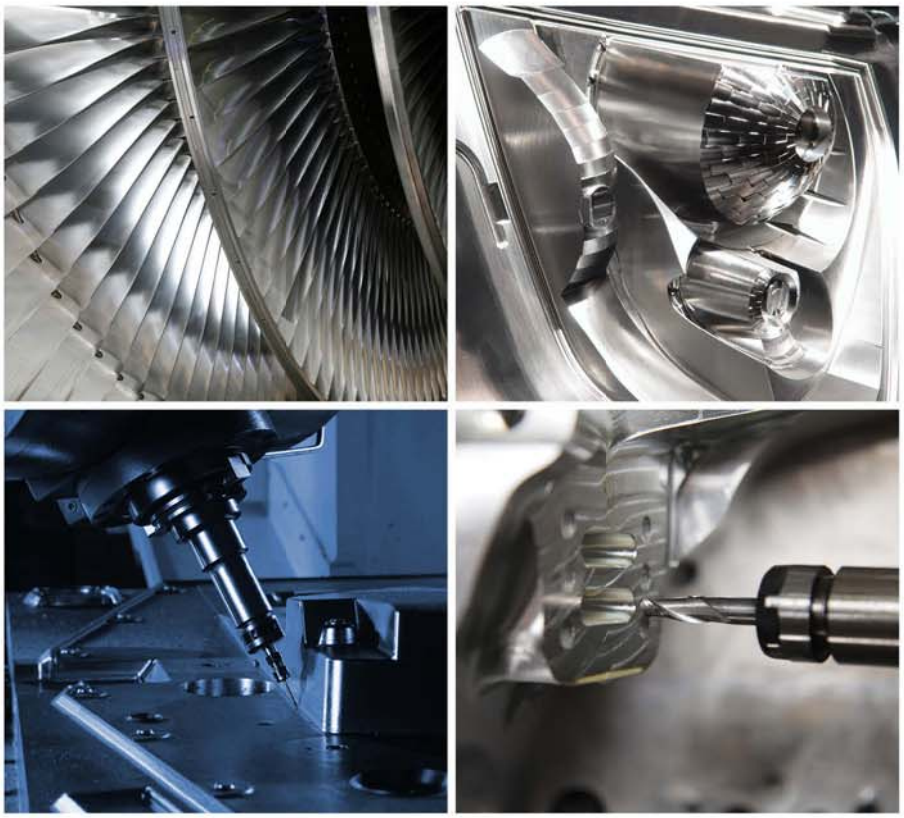
Delcam Advanced Manufacturing Solutions

Powering your productivity

PowerMILL

5-axis



Training Course

Important Notice

This document is supplied as part of a Delcam Training Course. It is not intended to be distance-learning material: rather as an aid for Tutors when presenting material to course delegates and as a subsequent aid memoir to those delegates.

Delcam does not accept responsibility for any personal belongings / valuables whilst on the premises. Delegates are advised to keep their belongings on their person at all times.

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

PowerMILL 2016 Five Axis Contents

Chapters Page Number

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, before performing standard **X Y Z** transitions. This is achieved either by manual adjustment or as part of the CNC control.

The most common method of creating a **3 + 2 Axis** is by using **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.

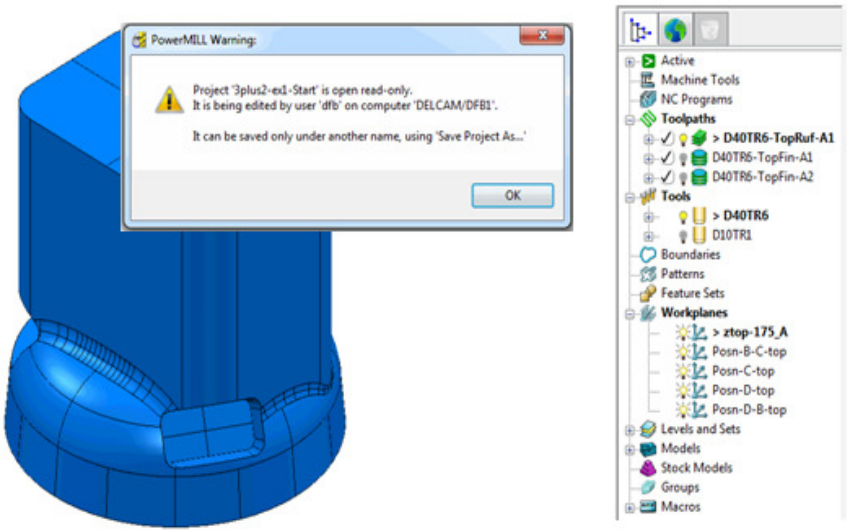
b Access to deep sidewalls with standard length tooling.

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:

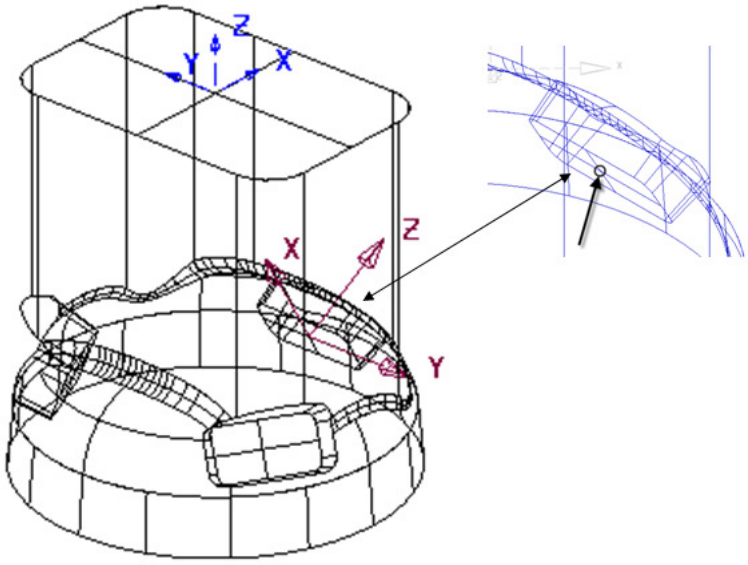
...\COURSEWORK\PowerMILL-Projects\3Plus2-ex1

*A **3-Axis Roughing** and two **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 **Z175** above the **Transform**. This **workplane** will be used as the main machining datum. The other 4 **workplanes** will be used later to control positional movements between multi-axis machining strategies.*

3 Select an **ISO 1** view and consider the machining options.

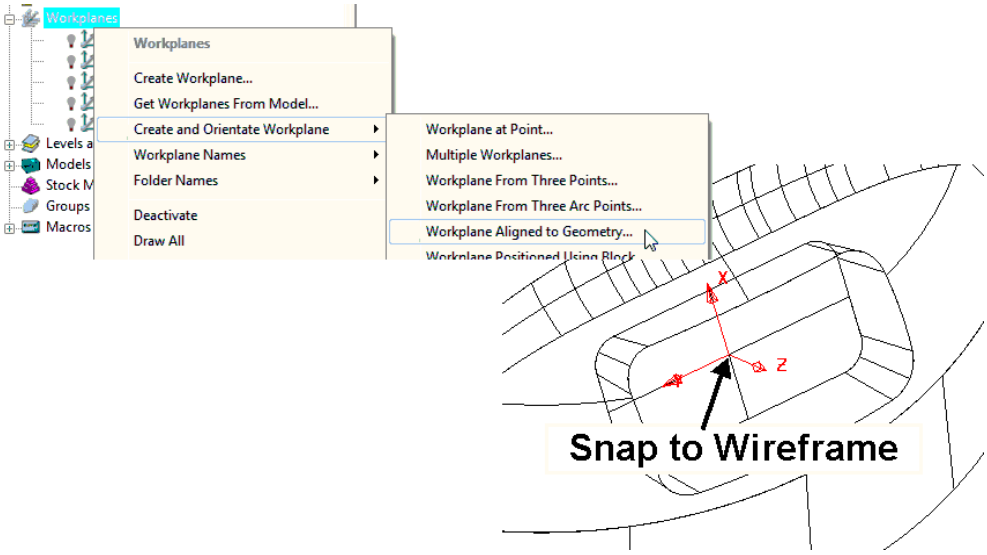
*Note the relatively high sides of the component and the orientation of the three recesses making it impossible to machine as **3 Axis** (with the tooling aligned to the **Z-Axis**).*



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

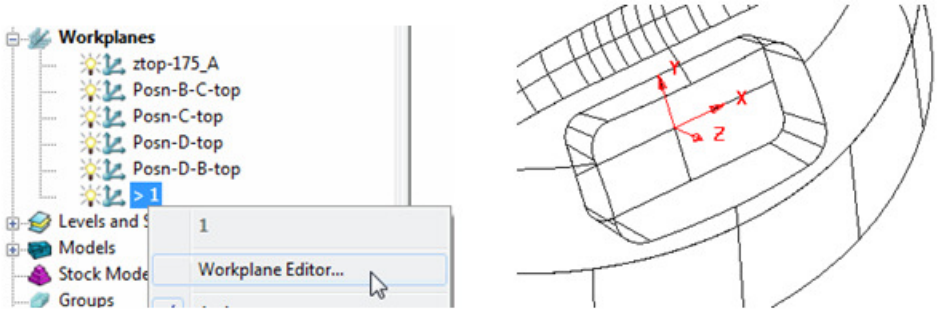
4 From the **PowerMILL *explorer*** select **Workplane** > **Create and Orientate Workplane** > **Workplane Aligned to Geometry**.

5 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***).

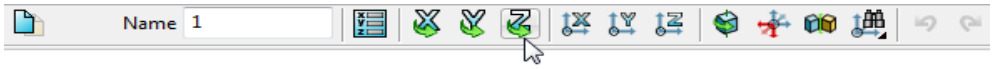


6 **Activate** the new **Workplane** 1.

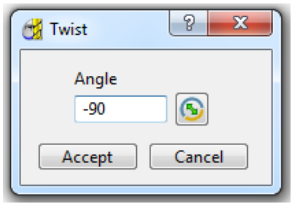
The **workplane** is automatically aligned to the wireframe with the **Z Axis** normal to the **surface**. It still requires further editing as it is advisable for the **X-Axis** to align anticlockwise around the component in reference to the global coordinates.

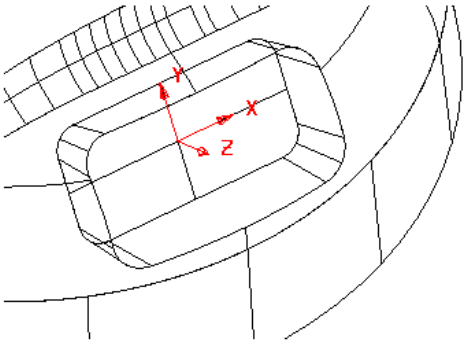


7 From the **PowerMILL *explorer***, right-click on **Workplane** 1 and from the local menu, select the ***workplane editor*** option to open the following toolbar.



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





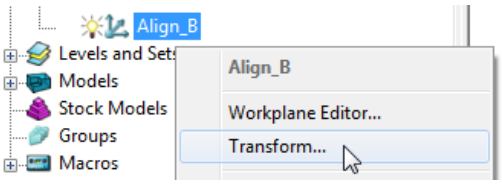
The **X-Axis** of the **workplane** should be pointing anticlockwise relative to the **Transform** (Global Datum) as viewed from the top of the component (If not already the case).

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

10 Rename **Workplane 1** as **Align_B**.

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

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

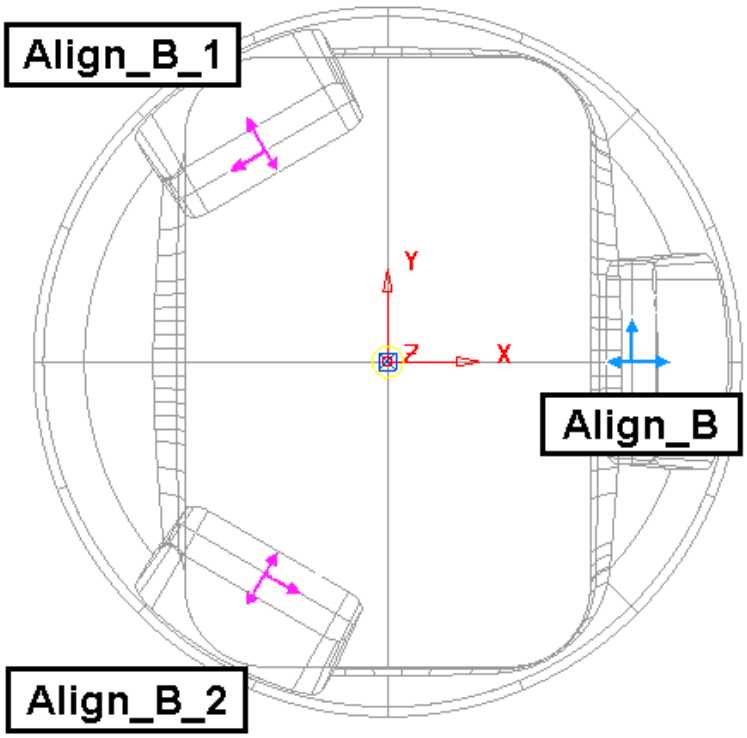


13 Select the **Rotate workplane** options (shown arrowed below).

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

15 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).

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



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

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

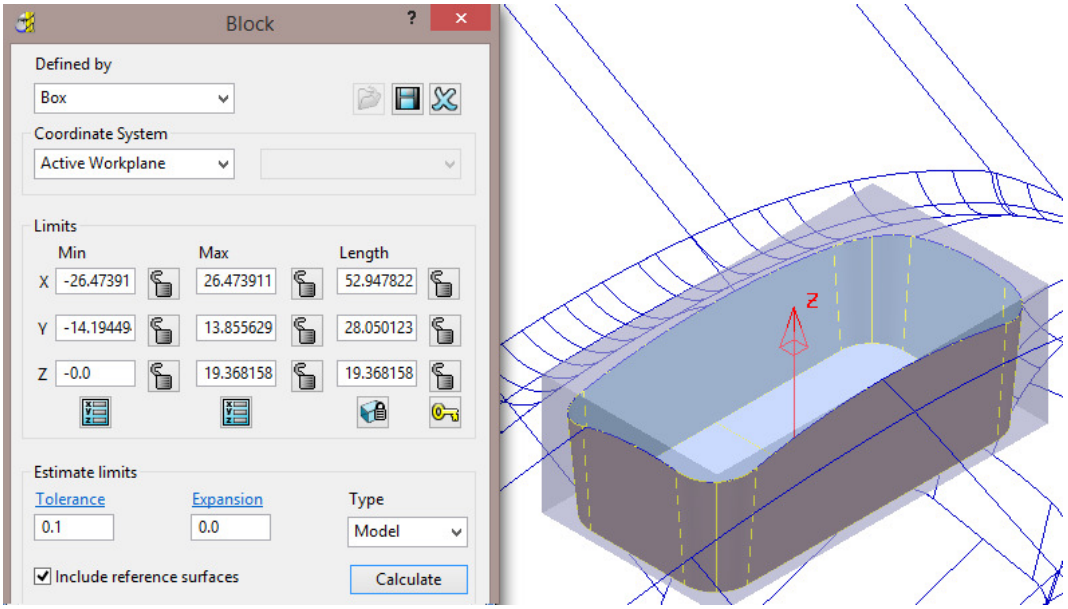
The component is now ready to have the **3 Plus 2** machining strategies created, relative to the three new workplane alignments. (**Align_B**, **Align_C**, and **Align_D**).

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.

1 **Activate** the **workplane *Align_B***.

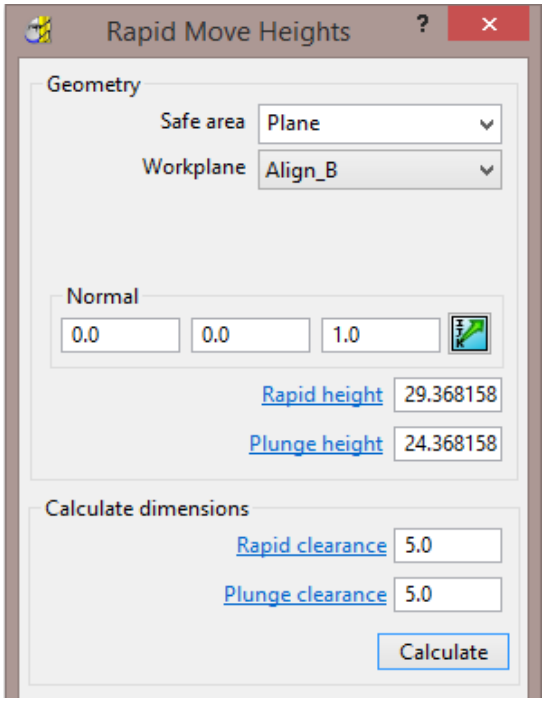
2 Select the 2 **surfaces** that define the pocket around **workplane *Align_B***.

3 Create a **Block** (***Defined by box***) to **Type Model** (***Model***).



4 Ensure that the ***pocket surfaces*** are still selected so that the **Rapid Move Heights** calculation ignores the de-selected **surfaces**.

5 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 correct coordinate system.



*Input the **Workplane** (**Align_B**) to achieve the correct **Rapid Move Heights** alignment.*

***Rapid Move Heights** are calculated based on the **Model** or **Block**, whichever has the **maximum Z height**.*

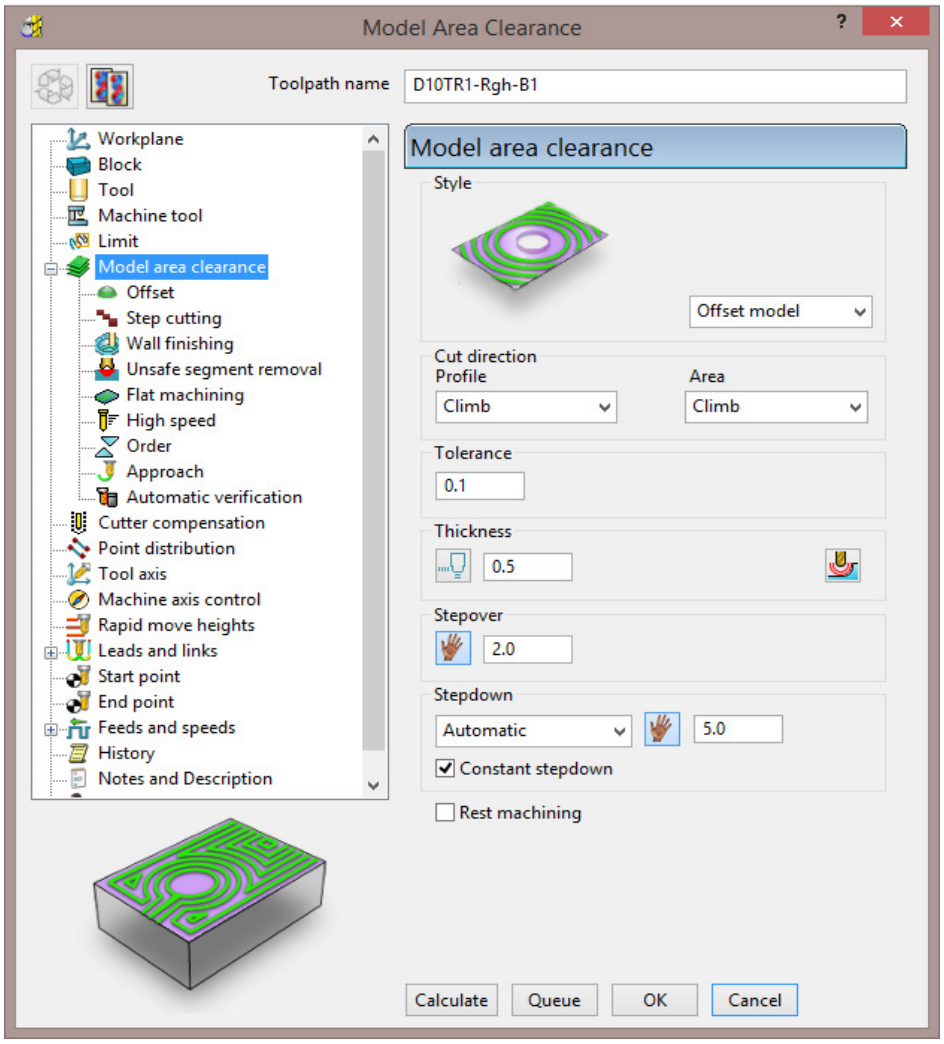
*If a group of **surfaces** are selected, the remainder will not be included in the calculation.*

6 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**).

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

8 Set **Lead In** to **Ramp** with **Ramp Options** as **Zig Angle 4**, **Follow Circle**, and **Diameter** (TDU) **0.6**

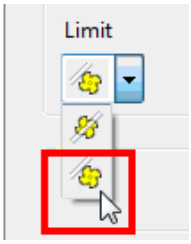
9 In the **Strategy Selector** dialog in **3D Area Clearance** select **Model Area Clearance** and fill in the options in the pages, exactly as shown below:



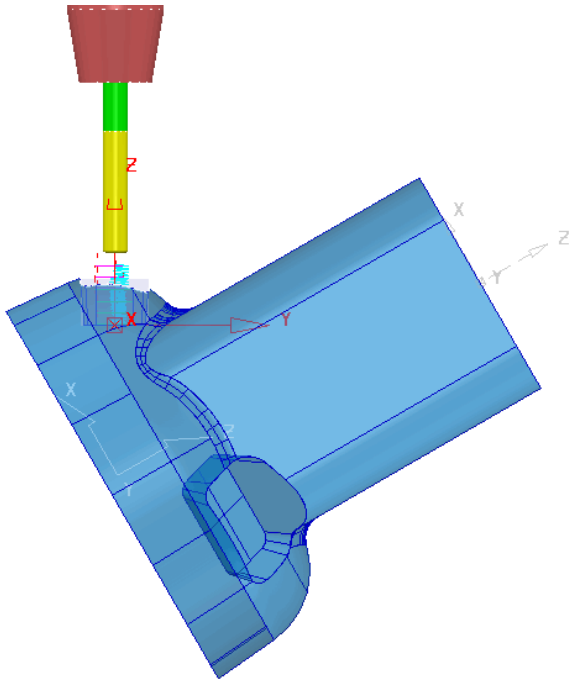
Toolpath name: **D10TR1-Rgh-B1**

Style: **Offset model**.

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



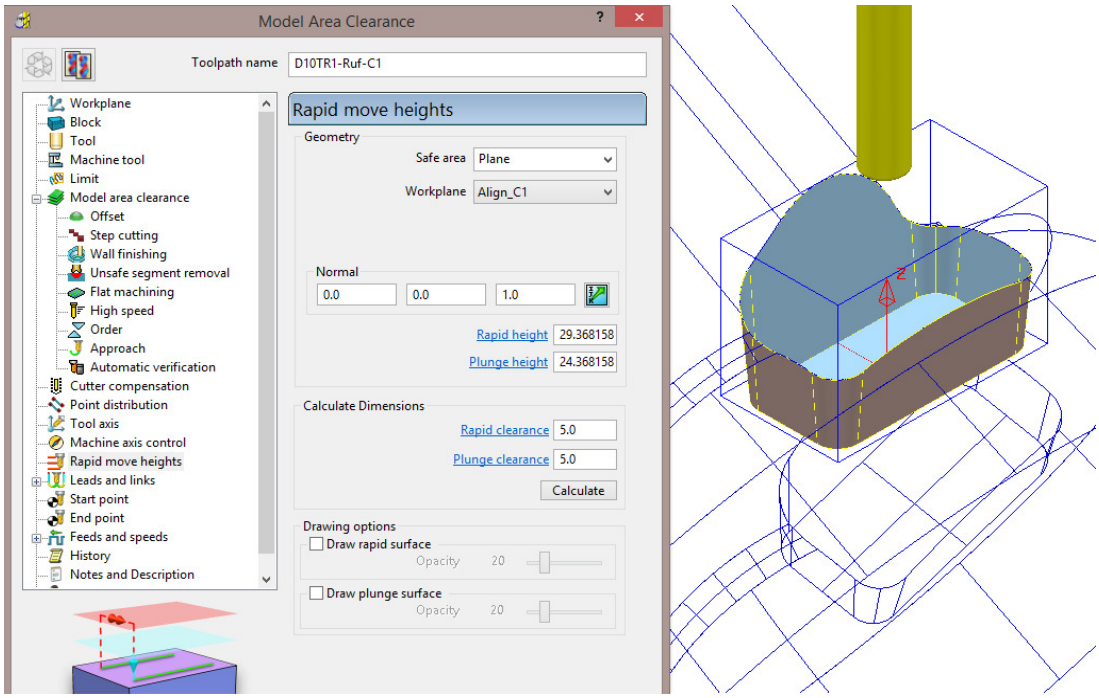
11 Select **Calculate** to create the **toolpath** and then **Cancel** the dialog.



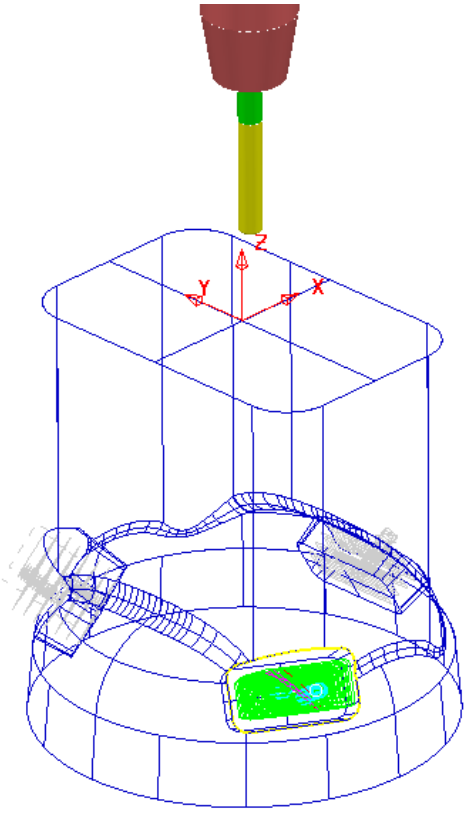
The toolpath plunge moves are aligned to the ***Active* workplane's**, Z Axis.

12 Use **Copy** to create cloned strategies on 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 the correct **Workplane** selected in the dialog (Remember to include the correct **Workplane** in the dialog).



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

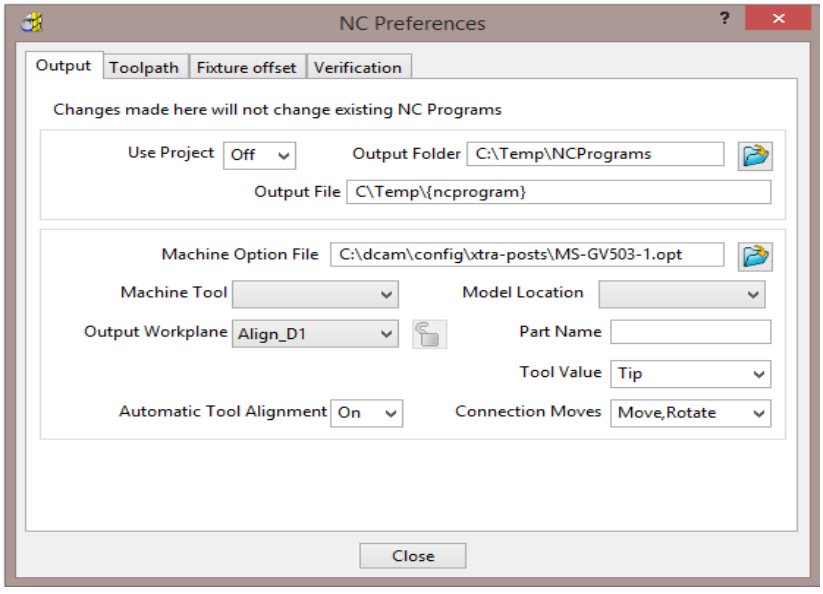


After the creation of **toolpaths** for **3 + 2 Axis** valid ncdata can only be output using a compatible, post-processor. For programs containing multi-alignment toolpaths the **NC Programs** output options create the ncdata from one datum (In this case the **Workplane** - **ztop175_A**). This option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

13 In the **PowerMILL *explorer***, right-click on **NC Programs** and select **Preferences**.

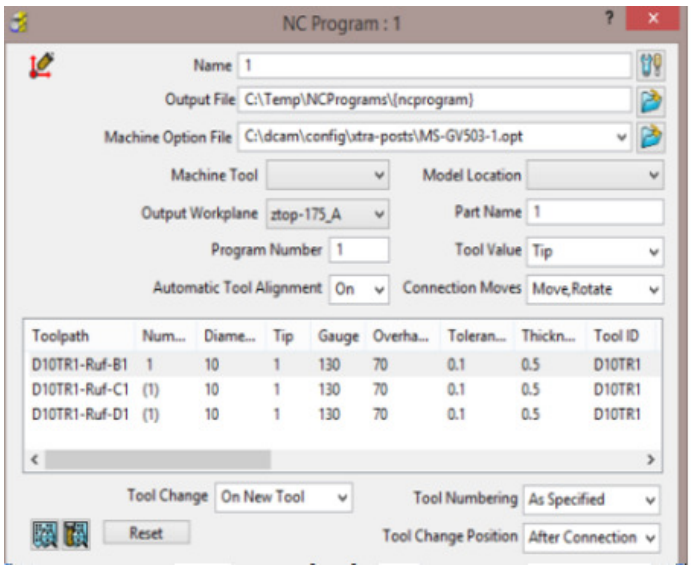
14 From the **Machine Option File** select **MS-GV503-1.opt** as the controller.

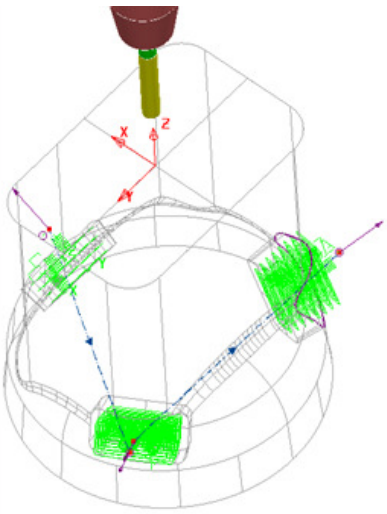
15 For the **Output Workplane** select **ztop-175_A**.



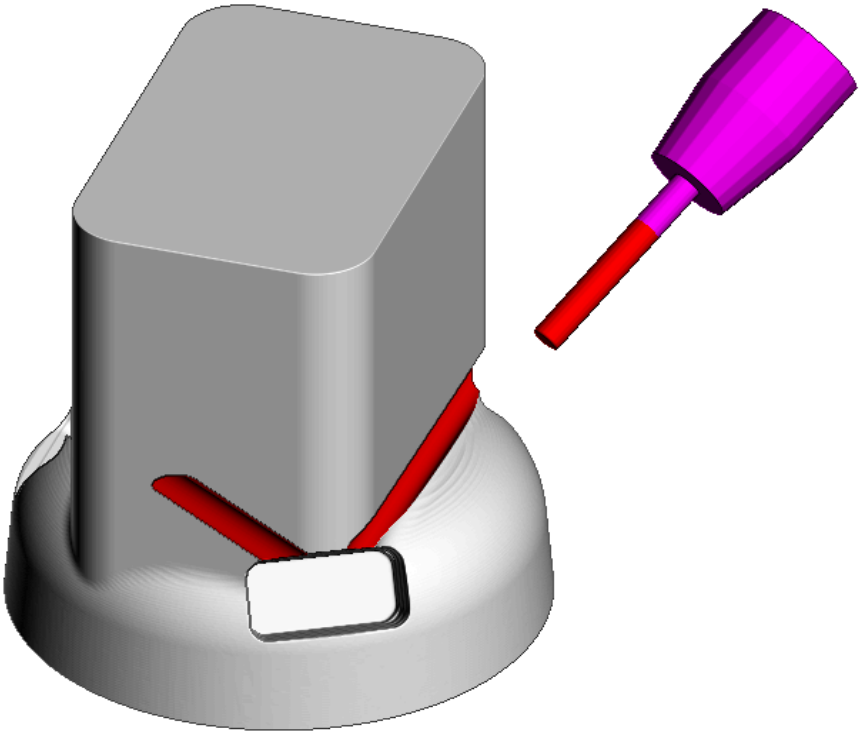
16 **Close** the dialog.

17 In the **PowerMILL *explorer***, **NCPrograms**, select **Create NC Program** and add to it the toolpaths, **D10TR1-Rgh-B1**, **D10TR1-Rgh-C1**, and **D10TR1-Rgh-D1**.





*The link moves between the three individual '**3Plus2**' toolpaths are displayed in the **NC Program** and in this case are passing straight through the component!*



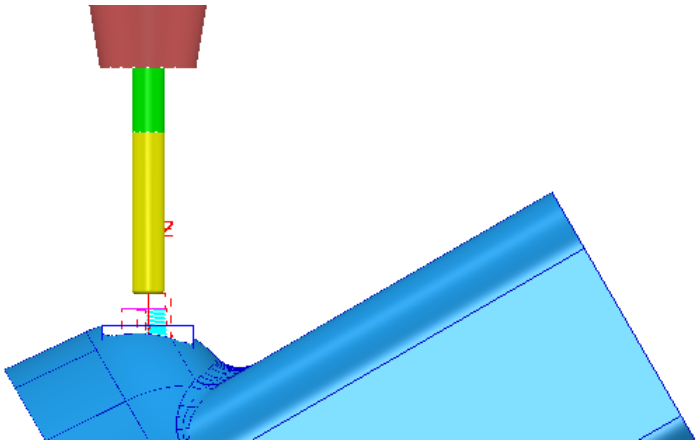
Automatic gouge protection only exists within a toolpath (Between the **Start and End Point**). After inserting the toolpaths into an **NC Program**, it is essential to independently control the **Positional Moves** outside the toolpaths to eliminate any collisions with the component and fixtures.

Link moves controlled by Absolute Start and End points

This method of creating safe link moves between separate 3 Plus 2 toolpaths is the most universal and safest method for all types of application and post-processor. The start and end point for each toolpath is defined as an absolute coordinate position above the maximum Z height of the component.

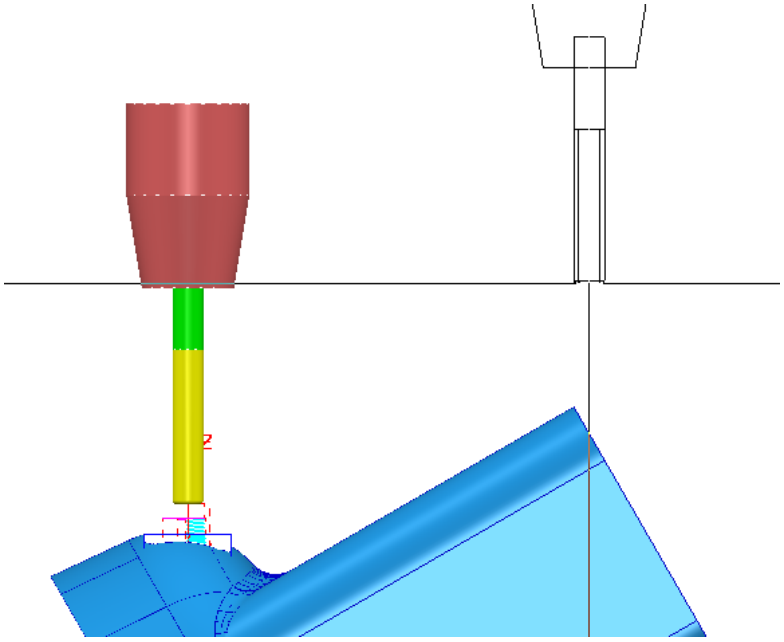
18 **Activate** the toolpath **D10TR1-Rgh-B1**.

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

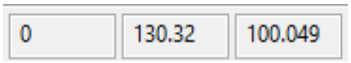


20 Switch on both **Draw** - **Cursor** – **Crosshair** (Ctrl H) and **Tool** (Ctrl T).

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

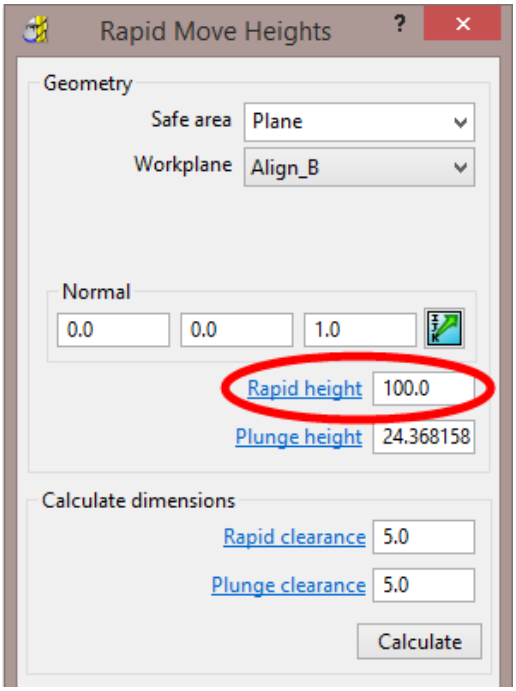


The X, Y, and Z coordinate values are displayed in the ***information*** **toolbar** located below the graphics area.



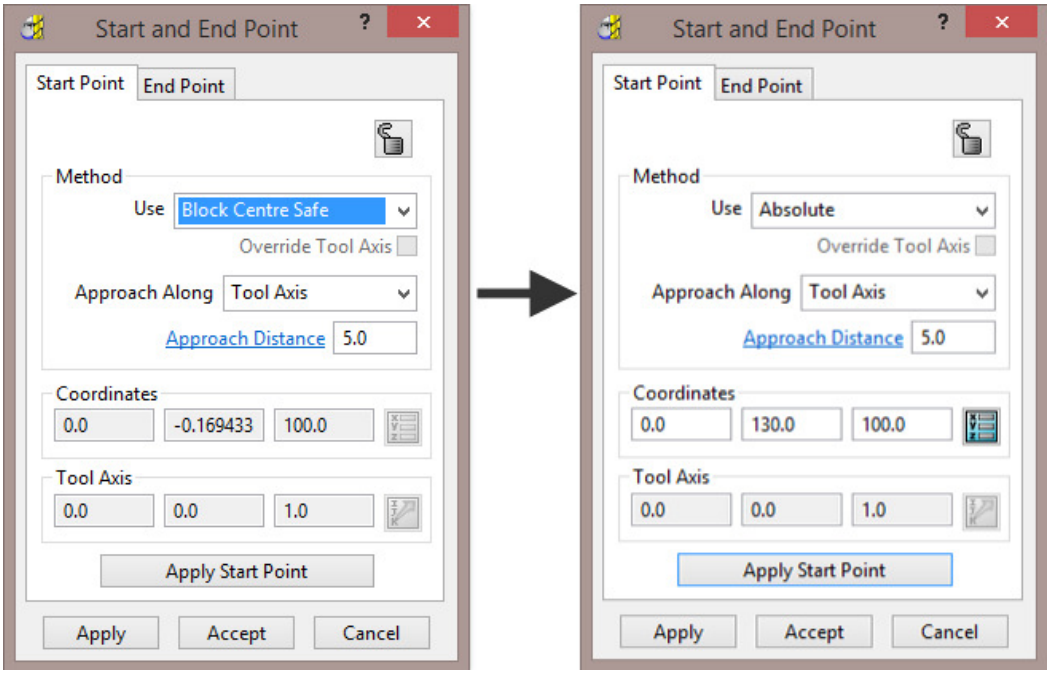
22 Make a note of the individual Y and Z values as integers **130** and **100**.

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



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

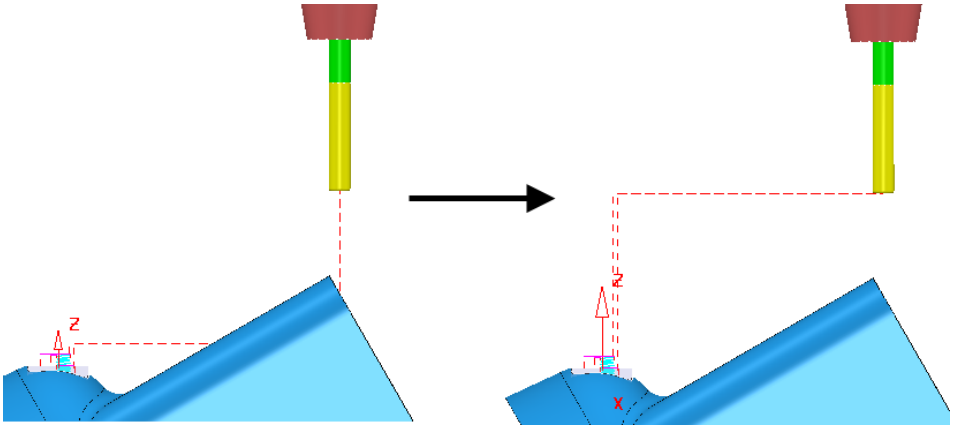
24 Open the **Start and End Point** dialog



25 For the ***Method*** select ***Use*** - **Absolute** and input the new Y value (**130**).

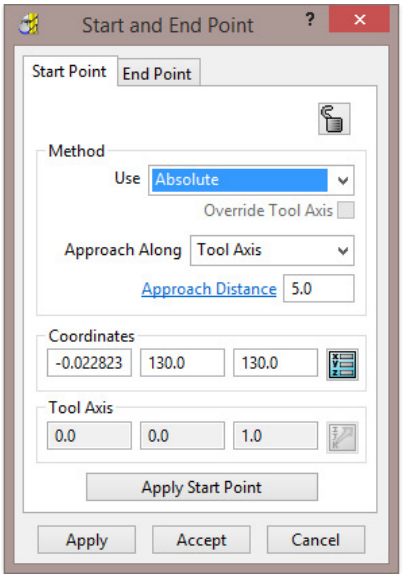
26 Select **Apply Start Point** (to update the toolpath).

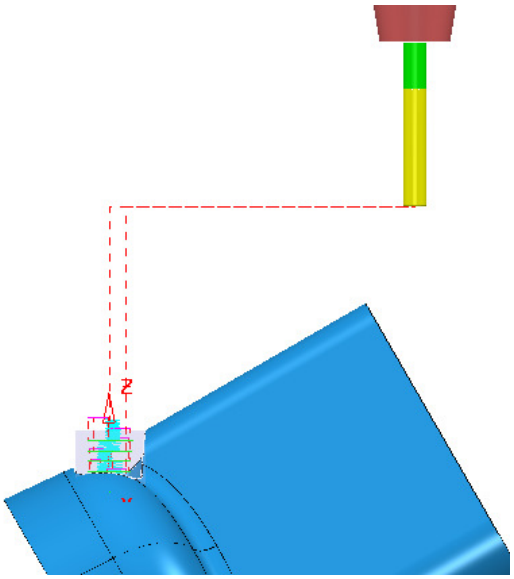
27 Select the **End Point** tab, repeat stage **25** above, and then select **Apply End Point** (to update the toolpath).

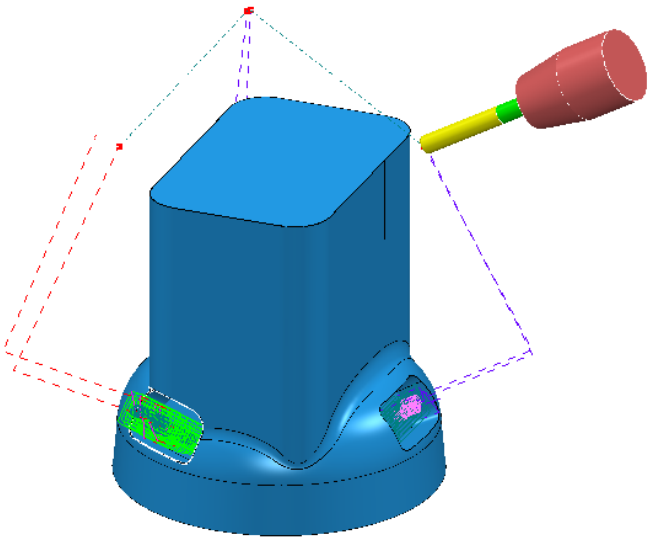


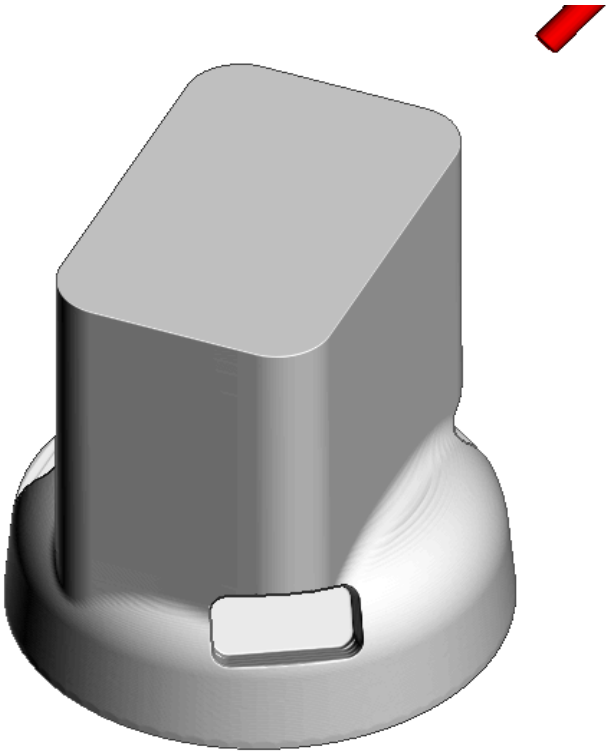
28 Repeat stages **19** to **27** on toolpaths **D10TR1-Rgh-C1** and **D10TR1-Rgh-D1** this time using the values **130** and **130**.

Note the new values (eg 130 130) will be different due to these pockets being in the corners of the main section.









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

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.

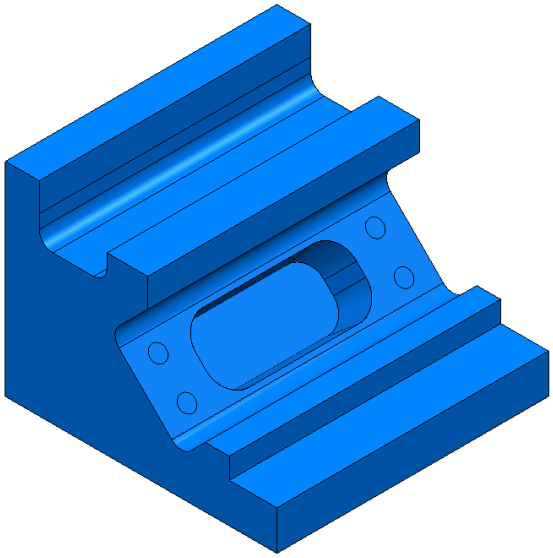
1 Select **File - Delete All** and **Tools** - **Reset** forms.

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

...\PowerMILL_Data\five_axis\AnglePad\StockModelRest.dgk

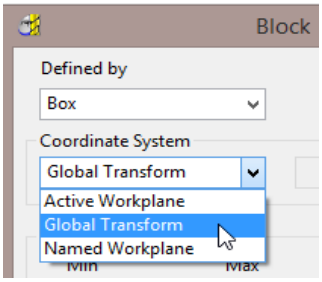
3 **Save Project As**:

...\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 Open the **Block** dialog and **Calculate** 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**

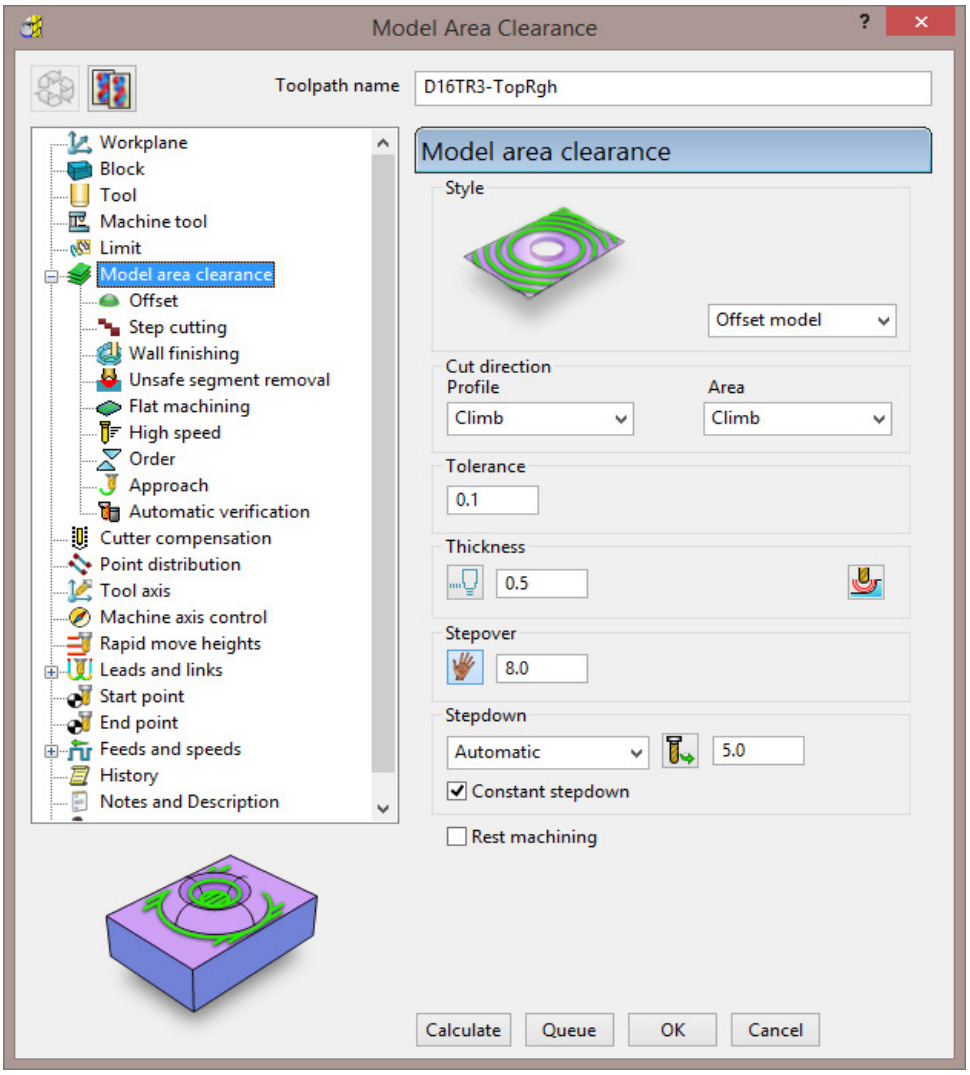
7 Create a **Dia 12 - tip radius 1** tool named **D12TR1**

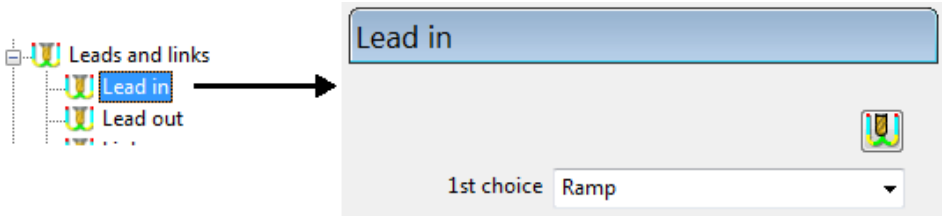
8 **Activate** the tool **D16TR3**

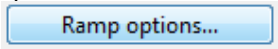
9 In the **Rapid Move Heights** dialog  select **Calculate**.

10 Use the default settings for both the **Start** and **End Points**  .

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

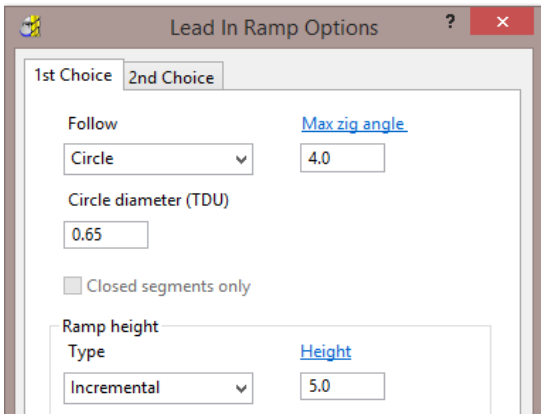




12 In the strategy **explorer**, select the **Lead in** option and in the new page select 1st choice **Ramp** followed by **Ramp options** 

.

13 In the **Ramp Options** dialog, Input **Max Zig Angle 4** using a **Circle** with **0.65** (TDU).



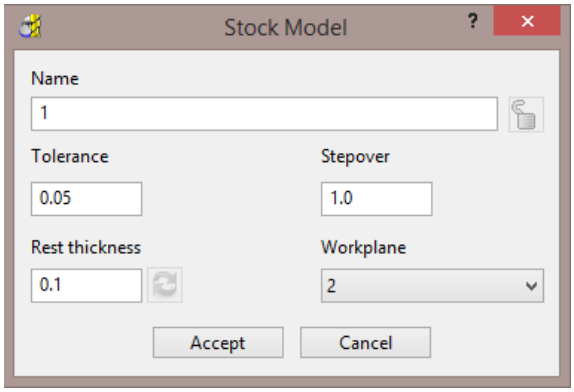
14 **Accept** both the **Ramp Options** and **Leads and Links** dialogs.

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

16 Select an **Iso1** view.

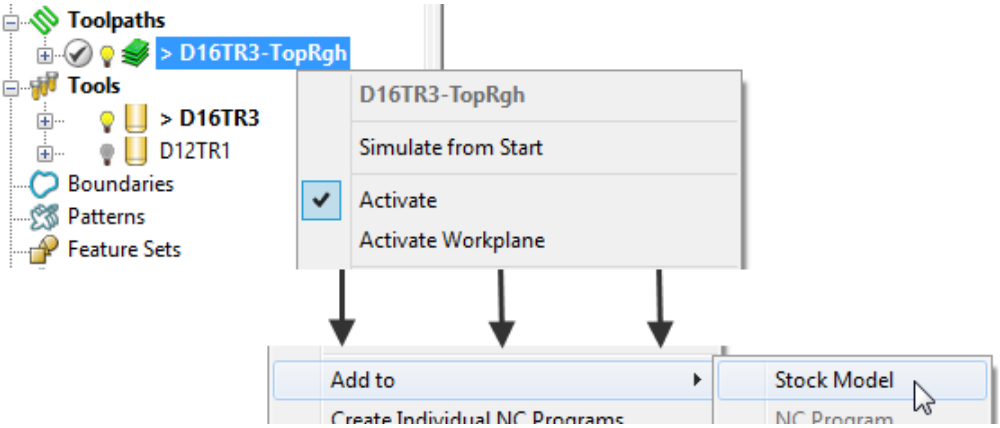
17 **Activate** > **Workplane 2** (The **Block** remains in situ!).

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



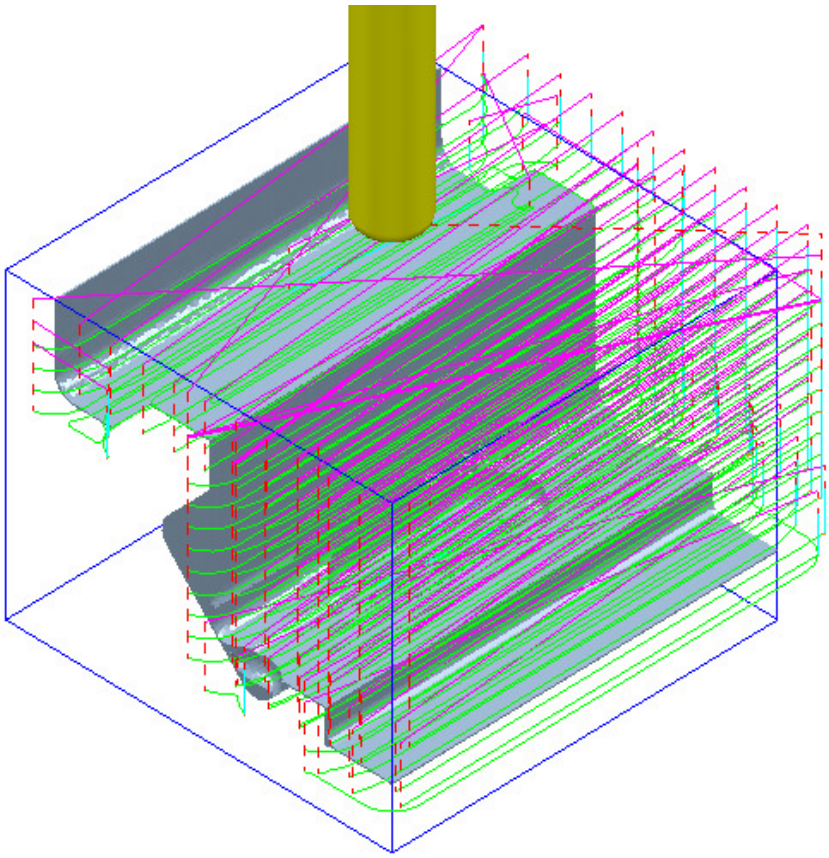
19 Input the above values and **Accept** the dialog.

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



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

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

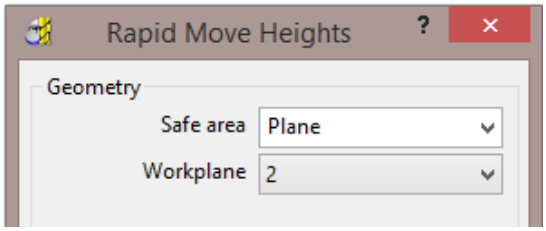


The **3-Axis Roughing** operation has removed all accessible material leaving a **0.5 thickness** on the component form. This is clearly visible on the displayed **Stock Model**.

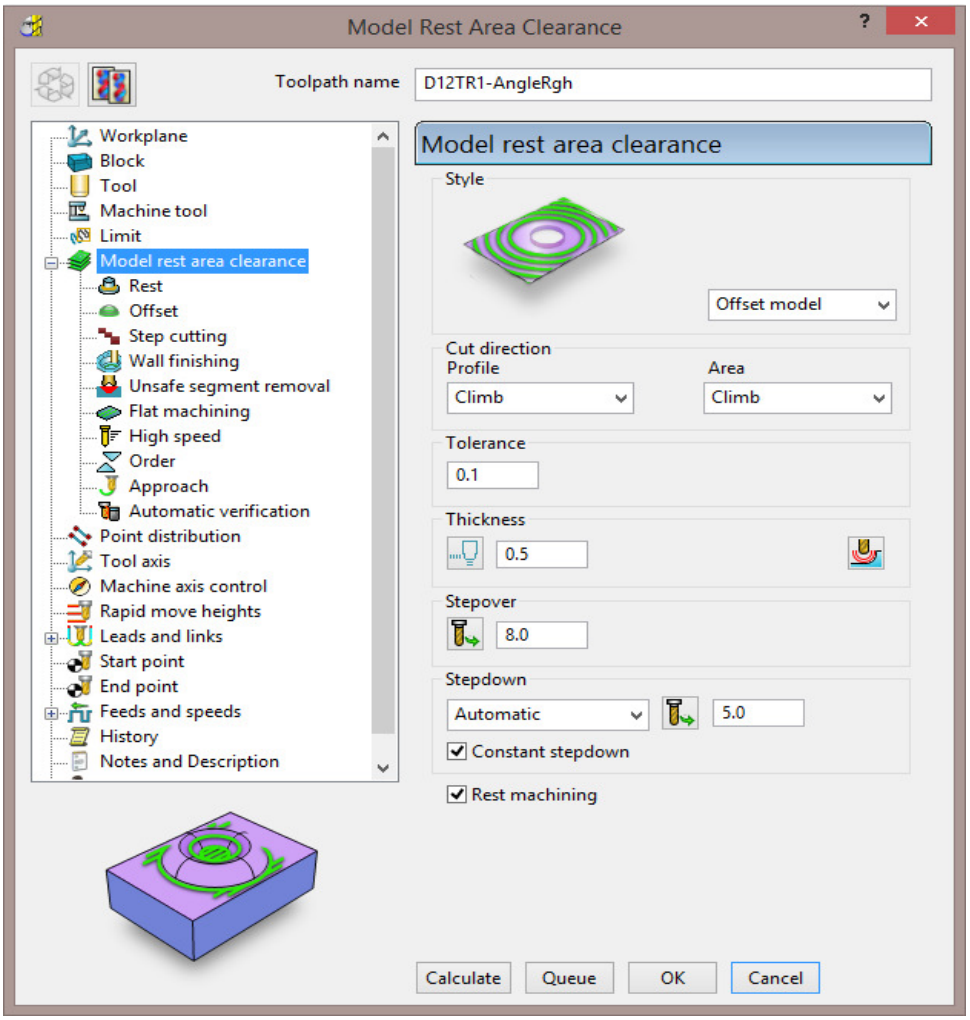
23 **Activate Workplane 2** to change the set up to a **3+2** orientation.

24 **Activate** the **tool *D12TR1***.

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

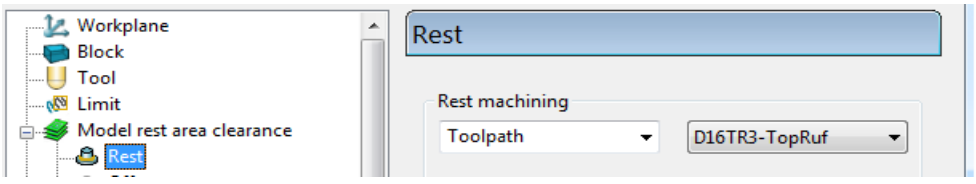


26 Select the **Strategy Selector** and from **3D Area Clearance** select **Model Rest Area Clearance** and input settings exactly as shown in the following illustration:

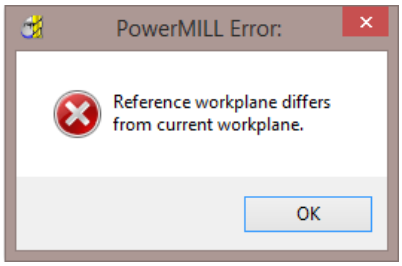


27 In the local explorer select the **Rest** page and input **Rest Machining** - **Toolpath** - **D16t3-TopRgh**.

28 Select **Calculate** to request the **toolpath** to be created.



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

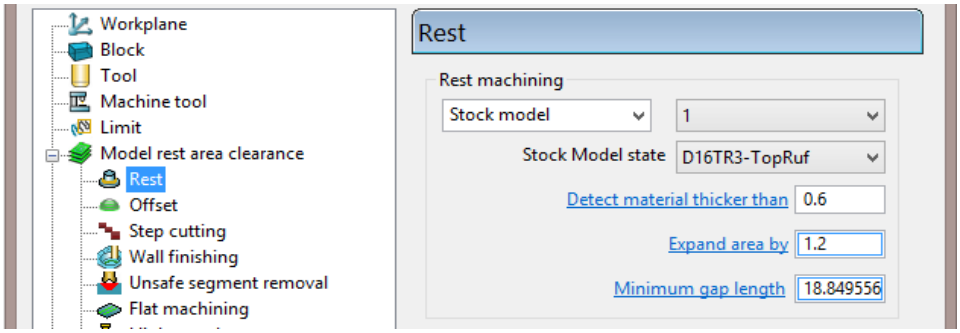


*It is not possible to apply **Area Clearance** - **Rest machining** using a reference toolpath that has been generated relative to a different **Workplane** alignment.*

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

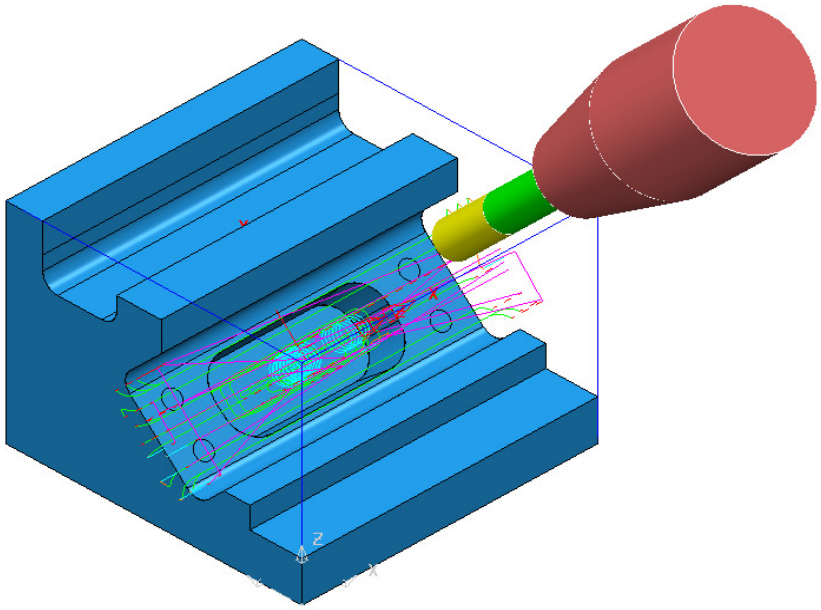
29 Select **OK** to close the **PowerMILL Error** message and return to the strategy (that is still in **Recycle** mode).

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



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

31 Select **Calculate** to create the **toolpath** and then **Close**.

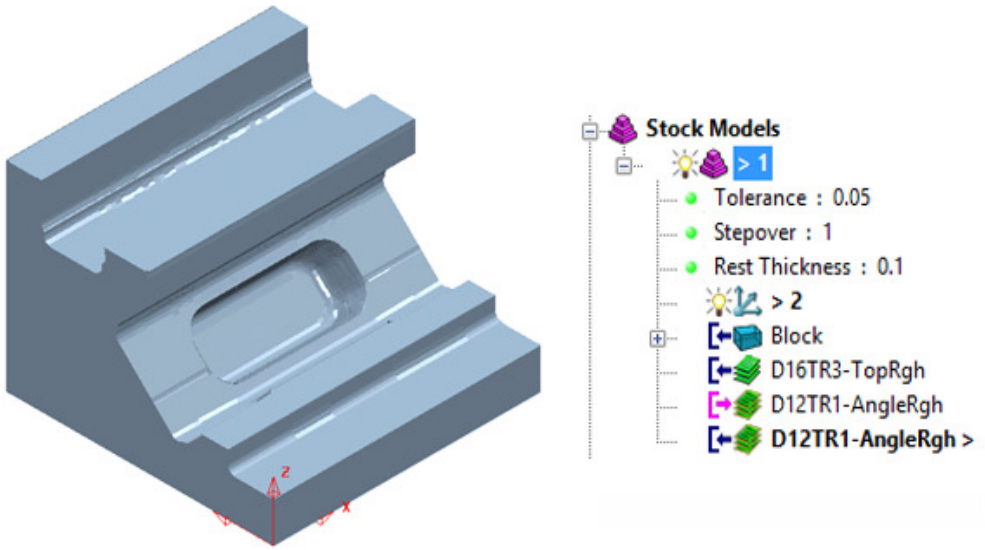


The modified **Rest Roughing** toolpath now successfully operates within the defined **Stock Model** volume (as shown above left).

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

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

33 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.

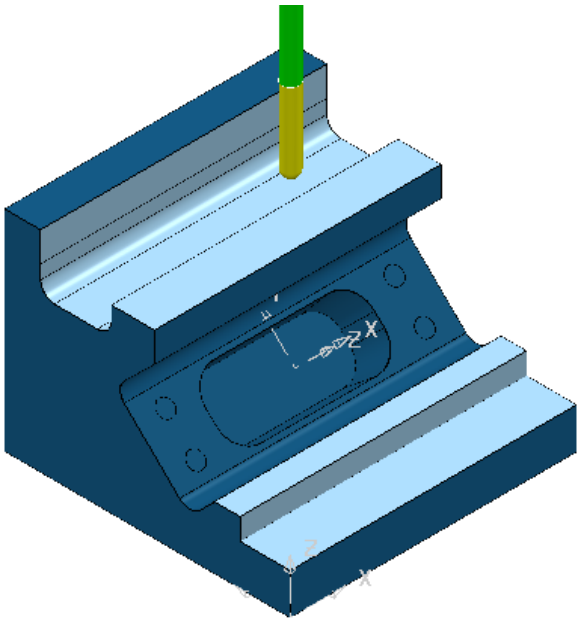
*Unlike **Area Clearance**, **Finishing** strategies cannot be directly **rest** machined to a **Stock Model**. However it is possible to create and apply **Stock Model Rest Boundaries** where required, providing suitable **rest** limits for subsequent finishing operations.*

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

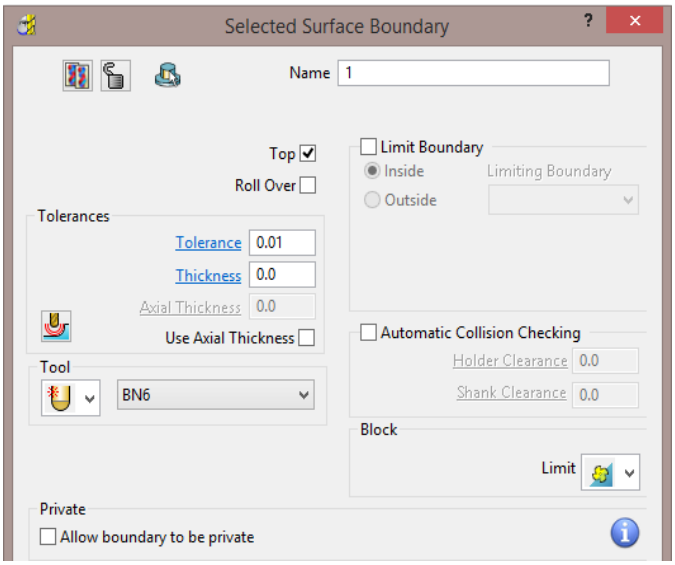
35 Create a Dia 6 Ball Nosed **Tool** named **BN6**.

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

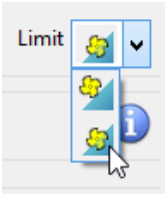
37 Select the **Surfaces** (shown shaded below) required for initial finish machining relative to **Workplane** 1.



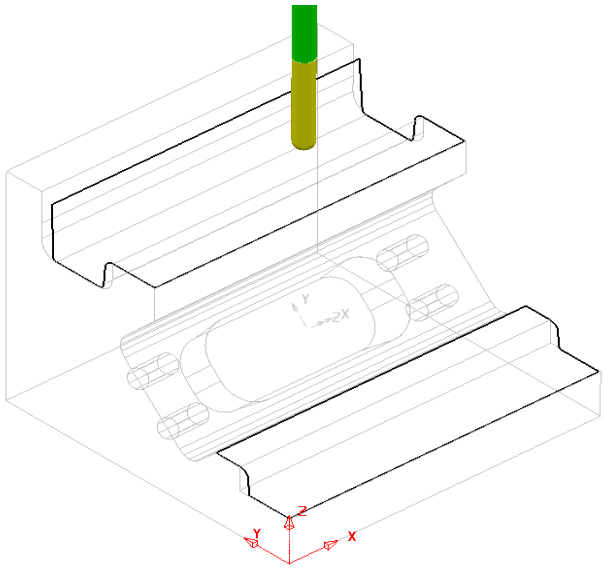
38 In the **PowerMILL *explorer***, right-click over **Boundaries** and select **Create Boundary** followed by **Selected Surface** to open the following dialog.



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



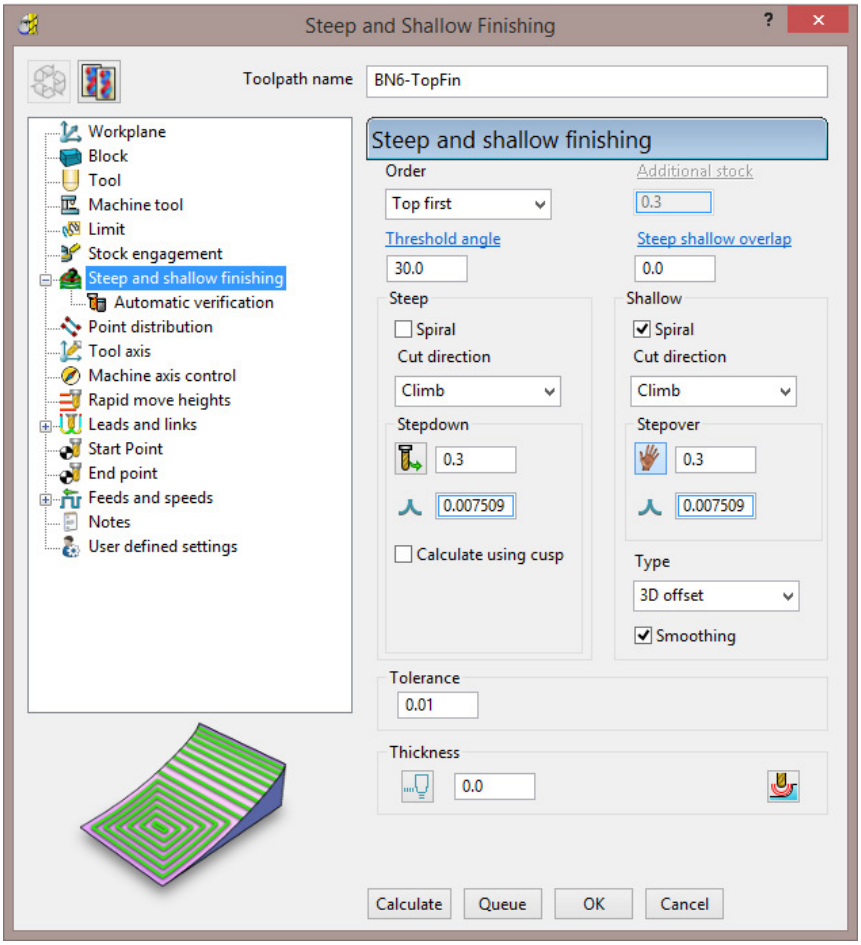
40 Select **Apply** and when processed, **Cancel** the dialog.



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

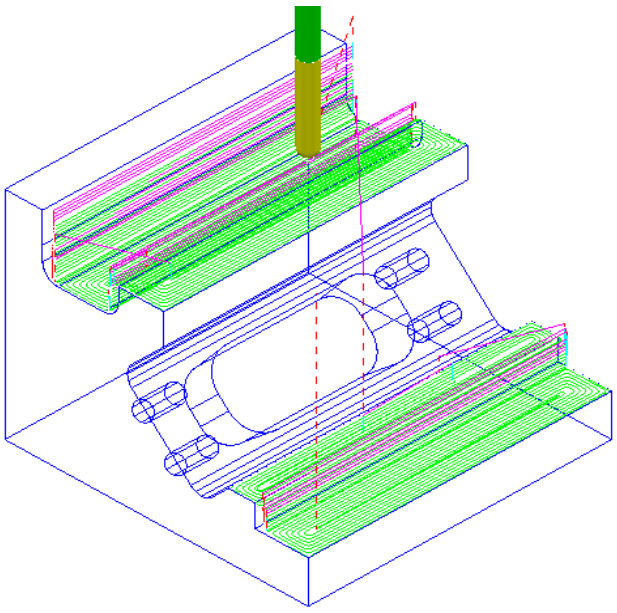
41 Select the **Strategy Selector** and from the **Finishing** dialog select the **Steep and Shallow** finishing option.

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



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

44 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**.

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

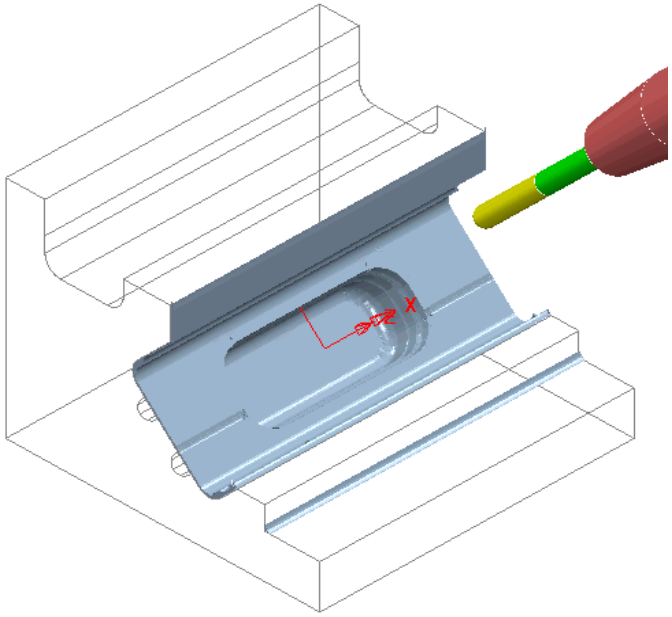
46 In the **PowerMILL *explorer***, right-click over the new **Stock Model 1** and from the local menu, select **Calculate**.

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

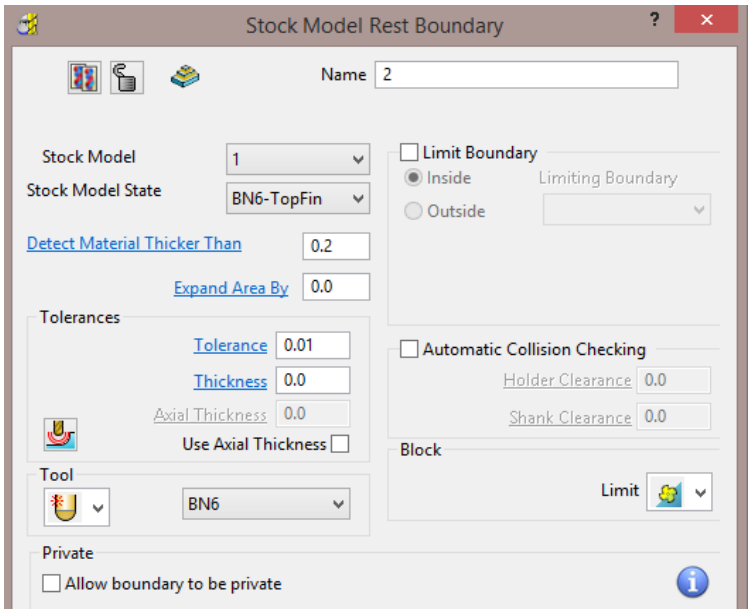
48 Select an **ISO 1**.

49 **Activate *Workplane* 2**.

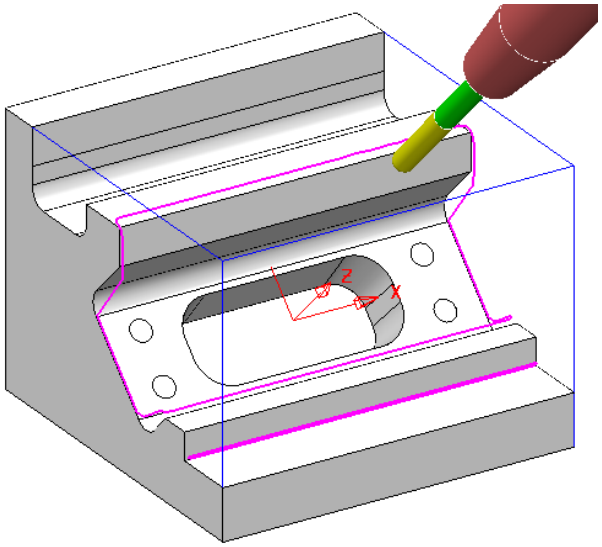
50 From the **main** toolbar select **Rapid Move Heights** and input the correct **workplane** (**2**) in the dialog before selecting **Calculate**.



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



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



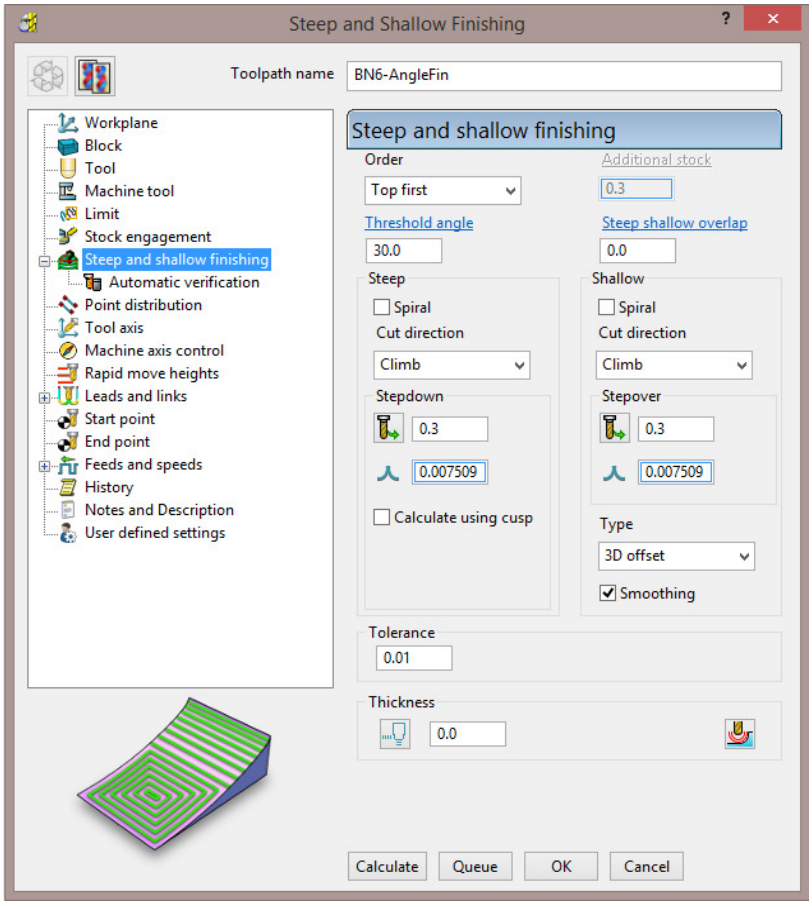
53 **Apply** and when processed, **Cancel**.

54 Select and **Delete** the narrow segment at the base of the lower vertical wall.

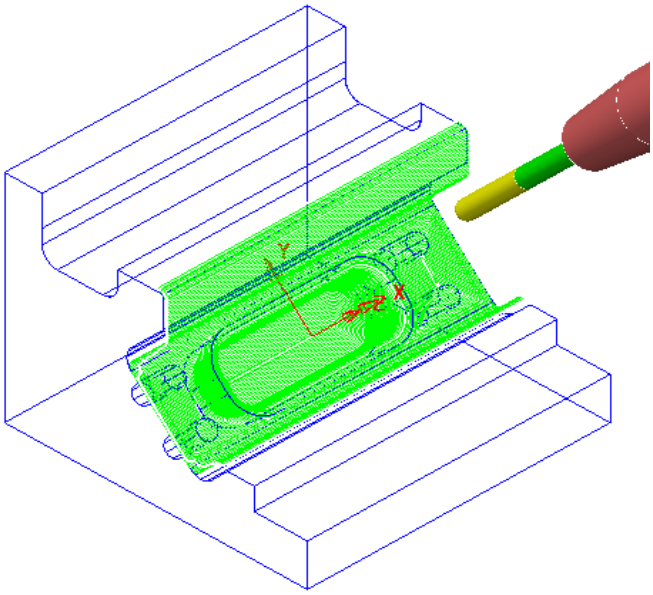
55 Ensure that new **Boundary 2** is **Active**.

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

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



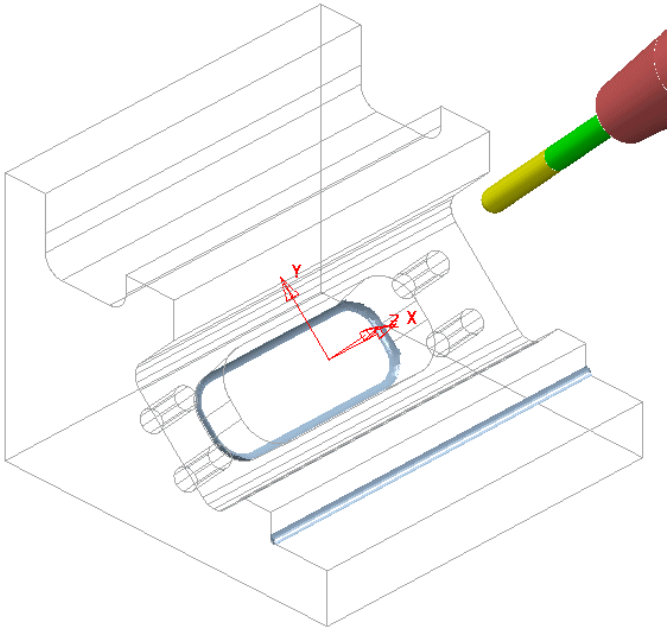
58 Select **Calculate** to create the **toolpath** and then **Close** the dialog.



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

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

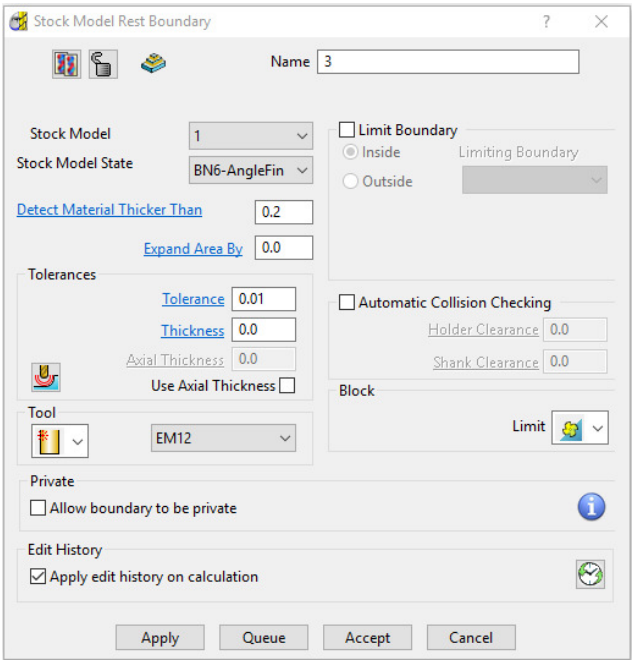
60 In the **PowerMILL *explorer***, right-click over the new **Stock Model 1** and from the local menu, select **Calculate**.



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.

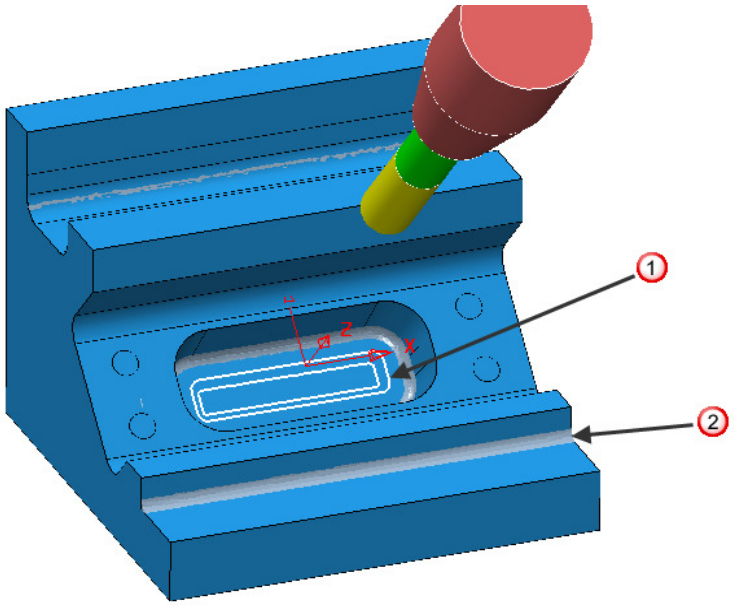
61 Create a Dia 12 End Mill named **EM12**.

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



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

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



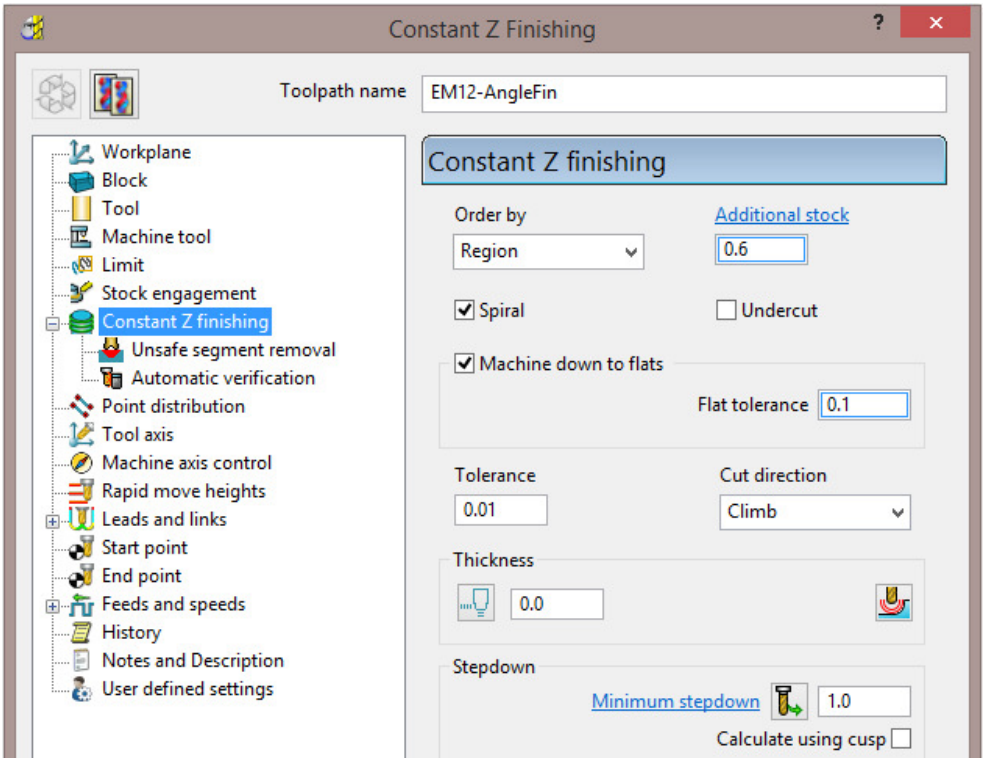
1 A new **Stock Model Rest *Boundary*** has appeared where the remaining material is accessible to the **EM12** tool.

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

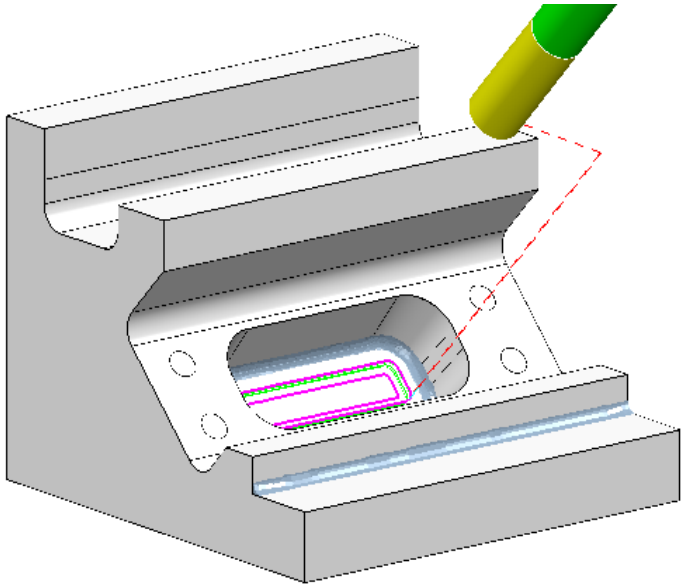
65 Ensure that the new **Boundary** (**3**) is **Active**.

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

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



68 Select **Calculate** to create the toolpath and then **Close** the dialoq.



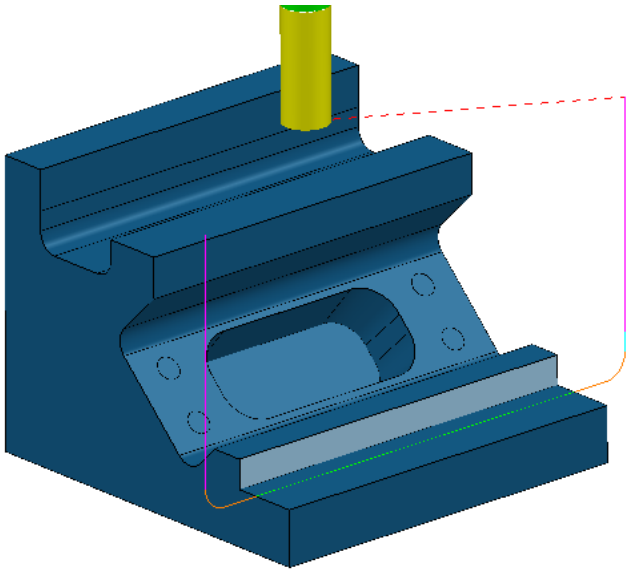
The angled pocket is now fully machined and the latest toolpath will now be added to the **Stock Model**.

69 In the **PowerMILL *explorer***, right-click on the **Toolpath EM12-AngleFin** and from the local menu select, **Add to** - **Stock Model**.

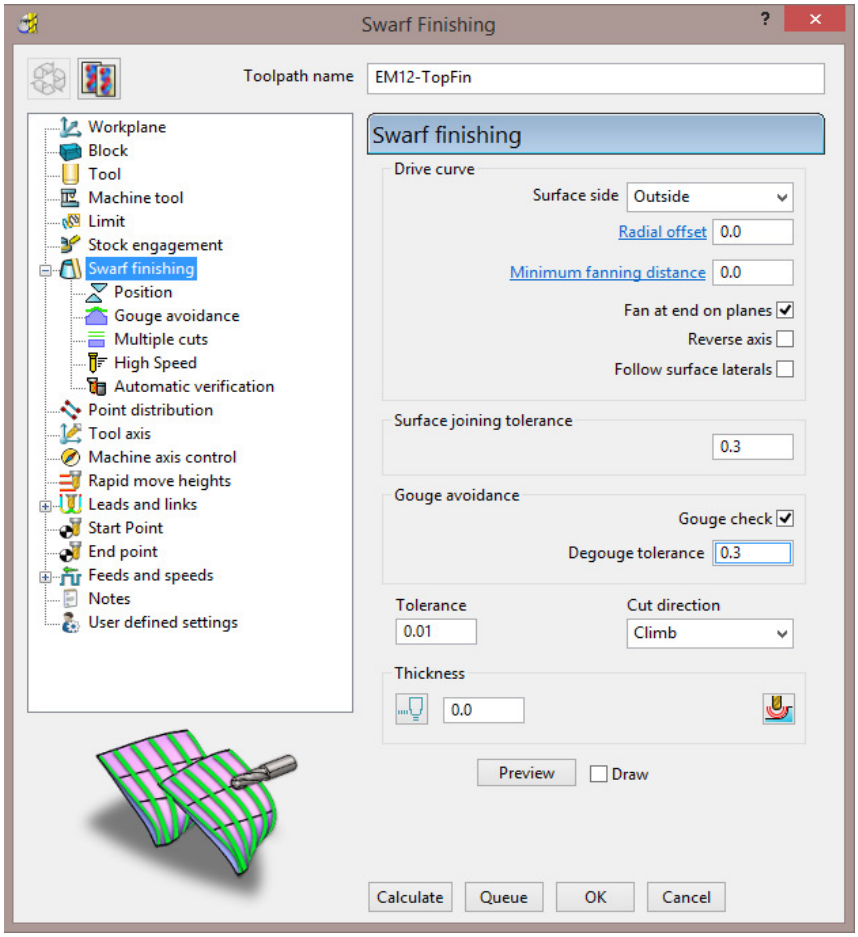
70 In the **PowerMILL *explorer***, right-click over the new **Stock Model 1** and from the local menu, select **Calculate**.

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

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

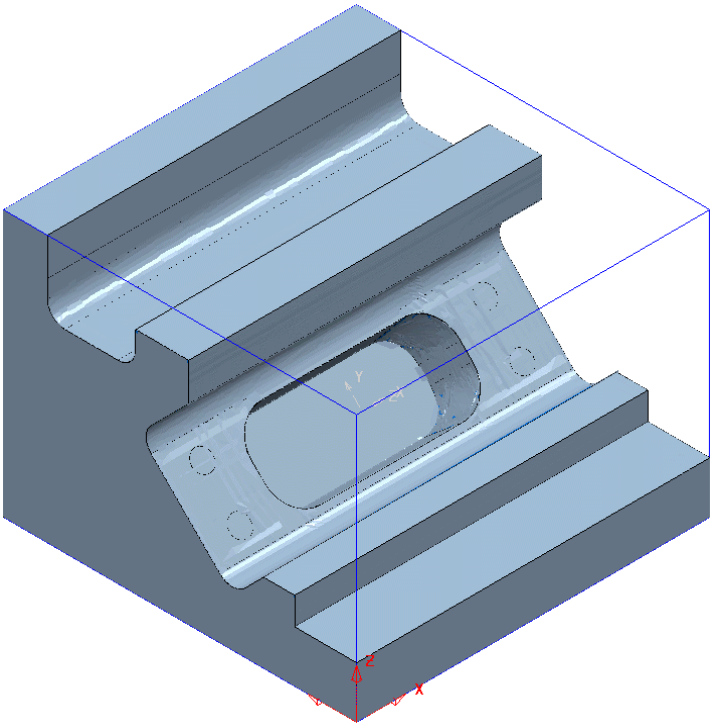


73 Create a **Swarf Finishing** strategy named **EM12-TopFin** on the selected, vertical **surface** (shown shaded above).



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

75 ***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.*

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

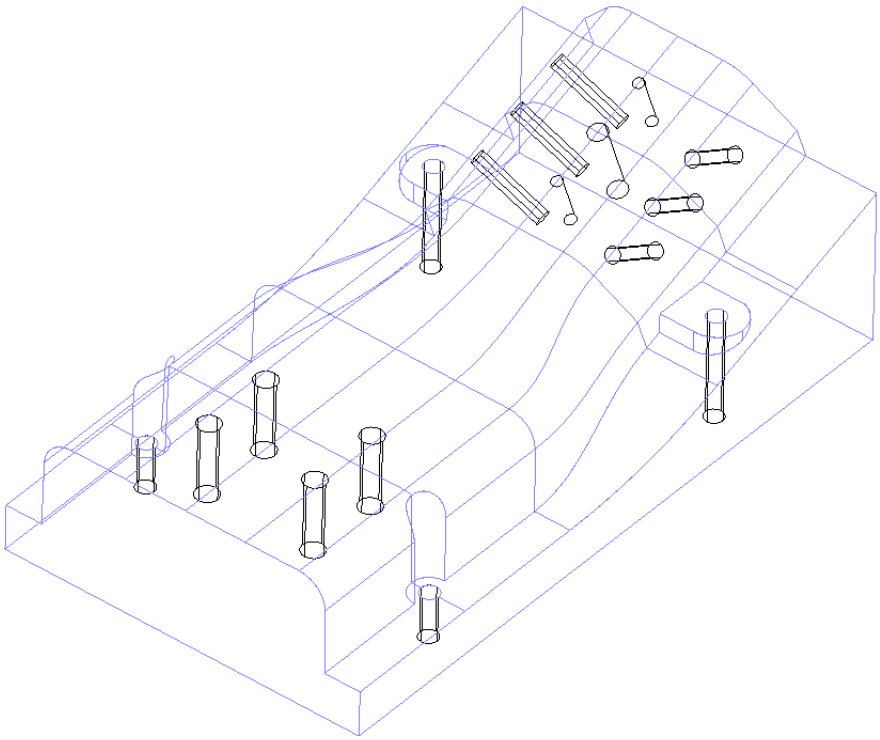
3+2 Axis - Drilling Example

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 **Feature Set** into which all cylindrical surfaces in the selected model are inserted as **Hole** features.

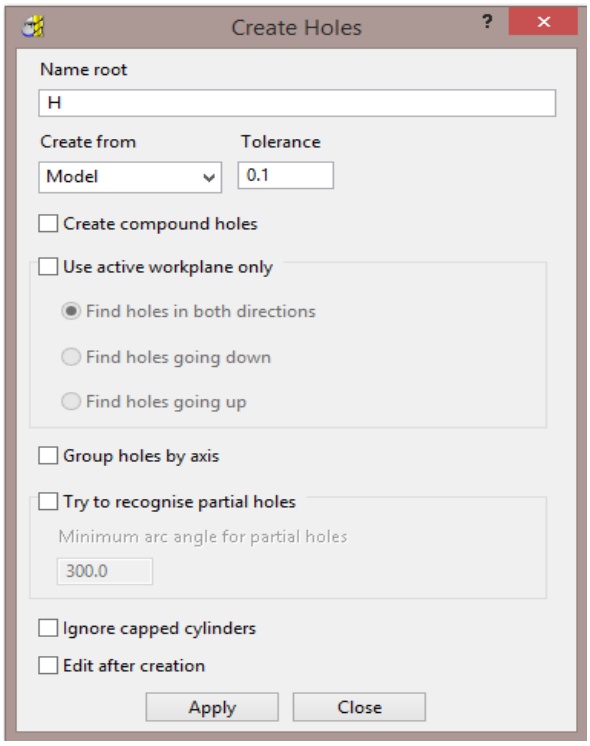
Any cylindrical surfaces within the selection will automatically be recognised as a **Hole Feature**. It is important that no **Block** exists to ensure that the **Hole Features** are arranged with the top at the **maximum Z height**.

*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 **Feature Sets**, and from the local menu select **Create Holes**.

