Delcam

PowerMILL

Training Course

Delcam

Talbot Way, Small Heath Business Park, Birmingham, B10 0HJ.
www.delcam.com

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.

The software described in this document is furnished under a license agreement and may be used only in accordance with the terms of such license.

Training Centre
Tel: 0121 683 1050
Fax 0121 7665511

Customer Support
Tel: 0121 683 1010
Fax: 0121 7665542

PowerMILL 9 Five Axis Contents

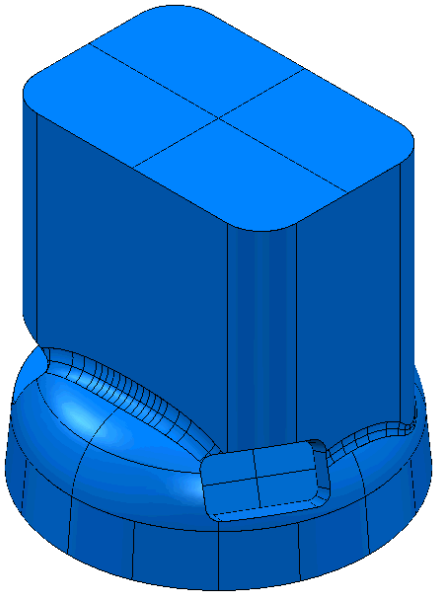
1. 3 + 2 Axis Machining

Introduction

On a **3 + 2 Axis** Machine it is possible to index the head and\or bed to realign the tool prior to performing standard **X Y Z** transitions. This is achieved either by manual adjustment or as part of the cnc control.
It is possible for customers who do not possess a **PowerMILL Multi-Axis** licence to create **3+2** strategies by using individual **Workplanes** to control **Tool Alignment** and output ncdata via the **NC Preferences** form with the **Automatic Tool Alignment** set to **Off**.
It is however both faster and easier to create **3 + 2** toolpaths if the **Multi-Axis** licence is available as it provides access to a larger range of options with minimal dependency on individual **Workplanes**. Either way **PowerMILL** enables components normally requiring a series of separate **3-Axis** operations to be machined in one set-up. This could include direct machining of undercut features or sidewalls deeper than the maximum tool length.
It is essential to apply suitable **Toolpath - Leads**, **Links**, and **Extensions** to eliminate any potential gouges.

3 + 2 Axis - Machining Example

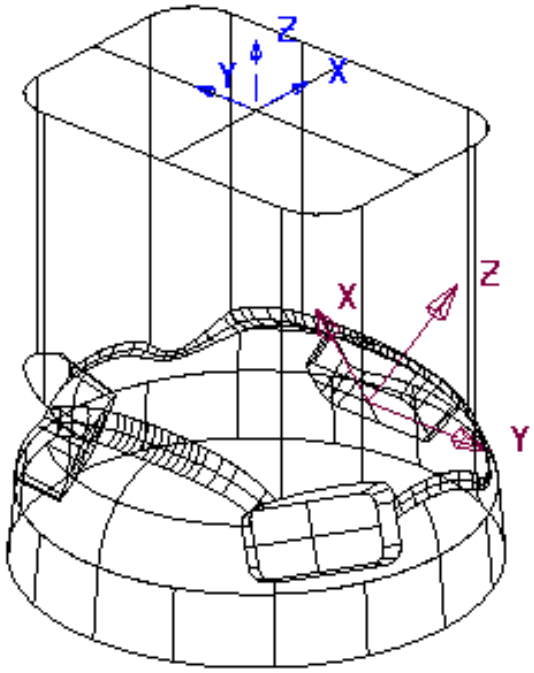
- **Import** the **Model**:-
  D:\users\training\PowerMILL_Data\five_axis\3plus2_as_5axis\3plus2b.dgk



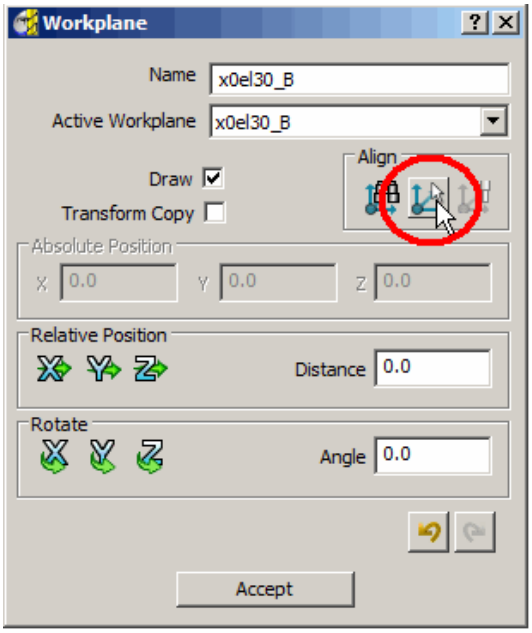
Note; The model is approx. **175mm** high.

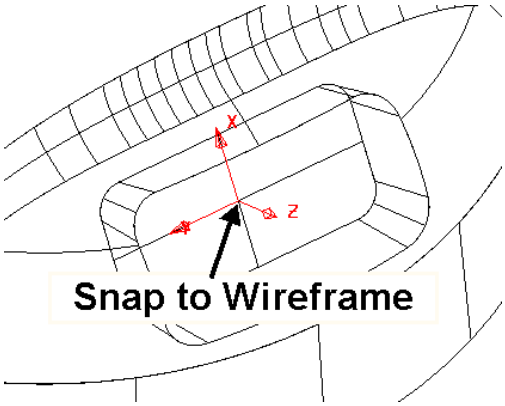
- **Select** an **Isometric** view and consider the machining options. Note the relatively high sides of the component and the orientation of the three recesses making it impossible to machine as **3 Axis** (with the tooling aligned to the **Z-Axis**).
- **Create** a **Workplane** and move it by a distance of **Z175** to clear the top of the component and ***Name*** it as **ztop175_A** and make it **Active**.

Note: **Workplane** alignment for compound angles is easier to achieve using **PowerSHAPE** (If familiar with the commands). A limited functionality version called **Wireframe modelling** is directly accessible (as standard) from **PowerMILL**. Create a **Pattern** to enable access to the **Wireframe modelling**, select the model and **Insert – Wireframe modelling**. Create, reposition, and re-orientate **Workplanes** dynamically as required. Otherwise use the direct method from within **PowerMILL** as described on the following page.



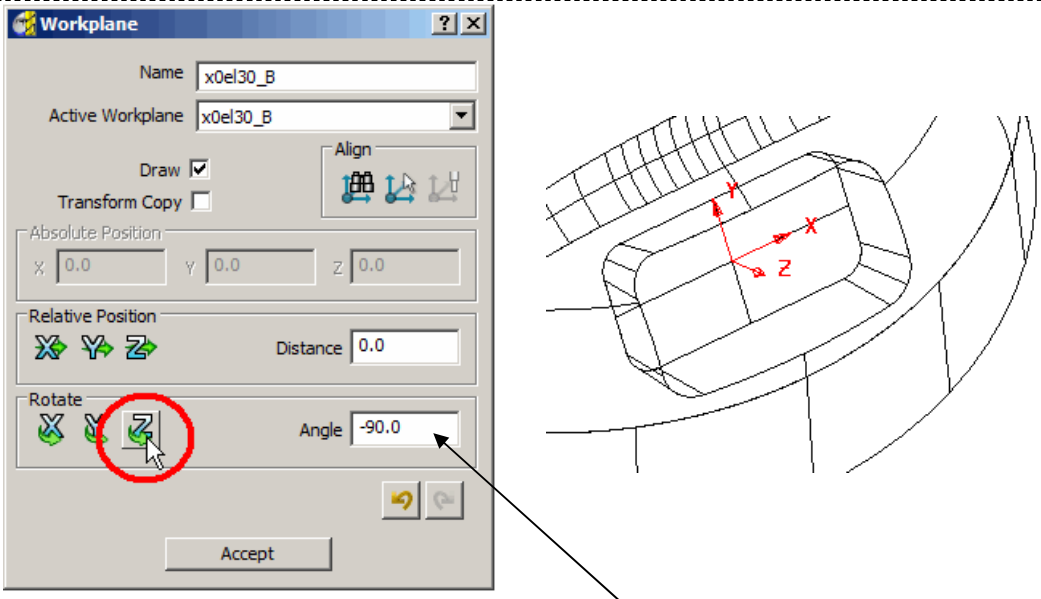
- **Create** a new **Workplane**, ***Name*** it as **x0el30_B**, select the **Align to Pick** icon and using the left mouse key **snap** or **box** the wireframe crossover at the base of the first pocket (located along global X).



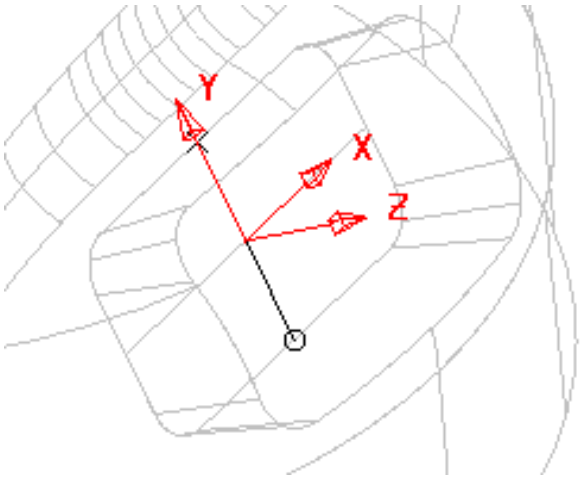


- Activate the **Workplane x0el30_B**.

The **Workplane** is automatically aligned to the wireframe with the Z Axis normal to the surface. It still requires further editing as it is required that the X-Axis points Anticlockwise around the component in reference to the global coordinates.



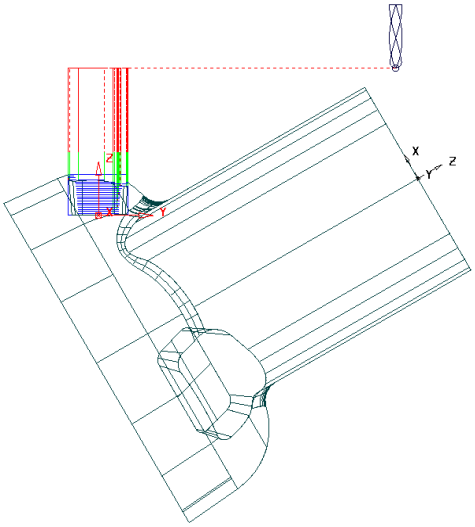
- ***Rotate*** the **Workplane - Around Z** by an ***Angle*** **-90** (normal to the base of the recess) ensuring that the **X-Axis** is pointing anticlockwise relative to the **Transform** (Global Datum) as viewed from the top of the component (If not already the case).



- Create another **Workplane** for the 2nd recess and ***Name*** it as **x120el30_C**.
- **Deactivate** the original **Workplane**, **Rotate** the **x120el30_C Around Z** by **120** degrees.
- **Activate** the **Workplane x120el30_C**.
- Repeat for the 3rd recess, rotating a copy of the **Workplane** a further **120** degrees and renaming it as **x240el30_D**.

The component is now ready to be machined creating separate strategies relative to the 4 different **Workplane** alignments. (**ztop175_A**, **x0el30_B**, **x120el30_C**, and **x240el30_D**).

For each of the 3 Pockets a rectangular material **Block** will be created locally, relative to the required **3+2 Workplane**. A **Model Boundary** will also be created around each pocket to be included in the machining strategies (Machine **Inside** Boundary).

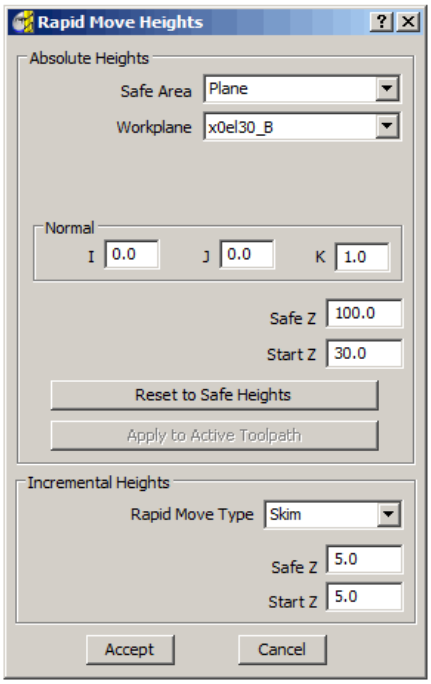


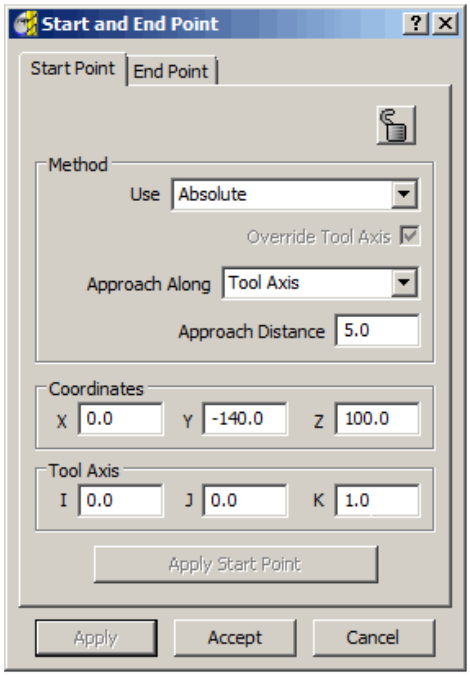
For users who are new to multiaxis work, it is advised that the **Rapid Move Heights** and **Start\End Point** for each toolpath are arranged to be on top of the component to guarantee safe rapid movement between individual machining **Workplanes** (as shown left).

Select a view along X and move the cursor to a suitable position for the Tool Start and End Point on the screen. The cursor X Z coordinate position is displayed to the bottom right of the graphics area.

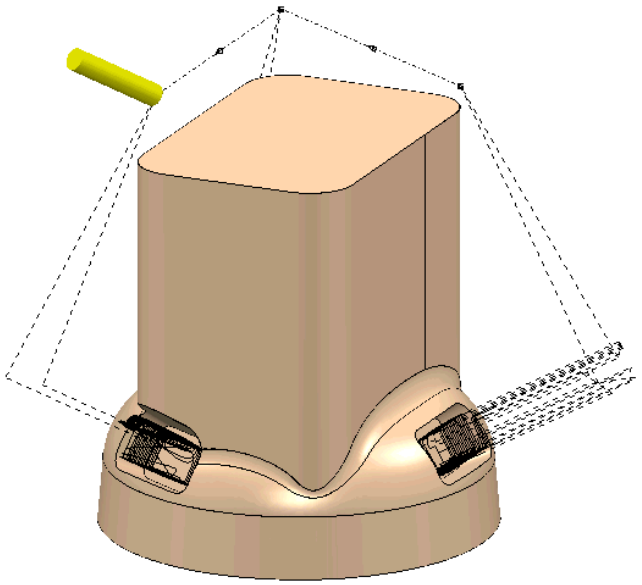
- Suitable values for **Rapid Move Heights** and **Start and End Point** to be applied to the local recesses are as illustrated above and as entered manually into the forms below.

Note:- Enter the same values for **End Point** that are shown input for the **Start Point**.





Once all **Workplanes** have been created a series of toolpaths can be created switching from one **Workplane** to the next to provide suitable **Tool Alignments**. Each individual toolpath is effectively a 3-Axis operation relative to the currently active **Workplane**.



- Create machining **Strategies** as listed below to the specified **3+2 Workplanes**.

TOOL	WORKPLANE	STRATEGY	STOCK	TOOLPATH
		3-AXIS ROUGHING		
DIA 40 Tiprad 6	ztop175_A	OFFSET Stepover 35 - Stepdown 10	1.0mm	D40t6rgh-a1
		3-AXIS SEMI-FINISH		
DIA 40 Tiprad 6	ztop175_A	CONSTANT Z Stepdown 2	0.5mm	D40t6sem-a1
		3+2 ROUGH RECESSES		
DIA 10 Tiprad 1	x0el30_B	OFFSET	0.5mm	D10t1rgh-b1
DIA 10 Tiprad 1	x120el30_C	OFFSET	0.5mm	D10t1rgh-c1
DIA 10 Tiprad 1	x240el30_D	OFFSET Stepover 3 - Stepdown 2	0.5mm	D10t1rgh-d1
		3+2 FINISH RECESSES		
DIA 10 Tiprad 1	x0el30_B	OPTIMISED CONST Z	0mm	D10t1fin-b1
DIA 10 Tiprad 1	x120el30_C	OPTIMISED CONST Z	0mm	D10t1fin-c1
DIA 10 Tiprad 1	x240el30_D	OPTIMISED CONST Z Stepover 2 - Stepdown 1	0mm	D10t1fin-d1

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

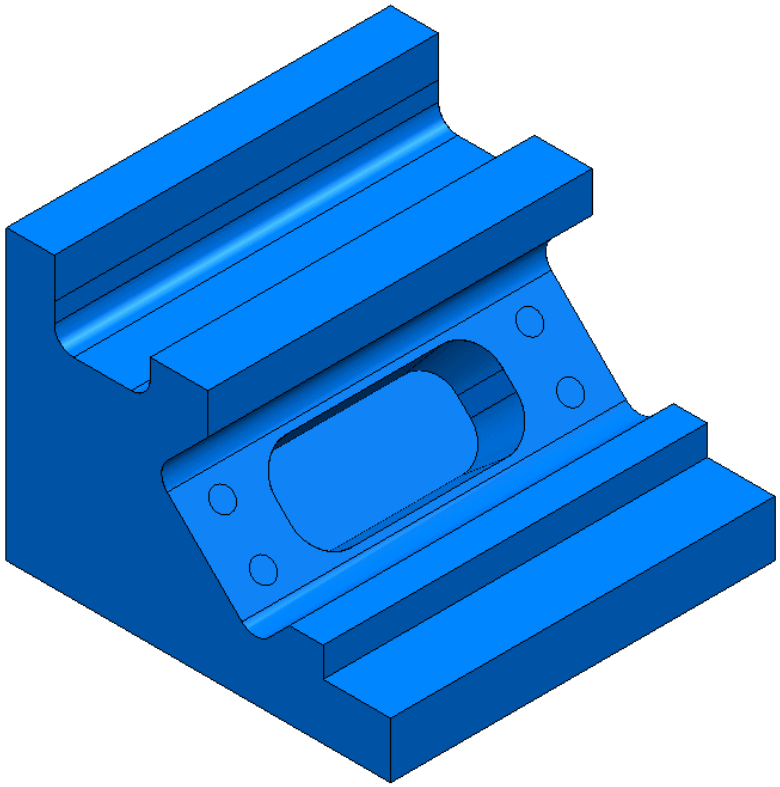
D:\users\training\COURSEWORK\PowerMILL-Projects\3+2example
(It will be used again later during the **Swarf Machining** chapter).

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** form.

3+2 Axis – Stock Model Application

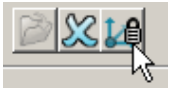
The **Stock Model** represents the un-machined material at any point in the machining process. An empty **Stock Model** is created, followed by **applying** the material **Block** and\or any number **toolpaths** to be considered in the process. The **Stock Model** is then updated by selecting **Calculate**,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

- **Delete all** and **Reset** forms.
- **Import** the **Model**:-
  D:\users\training\PowerMILL_Data\five_axis\AnglePad\StockModelRest



The model contains undercut pockets, 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 local **3+2** machining within each pocket.

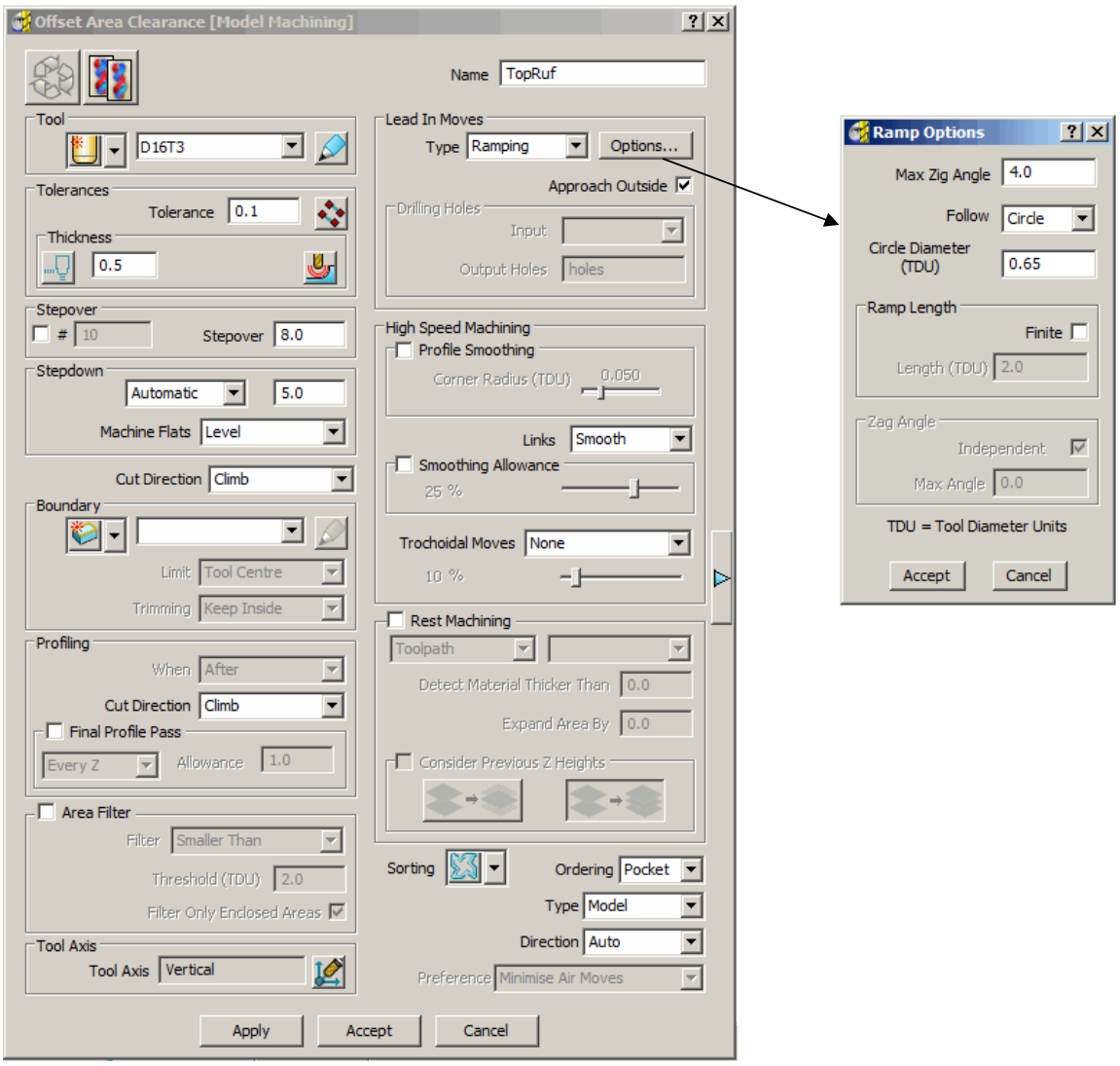
- **Open** the **Block** form and **Calculate** to **Min\Max** limits.
- Select **Lock** the **Block** (to the global co-ordinate system).
- **Accept** the form.



By **creating** and **locking** the material **Block** to the **Transform** (global co-ordinate system), it's orientation and position will remain unchanged when activating a **Workplane**.

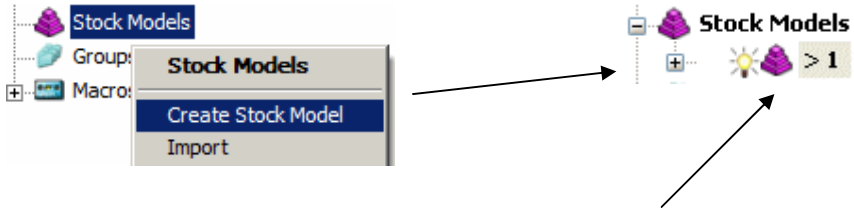
- **Create** a **Dia 12 - tip radius 1** tool and **Rename D12T1**.
- **Create** a **Dia 16 - tip radius 3** tool and **Rename D16T3**.

- In the **Rapid Move Heights** form  **Reset To Safe Heights** with ***Rapid Move Type*** set to **Skim**.
- Set both **Start Point** and **End Point** as **Block Centre Safe**. 
- **Activate** the tool **D16T3**.
- Select the **Toolpath Strategies** icon  and from the **3D Area Clearance** form select the **Offset AreaClear Model** option.
- Enter the **Name** - **TopRuf** along with the remaining values and settings ***exactly*** as shown below.

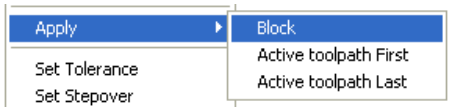


- **Apply** and **Cancel** the form.
- Select an **Iso1** view.

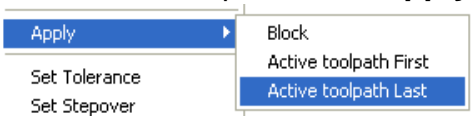
- In the **explorer**, right mouse click on **Create Stockmodel**.



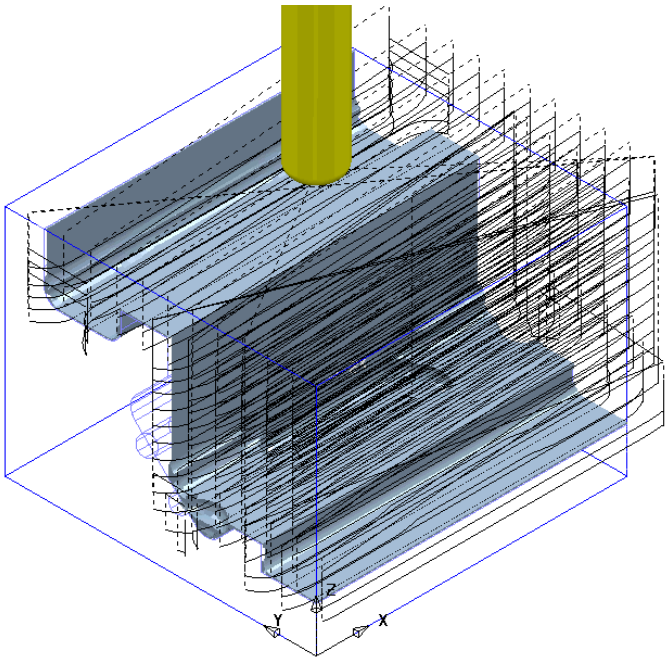
- Right mouse click over the new (empty**) Stock Model** and from the local menu select **Apply - Block**.



- With the local **Stock Model** menu still open select **Apply - Active toolpath Last**.



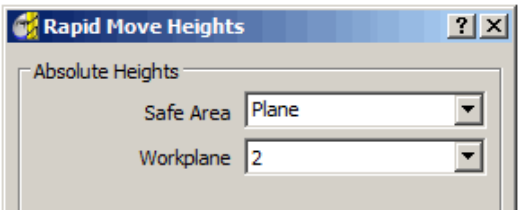
- With the local **Stock Model** menu still open, select **Show Rest Material**, followed by **Drawing Options - Shaded**, and finally **Calculate**.



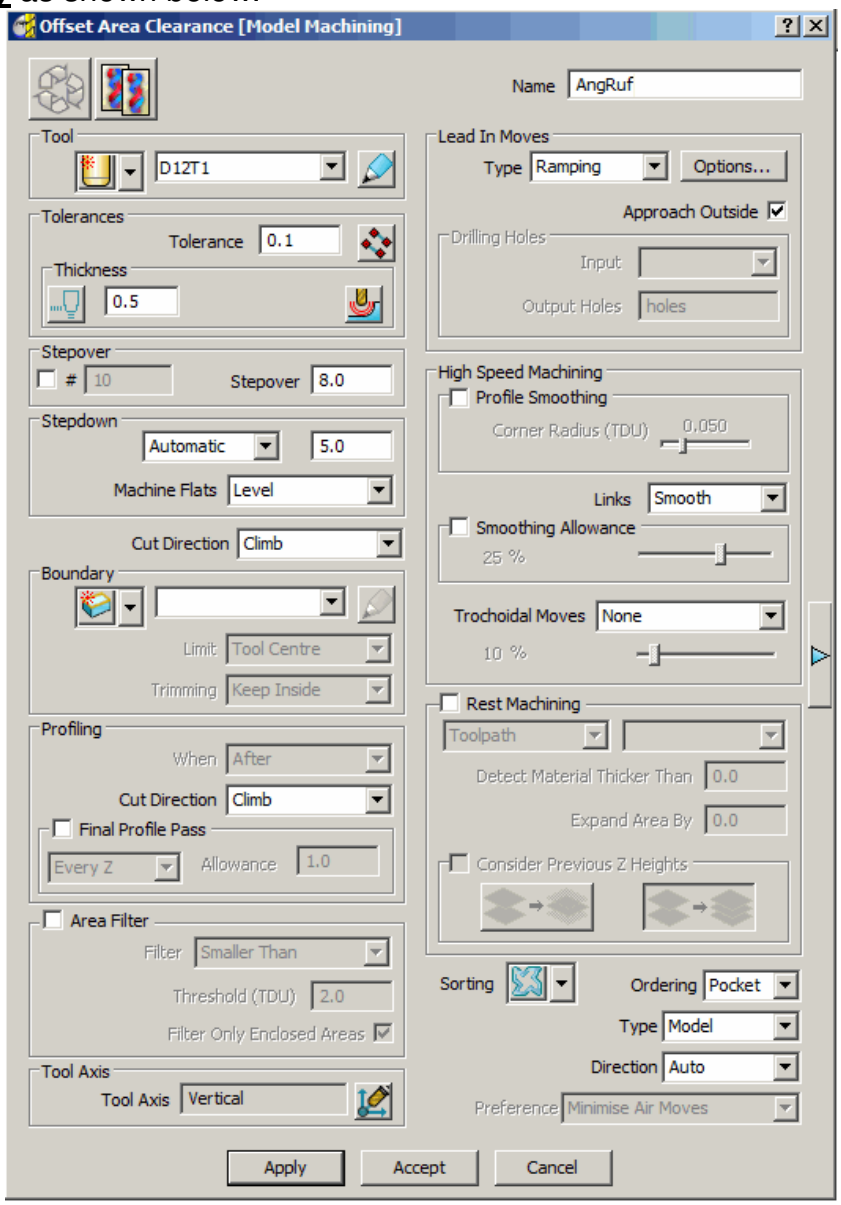
The **3-Axis** Roughing operation has removed all accessible material leaving a **0.5 thickness** on the component form. This is clearly visible on the displayed **Stock Model**.

- Right mouse click on the **active** toolpath **TopRuf** and select **Settings** to reopen the **Offset Area Clearance** form.
- Select the ***Copy Toolpath*** icon  ready to input some new parameters and settings for the **3+2** roughing strategy (keep the form open).
- **Activate - Workplane 2** to change the set up to a **3+2** orientation.
- **Activate** the tool **D12T1**.

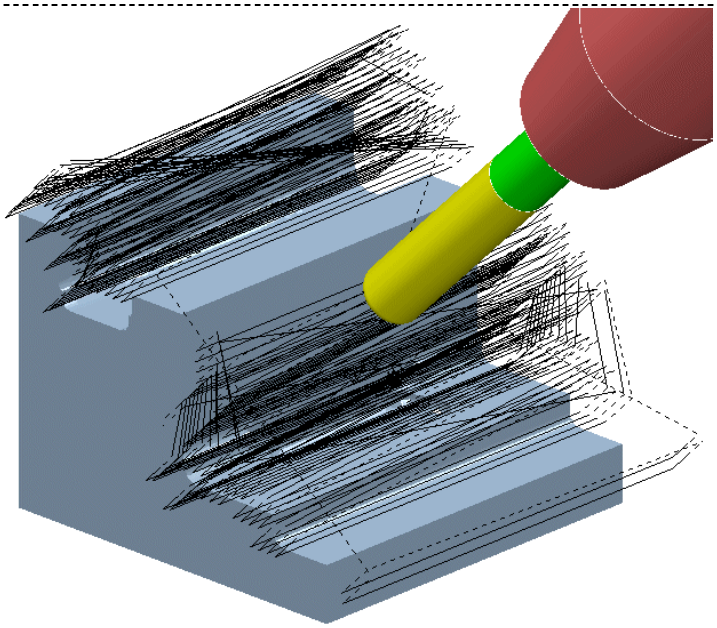
- From the main toolbar select **Rapid Move Heights** and input the correct **Workplane (2)** in the form before selecting **Reset to Safe Heights**.



- **Reset Start\End Point** as **Block Centre Safe**.
- Enter the **Name** - **AngRuf** along with the remaining values and settings ***exactly*** as shown below.

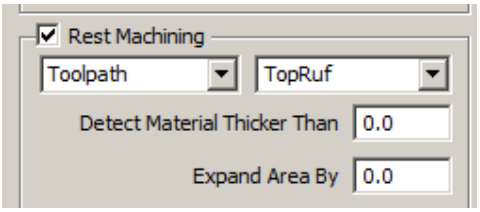


- **Apply** and **Cancel** the form.



The **3+2 Axis Roughing** operation has removed all the remaining material but at the expense of a lot of wasted time cutting fresh air. Most of the material has already been removed by the previous strategy. This is clearly visible on the illustration.

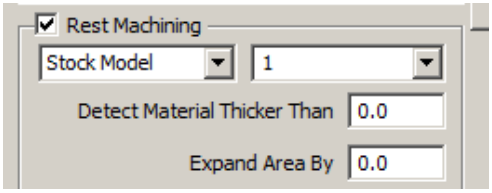
- Try to **Apply** the **Strategy** with **Rest Machining** to **Toolpath** applied as shown in the illustration, below left.



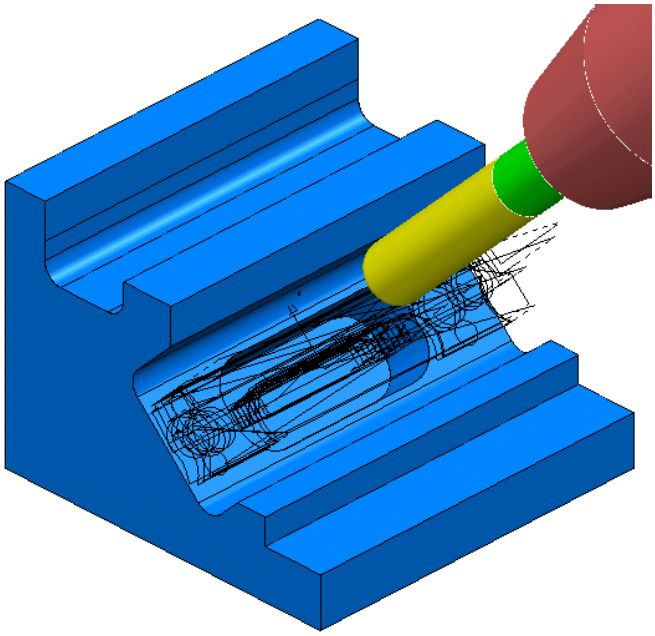
The strategy fails to calculate!

Note; It is not possible to apply **Rest Roughing** to an **Area Clearance** strategy if, as in this case, the **reference toolpath** has been generated relative to a different **Workplane** alignment. This is overcome by using the **Stock Model** to limit the **Rest Roughing** instead as shown in the next section.

- Right mouse click on the **Active** toolpath **AngRuf** and select **Settings** to reopen the 3+2 **Offset Area Clearance** form.
- Select the ***Recycle Toolpath*** icon  ready to input some new parameters (keep the form open).
- In advance setting  ***untick*** - **Allow Tool Outside Block**.
- Tick the box labelled **Rest Machining** and in the local selector boxes set to **Stock Model** and **1** as shown below before selecting **Apply**.

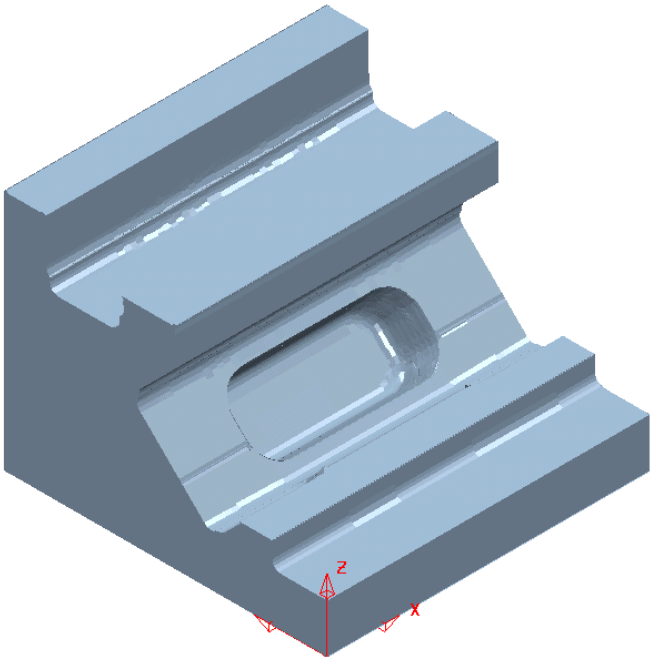


The **Rest Roughing** toolpath is successfully generated within the bounds of the **Stock Model**.



The modified **Rest Roughing** toolpath now successfully operates within the **Stock Model** limits (as shown left).
Note:- The material removed by the toolpath is not included as part of the actual **Stock Model** at this stage.

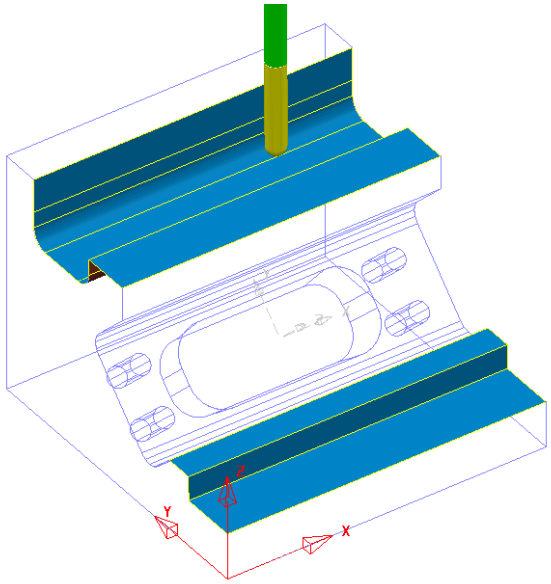
- In the **explorer** right mouse click on the **Stock Model** and in the local menu select **Apply - Active toolpath Last**, followed by **Calculate**.



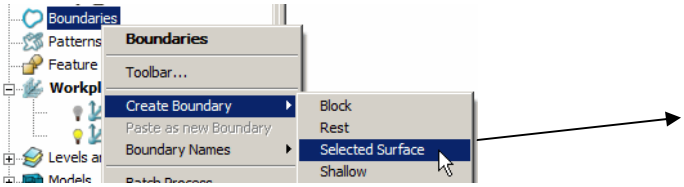
The **Stock Model** now displays the remaining material after both the **3-Axis Roughing** and **3+2 Roughing** operations.

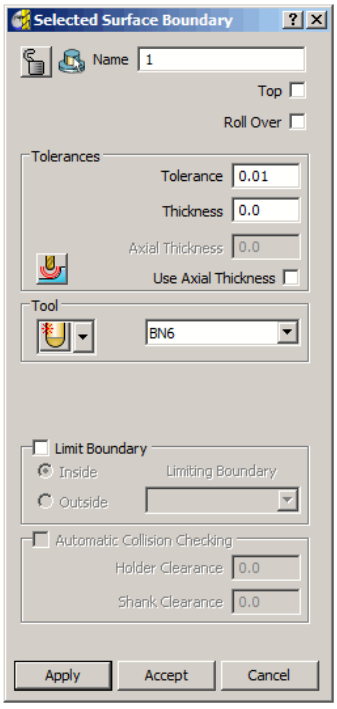
Unlike **Area Clearance**, rest machining with **Finishing** strategies cannot be directly referenced to a **Stock Model**. However it is possible to create and apply **Stock Model Rest Boundaries** where required, providing suitable **rest** limits for subsequent finishing operations.

- Activate **Workplane 1**.
- Create a **Dia 6 Ball Nosed** tool named **BN6**.
- Select the **Surfaces** (shown shaded below) required for initial **finish machining** relative to **Workplane 1**.

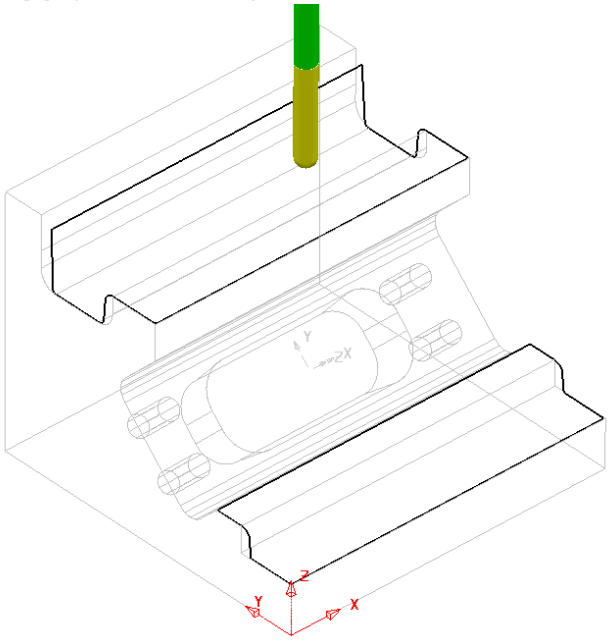


- In the **explorer** Right click over **Boundaries** and select **Create Boundary** followed by **Selected Surface** to open the following form.

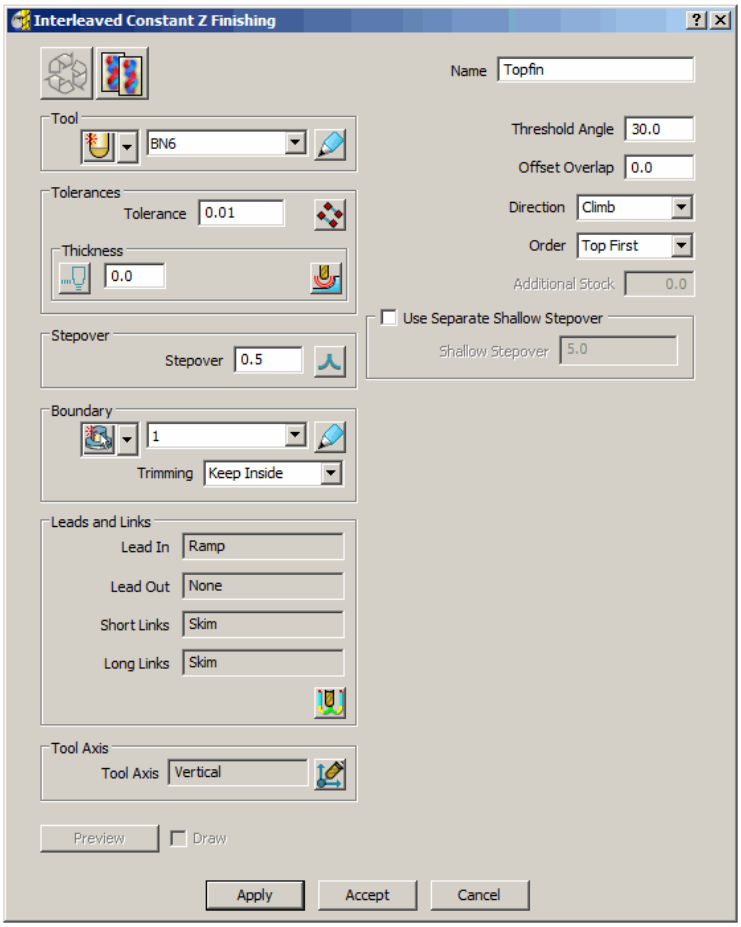




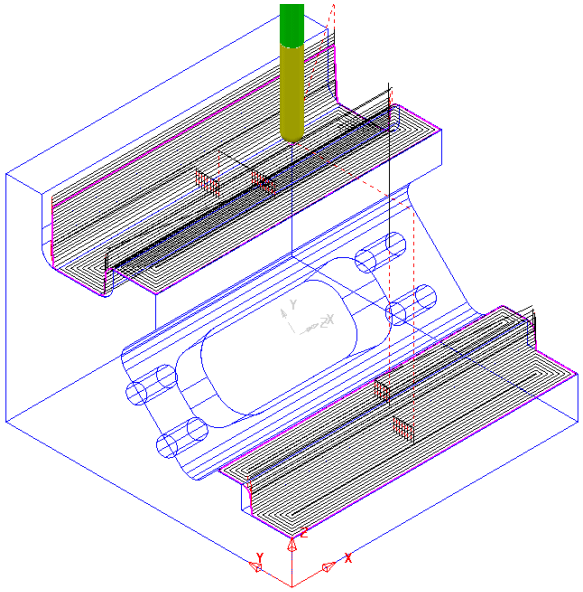
- Input data in the **Selected Surface Boundary** form exactly as shown with a ***tick*** in the box named ***Top*** and make sure that the **Boundary** has the ***Name*** **1**.
- **Apply** and when processed **Cancel**.



- Select the **Toolpath Strategies** icon  and from the **Finishing** form select the **Interleaved Constant Z** option.
- Enter the **Name** - **TopFin** along with the remaining values and settings ***exactly*** as shown below before selecting **Apply**.



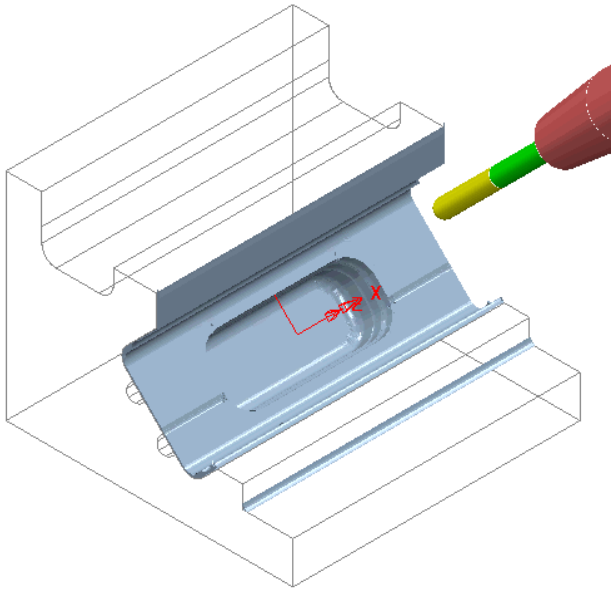
- **Cancel** the form.



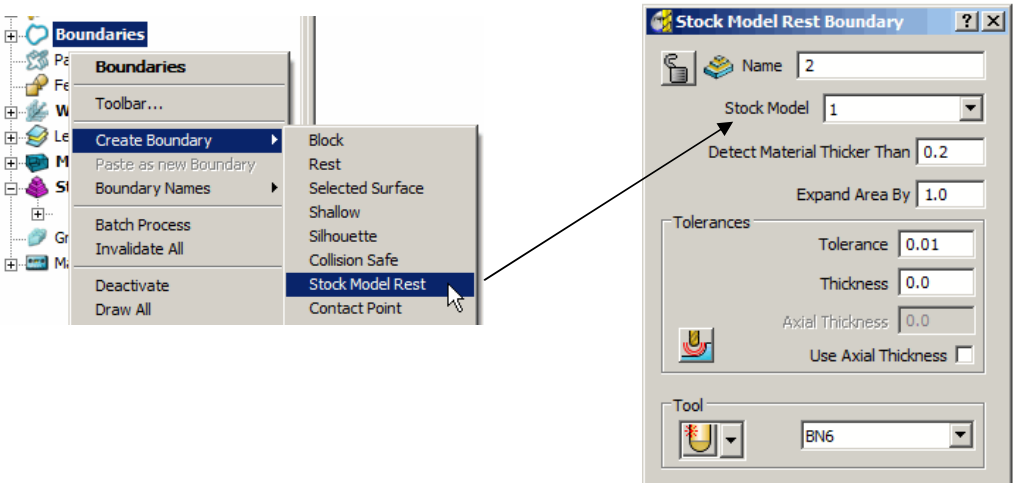
The features accessible from the top have now been finish machined. This **finishing** strategy will be added to the **Stock Model** ready for a **Stock Model Rest Boundary** to be created and applied to a 3+2 **finishing** strategy along **Workplane 2**.

- With the local **Stock Model** menu still open select **Apply - Active toolpath last**

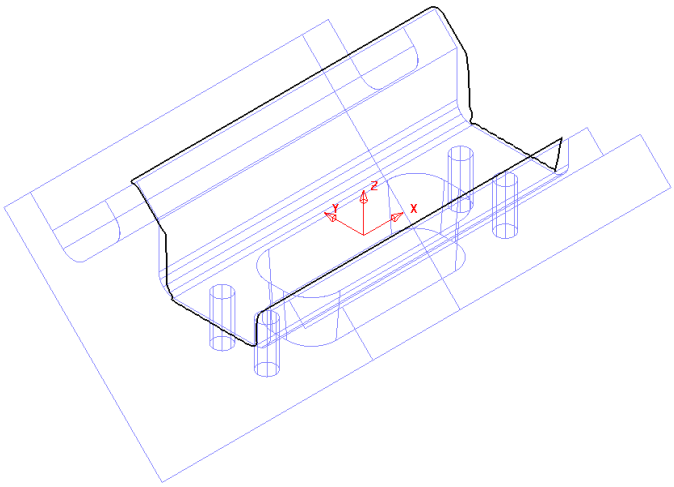
- With the local **Stock Model** menu still open, select **Show Rest Material**, followed by **Drawing Options - Shaded**, and finally **Calculate**.
- Activate **Workplane 2**.
- Select an **ISO 1** view to display the component relative to the **Workplane 2** orientation.



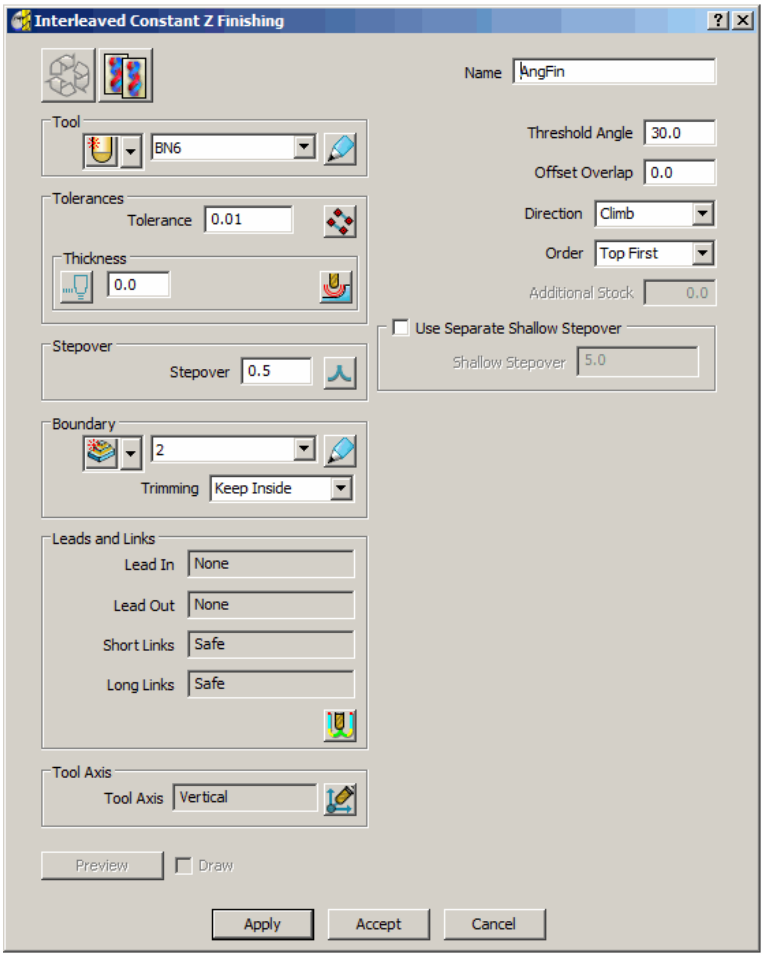
- In the **explorer** Right click over **Boundaries** and select **Create Boundary** followed by **Stock Model Rest** to open the following form.



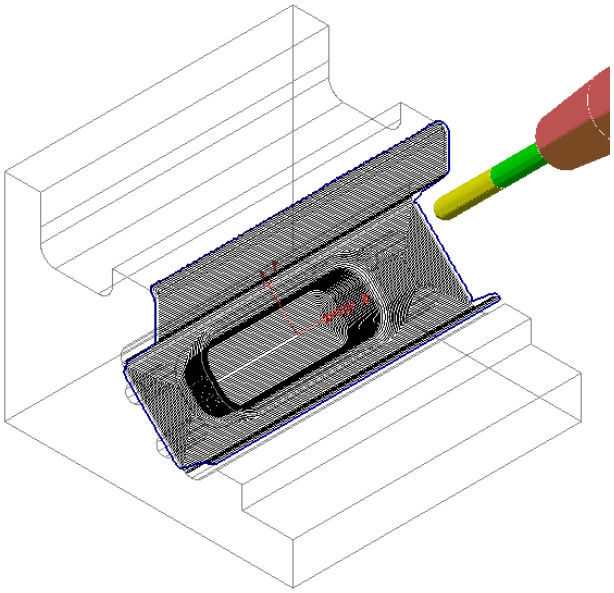
- Input data in the **Stock Model Rest Boundary** form exactly as shown and ensure that the **Boundary** has the ***Name* 2**.
- **Apply** and when processed **Cancel**.



- Select the **Toolpath Strategies** icon  and from the **Finishing** form select the **Interleaved Constant Z** option.
- Enter the **Name** - **AngFin** along with the remaining values and settings ***exactly*** as shown below before selecting **Apply**.



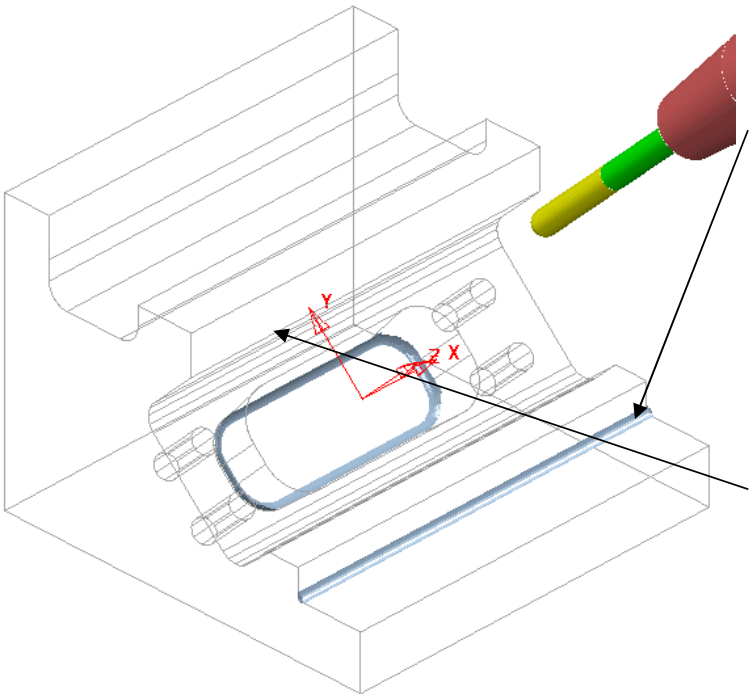
- **Cancel** the form.



The features accessible within the **Stock Model Rest Boundary** down **Workplane 2** have now been finish machined. This **finishing** strategy will be added to the **Stock Model** to confirm whether machining is now complete.

- With the local **Stock Model** menu still open select **Apply - Active toolpath Last**.

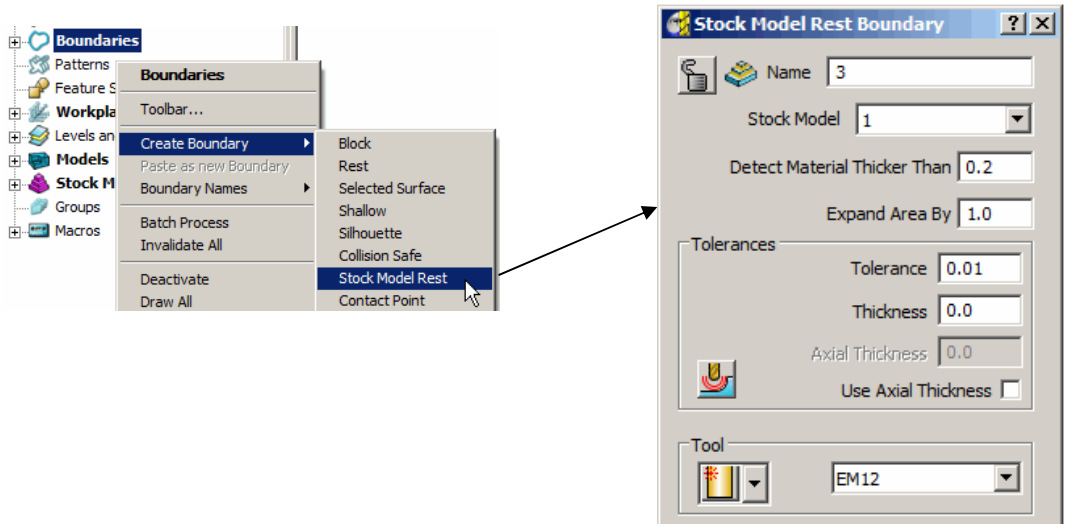
- With the local **Stock Model** menu still open, select **Show Rest Material**, followed by **Drawing Options - Shaded**, and finally **Calculate**.



