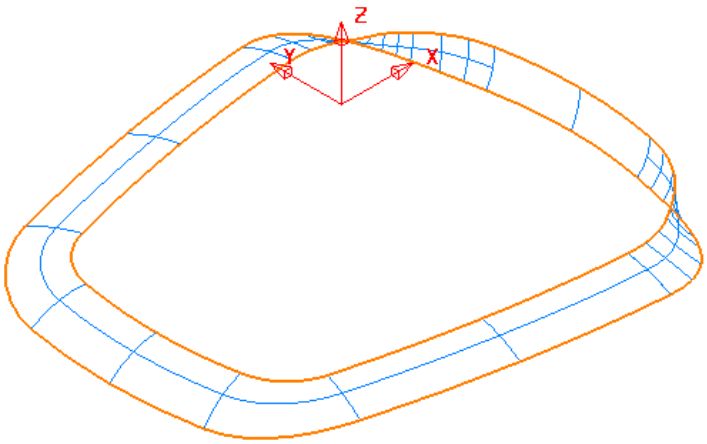
8 Select the following **5** surfaces and **Blank** the rest (**Ctrl K**).



9 Display the **Model** as **Wireframe** only.

10 Create a new (empty) ***Pattern*** **1**.

11 With the above **Surfaces** selected, Right click on the **Pattern** and from the local menu select **Insert** – **Model**.



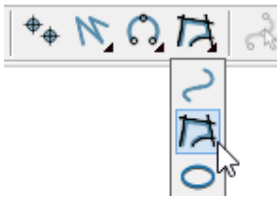
A Pattern extracted from a selection of surfaces will automatically become Embedded (Inherits the surface normals etc).

The Pattern symbol changes to identify it as being Embedded.

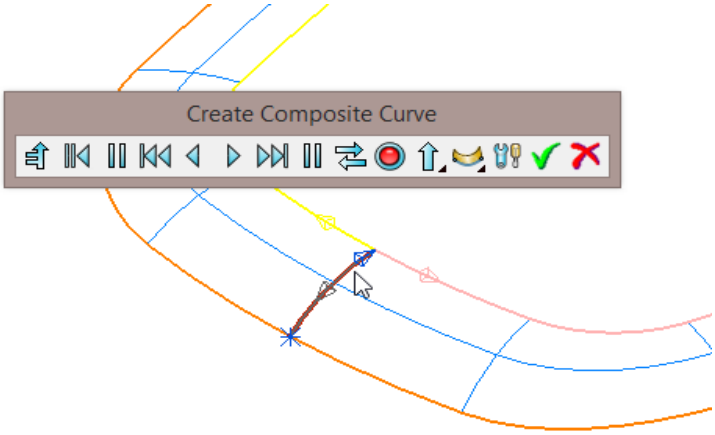
Copyright © Autodesk

12 Right click on the **Pattern** and from the local menu select **Curve Editor**.

13 Select the **Composite Curve** option from the **Curve Editor** toolbar.

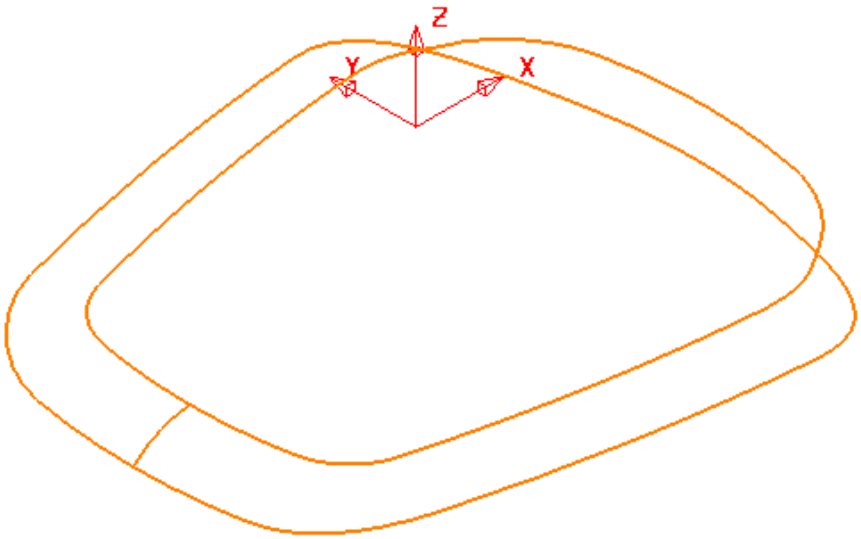


14 With the **Create Composite Curve** toolbar open click near to the upper end of the first surface lateral (As shown arrowed below).



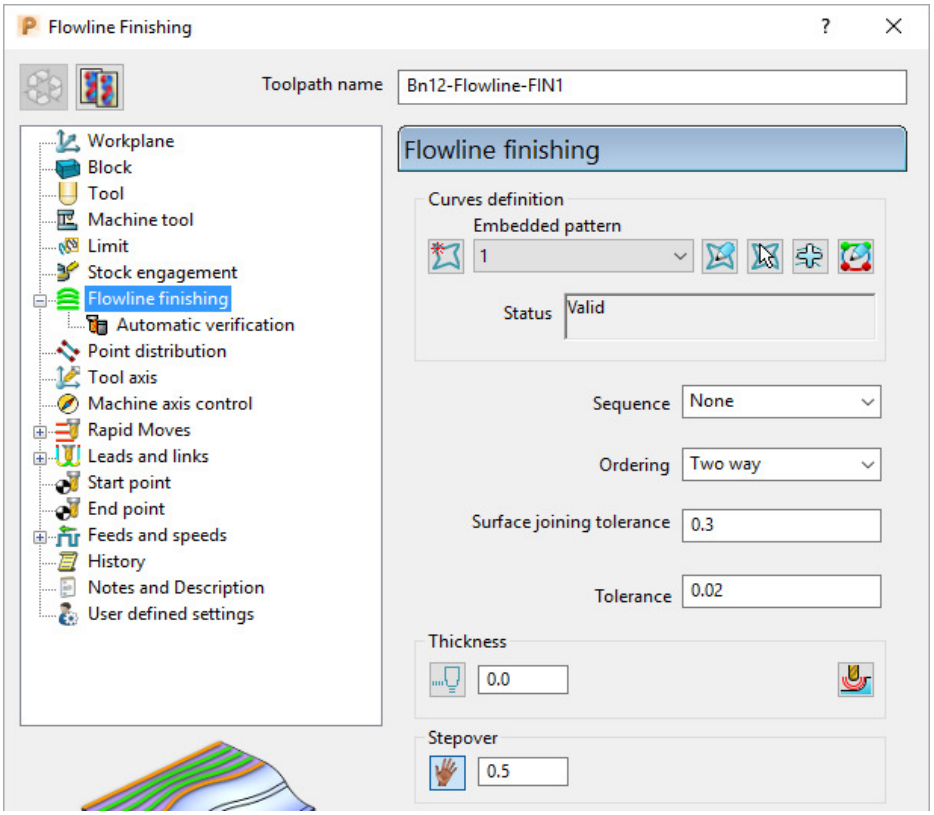
15 In the toolbar select the **red disc** followed by the **green tick** to both ***accept the new segment*** and ***close the toolbar***.

The new **Embedded Pattern** (**1**) is now made up of 3 segments (2 closed and 1 open).

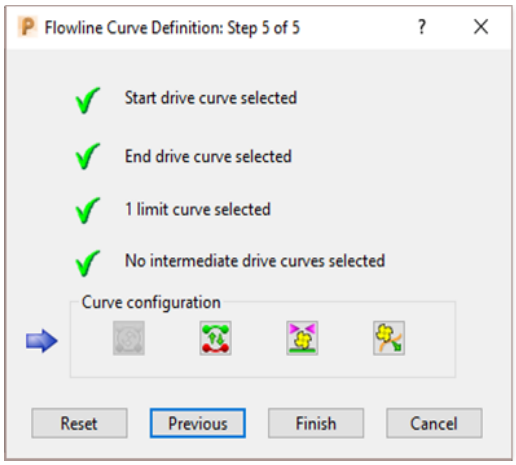


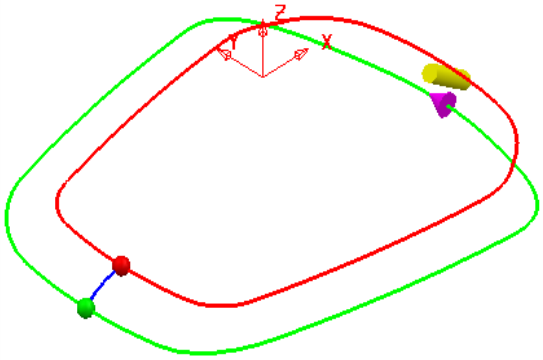
16 From **Toolpath Strategies** open the **Finishing** options and from the dialog select **Flowline Finishing**.

17 Enter data into the **main** page exactly as shown below.



18 Still on the **main** page select the **Interactively select pattern segments for the strategy** options.

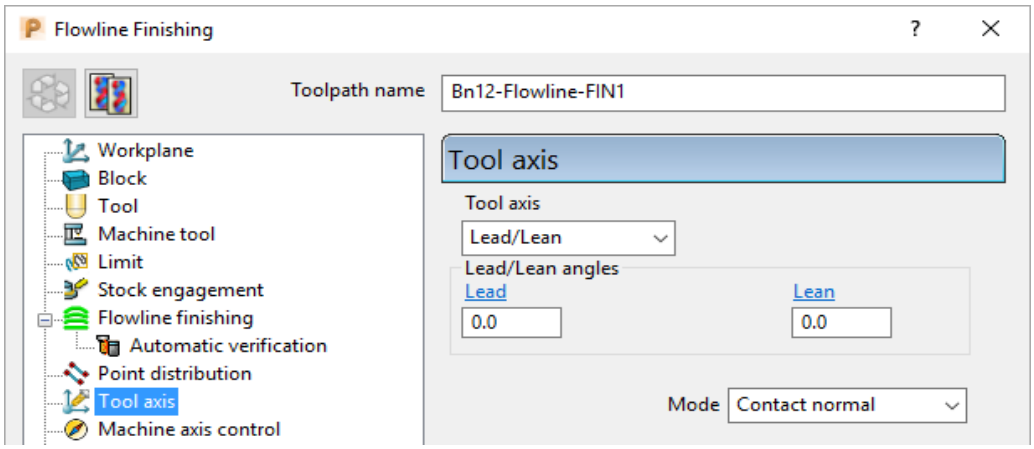




The options in the above form enable the toolpath ordering, directions, etc to be controlled before the toolpath is calculated. However, no changes are required in this instance.

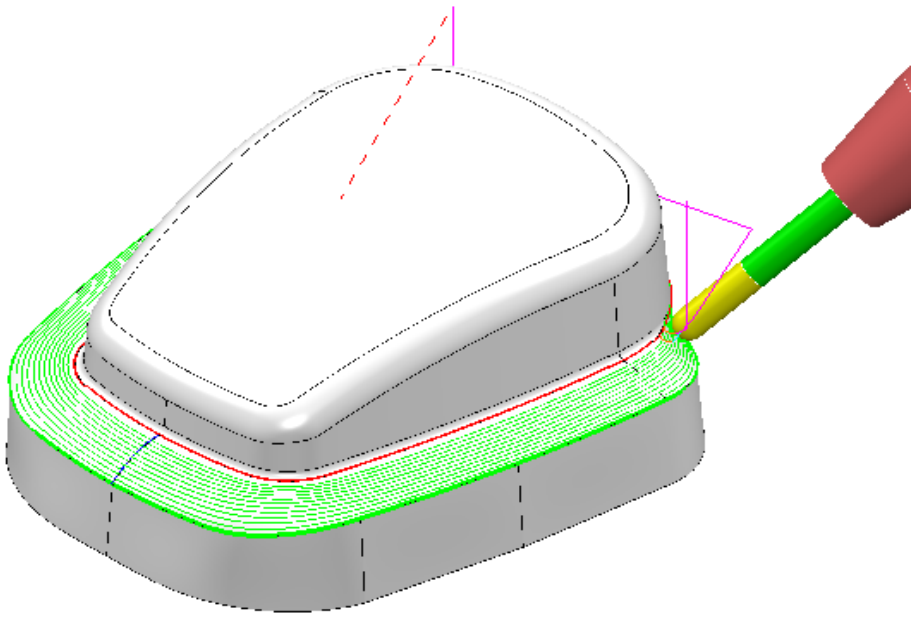
Copyright © Autodesk

19 Select the **Tool axis** page and select the **Lead/Lean** option with both the **Lead** and **Lean** angles as **0**.



20 Select **Calculate** to create the following **Flowline Finishing** toolpath.

*If the **Flowline Finishing** toolpath should fail to calculate try to **Embed** the **Pattern** again using **Closest point** with **Distance 0.1** and **Tolerance 0.01***



For visual reasons the above toolpath has been created with a 2mm stepover.

21 Right mouse click on the ***toolpath*** **BN12-Flowline-FIN1** and from the local menu select **Simulate from Start**.

22 From the **Simulation** toolbar select ***Play*** and observe the **Flowline Finishing** process.

23 Select **File** - **Save Project**.

24 Select **File** - **Delete All** and **Tools** - **Reset Forms**.

Copyright © Autodesk

Exercise

The purpose of this exercise is to create a suitable **Embedded Pattern** and use ***Five Axis*** - **Flowline Finishing** to machine the ***fillet surfaces*** running around the upper edge of a component model form.

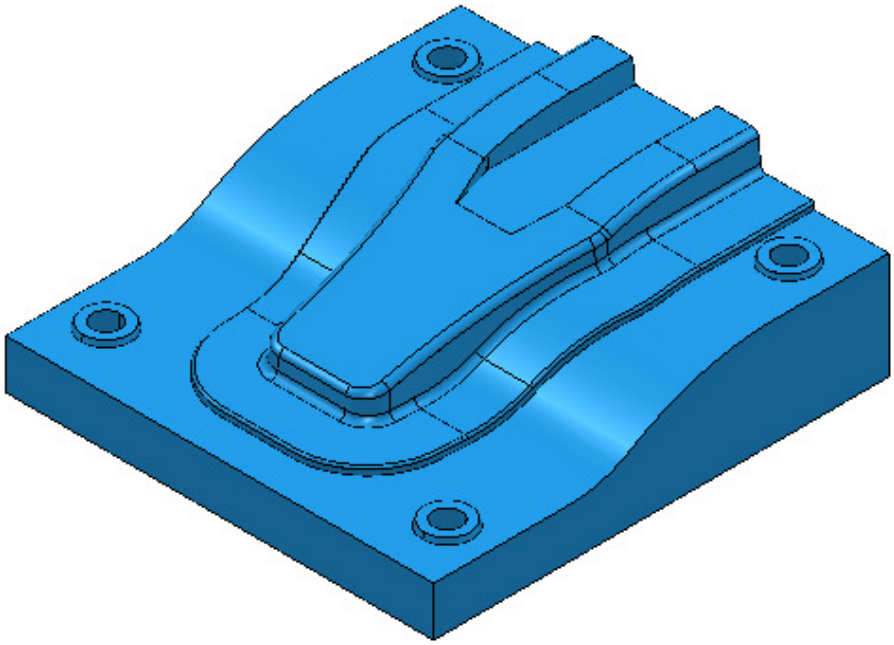
a A single segment, **Embedded Pattern** will be directly created along the edges of the ***fillet surfaces***.

b The **Embedded Pattern** will then be split into 4 suitable segments using the **Curve Editor** options.

c The user will then be asked to create a suitable **Flowline Finishing** strategy using the **Embedded Pattern**.

1 **Import** the **Model**:-

....\PowerMill_Data\Models\Vortex2.dgk



2 Select **File** > **Save Project As**:

...\COURSEWORK\PowerMill-Projects\FlowLineFillet-EX1

3 Select the whole of the **Model** and the select **Create and Orientate Workplane** – **Workplane** at **Top of Selection**.

4 Activate the new **Workplane**.

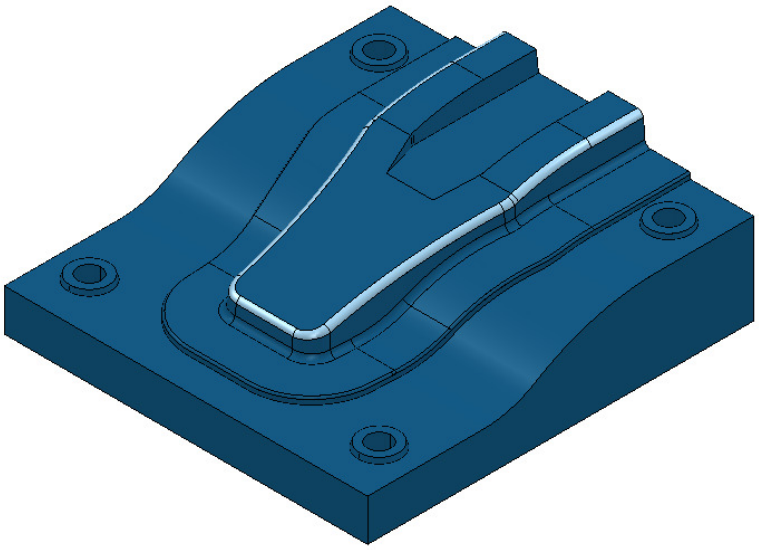
5 Create a *Dia* 8 - *Ball Nose* Tool, named **BN8**.

6 Create a **Block** using ***Defined by*** **Box** and ***Type*** **Model**.

7 Calculate the **Rapid Move Heights** using the default settings.

8 Select all of the **Surfaces** that define the fillet along the upper edge of the main 3D form (As shown below).

Copyright © Autodesk



9 In the **PowerMill *explorer*** right mouse click on **Patterns** and from the local menu select **Create Pattern**.

10 From the same menu select **Insert** – **Model**.

*This will automatically become an **Embedded Pattern** as it inherits the required data from the edges of the selected **Surfaces**.*

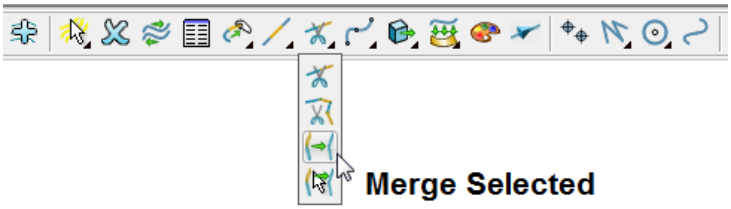
11 Select and **Blank** the whole of the **Model** and Un-draw the **Tool** and **Block**.

12 Right mouse click on the **Embedded Pattern** and from the local menu select **Curve Editor**.

13 Use **Cut item** to break the **Pattern** into separate segments at the four corners of the fillet form.

*In this example the **Embedded Pattern** must consist of 4 segments that specifically define the 2 ends and 2 sides of the enclosed fillet.*

14 If any of the drive or end curves contain multiple segments, select them and use **Merge** to reduce them to a single segment.

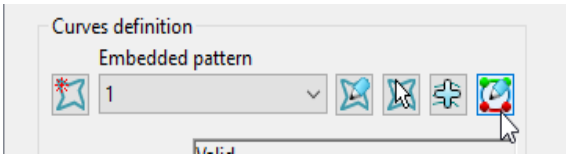


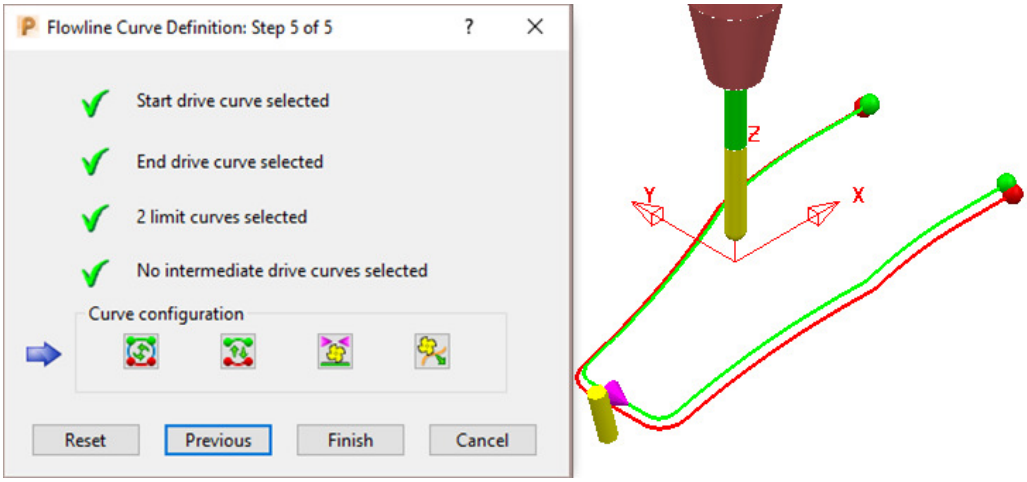
Copyright © Autodesk

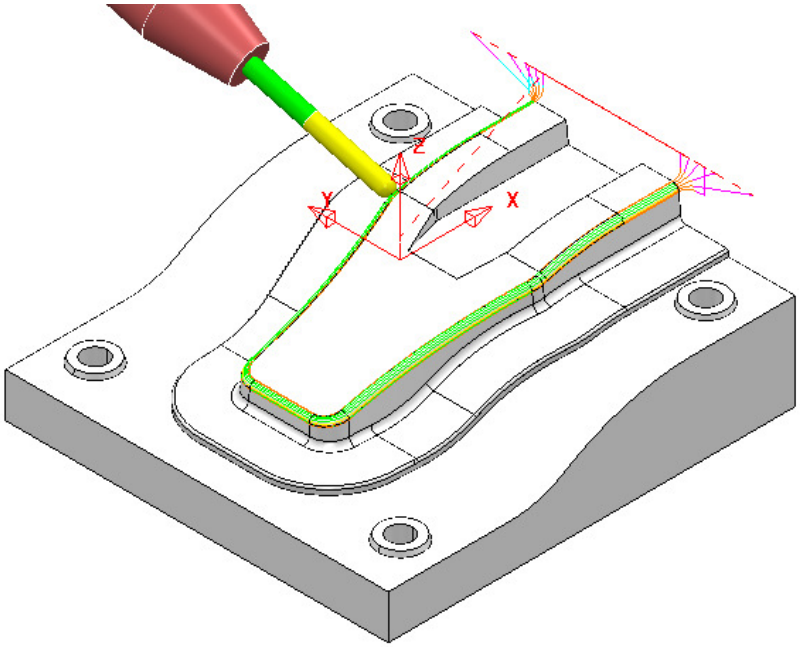
15 Accept the changes (**Green** tick) and return the **Embedded Pattern** from the **Curve Editor**.

16 Use the (4 segment) **Embedded Pattern** in a **Flowline Finishing** strategy to five axis machine (**Tool Axis** - **Lead/Lean 0** using **Mode** - **Contact normal**) the enclosed surfaces of the fillet form.

17 Use the options in **Interactively select pattern segments for the strategy** to control the required toolpath order and direction.







18 Select **File** > **Save Project As**:

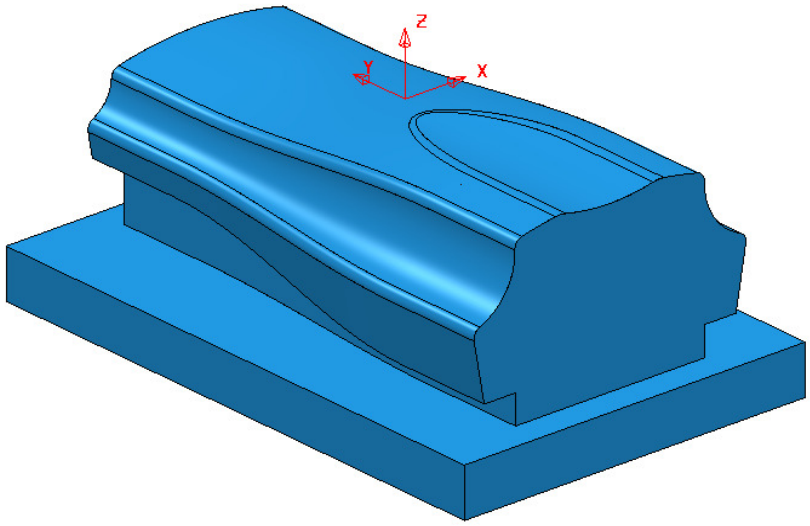
...\COURSEWORK\PowerMill-Projects\FlowLineFillet-EX1

Copyright © Autodesk

Flowline with intermediate drive curves Example

1. Select **File** - **Delete All** followed by **Tools** - **Reset Forms**.
2. **Open** the **Project**:

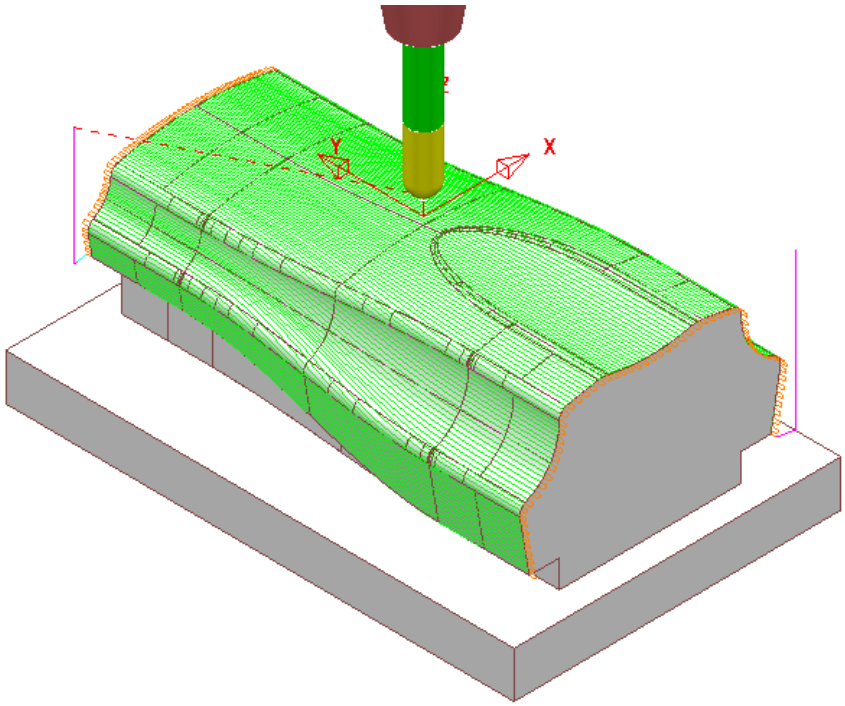
   ....\PowerMill_Data\five_axis\Flowline\FlowLine-ExtraDriveCurves-Start

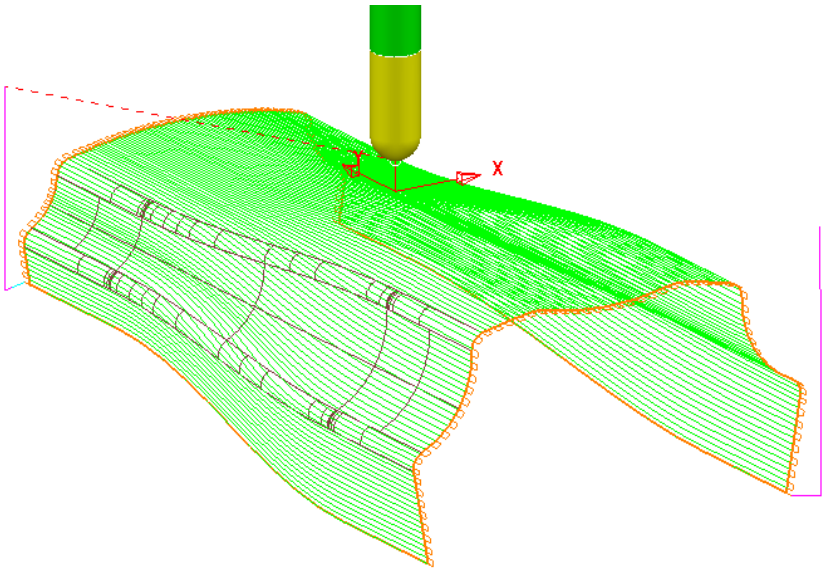


3. Select **File** - **Save Project As**:

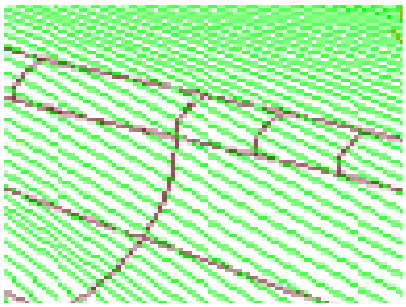
   ....\COURSEWORK\PowerMill-Projects\FlowlineWithExtraDriveCurves

4. Select an **ISO 1** view.
5. Activate the toolpath **BN16-Flowline-4Curves-All**.





Note that this basic 4 segment, **Flowline** toolpath does not follow the intermediate flowlines of the variable fillet within the model form.

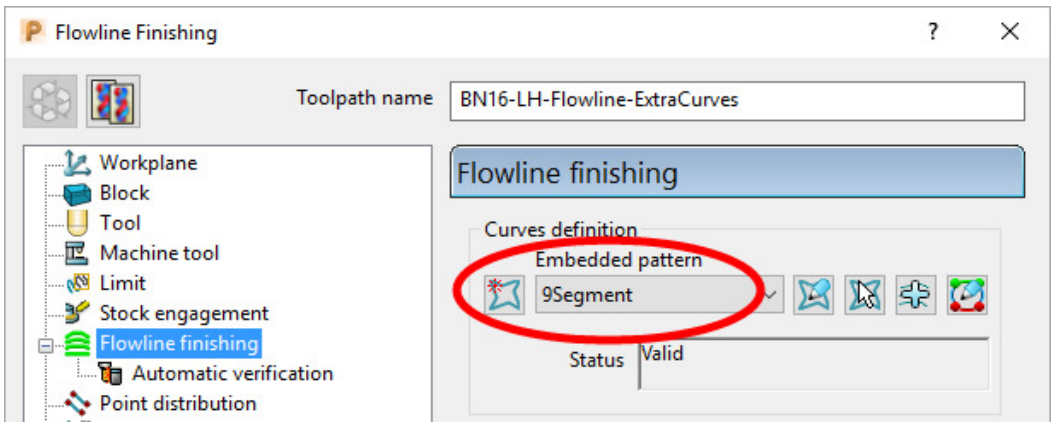


This can be changed if ***intermediate drive curves*** are included in the **Embedded Pattern**.

6 Right mouse click on the active toolpath **BN16-Flowline-4Curves-All** and from the local menu select **Settings** to open the **Flowline Finishing** dialog.

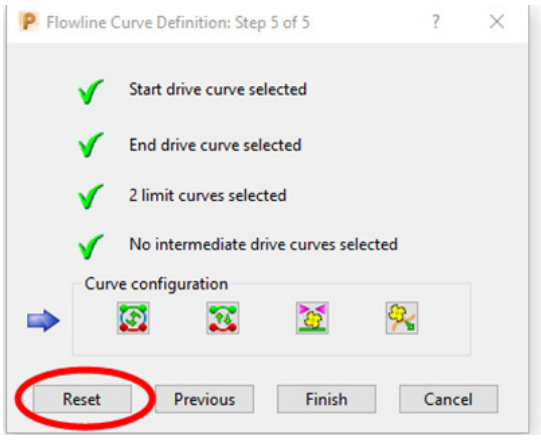
7 Select the **Copy** toolpath option and then rename the new toolpath as **BN16-LH-Flowline-ExtraCurves**.

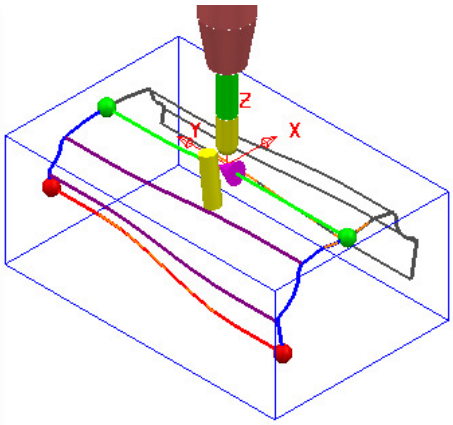
8 In **Curves definition**, change the selected **Embedded pattern** to **9Segment**.



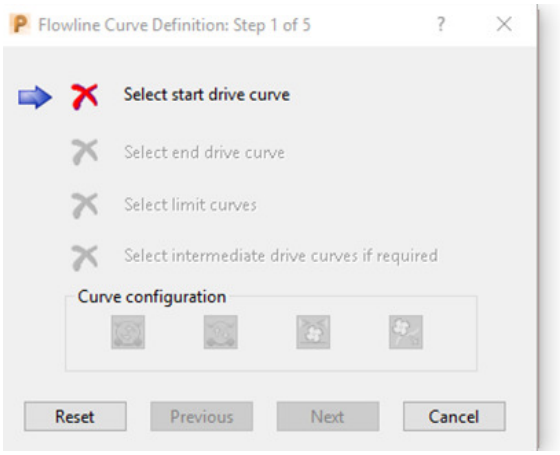
Copyright © Autodesk

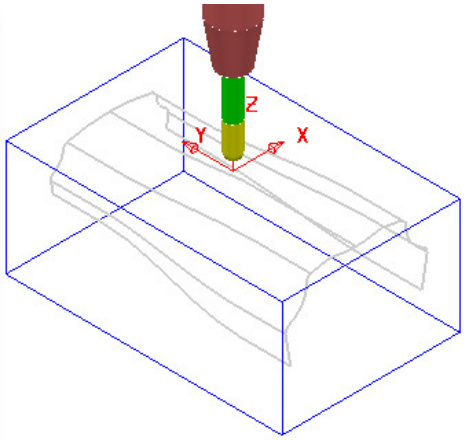
9 Select **Interactively select pattern segments for the strategy**.





10 Select the **Reset** button to clear the default, segment selection.



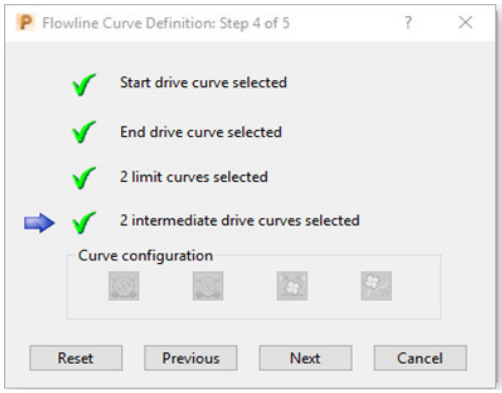


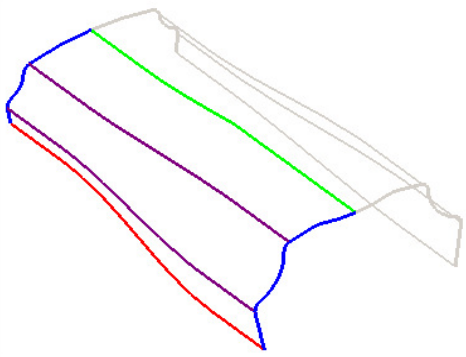
11 Select the upper central curve as the **start drive curve** (green)

12 Select the bottom left curve as the **end drive curve** (red).

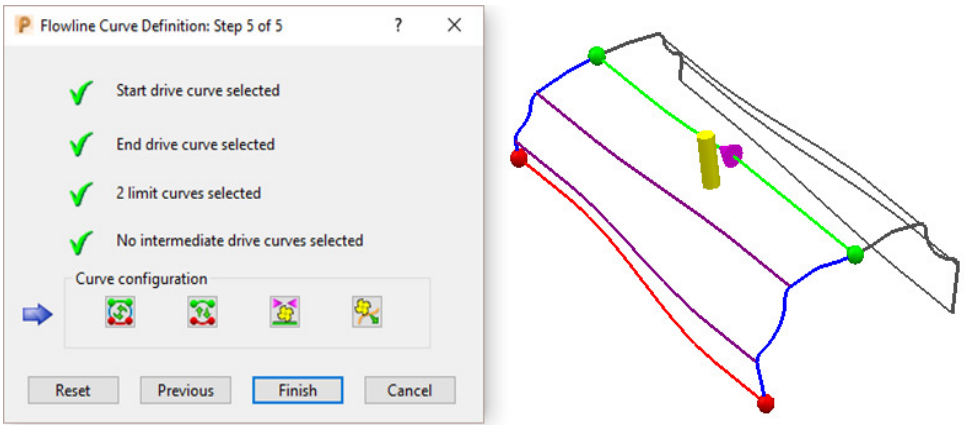
13 Shift – select the 2 **end curves** (blue).

14 Shift select the 2 nearest **intermediate curves** (purple).



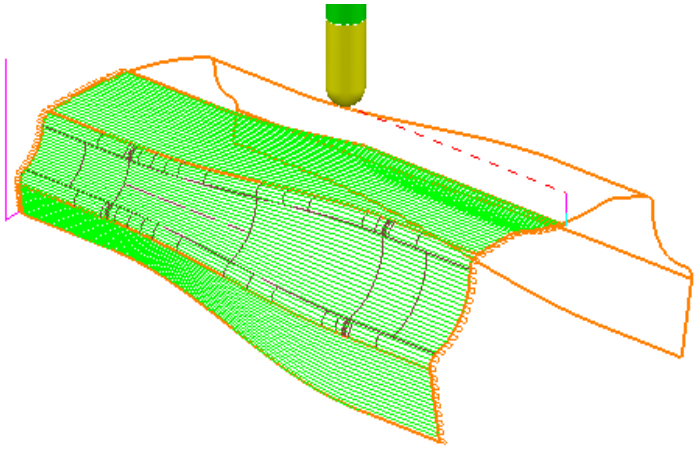


15 Select the **Next** button to return to the full preview stage.



16 Select the **Finish** button to return to the main Flowline finishing dialog.

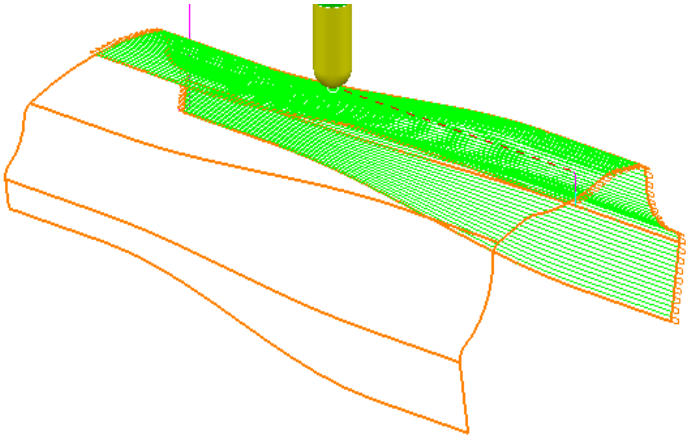
17 Select **Calculate** to generate the new toolpath then select **Close**.



Note that this 6 segment, **Flowline** toolpath follows the intermediate flowlines of the variable fillet within the model form.

Exercise

18 Create a ***Copy*** of the above toolpath named **BN16-RH-Flowline-ExtraCurves** and modify to machine the RH side of the component form.



19 Select **File** – **Save Project**.

20 Select **File** - **Delete All** and **Tools** - **Reset Forms**.

Copyright © Autodesk

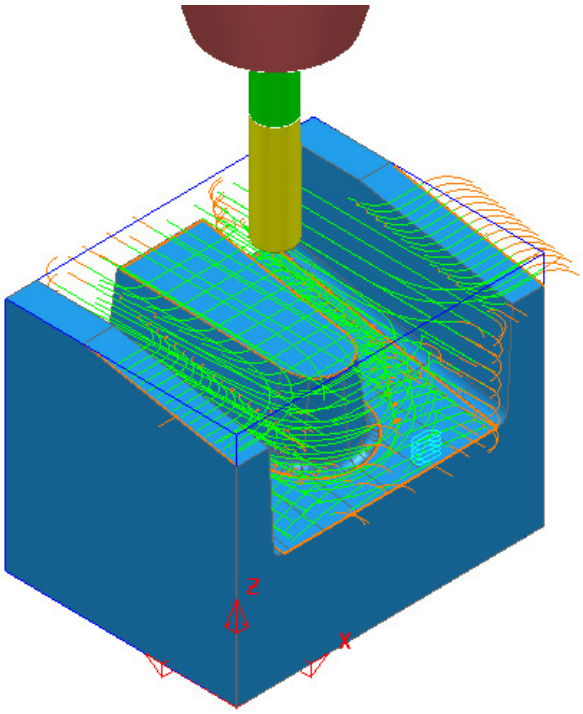
8. Auto Collision Avoidance

Introduction

Automatic Collision Avoidance can be applied to **Vertical** alignment operations in cases where the **Shank** would otherwise be rubbing on a sidewall and\or the Holder would clash with the component. If parts of the component still cannot be machined without a tool collision occurring then these areas will not be included as part of the toolpath. Note: **Automatic Collision Avoidance** is not applicable to **Swarf Finishing**.

1 **Open** the **Project**:

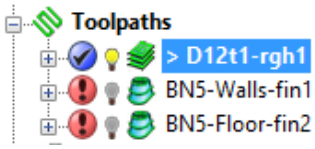
...\PowerMill_Data\five_axis\CollisionAvoidance\AutoCollisionAvoid1_Start



2 Select **File** > **Save Project as**:

...\COURSEWORK\PowerMill-Projects\AutoCollisionAvoid_1

The **Project** contains 3 toolpaths all of which have had **Collision Checking** applied with **Shank** and **Holder** clearances values of **1** and **2** respectively.



The **Area Clearance** strategy is **Gouge** and **Collision** safe while the two **3D**

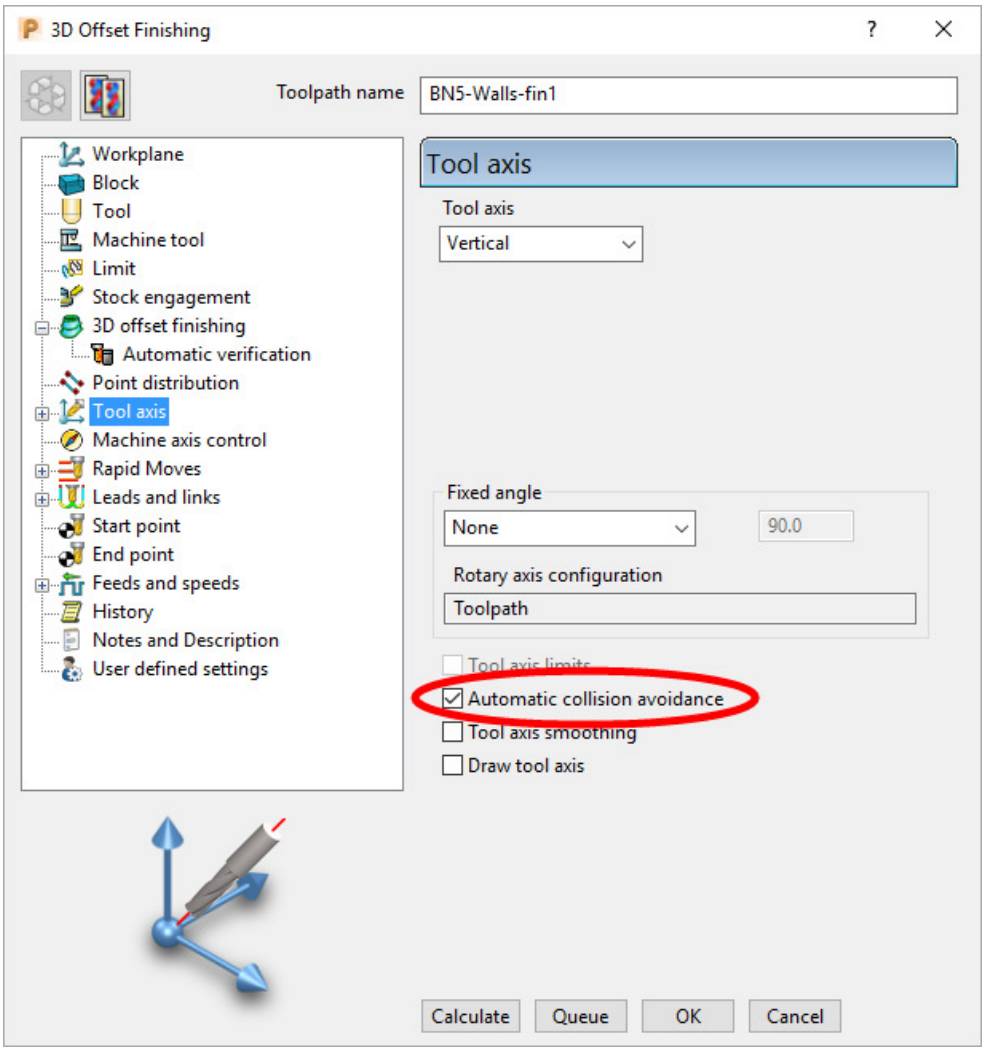
Offset finishing strategies although **Gouge** free are not **Collision** safe relative to the **Shank** and **Holder**.

3 In the **PowerMill *explorer***, activate the toolpath **BN5-Walls-fin1**.

4 Right mouse click on the ***toolpath*** **BN5-Walls-fin1** and from the local menu select **Settings**.

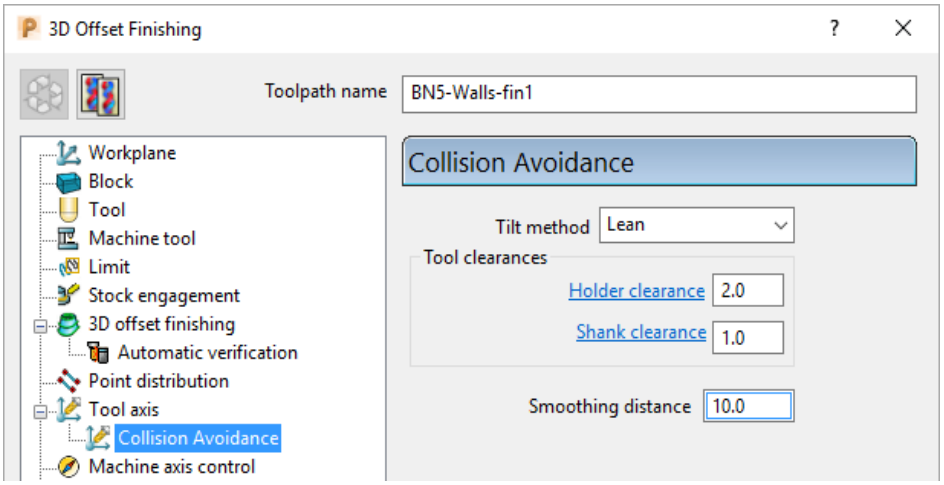
5 Select **Recycle** in the **3D Offset Finishing** dialog and select the **Tool Axis** page.

*Note that the selected **Tool Axis** option is **Vertical**.*



Copyright © Autodesk

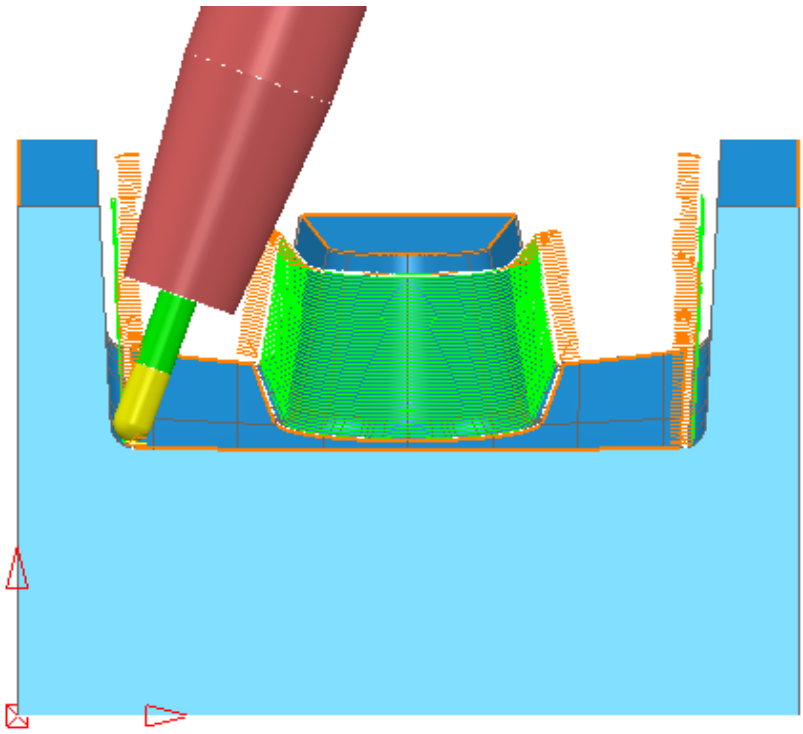
6 Select the newly available **Collision Avoidance** page (Below **Tool axis**).



- Input the ***Tilt method*** as **Lean**.
- Input ***Shank*** and ***Holder*** clearances of **1** and **2** along with a **smoothing distance** of **10**.

7 **Calculate** the **3D Offset Finishing** toolpath.

8 Select a **View from front** and **Simulate** the toolpath to observe the **Automatic Collision Avoidance** in action.

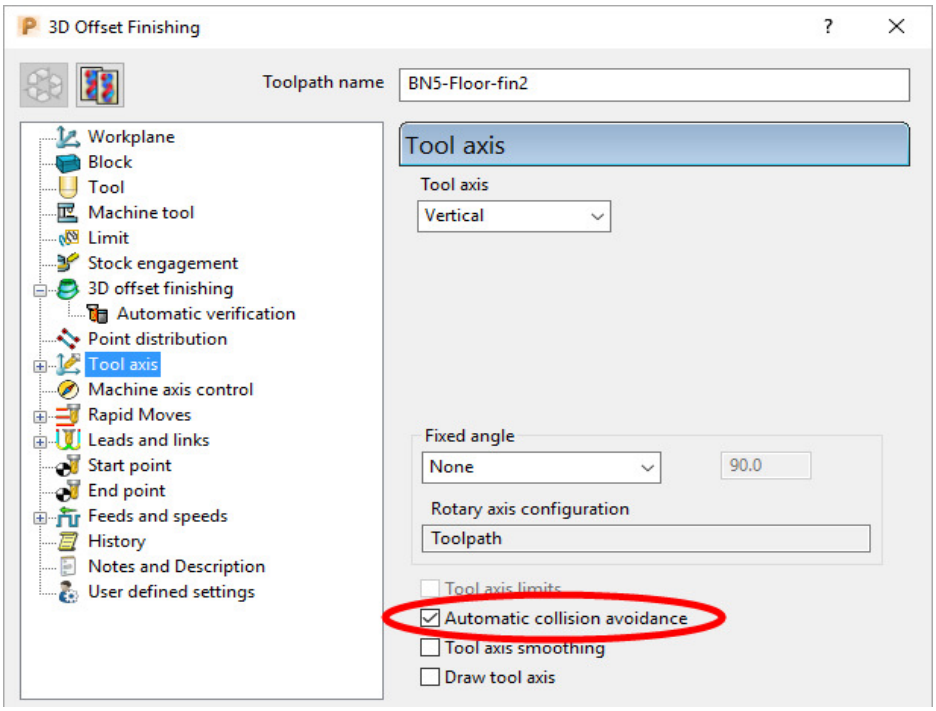


9 In the **PowerMill *explorer***, activate the toolpath **BN5-Floor-fin2**.

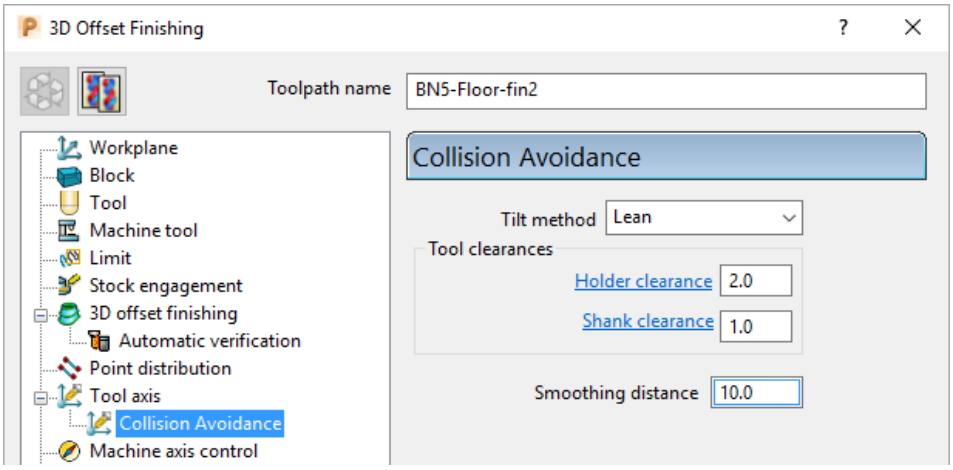
10 Right mouse click on the ***toolpath*** **BN5-Floor-fin2** and from the local menu select **Settings**.

11 Select **Recycle** in the **3D Offset Finishing** dialog and select the **Tool Axis** page.

*Note that the selected **Tool Axis** option is **Vertical**.*



12 Select the **Collision Avoidance** page.

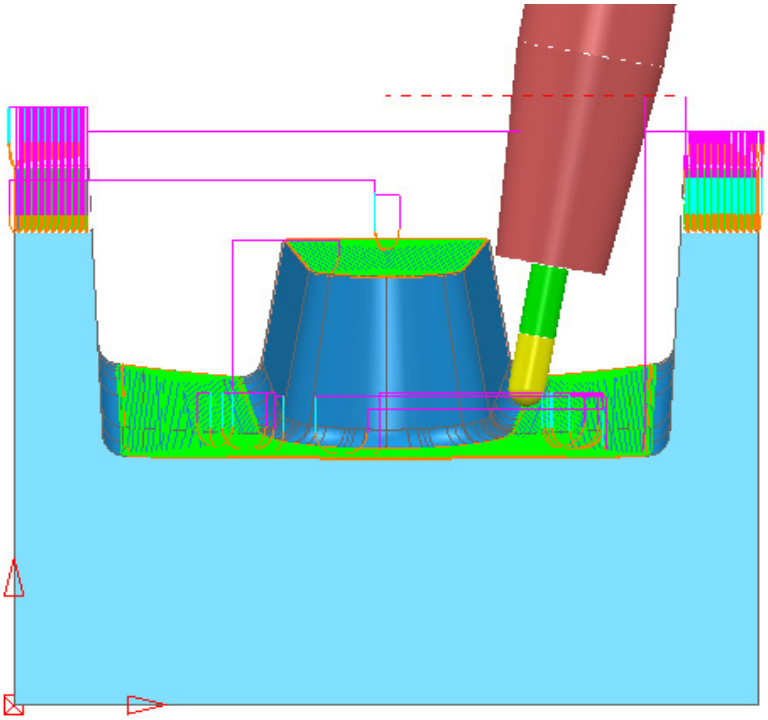


- Input the ***Tilt method*** as **Lean**.
- Input ***Holder*** and ***Shank*** clearances of **2** and **1** along with a **smoothing distance** of **5**.

