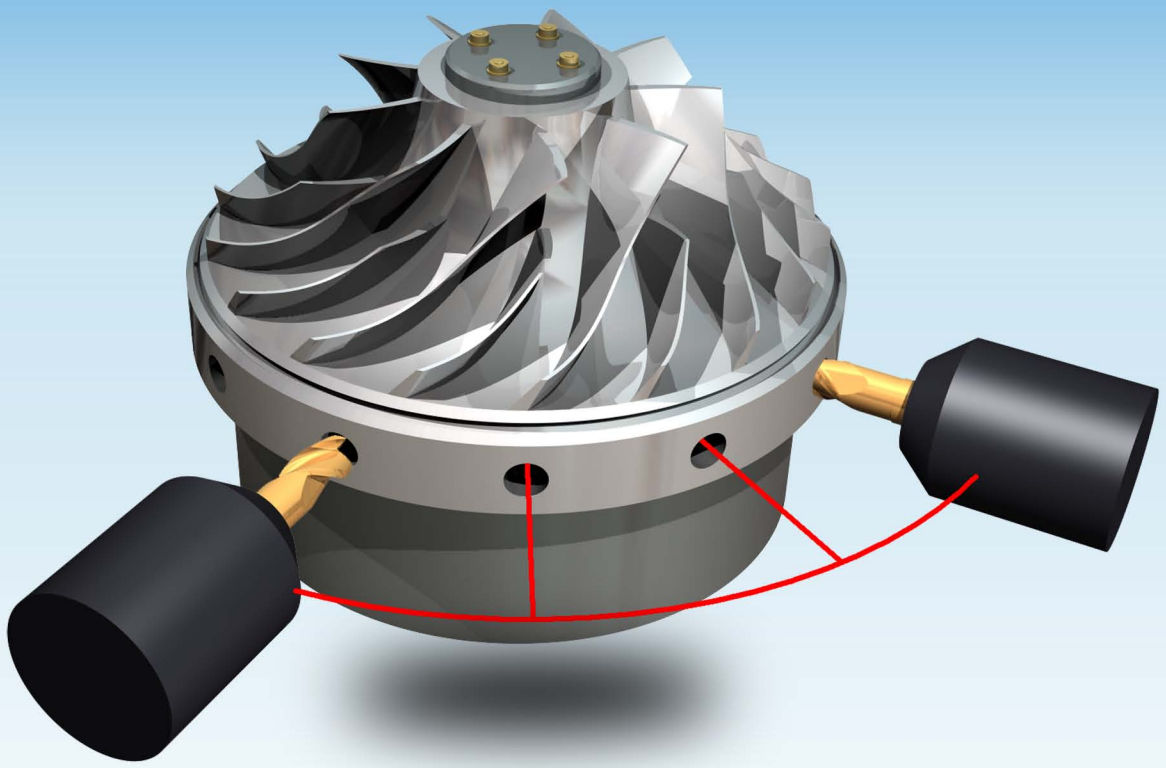
Delcam

PowerMILL

Training Course



Delcam

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

Important Notice

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

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

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

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

Training Centre
Tel: 0121 683 1050
Fax 0121 7665511

Customer Support
Tel: 0121 683 1010
Fax: 0121 7665542

PowerMILL 9 Contents

1. Getting Started

Introduction.

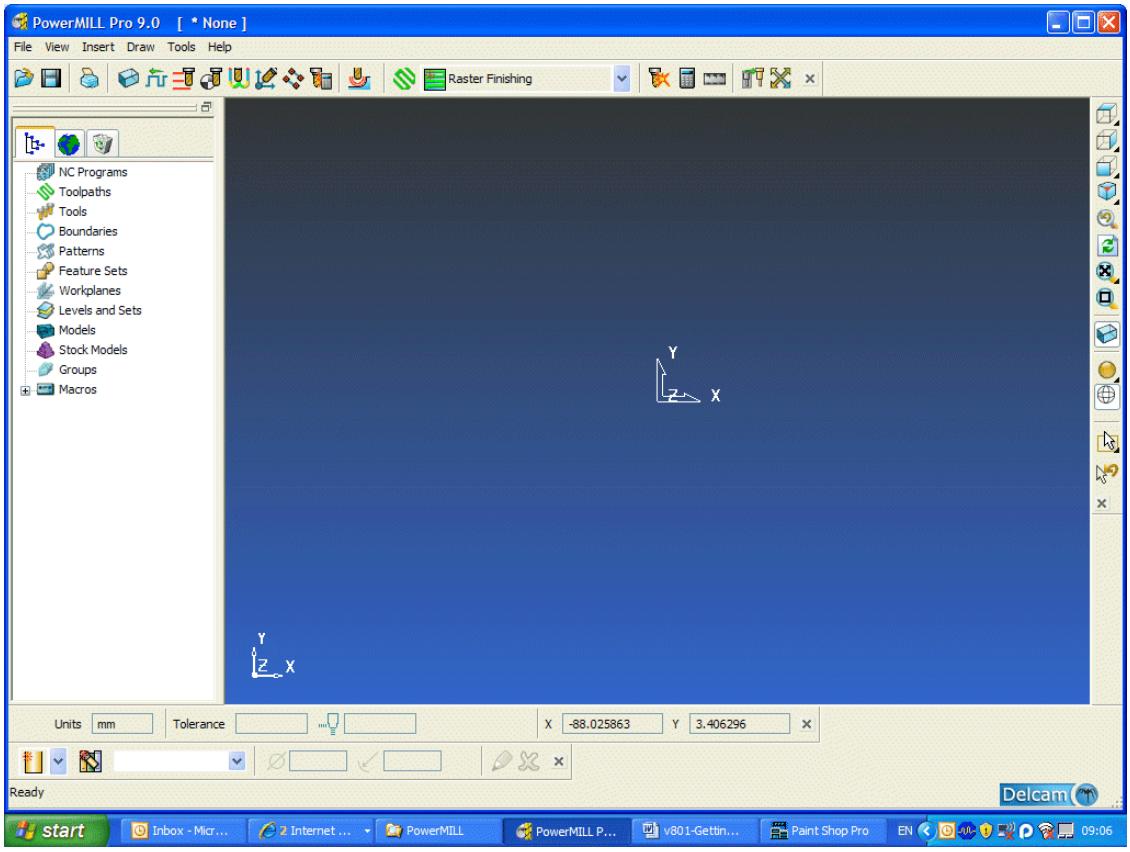
This course covers the 3-Axis functionality available in **PowerMILL**. The additional features available with **PowerMILL–Pro** and **Five Axis** licenses are covered in separate modules. **PowerMILL** will quickly create gouge free cutter paths on imported component data. It supports **Wireframe**,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

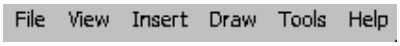
- Double click the relevant **PowerMILL** shortcut **icon** on the desktop:

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

The following screen is then displayed:



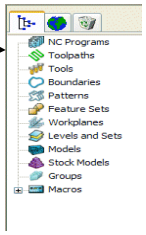
The screen is divided into the following main areas:

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 – 

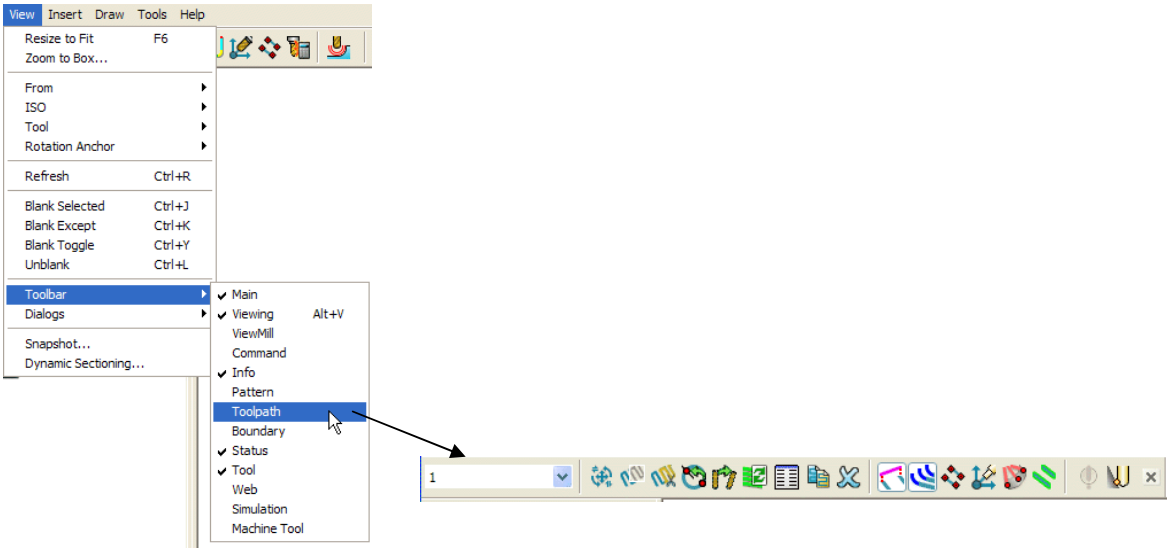
Provides quick access to standard view and shading options in **PowerMILL**.

6) Information Toolbar - 

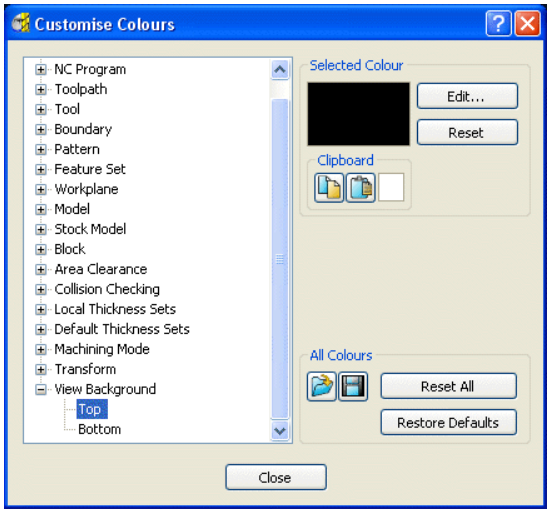
This area provides a reminder of some of the active setup options.

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 

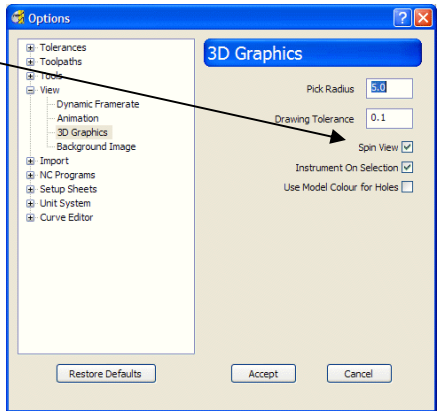
If this option is selected then multiple, selection of entities will occur by simply ***dragging*** the cursor across the required items. This is ideal for quick selection areas of the model consisting of multiple surfaces. To ***deselect*** the **Ctrl** key is depressed while dragging across an entity.

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.

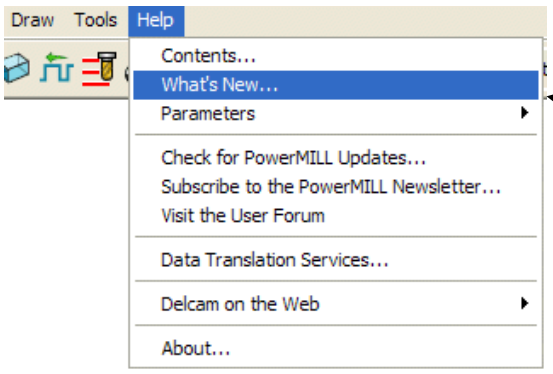
2. **Online Help**. Select **Help>Contents…** from the **Main Toolbar**, to access the online help documents. There is a full index and search facility provided.

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

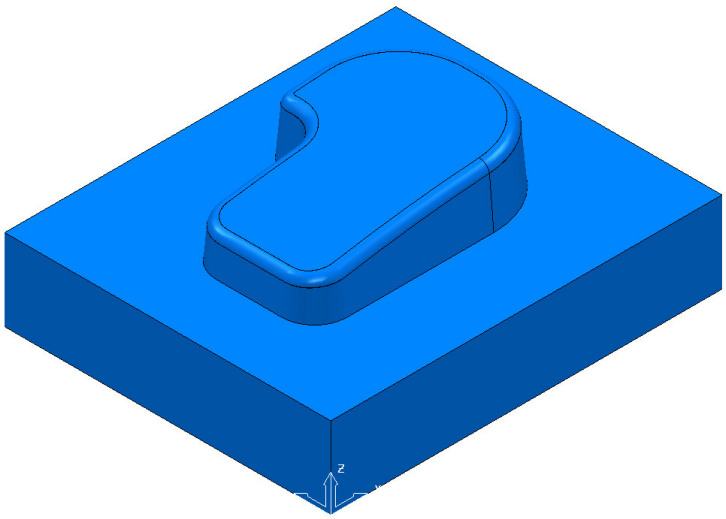
This example provides a quick overview of the machining process. It shows how to create and output a couple of simple toolpaths on a model of a valve chamber (using default settings wherever possible).

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

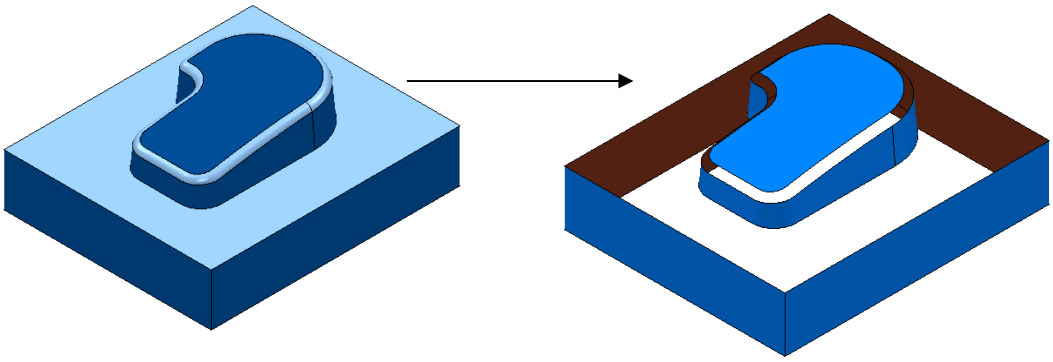
- From the **main pulldown** menus select, **File - Import Model** and browse for the model file:-
  D:\users\\training\ PowerMILL_Data\Models\PmillGettingStarted.dgk.



Blanking of Model entities

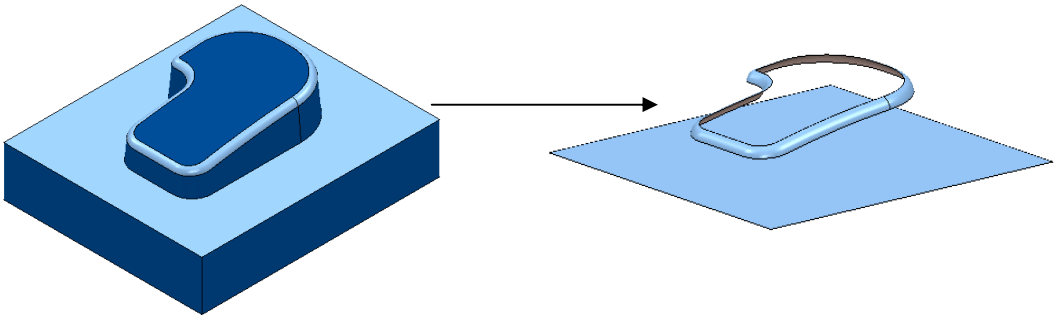
This provides the user with a quick and simple way to control which individual, model entities are displayed. In the illustrations below, the ***light blue*** surfaces are **selected**.

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**).



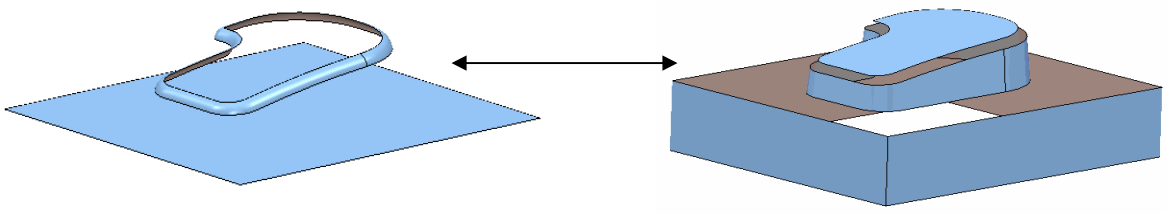
If one or more surfaces are selected all others can be removed from the graphics area by using the **Blank Except** option (**Ctrl K**) in the local menu.

The 2 selected items are light blue.



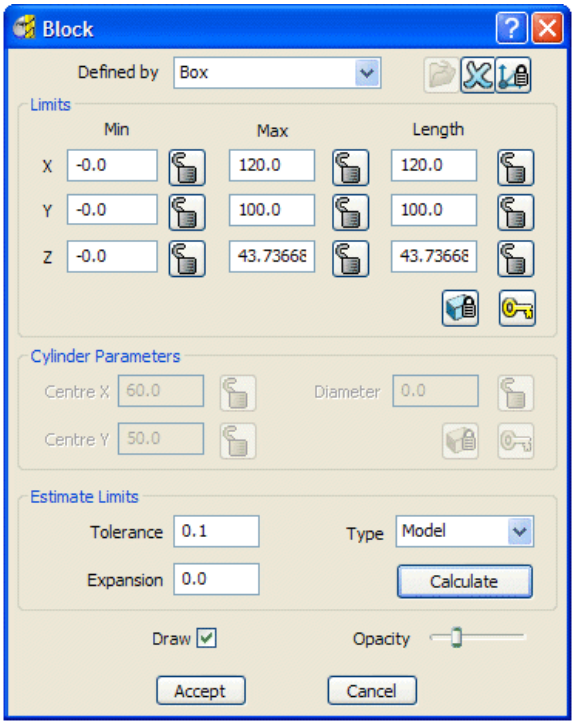
To return all ***Blanked*** items back to the graphics are the **Unblank** option (**Ctrl L**) is selected in the **Default** menu (accessed by right clicking in the graphics area).The **Blanked** items are returned to the graphics area and become selected (back to the left hand image above).

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

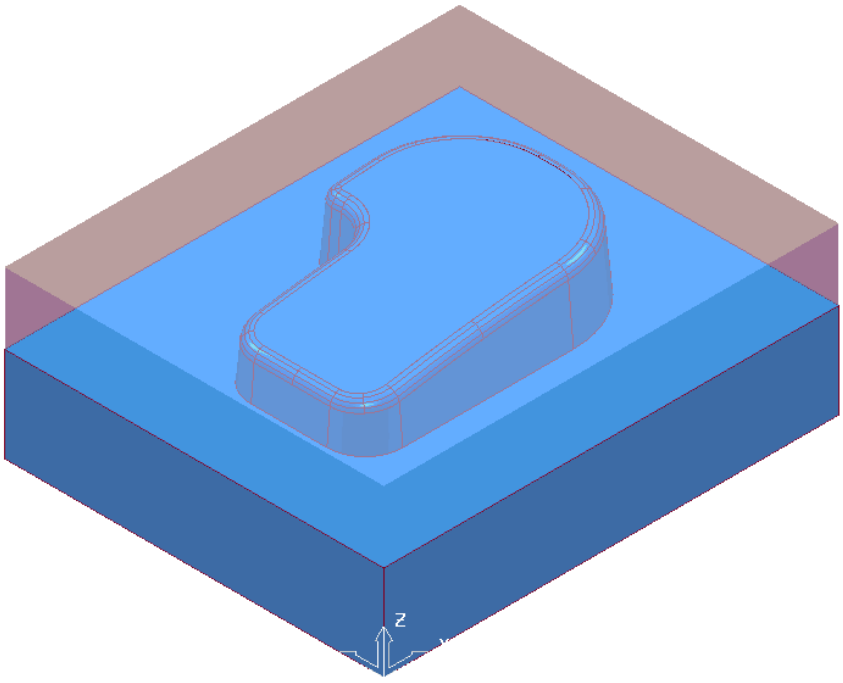
- Click on the **Block** icon  on the top toolbar.



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

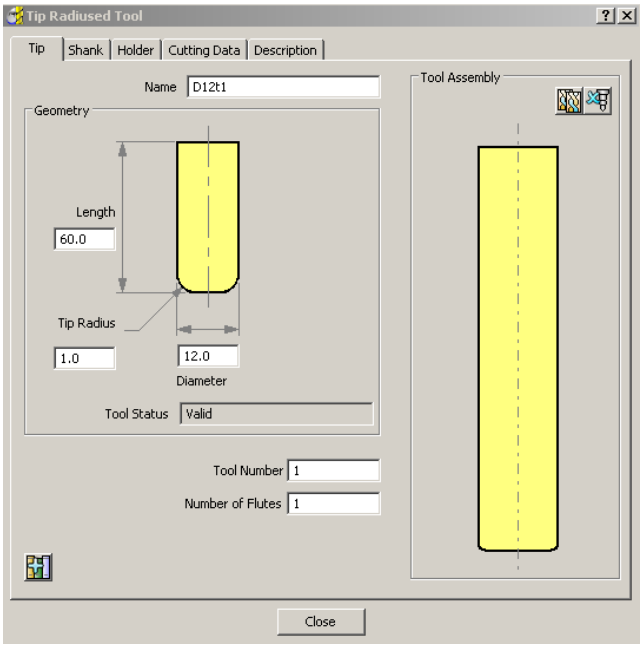
The **Tool definition** forms are accessed from the icons accessed from the **Tool toolbar** located to the bottom left corner of the graphics area.
For use with this example, 2 tools will be created, A **Tip Radiused** for roughing out and a **Ball Nosed** for finishing.

- 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**.

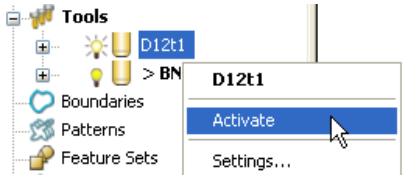
- Select the **Create a Tip Radiused tool** icon. 



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**.

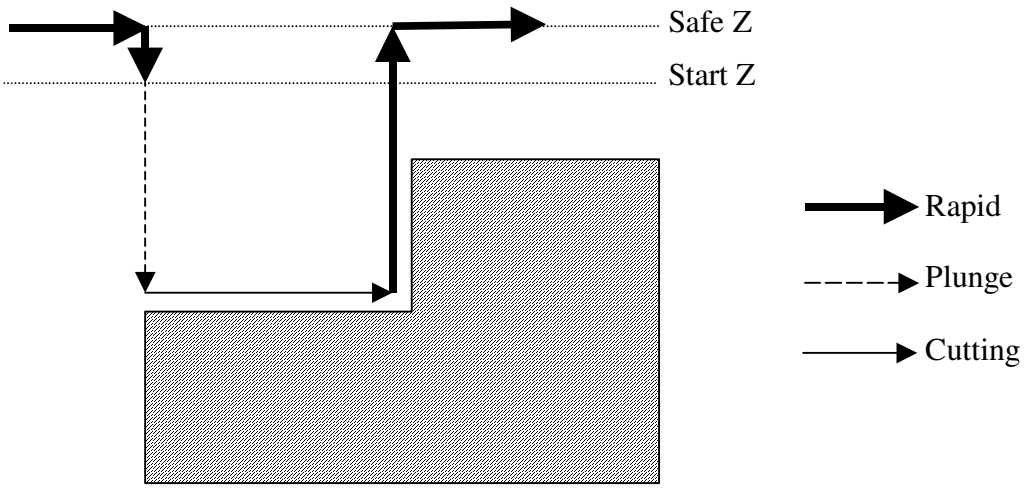
- Repeat the **Tool Definition** operation, this time selecting '**Create a Ball Nosed tool**' and in the form entering a ***Diameter* 12** with a ***Tool Number* 2** before and enter the ***name* BN12** before Clicking on **Close**.
- In the **explorer** panel on the left of the screen, open the **tools**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



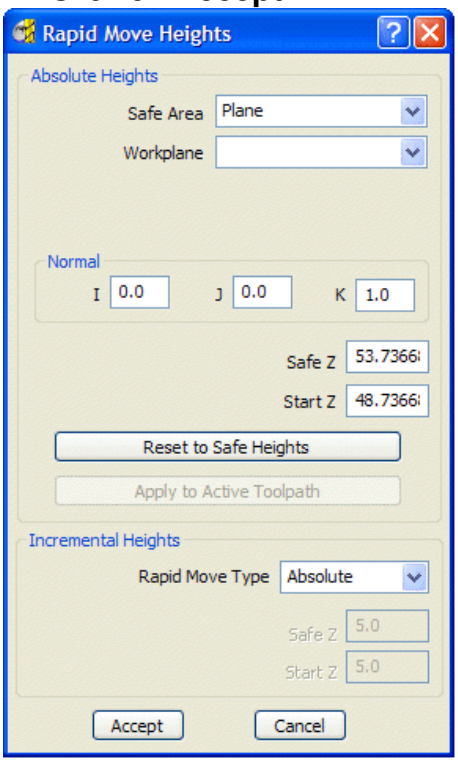
Only one tool can be **Active** at any one time and the word **Activate** in the local menu will be prefixed with a ***tick***. the ***active*** tool will automatically be included in the form when a ***machining strategy*** option is opened.
In the **explorer**, the **Active** tool will be displayed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



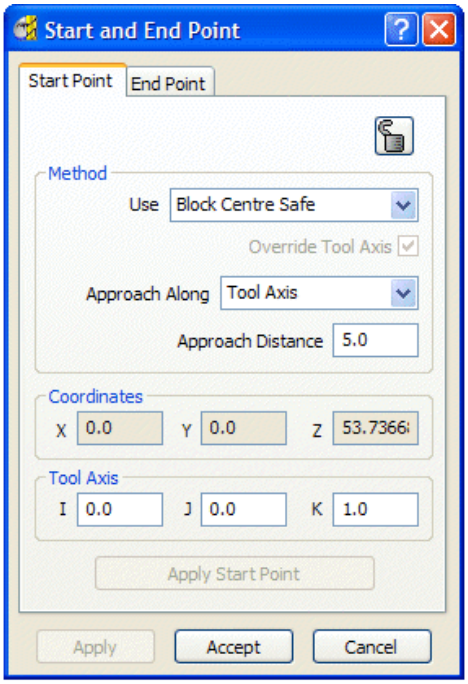
- Click on the **Rapid Move Heights** icon. 
- In the resultant form select the **Reset to Safe Height** button.
- Click on **Accept**.



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.

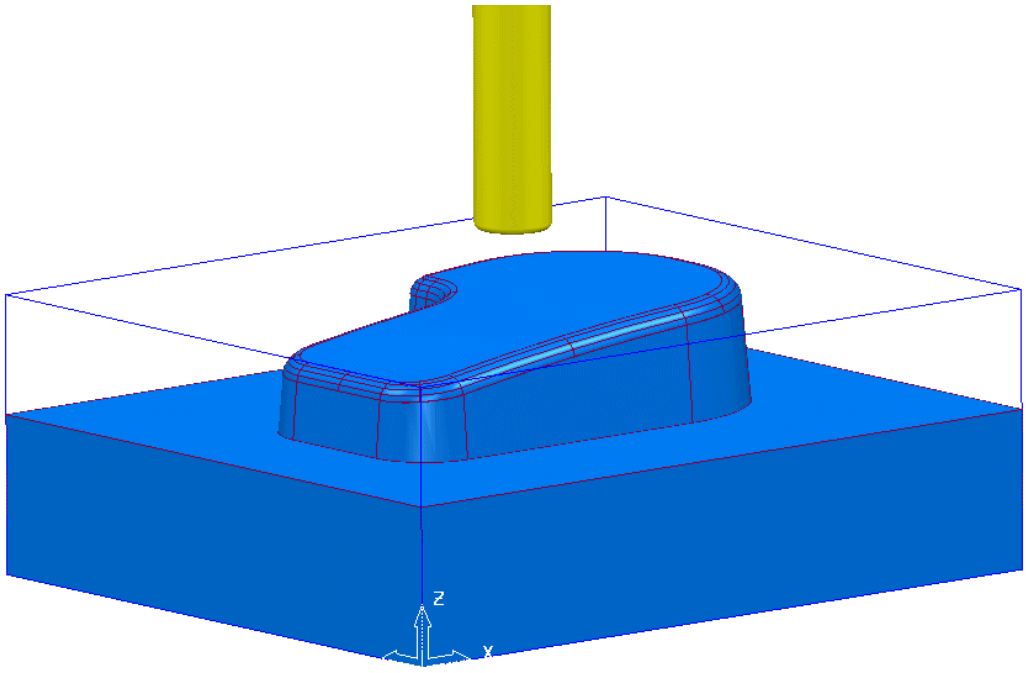
Tool Start and End Point.

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

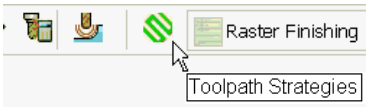
- **Accept** the form with the default settings.



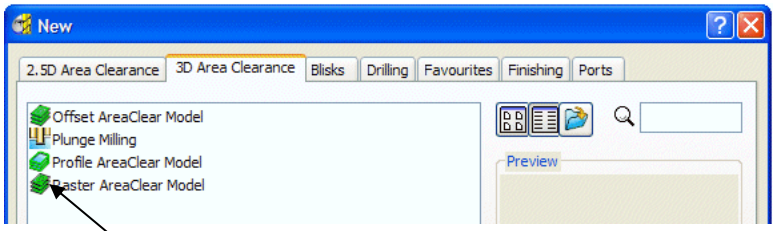
The **D12t1** tool is positioned at the **Block Centre Safe** position ready for the user to create the first toolpath.

Creating a Roughing Strategy

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

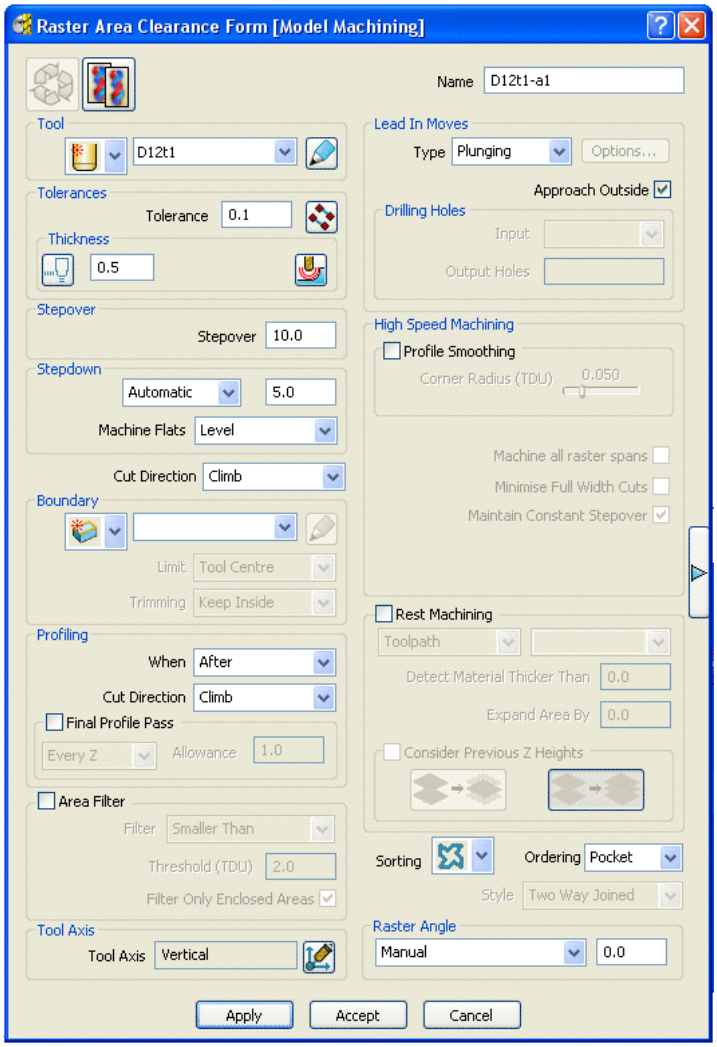


- Select the **3D Area Clearance Tab**.



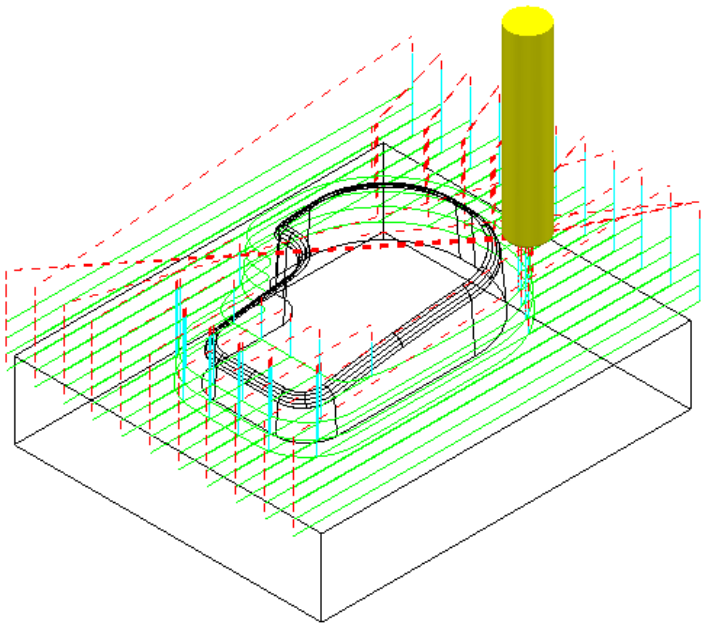
- Select the option **Raster AreaClear Model** to open the following form.

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



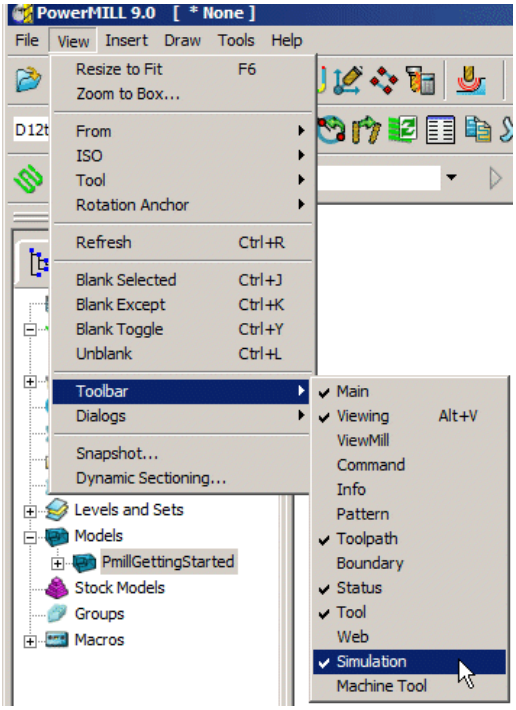
- Edit the ***Thickness*** value to **0.5**. This is the amount of material that will be left on the job
- Edit the ***Stepover*** value to **10**. This is the distance between each raster pass (the width of cut).
- The ***Stepdown*** value (depth of cut) is left at the default of 5 mm.

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

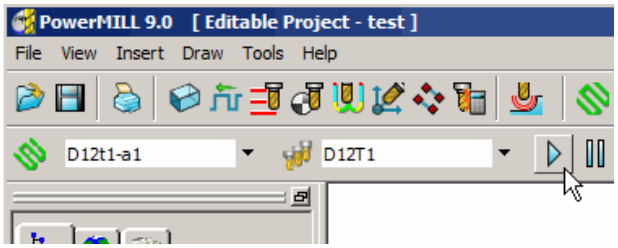


The resultant **3D Raster Area Clearance** can be simulated as follows:

- 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.

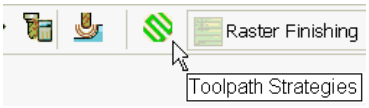


The other buttons on this toolbar can be used to rewind or step through the simulation.

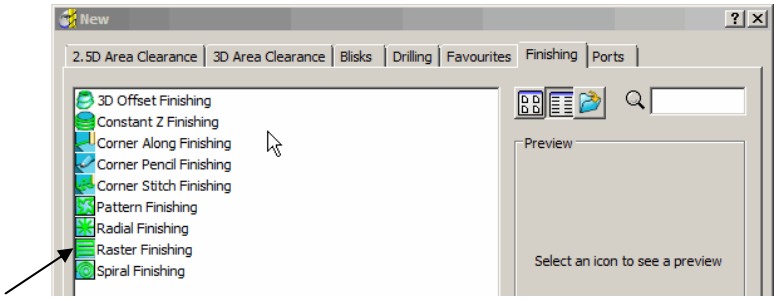
NB. The above strategy has been processed almost completely using the default values, the exceptions being the **Name**, **Thickness** and **Stepover**.

Creating a Finishing Strategy

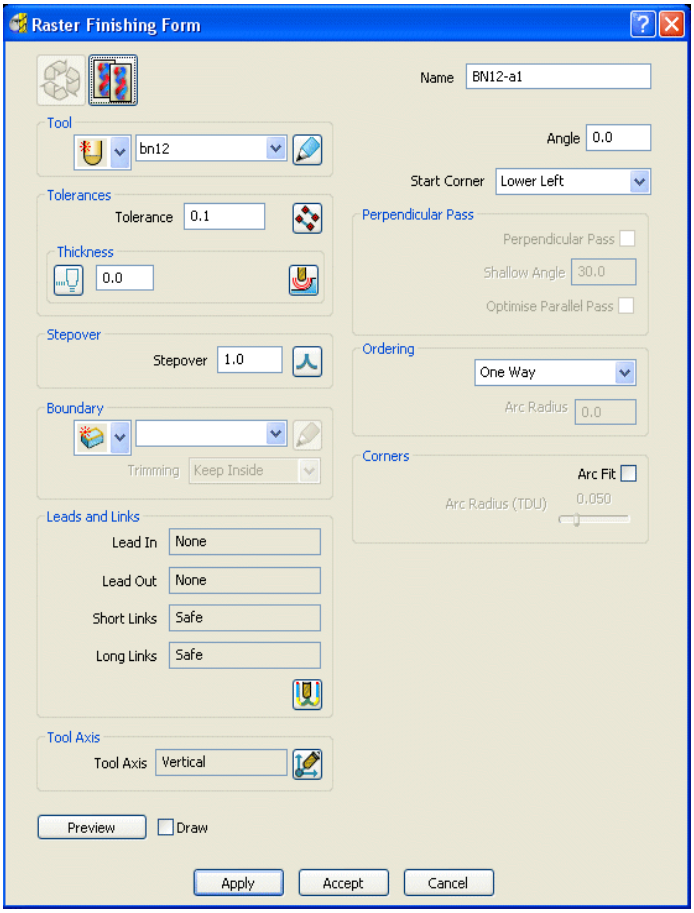
- In the **explorer** right mouse click on the **BN12** tool and in the local menu select **Activate**.
- From the **Main toolbar** select the **Toolpath Strategies** icon.



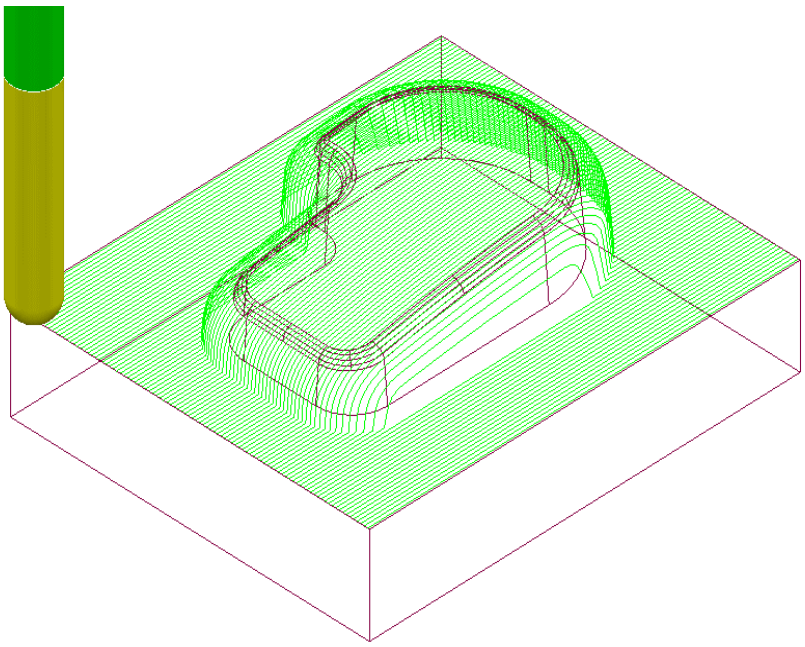
- Select the **Finishing Tab**.



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



- Input ***Name*** **Bn12-a1**
- Edit the ***Stepover*** value to **1.0**
- Click the **Apply** 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

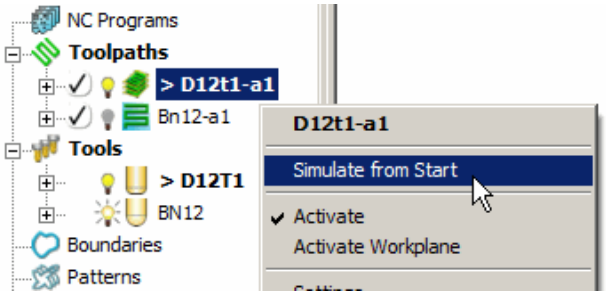
PowerMill provides two main options for simulating toolpaths. The first simulates the cutting tool as it progresses along the toolpath. The second includes a shaded image of the stock material being reduced as the tool progresses along the toolpath.

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.**

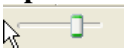


- The Toolpath Simulation toolbar will be displayed at the top of the screen. This displays the name of the toolpath and tool, together with buttons to control the simulation.

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 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.

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

 **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.

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

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

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

 **Unload -** stops the simulation and dims all the 'play' buttons.

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

- Animate the toolpath using the controls listed.
- Actvate the finishing toolpath **Bn12-a1** and repeat the animation process.
- Unload the toolpath when complete.

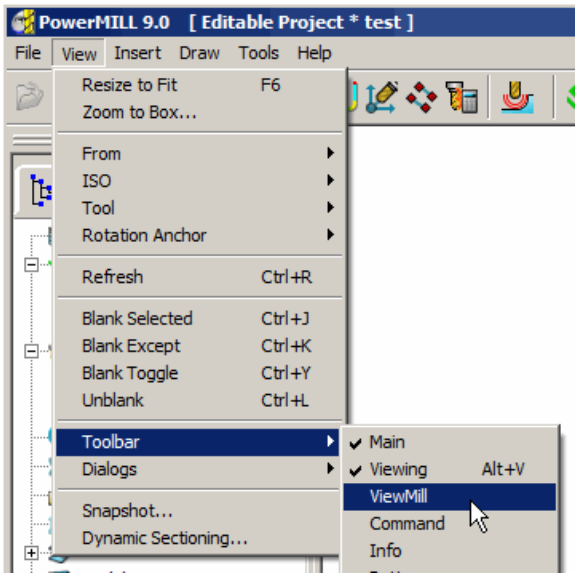
TIPS!

When selecting a path for animation from the toolbar, be sure that the light bulb next to the toolpath name in the tree is switched on.

The toolpath must be rewound to the beginning or unloaded and re-loaded before it can be played again.

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.

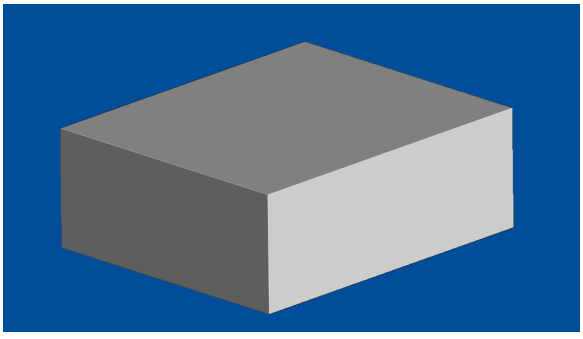


The **ViewMILL** toolbar will be displayed, although initially all the icons will be greyed out.

