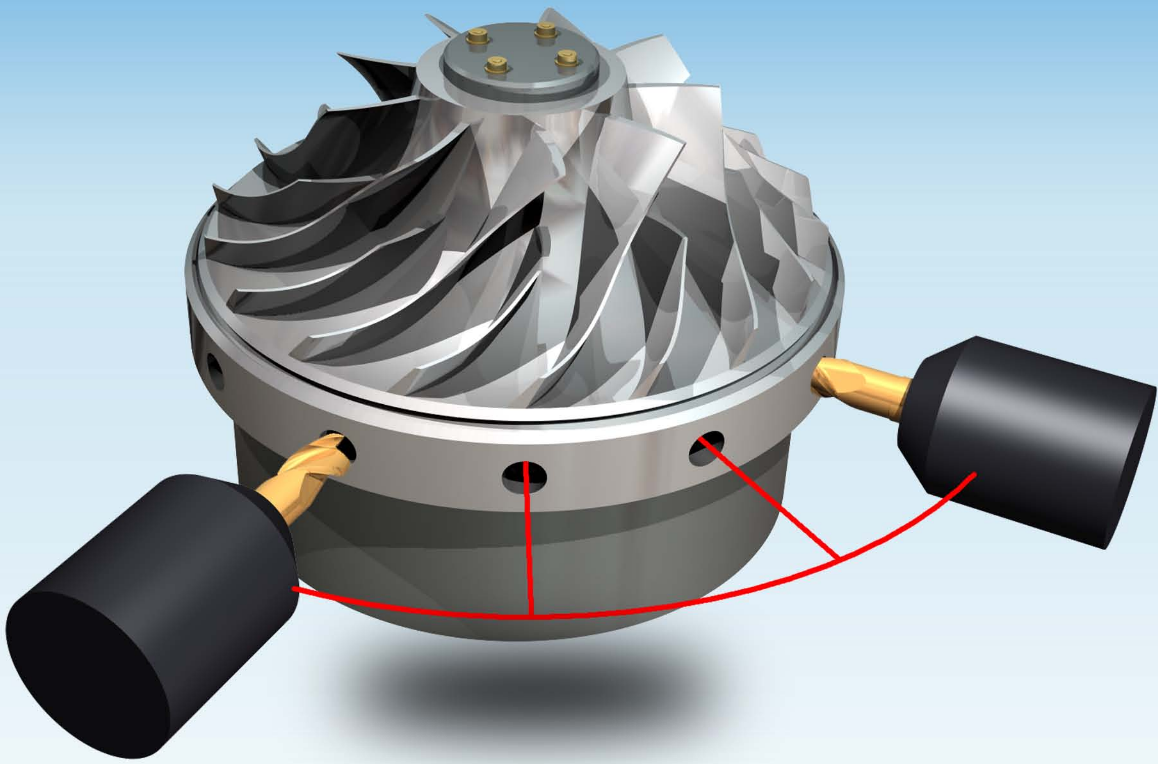
Delcam PowerMILL Training Course



Delcam

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

1. Getting Started

Introduction.

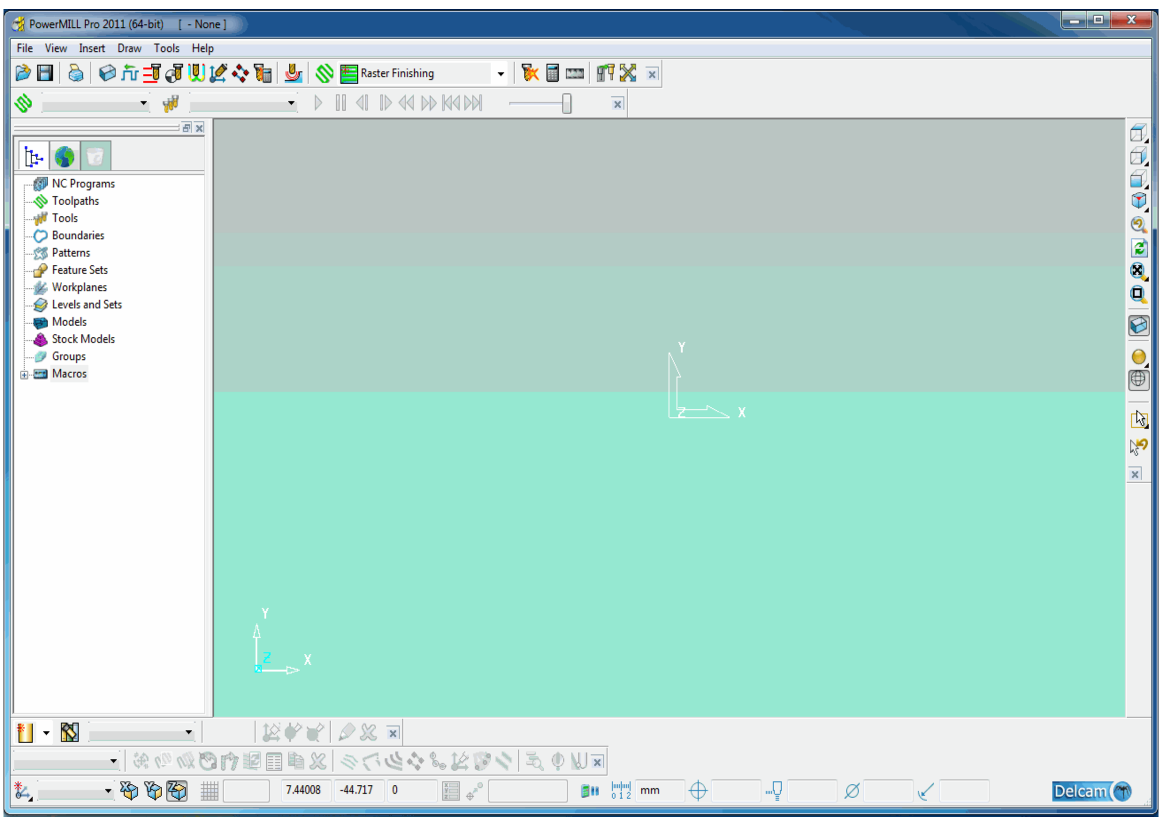
This course covers the 3-Axis functionality available in **PowerMILL**. The additional features available with **PowerMILL–Pro** and **Five Axis** licenses are covered in separate modules. **PowerMILL** will quickly create gouge free cutter paths on imported component data. It supports **Wireframe**, **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 also directly import data created by other of proprietary CAD packages.

Start PowerMILL

- Double click the relevant **PowerMILL** shortcut **icon** on the desktop:

Note:- On the training pc the **icon** will be displayed as **PowerMILL** .

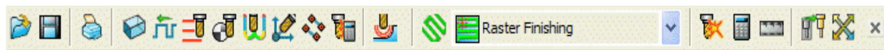
The following screen is then displayed:



The screen is divided into the following main areas:

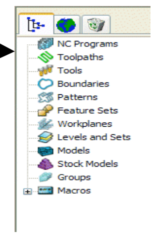
1) Menu Bar – 

Clicking one of the menu names on this bar (for example, **File**) opens a pull-down list of associated commands and sub-menus. A sub-menu is indicated by a small arrow to the right of the text (for example **File - Recent Projects >**). Highlighting this arrow generates a list of commands/names specific to that sub-menu (for example, **File - Recent Projects** displays a list of recently opened projects that will open directly when clicked).

2) Main Toolbar – 

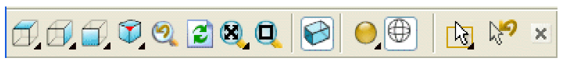
This provides quick access to the most commonly used commands in **PowerMILL**.

3) Explorer –



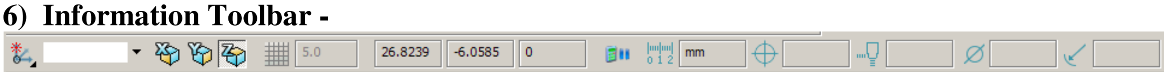
The **Explorer** provides control options and storage of **PowerMILL** entities created during the session.

4) Graphics Window – This is the large, visual display area to the right of the **Explorer** (Look at the illustration on previous page).

5) View Toolbar – 

Provides quick access to standard view and shading options in **PowerMILL**

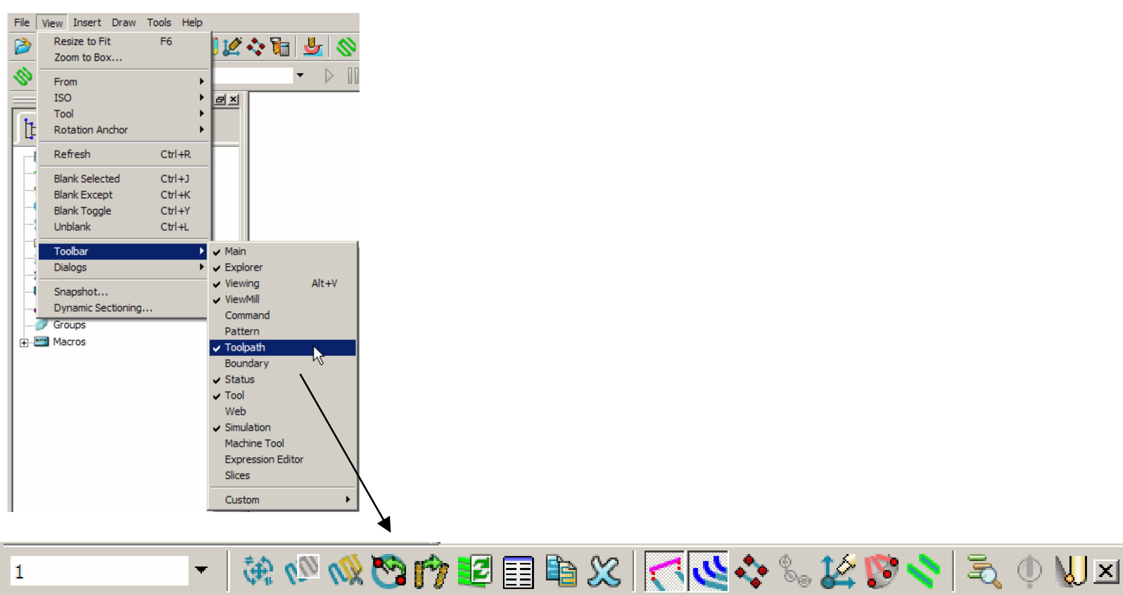
6) Information Toolbar -



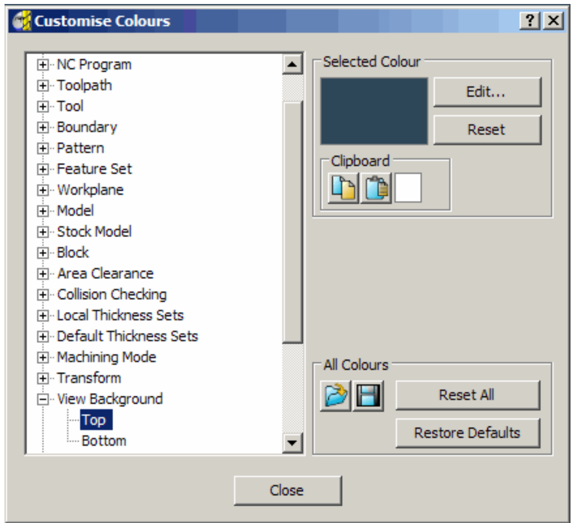
This area provides a reminder of some of the active setup options.

Tool Toolbar - facilitates the rapid creation of tools in **PowerMILL**.

The other toolbars are not factory defaults, and are therefore not shown at initial startup. To display any of these, select using the relevant option under **View - Toolbar**, for example **View - Toolbar - Toolpath** to display the **Toolpath Toolbar**:



To change the background colour of the graphics area, select **Tools - Customise Colours** and select **View Background**. The **Top** and/or **Bottom** colours can be changed independently and **Reset** using **Restore Defaults** to restore to the original settings:



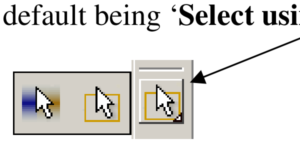
PowerMILL remembers **Toolbar** and **colour** selections from one session to the next, for example, if the **Toolpath Toolbar** is open when the session is closed, it will appear the next time that **PowerMILL** is opened.

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.
The method of selection is controlled by 2 options accessed from the **Viewing** toolbar the default being '**Select using a Box**'



Select using a box 

If the ***cursor*** is positioned on an entity, such as part of a **surface** model and the ***left mouse key*** is pressed, then the item will turn ***yellow*** signifying that it has been selected.

If the same process is applied to another **surface**, all currently selected items will be de-selected.

If the **Shift** key is held down during the process then the new selection will be added to the original selection.

If the **Ctrl** key is held down while clicking on a **surface** it will be removed from the total selection.

Select by dragging the cursor 

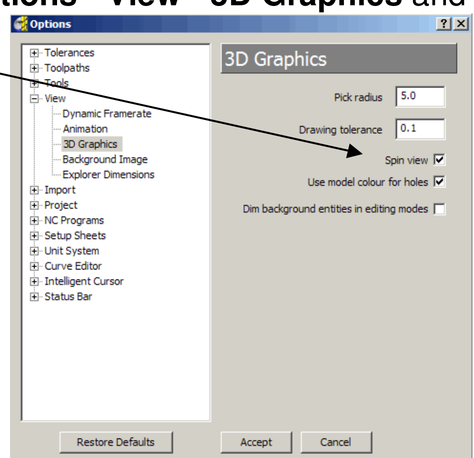
If this option is selected then multiple, selection of entities will occur by simply ***dragging*** the cursor across the required items. This is ideal for quick selection areas of the model consisting of multiple surfaces. To ***deselect*** the **Ctrl** key is depressed while dragging across an entity.

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 movement, the faster it will spin. This feature is switched off by default.

- Select **Tools -> Options - View - 3D Graphics** 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.

HELP!

PowerMill provides the user with help in a number of ways:

1. **Tooltips**. Place the mouse cursor over a menu button. A box will appear containing a description of what that button does.

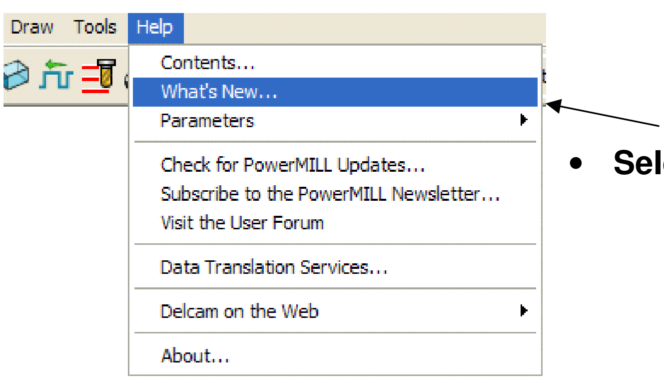
2. **Online Help**. Select **Help>Contents…** from the **Main Toolbar**, to access the online help documents. There is a full index and search facility provided.

3. **Context Sensitive Help**. Pressing the F1 key will display the help page for the currently active form. Clicking on the ? button in the top right hand corner of the form, followed by a left click in any of the input fields will focus on the help topic for that part of the form.

4. **PowerMill User Forum.** On any PowerMill PC with an Internet connection, selecting **Help** from the main toolbar, followed by **Visit the User Forum**, will enable you to participate in web based user discussions on **PowerMILL** issues. The forum can also be accessed from any other internet connection by going to **http://forum.delcam.com/**

5. **Telephone and Email Support**. UK customers with up-to-date software maintenance can call 01216831010 or mail support@delcam.com to get help or advice on specific application problems.

PowerMILL contains **On-Line Help** the main access being via the **Help** tab on the main **pulldown** menus.



- **Select Help -> What's New.**

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

Simplified PowerMILL Example

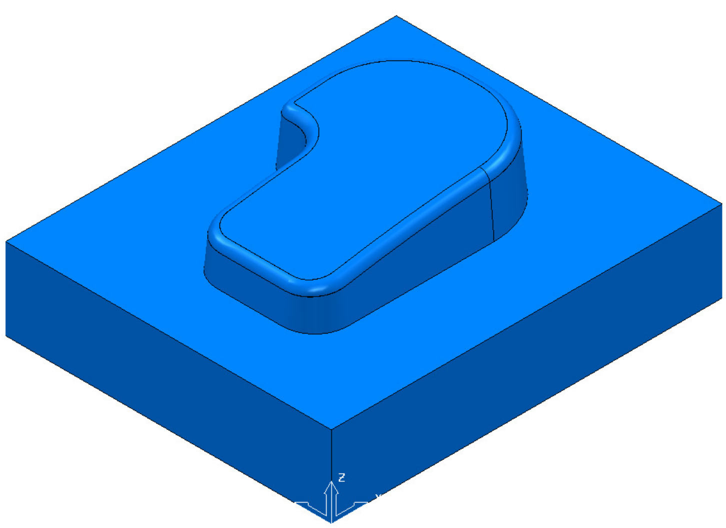
This example provides a quick overview of the machining process. It shows how to create and output a couple of simple toolpaths on a model of a valve chamber (using default settings wherever possible).

The basic procedure is:

1. Start PowerMILL.
2. Import a **Model**.
3. Define the **Block** from which the part will be cut.
4. Define the cutting **Tools** to be used.
5. Define Set up options (**Rapid Move Heights** – **Start and End Point**).
6. Create a **Roughing** Strategy.
7. Create a **Finishing** Strategy.
8. **Animate** and **Simulate** the toolpaths.
9. Create an **NCProgram** and output as a post-processed ncdata file.
10. **Save** the **PowerMILL Project** to an external directory.

Import a Model

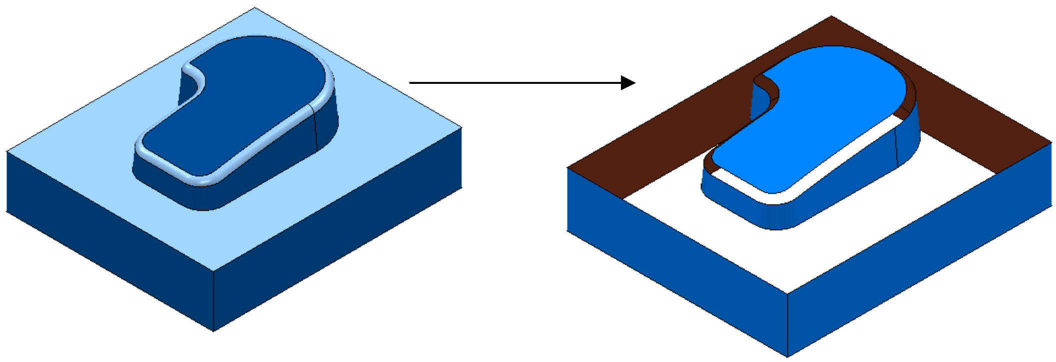
- From the **main pulldown** menus select, **File - Import Model** and browse for the model file:-
  D:\users\\training\ PowerMILL_Data\Models\PmillGettingStarted.dgk.



Blanking of Model entities

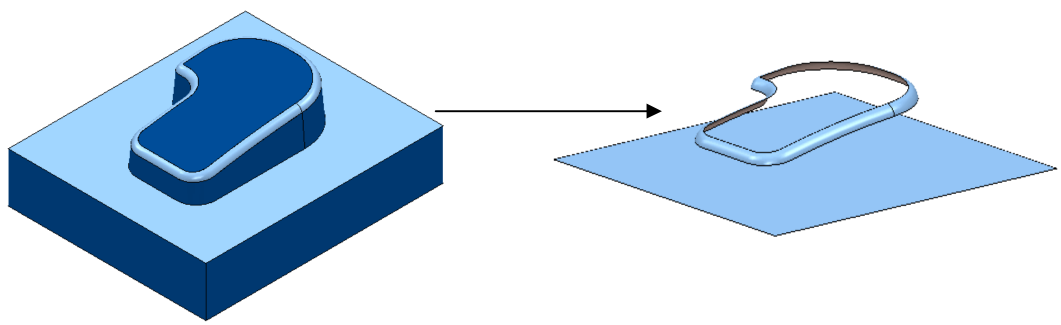
This provides the user with a quick and simple way to control which individual, model entities are displayed. In the illustrations below, the ***light blue*** surfaces are **selected**.

If one or more **surfaces** are selected they can temporarily be removed from the graphics area by using the **Blank Selected** option (**Ctrl J**) in the *local* **Model** *menu* (accessed by right clicking on a **surface**).



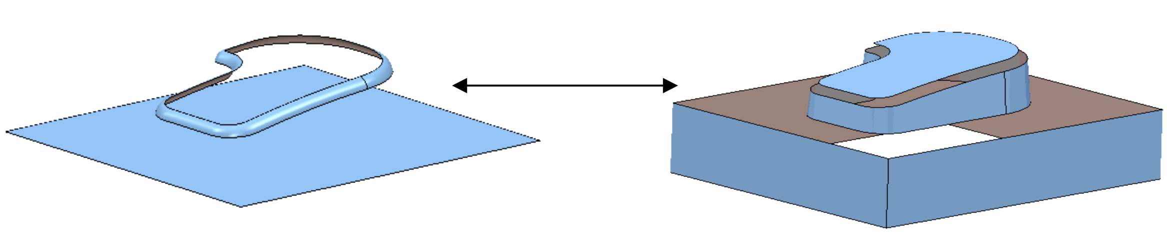
If one or more surfaces are selected all others can be removed from the graphics area by using the **Blank Except** option (**Ctrl K**) in the local menu.

The 2 selected items are light blue.



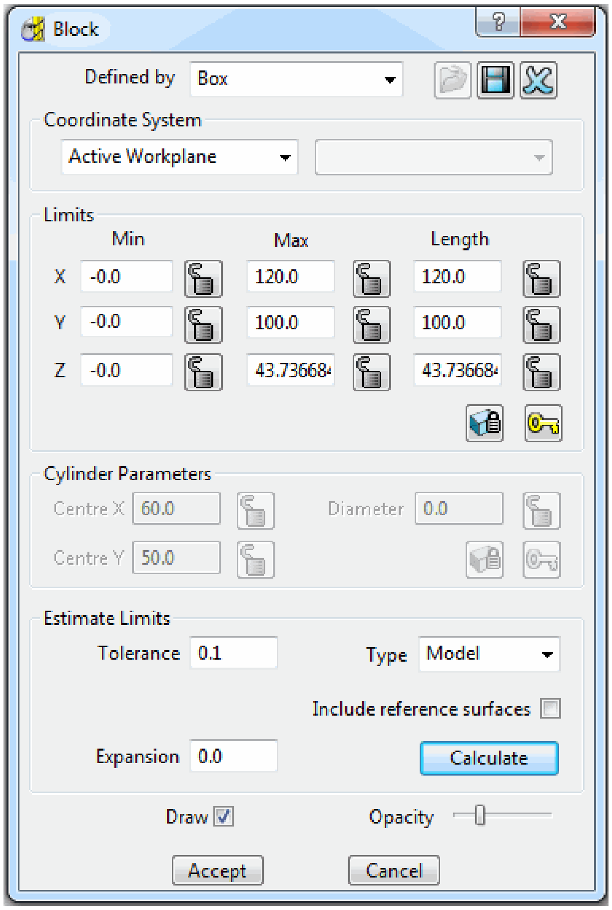
To return all ***Blanked*** items back to the graphics are the **Unblank** option (**Ctrl L**) is selected in the **Default** menu (accessed by right clicking in the graphics area).The **Blanked** items are returned to the graphics area and become selected (back to the left hand image above).

Also accessed form the **Default** menu is the **Blank Toggle** option (Ctrl Y) which if selected will switch the **Blanked** and **Unblanked** items to the other status.



Definition of the Block

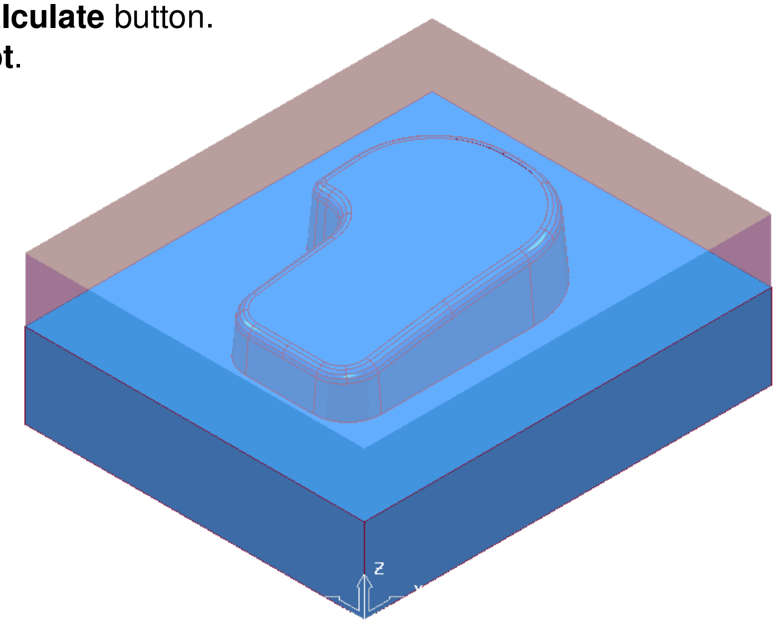
- Click on the **Block** icon  on the top toolbar.



The **Block Form** is used to define the 3D working limits. This could be the actual raw material size or a user defined volume, localised to a particular part of the component.

The **Block Form** default is **Defined by** - **Box** around the model dimensions on clicking the **Calculate** button. Individual values in the form 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**.

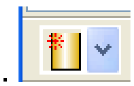
- Click on the **Calculate** button.
- Click on **Accept**.

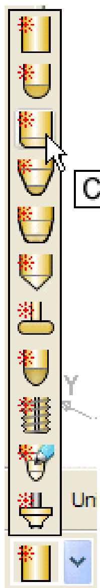


Cutting Tool definition

The **Tool definition** forms are accessed from the icons accessed from the **Tool toolbar** located to the bottom left corner of the graphics area.
For use with this example, 2 tools will be created, A **Tip Radiused** for roughing out and a **Ball Nosed** for finishing.

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

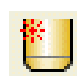


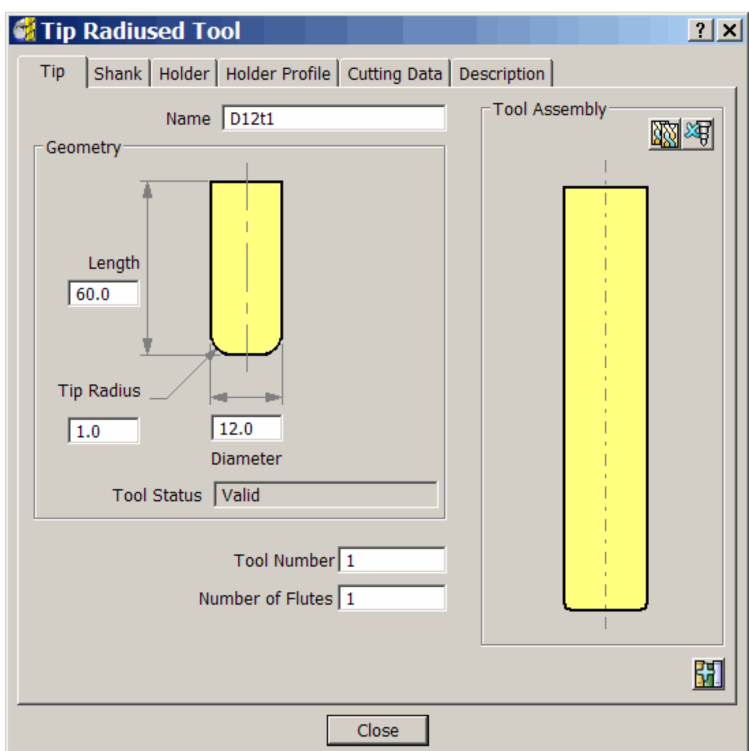


All of the tool types appear as icons.

Placing the cursor over an icon will open a small box containing a description of the tool type (Tool tips). Note the unavailable, greyed out tool definition icons are only available in **PowerMILL Pro**.

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

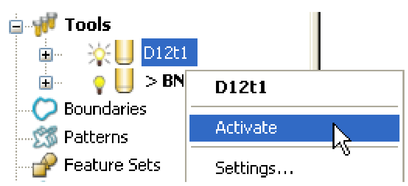




The **Tip Radiused Tool** form opens ready for the user to input the required values. When a diameter value is input the tool length automatically defaults to five times this value. This value can be edited if required.
It is highly recommended to input a more appropriate **Name** for the tool. In this case the tool has been renamed as ***Name*** **D12t1**.
If appropriate, a specified **Tool Number** can be output to the NC program. If the machine has a tool changer this number will represent the location in the carousel.

- Enter a **Diameter** of **12** a **Tip Radius** of **1**.
- Enter **D12t1** in the box marked **Name** before Clicking on **Close**.

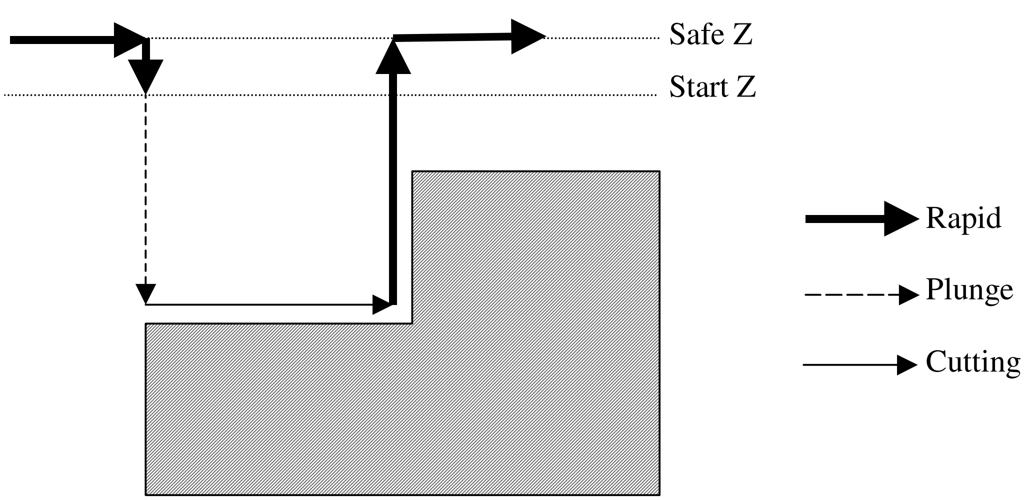
- Repeat the **Tool Definition** operation, this time selecting '**Create a Ball Nosed tool**' and in the form entering a ***Diameter*** **12** with a ***Tool Number*** **2** before and enter the ***name*** **BN12** before Clicking on **Close**.
- In the **explorer** panel on the left of the screen, open the **tools** and right mouse click on the **D12t1** tool to raise the local menu. Select **Activate**.



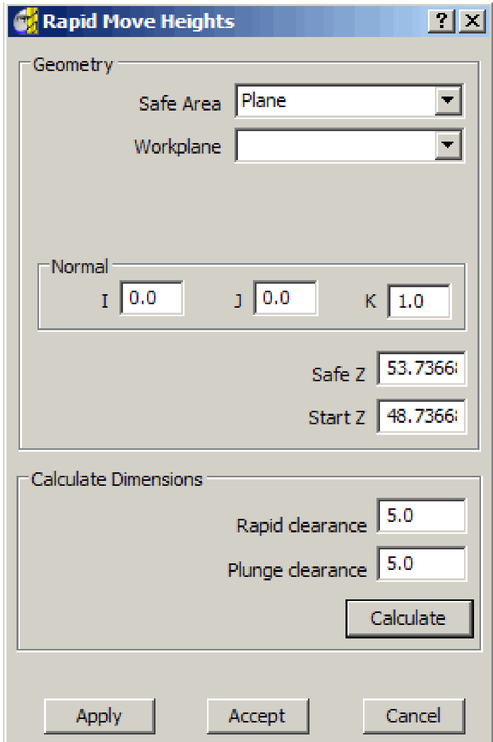
Only one tool can be **Active** at any one time and the word **Activate** in the local menu will be prefixed with a ***tick***. the ***active*** tool will automatically be included in the form when a ***machining strategy*** option is opened.
In the **explorer**, the **Active** tool will be displayed in ***bold text*** and prefixed with '> '.

Rapid Move Heights

The **Rapid move heights** form is essential to allow the user to safely control rapid tool movements across the component. **Safe Z** is the height above the job at which the tool can move at rapid feedrate, clear of any obstructions such as the workpeice or clamps. **Start Z** is the height to which the tool will descend, at rapid feed rate prior to applying the plunge feed rate. **PowerMill** displays rapid moves as dotted red lines, plunge as pale blue and cutting as green.



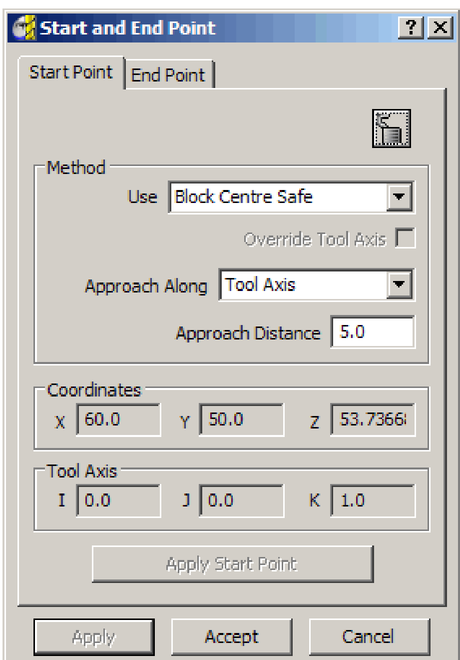
- Click on the **Rapid Move Heights** icon. 
- In the resultant form select the **Calculate** button.
- Click on **Accept**.



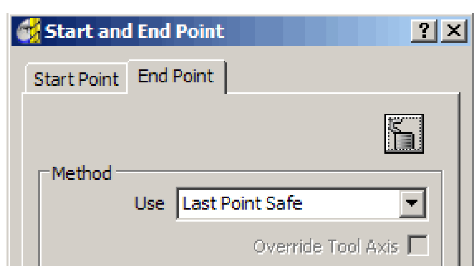
This will automatically set absolute **SafeZ** and **Start Z** values to be above the block by the distance in the incremental height fields shown at the bottom of the form.
An **Absolute** setting will always cause the tool to feed down from the same height.

Tool Start and End Point.

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

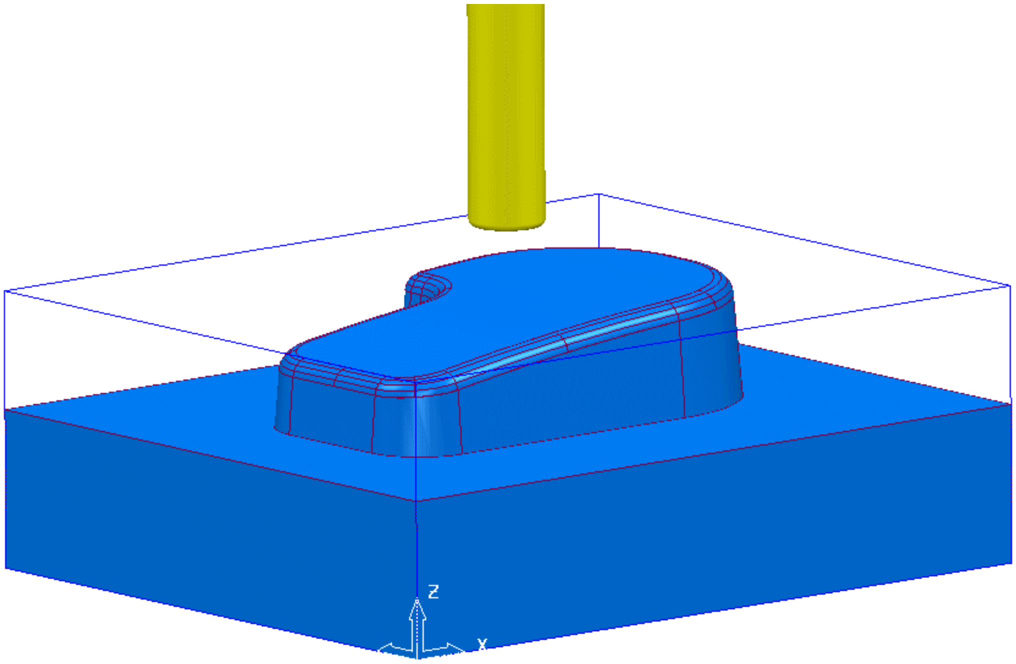


The **Start and End Point** form allows the user to define a position where the tool travels to before and after a machining strategy. By default the tool **Start Point** is set at **Block Centre Safe and the End Point is set to Last Point Safe**.



Other **Start and End Point** definitions are achieved by selecting different options in the ***Method*** area on the form.
These include **Block Centre Safe**, **First/Last Point Safe**, **First/Last Point**, and **Absolute**.

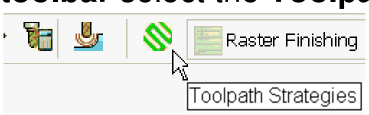
- **Accept** the form with the default settings.



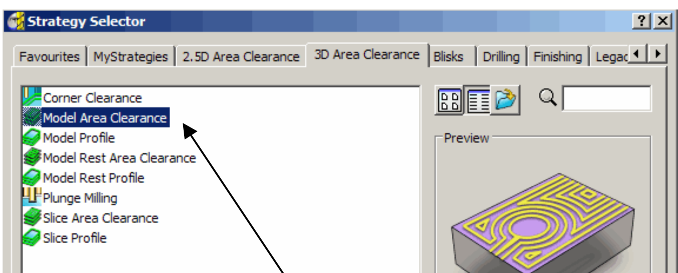
The **D12t1** tool is positioned at the **Block Centre Safe** position ready for the user to create the first toolpath.

Creating a Roughing Strategy

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

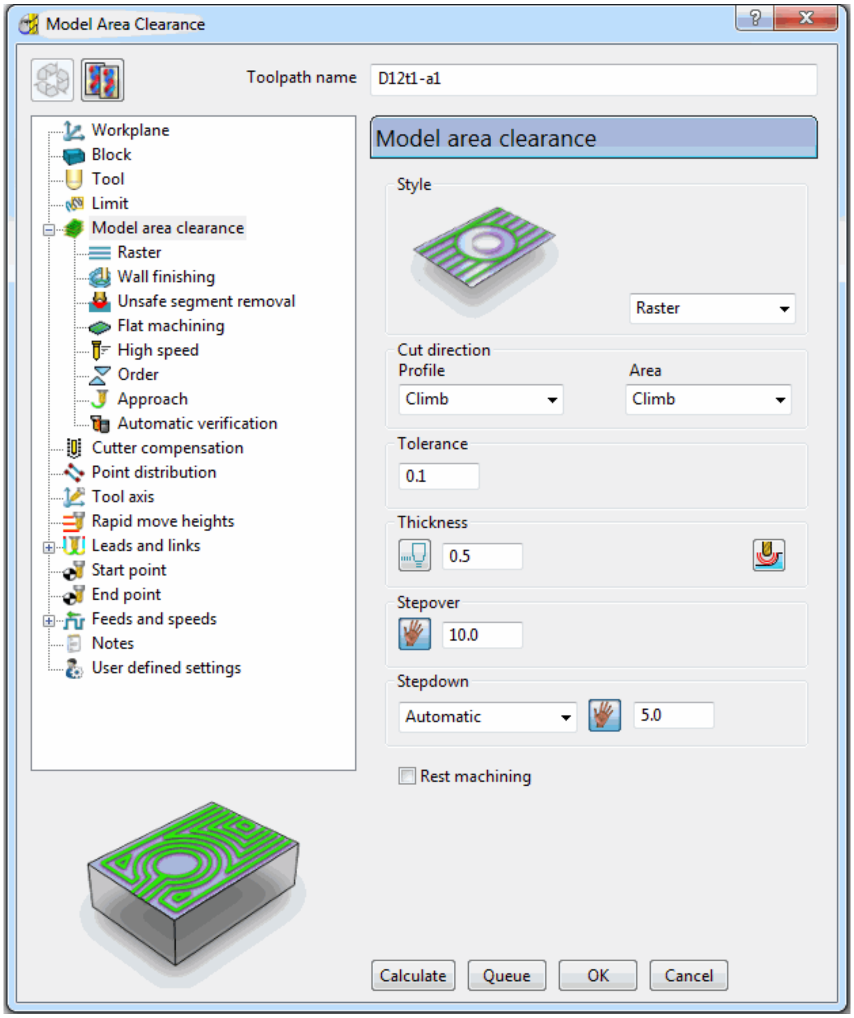


- Select the **3D Area Clearance** tab.



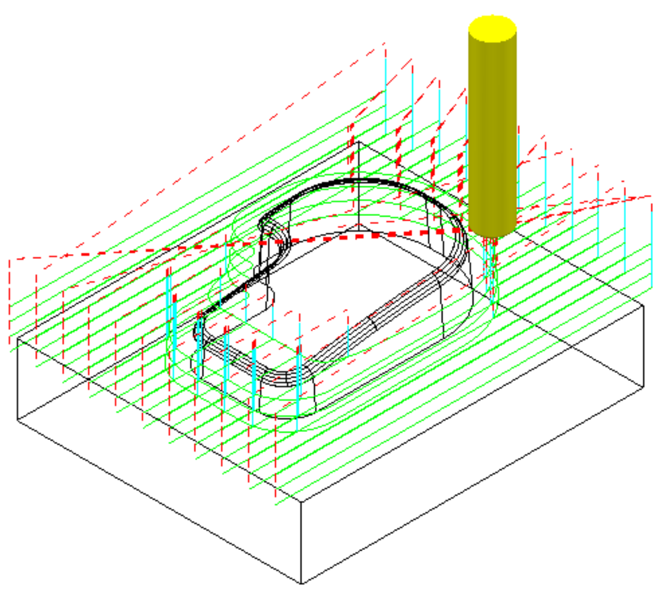
- Select the option **Model AreaClear Model** to open the form shown on the following page.

- Input the new ***Name*** **D12t1-a1** for the **toolpath** that will be created.



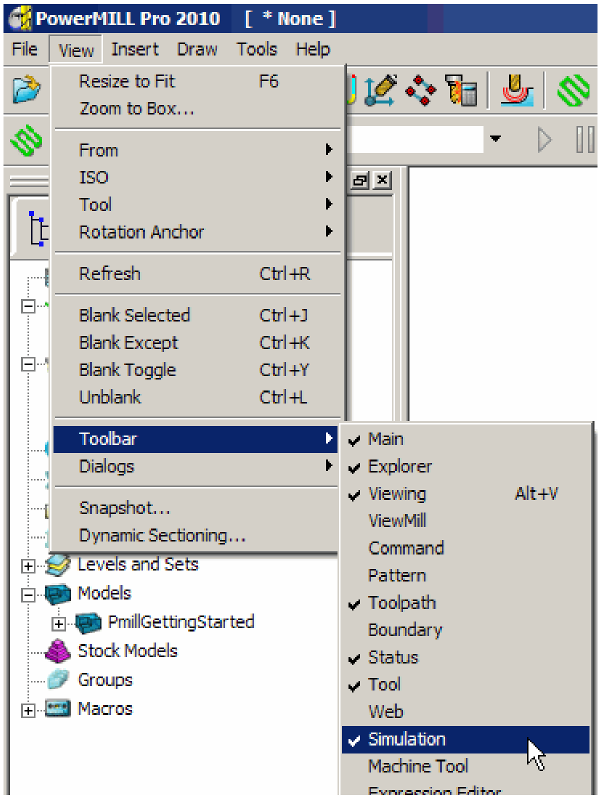
- Select ***Style*** **Raster**.
- Edit the ***Thickness*** value to **0.5**. This is the amount of material that will be left on the job
- Edit the ***Stepover*** value to **10**. This is the distance between each raster pass (the width of cut).
- The ***Stepdown*** value (depth of cut) is left at the default of **5**.

- Click the **Calculate** tab to process the machining strategy.

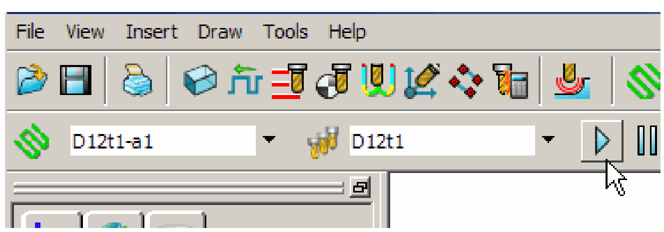


The resultant **3D Raster Area Clearance** can be simulated as follows:

- Raise the **Simulation Toolbar** by selecting **View>Toolbar>Simulation.**



- From the **Simulation Toolbar**, select toolpath **D12t1-a1** in the first field and then click the **Play** button to initiate the simulation.

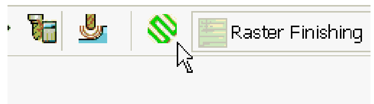


The other buttons on this toolbar can be used to rewind or step through the simulation.

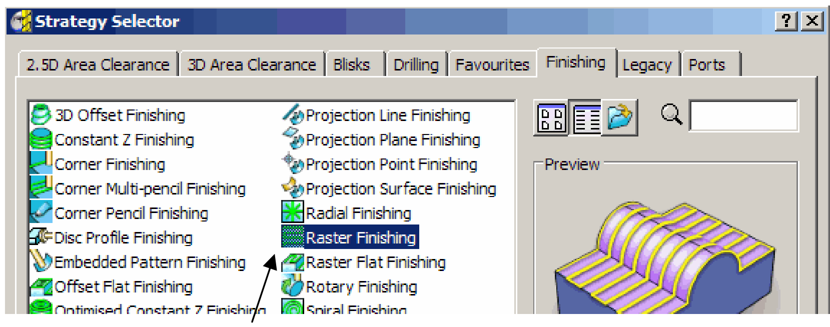
NB. The above strategy has been processed almost completely using the default values, the exceptions being the **Name**, **Thickness** and **Stepover**.

Creating a Finishing Strategy

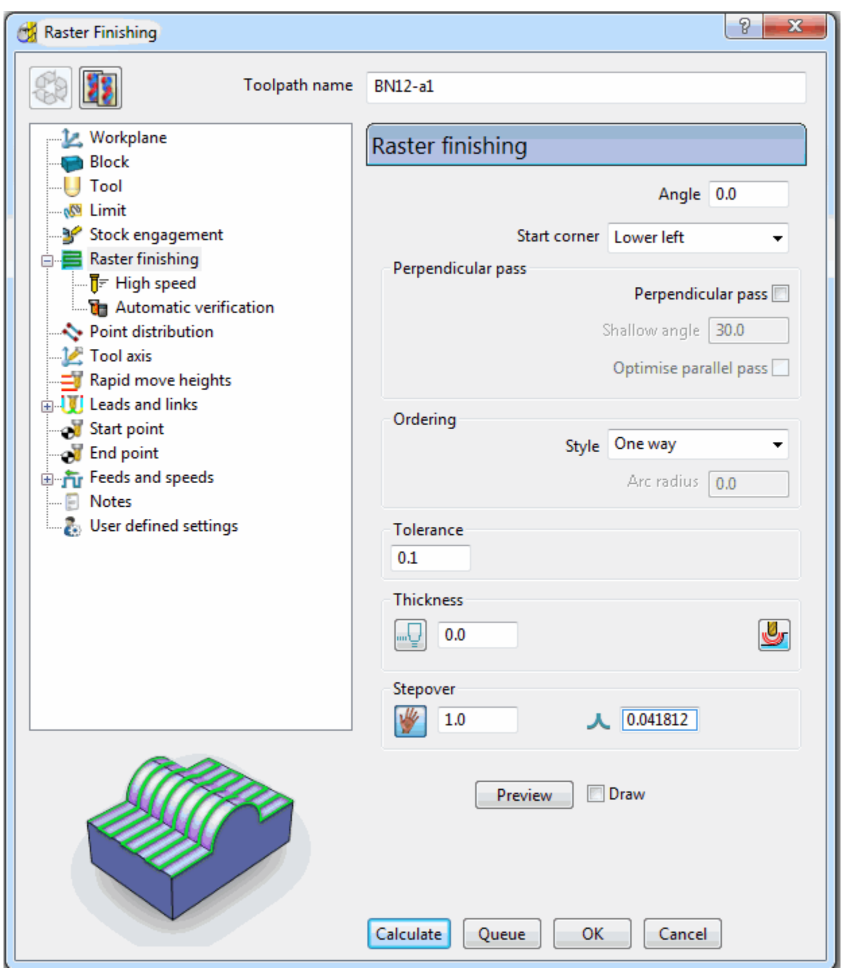
- In the **explorer** right mouse click on the **BN12** tool and in the local menu select **Activate** (or **double left mouse click** on the tool's explorer **icon**).
- From the **Main toolbar** select the **Toolpath Strategies** icon.



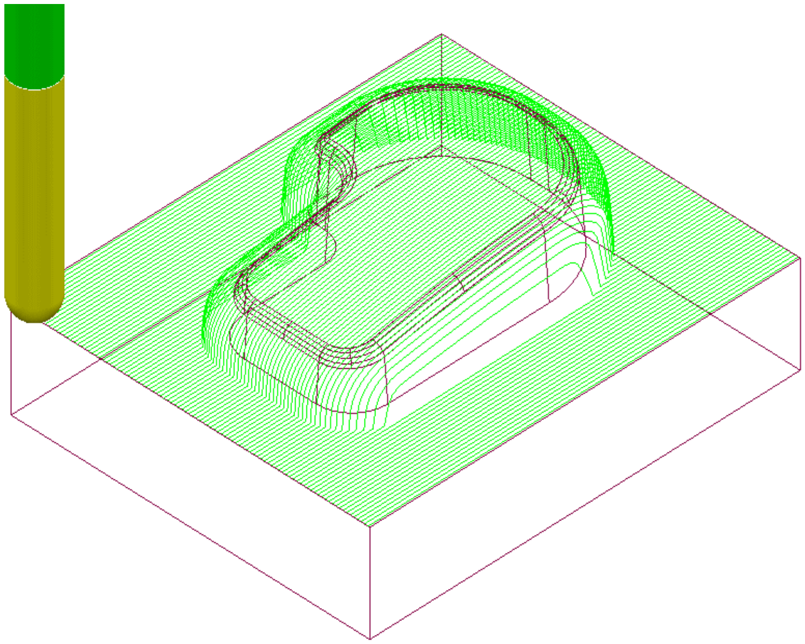
- Select the **Finishing Tab**.



- Select the option **Raster Finishing** follwed by **OK** to open the following form.



- Input ***Name*** **BN12-a1**
- Edit the ***Stepover*** value to **1.0**
- Click the **Calculate** tab to process the machining strategy.



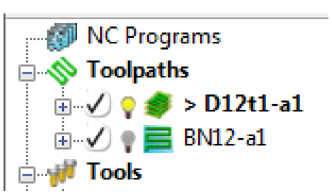
The **Raster Finishing** pattern is projected down Z onto the component taking into account tool geometry and machining settings. Note:- The toolpath link moves, clear of the job are not displayed in this illustration for clarity.

Toolpath Simulation and ViewMILL

PowerMill provides two stages of options for simulating toolpaths. The first simulates the cutting tool as it progresses along the toolpath. The second also includes a shaded image of the stock material being reduced as the tool progresses along the toolpath.

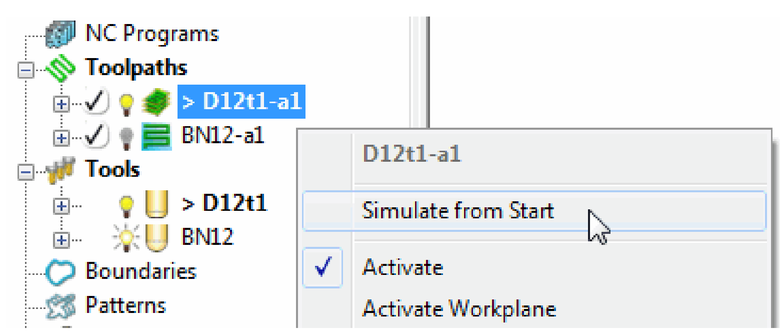
1 – Toolpath Simulation

- In the **explorer** right mouse click on the roughing toolpath **D12t1-a1** and from the pop-up menu click **Activate** to make the toolpath active (ticked).

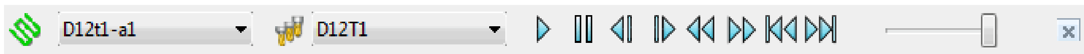


Note: The **Active** toolpath is displayed in bold text and prefixed with a **>** symbol.

- In the **explorer** right mouse click on the roughing toolpath **D12t1-a1** again and from the menu click **Simulate from start**.



- The **Toolpath Simulation** toolbar will be displayed at the top of the screen. This displays the **name** of the **toolpath** and **tool**, together with buttons to control the **simulation**.



The operations performed by each of the buttons are as follows:-

Play - starts the simulation and plays it in continuous mode.

Pause - pauses the simulation.

Step Back - steps the simulation back by tool moves. Click the **Play** button to resume continuous mode.

Step Forward - steps the simulation by tool moves. The faster the speed (defined using **Speed Control**) the bigger the step. Click the **Step Forward** button again to see the next move or click the **Play** button to resume continuous mode.

Search Backward - steps the simulation back to the previous toolpath segment. **Go to End -** moves to the end of the toolpath.

Search Forward - steps the simulation to the next toolpath segment. Click the **Search Forward** button again to see the next component or click the **Play** button to resume continuous mode.

Go to Beginning - moves to the start of the toolpath.

Go to End - moves to the end of the toolpath.

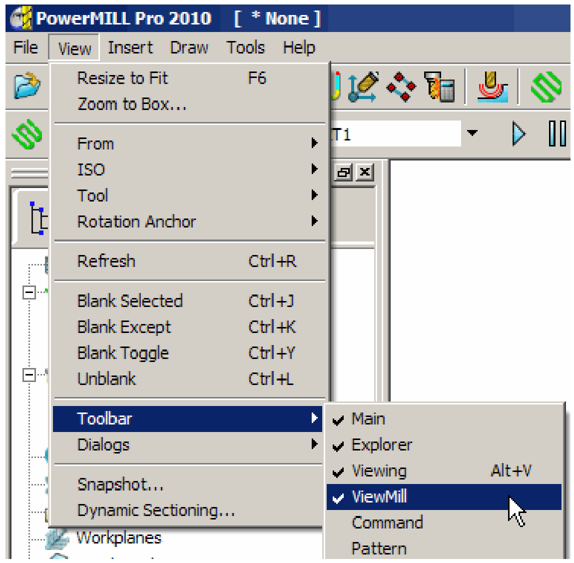
Speed Control - controls the speed of the simulation. The fastest setting is by having the slider at the right, the slowest at the left.

NB. Resting the mouse pointer over any button will also raise a tool-tip describing the button function.

- **Simulate** the **toolpath** by clicking on the **Play** button.
- **Activate** the finishing **toolpath BN12-a1** and repeat the simulation process.
- **Unload** the **toolpath** when complete.

2 – ViewMILL

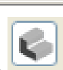
- Activate roughing toolpath **D12t1-a1** and select it in the **simulation toolbar.**
- Raise the **ViewMILL** toolbar by selecting **View>Toolbars>ViewMILL** from the top toolbar.

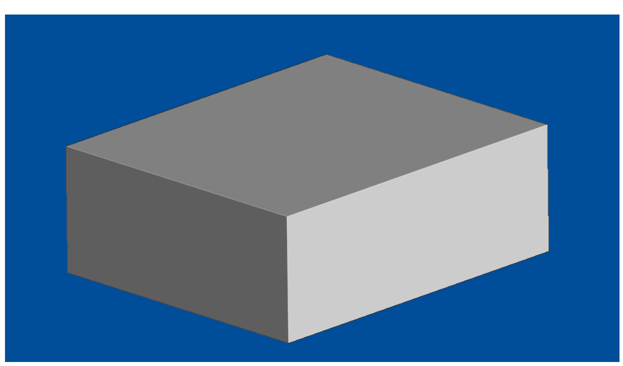


The **ViewMILL** toolbar will be displayed, although initially all the icons will be greyed out.

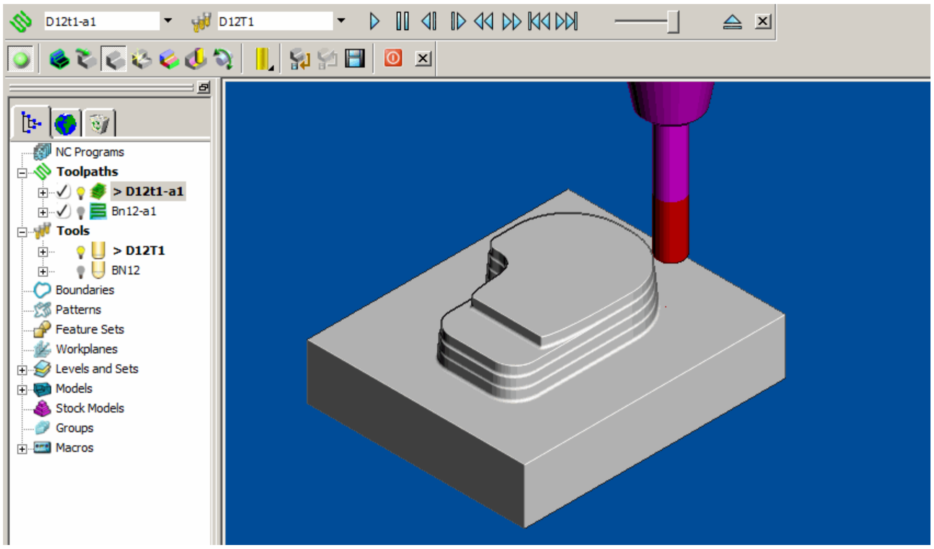
- Click the first button  to **Toggle ViewMILL Window** and enter ViewMILL mode.

The **ViewMILL** toolbar icons will become active (highlighted).

Click the fourth button  to select a plain shaded image.

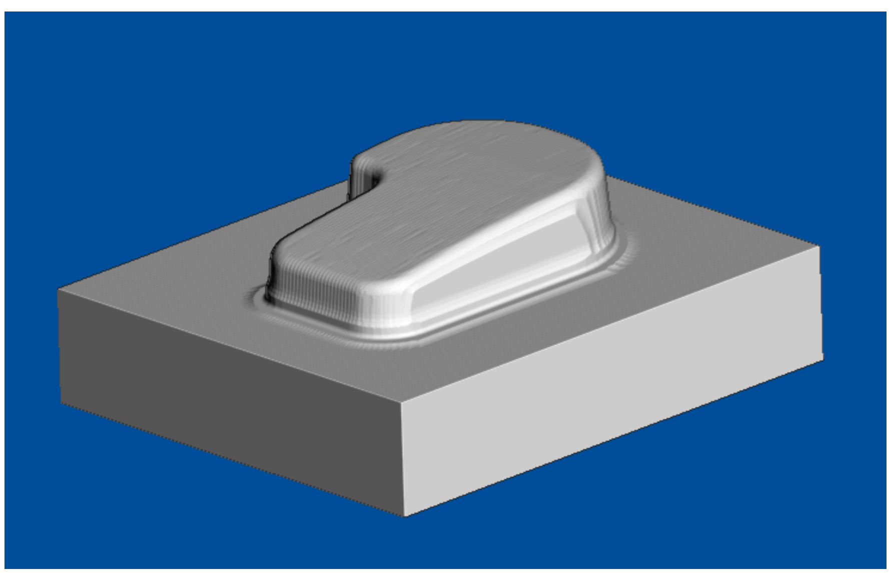


- Select the **tool** icon to display the tool followed by the **Play** icon .



In **ViewMILL** the machining of the material block is simulated as shown above.

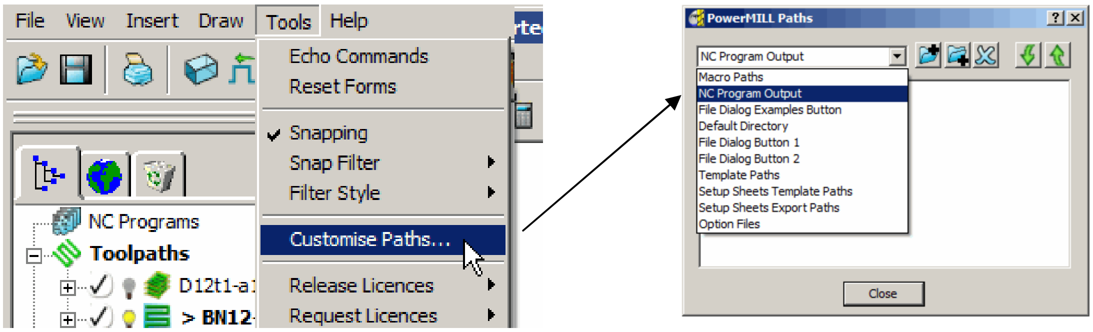
- When the above simulation is finished, in the **Simulation Toolbar**, select the finishing toolpath **BN12-a1** followed by the tool icon .
- Select **Play** icon again, to view the continued simulation of material removal by the finishing toolpath.



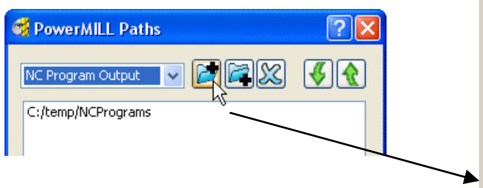
- In the **Simulation** toolbar select the **ViewMILL Exit** icon to exit the **ViewMILL** session.

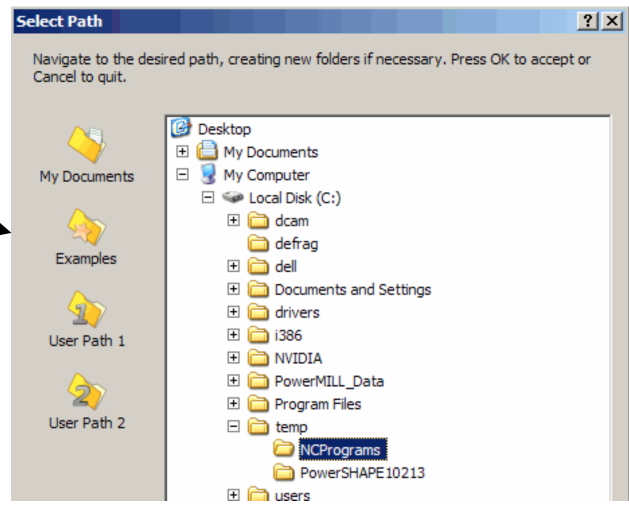
NC Programs (Post-Processing and Ncdata Output)

- In the main pull down menus select **Tools - Customise Paths** to open the ***PowerMILL Paths*** form (shown below right).



- In the ***Powermill Paths*** form select the option **NC Programs Output**.

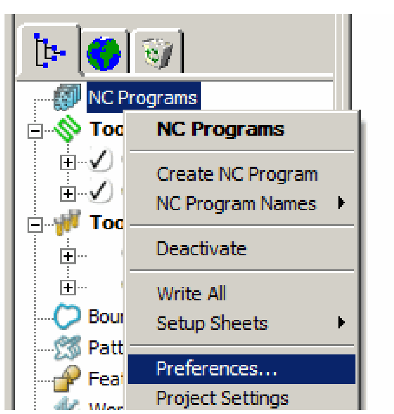




This controls where the post-processed, ncdata files are output ready for download to a machine tool controller.

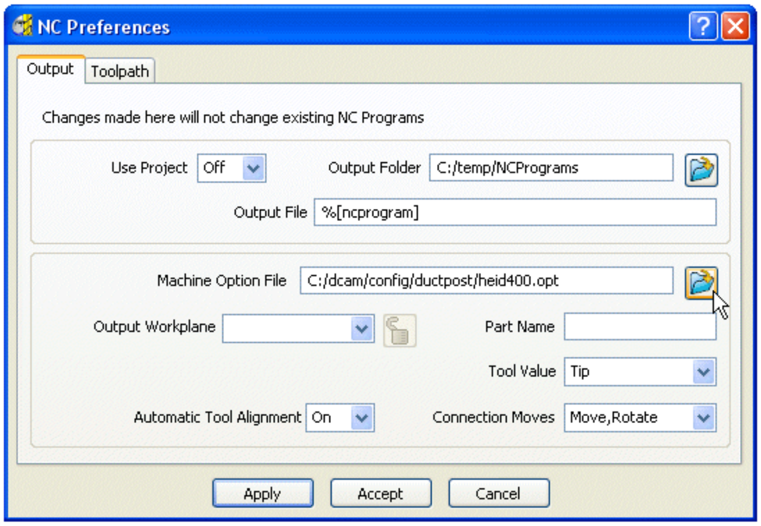
- Right mouse click the **Add path to top of list** icon and in the ***Select Path*** form browse to the required location **C:\temp\NCPrograms** and select **OK**.

- In the **explorer** right mouse click over **NC Programs** to open the following sub-menu.



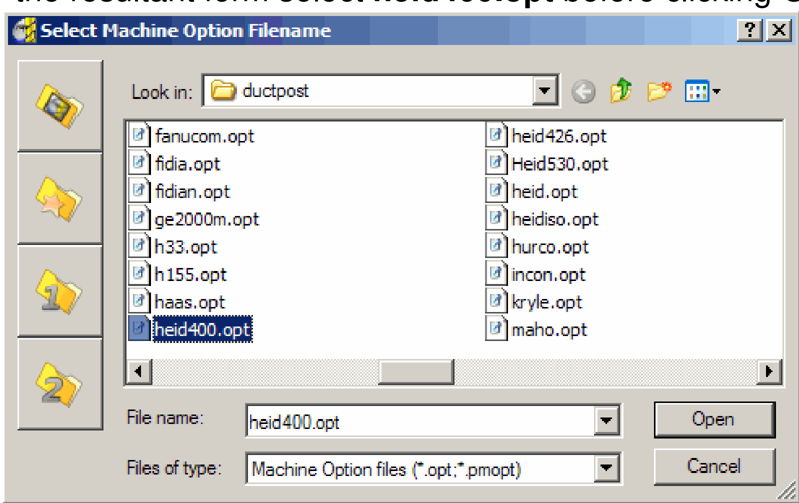
NC Preferences enable the user to control the content of output files for download to a Machine Tool.

- In the **NC Programs** sub-menu select **Preferences** to open the following form.

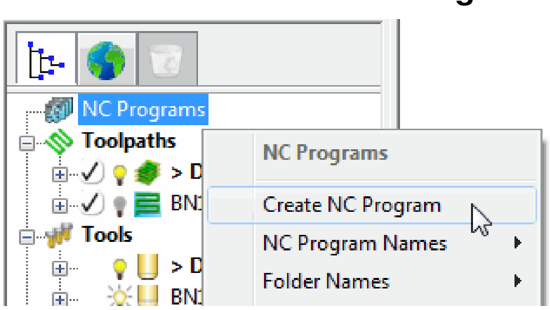


The **Output Folder** defaults to the location already defined in **Tools-Customise Paths**.

- In the above form click on the ***Machine Option File*** icon (arrowed) and in the resultant form select **heid400.opt** before clicking **Open**.



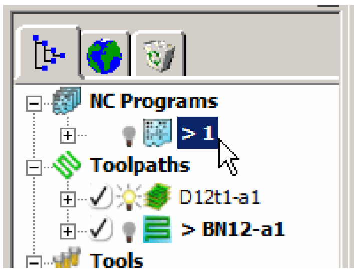
- On return to the **NC Preferences** form select the **Apply** tab to action the settings and then **Accept** the form.
- In the **explorer** right mouse click over **NC Programs** and from the sub-menu select **Create NC Program**.



An empty **NC Program** will appear in the **explorer** ready to have machining strategies assigned to it. The ***NC Program*** form will also open in the Graphics area.

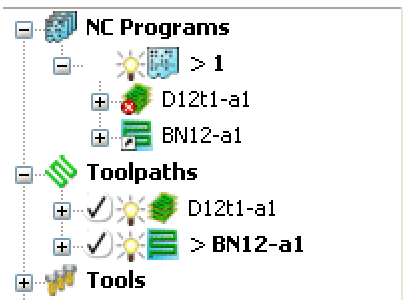
- In the **explorer** move the cursor over the ***toolpath*** **D12t1-a1** and while holding down the left mouse key drag a ghosted image onto the **NC Program** named **1**.

Note:- Alternatively, Right mouse click on ***toolpath*** **D12t1-a1** and from the local menu, select **Add to** > **NC Program**.



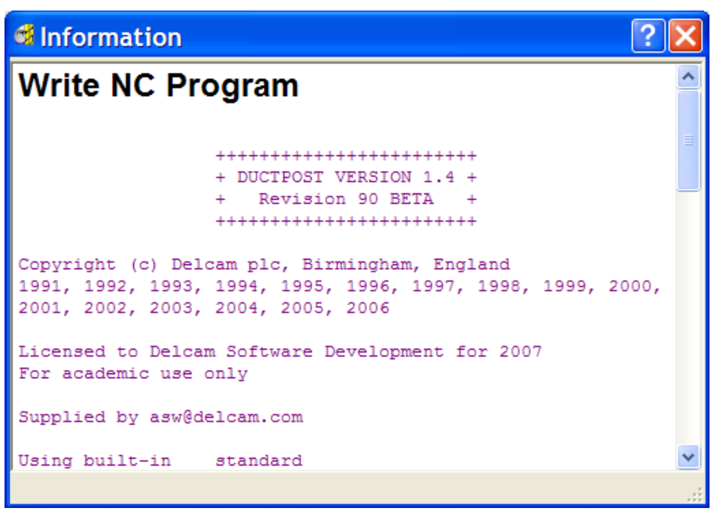
A copy of the toolpath name will appear in the **NC Program** indicating that it has been assigned as part of the output file.

- In the **explorer** drag a copy of the finishing toolpath name **BN12-a1** onto the **NC Program** named **1** and click on the small, adjacent boxed plus sign.



The toolpath names are listed in the **NC Program** ready to be post-processed.

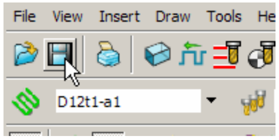
- In the **NC Program** form displayed in the graphics area, select the **Write** tab to start the post processing operation. The following Information form will open providing the user with a progress and confirmation summary.

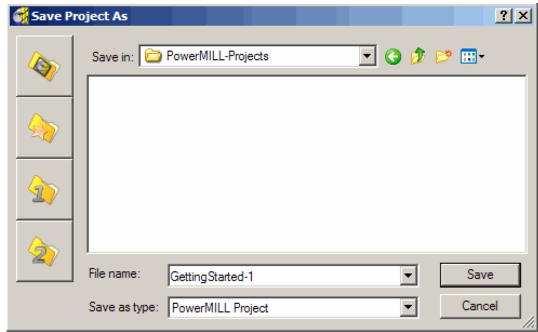


- Close both the **NC Program** and **Information** forms and using the ***windows explorer*** move to **C:\temp\NCPrograms** and note the existence of the **ncdata** output file **1.tap**.

Saving the Project

- Left mouse Click on the 2nd icon along the **Main** toolbar to open the **Save Project As** form.





If the **Project** has been **Saved** before then the **Project** will be updated without the following form being opened.

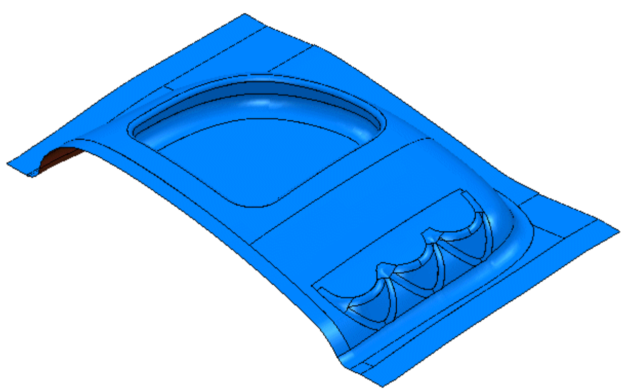
- Select **File** - **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Profects\GettingStarted-1

- Left mouse click in the **Save** tab to store the **Project** to a named external directory (the form will close automatically).
- In the **Main** toolbar select **File – Delete All** followed by **Tools – Reset Forms**.

The content of the **explorer** will be deleted and all **forms** will be reinstated to factory, default settings. The externally stored copy of the **Project** (**GettingStarted-1**) can be reopened as required.

Additional Exercise

- **Import** the ***model*** **facia.dgk** located in the **Examples** directory.



- **Save** the **Project** as:-
  C:\users\training\COURSEWORK\PowerMILL_Projects\Facia-1
- Use the same **Tooling** and create similar machining **Strategies** to those created in **GettingStarted_1**.
- Once completed and the **Project** finally **Saved**, select **File - Delete All** followed by **Tools - Reset Forms**.

2. Machining Setup in Detail

Additional Preparation before creating toolpaths

In the previous section the toolpaths were created using the default values wherever possible. We will now look at the machining process in more detail.
In particular:

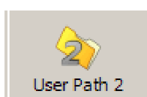
Setting up direct access to regularly used files
Repositioning of the component to a suitable alignment for machining
Detail Examination model features
Tool and Holder definition
Material Block Definition
Safe Z Heights

Setting up direct access to regularly used folders

To **Import a Model** the user can select **File > Import Model** from the top pulldown menu. A selection of sample model files are supplied and installed from within **PowerMILL** in a default directory called **Examples**.

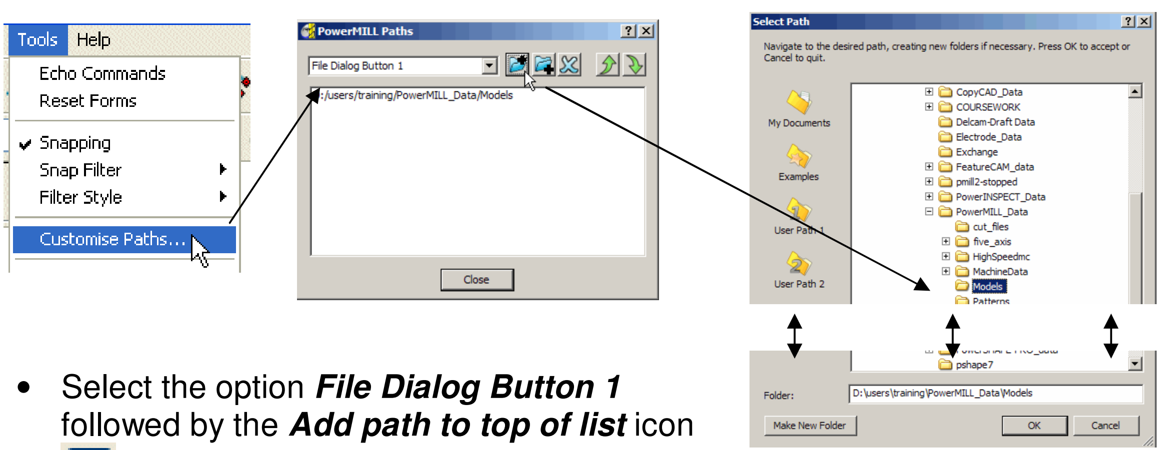
These are directly accessed via the icon in the **Import Model** form.
Alternatively, regularly used models can be accessed quickly with the user definable buttons

and



on the **Import Model** form.

- From the **Main** pulldown menus select **Tools - Customise Paths**.



- Select the option ***File Dialog Button 1*** followed by the ***Add path to top of list*** icon  and browse to:-

C:\users\training\PowerMILL_Data\Models

- Repeat the last process, but this time set ***File Dialog Button 2*** to provide direct access to:-

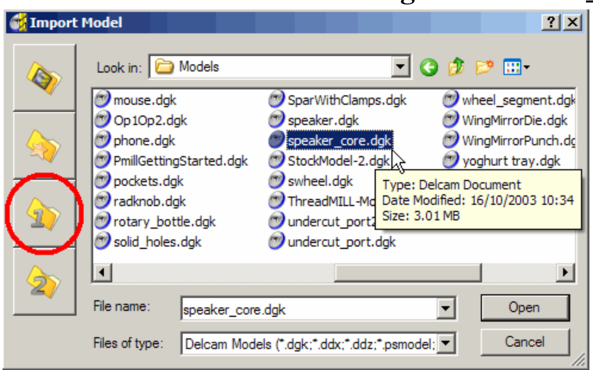
C:\users\training \PowerMILL_Data

Note:- Outside the training environment the location of the **PowerMILL_Data** directory will depend on where the user has installed it!

Loading a Model into PowerMILL

- From the **Main** pulldown menus select **File - Import Model**.
- Use **Shortcut Button 1** or browse to:-

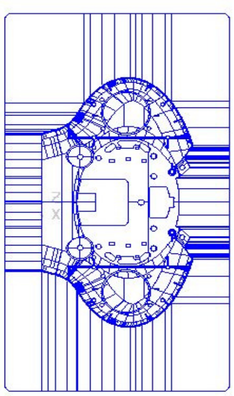
C:\users\training\PowerMILL_Data\Models



Note: 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 select the **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.

Saving the Project

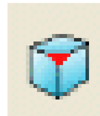
It is good practice to create a **Project** as soon as possible and then to update (**Save**) it at regular intervals both manually and automatically.

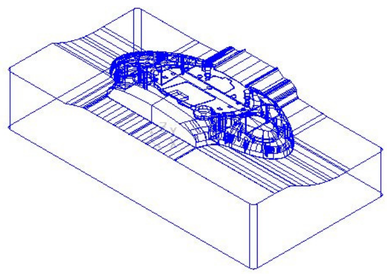
- From **File** select **Save Project As**:-

C:\users\training\COURSEWORK\PowerMILL-Projects\Spkr_Core

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. 

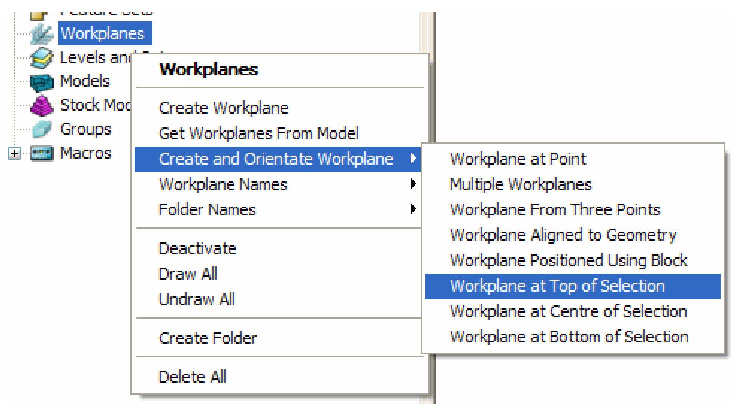


Although it is possible to rotate the actual component this is not generally regarded as the ideal approach. An additional moveable datum (**Workplane**) will be created and rotated through 90 Degrees to effectively create the condition that the longer side of the component is parallel with the front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

Orienting the Model – creating the machining datum using a Workplane

A **Workplane** will be created and rotated through 90 degrees about Z to effectively arrange the longest lower front edge of the model to be aligned to the front of the machine tool i.e. along the X-axis.

- Select the whole **model** by ***dragging a box over it*** while the ***left mouse key*** is depressed.
- Right click over **Workplanes** in the **PowerMILL explorer** and select:- **Create and Orientate Workplane > Workplane at top of Selection**

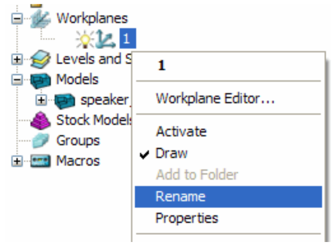


Workplanes are additional datums that can be moved and/or orientated within the global environment. One **Workplane** can be **Active** at any one time. If no **Workplanes** are active then the original global **Transform** is the datum.

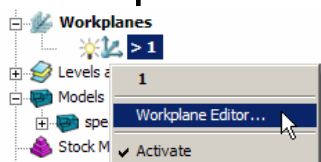
A **Workplane** will appear at the ***top centre*** of the selected **model** components.

The new **Workplane** will also be registered in the **PowerMILL explorer**. For easy identification it is recommended to **Rename** entities created in the **explorer** with something relevant to their application.

- In the **PowerMILL explorer** right mouse click on the actual **workplane** and from the ***local menu*** select **Rename**.

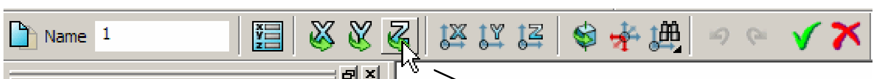


- In the form change the default ***Name*** to **Datum**.
- Right mouse click on the new ***Workplane*** named **Datum** and select ***Activate*** from the local menu.
- From the same local menu select **Workplane Editor** to open the **Workplane** editing toolbar.

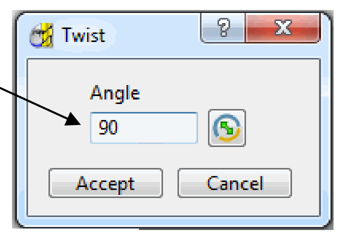


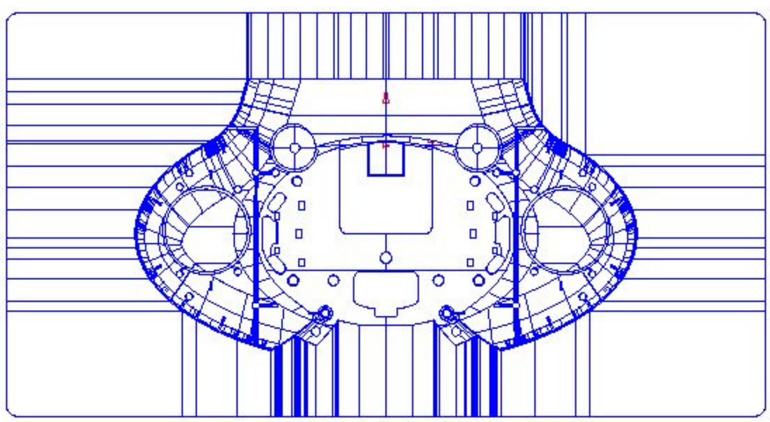
The next step will be to rotate the new **Active Workplane** to indirectly re-orientate the model such that the longer side is aligned to the **X-Axis**.

- Select the **Twist about Z** icon.



- Enter ***Angle*** **90.0** before selecting **Accept**.
- Select the **Green Tick** on the toolbar to finally accept the changes.





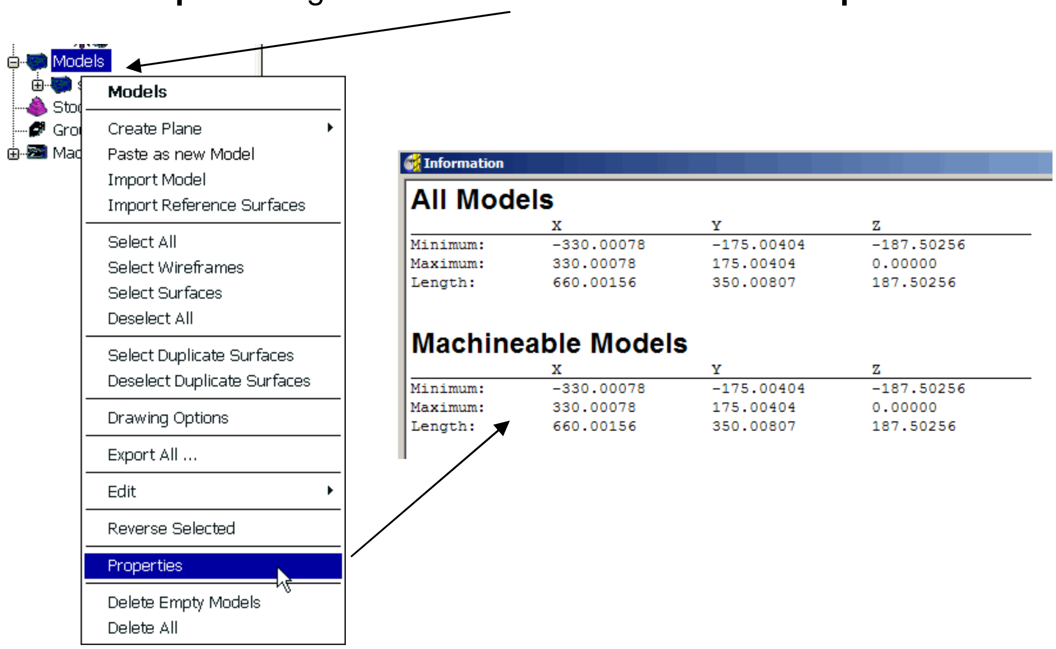
- Select a **View from top (Z)** to see the effect of the rotated, **Active Workplane** providing a more suitable machining position for the **model**.

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.

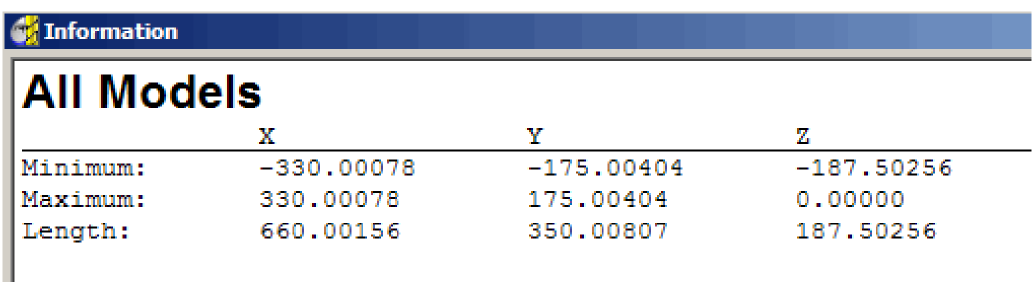
Examination of Model Properties

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**.



If required, the values in this form can be **copied** (**Ctrl C**) and then **pasted** (**Ctrl V**) into other forms (Such as to modify the position of a **Workplane**).

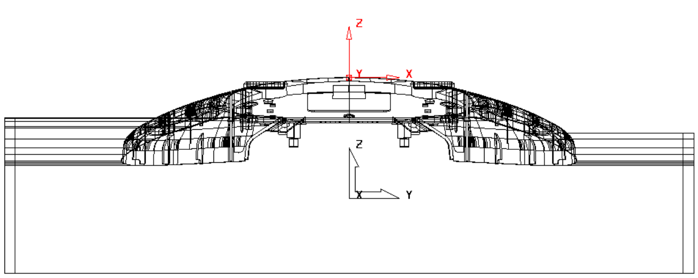


Information

All Models

	X	Y	Z
Minimum:	-330.00078	-175.00404	-187.50256
Maximum:	330.00078	175.00404	0.00000
Length:	660.00156	350.00807	187.50256

- **Eject** the **Model Properties** form by clicking **X** in the top right corner of the form.
- Select a **Y- view**. 



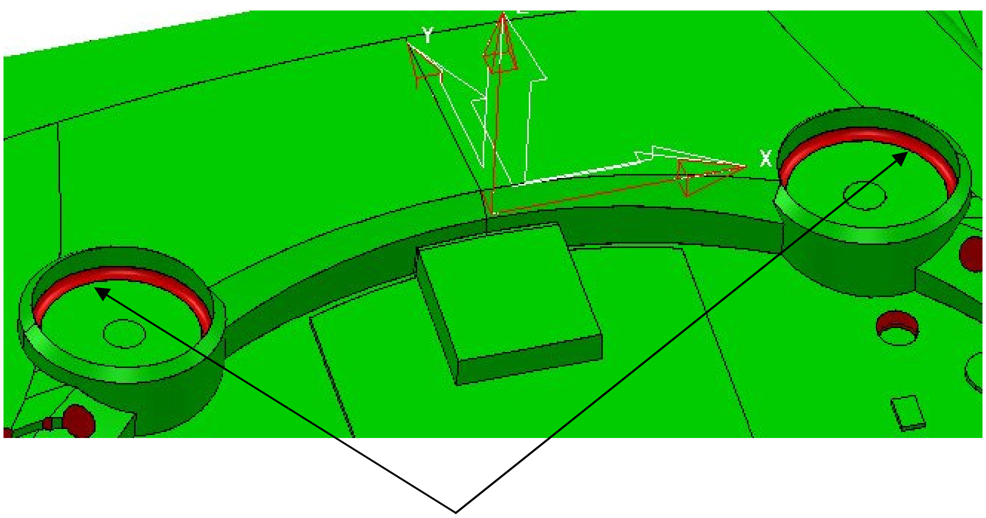
The **Workplane** is shown positioned on the top of the model, which has the longer edge aligned to **X**.

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*** in the bottom right corner of the main icon, as shown above.
- Select the **Minimum Radius Shade** icon on the **toolbar**. 

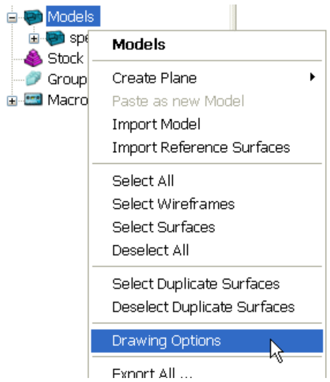
Any internal radii that are smaller than the specified **Minimum Tool Radius** will be shaded **RED**. The settings are located in the **explorer** - **Model** pull down menu in **Drawing Options**.
Examine the model to identify areas that are inaccessible to the specified radius (shaded Red).

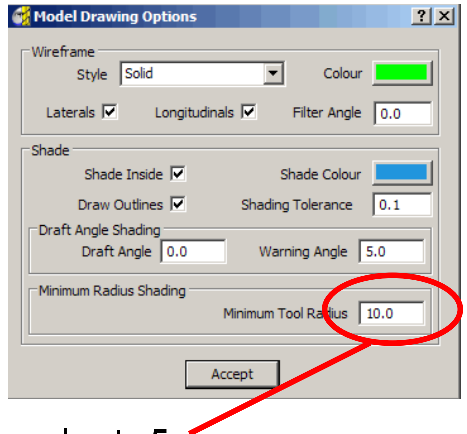


The two internal radii can be seen shaded **red** visually identifying that they will not be machined to their correct size if the default value tool was used.

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**.

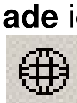
All of the red areas have now disappeared which suggests that the maximum tool size guaranteed to access all areas of the component would be a **Dia 4 Ball Nose** cutter.
The model can also be visually inspected for the size of draft angles and undercuts.

- 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.**

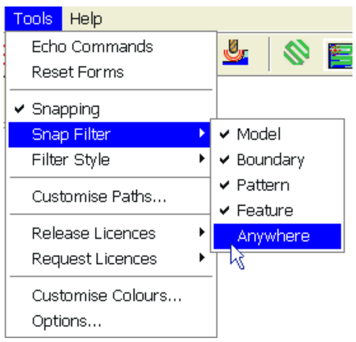
All of the red areas have disappeared and all that remain are green and yellow. If any red areas remain then these would indicate an undercut situation greater than –0.2 degrees.
The yellow areas indicate on or near vertical faces because the difference between the Draft and Warning Angles is so small.

- **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.

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.

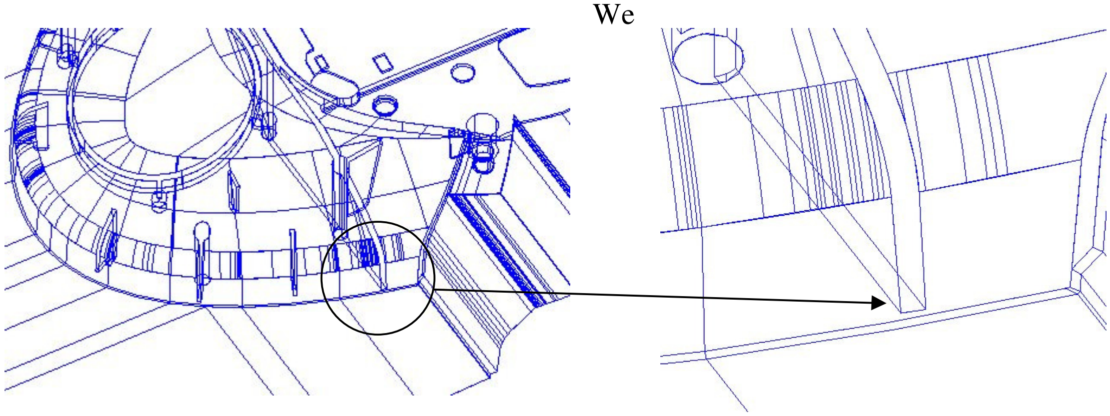
- From the top **Pull Down** menus select **Tools > Snap Filter** and use the left mouse key to **untick - Anywhere**.



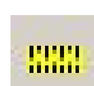
If **Anywhere** is unticked then measurements can be only be snapped onto the remaining ticked entities and not in free space.

- **Zoom** into the area shown by the arrow below.

We



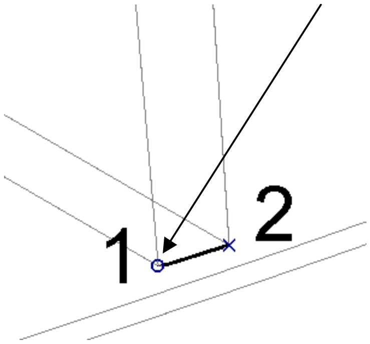
The gap at the bottom of the slot will be measured to determine its size and depth. Note that it is essential to **Zoom** in close to be able to discriminate the close proximity node points.

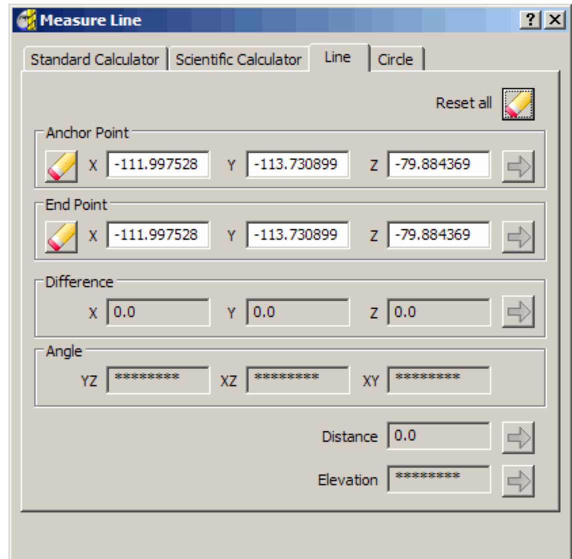
- From the **Main toolbar** select the **Measure model** icon. 

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**.



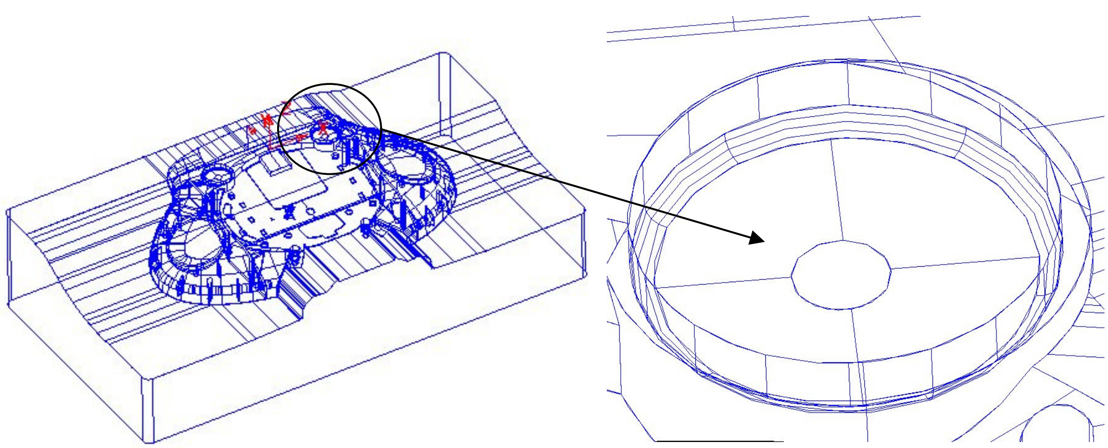


The **Anchor Point** is now selected and is represented by a small circle.
The x, y and z values seen in the above form are relative to the **Active Workplane** '**Datum**'.