6 Enter data into the **Feature** 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**.

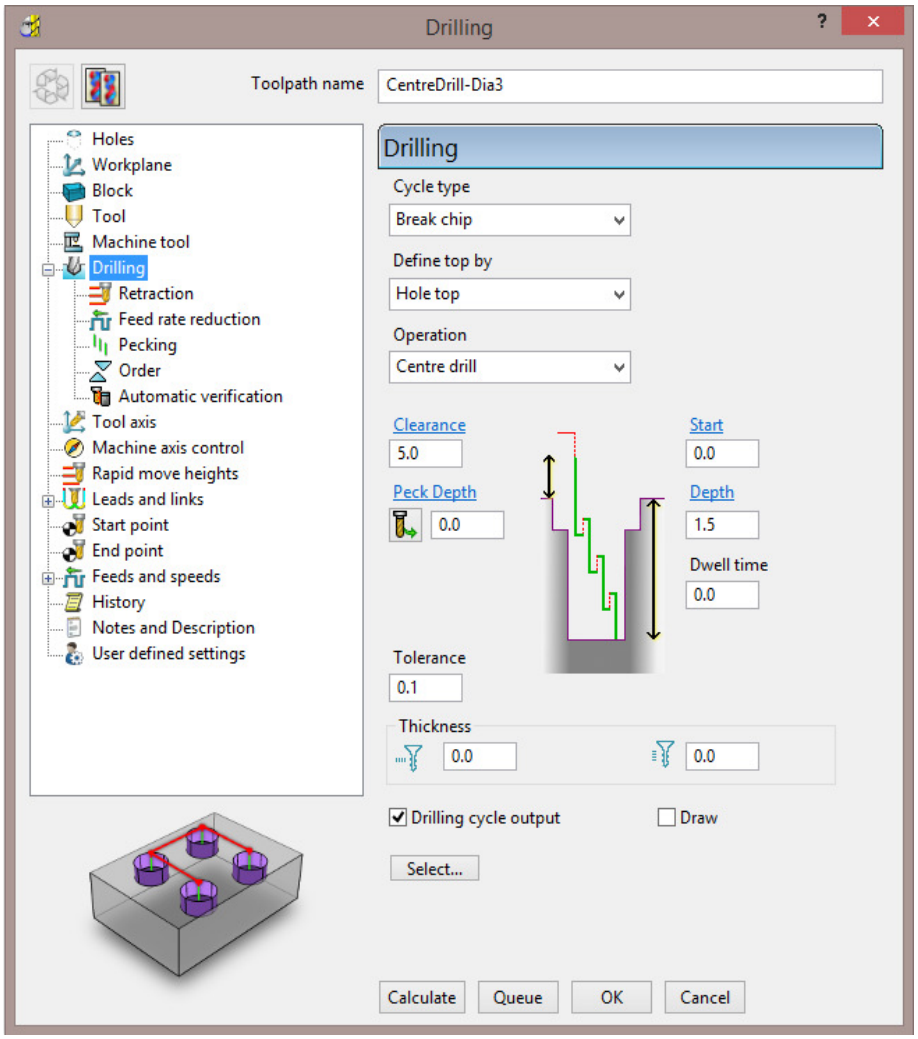
12 Add a **Shank** component Upper\Lower Dia **3** Length **40**.

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**.

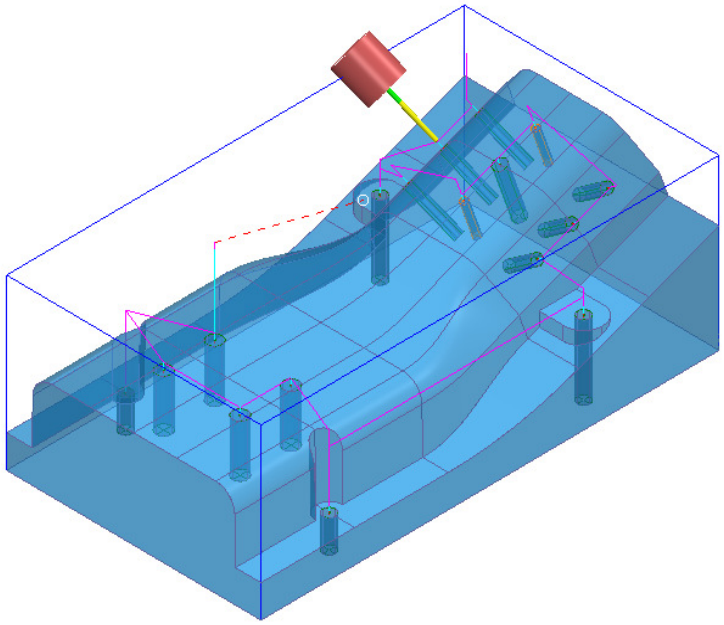
15 In the **Drilling** dialog, select the option **Drilling**.

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



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**

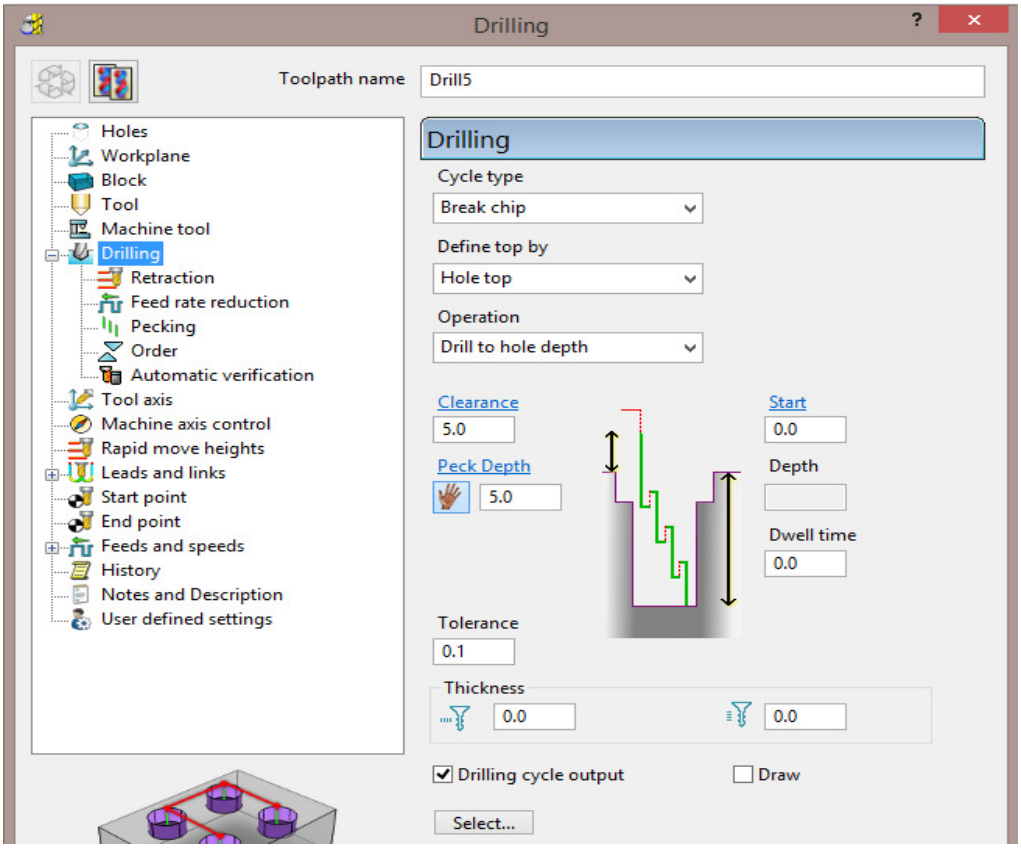
20 Add a **Shank**, Upper Dia **5**, Lower Dia **5**, Length **30**.

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

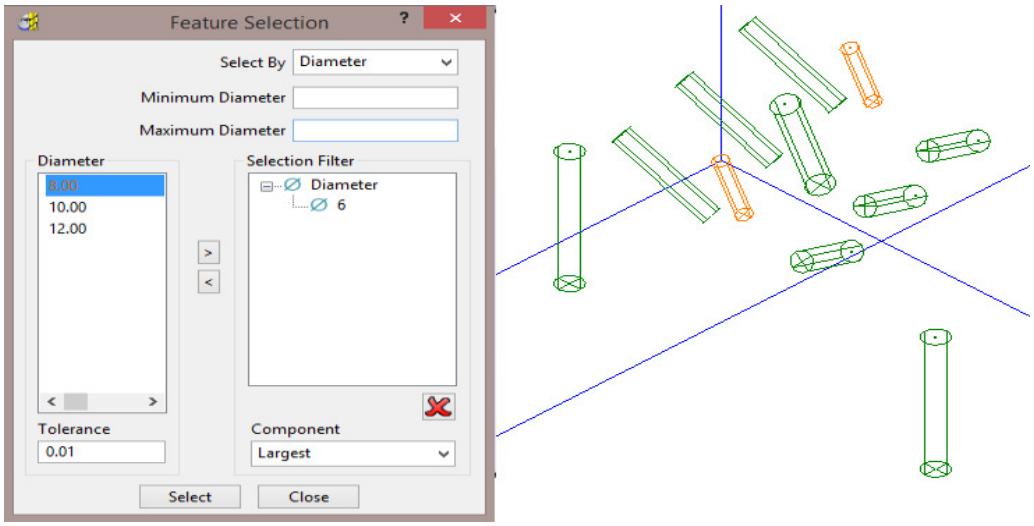
22 Select **Toolpath Strategies** and in the **Strategy Selector** dialog select **Drilling**.

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

24 Rename the toolpath as **Drill5**.



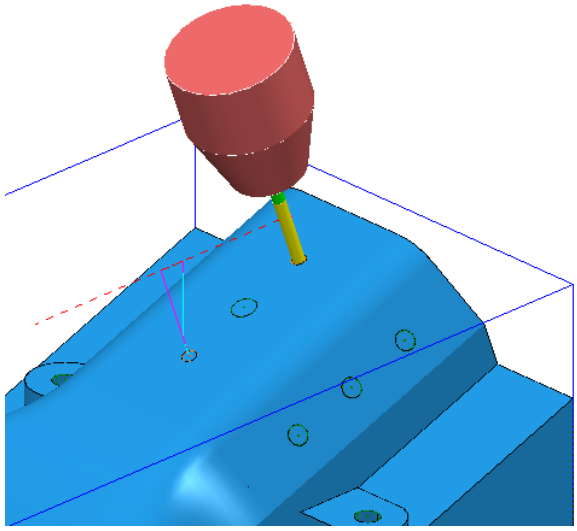
25 Select the two **Dia 6 *Hole Features*** (shown coloured orange below).



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**.

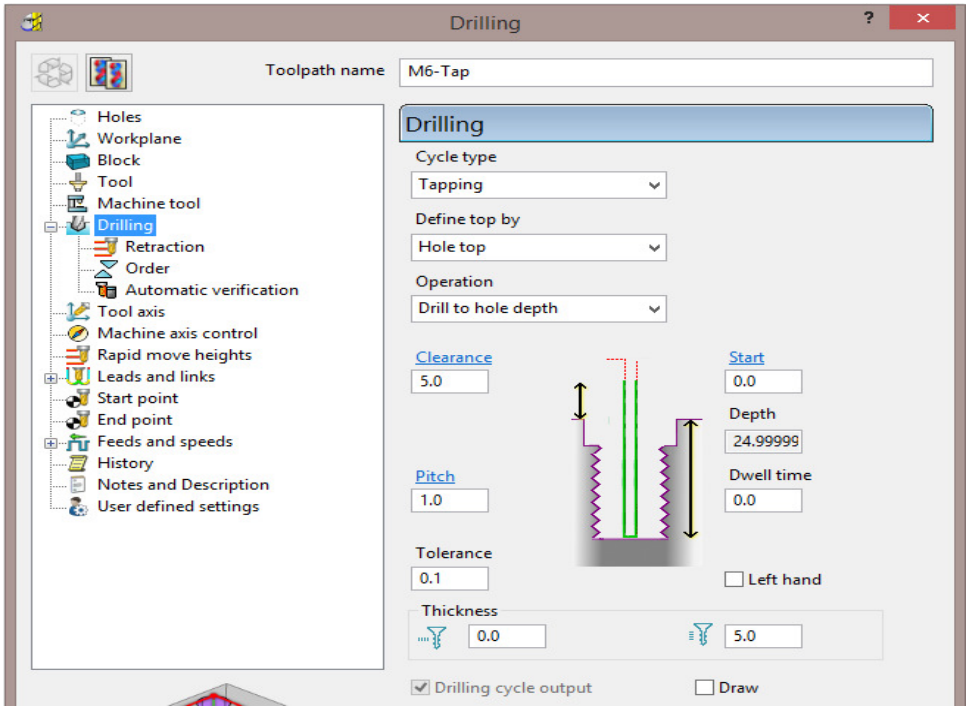
30 Add a **Shank**, Upper Dia **4**, Lower Dia **4**, Length **30**.

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**.

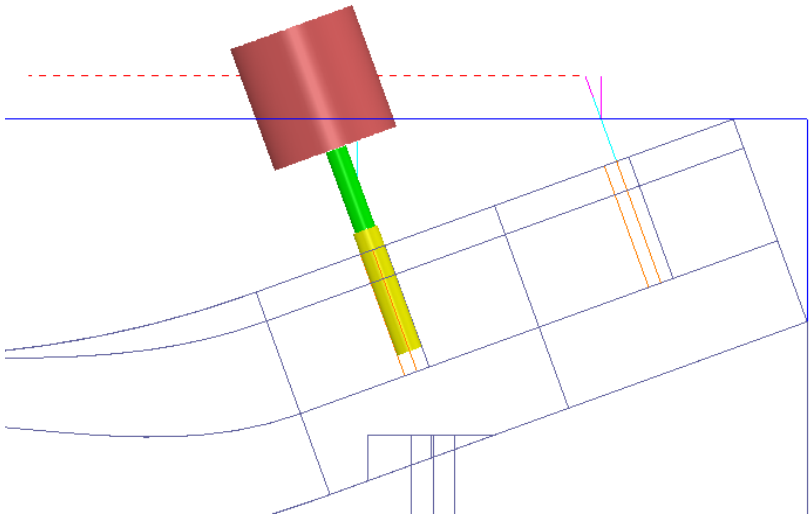
38 Set **Cycle Type** - **Tapping**, **Operation** - **Drill to hole Depth**, and **Pitch 1**.

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.

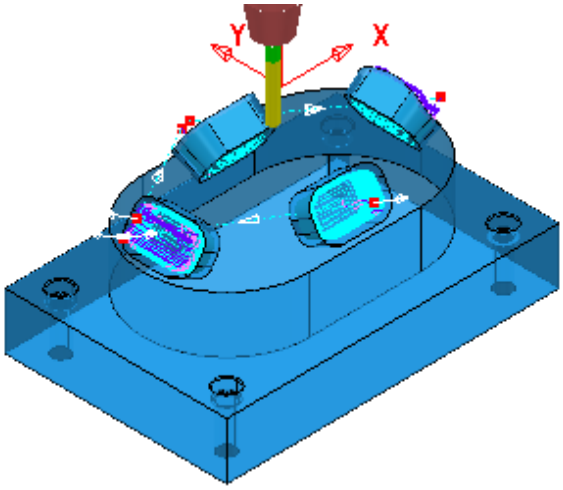
43 From the **main** pulldown menus, select **File** > **Save** to update the contents of the previously stored **Project**.

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**.

Postprocessor option file – Safe Z Positional Moves

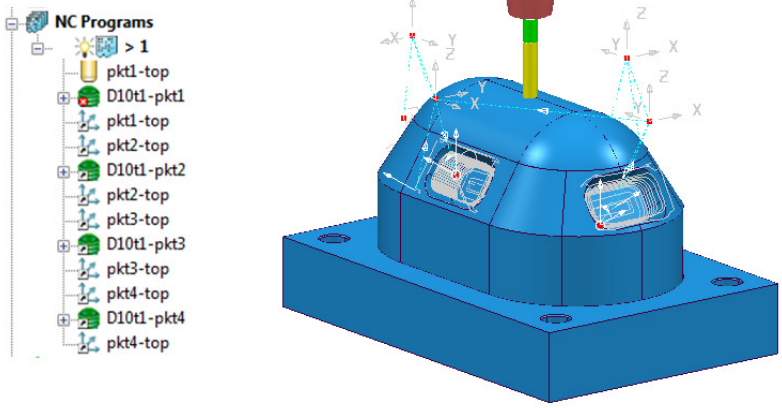
There are two different ways that a specific **Postprocessor Option file** can be created to deal with positional moves (Method 1 is the default).

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**

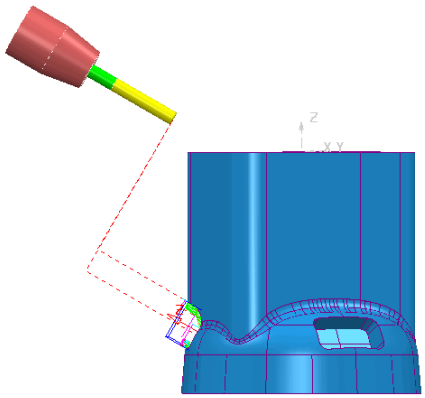
b Method 2 – The **Postprocessor Option file** does not automatically output Machine Tool **Safe Z** moves. All additional, safe positional, moves must be added by the user in **PowerMILL** (In this case a network of **workplanes** can be used as shown in the illustration below).

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).

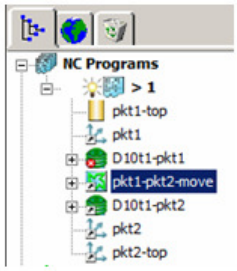


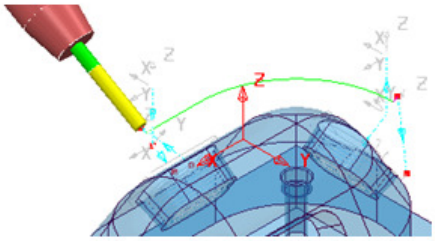
Start and End Points input as **Absolute** coordinates, above the component where it is safe to re-align the tool and/or perform rapid XY moves.

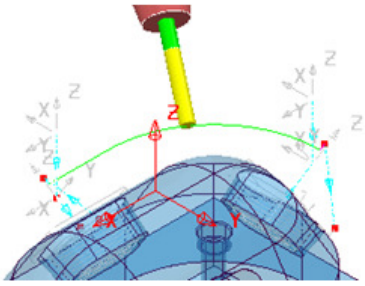
Tool Moves using a Pattern Finishing in 3D Space

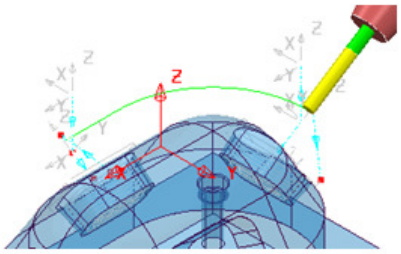
Positional Tool Moves can be controlled by running the tool along a **Pattern Finishing** strategy used as a ***Drive Curve*** in 3D space.

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





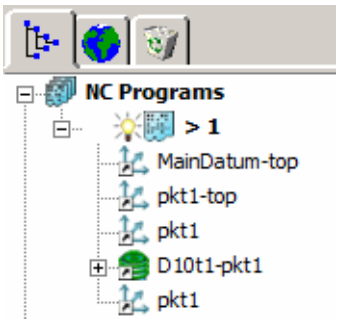




Tool Move via Workplanes in an NC Program

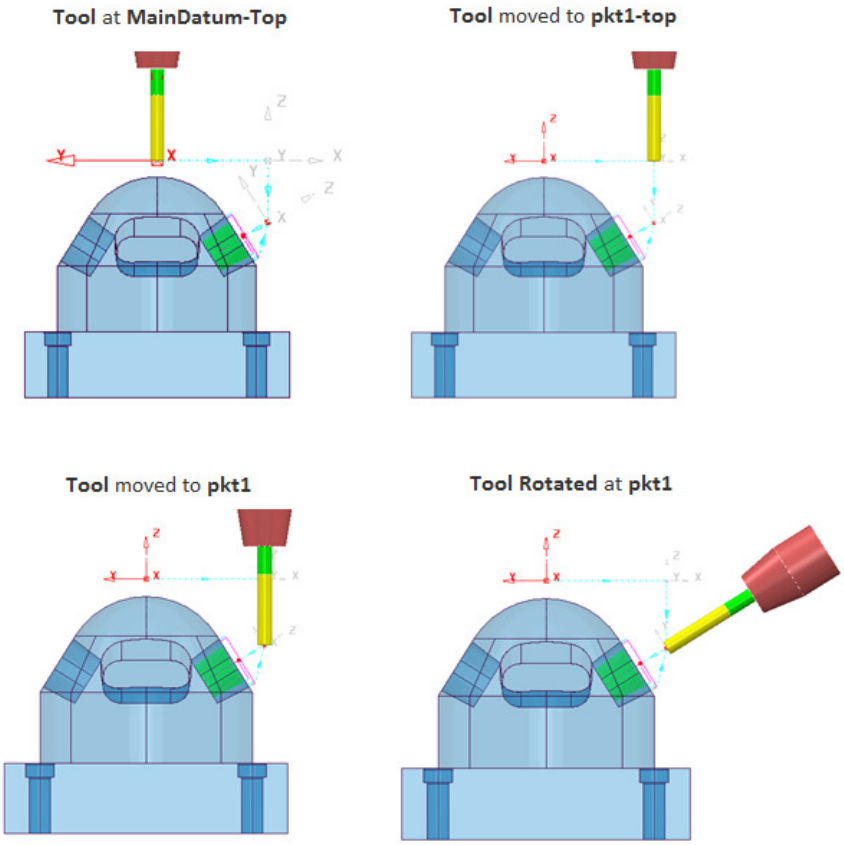
Positional Tool Moves can be controlled by strategically placed **Workplanes** inserted between the **Toolpaths** in the **NC Program** list. A **Workplane** in the **ncprogram** list can also be registered as a **Toolchange Point** if required.

*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.

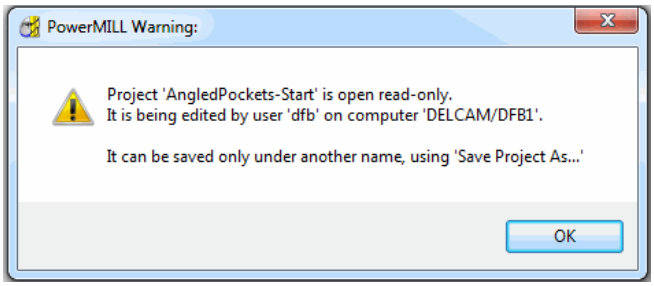


Example

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**:-

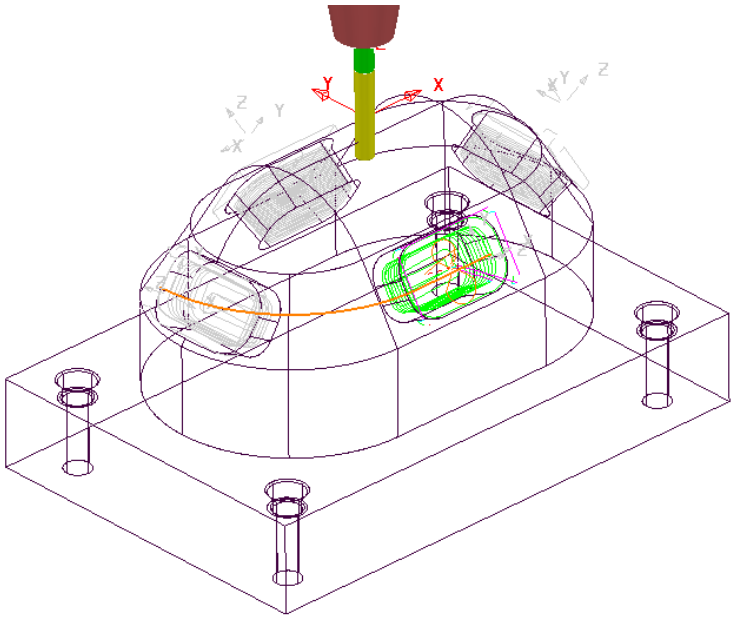
....\PowerMILL_Data\FiveAxis\PositionalMoves\AngledPockets-Start



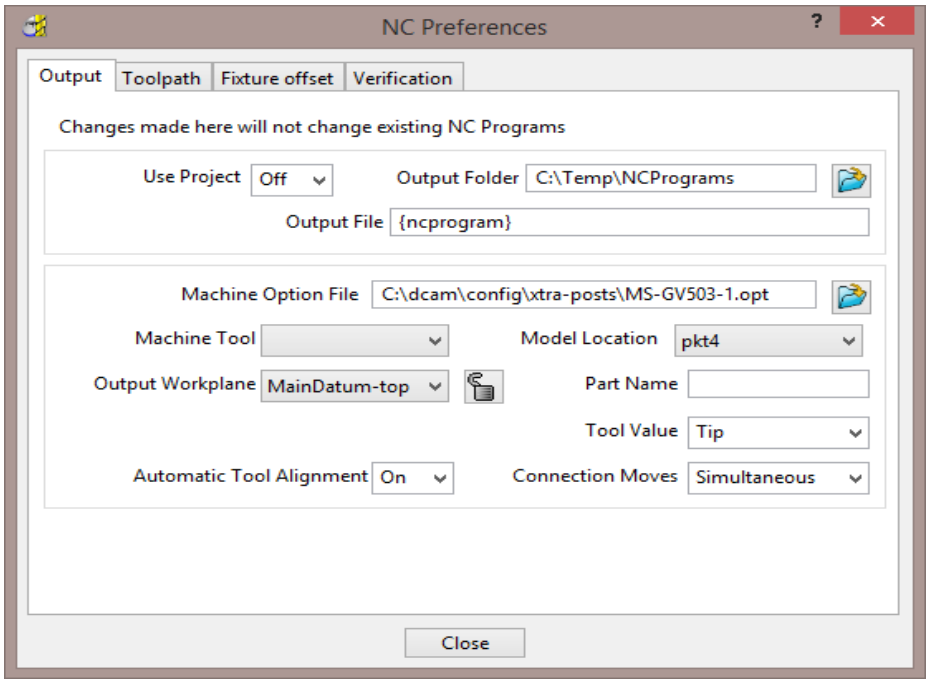
2 Select **File** - **Save Project As**:-

....\COURSEWORK\PowerMILL_Projects\AngledPockets

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**.



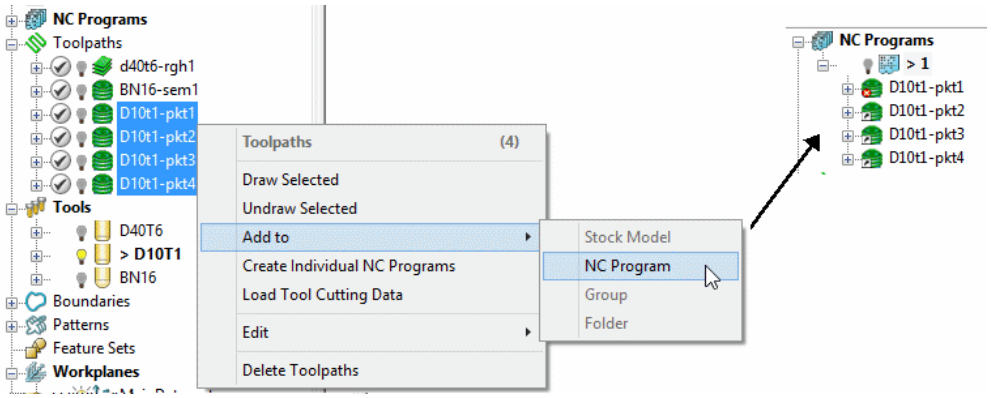
4 Select a suitable ***Five Axis***, Post-Processor ***Option File***:

C:\dcam\config\xtra-posts\MS-GV503-1.opt

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

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

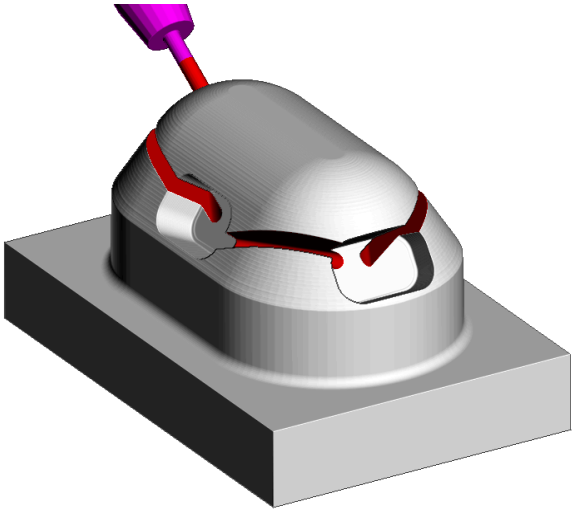
7 Select the four ***3plus2*** **Toolpaths** (***D10t1-pkt….***) then right-click over them and in the local menu select **Add to** > **NC Progam**.



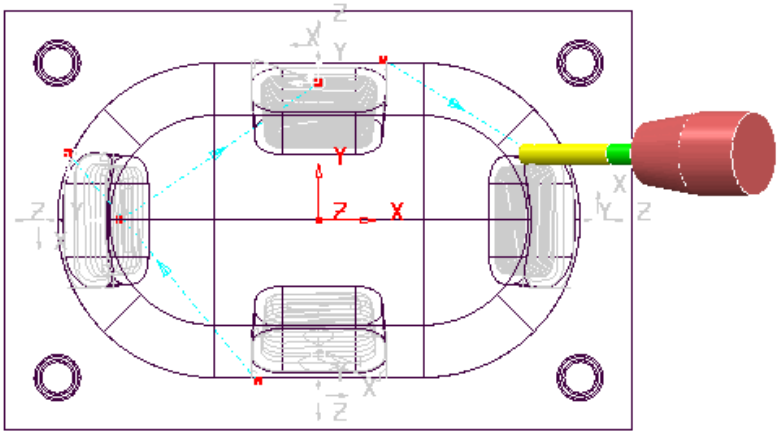
8 Select an **ISO 4** view.

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.*



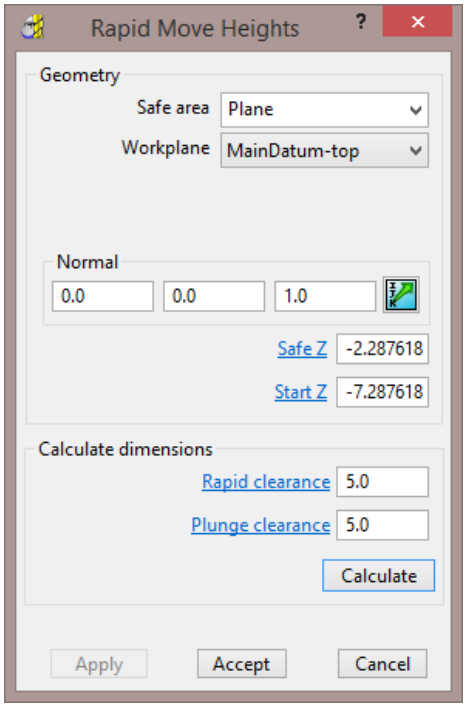
*If the **Model** is displayed as **wireframe** then the displayed **NC Program** will also include the **link** moves (Pale Blue dashed lines) between the 4 stored **Toolpaths**.*



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

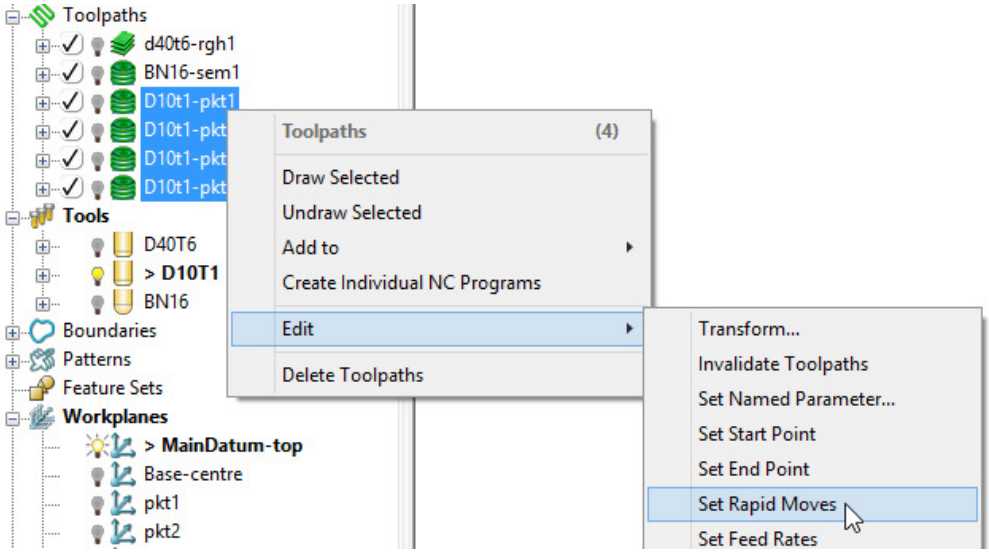
11 **Calculate** a **Block** using ***Defined by*** **Box** to the **Model** dimensions.

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



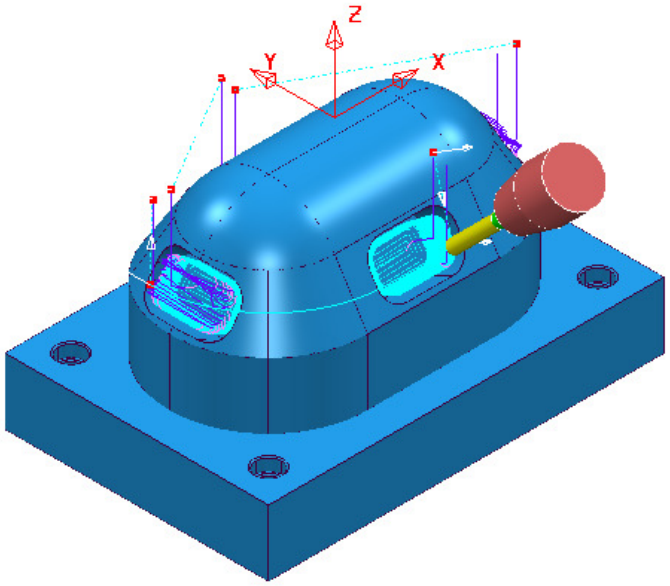
13 Input ***Workplane*** **MainDatum-top** and **Calculate** before selecting **Accept**.

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

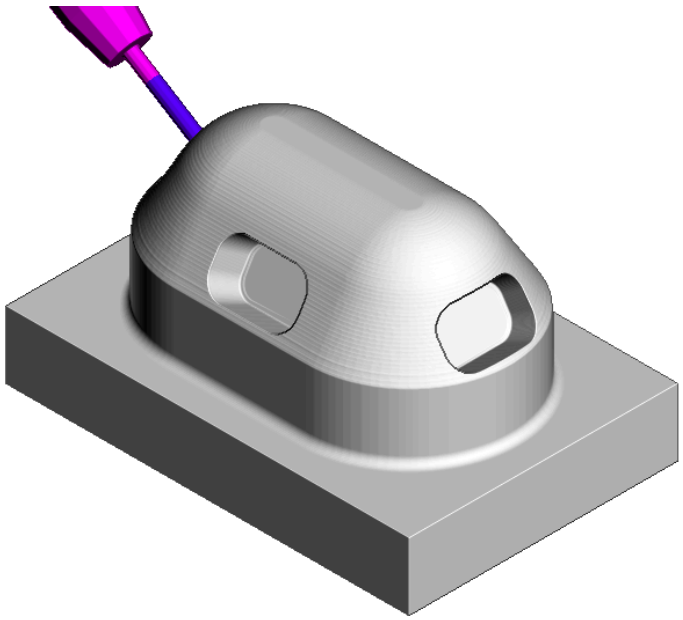


15 Right mouse click anywhere on the four selected **toolpaths**, and from the local menu select **Edit** – **Set Rapid Moves**.

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



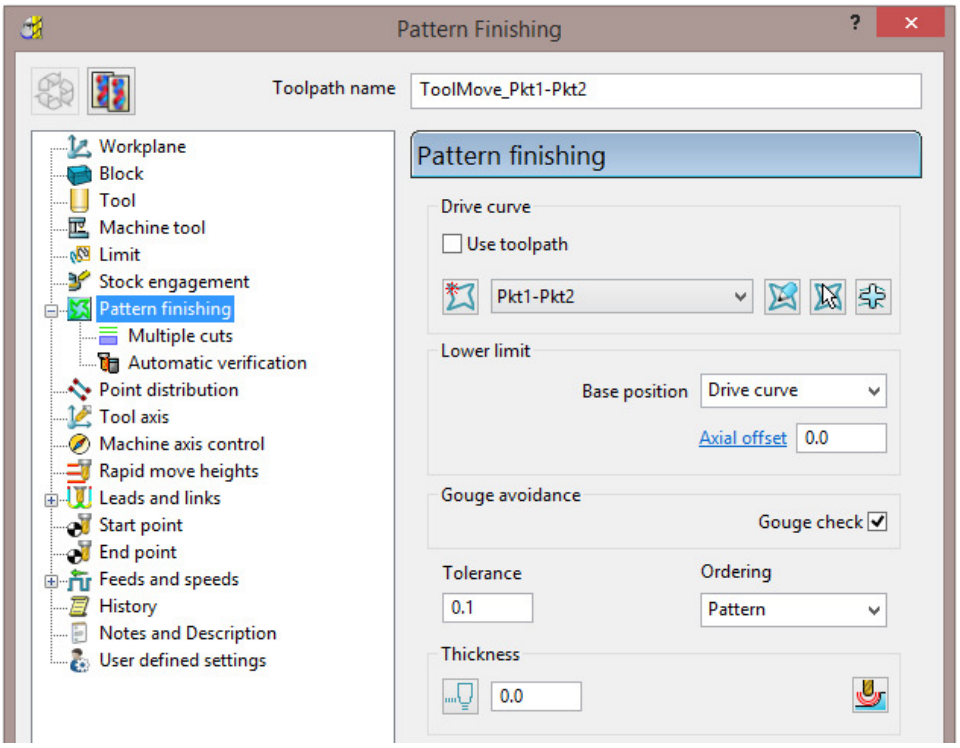
16 Run a **ViewMILL** simulation on the first two, 3 Axis toolpaths; **D40t6-rgh1**, **Bn16-sem1**, followed by the **NC Program** containing the four 3plus2 toolpaths.



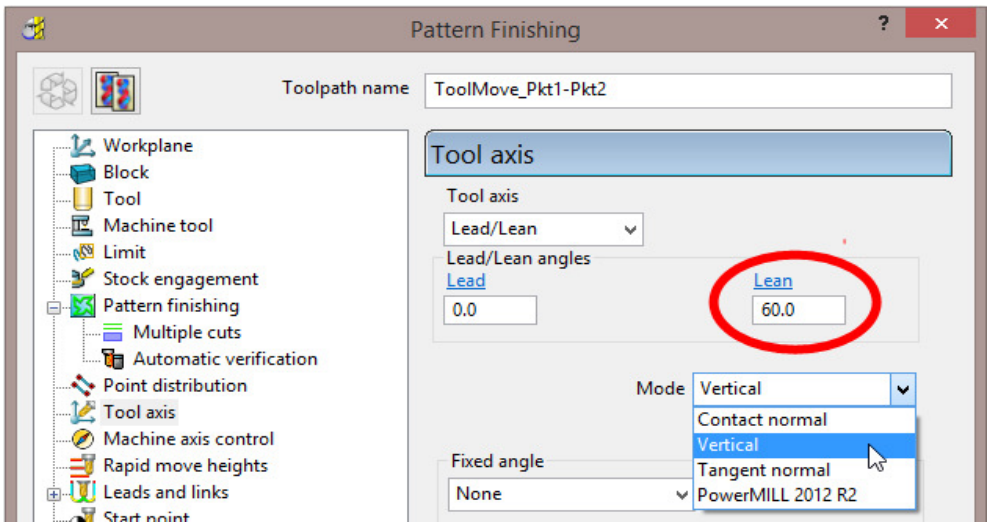
The toolpath connection moves no longer collide with the model.

Connection moves using (Drive Curve) Pattern toolpaths

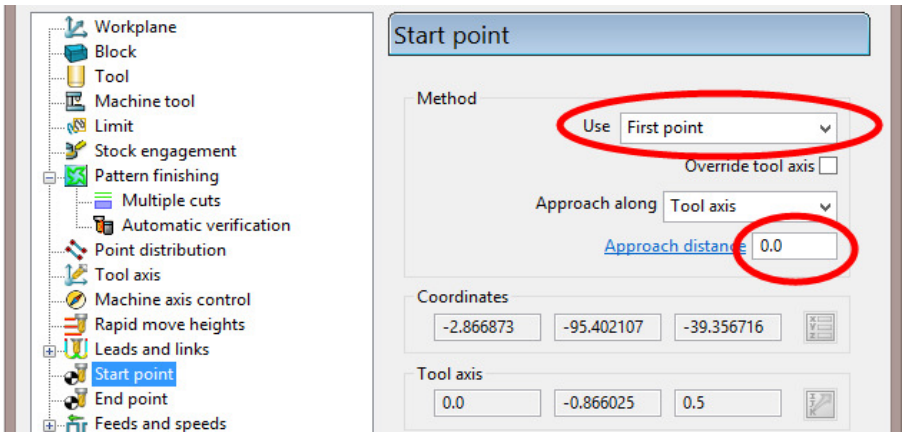
17 From the **Strategy Selector** - **Finishing** select the **Pattern** finishing strategy and enter data exactly as shown below.



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



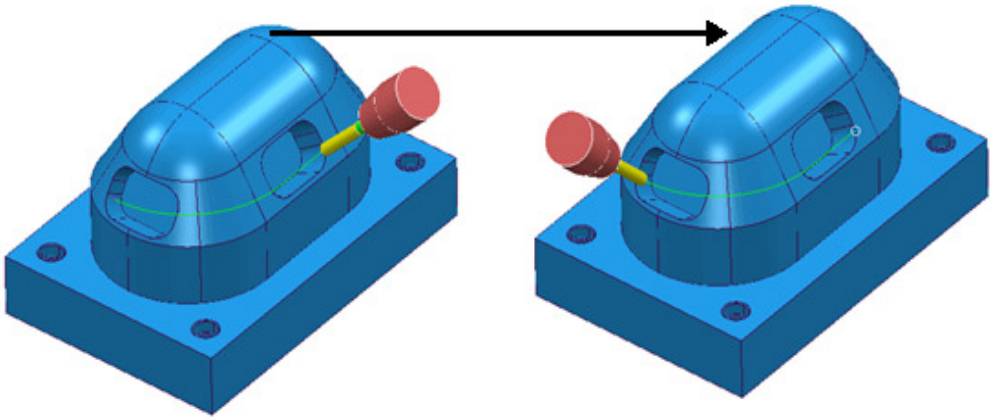
19 Select the **Start point** page, and ***Use*** **First point** with ***Approach distance*** **0**.



20 Select the **End point** page and ***Use*** **Last Point** with ***Approach distance*** **0**.

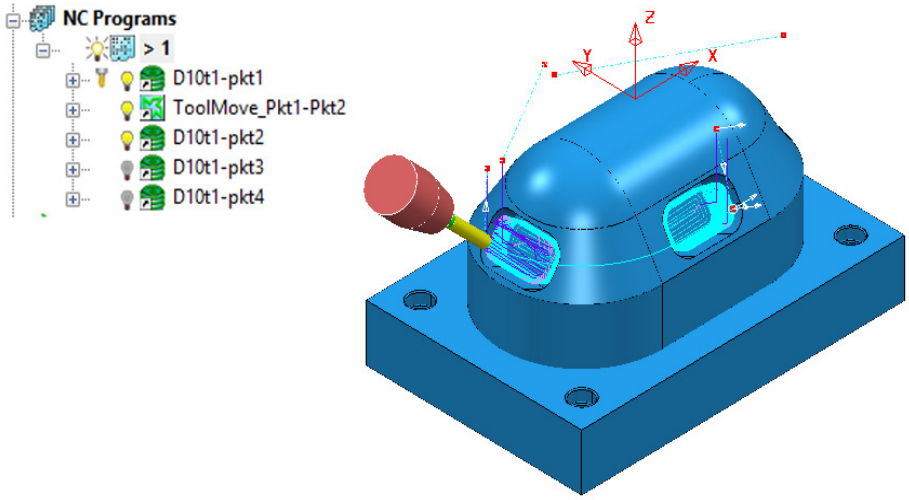
21 In **Leads and links** set both **Lead in** and **Lead out** to **None**.

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**.

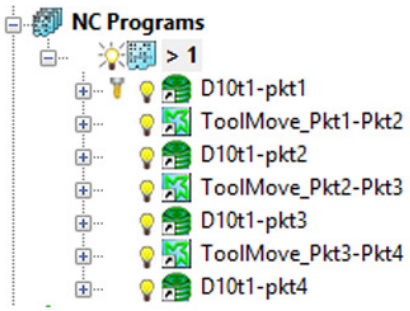


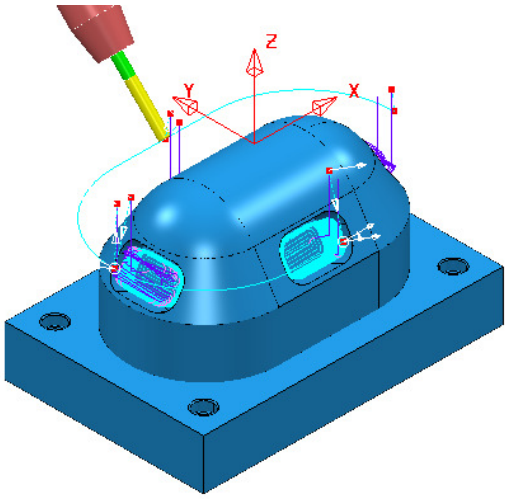
Exercise

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

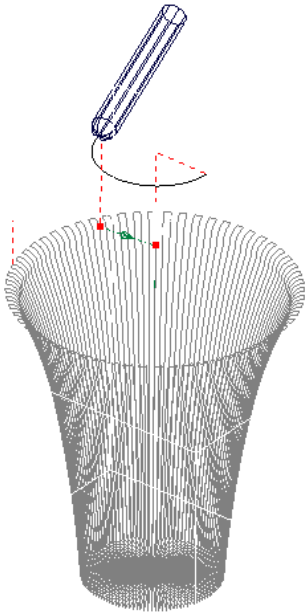
*Check the **direction** of each new **Pattern Strategy** and if necessary, reverse.*

25 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 use to '**Rewind**' the **tool** in 3D space, back to the start of its rotary travel limits.*

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**.

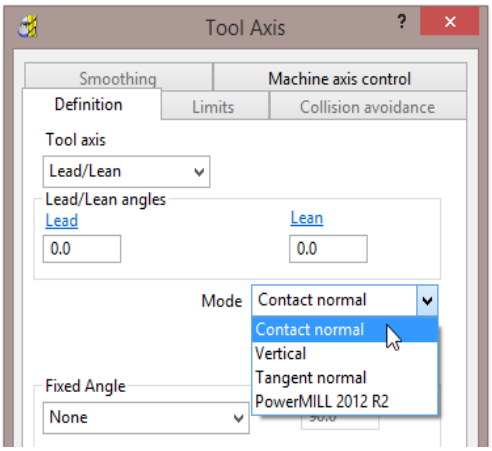
Five Axis Tool Alignment and Machining Options

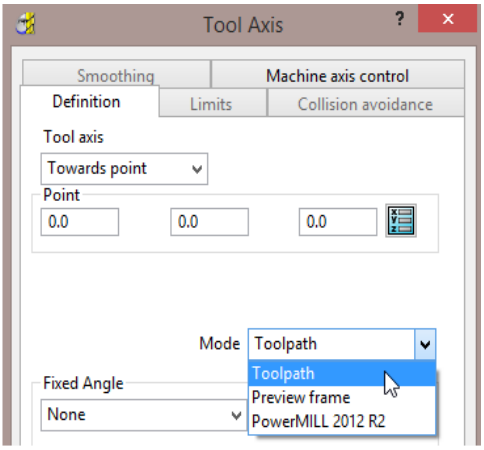
By default the **Tool Axis** alignment in **PowerMILL** is set to **Vertical** for **3-Axis** applications and other options will only be available to users with a multiaxis licence.

*Some strategies only support multiaxis **Tool Axis** alignments when operating 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 **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.

f **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.

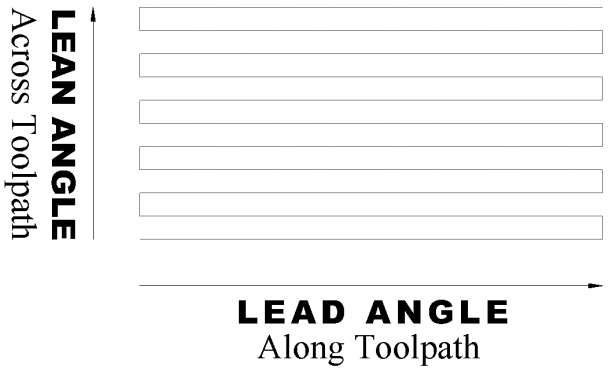
g **Undercut angle** – This measures the **Lead** angle from the projection direction corresponding to the specified undercut angle in a **Parametric Spiral** strategy.

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

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

Lead/Lean - Example1

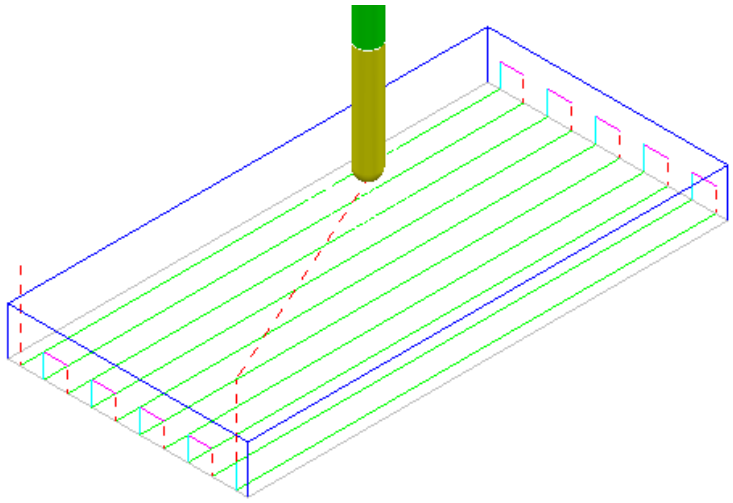
Lead allows the tool to be aligned to a specified angle ***along*** the ***toolpath direction*** and **Lean** a specified angle ***across*** the ***toolpath direction***. If both angles are zero the **tool** will be aligned along the direction that a **toolpath** is projected onto the model during creation. 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**.

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

….\Five_Axis\ToolAlignments\PlaneAlignments_Start



3 **Save Project As**:-

….\COURSEWORK\PowerMILL_Projects\PlaneAlignments_Demo

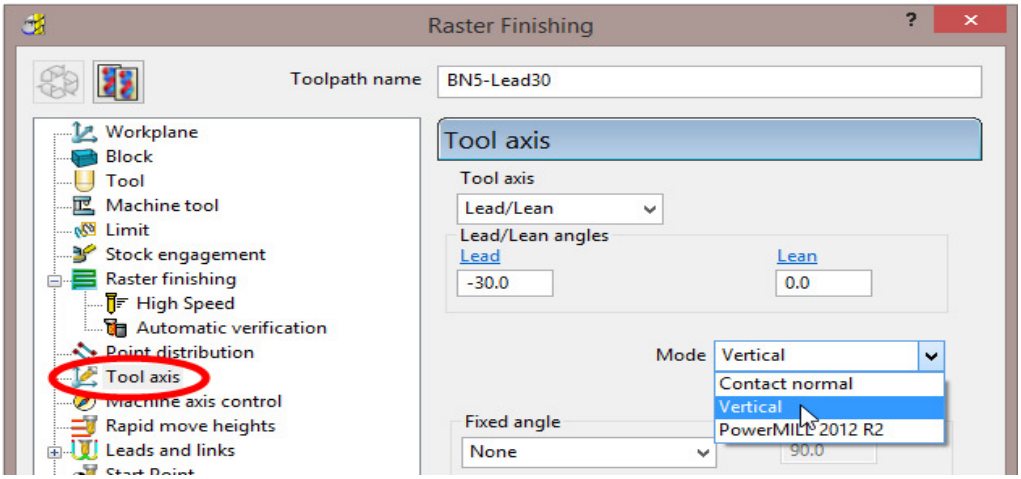
4 **Simulate** the **Toolpath BN5-Vertical**.

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.

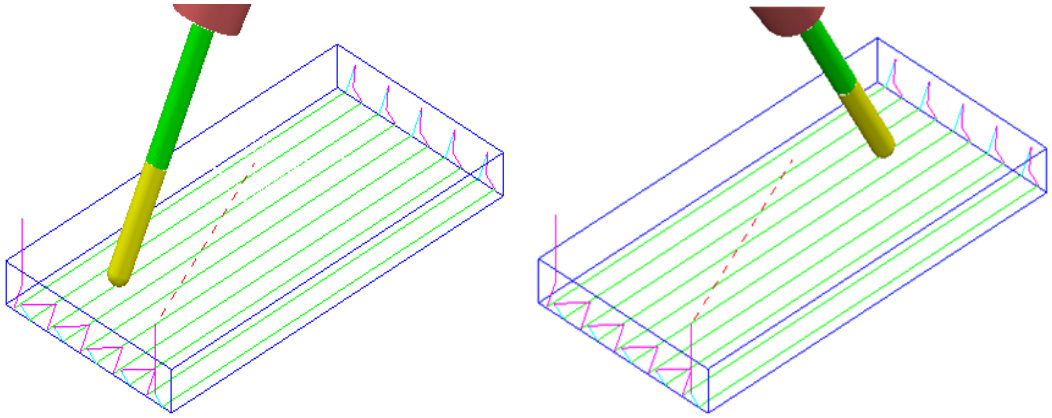
6 **Make a Copy**  of the **Toolpath** and rename it as **BN5-Lead30**.

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**).



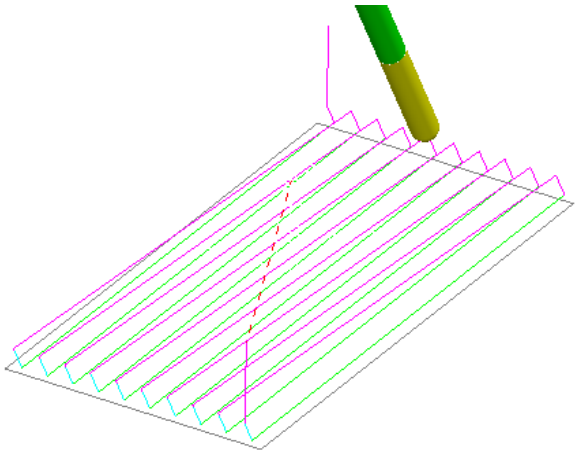
A **Raster** toolpath has been created with the **Tool axis** direction set to **Lead** at **-30** degrees relative to the ***vertical axis*** (**Z**).

*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.



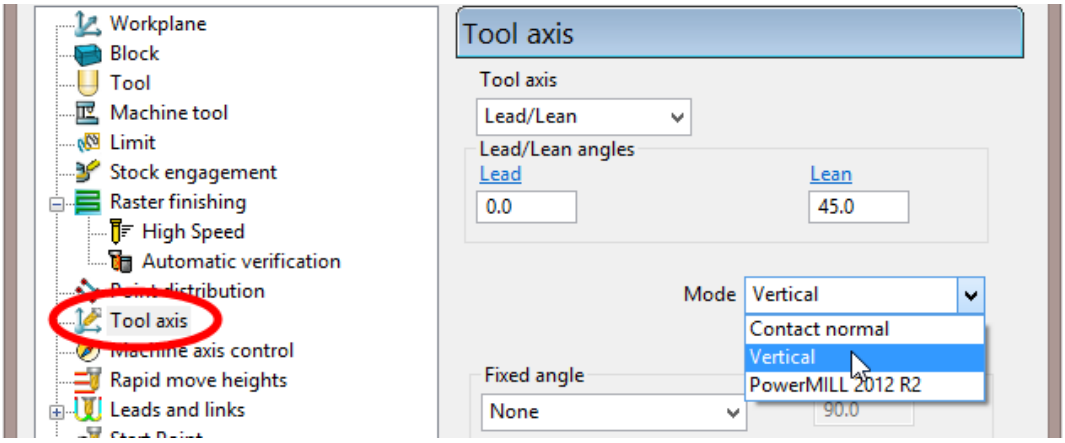
*With the **Style** set to **One Way** the **Tool Axis** direction remains fixed.*

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**.

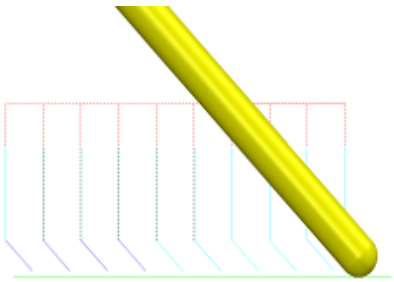
15 Select the **Tool Axis** icon to open the **Tool Axis Direction** dialog.

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.

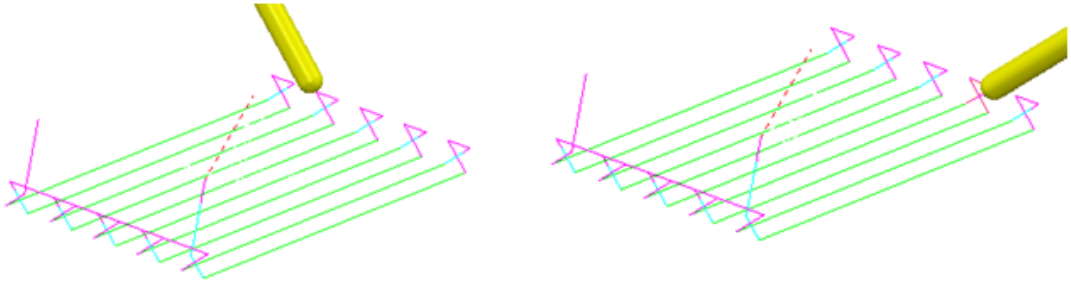
18 **Simulate** the **Toolpath**.



View from left -X

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

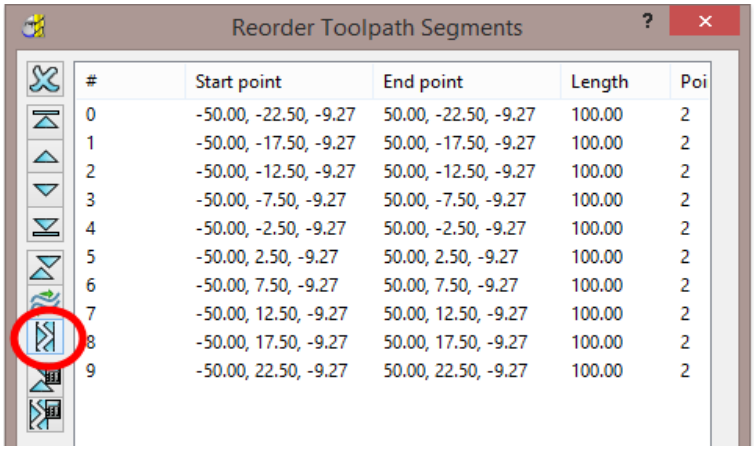
If a **Two Way** had been directly applied in the strategy dialog, the tool axis would lean in the opposite direction across alternate tool tracks.

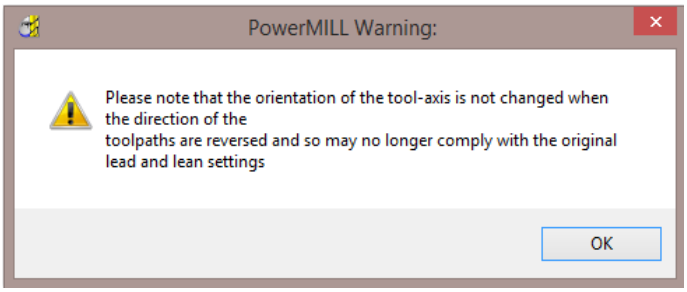


It is possible to create a **Two Way** strategy with a constant **Lean** direction by retrospectively editing a ***one way*** - **Toolpath**.

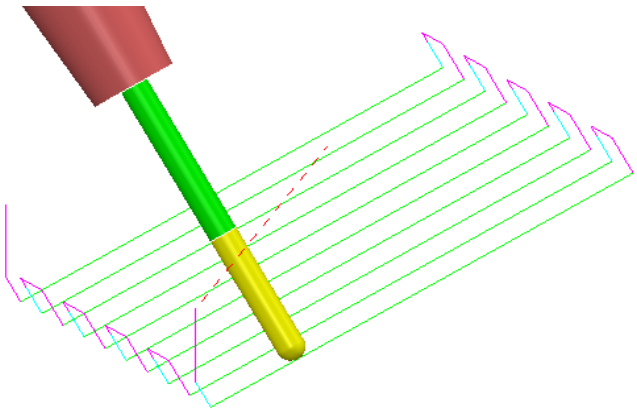
19 ***Right-click*** on the (***One Way***) ***Toolpath*** **BN5-Lean45** and from the local menu, select **Edit** > **Reorder** to open the **Reorder Toolpath Segments** dialog.

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





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



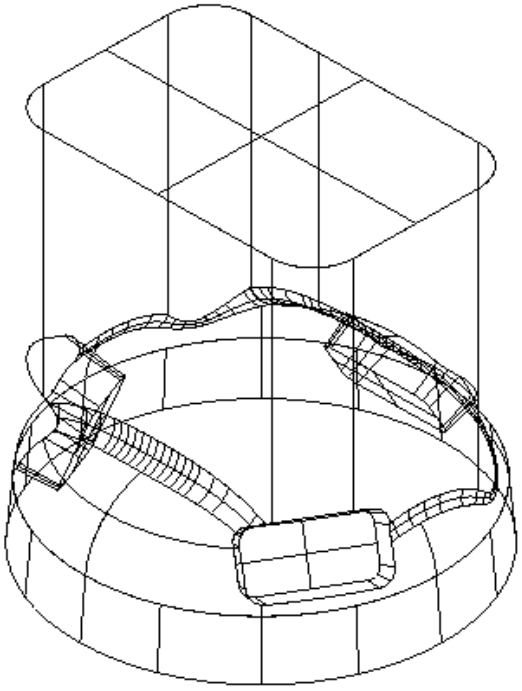
22 **Save Project** to update:-

….\COURSEWORK\PowerMILL_Projects\PlaneAlignments_Demo

Lead/Lean - Example2

1. Select **File** - **Delete all** and **Tools** - **Reset Forms**.
2. **Open** the **Project** (saved earlier during Chapter 1) from:

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



3. Define a **Dia 15mm Ball Nose** cutter named **BN15**.
4. **Activate** the **Workplane** - **ztop175_A**.
5. 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 -70 Xmax-52.5 Ymin -50 Ymax 50

6. Open the **Rapid Move Heights** dialog and select **Calculate**, then **Accept**.
7. In the **Start and End Point** dialog in ***Start Point*** set **First Point Safe** and in ***End Point*** set **Last Point Safe**.
8. Set **Leads\Links**  as follows:

   Z heights: ***Skim*** **15**, ***Plunge*** **5**

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

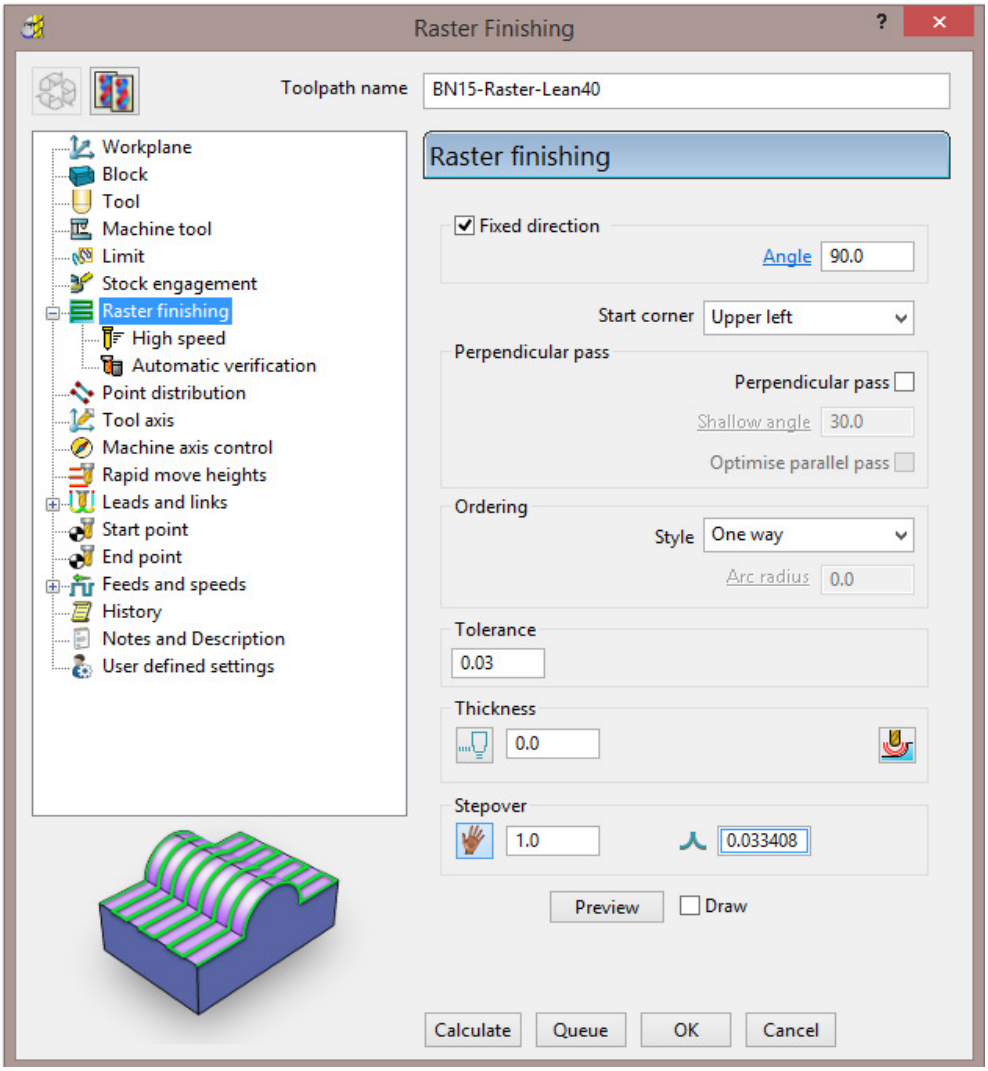
   Links: **Short\Long** ***Skim*** and ***Default*** **Incremental**

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

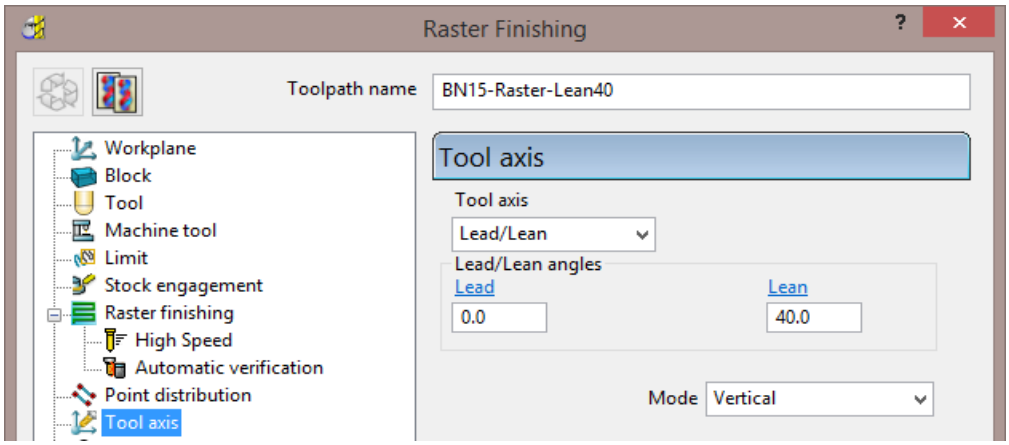
An area of the component will be machined using **Raster Finishing**. The **Tool Axis** will be aligned to a fixed ***Lean angle*** of **40** degrees relative to the **Vertical** axis.

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

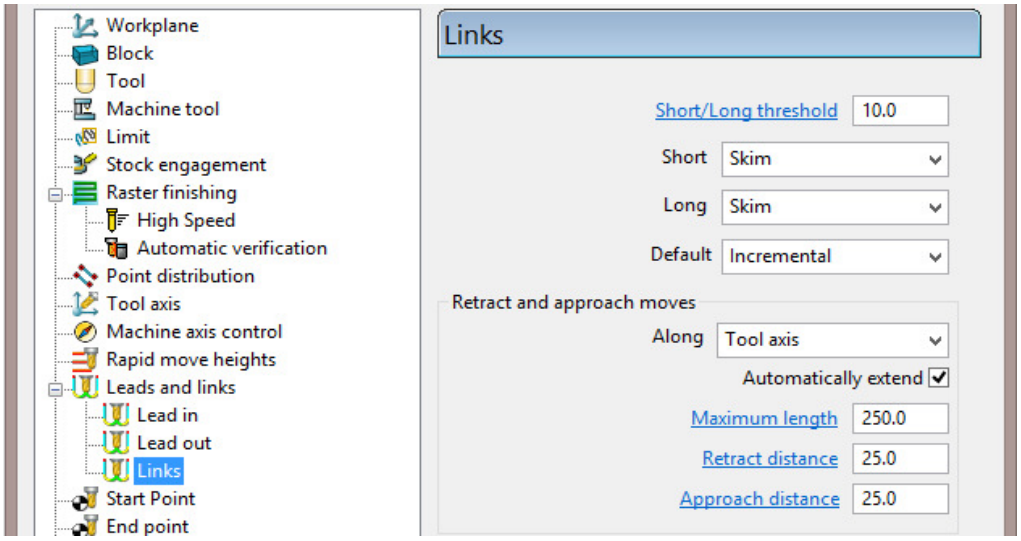
11 Select the **Raster Finishing** dialog and enter data exactly as shown below:



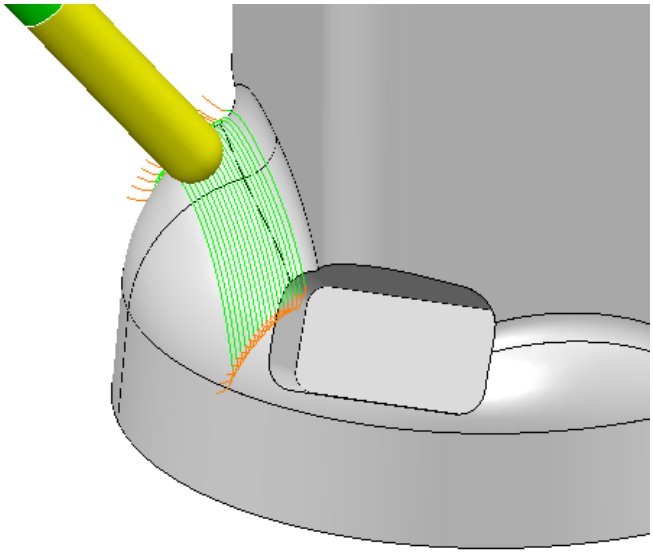
12 Select the **Tool axis** page and enter data into the dialog exactly as shown in the following illustration.



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



14 **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**.

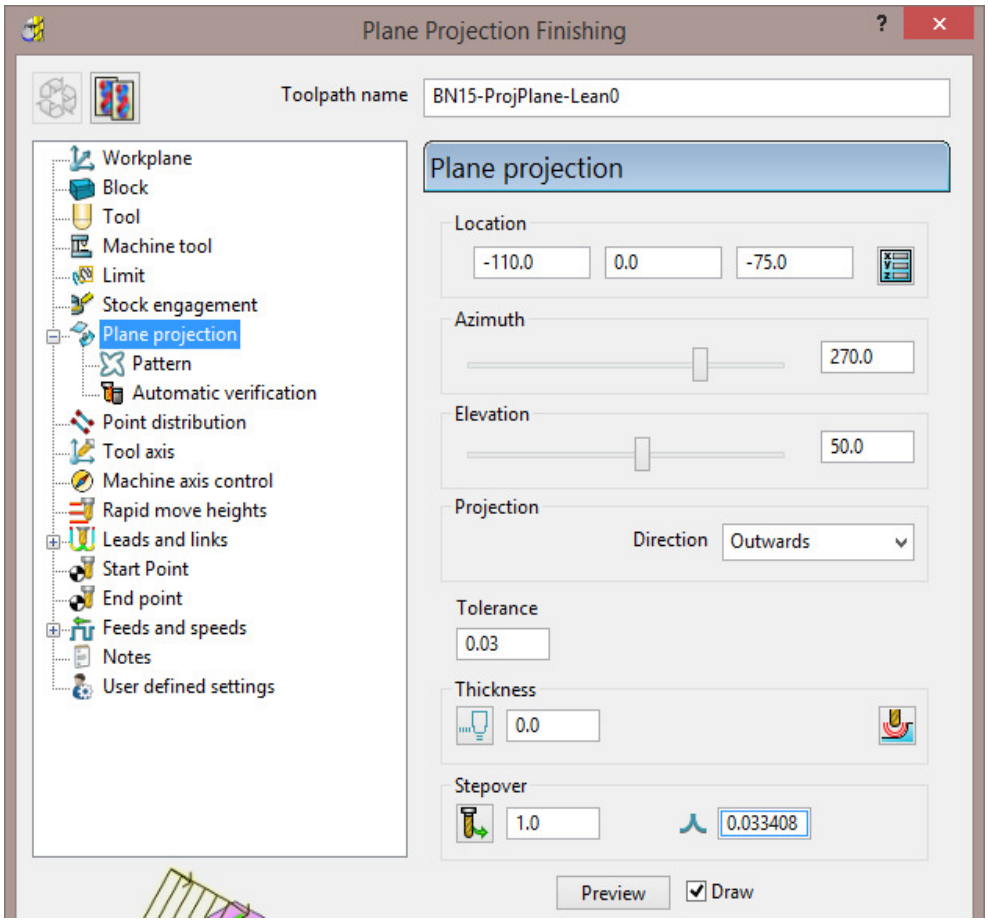
To compare the effect of applying a different machining strategy, the same area of the component will be machined using **Projection Plane Finishing**. This time the **Tool Axis** will be aligned using a **Lean** of **0** degrees relative to the ***projection direction*** of the strategy.

15 **Calculate** a **Block**, *Defined by* - **Cylinder** using ***Type* - Model**, to the ***Coordinate System*** - ***Named Workplane*** - **ztop175_A**.

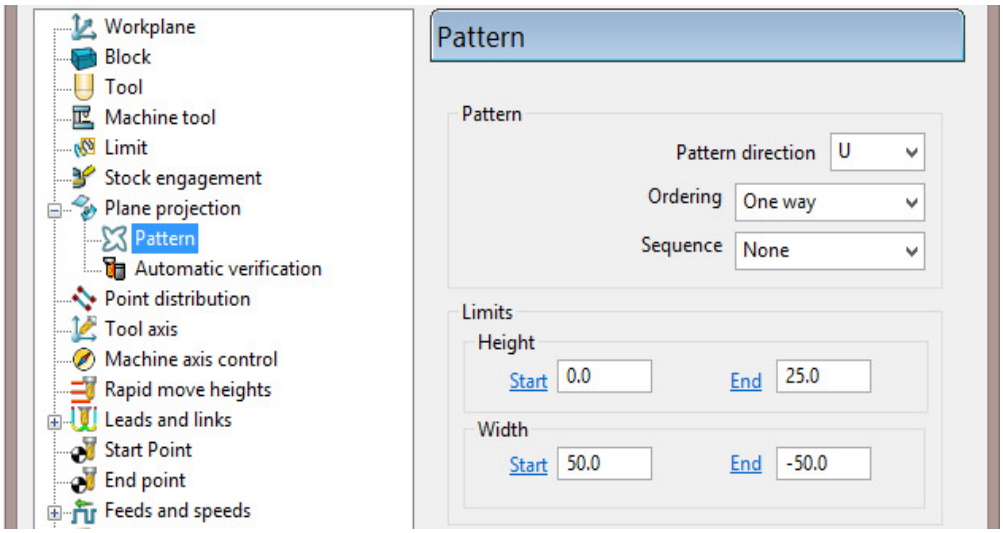
16 Select the **Toolpath Strategies** icon and in the ***Strategy Selector*** dialog select the **Finishing** tab.

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

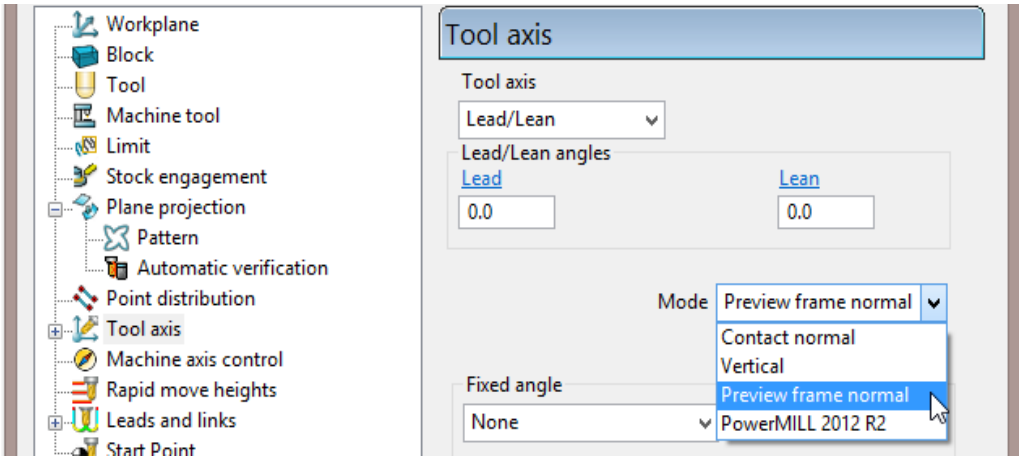
18 Enter data into the following 3 pages of the **Plane Projection Finishing** dialog, exactly as shown below:



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

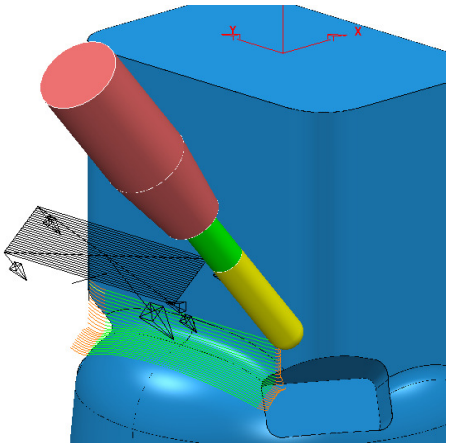


19 Set **Pattern** direction to **U**



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

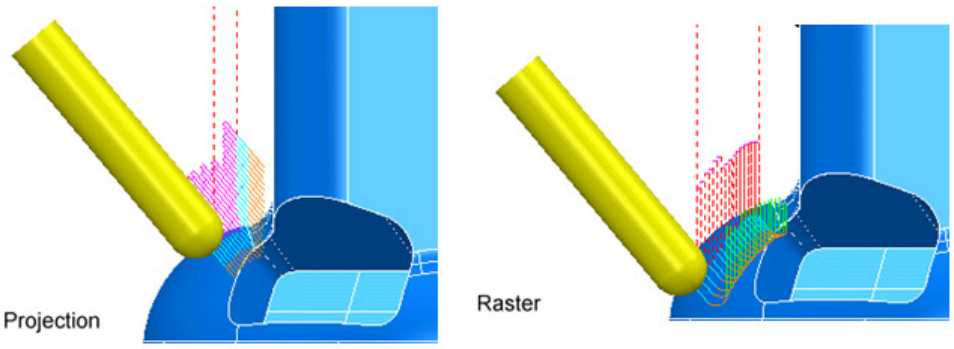
21 **Simulate** the **toolpath** and observe the associated tool alignment.



*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.*

22 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 suitable **Lean** value of **40** being applied to the **Raster** strategy.*

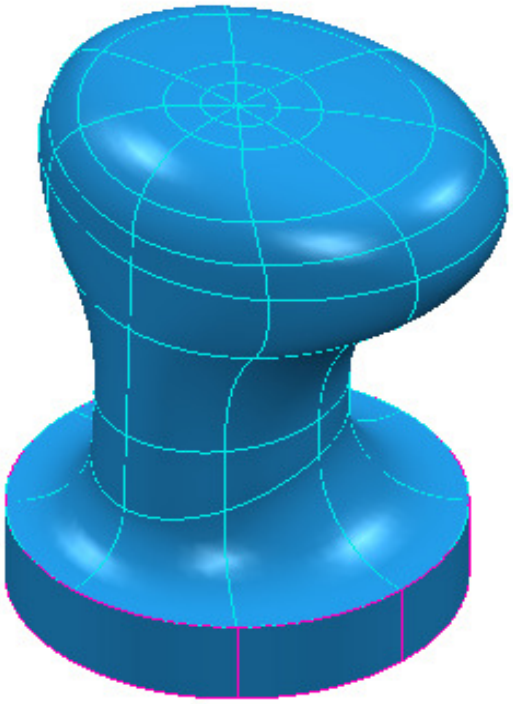


23 Select **File** - **Save Project**.

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



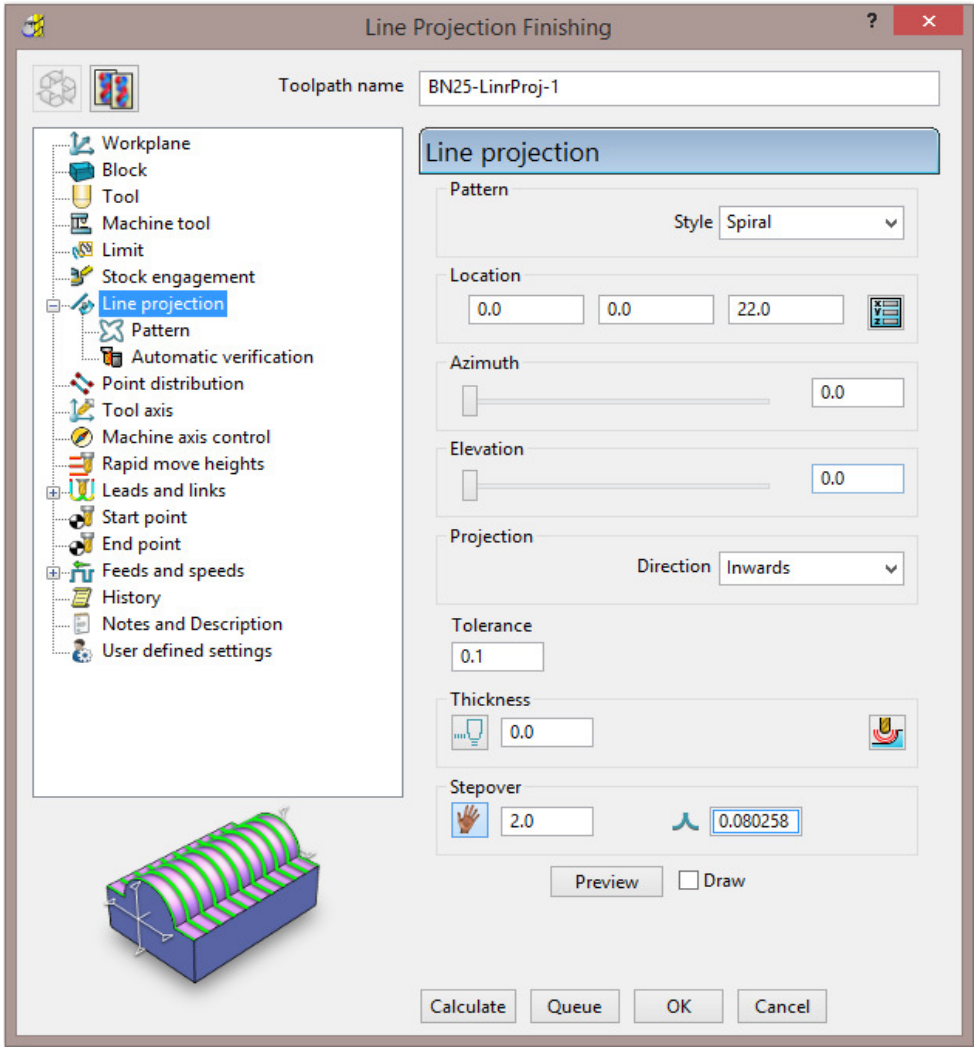
3. **Save** the **Project As**:-

   ...\COURSEWORK\PowerMILL_Projects\Joint-EX1

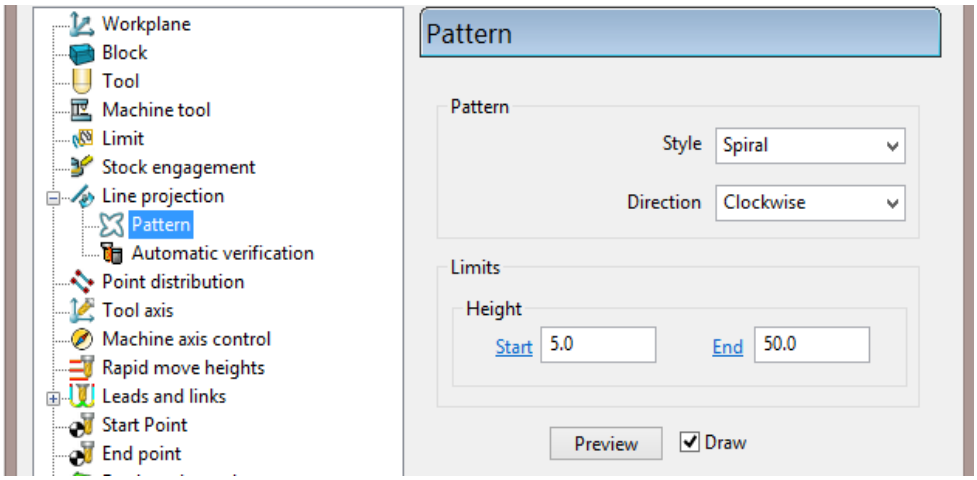
4. Create the material **Block** using ***Defined by*** - **Box** to the **Model** dimensions and then ***expand*** it by **15** in **X** and **Y** only.
5. Create a **Dia 25 Ball Nosed** cutter (**BN25**).
6. Select **Calculate** in the **Rapid Move Heights** dialog.
7. Use the **Start and End Point** default settings.
8. Modify the settings in **Leads and Links**  as follows:
   - For ***Z heights*** use **Skim 45** and **Plunge 10**
   - For ***Links*** set both ***Short*** and ***Long*** **- Links** to **Skim**.

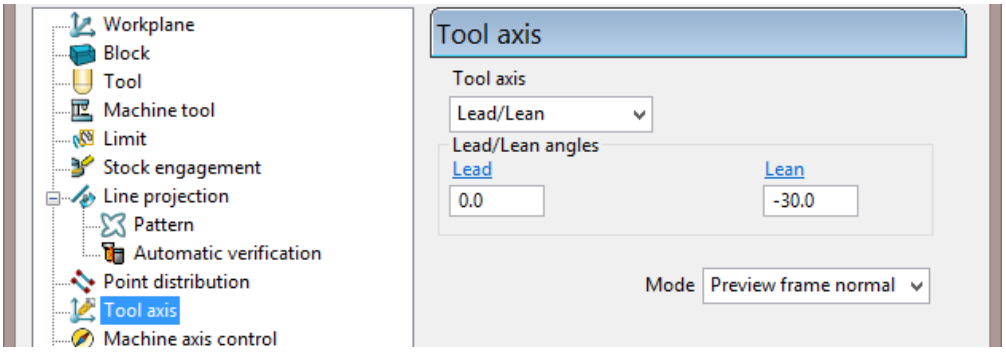
9 Select the **Toolpath Strategies** icon and in the **New Strategies** dialog, select the **Finishing** tab.

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

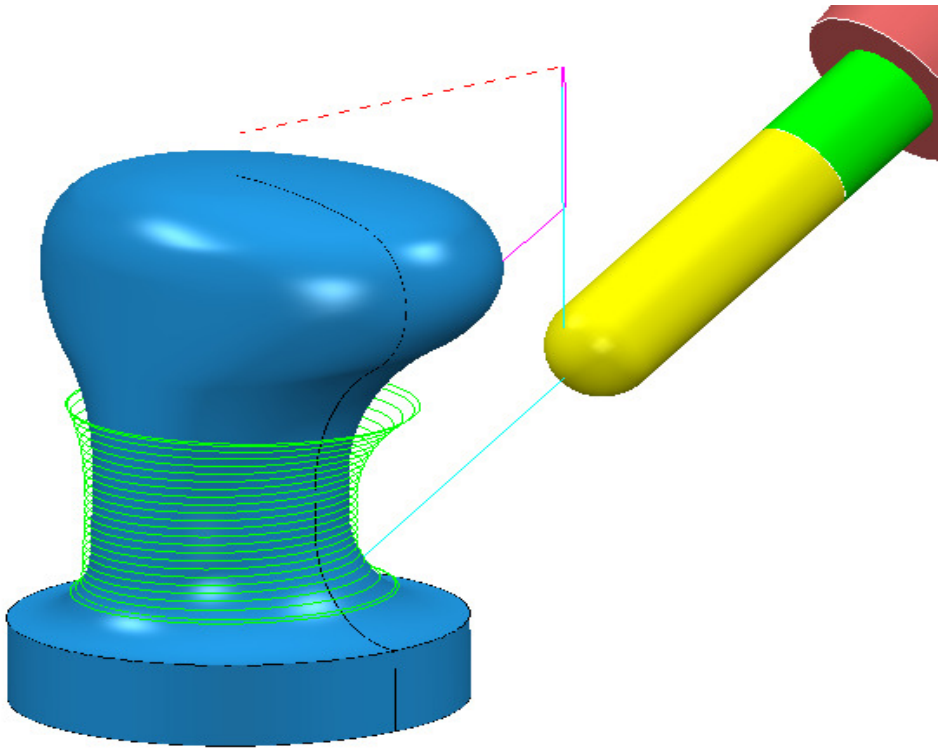


- Set **Azimuth** to **0**
- Set **Elevation** to **0**





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

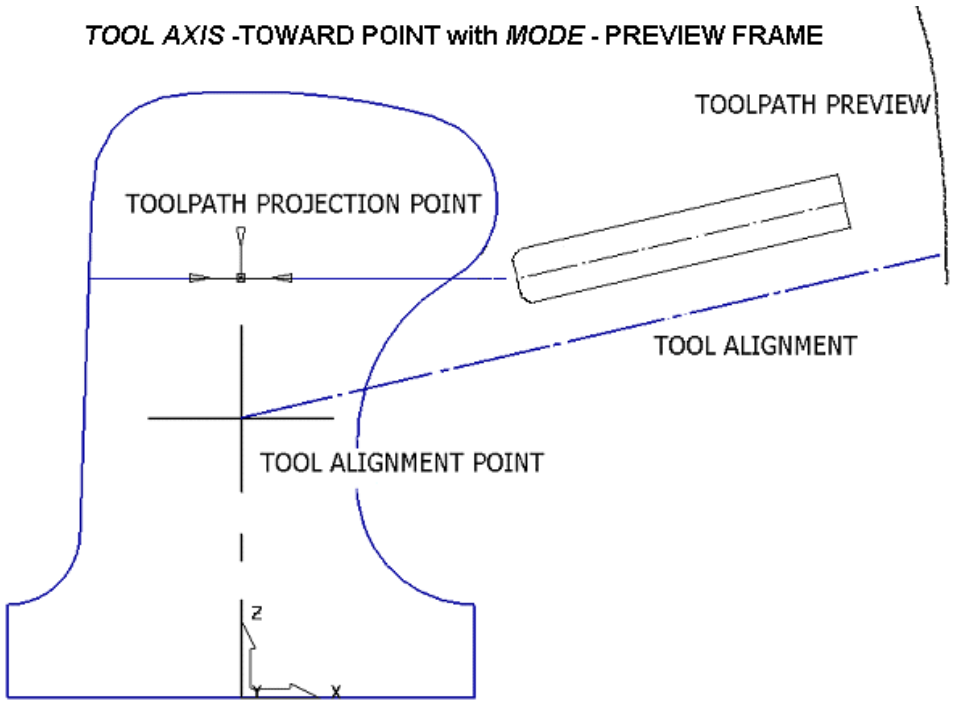


The next section continues with the machining of the upper part of this component.

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

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.



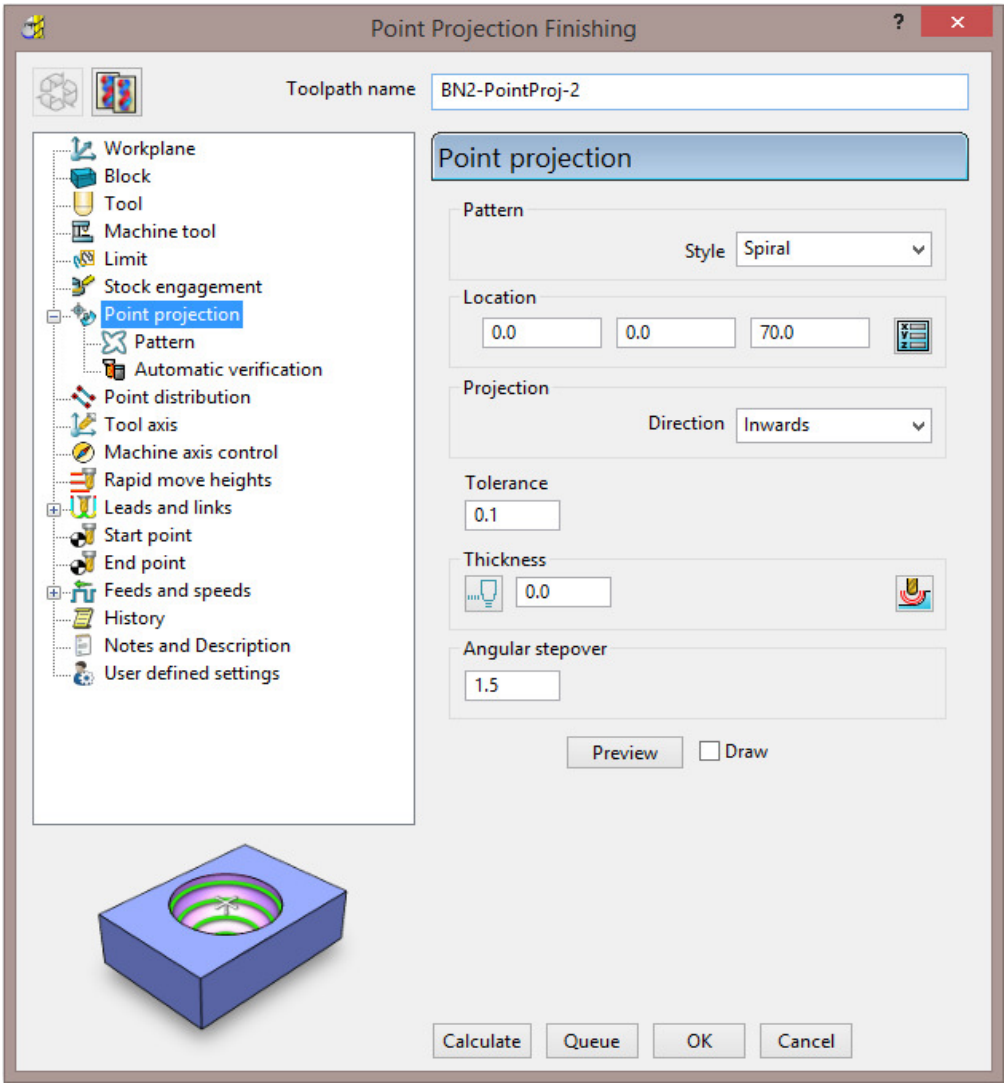
13 Set **Leads/Links** as follows:

- **Z heights**: ***Skim*** **45** ***Plunge*** **10**
- **Lead In/Out**: **Surface normal arc**, ***Angle*** **90** ***Radius*** **6**
- **Extensions**: **Inwards/Outwards**, ***Extended Move***, **Distance 30**
  Links: **Skim**
-

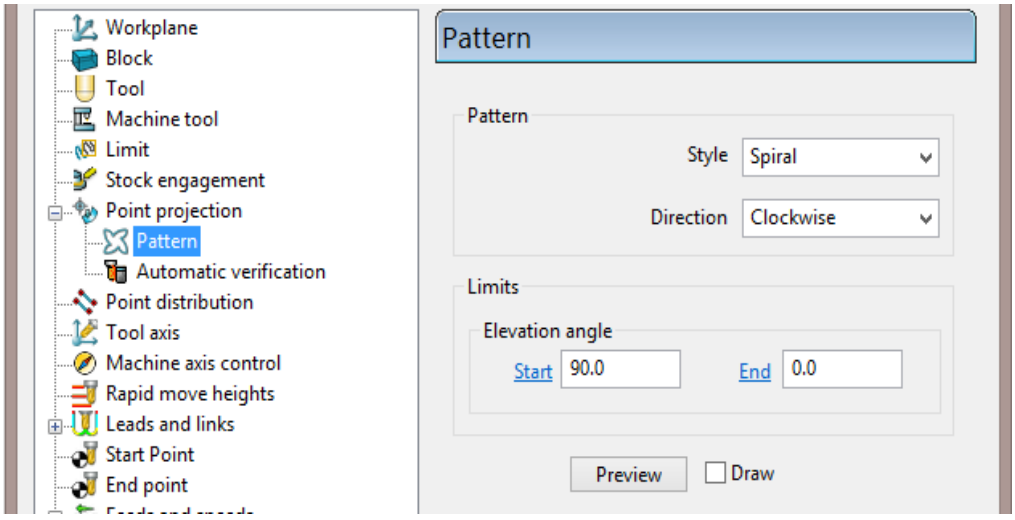
14 Select the **Toolpath Strategies** icon and select the **Finishing** tab located in the new **Strategy Selector** dialog.

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.

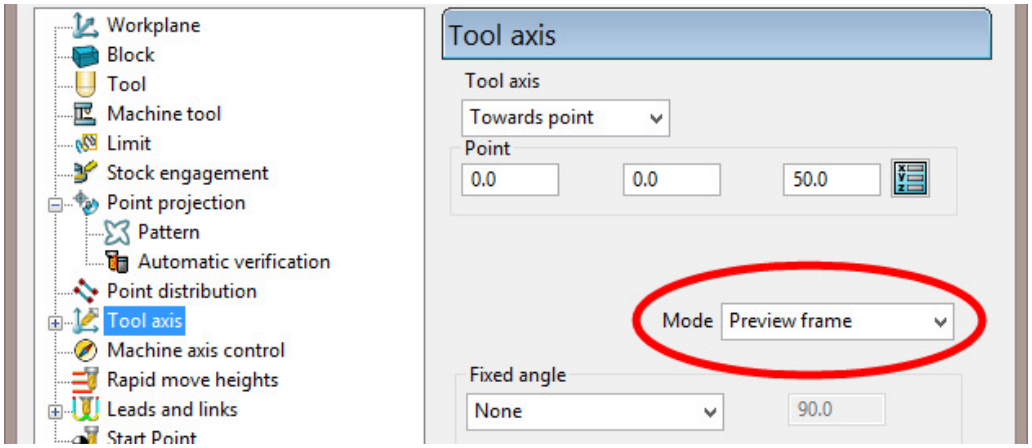


17 Select the **Pattern** page and enter data into the dialog exactly as shown in the following illustration.

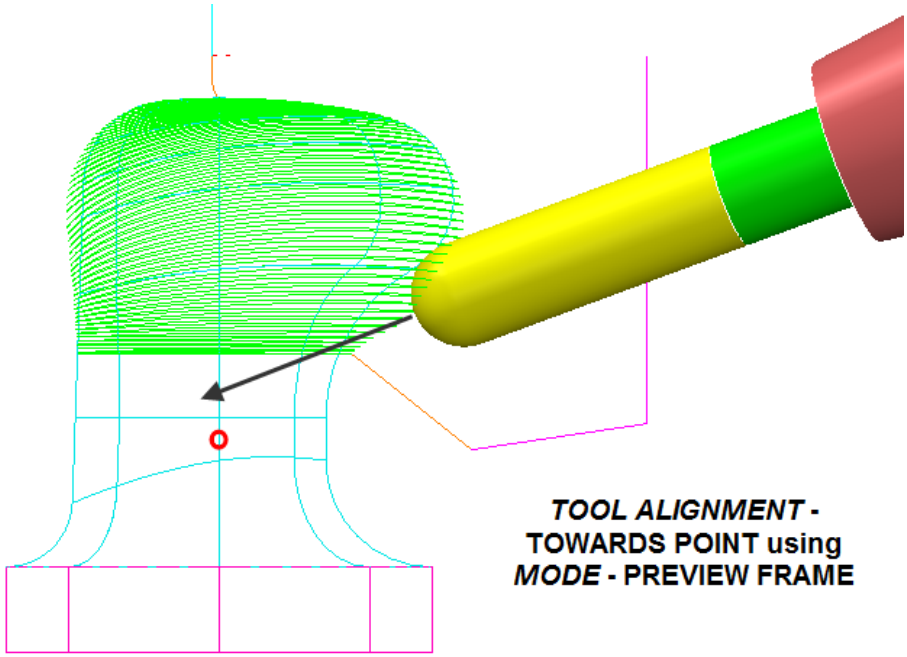


18 Input **Elevation Angle** data as:- **Start** 90 and **End** 0

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.