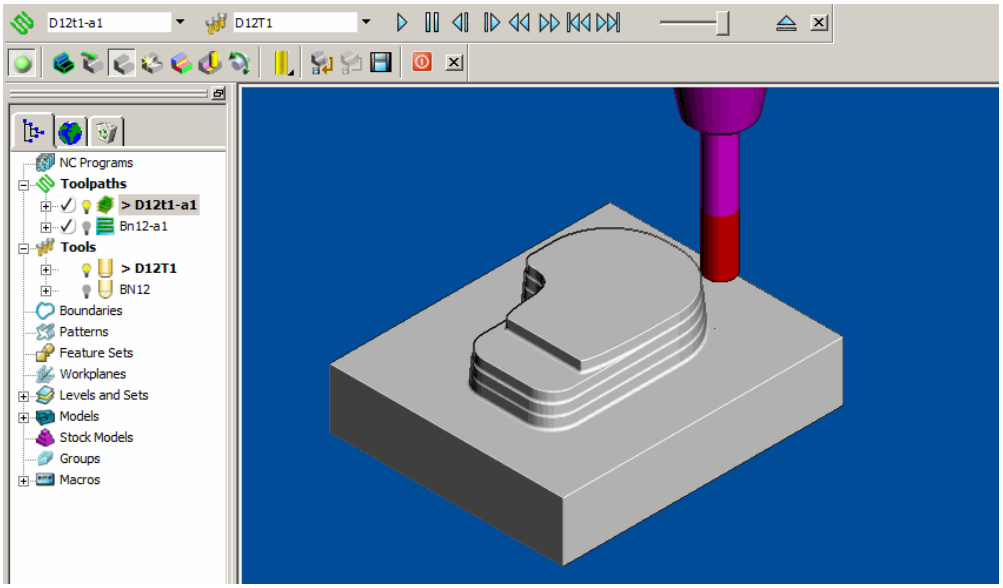
- Click the first button  to **Toggle ViewMILL Window** and enter ViewMILL mode .

The ViewMILL toolbar will then highlight.

Click the fourth button  to select a plain shaded image.



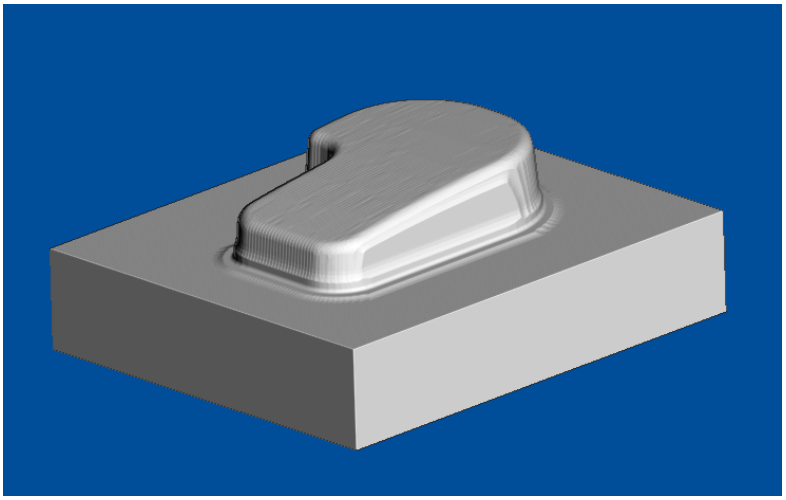
- Select the **tool** icon to display the tool followed by the **Play** icon .



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**

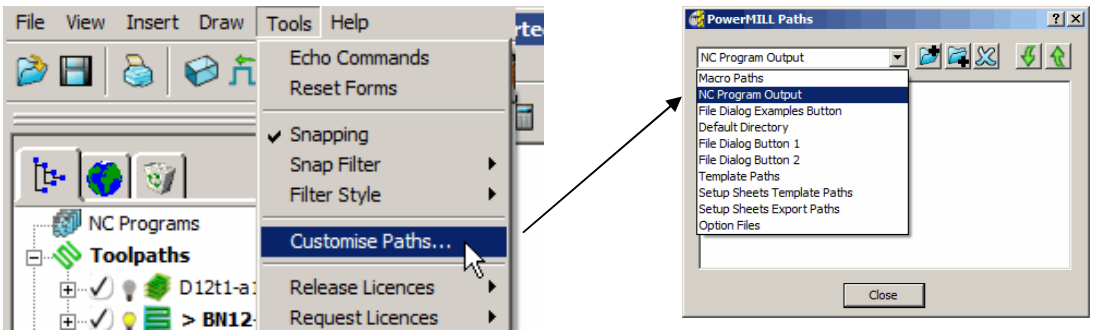
followed by the tool icon and **Play** icon again, to view the continued simulation of material removal by the finishing toolpath.



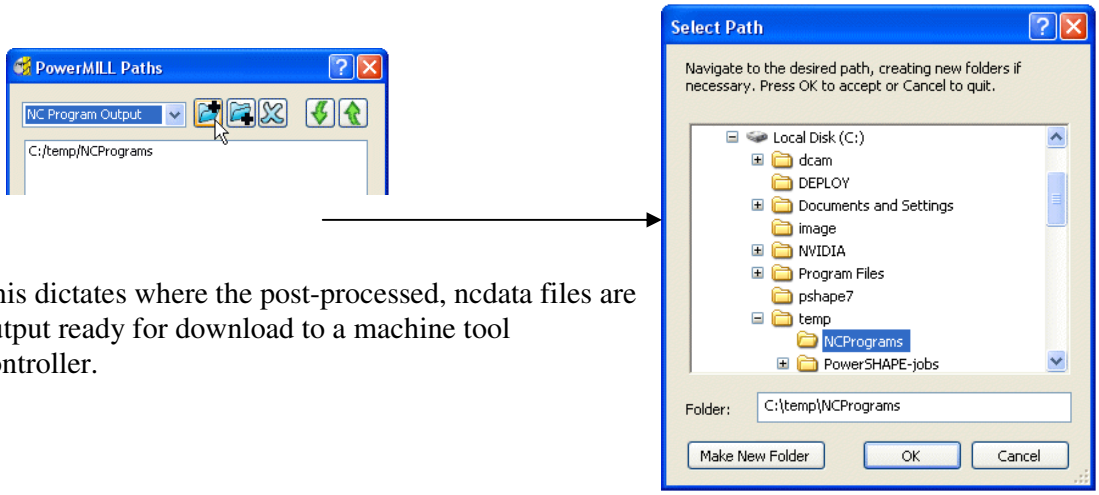
- In the **Simulation** toolbar select the **ViewMILL Exit** icon to exit the **ViewMILL** session.

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).



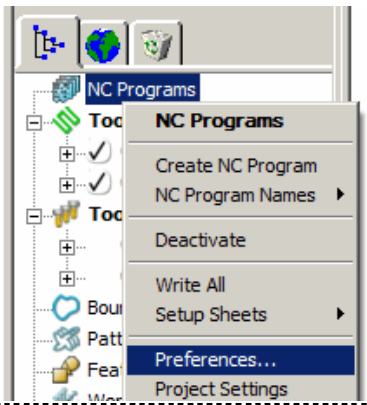
- In the ***Powermill Paths*** form select the option **NC Programs Output**.



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

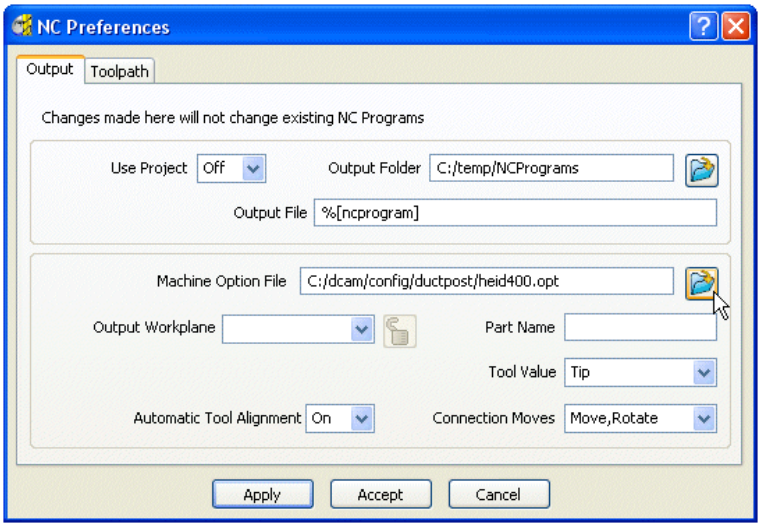
- Right mouse click the **Add path to top of list** icon and in the ***Select Path*** form browse to the required location **C:\temp\NCPrograms** and select **OK**.

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



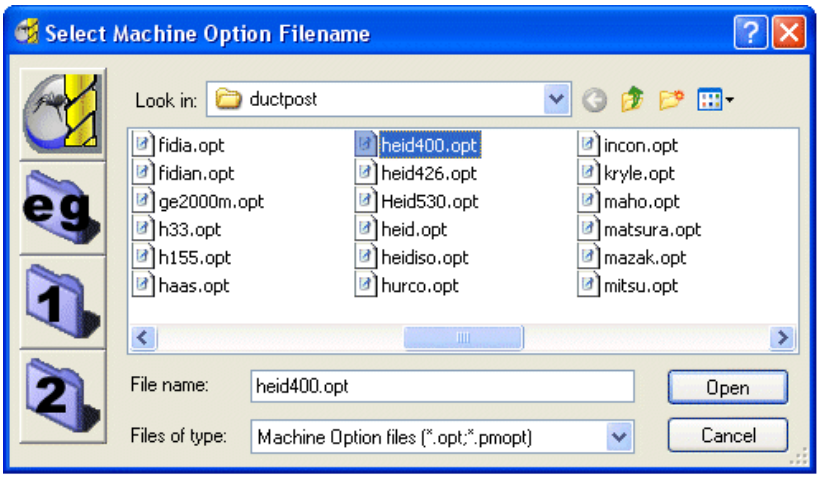
NC Preferences enable the user to control the content of output files for download to a Machine Tool.

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

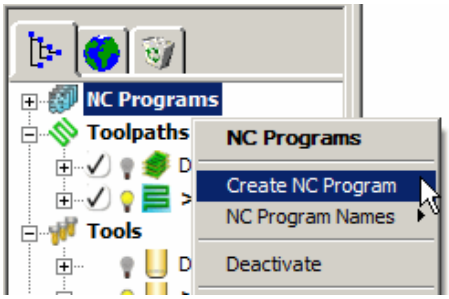


The **Output Folder** defaults to the location already defined in **Tools-Customise Paths**.

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

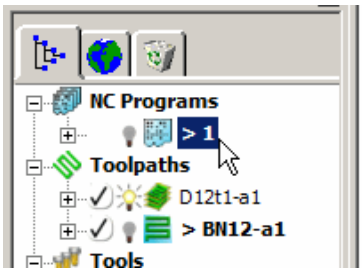


- On return to the **NC Preferences** form select the **Apply** tab to action the settings and then **Accept** the form.
- In the **explorer** right mouse click over **NC Programs** and from the sub-menu select **Create NC Program**.



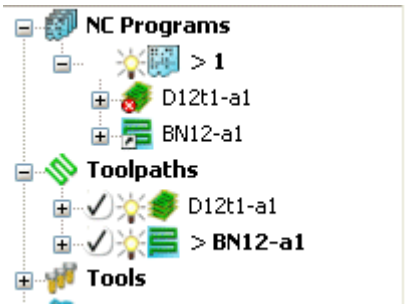
An empty **NC Program** will appear in the **explorer** ready to have machining strategies assigned to it. The ***NC Program*** form will also open in the Graphics area.

- 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**.



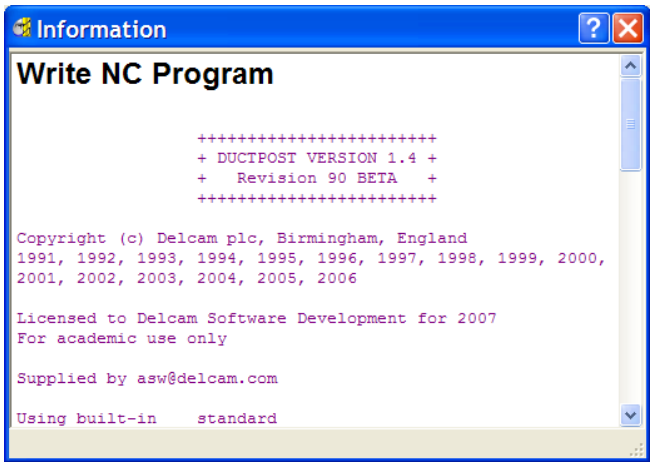
A copy of the toolpath name will appear in the **NC Program** indicating that it has been assigned as part of the output file.

- 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.

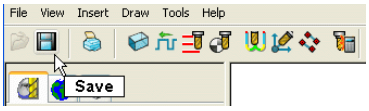
- In the **NC Program** form displayed in the graphics area, select the **Write** tab to start the post processing operation. The following Information form will open providing the user with a progress and confirmation summary.



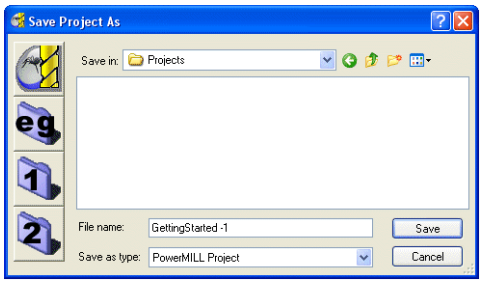
- Close both the **NC Program** and **Information** forms and using the ***windows explorer*** move to **C:\temp\NCPrograms** and note the existence of the **ncdata** output file **1.tap**.

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.



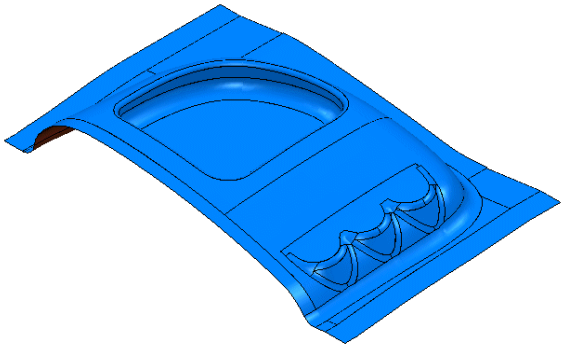
- In the **Save Project As** form, in the ***Save in*** box browse to **D:\users\training\COURSEWORK\PowerMILL-Profects\GettingStarted-1**

- 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:- **D:\users\training\coursework\PowerMILL\Facia-1**

Use the same tooling and strategies as applied in the previous worked example.

- 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 we created toolpaths using the default values whenever possible. We will now look at the machining process in more detail.
I particular:

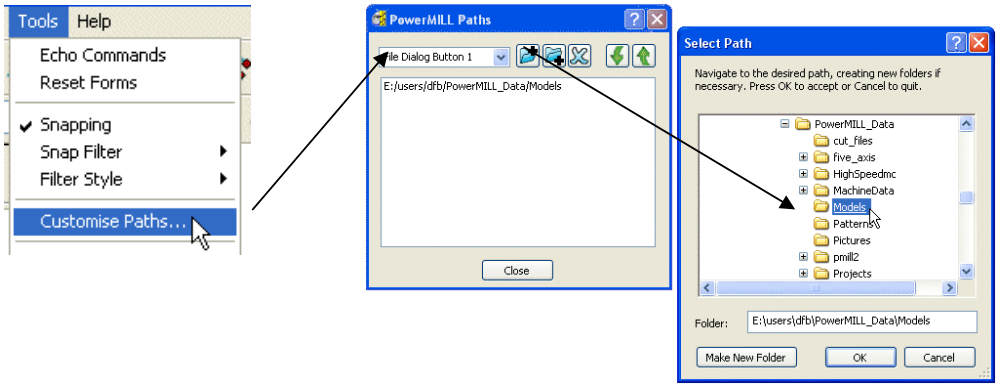
Setting up direct access to regularly used files
Orienting the job for machining
Detail Examination model features
Tool and Holder definition
Material Block Definition
Setting safe Z Heights

Setting up direct access to regularly used files

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 with **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**.



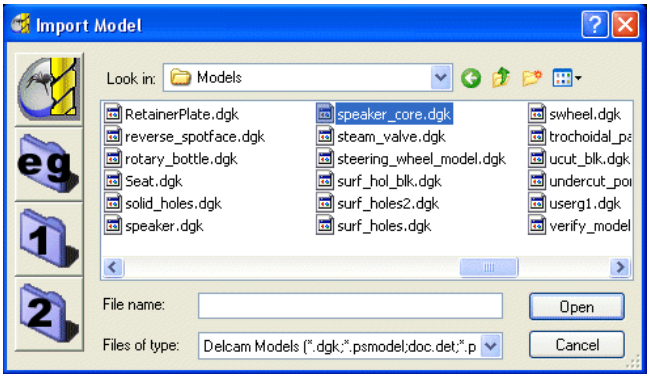
- Select the option ***File Dialog Button 1*** followed by the ***Add path to top of list*** icon  and browse to**:-**
  D:\users\training\PowerMILL_Data\Models.

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

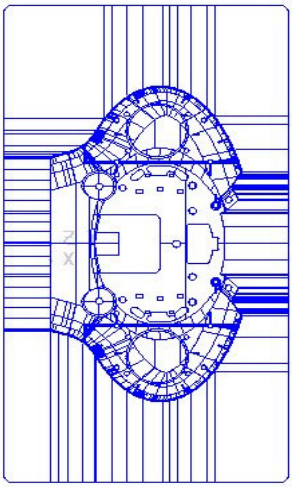
- From the **Main** pulldown menus select **File - Import Model**.
- Use **Shortcut Button 1** or browse to **D:\users\training\PowerMILL_Data\Models**



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.

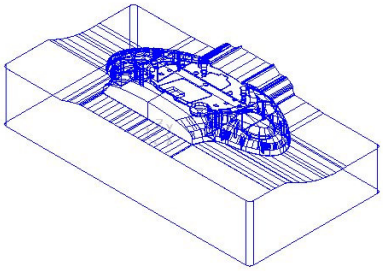


The model will be displayed (as shown) in the **PowerMILL** graphics area looking down the Z-axis with X aligned from left to right and Y from bottom to top.
In most cases the X dimension of a machine tool table will be greater than Y in which case the longer side of the component may be in excess of 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.

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. 

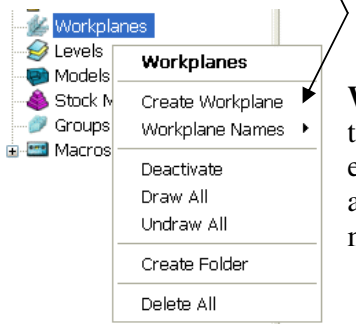


Although it is possible to rotate the actual component this is not generally regarded as the ideal approach.
An additional moveable datum (**Workplane**) will be created and rotated through 90 Degrees to effectively create the condition that the longer side of the component is parallel with the front 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.

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



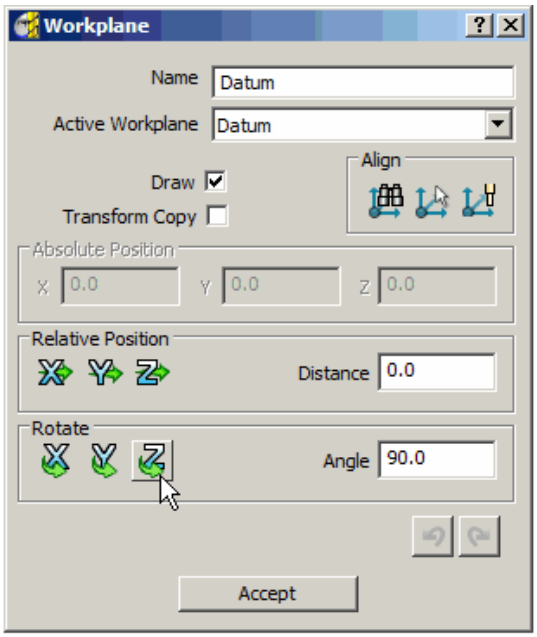
Workplanes are alternative or additional component datums that can be moved and/or orientated within the global environment. They are frequently used in **3 Axis** machining and are an essential item in the application of **3plus2** and **5-Axis** machining strategies.

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

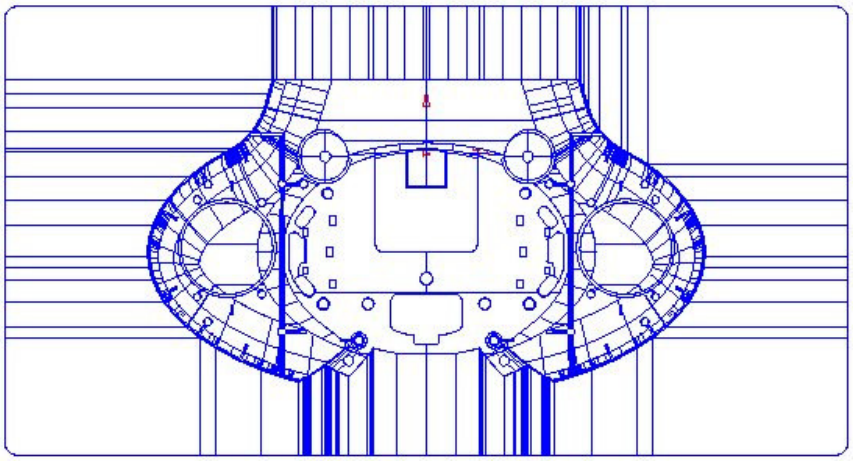
For easy identification it is recommended to appropriately rename **any** entity created in the **Explorer**.

- In the form modify the default ***Name*** to **Datum**.
- Set the ***Active Workplane*** to **Datum**.

The next step will be to rotate the new **Active Workplane** to indirectly re-orientate the **model**.



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



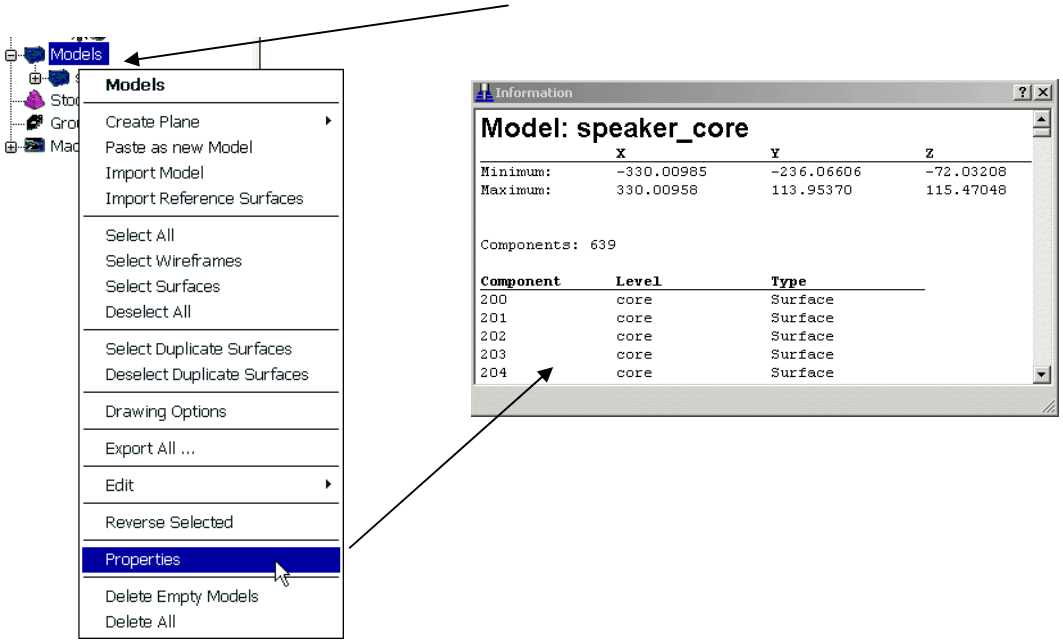
- Select a **View from top (Z)** and observe the effect of the rotated, **Active Workplane** providing a more suitable machining position for the **model**.

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.

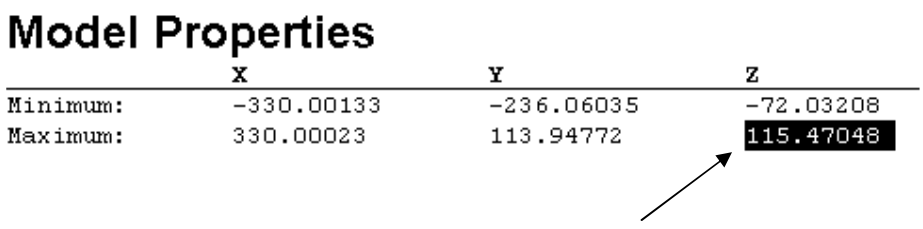
Examination of Model Properties

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**.



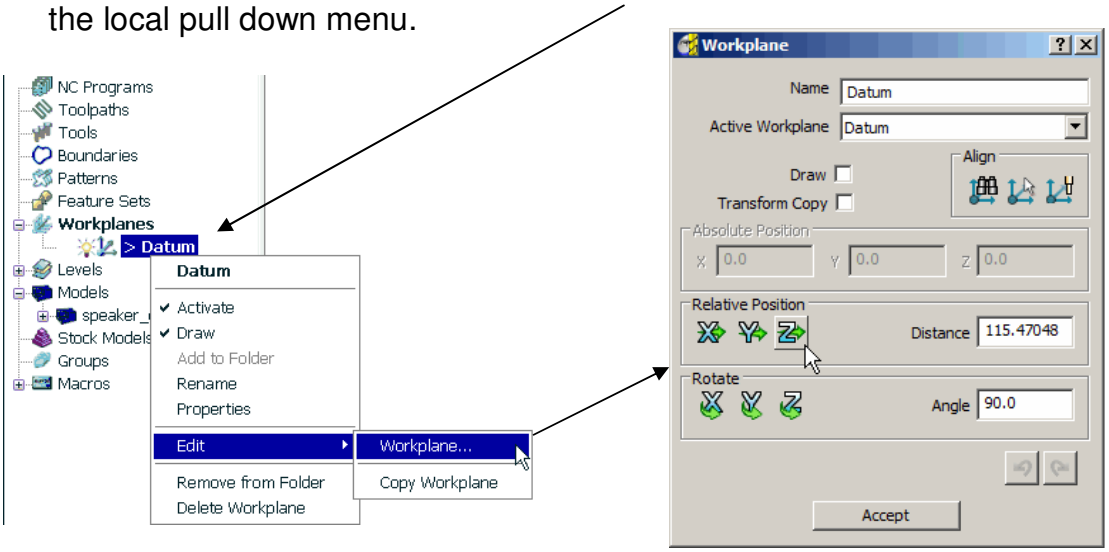
The values in this form can be **copied** (**Ctrl C**) and then **pasted** (**Ctrl V**) into other forms.
The **Workplane is moved** up in the **Z plane** so that it is situated at the maximum height of the model.
The form shows that the current maximum **Z** value is **115.47048mm**. It will be necessary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



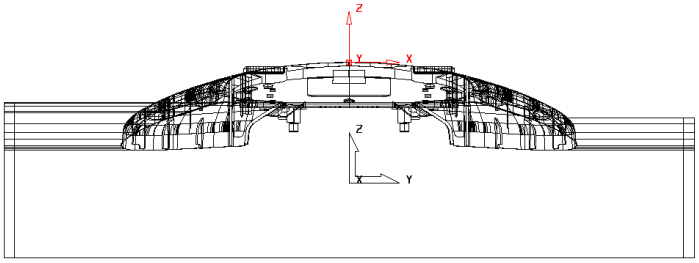
- Highlight the **Maximum Z** value (**115.47048**) by swiping over it with the left mouse button and press **Ctrl C** on the keyboard to **Copy** the value to the buffer.

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

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



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



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

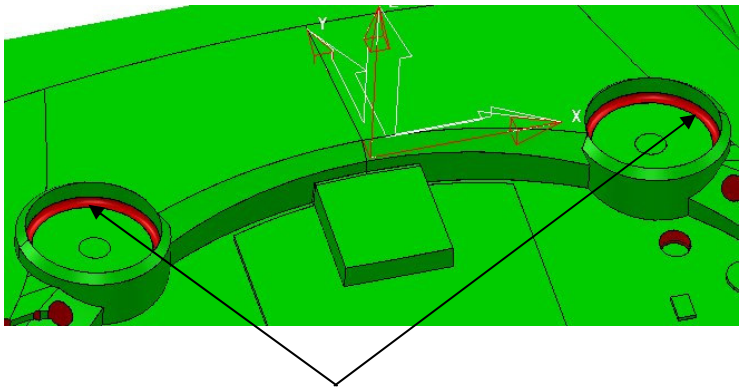
Minimum Radius and Draft Angle shading views.

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

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

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

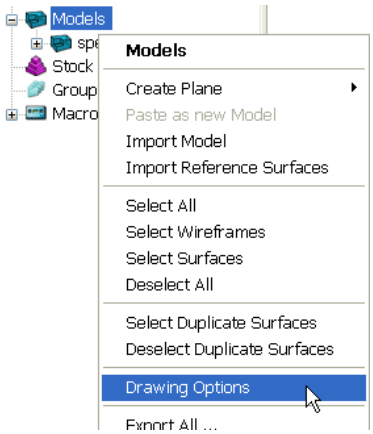
Any internal radii that are smaller than the specified **Minimum Tool Radius** will be shaded **RED**. The settings are located in the **explorer** - **Model** pull down menu in **Drawing Options**.
Examine the model to identify areas that are inaccessible to the specified radius (shaded Red).

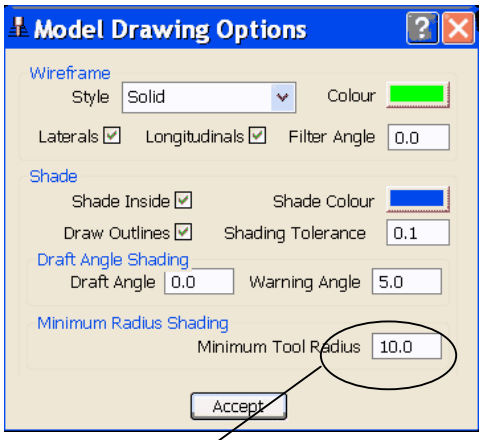


The two internal radii can be seen shaded **red** visually identifying that they will not be machined to their correct size if the default value tool was used.

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

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





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

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

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

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

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

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

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

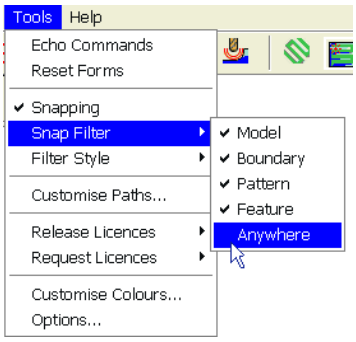
All of the red areas have disappeared and all that remain are green and yellow. If any red areas remain then these would indicate an undercut situation greater than –0.2 degrees.
The yellow areas indicate on or near vertical faces because the difference between the Draft and Warning Angles is so small.

- **Accept** the **Drawing Options** form
- Select the **Draft Angle Shade** icon  again to turn off the shading.
- Make sure the **Wireframe**  icon from the **View toolbar** is on so the model is displayed in Wireframe only.

Measuring the model.

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

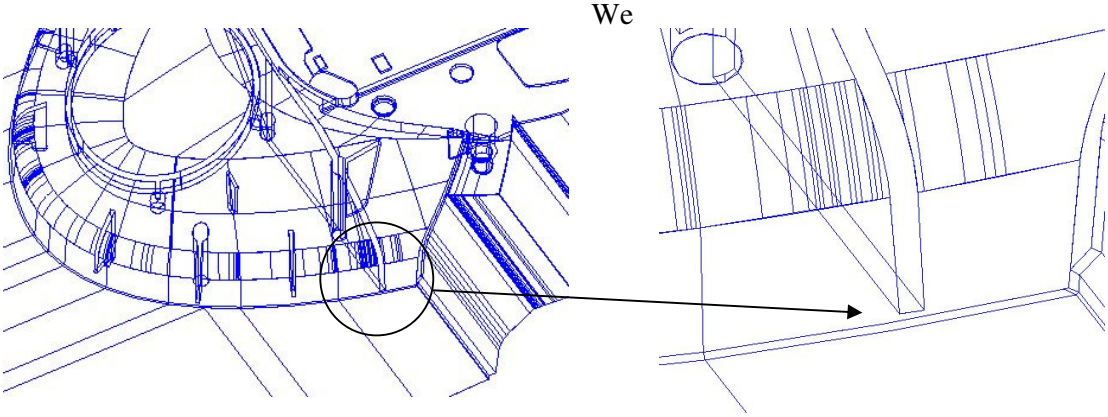
- From the top **Pull Down** menus select **Tools > Snap Filter** and use the left mouse key to **untick - Anywhere**.



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

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

We



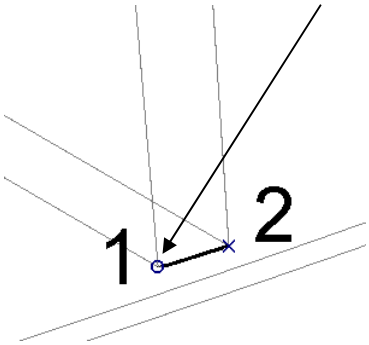
The gap at the bottom of the slot will be measured to determine its size and depth.

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

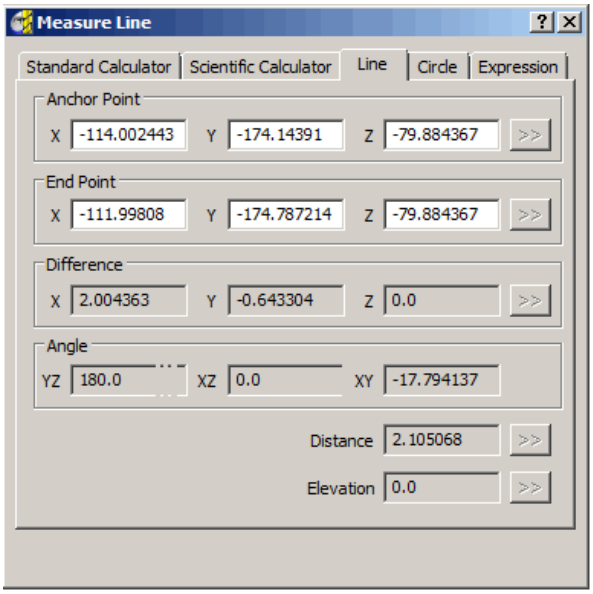
The Model Measurement for appears defaulted to **Line**. An Anchor Point is required to commence measuring.

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

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



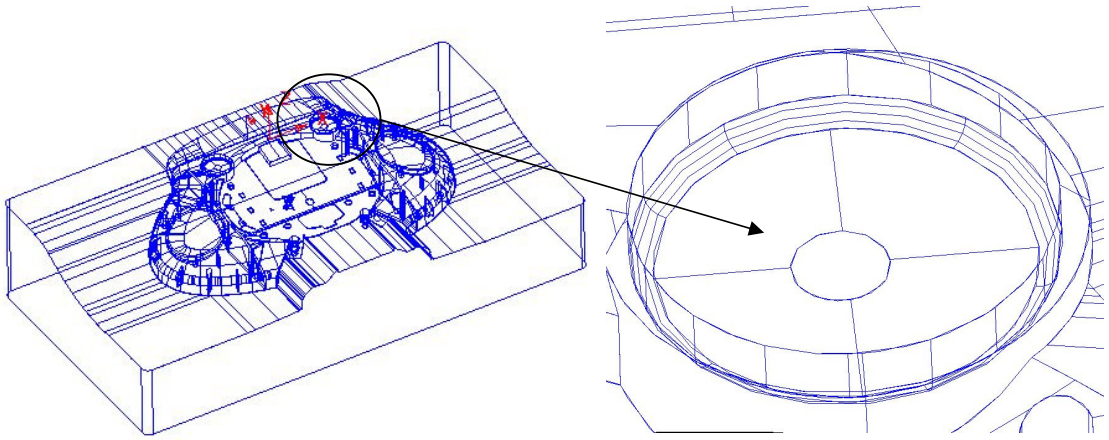
The **Anchor Point** is now selected and is represented by a small circle.
The x, y and z values seen in the above form are relative to the **Active Workplane** '**Datum**'.



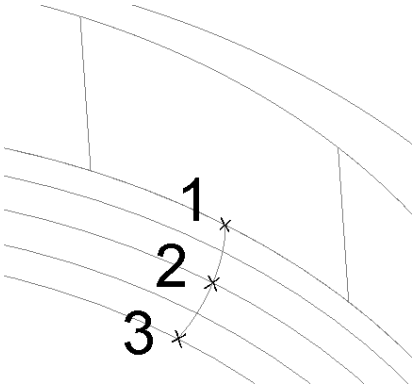
- Drag a window around (or snap) point **2** to obtain the final 'point to point' measurement information.

A temporary line appears connecting the two points, and the information relating to the line is displayed in the form.
Minimum Radius is measured via the **Circle** tab combined with snapping three points on the model.

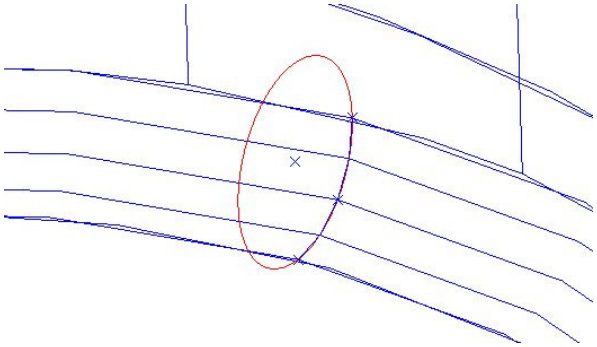
- Select the **Circle** tab on the **Model Measurement** form and zoom into the area shown below.

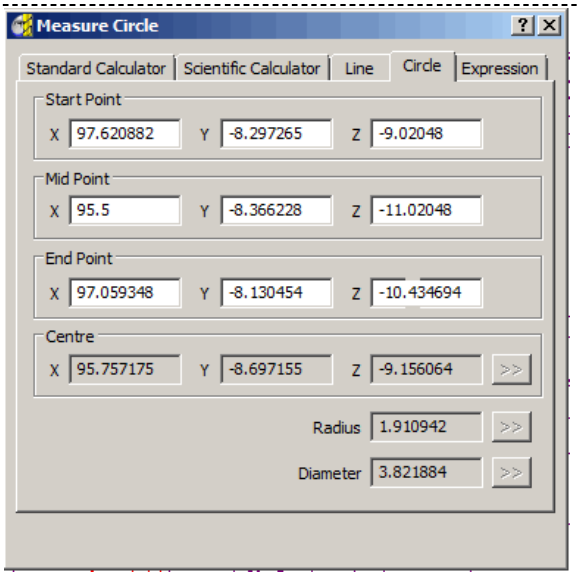


- Select **three points** along the arc either by dragging a small window around, or snapping onto each one as shown below.



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

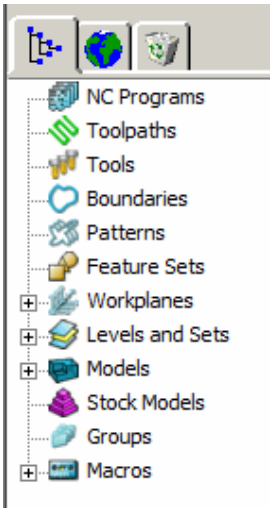




The **Circle** measurement form will display details of the arc as shown.

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

PowerMILL Panes



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

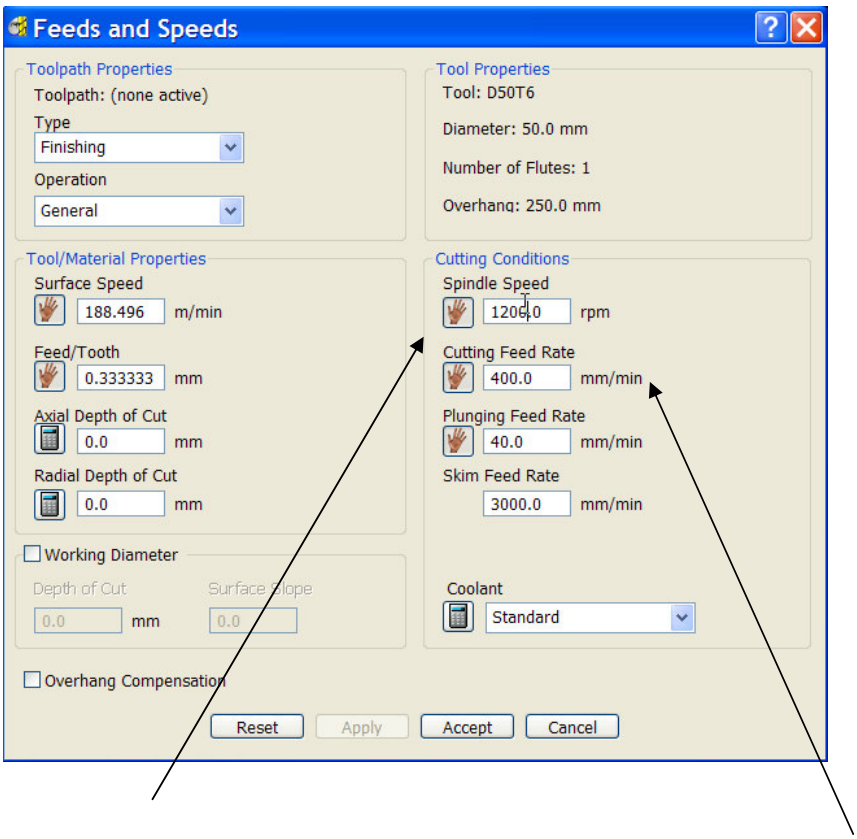
The standard format pane is denoted by the **PowerMILL** symbol and contains the explorer tree categorised into NC Programs,Toolpaths, Tools, Boundaries, Patterns, Feature Sets, Workplanes, etc.

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.

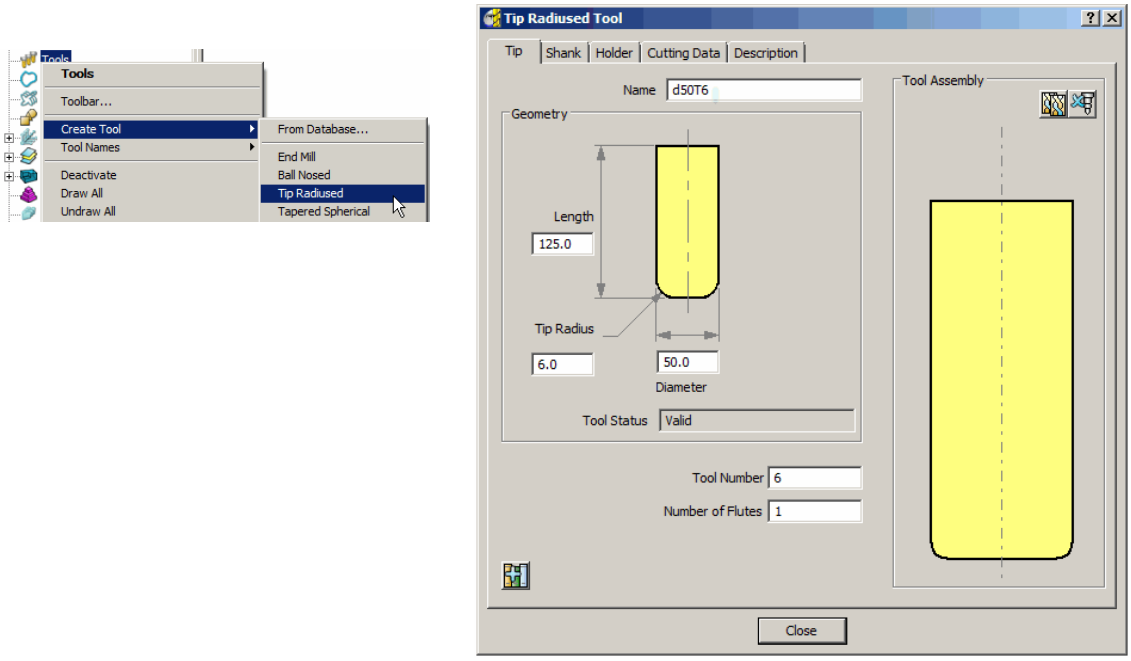


- In the **Cutting Conditions** section of the form, enter a ***Spindle Speed*** of **1200** and a ***Cutting Feed Rate*** of **400** (as shown above).
- **Accept** the form.