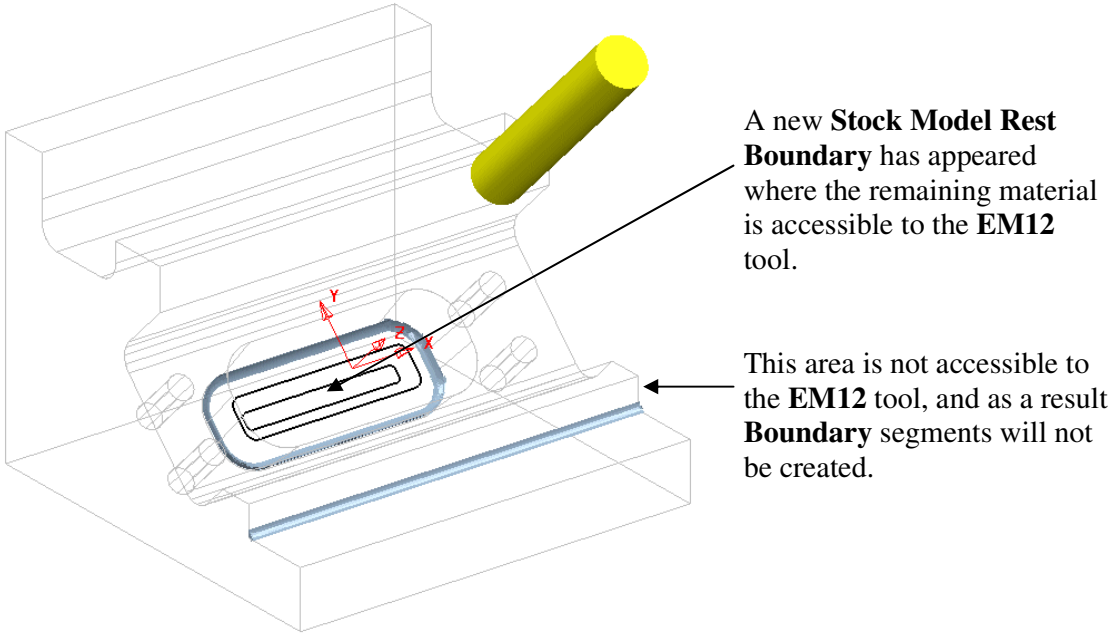
This area was not recognised as part of the **Stock Model Rest Boundary** as the material remaining in this area is totally inaccessible to the active **BN6** tool used in the calculation.

The other area is also inaccessible to **BN16** tool but was within the original **Stock Model Rest Boundary**. It is now visible since the toolpath **AngFin** has been added to the **Stock Model**.

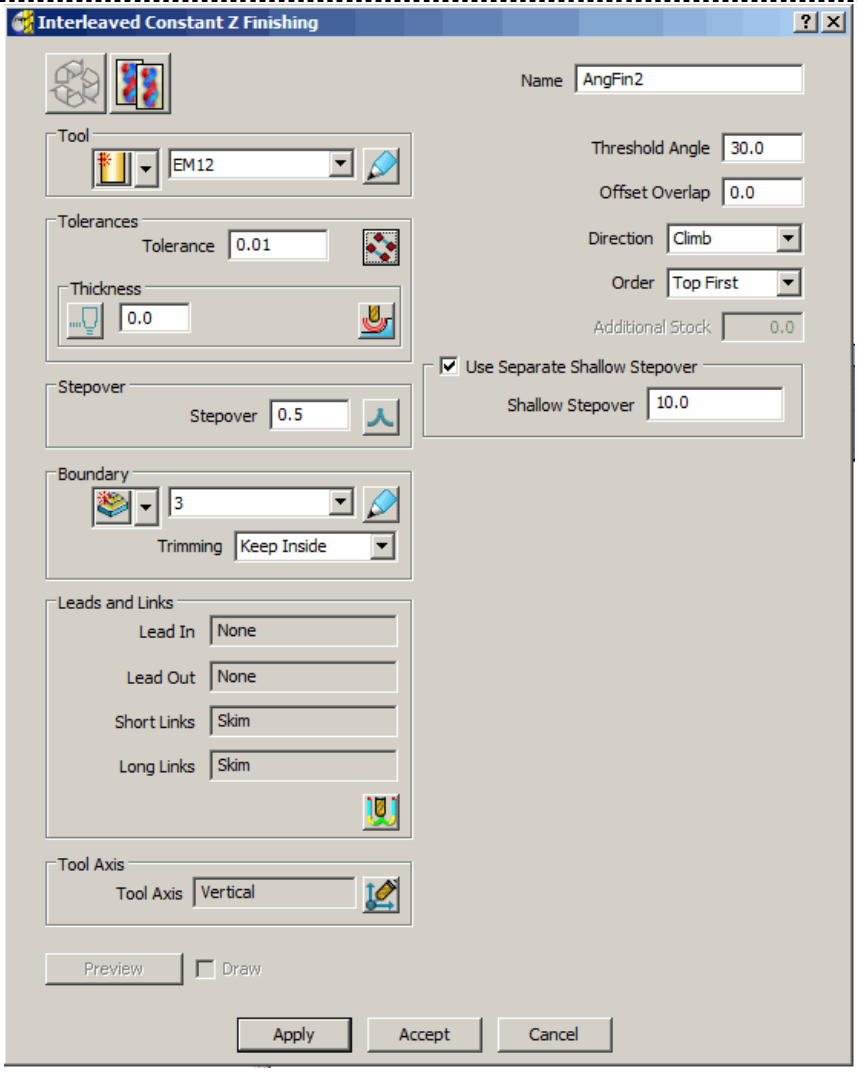
- Create a **Dia 12 End Mill** tool named **EM12**.
- In the **explorer** Right click over **Boundaries** and select **Create Boundary** followed by **Stock Model Rest** to open the following form.



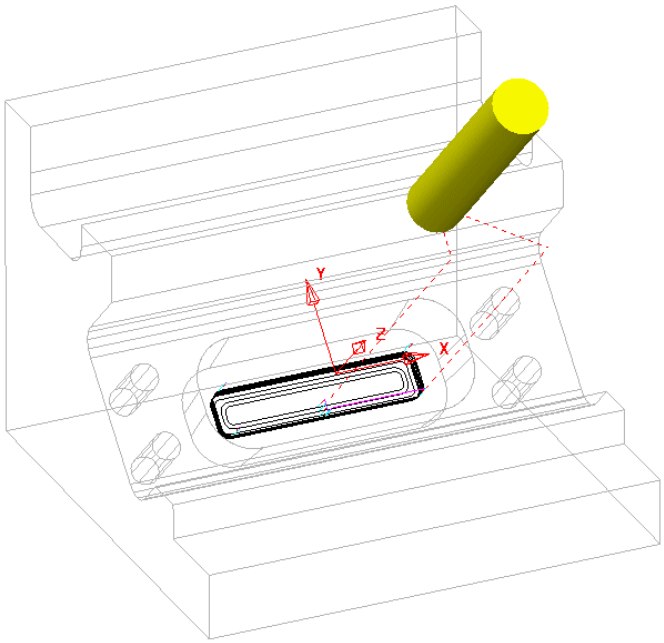
- Input data in the **Stock Model Rest Boundary** form exactly as shown and make sure that the **Boundary** has the ***Name*** **3**.
- **Apply** and when processed **Cancel**.



- Select the **Toolpath Strategies** icon  and from the **Finishing** form select the **Interleaved Constant Z** option.
- Enter the **Name** – **AngFin2** along with the remaining values and settings ***exactly*** as shown on the following page before selecting **Apply**.



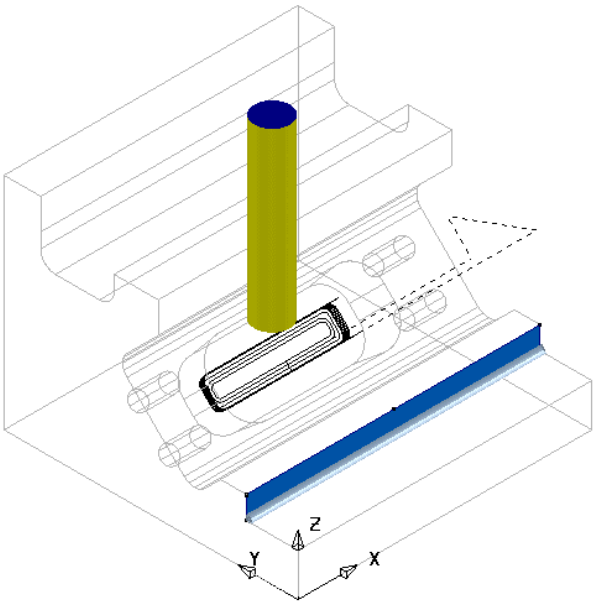
- **Cancel** the form.



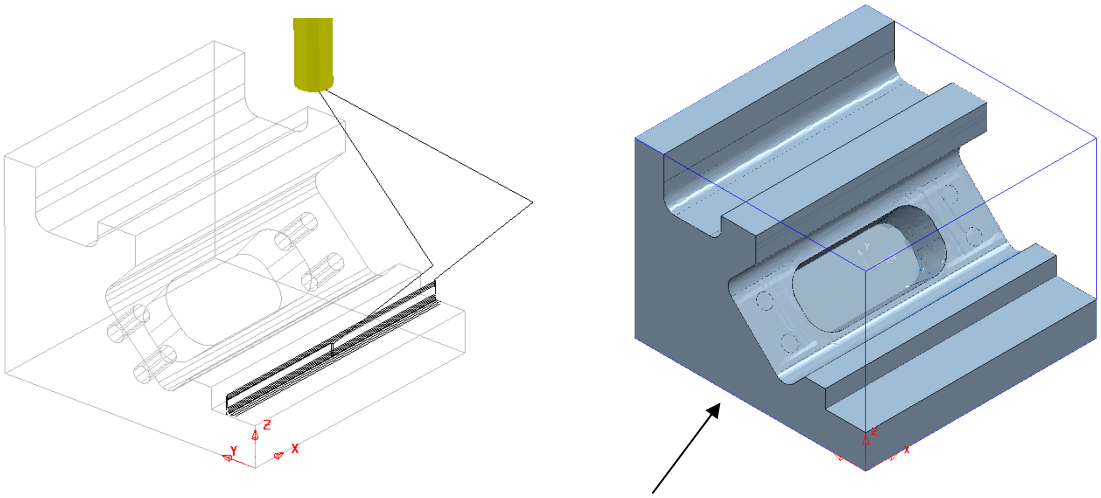
The Angled pocket is now fully machined and to confirm this, the latest toolpath will now be included in the **Stock Model**.

- With the local **Stock Model** menu still open select **Apply - Active toolpath Last**.

- With the local **Stock Model** menu still open, select **Show Rest Material**, followed by **Drawing Options - Shaded**, and finally **Calculate**.
- Activate **Workplane 1** and select an **ISO 1** view.



- Create a **Swarf Finishing** strategy named **TopSwarf** on the vertical surface as shown shaded above (Do not include the **Boundary** in the form).
- Add the new toolpath to the **Stock Model** and **Calculate** to confirm that all excess material has now been removed.

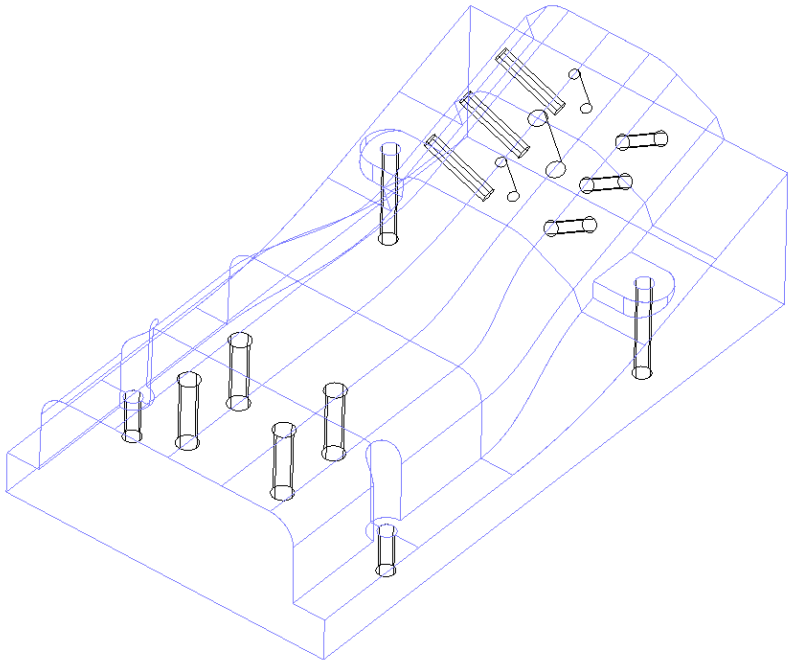


The **Stock Model** will only be visible if **Show Rest Material** is switched off.

3+2 Axis - Drilling Example (For users with MultiAxis licence)

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.

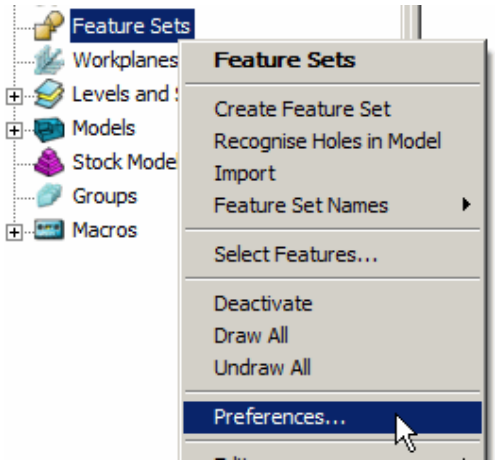
- **Delete** all entities and **Import** the model:- **D:\users\training\PowerMILL_Data\five_axis\drill_5axis\drill5ax_ex1**



- Do Not define a material **Block** and if one already exists, delete it (**Blue Cross** at top right corner of form). 

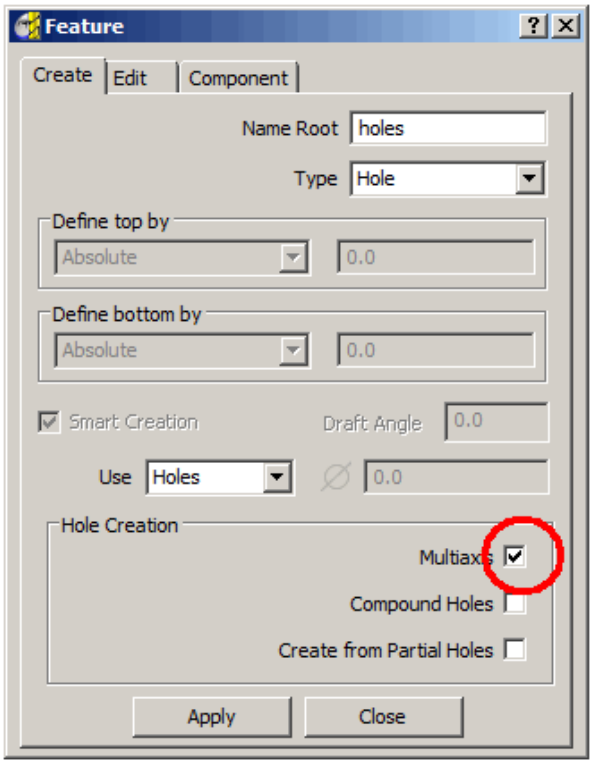
Any cylindrical surfaces within the selection will automatically be recognised as a **Hole Feature**. In this example, with no **Block** defined, the **Hole Features** will be arranged with the **top** at the end of **maximum Z height**.
If however, a **Block** is pre-defined, the orientation of an individual **Hole Feature** occurs with the **top of the hole** being nearest to the **upper Z** or **lower Z**, face of the material **Block**.
Note: It is possible, if required, to **Reverse** the **Holes** in a **Feature Set** using the local **Edit** options combined with dynamically selecting the affected **Hole Features**.

- Reset the **Rapid Move Heights**  (**Safe Z**, **Start Z)** and then, set the **Start\End Points**  to ***Use*** **- Block Centre Safe**.
- **Select** all the surface data in the graphics window and then **right** mouse click **Feature Sets** in the **PowerMILL Explorer**.
- **Select** the option **Preferences**.



This will open the **Feature Form**.

- **Create** the **Feature Set** entering the values into the form **Exactly** as shown.

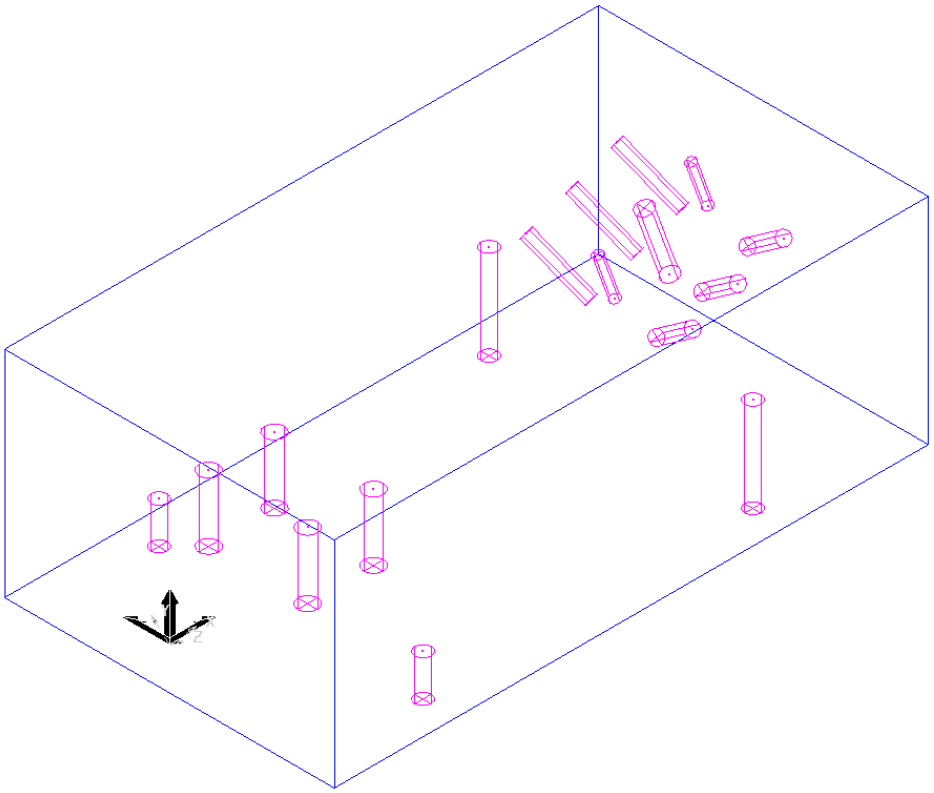


Once the option ***Type*** **Hole** has been selected the **Multiaxis** option must be **ticked** for **5 Axis drilling** to operate (All selected holes including those at different orientations will be input into the same **Feature Set**).

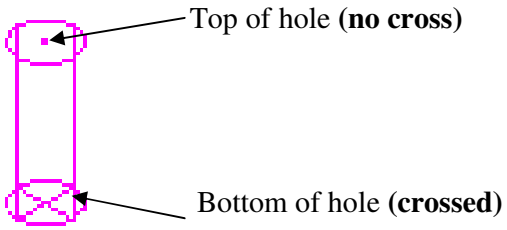
- **Apply** and **Close** the form.

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

- **Undraw** the model to view the newly created **Features**.



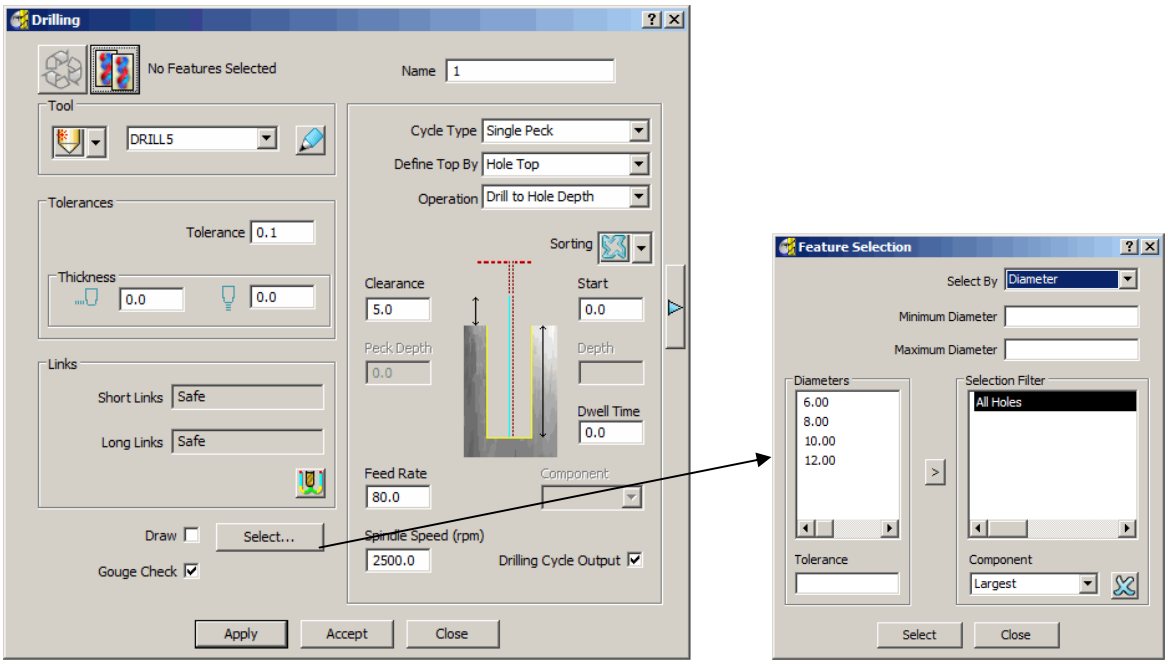
Hole features are defined with a specific top and bottom.



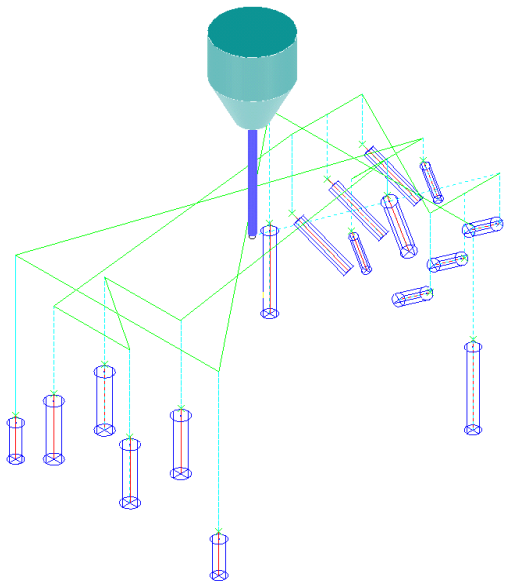
To reverse one or more **Hole Features**, select them and **click** over one (or more) with the **right mouse button** to open the local menu and select **Edit - Reverse Holes**.

- Create a material **Block - *Defined by* - Box** to the model limits.
- **Create** a **5mm Drill** of length **60**.
- **Add** a **shank** component ***Upper\Lower* dia 5 *Length* 30**.
- **Add** a **holder** component ***Upper Dia* 50 *Lower Dia* 30 *Length* 30 *overhang* 75**.
- **Add** a **holder** component ***Upper Dia* 50 *Lower Dia* 50 *Length* 30**.

- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Drilling** form.
- In the **Drilling form** select the option **Drilling**.
- **Rename** the toolpath **DRILL5**.
- In the **Drilling** form click the **Select** tab to open the **Feature Selection** form.



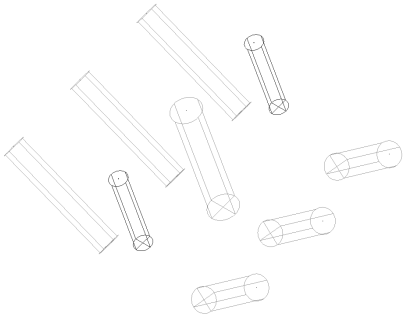
- By clicking the **Select** tab in the **Feature Selection** form all the **Hole Features** will be selected in the **Active - Feature Set**.
- **Simulate** the toolpath.



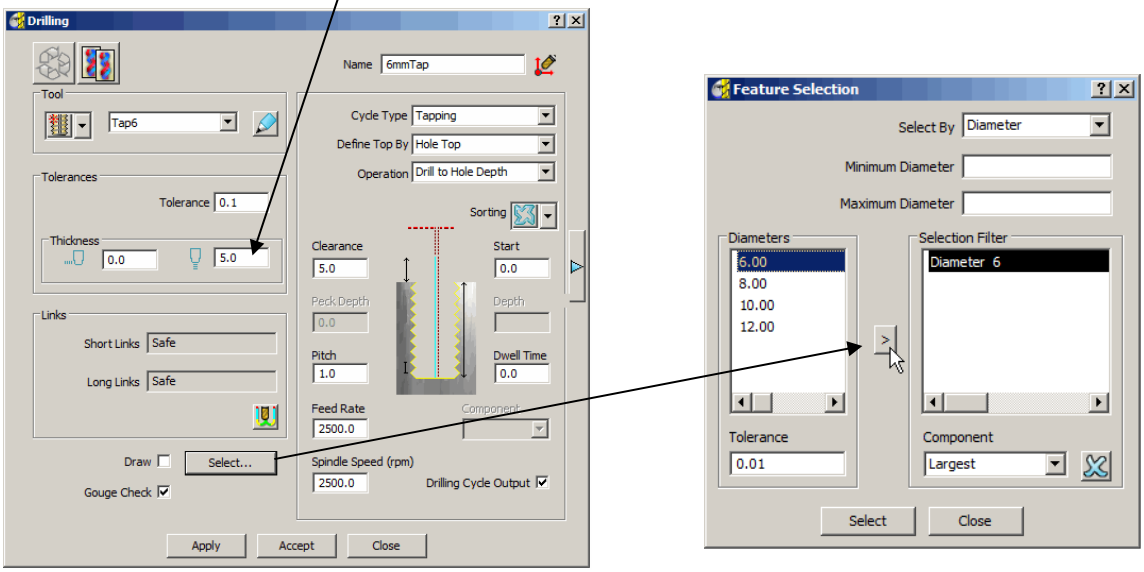
The **Multiaxis** options are automatically recognised enabling the user to create a single **Feature Set** from components that exist at different tool alignments and machine them in one go. Without the licence the **Recognise Holes in Model** option can be applied from the **Feature Set** menu to create separate **3+2 - Hole Features**. This command segregates the **Features** into separate **Feature Sets** each with it's own **Workplane**, to provide the necessary **3+2 - Z Axis** alignment.

The two **6mm Hole Features** are to be **Tapped**. The point angle of the **5mm Drill** has left a conical shape at the bottom of the holes. When the holes are **Tapped** it will be necessary to stop short within the full diameter range by applying a suitable **Axial Thickness** value.

- **Create** a **6mm Tapping Tool** of length **25**.
- **Add** a **Shank**, ***Upper - Dia*** **4**, ***Lower Dia*** **4**, ***Length*** **40**
- **Add** a **Holder**, ***Upper- Dia*** **30**, ***Lower Dia*** **30**, ***Length*** **20**, ***Overhang*** **60**.
- **Select** the two **6mm Hole Features** in the **Graphics Window**.

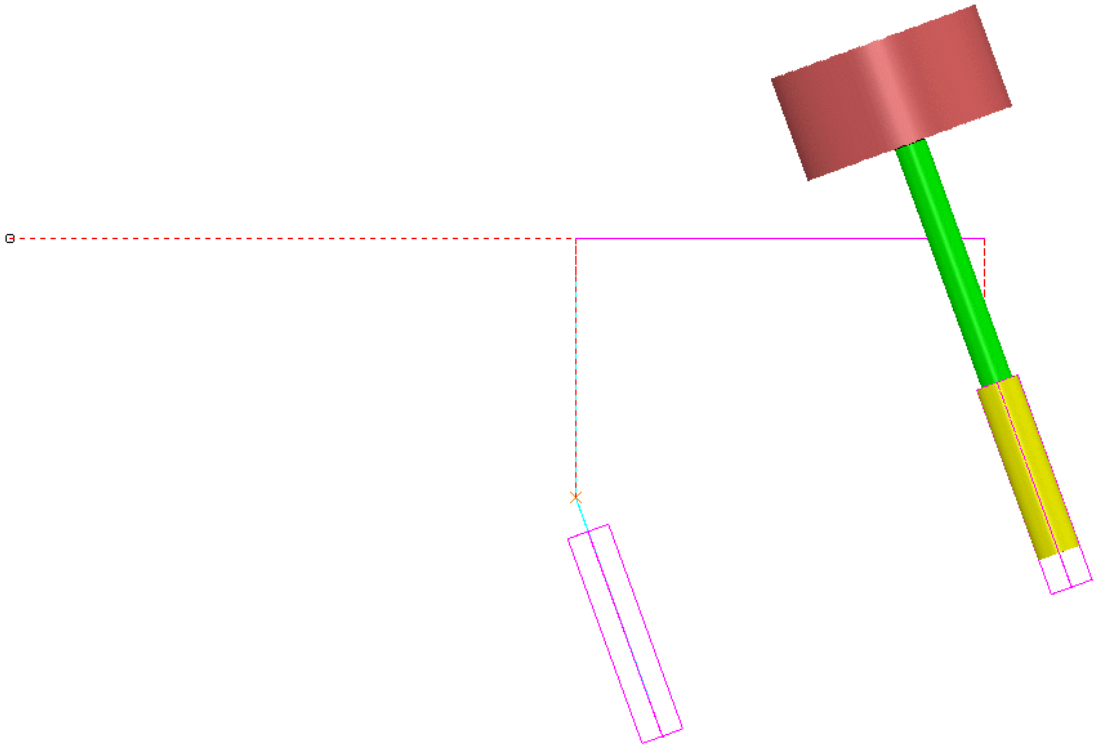


- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Drilling** form.
- In the **Drilling form** select the option **Drilling**.
- **Rename** toolpath **6mmtap**.
- Set ***Cycle Type*** - **Tapping**, ***Operation*** - **Drill to Hole Depth**, and ***Pitch*** - **1mm**.
- **Input** an **Axial Thickness** value of **5mm**.



- **Select** the two **Dia 6** ***Hole Features***.
- **Apply** and **Close** the form to create the toolpath.

- **View** the model along the **-Y** axis.
- **Right click** over the **6mmtap toolpath** in the **Explorer** window and select **Simulate from Start**.
- **Left click** in the graphics window and use the **Right\Left Cursor keys** to step the toolpath one move at a time.



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

2. Five Axis Tool Alignment

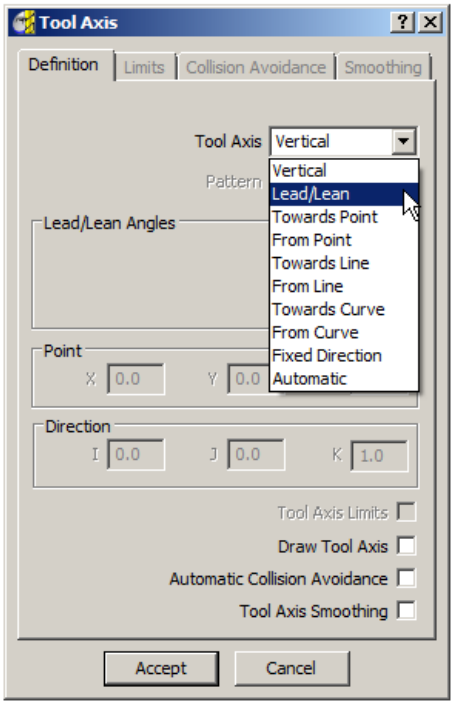
Introduction

For **5-Axis** applications where the machine tool head and\or table, rotates simultaneously with the linear axis movements, **PowerMILL** provides a range of suitable **Tool Alignments** and **Machining Strategies**.
5-Axis machining enables components normally requiring a series of **3-Axis** operations to be machined in one set-up. Tools can be re-aligned using **5-Axis** control to provide access to the base of steep or undercut features, which would otherwise inaccessible down the **Z-Axis**.
In **5-Axis** applications, as well as the normal, default gouge checking, a range of options exist to ensure that no part of the head, spindle or tooling clash with the component between different strategies. In all cases it is essential to carry out a thorough visual inspection of the results.

Five Axis Tool Alignment and Machining Options

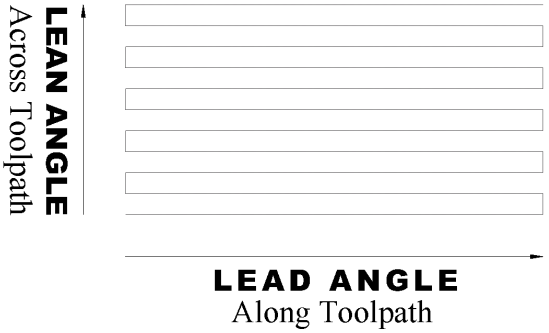
By default the **Tool Axis** alignment in **PowerMILL** is set to **Vertical** for **3-Axis** applications and other options will only be available to users with a multiaxis licence.

The **Tool Axis Direction form** is accessed via the **Tool Axis** icon  located in the **Main** toolbar or directly from supported **Machining Strategy** forms. Note: some strategies only support multiaxis **Tool Axis** alignments when operating with **Ballnose** or **Spherical** tools.

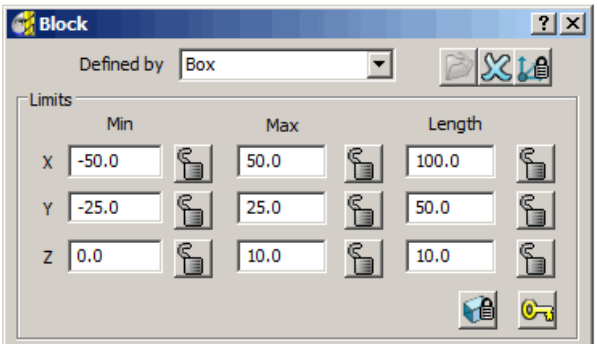


Lead\Lean

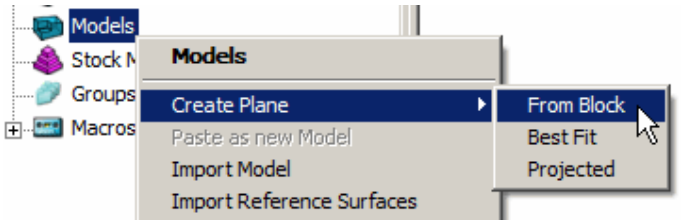
Lead allows the tool to be aligned to a specified angle **along** the toolpath direction and **Lean** a specified angle **across** the toolpath direction. If both angles are zero the tool will be aligned along the **normal** of the toolpath. The **normal** of the toolpath is the direction along which it was originally, projected onto the surface data during creation. For **Pattern finishing** this will always be vertical and for **Projection Finishing** it will vary depending on the defined projection, directional options.



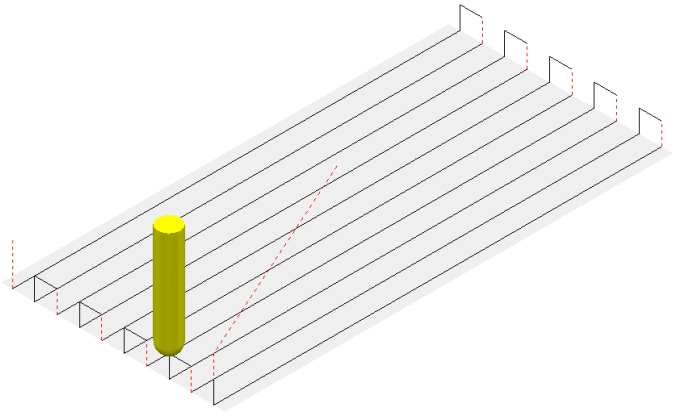
- **Delete** all and **Reset** forms.
- **Create** a **Block** with the manually input values displayed in the form below.



- Reset the **Rapid Move Heights** and **Start and End Point** forms.
- **Right Click** the **Models** option in the **Explorer Window** and **Create** a **Plane** from **Block** at a **Z** limit of **0**.

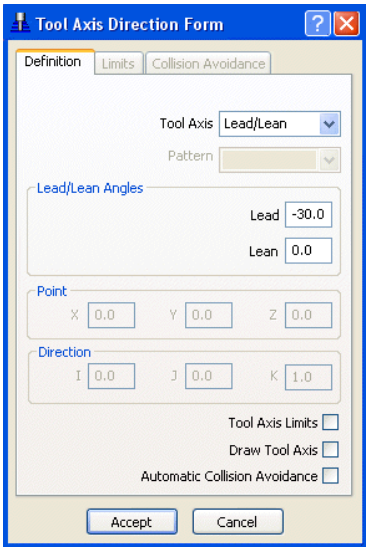


- **Create** a **Dia 5 Ballnose** tool of ***Length*** **25** and ***Rename*** **BN5**.
- **Create** a **Raster Finishing** Strategy, ***Rename*** **- Raster Vertical**, and set ***Tolerance*** **0.02** ***Thickness*** **0** ***Stepover*** **5** ***Angle*** **0** ***Style*** - **Two Way** ***Short Links*** - **Skim**.
- **Apply** the toolpath and **Cancel** to close the form.
- **Simulate** the Toolpath.

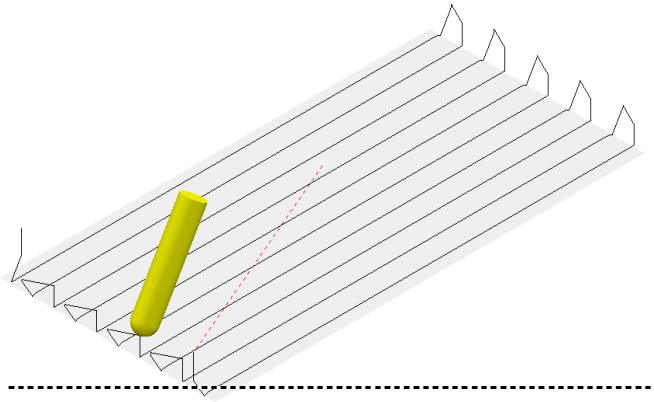


A **Raster** toolpath has been created with the tool aligned vertically to the plane.

- **Right Click** the Toolpath **Raster Vertical** in the **explorer** and select **Settings** to open the toolpath form.
- Make a **Copy**  of the toolpath and rename **Raster Lead@-30**.
- **Select** the **Tool axis** icon  to open the **Tool Axis Direction Form**.
- **Define** the **Tool Axis** as **Lead\Lean** with the **Lead** angle set to **-30**.

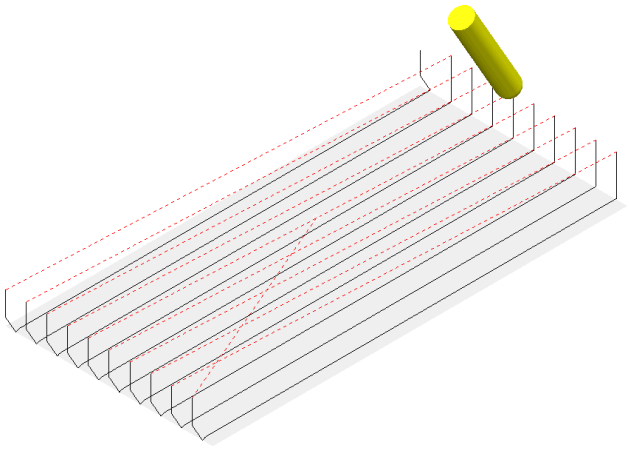


- **Accept** the **Tool Axis Direction Form**, **Apply** the toolpath and **Cancel** to close the form.
- **Simulate** the Toolpath.



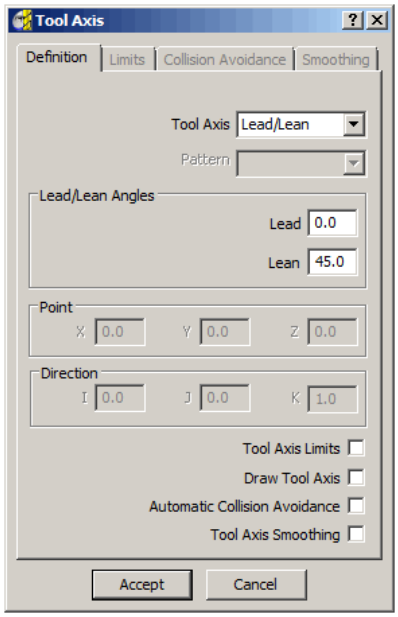
A raster toolpath has been created with the tool axis direction set to **Lead -30°** Along the Toolpath. Using the **Two Way** option the tool axis direction will alternate at the end of each pass.

- **Right Click** the Toolpath **Raster Lead@-30** in the **Explorer Window** and select **Settings** to open the toolpath form.
- **Re-cycle**  the toolpath and change **Style** from **Two Way** to **One Way**.
- **Apply** the toolpath and **Cancel** to close the form.

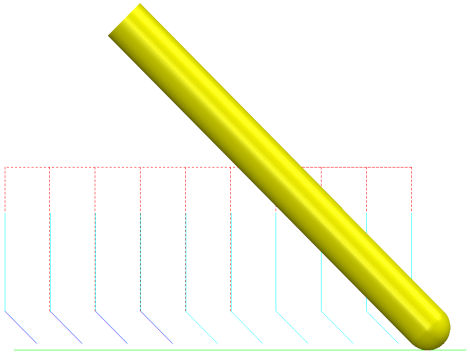


With the **Style** set to **One Way** the tool axis direction remains constant.

- **Right Click** the Toolpath **Raster Lead@-30** in the **Explorer Window** and select **Settings** to open the toolpath form.
- Make a **Copy**  of the toolpath and rename **Raster Lean@45**.
- **Select** the **Tool axis** icon  to open the **Tool Axis Direction Form**.
- **Define** the **Tool Axis** as **Lead\Lean** with the **Tool Lead Angle** set to **0** and **Tool Lean Angle** of **45**.



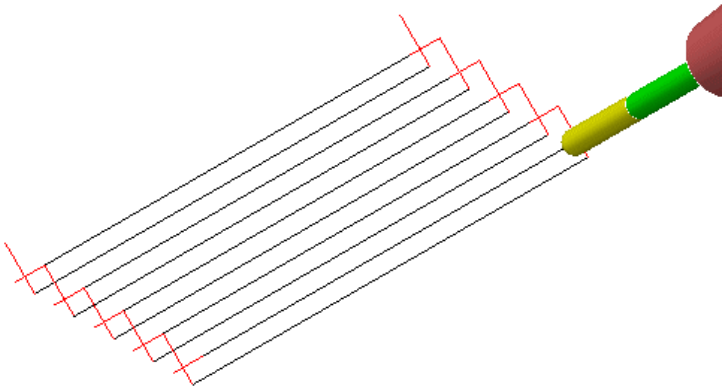
- **Accept** the **Tool Axis Direction Form**, **Apply** the toolpath and **Cancel** to close the form.
- **Simulate** the Toolpath.



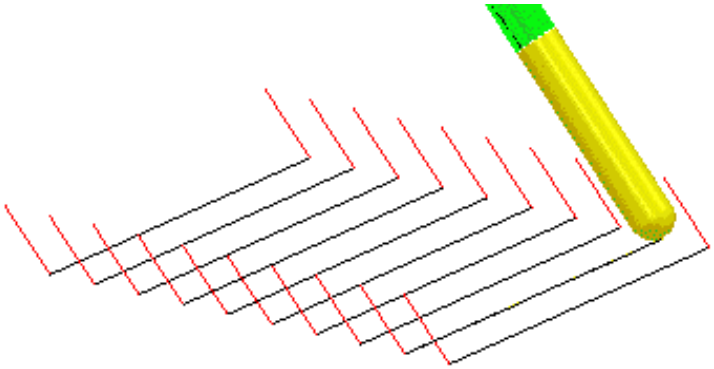
View from left -X

A **Raster** toolpath has been created with the **Tool Axis Direction** set to **Lean 45°** Across the Toolpath.

If a ***two way*** strategy had been directly applied in the form, the tool axis will lean in the opposite direction across alternate tooltracks.

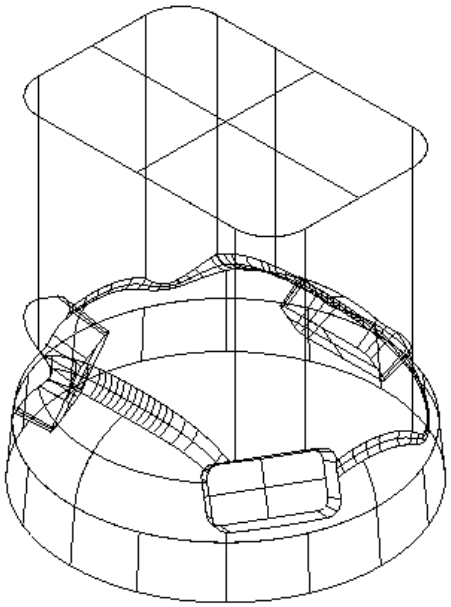


It is possible to create a ***two way*** strategy with a ***constant lean direction*** by retrospectively, editing a ***one way*** toolpath.
The **one way** strategy is modified in the toolpath **Edit - Reorder** options by clicking on the **Alternate Directions** icon  . The original ***tool axis alignment*** will remain unaffected.



Example2

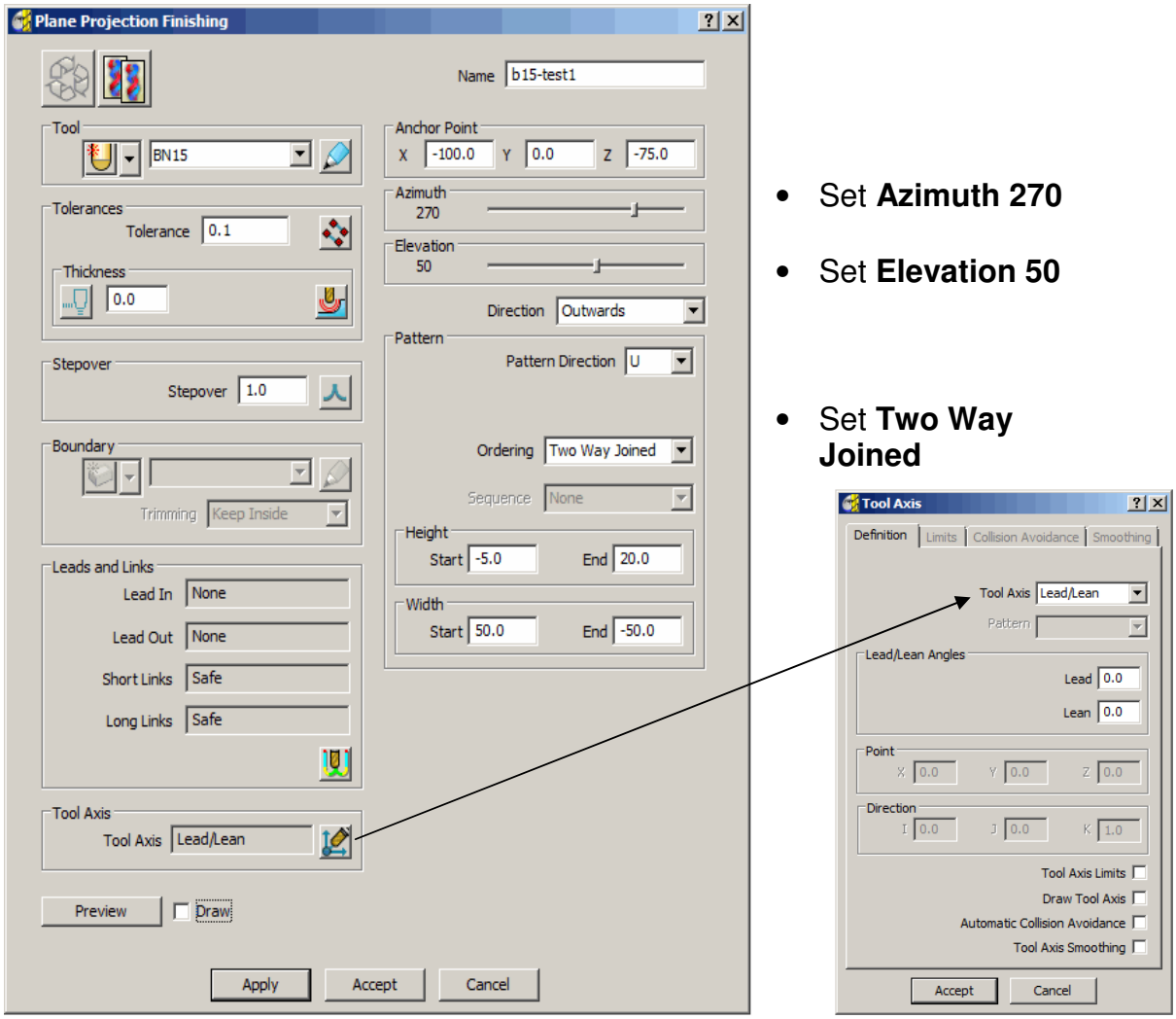
- **Delete all** and **Reset** forms.
- **Import** the **Project** saved earlier during **Chapter 1** from:- **D:\users\training\COURSEWORK\PowerMILL-Projects\3+2example**.



- Define a **15mm** diameter **Ball Nose** cutter **BN15**.
- Check the **Cylindrical Block** definition is **Locked** to the **Global** coordinates.
- **Activate** the **workplane - ztop175_A**.
- Reset **Safe Z** and **Start Z**.
- In the tool **Start and End Point** form set ***Use* - Absolute** with the positional ***Coordinates* X-100 Y0 Z10** for both the **Start** and **End Points**.
- In the **Main Toolbar** set the **Tool Axis**  - **Lead\Lean** values both set to **0**. This will create a **tool alignment** relative to the direction used to project the machining strategy onto the model.
- Set **Leads\Links**  as follows:-

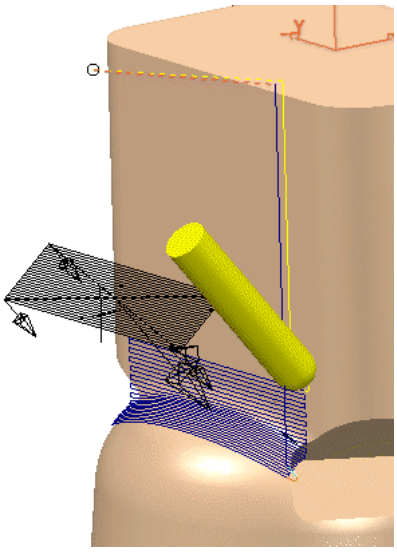
Zheights:	------ **Skim 15**	**Plunge 5**
Lead In\Out:	------ **Vertical Arc:**	**Angle 90** **Radius 6**
Links:	------ **Short\Long\Safe:**	**Skim**

- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Finishing** option.
- Enter the values into the **Plane Projection Finishing** and **Tool Axis** forms exactly as shown on the following page and **Apply**.



- Set **Azimuth 270**
- Set **Elevation 50**
- Set **Two Way Joined**

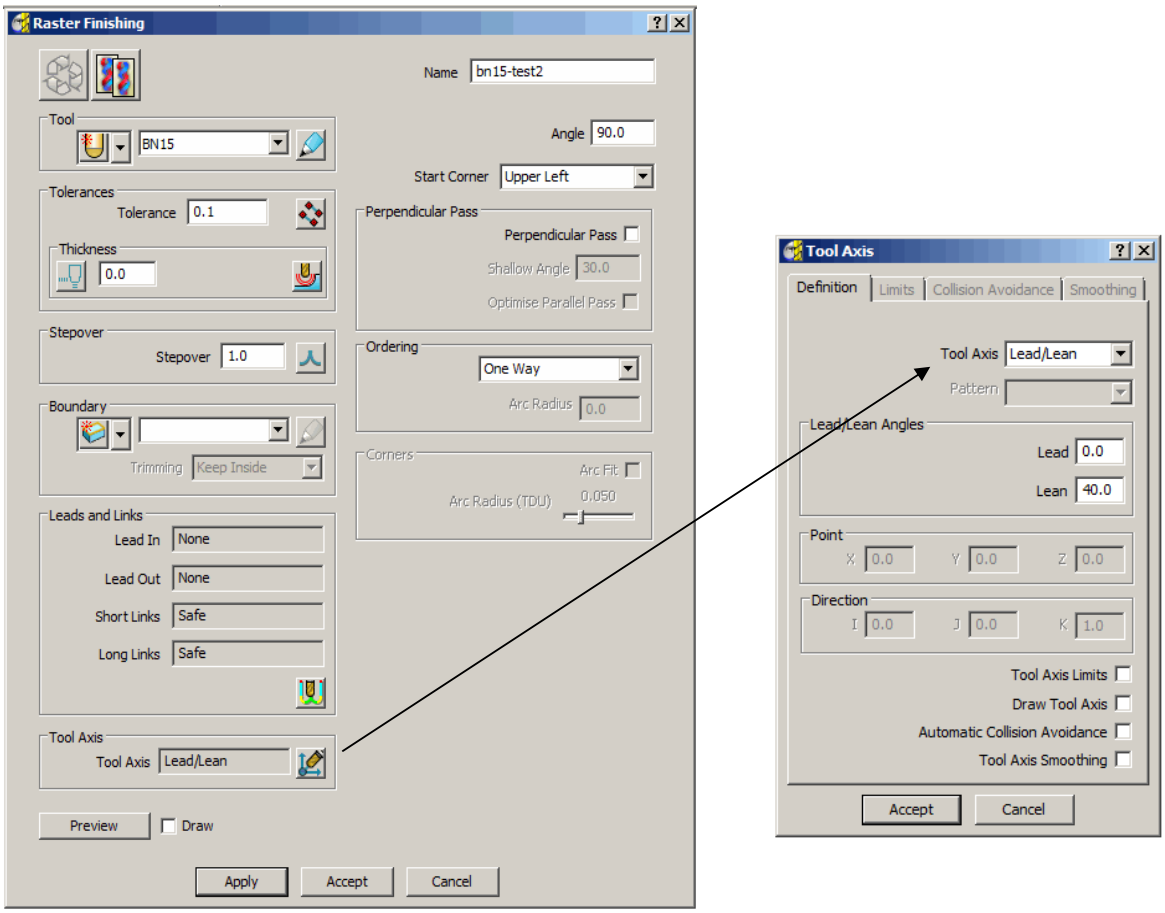
- **Simulate** the toolpath and observe the associated tool alignment.



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 joined up strategy is a feasible option.

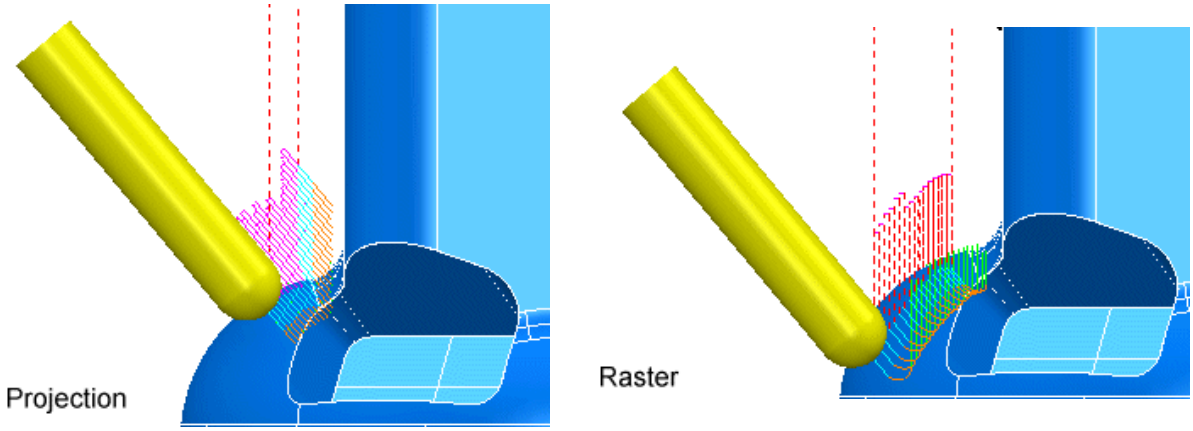
To compare the effect of applying a different machining strategy the same area of the component will be machined using **Raster Finishing**. This time the **Tool Axis** will be aligned with a **Lean** of **40** degrees relative to the downward projection of the **Raster Finishing** strategy.

- Define a material **Block** to the **Max\Min Limits** of the **Model** and edit the following values as shown:- **Xmin -70 Xmax -57.5 Ymin -50 Ymax 50**
- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Finishing** option.
- Open the **Raster Finishing** and **Tool Axis** forms and enter data exactly as shown below and **Apply** and then **Cancel**.