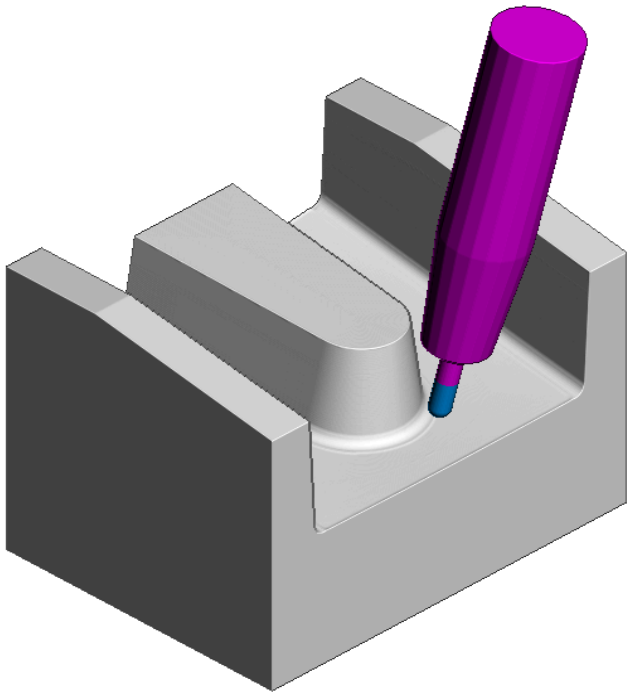
13 **Calculate** the **3D Offset Finishing** toolpath.

Copyright © Autodesk

14 Select a **View from front** and **Simulate** the toolpath to observe the **Automatic Collision Avoidance** in action.



Vertical tool alignment is applied as default and the **Lean** angle is progressively changed as required in areas where a collision would otherwise occur.



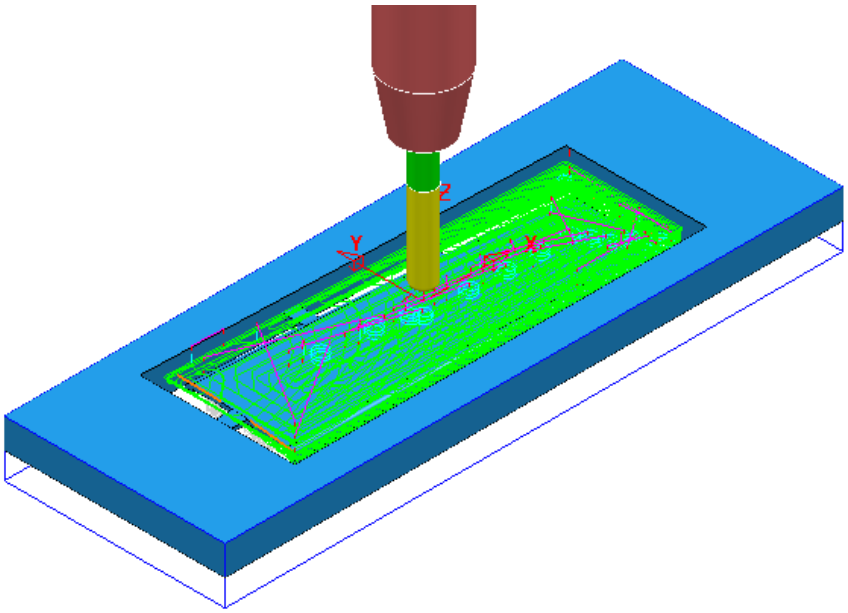
15 Select **File** > **Save Project**.

EXAMPLE 2

1 Select **File** - **Delete All** and **Tools** - **Reset Forms**.

2 **Open** the (read only) **Project**:

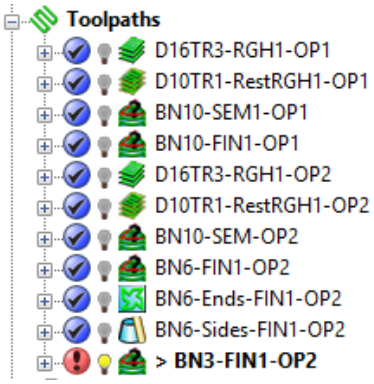
....\PowerMill_Data\five_axis\CollisionAvoidance\DashboardPanel-A1_Start



3 Select **File** > **Save Project as**:

....\COURSEWORK\PowerMill-Projects\Dashboard-A1_EX1

The **Project** contains 11 toolpaths all of which have had **Collision Checking** applied with **Shank** and **Holder** clearances values of **1** and **2** respectively.



All have been confirmed to be safe with the exception of **BN3-FIN1-OP2**.

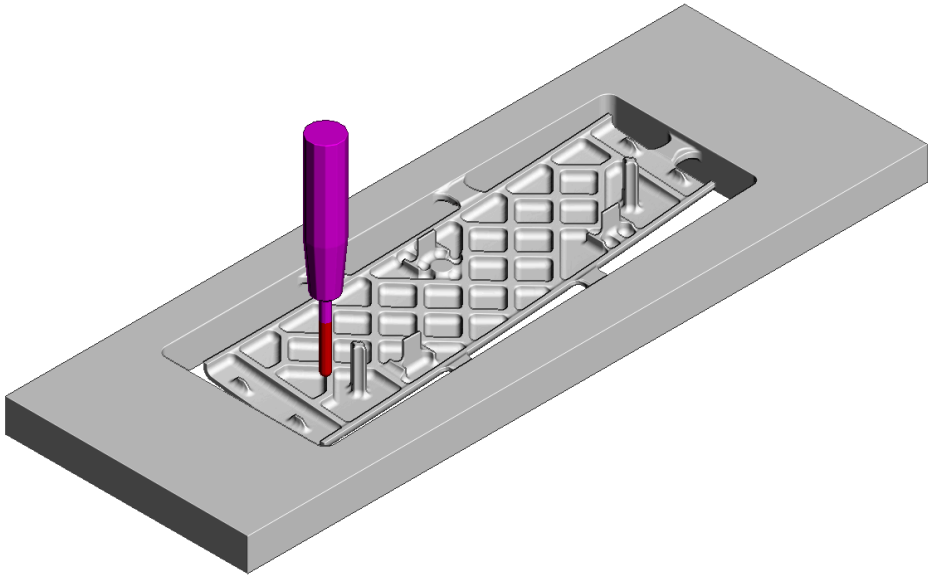
The toolpath is ***gouge free*** relative to the flutes but not ***collision safe*** relative to the **Shank** and **Holder**.

4 **Activate** the 5th toolpath, **D16TR3-RGH1-OP2** to reinstate the ***OP2*** setup.

5 Select an **ISO1** view.

Copyright © Autodesk

6 Run a full **ViewMILL** simulation on both ***NCPrograms*** **OP1** and **OP2**.



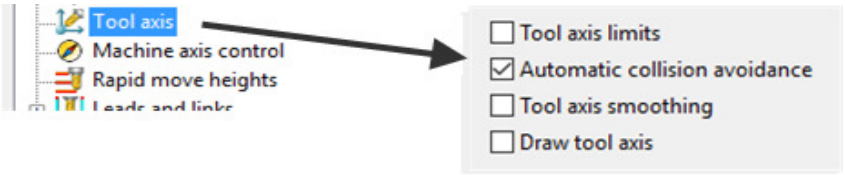
The final toolpath **BN3-FIN1-OP2** is not currently included in the **OP2** ***ncprogram***. The above **ViewMILL** simulation displays the result of all toolpaths except **BN3-FIN1-OP2**.

7 Put **ViewMILL** on suspend and return the graphics area to the **PowerMill** page.

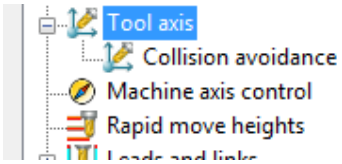
8 **Activate** the toolpath **BN3-FIN1-OP2** and from the local menu select **Settings**.

9 Select **Recycle** in the **Steep and Shallow Finishing** dialog and select the **Tool Axis** page.

10 In the **Tool axis** page ***tick*** the box named **Automatic collision avoidance**.

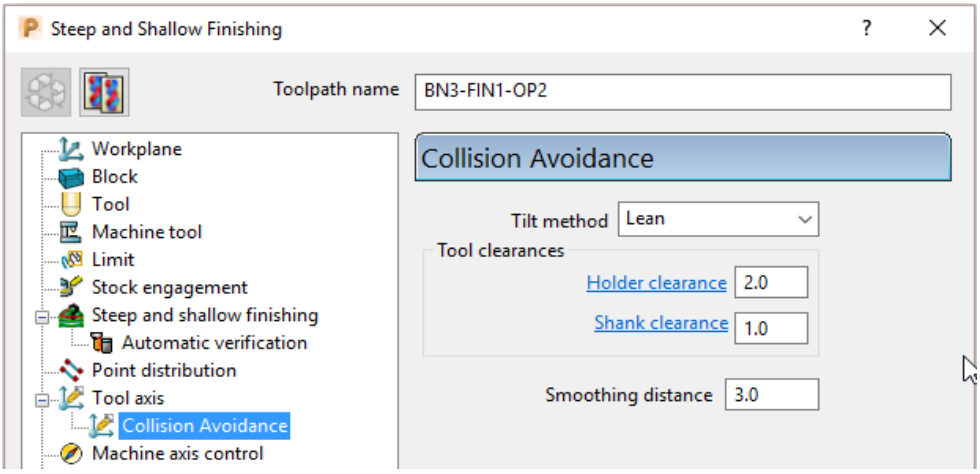


The **Collision avoidance** page appears in the local **explorer**.

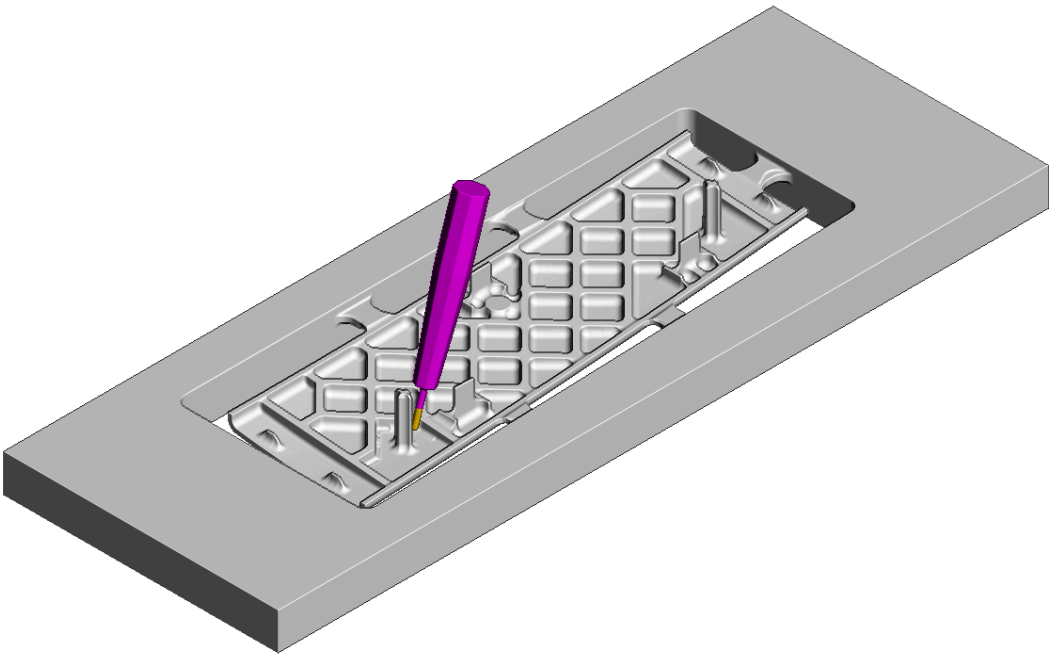


11 Select the **Collision avoidance** page.

12 Set ***Tilt Method*** to **Lean** with a ***Holder clearance*** of **2.0**, ***Shank clearance*** of **1.0**, and ***Smoothing distance*** **3.0**.



13 Select **Calculate** and note that the collision status of the toolpath **BN3-FIN1-OP2** (as displayed in the local explorer) changes from a ***Red*** to ***Blue***.



14 Run a **ViewMILL** simulation on the toolpath **BN3-FIN1-OP2**.

If a potential collision situation is encountered the tool automatically ***leans*** away from the wall (as shown in the above image).

15 **Save** the **Project**.

Copyright © Autodesk

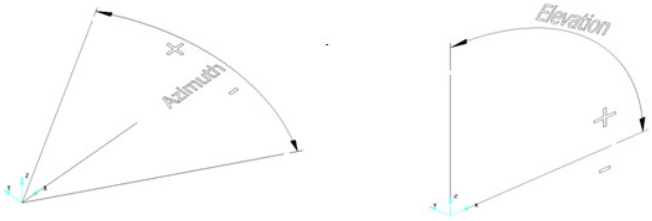
9. Tool Axis Limits

Introduction

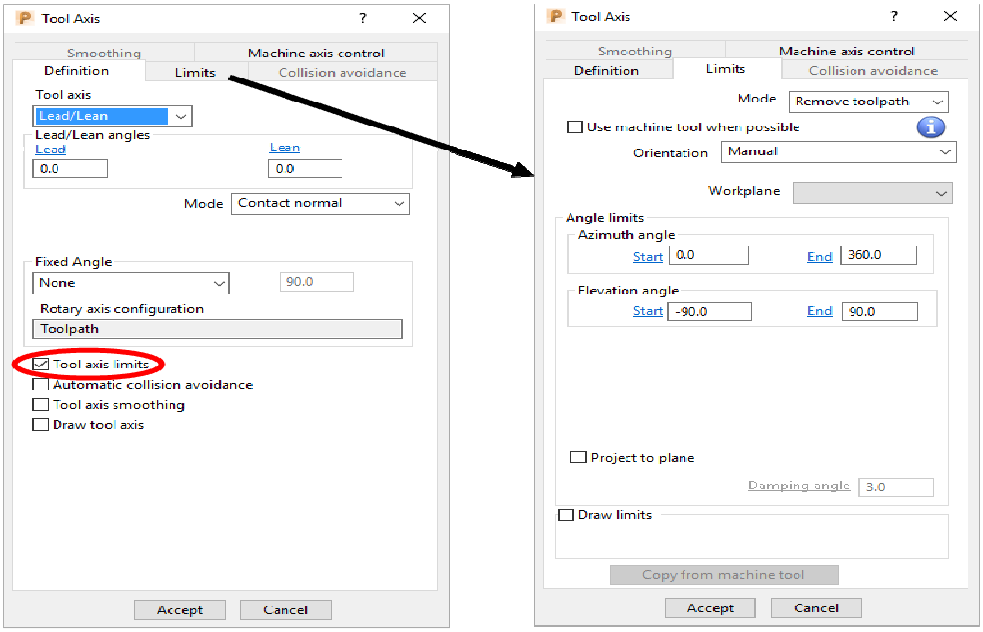
It is possible to control the rotary **Tool axis limits** of a machine tool within **PowerMill**. This enables the rotary working envelope of a specific machine not to be exceeded when creating multi-axis toolpaths. The angular limits are defined in terms of **Azimuth** and **Elevation** angles.

Azimuth and Elevation

The **Azimuth** is the angle from X 0° anticlockwise around the XY plane. The **Elevation** is the angle that is lifted upwards (+90°) or downwards (-90°) from the XY plane.



The **Tool Axis Limits** dialog is accessed from within the **Tool Axis** direction dialog. **Tool Axis** Limits are accessible if the **Tool Axis** box is ticked. This tick box will not be available if the Tool axis is set to **Vertical** or **Fixed Direction**.



1 Select **File** - **Delete all** and **Tools** - **Reset Forms**.

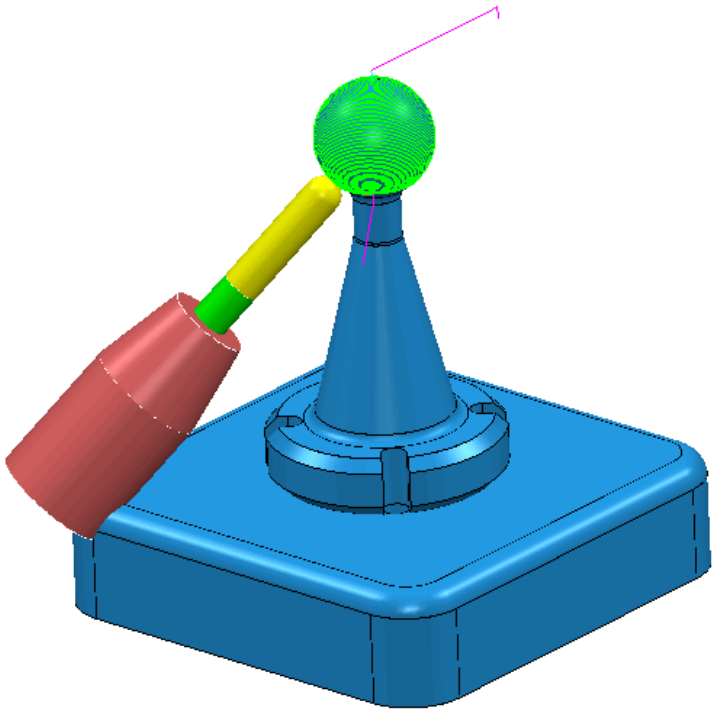
2 **Open** the 'read only' **Project**:

….\PowerMill_Data\five_axis\Tool_Limit\JoyStick_Start

3 Select **File** > **Save Project As**:

….\COURSEWORK\PowerMill-Projects\JoyStick-Example

4 In the graphics area, right-click on the toolpath at the underside of the sphere, and from the local menu select, **Simulate from Here**.



Note that the tool is cranked over way beyond the rotational limits of the machine tool and the tool holder is visibly colliding with the base form.

To further illustrate the machining process exceeding the rotary limits, the strategy will be simulated using a **DMU50 Evolution** machine tool.

5 In the **PowerMill *explorer***, right mouse click on the Toolpath **BN16-NoToolAxisLimits** and from the local menu select **Simulate from Start**.

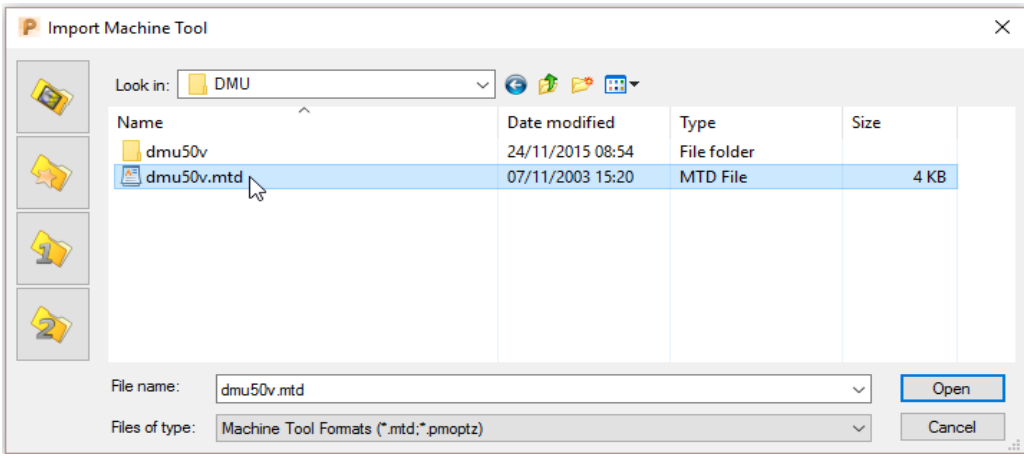
This command will raise the **Simulation** toolbar (if it is not already open).

6 In the **PowerMill *explorer*** right mouse click on **Machine Tools** and from the local menu select **Toolbar**.

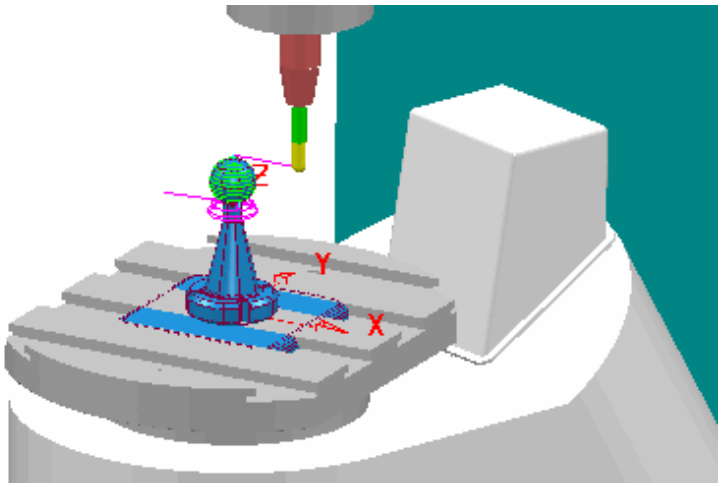
7 Select **Import the Machine Tool Model** and select the file:

....\PowerMill Data\Machine Data\DMU\dmu50v.mtd

Copyright © Autodesk



8 Ensure the **Draw\Undraw** machine tool icon is on to display the machine tool model.



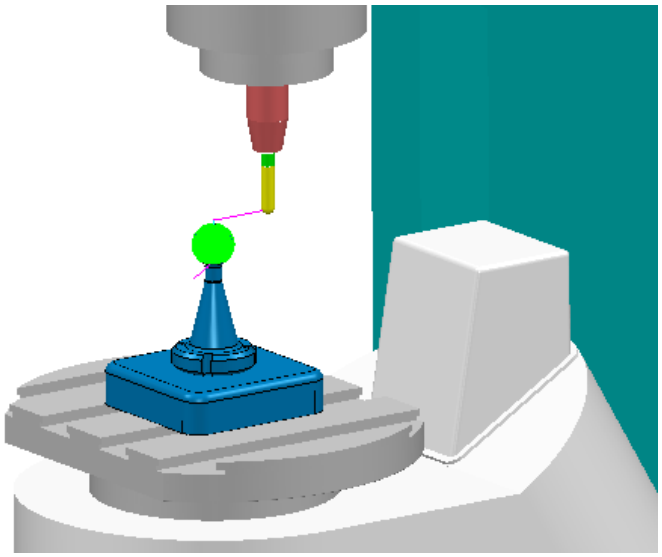
The current component datum (**Transform**) is matched to that of the Machine Tool model (Top - Centre of Table). As a result, the base of the component is currently embedded into the machine tool bed. To compensate for this, a new, suitably positioned Workplane is created. This new **Workplane** is then used in the **Machine Tool definition** toolbar.

9 Create a new **Workplane** named **MTD-datum** and move it by **Z-50**.

10 Input the new **Workplane**, **MTD-datum** into the **Machine Tool definition** dialog.

*It is not necessary to activate the new **Workplane.***

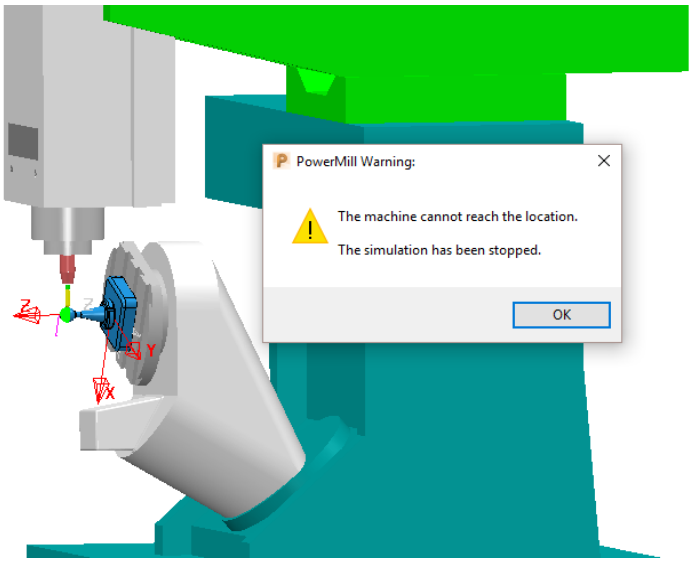
The component will immediately be repositioned, relative to the new **Workplane**.



11 Select **Floor View** on the **Machine Tool** toolbar.

12 Start the simulation to observe the virtual machining of the component.

The **DMU50** angular limits are **X ±90 Y ±360** and this information is stored in the **MTD** file. This translates to **Azimuth Angle** limits of **0** to **360** and **Elevation Angle** limits of **0** to **90**. When the toolpath simulation attempts to go beyond this range an error message will be displayed indicating that the machine, **Tool Axis Limits** will be exceeded.



*To allow for this appropriate **Tool Axis Limits** will be applied to the toolpath during calculation.*

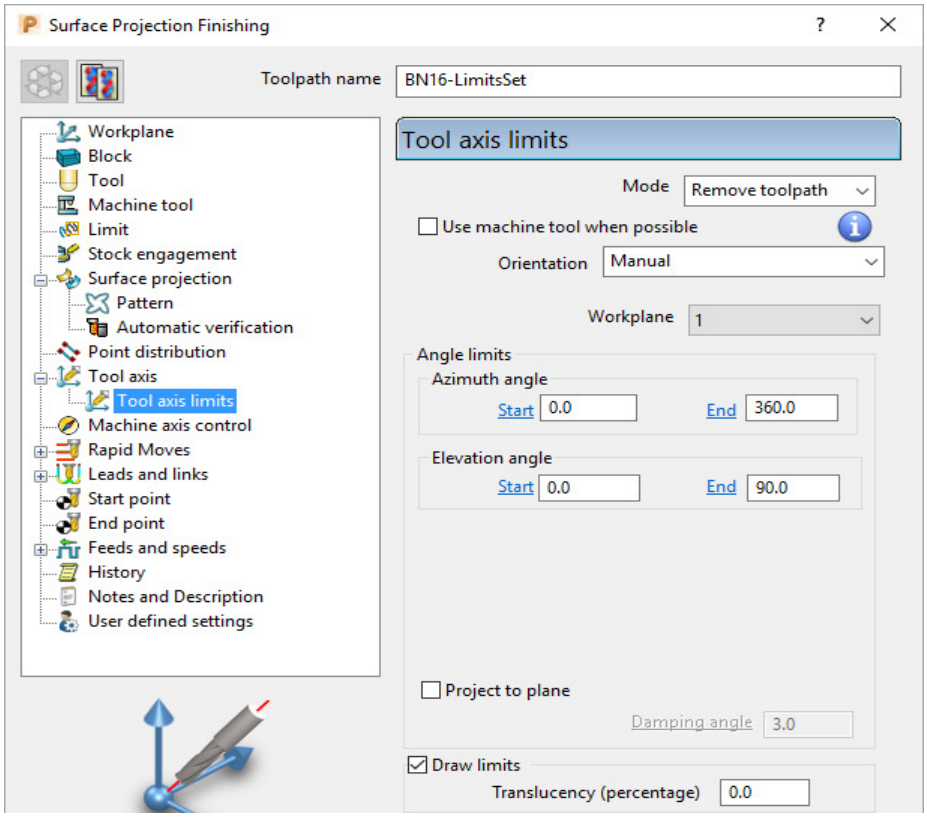
Copyright © Autodesk

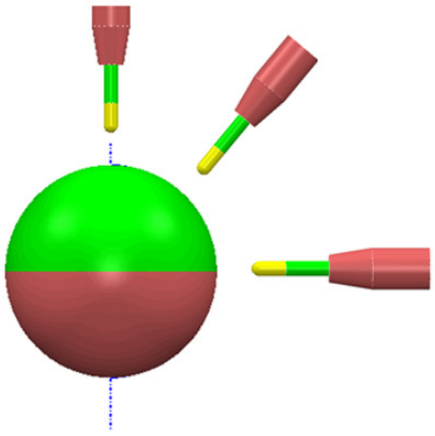
13 In the **PowerMill *explorer***, Machine Tools folder right mouse click on **dmu50v** and from the local menu select **Go Home** to return the machine tool model to the home position.

14 Make a **Copy** of the original toolpath and rename it as **BN16-LimitsSet**.

15 In the local explorer **Tool Axis** folder, select the **Tool Axis** - **Limits** option to activate the **Tool Axis** Limits page.

16 Enter the values into the dialog exactly as shown (select **Mode** - **Remove toolpath** and tick the **Draw limits** box).



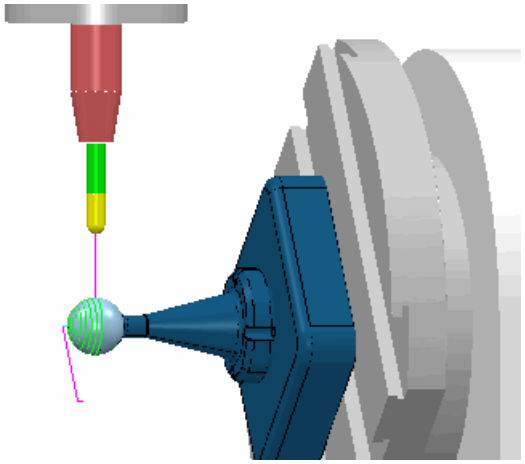


The green area of the sphere represents the permissible angular alignment of the **Tool Axis**.

17 With the same **Reference Surface** selected, **Calculate** and **Close** the dialog.

18 In the **PowerMill *explorer*** right-click the toolpath, **BN16-LimitsSet** and from the local menu select **Simulate from Start**.

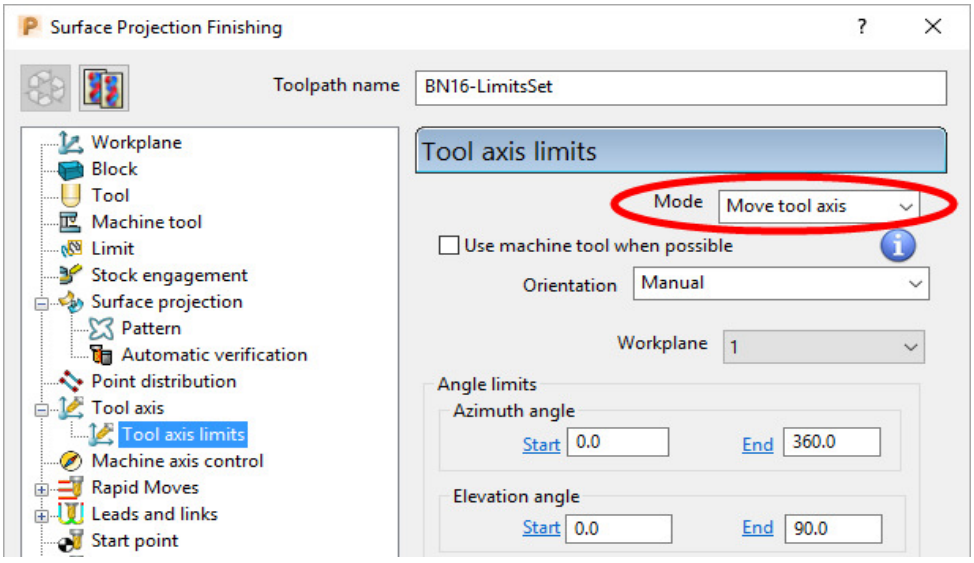
19 Select **Play** .



*With **Mode** set to **Remove Toolpath** in the **Limits** dialog, only the portion of the surface within the rotary **Tool Axis Limit** range is machined.*

20 Right-click on the toolpath, **BN16-LimitsSet**, and select **Settings** to access the **Surface Projection Finishing** dialog.

21 Select recycle and in the **Tool axis** page select the **Tool axis limits** and enter the values into the form exactly as shown below (select **Mode** – **Move tool axis**).

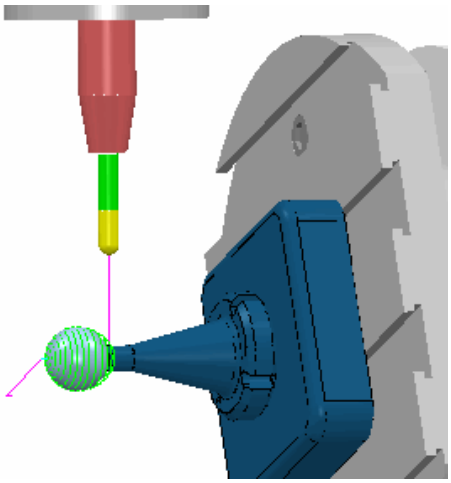


22 With the original reference surface selected, **Calculate** the toolpath.

23 Right-click the toolpath **BN16-LimitsSet**, in the **PowerMill *explorer*** and select **Simulate from Start**.

24 Select **Play** .

Copyright © Autodesk



With **Mode** set to **Move tool axis** in the **Limits** dialog, the surface is fully machined with the **Tool Axis** becoming fixed when it exceeds the maximum rotary Limit.

25 Select **File** > **Save Project** to update the stored data.

Defining Limits for a Multi-Axis Machine

The Tool Axis Limits option allows the user to control the angular limits of a tool while creating a multi-axis toolpath. The specified limits will differ in format depending on the type of rotary axis configuration. As a result they will have to be translated as universal Azimuth and Elevation angles to be compatible with PowerMill.

The configuration of the rotary axes varies widely, however the differences between many of these are relatively minor and there are really only three fundamentally different machine configurations:

Table – Table Both rotary axes move the table.

Head – Head Both rotary axes move the head.

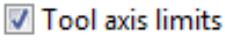
Head – Table One rotary axis moves the head, the other moves the table.

The next examples will show how to transpose angular limits of a machine tool into Azimuth and Elevation angles.

1 Select **File** - **Delete all** and **Tools** - **Reset Forms**.

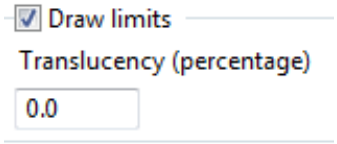
2 Select the **Tool Axis** icon from the **main** toolbar.

3 Define **Tool Axis** as **Lead\Lean** and set both the **Lead** and **Lean** angles to **0**.

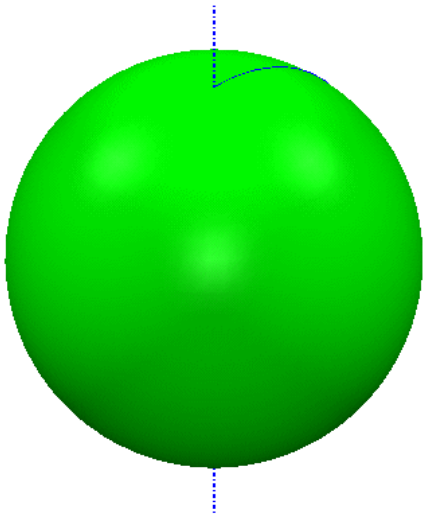
4 Select the **Tool axis limits** option  to activate the **Limits** tab.

5 Open the **Limits** tab.

6 Select the **Draw Limits** option from the dialog.



7 Select view **ISO 1**.

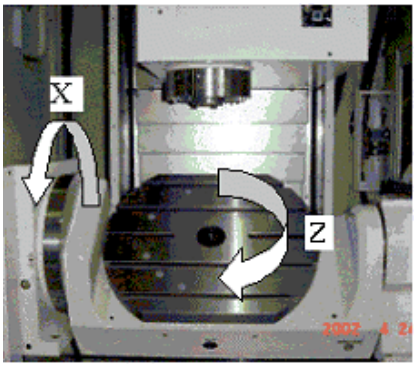


With the **Draw Limits** option selected a sphere will be displayed in the graphics window representing the angular machining limits available.

Green indicates a machinable portion and Red a non-machinable portion. With the default settings selected the total machining range is covered so the whole sphere will be Green.

Table – Table

Both rotary axes operate on a table.



With the above **Table-Table** machine tool the angular limits are specified as:

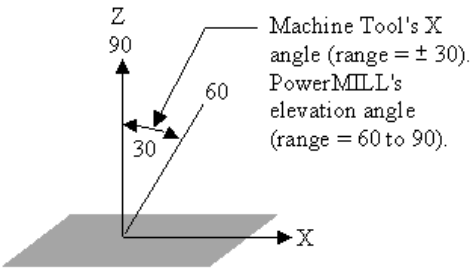
X ± 30

Z ± 360

The machine tool **Y** limits are equivalent to the **Azimuth Angle** or the angular limits normal to the **XY** plane. The **Y** limit of **± 360** translates to **Azimuth Angle** limits of **0** to **360**.

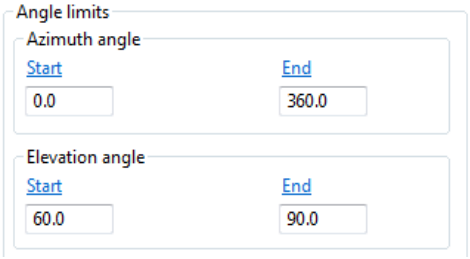
The machine tool **X** limits are equivalent to the **Elevation** angle above the **XY** plane. However they are not the same angle. This is best described using the diagram below. The machine tool measures the angular range relative to the **Z Axis** and **PowerMill** measures it relative to the **XY** plane, therefore the angle required for the limit in PowerMill is the complementary angle to the one given for the machine tool.

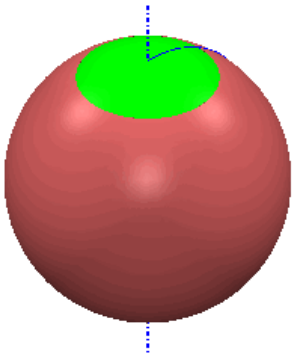
Copyright © Autodesk



The **X** limit of **±30** translates to **Elevation angle** limits of **60** to **90**.

1 Retain the default **Azimuth Angle** and modify the **Elevation Angle** values in the form as shown below to update the machining limits.





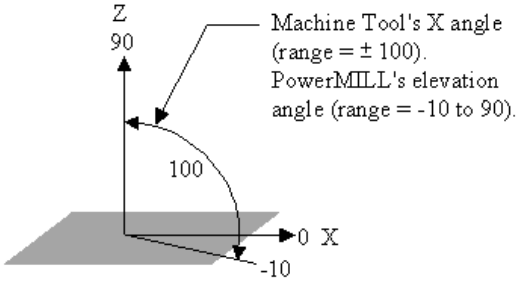
The **Tool Axis Limits** sphere has been visually updated with the modified values.

An Alternative **Table -Table** machine tool has the following angular limits:

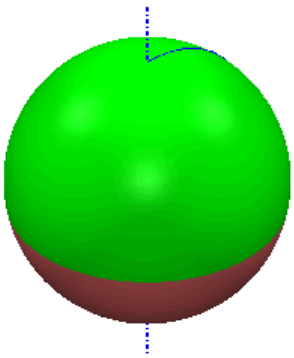
X ± 100

Y ± 360

This translates to **Azimuth Angle** limits of **0** to **360** and **Elevation Angle** limits of **–10** to **90**.



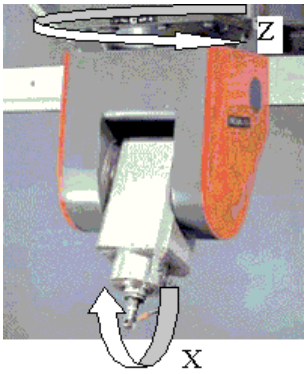
2 Modify the **Elevation Angle** values (**-10** to **90**) in the **Angle Limits** dialog to update the machining limits (as shown).



The **Tool Axis Limits** sphere has been visually updated with the modified values.

Head – Head

Both rotary axes move the head.



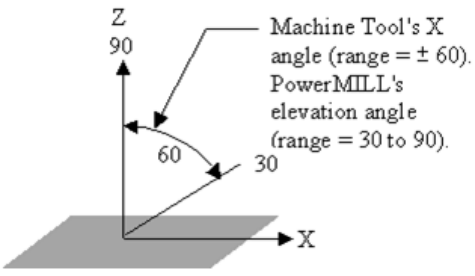
With the above **Head-Head** machine tool the angular limits are specified as:

X ± 60

Z ± 360

The machine tool **Z** limits are equivalent to the **Azimuth Angle** or the angular limits normal to the **XY** plane. In **PowerMill** the **Z** limit of **± 360** translates to **Azimuth Angle** limits of **0** to **360**.

The machine tool **X** limits are equivalent to the **Elevation Angle** above the **XY** plane. The angular range of the machine tool is relative to the **Z** axis, however **PowerMill** measures it relative to the **XY** plane. Therefore the angle required for the limit in **PowerMill** is the complementary angle to the one given for the machine tool. The **X** limit of **± 60** translates to **Elevation Angle** limits of **30** to **90**.



Alternative **Head-Head** machine tool angular limits:

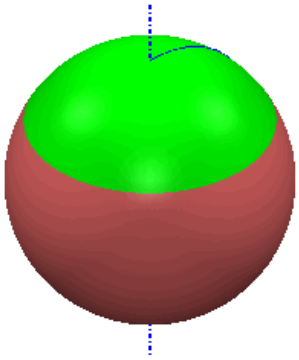
X –50 to **+60**

Z ± 360

Copyright © Autodesk

This translates to **Azimuth Angle** limits of **0** to **360** and **Elevation Angle** limits of **30** to **90**. In this case the machine tool limits across the **XZ** plane differ. **PowerMill** will use the largest rotational value (**+60**). This is allowed by rotating the head **180º** about **Z** to provide access to the maximum range **+60** (which otherwise would be **-50**).

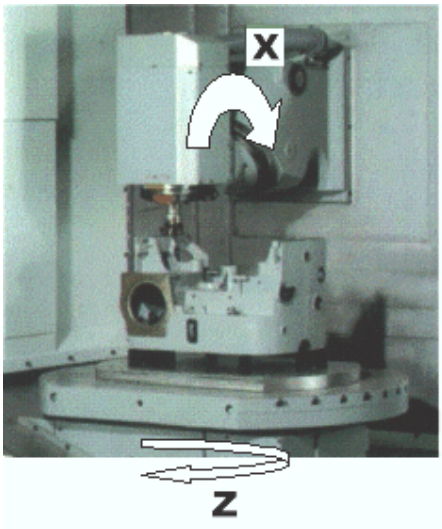
1 Modify the **Elevation Angle** values (**30** to **90**) in the dialog to update the machining limits as shown.



The **Tool Axis Limits** sphere has been visually updated with the modified values.

Head – Table

One rotary axis moves the head, and the other moves the table.



With the above **Head-Table** machine tool, the angular limits are specified as:

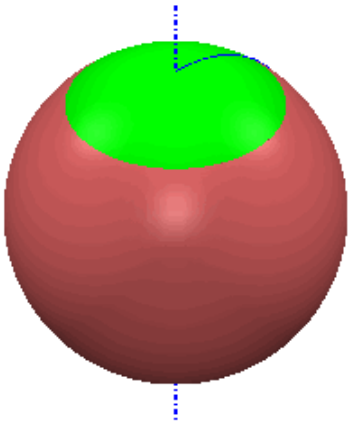
X ± 40

Z ± 360

The machine tool **Z** limits are equivalent to the **Azimuth Angle** or the angular limits in the **XY** plane. The **Z** limit of **±360** translates to **Azimuth Angle** limits of **0** to **360**.

The machine tool **X** limits are equivalent to the **Elevation Angle** above the **XY** plane. It is the complimentary angle to the **Elevation Angle**. The **X** limit of **± 40** translates to **Elevation Angle** limits of **50** to **90**.

1 Modify the **Elevation Angle** values (**50** to **90**) in the dialog to update the machining limits as shown below.

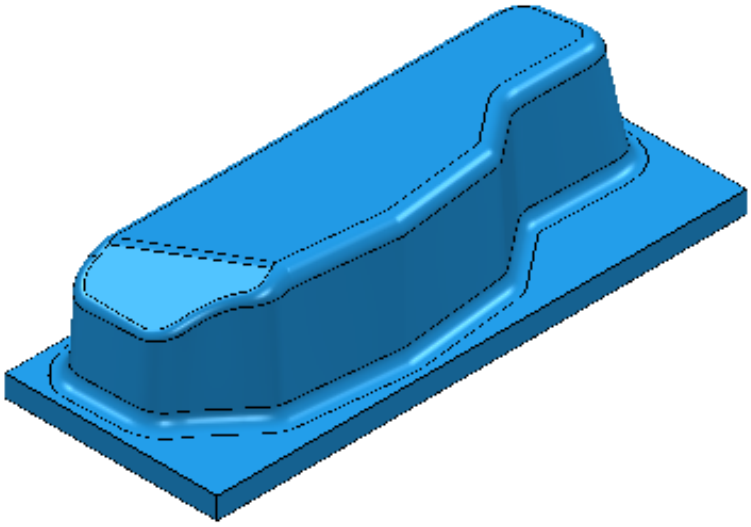


The **Tool Axis Limits** sphere has been visually updated with the modified values.

Applying tool Axis limits on a Steep Sidewall base Fillet

1. Select **File** - **Delete all** and **Tools** - **Reset Forms**.
2. **Import** the **Model**:

   ...\PowerMill_Data\five-axis\punch2\punch2_insert.dgk



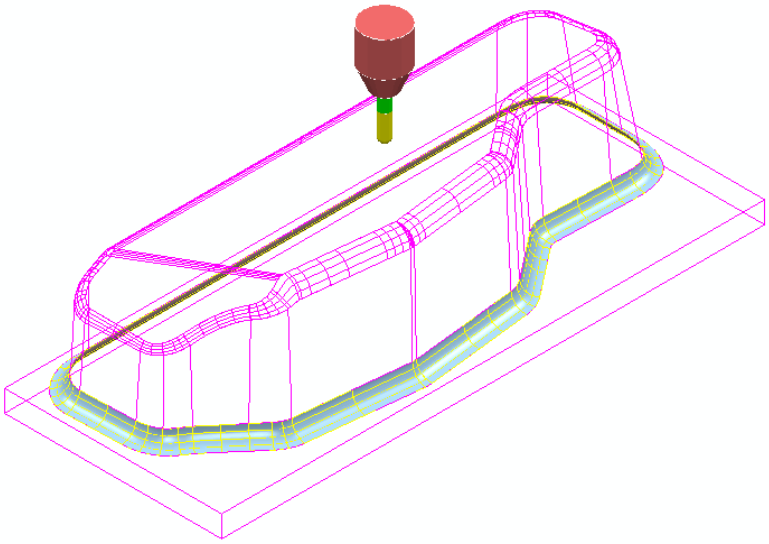
3. Select **File** > **Save Project As**:

   ...\COURSEWORK\PowerMill-Projects\Punch2-ToolAxisLimits

4. Create a Diameter **20** Ballnose tool - Length **50** - named **BN20**.
5. Add a **Shank** component - Upper Dia **20** - Lower Dia **20** - Length **40**
6. Add a **Holder** component - Upper Dia **70** - Lower Dia **40** - Length **40**
7. Add a **Holder** component - Upper Dia **70** - Lower Dia **70** - Length **60** - Overhang **80**
8. Calculate a **Block** using **Defined by** - **Box** and **Type** - **Model**.
9. Calculate the **Rapid Move Heights** using the default settings.

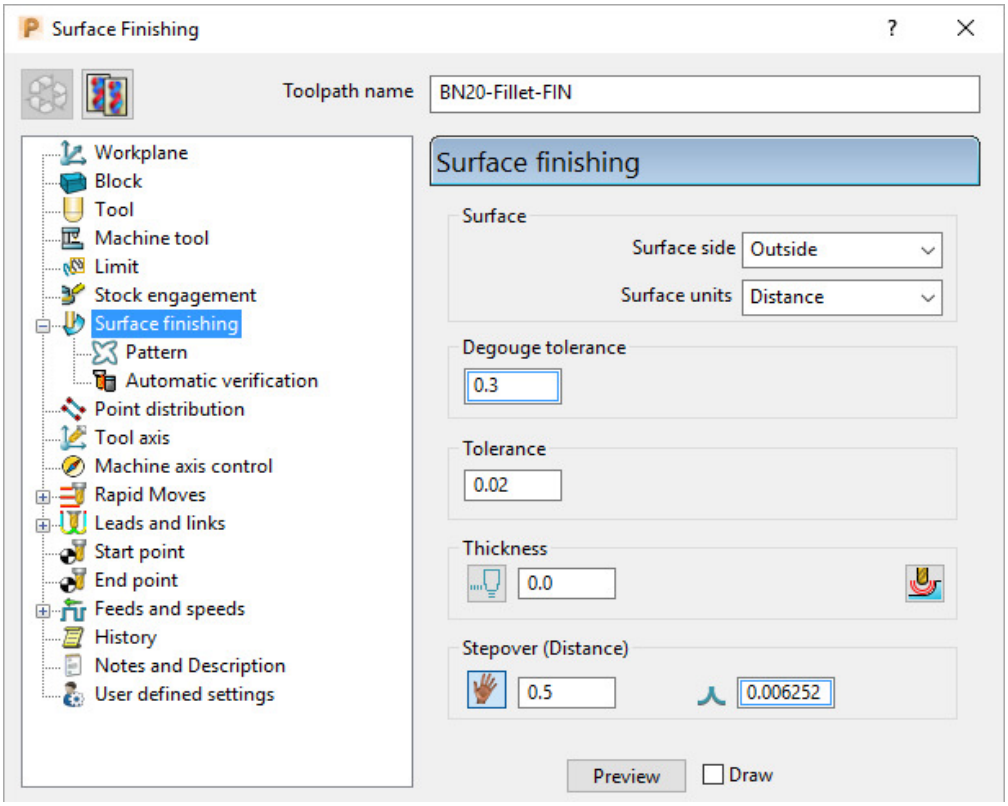
Copyright © Autodesk

10 Select the fillet running around the base of the main component sidewalls.

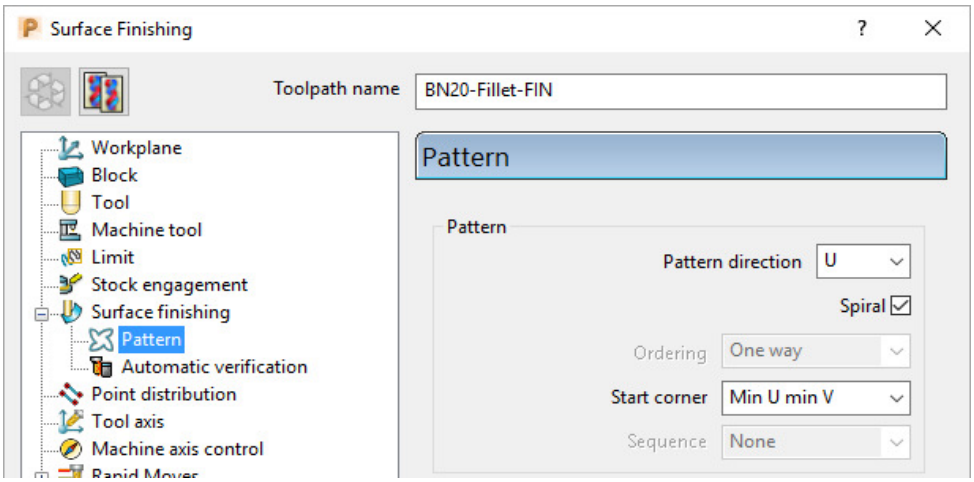


11 Select the **Toolpath Strategies** icon and in the **Strategy Selector** - **Finishing** options select **Surface Finishing**.

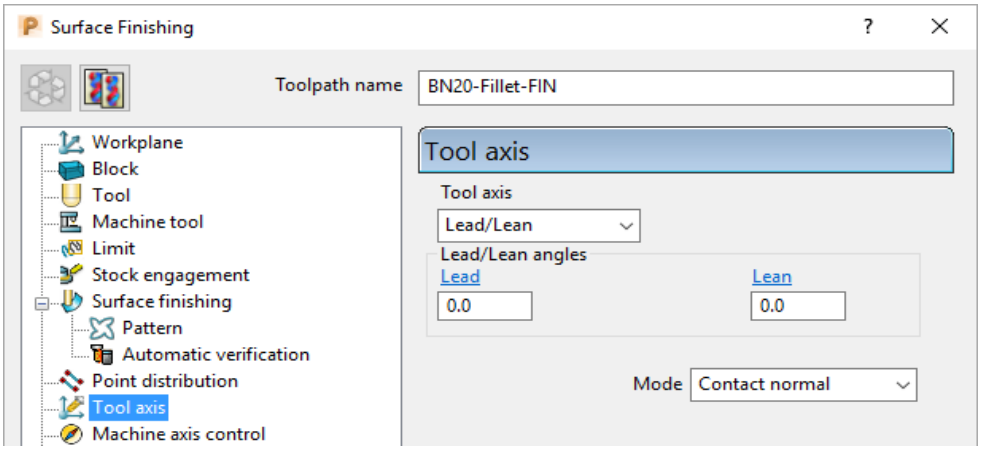
12 Input data into the **Surface Finishing** dialog exactly as shown:



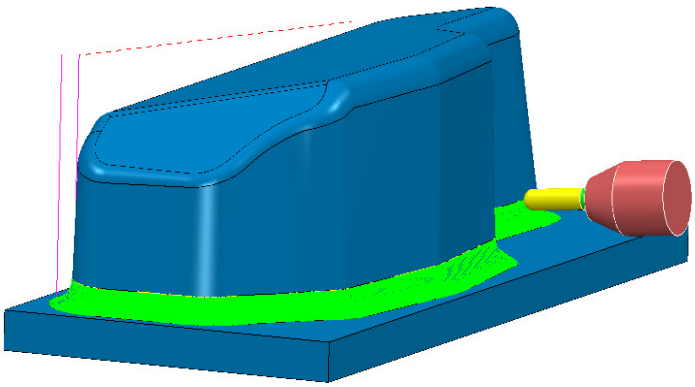
- Toolpath name: **BN20-Fillet-FIN**
- Stepover: **0.5**



- Pattern Direction: **U**
- Spiral: ***Ticked***



13 Select **Calculate** to create the **Surface Finishing** toolpath.



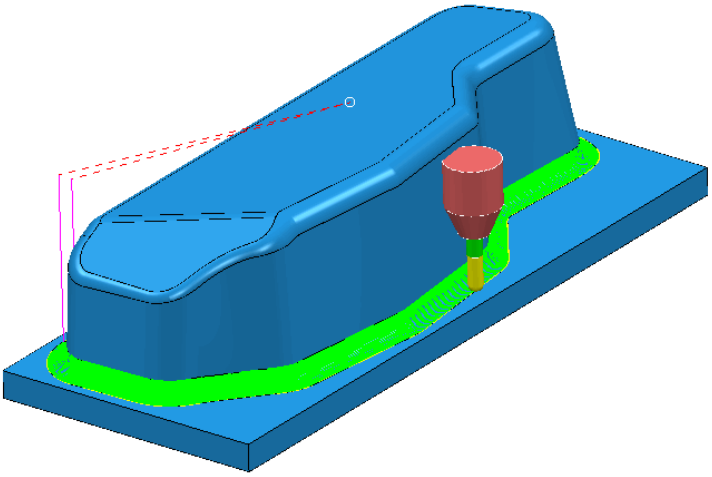
- The Tool Holder is in Collision with the Base.

