The Basics of Milling

Introduction

This chapter is specifically for those who have not yet experienced the Milling process and will endeavour to explain the fundamentals i.e. Machine Tool Axis Configuration, Tolerance and Thickness, Tool Stepover, Climb and Conventional Milling, Cusp Heights etc.

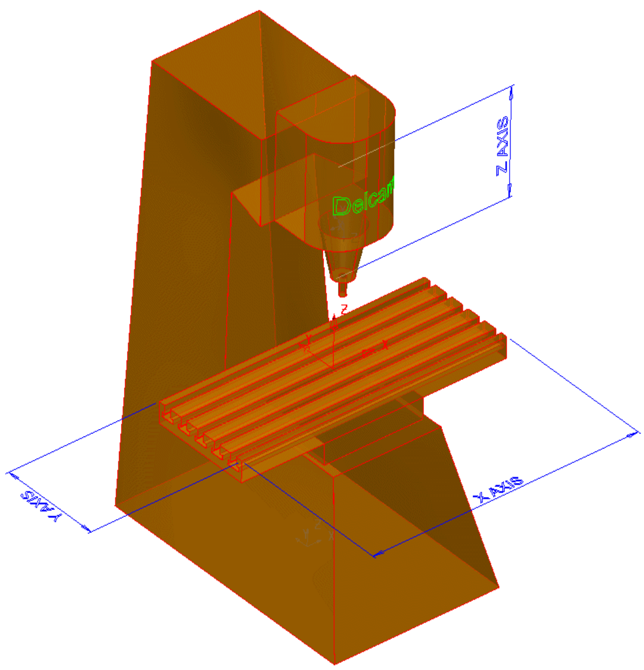
WHAT IS MILLING?

Milling is the process of cutting away material by feeding a workpiece past a rotating multiple tooth cutter. The cutting action of the many teeth around the milling cutter provides a fast method of machining.

THE MACHINE TOOL

In the present climate many different configurations of machine tool exist. Some machines have the table/workpiece stationary whilst the X, Y and Z Axes move and others may be constructed to allow the workpiece/table to be the moving part whilst the axes are fixed.

In any condition the X, Y and Z-axes directions are always configured the same.



The X-axis is always considered as the longest axis, where **X+** will be the table motioning to the left and **X-** to the right.

The Y-axis moves from front to back of the machine with the table motioning towards the operator as the **Y+** (positive) direction and away being the **Y–** (negative) direction.

The Z-axis where the tool normally is located, has the **Z+** (positive) axis motioning up and away from the workpiece/table and **Z–** (negative) direction down towards the workpiece/table.

Some machine tools possess further axes to allow greater versatility when machining complex components.

There could be a fourth axis added e.g. Rotary Table or a five axis configuration. Both can be either manual set-up (operator controlled) or computer numerically controlled (CNC).

3-axis motion normally involves linear moves along the three principal directions (x, y and z), with the tool being at the same orientation for all moves.

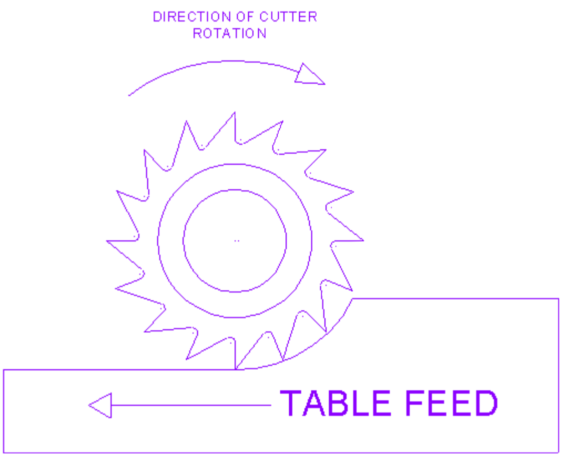
In 5-axis motion there are normally two more rotary axes in addition to the three linear axes (and of course in addition to the tool rotation about its centre line).

METHODS OF MILLING

There are generally two methods of milling, Climb and Conventional.

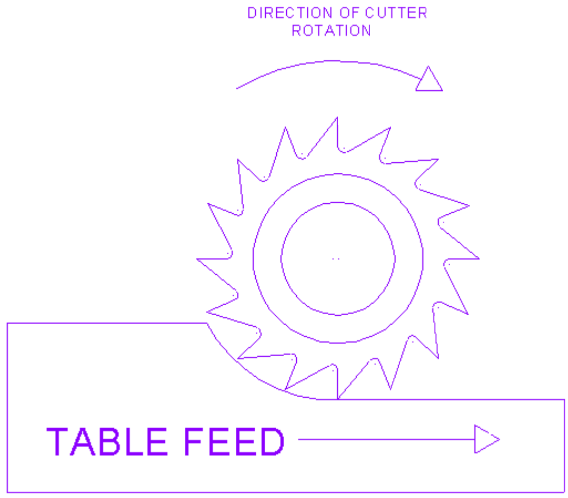
Climb milling, is sometimes referred to as Down milling, where the direction of the cutter rotation is the same as the feed direction. This method is probably the most common option on the shop floor and will normally produce a better surface finish.

Climb Milling



Conventional milling is also sometimes referred to as Up milling where the direction of the cutter opposes the feed direction.

Conventional Milling



TOLERANCE AND THICKNESS

Suitable values are required to control the accuracy and amount of excess material to be left on a component by a toolpath. The parameters used for this purpose are preset and are called **Tolerance** and **Thickness**.

There are two independent locations for setting these values depending on whether the programmer is performing an **Area Clearance** (Roughing) or **Finishing** operation.

The **Tolerance** controls the accuracy to which the cutter path follows the shape of the component. It can be above or below the stated Thickness value.

The **Thickness** is the amount of additional material to remain on the component surface after machining. Again this amount can vary depending on the current Tolerance.

WHAT IS ‘STEPOVER’

The Stepover can be defined as the distance a tool moves between adjacent toolpath tracks. The distance or Stepover value determines whether the surface finish on a component is rough or smooth.

When using a flat-bottomed tool such as an End Mill, the **Stepover** value normally ranges from around **70%** of the cutter diameter.

So a tool with a diameter of **10mm** may have a **Stepover** of approximately **7mm**.

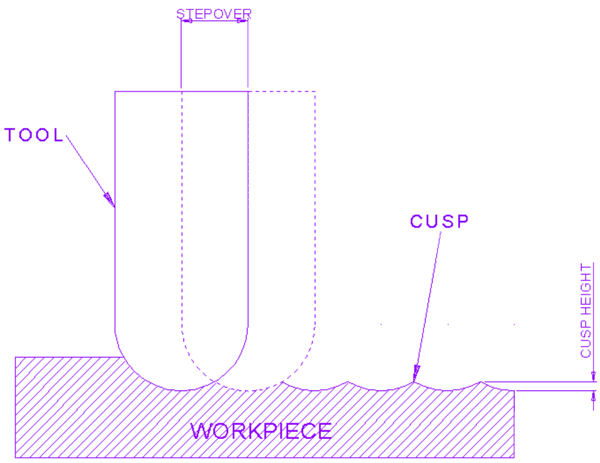
When using a Ball nose cutter the **Stepover** will be considerably smaller when Roughing and Finishing mainly due to the geometry of the tool.

A larger **Stepover** will ultimately give a more rough surface finish than a small **Stepover**.

WHAT IS 'CUSP HEIGHT'

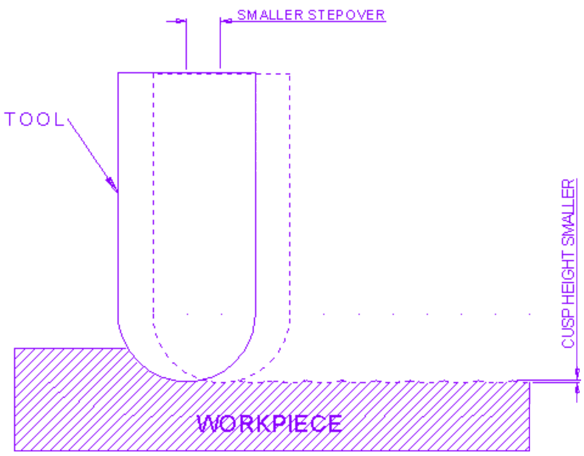
The Cusp is the height of material remaining after subsequent tool stepovers and is directly related to the diameter of tool used, Stepover value, and the Tolerance and Thickness values.

Ultimately a very small Stepover using a Ball Nose cutter will give a smaller cusp height and smoother surface finish than if a larger Stepover was used with the same cutter.



The two diagrams here represent the difference in cusp heights.

Both use the same diameter Ball Nose cutter but the lower diagram shows a smaller Cusp height because the Stepover is reduced hence a smoother surface finish.



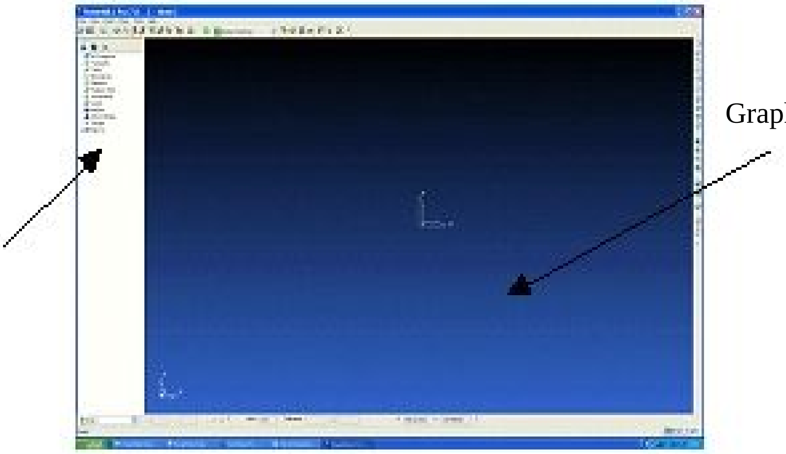
1. Introduction

Introduction.

PowerMILL is a stand-alone machining package, which can quickly create gouge free cutter paths on imported component data. **PowerMILL** supports **Wireframe**, **Triangle**, **Surface**, and **Solid** models created by other Delcam products or from neutral formats such as IGES. If the relevant **PS-Exchange** translators are purchased **PowerMILL** will directly import data created by the majority of non-Delcam packages.

PowerMILL Environment

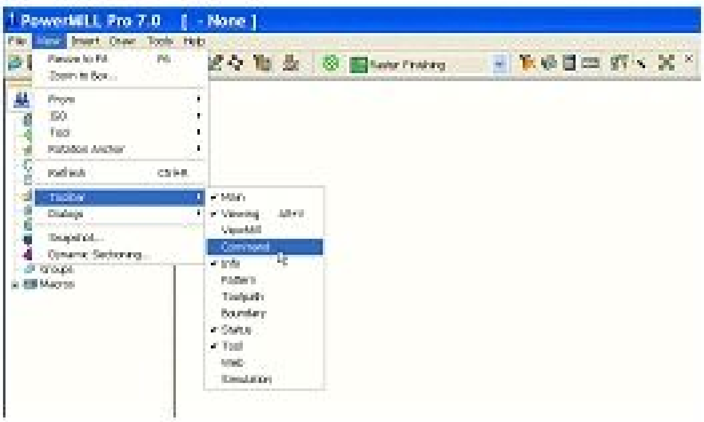
Double click on the **PowerMILL** icon.



Graphics Area

Explorer area

Pull Down menus are located across the top of the **PowerMILL** window. By placing the mouse over the menu and clicking with the left mouse key, this will open up the relevant sub menu. Further selection can be done by moving the cursor along a right arrow ( )



The **Main** toolbar is as shown on the following page. Each icon has a specific function and by holding the cursor over it, an appropriate description (or ToolTips) is displayed.

On the right hand side of the screen is the **Viewing** toolbar. By selecting one of the icons a different view of the model and global transform is displayed in the central or graphics area.

View along the X-axis
View along the -Y axis
View along the Z-axis
View along the -X axis
View along the Y-axis
View along the-Z axis
Isometric View 1
Isometric View 2
Isometric View 3
Isometric View 4
Resize to fit
Zoom In
Zoom Out
Zoom to Box

Last View
Refresh
Block

Plain Shade and access to other shading options
Wireframe

Shade options from left to right:-

Thickness - Machining Mode - Default Thickness - Default Machining Mode - Draft Angle - Minimum Radius - Multicolour - Plain

Mouse buttons

Each of the three mouse buttons perform a different dynamic operation in **PowerMILL**.

Mouse button 1: Picking and selecting

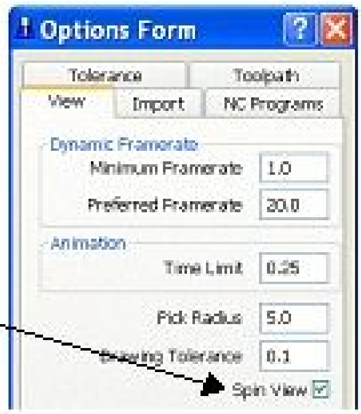
This button is used for selecting items off the pull down menus, options within forms, and entities in the graphics area.

Mouse button 2: Dynamics

Zooming in and out: - Hold down the **CTRL** key and mouse button 2. Move the mouse up and down to zoom in and out.
Pan around the model: -Hold down the **SHIFT** key with mouse button 2. Move the mouse in the required direction.
Zoom Box - hold down the **Ctrl** and **shift** key, drag a box around the area to zoom into using the middle mouse button.
Rotate mode: Hold down mouse button 2 and move the mouse, and the rotation is centered about the trackerball.

View Spinning- Dynamically rotate the view and quickly release the mouse. The faster the mouse the faster it will spin. This feature is switched off by default.

Select **Tools -> Options**, select the **View tab** and tick the option **Spin View**.



Mouse button 3: Special Menus & PowerMILL Explorer Options

When this button is pressed it brings up a local menu relating to whatever the mouse is over, such as a named item in the **PowerMILL Explorer** or a physical entity in the graphics area. If nothing specific is selected the **View** menu appears.

Example 1

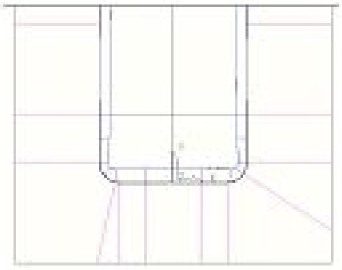
For the first exercise an existing model will be imported and used to illustrate some of the basic visual display options.

Select **File -> Import Model**.

The open examples form appears and the large icon **eg** provides access to an area within the **PowerMILL** product software tree where sample models are stored. The icons marked **1** and **2** can be customised by the user to locate areas where data for live jobs are stored.

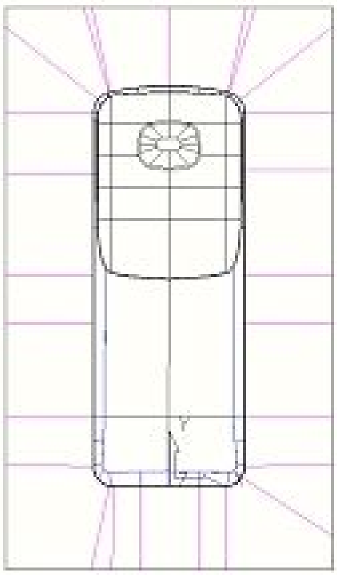
Click on the **eg** icon.

Select the **phone.dgk** model and then click on **Open**.

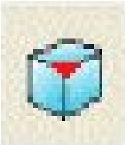


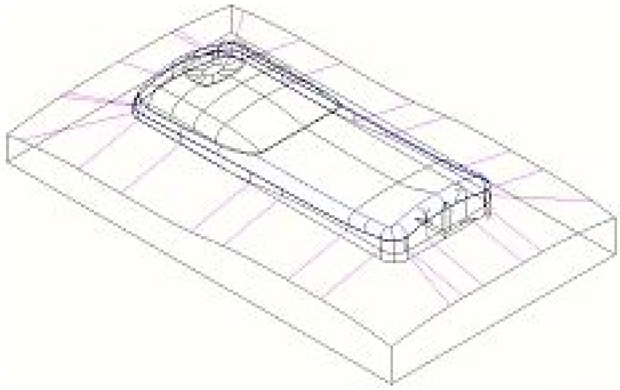
The phone model is displayed as a wireframe viewed down the Z-axis. Only part of the model is visible, as it is too large to fit in the current view. To display the whole model the **Resize to fit** icon in the **Viewing** toolbar is selected.

In the **Viewing** toolbar, select the **Resize to fit** icon.

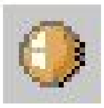


The view of the model is scaled to the full screen.

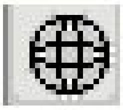
Select the **ISO 1** icon. 



You can now see the entire model quite clearly in an isometric wireframe view. To see it shaded you need to select the shaded wire option.

Select the **Shaded Model** icon. 

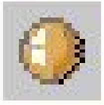
This displays the shading on top of the wire. To remove the wire view and show the model as shaded only, then click on the pressed in wire button.

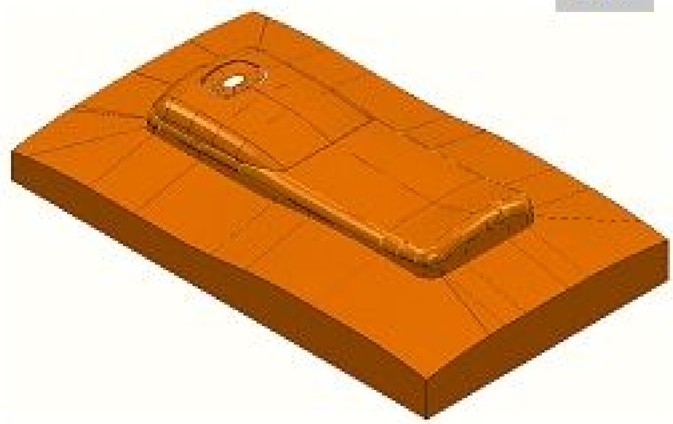
Select the **Wire Model** icon. 

By pressing the wire icon again, the wireframe is hidden showing the model only in shaded.

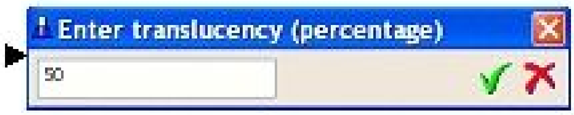
Click on the **Shaded Model** icon and then click on the **wire view** icon.
Try the other **Viewing** icons and observe the results.

Sometimes it is useful to be able to see inside a model. In order to do this, the model can be made translucent.

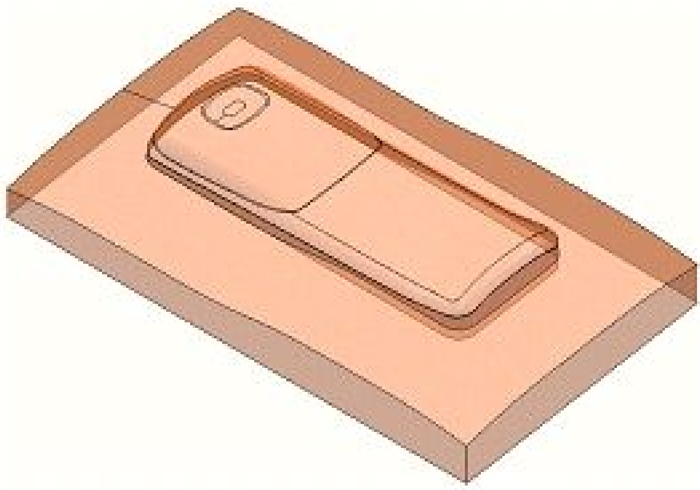
Select the **Shaded Model** icon. 



Right Click over the **Mode**l and select **Translucency** from the menu



Enter the percentage **translucency** you require
(0 percent = opaque, 100 percent = transparent)



The model is displayed in translucent shaded mode allowing the user to see internal details

To return to normal shading set the **translucency** to **Zero**.

PowerMILL Panes

On the left hand side of the screen above the Explorer are the PowerMILL panes.

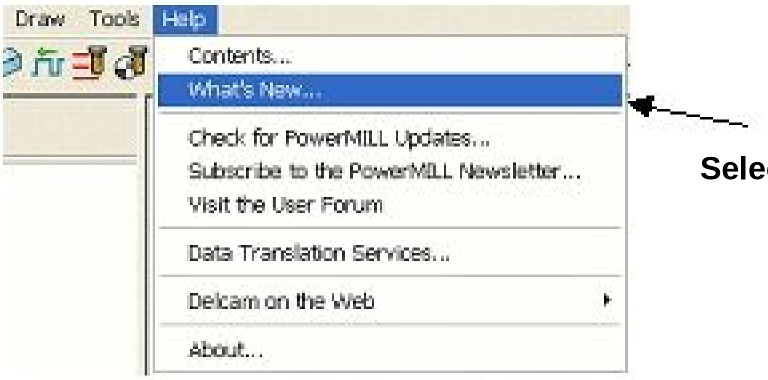
These are used extensively to help organise your machining. The standard format pane is denoted by the **PowerMILL** symbol and contains the explorer tree categorised into NC Programs,Toolpaths, Tools, Boundaries, Patterns, Feature Sets, Workplanes, etc.

The second pane is the HTML browser used for viewing HTML files or Help files and the third pane opens the **PowerMILL** Recycle Bin.

Select **File -> Delete All.**

PowerMILL Help

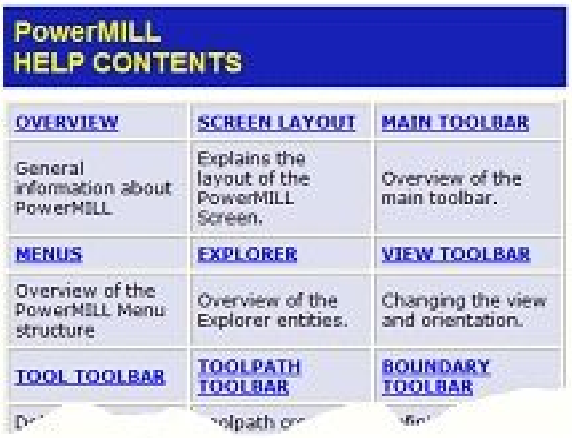
PowerMILL comes complete with it's own **On-Line Help** which is accessed via the help menu.



Select Help -> What's New.

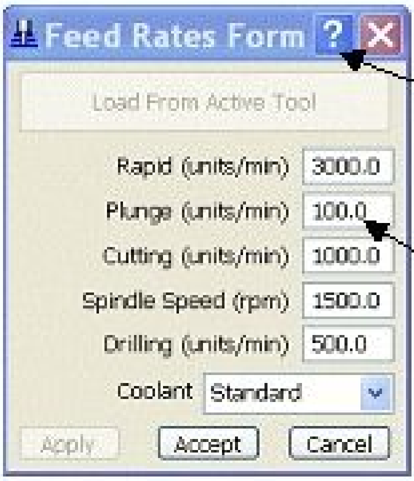
An summary of all the new functionality available in the current version of **PowerMILL** is loaded into the html pane.

Select **Help -> Contents**



Another extremely useful feature is the Context sensitive Help available on all forms.

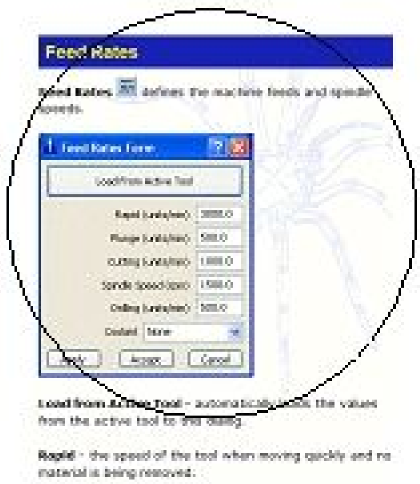
Select the **Feedrate** icon from the top toolbar to bring up the **Feedrate** form below



Select the Question mark at the top of the form

The cursor will be displayed with a question mark next to it.

Click on any of the white areas on the form.



Help on that specific form or topic will be displayed

Select **File Close Project**.

2. Machining Set up

Preparation before creating toolpaths

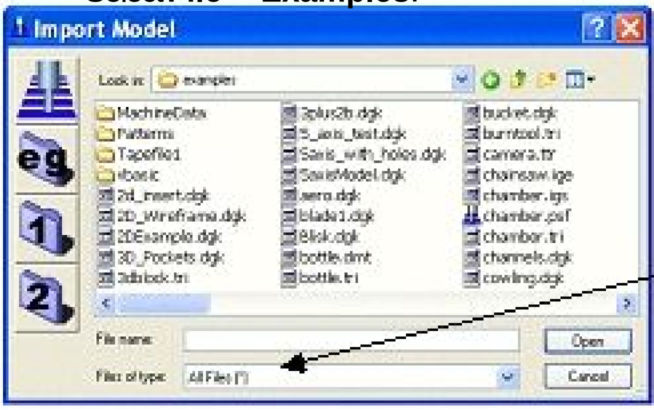
The following list is a summary of basic setting up procedures required before the toolpaths are created.
It is important to spend a little time gathering as much information as possible about the model before actually applying toolpaths.
PowerMILL has some unique tools that allow parts of the model to be measured whether it is a gap or an internal radius.
Various shading methods can also be used to get a quick idea visually of the minimum radius and draft angles.
An imported model might be in an unsuitable orientation for machining in which case **PowerMILL** allows the user to change the coordinate system as required.
Some of these methods will be used within the following chapter.
The following list could be used as a check list for setting up any job within **PowerMILL** and generally represent the basics that should be followed before applying any toolpaths.
Note:- Items 8 9 &10 below can be redefined retrospectively on an existing toolpath.

1. **Loading a model into PowerMILL.**
2. **Viewing the model**
3. **Orientating the model around an Active Workplane where required.**
4. **Gather information on the model i.e. Minimum radius / Draft angles.**
5. **Measure the model.**
6. **Material block definition.**
7. **Cutting tool definition.**
8. **Feed rate and Spindle Speed Settings.**
9. **Rapid Move Heights.**
10. **Tool Start and End Point.**
11. **Save the Project.**

1. Loading a Model into PowerMILL

A selection of sample model files are supplied and installed with **PowerMILL** and these are retrieved from a default directory called **Examples**.
The usual method of importing a model into **PowerMILL** is by selecting **File > Import Model** from the top pulldown menus.

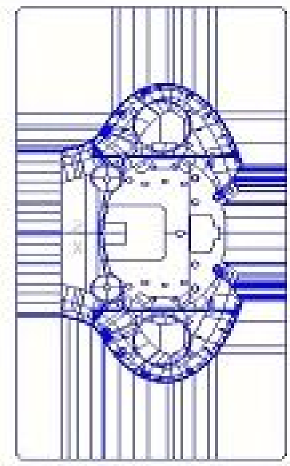
Select **File ->Examples**.



A variety of different types of **Model** format can be **Imported** into **PowerMILL**. These can easily be discriminated on the form using the filter **Files of type** to widen or narrow the choice for file extension.

Click on the file name **speaker_core.dgk** and then **Open**.

Select **View from top (Z)**  followed by **Resize to fit**  from the **Viewing toolbar** to the right of the graphics area.

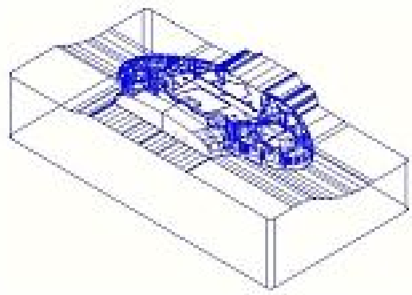


The model will be displayed (as shown) in the **PowerMILL** graphics area looking down the Z-axis with X aligned from left to right and Y from bottom to top.
In most cases the X dimension of a machine tool table will be greater than Y in which case the longer side of the component may be in excess of the travel limits in Y.
If this is the case it will be necessary to align the component with the longest side along X to ensure that it can be positioned within the travel limits.

2. Viewing the Model

Although the model is displayed it is a good idea to look at it from all angles to fully understand its size and features.

Select an isometric **ISO 1** view. 



Although it is possible to rotate the actual component this is not the generally the most effective approach. An additional moveable datum (**Workplane**) will be created and rotated through 90 Degrees to effectively create the condition that the longer side of the component is parallel with the front of the machine.
The original coordinate set-up can then easily be re-activated for tasks such as checking dimensions.

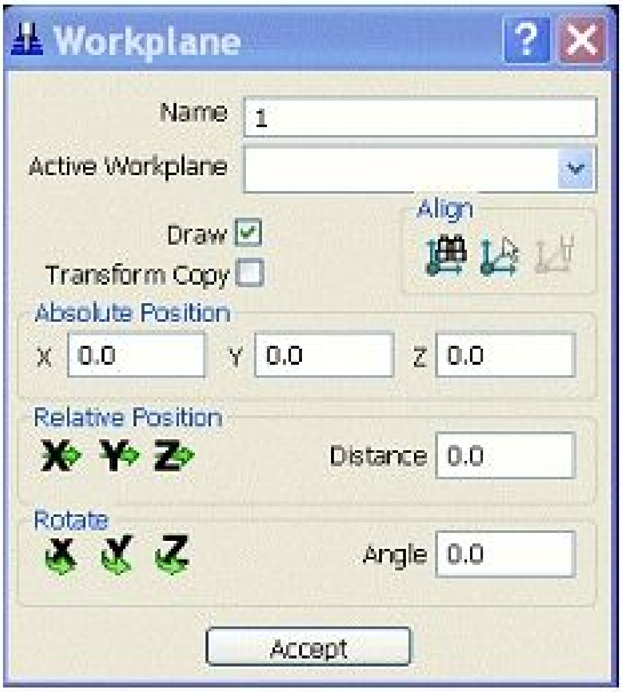
3. Orientate the Model using a Workplane.

The model needs to be rotated to leave the longest lower front edge of the model aligned to the front of the machine tool i.e. along the X-axis.
Note. It is best practise to create a **Workplane** (moveable datum) and **Rotate** it about the **model**.

Right click over **Workplanes** in the **PowerMILL Explorer** and select **Create Workplane**.

Workplanes are alternative or additional component datums that can be moved and/or orientated within the global environment. They are frequently used in **3 Axis** machining and are an essential item in the application of **3plus2** and **5-Axis** machining strategies.

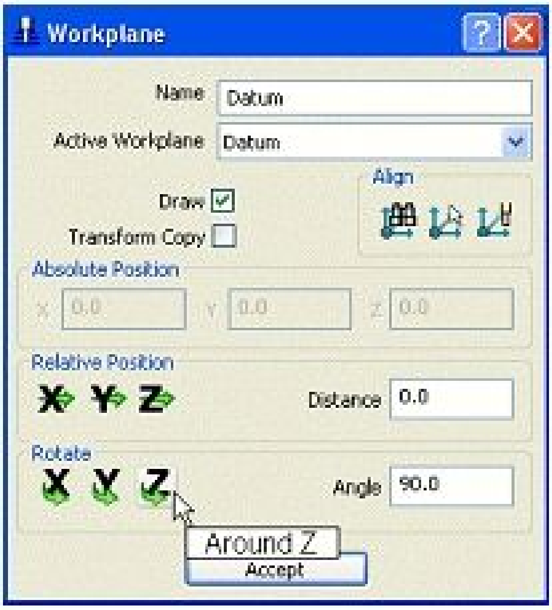
The **Workplane** creation and editing form will appear.



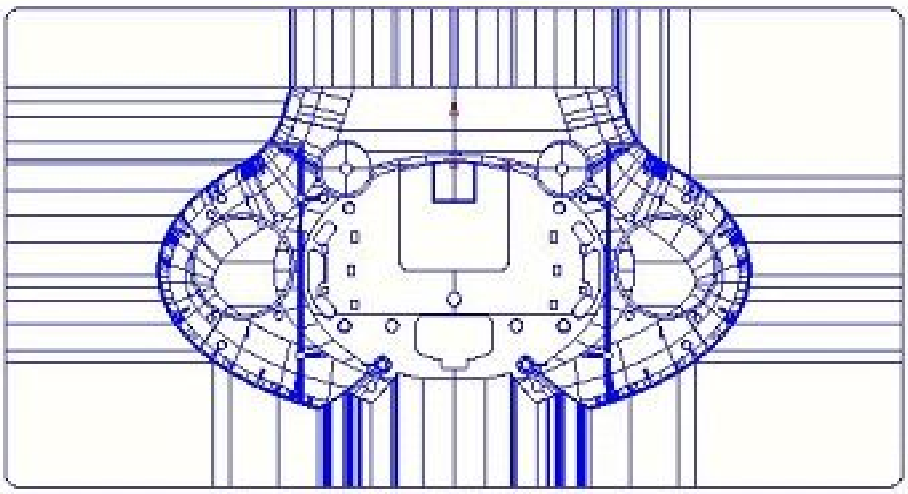
For easy identification it is recommended to appropriately rename **any** entity created in the **Explorer**.