The resultant toolpath starts at the lower corner and progresses towards the centre using a climb milling action (**One Way**). It would not be feasible to use **Two Way** strategy due to the applied **Lean Angle** (40) being controlled by the direction of the toolpath.

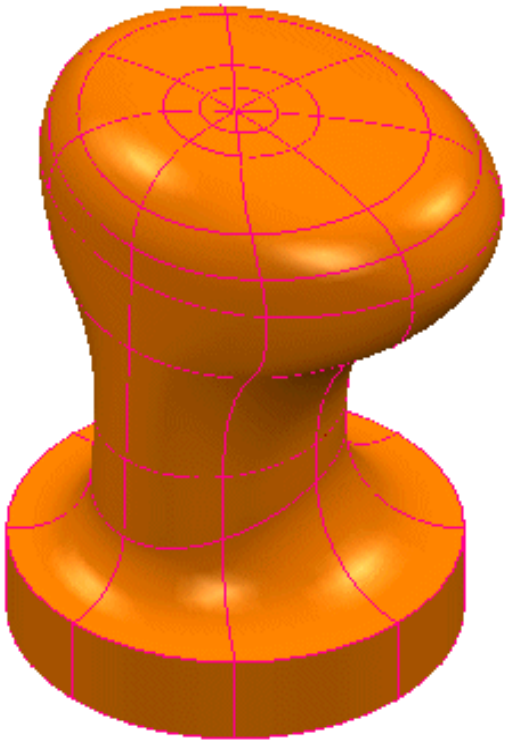
- View along the Y-Axis and **Simulate** both toolpaths in turn to compare the results of the **lead\lean** option. Note; the tool alignment is the same for both toolpaths due to a suitable **Lean** value of **40** being applied to the **Raster** strategy.



Lead\Lean is designed for unidirectional toolpaths the main application being to maintain a suitable angle of the **Tool Axis** away from steep features as well as the machine tool table. The lower part of the component form in the next example is an ideal application for applying a suitable **Lean** value using **Lead\Lean - Tool Axis** alignment.

Example 3

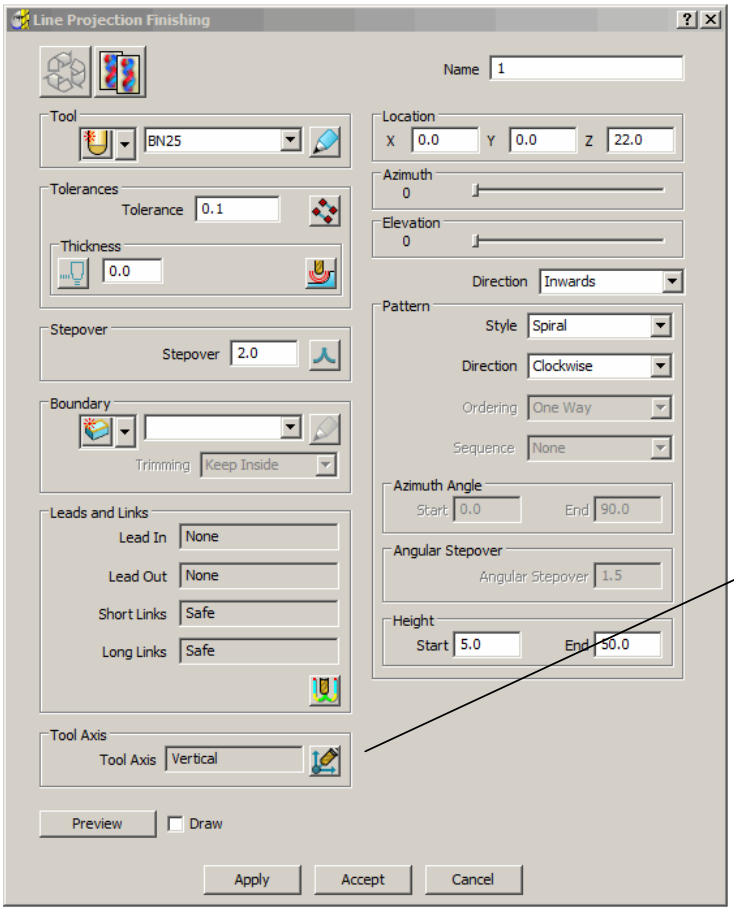
- **Delete** all and **Reset** forms.
- **Import** the model **joint5axis.dgk** from the directory **D:\users\training\PowerMILL_Data\five_axis\joint_5axismc**.



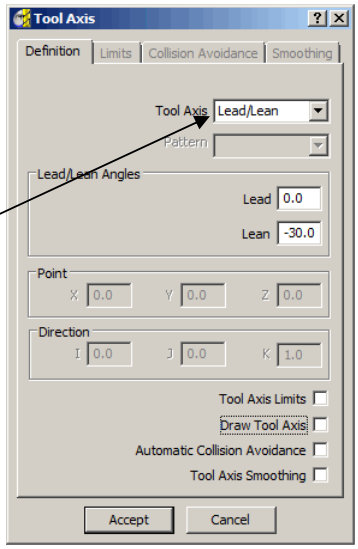
- Create the material **Block** to component size and expand by **15mm** in **X** and **Y** only.
- Define a **25mm** diameter **Ball Nosed** cutter (**bn25**).
- Reset **Safe Z** and **Start Z**.
- For the **Start Point** ***Use*** **- Block Centre Safe** and **End Point** set ***Use*** - **Last Point Safe**.
- Modify **Leads\Links**  as follows:-

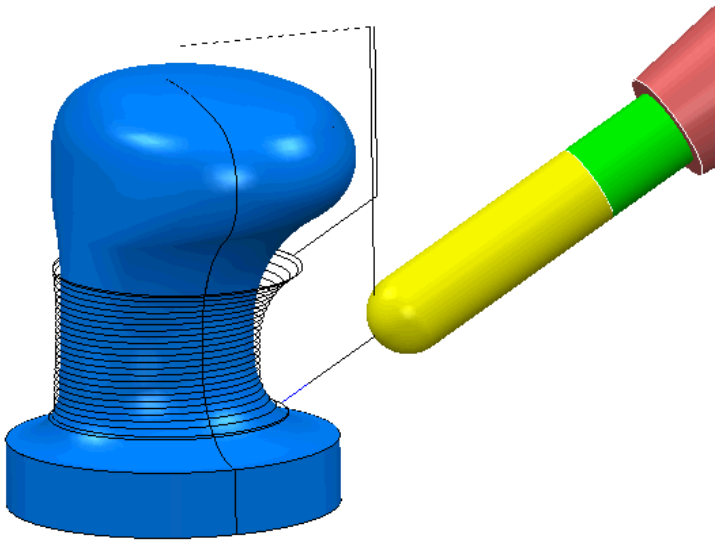
  Zheights: **Skim 45 Plunge 10** **Links:** **Skim**

- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Finishing** option.
- Open the **Line Projection Finishing** and **Tool Axis** forms and enter data <u>exactly</u> as shown and **Apply**.



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

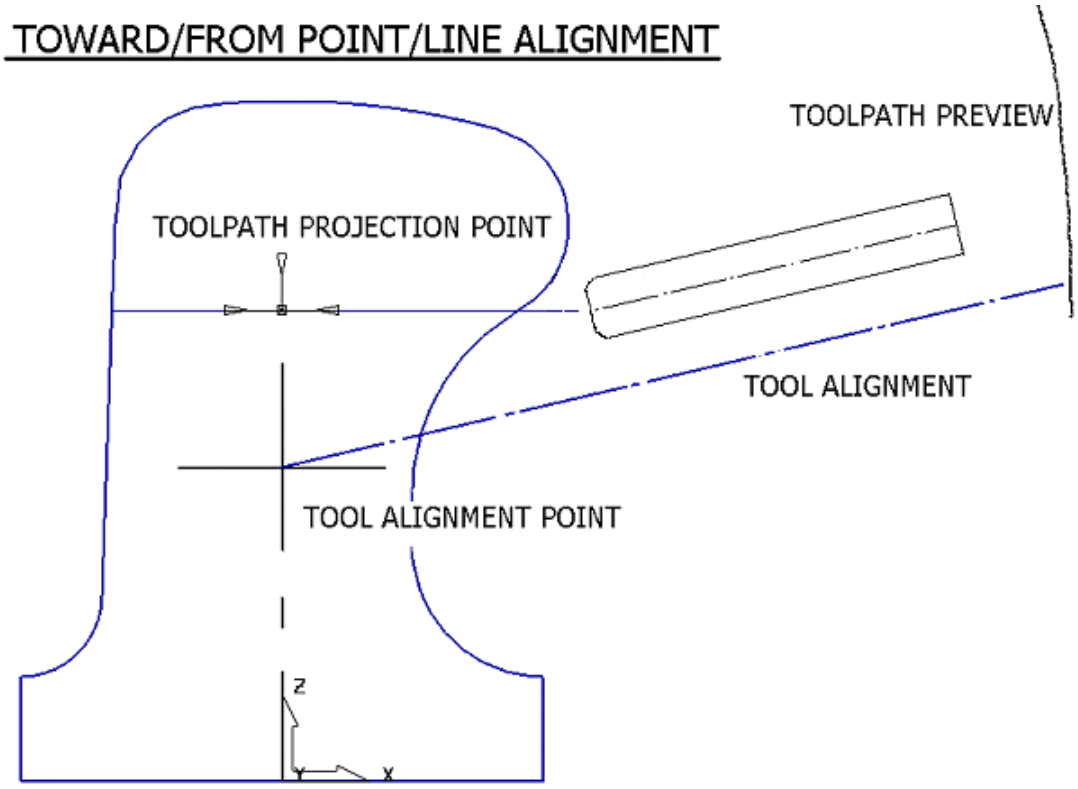




Note: The next section continues with the machining of the upper part of the component.

Toward \ From Point

These options allow the **Tool Axis** alignment to be based on a user-defined point during the generation of toolpaths. The actual alignment is taken relative to the **Preview** pattern for the toolpath and not the actual toolpath. **Toward Point** is suitable for aligning to external forms (Upstands) while **From Point** is suitable for aligning to internal forms (Pockets). The upper part of the existing component is an ideal application for the **Toward Point** tool alignment.

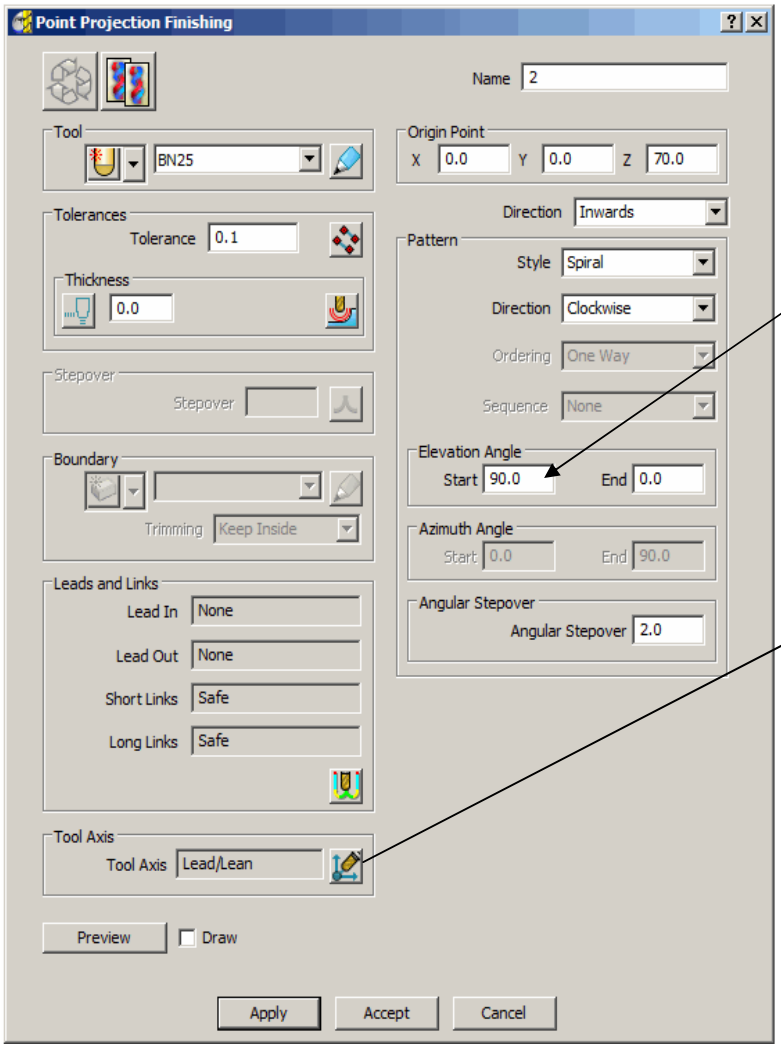


Note:- The same alignment condition (above) applies to **Toward \ From Line** (page 8)

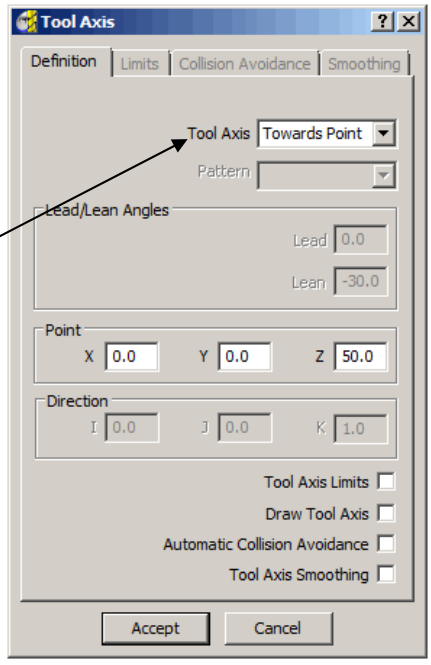
- Set **Leads\Links** as follows:-

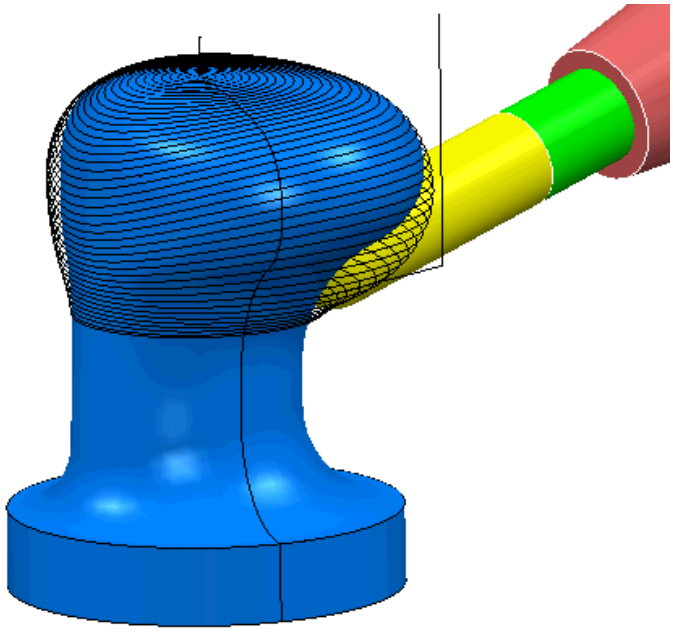
Zheights:	**------ Skim 45**	**Plunge 10**	
Lead In;	**------ Horizontal Arc**	**Angle 90**	**Radius 6.0**
Lead Out:	**------ Vertical Arc**	**Angle 90**	**Radius 6.0**
Extensions;	**------ Inwards\Outwards**	**Extended Move**	**Distance 30**
Links;	**------ Skim**		

- Select the **Toolpath Strategies** icon and select the **Finishing** tab located in the strategies form.
- Open the **Point Projection Finishing** and **Tool Axis** forms and enter data exactly as shown on the following page and **Apply**.



- Set **Elevation Angle:- Start 90** & **End 0**





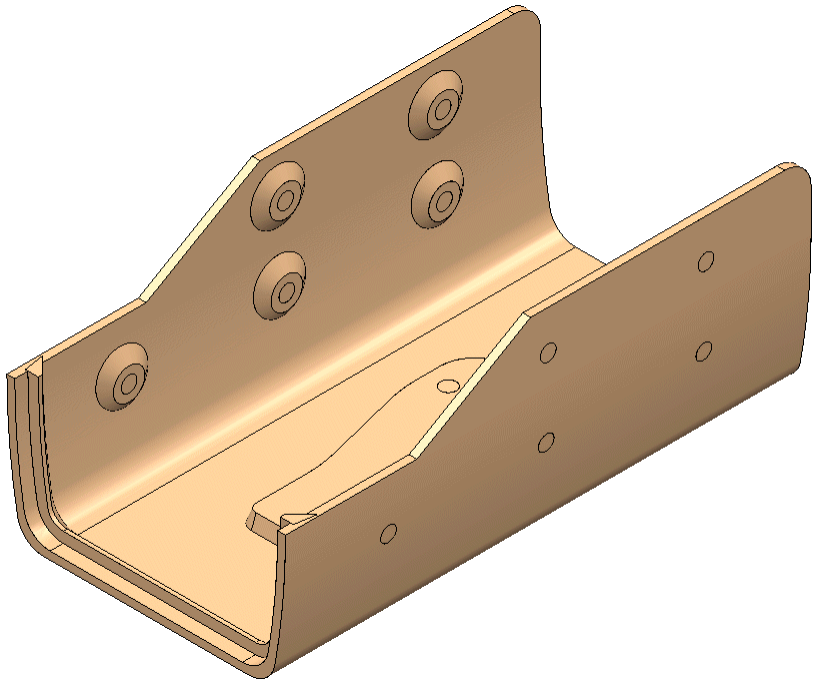
Note the **Tool Alignment** point is approximately 10mm below the **Projection Finishing** focal point to ensure that the spindle axis remains at an elevation angle to the machine tool table during machining hence avoiding collision.
The resultant machining is illustrated after suitable gouge removal has been applied to the start and end of the toolpath.

Toward\From Line

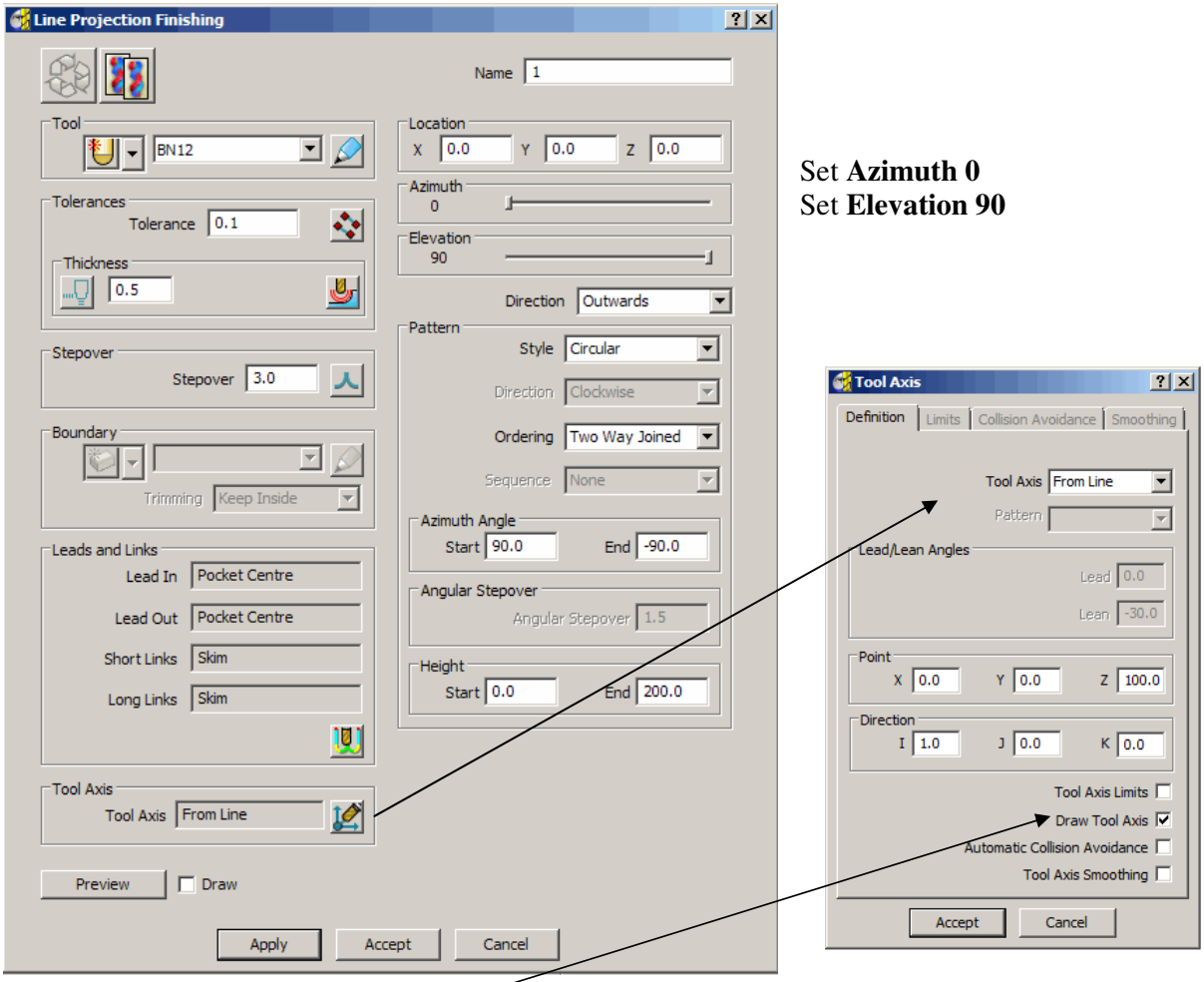
These options allow tool axis alignment to be based on a user-defined line, specified by a Vector direction through a suitably positioned XYZ coordinate. In this example the actual alignment is towards the **Preview** pattern for the toolpath and not the final toolpath. **Toward Line** is suitable for aligning to external forms (Upstands) while **From Line** is suitable for aligning to internal forms (Pockets).

- From the main pulldown menus select **File -Delete All**.
- **Import** the **model**:-

D:\users\training\PowerMILL_Data\five_axis\Casing from-line-model.dgk.

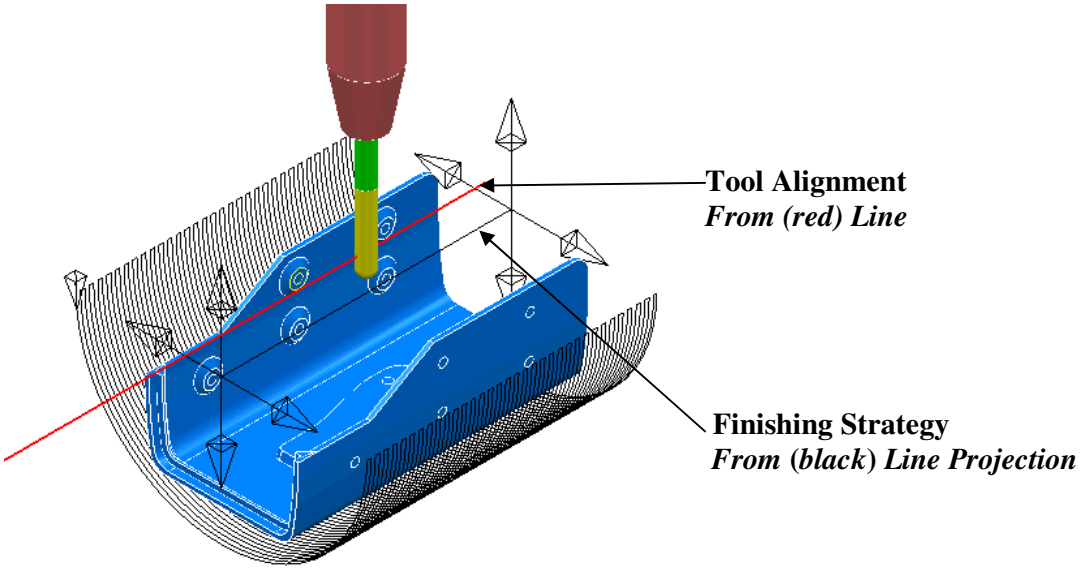


- Create a **Dia 12mm Ball Nose** cutter of **Length 55** with **Dia 12 Shank** of **Length 40**, 1st **Holder** component **Lower Dia 25** - **Upper Dia 40** - **Length 40**, 2nd **Holder** component **Upper\Lower Dia 40** - **Length 60**, **Overhang 90**.
- Define the **Block** (use **Box**) to the component limits.
- **Reset Safe Heights**.
- For both the **Start Point** and **End Point** set ***Use*** - **Block Centre Safe**.
- Set all **Leads** and **Extensions** to **None**, **Zheights – Skim distance** and **Plunge distance** to **5**, **Links - Short - Circular Arc** and **Long\Default - Skim**.
- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Finishing** option.
- Open the **Line Projection Finishing** and **Tool Axis** forms and enter data exactly as shown on the following page and select **Preview**.



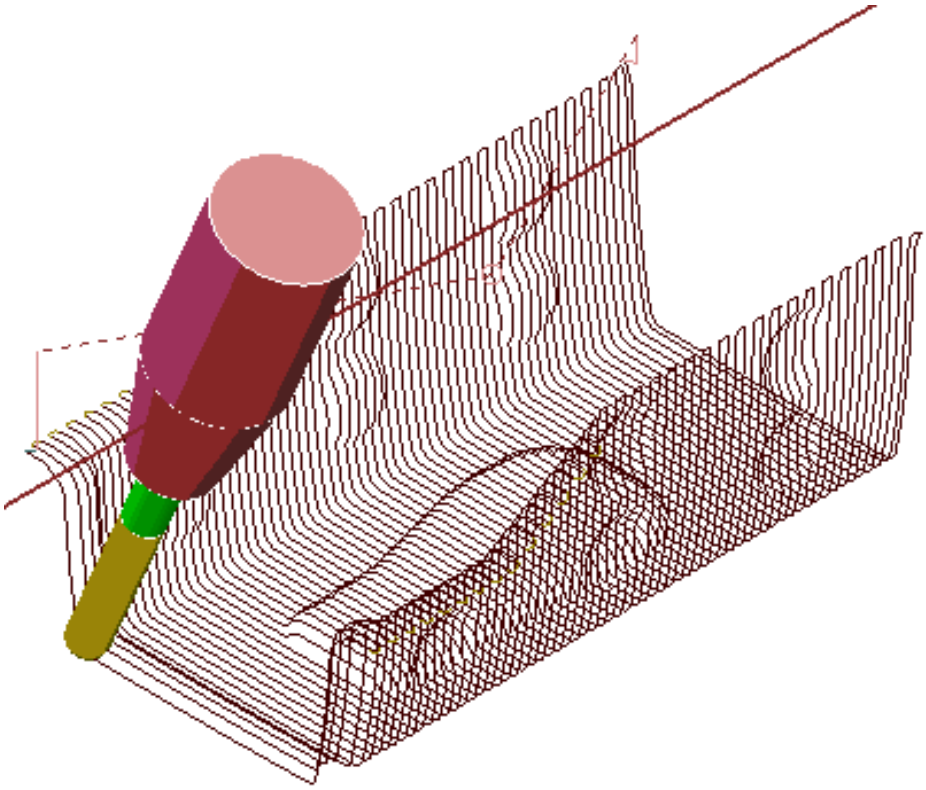
Set **Azimuth 0**
Set **Elevation 90**

- Tick the **Draw Tool Axis** box to display the ***Tool Alignment*** relative to the model.



- Click the **Preview** tab to view the strategy before selecting **Apply**.

The end result is as shown in the following illustration.

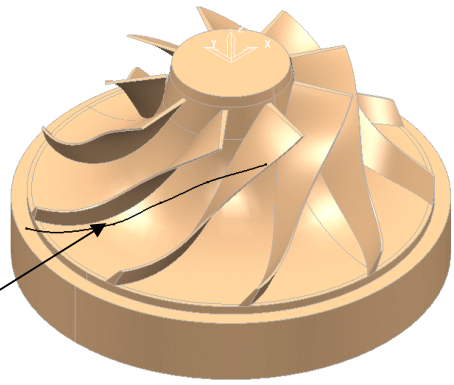


The whole of the internal form including undercut areas is accessible to this **Tool Axis** (**From Line**) alignment combined with the **Line Projection** - **Finishing** strategy.

Toward\From Curve

These options allow the tool axis **alignment** to be through a user-defined curve (**pattern**), during the creation of a **5-Axis** toolpath. Note; The following chapter provides a more detailed look at **Projection Surface Finishing**, the strategy used during this example.

- From the main pulldown menus select **File -Delete All**.
- **Import** the model **impeller+Curve.dgk** from the local directory:- **D:\users\training\PowerMILL_Data\five_axis\Impeller**
- Create an empty **Pattern** and rename it as **Align2Curve**.
- **Select** the alignment curve (imported with the model) in the graphics area and in the local **Pattern** (**Align2Curve**), menu select **Insert** - **Model** to make a copy of it as the **Pattern** segment.



- Create a **Dia 3 Ball nosed** cutter (**BN3-LR**) length **35** with the following **Shank** and **holder** dimensions:-

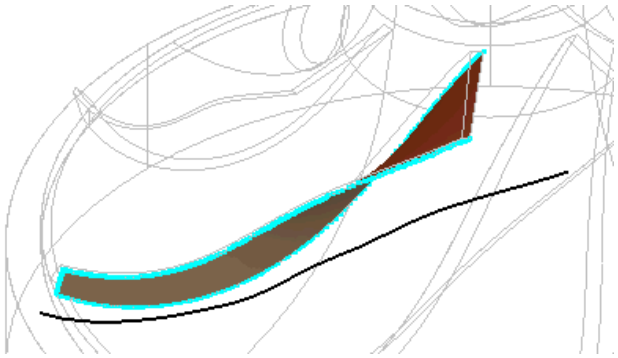
  Shank - Upper\Lower Dia 3 - Length 25
  Holder 1 - Upper Dia 15- Lower Dia 10 - Length 50
  Holder 2 - Upper\Lower Dia 15 - Length 35
  Overhang 50

- Create a **Block** defined by **Cylinder** to the **Model** dimensions.
- Set **Lead In\Out** to **Vertical Arc - Distance 0 -Angle 90 - Radius 3** and set all **Links** to **Skim**.
- In the **Rapid Move Heights** form select **Reset to Safe Heights**.
- In the **Start and End Point** form set both ***Start Point*** and ***End Point*** to **Automatic** and **Block Centre Safe**.
- From the main pulldown menus, select **View - Toolbar - Command**.
- In the ***command window*** type the following 3 lines:-

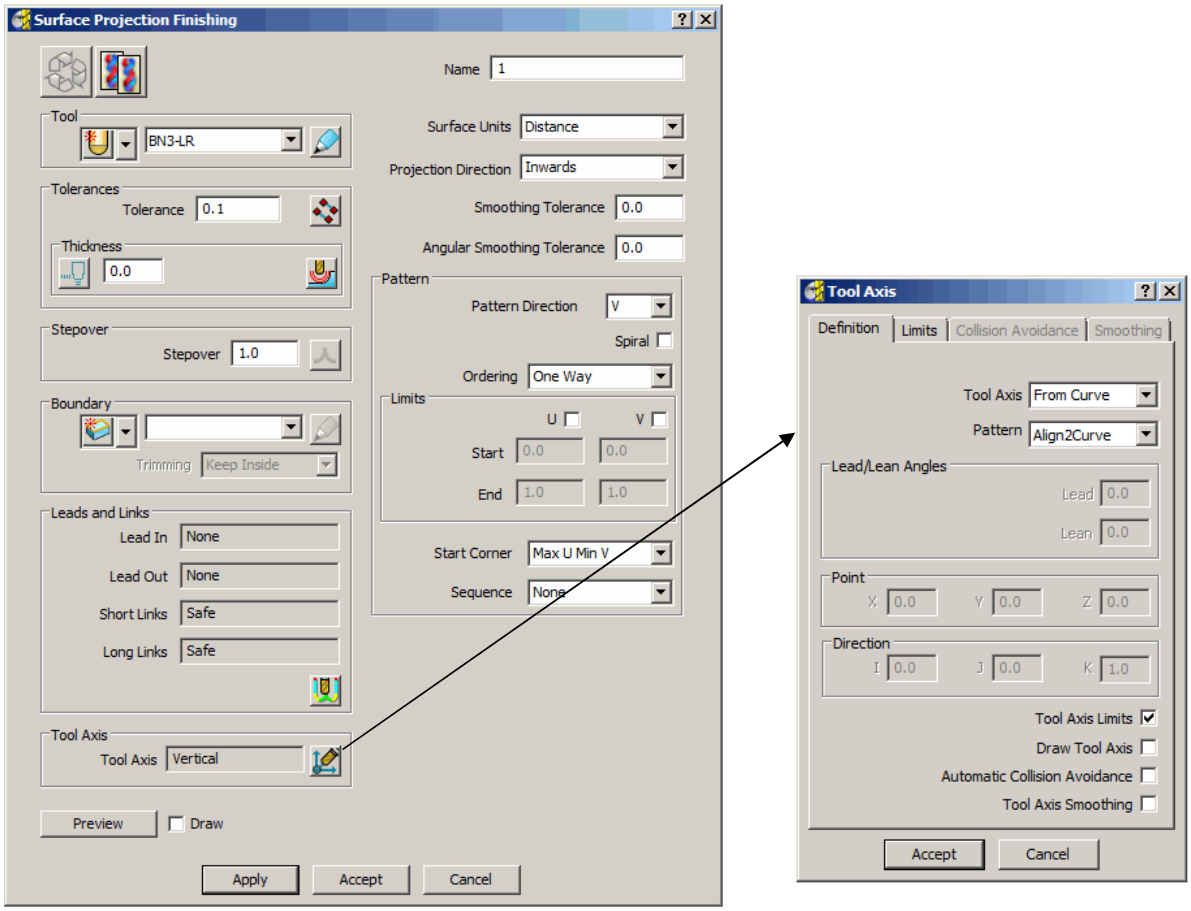
  EDIT SURFPROJ AUTORANGE OFF
  EDIT SURFPROJ RANGEMIN -1
  EDIT SURFPROJ RANGEMAX 1

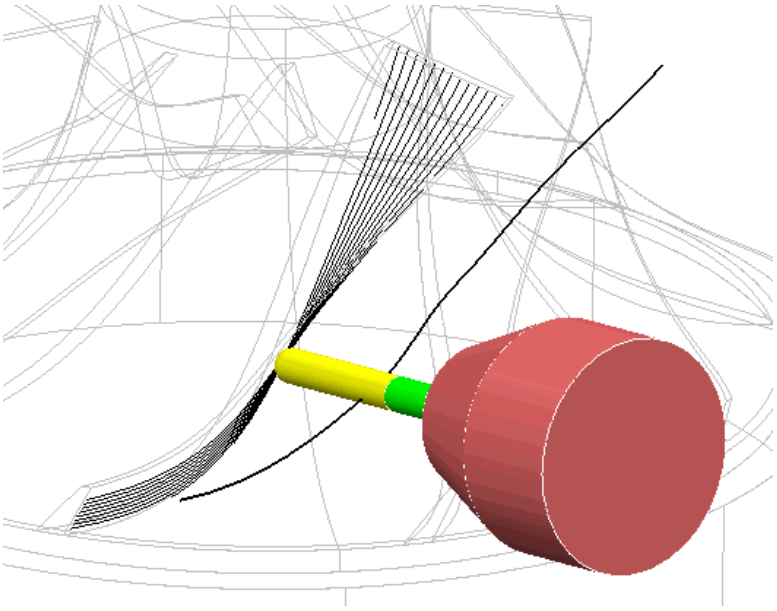
This will make the projection effective only within 1mm of the selected surface.
Note: **Surface Projection Range** is covered in more detail in **Chapter 3 - Page 57**.

- Close the ***command window*** by clicking on the **black cross** at the top left corner.
- Select the underside, blade surface nearest to the **Pattern** for use with the **Projection Surface** strategy.



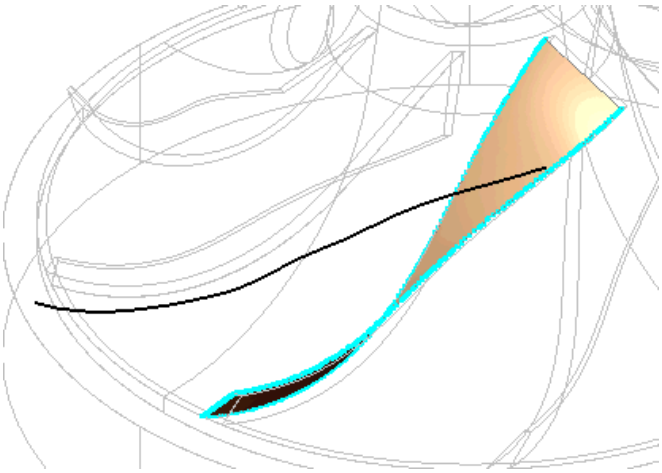
- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Finishing** option.
- Open the **Projection Surface Finishing** and **Tool Axis** forms entering the data <u>exactly</u> as shown below and select **Apply**.



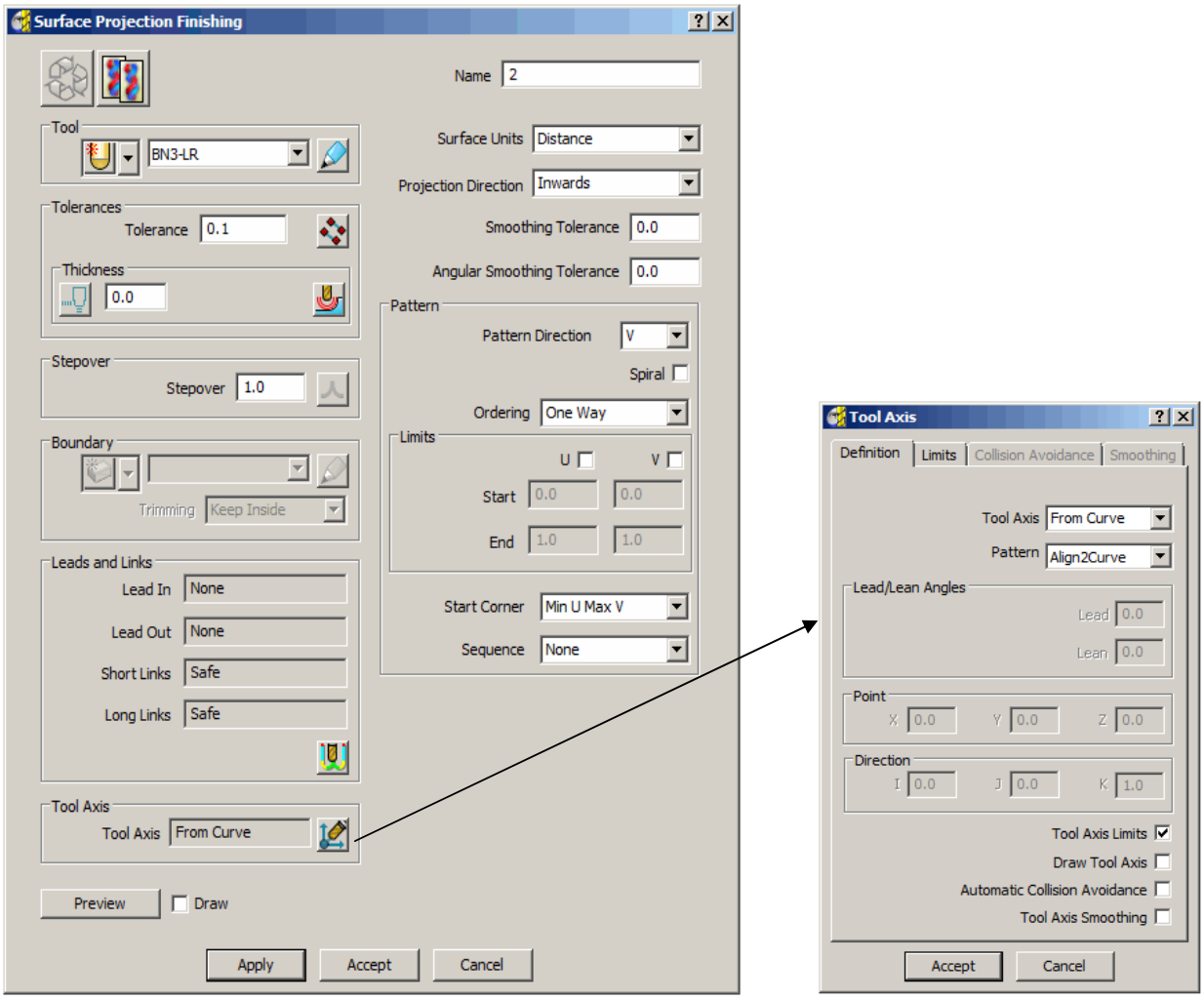


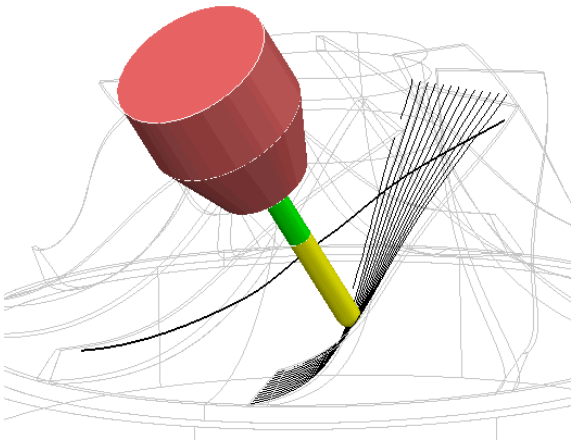
The tool axis is always aligned through the specified **Pattern** (Curve) while performing the chosen machining strategy (**Projection Surface**).

- Select the upper side, blade surface nearest to the **Pattern** for use with the **Projection Surface** strategy.



- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Finishing** option.
- Open the **Projection Surface Finishing** and **Tool Axis** forms entering the data exactly as shown on the next page and select **Apply**.





The ***tool axis*** is always aligned through the specified **Pattern** (Curve) while performing the chosen machining strategy (**Projection Surface**).

- From the main pulldown menus, select **View - Toolbar - Command**.
- In the ***command window*** type the following line (to reinstate the default, infinite projection range):-

EDIT SURFPROJ AUTORANGE ON

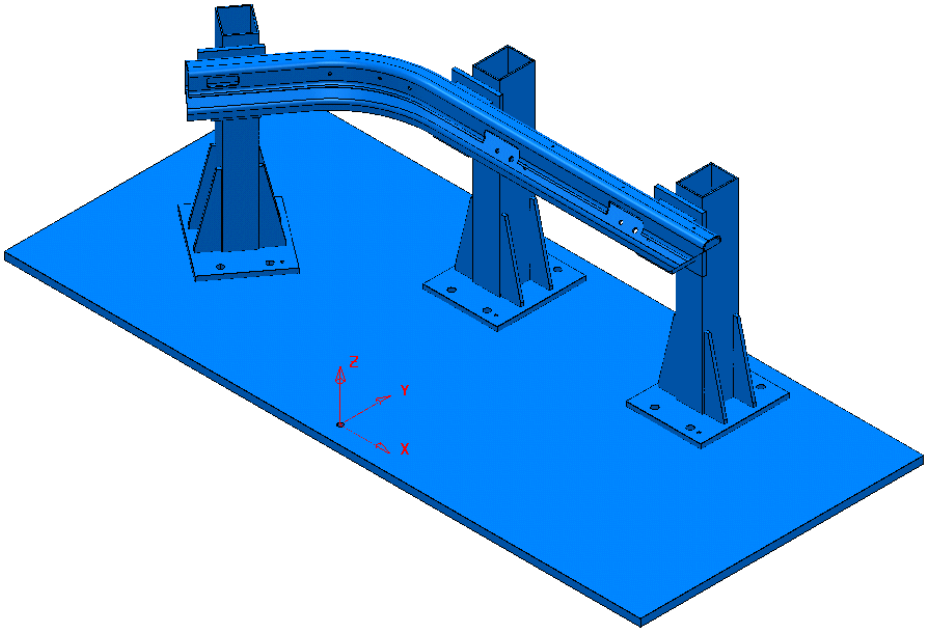
Fixed Direction

This allows the tool axis to be set to a fixed angle, specified by Vector, defined by the user. In this case it is applied to the finishing of an undercut form on the 5axis_fixture.dgk model.

- From the main pulldown menus select **File -Delete All**.
- **Reset** forms.
- **Import** the **model**

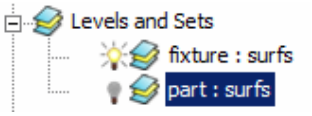
D:\users\training\PowerMILL_Data\five_axis\Autorail_and_Fixture\5axis_fixture.dgk

- Create a **Dia 16 Ball nosed** cutter (**BN16**).
- Set all **Leads** and **Extensions** to **None**.



The model consists of components existing on two separate **levels**, ***Fixture:surfs*** and ***Part:surfs.*** All items on a specific **level** can be drawn or undrawn from the **levels** area in the **explorer** window. In this example it is only required to machine the part of the component stored on **Fixture:surfs** using a **Fixed Direction** alignment which means that the items stored on **Part:surfs** must be temporarily discarded to allow access.

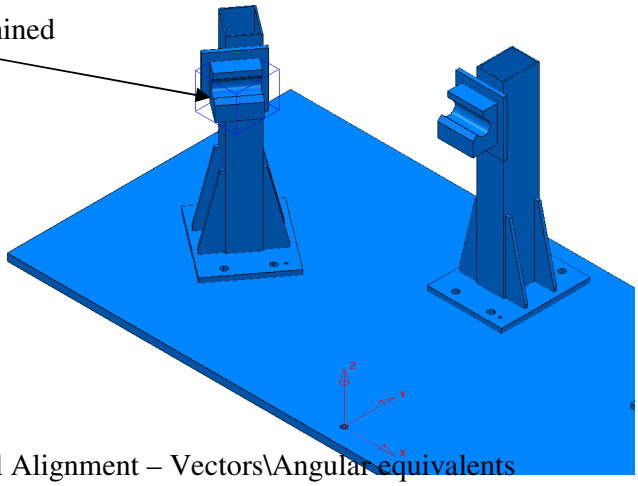
- **Undraw** the **Part: surfs** level from within the **explorer** window.



It is not sufficient to simply undraw a **level** to prevent the associated part of the model being included in a machining strategy. To stop **Powermill** machining data stored on a particular **level**, the contents are **Aquired** to a selected row in the **Components Thickness** list, which is then set to **Machining Mode - Ignore** before the toolpath is calculated.

- **Activate** the workplane **Car Line datum.**
- **Select** the **surfaces** of the location to be machined and calculate a **Block**.

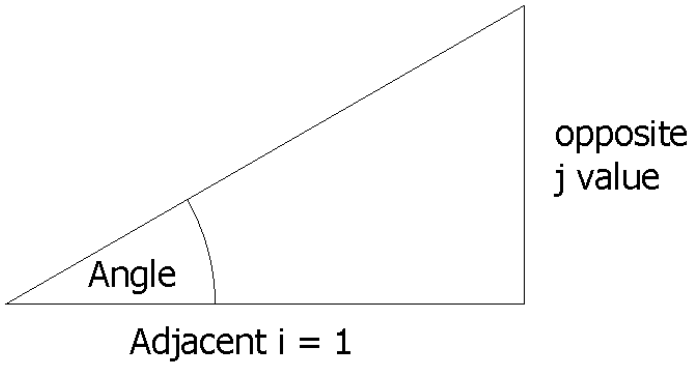
Location to be machined (**Block** drawn).



Fixed Direction Tool Alignment – Vectors\Angular equivalents
The table represents angular directions and vectors on the XY plane (Z=0).

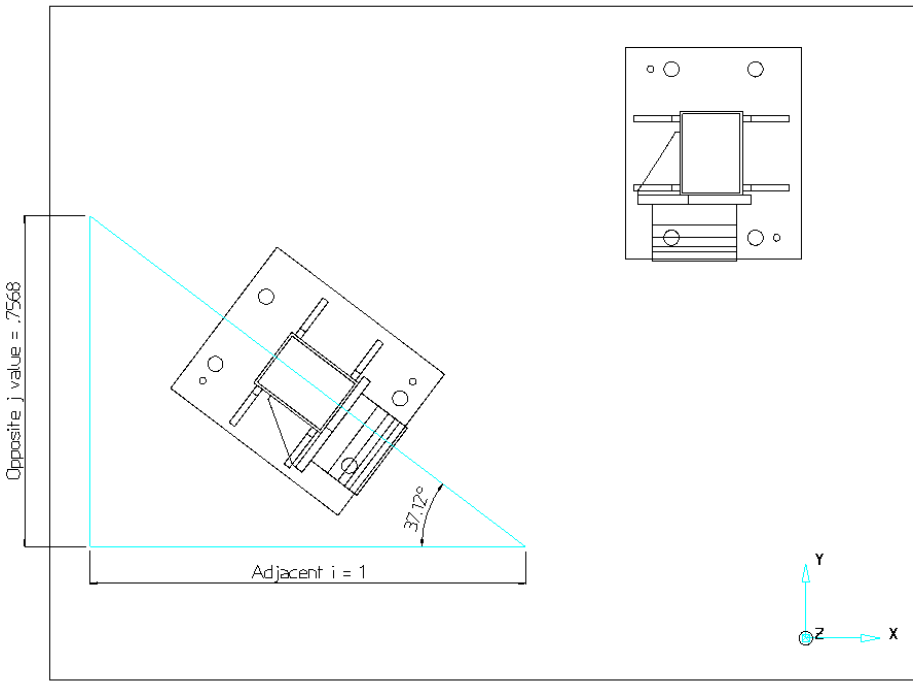
ANGLE	**VECTORS**		
(Degrees)	**(I**	**J**	**K)**
0	1	0.0000	0
5	1	0.0875	0
10	1	0.1760	0
15	1	0.2680	0
20	1	0.3640	0
25	1	0.4660	0
30	1	0.5770	0
35	1	0.7000	0
40	1	0.8390	0
45	1	1.0000	0
50	1	1.1920	0
55	1	1.4280	0
60	1	1.7320	0
65	1	2.1450	0
70	1	2.7470	0
75	1	3.7320	0
80	1	5.6710	0
85	1	11.4300	0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

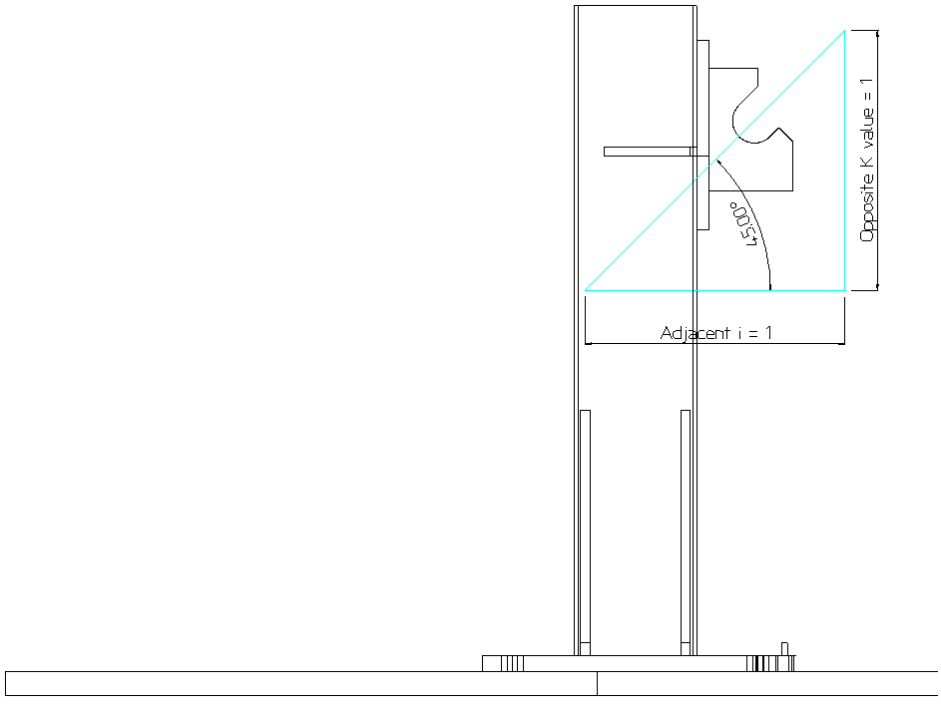
$$\text{Tan (Angle)} = \frac{\text{(opposite)}}{1}$$



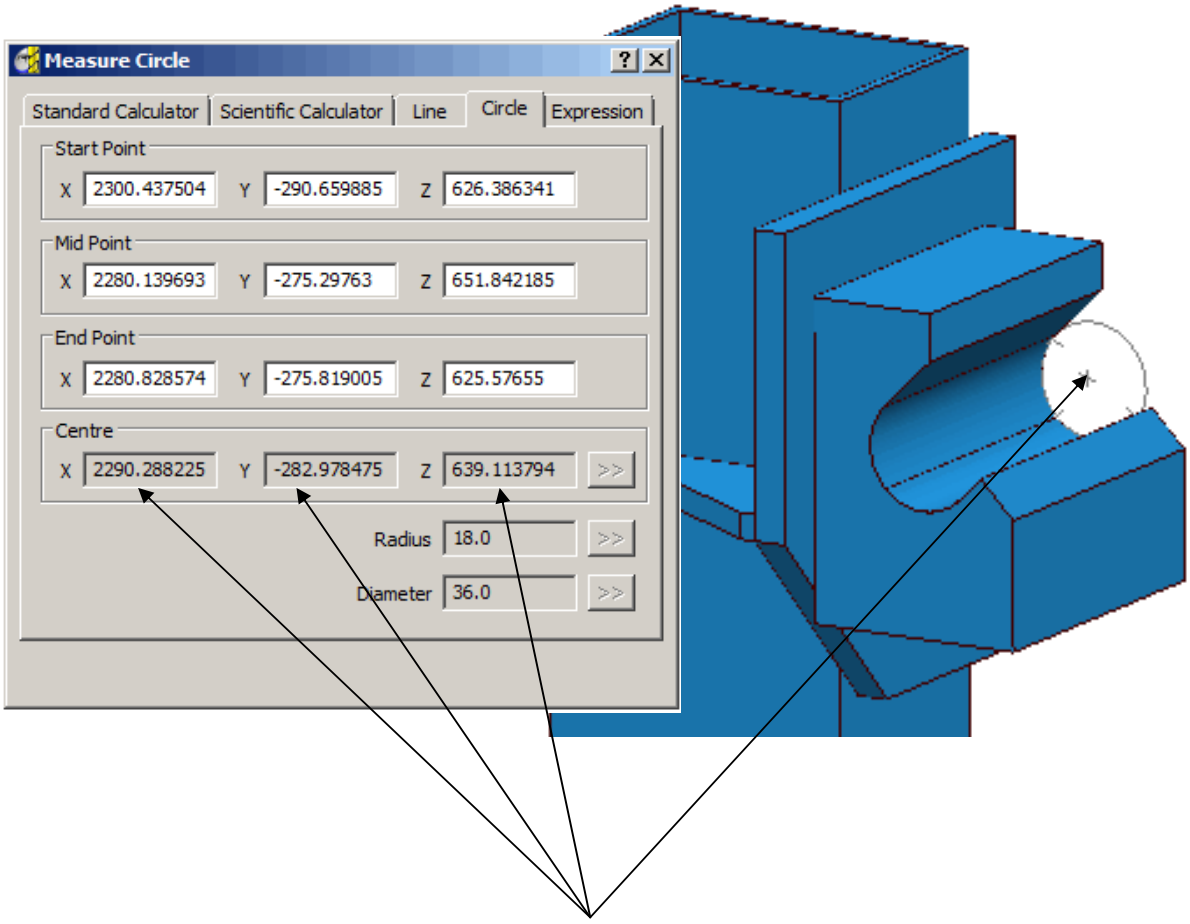
The **tool alignment** is set to a **Fixed Direction** relative to the currently active **workplane** by inputting suitable values to define the **IJK** vector along the **Tool Axis** (towards the spindle). Although obtaining individual values for a vector may require the user to exercise their trigonometry skills, this method does provide full flexibility for defining compound angles.

Note: on the previous page an **Angle to Vector** conversion table has been provided.