14 From the local toolpath menu, select **Verify** – **Toolpath** to open the **Toolpath Verification** dialog.

15 Un-tick both **Split Toolpath** and **Adjust Tool**.

16 Then check for **Collisions** using a **Shank Clearance** of **1** and a **Holder Clearance** of **2**.

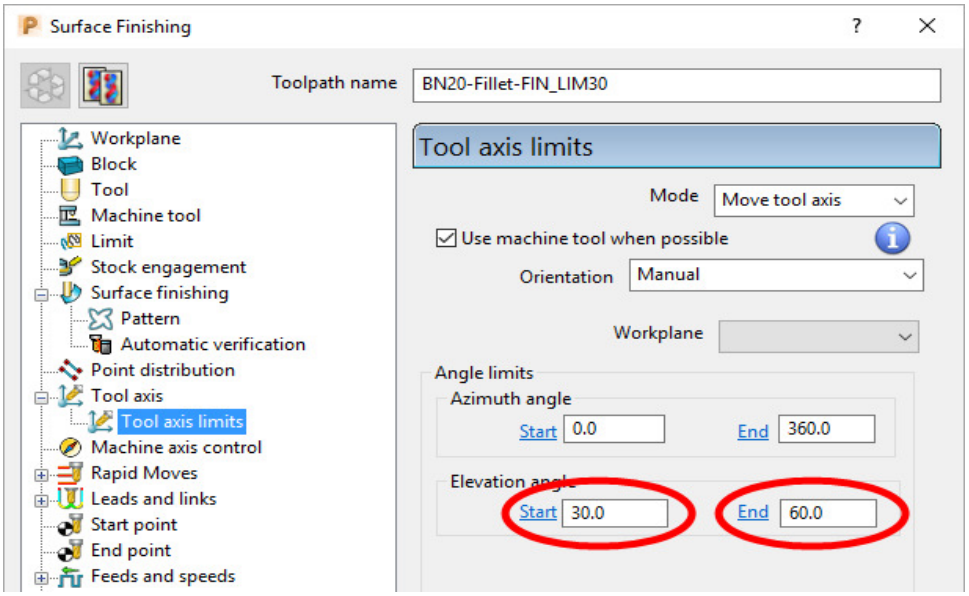
Copyright © Autodesk



- The **Tool Holder** is also in collision with the Sidewall.
- During **Projection Surface Finishing** the tool (**BN20**) is aligned normal to the Fillet Surface (if default **Lead\Lean 0** is applied). This is creating a Collision situation with both the sidewall and the base. **Tool Axis limits** will be applied to prevent this immediate problem as well as to keep the tool alignment within the machine tool rotary limits.

17 In the **PowerMill *explorer***, right-click the toolpath, **BN20-Fillet-FIN**, and select **Settings** to re-open the **Surface Finishing** dialog.

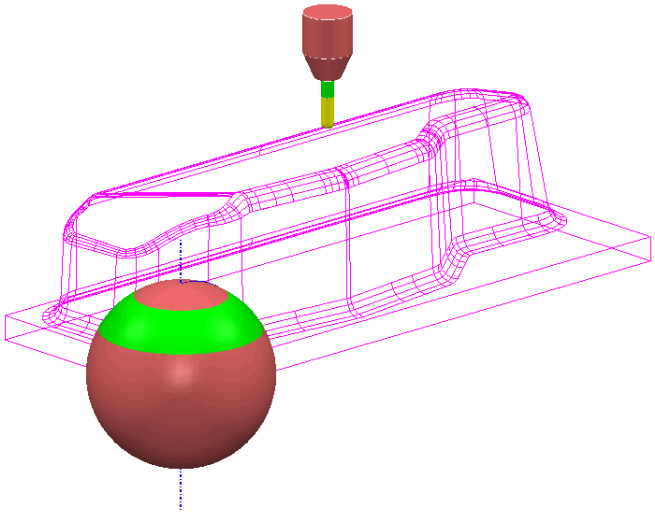
18 In the **Surface Finishing** dialog select to create a 'copy' and enter the toolpath name **BN20-Fillet-FIN-LIM30**.



19 Tick **Tool Axis limits** and move to the **Tool axis limits** page.

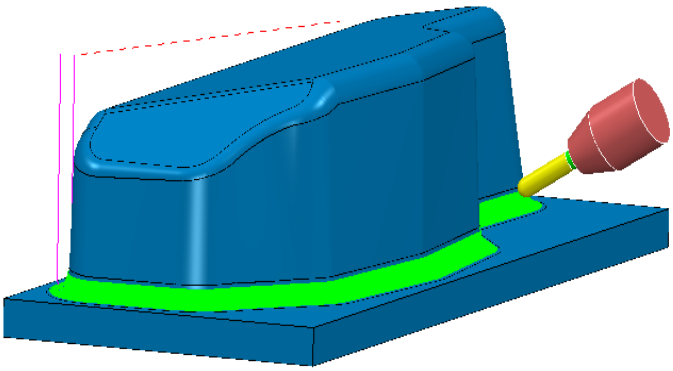
20 In **Mode**, select **Move tool axis** and for **Elevation angle** input:- **Start 30** and **End** 60.

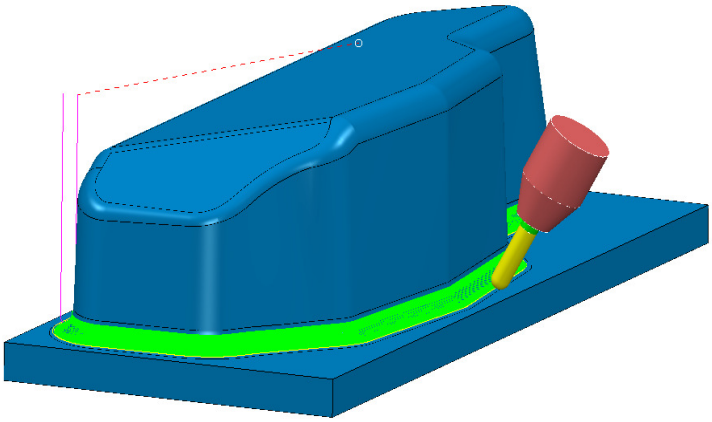
21 Select **Calculate** to create the new toolpath.



With the **Tool Axis Limits** applied the alignment will be restricted to operate between 30 and 60 degrees (**Elevation Angle**) relative to the **XY** plane. The sphere displaying the **Tool Axis** alignment limits (shaded green and pink) becomes visible when the box labelled **Draw Limits** is ticked.

The two illustrations below show the toolpath with the tool attached on the upper and lower tracks and the effect of the specified limits.





- The **Tool Holder** is no longer in collision with the Base or Sidewalls.

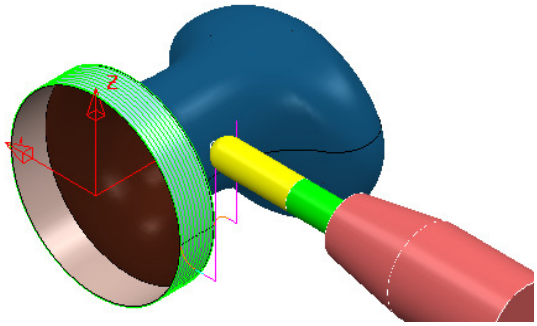
22 **Save** the **Project**.

Copyright © Autodesk

Applying Tool Axis limits to maintain a 4-Axis toolpath.

1. Select **File** - **Delete all** and **Tools** - **Reset Forms**.
2. **Open** the 'read only' **Project**:

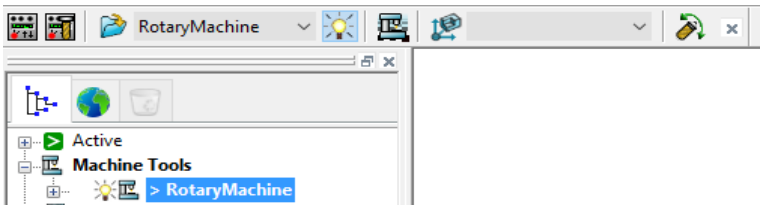
   ....\PowerMill_Data\five_axis\Tool_Limit\FixTo4Axis_Start



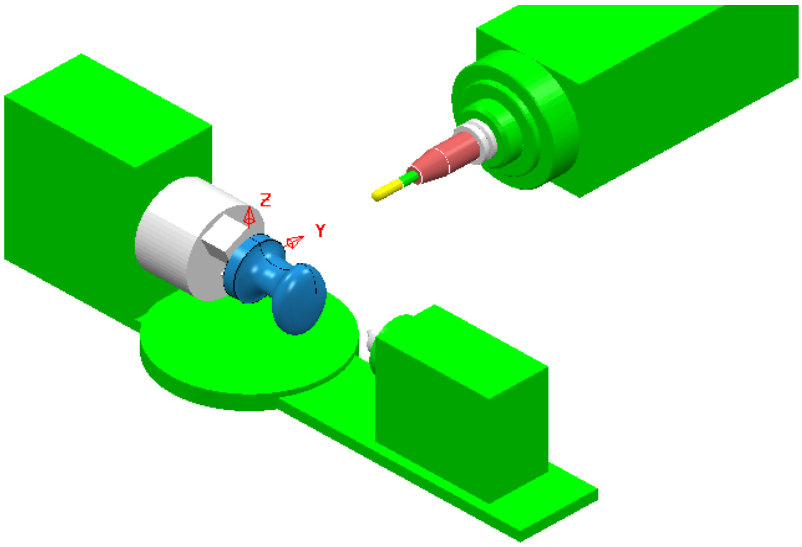
3. Select **File** > **Save Project As**:

   ....\COURSEWORK\PowerMill-Projects\FixTo4Axis-example1

4. If not already displayed, from the main pull down menus select **View** - **Toolbar** and tick **Machine Tool**.
5. In the **PowerMill *explorer*** open the **Machine Tools** folder and activate the **RotaryMachine** (double left click the icon).



6. Input **workplane 1** as the **Machine Tool** output datum.
7. Select an **ISO 2** view.



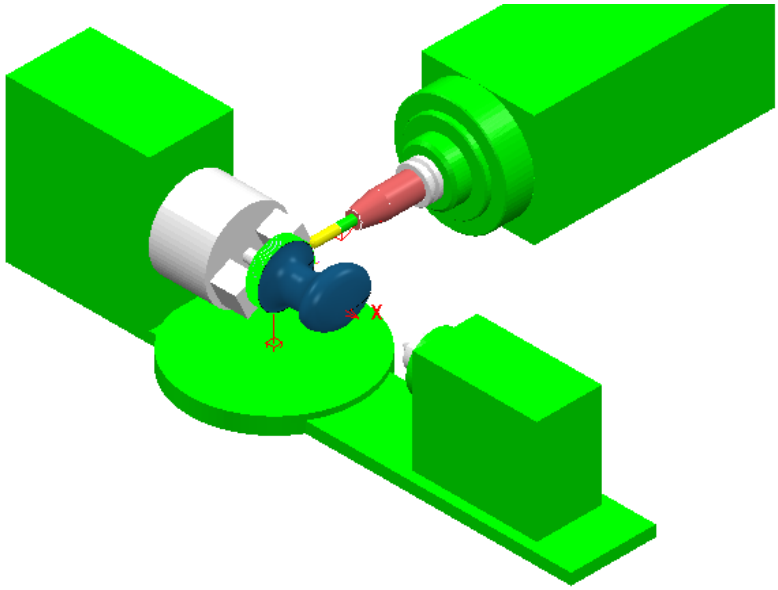
The **Project** contains a model of a 4-Axis Rotary Machine and two toolpaths.

The first toolpath is inherently 4 Axis while the second toolpath is 5 Axis.

The Rotary Machine is unable to produce the **Tool Axis** moves required by the second **Five Axis** toolpath. This toolpath can however easily be converted to **4-Axis** using the appropriate **Tool Axis Limits** options.

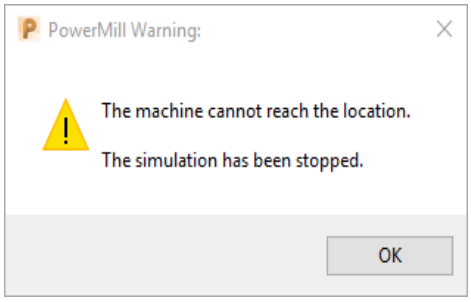
8 In the **PowerMill *explorer***, right mouse click on the toolpath **BN16-Base-SEM1** and select **Activate** followed by **Simulate from Start**.

9 Click the **Play** button and the simulation successfully runs all the way to the end.



10 In the **PowerMill *explorer***, right mouse click on the toolpath **BN16-Form-SEM2** and select **Simulate from Start**.

11 Click the **Play** button and the simulation immediately fails to proceed with the following message being displayed.



As the toolpath is fully ***Five Axis*** it impossible for the **Rotary Machine** to perform a **simulation**.

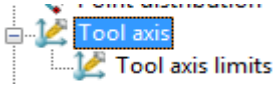
12 Abandon the simulation.

13 Right mouse click on the toolpath **BN16-Form-SEM2** and from the local menu select **Activate** it before selecting **Settings** to open the **Projection Surface** dialog.

Copyright © Autodesk

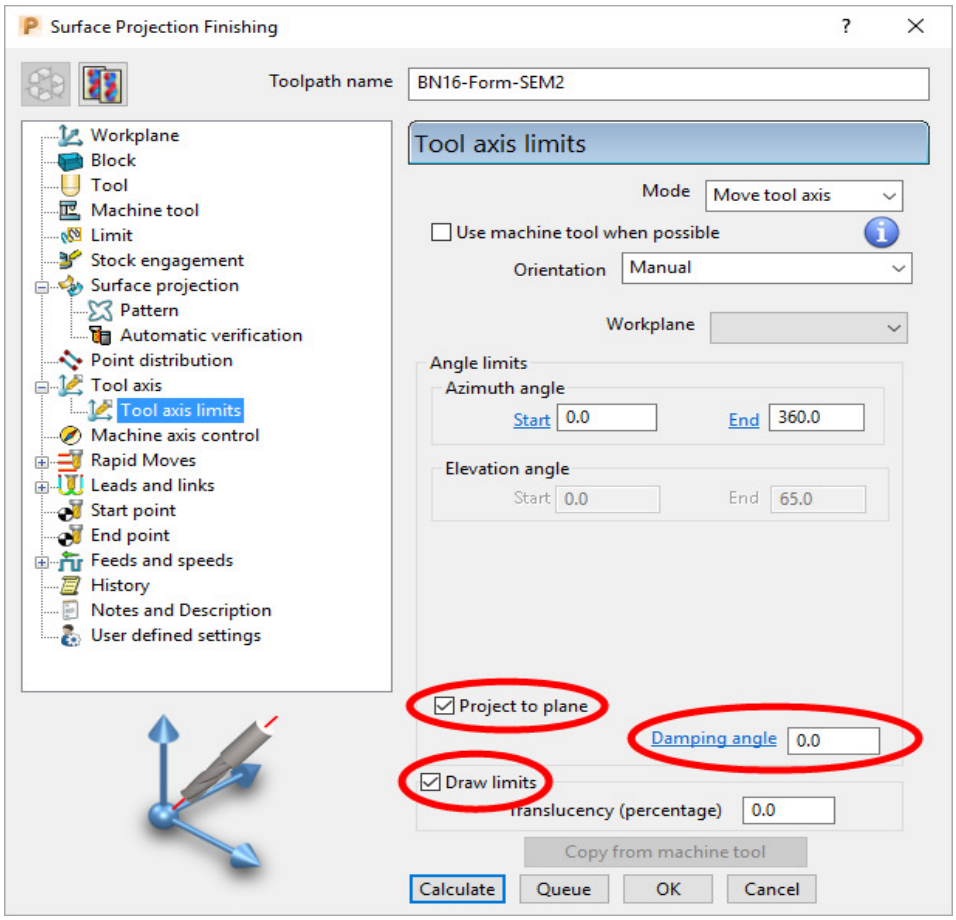
14 In the **Projection Surface** dialog select the **Recycle** toolpath option .

15 In the local **explorer** select **Tool axis** and tick the **Tool axis limits** box.

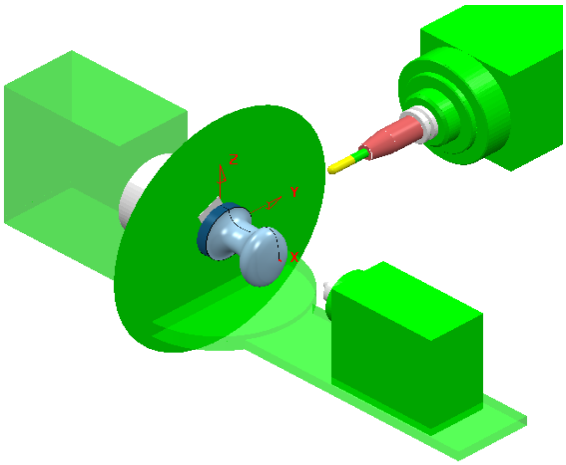


The **Tool axis limits** page will become available in the local **explorer**.

16 Select the **Tool axis limits** page and enter data exactly as shown below.

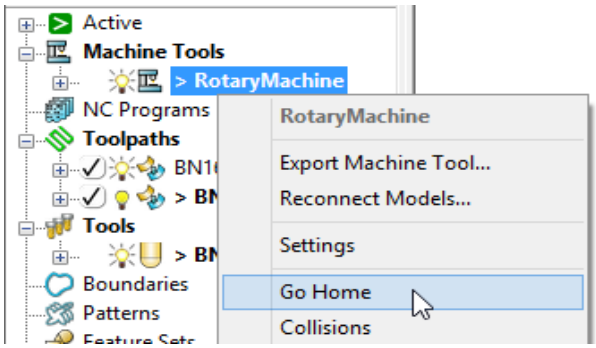


With **Project to plane** ticked the **Tool axis limits** are flattened to create a 4 axis limit (about the global transform in this case). A zero **Damping angle** will ensure that the 4 axis alignment is fully maintained.



17 In the **Tool axis limits** page, un-tick **Draw limits**.

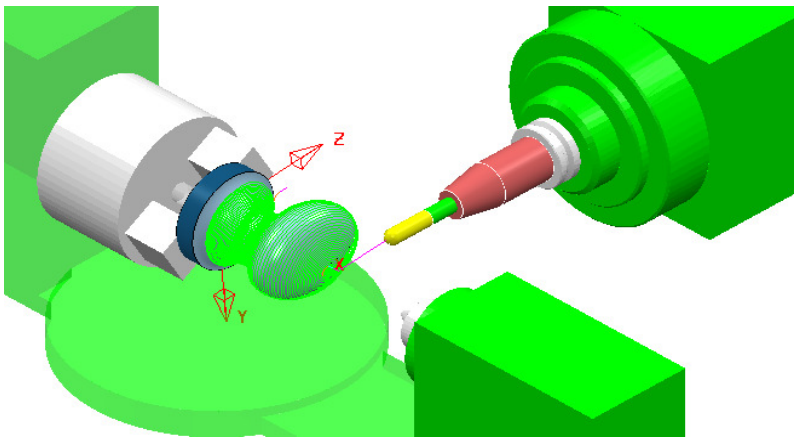
18 With the 3D form, **surface** selected **Calculate** the new 4-Axis toolpath.



19 In the local **explorer**, right mouse click on **Rotary Machine** and from the local menu select **Go Home**.

20 From the **PowerMill *explorer***, right mouse click on the new toolpath **BN16-Form-SEM2** and from the local menu select **Simulate from Start**.

21 Click the **Play** button and the 4-Axis toolpath simulation successfully runs all the way to the end.



22 **Save** the **Project**.

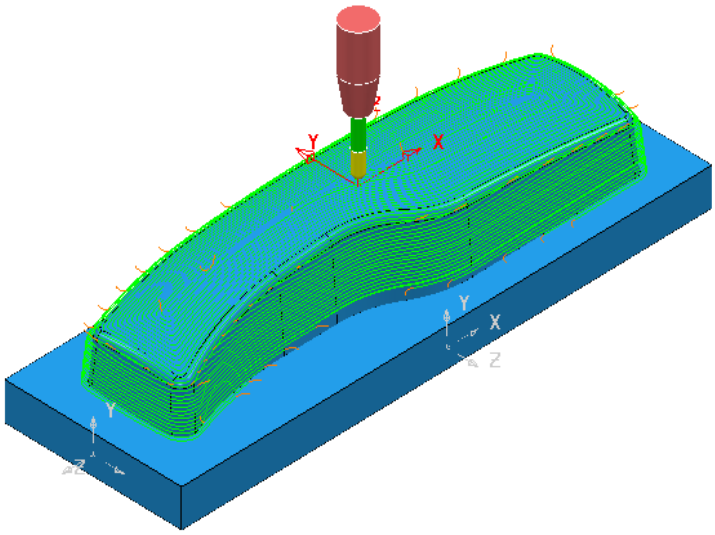
Copyright © Autodesk

Exercise 1

1 Select **File** - **Delete all** and **Tools** - **Reset forms**.

2 **Open** the 'read only' **Project**:

….\PowerMill_Data\five_axis\Punch3\Punch3_Start

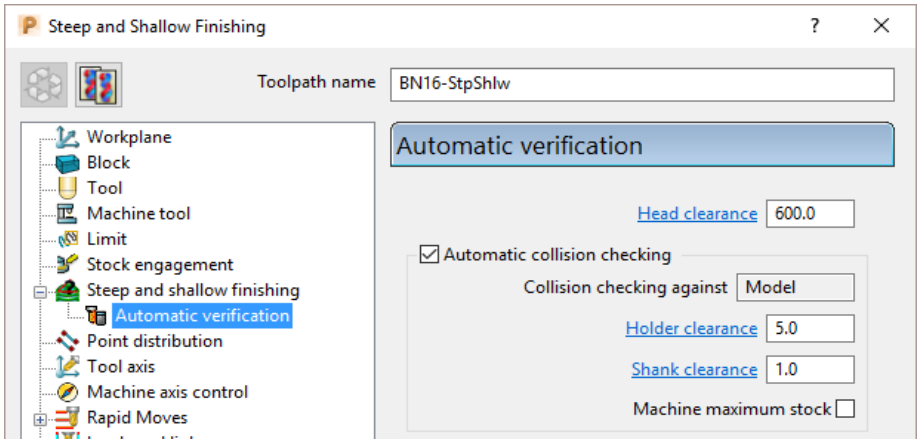


3 **Save Project As**:-

....\COURSEWORK\PowerMill_Projects\Punch3_EX1

The Project contains a **Steep and Shallow Finishing** toolpath **BN16-StpShlw** with a **Tool Axis** set to **Vertical**. The **Tool** is too short to safely machine some areas of the sidewalls. These potential collisions are avoided by the using **Automatic verification** (**Shank clearance 1** and **Holder clearance 5**).

The upper base surface is set with **Machining Mode** as **Collision** with a **Thickness** value of **0.5**.

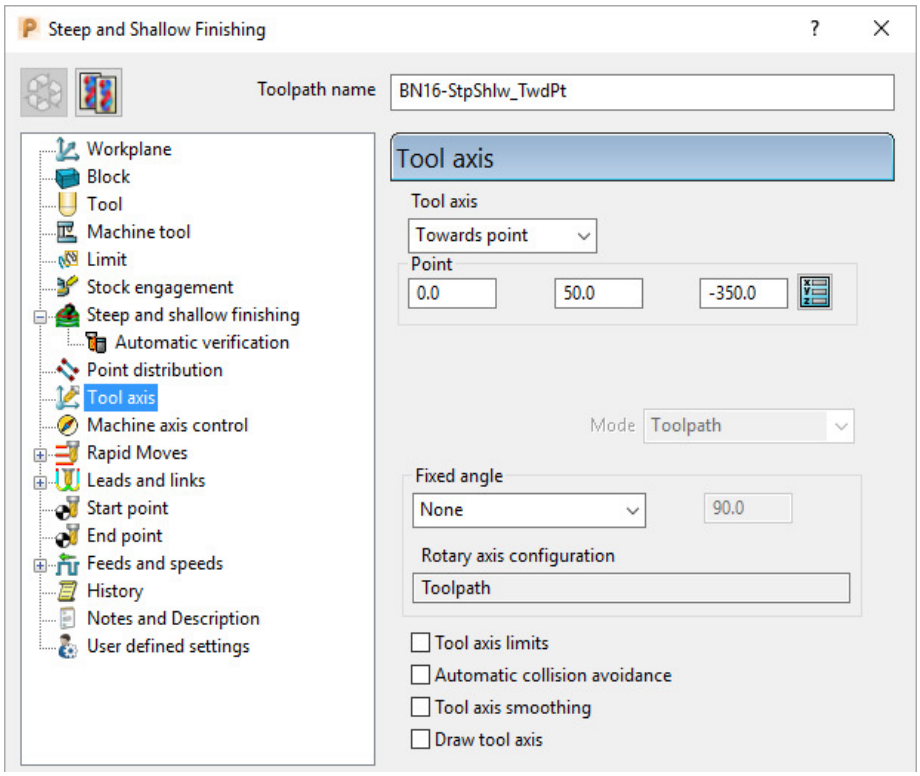


The object of the exercise is to create 2 copies of the existing toolpath the first with a suitable **Toward Point** - **Tool Axis alignment** applied and the second that is further limited to being **4 Axis** only (applied about the X axis).

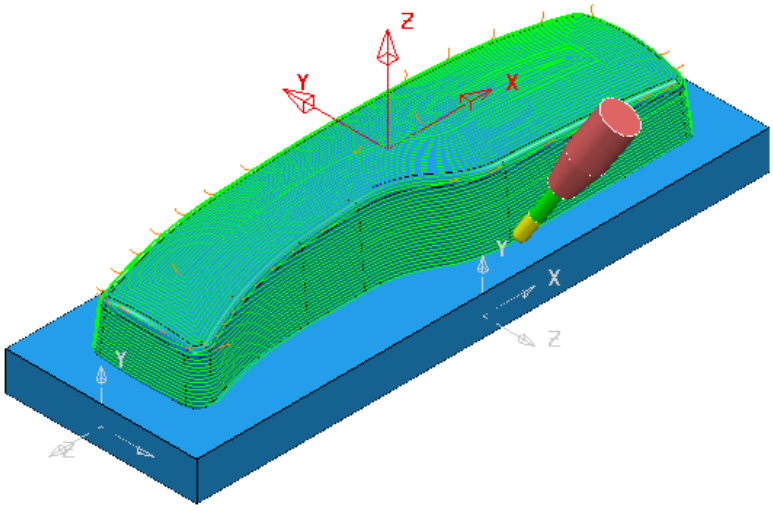
4 Activate the toolpath **BN16-StpShlw** and from the local menu select **Settings** to re-open the strategy dialog.

5 Create a **Copy** of the toolpath and rename it as **BN16-StpShlw_TwdPt**.

6 Select the **Tool Axis** page and select the option, **Towards Point** with the **X Y Z** positional values as **0 50 -350**.



7 **Calculate** the new toolpath.



With a suitable **Toward Point** - **Tool Axis alignment** applied the **Tool** is now angled away from the sidewalls providing collision free access to the deeper areas.

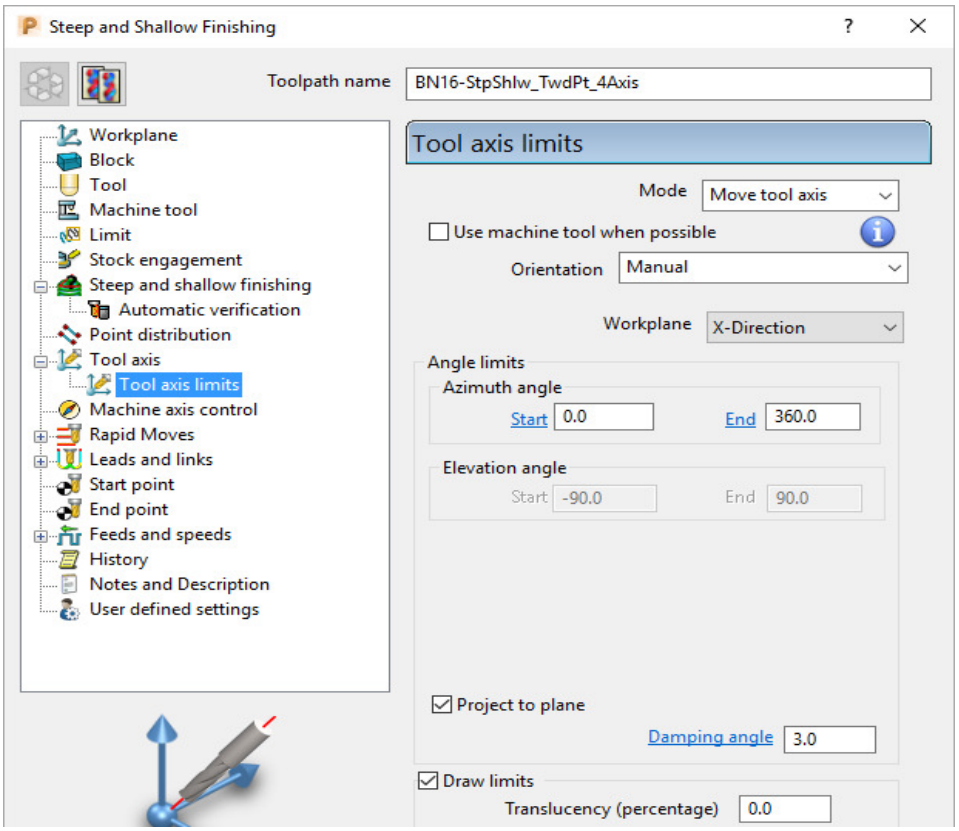
8 Activate the toolpath **BN16-StpShlw_TwdPt** and from the local menu select **Settings** to re-open the strategy dialog.

Copyright © Autodesk

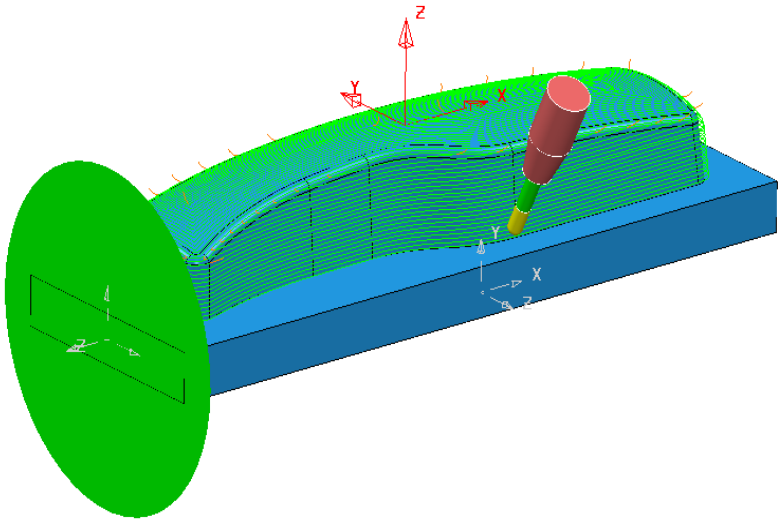
9 Create a **Copy** of the toolpath and rename it as **BN16-StpShlw_TwdPt_4Axis**.

10 Select the **Tool Axis** page and tick the **Tool axis limits** box.

11 Select the newly appeared **Tool axis limits** page in the local **explorer** and input the data into the dialog exactly as shown below.



12 **Calculate** the new toolpath.



This strategy is limited to operate as 4 Axis only.

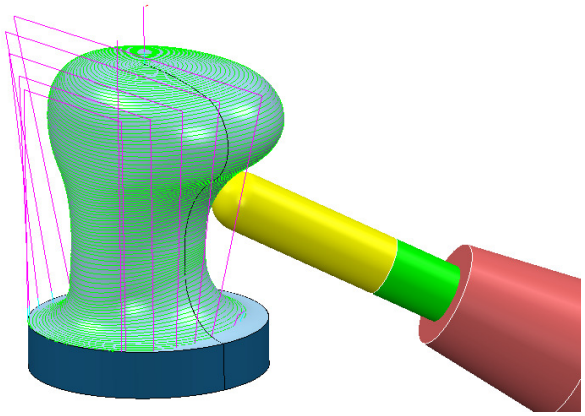
13 **Save** the **Project**.

Exercise 2

14 Select **File** - **Delete all** and **Tools** - **Reset forms**.

15 **Open** the 'read only' **Project**:

....\PowerMill_Data\five_axis\joint_5axismc\JointAxisLimits_Start



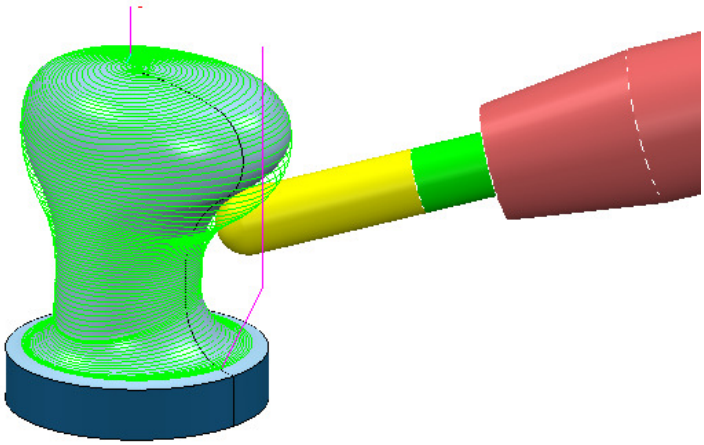
16 **Save Project As**:-

….\COURSEWORK\PowerMill_Projects\JointAxisLimits-EX1

The **Project** contains a **Surface Finishing** toolpath with a **Tool Axis** setting of **Lead 0** and **Lean 0**. This causes the **Tool** to be aligned to the ***surface normals*** at all points on the toolpath adjacent to the selected surface. The tool alignment is not prevented from exceeding the machine tool, rotary limits or to clear overhanging areas of the model.

17 Recycle the **Surface finishing** toolpath using **Tool axis limits** between **Elevation Angles** of **15** and **65** degrees using **Mode** - **Move tool axis**.

18 Perform a toolpath **Simulation** followed by a full **Gouge** and **Collision** (**Shank Clearance 2** – **Holder Clearance 3**) check on the modified **Surfacing Finishing** toolpath.



19 **Save** the **Project**.

Copyright © Autodesk

10. Tool Axis Editing

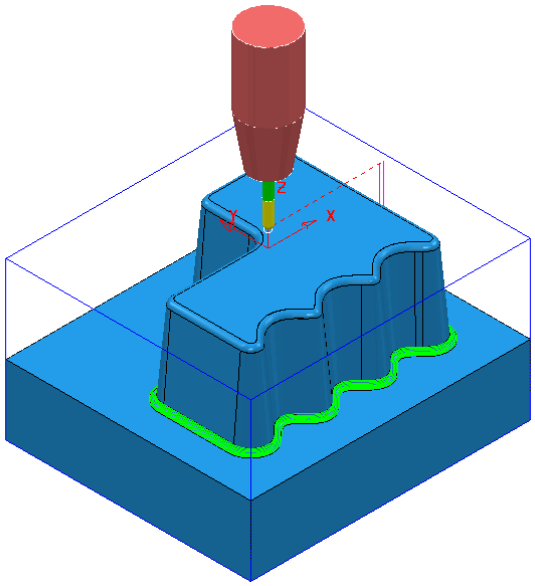
Introduction.

In some applications the type of **Tool Alignment** option applied to a **Five Axis** strategy can result in unnecessary, exaggerated rotary movements while the cutter runs along a toolpath.

In some cases, these exaggerated movements can result in **Tool Holder** or **Machine Tool** collisions. To reduce this, it is possible to edit the toolpath to have an alternative **Tool Alignment** within a user defined, area.

1 **Open** 'read only' the **Project**:

....\PowerMill_Data \five_axis\ToolAxisEditing\EditToolAxis_Start



The **Project** contains a **Corner Along** strategy created using a **Dia 5 Ball Nosed** tool with a **Lean Angle** of **45** Degrees applied.

2 From the **main** pulldown menus select **File** > **Save Project as**:

....\COURSEWORK\PowerMill-Projects\EditToolAxis

3 In the **main** pull down menus select **View** > **Toolbars** > **Machine Tool** to raise the **Machine Tool** definition toolbar.

4 Click on the **Import Machine Tool Model** icon and from the **...\PowerMill_Data\MachineData** directory, select the file **dmu50v.mtd**.

5 Input the **workplane *Base*** as the datum for the **Machine Tool Simulation**.

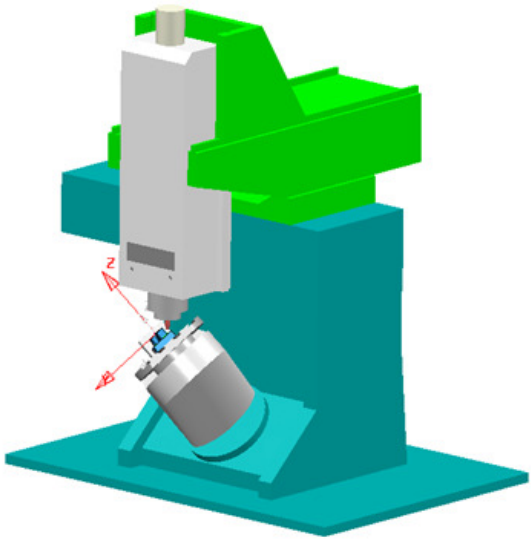
6 Ensure the **Draw\Undraw** machine tool icon is selected to display the machine tool.

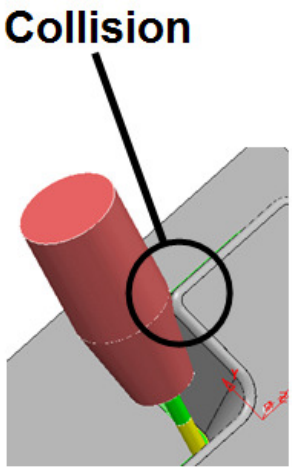
7 Select an **ISO 1** view and zoom the machining area.

8 Select **Floor View** from the **Simulation** toolbar.

9 Right-click on the toolpath **BN5-Rest-Lean45** and from the local menu, select **Simulate from Start**.

10 Move the simulation speed, ***slider*** to a slow setting and then click the ***Play*** icon and observe the machine tool movement.



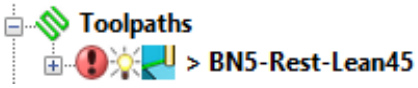


In addition to there being an excessive amount of table rotation while the undulating sidewall is machined there is also a **Tool Holder collision** on the sidewall internal corner (see above right graphic).

11 From the local **BN5-Rest-Lean45 *toolpath*** menu, select **Verify** - **Toolpath** and in the dialog, un-tick both **Split Toolpath** and **Adjust Tool**.

12 Select ***Check*** **Collisions**, with a **Shank Clearance 1** and **Holder Clearance 2** and **Apply** the dialog.

The **Toolpath** is now displayed red in the **PowerMill explorer** confirming that the **Shank** and/or **Holder** are colliding with the model.



Copyright © Autodesk

The 2 issues will be fixed by applying localised Tool Axis alignment modifications on the toolpath. To enable normal viewing while the changes are made, the Machine Tool model will temporarily be removed from the Machine Tool Simulation toolbar.

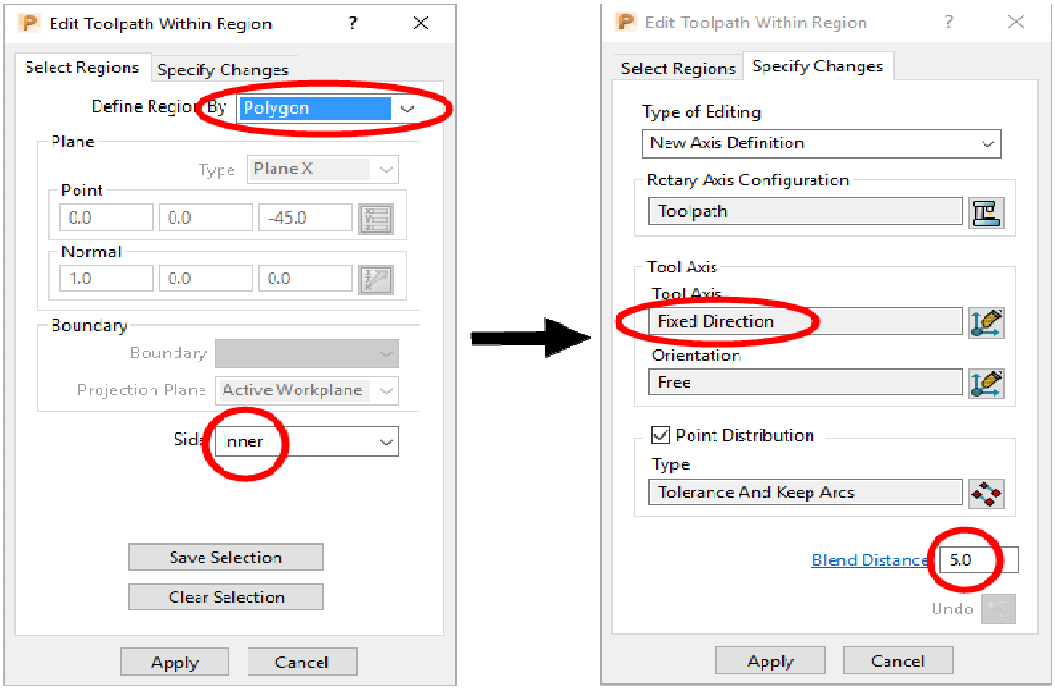
13 In the **Machine Tool Simulation** toolbar select **None** to remove **dmu50v.mtd** from the selection input box leaving it blank.

14 Select a **View from top** ready for snapping the corners of a ***polygon*** to define the first area of the toolpath to undergo ***Tool Axis Editing***.

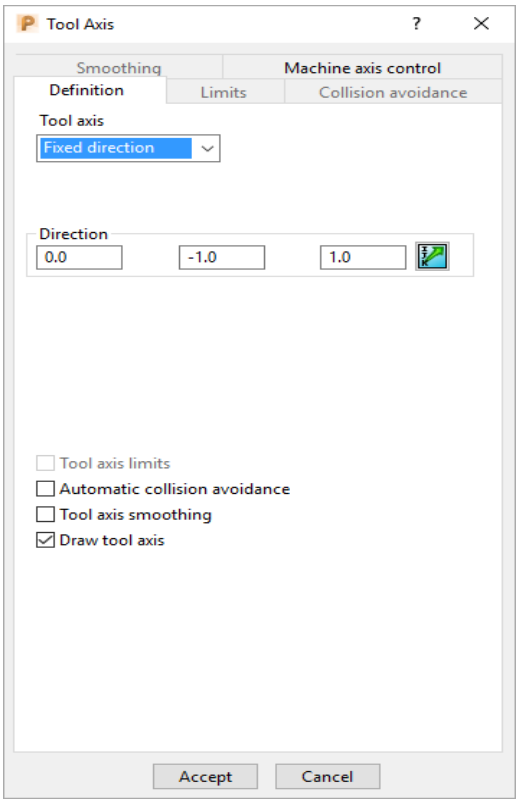
15 On the keyboard select **Ctrl H** to display the **Cross Hair** on the cursor position.

16 In the **PowerMill *explorer***, right-click on the toolpath **BN5-Rest-Lean45** and select **Edit** > **Edit Within Region** from the local menus to access the following dialogs.

17 In the (default) **Select Region** dialog set ***Define Region By*** - **Polygon** and ***Side*** - **Inner** (as shown below left).



18 Select the **Specify Changes** tab, and set **Tool Axis** as **Fixed Direction** and input a **Blend Distance** of **5**, before selecting the **Tool Axis** icon .



19 Input values into the **Direction** vector (Along the tool axis) as:-

I **0.0** ***J*** **-1.0** ***K*** **1.0**

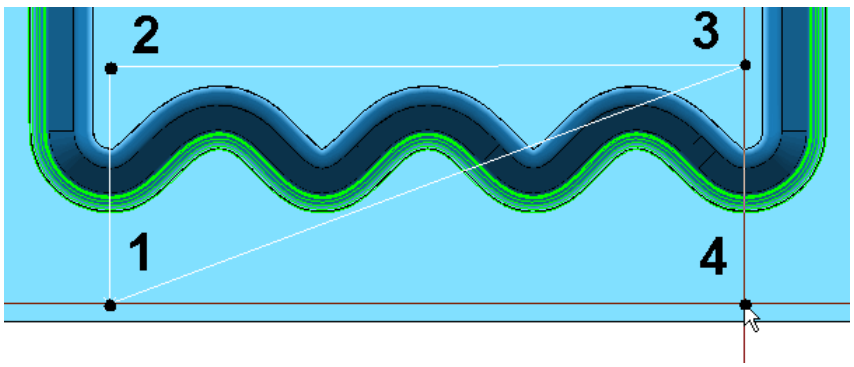
20 In the **Tool Axis** dialog, tick the **Draw Tool Axis** box before selecting **Accept** (to return to the **Edit Toolpath Within Region** dialog).

21 From the **main** pulldown menus select **Draw** > **Cursor** > **Cross Hair**.

*This will display **cross hairs** running through the **cursor position**, to assist visual alignment when 'snapping' points.*

22 Click on the **Select Regions** tab to access the original options.

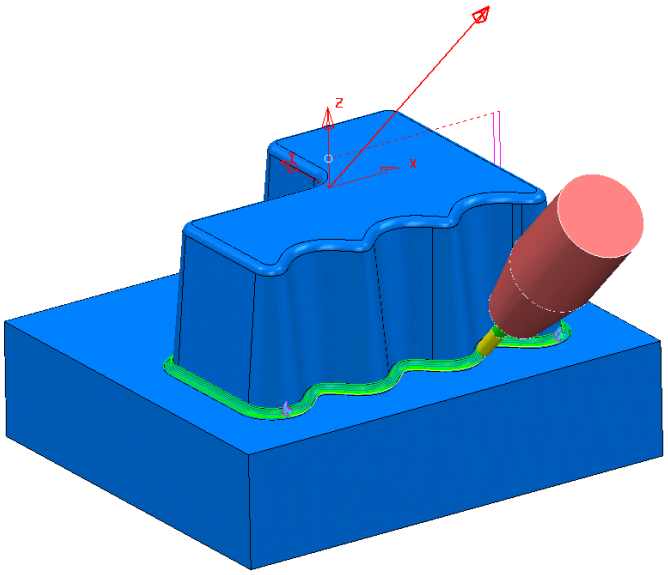
23 Left-click to snap 4 corners of the **Polygon** inside where the ***Tool Axis Edit*** is to occur, and click **Apply**.



The above illustration is at the stage where the user is ready to **snap** the **4th** and final point on the **Polygon** (Cross hairs are mid grey).

Copyright © Autodesk

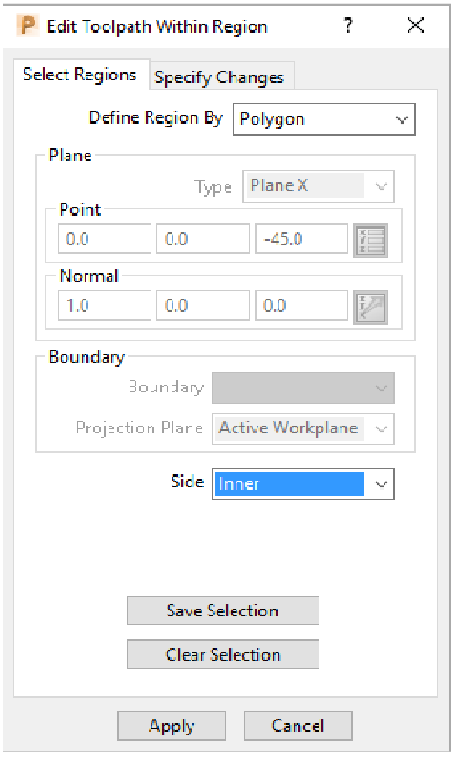
24 Select an **ISO 1** view and **simulate** the toolpath to observe that the **Tool Axis Alignment** is no longer performing a series of rotary movements along the undulating sidewall.

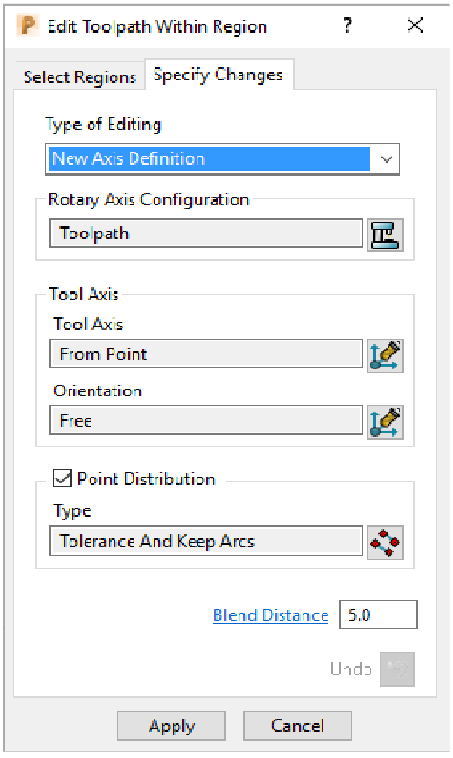


25 Select a **View from top**.

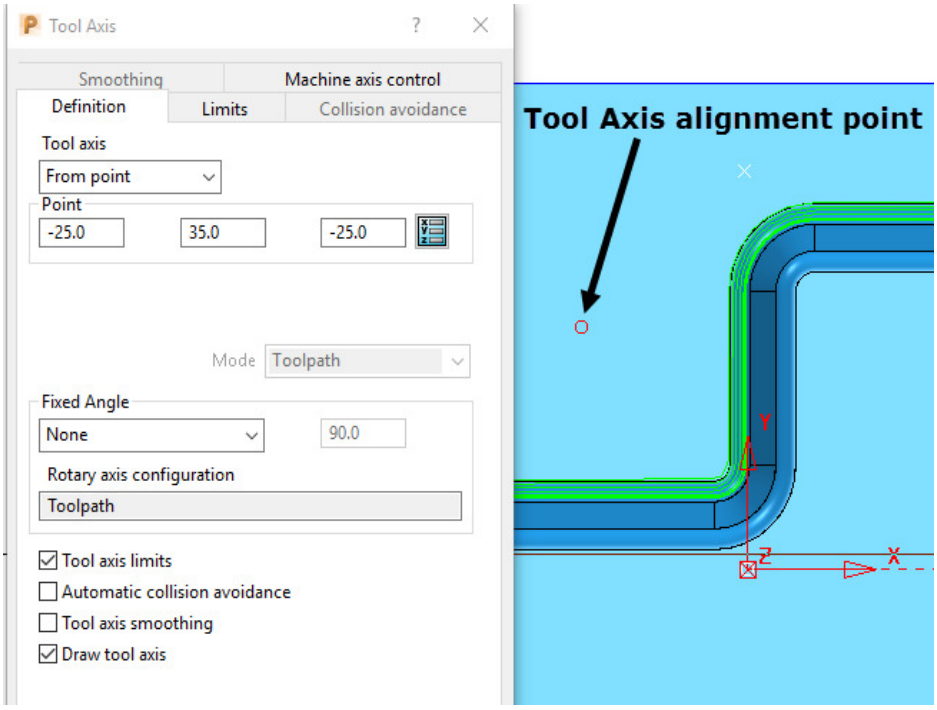
26 In the **PowerMill *explorer***, right-click on the toolpath **BN5-Rest-Lean45** and select **Edit** > **Edit Within Region** to open the **Edit Toolpath Within Region** dialog again.

27 In **Select Regions** select **Polygon** with ***Side*** - **Inner**, and then click the **Specify Changes** tab.

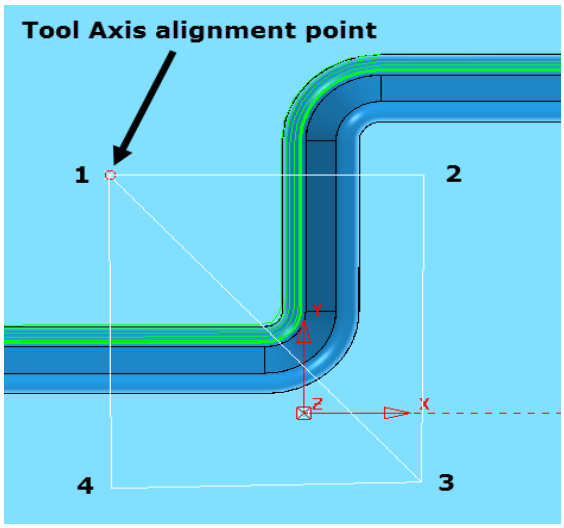




28 Set ***Tool Axis*** - **From Point** with the coordinate values **X-15 Y 25 Z-25**, and then tick the box labelled **Draw Tool Axis** before selecting **Accept**.



29 This time snap 4 corners of the polygon to form a square, aligned as close as possible to the **Tool Axis Alignment** point, and click **Apply**.



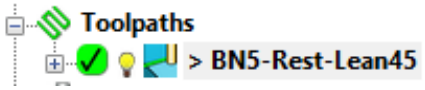
30 Select an **ISO 1** view.

31 From the local **BN5-Rest-Lean5** ***toolpath*** menu, select **Verify** - **Toolpath** and in the dialog, un-tick both **Split Toolpath** and **Adjust Tool**.