In the above form modify the ***Name*** to **Datum**.
In the above form set the ***Active Workplane*** to **Datum**.

The next step will be to rotate the new **Active Workplane** to indirectly re-orientate the **model**.
Note:- It is also possible (but not generally good practise in the long term) to **Rotate** and/or **move** the actual **model** relative to the active coordinate system.



Enter ***Angle*** **90.0** before selecting the ***Rotate*** **- Around Z** icon.
Accept the form.

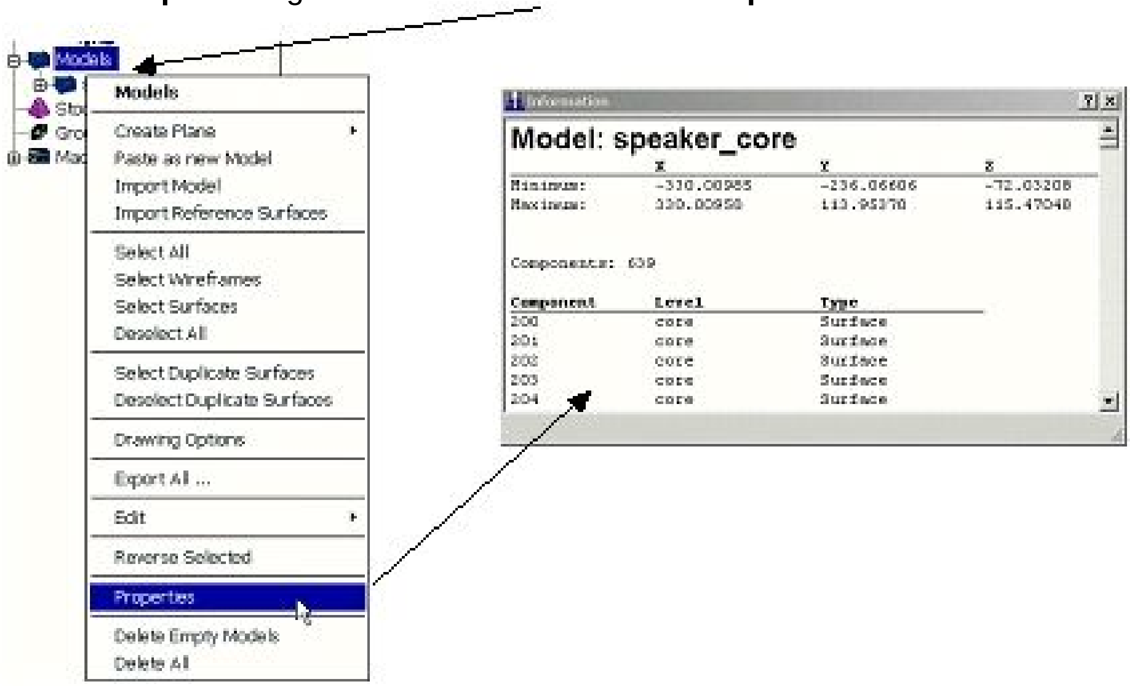


Select a **View from top (Z)** and observe the effect of the rotated, **Active Workplane** producing a more suitable machining position for the **model**.

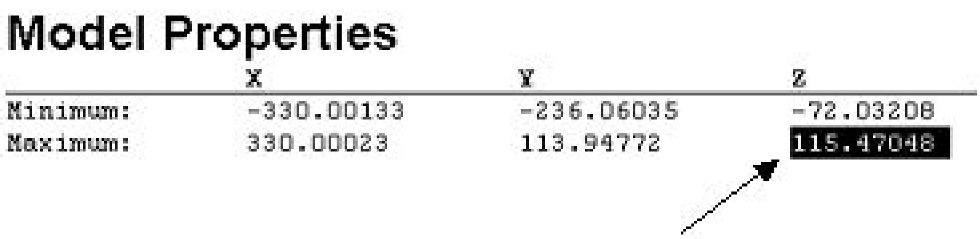
It will not always be necessary to create and rotate or move a **Workplane** after import into **PowerMILL**. It depends on the original, orientation of the model when exported from the CAD software.

Information regarding the model dimensions in relation to the world datum (**Transform)** or (if present), an **Active Workplane** can be obtained.

In the **explorer** Right click over **Models** and select **Properties**.



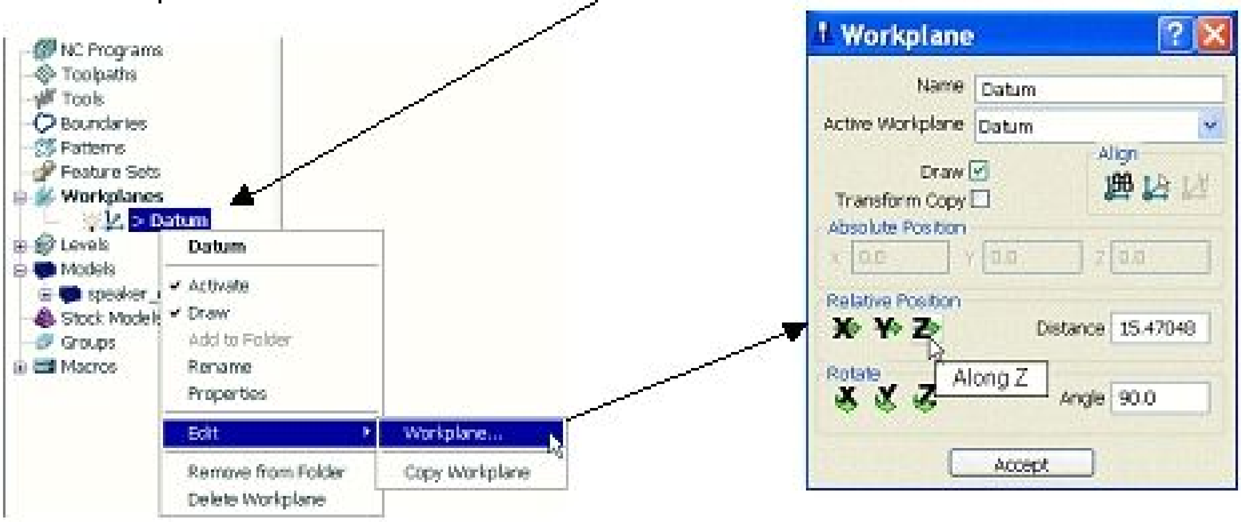
The values in this form can be **copied** (**Ctrl C**) and then **pasted** (**Ctrl V**) into other forms.
The **Workplane is moved** up in the **Z plane** so that it is situated at the maximum height of the model.
The form shows that the current maximum **Z** value is **115.47048mm**. It will be necessary to move the **Workplane** by **115.47048** to position it at the maximum height of the model.



Highlight the **Maximum Z** value (**115.47048**) by swiping over it with the left mouse button and press **Ctrl C** on the keyboard to **Copy** the value to the buffer.

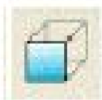
Eject the **Model Properties** form by clicking **X** in the top right corner.

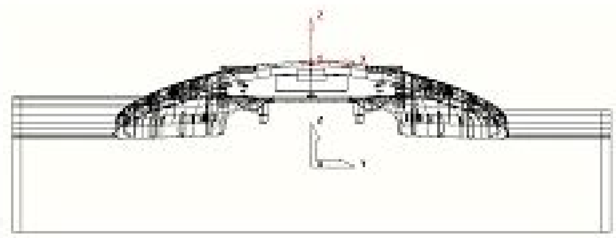
In the **explorer** Right Click over the **Workplane** named **(Datum)** to access the local pull down menu.



Select **Edit - Workplane** to open the form (above right) and use **Crtl V** to paste the previously stored value (**115.47048**) into the ***Distance*** box before selecting the ***Relative Position*** **- Along Z** icon.
Accept the form.

Select a **Y- view**.

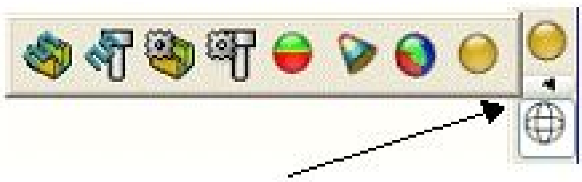




The **Workplane** is now in position on the top of the model.

4. Minimum Radius and Draft Angle shading views.

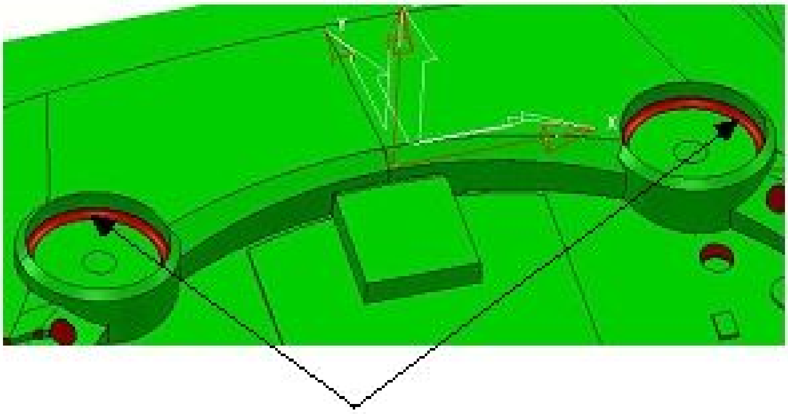
Visual checks can be made quickly with the use of two shading options found in the views menu on the right hand side of the screen.
It is useful to know before generating tools and toolpaths what the minimum radius is on the model and also whether there are any undercuts or draft angles.



Open the **Shading Toolbar** by clicking the small arrow as shown above.

Select the **Minimum Radius Shade** icon on the **toolbar**.

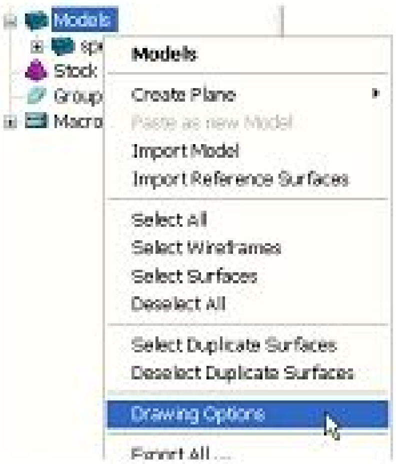
Any internal radii that are smaller than the specified **Minimum Tool Radius** will be shaded **RED**. The settings are located in the **explorer - Model** pull down menu in **Drawing Options**.
Examine the model to identify areas that are inaccessible to the specified radius (shaded Red).

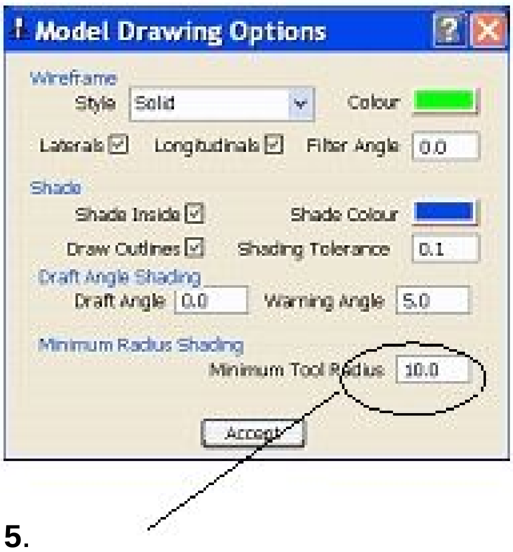


The two internal radii can be seen shaded **red** visually identifying that they will not be machined to their correct size if the default value tool was used.

The specified **Minimum Tool Radius** can be modified to suit in the **Drawing Options** area within the **Models** menu in the **explorer**.

In the **explorer** Right click over **Models** and select **Drawing Options**.





Change the **Minimum Tool Radius** value to **5**.

The shading on some parts of the model has changed from Red to Green signifying that from a finishing viewpoint these local areas are fully accessible to a **Dia 10 Ball Nose** cutter.

Change the **Minimum Tool Radius** value to **2**.

All of the red areas have now disappeared which suggests that the maximum tool size guaranteed to access all areas of the component would be a **Dia 4 Ball Nose** cutter.

The model can also be visually inspected for the size of draft angles and undercuts.

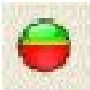
Select the **Draft Angle Shade** icon on the shading toolbar. 

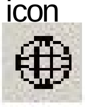
The model is shaded in three different colours, red, green and yellow.
The red areas represent angles equal to or less than the current **Draft Angle** specified in the **Drawing Options** form (default is 0).
The green areas represent angles above the current **Warning Angle** specified in the **Drawing Options** form (default is 5).
The yellow areas represent the areas between the current **Draft Angle** and **Warning Angle**.
On this particular model the yellow areas represent angles between 0 and 5 degrees.

To check for undercuts change the **Draft Angle** to **-0.2** and the **Warning Angle** to **0.2.**

All of the red areas have disappeared and all that remain are green and yellow. If any red areas remain then these would indicate an undercut situation greater than -0.2 degrees.
The yellow areas indicate on or near vertical faces because the difference between the Draft and Warning Angles is so small.

Accept the **Drawing Options** form

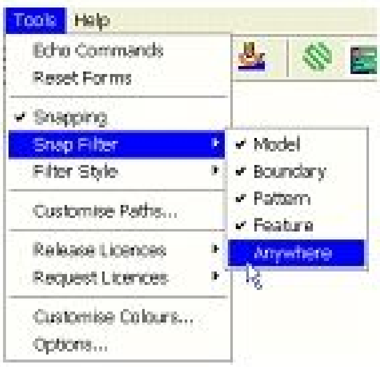
Select the **Draft Angle Shade** icon  again to turn off the shading.

Make sure the **Wireframe**  icon from the **View toolbar** is on so the model is displayed in Wireframe only.

5. Measuring the model.

The user may require dimensional information relating to certain features on the model. A measuring tool is provided in **PowerMILL** that allows the user to snap in the graphics area to obtain dimensions based on points lines and arcs.
Before any such measurements can be taken the **PowerMILL**, **Snap Filter** will need to be modified.

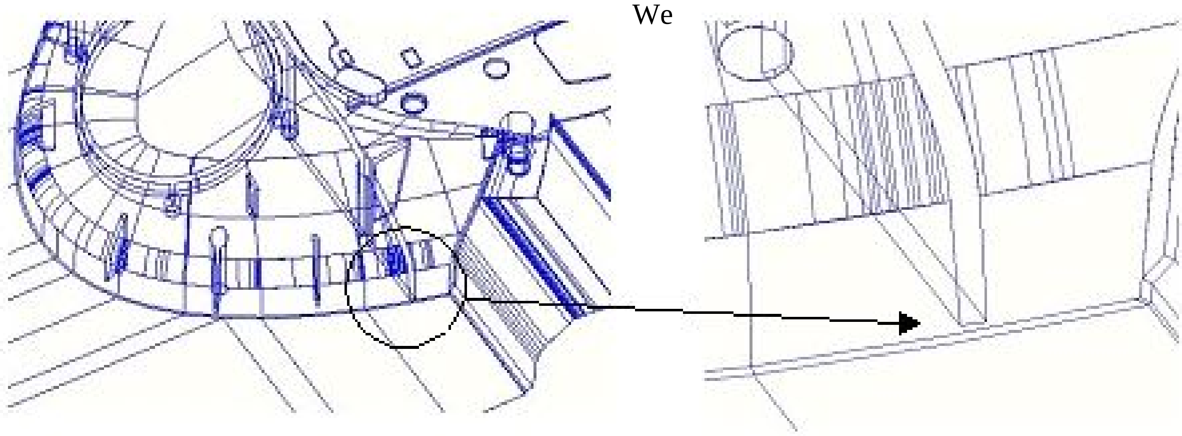
From the top **Pull Down** menus select **Tools > Snap Filter** and use the left mouse key to **untick - Anywhere**.



If **Anywhere** is unticked then measurements can be only be snapped onto the remaining ticked entities and not in free space.

Zoom into the area shown by the arrow below.

We



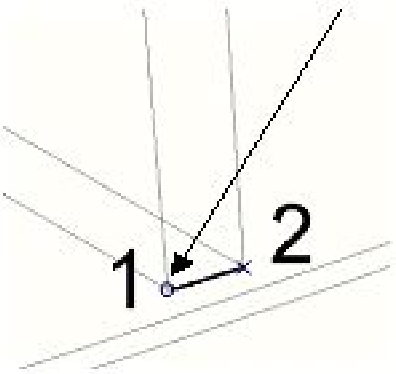
The gap at the bottom of the slot will be measured to determine its size and depth.

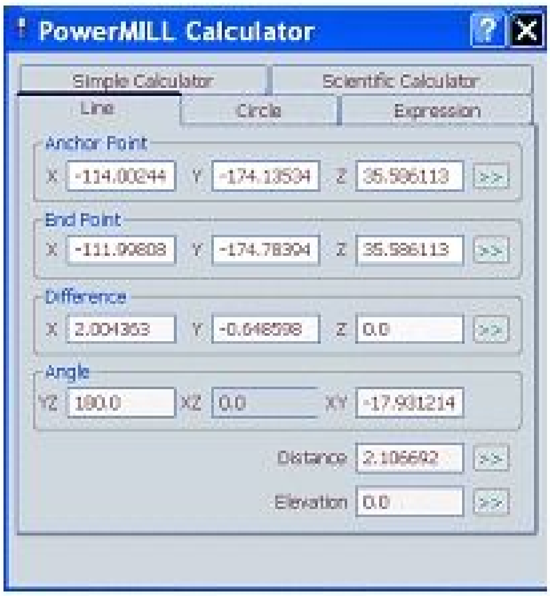
From the **Main toolbar** select the **Measure model** icon.

The Model Measurement for appears defaulted to **Line**. An Anchor Point is required to commence measuring.

The **PowerMILL Calculator** form will open in **Line** measurement mode. Using the left mouse button drag a window around (or snap) point **1** shown below to display the XYZ coordinates in the form.

Drag a window around (or snap) point **1**.





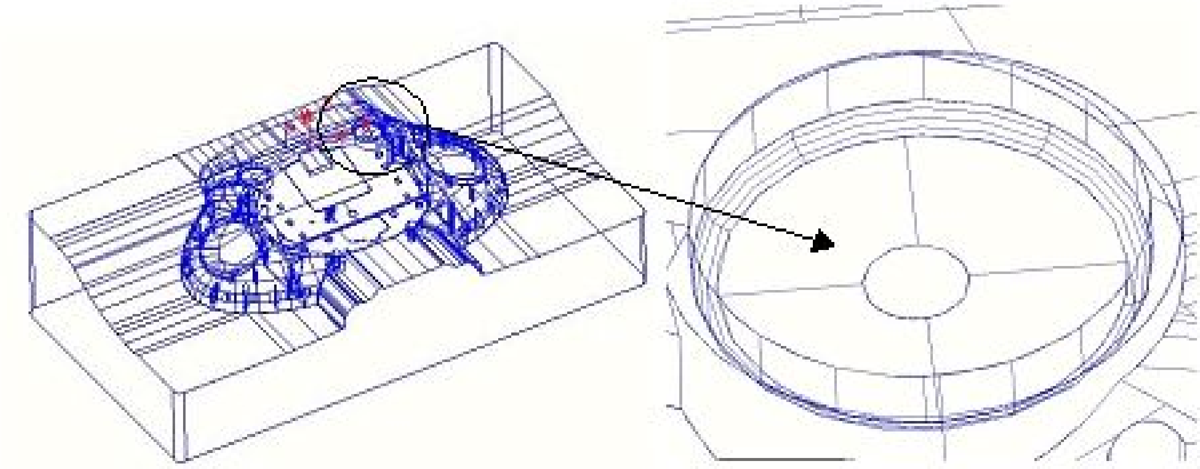
The **Anchor Point** is now selected and is represented by a small circle.
The x, y and z values seen in the above form are relative to the **Active Workplane** '**Datum**'.

Drag a window around (or snap) point **2** to obtain the final 'point to point' measurement information.

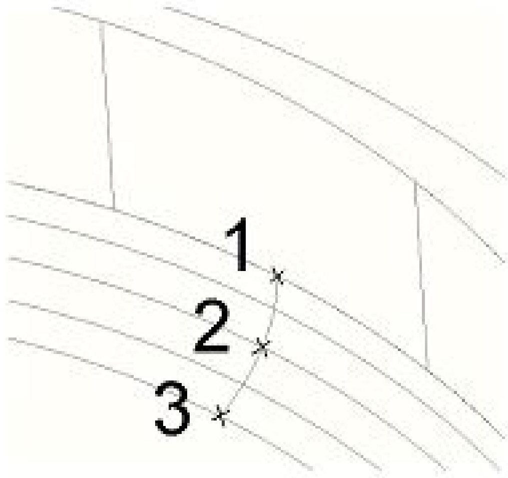
A temporary line appears connecting the two points, and the information relating to the line is displayed in the form.

Minimum Radius is measured via the **Circle** tab combined with snapping three points on the model.

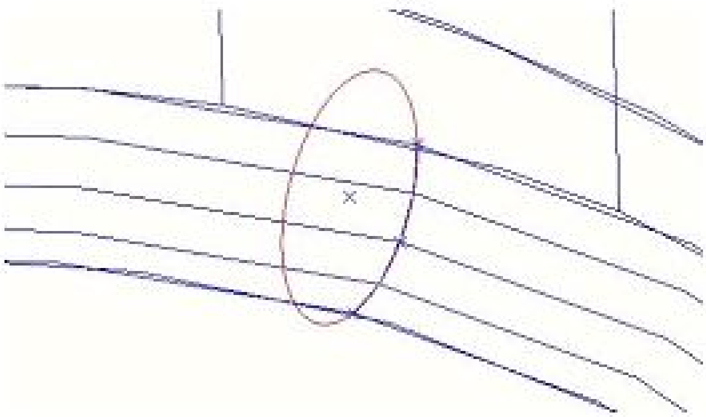
Select the **Circle** tab on the **Model Measurement** form and zoom into the area shown below.

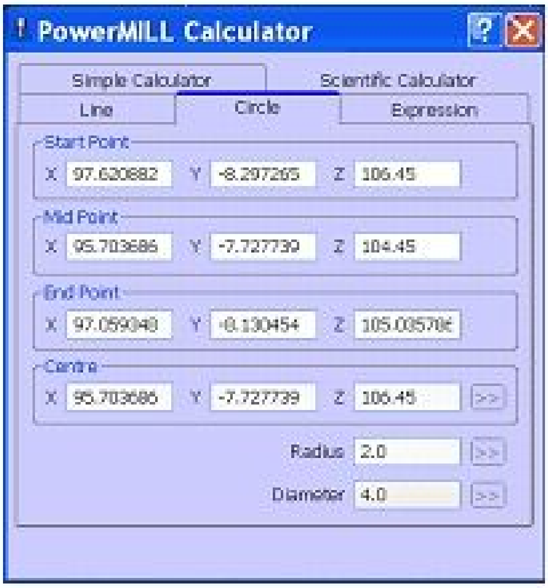


Select **three points** along the arc either by dragging a small window around, or snapping onto each one as shown below.



A temporary circle will appear after the third point is selected as shown.





The **Circle** measurement form will display details of the arc as shown.

Close the **Model Measurement** form and select an **ISO 1** view.

6. Material Block definition.

The **Block** usually starts out as the initial material size prior to machining the component. It is often created with more specific dimensions to provide a more localised limit for individual machining strategies. There are also other methods available for a more complex definition of the area in which a machining strategy operates.

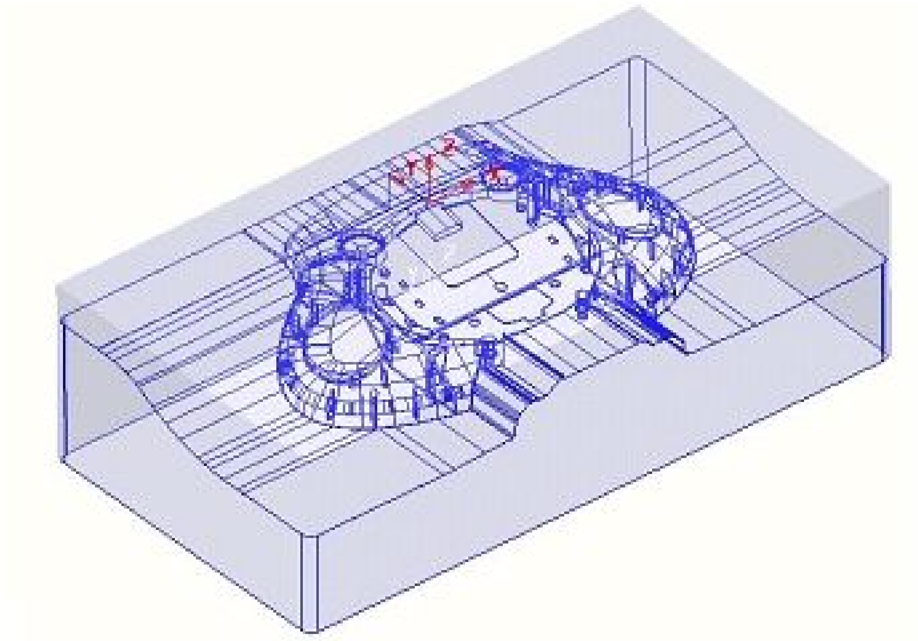
Click on the **Block** icon  on the top toolbar

The **Block** form appears allowing the user to define the **Block** size either to the default, **Max/Min Limits** or by manually entering the minimum and maximum X, Y and Z values (or a combination of both).

The **Block** size is automatically set to the model dimensions by clicking the **Calculate** button with **Max/Min Limits** set. Individual values can be edited or locked (greyed out) as required in addition to being calculated to include an offset by entering a suitable value in the box marked **Expansion**.

Click on the **Calculate** button.

Click on **Accept.**



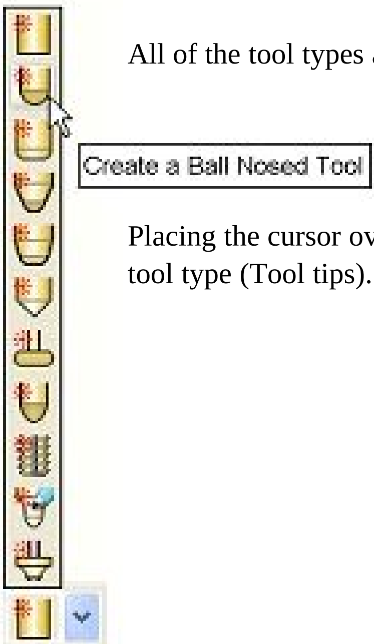
The **Block** is displayed shaded by default. This can be shown as solid, partially transparent, or as a blue wireframe, as controlled by the **Opacity** slider on the **Block** form.

The **Block** can be turned off graphically by selecting the block icon  from the **View toolbar** on the right

7.Cutting Tool definition

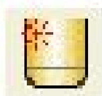
The **Tool definition** forms are accessed from the icons in the **Tool toolbar** located to the bottom left corner of the graphics area.

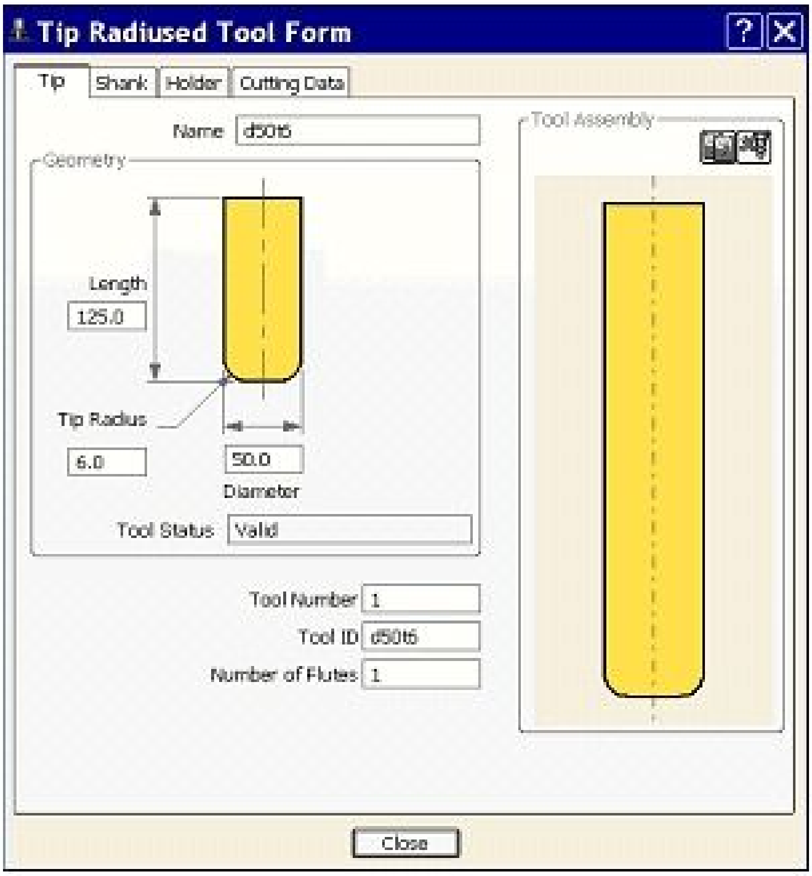
Click on the **down arrow** to display all of the **Create Tool** icons.



All of the tool types appear as icons.

Placing the cursor over an icon will open a small box containing a description of the tool type (Tool tips).

Select the **Create a Tip Radiused tool** icon. 



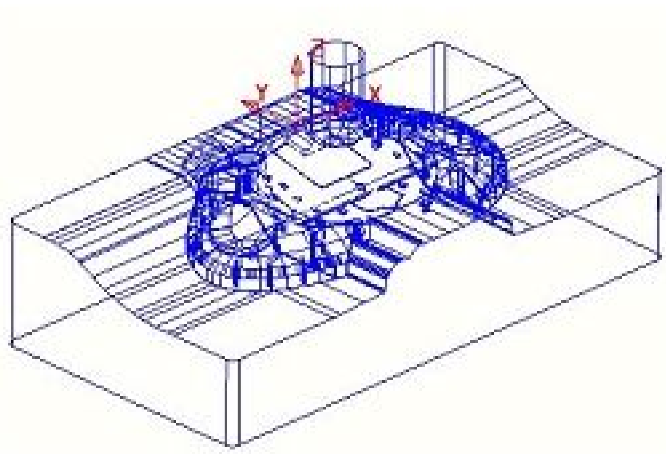
The **Tip Radius Tool** form opens into which the required tool data is entered. When the diameter is entered the tool length automatically defaults to five times the diameter, and this can be changed if not desirable.

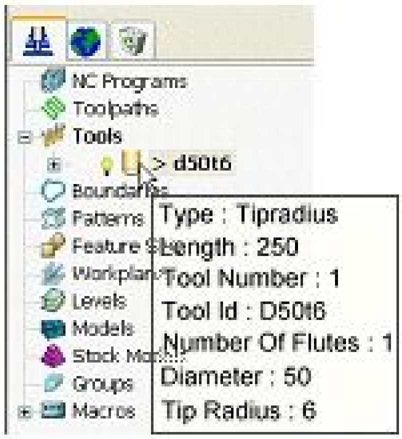
It is a good idea to input a suitable **Name** for the tool. This tool for example with a diameter of 50 a length of 100 and a tip radius of 6 could be called **D50T6**.

The specified **Tool Number** can be output in the NC program. If the machine has a tool changer it will represent the carousel number.

Shank and **Holder** definition will be covered later in the course during the chapter on **Collision Checking**.

Enter a **Diameter** of **50** a **Tip Radius** of **6** and modify the **Length** to **100**.
Enter **D50T6** in the box marked **Name**.
Click on **Close.**



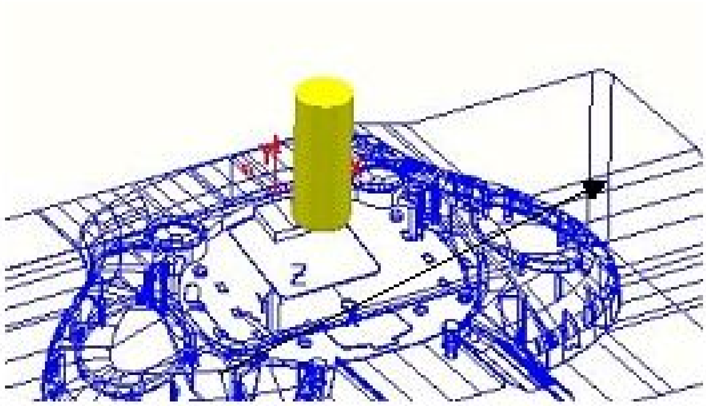


The tool is now displayed on the screen and also appears as an entity in the **Tools** section of the **explorer**. Note: If the mouse is positioned over the tool in the **explorer** a **Tooltips** window will open displaying details of the tool definition.