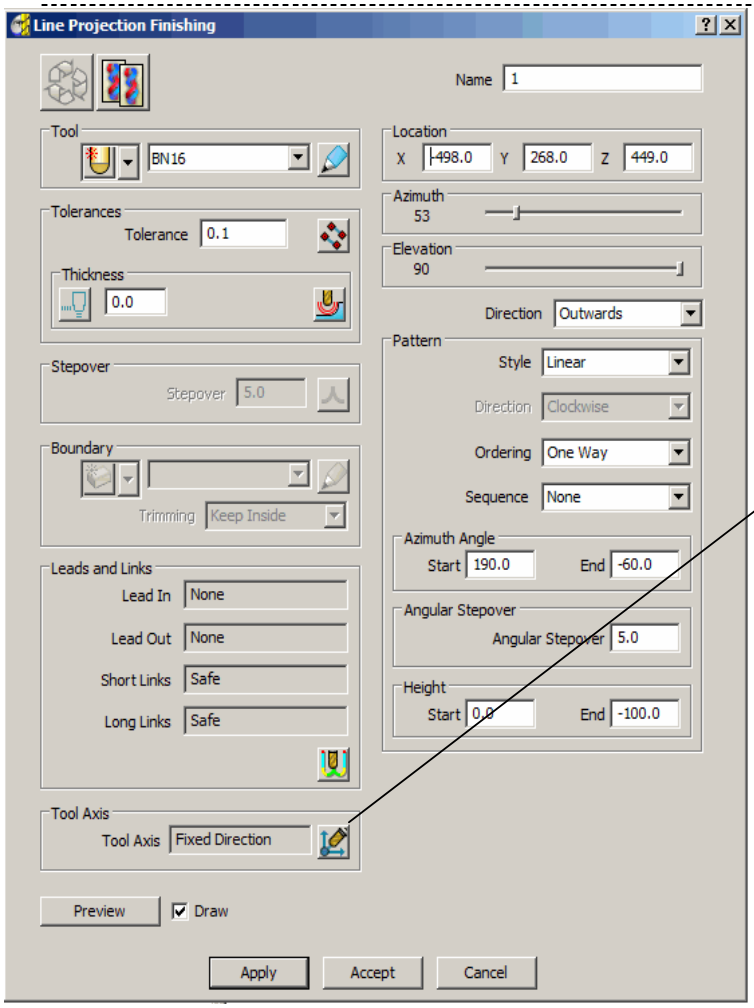


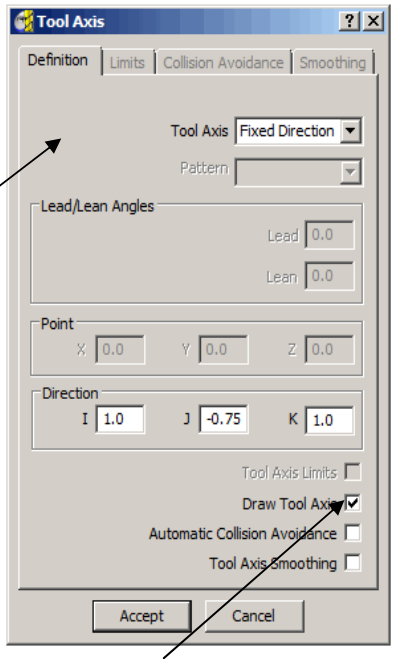
- From the **Main toolbar** click  to open the **Calculator** form and using the **Circle** option click 3 points around the circular edge at the end of the location block (as shown below).



The displayed **Centre** - **XYZ** coordinate values will be used as the **Location** values in the **Line Projection** form (as illustrated on the next page).

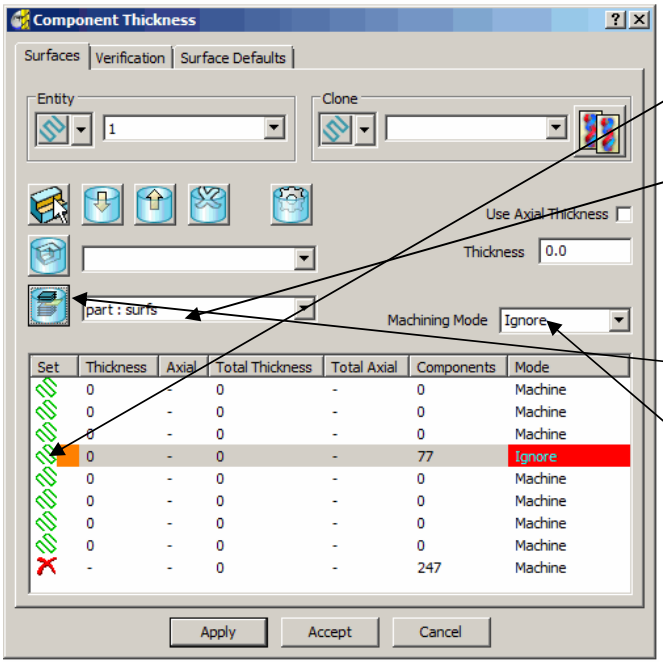
- Select the **Toolpath Strategies** icon  and in the **New** strategies form select the **Finishing** option.
- Open the **Line Projection Finishing** and **Tool Axis** forms and enter data exactly as shown on the next page.





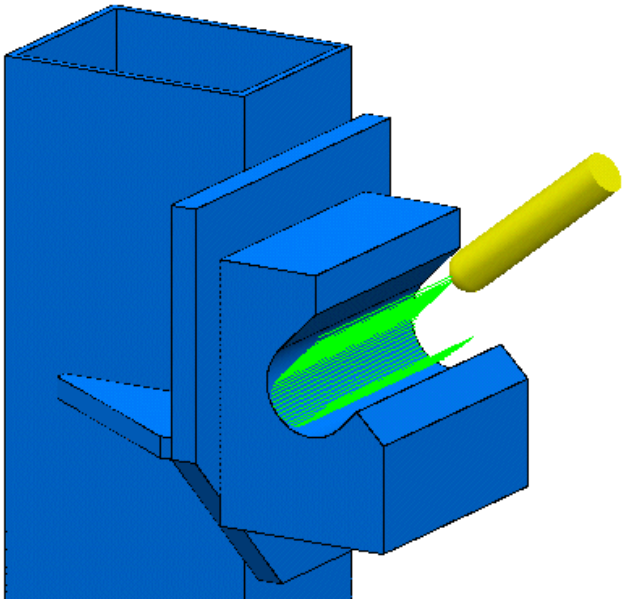
- ***Azimuth* 53**
- ***Elevation* 90**
- **Draw Tool Axis** (ticked).

- Select the **Thickness** icon and then the **Surfaces** tab to access the **Component Thickness** form below.

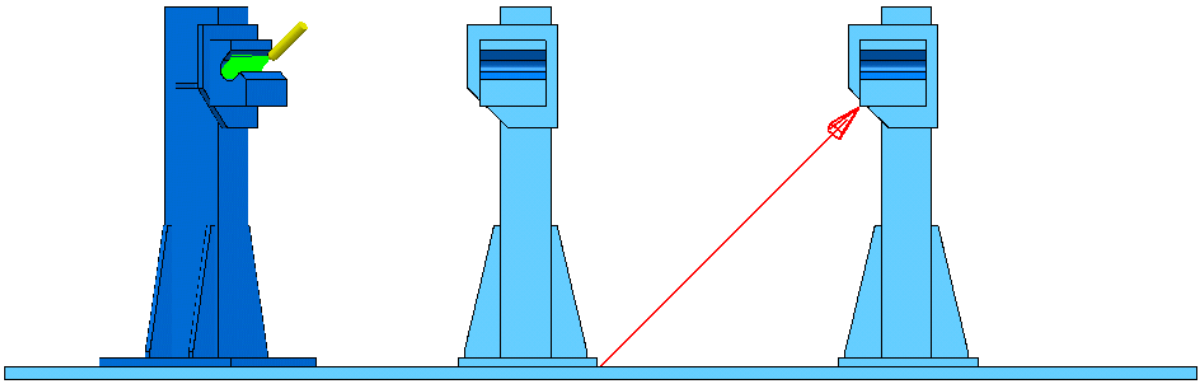


- **Highlight** a row (left mouse click).
- **Select** the level **part : surfs**.
- Click the **Level** icon to acquire the contents to the selected, **Component Thickness** row.
- **Select Ignore** from the machining mode pull down option.
- **Apply** and **Accept** the form.

- **Apply** the **Line Projection** finishing
- **Simulate** the Toolpath.



The tool axis is fixed to the direction specified by the IJK vector. By setting the machining mode to **ignore** for the part surfaces only the fixture is machined. The **tool axis** has also been drawn indicating the vector direction.



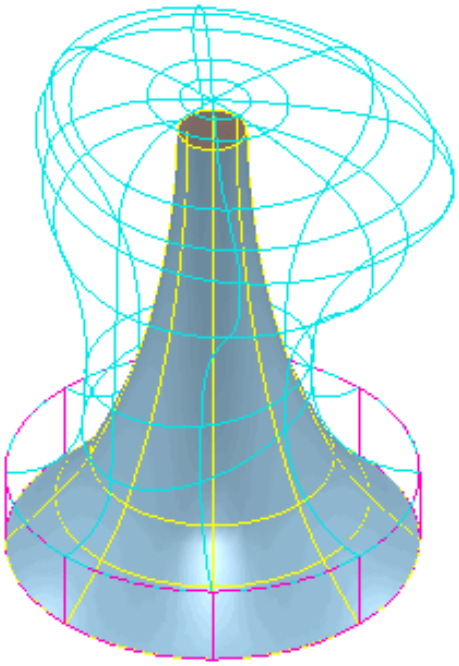
Note: The **vector** must be defined pointing up the tool axis towards the spindle.

3. Projection Surface Finishing

Introduction

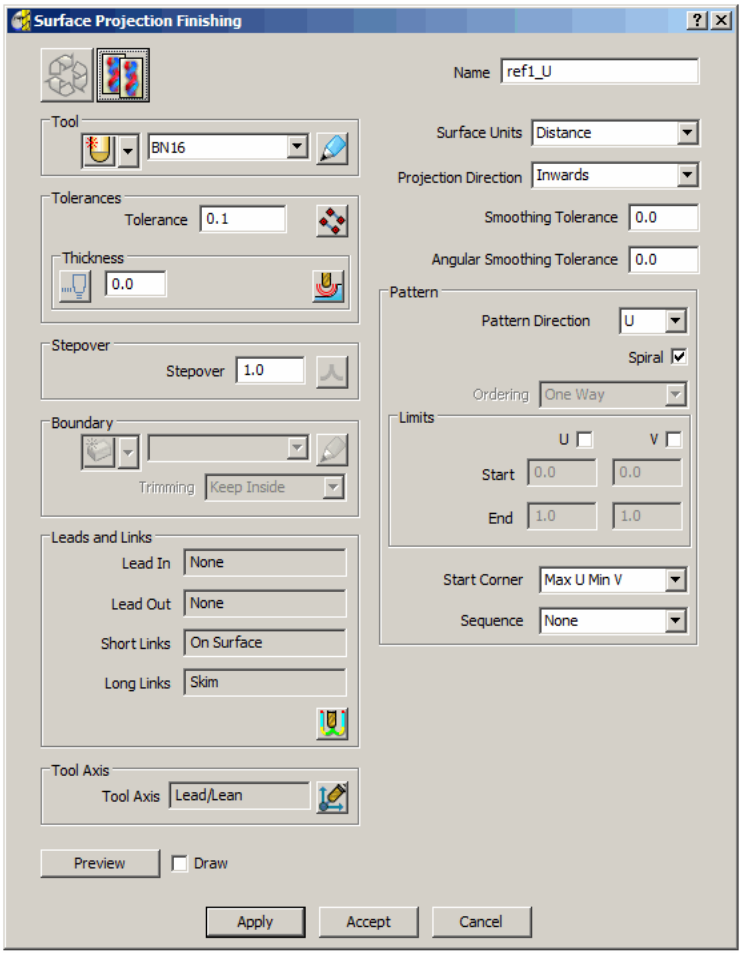
The strategy is projected along the normals of a **Reference Surface** onto the main component with tool alignment as specified by the user. The toolpath runs either across or along the **Reference Surface** directions (**U** or **V**) with the **Stepover** being defined by unit **Distance** or **Parametric** 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.

- **Import** the two **models** ***joint5axis.dgk*** and ***joint_ template1.dgk*** from:-
  D:\users\training\PowerMILL_Data\PowerMILL_data\five_axis



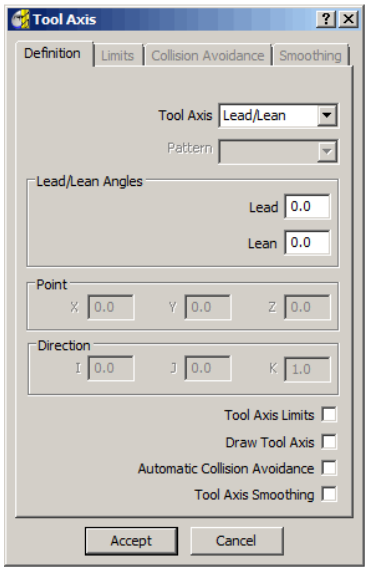
- Create a **Block** to the component dimensions.
- Define a **16mm** diameter **Ball Nosed** cutter (**BN16**).
- In the **Rapid Move Heights** form select the **Reset to Safe Heights**.
- In the **Start and End Point** form **Use - Block Centre Safe** for both.
- In the **Leads and Links** form  set ***Short Links*** to **On Surface** and ***Long\Safe Links*** to **Skim** adjusting the **Zheights** to a **Skim distance** of **30** and **Plunge distance** of **5**.
- Select the ***Reference Surface*** - **joint_ template1.dgk** (shown shaded above).

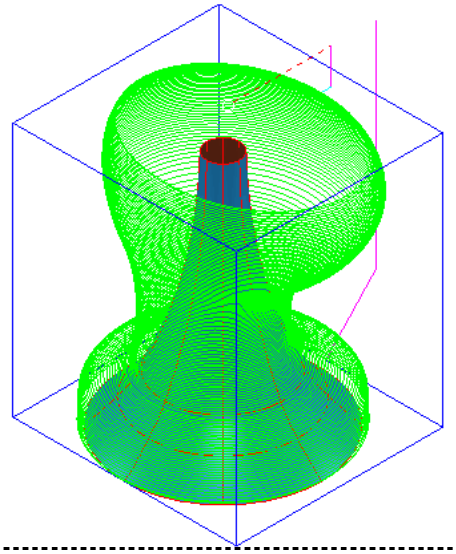
- Select the **Toolpath Strategies** icon in the **Main** toolbar to open the **New** strategies form.
- Select the **Finishing** tab followed by the **Projection Surface Finishing** option.
- Enter the values into the **Surface Projection Finishing** and **Tool Axis** forms <u>exactly</u> as shown below and **Apply** but **<u>do not close the form</u>**.



- Enter the ***Name*** - **ref1_U**

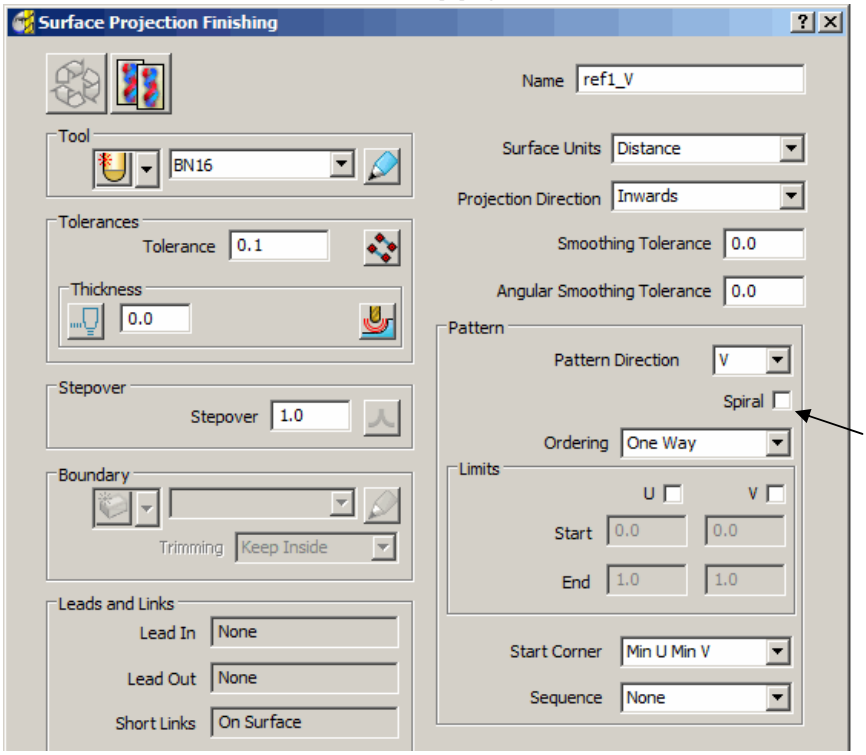
- Tick **Spiral**





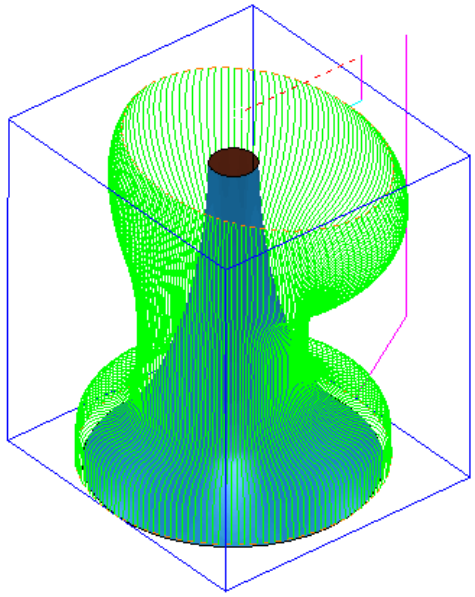
With **Pattern Direction** set to **U** the toolpath will appear <u>along</u> and <u>aligned</u> to the **Reference Surface** - **Longitudinal** direction (as shown left).

- Select the **Copy** icon  to re-activate the **Settings** and input the new ***Name*** - ***Ref1_V***
- Untick **Spiral** and **Set** the **Ordering** option to **One Way**, set the **Pattern Direction** to **V** and select **Apply**.



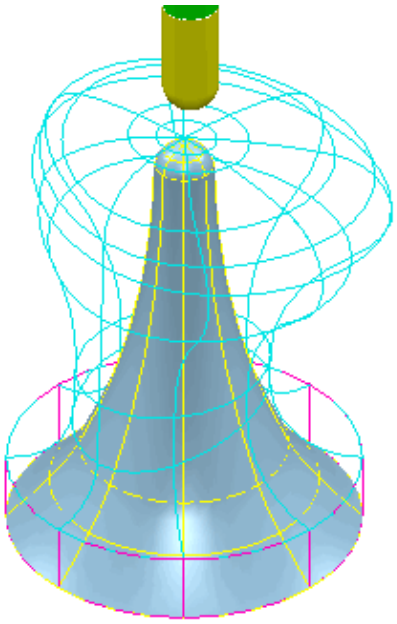
Untick Spiral

This time the toolpath will appear along and aligned to the **Reference Surface** - **Lateral** direction (as shown below).

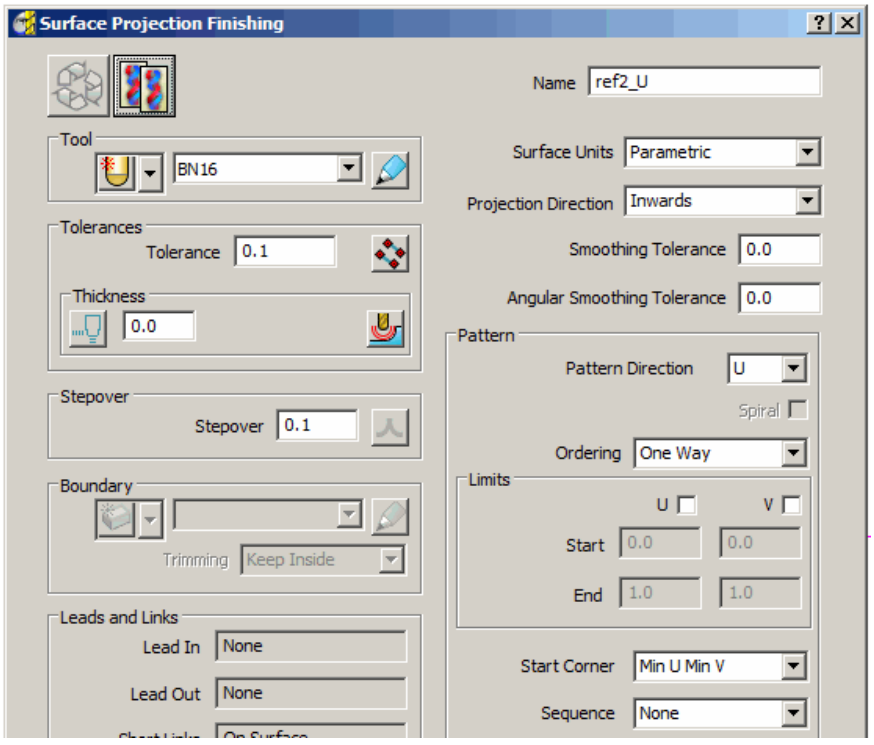


Both the above toolpaths have a **Stepover** based on a unit **Distance** value across the **Reference Surface**. The next **Toolpath** to be produced will use **Parametric** divisions across the **Surface Curves**.

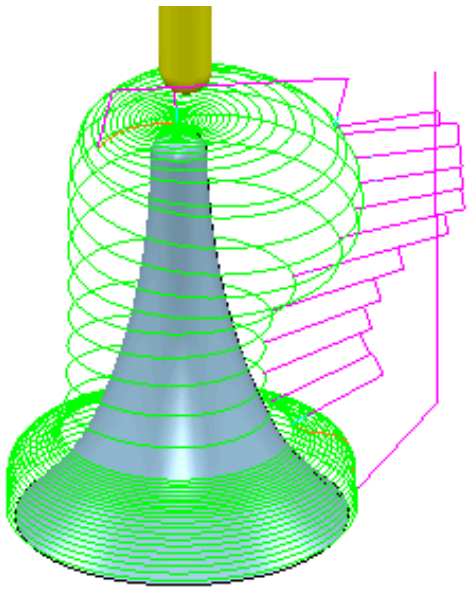
- **Delete** the ***Reference Surface*** – **joint_ template1.dgk** and as a replacement, ***Import*** - **joint_ template2.dgk** making sure that it is **selected**.



- **Apply** another **Copy** of the **toolpath** named **Ref2_U** with ***Surface Units*** - **Parametric**, ***Pattern Direction*** **- U**, and ***Stepover*** - **0.1**.



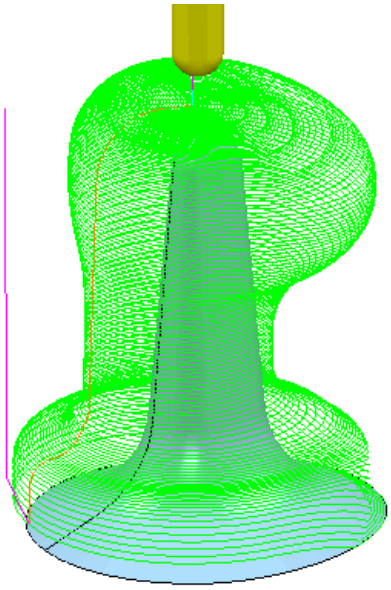
The **Stepover** - **0.1** creates **10** equally spaced tool tracks between each pair of ***Reference Surface***, **Curves** parallel with the machining direction.



To even out large variations in the parametrically defined **Stepover** across the component the **Reference Surface** requires additional, suitably placed curves.

A modified surface has already been created in **PowerSHAPE** to replace the current **Reference Surface** as instructed below.

- **Delete** the ***Reference Surface*** – **joint_ template2.dgk** and as a replacement, ***Import*** - **joint_ template3.dgk** making sure that it is **selected**.
- **Apply** a **Copy** of the above **Surface Projection** toolpath named **Ref3_U** to create the following result.

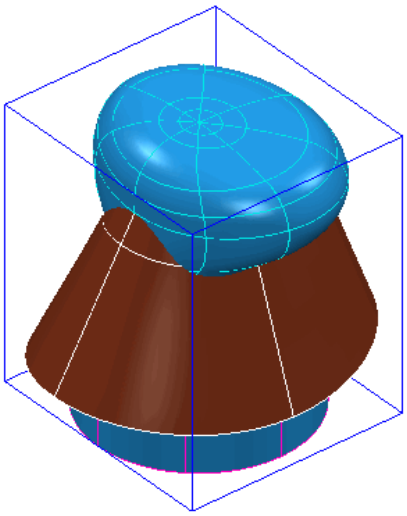


The new **toolpath** is greatly improved with a more consistant **Stepover**.

To provide full flexibility for the shape of the **Reference Surface** it is possible to allow it to exist outside the component to be machined.
This is achieved by opening up the **Component Thickness** options from the strategy form, and assigning the **Reference Surface** to a row, with the ***Machining Mode*** set to **Ignore** (it is still used to control the **Tool Axis** alignment).

- **Delete** the ***Reference Surface*** - **joint_ template3.dgk**.

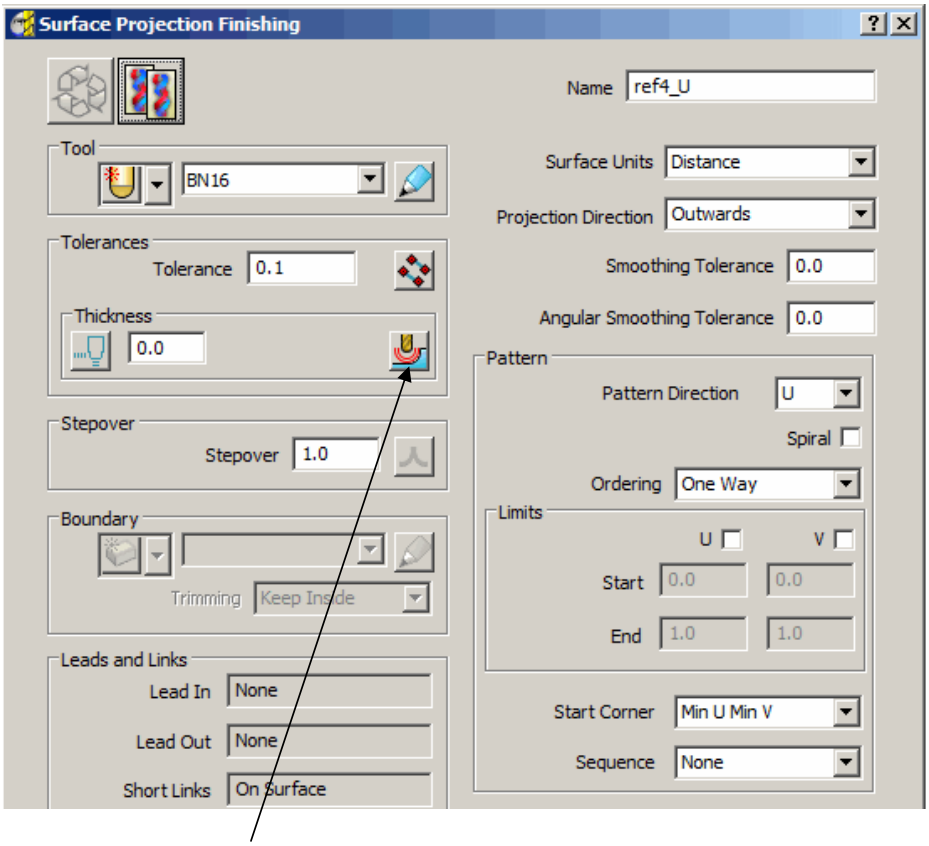
- Right click on **Models** in the **PowerMILL explorer** and select **Import Reference Surfaces**.
- In the form select the surface model **joint_template4.dgk**.



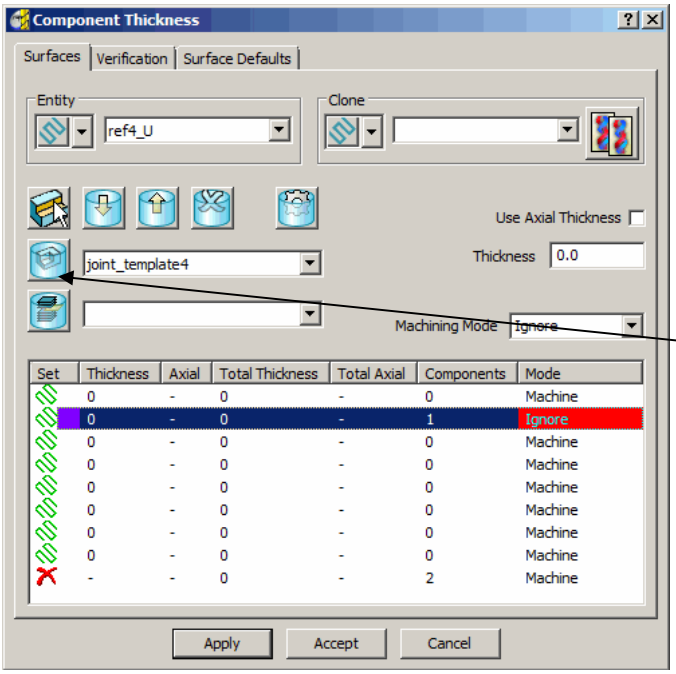
Note: Most of the new **Reference Surface** exists outside the component to be machined. As a result of using the command **Import Reference Surfaces** the **Machining Mode** is inherently set to **Ignore** for the **Reference Surface**. To remove this setting the surface must first be aquired to another row.

Also the ***reference*** **surface** is inside out (dark brown in normal shading mode) requiring the **Projection Surface** Strategy to use ***Projection Direction*** - **Outwards**.

- Open a **Surface Projection** strategy named **Ref4_U** with ***Surface Units*** - **Distance**, ***Projection Direction*** **- Outwards**, ***Pattern Direction*** **- U**, ***Stepover*** - **1.0** and ***One Way*** ticked.

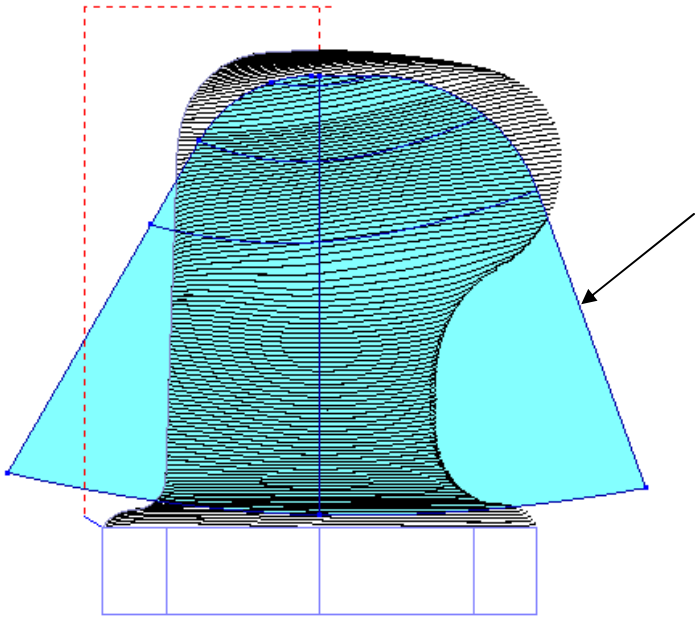


- Select the **Component Thickness** options.



- Select a **row** and set the **Machining Mode** to **Ignore**.
- Select the ***Model*** **Joint_template4** and click the icon to ***aquire*** it to the selected row.

- **Apply** and **Accept** the **Component Thickness** settings.
- **Apply** the **Surface Projection Finishing** form to produce the following toolpath.



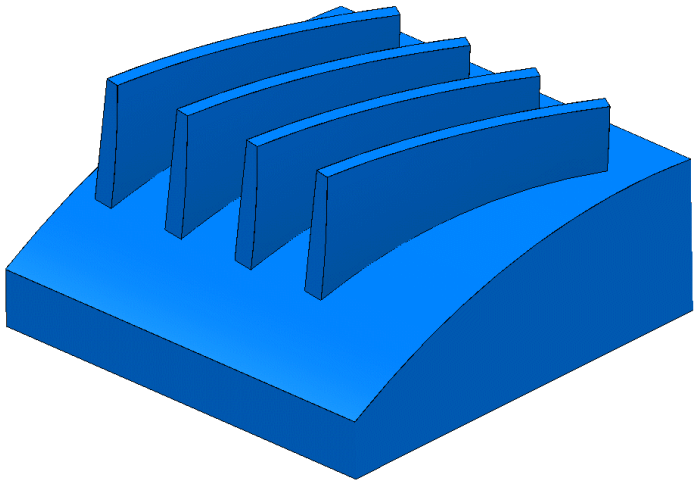
Note: The **Reference Surface** used to provide the **Tool Alignment** has been ***ignored*** to enable the ***toolpath*** projection onto the component.

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

Surface Projection Range.

It may be required during some applications, to **limit** the 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 way of the defined projection options. This command is, at present, only available via typed input into the ***PowerMILL***, **Command Window**. A more efficient way to control the **Projection Range** limits is to store the command lines for different distances in a series of **macros**, which in turn can be accessed via the **user menu**.

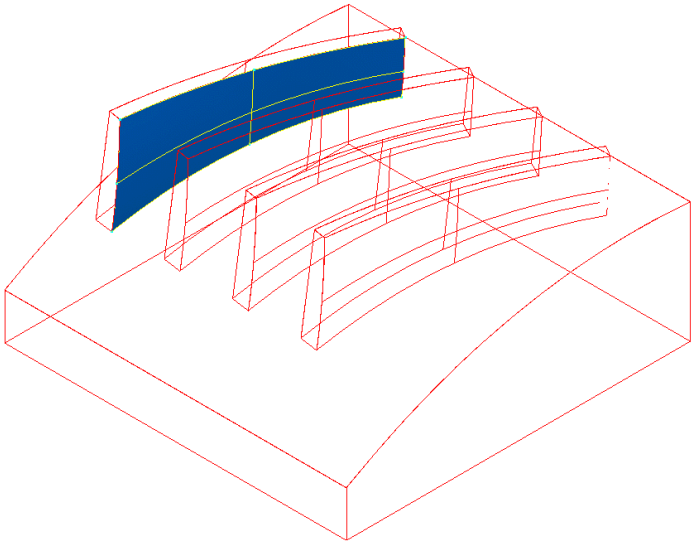
- **Import** the **model**:-
  D:\users\training\PowerMILL_data\five_axis\Blade_Sub_Assembly\Blade Inserts



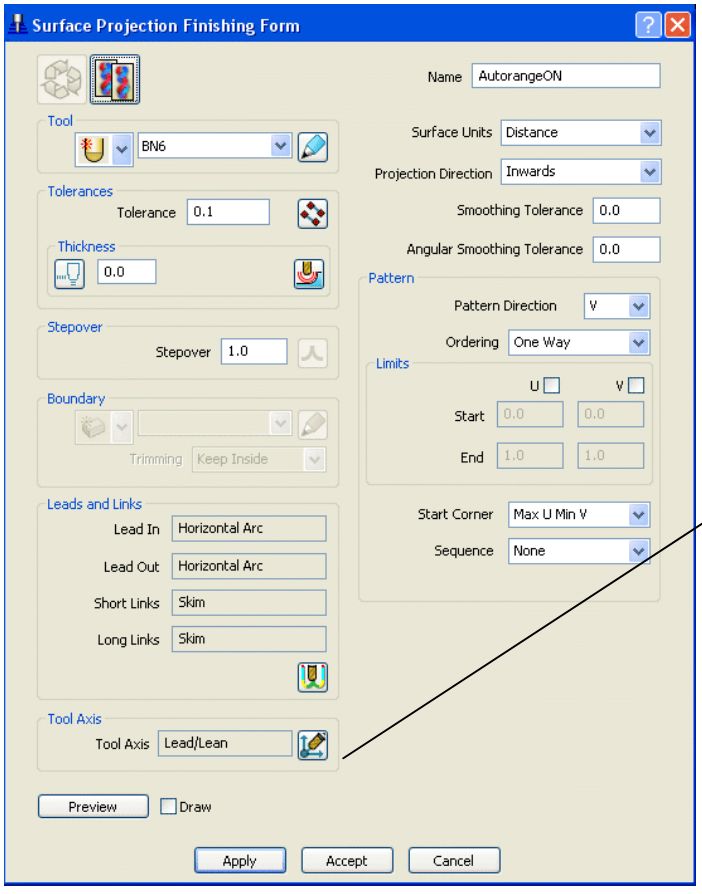
- Create a **Block** to the component dimensions.
- Define a **6mm** diameter **Ball Nosed (BN6)** cutter of **Length 30**.
- Create a **Shank** with **Upper Dia 6 Lower Dia 6 Length 20**.
- Create a **Holder** with **Upper Dia 20 Lower Dia 16 Length 30**
- **Add** a **Holder** component **Upper Dia 30 Lower Dia 30 Length 20 Overhang 40**
- Reset **Safe Z** and **Start Z**.
- Set the **Start and End Point** form for both to ***Use* - Block Centre Safe**.
- Set **Leads** and **Links** as follows:-

Lead in\out **Horizontal arc** **Distance 0** **Angle 90** **Radius 3**
Links **Short\Long\Safe** **Skim**

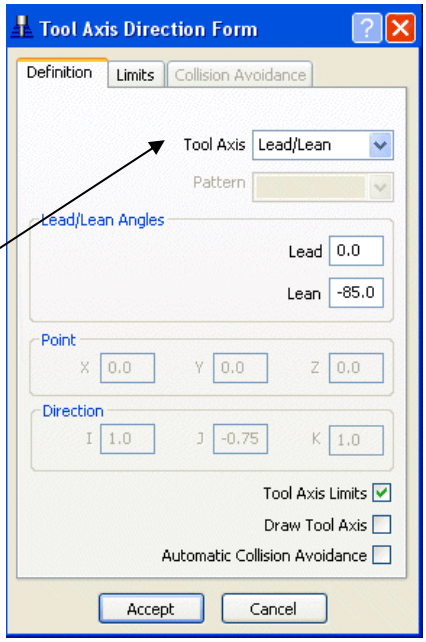
- Select the **Blade Surface** to be machined (shown shaded below).



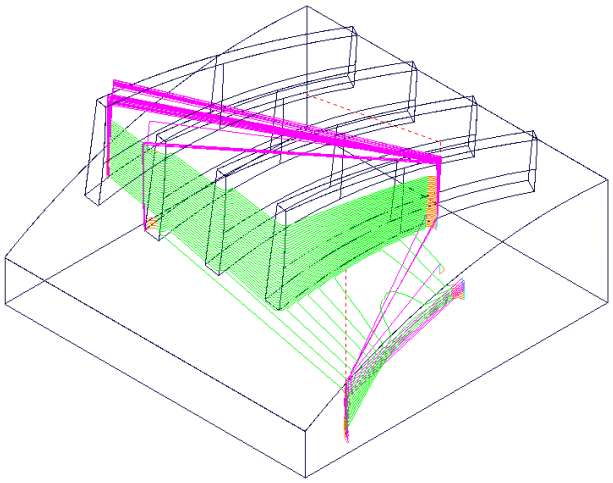
- Select the **Finishing icon**  followed by the **Projection Surface Finishing** option.
- Enter the values into the **Surface Projection Finishing** and **Tool Axis** forms ***exactly*** as shown below and **Apply**.



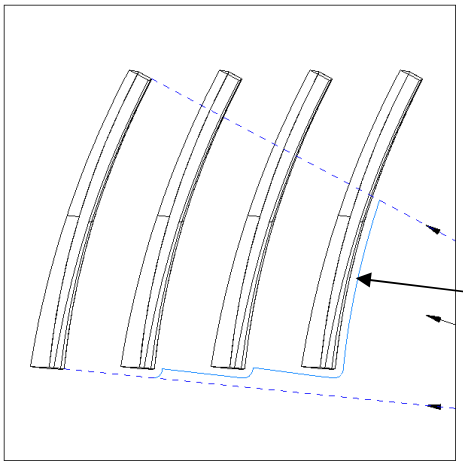
- Rename as **AutorangeON**



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 it will appear on neighbouring surfaces instead of those intended to be machined and create the toolpath as shown.



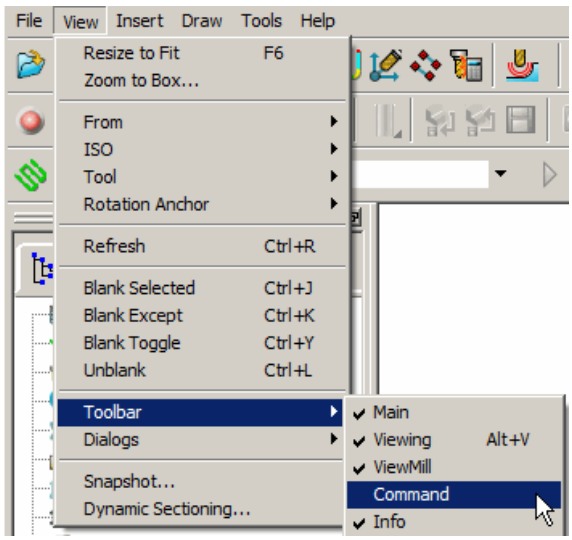
Projection Range (Unlimited)



Tool tracks are created on first face of **model** in line with the **projection direction**.

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

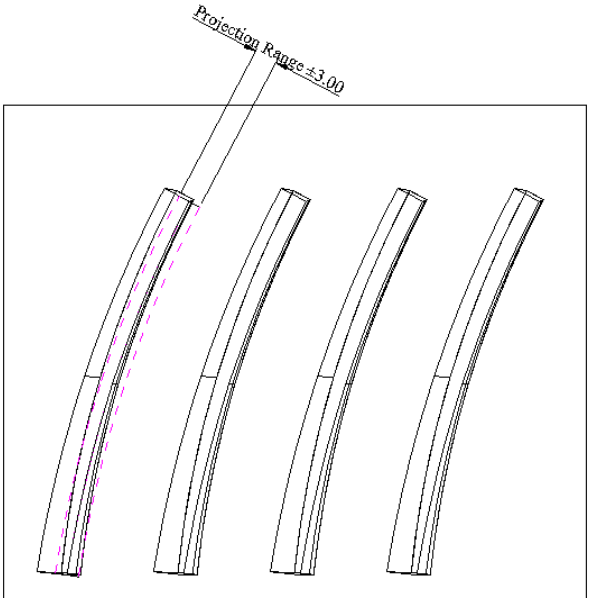
- From the **View menu** at the top of the screen Select the option **Toolbar - Command Window**.



- Select **Settings** from the toolpath in the explorer window and make a **Copy** of the original toolpath. 

- **Rename** the toolpath:- **3mm_projection_range**.
- **Click** the mouse into the **Command Window** at the bottom of the screen and **enter** the following commands.

EDIT SURFPROJ AUTORANGE OFF
EDIT SURFPROJ RANGEMIN –3
EDIT SURFPROJ RANGEMAX 3

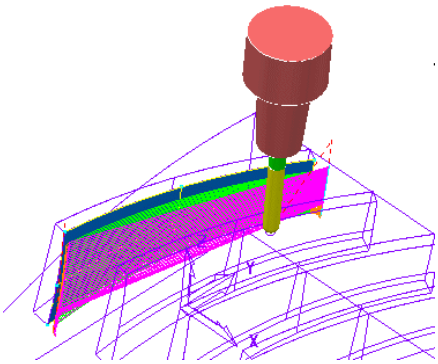


The above command input limits the **Surface Projection Range** to **+ \ - 3mm**.

- **Apply** the form to calculate the toolpath.

- **Simulate** the toolpath to observe the effect of the limited projection range.

- In the **Command Window** type the following to restore the ***default***, infinite projection range.



EDIT SURFPROJ AUTORANGE ON

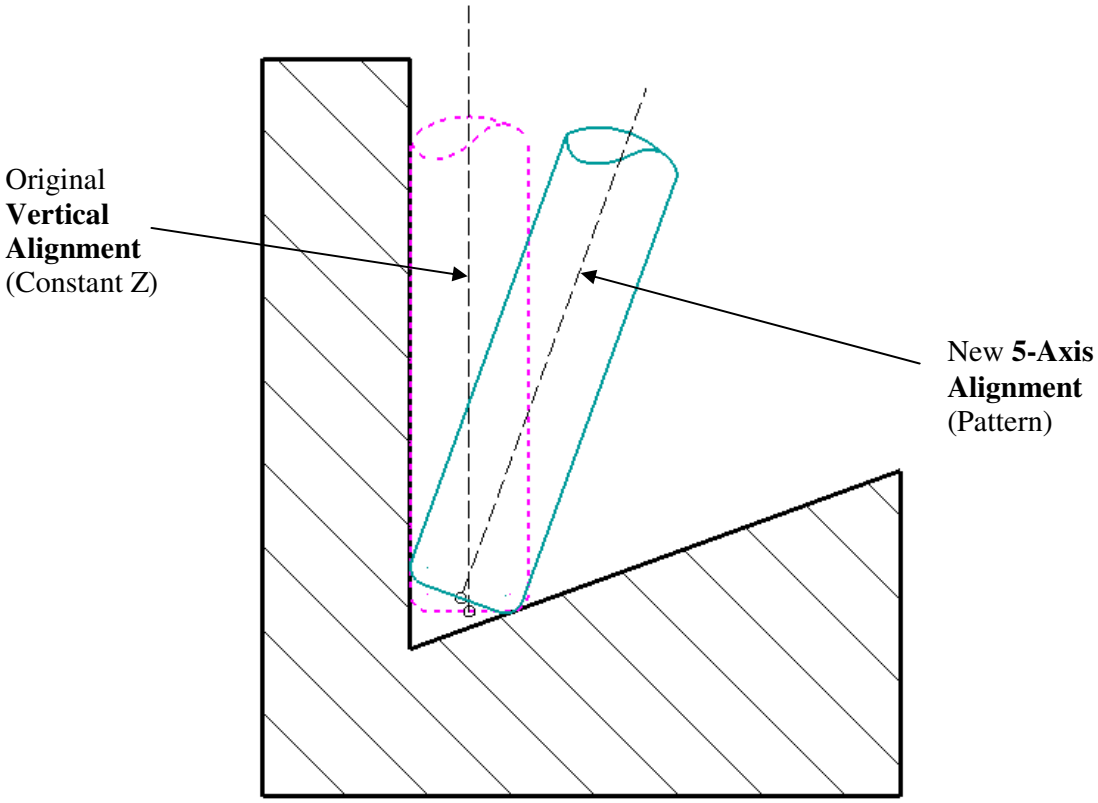
This command returns to default no limit to the **Surface Projection** distance range.

Note: A series of **macros**, ideally accessed via the **user menu** could easily be created so that the user does not have to keep on typing in the required **Surface Projection Range** data into the **Command Window**.

4. Five Axis Pattern Finishing

Introduction

All strategies with the exception of **Plunge Milling** and **Drilling** will directly support **5-Axis tool alignments**. However, certain finishing strategies will only do so where a **Ball Nose** tool is being used. For these strategies if an **End Mill** or **Tip Radiused** tool is in use a **5-Axis** conversion is achieved by re-machining using the **Pattern** finishing strategy. If **Base Position - Automatic** is specified the toolpath will be recreated to the selected **5-Axis Tool Alignment** (as illustrated below).

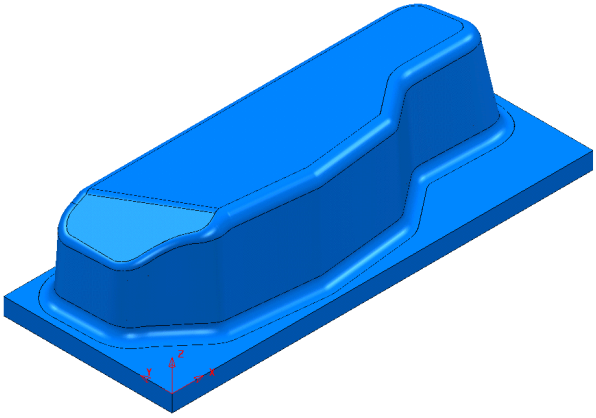


The above diagram represents a **Tip Radiused** cutter applied to a **Constant Z** finishing strategy. This will only operate with a **Vertical Alignment** and as a result will have to be regenerated as a **Pattern** finishing strategy with a **Lead\Lean Alignment** selected.

Constant Z Finishing - conversion to 5-Axis

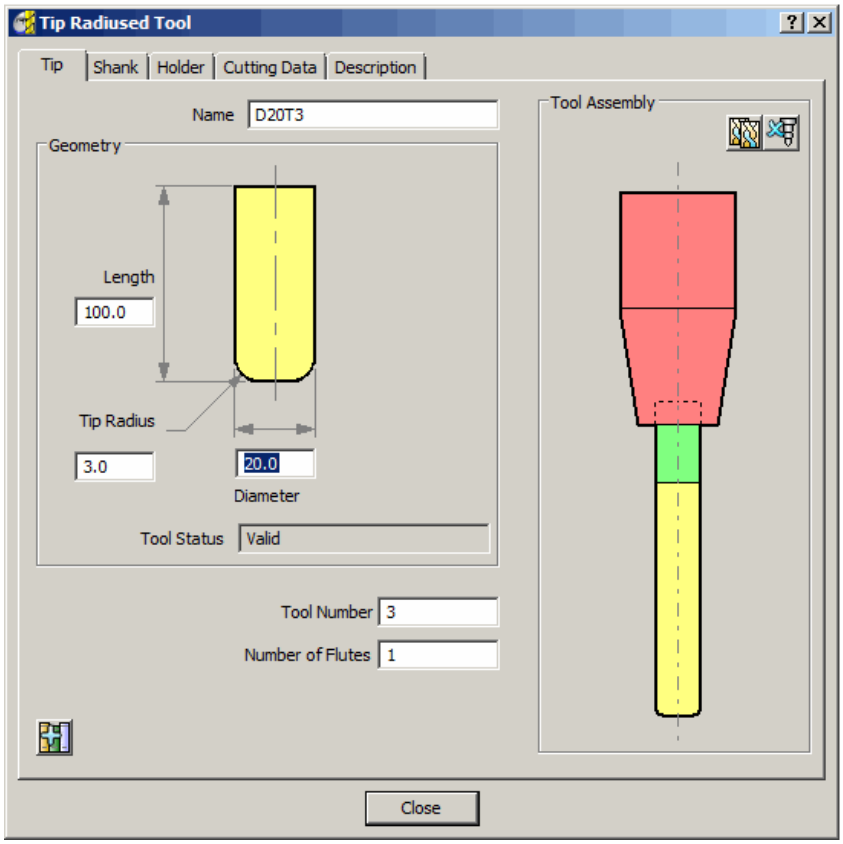
- **Import** the **model**:-

D:\users\training\PowerMILL_Data\five_axis\punch2\ punch2_insert.dgk

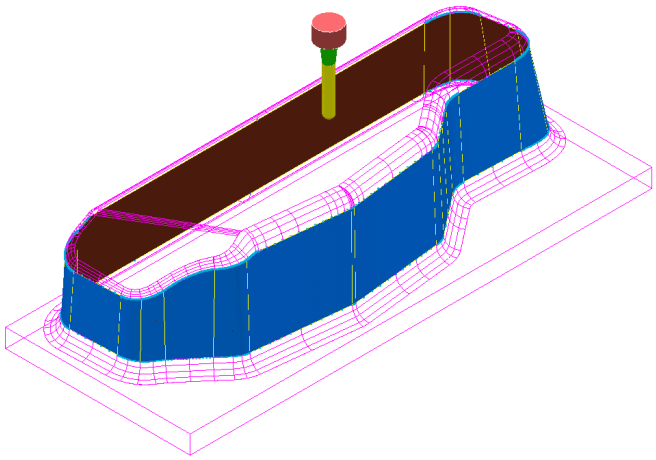


Initially a 3-Axis, **Constant Z finishing** strategy will be created, which when checked will be found to be in collision with the component form. The strategy will then be re-machined as a **Pattern finishing** strategy along with a suitable ***5-Axis*** **Tool Alignment**. A full collision check will again be performed on the final toolpath.

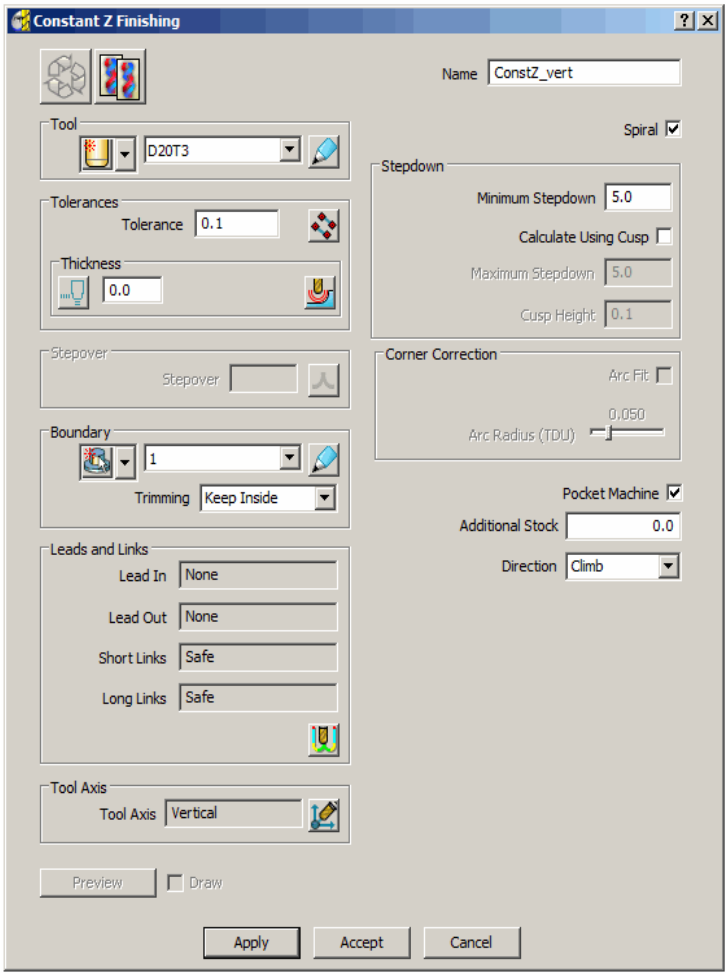
- Create a **Block** to the **Max\Min Limits** of the **Model**.
- Reset **Safe Z** and **Start Z**.
- In the **Start\End Point** form set ***Use*** **- Block Centre Safe** for both.
- Create a **Dia 20 tip radius 3** tool, ***Length*** **100**, ***Name*** **D20T3**.
- Add a **Shank** with ***Upper*** **Dia 20**, ***Lower*** **Dia 20** and ***Length*** **35**.
- Add a **Holder** with ***Upper*** **Dia 50**, ***Lower*** **Dia 35**, ***Length*** **50** and **Overhang 125**.
- Add a further **Holder** with ***Upper*** **Dia 50**, ***Lower*** **Dia 50**, ***Length*** **50**.



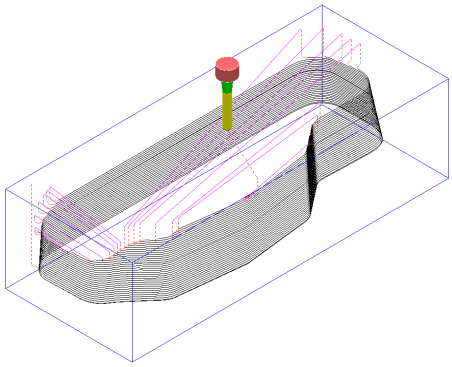
- Select the Sidewall **Surface** (shown shaded dark below) and **Create** a **Selected Surface Boundary** with the name **1**.



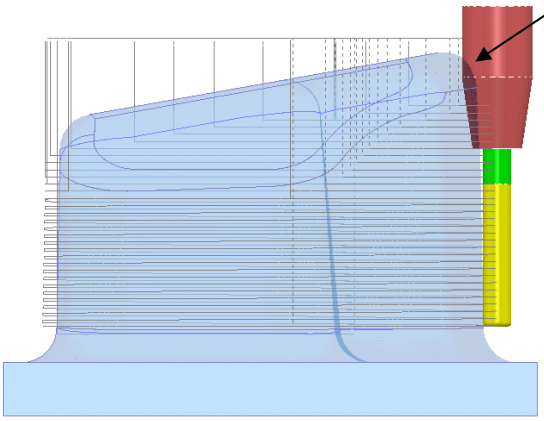
- Select the **Toolpath Strategies** icon  followed by the **Finishing** option located in the strategies form.
- Enter data into the **Constant Z Finishing** form <u>exactly</u> as shown below and **Apply**.



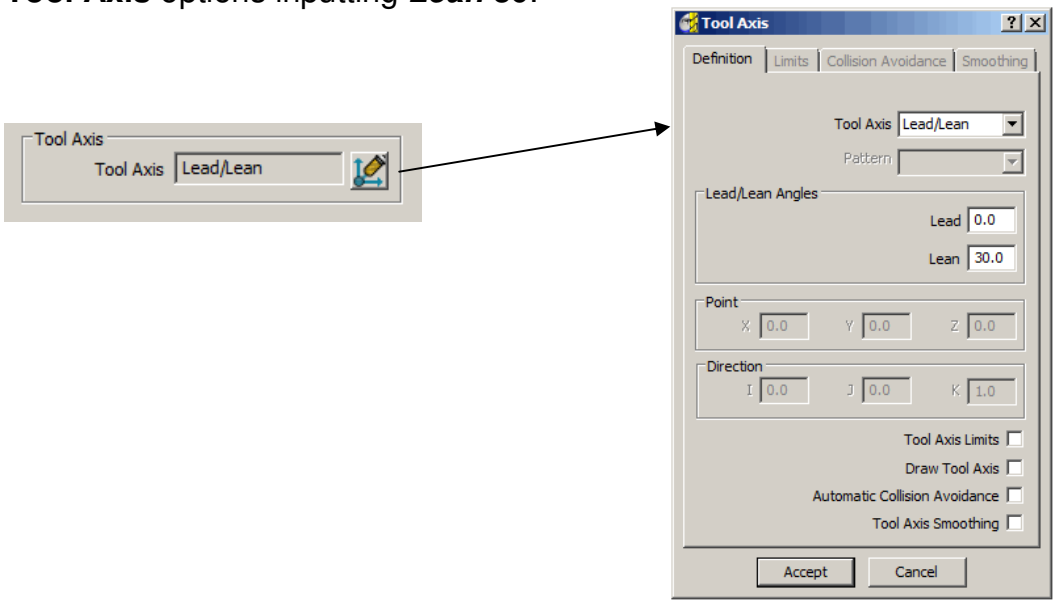
Toolpath ***Name*** is **ConstZ_vert**.