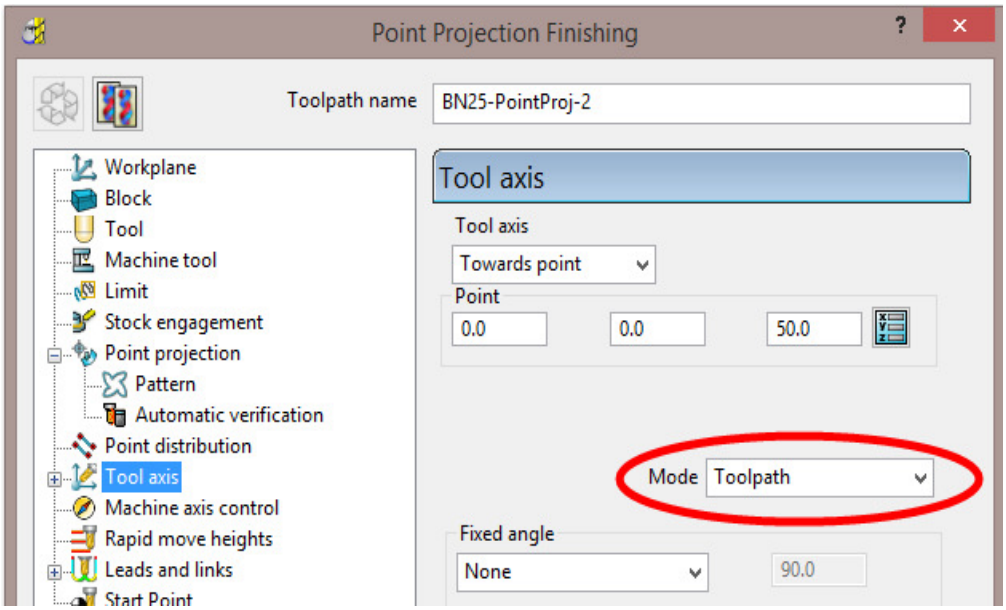


*The **Tool Alignment** point is approximately **20mm** below the **Projection Finishing** focal point to ensure that the spindle axis remains at an elevation angle to the machine tool table during machining hence avoiding collision.*

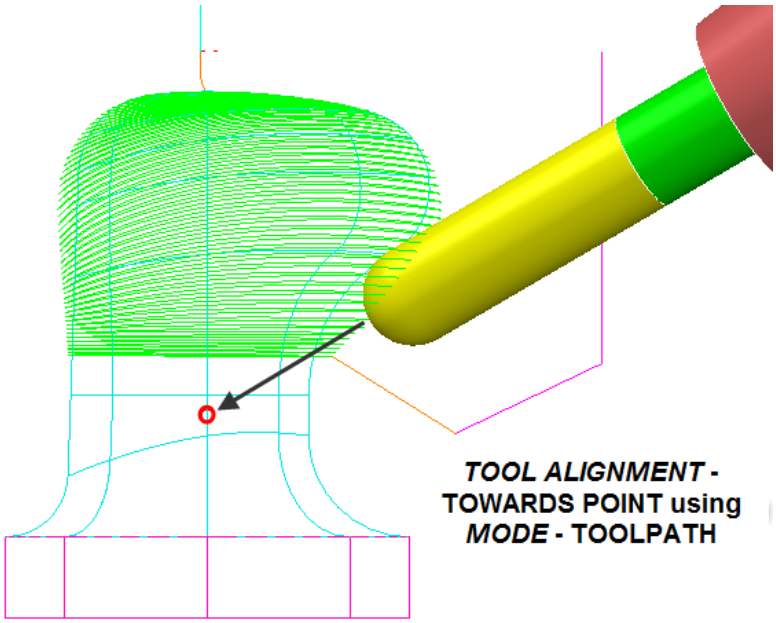
Mode – Preview Plane:- The actual **Tool** alignment occurs on the **Projection Point Finishing - Preview** pattern, after which the tool is then projected on to the model (towards the **strategy - focal point**).

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**.

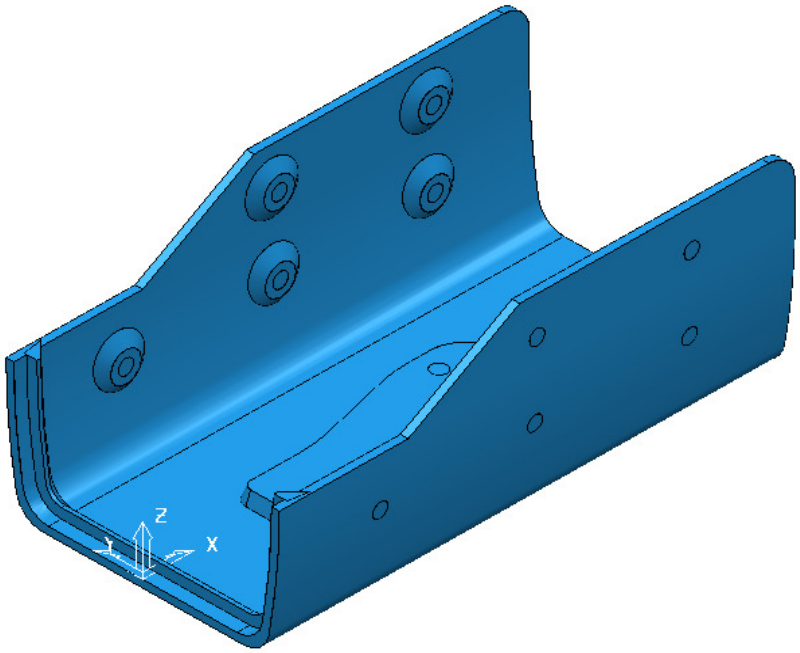
Toward\From Line

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** the **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.

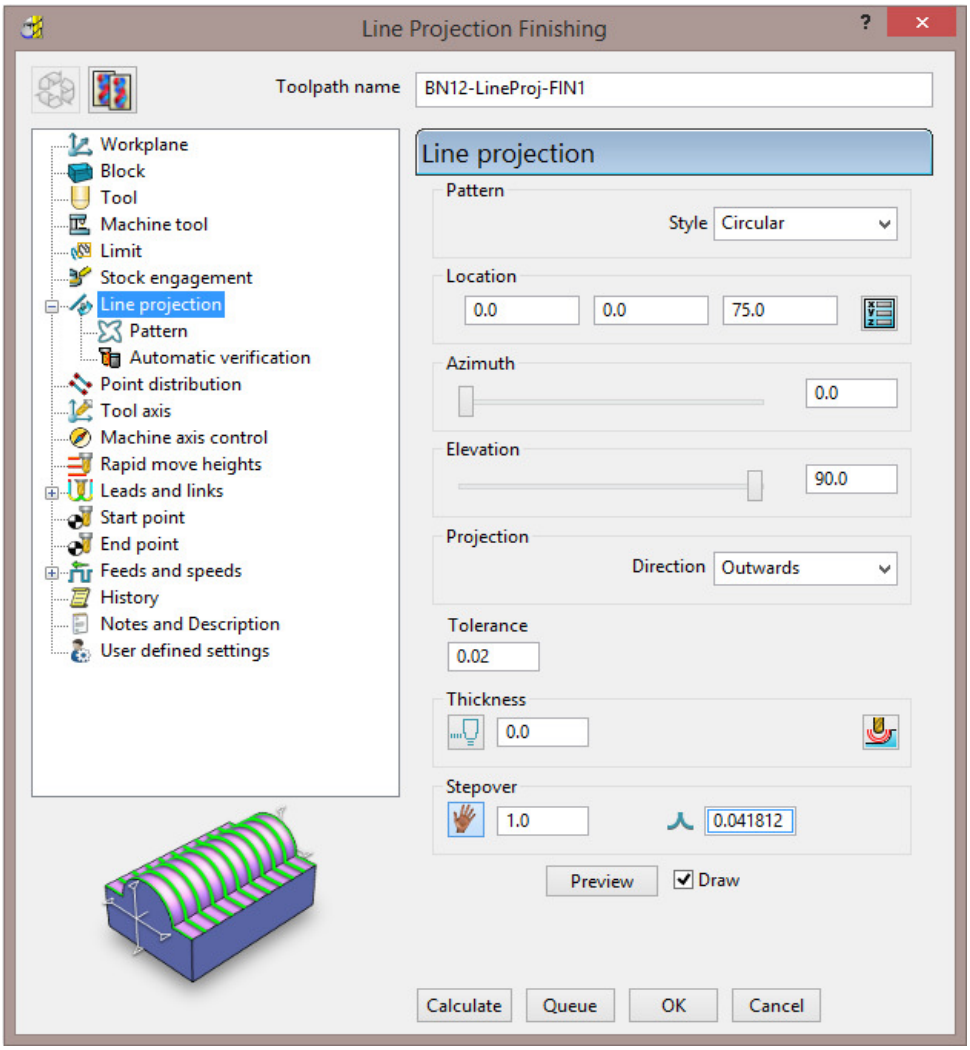
7 Set **Leads\Links**  as follows:

- **Z heights**: ***Skim*** **5**, ***Plunge*** **5**
- **Lead In/Out**: ***1st Choice*** – **Surface normal arc**, ***Angle*** **45**, ***Radius*** **5.0**
- **Links**: **Threshold 3**, ***Short*** – **Circular Arc**, and ***Long*** – **Skim**.

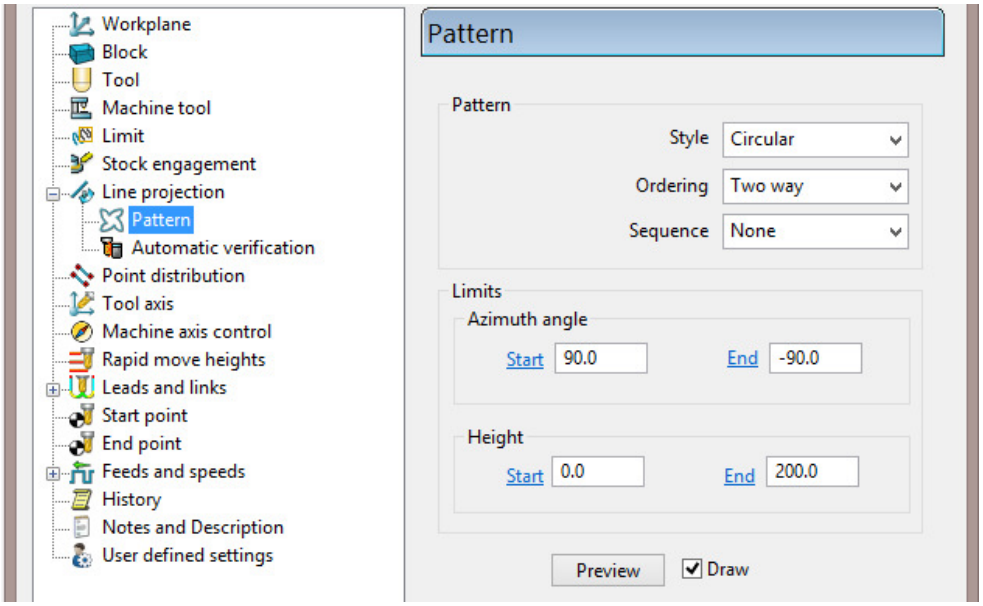
8 Select the **Toolpath Strategies** icon and in the new **Strategy Selector** dialog, select the **Finishing** tab.

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.

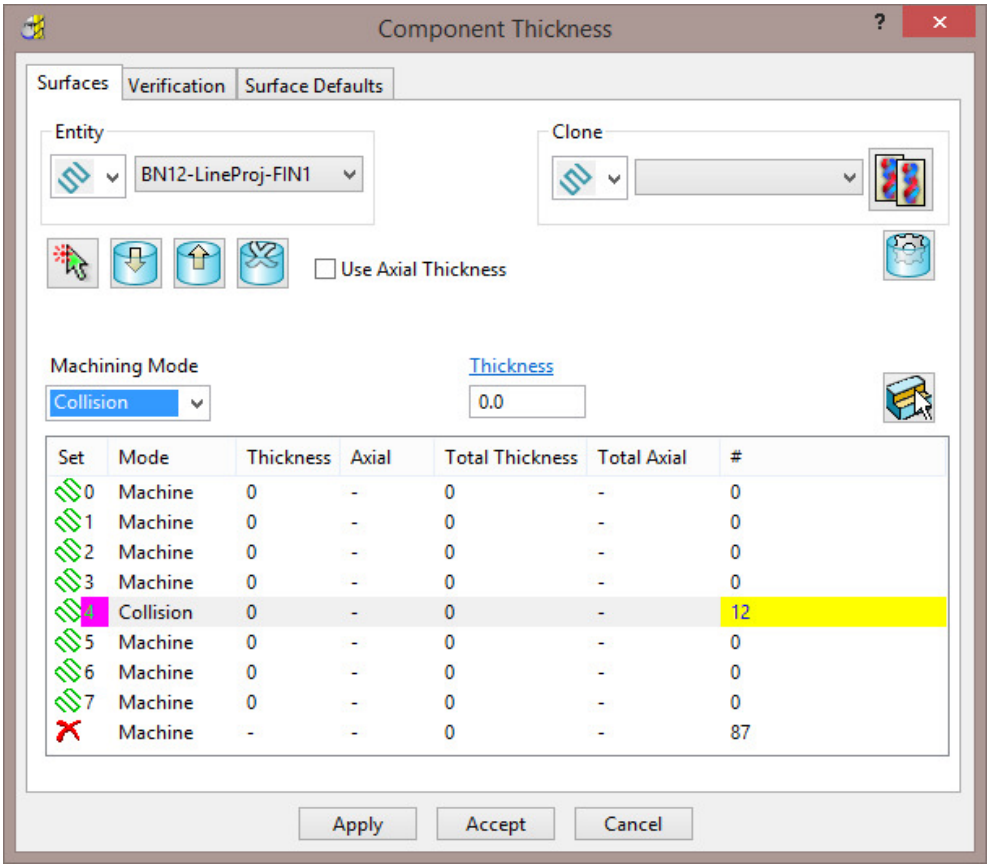


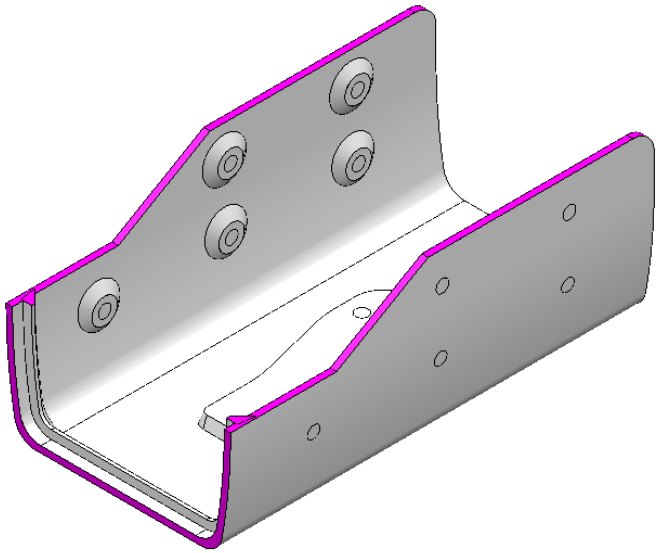
- Set ***Style*** to **Circular**
- Input ***Location*** as **0 0 75**
- Set **Azimuth** to **0**
- Set **Elevation** to **90**
- Set ***Direction*** **Outwards**
- Use ***Tolerance*** **0.01**
- Use ***Stepover*** **1**



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

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



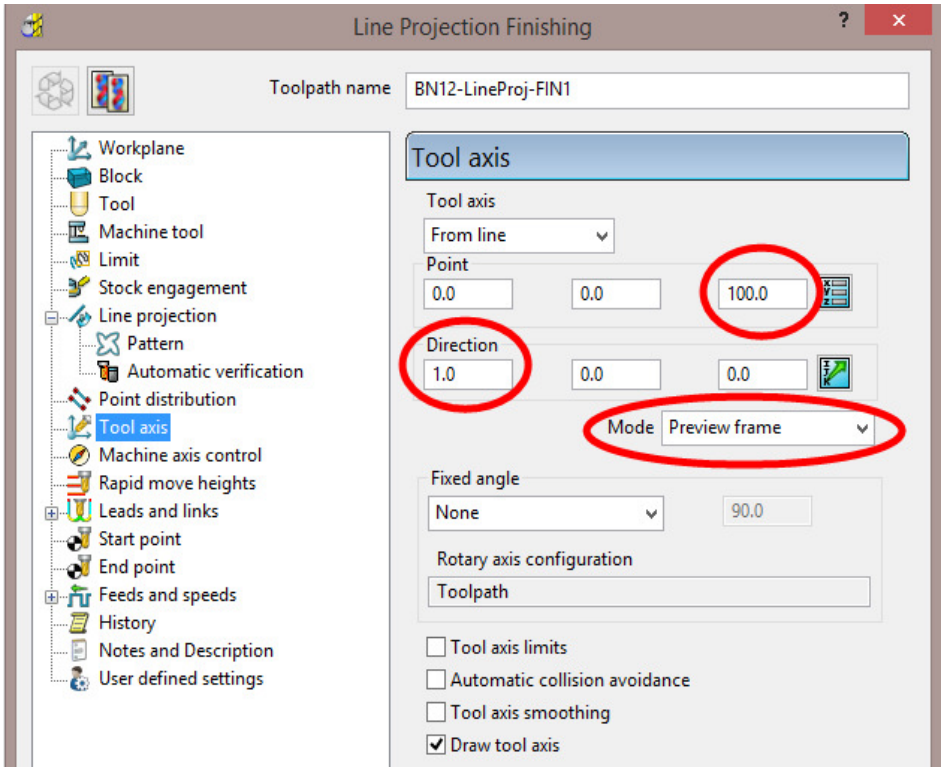


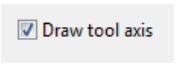
13 Select all of the edge **surfaces** on the **Model** and acquire  them to the selected row in the **Component Thickness** dialog.

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

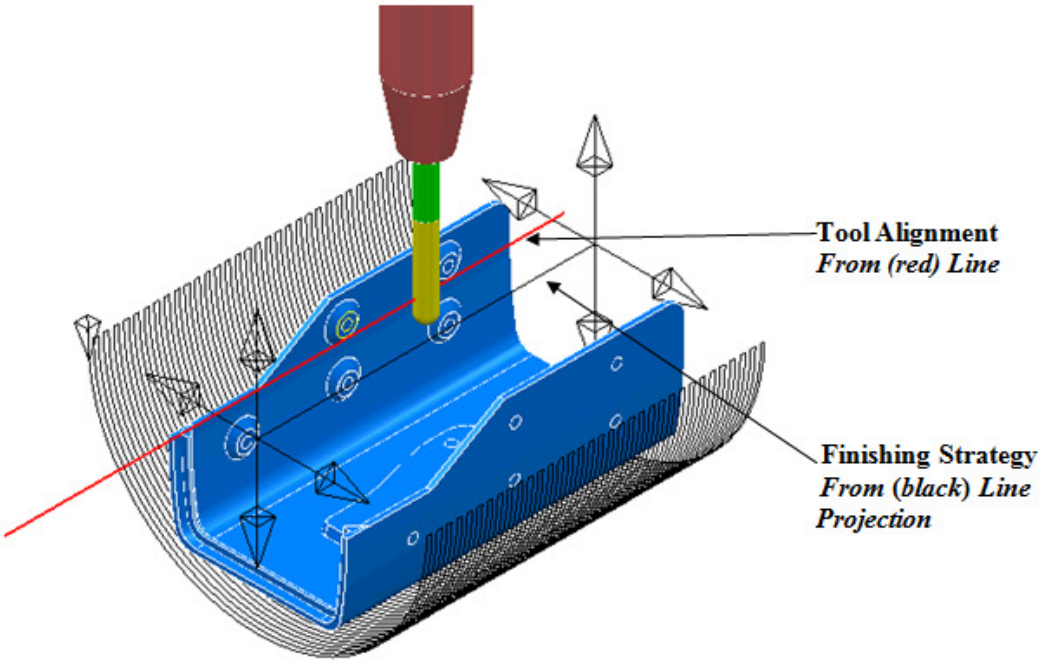
15 **Apply** and **Accept** the **Component Thickness** dialog.

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 

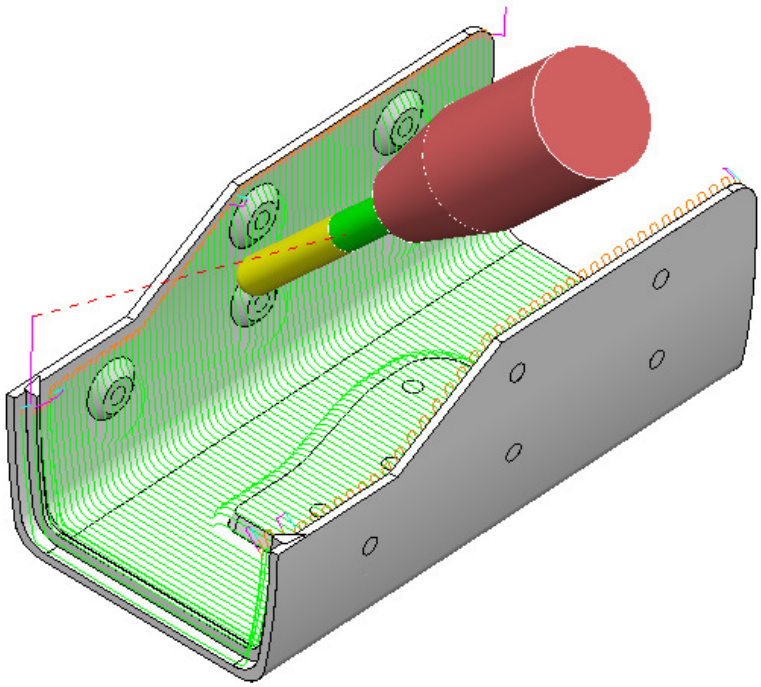
to display the red, **Tool Alignment** line relative to the **Model** (as shown in the following image).



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

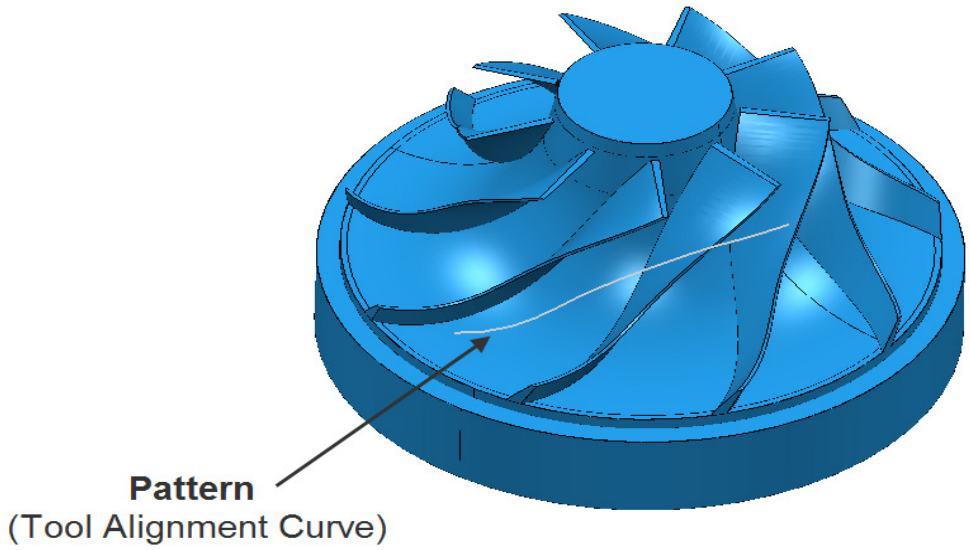
These options allow the ***tool axis alignment*** to be through a user-defined curve (**Pattern**).

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**:

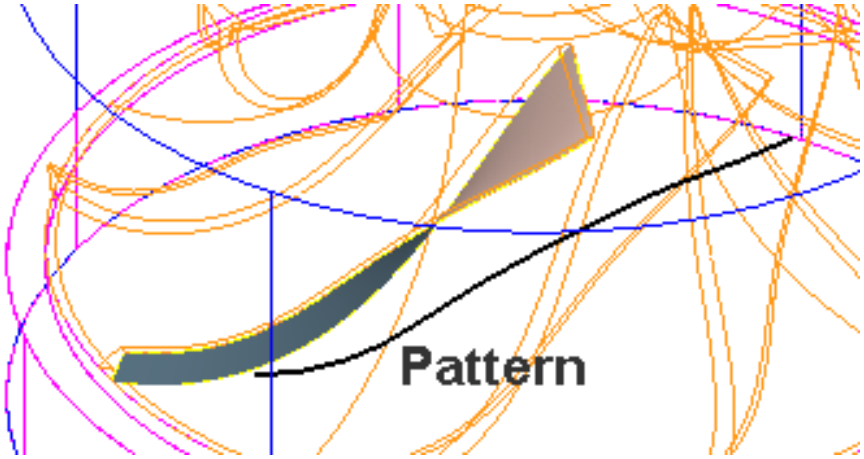
   ....\PowerMILL_Data\five_axis\Impeller\ impeller+Curve.dgk
3. **Save** the **Project** as:

   ....\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.



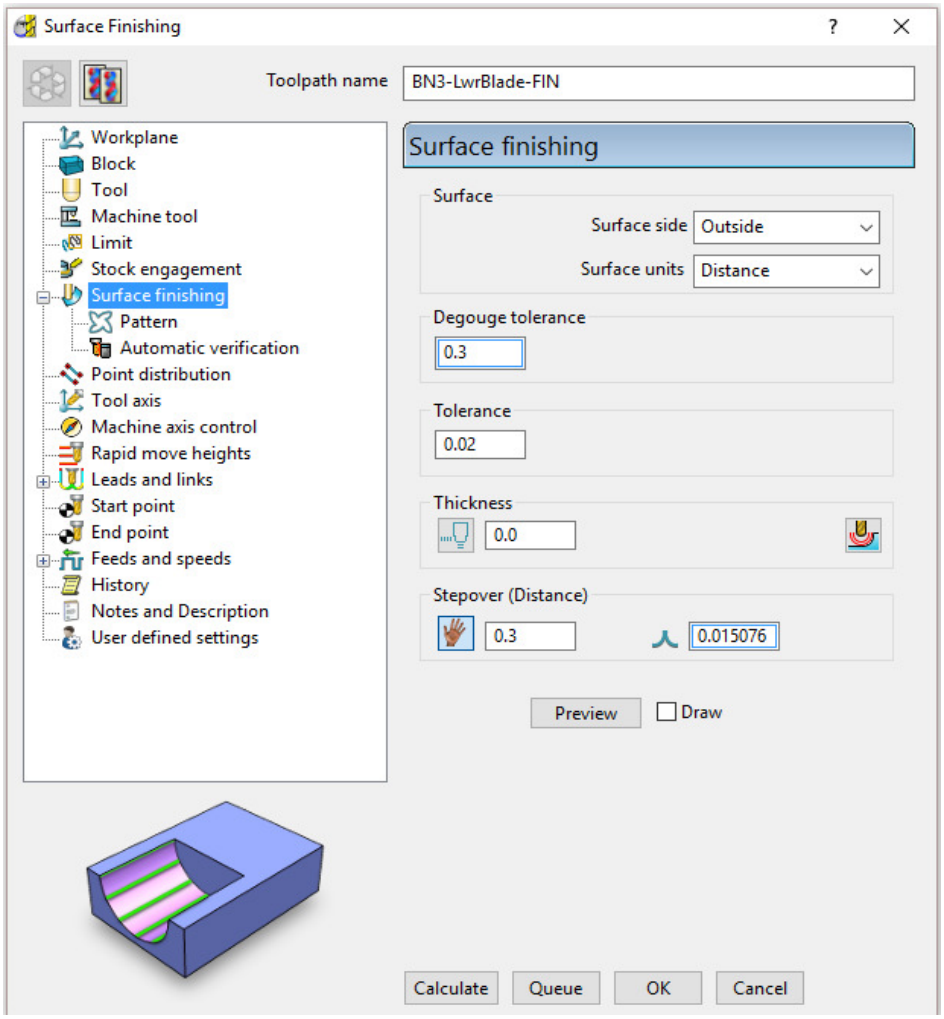
7. Create a **Dia 3 Ball Nosed** cutter (named **BN3-LR**) of **Length** 35 with the following **Shank** and **Holder** dimensions:
   - **Shank** - ***Upper\Lower*** **Dia 3** - ***Length*** **25**
   - **Holder 1** - ***Upper*** **Dia 15** - ***Lower*** **Dia 10** - ***Length*** **50**
   - **Holder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

11 Select the underside face of the blade (nearest to the **Pattern)** to be used in the 1st **Surface Finishing** strategy.

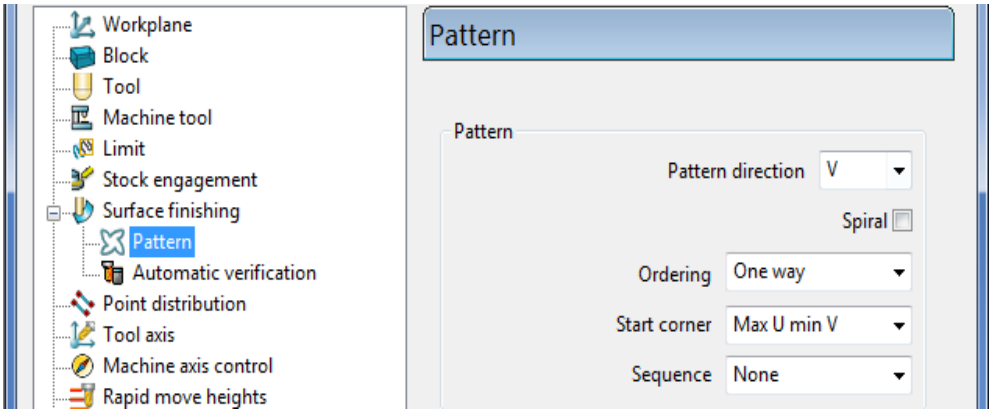


12 Select the **Toolpath Strategies** icon and in the **Strategy Selector** dialog select the **Finishing** tab.

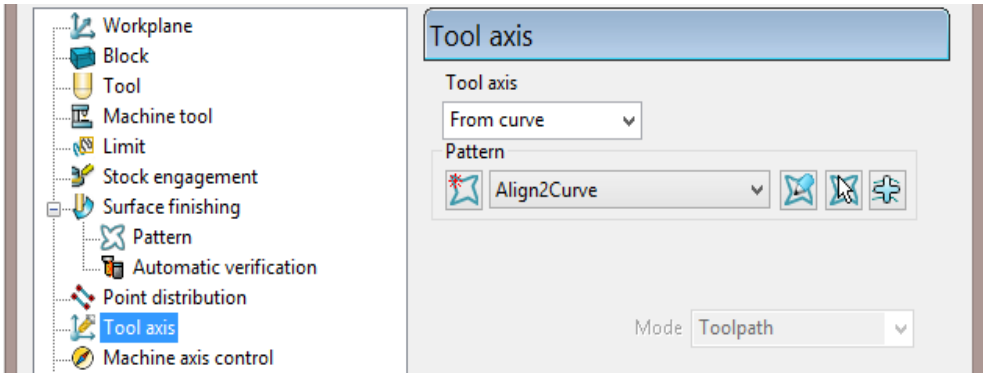
13 Select **Surface Finishing** and fill in the pages of the dialog exactly as shown in the following illustration.



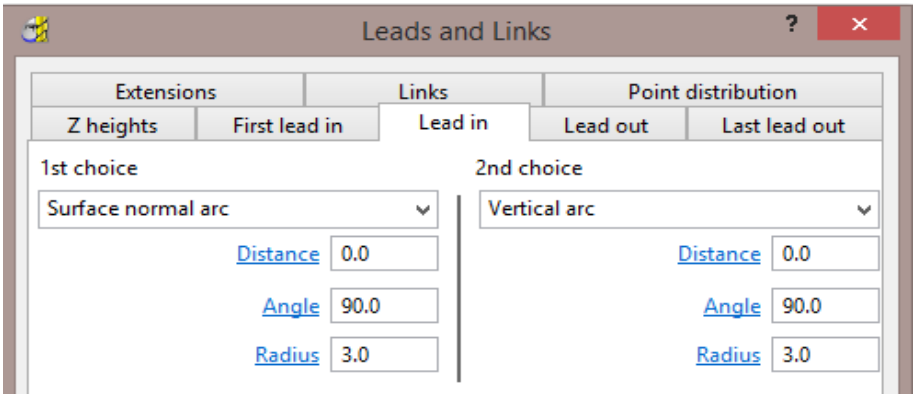
- ***Surface side*** - **Outside**



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

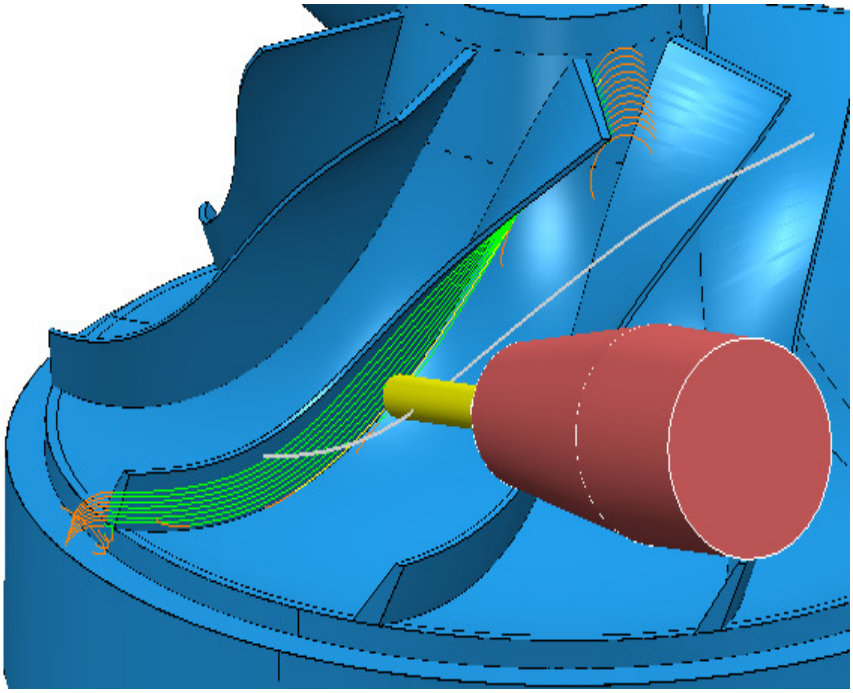


14 For both **Lead in** and **Lead out** use the following for ***1st*** and ***2nd*** **choice**.



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

16 Close the **Surface Finishing** dialog.

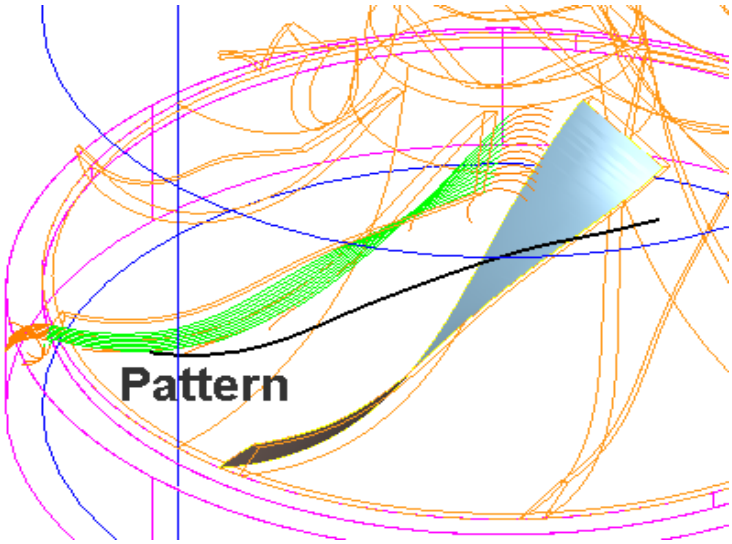


The **Tool Axis** is always aligned through the specified **Pattern** (Curve) while performing the **Surface Finishing** strategy.

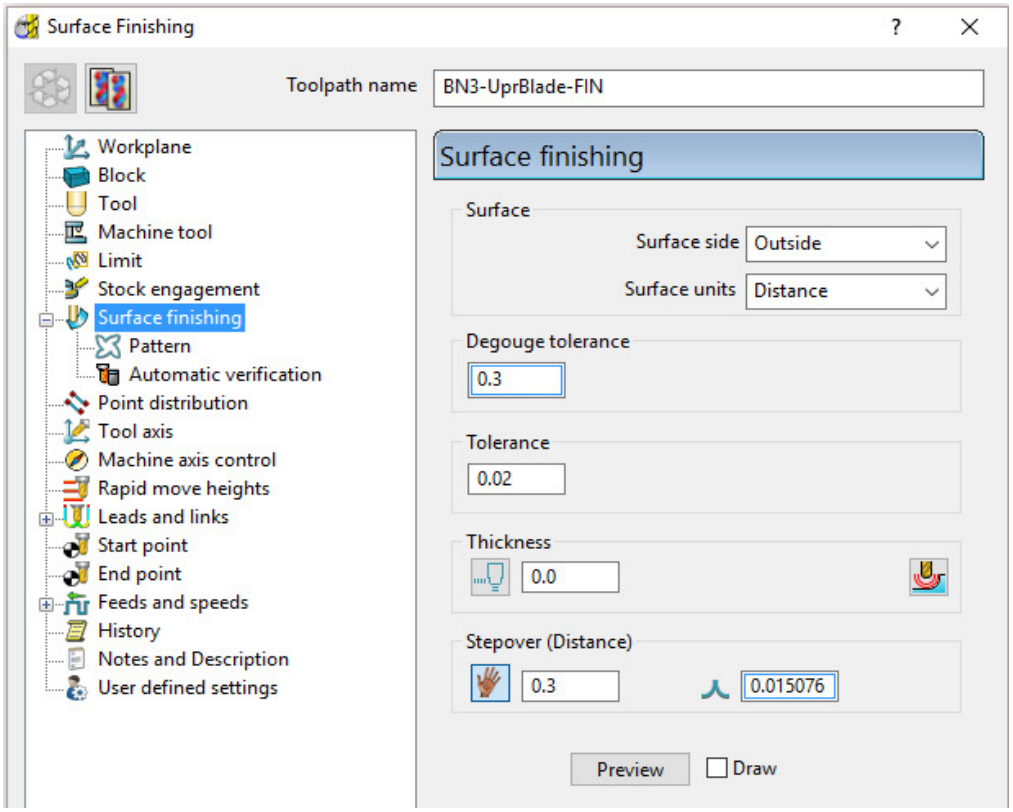
17 Right-click on the existing **Surface Finishing** strategy and from the local menu, select **Settings** to re-open the dialog.

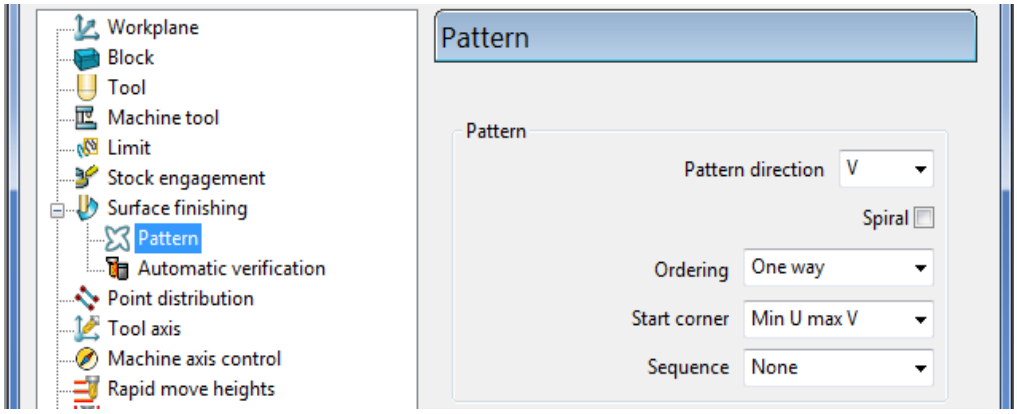
18 In the dialog, select **Make a copy of the Toolpath**  .

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**.

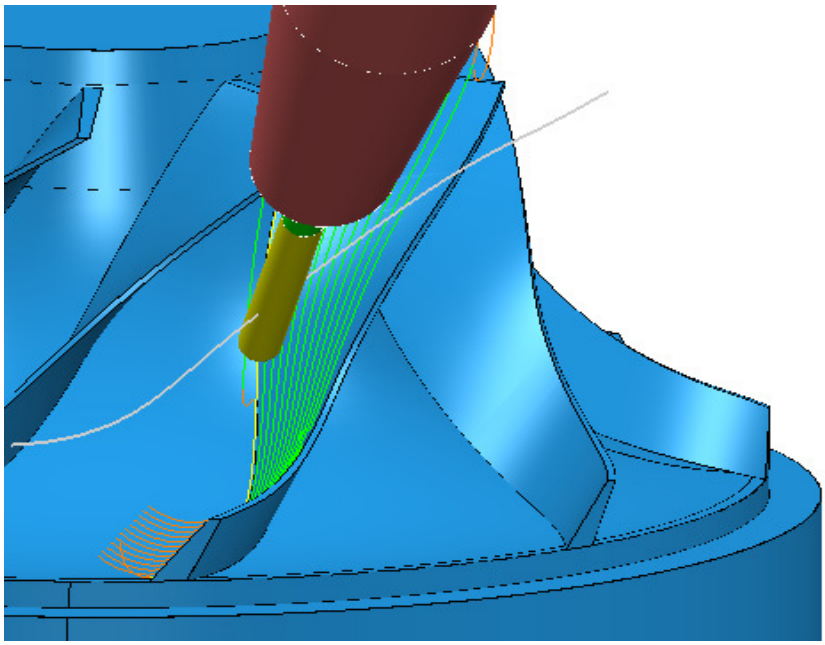




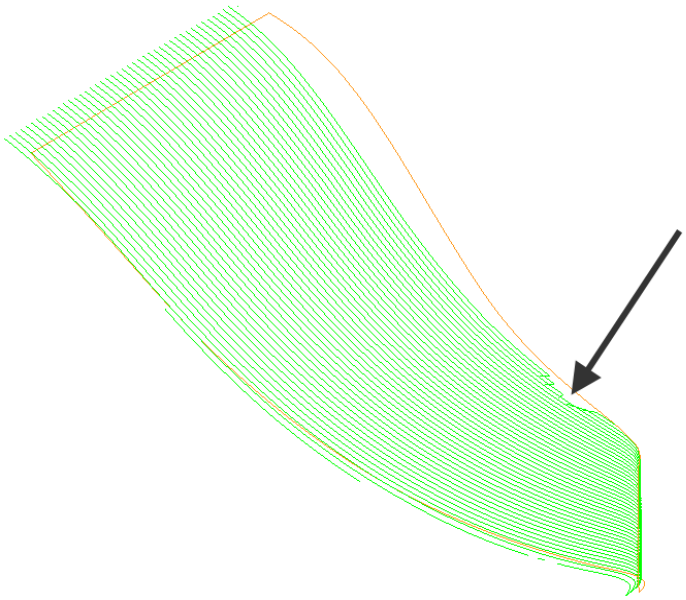
- ***Pattern direction* - V**
- ***Start corner*** - **Min U max V**

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

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



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

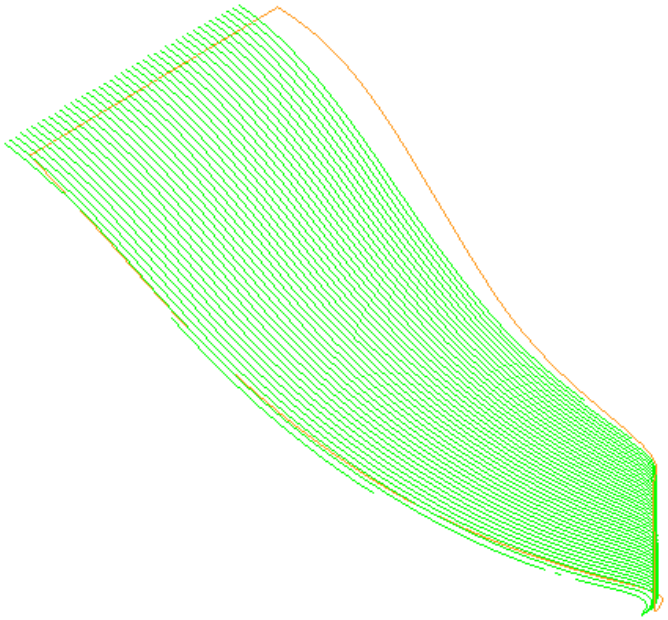


A small area on the latest toolpath is fragmenting along the top edge as arrowed above. This is caused by one of the other blades getting in the way of the projection process at this point. In this instance an easy fix can be applied by using a copy of the existing **Pattern** and moving it upwards by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



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**.

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**.

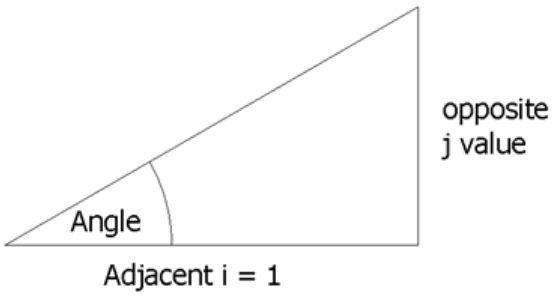
Angle to Vector conversion table

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

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

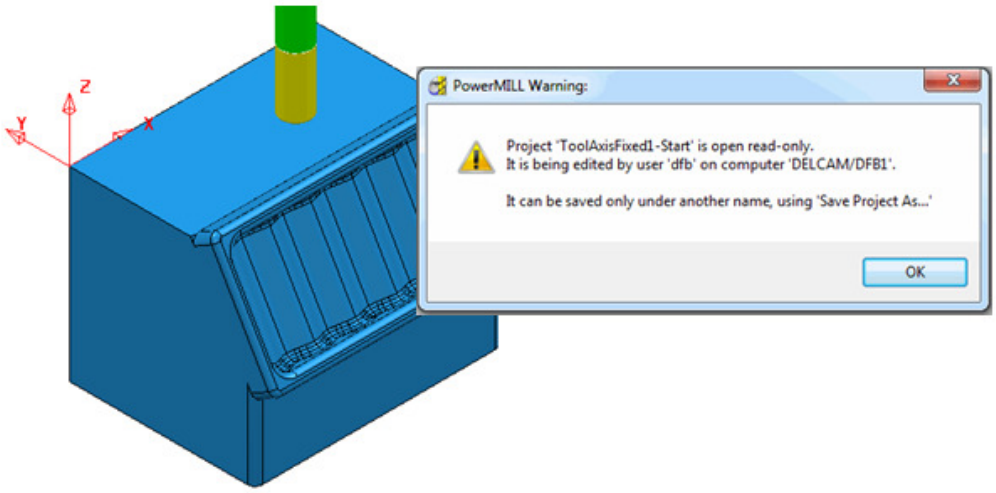


The ***Tool Alignment*** is set to a **Fixed Direction** relative to the currently active **Workplane** by inputting suitable values as an ***I J K vector*** along the **Tool Axis** (towards the spindle). Although the correct values for a ***vector*** may require the user to exercise their trigonometry 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** the **Project As**:

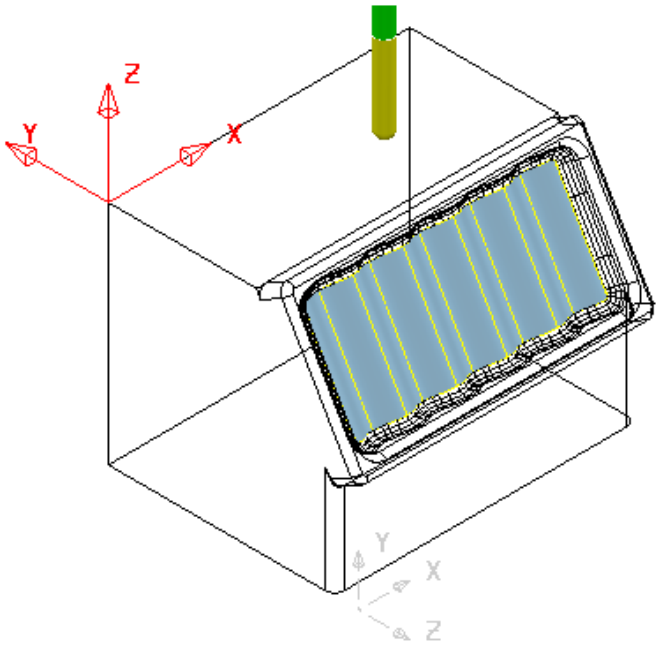
...\COURSEWORK\PowerMILL_Projects\ToolAxisFixed-EX1

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

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

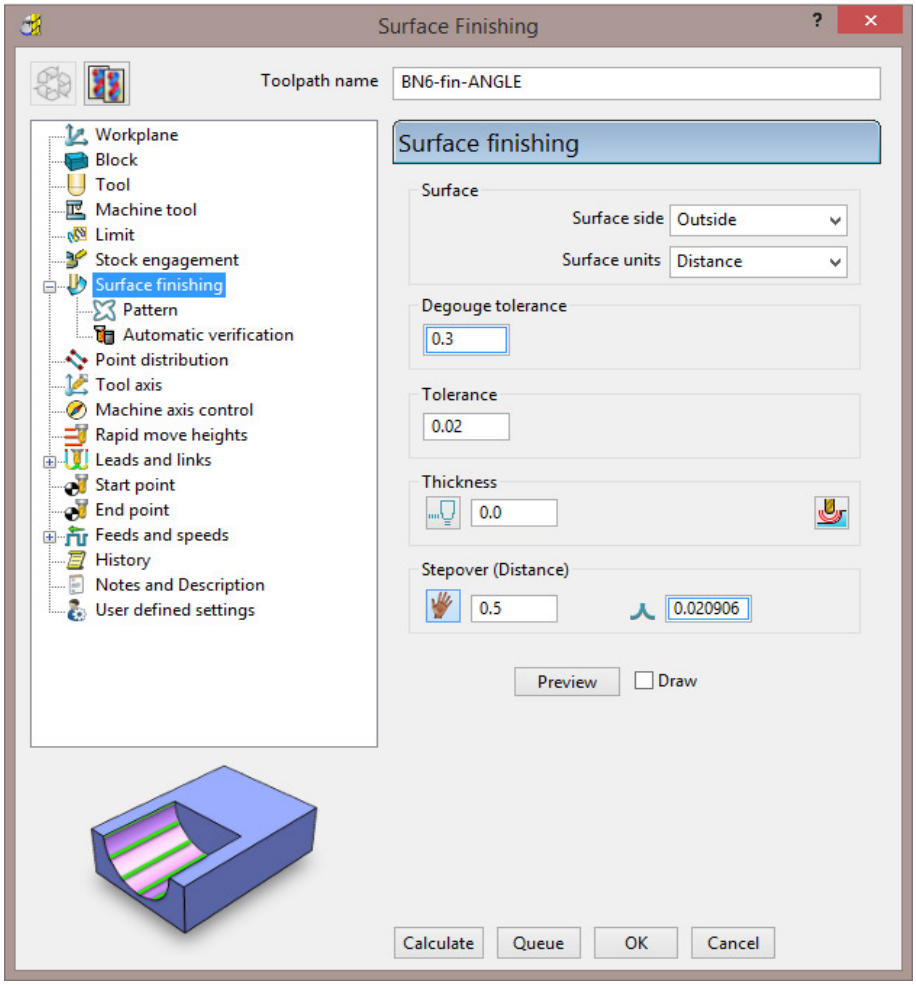
6 **Activate** the **BN6** tool.

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

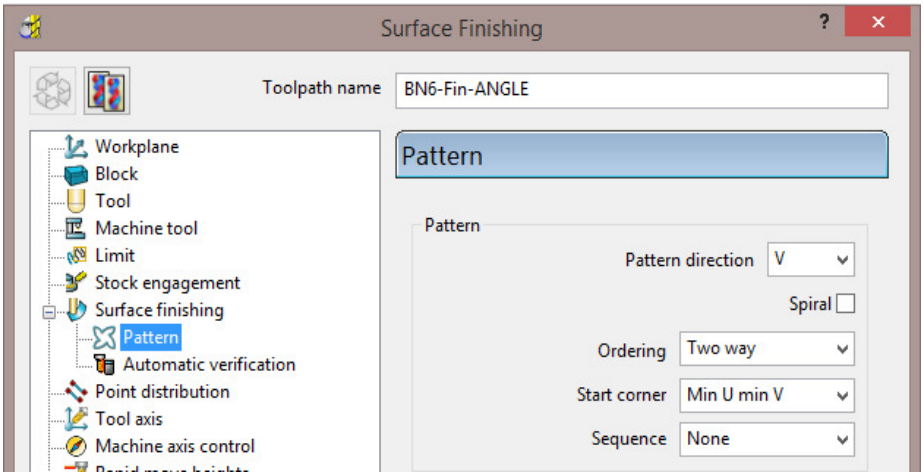


8 Select the **Toolpath Strategies** icon and in the **Strategy Selector** dialog select the **Finishing** tab.

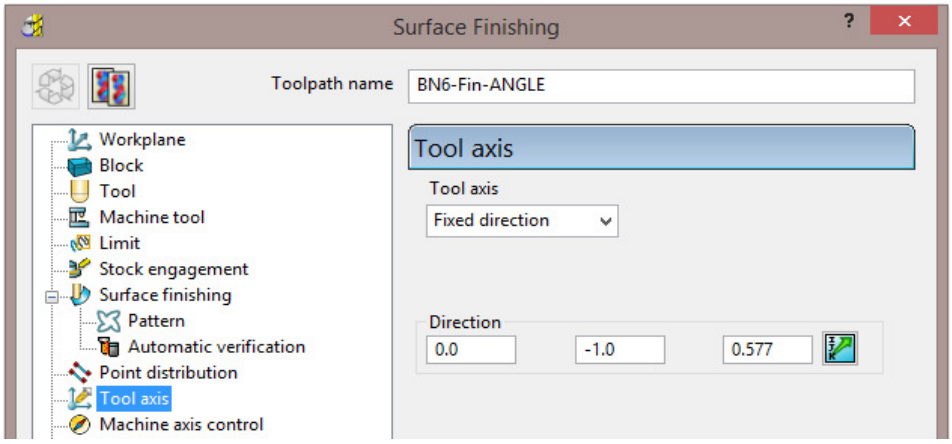
9 Select the **Surface Finishing** option and fill in the pages of the dialog exactly as shown in the following illustrations.



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



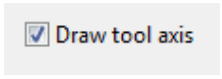
- ***Pattern direction*** - V
- ***Ordering*** - **Two way**

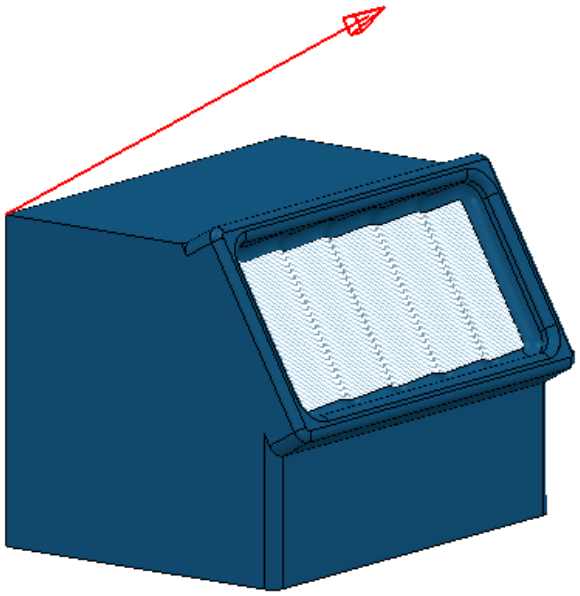


10 Set the **Tool Axis** to **Fixed direction**.

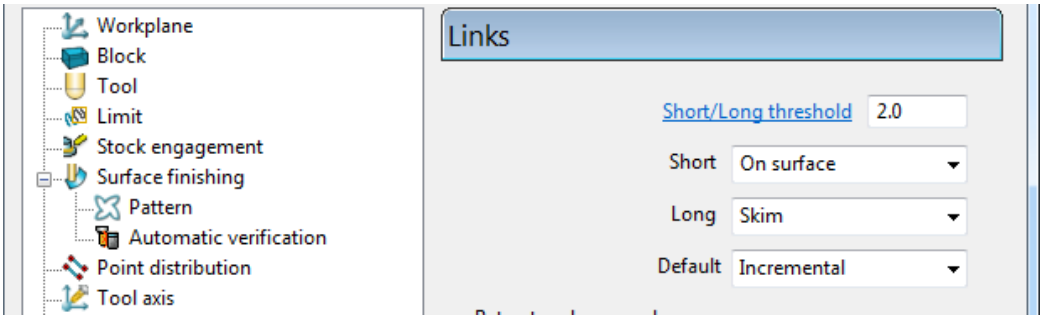
11 Input the vector ***Direction*** values: ***I* 0 *J* -1 *K* 0.577**

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



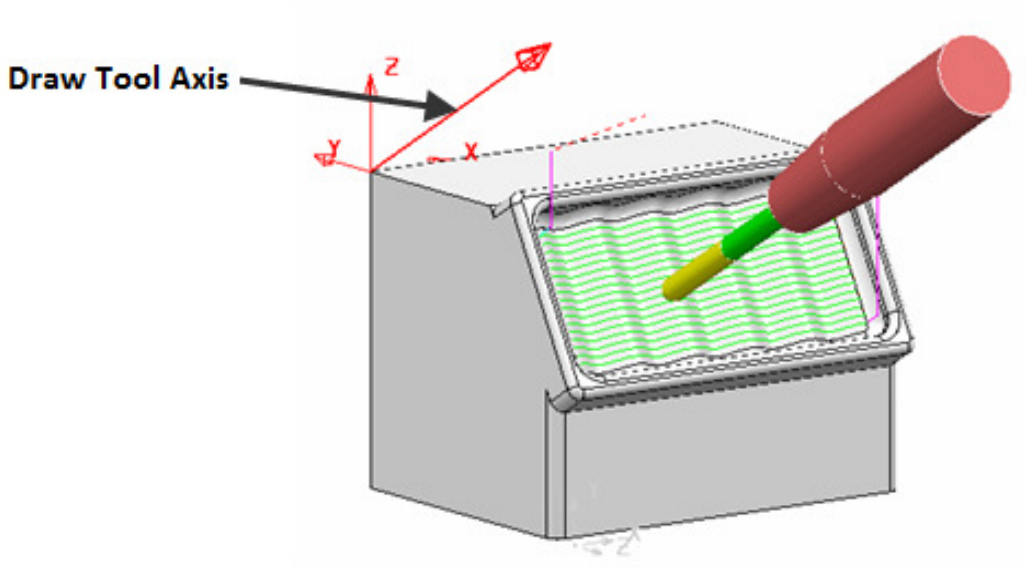


13 From **Leads and links** select the **Links** dialog and enter data exactly as shown in the following illustration.



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

15 **Close** the **Surface Finishing** dialog.



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

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

...\COURSEWORK\PowerMILL_Projects\ToolAxisFixed-EX1

4. Projection Surface Finishing

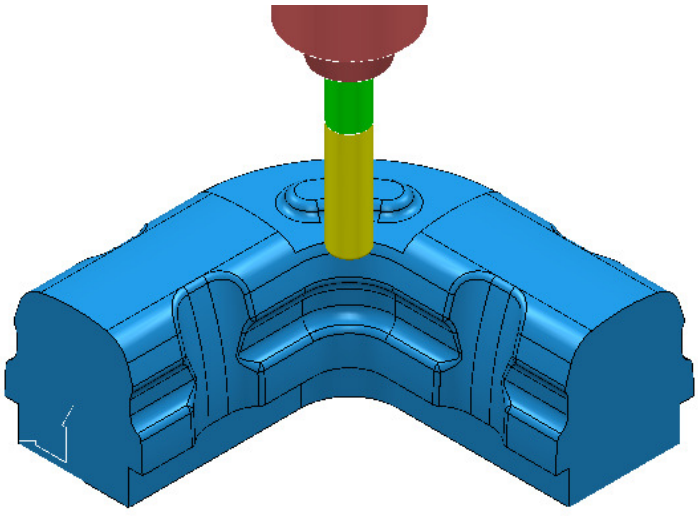
Introduction

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**:-

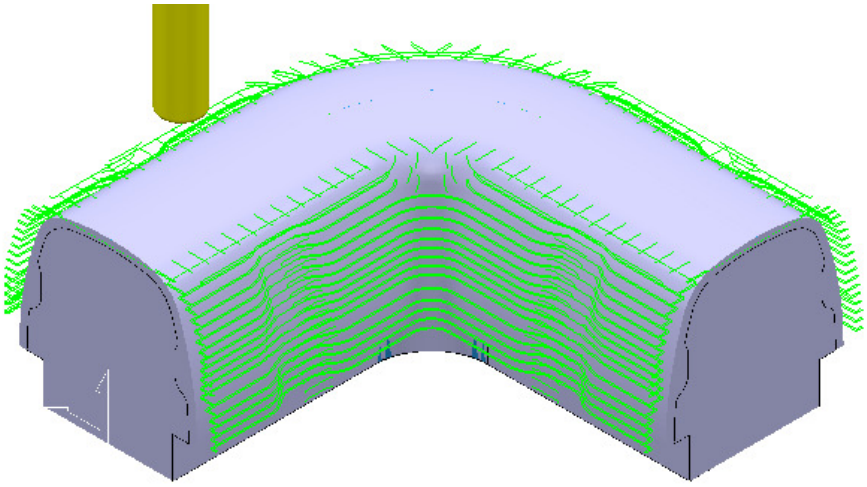
...\PowerMILL_Data\five_axis\CornerFixing\ProjectionSurfaceEX1-Start



2 **Save** the **Project** as:

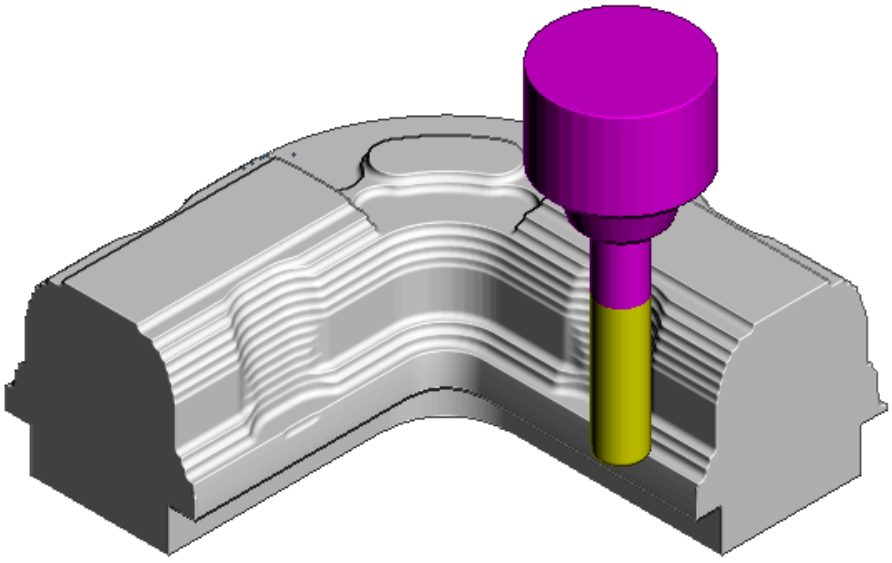
...\COURSEWORK\PowerMILL_Projects\ProjectionSurface-Example1

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



*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**.

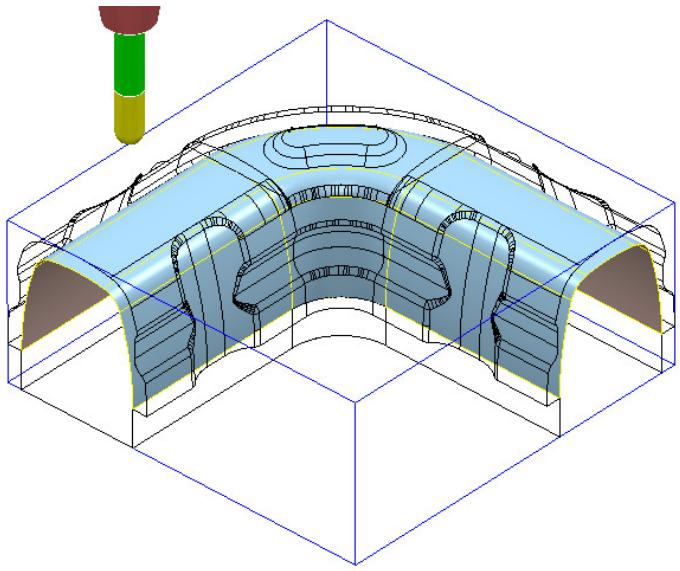
7 **Calculate** a new **Block** - ***Defined by*** **- Box** using ***Type*** – **Model**.

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

…..\PowerMILL_Data\five_axis\CornerFixing\CornerFixingRefSurf1.dgk

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**.

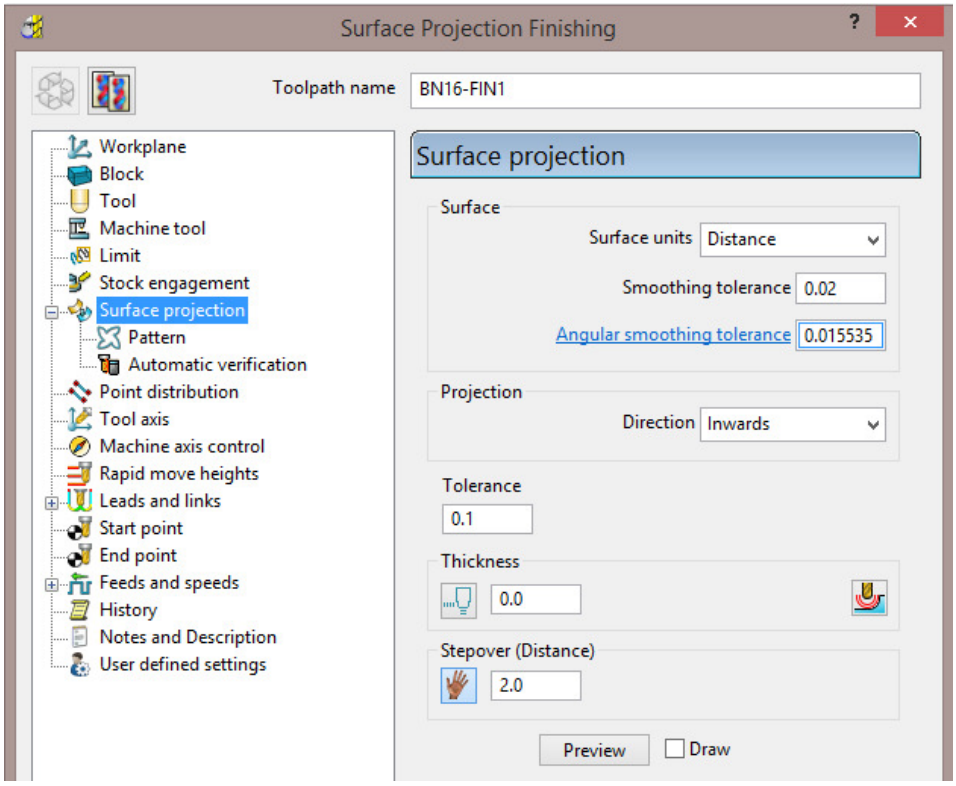


The newly ***imported*** (reference) **surface** (shown shaded above) is fully located inside the original component **model**.

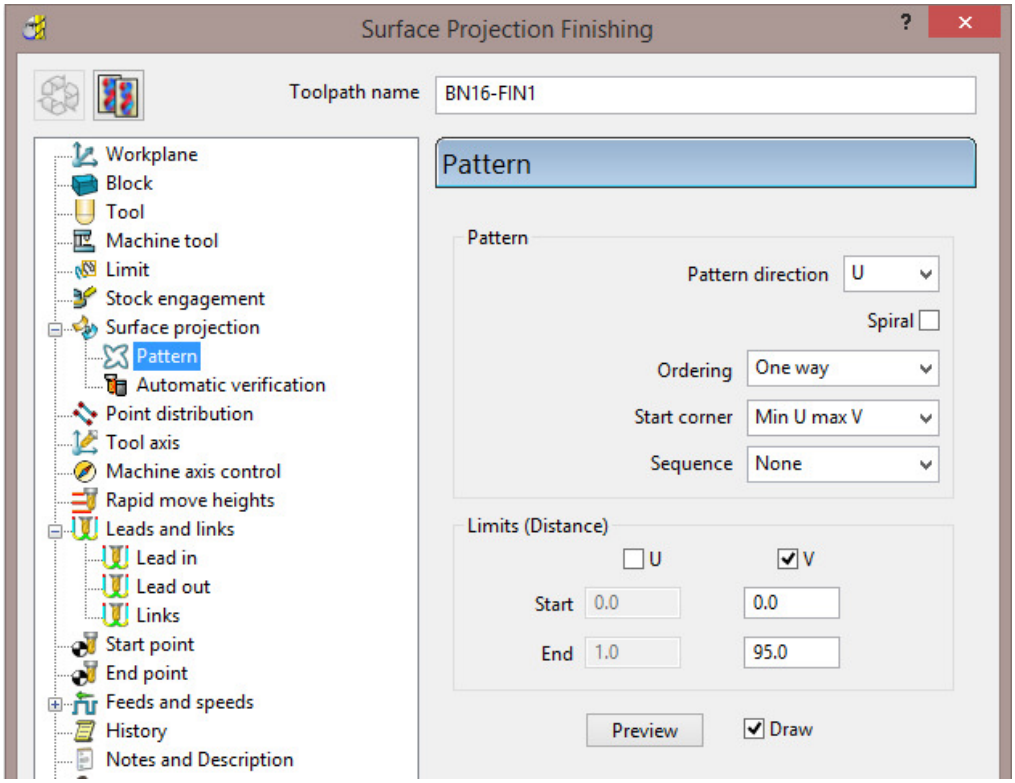
11 From the **main** toolbar select the **Toolpath Strategies**  .

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.

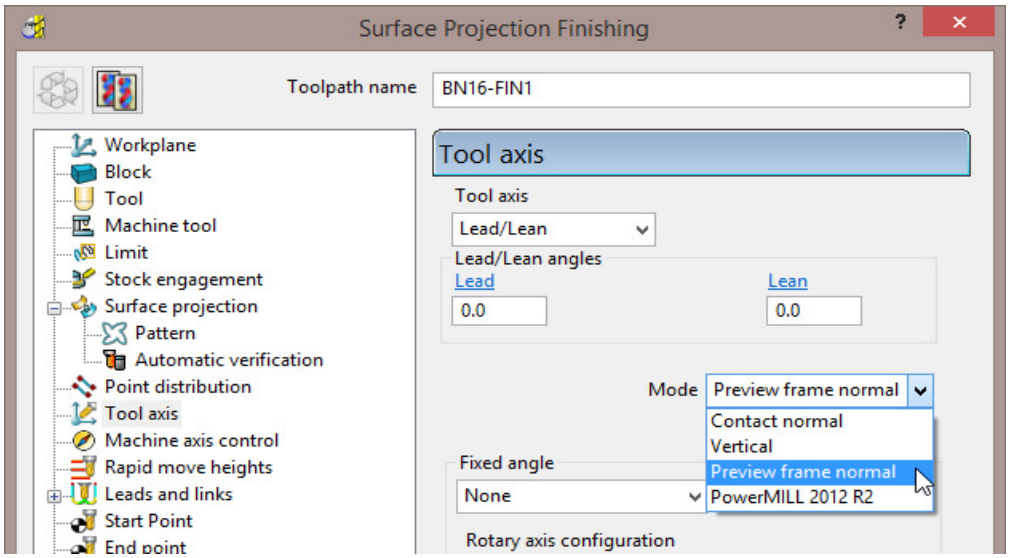


14 Enter the values into the **Pattern** page of the **Surface Projection Finishing** exactly as shown below.



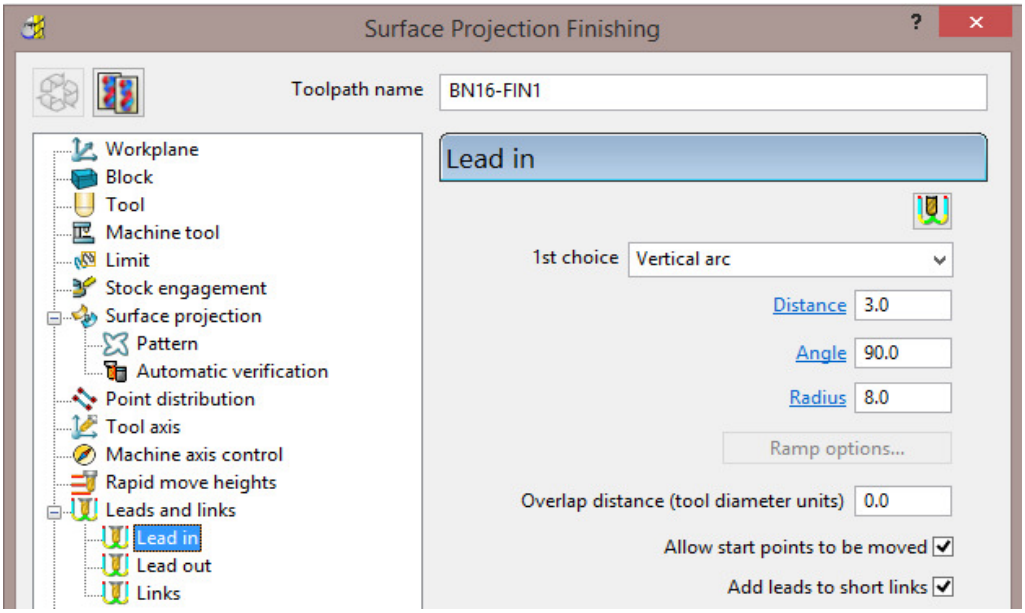
- ***Pattern direction*** – **U**
- ***Start corner*** – **Min U max V**
- ***Limits (Distance)*** – **V** (ticked):- **Start 0 End 95**

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**.

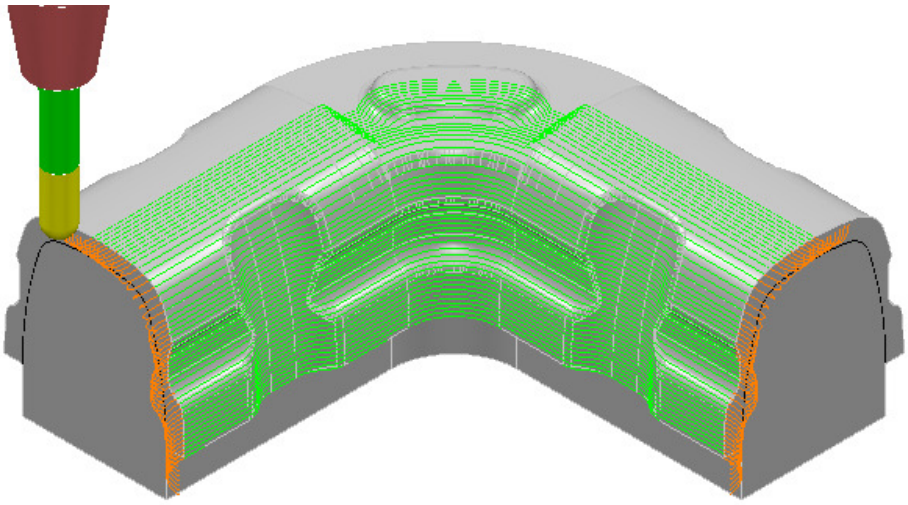
17 Enter the values into the **Tool axis** page of the **Surface Projection Finishing** exactly as shown below.



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*

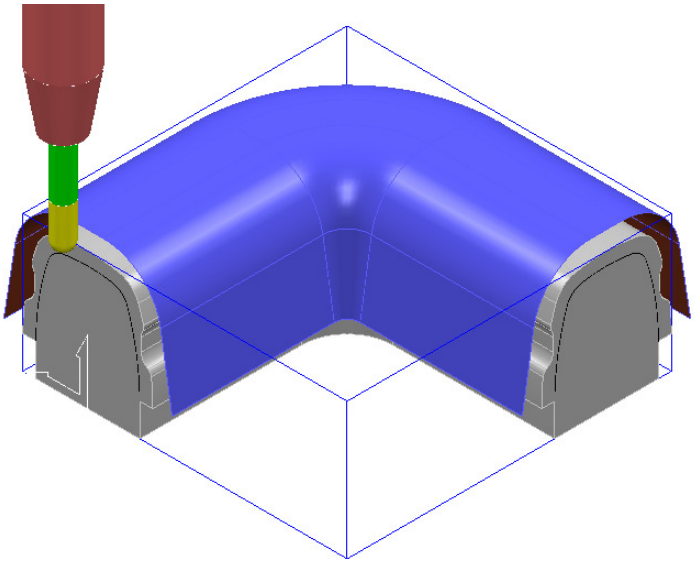
20 In the **Surface Projection** dialog select ***Make a Copy***  to re-activate the **Settings**.

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.

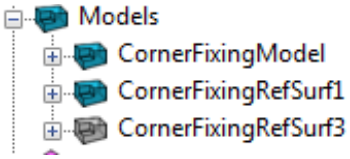
21 Right-click on **Models** in the **PowerMILL *explorer*** and select **Import Reference Surfaces**.

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**.*

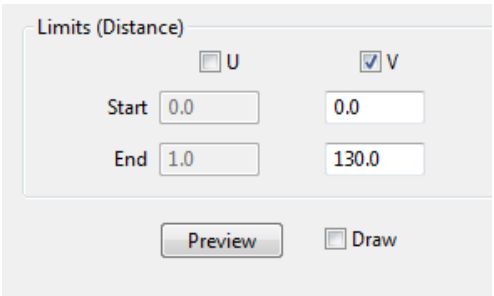


*The imported **Reference Surface** is displayed in the **PowerMILL *explorer*** with a different colour (grey).*

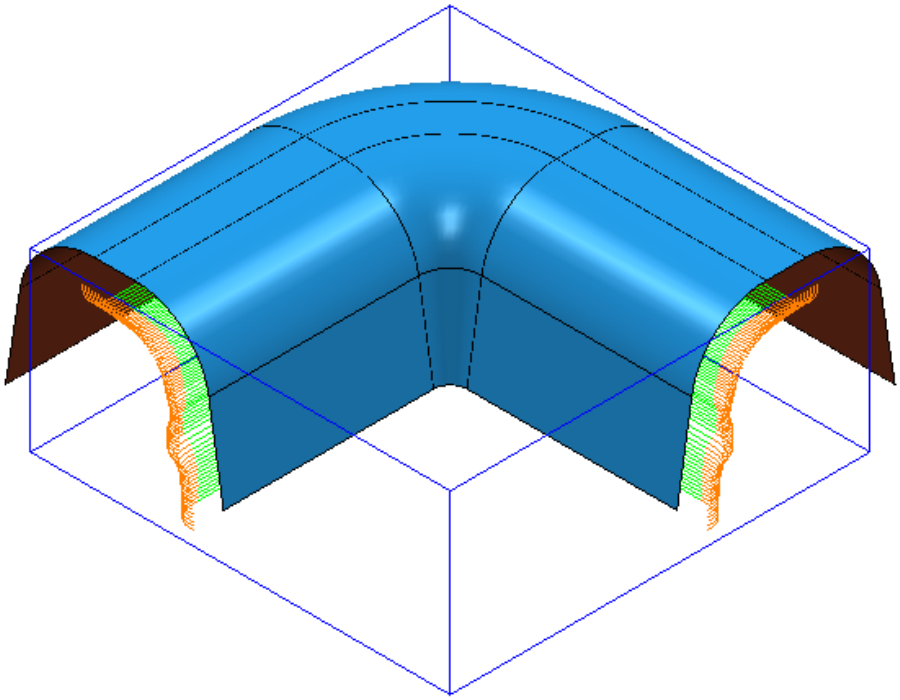
23 Select the ***Reference Surface*** - **CornerFixingRefSurf3**.

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**.

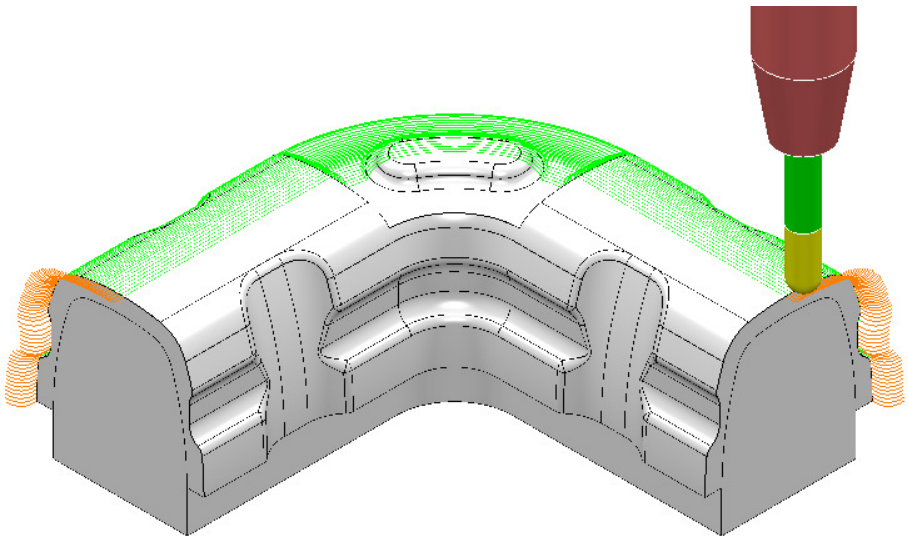


27 To update the stored **Project**, select **File > Save Project**:

...\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**:

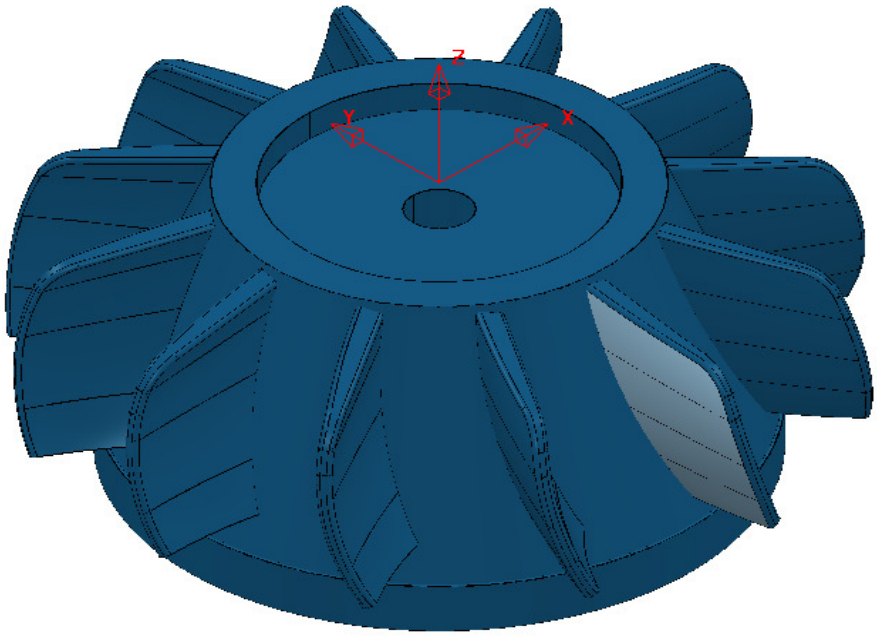
Surface Projection Range.

Depending on the application, it may be necesary 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** the **Project** as:-

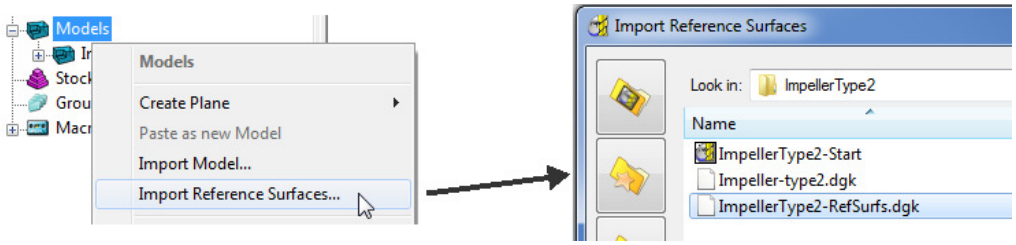
….\COURSEWORK\PowerMILL_Projects\ImpellerType2-EX1

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.

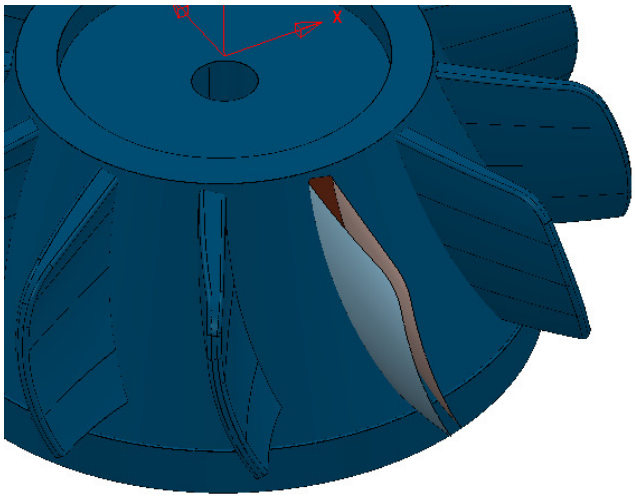
5 In the **PowerMILL *explorer*** right mouse click on **Models** and from the local menu select **Import Reference Surfaces**.



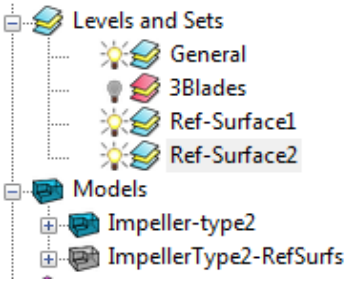
6 In the dialog browse to:-

...\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.

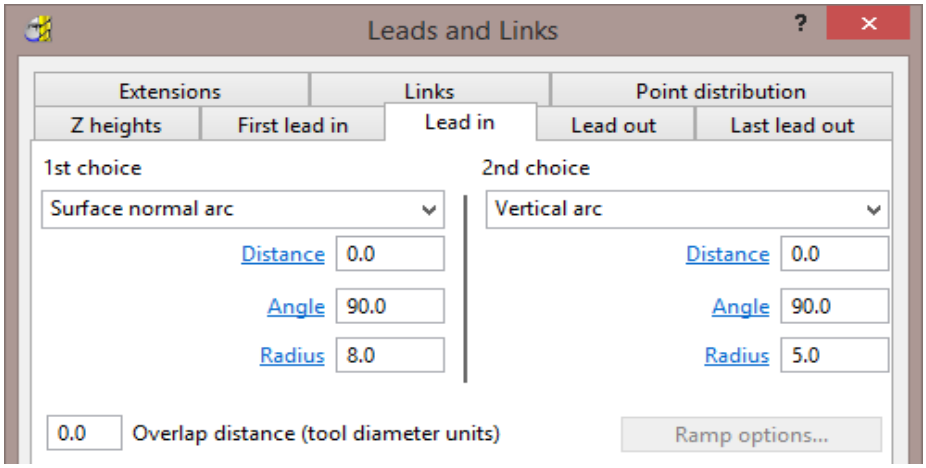
10 Switch off the **Level**, **Ref-Surface2**.

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.

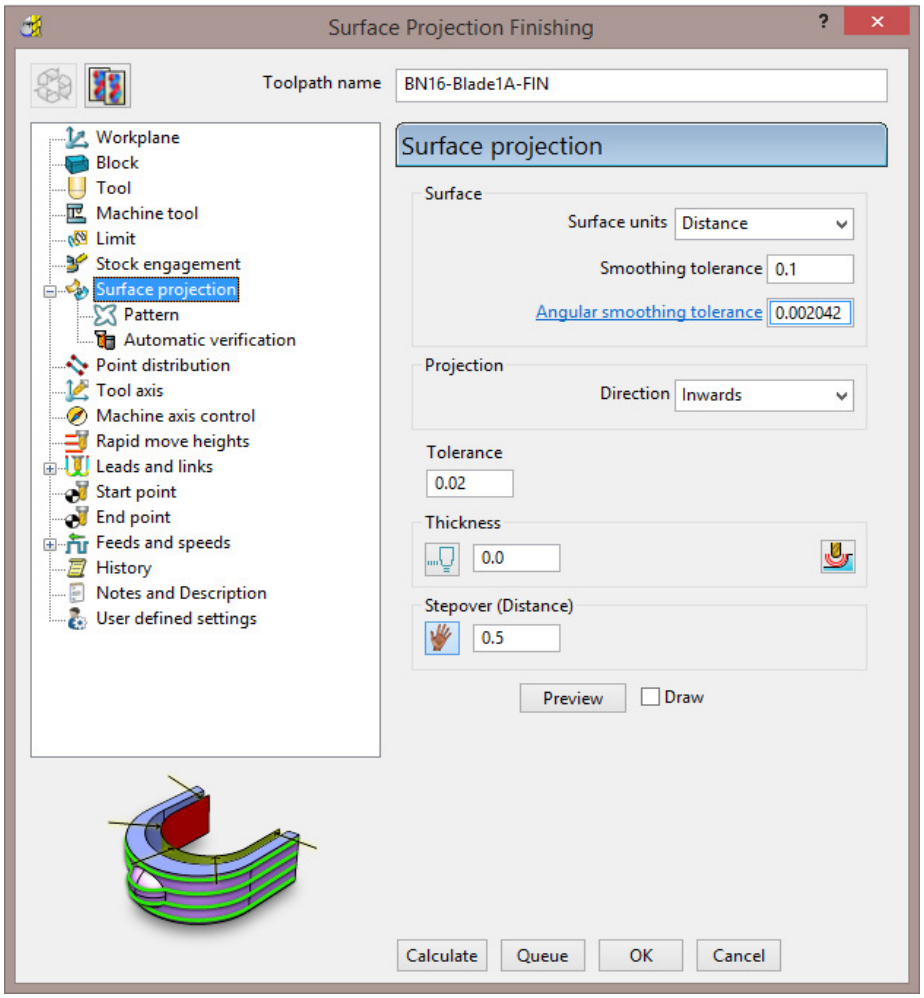
14 Set **Lead In** and **Out** to use **Surface normal arc**.

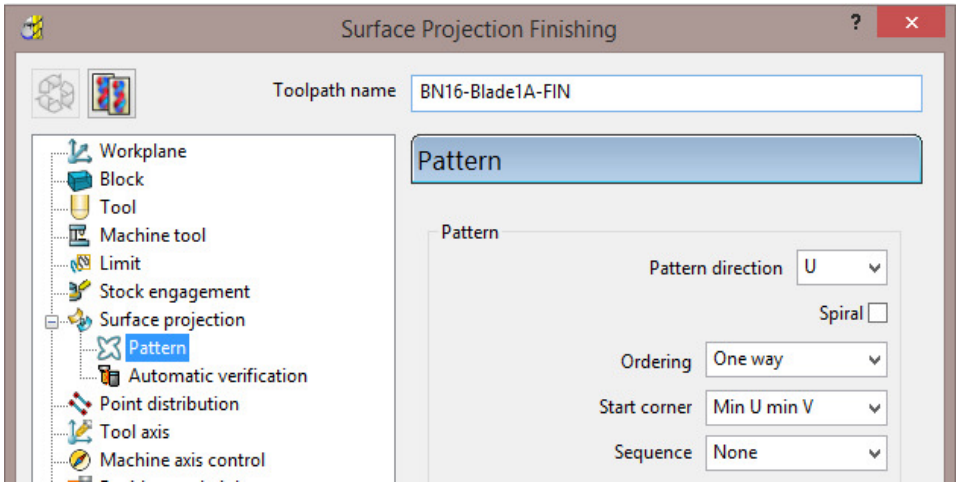


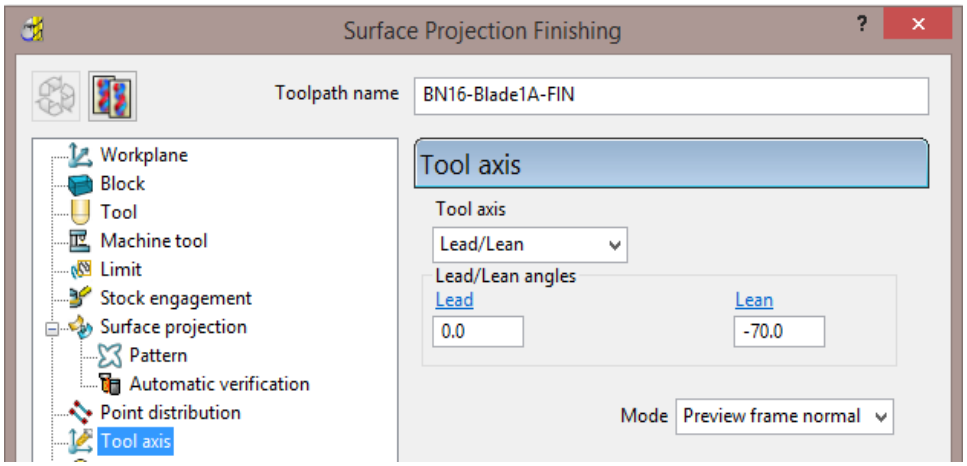
15 From the **main** toolbar select the **Toolpath Strategies**  .

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

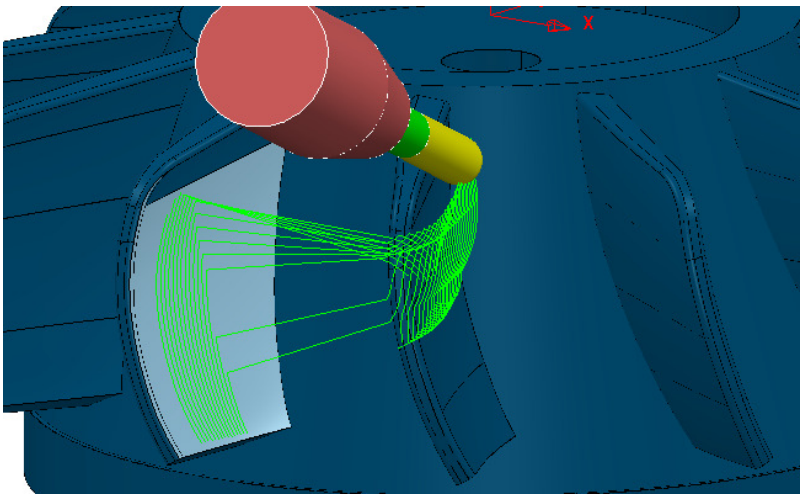
17 Enter data into the following 3 pages of the **Surface Projection** dialog exactly as shown below.



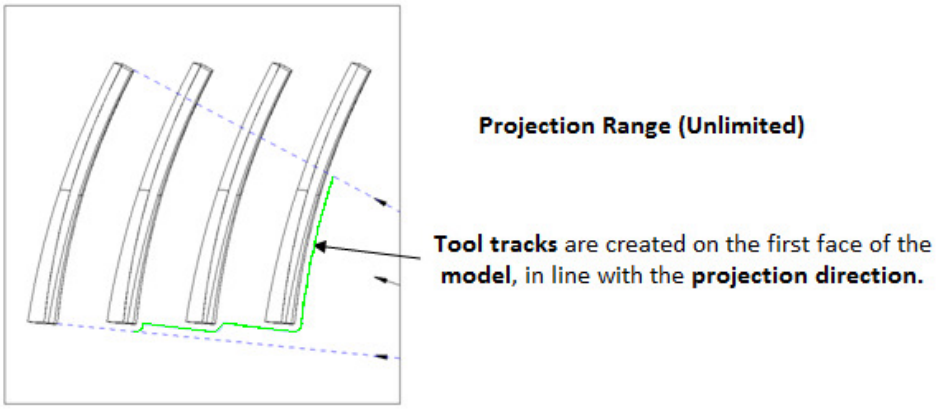




18 Select **Calculate** in the **Surface Projection Finishing** dialog to produce the following toolpath, and then **Close** the dialog.



*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.

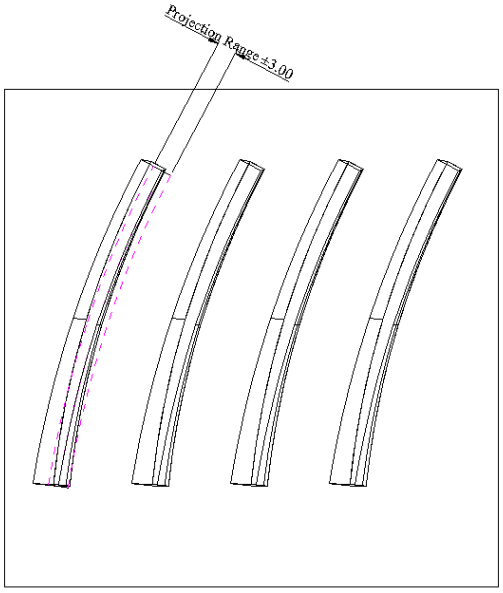
19 Right mouse click on the toolpath **BN16-Blade1A-FIN** and from the local menu select **Settings** to re-open the strategy dialog.

20 Select **Recycle**  .

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

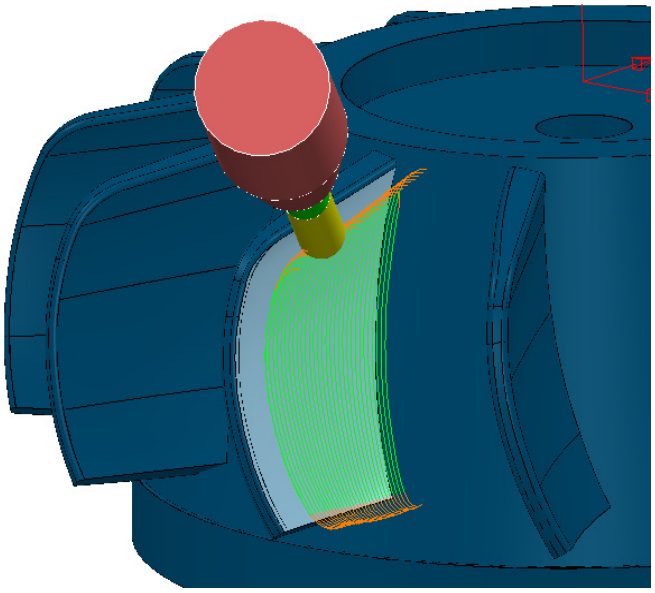
22 Enter the following commands into the **Command Window** that has appeared at the below of the graphics area:

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



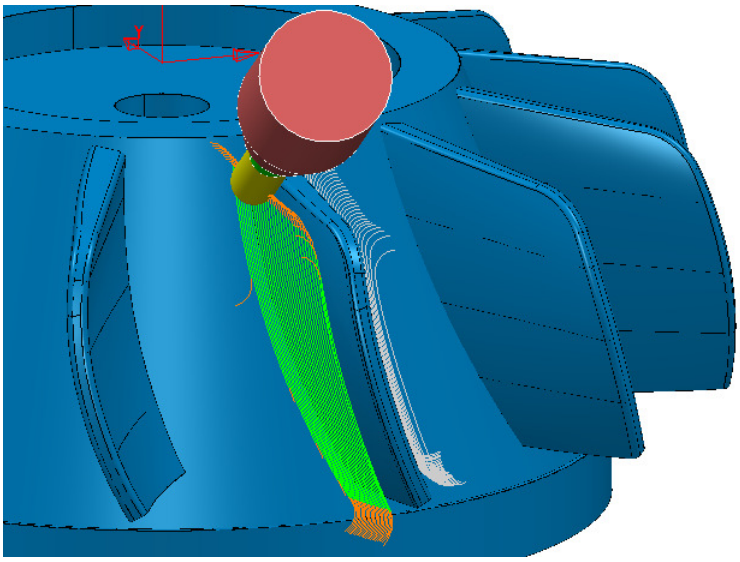
The above command input limits the **Surface Projection** Range to **+** and **-3mm**.

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



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.

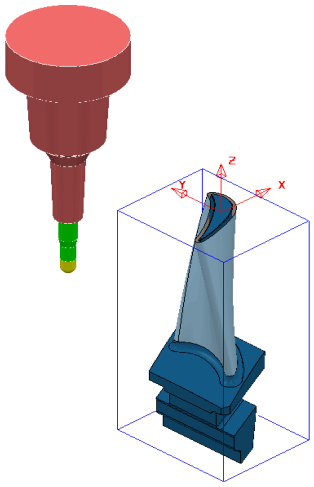
If you select a **Surface Units** of **Distance**, then the first and last passes will match the edges of the surface. To achieve this, the intermediate passes will adjust to less than or equal to the specified **stepover**.

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

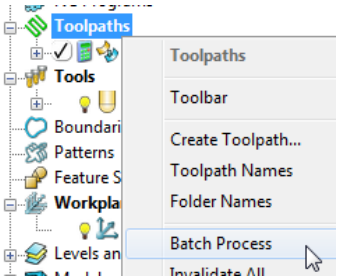


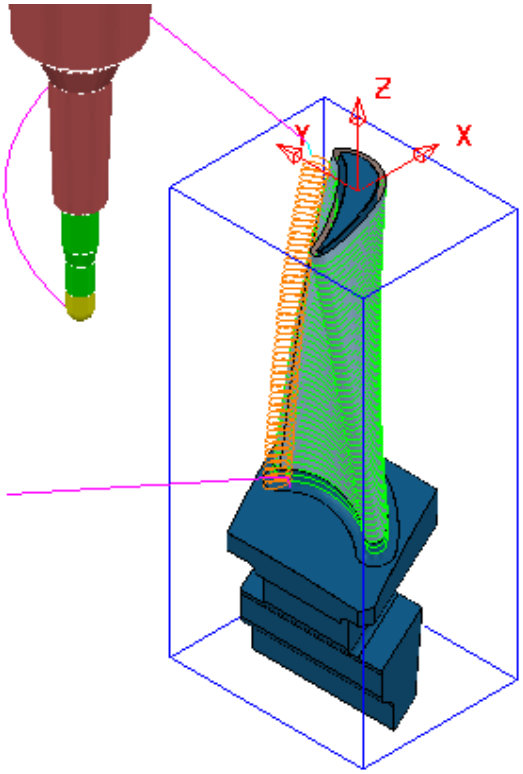
3 **Save** the **Project** as:

...\COURSEWORK\PowerMILL_Projects\SurfMachine-Spiral

*The Strategy in the above **Project** is unprocessed (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.