Any future changes to the tool are initiated through the explorer. To **undraw** the tool click the **yellow light bulb** icon. To **deactivate** the tool **right click** anywhere on the tool data in the explorer and click the name **Activate** (the tick will disappear). Selecting the **Parameters** option will open **the tool definition** form.

The tool can be seen either in wireframe or shaded in the graphics window.

Right click over the tool in the **Explorer** and select **Shaded.**



8. Feedrate and Spindle speed settings

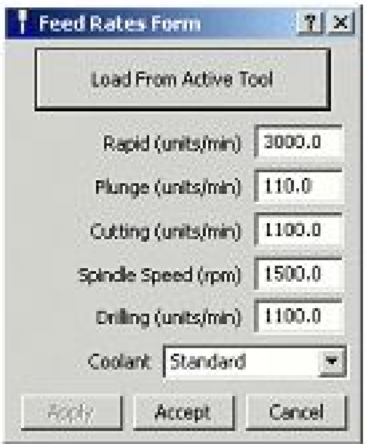
The speed and feed rate settings need to be defined.

Click on the **Feed rates** icon  on the **Main toolbar**.

The **Feed Rates** form opens enabling the user to enter appropriate feeds and speed for the machining strategy.

Cutting Feed Rate and **Spindle Speed** can be stored within the **Tool definition** and input by clicking the box **Load From Active Tool**.

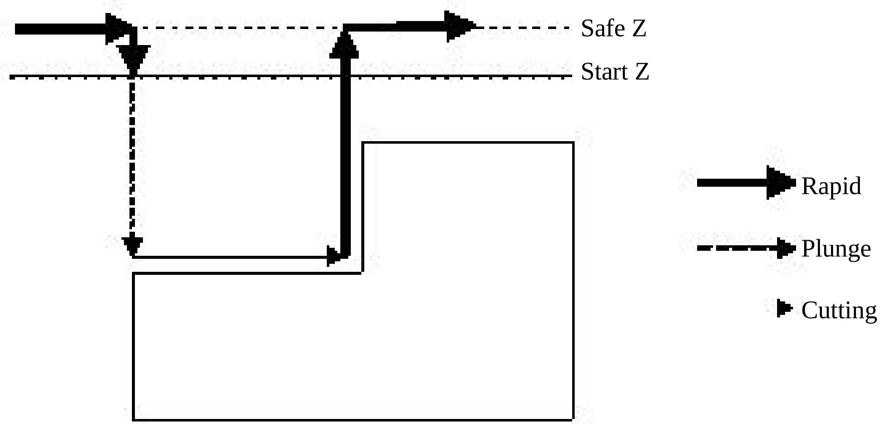
The value for the **Plunge** feed rate is controlleddefined as a proportion of the **Cutting** feed rate controlled by changing the value in **Tools - Options - Toolpath -Tools - Plunge Factor - 0.1** (default). If the **Autoload** box is ticked the **Feed Rates** form will update automatically when a **Tool** is activated.



Leave the settings as default and click **Accept**.

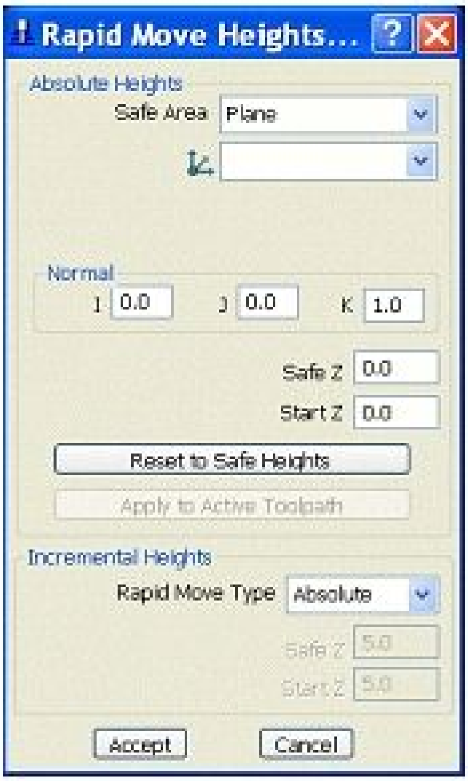
9. Rapid move heights

Safe heights in Z must be set at which the tool can travel at rapid feed rates without contacting the component or work holding devices.



Safe Z is the height to which the tool will retract for rapid moves across the work. **Start Z** is the height to which the tool will descend, at rapid feed rate prior to applying the plunge feed rate.

Click on the **Rapid Move Heights** icon.
In the resultant form select the **Reset to Safe Height** button.
Click on **Accept.**



The same principle can also be defined incrementally to enable safe rapid moves, locally within the component. In the section of the form labelled **Incremental Heights**, in addition to **Absolute**, two additional options are available **Plunge** and **Skim**.

Plunge will enable the rapid feed rate to continue to a specified distance above the full plunge depth before the plunge feed rate cuts in.

Skim will operate in a similar way to **Plunge** but with the addition of applying rapid horizontal moves, at a specified distance above the highest point across the component to the next plunge position.

10. Tool Start and End Point.

This is a safe position for the tool to move to before and after each tool change or machining operation. Depending on the type of machine tool it may be the actual tool change position.

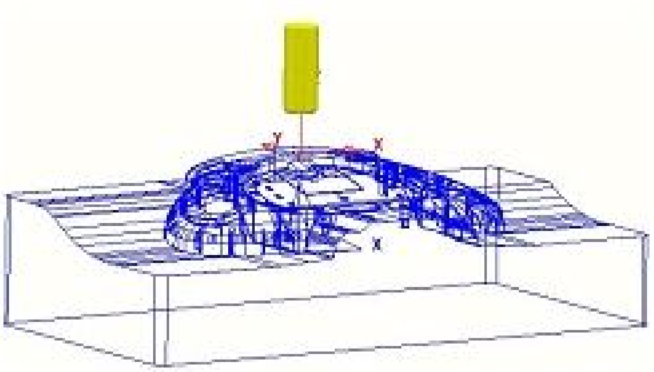
Click on the Tool **Start and End Point** icon.

The tool **Start and End Point** form appears in which the user full control in defining a safe position for the tool to start from following a tool change or previous toolpath.

By default the tool position is set automatically at the **Block Centre Safe**. A more specific position can be defined by selecting a different option in the **Method** area on the form (**First Point**, **First Point Safe**, and **Absolute**).

The tool **End Point** is also defined from this form independently from the **Start Point**.

Select **Start Point** and in the ***Method*** area set ***Use*** to **Absolute**.
Set the co-ordinates as **X0, Y0** and **Z 50**.
Click on **Accept**.



The tool is now located at the new **Start Point** above the active **workplane**.

Setting up is now complete and **PowerMILL** is ready to start generating toolpaths.

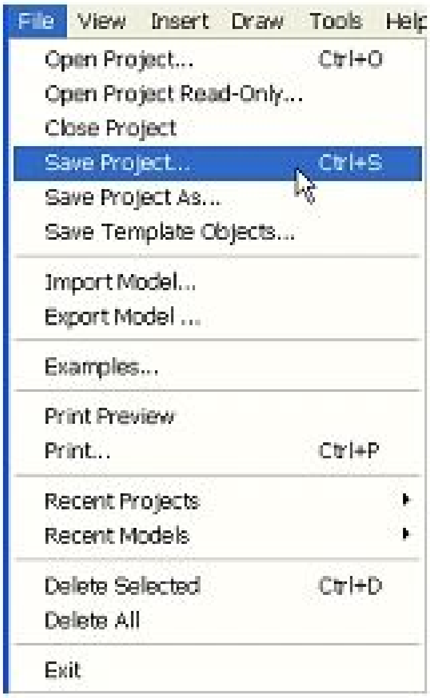
11. Projects

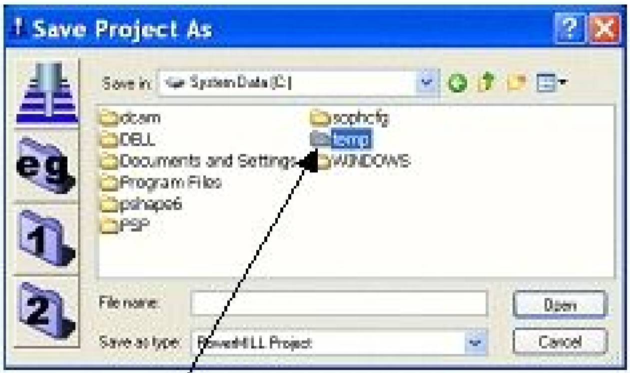
A **Project** is a folder used for maintaining a permanent copy of the data created in the **PowerMILL explorer**. This could include data such as **Toolpaths**, **Tools**, **Workplanes**, and other entities related to the machining strategies. The project will also retain the activation status of each entity when saved. The **Model** is also stored in the **Project**.

The commands for the project are accessed through the **File** pull down menu and it is good policy to initially **Save Project** as early as possible during programming and then at suitable, regular intervals.

When initially saving a **Project** the user has to decide on a suitable directory location on the computer.

Select **File -> Save Project**.





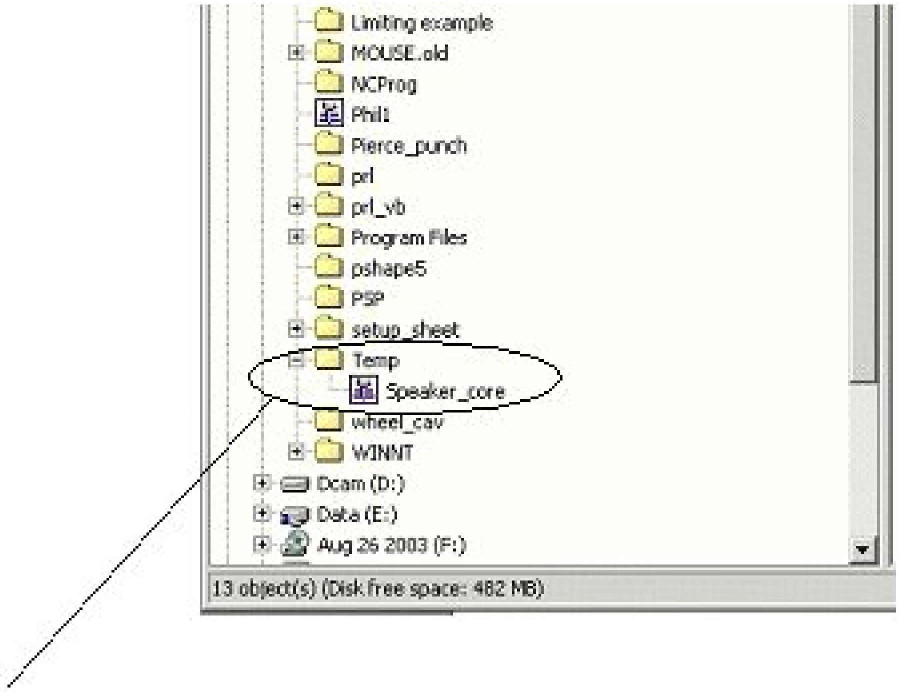
In the form locate the folder **C:\temp** (or if it does not exist with **C** open select the create **New Folder** tab and rename it as **temp**).

Once **C:\temp** is accessed, enter the name, **Speaker_Core** in ***File name*** and then select **Save**.

To update the stored **Project** apply **Save Project** as and when required. To create an additional copy, use **Save Project As** (This will require a new **File name**).

The use of **Projects** to store component data, machining data, and associated settings is essential to enable the user to return and modify an existing machining process. In many cases an existing **Project** can be used as the start of a new one, using the existing settings as the basis for the creation of new or recycled toolpaths on a different component.

Project data is stored in a folder that is identifiable by a **PowerMILL** icon.

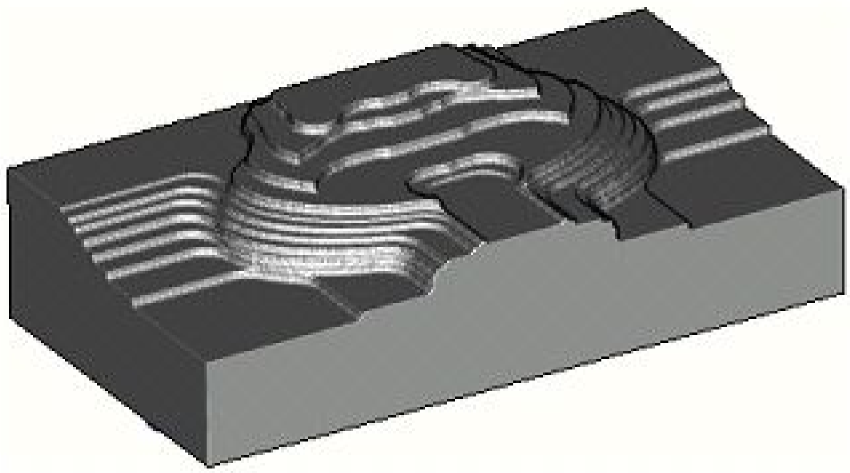


How the **Project** appears in Windows **Explorer**.

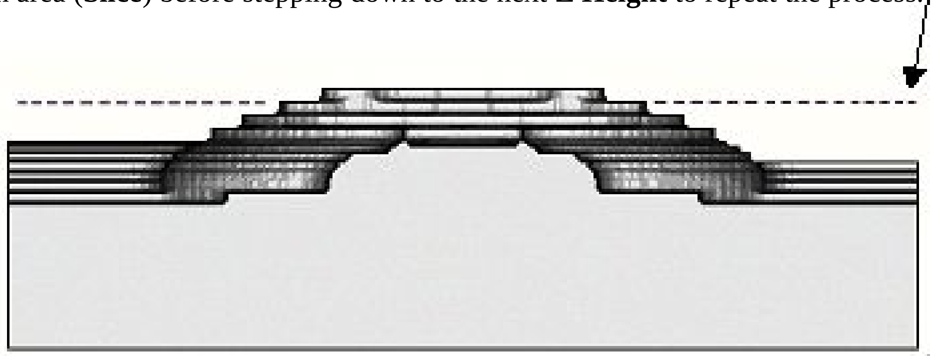
3. 3D Area Clearance

Introduction

The main strategies for roughing a **3D** component **Model** are called **3D Area Clearance**. These provide a choice of 2D material removal methods, which progressively machine the area (**Slice**), up to the component contour, down a sequence of user-defined **Z Heights**. There is also a similar group of strategies, **2.5D Area Clearance** for use, exclusively with **PowerMILL 2.5D Feature** machining (covered on the final day of the course).



Sometimes known as Waterline Roughing the cutter steps down to a specified **Z Height** and fully clears an area (**Slice**) before stepping down to the next **Z Height** to repeat the process.



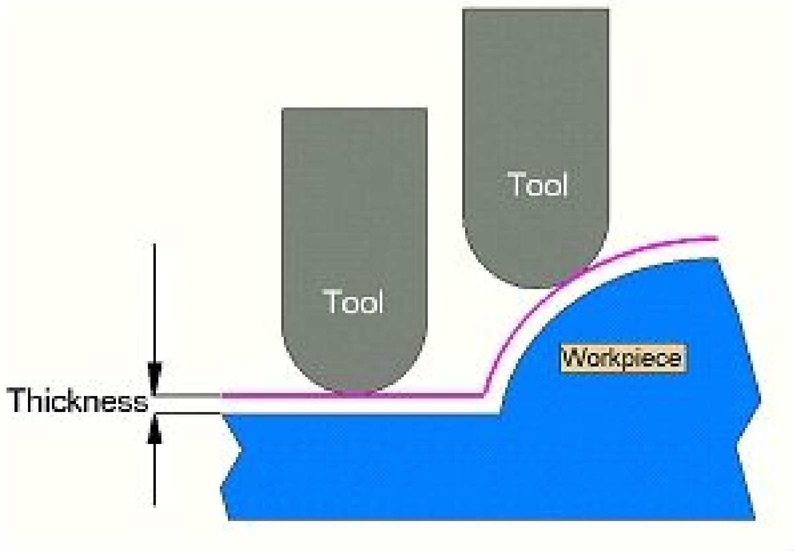
For some components a secondary **Area Clearance** strategy is applied using the **Rest Machining** options in conjunction with a smaller roughing tool. This will locally remove pockets of excess material inaccessible to the original Tool **Reference Toolpath** or **Stock Model**. This will reduce the degree of tool overload and provide a more consistent material removal rate for any subsequent **Finishing** operations.

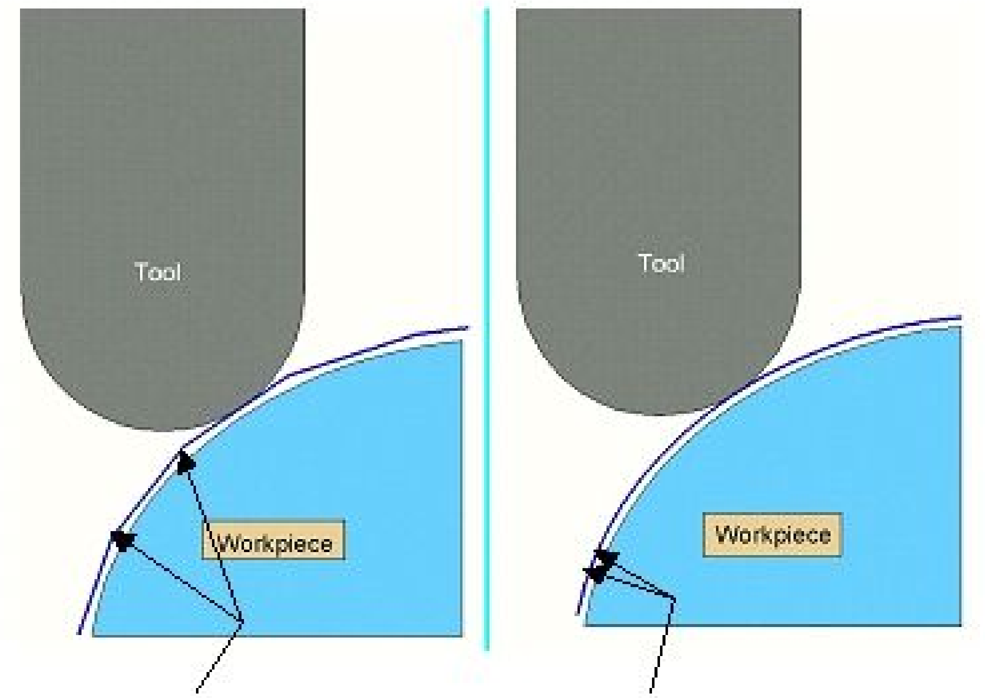
If the original material is in the form of a casting or fabrication then it may not be necessary to apply any **Area Clearance** machining but to go directly for a semi-**Finishing** strategy.

Toolpaths

Suitable values are required to control the accuracy and amount of excess material to be left on a component by a toolpath. The parameters used for this purpose are preset and are called **Thickness** and **Tolerance**.

Thickness is the amount of extra material specified to remain on the work-piece after machining. This can be applied generally (as shown), or independently as separate **Radial** and **Axial** values within the machining options.
It is also possible to assign additional **Thickness** values to groups of **Surfaces** on the actual model.





Coarse Tolerance **Fine Tolerance**

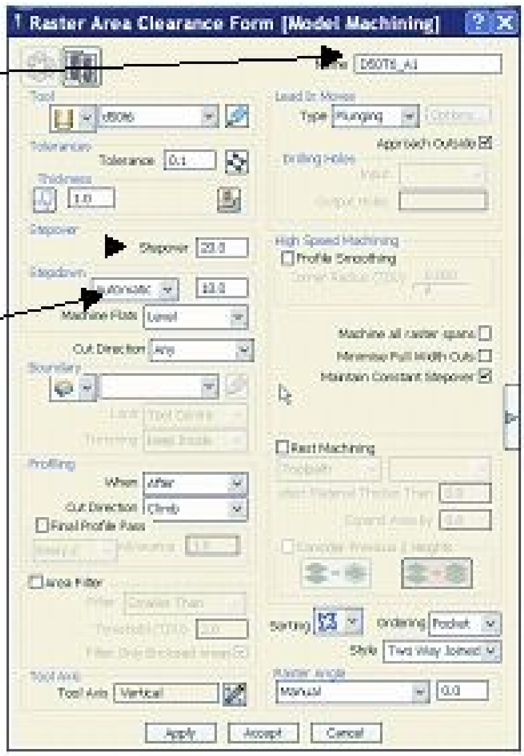
Tolerance controls the accuracy to which the cutter path follows the shape of the work-piece. For roughing a Coarse tolerance can be used but for finishing a Fine tolerance must be used.

<u>**Note**</u> where the **Thickness** value is greater than **0** it should always be more than the **tolerance** value

Raster Area Clear example

Activate the **D50t6** tool in the **explorer**.
From the **Main toolbar** select the **Toolpath Strategies** icon.
Select **3D Area Clearance Tab**.

Select the option **Raster AreaClear Model** to open the following form.



Enter the Name **D50T6_A1**.

Set **Stepover** to **20**.

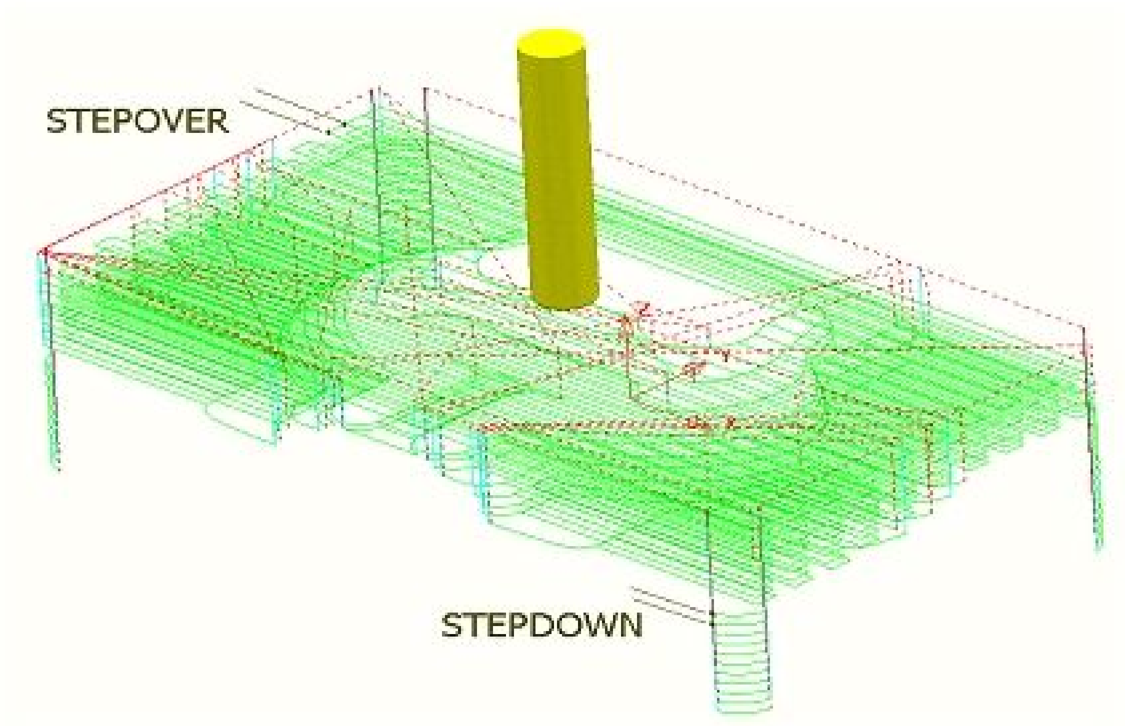
Set **Stepdown** to **10**.

Keeping all other values as default, **Apply** the form.
After the processing is complete **Cancel** the form.

As soon as the **Raster AreaClear Model** form is opened an unprocessed **Toolpath** appears in the **Explorer** (the default name has been changed to **D50T6_A1**).

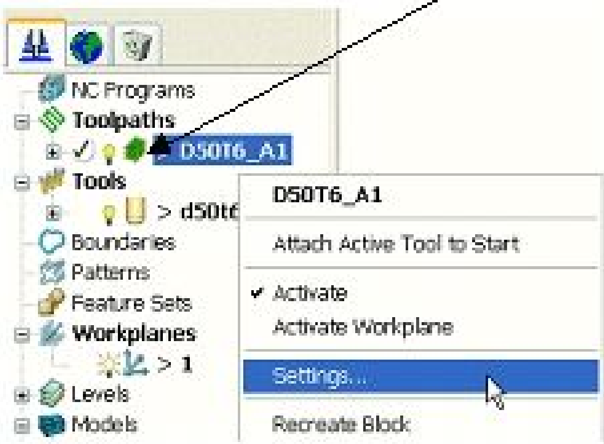
The Toolpath icon can now be double-clicked to **Activate** and de-activate the toolpath.

The + symbol can be clicked to open a full record of the data used to create the toolpath.



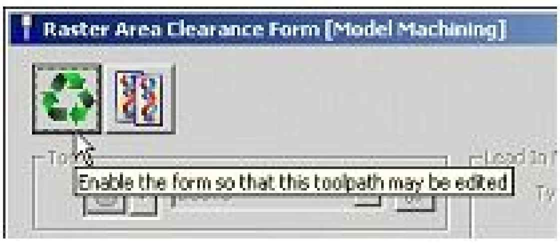
The dotted red lines represent **Rapid** moves and the light blue lines are the **Plunge** feed moves. The **Plunge** moves at present are longer than necessary resulting in time wasted by the tool feeding down in free space where most of the move could be **Rapid**.
The existing toolpath will now be **Recycled** with new settings to improve the efficiency of Rapid movements across the component.

Right mouse click over the **Toolpath** icon in the **explorer** to open the local pull down menu.

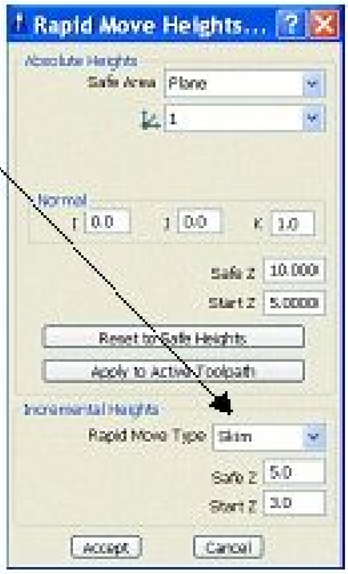


Note the toolpath can also be activated or deactivated from the **Activate** switch in the pulldown menu.

Select **Settings** to reopen the **Raster AreaClear Model** form.
Select the 'Recycle toolpath' icon (shown arrowed below).

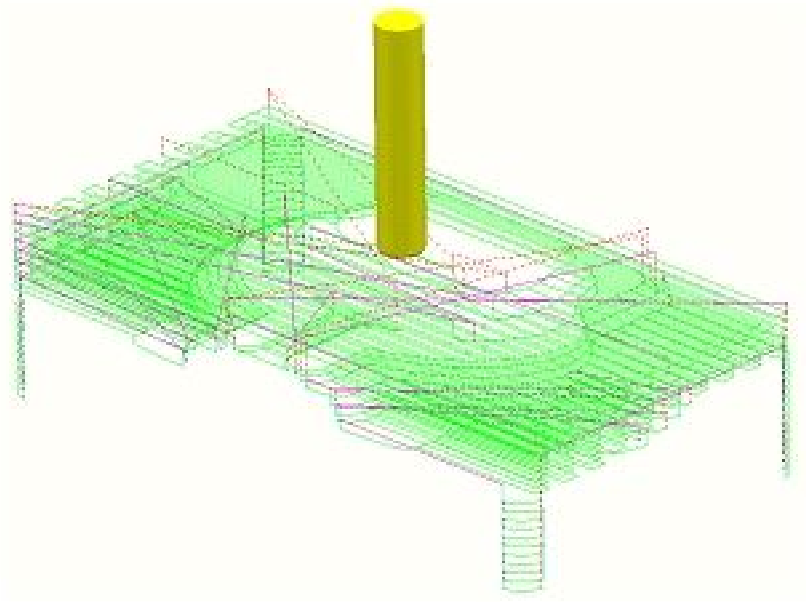


Select the **Rapid Moves Height** icon  from the main toolbar
Select the **Skim** option and change the **Safe Z** to **5** and **Start Z** to **3** as shown below.



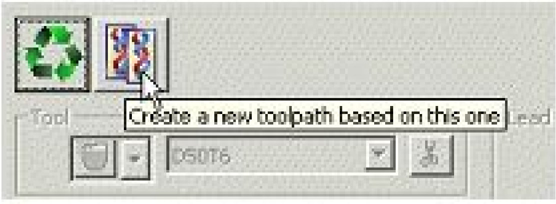
Accept the above form and then click **Apply** in the **Raster AreaClear Model** form and once processing is complete click **Cancel** to close the form.

The tool will now plunge locally (pale blue move) from the defined ***Incremental*** **- Start Z** above each slice and rapid across the roughed areas by the ***Incremental*** **- Safe Z** (purple move).



The toolpath has been recycled with the **Skim** option set in the **Rapid Move Heights** form and now uses Incremental **Safe Z** and **Start Z** values (Note; In practice more than one change can be implemented when Recycling or Copying a toolpath).
This time a **Copy** of the above toolpath will be created using the **Area Filter** options to remove the single pass tooltracks which are stepping down each of the four corners.

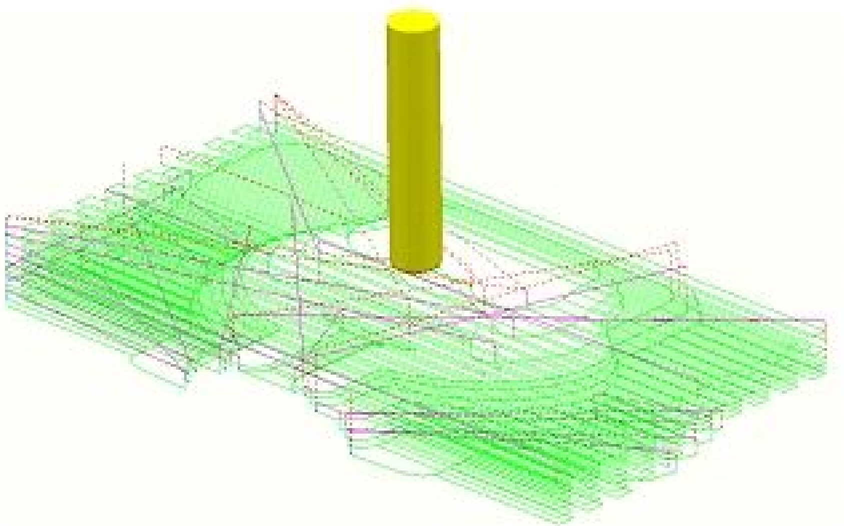
Select **Settings** to reopen the **Raster AreaClear Model** form.
Select the 'Copy toolpath' icon (shown arrowed below).



In the form locate the **Area Filter** section (lower left corner) and input the settings as shown below (Note **Filter Only Enclosed Areas** is unticked).

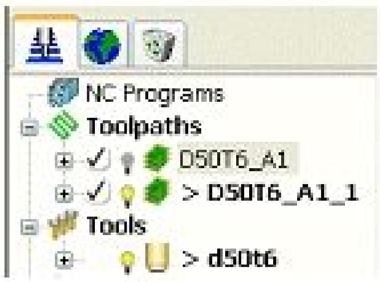
All tooltracks spanning a distance less than the Tool Diameter (TDU) will be filtered out and not appear in the final Toolpath.
The effect of unticking **Filter Only Enclosed Areas** means that the areas of toolpath outside the **Block** will be included in the filtering process.

Apply the **Raster AreaClear Model** form and once processing is complete click **Cancel** to close the form.



The default **Raster Area Clear** strategy steps across the model with parallel tooltracks. The remaining edges around the model, contour are then removed by a single profiling pass.

A new toolpath called **D50T6_A1_1** is created with the corner machining filtered out.