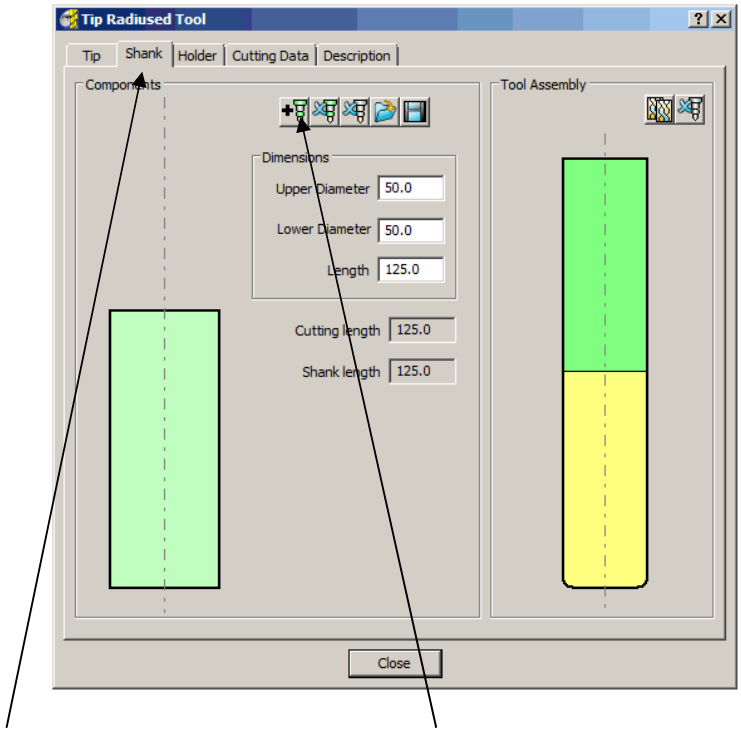
- ***Save Project As***:- **D:\users\training\COURSEWORK\PowerMILL-Projects\Spkr-Core** (but do not close).

Tool and Holder Definition

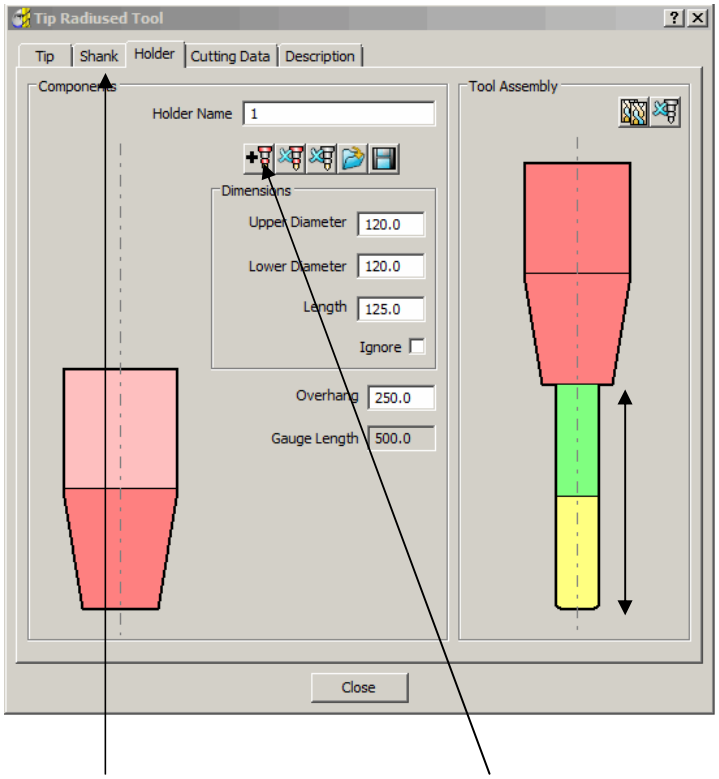
- **Open** the ***Tip Radiused*** tool form.



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



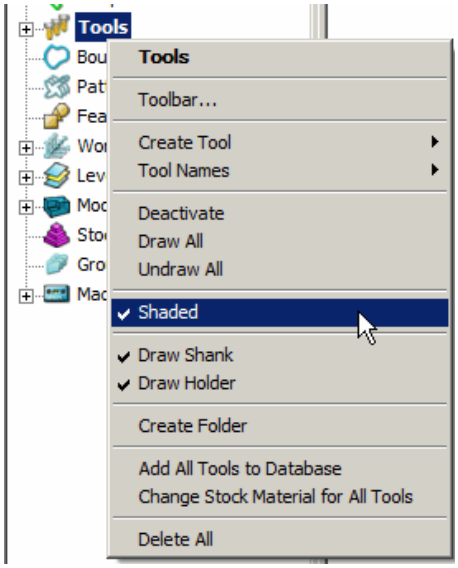
- In the ***Shank*** form, click the ***Add a shank component*** icon and enter ***Upper/Lower Diameter*** **50** and ***Length*** **125**.



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

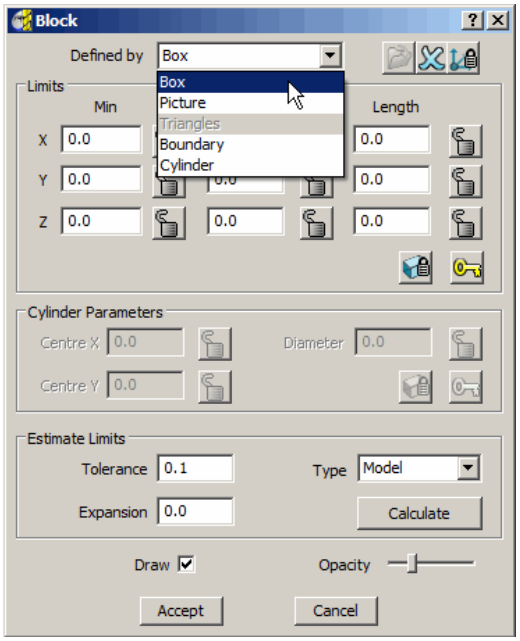
- In the ***Holder*** form, click the ***Add holder component*** icon entering ***Upper Diameter 120*** - ***Lower Diameter*** **80** - ***Length*** **125** and ***Overhang*** **250**.
- In the ***Holder*** form, click the ***Add holder component*** icon entering an ***Upper/Lower Diameter*** **120**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

The way 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.



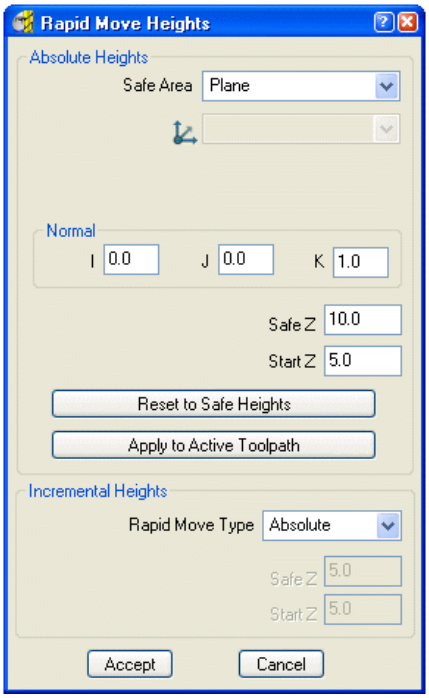
The default option for **Block** is **Box** (A rectangular volume). Other options include **Cylinder** (a Cylindrical volume), a **Triangle** model (Casting) available in PowerMILL PRO only, and **Picture/Boundary** (Extruded 2D wireframe contours).

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

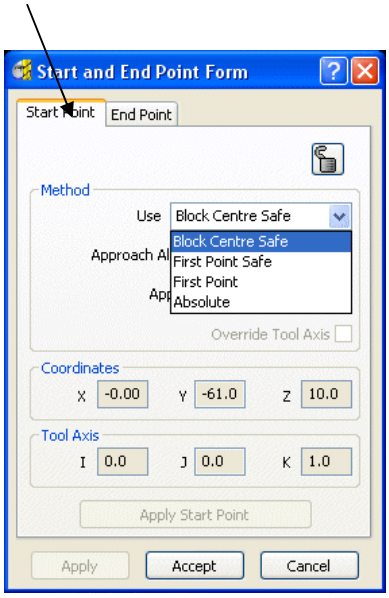
- In the **Rapid Move Heights** form click the **Reset to Safe Heights** tab and ensure that in the ***Incremental Heights*** area that ***Rapid Move Heights*** is set to the **Absolute** option.



The **Absolute** option enables ***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**.



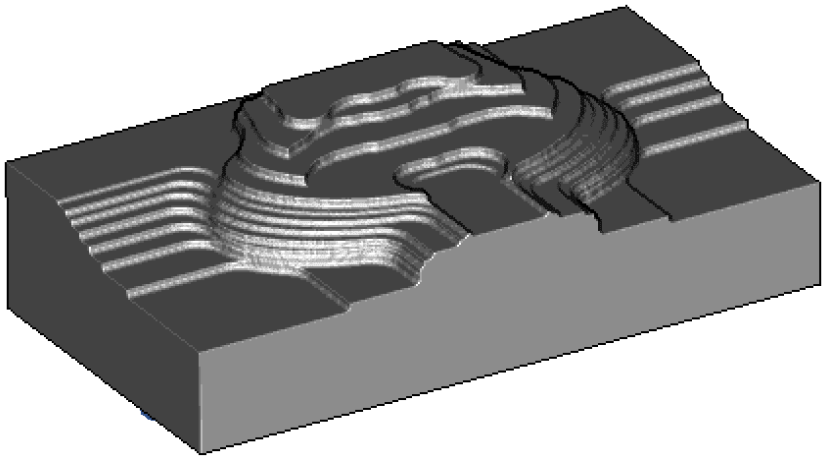
- Select ***Save Project*** to update the saved ***PowerMILL*** **Project** (***Do not close*** the **Project**).

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

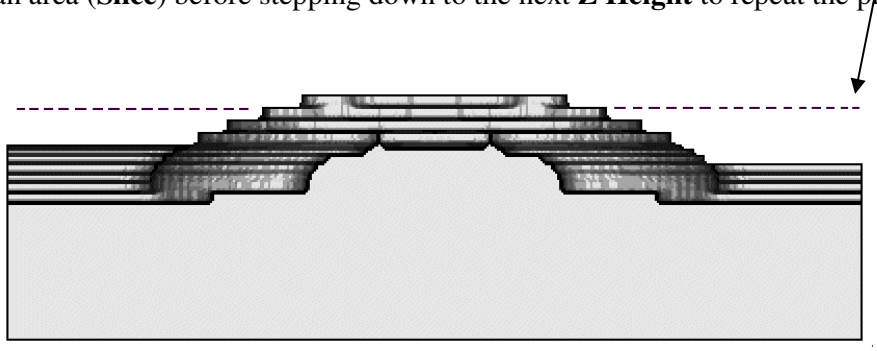
3. 3D Area Clearance

Introduction

The main strategies for roughing a **3D** component **Model** are called **3D Area Clearance**. These provide a choice of 2D material removal methods, which progressively machine the area (**Slice**), up to the component contour, down a sequence of user-defined **Z Heights**. There is also a similar group of strategies, **2.5D Area Clearance** for use, exclusively with**PowerMILL 2.5D Feature Sets** (covered in Chapter 9:- **2D Feature Set machining**).



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.



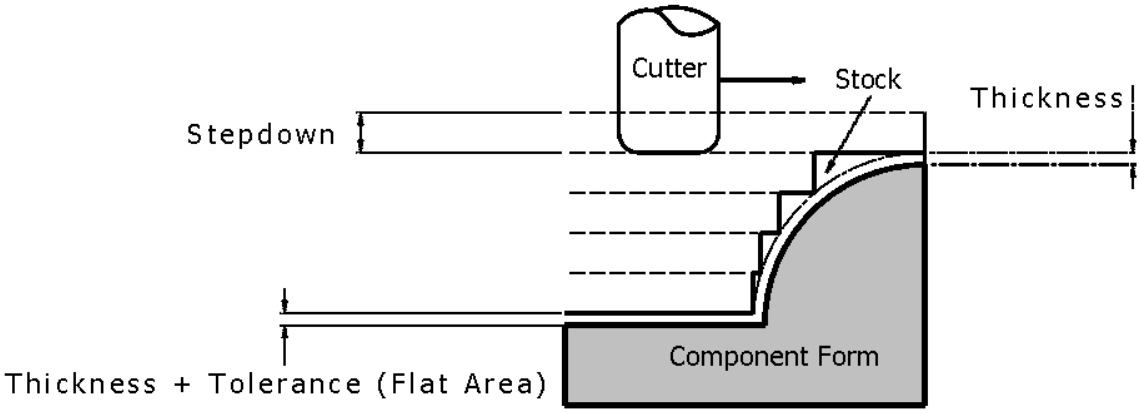
For some components a secondary **Area Clearance** strategy is applied using the **Rest Machining** options in conjunction with a smaller roughing tool. This will locally remove pockets of excess material inaccessible to the original **Reference Toolpath** or **Stock Model**. This will reduce the degree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

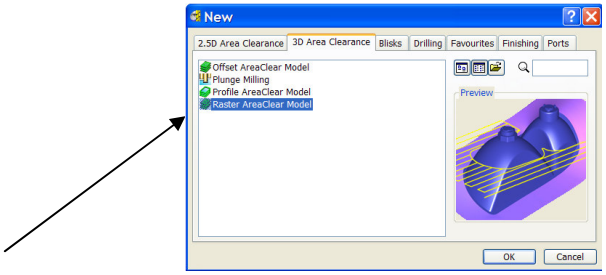
Thickness is the amount of extra material specified to remain on the work-piece after machining. This can be applied generally (as shown), or independently as separate **Radial** and **Axial** values within the machining options.



Raster Area Clear example

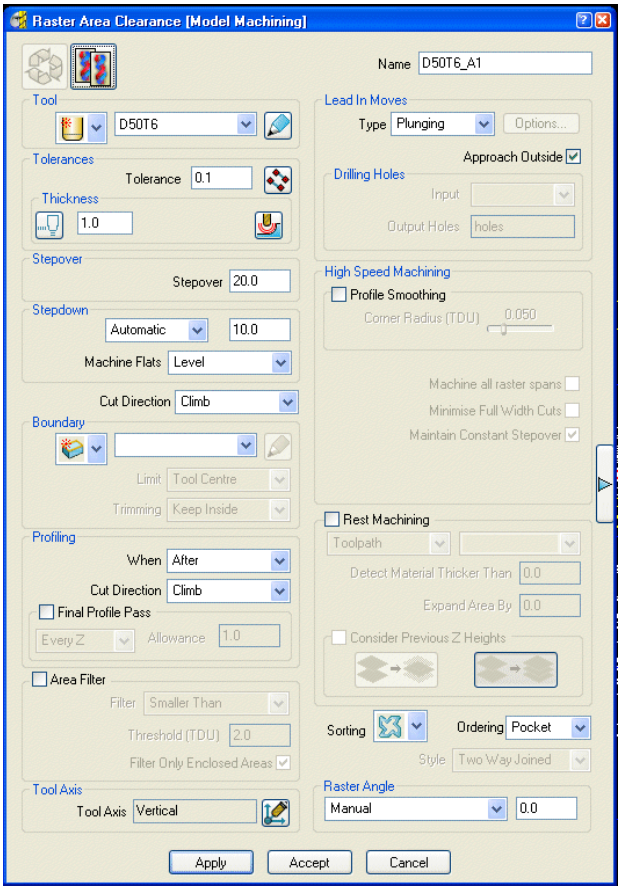
The **Raster Area Clear** strategy follows a series of ***linear moves*** across the **Block** limiting to the **Component form** at the ***active Z height***. It then, (if required) performs a ***Profile pass*** around the component to leave a constant **thickness** around the **Slice**. Other options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

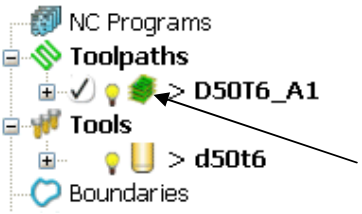
- From the **Main toolbar** select the **Toolpath Strategies** icon. 
- Select **3D Area Clearance** tab.



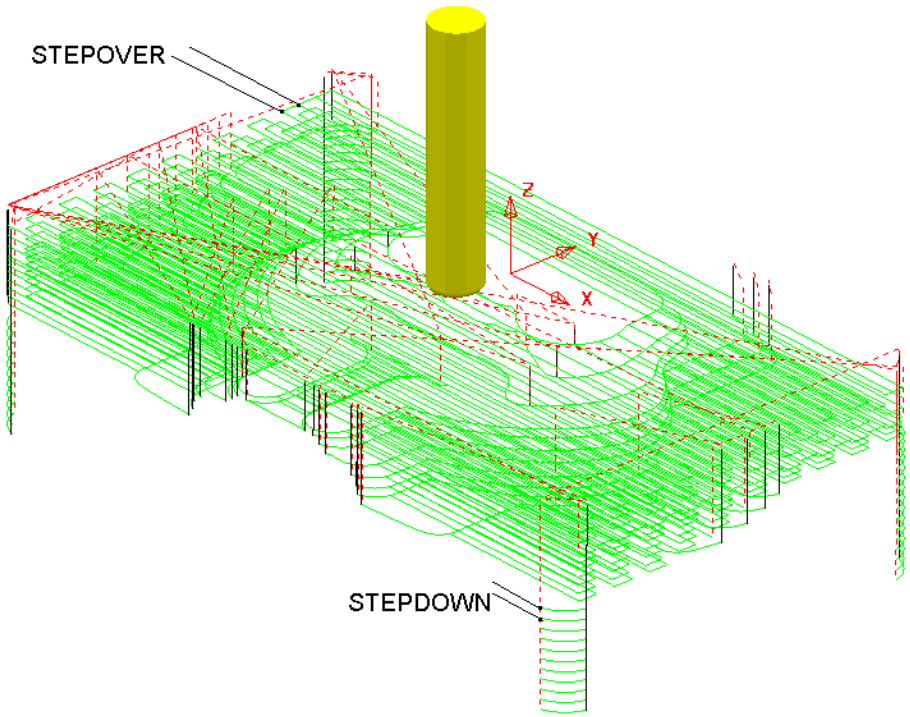
- Select the option **Raster AreaClear Model** to open the following form.

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





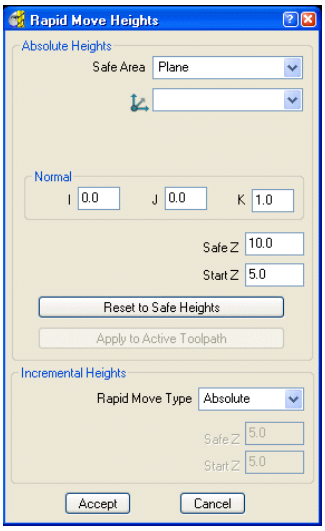
As soon as the **Raster Area Clearance (Model)** form is opened an unprocessed **Toolpath** appears in the **Explorer** (the default name has been changed to **D50T6_A1**).

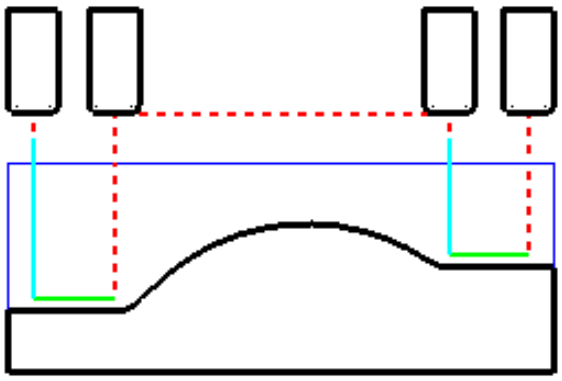


The Toolpath icon can now be double-clicked to **Activate** and de-activate the toolpath.
The **+** symbol can be clicked to open a full record of the data used to create the toolpath.
The dotted red lines represent **Rapid** moves and the light blue lines are the **Plunge** feed moves.

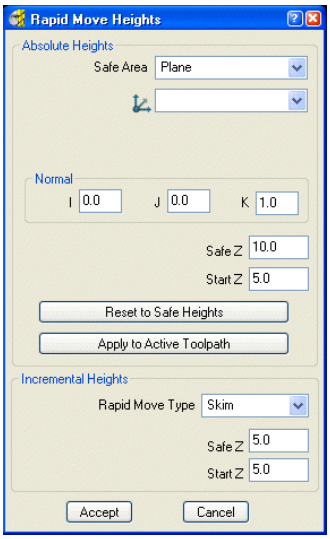
Rapid Move Heights in detail

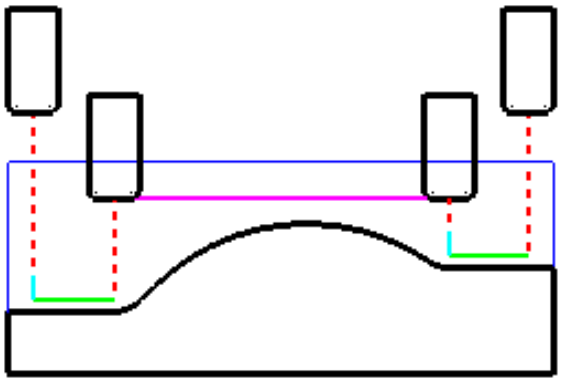
The **Rapid Move Heights** form provides **Safe Z** and **Start Z** input boxes. Suitable values are entered to define a safe height (**Safe Z**) at which a tool can safely perform, horizontal **Rapid Moves** above the model as well as (**Start Z**) where a **Rapid** ***plunge move*** changes to a ***plunge*** **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). These ***Default*** distances will be **Safe Z**, **10** and **Start Z**, **5** and are applied with the ***Incremental Heights*** options switched off (set to **Rapid Move Type** -**Absolute**).



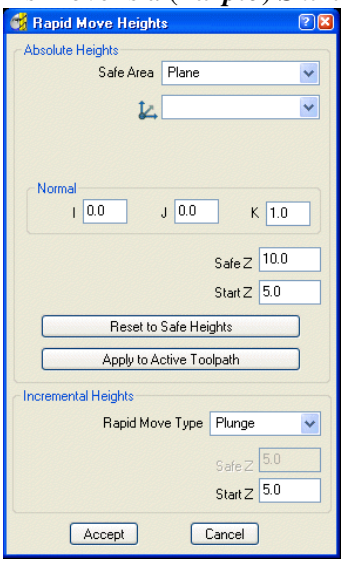


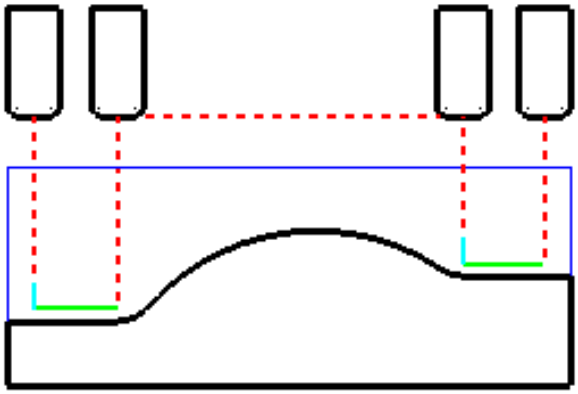
Absolute (default) sets the **plunge feed rate** to apply at a specific height above the job. This is more predictable and reassuring for the ***machine tool operator*** but the time take by the non-cutting (air) moves is inefficient especially in the case of large, deep components.
In the section of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





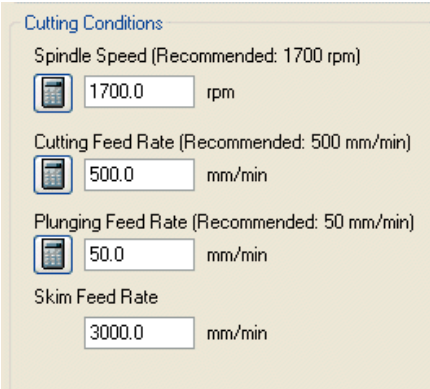
Plunge applies the ***rapid feed rate*** all the way down to an ***Incremental*** **Start Z** measured from the ***full plunge depth*** at which point the slow **plunge feed rate** 'cuts in'. The **Plunge** option differs from **Skim** in that all **rapid link** moves occur at the ***Absolute*** **Safe 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 value **(G1) Feed Rate** moves:-



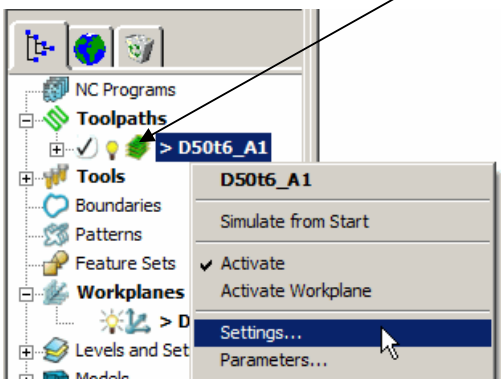
Green/Orange - Toolpath ***Cutting Feed*** moves.

Pale Blue - Toolpath ***Plunging Feed*** moves.

Purple - Toolpath ***Skim Feed*** moves.

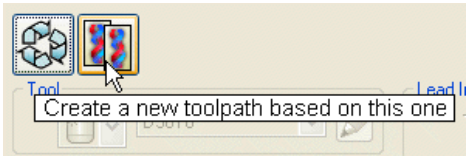
Also, ***local areas*** of a **toolpath** can be assigned with additional **Cutting Feed Rate** values via the **Toolpath Editing** options (***See Chapter 8***) as a **percentage** of the nominal value. **PowerMILL** will assign a ***different colour*** to areas of the **toolpath** edited to have a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

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

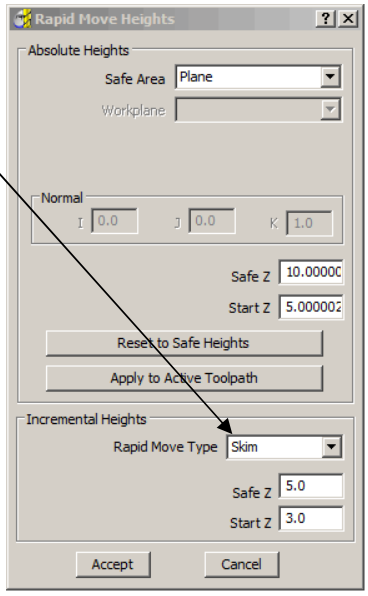


Note the toolpath can also be activated or deactivated from the **Activate** switch in the pulldown menu.

- Select **Settings** to reopen the **Raster AreaClear Model** form.
- Select the '***make a copy***' of the **toolpath** icon (shown arrowed below).

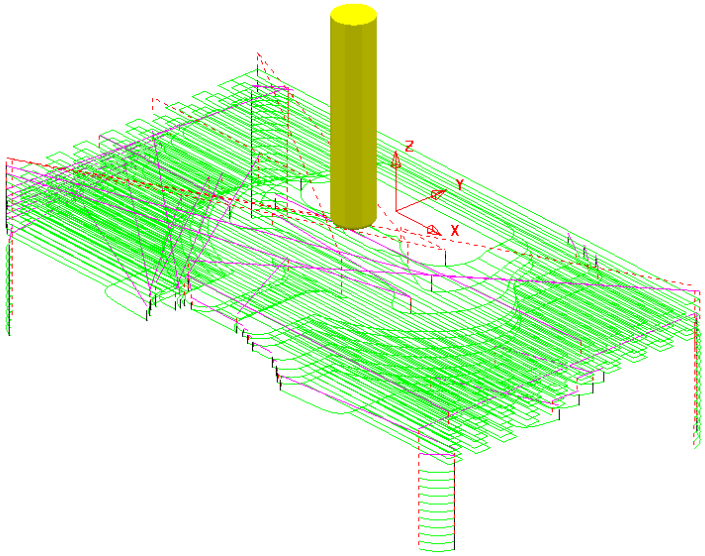


- Select the **Rapid Moves Height** icon  from the main toolbar.
- Select the **Skim** option and change the **Safe Z** to **5** and **Start Z** to **3** as shown below.



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

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

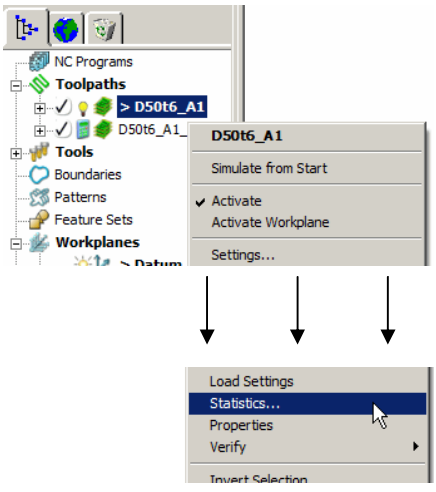


The toolpath has been recycled with the **Skim** option set in the **Rapid Move Heights** form and now uses Incremental **Safe Z** and **Start Z** values.

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 **8 hrs**.

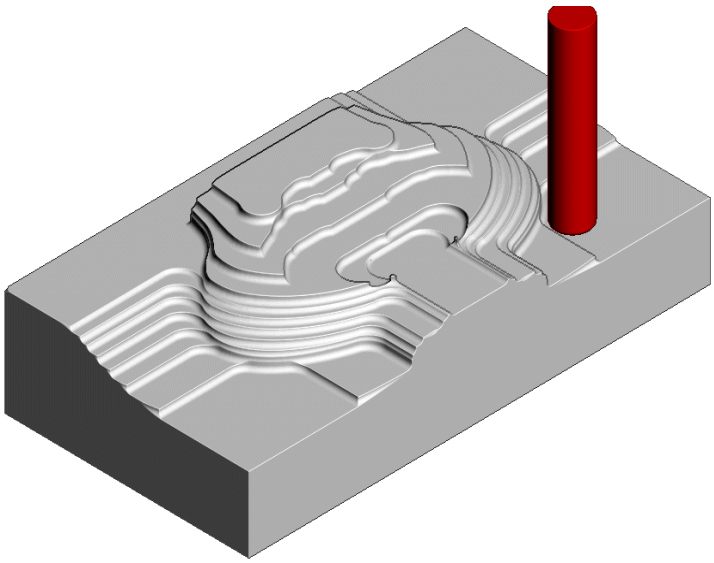
- **Activate** the second ***toolpath*** (**D50T6_A1_1**) and obtain the **Statistics**.

Note; In this case the ***total machining time*** will be around **4.5hrs**.

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.

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.

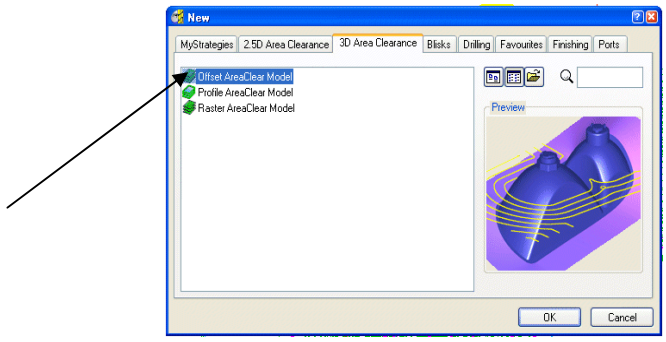
The **Project** (**D:\users\training\COURSEWORK\PowerMILL-Projects\Spkr-Core**) has now been updated to include the **3D Area Clearance** toolpaths.

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

Offset Area Clear example

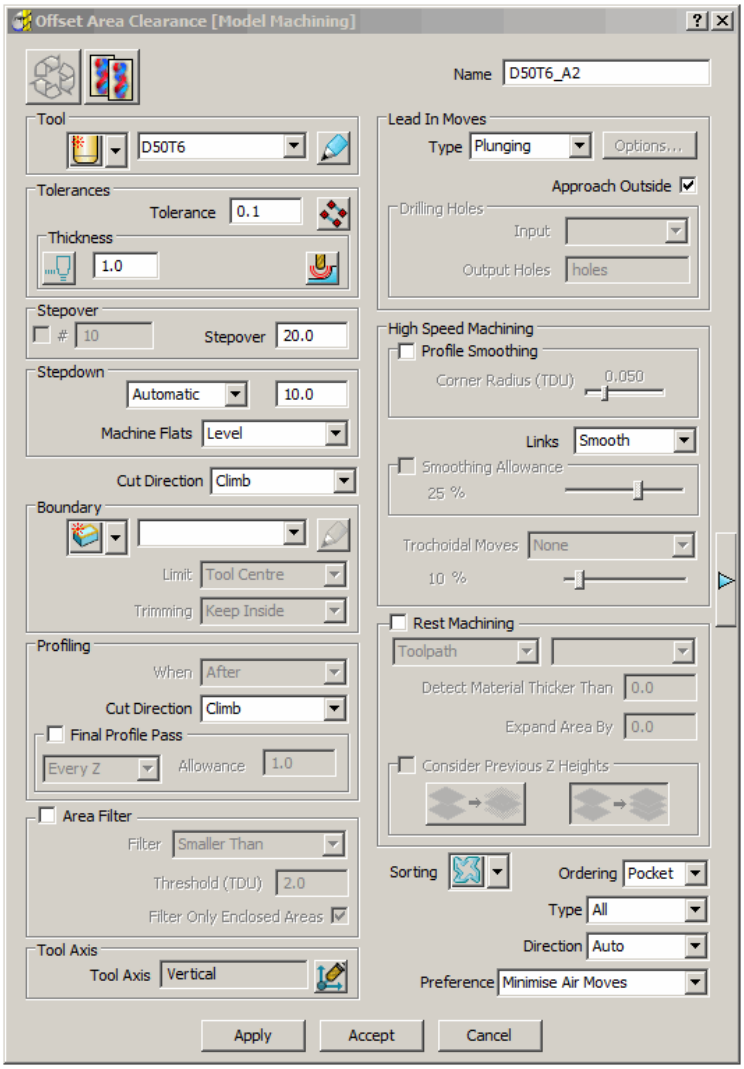
The **Offset Area Clear** strategy immediately follows the contour of the both the **Block** and **Component form** at the ***active Z height***. It then progressively offsets across into any remaining areas of stock.

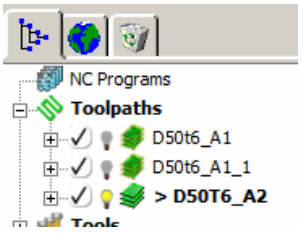
- From the **Main toolbar** select the **Toolpath Strategies** icon.
- Select **3D Area Clearance** tab.



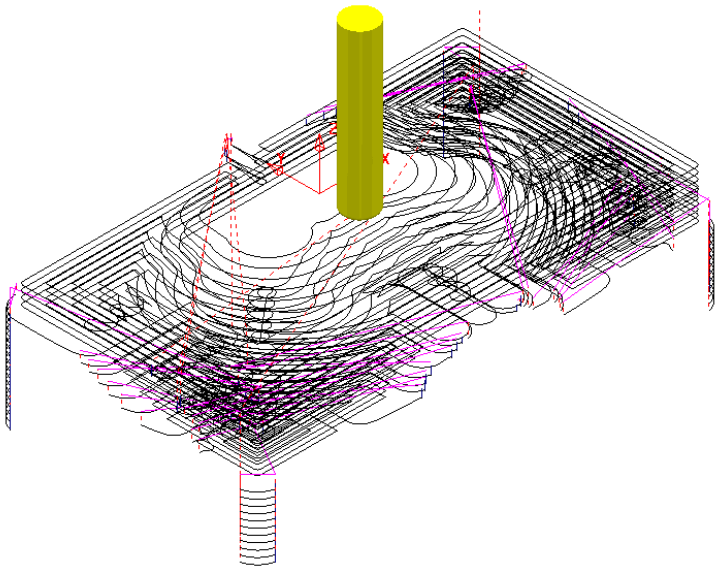
- Select the option **Offset AreaClear Model** to open the following form.

- Enter the ***Name*** **D50T6_A2**.
- Set **Stepover** to **20**.
- Set **Stepdown** to **10**.
- Keeping all other values as default, **Apply** the form.
- After the processing is complete **Cancel** the form.

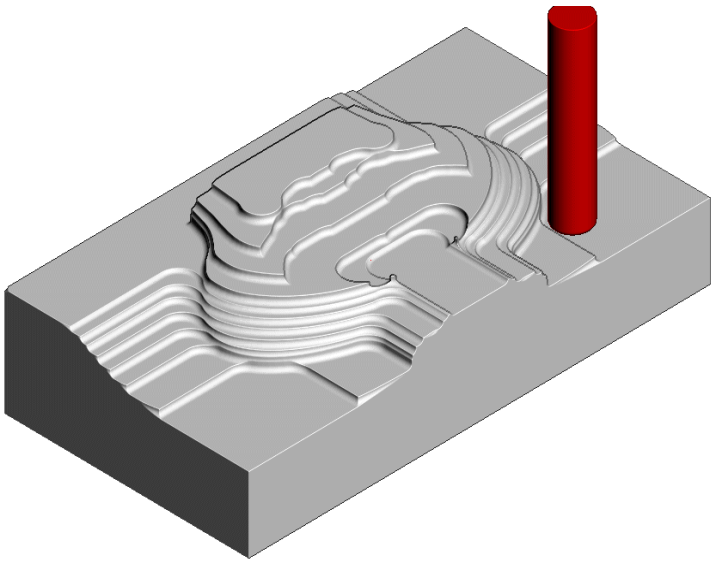




As soon as the **Offset Area Clearance (Model)** form is opened an unprocessed **Toolpath** appears in the **Explorer** (the default name has been changed to **D50T6_A2**).



- 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.

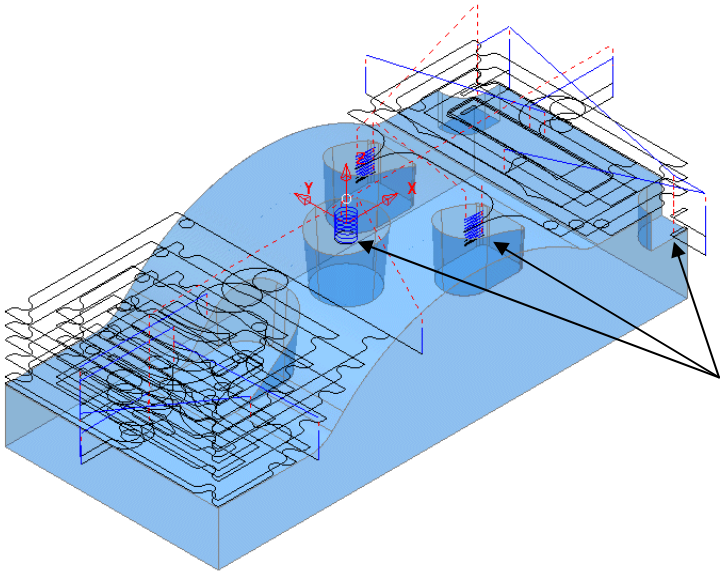
The **Project** (**D:\users\training\COURSEWORK\PowerMILL-Projects\Spkr-Core**) has now been updated to include the **3D Area Clearance** toolpaths.

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

Area Clearance - Area Filter

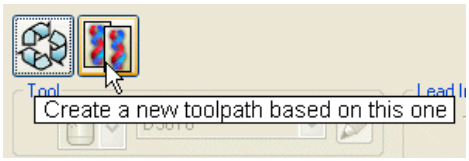
This option is designed to filter out machining of confined areas that would involve small movement of the cutting tool. In the following example it is applied to prevent a relatively large tool attempting to plunge into a deep pocket area where a smaller tool would be a better choice.

- **Open** the '*Read Only*' **Project**:- **D:\users\training\PowerMILL_Data\Projects\MountingBlock-Start**
- **Save Project As**:- **D:\users\training\COURSEWORK\PowerMILL-Projects\MountingBlock**
- **Activate** the ***Toolpath*** named **No-AreaFilter**

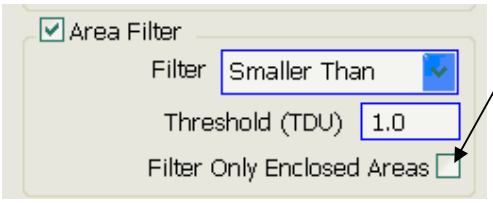


The existing toolpath is allowing the cutter to machine into confined areas. This is resulting in sudden sharp changes of cutting direction and excess loading on the tool.

- 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** icon (shown arrowed below).

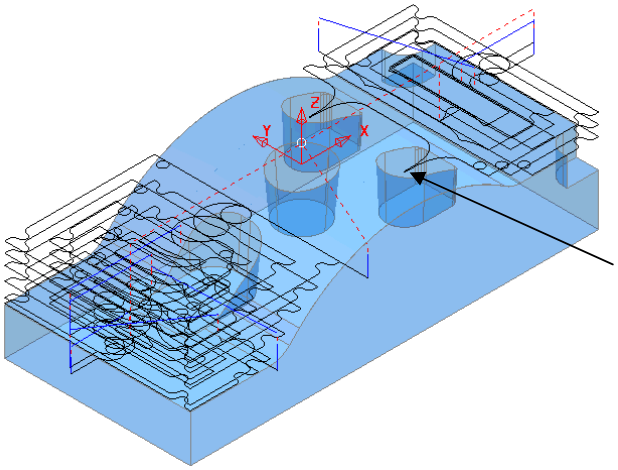


- ***Rename*** the copy of the toolpath as **AreaFilter**.
- In the form locate the **Area Filter** section (lower left corner) and input the settings as shown below (Note; **Filter Only Enclosed Areas** is unticked).



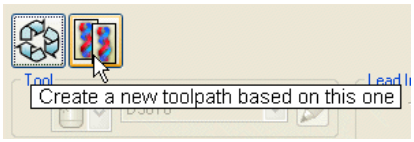
All tooltracks spanning a distance less than the Tool Diameter (TDU) will be filtered out and not appear in the final Toolpath.
The effect of unticking **Filter Only Enclosed Areas** means that the ***recessed areas*** running out to the **Block** will be included in the filtering process.

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

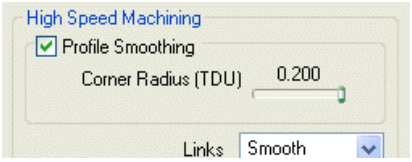


Area Filter has been applied to prevent the cutter attempting to machine into confined areas.
There are however a couple of undesirable spikes on the unfiltered part of the toolpath. These can be reduced by applying the **Profile Smoothing** option.

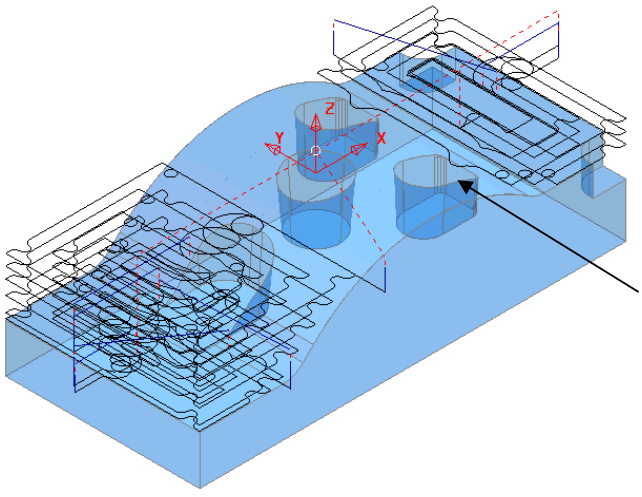
- Right click on the **toolpath *named* AreaFilter** and from the local menu select **Settings** to open the original **Offset Area Clearance** form.
- Select the '***make a copy***' of the **toolpath** icon (shown arrowed below).