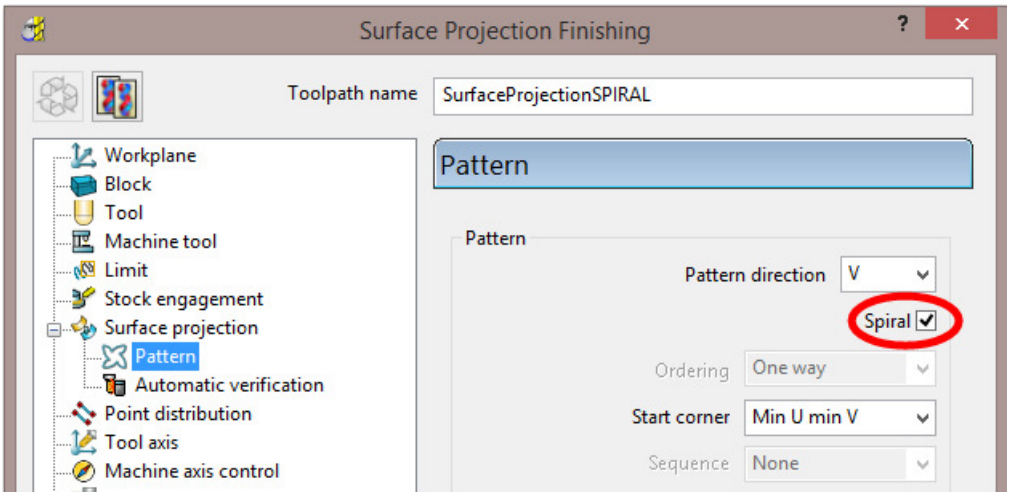


The strategy automatically reinstates the original selected **surface** used on a previous occasion. The resultant **Toolpath** contains independent, stacked tool tracks and **Surface Normal Arcs** for **Lead In/Out**. For this strategy it, would be both possible and more appropriate to use the **Spiral**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

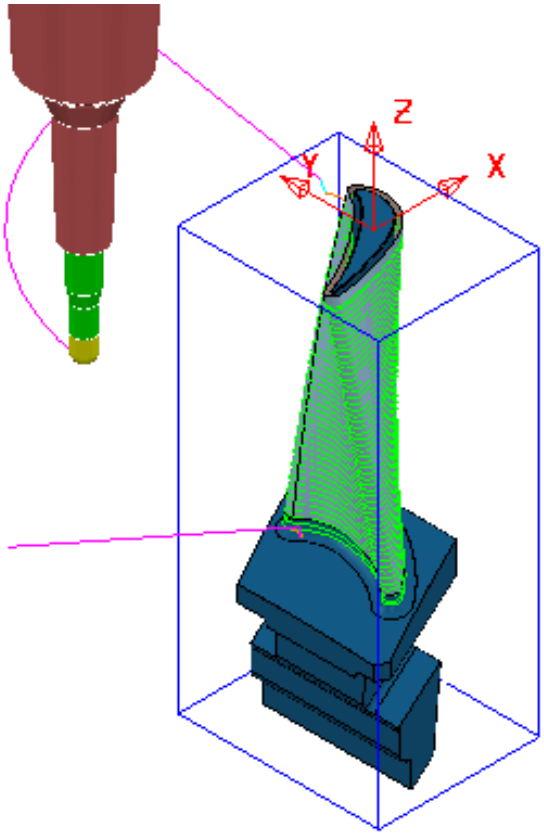
5 Right-click on the **Active** toolpath to open the local menu and select **Settings** to open the original **Surface Projection Finishing** dialog.

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 the stored **Project**:-

...\COURSEWORK\PowerMILL_Projects\SurfMachine-Spiral

5. Embedded Pattern Finishing

Introduction

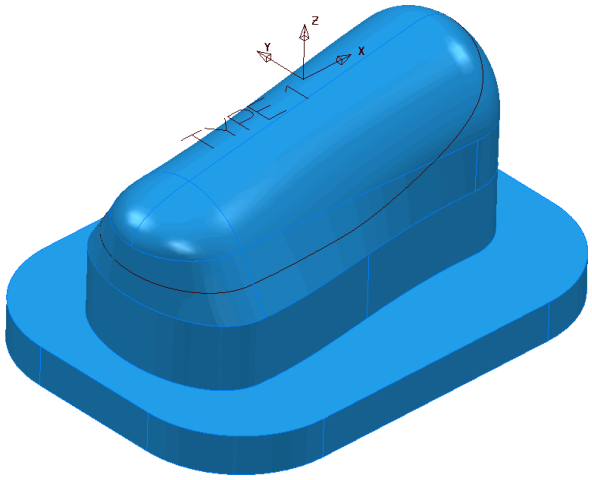
This **Five Axis** strategy not only tracks along the **Embedded Pattern** but automatically applies a **Tool Axis** alignment relative to the ***Surface Normals***.

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 ***Parameters*** and ***Settings*** for the **Embedded Pattern**.

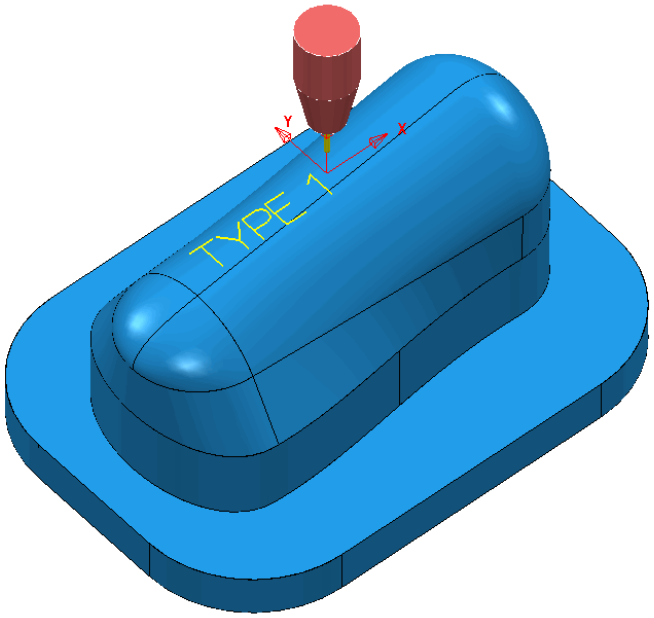
4 **Activate** the ***Tool*** **BN2**.

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

6 In the graphics area, select the 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**'.

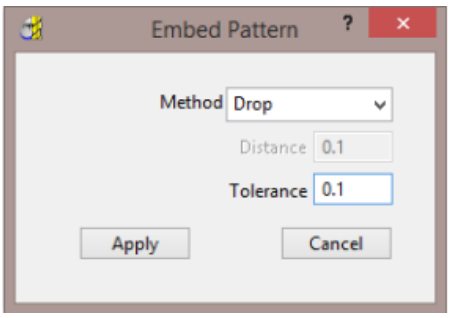


The first **Pattern** contains the text '**TYPE 1**' and is positioned above the **Model**. This **Pattern** is then used to create an **Embedded Pattern** which projects down **Z** onto the **model**. The **Embedded Pattern** will also pick up and store the **Surface Normals**.

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.

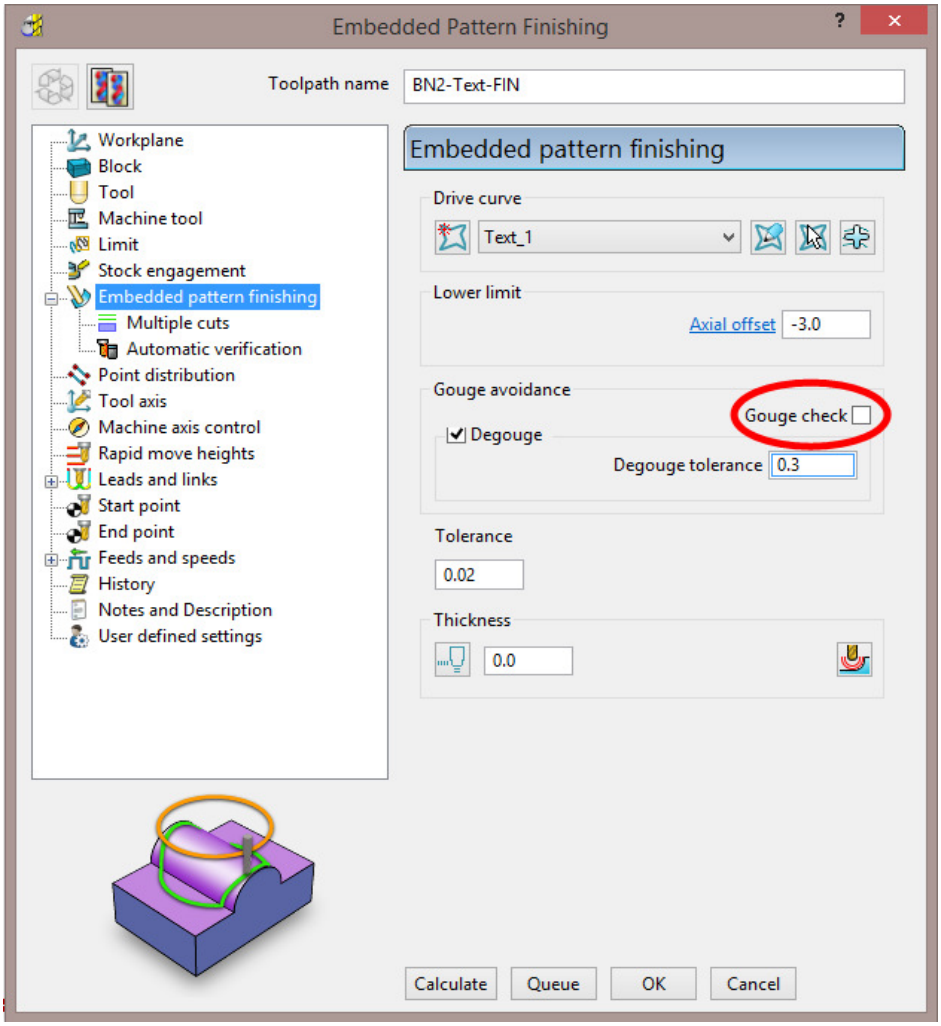


*An Embedded Pattern called **Text_1** is created as projected down Z and is identified by the icon in the explorer. The original Pattern is retained.*

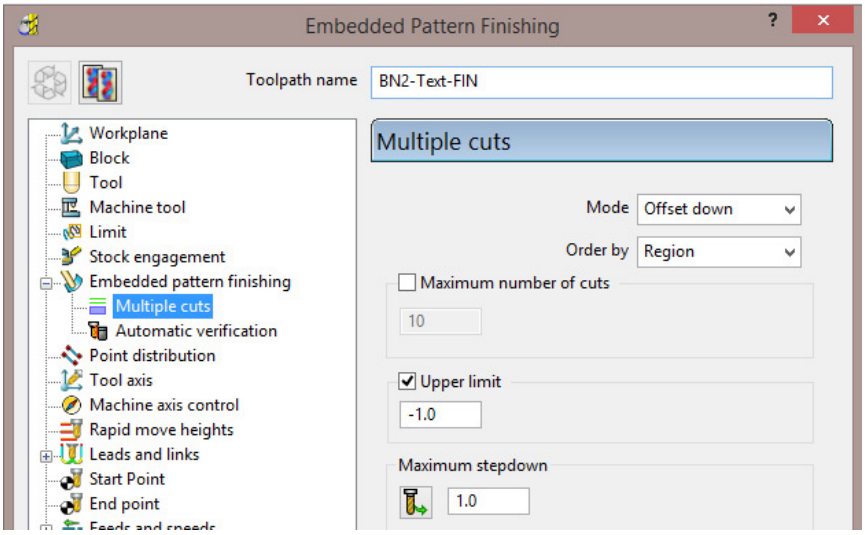
12 Select **Toolpath Strategies** and from the **Strategy Selector** dialog sect the **Finishing** tab.

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

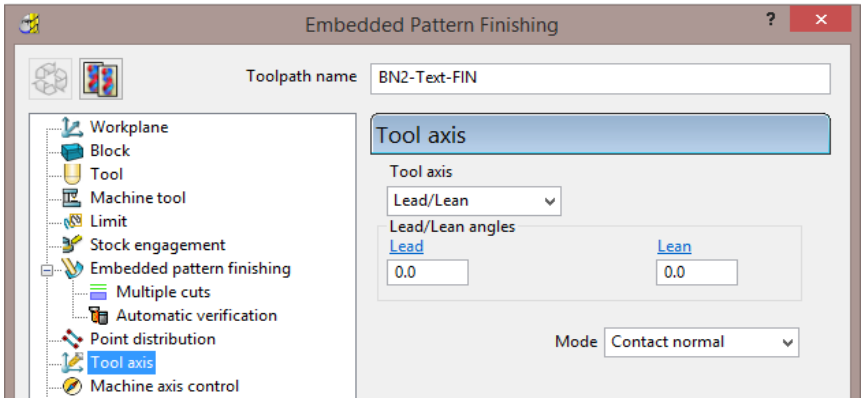
14 Enter the values into the following pages of the **Embedded Pattern Finishing** dialog exactly as shown.



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



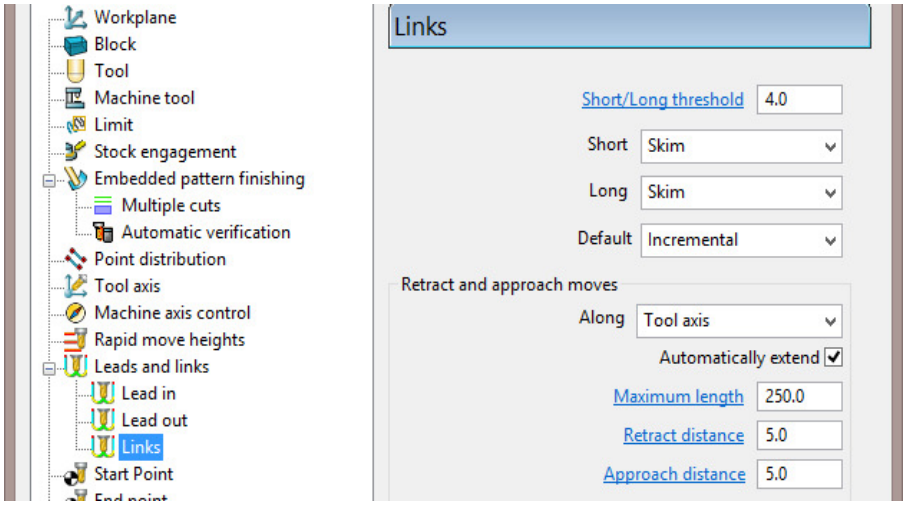
15 Tick **Upper limit** with a value of **-1**



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 and Approach distance 5).

17 Under **Leads and Links**, select the **Links** page.



18 In both **Retract** and **Approach distance**, enter **5** (as shown above).

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

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

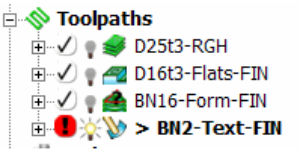
The text '**TYPE 1**' is machined to a total depth of **-3** with the tool aligned to the ***Surface Normals***.

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**.

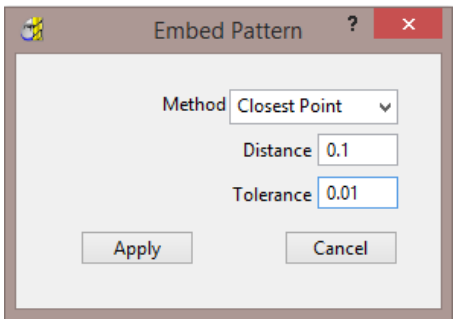
*If the toolpath is **verified** for **gouges** it will take on a **Red** 'gouge' warning. It is not possible to create a toolpath using a -ve thickness value that is higher than the tool tip radius while **Gouge check** is switched on.*

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

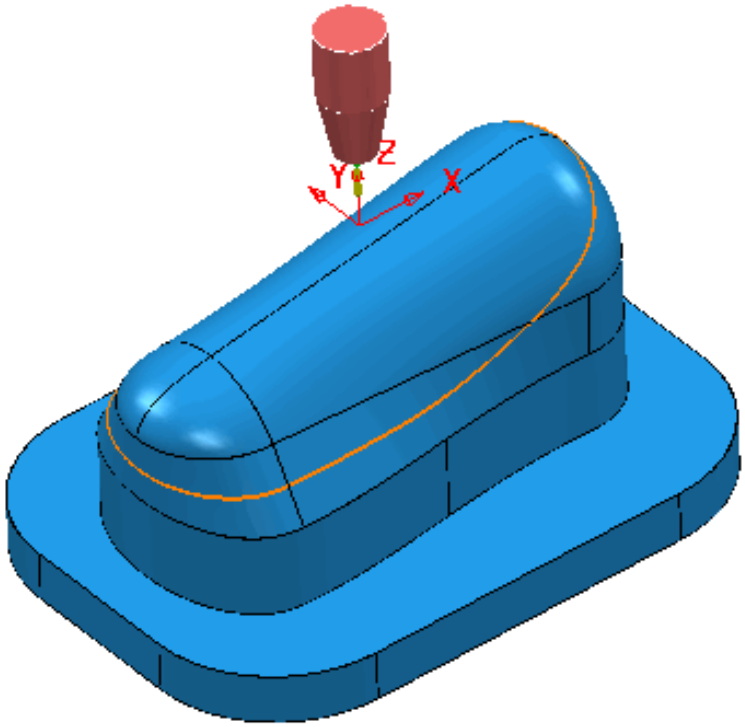


22 Create a new **Pattern** named **ScribeLine** and **Insert** the wireframe ***Curve***, running around the upper wall of the **model**.

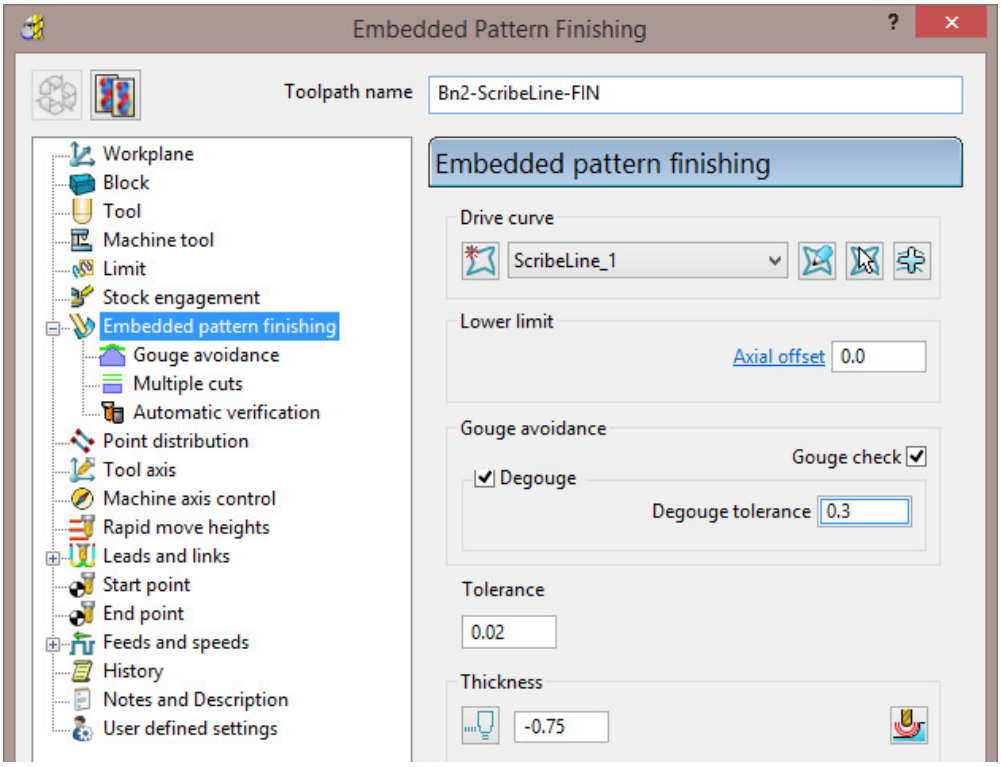
23 Create an **Embedded Pattern** from the **Pattern** (**ScribeLine***)* but this time select ***Closest Point*** in the dialog.



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

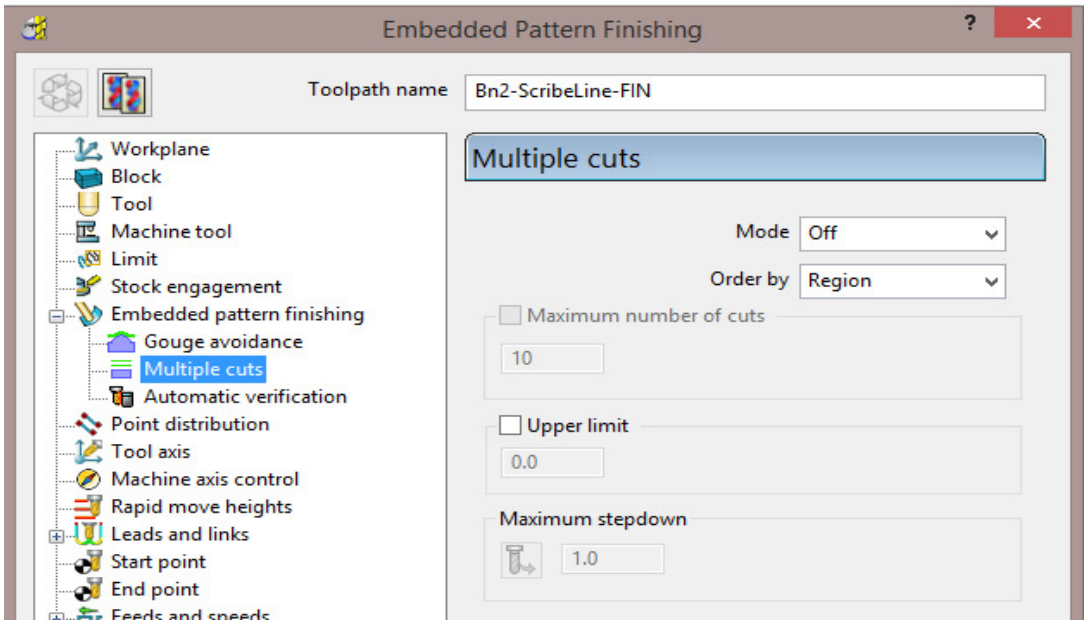


24 Open a new **Embedded Pattern** strategy named **BN2-ScribeLine-FIN** using **Drive curve** - ***ScribeLine_1***.



- ***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**.



- **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**.

6. Five Axis Swarf Machining

Introduction

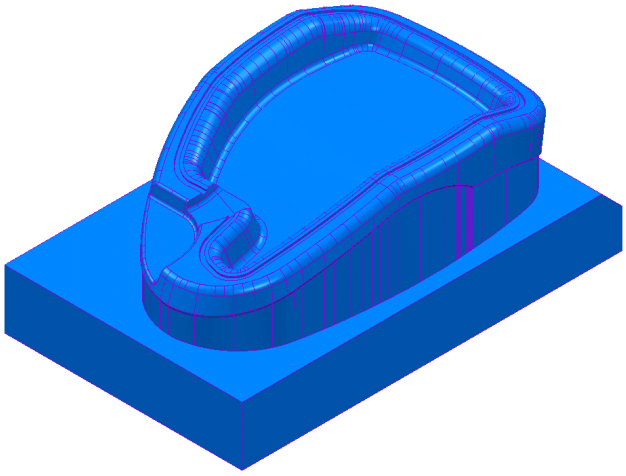
Five Axis - **Swarf Finishing** is designed to create a toolpath where the selected **surfaces** are machined with the side of the tool (the default **Tool Alignment** option is **Automatic**). A **Swarf Finishing** toolpath will only exist where the tool is able to remain in full contact with the selected **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**:

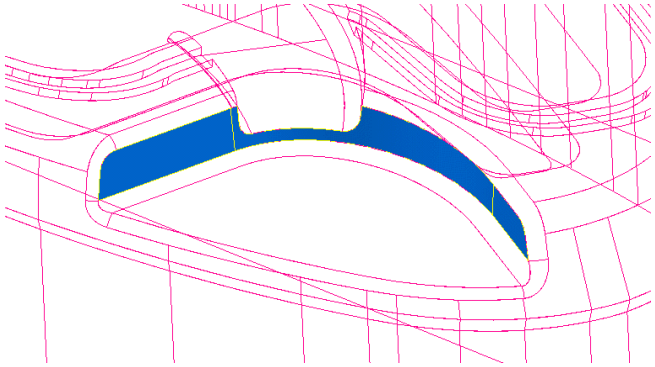
   ...\PowerMILL_Data\five_axis\swarf_mc\swarf_model.dgk



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**.

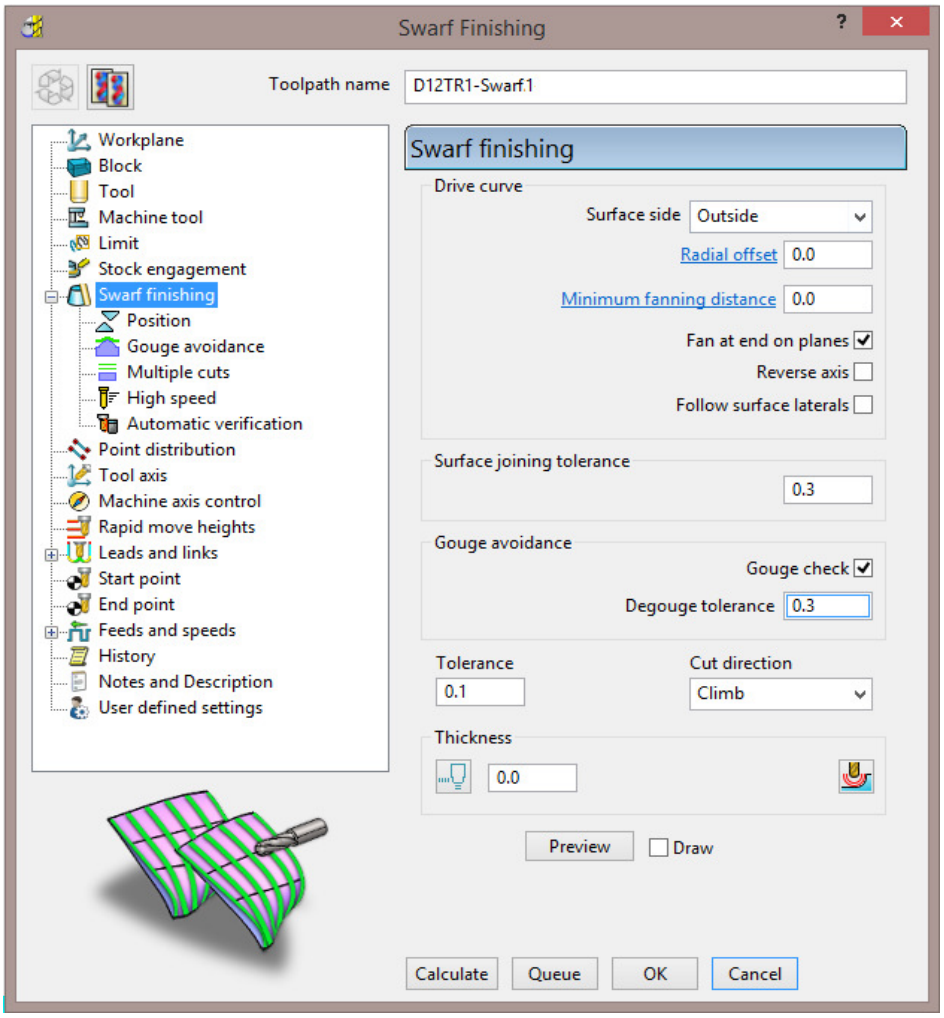
6 Change the **View** to **ISO 4** and select the local surfaces to be **Swarf** machined (as shown below shaded).

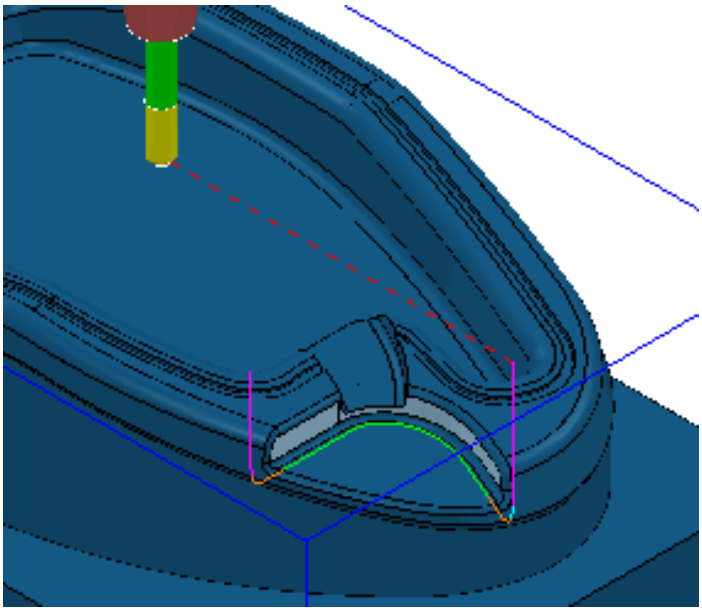


7 Select **Toolpath Strategies** and from the **Finishing** dialog select the **Swarf Finishing** option.

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.*





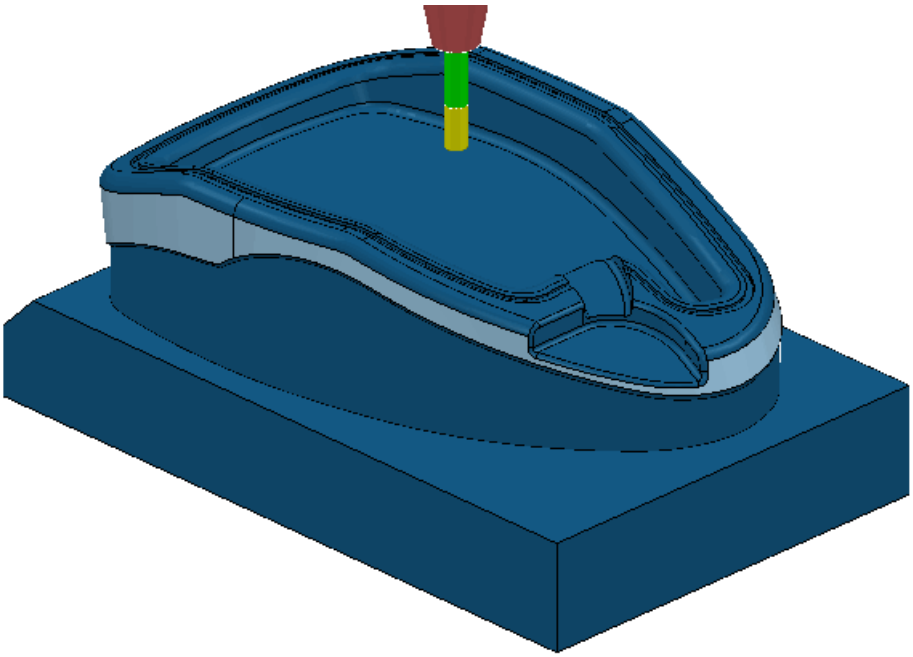
9 Open the **Leads and links** dialog via the icon  and **Apply** the following settings:

Lead In/Out: ***Vertical Arc*** – **Distance 10** – **Angle 90** – **Radius 5.0**

Links: ***Short\Long*** – **Skim** ***Default*** – **Incremental**

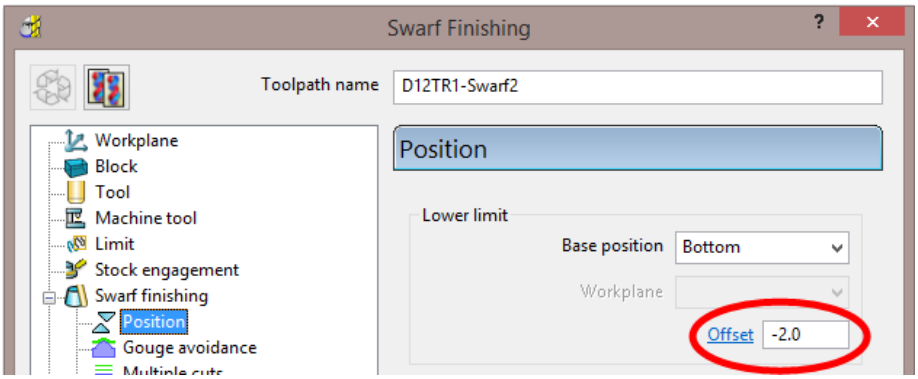
10 Deselect the local surfaces, **Simulate** the resultant toolpath, and observe the changing **tool alignment** as it **Swarf** machines the selected **surfaces**.

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

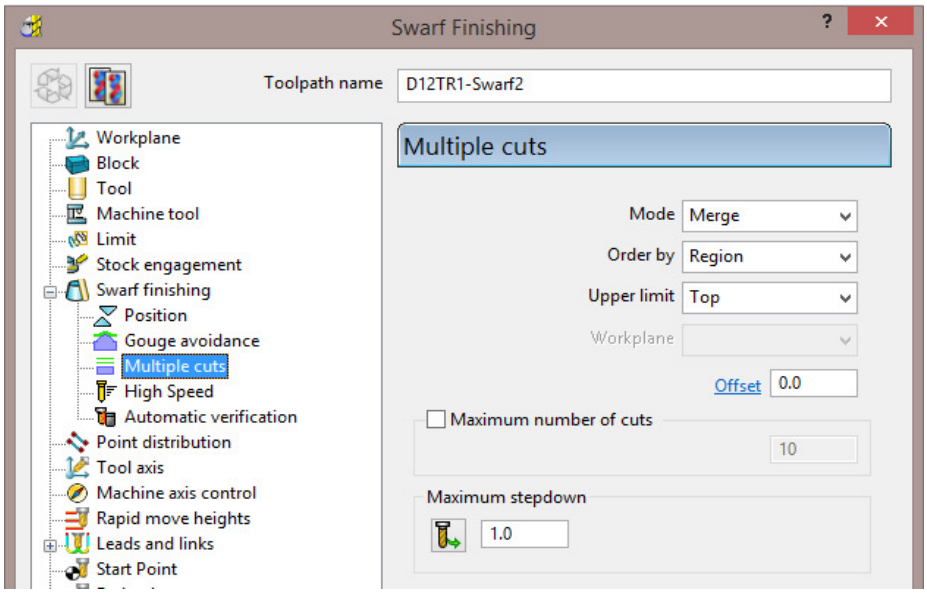


12 Select the, upper sidewall, **surface** (as shown above).

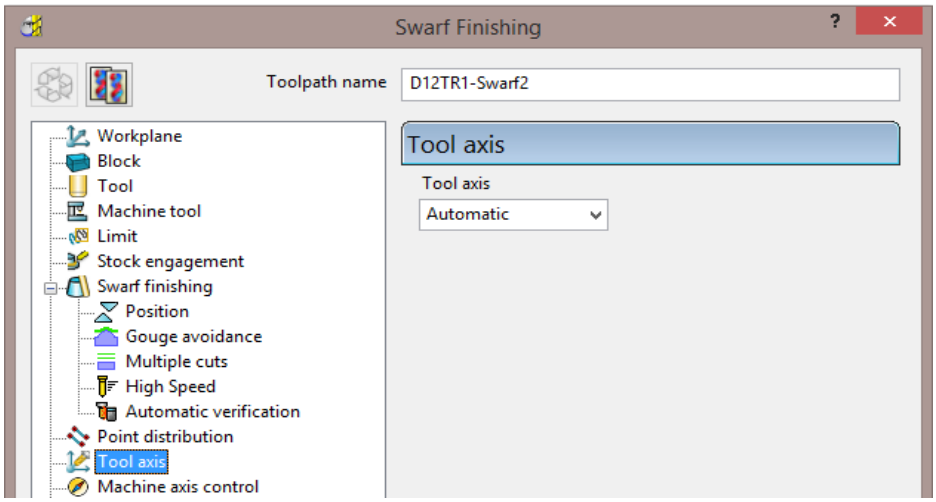
13 Select the **Position** page and set ***Base position*** **- Bottom** with **Offset** -2.



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**.

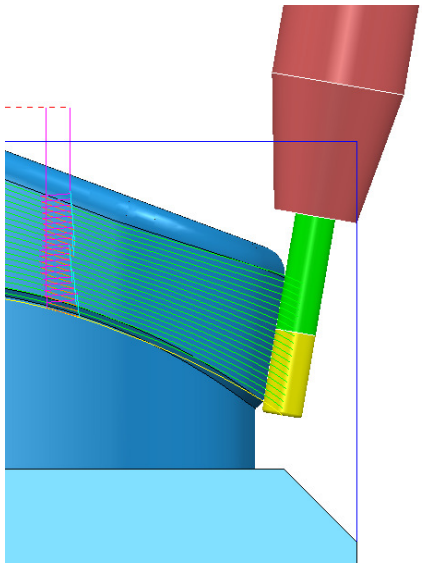


16 Select **Calculate** at the bottom of the dialog to create the toolpath.

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: **Short\Long** – **Skim** **Default** – **Incremental**

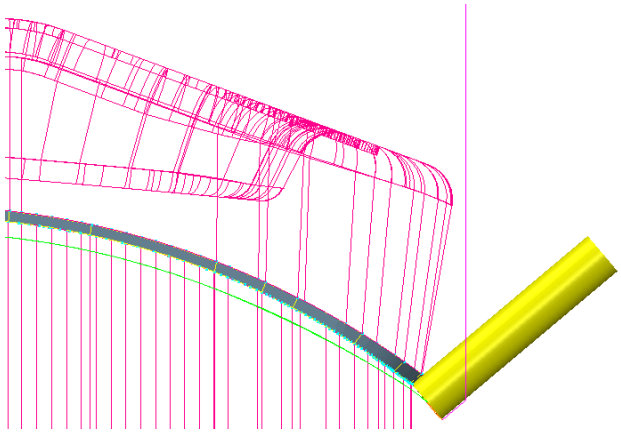


The new **Swarf Finishing** strategy steps down the selected **surfaces** while merging the **stepover** between the ***Upper*** and ***Lower contours***.

Exercise

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

2 Create a **Copy** of the original, single pass, **Swarf Finishing** strategy to create a machining path, this time using the **Dia 10 End Mill** and aligned to the underside of the undercut selected, **recess surface** 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**.

Swarf Finishing - Example 2

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

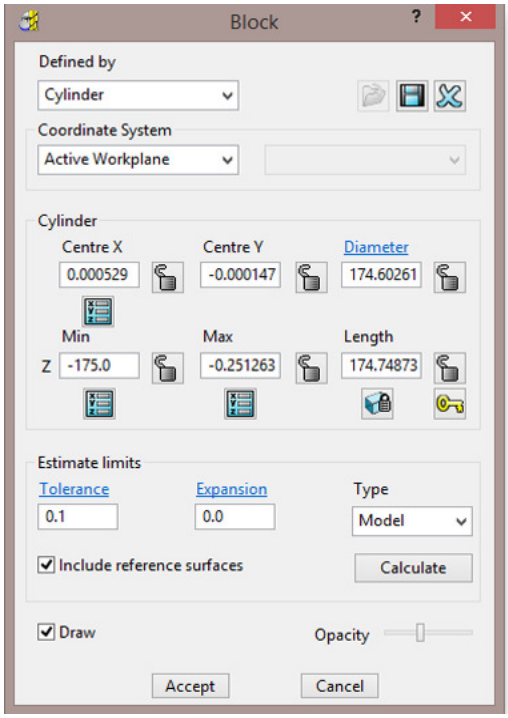
...\COURSEWORK\PowerMILL-Projects\3Plus2-EX1

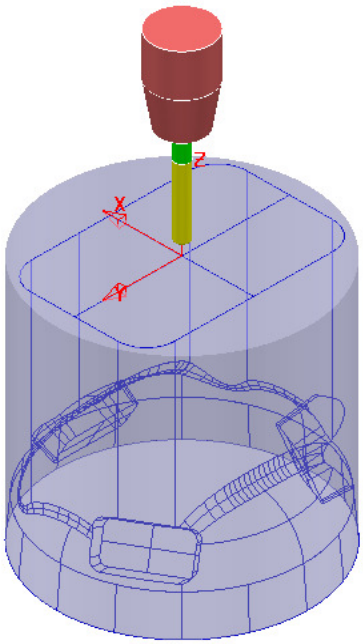
2 **Activate** the **Dia 10 tiprad 1** tool (**D10T1**).

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

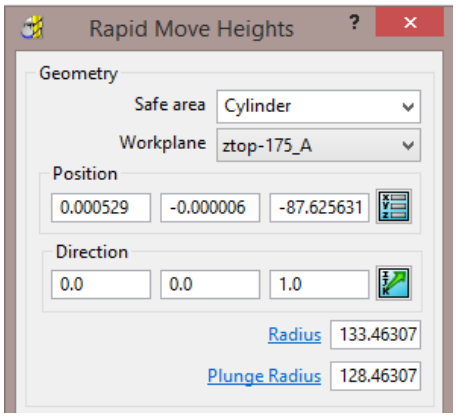
4 Make sure that no **Boundary** is **Active**.

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**.

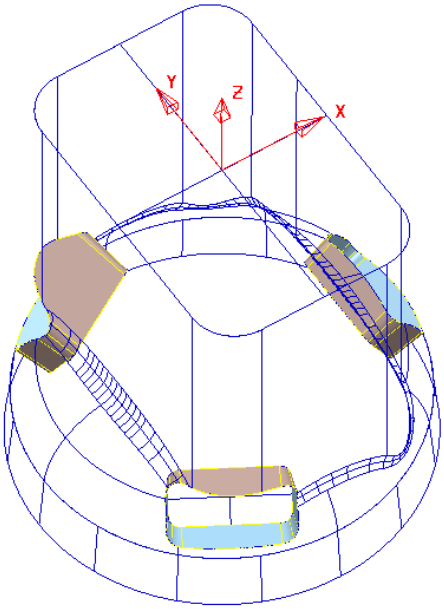


- Input the ***Workplane*** **ztop-175_A** and set a ***Direction*** (vector) **0 0 1**

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

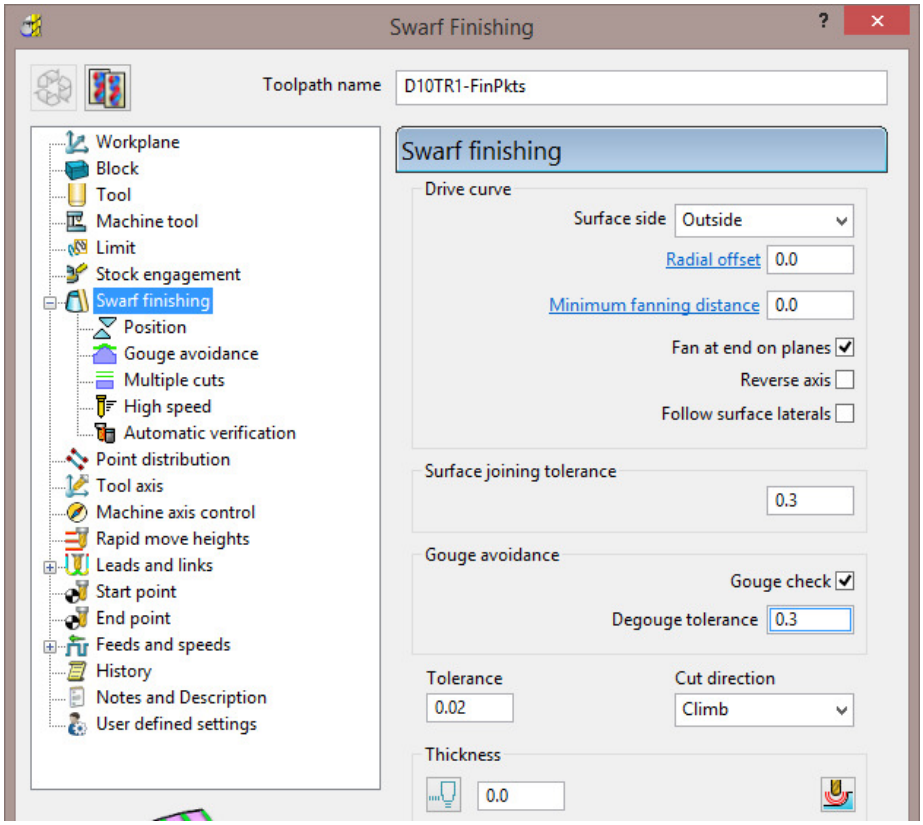
7 Set both **Lead in/out** to **Surface normal arc** with ***Angle* 90** and ***Radius* 4**.

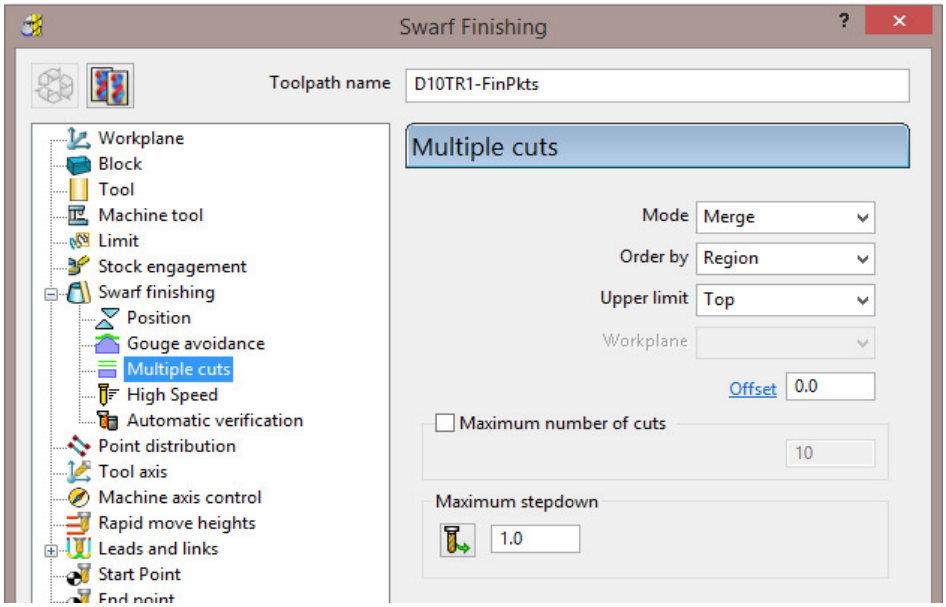
8 Select all three ***sidewall surfaces*** for the three pockets.



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.

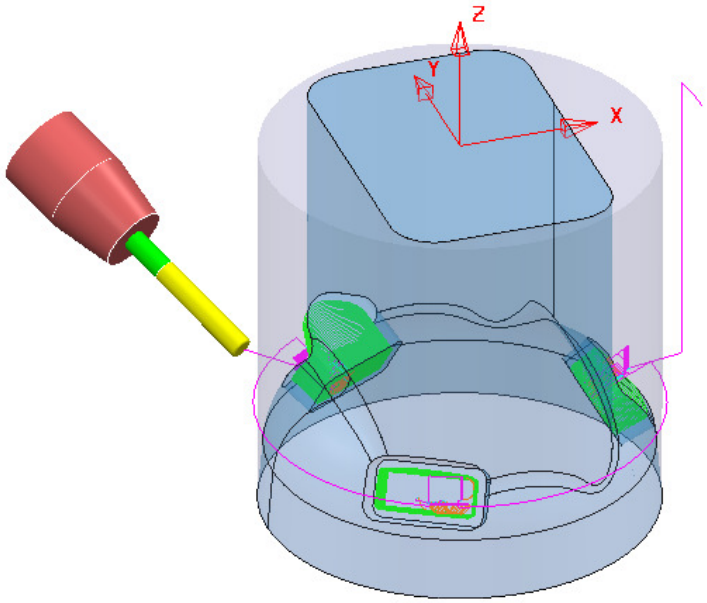




- **Mode - Merge** provides continuous tool tracks as variable offsets between the upper and lower edges of the swarfable, **surfaces**.
- Use: **Maximum stepdown 1.0**

*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

Swarf Finishing - Example 3

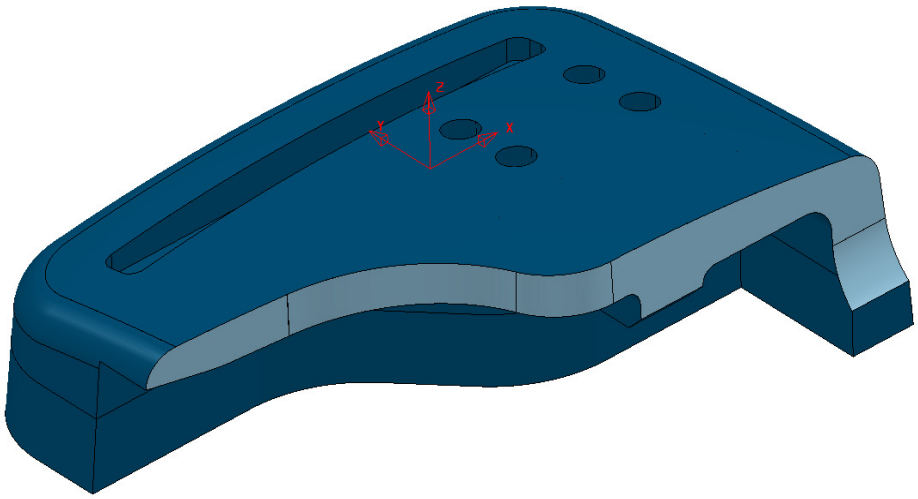
There are situations where ***swarfable surfaces*** do not provide an acceptable **tool alignment** around the component form. In these cases the only obvious solution is to use a separate CAD package (eg **PowerSHAPE**) to re-model the affected **surfaces**.

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 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**.

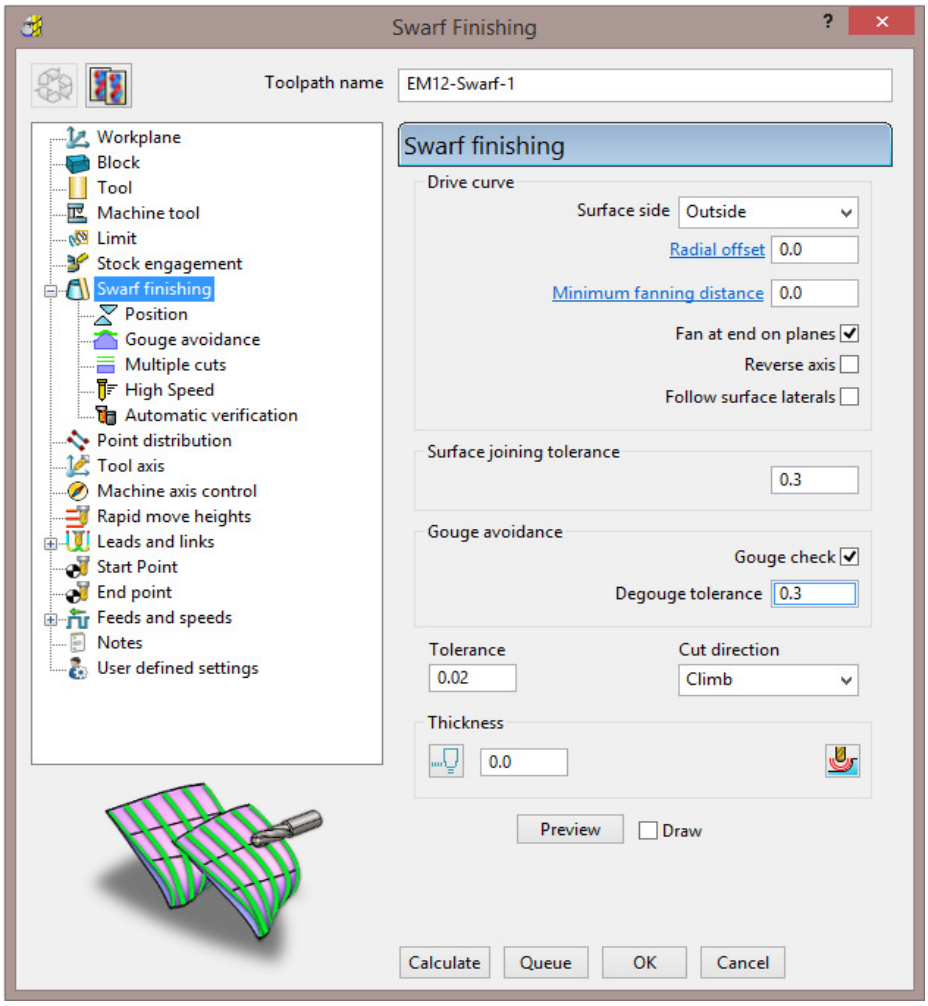
9 Set the ***Start Point*** to **First Point Safe** and ***End Point*** to **Last Point Safe**.

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

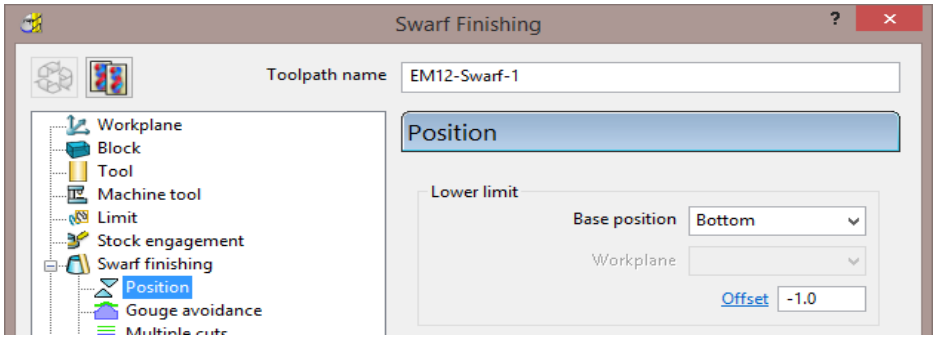
11 ***Select*** the 5 **Surfaces** required to be **Swarf** machined as shown ***pale blue*** in the above illustration.

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.

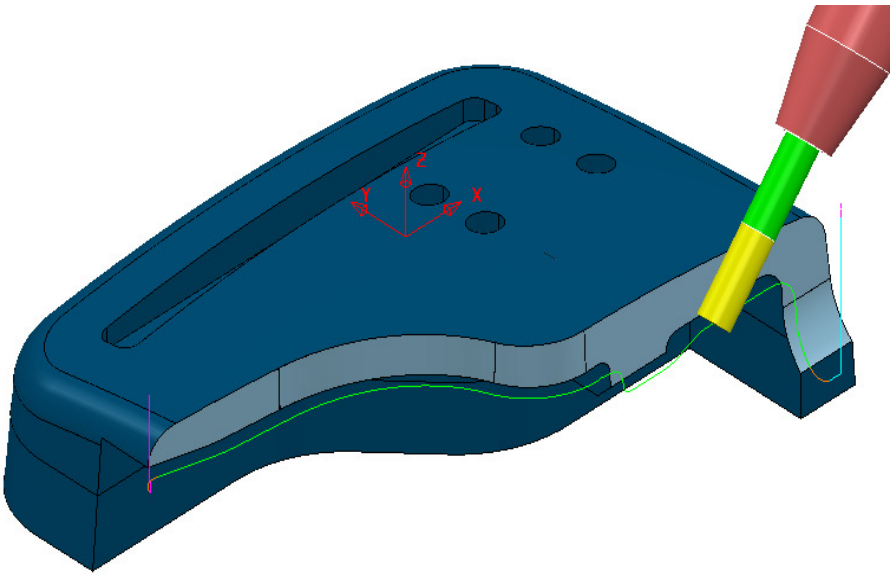


14 Select the **Position** page and input ***Offset*** **-1.0**



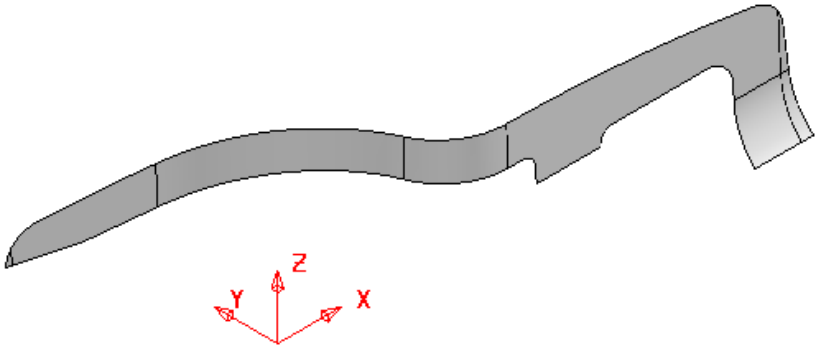
15 Select **Calculate** to process the toolpath.

16 Run a slow, ***Point to Point***  , **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.

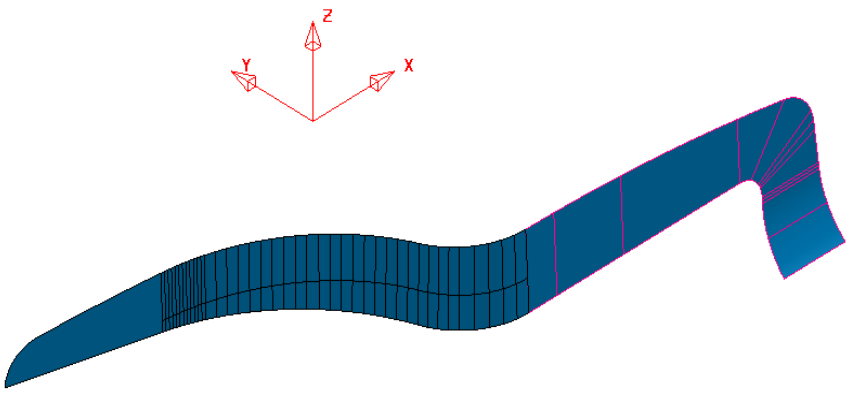
This strategy can be improved by using different **Surfaces** that are specifically designed to provide a better tool alignment during **Swarf Finishing**.



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**:

18 **...\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.

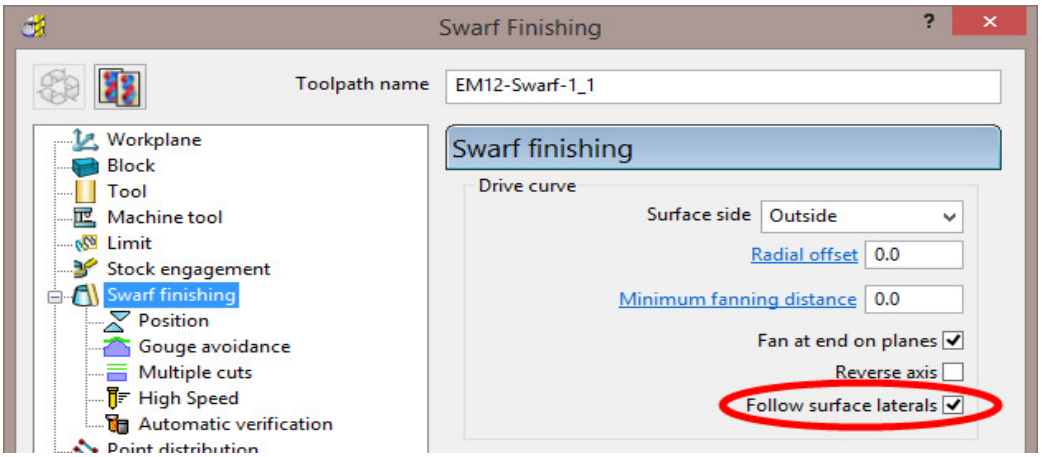
Do Not Delete *the **5 surfaces** used in the original **Swarf Finishing** strategy. They are still required for accurate gouge checking.*

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

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

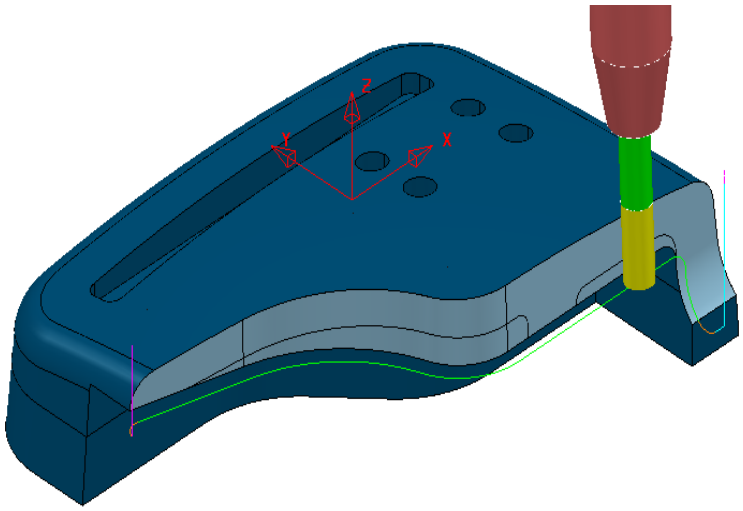
21 ***Select*** the new, imported reference **Surfaces**.

22 In the main page select **Follow surface laterals** to make the **tool** to align to the **surface curves** wherever possible.



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

24 Run a slow, ***Point to Point***  , **Simulation** and observe the changes in **tool alignment** as it runs along the tool track.



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

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

...\COURSEWORK\PowerMILL-Projects\SwarfSurf-EX3

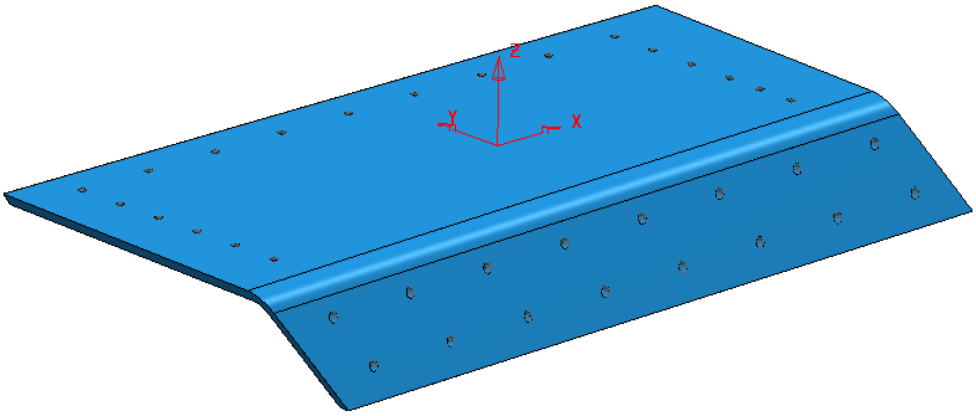
Swarf Finishing - Example 4

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



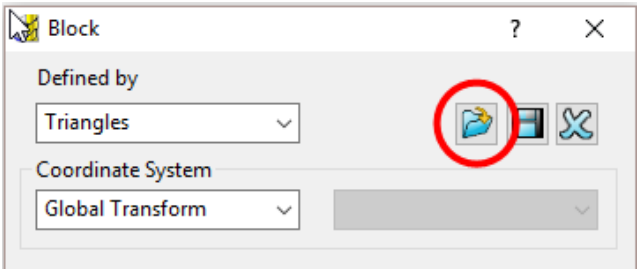
3 Select **File > Save Project As**:

...\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***.

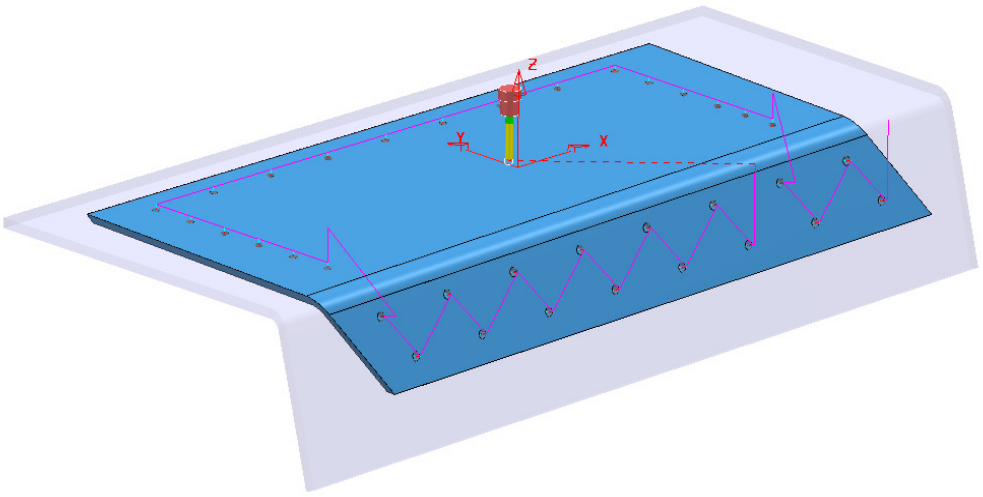


7 Select the ***Load block from file*** option. 

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

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

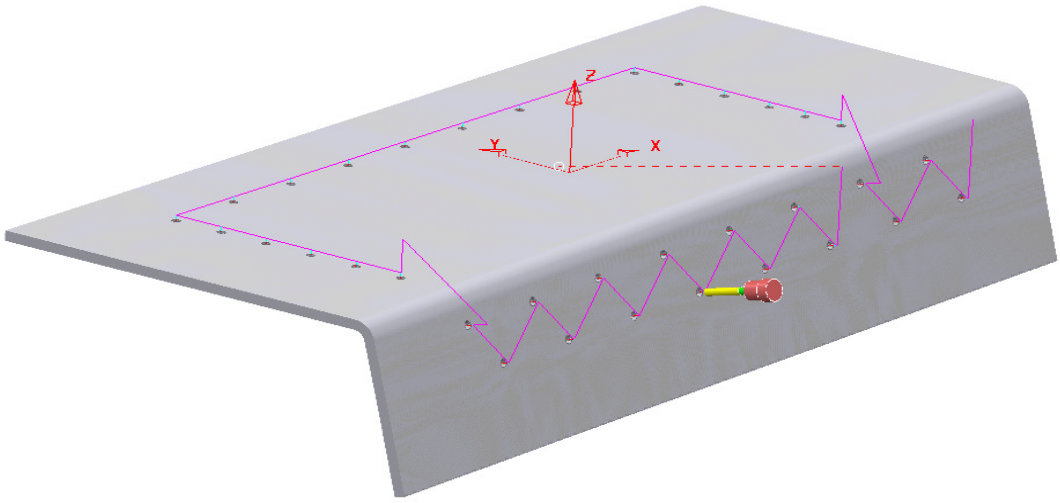
9 Select **Accept** to close the dialog.



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 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.

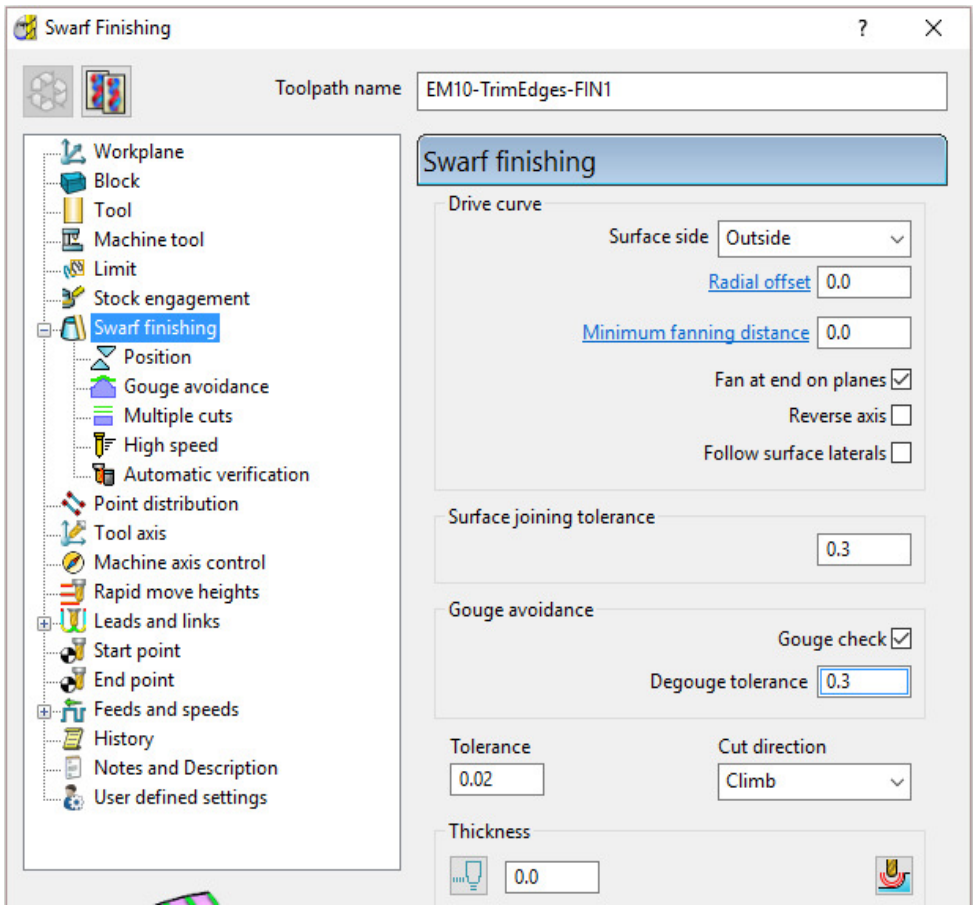
14 **Activate** the **Tool *EM10***.

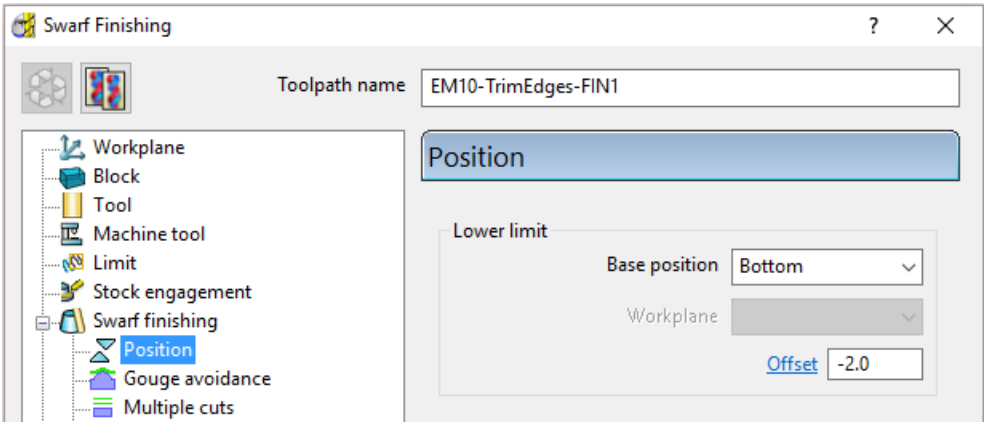
15 In the **Leads and Links** dialog set both **Lead in** and **Lead out** to **Vertical arc** with **Angle 90** and **Radius 5**.

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

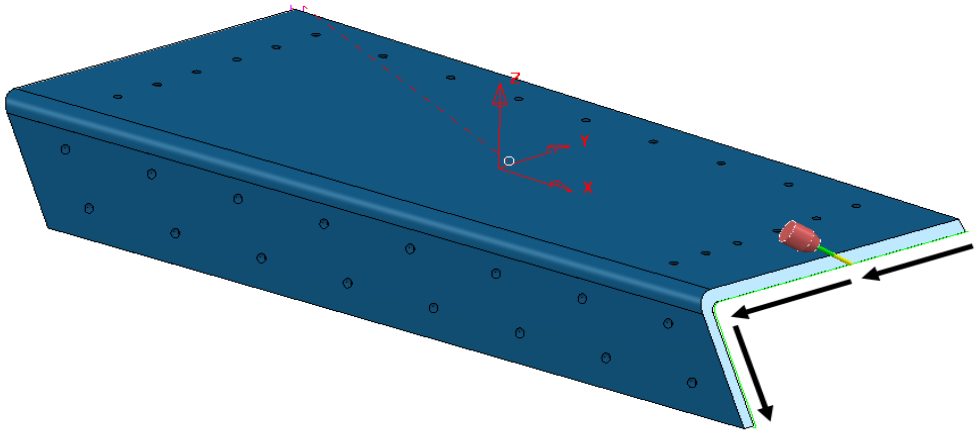
18 Enter the values into the pages of the **Swarf Finishing** dialog exactly as shown below.





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

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

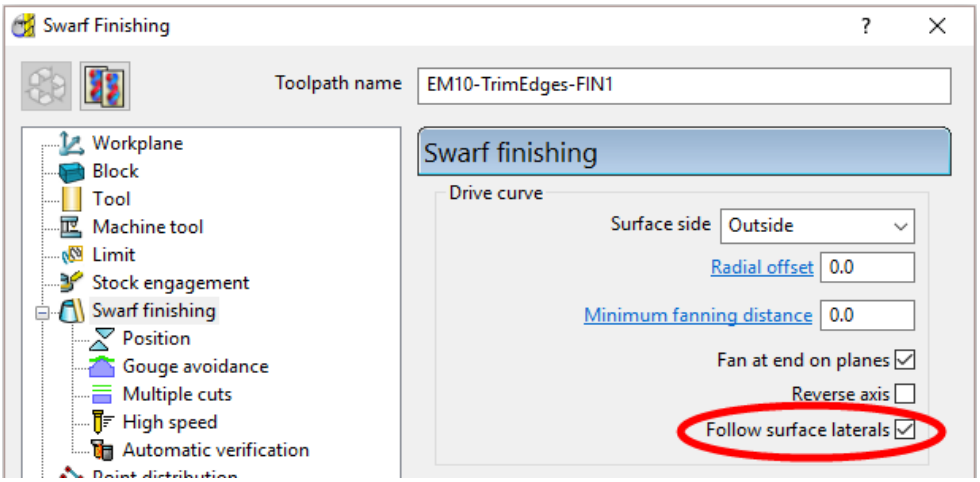


Due to the component edges being planer the alignment of the **EM12** tool is not compelled to remain ***Normal*** to the main ***Angle Plate*** surfaces. In the above image the tool has remained cranked over after tracking around a curve in the edge form. There are too many choices for the cutter to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

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

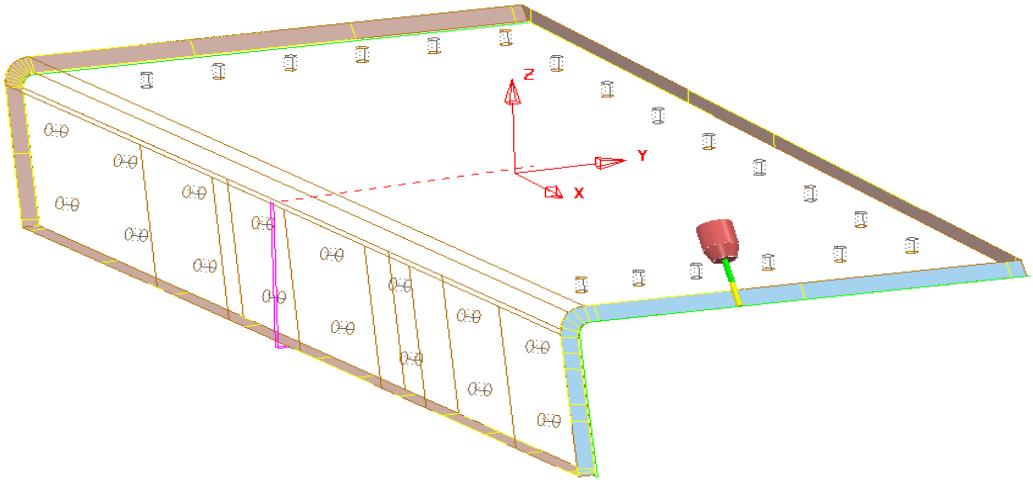
22 Select the option to ***Recycle***  the toolpath.

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



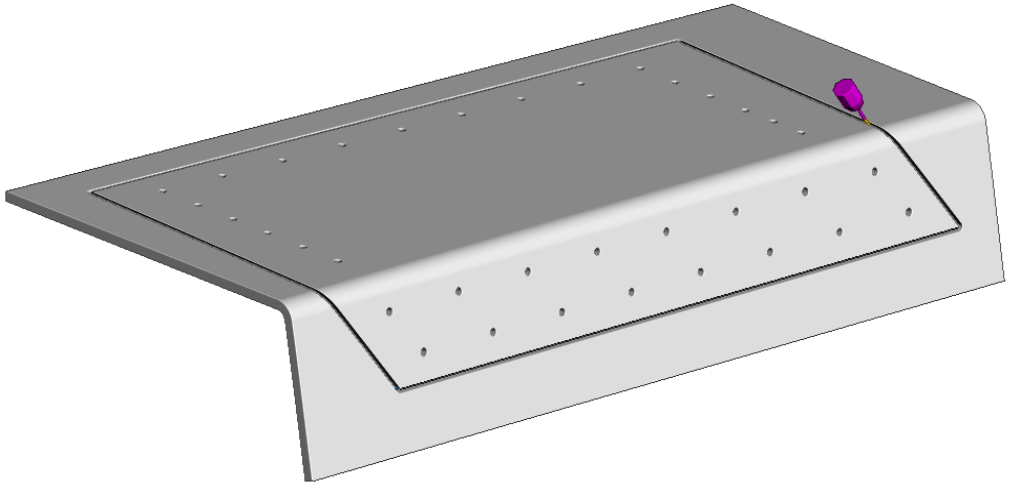
24 In the **PowerMILL *explorer***, **Levels and Sets** folder, right mouse click on the **Level *NewEdges*** and from the local menu click on **Select Surfaces**.

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 and have been designed with ***laterals*** that produce the above tool alignment.

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



27 Select **File** - **Save Project** to update the stored **Project**.

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**)).

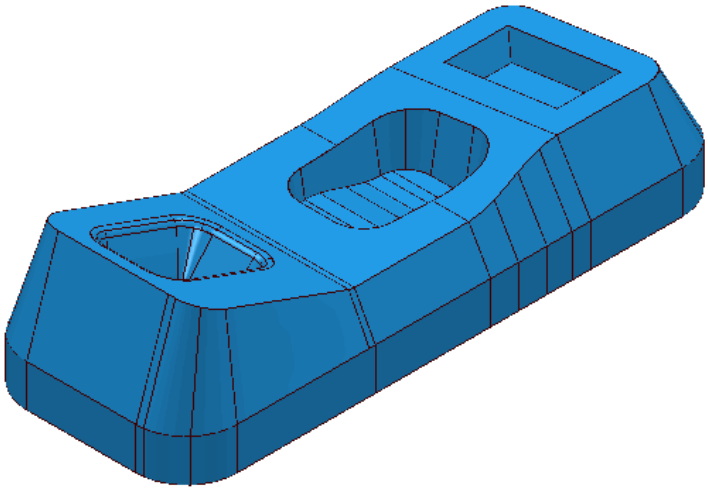
The other alternative is to create 2 separate **Patterns** that define both edges running along the area to be ***swarf machined***. The user can then use the **Wireframe Swarf Finishing** strategy.

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** - **Datum**.

5 **Calculate** a **Block** using ***Defined by*** - **Box** to the **model** dimensions.

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* 15**

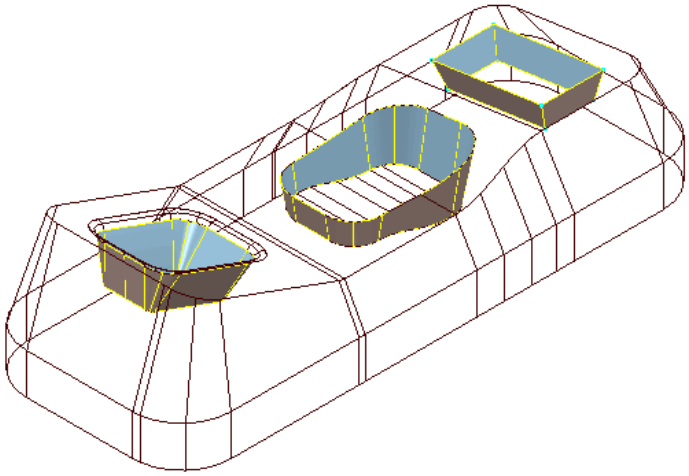
Holder-1 ***Lower Dia* 15** ***Upper Dia* 25** ***Length* 15**

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

Overhang 50

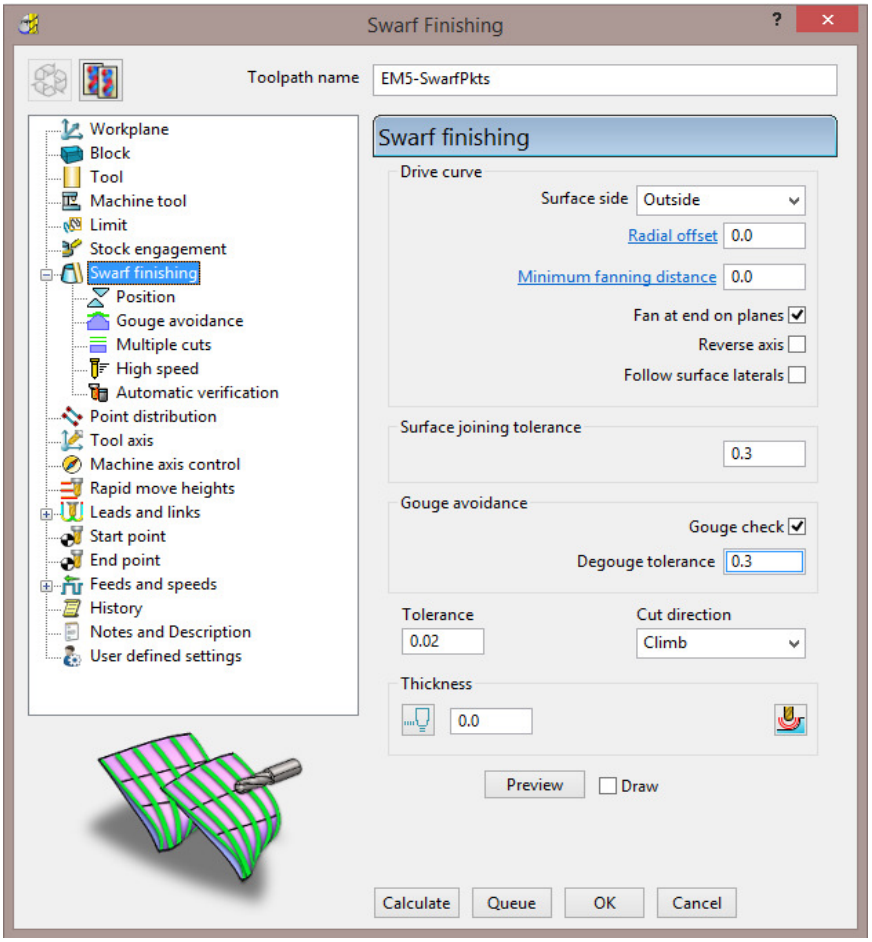
7 **Calculate** the **Rapid Move Heights**.

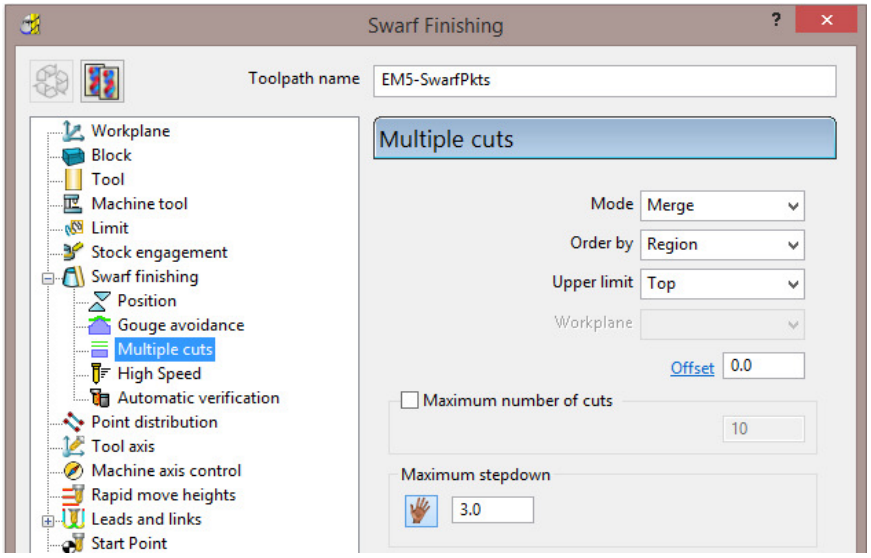
8 Select the ***sidewall surfaces*** on each of the 3 pockets in the component.



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

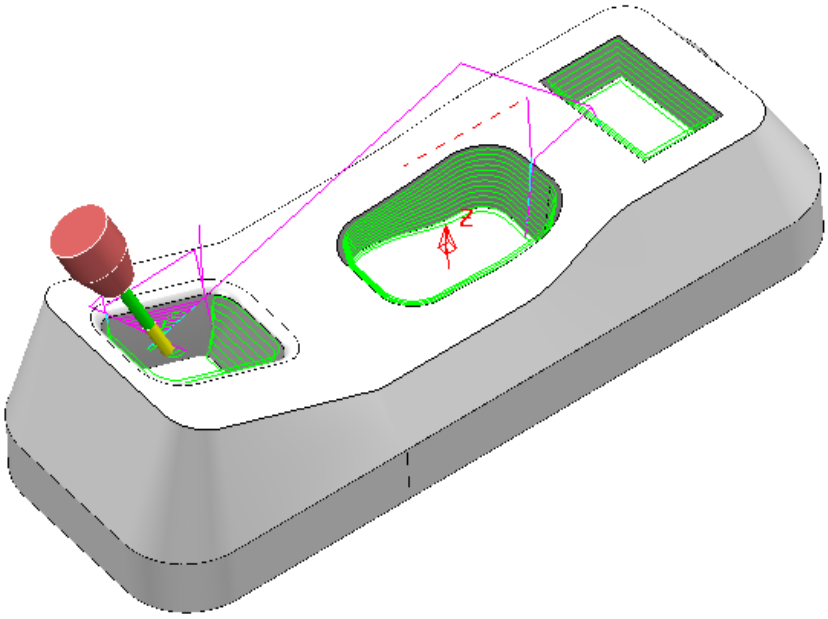
10 Enter the values into the 2 pages of the **Swarf Finishing** dialog exactly as shown.



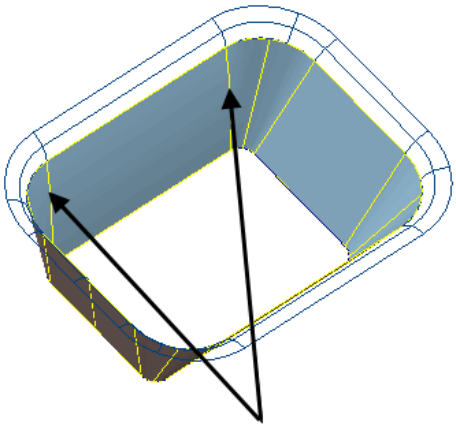


- ***Mode*** – **Merge** and ***Max Stepdown*** - 3.0

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.



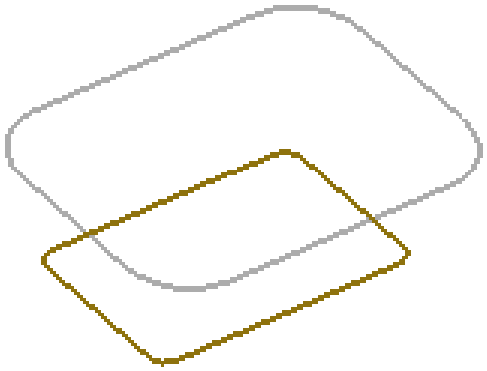
On closer inspection it is evident that some of the curves linking across the ***sidewall surface*** are convex and as a result make this area un-swarfable.

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)**.

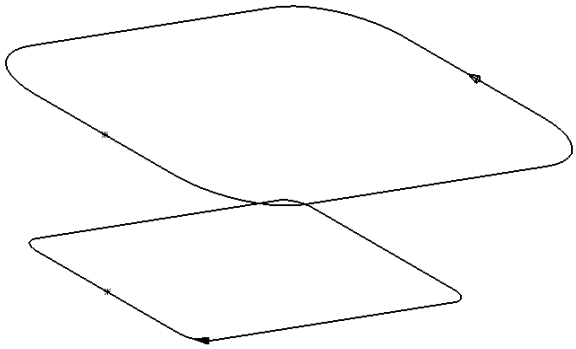
16 ***Rename*** the new, copied **Pattern** as **Lower**.

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.*

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.*

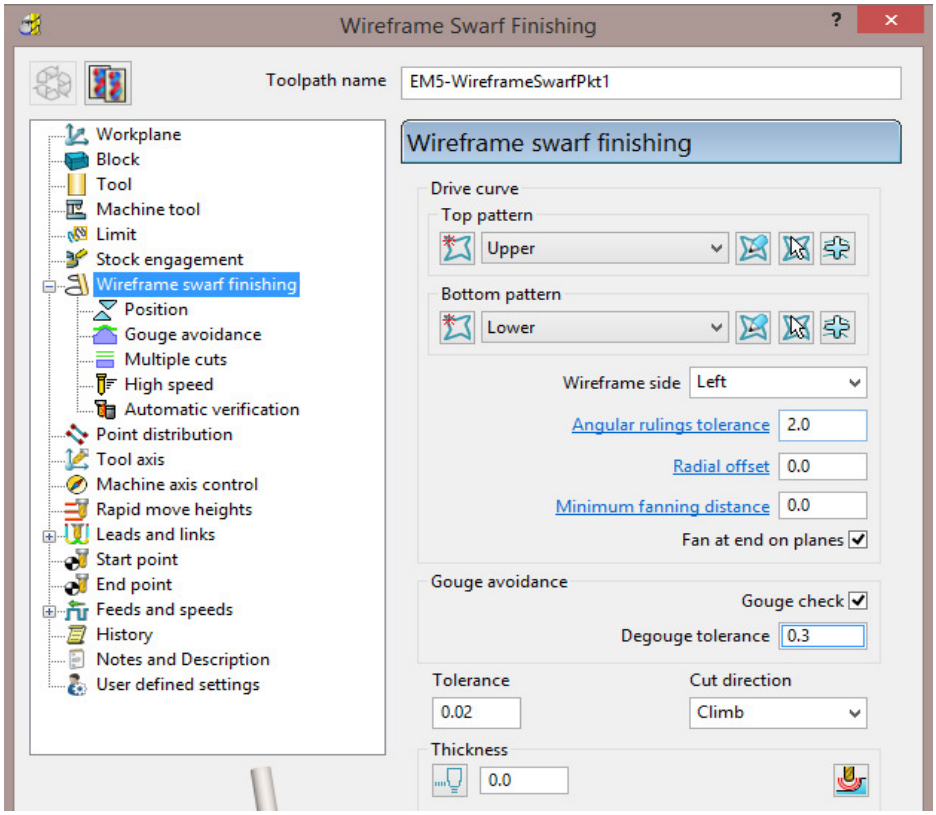
*In this case, to create a **Climb** milling strategy, the lower **segment** must be reversed.*

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**.

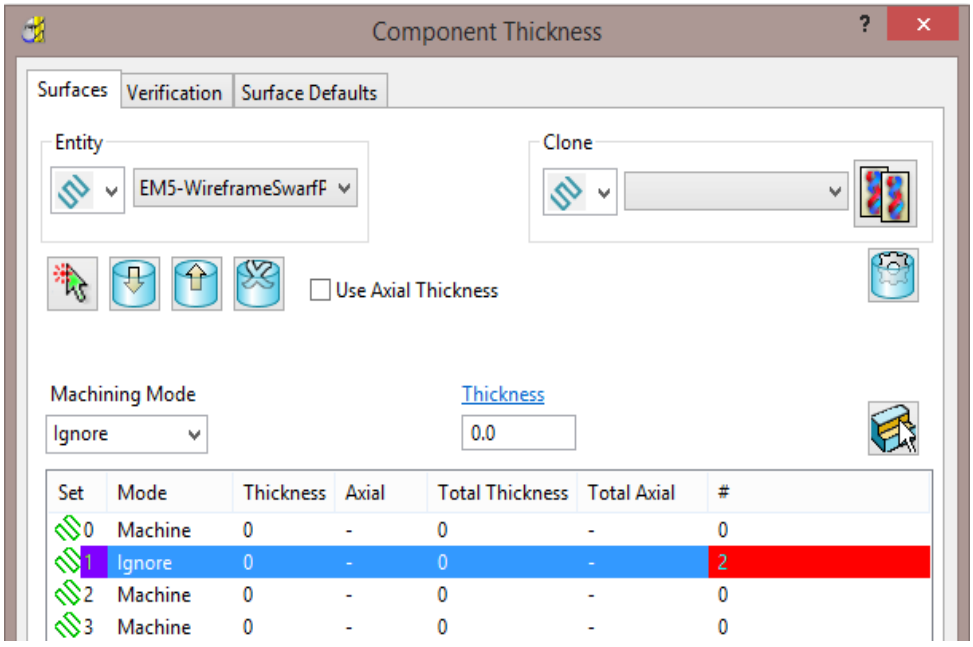
21 Select the **Toolpath Strategies** and from the **Strategy Selector > Finishing** options, select **Wireframe Swarf Finishing**.

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

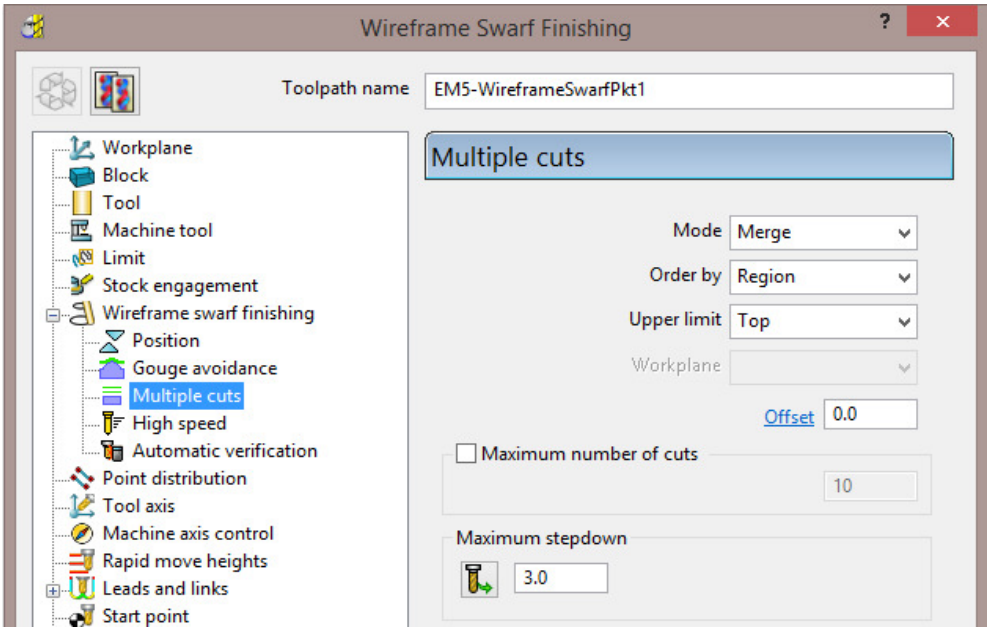


23 Select **Component Thickness**  to open the following dialog.

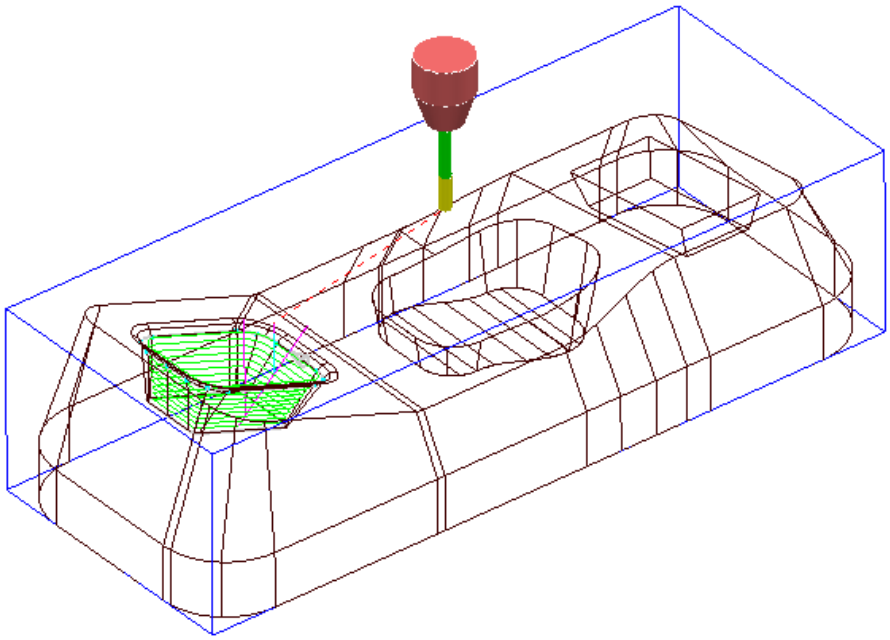
24 Select both the ***sidewall*** and ***fillet*** **surface** (flanked by the **Patterns**) and ***acquire*** them to a selected **row** (with the **Machining Mode** set to **Ignore**) in the **Component Thickness** dialog.



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



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



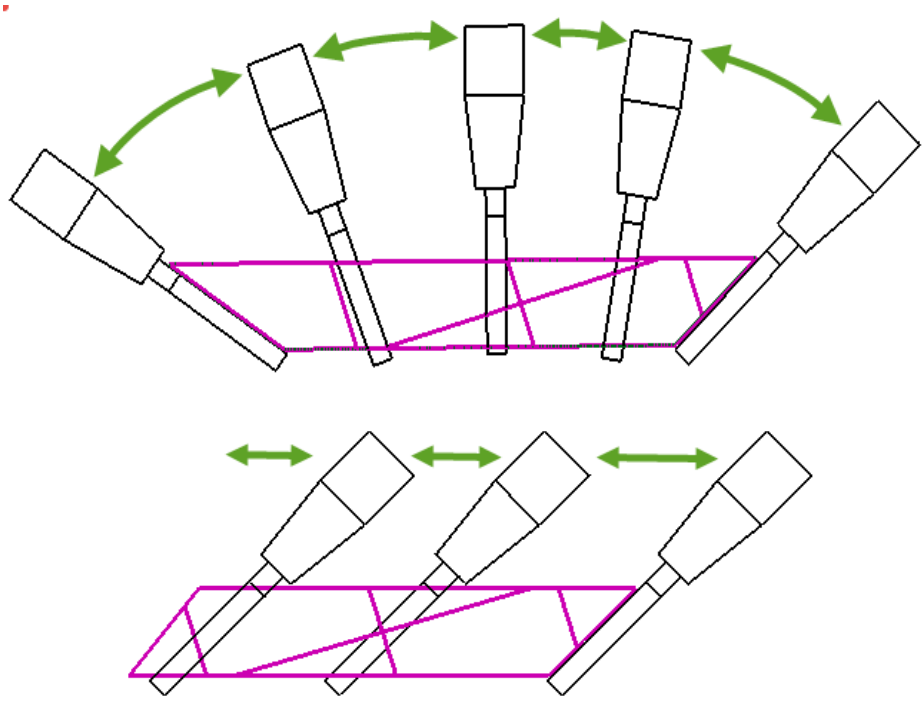
The new **Wireframe swarf finishing** strategy has successfully been applied between the ***Upper*** and ***Lower*** **Patterns**.

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

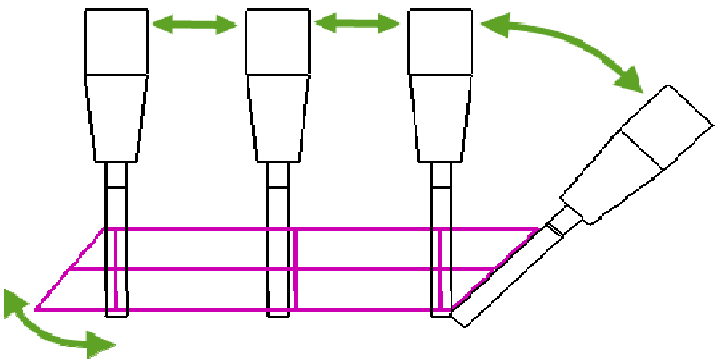
...\COURSEWORK\PowerMILL-Projects\Wframe-Swarf

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**.

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:

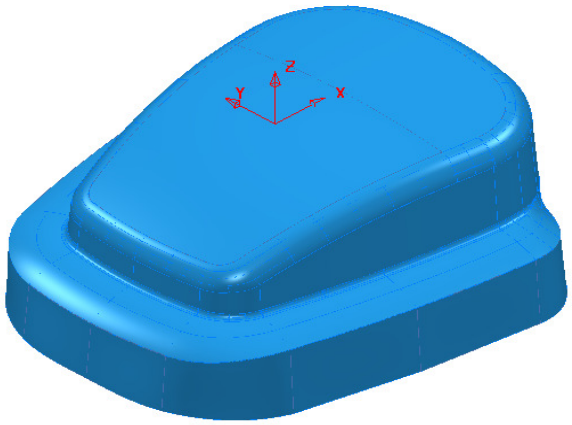
- Multiple surfaces can be machined.
- Machining of undercut regions is possible.
- The toolpath passes are not determined by the surface parameterisation.
- The region being machined is not forced to be an entire surface or parametric box.
- The stepover is defined in 3D, giving total control over the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

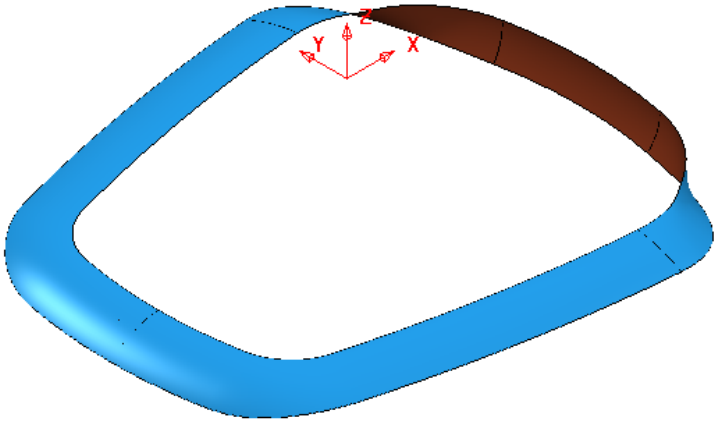
4 **Activate** the **Workplane MC-Datum**.

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.

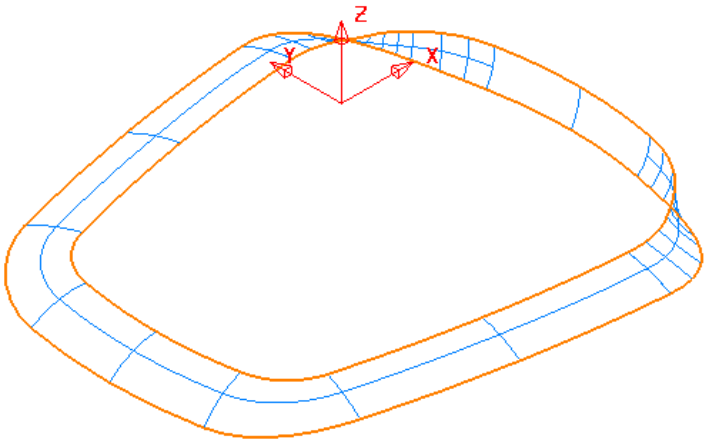
8 Select the following 5 **Surfaces** and **Blank** the rest (Ctrl K).



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**.

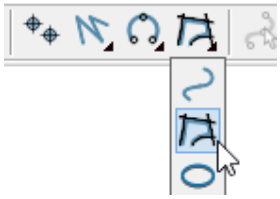


A Pattern extracted from a selection of surfaces will automatically become Embedded (Inherits the surface normals etc).