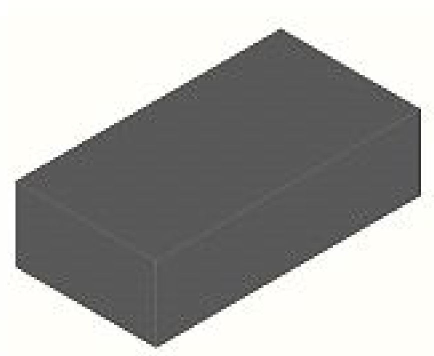
Simulating the toolpath

ViewMill provides a 3D graphical simulation that allows toolpaths to be checked before they are machined. **ViewMill** has it's own separate toolbar that can be accessed from the **ViewMill** button on the **Main** toolbar. There are currently 2 different versions of **ViewMILL** the intention being that the original will be phased out once the new version is fully developed. For the time being there is no real benefit from using the new version for 3-Axis applications. Below is an example using the original **Viewmill** and in the following example on p39 the new version is applied.

Select the (Original Version) **ViewMill** icon located towards the right in the **Main Toolbar** at the top of the screen

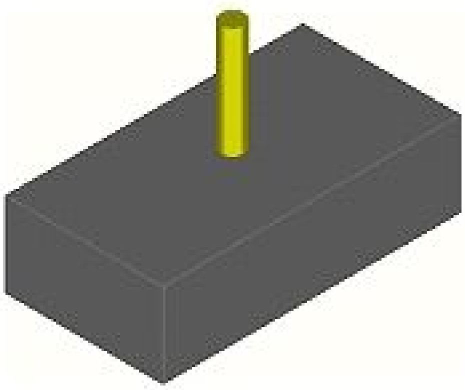
The simulation toolbar appears.

Select **ISO 1** view of the model and scale it to be as large as possible.
Select the **ViewMill toggle** icon.



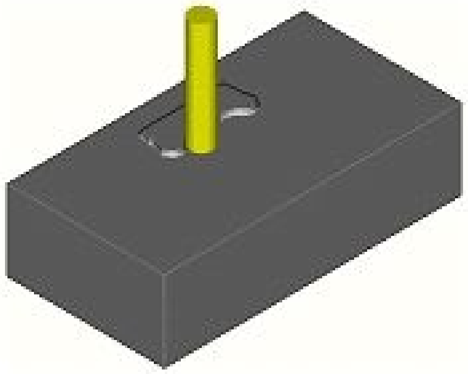
This toggles the display to the Viewmill environment and generates a shaded block.

Select the **Shaded Tool** icon.

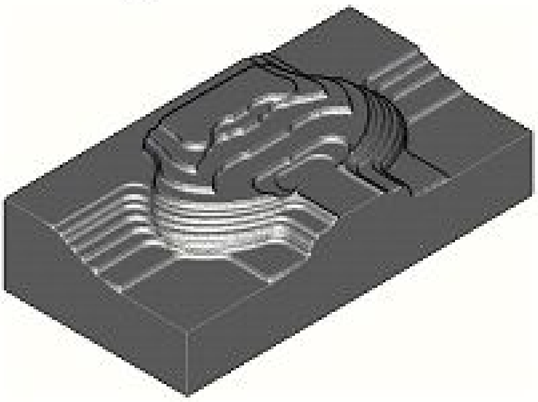


With the tool shaded, it is clear where the tool datum position is. The speed of the simulation is reduced if the tool is shaded.

Select the **Start/Restart** icon.

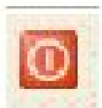


The toolpath is simulated displaying the material removal from the defined block of material.
Click the **Shaded Tool** icon again to switch it off which in turn will speed up the simulation.



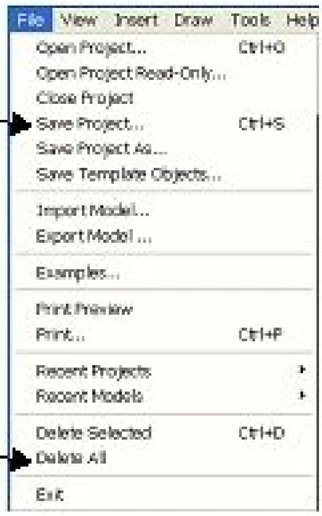
The complete machined block.

Select the **ViewMill toggle** icon to return to the **PowerMILL** session.



The above **ViewMILL** session will not be cancelled but temporarily put into the background so that any subsequent toolpaths can be simulated as a continuation from the above.

From the **Main** toolbar select **File**.
Save Project to update the stored data.



The **Speaker_Core** Project has now been updated to include
The 3D Area Clearance toolpaths.

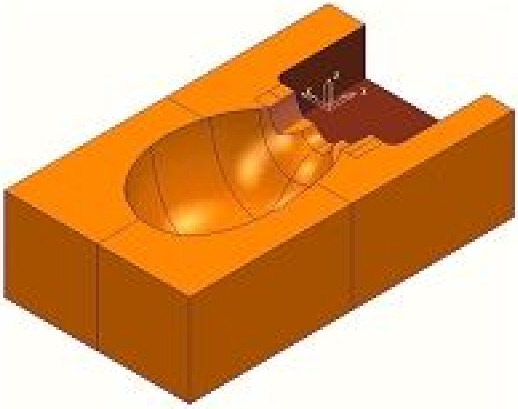
From the **Main** toolbar select **File**
Delete All.

From the **Main** toolbar select **Tools - Reset Forms**.

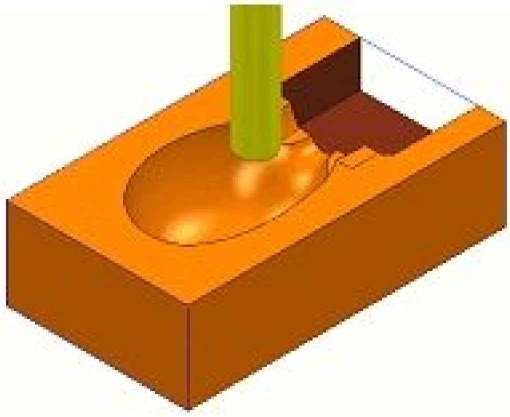
This restores all default **PowerMILL** settings in the forms without having to exit and re-enter.

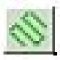
Offset Area Clear example

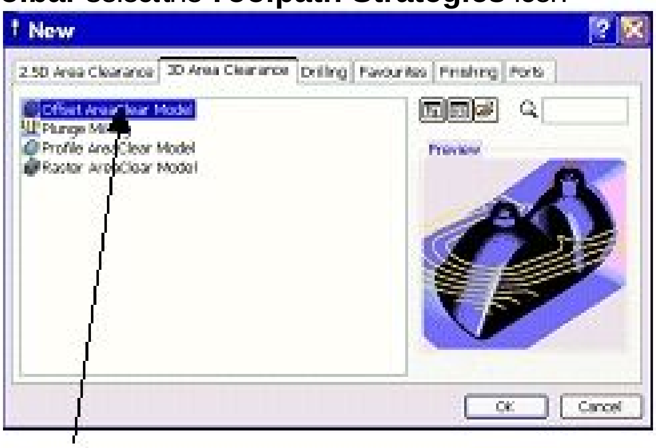
Import the model **WingMirrorDie.dgk** from **PowerMILL_Data\models**.



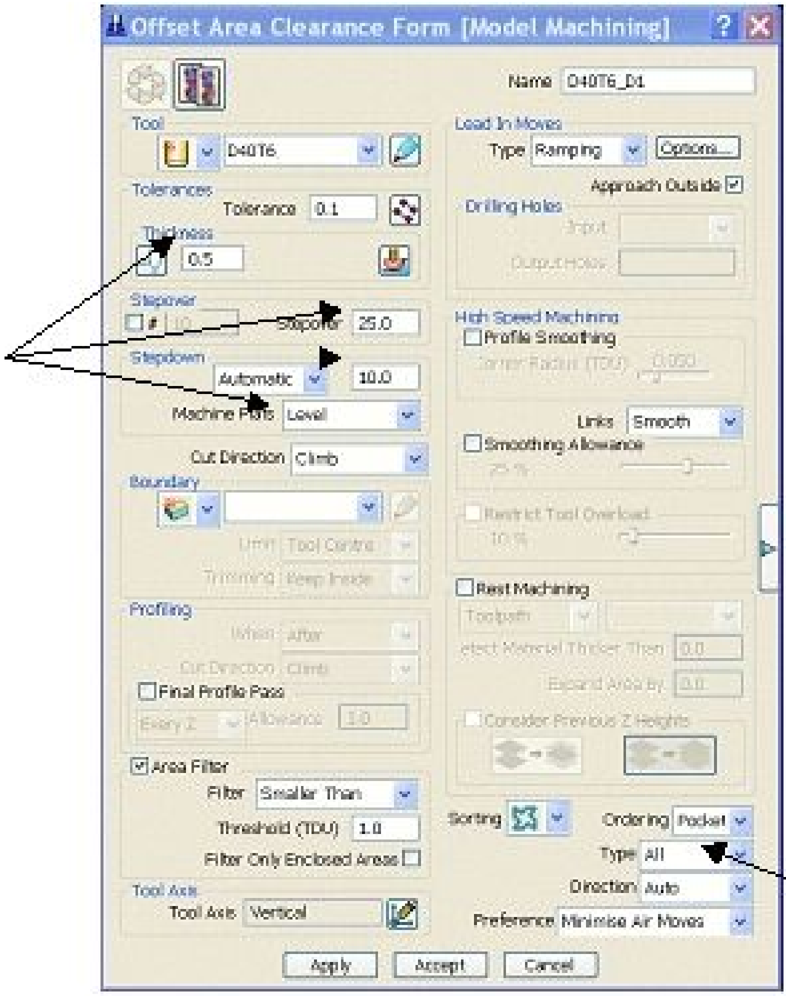
Create a **Tip Radiused** tool of **Dia 40 tiprad 6** and ***Name*** **d40t6**.
From the **Main toolbar** open the **Block** form and **Calculate** a material **Block** to the full model dimensions.
Reset the **Rapid Move Heights** and set the **Incremental** moves to **Skim**.
In the **Start Point** form set the ***Start Point*** to **Block Centre Safe** and the ***End Point*** to **Last Point Safe**.



From the **Main toolbar** select the **Toolpath Strategies** icon 



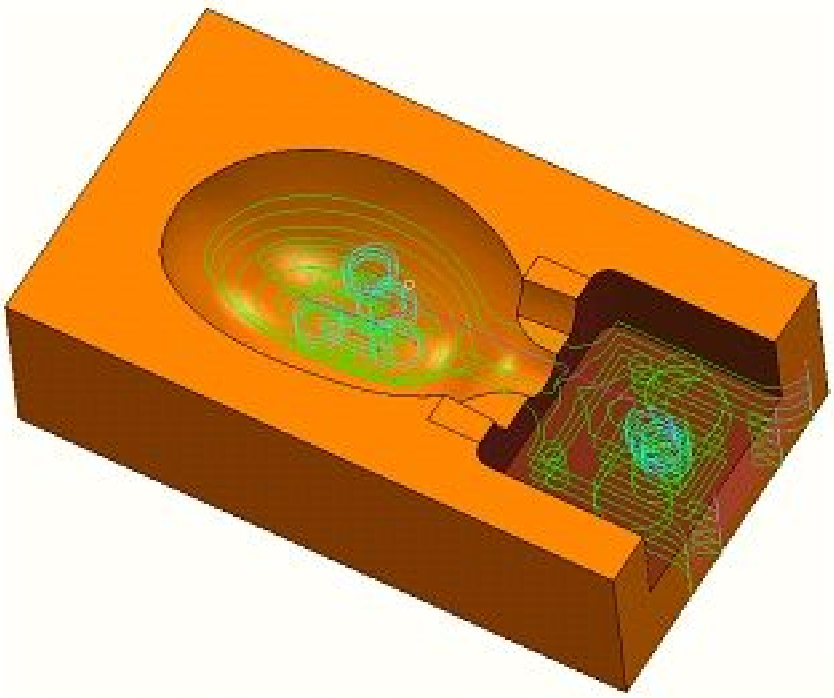
Select the option **Offset AreaClear Model** to open the following form.



Enter ***Name*** **D40T6_D1**.
Select **Ramping**.
Select **Ramping** ***Options***
Entering;
Max. Zig Angle **4**,
Follow **Circle**, and
Circle Diameter **0.6**
As shown below.

Input *Type* - **All** (default).

Input or modify the data as shown in the sections arrowed above and click **Apply** to create the **Offset Area Clear** toolpath shown below.

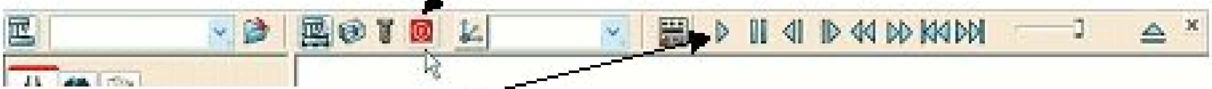


The **Offset Area Clear** strategy using *Type* **All** follows both the contours of both the **Model** and **Block** then gradually Offsets into the remaining material at each **Z Height**.

RMBon the toolpath and select **Attach Active Tool to Start** to bring up the Simulation toolbar

The **toolpath simulation** toolbar will open.

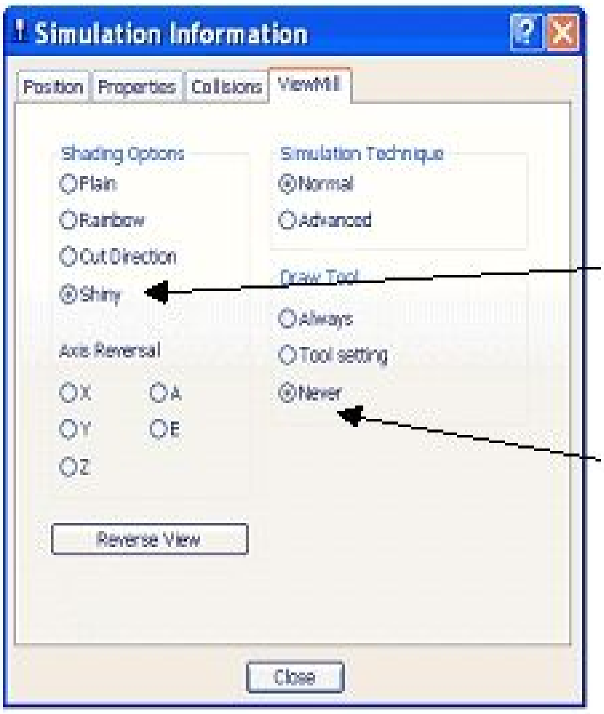
Select the (New Version) **ViewMill toggle** icon.



Press the **Play** button to start the simulation.

The simulation of the toolpath will start with tool displayed.

- Click the **Display** button to bring up **Simulation info** form



Select the **Shiny** option

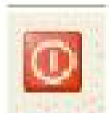
Select **Never** to undraw the tool (This will speed up the simulation)

Close the form

Press the **Play** button again on the Simulation toolbar

The finished result indicates that the current tool geometry is not suitable to fully access some features (arrowed) on the model.
As a result a further **Area Clearance** strategy is required using a smaller tool to continue locally into the remaining areas.
This technique is known as **Rest Machining**.

Select the **ViewMill toggle** icon to return to the **PowerMILL**.



Rest Machining

Rest Machining is localised removal of the remaining material inaccessible to previous toolpaths. The 3D Area Clearance forms contain options to apply Rest Machining either based directly on a previously defined Reference Toolpath or a Stock Model.
The following examples illustrate Rest Machining firstly using a Reference Toolpath and secondly by using the Stock Model.
The Stock Model represents the un-machined stock at any point in the machining process. It is applied by first creating a Stock Model entity by applying a Block, followed by various toolpaths. After each update, the stock model adjusts itself to show the un-machined material remaining on the block.

Rest Machining using a Reference Toolpath

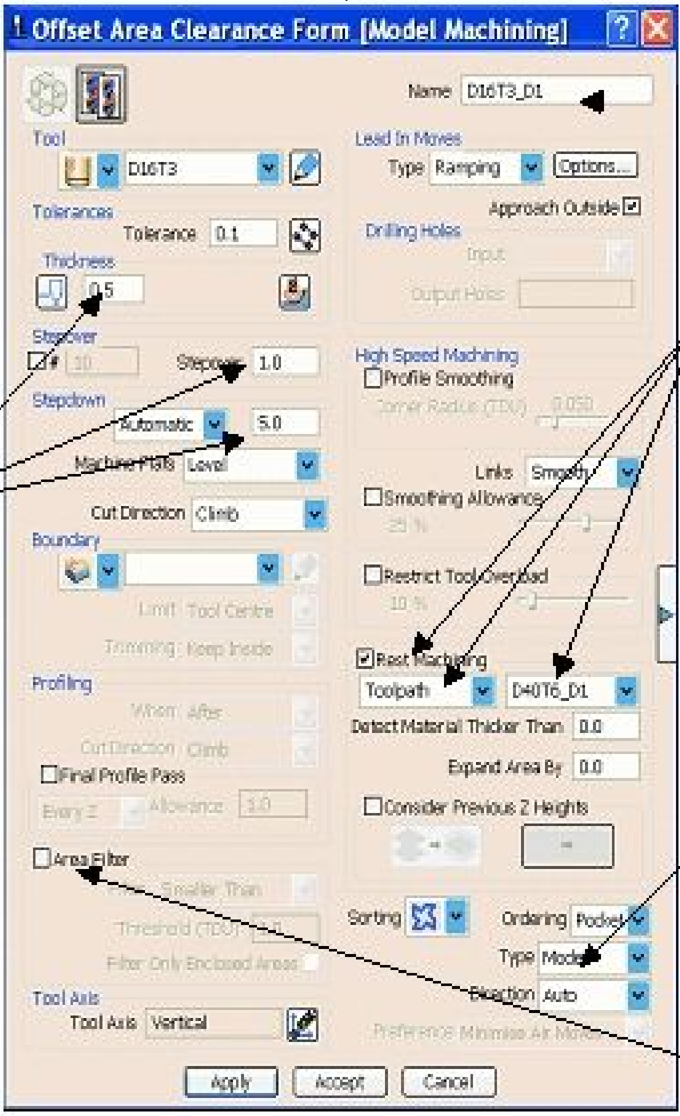
Create a **Tip Radiused** tool of **Dia 16**, **tiprad 3** and *name* **d16t3**.
RMB over the Toolpath icon in the **explorer**.

Select **Settings** to reopen the **Offset AreaClear Model** form.

Select the '**Copy toolpath**' icon.

Note; all associated items originally used to create the toolpath will be activated.

Activate the new tool, **d16t3**.



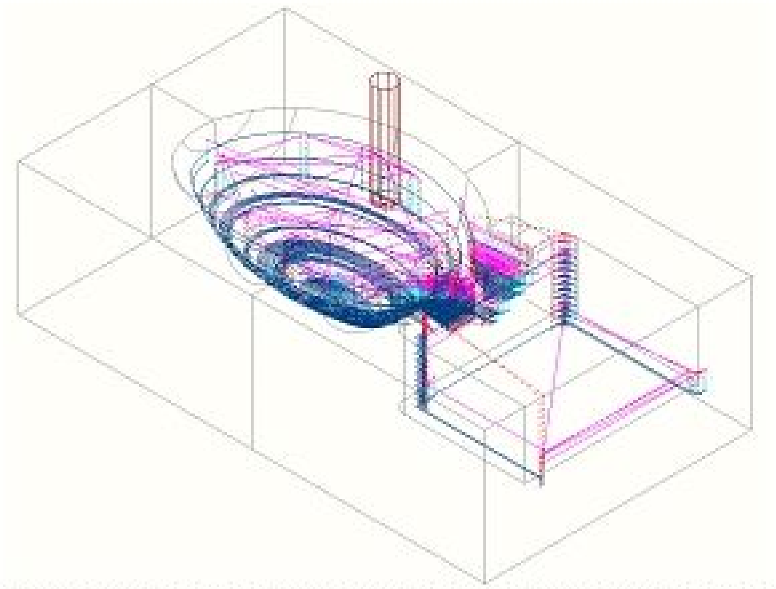
Enter a new name **D16T3_D1**.

The options arrowed control the **Rest Machining** limits by comparison with the previously defined toolpath **D40T6_D1**.

Set *Type* to **Model** (This generates offset tooltracks, based solely on the **Model** contours).

Untick **Area Filter**.

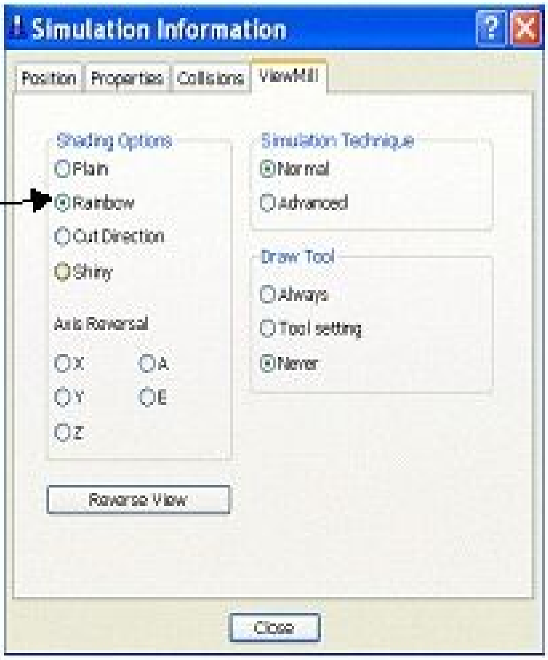
Input or modify the data as shown arrowed above and click **Apply** to create the new **Offset Area Clear** toolpath shown on the following page. **Cancel** the form.



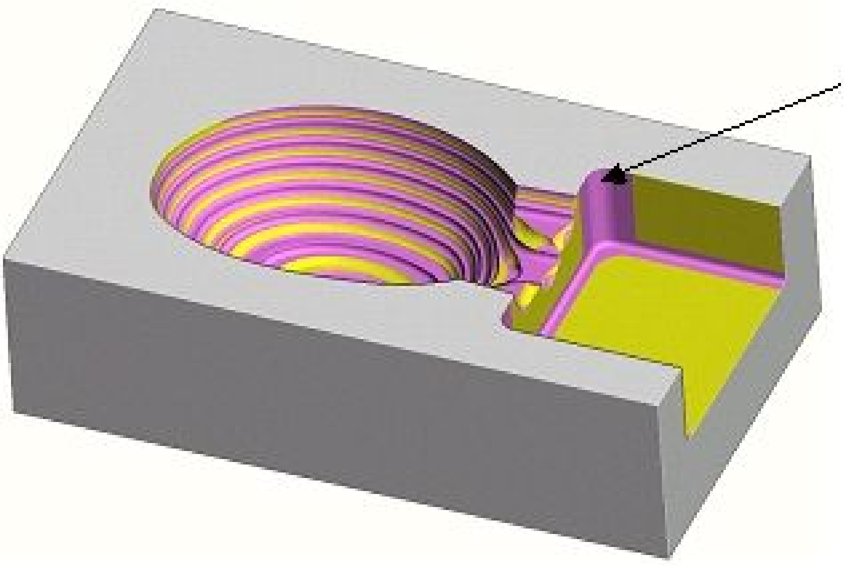
RMB over the Toolpath icon in the **explorer**.
Select **Attach Active Tool to Start**.
Select the **ViewMill toggle** icon access the existing **ViewMILL** session.

- Click the **Switch on Display** button to bring up **Simulation info** form

Select the **Rainbow** option.



Close form and press play on the **ViewMill** simulation toolbar.



The **ViewMILL** simulation shows this next toolpath shaded in a different colour where it has machined in areas the previous toolpath did not cover

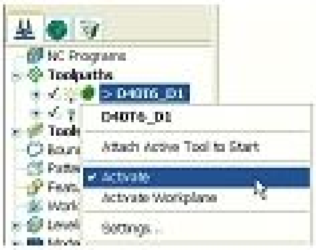
The **Reference Toolpath** finished to rough out material closer to the component form.
This will reduce the risk of excessive wear or damage to tools used for the subsequent finishing operations.

Select the **ViewMill toggle** icon to return to **PowerMILL**

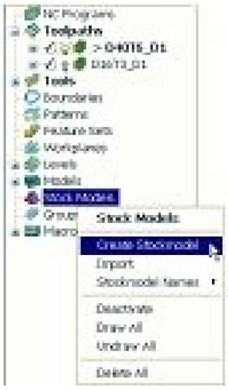
Rest Machining using a Stock Model

An alternative method of **Rest Machining** is the use of a **Stock Model** instead of a **Reference Toolpath**. One of the benefits of using a **Stock Model** is that the remaining material can be visualised and registered along with the associated toolpaths.
The following example will demonstrate the creation of a **Stock Model** up to the end of the toolpath **D40T6_D1** at which point it will be displayed as the **Rest Material**. The **Rest Machining** toolpath **D16T3_D1** will then be used as the basis for creating a new toolpath but this time using the **Stock Model**.

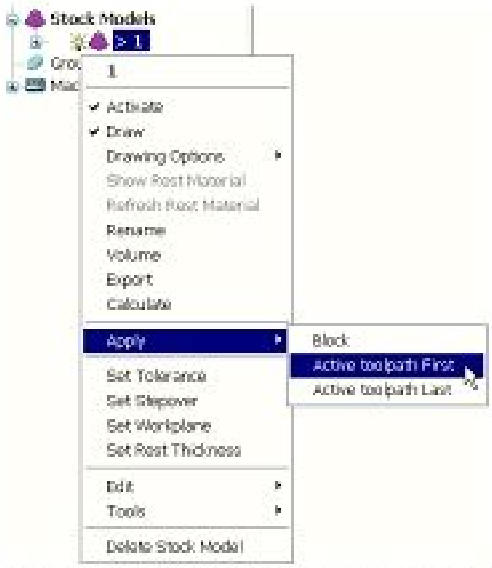
In the **explorer**, **Activate** the toolpath **D40T6_D1**.



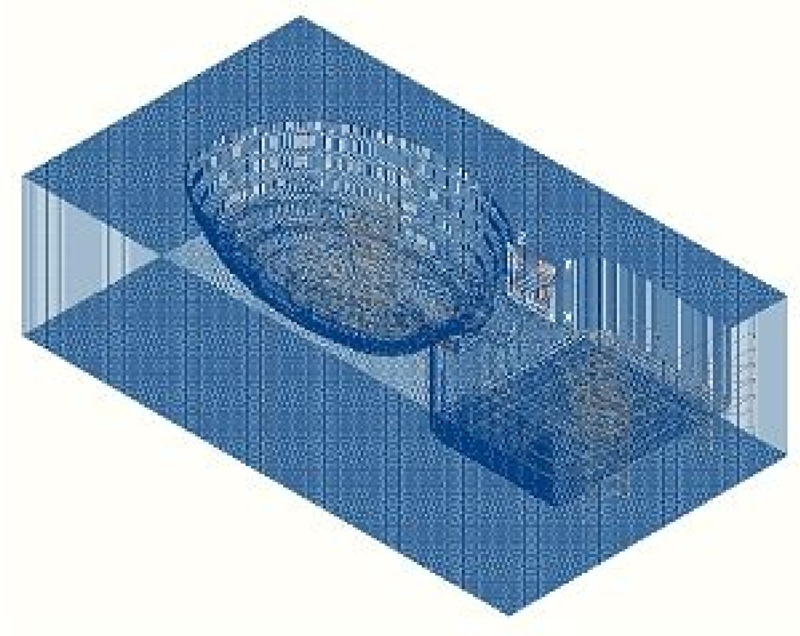
In the **PowerMILL explorer** right mouse click **Stock Models** to open the local menu and select **Create**.



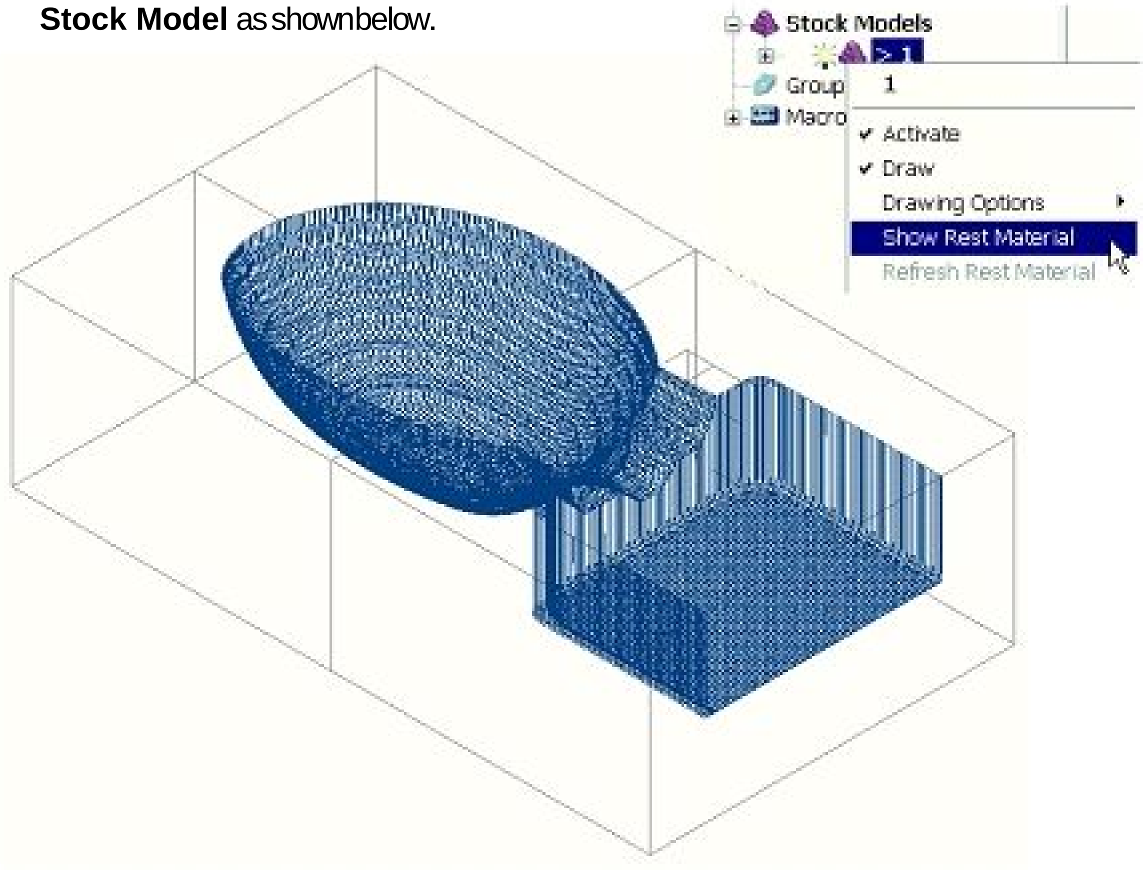
In the **PowerMILL explorer** right mouse click the newly created **Stock Model** icon and from the local menu select **Apply - Active Toolpath First**.



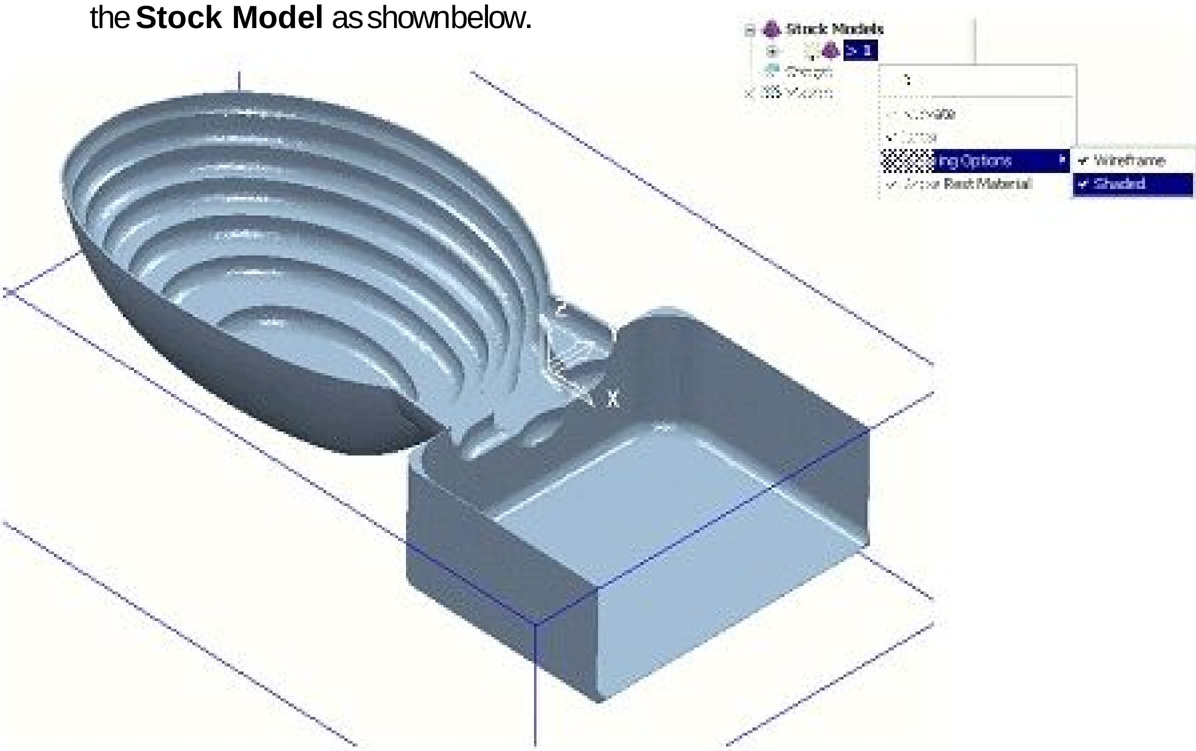
From the same menu select **Calculate** to create the **Stock Model** as shown below.