- ***Rename*** the copy of the toolpath as **AreaFilter-ProfileSmooth**.
- Keep the same **Area Filter** settings switched on, and ***tick*** the **Profile Smoothing** box with the **Corner Profile** slider set to **0.200**.



- **Apply** 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.

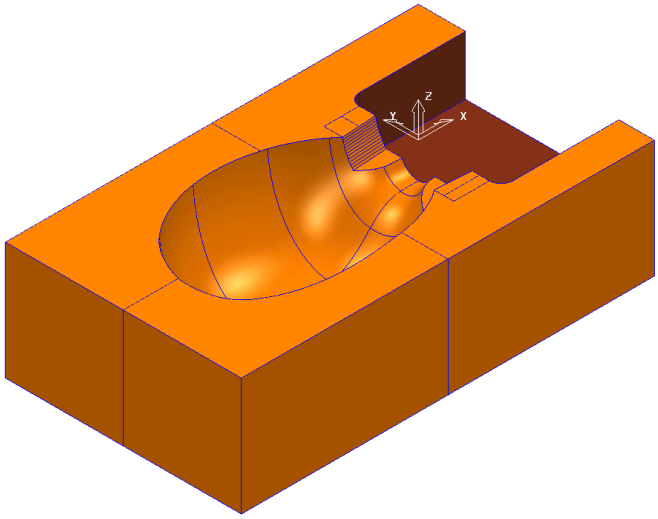
The **Project** (**D:\users\training\COURSEWORK\PowerMILL-Projects\MountingBlock**) has now been updated to include the **3D Area Clearance** toolpaths.

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

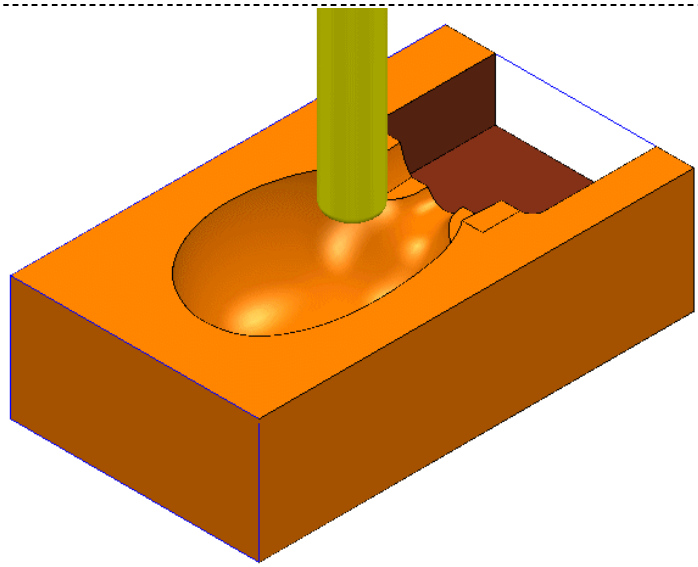
Rest Roughing - Area Clear example

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.
The **Rest Roughing** option creates a **Roughing Strategy** using a **smaller diameter tool** ***referenced*** to a previously created ***machining strategy*** such that ***tool tracks*** will only be produced locally within the remaining material (stock).

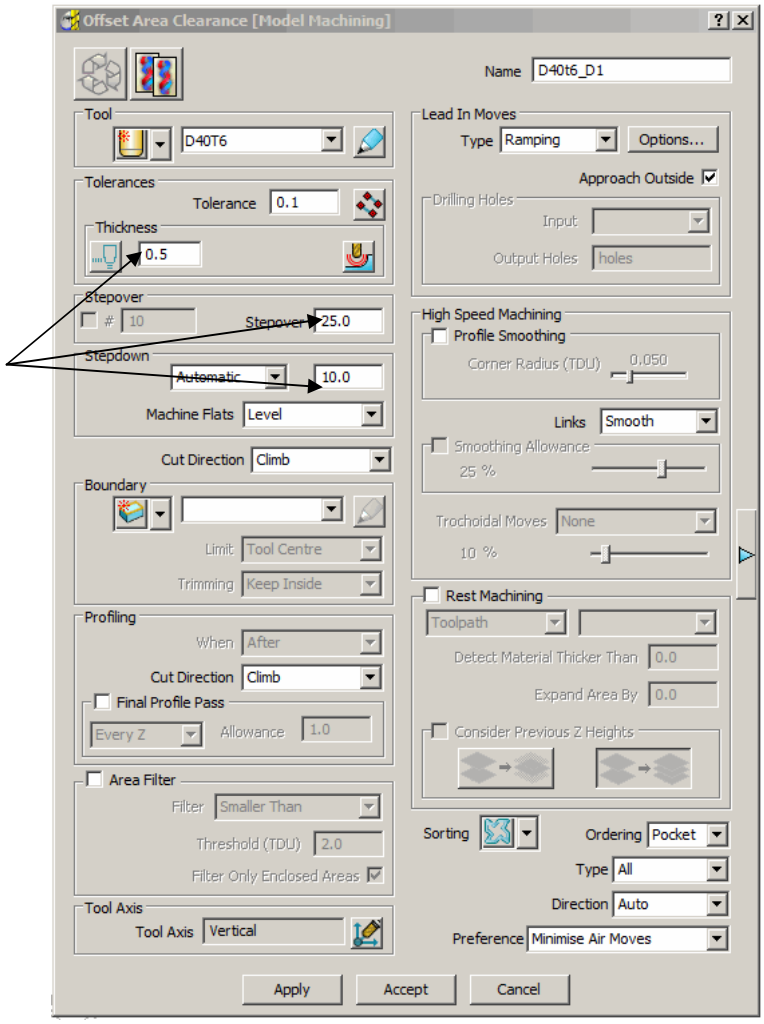
- **Import** the model **WingMirrorDie.dgk** from **D:\users\training\PowerMILL_Data\models**.



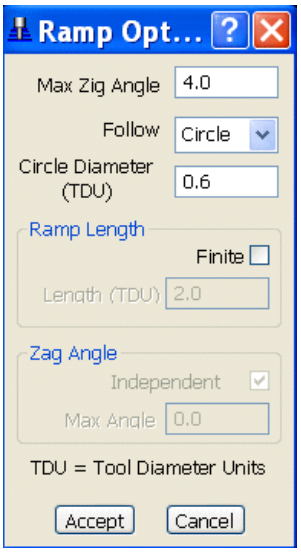
- Create a **Tip Radiused** tool of **Dia 40 tiprad 6** and ***Name*** **d40t6**.
- From the **Main toolbar** open the **Block** form and **Calculate** using ***Defined by*** **- Box** to the full model dimensions.
- Reset the **Rapid Move Heights** and set the **Incremental** moves to **Skim**.
- In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



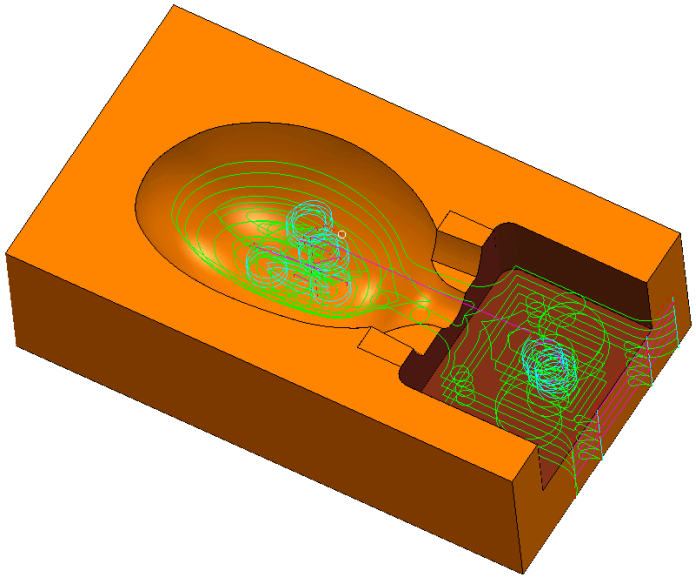
- From the **Main** toolbar select the **Toolpath Strategies** icon .
- In **3D Area Clearance** select the option **Offset AreaClear Model** to open the following form.



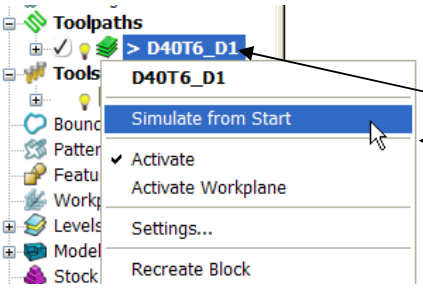
- Enter ***Name*** **D40T6_D1**.
- Select **Ramping**.
- Select ***Options***
- Enter:- ***Max. Zig Angle*** **4**, ***Follow*** **Circle**, and ***Circle Diameter*** **0.6** As shown below.



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



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



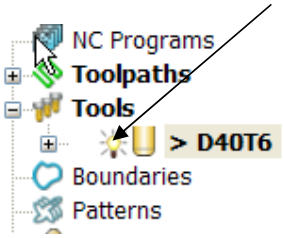
- RMB on the toolpath and select **Simulate from Start** to bring up the Simulation toolbar

The **toolpath simulation** toolbar will open.

- 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.



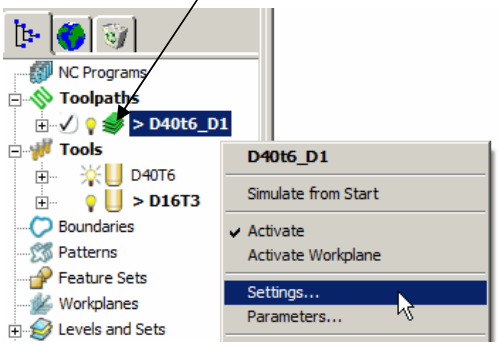
NB.Undrawing the tool will speed up the simulation.

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

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

Rest Machining 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**.



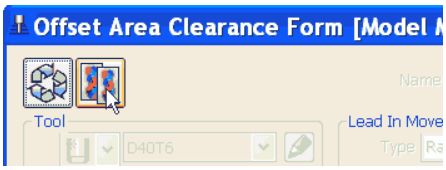
- Select **Settings** to reopen the **Offset AreaClear Model** form.

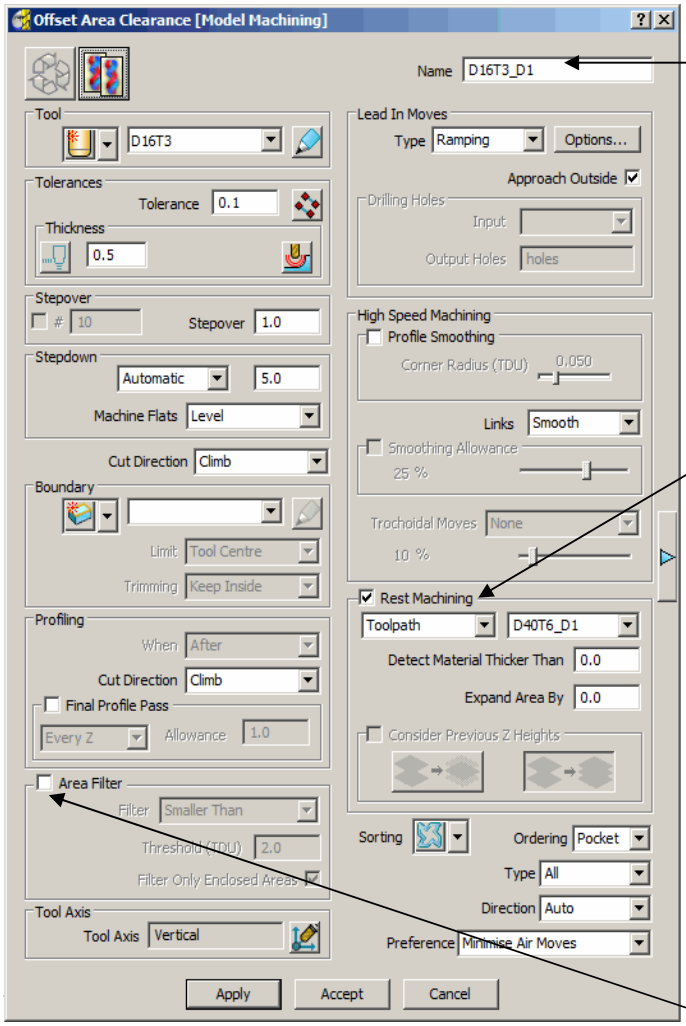
- Select the '**Copy toolpath**' icon.

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

ew toolpath based on this one

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





Enter a new name **D16T3_D1**.

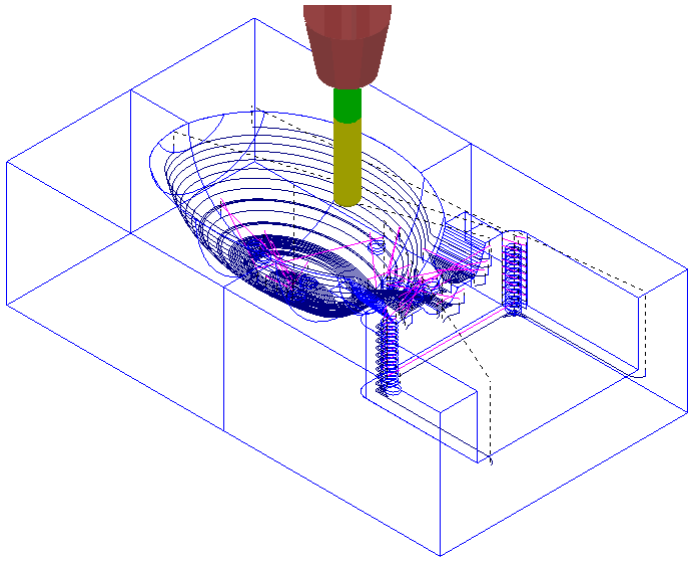
Enter:- ***Thickness*** **0.5** ***Stepover*** **1.0** ***Stepdown*** **5.0**

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

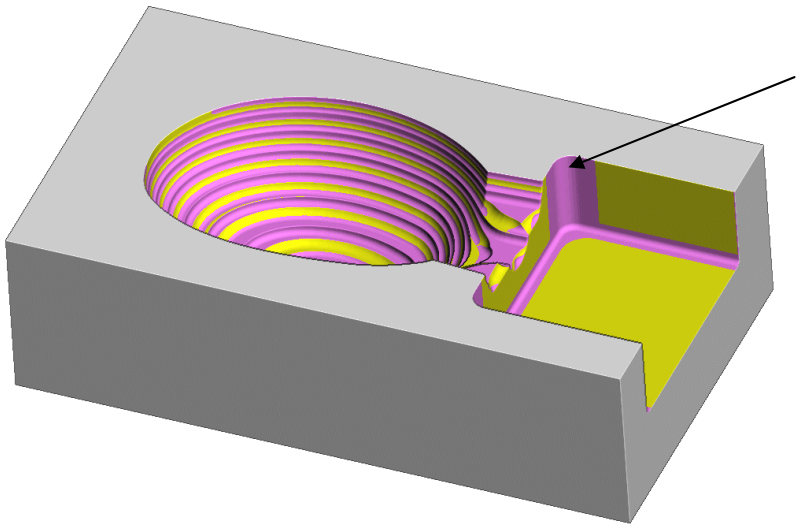
Type **All** is the only available option for the basic **PowerMILL** license.

Untick **Area Filter**.

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



- ***Save Project*** *as:-*
  D:\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

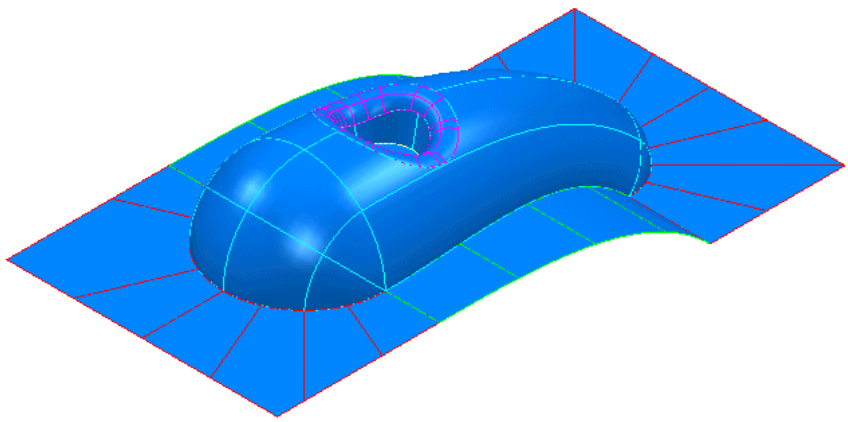
The **Reference Toolpath** finished to rough out material closer to the component form.
This will reduce the risk of excessive wear or damage to tools used for the subsequent finishing operations.

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

EXERCISE:-

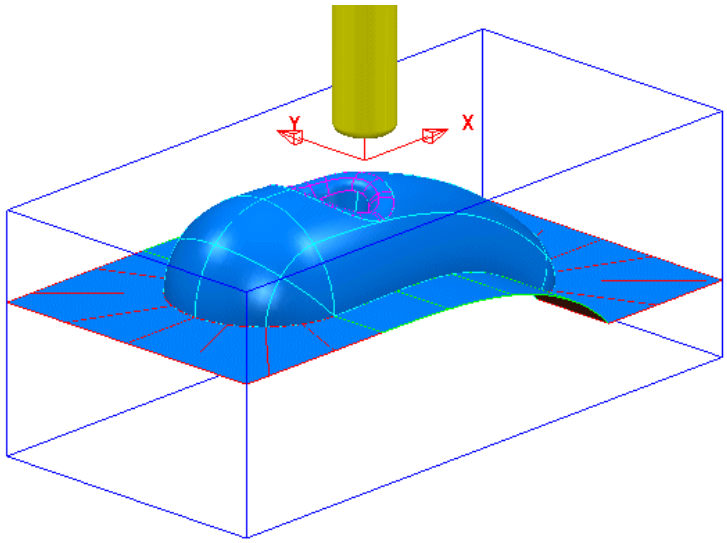
Area Clearance with Area Filter and Rest Roughing

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



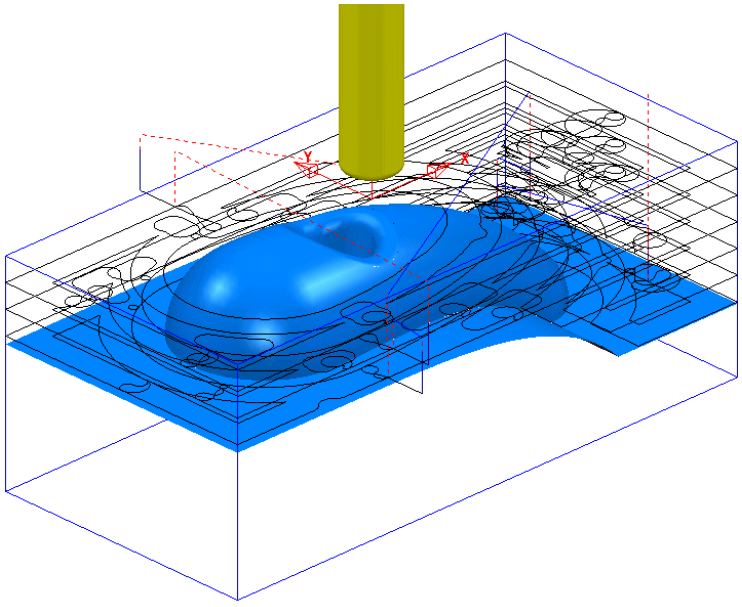
Machining Set Up

- 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 the **Box** option.
- Apply **Reset to Safe Heights** in the **Rapid Move Heights** form.

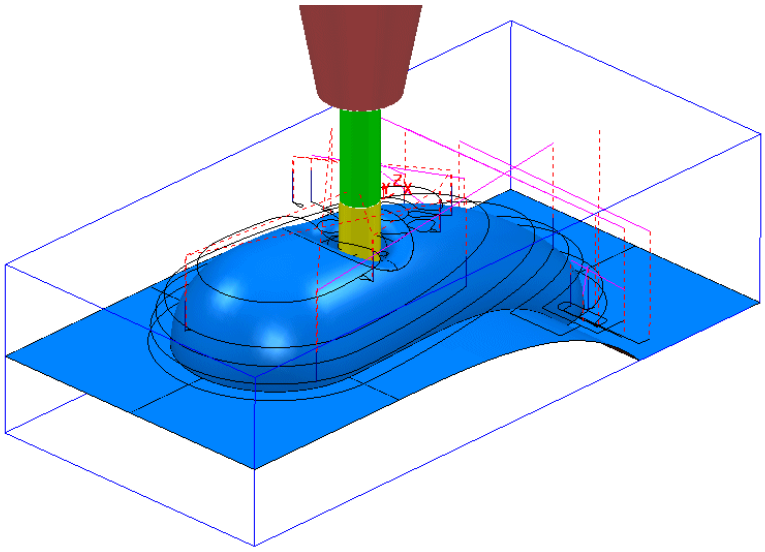


Area Clearance Strategies

- Create an **Offset Area Clearance** strategy using the **Area Filter** options to ***prevent*** the ***D20t3*** **tool** from attempting to machine into the ***central pocket***.



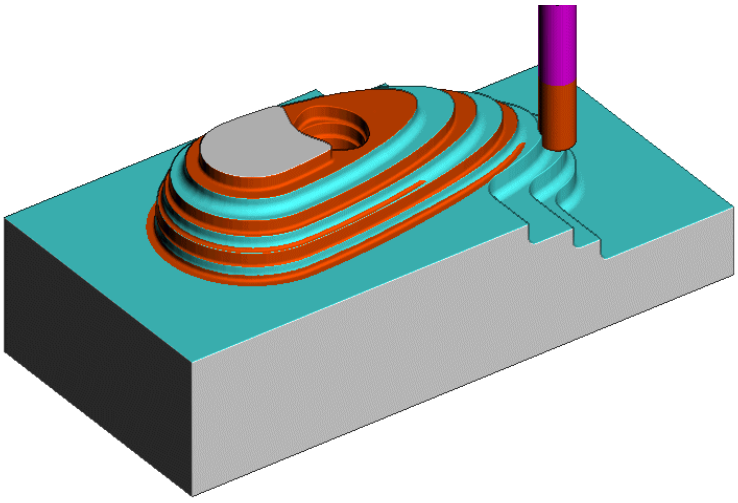
- Create another **Offset Area Clearance** strategy this time with the **D12t1** tool with the **Area Filter Options** switched off and the **Rest Machining** options switched on and ***referenced*** to the previous toolpath.



- **Save Project As**:-
D:\users\training\COURSEWORK\PowerMILL_Projects\AreaClear-Exercise

ViewMILL Simulation

- Perform a **ViewMILL *simulation*** on both of the **Raster 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.



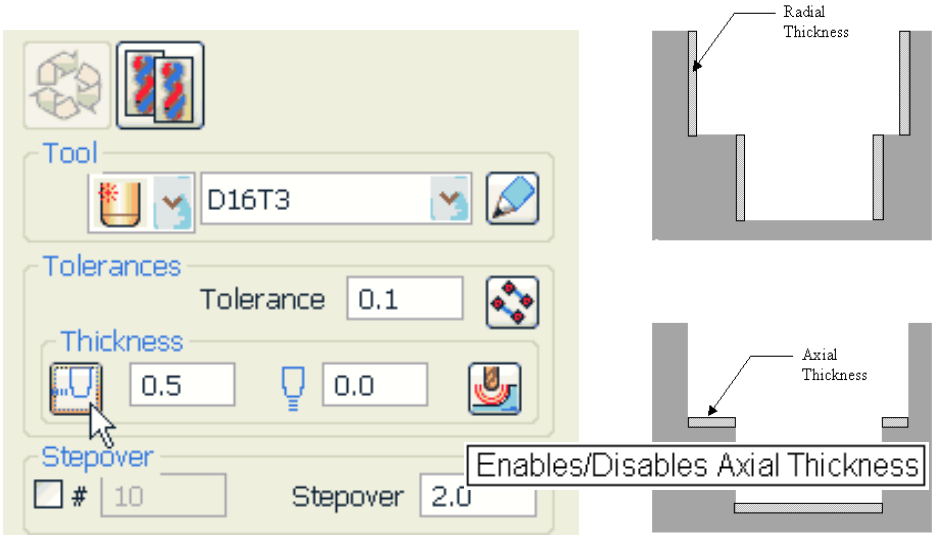
The ***Rainbow*** **ViewMILL *Shading*** option displays the resultant material removal for each **toolpath** as a different colour.

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

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

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

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

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

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

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

Appending Z Heights

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

Profiling

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

When

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

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

Cut Direction

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

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

Final Profiling Pass

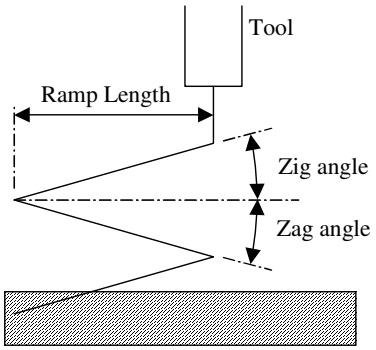
This option is held in the profiling area of the main area clearance toolbar and allows the user to make an additional, final profiling pass to further reduce tool wear.

Allow tool outside block

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

Ramping

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



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

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

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

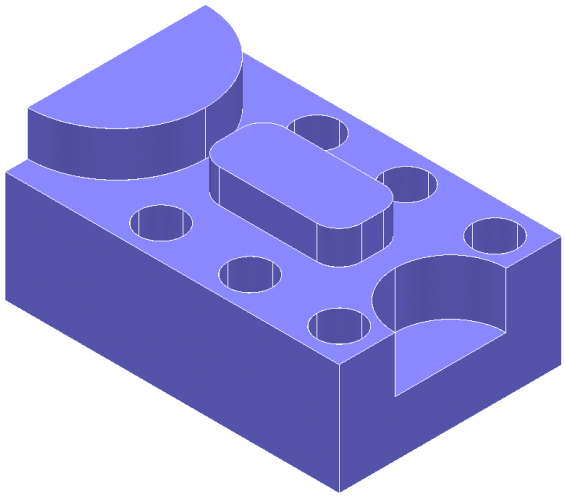
If **Approach Outside** is *set*, and where it is practical for it to operate without gouging it will take priority over **Ramping**.

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

Machining Flats

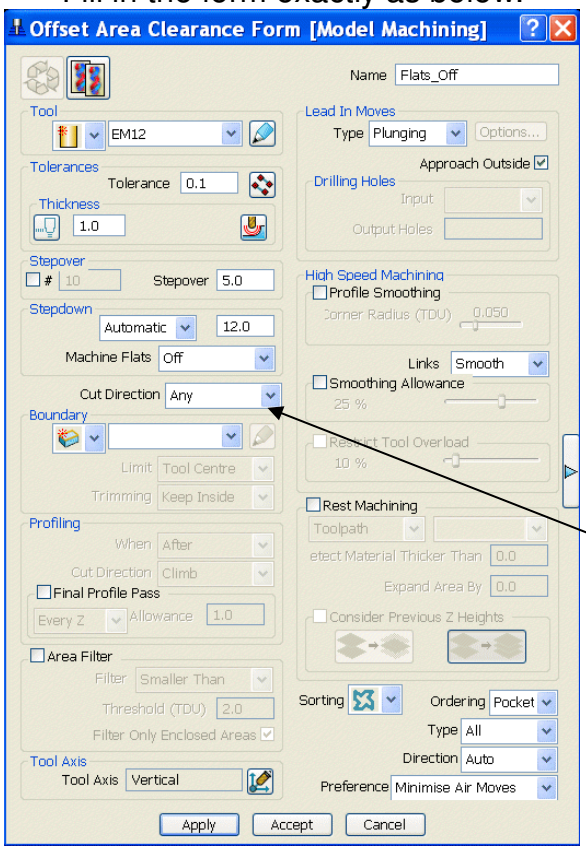
The **area clearance** strategies in **PowerMILL** have an option that allows the user to control the way in which flat areas of the model are rough machined. These are found on the area clearance form under **Machine Flats**.

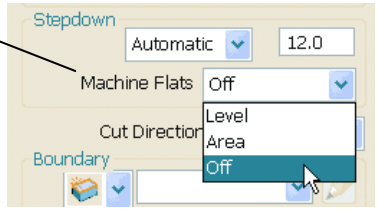
- **Import** the model **D:\users\training\PowerMILL_Data\Models\Flats.dgk**
- Create a **12mm** diameter **End Mill** tool and name it **EM12**
- **Calculate** the **Block** using the default settings.
- Set the **Rapid Move Heights** and check **Start/End Point** is set to default; ***Start Point***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



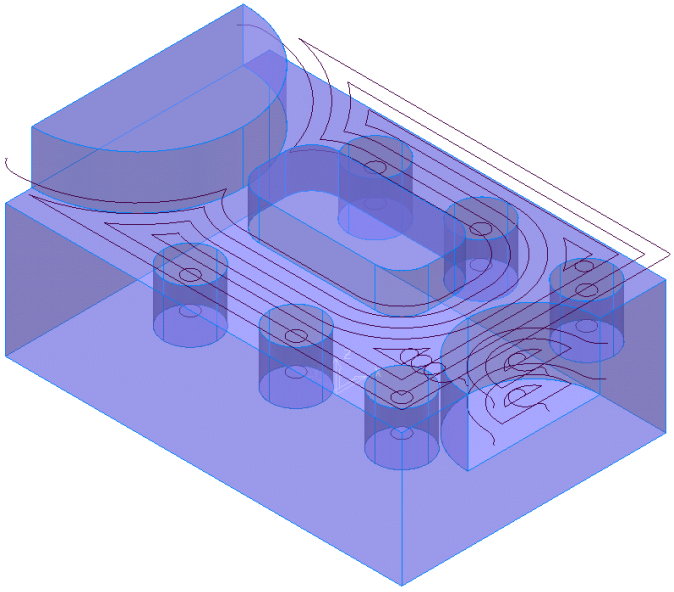
- From the **Toolpath Strategies** form, select **Offset Area Clearance**.

- Fill in the form exactly as below.



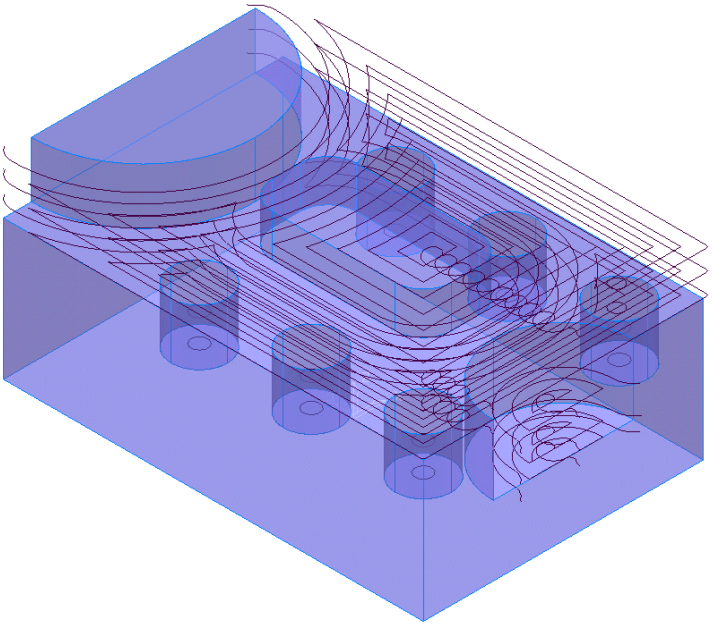


- **Apply** and then **Cancel** the form



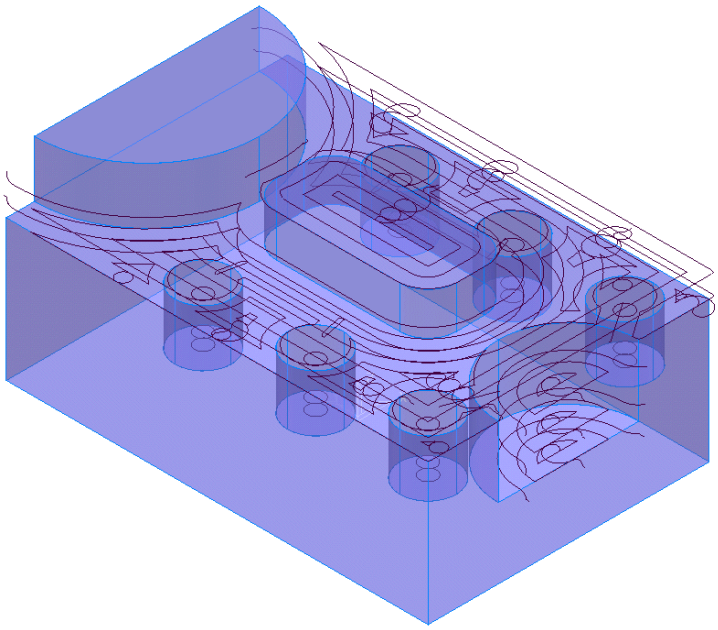
It can be seen that with **Machine Flats - Off** the toolpath has ignored the flat surfaces of the model. It has maintained a constant **Stepdown** value and completely performed area clearance across the material **Block** at each **Z Height**.

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



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

- Right mouse click over the **Active** toolpath and in the local menu select **Settings**
- Select make a **Copy** of the **toolpath**.
- Change the **Machine Flats** option to **AREA**.
- Change the **Name** to **Flats_Area**.
- **Apply** and then **Cancel** the form.

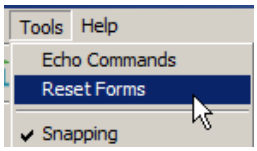


The Component is fully area cleared at the general **Stepdown** heights and locally to the edge of the component **Flat** areas. This provides a shorter toolpath compared with using the **Level** option.

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

PowerMILL Recycle Bin

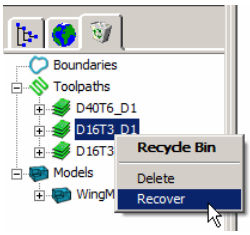
- In the **explorer** Right click over **Toolpaths** and select **Delete All** from the local menu.
- Use the same method to **Delete All** - **Models** and **Stock Models** (Do not apply to the **Tools** section in the **explorer**).
- From the **Main** pulldown menus select:- **Tools - Reset Forms**.



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

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

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



- Right mouse click over an entity in the **Recycle Bin** to view the local menu options to **Recover** or permanently **Delete** an item.
- **Recover** a **Toolpath** from the **Recycle Bin** and check that it has been reinstated in the **Toolpaths** section of the **PowerMILL explorer**.
- Return to the **PowerMILL explorer** and if necessary, apply **Delete All** separately again on **Toolpaths** 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**.

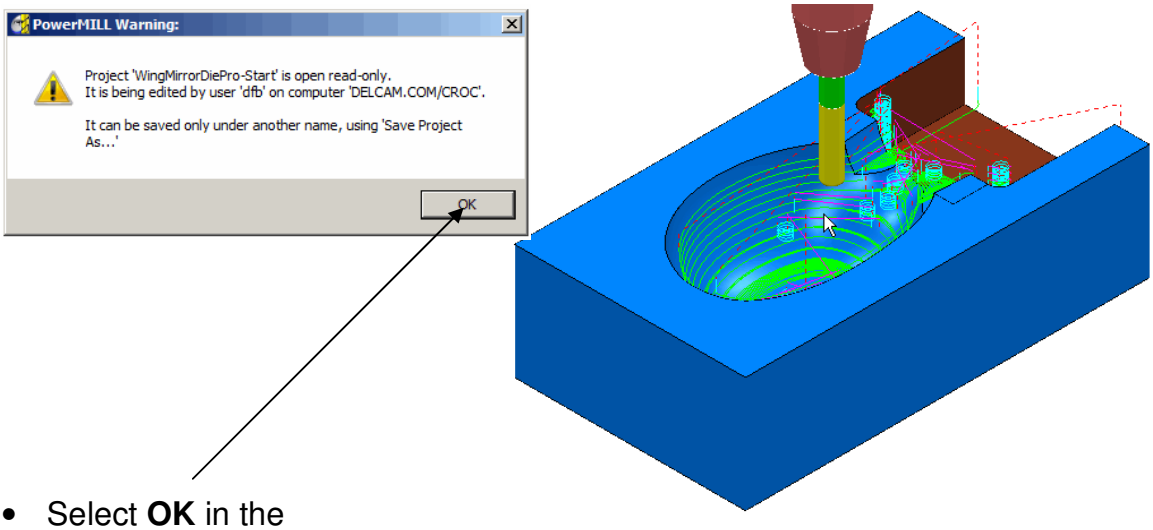
Rest Machining using a Stock Model

An alternative method of **Rest Machining** is the use of a **Stock Model** instead of a **Reference Toolpath**.
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 Path** based on a stock model will account for all the previous operations on the stock model not just one area clearance path.
- Operations created to different active **Workplanes** (3+2) can be used to create a stock model. Toolpath Rest Roughing is restricted to paths using the same workplane.

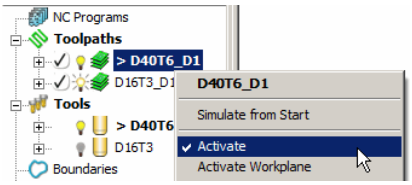
The following example will demonstrate the creation of a **Stock Model** up to the end of the toolpath **D40T6_D1** at which point it will be displayed as the **Rest Material**. The **Rest Machining** toolpath **D16T3_D1** will then be used as the basis for creating a new toolpath but this time using the **Stock Model**.

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

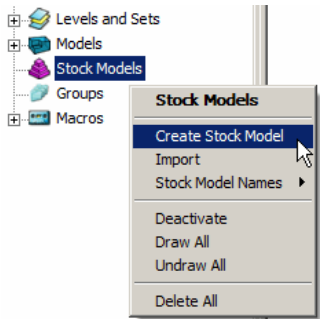


- Select **OK** in the ***PowerMILL Warning*** form.

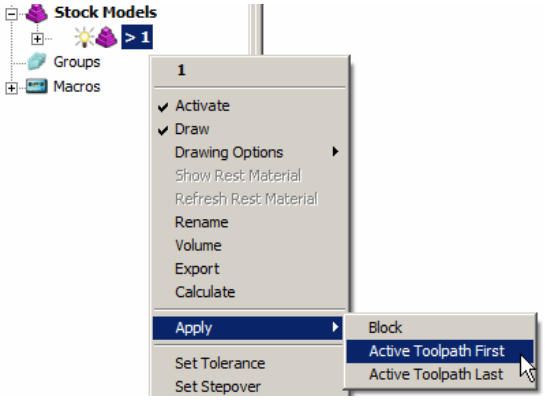
- **Save Project As**:- **D:\users\training\COURSEWORK\PowerMILL-Projects\WingMirrorDie**
- In the **explorer**, **Activate** the toolpath **D40T6_D1**.



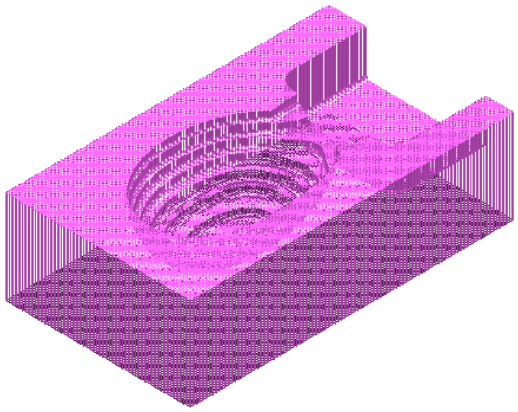
- In the **PowerMILL explorer** right mouse click **Stock Models** to open the local menu and select **Create Stock Model**.



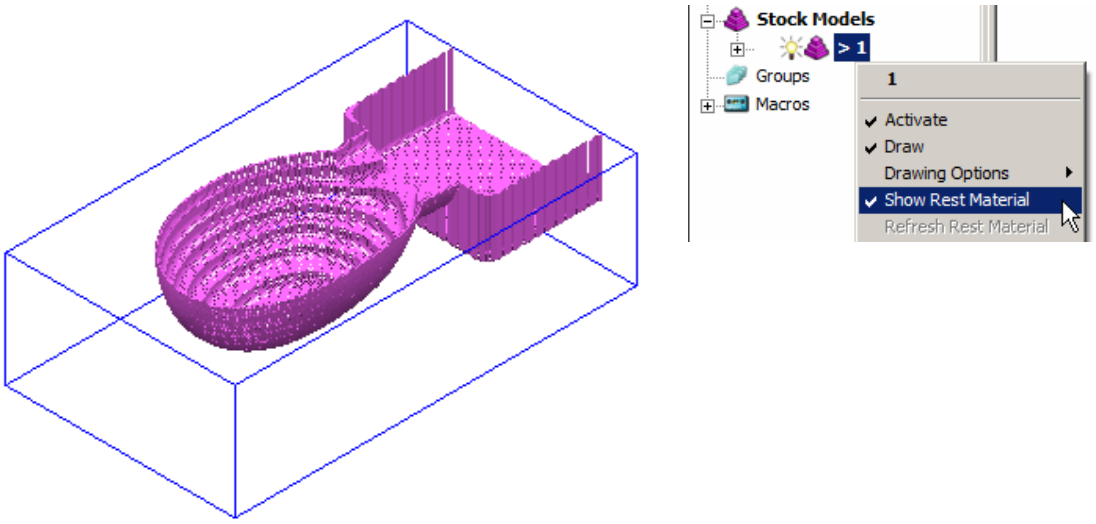
- In the **PowerMILL explorer** right mouse click the newly created **Stock Model** icon and from the local menu select **Apply** – **Active Toolpath First**.



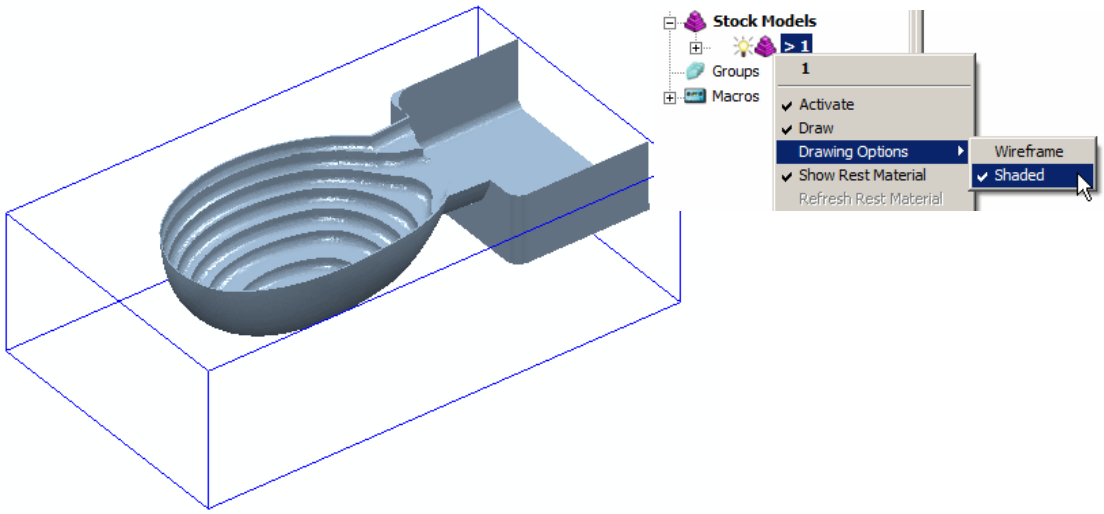
- From the same menu select **Calculate** to create the **Stock Model** as shown below.