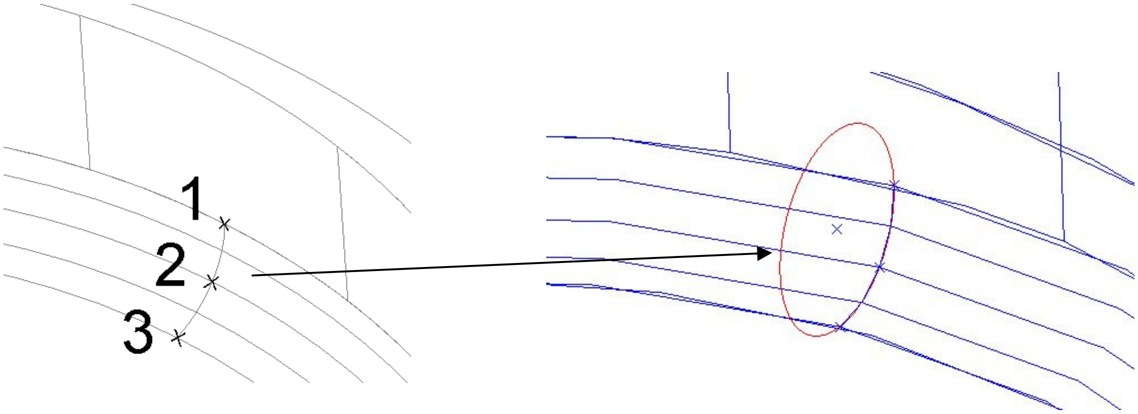
- Drag a window around (or snap) point **2** to obtain the final 'point to point' measurement information.

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.

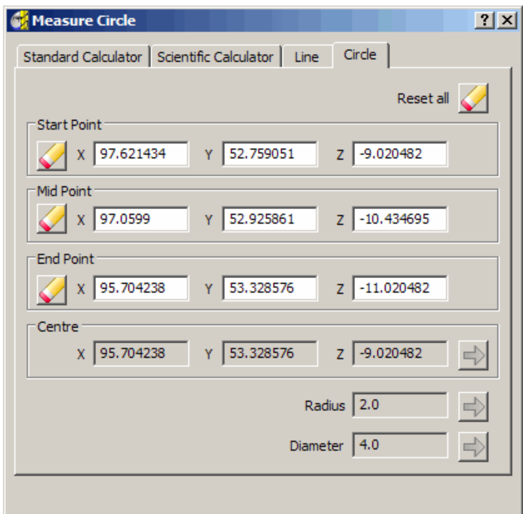
- Select the **Circle** tab on the **Model Measurement** form and zoom into the area shown below.



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



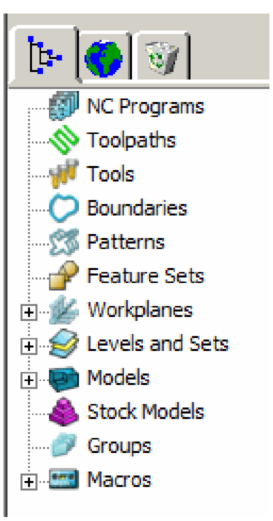
A temporary circle will appear after the third point is selected as shown above.



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

- **Close** the **Model Measurement** form.

<u>PowerMILL Panes</u>



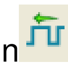
On the left hand side of the screen above the **explorer** are the **PowerMILL** panes.

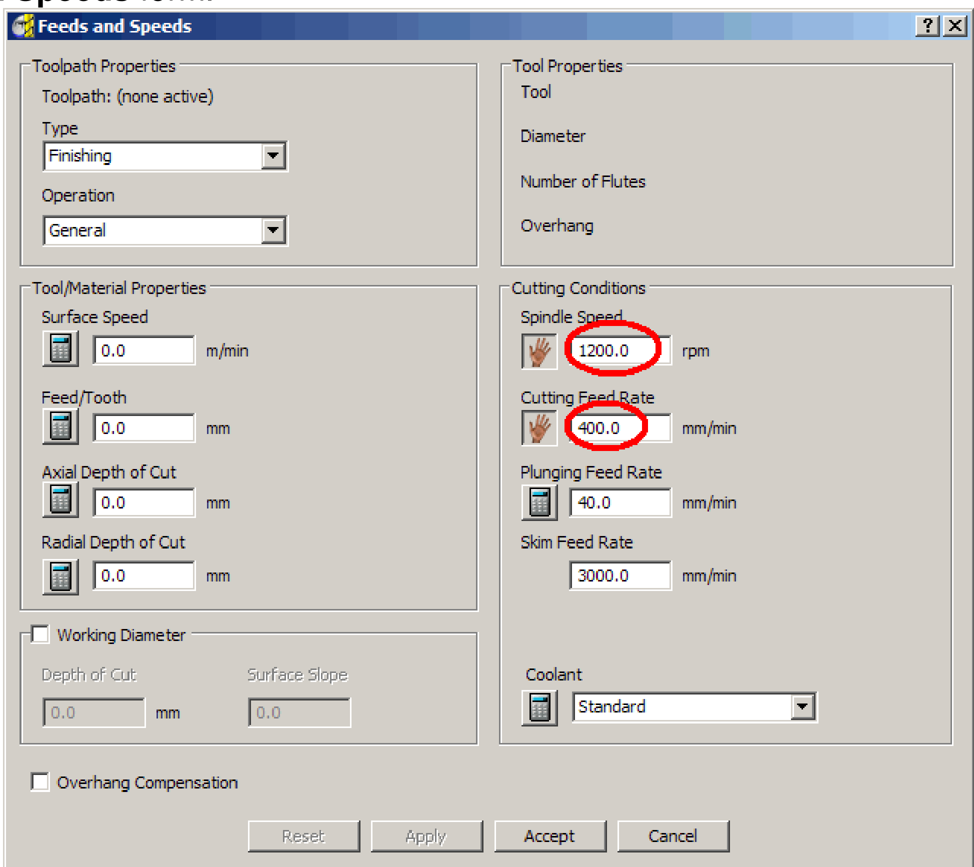
The standard format pane is denoted by the **PowerMILL** symbol and contains the explorer tree categorised into NC Programs,Toolpaths, Tools, Boundaries, Patterns, Feature Sets, Workplanes, etc.

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

Setting Feedrates

Feedrates can be set individually for the current tool and toolpath or loaded from a database of predefined values. At this stage we will look at setting the feedrate individually. The Tool Database will be covered later in the course.

- Click the **Feeds and Speeds** icon  on the top toolbar to raise the **Feeds and Speeds** form.

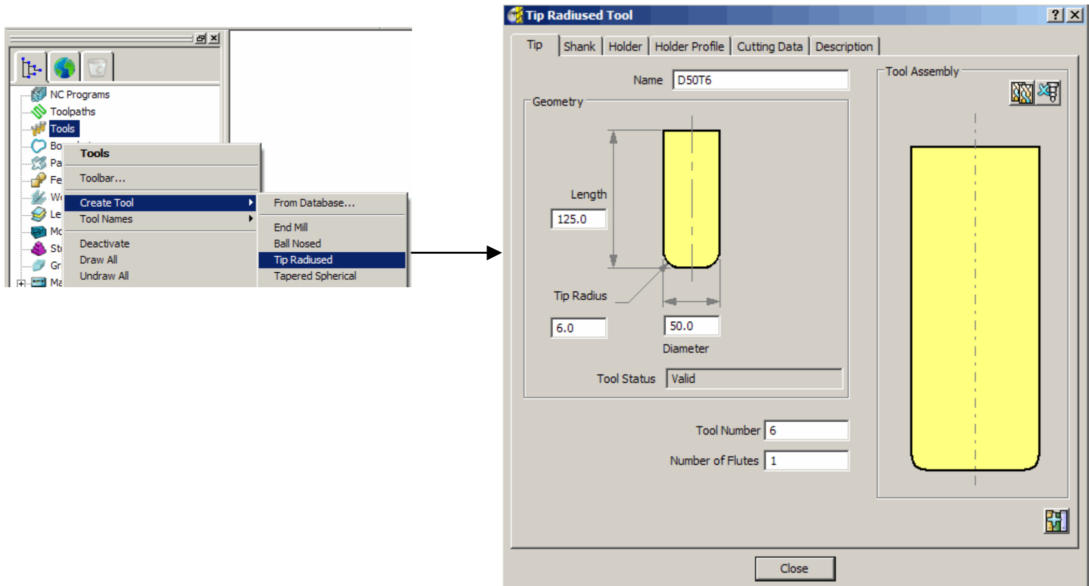


- In the **Cutting Conditions** section of the form, enter a ***Spindle Speed*** of **1200** and a ***Cutting Feed Rate*** of **400** (as shown above).
- **Accept** the form.

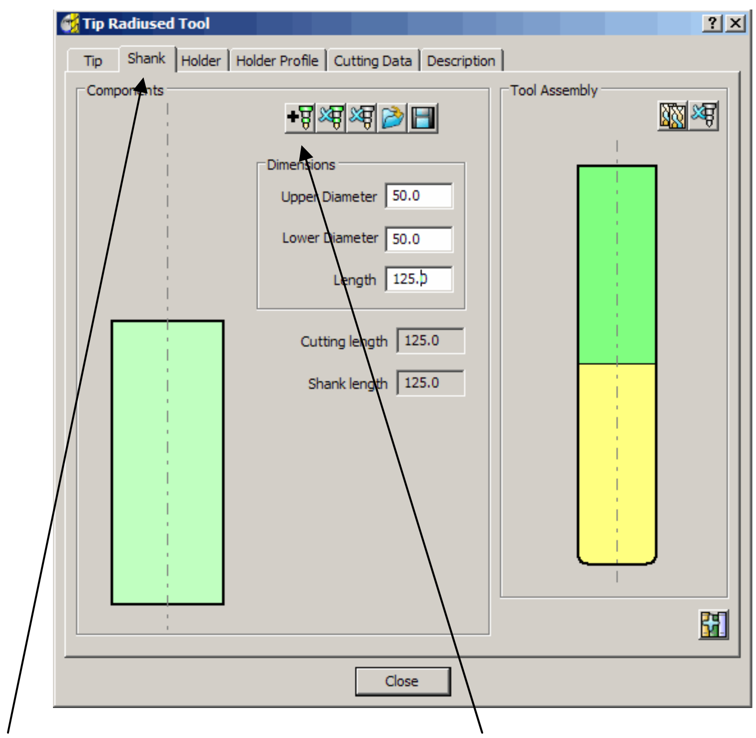
- Select ***Save Project*** to update the content of the externally stored **Spkr-Core** folder (Do not close).

Tool and Holder Definition

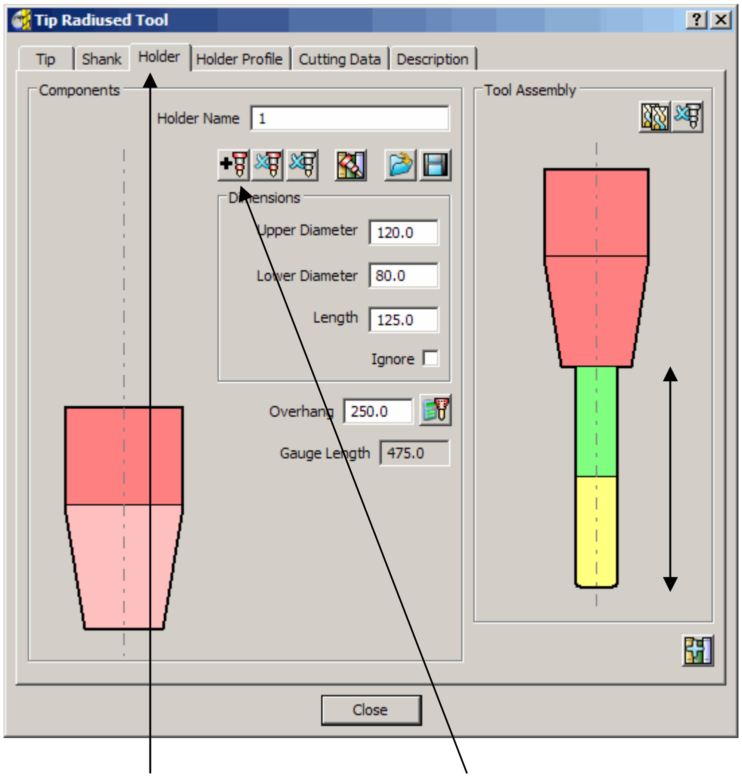
- **Open** the ***Tip Radiused*** tool form.



- In the (above) default ***Tip*** form define a ***Dia*** **50** ***Tiprad*** **6** - ***Length*** **125** - ***Named*** **D50T6 -** ***Tool Number*** **6**.



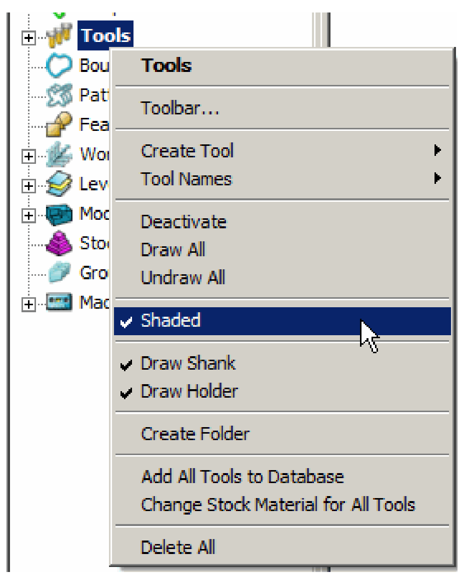
- In the ***Shank*** form, click the ***Add a shank component*** icon and enter ***Upper/Lower Diameter*** **50** and ***Length*** **125**.



The ***Overhang*** is the vertical distance from the bottom of the holder to the tip of the cutter.

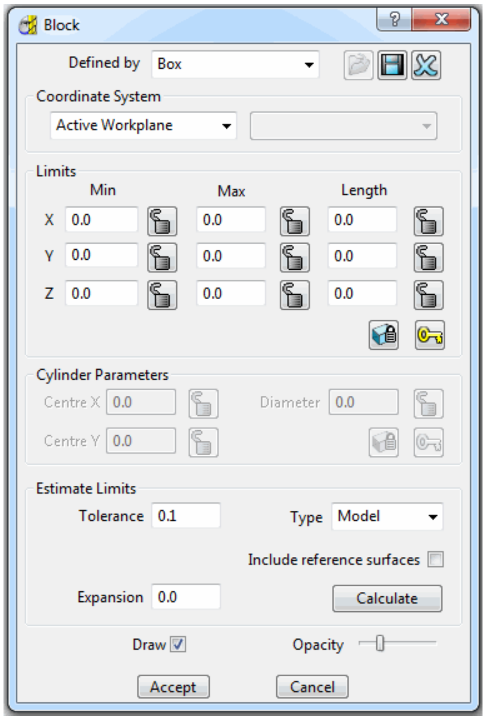
- In the ***Holder*** form, click the ***Add holder component*** icon entering ***Upper Diameter 120*** - ***Lower Diameter*** **80** - ***Length*** **125** and ***Overhang*** **250**.
- In the ***Holder*** form, click the ***Add holder component*** icon entering an ***Upper/Lower Diameter*** **120** and ***Length*** **100**.
- Close the form

The style in which the tool is displayed can be controlled by right clicking on TOOLS in the tree browser pane and selecting **Shaded**.



Material Block Definition

- **Calculate** a 3D working volume (**Block**) to actual model dimensions using the ***Defined by* - Box** option.



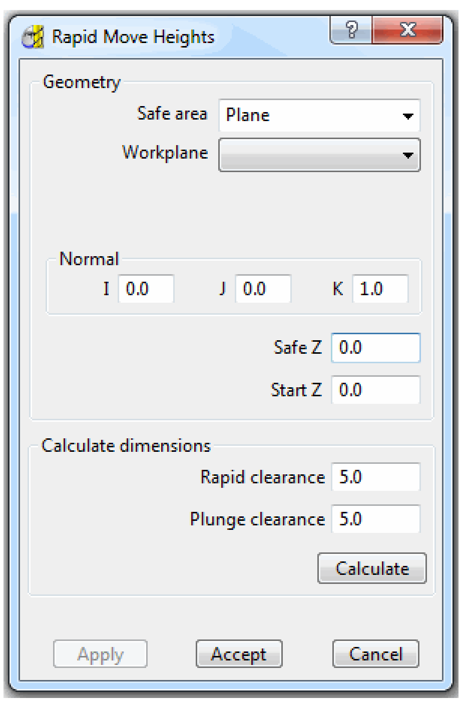
The default option for **Block** is **Box** (A rectangular volume). Other options include **Cylinder** (a Cylindrical volume), a **Triangle** model (Casting) available in PowerMILL PRO only, and **Picture/Boundary** (Extruded 2D wireframe contours).

The dimensions of the **Block** can be entered manually or calculated directly to the ***Type*** of entity:-
Model, **Boundary**, **Pattern**, or **Feature**.

The **Opacity** slider controls the degree of shading (clear to dense).

Rapid Move Heights

- In the **Rapid Move Heights** form click the **Calculate tab** to set the default values for ***tool rapid moves*** to clear the component by **10**.

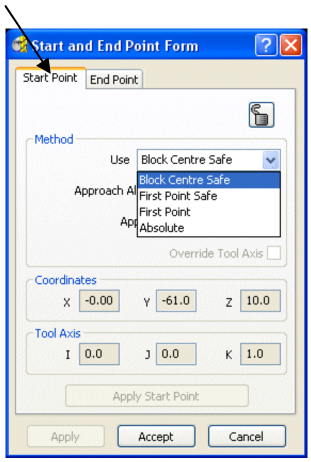


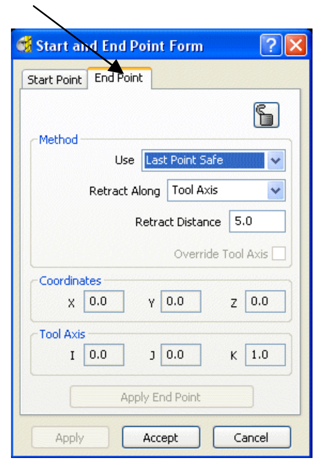
- Input the active ***Workplane* Datum**.

The **Rapid Move Heights** options enable ***safe rapid moves*** between tool tracks clear of the top of the **Component** or **Block** (whichever is the higher).

Start and End Point

- In the **Start and End Point** form use the default settings :-
Start Point - **Block Centre Safe** and ***End Point*** - **Last Point Safe**.





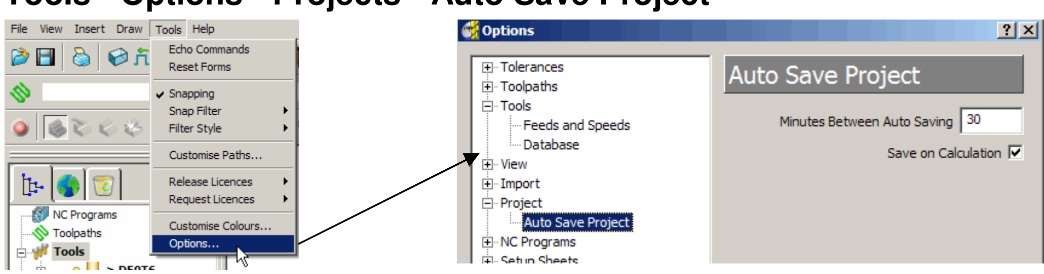
Save Project

- Select ***Save Project*** to update the saved ***PowerMILL*** **Project** (***Do not close*** the **Project**).

Note: The **Project** is stored in:-
C:\users\training\COURSEWORK\PowerMILL-Projects\Spkr-Core

Up to **PowerMILL 9** it has been the responsibility of the user to **Save the Project** at suitable times during the **PowerMILL** process. In addition to this, **Tools** - **Options - Projects** can be set to include **Save the Project** automatically either on ***calculation*** of each toolpath or after ***a set time period*** (or both). From **PowerMILLL 10**, if one or both <u>is set,</u> then even if a **Project** has not been created the session will now **auto recover** after a crash.

- In the main **Pull Down** menus select:-
Tools - Options - Projects - Auto Save Project



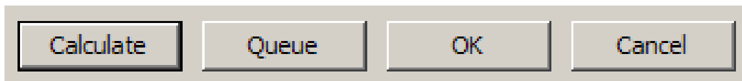
- Input **Minutes Between Auto Saving** as **30** (minutes) and **tick** the ***Save on Calculation*** **Box**.

PowerMILL will now automatically apply **Save Project** after a toolpath has ***calculated*** as well as every time **30** minutes has elapsed since the previous **Save Project**.

Toolpath Processing Options

Calculate

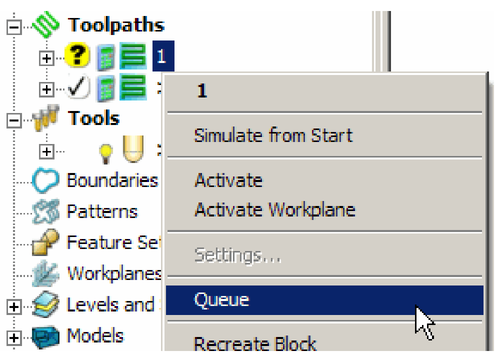
Once a machining strategy has all the required ***settings*** and ***values*** ready for processing, the **Calculate** tab can be clicked to start the calculation process. During this time the user is unable to continue working in **PowerMILL** until the **Toolpath** has been created. In this case the ***strategy form*** remains open with just the ***Recycle*** and ***Copy*** options active.

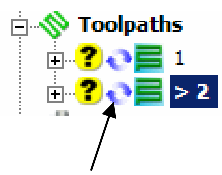


There however two further options:-

Queue (PowerMILL PRO users only)

The second is **Queue** that allows the user to continue working in **PowerMILL** while the **Toolpath** is being processed in the background. (One or more **Strategies** can be added to the **Queue** and they are calculated in the background, in sequential order). As soon as a **strategy** is applied to the **Queue** the form closes.



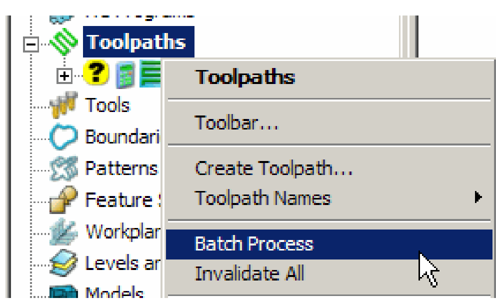


Note; The '***Calculator***' symbol changes to the '***Queue***' symbol as soon as the **Queue** option is selected.

Queue can either be applied directly from within a **Strategy** form, or from the **explorer** menu (as shown left).

OK - Batch Process

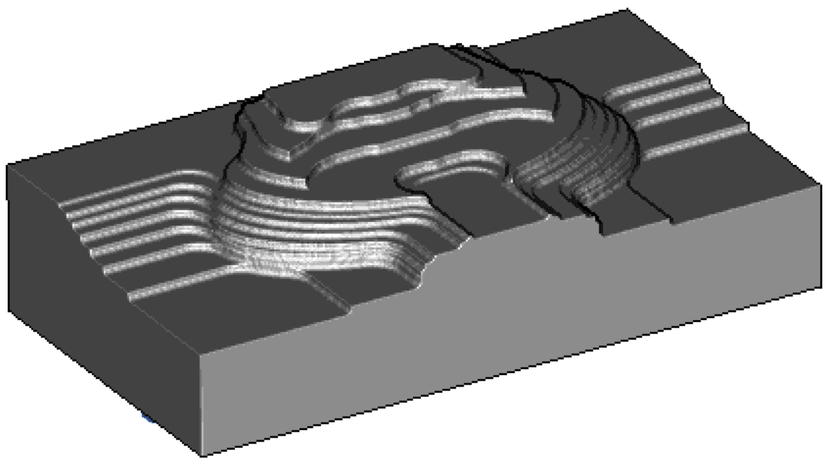
The third is **OK** which closes the unprocessed, toolpath form with the current settings and values, stored and ready for the **Batch Process**, menu option to be applied. One or more unprocessed, machining strategy can be saved to be sequentially calculated later, on selection of the **Batch Process** option. While ***Batch Processing*** in taking place the user is unable to continue working in **PowerMILL** until all the included **Toolpaths** have been created.



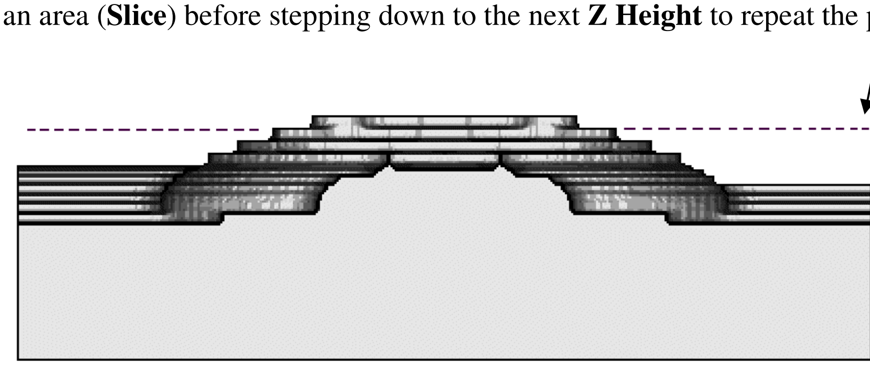
3. 3D Area Clearance

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**.



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.



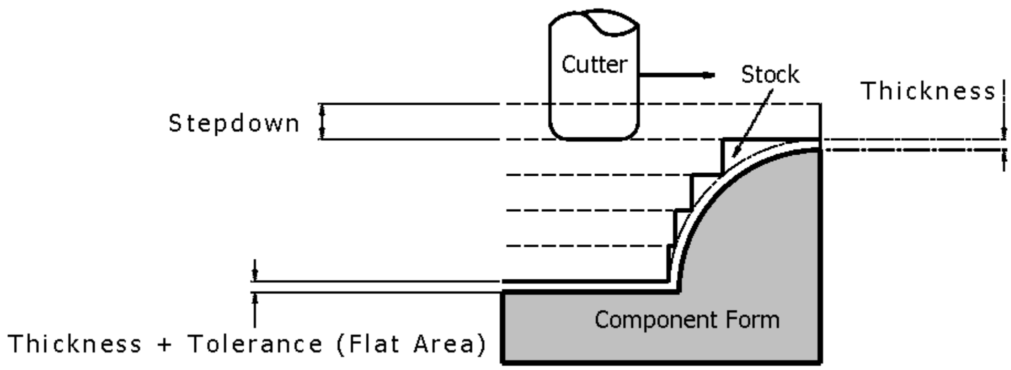
For some components a secondary **Area Clearance** strategy is applied using the **Rest Machining** options in conjunction with a smaller roughing tool. This will locally remove pockets of excess material inaccessible to the original **Reference Toolpath** or **Stock Model**. This will reduce the degree 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.

Thickness and Tolerance (Applied to 3D Area Clearance)

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**.

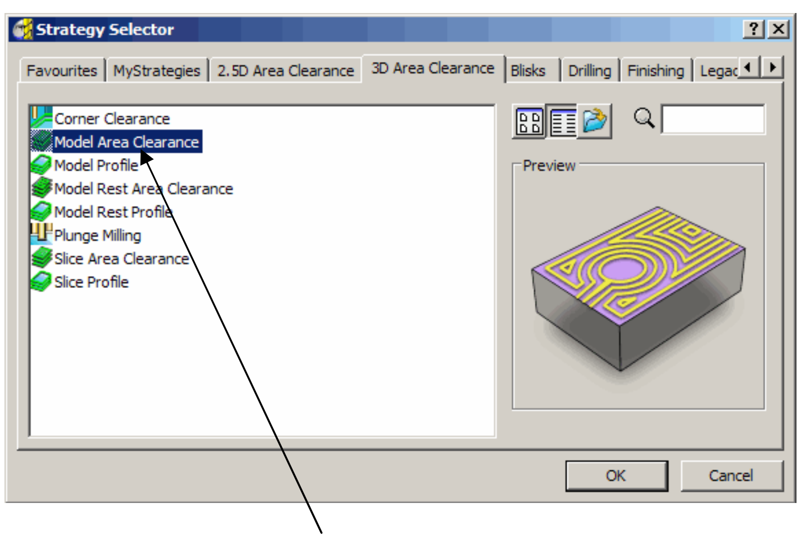
Thickness is the amount of extra material specified to remain on the work-piece after machining. This can be applied generally (as shown), or independently as separate **Radial** and **Axial** values within the machining options.



Raster Area Clear example

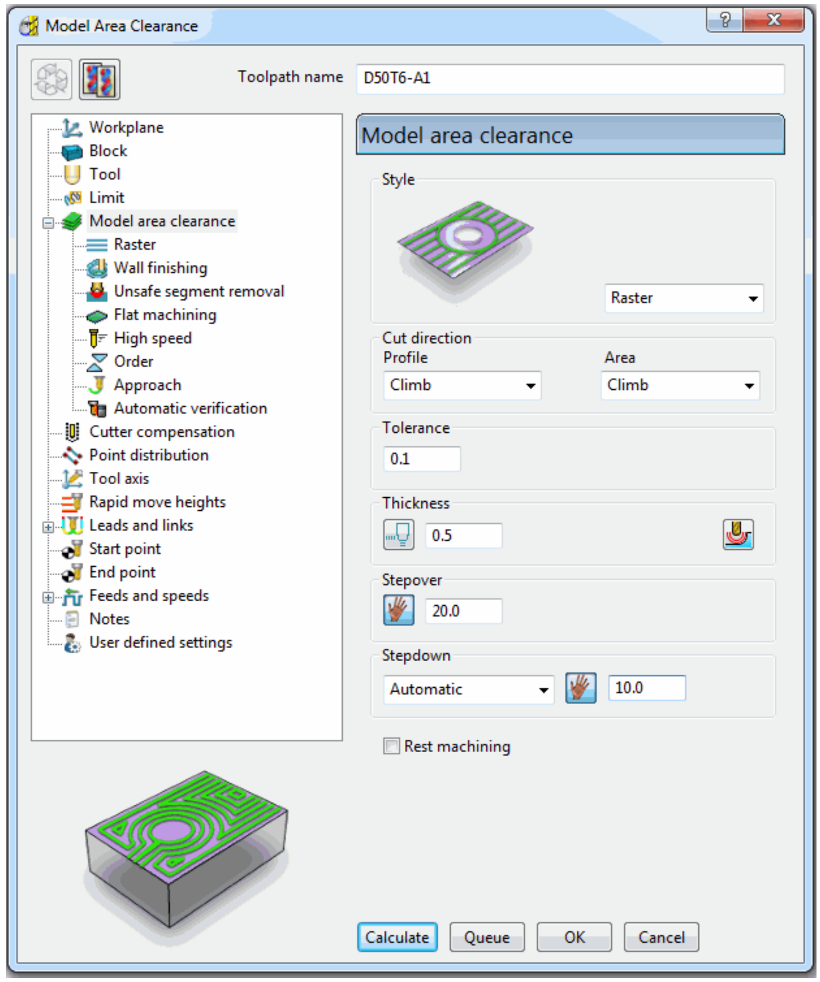
The **Raster Area Clear** strategy follows a series of ***linear moves*** across the **Block** limiting to the **Component form** at the ***active Z height***. It then, (if required) performs a ***Profile pass*** around the component to leave a constant **thickness** around the **Slice**. Other options provide the abilty to further fine tune the final strategy.

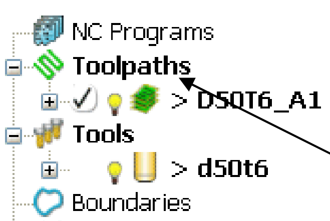
- From the **Main toolbar** select the **Toolpath Strategies** icon. 
- Select **3D Area Clearance** tab.



- Select the option **Model Area Clearance** to open the following form.

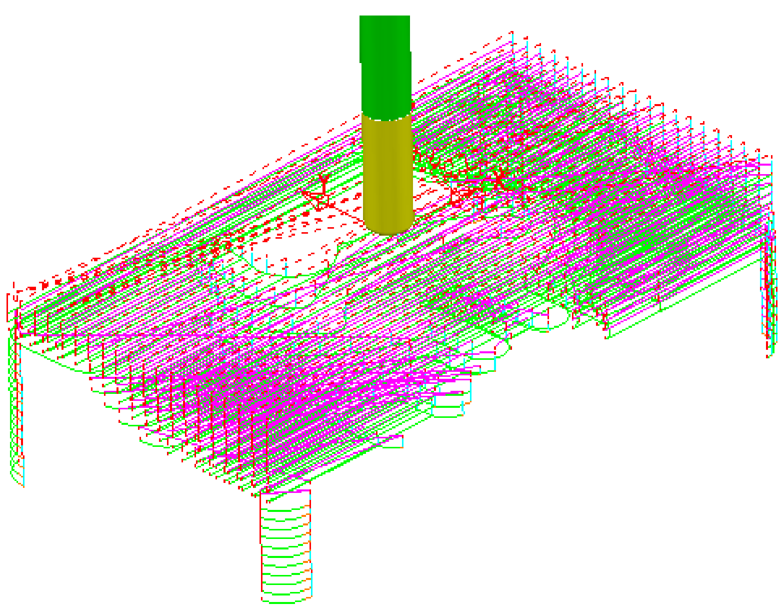
- Enter the ***Toolpath Name*** **D50T6_A1**
- Set **Thickness** to **0.5**
- Set **Stepover** to **20**
- Set **Stepdown** to **10**
- Keeping all other values as default, **Calculate** the form.
- After the processing is complete **Cancel** the form.





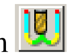
As soon as the **Raster Area Clearance (Model)** form is opened an unprocessed **Toolpath** appears in the **Explorer** (the default name has been changed to **D50T6_A1**).

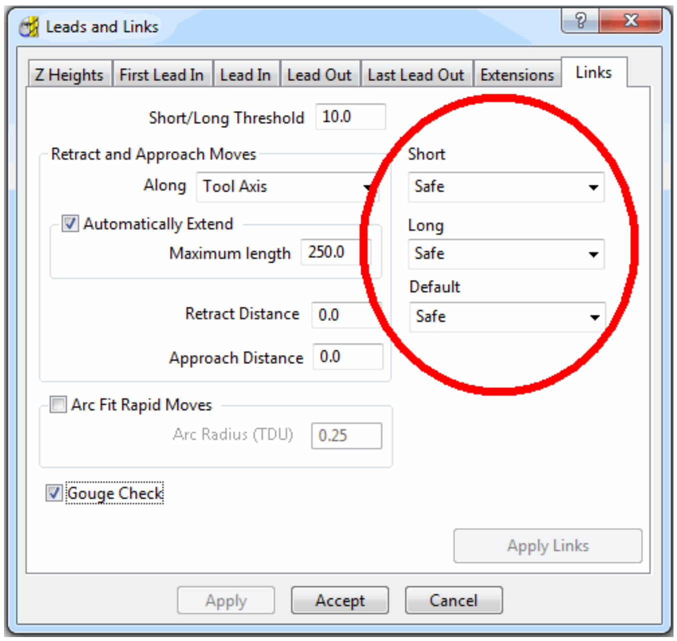
The **Toolpath** icon can be double-clicked to **Activate** and **de-activate** the toolpath.
The + symbol can be clicked to open a full record of the data used to create the toolpath.
The dotted red lines represent **Rapid** moves and the light blue lines are the **Plunge** feed moves.

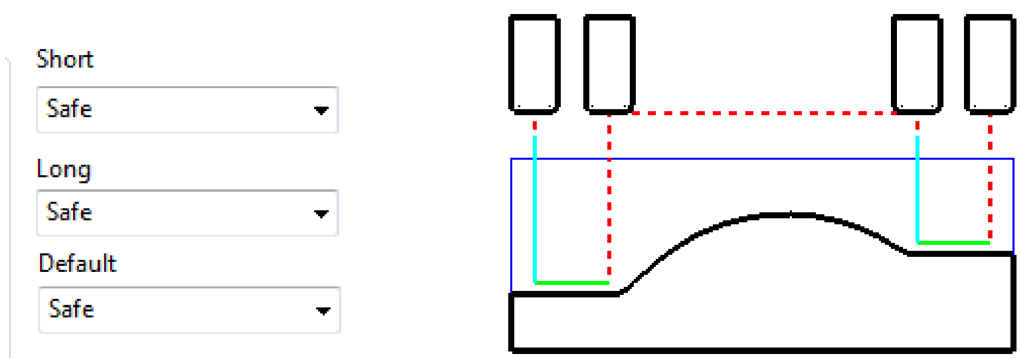


Rapid Move Heights in detail

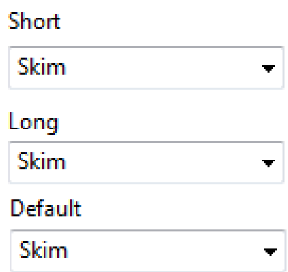
The **Rapid Move Heights** form provides **Safe Z** and **Start Z** input boxes. Suitable values are entered to define a safe height (**Safe Z**) at which a tool can safely perform, horizontal **Rapid Moves** above the model as well as (**Start Z**) where a **Rapid** ***plunge move*** changes to a ***plunge*** **Feed Rate**. If the **Reset to Safe Heights** tab is clicked then **PowerMILL** will set the **Safe Z** and **Start Z** to be a safe distance above the ***Top of the Model or Block*** (whichever is the highest).

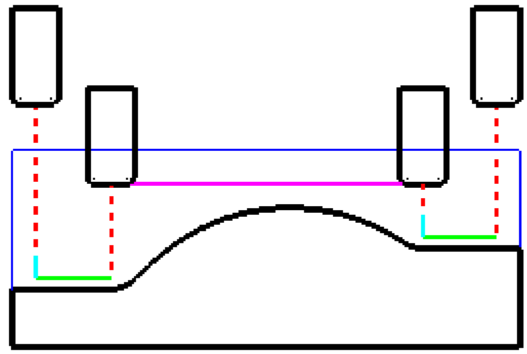
Rapid Move Heights works in conjunction with the **Leads and Links** form  which in turn provides a more flexible choice of toolpath **Link** moves.



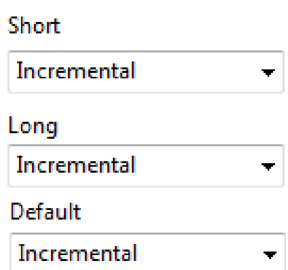


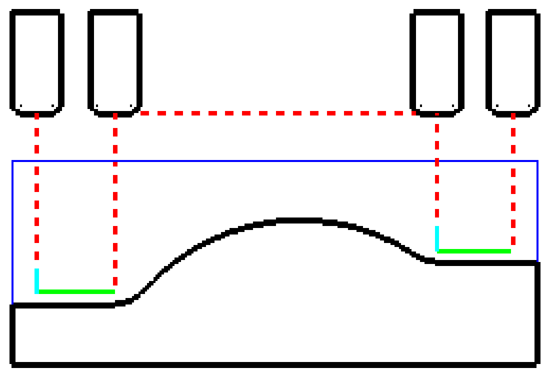
Safe (default) sets the **plunge feed rate** to apply at a specific height above the job. This is more predictable and reassuring for the ***machine tool operator*** but the time take by the non-cutting (air) moves is inefficient especially in the case of large, deep components.
In the section of the form labelled **Incremental Heights**, in addition to **Safe**, two other options **Skim** and **Incremental** are available.





Skim enables the downward, ***rapid feed rate*** to continue to a specified ***Incremental* Start Z** above the ***full plunge depth*** before the slow **plunge feed rate** 'cuts in'. **Skim** then applies a **rapid retract** to an ***Incremental* Safe Z** above the ***highest point*** on the component 'in line' with a ***linear* link** move to the next ***plunge*** position. To cater for all types of ***machine tool*** this move is a (***Purple***) ***Skim Feed Rate*** (G1) as instead of a (***Dashed red***) ***Rapid*** (G0).



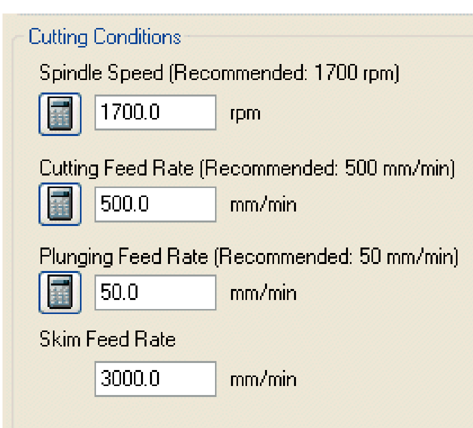


Incremental applies the ***rapid feed rate*** all the way down to an ***Incremental* Start Z** measured from the ***full plunge depth*** at which point the slow **plunge feed rate** 'cuts in'. The **Plunge** option differs from **Skim** in that all **rapid link** moves occur at the ***Absolute* Safe Z**.

Feed Rates assigned to Toolpath element colour

The **Feeds and Speeds** form uses the ***Style*** and ***Colour*** of elements along a ***toolpath*** to register the correct type of ***Rapid Move*** or ***Feed Rate*** settings.

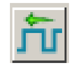
Fixed **(G0) Rapid** moves:- ***Red Dashed*** - Toolpath elements

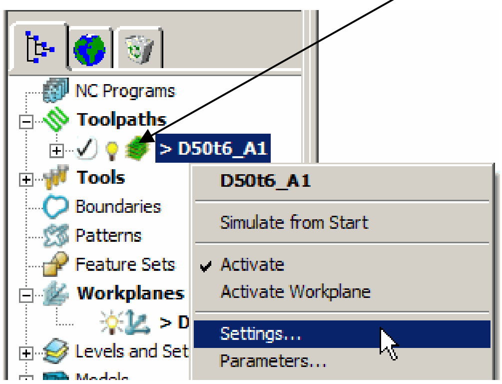


Variable **(G1) Feed Rate** moves:-

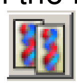
Green/Orange - Toolpath ***Cutting Feed*** moves.
Pale Blue - Toolpath ***Plunging Feed*** moves.
Purple - Toolpath ***Skim Feed*** moves.

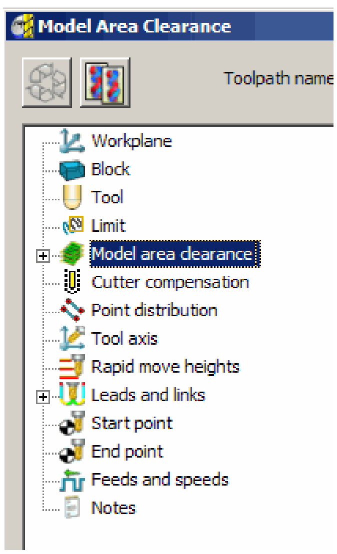
Also, ***local areas*** of a **toolpath** can be assigned with additional **Cutting Feed Rate** values via the **Toolpath Editing** options (***See Chapter 8***) as a **percentage** of the nominal value. **PowerMILL** will assign a ***different colour*** to areas of the **toolpath** edited to have a new **Feed Rate**.

- From the **Main** toolbar, select **Feeds and Speeds**  and input the values shown above before selecting **Accept**.
- Right mouse click over the **Toolpath** icon in the **explorer** to open the local pull down menu.



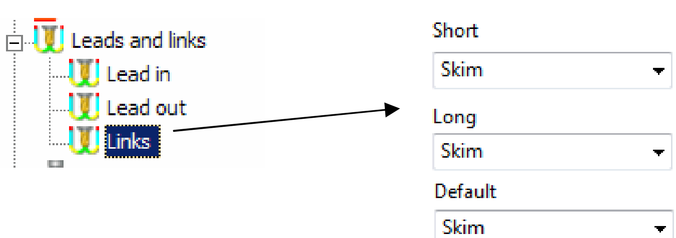
Note the toolpath can also be activated or deactivated from the **Activate** switch in the pulldown menu.

- Select **Settings** to reopen the **Raster AreaClear Model** form.
- Select the '***make a copy***'  of the **toolpath** icon.

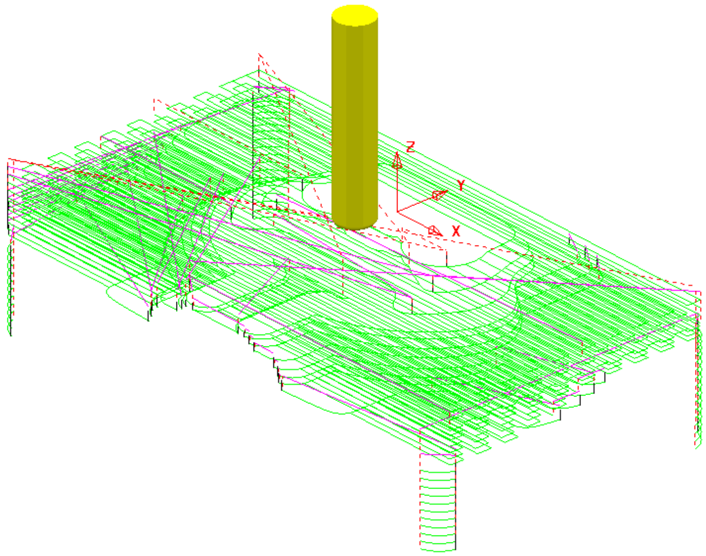


On the left of the **Model Area Clearance** form is an **explorer** window. When an individual option relating to the strategy is selected all related settings etc are ***exclusively*** displayed in the main form. This makes it easier for the user to navigate, with the minimum settings on the form at any one time along with larger tabs and icons.

- From the **Toolpath** form **explorer** select the **Links** option and change the ***Short***, ***Long***, and ***Default*** **Links** to **Skim** as shown below.



- Select the **Calculate** tab at bottom left the **Model Area Clearance** form and once processing is complete click **Cancel** to close the form.

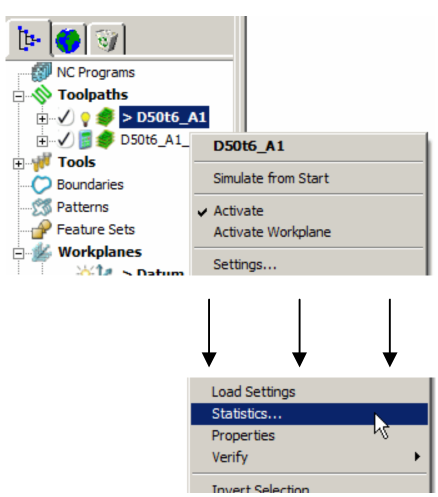


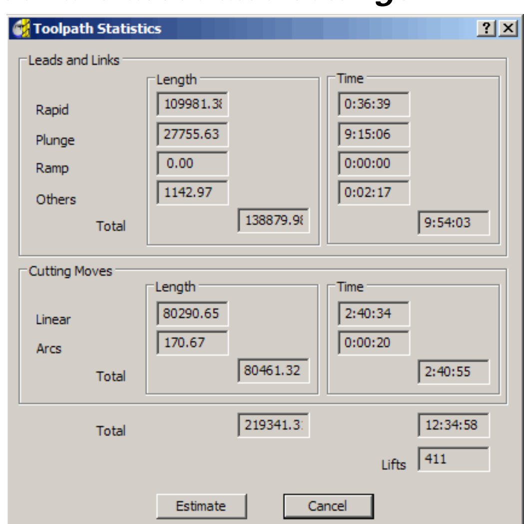
The tool will now plunge locally (pale blue move) from the defined ***Incremental* - Start Z** relative to each ***slice*** and rapid across the roughed areas using an ***Incremental* - Safe Z** (purple move).

Statistics

Provides the user with essential ***information*** about the **Active *toolpath and associated parameters***.

- Right mouse click on the original ***toolpath*** (**D50T6_A1**) in the **PowerMILL explorer** and from the local menu select **Activate**.
- In the same menu select **Statistics** and a form will open displaying information relating to the ***toolpath*** and associated ***settings***.





Note; In this case the ***total machining time*** is displayed as **12:35 hrs**.

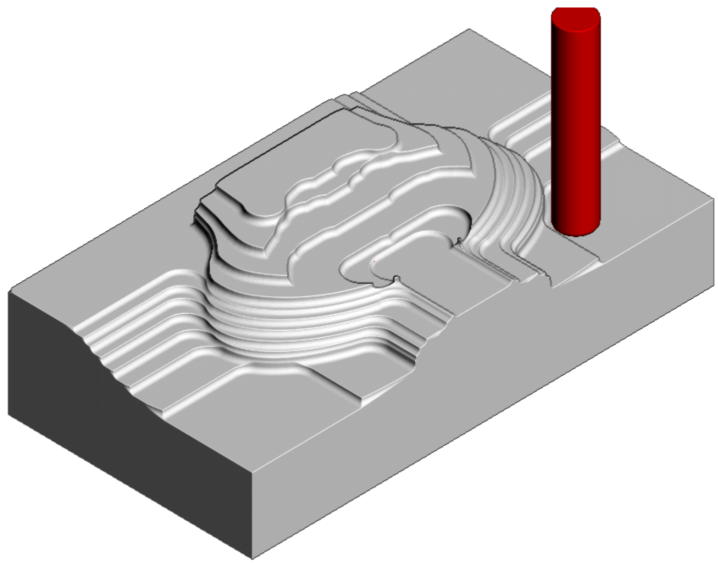
- **Activate** the second ***toolpath*** (**D50T6_A1_1**) and obtain the **Statistics**.

Note: In this case the ***total machining time*** will be around **5:12 hrs**.

This large saving is achieved simply by using **Skim** in the **Rapid Move Heights** form. In the original toolpath **Absolute** was used in the **Rapid Move Heights** form and this includes a large amount of **toolpath link moves** running in fresh air.

Simulating the toolpath

- Perform both a **toolpath** and **ViewMILL *simulation*** on the final **Raster Area Clearance** toolpath.



- Switch the **ViewMILL** to **No Image**  to return to the **PowerMILL** session and toggle the **ViewMILL On/Suspend**  to ***red*** (**Suspend**).

Note:-By toggling back to **PowerMILL**, the **ViewMILL** session will still exist in the background so that any subsequent toolpaths can later be used to continue the ***simulation***. If the **Viewmill** session is still set to **On**, then even though it is set to **No Image**, the **Viewmill** simulation will continue to update parallel with any further, **toolpath** simulations.

Saving the Project

- From the **Main** pull down menus, select **File** - **Save Project** to update the stored data.

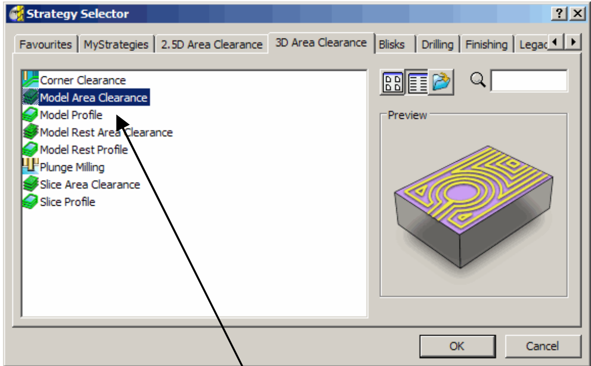
The **Project** (**C:\users\training\COURSEWORK\PowerMILL-Projects\Spkr-Core**) has now been updated to include the **3D Area Clearance** toolpaths.

- ***Do Not Close*** the **Project** as it is to be continued in the next example.

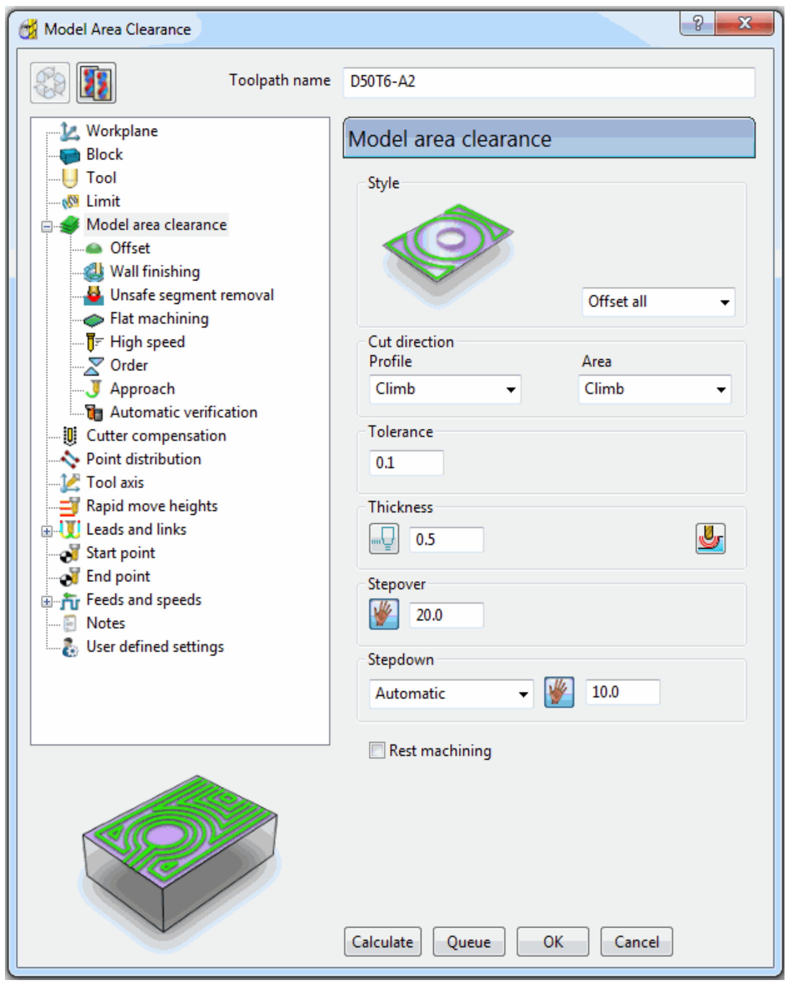
Offset All – Model Area Clearance example

The **Offset All** strategy starts by following the contour of the both the **Block** and **Component form** at the ***active Z height***. It then progressively ***offsets*** into the remaining areas of stock.

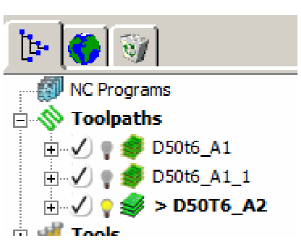
- From the **Main toolbar** select the **Toolpath Strategies** icon. 
- Select **3D Area Clearance** tab.



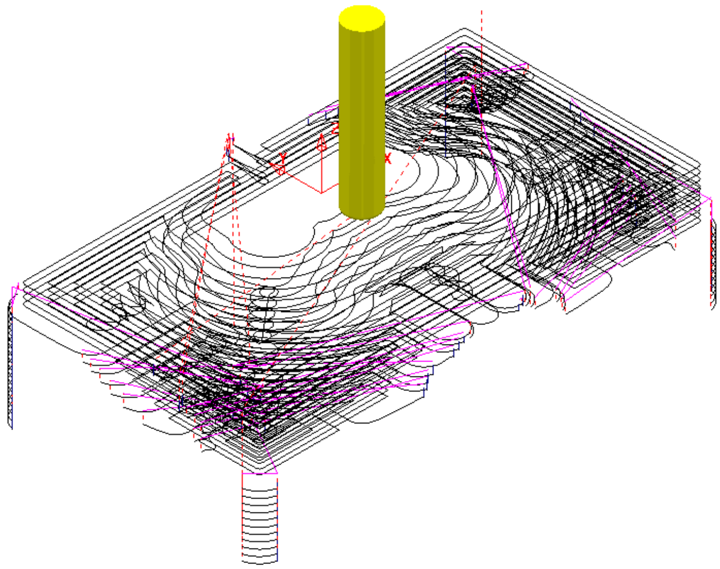
- Select the option **Model Area Clearance** to open the following form.



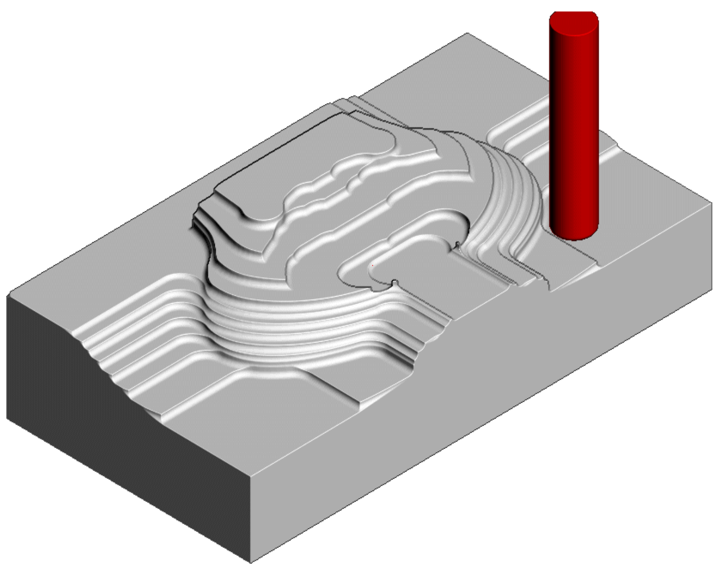
- Enter the ***Name*** **D50T6_A2**.
- Select ***Style*** – **Offset All**
- Set **Thickness** to **0.5**
- Set **Stepover** to **20**
- Set **Stepdown** to **10**
- Keeping all other values as default, **Calculate** the form.
- After the processing is complete **Cancel** the form.

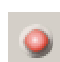


As soon as the **Offset Area Clearance (Model)** form is opened an unprocessed **Toolpath** appears in the **Explorer** (the default name has been changed to **D50T6_A2**).



- Perform both a **toolpath** and **ViewMILL *simulation*** on the final **Raster Area Clearance** toolpath.



- Select the **ViewMILL Suspend** icon  and **No Image**  to return to the **PowerMILL** session.

- From the **Main** pull down menus, select **File** - **Save Project** to update the stored data.

The **Project** (**C:\users\training\COURSEWORK\PowerMILL-Projects\Spkr-Core**) has now been updated to include the **3D Area Clearance** toolpaths.

- From the **Main** pull down menus select **File - Delete All**
- From the **Main** pull down menus select **Tools - Reset Forms**

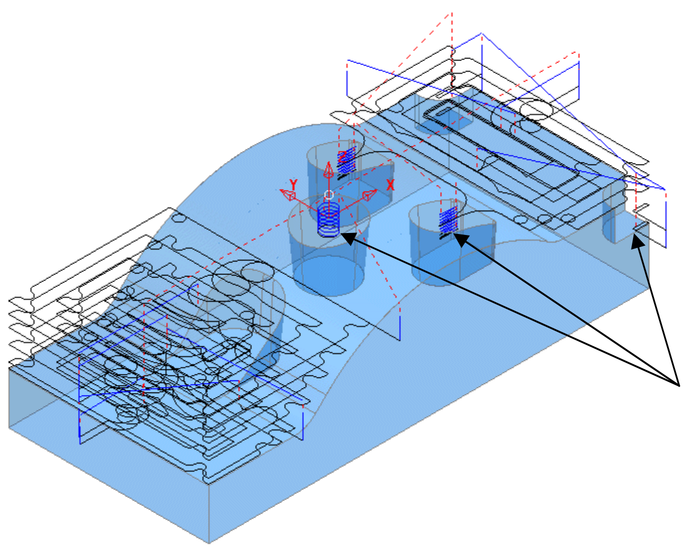
Area Clearance – Unsafe Segment Removal

This option is designed to filter out machining of confined areas that would involve small movement of the cutting tool. In the following example it is applied to prevent a relatively large tool attempting to plunge into a deep pocket area where a smaller tool would be a better choice.

- **Open** the '*Read Only*' **Project**:- **C:\users\training\PowerMILL_Data\Projects\MountingBlock-Start**
- **Save Project As**:-

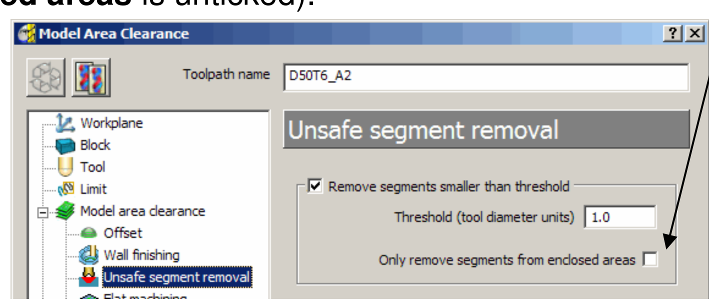
C:\users\training\COURSEWORK\PowerMILL-Projects\MountingBlock

- **Activate** the ***Toolpath*** named **No-AreaFilter**



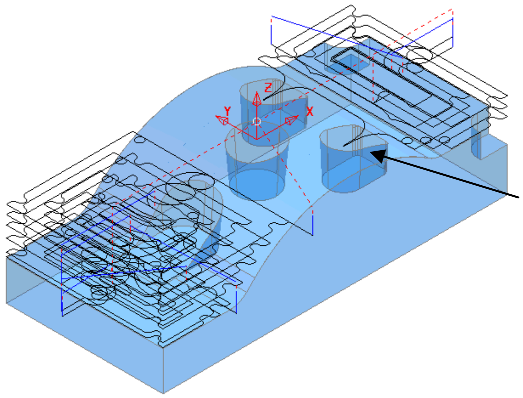
The existing toolpath is allowing the cutter to machine into confined areas. This is resulting in sudden sharp changes of cutting direction and excess loading on the tool.

- Right click on the **toolpath** named **No-AreaFilter** and from the local menu select **Settings** to open the original **Offset Area Clearance** form.
- Select the '***make a copy***' of the **toolpath**. 
- ***Rename*** the copy of the toolpath as **AreaFilter**.
- From the toolpath form **explorer**, select **Unsafe segment removal** and input the settings as shown below (Note; **Only remove segments from enclosed areas** is unticked).



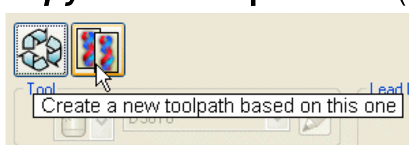
All tooltracks spanning a distance less than the **Tool Diameter (TDU)** will be filtered out and not appear in the final **Toolpath**.
The effect of unticking **Only Remove segments from enclosed areas** means that the ***recessed areas*** running out to the **Block** will be included in the filtering process.

- **Calculate** the **Offset Area Clear Model** form and once processing is complete click **Cancel** to close the form.

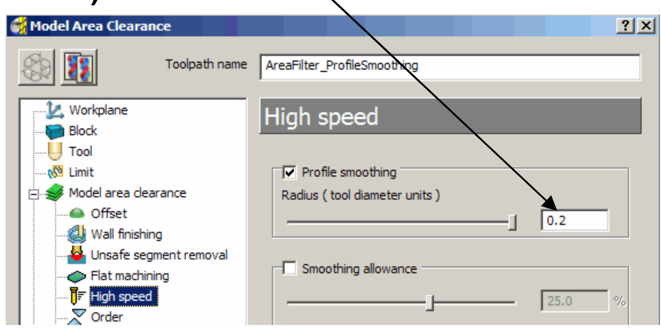


Area Filter has been applied to prevent the cutter attempting to machine into confined areas.
There are however a couple of undesirable spikes on the unfiltered part of the toolpath. These can be reduced by applying the **High Speed** - **Profile Smoothing** option.

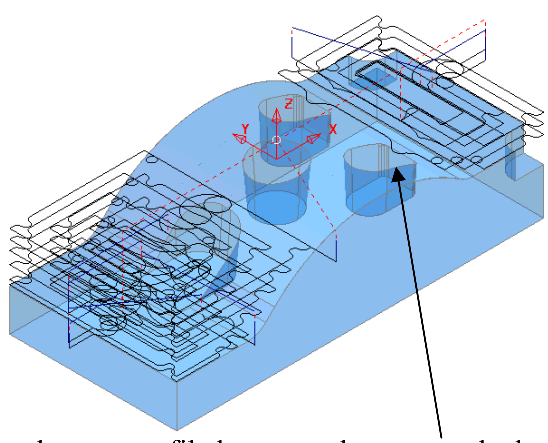
- Right click on the **toolpath *named* AreaFilter** and from the local menu select **Settings** to open the original **Offset Area Clearance** form.
- Select the '***make a copy***' of the **toolpath** icon (shown arrowed below).



- ***Rename*** the copy of the toolpath as **AreaFilter-ProfileSmooth**.
- Keep the same **Unsafe Segment Removal** settings switched on, and in ***High Speed***, ***tick*** the **Profile Smoothing** box with the **Radius (tool diameter units)** slider set to **0.2**



- **Calculate** the **Offset Area Clear Model** form and once processing is complete click **Cancel** to close the form.



The previously sharp corners around the toolpath outer profile have now been smoothed.

- From the **Main** pull down menus, select **File** - **Save Project** to update the stored data.

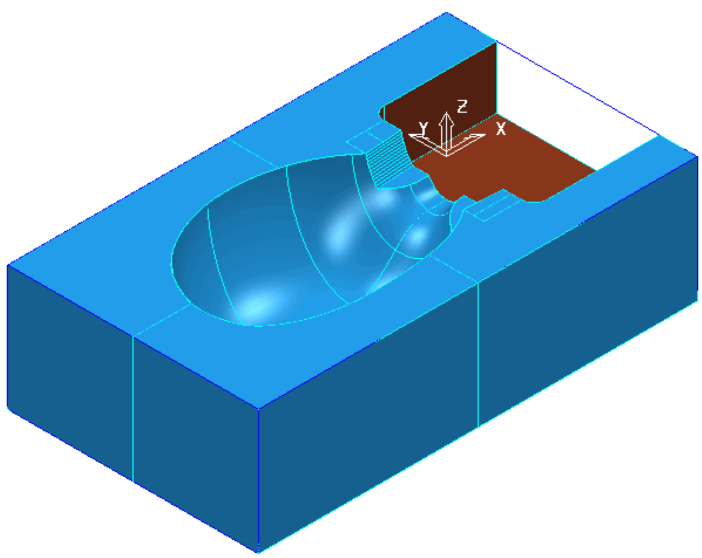
The **Project** (**C:\users\training\COURSEWORK\PowerMILL-Projects\MountingBlock**) has now been updated to include the **3D Area Clearance** toolpaths.

- From the **Main** toolbar select **File - Delete All**.
- From the **Main** toolbar select **Tools - Reset Forms**.

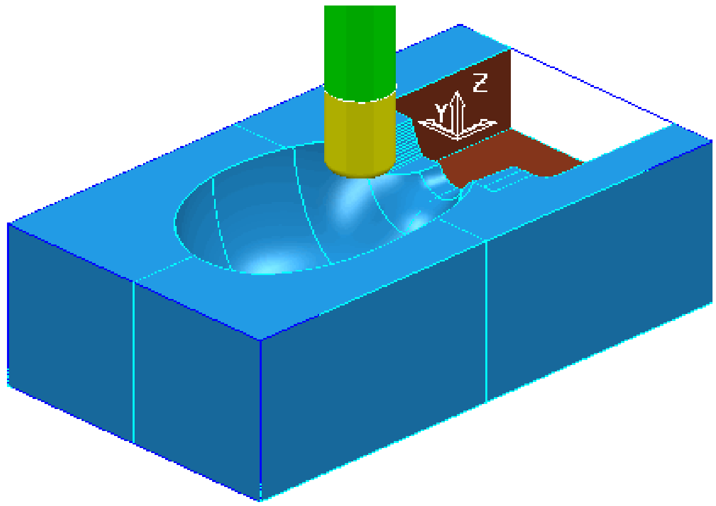
Model Rest Area Clearance example

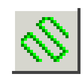
It is generally good practice to use as ***larger diameter*** **tool** as possible for the initial **Area Clearance** operation. This ensures that the maximum amount of material is removed as quickly as possible. In many cases however the ***larger diameter*** **tool** may not have full access to certain internal corners or pockets within the component. As a result these areas will require further **roughing** out with one or more, **smaller diameter tool** before sufficient material is removed prior to running the **Finish Machining** strategies.
In the **Model Rest Area Clearance** options a **smaller diameter tool** is ***referenced*** to a previously created ***machining strategy*** such that ***tool tracks*** will only be produced locally within the ***remaining material*** (**stock**).

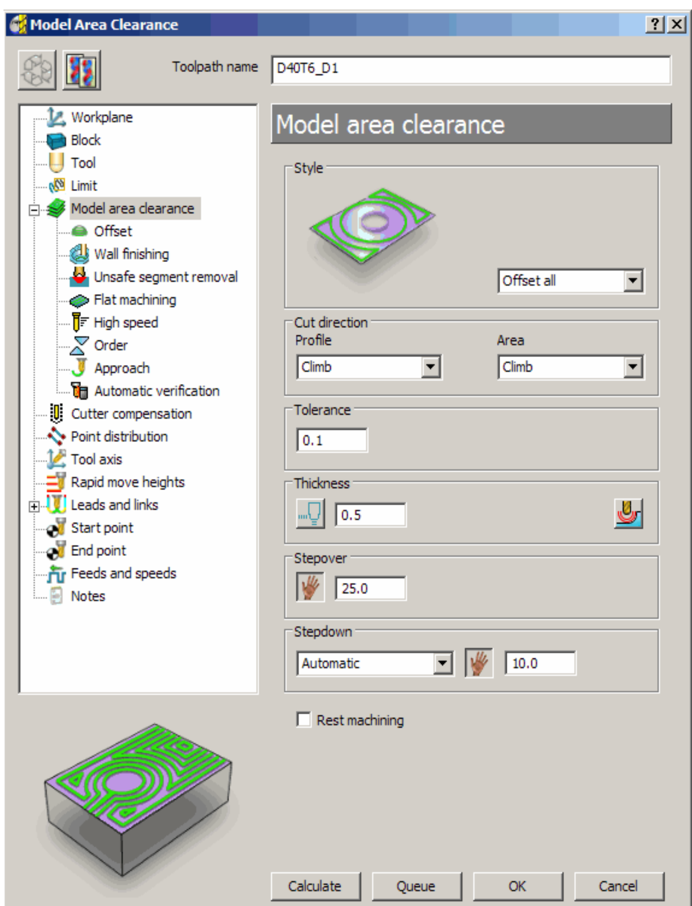
- **Import** the **model**:-
C:\users\training\PowerMILL_Data\models\ WingMirrorDie.dgk



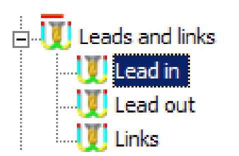
- Create a **Tip Radiused** tool of **Dia 40 tiprad 6** and ***Name*** **d40t6**.
- From the **Main toolbar** open the **Block** form and **Calculate** using the ***Defined by*** **- Box** to the full **model** dimensions.
- **Calculate** the **Rapid Move Heights**.
- In the **Start and End Point** form set the ***Start Point*** to **Block Centre Safe** and the ***End Point*** to **Last Point Safe**.



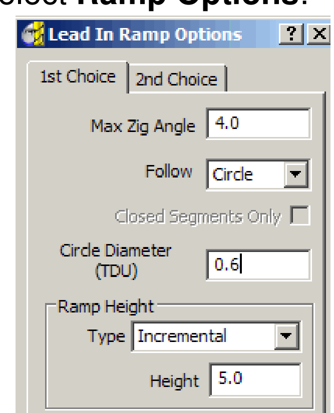
- From the **Main** toolbar select the **Toolpath Strategies** icon .
- In **3D Area Clearance** select the option **Model Area Clearance** to open the following form.



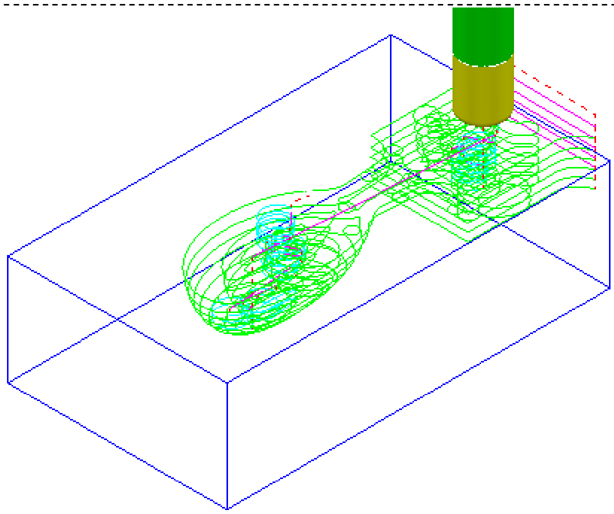
- Enter ***Name*** **D40T6_D1**.
- Select ***Style*** - **Offset all**
- From the ***explorer*** - **Leads and Links**, select **Lead In**.



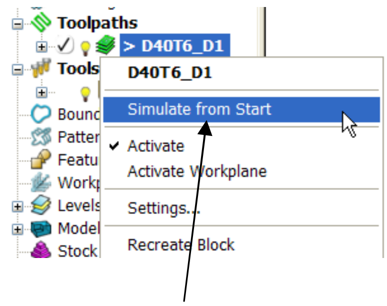
- In the **Lead in** window select ***1st Choice*** - **Ramp**
- Select **Ramp Options**:-



- Click **Calculate** to process the above specified **Area Clearance** toolpath.

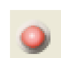
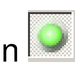


The **Offset all - Area Clearance** strategy follows both the contours of both the **Model** and **Block** then gradually offsets into the remaining material at each **Z Height**.



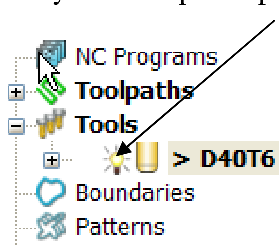
- **Right mouse click** on the toolpath and select **Simulate from Start** to open the **Simulation** toolbar.

- Ensure that the **Viewmill** toolbar is also active.

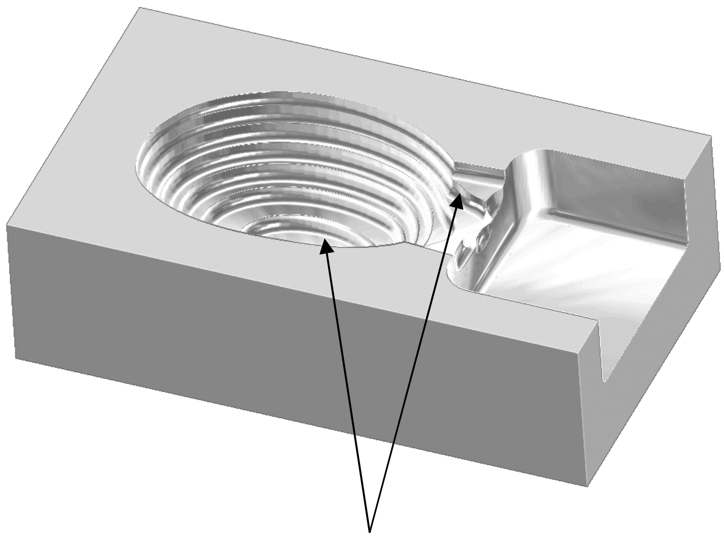
- Switch the **ViewMill On/Suspend** icon from red  to green  on the **ViewMill Toolbar**, followed by the **Shiny Shaded** image icon .

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

The simulation of the toolpath will start with tool displayed, but this can be controlled by toggling the light bulb on the tool entity in the explorer pane.



Note: Undrawing the tool will speed up the simulation.

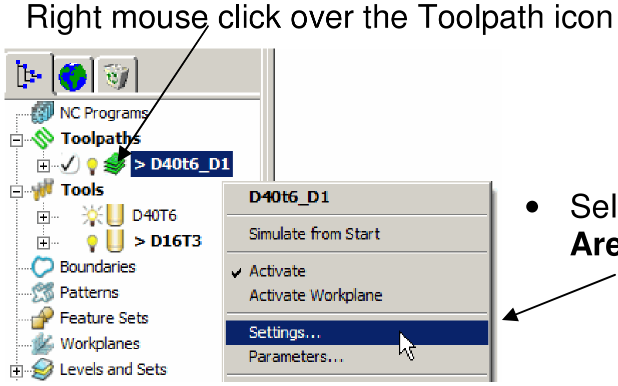


The finished result indicates that the current tool has not been able to remove enough material from some areas of the model.
As a result a further **Area Clearance** strategy is required using a smaller tool to continue locally into the remaining areas.
This technique is known as **Rest Machining**.

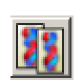
- Suspend **ViewMill** 
- Switch off the ***shaded image***. 

Rest Roughing using a Reference Toolpath

- Create a **Tip Radiused** tool of **Dia 16**, **tiprad 3** and *name* **D16t3**.
- Right mouse click over the Toolpath icon in the **explorer**.

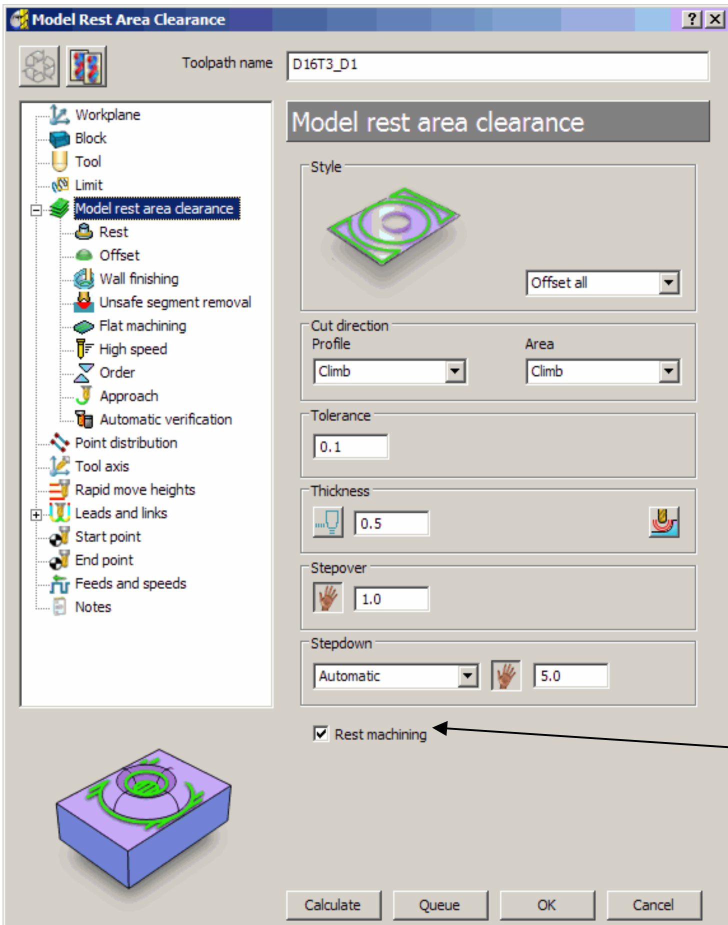


- Select **Settings** to reopen the **Model Area Clearance** form.

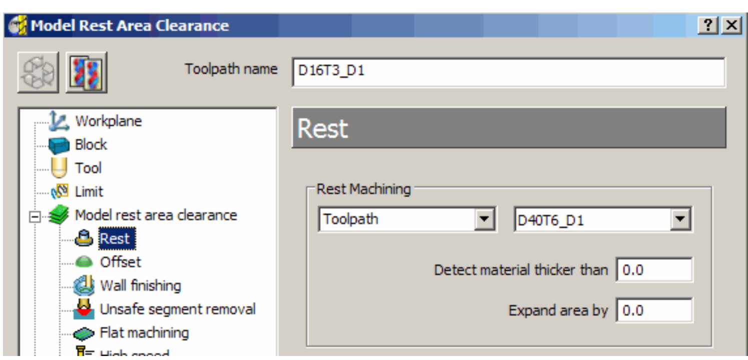
- Select the '***make a copy of the toolpath***' icon .

Note: All settings originally used to create the toolpath will become re-activated.

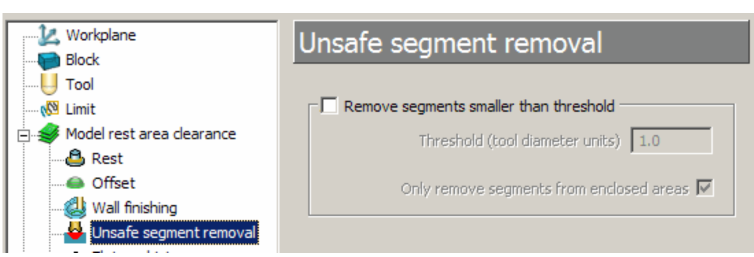
- **Activate** the new tool, **D16t3**.



- Enter a new name **D16T3_D1**.
- Enter:- ***Thickness* 0.5**
  ***Stepover* 1.0**
  ***Stepdown* 5.0**
- **Tick** the box labelled **Rest machining** to display the **Rest** option in the **explorer**.

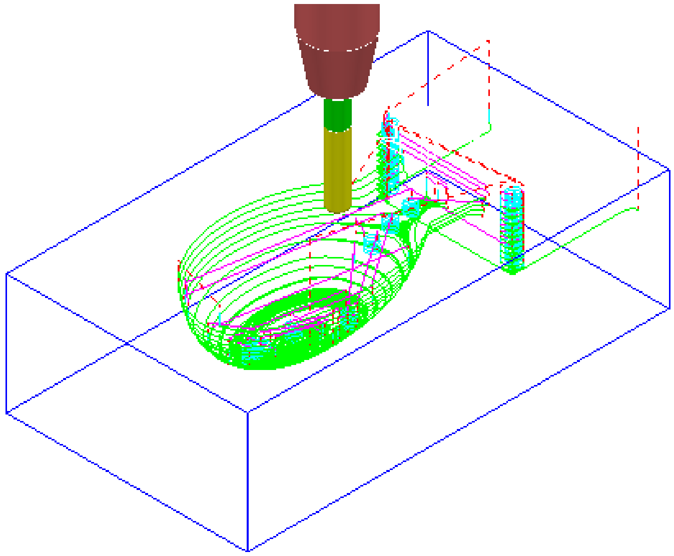


- Select the ***Rest Machining*** type as **Toolpath** and enter **D40T6_D1** as the reference item

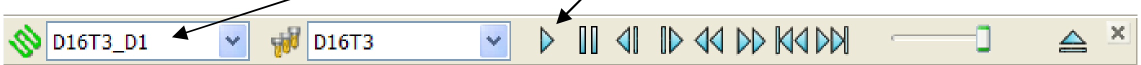


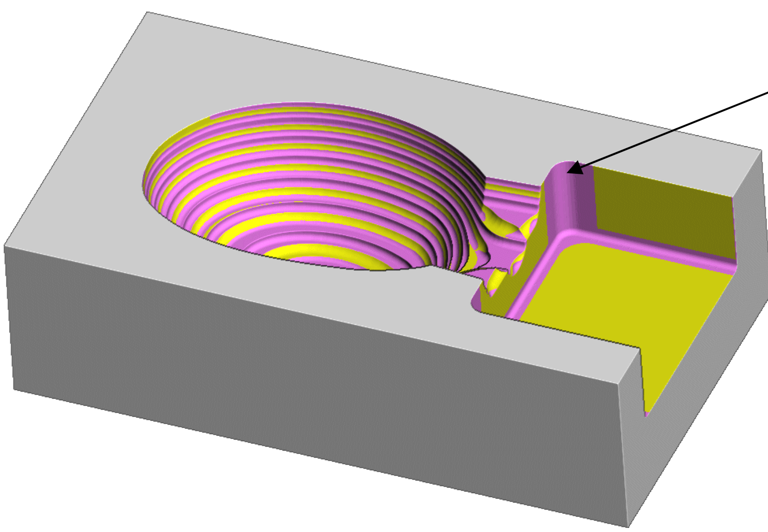
- In the **explorer** select **Unsafe segment removal**.
- Untick **Remove segments smaller than threshold**.

- Select **Calculate** to create the additional **Offset Area Clear** toolpath shown on the following page.
- **Cancel** the form.



- ***Save Project as***:- **C:\users\training\COURSEWORK\PowerMILL-Projects\Wing_Mirror_Die**.
- Turn **ViewMill On** (Green) → .
- Select the **Rainbow Shaded Image** .
- Select the toolpath **D16t3_D1** and **Play** the simulation.





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

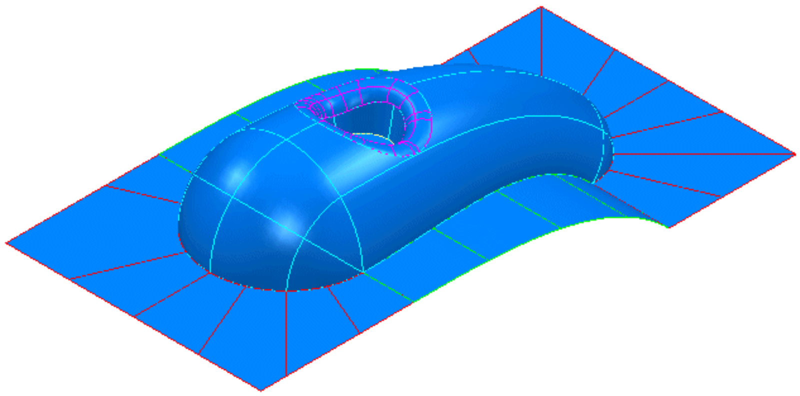
The **Reference Toolpath** has roughed out material closer to the component form. This will reduce the risk of excessive wear or damage to tools used for the subsequent finishing operations.

- Select the **Suspend ViewMill** icon to return to **PowerMILL**.
- From the **Main** toolbar select **File** - **Delete All**
- From the **Main** toolbar select **Tools** - **Reset Forms**.

EXERCISE:-

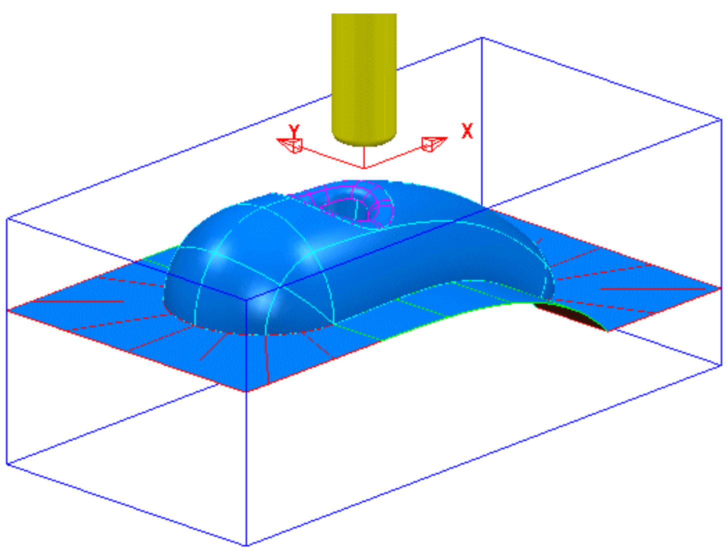
Area Clearance with both Unsafe Segment Removal and Rest Roughing

- **Import** the **Model**:-
  C:\users\training\PowerMILL_Data\Models\Cowling



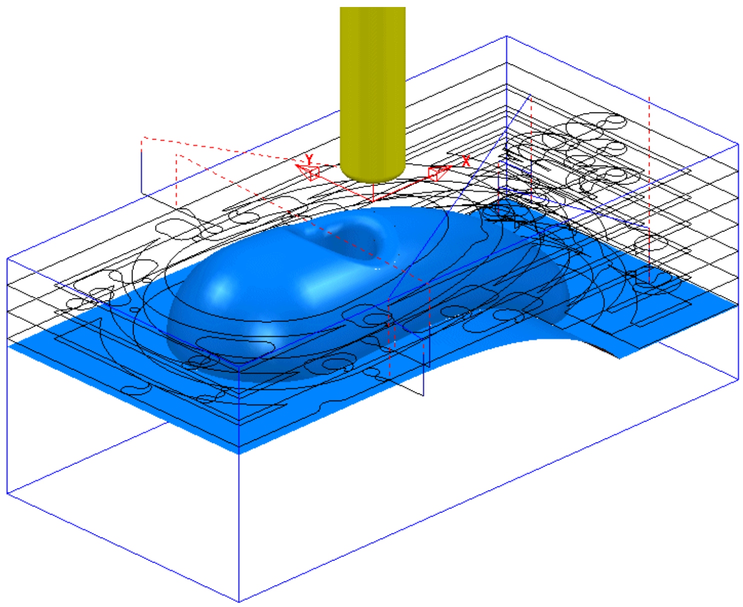
Machining Set Up

- Create the following **Tools**:-
  Dia 20 tiprad 3 named **D20t3**
  Dia 12 tiprad 1 named **D12t1**
- **Create** and position a **Workplane** centrally positioned to the **top centre** of the **model** as a more suitable machining datum.
- Define a suitable **Block** using ***Defined by*** - **Box** to ***Type*** - **Model**.
- Apply **Calculate** in the **Rapid Move Heights** form.

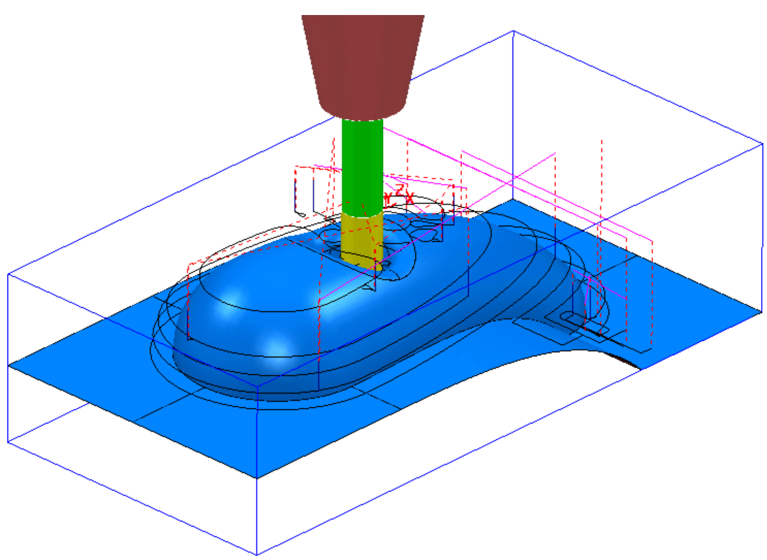


Area Clearance Strategies

- Create an **Offset All** - **Model Area Clearance** strategy using the **Unsafe segment removal** options to ***prevent*** the ***D20t3*** **tool** from attempting to machine into the ***central pocket***. Use a **Thickness** of **0.5**.



- Create another **Offset All** - **Model Area Clearance** strategy this time with the **D12t1** tool.
- Before calculation switch off the **Unsafe segment removal** option and switch on the **Rest Machining** options switched on, ***referenced*** to the previous toolpath.

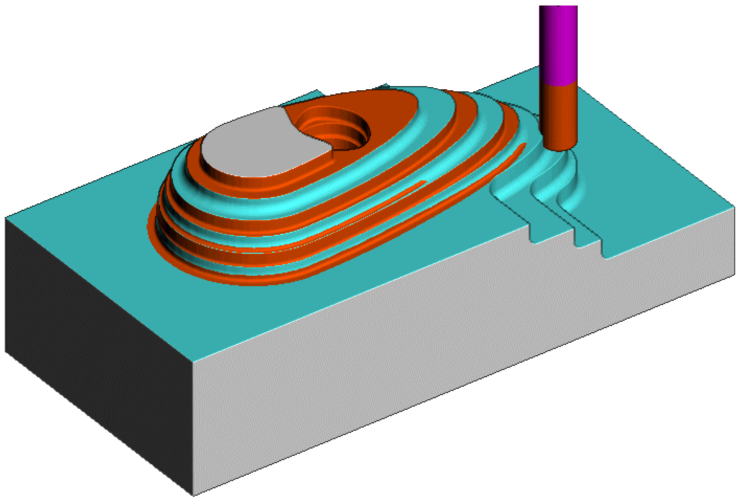


- **Save Project As**:-

C:\users\training\COURSEWORK\PowerMILL_Projects\ModelAreaClear-Exercise

ViewMILL Simulation

- Perform a **ViewMILL *simulation*** on both of the **Model Area Clearance** toolpaths with the **Rainbow** shading option active. This will enable the user to ***see exactly*** where the ***second toolpath (different colour)*** is removing further material from the partially machined block.



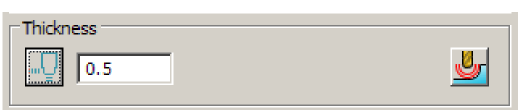
The ***Rainbow*** **ViewMILL *Shading*** option displays the resultant material removal for each **toolpath** as a different colour.

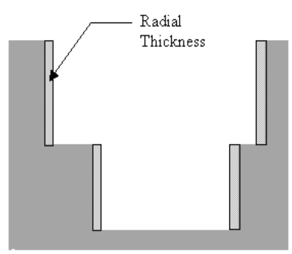
- From the **Main** pull down menus select **File** - **Delete All**.
- From the **Main** pull down menus select **Tools** - **Reset Forms**.

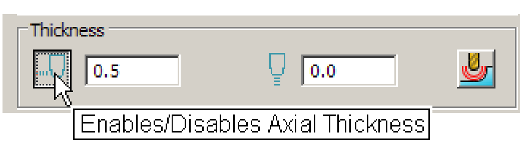
General information on Area Clearance Machining

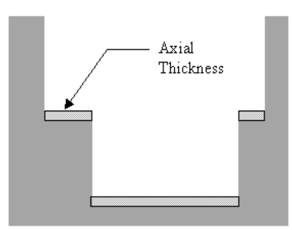
The following is reference information for the many different options contained in the Area Clearance form. This can also be found by using **Help**.

Thickness









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.

Z Heights – Manual Input

If **Stepdown** is set to **Manual** on the **Model Area Clearance** form the **Area Clearance Z Heights** form will open. In here are five alternative **Defined by** options:- **Z Heights**:- **Number**, **Stepdown**, **Value, Intermediate** and **Flat**.

Number - divides the block equally into the defined number of **Z Heights**.

Stepdown - creates a **Z Height** at the base of the **Block** and then steps upwards using a fixed incremental value. If active the ***Tick Box*** **- Constant Stepdown** causes the distance between all levels to remain constant. To achieve this, the actual **stepdown** value will more than likely have to be automatically, modified to be ***less than*** the **nominal value**.

Value - creates a single **Z Height** at the defined **value**.

Flat - Identifies **flat areas** of the model and creates a **Z height** (+ thickness) at these values. It can also be set to machine the specific **flat areas** instead of the ***whole of the slice***.

Intermediate - adds a specified number of **Z Heights** between the current ones.

Appending Z Heights - Z Heights can be also be cloned from an existing **Model Area Clearance** toolpath from the **Reference** section of the **Area Clearance Z Height** form.

.

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** - *Model 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.

When

This option is in the **Raster** form accessed from **Model Area Clearance** explorer. It allows the user to determine when the profiling pass occurs during machining.

- **None** – No Profiling pass is performed
- **Before** – The Profiling pass is performed ***before*** the ***Raster*** path.
- **During** – The Profiling is performed at suitable intervals ***during*** the Raster path
- **After** – The Profiling pass is performed last.

Cut Direction

This controls the direction of the tool along the toolpath:-

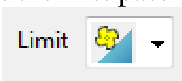
- **Any –** This instructs the cutter to machine in both directions.
- **Climb –** This will instruct the cutter to profile such that the material is to the right of the direction of travel (**Climb Milling**).
- **Conventional –** This will the cutter to profile such that the material is to the left of the direction of travel (**Conventional Milling**).

Wall Finishing

This option is accessed from the **Wall Finishing** Page in the **Model Area Clearance** explorer. It allows the user to make an additional, ***final profiling pass*** to further reduce tool wear.

Allow tool outside block

The **Allow tool outside block** switch is located in the **Limit** page Of the **Model Area Clearance** form

This enables the first pass of an **Area Clearnace** strategy to be performed to the specified **Stepover**, 

rather than the ***full radius of the tool***. 