From the same menu again select **Show Rest Material** to display the **Stock Model** as shown below.



From the same menu again select **Drawing Options - Shaded** to display the **Stock Model** as shown below.

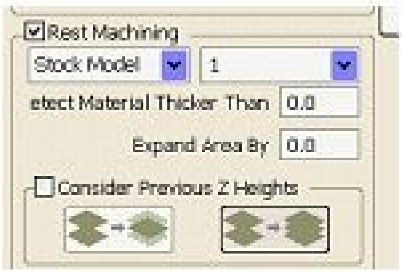


In the **explorer Activate** the toolpath **D16T3_D1**.

Right click the toolpath icon and from the local menu select **Settings** to reopen the **Offset AreaClear Model** form.

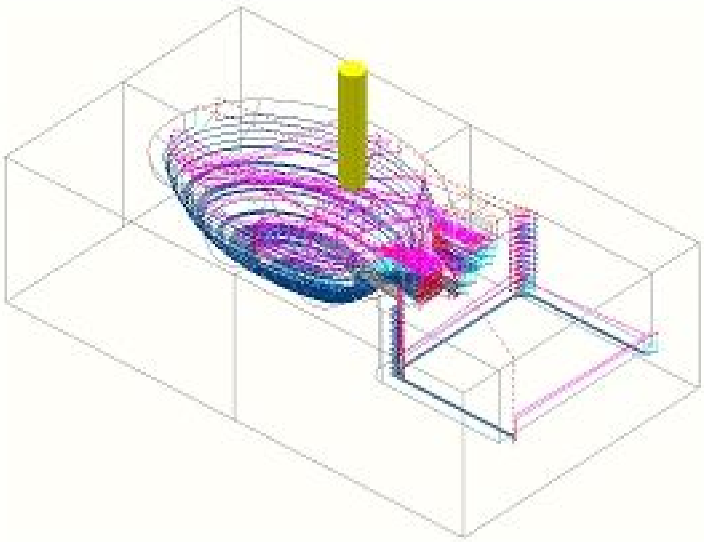
Select the '**Copy toolpath**' icon (shown arrowed below).

In the form locate the **Rest Machining** section (lower right corner) and change the settings to use **Stock Model - 1** as shown below.



Click **Apply** to create an alternative (more efficient) **Rest Machining** toolpath named **D16T3_D1_1**.

Cancel the **Offset Area Clearance** Form to reveal the following toolpath.

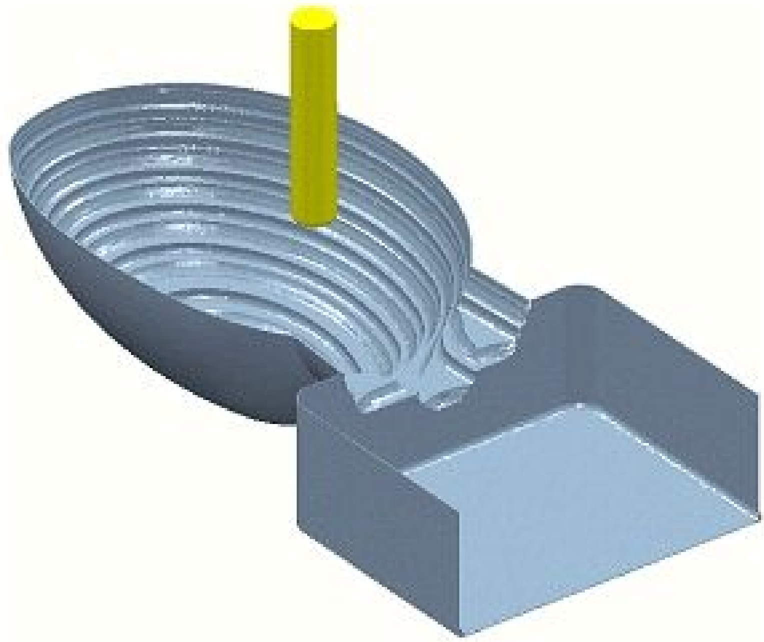


Although the **Stock Model** has been used to define the area to be machined the new toolpath is not automatically added to the **Stock Model**. This is carried out as a secondary operation if required by the user.

Right click over the active toolpath, **D16T3_D1_1** in the **explorer** and select **Add to - Stock Model**.

It is also possible to Right Click on the named, **Stock Model** in the **explorer** and select **Apply - Active toolpath Last**. At this stage the **Stock Model** will disappear from the screen.

Right click over the named, **Stock Model** in the **explorer** and select **Calculate** to create the updated **Stock Model** as shown below.



From the **Main** toolbar select **File - Save Project** and save the **Project** as **C:\temp\Wing_Mirror_Die**. (See Chapter 2 for full details of method)

In the **explorer** Right click over **Toolpaths** and select **Delete All** from the local menu.

Use the same method to **Delete All** - **Models** and **Stock Models** (Do not Apply to the **Tools** section in the **explorer**).

From the **Main** pulldown menus select **Tools - Reset Forms**.

This restores all default settings in the forms without having to exit and re-enter.

Note: The above items **deleted** from the **explorer** can at this stage be recovered as they have been transferred to the ***PowerMILL*** **Recycle Bin**. If **File - Delete All** is applied from the main pulldown menus all entities by-pass the **Recycle Bin** and as a result cannot be recovered.

Left mouse click on the **Entity Recycler** pane to open the **Recycle Bin**.

Right mouse click over an entity in the **Recycle Bin** to view the local menu options to **Recover** or permanently **Delete** an item.

Recover a **Toolpath** from the **Recycle Bin** and check that it has been reinstated in the **Toolpaths** section of the **PowerMILL explorer**.

Return to the **PowerMILL explorer** and if necessary, apply **Delete All** separately again on **Toolpaths**, **Models** and **Stock Models**. (Do not Apply to the **Tools** section in the **explorer**).

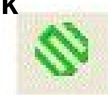
Offset Area Clearance - Type Model

This variant of the **Offset Area Clearance** is designed for High Speed machining. It provides a very consistent material removal rate at the expense of an increased number of rapid moves across the component (This is accepted practice in High Speed applications). This strategy if applied correctly will dramatically help to minimise wear to both the tooling and machine. The strategy is based on the profile around the component at each **Z Height** being continually offset up to the material **Block**.

Import the model **Handle.dgk** located in **PowerMILL_Data\models**.

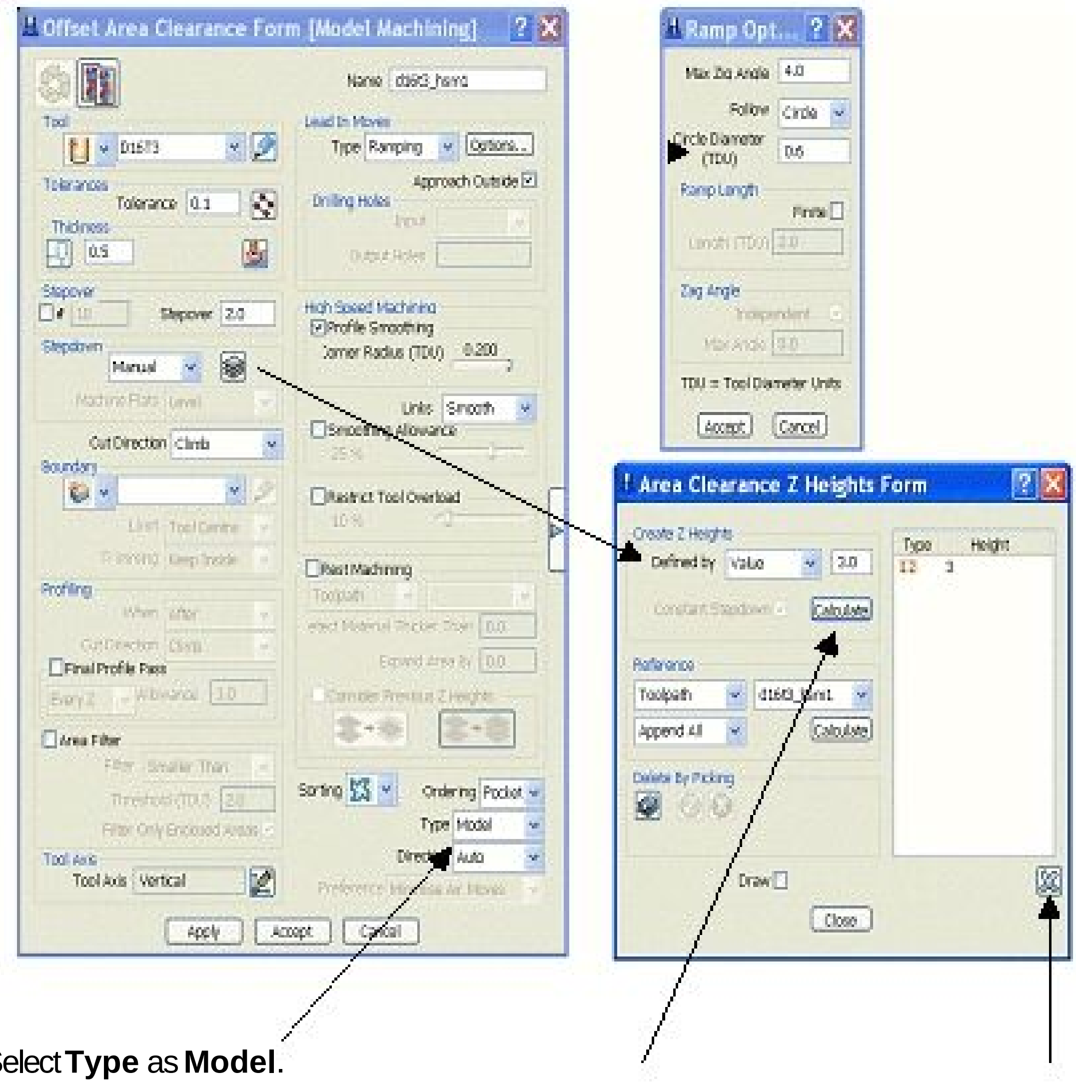
In the **explorer - Activate** the **Dia 16 tiprad 3** tool **d16t3**.

From the **Main toolbar** open the **Block** form and **Calculate** a material **Block** to the full model dimensions. **Lock** the **Z max** and **Z min** values, enter an **Expansion** value of **10** and **Calculate** again.
In **Rapid move heights**, click **Reset to Safe Heights**, and in ***Incremental Heights*** set the ***Rapid Move Type*** to **Skim**.
In the **Start Point** form set both the **Start Point** and **End Point** to **Block Centre Safe**.



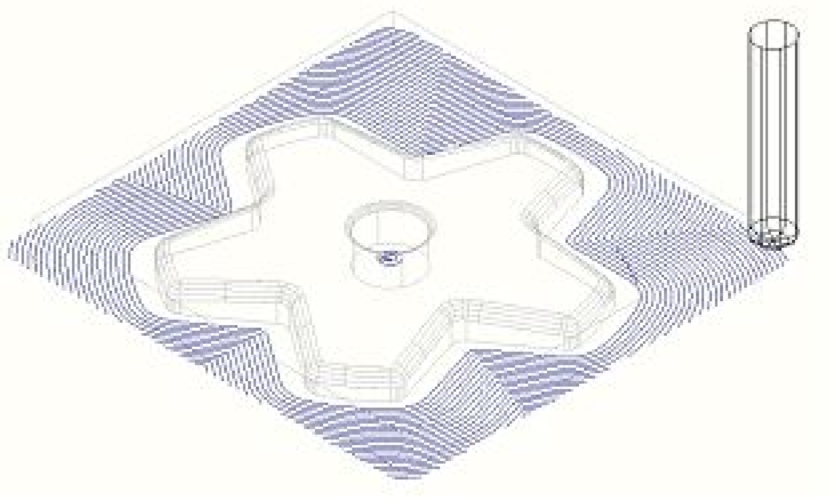
From the **Main toolbar** select the **Toolpath Strategies** icon.
From **3D Area Clearance** select the option **Offset AreaClear Model** to open the following form.
Enter data in the forms exactly as shown below.
The **Stepdown** is to be defined manually as a single level from the **Area Clearance Z Heights Form** 

(If **Z Heights** already exist, **Delete** them by clicking the **red cross** located to the lower right of the form).



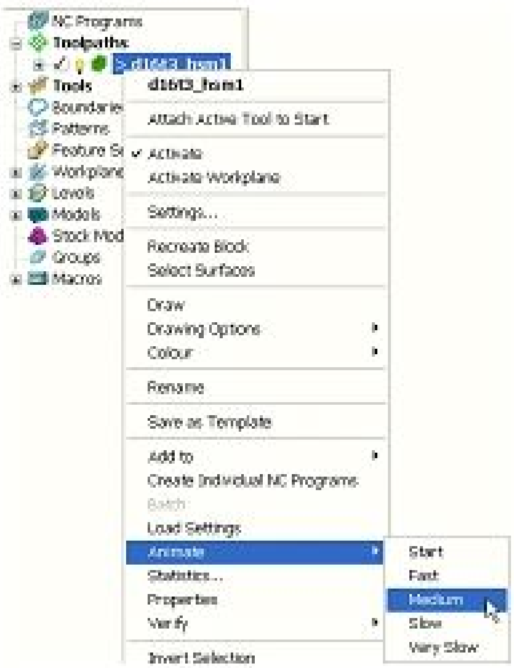
Select **Type** as **Model**.
Input Defined by ***Value*** **3.0** and select **Calculate** (After selecting **Delete All** existing **Z Heights**).

Check the data as shown previously and click **Apply** then **Cancel.**



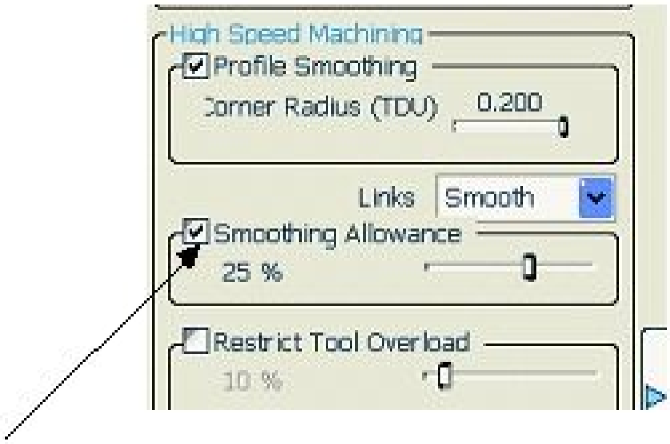
Note: Shown with leads and links undrawn.

Select an **Iso 1** view as shown and right click over the toolpath icon in the **PowerMILL explorer** to open the local menu.
Select **Animate - Medium** and observe the animation of the toolpath (select the **Esc** key to stop the animation).

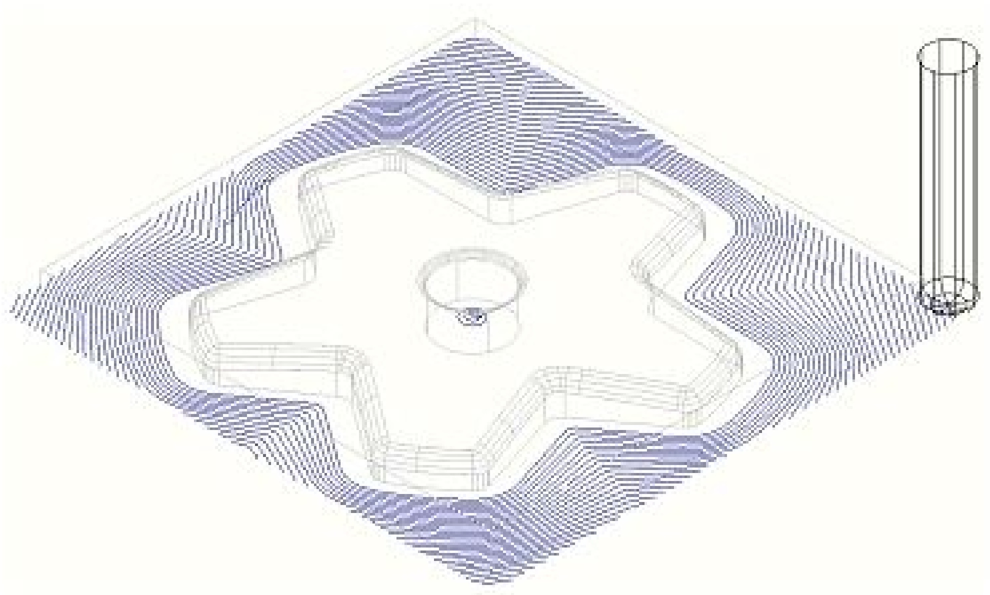


The tool will continuously climb mill, starting each tool track a distance from the material block to allow for acceleration to optimum Feed Rate. Each pass will maintain a very consistent material removal rate. The only negative side to the strategy is the occasional sudden changes of direction along the tool tracks. This can be improved using advanced settings to progressively straighten tooltracks as they offset further away from the Component form. A specified deviation from the nominal stepover will be required to allow this.

Right mouse click over the Toolpath icon in the **explorer** to open the local pull down menu.
Select **Settings** to reopen the **Offset AreaClear Model** form.
Select the '**Copy toolpath**' icon (shown arrowed below).



With the **Offset Area Clearance** form open again, **tick** the **Smoothing Allowance** option leaving the slider value as default (**25%**).
Click **Apply** to create a new toolpath **d16t3_hsm1_1** with the improved 25% smoothed strategy.



Compare the 2 **3D Offset Area Clearance** (Type - Model) toolpaths noting the progressive straightening of tool tracks on the second strategy (25% smoothing applied).

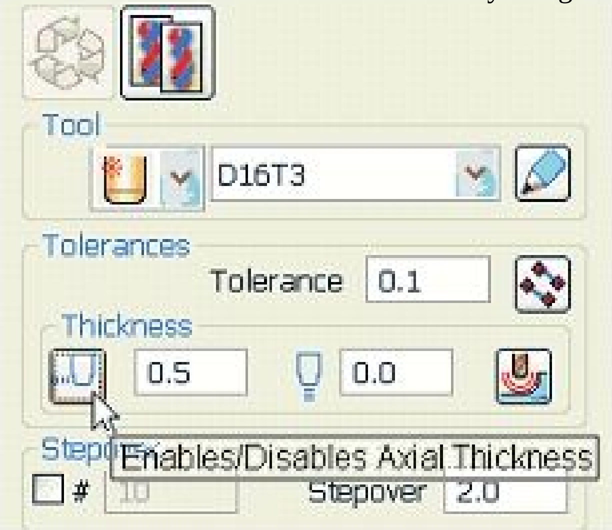
From the **Main** toolbar select **File - Delete All** to delete all data from the current project

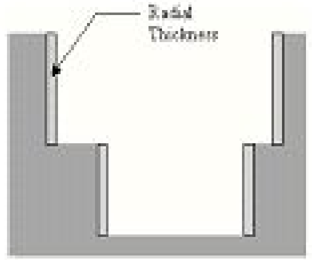
From the **Main** toolbar select **Tools Reset Forms**

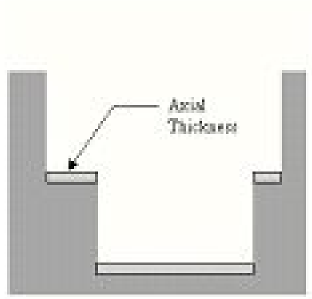
This restores all default PowerMILL settings in the forms without having to exit and re-enter.

General information on Area Clearance Machining

The following is reference information for the many different options contained in the Area Clearance form. This can also be found by using **Help**.

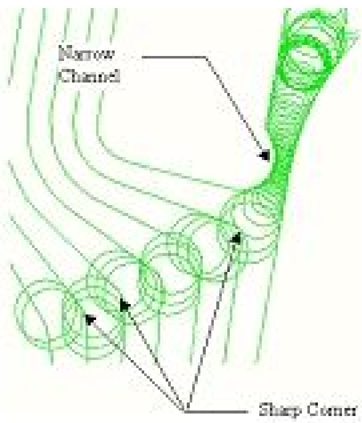






Clicking the **Thickness** button on the **Area Clearance** forms opens the **Axial Thickness** box allowing the user to set separate values for **Radial** and **Axial** thickness. This faclity is also available on the finishing forms.

Restrict Tool Overload



With **Restrict Tool Overload** ticked as the tool reaches an specified overload situation **PowerMILL** automatically inputs a **trochoidal** path to eliminate full width cuts. This will occur in corners, narrow channels, slots, etc and the degree of movement is controlled with the slider as a percentage value in the form.

Z Heights

If **Stepdown** is set to Manual on The Area Clearance form, there are five ways of generating Z Heights; **Number**, **Stepdown**, **Value, Intermediate** and **Flat**.

Number - divides the block equally into the defined number of Z Heights, the lowest of which will be at the bottom of the block.

Stepdown - creates a Z Height at the base of the block and then steps up a defined Height in Z. The setting **Maintain Constant Stepdown** causes the distance between all levels to remain constant and will modify the stepdown to create evenly spaced levels as near to the specified value as possible.

Value - creates a single Z Height at the defined value. You can specify as many Z Heights as is required, but when using Value you must do so one at a time.

Flat - Identifies flat areas of the model and creates a Z height (+ thickness) at these values.

Intermediate - adds the specified number of Z Heights between existing Z Heights.

Appending Z Heights
Z Heights can be also be used from saved Area Clearance Toolpaths. When a toolpath is activated the Append button becomes active.

Profiling

A profile can be performed at each level to remove steps that will be left by the cutter **Before**, **During**, or **After** a **Raster - Area Clearance** strategy. Additional profile passes can be applied when machining either on either **Every Z**, or the **Last Z** level with **Offset**, **Profile** or **Raster** strategies. Note: **Offset** and **Profile** strategies inherently follow the component profile.

When

This determines when the profile pass takes place during machining. There are 4 options

None - No profiling pass is performed
Before - PowerMILL will perform the profiling first and then the raster path.
During - As the raster path is generated it will find profile paths as it goes.
After - PowerMILL performs the profile pass last.

Cut Direction

This determines the direction of the tool. Choosing a single direction will more than likely lead to more lifts generated.

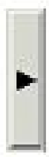
Any - this allows the cutter to travel in both directions allowing it to climb mill and conventional mill.
Climb - this will force the cutter to only travel in one direction so that it is always climb milling.
Conventional - this will force the cutter to only travel in one direction so that it is always conventionally milling.

Final Profiling Pass

This option is held in the profiling area of the main area clearance toolbar and allows the user to make an additional, final profiling pass to further reduce tool wear.

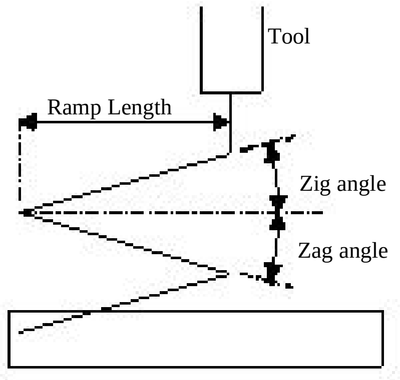
Allow tool outside block

The **Allow tool outside block** tick box is located in the **Expert Area Clearance** form, which is opened by selecting the tab midway down the right hand side of the main form. This enables the first pass of an **Offset** or **Raster** pass to be performed to the specified **Stepover**, rather than the full radius of the tool.



Ramping

This provides a way to lead down onto a tooltrack where it is impossible to approach from outside the **Block** at the full machining depth (eg within a pocket).



The **Zig** angle is the angle of descent along the machining direction as the tool ramps into the material. There are 3 different types of ramp move following the geometry of the **Toolpath**, a **Circle**, or a **Line**. If the **length** of the **Zig** angle is limited to a **finite** distance a ramp move in the opposite direction, **Zag** angle can be applied.

The **Ramp Length** is defined as '**Tool Diameter Units**' (**TDU**). For example, with a 10mm diameter tool, A **Ramp Length** of 2 TDU's would equal 20mm. Normally the Ramp Length should be greater than the tool diameter to allow swarf to clear from beneath the tool.

Zag Angle
If a **finite** ramp length has been specified, then PowerMILL will insert **Zag** moves. The default setting for **Zag** angle has the **Independent** flag *set* - which means the angle, is defined manually. The default angle is **0** degrees. When *unset*, it will be the same value as the **Zig** angle.

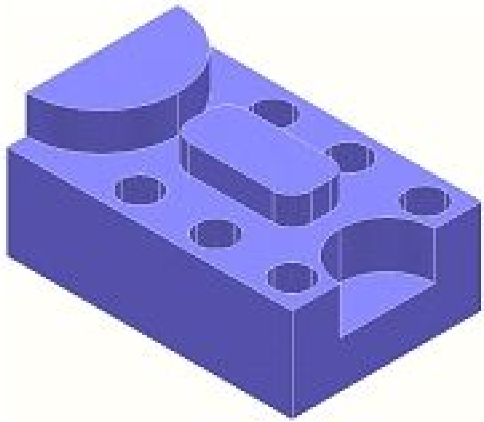
If **Approach Outside** is *set*, and where it is practical for it to operate without gouging it will take priority over **Ramping**.

If the defined geometry for a **Ramp** move is such that it would cause a gouge then it will be replaced by a **Plunge** move.

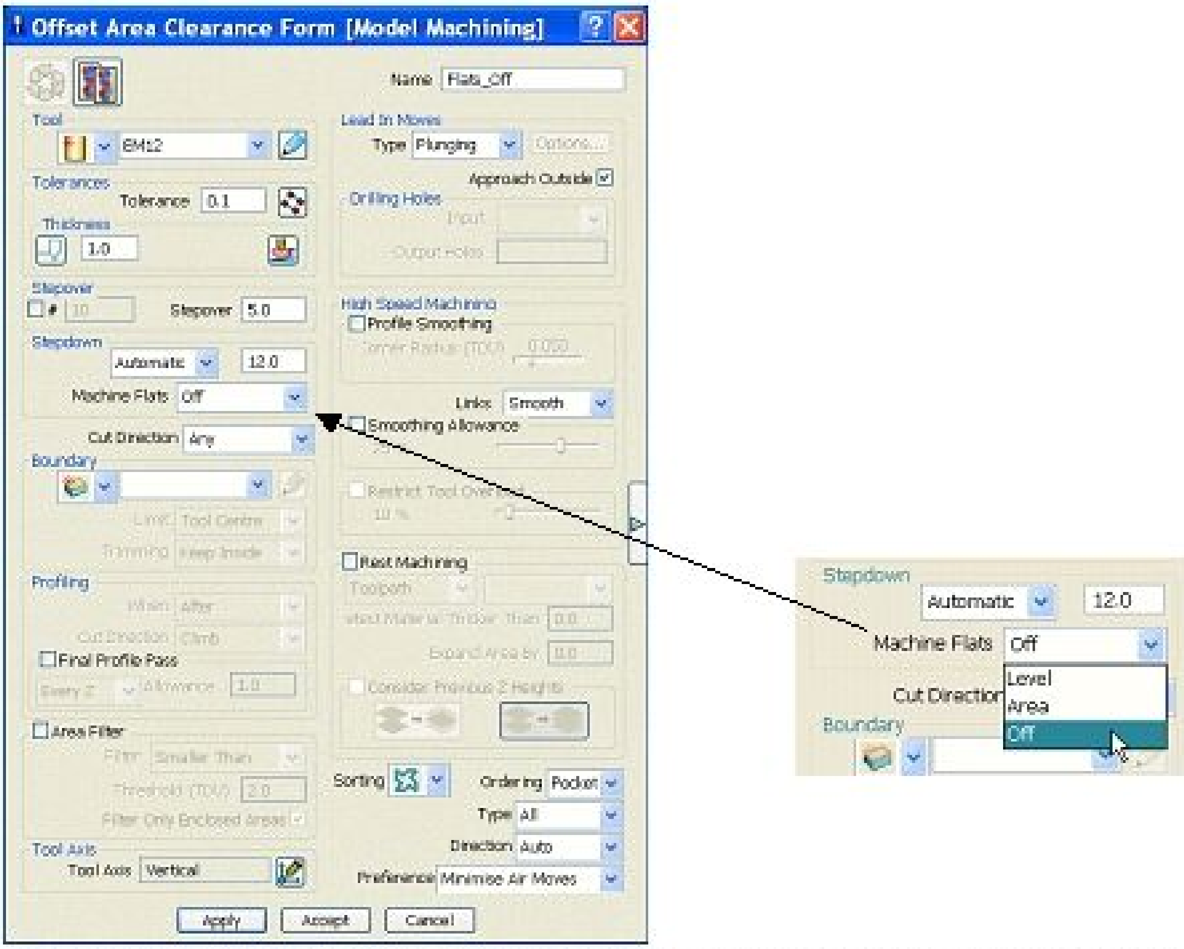
Machining Flats

The area clearance strategies in PowerMILL have an option that allows the user to control the way in which flat areas of the model are rough machined. These are found on the area clearance form under Machine Flats.

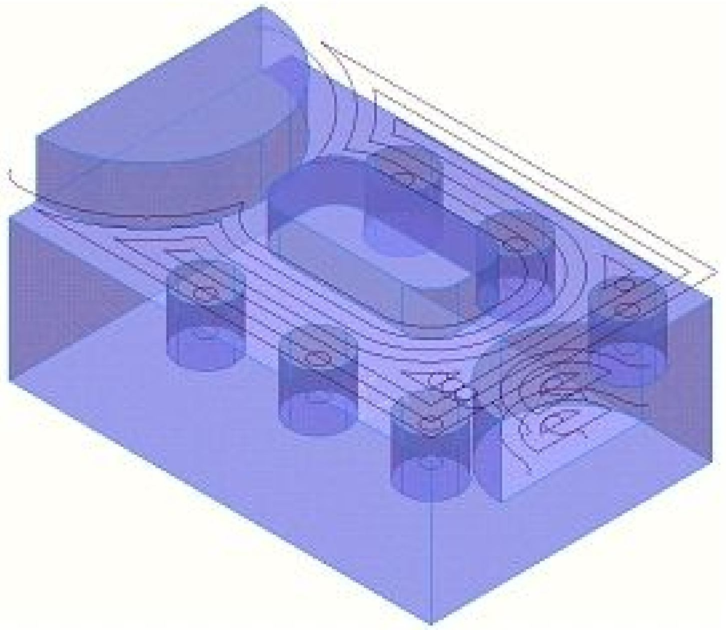
Import the model **PowerMILL_Data\Models\Flats.dgk**
Create a 12mm diameter **End Mill** tool and name it **EM12**
Calculate the **Block**
Set the **Rapid Move Heights** and check **Start/End Point** is set to default;
***Start Point* - Block Centre Safe** and ***End Point* - Last Point Safe**.



From the **Toolpath Strategies** form, select **Offset Area Clearance**.
Fill in the form exactly as below.

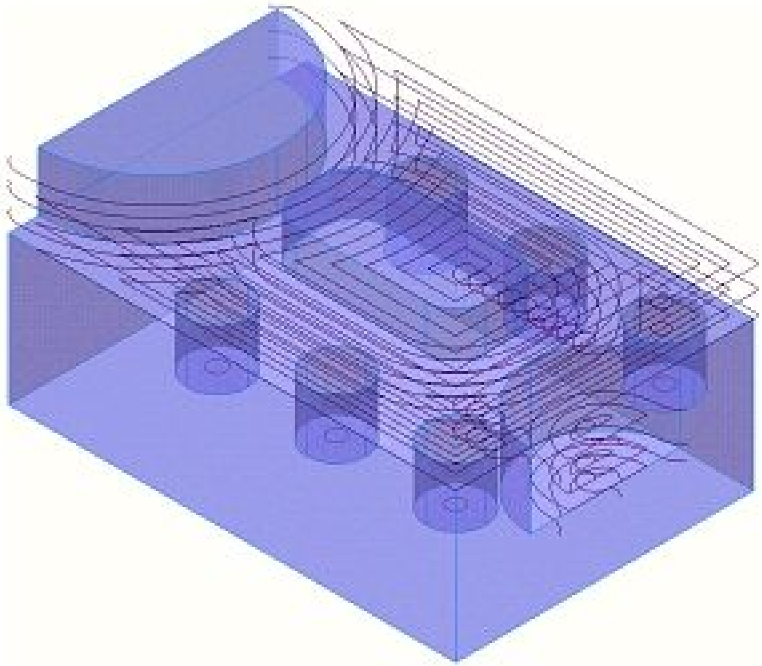


Apply and then **Cancel** the form



It can be seen that with **Machine Flats - Off** the toolpath has ignored the flat surfaces of the model. It has maintained a constant **Stepdown** value and completely performed area clearance across the material **Block** at each **Z Height**.