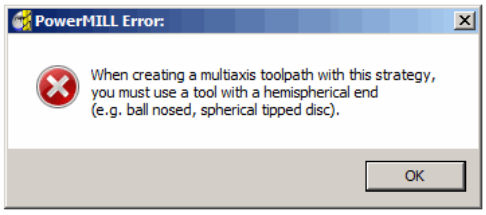
- **Attach** the **Active** tool, **D20T3** to a point along the base of the toolpath near to the highest point of the model to observe the **Collision** condition.



- Right click over the toolpath **ConstZ_vert** and from the local menu select **Settings**.
- Select make a **Copy**  of the toolpath and select **Lead\Lean** from the **Tool Axis** options inputting ***Lean* 30**.



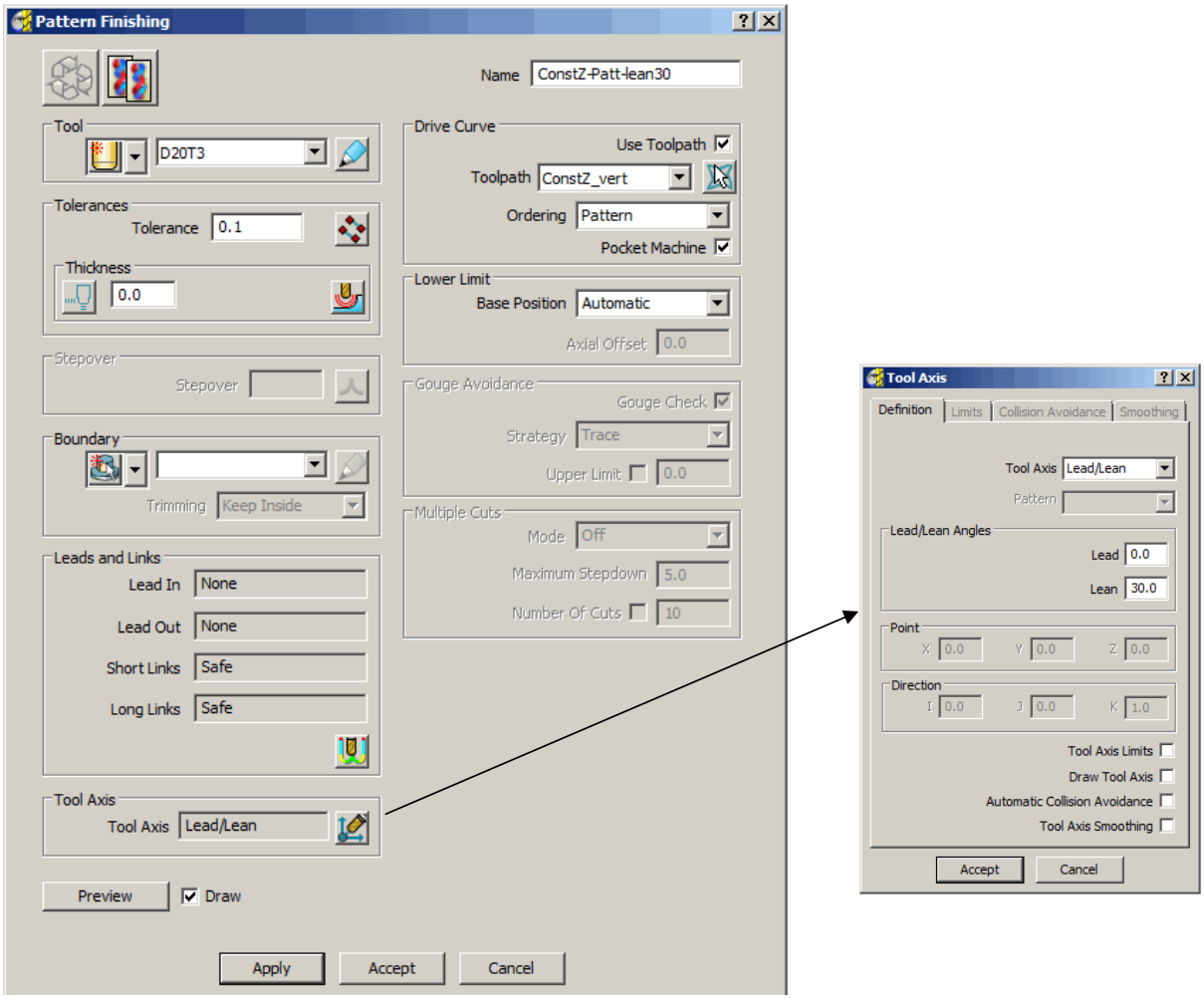
- Click **Apply** on the **Constant Z strategy** form and it will fail to produce a toolpath using a **tipped cutter** with a **multiaxis alignment** active.

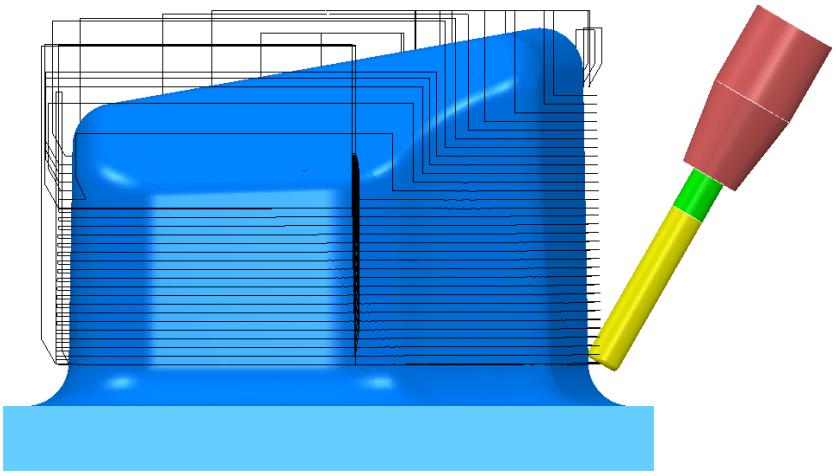


This confirms that multi-axis, **Tool Axis** definitions cannot be applied directly with a **Constant Z** finishing strategy where ***non-spherical*** tools are used.

- **Delete** the unprocessed **Constant Z** finishing strategy:- **ConstZ_vert_1**

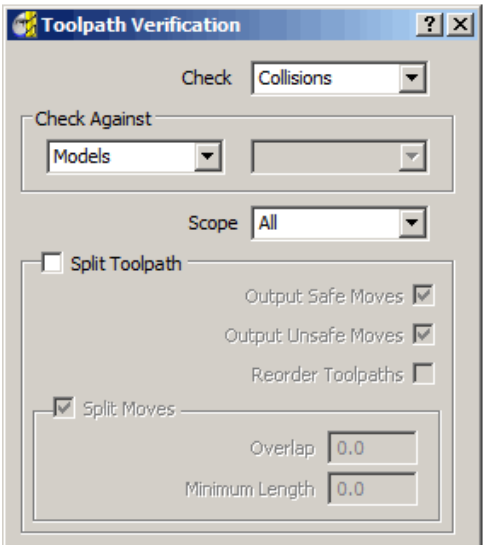
- Select the **Toolpath Strategies** icon followed by the **Finishing** option located in the strategies form.
- Enter data into a **Pattern Finishing** form <u>exactly</u> as shown below and **Apply**.





The original **Constant Z** finishing strategy has been used as the basis for a **Pattern** finishing strategy with one of **5-Axis** tool alignments applied.

- Open the **Collision checking** form by selecting the  icon in the top toolbar.
- Select the option ***Check* Collisions**.
- Set ***Scope* All**.
- ***Untick*** the box, **Split Toolpath**.
- **Apply** the form to receive the following message confirming that the toolpath is collision safe.



The lower fillet is yet to be machined and again requires a suitable **5-Axis** strategy to avoid a collision situation (This will be covered later in ***Chapter 9 - Tool Axis Limits).***

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

D:\users\training\COURSEWORK\PowerMILL-Projects\Punch2

Note:- This **Project** will be continued later during ***Chapter 8 -Tool Axis Limits***.

5. Embedded Pattern Finishing

Introduction

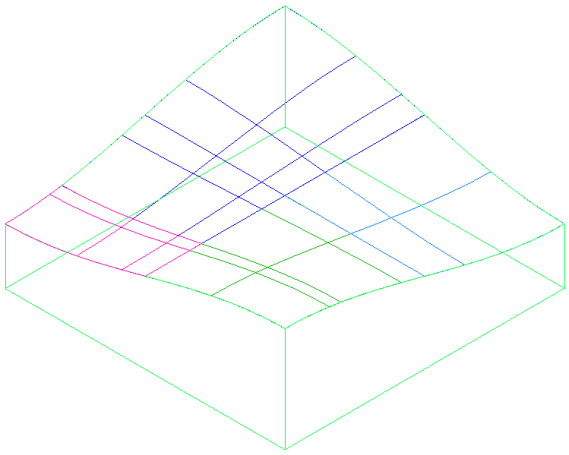
This strategy allows the user to produce an **Embedded Pattern Finishing** Toolpath using an **Embedded Pattern** to define the contact points of the toolpath. An **Embedded Pattern** is a curve lying on the model linked to its associated surface (or surfaces).

An **Embedded Pattern Finishing** toolpath can be used to specify the exact position of contact point or to use information about the underlying surface (for example surface normal) to determine the Tool Axis orientation when engraving.

Embedded Pattern Finishing – Engraving

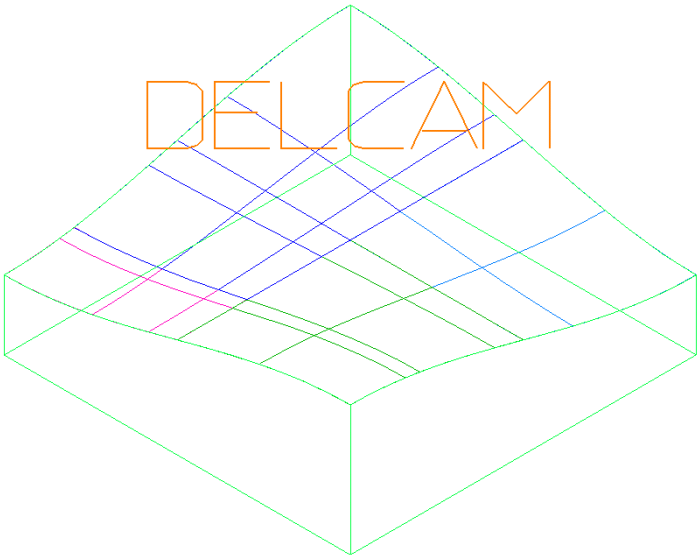
- ***Import*** the **model**:-

D:\users\training\PowerMILL_data\five_axis\ 5axis_Embedded_Pattern\ Embedded.dgk

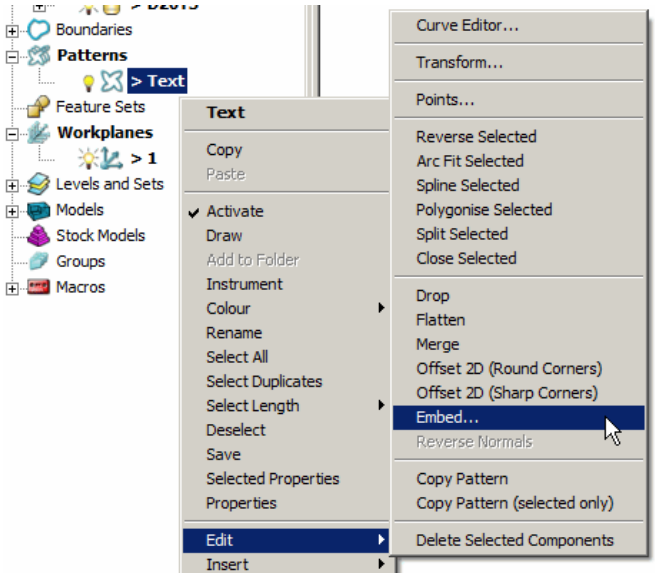


- Create a **Box** type **Block**, ***Calculated*** to the **model** dimensions.
- Reset **Safe Z** and **Start Z**.
- In the **Start and End Point** form set **Use - Block Centre Safe** for both.
- Create a **Taper Tipped Form** tool **Dia 5** – **Tip rad 0.5**, ***Length*** **25**, ***Taper height 15, Taper Angle 7.5,Name*** **TTR0.5A7.5**
- Add a **Shank** with ***Upper\Lower*** **Dia 5** and ***Length*** **10**.
- Add a **Holder** with ***Upper*** **Dia 30**, ***Lower*** **Dia 20**, ***Length*** **5** and **Overhang 35**.
- Add a **Holder** with ***Upper*** **Dia 30**, ***Lower*** **Dia 30** and ***Length*** **20**.
- **Right click** over **Patterns** in the **Explorer Window** and select **Toolbar** to display the **Pattern toolbar** at the top of the screen.
- **Create** a **Pattern**  and from **Insert** from **File** select the geometry:- **PowerMILL_data\five_axis\5axis_Embedded_Pattern\ Delcam_Pattern.dgk**

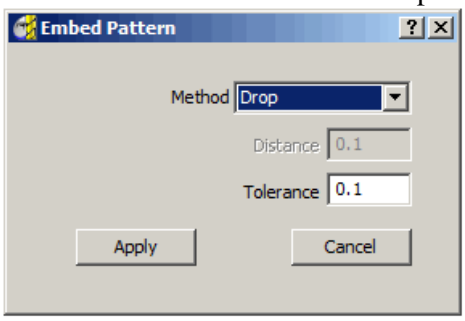
The **Pattern** contains the Text DELCAM and is positioned above the component surfaces. To inherit the surface normals for tool alignment it needs to be **embedded** before it can be used.



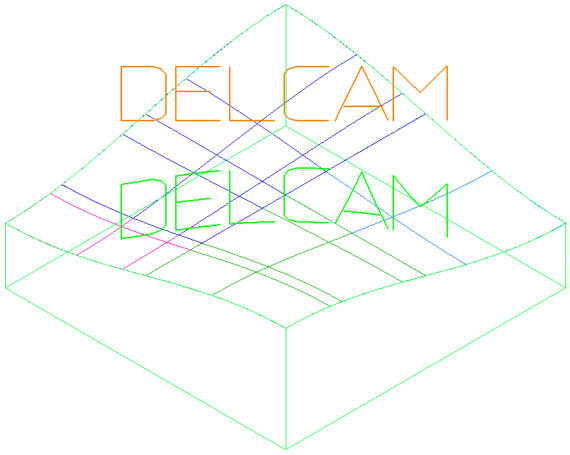
- **Rename** the Pattern '**Text**'.
- **Right click** the **Patterns** menu in the **explorer window** and select **Edit – Embed**.

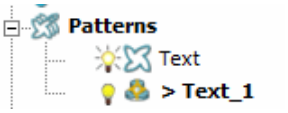


The **Embedded Pattern** form is opened.



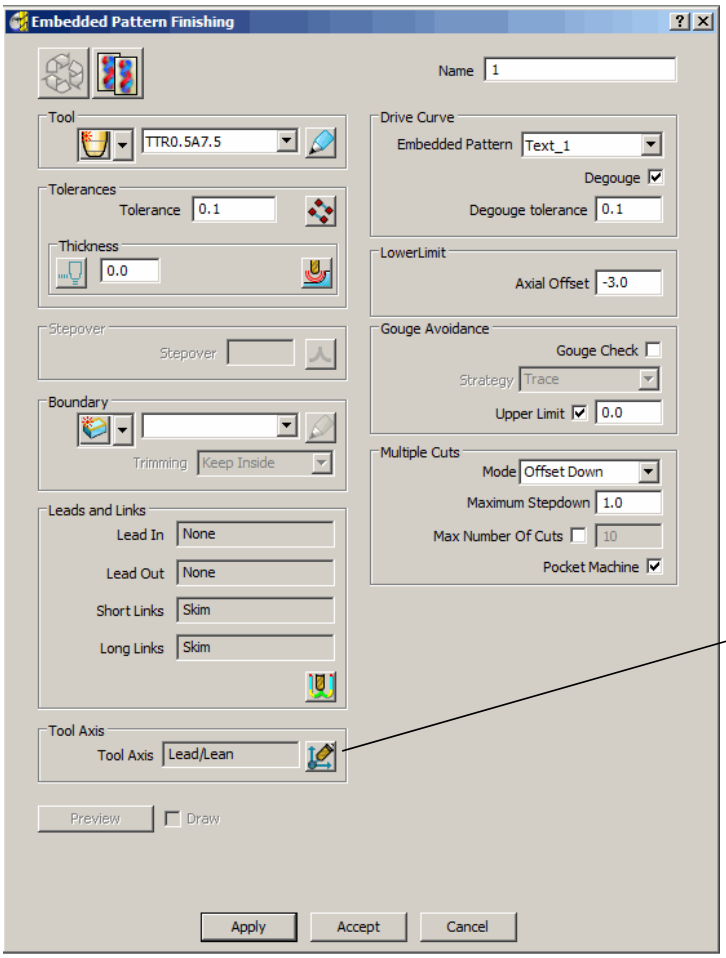
- Select **Drop** as the method and **Apply**.



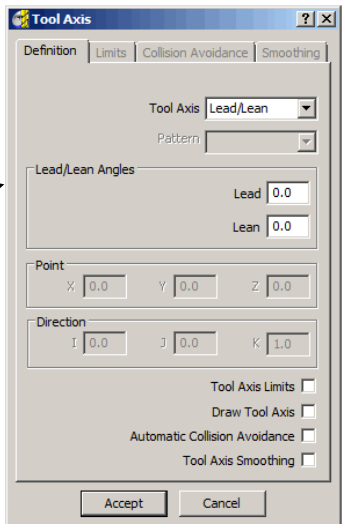


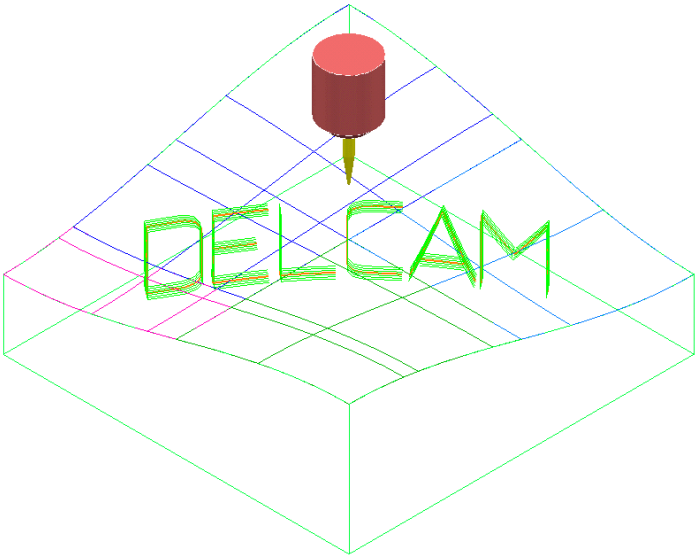
An **Embedded Pattern** called **Text_1** is created and identified by  in the **explorer**. The original **Pattern** is retained.

- Select the **Toolpath Strategies** icon  to open the **New** strategies form and select the **Embedded Pattern Finishing** option.
- Enter the values into the **Embedded Pattern Finishing** form <u>exactly</u> as shown and **Apply**.



- Select the ***Embedded Pattern*** **Text_1**.
- Untick **Gouge Check**.
- Tick **Upper Limit**.





- **Simulate** the toolpath.

The **Embedded Pattern** toolpath has been produced with the **tool** aligned ***normal*** to the **surface model**. Note: The toolpath is highlighted in the **explorer** window with 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**. To overcome this restriction, a **–ve Axial Offset** has been applied which results in the strategy registering a ***gouge warning***.

6. Five Axis Swarf Machining

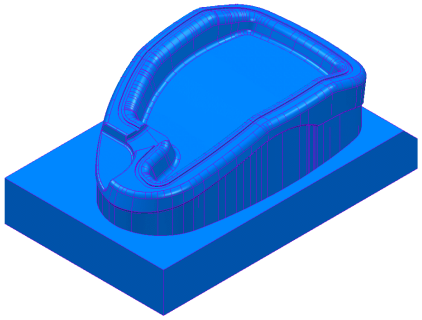
Introduction

The **Swarf Finishing** option creates a toolpath where (by default) the selected surfaces are machined with the side of the tool (**Tool Alignment** is **automatic**). A **Swarf Finishing** toolpath will only exist where the tool is able to remain in contact with the surfaces for the whole cutting depth. This means the **surfaces** to be machined must be **Swarfable** (Not Convex or Concave but Linear relative to an **automatic** tool alignment). It is possible for the user to apply a different tool alignment (such as **Lead\Lean** for deep sidewalls) but the selected surfaces must still be **Swarfable** for machining to occur. In cases where imported surfaces are intended 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**.

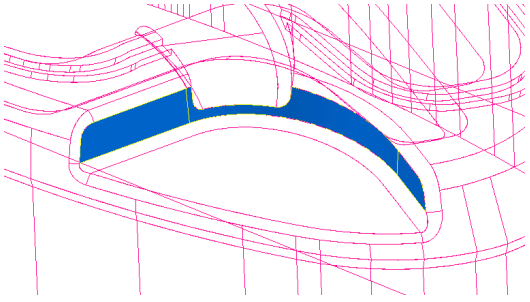
Swarf Finishing - example 1

- Select **File - Delete All** followed by **Tools - Reset Forms**.
- ***Import*** the model:-

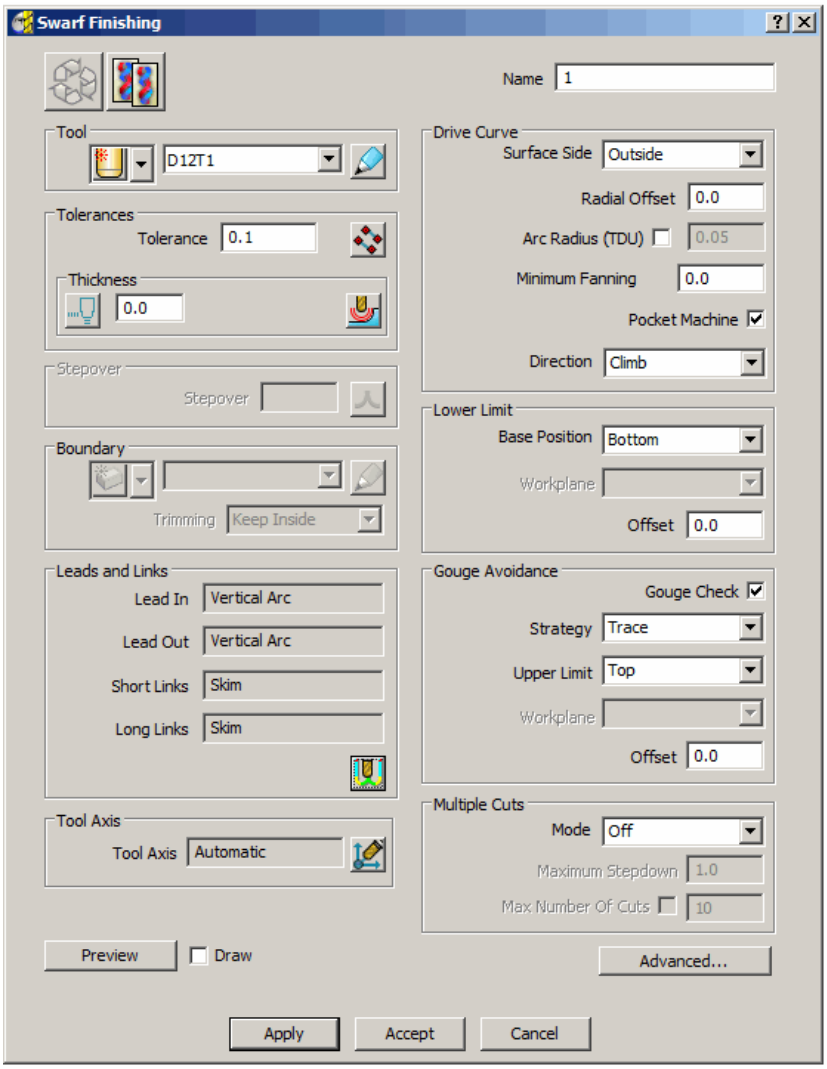
D:\users\training\PowerMILL_data\five_axis\swarf_mc\swarf_model.dgk

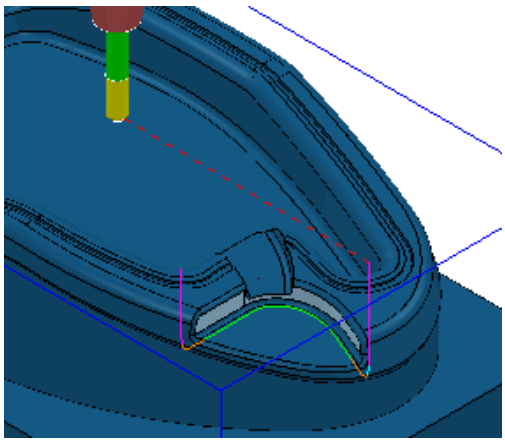


- Create a **Block** to the **Max\Min Limits** of the component **Model**.
- **Create** a **Dia 12mm x 1tiprad** cutter (**D12t1**).
- Reset **Safe Z** and **Start Z**.
- In the **Start\End Point** form ***Use*** - **Block Centre Safe**.
- Change the **View** to **ISO 4**  and ***select*** the ***local*** **surfaces** to be **Swarf** machined (as shown below shaded).



- Select the **Toolpath Strategies** icon  and from the **Finishing** form select the **Swarf Finishing** option.
- Enter the values into the exactly as shown below and **Apply** (Note:- The default **Tool Alignment** is **Automatic** for this strategy).

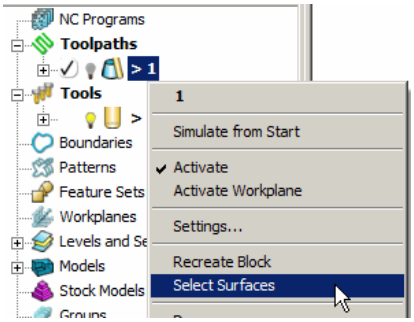


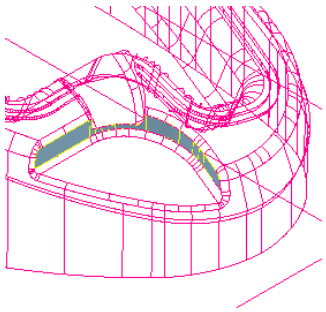


- Open the **Leads and links** form via the icon  and **Apply** the settings as illustrated in the following table:-

Z Heights:	**Skim distance** - **5.0** - **Plunge distance 5.0**
Lead In/Out:	**Vertical Arc** - **Distance 10** - **Angle 90** - **Radius 5.0**
Links:	**Short\Long\Safe** - **Skim**

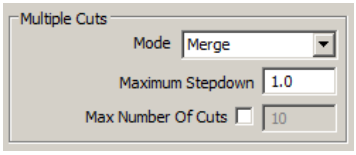
- **Deselect** the local surfaces, **Animate** the resultant toolpath and observe the changing angle of the tool as it **Swarf** machines the selected surfaces (Toolpath shown on previous page).
- Using the **Right** mouse key select the toolpath in the **explorer** to open the pull down menu the top half of which is shown below left.
- Use the **Left** mouse key to pick the option **Select Surfaces**, which will prompt the **surfaces** used during the toolpath creation to become ***selected*** again.

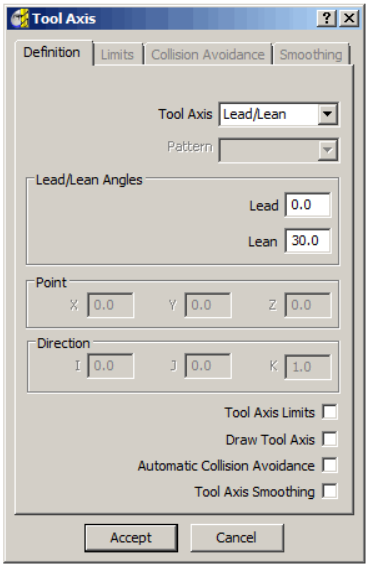


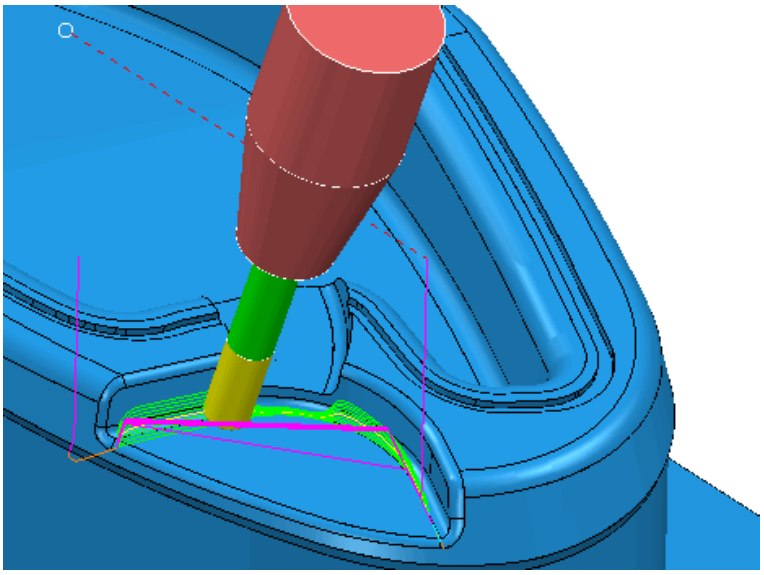


- Select **Settings** to reopen the existing **Swarf Finishing** strategy and select the icon to make a copy. 

- Select the **Multiple Cuts - *Mode*** to **Merge** in the ***main form*** along with a **Tool Axis Direction** set to **Lead 0** and **Lean 30** as shown below and **Apply**.

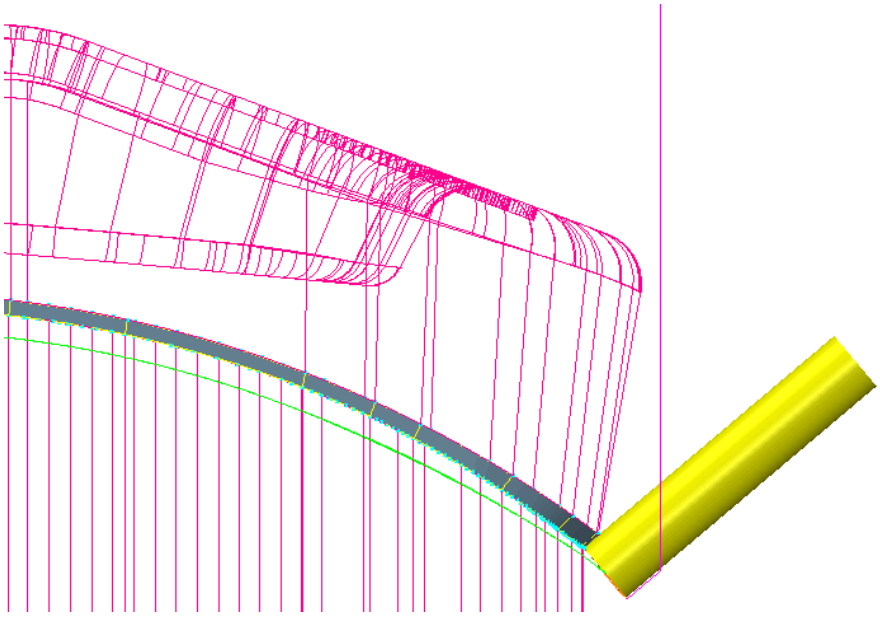






The new strategy steps down the selected surfaces while ***merging*** the **stepover** between the Upper and Lower contours. As a result there is less fragmentation of the toolpath.
Also, a **lean** angle of **30 degrees** has been applied relative to the **automatic (Swarf)** alignment (This makes it possible to machine deep sidewalls without having to use ***long reach tooling***).

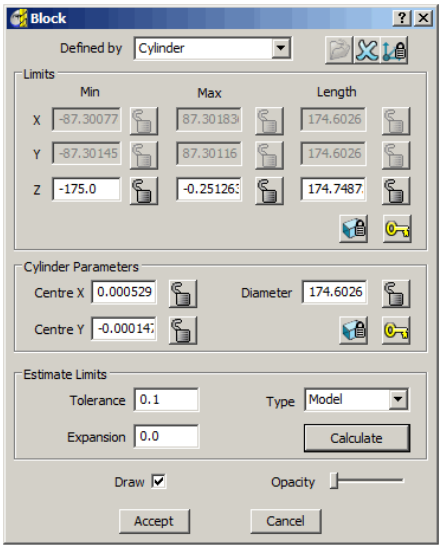
- Create a **Dia 10mm End Mill** (**em10**).
- Set up a **Copy** of the original, single pass, **Swarf Finishing** strategy to create a machining path, this time using the **Dia 10 End Mill** and aligned to the underside of the undercut **selected**, ***recess surface*** on the outer sidewall.

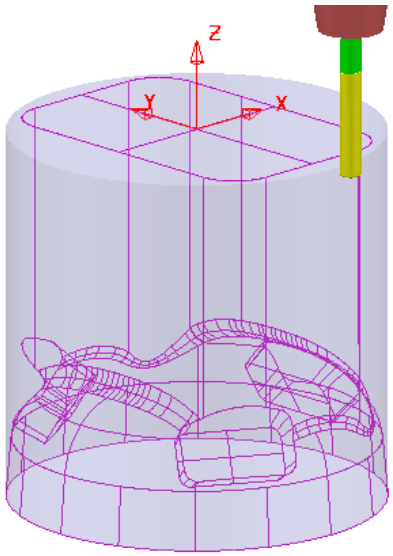


- **Deselect** the local surfaces, **Animate** the resultant toolpath and observe the changing angle of the tool as it **Swarf** machines the selected surface.
- **Delete** the **Model** and **Delete all - toolpaths**.

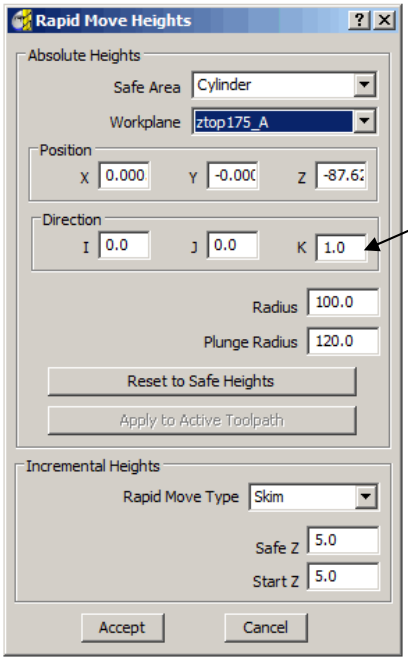
Swarf Finishing - example 2

- **Open** the **Project** - created earlier and **Saved** in:- **D:\users\training\COURSEWORK\PowerMILL-Projects\3+2example**
- **Activate** the **Dia 10 tiprad 1** tool **D10T1**.
- **Activate** the **Workplane - ztop175_A**.
- Open the **Block** form and select the option ***Defined by*** **- Cylinder** followed by clicking the **Calculate** tab (A cylindrical **Block** based on the **Model** dimensions will be created).





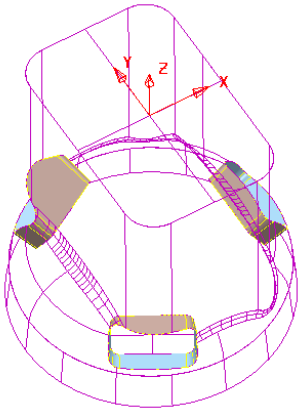
- Open the **Rapid Move Heights** form and set ***Safe Area*** **– Cylinder** and enter all values into the form **exactly** as shown below, before selecting **Accept**.



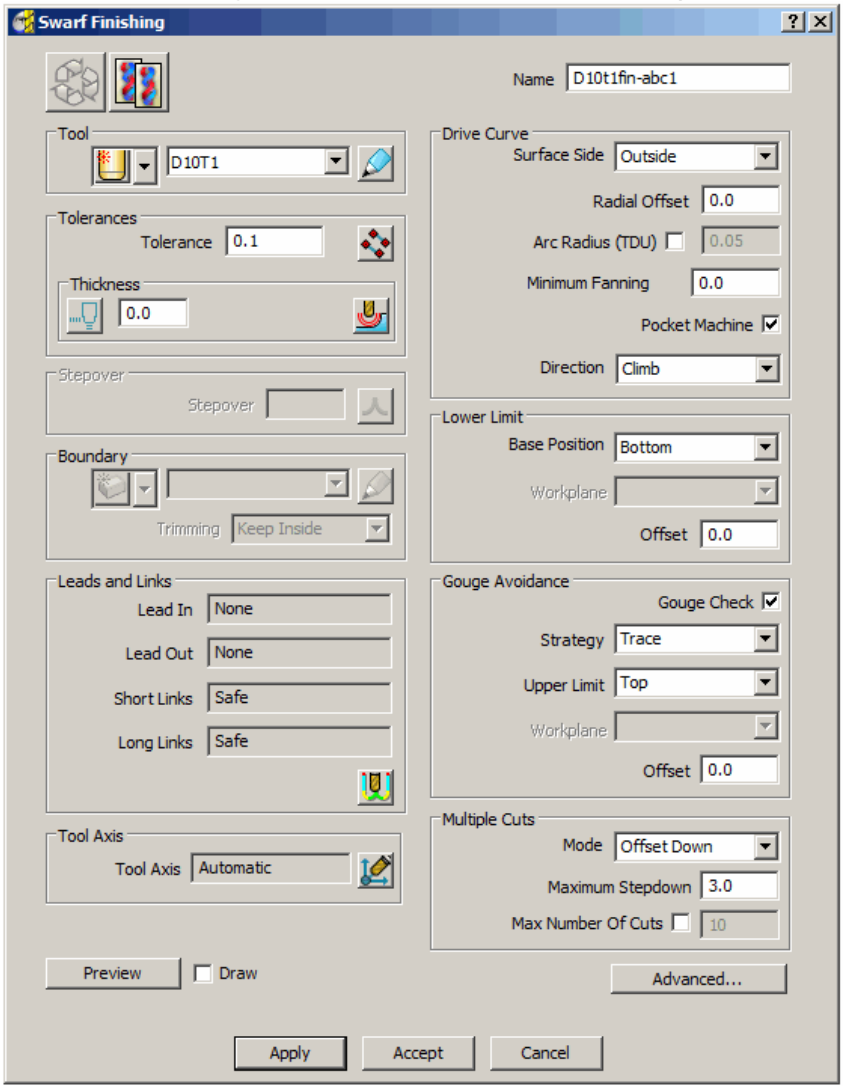
- Set the **Direction** (vector) to **I0 J0 K1**.

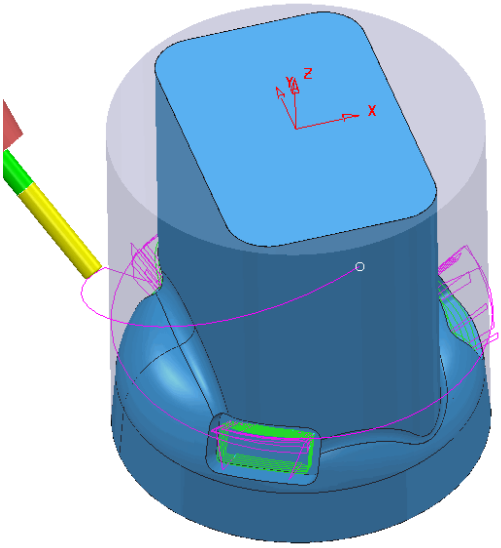
The use of **Safe Area - Cylinder** will produce smoother **link** moves that follow the defined ***cylindrical*** form instead of ***point to point***, linear moves around the component.

- **Select** the ***swarfable***, sidewall **surfaces** for all of the three Pockets.



- Select the **Toolpath Strategies** icon  and select the **Finishing** option located in the **New** strategies form.
- Enter the values into the **Swarf Finishing** form exactly as shown and **Apply** (Note:- **Tool Alignment** is **automatic** for this option).





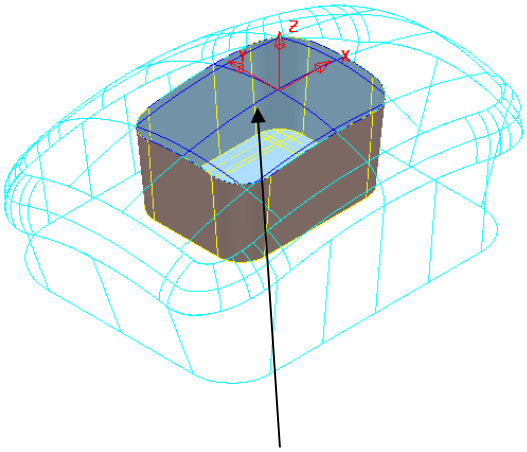
Note the **link** moves follow a path around the **Safe Area – Cylinder** as defined in the **Rapid Move Heights** form. This will provide both a smoother tool movement and re-orientation between separate machining areas.

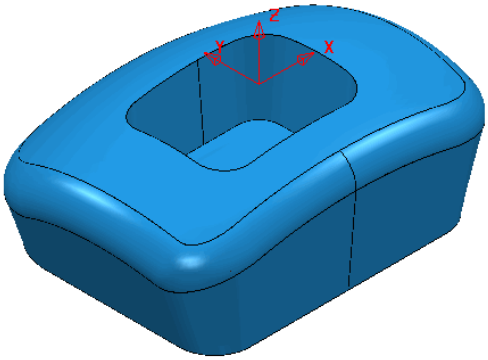
- **Simulate** the resultant toolpath and observe the changing angle of the tool as it **Swarf** machines the ***selected surfaces***.

Radial and Axial Thickness

- Select **File - Delete All** and **Tools - Reset Forms**.
- **Import** the <u>two</u> **models**, ***locnpad.dgk*** and ***pocket.dgk*** from :-

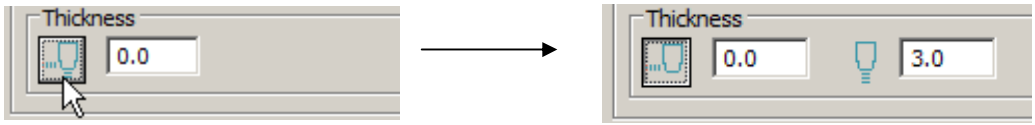
D:\users\training\PowerMILL_data\five_axis\locnpad_5axismc



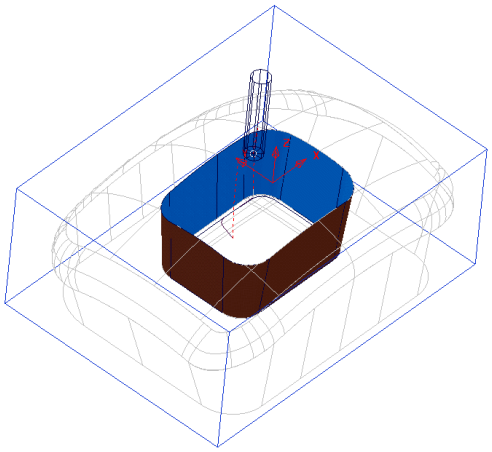


- **Delete** the **surface** covering the pocket.
- Create a **Block** to the component dimensions.
- Reset **Safe Z** and **Start Z**.
- In the **Start and End Point** form **Use - Block Centre Safe** for both.
- Create a **Dia 10 End Mill** with the *default* tool **Length** of **50**.

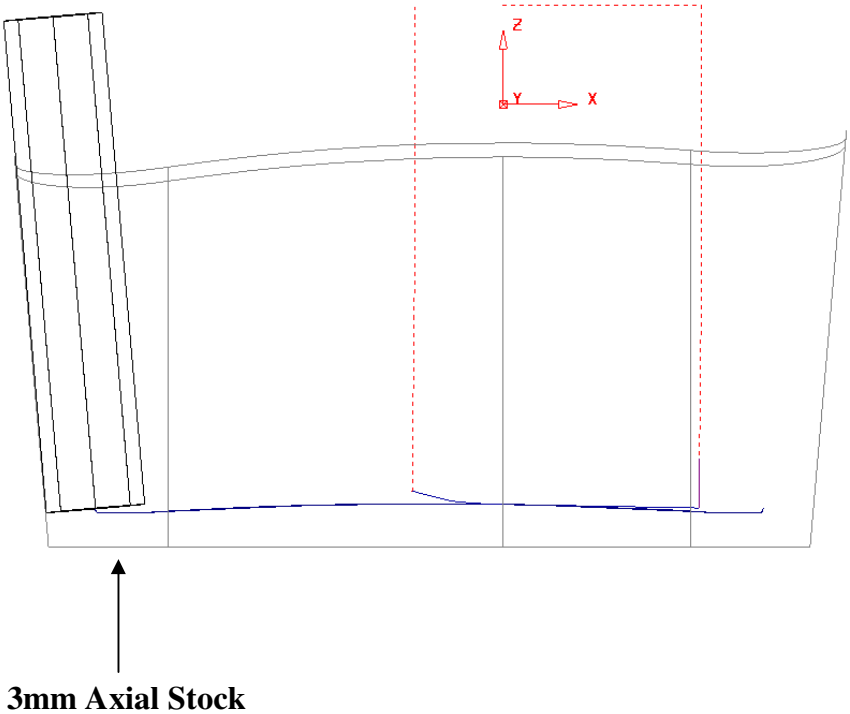
- Select the **Toolpath Strategies** icon and select the **Finishing** option located in the **New** strategies form.
- Select the **Swarf Finishing** form and select **Axial Thickness** option.
- Set **Thickness** settings as **Radial 0** and **Axial 3**.



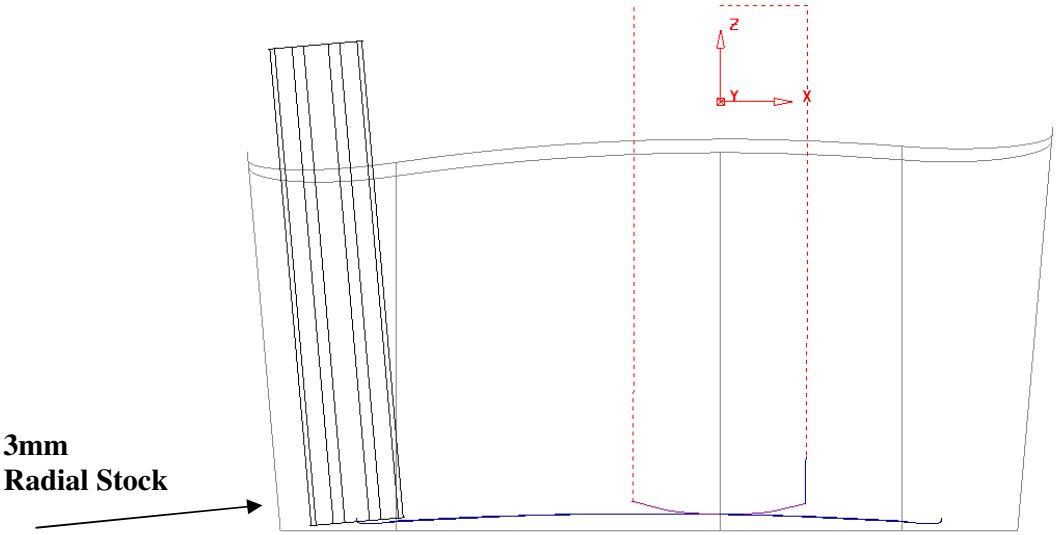
- **Select** the local **Surface** defining the wall of the pocket (shown below).



- Apply the **Swarf Finishing** form.



- Create another similar **Swarf Finishing** strategy apart from swapping the **Thickness** settings to **Radial 3** and **Axial 0**.

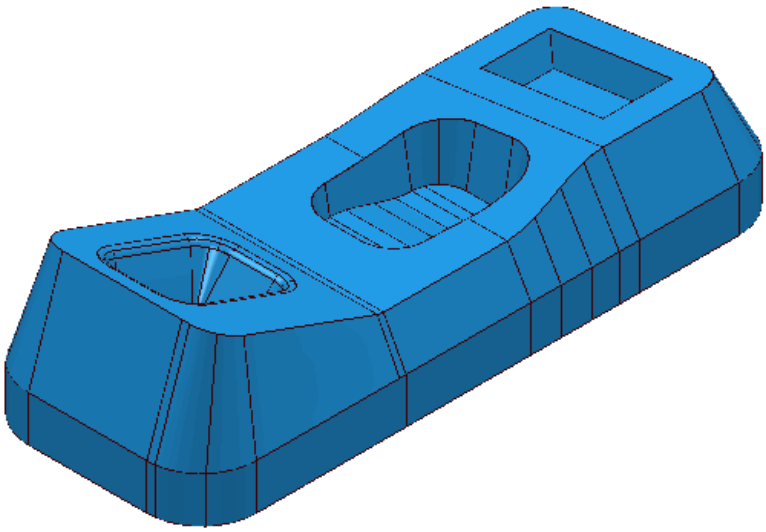


- **View** along the **Y Axis** to Compare the results as shown in the above two illustrations.

Wireframe Swarf Finishing

It is not uncommon for an imported model to be of a poor standard with such problem areas as small gaps, mismatching surfaces, and intended planer, surfaces that bulge slightly. One or more of these can create a range of problems in creating a high quality finish during the machining process. In some cases a poor quality model will cause a particular machining strategy to be unusable resulting in the user having to find an alternative method.
The model used in the following example contains a pocket in which part of the sidewall is unswarfable due to being slightly convex between the top and bottom edges.

- From the **main** pulldown menus select **File - Delete all**.
- **Import** the models **Wfrm-Swarf.dgk** from the local directory: **D:\users\training\PowerMILL_data \five_axis\Swarf_mc**.

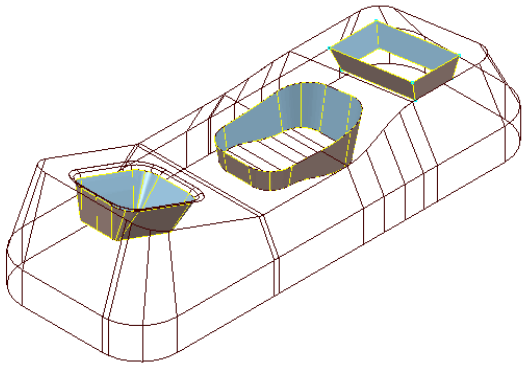


- **Activate** the **Workplane - Datum**.
- Create a **Block** to the **Max\Min Limits** of the component **Model**.
- Create a **Dia 5mm End Mill** cutter (**EM5**) using the following dimensions for the **Tool**, **Shank** and **Holder**:-

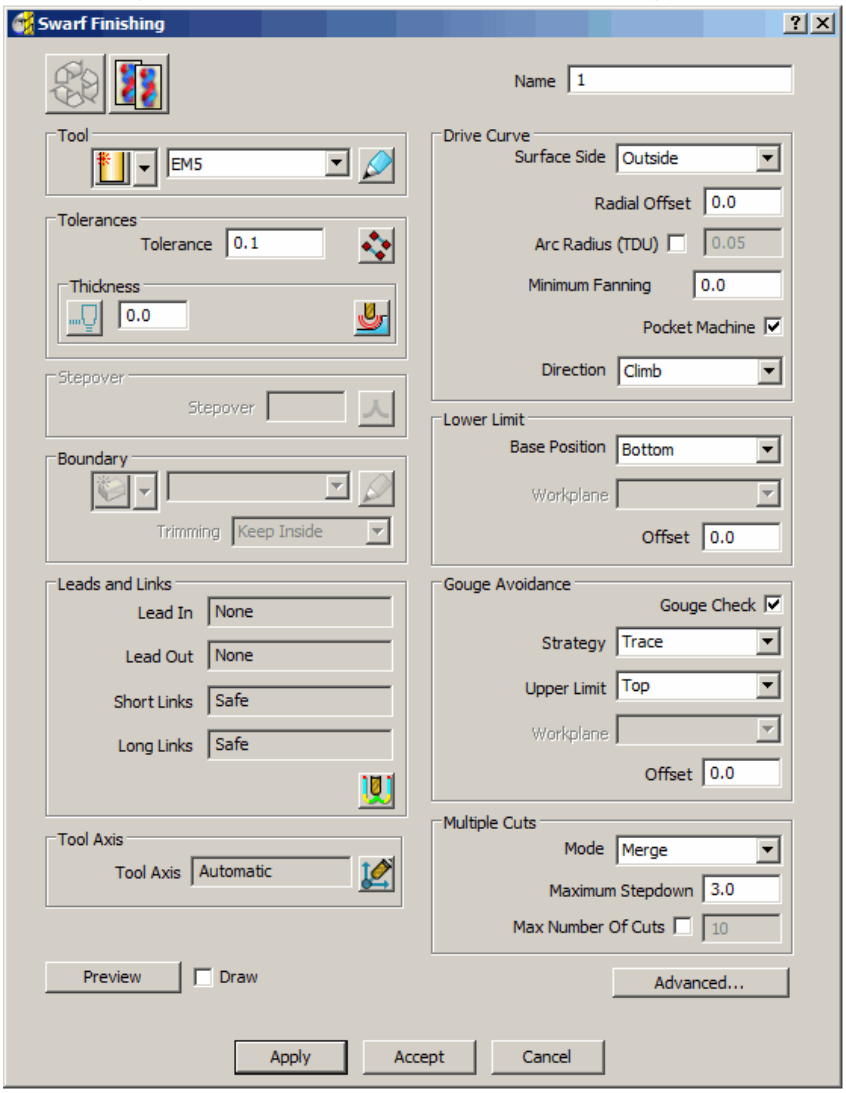
Tool	**Dia 5**	**Length 35**	
Shank	**Lower Dia 5**	**Upper Dia 5**	**Length 15**
Holder-1	**Lower Dia 15**	**Upper Dia 25**	**Length 15**
Holder-2	**Lower Dia 25**	**Upper Dia 25**	**Length 15**
Overhang 50			

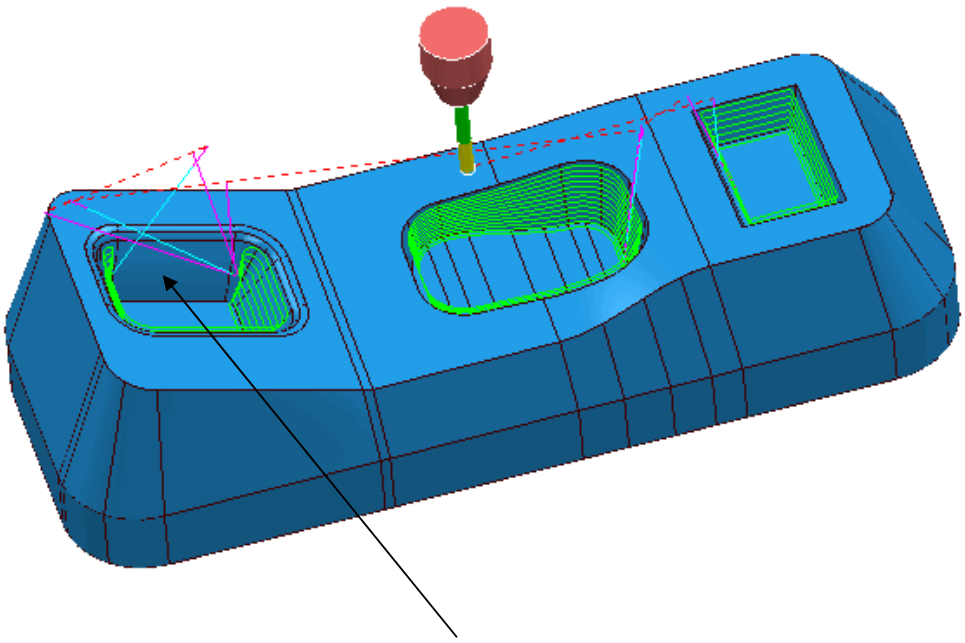
- Reset **Safe Z** and **Start Z**.
- In the **Start and End Point** form ***Use*** - **Block Centre Safe** for both.

- **Select t**he sidewall surfaces on each of the 3 pockets in the component.

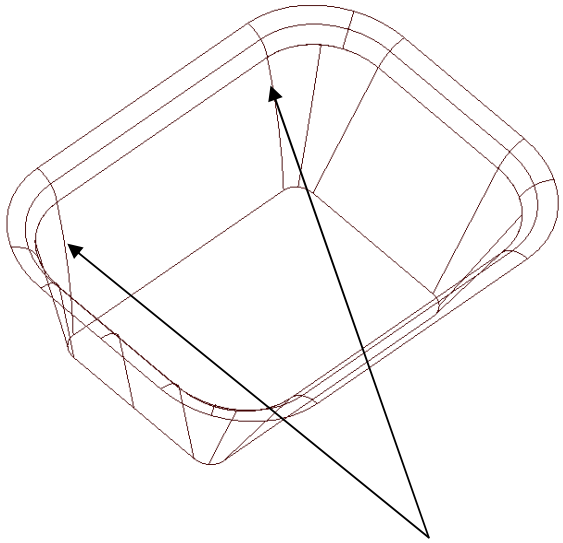


- Select the **Toolpath Strategies** icon  and from the **Finishing** form select the **Swarf Finishing** option.
- Enter the values into the <u>exactly</u> as shown below and **Apply** (Note:- The default **Tool Alignment** is **Automatic** for this strategy).