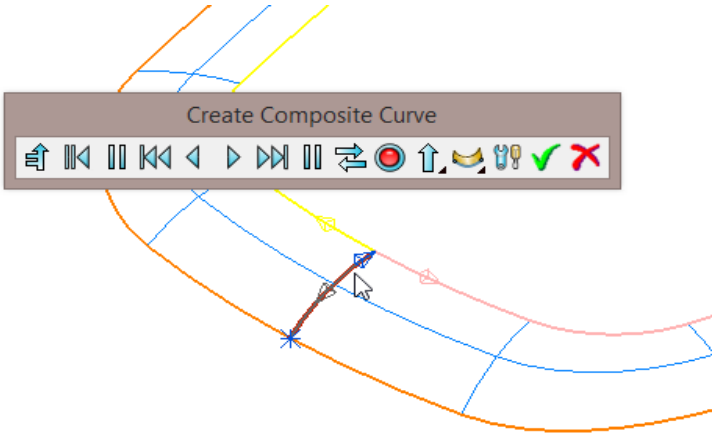
The Pattern symbol changes to identify it as being Embedded.

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.

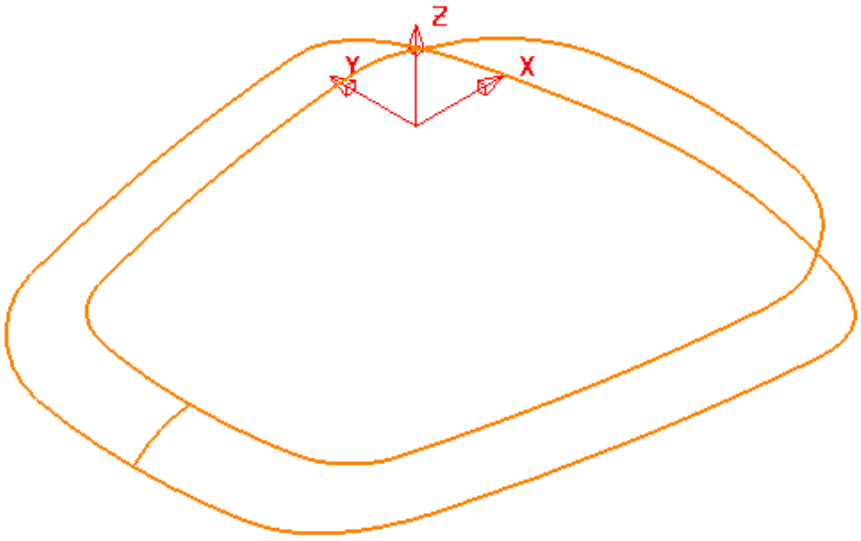


14 With the **Create Composite Curve** toolbar open click near to the upper end of the first surface lateral (As shown arrowed below).



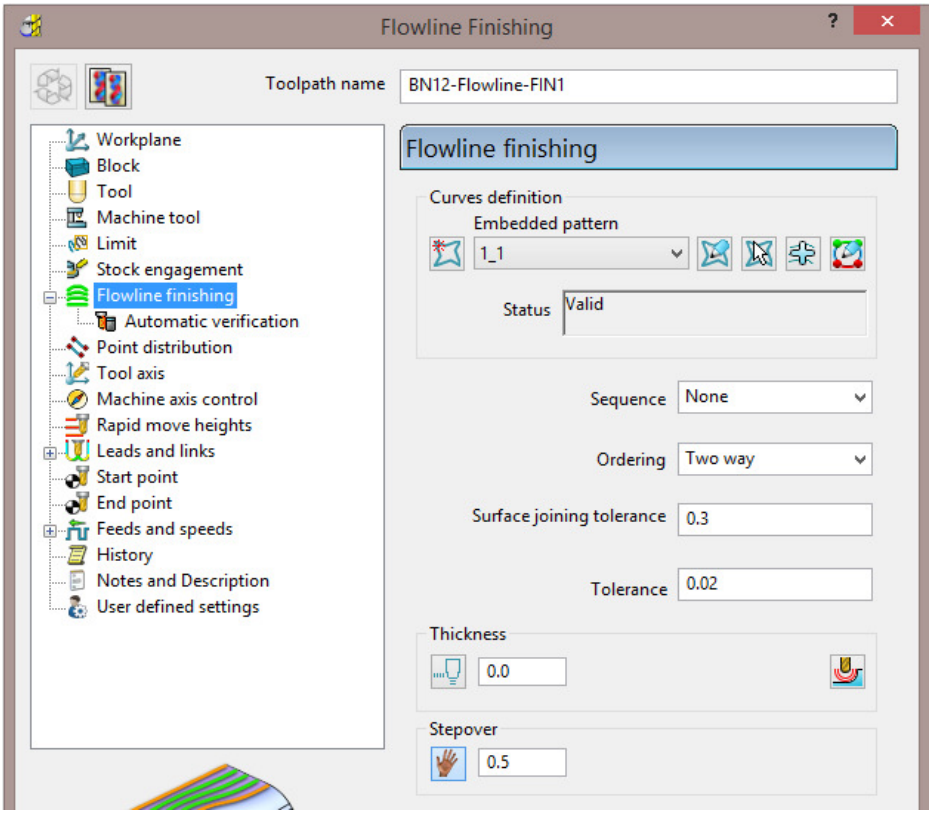
15 In the toolbar select the **red disc** followed by the **green tick** to both accept the new segment and close the toolbar.

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

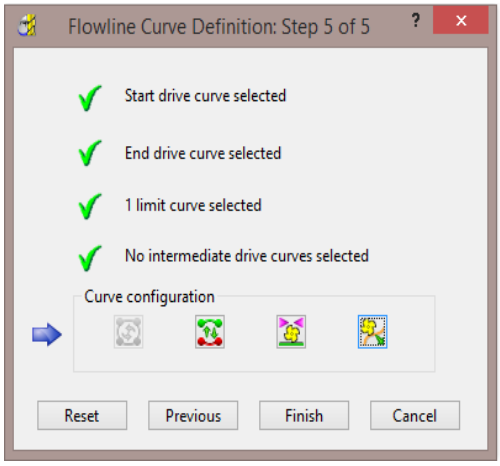


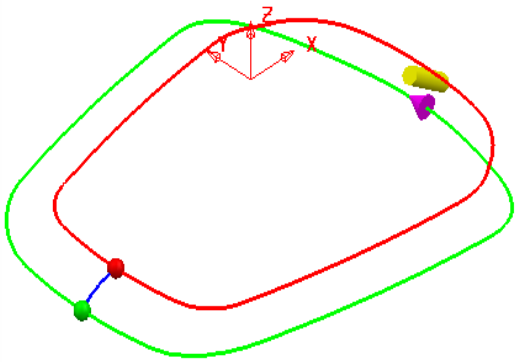
16 From **Toolpath Strategies**  open the **Finishing** options and from the dialog select **Flowline Finishing**.

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



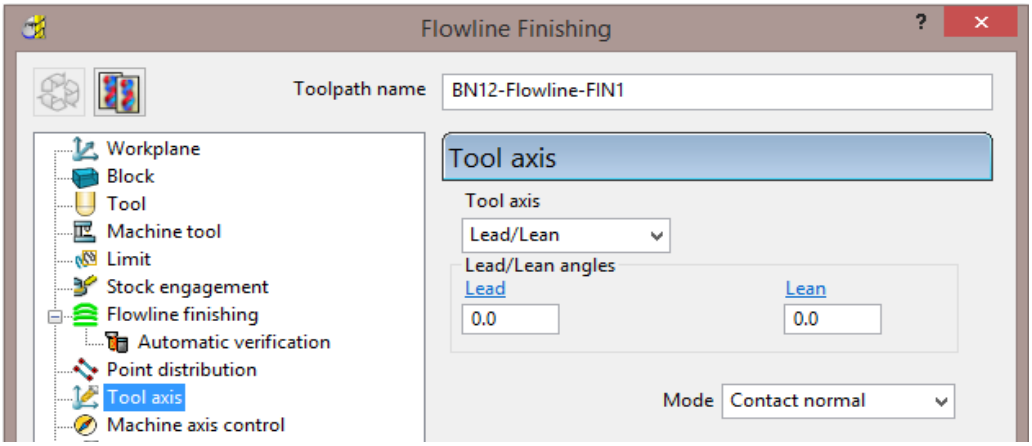
18 Still on the **main** page select the **Interactively select pattern segments for the strategy** options. 





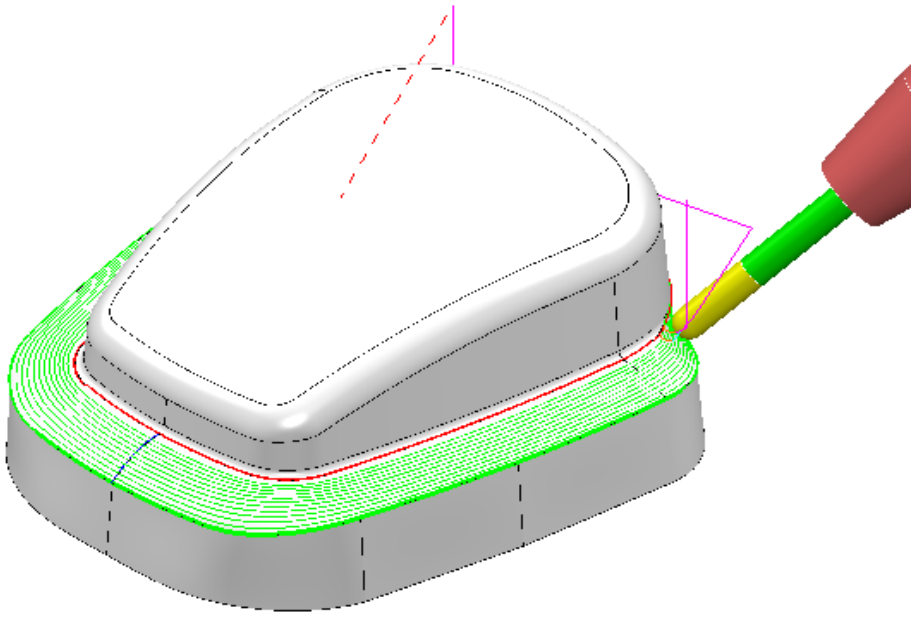
The options in the above form enable the toolpath ordering, directions, etc to be controlled before the toolpath is calculated. However, no changes are required in this instance.

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**.

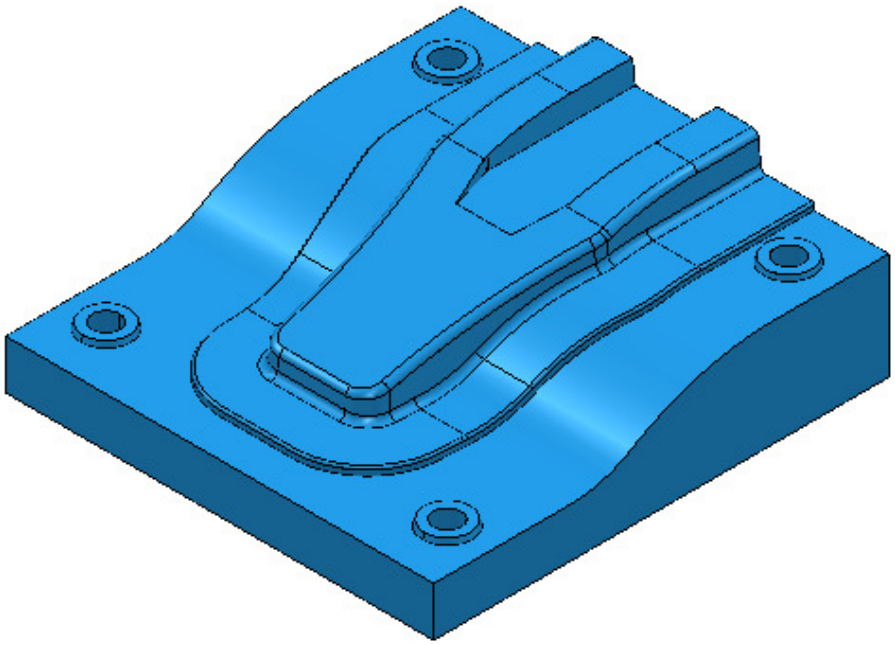
Exercise

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**.

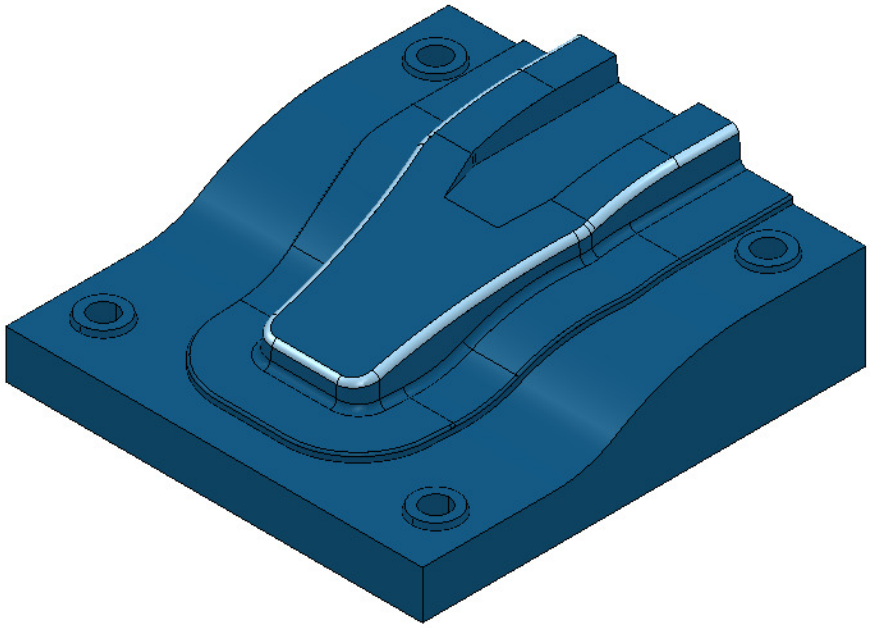
4 **Activate** the new **Workplane**.

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).



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**.*

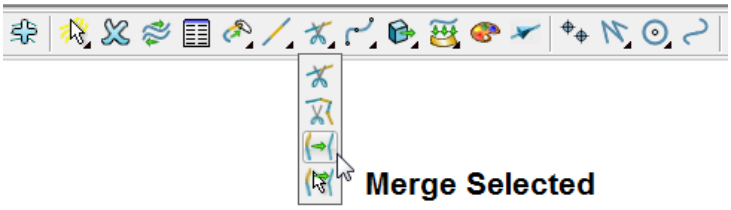
11 Select and **Blank** the whole of the **Model** and **Un-draw** the **Tool** and **Block**.

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.*

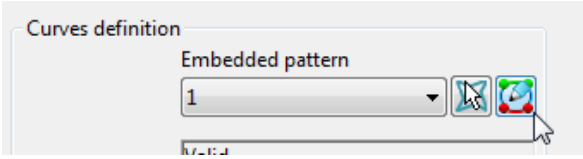
14 **Select** any multiple **segments** that define a single, ***drive*** or ***end*** curve and then use **Merge selected** to reduce these to a single segment.

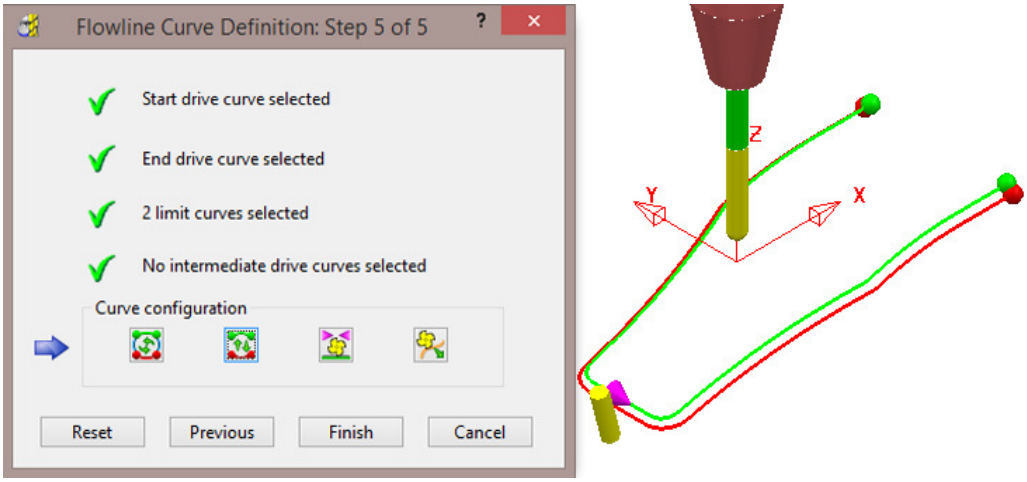


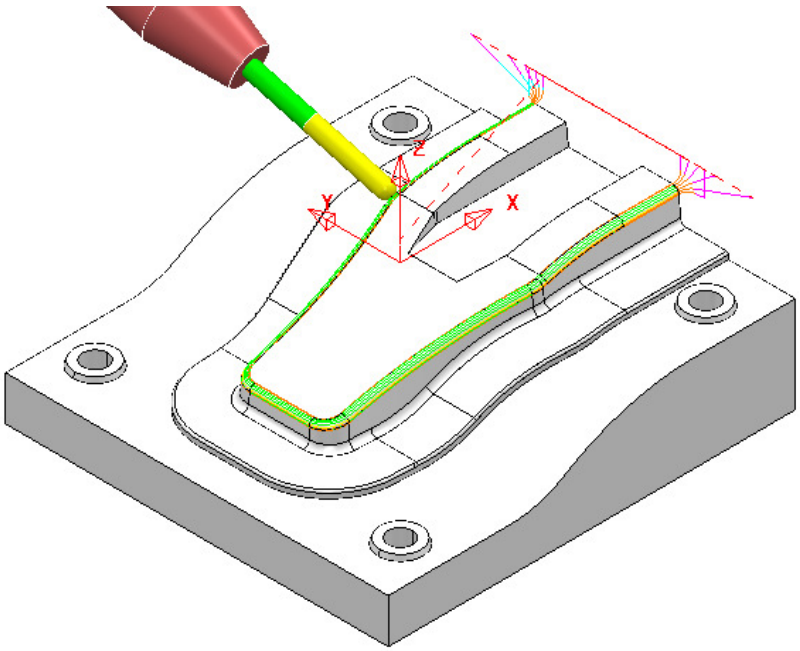
15 Accept the changes (***Green tick***) and return the **Embedded Pattern** from the **Curve Editor**.

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**:

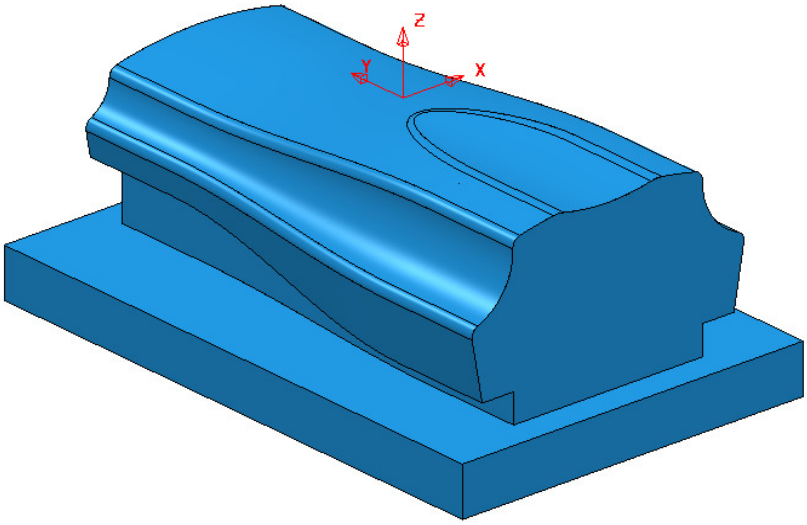
...\COURSEWORK\PowerMILL-Projects\FlowLineFillet-EX1

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

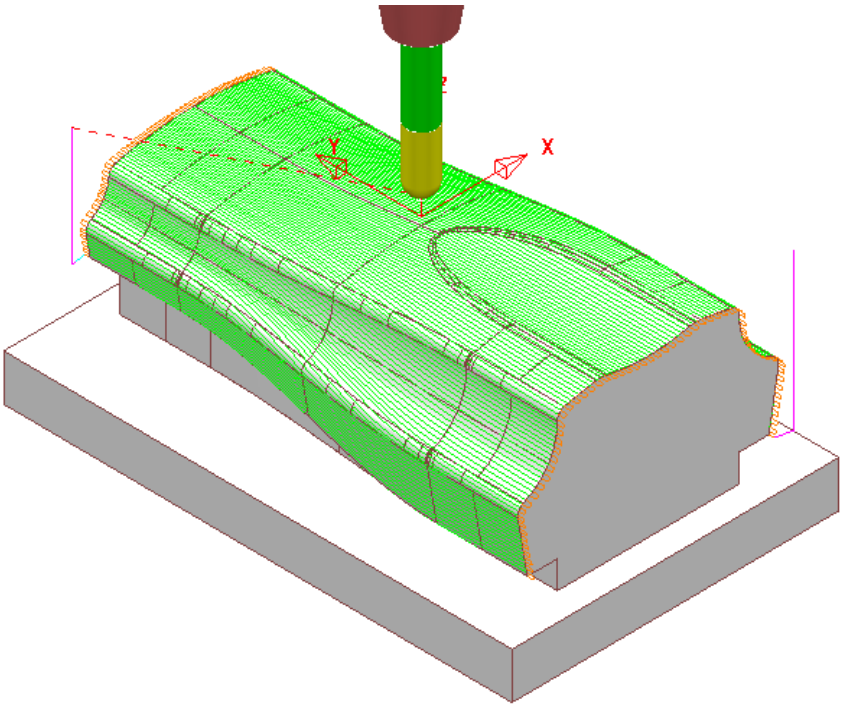


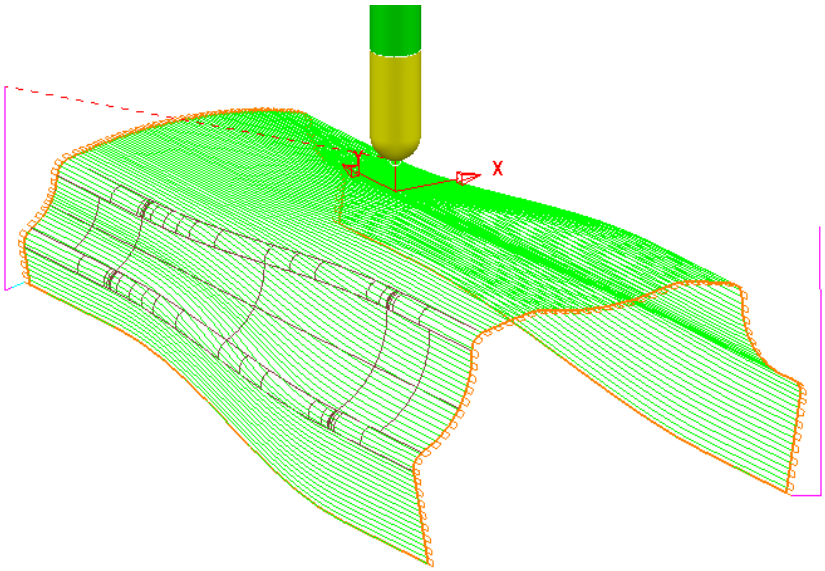
3 Select **File** > **Save Project As**:

...\COURSEWORK\PowerMILL-Projects\FlowlineWithExtraDriveCurve

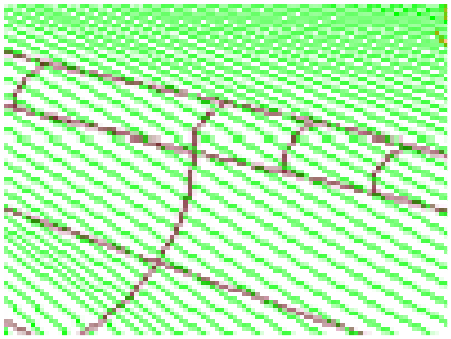
4 Select an **ISO 1** view. 

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.

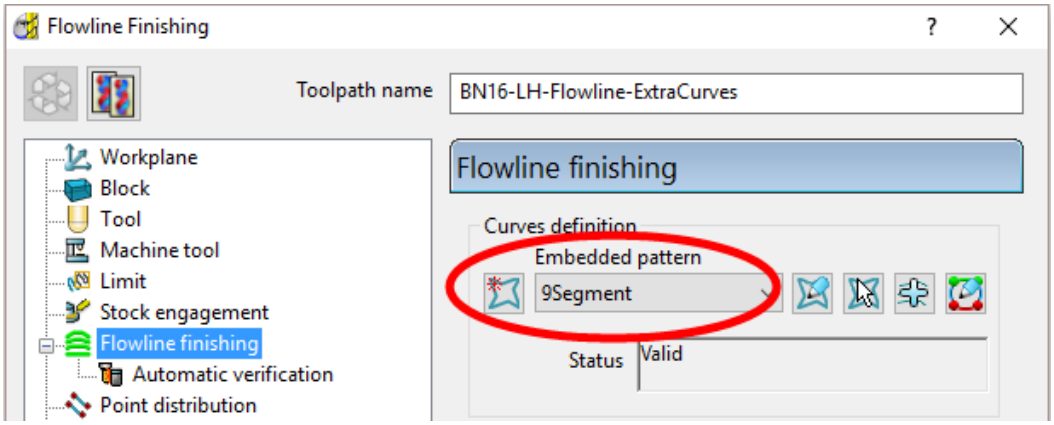


This can be changed if ***intermediate drive curves*** are included in the **Embedded Pattern**.

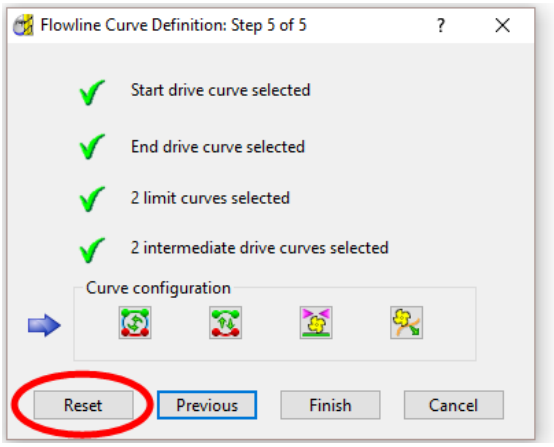
6 Right mouse click on the active toolpath **BN16-Flowline-4Curves-All** and from the local menu select **Settings** to open the **Flowline Finishing** dialog.

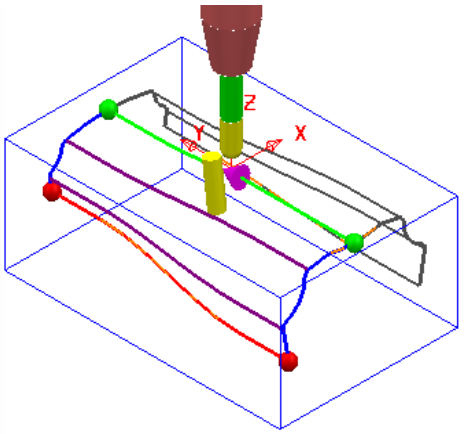
7 Select the ***Copy*** toolpath  option and then rename the new toolpath as **BN16-LH-Flowline-ExtraCurves**.

8 In ***Curves definition***, change the selected **Embedded pattern** to **9Segment**.

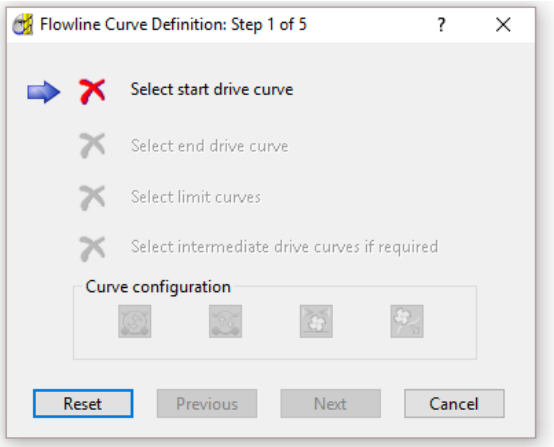


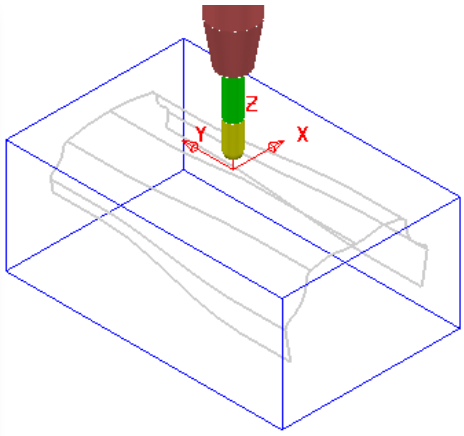
9 Select ***Interactively select pattern segments for the strategy***. 





10 Select the **Reset** button to clear the default, segment selection.



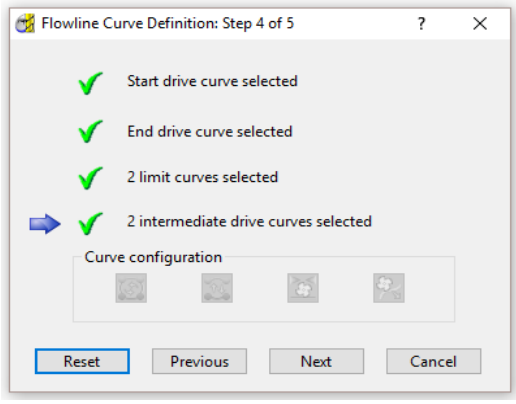


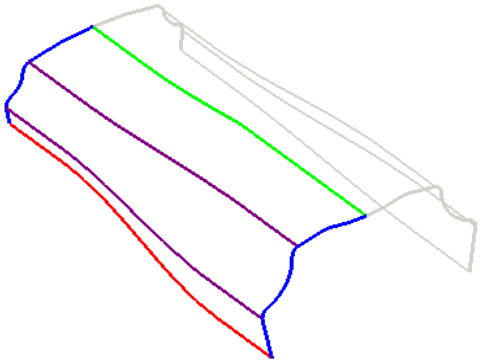
11 Select the upper central curve as the ***start drive curve*** (green)

12 Select the bottom left curve as the ***end drive curve*** (red).

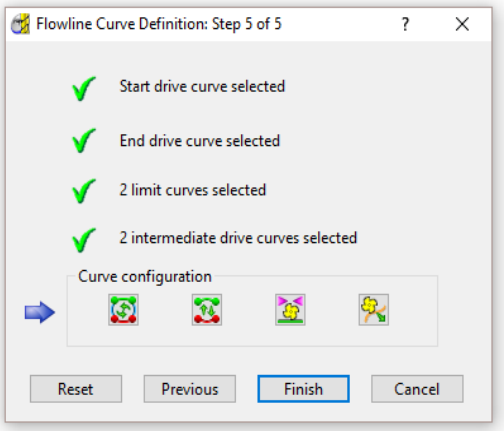
13 Shift – select the 2 ***end curves*** (blue).

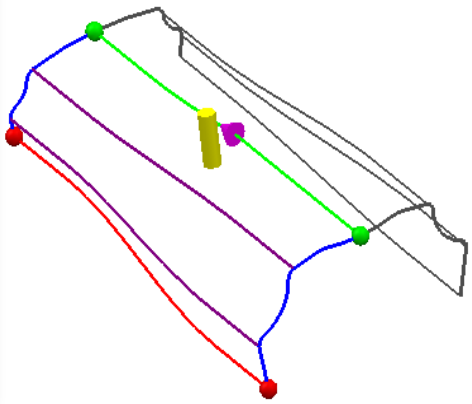
14 Shift select the 2 nearest **intermediate curves** (purple).





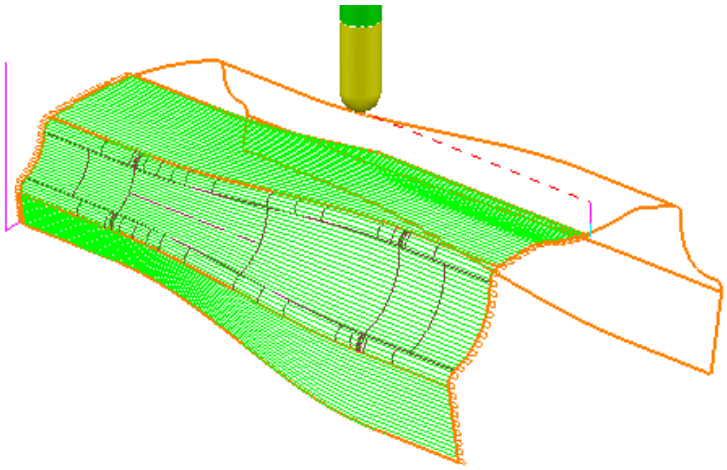
15 Select the **Next** button to return to the full preview stage.





16 Select the **Finish** button to return to the main **Flowline finishing** dialog.

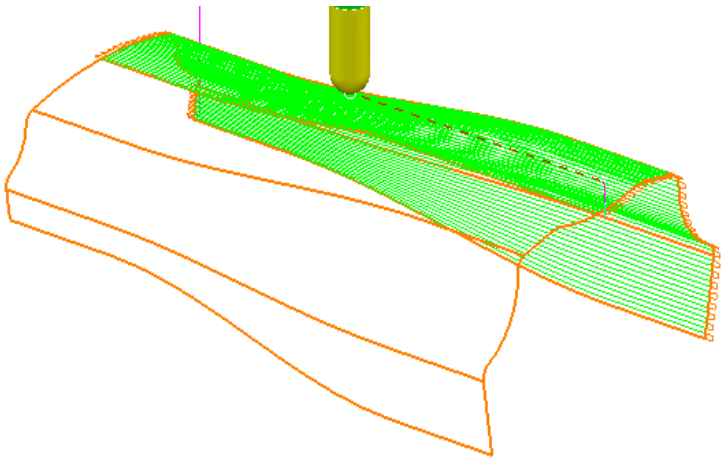
17 Select **Calculate** to generate the new toolpath then select **Close**.



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**.

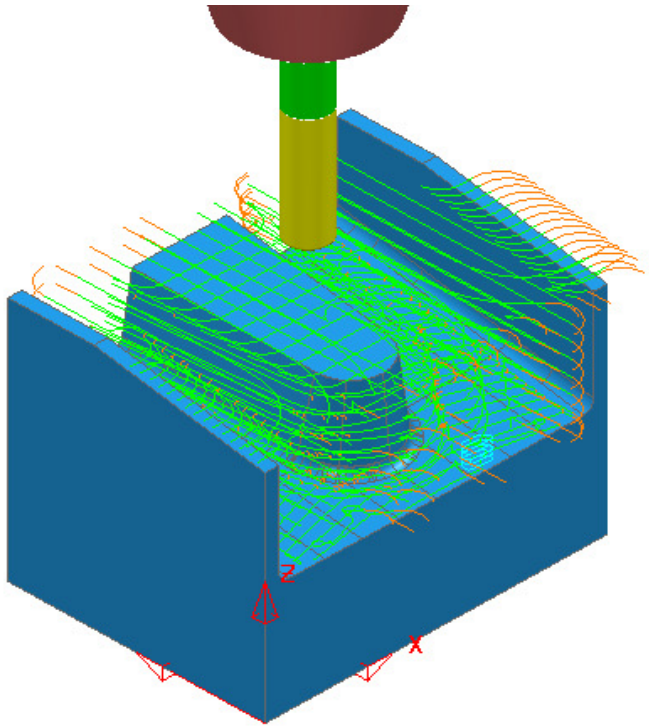
8. Auto Collision Avoidance

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**:

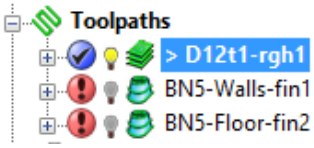
...\PowerMILL_Data\five_axis\CollisionAvoidance\AutoCollisionAvoid1_Start



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**

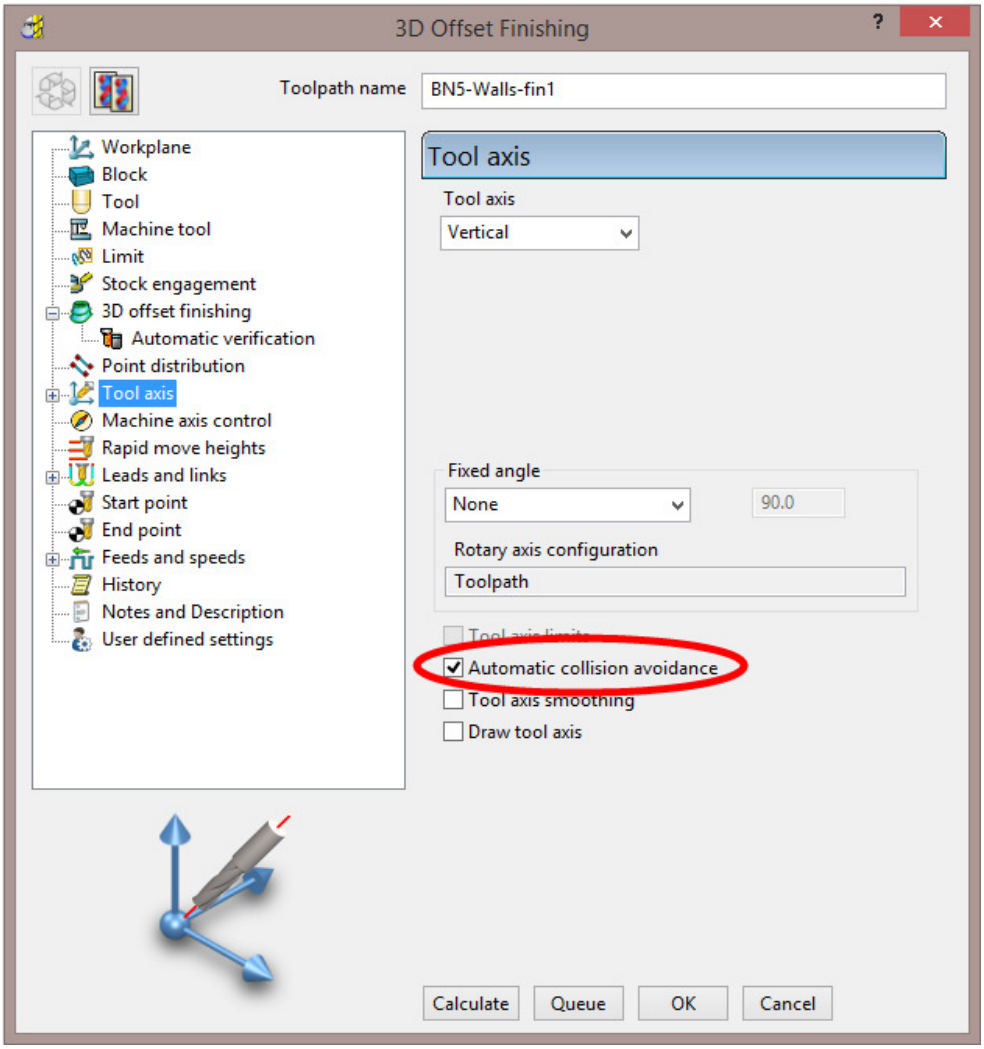
Offset finishing strategies although **Gouge** free are not **Collision** safe relative to the **Shank** and **Holder**.

3 In the **PowerMILL *explorer***, activate the toolpath **BN5-Walls-fin1**.

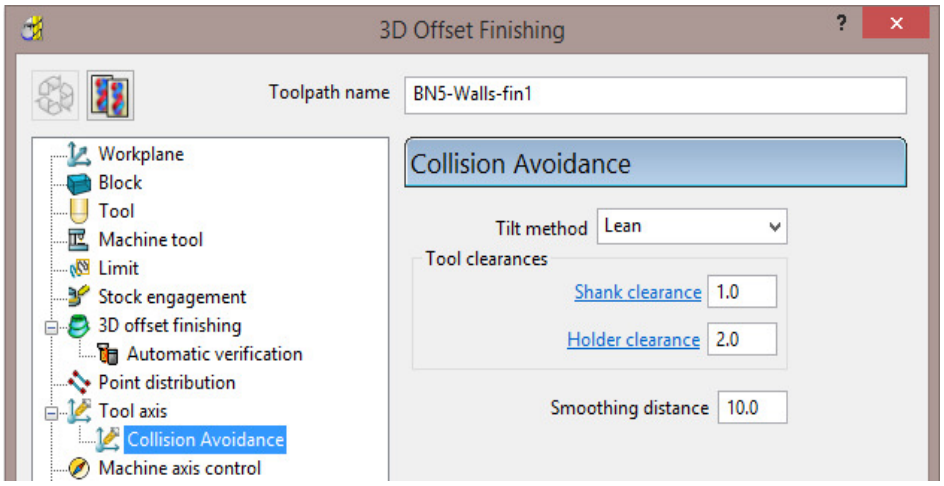
4 Right mouse click on the ***toolpath*** **BN5-Walls-fin1** and from the local menu select **Settings**.

5 Select **Recycle**  in the **3D Offset Finishing** dialog and select the **Tool Axis** page.

*Note that the selected **Tool Axis** option is **Vertical**.*



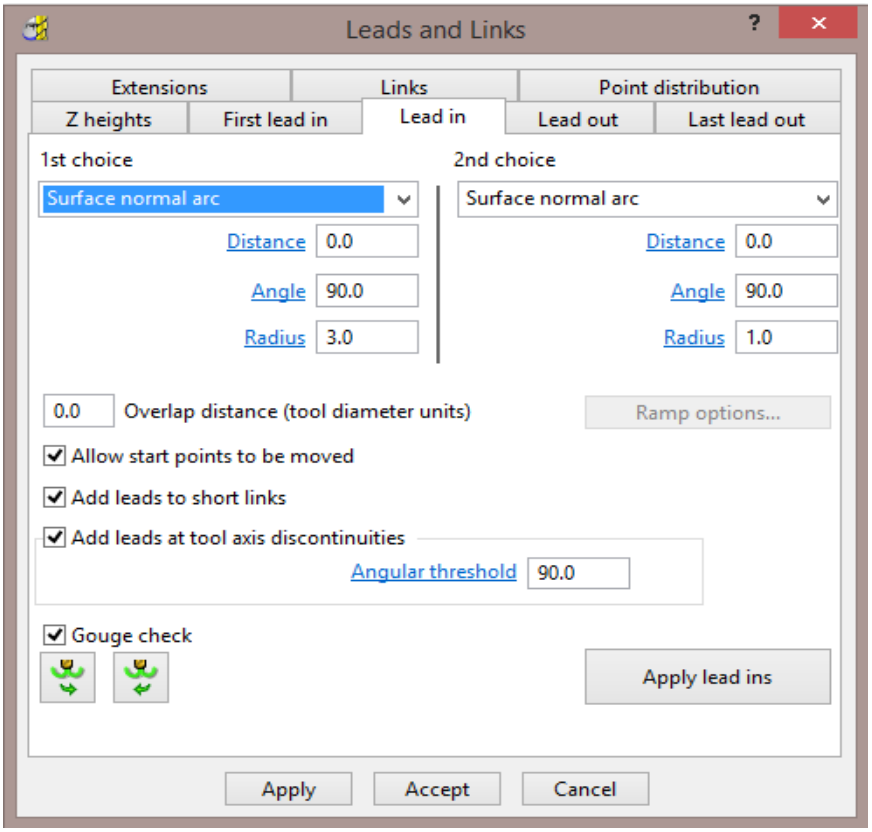
6 Select the newly available **Collision Avoidance** page (Below **Tool axis**).



- Input the ***Tilt method*** as **Lean**.
- Input ***Shank*** and ***Holder*** clearances of **1** and **2** along with a **smoothing distance** of **10**.

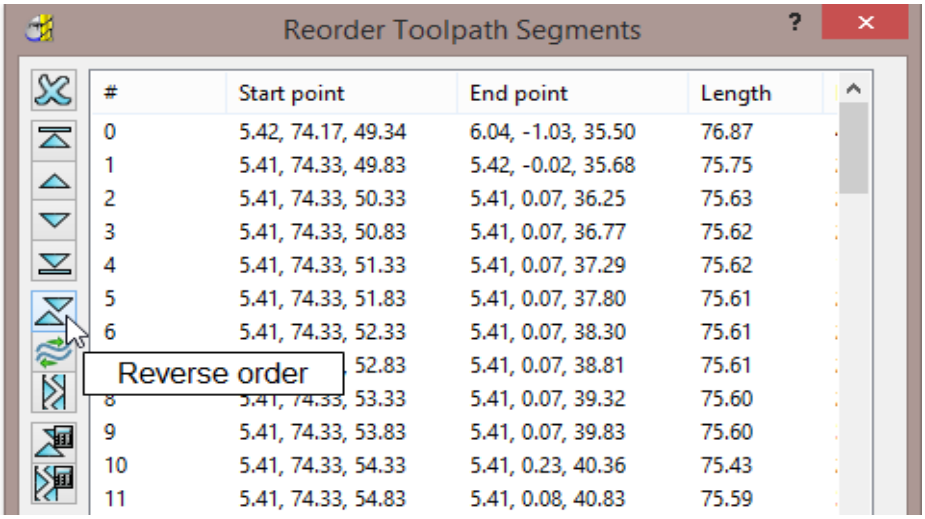
7 **Calculate** the **3D offset Finishing** toolpath.

8 Apply **Lead In** and **Lead Out** with the same **1st**/**2nd choice** – **Surface normal arc** as shown in the following dialog.



This recycled **3D Offset Finishing** toolpath will need to be retrospectively edited to make the strategy ordering start at the top of the walls.

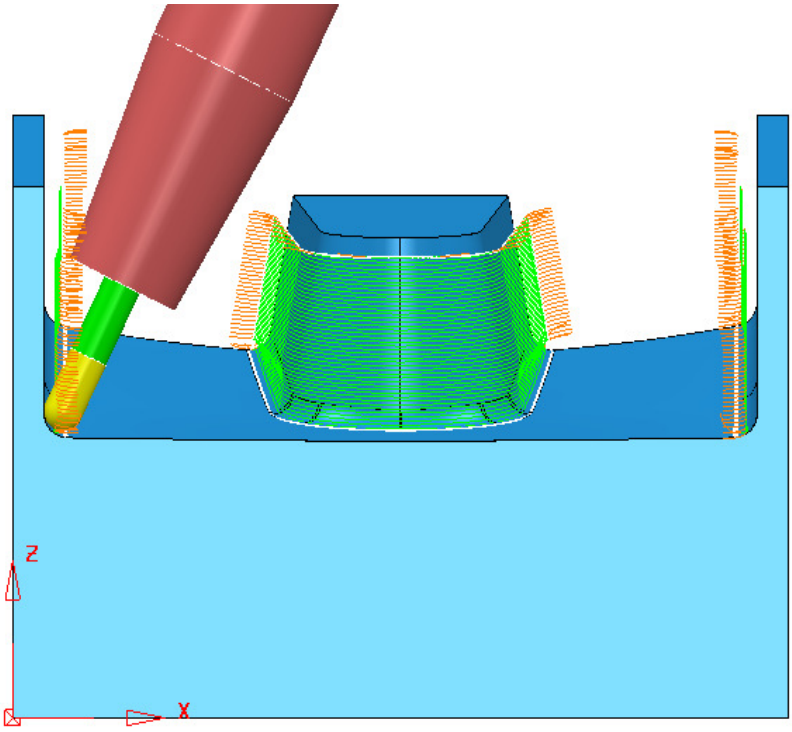
9 Right mouse click on the toolpath and from the local menu select **Edit** – **Reorder**.



#	Start point	End point	Length
0	5.42, 74.17, 49.34	6.04, -1.03, 35.50	76.87
1	5.41, 74.33, 49.83	5.42, -0.02, 35.68	75.75
2	5.41, 74.33, 50.33	5.41, 0.07, 36.25	75.63
3	5.41, 74.33, 50.83	5.41, 0.07, 36.77	75.62
4	5.41, 74.33, 51.33	5.41, 0.07, 37.29	75.62
5	5.41, 74.33, 51.83	5.41, 0.07, 37.80	75.61
6	5.41, 74.33, 52.33	5.41, 0.07, 38.30	75.61
	52.83	5.41, 0.07, 38.81	75.61
	53.33	5.41, 0.07, 39.32	75.60
9	5.41, 74.33, 53.83	5.41, 0.07, 39.83	75.60
10	5.41, 74.33, 54.33	5.41, 0.23, 40.36	75.43
11	5.41, 74.33, 54.83	5.41, 0.08, 40.83	75.59

10 In the ***Reorder Toolpath Segments*** dialog select **Reverse Order**.

11 Select a **View from front** and **Simulate** the toolpath to observe the **Automatic Collision Avoidance** in action.

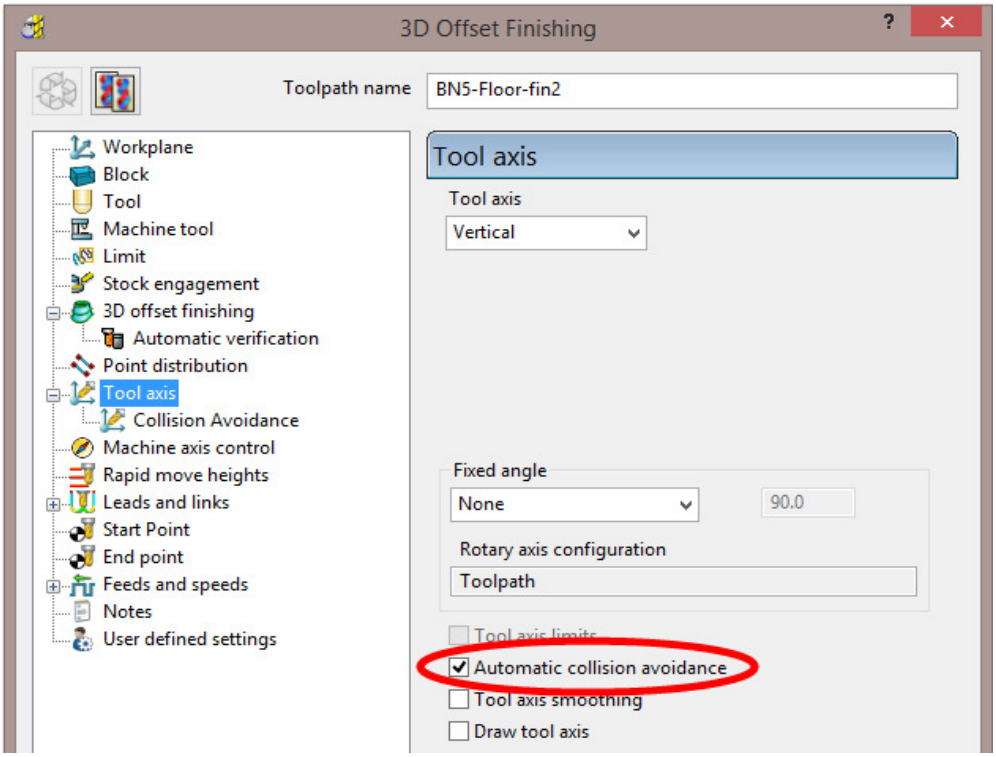


12 In the **PowerMILL *explorer***, activate the toolpath **BN5-Floor-fin2**.

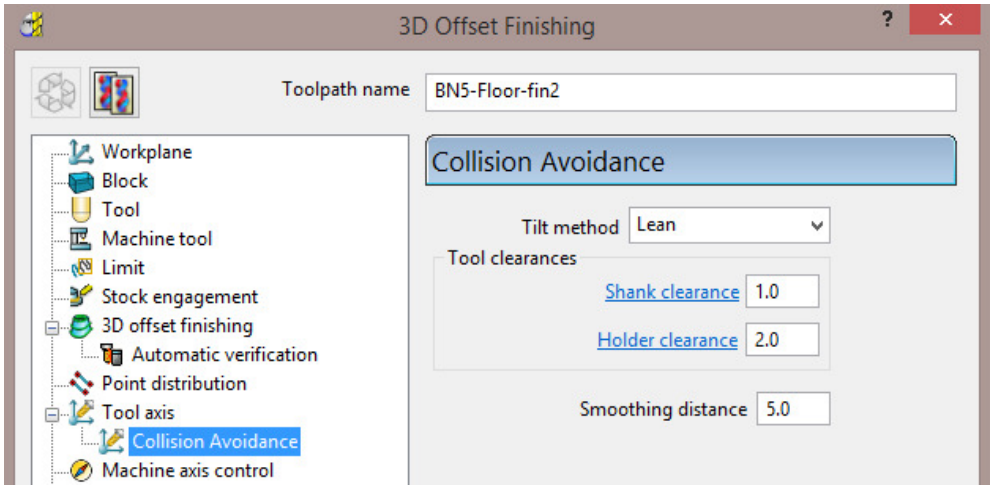
13 Right mouse click on the ***toolpath*** **BN5-Floor-fin2** and from the local menu select **Settings**.

14 Select **Recycle** in the **3D Offset Finishing** dialog and select the **Tool Axis** page.

*Note that the selected **Tool Axis** option is **Vertical**.*



15 Select the **Collision Avoidance** page.

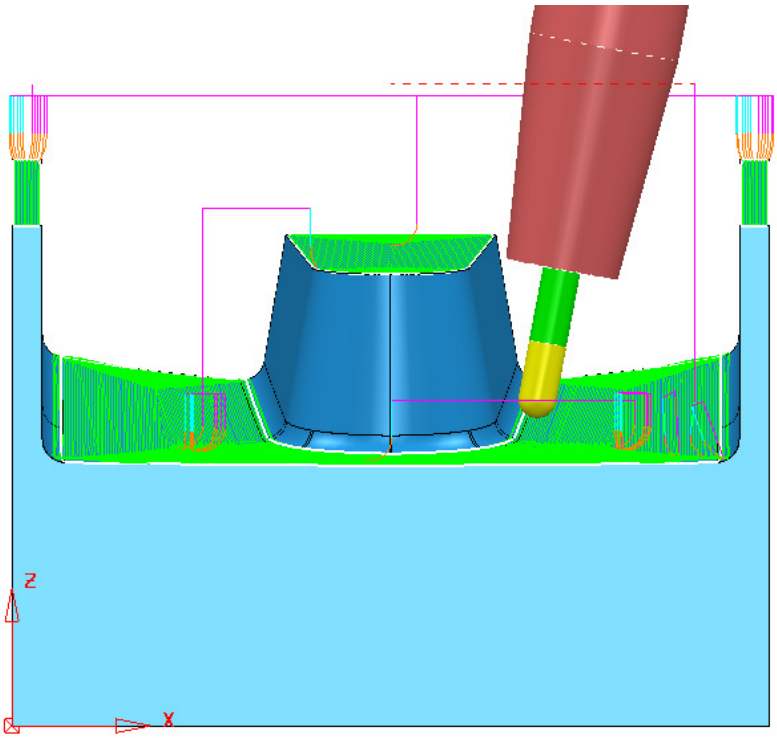


- Input the ***Tilt method*** as **Lean**.
- Input ***Shank*** and ***Holder*** clearances of **1** and **2** along with a **smoothing distance** of **5**.

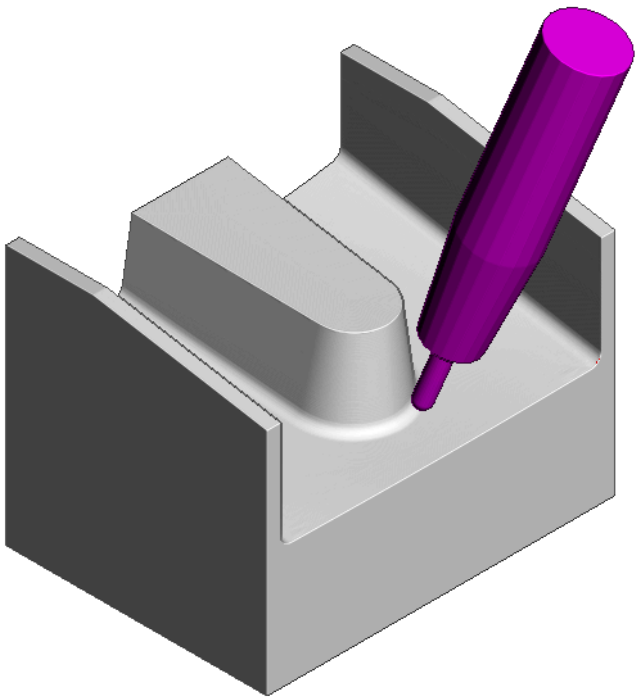
16 **Calculate** the **3D Offset Finishing** toolpath.

17 Apply **Lead In** and **Lead Out** as **1st choice** – **Surface normal Arc** as used in the previous toolpath (**BN5-Walls-fin1**).

18 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.



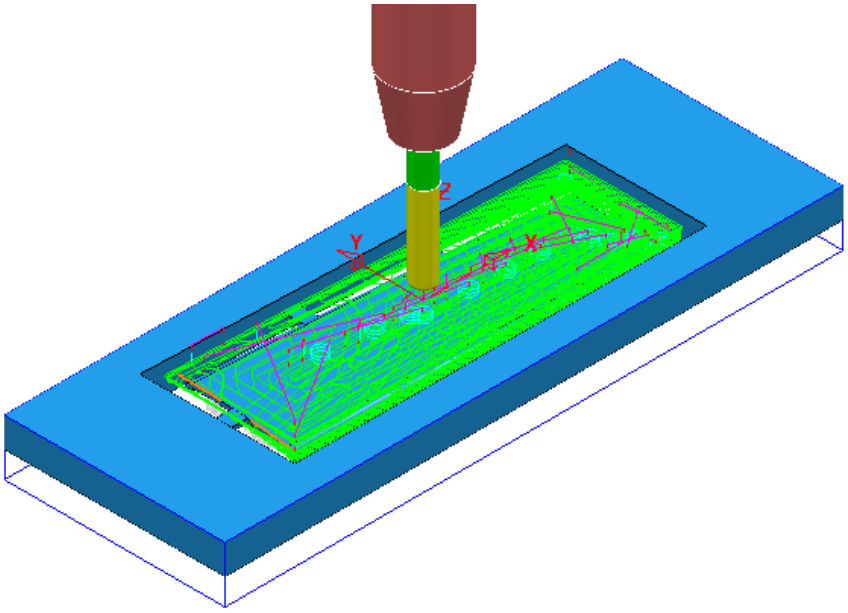
19 Select **File** > **Save Project**.

EXAMPLE 2

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

2 **Open** the (read only) **Project**:

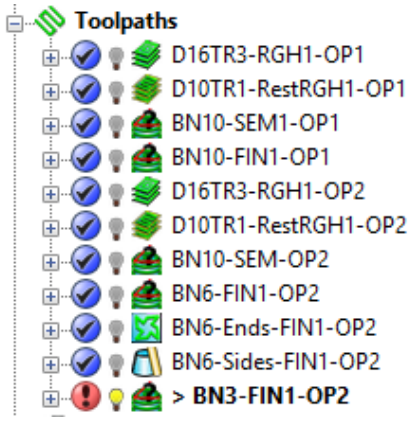
....\PowerMILL_Data\five_axis\CollisionAvoidance\DashboardPanel-A1_Start



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.



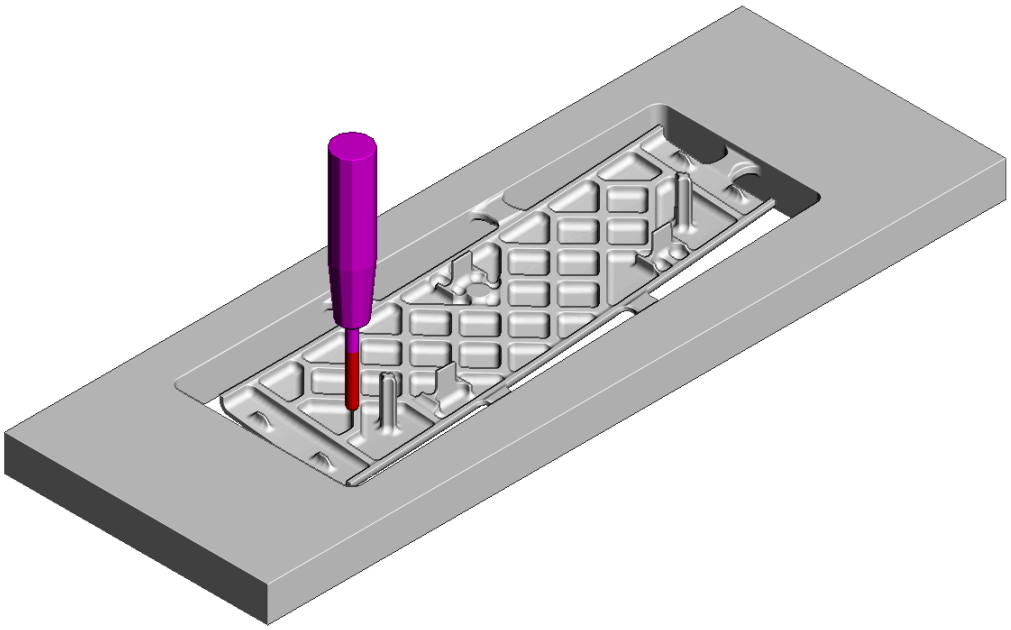
All have been confirmed to be safe with the exception of **BN3-FIN1-OP2**.

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.

6 Run a full **ViewMILL** simulation on both ***NCPrograms*** **OP1** and **OP2**.

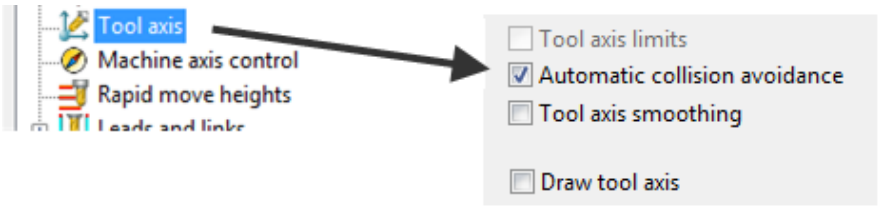


The final toolpath **BN3-FIN1-OP2** is not currently included in the **OP2** ***ncprogram***. The above **ViewMILL** simulation displays the result of all toolpaths except **BN3-FIN1-OP2**.

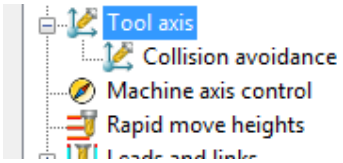
7 Put **ViewMILL** on suspend and return the graphics area to the **PowerMILL** page. 

8 **Activate** the toolpath **BN3-FIN1-OP2** and from the local menu select **Settings**.

9 In the **Tool axis** page ***tick*** the box named **Automatic collision avoidance**.

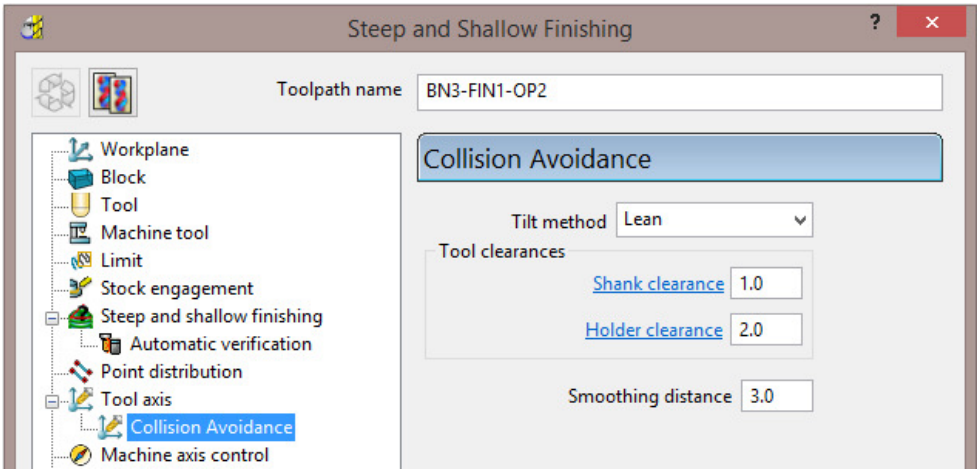


The **Collision avoidance** page appears in the local **explorer**.

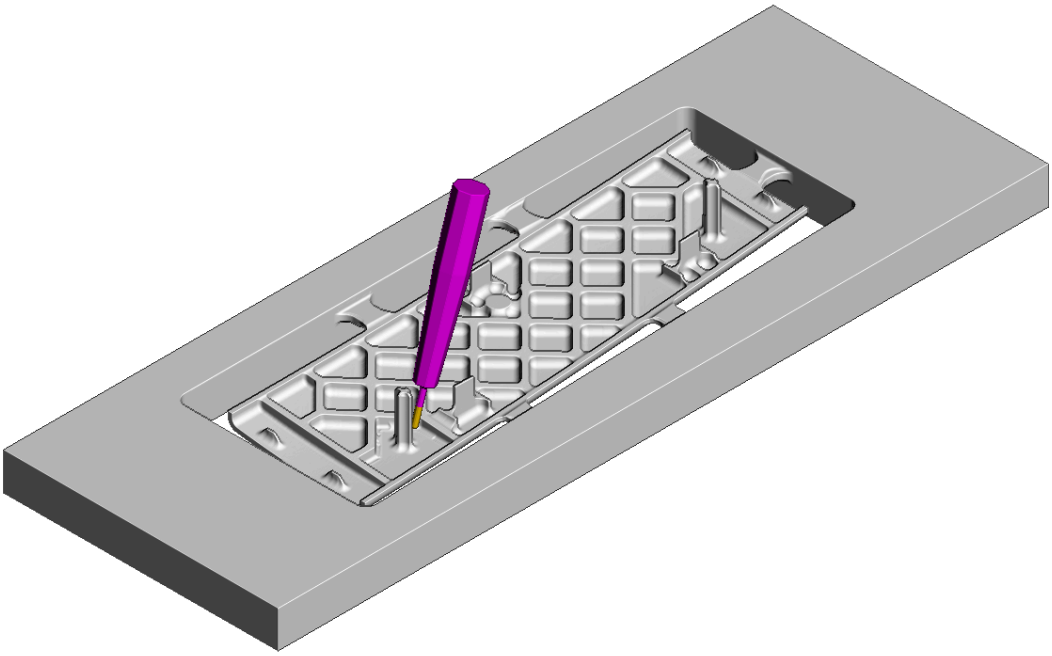


10 Select the **Collision avoidance** page.

11 Set ***Tilt Method*** to **Lean** with a ***Holder clearance*** of **2.0**, ***Shank clearance*** of **1.0**, and ***Smoothing distance*** **3.0**.



12 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***.



13 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).

14 **Save** the **Project**.

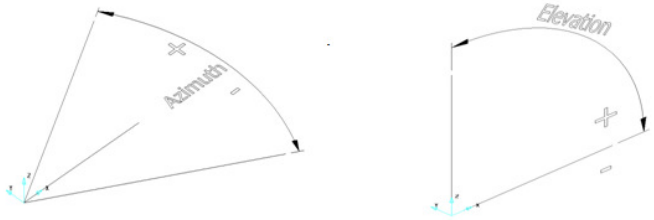
9. Tool Axis Limits

Introduction

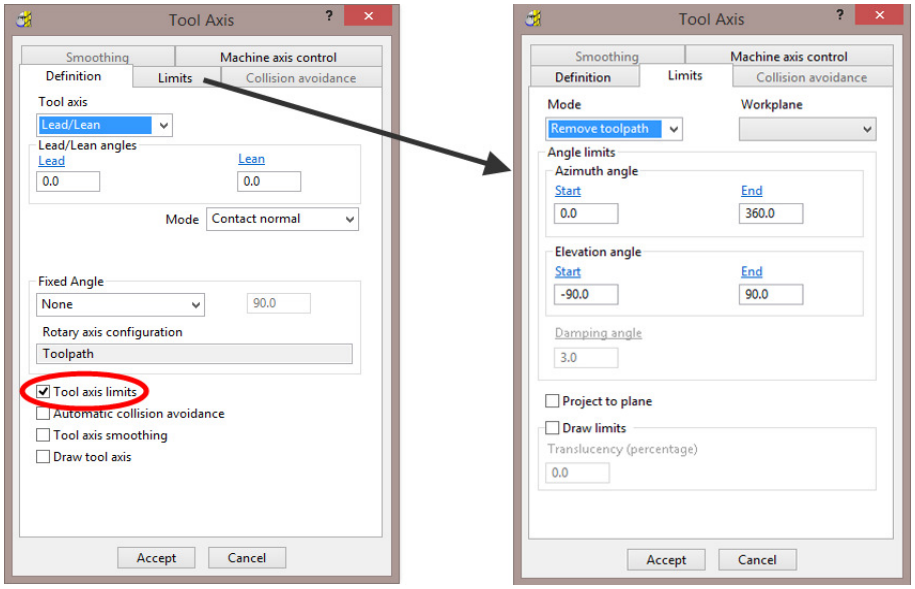
It is possible to control the **Tool axis limits** of the machine tool within **PowerMILL**. This enables the rotary working envelope of a specific machine not to be exceeded when creating multi-axis toolpaths. The angular limits are defined in terms of **Azimuth** and **Elevation** angles in **PowerMILL**.

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**.

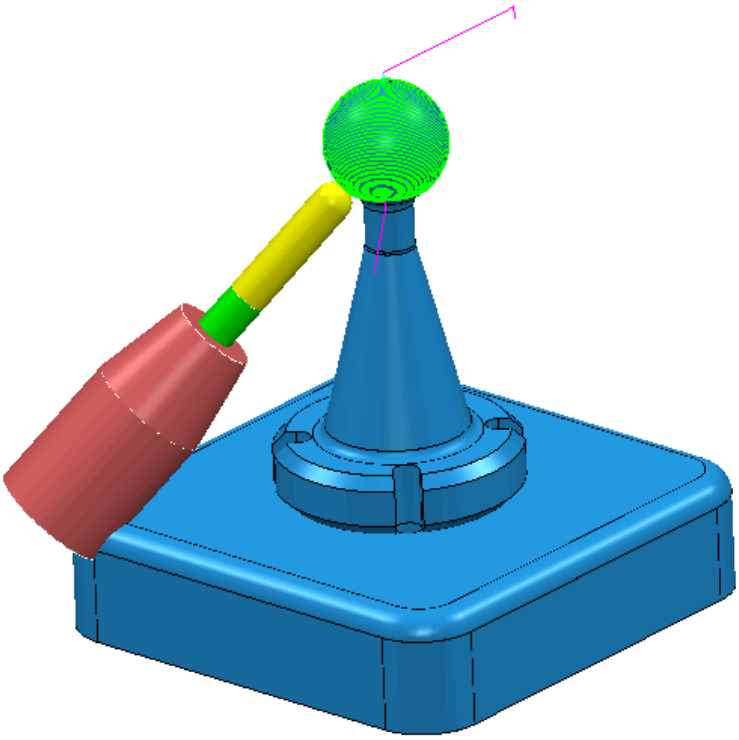
2 **Open** the 'read only' **Project**:

...\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 **Nearest Point**.



*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.

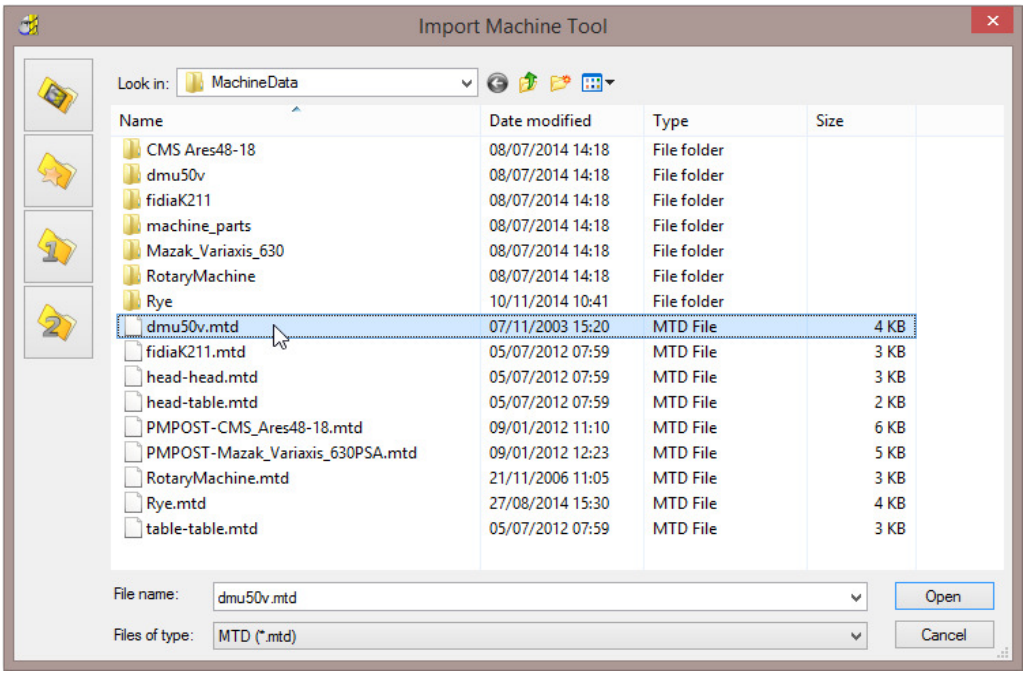
5 In the **PowerMILL *explorer***, right mouse click on the ***Toolpath*** **BN16-NoToolAxisLimits** and from the local menu select **Simulate from Start**.

This command will raise the **Simulation** toolbar (if it is not already open).

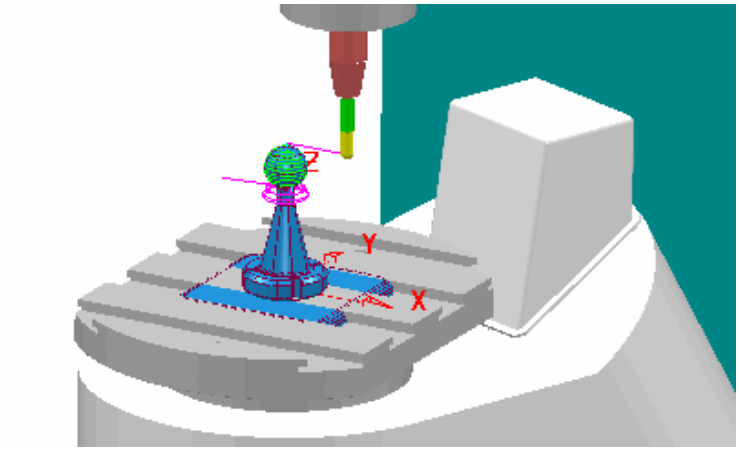
6 In the **PowerMILL *explorer*** right mouse click on **Machine Tools** and from the local menu select **Toolbar**.

7 Select **Import the Machine Tool Model** and select the file:

....\PowerMILL Data\Machine Data\DMU\dmu50v.mtd



8 Ensure the **Draw\Undraw** machine tool icon  is on to display the machine tool model.

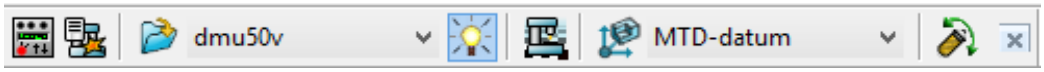


The current component datum (**Transform**) is matched to that of the Machine Tool model (Top - Centre of Table). As a result, the base of the component is currently embedded into the machine tool bed. To compensate for this, a new, suitably positioned Workplane is created. This new **Workplane** is then used in the **Machine Tool definition** toolbar.

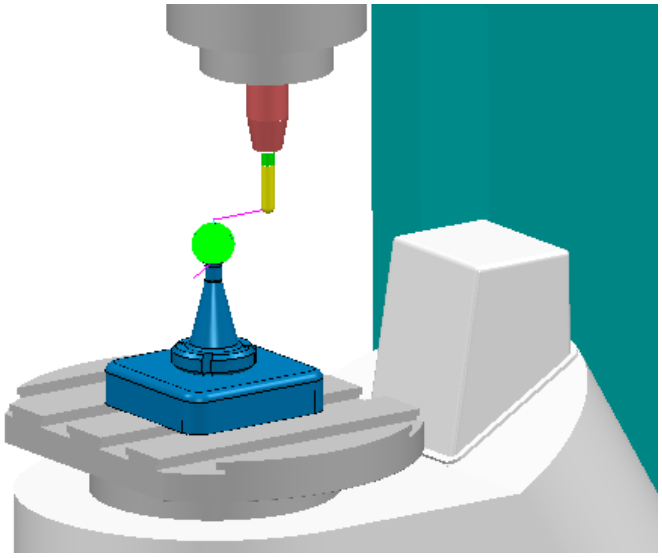
9 Create a new **Workplane** named **MTD-datum** and move it by **Z-50**.

10 Input the new **Workplane**, **MTD-datum** into the **Machine Tool definition** dialog.

 *It is not necessary to activate the new workplane.*



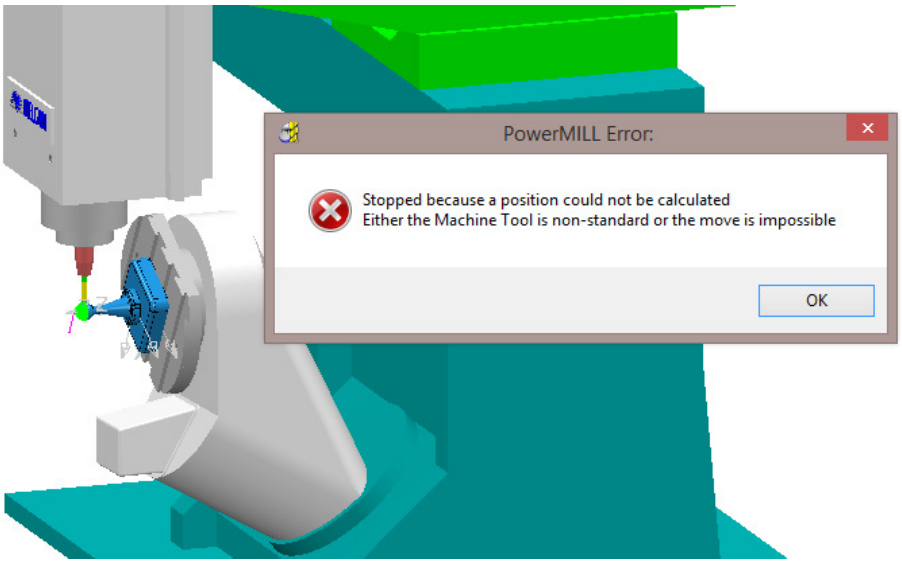
The component will immediately be repositioned, relative to the new **Workplane**.



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.



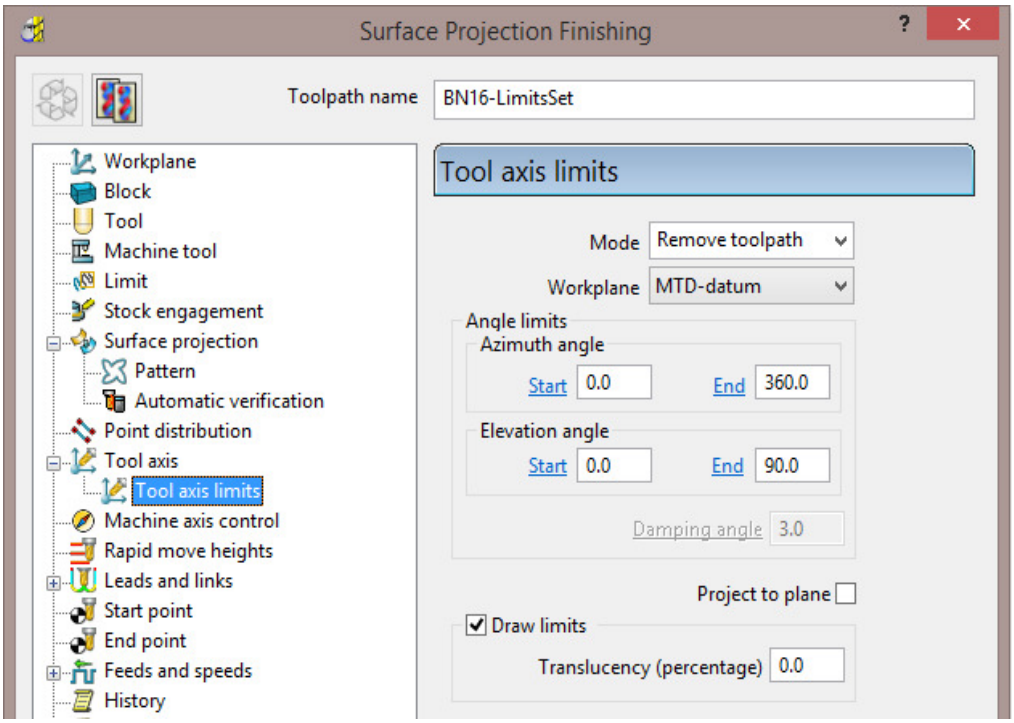
*To allow for this appropriate **Tool Axis Limits** will be applied to the toolpath during calculation.*

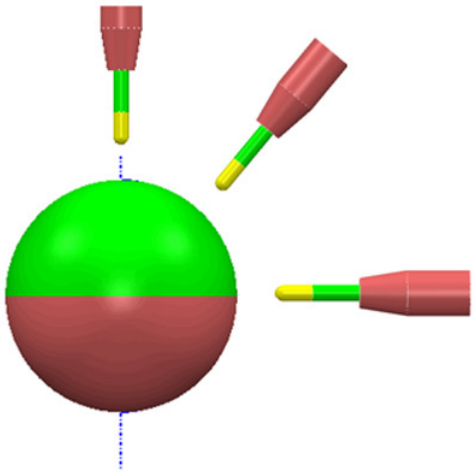
13 In the **PowerMILL *explorer***, **Machine Tools** folder right mouse click on **dmu50v** and from the local menu select **Go Home** to return the machine tool model to the home position.

14 Make a **Copy**  of the original toolpath and rename it as **BN16-LimitsSet**.

15 In the local ***explorer*** below **Tool Axis**, 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).



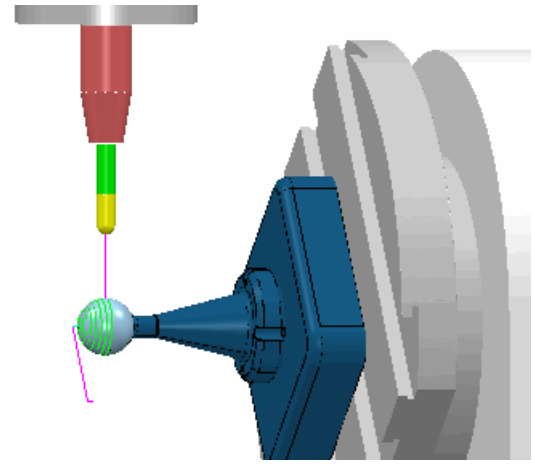


The green area of the sphere represents the permissible angular alignment of the **Tool Axis**.

17 With the same **Reference Surface** selected, **Calculate** and **Close** the dialog.

18 Right-click the ***Toolpath***, **BN16-LimitsSet**, in the **PowerMILL *explorer*** and select **Simulate from Start**.

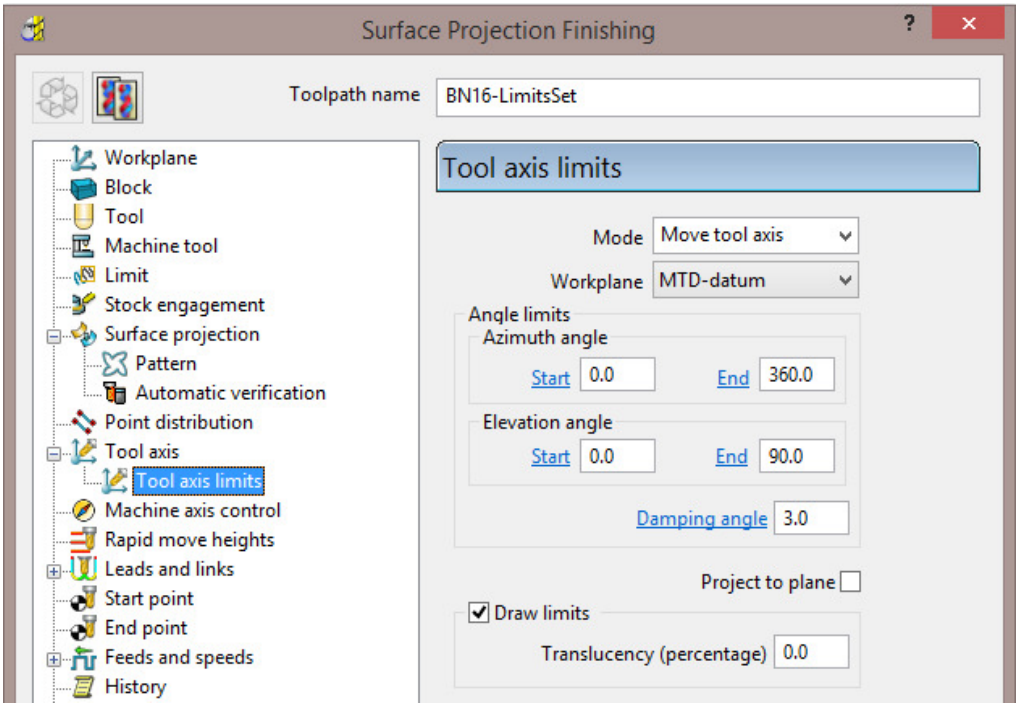
19 Select **Play** .



*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.

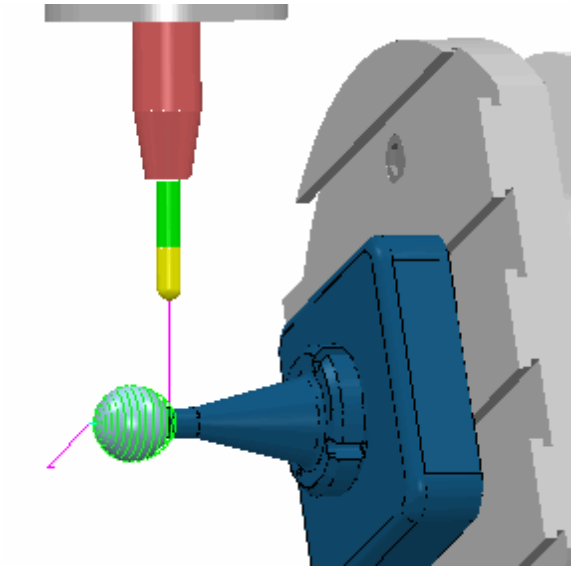
21 Select recycle and from the **Tool Axis** dialog select the **Limits** page and enter the values into the form exactly as shown below (select **Mode** – **Move tool axis**).



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** .



With **Mode** set to **Move tool axis** in the **Limits** dialog, the **surface** is fully machined with the **Tool Axis** becoming fixed when it reaches the maximum, **Tool Axis 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.

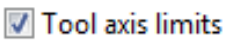
Head – **Table** One rotary axis moves the head, the other moves the table.

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.

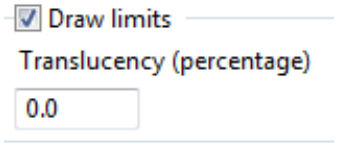
3 Define **Tool Axis** as **Lead\Lean** and set both **Lead** and **Lean** angles to **0**.

4 Select the **Tool axis limits** option 

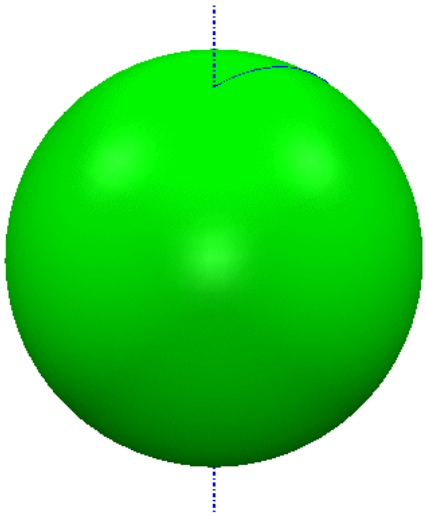
to activate the **Limits** tab.

5 Open the **Limits** tab.

6 Select the **Draw Limits** option from the dialog.



7 Select view **ISO 1**.

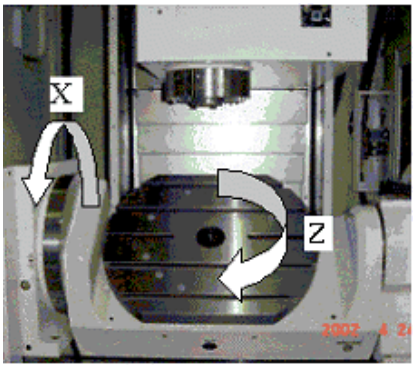


With the **Draw Limits** option selected a sphere will be displayed in the graphics window representing the angular machining limits available.

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.



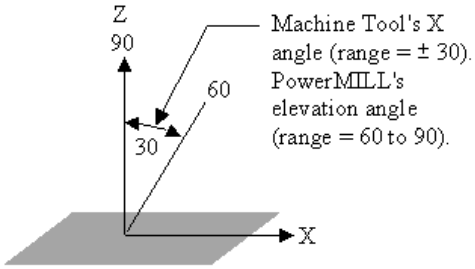
With the above **Table-Table** machine tool the angular limits are specified as:

X ± 30

Z ± 360

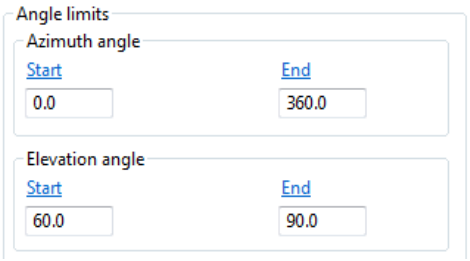
The machine tool **Y** limits are equivalent to the Azimuth angle or the angular limits normal to the **XY** plane. The **Y** limit of **± 360** translates to **Azimuth** angle limits of **0** to **360**.

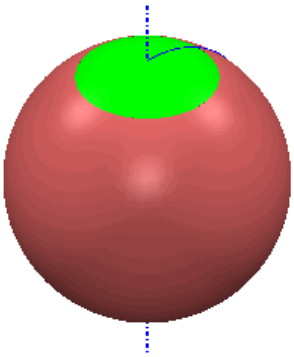
The machine tool **X** limits are equivalent to the **Elevation** angle above the **XY** plane. However they are not the same angle. This is best described using the diagram below. The machine tool measures the angular range relative to the **Z** Axis and **PowerMILL** measures it relative to the **XY** plane, therefore the angle required for the limit in PowerMILL is the complementary angle to the one given for the machine tool.



This means that 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.





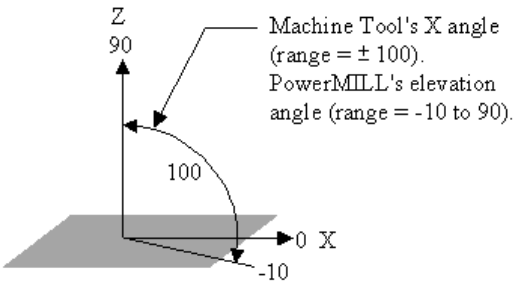
The **Tool Axis Limits** sphere has been visually updated with the modified values.

An Alternative **Table -Table** machine tool has the following angular limits:

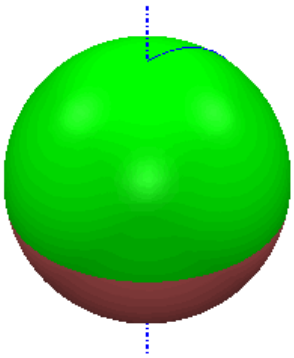
X ± 100

Y ± 360

This translates to **Azimuth Angle** limits of **0** to **360** and **Elevation Angle** limits of **−10** to **90**.



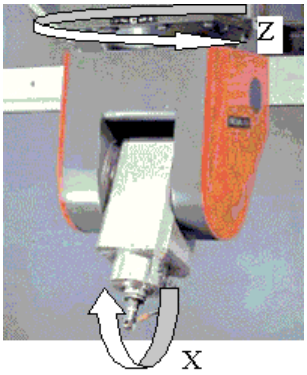
2 Modify the **Elevation Angle** values (-10 to 90) in the **Angle Limits** dialog to update the machining limits (as shown).



The **Tool Axis Limits** sphere has been visually updated with the modified values.

Head – Head

Both rotary axes move the head.



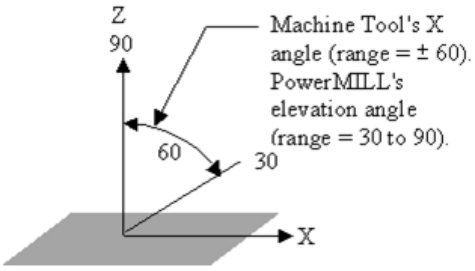
With the above **Head-Head** machine tool the angular limits are specified as:

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**.



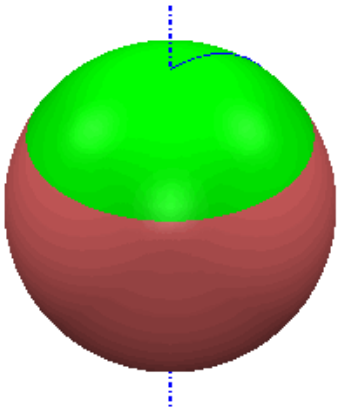
Alternative **Head-Head** machine tool angular limits:

X –50 to +60

Z ± 360

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**).

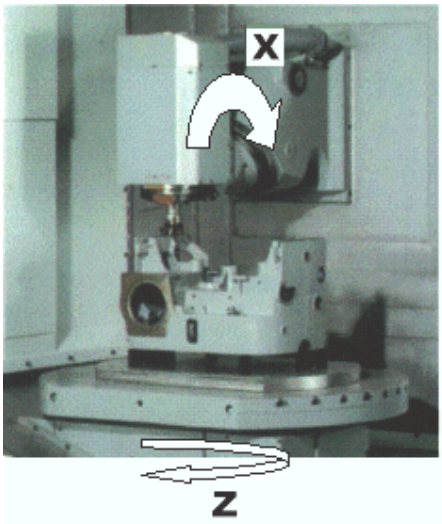
1 Modify the **Elevation Angle** values (**30** to **90**) in the dialog to update the machining limits as shown.



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:

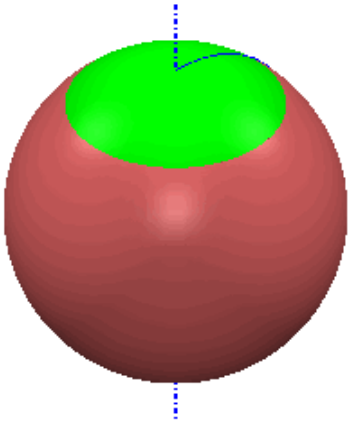
X ± 40

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.

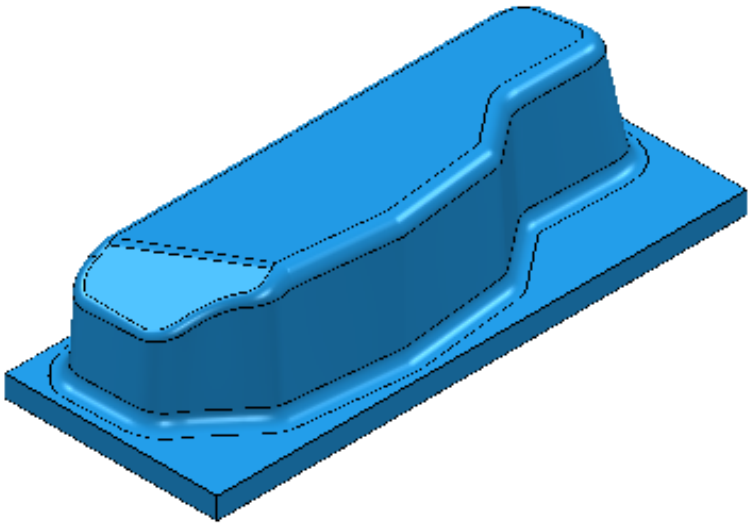


The **Tool Axis Limits** sphere has been visually updated with the modified values.

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

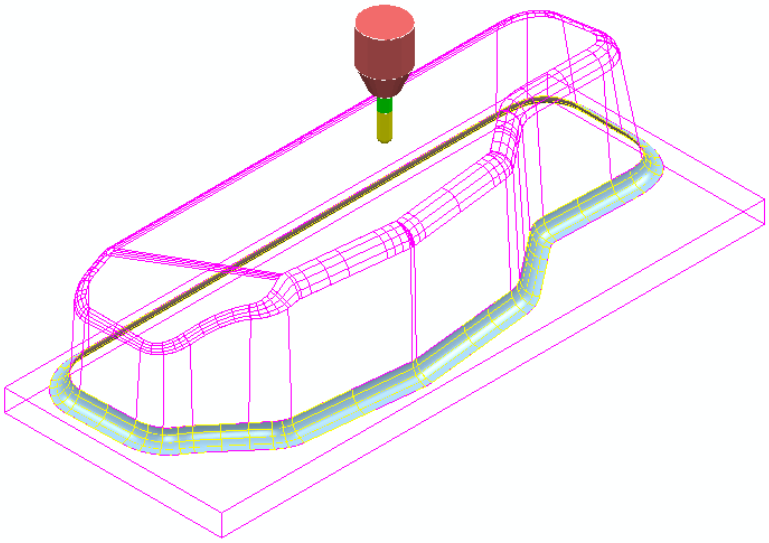


3. Select **File > Save Project As**:

   ...\COURSEWORK\PowerMILL-Projects\Punch2-ToolAxisLimits

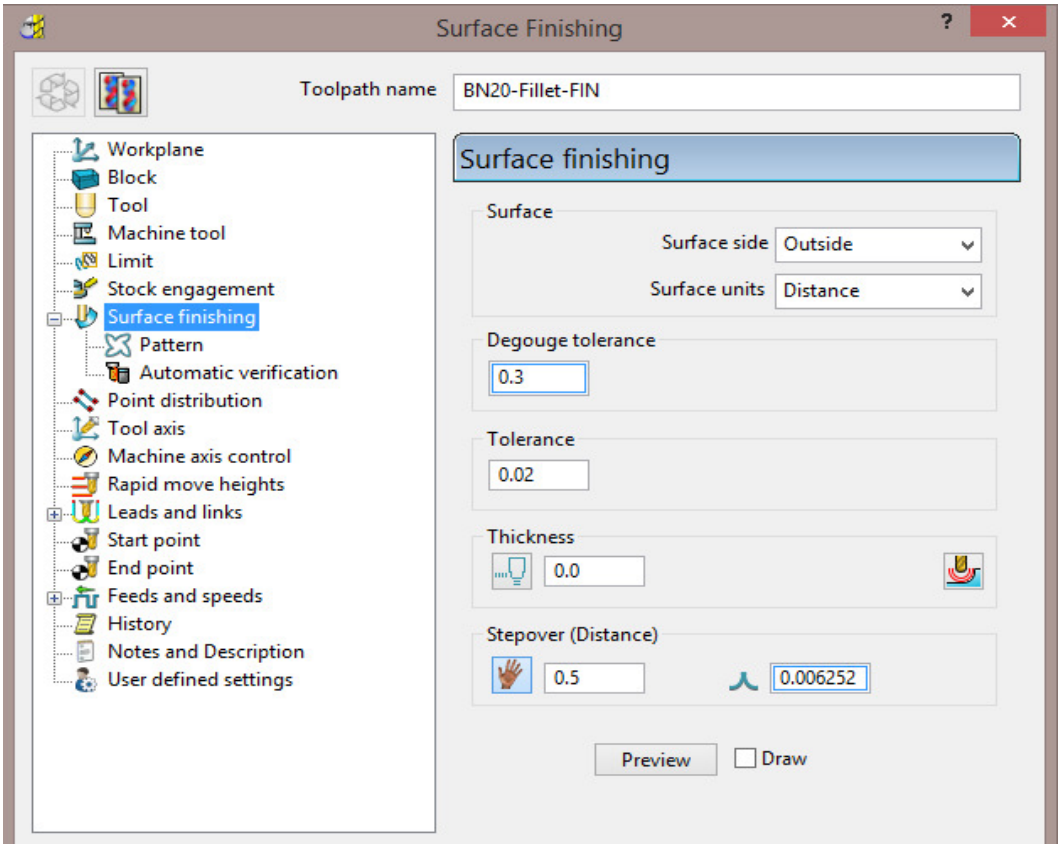
4. Create a Diameter **20** Ballnose tool - Length **50** - named **BN20**.
5. Add a **Shank** component - Upper Dia **20** - Lower Dia **20** - Length **40**
6. Add a **Holder** component - Upper Dia **70** - Lower Dia **40** - Length **40**
7. Add a **Holder** component - Upper Dia **70** - Lower Dia **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.

10 Select the fillet running around the base of the main component sidewalls.

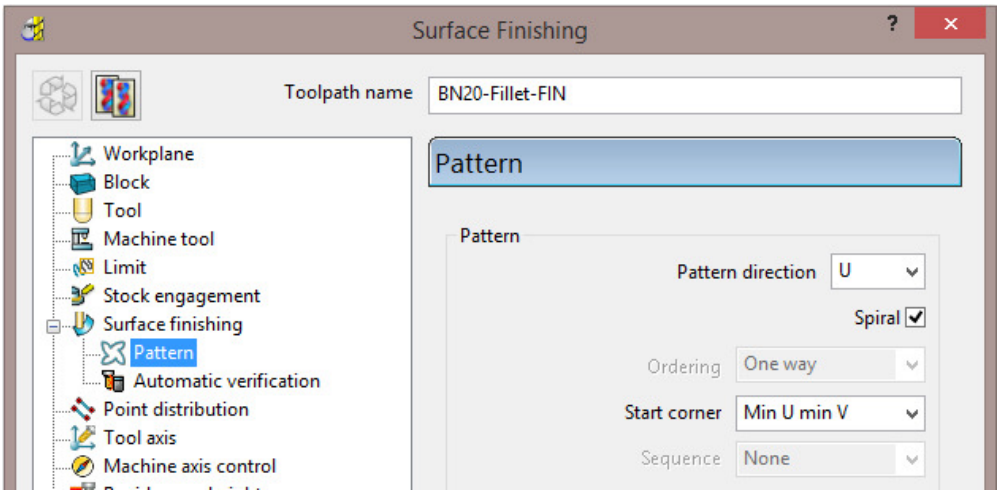


11 Select the **Toolpath Strategies** icon and in the **Strategy Selector** - **Finishing** options select the **Surface Finishing**.