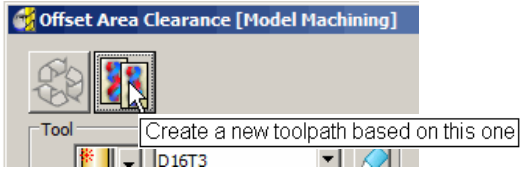
- From the same menu again select **Show Rest Material** to display the **Stock Model** as shown below.



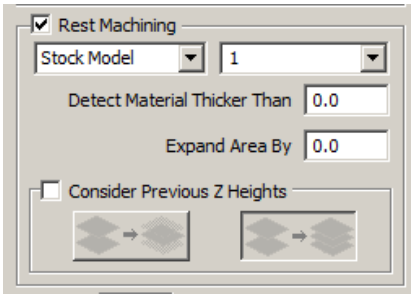
- From the same menu again select **Drawing Options** - **Shaded** to display the **Stock Model** as shown below.



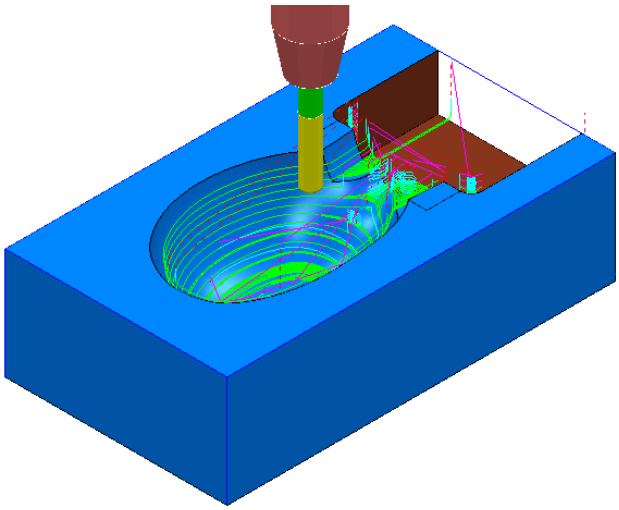
- In the **explorer Activate** the toolpath **D16T3_D1**.
- Right click the toolpath icon and from the local menu select **Settings** to reopen the **Offset AreaClear Model** form.
- Select the '**Copy toolpath**' icon (shown arrowed below).



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

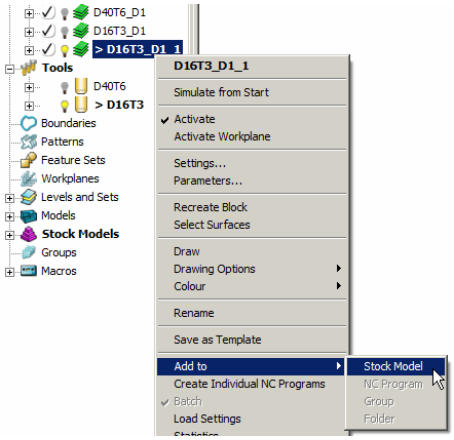


- Click **Apply** to create an alternative (more efficient) **Rest Machining** toolpath named **D16T3_D1_1**.
- **Cancel** the **Offset Area Clearance** Form to reveal the following toolpath.



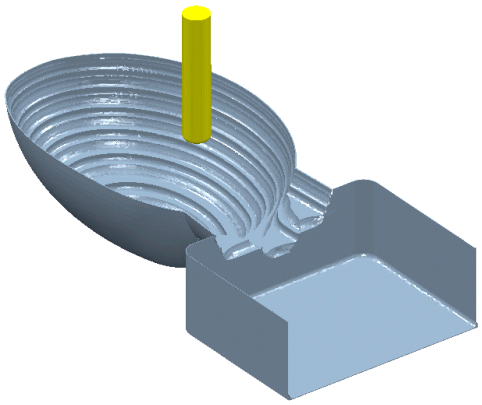
Although the **Stock Model** has been used to define the area to be machined the new toolpath is not automatically added to the **Stock Model**. This is carried out as a second operation if required.

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



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

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

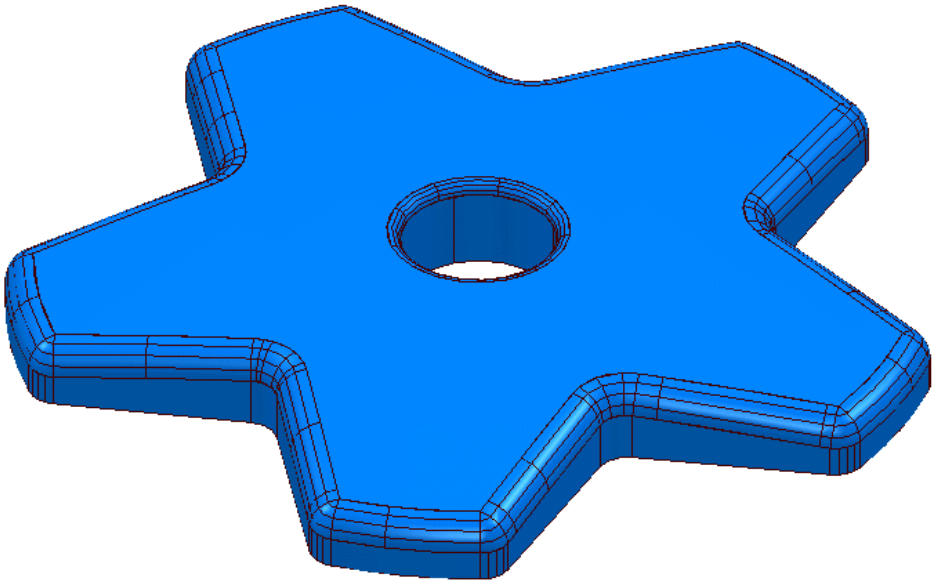


- From the Main toolbar select **File - Save Project** to update the Project:- **D:\users\training\COURSEWORK\PowerMILL-Projects\WingMirrorDie**

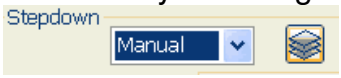
Offset Area Clearance – Offset Type Model

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

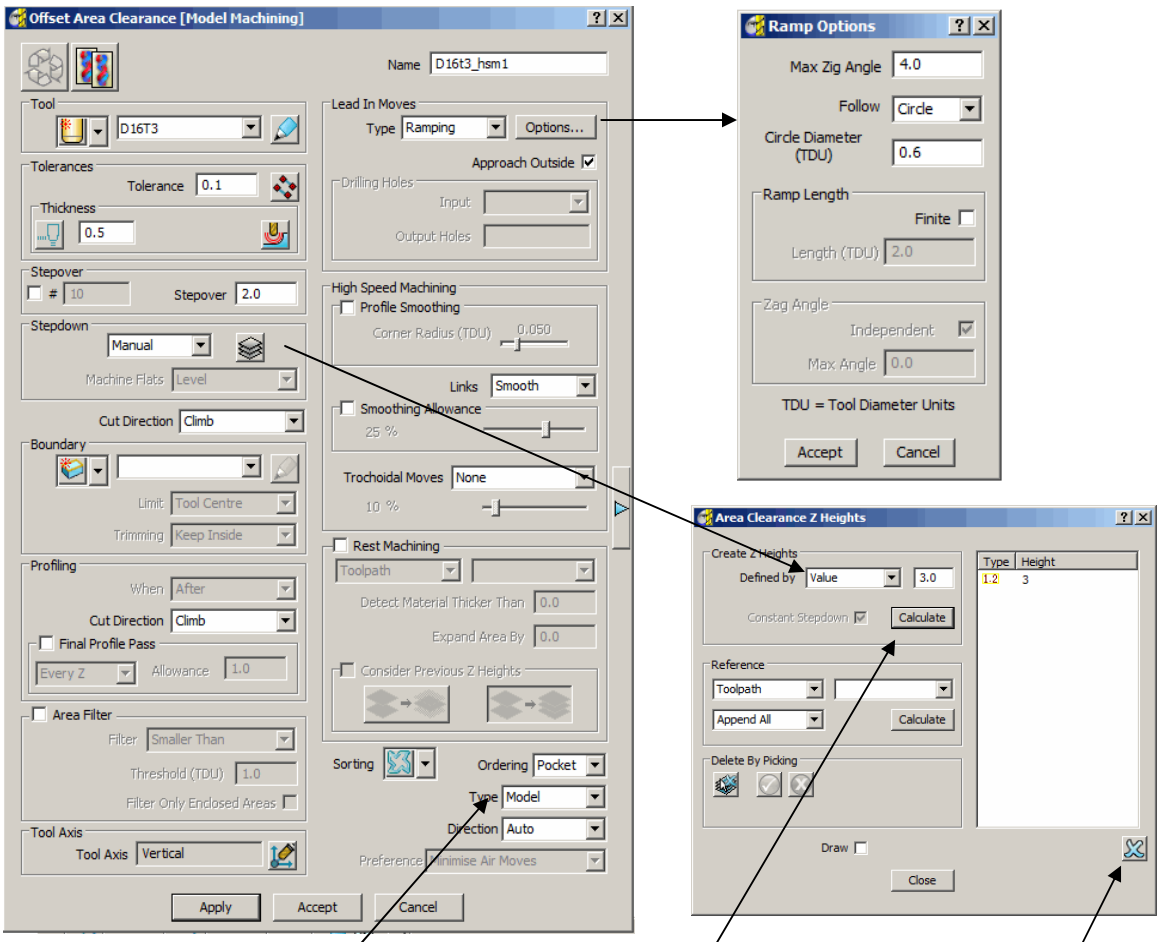
- **Import** the model **Handle.dgk** located in **PowerMILL_Data\models**.



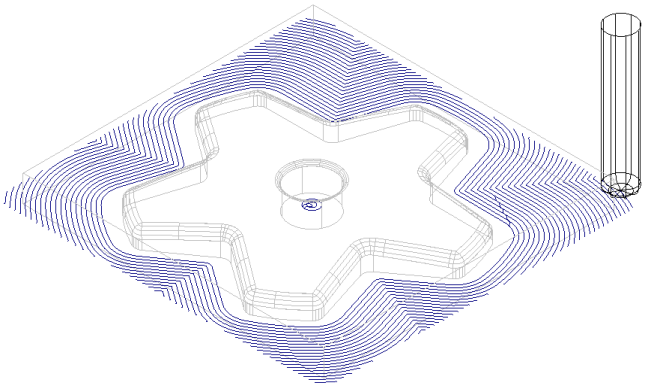
- In the **explorer - Activate** the **Dia 16 tiprad 3** tool **d16t3**.
- From the **Main toolbar** open the **Block** form and **Calculate** a material **Block** to the full model dimensions. **Lock** the **Z max** and **Z min** values, enter an **Expansion** value of **10** and **Calculate** again.

- In **Rapid move heights**, click **Reset to Safe Heights**, and in ***Incremental Heights*** set the ***Rapid Move Type*** to **Skim**.
- In the **Start Point** form set both the **Start Point** and **End Point** to **Block Centre Safe**.
- From the **Main toolbar** select the **Toolpath Strategies** icon. 
- From **3D Area Clearance** select the option **Offset AreaClear Model** to open the following form.
- Enter data in the forms exactly as shown on the next page.
- The **Stepdown** is to be defined manually as a single level from the **Area Clearance Z Heights Form** 

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

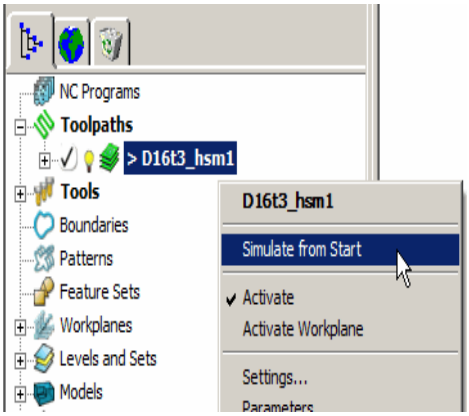


- Select **Type** as **Model**.
- Input Defined by ***Value*** **3.0** and select **Calculate** (After selecting **Delete All** existing **Z Heights**).
- Check the data as shown previously and click **Apply** then **Cancel.**



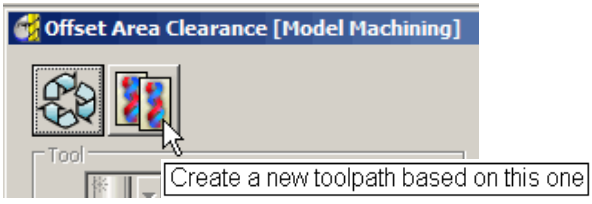
Note: The toolpath is shown with the **Leads and Links** undrawn.

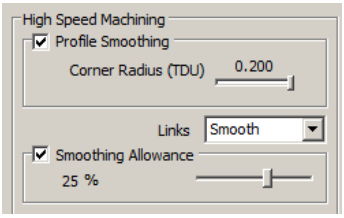
- Select an **Iso 1** view as shown and right click over the toolpath icon in the **PowerMILL explorer** to open the local menu.
- Select **Simulate from Start** and click the > to run the animation of the toolpath (select the **Esc** key to stop the **simulation**).



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. A specified deviation from the nominal **stepover** will be required to allow for this.

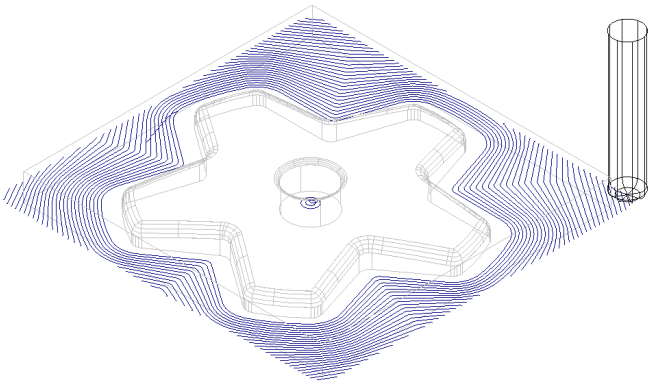
- Right mouse click over the Toolpath icon in the **explorer** to open the local pull down menu.
- Select **Settings** to reopen the **Offset AreaClear Model** form.
- Select the '**Copy toolpath**' icon (shown arrowed below).





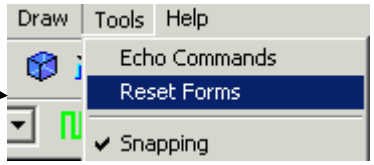
- With the **Offset Area Clearance** form open again, **tick** the **Smoothing Allowance** option leaving the slider value as default (**25%**).

- Click **Apply** to create a new toolpath **d16t3_hsm1_1** with the improved 25% smoothed strategy.



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

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

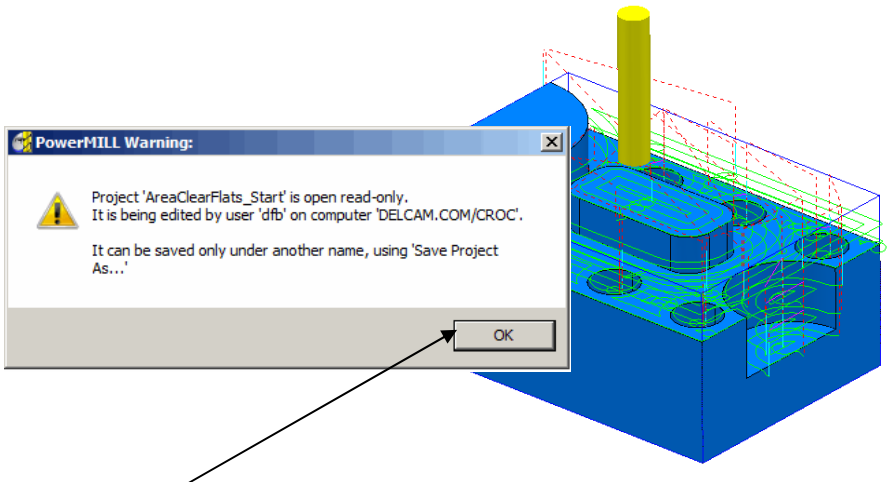


This restores the original **PowerMILL** settings in the forms without having to **Close** and restart.

Machining Flats

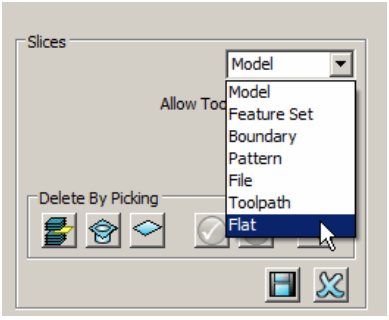
In **PowerMILL Pro** it is possible to locally **Area Clear** machine the flat areas to their exact height. This allows the user to rough the part first using Machine Flats set to OFF and then clear the flats in a second operation.

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

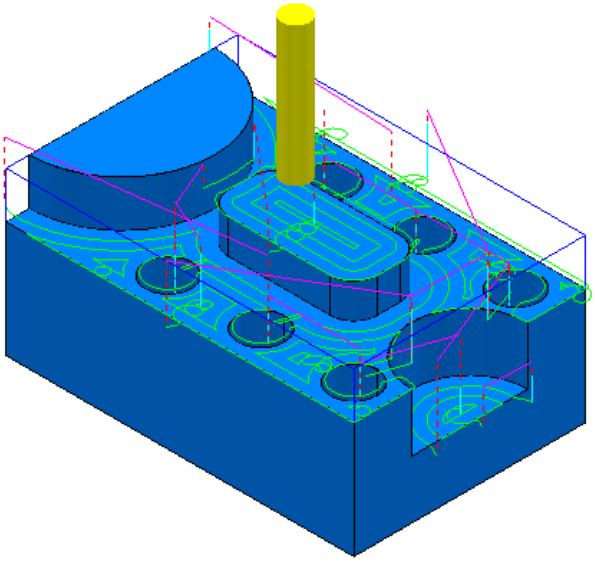


- Select **OK** in the ***PowerMILL Warning*** form.
- **Save Project As**:-
  D:\users\training\ COURSEWORK\PowerMILL-Projects\AreaClearFlats_Example

- **Right mouse click** on the toolpath **FlatsArea** and from the local menu, select **Activate** before selecting **Settings**.
- Make a **Copy** of the **toolpath**.
- Open the **Expert** part of the form by pressing the arrow at the right.
- Select **Flat** from the ***Slices*** options pulldown.

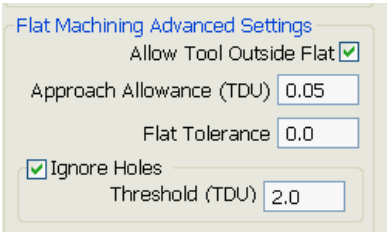


- **Calculate** the slices.
- **Apply** and **Cancel** the form.



The **Area Clearance** 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** and pass straight over them.

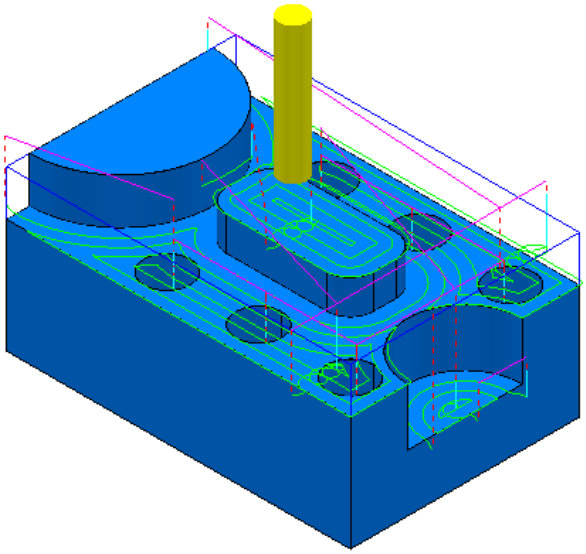
- Make a **Copy** of the **toolpath**.
- Open the **Expert** part of the form by pressing the arrow at the right. 
- **Tick** the box next to **Ignore Holes**.



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

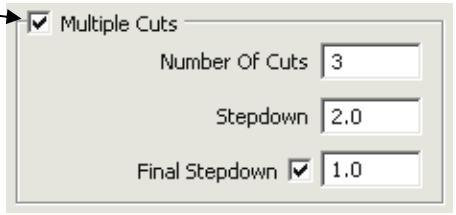
- **Apply** and **Cancel** the form.

The cutter now passes over the top of the holes without lifting, 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 the cutting tool daimeter is small enough to fit the bore.

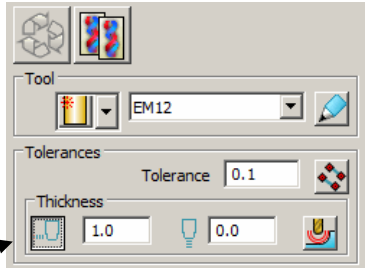


If there is a large amount of material remaining on the top of the ***flat areas***, then several passes at different heights can be made to reduce the tool loading during the **Area Clearance**.

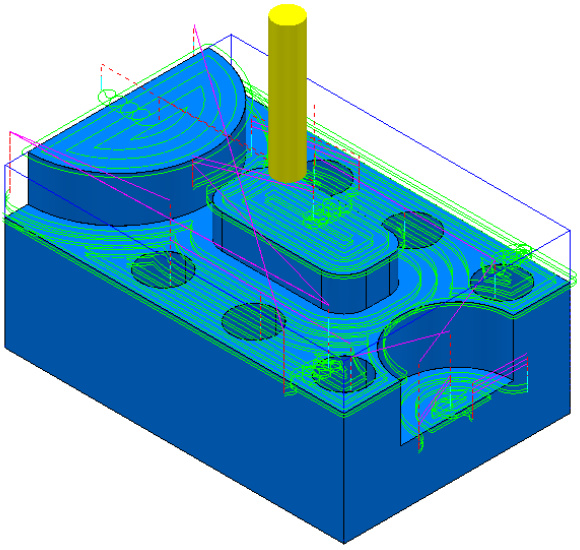
- Make a **Copy** of the **toolpath**.
- Open the **Expert** part of the form. 
- Select **Multiple Cuts**.
- Fill in the remaining options as shown.



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



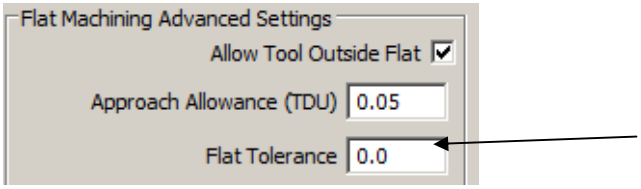
- Click the **Thickness** icon to activate **Axial Thickness** (**0**).
- **Apply** and **Cancel** the **Area Clearance** form.



The **Flat**, **Area Clearance** strategy is repeated at the 3 specified heights controlled by the **Stepdown** options.

Flat Tolerance

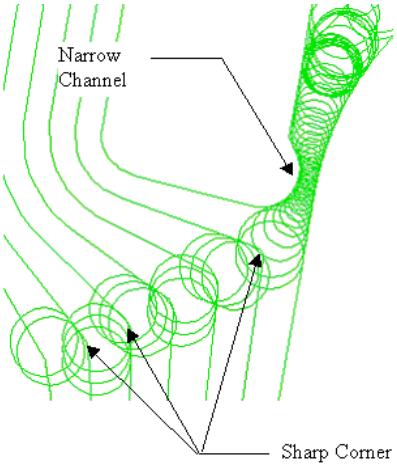
It should be noted that by default, the **PowerMILL - 3D Area Clearance** strategies will only detect perfectly flat **surfaces**. **Surfaces** that are not quite flat dimensionally will not be recognised unless the **Flat Tolerance** is set with a suitable value to allow for the deviation. This option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



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**.

Restrict Tool Overload (PowerMILL Pro)



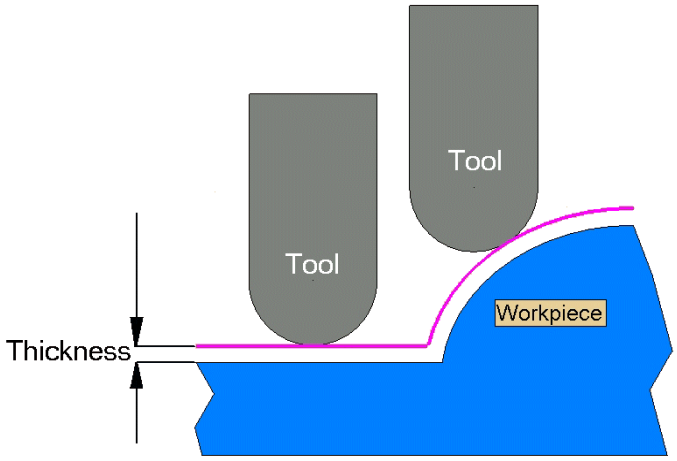
With **Restrict Tool Overload** ticked as the tool reaches an specified overload situation **PowerMILL** automatically inputs a **trochoidal** path to eliminate full width cuts. This will occur in corners, narrow channels, slots, etc and the degree of movement is controlled with the slider as a percentage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

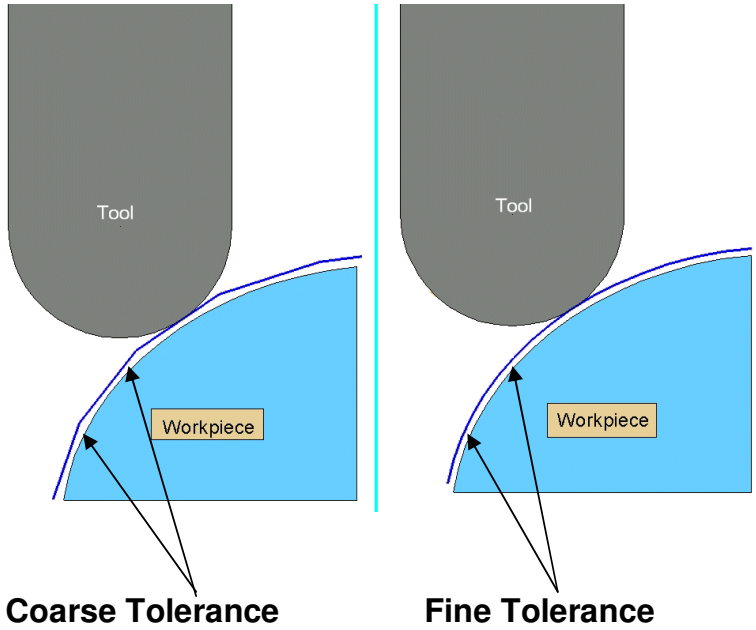
4. Finishing Strategies

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.

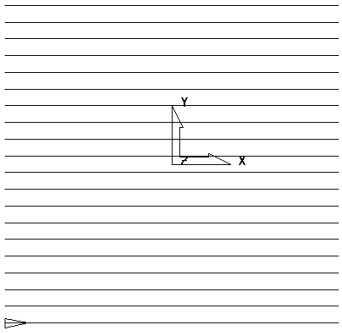
Introduction.

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**.

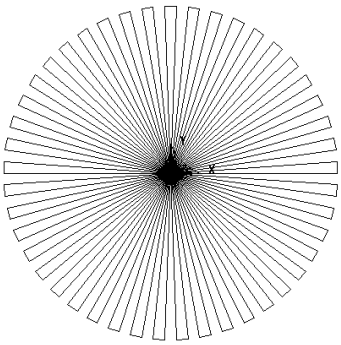
PowerMILL generates the toolpaths by projecting a wireframe form **down the Z-axis** onto the model. The standard patterns applied in **Raster**, **Radial**, and **Spiral** are achieved by entering values directly into the **Finishing** Form. The resultant **Pattern** can be displayed by selecting **Preview** before executing the command by selecting **Apply**. 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.

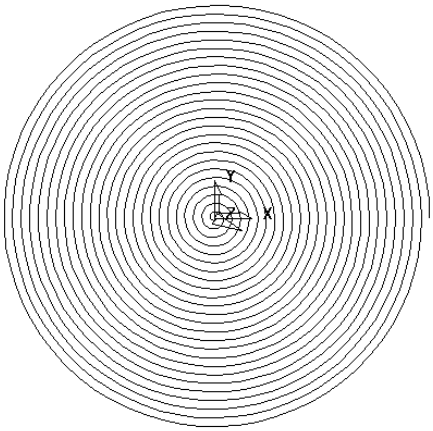
Raster



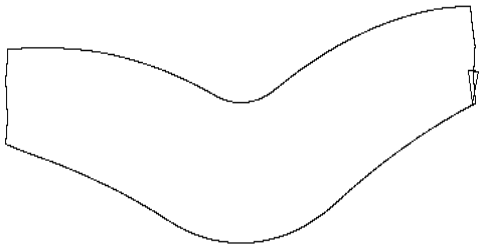
Radial



Spiral



Pattern (User Defined)



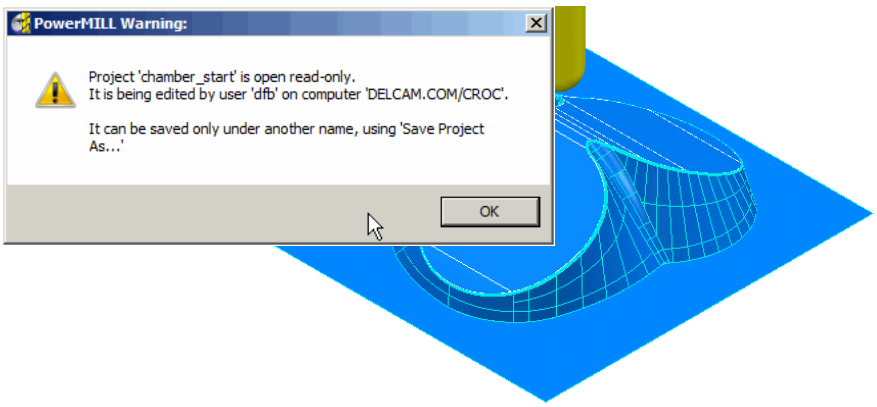
Radial, **Spiral,** and **Raster** finishing operate in exactly the same way by projecting the standard **Pattern** down **Z** onto the model. The more commonly used **Raster** finishing strategy will be used as an example in the next section.

Raster Finishing

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

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

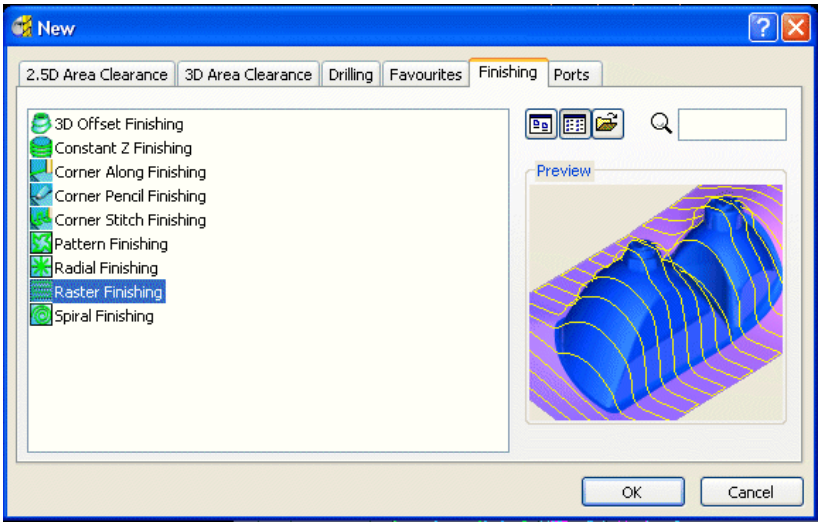
- Select **OK** on the form that appears informing that the source **Project** is **read-only**.



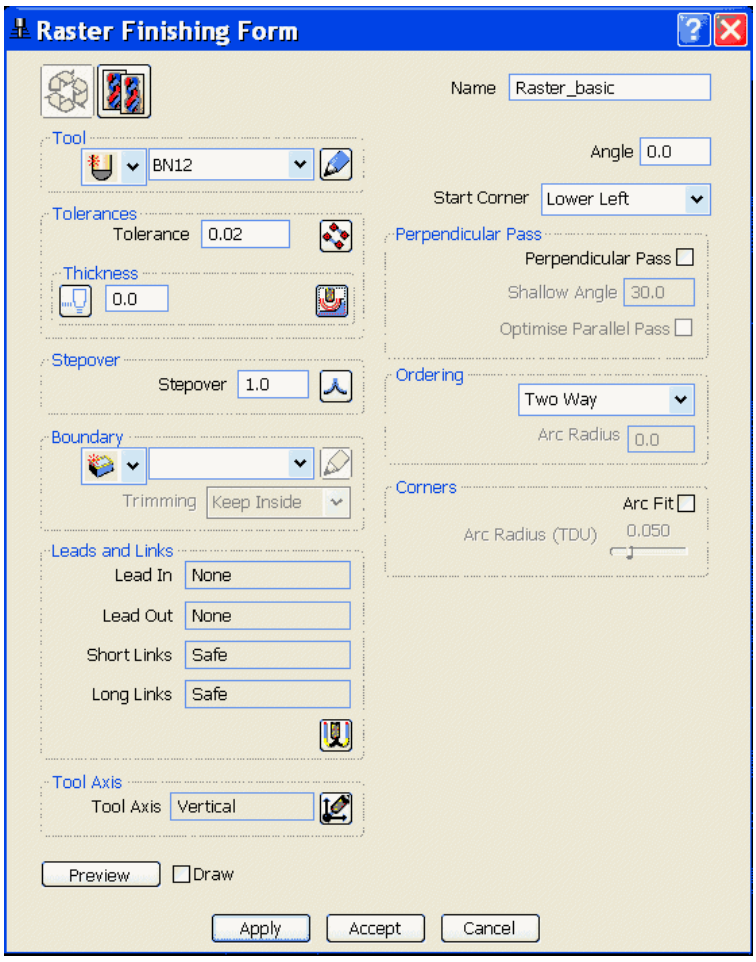
The model and tool stored in the imported **Project** are displayed.

- From **File** select **Save Project As**:- **D:\users\training\COURSEWORK\PowerMILL-Projects\chamber**
- Calculate a **Block** to default **Box – Model** settings.

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



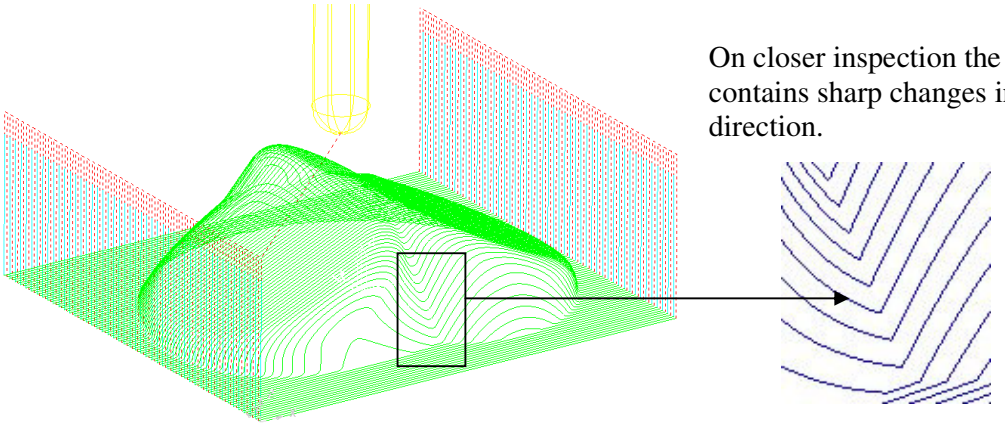
- Select the **Raster Finishing** icon then **OK**.



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

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

- Select a tool **Stepover** of **1mm**.
- **Apply** and **Cancel** the Form.

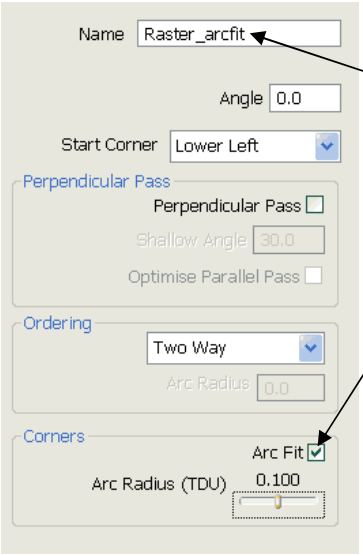


On closer inspection the toolpath contains sharp changes in direction.

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

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

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



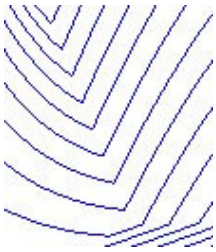
- Rename the toolpath **Raster_arcfit**.

- Tick the box **Arc Fit** and edit the **Arc Radius** to **0.1** (Use ***keyboard*** ← → to fine tune to exact value)

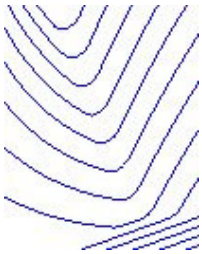
The Arc Radius (0.1) is multiplied by the tool diameter (12mm) to give a final radius of 1.2mm.
This option is particularly good for HSM (High Speed Machining).

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

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



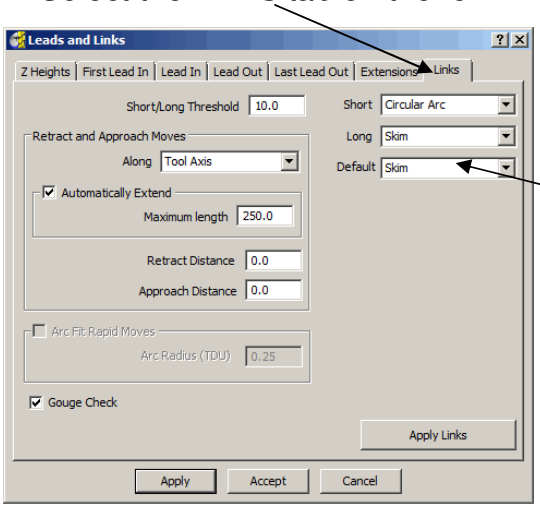
Before Arc Fit



After Arc Fit

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

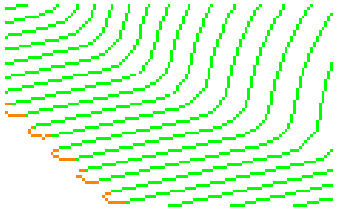
- Select the **Leads and Links** icon from the top toolbar. 
- Select the **Links** tab on the form.



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

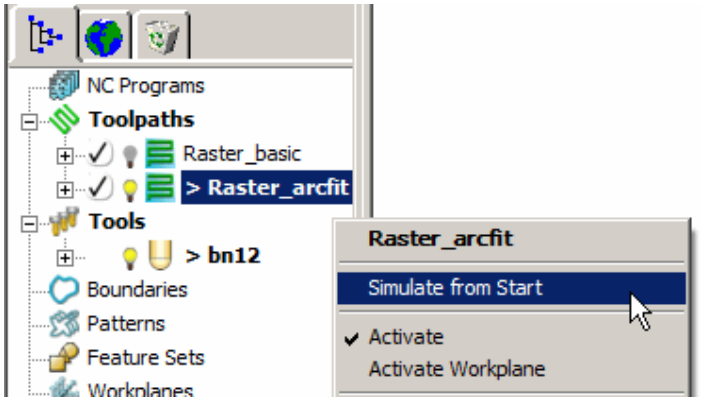
- Change the **Long** and **Default** links to **Skim.**

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



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

The next stage is to **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.

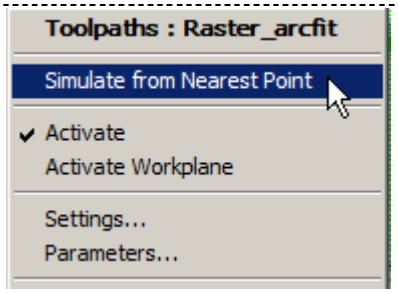
When the **simulation** has finished it will be observed that the tool remains at the end of the last retract move. To send the tool back to the **Start Point** either press the **Home** key on the keyboard or select the **Go to beginning** button on the **Simulation** toolbar.

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.**

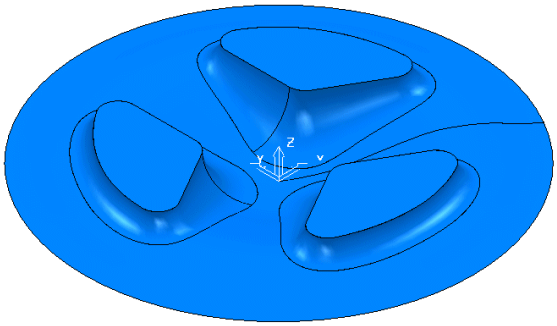


- Use the **Left** and **Right** arrow keys to move the tool forwards and backwards along the toolpath.

Pattern Finishing

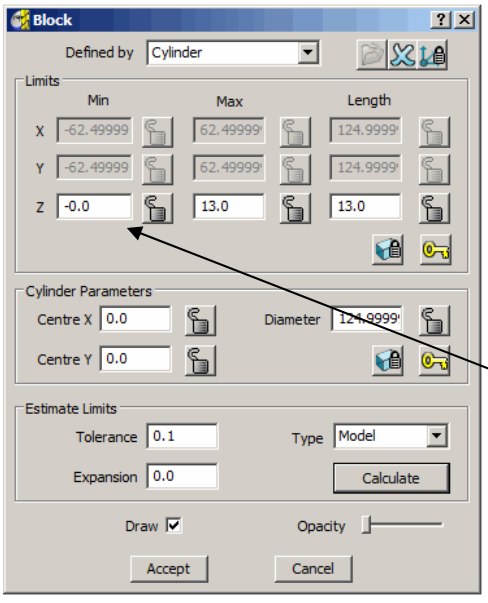
This technique requires a user defined **Pattern** for projection onto the model as a toolpath. This option can be used for applications such as scribe lines, lettering, and non-standard tool strategies. A **Pattern** is created either from within **PowerMILL** or as imported **Wireframe** data.

- **Delete All** and **Reset forms**.
- **Import** the **Model**: **D\: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 top toolbar. 



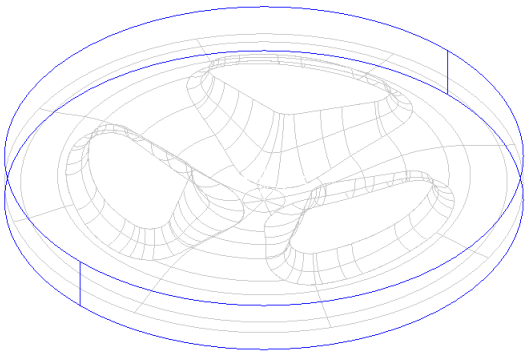
- Select ***Defined by*** **– Cylinder**.