Flat Machining

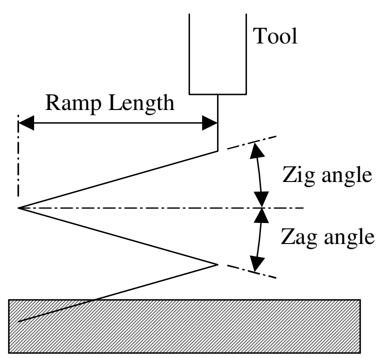
The **Flat Machining** page of the **Model Area Clearance** form contains options to control if and how the flat areas of a component are machined. These include **Off**, **Area**, and **Level**.

If **Level** is selected ***Flat Machining*** will occur across the whole area defined by the **Block**. The nominal **Stepdown** value will be locally reduced as required to accommodate each ***Flat areas*** as part of an equally spaced, group.

If **Area** is selected ***Flat Machining*** will be confined to the individuall **Flat areas**. Note: The nominal **Stepdown** value will remain constant (independant of the ***Flat areas***).

Ramping

This option is accessed via the **Leads and Links** form down to be applied 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 also be applied.

The **Ramp Length** is defined as '**Tool Diameter Units**' (**TDU**). For example, A **Ramp Length** of **2 TDU** would equal **20mm** for a **Dia 10mm tool**.

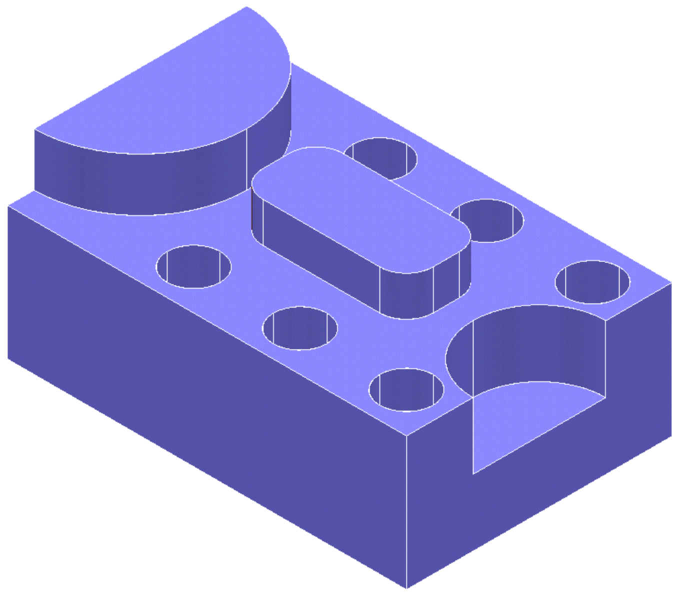
If **Approach Outside** is *set*, this will take priority over **Ramping**.

If the defined geometry for a **Ramp** move is such that it would cause a gouge, then the tool will approach using a default, **Plunge** move.

Machining Flats

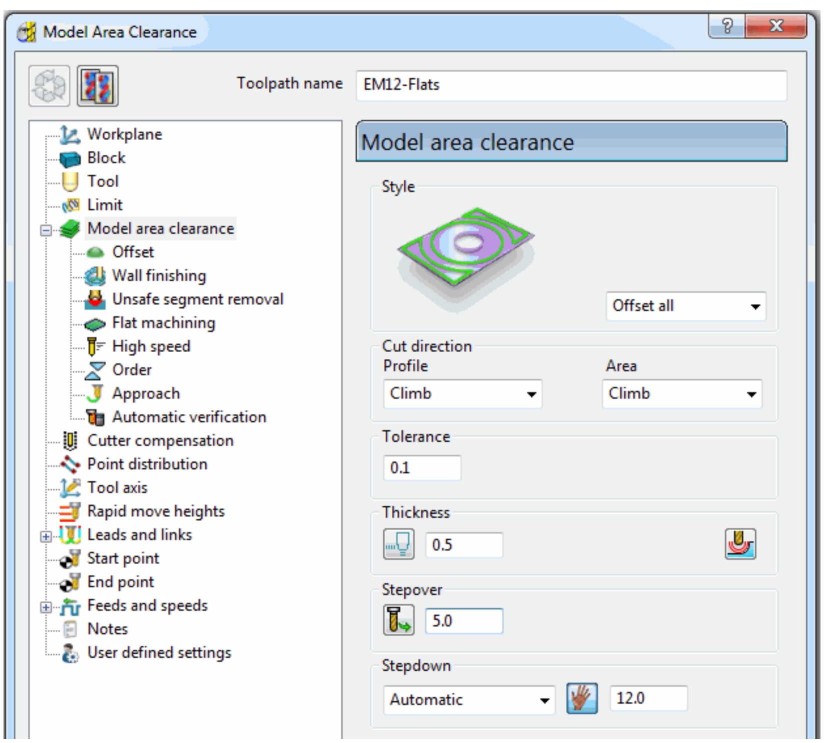
The **area clearance** strategies in **PowerMILL** have an option that allows the user to control the way in which flat areas of the model are rough machined. These are found on the area clearance form under **Machine Flats**.

- **Import** the **model**:-
  C:\users\training\PowerMILL_Data\Models\Flats.dgk

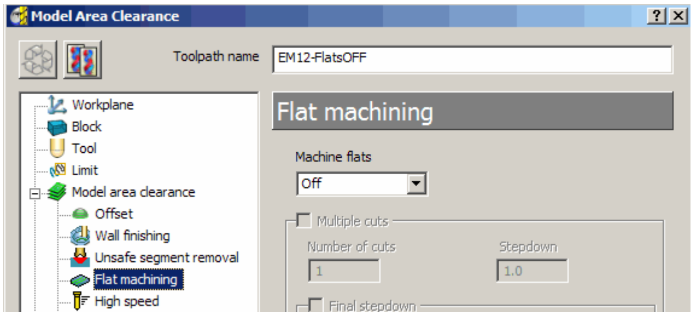


- Create a **12mm** diameter **End Mill** tool named **EM12**
- **Calculate** the **Block** using the default settings.
- Set the **Rapid Move Heights** and check **Start/End Point** is set to default; ***Start Point* - Block Centre Safe** and ***End Point* - Last Point Safe**.

- From the ***Strategy Selector*** - **3D Area Clearance** options, select **Model Area Clearance**.
- Fill in the form exactly as below.

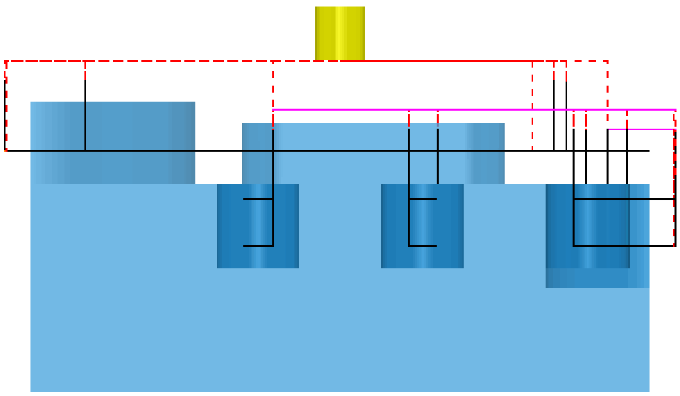


- Name:-
  EM12-Flats
- Thickness **0.5**
- Stepover **5**
- Stepdown **1**



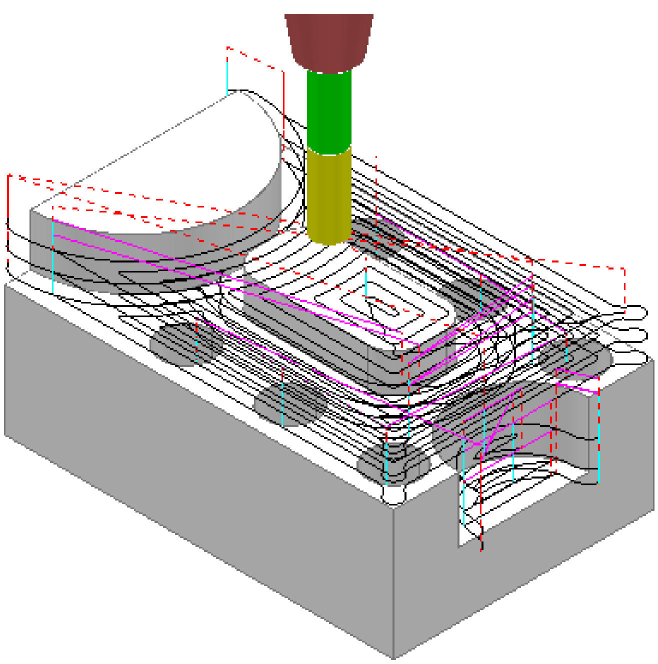
- In the **Model Area Clearance** explorer select the **Flat machining** options.
- Set **Machine flats** to **Off**.

- **Select Calculate** to process the **toolpath** and then **Cancel** the form



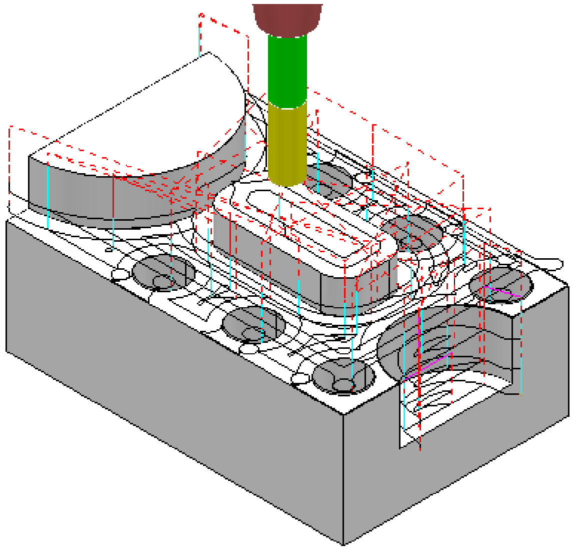
With **Machine flats - Off** the toolpath has totally ignored the ***flat surfaces*** of the **model**. It has maintained the defined, **Stepdown** value and performed ***area clearance*** across the material **Block** at each **Z Height**.

- Right mouse click over the **Active** toolpath and in the local menu select **Settings**.
- Select make a **Copy** of the **toolpath**.
- Change the **Flat machining** option to **LEVEL** (This is the default).
- Change the toolpath **name** to **EM12-Flats_Level**.
- **Calculate** to process the **toolpath** and then **Cancel** the form.



The **Model Area Clearance** toolpath now removes material across the ***whole block area*** leaving **0.6mm** stock above each **Flat** face (The **thickness** plus the **tolerance**).

- Right mouse click over the **Active** toolpath and in the local menu select **Settings**.
- Select make a **Copy** of the **toolpath**.
- Change the **Flat machining** option to **AREA**.
- Change the **Name** to **EM12-Flats_Area**.
- **Calculate** to process the **toolpath** and then **Cancel** the form.

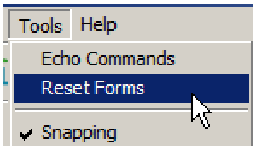


The **Model Area Clearance** toolpath now removes material locally across each **Flat** face leaving **0.6mm** stock (The **thickness** plus the **tolerance**).

- ***Save Project as:-***
  C:\users\training\COURSEWORK\PowerMILL-Projects\AreaClearFlats

PowerMILL Entity Recycler

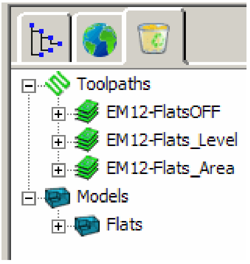
- In the **explorer** Right click over **Toolpaths** and select **Delete All** from the local menu.
- Use the same method to **Delete All** - **Models** (Do not apply to the **Tools** section in the **explorer**).
- From the **Main** pulldown menus select:- **Tools - Reset Forms**.



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

Note: The **Model** and **Toolpaths deleted** from the **explorer** can at this stage be recovered as they have been transferred to the ***PowerMILL*** **Entity Recycler**. If **File - Delete All** is applied from the **main** pulldown menus all items by-pass the **Entity Recycler** and as a result cannot be recovered. Note: Certain items such as **Stock Models** cannot be transferred to the **Entity Recycler**.

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



- Right mouse click over an item in the **Entity Recycler** 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** and **Models**. (Do not apply to the **Tools** section in the **explorer**).

PRO - 3D Area Clearance

Introduction

Projects Similar to those saved earlier during the (basic) **PowerMill** part of the course will be continued using previously unavailable options, exclusive to **PowerMILL Pro**.

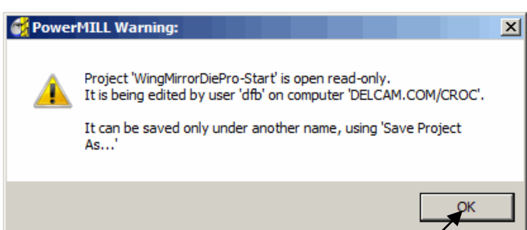
Rest Machining using a Stock Model

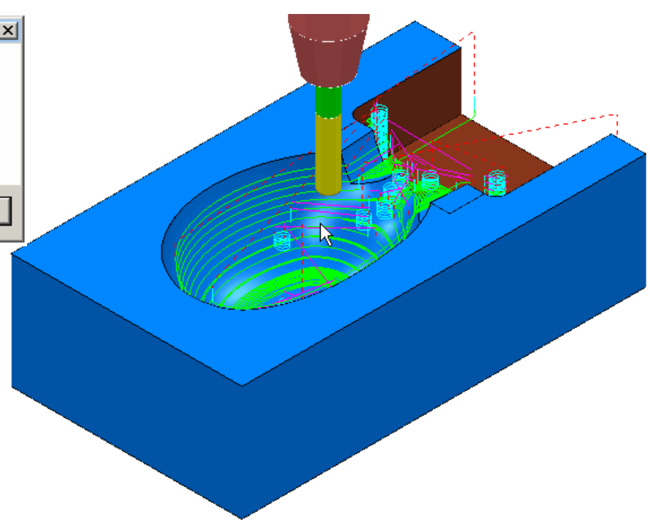
An alternative method of **Rest Machining** is the use of a **Stock Model** instead of a **Reference Toolpath**.
The benefits of using a **Stock Model** compared to a **Reference Toolpath** are:

- The remaining surplus material can be visualised and registered along with the associated toolpaths.
- A **Rest** machining strategy ***referenced*** to a **Stock Model** will account for all previous operations registered in that **Stock Model**.
- A **Rest** machining strategy created to different **Workplane** (eg **Multi Axis**) can be ***referenced*** to a **Stock Model**. (Note: It is not possible to apply a **Rest** machining strategy ***referenced*** to a **Toolpath** that was created to a different **Workplane**).

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**.

- **Open** the ***Project***:-
  C:\users\training \PowerMILL_Data\Projects\WingMirrorDiePro_Start.

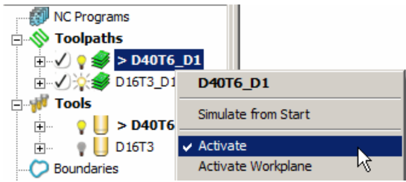




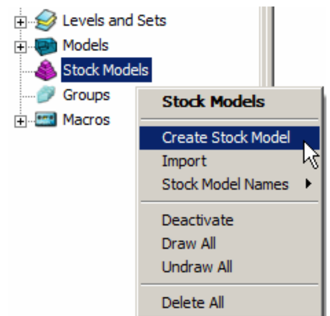
- Select **OK** in the ***PowerMILL Warning*** form.

- **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\WingMirrorDie

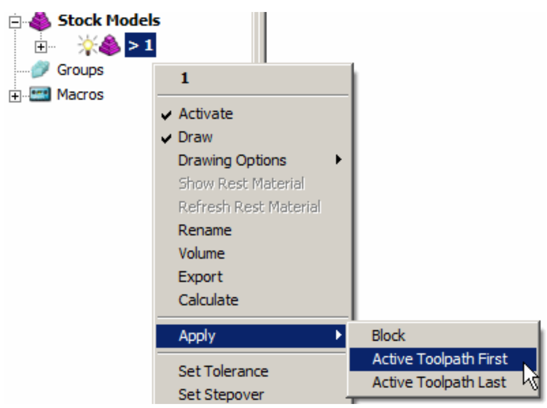
- In the ***PowerMILL* explorer**, **Activate** the toolpath **D40T6_D1**.



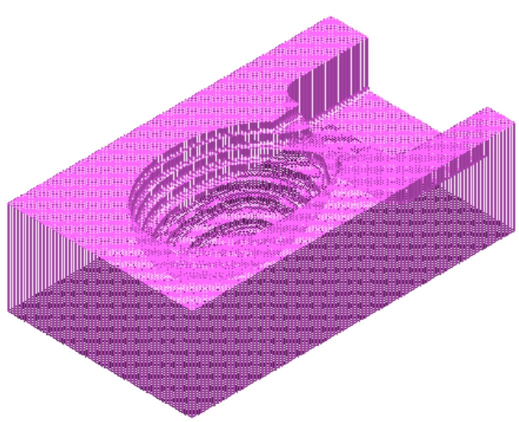
- In the ***PowerMILL* explorer** right mouse click **Stock Models** to open the local menu and select **Create Stock Model**.



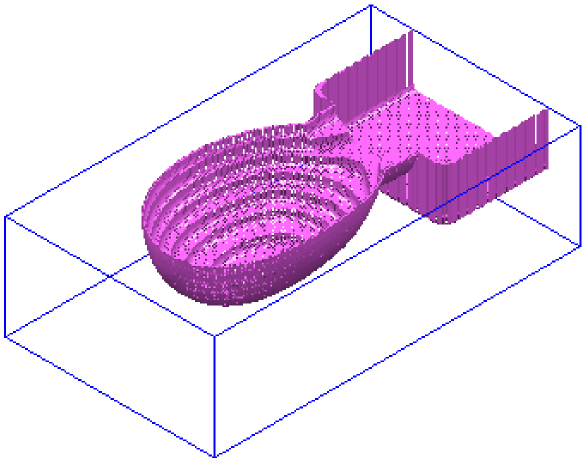
- In the ***PowerMILL* explorer** right mouse click the newly created **Stock Model** icon and from the local menu select **Apply** - **Active Toolpath First**.

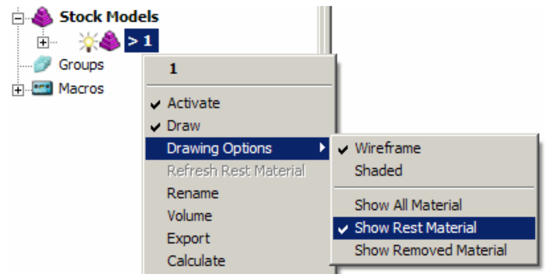


- From the same menu select **Calculate** to create the **Stock Model** as shown below.

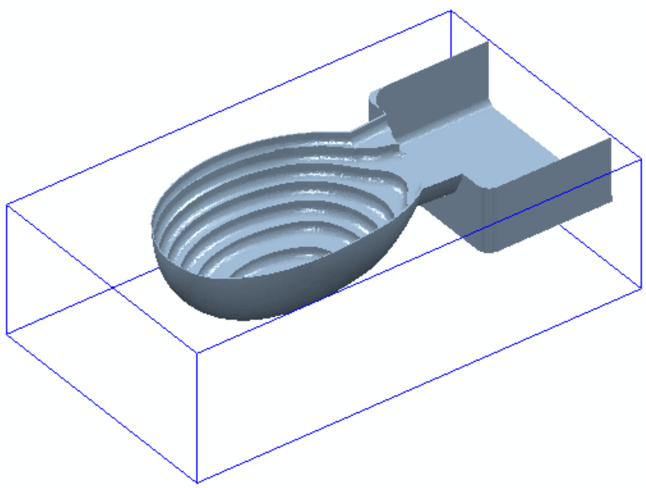


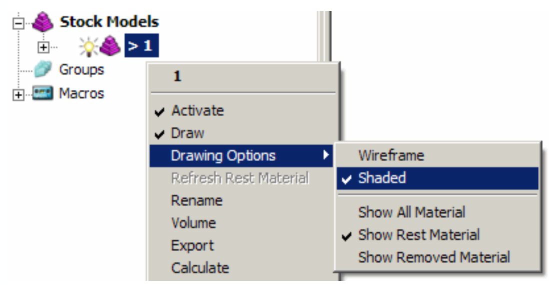
- From the same menu again select **Show Rest Material** to display the **Stock Model** as shown below.



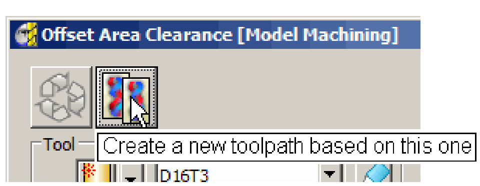


- From the same menu again select **Drawing Options** - **Shaded** to display the **Stock Model** as shown below.

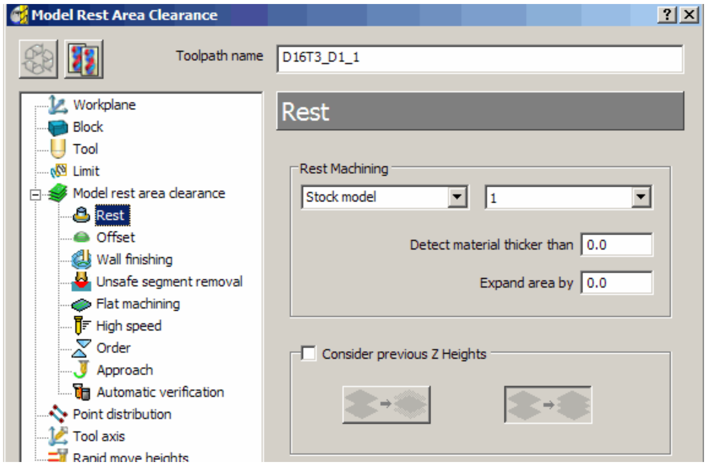




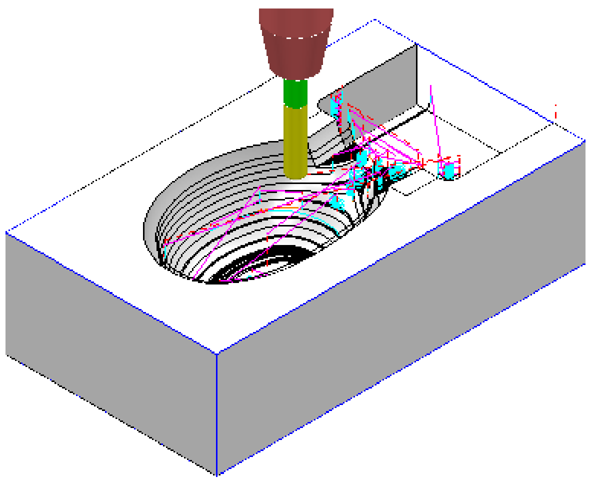
- In the ***PowerMILL* explorer** - **Activate** the toolpath **D16T3_D1**.
- Right click the **D16T3_D1** toolpath icon and from the local menu select **Settings** to reopen the **Model rest area clearance** form.
- Select the '**Copy toolpath**' icon (shown arrowed below).



- In the local form's ***explorer*** select the **Rest** option and change the settings to use **Stock Model** - **1** as shown below.

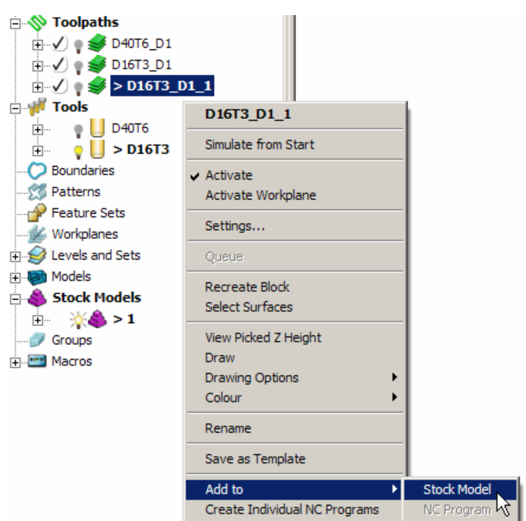


- Click **Calculate** to create an alternative (more efficient) **Rest Machining** toolpath named **D16T3_D1_1**.
- **Cancel** the **Offset Area Clearance** Form to reveal the following toolpath.



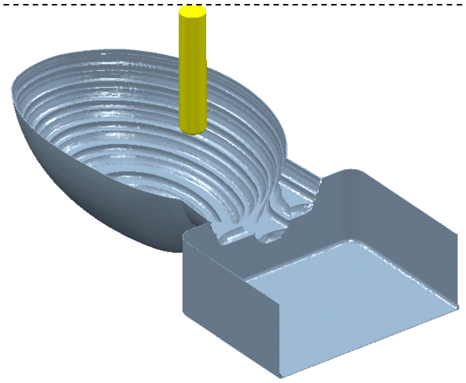
Although the **Stock Model** has been used to define the machining limits, the new toolpath is not automatically added to the **Stock Model**. This is carried out as a second operation if required.

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



It is also possible to right mouse click on the named, **Stock Model** in the ***PowerMILL*** **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 ***PowerMILL* explorer** and select **Calculate** to create the updated **Stock Model** as shown.



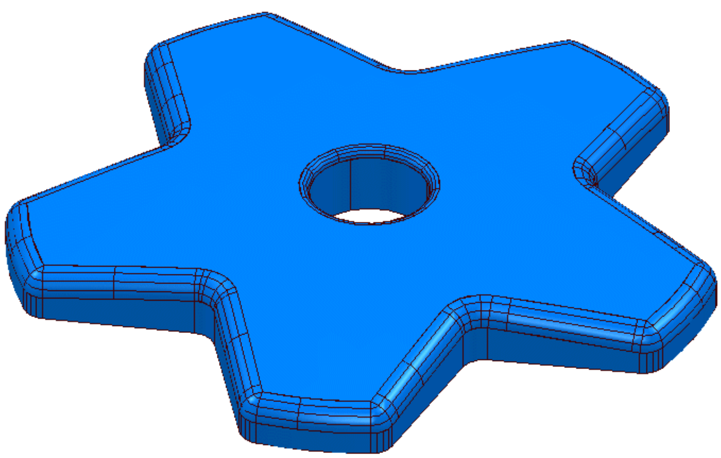
- From the **Main** toolbar select **File - Save Project** to update the **Project**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\WingMirrorDie

- **Delete** all entities in the ***PowerMILL*** **explorer** <u>except</u> the **D16T3** tool.

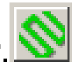
Offset Area Clearance – Offset Type Model

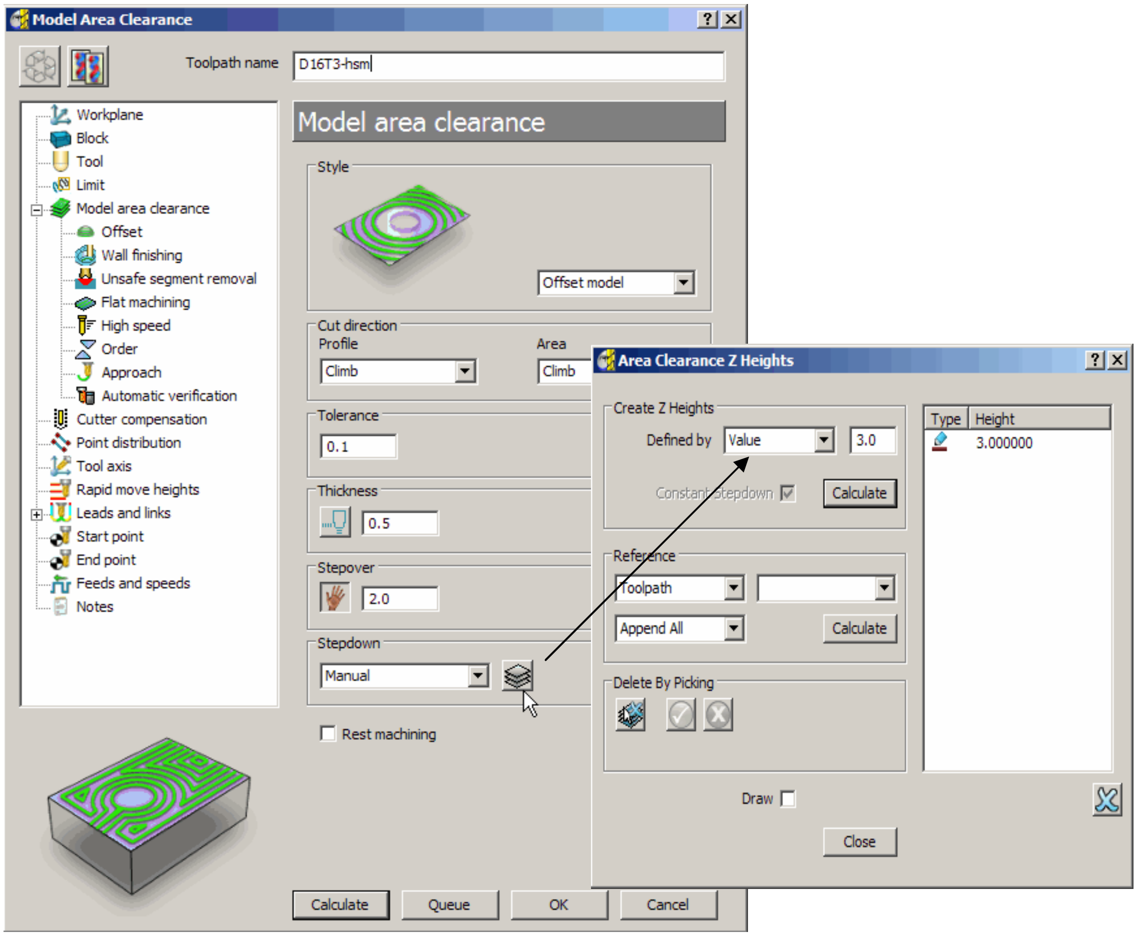
This variant of the **Offset Area Clearance** is mainly designed for ***High Speed*** machining. It provides a very consistent tool loading at the expense of an increased number of rapid, air moves across the component. This strategy if applied correctly will dramatically help to minimise wear to both the tooling and machine.
The strategy is based on the profile around the component at each **Z Height** being continually offset outwards to be trimmed to the limits of the material **Block**.

- **Save Project As:-**
  C:\users\training\COURSEWORK\PowerMILL_Projects\Handle
- **Import** the ***model***:-
  C:\users\training\PowerMILL_Data\models\Handle.dgk

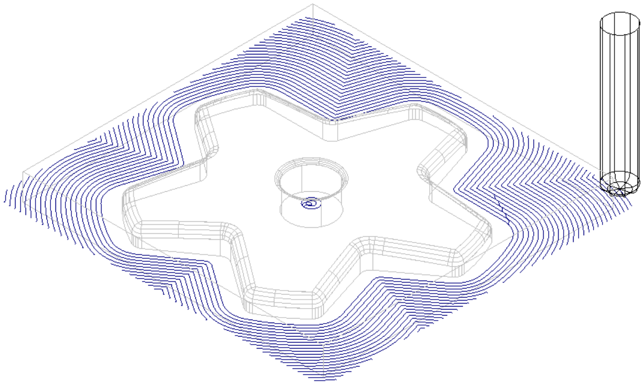


- In the **explorer - Activate** the **Dia 16 tiprad 3** tool **D16T3**.
- From the **Main** toolbar open the **Block** form and **Calculate** a material **Block** to the full **model** dimensions. **Lock** the **Z max** and **Z min** values, enter an **Expansion** value of **10** and **Calculate** again.

- In the **Rapid Move Heights** , click **Calculate**.
- From the **Main toolbar** click on the **Strategy Selector**. 
- From **3D Area Clearance** select the option **Model Area Clearance** to open the following form then enter data in the main form exactly as shown below.

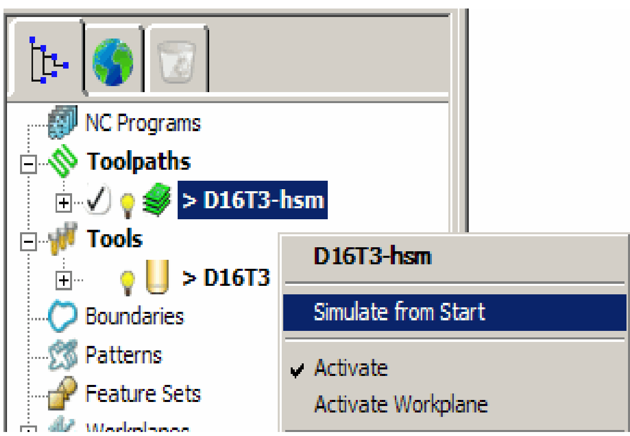


- In the **Model Area Clearance** form select **Stepdown** - **Manual** and click on the **Z Heights** icon  to open the **Area Clearance Z Heights** form.
- In the **Area Clearance Z Heights** form input ***Defined by*** - ***value*** - **3.0** and then select **Calculate** (Note: If **Z Heights** already exist, **Delete** them by clicking the ***blue cross*** located to the lower right of the form).
- **Close** the **Area Clearance Z Heights** form and then in the main **Model Area Clearance** form select **Calculate** to process the toolpath.



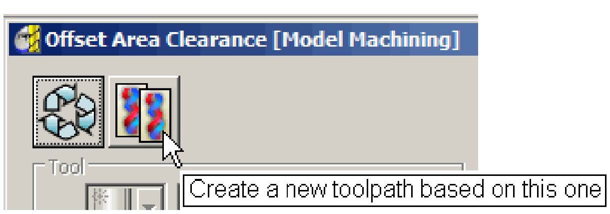
Note: The toolpath is shown with the **Leads and Links** undrawn.

- Select an **Iso 1** view as shown and right click over the toolpath icon in the **PowerMILL explorer** to open the local menu.
- Select **Simulate from Start** and in the **Simulation** toolbar select ***Play***  to observe the toolpath being run (If required, select the **Esc** key to stop the **simulation**).

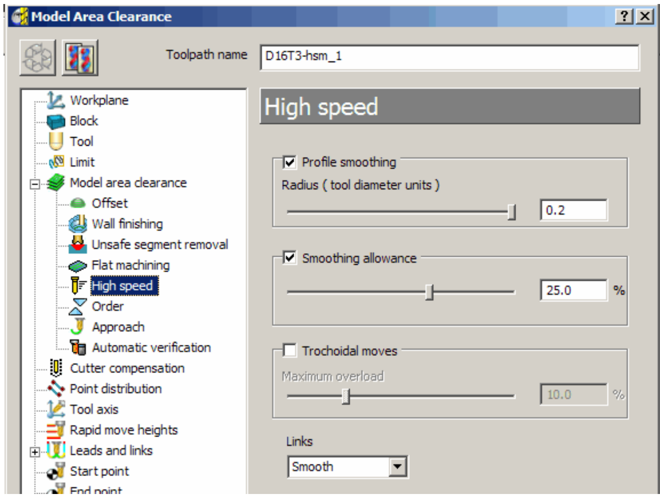


The tool will continuously **climb mill**, starting each tool track a distance from the material **block** to allow the **tool** to reach optimum **Feed Rate** before contact. Each pass will benefit from having a consistent material removal rate. The strategy can be further improved using advanced settings to progressively **smooth** away sharp corners and **straighten** tool tracks as they offset further away from the component form.

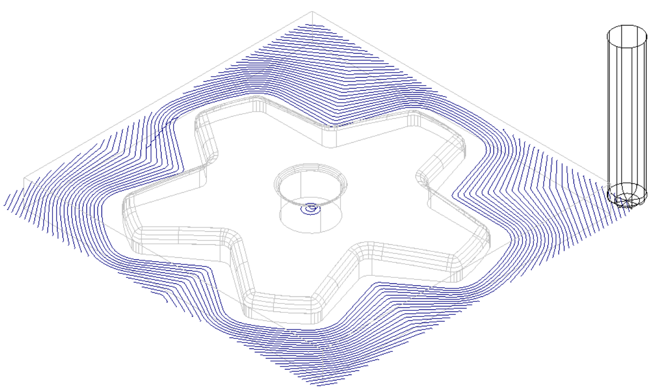
- Right mouse click over the ***Active***, **D16T2-hsm** toolpath icon in the **explorer** to open the local pull down menu.
- Select **Settings** to reopen the **Offset AreaClear Model** form.
- Select the '**Copy toolpath**' icon (shown arrowed below).



- With the **Model Area Clearance** form open again, select **High speed** from the local **explorer** options.



- **Tick** and set **Profile smoothing** to the maximum slider value of **0.2** (TDU).
- **Tick** the **Smoothing allowance** option and set the slider value to **25** (%).
- Click **Calculate** to create a new toolpath **d16t3_hsm1_1** with the improved **25% smoothing allowance** added.



Compare both of the **Model Area Clearance** toolpaths noting the progressive straightening of tool tracks on the second strategy (25% smoothing applied).

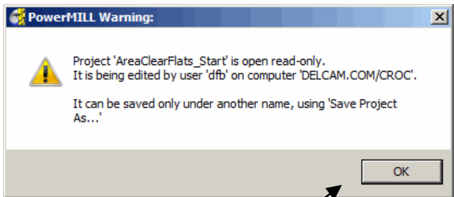
- From the **Main** toolbar select **File - Delete All** to delete all data from the current project.
- From the **Main** toolbar select **Tools Reset Forms**

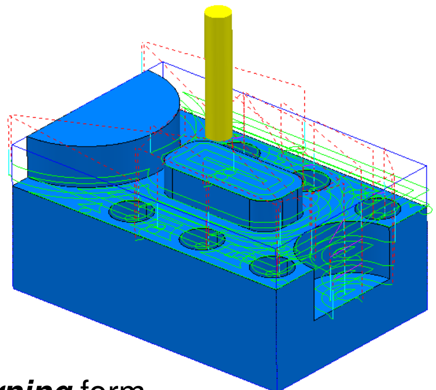
This restores the original **PowerMILL** settings in the forms without having to **Close** and restart.

Machining Flats

In **PowerMILL Pro** it is possible to locally **Area Clear** machine the flat areas to their exact height. This allows the user to rough the part first using Machine Flats set to OFF and then clear the flats in a second operation.

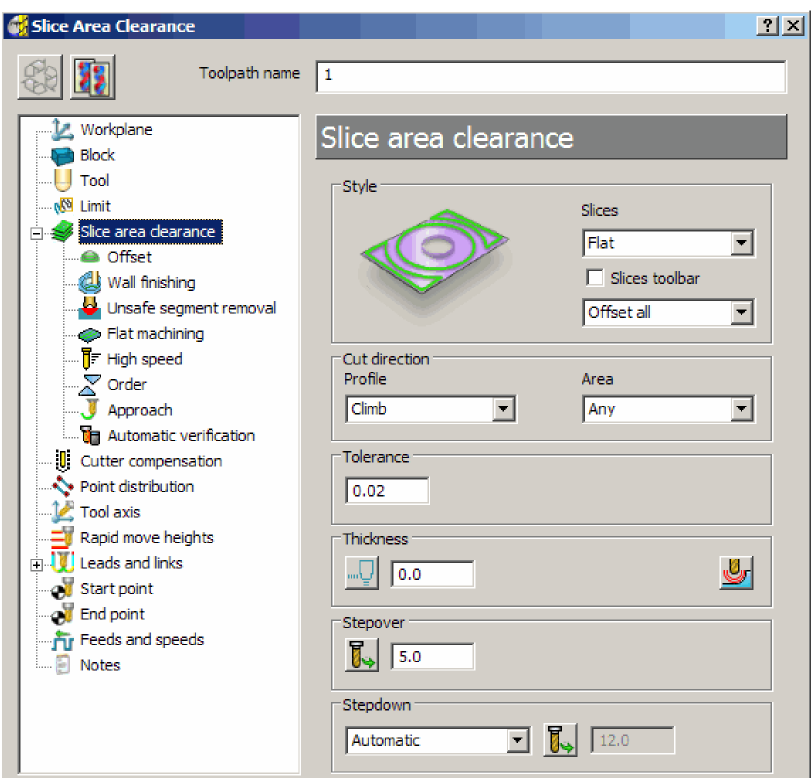
- **Open** the ***Project***:-
  C:\users\training\PowerMill_Data\Projects\AreaClearFlats_Start





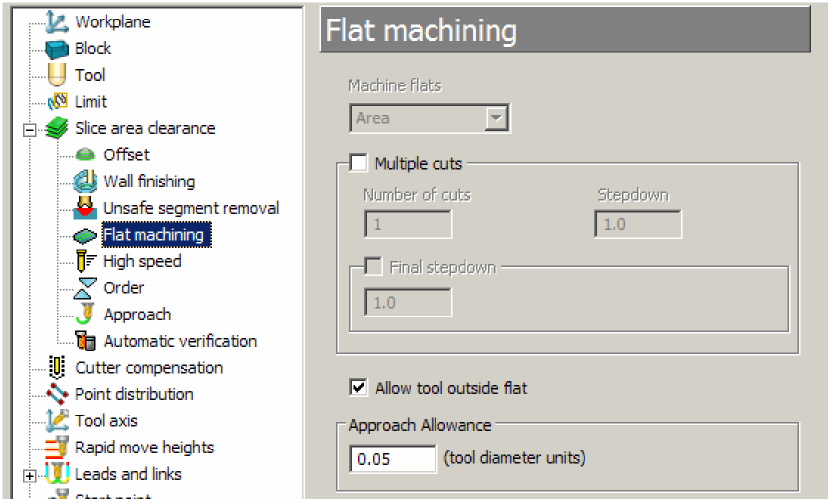
- Select **OK** in the ***PowerMILL Warning*** form.
- **Save Project As**:-
  C:\users\training\ COURSEWORK\PowerMILL-Projects\AreaClearFlats_Example

- **Right mouse click** on the toolpath **FlatsArea** and from the local menu, select **Activate** to reinstate all **settings** and **parameters** used.
- From the **Main toolbar** click on the **Toolpath Strategies**. 
- From the **3D Area Clearance** options select **Slice Area Clearance**.



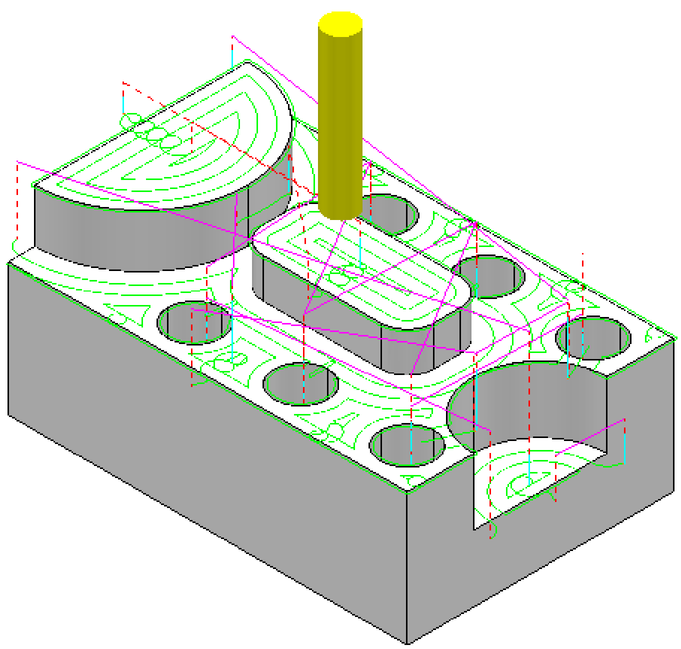
- Set ***Slices*** as **Flat**.
- Set **Offset all**.
- Set **Tolerance 0.02**
- Set **Thickness 0**
- Set **Stepover 5**

- In the local ***explorer*** select **Flat machining**.

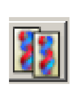


- Tick ***Allow tool outside flat***.

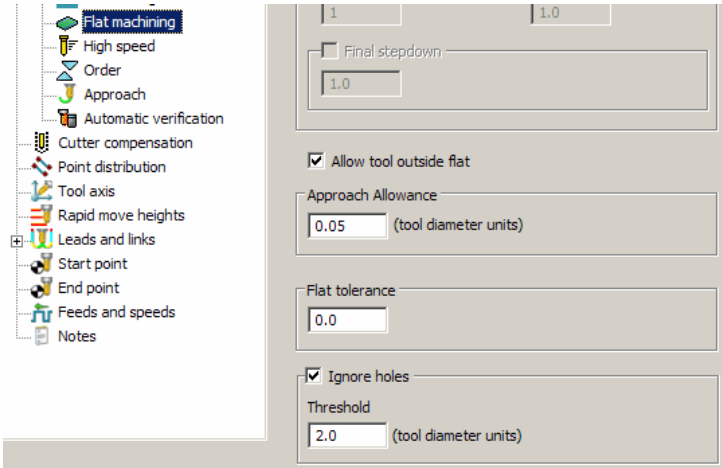
- Select **Calculate** to process the toolpath and then **Cancel** the form.



The **Area Clearance** now creates only the **Flat** areas.
At the moment the cutter profiles around the edge of the holes. If preferred the strategy can be changed to **Ignore Holes** causing the tool to pass straight over them.

- Right mouse click on the **Active** toolpath and select **Settings** to re-open the **Slice Area Clearance** form.
- Select make a **Copy** of the **toolpath**. 

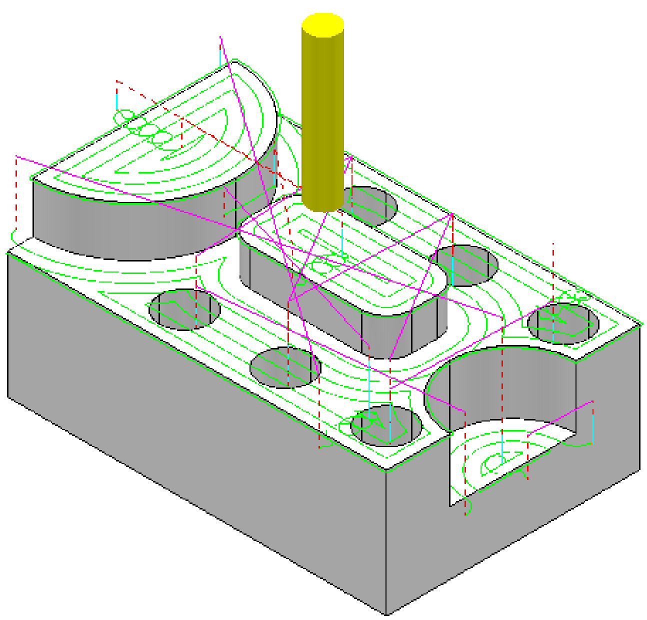
- In the **Slice Area Clearance** form, local ***explorer*** select **Flat machining**.



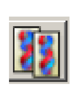
- **Tick** the box next to **Ignore holes**.

The **Threshold** in TDU (tool diameter units) informs PowerMILL the maximum size of holes to ignore. With the tool **EM12** selected and the **Threshold** set to **2**, PowerMILL will only ignore holes that are less than **Dia 24mm**.

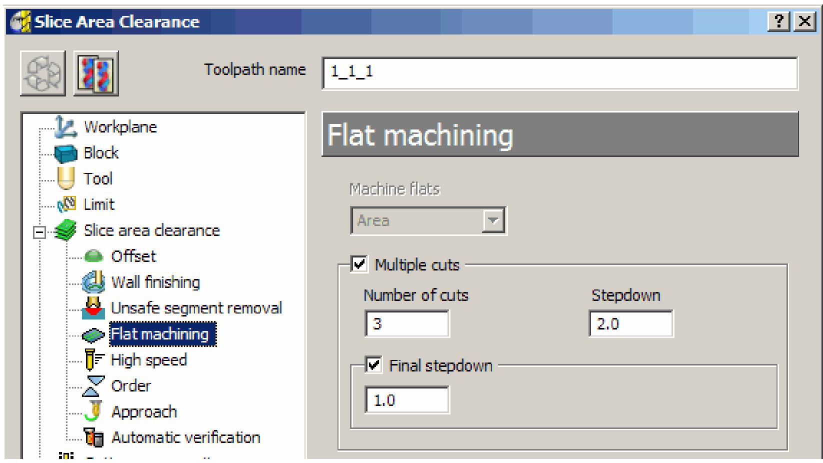
- Select **Calculate** to process the toolpath and then **Cancel** the form.



The cutter now passes across the top of the holes without lifting, producing a smoother toolpath. Note that **Ignore Holes** refers to the slice definition at the top of the holes. The flat base of any hole will still be machined.
If there is a large amount of material remaining on the top of the ***flat areas***, then several passes at different heights can be made to reduce the tool loading during the **Area Clearance**.

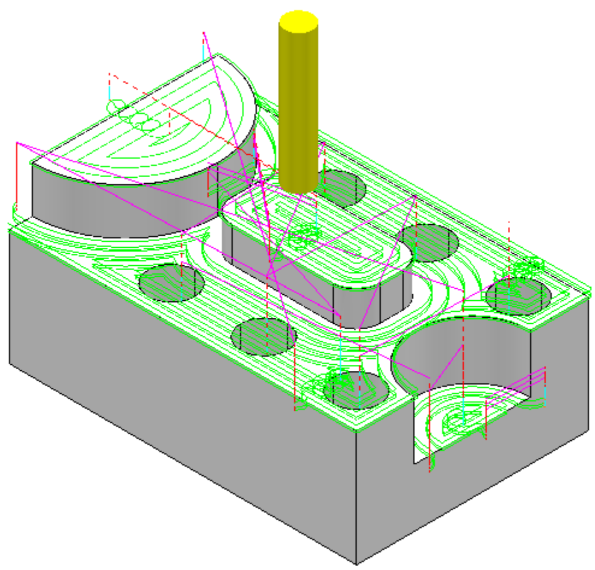
- Right mouse click on the **Active** toolpath and select **Settings** to re-open the **Slice Area Clearance** form.
- Select make a **Copy** of the **toolpath**. 

- In the **Slice Area Clearance** form, local ***explorer*** select **Flat machining**.



- Tick ***Multiple cuts*** with ***Number of cuts*** as **3**
- Set ***Stepdown*** as **2.0**
- Tick **Final stepdown** with the value **1.0**

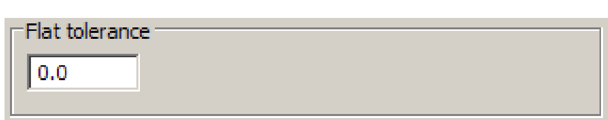
- Select **Calculate** to process the toolpath and then **Cancel** the form.



Note the 'D' shaped ***Flat*** area at the very top level is not using **Multiple Cuts** as the top of the **Block** is the overall **Z limit** for the strategy.

Flat Tolerance

It should be noted that by default, the **PowerMILL - 3D Area Clearance** strategies will only detect perfectly flat **surfaces**. **Surfaces** that are not quite flat dimensionally will not be recognised unless the **Flat Tolerance** is set with a suitable value to allow for the deviation. This option is found in the **Flats** page of the **Slice Area Clearance** form.



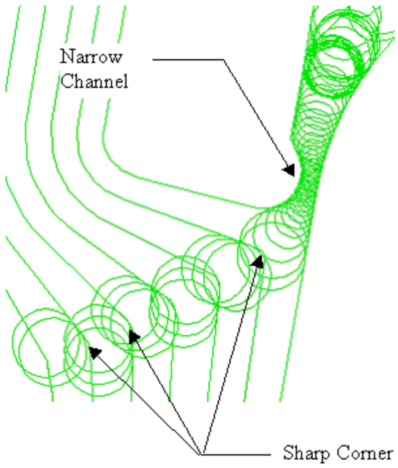
Note the **Flat tolerance** value entered will be added to the existing nominal **Z height** values.

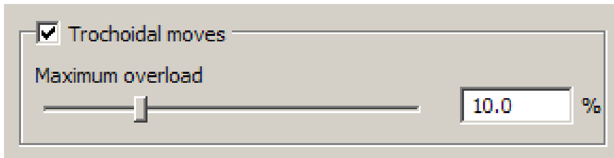
Further information on Model Clearance Machining

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

Restrict Tool Overload – Trochoidal moves (PowerMILL Pro)

This option is available to **Area Clearance** strategies in which ***Style*** - **Offset model** is set. The form for ***activating*** and ***inputting*** the **Maximum overload** value (%) is located on the **High speed** options page.





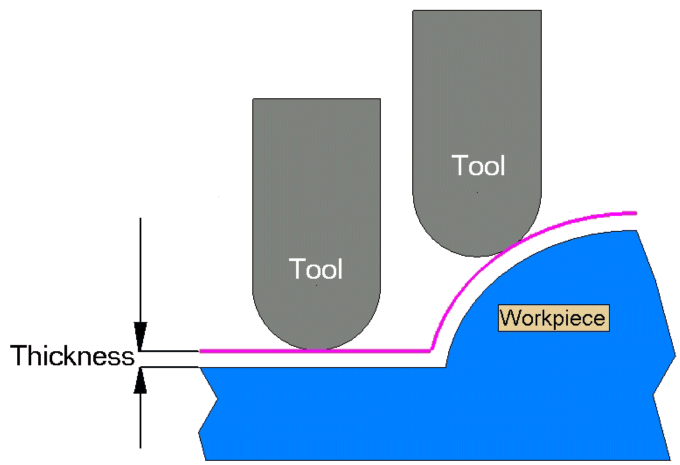
If the tool reaches a specified overload situation **PowerMILL** automatically inputs a **trochoidal** path to eliminate full width cuts. This will occur in corners, narrow channels, slots, etc and the degree of movement is controlled with the slider as a percentage value in the form.

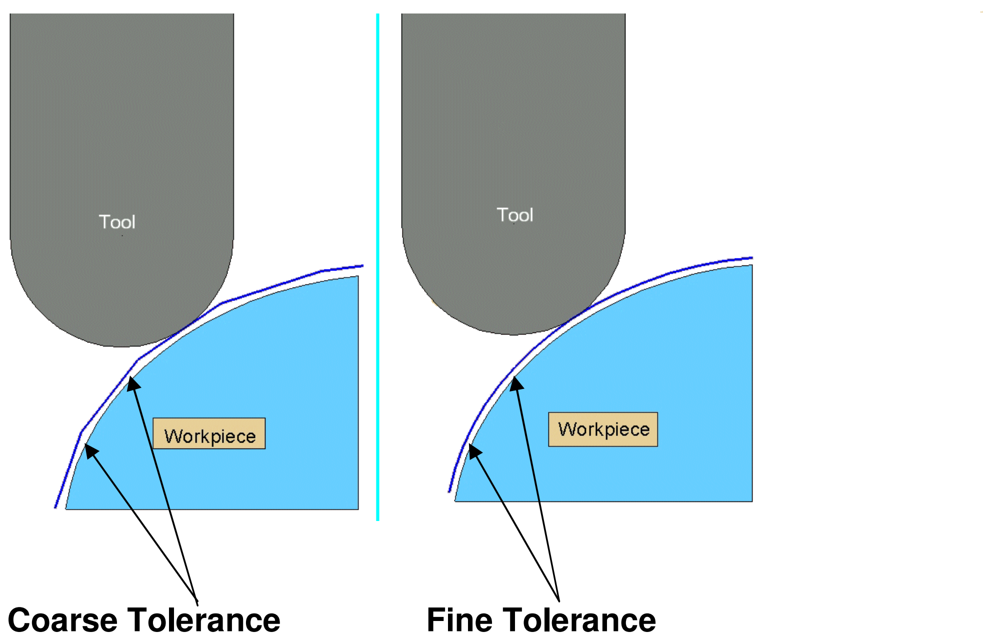
4. Finishing Strategies

Introduction to Semi Finishing / Finishing Strategies

Finishing strategies machine the actual component form and where applicable, follow on from the **Area Clearance** operation. 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 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.





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.

Note where the **Thickness** value is greater than **0** it should always be greater than the **tolerance** value

Raster, Radial, Spiral, and Pattern Finishing.

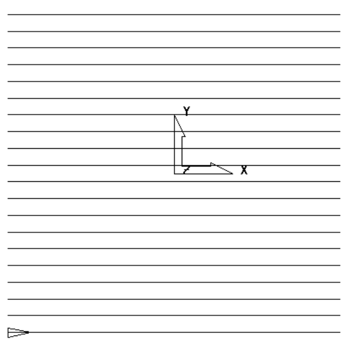
Introduction.

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

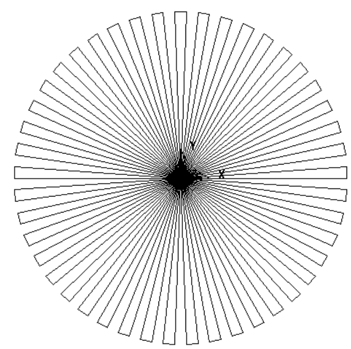
PowerMILL generates the toolpaths by projecting a wireframe form **down the Z-axis** onto the model. The standard patterns applied in **Raster**, **Radial**, and **Spiral** are achieved by entering values directly into the **Finishing** Form. The resultant **Pattern** can be displayed by selecting **Preview** before executing the command by selecting **Calculate**. The **Pattern** option requires a user-defined geometric form (active **Pattern**), which is projected down **Z** onto the model as a toolpath.

Typical previews of the four **Pattern** strategies are shown below as viewed down **Z**.

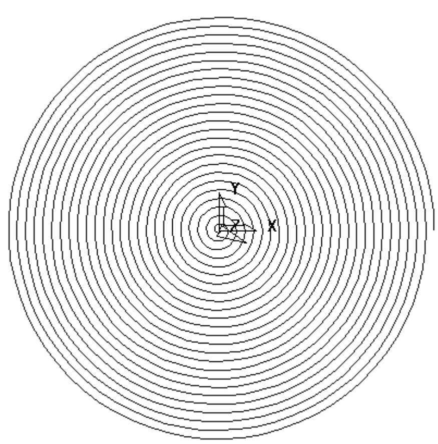
Raster



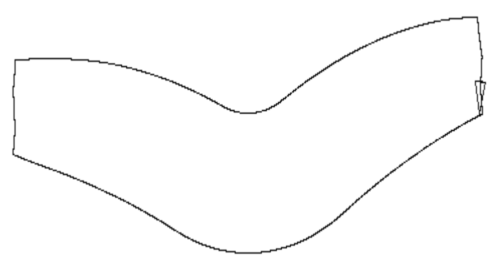
Radial



Spiral



Pattern (User Defined)



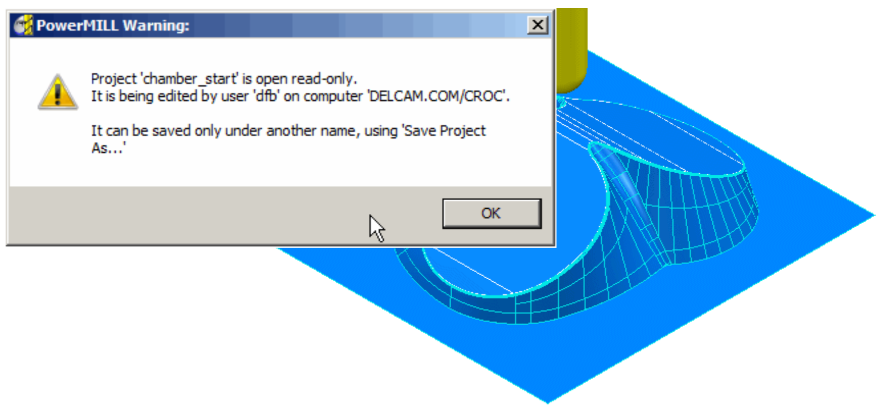
The most frequently used **Raster** finishing strategy will be used as an example in the next section.

Raster Finishing

- **Delete All** and **Reset forms**.
- From **File** select **Open Project** and browse the form, to **select** the ***Project:-*** **C:\users\train\PowerMILL_Data\Projects\Chamber_Start**.

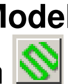
This time the example starts from an existing **Project**.

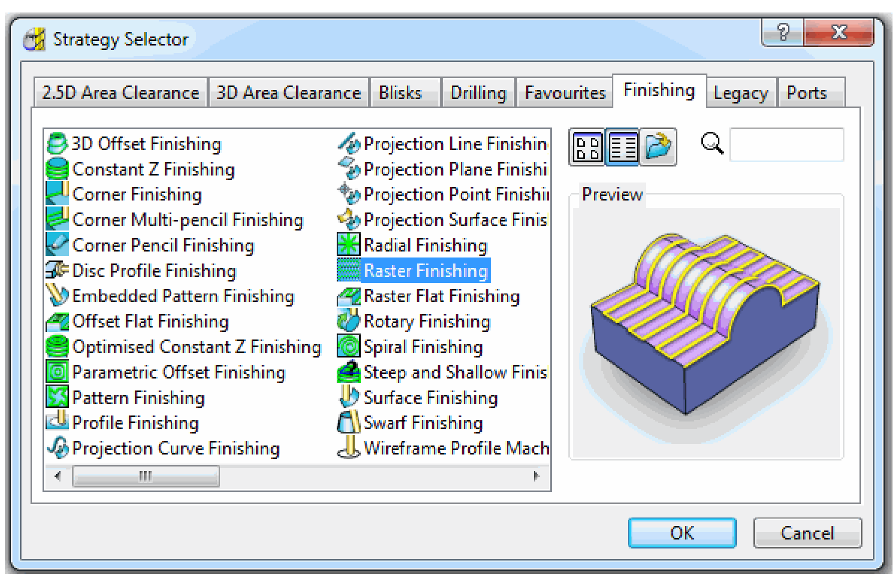
- Select **OK** on the form that appears informing that the source **Project** is **read-only**.



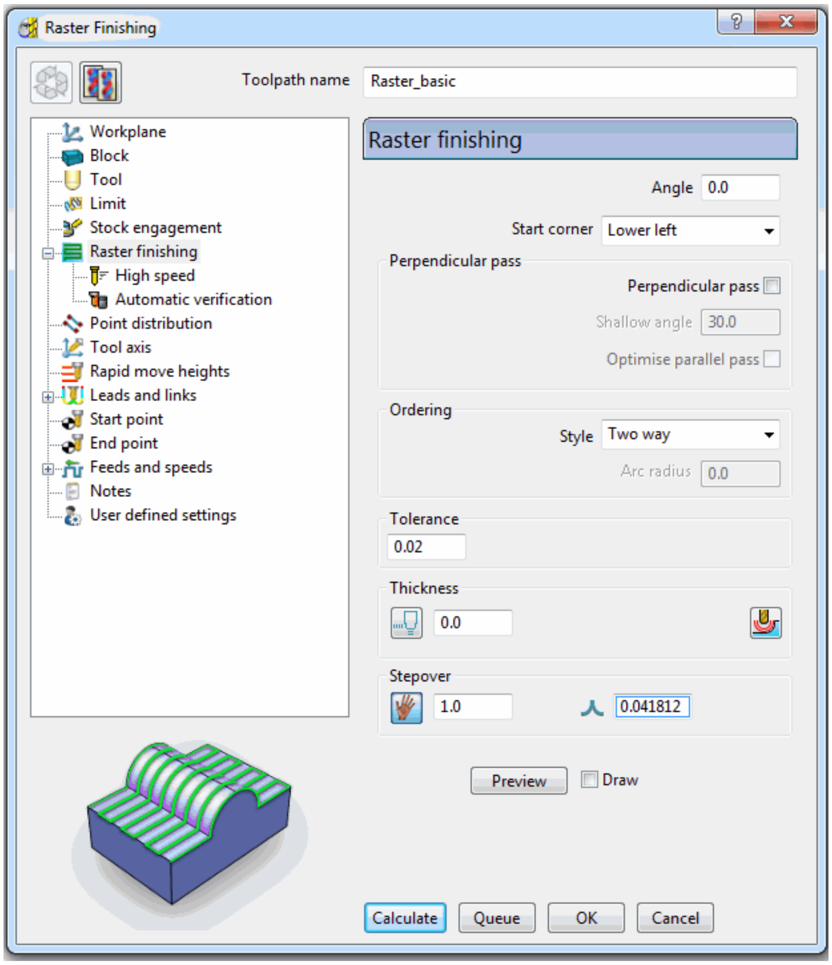
The model and tool stored in the imported **Project** are displayed.

- From **File** select **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\chamber

- Calculate a **Block** to default **Box - Model** settings.
- Select the **Toolpath Strategies** icon  from the top toolbar.



- From the **Strategy Selector** select the **Raster Finishing** icon then **OK**.



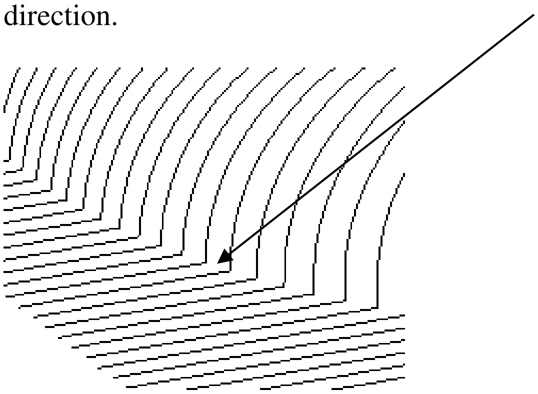
- Name the toolpath **Raster_basic**.
- Enter the **Tolerance** as **0.02** and **Thickness** as **0**.
- Enter **Ordering - Two Way**.

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**.

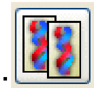
- **Calculate** to process the toolpath and then **Cancel** the Form.

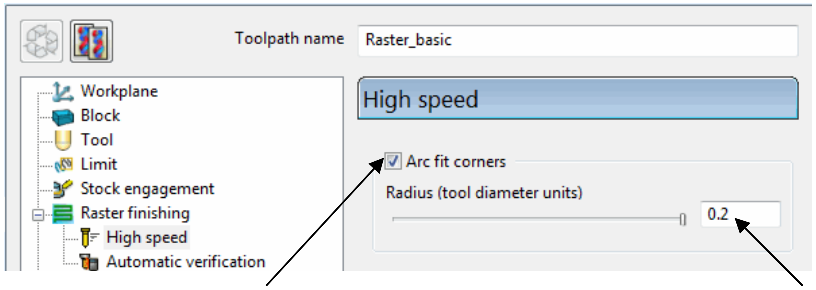
On closer inspection the toolpath contains **sharp corners** in along the vertical plane, direction.



Selecting the **High speed** page **Arc fit corners** option in the form will ***arcfit*** these to a maximum radius of **0.2** TDU.

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

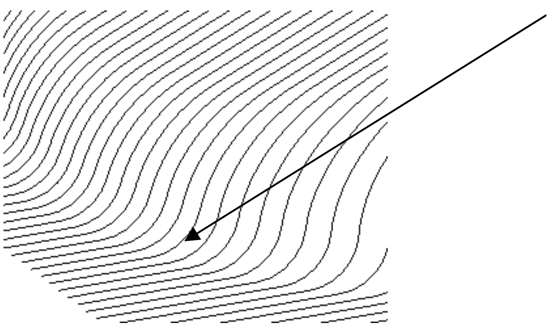
- Select the **Copy Toolpath** icon from the form. 
- Rename the new toolpath as **Raster_arcfit**.
- From the local **Raster Finishing**, ***explorer***, select the **High speed** options page.



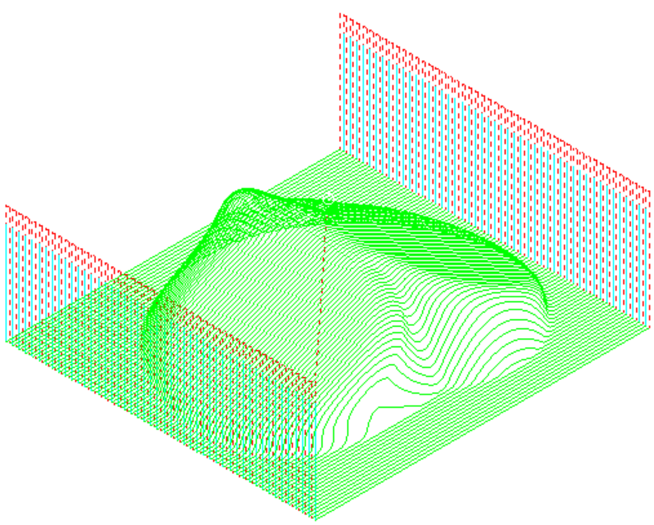
- Tick the box **Arc Fit** and edit the **Arc Radius** to the maximum value **0.2**

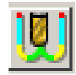
The value **0.2** (**TDU**), multiplied by the tool diameter (**12mm**) produces a **radius** of **2.4mm**.

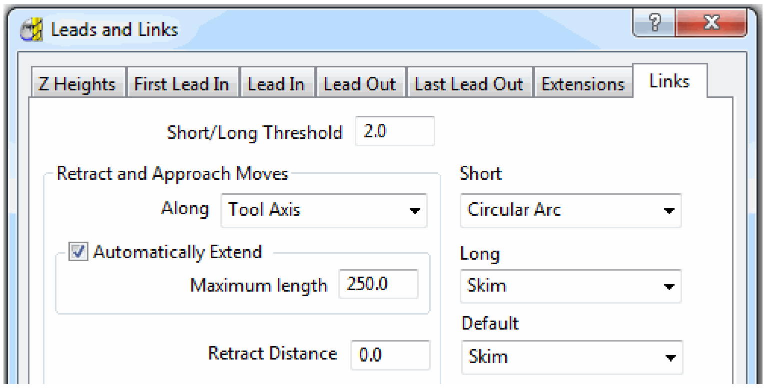
- Press **Calculate** to process the **toolpath** and then **Cancel**.
- **Zoom** into the same area to view the **arc fitted** toopath..



At the end of each **tool track** the cutter ***retracts*** to the **Safe Z height**. This is a waste of time as these lifts are unnecessary. To make the toolpath a more efficient **Link** option can be applied.



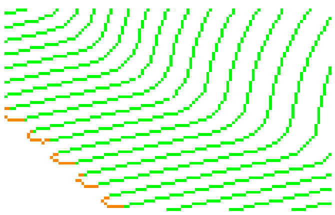
- Select the **Leads and Links** icon  from the **main** toolbar.
- Select the **Links** tab on the form.



- Set the **Short/Long Threshold** to **2.0**

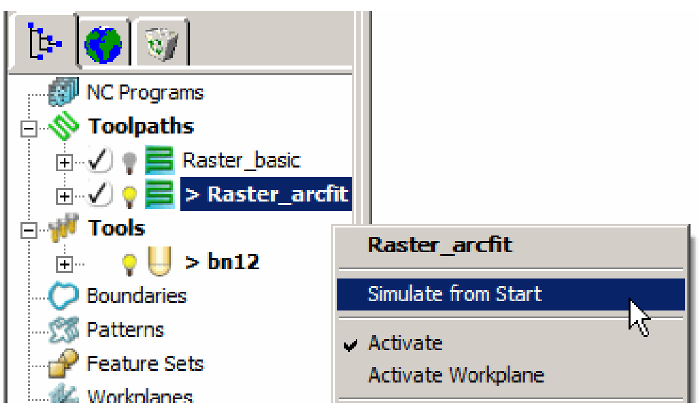
If the distance from the end of a **tool track** is within **2mm** from the start of the next, then the **Short** links option (**Circular Arc**) will apply.

- Set the **Short** links to **Circular Arc**.
- Set the **Long** and **Default** links to **Skim**.
- **Apply** the ***Leads and Links*** and then **Accept** the form.



All unnecessary **retract** moves have been eliminated with a **circular arc** move now forming the **link** between adjacent tool tracks.

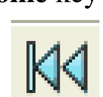
The next stage is to **Simulate** the latest, **Active** toolpath, **Raster_arc fit**.



To **Simulate** the toolpath, **right click** over the toolpath icon in the **explorer** area, and select **Simulate from Start**.

When the **Simulate from Start** is selected the toolbar will appear at the top of the screen, and the toolpath can be **Simulated** using the ***Play*** button.

When the **simulation** 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 **Simulation** toolbar.

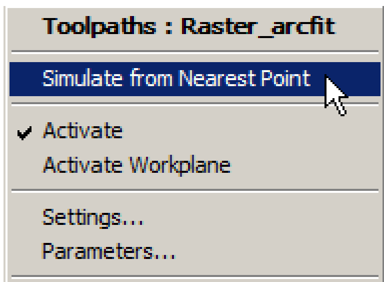


To **simulate** the cutting moves locally and in more detail, it is possible to attach the tool to any position along the toolpath and use the **Left/Right Arrow keys** on the keyboard to make the tool move backwards or forwards along the toolpath.

- Position the cursor at the required start point along the toolpath and right click to open the **Toolpath** pull down menu.

Note; When the menu is accessed directly from the **toolpath** in the graphics area, the first option is **Simulate from Nearest Point** instead of **Simulate from Start**.

- Select **Simulate from Nearest Point.**



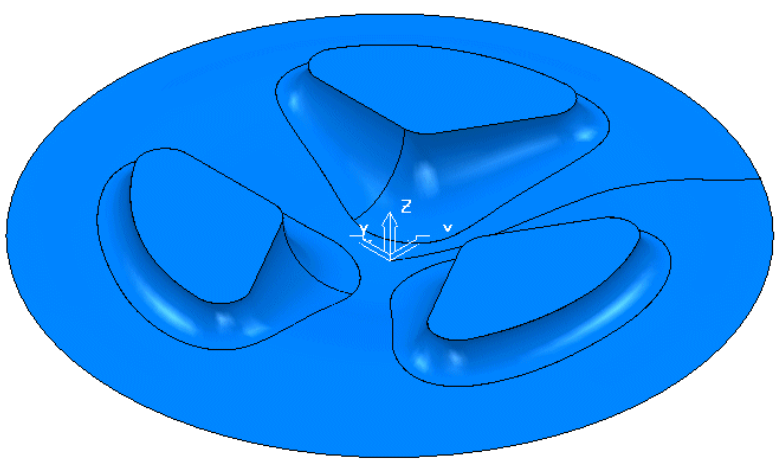
- Use the **Left** and **Right** arrow keys (***bottom right corner of keyboard***) to ***increment*** or ***move*** the tool forwards and backwards along the toolpath.

Pattern Finishing

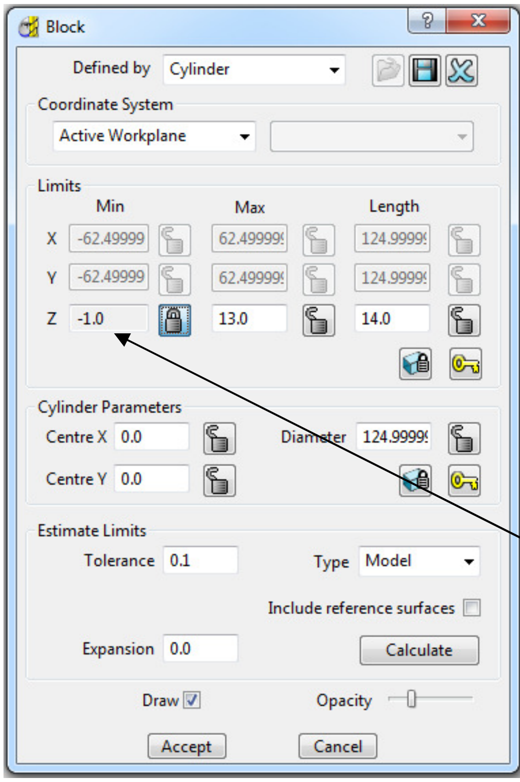
This technique requires a user defined **Pattern** for projection onto the model as a toolpath. This option can be used for applications such as scribe lines, lettering, and non-standard tool strategies. A **Pattern** is created either from within **PowerMILL** or as imported **Wireframe** data.

- **Delete All** and **Reset forms**.
- **Import** the **Model**:

  C:\users\training\PowerMILL_Data\Models\swheel.dgk



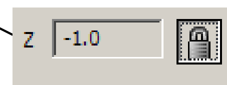
The component is circular as a result of which the **Block - Z Minimum** and **Z Maximum** will be developed as a **Circular** billet.



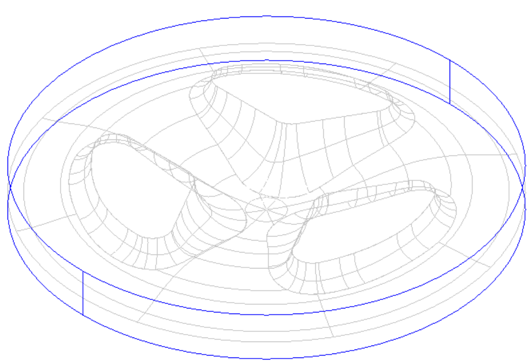
- Select the **Block** icon from the **main** toolbar.
- Select ***Defined by* - Cylinder**.

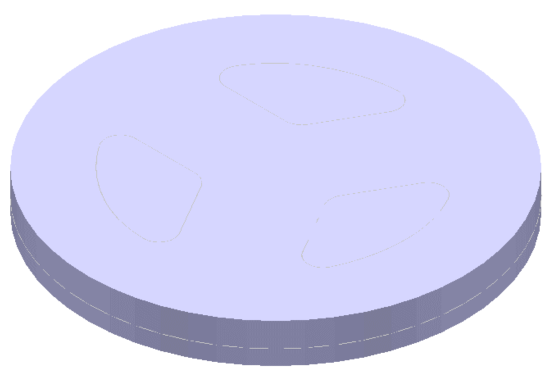
The default ***Radius*** value represents the best cylindrical fit around the component achieved when the **Calculate** button is clicked

- 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 variable **Opacity** up to solid, depending on the position of the slider switch.

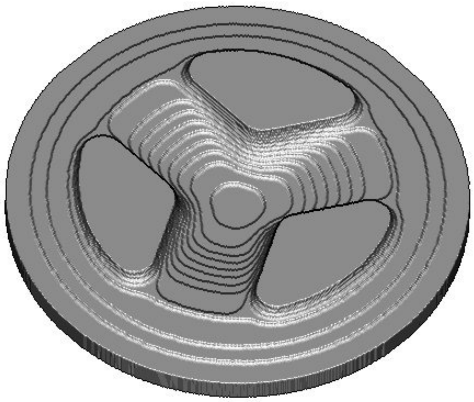




Before the **Pattern** finishing strategy is created, a preliminary **Area Clear** strategy is required to remove the bulk of the material, leaving **0.5mm**, followed by a **Semi Finishing** strategy leaving **0.2mm** prior to the final **Finishing**.

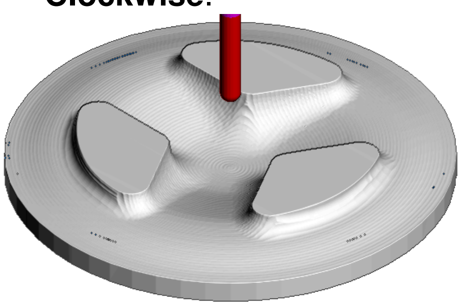
- Define a tool of ***Diameter* 10** and ***Tip Radius* 1**, ***Named* D10T1** with ***Tool Number* 1**.
- Define a ***Diameter* 6** Ball Nose tool ***Named* BN6** with ***Tool Number* 2**.
- **Activate** the **D10T1 Tip Radius tool**.
- In the **Rapid Move Heights**  select **Calculate**.

- **Calculate** a **Model Area Clearance** toolpath using the following parameters:- ***Style*** - **Offset all**, ***Name*** - **RoughOp1**, **Tolerance 0.1**, **Thickness 0.5**, **Stepover 5.0**, **Stepdown 1.0**, and **Cut Direction Climb**.

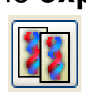


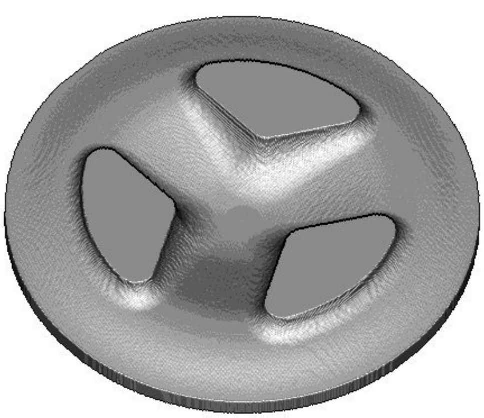
- **Simulate** the toolpath in **Viewmill** to give a result similar to as shown left.
- Return the **Viewmill** display to **No Image**. 

- **Activate** the **BN6** Ball Nose tool.
- **Calculate** a **Spiral Finishing** toolpath using the following parameters: ***Name*** - **SemiFinishOp1**, ***Centre Point*** **X 0 Y 0**, ***Radius*** **- Start 62.0**, **End 0.0**, ***Tolerance*** **0.1**, ***Thickness*** **0.2**, ***Stepover*** **1.0**, and ***Direction*** - **Clockwise**.



- **Simulate** this new toolpath in **ViewMILL**.
- Return the **Viewmill** display to **No Image**. 

- **Right click** over the **SemiFinishOP1** toolpath in the **explorer** and select **Settings**.
- In the **Spiral Finishing** form select the **Copy** icon. 
- Enter a new ***Name*** - **FinishOP1**, Use ***Tolerance*** **0.01**, **Thickness 0.0**, **Stepover 0.3** and then **Calculate** and **Close** the form.
- **Simulate** the final toolpath in **Viewmill** to display the effects of the final toolpath.

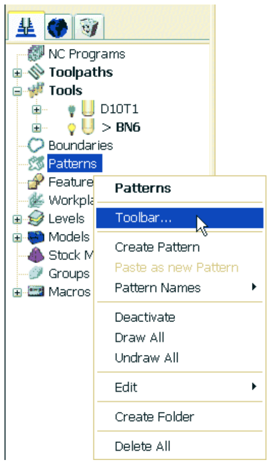


A **Pattern** finishing strategy will be now applied to machine the text, **Engraved** into the component.

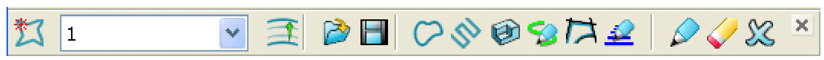
- Return the **Viewmill** display to **No Image**. 

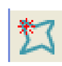
Next a pre-defined **Pattern** will be **imported** for use with the strategy.

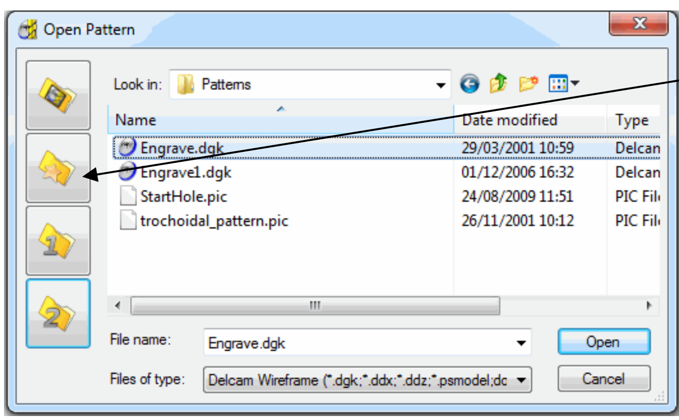
- Right click the **Pattern icon** in the **explorer** and select **Toolbar….**



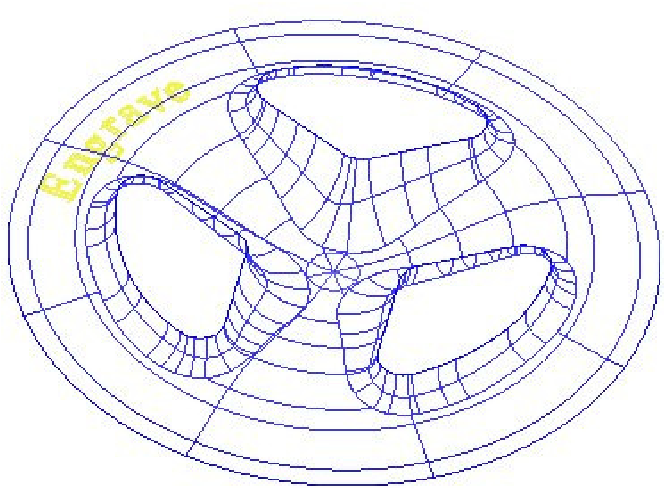
When **Toolbar** is selected off the **Pattern** menu it loads a new toolbar into **PowerMILL**. The toolbar contains icons to create an empty **Pattern** and **insert** different types of wireframe entities. The toolbar can be removed from the screen by clicking the small **x** on the right.



- Select the **Create pattern** icon .
- Click on the **Insert - File** into the **Active Pattern** icon .

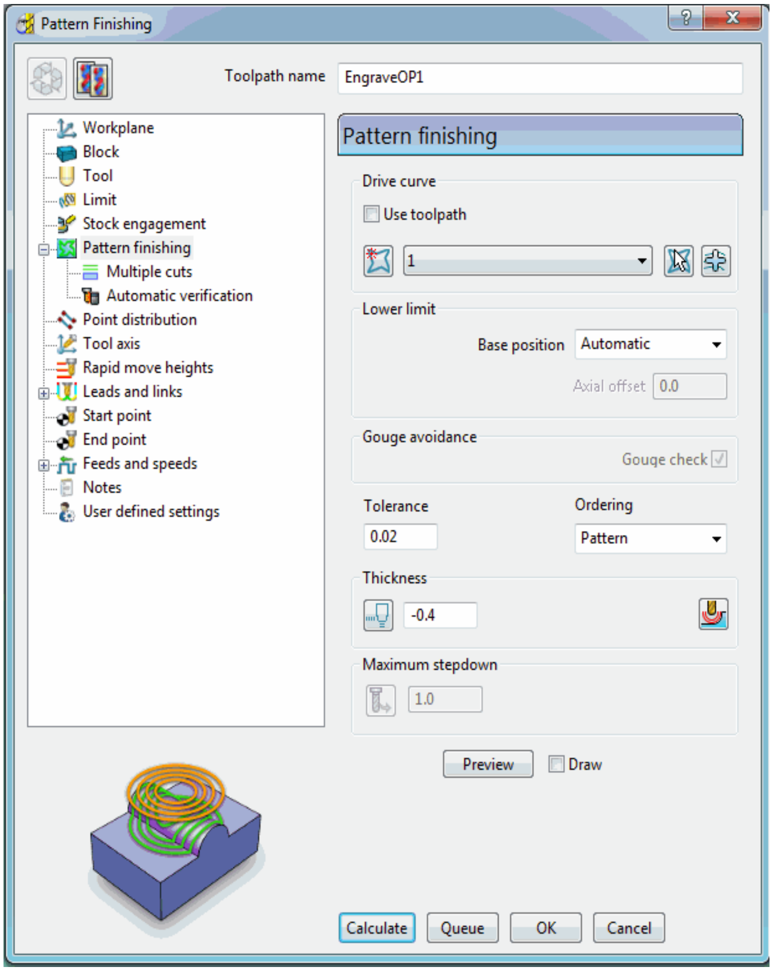


- Select the examples shortcut button.
- Double click the **Patterns** folder and then select **Engrave.dgk**.
- Select **Open**.



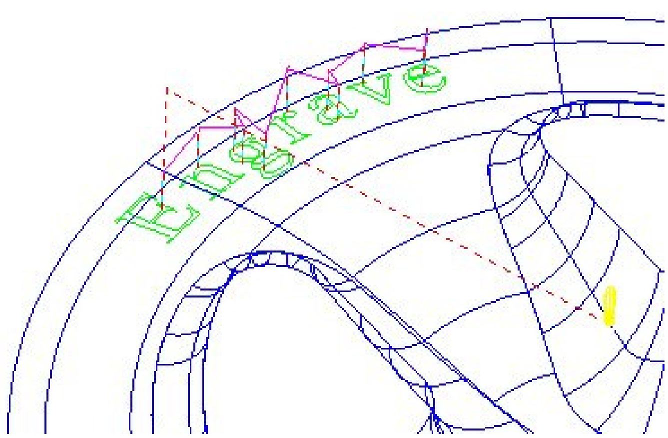
The **Pattern** is made up of **2D geometry** and is positioned at the bottom of the **Block**. This will be projected along **Z** through the **3D model** form to create a single **toolpath** with a negative **Thickness** value.

- Define a **Ball Nose** tool with **Diameter 1** called **BN1** and make this **Tool Number 3**.
- Create a **Pattern Finishing** toolpath.



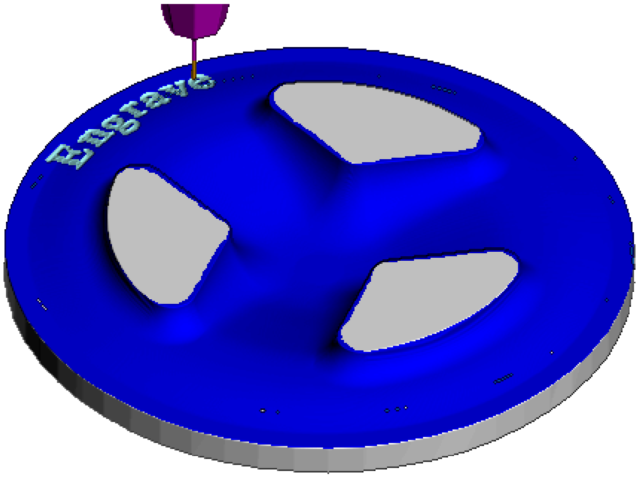
- Enter the ***Name*** - **EngraveOp1**.
- Select **Pattern 1** (if not already selected).
- With ***Base position*** set to either **Automatic** or **Vertical**, the **Pattern** will project down **Z** onto the model.
- Enter a ***Tolerance*** **0.02** and ***Thickness*** **-0.4**

- Select **Calculate** the toolpath then **Cancel** the form.



By entering a negative **Thickness** the tool will machine into the previously finished surface. This is frequently used for engraving. Note: It is not possible to use a ***–ve thickness*** **value** that is greater than the **Tool Tip Radius**. There is however a way to achieve this if the user has a **PowerMILL PRO** license (Chapter 11 **Patterns**, page 20 - **Deep Pattern Machining**).

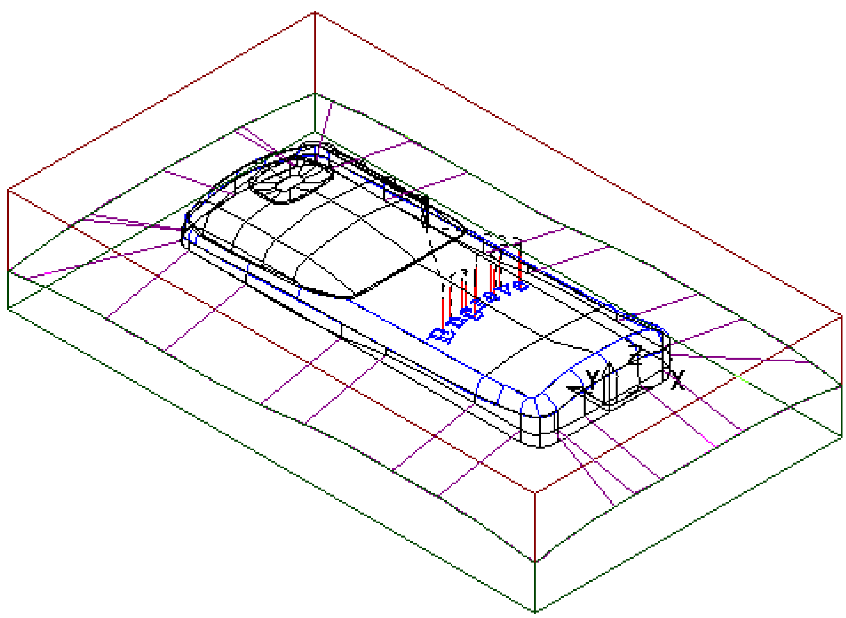
- **Simulate** the path in **Viewmill**.



- **Save Project as**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\PatternMachining

Pattern Exercise

The same imported **Pattern** will be used to engrave text into a telephone handset.
The **Pattern**, '***Engrave***' used earlier will be out of position, but can be moved using the options on the **Pattern** - right click menu.



- 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.

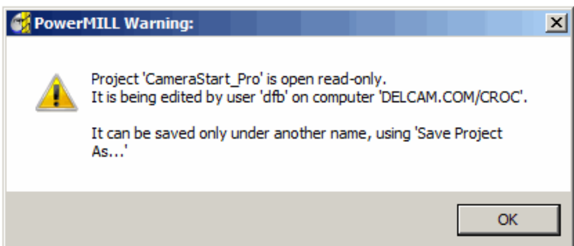
3D Offset and Constant Z Finishing

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. A **Boundary** will be created and used to limit the **3D Offset** toolpath to the flatter areas leaving the remaining steep areas to be machined using the **Constant Z** strategy.

3D Offset Finishing

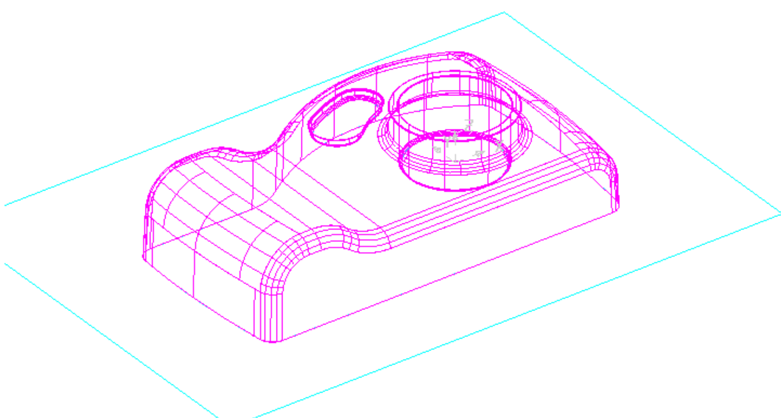
3D Offset Machining defines the tool **Stepover** relative to the 3D model surface providing consistency over both flat areas and steep sidewalls. In this exercise applying this toolpath to the complete model without using **Boundaries** would not be recommended. Although this toolpath maintains a constant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

- **Delete All** and **Reset forms**.
- From **File** - **Open Project** and in the form browse to:-
  C:\users\training\PowerMILL_Data\Projects\Camera_Start



The **Project** is **Locked** and cannot be modified unless saved as a new file (or the original **Project's *Lock*** file is deleted).

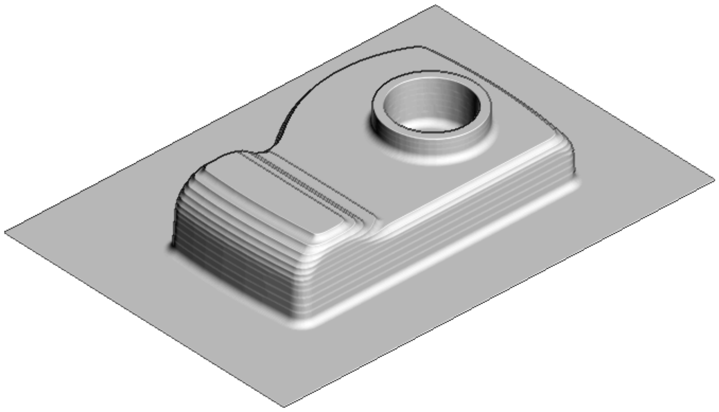
- Select **OK** to load the existing **Project** into **PowerMILL**.
- From **File** – **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL_Projects\Camera_Example



The **Project** already contains an **Offset Area Clearance** strategy along with any associated Settings.

This model is a good example where it is a better option to use more than one finishing strategy. **3D Offset** (for shallow areas) and **Constant Z** (for Steep areas) will be used and they will be they will be kept separate using a **Boundary**.

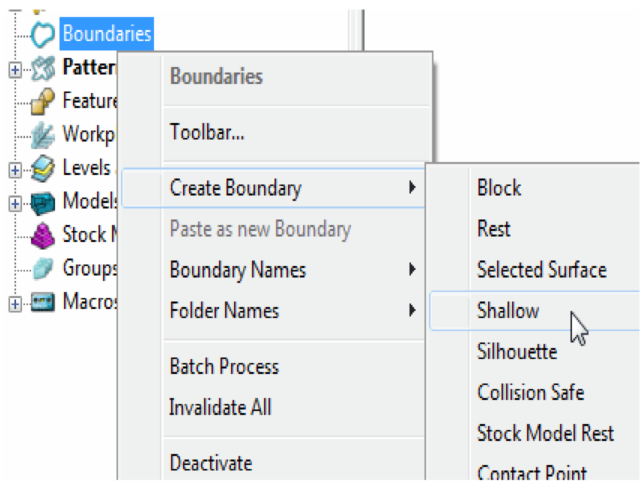
- **Activate** the existing **Model *Area Clearance*, Toolpath** and **Simulate** the toolpath in **Viewmill**.