32 Select ***Check*** **Collisions**, with a **Shank Clearance 1** and **Holder Clearance 2** and **Apply** the dialog.

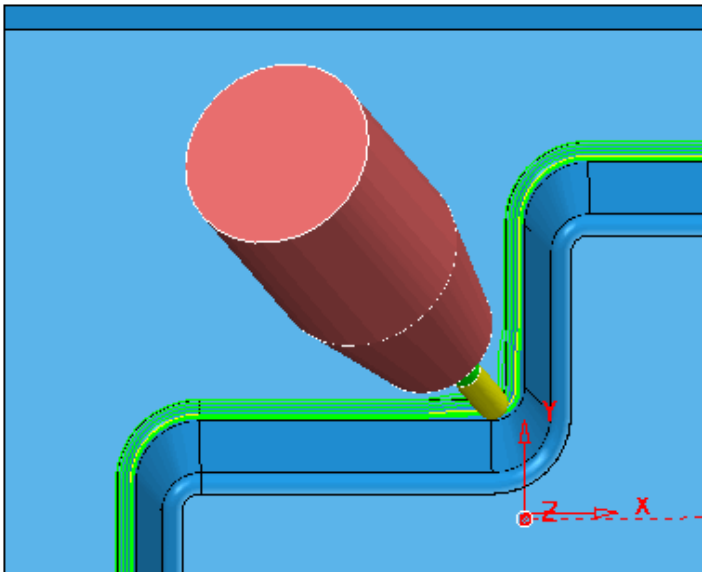
33 Select ***Check*** **Gouges** and **Apply** the dialog.

The **Toolpath** is now displayed green in the **PowerMill explorer** confirming that the toolpath is free of both **Gouges** and **Collisions**.

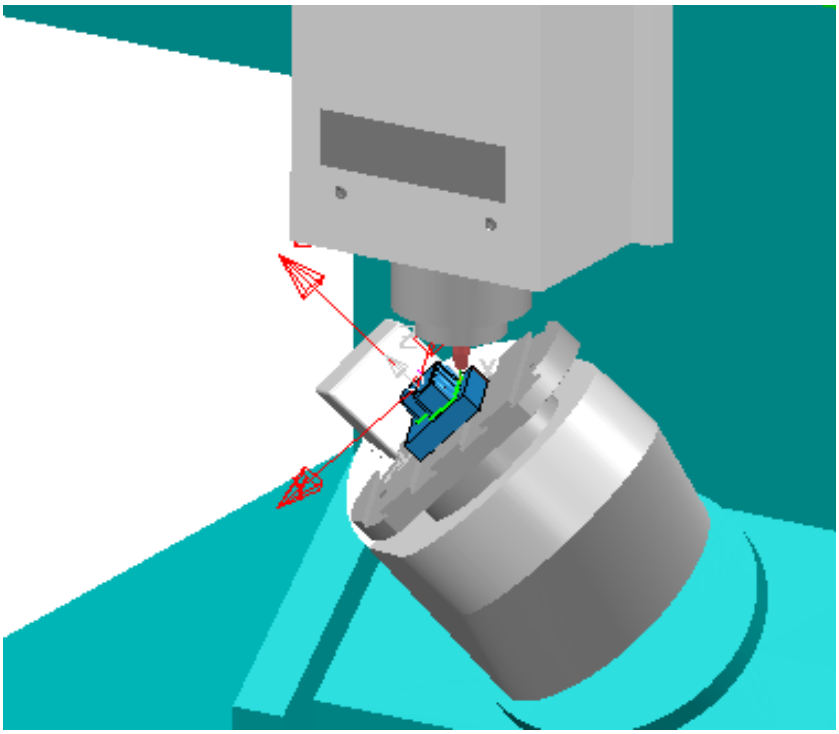


Copyright © Autodesk

34 **Simulate** the toolpath to observe that with the new **Tool Axis Alignment** the tool holder is no longer in a ***collision*** condition with the sidewall internal corners.



35 Perform a **Machine Tool Simulation** again using the **DMU50V** with the **Floor View** selected.



This time the unnecessary, 'rocking' movements about the rotary axis are minimised and the **Tool Holder collision** removed.

36 Select **File** - **Save Project**.

Copyright © Autodesk

11. Machine Tool Simulation and Verification

Introduction

The ability to check for potential **Machine Tool** - **Component collisions** is an essential requirement for **Five Axis** applications. To provide this capability **PowerMill** contains an additional **Machine Tool** toolbar for use with the toolpath **Simulation** options.

The standard **Machine Tool Simulation** is purely visual and it is the user's responsibility to identify collisions. An additional, 'cost option' is available where the **Machine Tool Simulation** will stop if a collision situation is identified. At this point a warning box will be displayed. The tool along with the affected parts of the ***Machine Tool model*** will turn red, and once acknowledged (by clicking **OK**) all moves in a collision condition will be registered in a list.

The individual component parts of a **machine tool** (eg; Main Body, Head, Rotary Table, Cradle, etc) are stored as a set of individual triangle models. These are registered within an **mtd** file that controls the **orientation** and **position** of the individual triangle models during a **simulation**.

Three basic, multi-axis **Machine Tool Simulation** (**.mtd**) files are supplied within the **PowerMill** installation data. A typical location for a **C** drive install:

C:\Program Files\Delcam\PowerMill13.0.06\file\examples\MachineData

A further selection of sample **mtd** files, based on actual **machine tools** are located on the training PCs in:

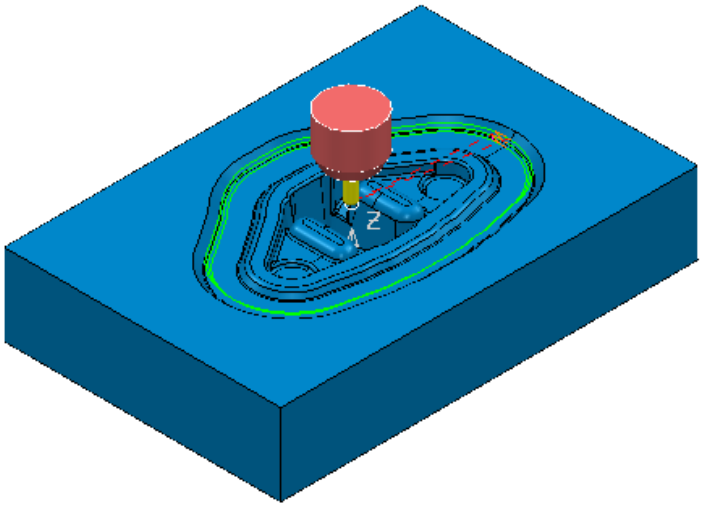
….\PowerMill_Data\MachineData

*It is essential that all models along with controlling moves and limits used in a **Machine Tool Simulation** (**.mtd**) file are an accurate copy of the actual Post-Processor and machine tool in use. Due to design variations, different set up criteria, and tolerance issues, each **Machine Tool Simulation** (**.mtd**) file and associated models must be tailor made and fine-tuned for each individual application.*

1 Select **File** - **Delete all** and **Tools** - **Reset forms**.

2 **Open** the read only **Project**:

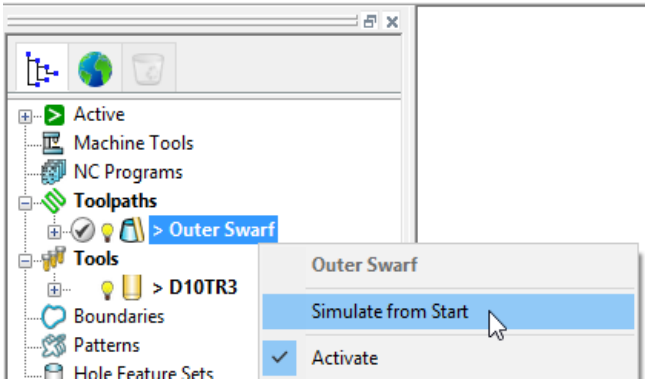
....\PowerMill_Data\five_axis\Collision_Simulation\Swarf_check



3 Select **File** - **Save Project As**:

....\COURSEWORK\PowerMill-Projects\MCTool-simulation

4 In the **PowerMill *explorer***, right mouse click the toolpath **Outer Swarf** and select **Simulate from Start**.



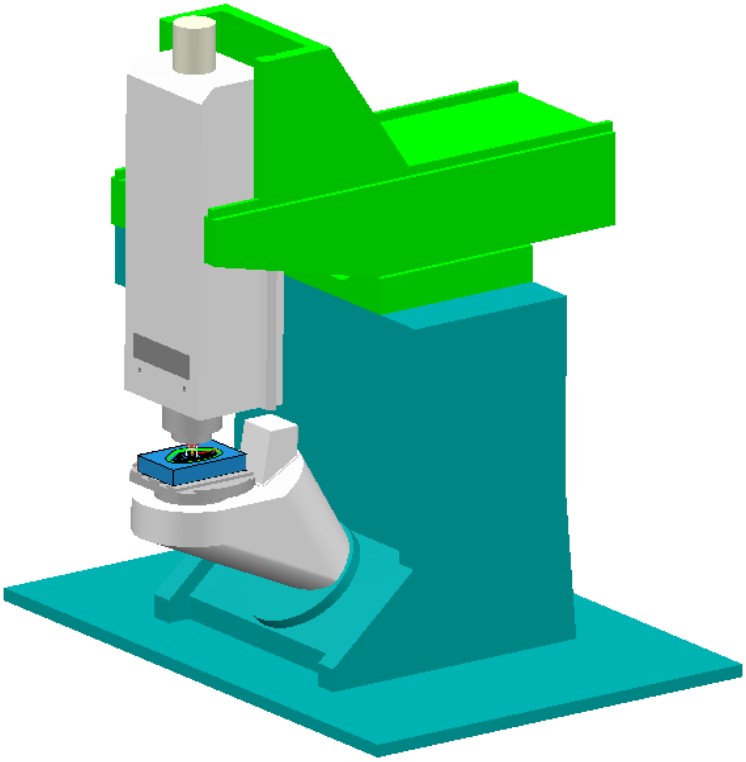
5 The **Simulation** toolbar will appear (if it is not already displayed).

6 In the **PowerMill *explorer*** right mouse click on **Machine Tools** and from the local menu select **Toolbar**.

7 Select the ***Import the Machine Tool Model*** icon and select the file **dmu50v.mtd** from:

....\PowerMill_Data\Machine Data\DMU\dmu50.mtd

Copyright © Autodesk



The **MTD** file contains the **Positional** and **Rotational** details for the individual, **Machine Tool**.

It is normal practice to create the **machine tool model**, with the global datum (Transform) positioned at the Top - Centre of the table.

The active cutting **Tool** is automatically positioned in the machine tool head.

8 Ensure the ***Draw\Undraw*** machine tool icon is selected to display the machine tool.

9 Select an **ISO 1** view and zoom into the machining area.

10 Select the **Floor View** icon from the **Simulation** toolbar.

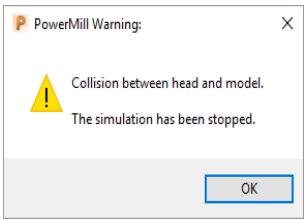
11 Select ***Display the simulation issues*** from the **Simulation** toolbar.

During a toolpath **Simulation**, the **Simulation Issues** dialog will display information on collision positions.

With the **Position** tab selected machine tool positions will be displayed. The values on the left hand side refer to the **Axis Address Letters** and their associated values.

This machine tool has five axes, **A** and **B** are rotary with **X**, **Y** and **Z** as linear.

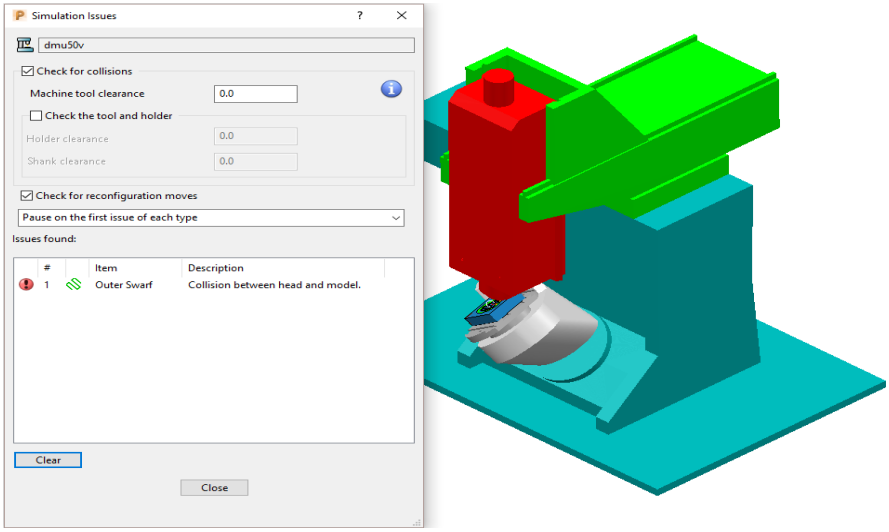
12 Start the **Simulation** to observe the virtual machining of the component.



A collision has been detected and a ***PowerMill Warning*** message displayed.

The warning message is only displayed for the first collision.

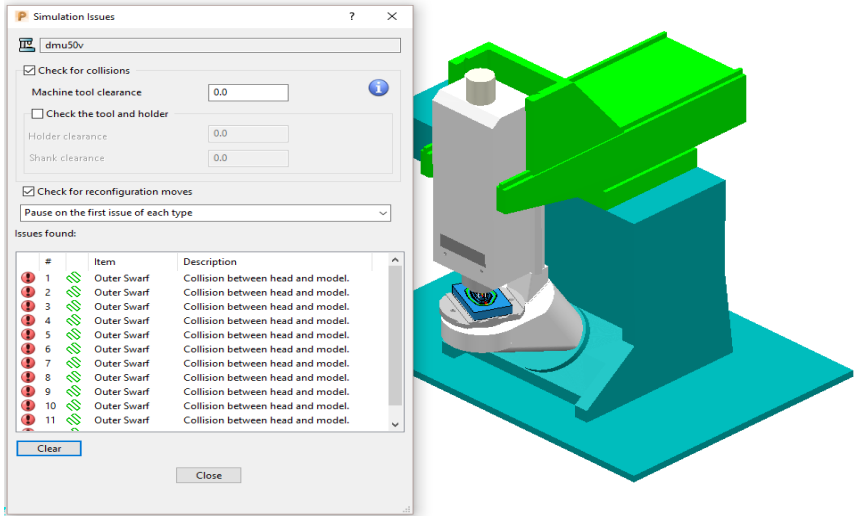
13 Select **OK** to close the **PowerMill Warning** dialog.



The first collision move is registered and displayed in the **Issues found** pane.

At present only the first collision is displayed until the **Simulation** is continued.

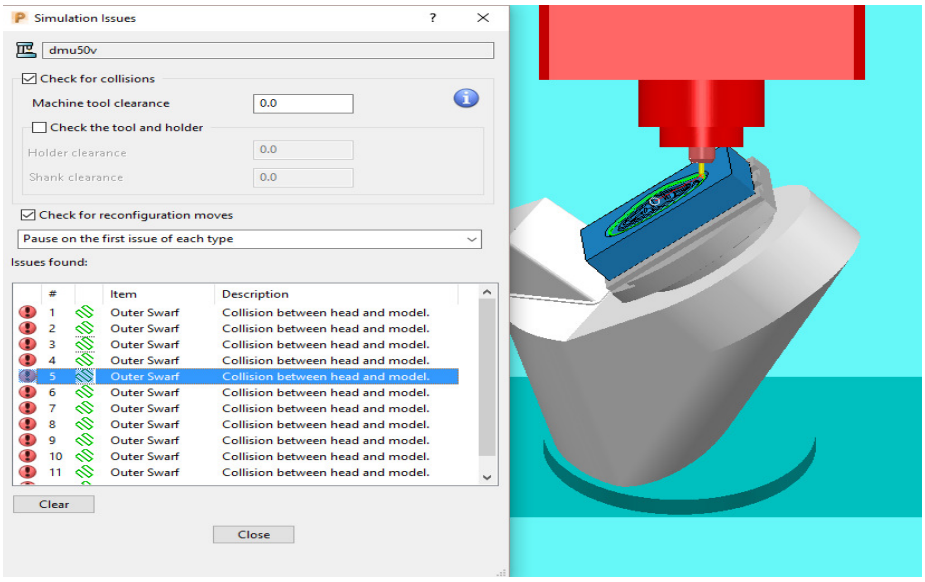
14 Continue the **Simulation**.



On completion of the simulation the machine tool is displayed in the normal colour scheme.

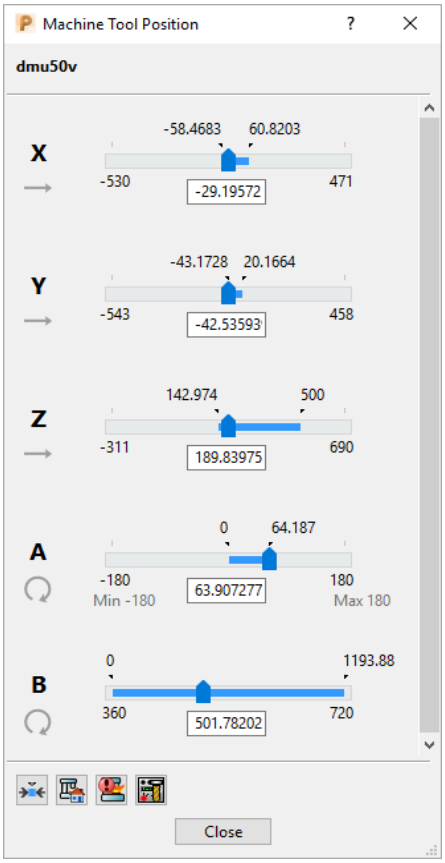
15 Select a ***Collision Move*** from the list.

Copyright © Autodesk



The **simulation** will move directly to the selected position in the form so that the collision (top right corner of component) can be viewed.

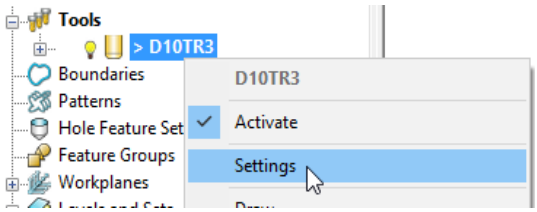
16 Select ***View and adjust the Machine Tool position*** from the **Simulation** toolbar.



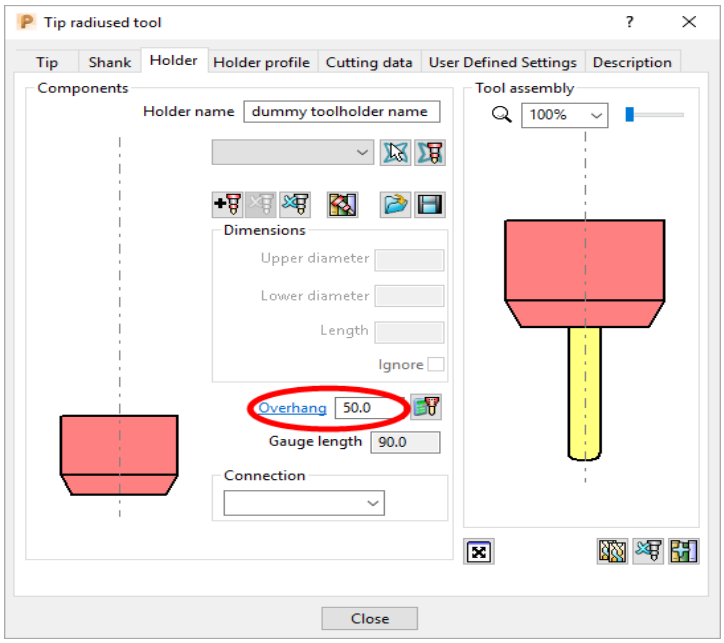
The current positional values are displayed in the **Machine Tool Position** dialog.

With a clear view of the collision, the user can assess how best to make the appropriate changes to avoid it. In this case, substituting the cutter with one with increased Tool Length would be the easiest solution.

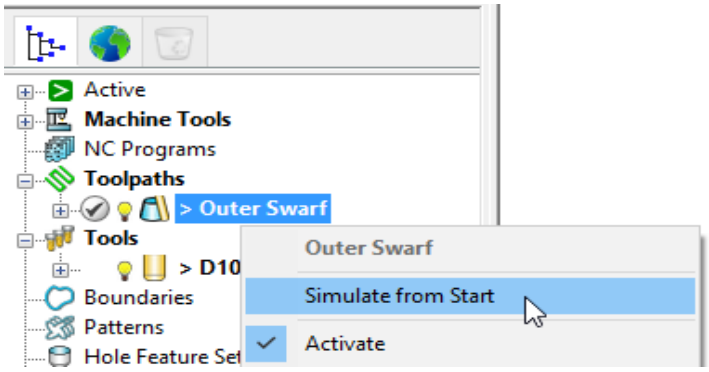
17 In the **PowerMill *explorer*** right mouse click on the tool **D10TR3** and select **Settings**.



18 Select the **Holder** tab on the tool dialog and modify the **Overhang** to **50**.



19 Close the dialog and in the **PowerMill *explorer*** select **Simulate from Start** on the **toolpath** named '**Outer Swarf**'.



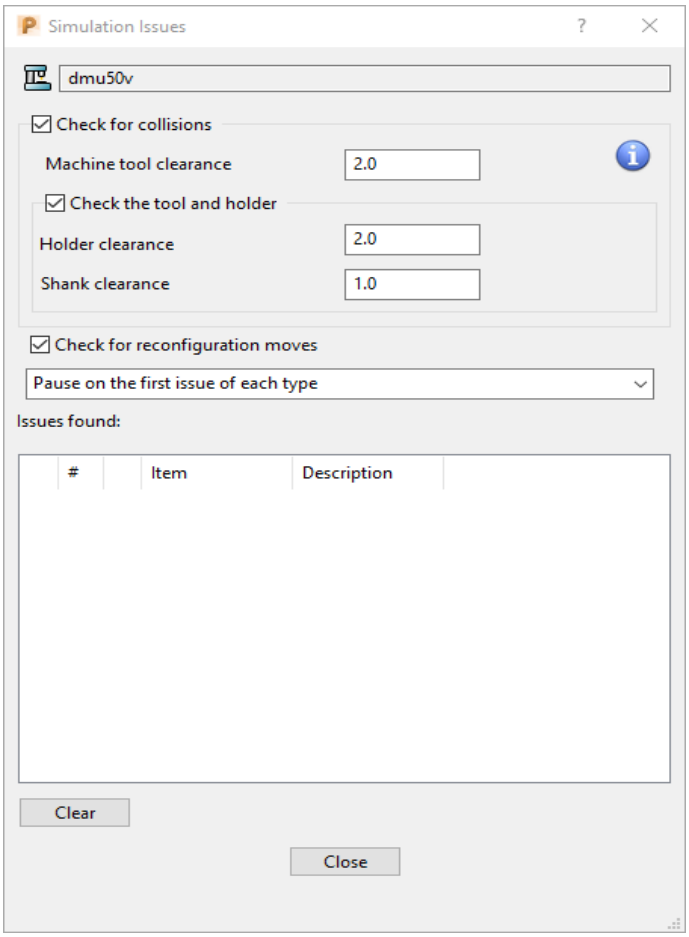
20 Select ***Switch on the Machine Tool Display*** from the **Simulation** toolbar.

21 Select the **Collisions** tab on the dialog.

22 Select **Clear** on the dialog to clear any existing collisions.

Copyright © Autodesk

23 Start the **Simulation** .



The **Machine Tool Collisions** dialog remains blank indicating that no collisions have been detected.

24 Select **File** > **Save Project**.

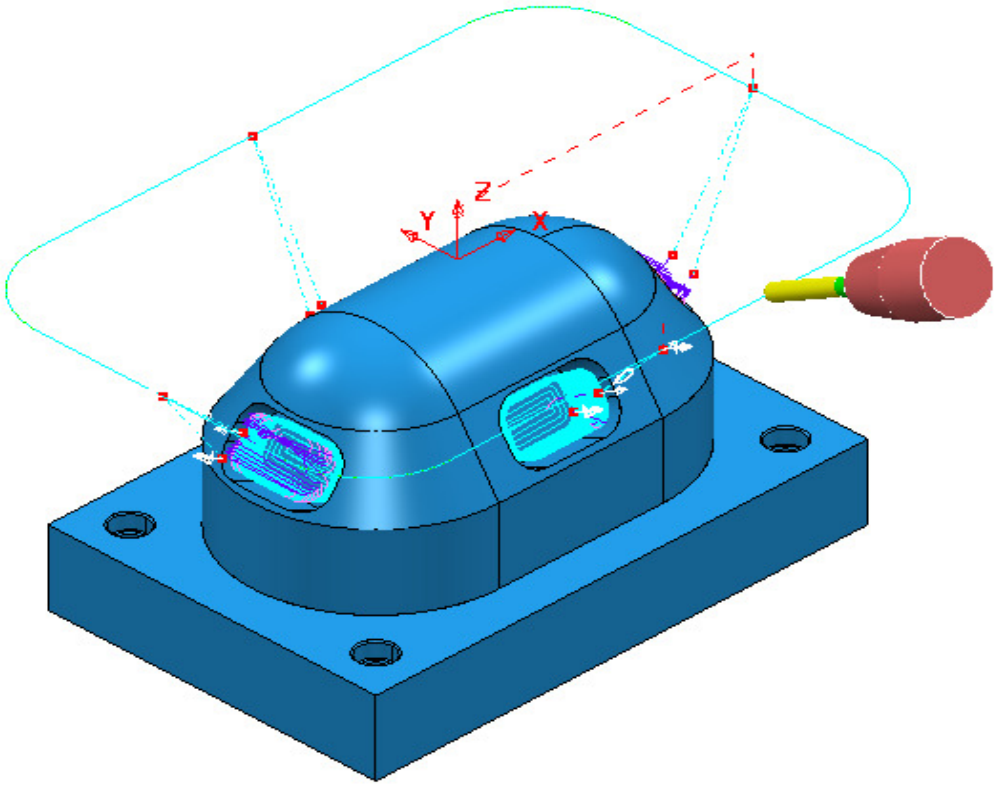
25 Select **File** – **Delete All** and **Tools** – **Reset Forms**.

NCProgram Verification (Including a Machine Tool Model)

It is possible to **Verify** the whole contents of an **NC Program** including an option to collision check a ***Machine Tool Model***.

1 **Open** the read only **Project**:

....\PowerMill_Data\five_axis\Collision_Simulation\AnglePockets-FullVerify_Start



Select **File** - **Save Project As**:

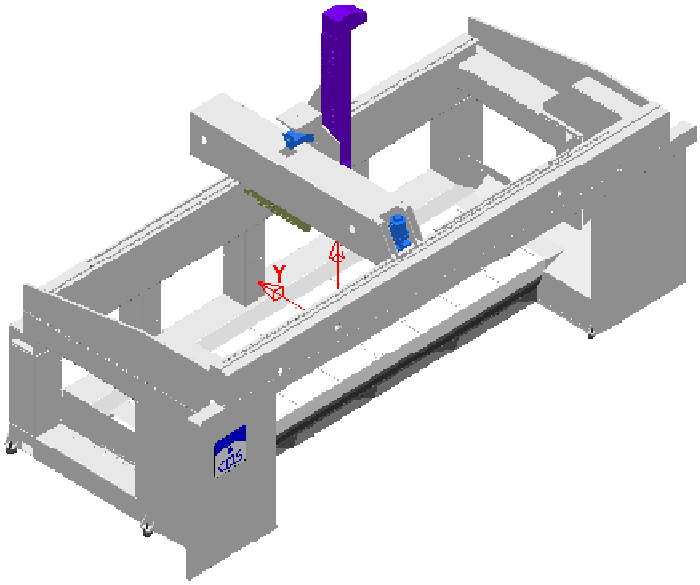
....\COURSEWORK\PowerMill-Projects\AnglePockets-FullVerify

2 In the **PowerMill *explorer*** right mouse click on **Machine Tools** and from the local menu select **Toolbar**.

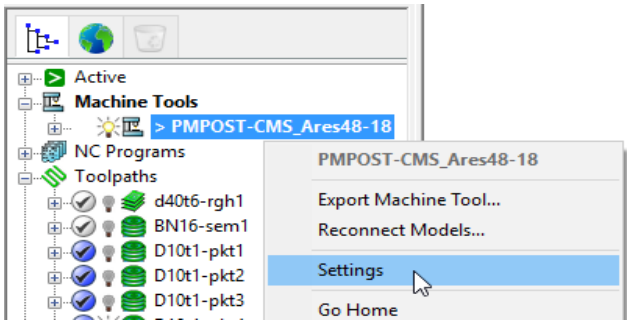
3 Select the ***Import the Machine Tool Model*** icon and select the file **PMPOST-CMS_Ares48-18.mtd** from:

….\PowerMill_Data\MachineData\CMS\.....

Copyright © Autodesk

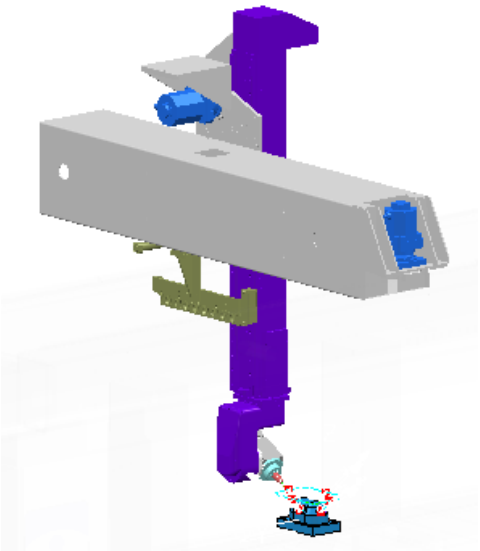


4 In the **PowerMill *explorer*** right mouse click on the **Machine Tool *PMPOST-CMS_Ares48-18*** and fro the local menu select **Settings**.



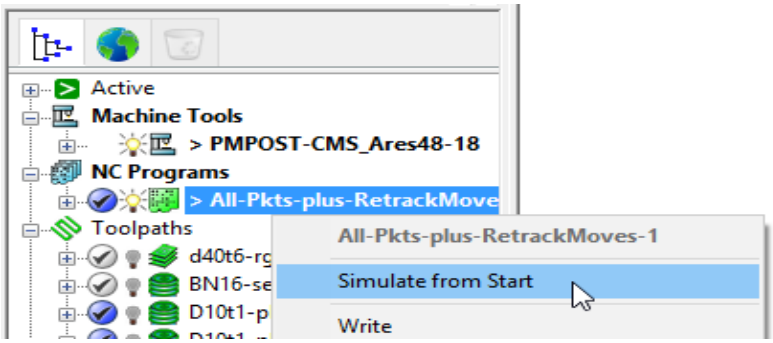
5 In the dialog, select **Part** and move the **Opacity** slider fully to the left to make the static machine tool parts almost invisible.

6 **Apply** and **Accept** the dialog.

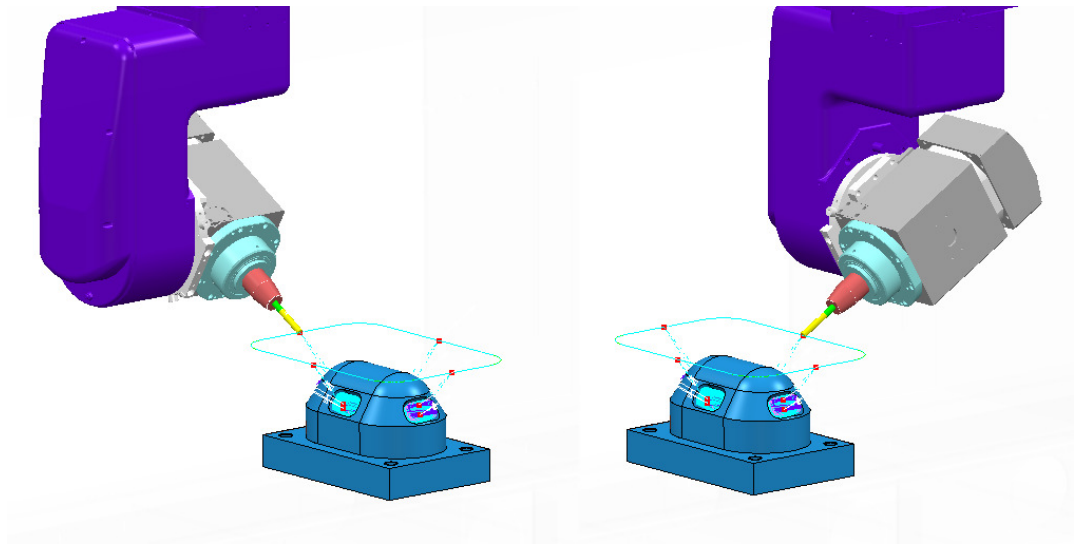


7 In the **Machine Tool** toolbar select the **Workplane *MC-Table-location***.

8 In the **PowerMill *explorer***, right mouse click the **NC Program** named **All-Pkts-plus-RetrackMoves-1** and from the local menu select **Simulate from Start**.



9 In the **simulation** toolbar set the speed slider to medium before selecting **Play**.

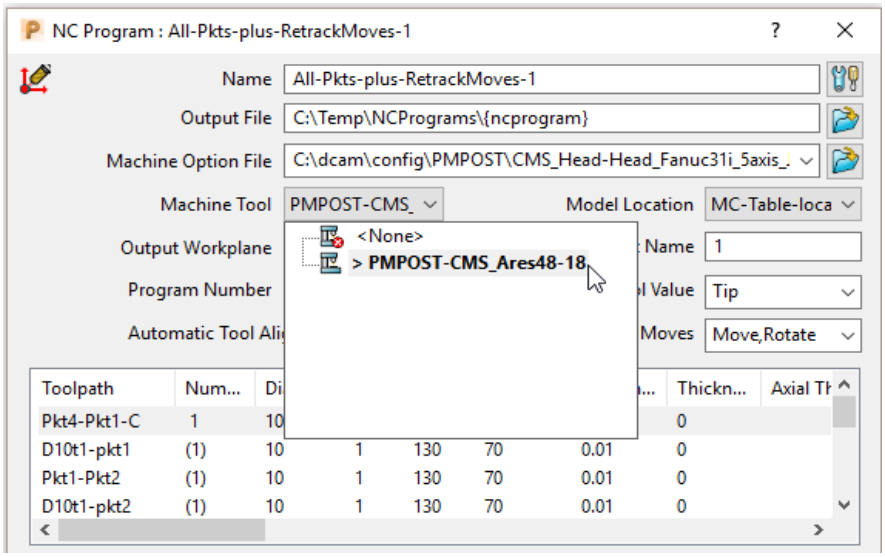


Observe that the total CMS machine's Head rotation (C Axis) is currently 450 degrees (This CMS has a maximum C axis rotation limit of 360 degrees).

10 Right mouse click on the **NC Program *All-Pkts-plus-RetrackMoves-1*** and from the local menu select **Settings** to open the **NC Program** dialog.

11 In the ***Machine Tool*** box select the **PMPOST-CMS_Ares48-18** and for ***Model Location*** select **MC-Table-location**.

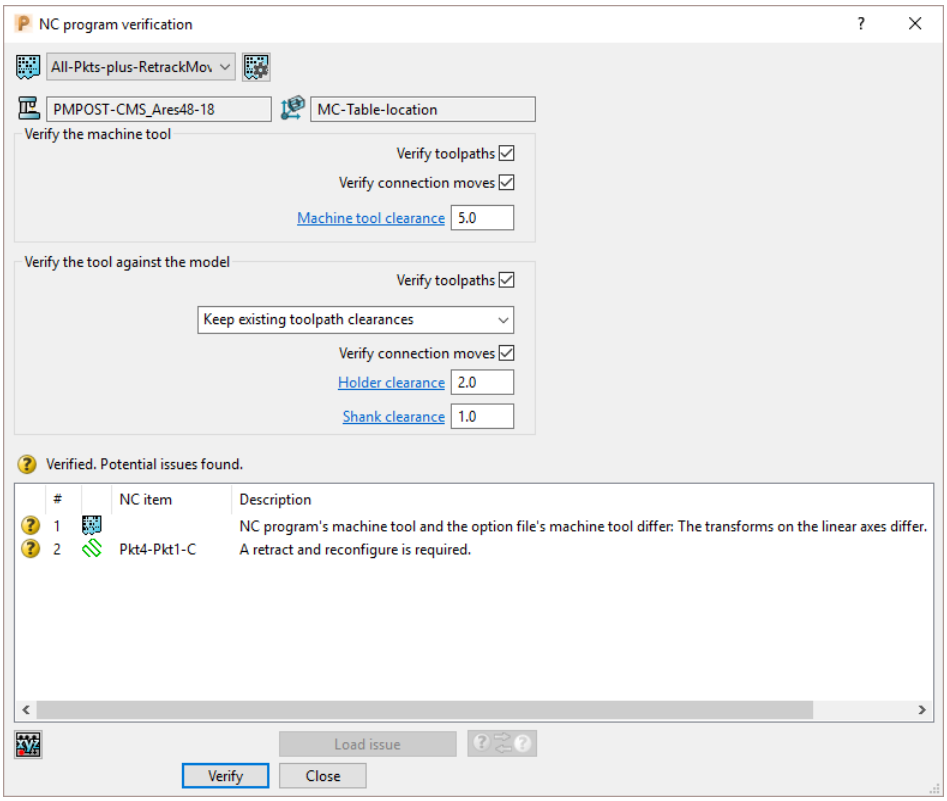
Copyright © Autodesk



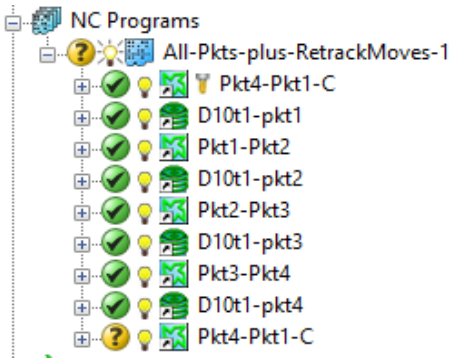
12 At the base of the dialog select **Apply** followed by **Accept**.

13 In the **PowerMill *explorer*** right mouse click on the **NC Program** ***All-Pkts-plus-RetrackMoves-1*** and from the local menu select **Verify**.

14 In the **NC program** verification dialog input the **Machine Tool**, **Holder**, **Shank** clearances as **5**, **2** and **1** respectively before selecting the **Verify** tab.



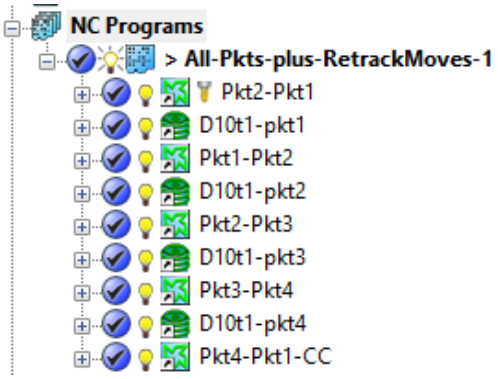
In the above **NC Program** dialog it has been reported that one of the toolpaths is causing the rotary limits to be exceeded. This CMS machine tool can only perform a total of one 360 Degrees rotation about the C axis. The total of C Axis rotation currently required by the **NC Program** is 450 Deg.



This can be fixed by using different positional move toolpaths at the ends of the **NC Program** that will start well within the main machining areas and then re-track the tool back to the original start position.

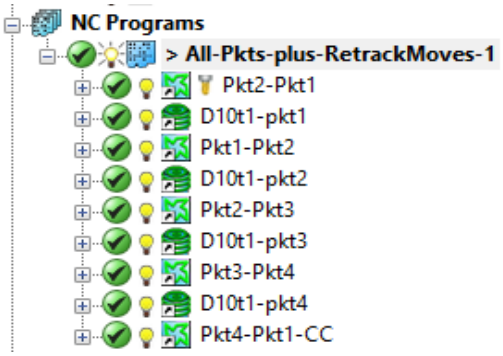
15 In the **PowerMill *explorer*** right mouse click on the **NC Program** ***All-Pkts-plus-RetracMoves-1*** and from the local menu select **Clear Verification**.

16 In the **NC Program *All-Pkts-plus-RetracMoves-1*** replace the common first and last positioning toolpath (**Pkt4-Pkt1-C**) with **Pkt2-Pkt1** and **Pkt4-Pkt1-CC** respectively.



The replacement toolpaths preform repositioning with counter clockwise rotation about the C axis. This will keep the total C rotation moves well within the CMS machine limits.

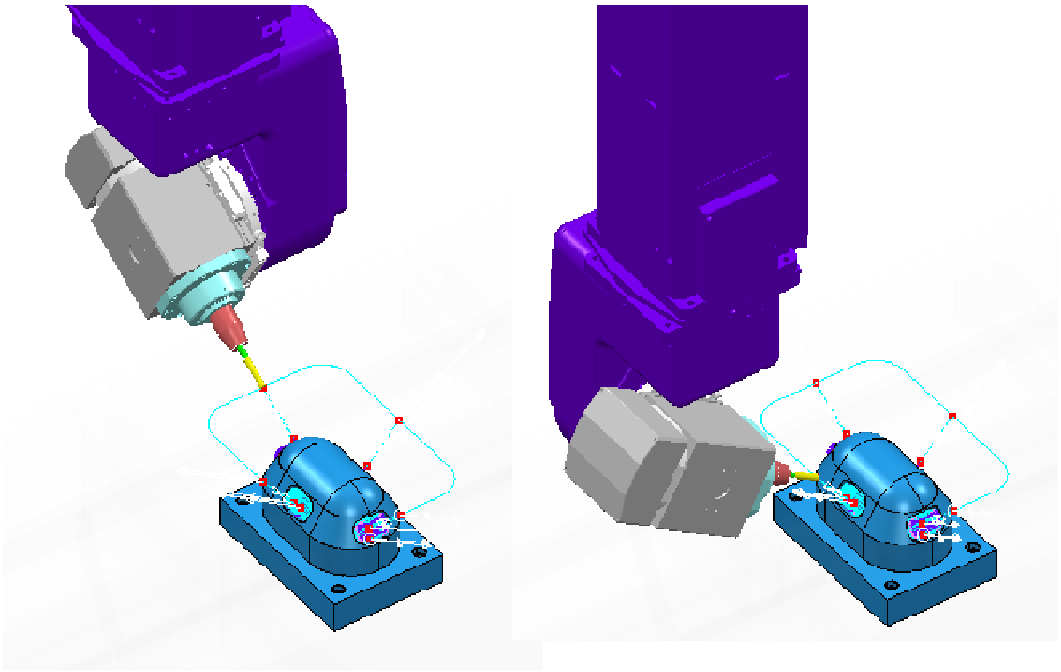
17 In the **PowerMill *explorer*** right mouse click on the updated **NC Program *All-Pkts-plus-RetracMoves-1*** and from the local menu select **Verify**.



Copyright © Autodesk

This time all toolpaths are displayed with a green verification status and no issues are reported in the ***NC program verification*** dialog. The green status also confirms that within the **NC Program** the ***Machine Tool*** is collision safe in addition to the included, gouge free **toolpaths**.

18 Run another **NC Program** simulation and observe that this time the whole process is now contained within a total C axis rotation of 270 degrees (max permissible 360 degrees).



19 Select **File** > **Save Project**.

20 Select **File** – **Delete All** and **Tools** – **Reset Forms**.

Copyright © Autodesk

12. Machine Axis Control

Introduction.

The **PowerMill** ***verification*** options will ensure collision safe, toolpaths relative to the ***Tooling components***. However, In 5 Axis applications it is also essential to check that the actual moving parts of the ***Machine Tool*** are also collision safe. For this to work effectively, a suitable machine tool model (**MTD** file) is required in the **simulation** process. If a collision between the component part and machine tool model is detected then the user can **recycle** the strategy with appropriate **Machine Axis Control** options applied.

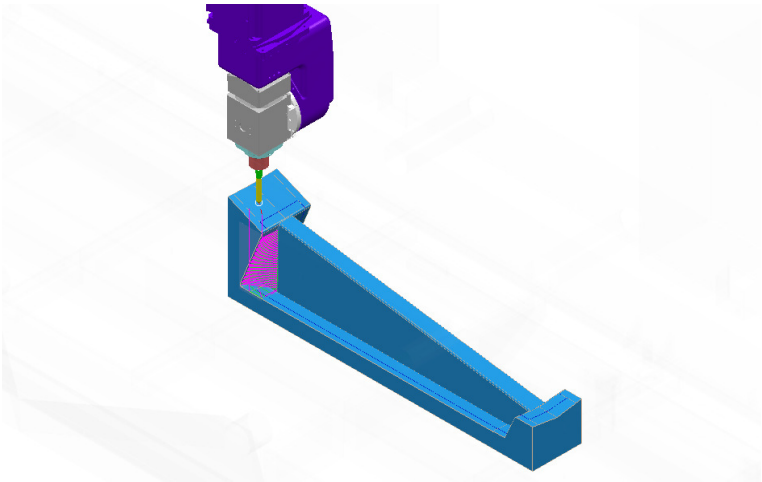
Machine Axis Control options allow the user to avoid collisions by controlling the ***orientation/movement*** of the independent parts of a machine tool within the machining strategy.

Machine Axis Control is not just used for collision avoidance but can also be used to enable an oversized component to be machined by applying **Polar** to force the use of a ***rotary axis*** in preference to a ***linear axis***. This option can also be used in cases where it is simply more efficient for a specific machine tool to use a rotary axis in place of a linear axis.

Tool Axis Control - Re-Orientation - Example

1 **Open** 'read only' the **Project**:

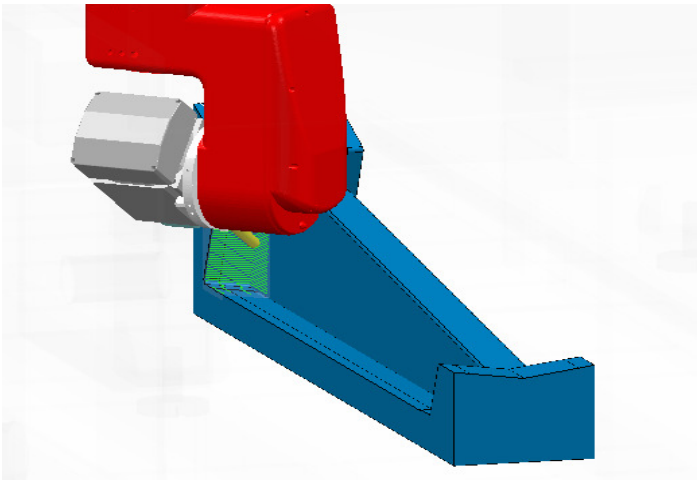
….\PowerMill_Data \five_axis\MachineAxisCtrl\CMS-AxisControl_Start



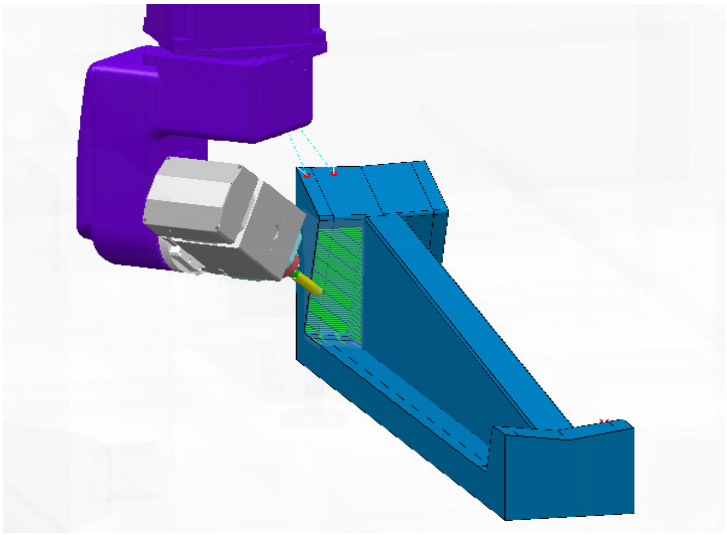
2 From the **main** pull down menus select **File** > **Save Project as**:

….\COURSEWORK\PowerMill-Projects\ CMS-AxisControl-ex1

The **Project** contains 2 **NC Programs** that contain the same machining toolpaths. If the **NCProgram** ***No-OrientationMoves*** is run in a ***machine tool simulation*** the Machine Tool Head clashes with the component.



If the **NCProgram** ***With-External-OrientationMoves*** is run in a ***machine tool simulation*** there are no collisions between the machine tool parts and component. This is due to the addition of repositioning toolpaths inserted as necessary to provide machine tool head **re-orientation**.



Although the last **NC program** provides an acceptable collision free solution the method is labour intensive and dependant on the user correctly creating and inserting positional move toolpaths in the correct order.

3 Select an **ISO 2** view.

4 In the **PowerMill** ***explorer*** right mouse click on **Machine Tools** and from the local menu select Toolbar.

5 Input **Base-Centre** in the ***workplane*** input box of the **Machine Tool** toolbar.

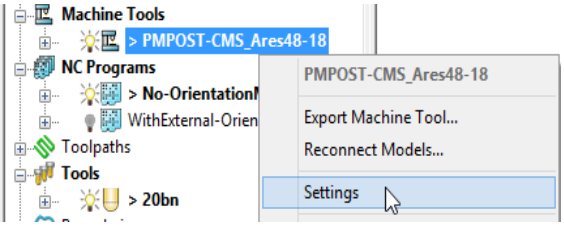
6 In the **Machine Tool** toolbar input the machine tool model:-

....\PowerMill_Data\MachineData\CMS\PMPOST-CMS_Ares48-18.mtd

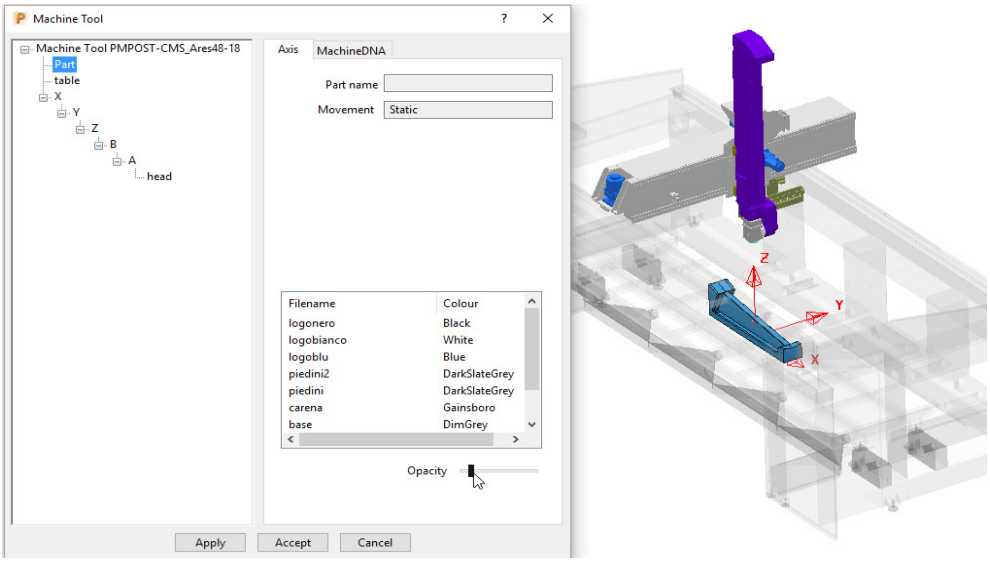
Copyright © Autodesk

7 In the **PowerMill *explorer***, right mouse click **Machine Tools** > **PMPOST-CMS_Ares48-18,** to open the local menu.

8 In the local menu select **Settings**.



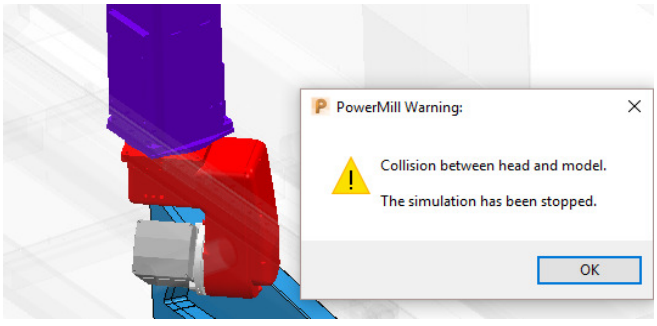
9 In the dialog select the item, named **Part** and then move the **Opacity** slider to make the static part of the machine tool model, translucent.



10 Accept the **Machine Tool** dialog.

11 In the **PowerMill *explorer***, right mouse click on the **NC Program** named **No-Orientation Moves** and from the local menu select **Simulate from Start**.

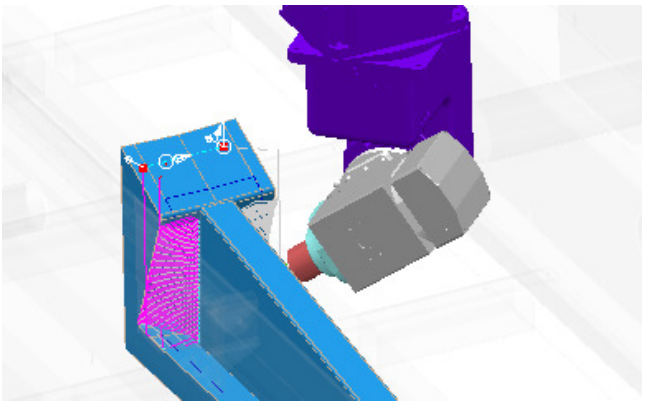
12 Select the **Play** button in the simulation toolbar.



After several passes the machine tool head is in collision with the component. The machine tool head changes colour to ***red*** and a **PowerMill Warning** box appears as soon as the first collision is detected.

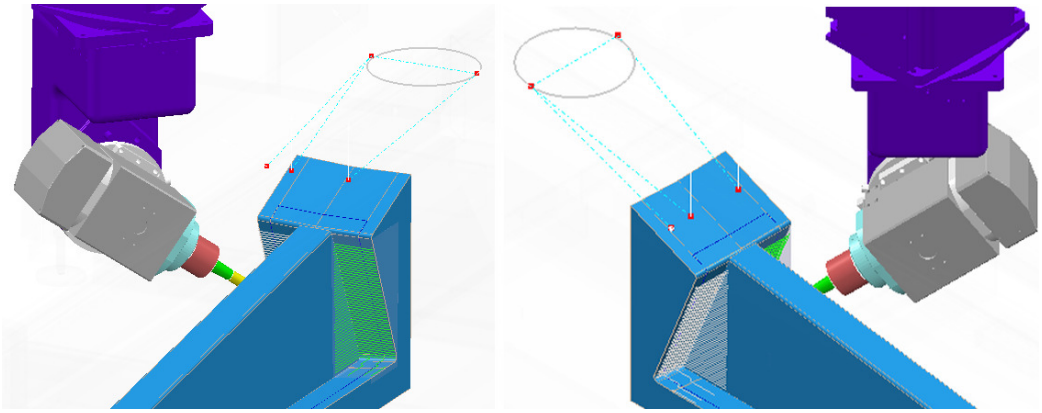
13 Select **OK** and the simulation will continue even though the head is still colliding (The head will change colour to red during each collision).