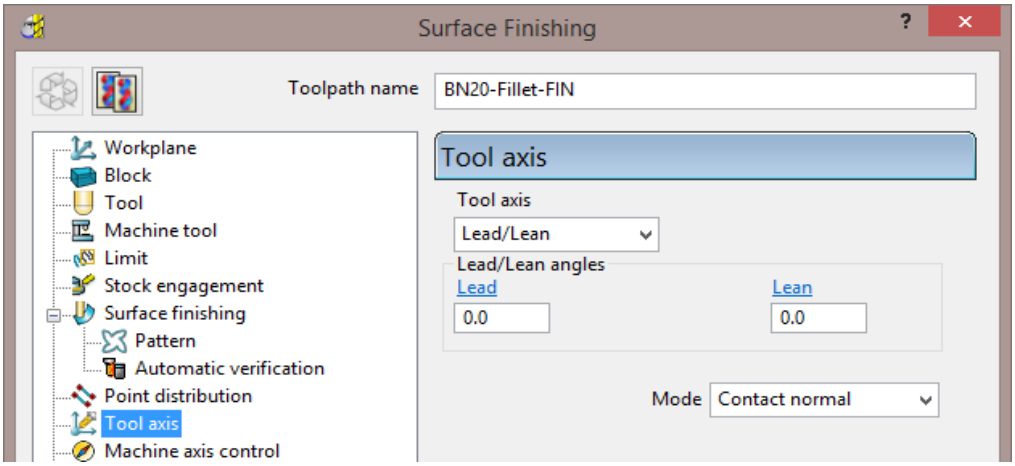
12 Input data into the **Surface Finishing** dialog exactly as shown:



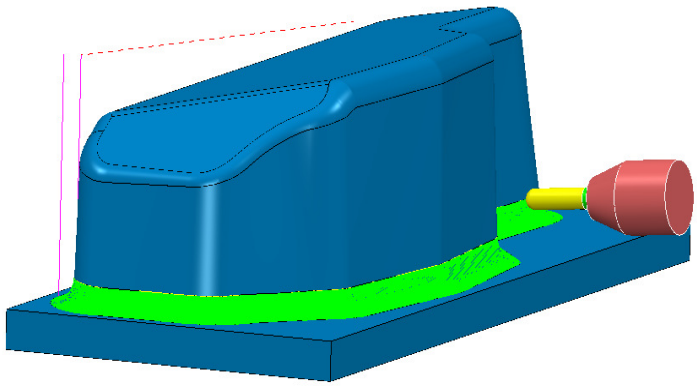
- Toolpath name: **BN20-Fillet-FIN**
- Stepover: **0.5**



- Pattern Direction: **U**
- Spiral: ***Ticked***



13 Select **Calculate** to create the **Surface Finishing** toolpath.

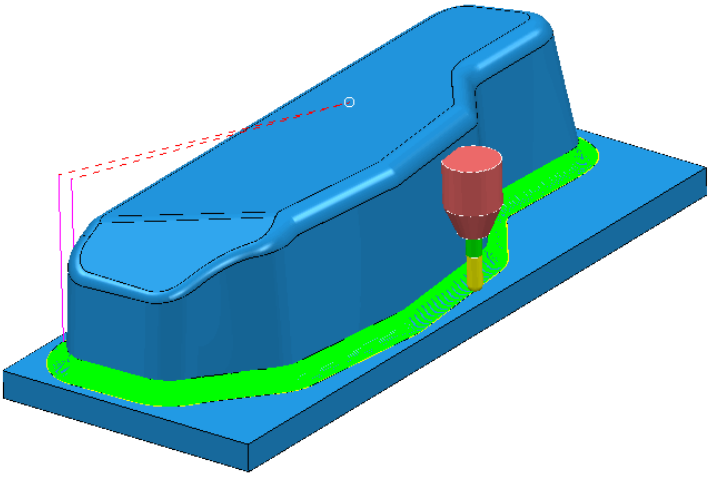


- The **Tool Holder** is in Collision with the Base.

14 From the local toolpath menu, select **Verify** – **Toolpath** to open the **Toolpath Verification** dialog.

15 ***Un-tick*** both **Split Toolpath** and **Adjust Tool**.

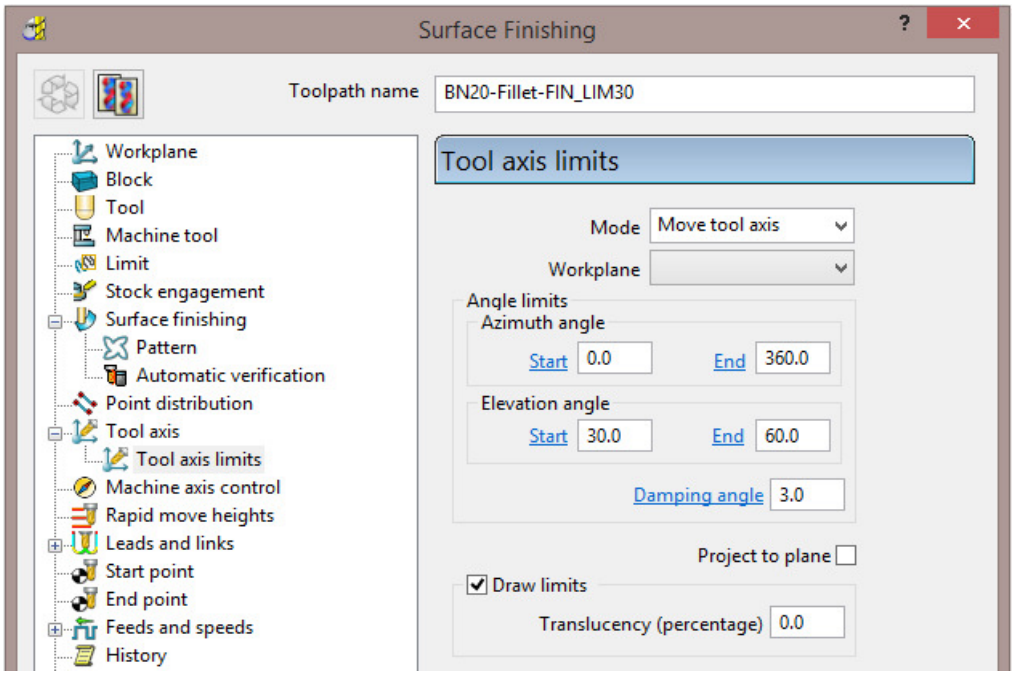
16 Then check for **Collisions** using a **Shank Clearance** of **1** and a **Holder Clearance** of **2**.



- 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.

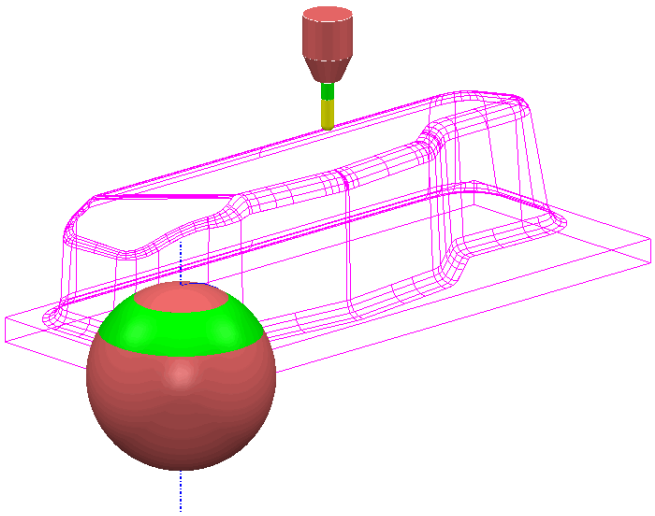
18 In the **Surface Finishing** dialog select  to create a 'copy' and enter the toolpath name **BN20-Fillet-FIN-LIM30**.



19 Tick **Tool Axis limits** and move to the **Tool axis limits** page.

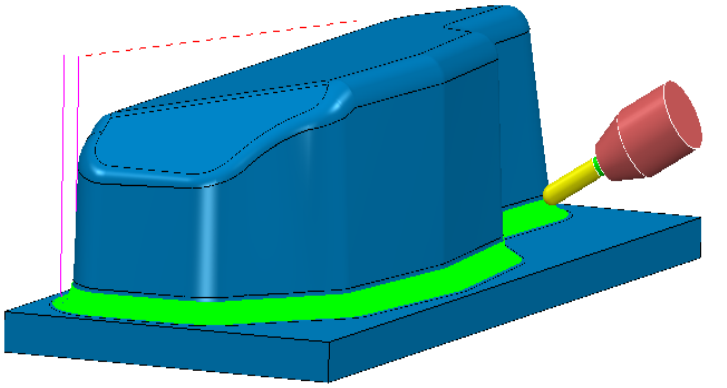
20 In **Mode**, select **Move tool axis** and for **Elevation angle** input:- ***Start* 30** and ***End* 60.**

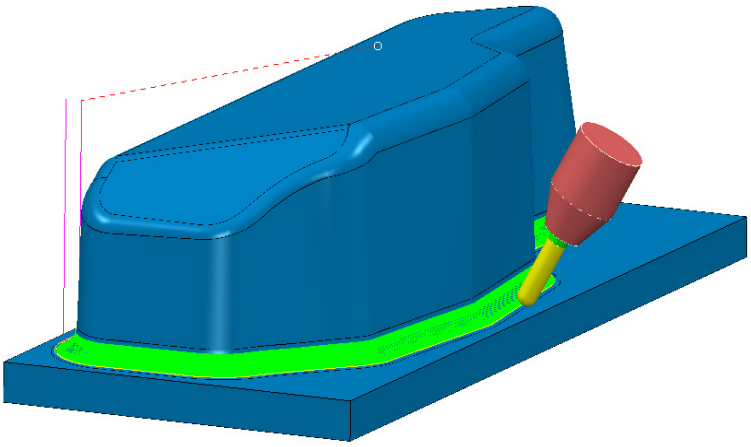
21 Select **Calculate** to create the new toolpath.



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.

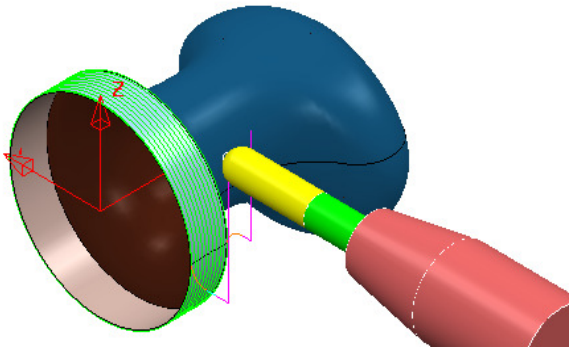
22 **Save** the **Project**.

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

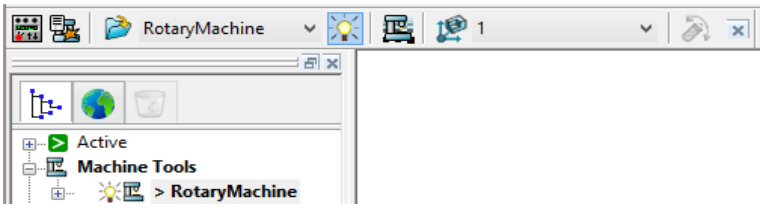


3 Select **File** > **Save Project As**:

....\COURSEWORK\PowerMILL-Projects\FixTo4Axis-example1

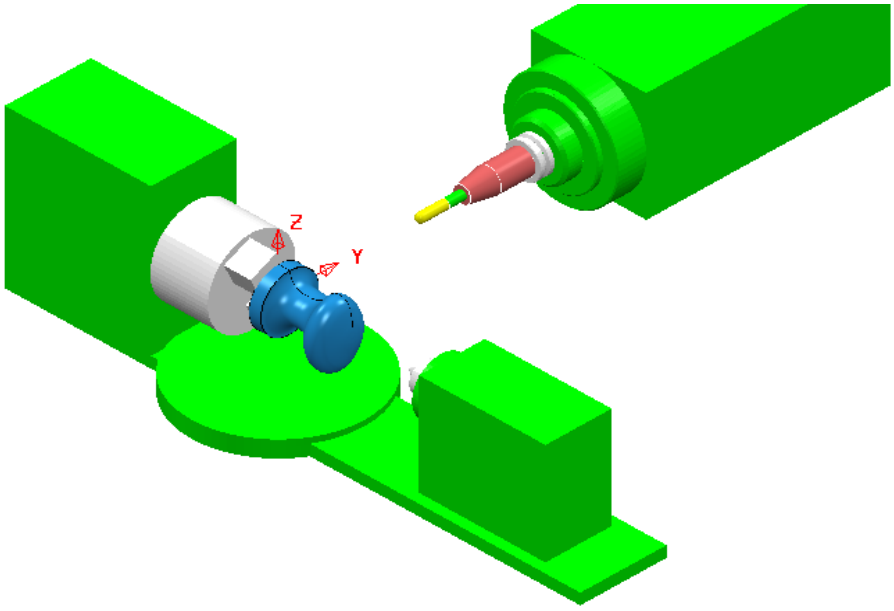
4 If not already displayed, from the main **pull down** menus select **View** - **Toolbar** and tick **Machine Tool**.

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.



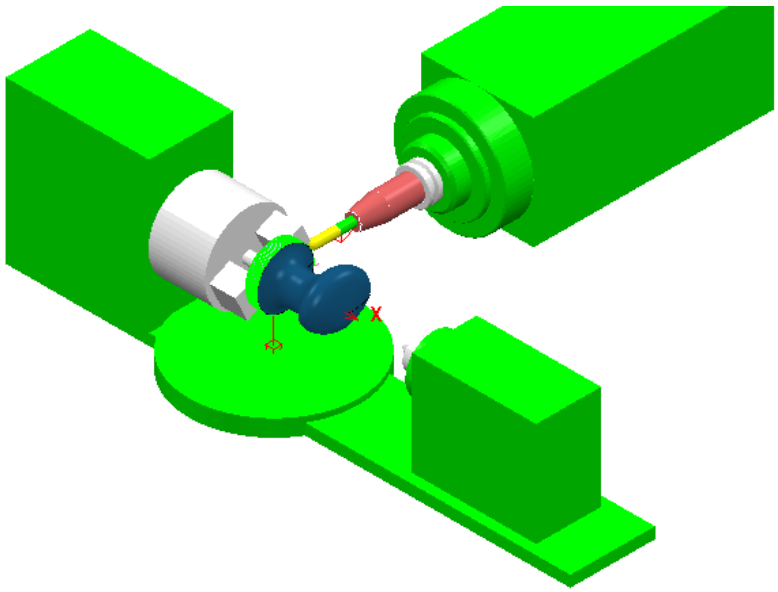
The **Project** contains a model of a 4-Axis Rotary Machine and two toolpaths.

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 **Tool Axis Limits** options.

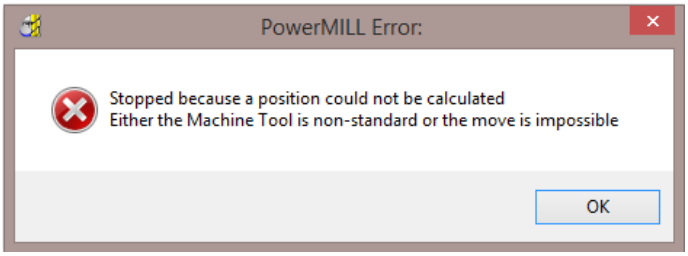
8 In the **PowerMILL *explorer***, right mouse click on the ***toolpath*** **BN16-Base-SEM1** and select **Activate** followed by **Simulate from Start**.

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.



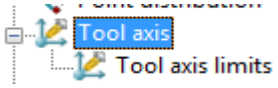
As the toolpath is fully ***Five Axis*** it impossible for the **Rotary Machine** to perform a **simulation**.

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.

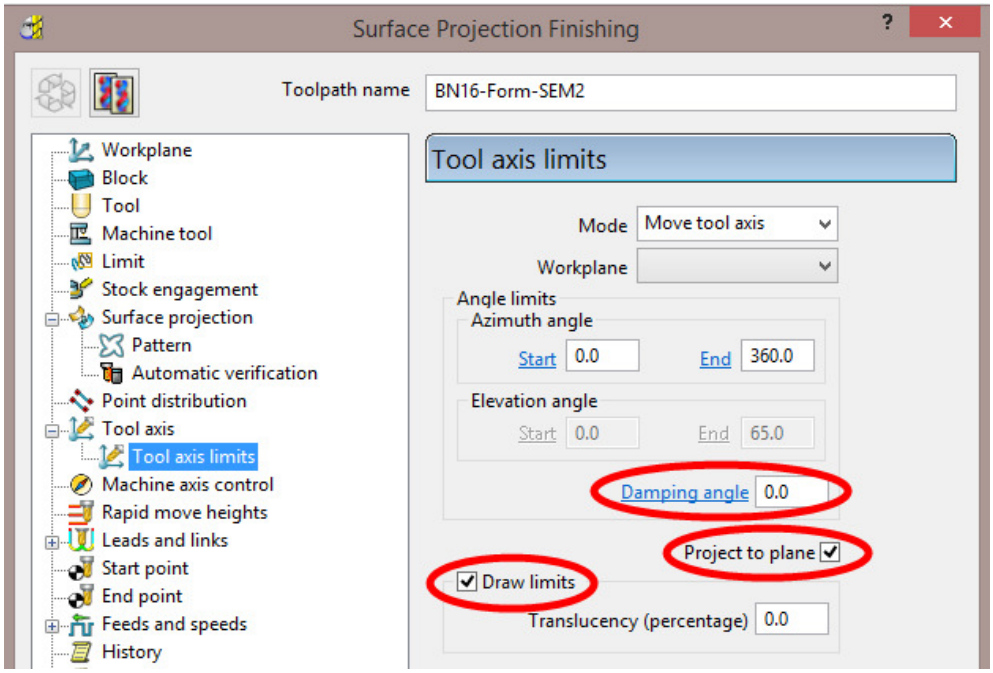
14 In the **Projection Surface** dialog select the Recycle toolpath option .

15 In the local ***explorer*** select **Tool axis** and ***tick*** the **Tool axis limits** box.

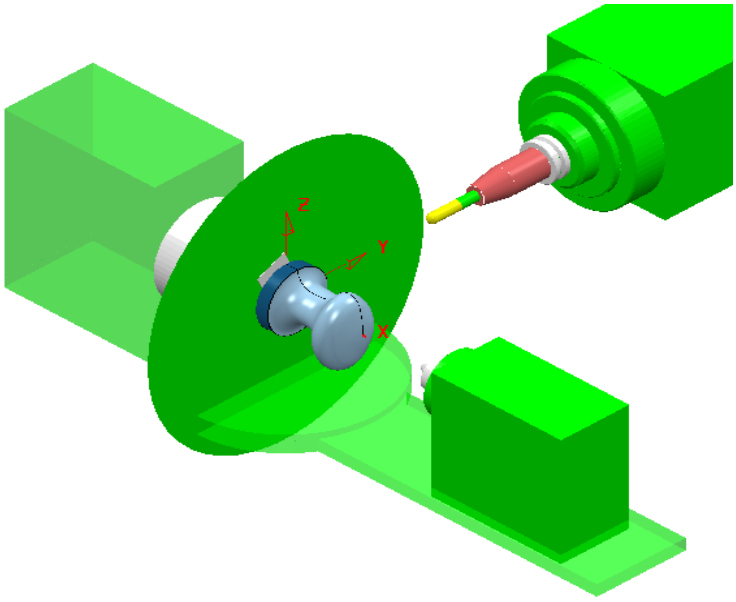


The **Tool axis limits** page will become available in the local ***explorer***.

16 Select the **Tool axis limits** page and enter data exactly as shown below.

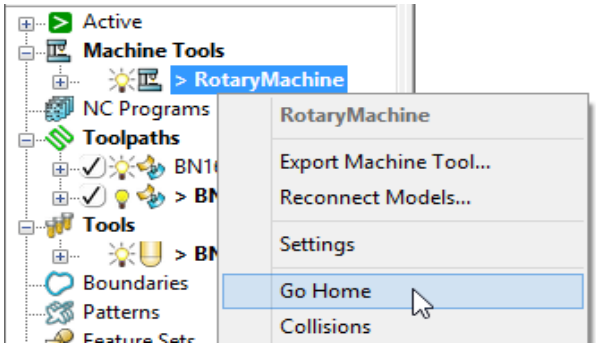


With **Project to plane** ticked the **Tool axis limits** are flattened to create a 4 axis limit (about the global transform in this case). A zero **Damping angle** will ensure that the 4 axis alignment is fully maintained.



17 In the **Tool axis limits** page, un-tick **Draw limits**.

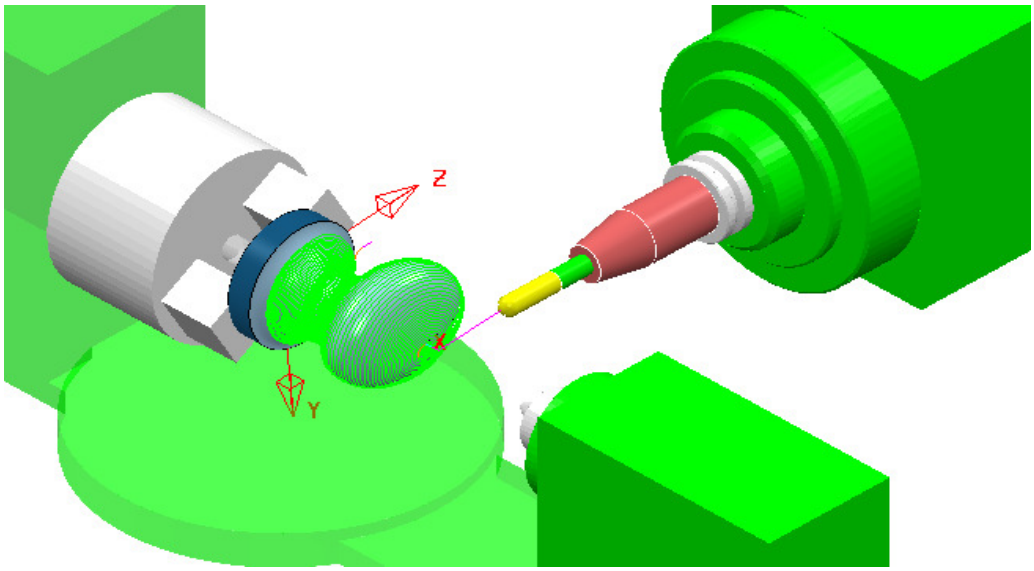
18 With the 3D form, **surface** selected **Calculate** the new 4-Axis toolpath.



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.



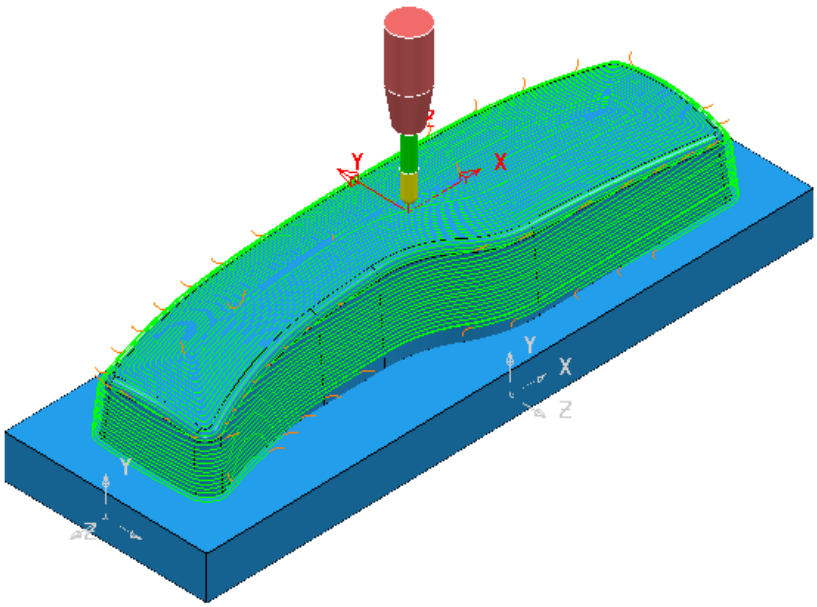
22 **Save** the **Project**.

Exercise 1

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

2 **Open** the 'read only' **Project**:

...\PowerMILL_Data\five_axis\Punch3\Punch3_Start

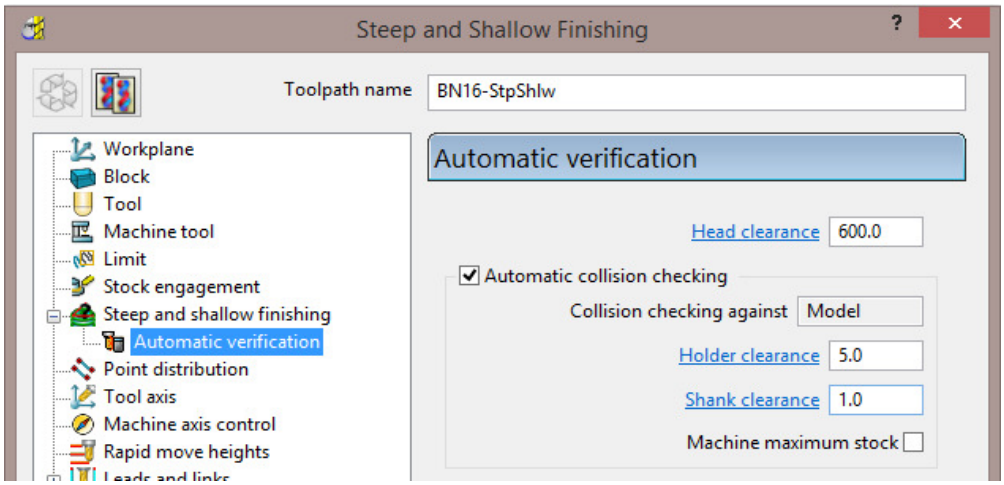


3 **Save** the **Project As**:-

...\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**.

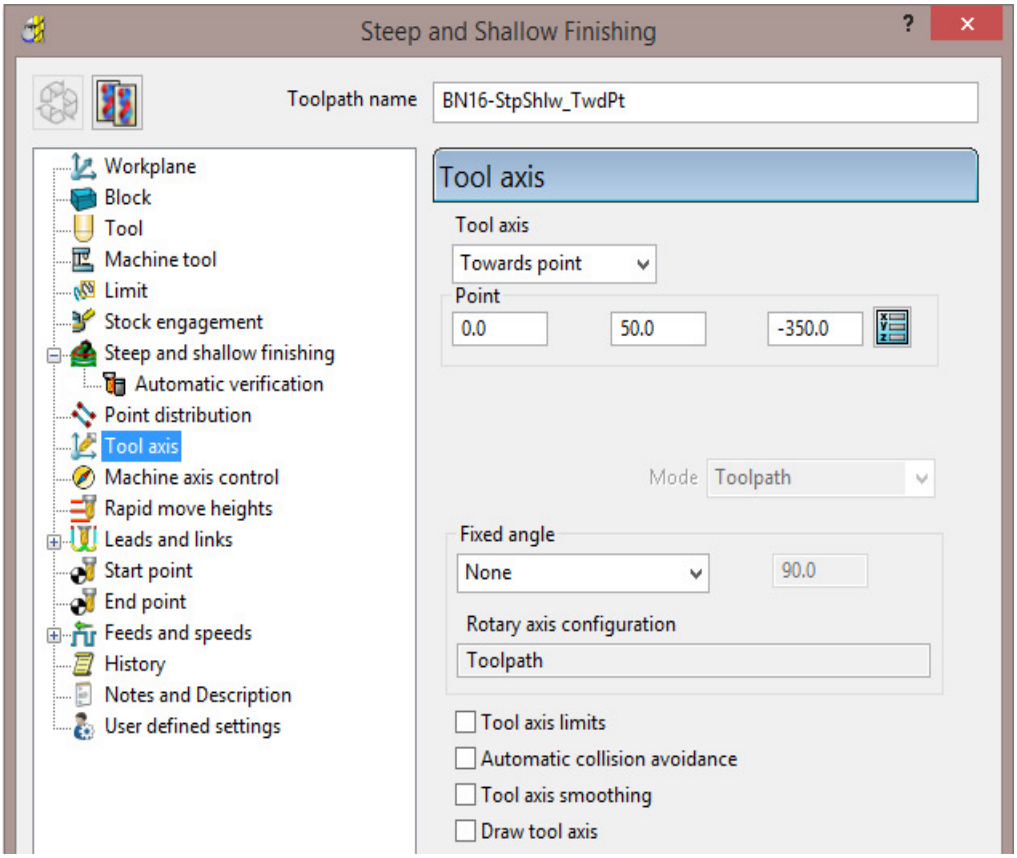


The object of the exercise is to create 2 copies of the existing toolpath the first with a suitable **Toward Point** - **Tool Axis alignment** applied and the second that is further limited to being ***4 Axis*** only (applied about the X axis).

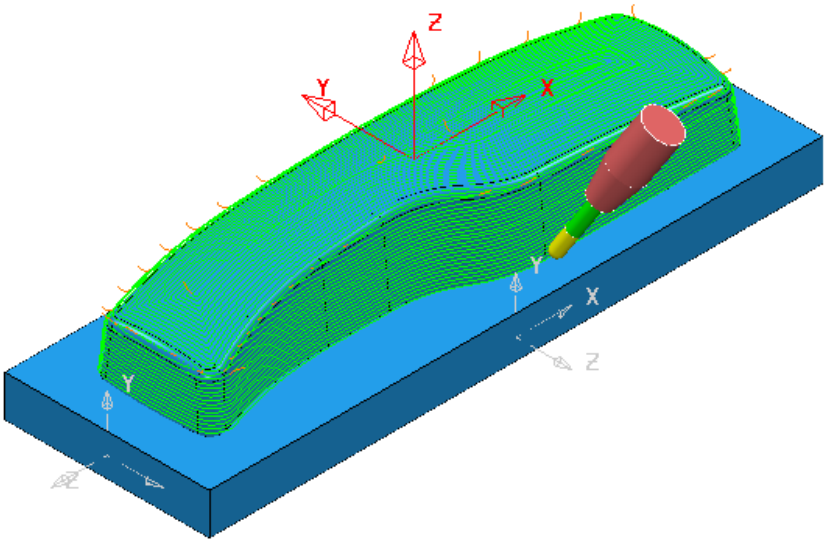
4 **Activate** the toolpath **BN16-StpShlw** and from the local menu select **Settings** to re-open the strategy dialog.

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.



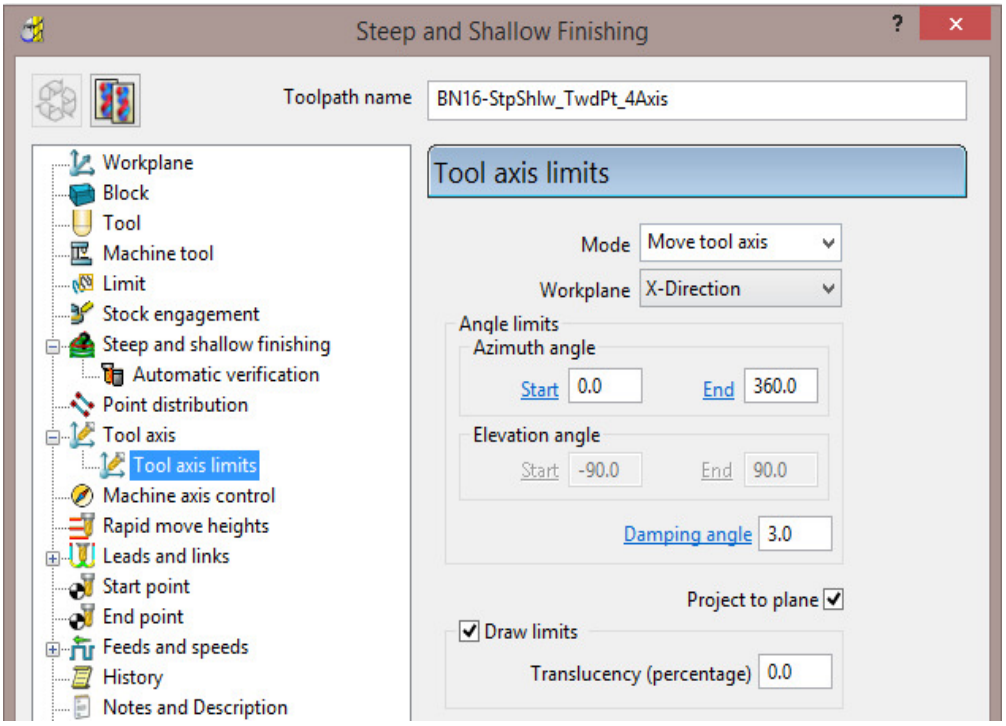
With a suitable **Toward Point** - **Tool Axis alignment** applied the **Tool** is now angled away from the sidewalls providing collision free access to the deeper areas.

8 **Activate** the toolpath **BN16-StpShlw_TwdPt** and from the local menu select **Settings** to re-open the strategy dialog.

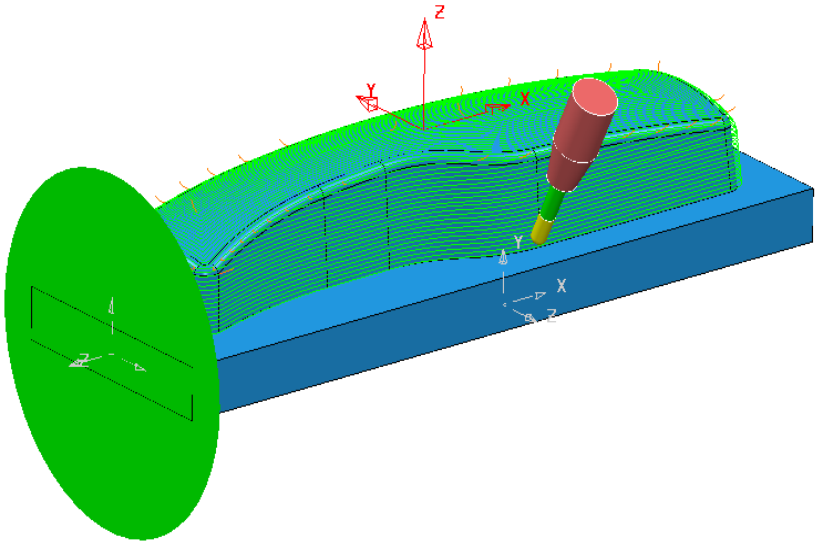
9 Create a ***Copy*** of the toolpath and rename it as **BN16-StpShlw_TwdPt_4Axis**

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.

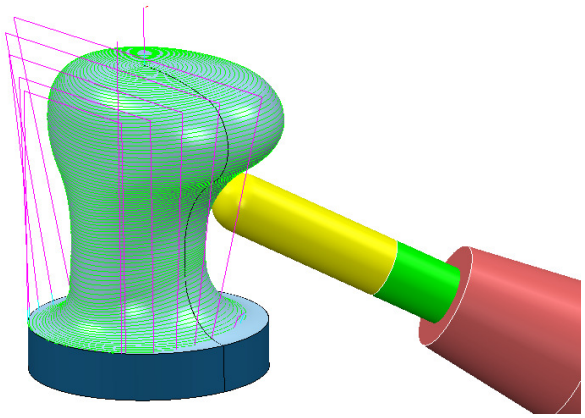
13 **Save** the **Project**.

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



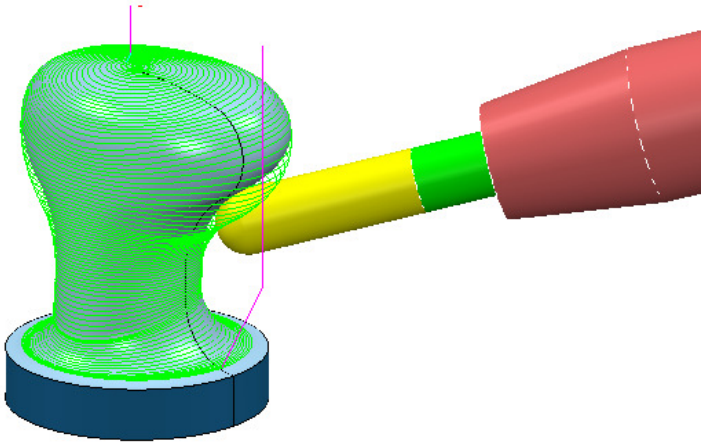
16 **Save Project As**:-

...\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** on the selected **Surface** 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**.

10. Tool Axis Editing

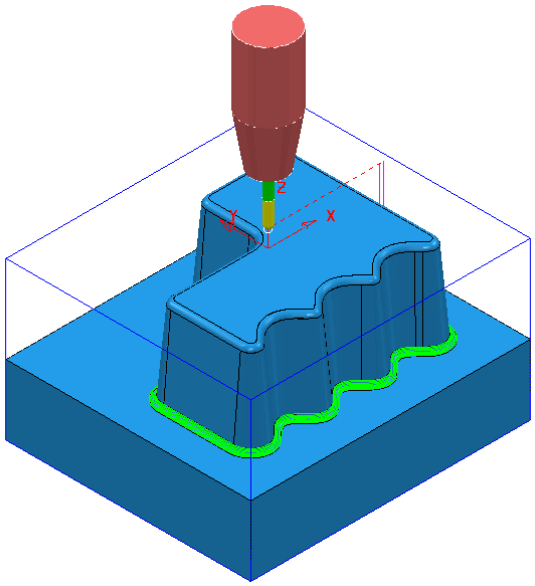
Introduction.

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**.

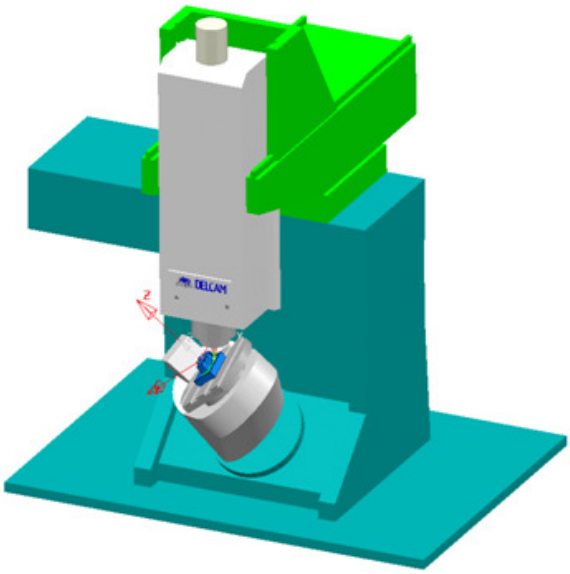
6 Ensure the **Draw\Undraw** machine tool icon is selected to display the machine tool.

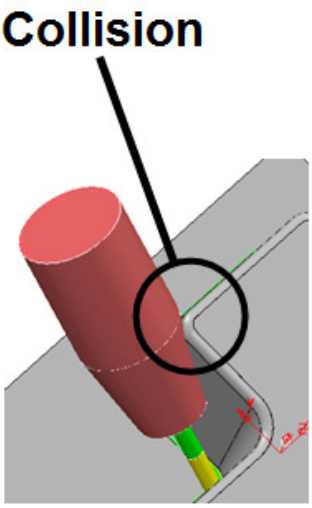
7 Select an **ISO 1** view and zoom the machining area.

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.



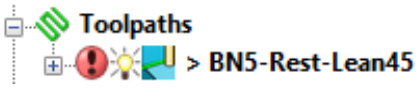


In addition to there being an excessive amount of table rotation while the undulating sidewall is machined there is also a **Tool Holder collision** on the sidewall internal corner (see above right graphic).

11 From the local **BN5-Rest-Lean5 *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.



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.

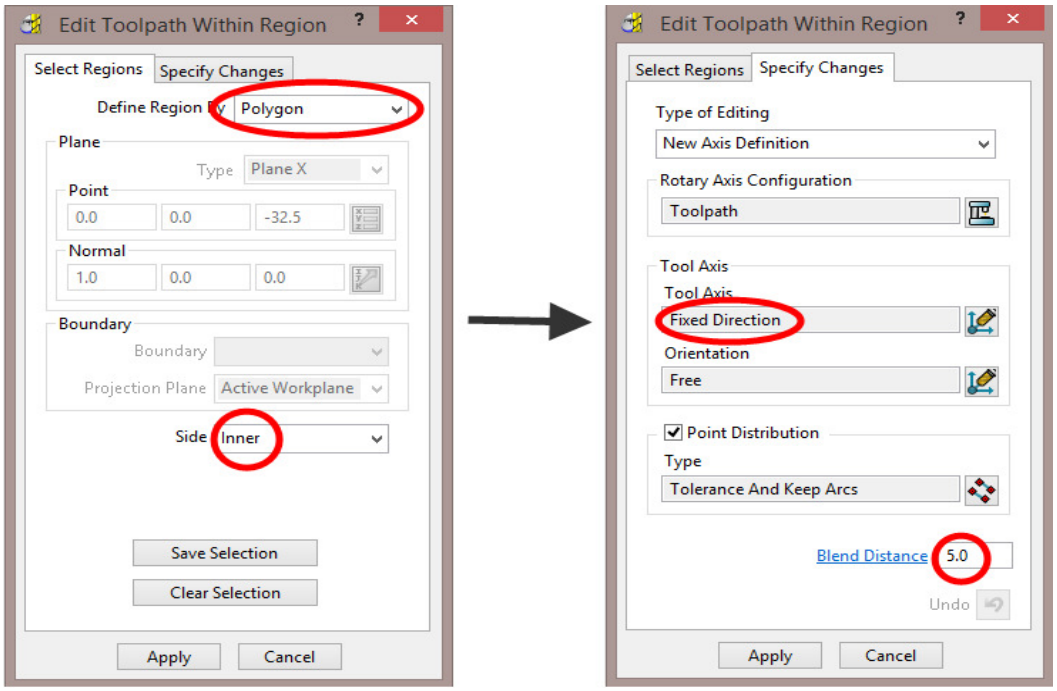
13 In the **Machine Tool Simulation** toolbar select **None** to remove **dmu50v.mtd** from the selection input box leaving it blank.

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***.

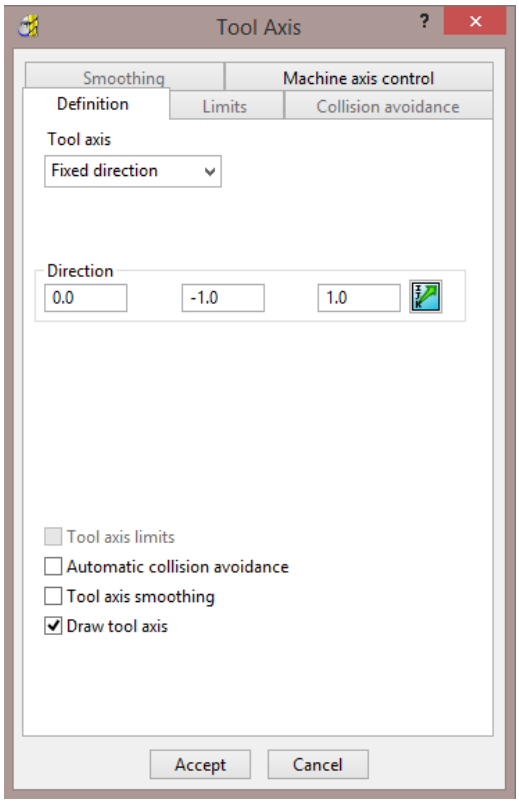
15 On the keyboard select **Ctrl H** to display the **Cross Hair** on the cursor position.

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**

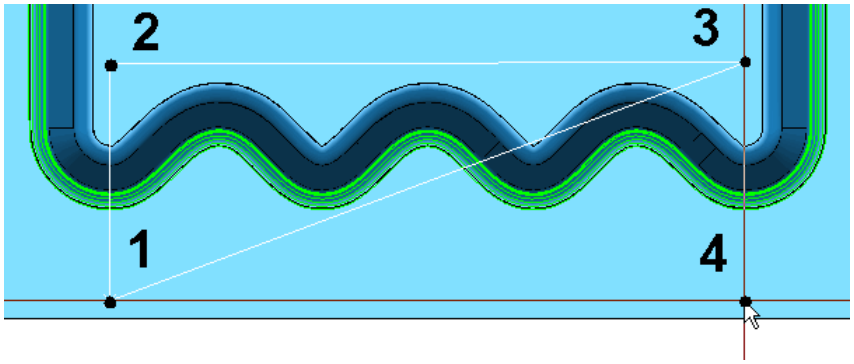
20 In the **Tool Axis** dialog, tick the **Draw Tool Axis** box before selecting **Accept** (to return to the **Edit Toolpath Within Region** dialog).

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.*

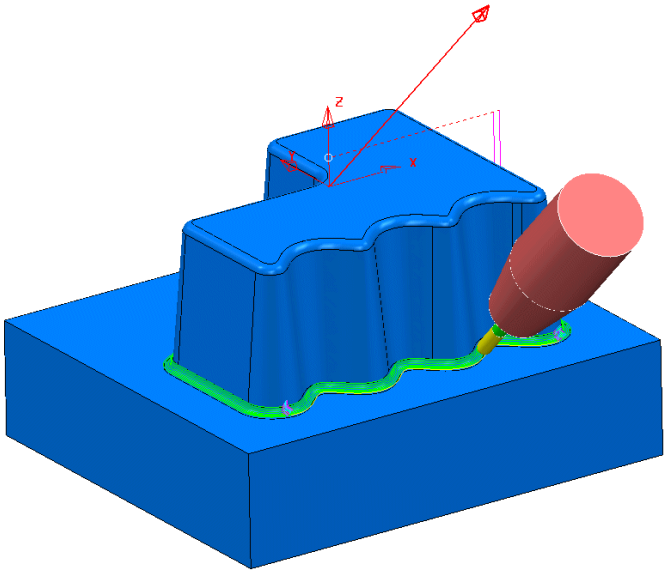
22 Click on the **Select Regions** tab to access the original options.

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).

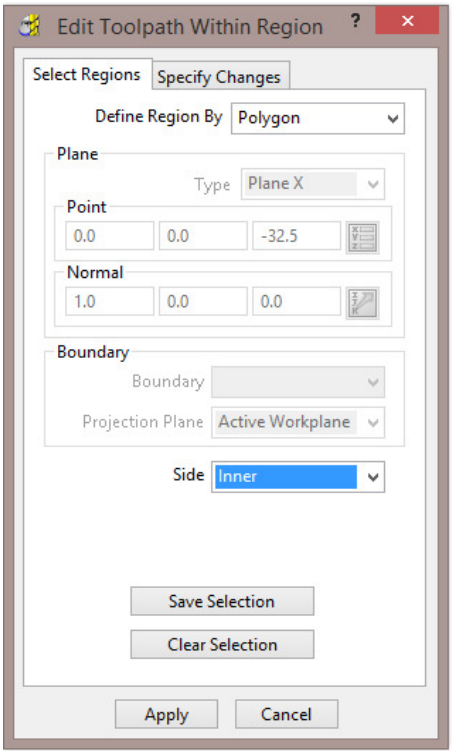
24 Select an **ISO 1** view and **simulate** the toolpath to observe that the **Tool Axis Alignment** is no longer performing a series of rotary movements along the undulating sidewall.

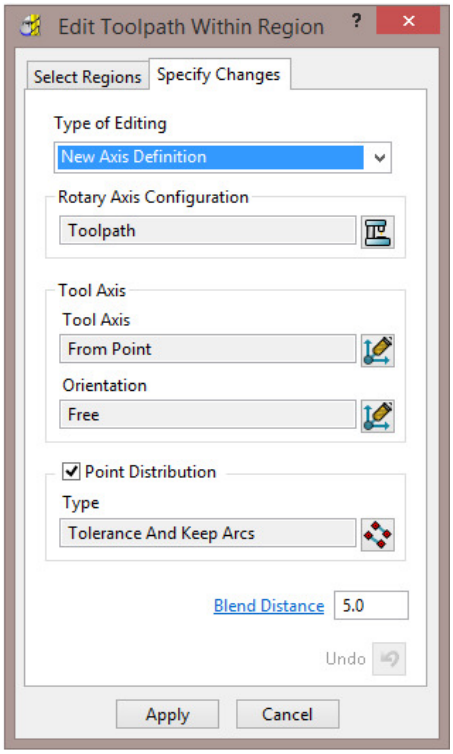


25 Select a **View from top**.

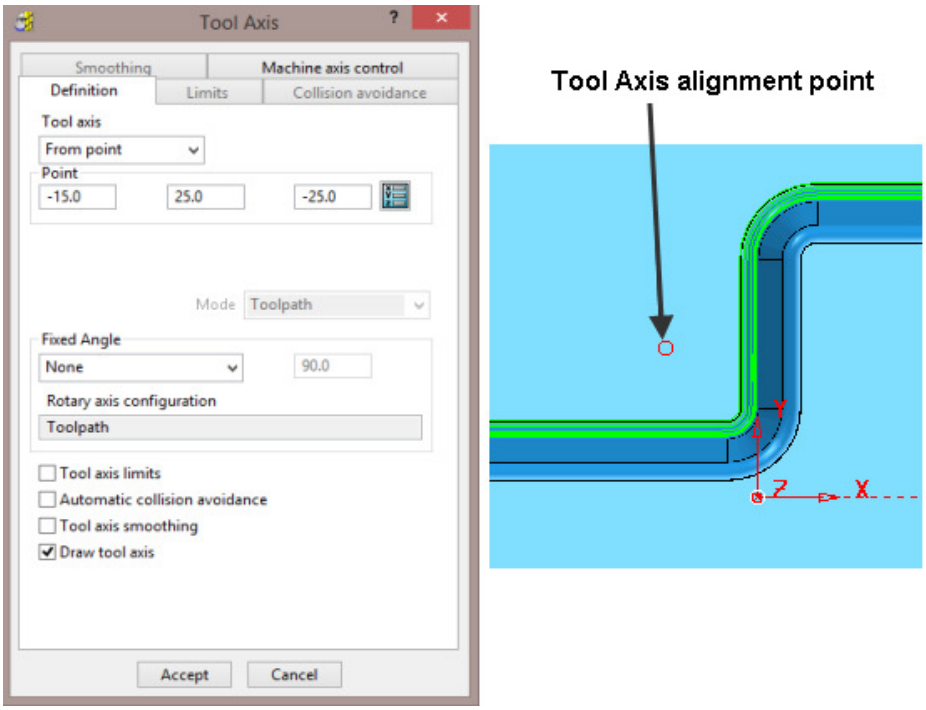
26 In the **PowerMILL *explorer***, right-click on the toolpath **BN5-Rest-Lean45** and select **Edit** > **Edit Within Region** to open the **Edit Toolpath Within Region** dialog again.

27 In **Select Regions** select **Polygon** with ***Side*** - **Inner**, and then click the **Specify Changes** tab.

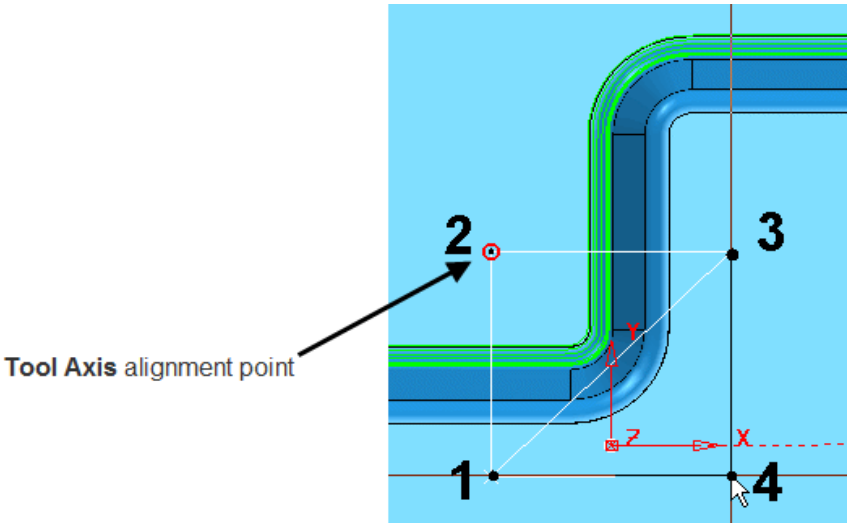




28 Set ***Tool Axis*** - **From Point** with the coordinate values **X-15 Y 25 Z-25**, and then tick the box labelled **Draw Tool Axis** before selecting **Accept**.



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**.



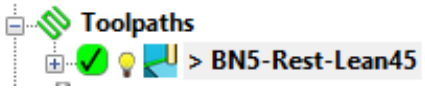
30 Select an **ISO 1** view.

31 From the local **BN5-Rest-Lean5** ***toolpath*** menu, select **Verify** - **Toolpath** and in the dialog, un-tick both **Split Toolpath** and **Adjust Tool**.

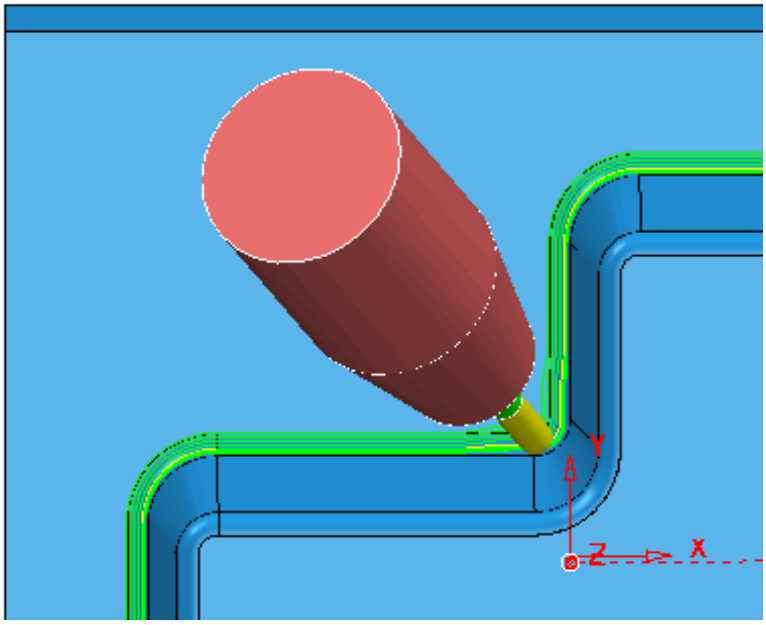
32 Select ***Check*** **Collisions**, with a **Shank Clearance 1** and **Holder Clearance 2** and **Apply** the dialog.

33 Select ***Check*** **Gouges** and **Apply** the dialog.

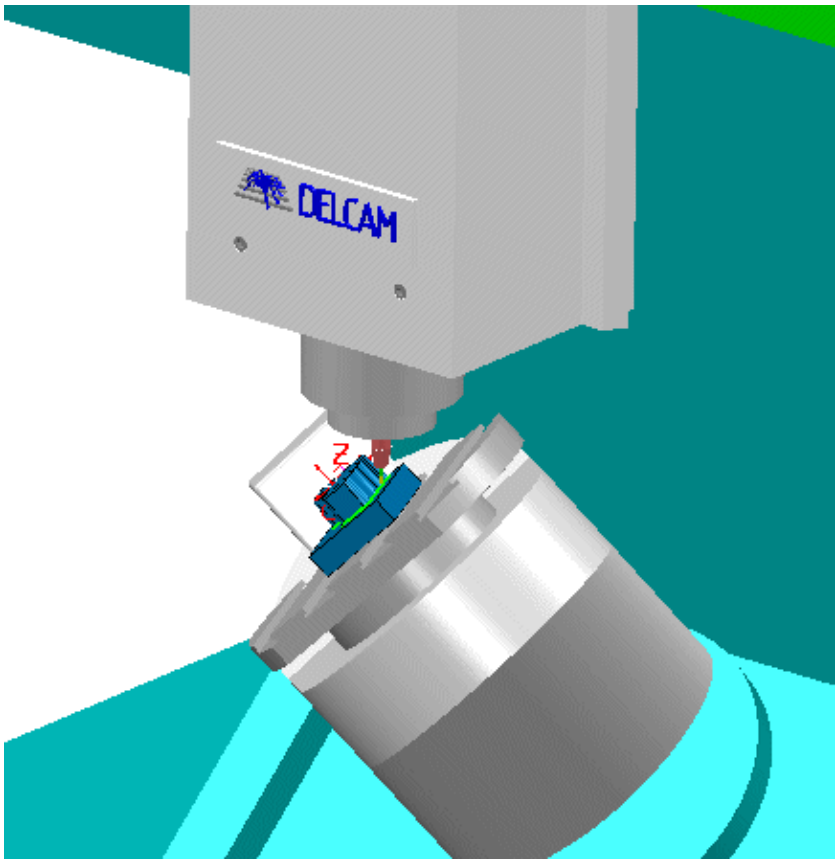
The **Toolpath** is now displayed green in the **PowerMILL explorer** confirming that the toolpath is free of both **Gouges** and **Collisions**.



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**.

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:

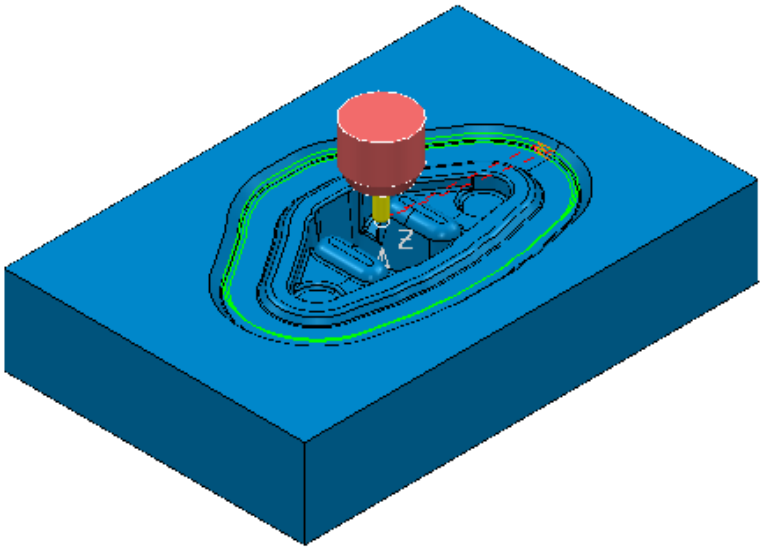
….\PowerMILL_Data\MachineData

*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**:

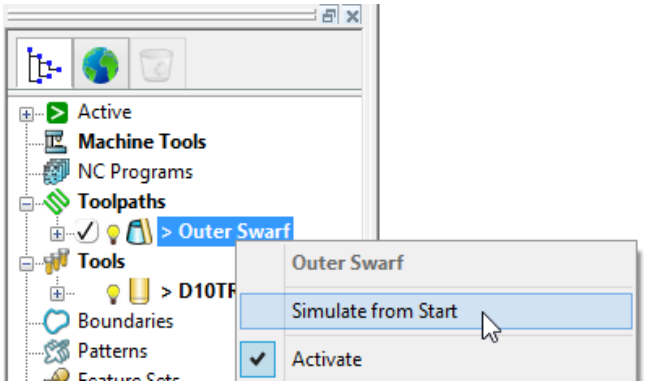
....\PowerMILL_Data\five_axis\Collision_Simulation\Swarf_check



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**.

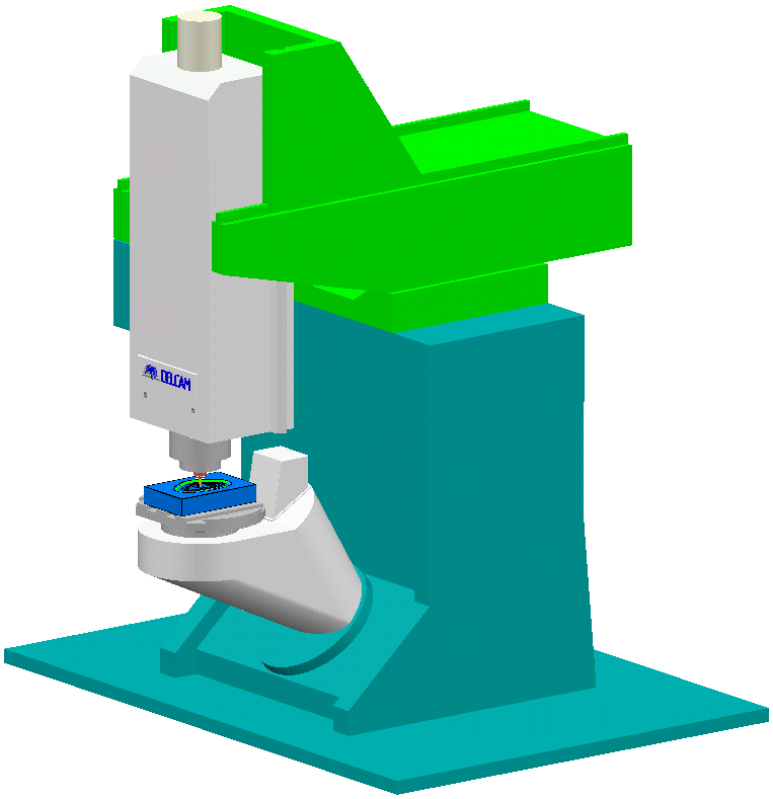


5 The **Simulation** toolbar will appear (if it is not already displayed).

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



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.

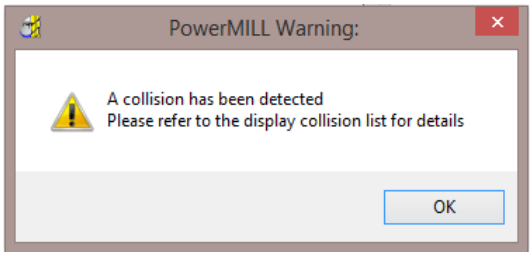
11 Select ***Display the machine tool issues*** from the **Simulation** toolbar.

During a toolpath **Simulation**, the **Machine Tool 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.

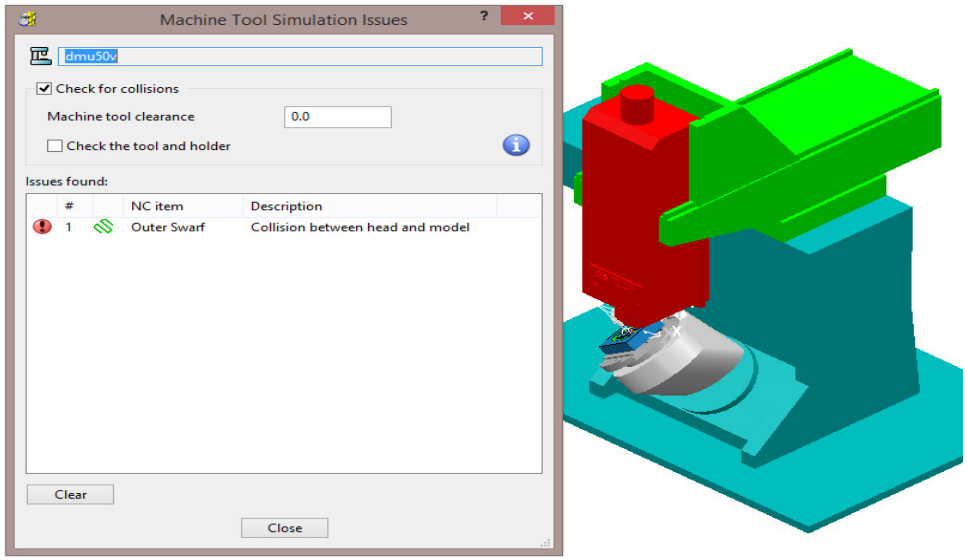
12 Start the **Simulation** to observe the virtual machining of the component.



A collision has been detected and a ***PowerMILL Warning*** message displayed.

The warning message is only displayed for the first collision.

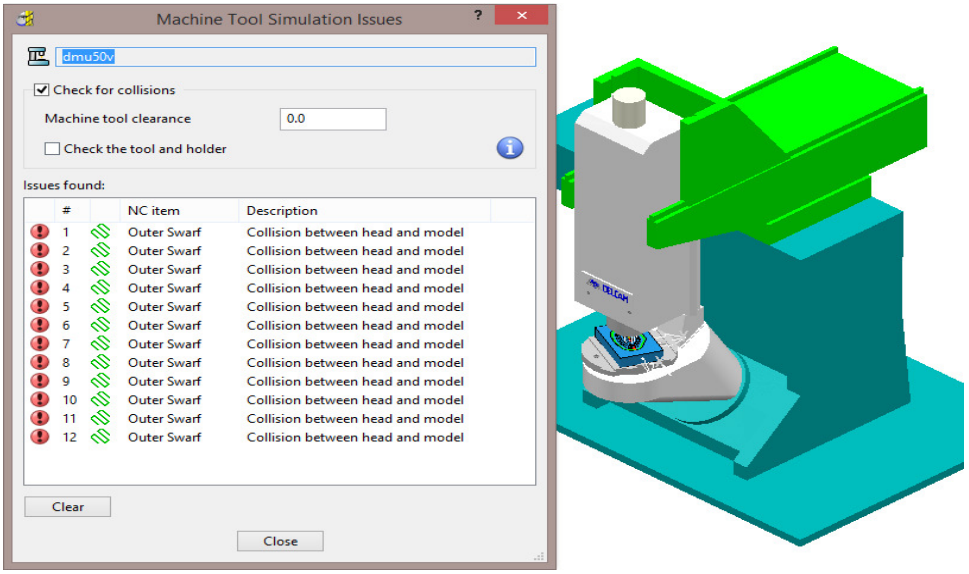
13 Select **OK** to close the **PowerMILL Warning** dialog.



The first collision move is registered and displayed in the **Issues found** pane.

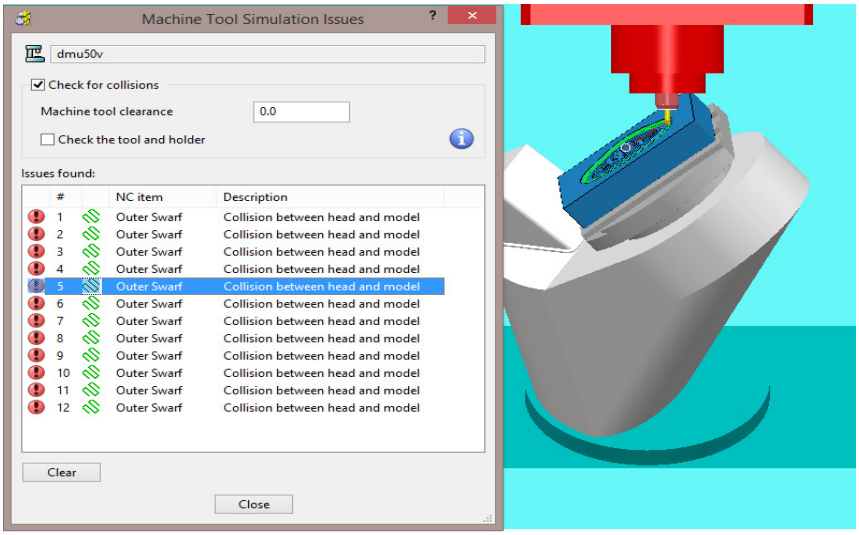
At present only the first collision is displayed until the **Simulation** is continued.

14 Continue the **Simulation**. 



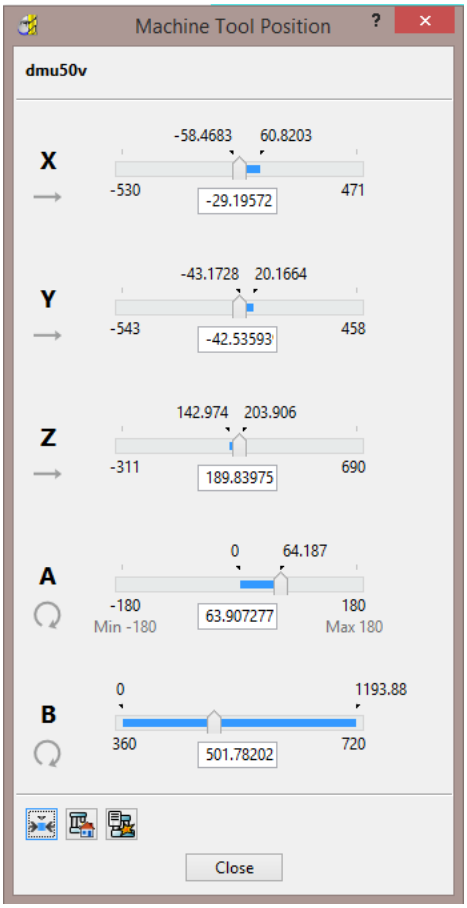
On completion of the simulation the machine tool is displayed in the normal colour scheme.

15 Select a ***Collision Move*** from the list.



The **simulation** will move directly to the selected position in the form so that the collision (top right corner of component) can be viewed.

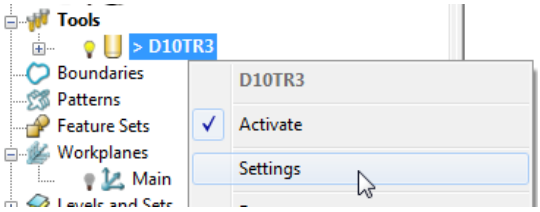
16 Select ***View and adjust the Machine Tool position***  from the **Simulation** toolbar.



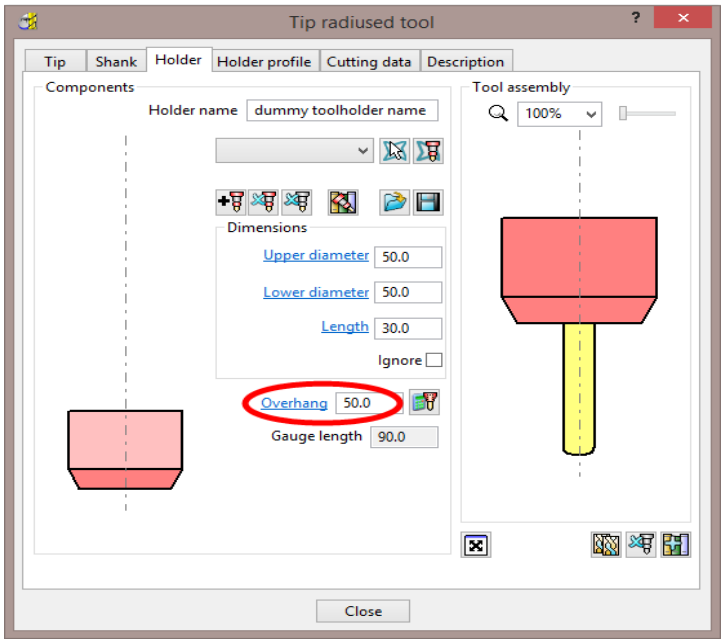
The current positional values are displayed in the **Machine Tool Position** dialog.

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.

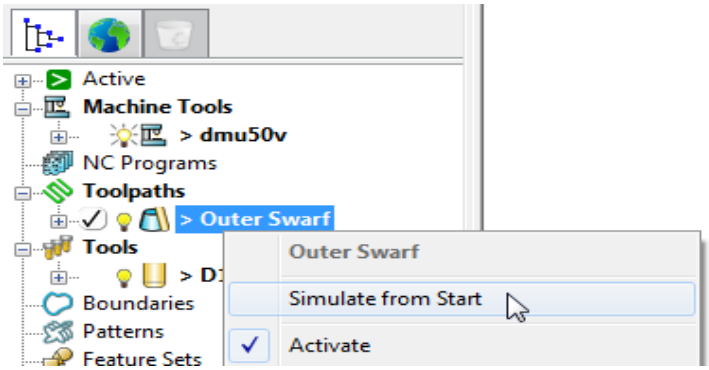
17 In the **PowerMILL *explorer*** right mouse click on the tool **D10TR3** and select **Settings**.



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**'.

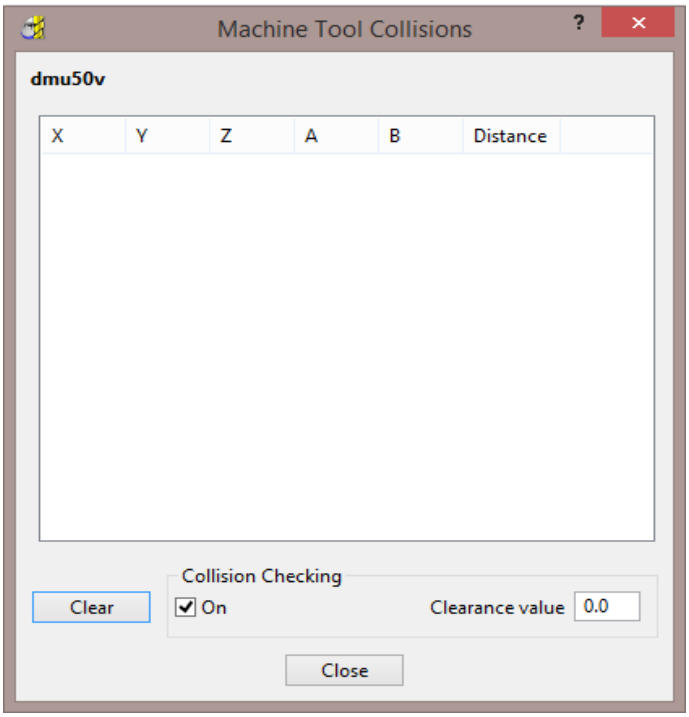


20 Select ***Switch on the Machine Tool Display***  from the **Simulation** toolbar.

21 Select the **Collisions** tab on the dialog.

22 Select **Clear** on the dialog to clear any existing collisions.

23 Start the **Simulation** .



The **Machine Tool Collisions** dialog remains blank indicating that no collisions have been detected.

24 Select **File** > **Save Project**.

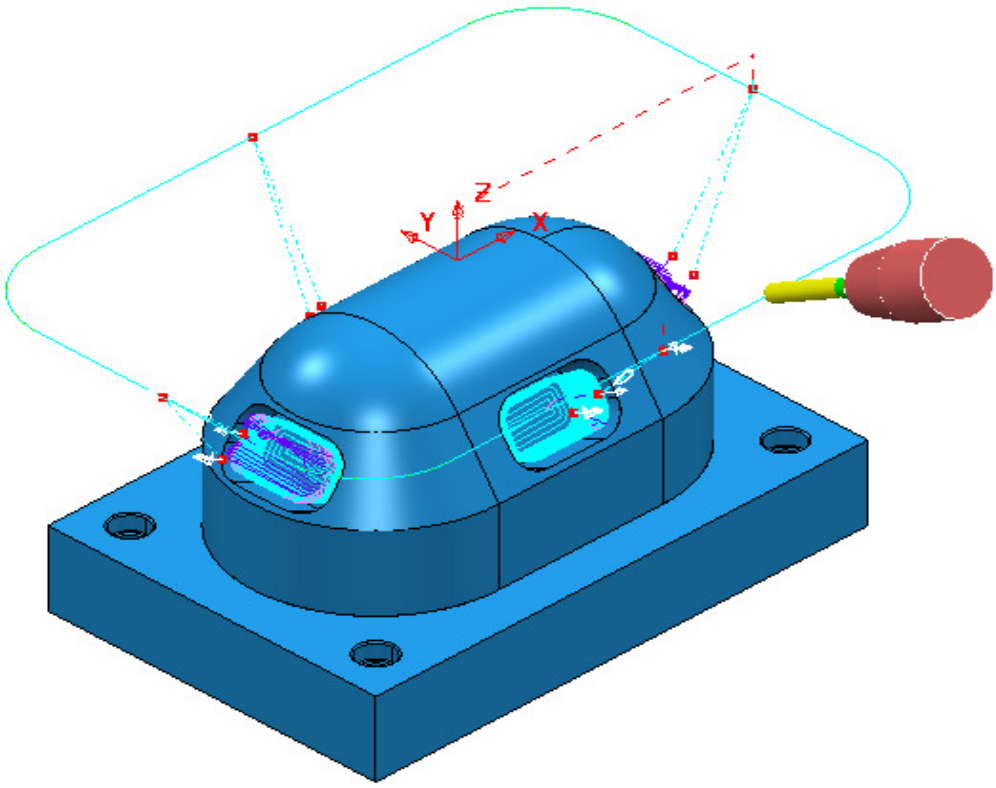
25 Select **File** – **Delete All** and **Tools** – **Reset Forms**.

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



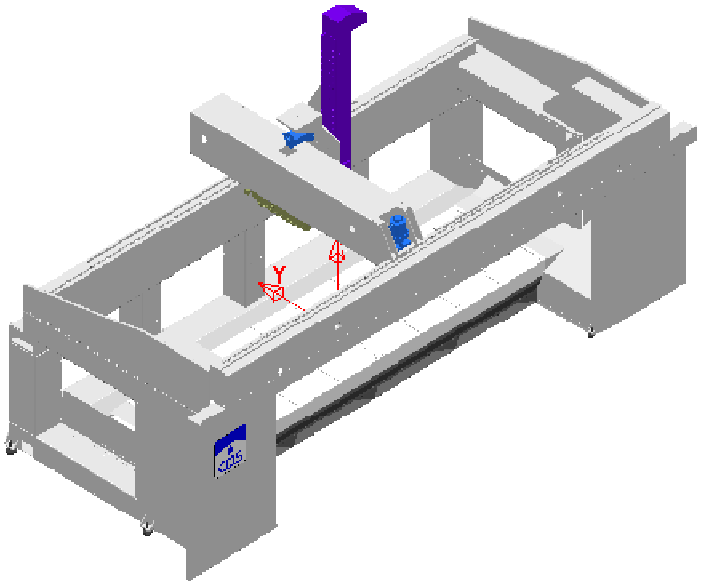
Select **File** - **Save Project As**:

....\COURSEWORK\PowerMILL-Projects\AnglePockets-FullVerify

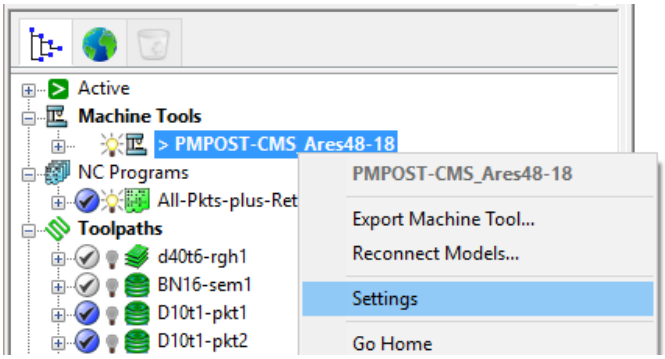
2 In the **PowerMILL *explorer*** right mouse click on **Machine Tools** and from the local menu select **Toolbar**.

3 Select the ***Import the Machine Tool Model*** icon  and select the file **PMPOST-CMS_Ares48-18.mtd** from:

….\PowerMILL_Data\MachineData\CMS\.....

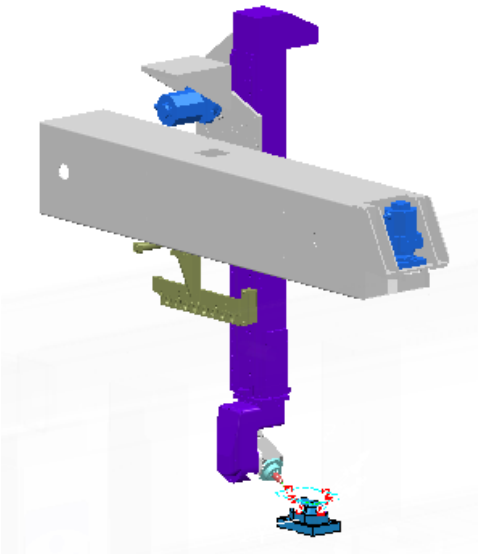


4 In the **PowerMILL *explorer*** right mouse click on the **Machine Tool *PMPOST-CMS_Ares48-18*** and fro the local menu select **Settings**.



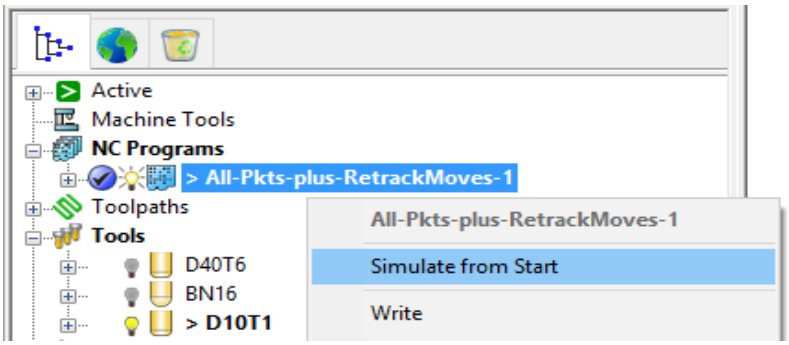
5 In the dialog, select **Part** and move the **Opacity** slider fully to the left to make the static machine tool parts almost invisible.

6 **Apply** and **Accept** the dialog.

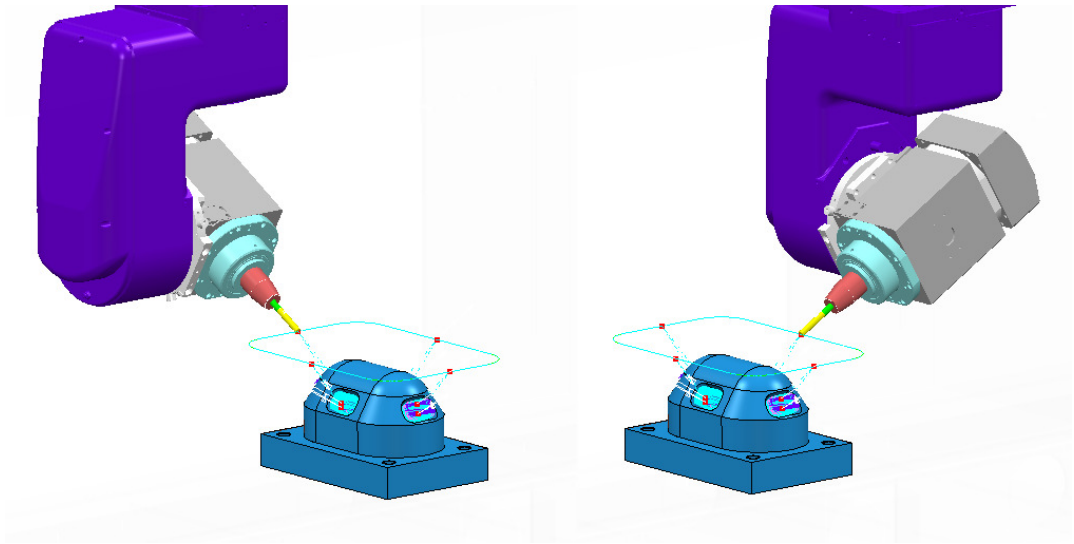


7 In the **Machine Tool** toolbar select the **Workplane *MC-Table-location***.

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**.



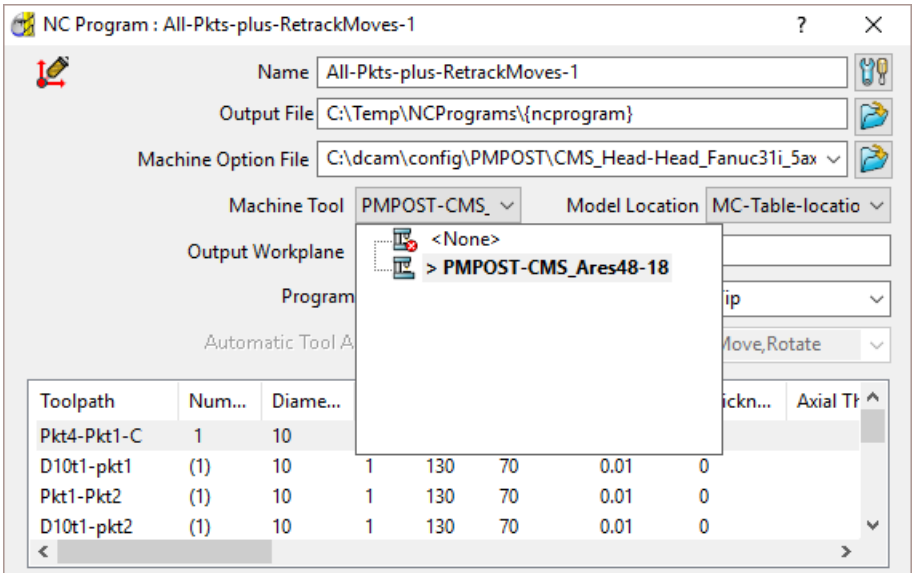
9 In the **simulation** toolbar set the speed slider to medium before selecting **Play**.



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.

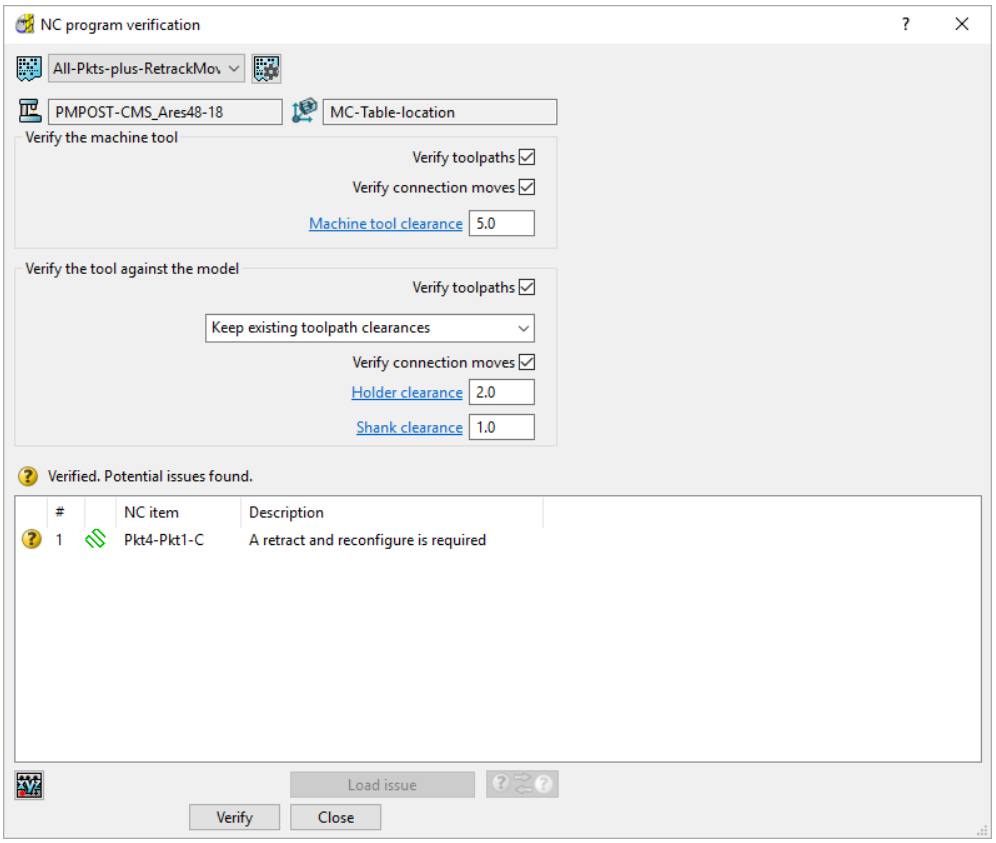
11 In the ***Machine Tool*** box select the **PMPOST-CMS_Ares48-18** and for ***Model Location*** select **MC-Table-location**.



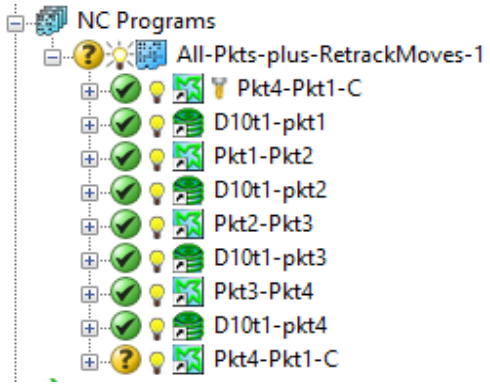
12 At the base of the dialog select **Apply** followed by **Accept**.

13 In the **PowerMILL *explorer*** right mouse click on the **NC Program** ***All-Pkts-plus-RetracMoves-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.



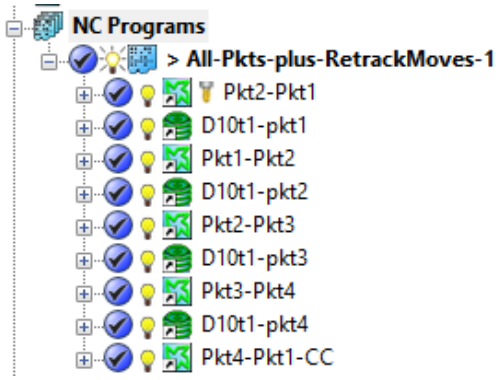
In the above **NC Program** dialog it has been reported that one of the toolpaths is causing the rotary limits to be exceeded. This CMS machine tool can only perform a total of one 360 Degrees rotation about the C axis. The total of C Axis rotation currently required by the **NC Program** is 450 Deg.



This can be fixed by using a different positional move toolpath at the ends of the NC Program that will start well within the main machining areas and re-track the tool back to the original start position.

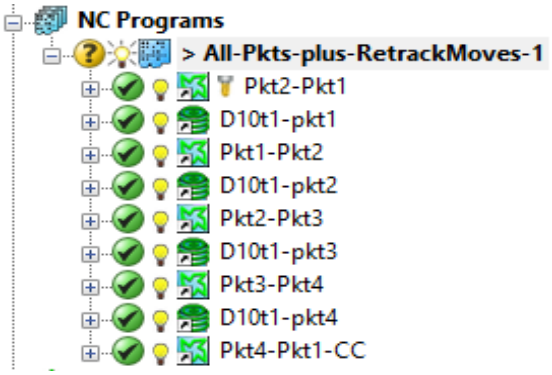
15 In the **PowerMILL *explorer*** right mouse click on the **NC Program** ***All-Pkts-plus-RetracMoves-1*** and from the local menu select **Clear Verification**.

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.



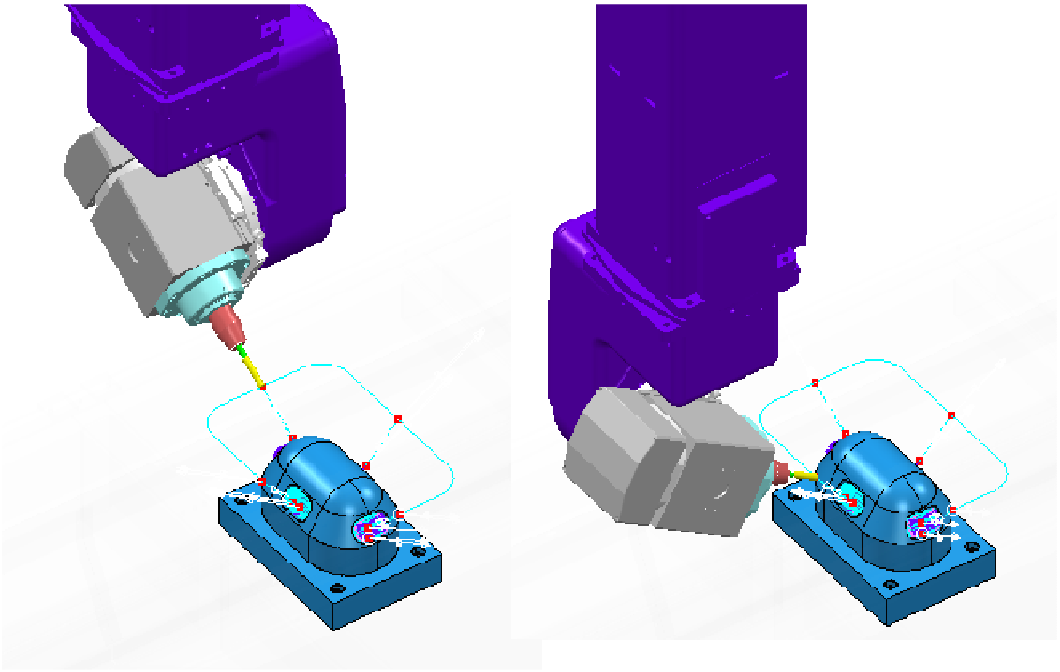
The replacement toolpaths preform repositioning with counter clockwise rotation about the C axis. This will keep the total C rotation moves well within the CMS machine limits.

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**.



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**.

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.

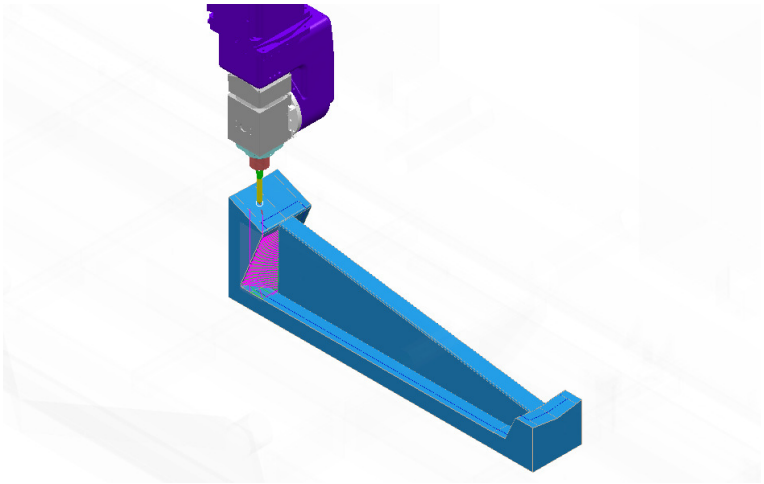
Machine Axis Control options allow the user to avoid collisions by controlling the ***orientation/movement*** of the independent parts of a machine tool within the machining strategy.

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 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**:

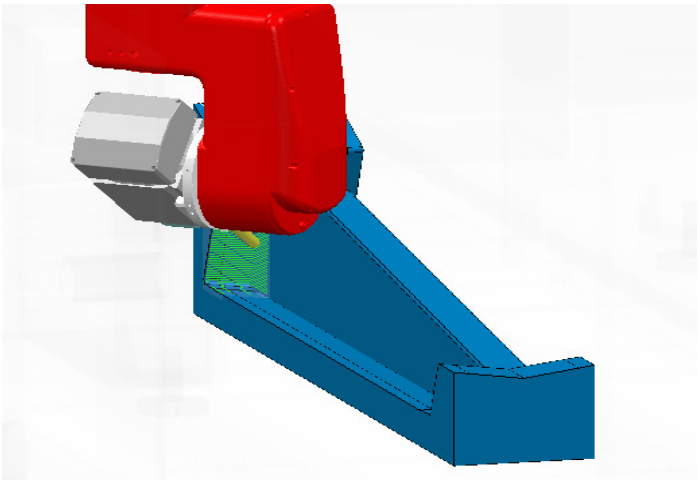
….\PowerMILL_Data \five_axis\MachineAxisCtrl\CMS-AxisControl_Start



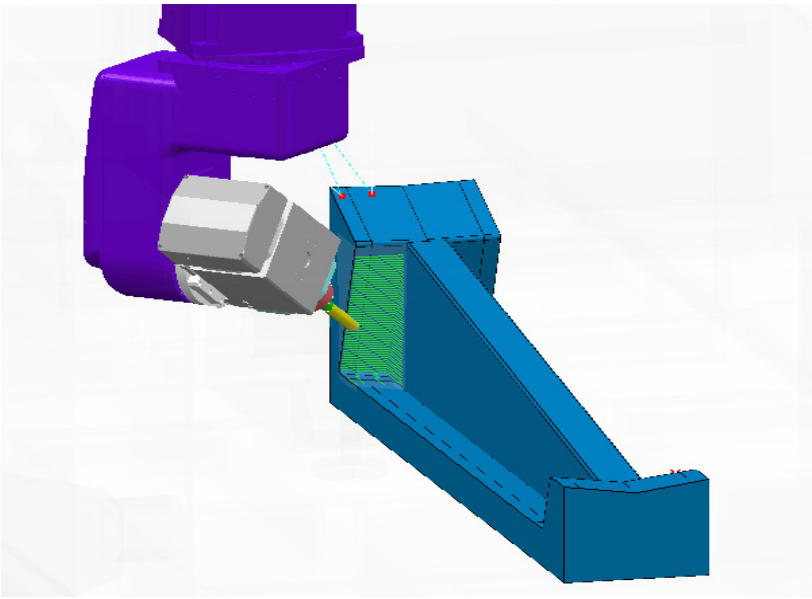
2 From the **main** pull down menus select **File** > **Save Project as**:

….\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**.

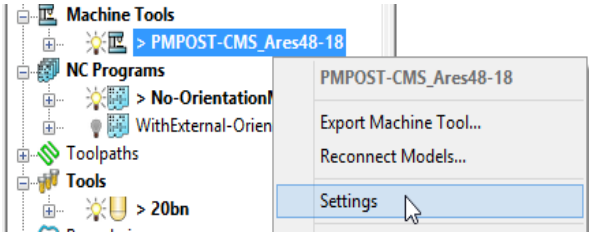
5 Input **Base-Centre** in the ***workplane*** input box of the **Machine Tool** toolbar.

6 In the **Machine Tool** toolbar input the machine tool model:-

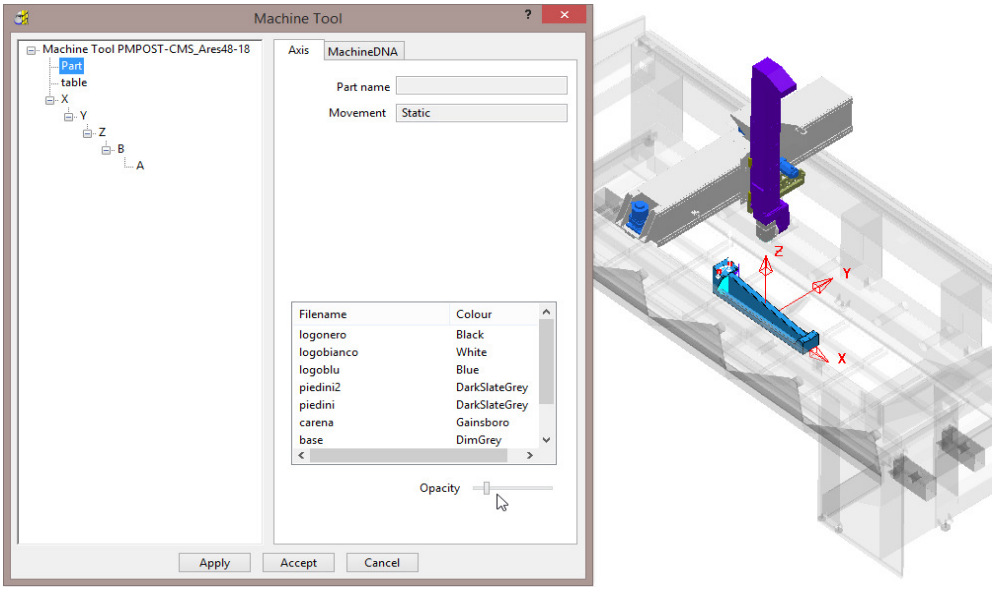
....\PowerMILL_Data\MachineData\CMS\PMPOST-CMS_Ares48-18.mtd

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**.



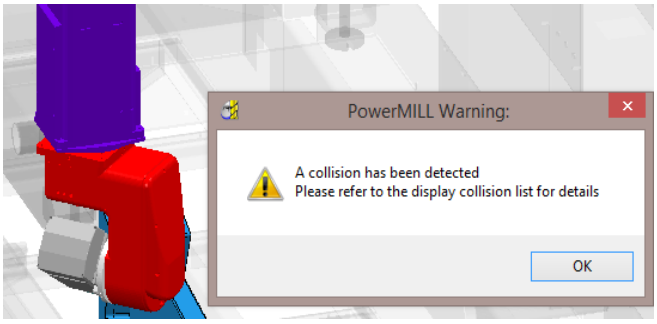
9 In the dialog select the item, named **Part** and then move the **Opacity** slider to make the static part of the machine tool model, translucent.



10 **Accept** the **Machine Tool** dialog.

11 In the **PowerMILL *explorer***, right mouse click on the **NC Program *No-Orientation Moves*** and from the local menu select **Simulate from Start**.

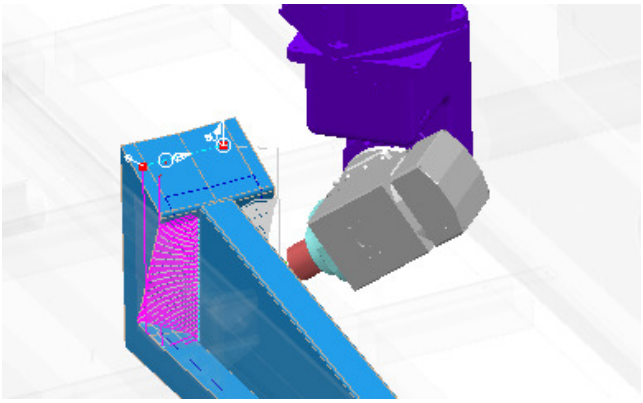
12 Select the **Play** button in the **simulation** toolbar.



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.

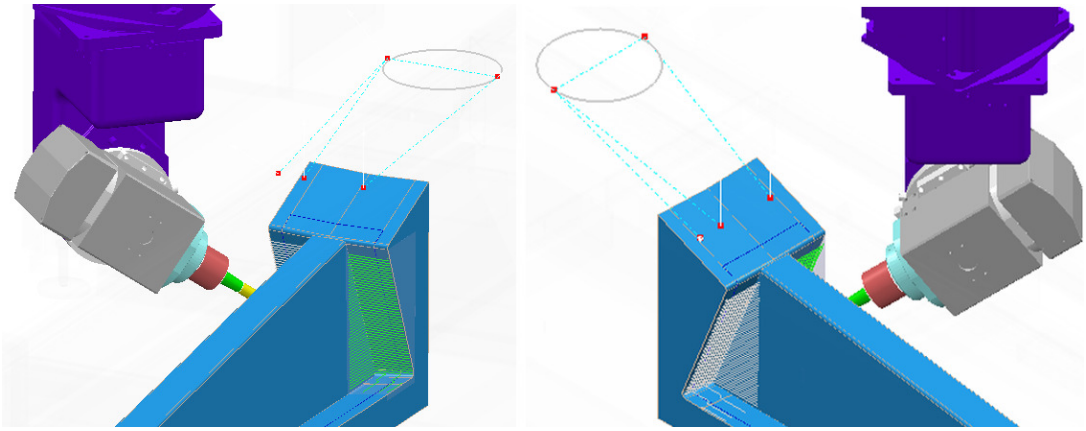
13 Select **OK** and the **simulation** will continue even though the head is still colliding (The head will change colour to ***red*** during each collision).