The **Viewmill** simulation is as shown left.

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

Use of Shallow Boundary to define specific Machining Areas



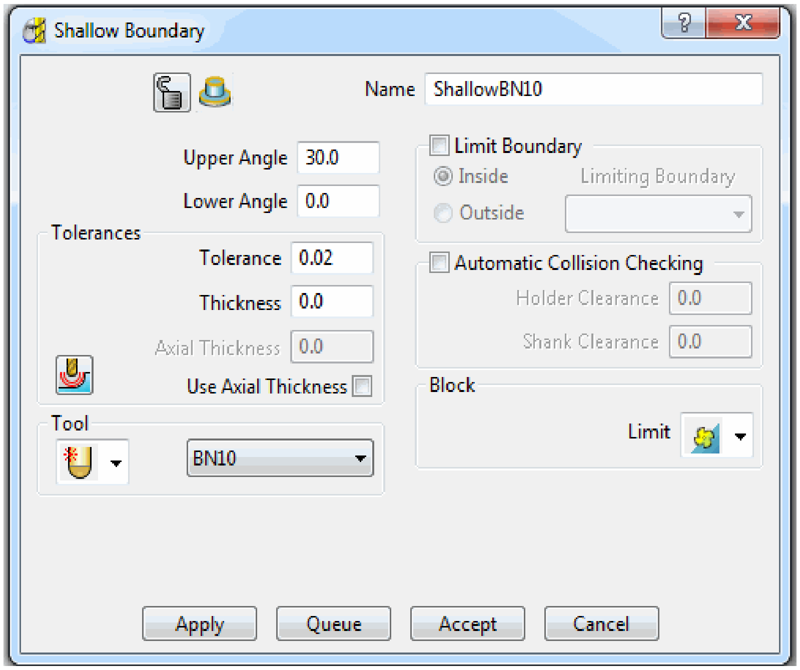
Specific ***Finishing strategies*** are more effective on either **Steep** or **Shallow** parts of the ***3D component model***. For example, ***Constant Z finishing*** is most effective on ***steep sidewalls*** while ***Raster Finishing*** is most effective on ***shallow areas***.
A **Shallow Boundary** will be created to discriminate the areas most suited to the individual machining strategies used. This type of **Boundary** is calculated taking into account the **Active** tool.

There are several other types of ***Boundary options*** available to suit various applications and these are covered later in ***Chapter 6***.

- With the **BN10** tool **Active**, right click on the **Boundaries icon** in the **explorer** and select **Create Boundary -> Shallow**.

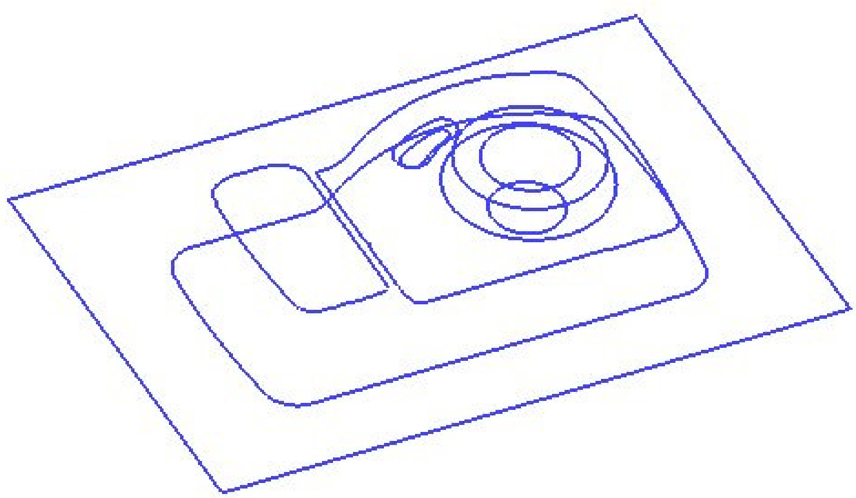
A **Shallow Boundary** defines segments from areas on the model that are defined by an **Upper** and **Lower** threshold angle. It is therefore specifically suited to steep walled and shallow surface machining techniques.

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



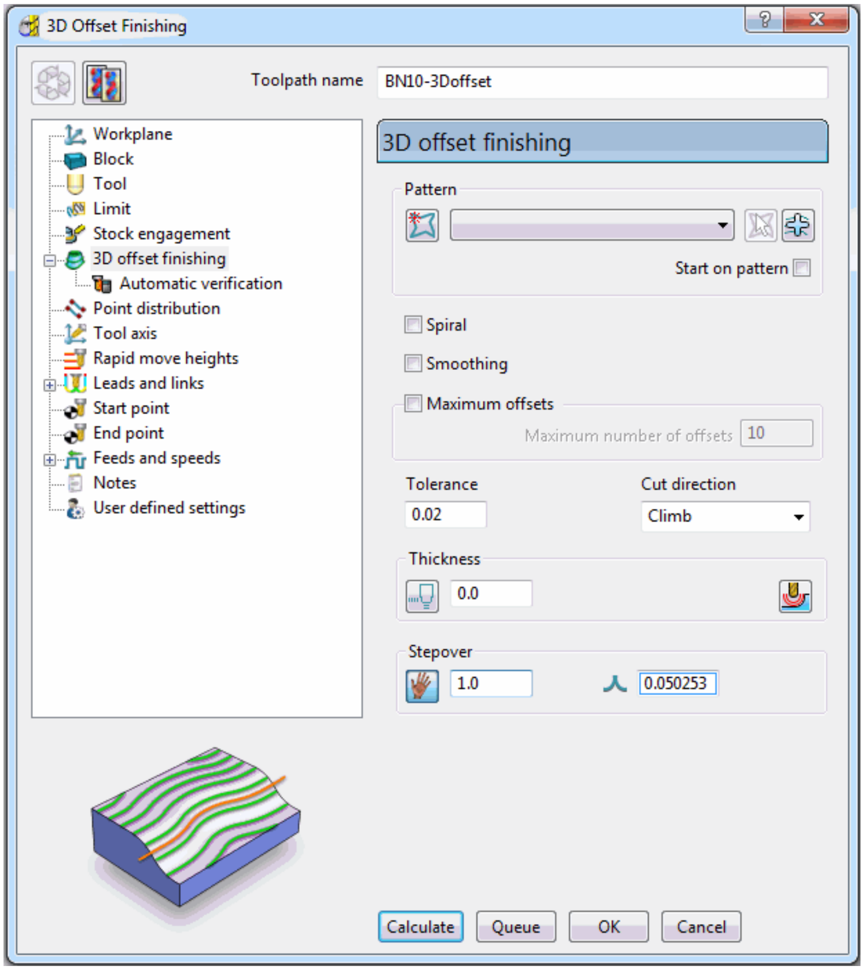
- Input the **Name** - **ShallowBN10**.
- Input ***Upper Angle* 30** and ***Lower Angle*.0**
- Input**Tolerance 0.02**.
- Input ***Thickness* 0**
- Ensure the correct tool **BN10** is **Active**.

- **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 except when **Locked** due being assigned to a toolpath.

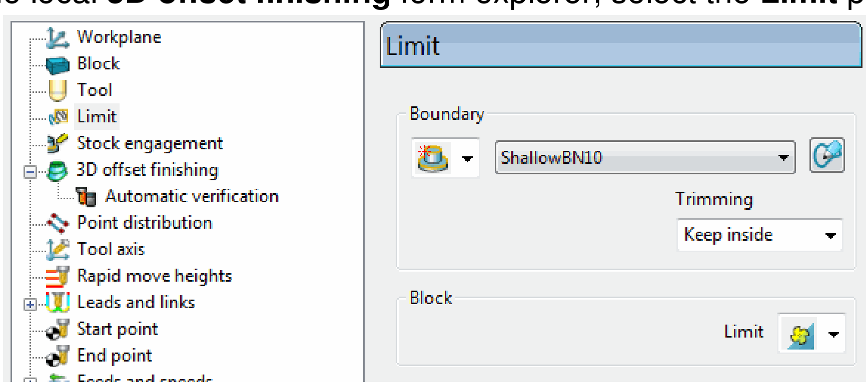
- Select the **Toolpath Strategies** icon from the top of the screen. 
- Select a **3D Offset Finishing** strategy from the form then **OK**.



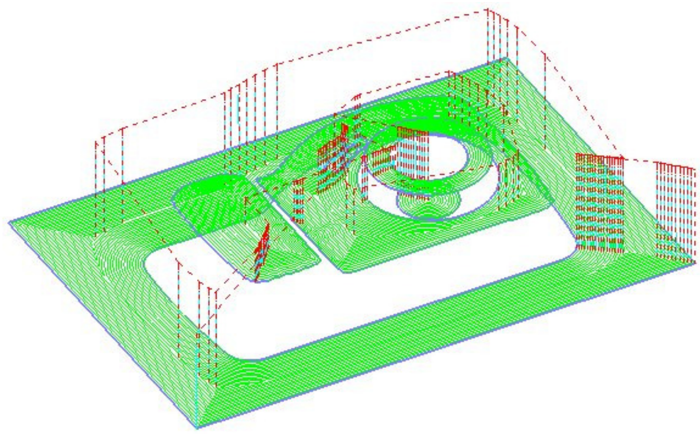
- Enter the ***Name* Bn10-3DOffset**
- Select ***Direction* to Climb**.
- Enter a ***Tolerance* 0.02**.
- Enter a ***Thickness* 0**.
- Enter ***Stepover* 1.0**

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.

- In the local **3D offset finishing** form explorer, select the **Limit** page.

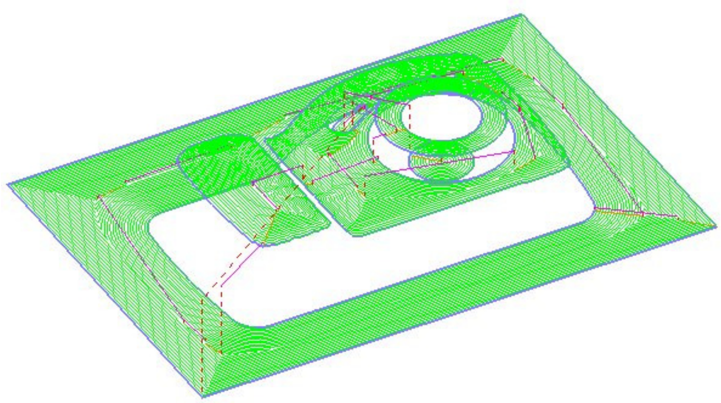


- Input the **Boundary *ShallowBN10*** and **Calculate** to process the **toolpath**.
- **Cancel** the form.



The toolpath follws the contours of the **Boundary** segments and is produced only on the shallow areas of the model (**Inside** the **Boundary**).

- 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 **Default Links** to **Skim.**
- **Apply** and **Accept** the form.

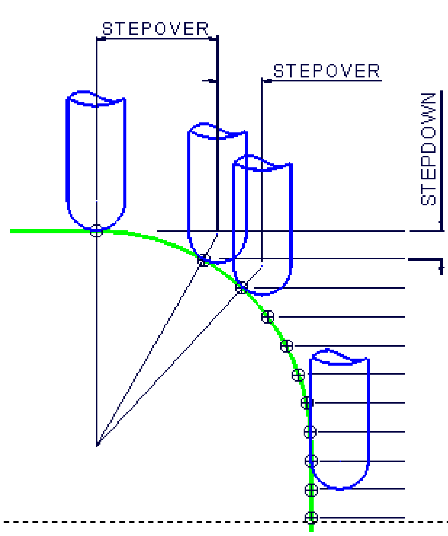


The toolpath has now is now more efficient with the rapid moves at **skim** height and the **short links** being forced onto the surface.

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

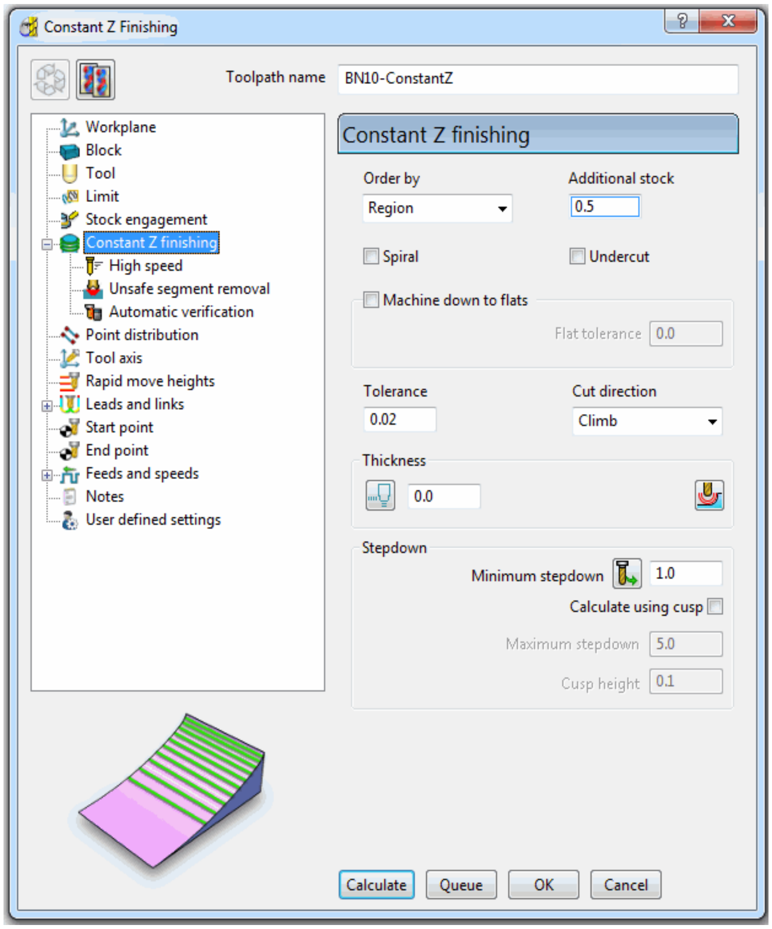
Constant Z machining

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.

- Select the **Toolpath Strategies** icon from the **main** toolbar.
- Select a **Constant Z Finishing** strategy from the form then **OK**.

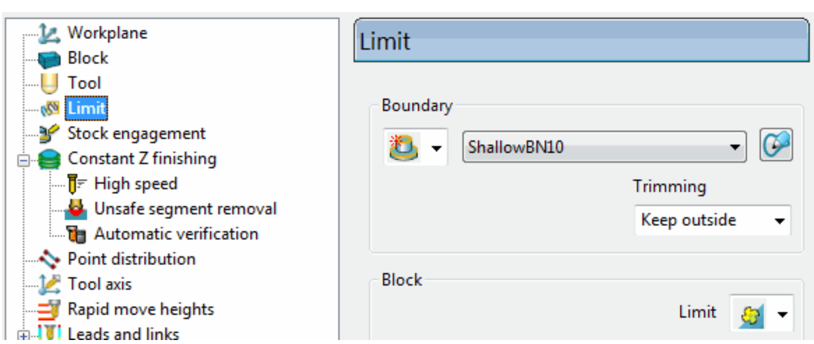


- Enter a ***Name*** as **ConstantZ_BN10.**
- Input ***Tolerance*** **0.02**

.

- Set the ***Direction*** to **Climb**.
- Input ***Stepdown*** **1**

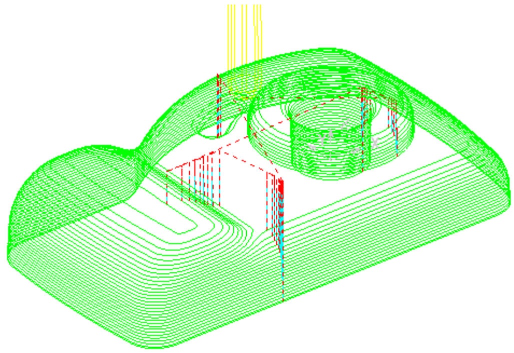
- In the local **Constant Z finishing** form **explorer**, select the **Limit** page.



- Set ***Trimming*** to **Keep outside**.

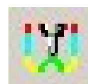
- Select **Calculate** to process the **toolpath** and then **Cancel** the form.

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

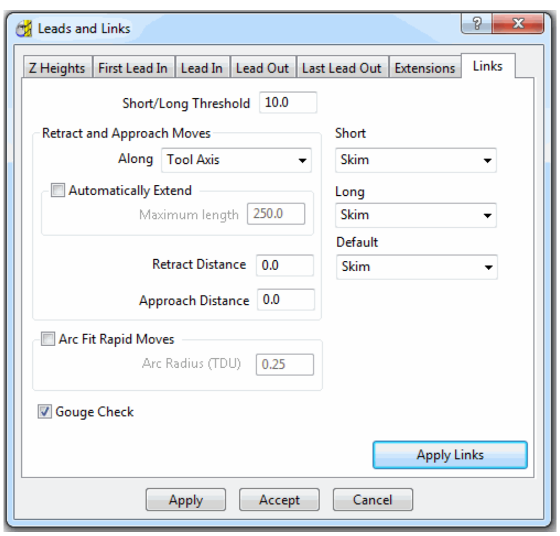


If the **Boundary** had not been used the toolpath would have looked like this.
It can be seen that the parts of the toolpath on the shallow areas have an excessive **Stepover**.

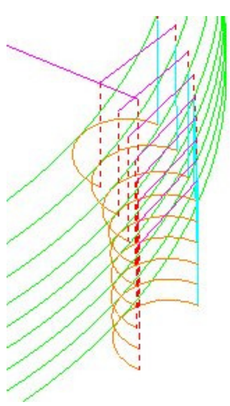
- Select the **Leads and Links** icon at the top of the screen.



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

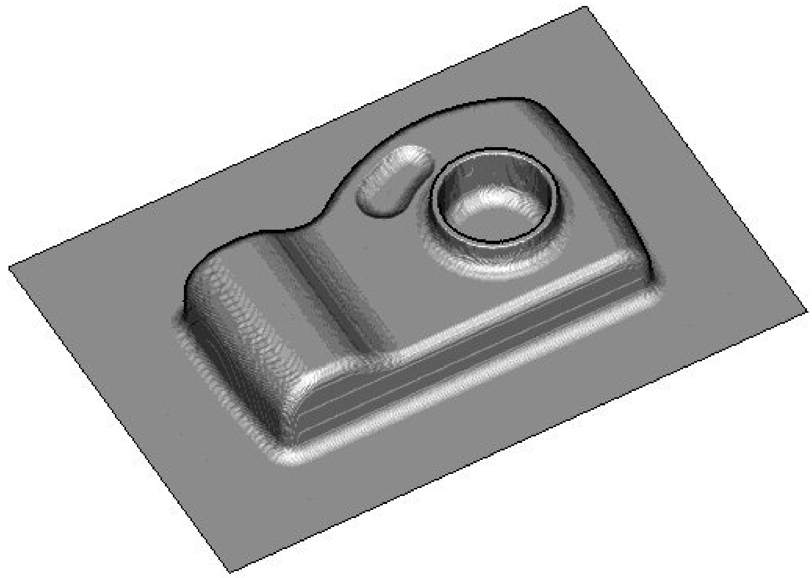


- Select the **Links** tab and change the **Short**, **Long**, and **Default** to **Skim**.
- **Apply** and **Accept** the form.



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**.



- **Save** the **Project** (***C:\users\training\COURSEWORK\PowerMILL-Projects\camera***) but do not **Close**.

Other Constant Z Options

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

The **Order By - Region** (default) option will make **Constant Z** machine local features, such as ***pockets*** completely from top to bottom. The **Order By - Level** option will make **Constant Z** fully machine the model at each **stepdown** height.

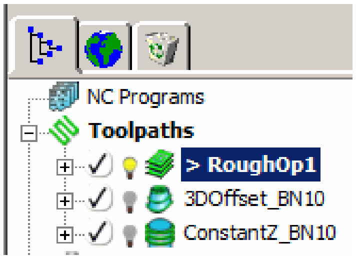
The **Undercut** option can be selected (if a ***spherical*** or ***disc cutter*** is used) to enable the machining of otherwise inaccessible areas.

The **Spiral** option will make the **Constant Z** strategy produce continuous spiral tool tracks wherever possible (closed contours).

The **Machine down to flats** option will ensure that the final pass at the base of a wall is exactly level with a flat area of the model (if applicable).

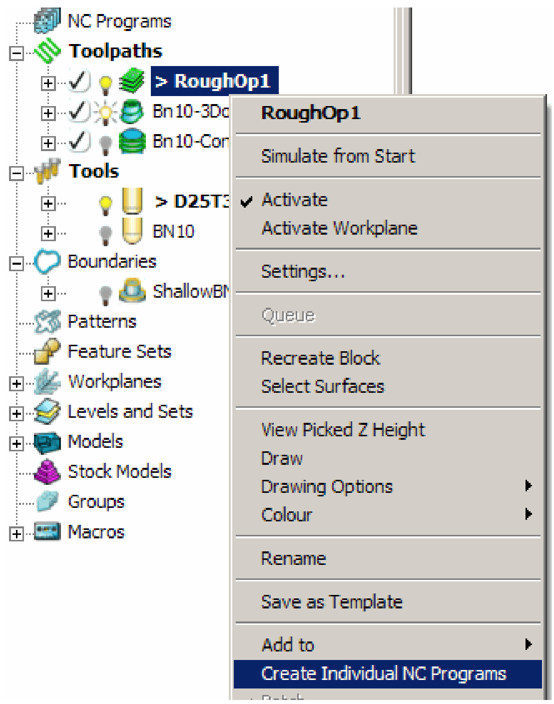
Post Processing and NC Programs.

At this stage we will post process a single toolpath from the explorer as an NC Program.

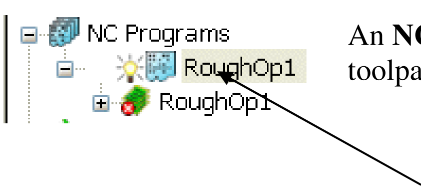


All of the toolpaths that have been created in this chapter should appear in the explorer like this.
This example will illustrate with the output of one single toolpath – **Rough Op1** to be output as a post-processed ncdata file.

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

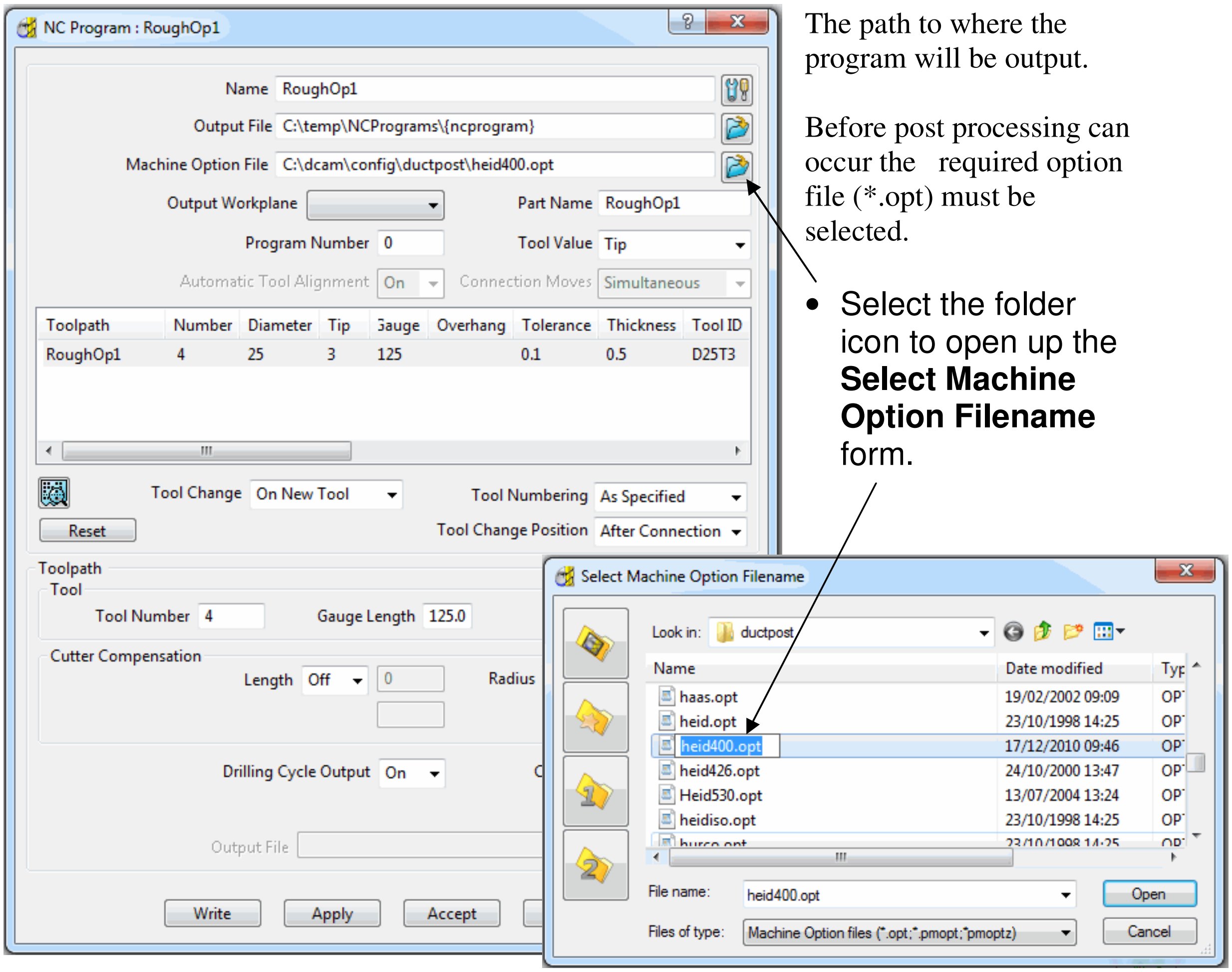


- Select the **Create Individual NC Programs** option.



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

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



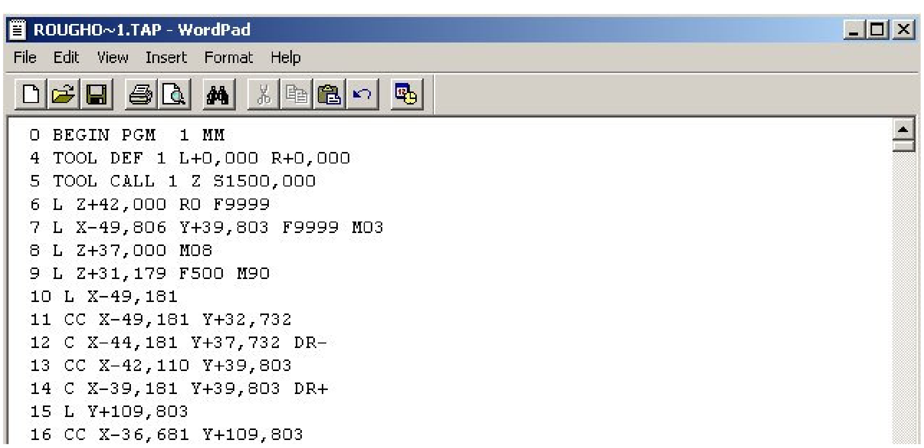
The path to where the program will be output.

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 output, NC Program can be viewed by double clicking on it in the Windows **C:\ Temp\NCPrograms** folder and view it in WordPad.