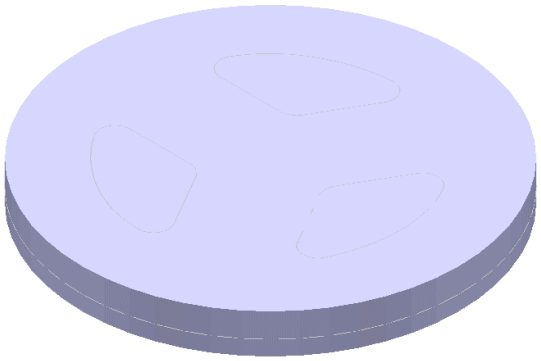
The default ***Radius*** value represents the best cylindrical fit around the component achieved when the **Calculate** button is clicked

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

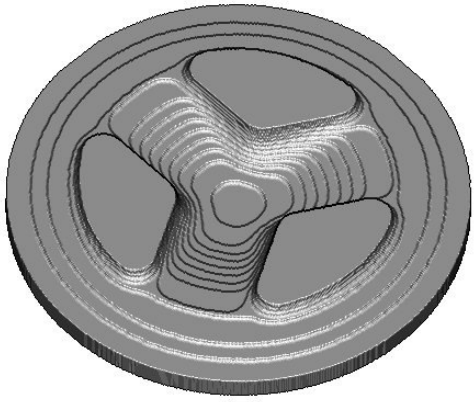




Before the **Pattern** finishing strategy is created, a preliminary **Area Clear** strategy is required to remove the bulk of the material, leaving 0.5mm, followed by a **Semi Finishing** Strategy leaving 0.2mm prior to the final **Finishing**.
The object of this **Pattern** exercise is to consolidate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

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

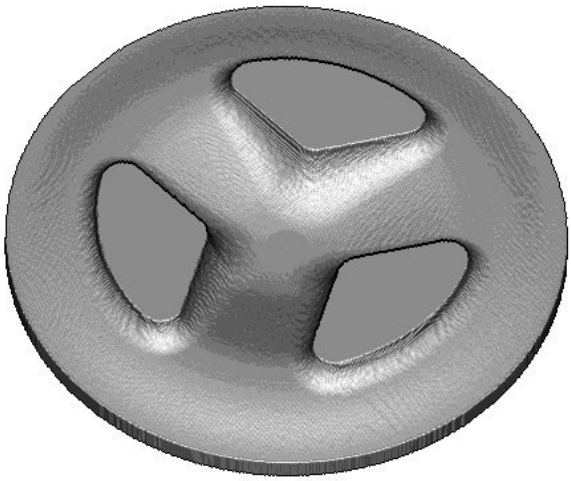


- **Simulate** the toolpath in **Viewmill** to give a result similar to as shown left.
- **Activate** the **BN6 Ball Nose tool**.
- Calculate a **Spiral Finishing** toolpath using the following parameters: ***Name*** – **SemiFinishOp1**, **Tolerance 0.02**, **Thickness 0.2**, **Stepover 1.0**, **Centre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

- **Simulate** this new toolpath in **Viewmill**.

- **Right click** over the **Semi Finish** toolpath in the **explorer** and select **Settings**.
- In the **Spiral Finishing** form select the **Copy** icon. 
- Enter a new ***Name*** - **Finish**, change the **Thickness** to **0.0** and **Stepover** to **0.5** then **Apply** and **Close** the form.

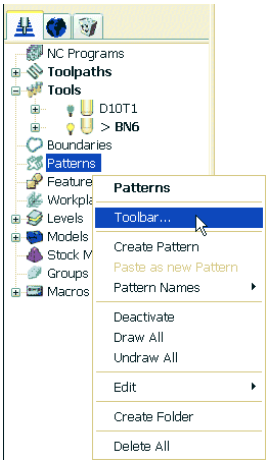
- **Simulate** the final toolpath in **Viewmill** to give this result.



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

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

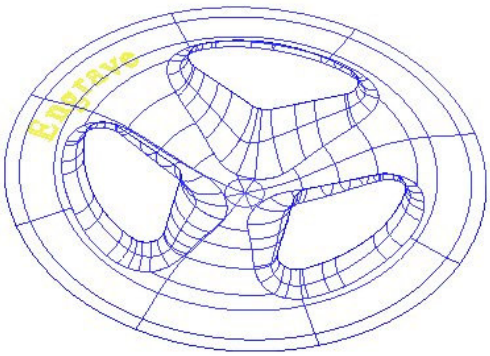
- Right click the **Pattern icon** in the **explorer** and select **Toolbar….**



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

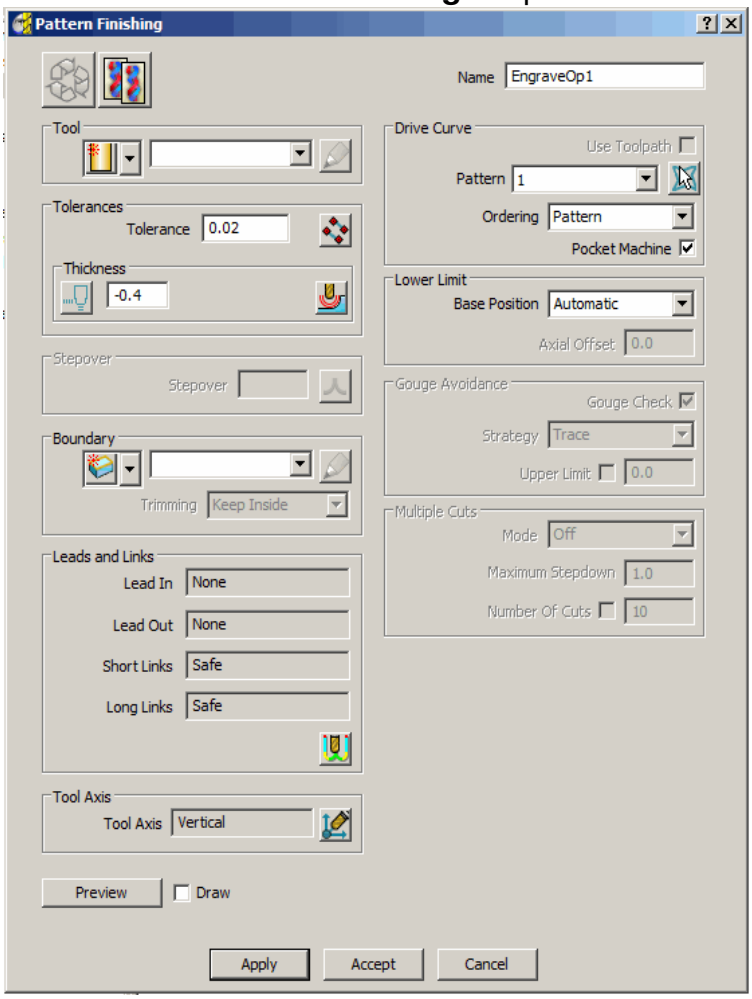
- Select the **Create pattern** icon  .
- Click on the Insert file into active pattern icon 

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



The **Pattern** is made up of **2D 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.

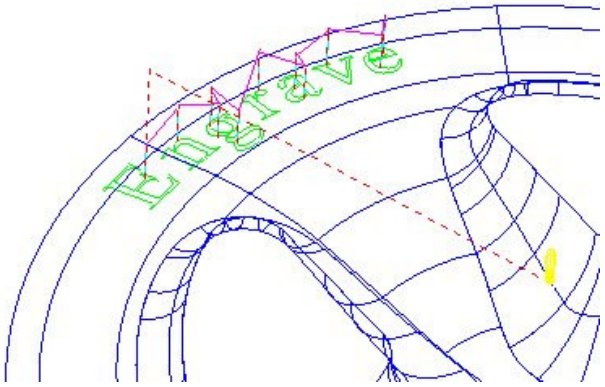
- Create a **Pattern Finishing** toolpath.



- Define a **Ball Nose** tool with **Diameter 1** called **BN1** and make this **Tool Number 3**.
- Select a **Pattern Finishing** strategy.
- Enter the ***Name*** – **EngraveOp1**.
- Enter the **Tolerance** as **0.02** and **Thickness** as **-0.4**
- Select **Pattern 1** (if not already selected).
- With **Automatic** set the pattern will be Dropped / Projected onto the model.
- Select **Apply** then **Cancel** the form.

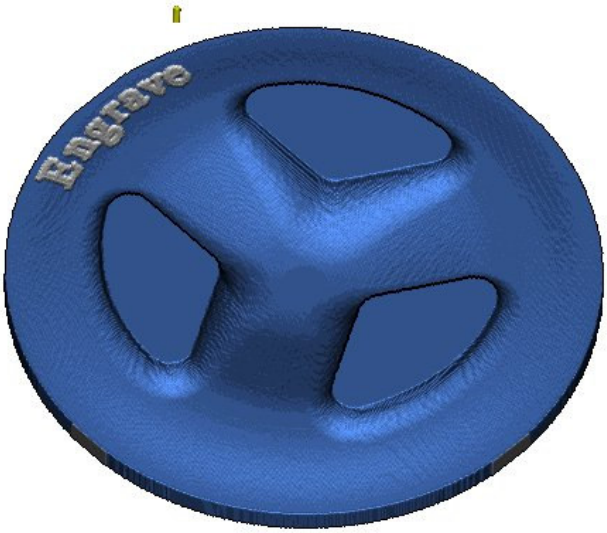
By entering a negative **Thickness** the tool will machine into the previously finished surface. This is frequently used for engraving, or to achieve a spark gap when machining part of a component as an electrode.

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



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

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

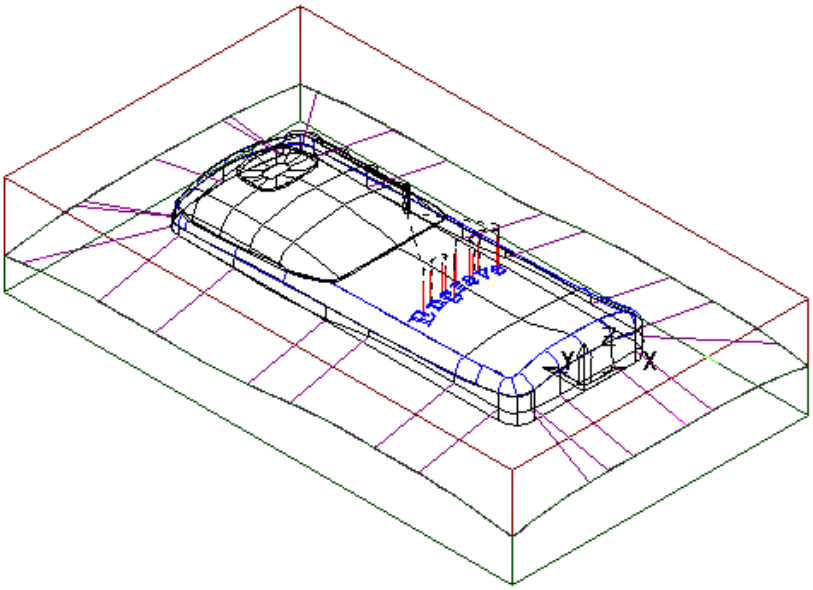


- **Save Project as**:-

D:\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.

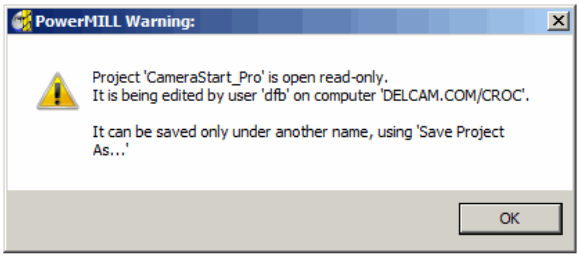
3D Offset and Constant Z Finishing

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

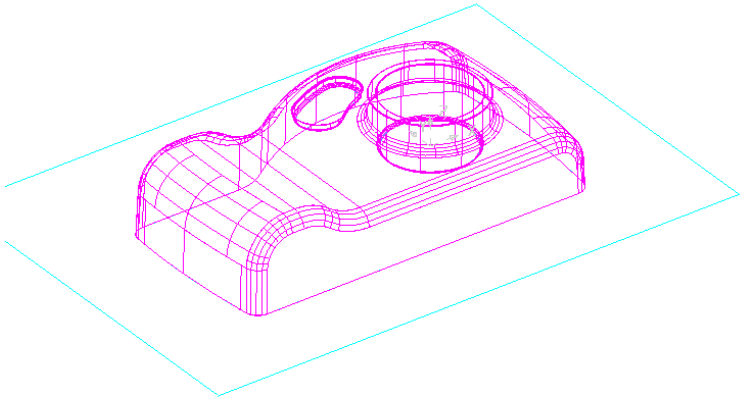
3D Offset Machining defines the tool **Stepover** relative to the 3D surface shape providing consistency over both flat areas and steep sidewalls. In this exercise applying this toolpath to the complete model without using **Boundaries** would not be recommended. Although this toolpath maintains a constant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

- **Delete All** and **Reset forms**.
- From **File** – **Open Project** and in the form browse to:-
D:\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).

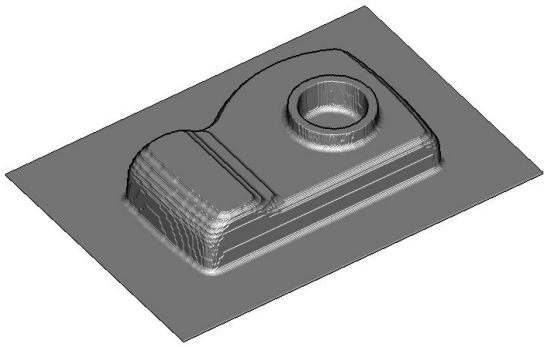
- Select **OK** to load the existing **Project** into **PowerMILL**.
- From **File** – **Save Project As**:-
D:\users\training\COURSEWORK\PowerMILL_Projects\Camera_Example



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**.

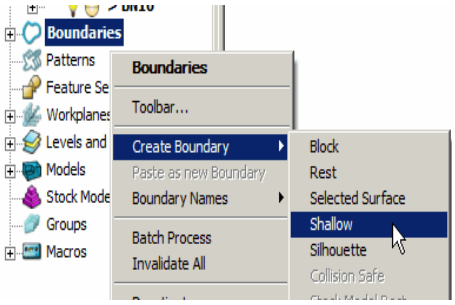
- **Activate** the existing **Offset *Area Clearance***, **Toolpath** and **Simulate** the toolpath in **Viewmill**.



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

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

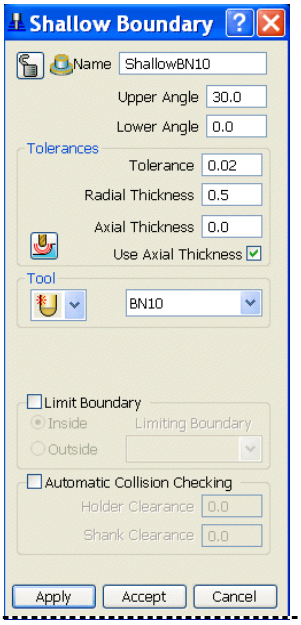
Use of Shallow Boundary to define specific Machining Areas



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.

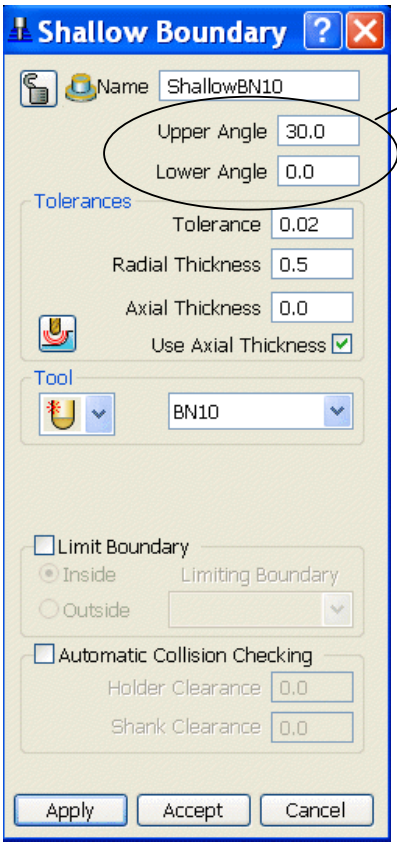
There are several other types of ***Boundary options*** available to suit various applications and these are covered later in ***Chapter 6***.

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



- Input the **Name** - **ShallowBN10**.
- Input a **Tolerance 0.02**.
- Tick the ***Use Axial Thickness*** box and enter ***Radial Thickness*** **0.5** and ***Axial Thickness*** **0**.

.

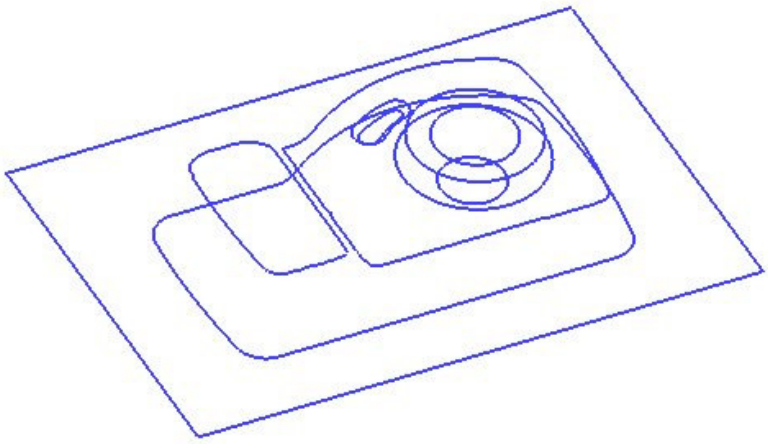


A **Shallow Boundary** defines segments from areas on the model that are defined by an **Upper** and **Lower** threshold angle. It is therefore specifically suited to steep walled and shallow surface machining techniques.

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

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

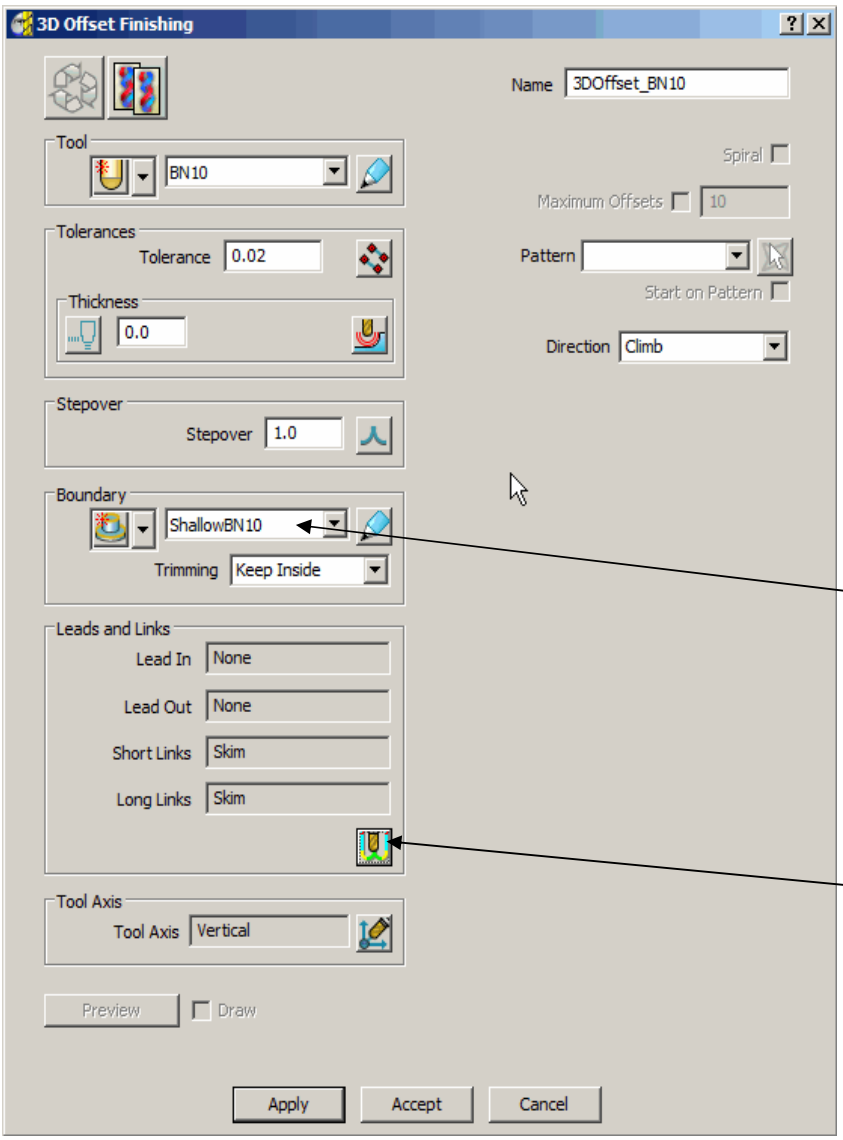
- **Apply** and **Accept** the form.



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

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

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

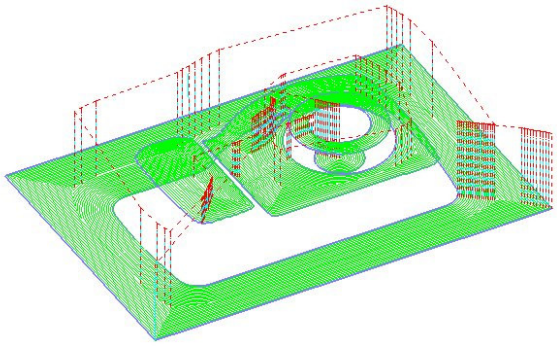


- Enter the **Name 3DOffsetBN10**.
- Select ***Direction* to Climb**.
- Enter a **Tolerance 0.02**.
- Enter a **Thickness 0**.

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

- Select the **Leads and Links** icon and set the **Lead In** & **Lead Out** to **None**, and **Short Links** & **Long Links** to **Skim**.

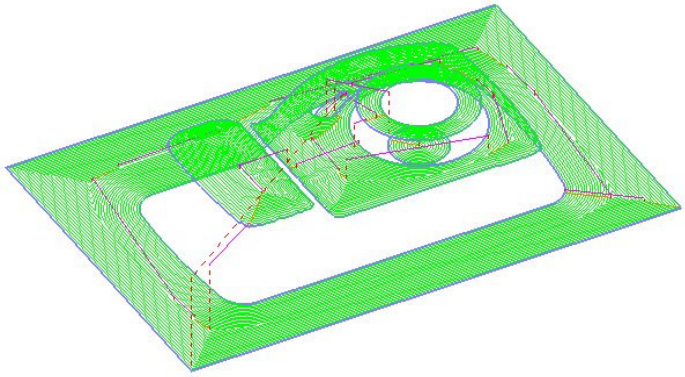
- **Apply** and **Cancel** the form.



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

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

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

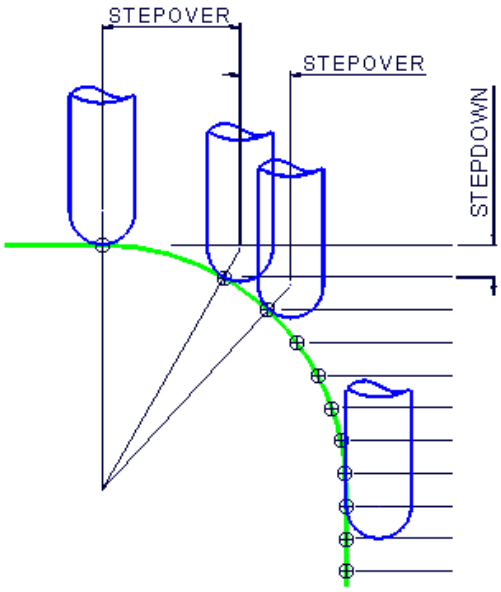


The toolpath has now is now more efficient with the rapid moves at **skim** height and the **short links** being forced onto the surface.

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

Constant Z machining

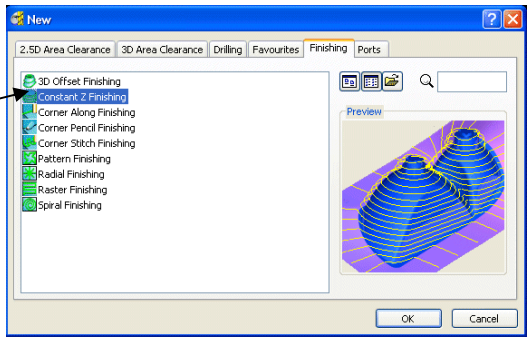
Constant Z machining projects each tool track horizontally onto the component at fixed heights defined by the **Stepdown**.

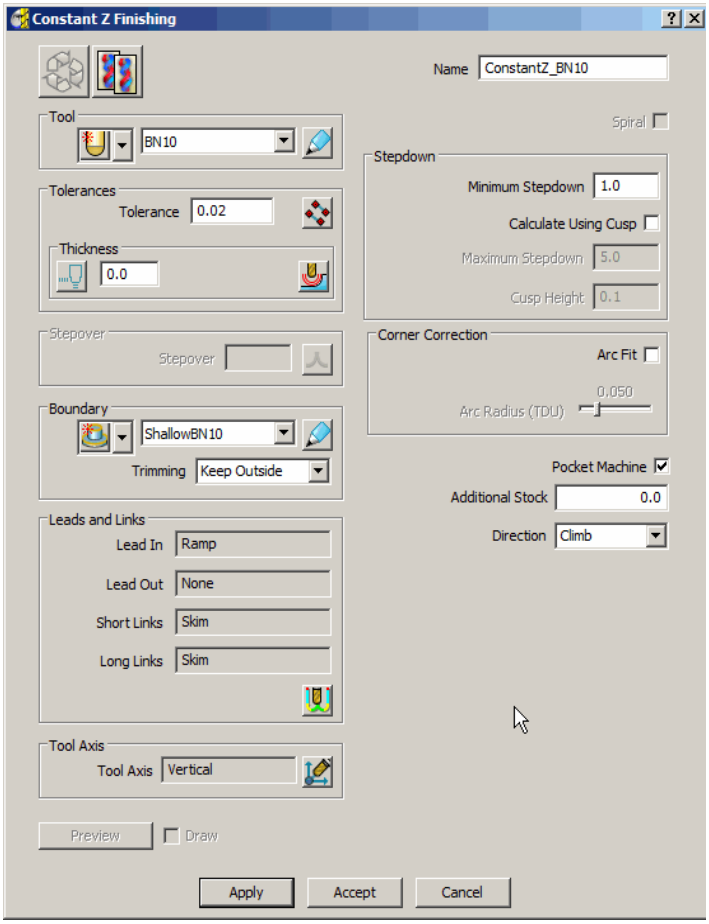


As the component surface becomes shallow the actual tool step over increases until it becomes non-existent on flat areas.
It is possible within the **Constant Z** finishing form to apply a variable stepdown by applying a **Cusp tolerance** in conjunction with a **max** and **min stepdown**. While this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

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

Select the **Constant Z Finishing** strategy.

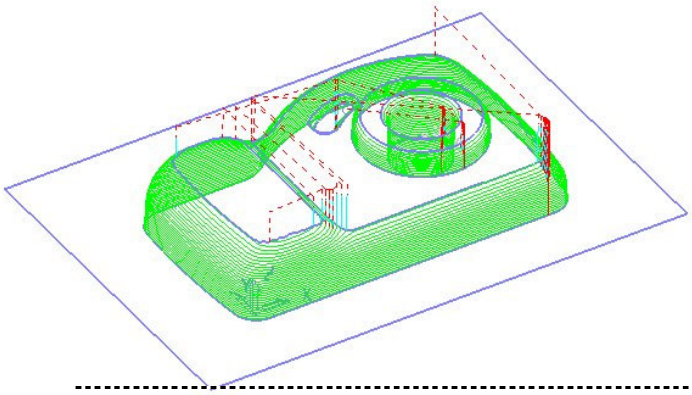




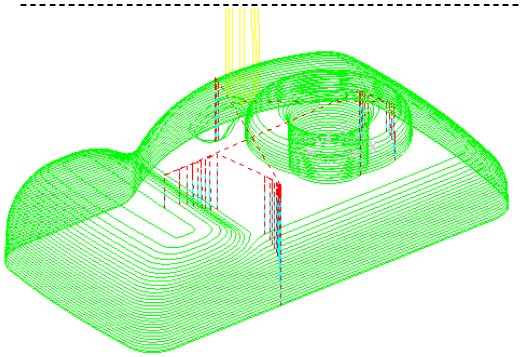
- Enter a ***Name*** as **ConstantZ_BN10.**
- Set a **Stepdown** of **1.**
- Enter the **Tolerance** as **0.02**

The ***Active*** **Boundary** will automatically be selected in the form.

- Set the ***Direction*** to **Climb**.
- Set the ***Trimming*** to **Keep Outside**.
- **Apply** then **Cancel**

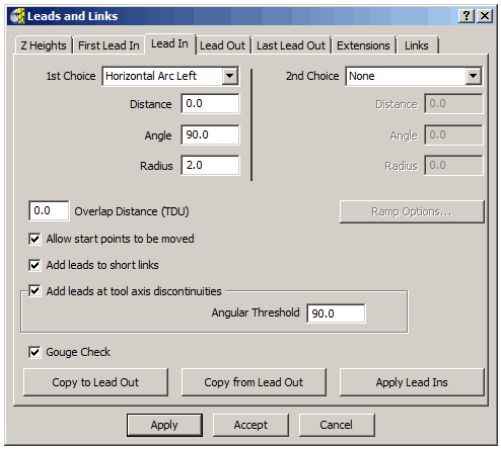


By using the **Boundary Trimming** option **Keep Outside**, the toolpath is correctly limited to the steep areas of the model.



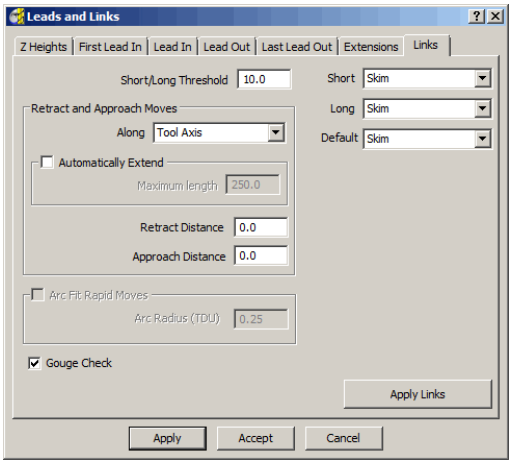
If the **Boundary** had not been used the toolpath would have looked like this.
It can be seen that the parts of the toolpath on the shallow areas have an excessive **Stepover**.

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

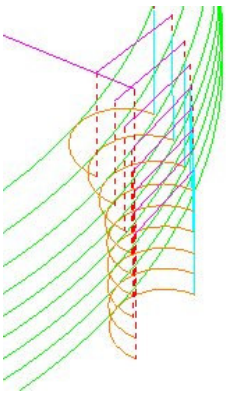


- Select the **Leads and Links** icon at the top of the screen.

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

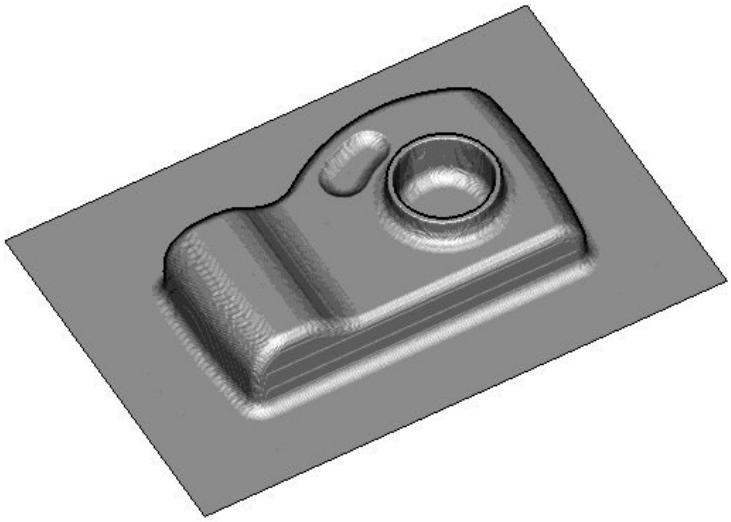


- Select the **Links** tab and change the **Short**, **Long**, and **Default** to **Skim**.
- **Apply** and **Accept** the form.



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** (***D:\users\training\COURSEWORK\Projects\camera***) but do not **Close**.

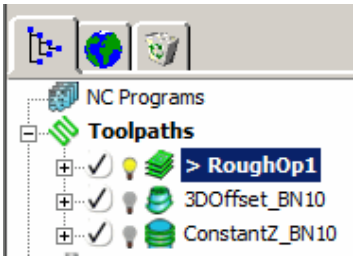
Other Constant Z Options

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

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

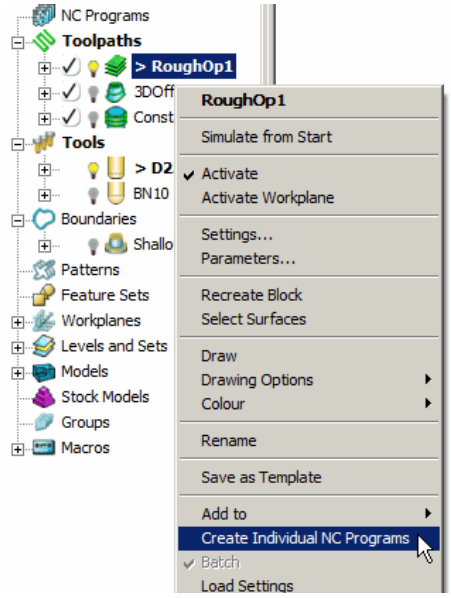
Post Processing and NC Programs.

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

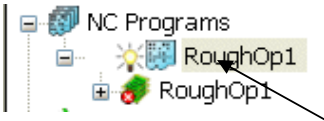


All of the toolpaths that have been created in this chapter should appear in the explorer like this.
This example will illustrate with the output of one single toolpath – **Rough Op1** to be output as a post-processed ncdata file.

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

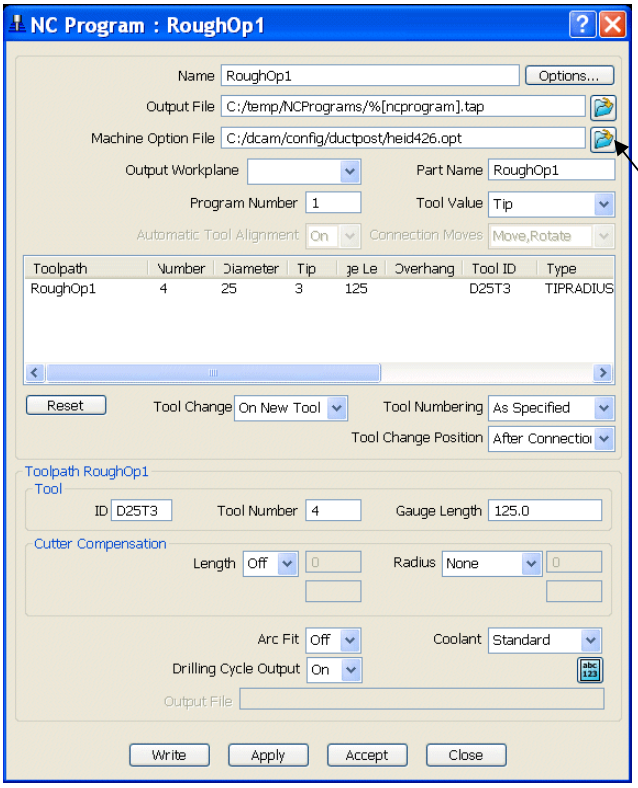


- Select the **Create Individual NC Programs** option.



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

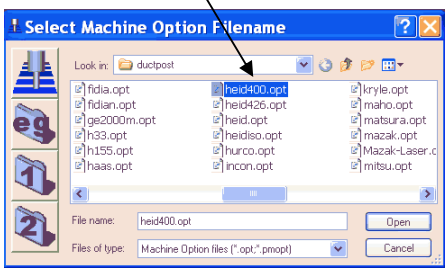
- Right click over the NC Program and select **Settings**.



The path to where the program will be output.

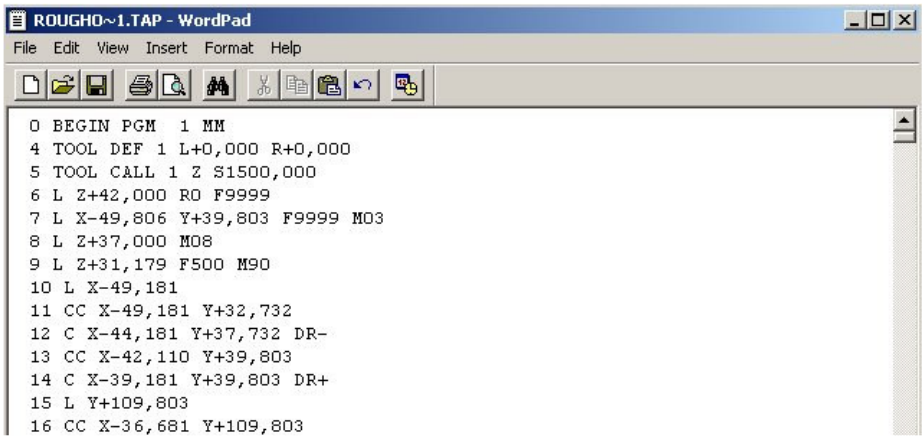
Before post processing can occur the required option file (*.opt) must be selected.

- 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.

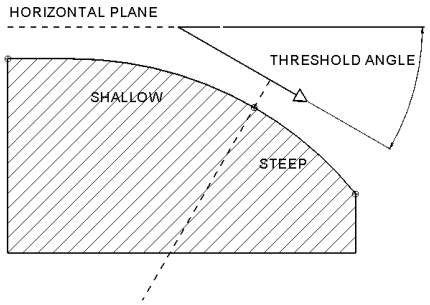


Corner Finishing.

Introduction.

Corner Finishing strategies are mainly used to remove material in corners that could not be removed by larger tools in previous paths. 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).

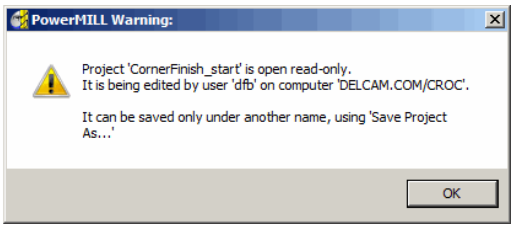
All **Corner Finishing** strategies allow a **Threshold Angle** to be specified. The **Threshold Angle -** determines the angle, from the horizontal, at which steep and shallow portions are split when you select a type of **Steep** or **Shallow**. This avoids problems associated with the tool running up or down steep slopes. For instance the user could apply a stitch strategy to track across the steep areas and parallel strategy for shallow areas. Also a higher feed rate could be used on the shallow areas and a lower feed rate on the steep areas, which if applied first is more likely to be taking a heavy cut.



Corner Pencil Finishing

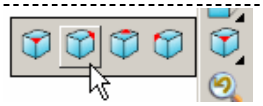
This option produces single pass tool tracks along the intersection between sharp internal corners of the component surface.

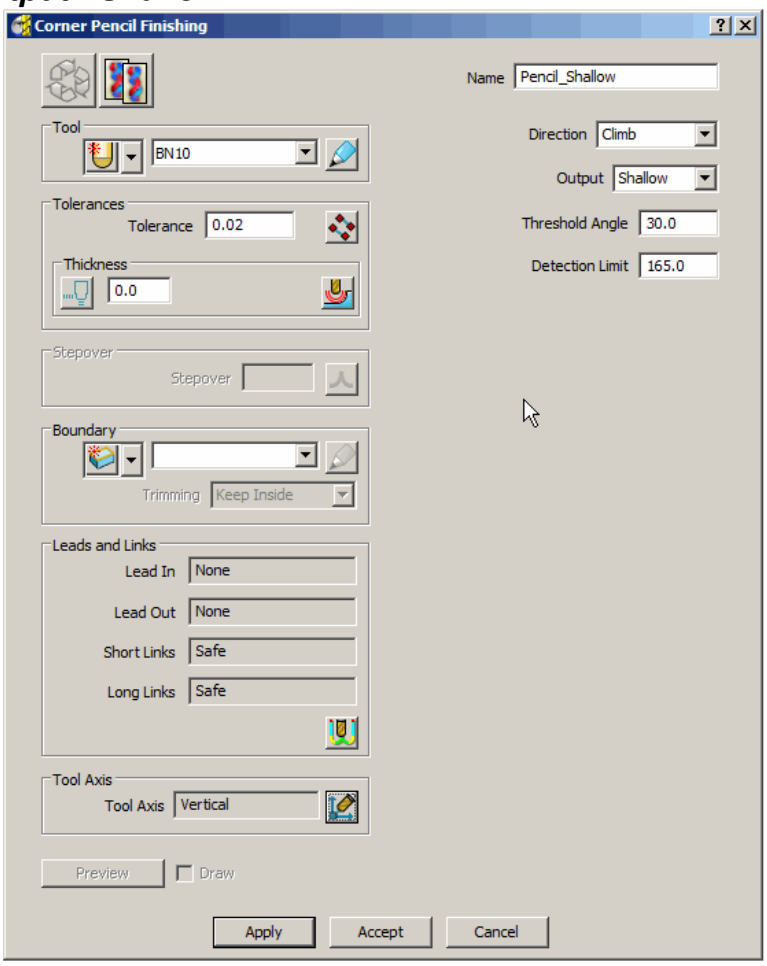
- From the **File** pull down menu select **Delete All** and from the **Tools** pull down menu select **Reset** forms.
- From **File – Open Project** and in the form browse to:- **D:\users\training\PowerMILL_Data\Projects\CornerFinish_Start**
- Select **OK** to load the existing **Project** into **PowerMILL**.



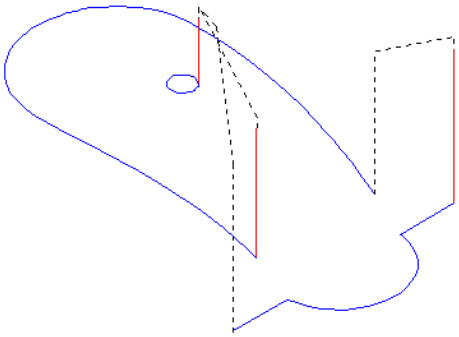
The **Project** is **Locked** and cannot be modified unless saved as a new file (or the original **Project's *Lock*** file is deleted).

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

- Select an **Iso2** view. 
- Reset the Rapid Move Heights from.
- Open the **Toolpath Strategies** form  and from **Finishing** select **Corner Pencil**.
- Input ***Name*** as **Pencil_Shallow**.
- Select ***Output*** - **Shallow**.



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

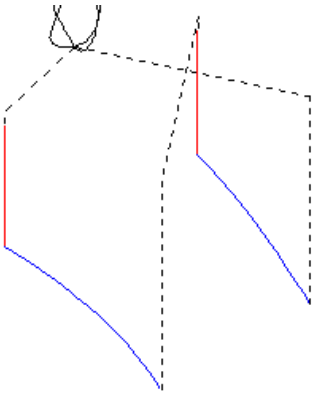


The toolpath is produced. The steep area is not machined.

- **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**.
- **Apply** the **pencil** machining toolpath (in the **Steep** areas only).



Only the steep area is machined.

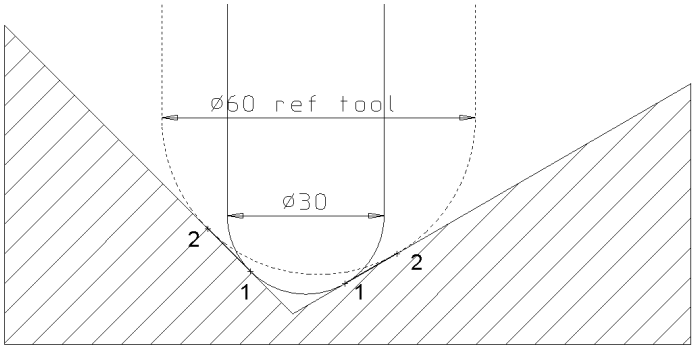
- **Simulate** the toolpath. **Rename** the toolpath as **Pencil_Steep**.
- In the main toolbar select **File** - **Save Project**.