Collision checking and reporting to a Machine Tool Model simulation is a cost option.

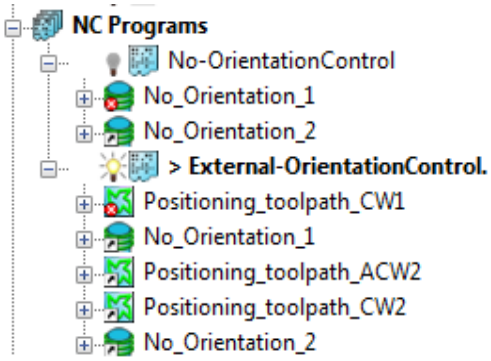


When the ***NC Program*** **simulation** moves to the other side of rib, the machine tool head happens to be in a collision safe, orientation relative to the component. This is not easily predictable and could easily change to a collision situation on switching the order or inserting new toolpaths into the **NC Program**.

14 In the **PowerMill *explorer***, right mouse click on the **NC Program WithExternal-OrientationMoves** and from the local menu select **Simulate from Star**t.



This time the machine tool head is collision safe throughout the simulation.



In this case, drive curve toolpaths have been inserted as required into the **NC Program** to force appropriate re-orientation moves to occur.

The only problem is that these moves are added retrospectively and as mentioned earlier, will be unpredictable if the toolpath order is changed or additional toolpaths are inserted into the **NC Program**.

A more efficient and direct method to apply machine tool head alignment within an individual strategy is to use the **Machine axis control** dialog.

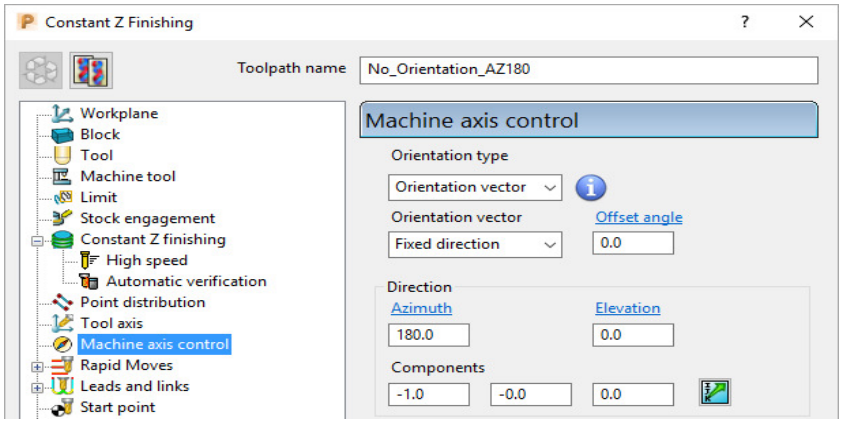
Copyright © Autodesk

15 Ensure that neither **NC Program** is active.

16 Activate the toolpath **No_Orientation_1** and then select **Settings** from the local menu.

17 In the **Constant Z Finishing** dialog, select ***Make a Copy*** and rename the new toolpath as **Orientation_AZ180**.

18 Select the **Machine axis control** page in the dialog and select the **Orientation type** as **Orientation vector**, the **Orientation vector** as **Fixed direction**, and input a **Direction** - **Azimuth 180.**



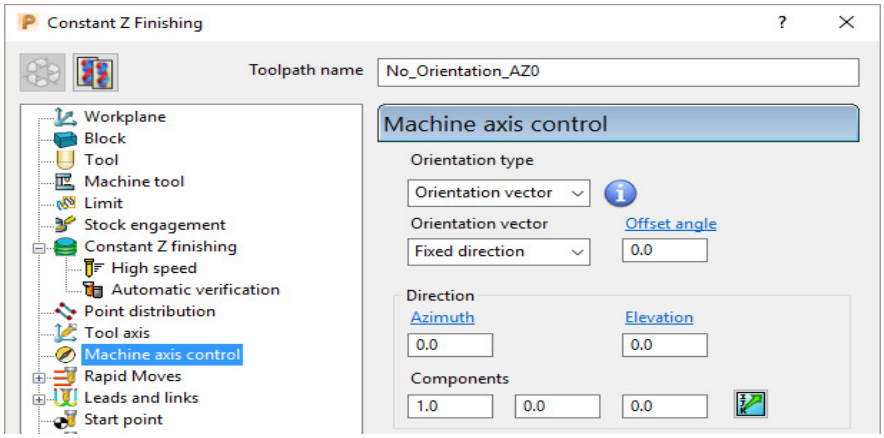
By changing the default **Orientation type** from **Free** to **Orientation vector** the machine head alignment will now be controlled directly within the strategy. The **Machine axis control** will cause the head to be rotated by **180** Degrees about the **Z** axis compared with the original toolpath.

19 Select **Calculate** to create the new toolpath and close the dialog.

20 Activate the toolpath **No_Orientation_2** and then select **Settings** from the local menu.

21 In the **Constant Z Finishing** strategy dialog, select ***Make a Copy*** and rename the new toolpath as **Orientation_AZ0**.

22 Select the **Machine axis control** page in the dialog and enter data exactly as shown below but this time with **Direction** - **Azimuth 0**.



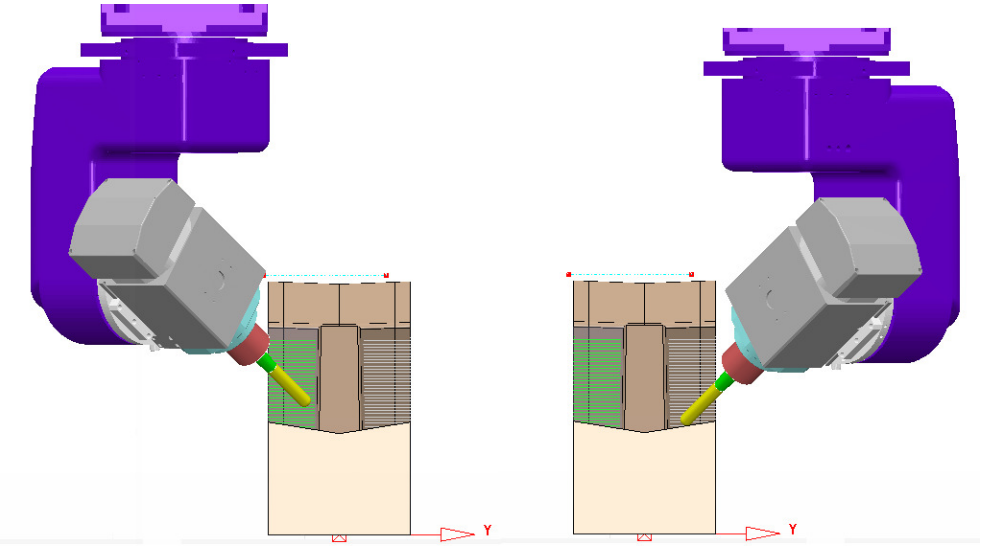
The **Machine axis control** will use the default values for an **Orientation Vector** of **Azimuth 0** and **Elevation 0**.

23 Select **Calculate** to create the new toolpath and close the dialog.

24 Create a new **NC Program** named **Internal-OrientationControl** containing the 2 new toolpaths **Orientation_A180** and **Orientation_A0**.

25 With the machine tool model, **PMPOST-CMS_Ares48-18** still active, run a full simulation on the new **NC Program** and check that the head movement is collision free throughout.

26 Reverse the order of the toolpaths in the **NC-Program** named **Internal-OrientationControl** and run the full **simulation** again and check that the head movement is collision free throughout.



The application of **Machine axis control** directly in the strategies has eliminated the need for the additional positioning toolpaths. As shown in the above illustration, the **machine axis control** is now independently controlled within the 2 new strategies.

Dynamic Machine Control (Requires VeriCL - The Advanced Simulation cost option)

The **Dynamic Machine Control** options are accessed from the **Machine Tool** toolbar.

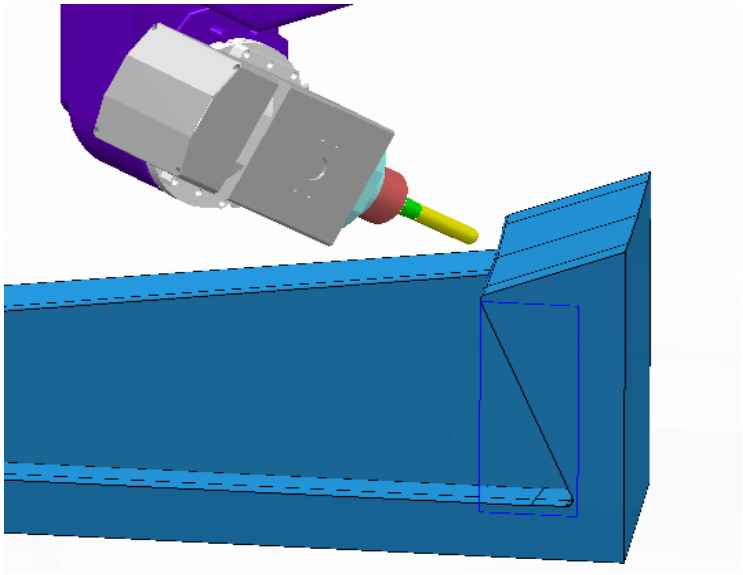
It is possible to apply machine axis control dynamically using options from the **Dynamic machine control** toolbar (shown below).

Copyright © Autodesk

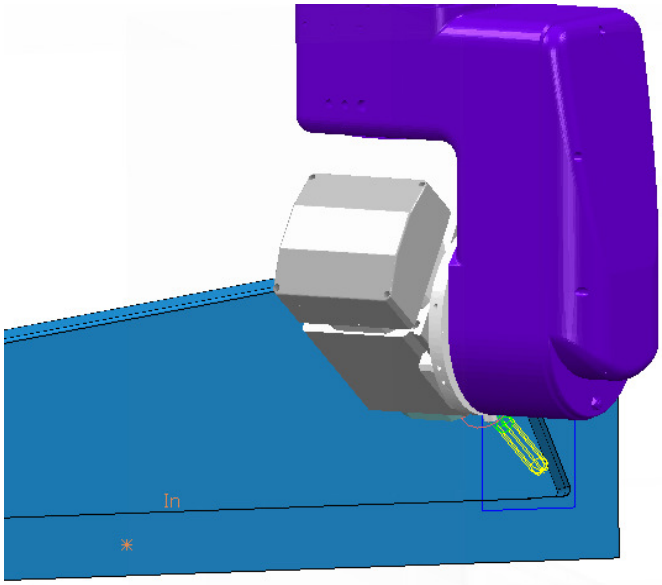
Visual checking for Machine Tool accessibility

The active **Tool** and **Machine Tool** head can be dynamically moved around the part to visually check for accessibility relative to the current orientation

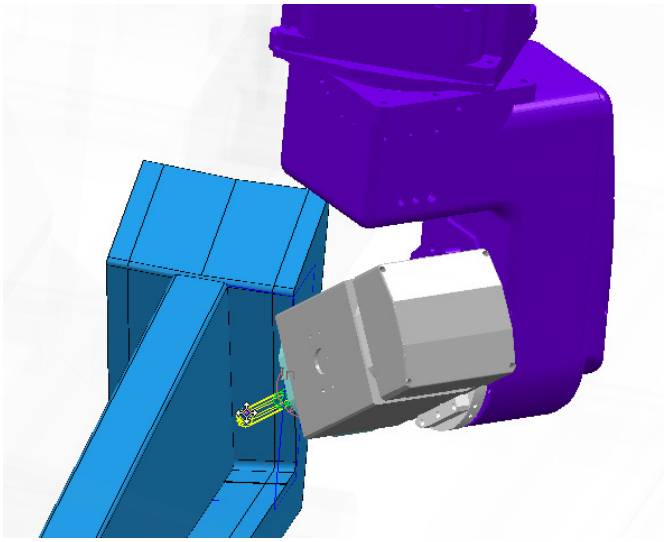
1. In the **PowerMill *explorer*** right mouse click on the toolpath **No_Orientation_2** and select **Simulate from start**.
2. **Jog** the tool to the start of the fixed machining alignment position and then from the **Machine Tool** toolbar select **Dynamic Machine Control**.



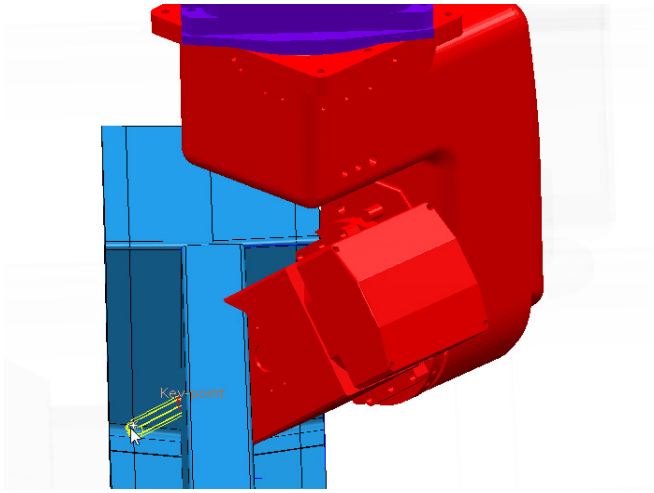
3. In the **Dynamic machine control** toolbar select use **tool positioning graphics**.
4. Left click on the tip of the tool and then dynamically move it down the angled model, undercut towards the base of the recess.



5 Rotate the view to observe the new position from different angles.



6 Dynamically drag the tool to a '***key point***' on the opposite side of the model.



If any part clashes with the model the **machine tool head** will change colour to red.

7 Select the green tick at the end of the **Dynamic machine control** toolbar to exit with the tool still displayed in the final position.

*The above final **Tool** position is cancelled if a new **Toolpath simulation** is selected, or if **Go Home** is selected in the **Machine Tool** menu.*

*If the **red cross** is selected then the tool will be positioned as it was before the **Dynamic machine control** toolbar was opened.*

8 Right mouse click on the machine tool model, **PMPOST-CMS_Ares48-18** and from the local menu select **Go Home**.

Copyright © Autodesk

Dynamic Machine Tool re-orientation applied to a Toolpath.

The default setting in the **Dynamic machine control** toolbar is ***Use machine tool graphics***.

The rotary axis components of the machine tool can be dynamically re-orientated and applied retrospectively to the active toolpath.

Other options include:-

a **Use machine tool graphics** - Applies dynamic rotation and linear moves relative to the Machine Tool model datum (Default).

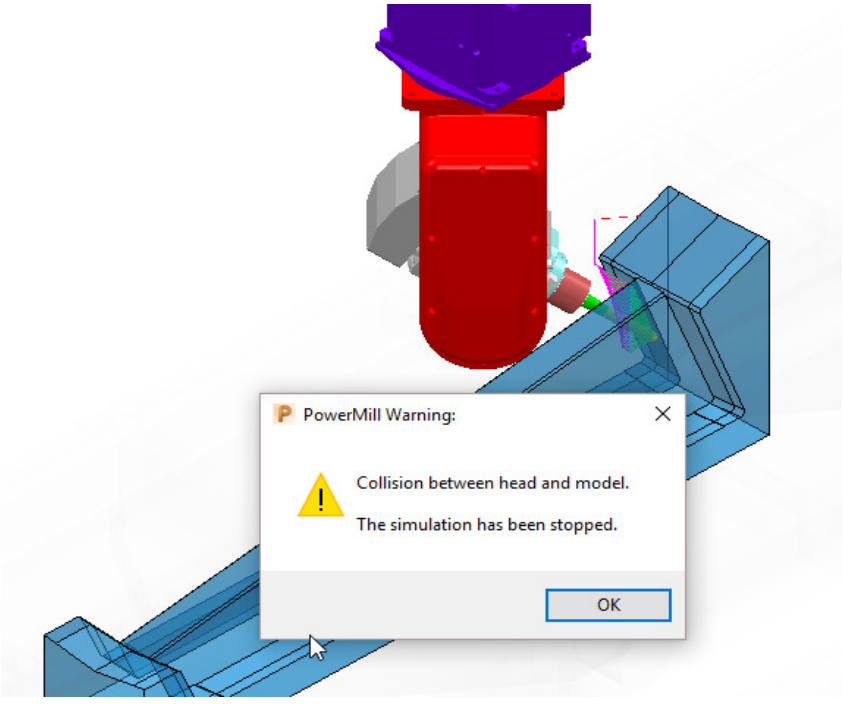
b **Active workplane graphics mode** - Applies dynamic rotation and linear moves relative to the active workplane.

c **Lock tool axis direction** - This locks the current rotary axis orientation, allowing only dynamic linear axis, moves to take place when used with **a** or **b** above.

1 From the local **PowerMill *explorer*** - **Toolpaths**, right mouse click on **No_Orientation_1** and from the local menu select **Edit** - **Copy Toolpath**.

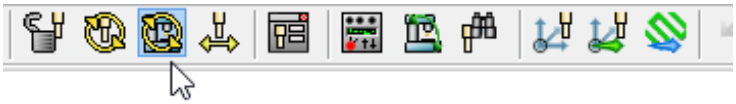
2 Rename the new toolpath as **Dynamic_Orientation_1**.

3 Activate the new toolpath and run a **Machine Tool simulation** until the collision with the ***Head*** is first detected.

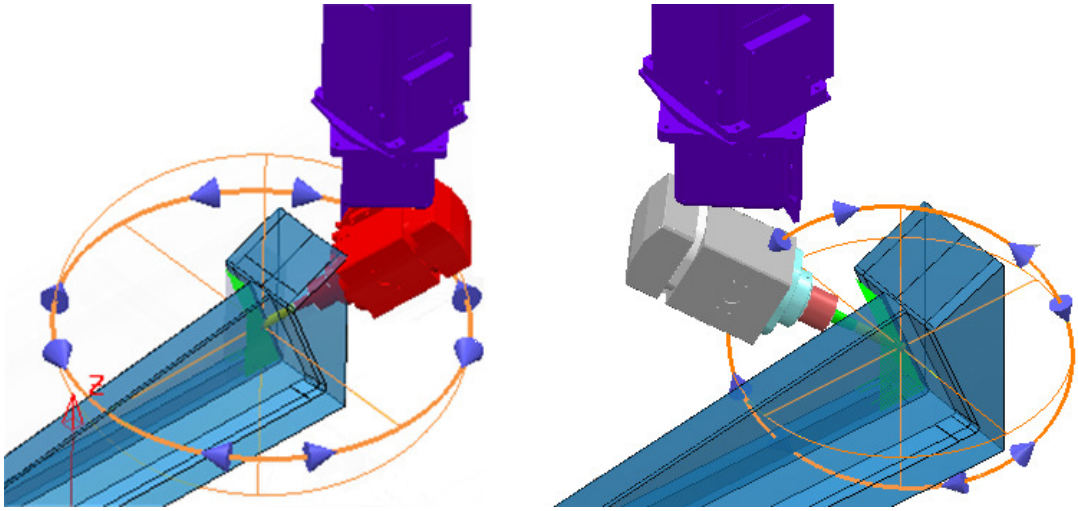


4 Select **OK** in the **PowerMill Warning** dialog.

5 On the **Machine Tool**, toolbar select **Dynamic machine control** .



6 **Use the default Use machine tool graphics option** .

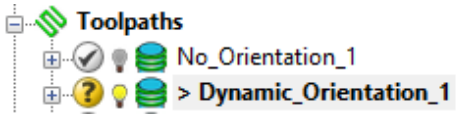


7 Use the upper circular handle to rotate the head to the above left, alignment, followed by the lower circular handle to re-orientate the tool to the above right alignment.

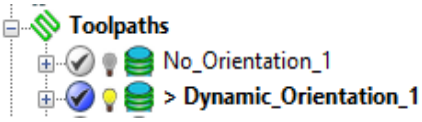
8 Once a suitable **Machine Tool Orientation** has been achieved select the **Update Toolpath** option.

9 Select the green tick to accept the changes and close the **Dynamic machine control** toolbar.

*The toolpath is now displaying **gouge** and **collision** status as **Unknown**.*



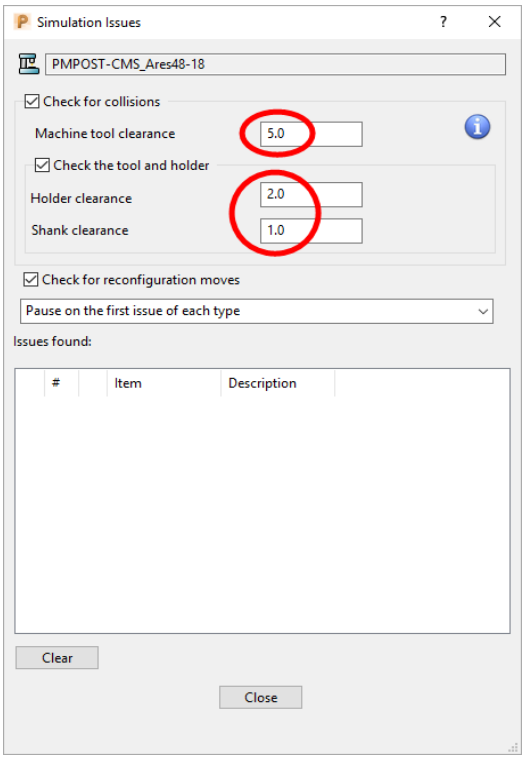
10 **Verify** the modified toolpath **Dynamic_Orientation_1** for both **Gouges** and **Collisions** (Clearances:- **Holder 2** and **Shank 1**).



The **Toolpath** is now confirmed to be both **Gouge** and **Collision** safe relative to the **Tool** assembly.

Copyright © Autodesk

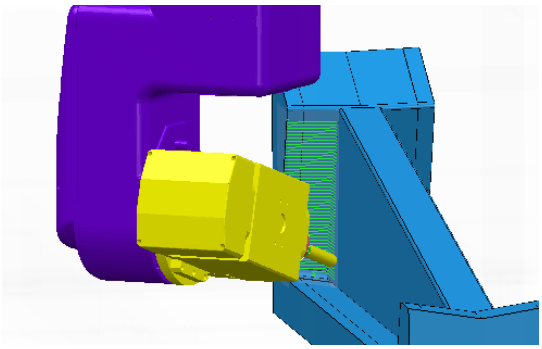
11 Right mouse click on the machine tool model and from the local menu select **Issues** to open the following dialog.

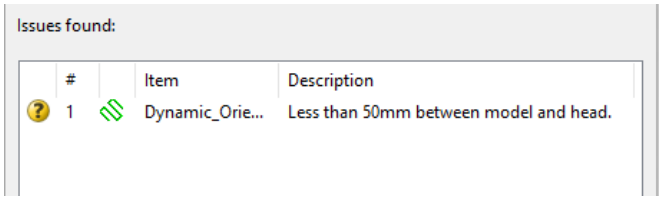


12 In the dialog input a suitable clearance value for the machine tool head (eg **5**).

13 **Clear** the issues found, list and perform further machine tool simulation to confirm that the ***head assembly*** is also collision safe (The **Holder** and **Shank** clearances (eg **2** and **1**) can also be included in the same dialog).

*During a simulation the machine tool **Head** will change colour yellow if a collision situation is identified and it will also be registered in the above dialog .*





14 **Save** the **Project**.

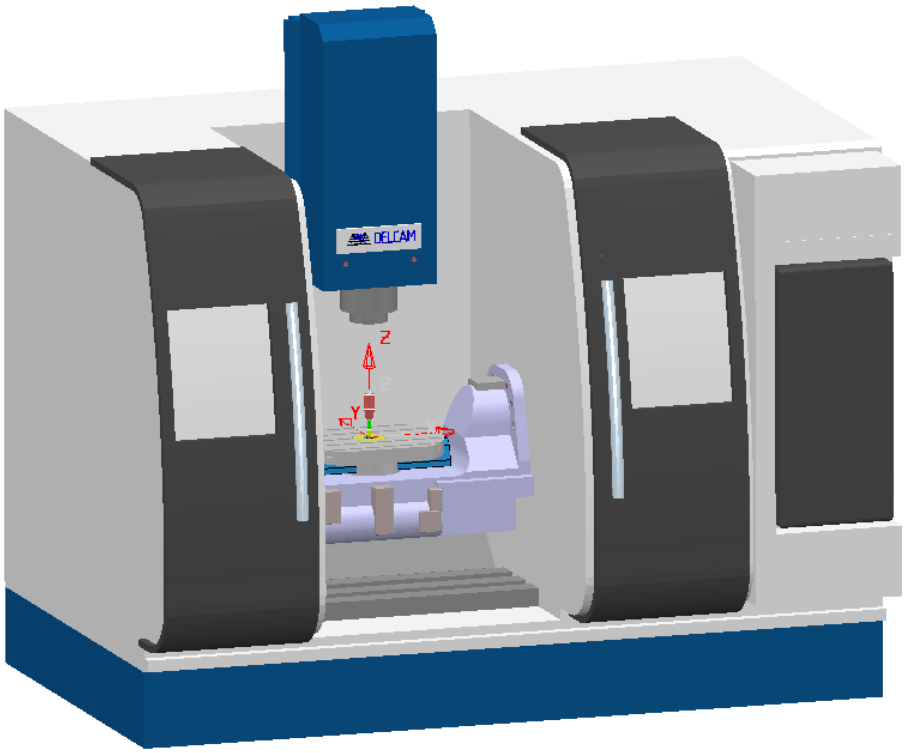
15 Select **File** - **Delete All** and **Tools** - **Reset Forms**.

Tool Axis Control - Polar - Example

The start up **Project** contains a **BN16** tool, 3 Axis **toolpath**, Mazak Variaxis **Machine Tool** model, and component **model**. The component model has a length dimension that exceeds the **X axis** and ***maximum* Y**, travel limits for the Mazak machine. However the model could be machined in 4 stages if the longest dimension, is controlled to align within the ***minimum* Y** axis travel limits. This will be achieved by applying the **Tool Axis Control** – **Polar** options to convert all **X** linear moves to **C Axis**, rotary moves (about the Z Axis).

1 **Open** 'read only' the **Project**:

...\PowerMill_Data \five_axis\MachineAxisCtrl\Mazak-OversizeCompt_Start



2 From the **main** pull down menus select **File** > **Save Project as**:

...\COURSEWORK\PowerMill-Projects\ Mazak-OversizeCompt-ex1

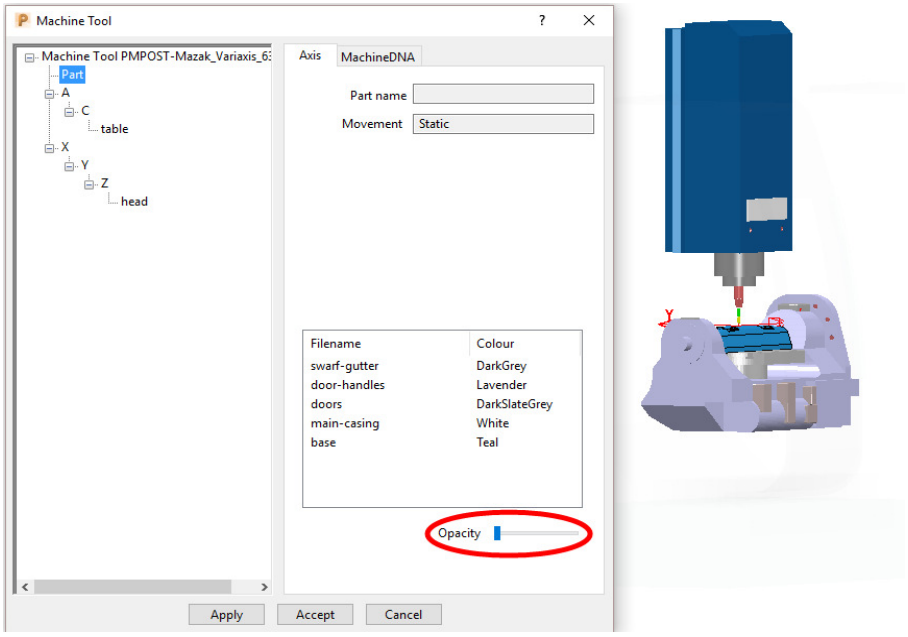
3 In the **PowerMill *explorer*** - Machine Tools area, activate the machine tool model, **PMPOST-Mazak_Variaxis_630PSA**.

4 From the **main** pull down menus select **View** – **Toolbar** and from the resultant menu ensure that the **Simulation**, and **Machine Tool** options are all ticked.

5 In the **Machine Tool** toolbar, input the machine tool named **PMPOST-Mazak_Variaxis_630PSA** and the **workplane** named **TableDatum**.

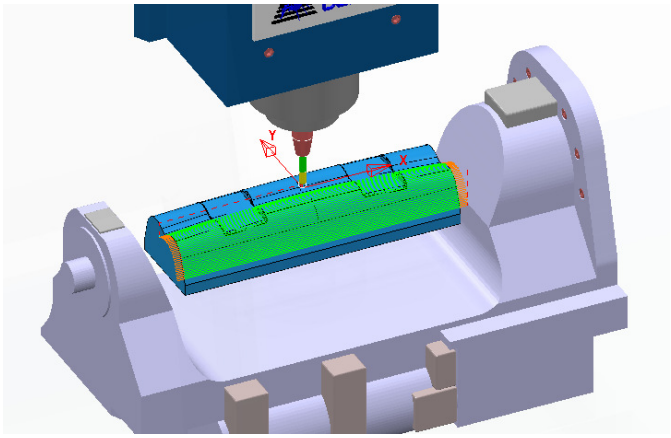
Copyright © Autodesk

6 In the **PowerMill *explorer*** right mouse click on the ***Machine Tool*** - **PMPOST-Mazak_Variaxis_630PSA** and from the local menu select **Settings**.

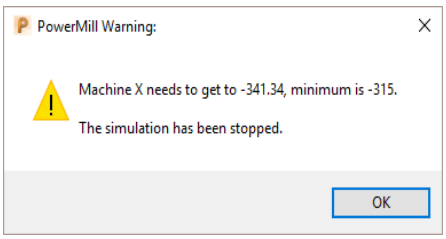


7 In the dialog, select **A** and move the ***Opacity*** slider to the left until the moving parts can be clearly seen behind the opaque, static parts of the **Machine Tool Model**.

8 From the **PowerMill *explorer***, right mouse click on the existing **toolpath** and from the local menu, select **Simulate from Start**.



9 In the **Simulation** toolbar, select the **Play** button and immediately the following ***PowerMill Error*** dialog will open.



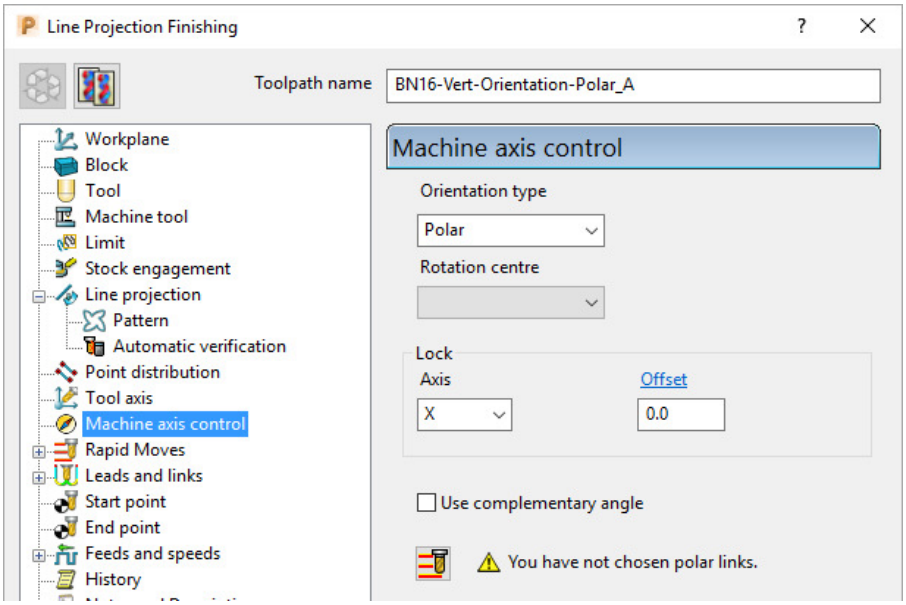
The **X** co-ordinates at the ends of the above **toolpath** are outside the ***X axis***, travel limits. Note, the travel limits are included in the **machine tool model** definition.

10 In the **PowerMill explorer**, right mouse click on the existing toolpath and from the local menu select **Activate** followed by **Settings** to open the dialog.

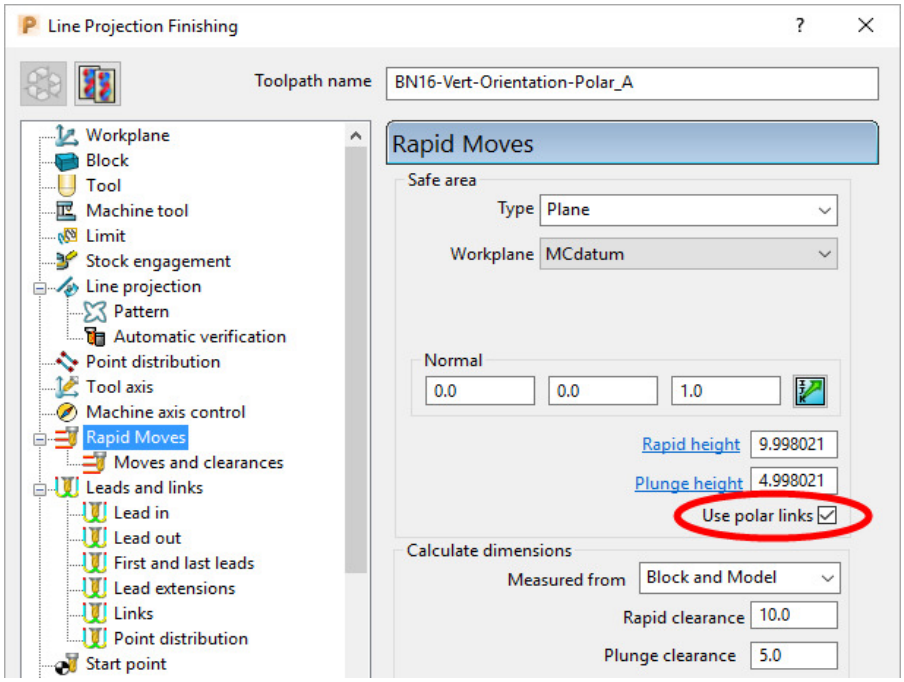
11 In the **Line Projection Finishing** dialog select the 'make a copy' option .

12 In the dialog, **Rename** the new toolpath as **BN16-Vert-Orientation-Polar_A**.

13 In the local explorer, select the **Machine axis control** page and input data as shown below.



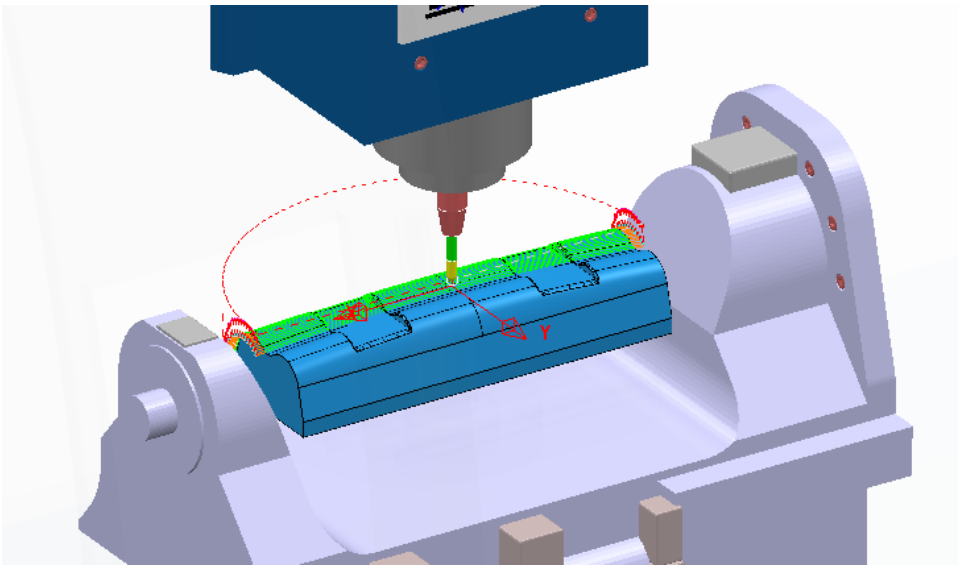
14 In the above dialog select to change to the **Rapid Moves** page and then tick the box, **Use polar links** as shown below.



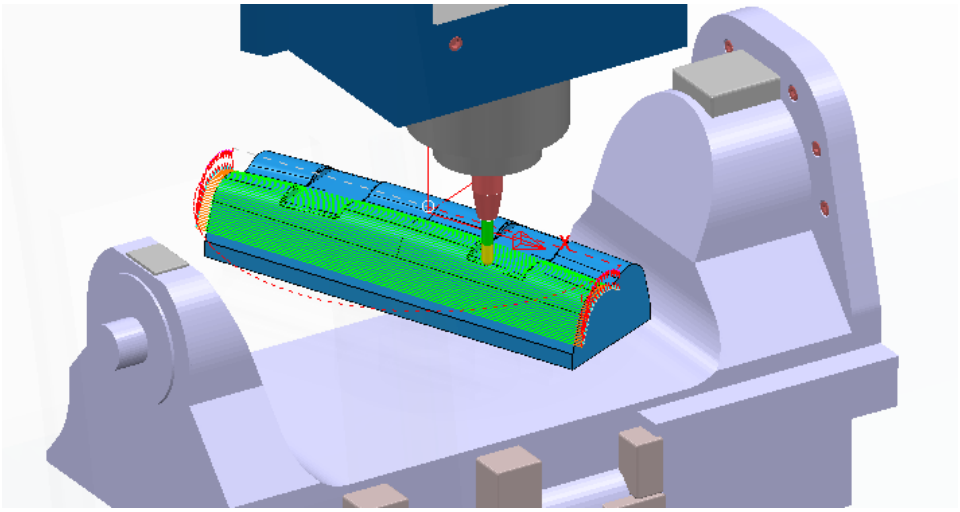
15 **Calculate** the dialog to create the new toolpath.

Copyright © Autodesk

16 From the **PowerMill *explorer***, right mouse click on the new toolpath and from the local menu, select **Simulate from Start**.



17 In the **Simulation** toolbar, select the **Play** button and this time the strategy will successfully run from start to end.



All the **X-Axis** cutting moves in the original toolpath have been converted to **C-Axis** rotary moves in the new toolpath. This has resulted in tool moves being confined to the area from the ***table centre*** towards the **min Y** travel limit.

18 Select **File** - **Save Project**.

Exercise

1 Create a copy of the above toolpath named **BN16-Vert-Orientation-Polar-B**.

2 **Calculate** after inputting the appropriate data to finish machine the other half of the component model keeping within the **Tool Axis limits** for the **Mazak** machine tool.

3 Select **File** - **Save Project**.

Copyright © Autodesk

Copyright © Autodesk

13. Machine Tool Models

Introduction.

The **PowerMill** toolpath **simulation** process can be expanded to include a **Machine Tool Model**. This capability becomes extremely important for multi-axis applications.

First of all **triangle** models (eg **dmt** or **stl** format) need to be obtained of all the independent moving parts of the machine tool. It is essential that the user obtains accurate dimensions along with any other details relating to the setup of a specific machine tool.

These separate **Triangle** files are stored in a folder adjacent to a ***Machine Tool Simulation*** schema file (**mtd**)

To create a new **mtd** file it is easier to modify a copy of an existing one based on a machine tool of similar configuration.

It is essential that the kinematics of a **Machine Tool Simulation model**, is the same as the relevant **Post-Processor** output.

How to configure a Machine Tool for simulation

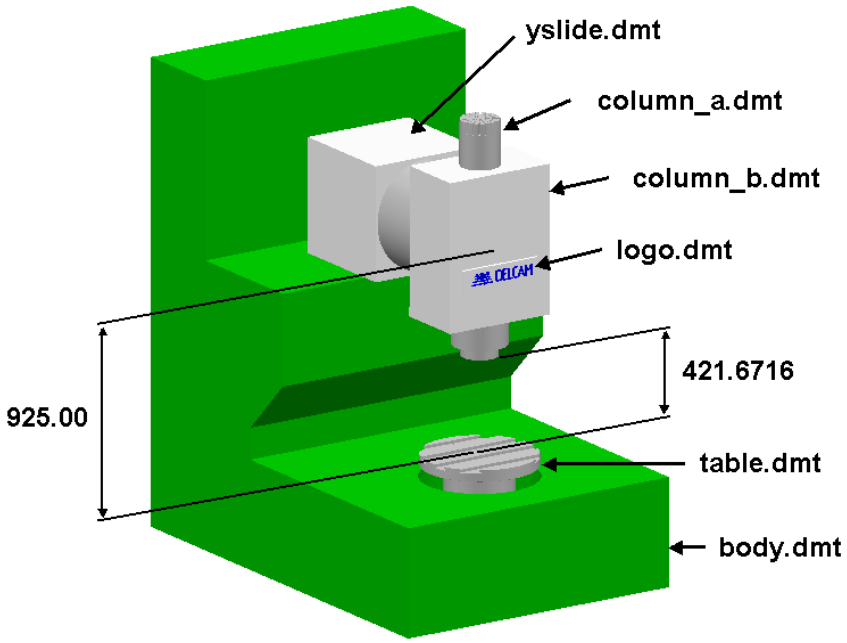
The first stage is to create the separate component models (ideally in **PowerSHAPE**) for each moveable axis for the machine tool. These must be exported as individual **triangle** models to be compatible for use in an **mtd** file. For the following example the models have already been created and are ready to be registered to a new **mtd** file.

An **mtd** file contains information that describes the kinematics of a particular machine.

For **mtd** files to work, they must be accompanied in the same folder by the ***schema*** file, **pmillmt.xsd**.

In the same location, a ***sub-folder*** must exist that contains separate **triangle** models **dmt** of all the necessary ***static*** and ***moving*** machine tool components.

Machine Tool Model for a dmu80t Five Axis



The above illustration identifies 6 separate **triangle** models that form a ***dmu80t*** five axis machine tool. Also shown are the vertical distances from the top of the ***table*** to the base of the ***head*** (**421.6716**), and the centre of the ***rotational axes*** (**925.00**).

machine_part

The **mtd** file is made up of objects called ***machine_part***. There are 2 compulsory ***machine_part*** objects which must be identified as "**head**" and "**table**". Note; their names in the **mtd** file must be in lower case.

In the above machine tool the **X**, **Y**, and **A** axis (rotation about Y) are controlled on the "**head**", and the **Z** and **B** axis (rotation about Z) are controlled on the "**table**".

Linear Axis Controls

X, **Y**, and **Z** must exist and only right handed axis systems are supported.

These can be in any orientation as long as they are orthogonal.

Type of axis that can be configured is:-

Simple_linear:- The definition requires ***direction***.

Rotary Axis Controls

Rotary controls are typically defined by **A**, **B**, or **C** and ideally should be selected to match the same output as output in the associated ***Post-Processor***. Note; where applicable, angular limits can be assigned to an individual rotary axis.

Type of axis that can be configured is:-

simple_rotary:- The definition requires ***position*** and ***direction.***

Copyright © Autodesk

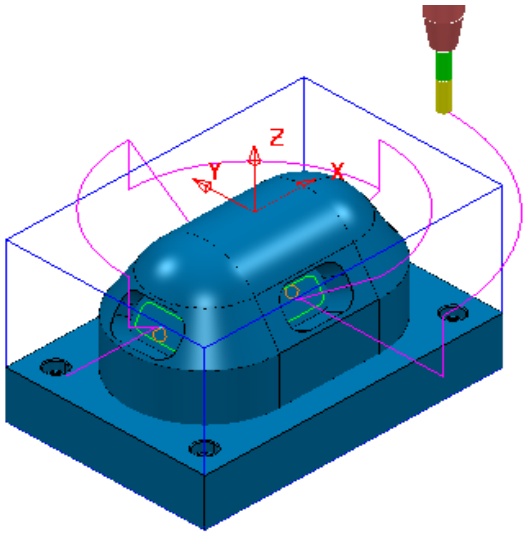
MTD file initial state

Although all the separate **triangle** models are correctly positioned relative to the assembled machine tool model, it is still required to input key dimensions into the mtd file. In this example, these include the 'gauge' distance from the top of the table (**0 0 0**) to the base of the head (**0 0 421.6716**), and the coordinates of the centre of the rotary axes (**0 0 925**).

EXAMPLE:- MTD file for a Mazak five axis machine

1 Select **File** - **Open Project**:-

....\PowerMill_Data\five_axis\MachineToolModels\MTD-Mazak-Ex1_Start



2 Select **File** – **Save Project As**:-

....\COURSEWORK\PowerMill_Projects\MTD-Example1

A **Machine Tool Model** will be loaded into the above **Project** that correctly defines:-

a The position of all the separate machine tool components.

b The direction of rotation for the 2 rotary and +/- movement for the 3 linear axes.

c The max/min travel limits for each of the 5 axes.

3 From the main pull down menus select **View** – **Toolbar** – **Machine Tool** to open the **Machine Tool** toolbar (Or alternatively right click on **Machine Tools** in the **PowerMill *explorer*** and from the local menu select **Toolbar**).

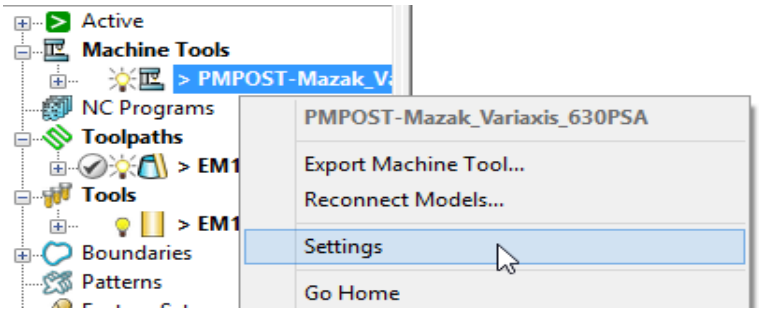
4 In the **Machine Tool** toolbar select the **Import Machine Tool Model** option.

5 In the dialog select the **Machine Tool Model** file:-

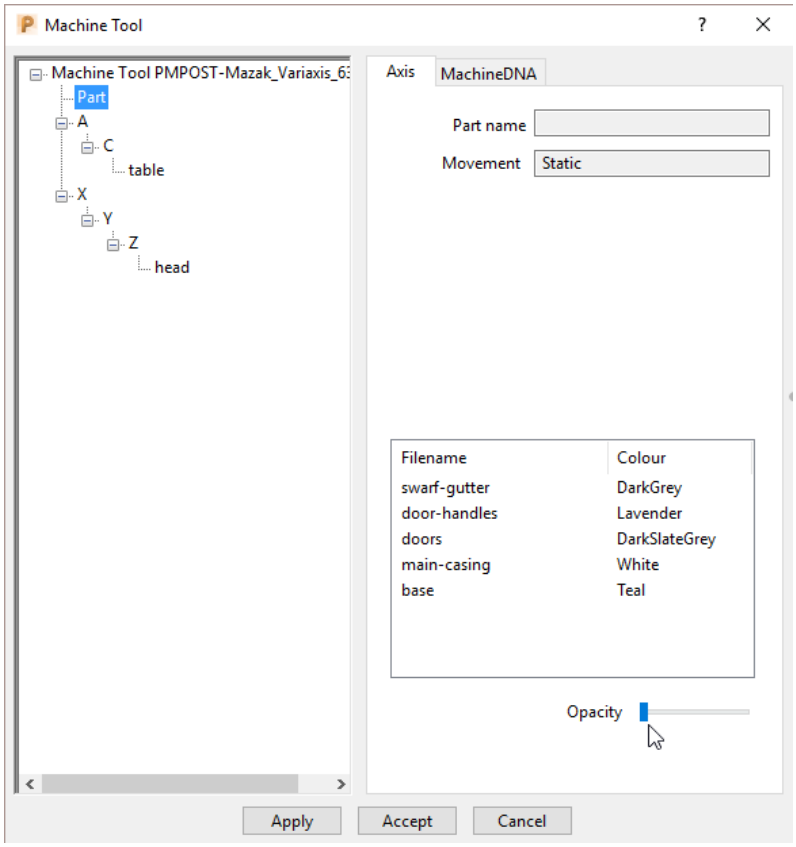
….\PowerMill_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA.mtd

6 Select **Workplane 2** as the **Current Model location** point.

7 In the **PowerMill *explorer*** right mouse click on the **Machine Tool Model** name **PMPOST-Mazak_Variaxis_630PSA** and from the local menu select Activate followed by **Settings**.

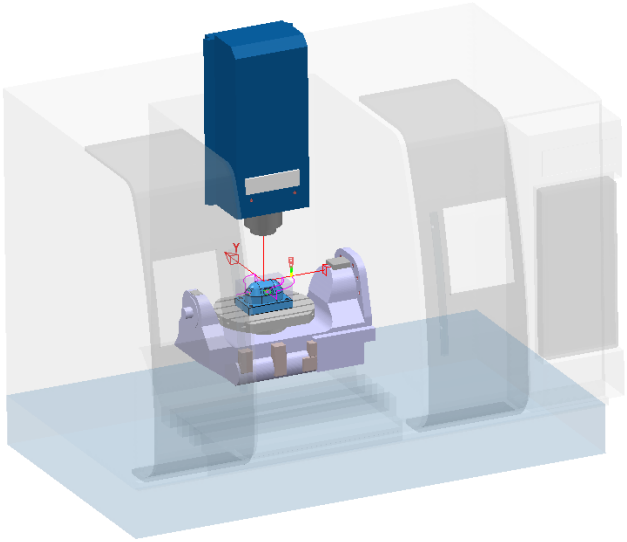


8 In the **Machine Tool** dialog select **Part**.



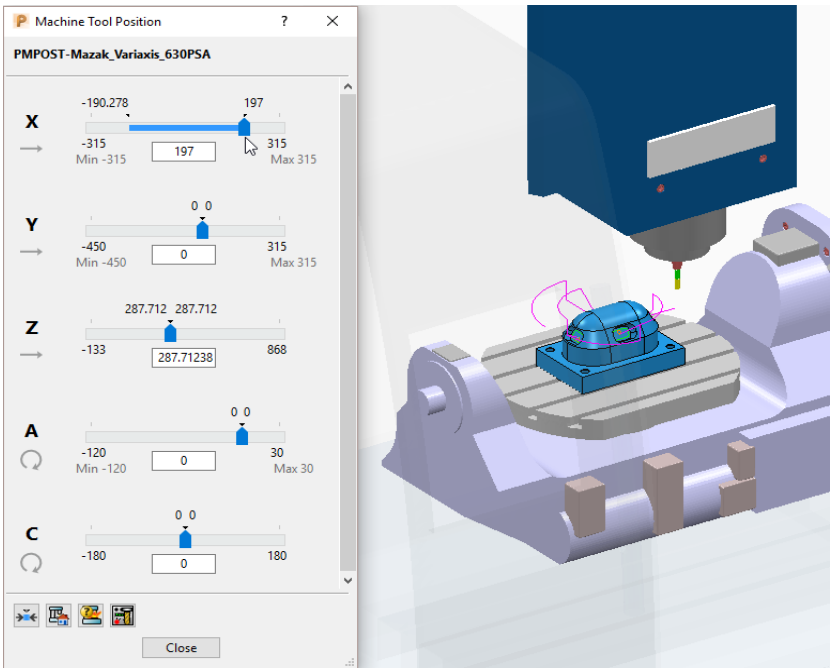
9 Move the **Opacity** slider to the left until the ***Machine Tool Body*** is almost invisible.

Copyright © Autodesk

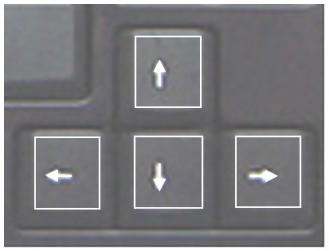


10 Right mouse click on the toolpath **EM12-FinPkts** and from the local menu select **Simulate from Start**.

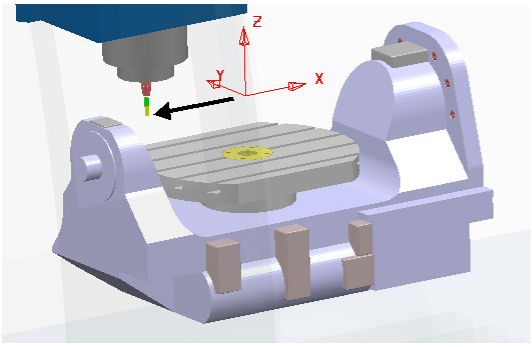
11 In the **Machine Tool** toolbar select the **Machine Tool Display** options.

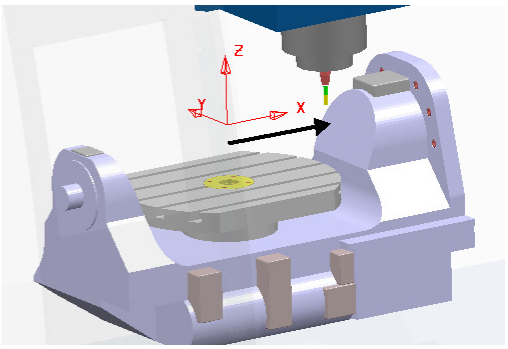


12 To test the **X-Axis** travel, select the **Cursor** on the **X** row slider and drag it to the left, and/or use the LH/RH keyboard ***arrow*** keys for exact **0.5** incremental moves.



13 On the keyboard hold while holding down the **Shift** key, press the ***left hand arrow*** key to perform continuous, incremental ***-ve* X** moves on the **machine tool model**.



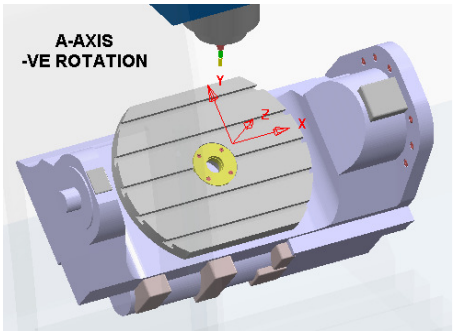


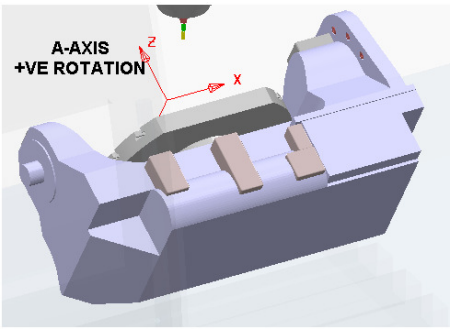
14 While holding down the **Shift** key, press the RH keyboard ***arrow*** key to perform a ***+ve* X** move on the **machine Tool Model**.