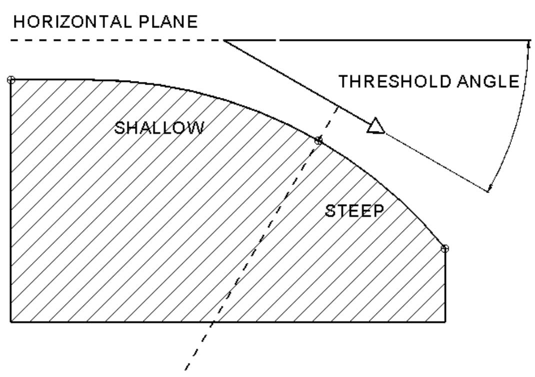

```
0 BEGIN PGM  1 MM
4 TOOL DEF 1 L+0,000 R+0,000
5 TOOL CALL 1 Z S1500,000
6 L Z+42,000 R0 F9999
7 L X-49,806 Y+39,803 F9999 M03
8 L Z+37,000 M08
9 L Z+31,179 F500 M90
10 L X-49,181
11 CC X-49,181 Y+32,732
12 C X-44,181 Y+37,732 DR-
13 CC X-42,110 Y+39,803
14 C X-39,181 Y+39,803 DR+
15 L Y+109,803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

Corner Finishing.

Introduction.

Corner Finishing strategies are used to remove material in corners that are inaccessible to previously used, larger tools. There are **3** different types of strategy: **Pencil, Stitch Corner,** and **Along Corner**. The **Pencil** strategy creates 'single pass' machining along sharp internal corners while the other two perform local machining of the entire area of material inaccessible by a larger tool (Rest Milling).

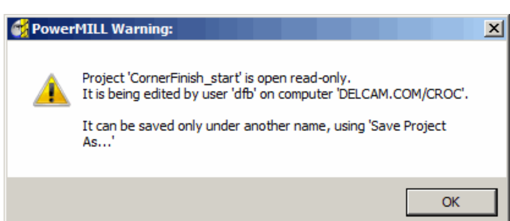
All **Corner Finishing** strategies allow a **Threshold Angle** to be specified. The **Threshold Angle -** determines the angle, from the horizontal, at which steep and shallow portions are split when you select a type of **Steep** or **Shallow**. This provides the user with the ability to prevent the tool running up or down steep slopes using editing techniques. For instance the user could easily apply a **Stitch** strategy to track across the ***steep*** areas while using the **Along** strategy in ***shallow*** areas.



Corner Pencil Finishing

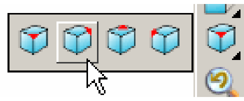
This option produces single pass tool tracks along the intersection between sharp internal corners of the component surface.

- From the **File** pull down menu select **Delete All** and from the **Tools** pull down menu select **Reset** forms.
- From **File - Open Project** and in the form browse to:- **C:\users\training\PowerMILL_Data\Projects\CornerFinish_Start**
- Select **OK** to load the existing **Project** into **PowerMILL**.

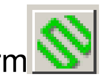


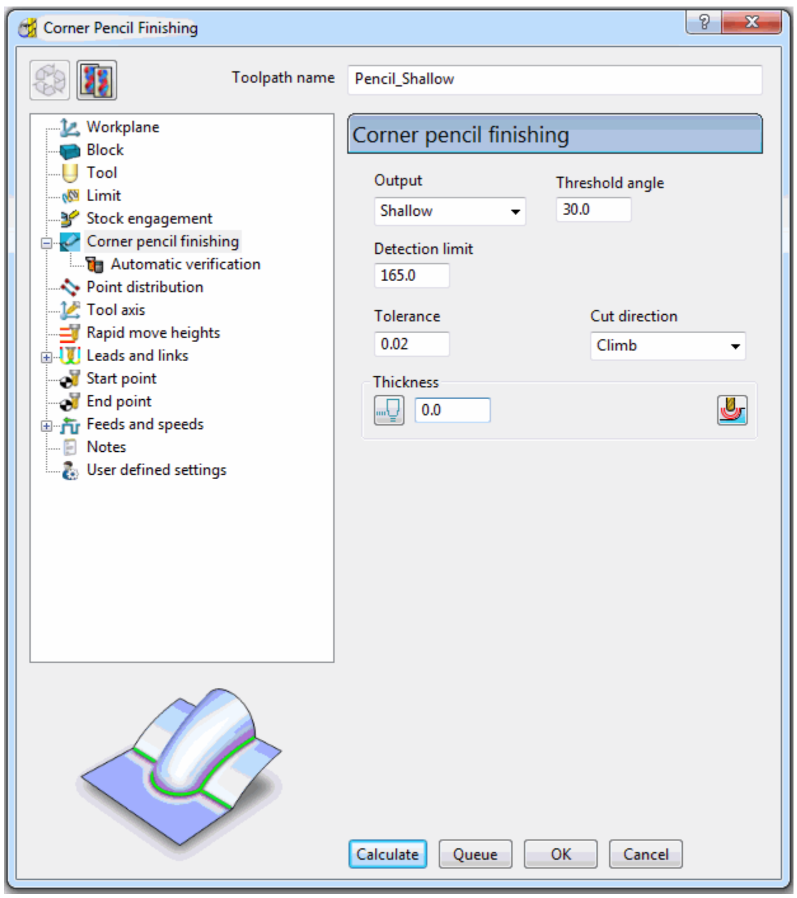
The **Project** is **Locked** and cannot be modified unless saved as a new file (or the original **Project's** ***Lock*** file is deleted).

- From **File** – **Save Project As**:- **C:\users\training\COURSEWORK\PowerMILL-Projects\CornerFinish**

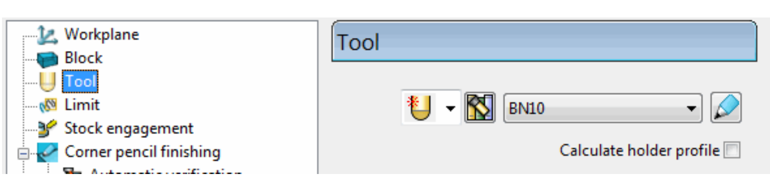
- Select an **Iso2** view. 

- **Activate** the ***toolpath*** **D16T2-rghA1** to restore the settings.
- From the **PowerMILL *explorer***, **Activate** the **Bn10** tool.

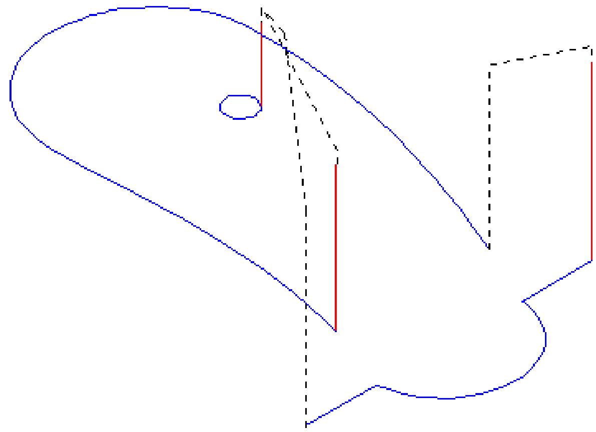
- Open the **Toolpath Strategies** form  and from **Finishing** select **Corner Pencil**.
- Input ***Name*** as **Pencil_Shallow**.
- Select ***Output*** - **Shallow** and ***Tolerance*** **0.02**



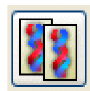
- In the **Corner Pencil Finishing** – ***explorer***, select **Tool** and check that **BN10** is selected as the active cutter.

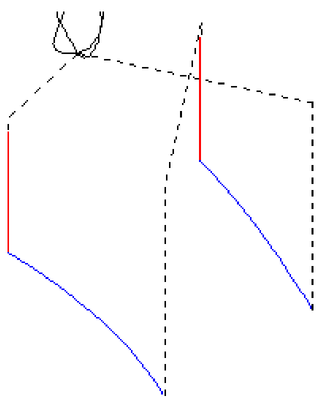


- Select **Calculate** to generate the **Shallow** slope machining only.



The toolpath is produced. The steep area is not machined.

- **Simulate** the toolpath.
- **Right click** over the toolpath in the explorer and select **Settings**.
- Select the **Copy toolpath** icon from the form. 
- Change the **Output** option from **Shallow** to **Steep**.
- **Calculate** the **pencil** machining toolpath (in the **Steep** areas only).



Only the steep area is machined.

- **Simulate** the toolpath. **Rename** the toolpath as **Pencil_Steep**.
- In the main toolbar select **File** - **Save Project**.

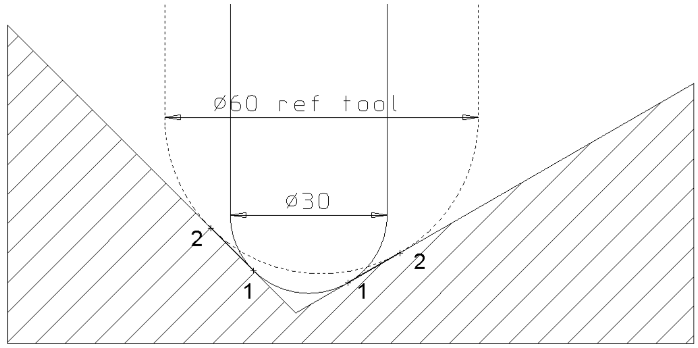
The contents of the **Project** will be updated**:-**

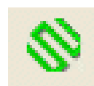
C:\users\training\COURSEWORK\PowerMILL-Projects\CornerFinish

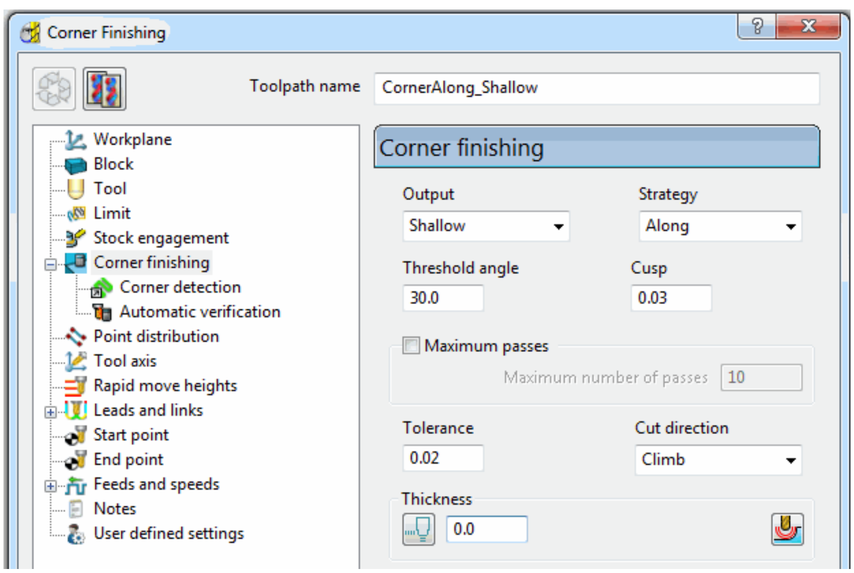
Corner Along Finishing

All the **Corner Finishing** strategies, (except for **Pencil**) are based on machining the area between the Reference tool **(2)** and the Active tool **(1).**

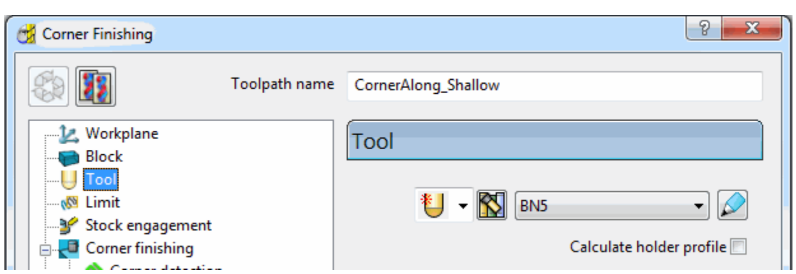
In this example the main finishing tool is a 12 Ball nose (reference) to be **Corner** finished with a diameter 5 Ball-nose.



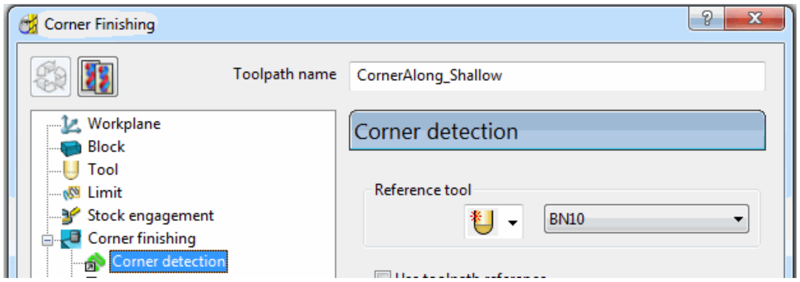
- Define a **dia 5 ball nose** tool named **BN5**.
- Open the **Toolpath strategies** form  and from **Finishing** select **Corner Finishing**.
- Input ***Name** as* **CornerAlong_Shallow**.



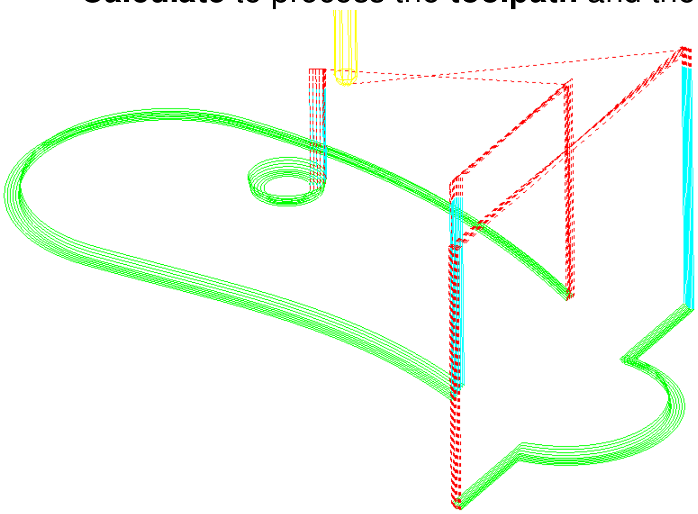
- Change ***Output*** to **Shallow** and ***Strategy*** to **Along**.
- Set **Cusp** to **0.03**.
- Set ***Cut direction*** to **Climb**.



- Check that the **BN5** tool is set (as above) on the **Tool** page.



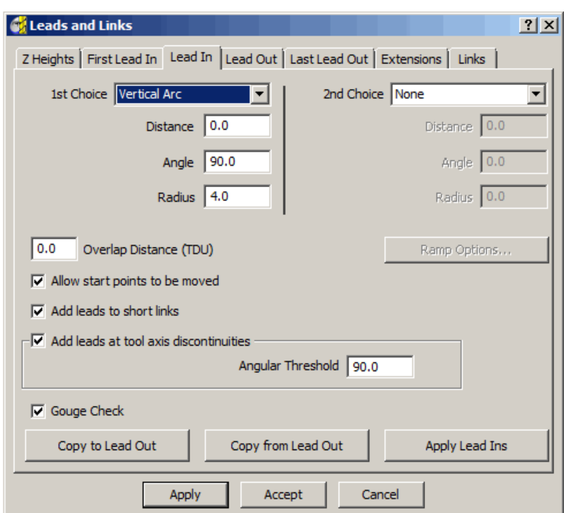
- Select the **Corner detection** page and input the **Reference tool** as **BN10**.
- **Calculate** to process the **toolpath** and then **Cancel** the form.



Simulate the toolpath and observe the order of the machining .

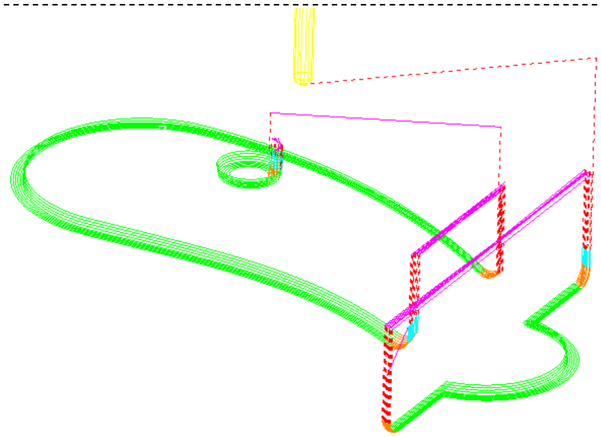
The **Toolpath** will be further improved by applying more suitable **Leads and Links**.

- Open the **Leads and Links** form  and Set all the ***Links*** to **Skim**.



- Select the **Lead In** tab.
- Select **Vertical Arc** and fill in the values shown.
- Click **Copy to Lead Out**.

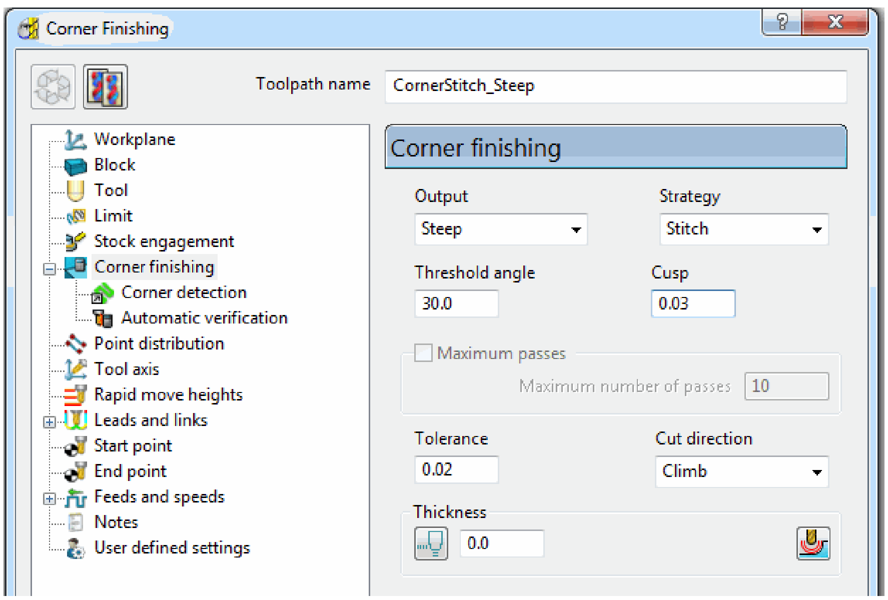
- **Apply** and **Accept**.



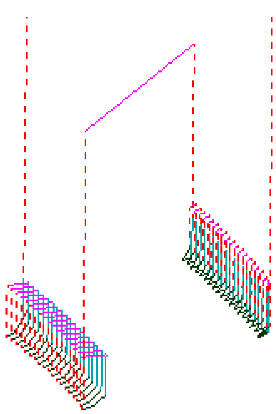
The **Vertical Arc** lead provides a smoother transition onto the work piece with the tool already moving at the specified cutting feed rate.

Corner Stitch Finishing

- From the **Finishing strategies** form, select **Corner Finishing**.
- Input ***Name*** *as* **CornerAlong_Steep**.



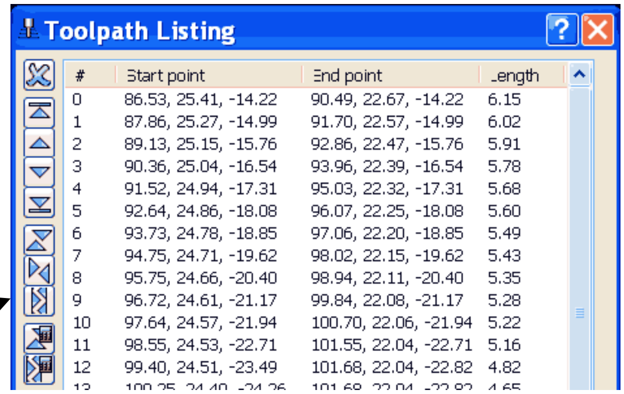
- Change ***Output*** to **Steep** and ***Strategy*** to **Stitch**.
- Check that the **BN5** tool is set on the **Tool** page.
- Check that the **Reference tool** is **BN10** on the **Corner detection** page.
- **Calculate** to process the **toolpath** and then **Cancel** the form.



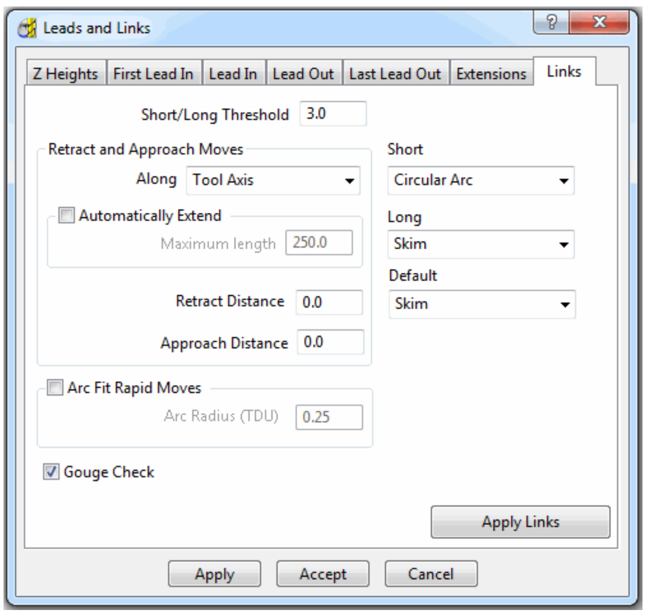
This time only the steep areas have been machined.

If required, it is possible to change the Direction and/or order of individual tool tracks via the **Toolpath - Edit** options.

- **Right click** on the toolpath and select **Edit > Reorder**.

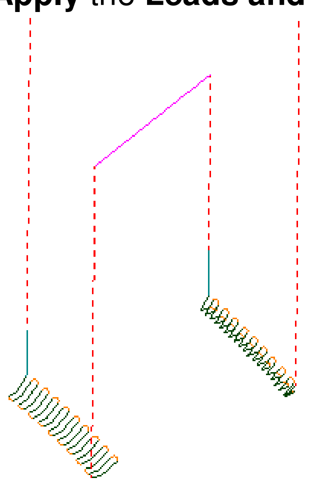


- Select **Alternate directions**.



- In the **Leads and Links** form set ***Short* Links** to **Circular Arc**.

- **Apply** the **Leads and Links** form.



The steep areas now running two ways and this time the **Short links** are defined as a **Circular Arc**.

- In the **main toolbar** select **File - Save Project**.

The contents of the **Project** will be updated:-

C:\users\training\COURSEWORK\Projects\CornerFinish

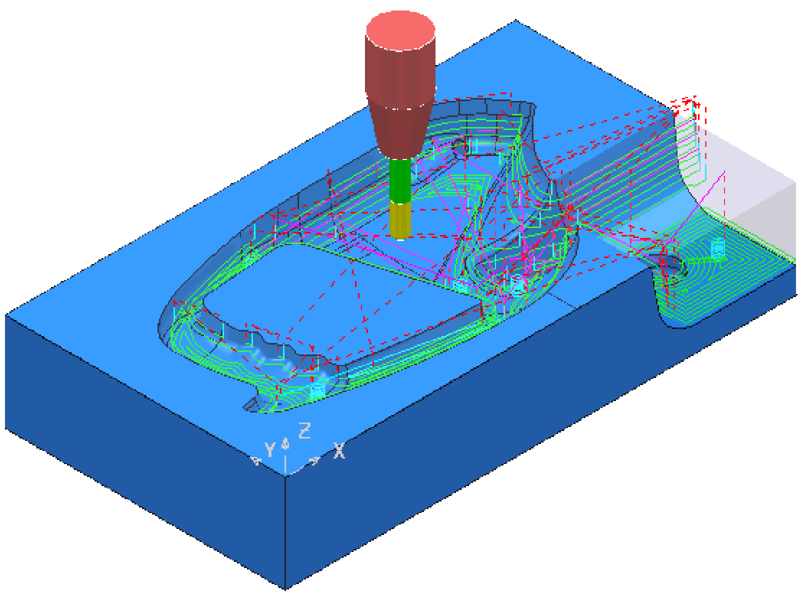
Exercise

Create Finish Machining for an existing *Rough Machined Project*

- **Delete All** and **Reset forms**.
- From **File** – **Open Project** and in the form browse to:-
  C:\users\training\PowerMILL_Data\Projects\ChainsawBodyDie_Start

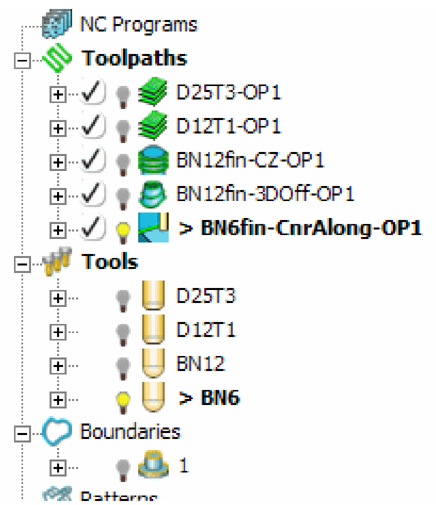
The **Project** is **Locked** and cannot be modified unless saved as a new file (or the original **Project's** ***Lock*** file is deleted).

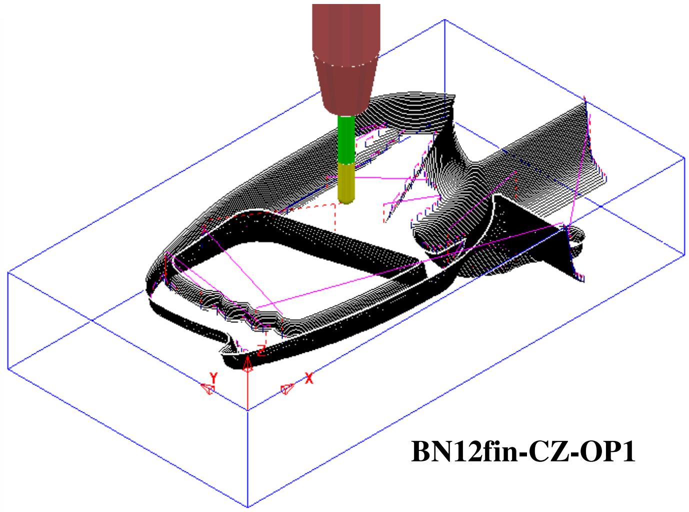
- Select **OK** to load the existing **Project** into **PowerMILL**.
- From **File** - **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL_Projects\ChainSawDie-Ex



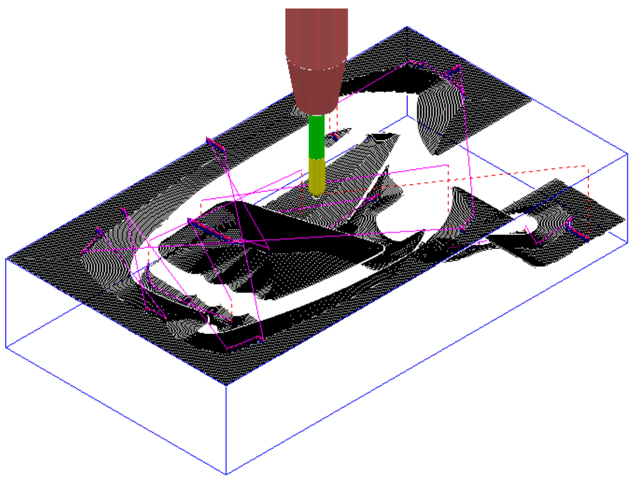
- **Activate** one of the existing **toolpaths** to reinstate the original settings.
- **Activate** the existing **Ball Nose tool - BN12**
- Create a **Shallow Boundary** with **Thickness 0**.
- Create a **Constant Z** finishing strategy named **BN12fin-CZ-OP1** ***limited*** to the steep areas of the model.
- Create a **3D Offset** finishing strategy named **BN12fin-3DOff-OP1** ***limited*** to the shallow areas of the model.
- **Activate** the existing **Ball Nose tool - BN6**
- Create a **Corner Finishing** - **Along** strategy, named **BN6fin-CnrAlong-OP1**, ***referenced*** to the larger **BN12** tool.
- Perform a full **ViewMILL** ***simulation***.
- **Save** the **Project** to update all the stored toolpaths.

Illustrations of typical **Finishing Strategy** solutions and corresponding **ViewMILL** solutions are as shown below.

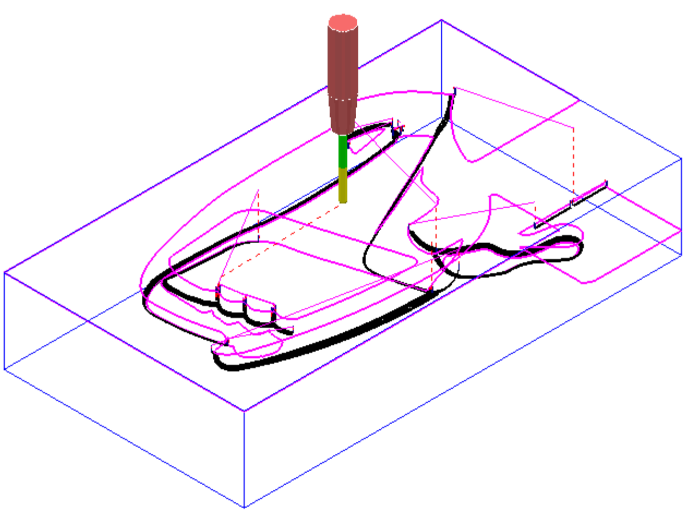




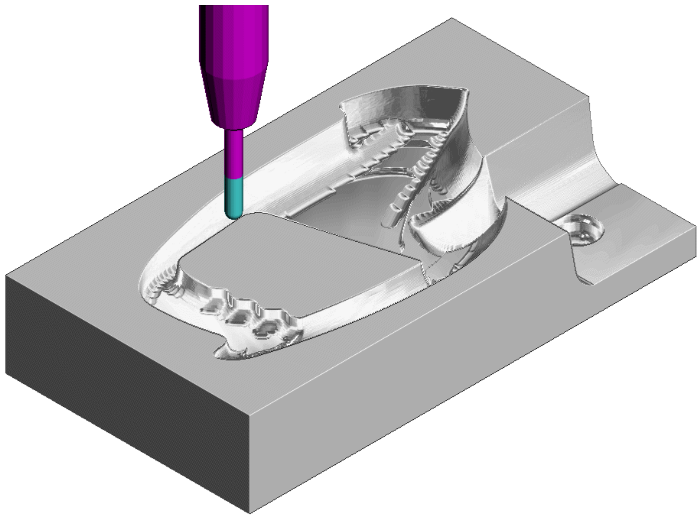
BN12fin-CZ-OP1



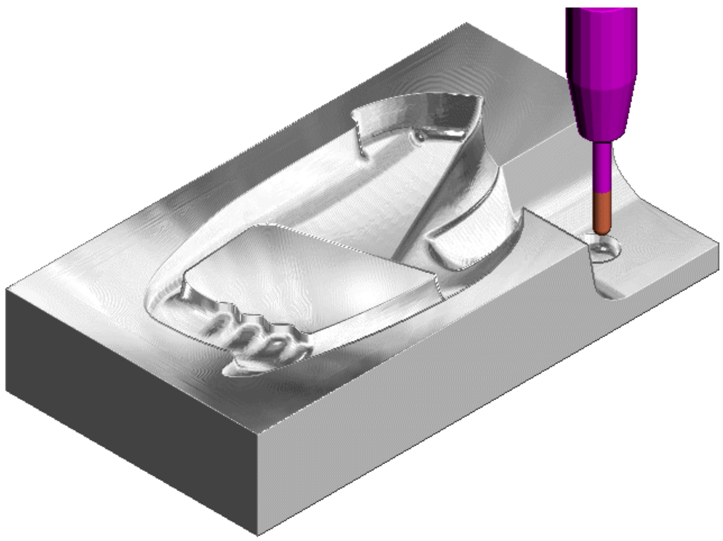
BN12fin-3DOff-OP1



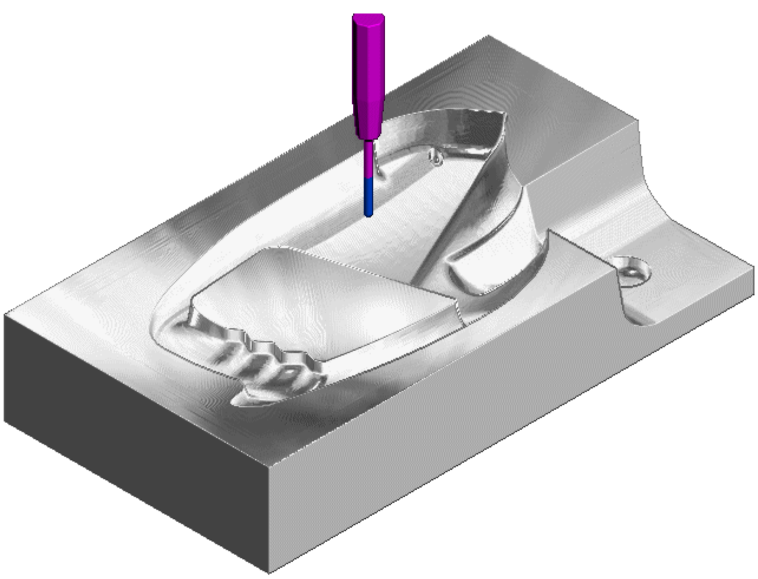
BN6fin-CnrAlong-OP1



BN12fin-CZ-OP1



BN12fin-3DOff-OP1



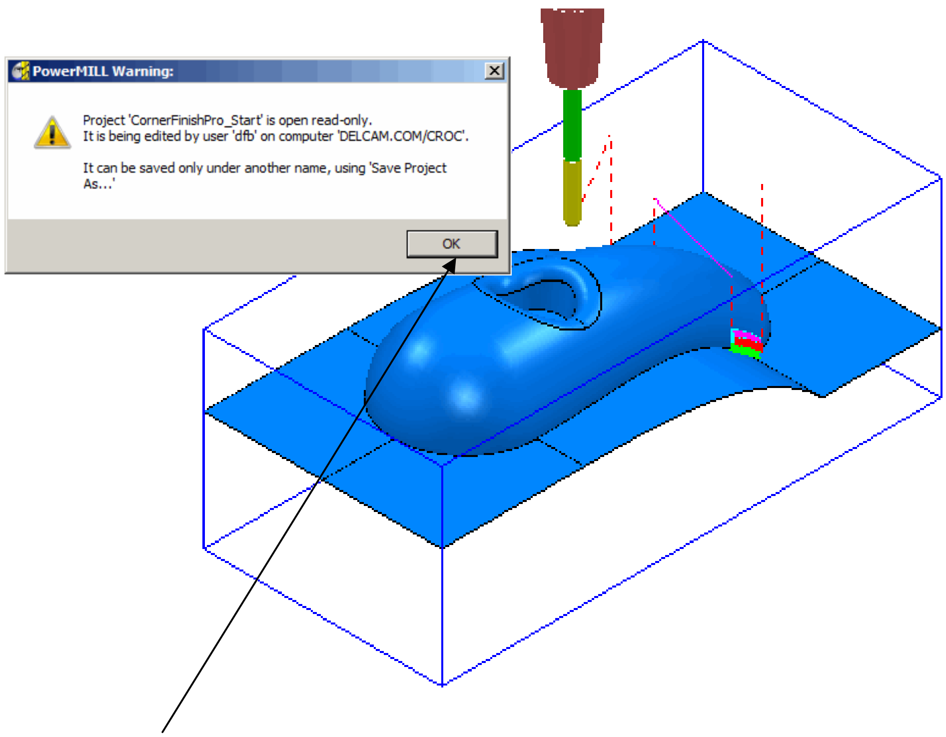
BN6fin-CnrAlong-OP1

PRO - Finishing Strategies

Further Corner Finishing strategies

In **PowerMILL Pro** two further **Corner Finishing** strategies are available. These include:- **Corner MultiPencil** (offsets tooltracks outwards from the **Pencil** intersection) and **Corner Automatic** (a combination of **Stitch** and **Along**). The option **Output - Both** also appears in all five of the **Corner Finishing** strategies. This creates separate groups of toolpath segments on both sides of the **Threshold Angle**.

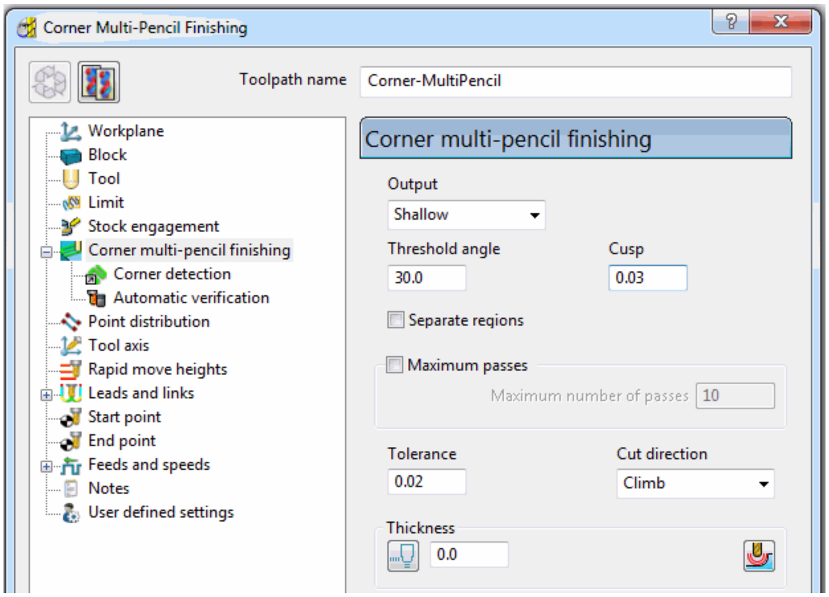
- **Delete All** and **Reset forms**.
- From **File - Open Project** select the ***read-only* Project**:- **C:\users\training\PowerMILL_Data\Projects\CornerFinishPro_Start**



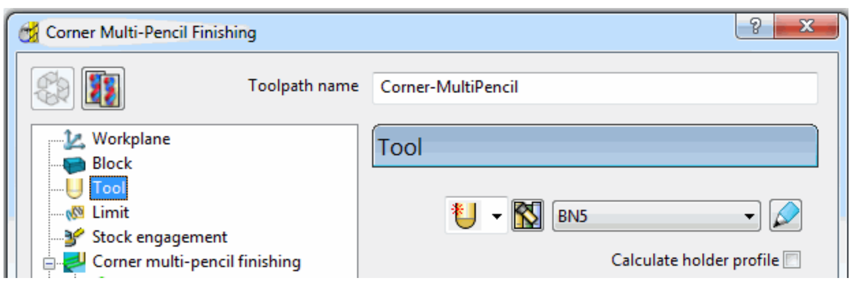
- Select **OK** in the ***PowerMILL Warning*** form.
- **Save Project As**:- **C:\users\training\COURSEWORK\PowerMILL-Projects\CornerFinishPro_Example**

Corner MultiPencil Finishing

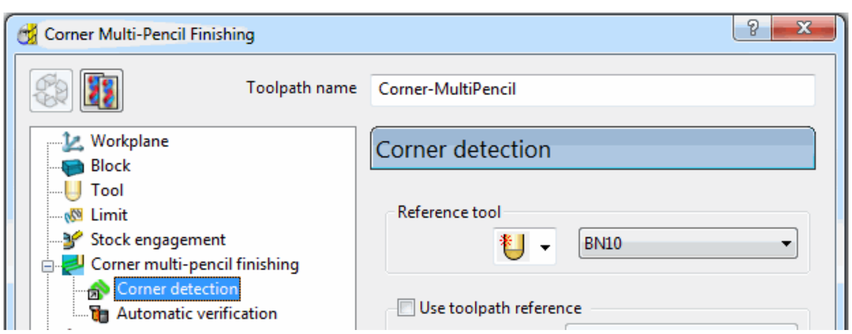
- From the **Toolpath strategies** form, select **Corner Multi-pencil Finishing**



- Change ***Output*** to **Shallow**.
- Input the rest of the ***settings*** in 3 the pages of the form exactly as shown.

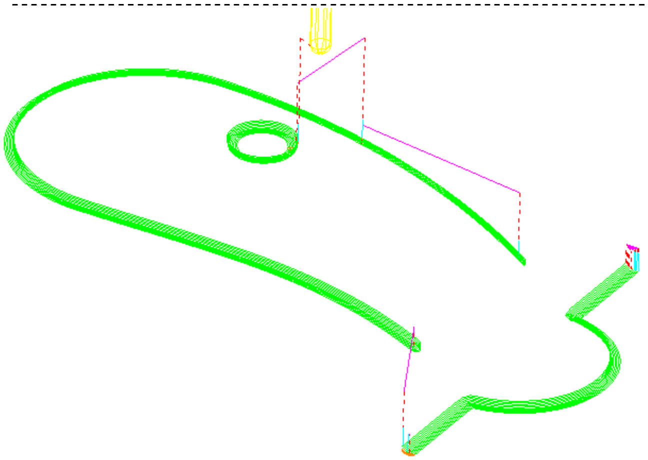


BN5 tool ***Active***

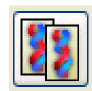


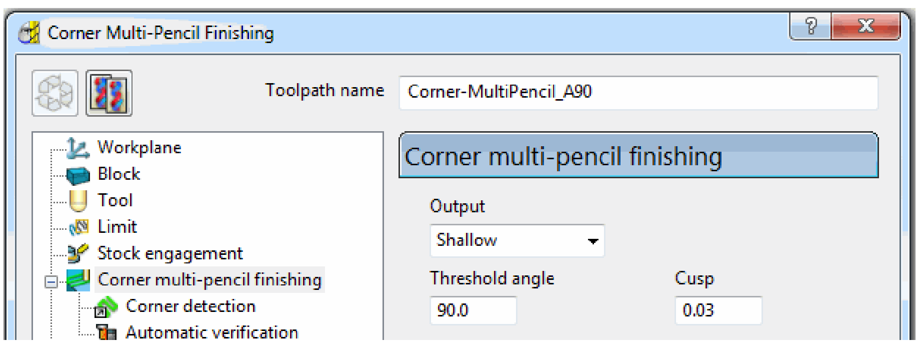
Reference tool **BN10**

- Select **Calculate** but do not close the form.



MultiPencil appears to be very similar to the **Corner Along** strategy. There however fundamental differences one being the toolpath segments in **Corner Along** are offset from the outer profile inwards, whereas in **MultiPencil** the toolpath segments are offset outwards from the true intersection.

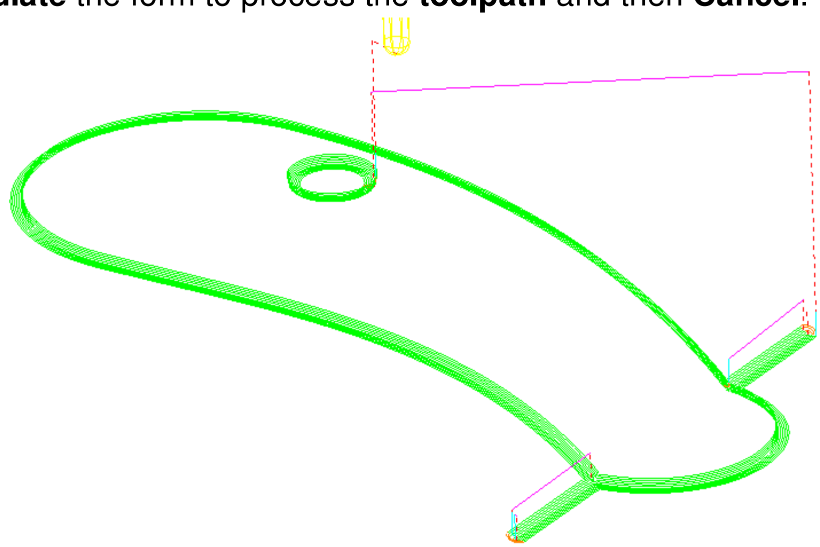
- Select the **Copy Toolpath** icon from the form. 



- Change the **Threshold Angle** to **90**.

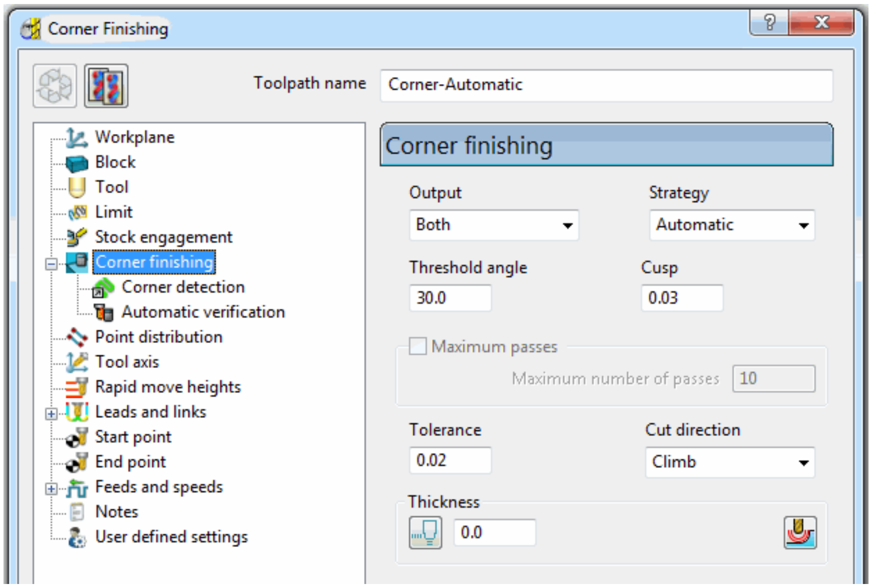
This has the effect of creating an unlimited **Shallow** machining area for the toolpath. No toolpath will be created if ***Output*** **- Steep** is selected with a **Threshold Angle** of **90**.

- **Calculate** the form to process the **toolpath** and then **Cancel**.



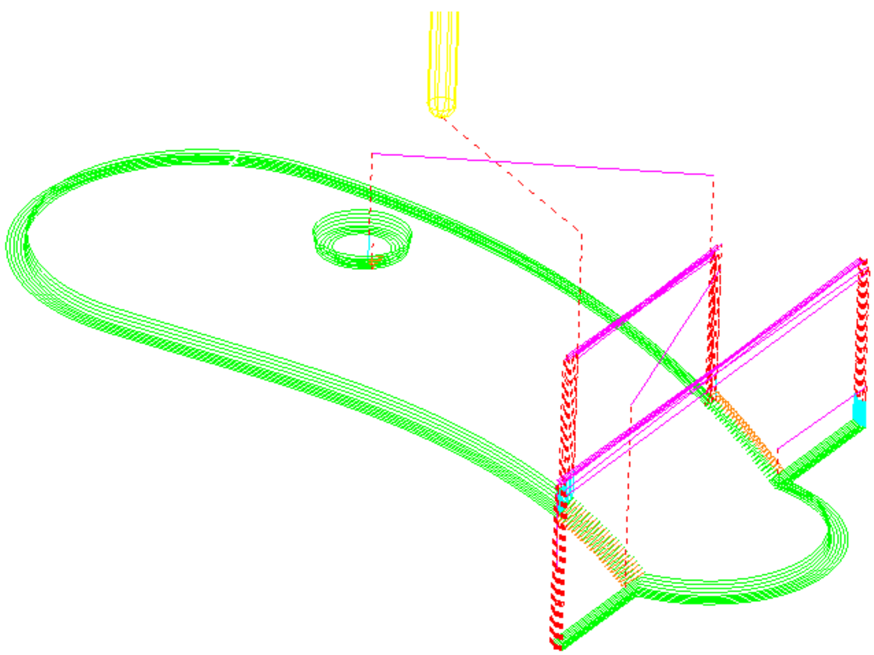
Corner Automatic Finishing

- From the **Toolpath strategies** form, select **Corner Finishing** and input data into the form exactly as shown below.



- Use ***Strategy*** **-Automatic**
- Change ***Output*** to **Both**.
- Change the ***Threshold angle*** back to **30**.

- **Calculate** the form to process the **toolpath** and then **Cancel**.

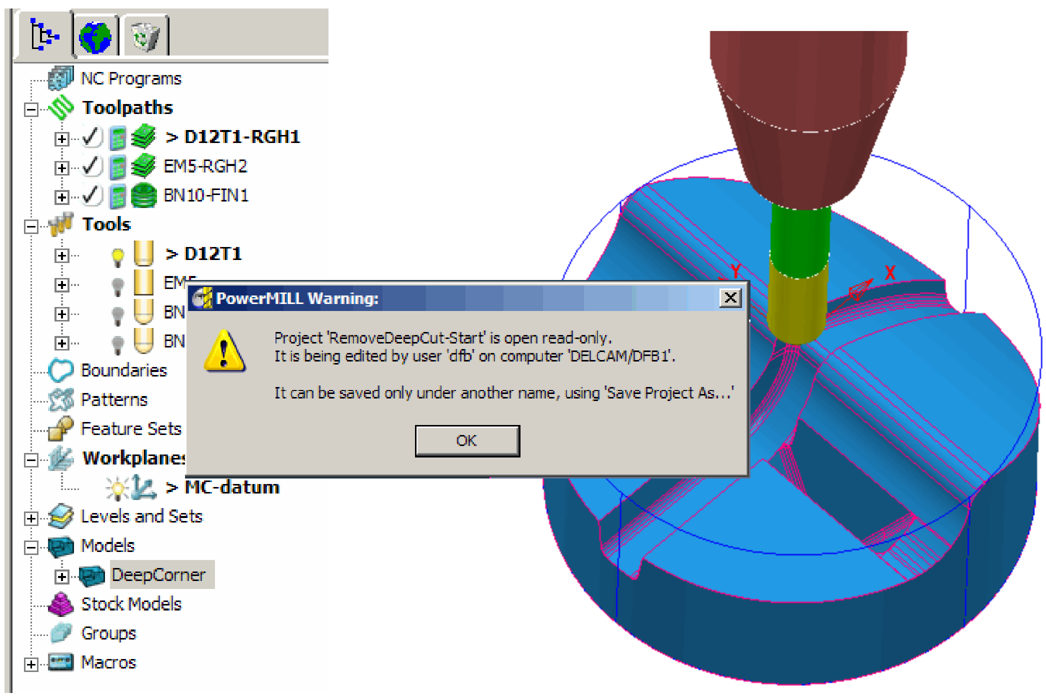


This strategy automatically uses **Along** for the **Shallow** areas and **Stitch** for the **Steep** areas.

Deep Cut Prevention

In cases were the **Reference Tool** is too large to machine all the way down a deep pocket the excessive stock remaining could cause a smaller ***Corner Finishing*** **Tool** to overload and break. A tick box option is available in the **Corner Finishing** strategies (Except ***Corner Pencil***) to **Remove Deep Cuts**. The user would then ***locally*** apply a more suitable strategy such as **Constant Z Finishing** into the deep areas.

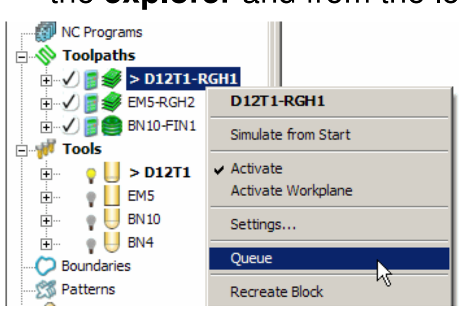
- **Delete All** and **Reset forms**.
- From **File - Open Project** select the read-only **Project** :-
  C:\users\training\PowerMILL_Data\Projects\RemoveDeepCut-Start



- Select **OK** in the ***PowerMILL Warning*** form.
- **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\RemoveDeepCut

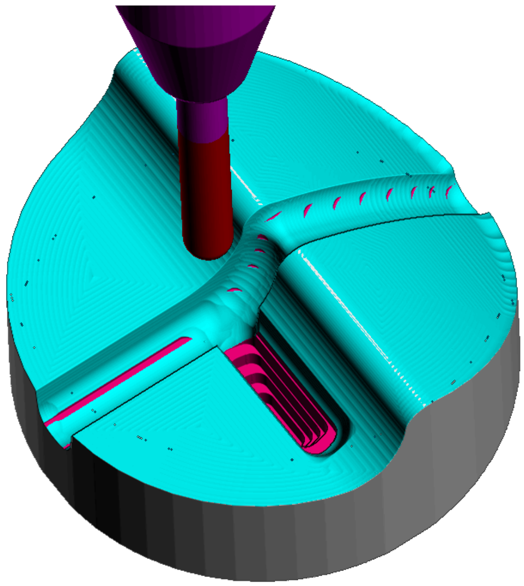
Note: The existing 3 strategies in the above **Project** are as yet, unprocessed (***Calulator*** icon).

- Right mouse click on each of the unprocessed **Toolpaths**, one at a time in the **explorer** and from the local menu select **Queue**.

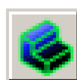


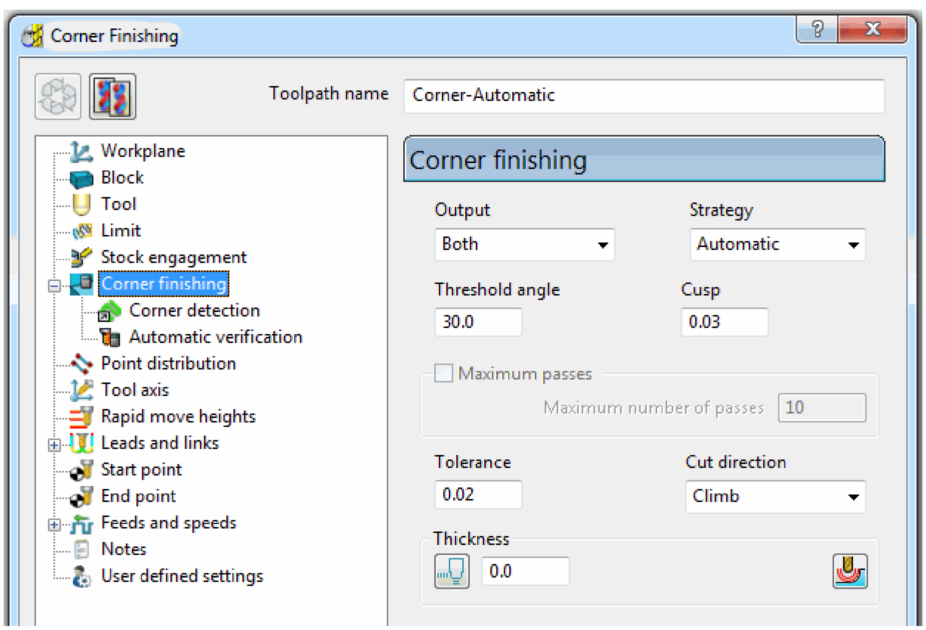
The 3 '***Queued***' **strategies** will start processing in sequential order while the user continues to work in the **PowerMILL Project**.

- **Activate** the first ***Roughing strategy*** **D12T1-RGH1** and from it's local menu select **Simulate from Start**.
- Open a **Viewmill simulation** session and **simulate** each toolpath in sequential order.



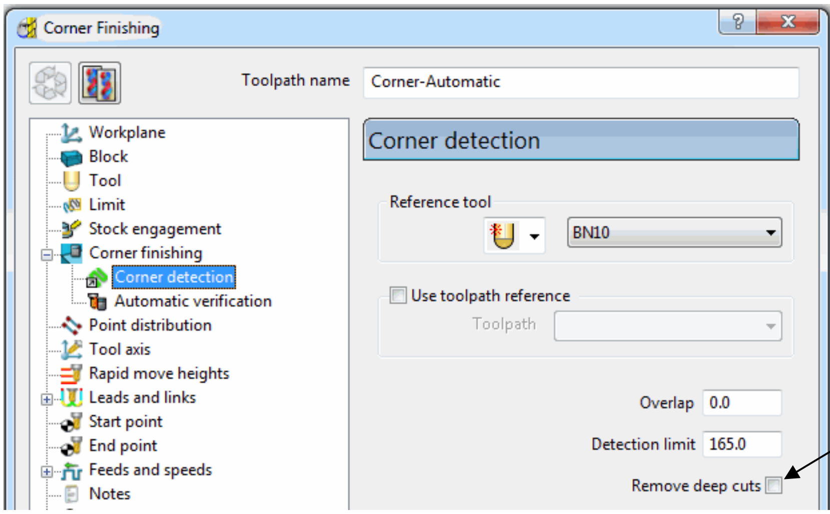
The ***Pale Blue*** area shows where the ***Dia 10 Ball Nosed*** tool has successfully machined. The deep pocket (**Purple**) is inaccessible and still includes large steps down the side wall. This excess stock could cause a smaller finishing tool to break if it attempts to **Corner Finish** down the pocket. To prevent this, the **Remove Deep Cuts** option will be used as demonstrated below.

- Select the **No Image** icon  on the **ViewMILL** toolbar to display **PowerMILL** only in the graphics area.
- **Activate** the **BN4** tool.
- From the **Toolpath Strategies** form, select **Corner Finishing**.
- Input settings in the main page of the form as shown below:-



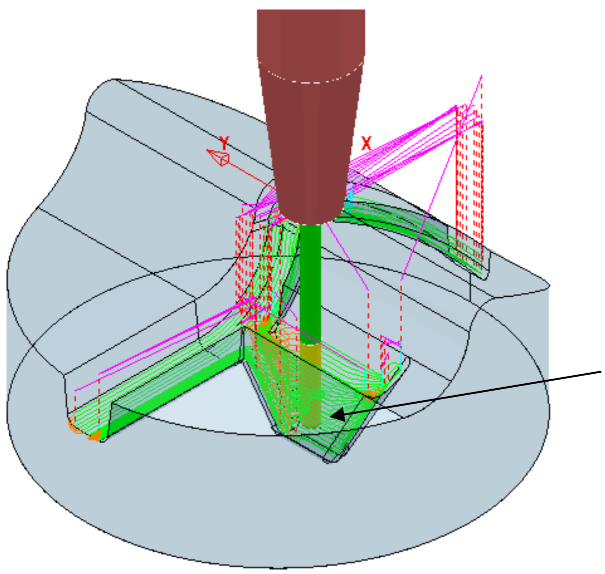
- ***Output*** **– Both**
- ***Strategy*** **- Along**
- ***Threshold angle*** **- 90**

- From the local ***explorer*** select the **Corner detection** page.



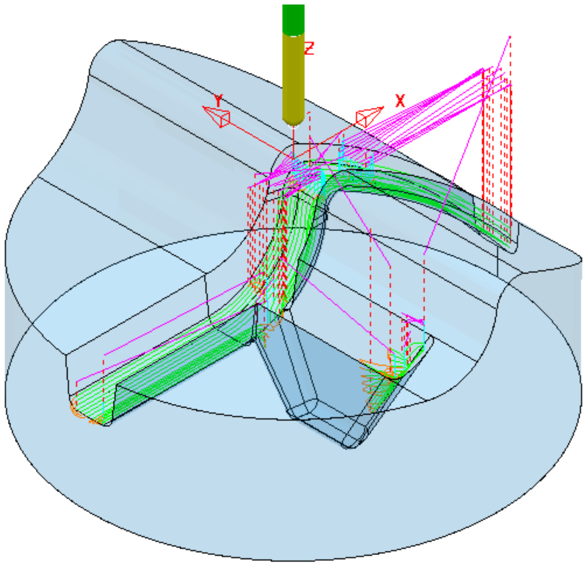
- ***Reference Tool*** - **BN10**
- ***Untick*** - **Remove deep cuts**

- Select **Calculate** to process the **toolpath** and then **Cancel** the form.



The **Corner Finishing** has operated all the way down to the base of the deep pocket.
Due to the ***excess stock*** this could cause the cutter to break.

- Re-open (***Settings***) the above **Corner Finishing - Along** strategy and select **Make a Copy**. 
- **Rename** the new strategy as **BN4-Fin1-NODeepCut** and this time ensure that the **Remove Deep Cuts** box is ***ticked*** before selecting **Calculate**.



The **Corner Finishing** has not occurred in the deep pocket that is inaccessible to the **BN10** reference tool

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

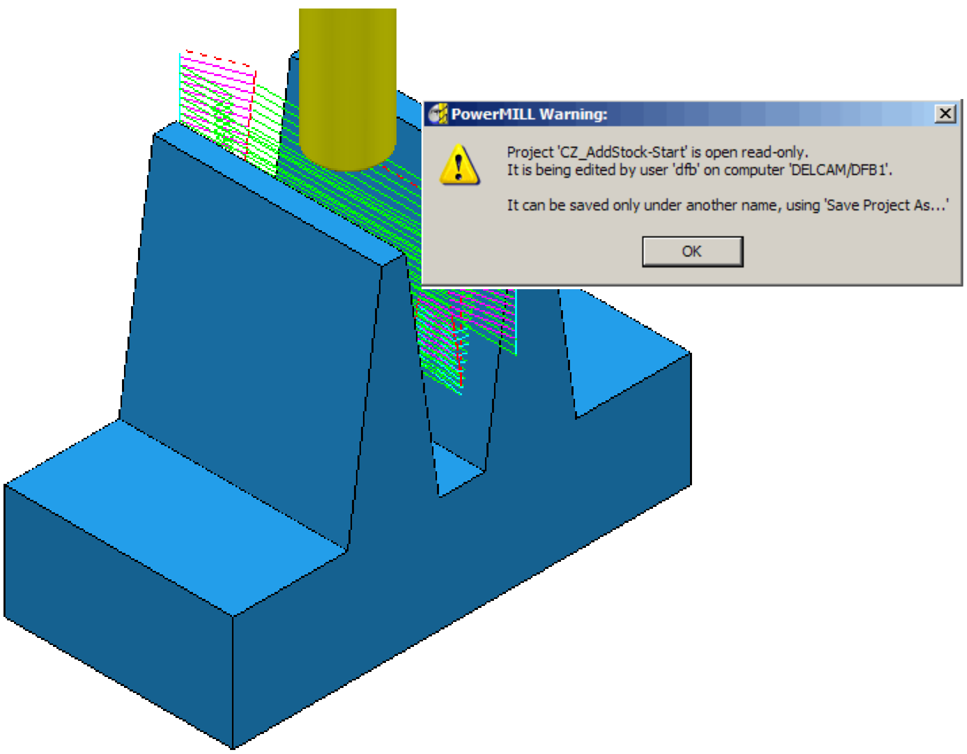
Exercise

- Create a material **Block** defined by a **Box**, locally around the **Pocket Surfaces** (selected).
- Using the same **BN4** tool, create a **Constant Z finishing** strategy to step down the sidewalls.

Constant Z Finishing – Additional Stock option

The **Additional Stock** option causes a **Constant Z** strategy to ignore the **Pocket** machining setting below the point where the width of a ***deep, angled slot*** reaches a ***specified width***. Above this point the **tool** will continuously machine down each angled face in turn. Below this point the ***tool tracks*** will alternate in descending, height order from one face to the other.

- **Delete All** and **Reset forms**.
- From **File - Open Project** select the read-only **Project**:-
  C:\users\training\PowerMILL_Data\Projects\CZ_AddStock-Start

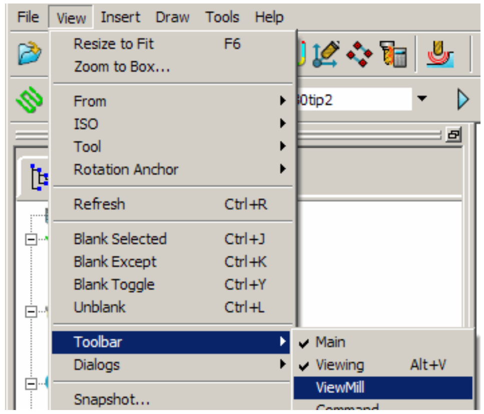


- Select **OK** in the ***PowerMILL Warning*** form.

- **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\CZ_AddStock-EX1

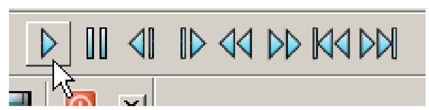
- ***Right mouse click*** on the **Toolpath** named **Roughing** and from the local menu select **Activate** to reinstate the original **Block**.
- ***Right mouse click*** on the **Toolpath** named **Roughing** and from the local menu select **Simulate from Start**.

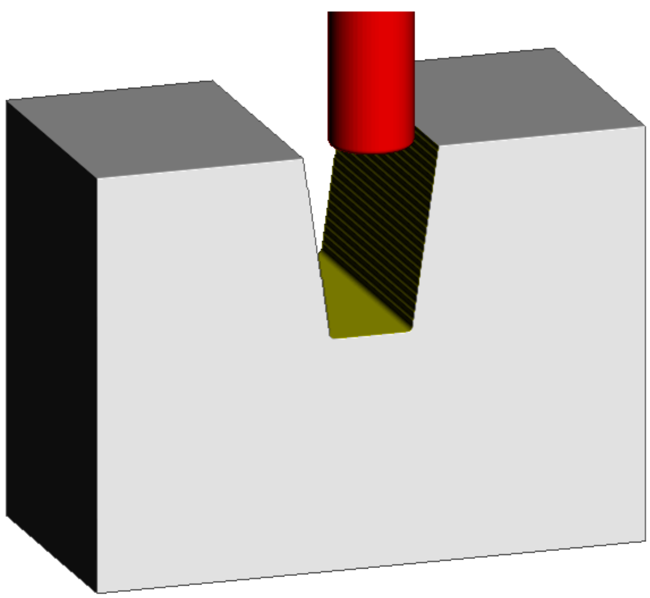
- From the top **Pulldown** menus select (tick) **View - Toolbar - ViewMILL** to open the **ViewMILL** toolbar (if not already open).



- On the **ViewMILL** toolbar switch the **Red** sphere icon to **Green** to connect the **simulation** to **ViewMILL**.

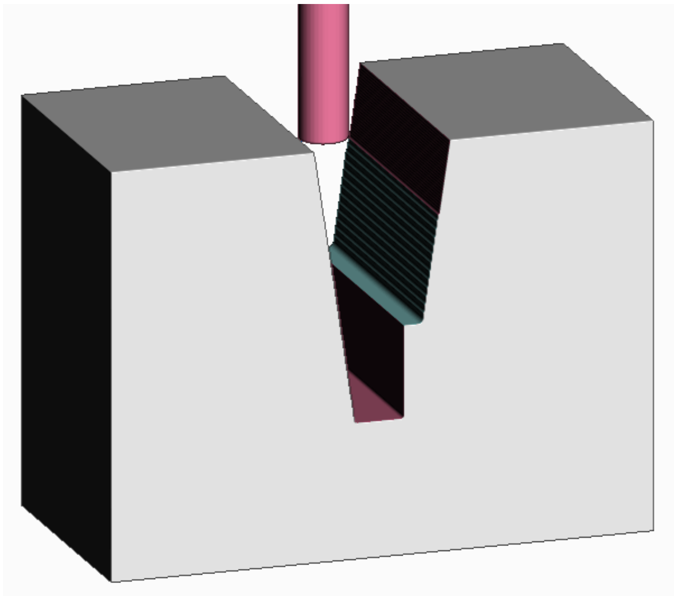
- Select the **Rainbow Shading** option in the **ViewMILL** toolbar and then select **Play** in the **simulation** toolbar.





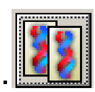
The **Roughing** strategy has not fully machined a large volume of **stock** towards the base of the slot. This will have to be removed as part of a **Constant Z** ***finishing*** strategy.

- ***Right mouse click*** on the **Toolpath** named **0_AdditionalStock** and from the local menu select **Simulate from Start** followed by **Play** in the **simulation** toolbar.

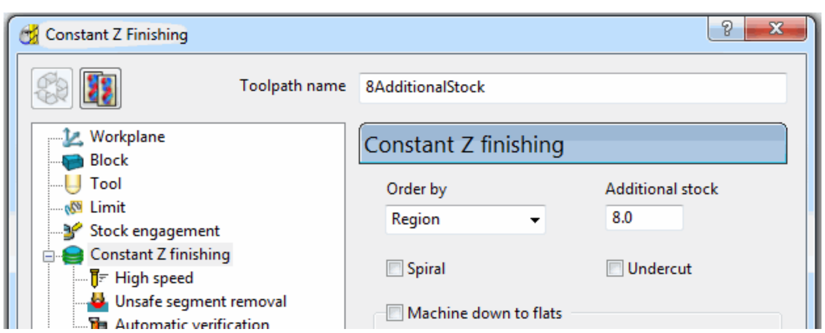


The existing **Constant Z** finishing strategy is ploughing into the stock that was inaccessible to the previously ***simulated*** **Roughing** toolpath. The excessive loading is likely to result in ***tool breakage***. To avoid this problem the **Constant Z** strategy will be modified to include the **Add Stock** option.

- In the **ViewMILL** toolbar select the **No Image** (***PowerMILL only***) view.
- ***Right mouse click*** on the **Toolpath** named **0_AdditionalStock** and from the local menu select **Activate** followed by **Settings** to open the original **Constant Z** finishing form.

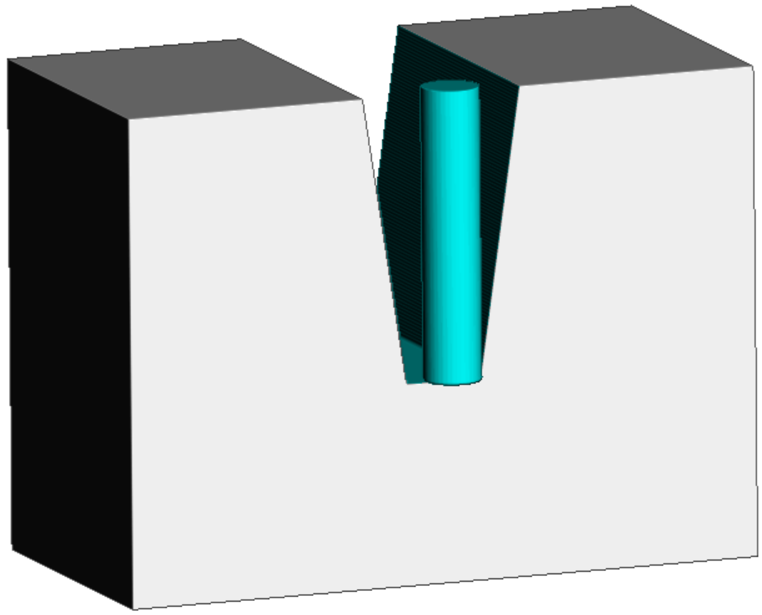
- Select the '**Create a new toolpath based on this one'** option. 

- ***Rename*** the new **Toolpath** as **8AdditionalStock** and in the **Additional Stock** box input the value **8.0** before selecting **Calculate**.



The choice of the selected **Additional Stock** value requires user intervention in estimating the width of the slot, just above the depth of material accessible to the **Roughing** strategy.

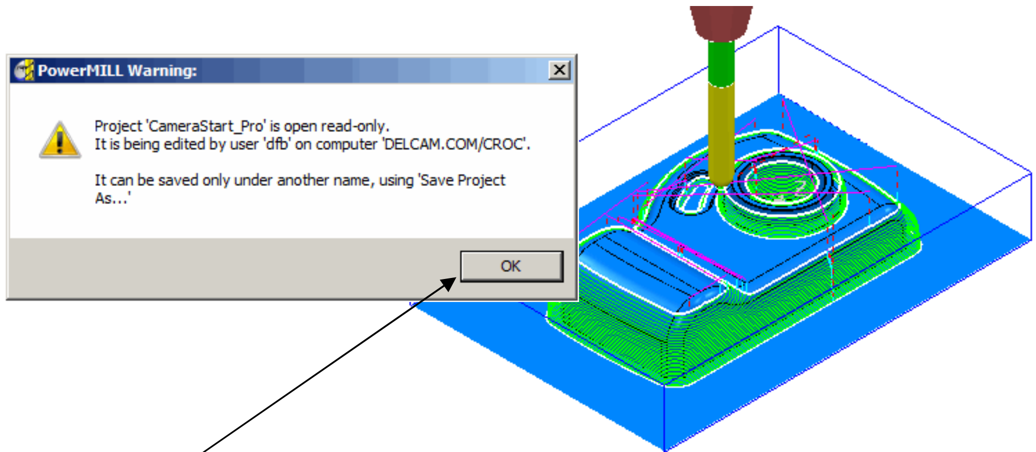
- Repeat the **ViewMILL simulation**, starting with the **Roughing** strategy as before but then following with the new **Constant Z** strategy that has the **Additional Stock** value of **8.0** included.



The new **Constant Z** finishing strategy **Pocket** machines continuously down each side of the slot up to the depth where the width of the slot is equal to **TDU + 8**. The strategy then continues but with the **Pocket** option ignored for the remaining **tool tracks**, which alternate between each side wall in descending order.

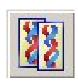
3D Offset Finishing

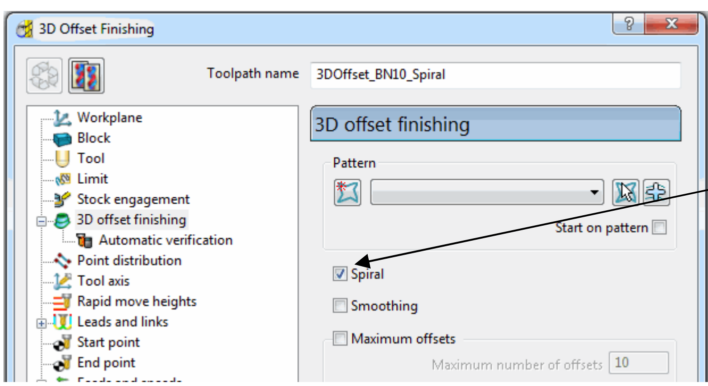
- **Open** the **Project**:-
  C:\users\training\PowerMILL_Data\projects\CameraStart_Pro



- Select **OK** to load the existing **Project** into **PowerMILL**.
- From **File** – **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL_Projects\Camera_Pro

- Right click over toolpath **3DOffset_BN10** in the **explorer** and select **Settings** from the pull down menu.

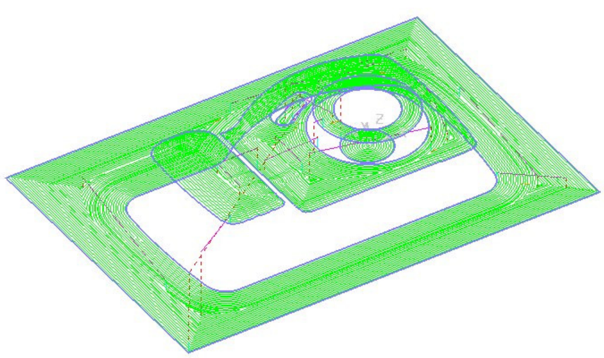
- Select the **Copy** icon on the form. 



- **Rename** the toolpath as **3DOffset_BN10_Spiral**.

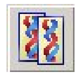
- Tick the box **Spiral**.

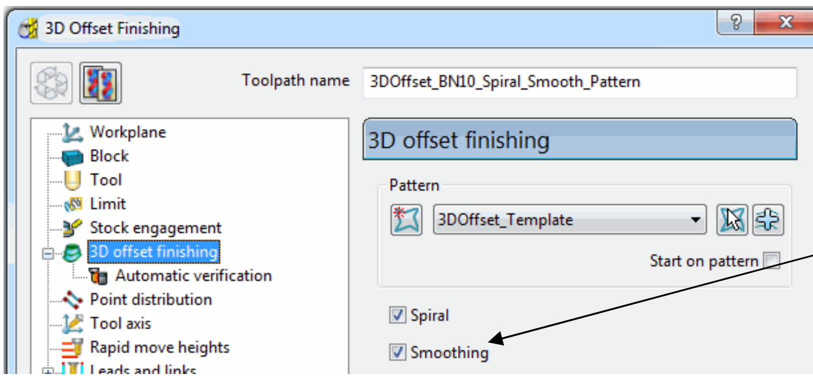
- Leave all other values the same then **Calculate** and **Cancel** the form.



The **Spiral** option is essential for HSM (High Speed Machining) applications where ideally toolpaths should be as continuous as possible.

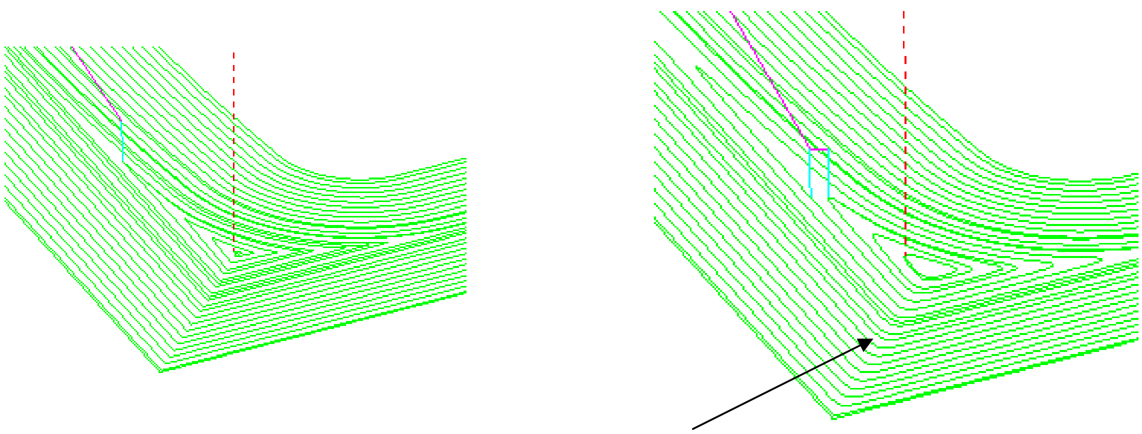
To further improve the compatibility of this strategy for HSM a **Smoothing** option can be **ticked** resulting in sharp corners being replaced with a ***smooth blend*** wherever possible.

- Right click over toolpath **3DOffset_BN10_Spiral** in the **explorer** and select **Settings** from the pull down menu.
- Select the **Copy** icon on the form. 
- **Rename** the toolpath as **3DOffsetBN10_Spiral_Smoothing**.



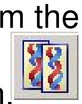
- Tick the box **Smoothing**.
- Leave all other values the same

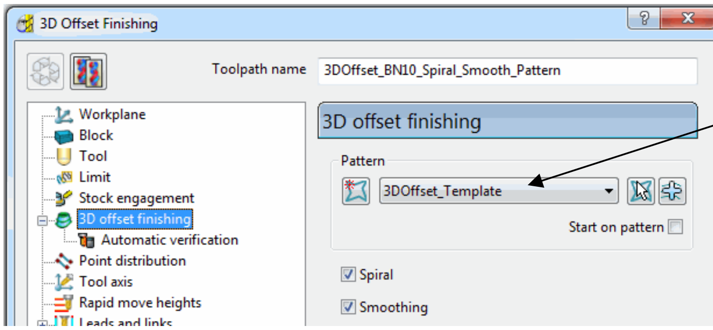
then **Calculate** and **Cancel** the form.



Smoothing applied

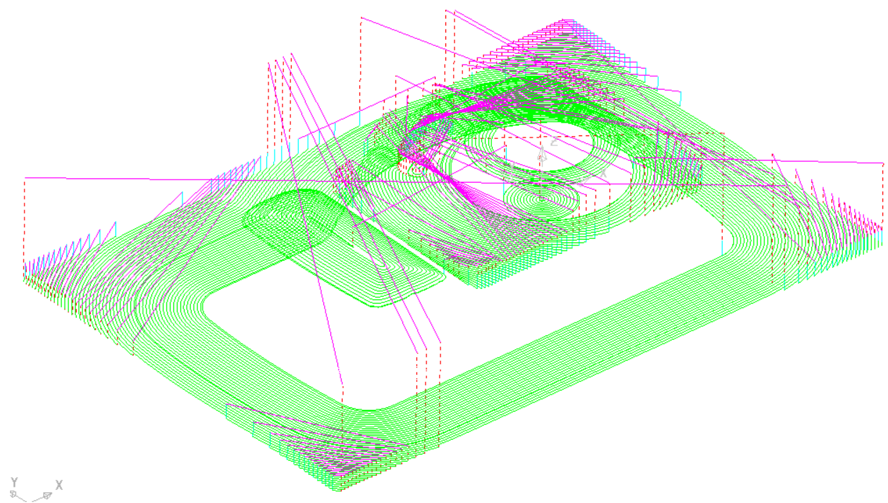
Note:- For optimum results the **3D Offset** toolpath shape can also be controlled with a suitable **Pattern** used as the basis for the initial contour of each set of tooltracks. The user will need to be able to create a suitable **Pattern** which in this case consists of ***selected segments*** copied from the **Boundary**.
Note: **Patterns** are covered in more detail in **Chapter 11**.

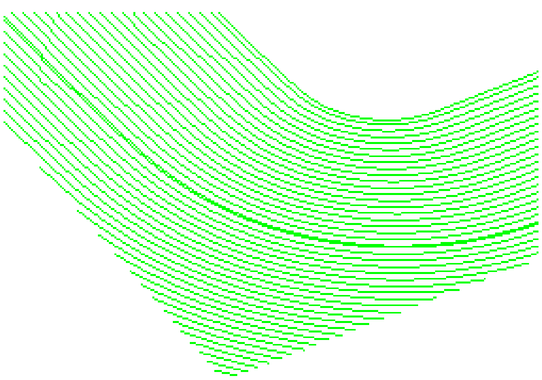
- Right click over toolpath **3DOffset_BN10_Spiral_Smoothing** in the **explorer** and select **Settings** from the pull down menu.
- Select the **Copy** icon on the form. 
- **Rename** the toolpath as **3DOffsetBN10_Spiral_Smoothing_Pattern**.



- Input the ***Pattern*** - **3DOffset_Template**.

- Leave all other values the same then **Calculate** to process the **toolpath** and then **Cancel** the form.





The **Pattern** segments have now taken over from the **Boundary** to ***control*** the actual toolpath shape.
The **Boundary** segments are now used solely to limit the areas covered by the toolpath.

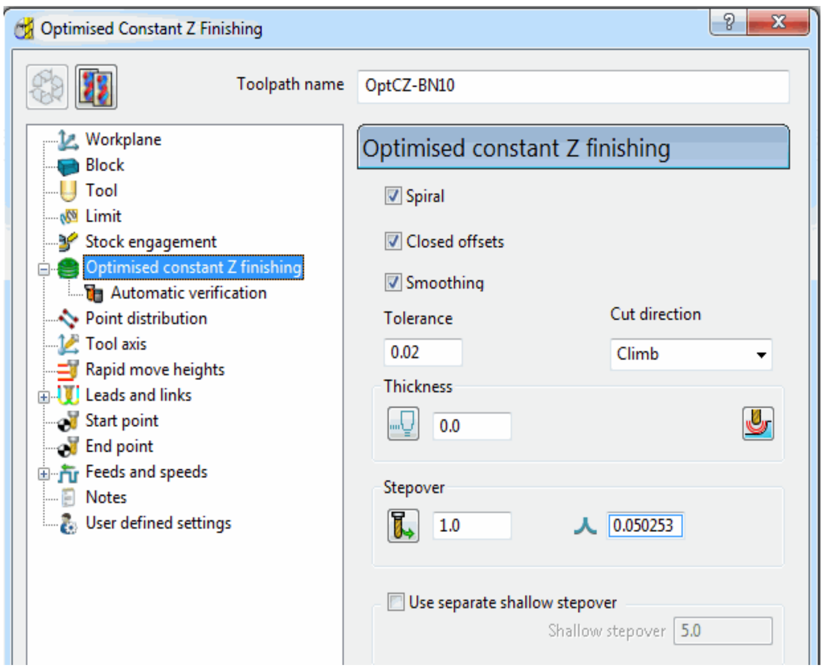
- **Save** but <u>do not close</u> the **Project** as it will be continued in the following section.

Optimised Constant Z Finishing

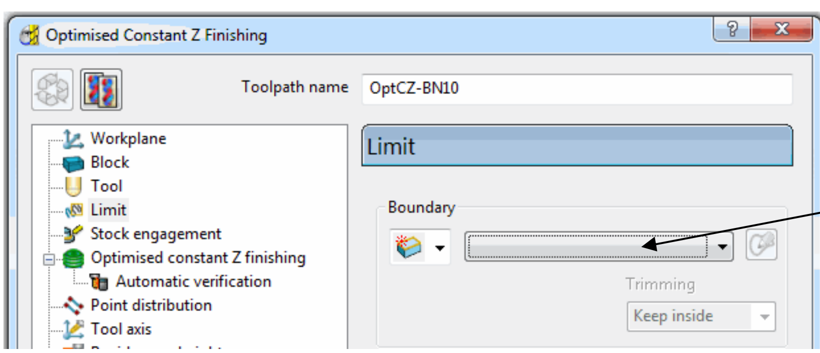
This strategy is a mixture of **Constant Z** and **3D Offset** machining. Where the model is steep, **Constant Z** is applied and for other areas, **3D Offset** is applied.
Closed Offsets if ticked will cause the **3D Offset** areas of the machining to be ordered to occur from outside to inside. The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

- **Deactivate** the ***Shallow Boundary***, **ShallowBN10** in the **explorer**.

- Open the **Toolpath Strategies** form  and from **Finishing** select **Optimised Constant Z Finishing** from the form then **OK**.



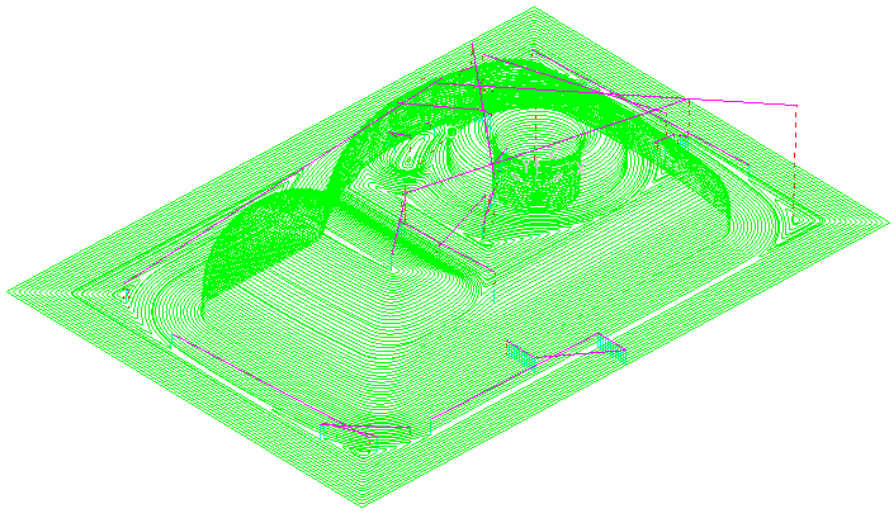
- Enter **Toolpath name** - **OptCZ-BN10**.
- ***Tick*** -**Spiral**, **Closed Offsets,** and **Smoothing**.
- Enter the **Tolerance** as **0.02**.
- Set the **Direction** to **Climb**.
- Input a **Stepover** value of **1**.



- Make sure that the **Boundary** is <u>not</u> selected.

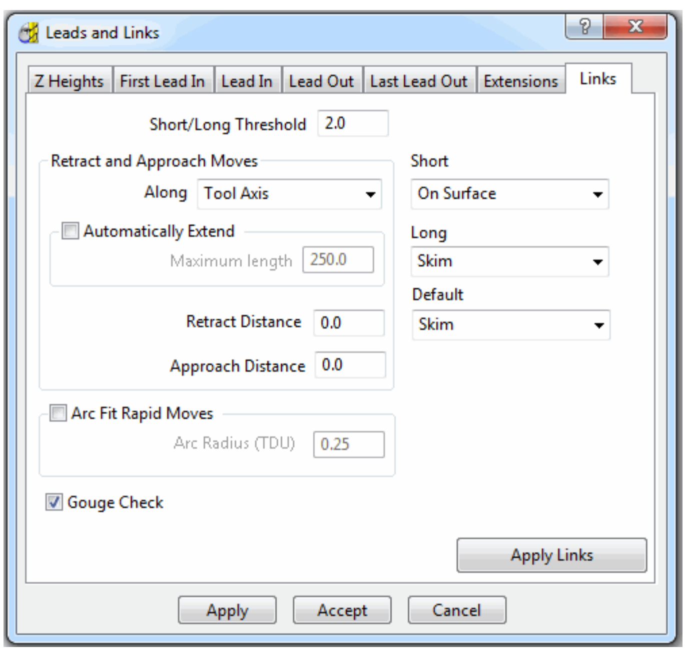
- Leave all other values the same then **Calculate** to process the **toolpath** and then **Cancel** the form.

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



Optimized Constant Z generally performs well on **Models** that contain extreme **Steep** and **Shallow** areas, with a minimum of areas around 45 degrees to the vertical. A typical application would include many types of Thermoform Packaging.

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

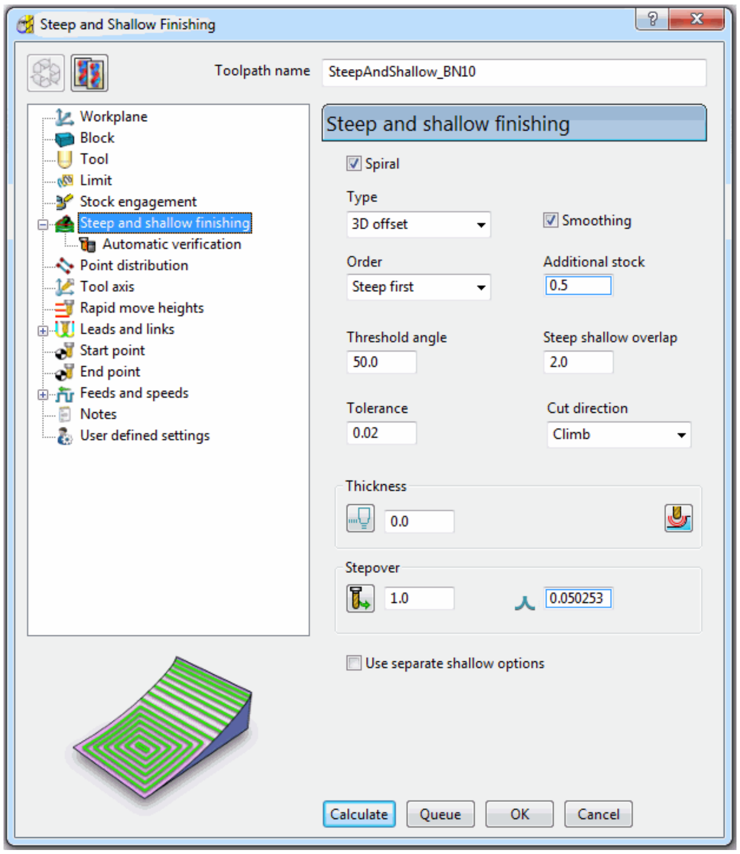


- Change the **Short/Long Threshold** to **2**.
- Change the **Short** links to **On Surface**.
- Change the **Long** and **Default** links to **Skim**.
- **Tick** the **Arc Fit Rapid Moves** box with an **Arc Radius** value of **0.5**
- **Apply** and **Accept** the form.

Steep and Shallow Finishing

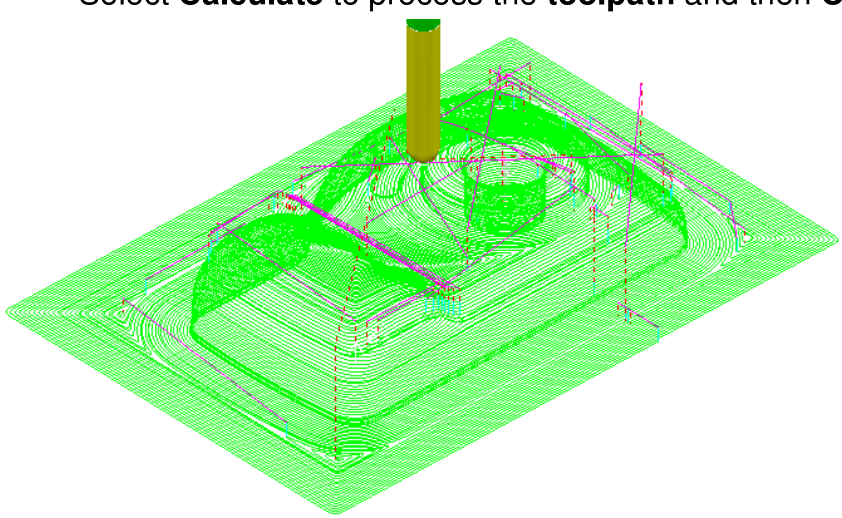
This is an alternative strategy to **Optimised Constant Z** with different options which include a user defined **Threshold Angle** between the **Constant Z,** and **3D Offset** areas of the toolpath as well the option to specify an **Overlap** value between them.

- From the **Finishing** strategies form, select **Steep and Shallow finishing**.
- Enter data exactly as shown in the following illustration.



- Enter ***Name*** as:- **SteepAndShallow_BN10**

- Select **Calculate** to process the **toolpath** and then **Cancel** the form.

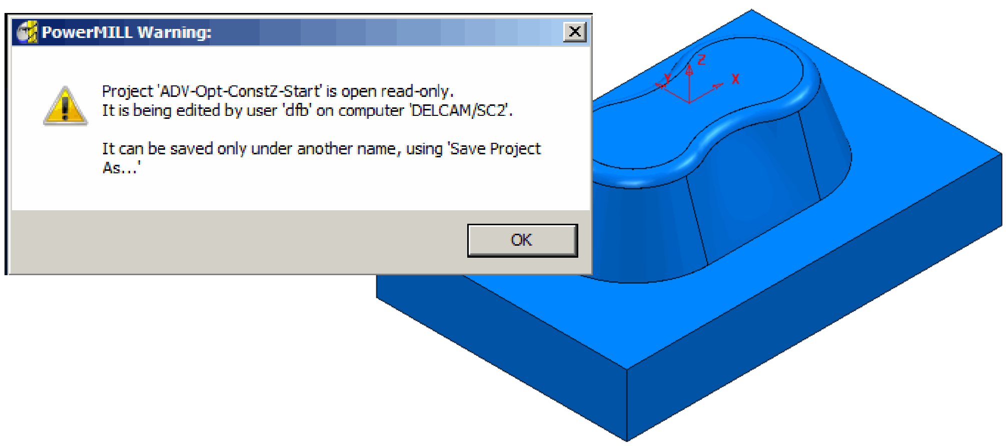


The user is able to control the angular threshold between the ***3D Offset*** and ***Constant Z*** components of the hybrid **Steep and Shallow finishing** strategy.

Steep and Shallow Constant Z with separate Shallow Stepover

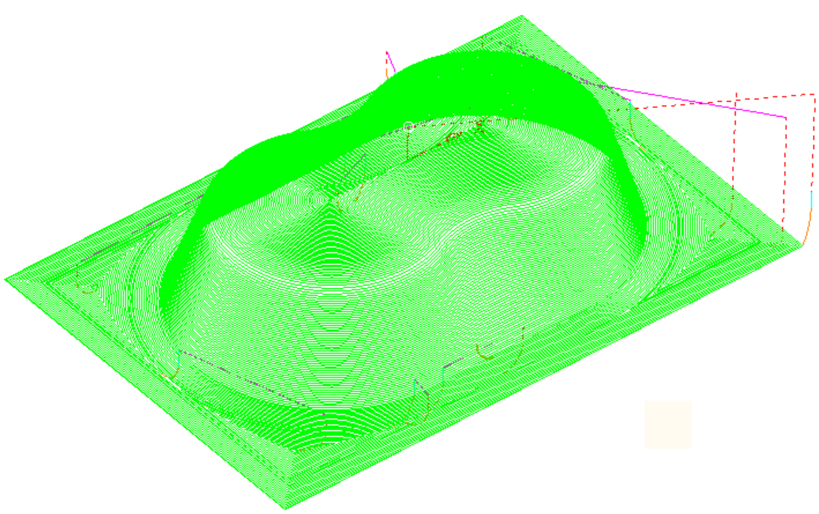
Both **Optimised Constant Z** and **Steep and Shallow** have an option to apply a separate, larger **Stepover** across the **Shallow** areas. This is designed to provide more efficient machining with a **Tip Radiused Tool**, where the **Shallow** areas are perfectly ***flat***. The **Steep** walls are machined using the **tip radius** with a ***fine*** **Stepover** and the ***flat*** areas using the base of the tool with a ***large*** **stepover**.

- **Delete All** and **Reset forms**.
- From **File - Open Project** select the read-only **Project** :-
  C:\users\training\PowerMILL_Data\Projects\ADV-Opt-ConstZ-Start

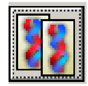


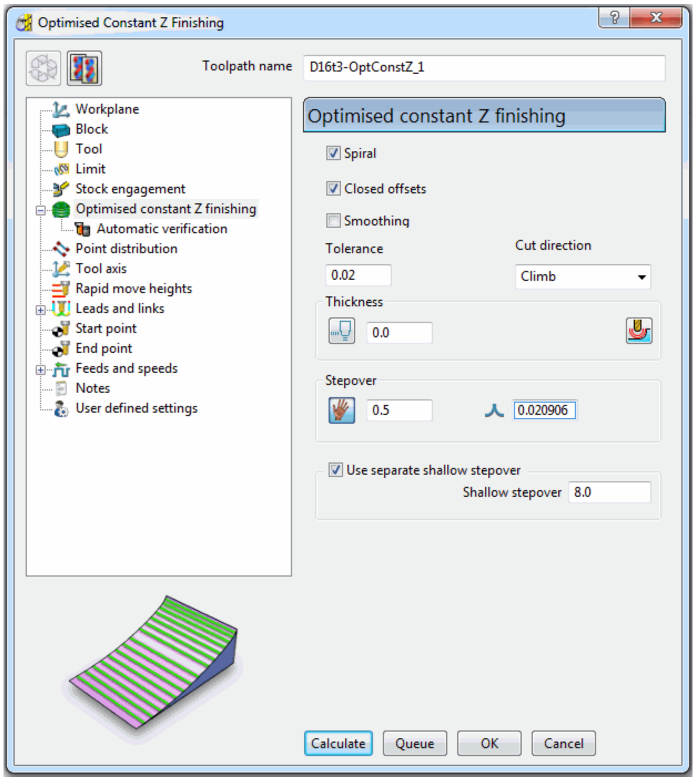
- Select **OK** to load the existing **Project** into **PowerMILL**.
- From **File – Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL_Projects\ADV-CZ-example

- Right click over toolpath **D16t3-OptConstZ** in the **explorer** and select **Settings** from the pull down menu.

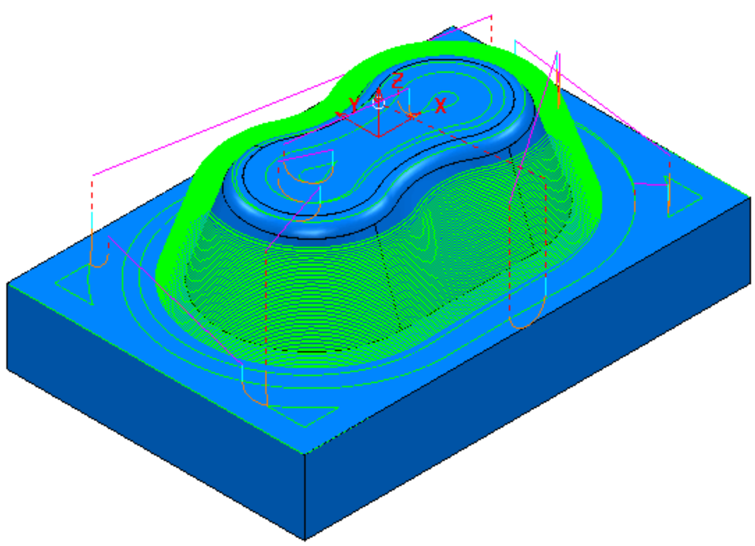


The component form includes 2 ***Flat*** *areas* which are currently machined with the same ***small*** **stepover** as the angled walls. For more efficiency, an option is available for a larger **stepover** to be independently applied to the **shallow** (***Flat***) areas.

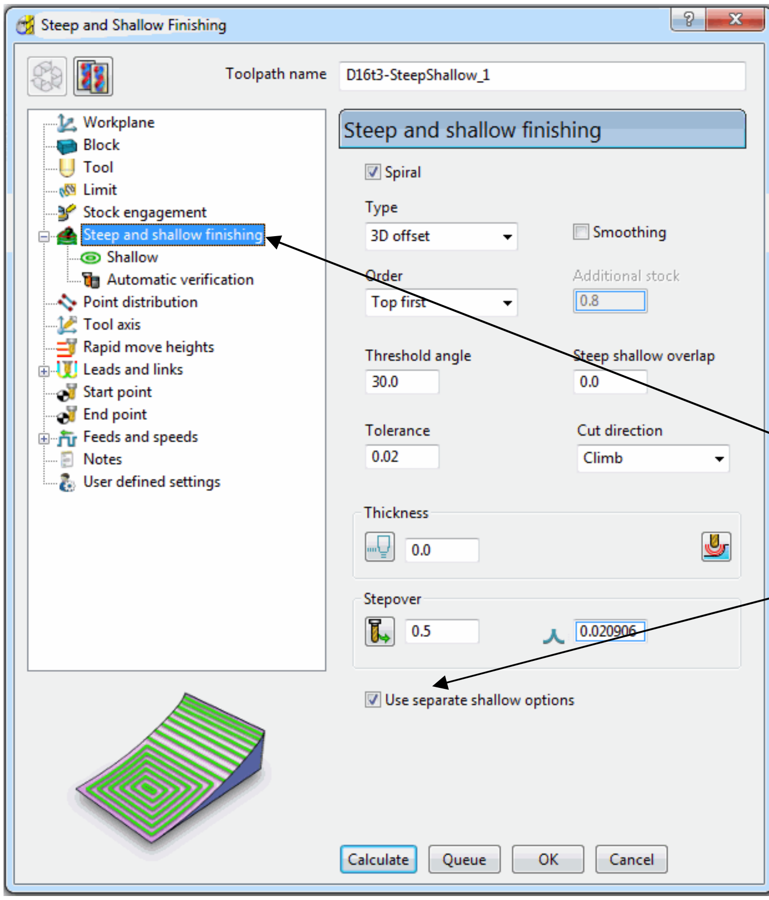
- Select the **Copy** icon on the form. 



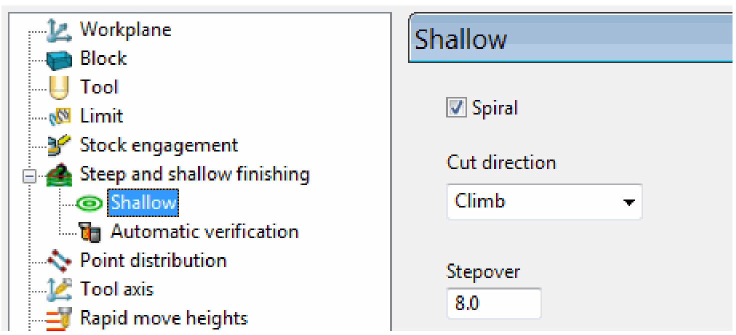
- **Tick** The **Box** named ***Use Separate Shallow Stepover*** and input a ***Shallow Stepover*** of **8.0**
- Select **Calculate** to process the **toolpath** and then **Cancel** the form.



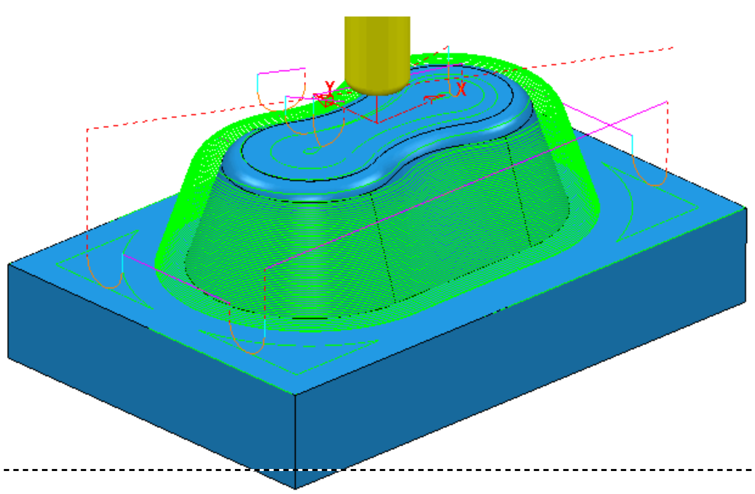
- Open a new **Steep and Shallow finishing** form and input the same **separate - Shallow stepover** as used in the previous ***Optimised Constant Z*** strategy.



The **Shallow** options only become available in the local **explorer** if this box is ***ticked***.



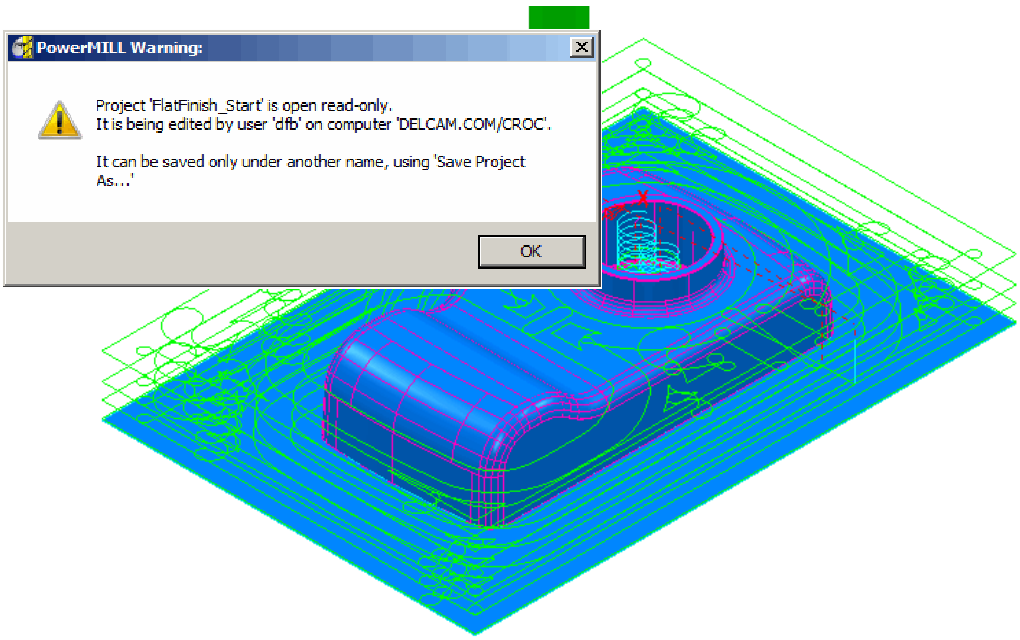
- Input the **Shallow** options as shown left.
- **Calculate** the toolpath before selecting **Cancel**.



Flat Finishing

These strategies finish flat areas only, the definition of this being controlled by a **Flat Tolerance** value. Where applicable, they are usually run immediately after the main **3D Area Clearance** operation. The **Project** imported for the next example already includes a preliminary **3D Area Clearance** strategy.

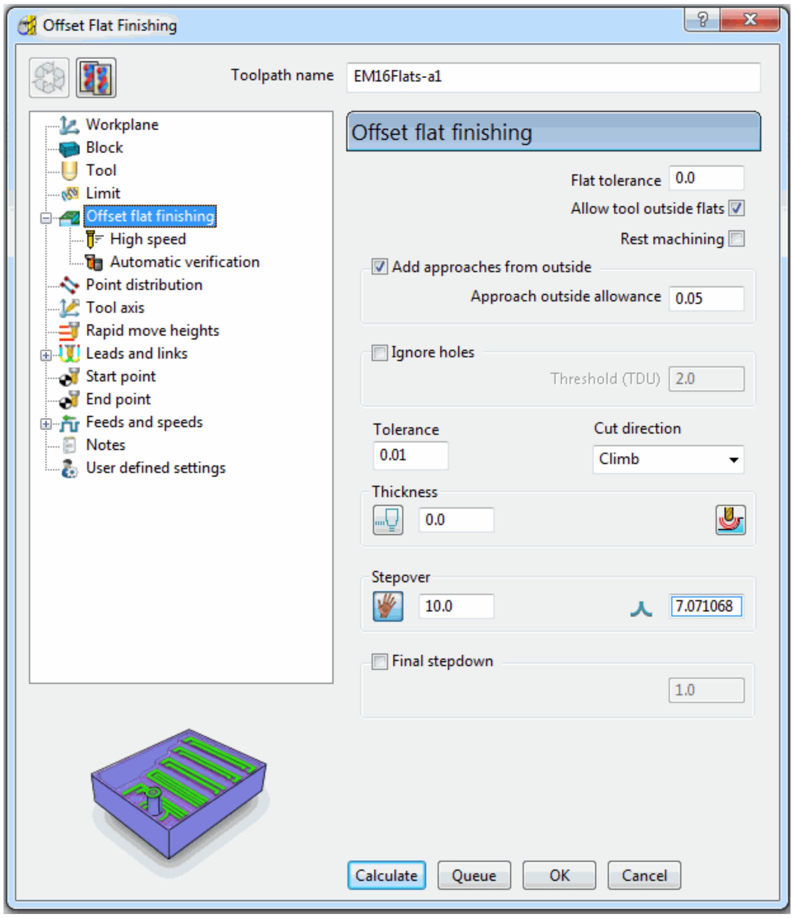
- **Delete All** and **Reset forms**.
- From **File - Open Project** select the read-only **Project**:- **C:\users\training\PowerMILL_Data\Projects\FlatFinish_Start**

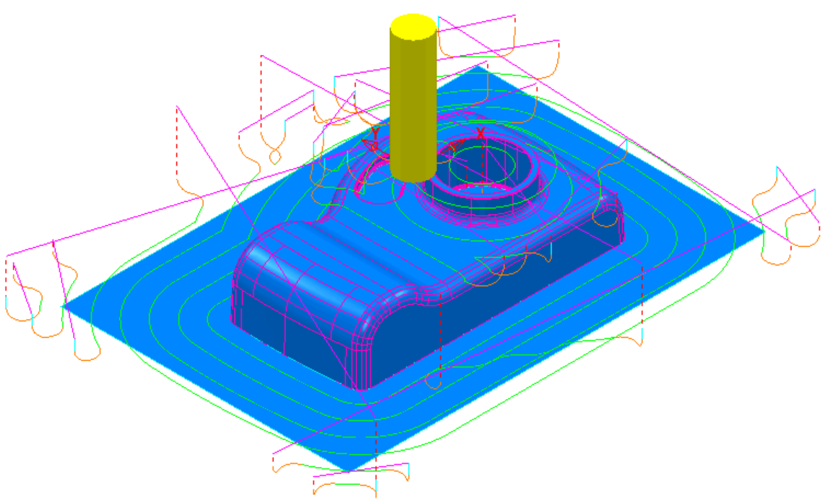


- Select **OK** in the ***PowerMILL Warning*** form.
- **Save Project As**:- **C:\users\training\COURSEWORK\PowerMILL-Projects\FlatFinish-Example**

Offset Flat Finishing

- In the **explorer - Activate** the **Dia 16** End Mill (**EM16**).
- From the ***Strategy Selector*** form, select **Offset Flat Finishing**.
- Fill in the form exactly as shown on the following page and select **Calculate**.



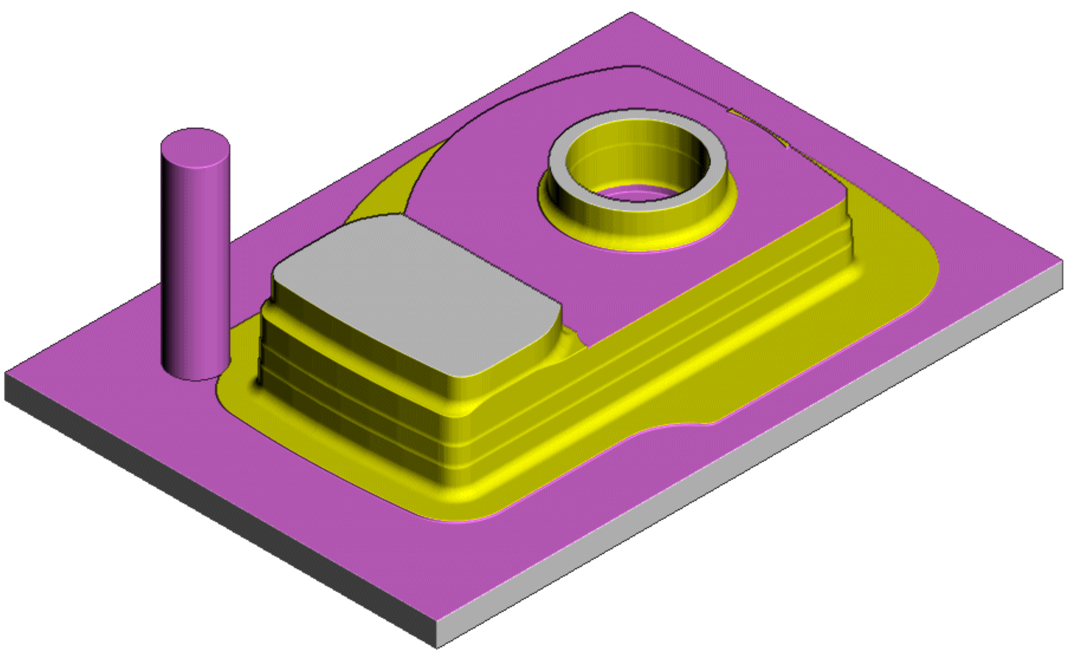


The **Offset Flats** finishing strategy will appear as shown left. In this case suitable **Leads** have been applied retrospectively:-
Lead In/Out –
***1st choice* -- Pocket Centre**
***2nd Choice* -- Horizontal Arc**

The original imported **Project** contains both an **Offset Area Clear** roughing strategy and an **Optimised Constant Z** finishing strategy. It is recommended that **Offset Flats** (***EM16Flats-a1***) is run directly after the **3D Area Clearance** (***D16t3-a1***). For this reason, it would be a good idea to move it one place upwards in the **explorer - toolpaths** area.

- In the **explorer**, move the **cursor** over the **Offset Flats** toolpath (***EM16Flats-a1***) and with the **left mouse key** depressed, **drag it** to the new position.

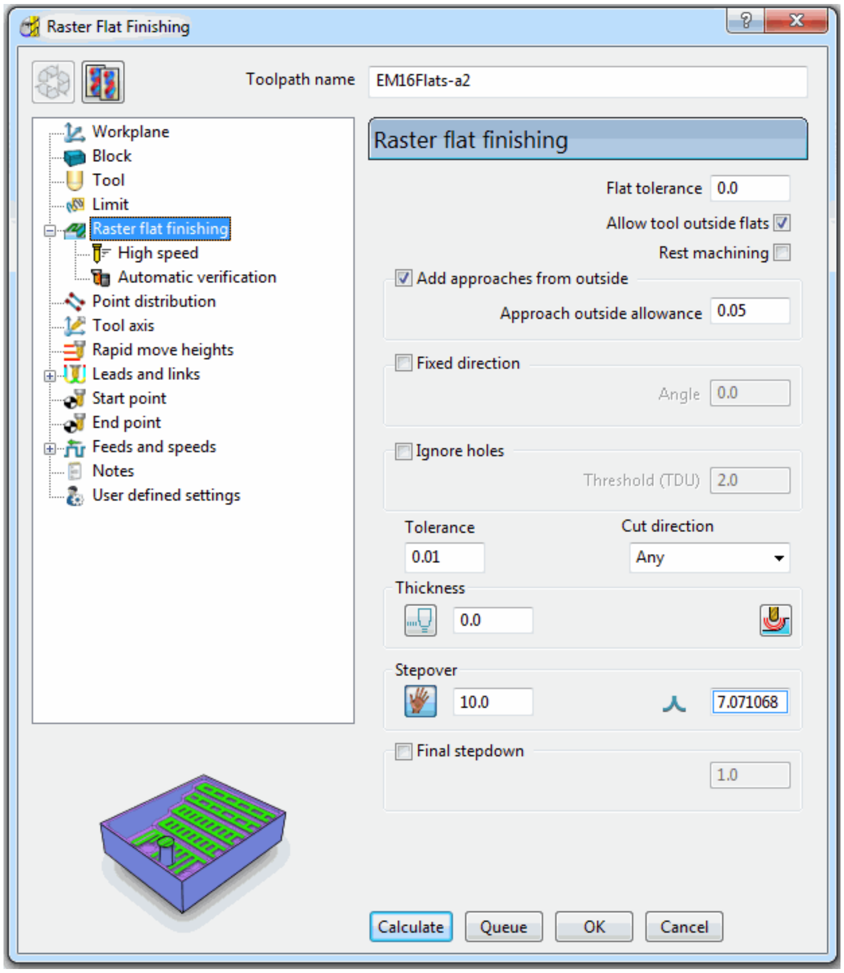
- Perform a **ViewMILL simulation** on all 3 toolpaths to check that method and result of material removal is acceptable.

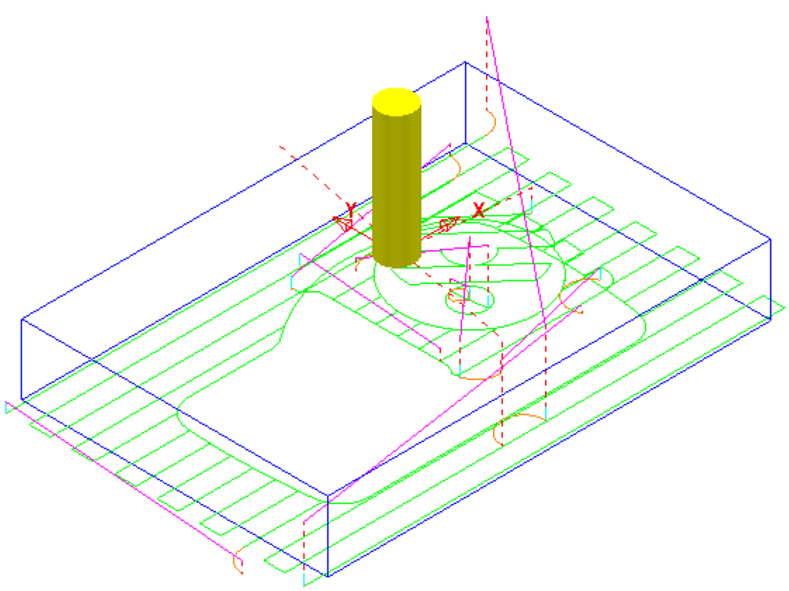


The **Offset Flats** strategy is shown above nearing the end of the **ViewMILL** simulation.

Raster Flat Finishing

- From the **Strategy Selector** form, select **Raster Flat Finishing**.
- Fill in the form exactly as shown below and select **Calculate**.

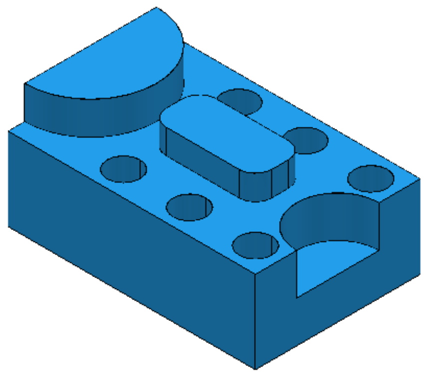




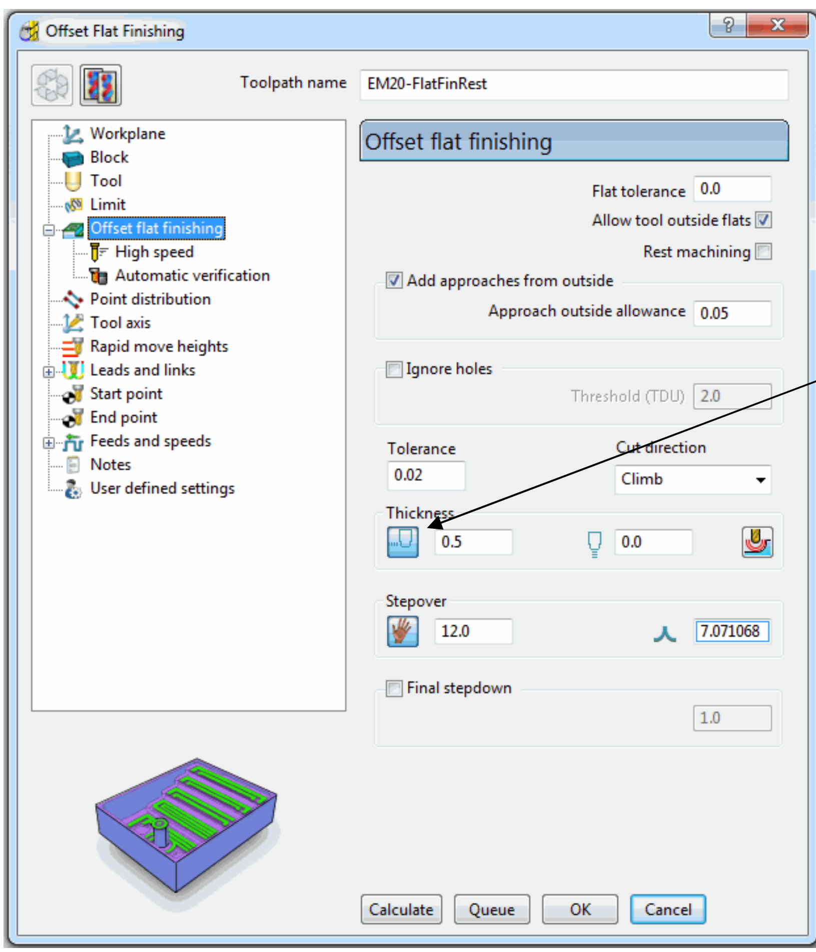
Raster Flat strategy

Flat Machining example

- **Delete All** and **Reset Forms**
- **Import** the **model**:-
  C:\users\training\PowerMILL_Data\Models\Flats.dgk

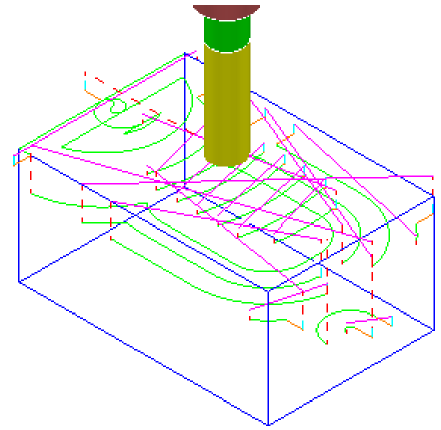


- Create a **20mm** diameter **End Mill** tool named **EM20**.
- **Calculate** the (default) **Block** and reset the **Safe** heights.
- Open the **Offset Flat Finishing** form and fill in as below.

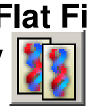


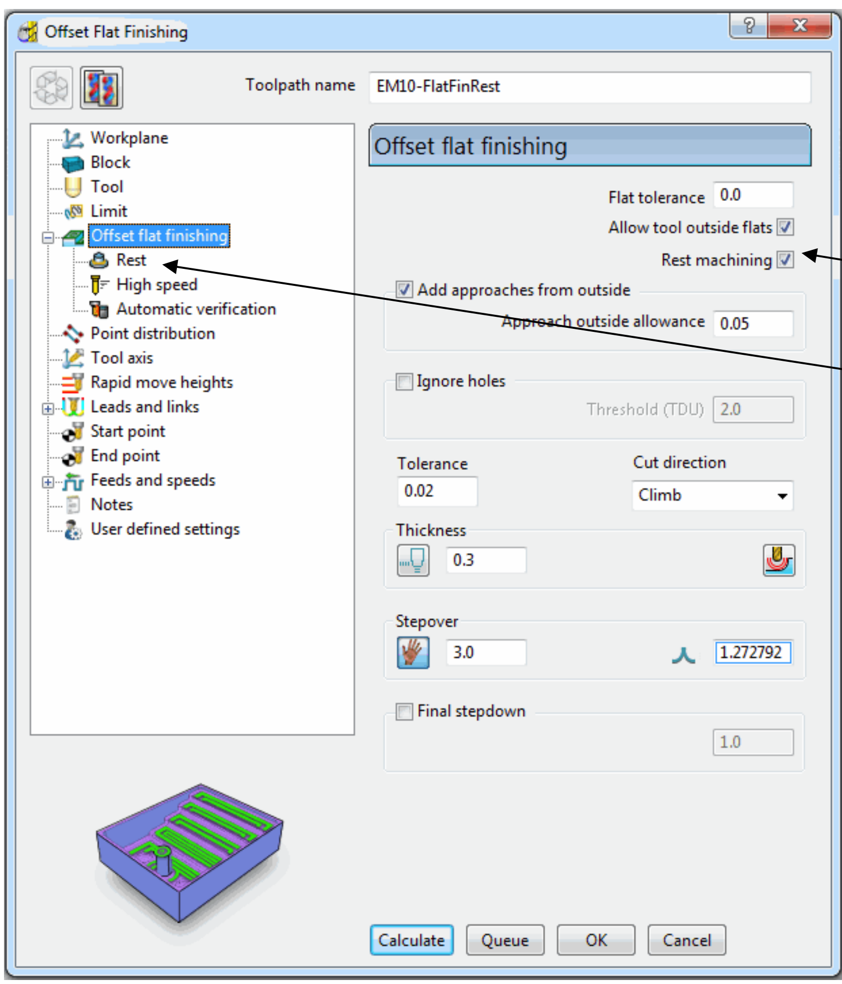
- ***Name*** the toolpath EM20-**FlatFin**.
- Select the **Use Axial Thickness** icon.
- Enter separate **Thickness** values for **Radial 0.5** and **Axial 0**.