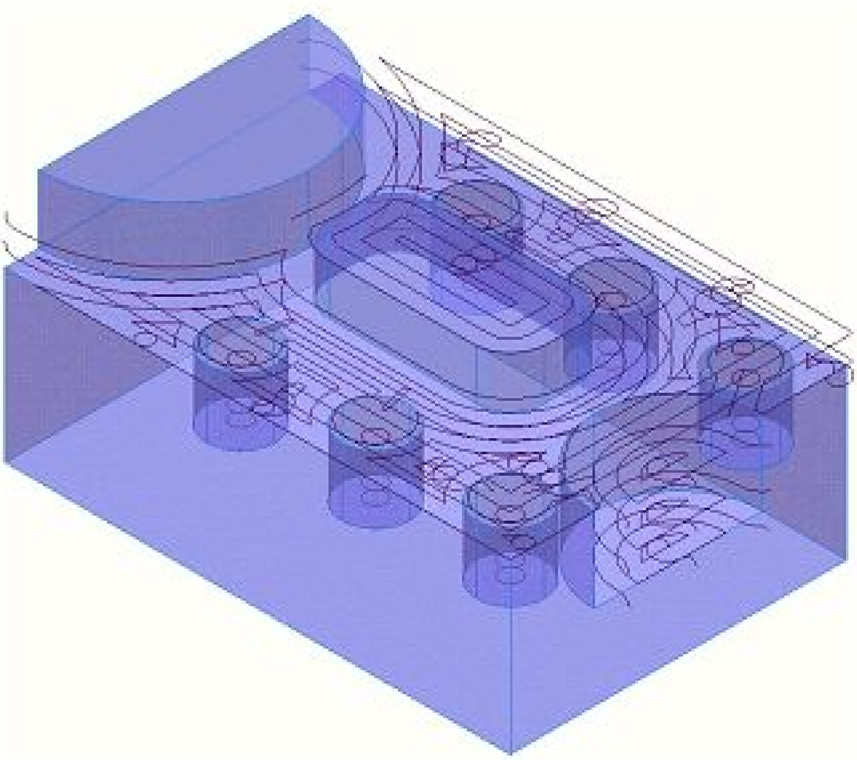
Make a **copy** of the **toolpath**.
Change the **Machine Flats** option to **LEVEL** (This is the default).
Change the **name** to **Flats Level**.
Apply and then **Cancel** the form.



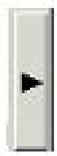
The area clearance toolpath now removes material from the flat surfaces leaving just 1.1mm this is equal to the **thickness** plus the **tolerance** set in the form. Where new slices have been added, the toolpath clears all the way to the edge of the block.

Make a **Copy** of the **toolpath**.
Change the **Machine Flats** option to **AREA**.
Change the **Name** to **Flats Area**.
Apply and then **Cancel** the form.

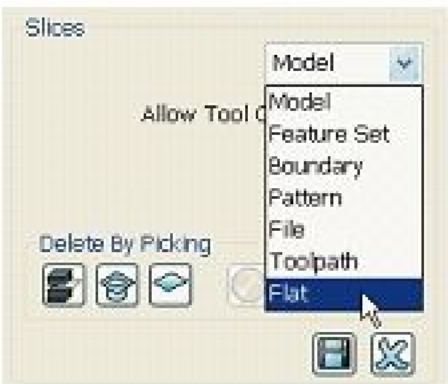
The toolpath now clears the flats as before but only extends to the edge of the component flat area. This provides a shorter toolpath than using the Level option.



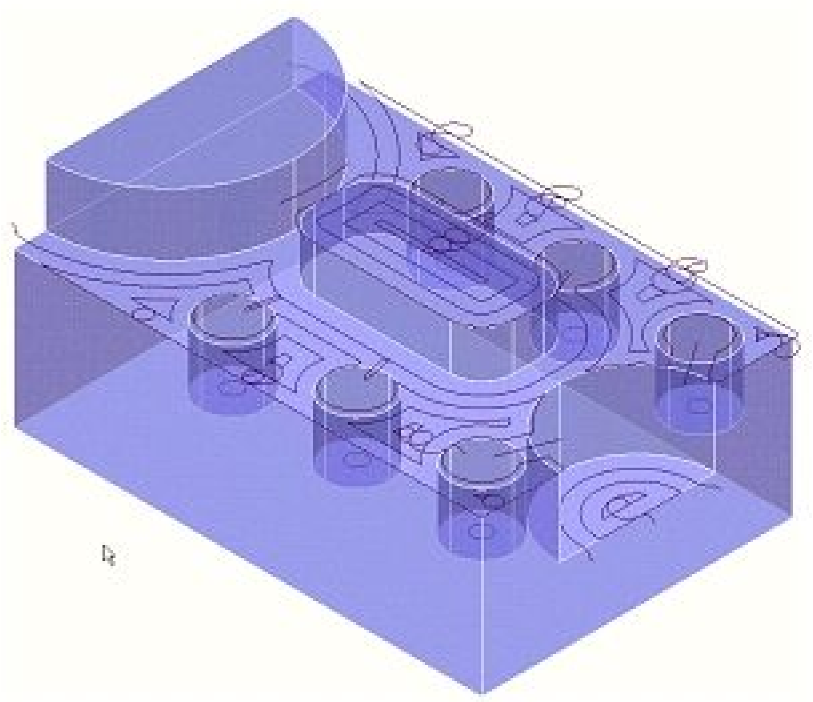
It is also possible to machine just the flat areas. This allows the user to rough the part first using Machine Flats set to OFF and then clear the flats in a second operation.



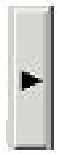
Make a **copy** of the **toolpath**.
Open the **Expert** part of the form by pressing the arrow at the right.
Select **Flat** from the ***Slices*** options pulldown.



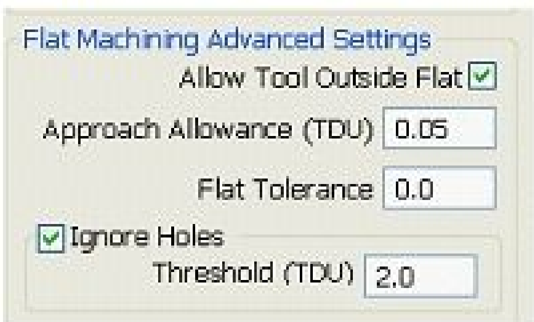
Calculate the slices.
Apply and **Cancel** the form.



The area clearance has now machined only the flat areas, the stepdown has been ignored The cutter avoids the holes resulting in a fragmented toolpath requiring many lifts. The area clearance can be made to ignore the holes if required.



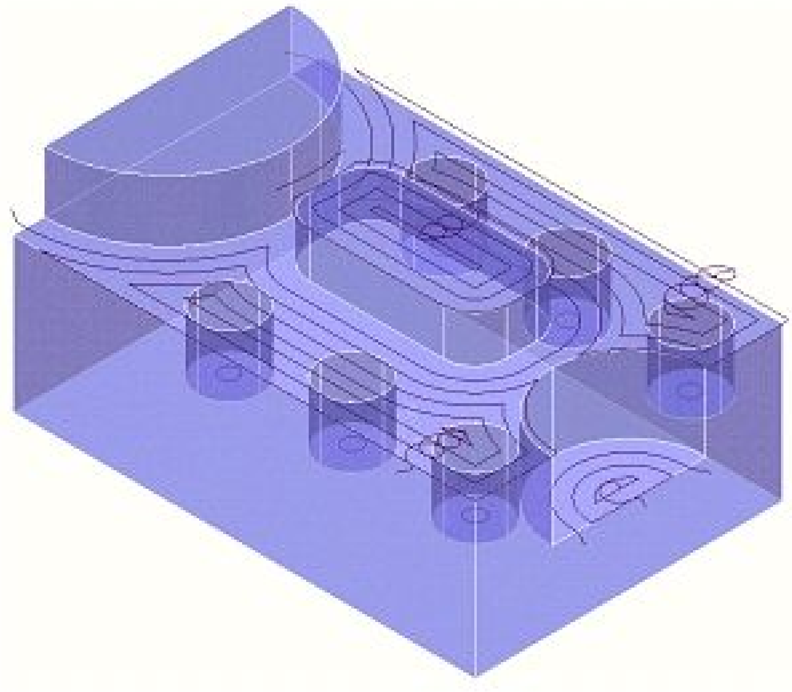
Make a **copy** of the **toolpath**.
Open the **Expert** part of the form by pressing the arrow at the right.
Tick the box next to **Ignore Holes**.



The Threshold in TDU (tool diameter units) tells PowerMILL what size of holes to ignore. With the tool EM12 selected and the Threshold set to 2, PowerMILL will ignore any hole less than 24mm in diameter.

Apply and **Cancel** the form.

The cutter now passes over the top of the holes without lifting giving a smoother toolpath. It should be noted that the flats at the bottom of the holes are still machined.

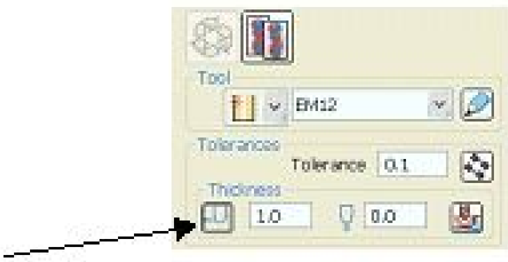


If the amount of material left on the flats is large then several passes can be made to reduce the tool loading during the area clearance.

Make a **copy** of the **toolpath**.
Open the **Expert** part of the form.
Select **Multiple Cuts**.

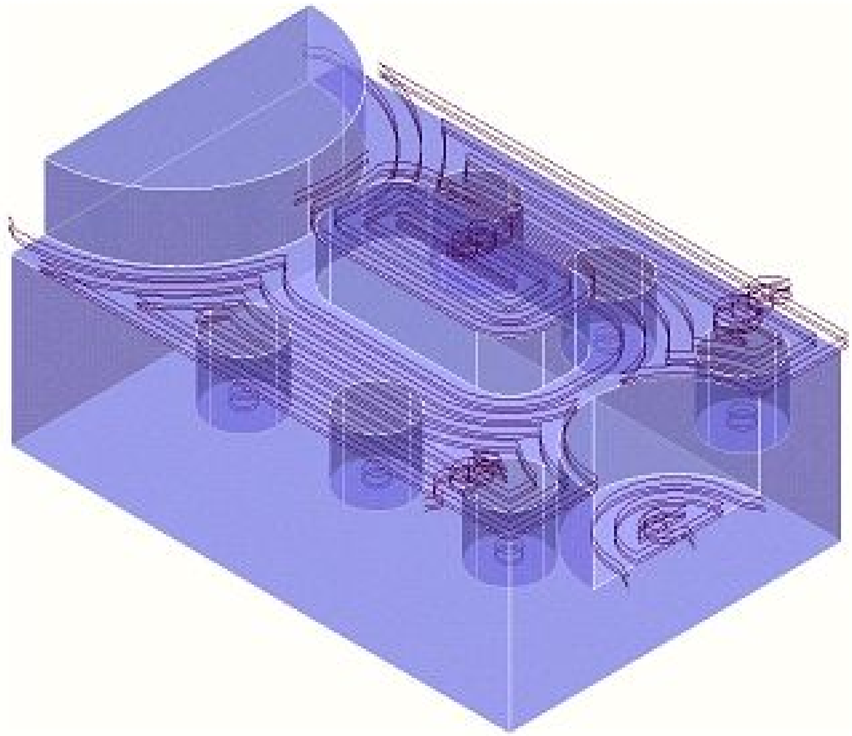
Fill in the remaining options as shown.

As the Flats are being machined using several passes it is a good opportunity to finish them to size on the depth (Axial thickness 0) but still leave 1mm on the sides (Radial thickness 1).



Click the **Thickness** icon to activate **Axial Thickness** (**0**).

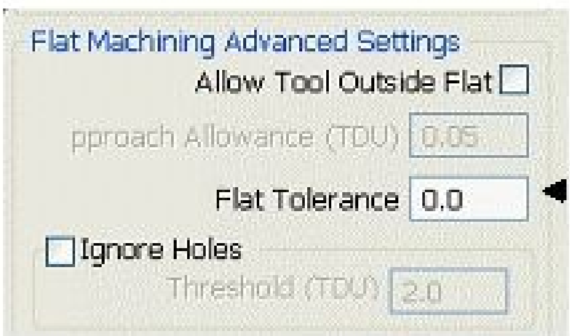
Apply and **Cancel** the **Area Clearance** form.



Three passes are made at each flat area, the final one with a 1mm depth of cut that cuts to size in the Axial direction and leaves 1mm stock on in the Radial direction.

INFORMATION

It should be noted that by default, the **PowerMILL - 3D Area Clearance** strategies will only detect completely flat surfaces. If a model contains surfaces that are almost flat then they will not be recognised unless the **Flat Tolerance** is set with a suitable, higher value. This option is found on the expert page of the **3D Area Clearance** form.



4. Raster, Radial, Spiral, and Pattern Finishing Strategies.

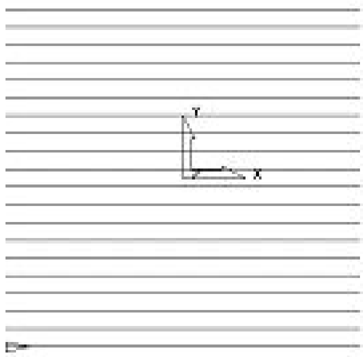
Introduction.

This chapter will cover **Finishing** strategies created by the **downward projection of a Pattern**, which include four types, **Raster, Radial, Spiral** and **Pattern**.

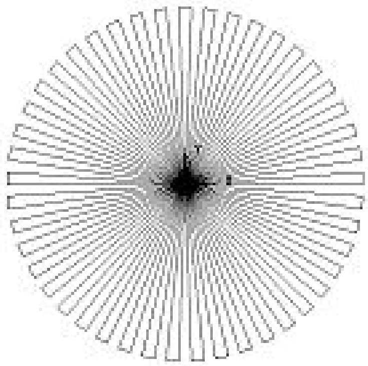
PowerMILL generates toolpaths by projecting a pre-defined strategy **down the Z-axis** onto the model. The standardised geometry for **Raster**, **Radial**, and **Spiral** are generated by entering values into the **Finishing** Form. The resultant **Pattern** can be displayed by selecting **Preview** before executing the command by selecting **Apply**. The **Pattern** option requires a user-defined geometric form (active **Pattern**), which is projected down Z onto the model as a toolpath.

A Preview of the four pattern strategies are shown, looking down the Z.

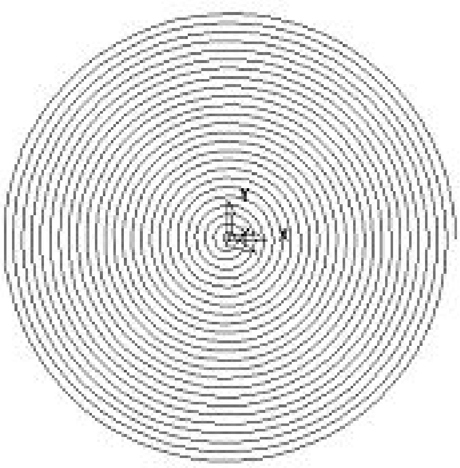
Raster



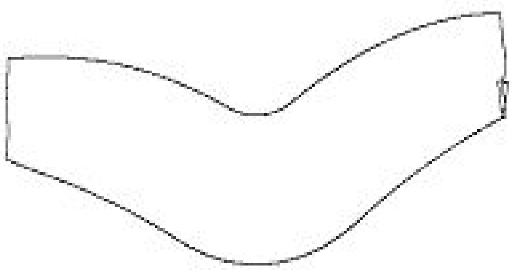
Radial



Spiral

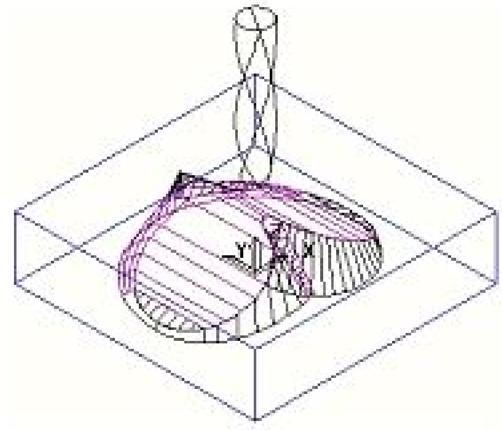


Pattern (User Defined)



Raster Example

Delete All and **Reset forms** and from **File > Examples** select the model **chamber.dgk**.
Calculate the **Block** and define a **Dia 12 Ball Nose** tool named **bn12**.



The model and tool are displayed..

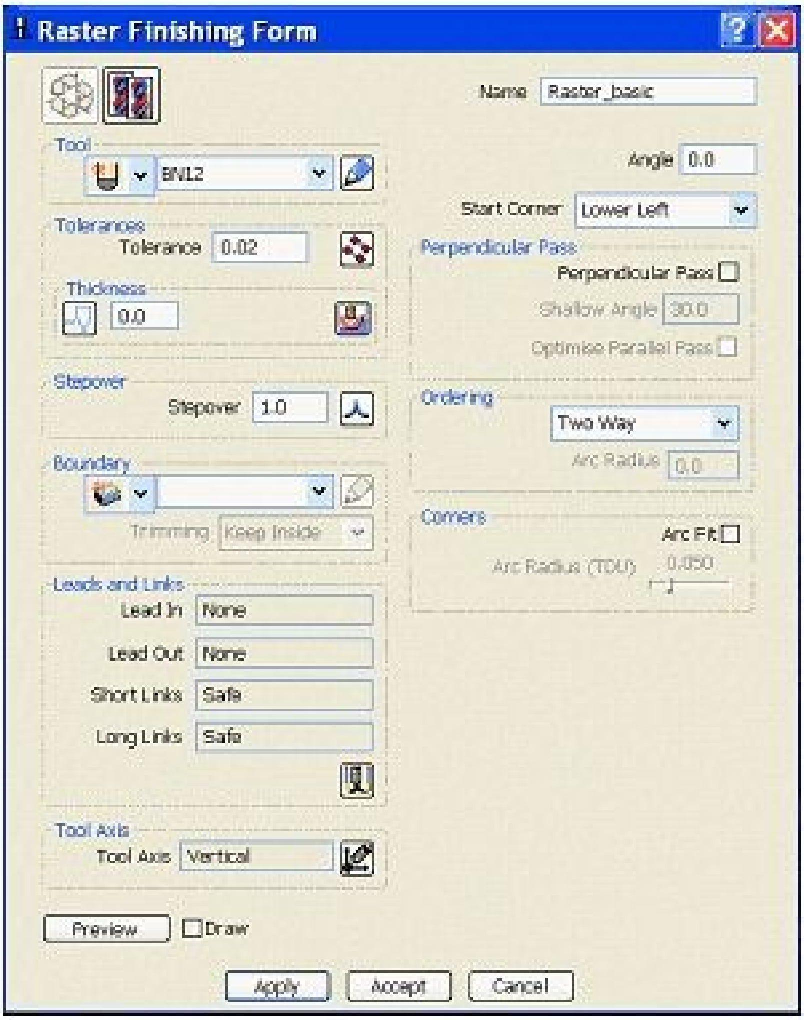
Select the **Rapid Move Heights** icon  from the top toolbar and in the form click **Reset to safe heights** then **Accept**.

Select the **Start and End Point** icon  from the top toolbar then **Accept**.

Note that the default settings are **Start Point - Block Centre Safe** and **End Point - Last Point Safe**.

Select the **Toolpath Strategies** icon from the top toolbar.

Select the **Raster Finishing** icon then **OK**.



Name the toolpath **Raster_basic**.

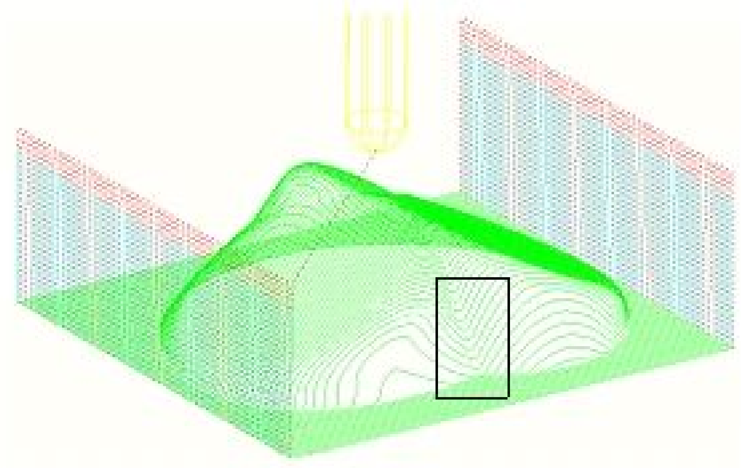
Enter the **Tolerance** as **0.02** and **Thickness** as **0**.

Enter **Ordering - Two Way**.

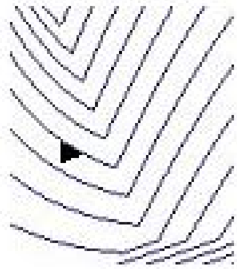
The default tolerance of 0.1 will give a fairly coarse surface finish. For a finer finish a lower value such as 0.02 is used. A thickness of 0 will machine the material to size, within the tolerance.

Select a tool **Stepover** of **1mm**.

Apply and **Cancel** the Form.



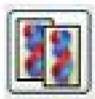
On closer inspection the toolpath contains sharp changes in direction.

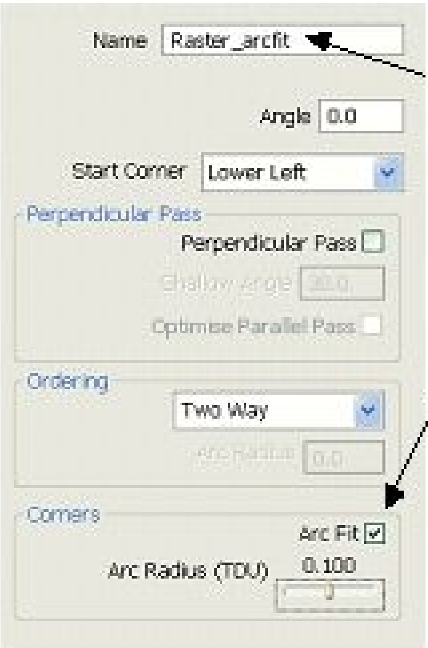


Selecting the **Arc Fit** option in the form can eliminate these.

Right click over toolpath **Raster_basic** in the explorer and select **Settings** from the available menu.

Select the **Copy Toolpath** icon from the form.





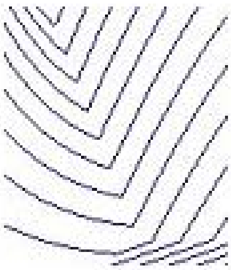
Rename the toolpath **Raster_arcfit**.

Check the box **Arc Fit** and change the **Arc Radius** to **0.1**.

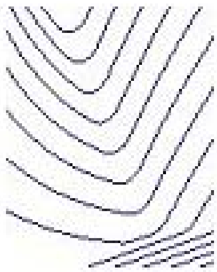
The Arc Radius (0.1) is multiplied by the tool diameter (12mm) to give a final radius of 1.2mm.
This option is particularly good for HSM (High Speed Machining).

Press **Apply** and **Cancel**.

Zoom into the same area as previous to see the changes.

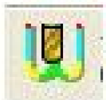


Before Arc Fit



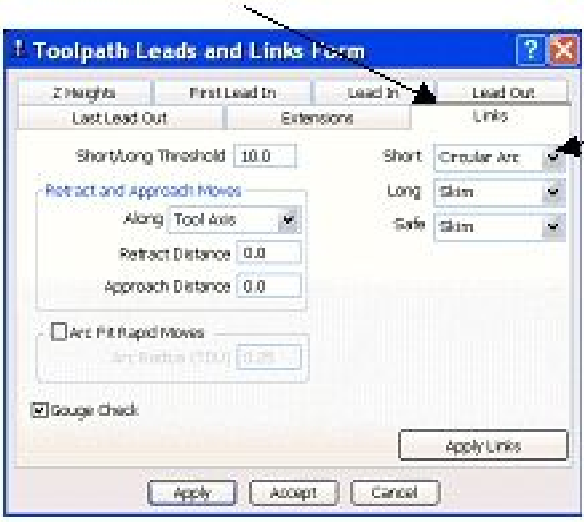
After Arc Fit

Where the toolpath is stepping over it is lifting up to the **Safe Z height** each time. This is wasting time putting in these unnecessary lifts. To make the toolpath more efficient the **Leads** and **Links** can be altered.



Select the **Leads and Links** icon from the top toolbar.

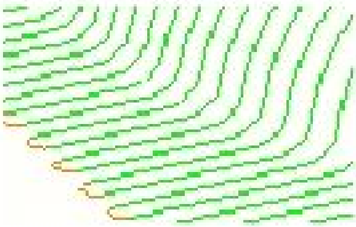
Select the **Links** tab on the form.



Change the **Short** links to **Circular Arc**

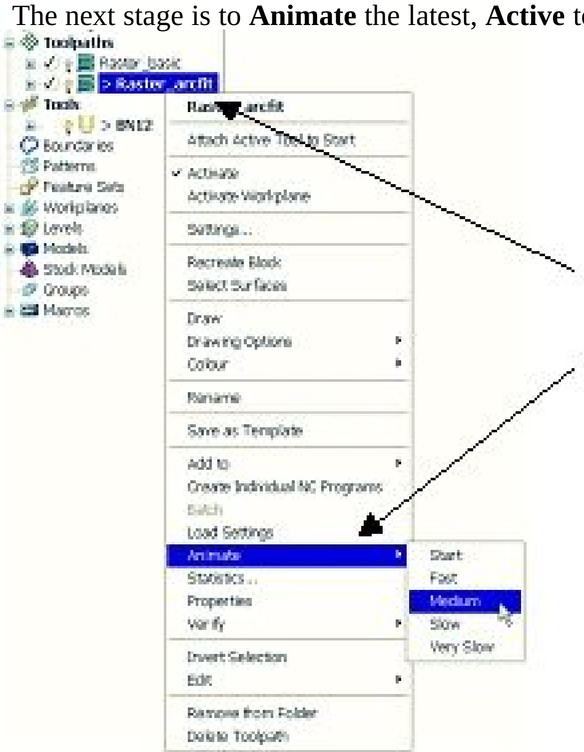
Change the **Long** and **Safe** links to **Skim.**

Apply Links and **Accept** the form.



Most of the lifts have been eliminated hence saving time and a circular arc has joined the links between adjacent toolpath tracks.

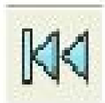
The next stage is to **Animate** the latest, **Active** toolpath, **Raster_arc fit**.



To **Animate** the toolpath, **right click** over the toolpath icon in the **explorer** area, move down to **Animate** and select the required simulation speed.

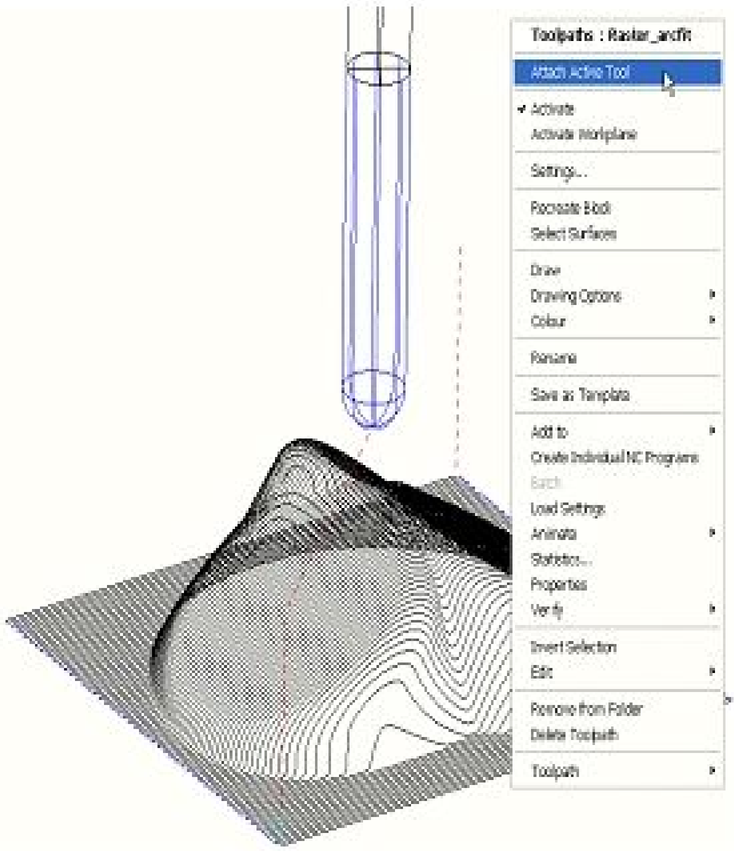
When the **Animation - Start** is selected a new toolbar will appear at the top of the screen. This is the **Simulation** toolbar.

When the **Animation** has finished it will be observed that the tool remains at the end of the last retract move. To send the tool back to the **Start Point** either press the **Home** key on the keyboard or select the **Go to beginning** button on the **Animation** toolbar.



To see how a certain part of the toolpath runs in more detail it is possible to attach the tool to an area and use the **Left/Right Arrow keys** on the keyboard to make the tool move backwards or forwards along the toolpath.

Position the cursor where required and **right click** on top of the toolpath.



Select **Attach Active Tool.**

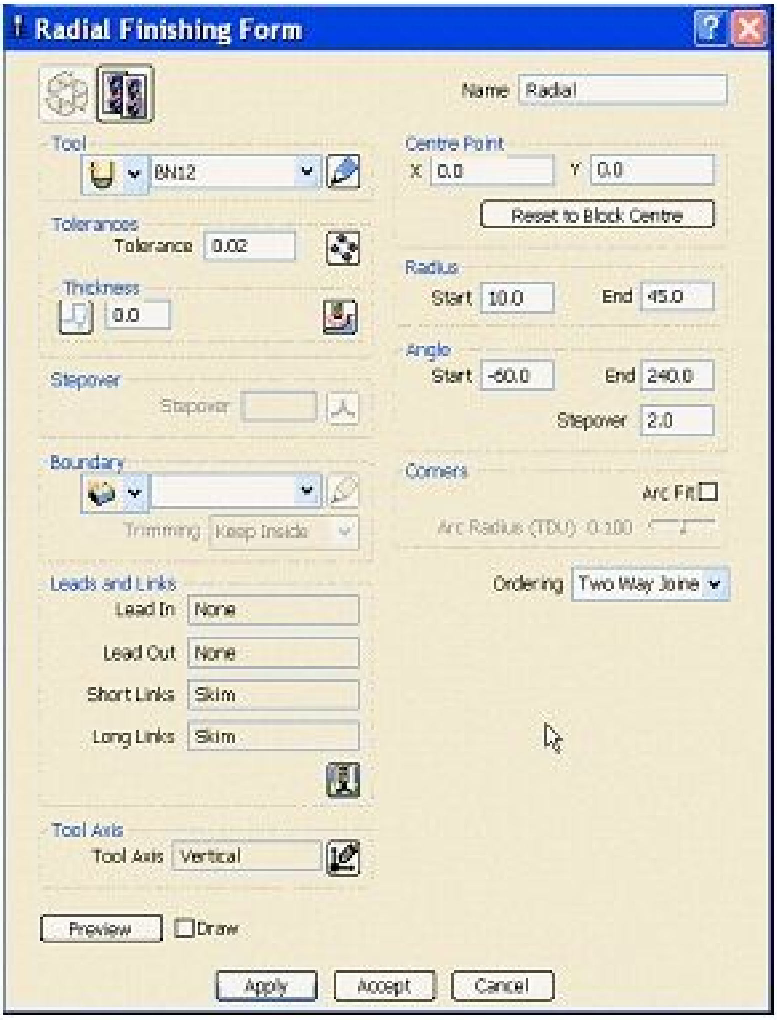
Use the **Left** and **Right** arrow keys to move the tool forwards and backwards along the toolpat

Tip: - Go to **Help** and in the Index area type Attach. You can then display more info on '**Attach Active Tool**'

Experiment with some of the other options available on the **Animation** toolbar.