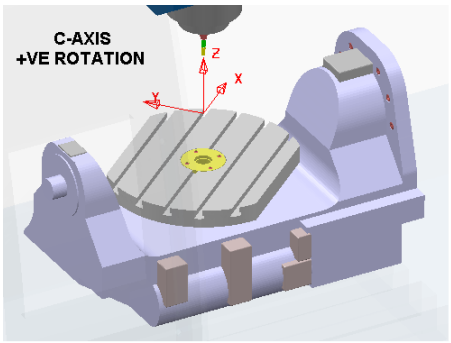
15 On the keyboard press the downward, ***arrow*** key to move down to the next row (or simply click on the **Y** row, cursor).

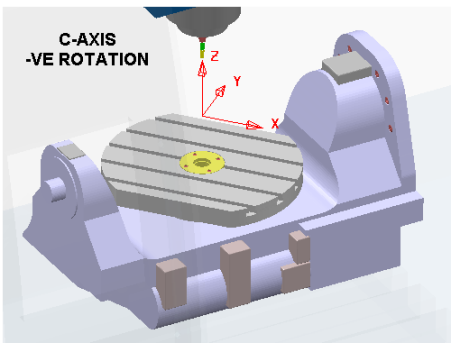
16 As before visually check both the ***-ve*** and ***+ve* Y** moves on the **Machine Tool Model**.

17 Systematically move down the rows and check out the movement of the remaining 3 axes (**Z**, **A**, and **C**) on the **Machine Tool Model**.







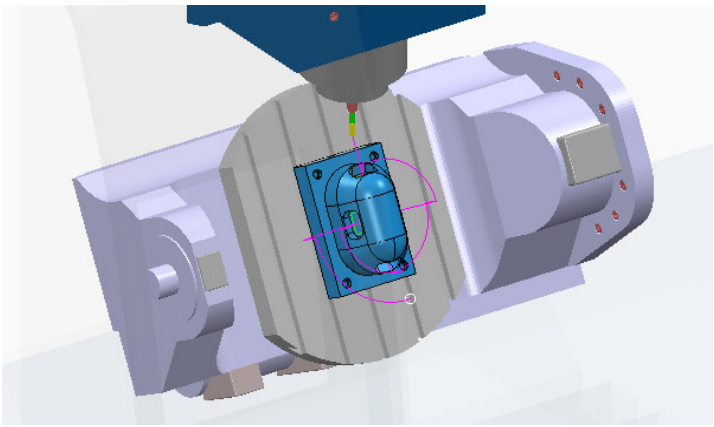


The above movements are exactly the same as on the actual **Mazak** machine tool to which the **MTD** file relates.

A full **simulation** will provide a visual confirmation of the all moves performed on the toolpath. It is also possible to check for ***Machine Tool*** component collisions (if an additional cost option is purchased).

Copyright © Autodesk

18 Close the ***Machine Tool Position*** dialog and then perform a full **Simulation** on the toolpath **EM12-FinPkts**.



19 In the **PowerMill *explorer*** right mouse click on the machine tool, ***PMPOST-Mazak_Variaxis_630PSA*** and from the local menu select **Go Home**.

Correcting a faulty MTD File:-

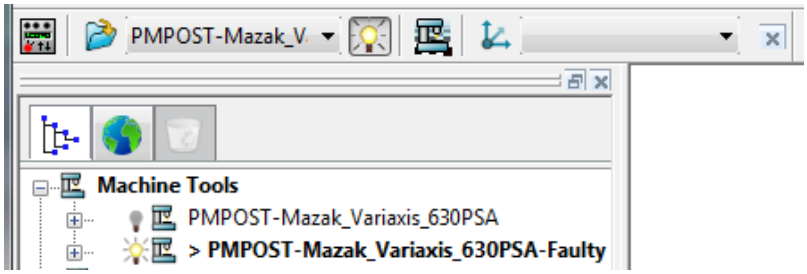
20 In the **Machine Tool** toolbar select the **Import Machine Tool Model** option.

21 In the dialog select the **Machine Tool Model** file:-

….\PowerMill_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA-Faulty.mtd

22 As before open the **Settings** dialog from the local menu and turn down the **Opacity** of the non-moving parts of the **Machine Tool Model**.

23 In the dialog select **Open** to import the new **Machine Tool Model**.

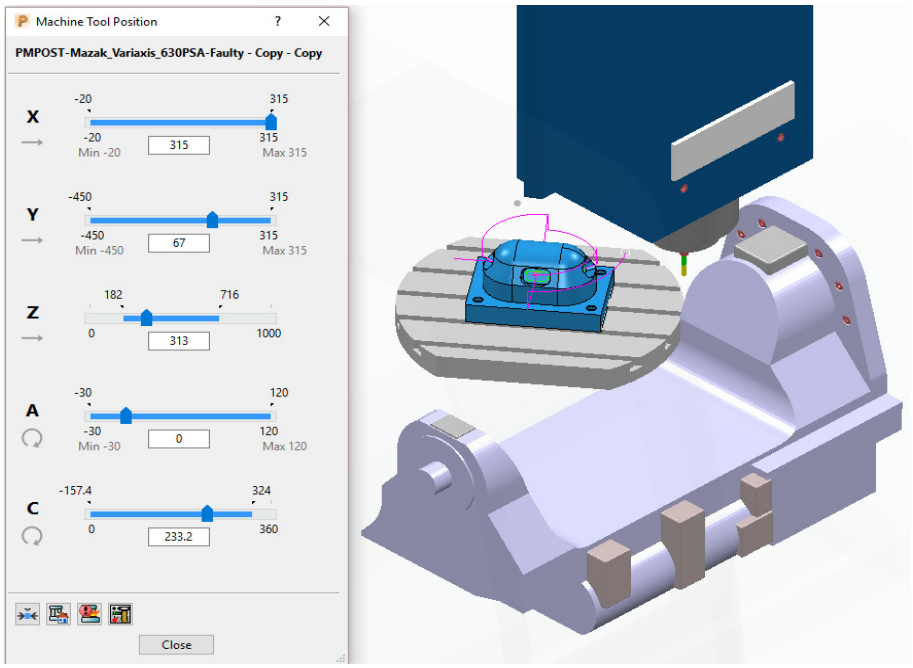


The newly imported **Machine Tool Model** file should perform exactly as the existing one. However the **MTD** file contains some incorrect values which need to identified and fixed. The user will need to systematically check the linear and rotary axis movements and the relative location of the trunnion and rotary table.

24 In the **Machine Tool** toolbar select the **Machine Tool Display** option.

25 Use the **Jog** options to check the **+ve** and **–ve** movement of all 5 axes (They should be the same as the **Machine Tool Model** imported with the original **Project**).

Copyright © Autodesk



The following C-Axis faults are exposed in the currently active MTD file:-

a The **+ve/-ve** rotational moves for the ***Rotary Table*** **C Axis** are reversed compared with the **Machine Tool Model** imported with the original **Project**.

b The **Rotary Table** rotational centre (**C-Axis**) is incorrectly aligned.

c The **–ve X Axis** moves are incorrectly limited to just beyond the table centre.

26 From the **Desktop** open a **File Explorer** dialog and browse to the folder:-

….\PowerMill_Data\five_axis\MachineToolModels

27 Create a copy of the mtd file:- **PMPOST-Mazak_Variaxis_630PSA-Faulty.mtd** and rename it as:- **PMPOST-Mazak_Variaxis_630PSA-Update.mtd**

If an MTD file of the same name already exists overwrite it.

28 Open the new copy of the **mtd** file in **Wordpad** and look for incorrect data in the section of the **mtd** file that defines the **Rotary Table** – **C-Axis**.

<machine_part>

<axis>

<control_info ADDRESS="C" />

<simple_rotary X="0" Y="250" Z="0" I="0" J="0" K="-1" />

</axis>

<model_list>

<dmt_file>

<path FILE="Mazak_Variaxis_630/table.dmt" />

Copyright © Autodesk

<rgb R="200" G="200" B="200" />

The previous row that defines the C-Axis requires the XYZ position to be in the centre of the table, and the +ve/-ve rotation directions to be reversed.

29 Modify the above incorrect values to those shown below:-

<simple_rotary X="0" Y="0" Z="0" I="0" J="0" K="1" />

30 **Save** the updated **mtd** file (but do not close it).

*This latest update will now be tested in the **PowerMill Project**.*

31 In the **Machine Tool** toolbar select the **Import Machine Tool Model** option.

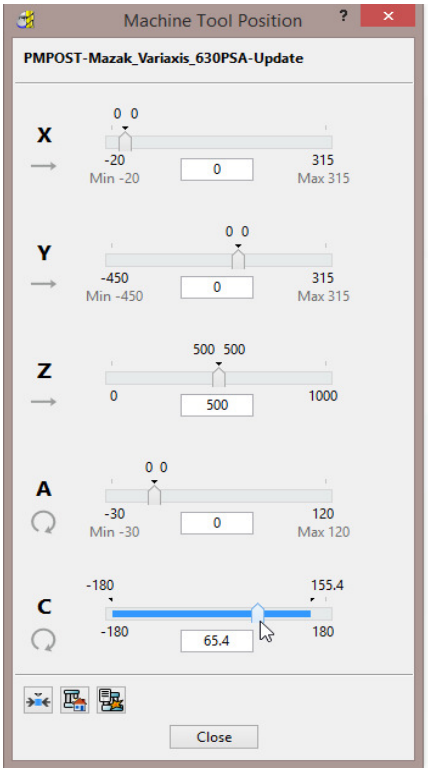
32 In the dialog select the **Machine Tool Model** file:-

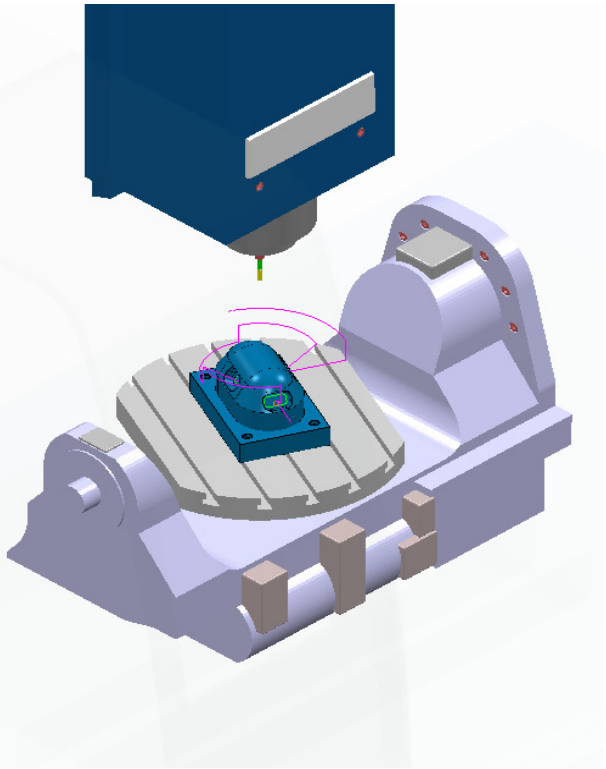
….\PowerMill_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA-Update.mtd

33 Turn down the **Opacity** on the static parts of the **Machine Tool model**.

34 In the **Machine Tool** toolbar select the **Machine Tool Display** option.

35 Use the **Jog** options to check the **+ve** and **–ve** movement of the **Rotary Table – C-Axis**.



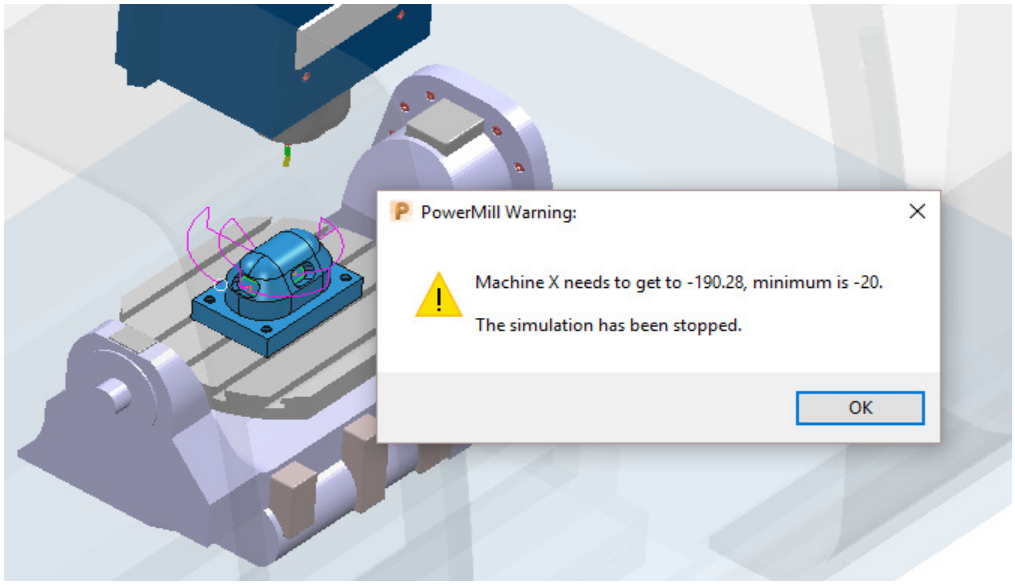


The updated **Rotary Table** – **C-Axis** movement is now exactly the same as in the original **Machine Tool Model** (***PMPOST-Mazak_Variaxis_630PSA***).

The next stage is to use a **Simulation** to check that all of the ***linear*** and ***rotary*** travel limits on the **Machine Tool Model** are correct.

36 Right mouse click on the toolpath **EM12-FinPkts** and from the local menu select **Simulate from Start**.

37 On the **Simulation** toolbar select the **Play** button.



The **Simulation** has immediately stalled due to one of the defined **X** axis the travel limits being exceeded.

38 Press the **Play** button again to allow the **Simulation** to continue but note that the **A axis** trunnion has been able to incorrectly rotate well beyond its **30** deg limit towards the back of the machine.

39 Return to the file in **Wordpad** and look for incorrect data in the section of the **mtd** file that defines the **Table** – **A-Axis**.

```
<!-- The table -->
<machine_part>
<axis>
<control_info ADDRESS="A" MIN="-30" MAX="120" />
<simple_rotary X="0" Y="0" Z="50" I="-1" J="0" K="0" />
</axis>
<model_list>
<dmt_file>
<path FILE="Mazak_Variaxis_630/Trunion1.dmt" />
 <rgb R="210 " G="210" B="255" />
```


Copyright © Autodesk

The above row that defines the A-Axis rotational travel limits and simply requires the MIN and MAX numbers to be swapped (The correct movement for +ve and -ve rotation has already been checked and confirmed to be okay).

40 Modify the above incorrect values to those shown below:-

<control_info ADDRESS="A" MIN="-120" MAX="30" />

41 **Save** the updated **mtd** file (but do not close it).

*This latest update will now be tested in the **PowerMill Project**.*

42 In the **PowerMill *explorer*** right mouse click on the machine tool ***PMPOST-Mazak_Variaxis_630PSA-Update*** and from the local menu select **Delete**.

*It is necessary to re-import the updated **MTD** file into the **Project**.*

43 In the **Machine Tool** toolbar select the **Import Machine Tool Model** option.

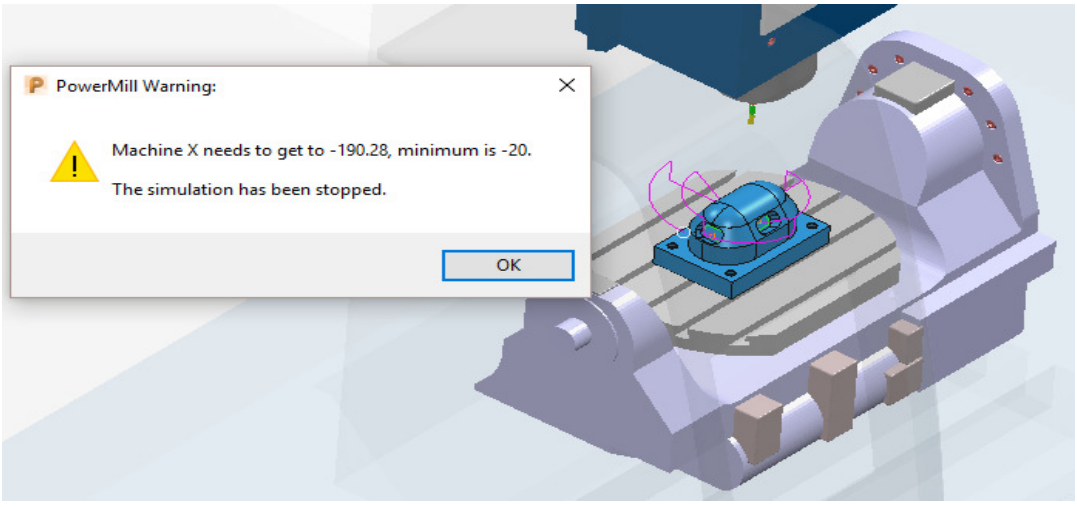
44 In the dialog select the **Machine Tool Model** file:-

….\PowerMill_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA-Update.mtd

45 Turn down the **Opacity** on the static parts of the **Machine Tool model**.

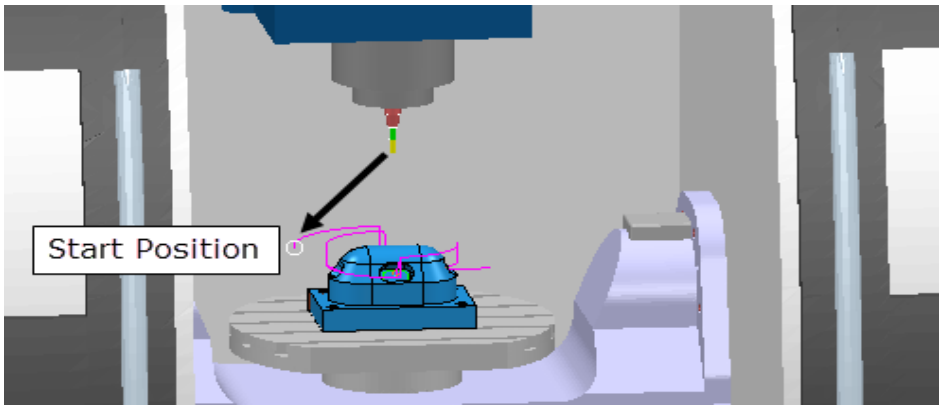
46 Right mouse click on the toolpath **EM12-FinPkts** and from the local menu select **Simulate from Start**.

47 On the **Simulation** toolbar select the **Play** button.



The **Simulation** immediately stalls and the **PowerMill Error** dialog is displayed informing that the current minimum limit for **X** is **-20**.

48 Select **OK** in the **PowerMill Error** box.



Note that the head is currently unable to move to the **-ve X** start position.

49 Continue to **Step Forward** and observe that the **Machine Tool Model** is only performing moves that exist from **X-20** to **X315**.

50 Return to the file in **Wordpad** and look for incorrect data in '**The head**' section of the **mtd** file that defines the **X** axis travel limits.

```
<!-- The head -->
<machine_part>
<axis>
<control_info ADDRESS="X" MIN="-20" MAX="315" />
<simple_linear I="1" J="0" K="0" />
</axis>
<machine_part>
<axis>
<control_info ADDRESS="Y" MIN="-450" MAX="315" />
<simple_linear I="0" J="1" K="0" />
 </axis>
 <machine_part>
```

The above row defining the MIN **X** travel limits contains an incorrect value of **"-20"**. This value must be changed to **"-315"**.

51 Modify the above incorrect value to that shown below:-

<control_info ADDRESS="X" MIN="-315" MAX="315" />

52 **Save** the file.

*This final update will now be tested in the **PowerMill Project**.*

53 In the **PowerMill *explorer*** right mouse click on the machine tool **PMPOST-Mazak_Variaxis_630PSA-Update** and from the local menu select **Delete**.

Copyright © Autodesk

*It is necessary to re-import the updated **MTD** file into the **Project**.*

54 In the **Machine Tool** toolbar select the **Import Machine Tool Model** option.

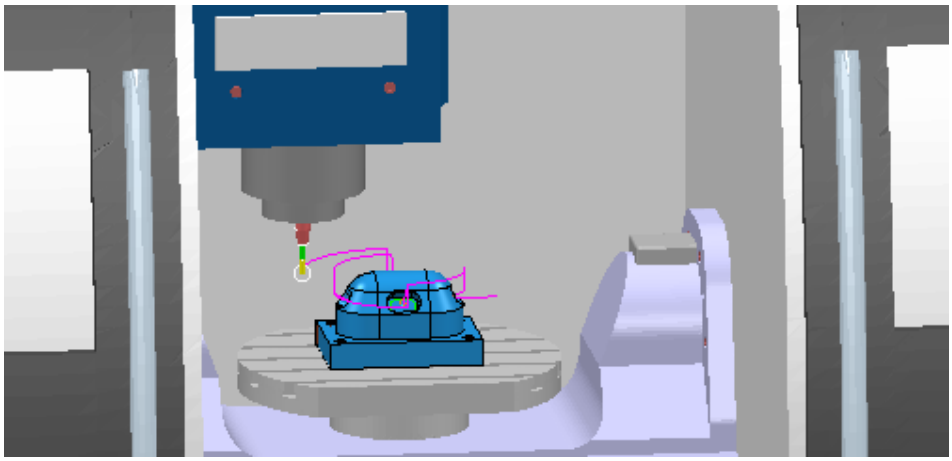
55 In the dialog select the **Machine Tool Model** file:-

….\PowerMill_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA-Update.mtd

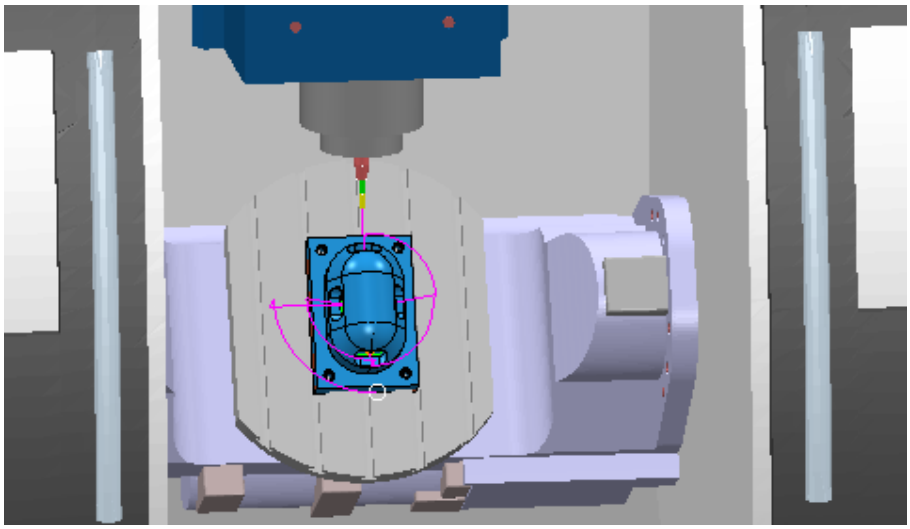
56 Turn down the **Opacity** on the static parts of the **Machine Tool model**.

57 Right mouse click on the toolpath **EM12-FinPkts** and from the local menu select **Simulate from Start**.

58 On the **Simulation** toolbar select the **Play** button.



The simulation will now successfully runs from start to finish without stalling. All linear and rotary moves should be an exact match compared with the original **mtd** file, **PMPOST-Mazak_Variaxis_630PSA**.



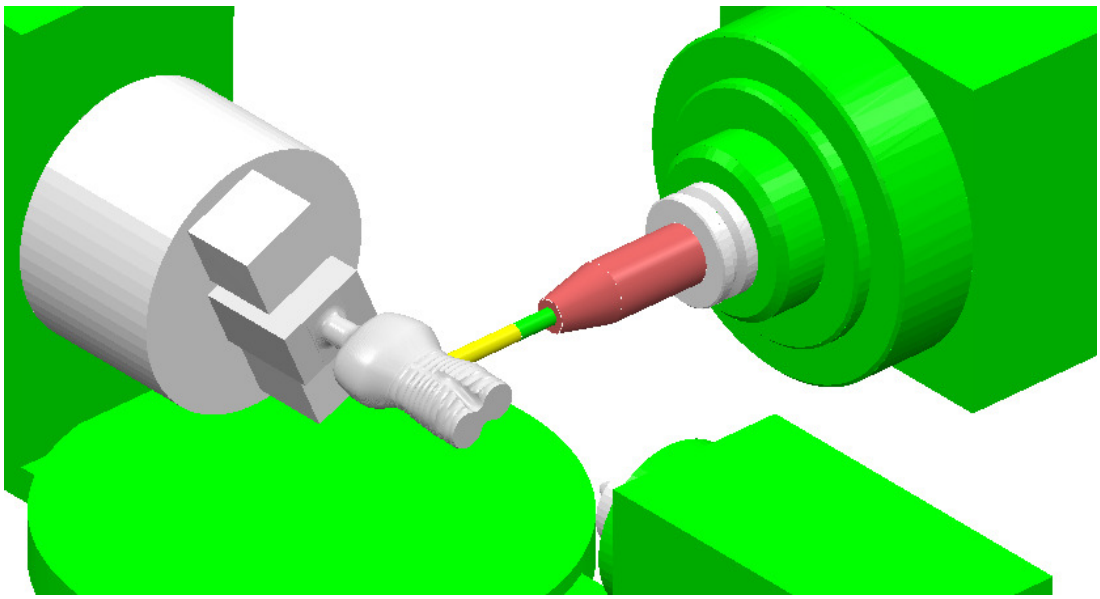
59 Select **File** - **Save Project**

Copyright © Autodesk

14. Four Axis Rotary Finishing

Introduction.

This **Finishing** strategy is designed for machining a component mounted on a single, programmable **Rotary Axis**. During milling, the component rotates around the rotational **X-axis** while the cutter performs simultaneous **3-Axis** movements.



The main options are summarised below.

X Limits

The **X Limits** define the absolute limits of the finishing path along the rotational, X-axis. These can be manually defined, or automatically set to the limits of the block.

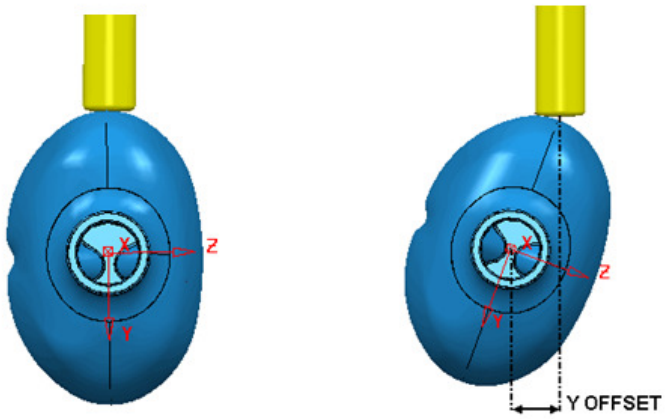
Pattern

This enables the cutting method to be specified for rotary milling either **Circular**, **Line**, or **Spiral**.

Copyright © Autodesk

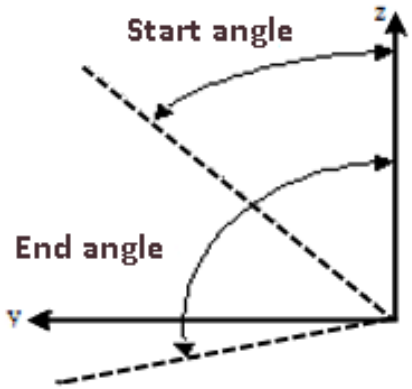
Y Offset

A **Y Offset** distance can be specified to avoid cutting with the tip of the tool. This view along the X-axis shows how the **Y Offset** (if active) is applied to the Rotary form:



Angular Limits

The **Angular Limits** section of the dialog is only available when using the ***Style*** as **Circular** or **Line**. The ***Angular limits*** are defined between a **Start** and **End** angle.



The angular limits are measured in an anti-clockwise direction when viewed along the positive X-axis. The area machined is between the start and end angles.

Cut Direction

This option determines whether **Climb**, **Conventional**, or **Any** milling directions are used.

Stepover

In the case of **Circular** and **Spiral** this is defined as the pitch for each programmed revolution of the component. For **Line** this is defined as the angular, stepover between adjacent tool tracks.

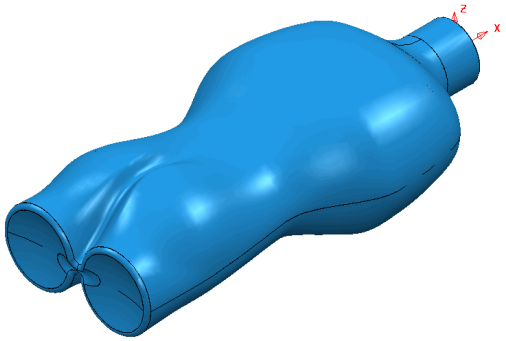
Copyright © Autodesk

Circular Rotary Machining

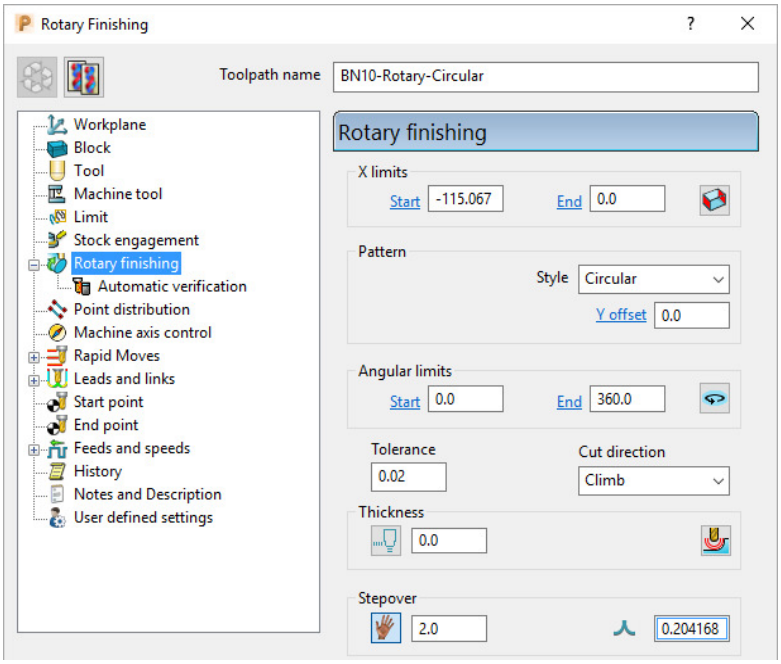
In this example a model of a bottle with its centre along the X-axis will be used. Using the ***Style*** as **Circular**, the job rotates with the tool aligned to a fixed direction. While the component rotates, the tool moves back and forth along its axis to generate the sectional form. The tool then steps over by the **Pitch** value and the rotational machining process is repeated.

1. Select **Delete All** and **Reset forms**.
2. **Import** the **Model**:

   ...\PowerMill_Data\Models\rotary_bottle.dgk



3. **Calculate** the **Block** using ***Defined by*** - **Box** and **Type** - **model**.
4. Create a Dia 10 Ball nosed tool named **BN10**.
5. In the **Rapid Move Heights** dialog select **Calculate**.
6. In the **Start** and **End Point** dialog **Use** – **Absolute** using the coordinates **20 0 40**.
7. Set the ***Short/Long*** **Links** to **Skim** and ***Default*** **Links** to **Incremental**.
8. In the **Strategy Selector** select **Finishing** > **Rotary Finishing**.

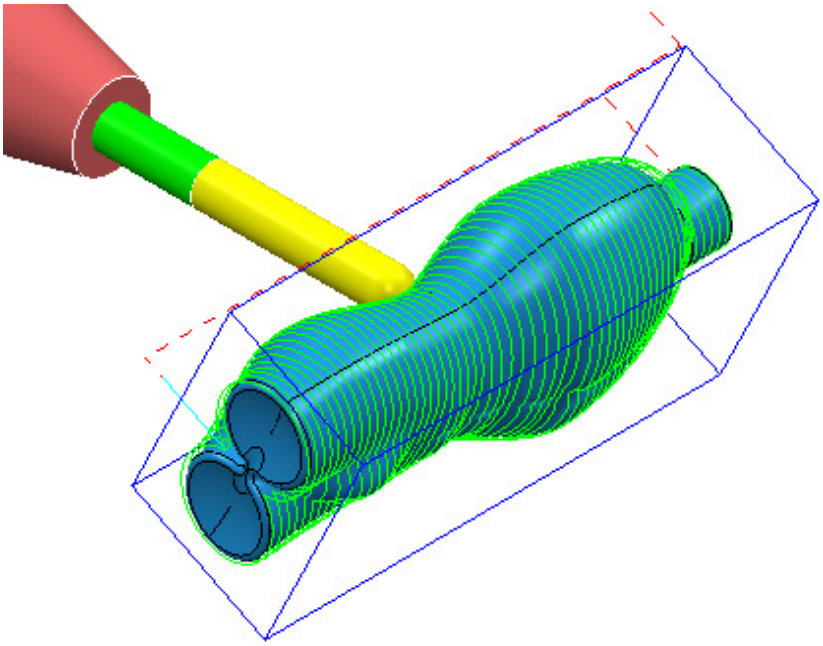


- Enter ***Name***: **BN10-Rotary-Circular**
- Click on **Defines X limits as block limits**.
- Set ***Style***: **Circular**
- Select ***Cut Direction***: **Climb**
- Input ***Stepover***: **2**

9 Select **Calculate** to create the toolpath then **Cancel** the dialog.

10 Right mouse click the toolpath **Rotary1_BN10** and from the local menu select **Simulate from Start** to open the **Simulation** Toolbar.

11 Select the **Play** button on the **Simulation** toolbar .



*By applying **Tool view** point the rotation of the component is simulated (As if viewing the actual machine tool).*

In the above example each section is machined in the same **Climb** milling direction. The entire length of the job is machined since the **X Limits** are set to the **Block** limits. Selecting **Conventional** will produce tool tracks travelling in the reverse direction and **Any** will produce alternate **Climb** and **Conventional** tool tracks along the job.

12 **Recycle** the toolpath, select **Any** for the **Cut direction** and select **Calculate** to create the toolpath before selecting **Cance**l to close the dialog.

13 **Simulate** the **toolpath** as before to observe the effect of the toolpath reversing direction with every new section machined.

Copyright © Autodesk

Line Rotary Machining

Using the ***Style*** as **Line**, the tool feeds in the X direction following the component form. At the end of each pass the tool retracts and moves above the start of the next pass. At the same time the rotary axis indexes by the angular stepover and the tool then leads onto the next machining move.

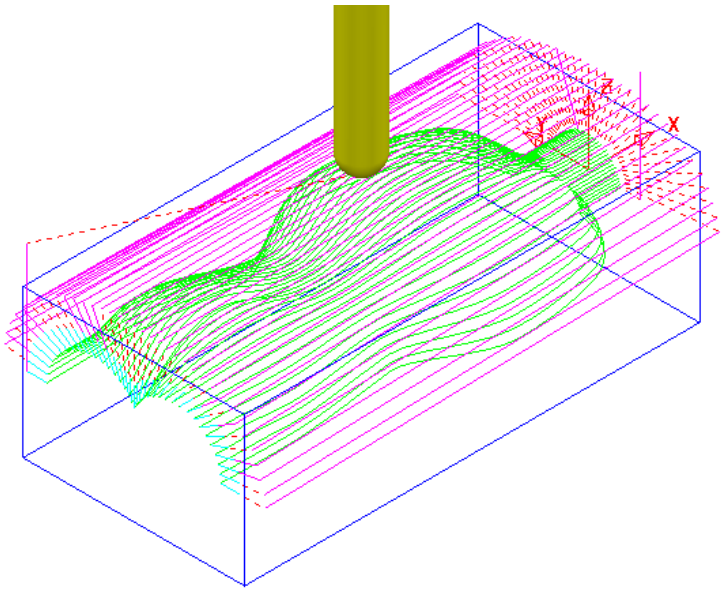
14 **Make a Copy** of the previous toolpath named **BN10-Rotary-Line** and select **Style** - **Line** using **Cut Direction** > **Climb milling**.

15 For **Angular Limits** - **Start Angle**, enter **90** and **End Angle**, enter **-90**.

16 From the **main** toolbar select **Leads and Links and** set all of the **Links** to **Skim**.

17 From the same dialog, select **Moves and Clearances** and set the **Axial clearance** to **20**.

18 Click on **Calculate** and **Close** the dialog.



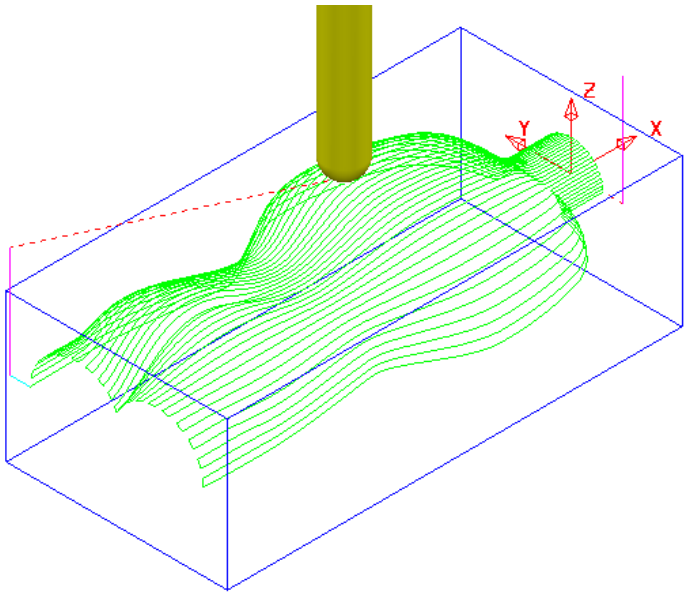
19 **Simulate** the new, uni-directional toolpath (shown above with a larger step over).

20 **Make a Copy** of the previous toolpath and name it as:-

BN10-Rotary-Line-2way.

21 Change the **Cut Direction** to **Any**.

22 Apply the dialog and **Simulate** the new, bi-directional toolpath.

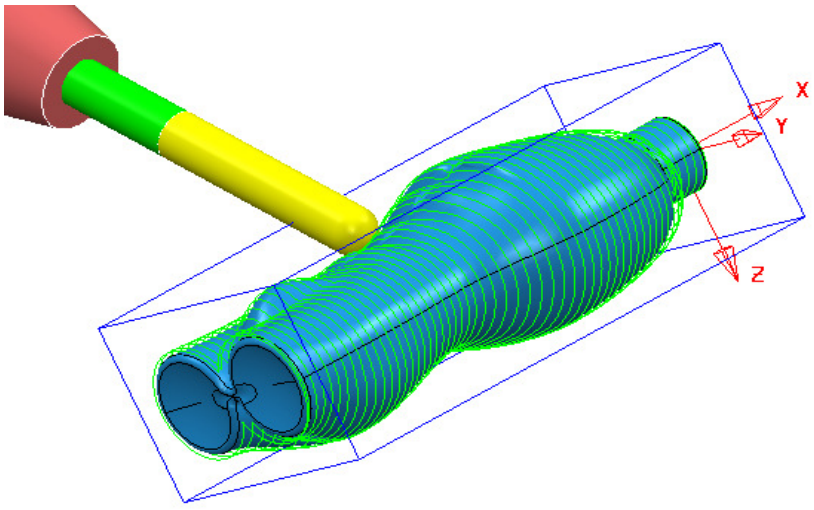


Spiral Rotary Machining

Using the **Spiral** technique, a continuous toolpath is generated around the form as the tool advances along the X-axis. To ensure a clean finish the toolpath starts and finishes with a constant X position, sectional pass. Due to a **Spiral** toolpath being a single, continuous track the cutting direction will be either **Climb** or **Conventional** milling. For the same reason the **Angular Limits** option is not viable and as a result is blanked out.

23 **Make a Copy of** the previous toolpath and name it as **BN10-Rotary-Spiral** and select **Style** - **Spiral** using **Cut Direction** > **Climb milling**.

24 Click on **Calculate** to produce the toolpath shown below.



25 **Simulate** the new toolpath.

26 **Save** the **Project** as:-

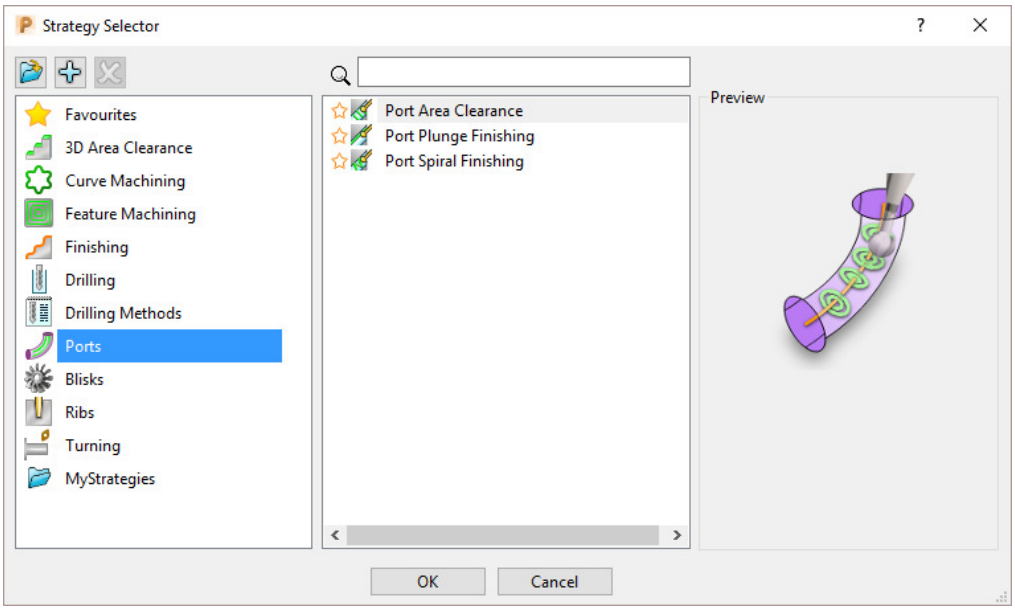
….\COURSEWORK\PowerMill_Projects\ Rotary_Bottle-EX1

Copyright © Autodesk

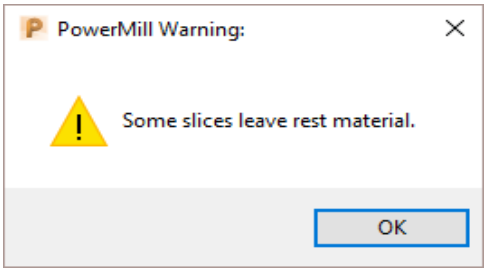
15. Port Machining

Introduction

In the **Strategy Selector** the **Ports** option provides 3 specialist strategies, designed for machining cylinder head, inlet/outlet ports.



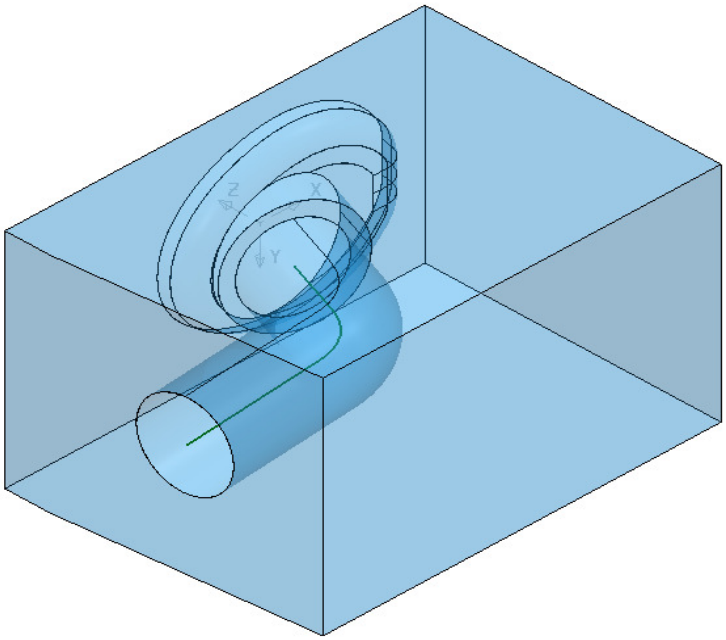
The **Port Machining** options automatically provide multi-axis, tool alignments where this is possible. A warning will be displayed if impossible alignment limits are reached and the port is not fully machined.



Basic Single Port Example

1 **Open** the 'read-only' **Project**:

...\PowerMill_Data\five_axis\Ports\SinglePort1-Start



2 **Save Project As**:-

...\COURSEWORK\PowerMill-Projects\SinglePort1-ex1

*The **Project** already contains all required cutting **Tools**, and preliminary **3Plus2** toolpaths.*

The following section will take the user through the specialist **Port Area Clearance** and **Port Spiral Finishing options**.

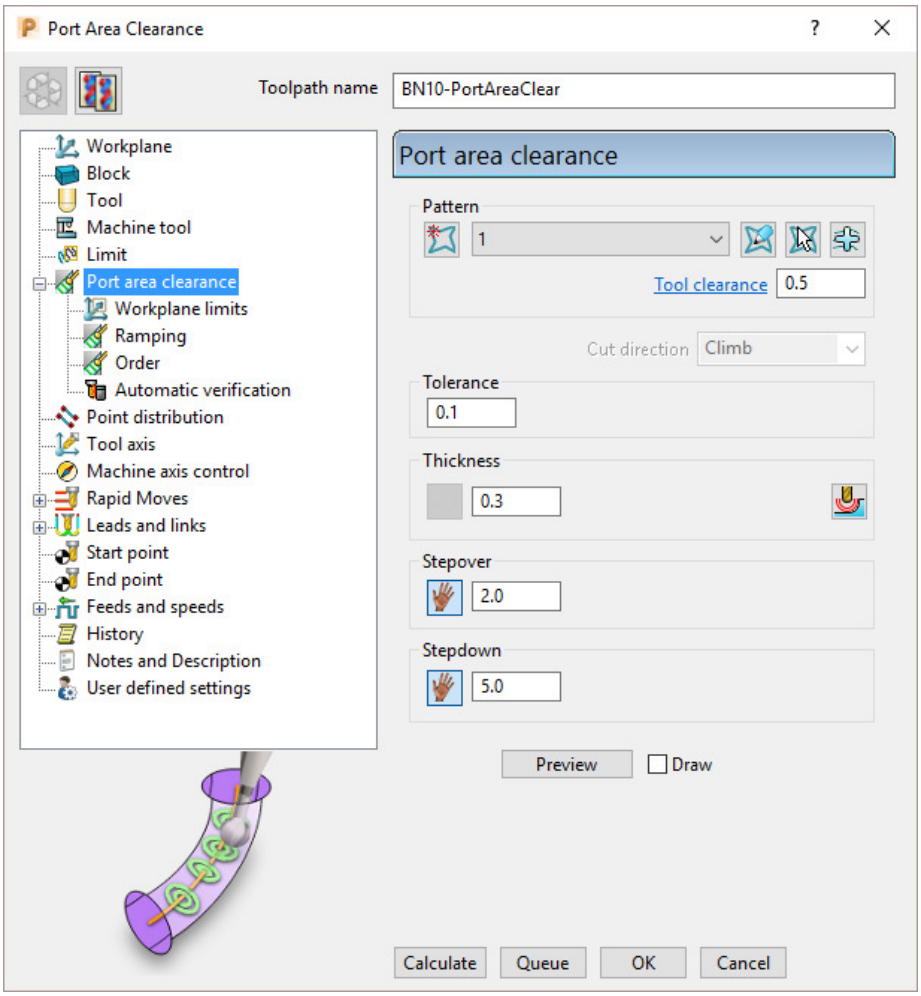
A **Pattern** (**1**) is included running down the centre of the **Port**. This is required to control the extent and alignment of the **Port Machining** strategies.

3 **Activate** the existing **Toolpath 1** to reinstate all the settings.

4 **Activate** the **BN10** tool.

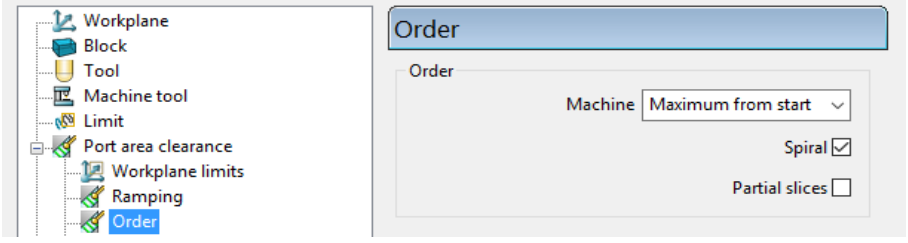
5 In **Toolpath Strategies** select **Ports** > **Port Area Clearance** and fill in the 3 pages of the dialog exactly as shown below.

Copyright © Autodesk



a Insert ***Pattern***: **4**

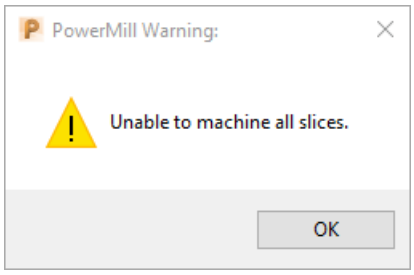
b Input ***Tool Clearance***: **0.5**



a Tick **Spiral**.

b Untick **Partial slices**.

6 **Calculate** and note the following advisory warning message:



The message is advising that the **Tool** is unable to fully machine the form to the current settings.

Later a visual check with **ViewMILL** will show that enough material has been removed to allow the finishing strategies to run.

7 Select **OK** to the message and wait for the toolpath to finish processing.

8 **Close** the dialog.

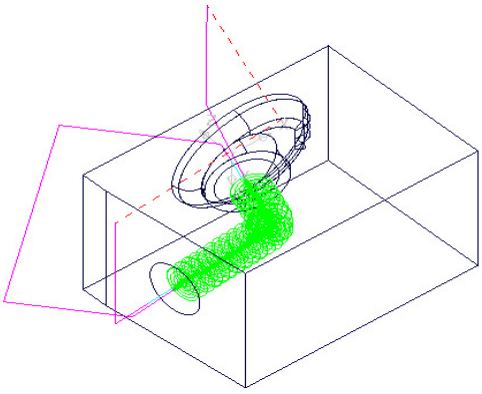
9 **Activate** Toolpath **1**.

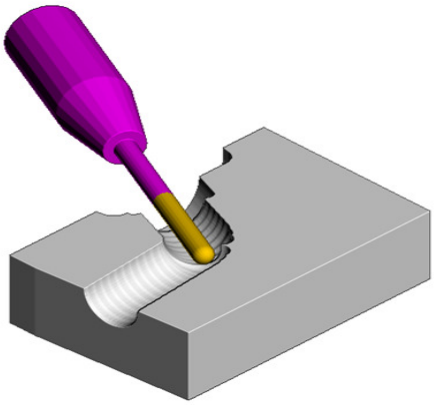
10 Right mouse click on toolpath **1** and from the local menu select **Simulate from start**.

11 Open the **Block** dialog and change the **Max Z** value from **40** to **0** before selecting **Accept**.

By defining the Block to be half the height of the model it will be possible to visualise the lower half of the port being machined.

12 Select an **ISO 2** view and run a **ViewMILL** simulation on all of the remaining toolpaths in sequential order.

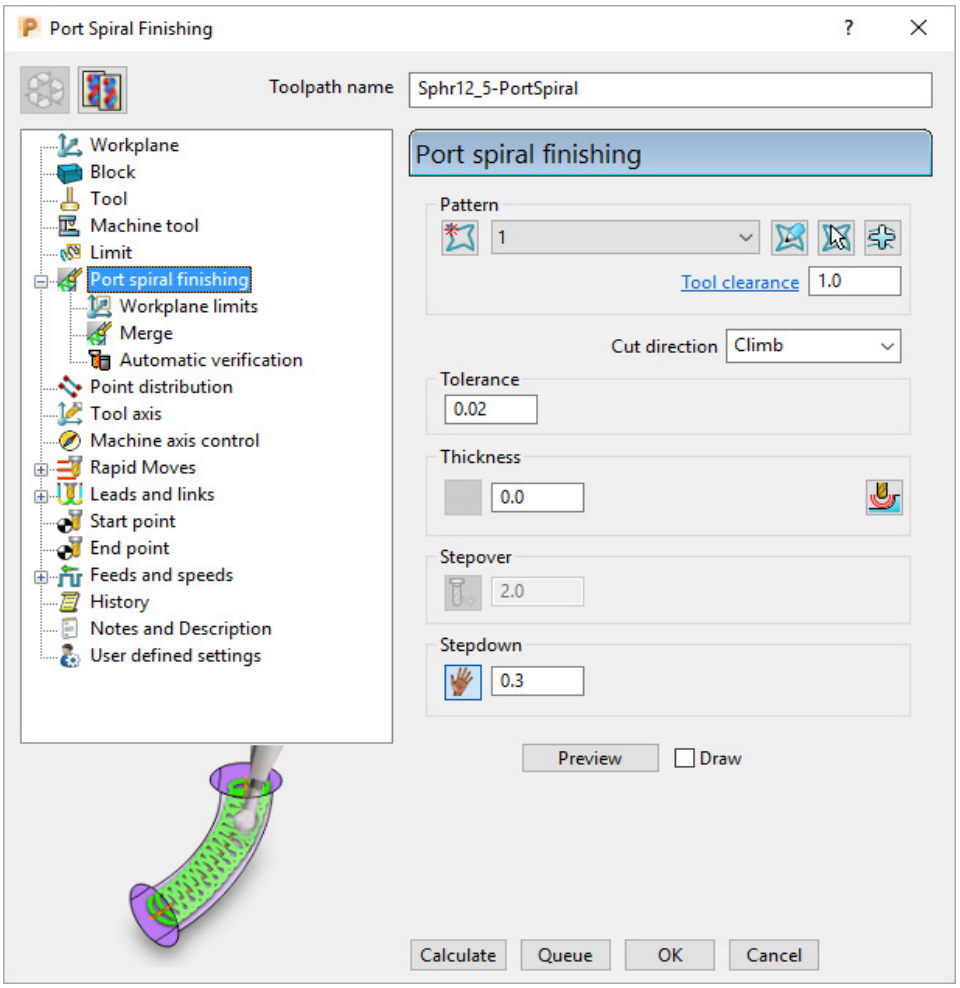


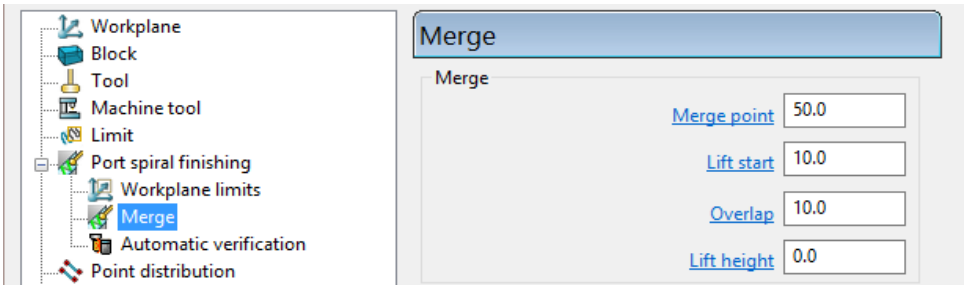


13 **Activate** the **SPHR12.5** tool.