- **Calculate** the **toolpath** and **Cancel** the **Offset Flat Finishing Form**.



The **Flat** areas have been finish machined apart from between the 'D' shaped and oval **Bosses** as well as at the base of the **holes** due to the 20mm diameter tool being too large. We will now use a smaller tool to **Rest** machine these areas.

- Create a **10mm** diameter **End Mill** tool named **EM10**.
- **Active** the toolpath **Flat Fin EM20** and select Settings.
- Select make a **Copy**  and fill in the form exactly as shown below.

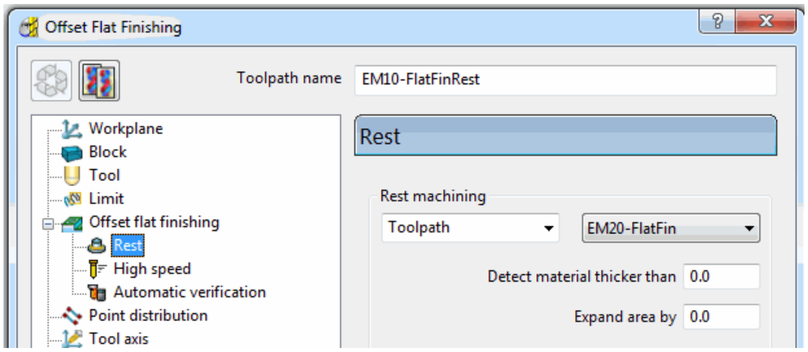


Toolpath Name:-
EM10-FlatFinRest

Tick **Rest machining** (The **Rest option** will appear in the **explorer**).

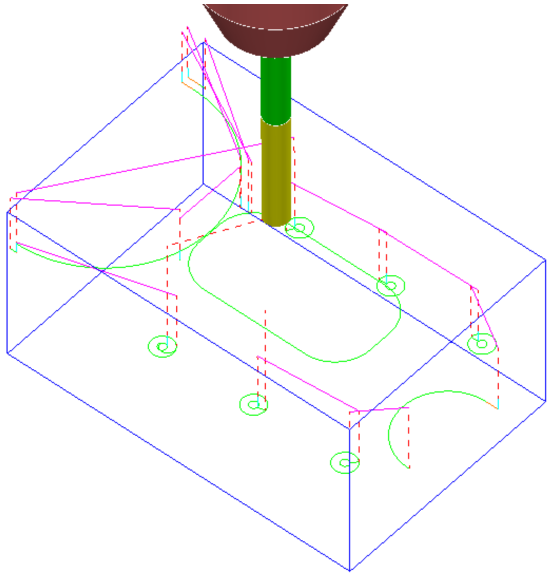
Stepover 3.0

- From the local ***explorer*** select the **Rest** option.



- Input the reference **Toolpath** as **EM20-FlatFin**

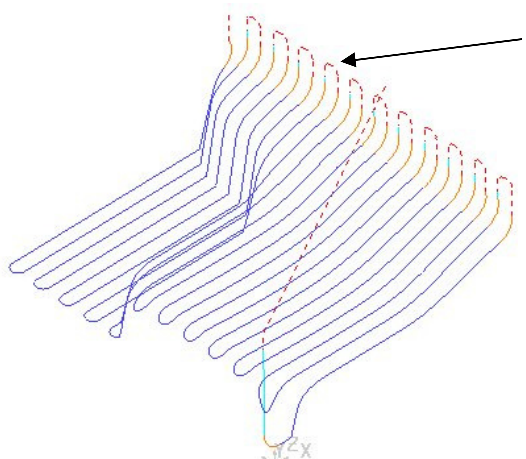
- **Calculate** the **toolpath** and **Cancel** the **Offset Flat Finishing** form.



The bottoms of the holes and the area behind the boss have now been finished as shown left.

Leads and Links - Arc Fit Rapid Moves

PowerMILL Pro contains a couple of additional options in the **Leads and Link**s form. These include **On Surface** link moves and the ability to **Arc Fit** rapid moves (provided the machine tool controller will support it).



If the **Short** links are set to **Skim** and **Arc Fit Rapid Moves** is set, then arcs are added to the rapid moves.

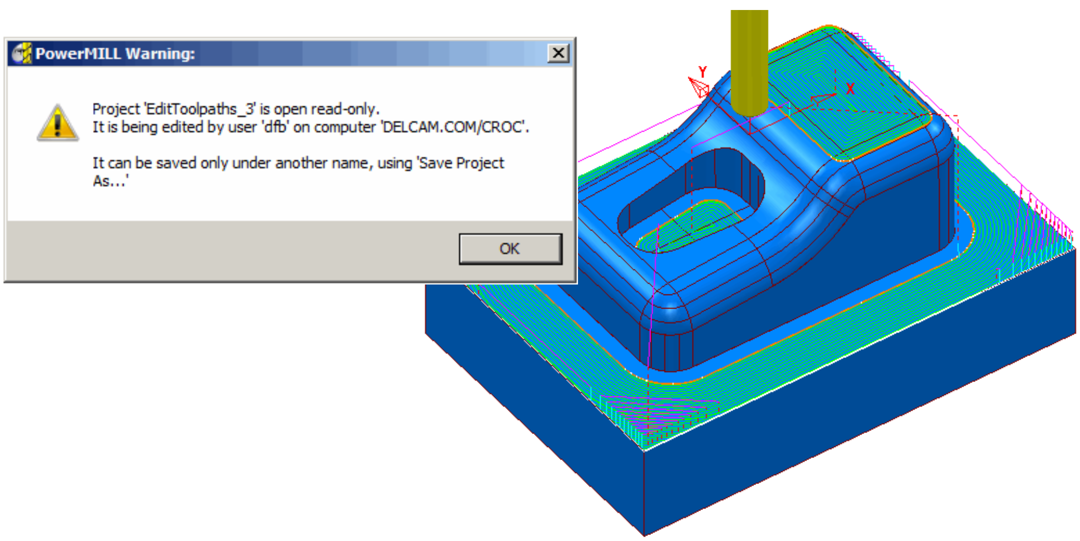
This option is ideal for applications where it is desirable to avoid sudden sharp changes in direction; for example (High Speed Machining).

Changing the Order and Direction within Toolpaths

For toolpaths containing internal link moves, the **order** and **direction** of the tool tracks can be changed. For example, if a machining sequence starts at the bottom of part, progressing upwards, reversing the order will change the tool track sequence to start at the top of the part and progress downwards. In this case the direction in which the tool travels is unchanged. It is also possible to apply **Reorder** and/or **Reverse** to selected tool tracks within a toolpath. Typical reasons for applying **Reorder** and **Reverse** to toolpaths include minimising fresh air, tool movements, or to comply with the recommended tooling specifications (it is often a requirement in **High Speed** applications for the toolpath to both climb mill and track upwards).
During applications where the base of a deep slot is to be climb milled, a uni-directional **Raster Finishing Strategy** will track across parallel, starting flush with one sidewall and tracking towards the other. By locally editing the **order** and **direction** a more desirable strategy can be created where tracking starts along the centre of the slot and progresses, climb milling, separately outwards towards both sidewalls.

Changing the Order/Direction of tooltracks

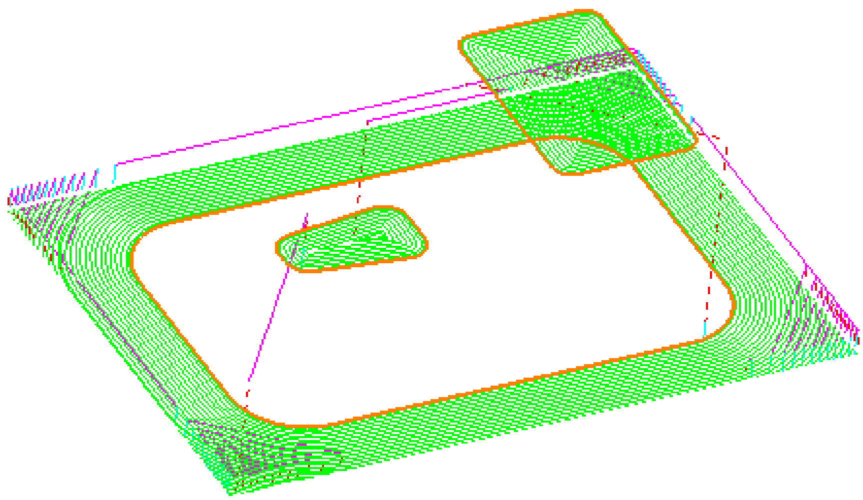
- **Delete All** and **Reset forms**.
- **Open** the read-only **Project**:-
  C:\users\training\PowerMILL_Data\Projects\EditToolpaths_3



- Click **OK** in the ***PowerMILL Warning*** form and **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\PunchForm_3

The **Project** contains a **Dia 10 Rad 1 Tipped Tool**, and a **3D Offset** finishing toolpath controlled by both **Boundary** and **Pattern** segments.

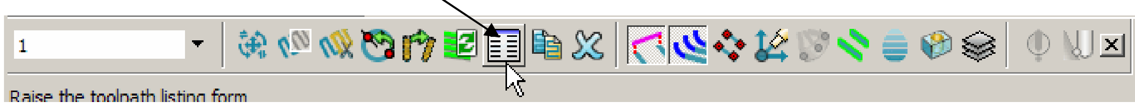
- **Simulate** the toolpath using the **Slow** setting.



Observe the current **direction** and **ordering** of the strategy considering potential improvements. For example, the lower area machining would benefit from climb milling inwards towards the component form and the pocket machining could start central and climb mill outwards towards the sidewall.

- Either right click over the toolpath and from the local menu select **Edit** - **Reorder** or select **Toolpaths** - **Toolbar** to activate the following **Toolbar**.

- Click on the **Reorder Toolpath** icon on the Toolpath Toolbar. 

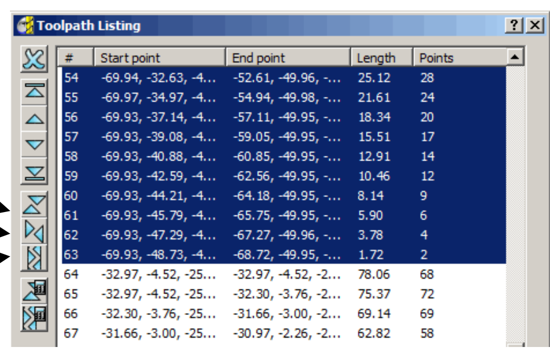


Note the selected segments are shown highlighted in the form.

Reverse order

Reverse direction

Alternate direction



Each segment is listed in order of execution. If a toolpath segment is selected on the list the corresponding segment is highlighted in the graphics area (and vice versa). As a result it can be modified or moved to another position in the pecking order.
The icons to the left hand side of the form are used for changing the order and direction of selected segments. If nothing is selected, the buttons when pressed will alter the whole toolpath. The 2 icons to the lower left of the form are **Automatic Reorder**  and **Automatic Reorder and Reverse** . These can only be applied to the whole toolpath and

not on selected tool tracks and are designed purely to minimise air moves (which in many cases could be produce a result which is detrimental to the order or direction of tool tracks).

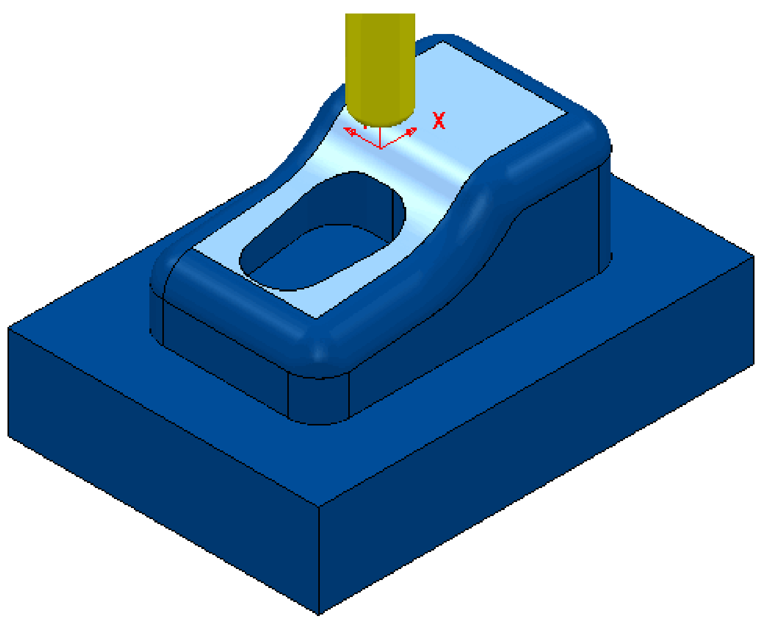
- Select all the tooltracks in the lower area and select **Reverse Order**. This part of the strategy should now climb mill inwards, towards the main component form (If not 'climb milling', then apply **Reverse Direction**).
- Select the tooltracks in the central pocket and again select **Reverse Order** followed by **Reverse Direction**. This part of the strategy should now climb mill from the centre of the pocket outwards.

Note:- Extra care is required when using the **Spiral** option in the selected finishing strategy as these are continuous tooltracks and can only be reversed and not internally reordered (If a spiral track is created to **Upcut** outwards then it can only be modified to **Upcut** inwards).

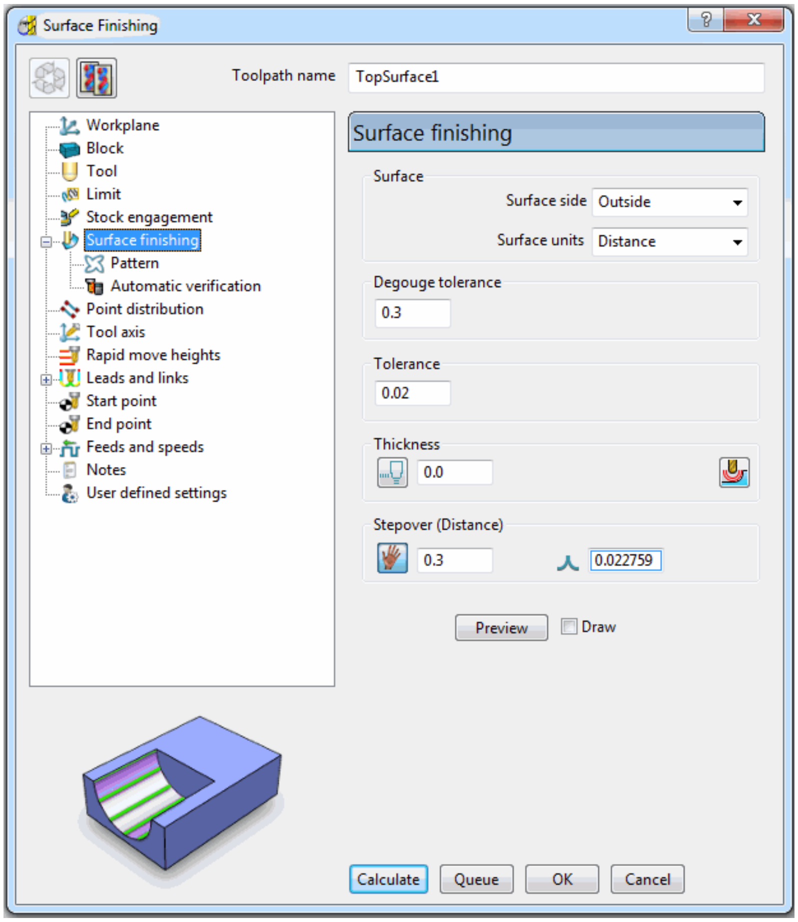
Surface Finishing

Surface Finishing creates a strategy that follows the **surface curves** on a selected **Surface**. Note:- The **strategy** will not run parallel to any ***trimmed, surface edges***.

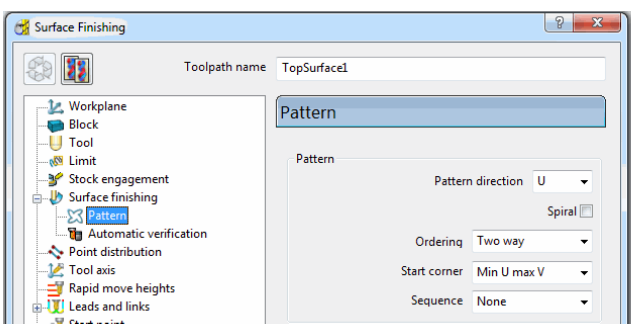
- **Activate** the **Dia 10**, **Rad 1**, **Tiprad tool** named **D10t1**.
- Select the ***upper surface*** on the punch form.



- Open the **Surface Finishing** form and fill in as shown below.

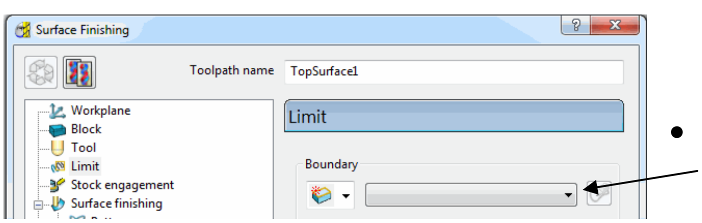


- From the local ***explorer*** select the **Pattern** option.



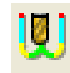
- Set **Pattern direction** - **U**
- Select **Two way**.
- Select:-
  Min U max V

- From the local ***explorer*** select the **Limit** option.

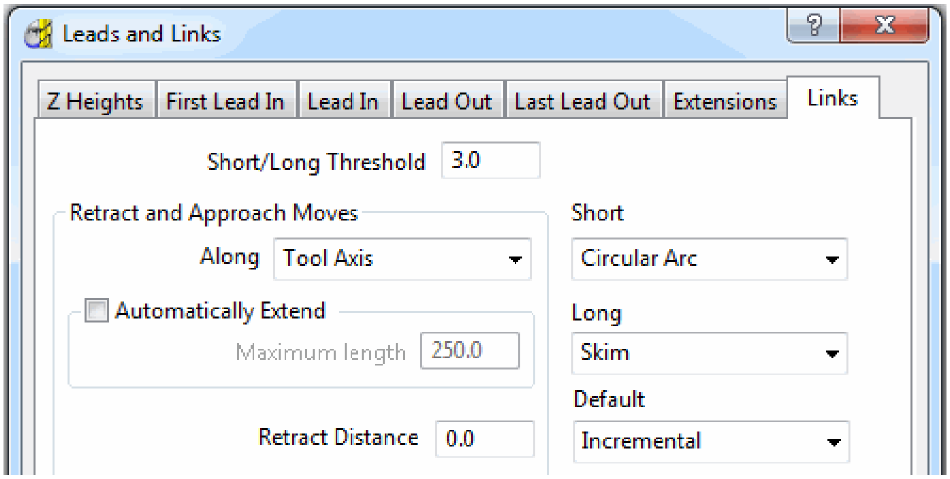


- Make sure that no **Boundary** is ***Active*** (Empty box).

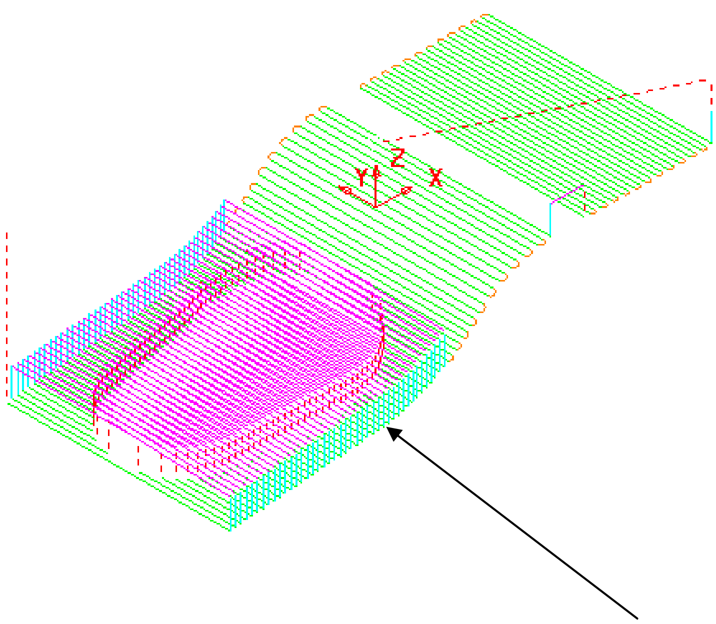
- Select **Calculate** to process the toolpath and **Close** the form.

- From the **Main** toolbar, Select the **Leads and Links** option. 
- In the **Links** page input:-

Short/Long - **Threshold 3** ***Short*** - **Circular Arc** ***Long* - Skim**
***Default* - Incremental**

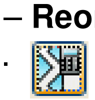


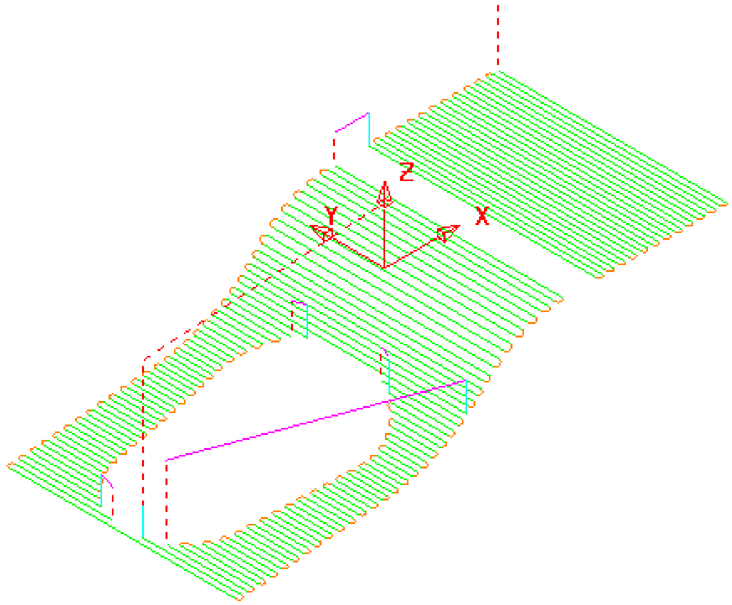
- **Apply** the above form.



Where the **Link** move from the end of a ***tool track*** to the start of the next is within 3mm, an (orange) arc move will be implemented.

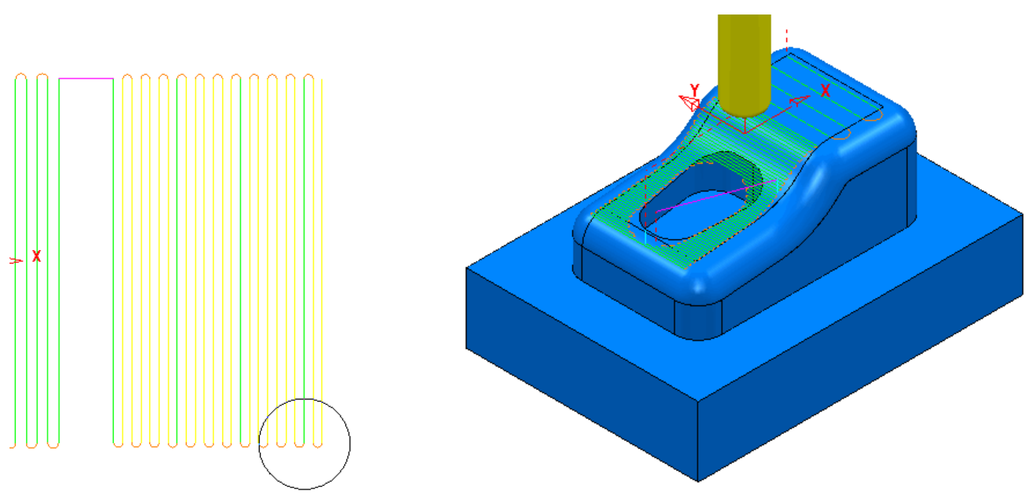
In the resultant **toolpath** the natural, order of the ***tool tracks*** and **link** moves across the 2 narrow areas is not very efficient.
This will be fixed by applying suitable toolpath **Edit - Reorder** options.

- From the local **toolpath menu** (Right mouse click), select **Edit** – **Reorder** and in the form, click the **Automatic reorder and reverse** icon. 



The **Toolpath** follows the curves of the selected **Surface**.
The toolpath illustrated has been created with **1mm Stepover** to help with visualisation.
It has also been edited using the **Automatic Reorder and Reverse** option to eliminate as many air moves as possible.
To further improve the efficiency of this strategy, several of the tool tracks on the upper flat area can be ***deleted*** to take advantage of using a bigger **Stepover** for the Dia 10mm flat area of the tool used.

- From the ***main*** pull down menus, select **Draw - Cursor - Tool** to help with the visualisation for the next action.
- Select groups of ***6 tool tracks*** leaving the **single tool track** outside each set (as shown below left).



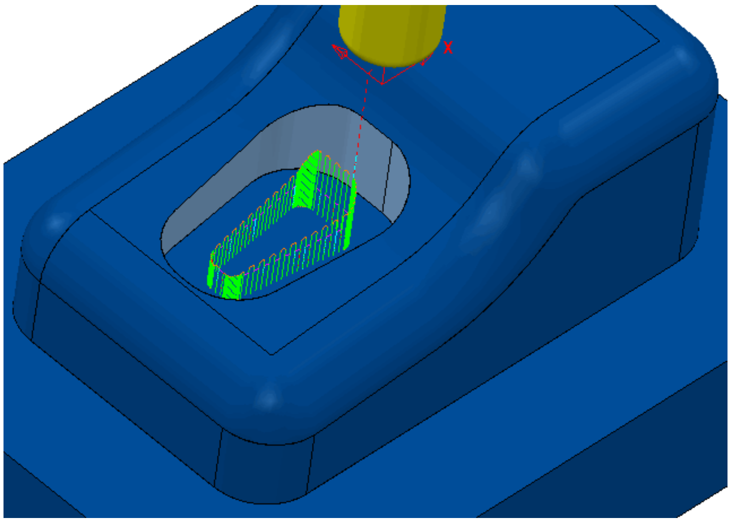
Tool outline displayed to visualise suitable **Stepover** (**Draw - Cursor - Tool**).

- From the local **toolpath menu** (Right mouse click), select **Edit - Delete Selected Components** to remove the selected ***tool tracks***.

Surface Finishing Exercise

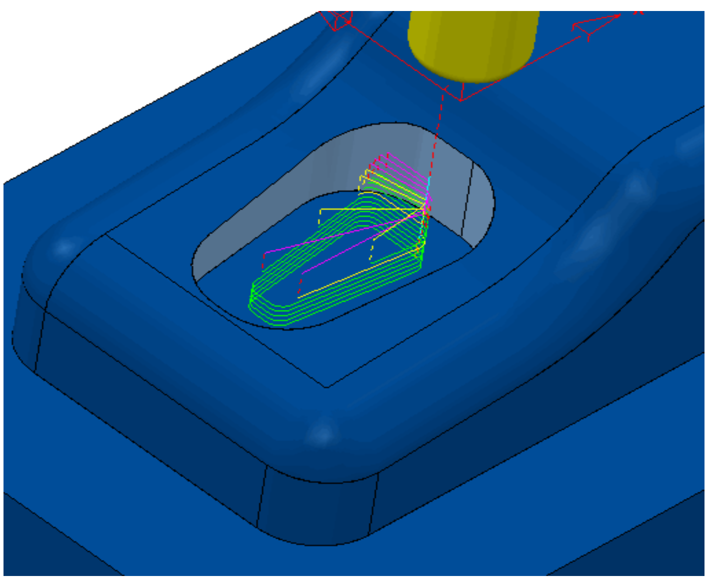
The direction of the curves on the ***selected*** **surface** will dictate whether the tool tracks are aligned **along** or **across** the model form.

- **Activate** the **D10t1** ***tip radiused*** tool.
- Create a **Surface Finishing** strategy with the pocket ***sidewall*** **surface** selected using a **1mm Stepover** (as shown below).



Due to the natural direction based on ***along or across*** the surface, the **tool tracks** run up and down the sidewalls.
This is not the correct choice as it is required to step the **tool tracks** down the sidewalls.

- ***Recycle*** the above **Surface Finishing** strategy and change the **Pattern Direction** option from **U** to **V** and **Calculate** the form.



The **Toolpath** now follows a direction, parallel to the base of the selected **Surface**.

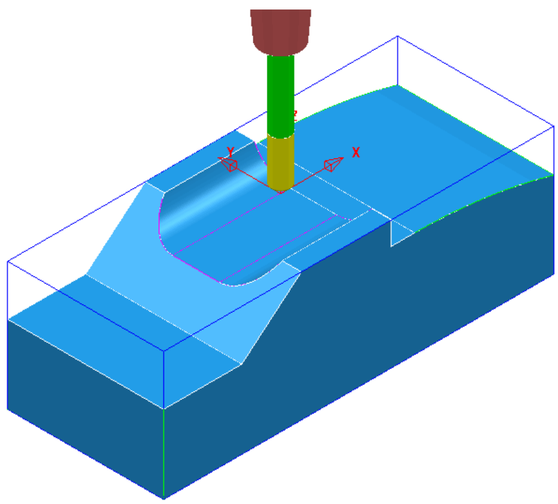
Note that the **tool tracks** do not run parallel to the ***trimmed*** top edge of the sidewall **surface**.

Note:- A **spiral** option in the form of a ***tick box*** is available in the **Pattern** page of the **Surface Finishing** form**.** This can be applied for a continuous **tool track** down closed **pocket** or **upstand** areas.

Toolpath Processing Queue

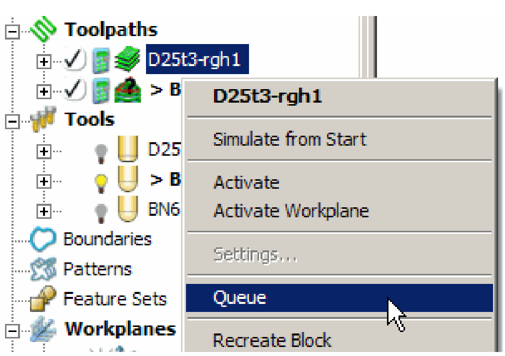
The **Toolpath** - **Queue** option allows the user to continue working in **PowerMILL** while the **Toolpath** is being processed in the background. (One or more **Strategies** can be added to the **Queue** and they are calculated in the background, in sequential order).
The following 'start up' **Project** contains two unprocessed **Toolpaths**. The form for each one will be opened in turn, but **Queue** will be used instead of **Calculate**. This **will** enable the user to continue work on a new **Corner Along** strategy while the original **toolpaths** are processing in the background.

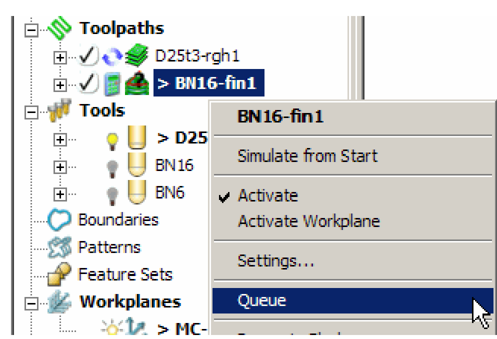
- **Delete All** and **Reset forms**.
- **Open** the read-only **Project**:-
  C:\users\training\PowerMILL_Data\Projects\ToolpathProcessing-Start



- From **File** select **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\ToolQueue-ex1

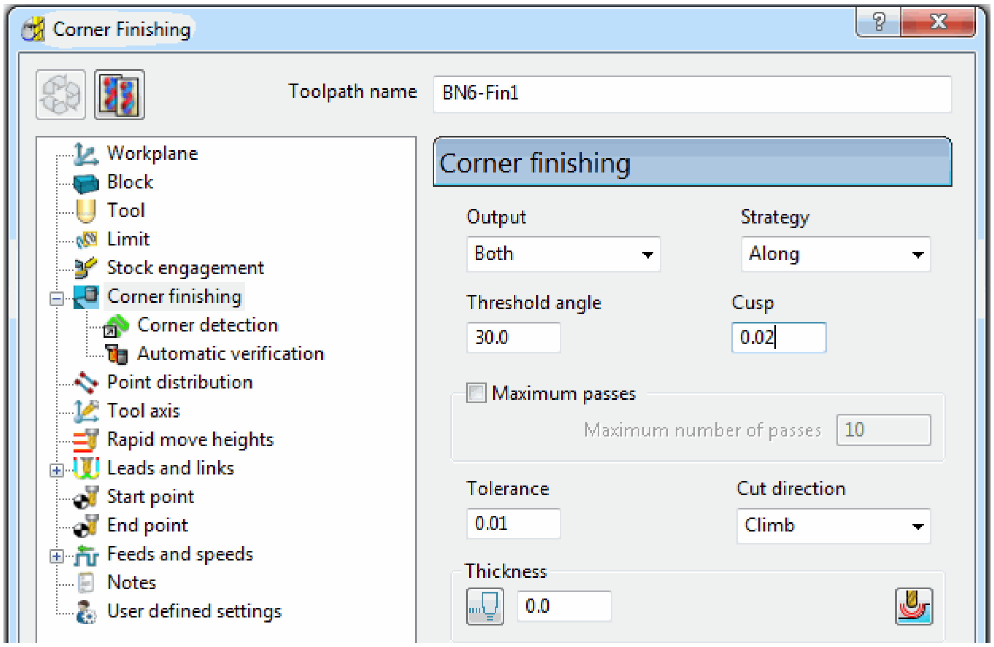
- In the ***PowerMILL*** **explorer**, right mouse click on the **Toolpath** named **D25t3-rgh** and select the option **Queue** from the local menu.
- Repeat this on the **Toolpath** named **BN16-fin1**.



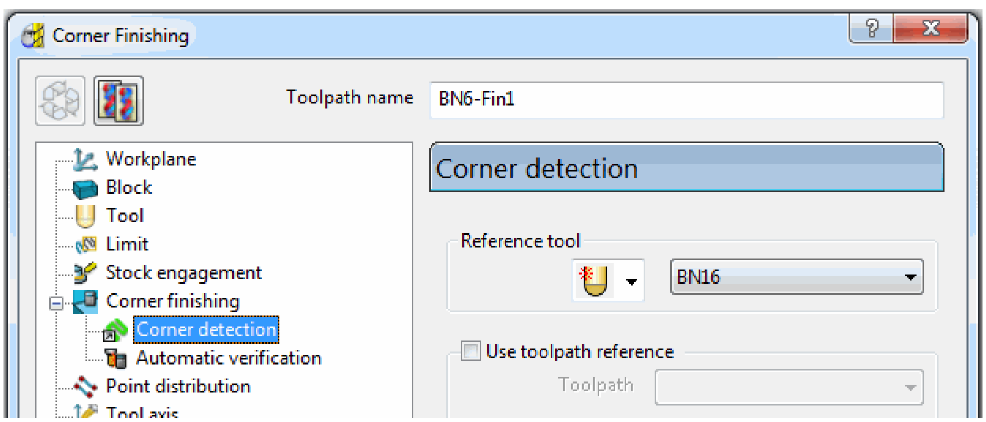


- While the original **Toolpaths** are being processed in the background, **Activate** the **Tool BN6**.

- From the **Finishing** options. select the **Corner Finishing** using ***Strategy* - Along** and enter the other values exactly as 'shown below'.

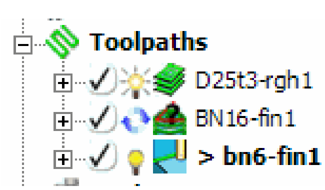


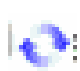
- In the local **explorer** select the **Corner detection** page and input the **Reference tool** as **BN16**.

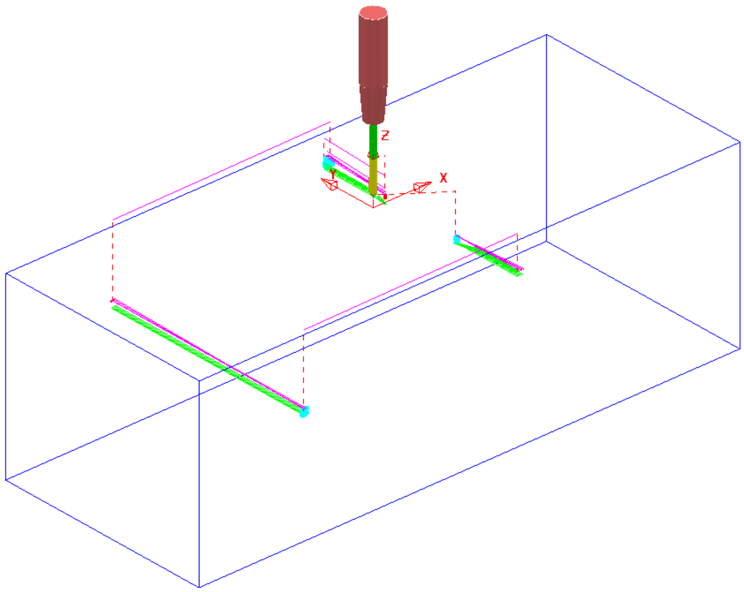


- Select **Calculate** to process the toolpath and then **Close** the form.

Note: The latest **Corner Finishing** strategy will start processing immediately and will more than likely, be completed before the **Interleaved Finishing** strategy currently pending in the **Queue**.



When a **Toolpath** has been added to the **Queue** the unprocessed status, ***Calculator symbol*** in the **explorer** changes to the ***Queue symbol***. 



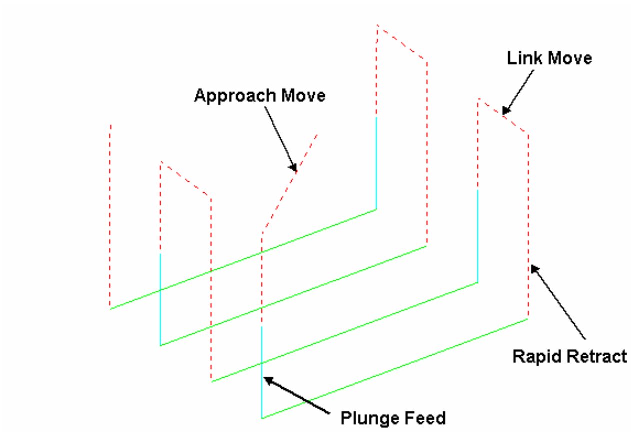
Bn6-fin1

- From **File** select **Save Project** to update the stored **Project**.

5. Toolpath Leads and Links

Introduction

If a tool is allowed to start machining directly on the end of a tool track, it first ploughs though the remaining stock depth before suddenly changing direction to run along the tool track. This is likely to result in machining marks, vibration, and excess wear both on the tool and machinery. To avoid sudden loading on the tool, appropriate **Lead** moves (at cutting federate) on and off tool tracks can be applied.
Fresh air (**Link)** moves between individual tool tracks can add a significant amount of extra time to a machining operation. This can be greatly reduced by applying alternative, **Link** move options.



< Default Leads and Links.

The **Leads and Links** options  are selected either from the **Main** toolbar or from within the **Finishing** strategies forms. They can be applied retrospectively to an existing toolpath.

Z Heights

Skim and **Plunge** distance provide variable control of rapid move heights within a component. These operate in conjunction with **Safe Z** and **Start Z** to minimise slow and unnecessary movement of the tool in fresh air while machining the component form.

Skim distance – An incremental distance above the model at which rapid moves occur from the end of one tool track to the start of the next. The tool traverses at a high feed rate (G1) across the model clearing the highest point along its route by the **Skim** value.

Plunge distance – An incremental distance above the local component surface where a downward rapid movement of a tool changes to plunge rate.

Lead In/Lead out Moves

Lead In controls the tool movement onto the start of a tool track and **Lead Out** the movement away from the end of a tool track. **Lead In** moves available include **None**, **Vertical Arc**, **Horizontal Arc**, **Horizontal Arc Left**, **Horizontal Arc Right**, **Extended Move**, **Boxed** and **Ramp**. The same options exist for **Lead Out** moves apart from the omission of **Ramp**.

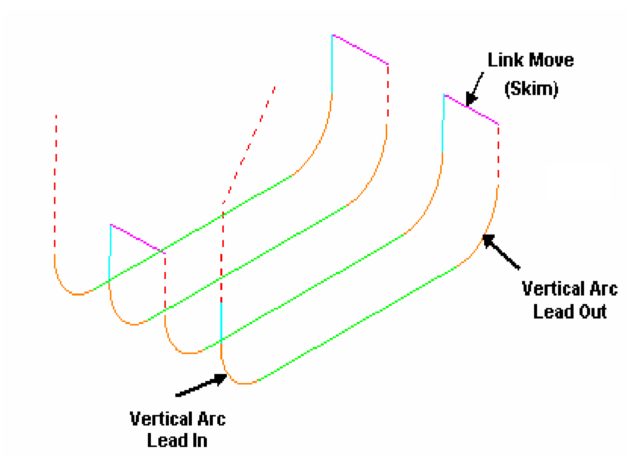


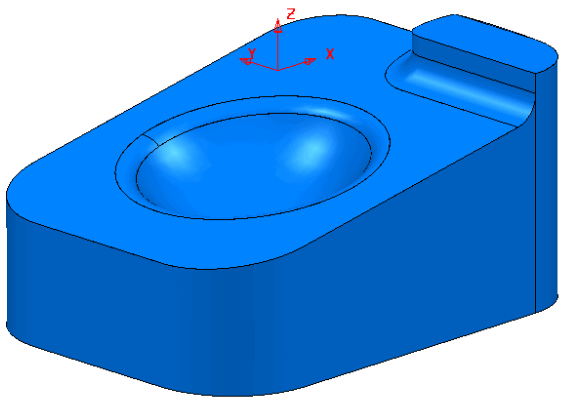
Illustration shows:-
Lead In/Out - **Vertical Arc** and
Incremental - **Skim** - ***Link*** moves.

Toolpath colour codes:-
Purple - Rapid Skim Feed Rate G1
Pale Blue - Plunging Feed Rate G1
Green/Orange - Cutting Feed Rate G1
Dotted Red - Full Rapid G0

Leads and **Links** are effectively extensions to tool tracks and as result must be gouge protected. To prevent gouging, the **Gouge Check** flag should be ticked (default) on the **Toolpath Leads and Links Form**. Any **Lead** that would result in a gouge will not be created. The following examples illustrate different leads and links, and unless otherwise stated, **Gouge Check** should always be set.
If any instances occur where the **1st Choice** cannot be applied then the software will apply the **2nd Choice**. If neither option is valid under gouge check conditions then the **Lead** will be locally applied as **None**.
The Current settings of **Leads and Links** are included in the creation of new machining strategies. Alternatively they can be applied later to the **Active** toolpath in the explorer.

Example

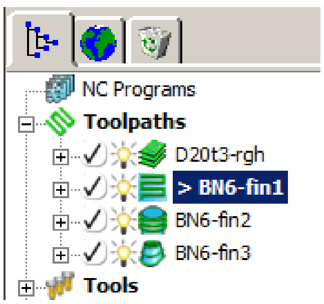
- **Delete All** and **Reset forms**.
- From **File** select **Open Project** and **select** the readonly ***Project:-***
  C:\users\train\PowerMILL_Data\Projects\LeadsLinks-Start



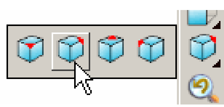
A message informing the user that the **Project** is **ReadOnly** will appear.

To continue, a copy of the **Project** will be created to allow the user to make changes (**Save As**).

- From **File** select **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\LeadsLinks-example



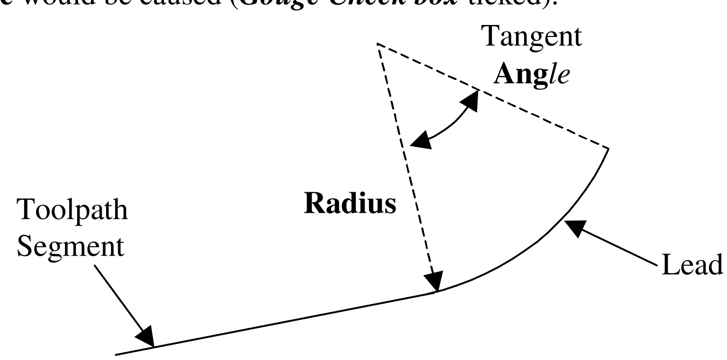
The **Project** contains 1 **Roughing toolpath** and 3 **Finishing toolpaths**. The following example will illustrate the retrospective addition of suitable **Leads and Links** to the 3 finishing toolpaths.

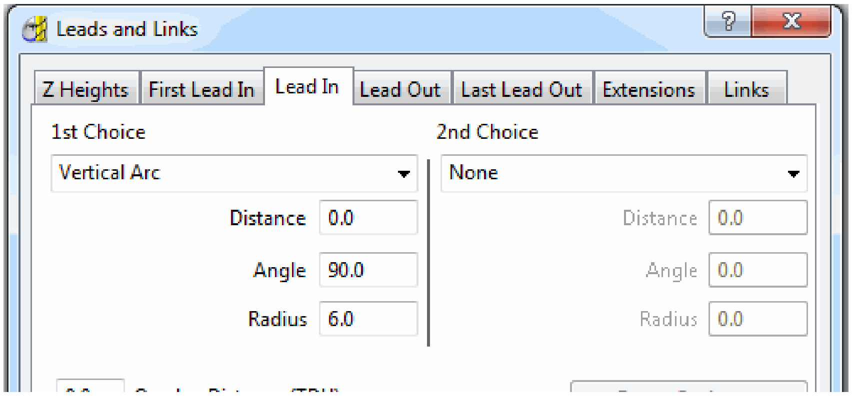


- Select an **ISO2** view.
- Right click on the ***Toolpath*** **BN6-fin1** and select **Activate**.
- Open the **Leads and Links** form. 

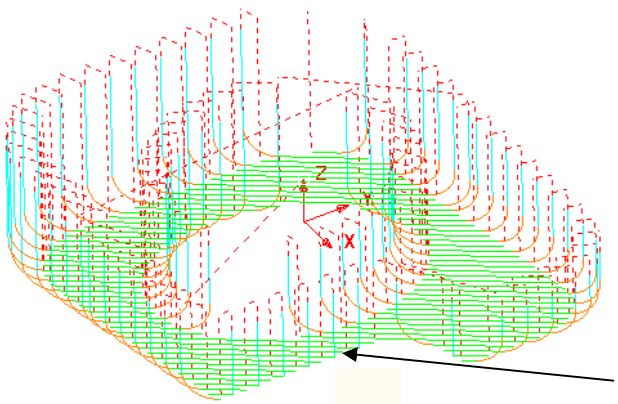
Vertical Arc Leads

Vertical Arc Leads create a circular movement downward at the start and upward at the end of each individual tool track. By default, the additional moves will not occur in instances where a **Gouge** would be caused (***Gouge Check box*** ticked).



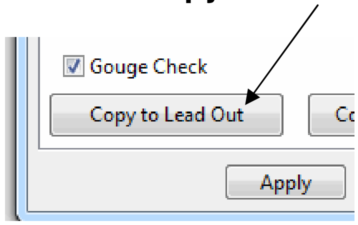


- Select the **Lead In** tab and for ***1st Choice*** input an ***Angle*** **90.0** and ***Radius*** **6.0** before clicking **Apply** at the bottom of the form.

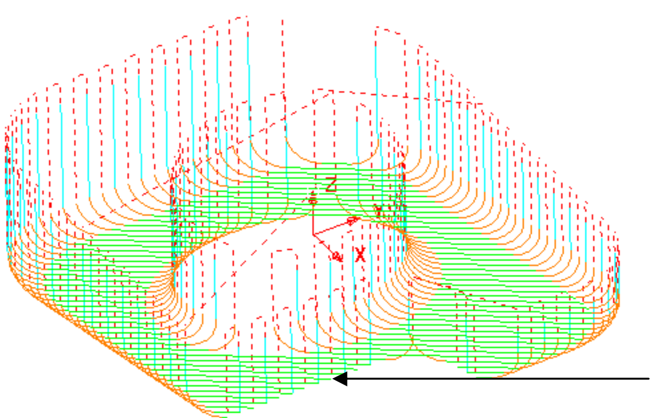


There is now a **Vertical Arc** leading onto most of the tool tracks. If the required **Lead Out** is to be exactly the same as the **Lead In** and vice versa the **Copy to Lead Out** option allows easy copying of the settings between the two.
The **Leads** in this area have remained unchanged as any **Vertical Lead In/Out** using the current **Radius** value would **gouge** the **model** if allowed to appear.

- Select the **Copy to Lead Out** button.

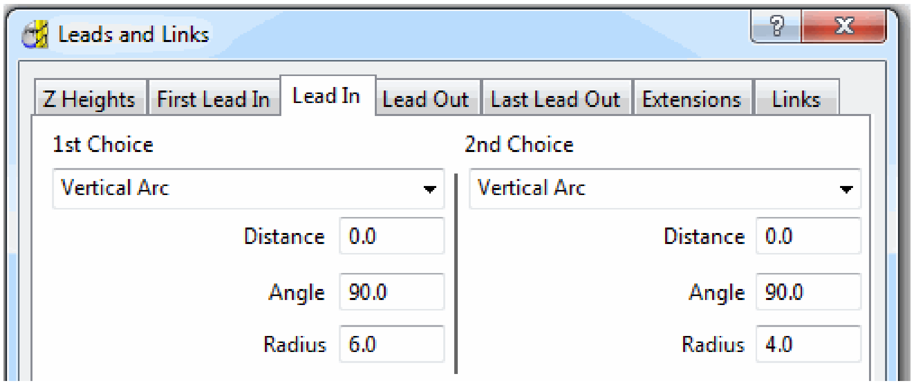


- **Apply** the form to update the **Active** toolpath.

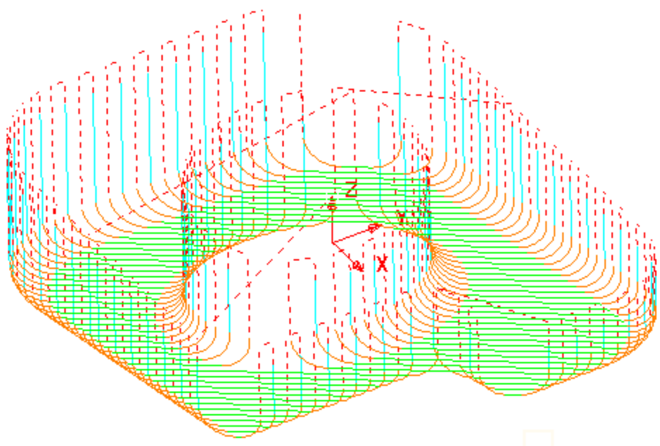


Where the operation is gouge free a **Lead Out** move, similar to the **Lead In** is applied at the end of each tool track.

The **Leads** in this area have remained unchanged as any **Vertical Lead In/Out** using the current **Radius** value would **gouge** the **model** if allowed to appear.



- Select the **Lead In** tab and for ***2nd Choice*** input **Vertical Arc** with an ***Angle*** **90.0** and a reduced value ***Radius*** **4.0**.
- As before, select the **Copy to Lead Out** button before clicking **Apply** at the bottom of the form.



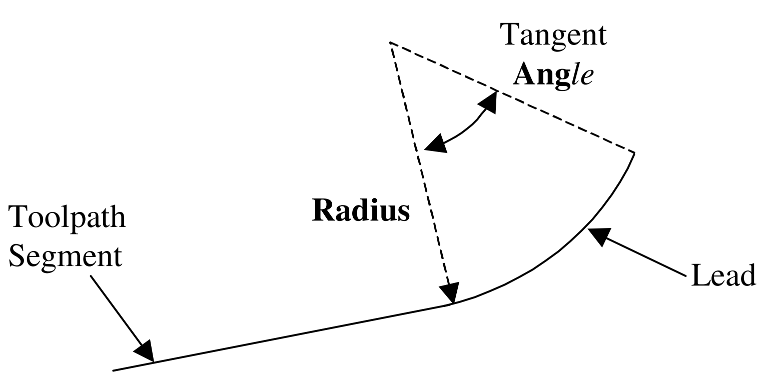
Wherever it is ***gouge free*** the ***1st Choice*** of a **Radius 6**, **Vertical Arc**, **Lead In/Out** is applied.

If this is not possible without a **gouge** occurring, then the ***2nd Choice*** of a **Radius 4**, **Vertical Arc** is attempted.

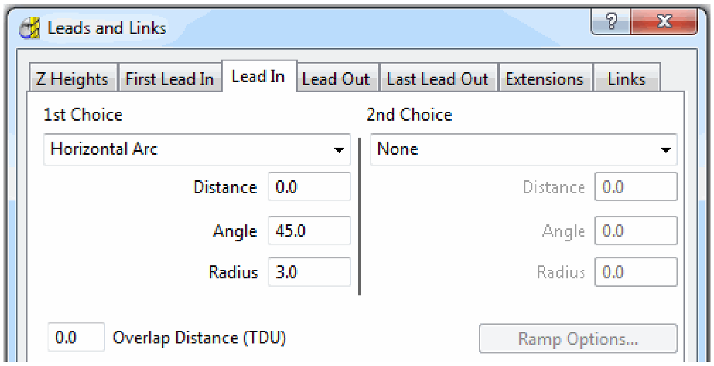
If neither the 1st or 2nd **Choice** is possible without a **gouge** occurring, then no **Lead In/Out** will be applied to that part of the toolpath.

Horizontal Arc Leads

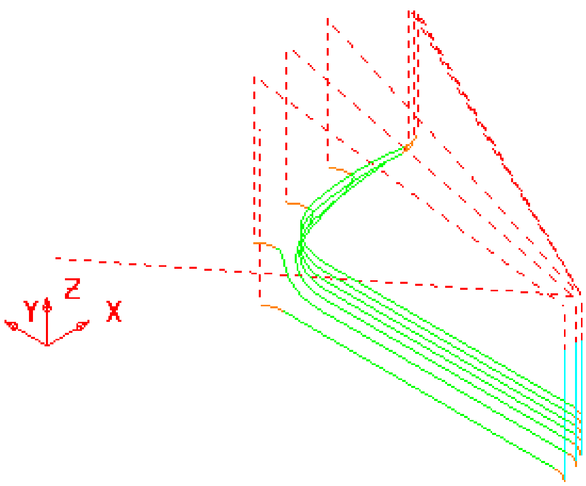
This produces circular leads on the horizontal plane, and the **Radius** and **Angle** value is similar to that for Vertical Arc. This type of lead is frequently used for toolpaths running at constant Z, or with only small changes in Z height.



- Right click on the ***Toolpath*** **BN6-fin2** and select **Activate**.
- Select an **ISO1** view.
- Open the **Leads and Links** form. 
- In the top ***Pull down*** menus select **Tools - Reset Forms** to return the **Leads and Links** settings to default.

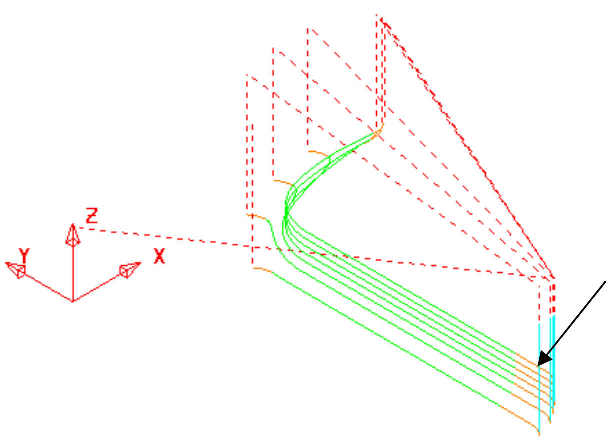


- Change the ***1st Choice*** **Lead In** to be a **Horizontal Arc** with ***Angle 45*** and ***Radius*** **3.0** then select the **Copy to Lead Out** button before selecting **Apply**.



There is no need to input a ***2nd choice*** as the **Horizontal Arc - Leads In/Out** have appeared on all the tool tracks. With **Horizontal Arc** set, **PowerMILL** decides whether to use **Left** or **Right** hand arcs. They are also calculated to comply with the **Gouge Check** flag. If it is not possible to apply the specified lead due to a gouge situation it will remain as the default **vertical** move unless the **Gouge Check** flag is unchecked (not recommended).

- In the **1st Choice** - **Lead In** input ***Distance*** **5** and **Apply** the form.

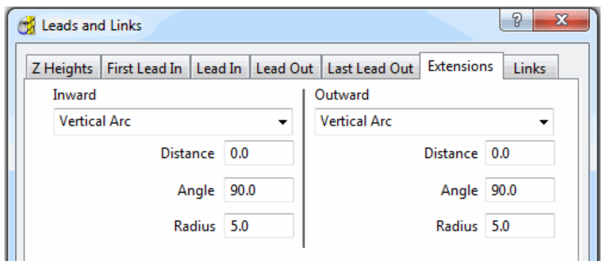


An additional (**Distance**) extension move is added to the ***tool track*** before the original **Lead In** option 'kicks in'.

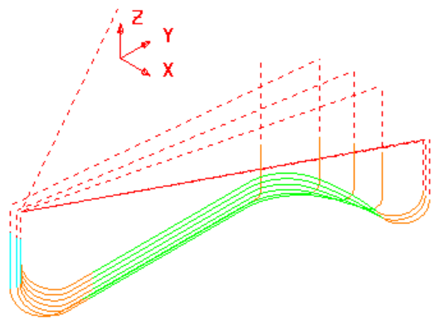
Extensions

Extensions provide the option for the user to add an additional ***lead move*** option onto the existing **Lead In/Out**. As an example, a **Vertical Arc** - **Extension** will be added to the **Lead In/Out** moves.

- Select an **ISO2** view.
- In the **Leads and Links** form, select the **Extensions** tab and for both the ***Inward*** *and* ***Outward*** options, input a **Vertical Arc** with an ***Angle*** **90.0** and ***Radius*** **5.0**.



- **Apply** the form.



A **Vertical Arc** ***Extension*** has successfully been added to all of the **Lead In/Out** moves.

Links

A **Link** is the movement from the end of a **tool track** to the start of the next. To provide for more efficient movement of the tool across the component the height of the **link moves** that connect adjacent **tool tracks** can be reduced to be closer to the local model form.

Short/Long Threshold - This defines the distance limit up to which the **Short** - **Links** apply. Any move from the end of a tool track to the start of the next, which exceeds this distance is defined as a **Long** - **Link** move.

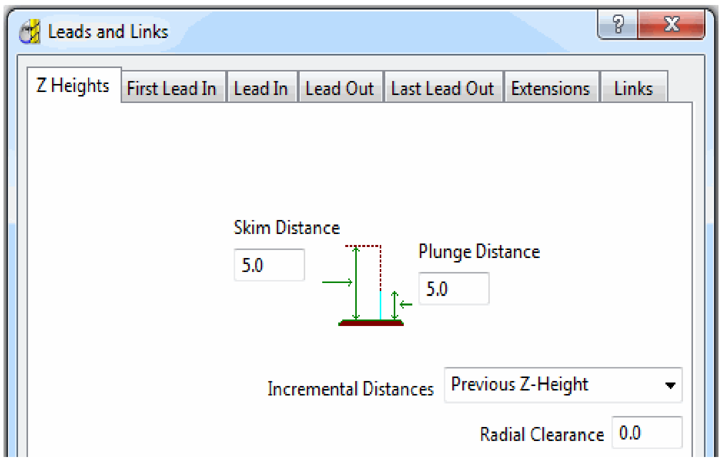
For **Short** links the available options include **Safe Z**, **Incremental**, **Skim**, **On Surface**, **Stepdown**, **Straight**, and **Circular Arc**.

For **Long** links the available options include **Safe Z**, **Incremental**, and **Skim**.

Safe links apply only at the start and end of a toolpath the available options being **Safe Z**, **Incremental**, and **Skim**.

For users with a **PowerMILL PRO** licence:-
Where a machine control system will permit arc fitting of rapid moves the **Arc Fit Rapid Moves** box can be ticked along with a suitable **Radius** based on the active tool diameter **TDU** (Of particular benefit to High Speed Machining).

- **Activate** the toolpath **bn6-fin1** created earlier in the **Leads** section.
- Open the **Leads and Links** form. 
- In the top ***Pull down*** menus select **Tools - Reset Forms** to return the **Leads and Links** settings to default.
- Select an **ISO 2** view.
- Select the **Z Heights** tab to access the **Skim** and **Plunge Distance** boxes.

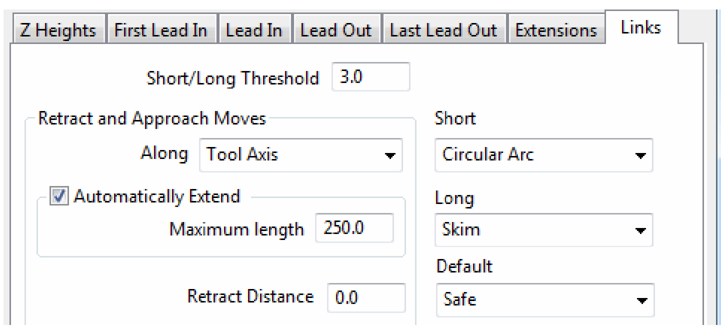


The values will be left as default for the time being.

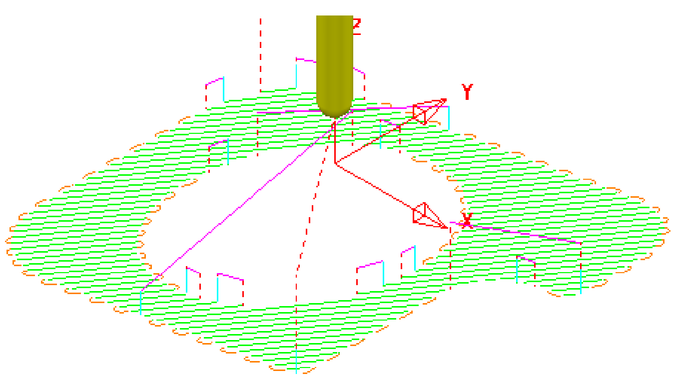
The **Skim Distance** is the ***Incremental height*** at which the **tool** clears the **model** form between tool tracks.

The **Plunge Distance** is the ***Incremental height*** at which the **tool** rapids down to before using the **Plunging Feed Rate**.

- Select the **Links** tab to access the **Link** move options.

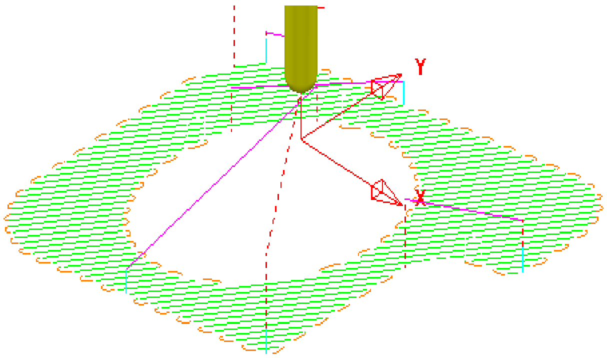


- Input the **Short/Long Threshold** as **3** (**Link** move distances ***less*** than this value will use the **Short** ***link*** option and those ***greater*** will use the **Long** ***link*** option).
- Select ***Short*** as **Circular** ad ***Long*** and Default as **Skim**.
- **Apply** the form.



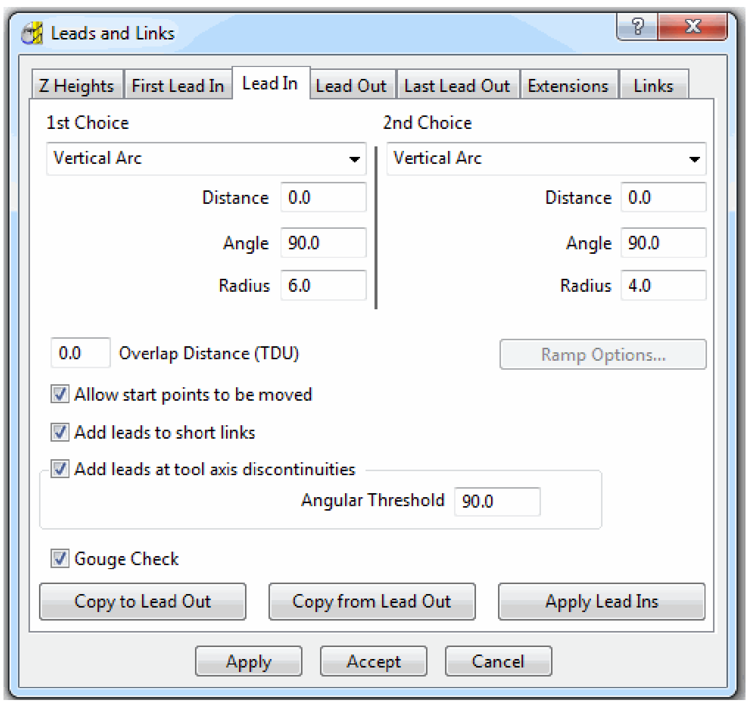
Note that the ***distance*** between the ends of adjacent ***tool tracks*** is greater than the **Short/Long Threshold** value in several areas of the toolpath (**Skim Link** moves apply).

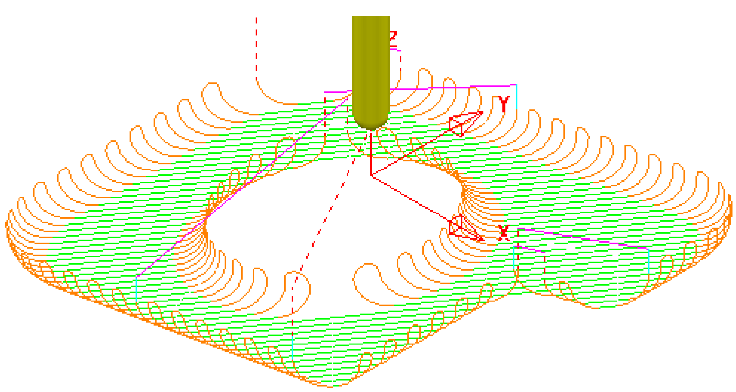
- Change the **Short/Long Threshold** to **5** and **Apply** the form again.



Note that by increasing the value of the **Short/Long Threshold** the number of ***Short Link*** **Cicular Arcs** has greatly increased.
The remaining **lift and plunge** moves would benefit from **Vertical Arc *Lead In/Out*** moves but <u>not</u> the existing **Circular Arc *Links***. This can be achieved locally by selecting the <u>individual</u> lift or plunge moves and then applying the **Leads and Links** form.

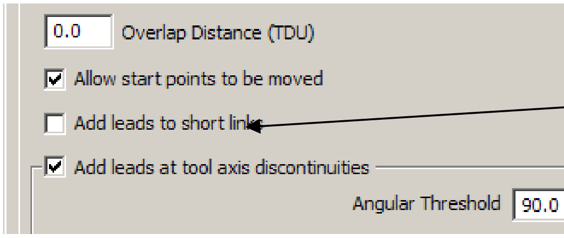
- **Apply** a **Vertical Arc** to both **Lead In** and **Lead Out** as shown below



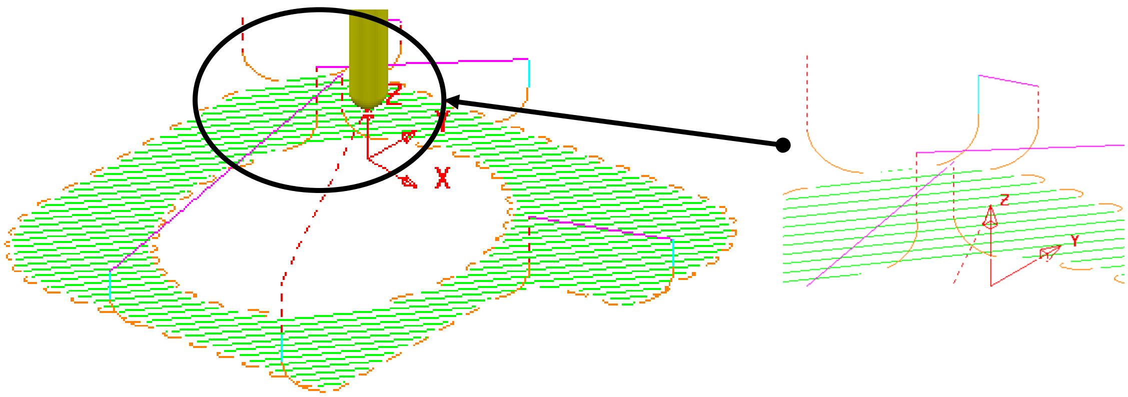


The **Vertical Arc** - ***Leads*** are currently adding to the **Circular Arc** - ***Short*** link moves.

This can be suppressed by **unticking** the **box** labelled ***Add leads to short links***.



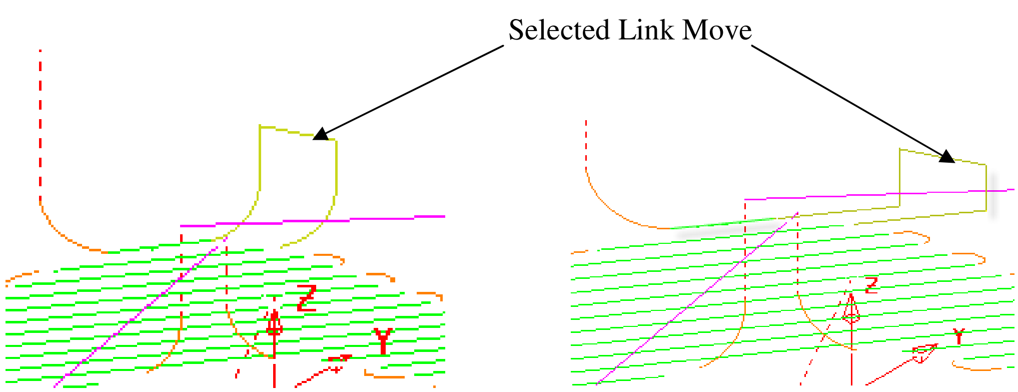
- **Untick** the **box** labelled ***Add leads to short links***.
- **Apply** the form.



As shown above the **Vertical Arc** - ***lead*** moves no longer exist on the **Short links** (***Circular Arcs***).

Applying Leads and Links locally to individual tool tracks

- With the **Shift key** depressed, select one of the individual ***link moves*** where the **Lead** is currently a **Vertical Arc**.
- In the **Leads and Links** form select the **Lead In** tab and select ***1st Choice*** as **Extended Move** with ***Distance*** **10**.
- Select the **Copy to Lead Out** tab and then **Apply** the form.



The selected **Vertical Arc** ***lead*** move is individually edited to become an **Extended** move As shown above.

- From **File** select **Save Project** to update the stored **Project**.

Additional Short Links options

In addition to **Safe**, **Incremental**, and **Skim** the following additional options are available in the **Short Links** section. These additions are not available in the **Long Links** options.

On Surface
This links the tool tracks with a gouge free, direct move that follows the surface form.

Stepdown
The link move remains at a constant height and performs a gouge checked, move over to the start of the next tool track where it then feeds down onto the surface.

Straight
In this case the link is a gouge checked, direct linear move to the start of the next tool track.

Circular Arc
This links the tool tracks with a gouge free, circular move that follows the surface form.

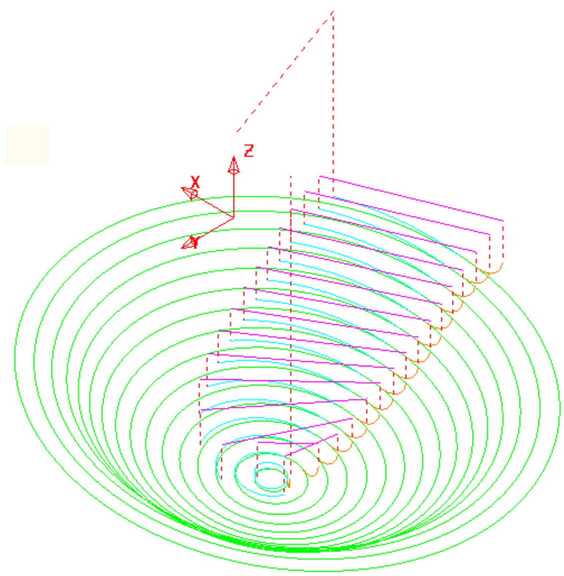
Exercise

- **Activate** the ***toolpath*** **BN6-fin3** and **Apply** the **Leads and Links** as follows:-

Links:-
Long/Short/Default - Skim

Lead In:-
1st Choice - Ramp (*Options* - *Max Zig Angle* 4 and *Ramp Height* 1)

Lead Out:-
1st Choice - Vertical Arc - *Angle* 90 - *Radius* 3 and *Overlap Distance* 0.2

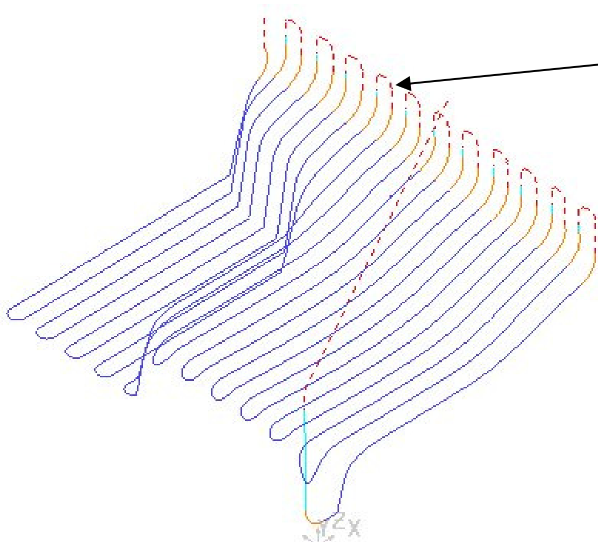


Overlap Distance continues the cutter before or after the natural end point (Along the original toolpath) by a specified distance based on the ***Tool Diameter*** (***TDU***).

The finished **Leads and Links** will be as shown above.

Pro – Arc Fitting Rapid Moves

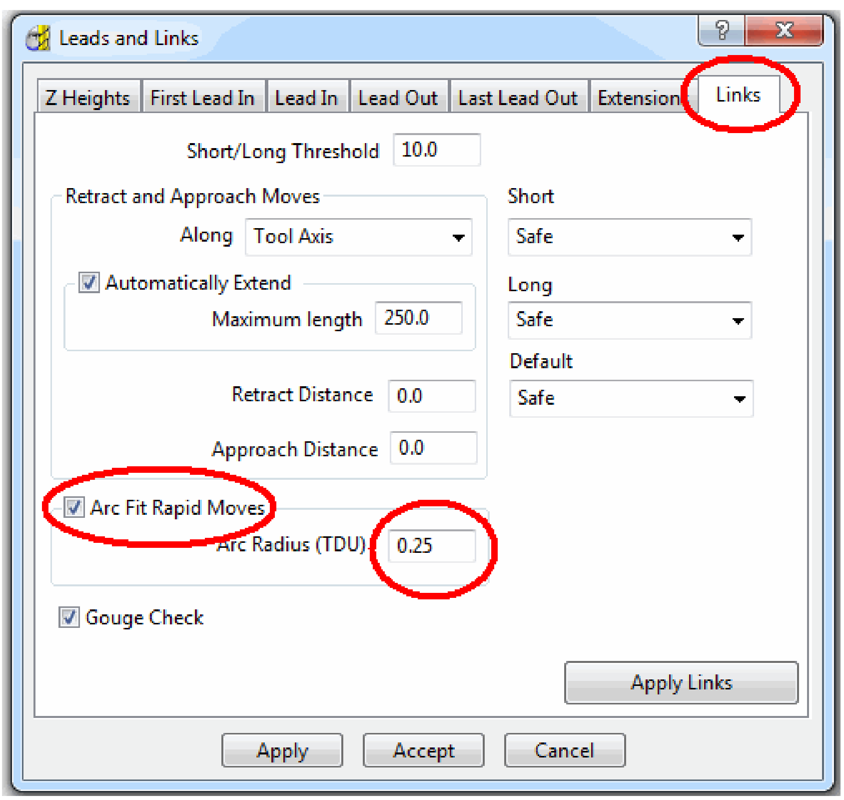
Leads and Links



Note that **arc**s can be applied to the **Rapid Moves** (If the individual Machine tool controller supports the option).

The **Arc Fitting** works on all **Rapid Moves** including **Skim** moves.
This option is ideal for applications where it is desirable to avoid sharp changes in direction; for example (High Speed Machining).

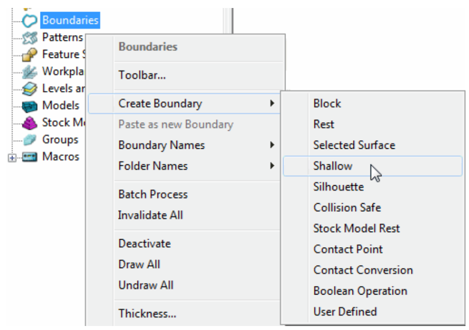
The **Arc Fit Rapid Moves** option is located in the **Links** form.
To activate the option the **Arc Fit Rapid Moves** box must display a ***tick***.
The **radius** of the **Arcs** is also controllable as a proportion of the active **Tool Diameter** (**TDU**).



6. Boundaries

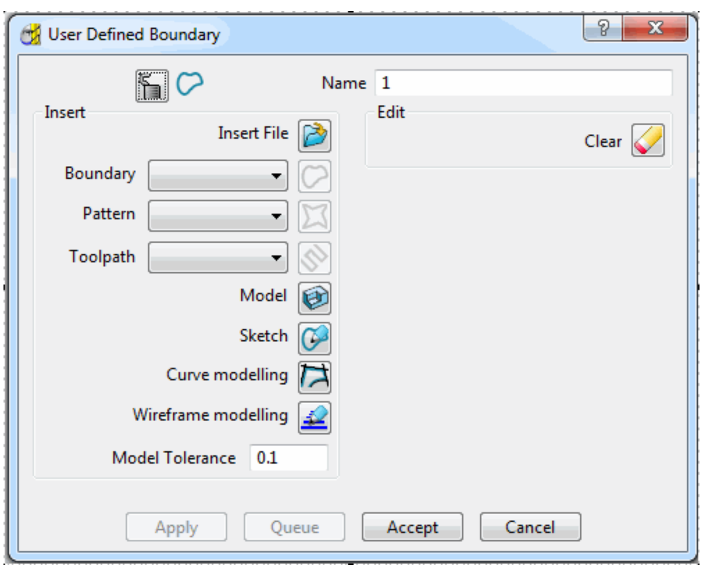
Introduction.

A **Boundary** consists of one or more closed (wireframe) segments, the main application being to limit machining strategies to specific areas of the component. Earlier in the course **Boundaries** have been utilised to limit machining strategies so that they occur in more compatible areas of the component. For example **Constant Z** finishing is more effective on **Steep** sidewalls and **Raster** finishing on **Shallow** areas.
There are several standard options available for **Boundary** creation.



User defined Boundary

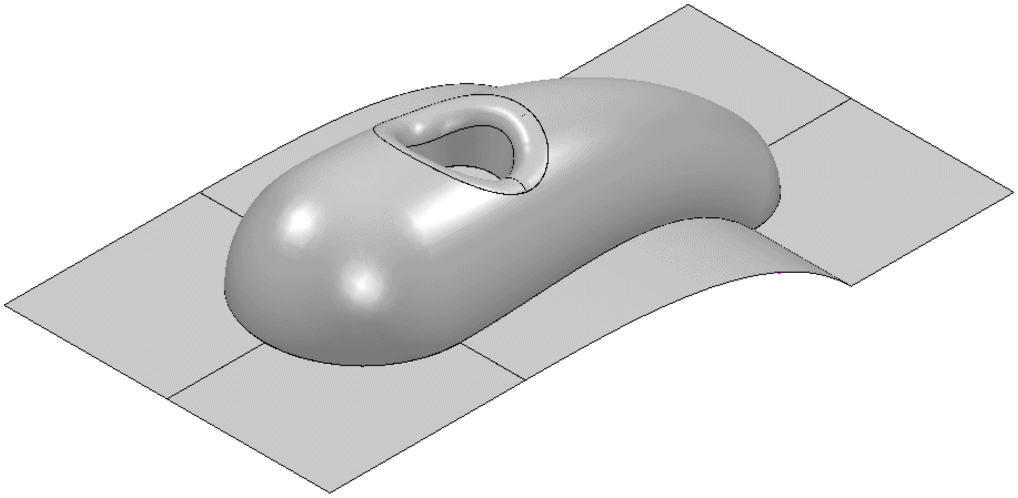
This type of **Boundary** is created from the options contained in an additional sub-menu. While all other main **Boundary** options involve interaction with other PowerMILL entities the **User Defined** options generally involve direct conversion of existing wireframes.



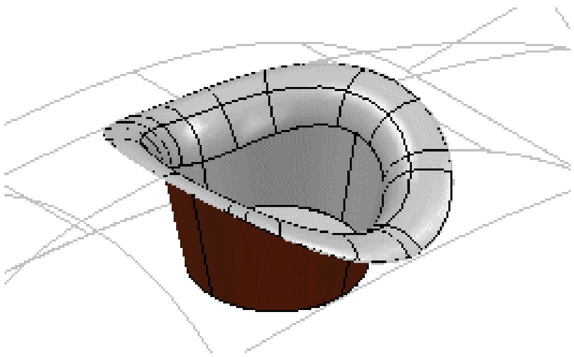
Boundary, **Pattern**, and **Toolpath** icons are greyed out unless a suitable entity is selected.

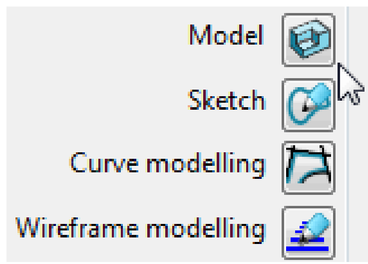
Model:- Inserts the edge of the selected **model**.
Sketch:- Enables free form coordinate input.
Curve Modelling:- Opens a **Composite Curve** generator.
Wireframe Modelling: Provides access to the wireframe part of **PowerSHAPE**.

- From the main **pulldown** menus select **File - Delete All**.
- From ***File*** **Import** the **model**:-
  C:\users\training\PowerMILL_Data\Models\cowling

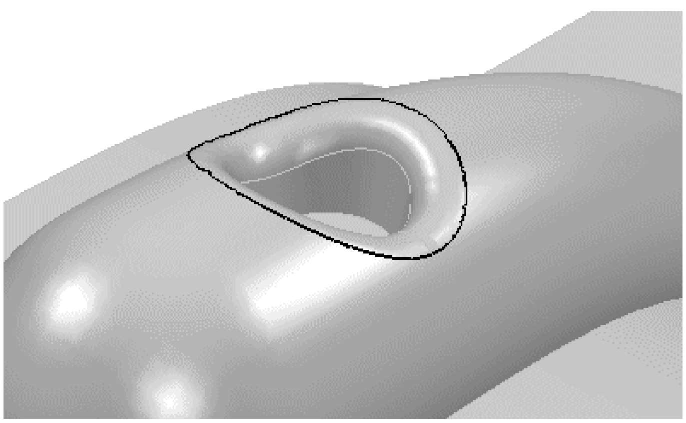


- Select the **surfaces** defining the central pocket and fillet.





- In the **explorer** right click over **Boundaries** and select **Create** - **User Defined** and in the **User Defined Boundary** form left click the **Model** icon (arrowed above).



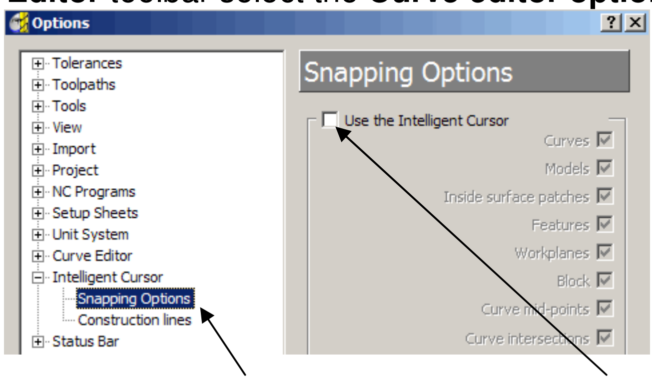
A **Boundary** segment is created around the edge of the selected part of the model.