Collision checking and reporting to a Machine Tool Model simulation is a cost option. Otherwise the checking process is purely visual.

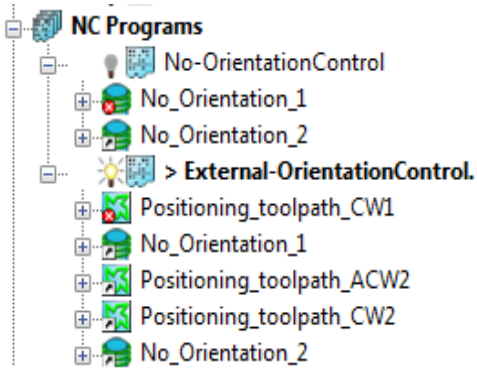


When the ***NC Program* simulation** moves to the other side of rib, the machine tool head happens to be in a collision safe, orientation relative to the component. This is not easily predictable and could easily change to a collision situation on switching the order or inserting new toolpaths into the **NC Program**.

14 In the **PowerMILL *explorer***, right mouse click on the **NC Program *WithExternal-OrientationMoves*** and from the local menu select **Simulate from Start**.



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**.

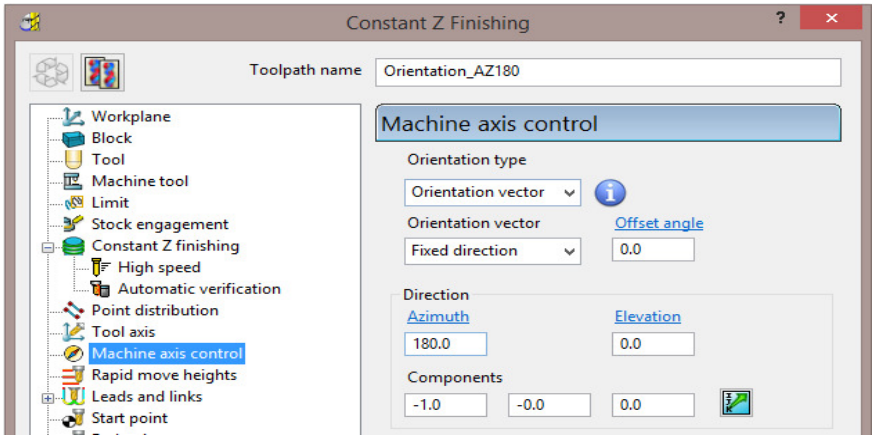
A more efficient and direct method to apply machine tool head alignment within an individual strategy is to use the **Machine axis control** dialog.

15 Ensure that neither **NC Program** is active.

16 **Activate** and then select **Settings** from the local menu for **toolpath *No_Orientation_1***.

17 In the **Constant Z** strategy 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**, ***Orientation vector*** as **Fixed direction**, and input a ***Direction*** - **Azimuth 180**.



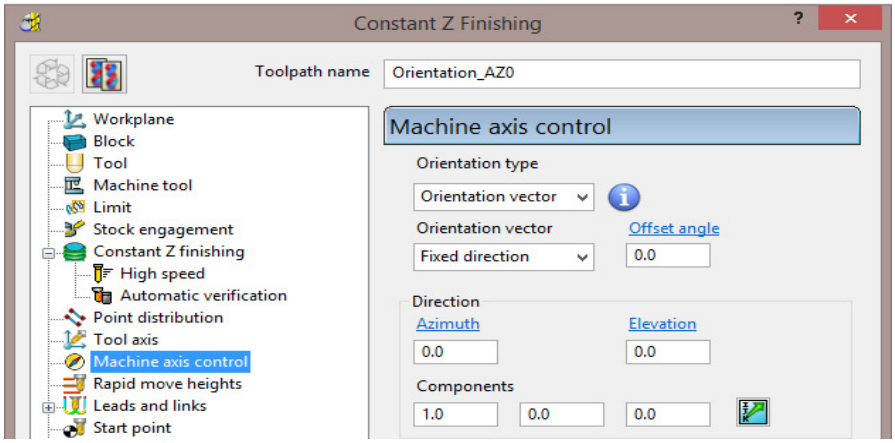
By changing the default **Orientation type** from **Free** to **Orientation vector** the machine head alignment will now be controlled directly within the strategy. The **Machine axis control** will cause the head to be rotated by **180** Degrees about the **Z** axis compared with the original toolpath.

19 Select **Calculate** to create the new toolpath and close the dialog.

20 **Activate** and then select **Settings** from the local menu for **toolpath *No_Orientation_2***.

21 In the **Constant Z** 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**.



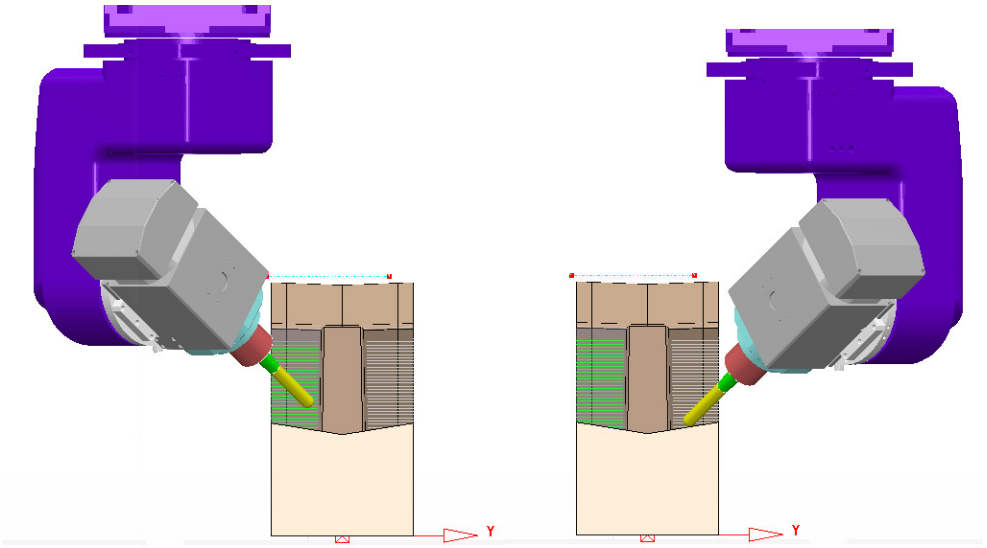
The **Machine axis control** will use the default values for an **Orientation Vector** of **Azimuth 0** and **Elevation 0**.

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 the Advanced Simulation cost option)

The **Dynamic Machine Control** options  are accessed from the **Machine Tool** toolbar.

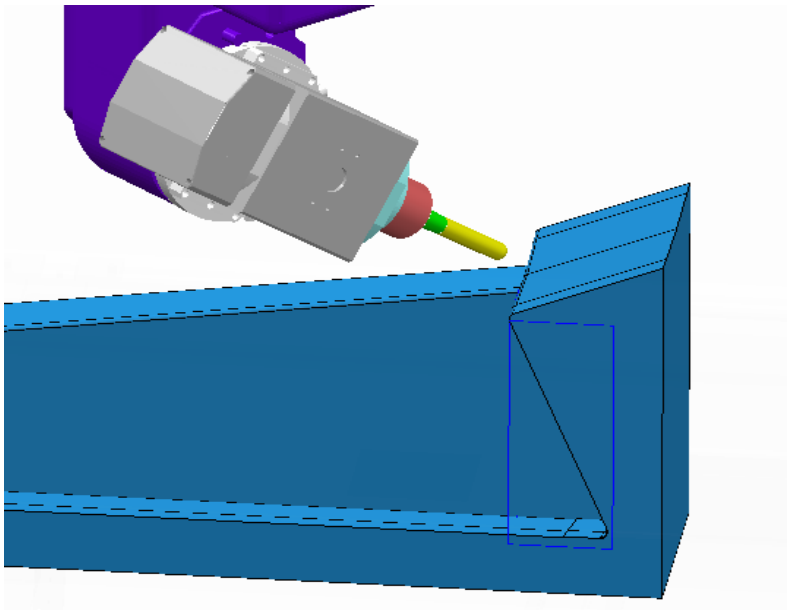
It is possible to apply machine axis control dynamically using options from the **Dynamic machine control** toolbar (shown below).

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

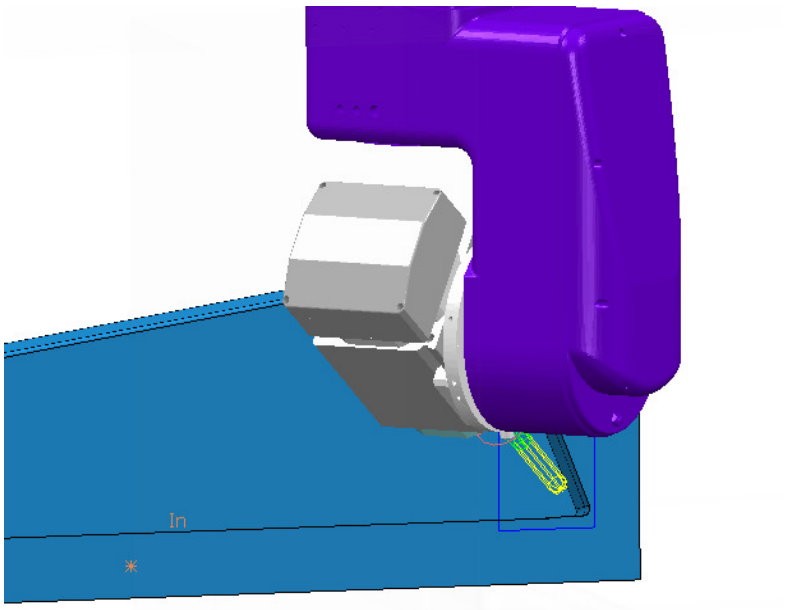
1 In the **PowerMILL explorer** right mouse click on the **Toolpath** ***No_Orientation_2*** and select **Simulate** from start.

2 **Jog** the tool to the start of the fixed machining alignment position and then from the **Machine Tool** toolbar select The **Dynamic Machine Control**. 

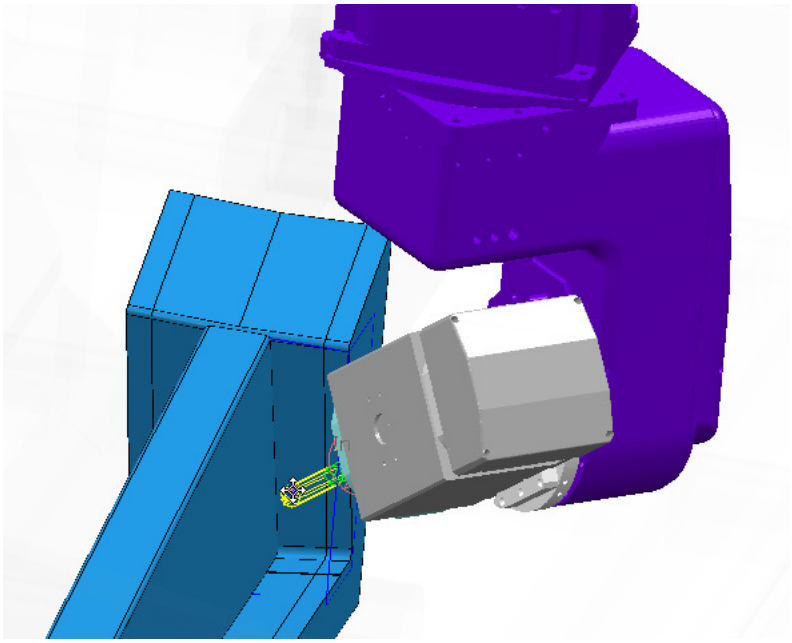


3 In the **Dynamic machine control** toolbar select **use tool positioning graphics**. 

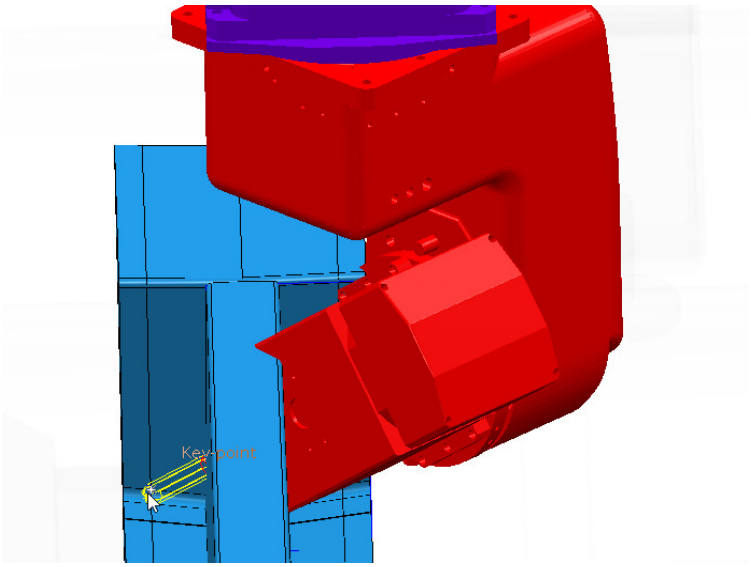
4 Left click on the tip of the **Tool** and then dynamically move it down the angled model, undercut towards the base of the recess.



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.*

Dymamic 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).

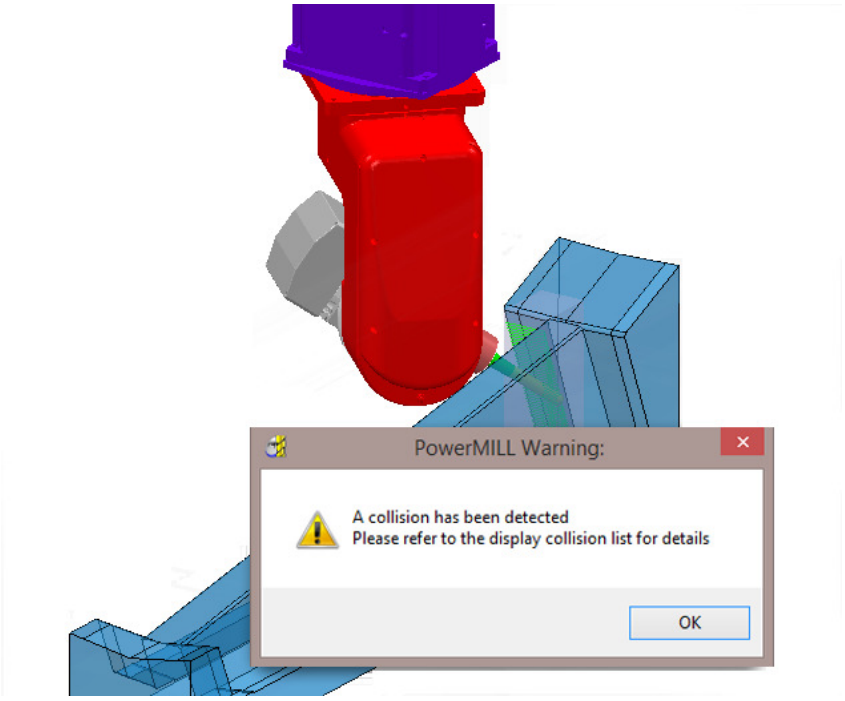
b **Active workplane graphics mode** - Applies dynamic rotation and linear moves relative to the active workplane.

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**.

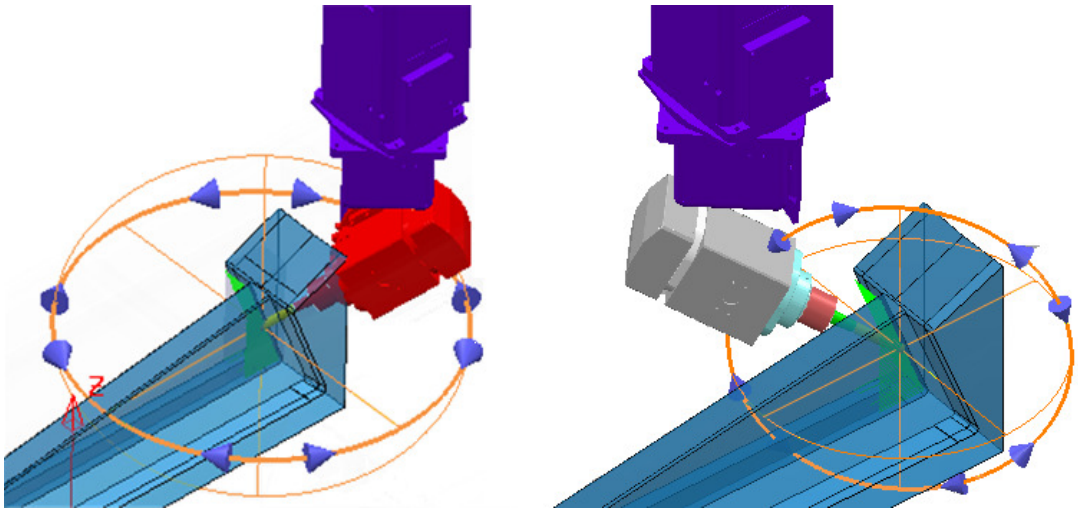
3 **Activate** the new **toolpath** and run a **Machine Tool simulation** until the collision with the ***Head*** is first detected.



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  .

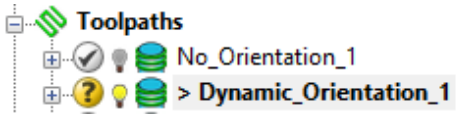


7 Use the upper circular handle to rotate the head to the above left, alignment, followed by the lower circular handle to re-orientate the tool to the above right alignment.

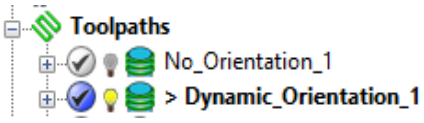
8 Once a suitable ***Machine Tool Orientation*** has been achieved select the ***Update Toolpath***  option.

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**.*

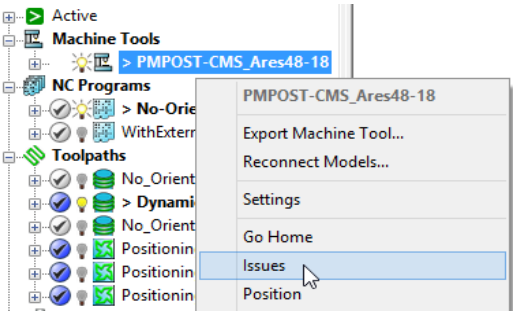


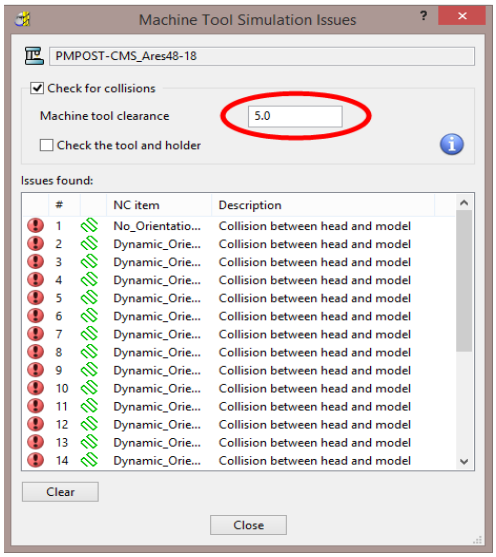
10 **Verify** the modified **Toolpath** ***Dynamic_Orientation_1*** for both **Gouges** and **Collisions** (Clearances:- ***Holder* 2** and ***Shank* 1**).



The **Toolpath** is now confirmed to be both **Gouge** and **Collision** safe relative to the **Tool** assembly.

11 Right mouse click on the machine tool model and from the local menu select **Collisions**.

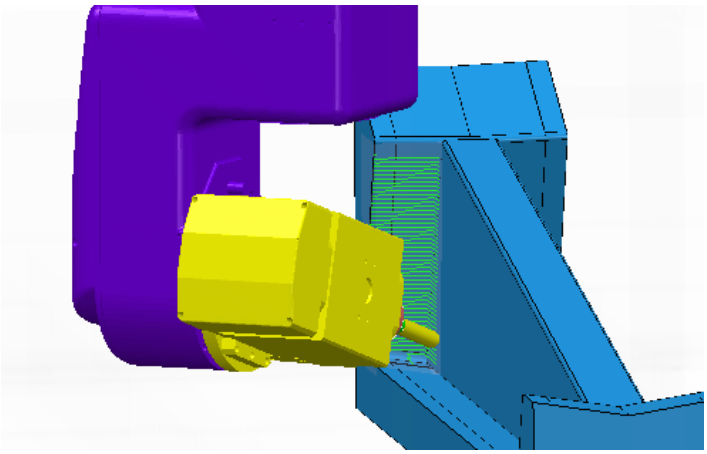




12 In the dialog input a suitable clearance value for the Machine Tool head.

13 **Clear** the issues found, list and perform further ***machine tool*** **simulation** to confirm that the **Head** assembly also collision safe.

During a simulation the machine tool Head will change colour yellow if a collision situation is identified.



14 **Save** the **Project**.

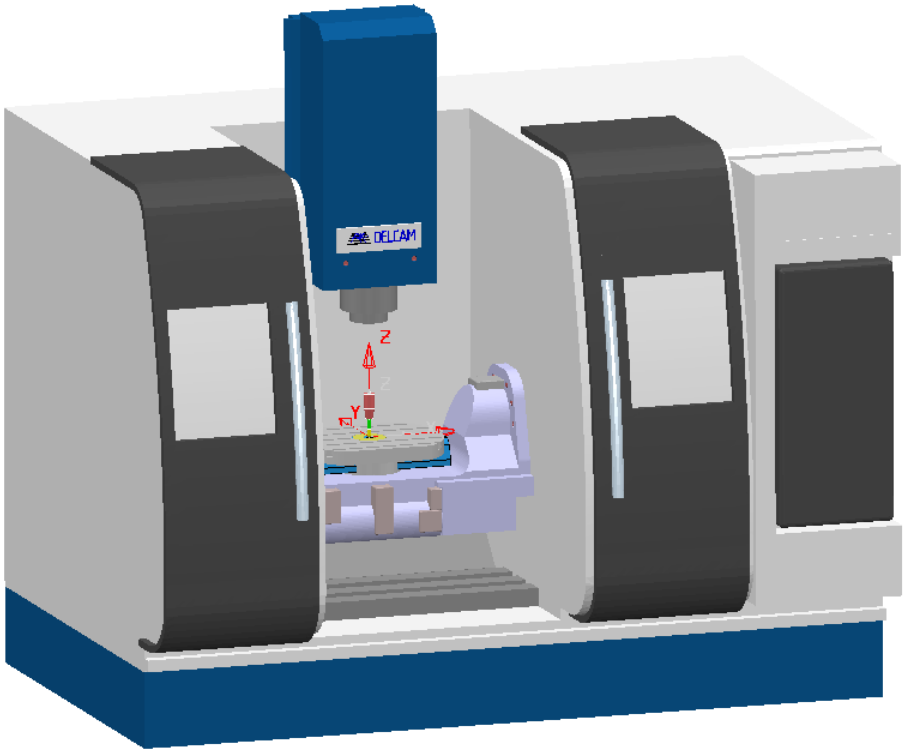
15 Select **File** - **Delete All** and **Tools** - **Reset Forms**.

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**:

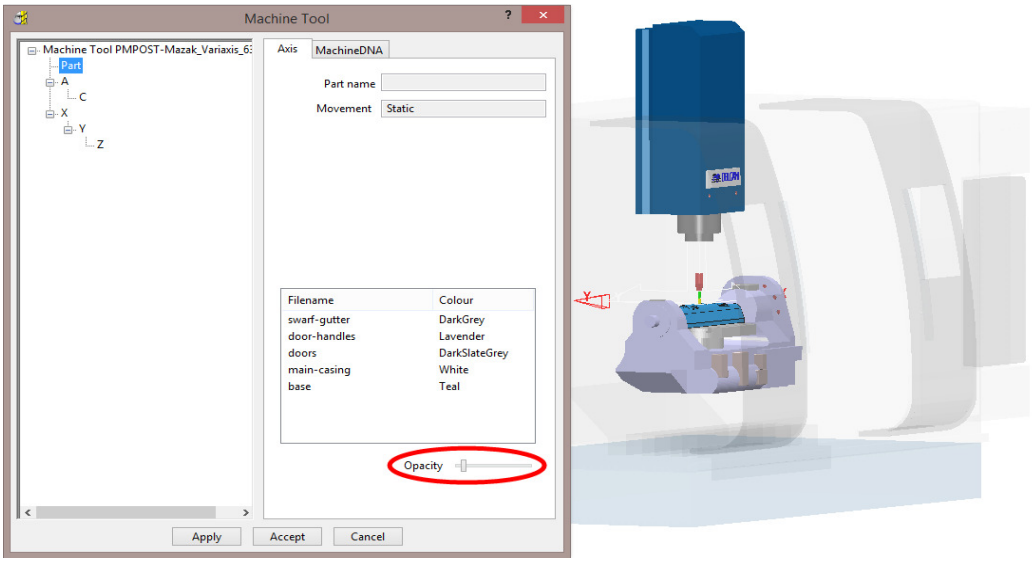
...\COURSEWORK\PowerMILL-Projects\ Mazak-OversizeCompt-ex1

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.

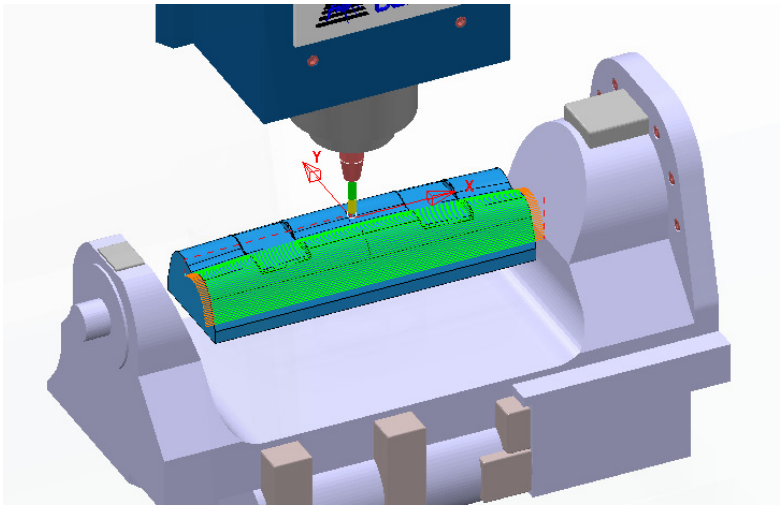
5 In the **Machine Tool** toolbar, input the ***machine tool*** named **PMPOST-Mazak_Variaxis_630PSA** and the ***workplane*** named **TableDatum**.

6 In the **PowerMILL explorer** right mouse click on the **Machine Tool** - **PMPOST-Mazak_Variaxis_630PSA** and from the local menu select **Settings**.

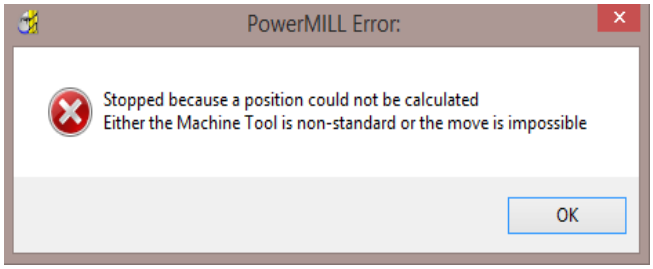


7 In the dialog, select **A** and move the ***Opacity*** slider to the left until the moving parts can be clearly seen behind the opaque, static parts of the **Machine Tool Model**.

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 the following **PowerMILL Error** dialog will open.



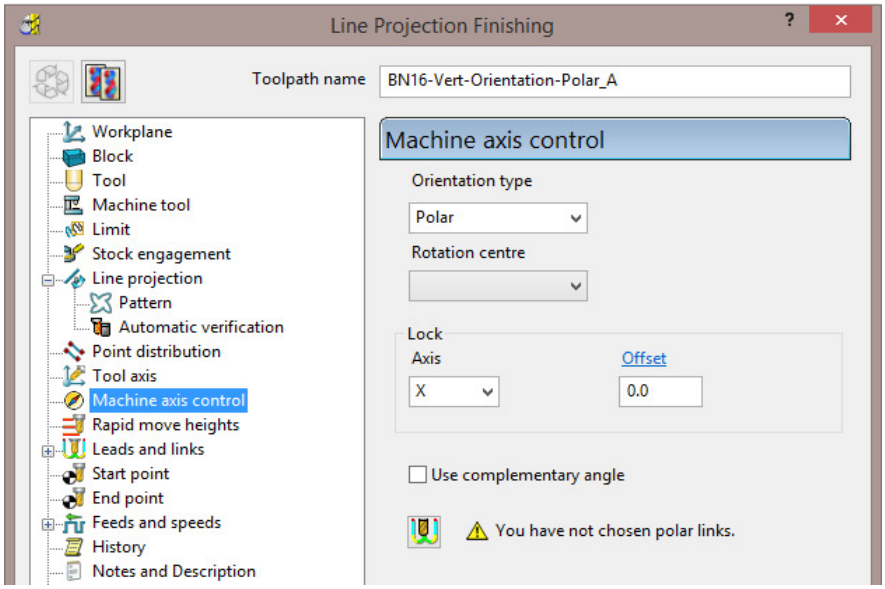
The **X** co-ordinates at the ends of the above **toolpath** are outside the ***X axis***, travel limits. Note, the travel limits are included in the **machine tool model** definition.

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.

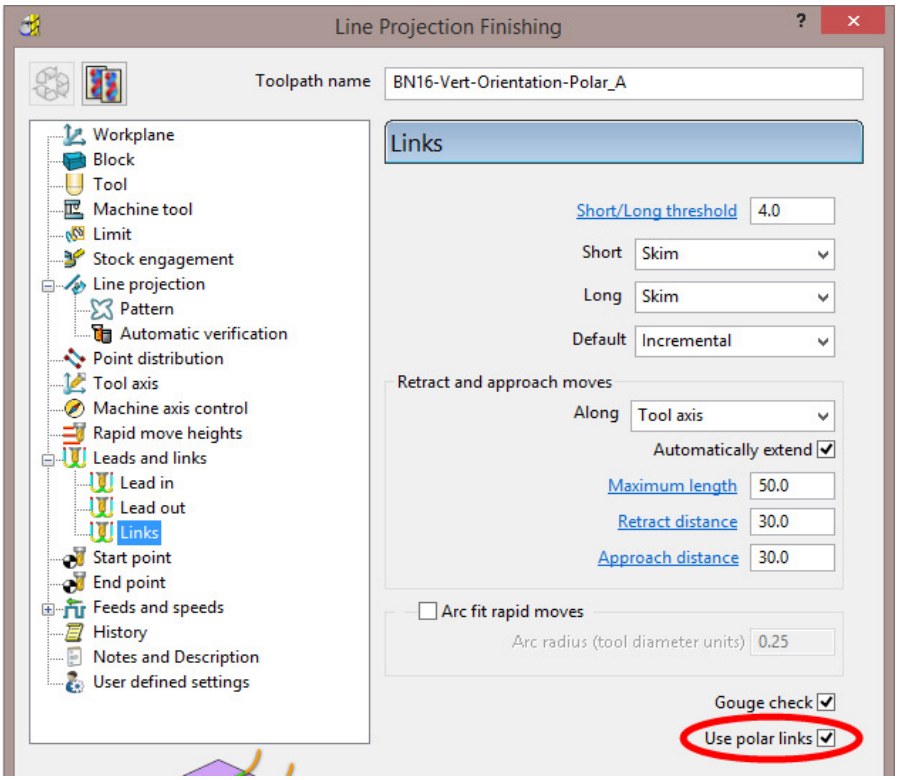
11 In the ***Line Projection Finishing*** dialog select the 'make a copy' option  .

12 In the dialog, ***Rename*** the new toolpath as **BN16-Vert-Orientation-Polar_A**.

13 In the local dialog explorer, select the **Machine axis control** page and input data as shown below.

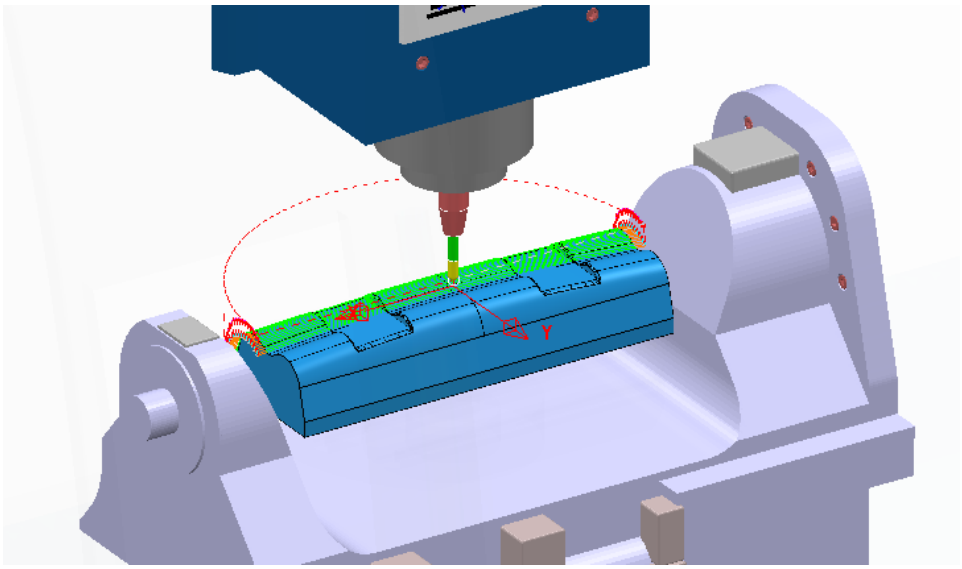


14 Select the icon  to change to the **links** page and input data with **Use polar links** - ticked as shown below.

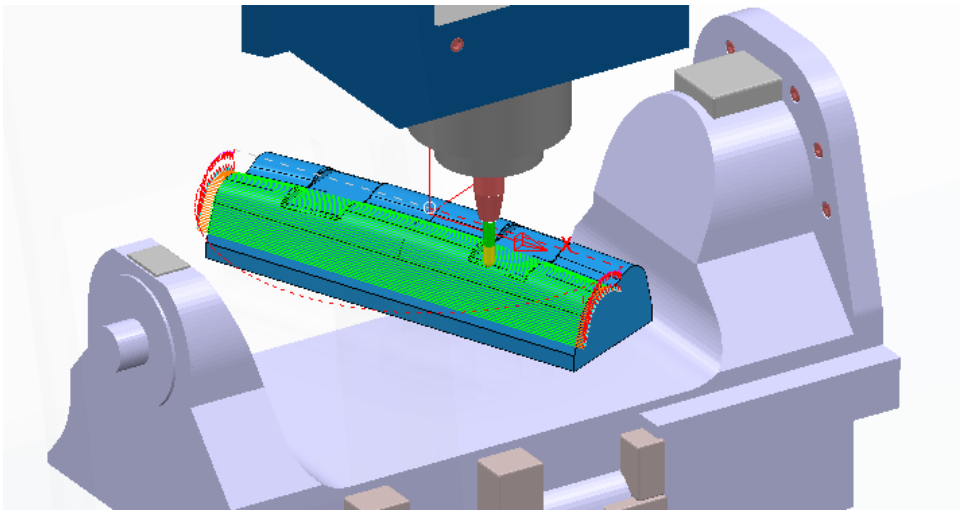


15 **Calculate** the dialog to create the new toolpath.

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**.

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 set up 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

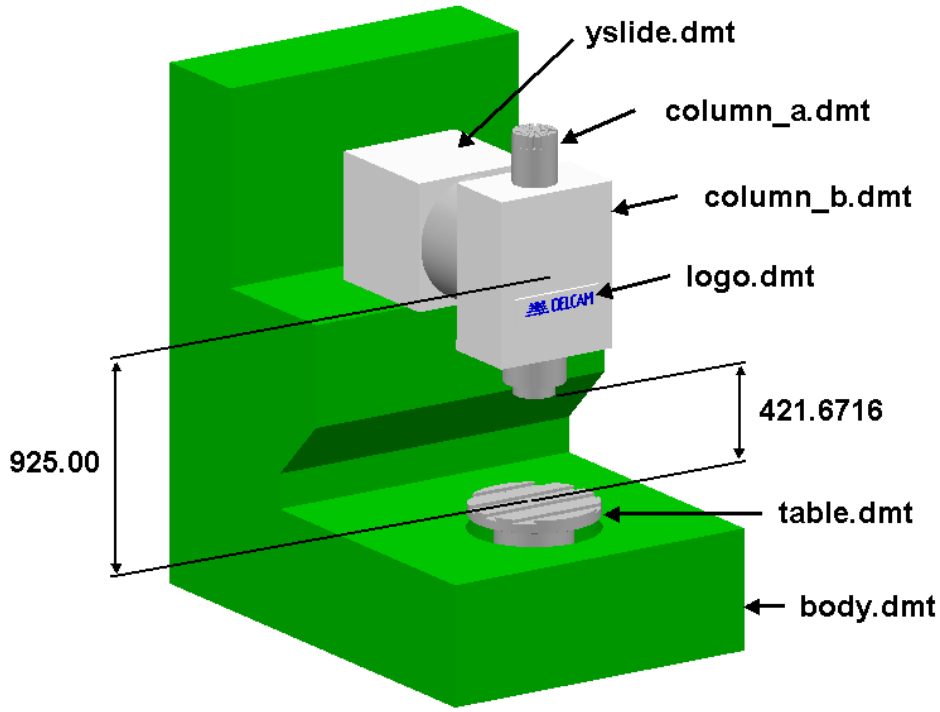
The first stage is to create the separate component models (ideally in **PowerSHAPE**) for each moveable axis for the machine tool. These must be exported as individual **triangle** models to be compatible for use in an **mtd** file. For the following example the models have already been created and 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.***

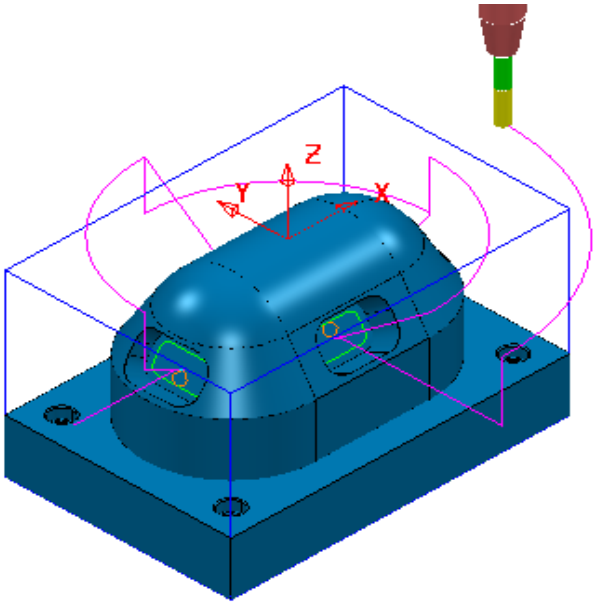
MTD file initial state

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.

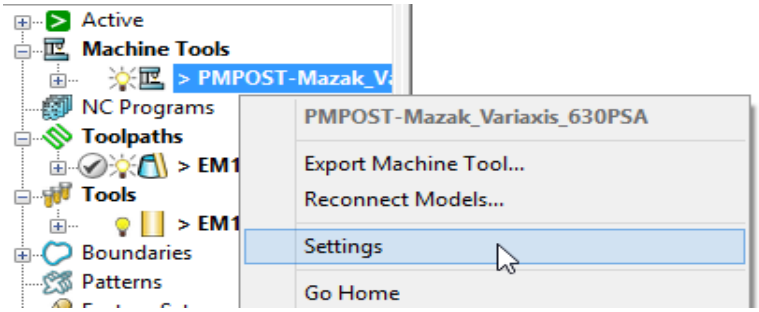
3 From the main pull down menus select **View** – **Toolbar** – **Machine Tool** to open the **Machine Tool** toolbar (Or alternatively right click on **Machine Tools** in the **PowerMILL *explorer*** and from the local menu select **Toolbar**).

In the **Machine Tool** toolbar select the **Import Machine Tool Model**  option.

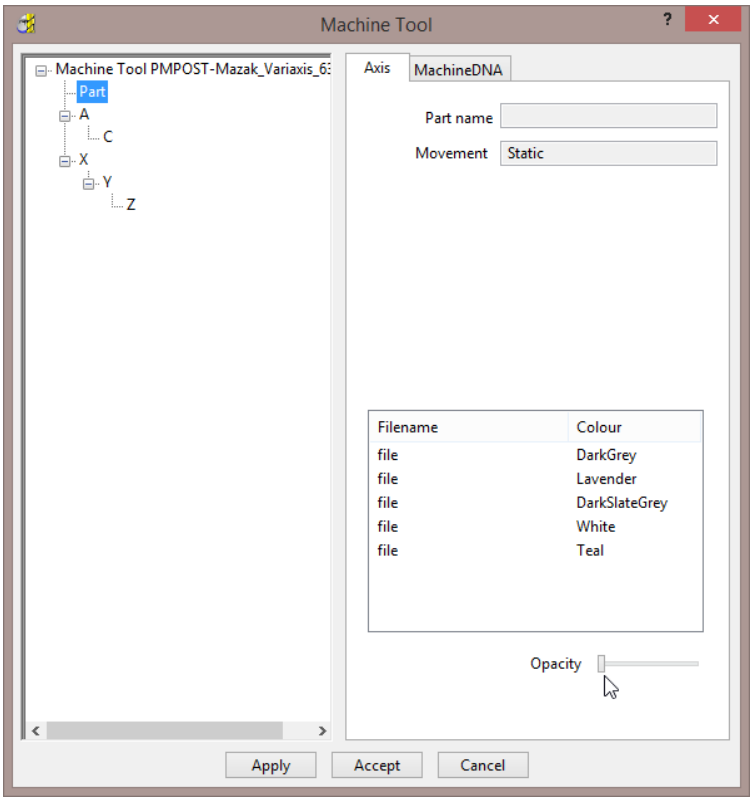
4 In the dialog select the **Machine Tool Model** file:-

….\PowerMILL_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA.mtd

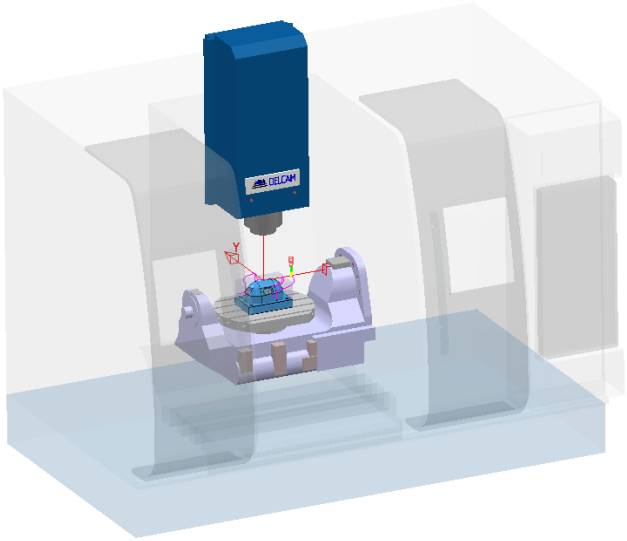
5 In the **PowerMILL *explorer*** right mouse click on the **Machine Tool Model** name **PMPOST-Mazak_Variaxis_630PSA** and from the local menu select **Activate** followed by **Settings**.



6 In the **Machine Tool** dialog select **Part**.

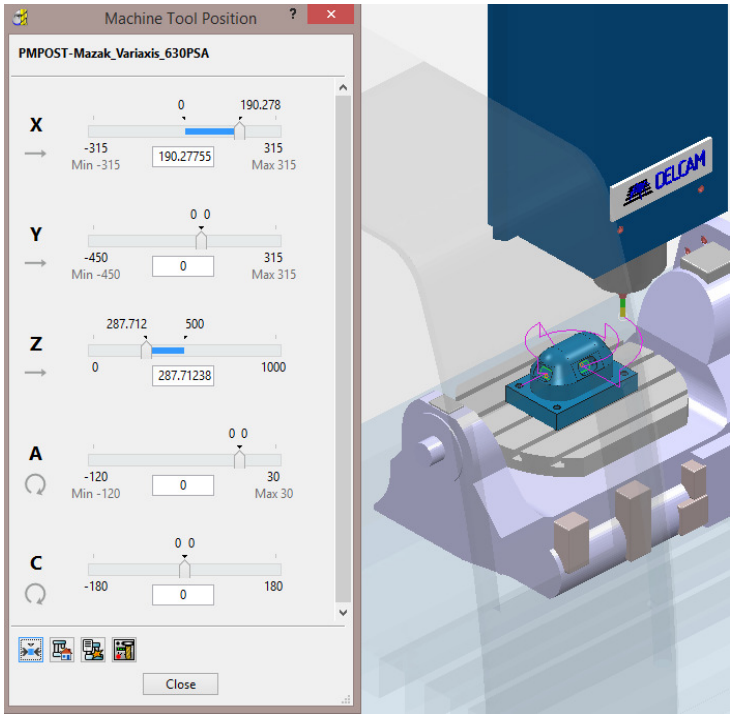


7 Move the **Opacity** slider to the left until the ***Machine Tool Body*** is almost invisible.

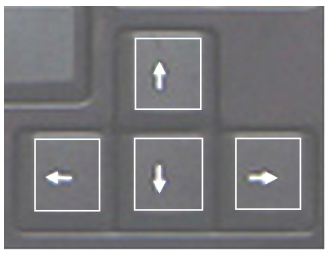


8 Right mouse click on the toolpath **EM12-FinPkts** and from the local menu select **Simulate from Start**.

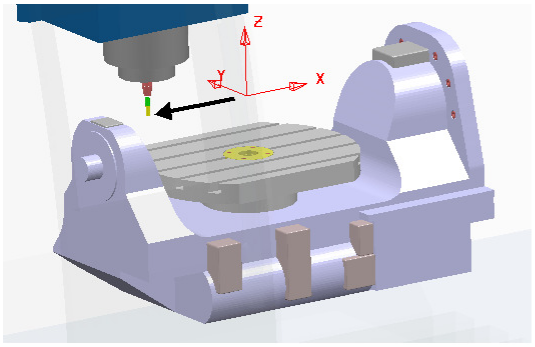
9 In the **Machine Tool** toolbar select the **Machine Tool Display**  options.

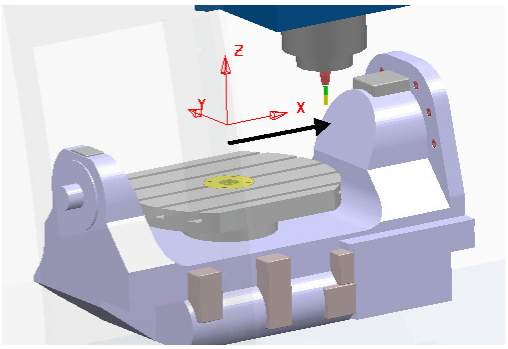


10 Select the **Cursor** on the **X** row slider and drag it to the required position, and/or use the keyboard arrow keys for exact **0.5** incremental moves.



11 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**.



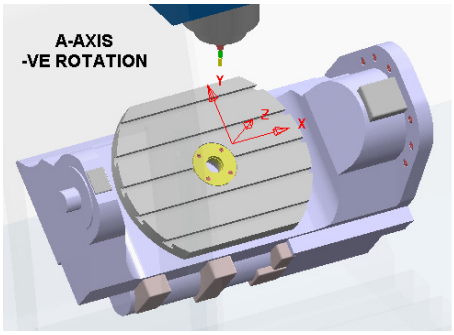


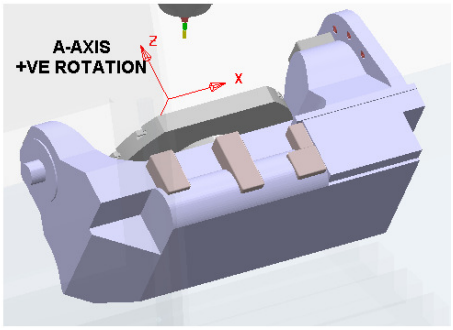
12 Repeat 8, but this time, while holding down the **Shift** key, press the ***right hand*** **Arrow** key to perform a ***+ve*** **X** move on the **machine Tool Model**.

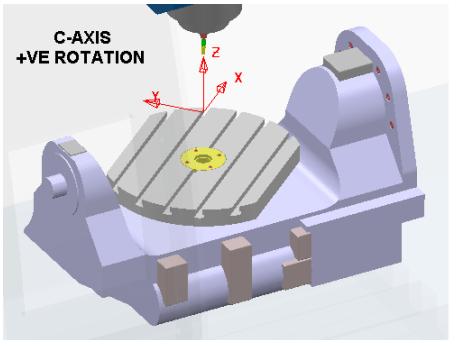
13 On the keyboard press the downward, **Arrow** key to move down to the next row (or simply click on the **Y** row, cursor).

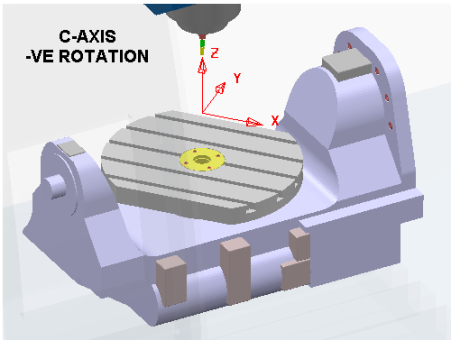
14 As in 8 and 9 above check the ***-ve*** and ***+ve*** **Y** moves on the **Machine Tool Model**.

15 Systematically move down the rows and check out the movement of the remaining 3 axes on the **Machine Tool Model**.





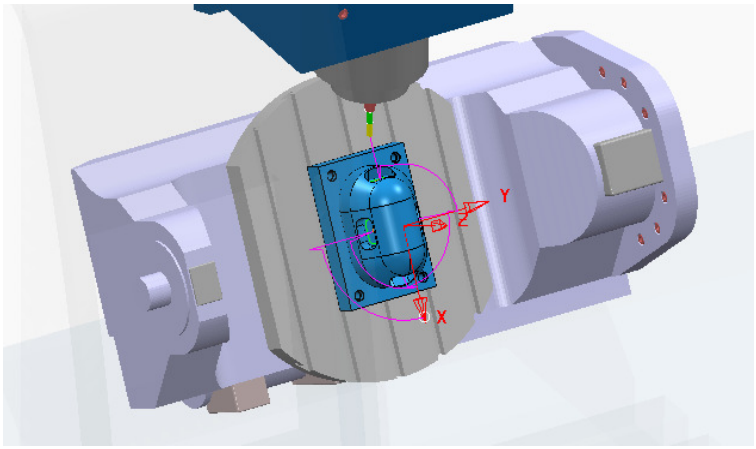




The above movements are exactly the same as on the actual **Mazak** machine tool to which the **MTD** file relates.

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).

16 Close the ***Machine Tool Position*** dialog and then perform a full **Simulation** on the toolpath **EM12-FinPkts**.



17 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:-

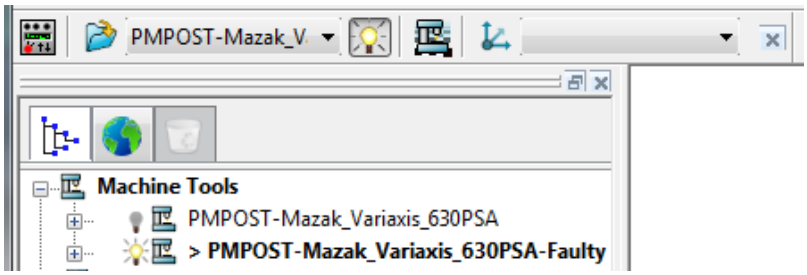
18 In the **Machine Tool** toolbar select the **Import Machine Tool Model** option.

19 In the dialog select the **Machine Tool Model** file:-

….\PowerMILL_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA-Faulty.mtd

20 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**.

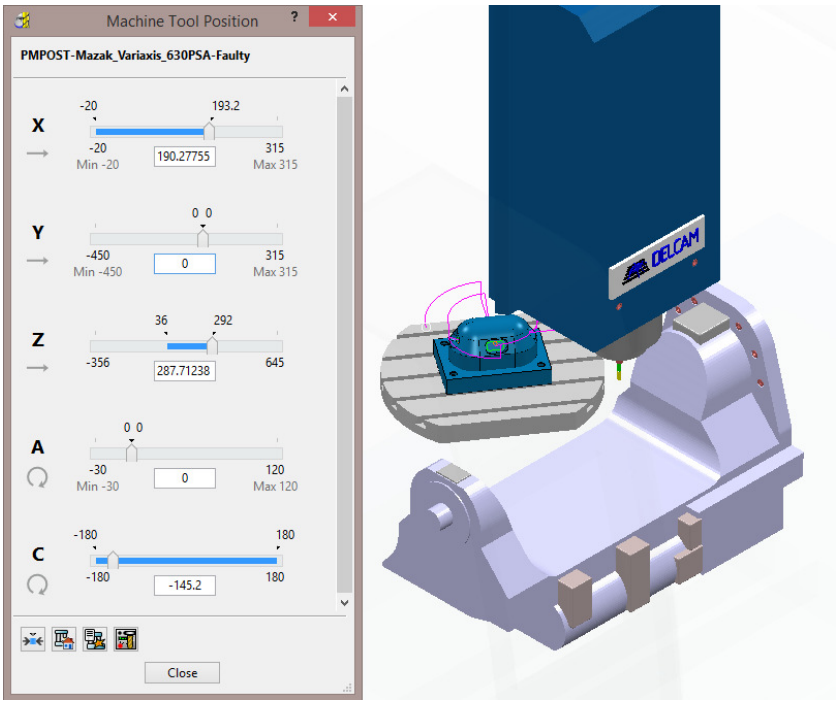
21 In the dialog select **Open** to import the new **Machine Tool Model**.



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.

22 In the **Machine Tool** toolbar select the **Machine Tool Display** option.

23 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**).



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 at some point along the active **Workplane**, **Y** direction.

24 From the **Desktop** open a **File Explorer** dialog and browse to the folder:-

….\PowerMILL_Data\five_axis\MachineToolModels

25 Create a copy of the **mtd** file:- **PMPOST-Mazak_Variaxis_630PSA-Faulty.mtd** and rename it as:- **PMPOST-Mbazak_Variaxis_630PSA-Update.mtd**

If an MTD file of the same name already exists overwrite it.

26 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" />
<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.

27 Modify the above incorrect values to those shown below:-

<simple_rotary X="0" Y="0" Z="0" I="0" J="0" K="1" />

28 **Save** the updated **mtd** file (but do not close it).

*This latest update will now be tested in the **PowerMILL Project**.*

29 In the **Machine Tool** toolbar select the **Import Machine Tool Model**  option.

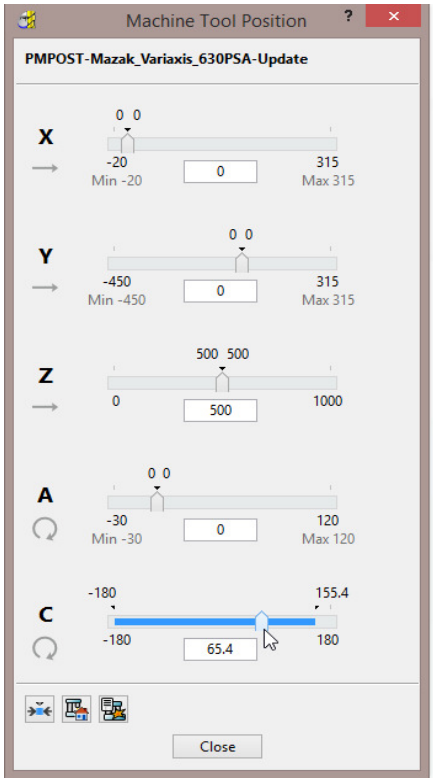
30 In the dialog select the **Machine Tool Model** file:-

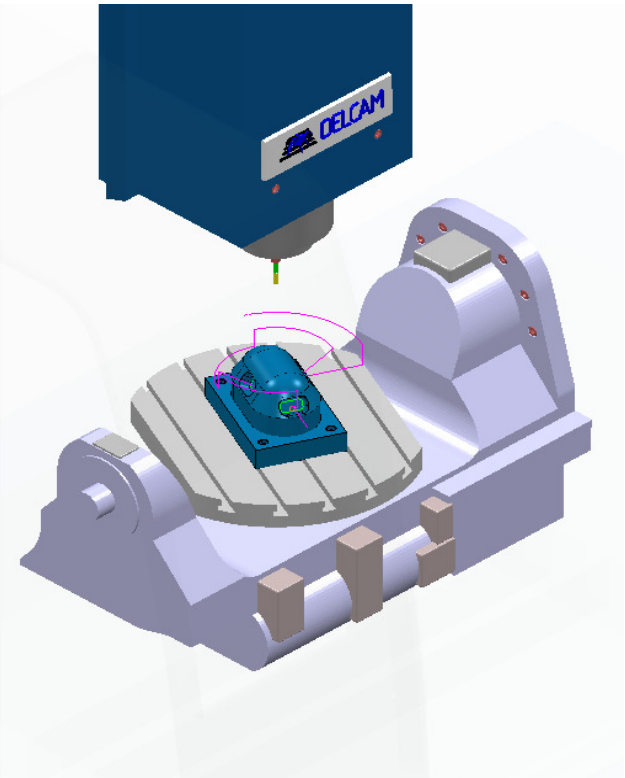
….\PowerMILL_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA-Update.mtd

31 Turn down the **Opacity** on the static parts of the **Machine Tool model**.

32 In the **Machine Tool** toolbar select the **Machine Tool Display**  option.

33 Use the **Jog** options to check the **+ve** and **–ve** movement of the **Rotary Table** – **C-Axis**.



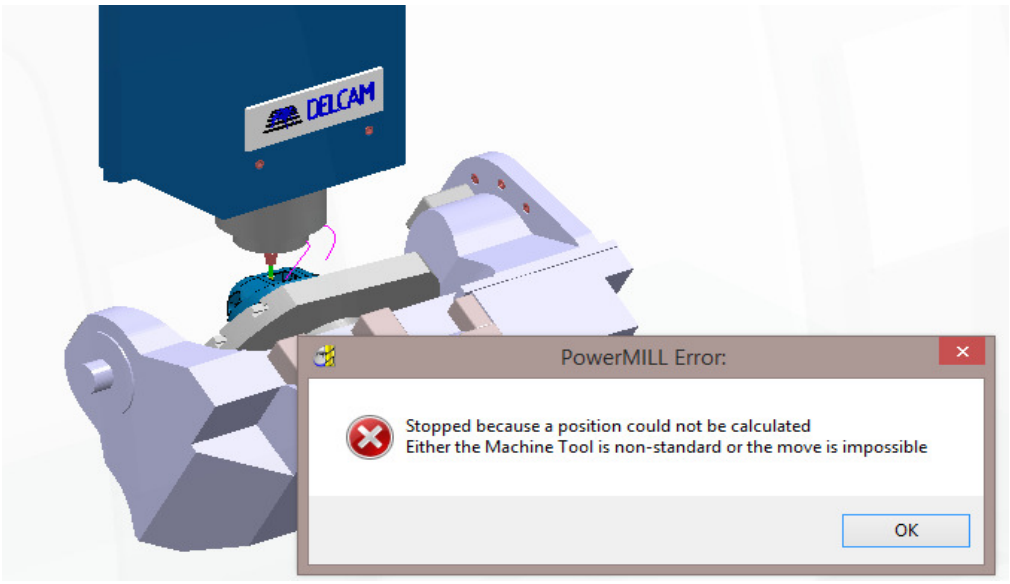


The updated **Rotary Table** – **C-Axis** movement is now exactly the same as in the original **Machine Tool Model** (***PMPOST-Mazak_Variaxis_630PSA***).

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.

34 Right mouse click on the toolpath **EM12-FinPkts** and from the local menu select **Simulate from Start**.

35 On the **Simulation** toolbar select the **Play**  button.



Not only has the ***Trunnion*** unexpectedly rotated towards the back of the machine but the **Simulation** has also stalled due to one of the travel limits being exceeded.

 *It cannot be assumed that these are due to the same error in the mtd file.*

36 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" />
```

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).

37 Modify the above incorrect values to those shown below:-

<control_info ADDRESS="A" MIN="-120" MAX="30" />

38 **Save** the updated **mtd** file (but do not close it).

*This latest update will now be tested in the **PowerMILL Project**.*

39 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**.*

40 In the **Machine Tool** toolbar select the **Import Machine Tool Model** option.

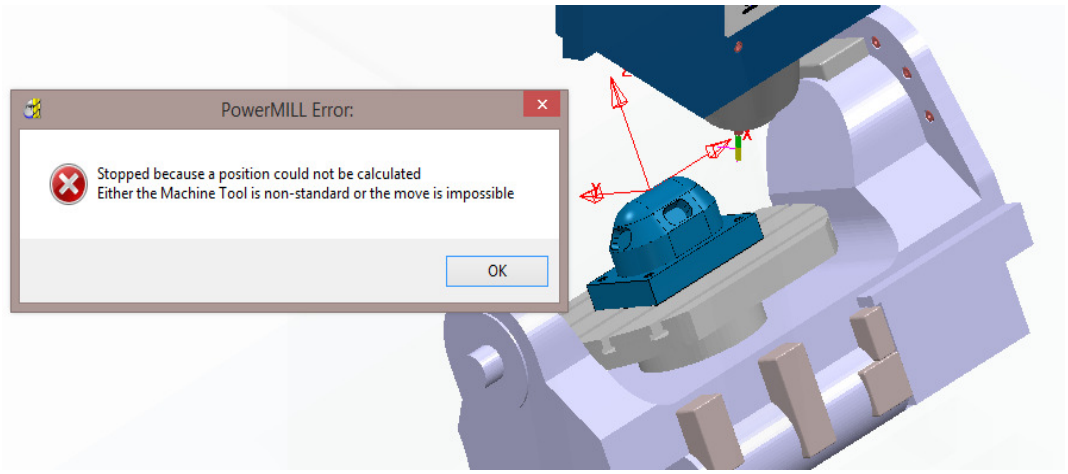
41 In the dialog select the **Machine Tool Model** file:-

….\PowerMILL_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA-Update.mtd

42 Turn down the **Opacity** on the static parts of the **Machine Tool model**.

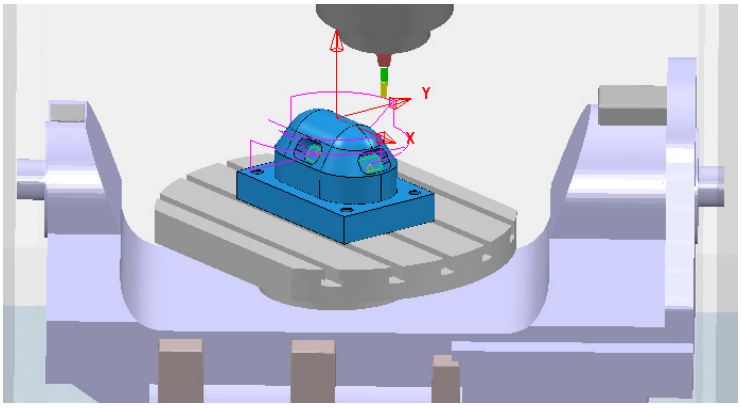
43 Right mouse click on the toolpath **EM12-FinPkts** and from the local menu select **Simulate from Start**.

44 On the **Simulation** toolbar select the **Play** button.



The **Simulation** immediately stalls and the **PowerMILL Error** dialog is displayed.

45 Select **OK** in the **PowerMILL Error** box and keep clicking on the **Step Forward** button and keep by-passing (**OK**) the warnings until they are no longer being displayed.



Note that the head is currently positioned along the **+ve Y** direction.

46 Continue to **Step Forward** and observe that all moves in the **–ve X** direction are stalling the **Simulation** while moves in **+ve X** and **-ve/+ve Y** are all okay.

47 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"**.

48 Modify the above incorrect value to that shown below:-