The contents of the **Project** will be updated**:-**
D:\users\training\COURSEWORK\PowerMILL-Projects\CornerFinish

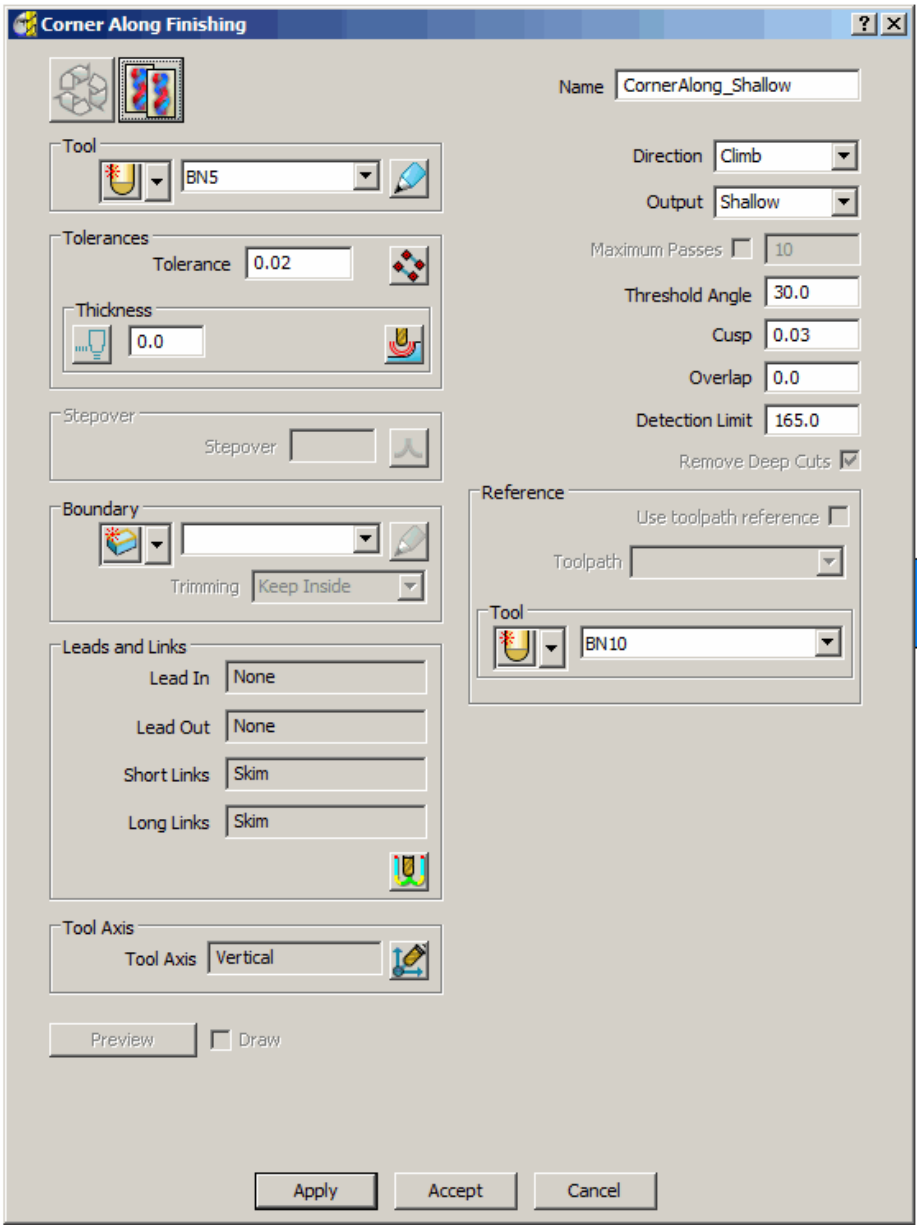
Corner Along Finishing

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

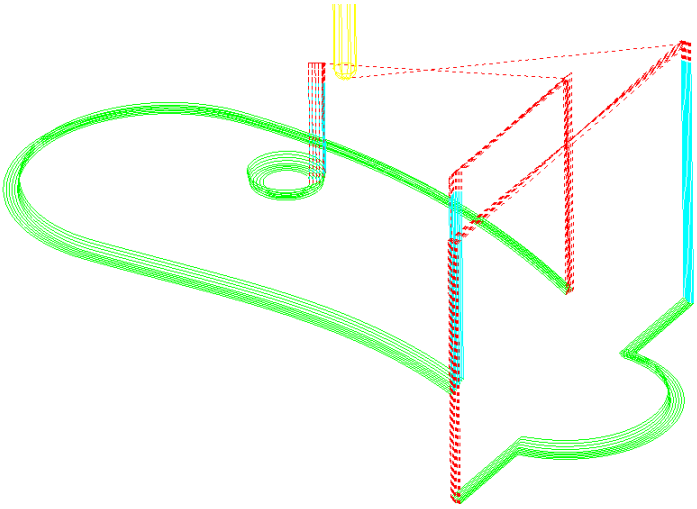
In this example the main finishing tool is a 12 Ball nose (reference) to be **Corner** finished with a diameter 5 Ball-nose.



- Open the **Toolpath strategies** form  and from **Finishing** select **Corner Along Finishing**.
- Define a **dia 5 ball nose** tool named **BN5**.
- In the **Reference Area**, select the tool **BN10**.
- Change ***Output*** to **Shallow**.
- Set **Cusp** to **0.03**.
- Check **Direction** is set to **Climb**
- Input ***Name*** *as* **CornerAlong_Shallow**.



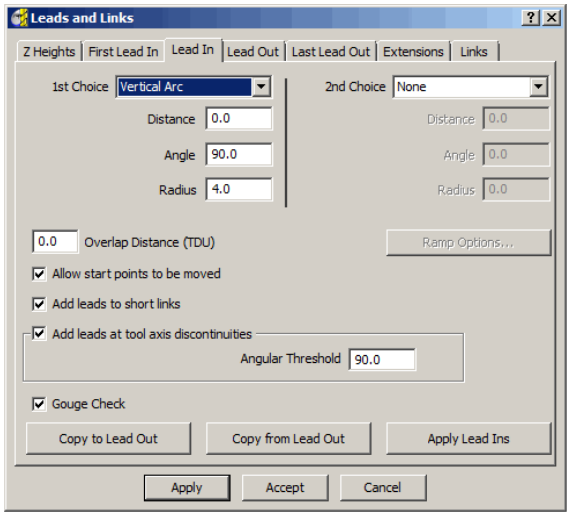
- **Apply** the form and **Cancel**.



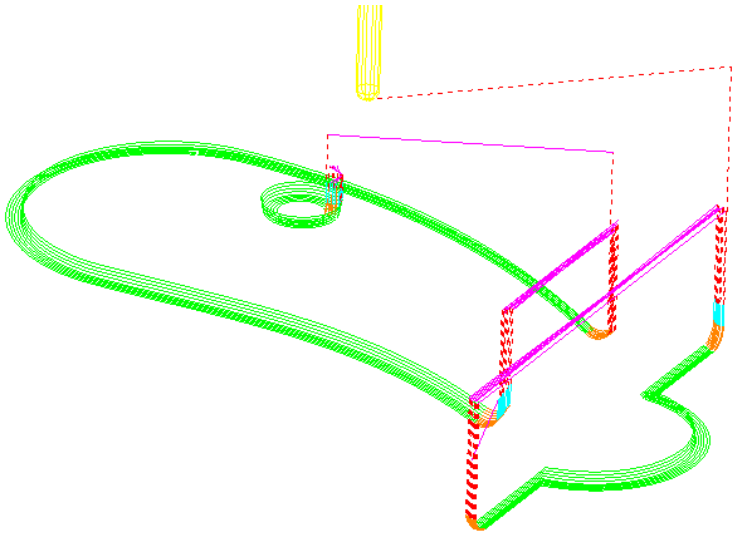
Animate the toolpath and note that it has machined the shallow areas from outside to inwards in a Climb milling direction.

The **Leads and Links** need to be improved.

- 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**.

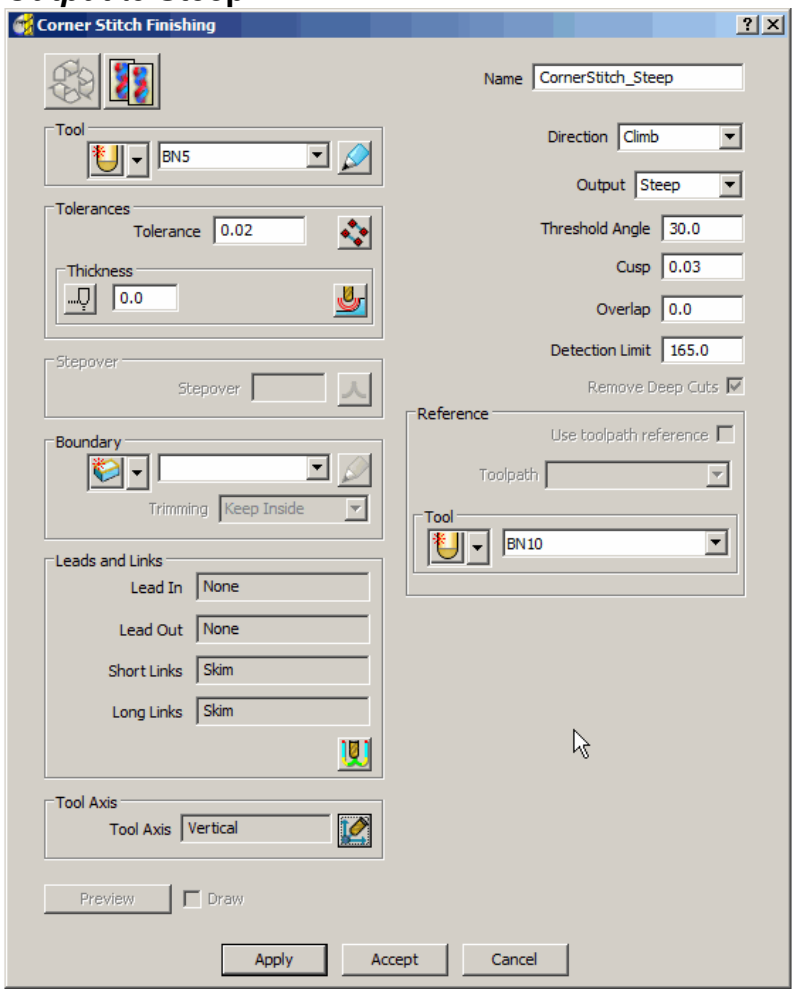


The **Vertical Arc** lead provides a smoother transition onto the work piece with the tool already moving at the specified cutting feed rate.

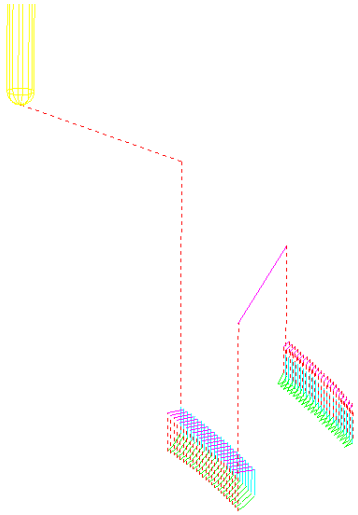
- Open the **Leads and Links** form changing the **Lead In/out** from **Vertical Arc** back to **None** and **Accept** (ready for the next toolpath).

Corner Stitch Finishing

- From the **Finishing strategies** form, select **Corner Stitch Finishing**.
- Change ***Output*** to **Steep**.



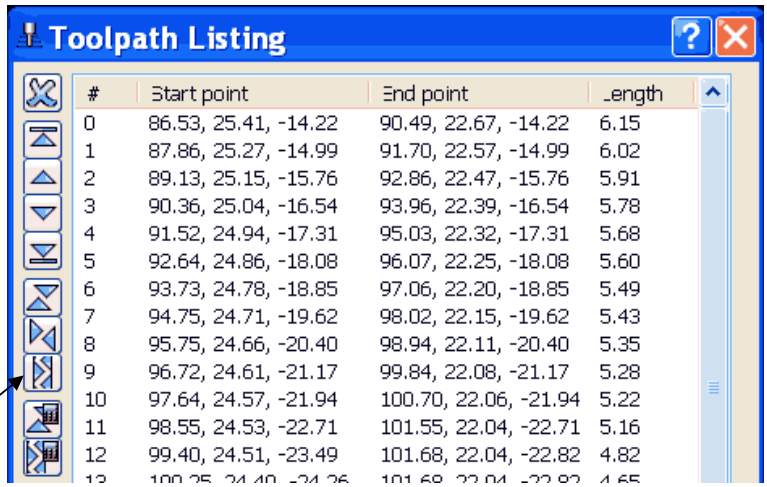
- **Apply** the toolpath and **Cancel**.



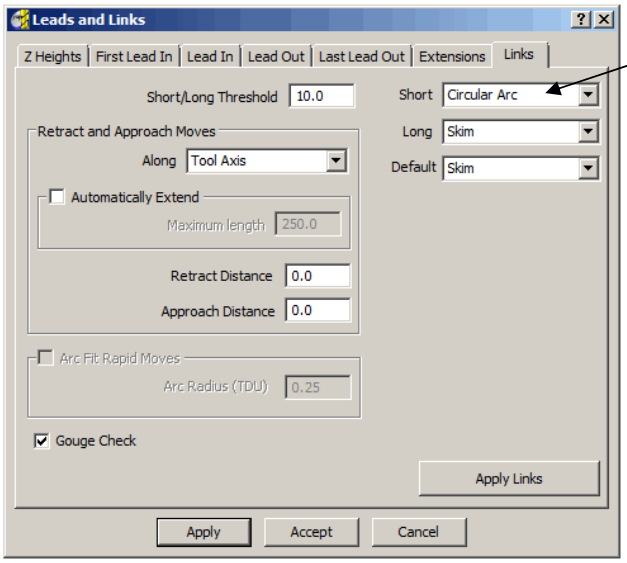
This time only the steep areas have been machined.

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**.

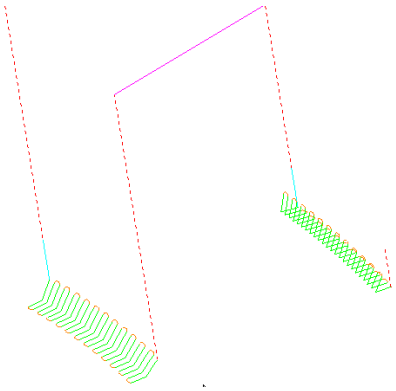


- Select **Alternate directions.**



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

The steep areas now run in a zig-zag direction rather than one way 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:-
D:\users\training\COURSEWORK\Projects\CornerFinish

Exercise

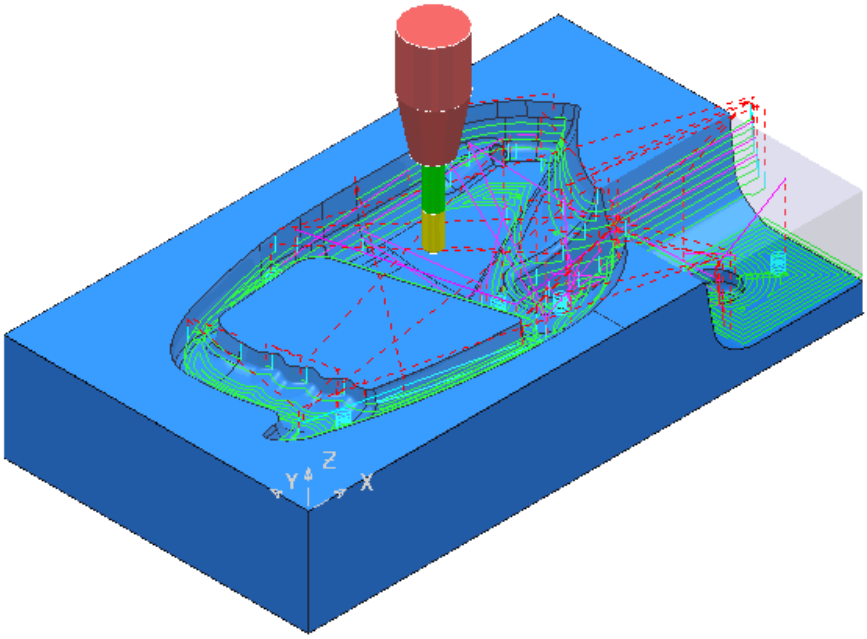
Create Finish Machining for an existing *Rough Machined Project*

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

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

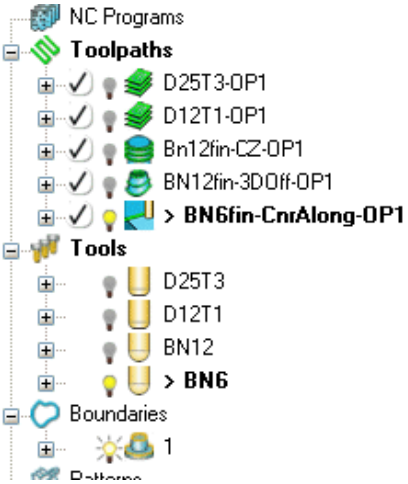
- Select **OK** to load the existing **Project** into **PowerMILL**.
- From **File** - **Save Project As**:-

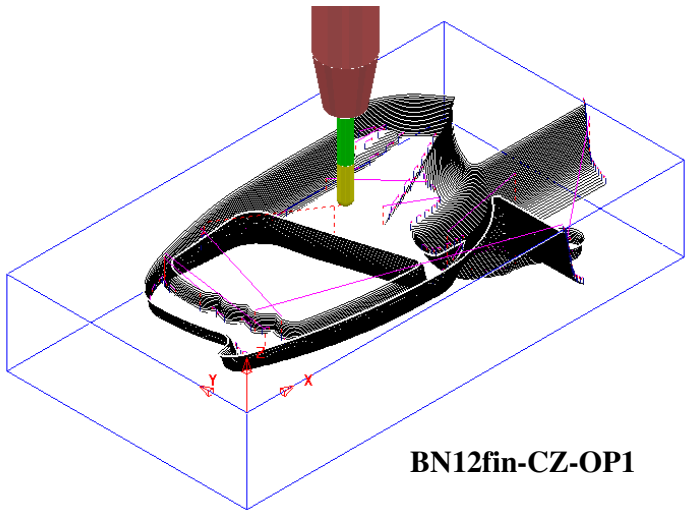
D:\users\training\COURSEWORK\PowerMILL_Projects\ChainSawDie-Ex



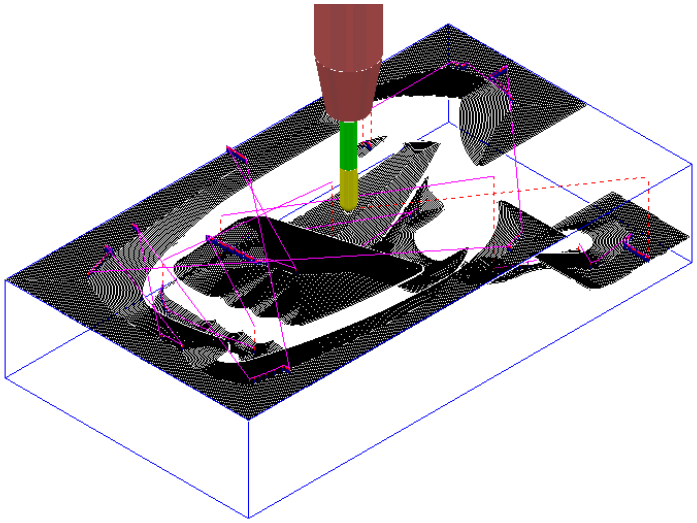
- **Activate** the existing **Ball Nose tool - BN12**
- Create a **Shallow Boundary** with **Thickness 0**.
- Create a **Constant Z** finishing strategy ***limited*** to the **Steep** areas and a **3D Offset** finishing strategy ***limited*** to the **Shallow** areas.
- **Activate** the existing **Ball Nose tool - BN6**
- Create a **Rest Finishing** strategy using **Corner Along** - ***referenced*** to the larger **BN12** tool.
- Perform a full **ViewMILL *simulation***.
- **Save** the **Project** to update.

Illustrations of typical **Finishing Strategy** solutions and corresponding **ViewMILL** solutions are shown on the next 2 pages.

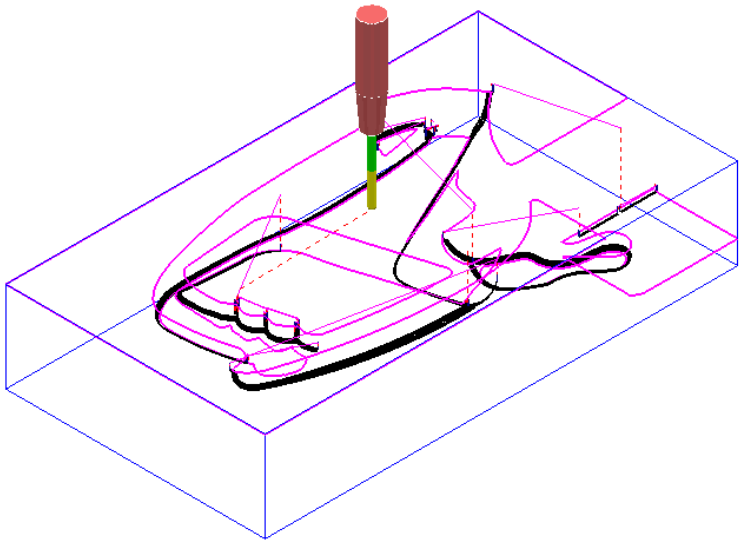




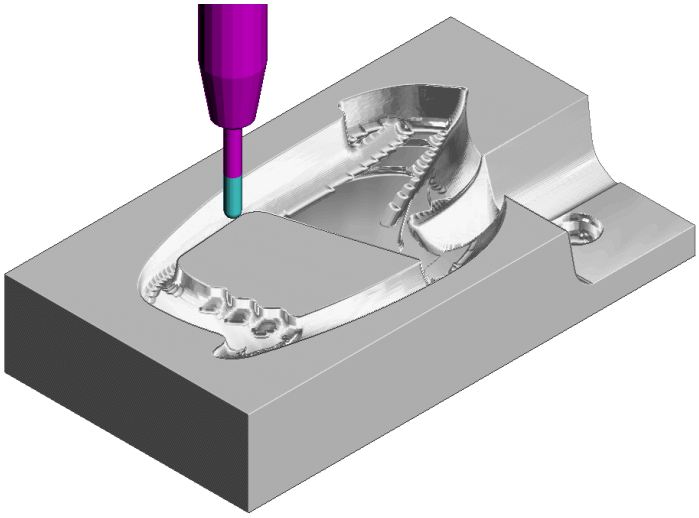
BN12fin-CZ-OP1



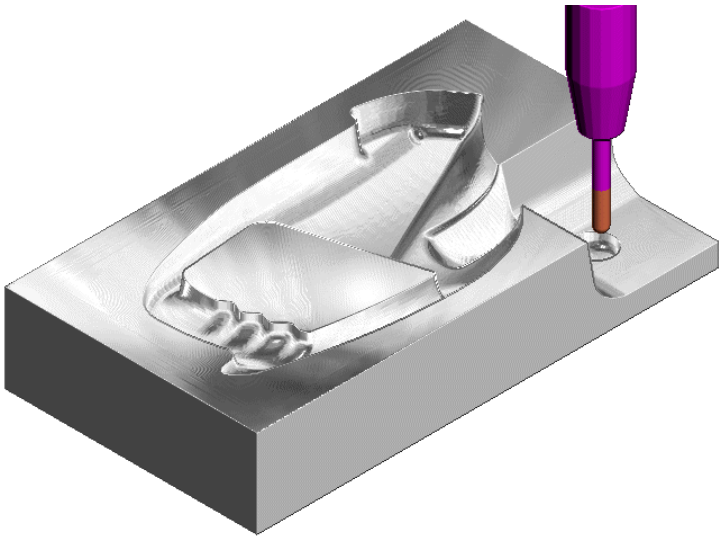
BN12fin-3DOff-OP1



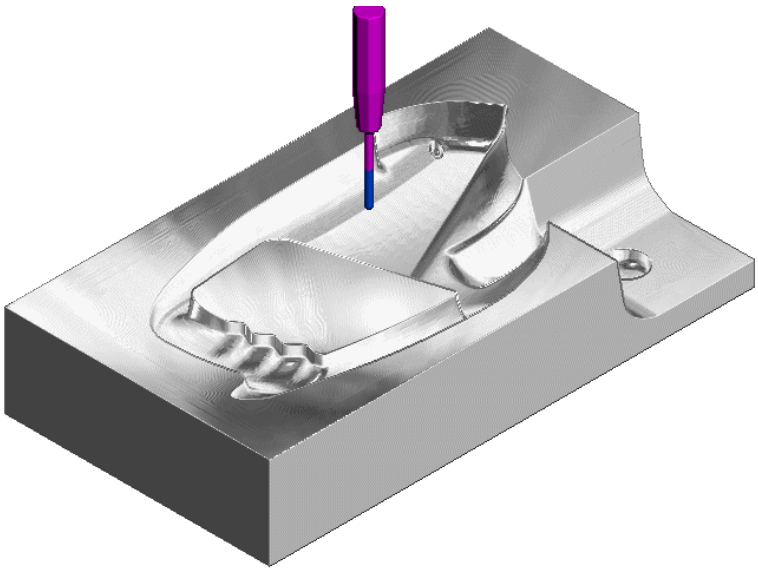
BN6fin-CnrAlong-OP1



BN12fin-CZ-OP1



BN12fin-3DOff-OP1



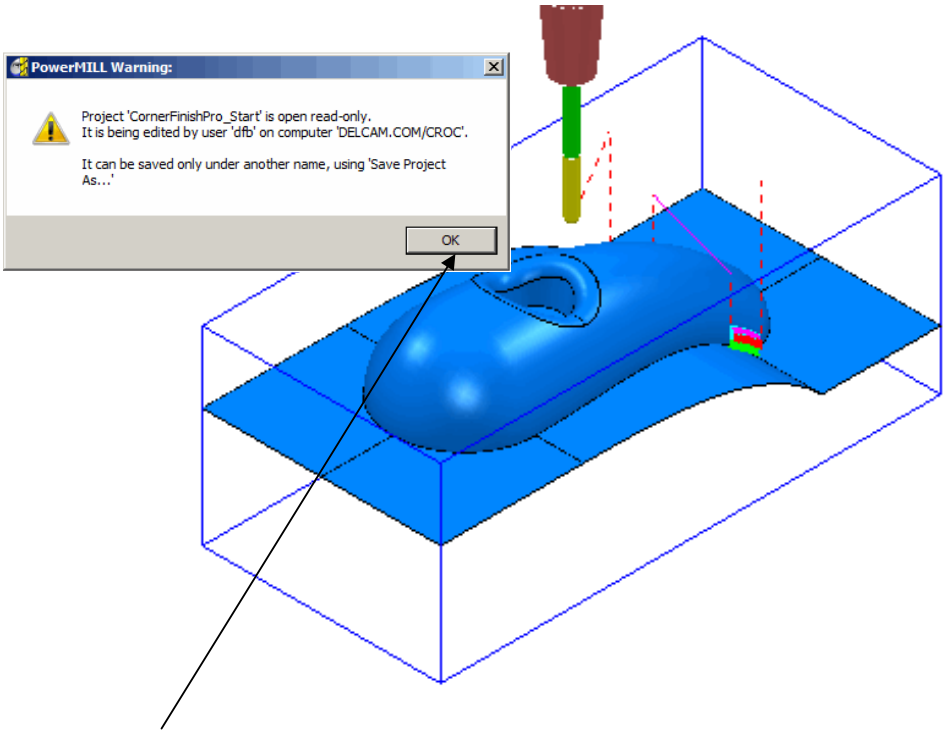
BN6fin-CnrAlong-OP1

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**.

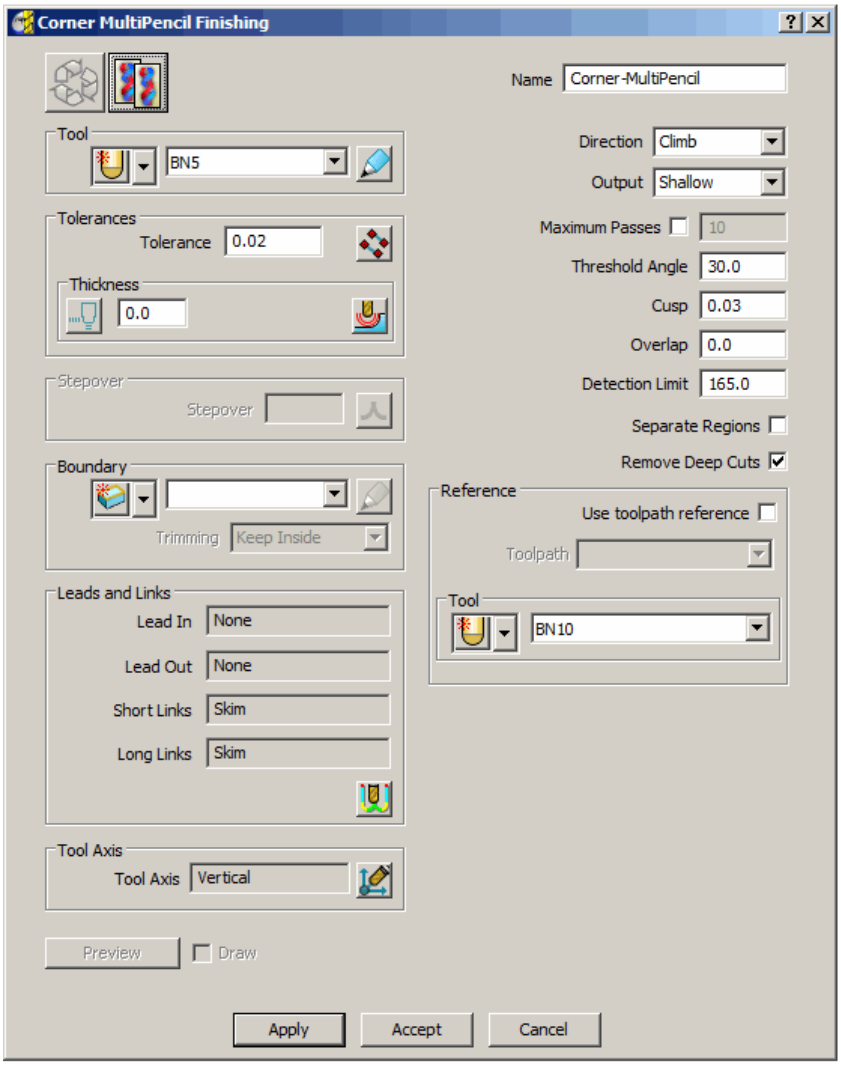
- **Delete All** and **Reset forms**.
- From **File – Open Project** select the read-only **Project** :- **D:\users\training\PowerMILL_Data\Projects\CornerFinishPro_Start**



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

Corner MultiPencil Finishing

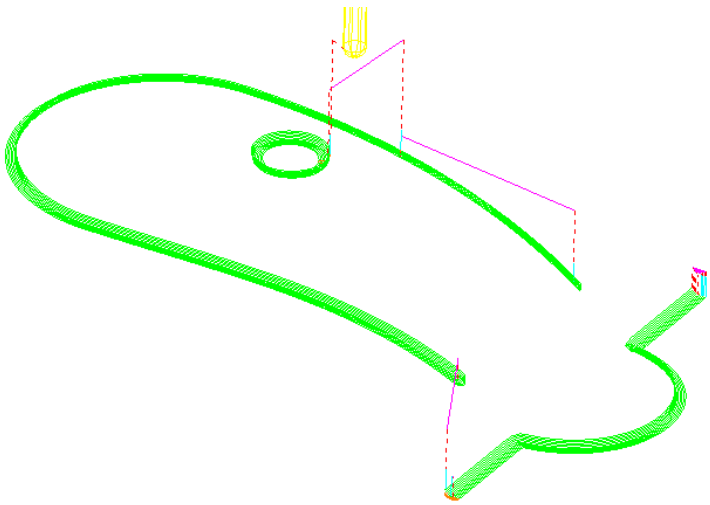
- From the **Toolpath strategies** form, select **Corner MultiPencil Finishing**



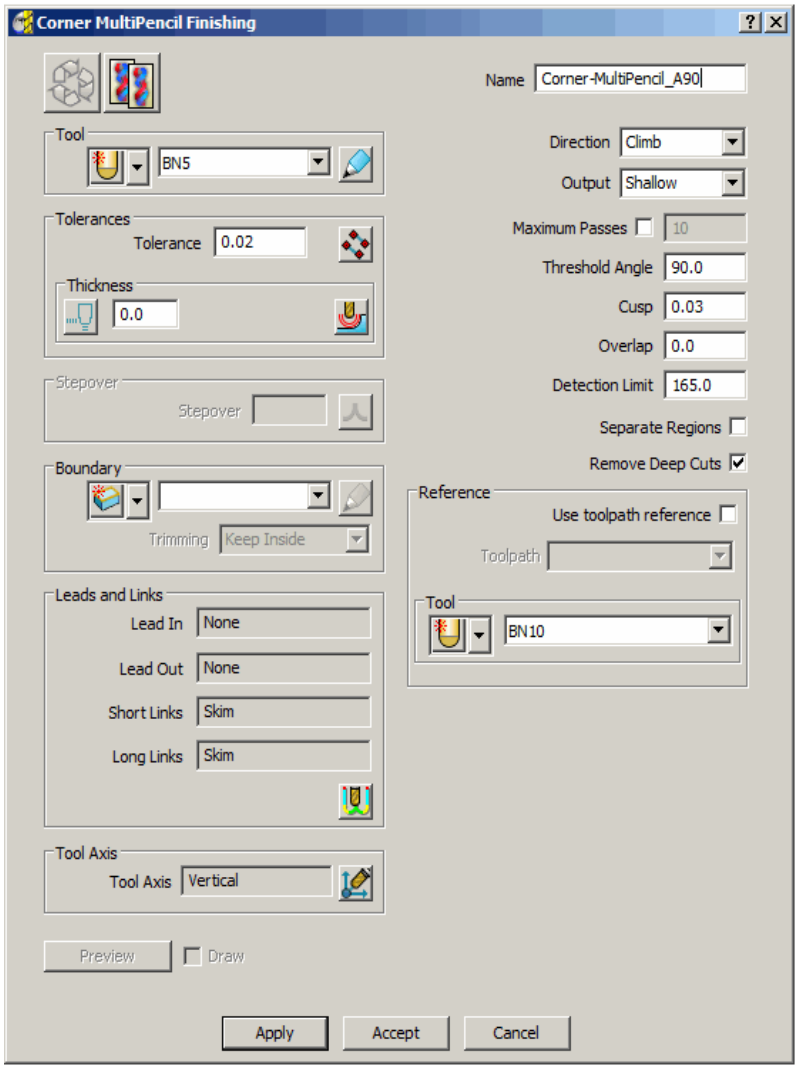
- Change **Output** back to **Shallow**

- Select **Apply** 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.



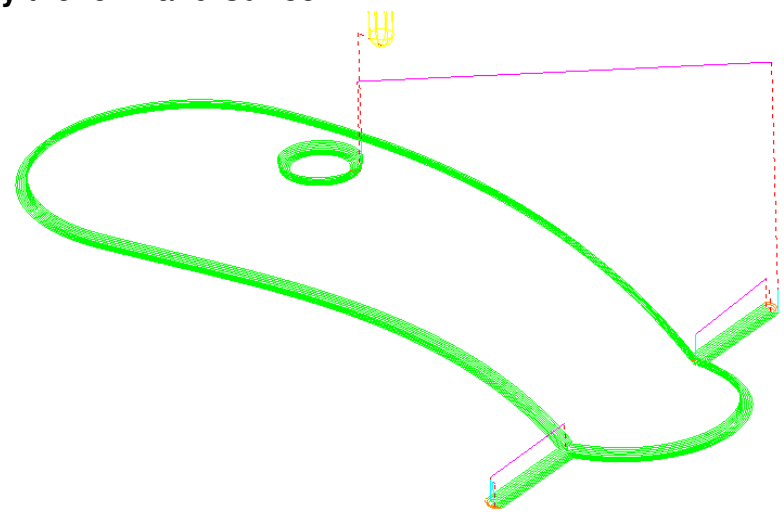
- Select the **Copy Toolpath** icon from the form. 



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

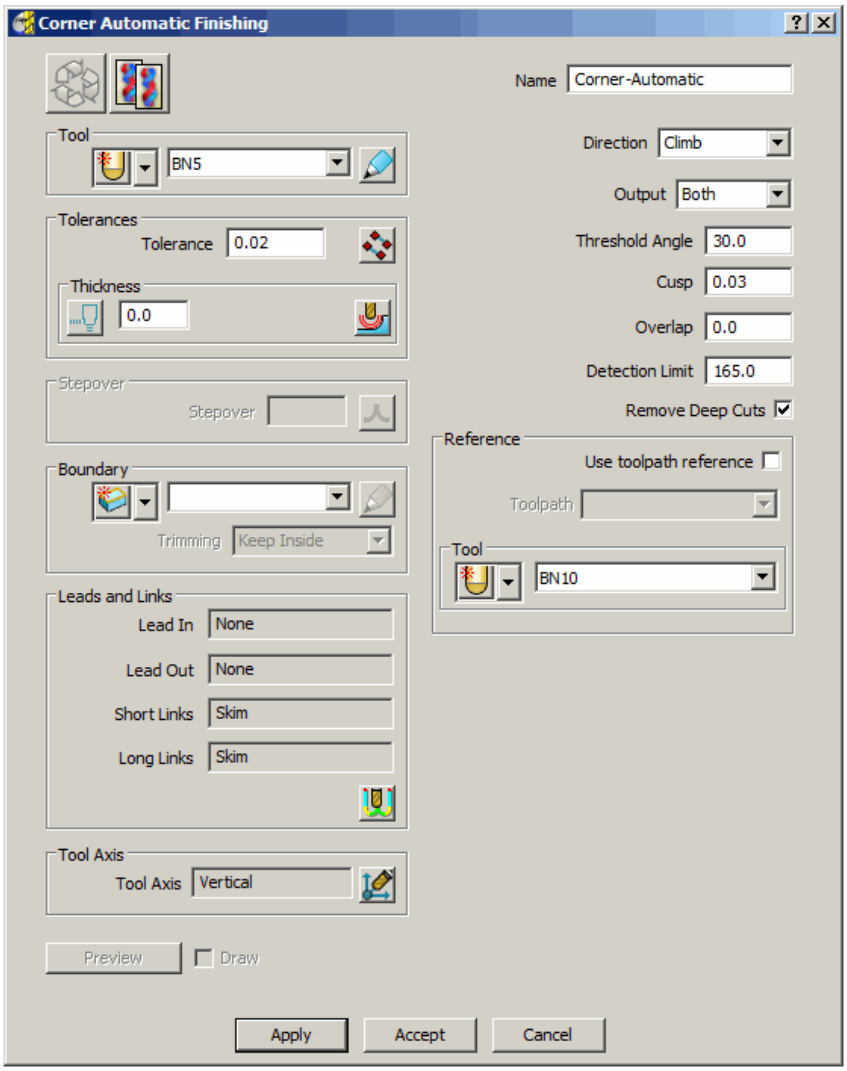
This has the effect of creating an unlimited **Shallow** machining area for the toolpath. Using these settings the same strategy would be created if the ***Output*** - **Both** is applied. If ***Output*** **- Steep** is applied no toolpath would be created.

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



Corner Automatic Finishing

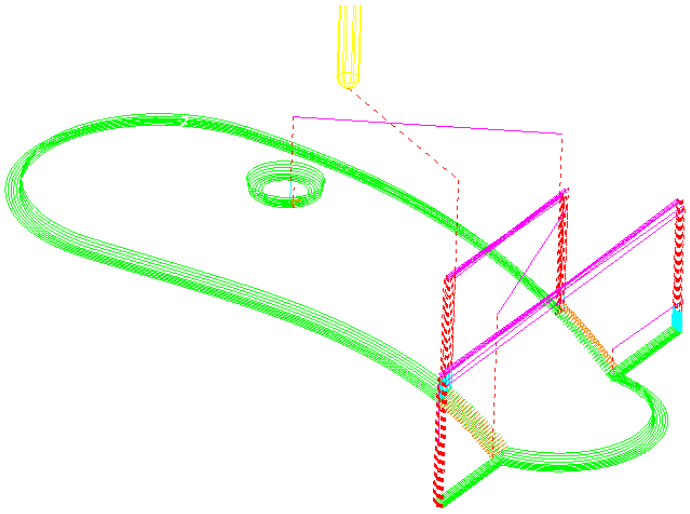
- From the **Toolpath strategies** form, select **Corner Automatic Finishing**.



- Change **Output** to **Both**.
- Change the **Threshold Angle** back to **30**.

- **Apply** the toolpath and **Cancel**.

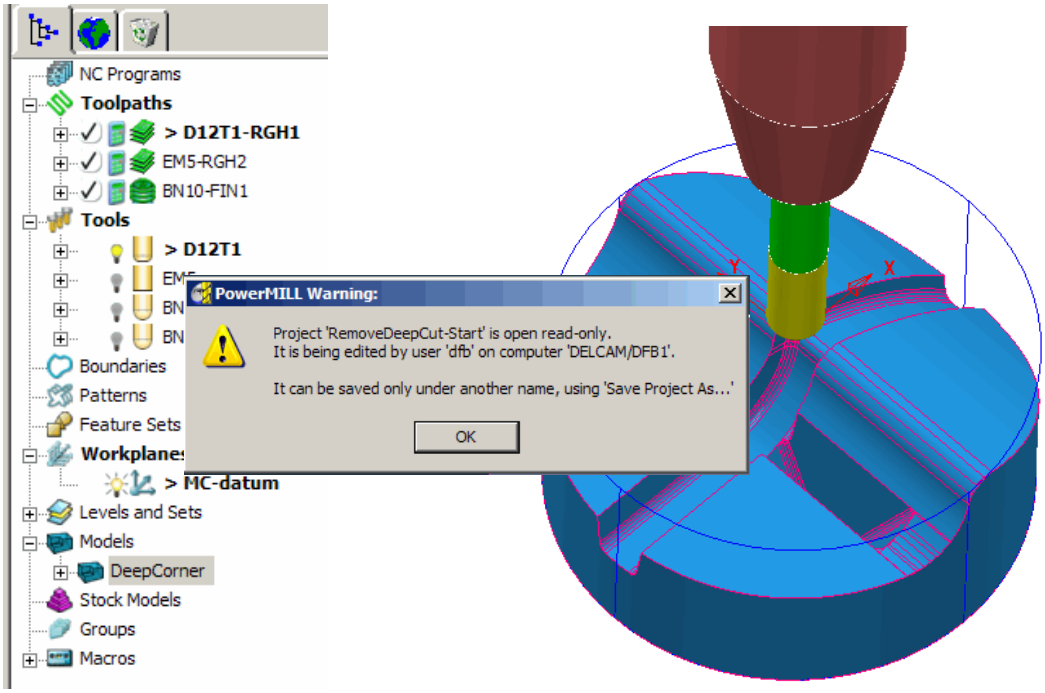
This strategy automatically uses **Along** for the **Shallow** areas and **Stitch** for the **Steep** areas. However, due to it being all one toolpath, it is not so easy to control the **Leads and Links** or the **Reordering**.



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.

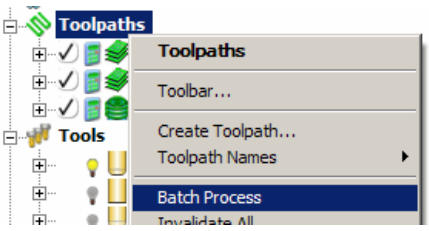
- **Delete All** and **Reset forms**.
- From **File – Open Project** select the read-only **Project** :- **D:\users\training\PowerMILL_Data\Projects\RemoveDeepCut-Start**



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

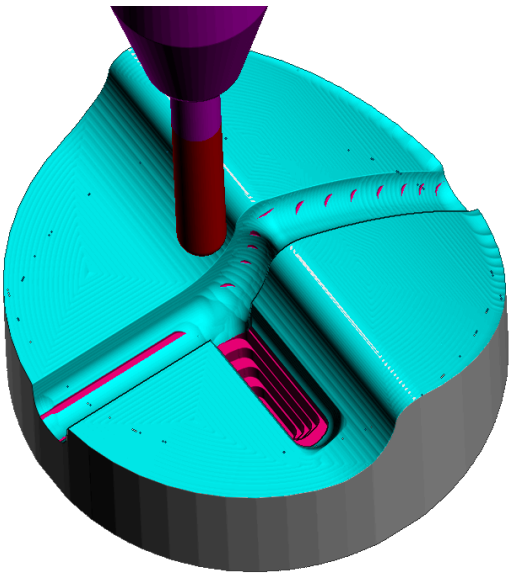
Note: The existing 3 strategies in the above **Project** are as yet, unprocessed (***Calulator*** icon).