- Select a **View from top (Z)** and switch off the **Shading** retaining the **Wireframe** view.

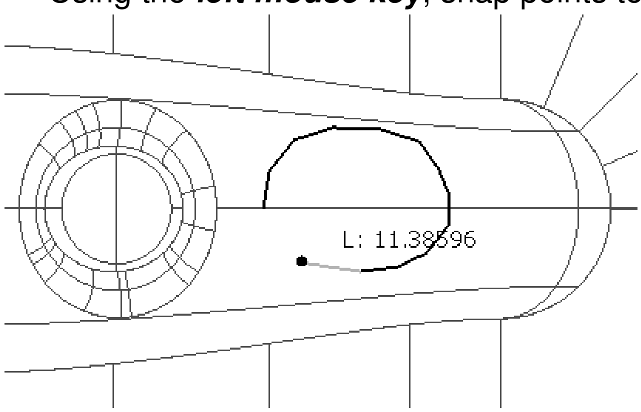
- In the **explorer** right click over **Boundaries** - **Create Boundary** - **User Defined** and in the **User Defined Boundary** form left click the **Sketch** icon. 

The **Curve Editor** toolbar (shown below) will appear above the Graphics area.

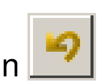
- In the **Curve Editor** toolbar select the **Curve editor options** icon. 



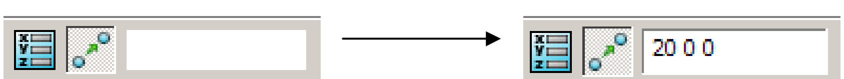
- Select **Intelligent Cursor** - **Snapping Options** and **untick** the **Use of Intelligent Cursor** box to enable full 'free form' sketching to occur.
- **Accept** the form.
- In the **Curve Editor** toolbar, select the create **Continuous Lines** option. 
- Using the ***left mouse key***, snap points to create a sketched **Boundary**.



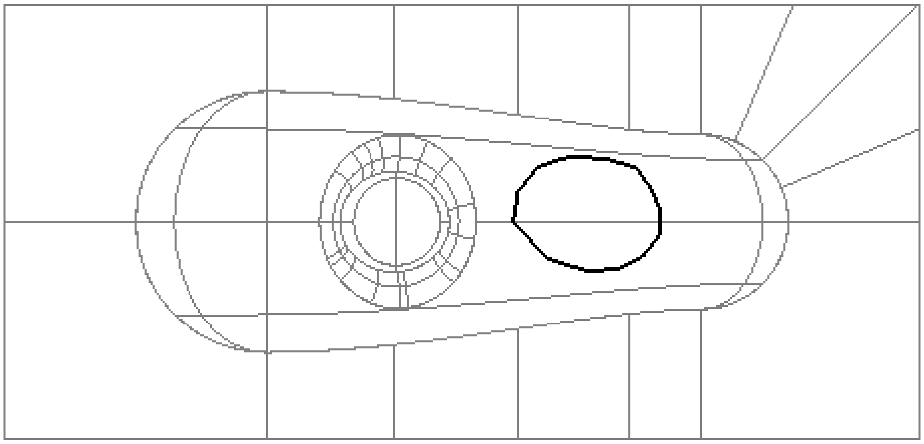
Note: <u>Do not</u> attempt to close the segment by trying to snap the 'final span' back on to the start point. Use the Close segment icon to achieve an exact match. 

- For any invalid points, sequentially remove using the **Undo** icon  and to insert the final span select the **Close segment** icon. 

Points can also be input as ***Incremental*** values, using ***X Y Z*** coordinates in the data input box, located below the graphics area.

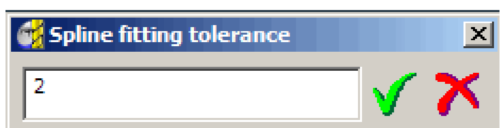


To insert as ***absolute*** coordinate values toggle the icon from  to .

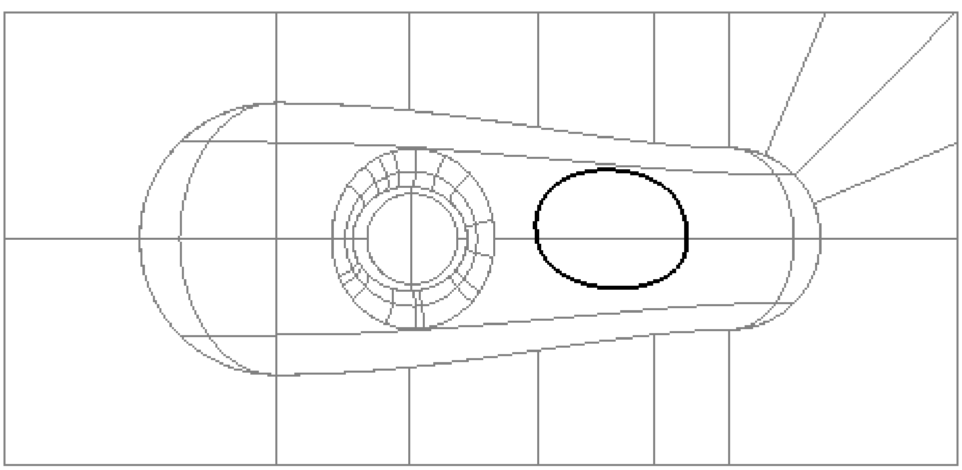


- Once the segment is closed, exit the **Continuous line mode** toolbar by selecting the small cross  at the far right. 

- In the **Curve Editor** toolbar, select the **Accept changes** icon  to accept and close the **Sketch *Boundary*** session.
- With the ***segment*** selected, right click on the new **Boundary** and in the local menu click on **Edit** - **Spline Selected** to open the following form.



- Enter a value of **2** before selecting the **green tick** to apply and close.

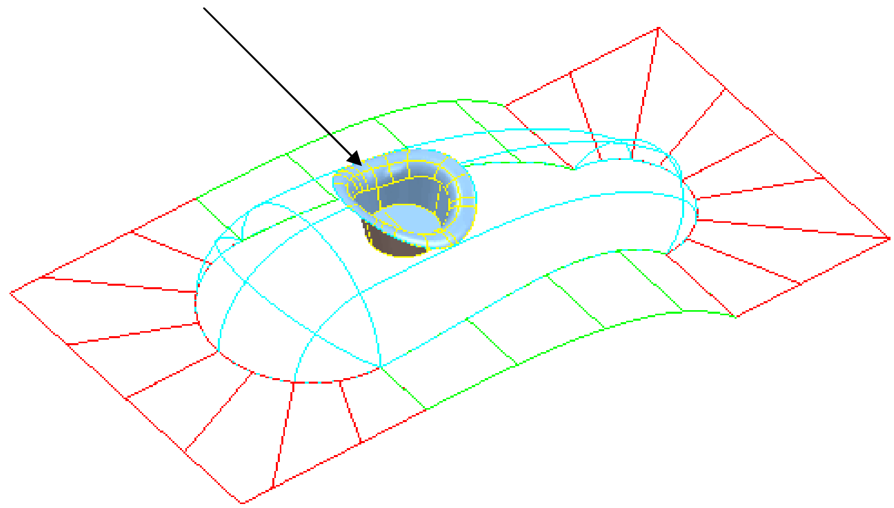


The **Boundary** segment is ***splined*** through the ***polyline points*** where doing so maintains the original form to within a ***maximum deviation*** of **0.5**. In instances where this is not possible a tangency will remain either side of the point.

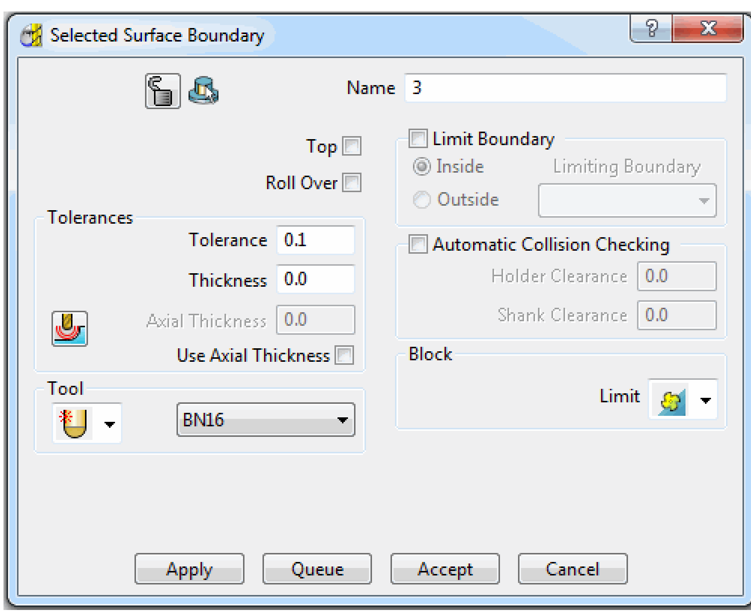
Selected Surface Boundary

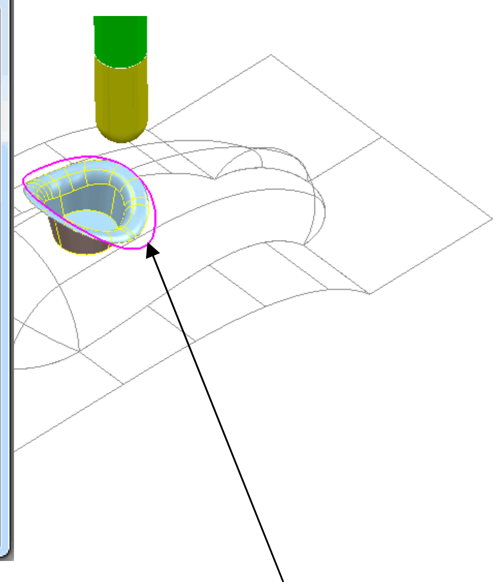
A **Selected Surface Boundary** defines one or more segments where the active tool looses contact with the **selected surfaces**. The segments represent the tip of the active tool.

- Create a **Block *Defined by* Box** using ***Type* Model**.
- Create a **Ball Nosed** tool of ***Diameter* 16** with the ***Name* bn16**.
- Select the **surfaces** defining the central pocket including the fillet.



- In the **explorer** right click over **Boundaries** and select the option **Create Boundary** - **Selected Surface**.



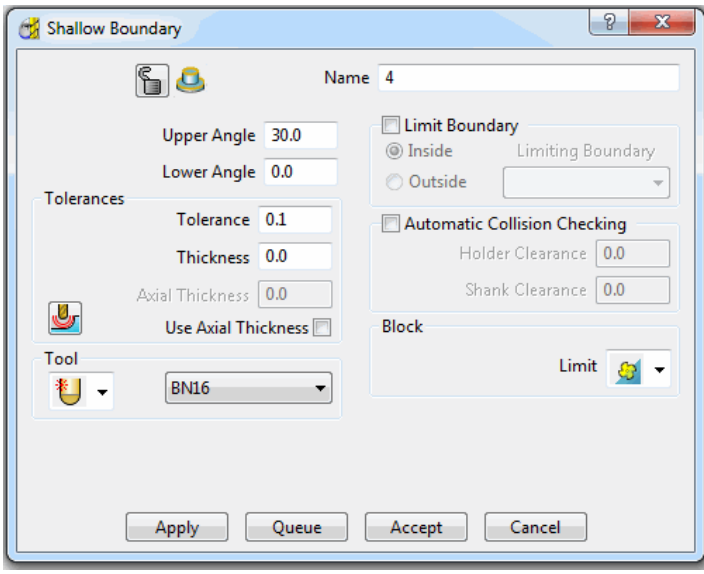


- Using the above settings **Apply** the form to create the above **Boundary** segment.

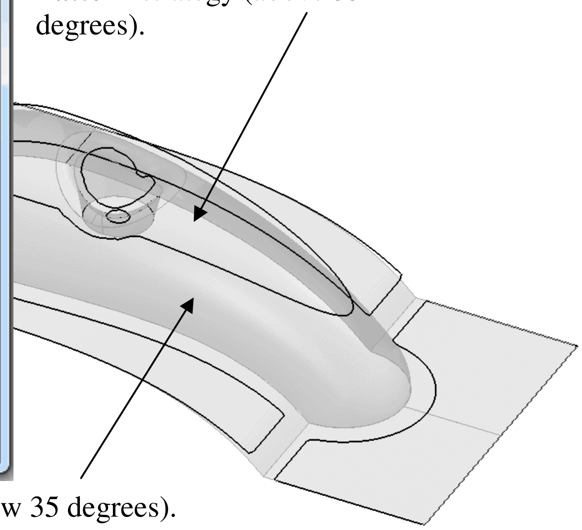
Shallow Boundary

This type of **Boundary** defines one or more segments where the model drops through a specified ***Threshold Angle*** downwards from the horizontal plane. It is designed to differentiate ***steep*** and ***shallow*** areas where ***Constant Z*** and ***Pattern*** strategies are respectively, more effective. The **Boundary** is calculated relative to the **Active** tooling parameters.

- In the **explorer** right click over **Boundaries** and select the option **Create Boundary** - **Shallow**.



Typical area more suited to a **Pattern** strategy (above 35 degrees).



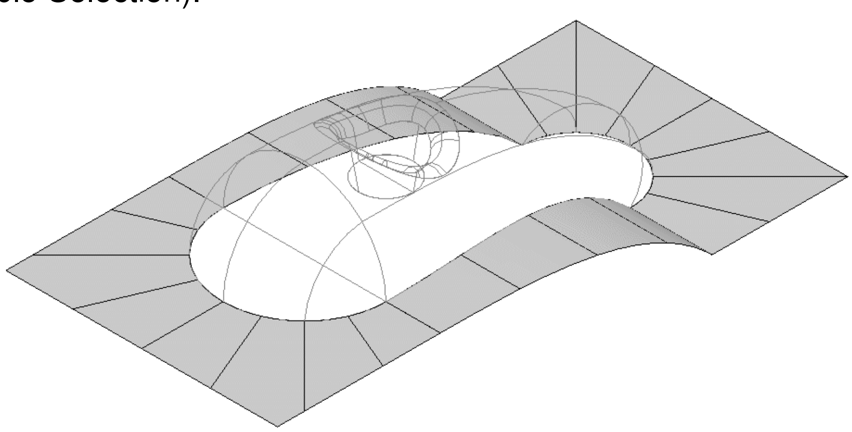
Typical area more suited to a **Constant Z** strategy (below 35 degrees).

- Using the above settings **Apply** the form to create the above **Boundary** segments.

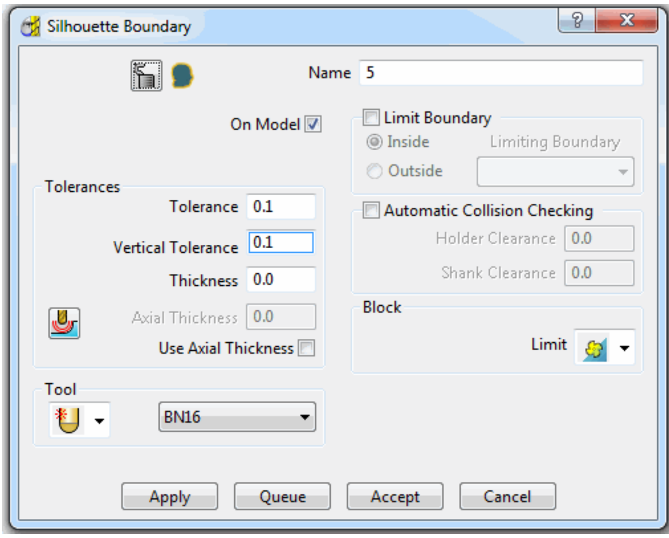
Silhouette Boundary

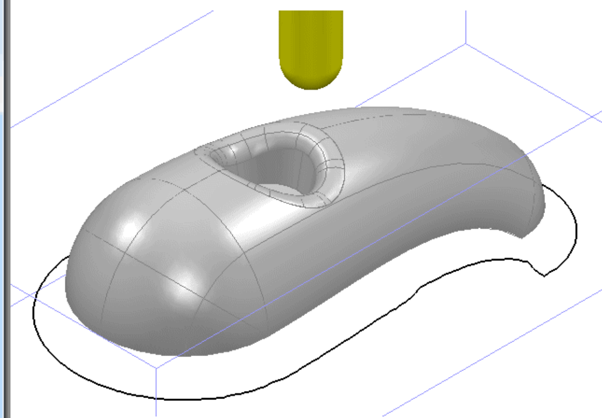
This type of **Boundary** defines the 2D the outline around the selected model adjusted to the contact point of the tool along Z.

- Select the following **surfaces** on the model (use the **Shift key** to enable multiple Selection).



- Right click over the model in the graphics area and in the local menu select **Edit - Delete Selected Components**.
- With the shutout areas now deleted, right click over **Boundaries** in the **explorer** and select the option **Create Boundary** - **Silhouette**.



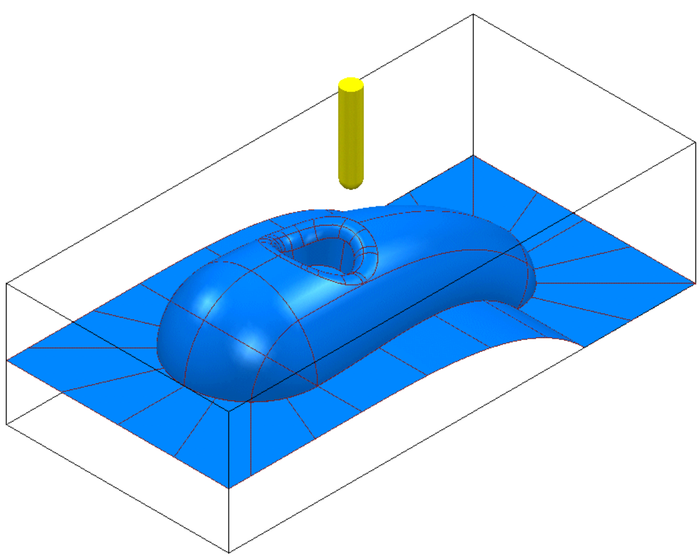


Rest Boundary

This type of **Boundary** defines the area inaccessible to a specified Reference tool. It also requires an (smaller) active tool to be specified otherwise it will not generate segments.

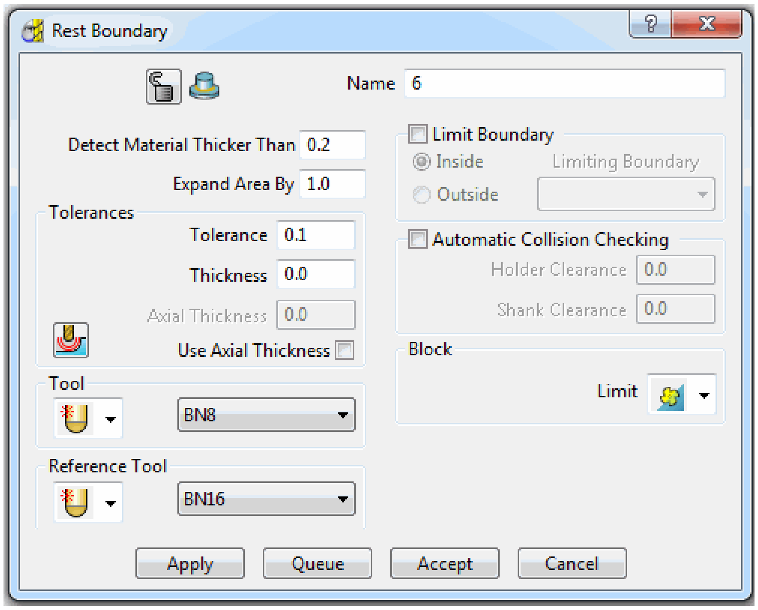
- Create a **Ball Nosed** tool of ***Diameter*** **8** with the ***Name*** **BN8**.
- In the **explorer** right mouse click over **Models** and select **Delete All** to remove the now (incomplete) component.
- **Import** the original (complete) **Model** back into the **Project**:-

  (**C:\users\training\PowerMILL_Data\Models\cowling**)

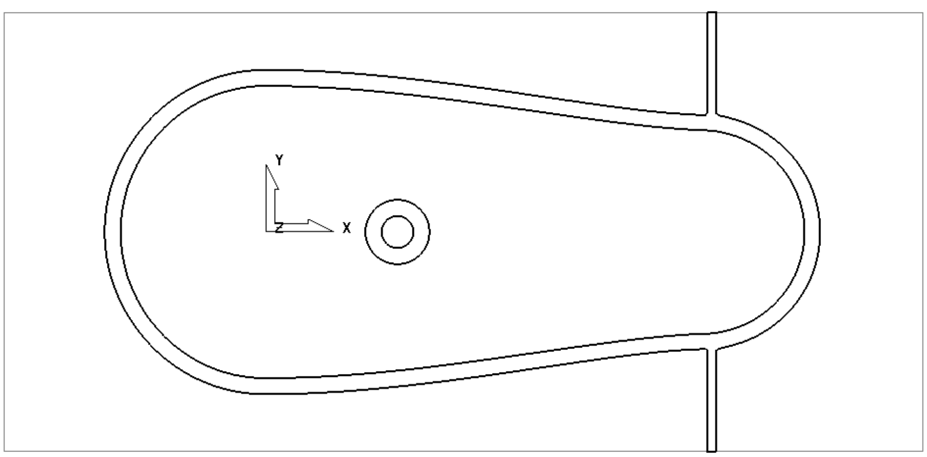


- In the **explorer** right click on **Boundaries**.

- Select **Create Boundary** and select **Rest** to open the **Rest Boundary** form.



- Modify the values in **Expand Area** to **0**, enter **bn8** as the **Tool**, and **bn16** as the **Reference Tool** and click **Apply** to create the following **Rest Boundary**.
- **Accept** the form.
- Select a **View** down the **Z Axis**  and **Undraw** the model.

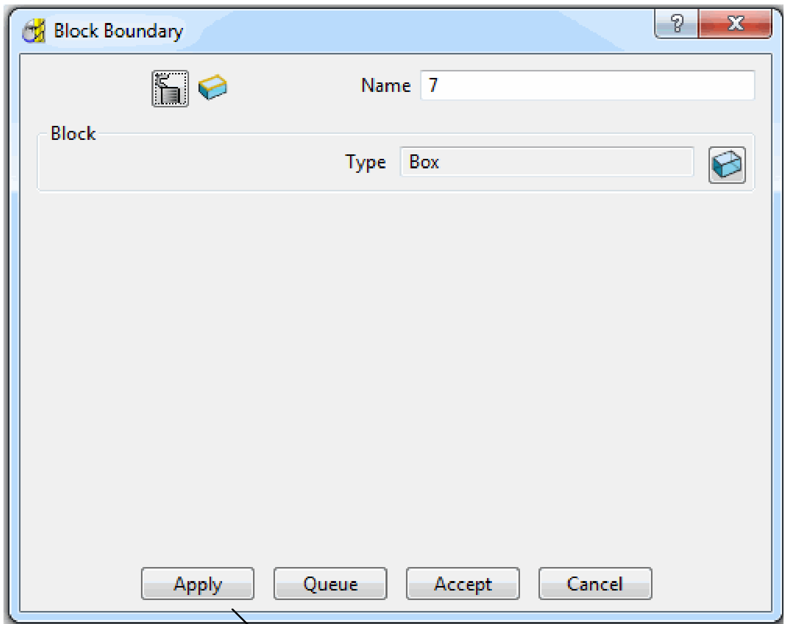


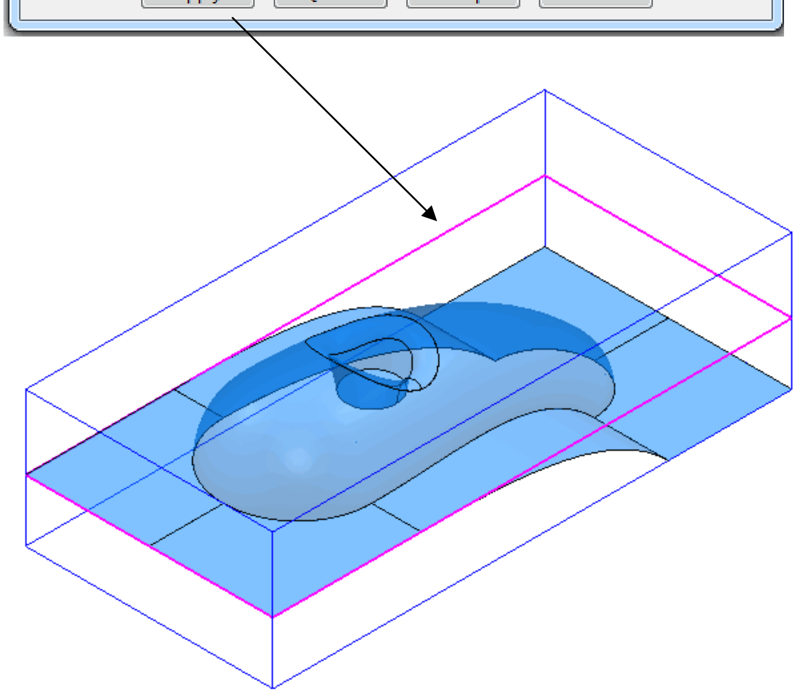
The above **Boundary** identifies areas on the component that are inaccessible to the **bn16** tool geometry to be locally machined with the **bn8** tool. On closer inspection the central pocket area would be more effectively machined with a flat bottomed tool and a smoother more continuous toolpath will be achieved if the 2 spurs were removed from the outer segment, to be machined later as a separate toolpath.

Block Boundary

This **Boundary** option creates segments as a profile around the **Block** definition.

- In **Rapid Move Heights** - **Reset to safe heights**.
- Use the default **Start and End Point** settings.
- **Calculate** a **Block**, ***Defined by*** **Box** with ***Type*** **Model**.
- In the **explorer** right click over **Boundaries** and select the option **Create Boundary - Block**.



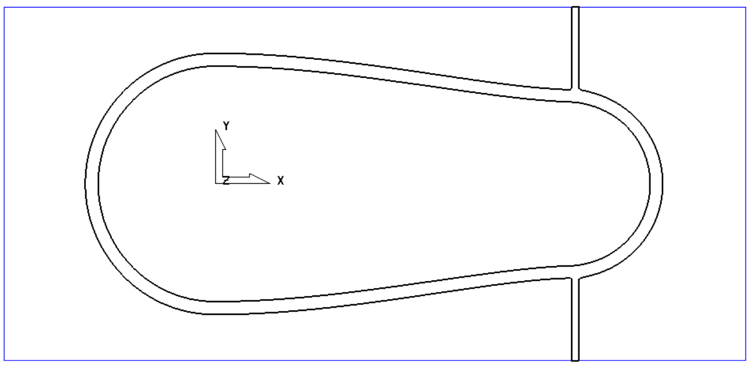


The resultant **2D Boundary** is defined around the outside edge of the **Block** at **Z0**.

Editing Boundaries

Several options are available that enable the user to modify the geometry of a **Boundary**. These include **Curve Modelling**, **Curve Editor**, and **Wireframe Modelling** (Chapter 12).

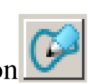
- In the **explorer** right click the **Rest *Boundary*** (**6**) icon to open the local pull down menu and select **Edit - Copy Boundary** to create a copy named (**6_1**).
- In the **explorer** right click over the original **Rest Boundary** (**6**) and **Rename** as **Master**.
- **Rename** the new copy of the **Rest Boundary** (**6_1**) as **Rest**.
- In the **explorer** click (off) the light bulb adjacent to the copy (**Master**) to remove it from view in the graphics area.
- Delete the 2 inner segments on **Boundary** (**Rest**) by boxing over them with the left mouse (select) and press the **Delete** key on the keyboard.



The next stage is to remove the two spurs (arrowed) which will be carried out after an explanation of the **Boundary Editor Form**.

Boundary - Curve Editor

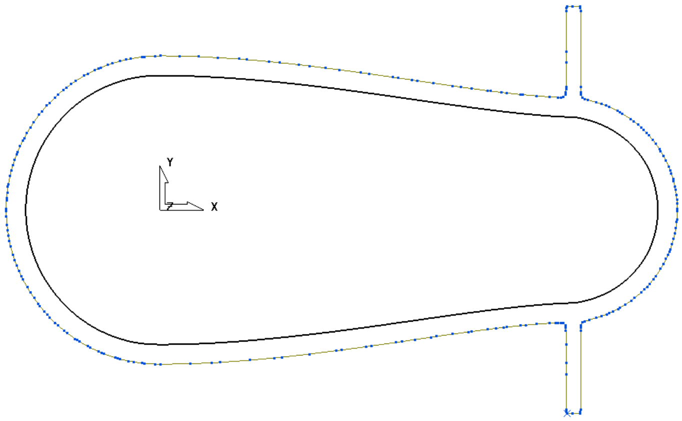
Once created, a **Boundary segment** can be modified using a series options accessed from the **Boundary Toolbar**.

The **Curve Editor** toolbar can be made Active by selecting the option  from the above **Boundary toolbar** (Alternatively, if a **Boundary** segment is ***double left mouse clicked*** the **Curve Editor** toolbar will become active).

Curve Point - editing

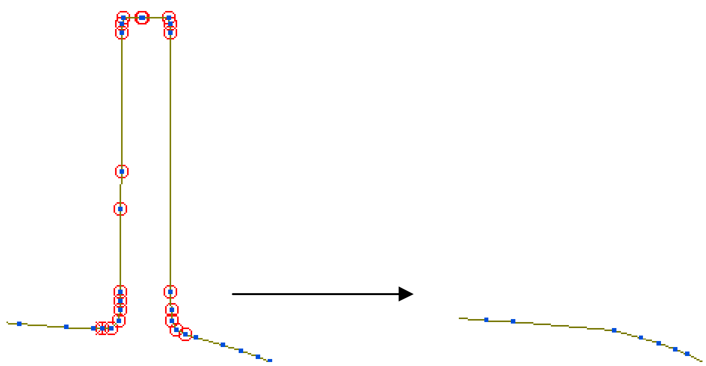
The **Point** editing options  can only be applied to **Continuous Line** (***Polygonised***) segments.

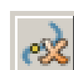
- From the ***PowerMILL* explorer - *Right Mouse Click*** on **Boundary** and from the local menu, select **Toolbar**.
- From the **Toolbar** select the **Curve Editor** option to open the ***Curve Editor toolbar***.
- Select the **outer** - ***Boundary segment*** in the graphics area.

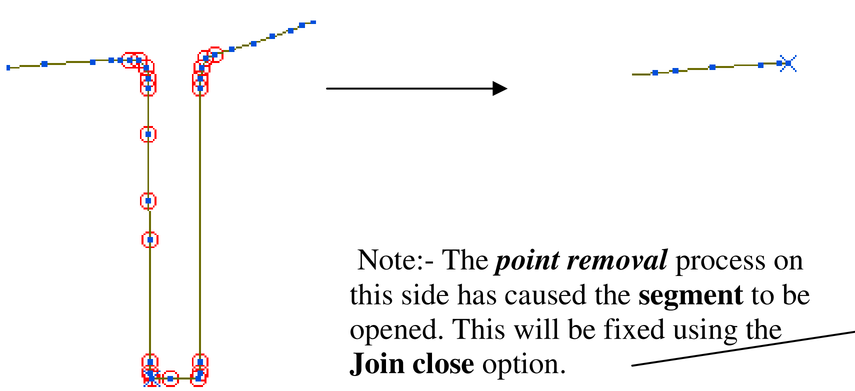


The **Points** will be displayed on the selected **Boundary** ***segment***.

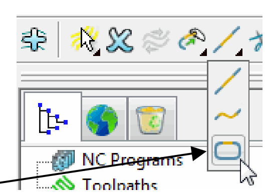
- **Zoom** into the area of the ***selected segment*** (shown below) and select all the **Points** that define the spur (Hold down the **Shift key** while selecting with the **left mouse key**).



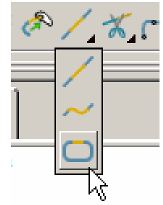
- From the **Curve Editor** toolbar select the **Delete points** option  to remove the spur from the segment.
- Repeat the procedure on the other spur (as shown below).

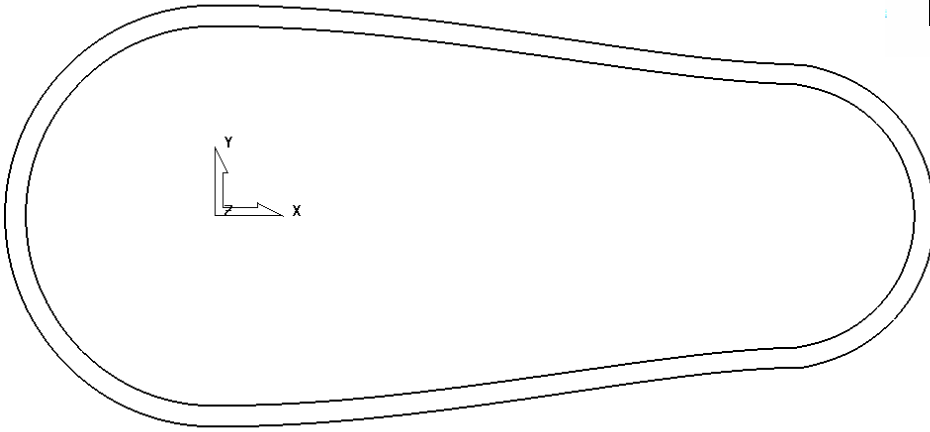


Note:- The ***point removal*** process on this side has caused the **segment** to be opened. This will be fixed using the **Join close** option.



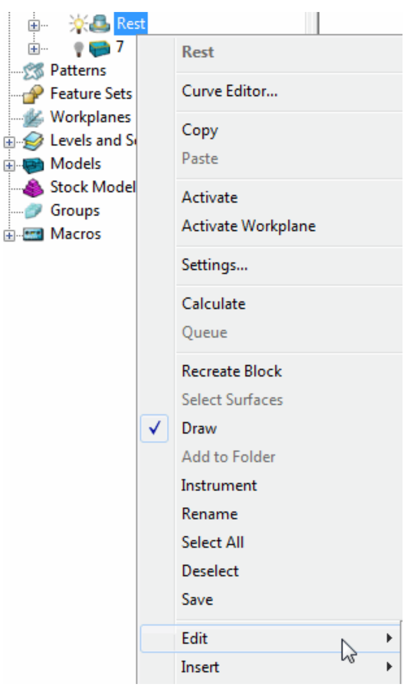
- From the **Curve Editor** toolbar, select the **Join close** option to re-create a ***closed segment***.

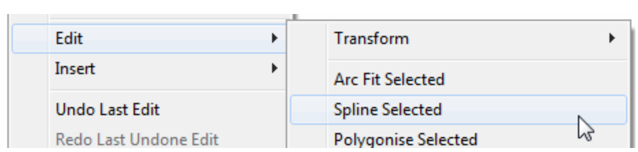




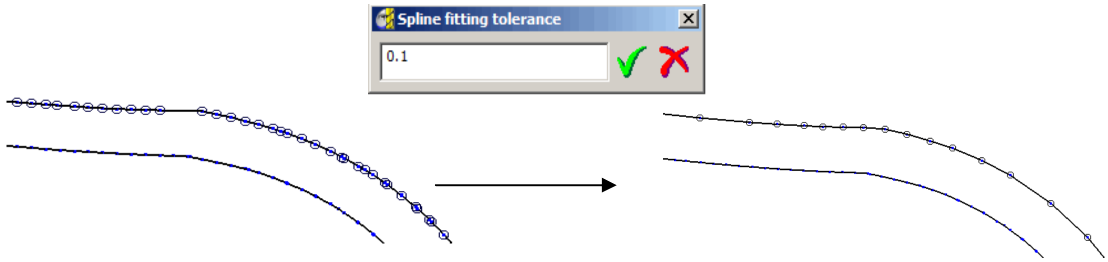
- Make sure that neither of the **2 seqments** is selected (***Unselected*** **segments** will not be returned from the **Curve Editor**).
- To accept the changes and exit the **Curve Editor**, select the **Green Tick** option. 

Other editing options (as shown below) are accessed by right clicking over a **Boundary** either in the ***graphics area*** or in the **explorer** and then selecting **Edit** from the local menu.





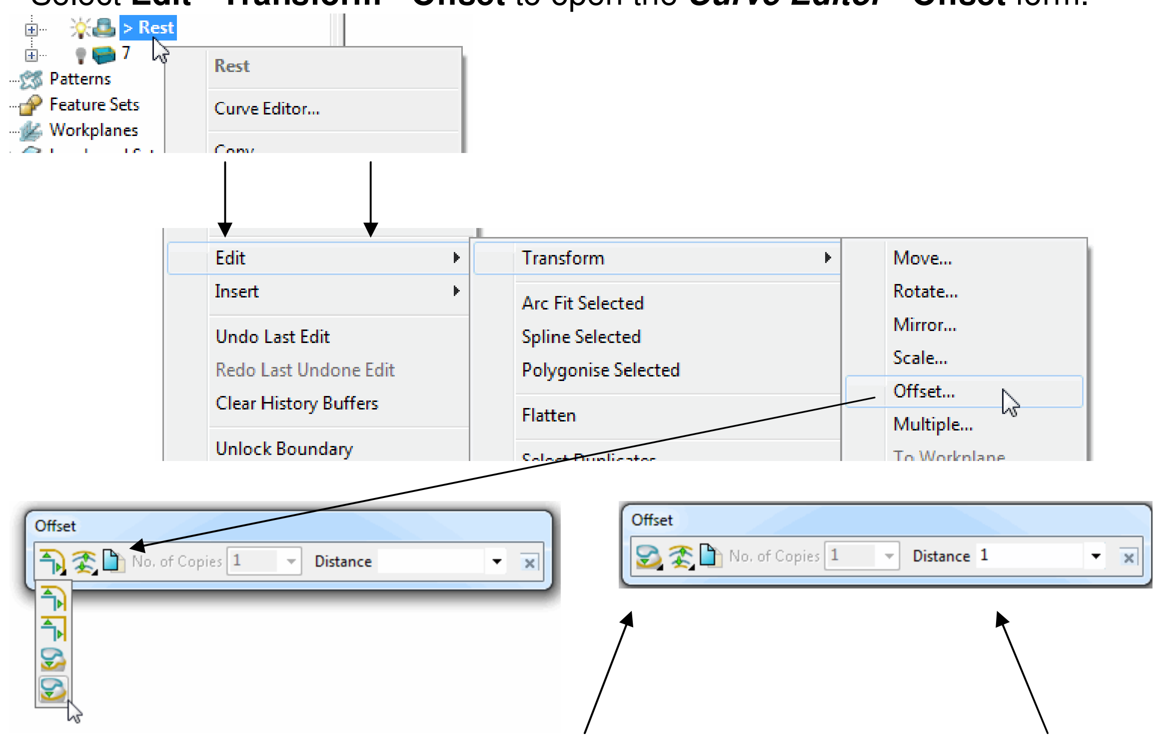
- Select the ***outer segment*** of the **Boundary** (**Rest**) and Right Click on it in the **explorer** to open the local pull down menu.

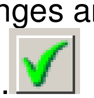


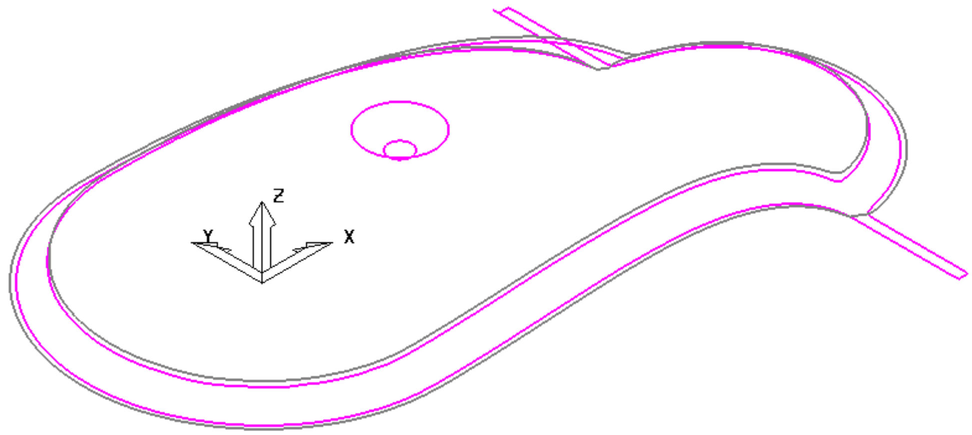
- Select **Edit** - **Spline Selected** entering a ***Spline fitting tolerance*** value of **0.1** in the form before clicking the **Green Tick** to apply.
- Re-select the outer segment to visualise the effect of the **Spline** (as shown above right).

The **Selected Segment** has now been **Splined** (***Curve fitted*** and the ***points re-distributed*** within the specified ***tolerance value***) to produce a smoother form with any unnecessary **points** removed. This will improve the quality of subsequent operations such as ***offsetting*** of the **Boundary**.
Note: It will not be possible to access the **Curve editor - Points** options on the 'Splined' segment unless it is converted back to ***Continuous line*** style (**Polygonise**).

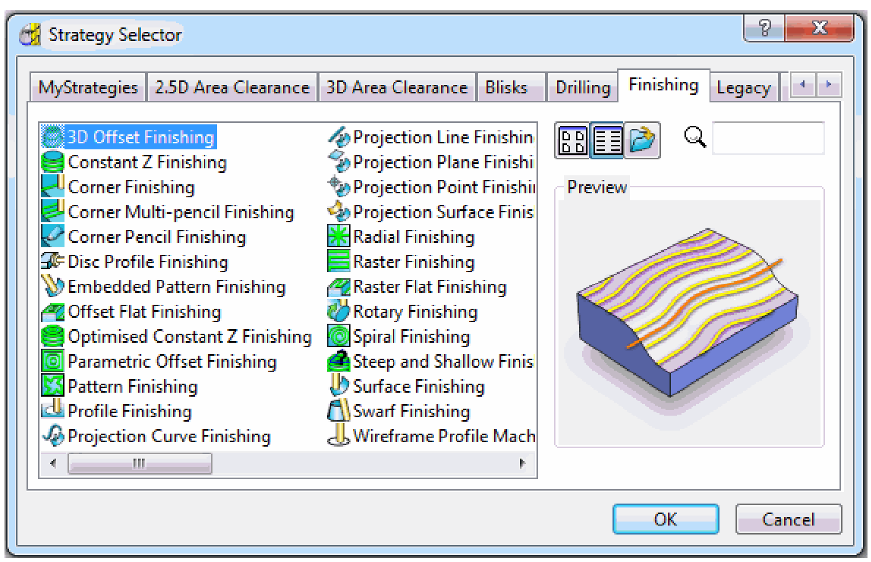
- Right click on the **Boundary** ***icon*** named **Rest** in the **explorer** to open the local menu again.
- Select **Edit - Transform - Offset** to open the ***Curve Editor*** - **Offset** form.



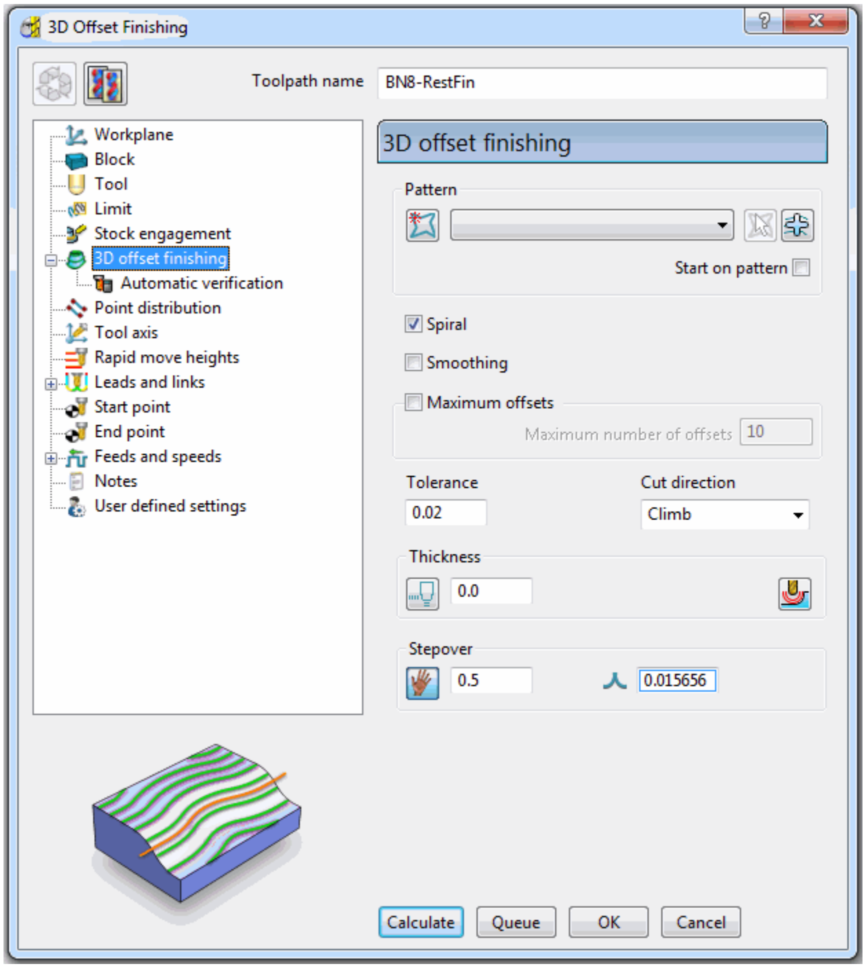
- In the ***Offset*** form select the **3D Smooth** option and input a ***Distance*** of **1** before applying the ***return key***.
- To accept the changes and exit, from the **Curve Editor** toolbar, select the **Green Tick** option. 



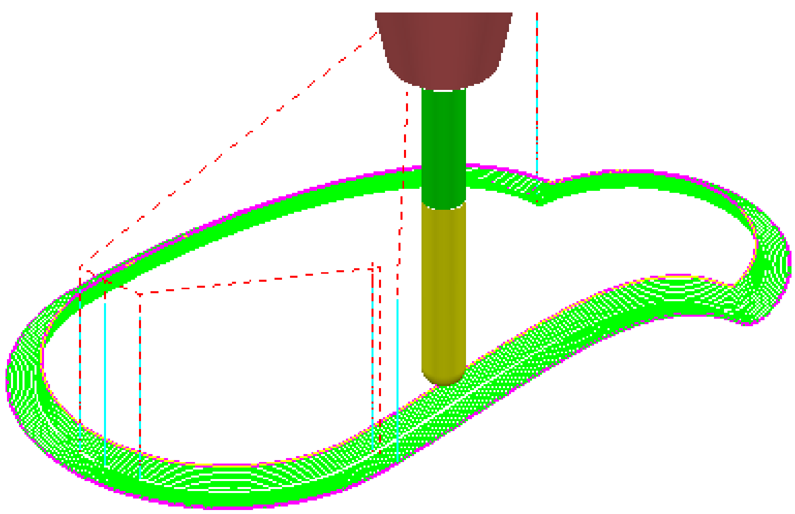
- In the **explorer** click (on) the light bulb adjacent to the copy of the original **Boundary** (**Master**) created earlier (It will be displayed alongside the edited **Boundary** (**Rest**) as shown above illustrating the effect of the 1mm **3D Offset**).
- Select the **Toolpath strategies** icon  to open the following form and select the **Finishing** options.



- Make sure that the **BN8** tool and **Boundary** named **Rest** are both **Active**.
- Select **3D Offset Finishing** to open the following form and enter data exactly as shown below.

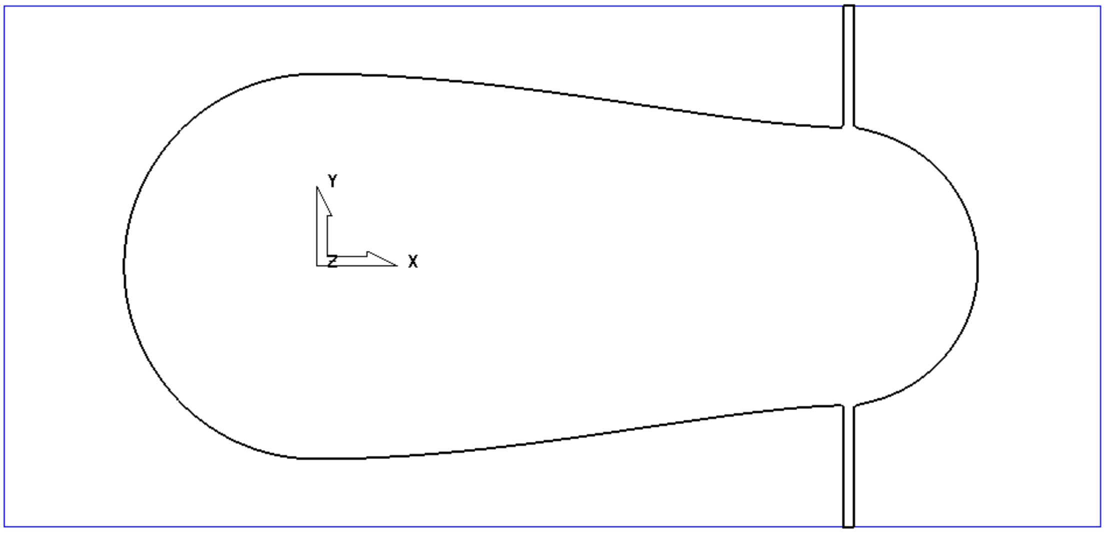


- **Calculate** the form to process the toolpath before selecting Cancel to exit the form.

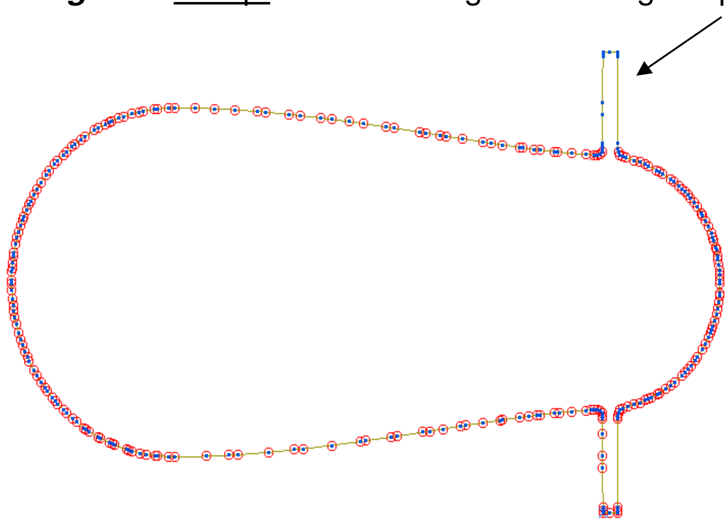


- Undraw the **Toolpath** (**BN8_RestFin**) and **Boundary** (**Rest**) by switching off the respective ***light bulb symbols*** in the **explorer**.
- **Select** the outer segment of the displayed **Boundary** (**master**).

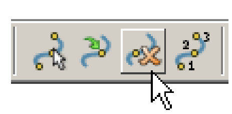
- Right click the **Boundary** icon (**Master**) in the **explorer** to open the local pull down menu and select **Edit - Copy Boundary (selected only)** to create a new **Boundary** (**master_1**) consisting of the outer segment only.
- Open the local pull down menu again and **Rename** the copy as (**Spur1**).
- In the **explorer** click (off) the light bulb adjacent to the **Boundary** (**master**) to remove it from view.

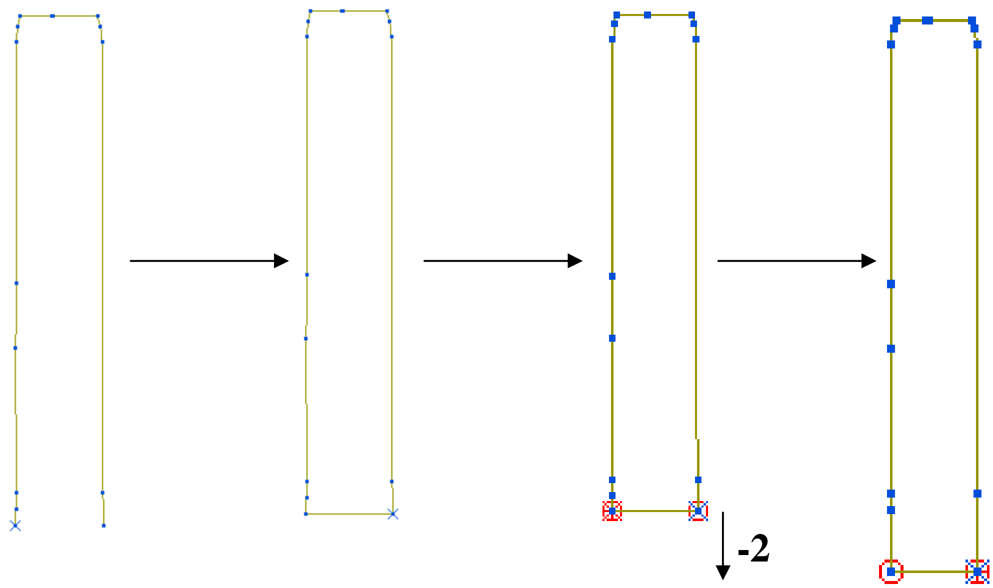


- Right click over the **Boundary icon** (**Spur1**) in the **Explorer** and select **Activate**.
- Double left click on the **Boundary *segment*** to activate the **Curve Editor** toolbar.
- With the **Shift key** depressed, Select all **points** along the **Boundary *segment*** except those defining the rectangular part off the upper spur.

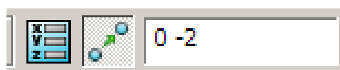


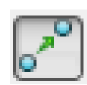
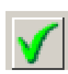
Notes:-
1/ The point selection process can be carried out in stages enabling the user to **Zoom in** as the bulk of the points have are deleted.
2/ If it is required to deselect one or more of the selected points the **CTRL key** is used during the selection process.

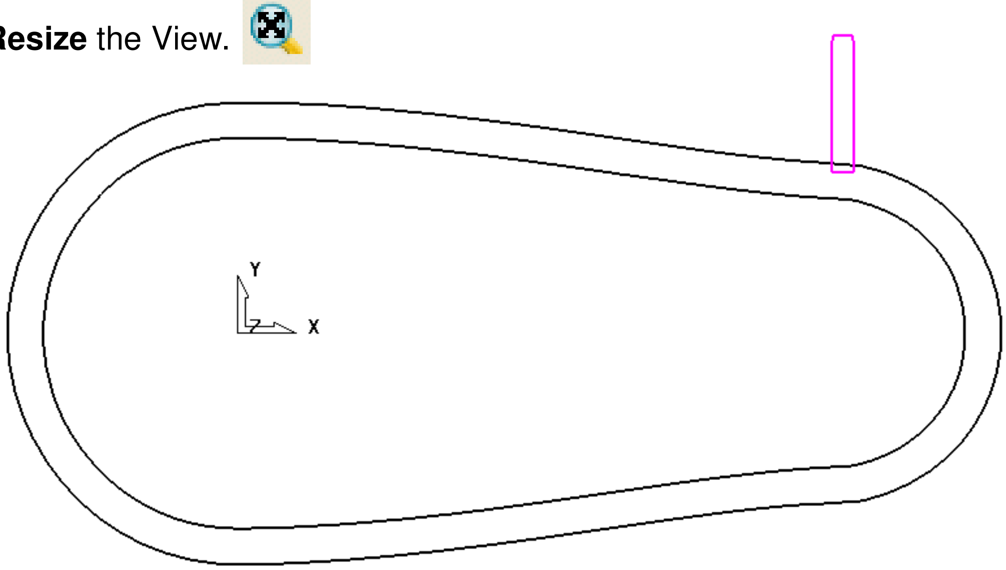
- In the **Curve Editor** toolbar select the **Delete Points** option. 



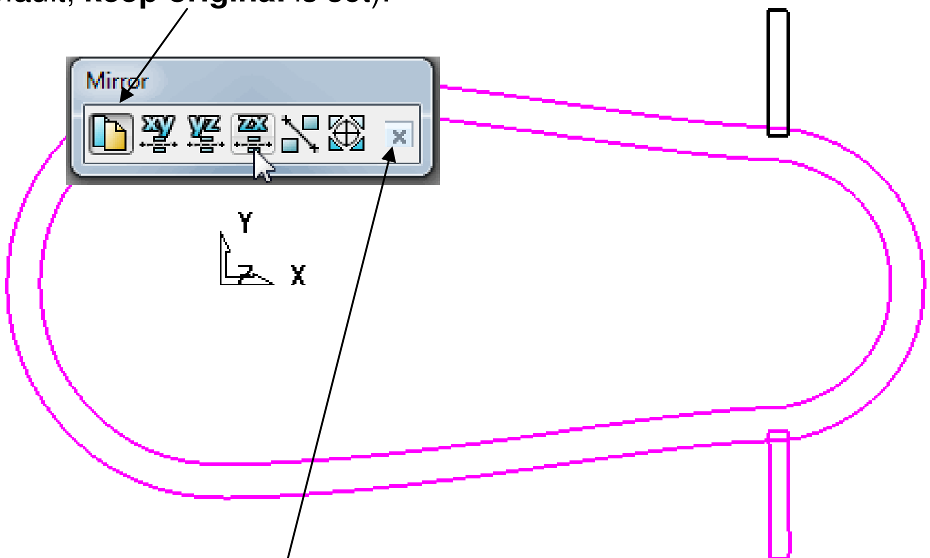
- Select the **Join Close** option to ***close*** the open **segment**.
- ***Shift - Select*** the **points** at the base of the rectangular **segment**.



- From below the **Graphics Area** with ***Relative*** move  selected, input **0 -2** in the ***Data input box*** and return (The selected points will move by a distance of **Y-2**).
- Select the **Green Tick**  to exit the **Curve Editor** toolbar.
- Display (light bulb on) both **Boundaries** - **Rest** and **Spur_1**
- **Resize** the View. 



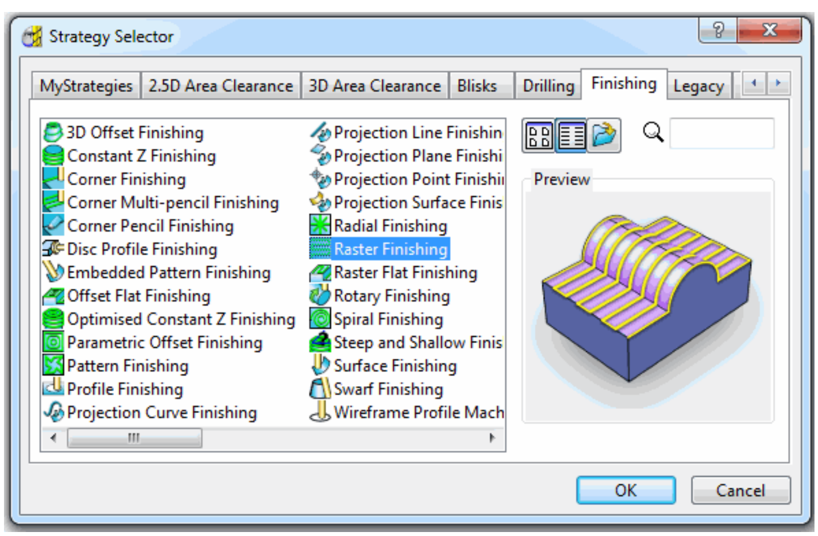
- In the **explorer** right click the **Boundary** (**Spur_1**) and from the local menu select **Edit - Transform - Mirror**.
- In the ***Transform Boundary*** form select the ***mirror* ZX** option (Note: The default, **keep original** is set).



- Click on the small ***cross*** to accept the ***mirror*** operation and close the form.
- Select the **Green Tick**  to accept and exit the **Curve Editor** toolbar.

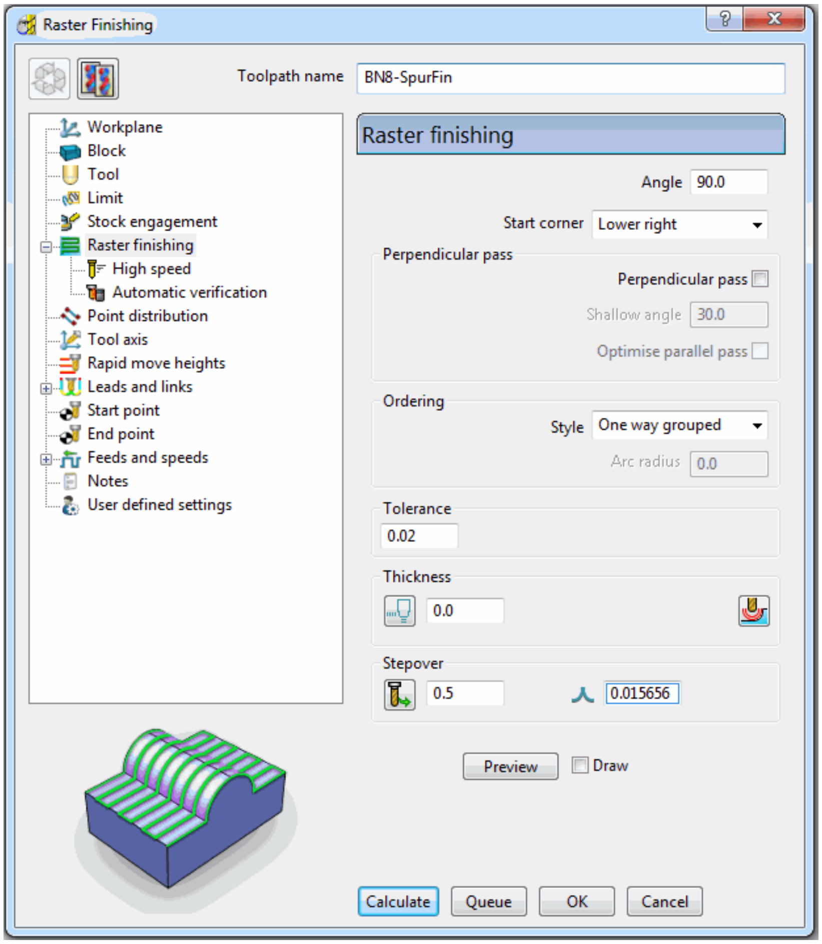
The 2 separate ***spur segments*** are now both contained in the **Boundary** named **Spur_1**

- Make sure that the ***tool*** - **BN8** and ***Boundary*** - **Spur_1** are both **Active**.
- Open the **Strategy Selector** form and select the **Finishing** options.

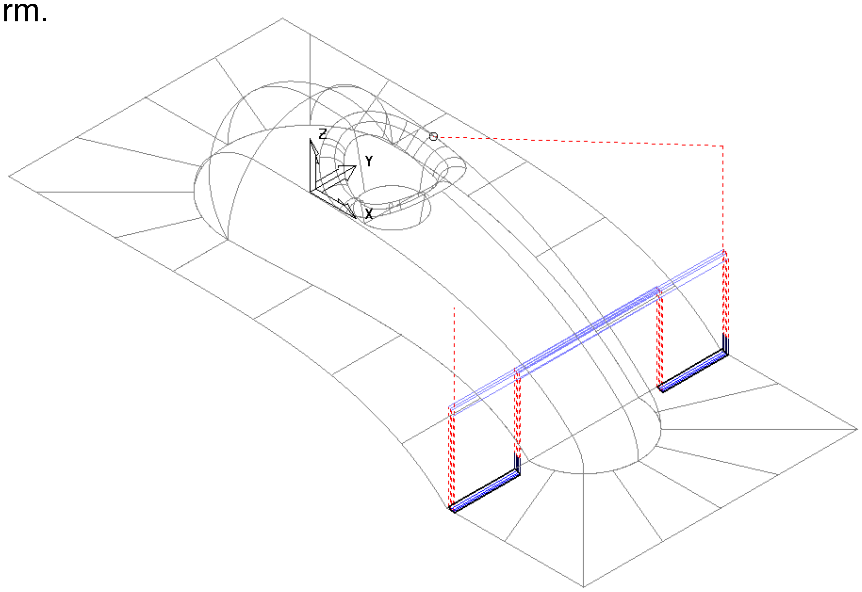


- Select the **Raster Finishing option.**

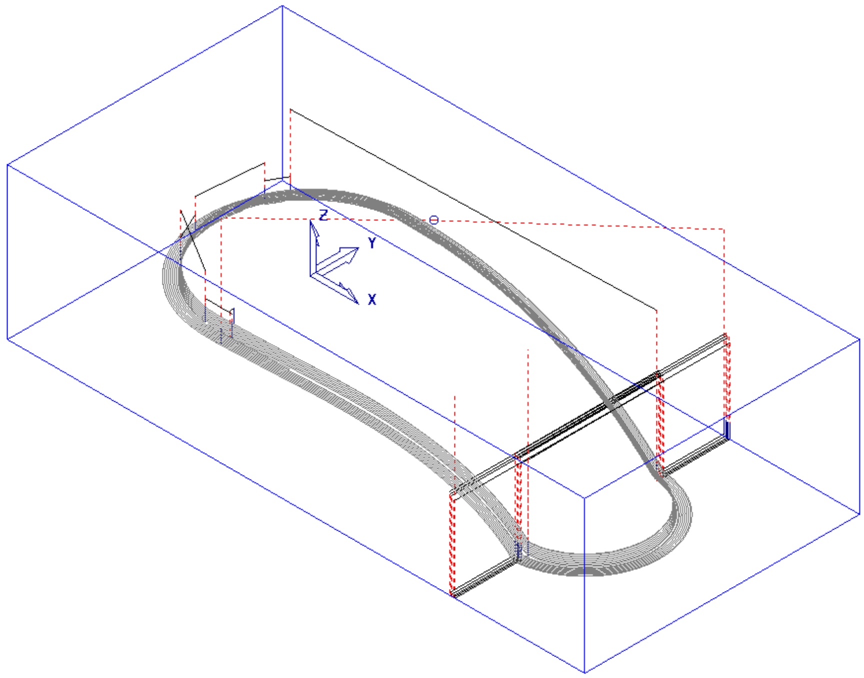
- In the **Raster Finishing** form and enter data exactly as shown below.



- Select **Calculate** to process the toolpath and then select **Cancel** to exit the form.



The machining strategies have been localised and the tooltrack shape controlled with the help of edited **Boundaries** originating from a single **Rest Boundary**.



- **Save** the **Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\Boundary-tests

Creating Blanking Surfaces using Boundaries

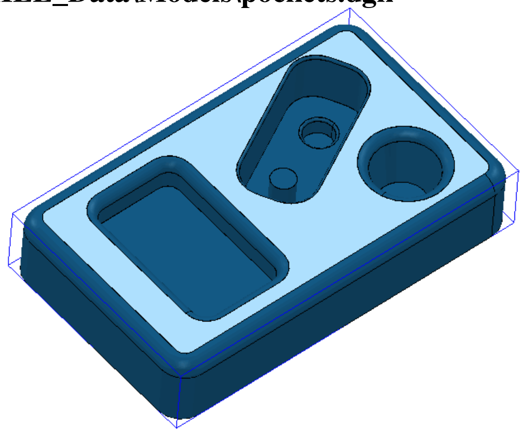
Blanking Surfaces are used to cap holes and pockets in a model to prevent the tool from machining into those areas. There are three types of **Plane** that can be created in **PowerMILL**.

1. **Block –** allows the creation of a ***plane*** at a specific ***Z Height*** after the material **Block** had been defined.
2. **Best Fit –** creates ***two planes***, one touching the **boundary** at its highest point and the other through its lowest point. The planes are parallel to the best fit plane (the plane that has the smallest maximum distance from the **Boundary**).
3. **Projected –** creates a plane parallel to the X Y axis of the active workplane. The plane will have the Z Height of the highest point of the boundary.

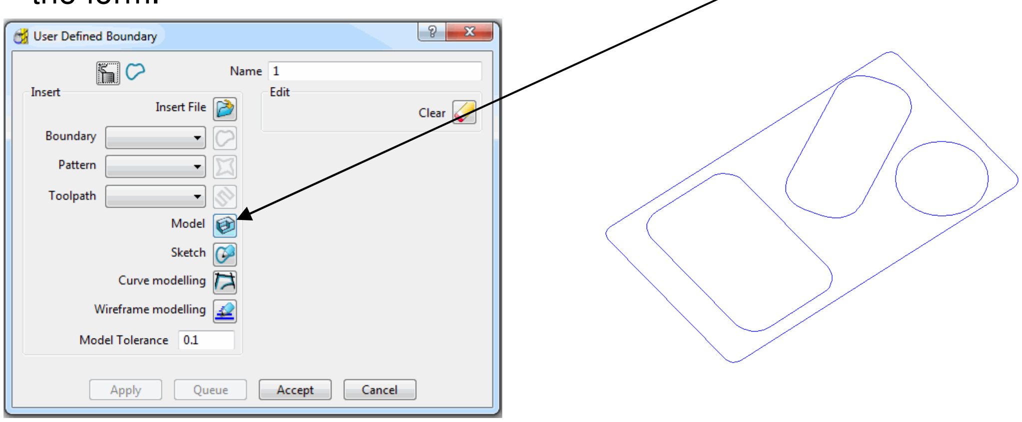
Example

- **Delete** the **current model, toolpaths and boundaries.**
- **Import** the **model**:-

    C:\users\training\PowerMILL_Data\Models\pockets.dgk

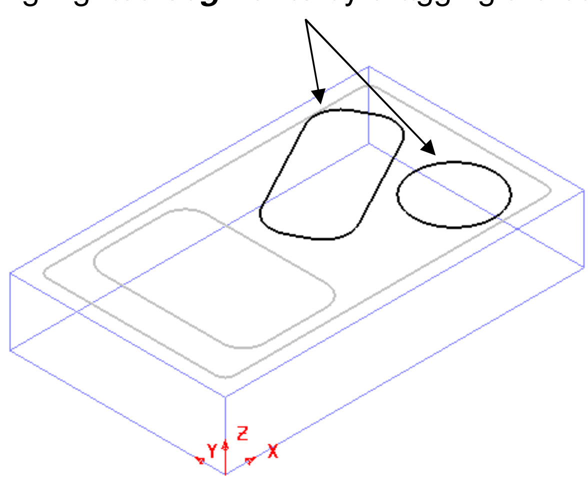


- Define the **Block** to **Min/Max limits**.
- Select the top **surface** of the ***model*** (shown shaded).
- Create a **User Defined Boundary**, select the icon **Model** and then **Accept** the form.

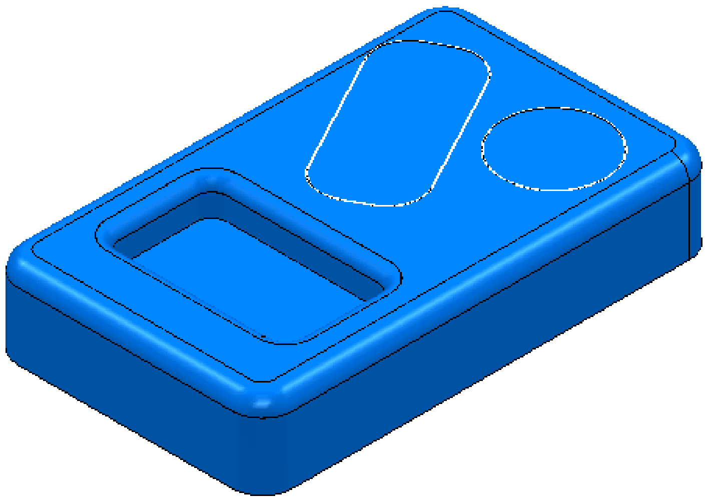


The **boundary** produced has 4 segments. For this example only the two smallest segments are required.

- Select the two highlighted ***segments*** by dragging the ***cursor*** over them.



- Right click over **Models** in the **explorer**, and select **Create Plane - Projected**.



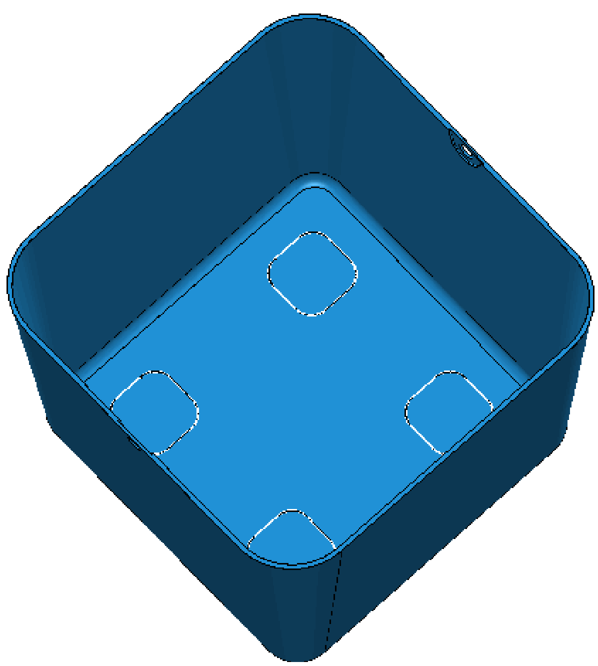
Two planer, blanking **Surfaces** are created as shown.

No need to Save the Project this time!

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

Exercise

- From **Examples**, open the model **bucket.dgk** and generate ***4 surface planes*** to cover the foot recesses on the upper base surface.



9. Feature sets / 2D machining

Introduction

PowerMILL has a range of **2D strategies** which operate specifically on entities called **Features**. These are ***extruded*** along Z from **wireframes** (**Pattern** or imported **model**) and are assigned as specific types such as **Boss**, **Pocket**, **Slot**, **Hole**, etc. **Features** are machined independently (not gouge checked) to any existing **surfaces/solids**. A **Feature** is displayed as an upper and lower contour linked by vertical lines. A ***2.5D Component*** is built up from a **Feature Set** consisting of one or more related **Features**. Milling will occur collectively in areas as dictated by the individual **Feature** types.

1. Features

Features are created from **2D geometry**, and are individually defined as a **Pocket**, **Slot**, **Boss** or **Hole**. It is also possible to extract **Hole** features directly from a **Surface** or **Solid** model as well as from **Area Clearance** strategies when using the **Drilling** option for **Lead In Moves**.

2. Area Clearance (2.5D machining strategies)

On completion of a **Feature Set**, the **2.5D Area Clearance** strategies are applied to create roughing, semi finishing, and finishing strategies.

3. Drilling

Drilling options can only be applied to **Hole Features**. Types of cycle supported include **Standard drilling**, **Boring**, **Helical milling**, **Tapping** and **Thread Milling**.

Features

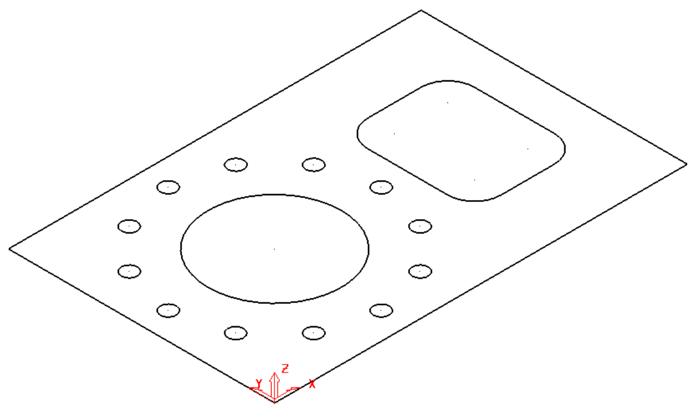
There are six different types of **Feature** which respond specifically to the **2.5D Area Clearance** options:-

1. **Pocket**, relates to the area inside a **Feature**. Machining will occur inside the **Feature**.
2. **Slot,** relates to the tool tracking along the **Feature** contour (with or without LH or RH cutter compensation).
3. **Boss**, relates to the area outside the **Feature**. Machining will occur outside the **Feature**.
4. **Hole**, this type of **Feature** is specifically used with **Drilling** strategies and can be defined from points, circles, curves, or directly from CAD model data.
5. **Circular Pocket -** a circular pocket is defined from points, circles or curves.
6. **Circular Boss -** a circular boss is defined from points, circles or curves.

Note: it is not possible to change an existing a **Feature** to one of a different type.
Stacked 2D/3D data containing pairs of wireframe **circles/curves/lines** or **cylindrical surface**s can be imported into **PowerMILL** to be directly defined as **Hole** features, thereby removing the need to manually input some or all of the dimensions.

Creating Features from Pattern segments

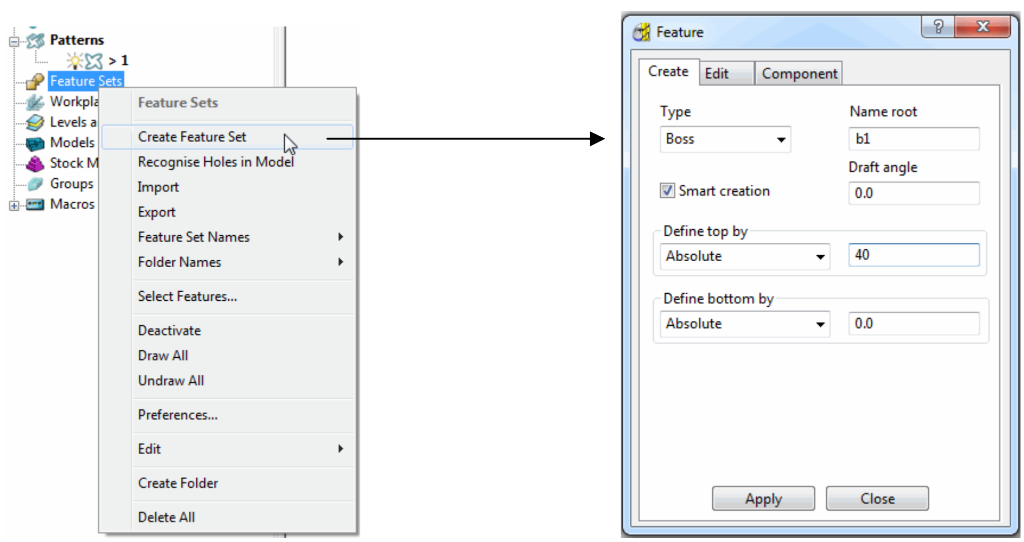
- Select **File →Delete All** and **Tools → Reset forms**.
- **Open** the ***Project***:-
  C:\users\training\PowerMILL_Data\Projects\2D-Drawing



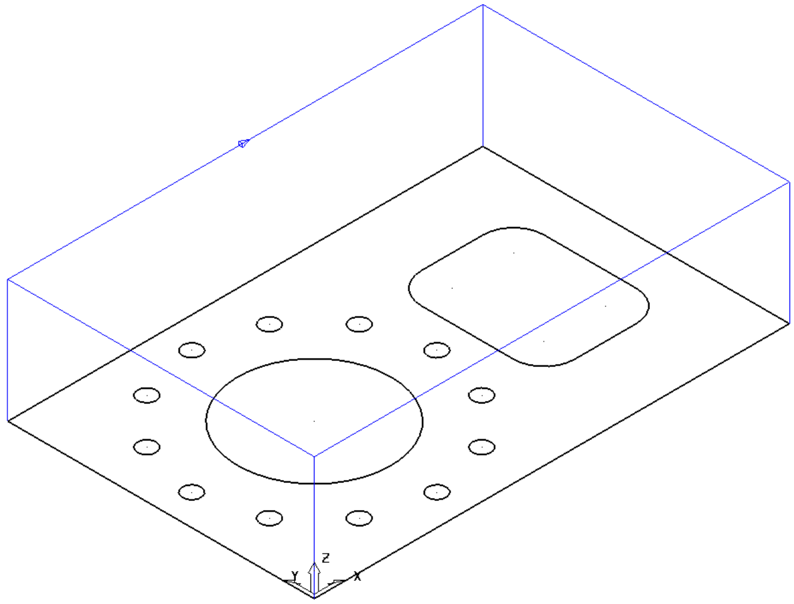
The imported **Project** contains a **Pattern** defining **2D geometry** to be used to create a **Feature Set**. The **Project** is **Locked** to prevent it from being altered hence the first step is to **Save As** a separate **Project** locally with a different name.

- Select **File** - **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\2DPatternExample

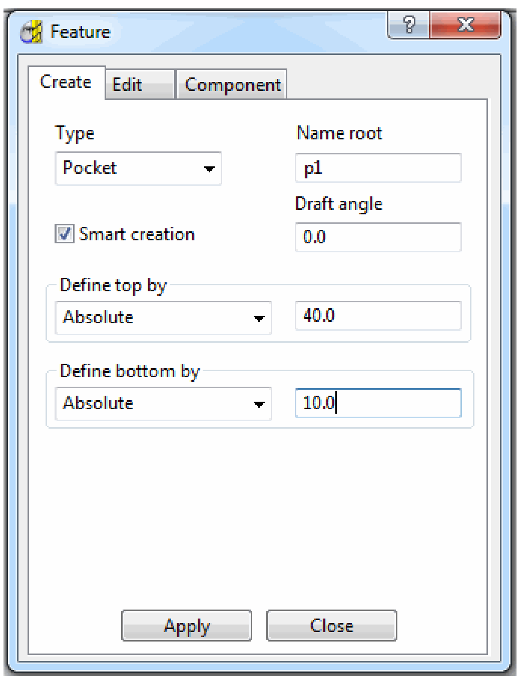
- In the **explorer** right mouse click on **Feature Sets** to access the local menu and select **Create Feature Set**.

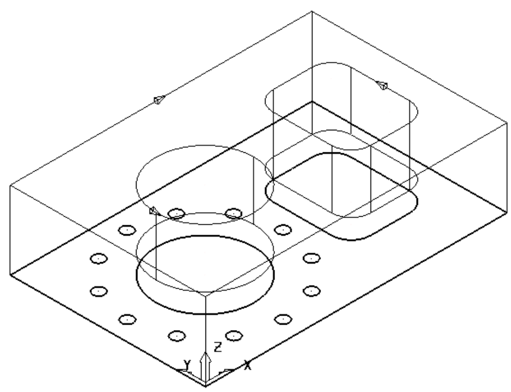


- A new (empty) **Feature Set** - **1** will appear in the **explorer** and the **Feature Form** will open ready to build the ***2.5D model***.
- **Select** the larger, rectangular **Pattern Segment** and enter values exactly as shown in the above right **Feature Form** before selecting **Apply** (Do not close the form).



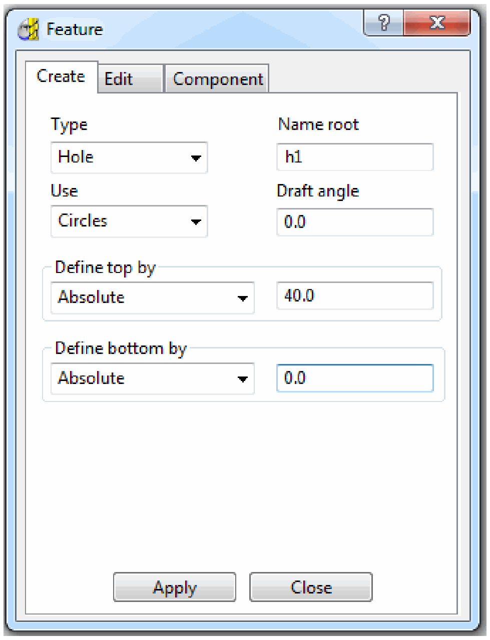
The first **Feature** is the main component body defined as a **Boss** (as shown above).

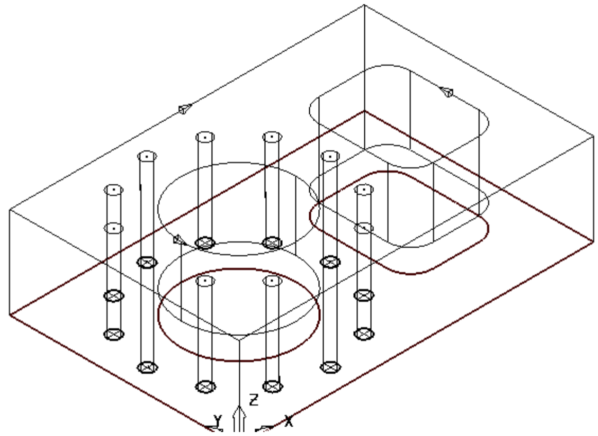




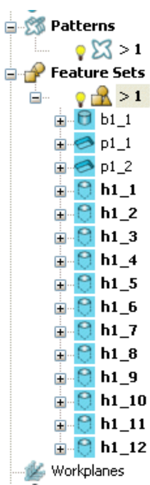
- **Select** both, the filleted rectangle and the large circle **Pattern Segments** and enter values exactly as shown in the above left **Feature Form** before selecting **Apply** (Do not close the form).

If the form is closed by mistake to reactivate on the same **Feature Set** - right mouse click on ***Feature Set*** named **1** in the **explorer** and in the local menu select **Settings**.





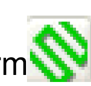
- **Select** all **12** of the **Dia 6 circles** and enter values exactly as shown in the above left **Feature Form** before selecting **Apply**.



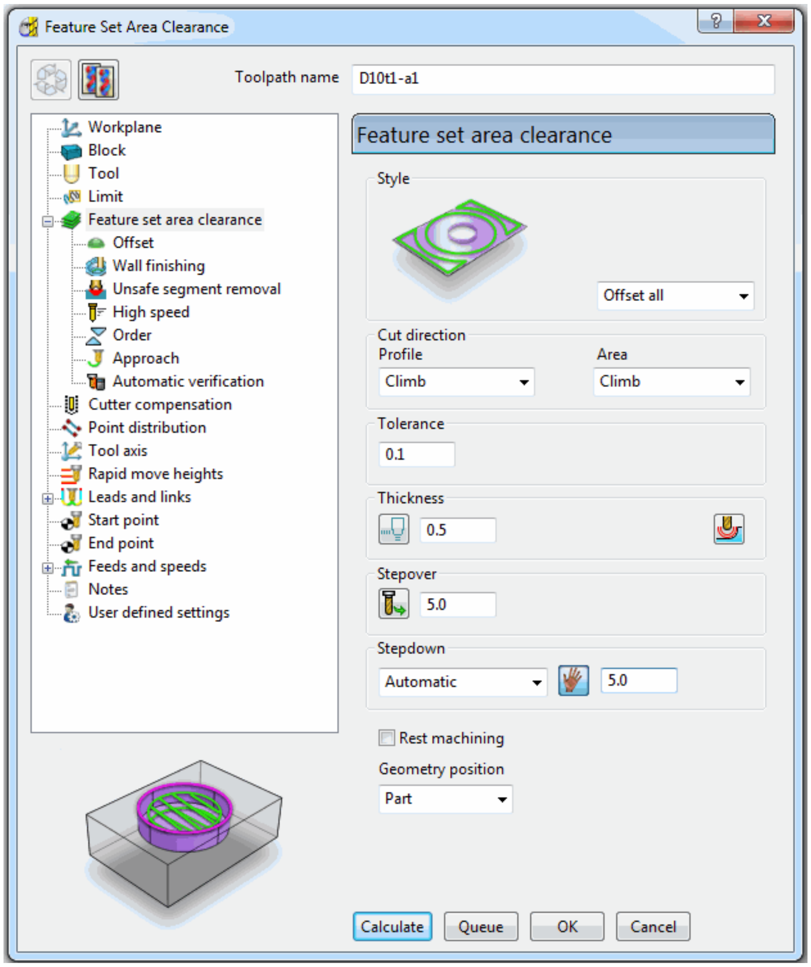
The contents of the **Feature Set** (**1**) should be similar to the illustration on the left (Note: The Author has opted to use a naming system where a **Boss** is prefixed with a **b**, a **Pocket** with a **p**, and a **Hole** with an **h**).

The **Feature Set** is now complete and ready for the creation of suitable **2.5D Machining Strategies**.

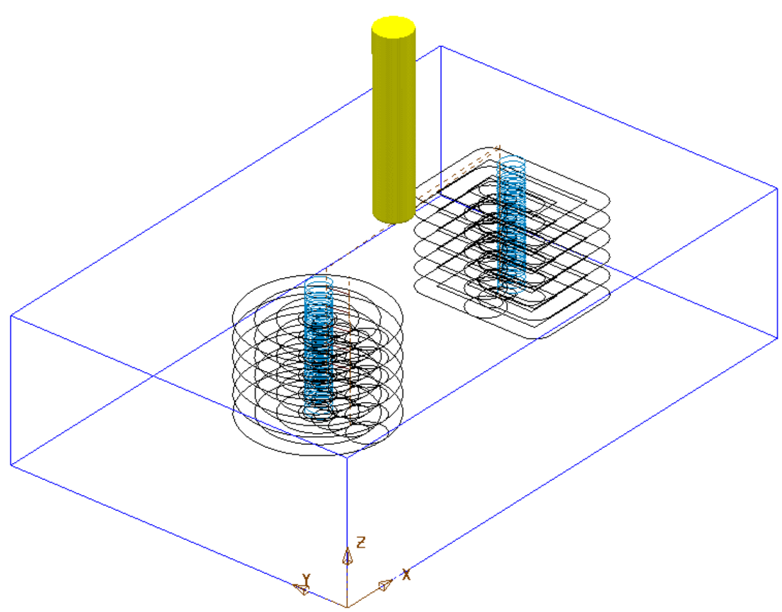
2D Feature Set Machining

- Define a **Dia 10 - tiprad 1** - **Length 50** tool named **D10T1**.
- In the **Block** form  set ***Type*** - **Feature** before selecting **Calculate**.
- In the **Rapid Move Heights** form  select **Calculate**.
- Open the **Toolpath Strategies** form  and select **2.5D Area Clearance**.

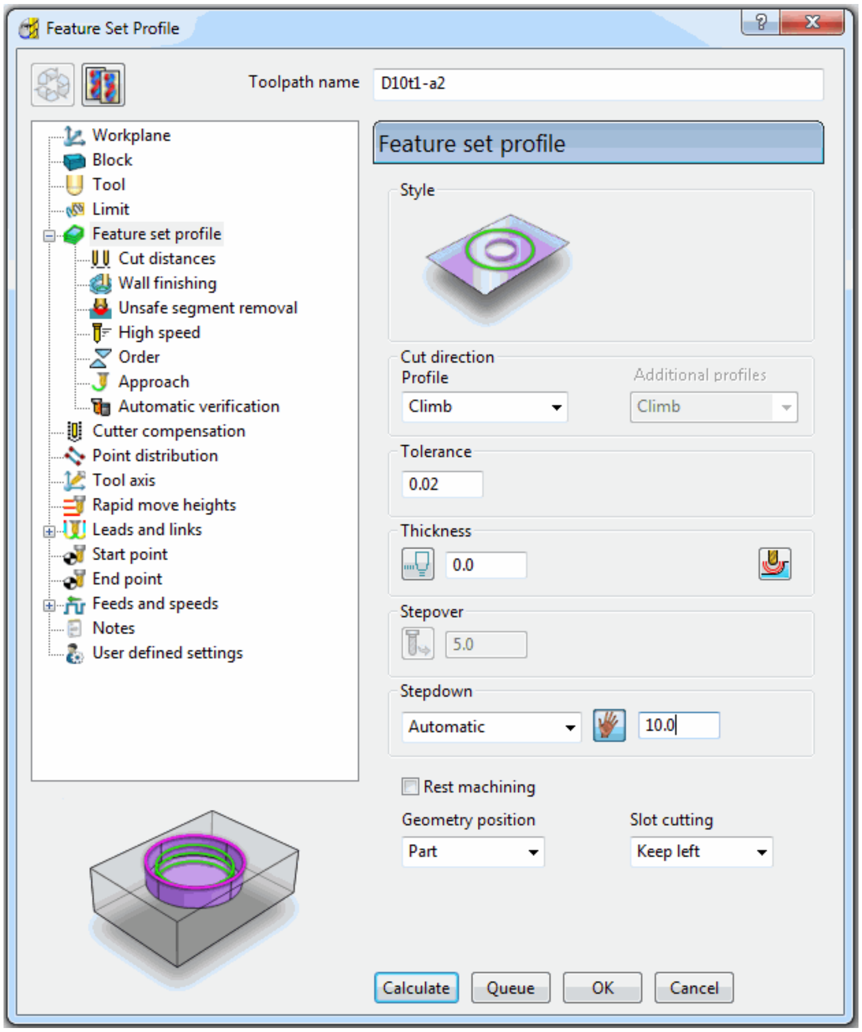
- Select **Feature Set Area Clearance** and input data exactly as shown in the following form.



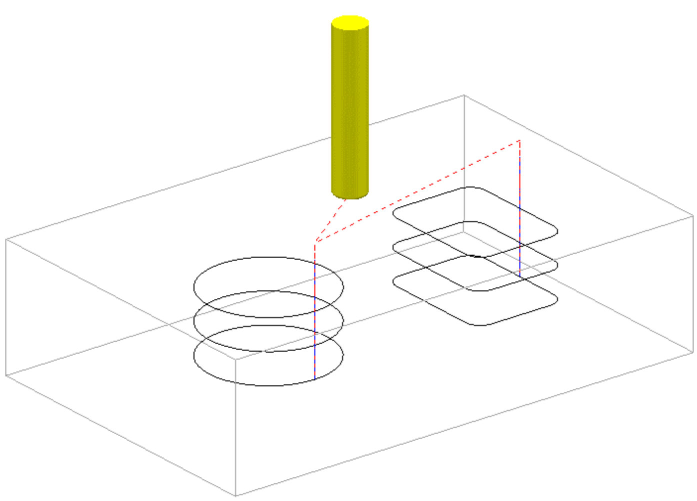
- Select **Calculate** to process the toolpath and then **Cancel** the form.



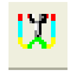
- Select **Feature Set Profile** and input data exactly as shown in the following form.

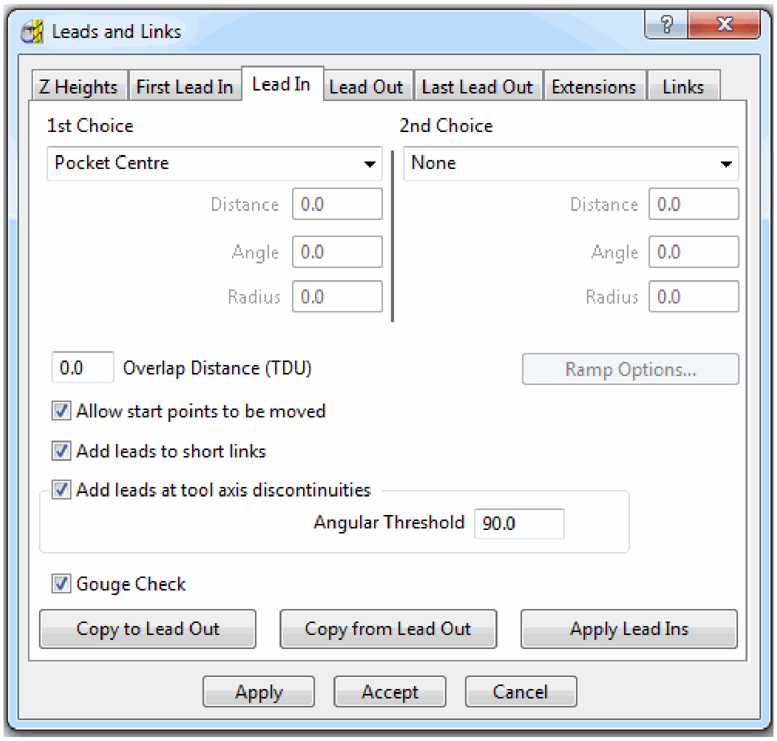


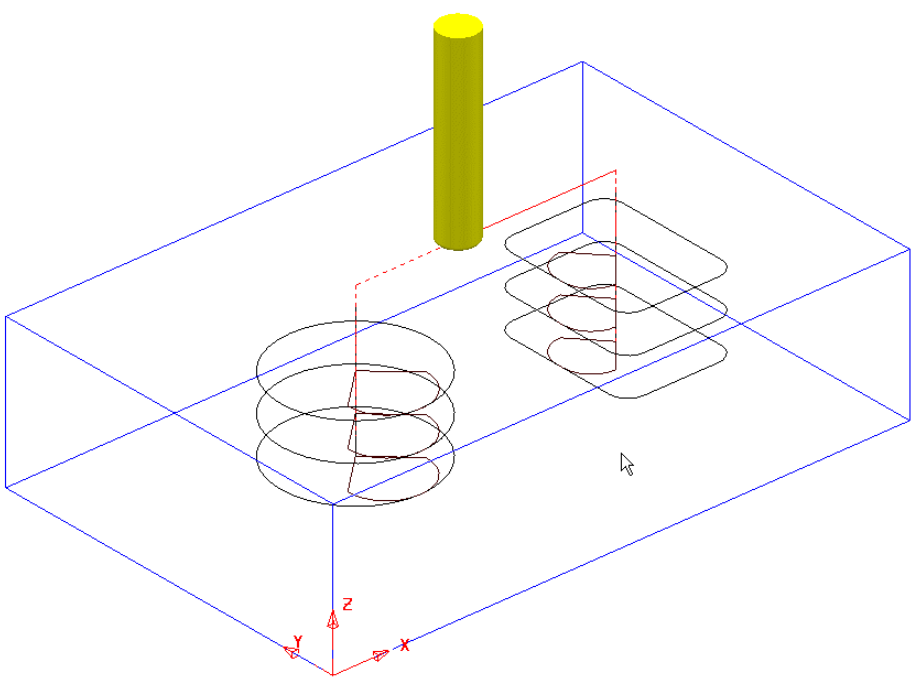
- Select **Calculate** to process the toolpath and then **Cancel** the form.



Note the tool is plunging directly onto the form down Z. This situation will be changed retrospectively by applying appropriate options in the **Leads and Links** form.

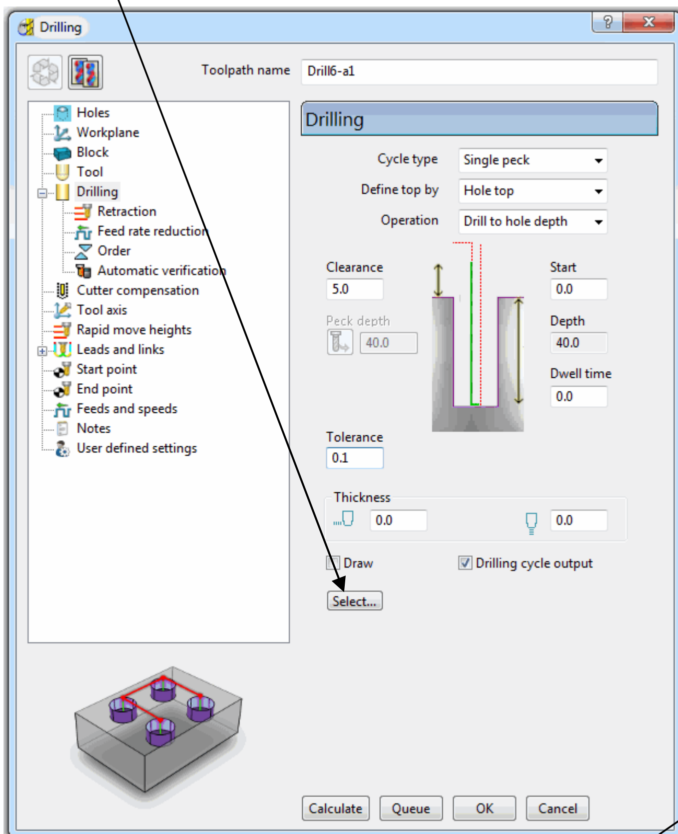
- Select the **Leads and Links** form  and for **Lead In** - ***1st Choice*** select **Pocket Centre** followed by **Copy to Lead Out** before selecting **Apply**.

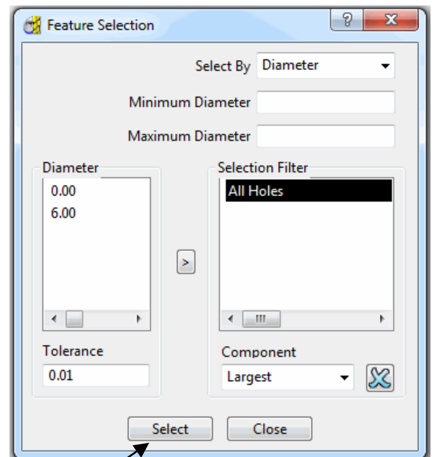




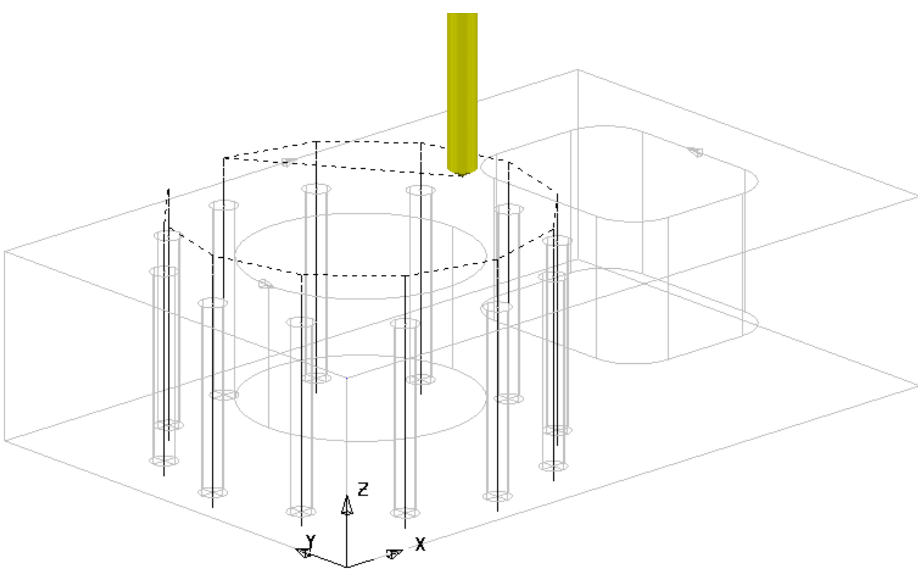
- Define a **Dia 6 Drill** - **Length 50** and named **Drill6**.

- Open the **Toolpath Strategies** form and select **Drilling**.
- In the form select **Drilling** to open the form (below) and then click on the **Select** tab to open the **Feature Selection** form.



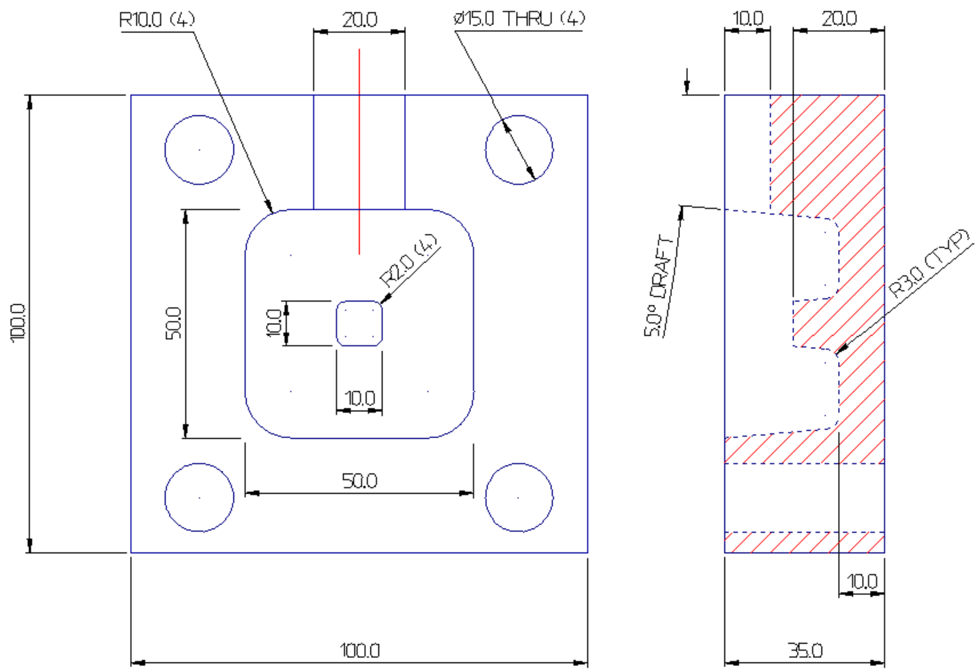


- In the **Feature Selection** form the **Select** tab is clicked to include all of the defined ***holes*** in the **Feature Set**.
- Enter other data into the **Drilling** form exactly as shown above before selecting **Calculate**.
- Close the **Drilling** and **Hole Selection** forms.



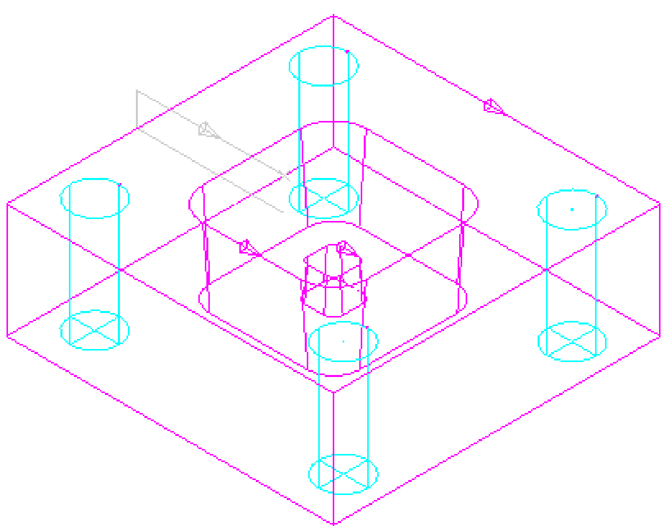
2D Machining Exercise

For this exercise the following ***2D drawing*** will be used.



- **Delete All** and **Reset forms**
- **Import** the **2D *wireframe* model** (as shown above):-
  C:\users\training\Powermill_Data\Models\2d_Wizard_Example.dgk

- Create appropriate **Feature Sets** to the above drawing dimensions.



1st Feature Set
The ***outer square*** and small tapered, ***central upstand*** are created as **Boss Features**.

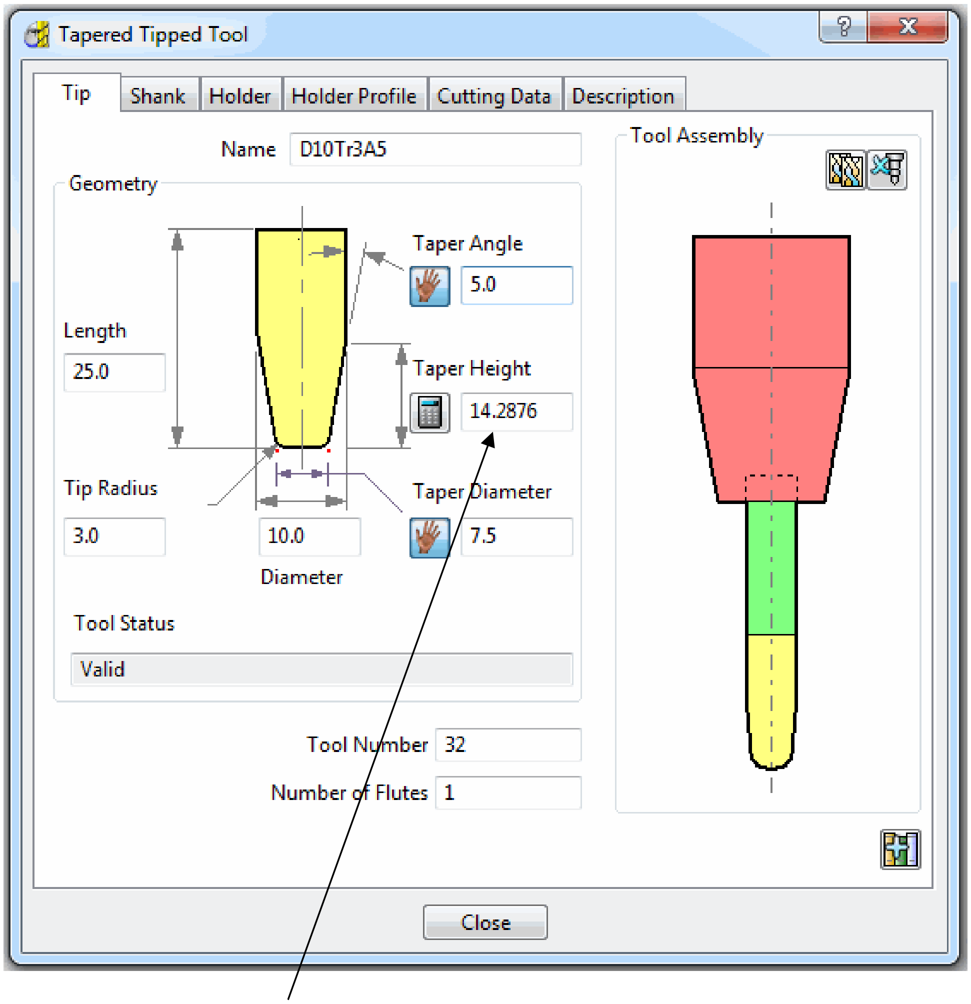
The **intermediate square** is created as a tapered **Pocket**.

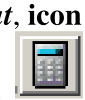
The four ***Dia 15 circles*** are created as **Hole Features**.

2nd Feature Set
The **Slot Feature** is best created in a second **Feature Set**.

- Create a **Dia10 *End Mill*** named **Em10** (***Roughing, Semi-finishing*** of ***main Pocket*** and **Finishing** of ***Slot Feature***).

- Create a **Dia 10** ***Tapered Tipped tool*** named **D10Tr3A5**, with **Tip Radius 3**, **Taper Angle 5Deg**, **Length 50**, **Taper Diameter 7.5** (***Final Finishing*** of tapered walls ***Main Pocket*** and ***small Boss***).

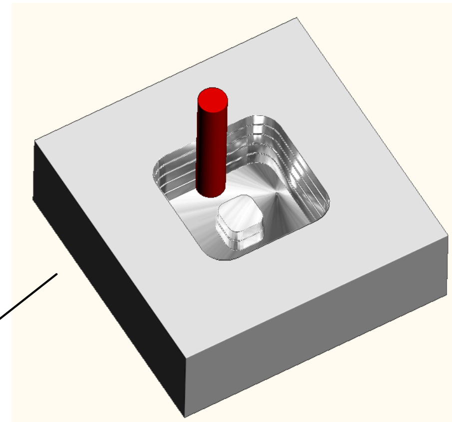


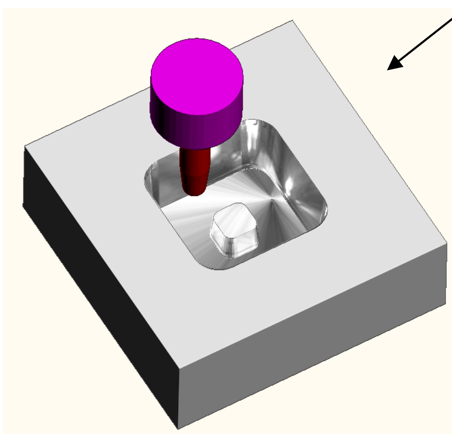
Note:- To calculate the **Taper Height** based on the **Taper Angle** and base **Taper Diameter** the ***Manual Input***, **icon**  adjacent to the **Taper Height** is **clicked** to apply the **Calculate** option  to this parameter.

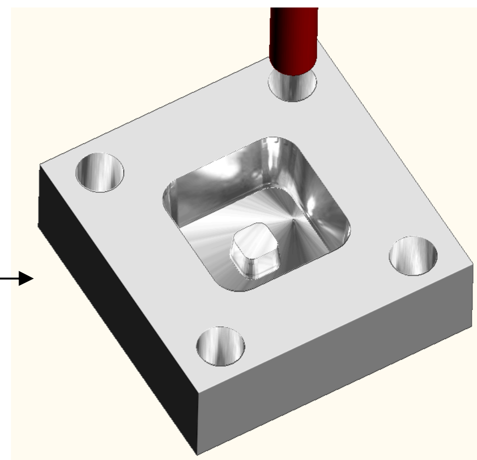
- Create a **Dia 15 Drill** named **Drill15**.
- Create suitable **2D machining strategies** to produce an end result similar to that shown in the following **ViewMill** illustrations.

Suggested Strategies:-

Main Block and Central Pocket
1/ EM10 - Feature Set Area Clear – Offset all
2/ D10Tr3A5 - Feature set Profile
3/ Drill15 - Drilling

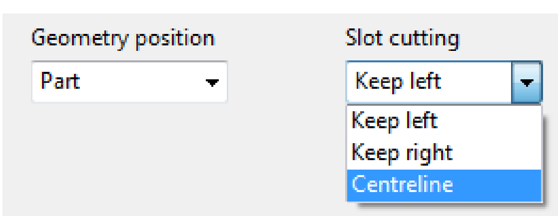




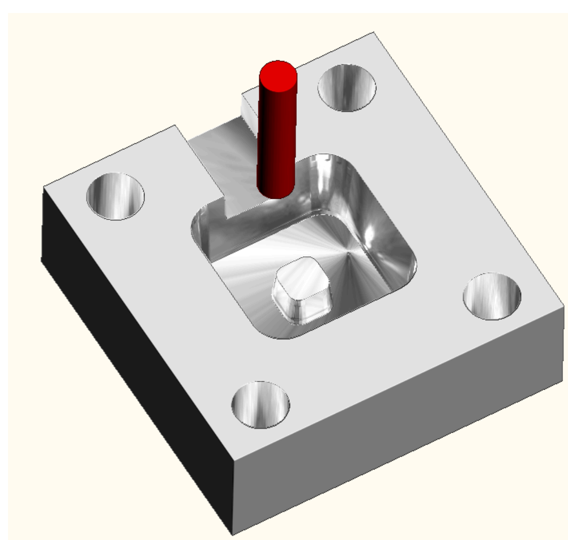


The **Slot** will be machined independently from the main pocket and for this reason it is better to define it in a separate **Feature Set**. Note; The Y dimension of the existing **Block** will have to be adjusted to fully, include the **Slot** Feature.

Slot
1/ EM10 - Feature Set Profile



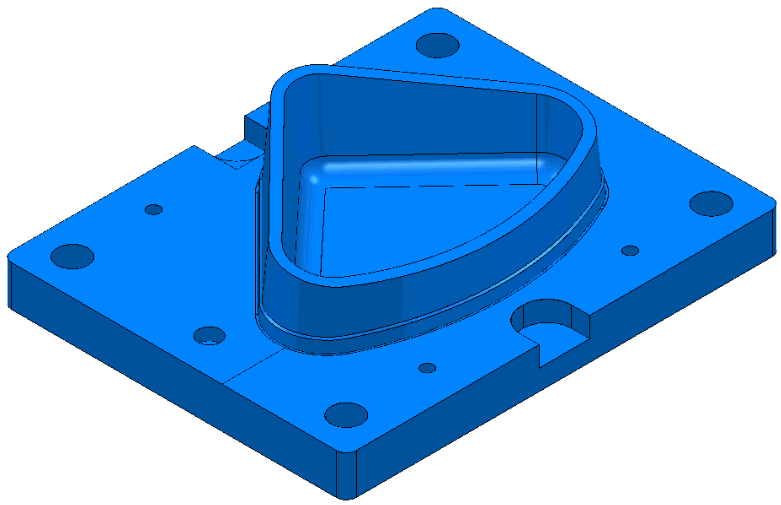
Width of Slot is achieved by creating ***multiple toolpaths*** that use the 3 different **Slot Cutting** options starting with **Centreline**.



Creating Features from Model holes

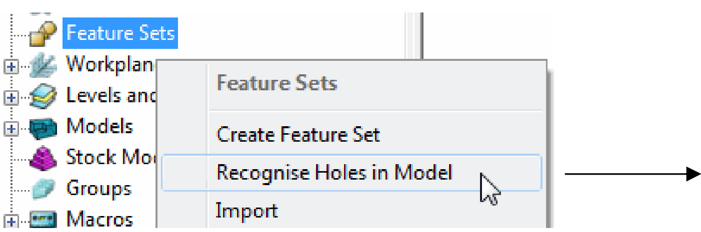
As well as being able to use imported ***2D geometry***, **Hole Features** can be automatically defined from selected **Holes** in the ***3D Surface or Solid*** **model**.

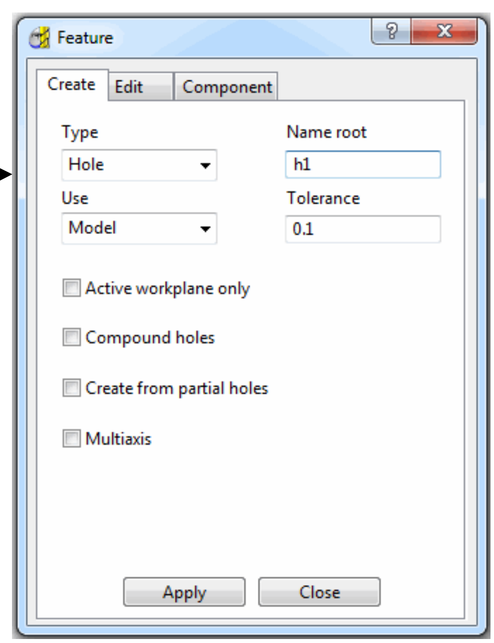
- **Delete All** and **Reset forms**.
- Select **File - Import Model:-**
  C:\users\training\PowerMILL_Data\Models\corner_bowl.dgk



- Make sure the **Block** is not defined at this stage, as the top of the **Hole *Features*** will be created at the nearest **Z** dimension (Max or Min) of the **Block**. This is likely to result in some holes being created the wrong way up (It is however possible to ***reverse*** a **Hole *Feature***).

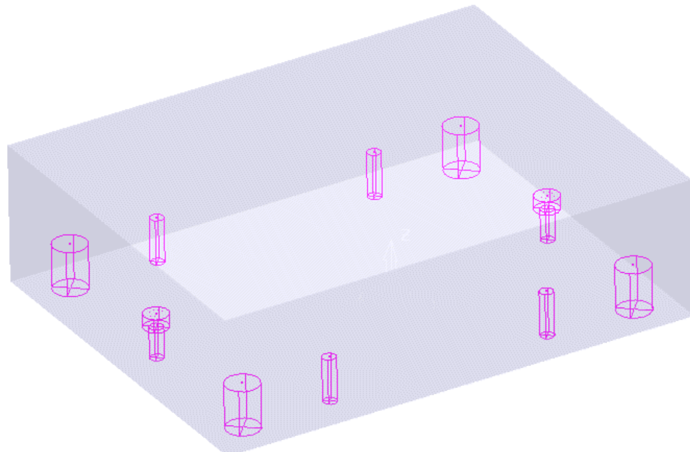
- Select the whole **model**.
- Right click over **Feature Sets** and select **Recognise Holes in Model**.





- Input data into the **Feature** form exactly as shown and select **Apply**.

- Calculate a **Block** defined by a **Box** around the **model**.
- Undraw the **model**.



A **Hole Feature** is created for each of the selected ***holes in the model*** (including the counter-bores).

- Define suitable tools, then drill and counter bore the holes using the **Drilling** options.

Hole Capping

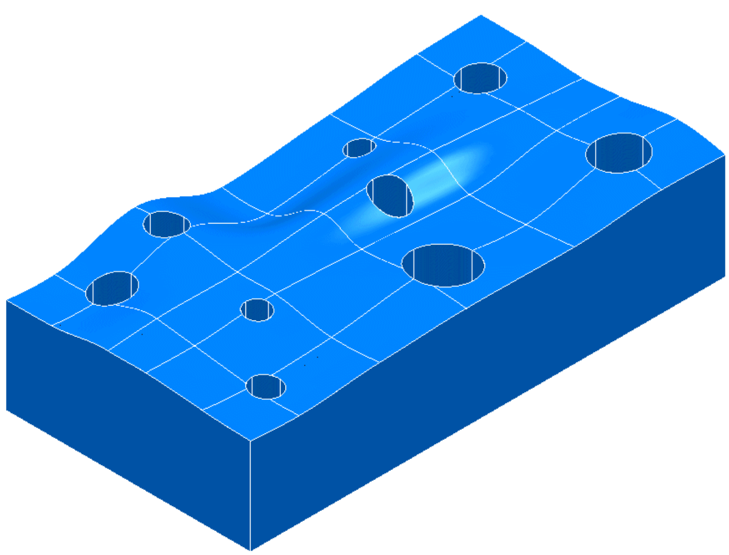
Hole Features can be used as the basis for capping holes in the model. If the intersection of the hole and the top face is non-planar, **PowerMILL** will try and cap the hole with a curved surface, matching the tangency of the surrounding surface. If it is not possible to create **Hole** features because the holes are not exact, circular sections then it will be necessary to follow the capping procedure in the **Edit Boundaries** chapter.

Example

The following example illustrates the process of ***capping*** cylindrical **Hole Features** defined from a **3D model**.

- **Delete All** and **Reset Forms**.
- **Import** the **model**:-

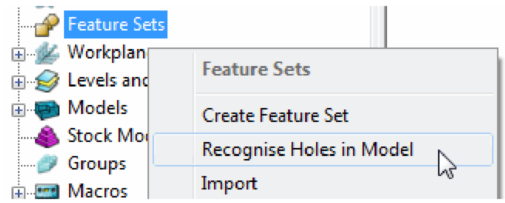
  C:\users\training\PowerMILL_Data\ Block_with_holes.dgk