```
<control_info ADDRESS="X" MIN="-315" MAX="315" />
```

49 **Save** the file.

*This final update will now be tested in the **PowerMILL Project**.*

50 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**.*

51 In the **Machine Tool** toolbar select the **Import Machine Tool Model** option.

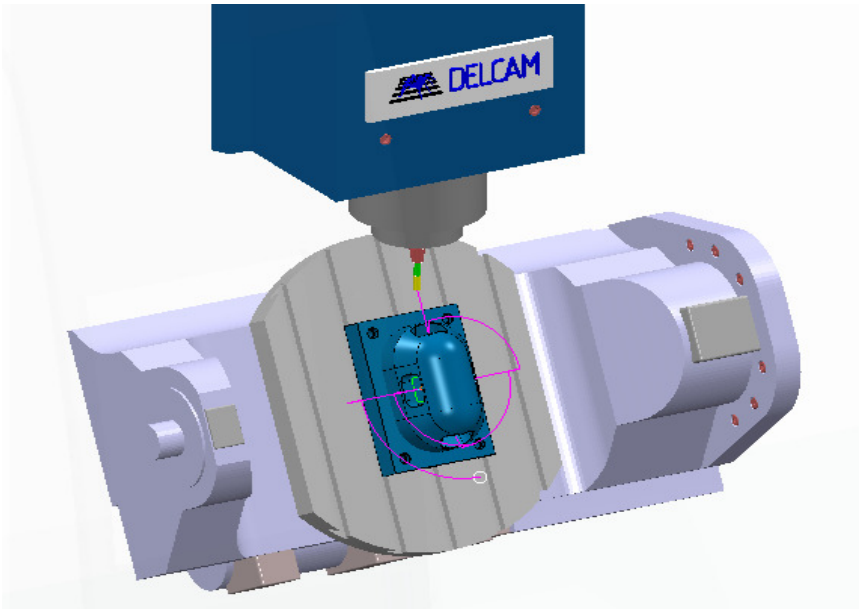
52 In the dialog select the **Machine Tool Model** file:-

….\PowerMILL_Data\five_axis\MachineToolModels\PMPOST-Mazak_Variaxis_630PSA-Update.mtd

53 Turn down the **Opacity** on the static parts of the **Machine Tool model**.

54 Right mouse click on the toolpath **EM12-FinPkts** and from the local menu select **Simulate from Start**.

55 On the **Simulation** toolbar select the **Play** button.



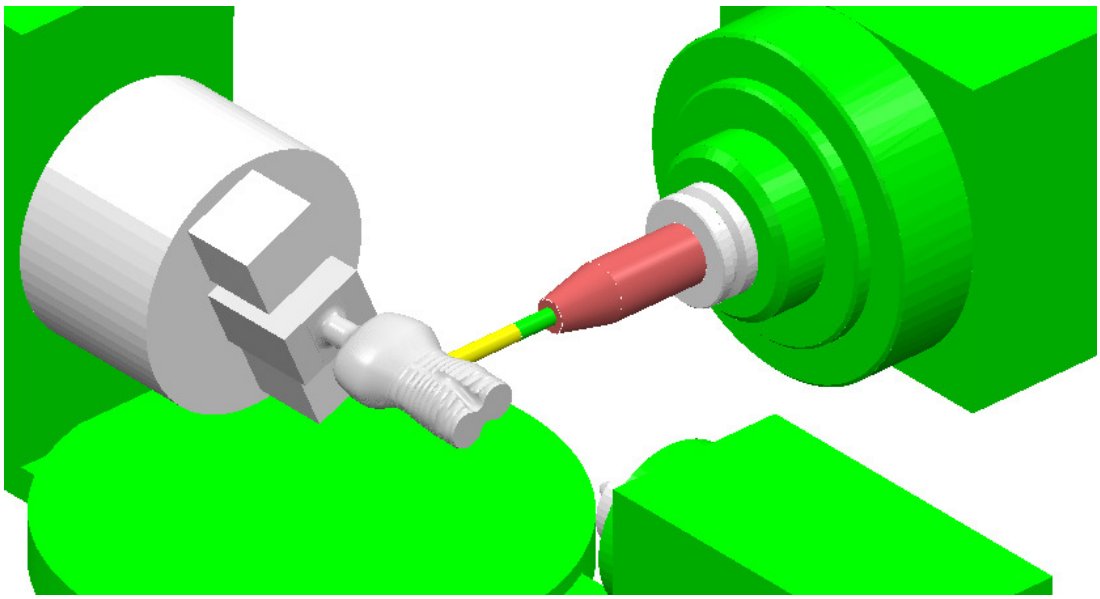
The **simulation** will now successfully run from start to finish without stalling. All linear and rotary moves should be an exact match comaperd with the original **mtd** file, **PMPOST-Mazak_Variaxis_630PSA**.

56 Select **File** - **Save Project**

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

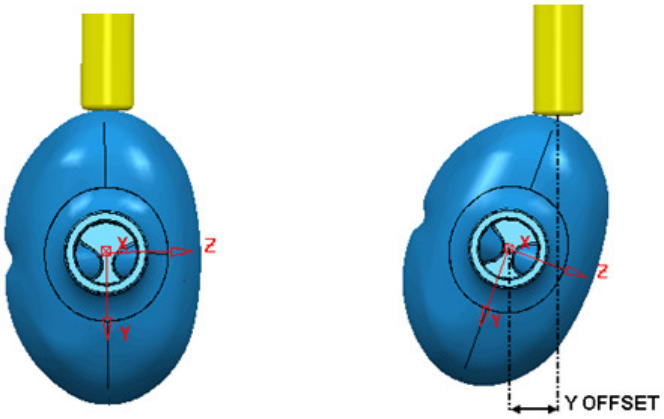
The **X Limits** define the absolute limits of the finishing path along the rotational, X-axis. These can be manually defined, or automatically set to the limits of the block.

Pattern

This enables the cutting method to be specified for rotary milling either **Circular**, **Line**, or **Spiral**.

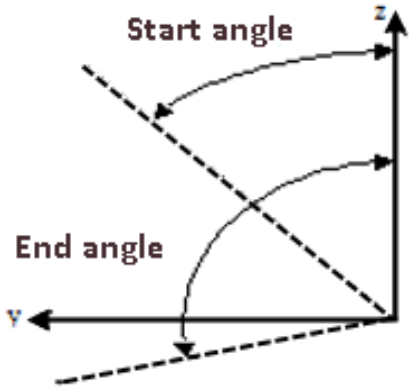
Y Offset

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.

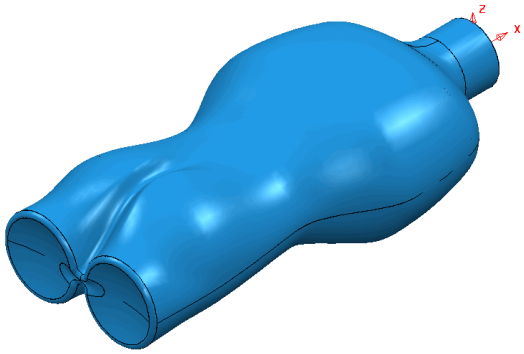
Circular Rotary Machining

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**.

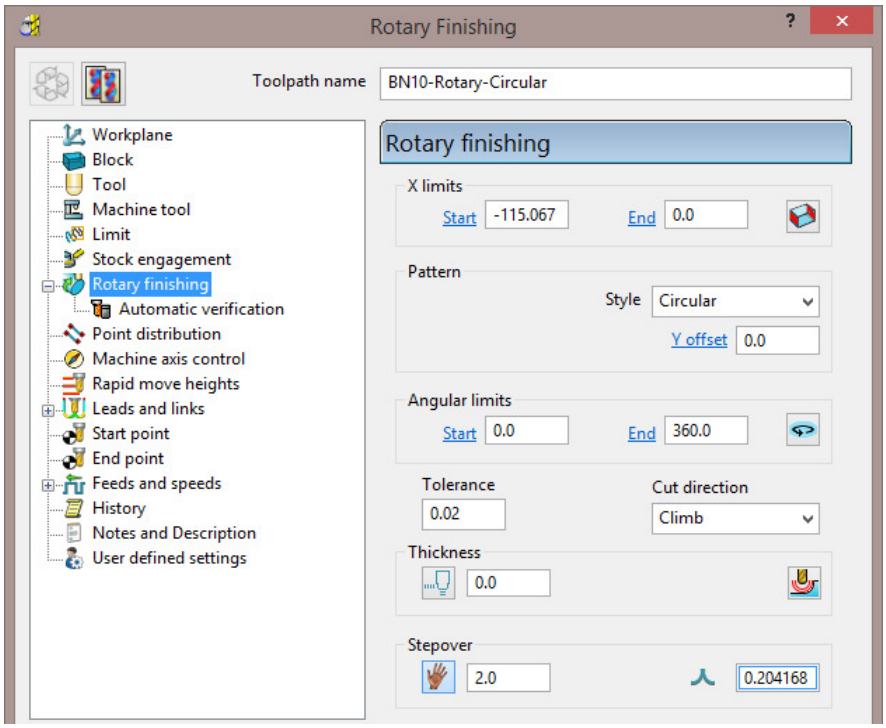
4 Create a Dia 10 Ball nosed tool named **BN10**.

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**.



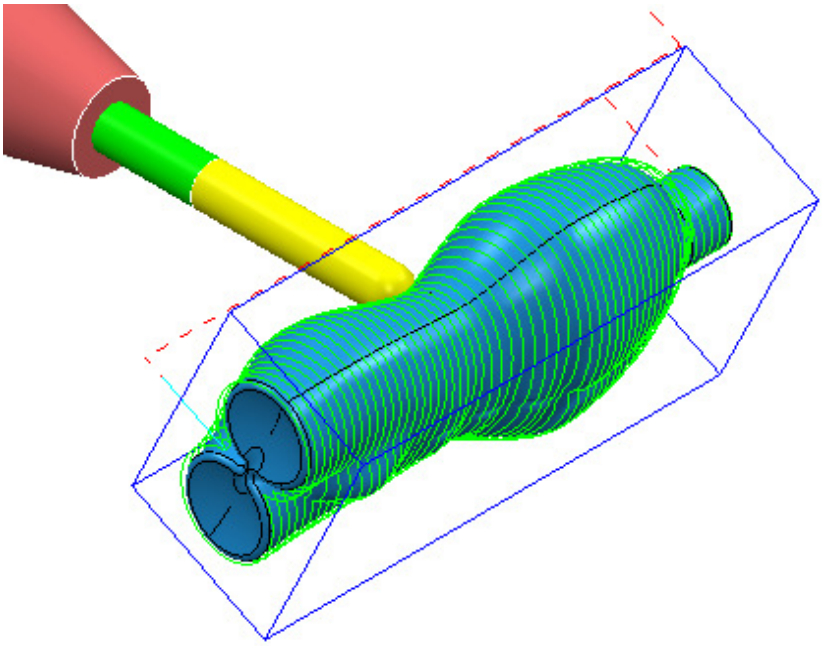
- Enter ***Name***: **BN10-Rotary-Circular**
- Click on **Defines X limits as block limits**. 
- Set ***Style***: **Circular**
- Select ***Cut Direction***: **Climb**
- Input ***Stepover***: **2**

9 Select **Calculate** to create the toolpath then **Cancel** the dialog.

10 In the **PowerMILL *explorer*** right mouse click the toolpath **Rotary1_BN10** and select **Simulate from Start** to open the **Simulation** Toolbar.

11 On the **Simulation** toolbar select the **Tool View** icon  .

12 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.

13 **Recycle** the toolpath, select **Any** for the **Cut direction** and select **Calculate** to create the toolpath before selecting **Cancel** to close the dialog.

14 **Simulate** the **toolpath** as before to observe the effect of the toolpath reversing direction with every new section machined.

Line Rotary Machining

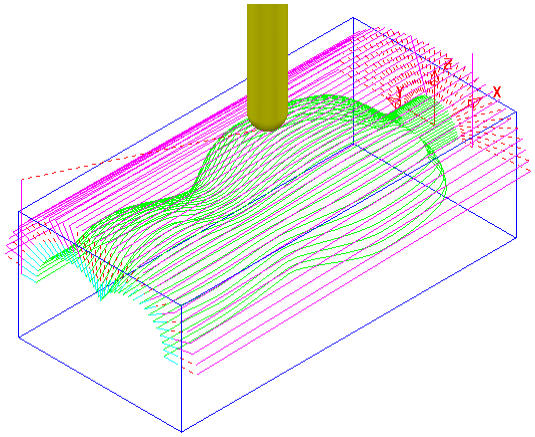
Using the ***Style*** as **Line**, the tool feeds in the X direction following the component form. At the end of each pass the tool retracts and moves above the start of the next pass. At the same time the rotary axis indexes by the angular stepover and the tool then leads onto the next machining move.

15 **Make a Copy** of the previous toolpath named **BN10-Rotary-Line** and select **Style** - **Line** using **Cut Direction** > **Climb milling**.

16 For **Angular Limits** - **Start Angle**, enter **90** and **End Angle**, enter **-90**.

17 In the **Leads and Links** dialog set the **Z Heights** - **Skim distance** to **20** and all of the **Links** to **Skim**.

18 Click on **Calculate** and **Close** the dialog.



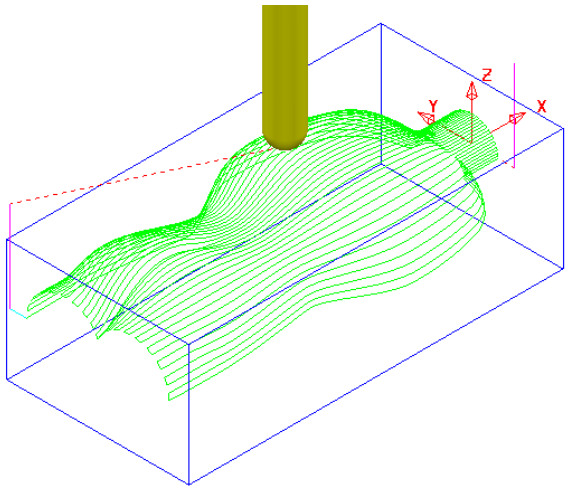
19 **Simulate** the new, uni-directional toolpath (shown above with a larger step over).

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.

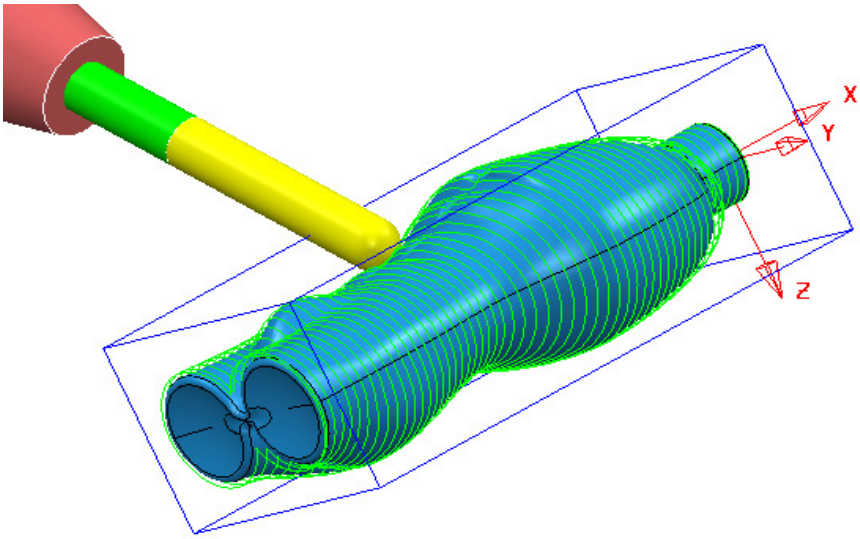


Spiral Rotary Machining

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.

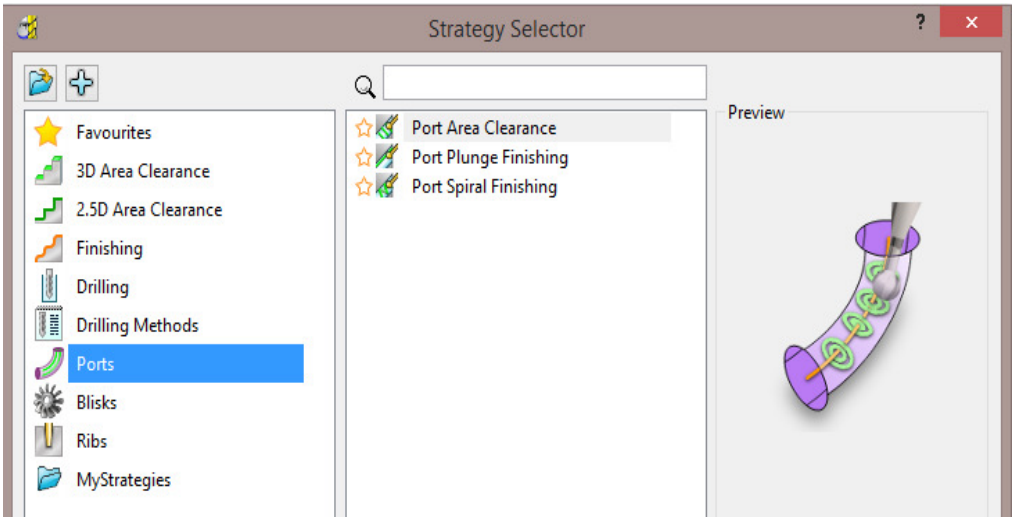
26 **Save** the **Project** as:-

...\COURSEWORK\PowerMILL_Projects\ Rotary_Bottle-EX1

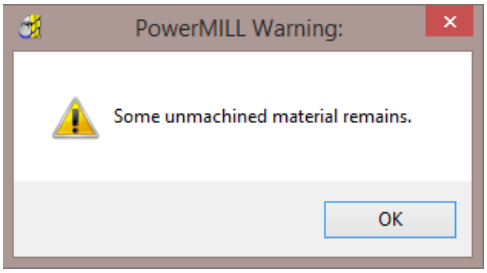
15. Port Machining

Introduction

In the **Strategy Selector** the **Ports** option provides 3 specialist strategies, designed for machining cylinder head, inlet/outlet ports.



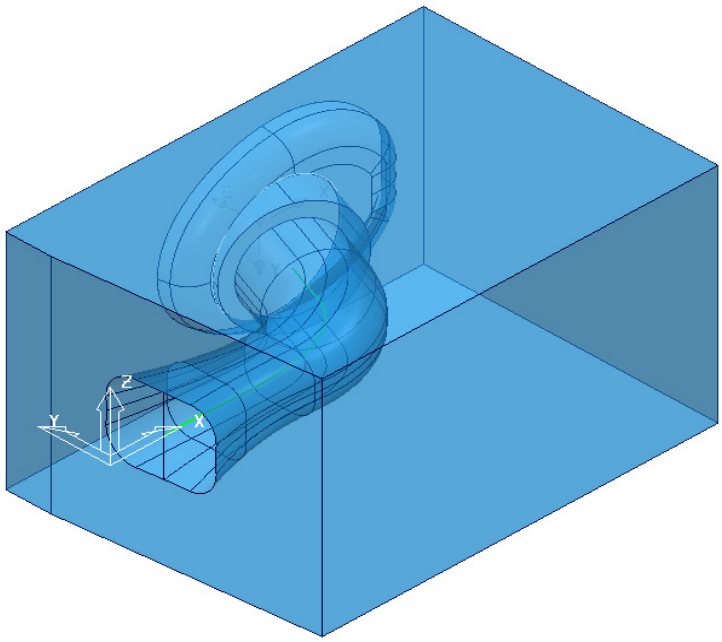
The **Port Machining** options automatically provide multi-axis, **Tool Alignments** where this is possible. A ***warning*** will be displayed if impossible alignment limits are reached and the port is not fully machined.



Basic Single Port Example

1 Open the 'read-only' **Project**:

...\PowerMILL_Data\five_axis\Ports\SinglePort2-Start



2 **Save Project As**:-

...\COURSEWORK\PowerMILL-Projects\SinglePort2-ex1

*The **Project** already contains all required cutting **Tools**, and preliminary **3Plus2 Toolpaths**.*

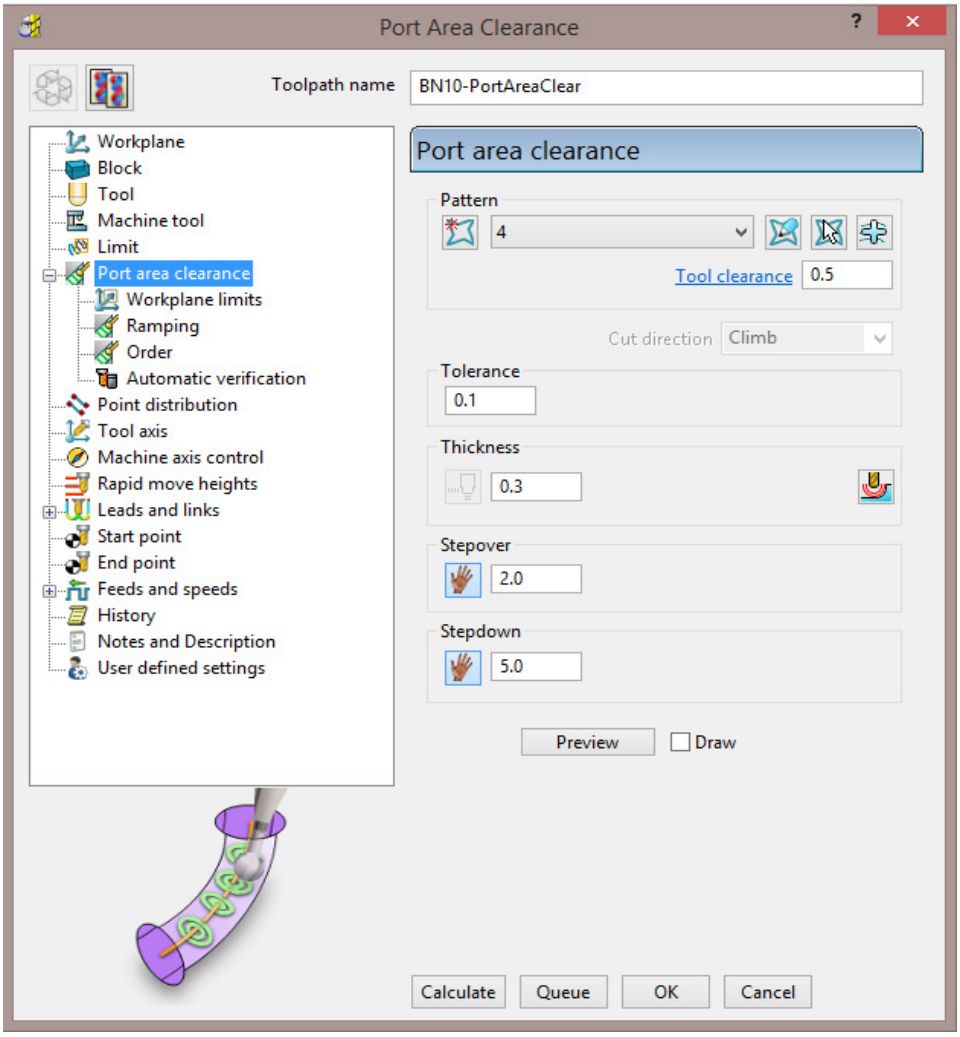
The following section will take the user through the specialist **Port Area Clearance** and **Port Spiral Finishing** options.

A **Pattern** (**4**) is included running down the centre of the **Port**. This is required to control the extent and alignment of the **Port Machining** strategies.

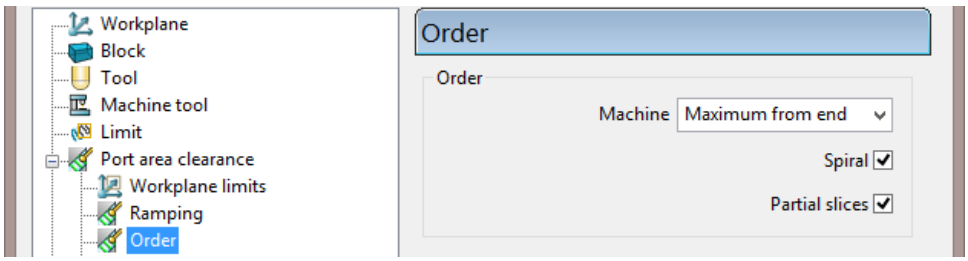
3 **Activate** the existing **Toolpath** 1 to reinstate all the settings.

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.

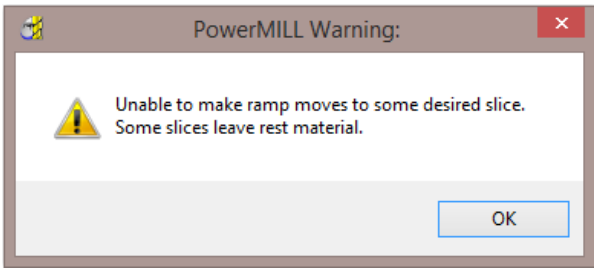


- Insert ***Pattern***: **4**
- Input ***Tool Clearance***: **1.0**



- Tick **Spiral**.
- Tick **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.

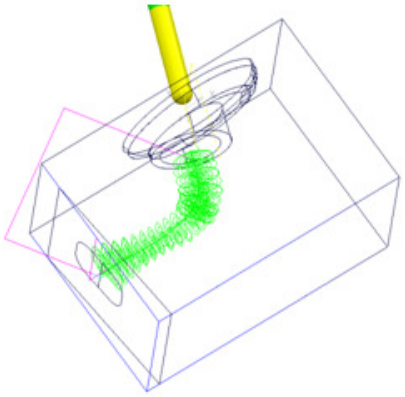
9 **Activate *Toolpath* 1**

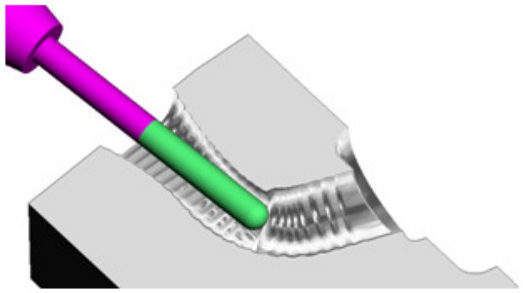
10 Right mouse click on **toolpath 1** and from the local menu select **Simulate from start**.

11 Open the **Block** dialog and change the **Max Z value** from **40** to **0** before selecting **Accept**.

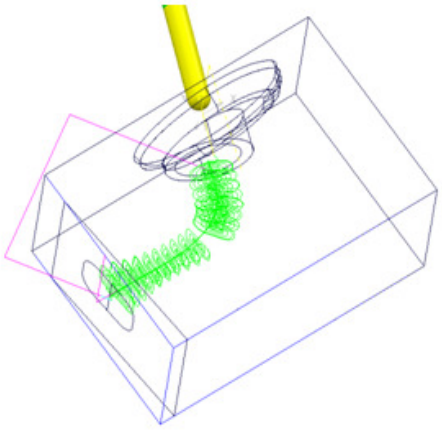
*By defining the **Block** to be half the height of the **model** it will be possible to visualise the lower half of the port being machined.*

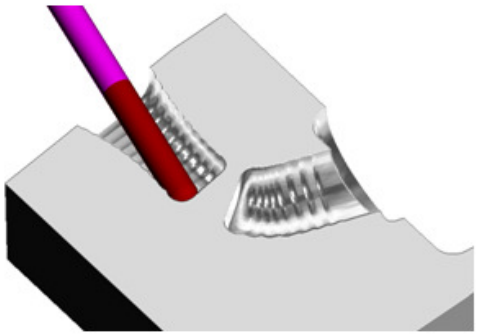
12 Select an **ISO 2** view and run a **ViewMILL simulation** on all the remaining toolpaths in sequential order.





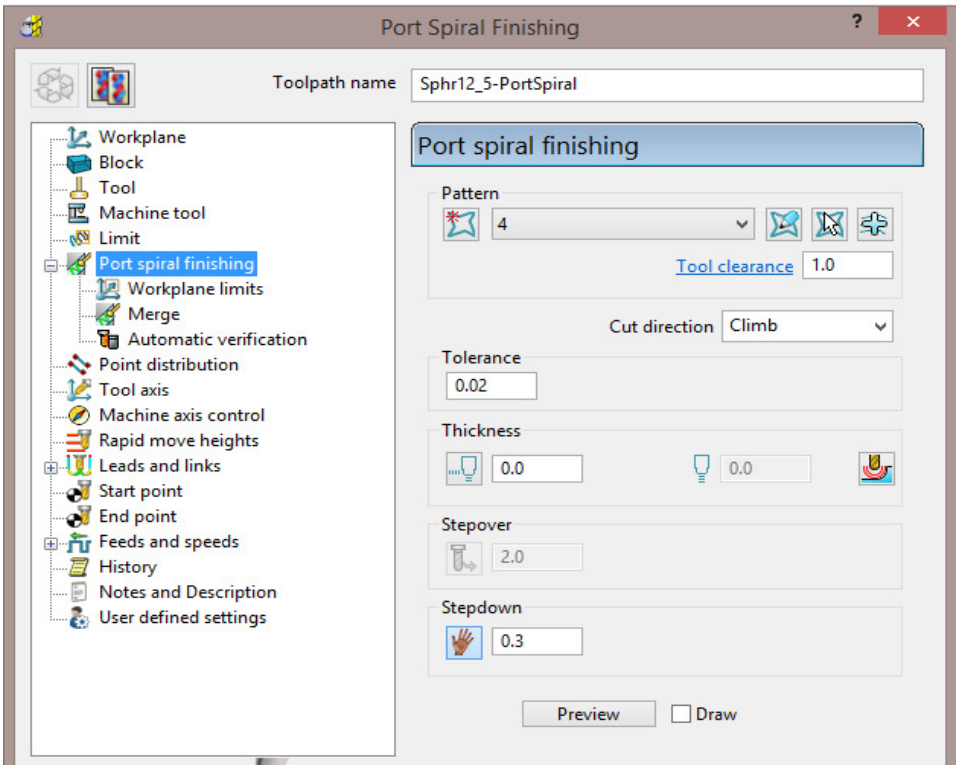
*If the **Partial Slices** option had not been ticked then the **Toolpath** would have been created as shown below with a wall of material remaining in the middle of the port.*

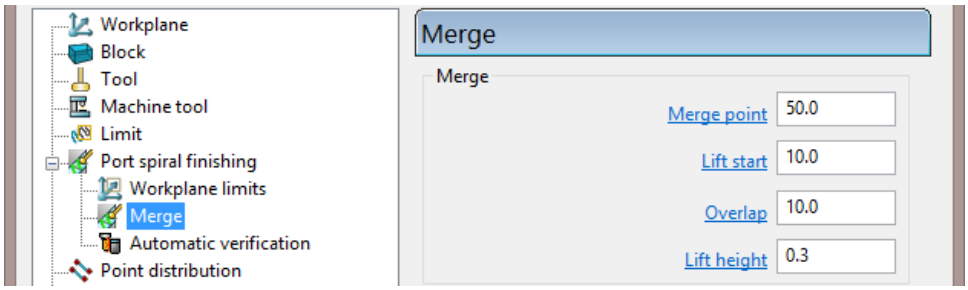




13 **Activate** the **SPHR12.5** tool.

14 In **Toolpath Strategies** select **Ports** > **Port Spiral Finishing** and fill in the two featured pages of the dialog exactly as shown below.

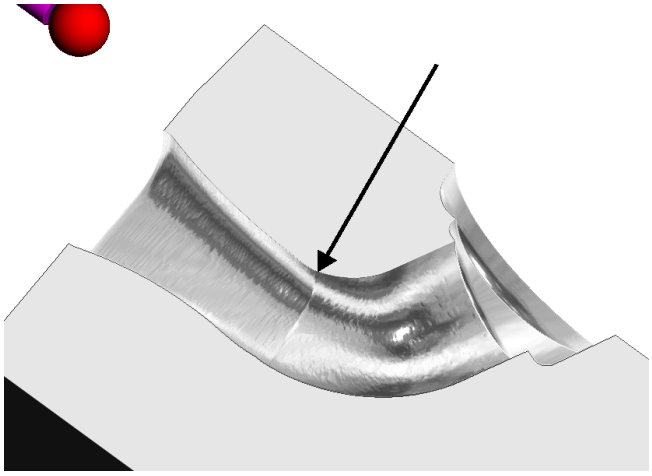




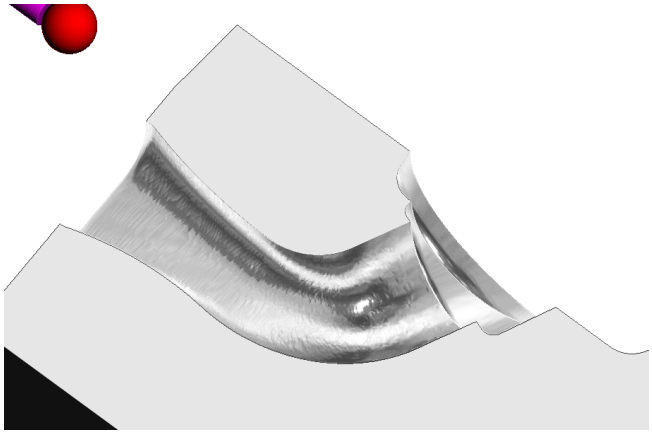
The above **Merge** settings causes the **thickness** to gradually increase to **0.3** as it merges into the areas previously machined from the first end. This is to compensate for the cutter tending to dig in when the loading decreases.

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**.

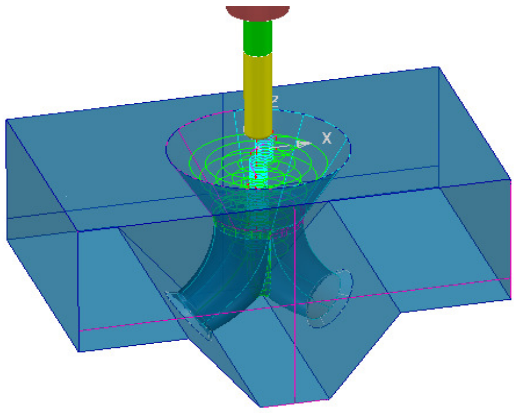
Twin Port Example

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**:

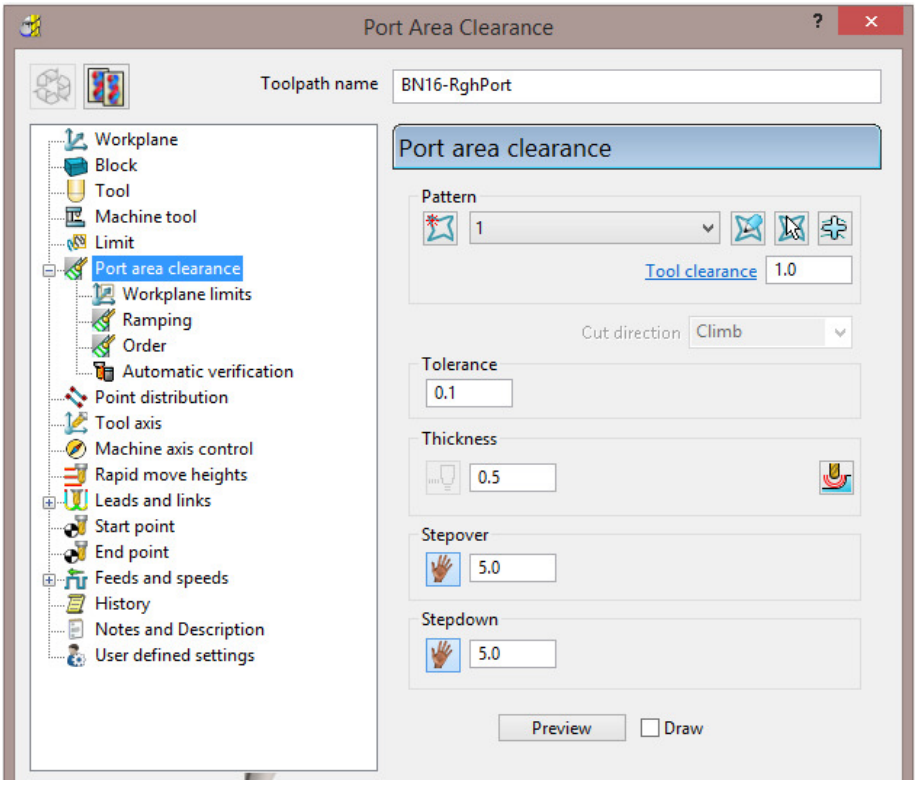
...\COURSEWORK\PowerMILL-Projects\DualPort-ex1

4 **Activate** the existing toolpath **RghPort-3Axis** to reinstate all the settings.

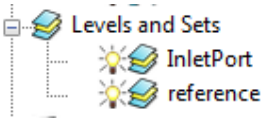
5 **Activate** the **BN16** tool.

6 **Calculate** a new **Block** to the full **Model** dimensions.

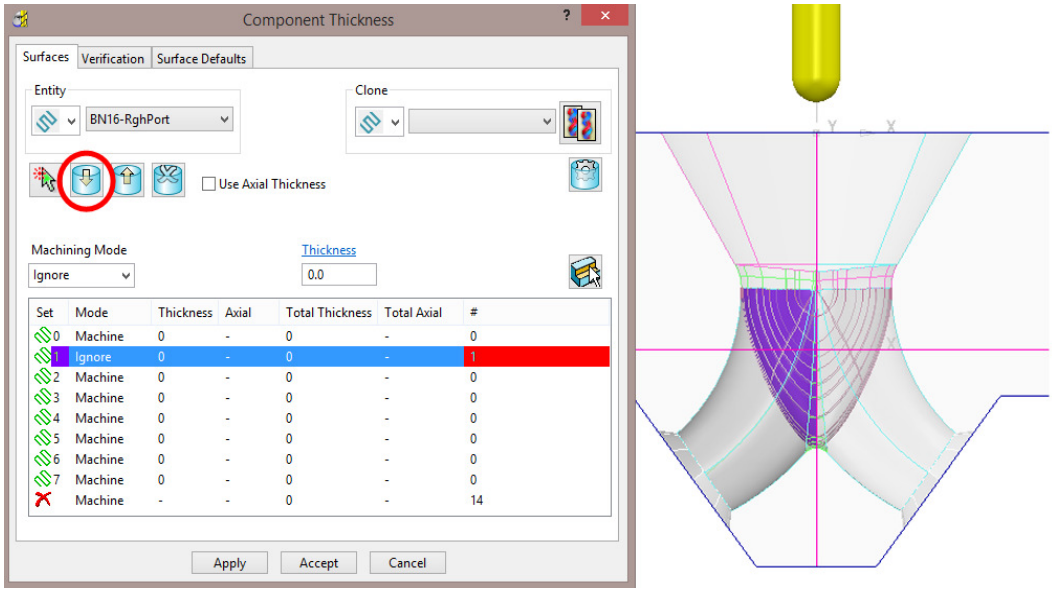
7 In **Toolpath Strategies** select **Ports** > **Port Area Clearance** and fill in the pages of the dialog exactly as shown below.



8 Switch on the **Level** named **reference**.

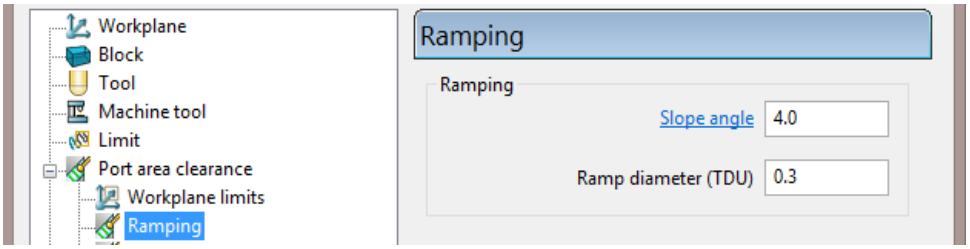


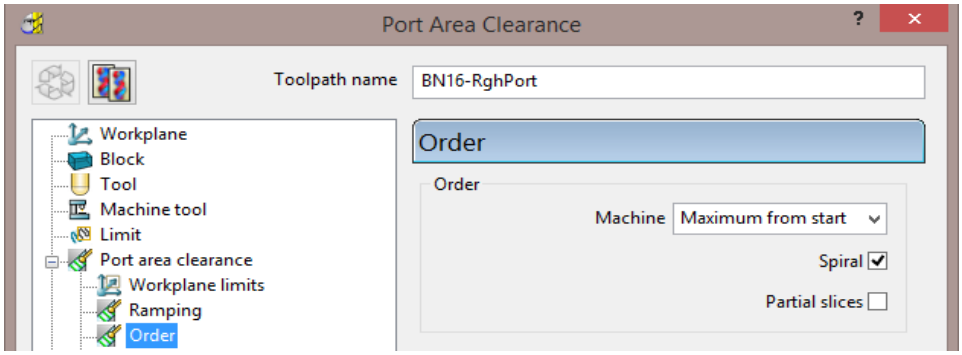
9 From the **Port Area Clearance** main page, select the **Component Thickness** icon  .



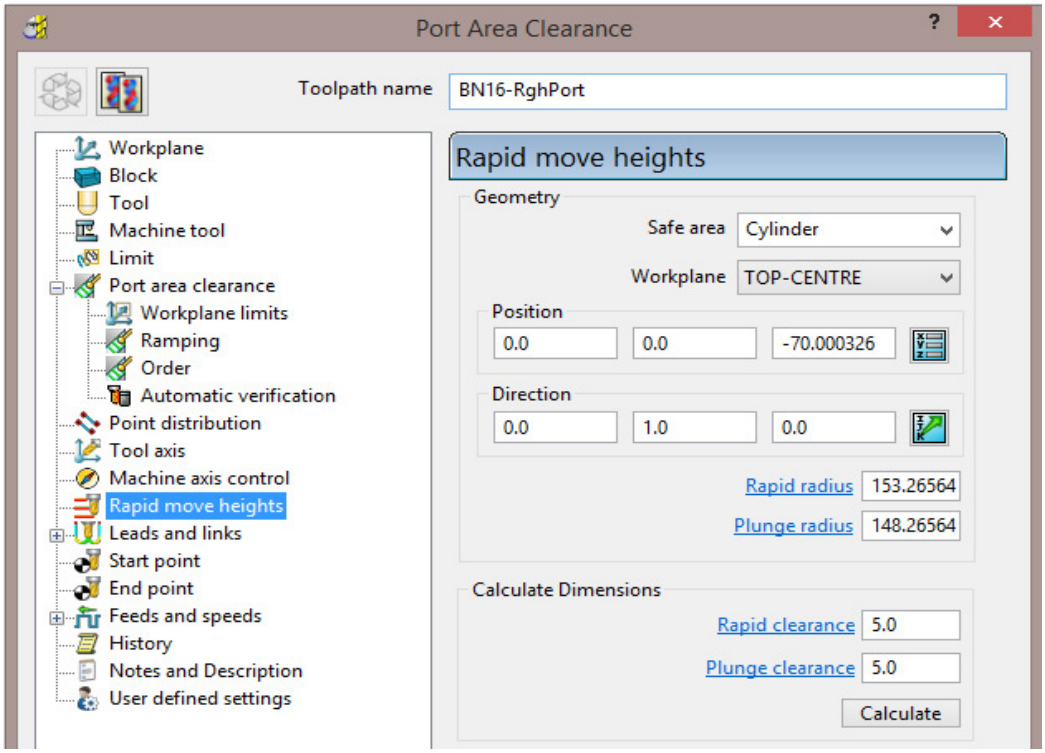
10 In the **Component Thickness** dialog, select and acquire the left hand ***reference surface*** (Shaded purple above right) to a row with the **Machining Mode** set to **Ignore**.

11 **Apply** the dialog.

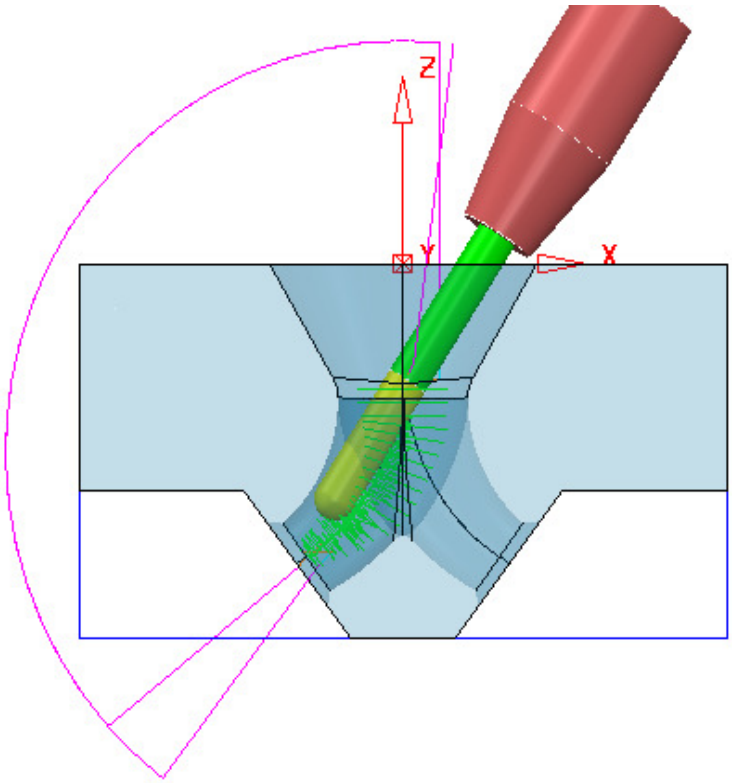




12 In the ***Order*** page tick **Spiral**.

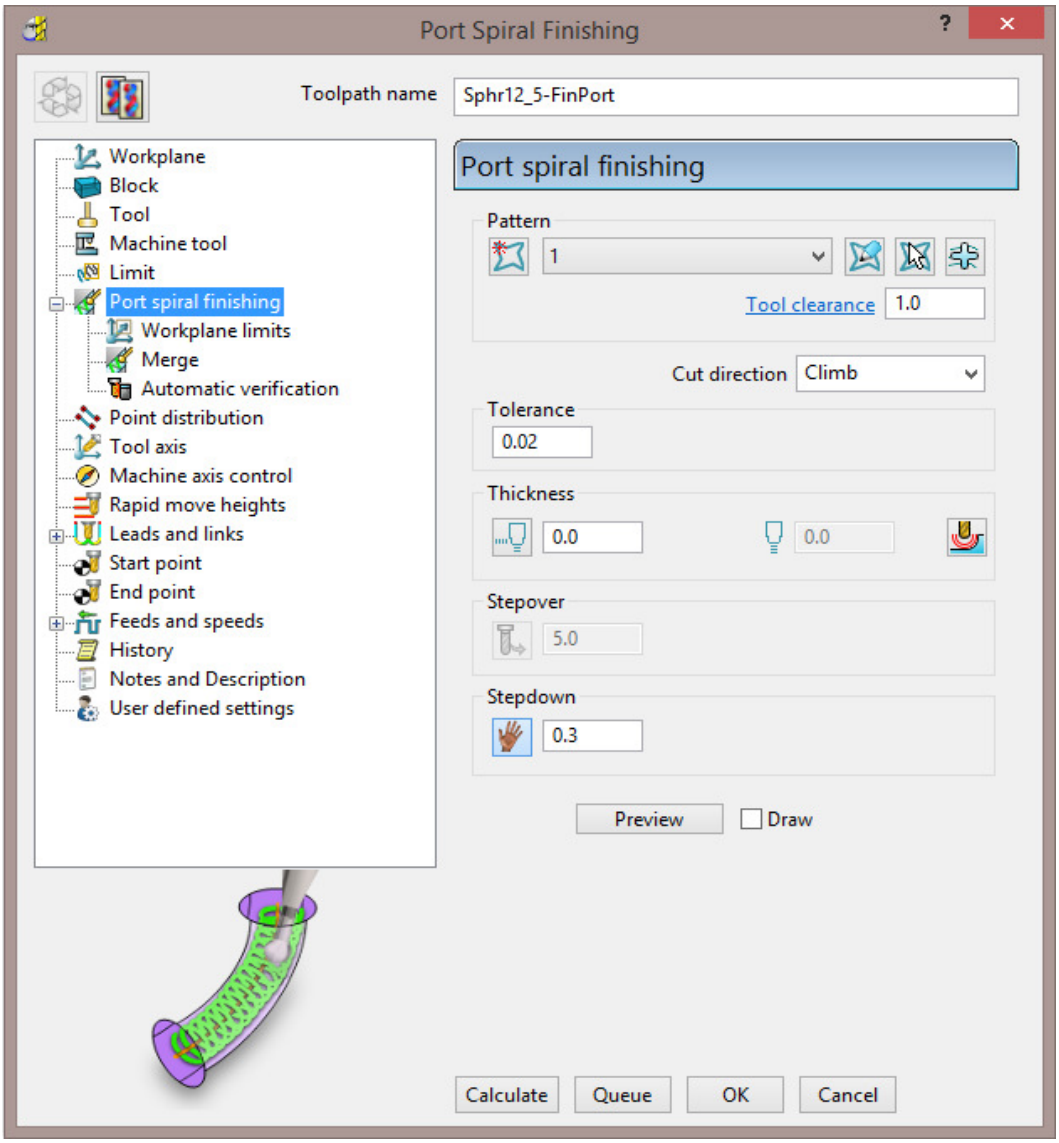


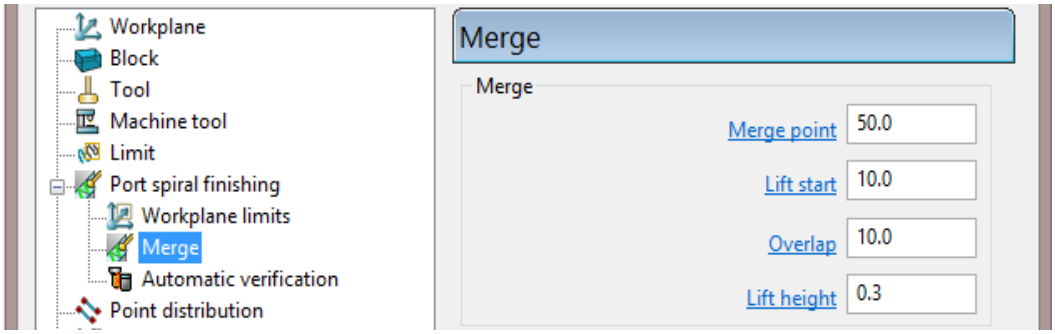
13 **Calculate** the **Port area clearance** dialog.



14 **Activate** the **SPHR25** tool.

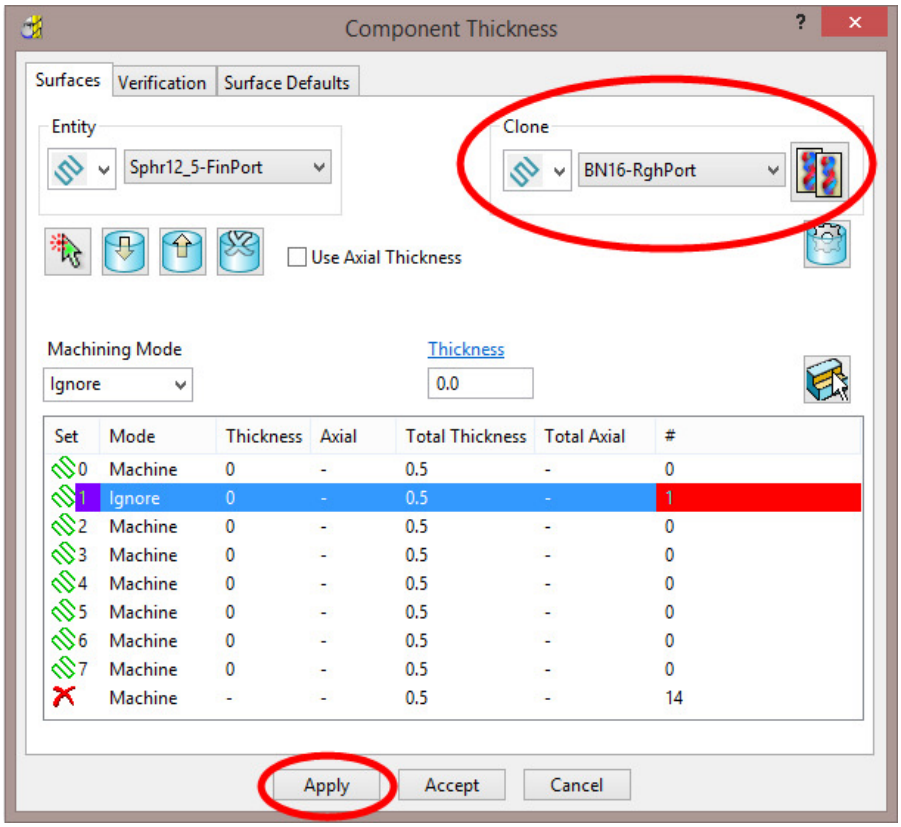
15 In **Toolpath Strategies** select **Ports** > **Port Spiral Finishing** and fill in the pages of the dialog exactly as shown below.



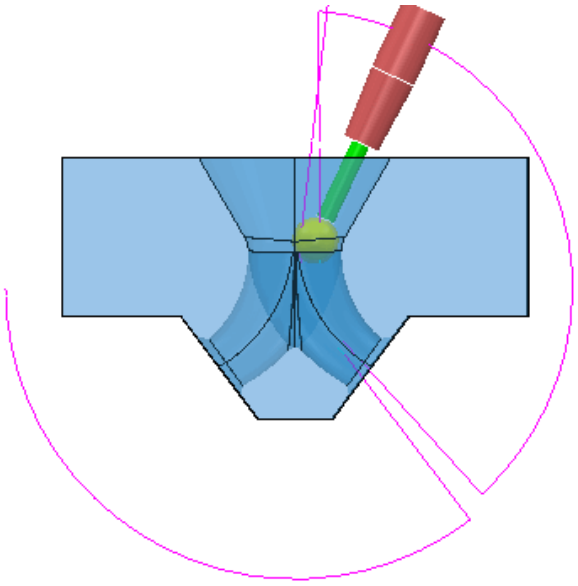


16 From the **Port Area Clearance** main page, select the **Component Thickness** icon  .

17 As shown in the following diagram, ***Clone***  the **Component Thickness** settings from the previously created toolpath **BN16-RghPort**.



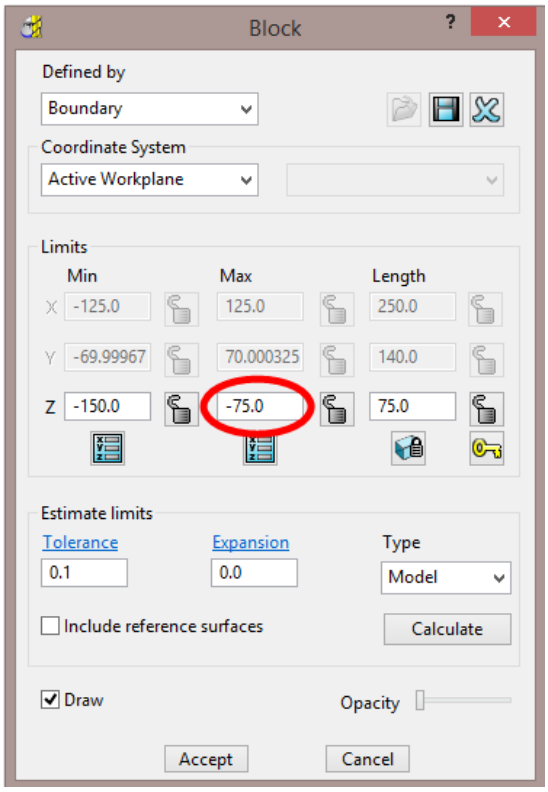
18 **Apply** and **Accept** the **Component Thickness** dialog, and then **Calculate** the **Port spiral finishing** strategy.



19 **Activate** the **Workplane** ***mc-OUTPUT***.

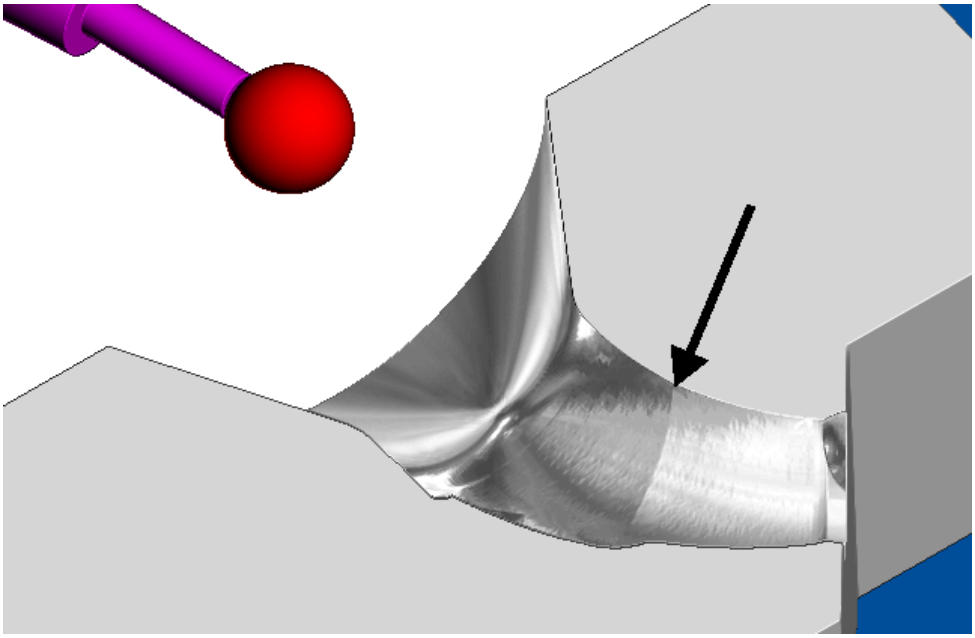
20 **Activate** the **Boundary 1**.

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**.



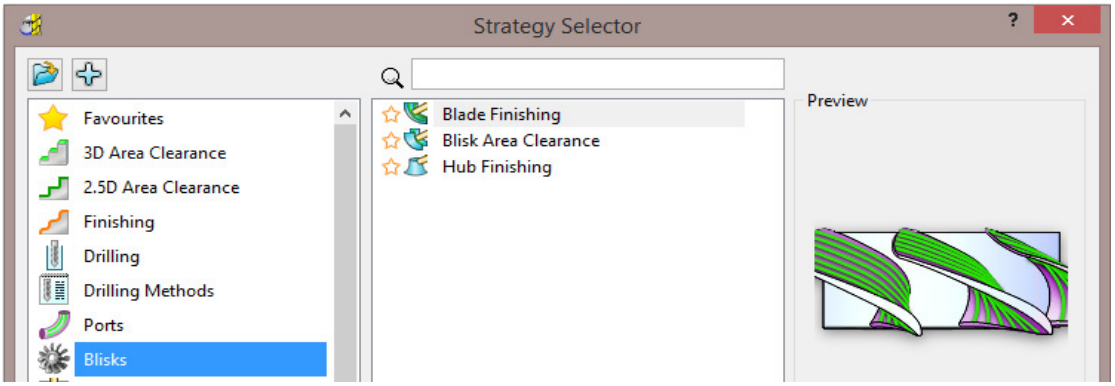
*Note the **0.3mm step** where the machining from the second end of the Port has gradually moved away from the form over the last **10mm** to the blend point.*

24 Select **File** > **Save Project**.

16. Blisks

Introduction

In new strategies, the **Blisks** option provides 3 specialist strategies for machining Impellers.

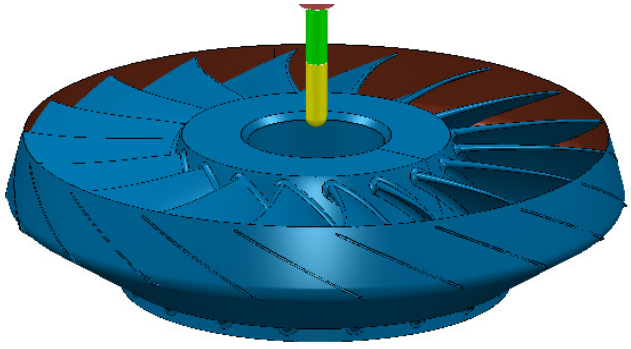


Impeller Example

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**.

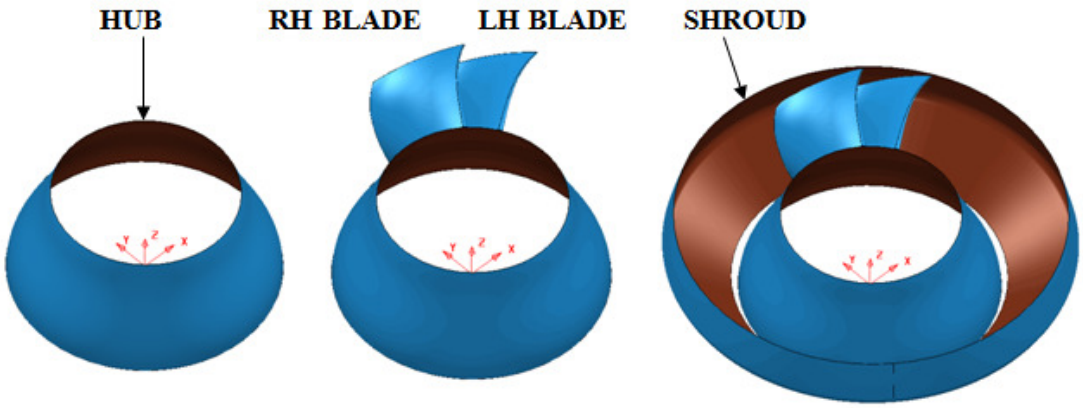
5 Create a ***Cylindrical*** **Block** to the **Model** dimensions.

6 **Calculate** the **Rapid Move Heights**.

7 Set both **Start** and **End Points** to **First/Last Point Safe**.

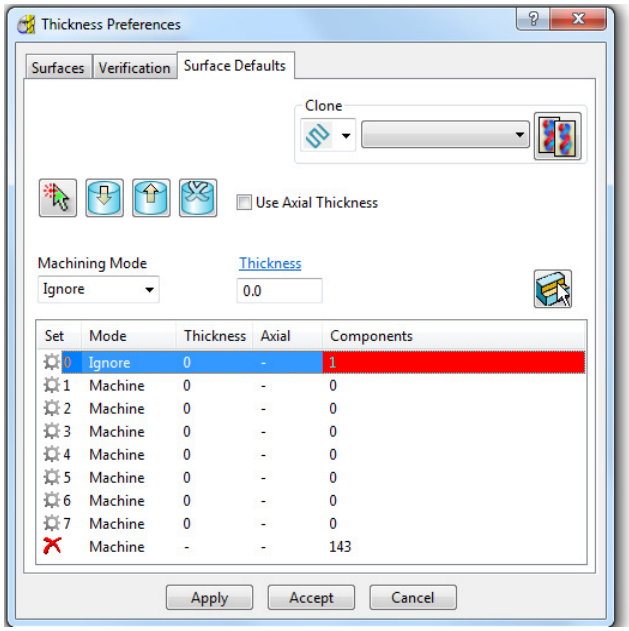
8 In the **Leads and Links** dialog, set **Links** to ***Short/Long* Skim** and ***Default* Incremental**.

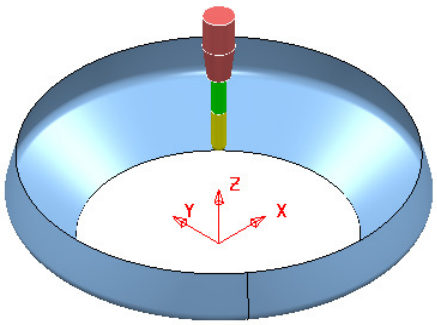
To be able to apply the **Blisks** machining options the component surfaces must first be acquired to a series of specifically named **levels**. In this example the **surfaces** in the imported model are already acquired to appropriate **levels**.



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.

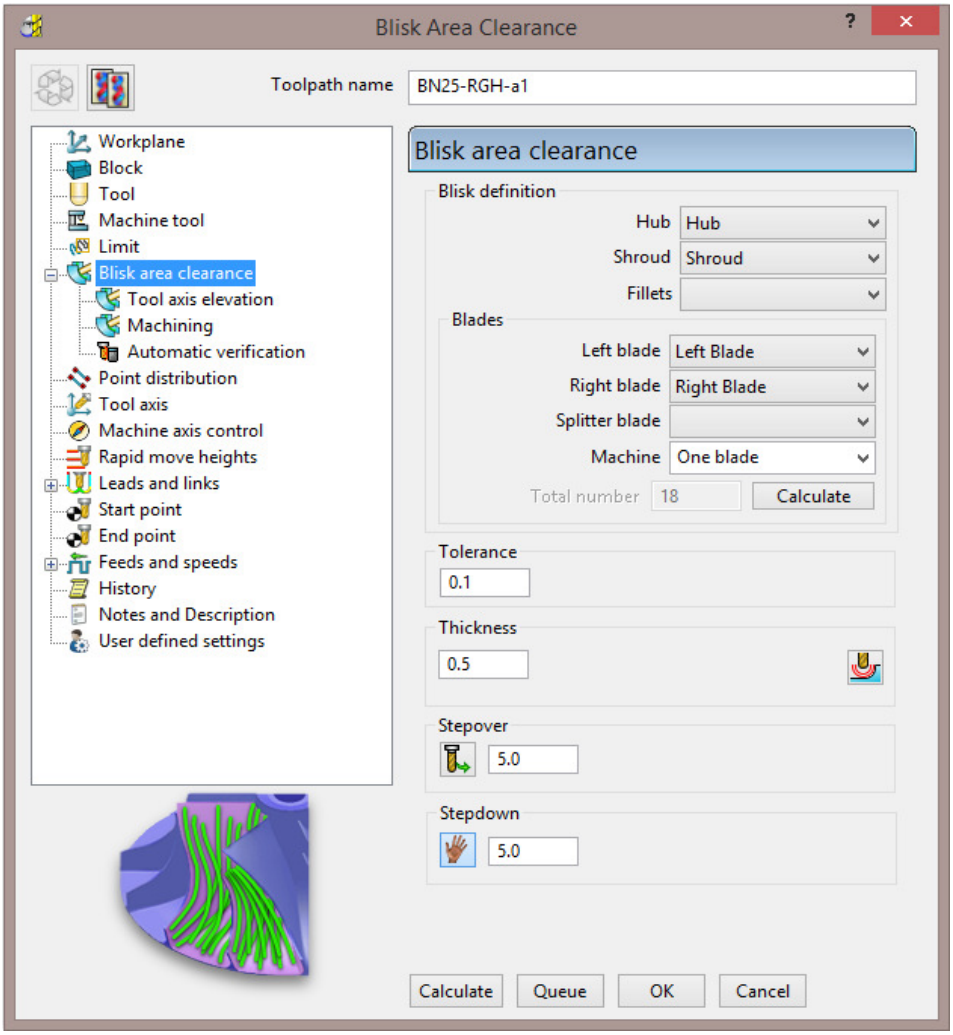
12 Set the first row to ***Machining Mode* - Ignore** and **Apply** the settings, before closing the dialog by clicking **Accept**.

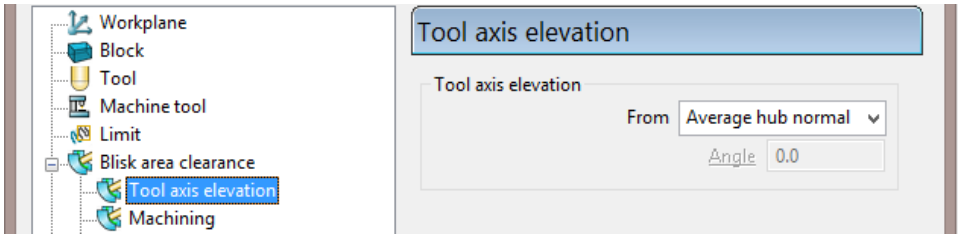
13 In the **Leads and Links** options input **Z heights** - **Skim distance 15**.

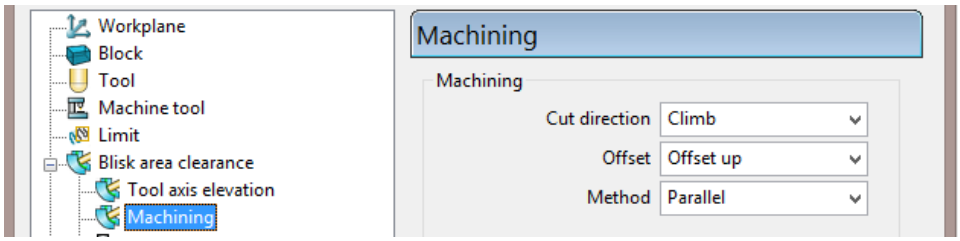
14 For both **Lead in** and **Lead out**, input ***1st choice*** – **Extended move 10** and ***2nd choice*** – **Surface normal arc** – **Angle 90** and **Radius 5**.

15 In the **Toolpath Strategies** dialog, select **Blisks** and from the three available options, select **Blisk Area Clearance**.

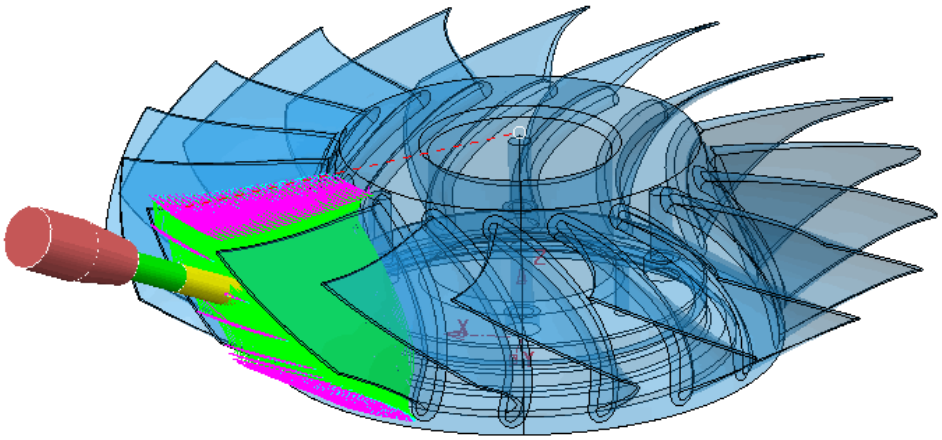
16 Enter data exactly as shown below with the **Fillets** and **Splitter Blade** options left blank.







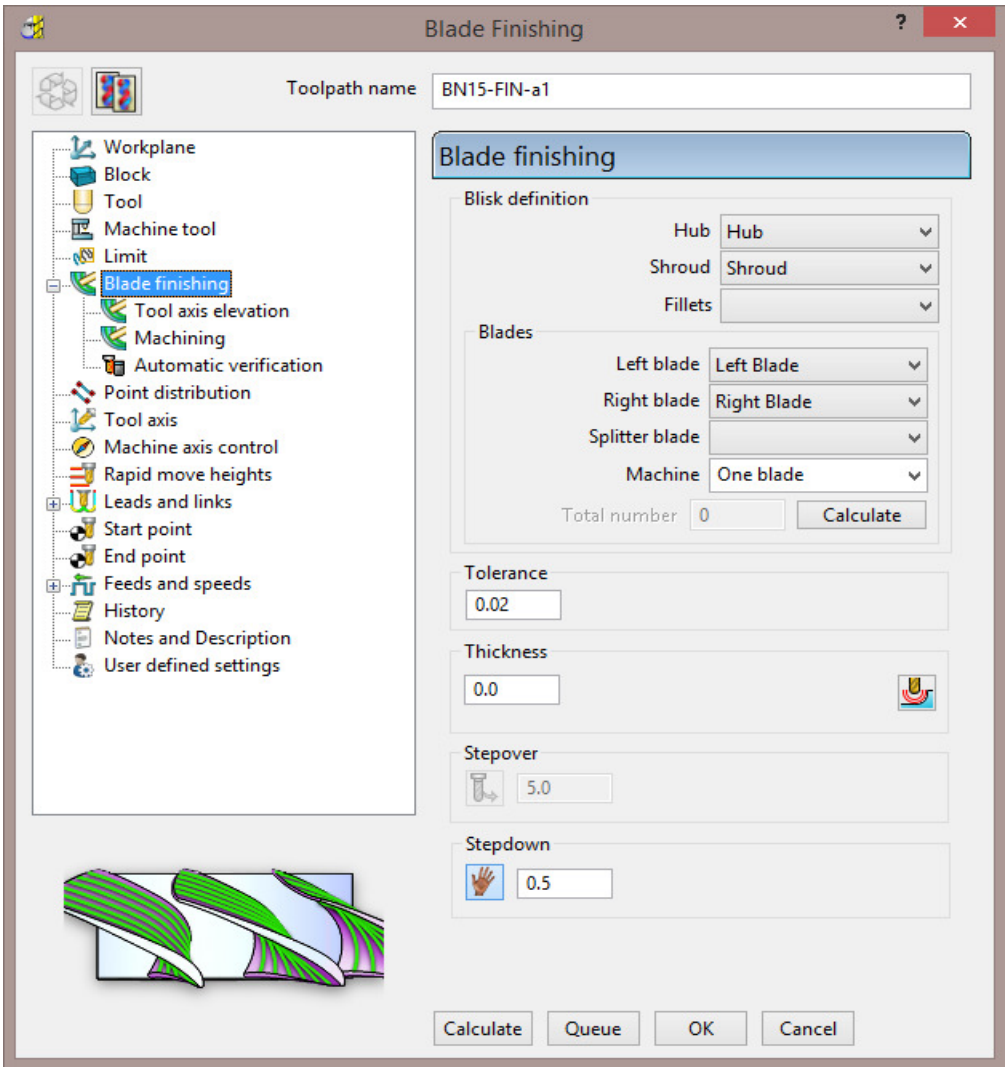
17 Select **Queue** in the dialog to commence background toolpath processing.

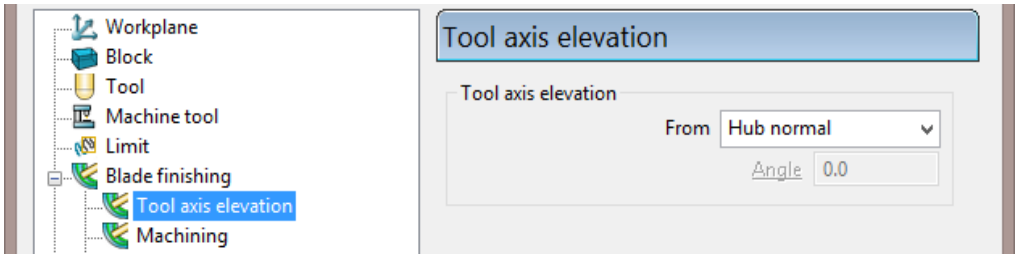


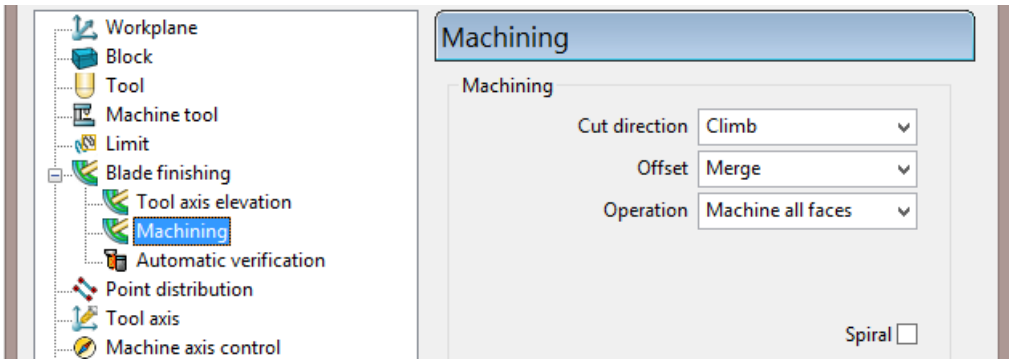
18 **Activate** the existing **Ball Nosed** tool named **BN15**.

19 In the **Toolpath Strategies** dialog, select **Blisks** and from the three available options, select **Blade Finishing**.

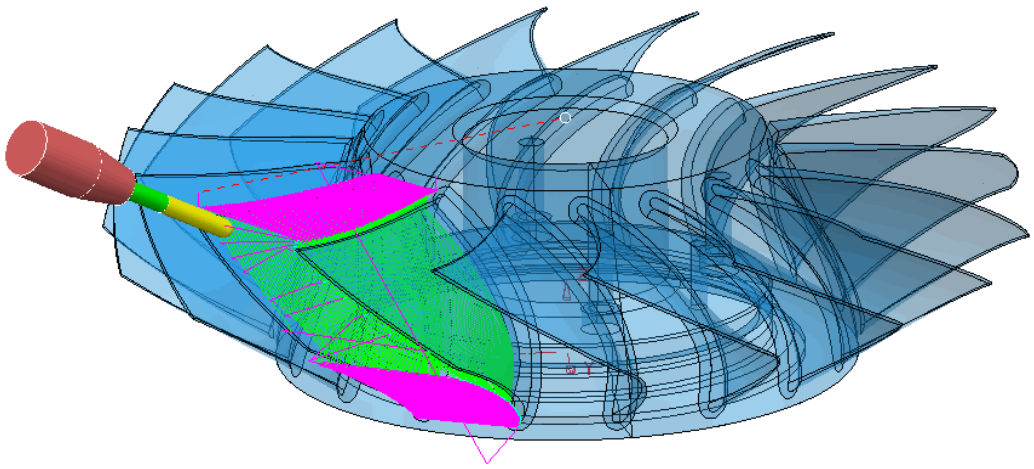
20 Enter data into the pages of the dialog exactly as shown below with the **Fillets** and **Splitter Blade** options left blank.





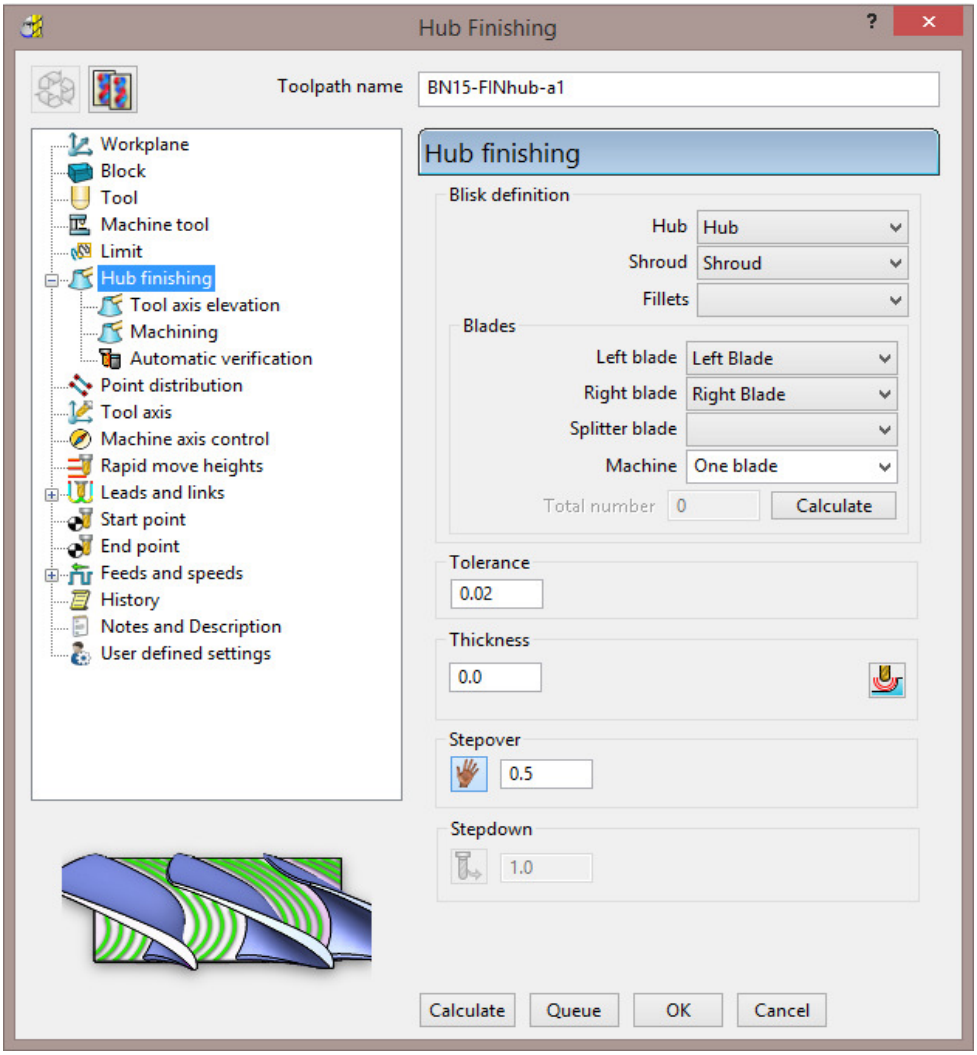


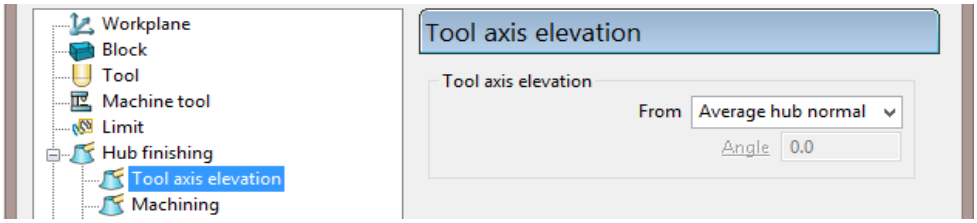
21 Select **Queue** in the dialog to commence background toolpath processing.

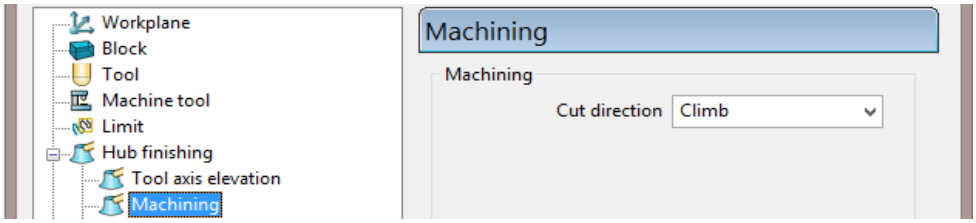


22 In the **Toolpath Strategies** dialog, select **Blisks** and from the three available options, select **Hub Finishing**.

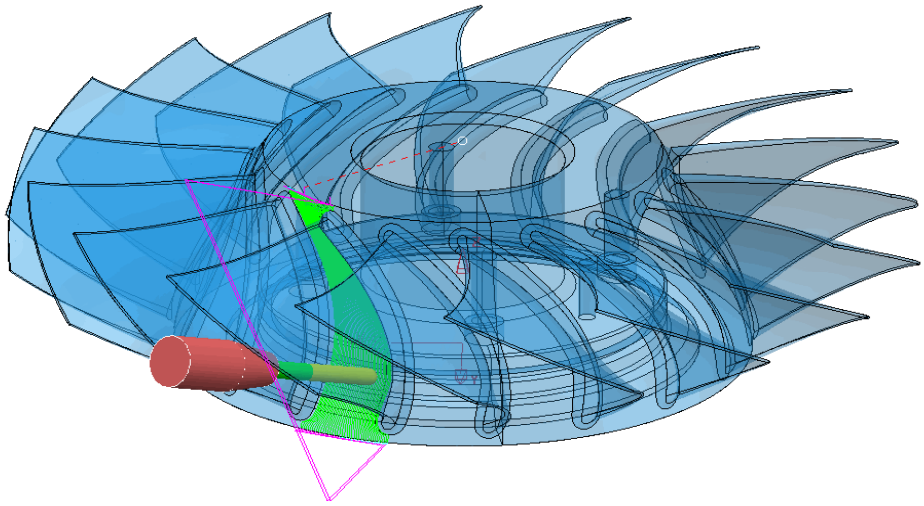
23 Enter data exactly as shown below with the **Fillets** and **Splitter Blade** options left blank.







24 Select **Queue** in the form to commence background toolpath processing.

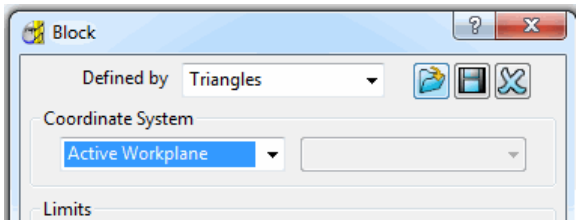


The next stage is to perform a full **ViewMILL simulation** of all 3 toolpaths.

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.

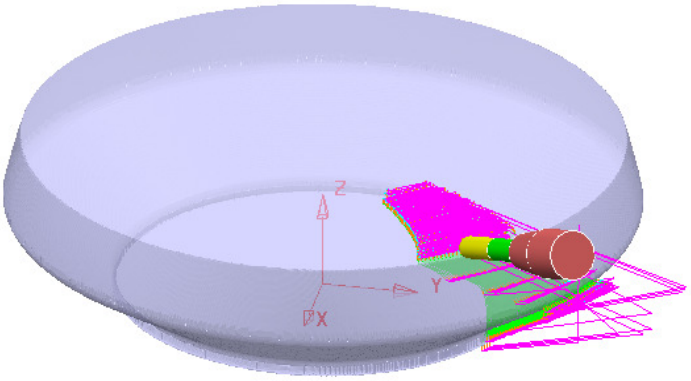
25 **Activate** and ***right mouse*** click on the toolpath **BN25-RGH-a1** and from the local menu select the option **Simulate from Start**.

26 Before starting the **Simulation**, open the **Block** dialog and select **Defined By** > **Triangles** followed by select the **Load block from file** icon.

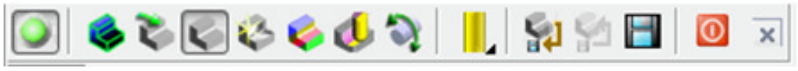


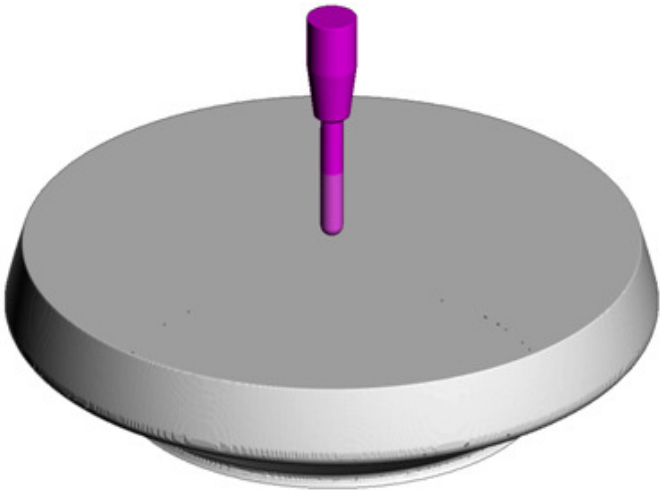
27 Browse and select the **Triangle** file:

...\PowerMILL_Data\FiveAxis\Blisks\TurnedShroud.dmt

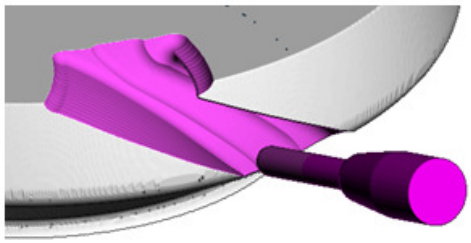


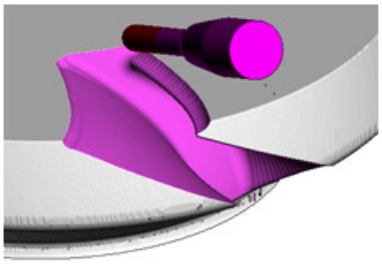
28 With the complex **Block** active, switch on a **ViewMILL** simulation.



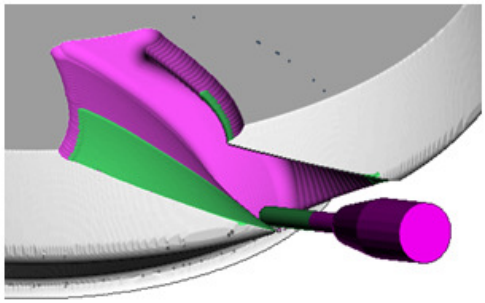


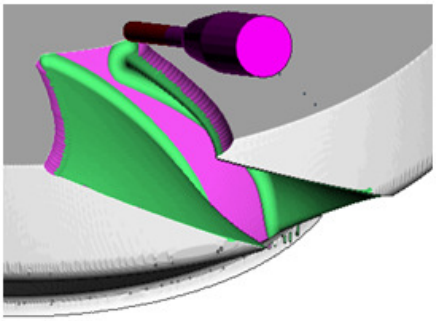
29 Select the **Play** button in the toolpath **Simulation** toolbar and observe the machining process for the toolpath **BN25-RGH-a1**.



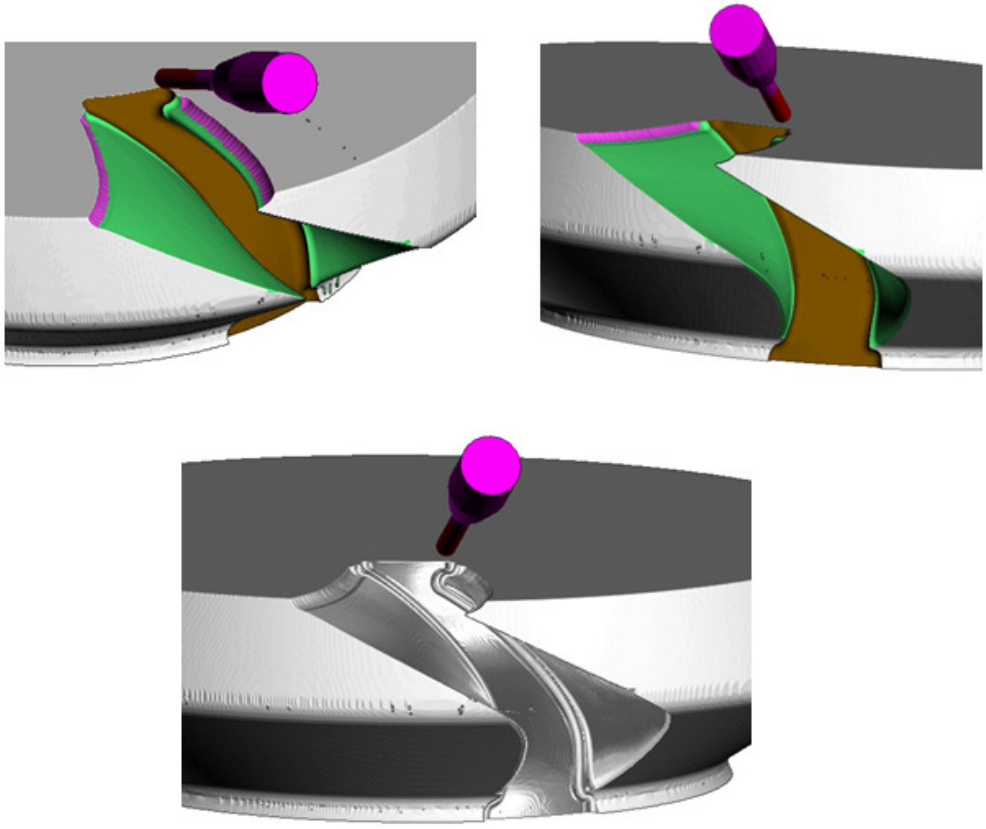


30 Continue the **ViewMILL Simulation** on toolpath **BN15-fin-a1**.

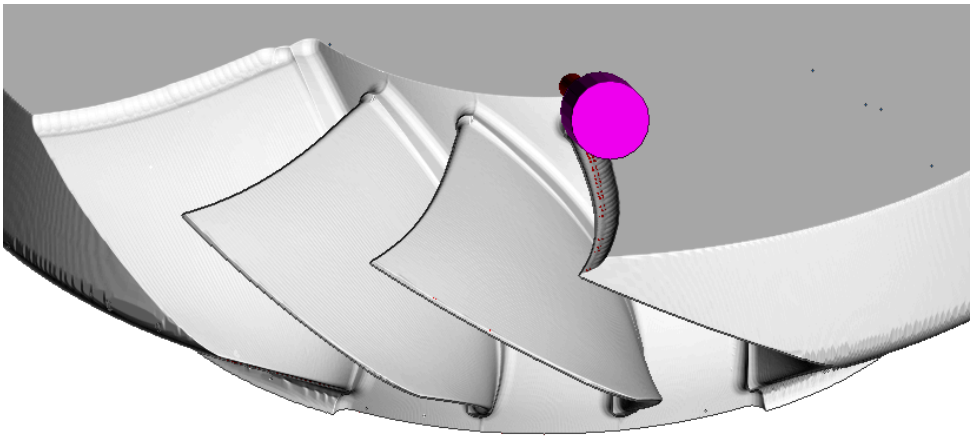




31 Continue the **ViewMILL Simulation** on toolpath **BN15-finHUB-a1**.



The final image (below) shows the result of a **ViewMILL simulation** including 17 copies of the **toolpaths** rotated by **20 Degree** increments around **Z**.

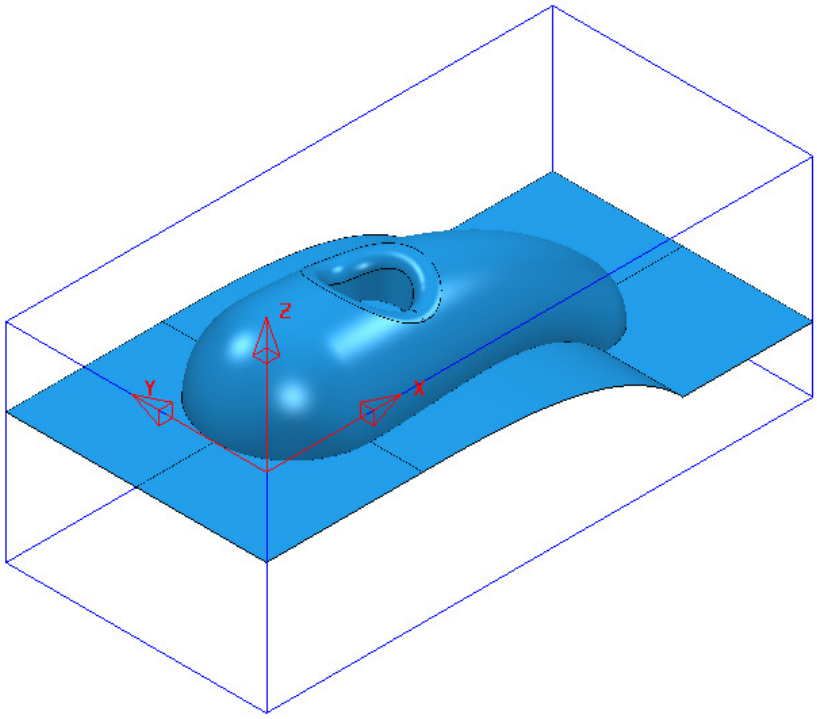


17. Five Axis-Tutorial

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

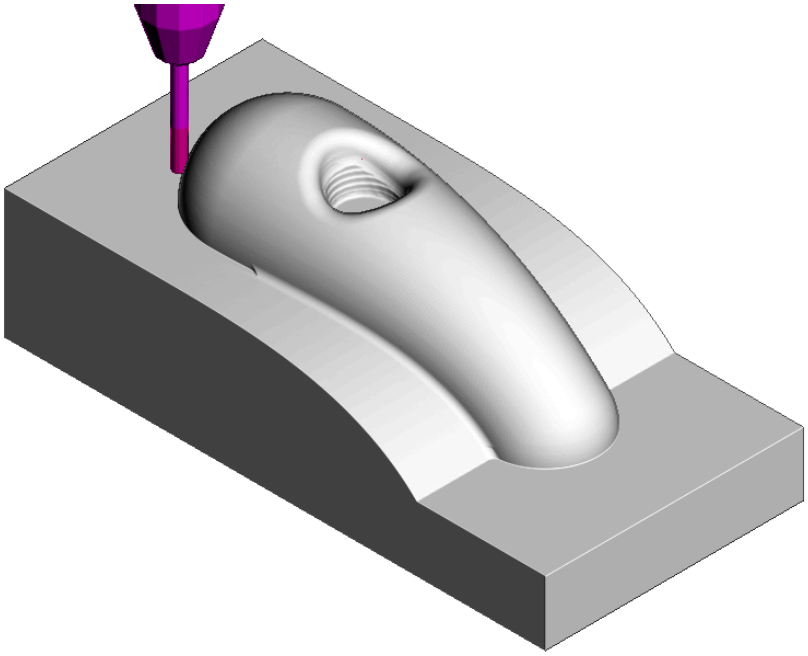
The **Project** currently contains 8, 3-Axis Toolpaths.

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 1**.

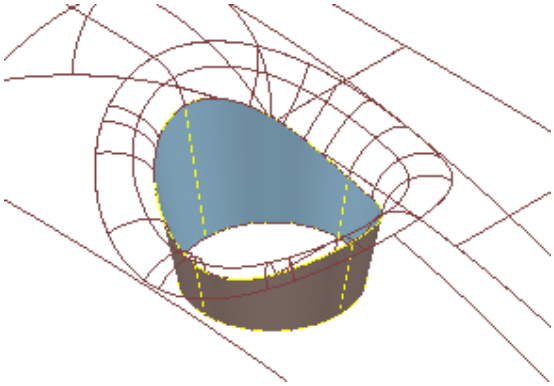


The remaining areas that require Five Axis machining include the central pocket (Swarf Machining), and the intersection between the main form and the 3D run-off (Embedded Pattern and Surface Finishing).

7 Select the ***ViewMILL*** **No-Image** display option and disconnect from the **simulation** process.

Central Pocket

8 **Activate** the tool **EM12**.

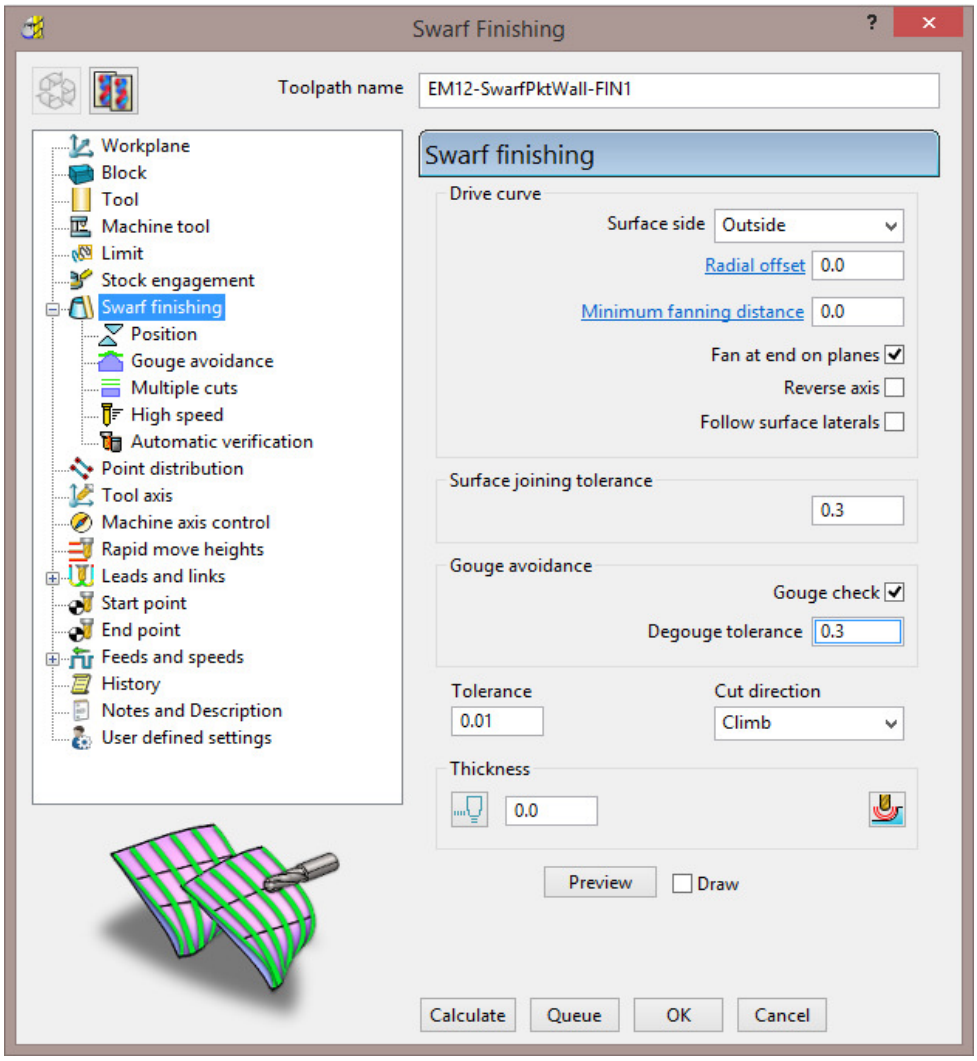


9 Select the **Pocket** wall.

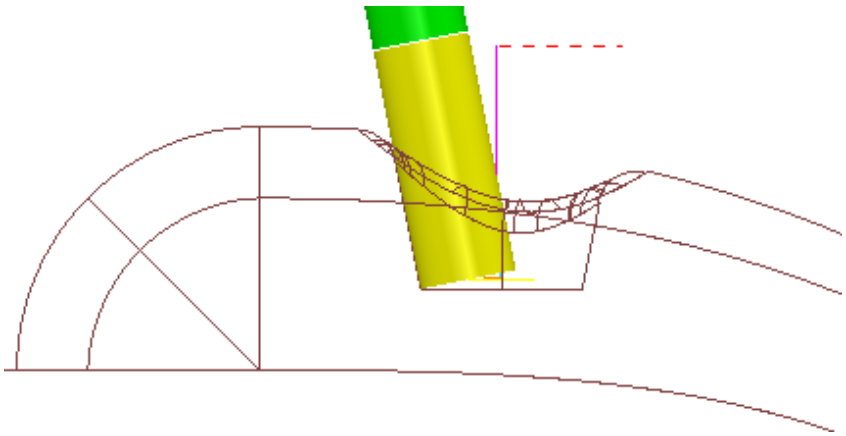
10 Select the **Toolpath Strategies** dialog select the **Finishing** tab.

11 From the list of strategies select **Swarf Finishing** and then **OK**.

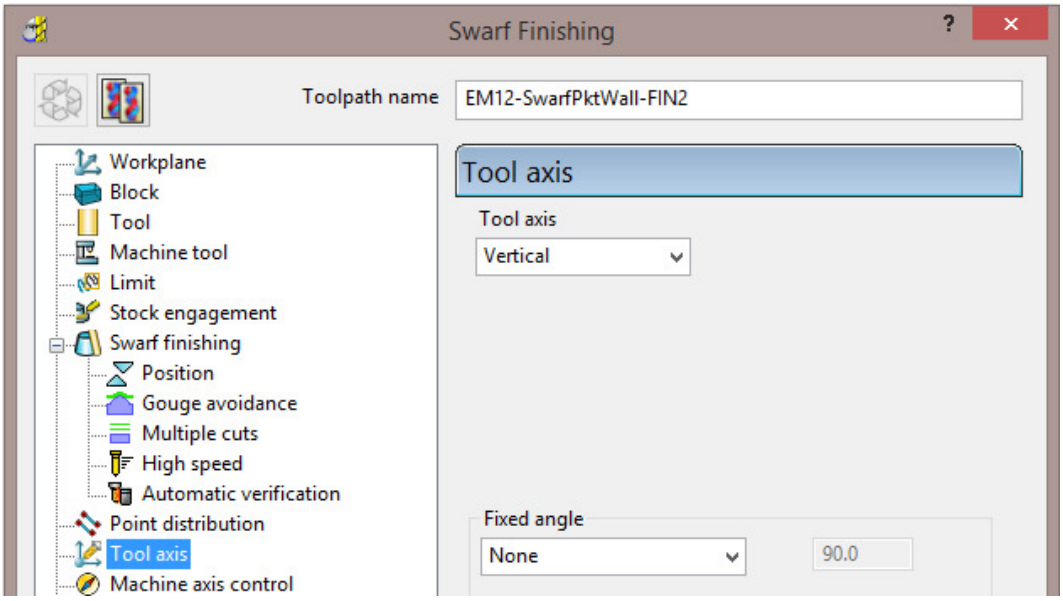
12 In the **Swarf Finishing** dialog enter data exactly as shown below:



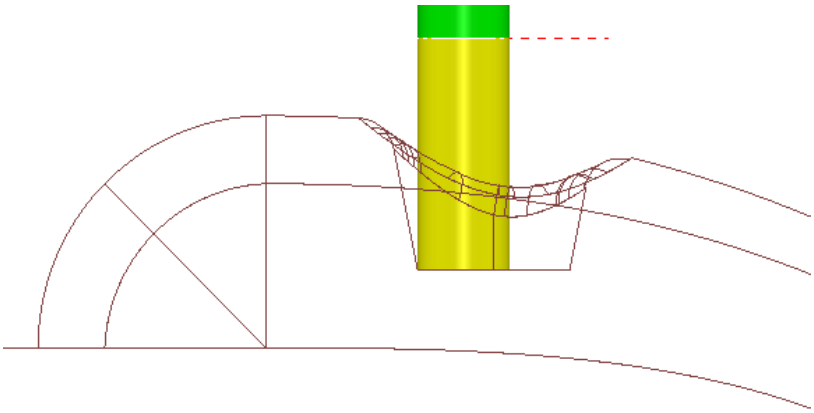
13 For both the **Lead in** and **Lead out** input **Surface normal arc** with ***Angle*** **180** and ***Radius*** **2**, then **Calculate** the toolpath.



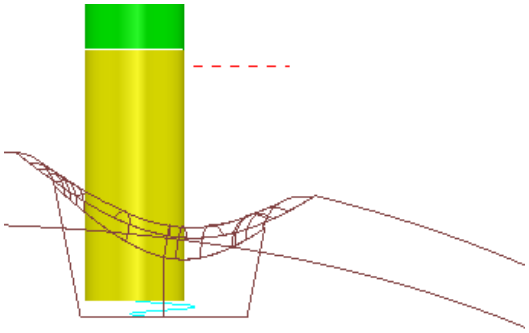
The **Swarf Finishing** strategy is created as ***Five Axis*** by default (The **Tool axis** option is set to **Automatic**). As shown in the above image this results in a wedge of material remaining on the flat base of the pocket. This can be removed with a second **Swarf Finishing** toolpath but this time with the **Tool Axis** set to **Vertical**.



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 **Lead in** move onto this toolpath to a ***circular*** **Ramp** with ***Angle*** **4**, ***Circle Dia (TDU)*** **0.65** and ***Ramp Height*** **2**.

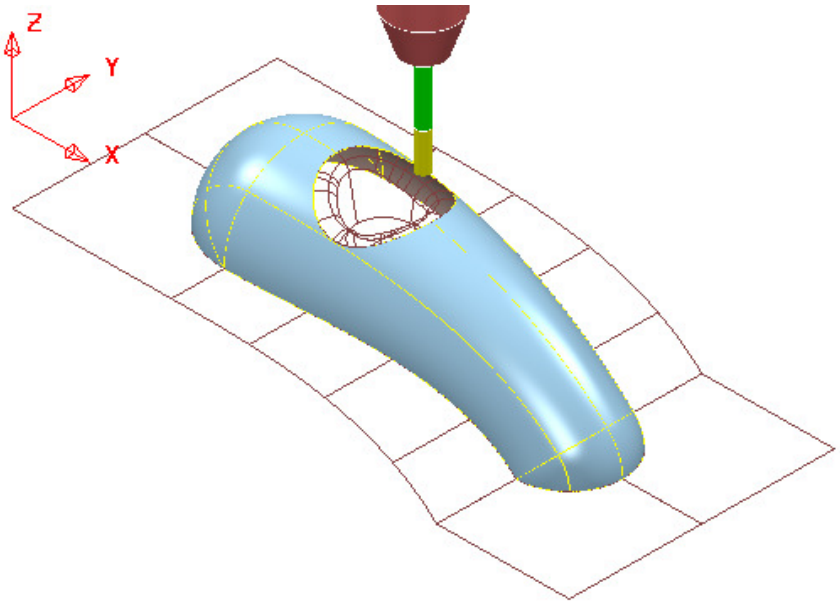


The Pocket is now fully machined.

16 **Save** (but do not close) the **Project**.

Main Form – Base Intersection

17 **Activate** the **EM5** tool.

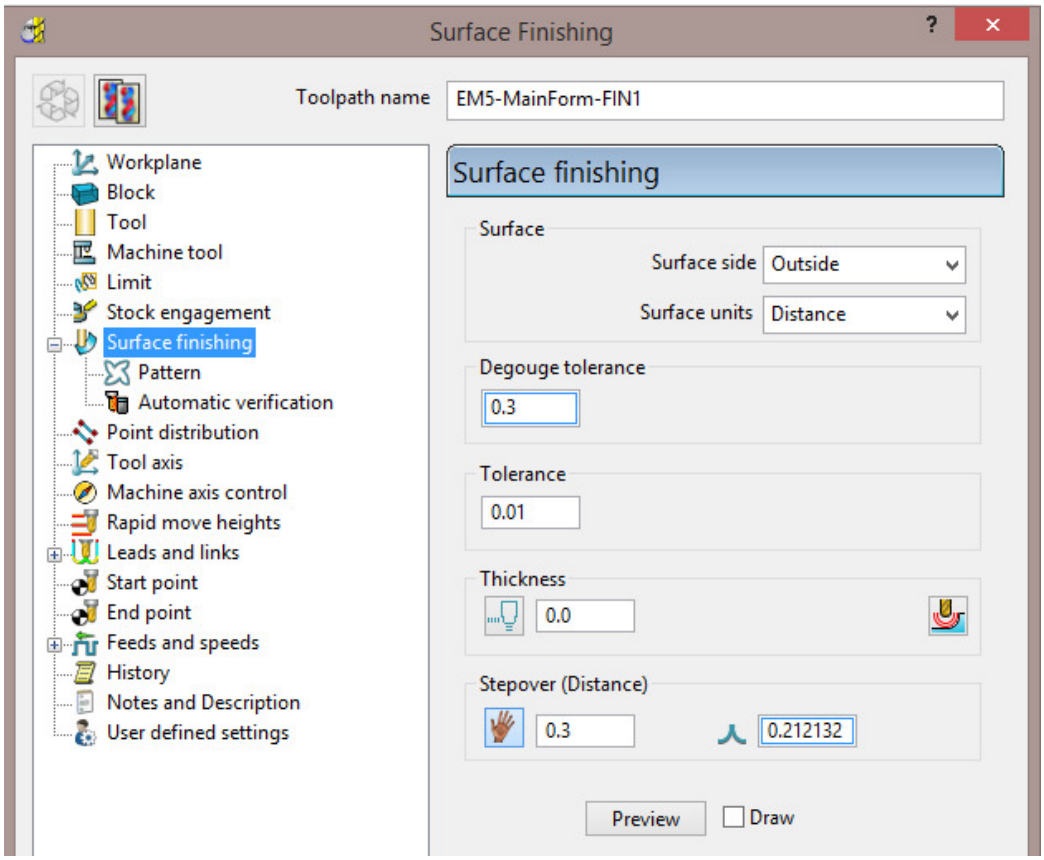


18 **Select** the main form **surface** (as shown in the above image).

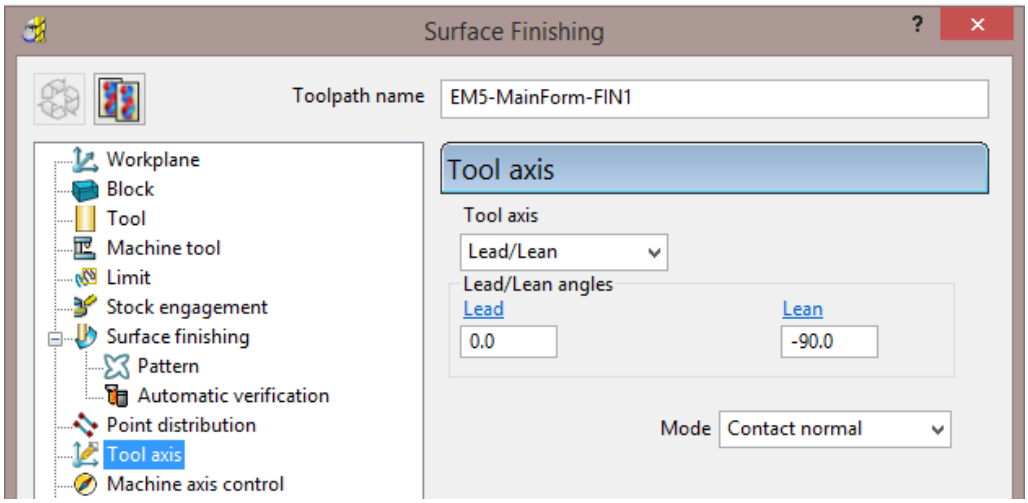
19 Select the **Toolpath Strategies** dialog  select the **Finishing** tab.

20 From the list of strategies select **Surface Finishing** and then **OK**.

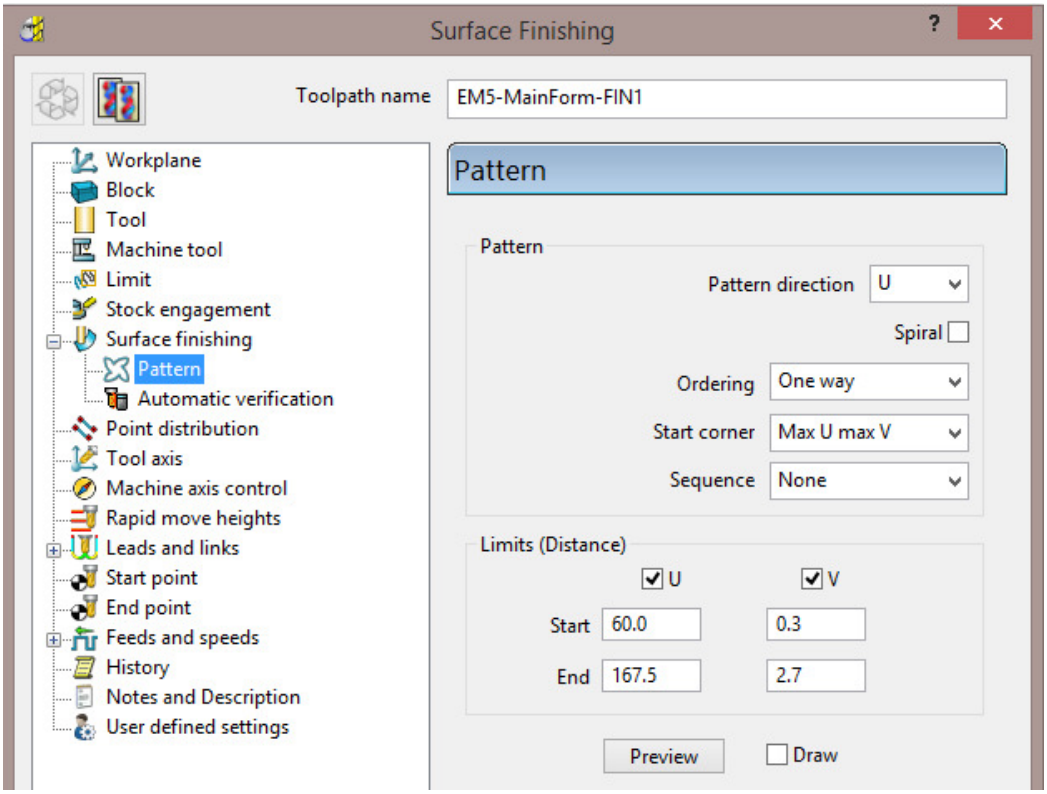
21 In the **Surface Finishing** page enter data exactly as shown in the image below.



22 Select the **Tool Axis** page and again enter data exactly as shown in the image below.

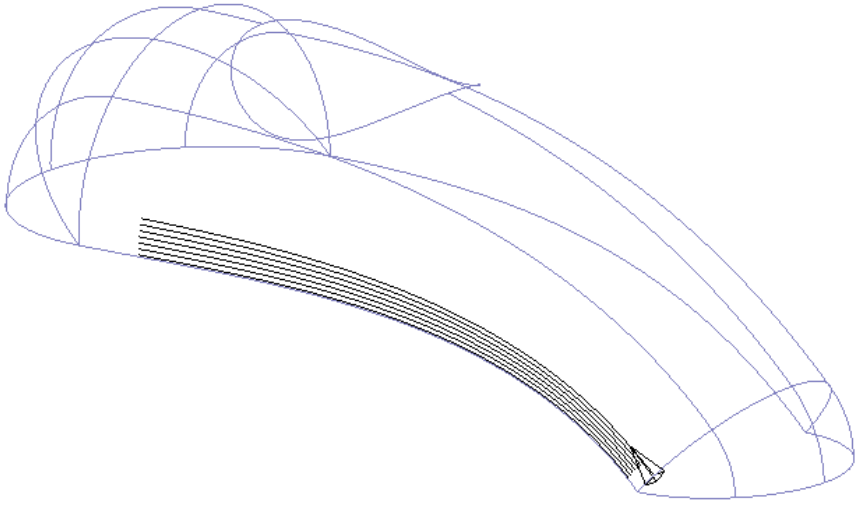


23 Select the **Pattern** page and again enter data exactly as shown in the image below.

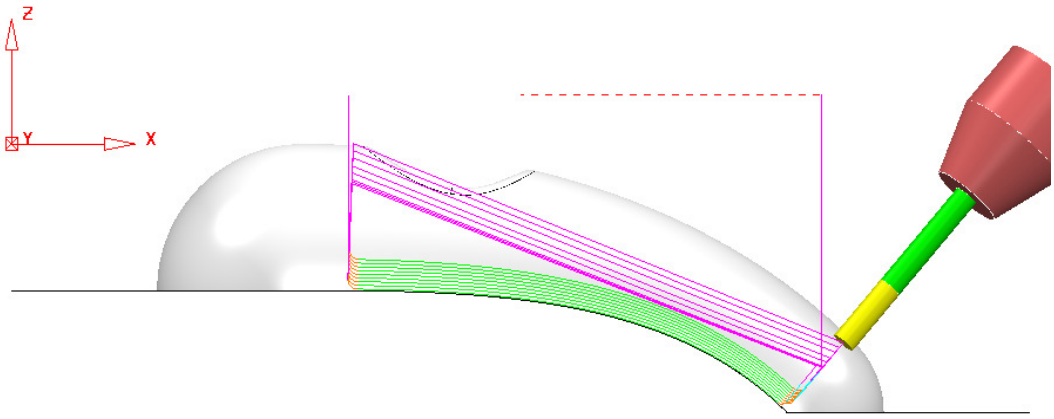


The values shown in the above page were finally arrived at after a trial and error process. The actual values are unit distance along and across the surface, curve network (U V).

24 Select **Preview** to visualise the defined pattern on the **Surface**.

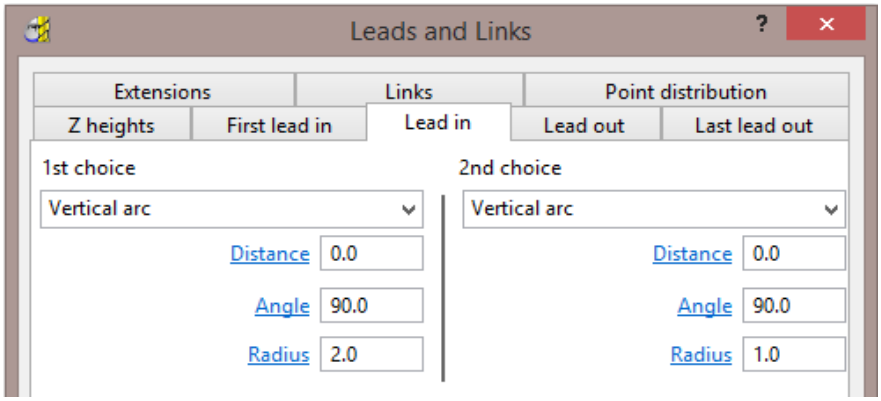


25 Select **Calculate** to process the toolpath.

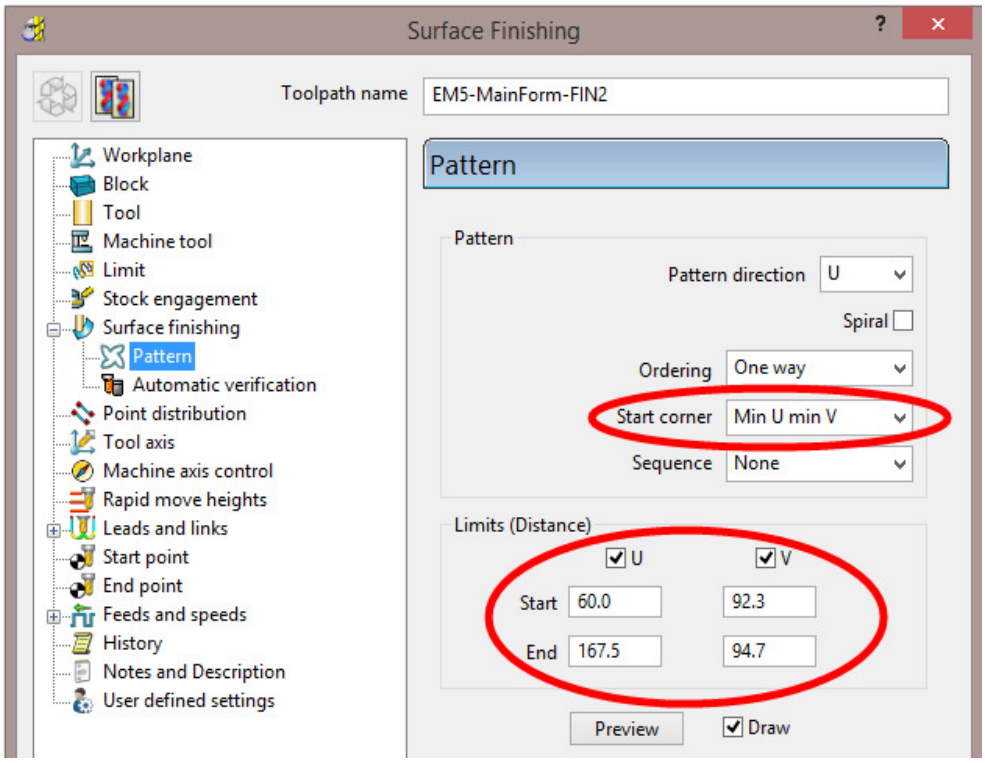


To help visualisation the above images show the preview with a larger area and stepover than specified in the instructions.

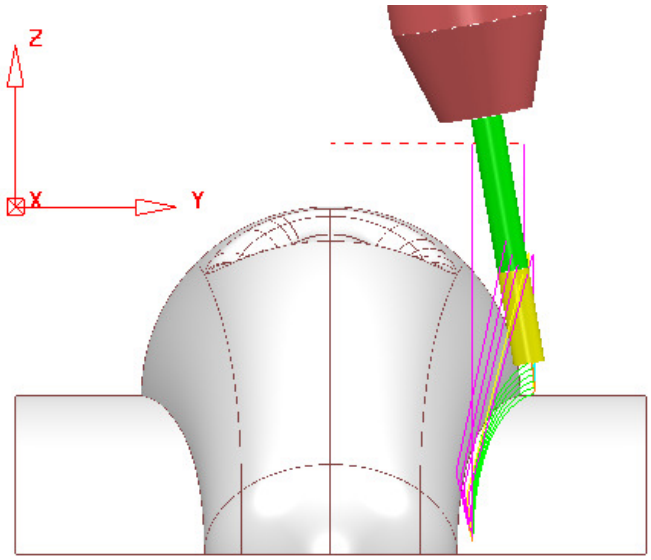
26 **Apply** the following **Lead in** and **Lead Out** settings.



27 Make a copy of the **Surface Finishing** toolpath and rename it as **EM5-MainForm-FIN2**.



28 Modify the original settings in the **Pattern Page** and select **Calculate** to perform the same strategy on the other side of the main form.

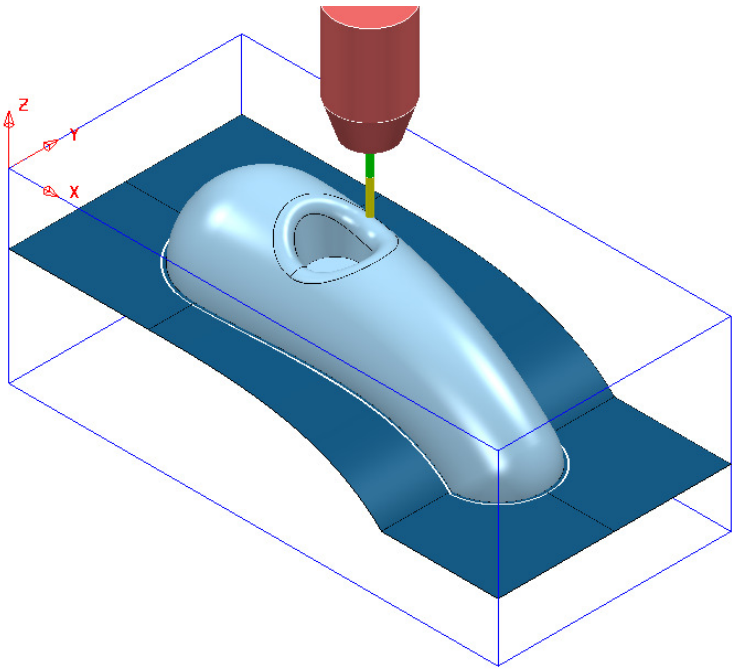


*The last 2 Surface Finishing strategies deliberately stop just short of the run-off surfaces. A more accurate, final pass will be created normal to the run-off surfaces using a separate **Embedded Pattern Finishing**.*

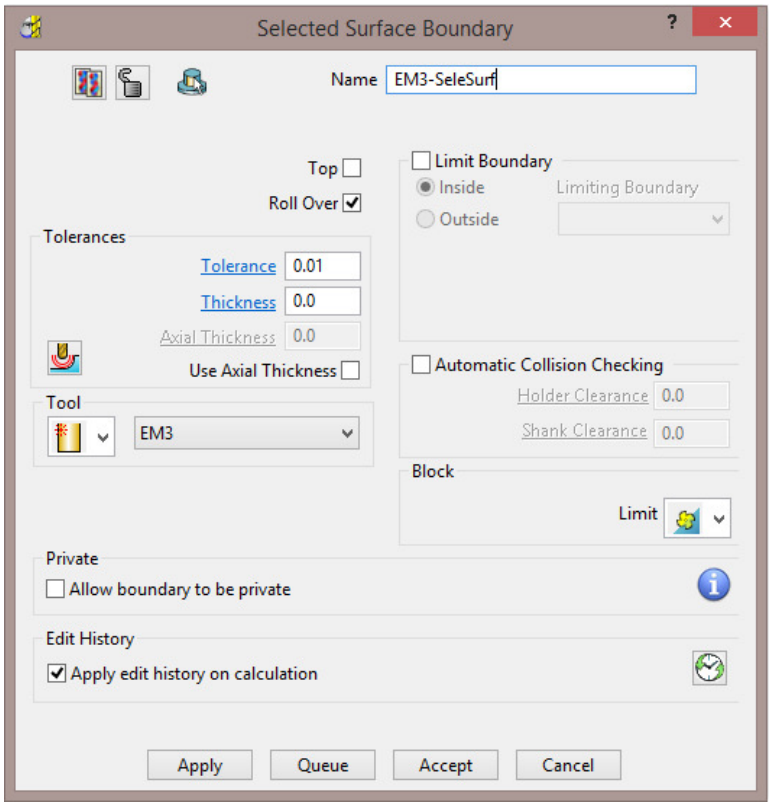
29 **Save** (but do not close) the **Project**.

Final Pass around Main Form - Base Intersection

30 **Activate** the **EM3** tool.



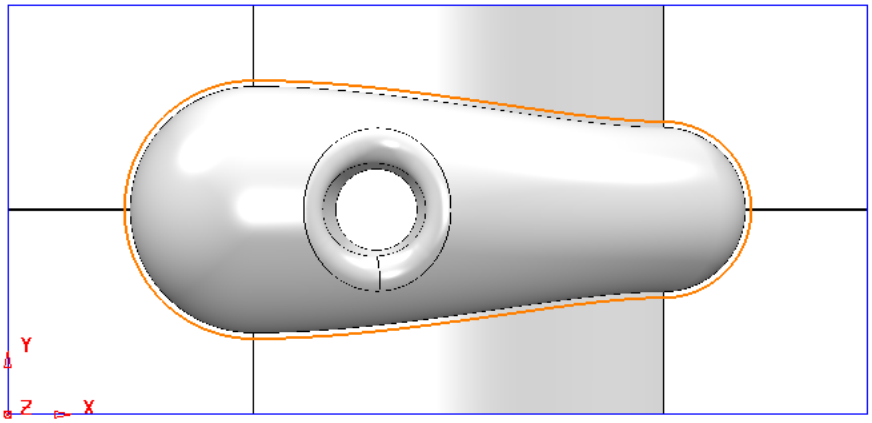
31 **Select** the ***main form*** and ***Pocket*** **surfaces** (as shown in the above image).



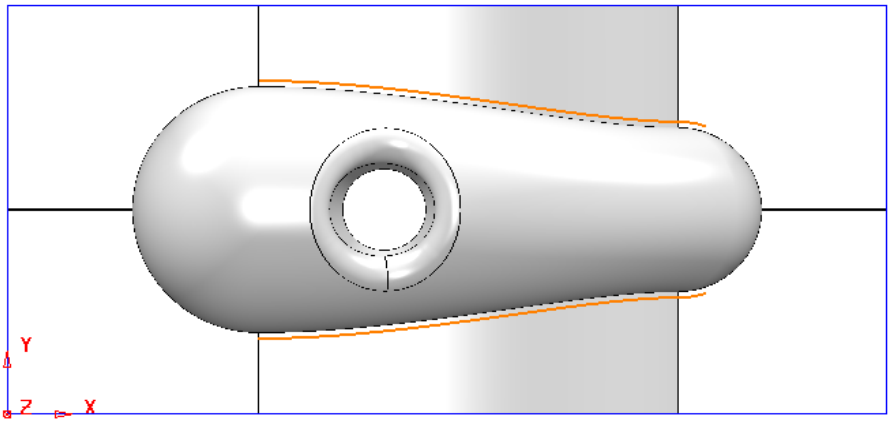
32 Create a ***Selected Surface*** **Boundary** (with **Roll Over** ticked).

33 Create an empty **Pattern** and insert the above **Boundary** segment into it.

34 **De-Activate** the **Boundary**.

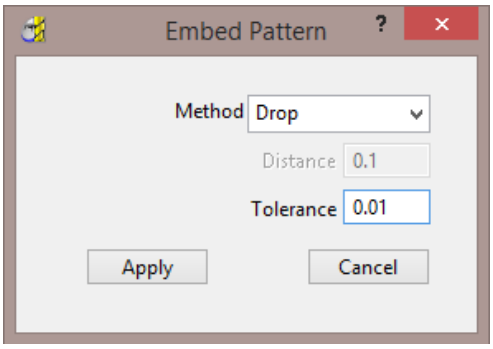


35 Use the **Curve Editor** to trim the **Pattern** into 2 segments that define the required extent for an **Embedded Pattern Finishing** strategy.



36 **Reverse** the **Pattern** segments as required to produce a **Climb Milling** direction (Material to the right of the tool feed direction).

37 Right mouse click on the new **Pattern** and from the local menu select **Edit – Embed**.



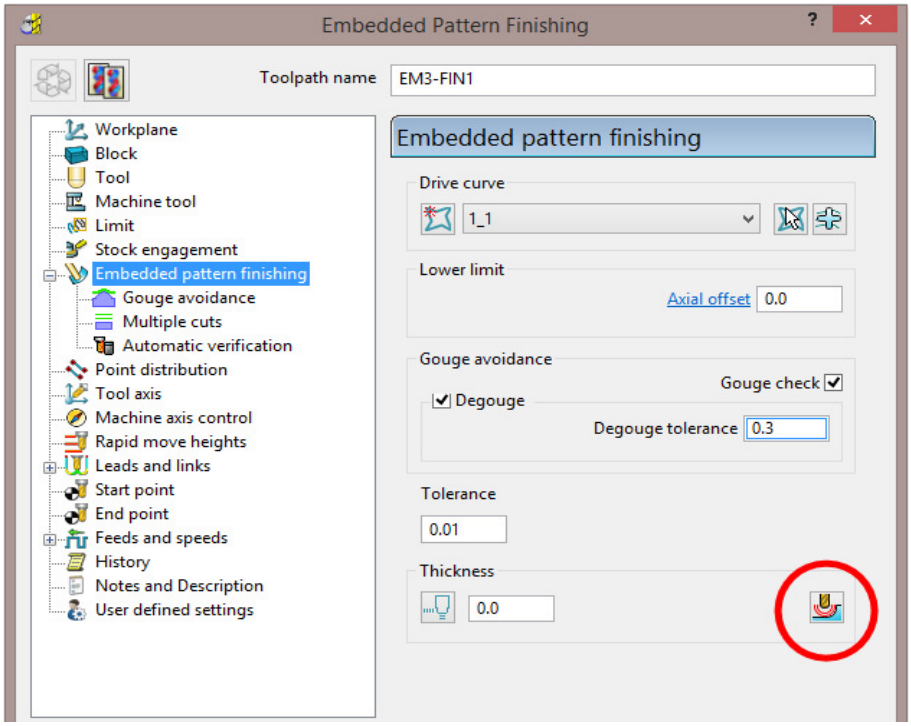
38 In the above dialog select ***Method*** as **Drop** with a **Tolerance** of **0.01** before selecting **Apply**.

A copy of the original ***Pattern* 1** is projected down Z onto the model as an ***Embedded Pattern* 1_1** (contains the surface normal).

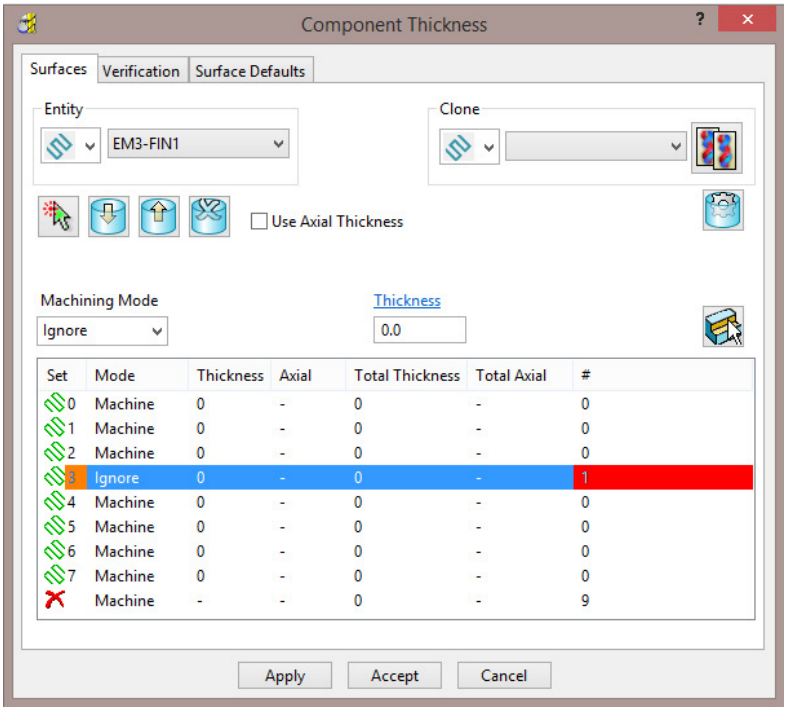
39 Select the **Toolpath Strategies** dialog  select the **Finishing** tab.

40 From the list of strategies select **Embedded Pattern Finishing** and then **OK**.

41 In the **Embedded Pattern Finishing** page enter data exactly as shown in the image below.



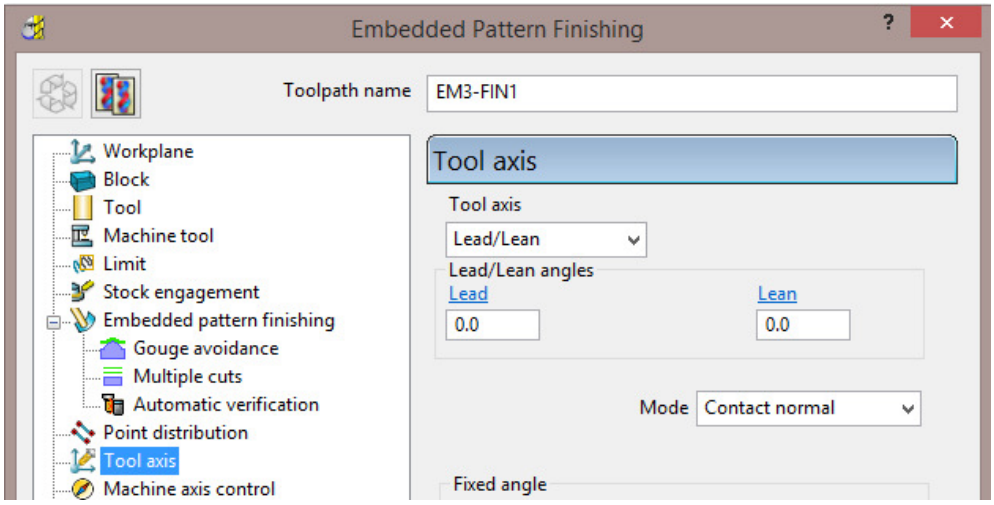
42 Open the **Component Thickness** dialog and acquire the ***Main Form*** **surface** to one of the rows.



43 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 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.

44 Back in the main **Embedded Pattern Finishing** dialog, open the **Tool Axis** page and select **Lead/Lean** both with a **0** angle.

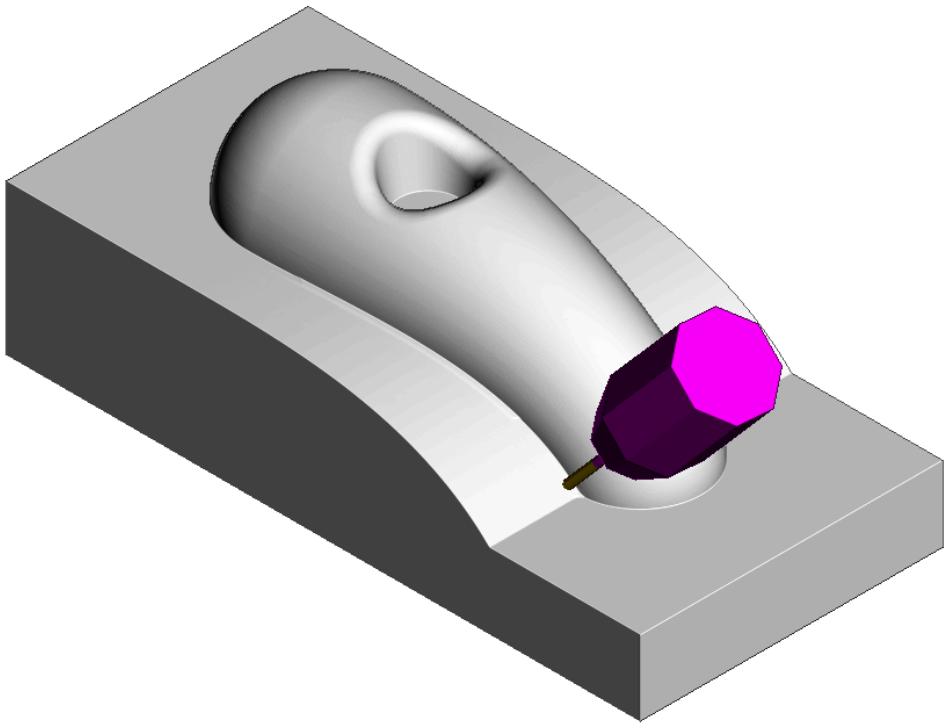


45 Set all **Link** moves (***Short***/***Long***/***Default***) to **Incremental**.

*This is to prevent the toolpath from linking straight through the main cowling surface (**Component Thickness** set to **Ignore**).*

46 **Calculate** the **Embedded Pattern** toolpath.

47 Run a full **ViewMILL simulation** (Final **EM3-FIN1** toolpath show below).



48 **Save** the **Project**.