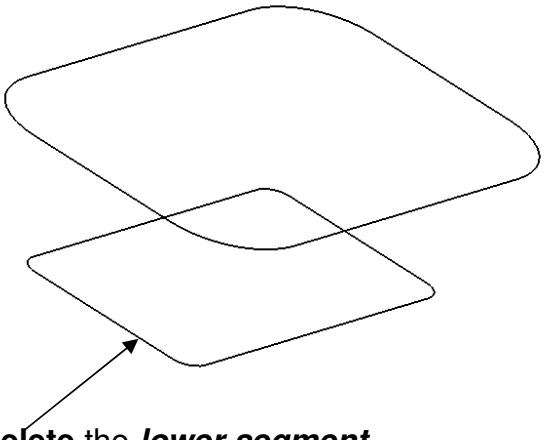


The **Swarf finishing** strategy has failed to completely machine all of the selected surfaces due to part of the sidewall being **unswarfable**. This could be resolved by fixing the affected surface using a **surface modeller**, ideally **PowerSHAPE**. Another alternative is to use **Wireframe Swarf Finishing** creates a strategy between a pair of wireframe curves defining the upper and lower edges of the surfaces to be machined.



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

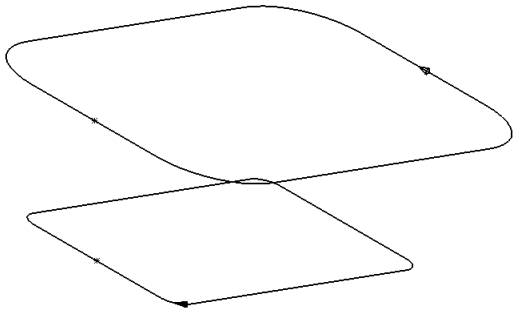
- **Select** the **surface** defining the ***angled sidewall*** of the left hand pocket (shown above).
- **Create** a **Pattern** named **Top** and right mouse click over it in the **explorer** selecting **Insert - Model** from the local menu.



- **Select** and **delete** the ***lower segment***.
- Create another, similar **Pattern** using **Insert Model** named **Bottom** and this time, **select** and **delete** the ***upper segment***.

For **Wireframe Swarf Finishing** to work the two **Patterns** must travel in the same direction. It is not necessary to align the start points on the upper and lower **Patterns** as it is the first point on the lower **Pattern** controls the start position of the resultant toolpath.

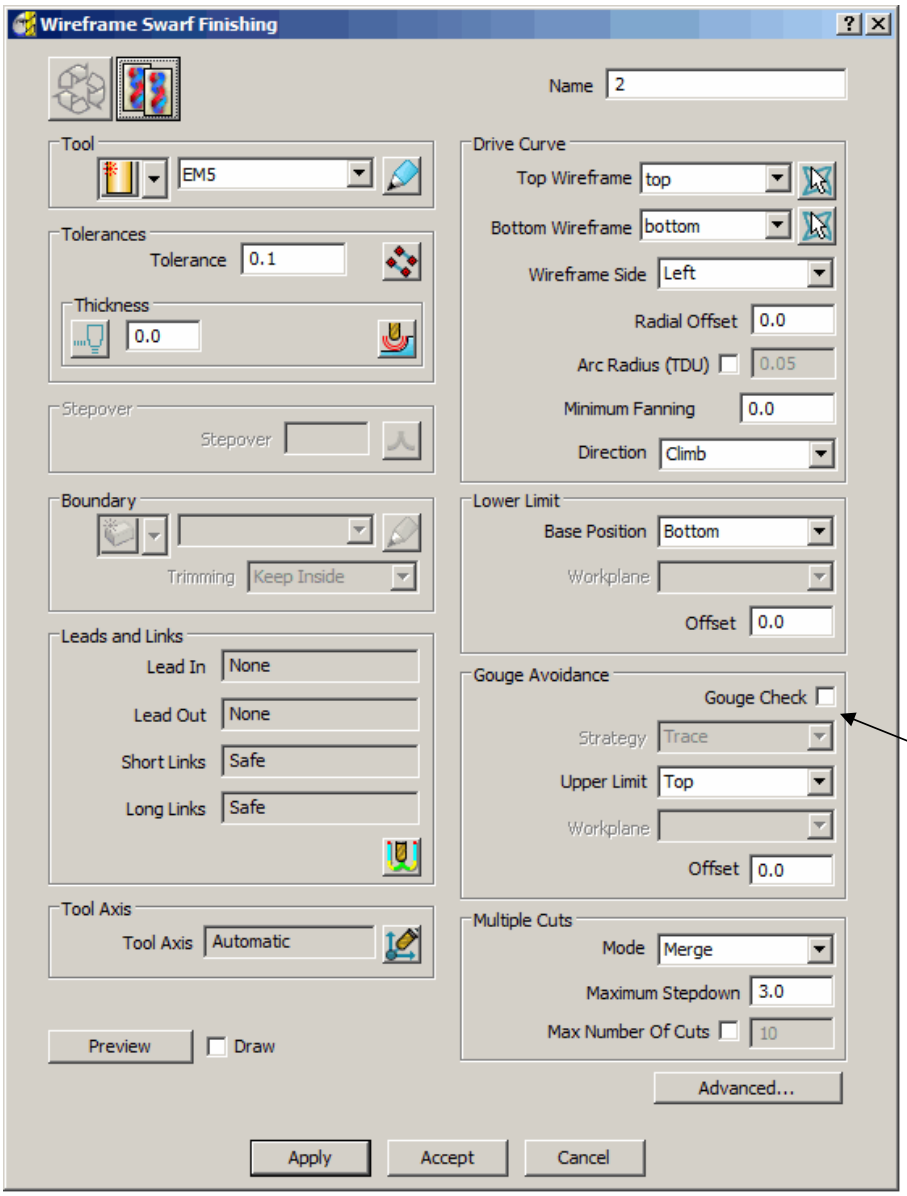
- To identify whether the two **Patterns** are suitable for use right click over each one in turn and select **Instrument** to display the start point and direction.



As can be clearly seen the two segments both already have a suitably positioned start point but 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.

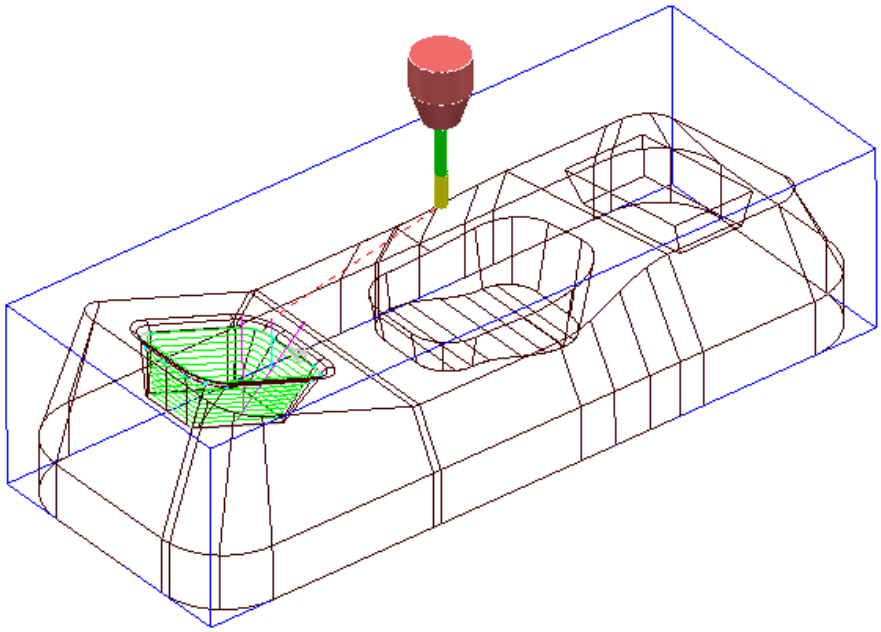
- **Select** the **lower segment** and Right Mouse click over the **pattern** to open the local menu and select **Edit - Reverse Selected**.
- With the **lower segment** still **selected** Right Mouse click over the **pattern** to open the local menu**,** select **Copy Selected Components** and **Rename** the resultant new **Pattern** as **Bottom**.
- **Select** and **Delete** the **lower segment** in the **Pattern** named **Top**.

- Select the **Toolpath Strategies** icon and from the **Finishing** form select the **Wireframe Swarf Finishing** option.
- Enter the values into the exactly as shown below (Note:- The default **Tool Alignment** is **Automatic** for this strategy).



- Untick **Gouge Check**.

Note:- An alternative to switching **Gouge Check** off would be to **Apply** - **Component Thickness** with the ***angled sidewall surface*** set to **Machining Mode - Ignore**.



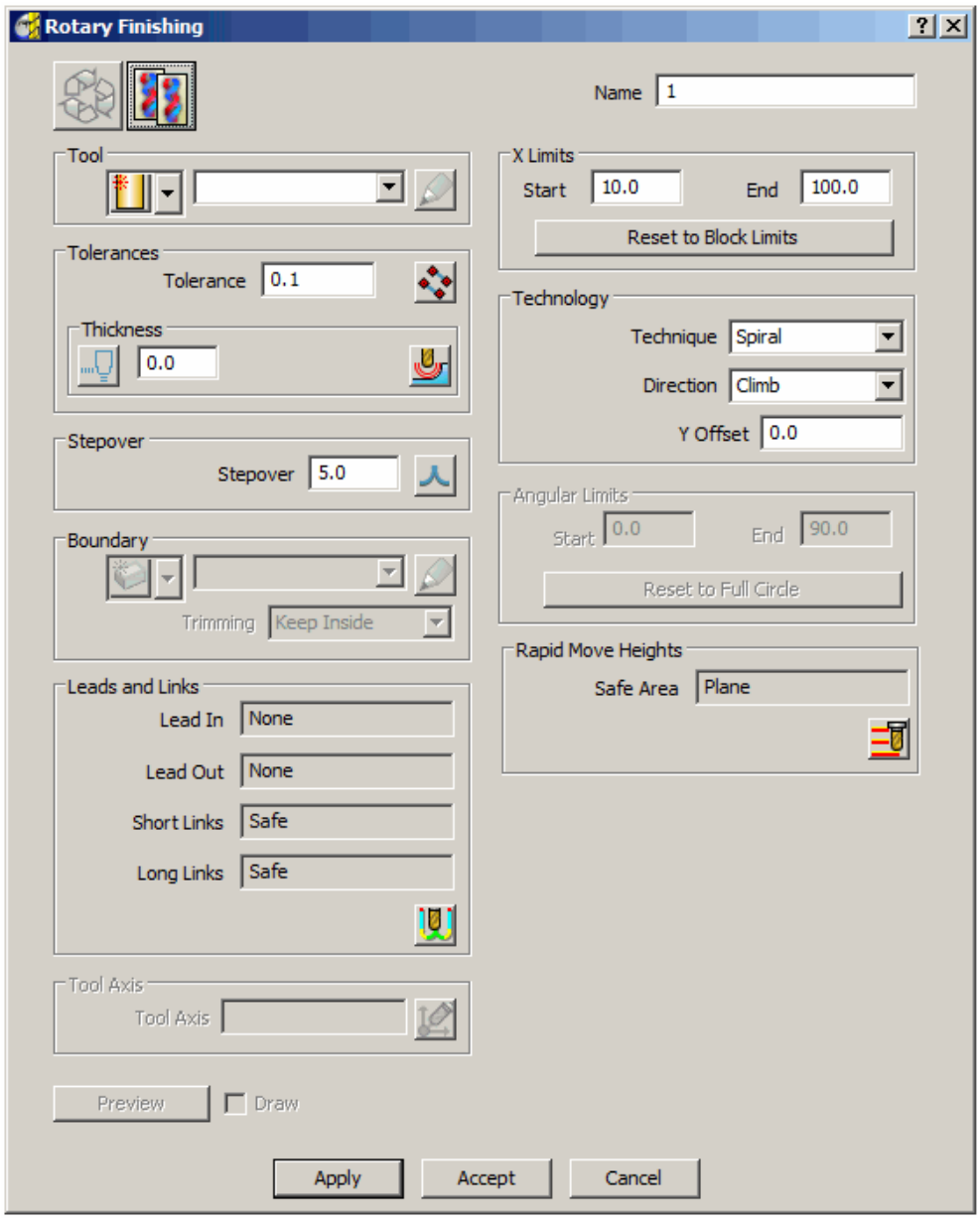
The new **Wireframe Swarfing** strategy has successfully been applied between the 2 **Patterns** and has been set to **Ignore** the original surface. This will register as a **gouge free** strategy.

It would have been possible to run the strategy with **Gouge Check** unticked, which would allow the same toolpath to be created without setting the selected surface (**Thickness**) to be **Ignored**. However the resultant toolpath would then be permanently registered with a **Gouge** warning.

7. Four Axis Rotary Machining

Introduction.

This **Finishing** strategy is designed for machining a component mounted on a fourth, programmable **Rotary Axis**. During milling, the component rotates around the rotational X-axis while the cutter performs simultaneous 3-Axis movements.



The main options available in the above form will be summarised on the following page.

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.

Technique

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

Direction

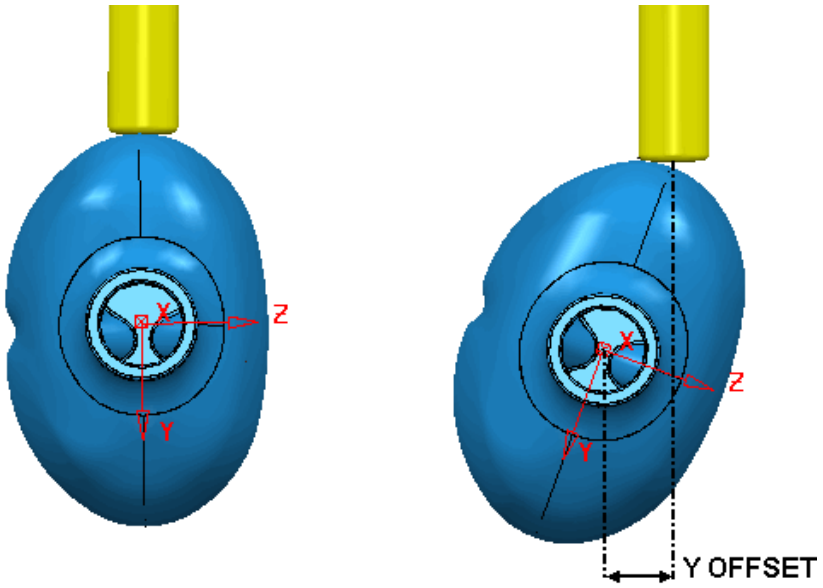
This option determines whether **Climb**, **Conventional**, or **Any** milling directions are used.

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.

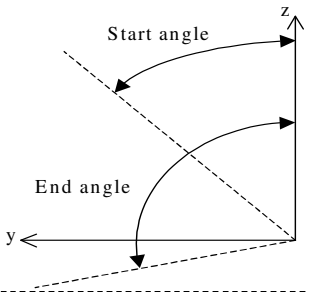
Y Offset

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 form is only available when using the **Circle** or **Line** technique. The angular limits are defined between a **Start** and **End** angle.

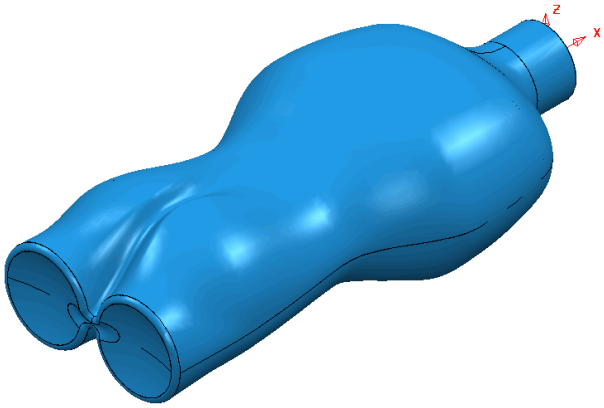


The angular limits are measured in an anti-clockwise direction when viewed along the positive X-axis. The area machined is between the start and end angles.

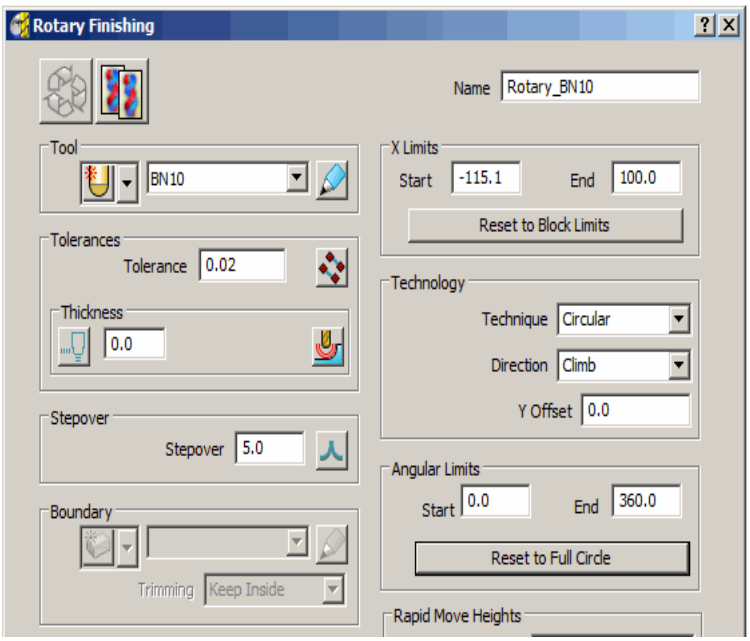
Circular Rotary Machining

In this example a model of a bottle with its centre along the X-axis will be used. Using the **Circle** technique, the job rotates with the tool aligned to a fixed direction. While the component rotates, the tool moves back and forth along it's axis to generate the sectional form. The tool then steps over by the **Pitch** value and the rotational machining process is repeated.

- Select **Delete All** and **Reset forms**.
- **Import** the model:-
  D:\users\training\PowerMILL_data\Examples\rotary_bottle.dgk

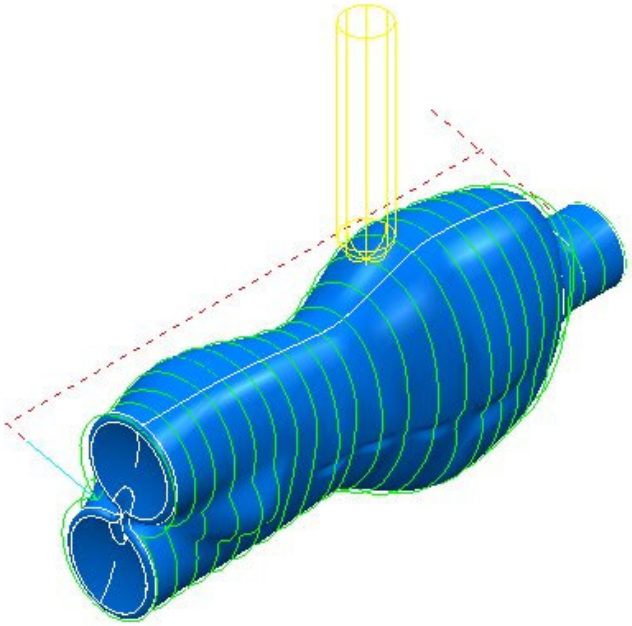


- Calculate the **Block** to the model limits and define a **Ball nosed** tool **Dia 10** named **BN10**.
- In the **Rapid Move Heights** form select **Reset to Safe Heights**.
- In the **Start and End Point** form set **Use - Absolute X0 Y0 Z40**.
- Select the **Toolpath Strategies** icon and choose the **Rotary Finishing** option then **OK**.



- Enter **Name** – **Rotary1_BN10**
- Click on the **Reset To Block Limits** button from the **X Limits** section.
- Define a **Stepover 5**.
- Select **Technology Direction - Climb**
- **Apply** and **Cancel** the form.

- Right click over the toolpath **Rotary1_BN10** in the **explorer** and select **Simulate from Start** to open the **Simulation Toolbar**.
- On the **Simulation Toolbar** select the **Tool view point** icon  .
- 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).

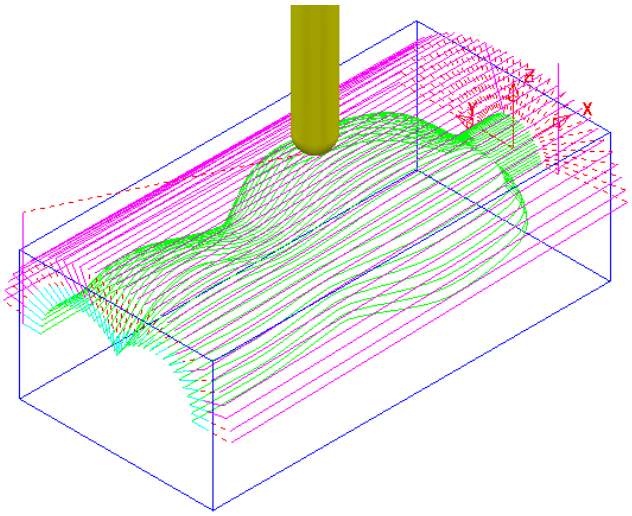
In the above example each section is machined in the same **Climb** milling direction. The entire length of the job is machined since the **X Limits** are set to the block limits. Selecting **Conventional** will produce tool tracks travelling in the reverse direction and **Any** will produce alternate **Climb** and **Conventional** tool tracks along the job.

- **Recycle** the toolpath, select **Any** from the **Technology Direction** pull-down button and click on **Apply** then **Cancel**.
- **Simulate** the toolpath as before to observe how the tool reverses direction with every new section machined.

Line Rotary Machining

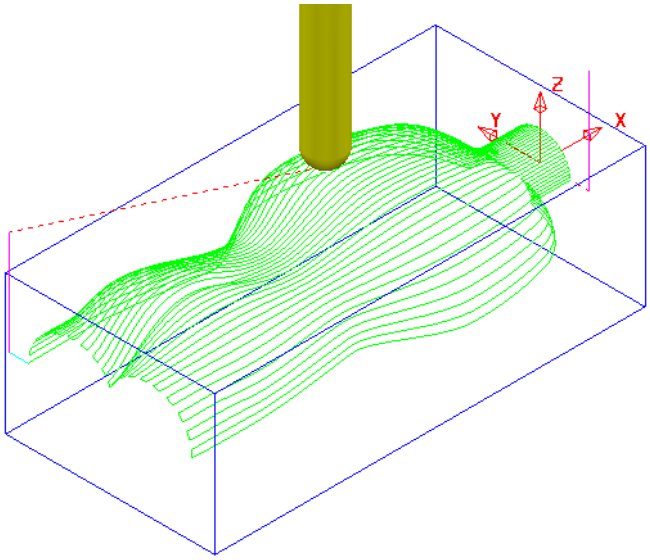
Using the **Line** technique, the tool feeds in the X direction following the component form. At the end of each pass the tool retracts and moves above the start of the next pass. At the same time the rotary axis indexes by the angular stepover and the tool then leads onto the next machining move.

- **Recycle** the same toolpath again as in the previous example, select the **Line** technique using **Technology Direction** - **Climb** milling.
- For the **Angular Limits** - **Start Angle**, enter **90** and **End Angle**, enter -**90**.
- Select the **Leads and Links** icon, set the Z Heights **Skim distance** to **20**.
- **Set Links** to **Skim**.
- Click on **Apply and Close**.



- **Simulate** the resultant uni-directional toolpath shown above.

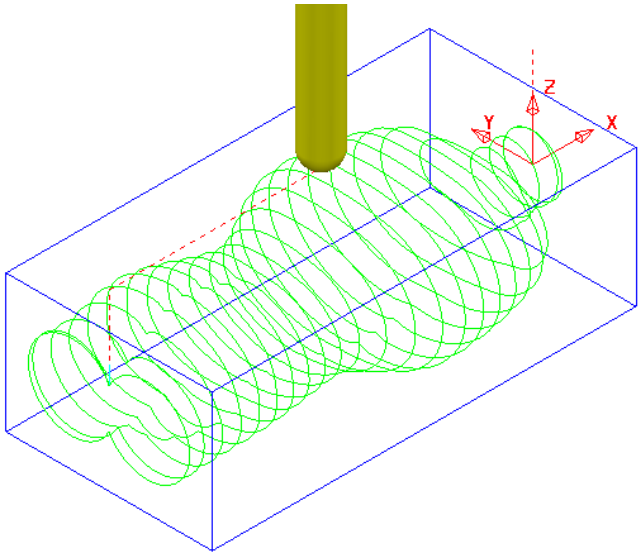
- Recycle the toolpath and change the **Technology Direction** to **Any**.
- **Apply** the form and **Simulate** the resultant bi-directional toolpath.



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.

- Recycle the toolpath again selecting **Technique - Spiral** and **Direction - Climb** milling.
- Click on **Apply** to produce the toolpath shown below.



- **Simulate** the new toolpath.

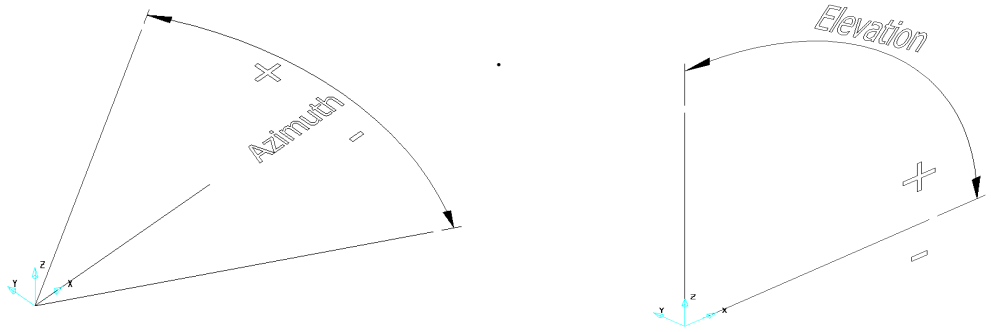
8. Tool Axis Limits

Introduction

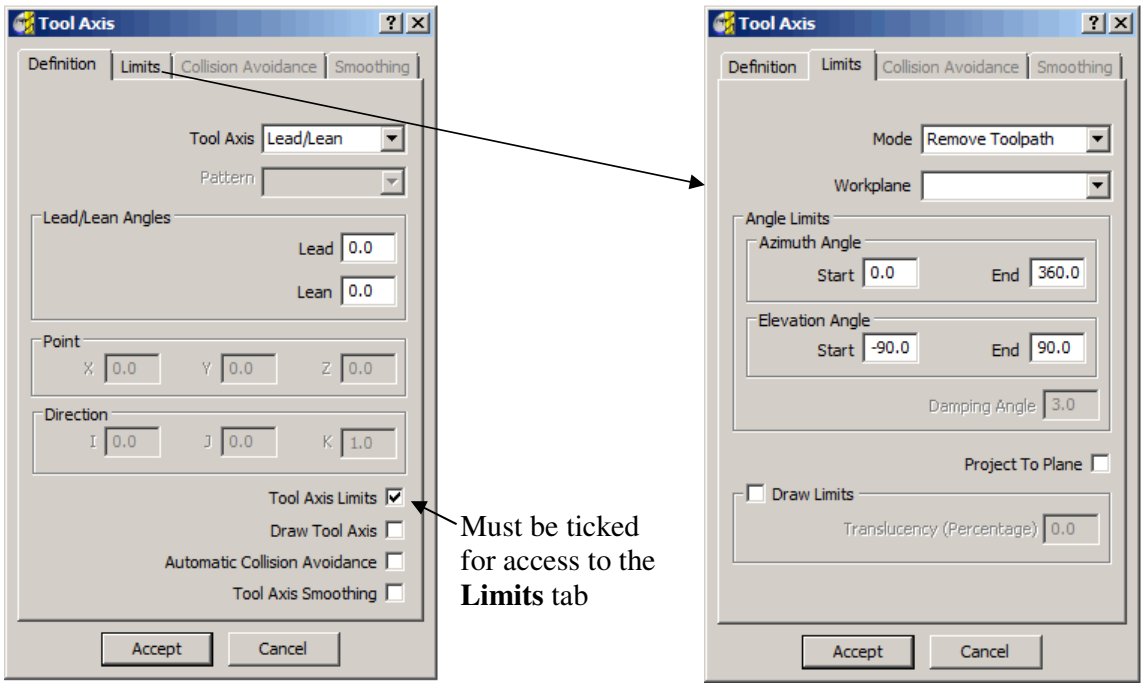
It is possible to set the **Toolaxis limits** of the machine tool within **PowerMILL**. This enables the rotary working envelope to be defined and not exceeded when creating multi-axis toolpaths. Due to differing configurations between different machine tools the angular limits are translated in terms of **Azimuth** and **Elevation** angles in **PowerMILL**.

Azimuth and Elevation.

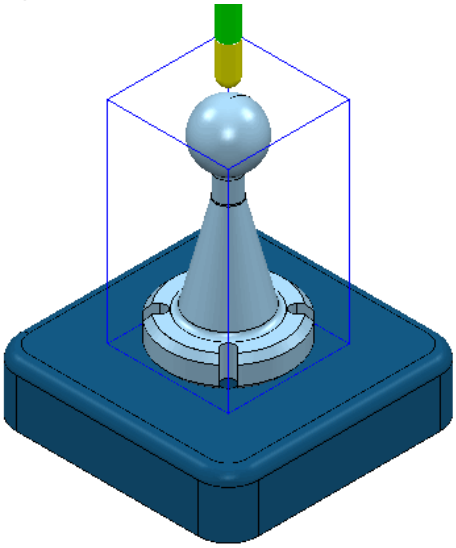
The **Azimuth** is the angle from X 0° anticlockwise around the **XY** plane. The **Elevation** is the angle that is lifted upwards (+90°) or downwards (-90°) from the **XY** plane.



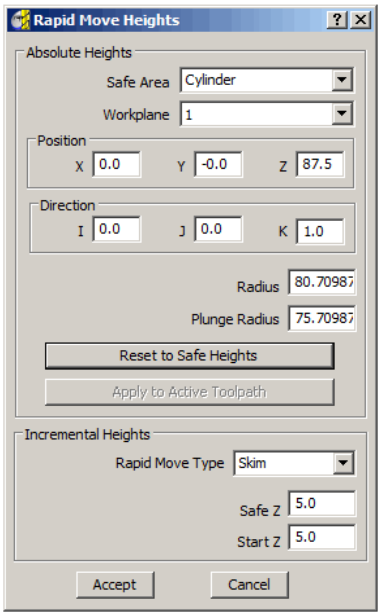
The **Tool Axis Limits** form is accessed from within the **Tool Axis** direction dialog. **Limits** can only be applied if the ***tool axis*** is set to anything other than **Vertica**l or **Fixed Direction** and the **Tool Axis Limits** box ***ticked***.



- **Delete all** and **Reset Forms**.
- **Import** the two **models**:- **D:\users\training\PowerMILL_Data\five_axis\Tool_Limit\ JoyStick.dgk** and **JoyStickBase.dgk**



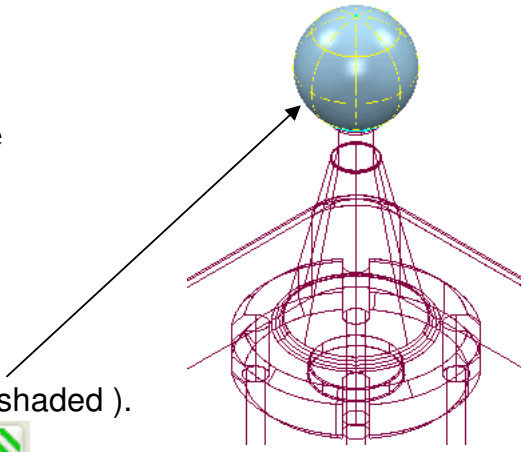
- With the ***JoyStick* model** form only (Not the ***Base***) selected, **Calculate** a **Block** to **Min\Max** limits.
- Create a **16mm Ballnose** tool **Length 60** named **BN16**.
- **Add** a **Shank** component **Upper Dia 16 Lower Dia 16 Length 40**
- **Add** a **Holder** component **Upper Dia 50 Lower Dia 35 Length 40**
- **Add** a **Holder** component **Upper Dia 50 Lower Dia 50 Length 60 Overhang 90**
- Open the **Rapid Move Heights** form selecting the **Safe Area** definition as **Cylinder** with a **Direction** vector **I 0 J 0 K 1** and click on **Reset to Safe Heights** to automatically set suitable values for both the **Radius** and **Plunge Radius**.



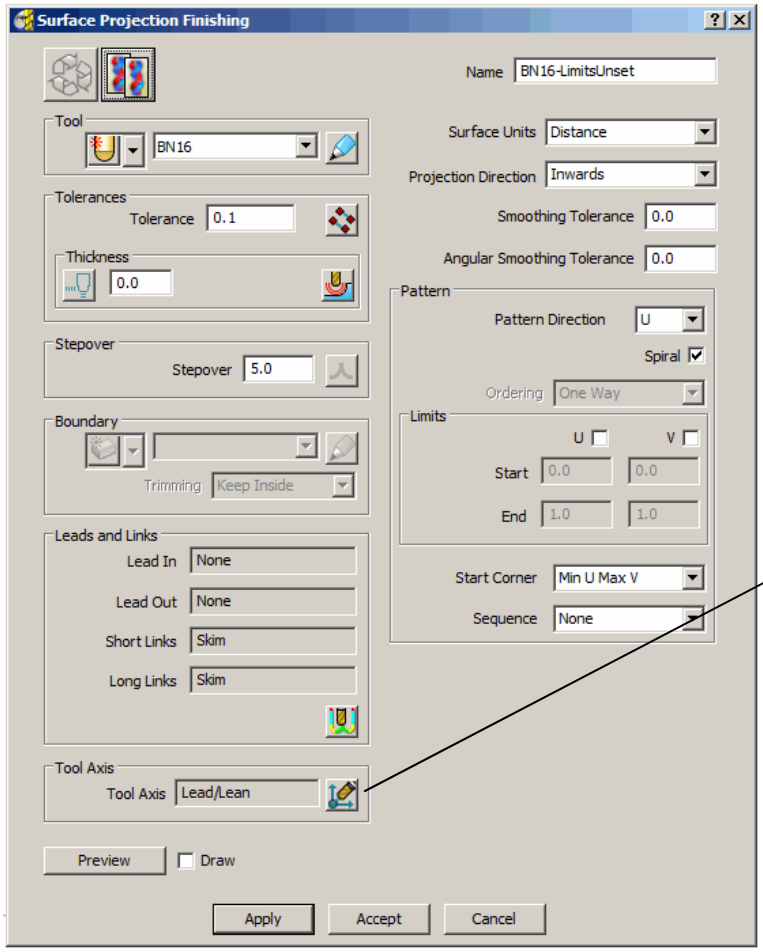
The use of **Safe Area – Cylinder** will provide a smoother tool axis, reorientation during **link** moves.

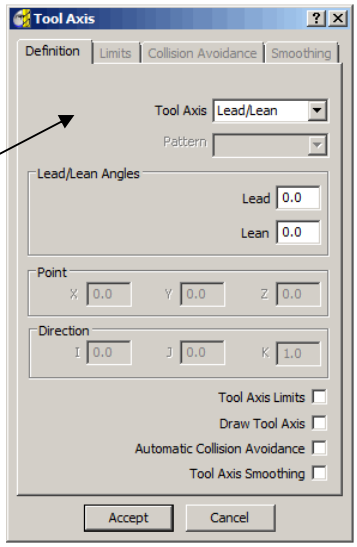
- Set both **Start and End Point** to **Use - Block Centre Safe**.
- Set **Leads** and **Links** as follows:-

Lead In\Out		**--None**
Extensions		**--None**
Links	**Short**	**--On Surface**
	Long	**--Skim**
	Default	**--Skim**

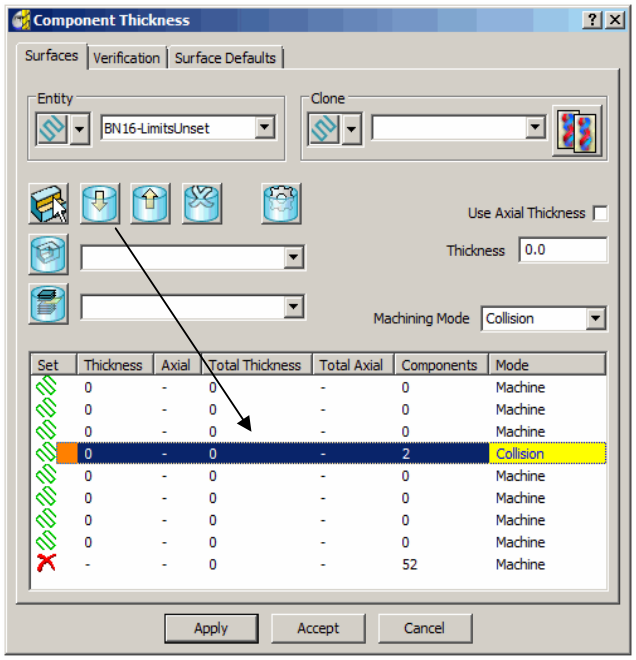


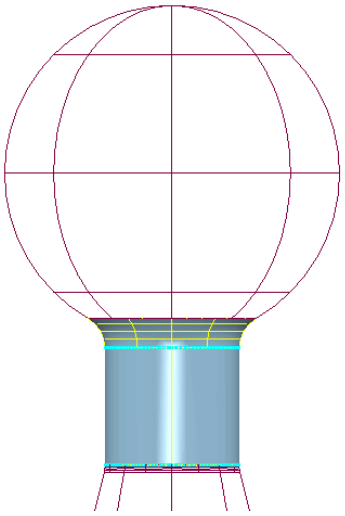
- Select the **Reference Surface** (shown shaded).
- From the **Toolpath Strategies** icon  select **Projection Surface Finishing** option from the **Finishing** form.
- **Rename** the Toolpath **BN16-LimitsUnset** and enter the values into the **Surface Projection Finishing** and **Tool Axis** forms <u>exactly</u> as shown below (<u>Do Not Apply</u> the form yet!).





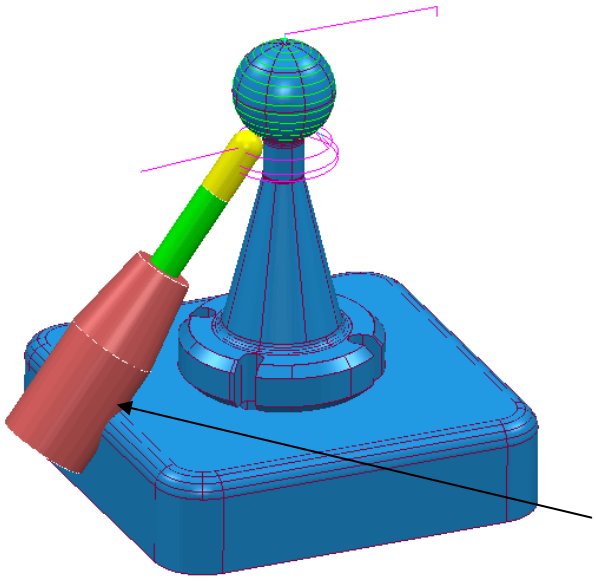
- Select the fillet and cylinder (**2 surfaces**) directly below the ***Sphere***.
- Select the **Thickness** form  and **Aquire**  the 2 selected **surfaces** to one of the rows.
- Set the row to **Collision** and **Apply** the form.





- **Click** the mouse into the **Command Window** at the bottom of the screen and **type** the following command lines:-

EDIT SURFPROJ AUTORANGE OFF
EDIT SURFPROJ RANGEMIN –2
EDIT SURFPROJ RANGEMAX 2



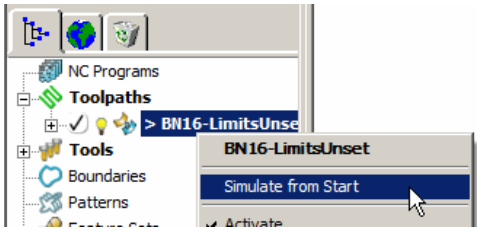
By setting the **surface projection range** to **+\-2** the toolpath is prevented from trying to project from infinity onto the underside of the model.

- **Apply** and **Accept** the form.
- ***Right click*** on one of the lowest tooltracks and from the local menu select **Simulate from Nearest Point**.

There are still problems due to the tool being cranked over way beyond the **rotational limits** of the ***machine tool*** and the **tool holder** 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.

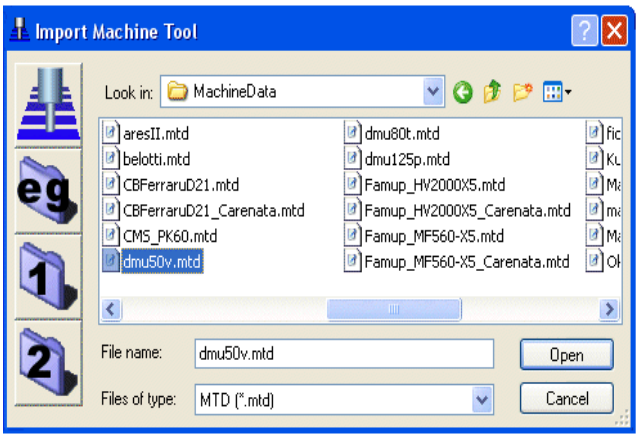
- Right click over the newly created **Toolpath** in the **explorer** window and select **Simulate from Start**.

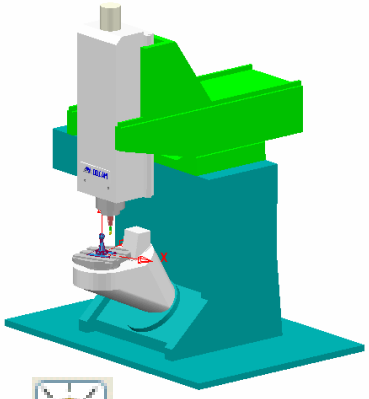


This command will raise the **Simulation** toolbar (if it is not already open).

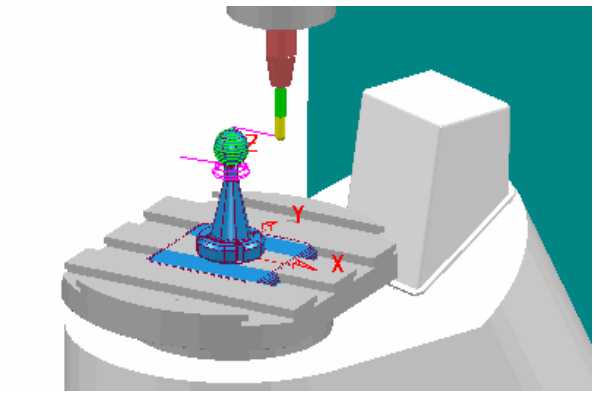
- In the main pull down menus select **View – Toolbars – Machine Tool** to raise the ***Machine Tool definition toolbar***.

- Select the **Import Machine Tool Model** icon and select:-
D:\users\training\PowerMILL Data\Machine Data\dmu50v.mtd



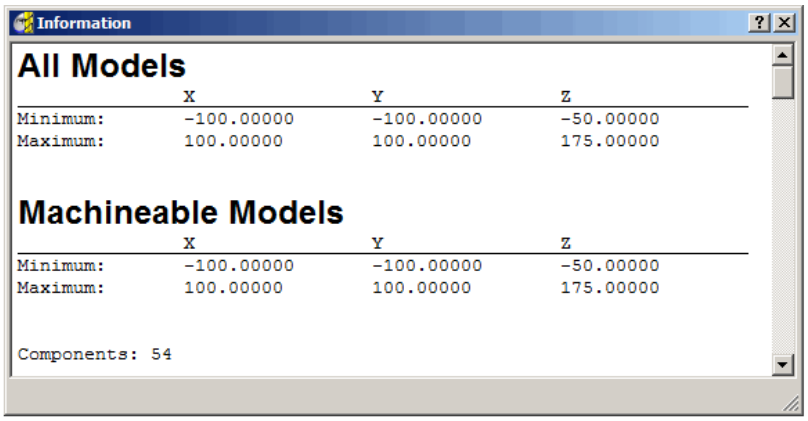


- Ensure the **Draw\Undraw machine tool** icon  is selected to display the machine tool.



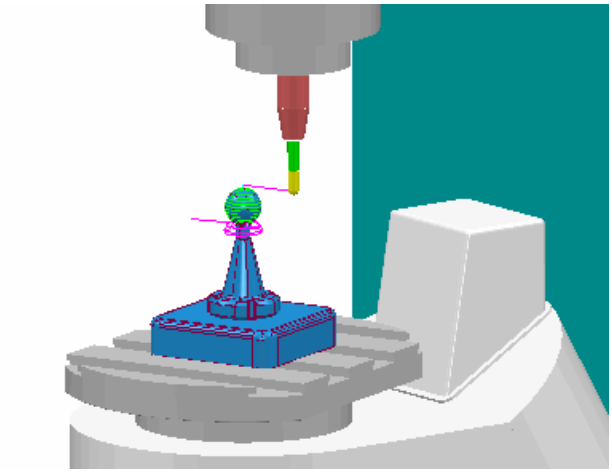
The current component datum (**Transform**) is matched to that of the Machine Tool model (Top - Centre of Table). As a result, the base of the component is currently embedded into the machine tool bed. To compensate for this, a new, suitably positioned **Workplane** is created. This new **Workplane**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

- In the **explorer**, right click on **Models** and from the local menu select **Properties** to obtain the dimensions.

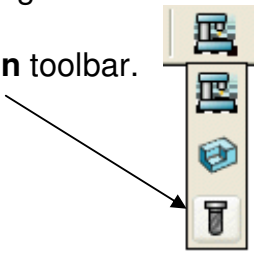


- Create a new **Workplane** named **MTD-datum** and move it by **Z-50**.
- Register the new **Workplane**, **MTD-datum** in the **Machine Tool definition** form (Note: It is not necessary to ***Activate*** the new **Workplane**).

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

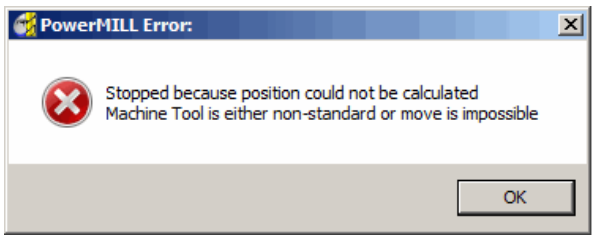


- Select **View from Front (-Y)** and **Zoom** the machining area.
- Select the **Tool View Point** icon from the **Simulation** toolbar.



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

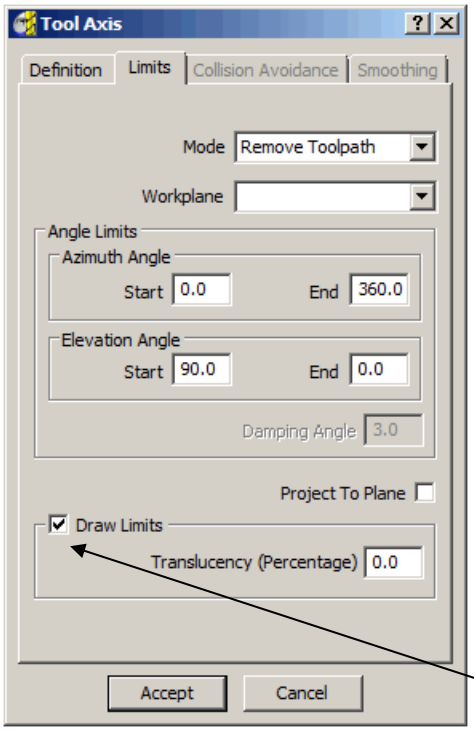
The **DMU50** angular limits are **X ±90 Y ±360** and this information is stored in the **MTD file**. This translates to **Azimuth** angle limits of **0** to **360** and **Elevation** angle limits of **0** to **90**. When the toolpath **simulation** attempts to go beyond this range an error message will be displayed indicating that the machine, **Tool Axis Limits** will be exceeded.

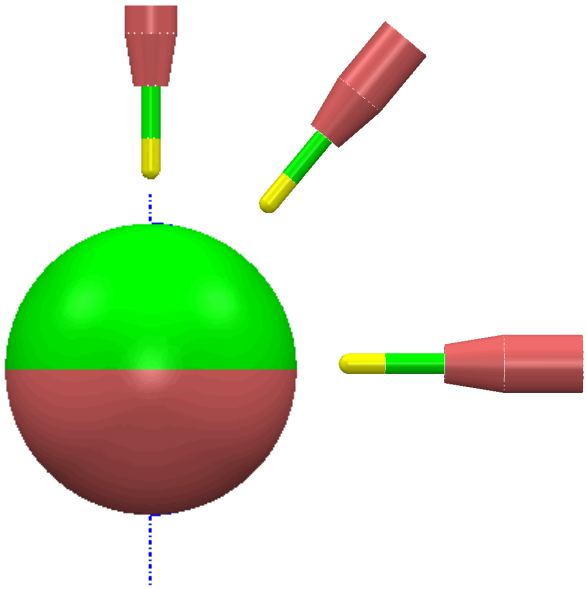


To allow for this, the true **Tool Axis Limits** will be applied to the toolpath on calculation.

- Select the toolpath **Settings** from the **explorer** window make a **Copy** of the original toolpath and rename **BN16-LimitsSet**.
- From the **Tool Axis Direction** form select the **Toolaxis limits** option to activate the Limits tab (Tick **Draw Toolaxis** to view limits).

- Select the **Limits** tab and enter the values into the form exactly as shown.

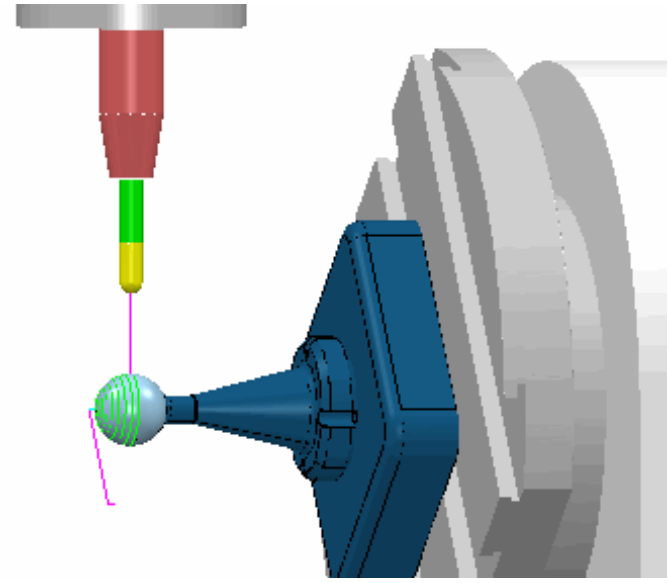




Draw Limits on (ticked)

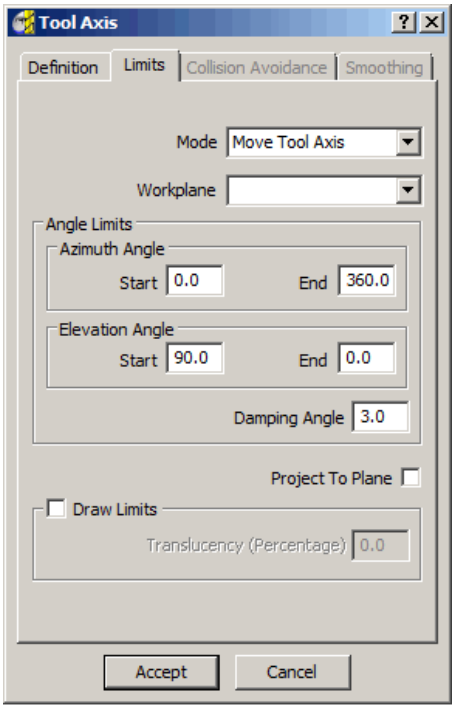
The ***green area*** of the sphere represents the permissible ***angular alignment*** of the **Tool Axis**.

- **Accept** the **Tool Axis Direction** form.
- With the same **Reference Surface** selected **Apply** and **Accept** the form.
- Right click over the ***Toolpath***, **BN16-LimitsSet** in the **explorer** window and select **Simulate from Start**.
- **Simulate** the **Toolpath**. 

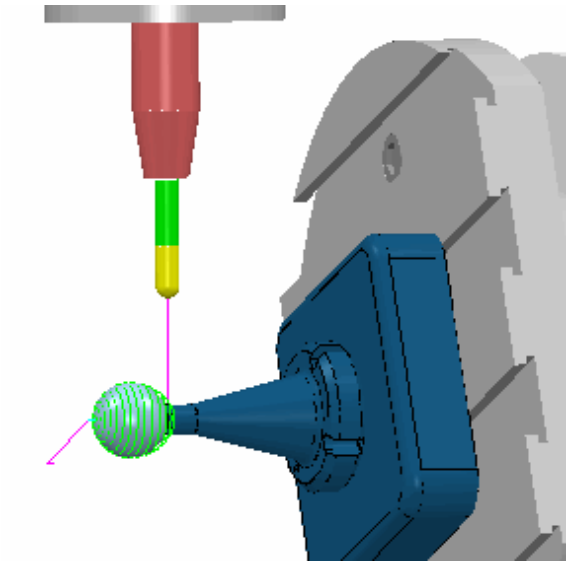


With **Mode** set to **Remove Toolpath** in the **Limits** form, only the portion of the surface within the rotary **Tool Axis Limit** range is machined.

- ***Right mouse click*** on the **toolpath**, **BN16-LimitsSet** and **Select Settings** to access the **Surface Projection Finishing** form.
- Select recycle  and from the **Tool Axis Direction** form  select the **Limits** tab and enter the values into the form exactly as shown below.



- **Accept** the **Tool Axis Direction** form.
- With the original surface selected **Apply** and **Accept** the toolpath.
- Right click over the Toolpath **10bnLimitsSet** in the **explorer** window and select **Attach Active Tool to Start**.
- **Simulate** the Toolpath. 



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

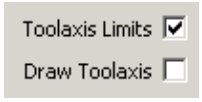
Defining Limits for a Multi-Axis Machine

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**.

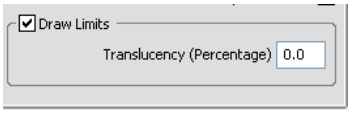
The configuration of the rotary axes varies widely, however the differences between many of these are relatively minor and there are really only three fundamentally different machine configurations:

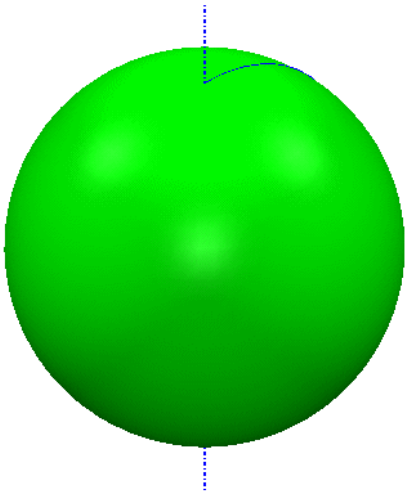
Table – Table Both rotary axes move the table.
Head – Head Both rotary axes move the head.
Head – Table One rotary axis moves the head, the other moves the table.

The next examples will show how to transpose angular limits of a machine tool into **Azimuth** and **Elevation** angles.

- **Delete all** and **Reset Forms**.
- Select the **Tool Axis** icon  from the main **PowerMILL** toolbar.
- Define Tool Axis as **Lead\Lean** and set **Lead\Lean** angles to **zero.**
- Select the **Toolaxis Limits** option 

to activate the **Limits** tab.
- Open the **Limits** tab.
- Select the **Draw Limits** option from the form.
- Select view **Iso 1**.

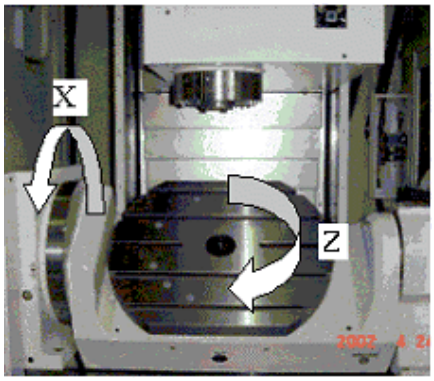




With the **Draw Limits** option selected a sphere will be displayed in the graphics window representing the angular machining limits available.
Green indicates a machinable portion and *Red* a non-machinable portion. With the default settings selected the total machining range is covered so the whole sphere will be *Green*.

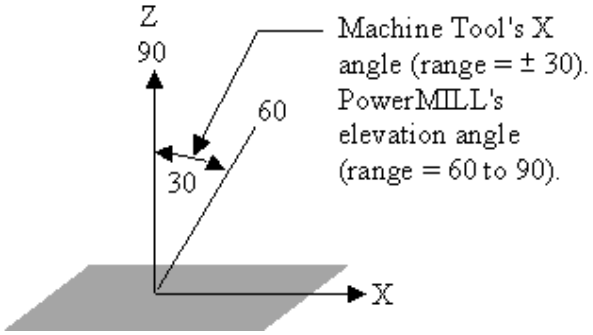
Table – Table

Both rotary axes operate on a table.