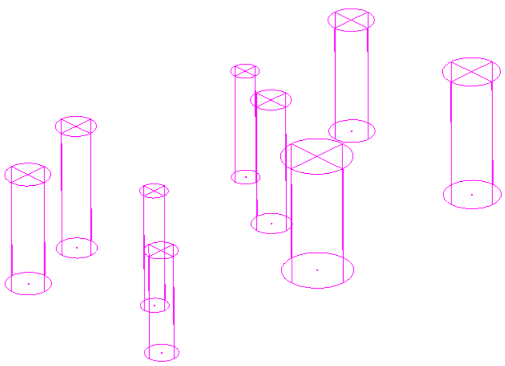


- Calculate a **Block** defined by a **Box** around the **model** (In this case the **Holes** will be created upside down).
- Select the whole **model** by holding down the left hand mouse button and dragging a box over it.

Next **Hole Features** will be created from the **model**.

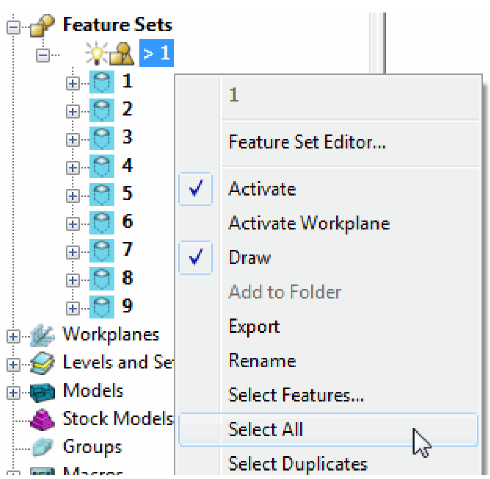
- Select **Recognise Holes in Model**.

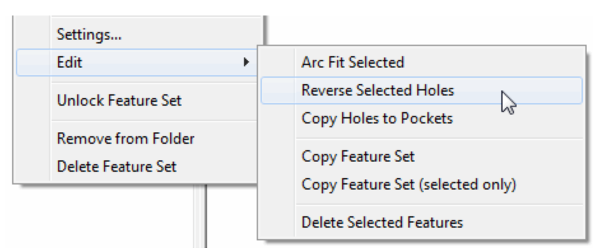




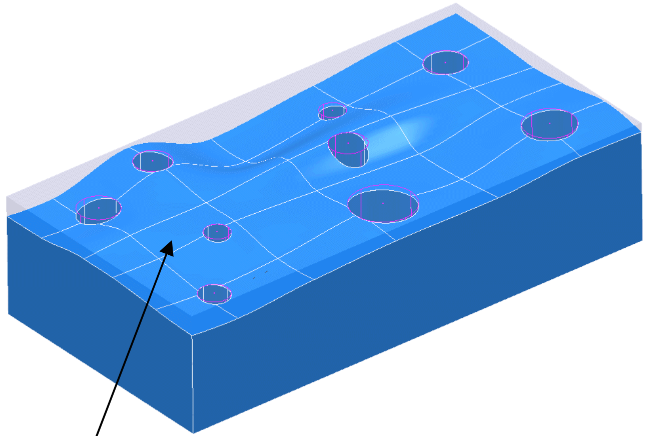
The **tops of the holes** are defined as the shortest distance from the **top** or **bottom** of the material **Block**.
If **Hole Features** are created to a different orientation from the natural **Z Axis**, They are assigned to a new **Feature Set**. A new **Workplane** is also created with the **Z Axis** aligned to the new **Feature Set**, **Hole** direction.

In this case the holes are created upside down to illustrate the **Reverse Holes** editing option.



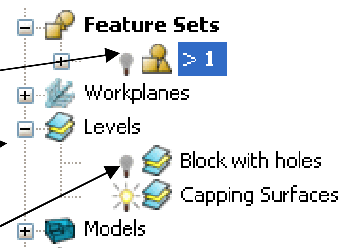


- Right click on **Feature Set 1** and in the menu click **Select All** followed by **Edit - Reverse Selected Holes**.



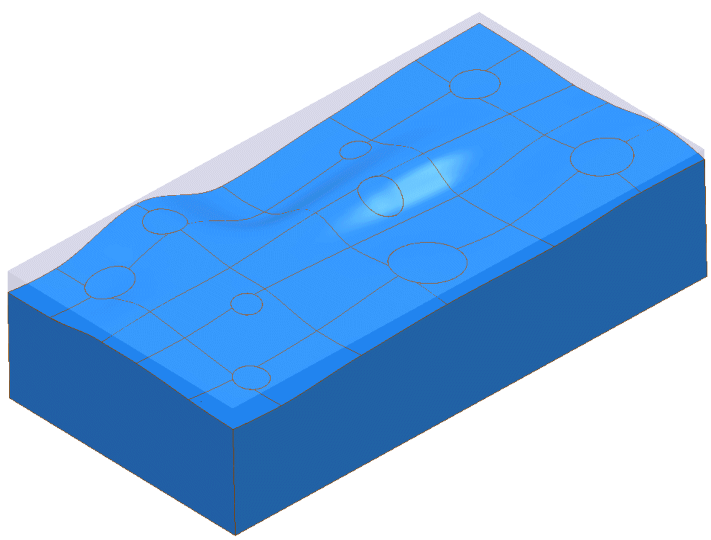
- Finally, select the **top surface**, then in the **explorer**, right click over **Feature Set 1** and select the **Cap Holes** option.

The new cap **Surfaces** will be generated (Tangential to the selected **top surface**) in a new **model** called ***Capping Surfaces*** created along with a **new level** of the same name.



- Click the light bulb to undraw **Feature Set 1**.
- Expand **Levels** by clicking on the + symbol.
- Click the light bulb to undraw ***main surfaces***.

The new **capping surfaces** may be created inside out (***Surface Normals reversed***). If required they can be **selected** and from the local **surfaces** menu **Reverse Selected** can be applied.

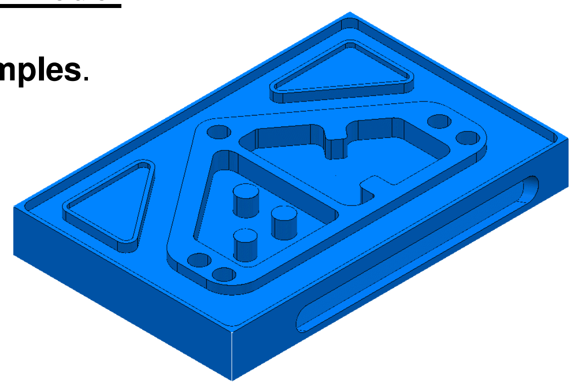


Note; the **Capping Surfaces** are automatically assigned to the new **Level** for easy selection.

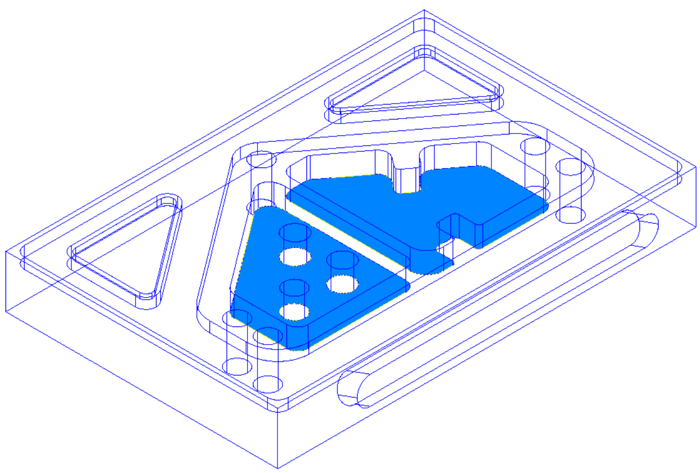
- Click on the ***light bulb*** to draw the **Block with holes** level back on.

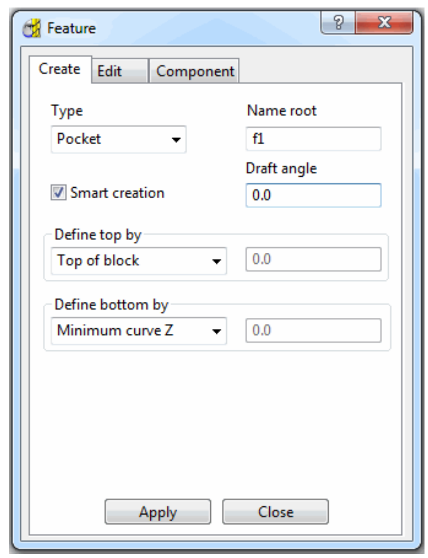
Creating Features from a 3D CAD Model

- **Delete All** and **Reset Forms**.
- **Import** - **2DExample.dgk** from **Examples**.

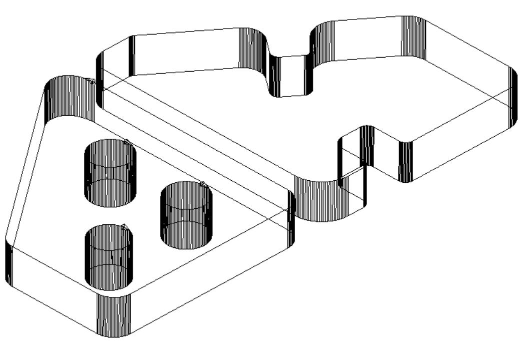


- ***Calculate*** a **Block** defined by a **Box** around the **model**.
- Select the **Two** surfaces shown (below).
- In the **explorer** right mouse click on **Feature Sets** and from the local menu select **Create Feature Set**.
- Rename the **Feature Set** as **Example2**.



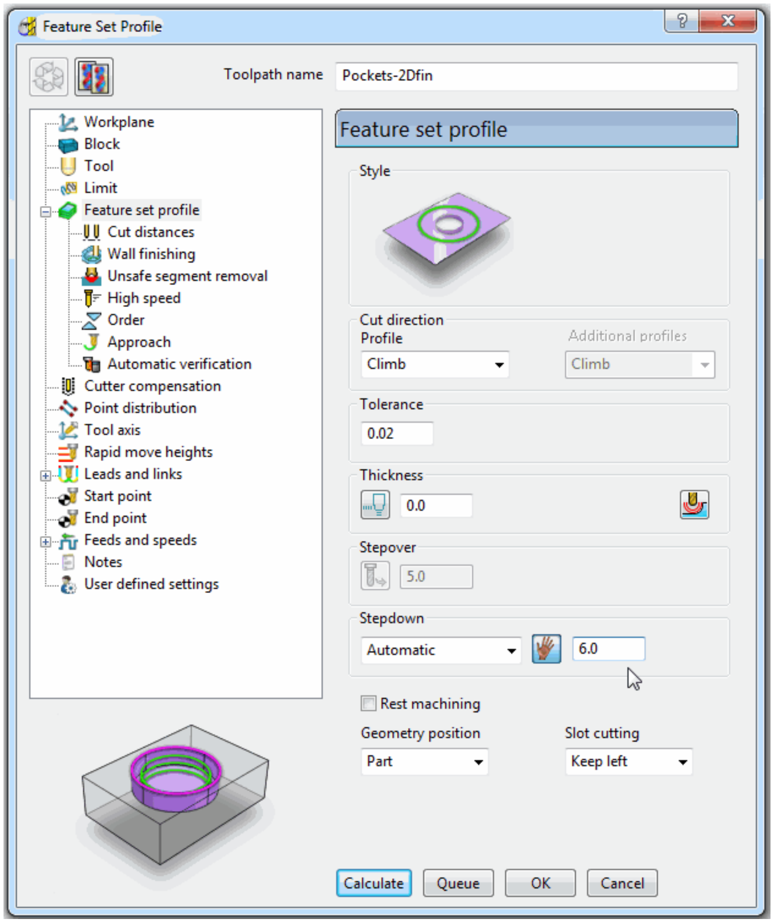


- In the **Feature** form input **Name root - F1** and select ***Type*** **Pocket**.
- Input ***Define top by*** **- Top of Block** and ***Define Bottom by*** **- Minimum curve Z** and **Apply** to create the **Features**.
- Create a **Dia 20 End Mill** named **EM20**.

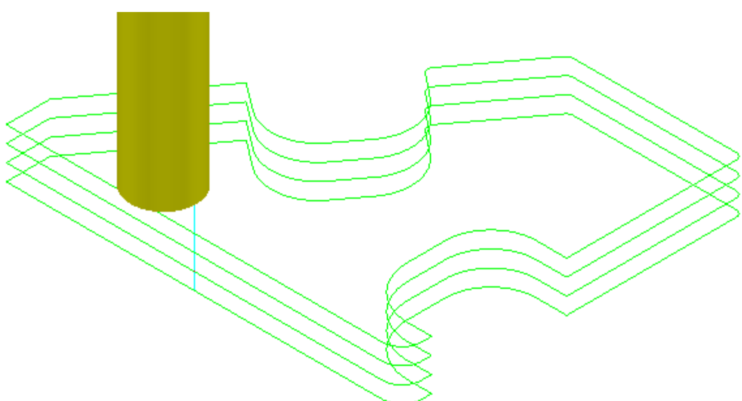


From the **2** selected **Surfaces** a total of **5 Features** are created (**2** large **Pockets** and **3** circular **Bosses**).

- Select the ***Feature*** that does not include the ***3 Boss Features***.
- From the **Strategy Selector** select the **2.5D Area Clearance** tab.
- Select **Feature Set Profile** and input data **exactly** as shown in the following form.



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



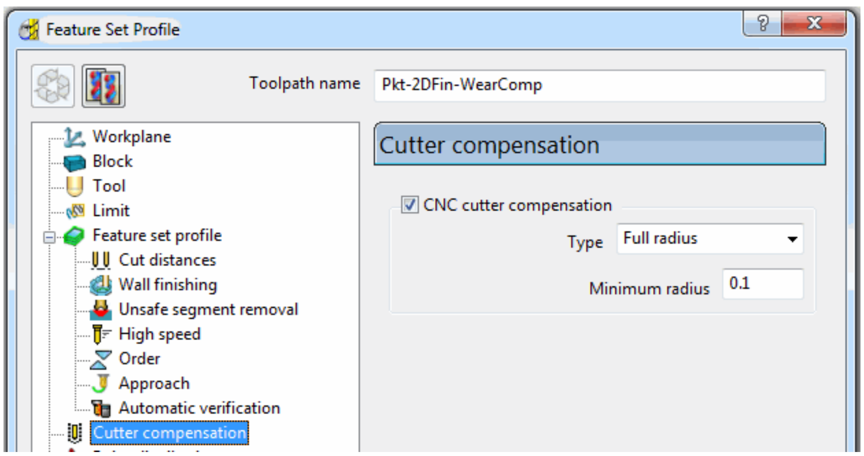
- **Save** the **Project** as:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\2Dtest

- <u>Do not close</u> the **Project** as it is to be continued in the next section.

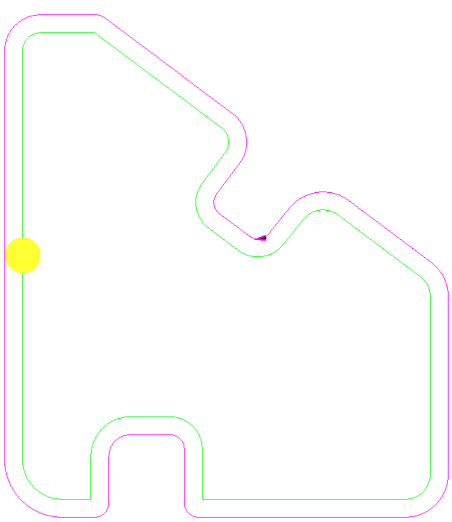
Area Clearance Cutter Compensation

There is an additional option for applying **Cutter Compensation** to a toolpath available from the **Expert** area of the **Area Clearance dialog**.

- **Activate** the ***Toolpath*** **Pocket-2DFin**
- Select **Settings** to open the toolpath form and then select the **make a Copy** option.
- **Rename** the ***Toolpath*** as **Pkk1-2DFin_FullRadComp**
- With the **Feature Set Profile** strategy form active select the **Cutter Compensation** page (as shown below)

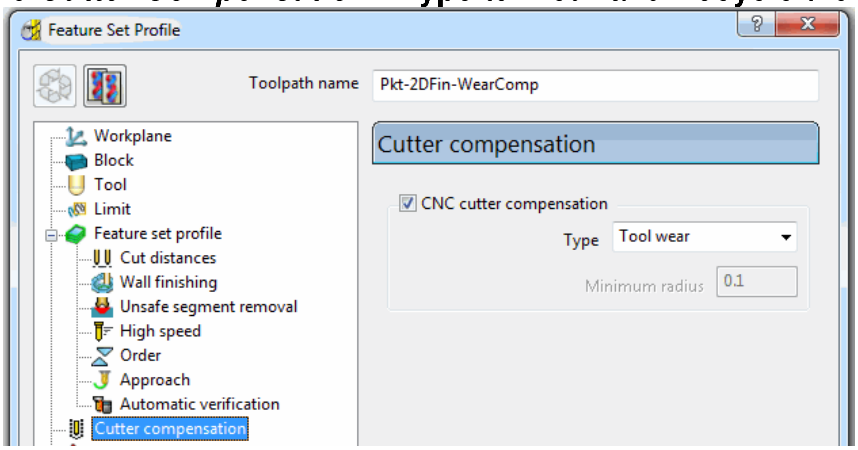


- Make sure the ***CNC cutter compensation*** box is ticked, and set ***Type* - Full radius** with ***Minimum radius*** **0.1**
- Select the **Leads and Links** page and select  to open the main **Leads and Links** form.
- Set **Lead In/Out** as **Horizontal Arc left** - **Radius 10** - **Angle 90**
- Set **Extensions - Inward/Outward - Extended Move 10**
- **Accept** the **Leads and Links** form.
- **Apply** the main **Feature Set Profile** form.



The effect of the **Cutter Compensation** is not visibly displayed in **PowerMILL** for this type of ***2D Strategy*** but is applied during post-processing for the ***ncdata*** output file.

- Select the **Settings** of the toolpath and make a **Copy**.
- Set the ***Cutter Compensation*** - **Type** to **Wear** and **Recycle** the toolpath.

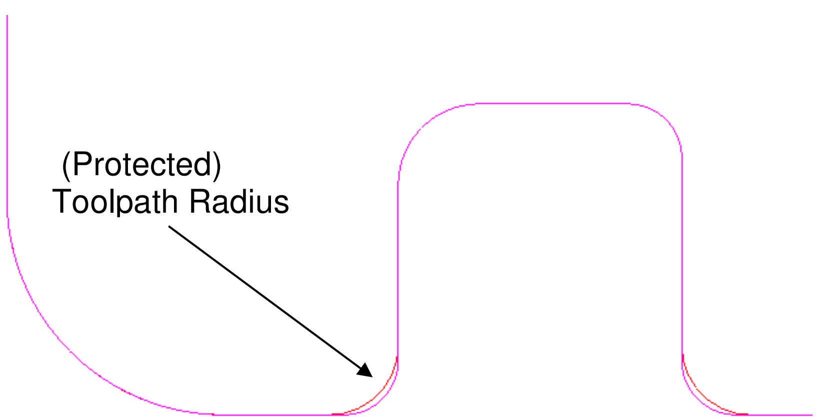


- Create a separate **NC Program** for each of the 3 strategies using a Heid option file

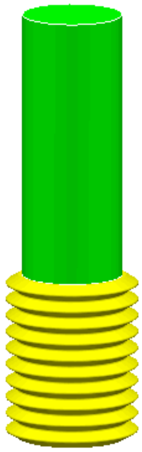
Note:-
The **X Y** positional cutting moves for the **Full Radius** compensation are dimensionally on the actual **2D Feature** form.
The **X Y** positional cutting moves for the **Tool wear** compensation, dimensionally relate to the tool centre position. The NCDate file includes an **RL** code to enable the operator to input small compensation values as required.

Protected Cutter Compensation

With ***Type*** **Wear** selected **PowerMILL** compensates for the nominal tool radius and the machine tool compensates for any difference. It corrects for the limitations of the **Off** option by ensuring that, in concave corners, a **Minimum Radius** field controls the size of the arc in corners after a tool radius offset.
Most machine tools will then be able to accurately represent the contour in concave corners. This allows **PowerMILL** to use the nominal tool size to check for collisions, but last-minute corrections for the physical tool size can be applied on the machine.
Protected compensation is naturally applied up to the **radius** of **Active Tool**. Provided the machinist does not use a compensation value (On the Machine tool controller) greater than the radius of the **Active Tool** use in **PowerMILL**, then potential gouging in the corners will be prevented.

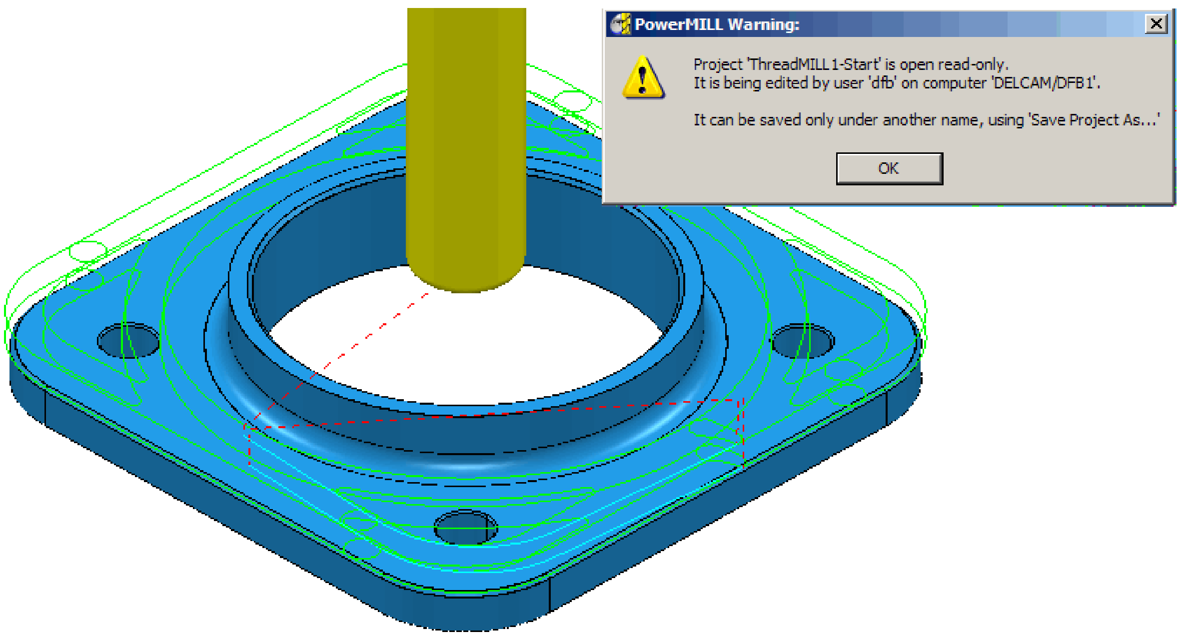


Thread Milling



Thread Milling is included as one of the **Drilling** options.
If the ***Cycle Type*** - **Thread Milling** option is selected, the tool plunges centrally to a specified depth, leads on to the sidewall, and then ***pitches upwards*** before leading off.
Within the options it is possible to apply a ***number of passes*** (stepping outwards by an **allowance** value) to avoid tool overload. It is also possible to choose either a **Right Hand** or **Left Hand** thread.

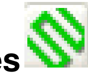
- Select **File ➔Delete All** and **Tools ➔ Reset forms**.
- **Open** the ***Project***:-
  C:\users\training\PowerMILL_Data\Projects\ThreadMILL1-Start

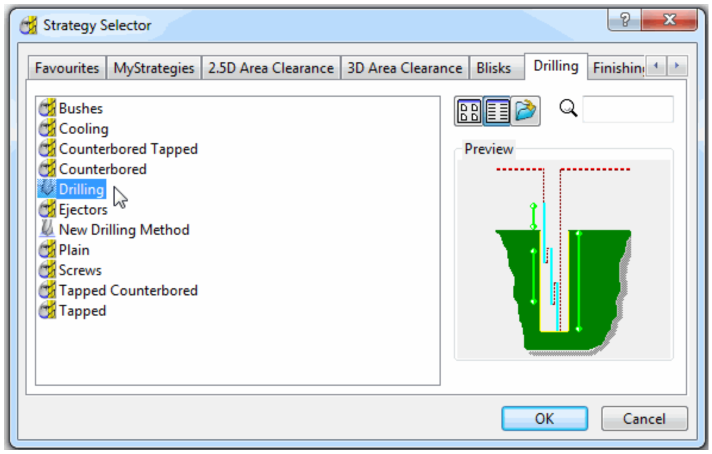


The imported **Project** basically contains a **Cad model**, a predefined **Hole Feature**, a **3D Offset Area Clearance** strategy using a **Tip Radius** cutter, and a **Thread Milling** tool. The original **Project** is **Locked** to prevent it from being altered, hence the first step is to **Save As** a separate **Project** locally with a different name.

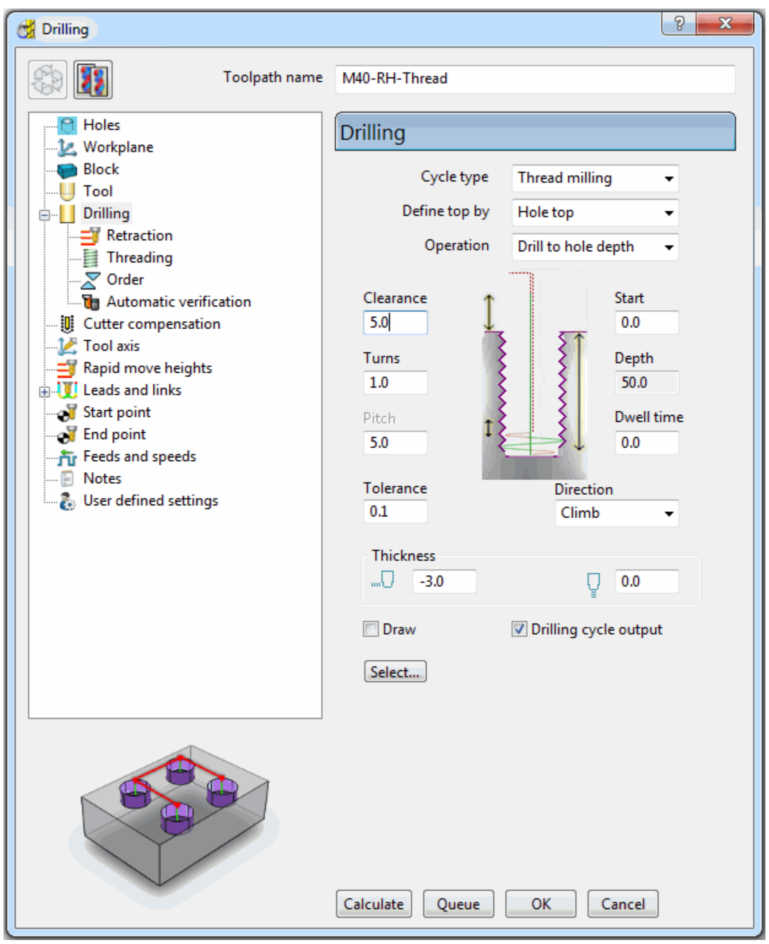
- Select **File** - **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\ThreadMILL-Example1

- **Activate** the ***Thread Milling*** **Tool** named **M40-Coarse-Pitch-5**.
- **Activate** the **Pocket *Feature*** around the centre wall of the component.

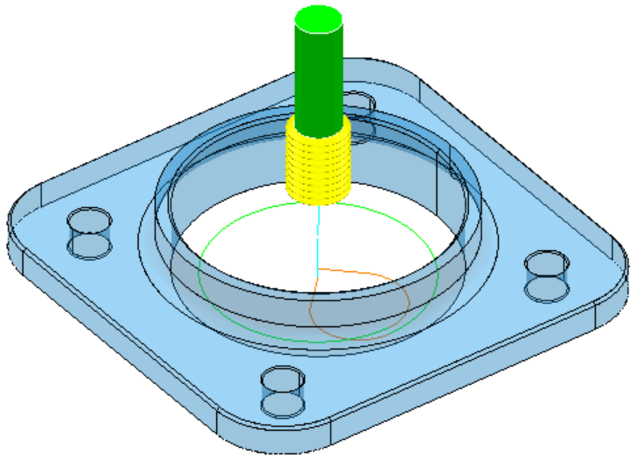
- Select **Toolpath Strategies**  and in the **Strategy Selector** form select **Drilling**.



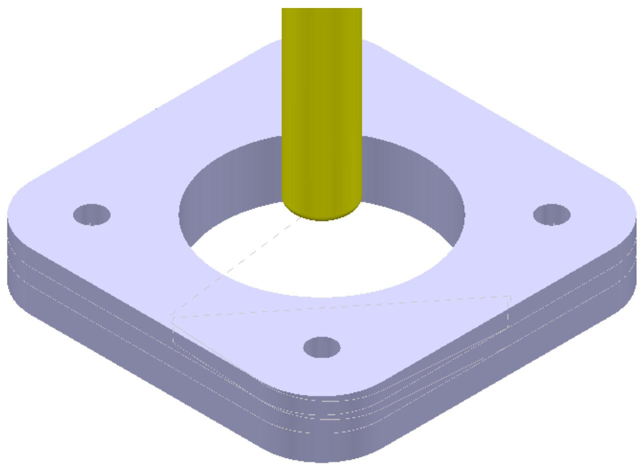
- In The **Drilling** form set the **Cycle Type** to **Thread Milling**, **Radial Thickness -3**, **Pitch 5**, ***Name*** **M40-RH-Thread**, leaving all other parameters as default (as shown below).



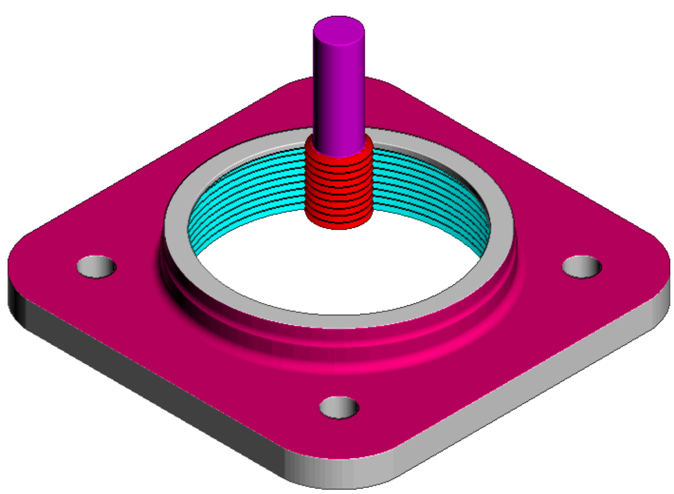
- **Calculate** the form to produce the ***Thread Mill*** **toolpath** and then **Cancel**.



- Create a new **Block** using ***Defined by*** **Boundary** along with ***Type*** **model** and **Calculate** to the exact component dimensions.



- Right click on the **toolpath D50t6-RGH-a1** (Do not Activate it) and in the local menu select **Simulate from Start**.
- Open a new **ViewMILL** session *and run a* **simulation** of both **toolpaths.** (Do not Activate either toolpath, so that the current **Block** definition is maintained).



PRO - 2D Machining Options

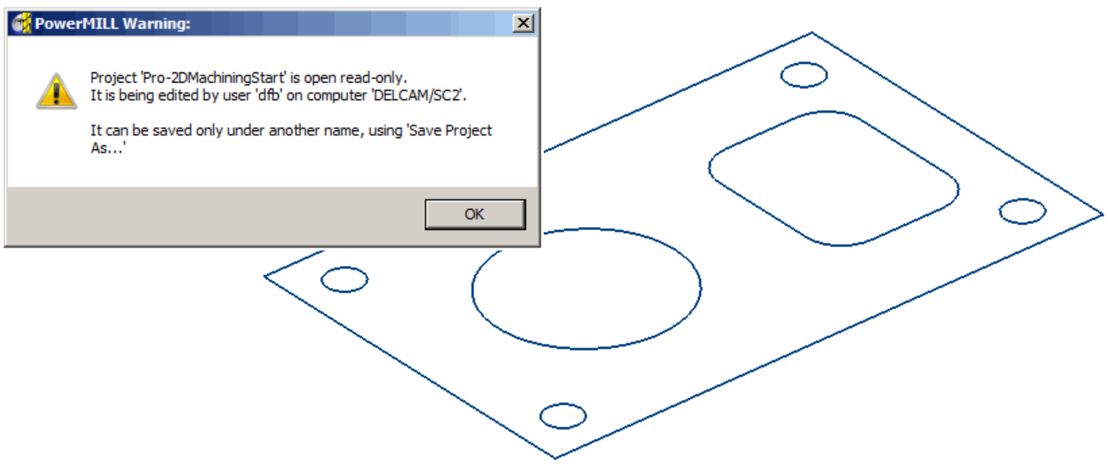
Introduction

PowerMILL Pro contains ***4 new strategies*** designed for more efficient **2D Machining** applications. These include:-

1. **Face Milling**.
2. **2D Curve Area Clearance**.
3. **2D Curve Profile**.
4. **Chamfer Milling**.

2D Curve Machining Example 1

- Select **File ➔Delete All** and **Tools ➔ Reset forms**.
- **Open** the ***Project***:-
  C:\users\training\PowerMILL_Data\Projects\Pro-2DMachiningStart

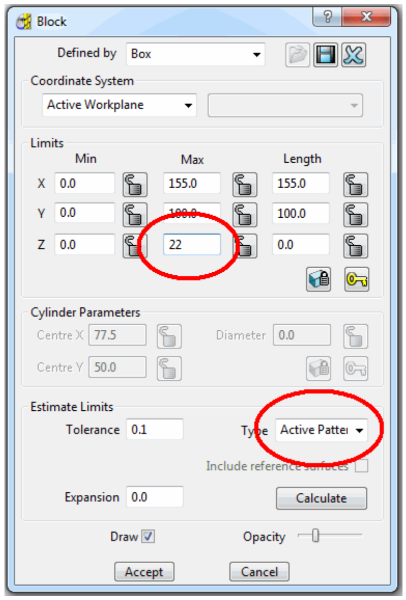


The imported **Project** contains **4 Tools** and **3 Patterns** defining the **2D geometry** to be used in the **2D Machining** options. The **Project** is **Locked** to prevent it from being altered hence the first step is to **Save As** a separate **Project** locally with a different name.

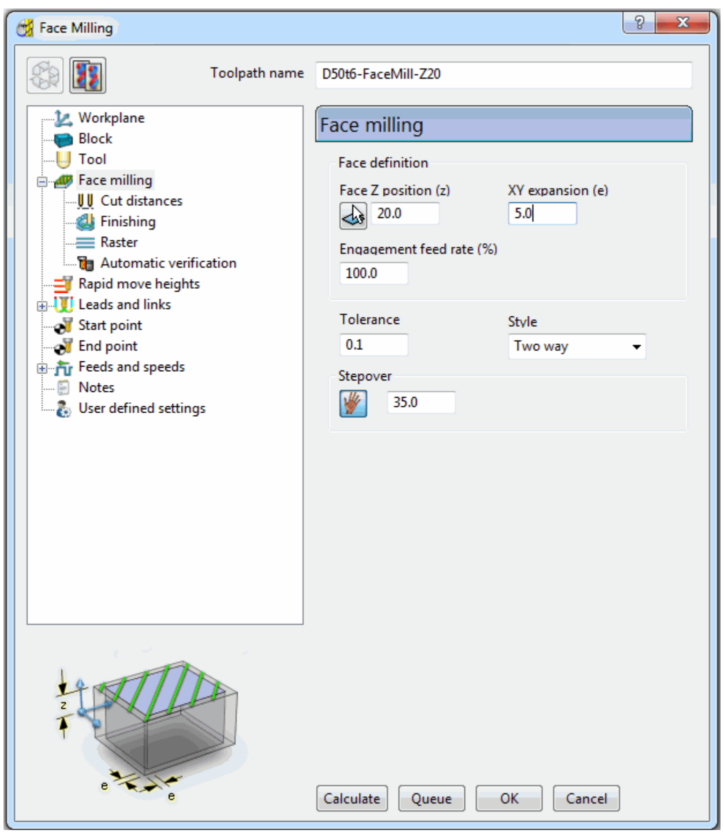
- Select **File** - **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\PRO-2DMachining

- **Activate** the **Pattern** named **CurveAll**.
- In the **Block** form, set ***Defined by*** - **Box** followed by ***Type*** - **Active Pattern** and select **Calculate** (Do not close the form).

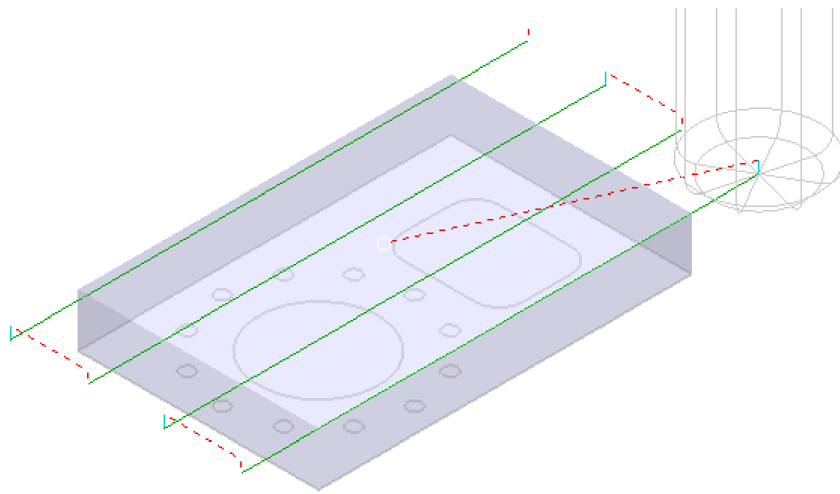
- In the ***Maximum Z Coordinate box*** input **22** before selecting **Accept** to close the form.



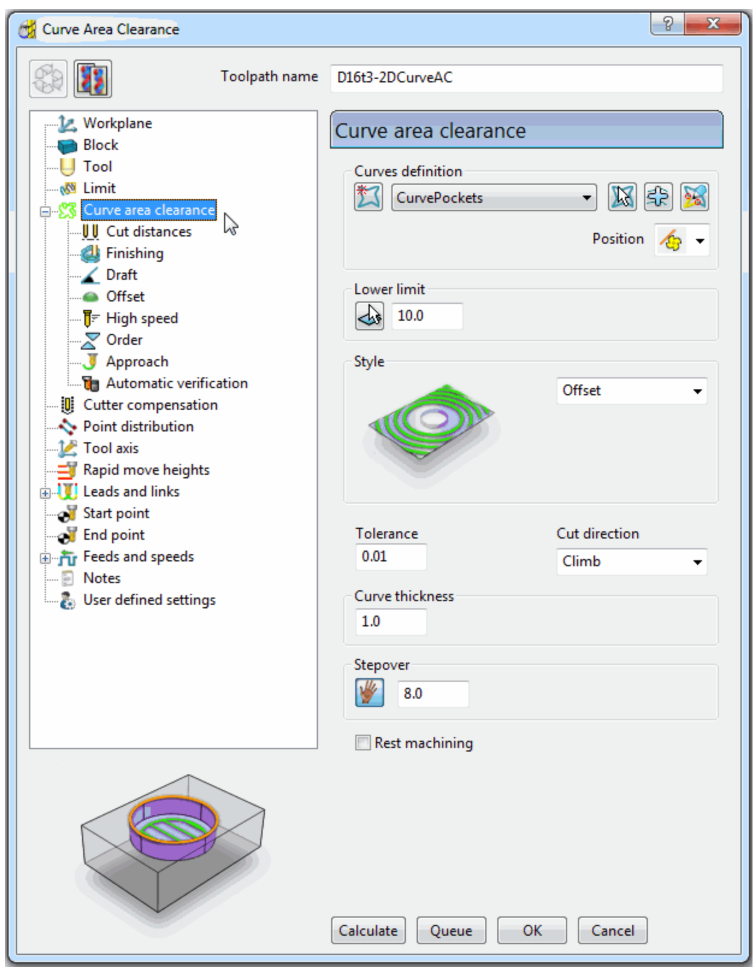
- In **Rapid Move Heights** click on **Reset to Safe Heights**.
- **Activate** the **Face Milling** tool named **D50T6**.
- From the **2.5D Area Clearance** options select **Face Milling** and enter data into the form exactly as shown below.



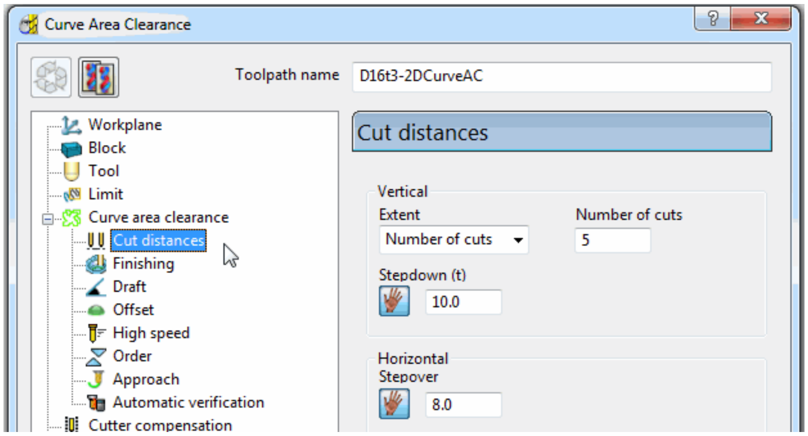
- Select **Calculate** to process the toolpath.

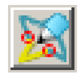


- **Activate** the **Tip Radius** tool named **D16T3**.
- From the **2.5D Area Clearance** options select **2D Curve Area Clearance** and enter data into the form exactly as shown below.

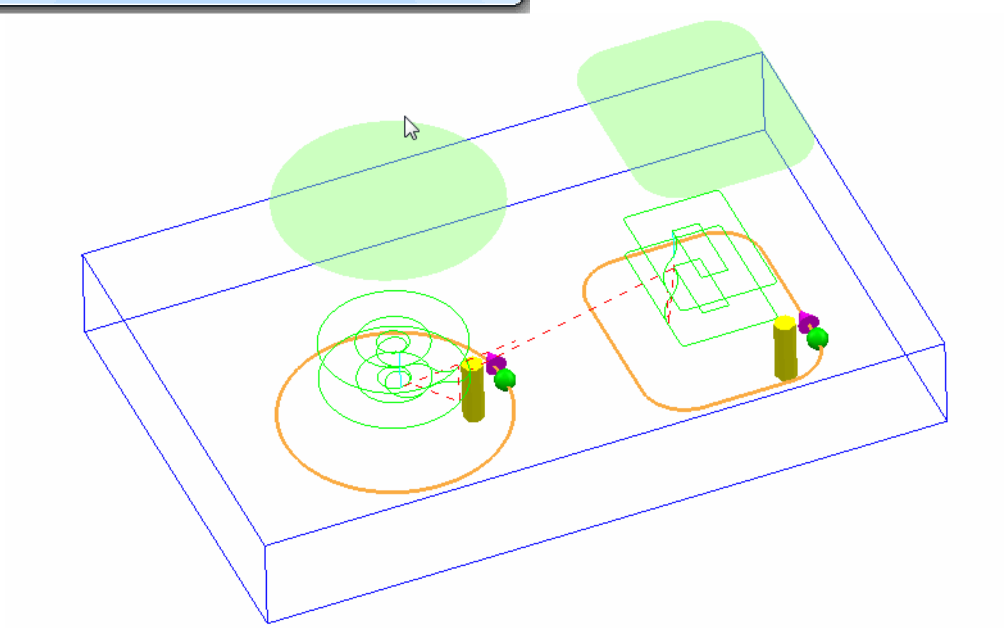


- Select the **Cut distances** page and input **Stepdown 10**.



- From the main **Curve Area Clearance** page, **Curve** options, select ***Interactively modify machinable sections***. 

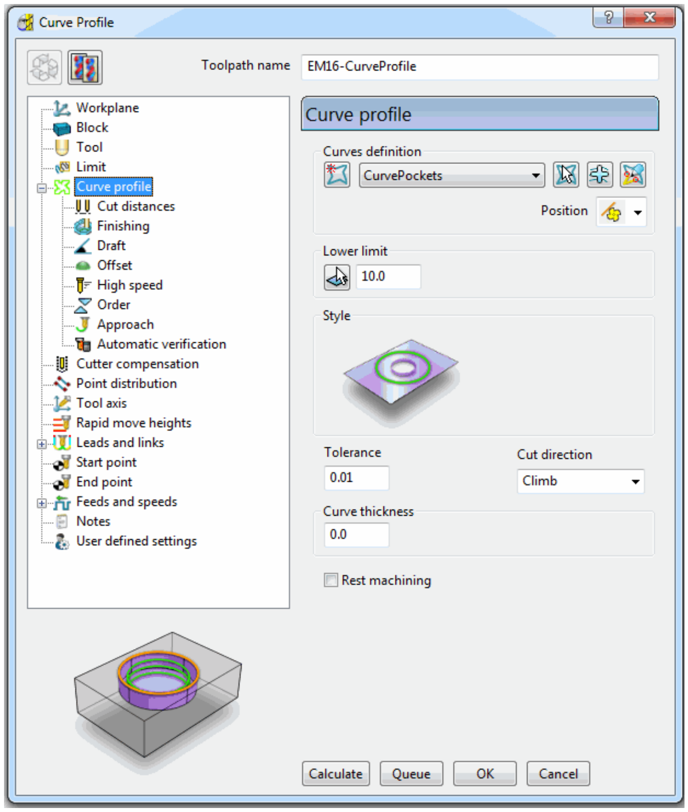
These options enable the user to have full control of direction of cut and the areas to be machined.



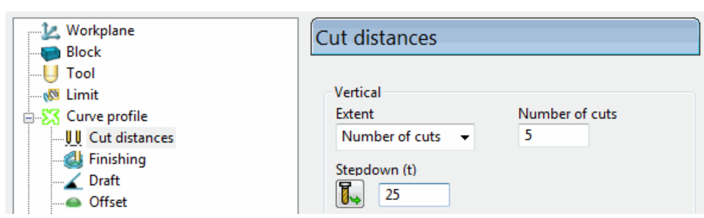
Note: While the **Curve Area Clearance** form is open, the areas to be machined are displayed as a shaded preview.

- Select the **Green Tick** to exit the **Edit Machining Sections** toolbar.
- Select **Calculate** to process the **toolpath** and then **Cancel** the form.

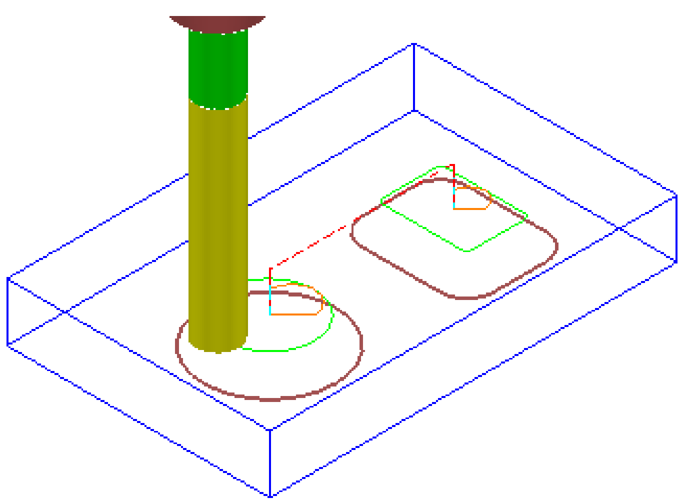
- **Activate** the **End Mill** named **EM16**.
- From the **2.5D Area Clearance** options select **2D Curve Profile** and enter data into the form exactly as shown below.



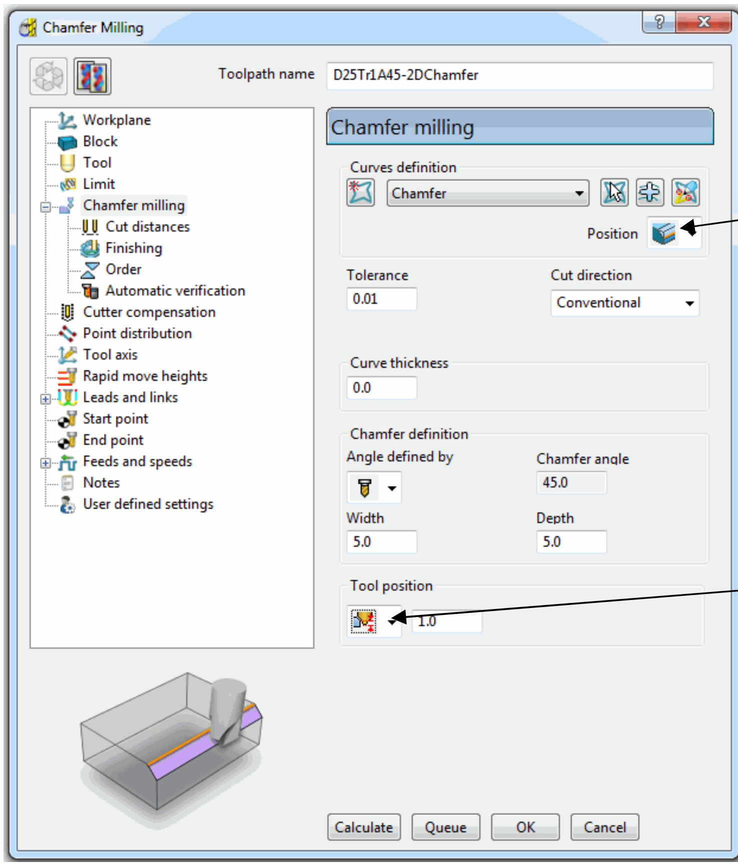
- Select the **Cut distances** page and input **Stepdown 25**.

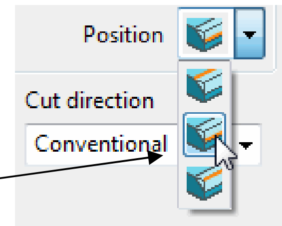


- Select **Calculate** to process the **toolpath** and then **Cancel** the form.

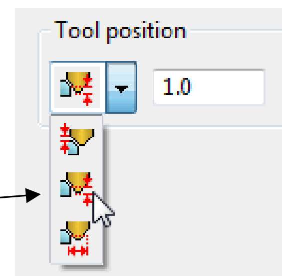


- **Activate** the **End Mill** named **D25TrA45**.
- From the **2.5D Area Clearance** options select **Chamfer Milling** and enter data into the form exactly as shown below.



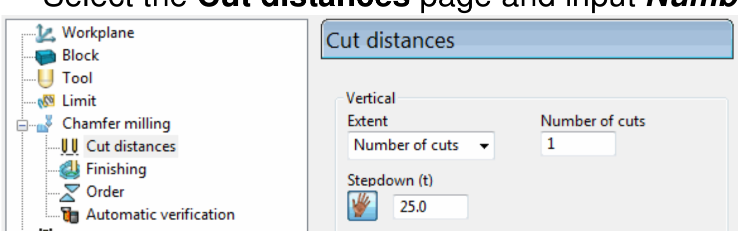


Set **Curve** position to **Bottom**

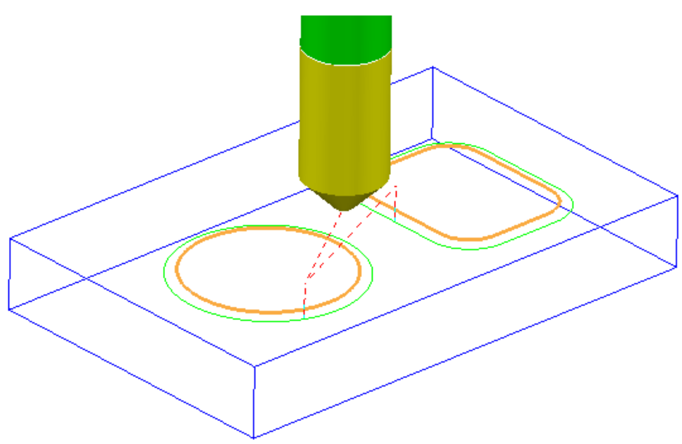


Set **Bottom axial depth**

- Select the **Cut distances** page and input ***Number of cuts*** **1**.



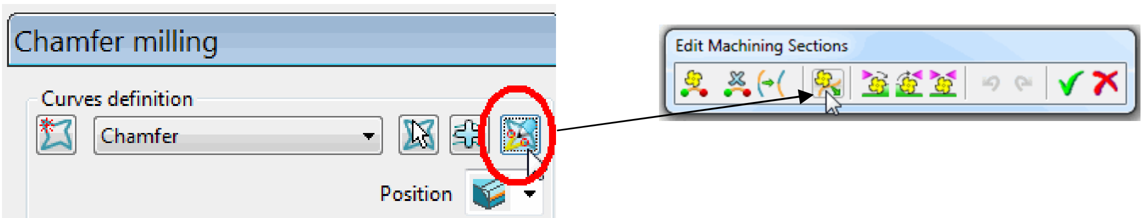
- Select **Calculate** to process the toolpath (Do not close the form).



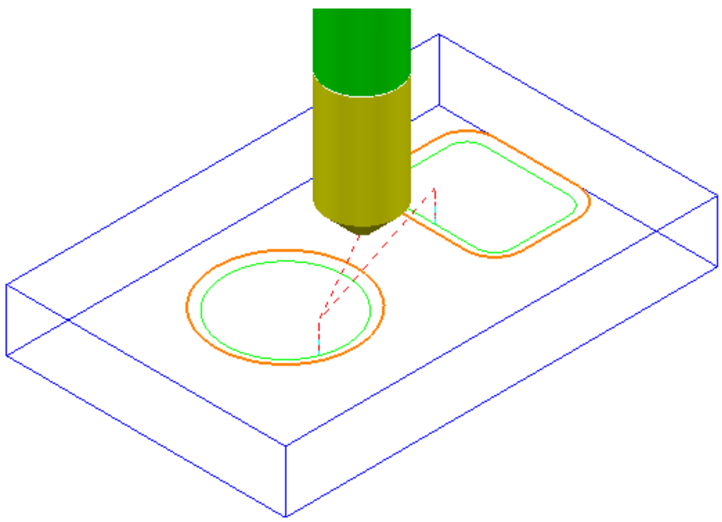
The resultant **Chamfer Milling** strategy has appeared on the wrong side of the **Pattern**.

The **toolpath** needs to be ***recycled*** with the **Chamfer Milling** tool tracks changed to appear on the inside of the **Pattern** segments.

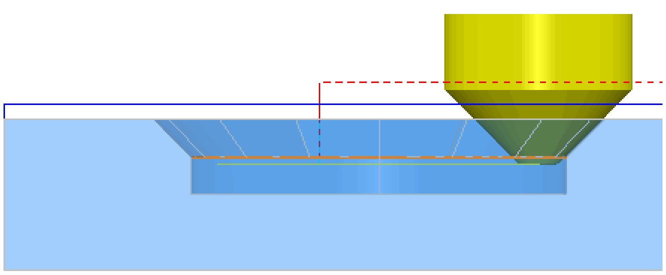
- In the **Chamfer Milling** form, select the ***Recycle*** icon to enable changes to be made to the above strategy.
- Select the ***Interactively modify machinable sections*** icon to access the ***Edit Machinable Sections*** toolbar.
- Select the ***Reverse machining side*** option, followed by the ***green tick*** to accept the changes and close the **toolbar**.

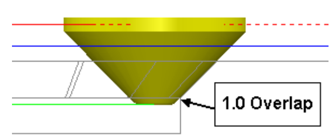


- In the main **Chamfer Milling** form, Select **Calculate** to process the toolpath (Do not close the form).



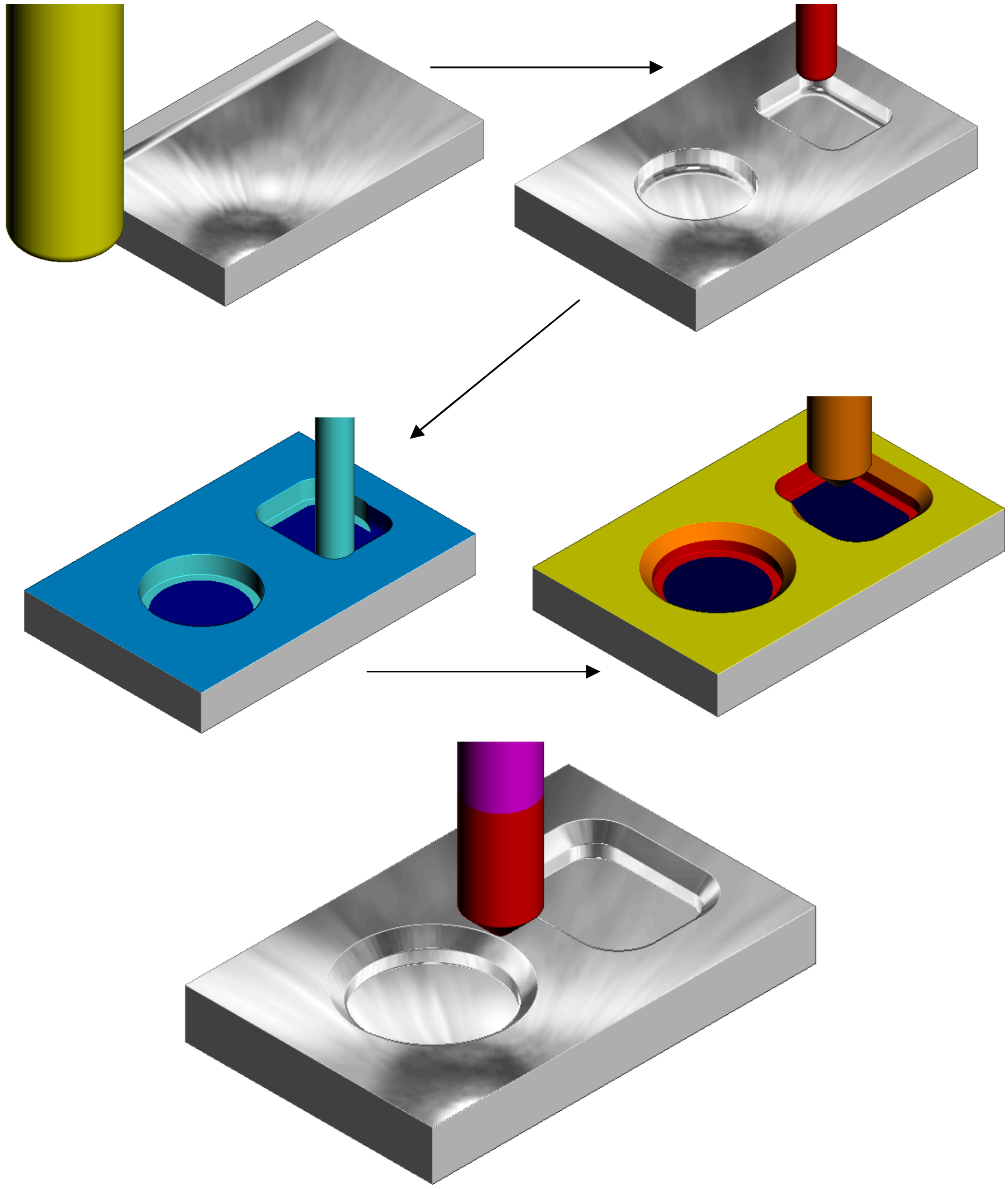
The **Chamfer Milling** options selected result in the angled part of the tool overlapping the base of the Chamfer by **1mm**.





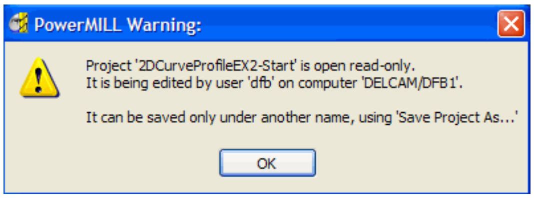
A **surface** model of the finished component has been imported to provide a visual check of the **Chamfer Machining**. The ***1mm tool overlap*** at the base of the chamfer is clearly visible in the above illustrations.

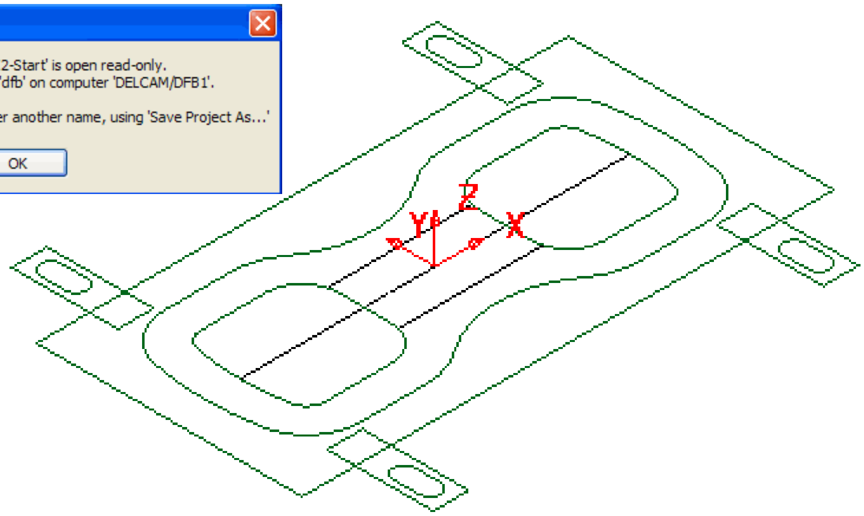
- Run a **ViewMILL *simulation*** of the whole machining process.



2D Curve Machining Example 2 – Inc. Clamp Avoidance

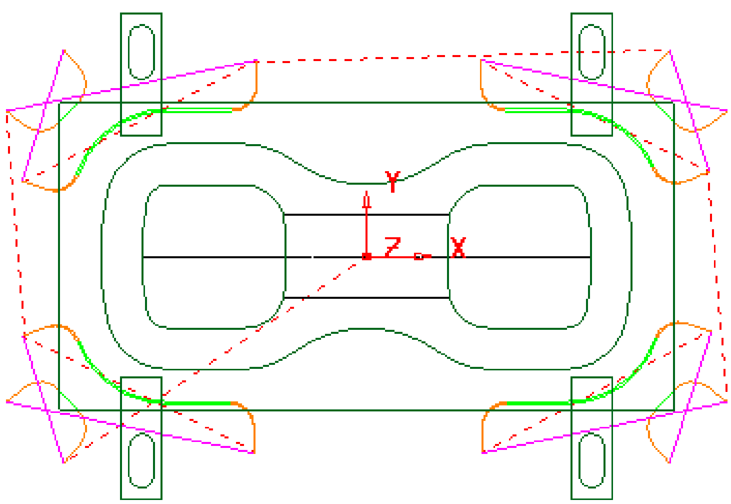
- Select **File ➔Delete All** and **Tools ➔ Reset forms**.
- **Open** the ***Project***:-
  C:\users\training\PowerMILL_Data\Projects\2DCurveProfileEX2-Start





The imported **Project** also contains **a Dia16 End Mill** and a **Pattern** defining the **2D geometry** used in the existing **2D Machining** Strategy. The **Project** is **Locked** to prevent it from being modified, hence the first step is to **Save As** a separate **Project** locally with a different name.

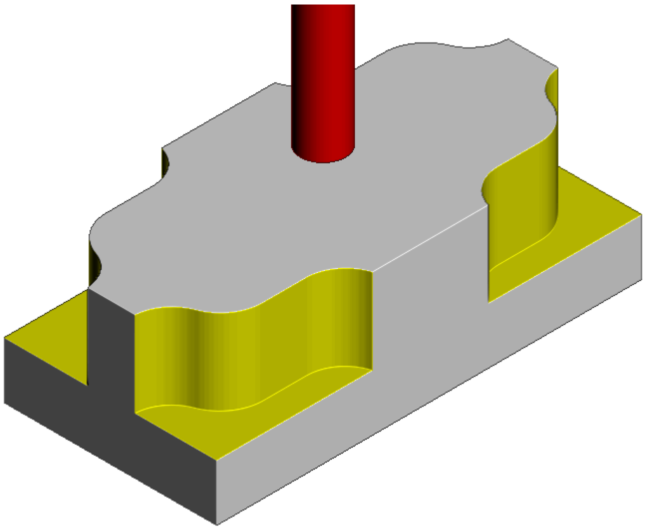
- Select **File** - **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\2DCurveProfileEX2



- In the **Viewing** toolbar ensure that ***Wireframe*** view is on.

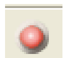
The existing preliminary, **Toolpath** machines away the area where the clamps are located ready for the remaining strategies.

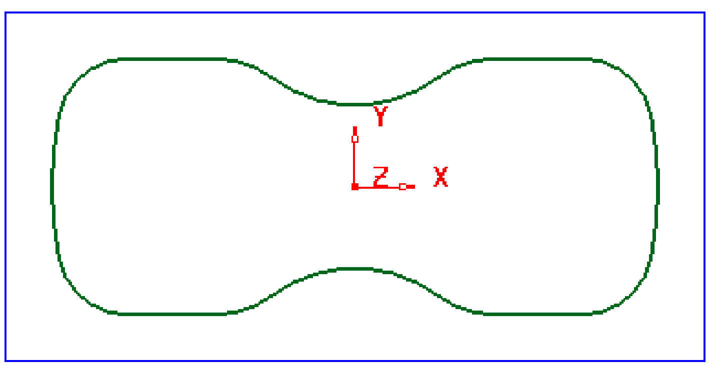
- Select an **Iso1 view** and perform a **ViewMILL simulation** on the existing **toolpath**.



The existing preliminary, toolpath (**MC-ClampArea-1**) requires the 4 clamps to be positioned midway along the sides and ends.

Once this machining operation has been performed, the clamps are then moved to the same position as defined by the **wireframe Model**.

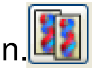
- Select the **No Image**  option in the **ViewMILL** toolbar and disconnect **ViewMILL**  from the simulation process (Toggle the ***Sphere*** from **Green** to **Red**)
- Select a **View from top (Z)**. 
- Undraw all entities except the **Block** and the **Pattern** (***OuterForm***).

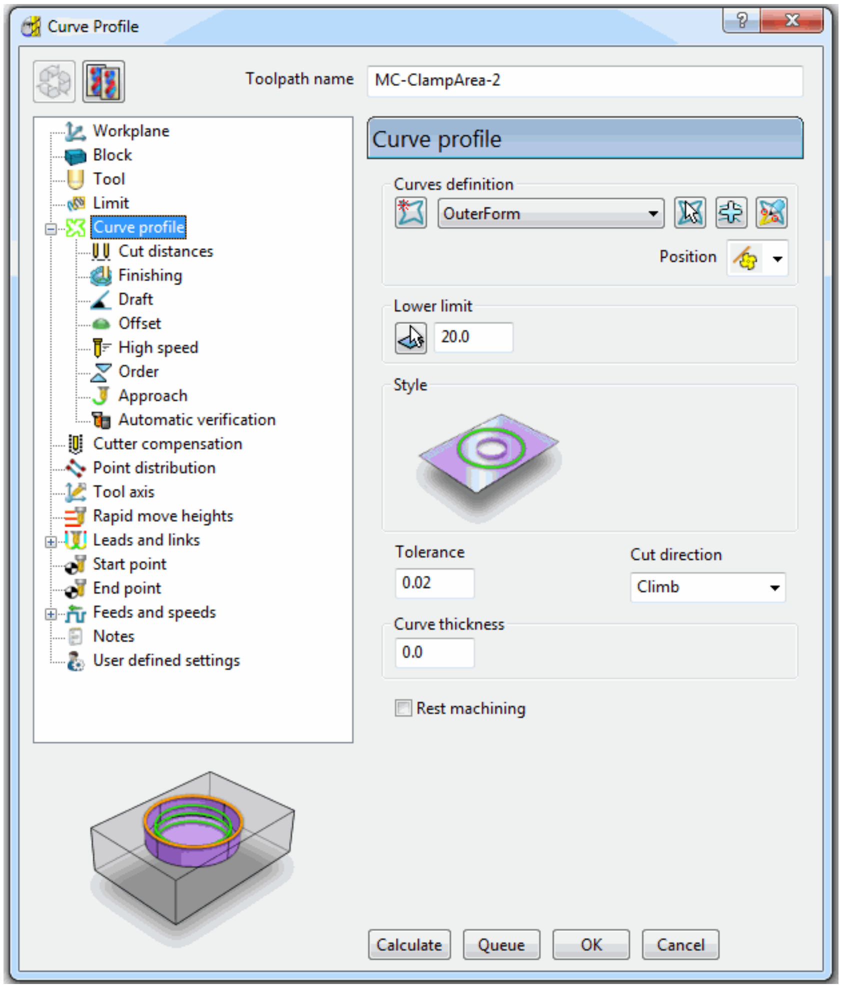


The above **Pattern** ***segment*** was created by using **Insert - Model** with the required **wireframe segment**.selected. The same **Pattern** will be used in the creation of the next **Toolpath**.

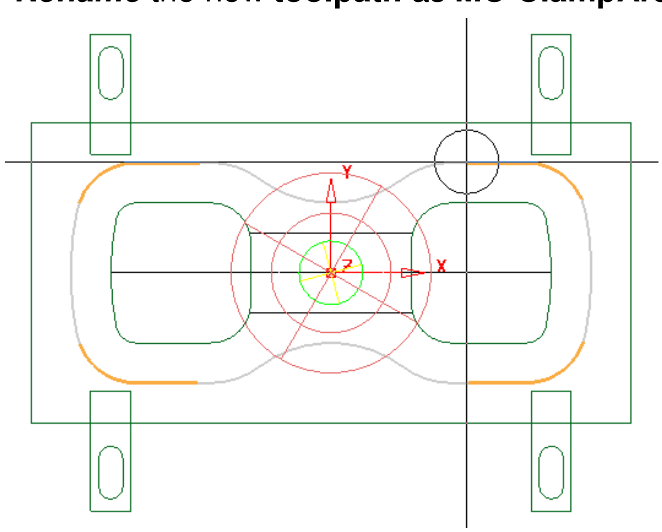
- From the ***main pulldown*** menus select:-
  Draw - Cursor - Cross Hair (***Ctrl H***) and **Draw - Cursor - Tool** (***Ctrl T***).

This will provide an essential visual aid while defining the parts of the toolpath **Pattern** to be machined. Note: Alternative ***Shortcut*** options are shown in brackets.

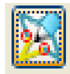
- With the ***toolpath*** (**MC-ClampArea-1**) ***activated*** select ***Settings*** (local menu) to open the existing **Curve Profile** form.
- At the top left corner of the form select the '***Make a Copy***' option. 

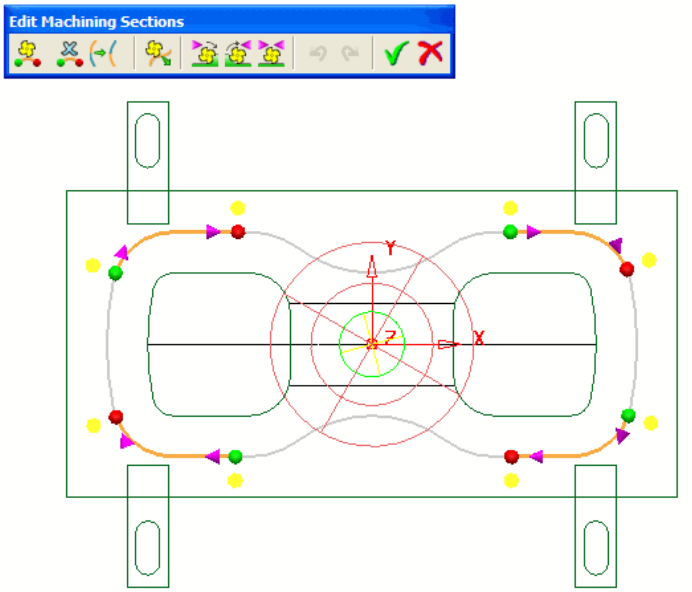


- ***Rename*** the new **toolpath** as **MC-ClampArea-2**.



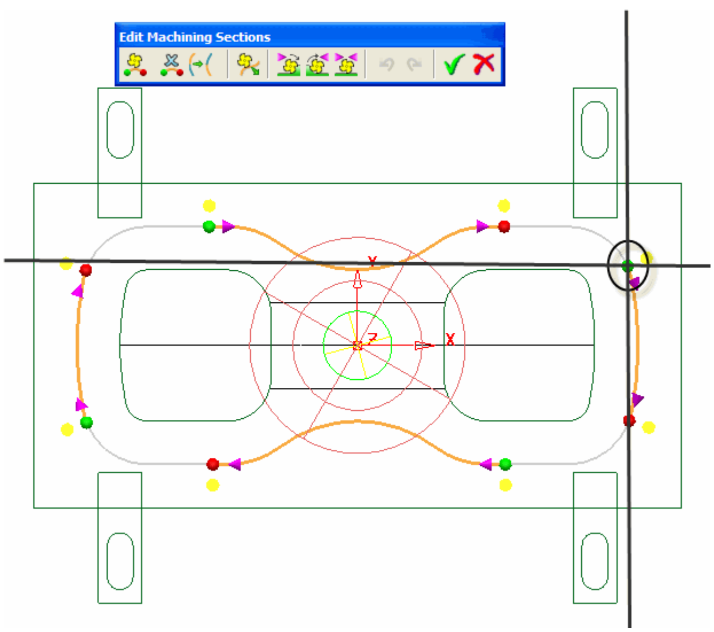
Note:-The **Tool Dia** and **Cross Hairs** are displayed on the ***cursor***. This will provide an essential visual aid while ***snapping*** new ***break points*** along the **Pattern**.

- Select  to open the **Curve** toolbar.



The sections of the **Pattern** included in the original toolpath are displayed, along with the user defined, ***Start*** (green) and ***End*** (red) points.

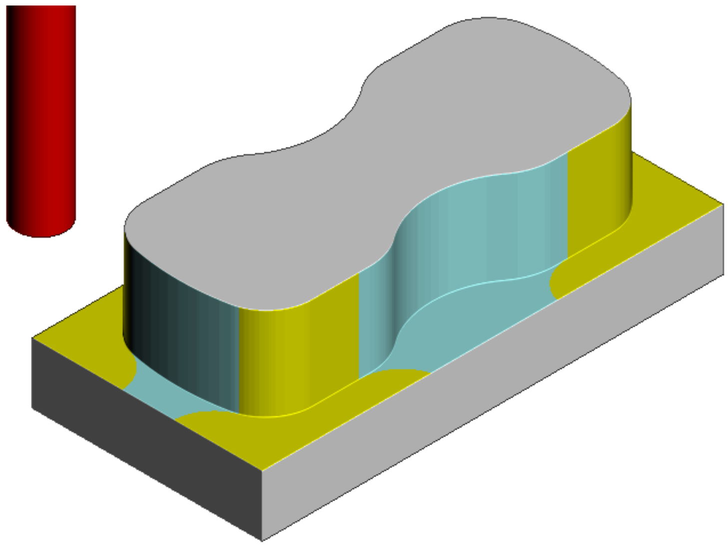
- Using the **left hand mouse button** (With the ***cursor*** visually assisted by the displayed **Tool** and **Cross Hairs**) reposition the **Start** and **End** points along the **Pattern** as shown below.



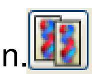
When defining the ***Start*** and ***End*** points along the **Pattern** allow room for suitable **Lead** moves.

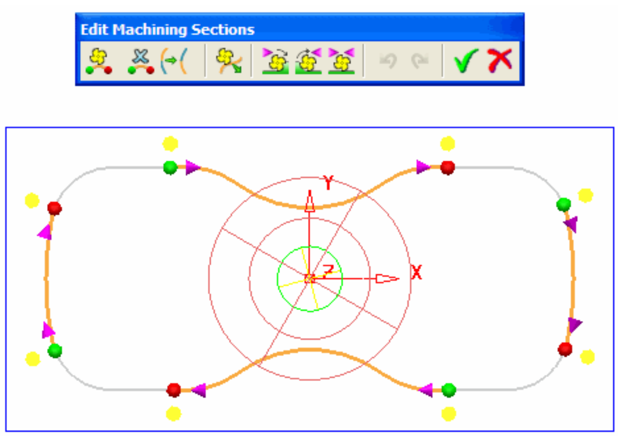
- Select the **green tick** to accept the 'user defined' ***toolpath segments***.
- Select **Calculate** in the main **Curve Profile form** to create the **toolpath**.
- Select **OK** to lose the **Curve Profile form**.
- Use ***toolpath*** **Simulation** to visually check that the tool does not clash with the clamp profiles.

- Reconnect the **ViewMILL** session (***Green Sphere***) and visually check that the **new toolpath** is not missing any material where the **original toolpath** overlaps.



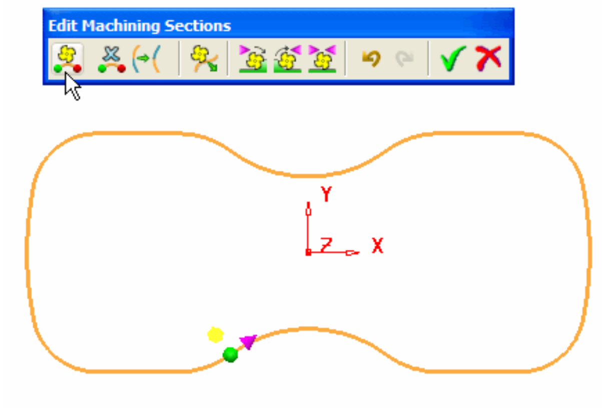
2D Curve Machining Example 2 – Without Clamp Avoidance

- With the ***toolpath*** (**MC-ClampArea-2**) ***activated*** select ***Settings*** (local menu) to open the existing **Curve Profile** form.
- At the top left corner of the form select the '***Make a Copy***' option. 
- Select  to open the **Curve** toolbar.



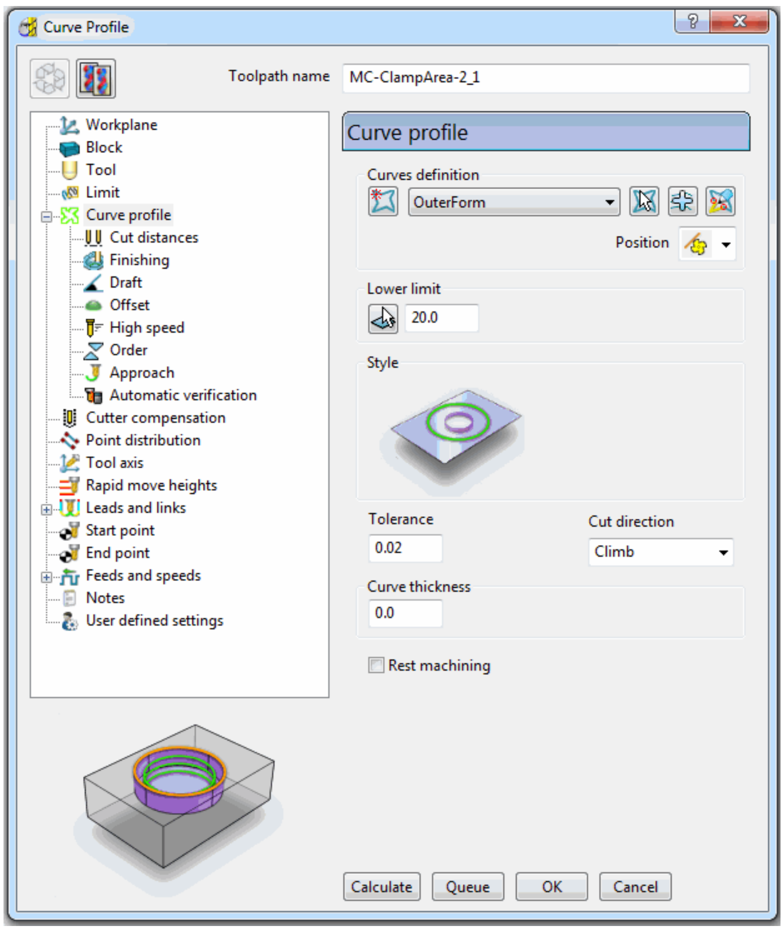
The sections of the **Pattern** included in the original toolpath are displayed, along with the user defined, ***Start*** (green) and ***End*** (red) points.

- To return the machining range to include the whole **Pattern**, click on **Remove selected machining sections**  (Note: This is the default setting).
- Select **Mark all segments for machining**. 

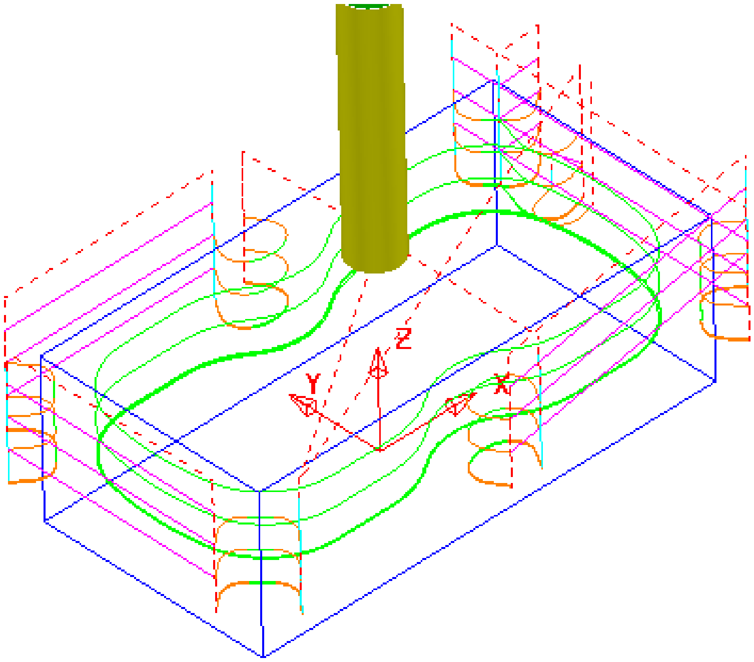


If the **Curve Profile** strategy is used from scratch, the **Mark all sections for machining** option must be selected to provide full control over which side and sections of the **Pattern** are to be machined.
Note the machining preview shows that the wrong side of the curve is currently selected (yellow sphere).

- Select the **Reverse machining side** option. 
- Select the **green tick** to accept the 'user defined' ***toolpath segments***.



- Select **Calculate** in the main **Curve Profile** form to create the **toolpath**.
- Select **OK** to close the **Curve Profile form**.



This time the whole **Pattern** is used to define the range of the machining. This is the default option when **Mark all sections for machining**  is selected followed immediately by the **Green tick**. 

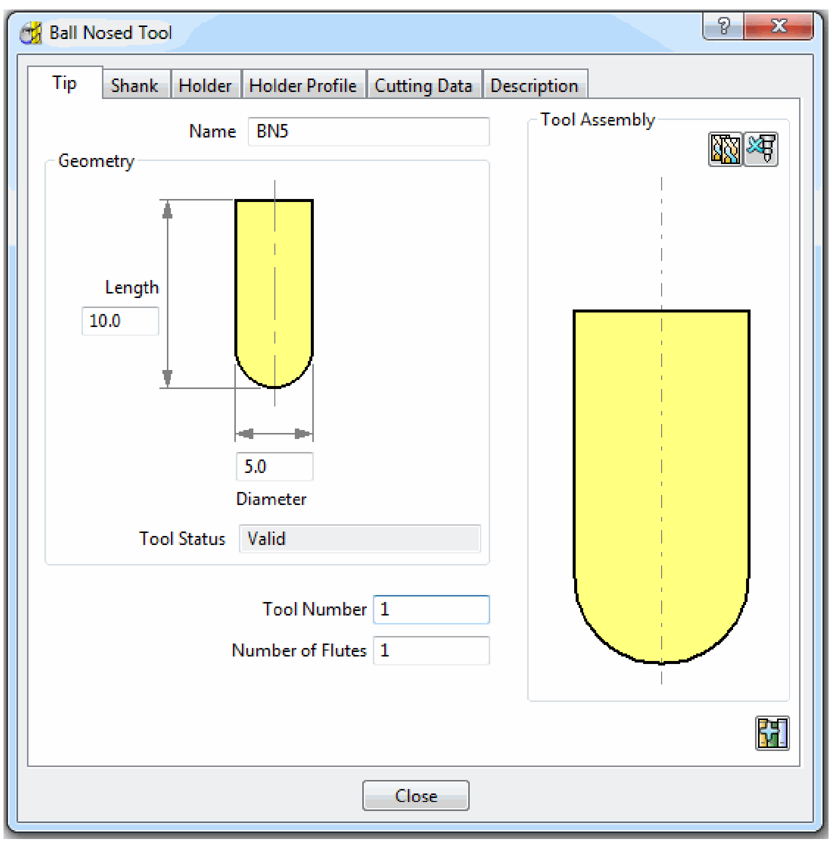
10. Tool Holder Collision Checking

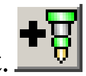
Collision Checking

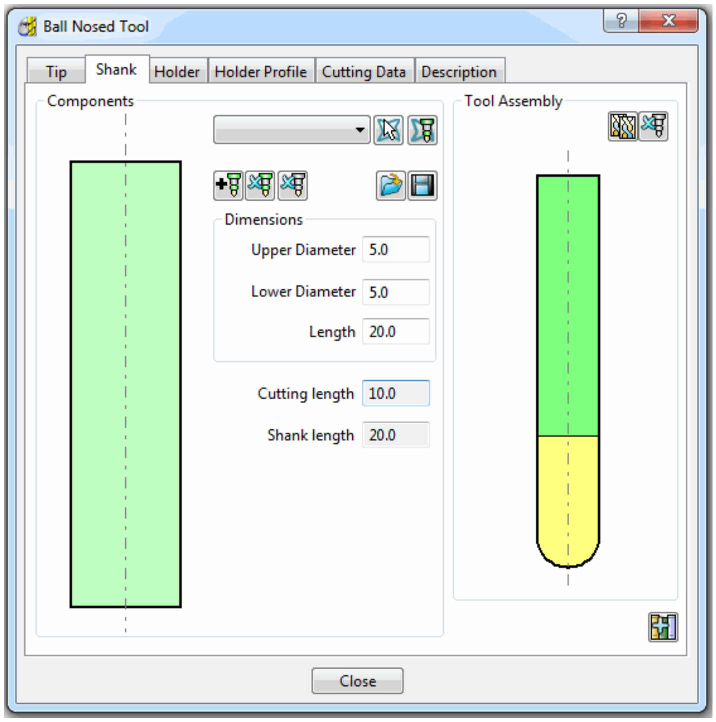
Collision Checking can be applied retrospectively, as long as a tool **Shank** and **Holder** are defined with the **Active Tool**. If the option **Verify - Collisions** is applied, two additional toolpaths will be created from the original, one being collision safe, and the other being in collision. At the same time a copy of the **Tool** with a suitably extended **Overhang** will be substituted into the Original and Collision status toolpaths.

- Select **File - Delete all** and **Tools - Reset Forms** from the **top** menu.
- **Import** the model:-
  C:\Users\training\PowerMILL_Data\Models\cowling.dgk

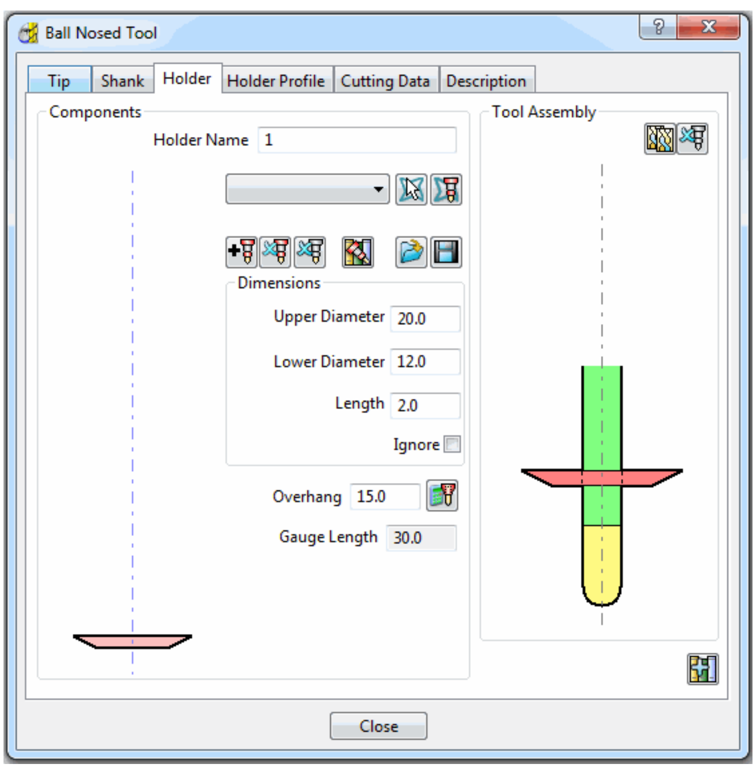
- Calculate the **Block** to **Min/Max limits**.
- Reset **Rapid Move Heights** and the tool **Start and End Point**.
- Create a **Dia 5mm Ball Nose** tool **Named BN5** with a **Length** of **10** and **Tool Number 1**.



- Select the **Shank** tab.
- Select **Add a shank component**. 
- Fill in the form exactly as shown in the image below.



- Select the **Holder** tab.
- Select **Add holder component**. 
- Fill in the form for the first **Holder** element exactly as shown in the image below.

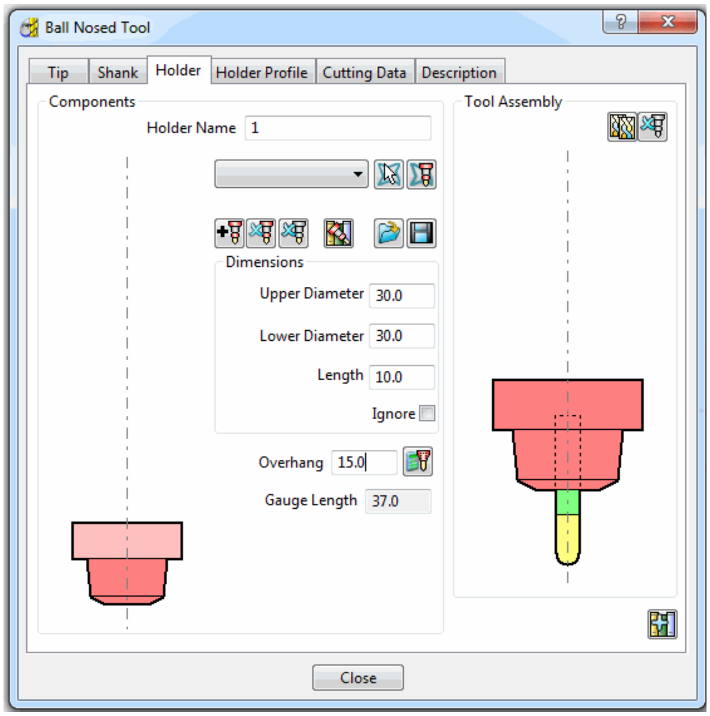


A picture of the current active **Tool** is displayed on the right hand side of the dialogue.
Upper Diameter – the diameter at the top of the current section of the tool **Holder**. This must be greater than or equal to the Lower Diameter.
Lower Diameter – the diameter at the bottom of the current section of the tool **Holder**. This must be less than or equal to the Upper Diameter.
Length – the vertical height of the current **Holder** element.
Overhang – the length of the cutter protruding from the base of the **Holder**.

- Select **Add holder component.**
- Enter an **Upper Diameter** of **22**, a **Lower Diameter** of **20** and a **Length** of **10**.

- Select **Add holder component.**
- Enter an **Upper Diameter** of **30**, a **Lower Diameter** of **30**, a **Length** of **10 and** an **Overhang** of **15**.

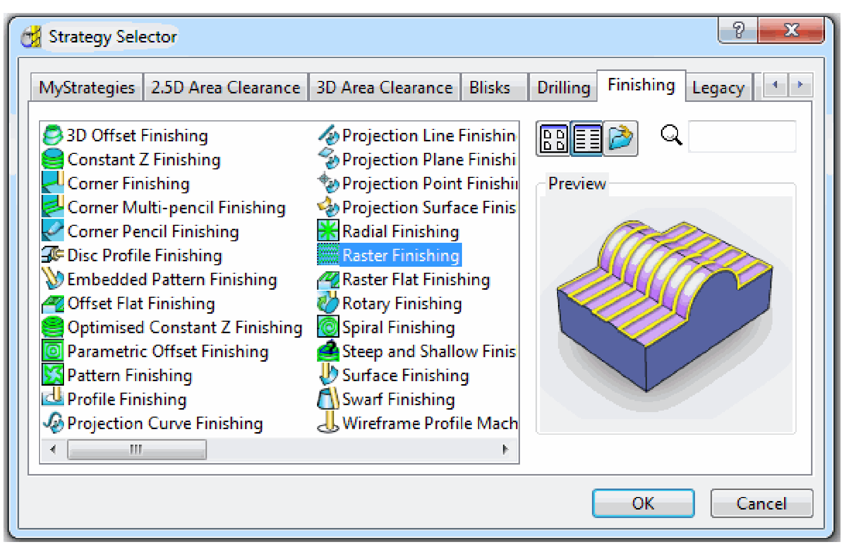
- Select **Save tool holder** in **C:\Temp** with a **File name** as **ToolHolder 1.pmlth**.

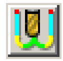


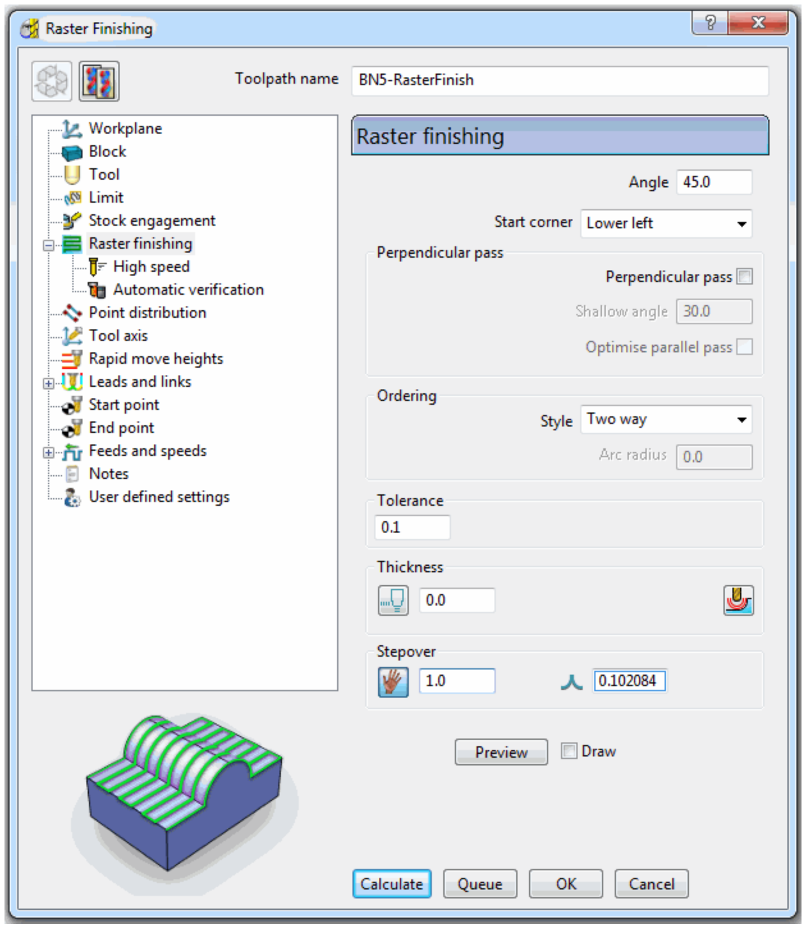
If it is required to modify or delete a **Holder** element, select (left click) on one of them in the assembly illustration and it will turn pale. The current values for that element will be displayed and available for modification.
The same procedure applies to modify **Shank** elements.

After creating the tool **Holder** you save it. It can then be loaded at a later date to check further toolpaths for collisions if required.

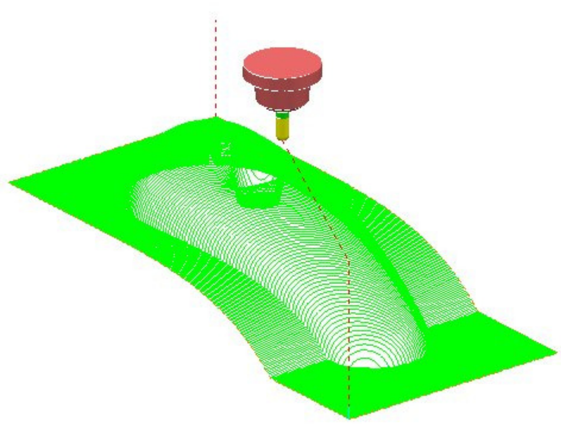
- Select **Close**.
- Select **Toolpath Strategies** on the main toolbar.
- From the **Finishing** options select a **Raster Finishing** strategy.



- Enter a **Name** as **RasterFinish**, an **Angle** of **45** and **Stepover** of **1**.
- Select **Lower Left** as the **Start Corner**.
- Select **Leads and Links**. 
- In the **Leads and Links** dialog box, select the tab **Lead In** and set the **1st** and **2nd Choices** to **None**.
- Do exactly the same in the **Lead Out** tab.
- In the **Leads and Links** dialog box select the **Links** tab, enter a **Short/Long Threshold** of **2**, set **Short** to **Circular Arc** and **Long** to **Skim**.

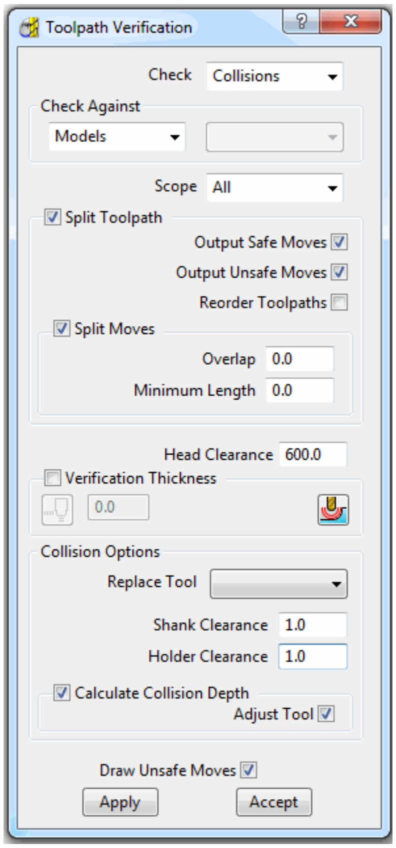


- Select **Calculate** to process the **toolpath** and then select **Cancel**.



The new, gouge free, **Toolpath is** generated but at this stage no collision checking has been applied in relation to the **Shank** and **Holder**.

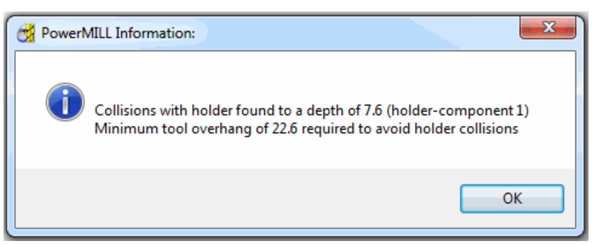
- From the **Main** toolbar select **Toolpath Verification**. 



The two **Check** options here are **Collisions** and **Gouges**. **Scope** contains options to control which actual elements of a toolpath are checked.

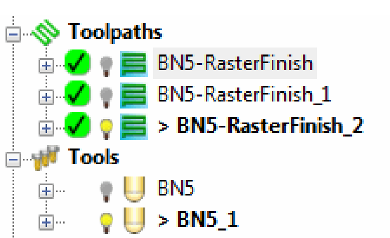
Shank Clearance and **Holder Clearance**, allows the user to specify a value for an additional, safe thickness around the tool **Shank** and **Holder** elements.

- For both ***Shank Clearance*** and ***Holder Clearance*** input **1**.
- Select **Apply**.
- Select **OK**.



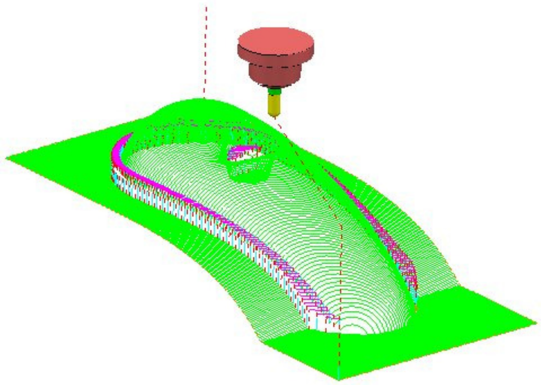
In this case **PowerMILL** has detected that with the current tool **Holder** and **Shank** settings collisions will be present at a **depth** of **6mm**. A suggested tool **Overhang** of **21mm** will be required to avoid this.

- Select **Accept** on the **Toolpath Verification** form.



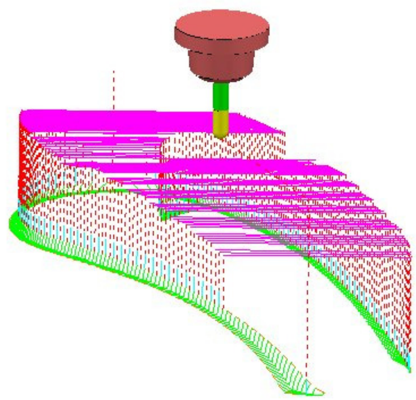
PowerMILL has created two new toolpaths, **BN5-Raster Finish_1** and **BN5-RasterFinish_2**. A new tool named **BN5_1** has also been created in the tools area of the explorer. This new tool has been created with the new valid adjustments made i.e. **Overhang 21**.

- Activate the toolpath **RasterFinish_1** in the **Explorer**.



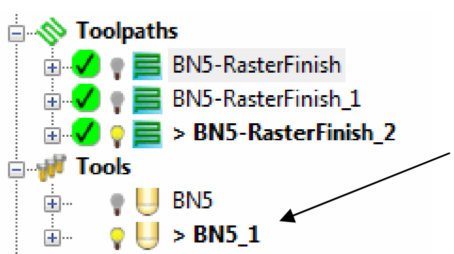
This toolpath only contains segments of the original toolpath, **RasterFinish**, that do not have any collisions so the original tool and tool holder, **BN5** is still associated with it.
If your preference is to run with the shorter overhang then you could run this toolpath on these areas shown.

- **Activate** the toolpath **RasterFinish_2** in the **Explorer**.



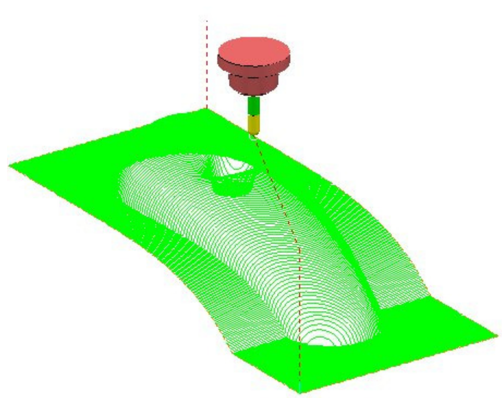
This toolpath contains segments of the original toolpath that can only be machined with the **newly** adjusted tool **Overhang**.

- **Activate** the original toolpath **RasterFinish** in the **Explorer**.



PowerMILL has **replaced** the ***newly created*** **Tool BN5_1** (with an extended overhang) into **BN5-RasterFinish** and **BN5-RasterFinish_2**.

- Select **File** - **Save Project As**:–
  C:\users\training\COURSEWORK\PowerMILL-Projects\ToolCollisionExample

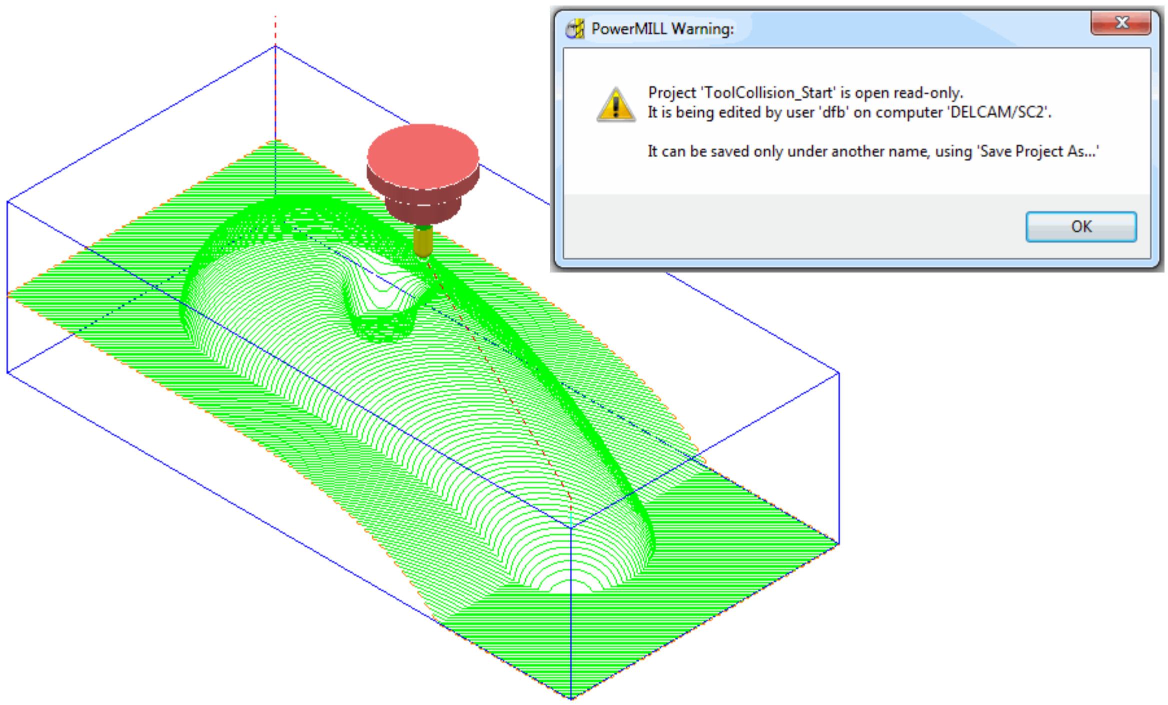


Pro - Tool Holder Collision Checking

Automatic Collision Checking

In ***PowerMILL Pro***, **Tool Holder Collision Checking** can be applied during **Toolpath** calculation.
If the **Toolpath** is calculated using a tool that includes a **Shank** and **Holder** any part of the ***toolpath segments*** that would otherwise be in a **collision** condition will not appear. These missing segments can then be machined later using a modified **Tool** in conjunction with a **Collision Safe Boundary**.

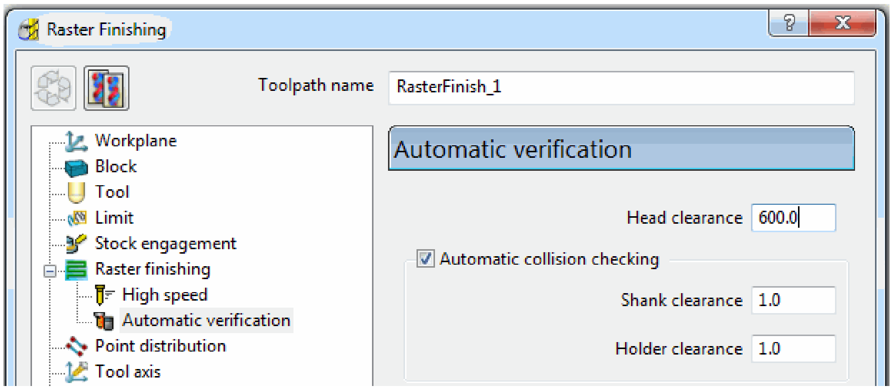
- Select **File - Delete All** and **Tools - Reset Forms**.
- **Open** the **Project**:-
  C:\users\training\PowerMILL_Data\Projects\ToolCollision_Start



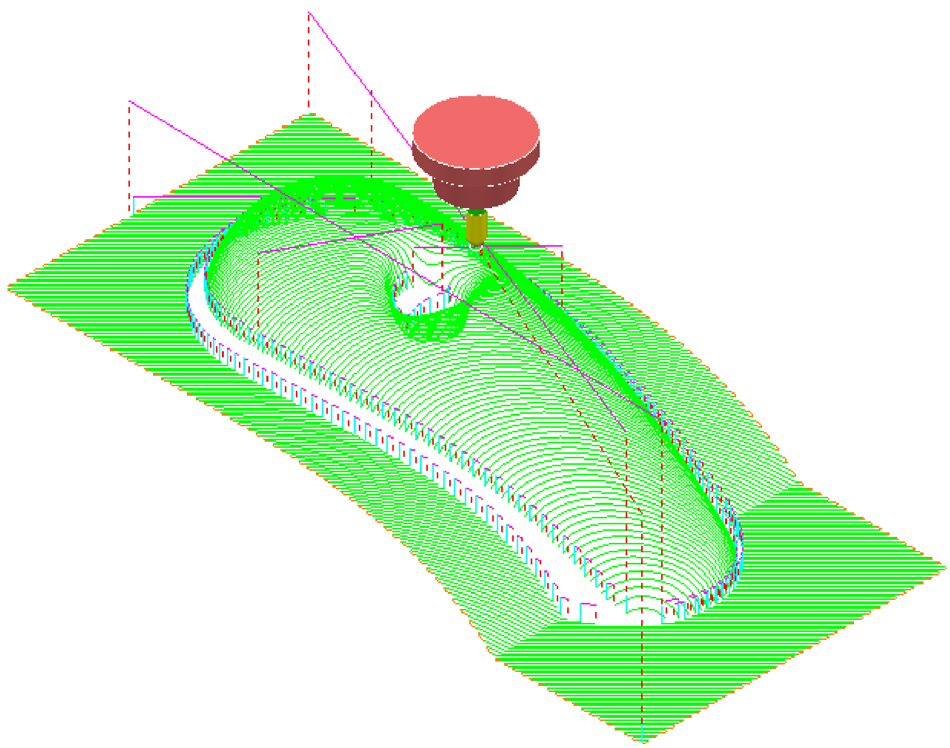
- Select **OK** on the ***PowerMILL Warning*** form.
- **Save Project As**:-
  C:\users\training\COURSEWORK\PowerMILL-Projects\ToolCollisionPro_1

- Right click on the toolpath **RasterFinish** in the **explorer**.
- From the local menu select **Activate** followed by **Settings** and in the **Raster Finishing** form select the **Copy toolpath** icon. 

- With the **Raster Finishing** form open, select the **Automatic Verification** page to open the following form.



- Tick the box ***Automatic collision checking*** and for both **Shank Clearance** and **Holder Clearance** input **1.0**.
- **Calculate** the **Raster Finishing** form to directly create a collision free toolpath, **RasterFinish_1**.

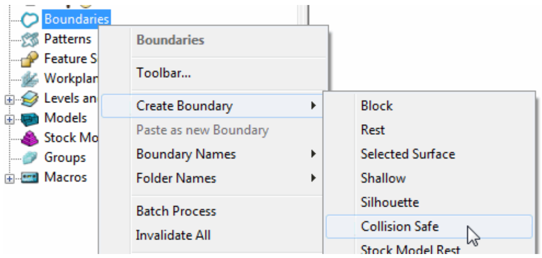


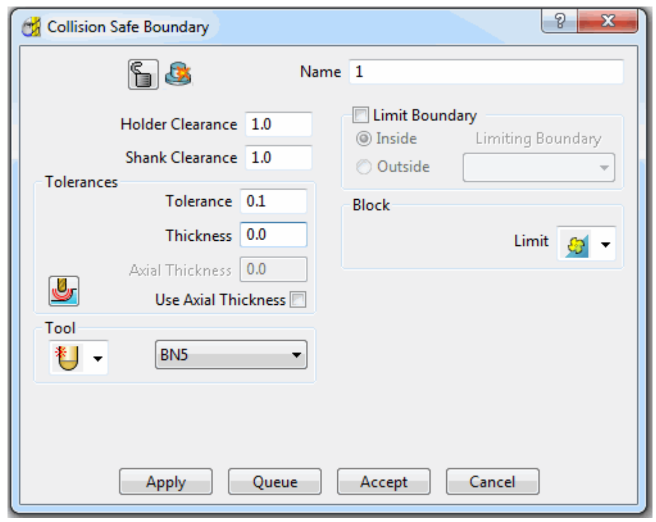
Note:- This method does not inform the user a of a recommended new **Shank** length for the missing area or automatically create a new compatible tool (**BN5_1**).

Collision Safe Boundary

A toolpath for the remaining areas will require a new **Dia 5 Ball Nosed** tool with increased **Shank** length. The additional **Toolpath** will be calculated to limits set by a **Collision Safe Boundary** based on the current tool (**bn5**).

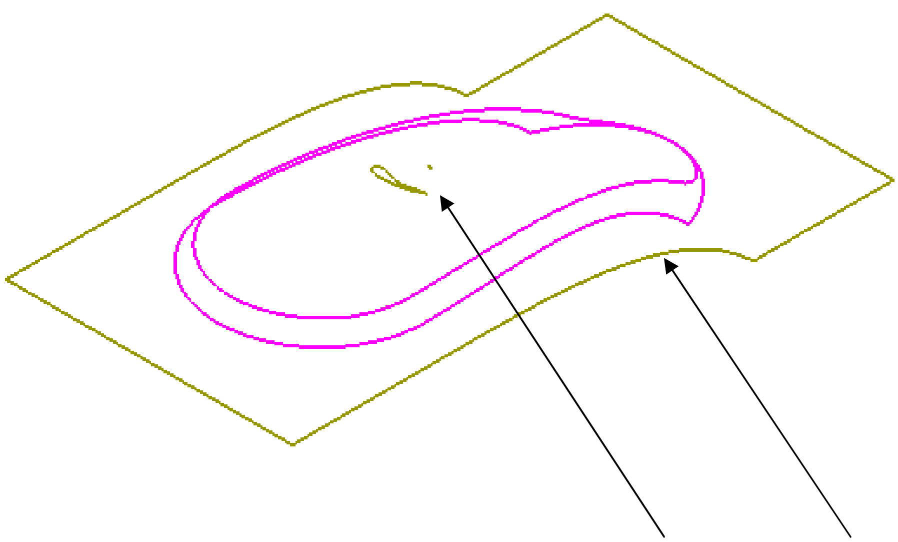
- Ensure that the tool **BN5** is **Active**.
- In the **explorer** right mouse click over **Boundaries** and in the local menu select **Create Boundary - Collision Safe**.





- **Apply** the form.

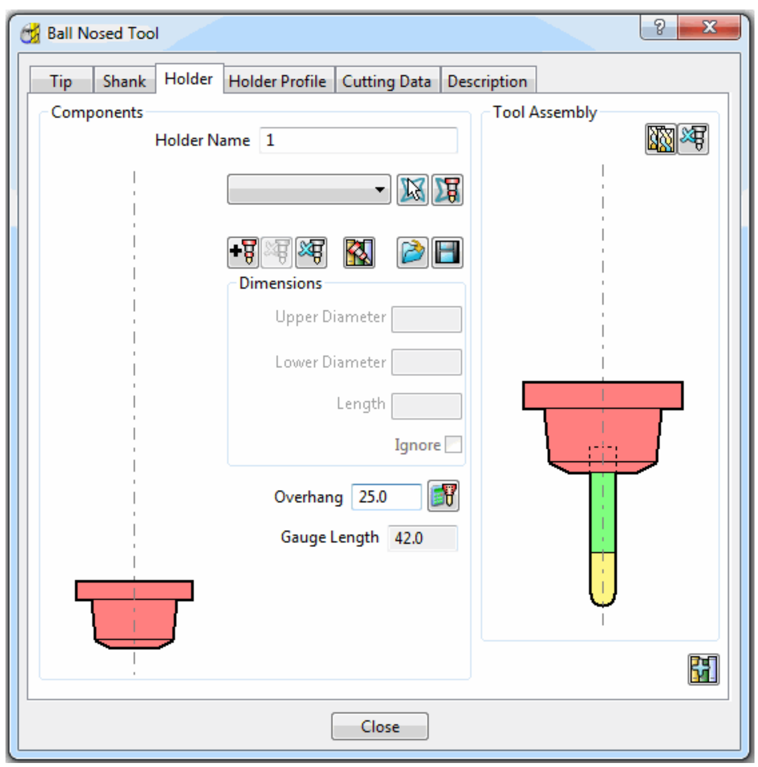
The lower part of the central pocket is more suited to be finish machined using an **End Mill**.



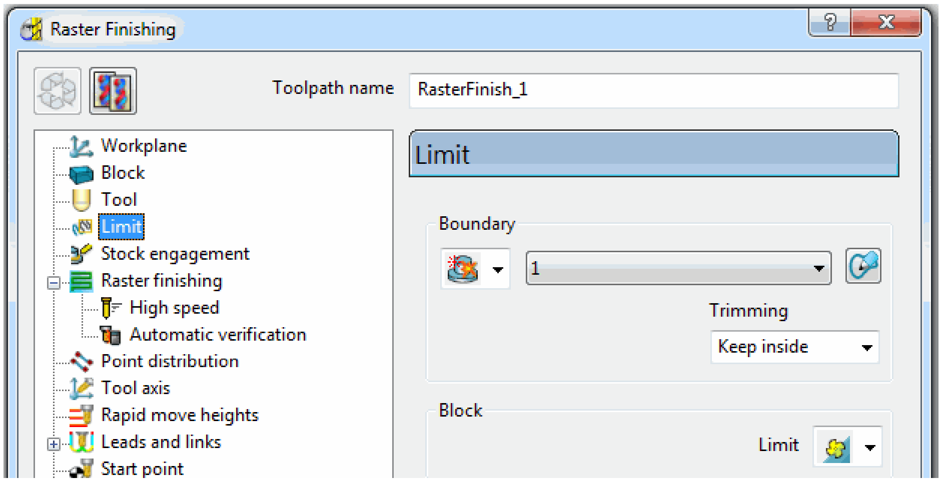
- **Select** and **Delete** the 2 segments in the central pocket and the outer one.

- In the **explorer** right mouse click over the **Tool** - **BN5** and from the local menu select **Edit - Copy Tool** (to create **BN5-1**).

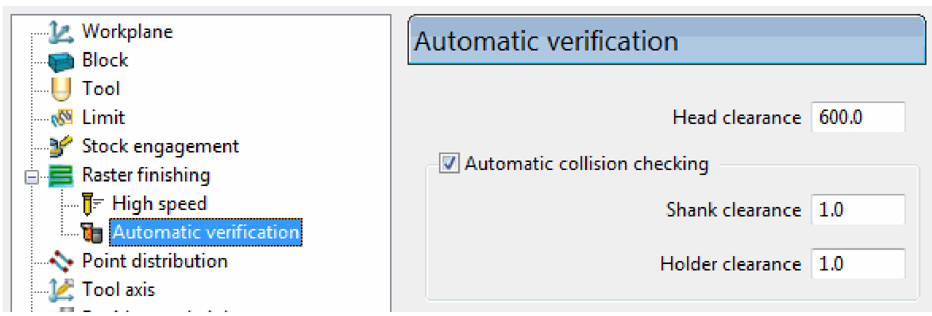
- Right mouse click over the new tool **BN5-1** and from the local menu select **Activate** followed by **Settings**.



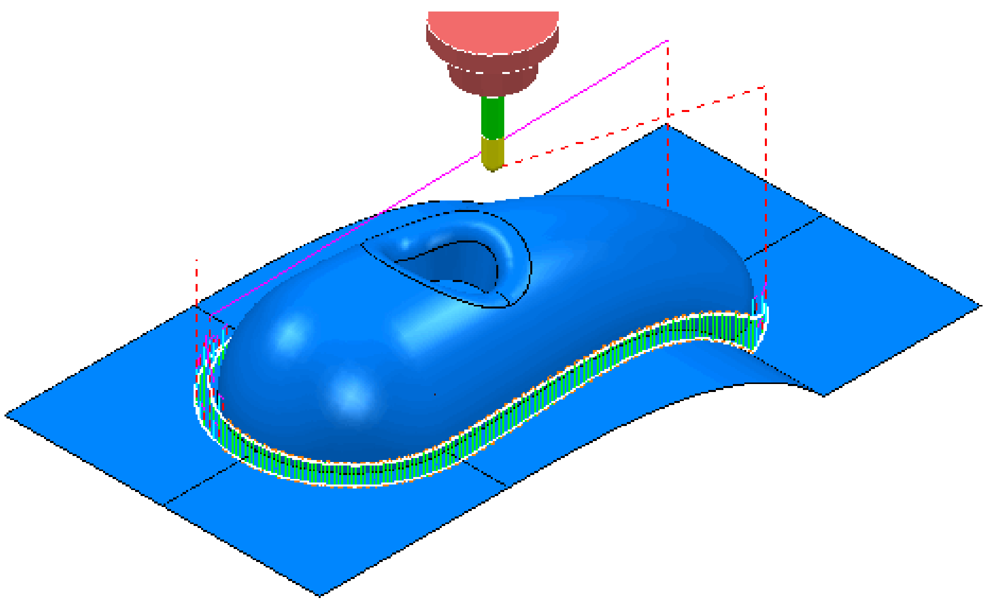
- In the **Tool Definition** form select the **Holder** tab and change the **Overhang** value to **25.0** before selecting **Close**.
- Right click on the toolpath **RasterFinish_1** in the **explorer**.
- From the local menu select **Activate** followed by **Settings** and in the **Raster Finishing** form select the **Recycle toolpath** icon.
- **Activate** the new 'long reach' Tool **BN5_1**



- Select the ***Limit*** page and select **Boundary 1** with ***Trimming*** - **Keep inside**.



- Keep the previously used **Automatic verification** settings.



Note that **Automatic Collision Checking** is still active and as can be seen in the above illustration there is no evidence of the **toolpath** being fragmented.