14 In **Toolpath Strategies** select **Ports** > **Port Spiral Finishing** and fill in the two featured pages of the dialog exactly as shown below.

Copyright © Autodesk



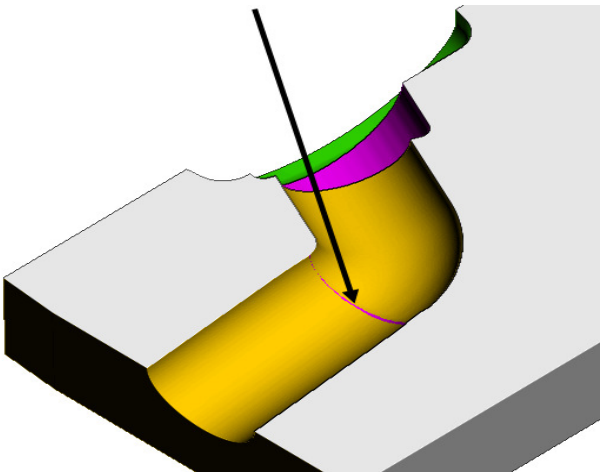


The above **Merge** settings enable the **thickness** to gradually increase as it merges into the areas previously machined from the first end. This is to compensate for the cutter tending to dig in when the loading decreases.

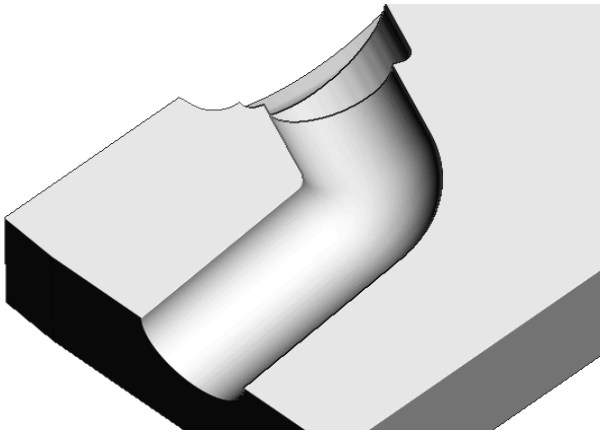
In this case a ***Lift height*** of **0** has been applied.

15 Select **Calculate** and once the toolpath has been created select Cancel.

16 Perform a **ViewMILL simulation** on the latest toolpath, **Sphr12_5-PortSpiral**.



Note the **0.3mm** step where the machining from the second end of the port has gradually moved away from the form over the last **10mm** to the blend point.



If the **Lift Height** is **0** then there will not be a visible step appearing on the toolpath running from the second end.

There is however a high risk of the **Tool** digging into this area if this **Toolpath** was to be run on the machine tool.

17 Select **File** > **Save Project**.

Copyright © Autodesk

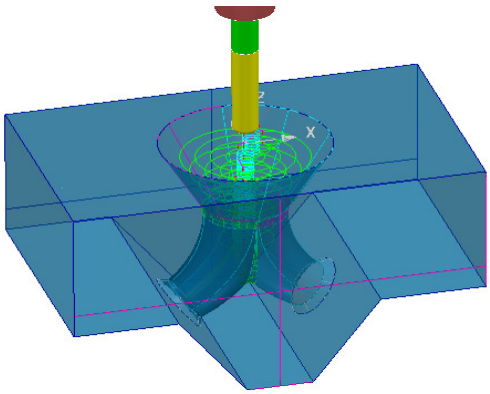
Twin Port Example

Only one Port at a time can be processed within the Port machining strategies. For this example, 2 reference surfaces have been created to blank access to the extra port while applying the **Port machining** strategies to the remaining ***single Port*** setups.

1 Select **File** > **Delete All** and **Tools** > **Reset Forms**.

2 **Open** the read only **Project**:

...\PowerMill_Data\five_axis\Ports\DualPort_Start



3 **Save Project As**:

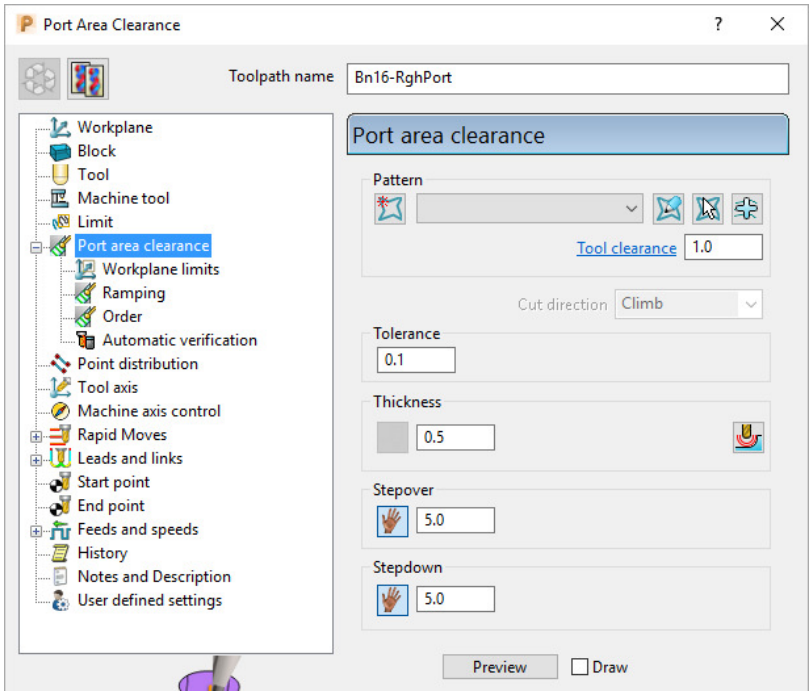
...\COURSEWORK\PowerMill-Projects\DualPort-ex1

4 **Activate** the existing toolpath **RghPort-3Axis** to reinstate all the settings.

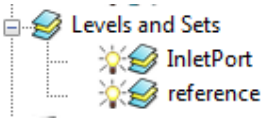
5 **Activate** the **BN16** tool.

6 **Calculate** a new Block to the full **Model** dimensions.

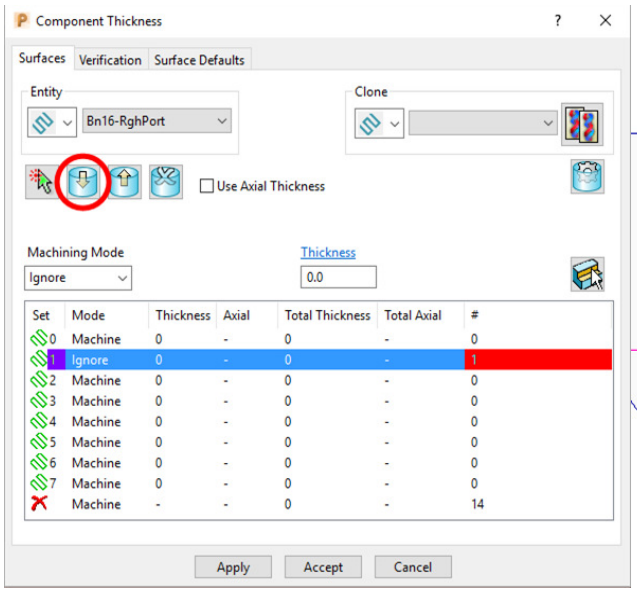
7 In **Toolpath Strategies** select **Ports** > **Port Area Clearance** and fill in the pages of the dialog exactly as shown below.

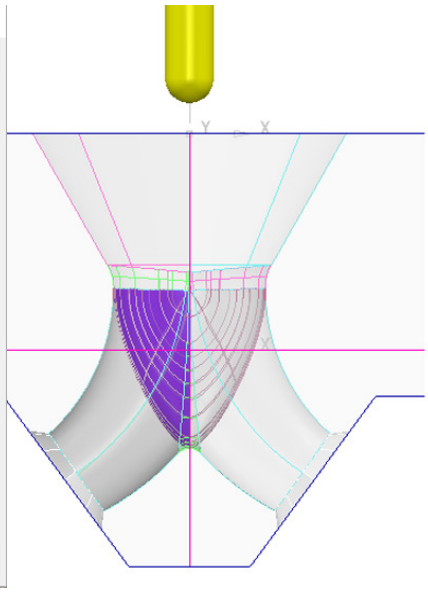


8 Switch on the **Level** named **reference**.



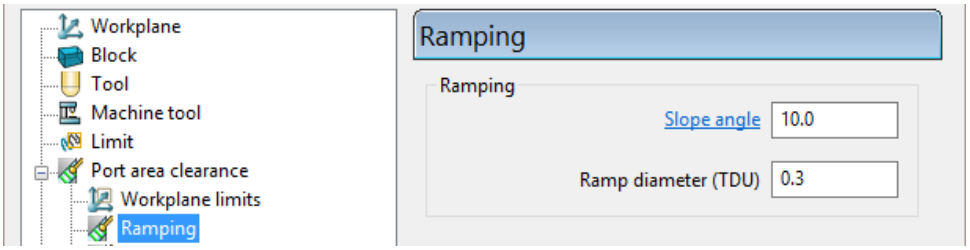
9 From the **Port Area Clearance** main page, select the **Component Thickness** icon .

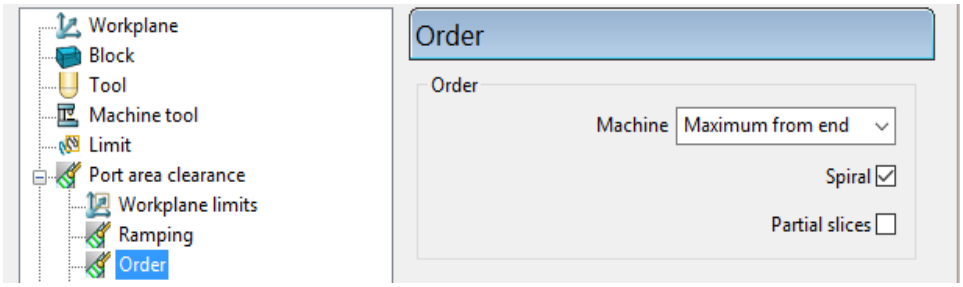




10 In the **Component Thickness** dialog, select and acquire the left hand reference surface (Shaded purple above right) to a row with the **Machining Mode** set to **Ignore**.

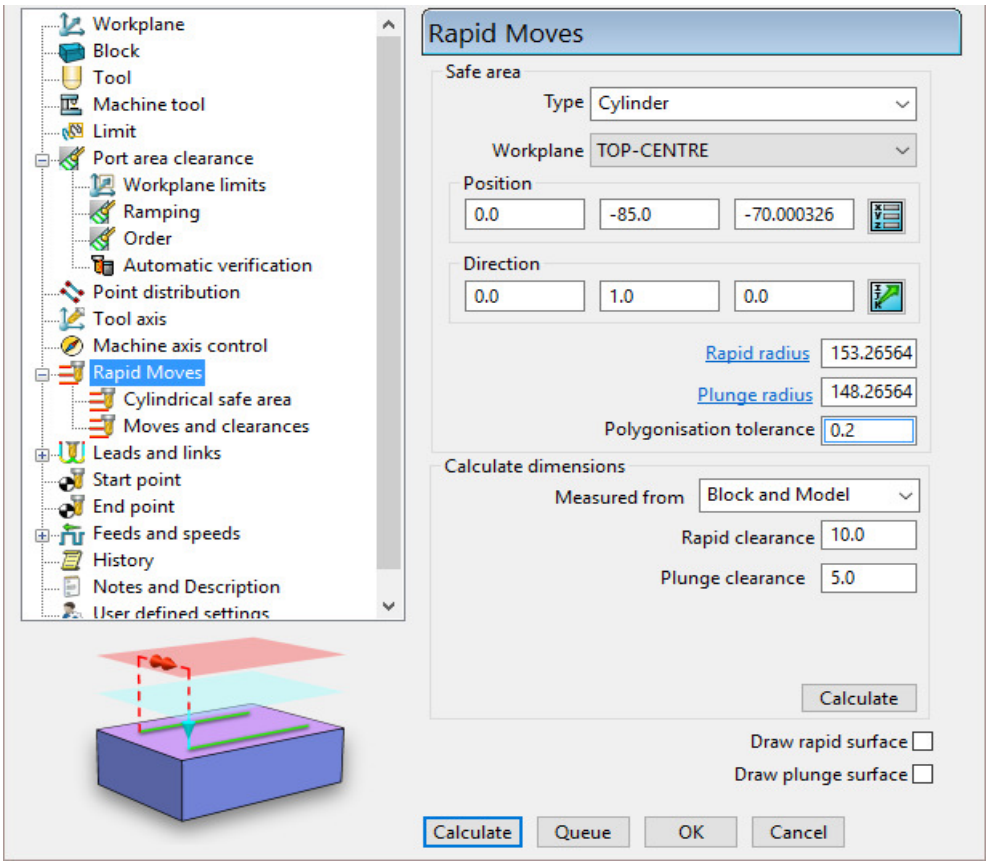
11 **Apply** the dialog.



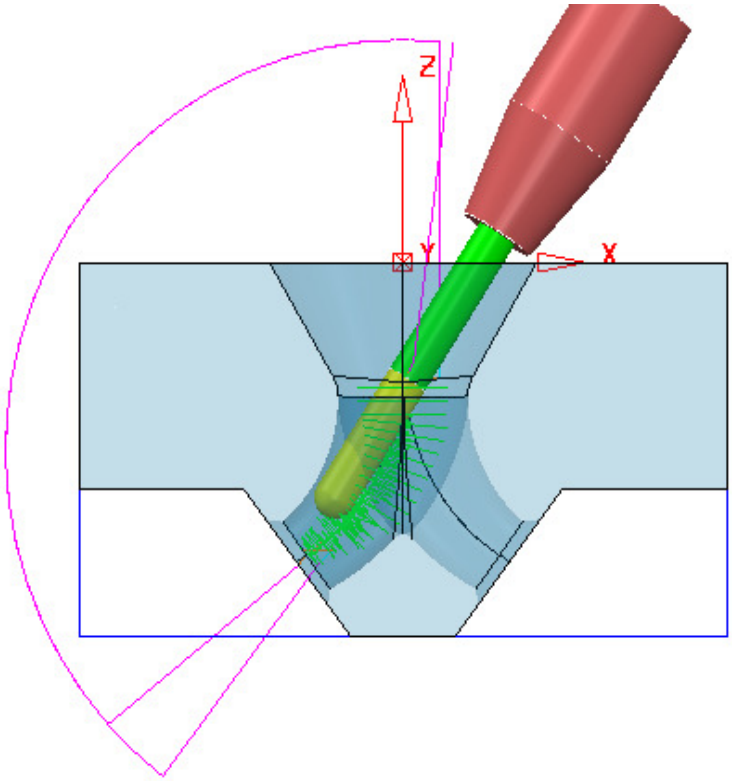


12 In the **Order** page tick **Spiral**.

Copyright © Autodesk

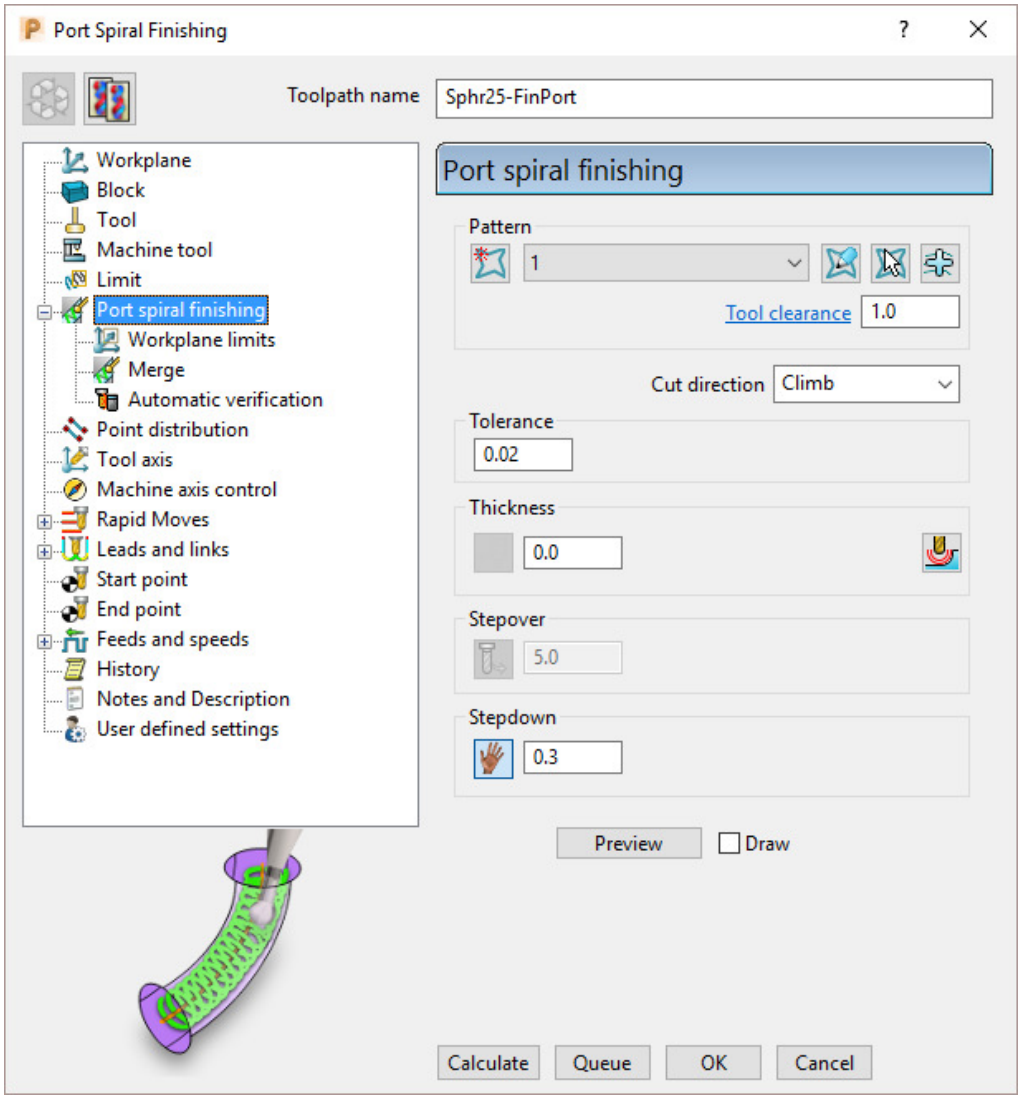


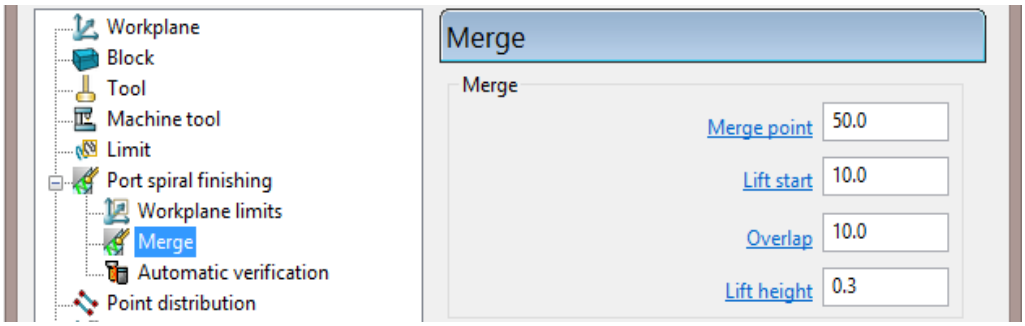
13 **Calculate** the **Port area clearance** dialog.



14 **Activate** the **SPHR25** tool.

15 In **Toolpath Strategies** select **Ports** > **Port Spiral Finishing** and fill in the pages of the dialog exactly as shown below.

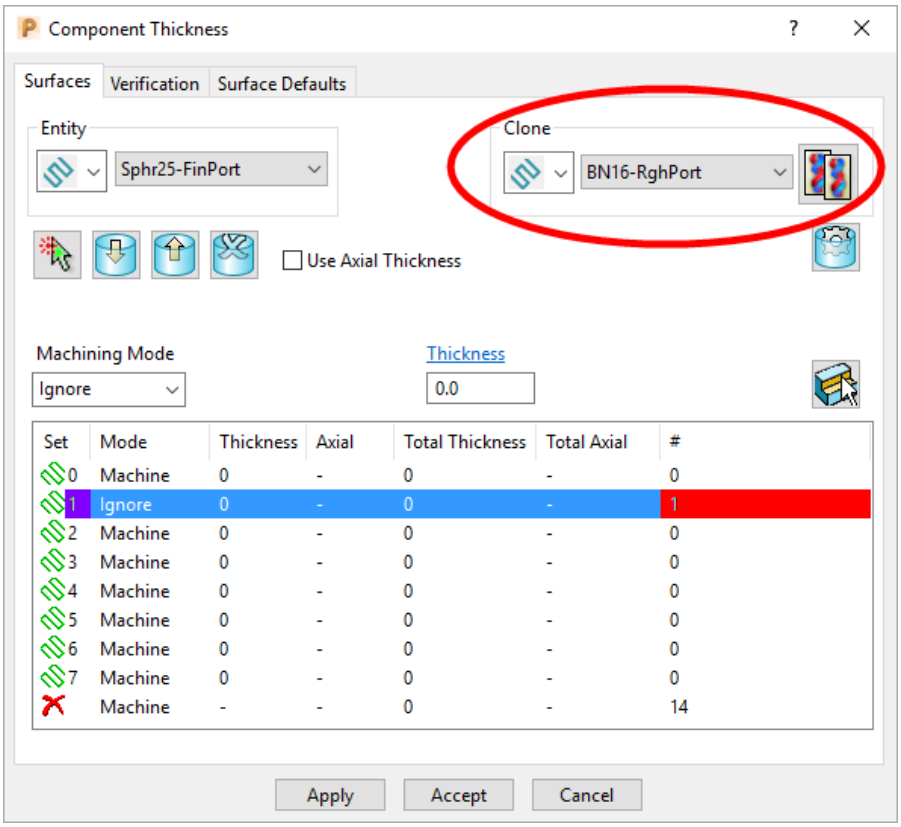




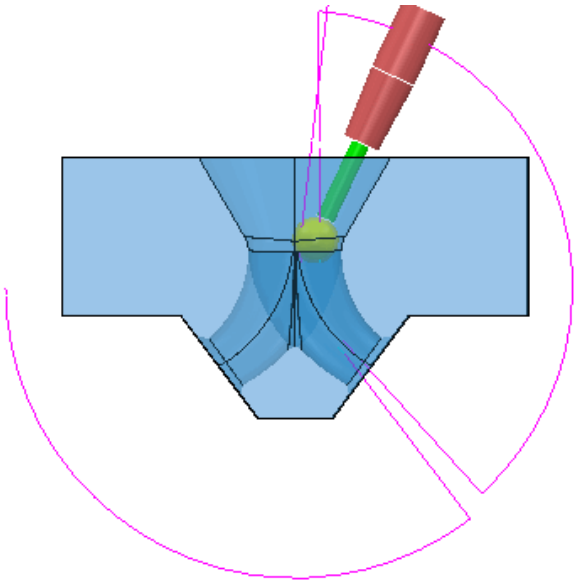
16 From the **Port Area Clearance** main page, select the **Component Thickness** icon .

Copyright © Autodesk

17 As shown in the following diagram, ***Clone*** the **Component Thickness** settings from the previously created toolpath **BN16-RghPort**.



18 **Apply** and **Accept** the **Component Thickness** dialog, and then **Calculate** the **Port spiral finishing** strategy.



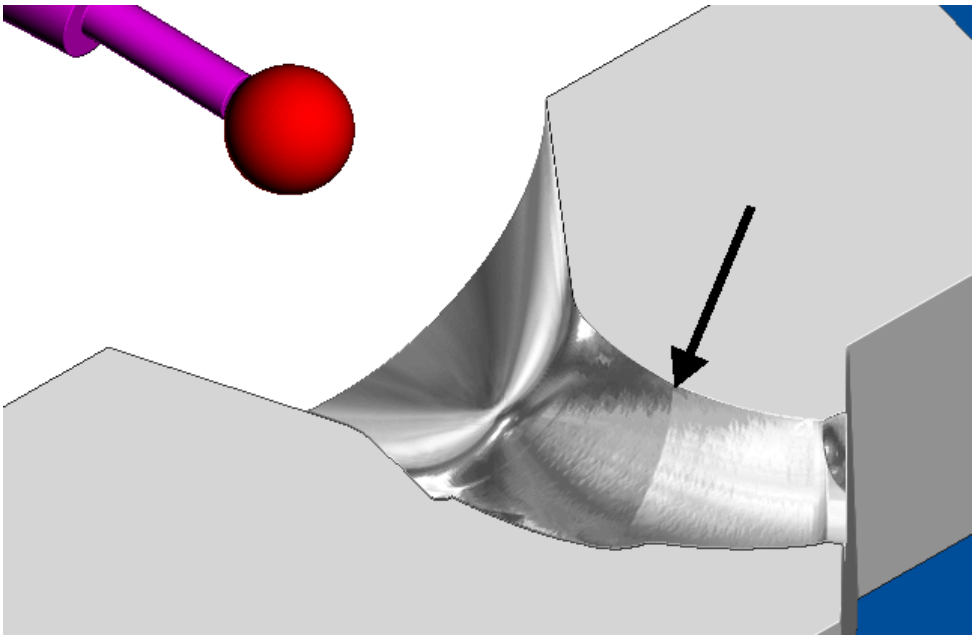
19 Activate **Workplane mc-OUTPUT**.

20 Activate ***Boundary*** **1**.

21 Calculate a **Block** - **Defined by** - **Boundary** and **Type** - **Model**.

22 Change the **Max Z** value to **-75.0**.

23 Select an **ISO 3** view, open and run a **ViewMILL simulation** of all the toolpaths.



24 Note the **0.3mm** step where the machining from the second end of the Port has gradually moved away from the form over the last **10mm** to the blend point.

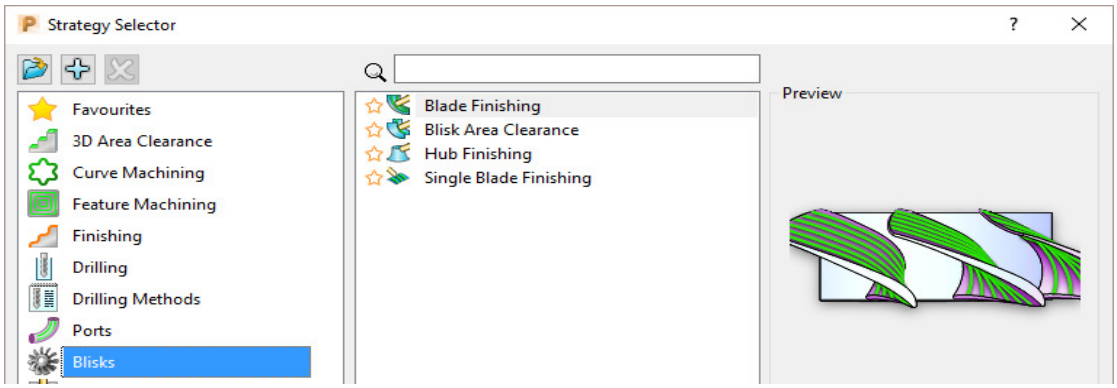
25 Select **File** > **Save Project**.

Copyright © Autodesk

16. Blisks

Introduction

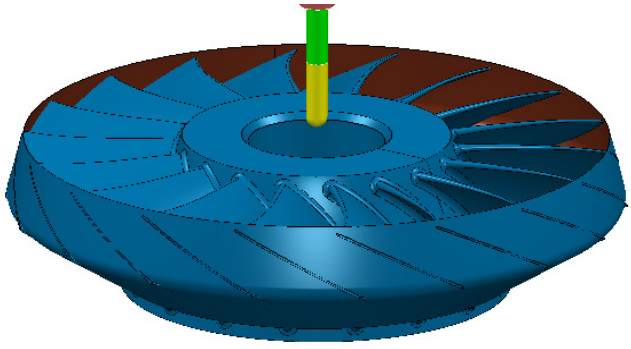
In toolpath strategies, the **Blisks** option provides 4 specialist strategies for machining Impellers.



Impeller Example

1. Select **File** > **Delete All** and **Tools** > **Reset Forms**.
2. **Open** the 'read only' **Project**:

   ...\PowerMill_Data\FiveAxis\Blisks\BliskSimple_Start



3. Select **File** > **Save Project As**:

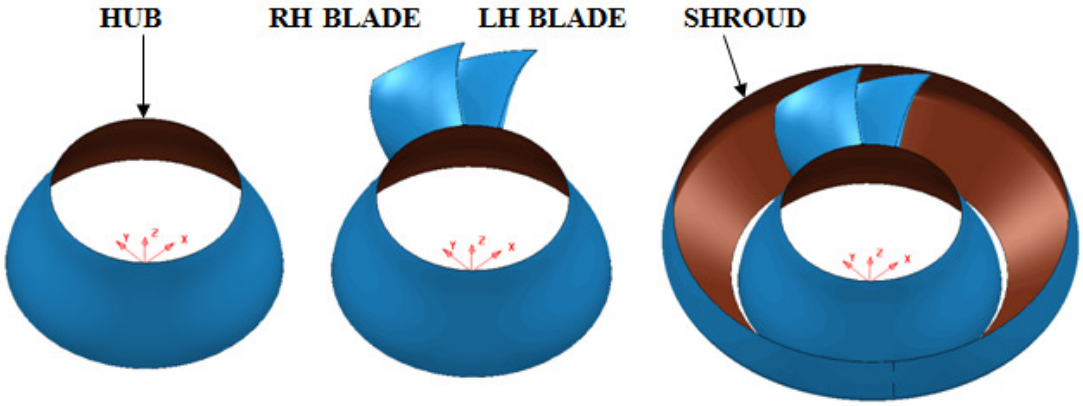
   ...\COURSEWORK\PowerMill_Projects\FiveAxis\Blisks\Blisk_Example
4. Activate **Workplane 1**.
5. Create a ***Cylindrical*** **Block** to the **Model** dimensions.

6 **Calculate** the **Rapid Move Heights**.

7 Set both **Start** and **End Points** to **First**/**Last Point** Safe.

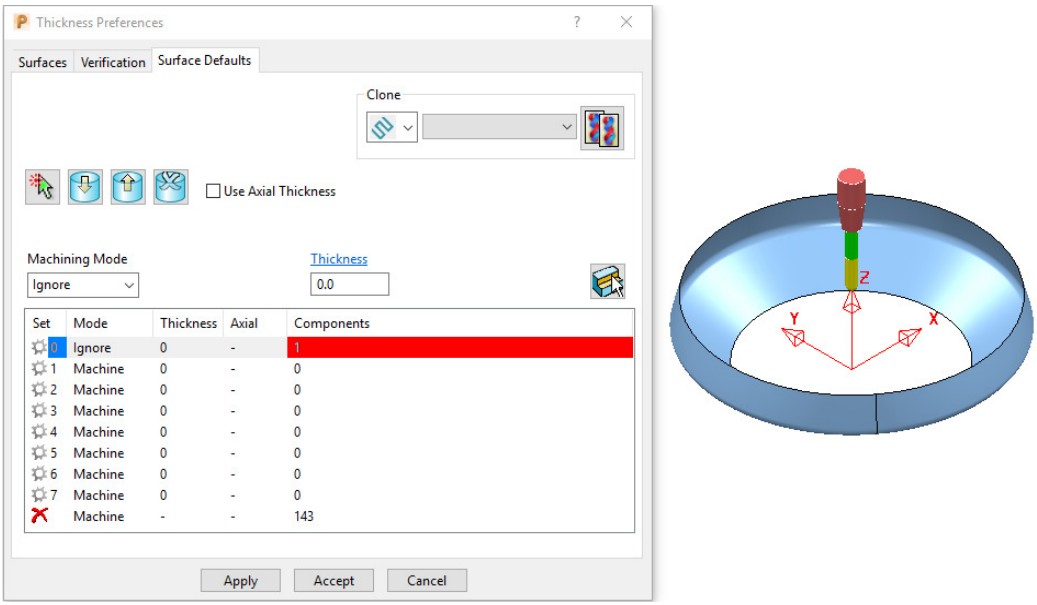
8 In the **Leads and Links** dialog, set **1st** and **2nd choice**, **Links** to **Short**/**Long Skim** and **Default Incremental.**

To be able to apply the **Blisk** machining options the component surfaces must first be acquired to a series of specifically named levels. In this example the surfaces in the imported model are already acquired to appropriate levels.



9 Activate the existing Ball Nosed tool named **BN25**.

10 From the **main** toolbar, select **Default Thickness** and in the **Thickness Preferences** dialog, select the **Surface Defaults** tab.



11 Select the single **Shroud Surface** and acquire it to the first row in the above dialog.

12 Set the first row to **Machining Mode** - **Ignore** and **Apply** the settings, before closing the dialog by clicking **Accept**.

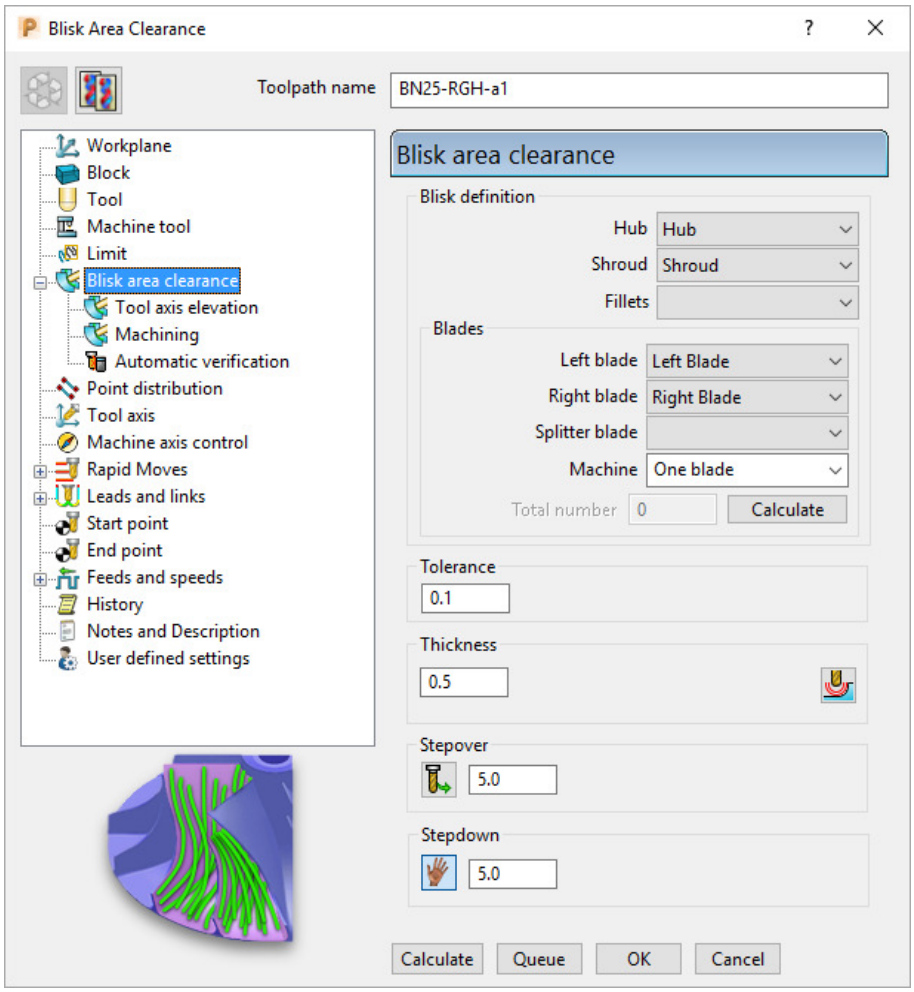
13 In the **Leads and Links** options input **Moves and clearances** set the **Axial clearance** to **15**, and set **Planar skim moves** to ***Plane*** - **Safe Area**.

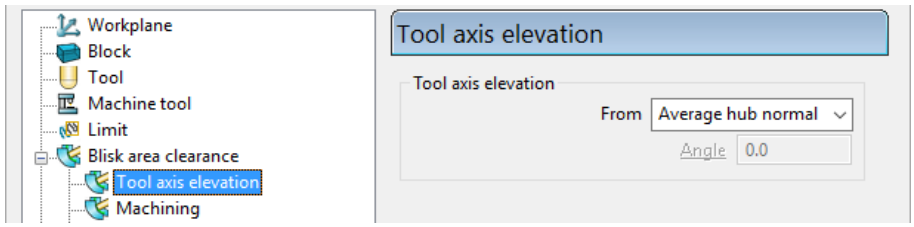
Copyright © Autodesk

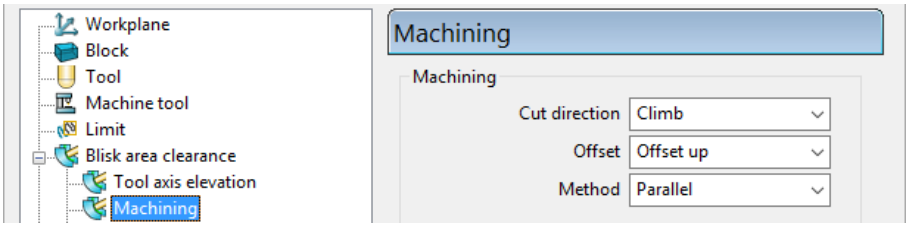
14 For both **Lead in** and **Lead out**, input **1st choice** – **Extended move 10** and **2nd choice** – **Surface normal arc** – **Angle 90** and **Radius 5**.

15 In the **Toolpath Strategies** dialog, select **Blisks** and from the four available options, select **Blisk Area Clearance**.

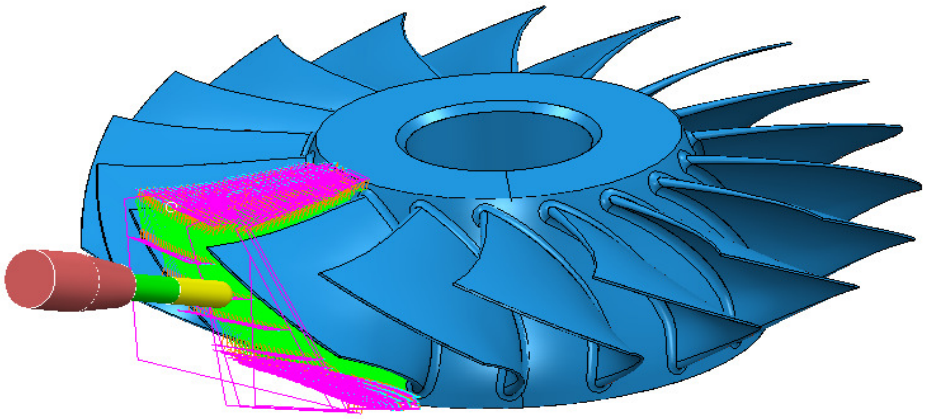
16 Enter data exactly as shown below with the **Fillets** and **Splitter Blade** options left blank.







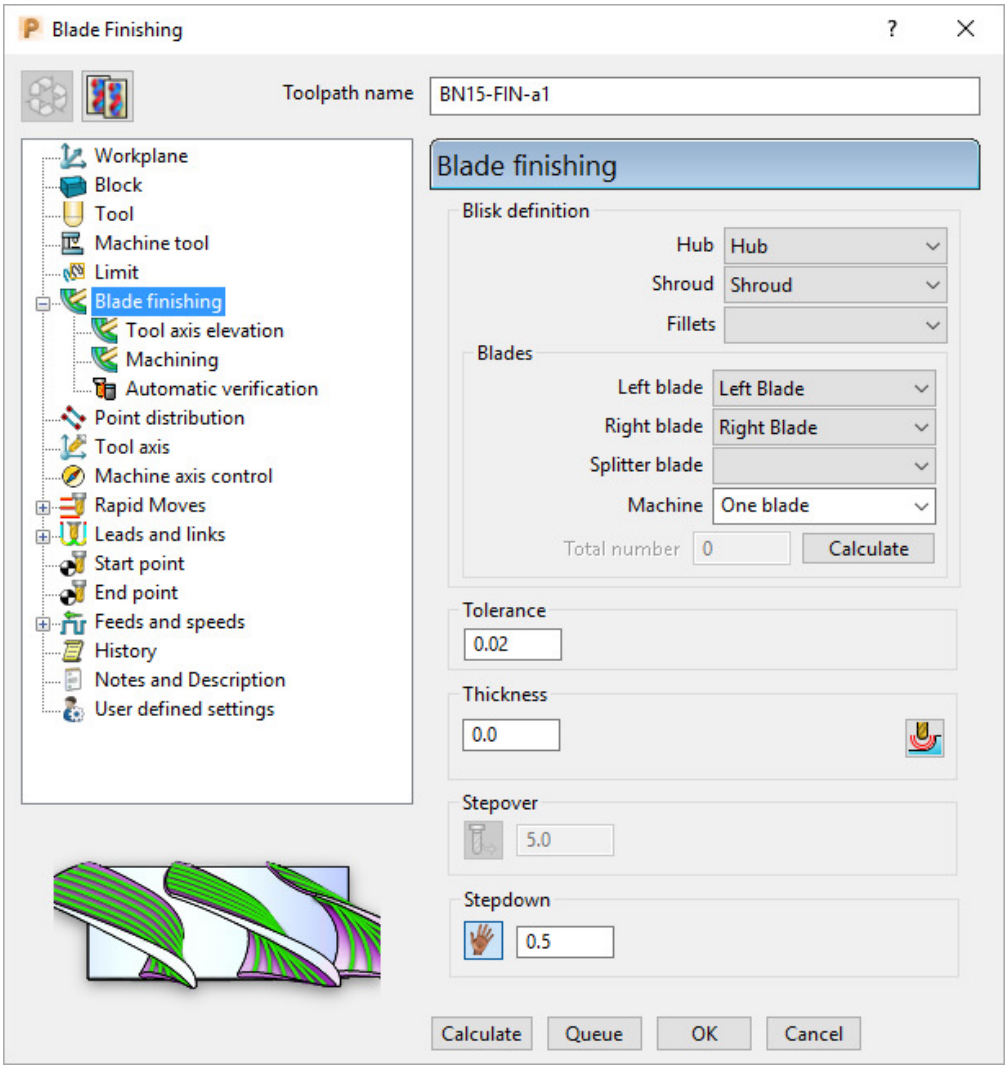
17 Select **Queue** in the dialog to commence background toolpath processing.



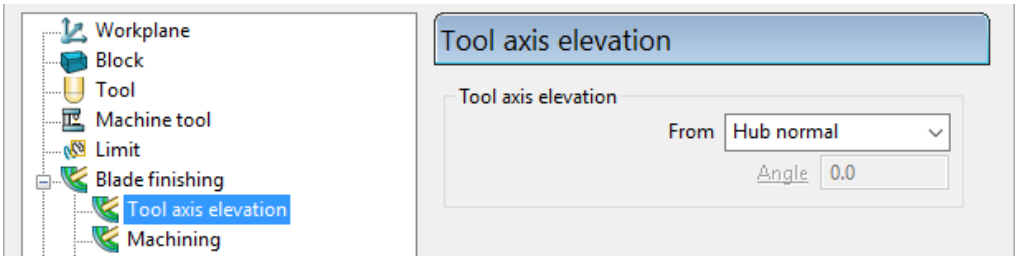
18 Activate the existing **Ball Nosed** tool named **BN15**.

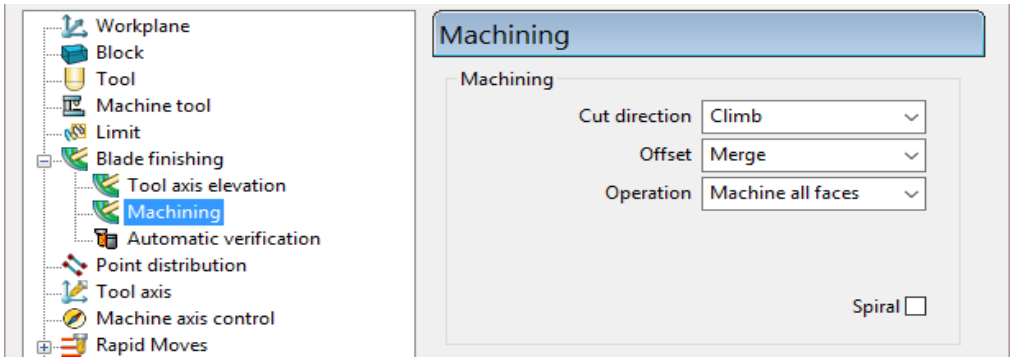
19 In the **Toolpath Strategies** dialog, select **Blisks** and from the three available options, select **Blade Finishing**.

20 Enter data into the pages of the dialog exactly as shown below with the **Fillets** and **Splitter Blade** options left blank.

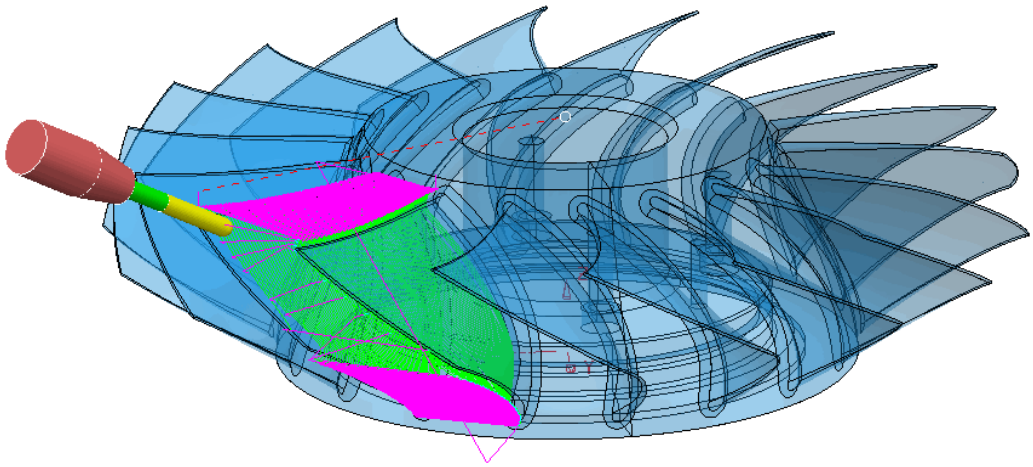


Copyright © Autodesk



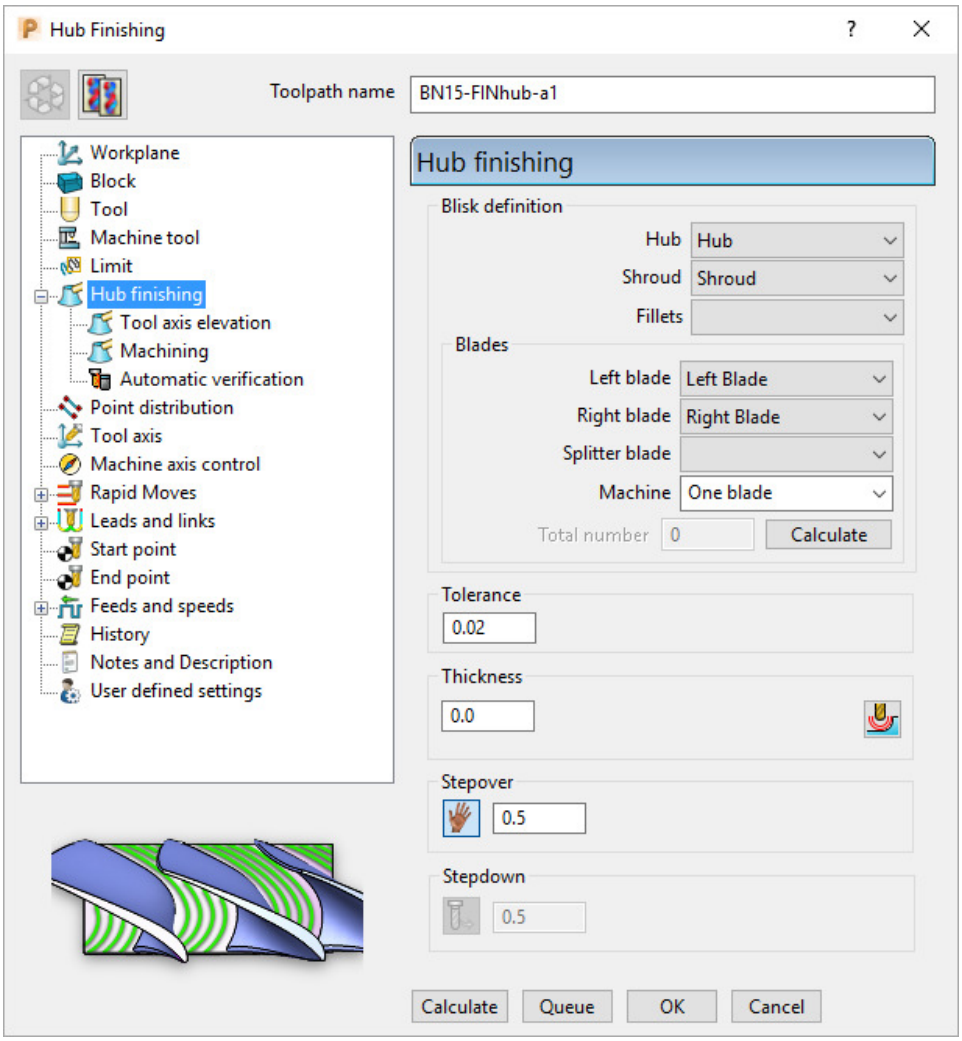


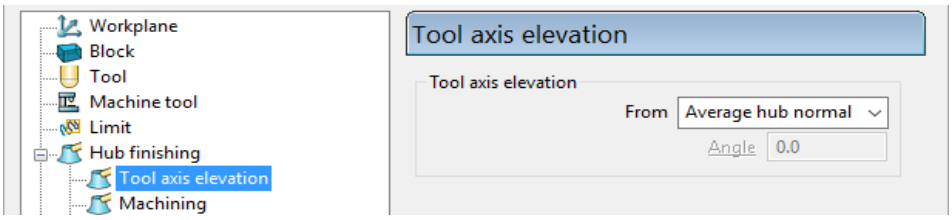
21 Select **Queue** in the dialog to commence background toolpath processing.

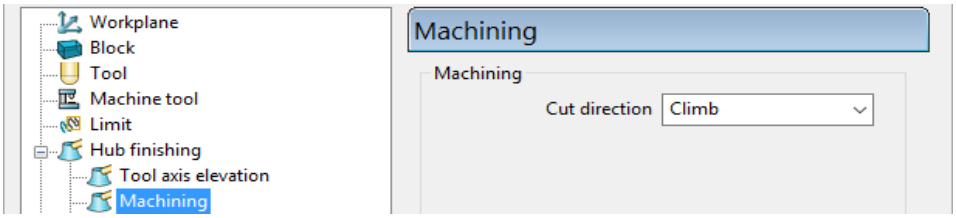


22 In the **Toolpath Strategies** dialog, select **Blisks** and from the four available options, select **Hub Finishing**.

23 Enter data exactly as shown below with the **Fillets** and **Splitter Blade** options left blank.

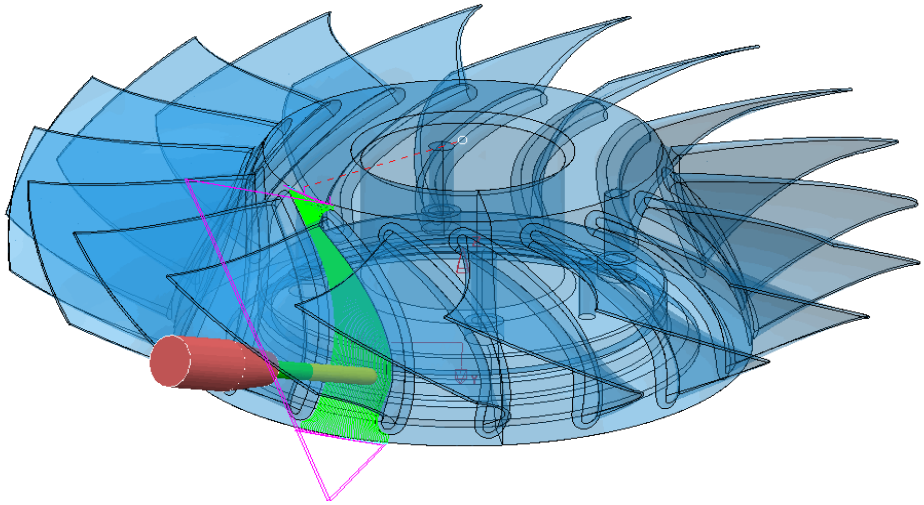






24 Select **Queue** in the form to commence background toolpath processing.

Copyright © Autodesk

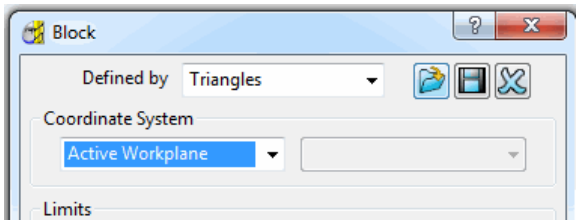


The next stage is to perform a full **ViewMILL** simulation of all 3 toolpaths.