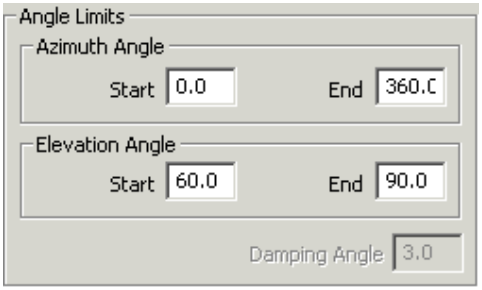
With the above **Table-Table** machine tool the angular limits are specified as:
X ± 30
Z ± 360
The machine tool **Y** limits are equivalent to the **Azimuth** angle or the angular limits normal to the **XY** plane. The **Y** limit of **± 360** translates to **Azimuth** angle limits of **0** to **360**.

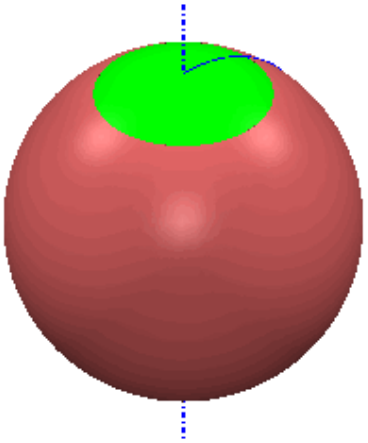
The machine tool **X** limits are equivalent to the **Elevation** angle above the **XY** plane. However they are not the same angle. This is best described using the diagram below. The machine tool measures the angular range relative to the **Z** Axis and **PowerMILL** measures it relative to the **XY**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

- Retain the default **Azimuth Angle** and modify the **Elevation Angle** values in the form as shown below to update the machining limits.

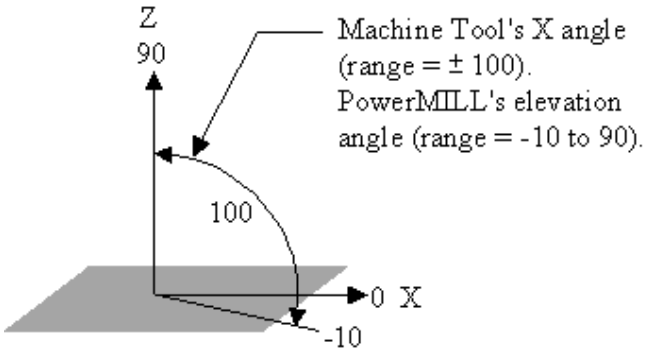




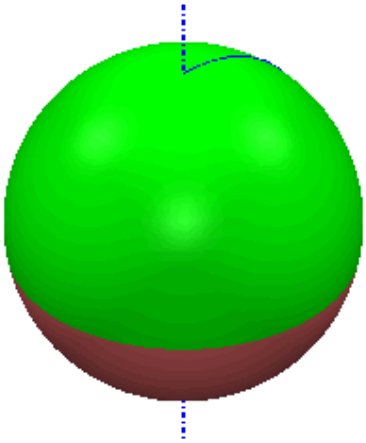
The **Tool Axis Limits** sphere has been visually updated with the modified values.

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**.



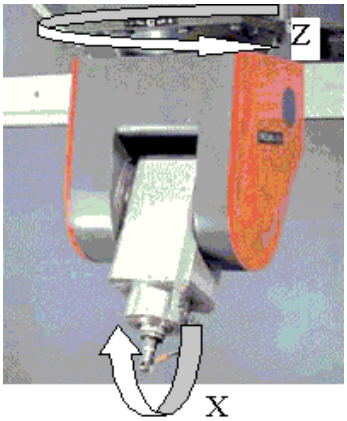
- **Modify** the **Elevation** angle values (**-10** to **90**) in the **Angle Limits** form to update the machining limits (as shown).



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

Head – Head

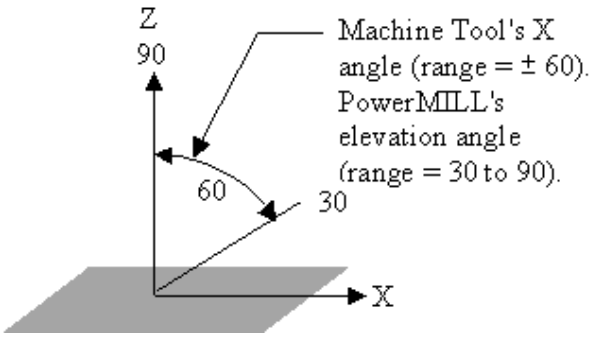
Both rotary axes move the head.



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**.

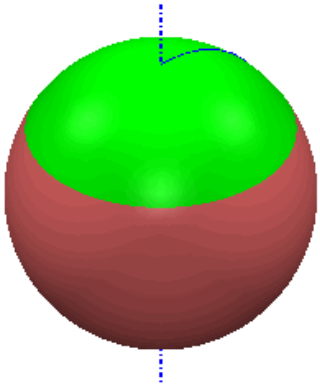


Alternative **Head**-**Head** machine tool angular limits:

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**).

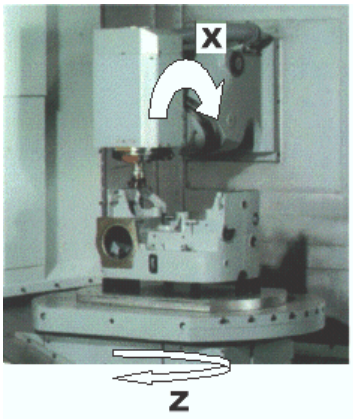
- **Modify** the **Elevation** angle values (**30** to **90**) in the form to update the machining limits as shown on the following page.



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.

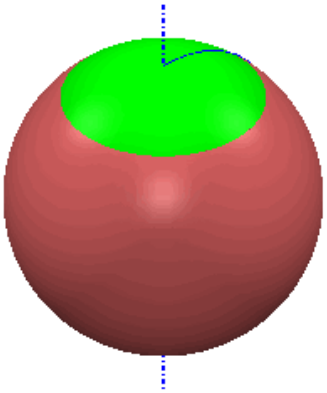


With the above **Head**-**Table** machine tool the angular limits are specified as:
X ± 40 Z ± 360

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 complementary angle to the **Elevation** angle. The **X** limit of **± 40** translates to **Elevation angle** limits of **50** to **90**.

- **Modify** the **Elevation** angle values (**50** to **90**) in the form to update the machining limits as shown below.

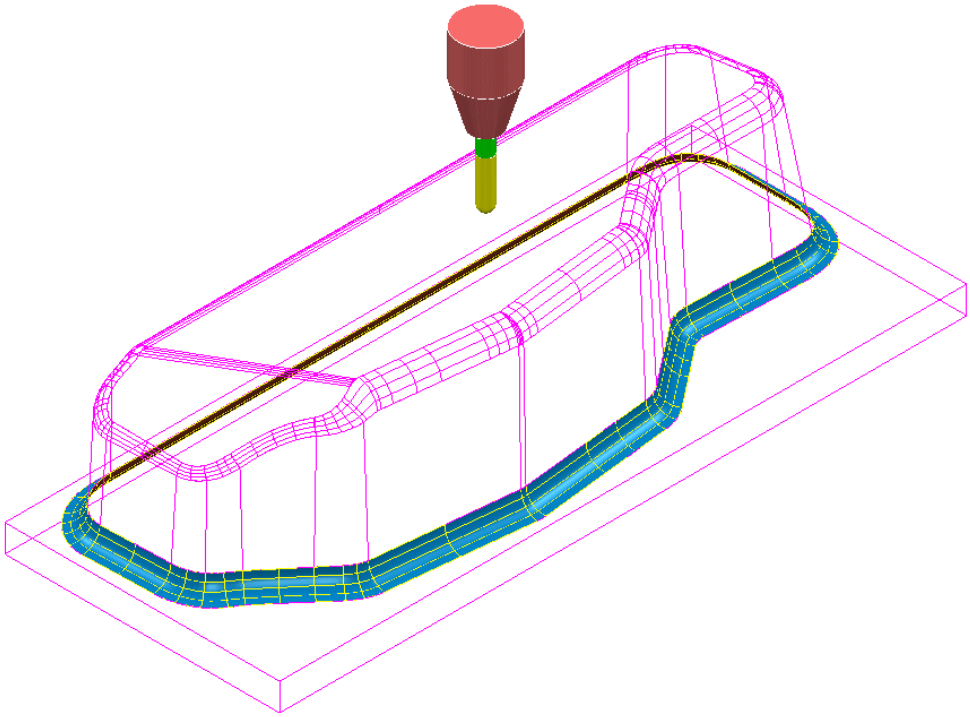


The **Tool Axis Limits** sphere has been visually updated with the modified values.

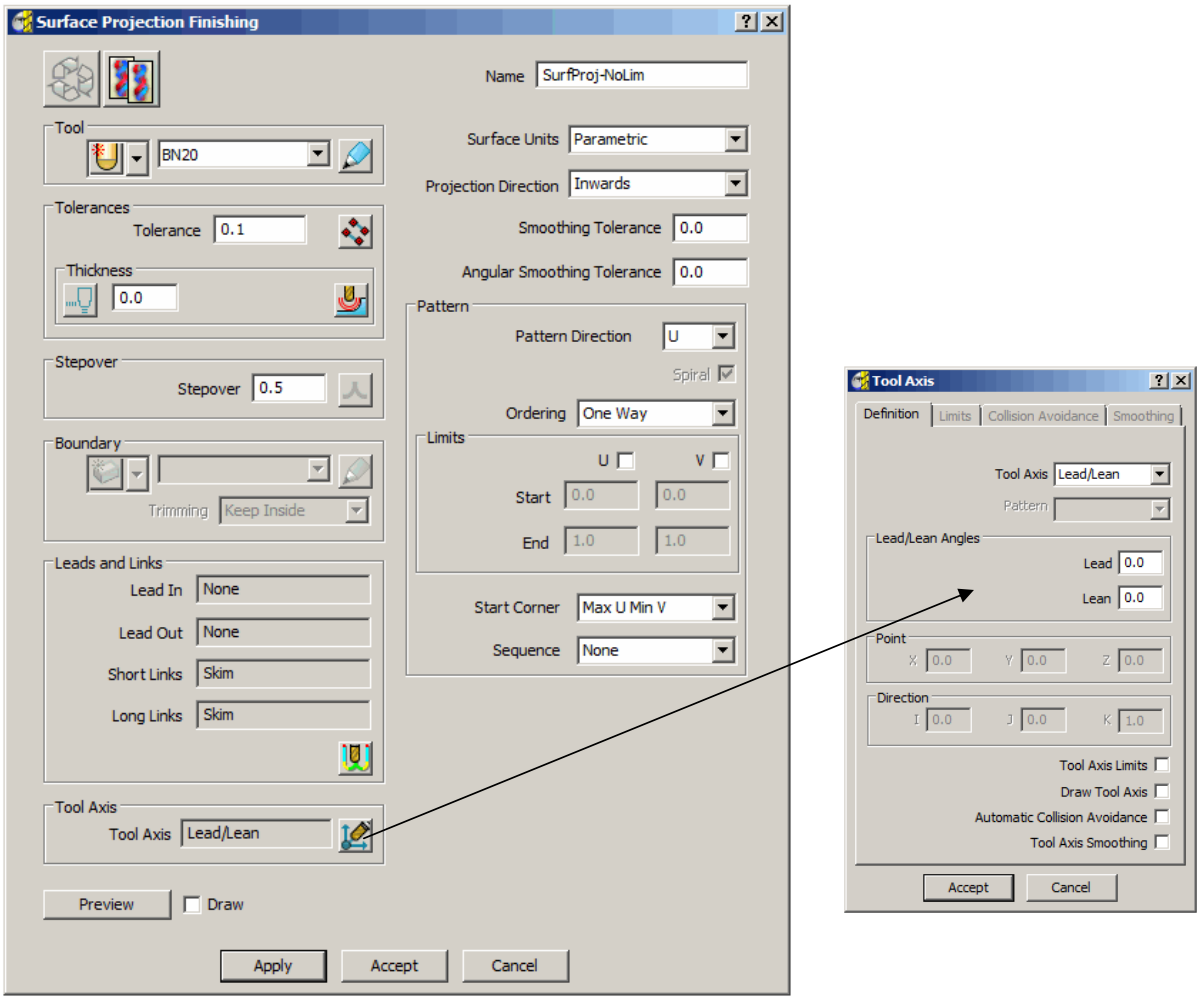
Applying tool Axis limits to a Steep Sidewall base Fillet

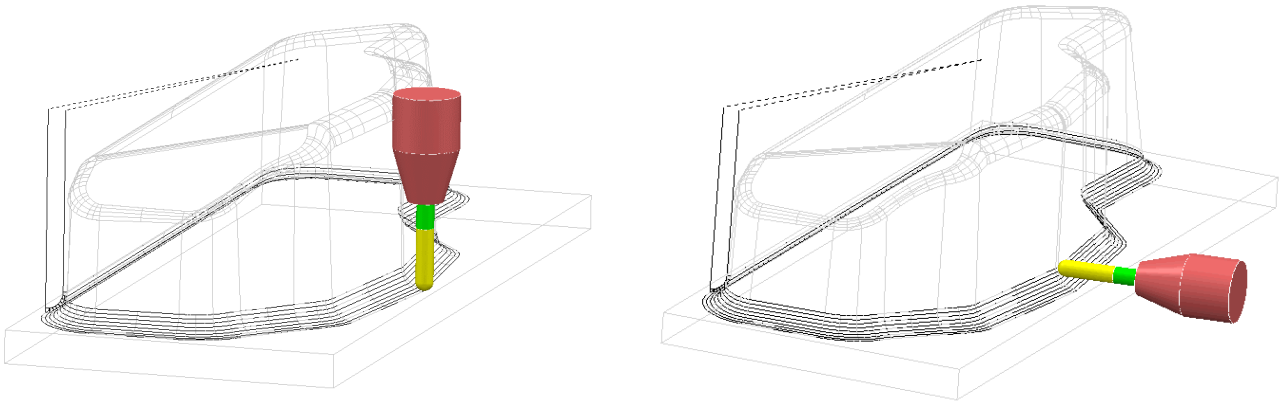
- **Delete all** and **Reset Forms**.
- **Open** the ***Project*** (As **Saved** earlier in Chapter 4)***:-***
  D:\users\training\COURSEWORK\PowerMILL-Projects\Punch2

- Create a **20mm Ballnose** tool **Length 70** named **BN20**.
- **Add** a **Shank** component **Upper Dia 20 Lower Dia 20 Length 40**
- **Add** a **Holder** component **Upper Dia 75 Lower Dia 40 Length 60**
- **Add** a **Holder** component **Upper Dia 75 Lower Dia 75 Length 60 Overhang 100**
- **Select** the fillet running around the base of the main component form.



- Select the **Toolpath Strategies** icon  and select the **Finishing** option located in the strategies form.
- Enter the remaining values into the **Projection Surface Finishing** form <u>exactly</u> as shown on the following page and **Apply**.

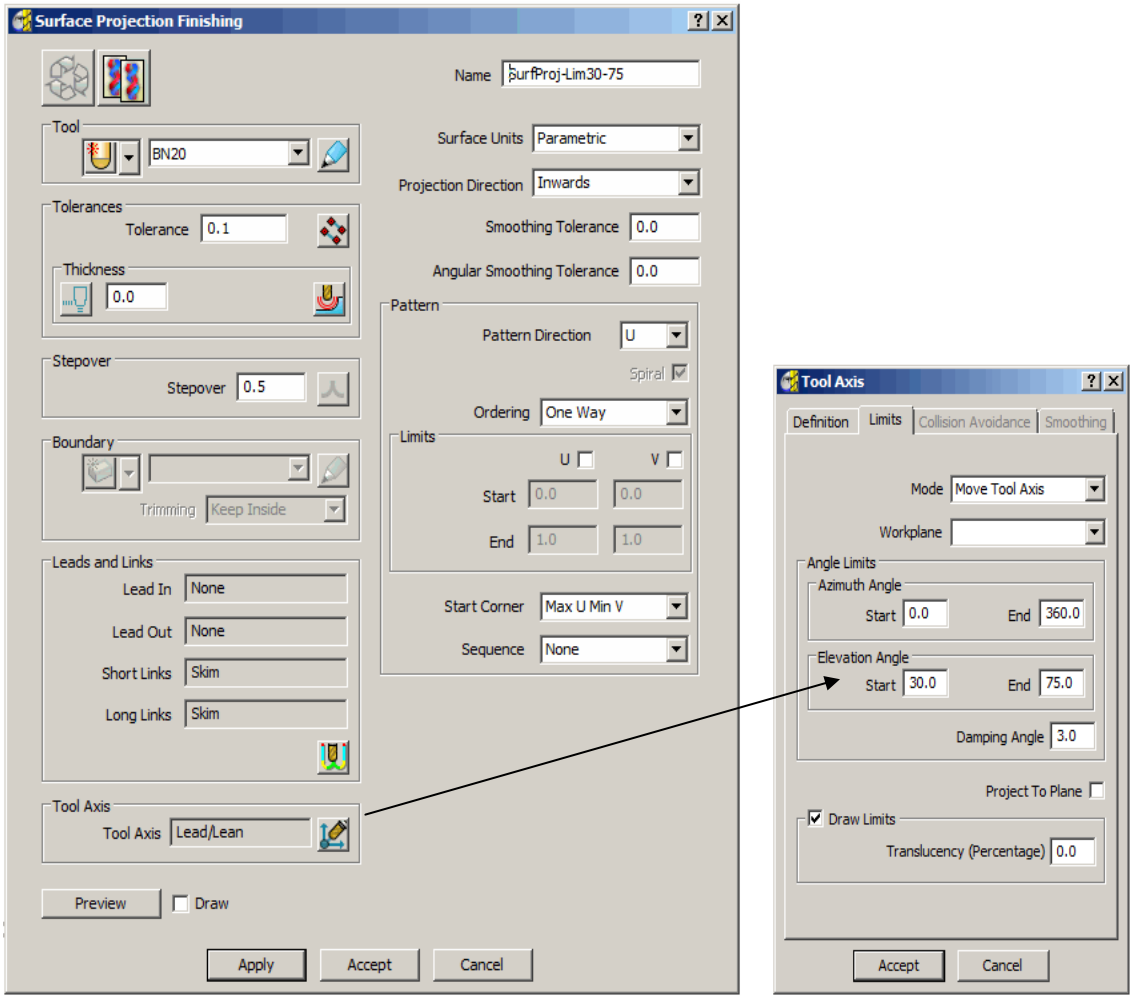




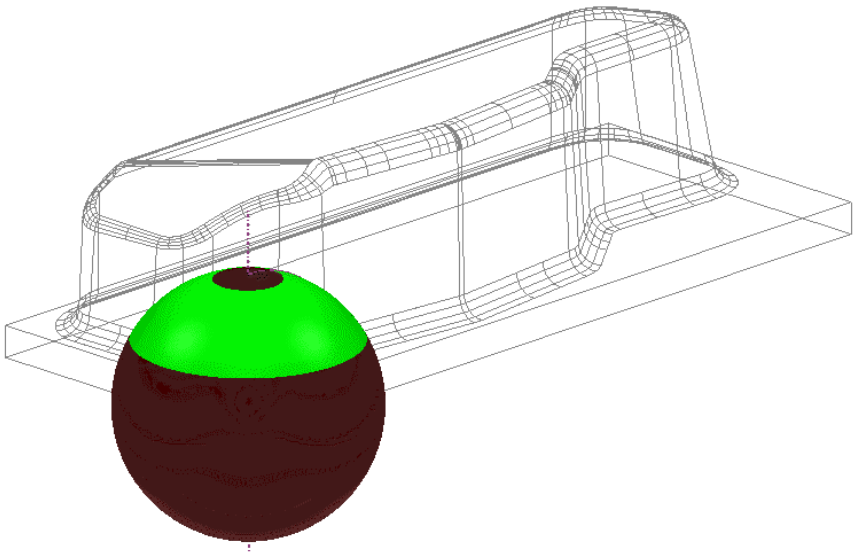
During **Projection Surface Finishing** the tool (**bn20**) is aligned normal to the **Fillet Surface** (if default **Lead\Lean 0** is applied). This is creating a **Collision** situation with both the sidewall and the base. **Tool Axis limits** will be applied to prevent this immediate problem as well as to keep the tool alignment within the machine tool rotary limits.

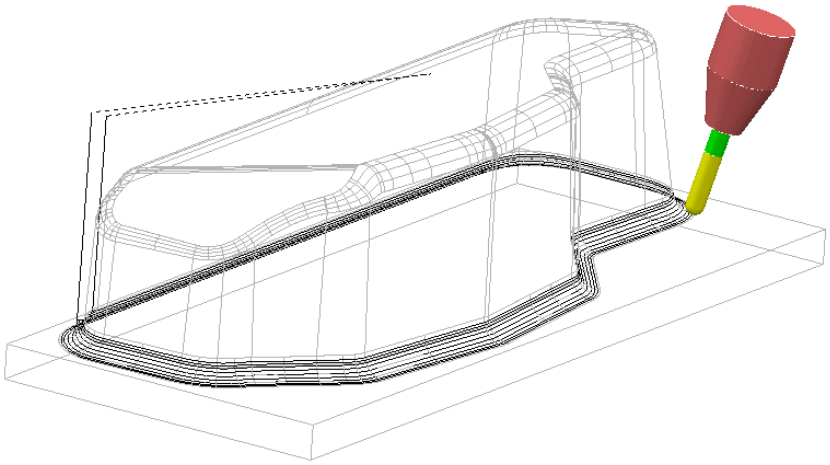
- In the **explorer** right click over the ***Toolpath*** - **SurfProj_NoLim** and select **Settings** to re-open the **Surface Projection Finishing** form used to create it.

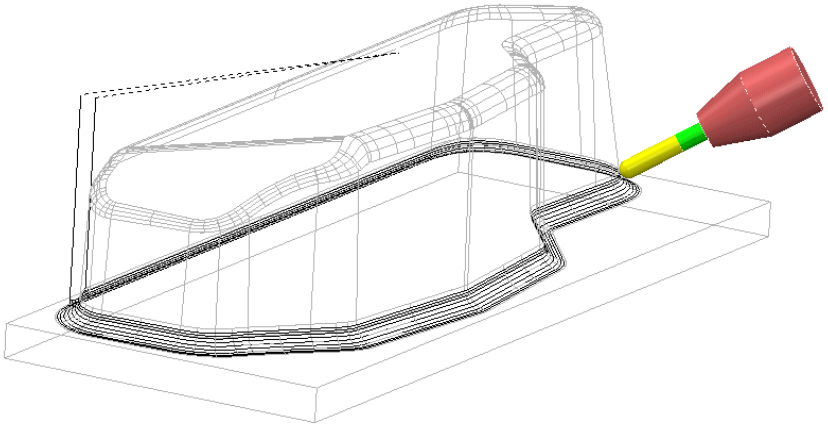
- In the **Surface Projection Finishing** form select the Icon  to create a copy and enter the ***Name*** - **SurfProj_Lim60-75**.
- Use the same values and settings in the **Projection Surface Finishing** form as used on the original toolpath with the addition of those shown (below left) in the **Tool Axis** form (including the **tick box** settings) and **Apply**.



With the **Tool Axis Limits** applied the alignment will be restricted to operate between **30** and **75** degrees (**Elevation Angle**) relative to the **XY** plane. A sphere (shaded pink) displaying the **Tool Axis** alignment limits (shaded green) becomes visible when the box labelled **Draw Limits** is ticked. The two illustrations on the following page show the toolpath with the tool attached on the upper and lower tracks to show the effect of the specified limits.
Compare these with the earlier illustrations of the previous toolpath created with no **Tool Axis Limits** applied.







- **Save** the **Project**.

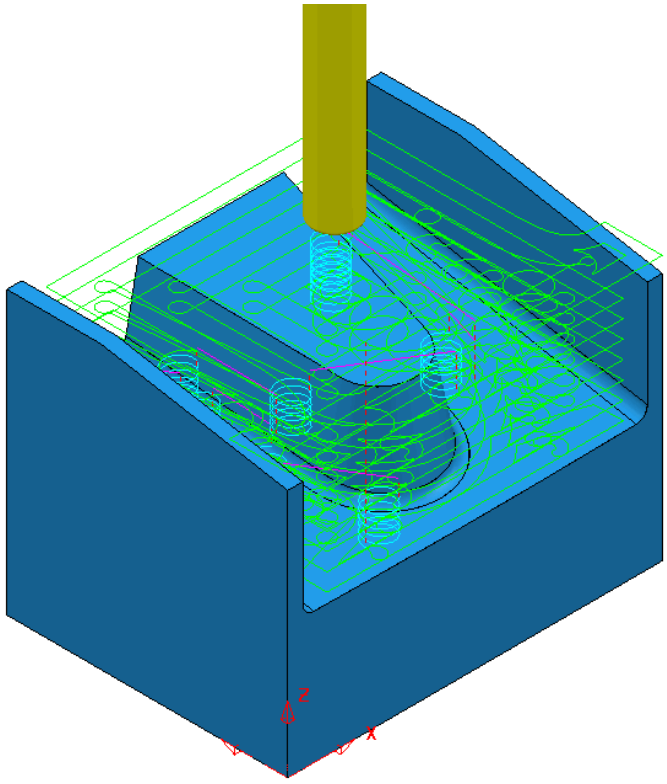
9. Auto Collision Avoidance

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; at present only a limited selection of **Finishing** strategies support **Automatic Collision Avoidance**. These include **Constant Z** and the 4 **Pattern** strategies.

- **Open** the stored training start up **Project**:-

D:\users\training\PowerMILL_Data\five_axis\CollisionAvoidance\Start-CollisionAvoid

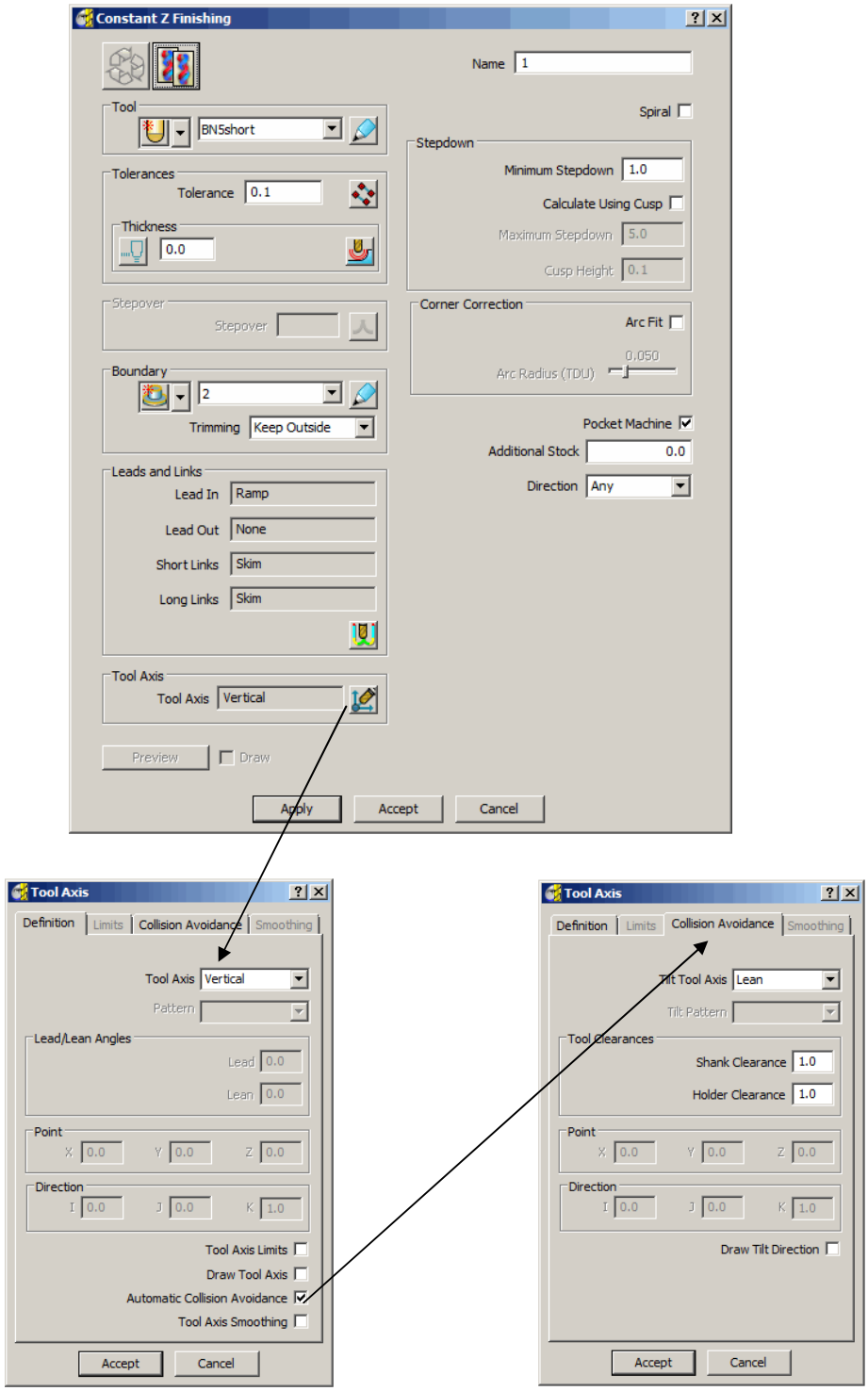


- From the main pulldown menus select **File - Save Project as**:-

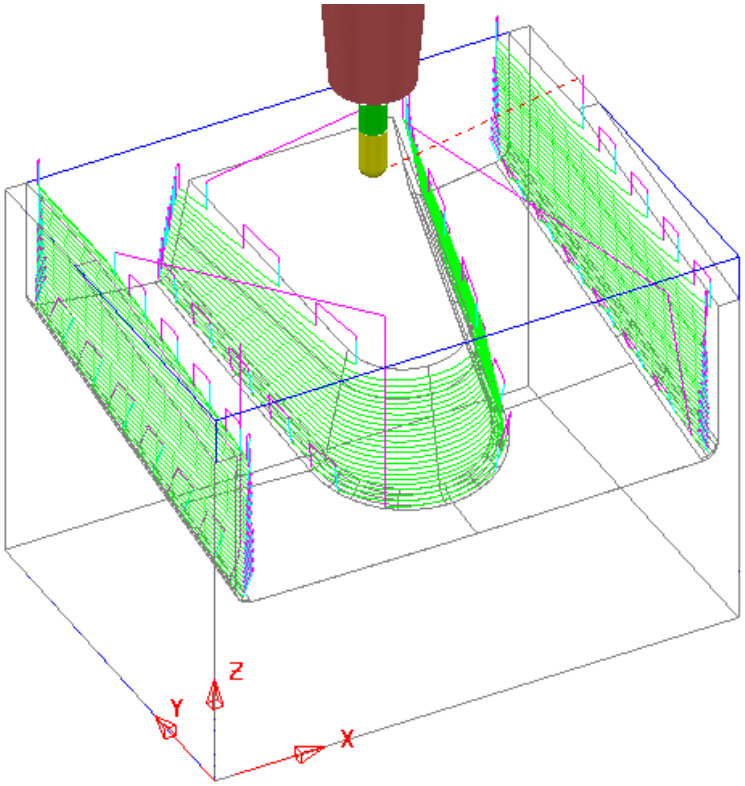
D:\users\training\COURSEWORK\PowerMILL-Projects*Collision-Avoid*

- In the **explorer - *Activate*** the tool **BN5short**.
- Select the **Toolpath Strategies** icon  in the **Main** toolbar to open the **New** strategies form.
- Select the **Finishing** tab and select the **Constant Z Finishing** option.

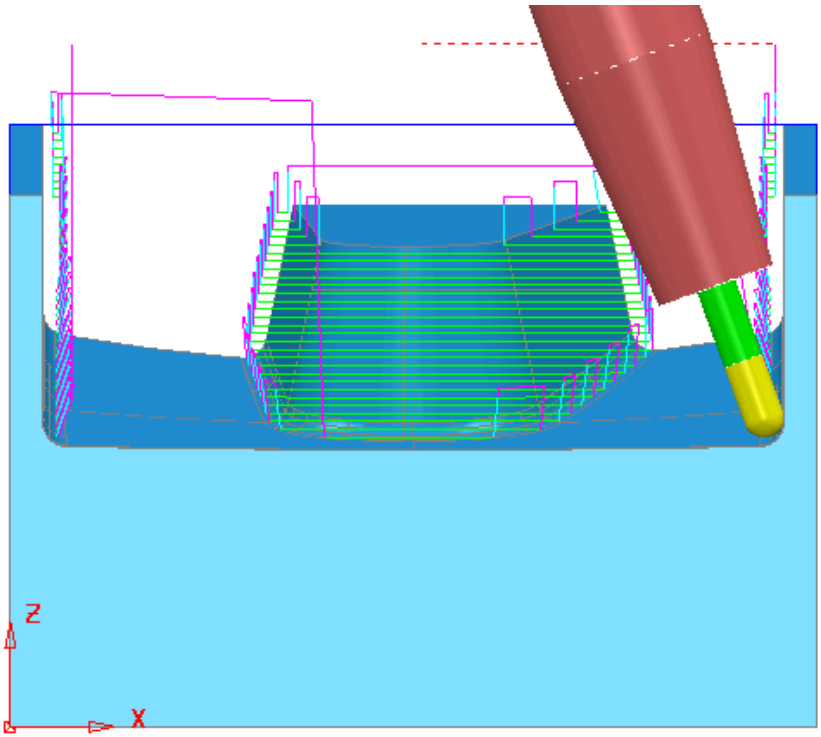
- Enter the values into the **Constant Z Finishing** and **Tool Axis** forms <u>exactly</u> as shown below.



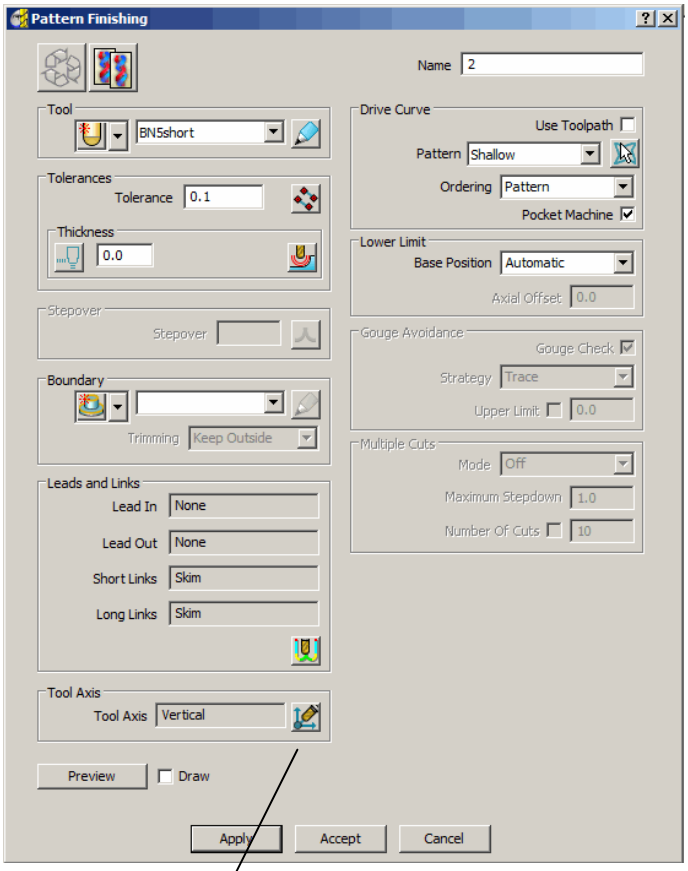
- In the **Tool Axis Direction** form tick **Automatic Collision Avoidance** and in ***Collision Avoidance*** - ***Tilt Tool Axis*** select **Lean** with **Shank Clearance 1** and **Holder Clearance 1**.
- **Accept** the **Tool Axis** form and **Apply** the **Constant Z Finishing** form.

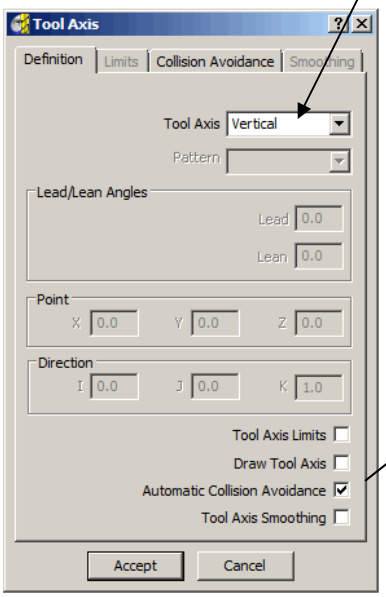


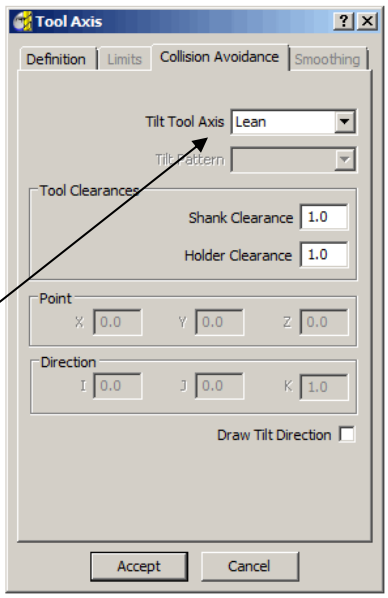
- Select a **View from front**  and **Simulate** the toolpath to observe the **Automatic Collision Avoidance** in action.

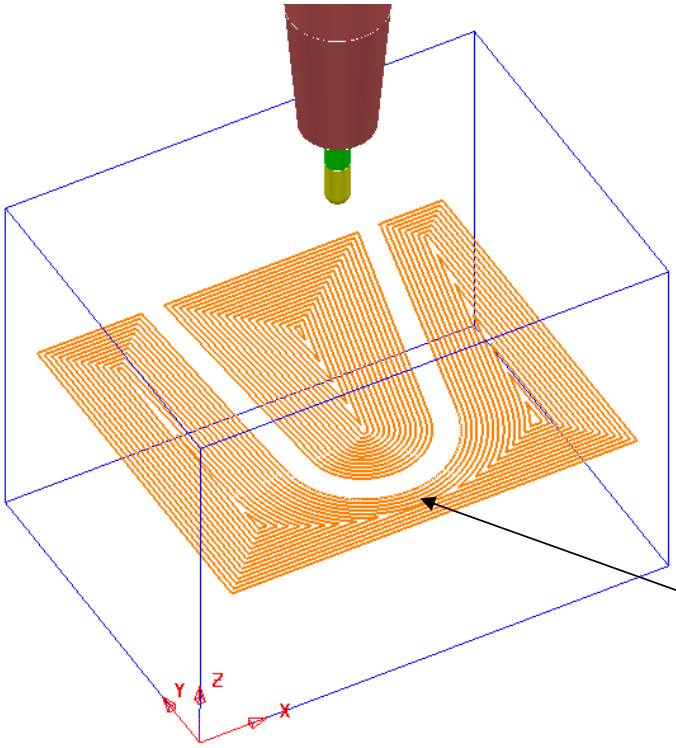


- Select the **Toolpath Strategies** icon  in the **Main** toolbar to open the **New** strategies form.
- Select the **Finishing** tab followed by the **Pattern Finishing** option.
- Enter the values shown below into the **Pattern Finishing form** and using the same **Tool Axis** options from the previous strategy select **Apply**.



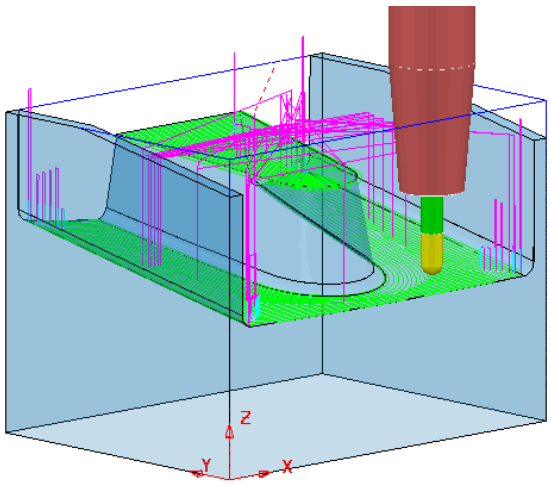


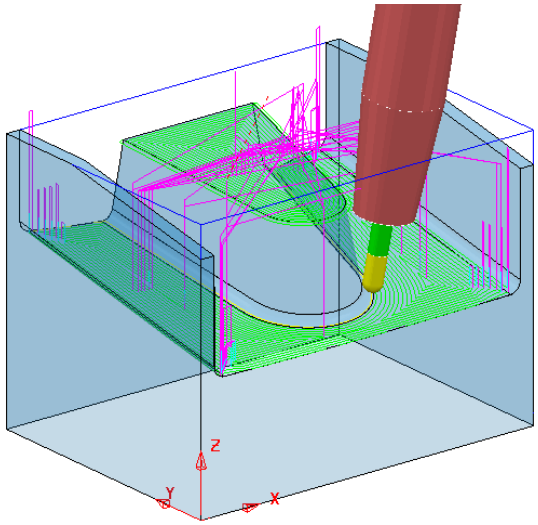




The start up **Project** contains a **Pattern** (***Shallow***) created using the **Pattern Maker** - **offset** option with some of the segments from **Pattern (*1*)** used as the ***drive curve***.
The segments of **Pattern (1)** are a direct copy of a **Shallow Boundary**.

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





Collision Avoidance applies ***collision safe***, **Vertical** tool alignment wherever possible and progressively applies a **Lean** angle to the **tool** in areas where it would otherwise collide with the component **model**.

10. Machine Tool Simulation

Introduction

The ability to check for potential **Machine Tool - Component** ***collisions*** is an essential requirement for ***Five Axis applications***. As a result, **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, 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, multiaxis **Machine Tool Simulation** (**.mtd**) files are supplied within the **PowerMILL** installation data. A typical location for a **C** drive install:-

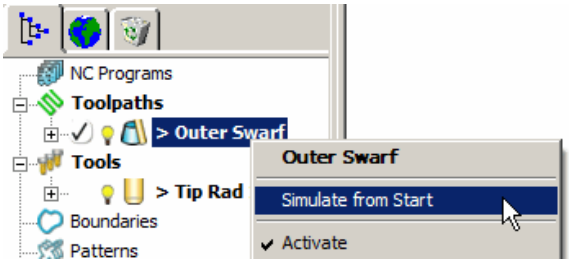
C:\Program Files\Delcam\PowerMILL9002\file\examples\MachineData

A comprehensive range of **mtd** files, based on actual **machine tools** are located on the training pc's in:-

D:\users\training\PowerMILL_Data\MachineData

Note: 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 **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 machine.

- **Delete all** and **Reset forms**.
- Open the **Project - Swarf Check** from the local directory:-
  D:\users\training\PowerMILL_data\five_axis\Collision_Simulation.
- Right click over the ***Toolpath*** **- Outer Swarf** in the ***explorer*** window and select **Simulate from Start**.

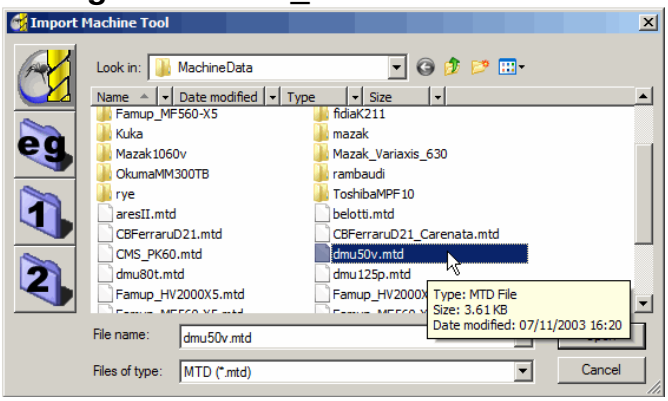


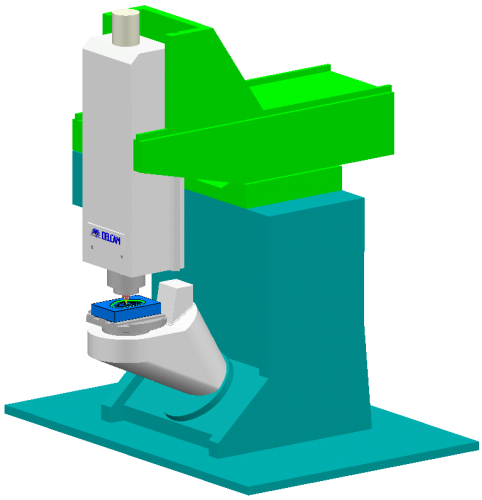
The **Simulation** ***toolbar*** will appear (if it is not already displayed).

- In the main pull down menus select **View – Toolbars – Machine Tool** to raise the **Machine Tool definition** toolbar.

- Select the **Import Machine Tool Model** icon and select **dmu50v.mtd** from the directory:-

  D:\users\training\PowerMILL_data\Machine Data



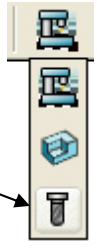


The **MTD** file contains the **Positional**, **Rotational** details for the individual, **Machine Tool**.

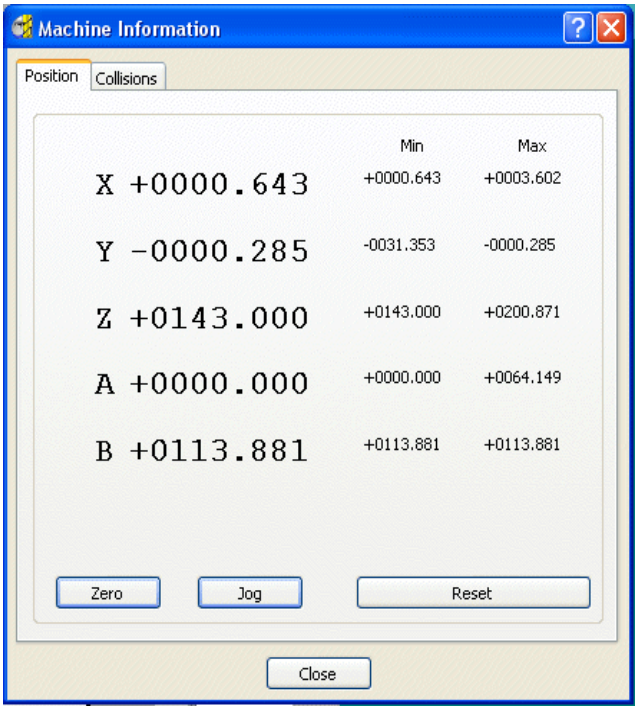
It is normal practice to create the **machine tool model**, with the global datum (**Transform**) position at the Top - Centre of the table.

The active **PowerMILL Tool** is automatically positioned in the **machine tool head**.

- Ensure the **Draw\Undraw machine tool** icon  is selected to display the machine tool.
- Select **View from Front (-X)** and **Zoom** the machining area.
- Select the ***tool view point*** icon from the **Simulation** toolbar.



- Select the option to **Open Display**  from the **Simulation** toolbar.

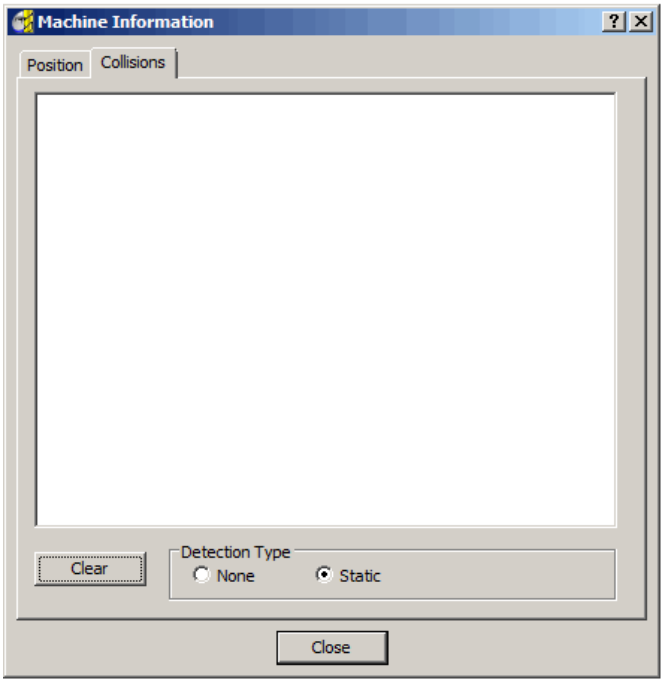


The **Simulation Information** form will display information on tool location and collision positions.

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.

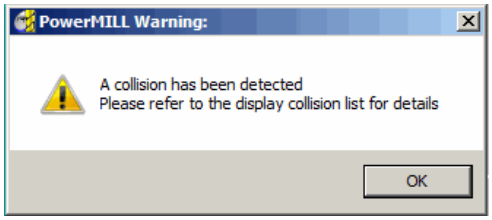
The values on the right hand side show the range of travel for each axis dependant on which toolpaths have been simulated. These are absolute values from a specific datum and can be reset by applying the **Zero** button.

- Select the **Collisions** tab  at the top left of the form.



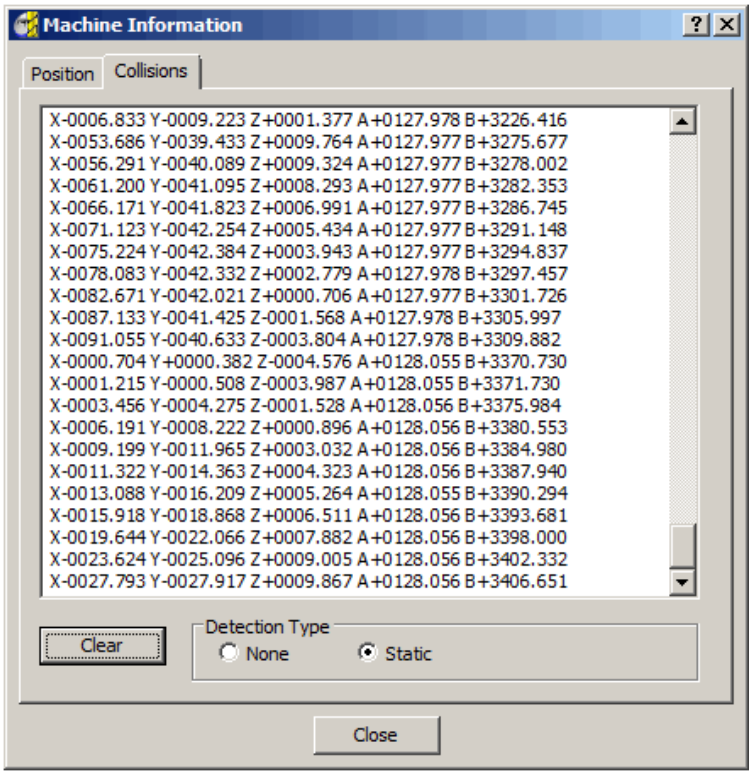
Any collisions encountered whilst running, or after the simulation has been completed, will be registered and displayed in the **Collisions** pane.

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



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

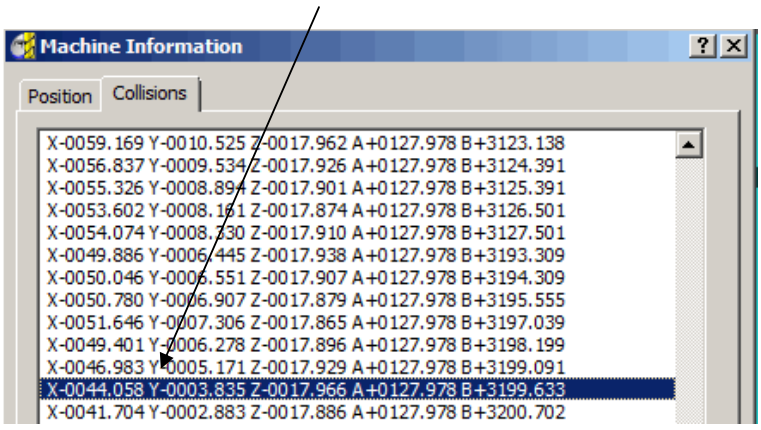
- Select **OK** to continue the **Simulation**.

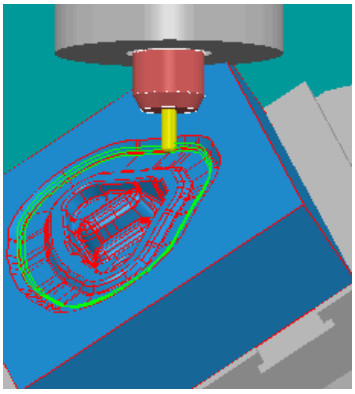


The above **Warning** message is only displayed for the first collision.

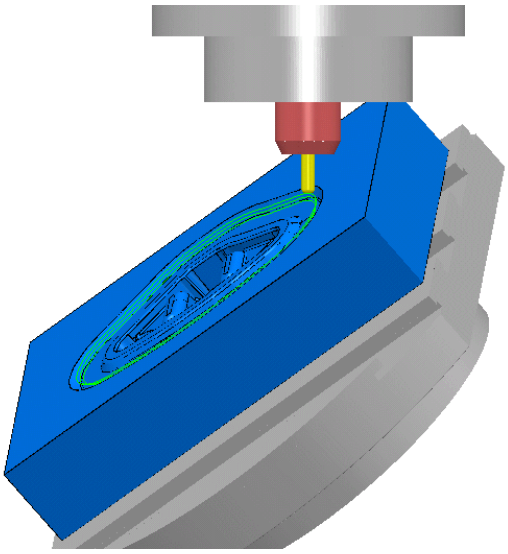
All collision moves will are registered and displayed in the **Collisions** pane.

- Press **Esc** to exit the **Simulation**.
- Select a **Collision Move** from the list.



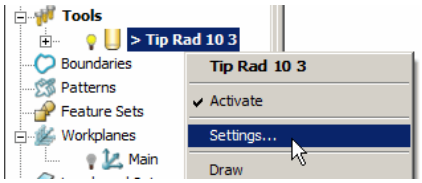


The **simulation** will move directly to the selected position in the form so that the collision can be viewed as shown above right.

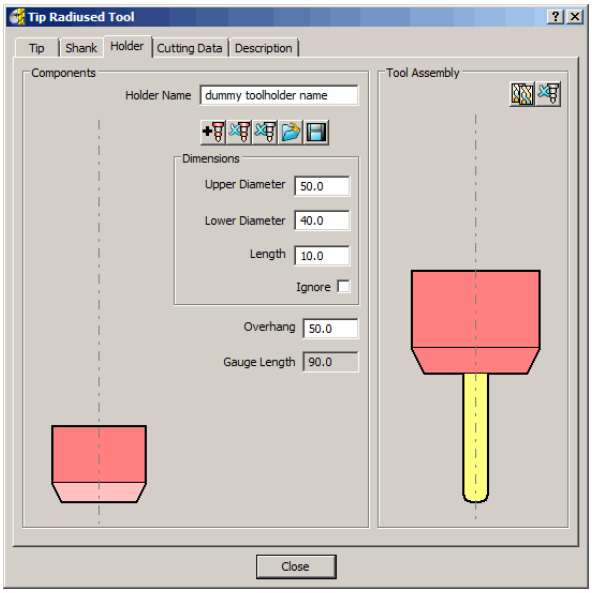


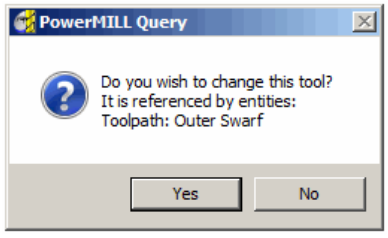
With a clear view of the collision, the user can assess how best to make the appropriate changes to avoid it. In this situation, substituting the cutter with one with increased **Tool Length** would be the easiest solution.

- **Right** click over the tool **Tip Rad 10 3** in the ***explorer window*** and select **Settings**.



- Select the **holder** tab on the tool form and modify the **Overhang** to **50**.

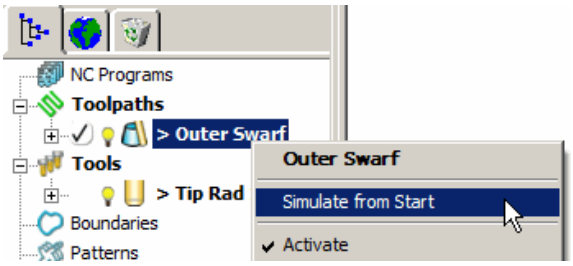




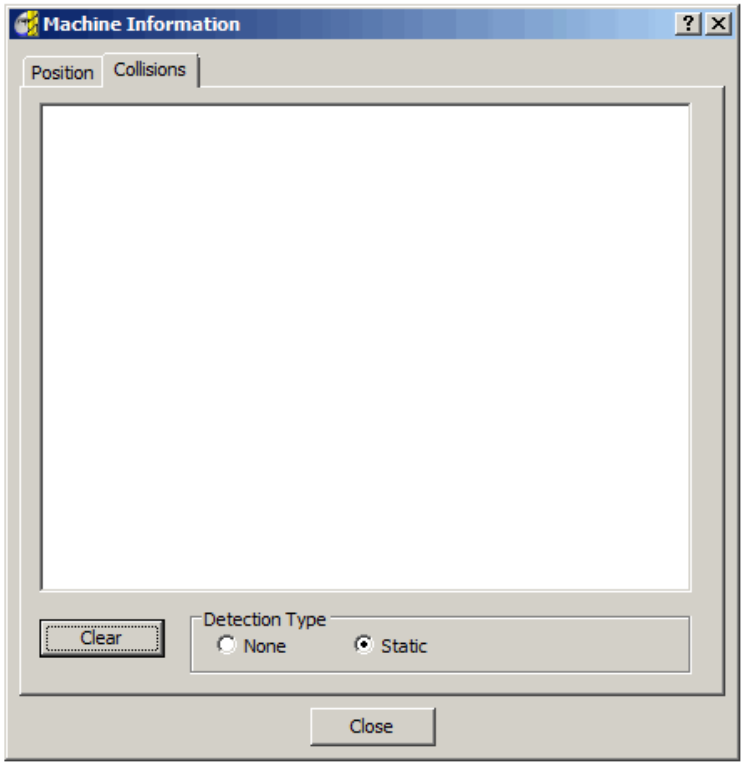
The above **PowerMILL Query** form only appears if the **tool** has been used in an existing ***machining strategy***.