Radial Machining

Select the **Toolpath Strategies** icon from the top toolbar. From the form select the **Radial Finishing** icon then **OK**.



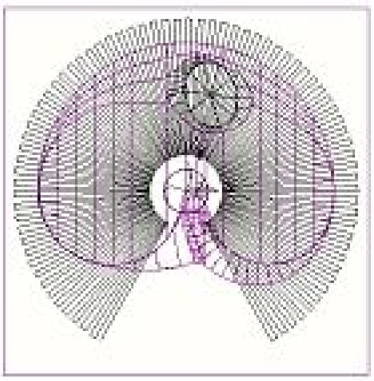
Name the toolpath **Radial**.

Make ***Radius* End 45**.

Make ***Angle* Start -60** and **End 240**.

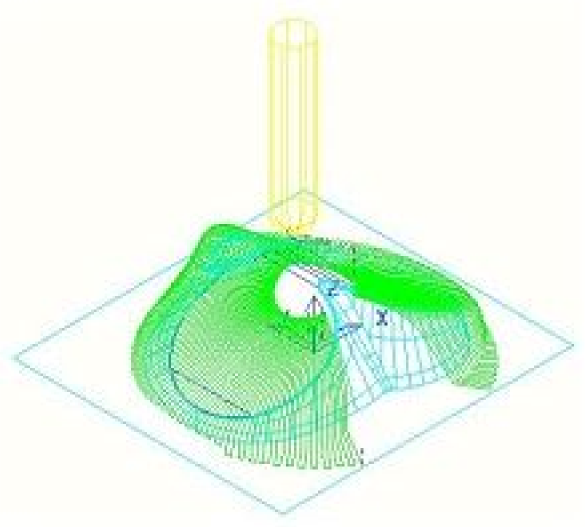
Set **Stepover** to **2** and **Two Way Joined Up** ticked.

Set the view to look down the Z direction and select **Preview**.



The route of the toolpath is shown with the **Stepover** being angular as determined by the **radial** pattern.

Apply and **Cancel** the form.



The toolpath follows the previewed **radial** pattern.

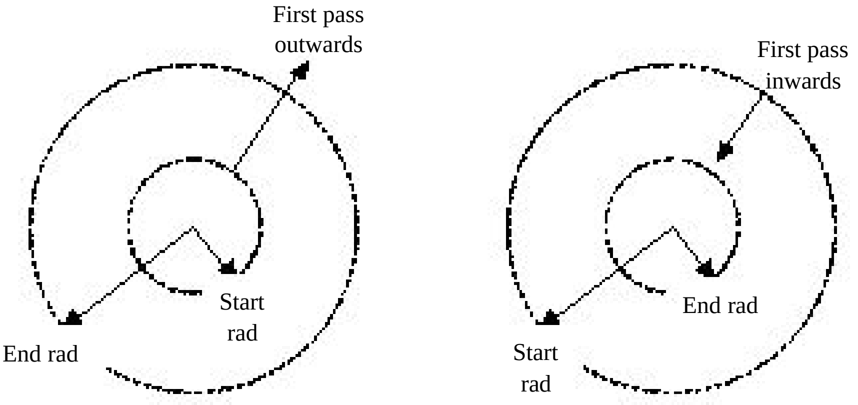
Animate the toolpath.

Centre Point

The **Centre Point** defines the origin of the radial pattern. By default the pattern is initially centred at the zero coordinates. This can be redefined if required to **user-defined** values or alternatively be centred to the middle of the block by clicking the **Reset to Block Centre** button.

Start and End Radius

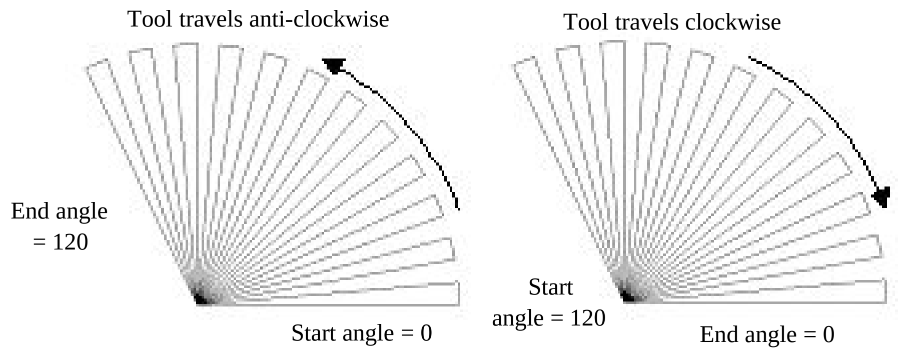
These parameters determine the dimensions of the pattern and the direction of the first pass:



Start angle and end angle

These parameters perform two functions: The first defines the portion of the circular area to be machined, and the second determines whether the tool tracks are generated in a clockwise or anticlockwise direction. Angles are measured anti-clockwise from the positive X-axis.

Start angle (120) > **end angle (0) -** tool tracks generate **clockwise**.
Start angle (0) < **end angle (120) -** tool tracks generate **anticlockwise**.

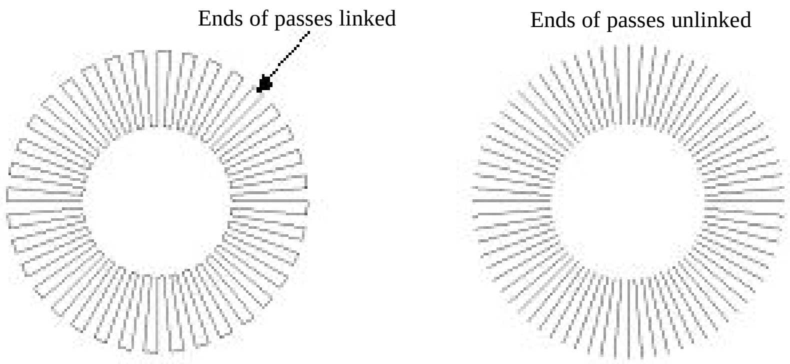


Stepover

The **Stepover** is the angle between consecutive passes. Note: - that further away from the centre point, the coarser the finish due to the passes becoming further apart.

Join Up

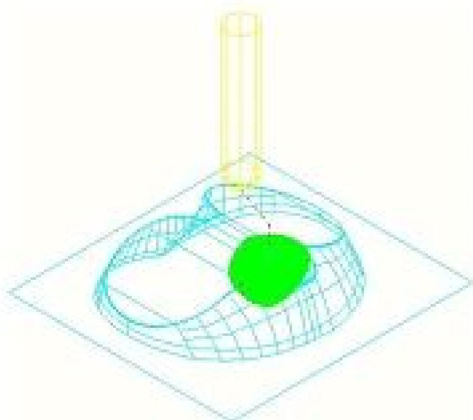
The tool tracks can be unidirectional or bi-directional depending on the setting of the **Join Up** flag. This is illustrated below:



Join Up flag *set*   **Join Up** flag *unset*

Radial Exercise

Generate a **Radial** toolpath called **Radial_1**, using the tool **bn12**, **Tolerance** of **0.01** and **Thickness** of **0.5**. Set the **Centre Point** at **X 6.5 Y 26**. Select a **Start Angle** of **0**, **End Angle** of **360, Start Radius** of **0, End Radius** of **12** with a **Stepover** of **1**.



The toolpath is generated in a small area.

Animate the toolpath.

Spiral Machining

Select the **Toolpath Strategies** icon from the top toolbar. From the form select the **Spiral Finishing** icon then **OK**.

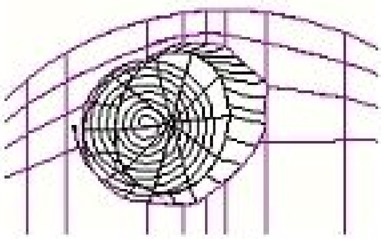
Name the toolpath **Spiral**.

Set the **Thickness** to **0** and **Stepover** to **0.5**.

Enter a **Centre Point** of **X 8 Y 26**, set **Start Radius** to **10**, **End Radius** to **0** and **Direction** as **Clockwise**.

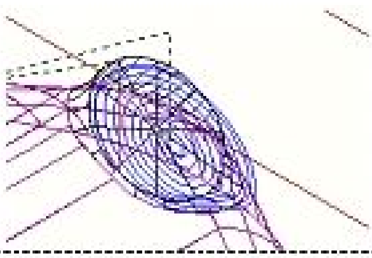
A spiral toolpath is a continuous one, where there are no links between adjacent toolpath tracks.

Set the view to look down the Z direction and select **Preview**.



The preview toolpath is displayed over an area of the model.

Apply and **Cancel** the form.

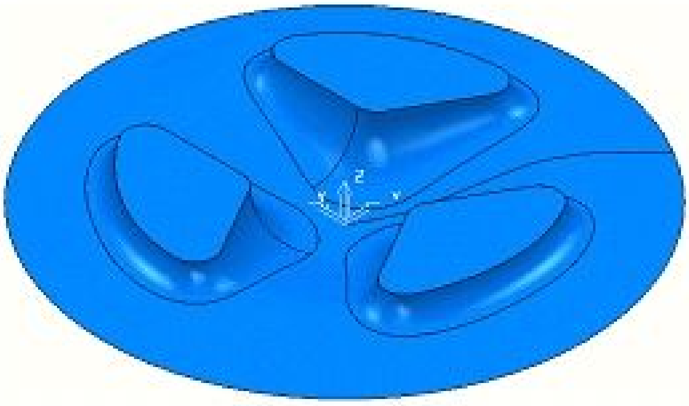


The spiral toolpath is produced.

Pattern Machining

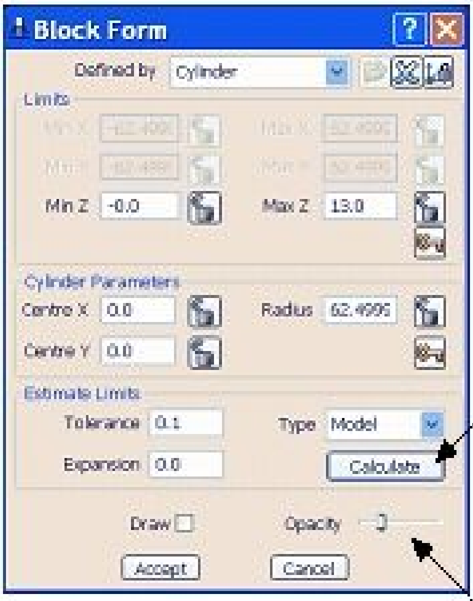
This technique requires a user defined **Pattern** for projection onto the model as a toolpath. This option can be used for applications such as scribe lines, lettering, and non-standard tool strategies. A **Pattern** is created either from within **PowerMILL** or as imported **Wireframe** data.

Delete All and **Reset forms** and from **File > Examples** select the model **swheel.dgk**.



The component is circular as a result of which the **Block - Z Minimum** and **Z Maximum** will be developed as a **Circular** billet.

Select the **Block** icon from the top toolbar.



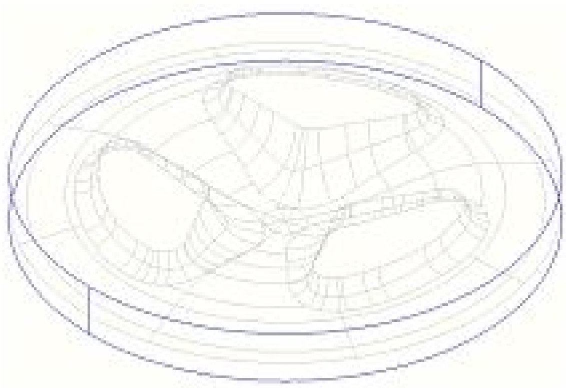
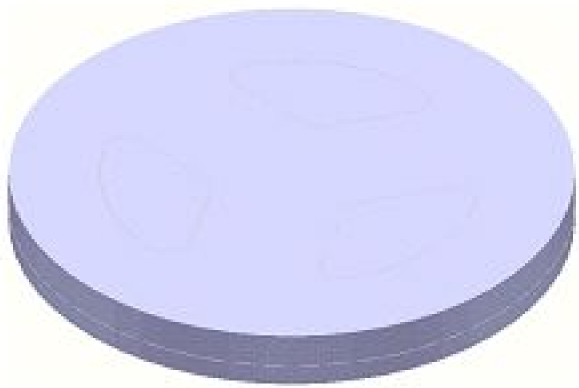
Select ***Defined by* - Cylinder**.

The Radius is worked out automatically when the Calculate button is clicked

Select **Calculate**.

To provide more stock on the base **Modify** the **Min Z** value to **-1**, **Lock** the value, and **Accept**.

The **Block** can be displayed as wireframe or of variable **Opacity** up to solid, depending on the position of the **Opacity** slider switch.

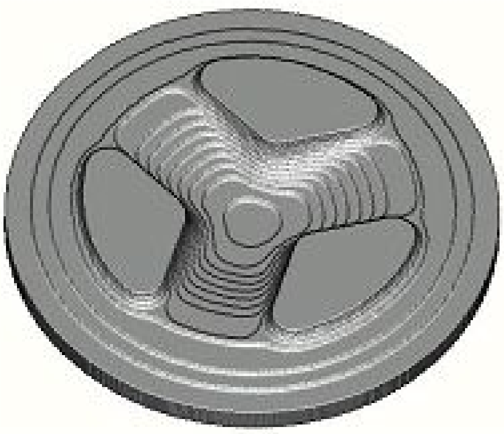
Before the **Pattern** finishing strategy is created, a preliminary **Area Clear** strategy is required to remove the bulk of the material, leaving 0.5mm, followed by a **Semi Finishing** Strategy leaving 0.2mm prior to the final **Finishing**.
The object of this **Pattern** exercise is to consolidate some of the training covered earlier as well as allowing the user to **simulate** the **Pattern** strategy at the correct stage in the overall machining process.

Define a **Tip Radiused** tool with **Diameter 10**, **Tip Radius 1** called **D10T1** and make this **Tool Number 1**.

Define a **Ball Nose** tool with **Diameter 6** called **BN6** and make this **Tool Number 2**.

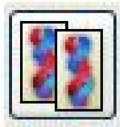
Activate the **D10T1 Tip Radius tool**.

In the **Rapid move heights**  select **Reset to Safe Heights**.
Calculate an **Offset AreaClear Model** toolpath using the following parameters: ***Name*** - **RoughOp1**, **Tolerance 0.1**, **Thickness 0.5**, **Stepover 5.0**, **Stepdown 1.0**, and **Cut Direction Climb**.



Simulate the toolpath in **Viewmill** to give a result similar to as shown left.
Activate the **BN6 Ball Nose tool**.
Calculate a **Spiral Finishing** toolpath using the following parameters:
Name - **SemiFinishOp1**, **Tolerance 0.02**, **Thickness 0.2**, **Stepover 1.0**, **Centre Point X 0 Y 0**, ***Radius*** **- Start 62.0** and **End 0.0**, and **Direction Clockwise**.

Simulate this new toolpath in **Viewmill**.
Right click over the **Semi Finish** toolpath in the **explorer** and select **Settings**.

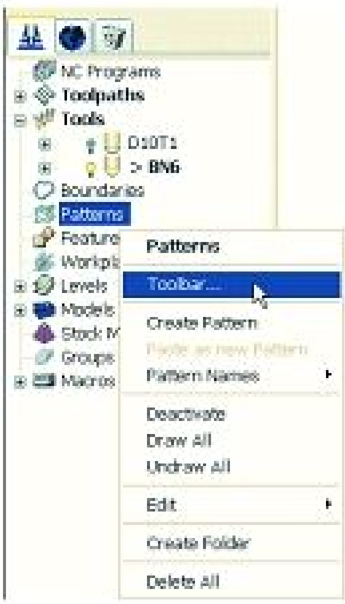


From the form select the copy icon.
Enter a new ***Name*** - **Finish**, change the **Thickness** to **0.0** and **Stepover** to **0.5** then **Apply** and **Close** the form.

Simulate the final toolpath in **Viewmill** to give this result.

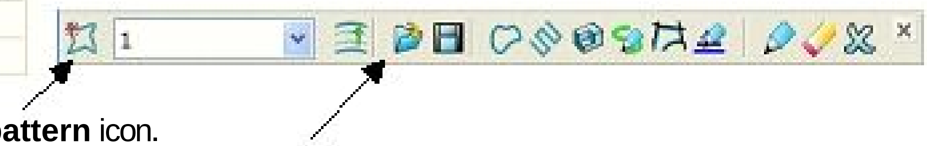
A **Pattern** finishing strategy will be now applied to machine the text, **Engraved** into the component.

A suitable, pre-defined **Pattern** will be **imported** for use with the strategy.



Right click the **Pattern icon** in the **explorer** and select **Toolbar….**

When **Toolbar** is selected off the **Pattern** menu it loads a new toolbar into **PowerMILL**. The toolbar contains icons to create an empty **Pattern** and load different types of entities into that empty pattern.
The toolbar can be removed from the screen by clicking the small x at the right.

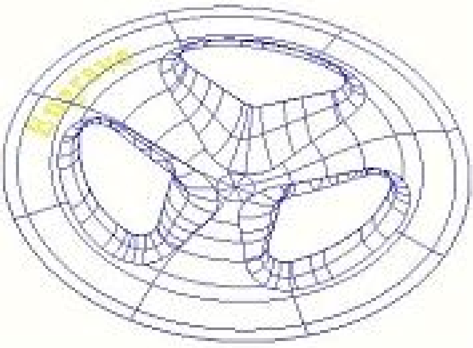


Select the **Create pattern** icon.
Click on the **Insert file into active pattern** icon.

Select the **eg** button.

Double click the **Patterns** folder and then select **Engrave.dgk**.

Select **Open**.



The **Pattern** is made up of 2D flat geometry and lies at the bottom of the block. This will be dropped onto the model in 3D form while creating a single pass toolpath with a negative **Thickness** value.

Define a **Ball Nose** tool with **Diameter 1** called **BN1** and make this **Tool Number 3**.

Select a **Pattern Finishing** strategy.

Enter the ***Name*** - **EngraveOp1**.

Enter the **Tolerance** as **0.02** and **Thickness** as **-0.4**.

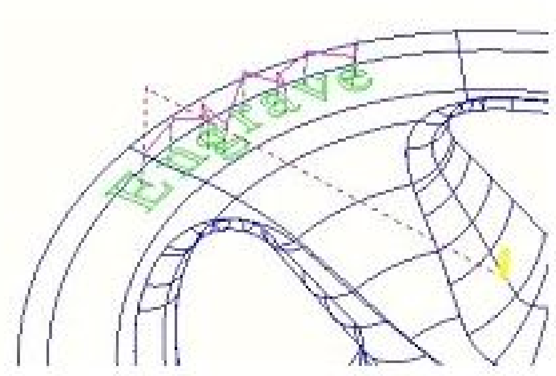
Select **Pattern 1** (if not already selected).

With **Automatic** set the pattern will be Dropped / Projected onto the model.

Select **Apply** then **Cancel** the form.

By entering a negative

Thickness the tool will machine into the previously finished surface. This is frequently used for engraving, or to achieve a spark gap when machining part of a component as an electrode.



Open the **Leads and Links** form and **Apply** the **Short, Long** and **Safe Links** to **Skim**.

The **Pattern** has been de-activated to enable a better visualisation of the toolpath.

Animate the toolpath and simulate it in **Viewmill**.

The **ViewMill** block has been painted so that the final **Pattern** toolpath appears more pronounced during the simulation.

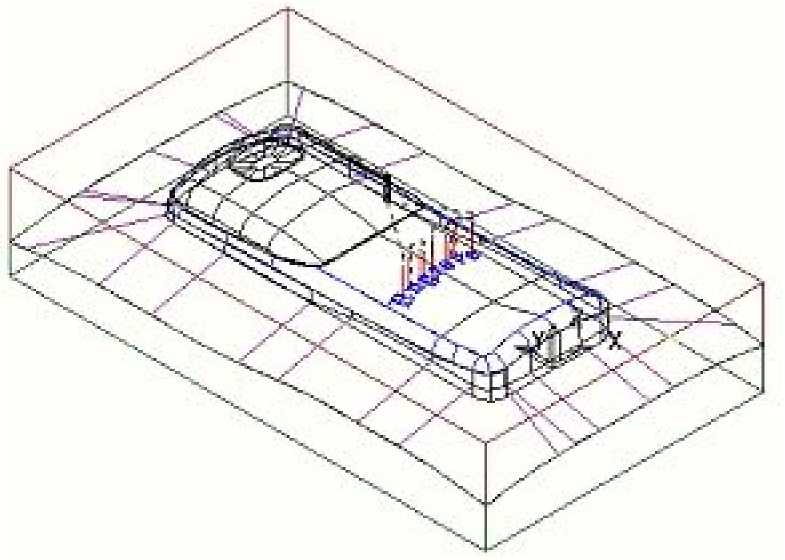
Save the Project as **Pattern Machining** in **C:\Temp**.

Pattern Exercise

The same imported **Pattern** will be used to engrave text into a telephone handset.

Right Click **Models** on the **Explorer** and Select **Delete All**.
Use **File - Examples** and load the model **phone.dgk**.
Deactivate Toolpath **EngraveOp1**.
Reset the **Block**, **Rapid Move Heights** and the **Tool Start Point**.
Generate a **Pattern Finishing toolpath** leaving all values as before.

The **Pattern**, 'Engrave' used earlier has been re-used on this model. If the pattern is in the wrong position, it can be moved using the options on the **Pattern** - right click menu.



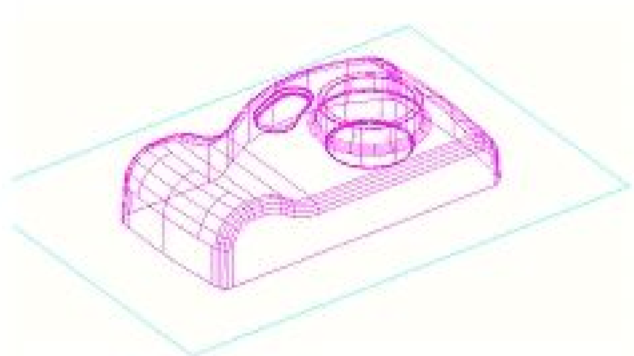
5. 3D Offset and Constant Z

Introduction

In this chapter **3D Offset** and **Constant Z** finishing strategies will be applied to a model that consists of a combination of flat and steep areas plus a pocket with vertical walls.

3D Offset Machining defines the tool **Stepover** relative to the 3D surface shape providing consistency over both flat areas and steep sidewalls. In this exercise applying this toolpath to the complete model without using **Boundaries** would not be recommended. Although this toolpath maintains a constant **Stepover** there is nothing to prevent the tool plunging with a full width cut into the deep pocket area.
Boundaries used correctly will limit the **3D Offset** toolpath to the flatter areas leaving the remaining steep areas to be machined using the **Constant Z** strategy.

Delete All and **Reset forms** and **Import** the model **PowerMILL_data\Models\camera.dgk**.



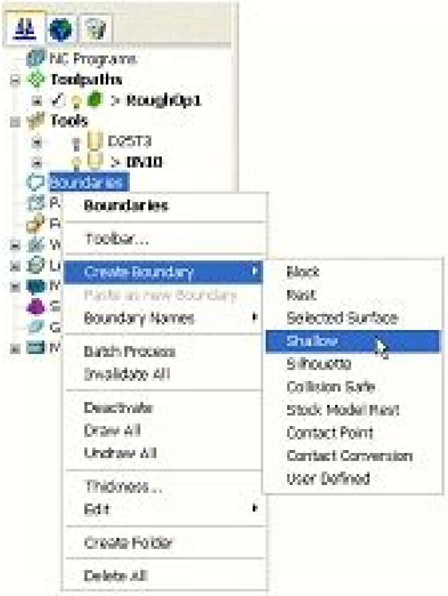
This model is a good example where it is a better option to use more than one finishing strategy. **3D Offset** and **Constant Z** strategies will be used and they will be they will be kept separate by being limited to a **Boundary**.
Before finishing can take place an **Area Clearance** toolpath will be created to remove the bulk of the material.

Calculate the **Block** to component size, in **Rapid Move Heights** apply **Reset to Safe Heights**, and use **Block Centre Safe** for both the **Start and End Point**.
Define a **Tipped Radius** tool with **Diameter 25** and **Tip Radius 3** called **D25T3**.
Calculate an **Offset Areaclear Model** toolpath called **RoughOp1** using a **Thickness 0.5**, **Stepover 10** and **Stepdown 3**.
Simulate the toolpath in **Viewmill**.

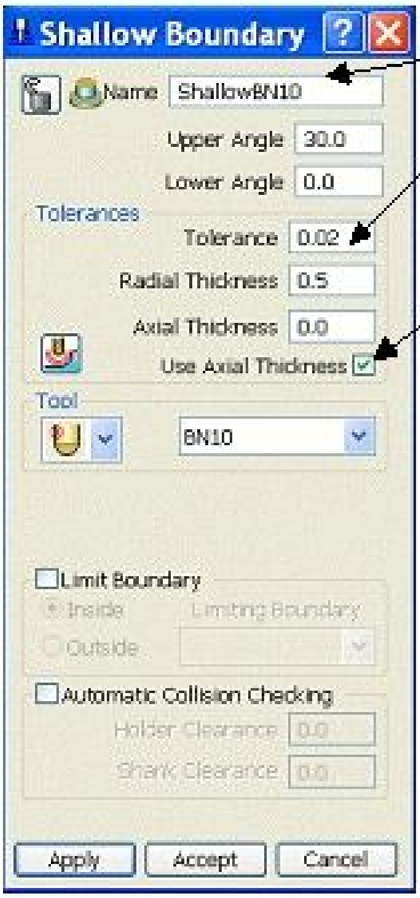
The **Viewmill** simulation should look something similar to this.

Define a **Ball Nose** tool with **Diameter 10** called **BN10**.

We are now in a position to define the **Boundary**. Certain types of **Boundary** are calculated based on the **Active** tool.
The **Tolerance** and **Thickness** values in the **Boundary** forms are important and normally reflect the same **Tolerance** and **Thickness** values with the subsequent toolpaths applied to them.



Right click on the **Boundaries icon** in the **explorer** and select **Create Boundary -> Shallow**.



Input the **Name - ShallowBN10.**

Input a **Tolerance 0.02**.

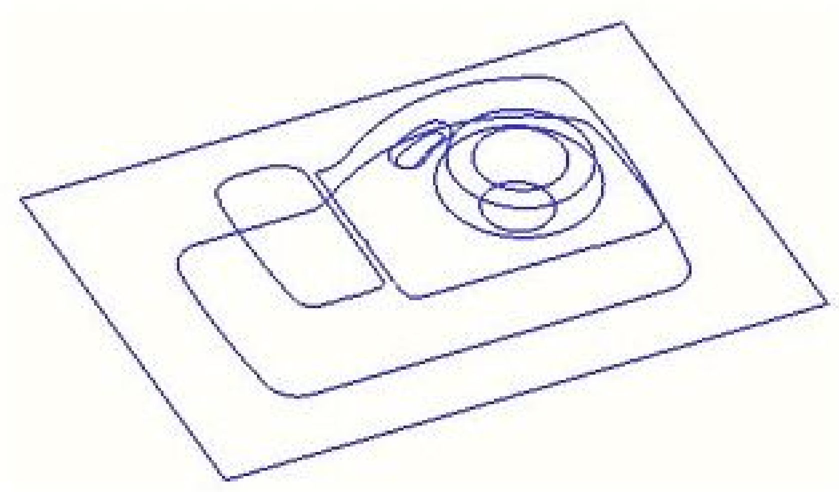
Tick the ***Use Axial Thickness*** box and enter a ***Radial Thickness* 0.5** and an ***Axial Thickness* 0**.

A **Shallow Boundary** defines segments from areas on the model that are defined by an **Upper** and **Lower** threshold angle. It is therefore specifically suited to steep walled and shallow surface machining techniques.

With **Radial Thickness** set to **0.5** the stock thickness left on the sidewalls from the roughing will not be machined at this stage.

Make sure that the correct tool is **Active**, in this case **BN10**.

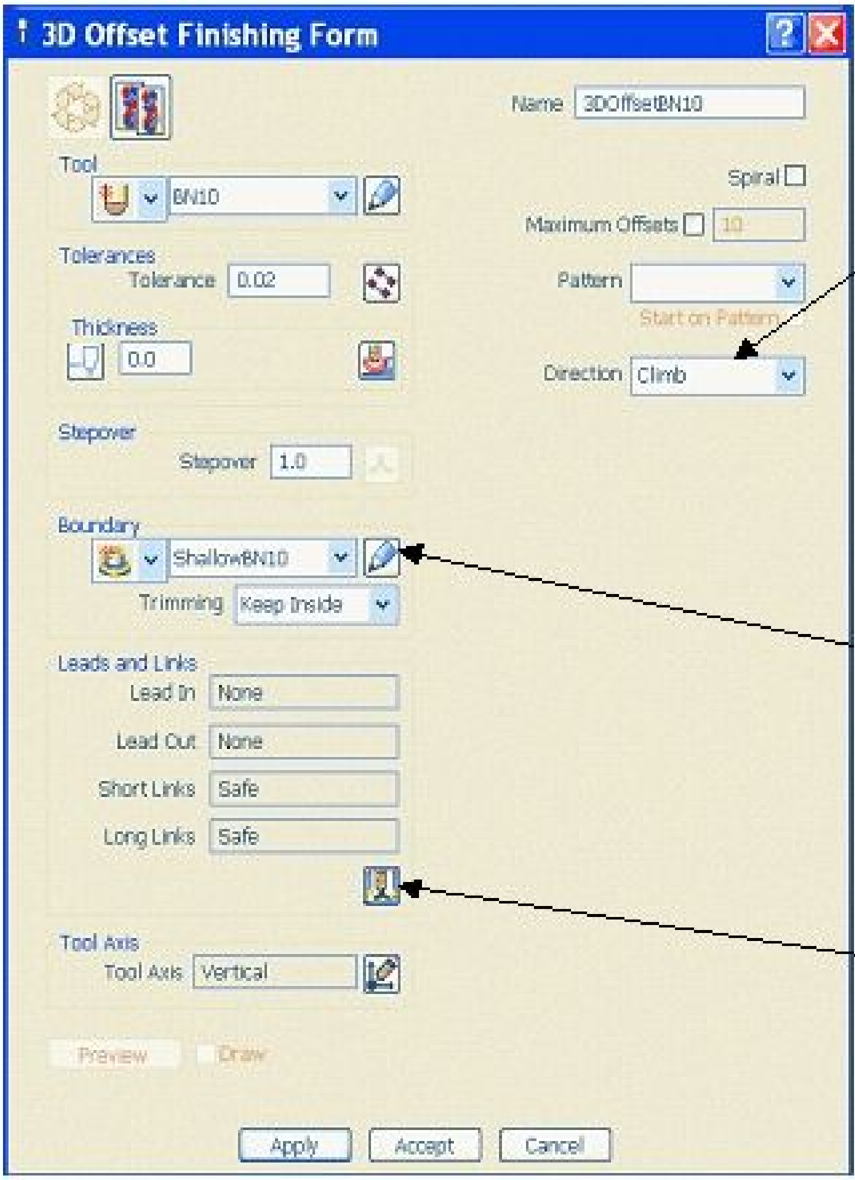
Apply and **Accept** the form.



With the model and toolpaths undrawn the **Boundary** should look something like this. It is made up of numerous segments each one dividing the model into steep and shallow areas. Any of which can be selected and individually deleted at any time (only before being assigned to a toolpath).

Select the **Toolpath Strategies** icon from the top of the screen.

Select a **3D Offset Finishing** strategy from the form then **OK**.



Enter the **Name - 3D Offset BN10**.

Select ***Direction* - Climb**.

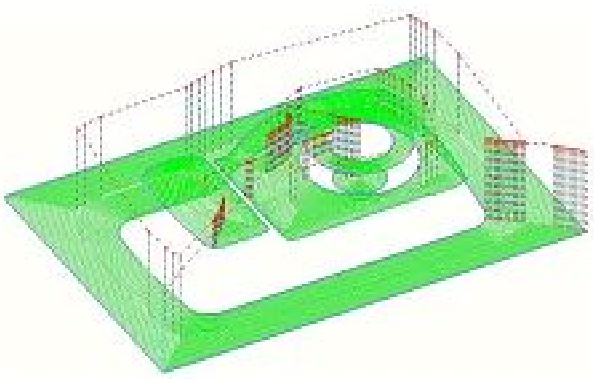
Enter a **Tolerance 0.02**.