Before running **ViewMILL** it is necessary to define the **Block** as a ***Triangle*** model of the pre-turned, Shroud for an accurate representation of the material prior to milling.

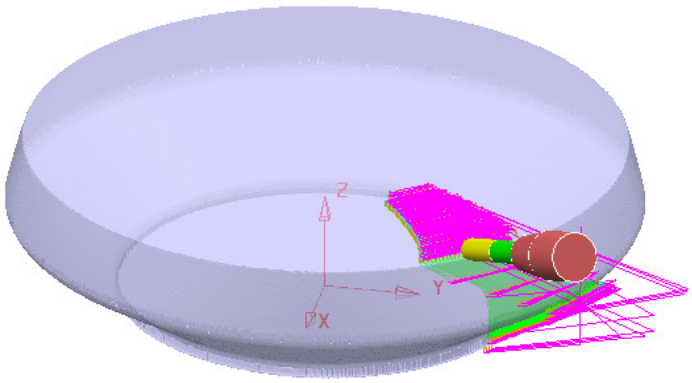
25 Activate and right mouse click on the toolpath **BN25-RGH-a1** and from the local menu select the option **Simulate from Start**.

26 Before starting the **Simulation**, open the **Block** dialog and select **Defined By** > **Triangles** followed by select the **Load block from file** icon.

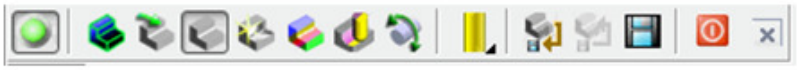


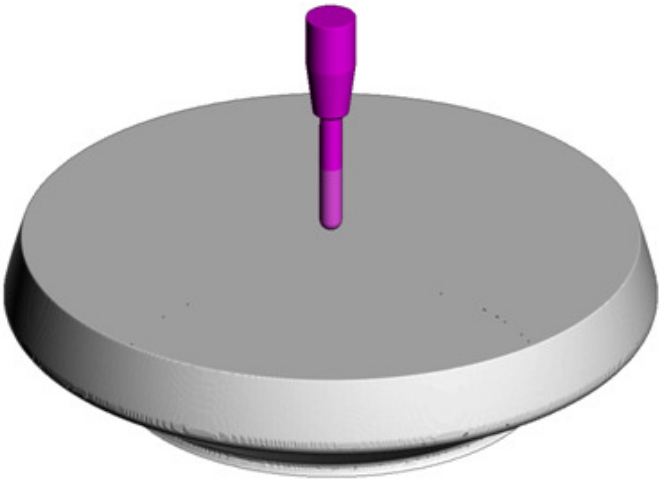
27 Browse and select the **Triangle** file:

...\PowerMill_Data\FiveAxis\Blisks\TurnedShroud.dmt

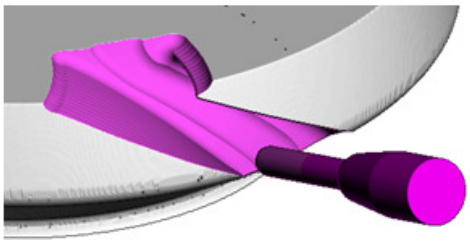


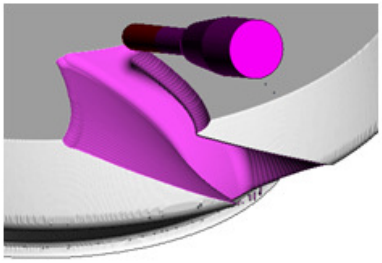
28 With the complex **Block** active, switch on a **ViewMILL** simulation.



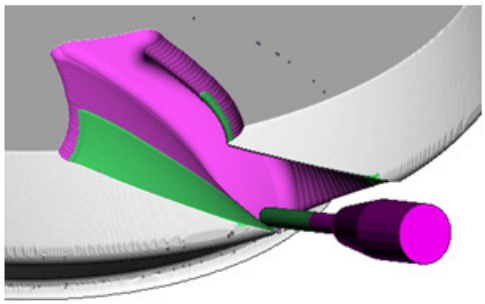


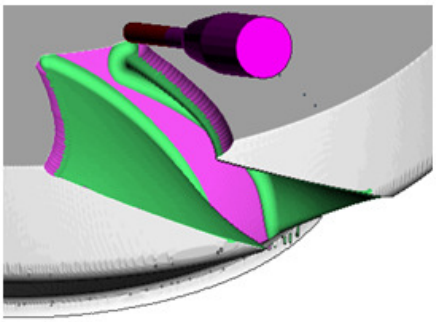
29 Select the **Play** button in the toolpath simulation toolbar and observe the machining process for the toolpath **BN25-RGH-a1**.





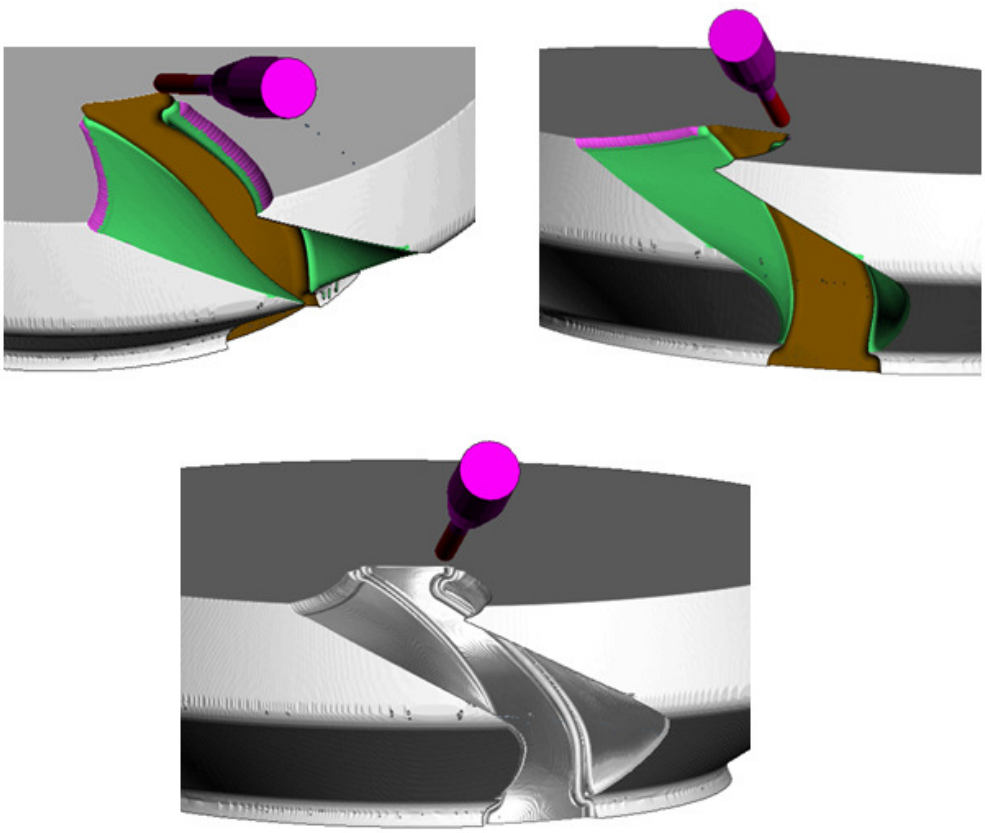
30 Continue the **ViewMILL** simulation on toolpath **BN15-fin-a1**.



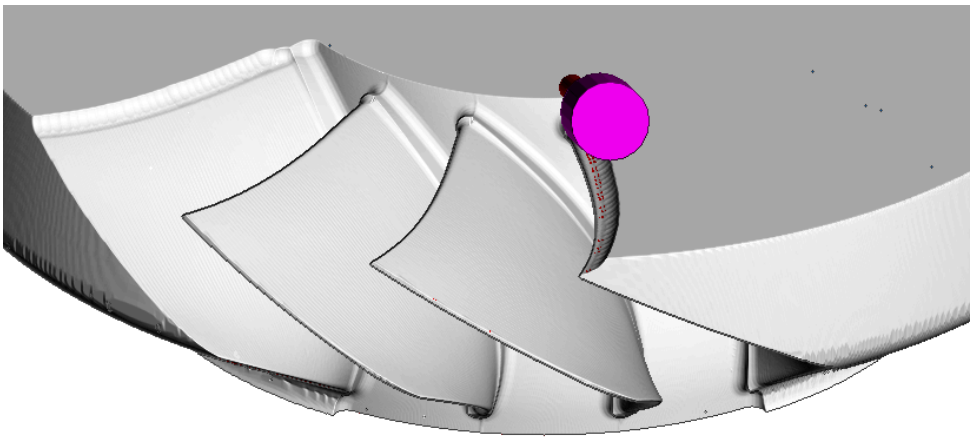


31 Continue the **ViewMILL** simulation on toolpath **BN15-finHUB-a1.**

Copyright © Autodesk



The final image (below) shows the result of a **ViewMILL** simulation including 17 copies of the **toolpaths** rotated by **20 Degree** increments around **Z**.



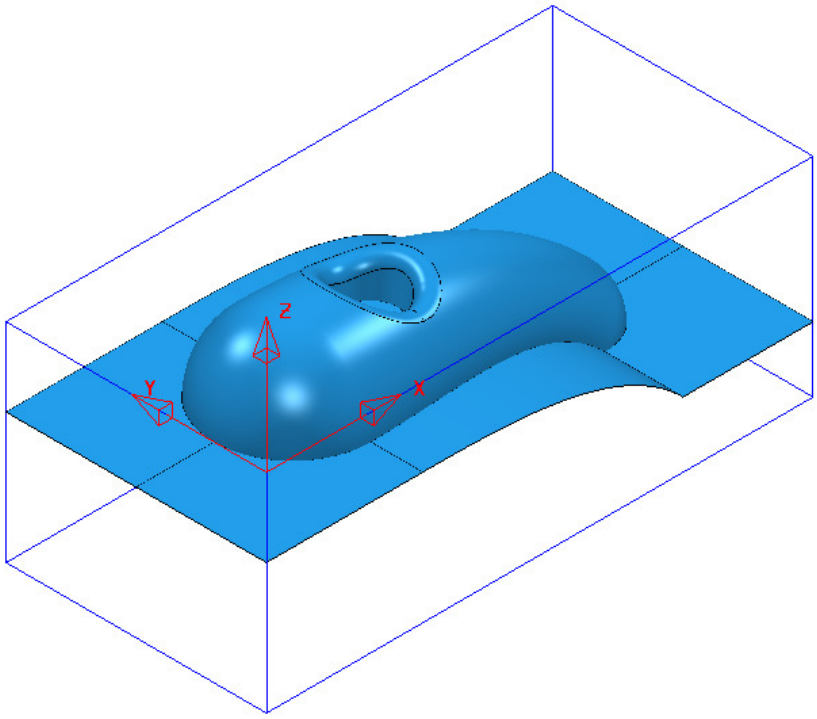
Copyright © Autodesk

17. Five Axis-Tutorial

1 Select **File** > **Delete All** and **Tools** > **Reset Forms**.

2 **Open** the (read-only) **Project**:-

….\PowerMill_Data\FiveAxis\Cowling\Cowling-5Axis-Start.dgk



3 **Save Project As**:

….\COURSEWORK\PowerMill_Projects\Cowling-5Axis

The **Project** currently contains 8, 3-Axis Toolpaths.

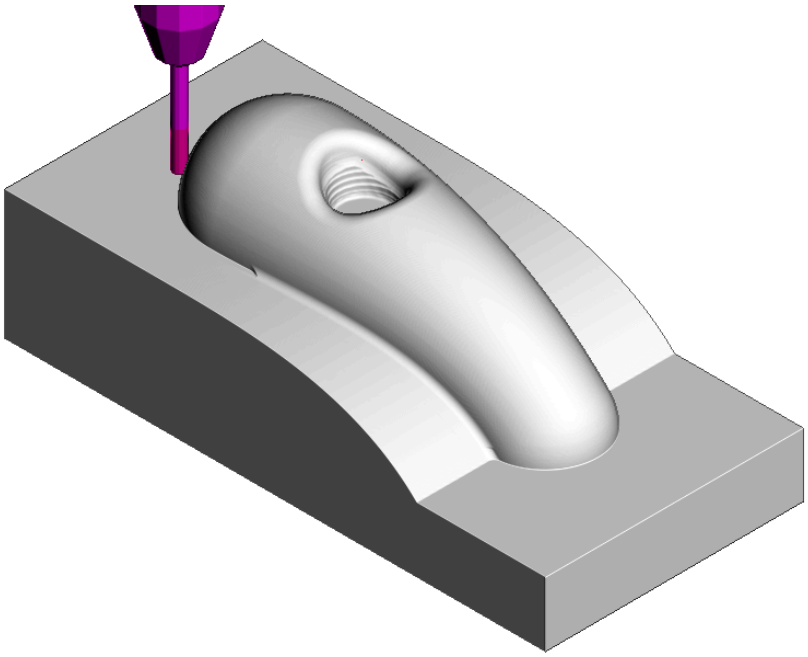
This has left remaining areas that are either more suited to, or can only be machined using 5-Axis strategies.

4 Select an **ISO2** view.

5 Activate the toolpath **D16TR6-RGH1_step** to restore the **Block** used in the strategy.

6 Perform a full **ViewMILL** simulation on the **NC Program** named **1**.

Copyright © Autodesk

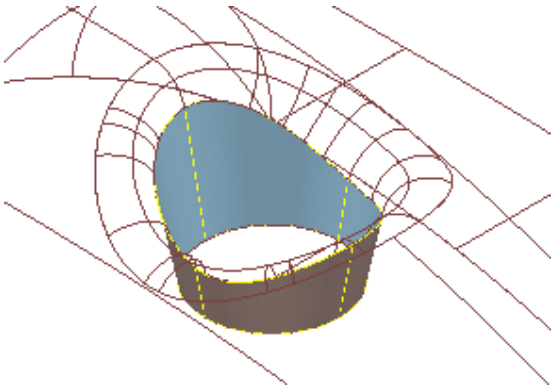


The remaining areas that require Five Axis machining include the central pocket (Swarf Machining), and the intersection between the main form and the 3D run-off (Embedded Pattern and Surface Finishing).

7 Select the **ViewMILL No-Image** display option to disconnect it from the simulation process.

Central Pocket

8 Activate the tool **EM12**.



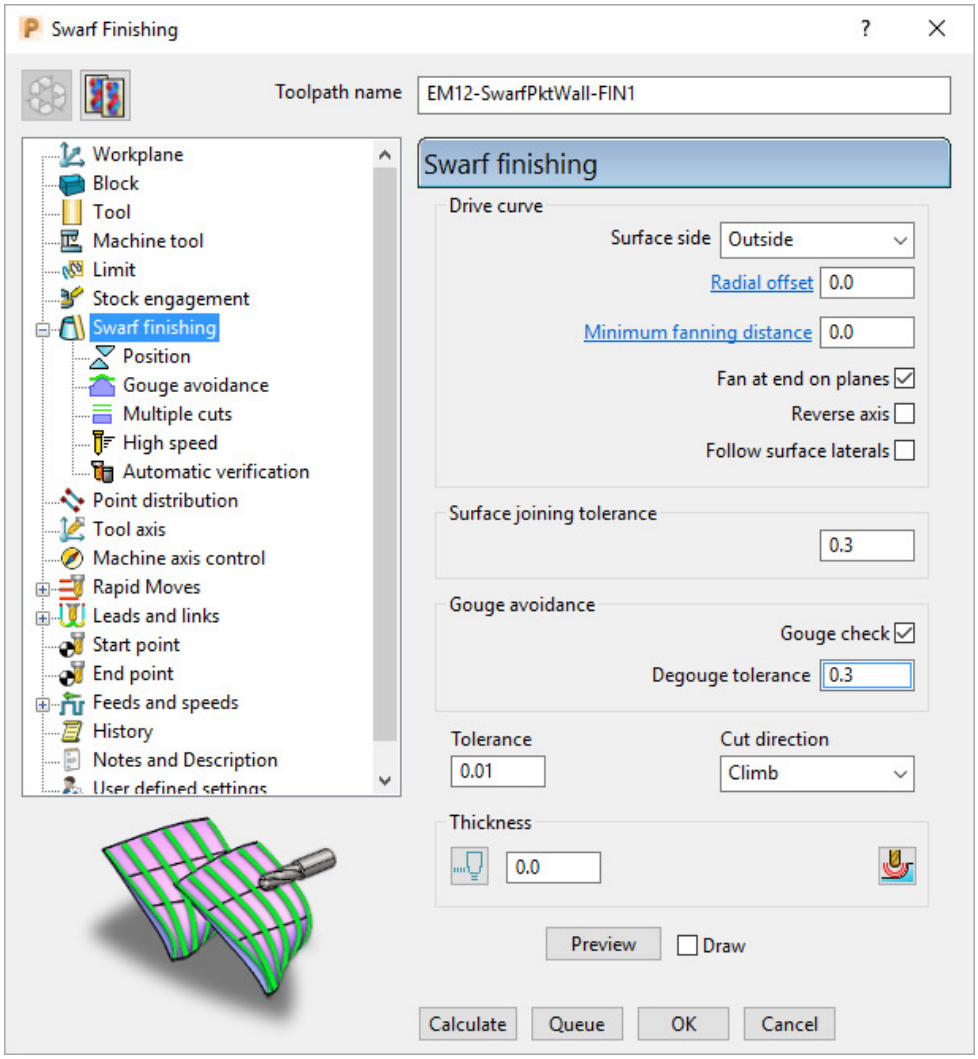
9 Select the pocket wall.

10 Select the **Toolpath Strategies** dialog select the **Finishing** tab.

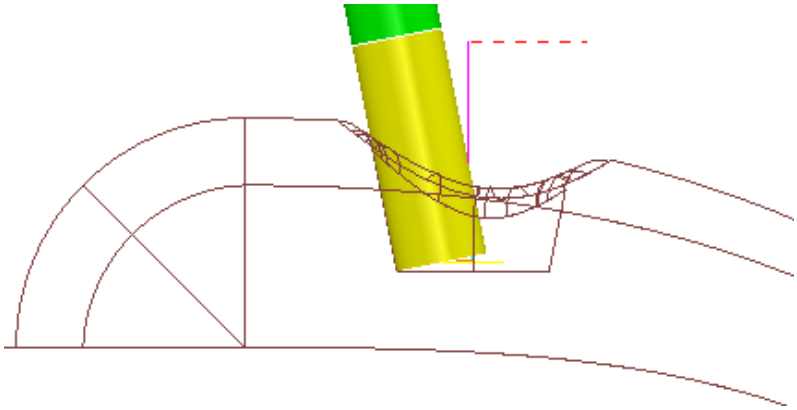
11 From the list of strategies select **Swarf Finishing** and then **OK**.

Copyright © Autodesk

12 In the **Swarf Finishing** dialog enter data exactly as shown below:



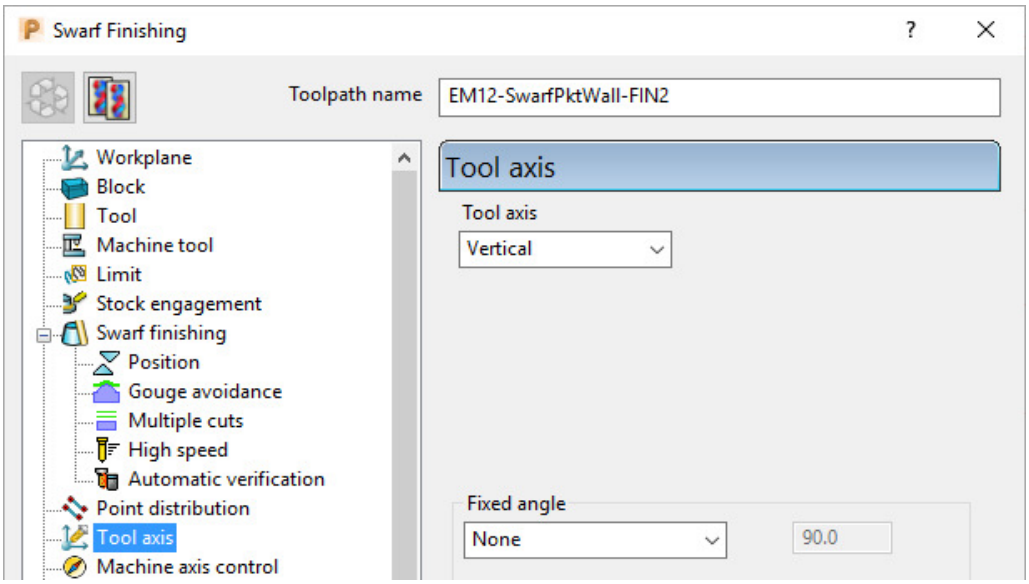
13 For both the **Lead in** and **Lead out** input **Surface normal arc** with **Angle 180** and **Radius 2**, then **Calculate** the toolpath.

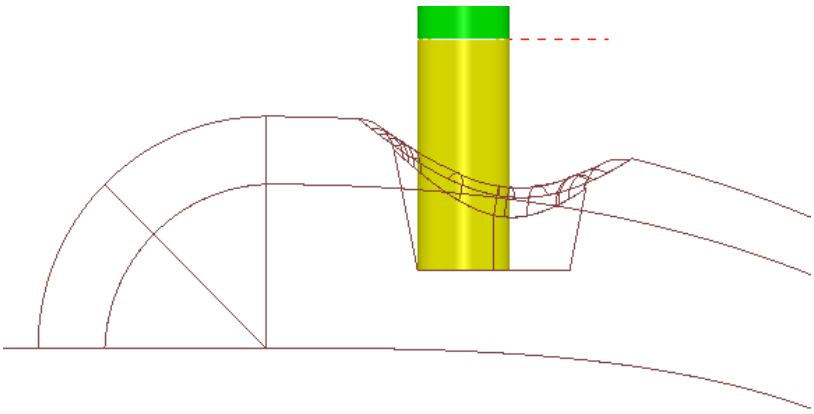


The **Swarf Finishing** strategy is created as ***Five Axis*** by default (The **Tool axis** option is set to **Automatic**). As shown in the above image this results in a wedge of material remaining on the flat base of the pocket. This can be removed with a second **Swarf Finishing** toolpath but this time with the **Tool Axis** set to **Vertical**.

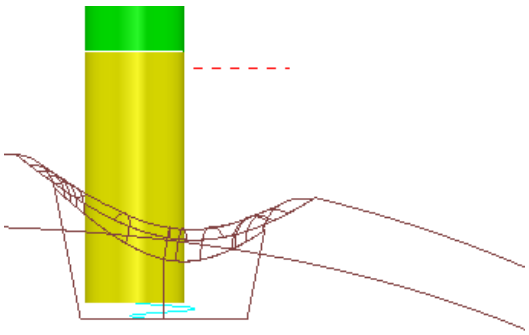
Copyright © Autodesk

14 Make a copy of the **Swarf Finishing** strategy named **EM12-SwarfPktBase-FIN2** but this time set the **Tool Axis** to **Vertical** before selecting **Calculate**.





15 Change the ***1st choice***, **Lead in** move onto this toolpath to a circular **Ramp** with **Angle 4**, **Circle Dia (TDU) 0.65** and **Ramp Height 2**.



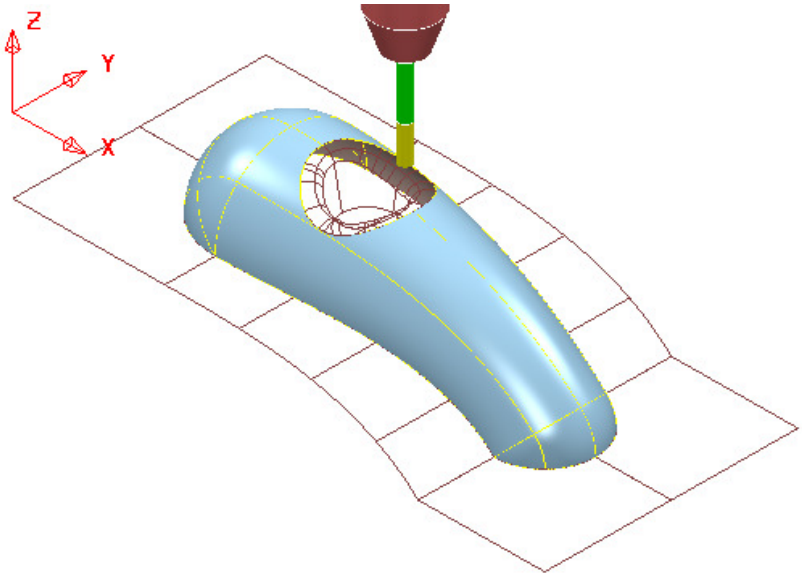
The Pocket is now fully machined.

16 **Save** (but do not close) the **Project**.

Copyright © Autodesk

Main Form – Base Intersection

17 Activate the **EM5** tool.

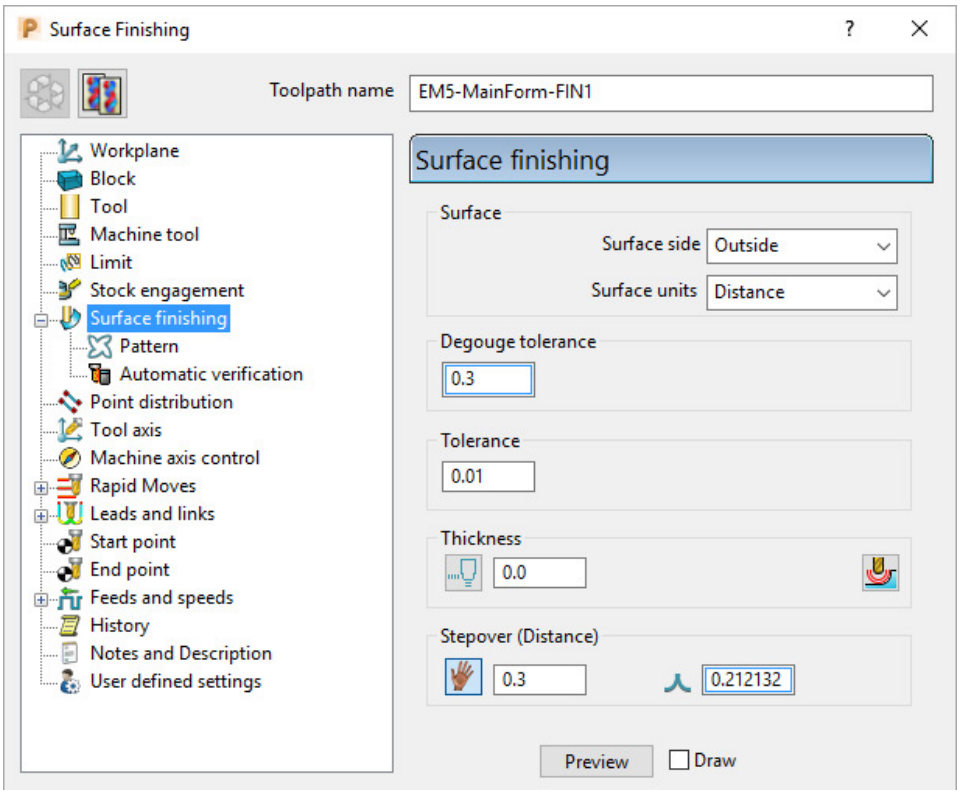


18 Select the main form surface (as shown in the above image).

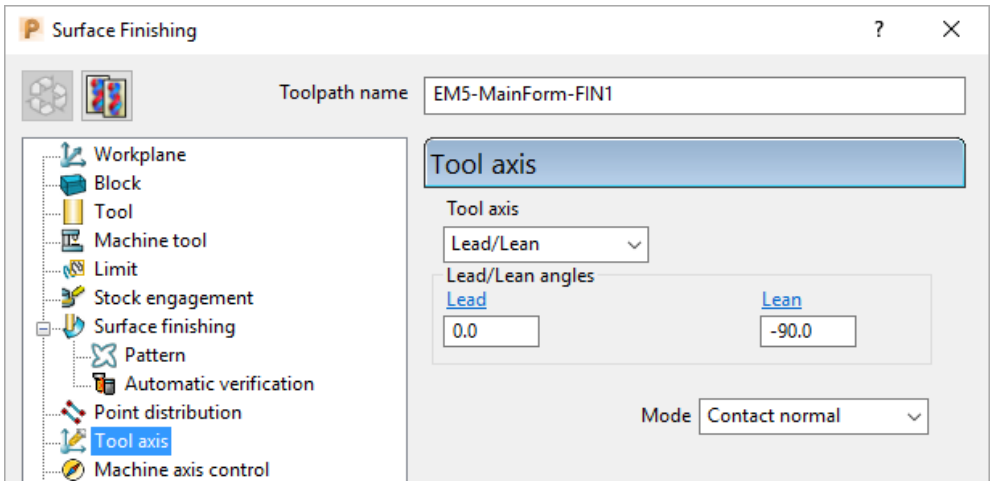
19 Select the **Toolpath Strategies** dialog select the **Finishing** tab.

20 From the list of strategies select **Surface Finishing** and then **OK**.

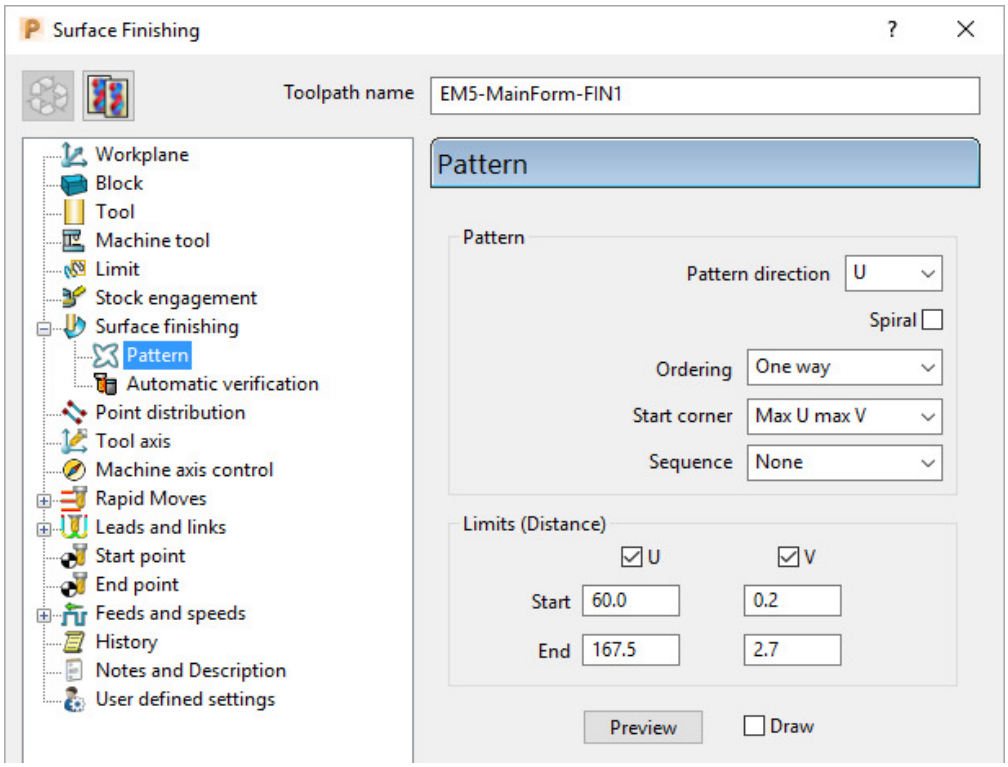
21 In the **Surface Finishing** page enter data exactly as shown in the image below.



22 Select the **Tool Axis** page and again enter data exactly as shown in the image below.



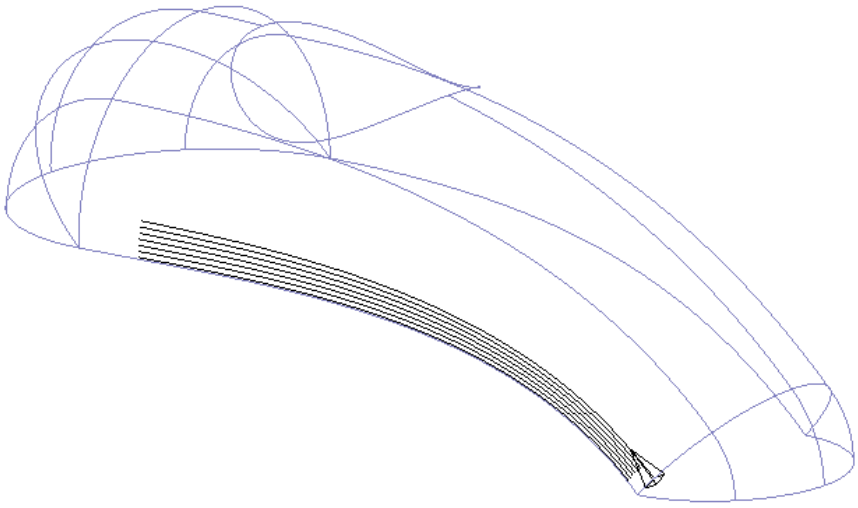
23 Select the **Pattern** page and again enter data exactly as shown in the image below.



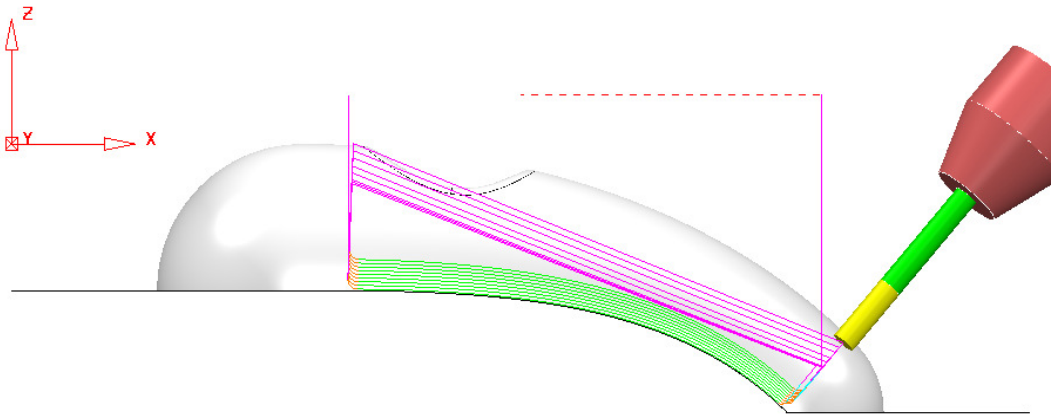
The values shown in the above page were finally arrived at after a trial and error process. The actual values are unit distance along and across the surface, curve network (U V).

24 Select **Preview** to visualise the defined pattern on the surface.

Copyright © Autodesk



25 Select **Calculate** to process the toolpath.

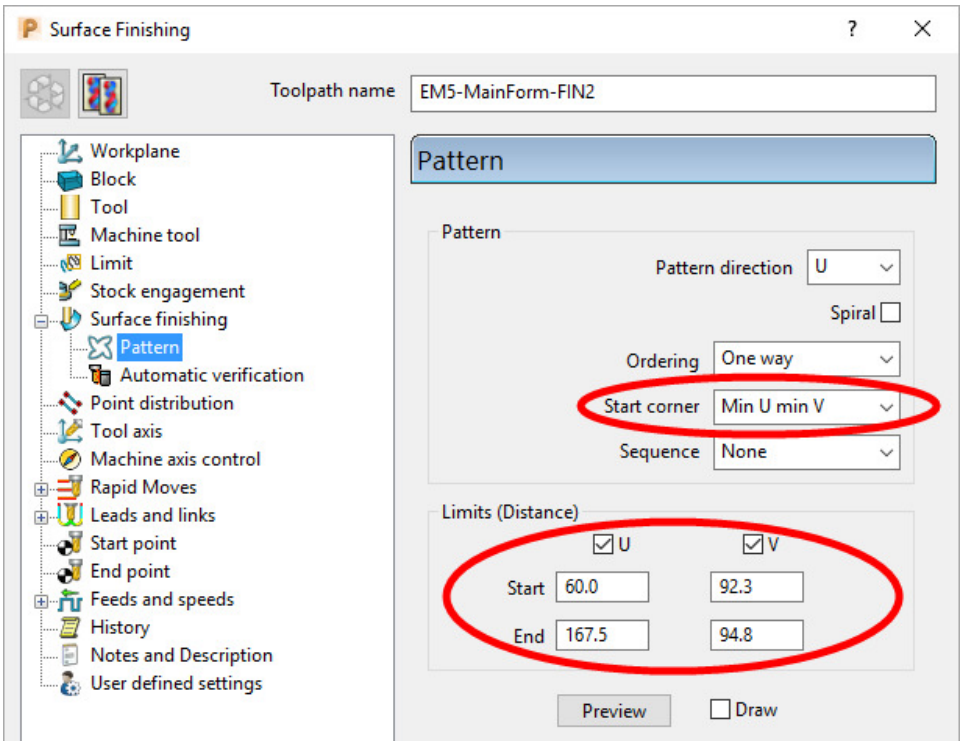


To help visualisation the above images show the preview with a larger area and stepover than specified in the instructions.

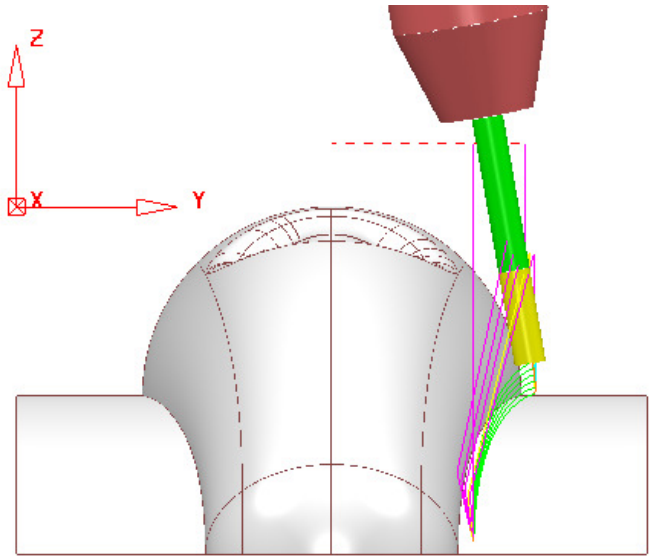
26 From the **main** toolbar select the **Leads and Links** dialog.

27 For both **Lead in** and **Lead out**, apply ***1st choice*** as **Vertical arc** – **Angle 90** – **radius 2** and ***2nd choice*** as **Vertical arc** – **Angle 90** – **radius 0.5**.

28 Make a copy of the **Surface Finishing** toolpath and rename it as **EM5-MainForm-FIN2**.



29 Modify the original settings in the **Pattern** page and select **Calculate** to perform the same strategy on the other side of the main form.



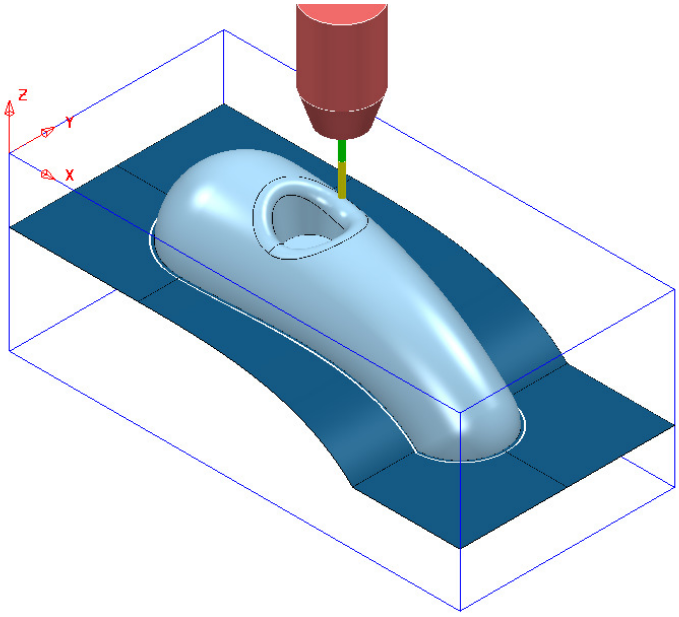
*The last 2 Surface Finishing strategies deliberately stop just short of the run-off surfaces. A more accurate, final pass will be created normal to the run-off surfaces using a separate **Embedded Pattern Finishing**.*

30 **Save** (but do not close) the **Project**.

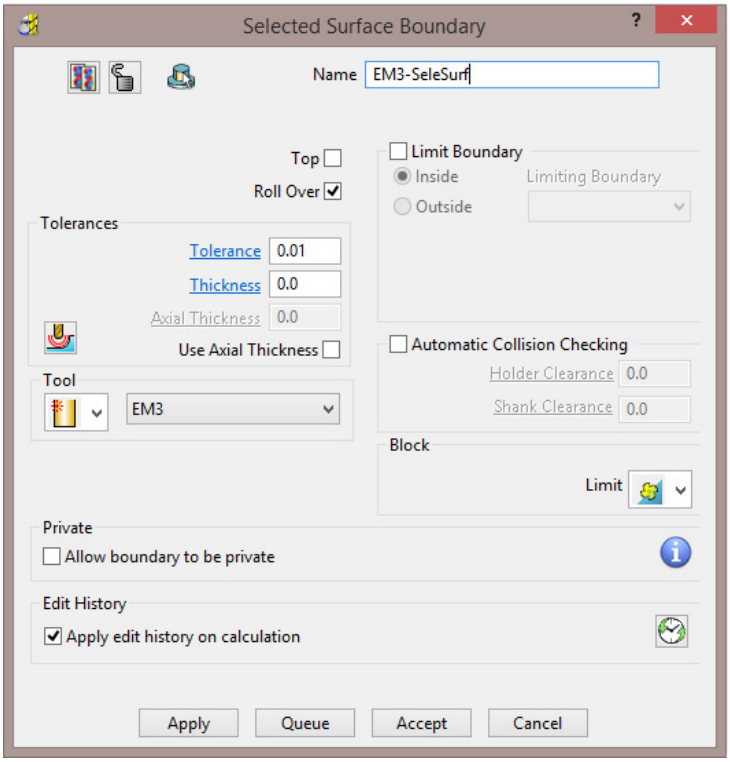
Copyright © Autodesk

Final Pass around Main Form - Base Intersection

31 **Activate** the **EM3** tool.



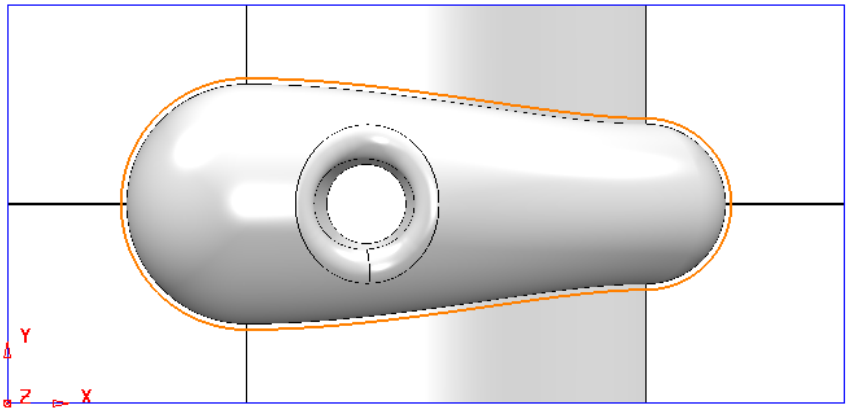
32 Select the main form and pocket **surfaces** (as shown in the above image).



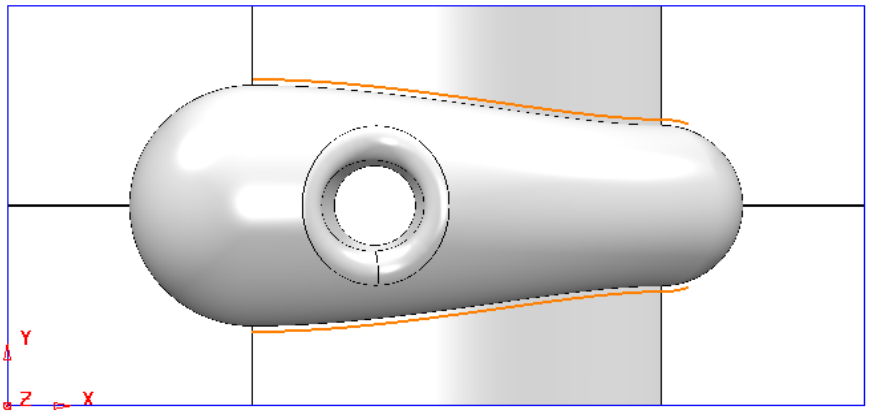
33 Create a **Selected Surface Boundary** (with **Roll Over** ticked).

34 Create an empty **Pattern** and insert the above **Boundary** segment into it.

35 Delete the **Boundary**.

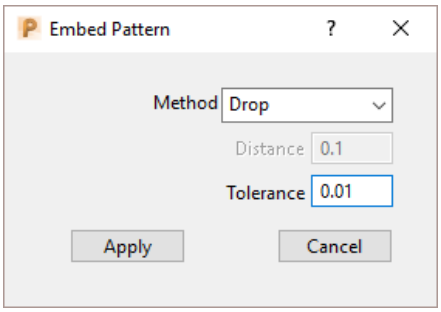


36 Use the **Curve Editor** to trim the **Pattern** into 2 segments (**Merge** if required) suitable for an **Embedded Pattern Finishing** strategy.



37 **Reverse** the **Pattern** segments as required to produce a **Climb Milling** direction (Material to the right of the tool feed direction).

38 Right mouse click on the new **Pattern** and from the local menu select **Edit** – **Embed**.



39 In the above dialog select **Method** as **Drop** with a **Tolerance** of **0.01** before selecting **Apply**.

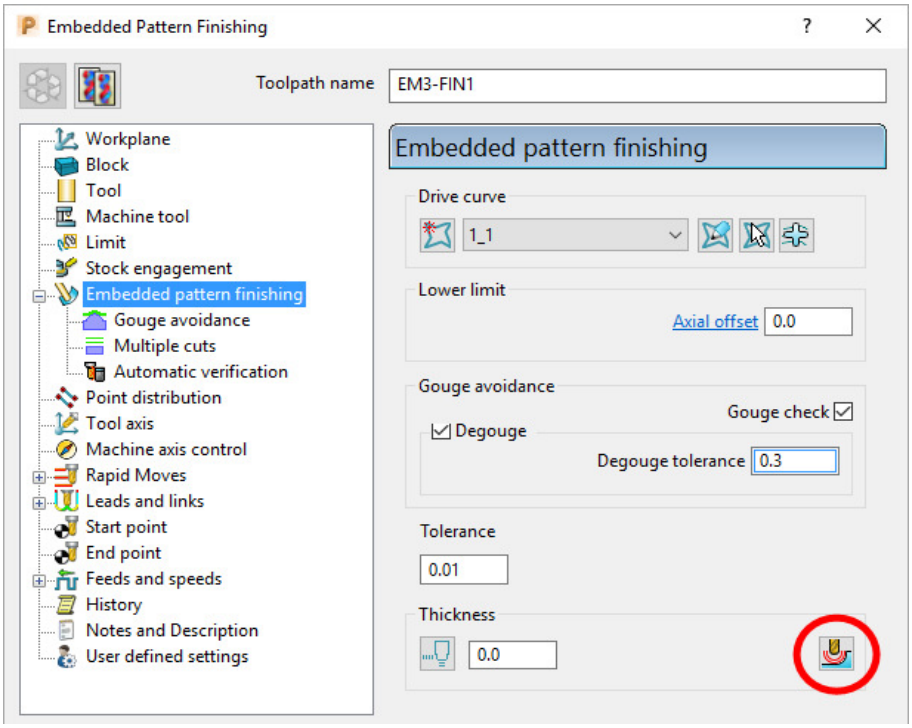
A copy of the original ***Pattern* 1** is projected down Z onto the model as an ***Embedded Pattern* 1_1** (contains the surface normal).

40 Select the **Toolpath Strategies** dialog select the **Finishing** tab.

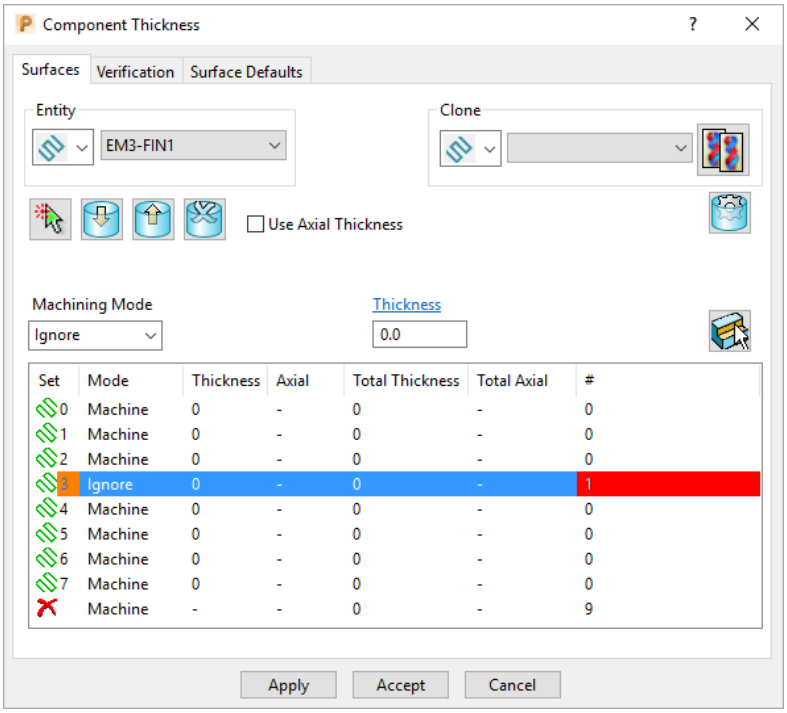
41 From the list of strategies select **Embedded Pattern Finishing** and then **OK**.

Copyright © Autodesk

42 In the **Embedded Pattern Finishing** page enter data exactly as shown in the image below.



43 Open the **Component Thickness** dialog and acquire the **Main Form surface** to one of the rows.

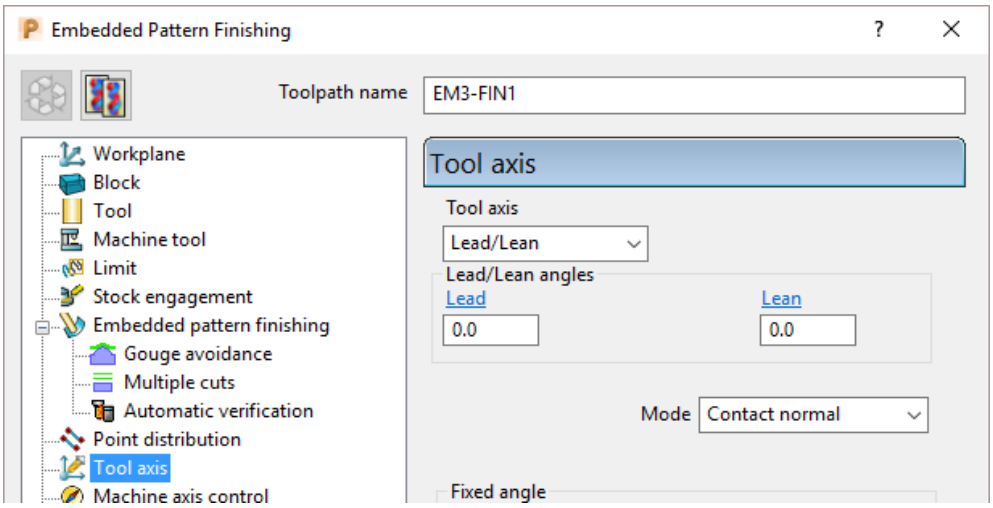


44 Set the selected row to **Machining Mode Ignore** and **Apply**.

The main form **surface** is set to be ignored as it would otherwise result in the toolpath to fragment. This in turn will result in the tool very marginally gouging the main form **surface**. If checked both visually and using the gouge checking options the actual results will be negligible.

Copyright © Autodesk

45 Back in the main **Embedded Pattern Finishing** dialog, open the **Tool Axis** page and select **Lead/Lean** both with a **0** angle.

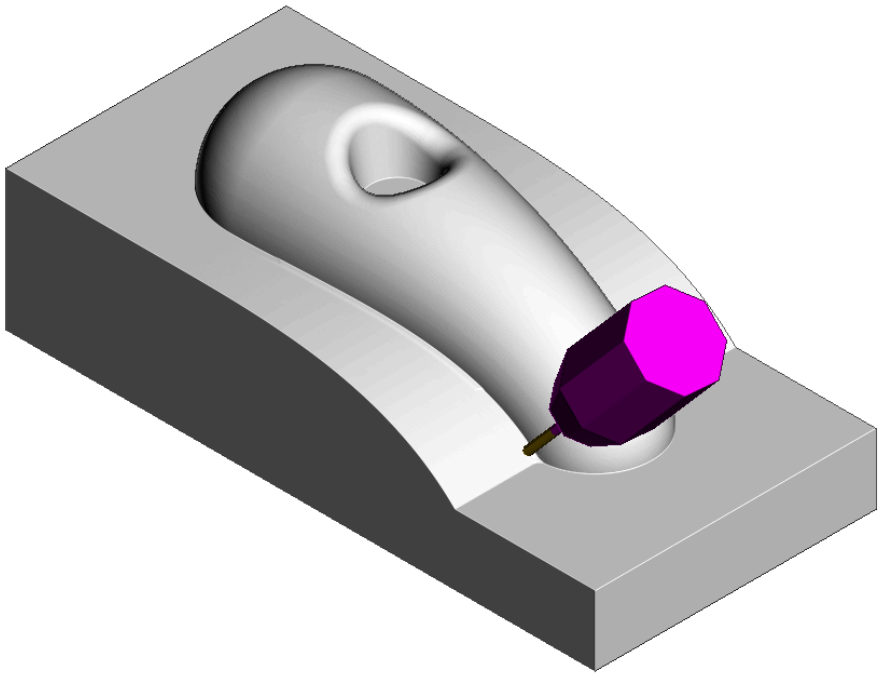


46 Set all **Link** moves (**1st choice**/**2nd choice**/**Default**) to **Incremental**.

*This is to prevent the toolpath from linking straight through the main cowling surface (**Component Thickness** set to **Ignore**).*

47 **Calculate** the **Embedded Pattern** toolpath.

48 Run a full **ViewMILL simulation** (Final **EM3-FIN1** toolpath show below).



49 **Save** the **Project**.

Copyright © Autodesk