- In the **PowerMILL Query** form select **Yes** to accept the changes to the tool.

- **Close** the form and **Attach Active Tool to Start** from the **toolpath** in the **explorer**.



- Select the option to **Open Display**  from the **Simulation** toolbar.
- Select the **Collisions** tab Collisions on the form.
- Select the **Clear** tab Clear on the form to clear any existing collisions.
- Start the **Simulation**  .



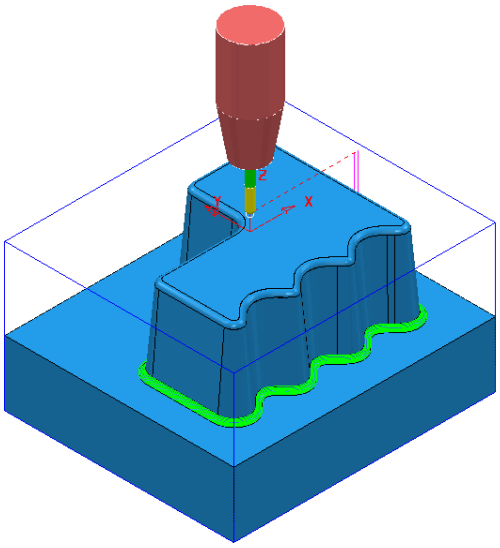
The ***Collision Pane*** remains blank indicating that no collisions have been detected.

11. 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.

- **Open** the **Project**:-
  D:\users\training\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.

As the **Project** is '**Locked'** it will be saved to a new one with a different name.

- From the ***main pulldown menus*** select **File - Save Project as**:-

D:\users\training\COURSEWORK\PowerMILL-Projects\EditToolAxis

- In the main pull down menus select **View - Toolbars - Machine Tool** to raise the ***Machine Tool definition toolbar***.

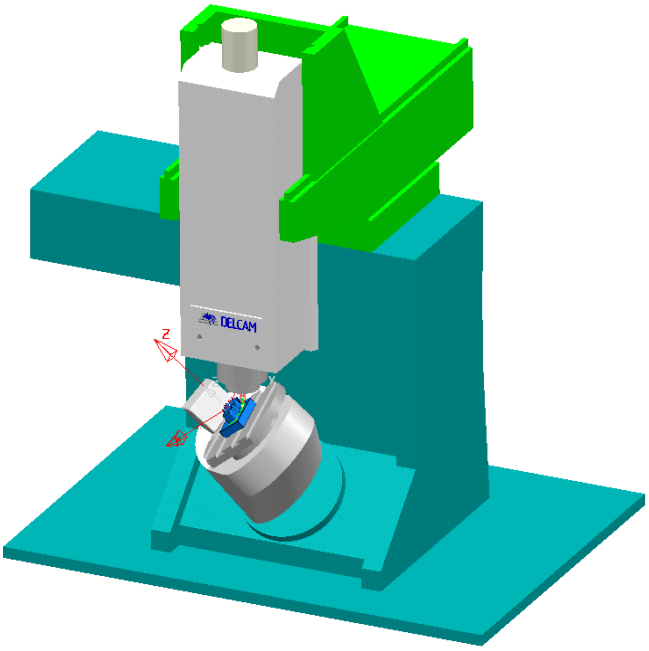
- Click on the **Import Machine Tool Model** icon and from the **PowerMILL Data\Machine Data** directory and select the file **dmu50v.mtd**

- Input the **Workplane - Base** as the component, datum for the **Machine Tool Simulation**.

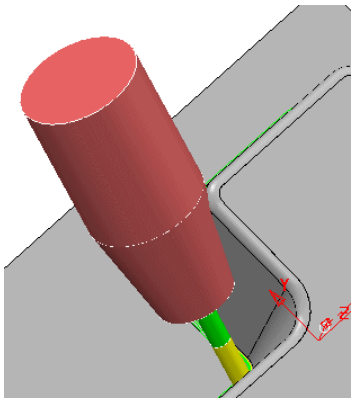
- Ensure the **Draw\Undraw machine tool** icon is selected to display the machine tool.

- Select **View from Front (-X)** and **Zoom** the machining area.
- Select the ***tool view*** icon from the **Simulation** toolbar.
- Select the option to **Open Display** from the **Machine Tool Simulation** toolbar.
- Right mouse click on the toolpath **BN5-Rest-Lean45** and from the local menu, select **Simulate from Start**.

- 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*** (Shown below).

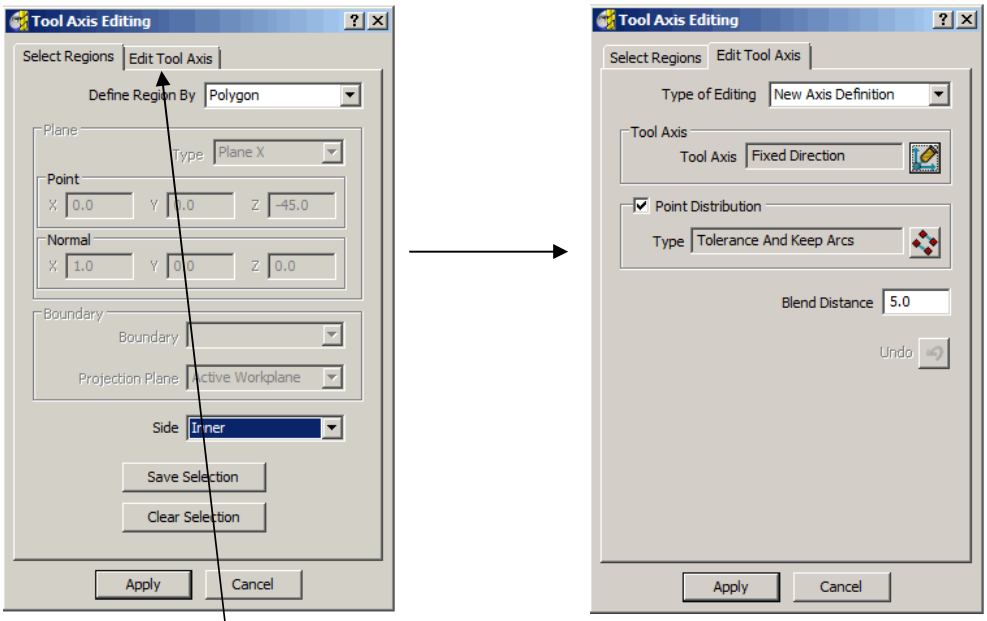


The above issues will be fixed by applying localised **Tool Axis** alignment modifications on the toolpath. To enable normal viewing while the changes are made, the **Machine Tool *model*** will temporarily be removed from the **Machine Tool Simulation** toolbar.

- In the **Machine Tool Simulation** toolbar remove **dmu50v.mtd** from the selection input box to leave it blank.

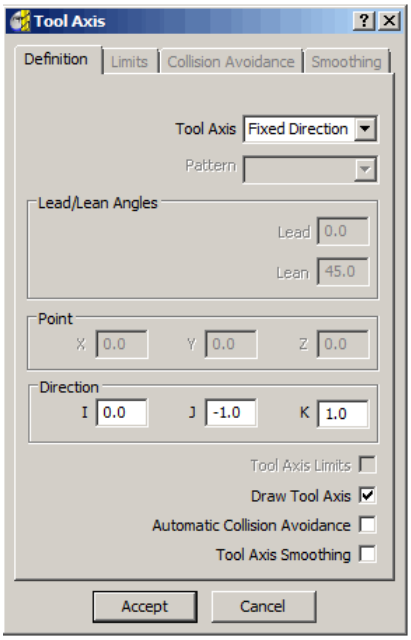
- Select a **View from top** ready for snapping the corners of a ***polygon*** around the first area of the **toolpath** to undergo **Tool Axis Editing**.

- In the **PowerMILL explorer**, right click on the ***toolpath*** **BN5-Rest-Lean45** and select **Edit – Tool Axis…** from the local menus to access the following forms.
- In the (default) **Select Regions** form set **Define Region By** – **Polygon** and **Side** – **Inner** (as shown below left).



- Select the **Edit Tool Axis** tab to change the options in the form, and set the **Blend Distance** to **5.0** before selecting the **Tool Axis** icon. 

Note:- The original **Tool Axis Alignment** will change gradually over the **Blend Distance** up to the new **Tool Axis Alignment**.

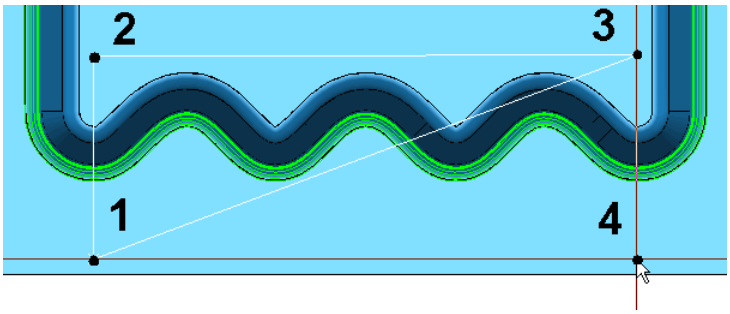


- Select **Tool Axis – Fixed Direction**.
- Input the **Direction** vector (Up tool axis) as:- **I 0.0 *J* –1.0 *K* 1.0**
- In the **Tool Axis** form, **Tick** the **Draw Tool Axis** box before selecting **Accept** (to return to the **Tool Axis Editing** form).

- From the **Main Pulldown** menus select **Draw – Cursor – Cross Hair**.

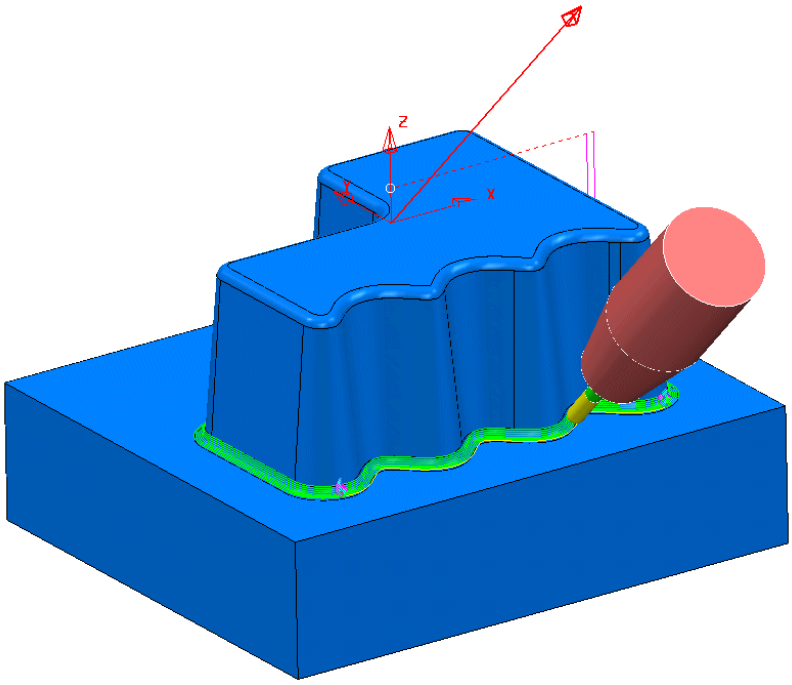
This will display **cross hairs** running through the ***cursor position***, to assist visual alignment when 'snapping' points.

- Click on the **Select Regions** tab to access the original options.
- Use the ***Left mouse key*** to ***snap*** **4** corners of the **Polygon** inside where the **Tool Axis Edit** is to occur, and click **Apply**.



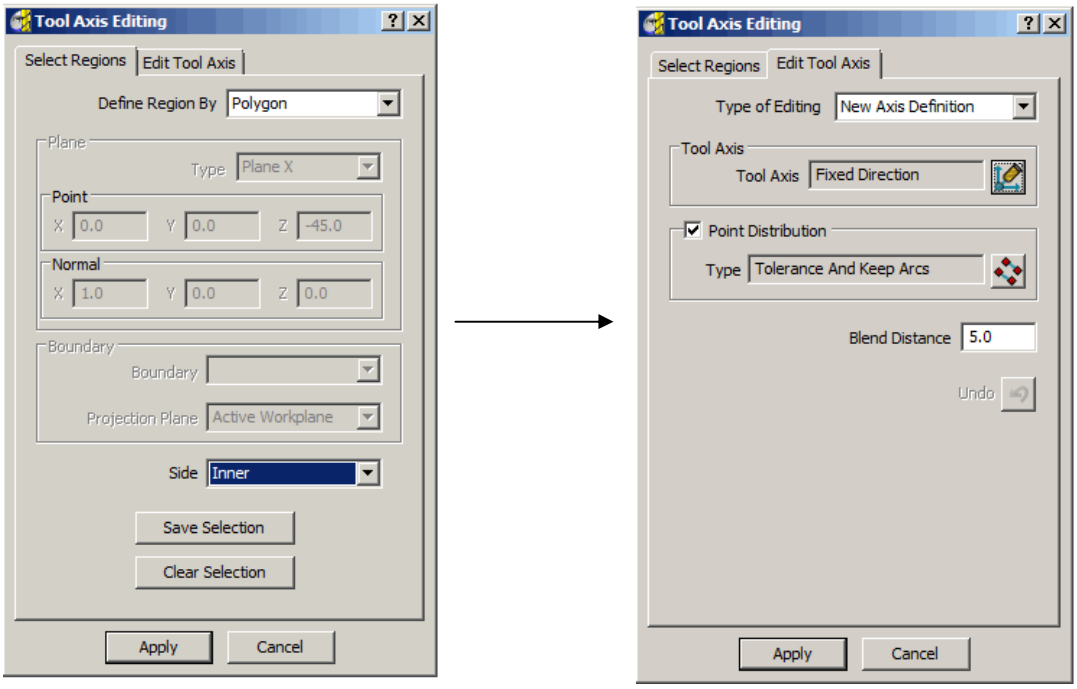
The above illustration is at the stage where the user is ready to ***snap*** the **4th** and final point on the **Polygon** (Cross hairs are mid grey).

- Select an **Iso 1 view** and **simulate** the **toolpath** to observe that the **Tool Axis** alignment is no longer causing repeated rotary movement along the undulating sidewall.

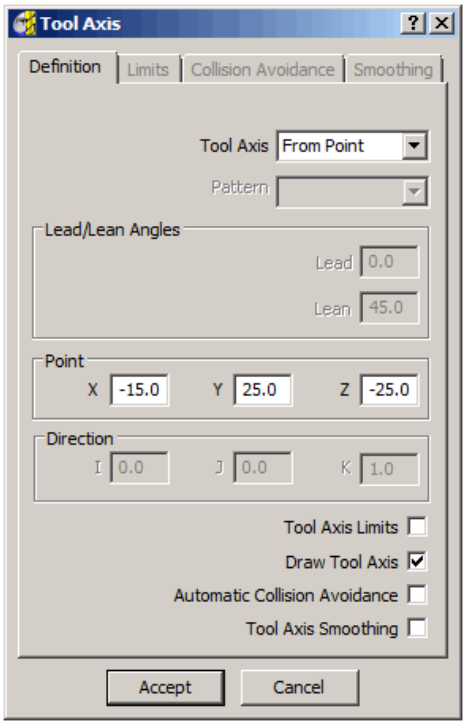


- Select a **View down Z**.
- In the **PowerMILL explorer**, right click on the ***toolpath*** **BN5-Rest-Lean45** and select **Edit – Tool Axis** to open the **Tool Axis Editing** form again.

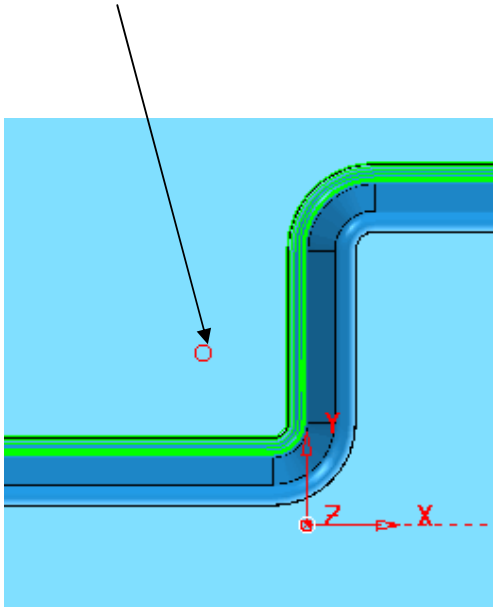
- In **Select Regions** select **Polygon** with **Side** – **Inner**, and then click the **Edit Tool Axis** tab.



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

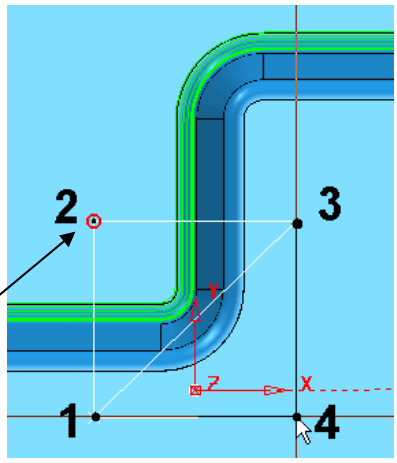


Tool Axis ***alignment point displayed***

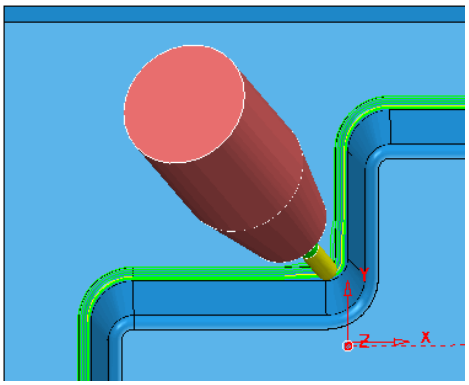


- This time ***snap* 4** corners of the **Polygon** to form a square, level with the **Tool Axis *alignment point*** and click **Apply**.

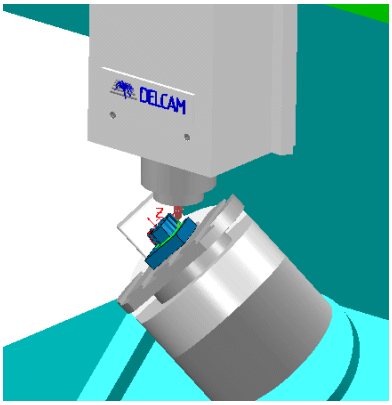
Tool Axis *alignment point*



- Select an **Iso 1 view**.
- **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 corner***.



- Perform a **Machine Tool Simulation** again using the **DMU50V** with the **Floor View** selected.



This time the unnecessary, 'rocking' movements about the **rotary axis** is minimised and the **Tool Holder** collision removed.

- From the ***main pulldown menus***, update the **Project** by selecting:- **File - Save**

12. Positional Tool Moves

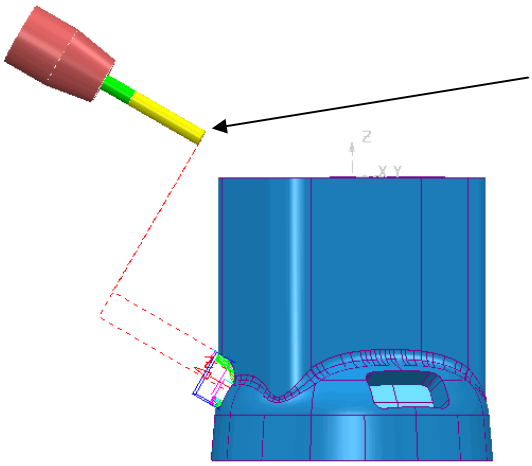
Positional Tool Moves

When performing **Positional Tool Moves**, it is essential to pay careful attention to preventing potential **collisions** and to ensure that the machine tool **rotational limits** are not exceeded. Three suggested methods to achieve this include:-

1/ Use of **Absolute** coordinates in the **Start and End Point** form.
2/ Insertion of strategically placed **Workplanes** into an **NC Program**.
3/ Use of a **Pattern Finishing** strategy in 3D space.

Tool Move with Start and End Point

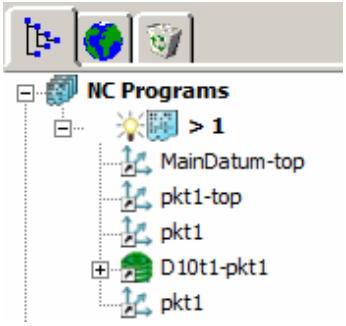
Positional Tool Moves can be controlled in the **Start and End Point** form  by use of **Absolute** (along with specific **XYZ** coordinates).
Note:- This method is used in the first example in **Chapter 1** : **3+2 Machining**.



Start and End Points input as **Absolute** coordinates, to be above to of component where it is safe to ***rotate*** the tool and perform ***rapid XY*** moves.

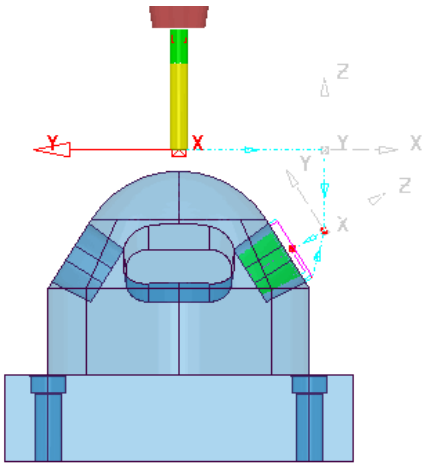
Tool Move via Workplanes in an NC Program

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

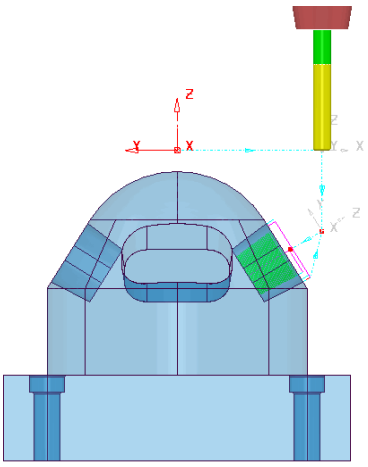


After a **Tool** moves to a **Workplane**, it will then **Rotate** (if applicable) to align to the **Workplane - Z Axis** (***Move***, ***Rotate*** is the **NC Preference**s - ***default*** setting).
The following 4 illustrations show the tool movements to 3 workplanes including a rotational move prior to performing the machining strategy.

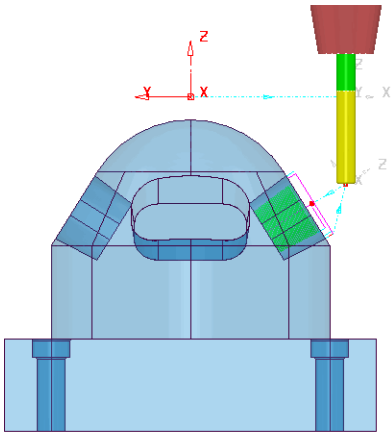
Tool at **MainDatum-Top**



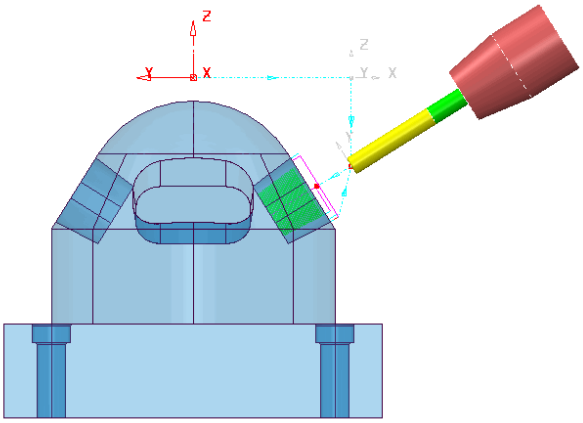
Tool moved to **pkt1-top**



Tool moved to **pkt1**



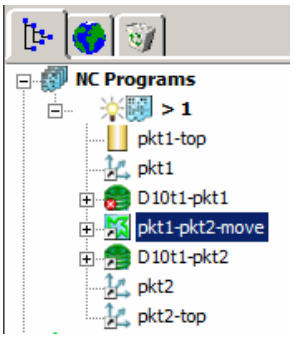
Tool Rotated at **pkt1**

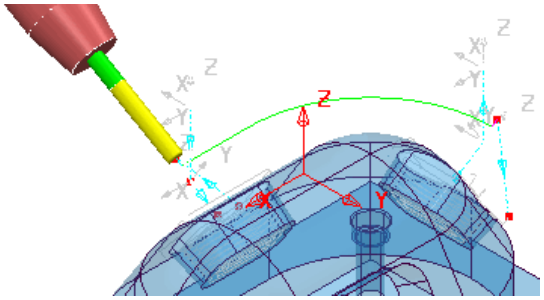


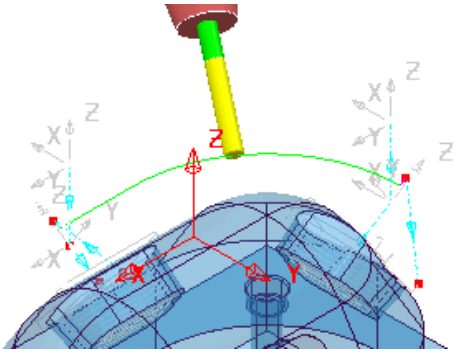
Note: When using **Workplanes** to control **Tool** movements around the component, it is normally feasible to use **First Point** and **Last Point** in the **Start and End Point** form on the individual **strategies** involved.

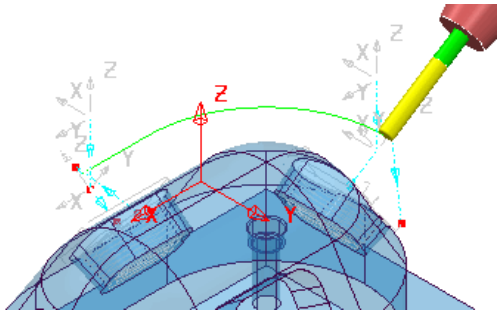
Tool Moves using a Pattern Finishing in 3D Space

Positional Tool Moves can be controlled by running the **Tool** along a **Pattern Finishing** strategy used as a **Drive Curve** in 3D space. Note:- A **lean** angle can be applied to maintain an angular **tool alignment** during transit.







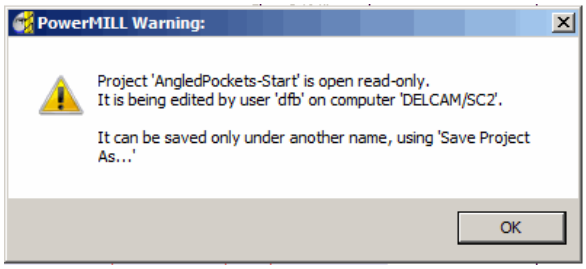


Example

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 when moving between **toolpaths**.

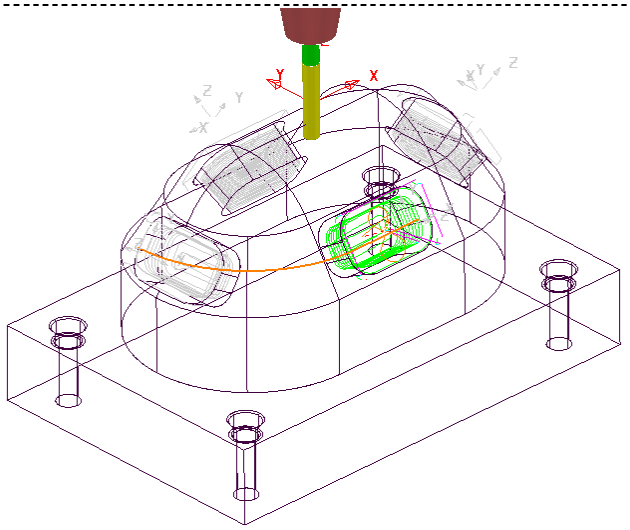
- **Import** the **Project**:-

D:\users\training\PowerMILL_Data\FiveAxis\PositionalMoves\AngledPockets-Start

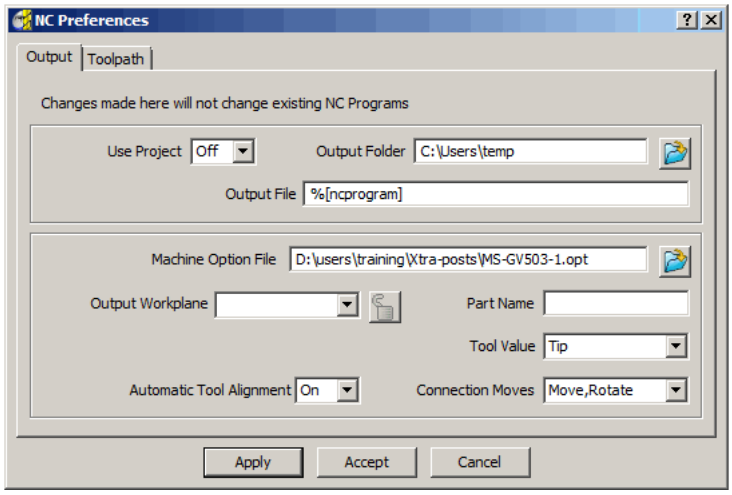


- **Save** the **Project** as:-

D:\users\training\COURSEWORK\PowerMILL_Projects\AngledPockets

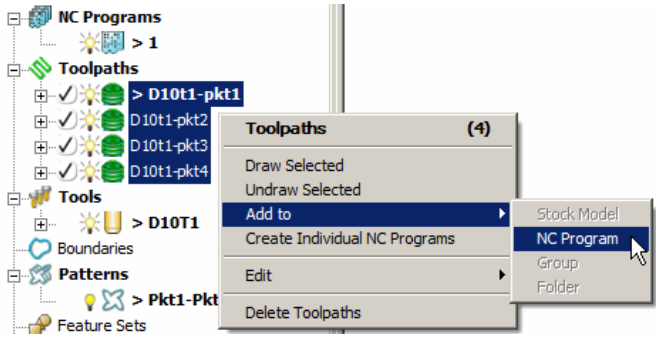


- In the **PowerMILL explorer** right mouse click on **NC Programs** and from the ***local menu*** select **Preferences**.

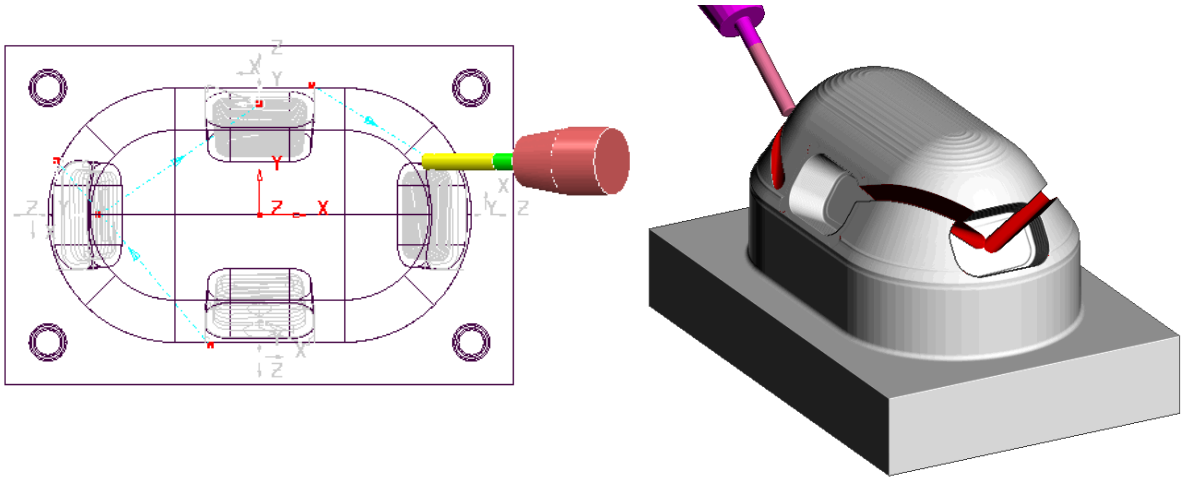


- Select a suitable Five Axis, **Machine Option File**:-
  D:\users\training\Xtra-posts\MS-GV503-1.opt

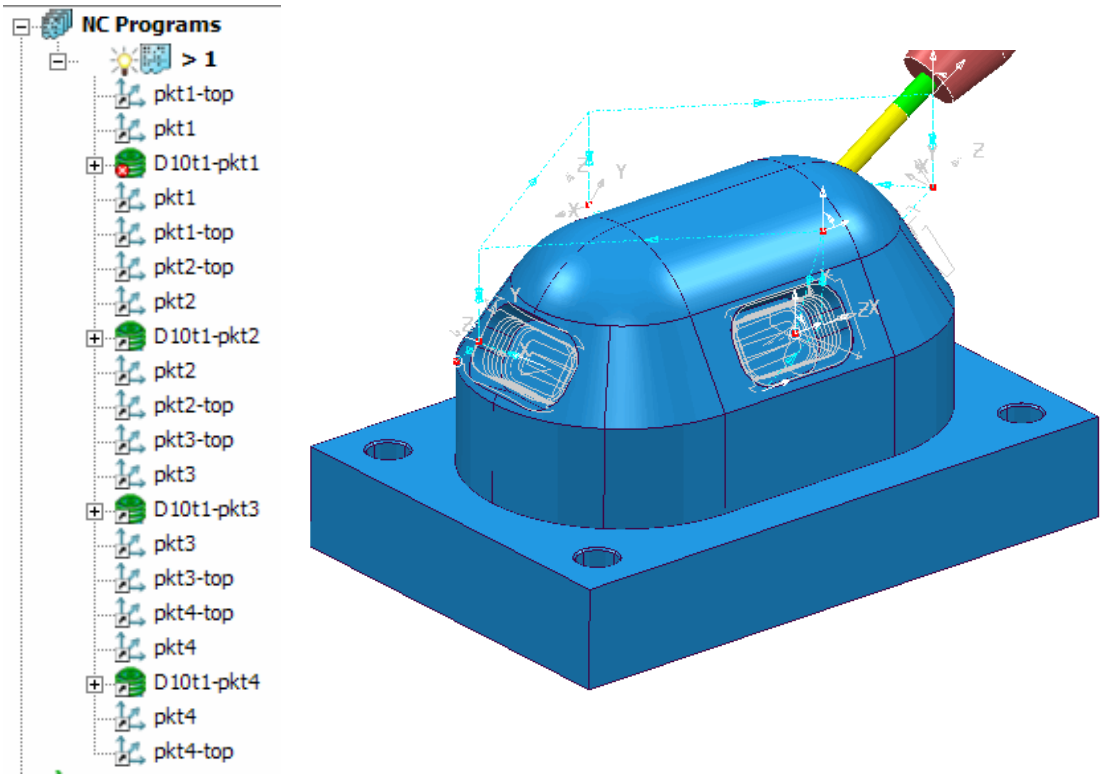
- Select **Apply** to update the **NC Preferences**.
- Create a new **NC Program**.
- Select all **4** existing **toolpaths** and **Add to** the new **NC Progam**.



On closer inspection it will be observed that the **Positional Tool Moves** are ploughing through the component between the individual **machining strategies**.



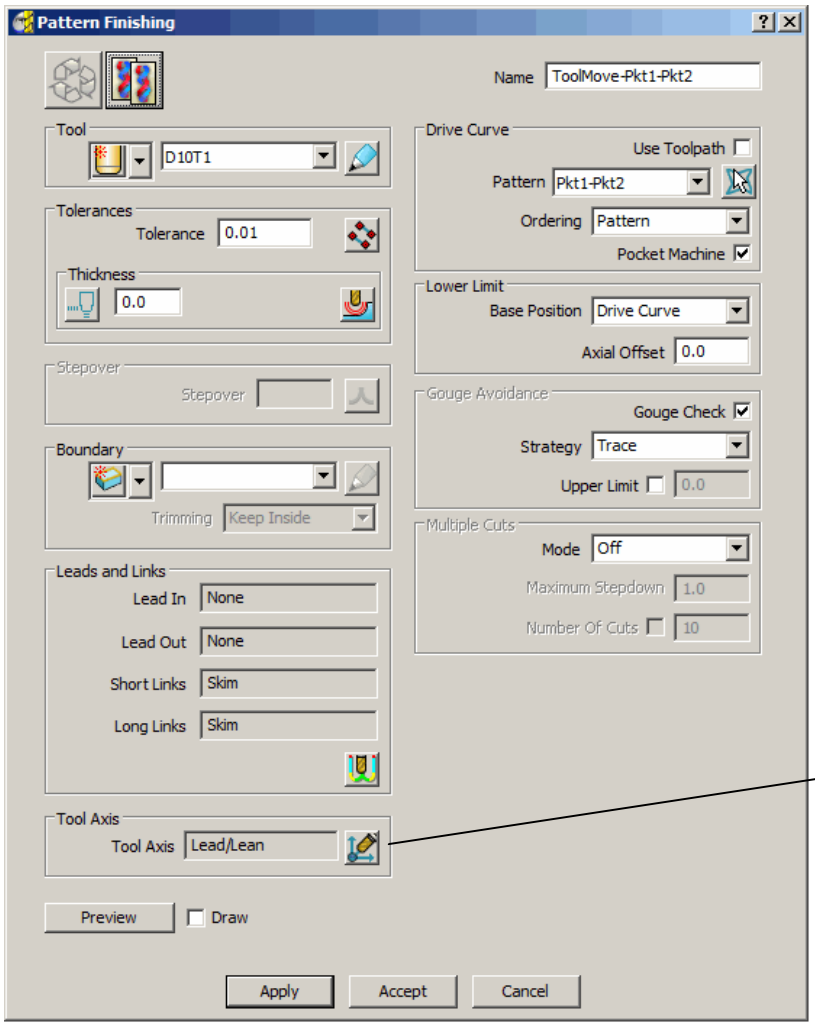
- Use the left mouse to ***drag*** and ***drop*** - **Workplanes** in between the **toolpaths** assigned to the **NC Program** as shown below.



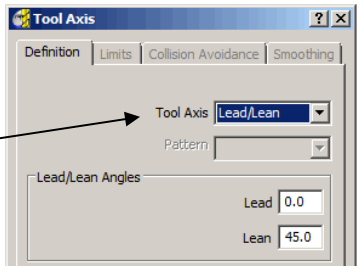
The **Tool** will perform a rapid move, and then align to the Z axis of each inserted **Workplane**. The **Positional Tool Moves** no longer pass through the component.

To further improve the above, **Pattern Finishing** strategies can be used in 3D space to produce a ***smoother transition*** that follows the component form more closely,

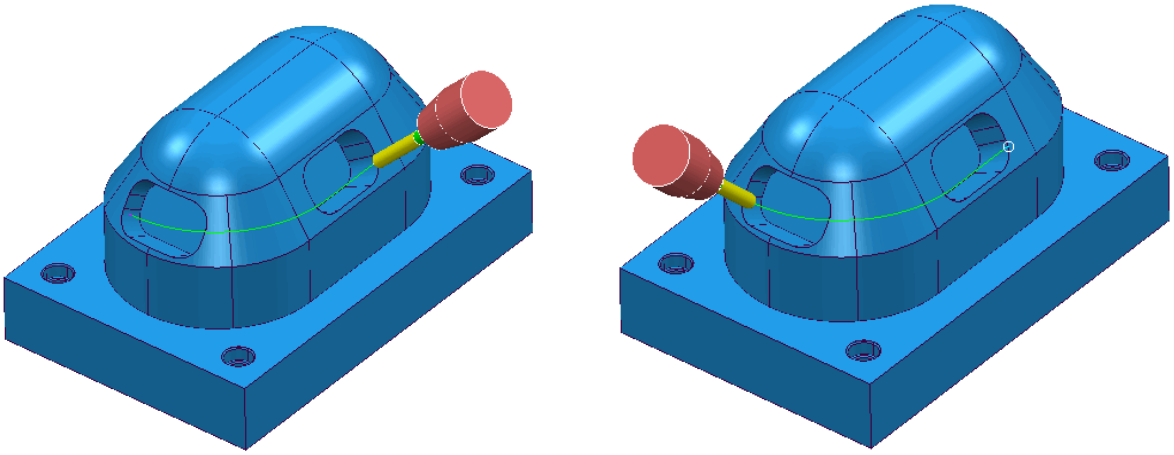
- In **New** strategies, select the **Finishing** tab, followed by the **Pattern Finishing** strategy and enter data exactly as shown below.



- Select **Drive Curve**.
- Use **Tool Axis – Lead/Lean**

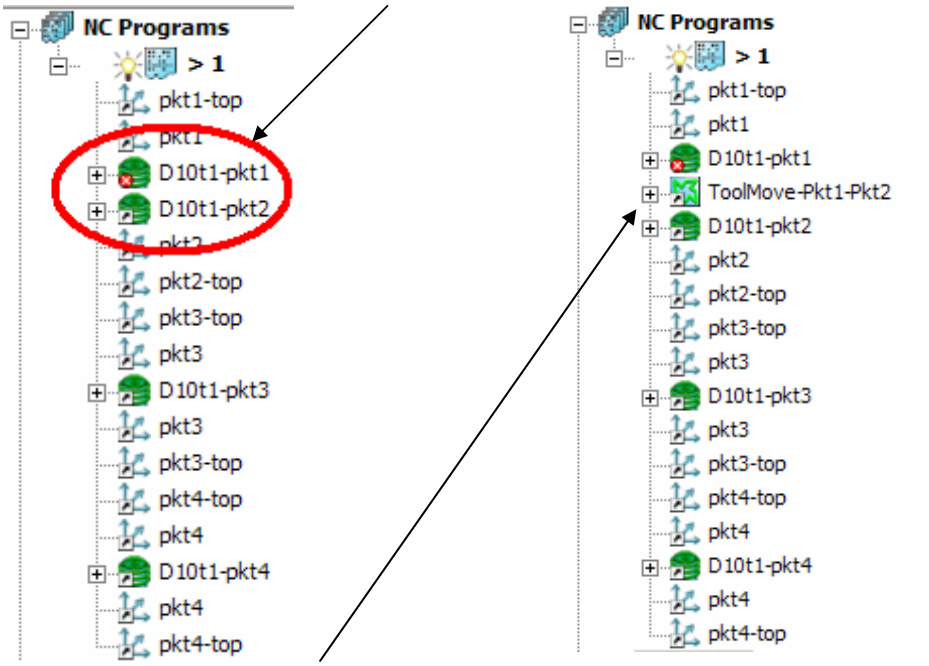


- Input **Lean** as **45**
- **Apply** the strategy.

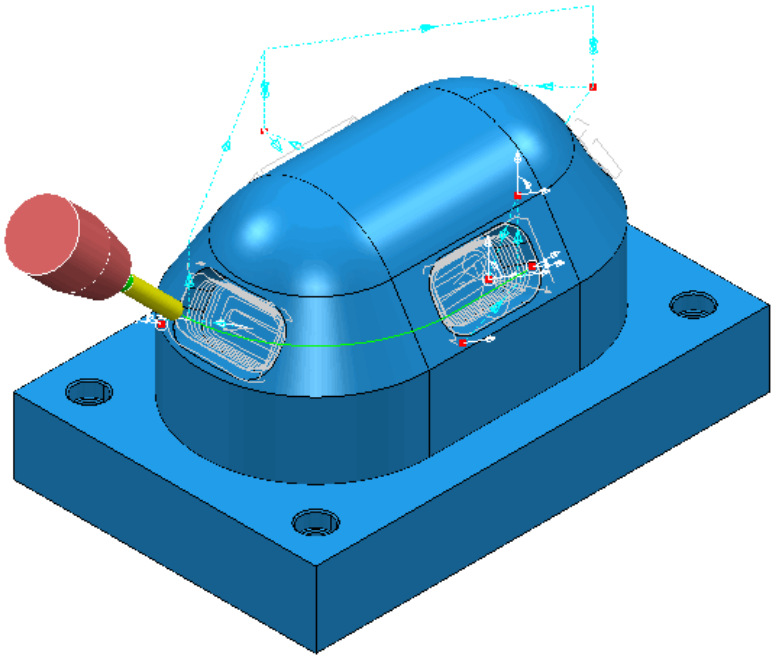


The **Tool** 'winds' along the **Pattern Finishing** strategy creating a collision safe, transition around the component.

- Open the **NC Program** remove all assigned **Workplanes**, **pkt1** between the ***toolpaths***, **D10t1-pkt1** and **D10t1-pkt2**.



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



Exercise

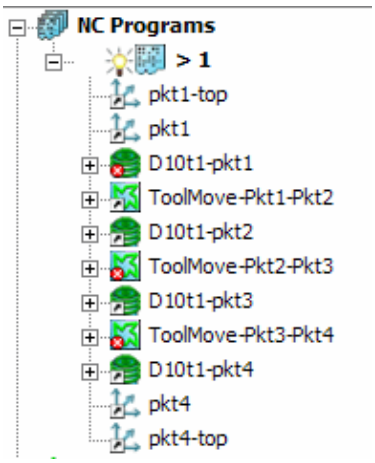
- Create **2** more **Pattern Finishing** strategies for **Positional Tool Moves** between the Toolpaths; **D10t1-pkt2** to **D10pkt3**, and **D10t1-pkt3** and **D10t1-pkt4**

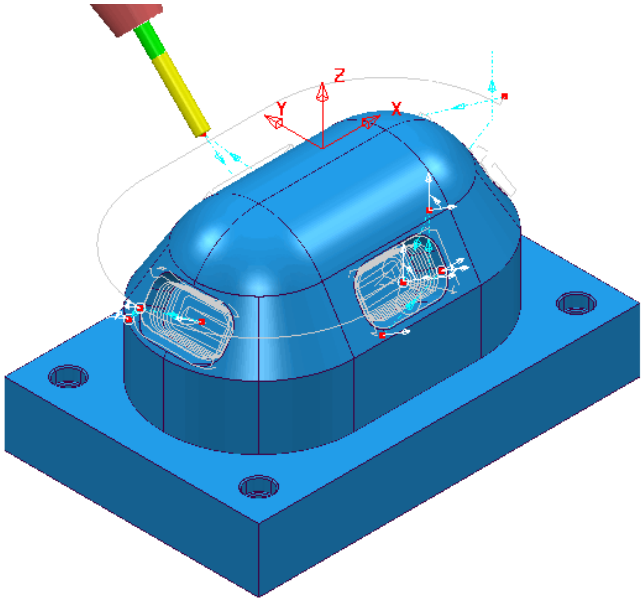
Suggested methods are:-

1. Work with existing toolpath using **Edit – Transform** (**Copy**) – **Mirror**
2. Create 2 copies of existing **Pattern** and reposition, to be used in ***Copies*** of the existing **Pattern Finishing** strategy.

- In the **NC Program**, replace the relevant **Workplanes** with the new **Pattern Finishing** strategies.

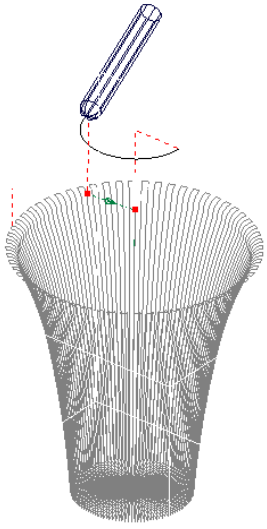
Note: Check the ***direction*** of each new **Pattern Strategy** and if necessary, reverse.





Tool Axis 'Rewind' Move

Note:- 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.



13. Hints and Tips

Useful commands

5-Axis Leads and Links

Leads and **Links** are applied in the same way as for **3-Axis** applications and will automatically take into account the applied **5-Axis** - **Tool Alignment** options .

Useful Preview Commands

There are a number of useful commands that may be used to help examine and check the **5-Axis** toolpaths created in PowerMILL. At present these are typed in via the **Command Window**.

EDIT TOOLPATH ; AXIAL_OFFSET
This command allows a new offset **5-Axis** toolpath to be created from the **Active**, **5-Axis** toolpath with an offset. All of the points of the new toolpath are calculated from the old toolpath but offset along the tool axis vector. The actual toolpath **name** can be inserted instead of **;** if the toolpath is not **Active**.

EDIT TOOLPATH SHOW_TOOL_AXIS 30 0
This command displays the tool axis vectors from an existing **5-Axis** toolpath. The '30' value in the command line above is the length of the vector. This value can be altered to any value.

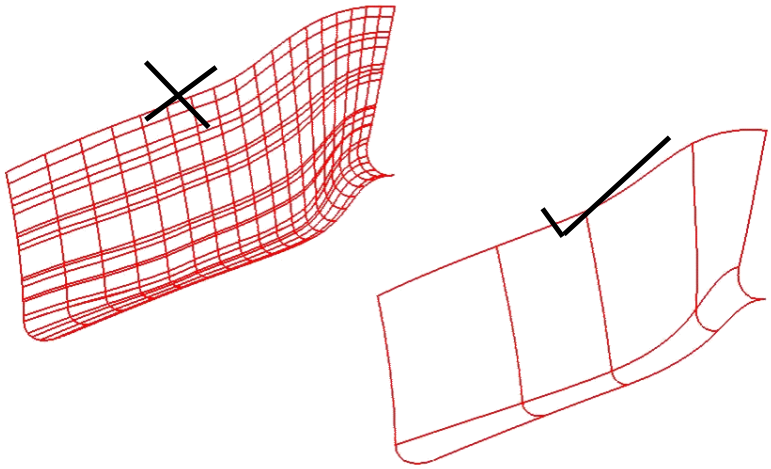
EDIT SURFPROJ AUTORANGE OFF
EDIT SURFPROJ RANGEMIN -6
EDIT SURFPROJ RANGEMAX 6
This command sequence limits the **Surface Projection** distance range to **+/- 6mm**.

EDIT SURFPROJ AUTORANGE ON
This command returns to default no limit to the Surface Projection distance range.

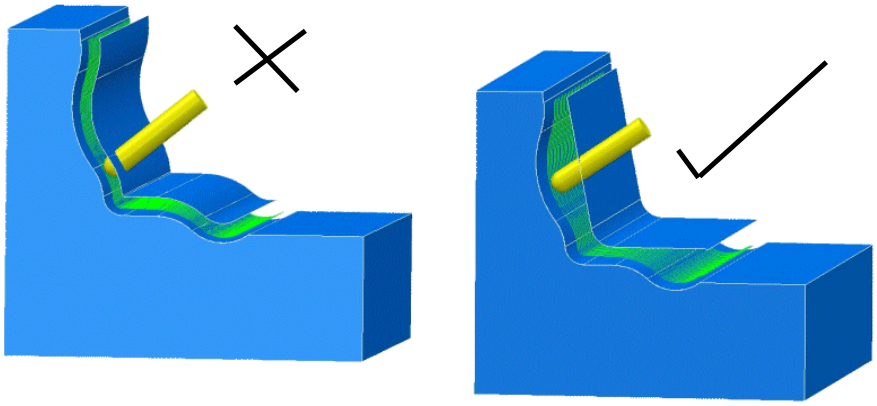
Reference Surface Rules

- Keep them simple.
- Don't follow the form too closely.
- Reference Surface can be Inside/Outside/Both of model, but must be within projection range.
- Avoid discontinuities.
- Aim for uniform parameterisation.
- Using a small projection range results in a quicker calculation.
- Avoid coincident lats/long as it causes duplicate toolpath segments.

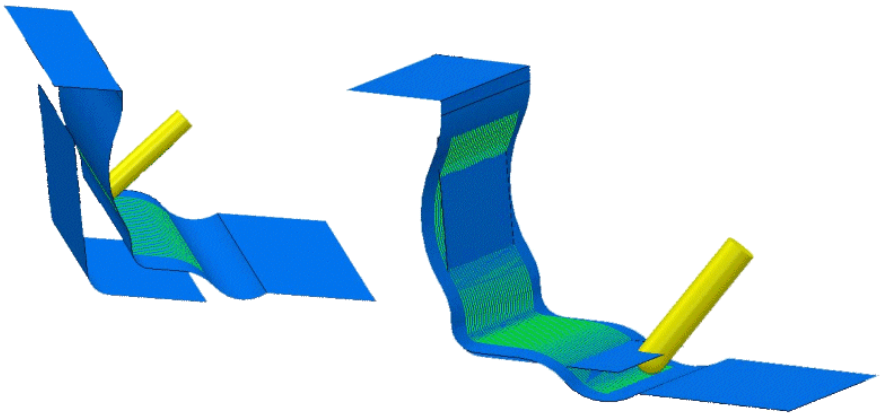
Keep them Simple



Don't Follow the Model form too closely.

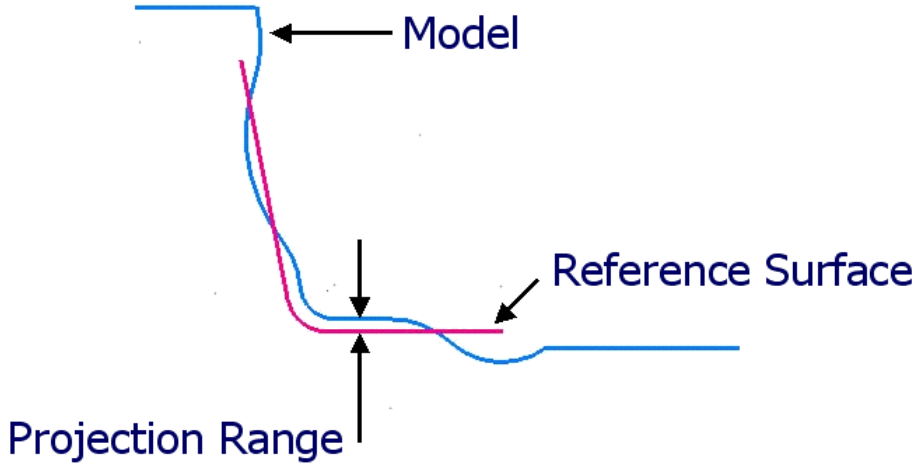


Reference Surface can be Inside/Outside/Both of model, but must be within projection range.



Remember to set the component thickness to ignore if the reference surface is outside or both.

What is the Projection Range?



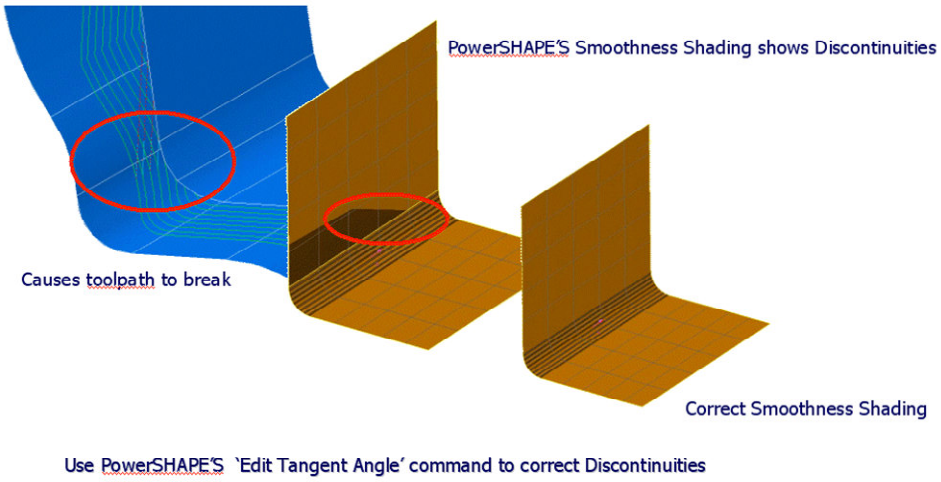
Setting the projection Range

EDIT SURFPROJ AUTORANGE OFF

EDIT SURFPROJ RANGEMIN -10

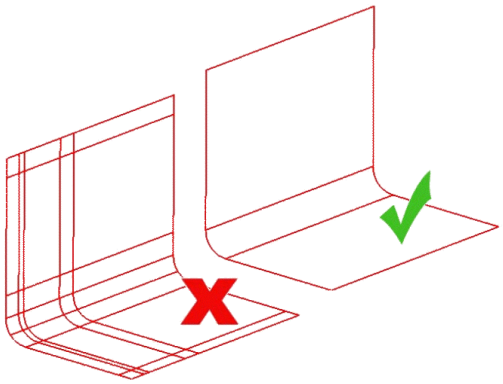
EDIT SURFPROJ RANGEMAX 10

Avoid discontinuities.



Use PowerSHAPE'S 'Edit Tangent Angle' command to correct Discontinuities

Aim for uniform parameterisation.



Surface internal curves can control the toolpath.