Enter a **Thickness 0**.

Notice that the newly created, **Active Boundary** is automatically selected for use. If a different **Boundary** is required it can be selected from the pull down menu.

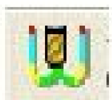
Select the **Leads and Links** icon and set the **Lead In** to **None**, **Links - Short and Long Links** to **Safe**

Apply and **Cancel** the form.

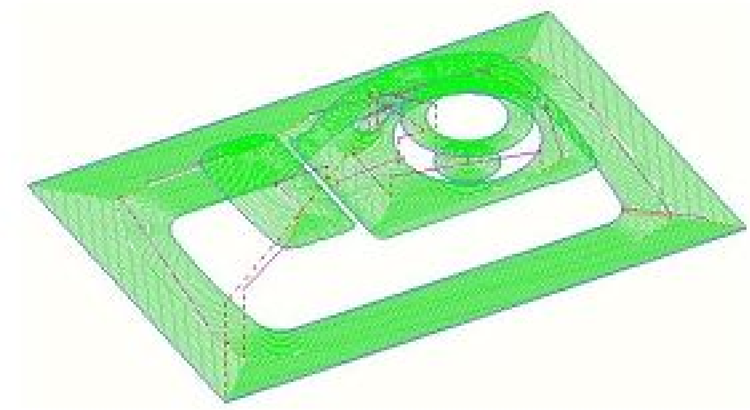


The toolpath is calculated following the contours of the **Boundary** segments and is produced only on the shallow areas of the model.
Further improvements can be made to this toolpath with respect to the **Links** between toolpath tracks. At the moment they are all at **Safe Z**.

Select the **Leads and Links** icon from the top of the screen.



Select the **Links tab** and change the **Short Links** to **On Surface**, the **Long** and **Safe Links** to **Skim.**
Apply and **Accept** the form.

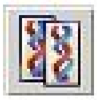


The toolpath has now is now more efficient with the rapid moves at **skim** height and the **short links** being forced onto the surface.

Simulate the **3D Offset** toolpath in **Viewmill**.

Right click over toolpath **3DOffsetBN10** in the **explorer** and select **Settings** from the pull down menu.

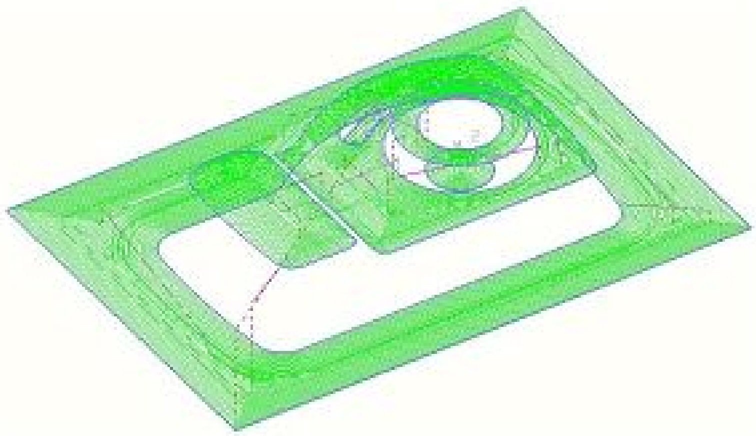
Select the **Copy** icon on the form.



Rename the toolpath as **3DOffsetBN10_Spiral**.

Tick the box **Spiral**.

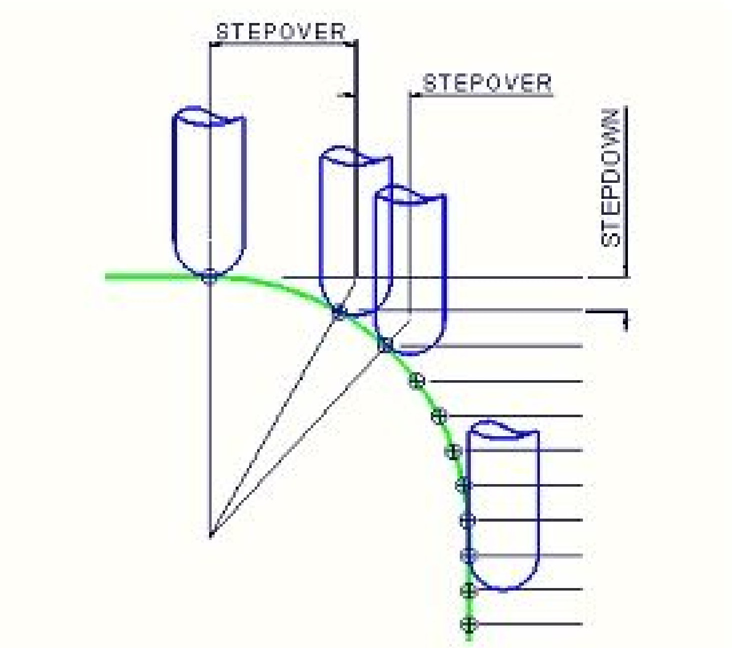
Leave all other values the same then **Apply** and **Cancel** the form.



The **Spiral** option is ideal for HSM (High Speed Machining) applications where ideally toolpaths should be as continuous as possible with the minimum sudden changes direction. **Leads** and **Links** are greatly reduced using this method.

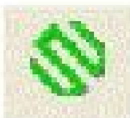
Constant Z machining

Constant Z machining projects each tool track horizontally onto the component at fixed heights defined by the **Stepdown**.

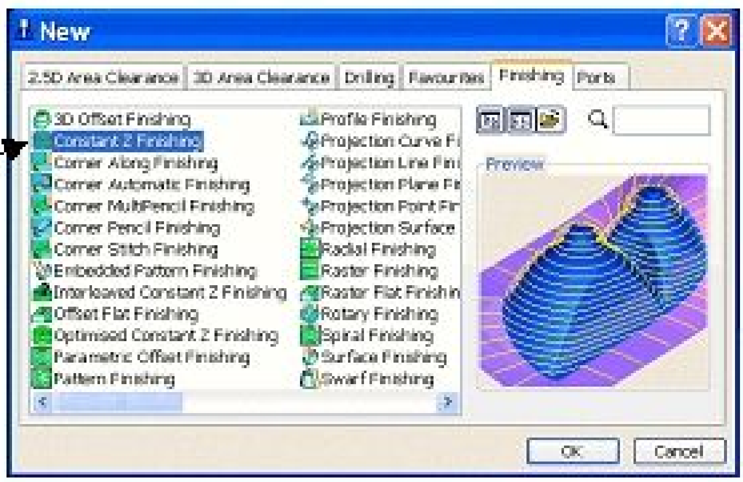


As the component surface becomes shallow the actual tool step over increases until it becomes non-existent on flat areas.
It is possible within the **Constant Z** finishing form to apply a variable stepdown by applying a **Cusp tolerance** in conjunction with a **max** and **min stepdown**. While this will generally provide a more consistant stepover relative to the angle of the model it will not help at all for very shallow or flat areas.

Select the **Toolpath Strategies** icon from the top toolbar.



Select the **Constant Z Finishing** strategy.



Enter a **Name** **ConstantZBN10**.

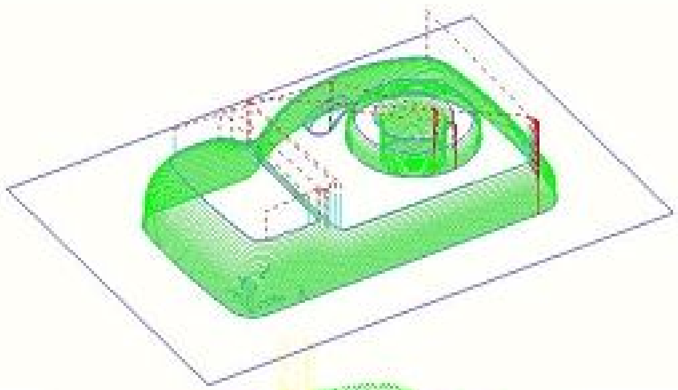
Set a **Stepdown** of **1**.

Enter the **Tolerance** as **0.02**

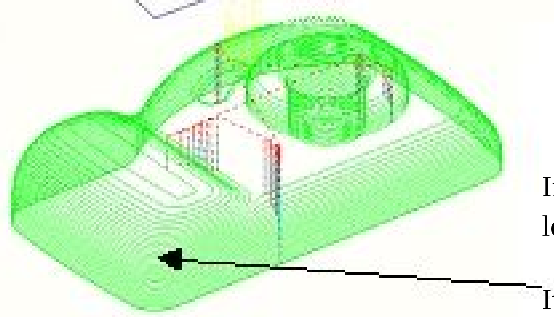
Set the **direction** to **Climb**.

Set the **Trimming** to **Keep Outside**.

Apply then **Cancel**.



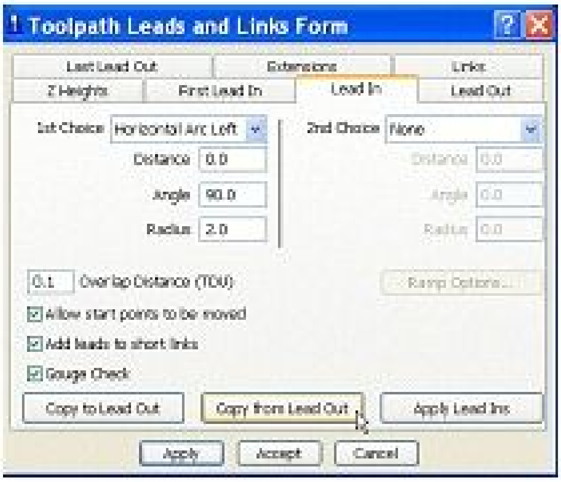
By using the **Boundary Trimming** option **Keep Outside**, the toolpath is correctly limited to the steep areas of the model.

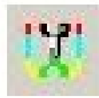


If the **Boundary** had not been used the toolpath would have looked something like this.

It can be seen that the parts of the toolpath on the shallow areas have an excessive **Stepover**.

To further improve the new toolpath the **Leads and Links** will be modified.





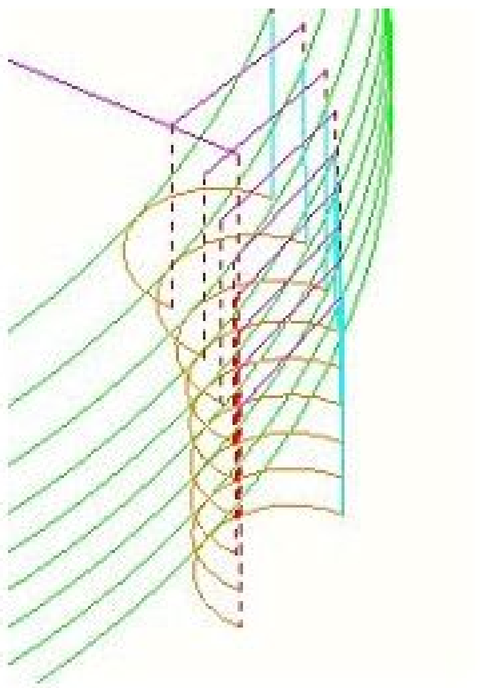
Select the **Leads and Links** icon at the top of the screen.

Select the **Lead In tab** and change the **1 st Choice** to **Horizontal Arc Left**, **Angle 90.0** and **Radius 2.0**.

Click the button **Copy to Lead Out**.

Select the **Links tab** and change the **Short, Long** and **Safe** to **Skim**.

Apply and **Accept** the form.



The tool now **leads in** and **out** of the toolpath with a **horizontal arc**.
If the tool lifts it will only lift by the **skim** distance taken from the first page of the **Leads and Links** form (**Z Heights** tab).
The **Rapid** moves at **skim** height are purple in colour while the **plunge** moves are light blue.

Simulate the toolpath in **Viewmill**.

Other Constant Z Options

Corner Correction to control whether changes of toolpath direction are **Arc fitted**, **Sharpened** or unchanged (**None**).

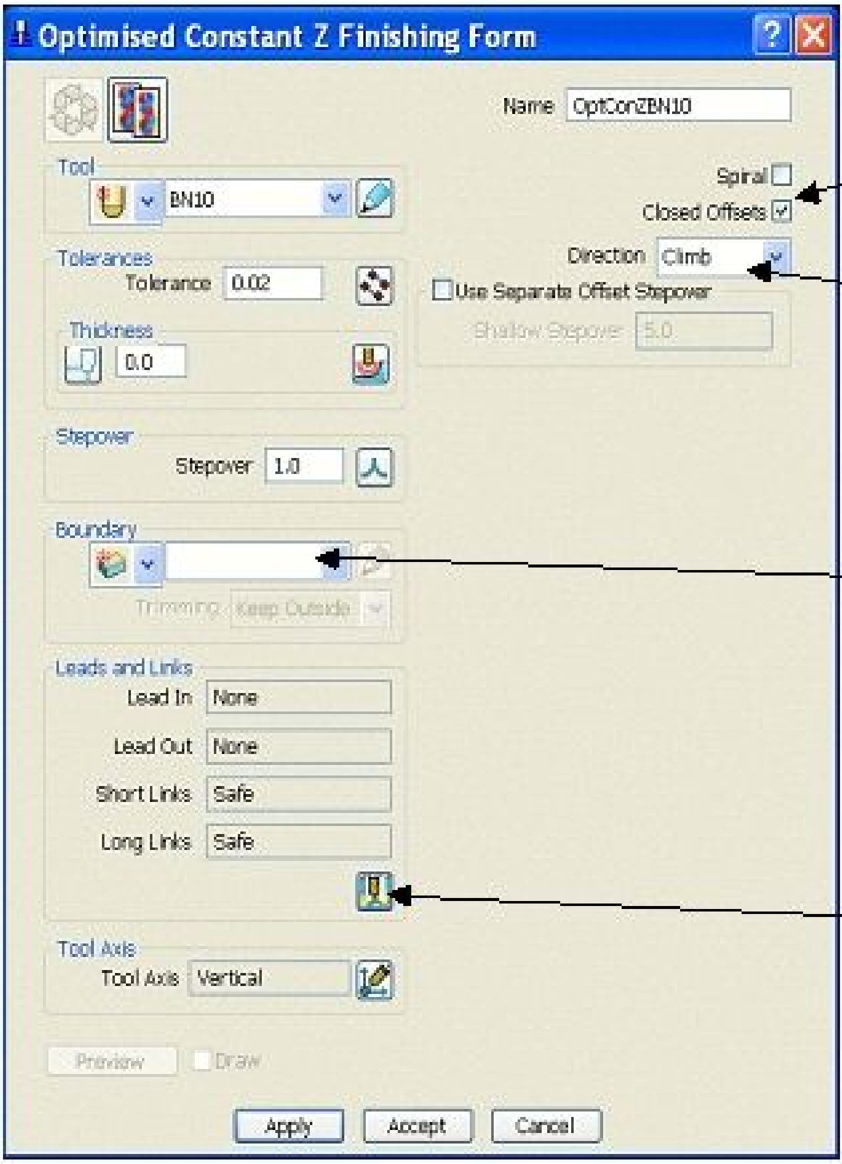
The **Pocket Machining** option if ticked will allow independent machining of local features such as pockets completely from top to bottom. If unticked the whole area of the component will be machined at each separate **stepdown** height.

Optimised Constant Z Machining

This strategy is a mixture of Constant Z and 3D Offset machining. Where the model is steep, Constant Z is used and for other areas, 3D offset is used.
Closed Offsets if ticked will cause the **3D Offset** areas of the machining to be ordered to occur from outside to inside. The reverse applies if **Closed Offsets** is unticked.

Deactivate the **Shallow boundary**, **ShallowBN10** in the **explorer**.

Select the **Toolpath Strategies** icon from the top of the screen.
Select **Optimised Constant Z** from the **Finishing** strategy form.



Enter **Name** - **OptConZBN10**.

Select **Closed Offsets**.

Set the **Direction** to **Climb**.

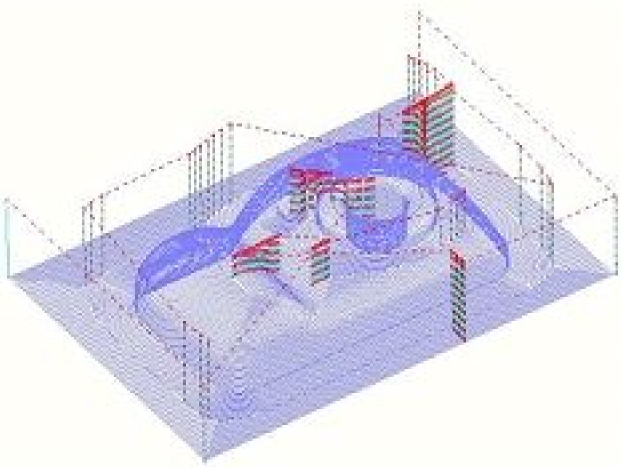
Input a **Stepover** value of **1**.

Make sure no **boundary** is selected.

Enter the **Tolerance** as **0.02**.

Reset the **Lead In** and **Lead Out** to **None**.

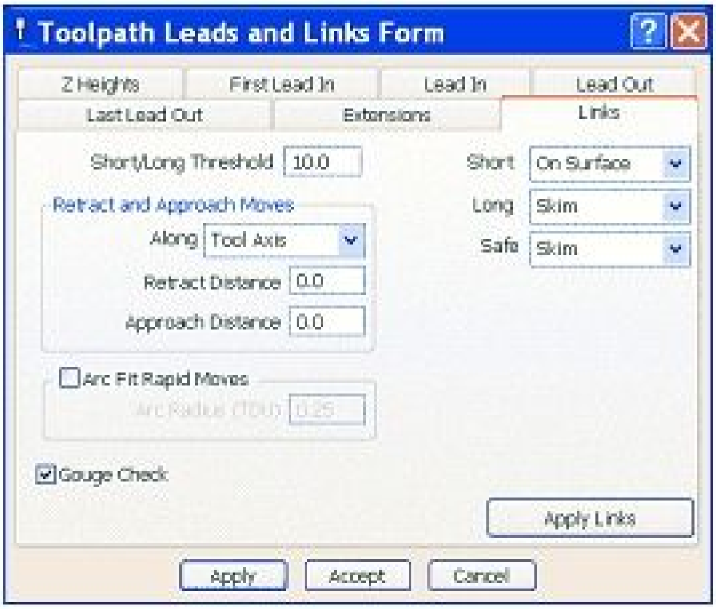
Apply and **Cancel**.



Note the consistent **Stepover** between tool tracks across the whole component.

Optimized Constant Z has performed well in this example but it does take longer to calculate. Sometimes it is better to use **Boundaries** with a combination of **3D Offset** and **Constant Z**.

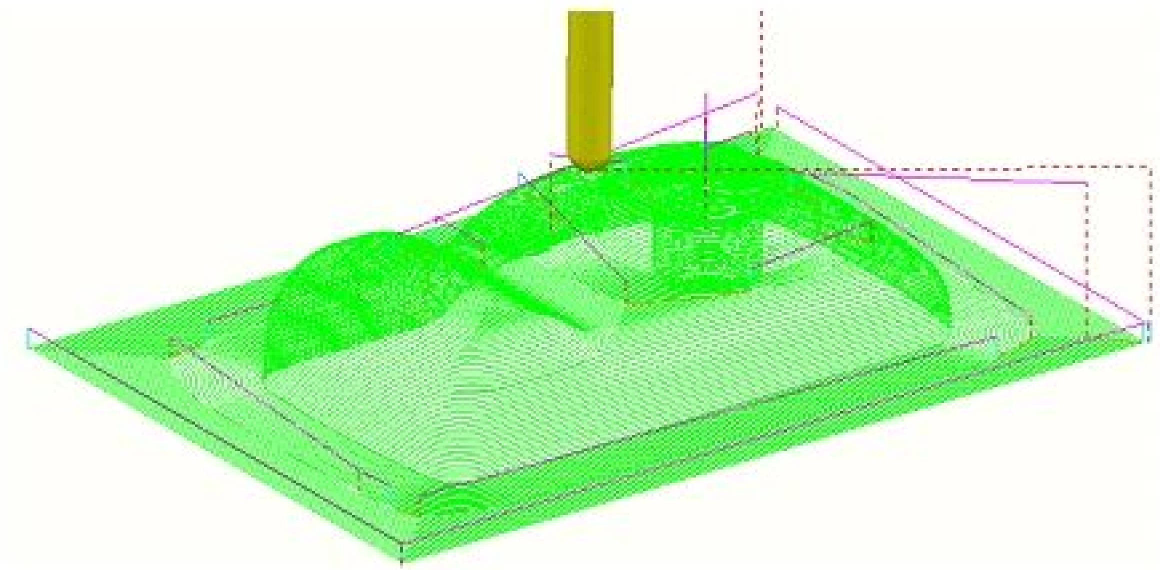
Select the **Leads and Links** icon from the top of the screen to bring up the form and select the **Links** tab.



Change the **Short** links to **On Surface**.

Change the **Long** and **Safe** links to **Skim**.

Apply and **Accept** the form.



Both the **Constant Z** and **3D Offset** parts of the toolpath currently use a **1mm Stepover**. By ticking the box **Use Separate Offset Stepover** it is possible to apply a different, larger **Stepover** value to the shallow areas created with the **3D Offset** strategy used in this hybrid form.

Interleaved Constant Z

This is a form of Optimised Constant Z finishing with additional options which include a user defined **Threshold Angle** between the Constant Z and 3D Offset areas of the toolpath as well the option to specify an **Overlap** value between them.

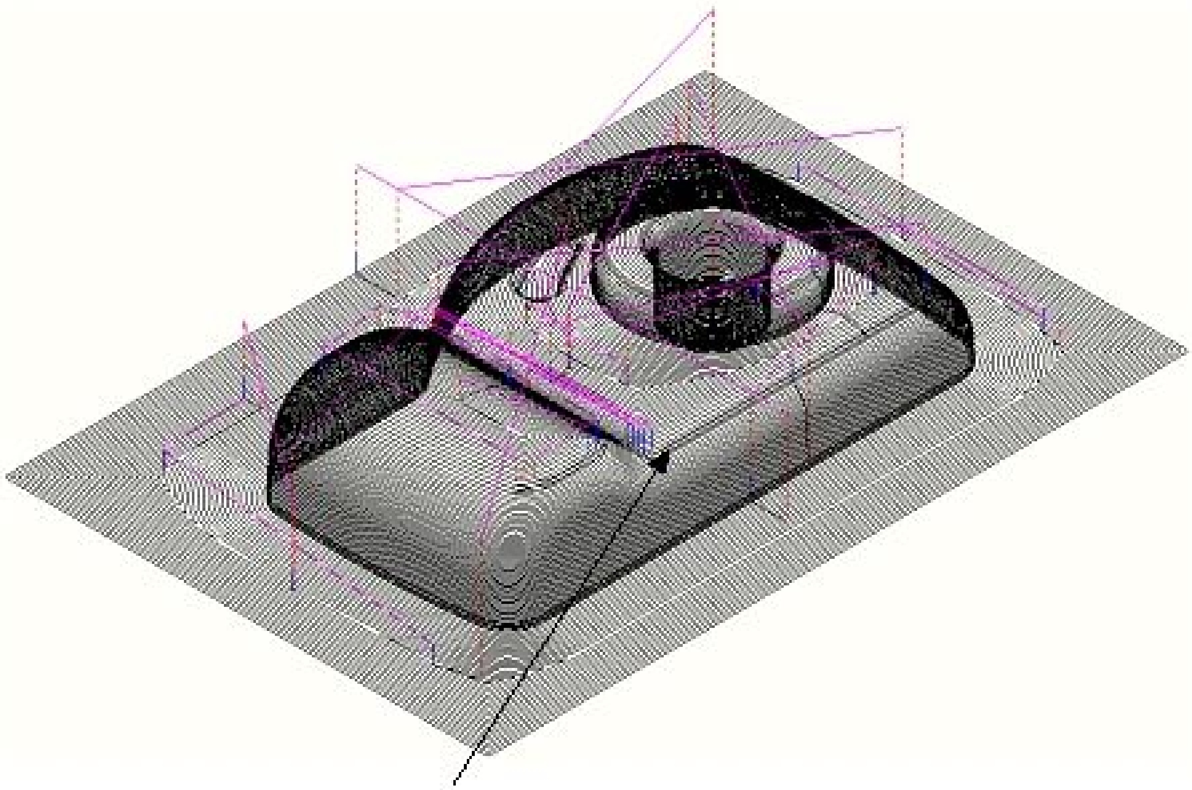
Select **Interleaved Constant Z** from the **Finishing** strategies form.
Enter data exactly as shown in the following illustration.

Enter **_Name_** as **InterleavedCZ_BN10**.

Input a **_Threshold Angle_** value as **25**.

Input an **_Offset Overlap_** value of **2**.

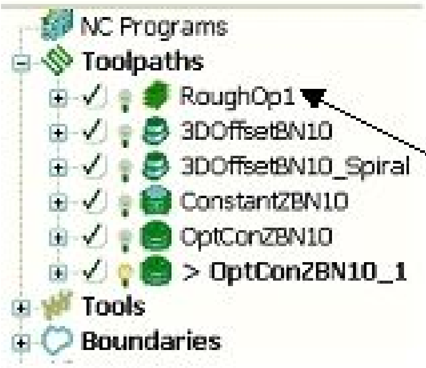
Apply and **Accept** the form.



The **Threshold Angle** and **Overlap** are specified by the user.

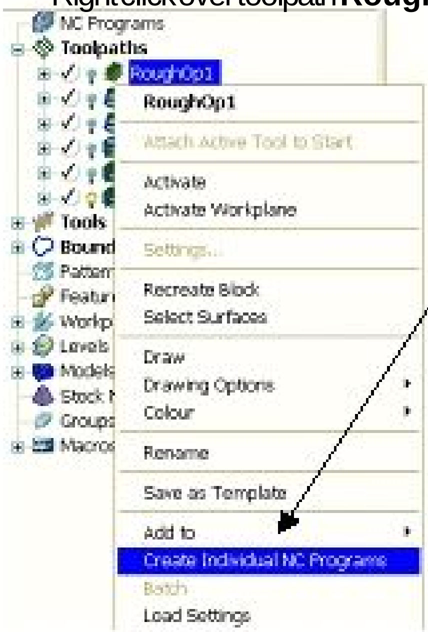
Introduction to NC Programs.

At this stage we will start looking at post processing a single toolpath from the explorer as an introduction to outputting NC Programs. NC Programs will be covered in more detail later in the course.



All of the toolpaths that have been created in this chapter should appear in the explorer like this.
This introduction will concentrate purely on the output of one single toolpath - **Rough Op1**.

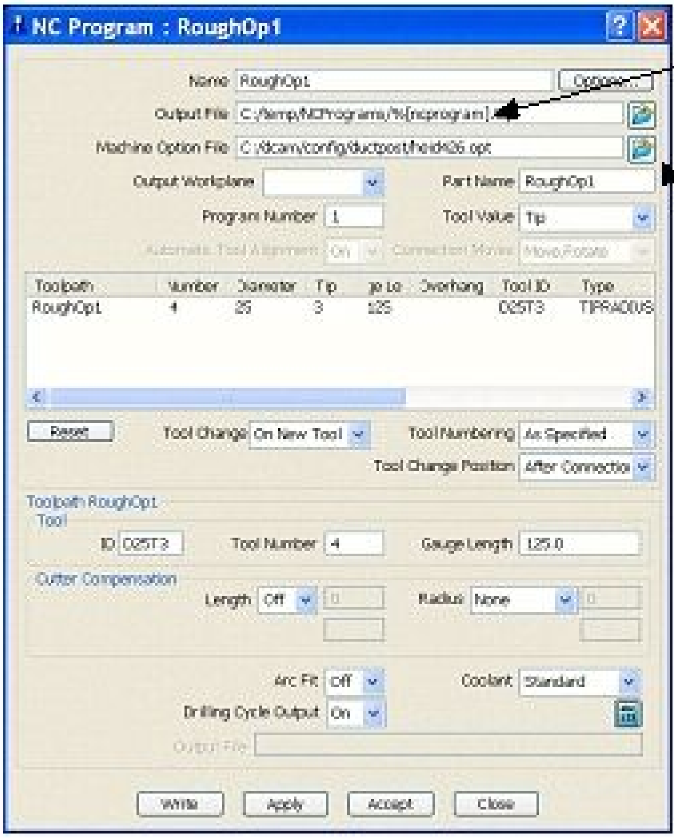
Right click over toolpath **Rough Op1** in the explorer.



Select the **Create Individual NC Programs** option.

An **NC Program** is created called Rough Op1 and contains the toolpath.

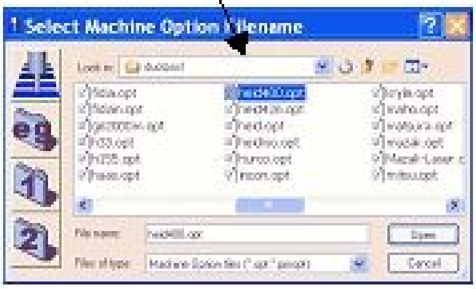
Right click over the NC Program and select **Settings**.



The path to where the program will be output.

Before post processing can occur the required option file (*.opt) must be selected.

Select the folder icon to open up the **Select Machine Option Filename** form.

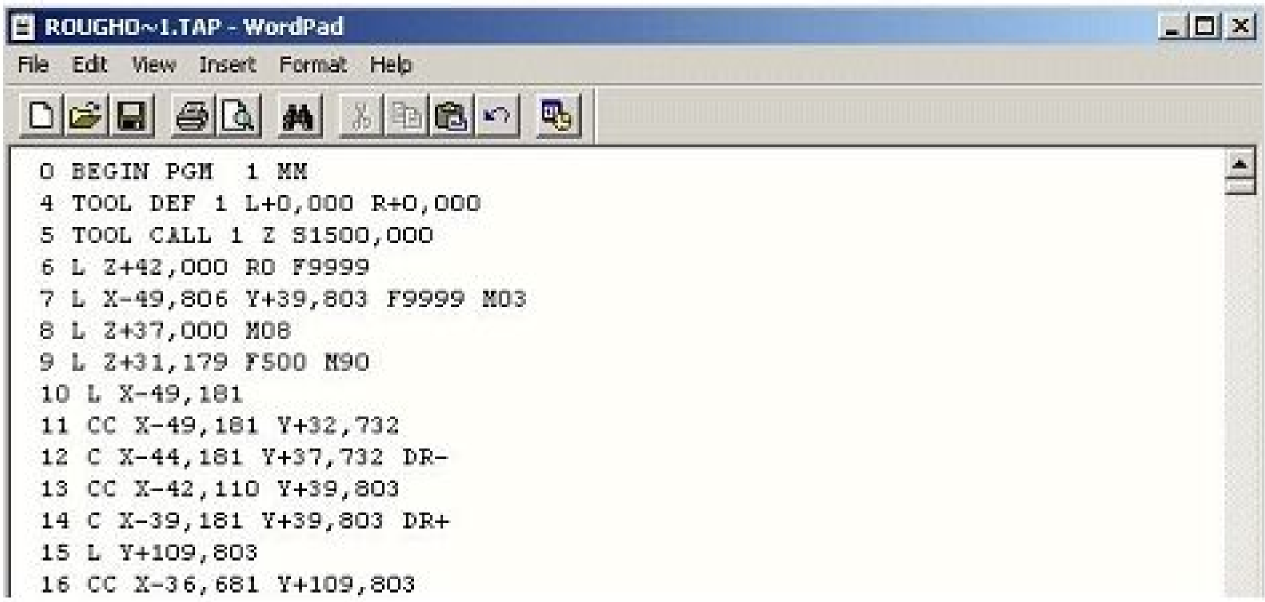


Select the **Heid400.opt** and then **Open**.

Select **Write** at the bottom of the **NC Program** form.

Close down the subsequent form, which confirms the output using.

The contents of the NC Program can be viewed by double clicking on it in the C:\ NC Programs folder and view it in WordPad.




```
0 BEGIN PGM  1 MM
4 TOOL DEF 1 L+0,000 R+0,000
5 TOOL CALL 1 Z S1500,000
6 L Z+42,000 R0 F9999
7 L X-49,806 Y+39,803 F9999 M03
8 L Z+37,000 M08
9 L Z+31,179 F500 M90
10 L X-49,181
11 CC X-49,181 Y+32,732
12 C X-44,181 Y+37,732 DR-
13 CC X-42,110 Y+39,803
14 C X-39,181 Y+39,803 DR+
15 L Y+109,803
16 CC X-36,681 Y+109,803
```