- Right mouse click on **Toolpaths** in the **explorer** and from the local menu select **Batch Process**.



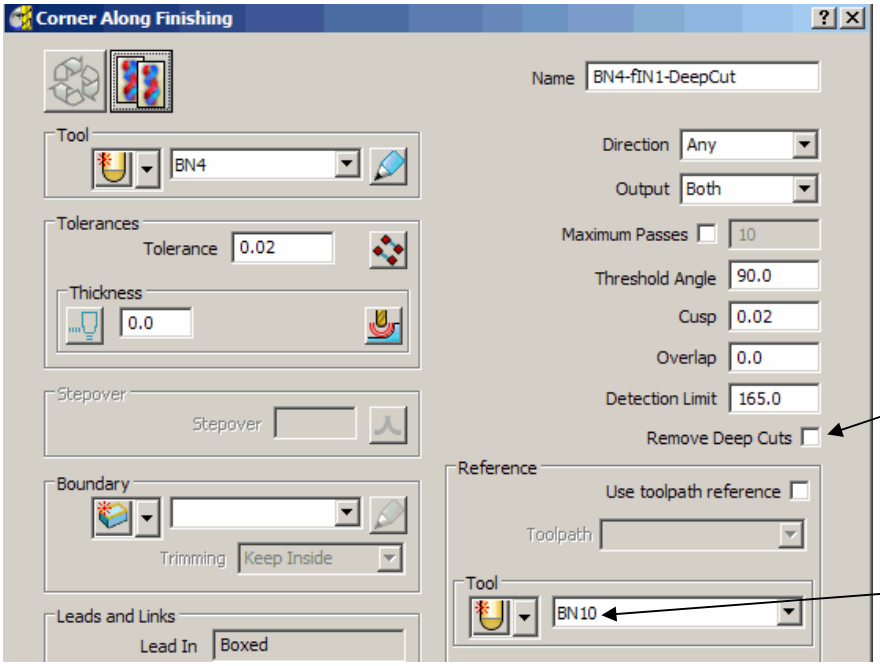
The 3 **strategies** will be processed in sequential order.

- **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.

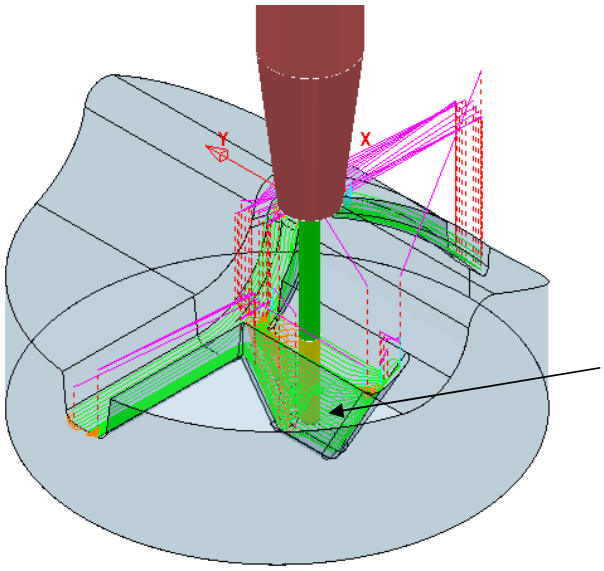


The ***Pale Blue*** area shows where the ***Dia 10 Ball Nosed*** tool has successfully machined. The deep pocket (**Purple**) is inaccessible and still includes large steps down the side wall. This excess stock could cause a smaller finishing tool to break if it attempts to **Corner Finish** down the 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 Along** finishing.

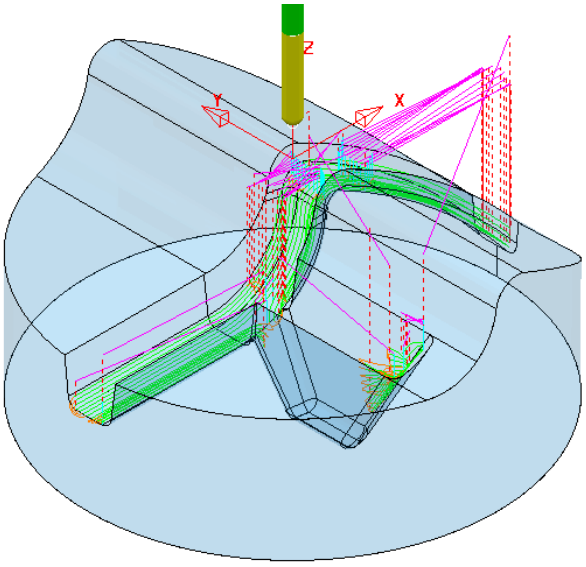


- Enter the data exactly as shown above and select **Apply**.



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 **Corner Along** finishing.form for the existing strategy and select **Make a Copy.**
- **Rename** the strategy as **BN4-fIN1-NODeepCut** and ensure that the **Remove Deep Cuts** box is ticked before selecting **Apply**.



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

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

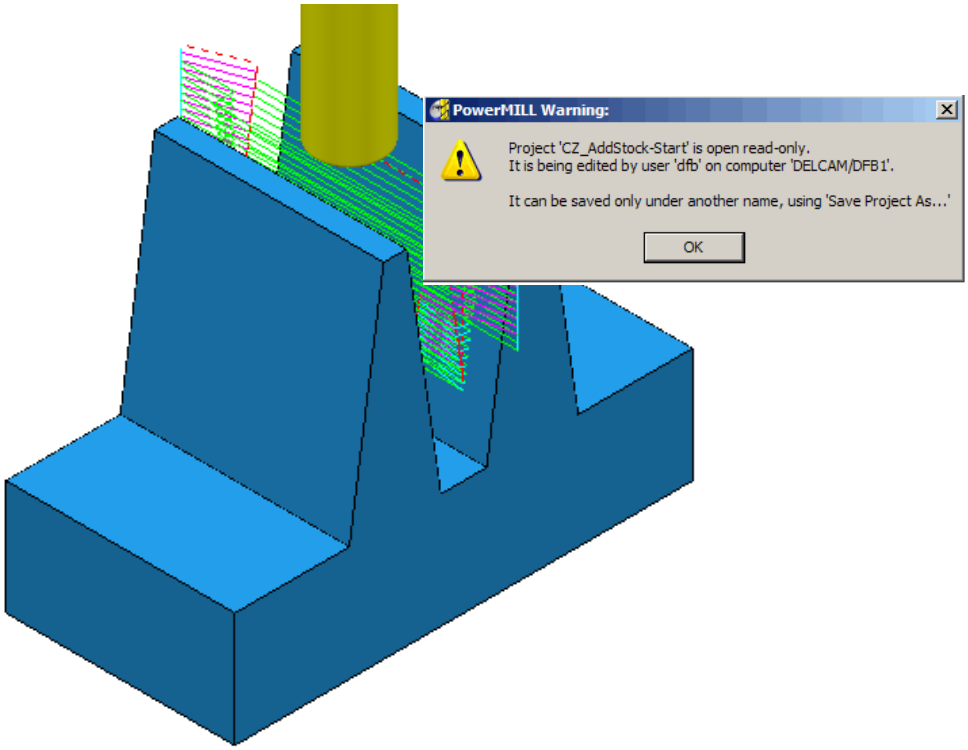
Exercise

- 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** :- **D:\users\training\PowerMILL_Data\Projects\CZ_AddStock-Start**

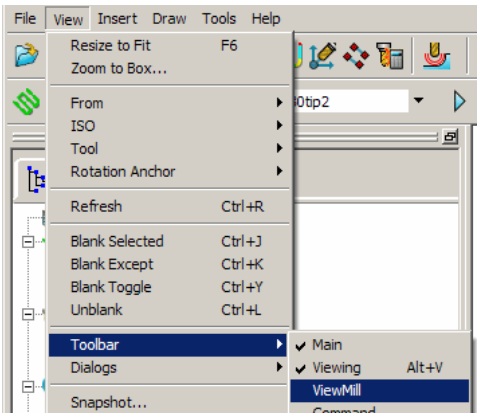


- Select **OK** in the ***PowerMILL Warning*** form.

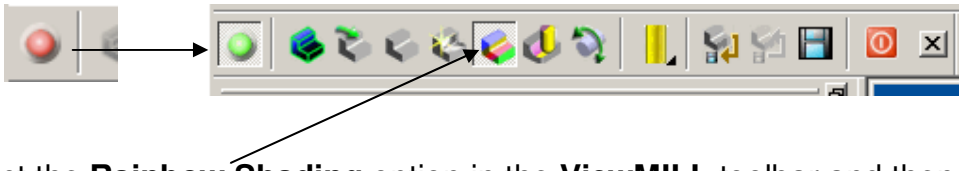
- **Save Project As**:- **D:\users\training\COURSEWORK\PowerMILL-Projects\CZ_AddStock-EX1**

- ***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**.

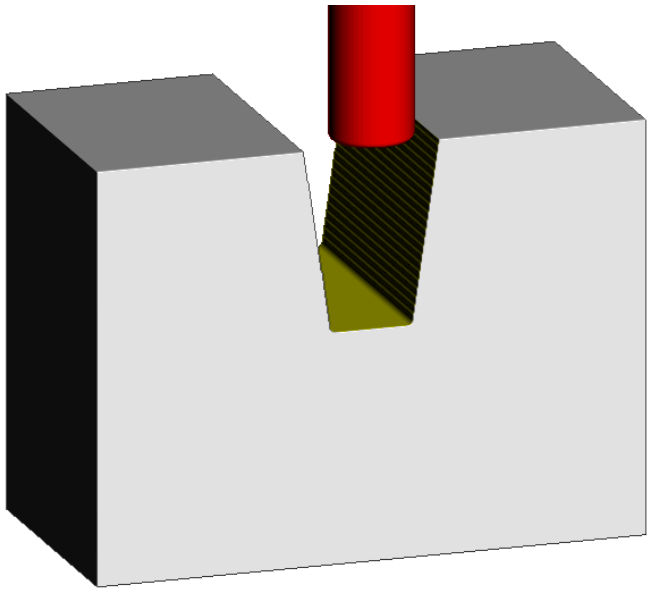
- From the top **Pulldown** menus select (tick) **View - Toolbar - ViewMILL** to open the **ViewMILL** toolbar (if not already open).



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

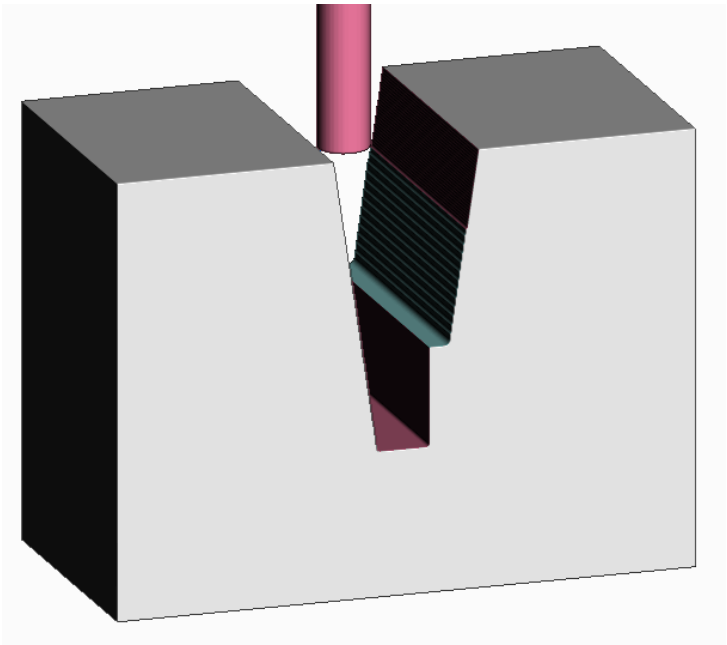


- Select the **Rainbow Shading** option in the **ViewMILL** toolbar and then select **Play** in the **simulation** toolbar.



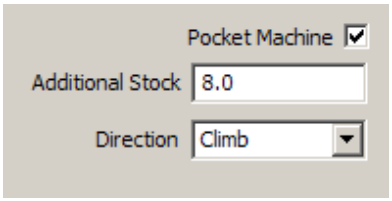
The **Roughing** strategy has not fully machined a large volume of **stock** towards the base of the slot. This will have to be removed as part of a **Constant Z *finishing*** strategy.

- ***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.



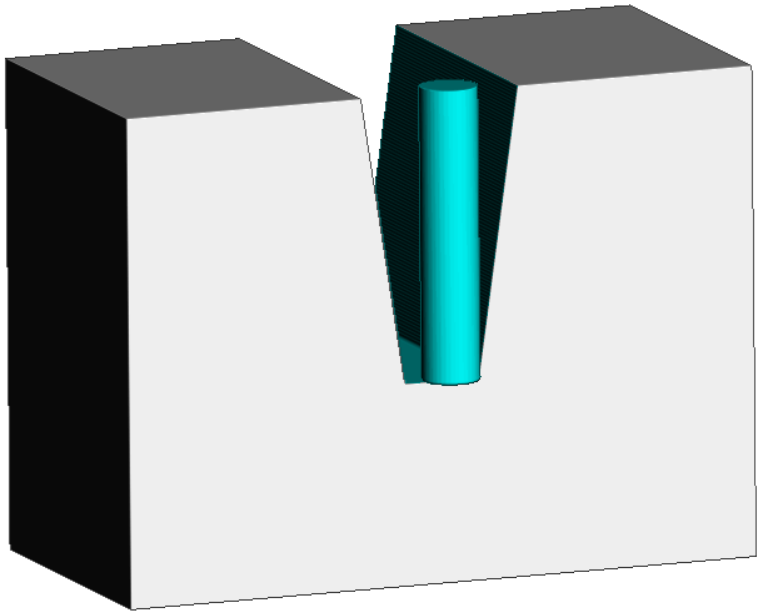
The existing **Constant Z** finishing strategy is ploughing into the stock that was inaccessible to the previously ***simulated*** **Roughing** toolpath. The excessive loading is likely to result in ***tool breakage***. To avoid this problem the **Constant Z** strategy will be modified to include the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



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.

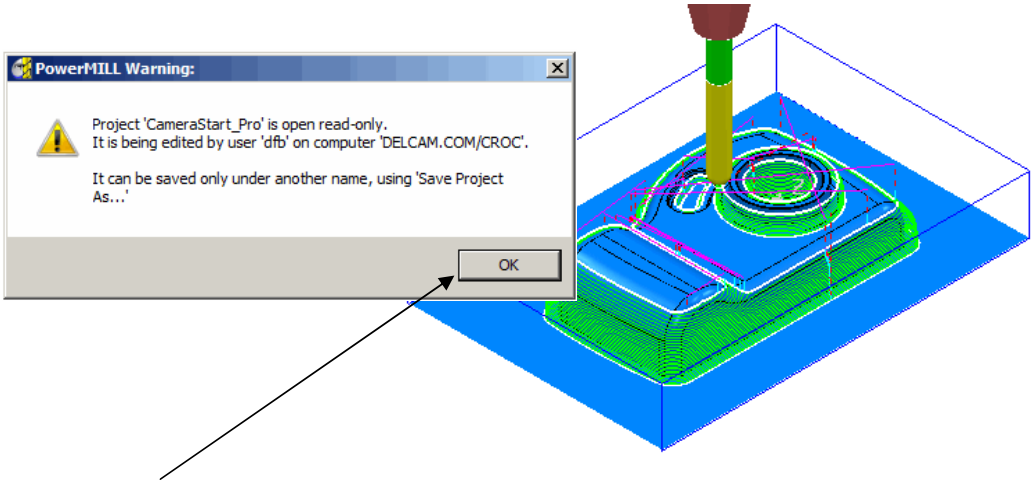
- Repeat the **ViewMILL simulation**, starting with the **Roughing** strategy as before but then following with the new **Constant Z** strategy that has the **Additional Stock** value of **8.0** included.



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.

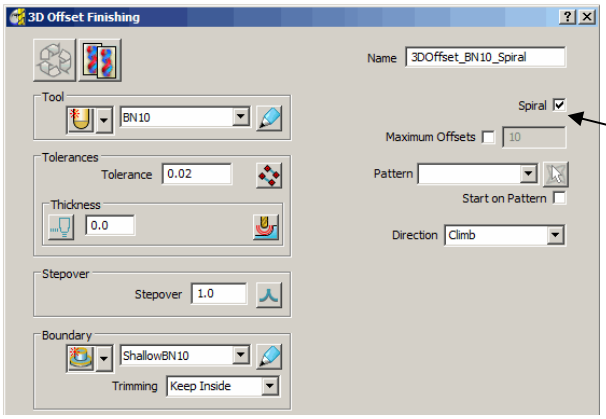
3D Offset Finishing

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

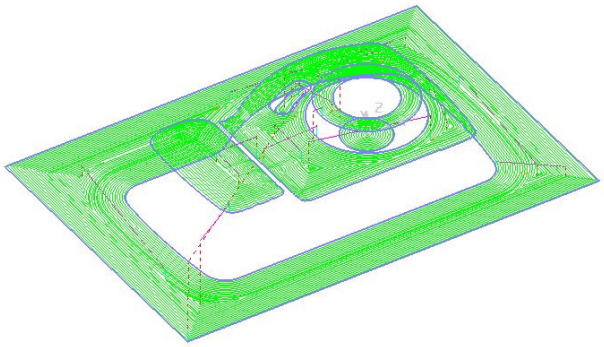


- Select **OK** to load the existing **Project** into **PowerMILL**.
- From **File** – **Save Project As**:- **D:\users\training\COURSEWORK\PowerMILL_Projects\Camera_Pro**

- 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 **3DOffsetBN10_Spiral**.
- Tick the box **Spiral**.
- Leave all other values the same then **Apply** and **Cancel** the form**.**



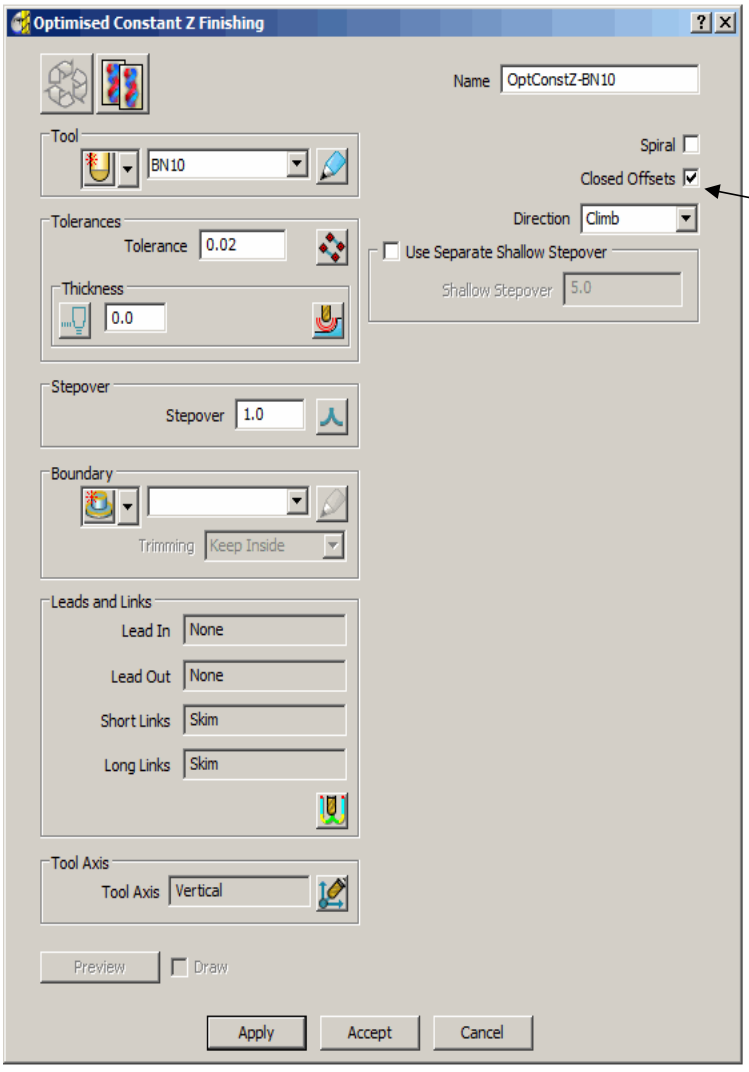
The **Spiral** option is ideal for HSM (High Speed Machining) applications where ideally toolpaths should be as continuous as possible with the minimum sudden changes direction. **Leads** and **Links** are greatly reduced using this method.

Optimised Constant Z Machining

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

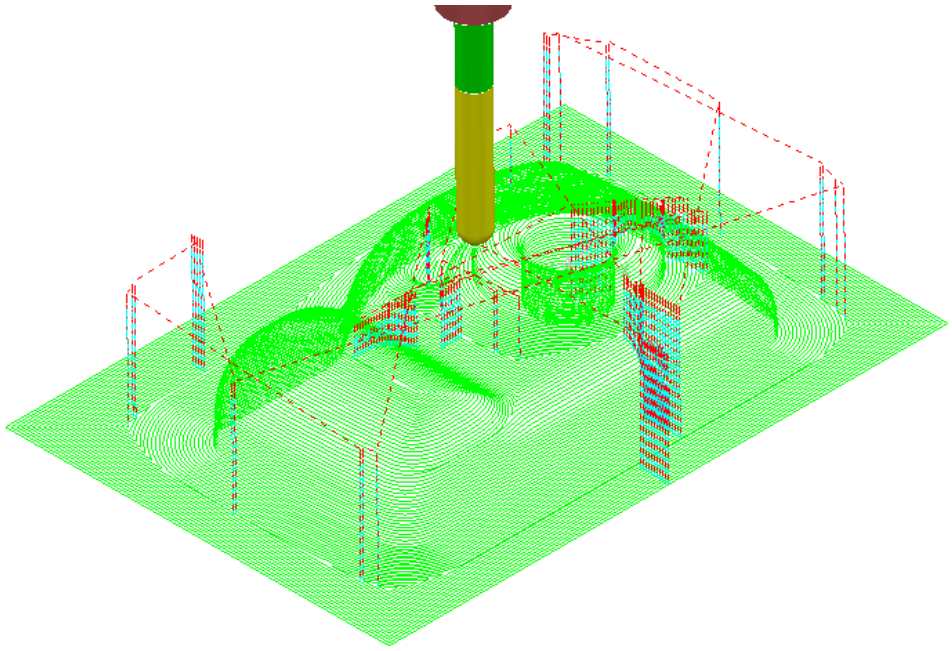
- **Deactivate** the **Shallow Boundary**, **ShallowBN10** in the **explorer**.

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



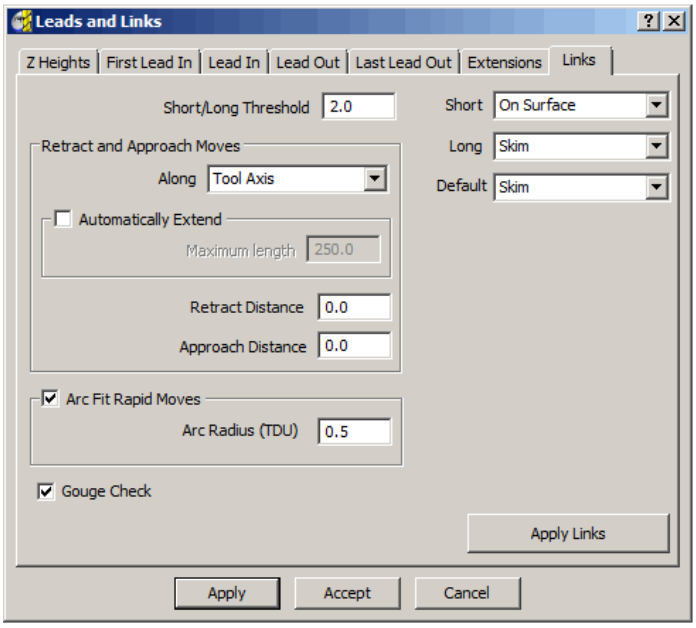
- Enter **Name** - **OptConZBN10**.
- Select **Closed Offsets**.
- Set the **Direction** to **Climb**.
- Input a **Stepover** value of **1**.
- Make sure no **Boundary** is selected.
- Enter the **Tolerance** as **0.02**.
- Reset the **Lead In** and **Lead Out** to **None**.
- **Apply** and **Cancel**.

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

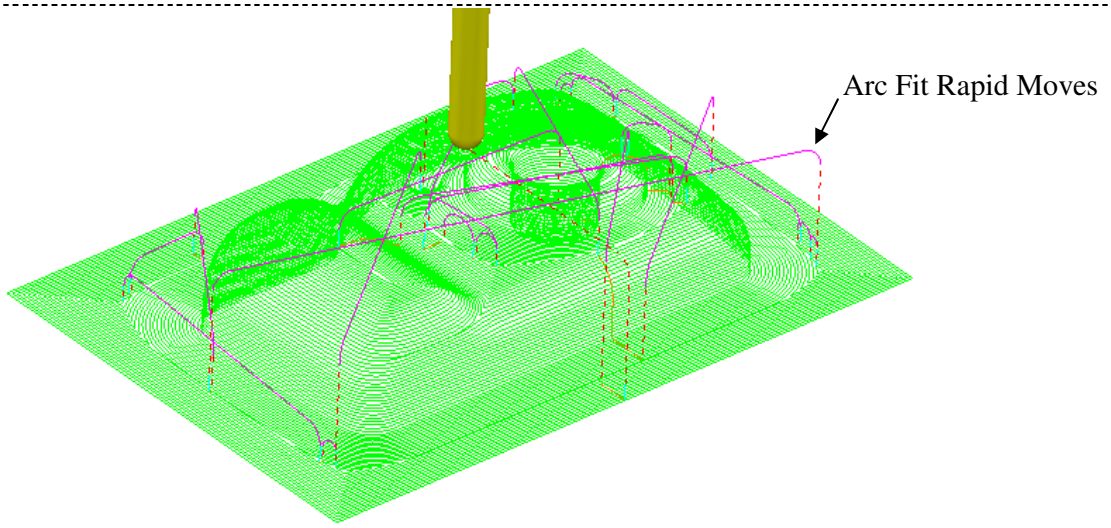


Optimized Constant Z has performed well in this example but it does take longer to calculate. Sometimes it is better to use **Boundaries** with a combination of **3D Offset** and **Constant Z**.

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



- Change the **Short/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.

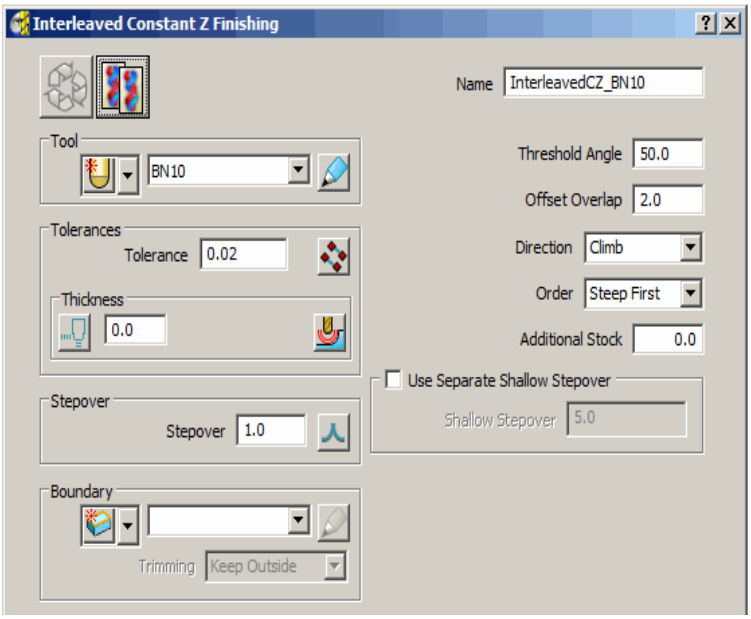


Both the **Constant Z** and **3D Offset** parts of the toolpath currently use a **1mm Stepover**. By ticking the box **Use Separate Offset Stepover** it is possible to apply a different, larger **Stepover** value to the shallow areas created with the **3D Offset** strategy used in this hybrid form.

Interleaved Constant Z

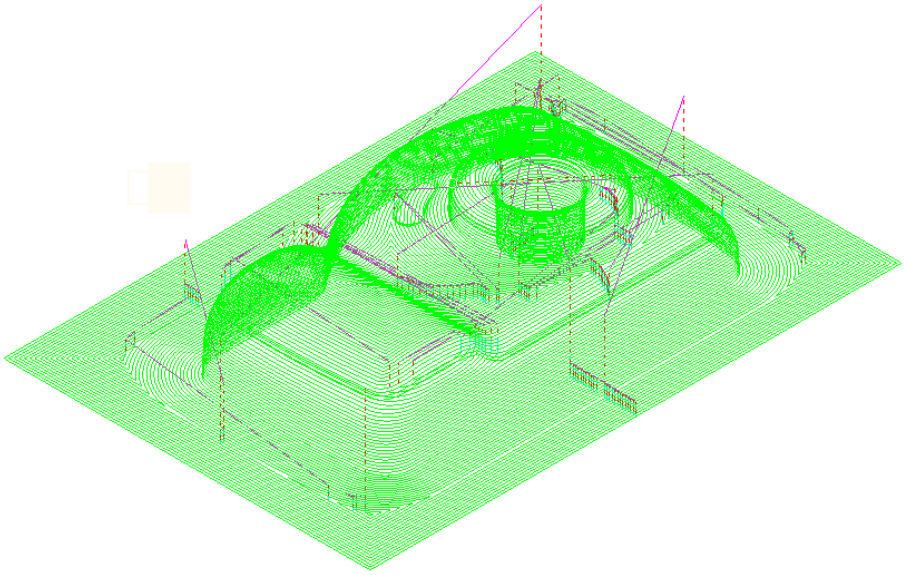
This is an alternative strategy to **Optimised Constant Z** with different options which include a user defined **Threshold Angle** between the **Constant Z,** and **3D Offset** areas of the toolpath as well the option to specify an **Overlap** value between them.

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



- Enter ***Name*** as **InterleavedCZ_BN10**.
- Input a ***Threshold Angle*** value as **50**.
- Input an ***Offset Overlap*** value of **2**.
- Set **Order** as **Steep First**.

- **Apply** and **Accept** the form.

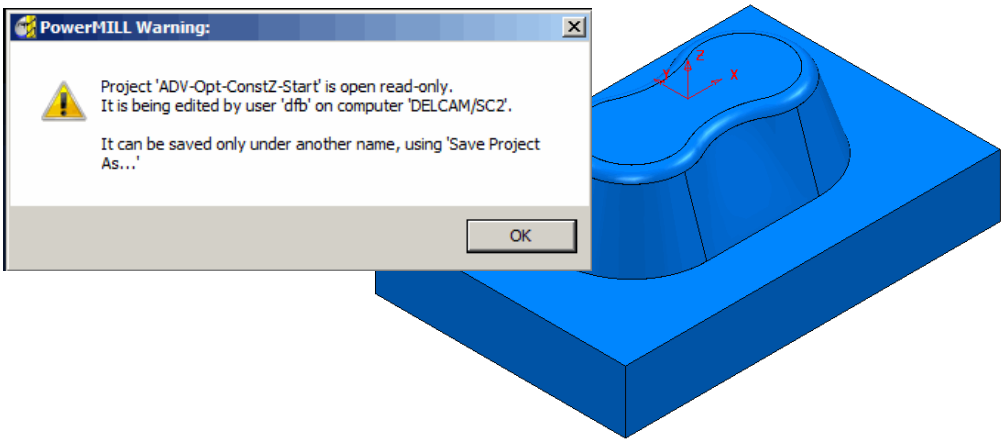


The user more has control over the extent of the ***3D Offset*** and ***Constant Z*** components of the hybrid **Interleaved Constant Z** strategy.

Optimised / Interleaved Constant Z with separate Shallow Stepover

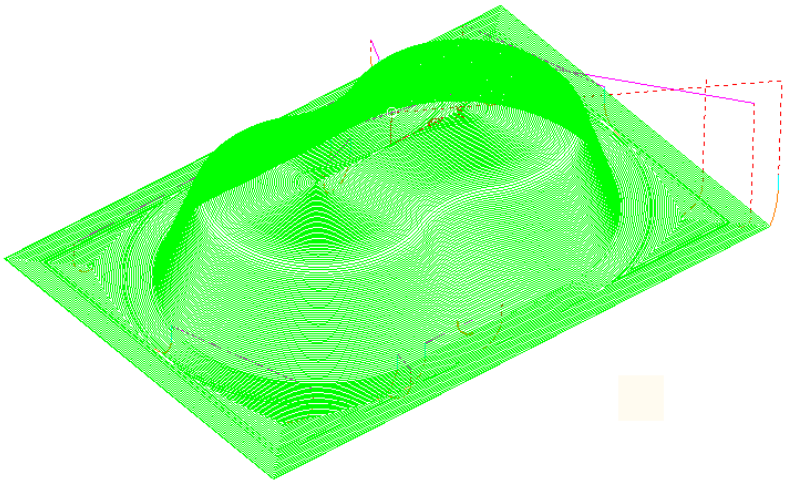
Both **Optimised** and **Interleaved**, **Constant Z** have an option to apply a separate, larger **Stepover** across the **Shallow** areas. This is designed to provide more efficient machining with a **Tip Radiused Tool**, where the **Shallow** areas are perfectly ***flat***. The **Steep** walls are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

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



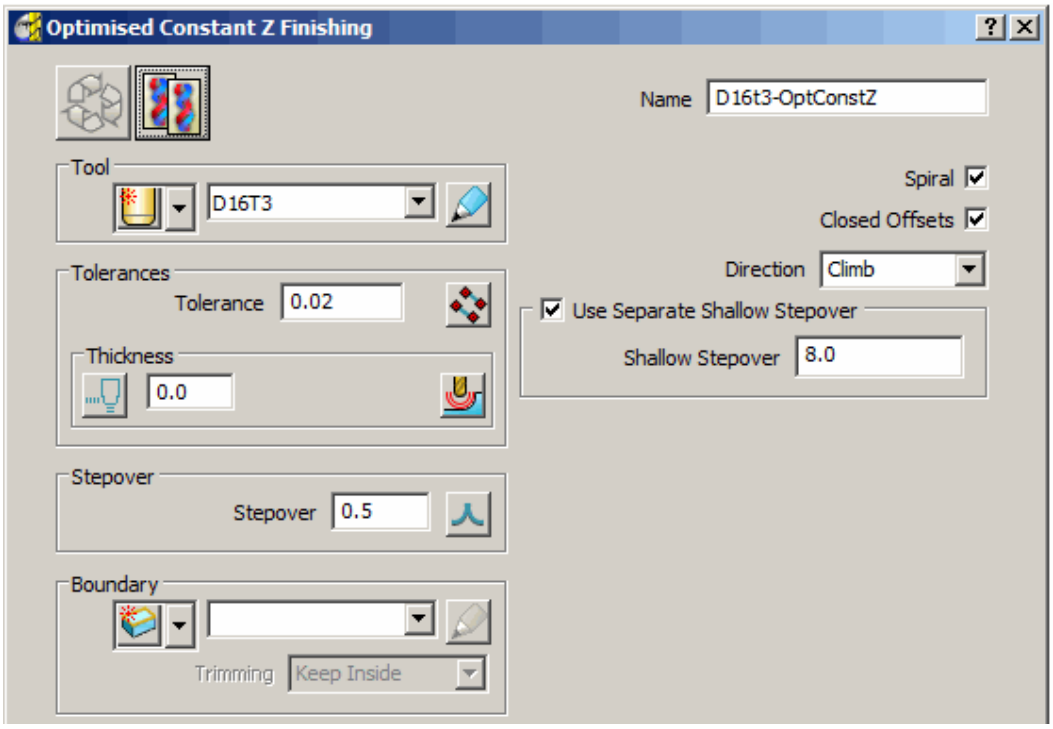
- Select **OK** to load the existing **Project** into **PowerMILL**.
- From **File – Save Project As**:- **D:\users\training\COURSEWORK\PowerMILL_Projects\ADV-CZ-example**

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

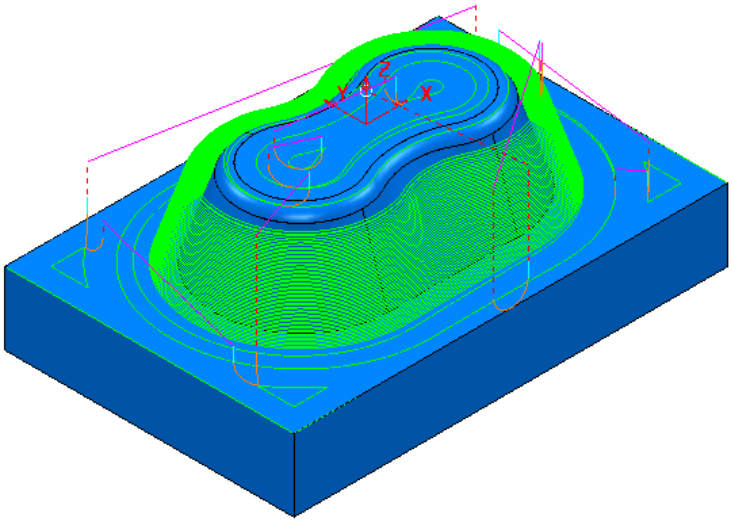


The component form includes 2 ***Flat*** *a*reas which are currently machined with the same ***small*** **stepover** as the angled walls. For more efficiency, an option is available for a larger **stepover** to be independently applied to the **shallow** (***Flat***) areas.

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

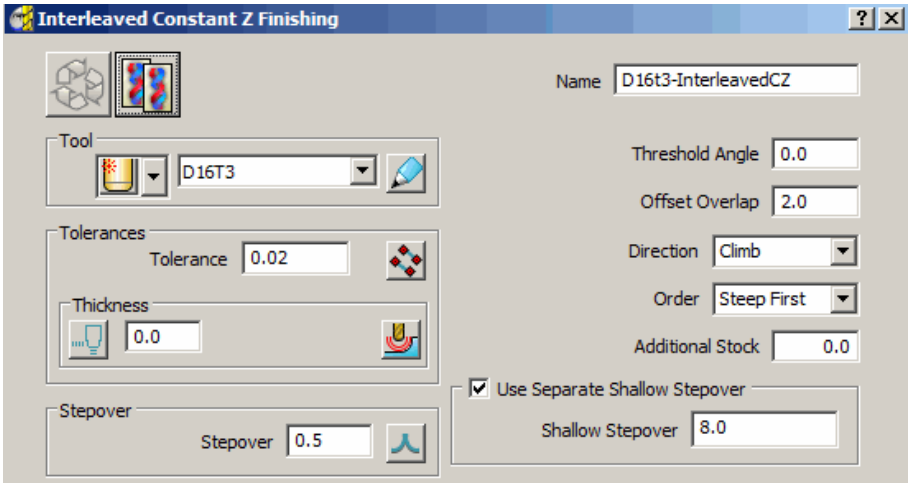


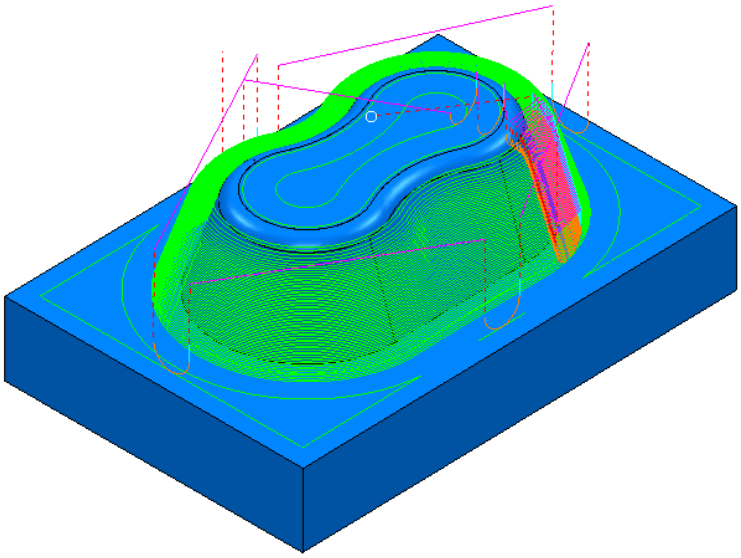
- **Tick** The **Box** named ***Use Separate Shallow Stepover*** and input a ***Shallow Stepover*** of **8.0** before selecting **Apply** and **Cancel**.



Optimised Constant Z provides an option to create **spiral** tool tracks suitable for **High Speed Machining** applications.

- **Apply** an **Interleaved Constant Z** strategy using the same cutting tool and the same basic settings as the previous ***Optimised Constant Z*** toolpath.



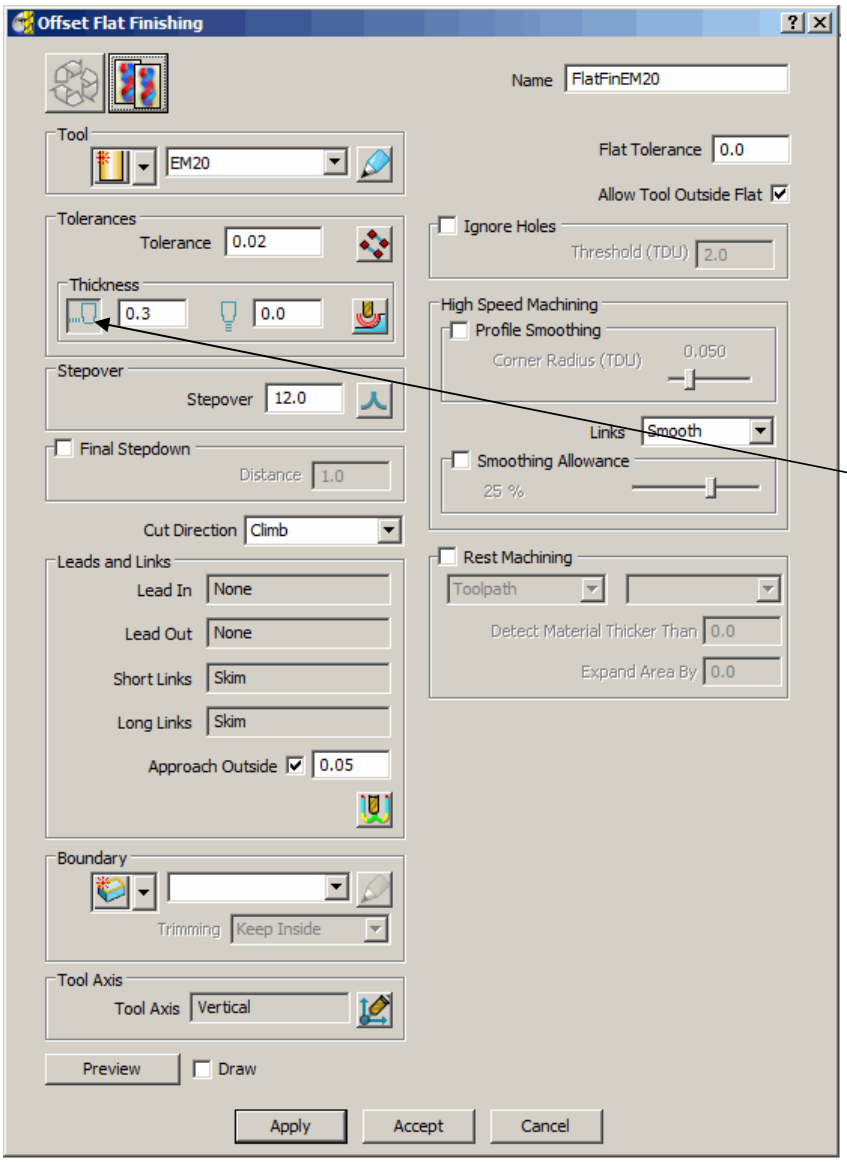


One drawback with the **Interleaved Constant Z** strategy is that there is no **Spiral** option. This would make it less desirable for **High Speed Machining** applications

Flat Machining

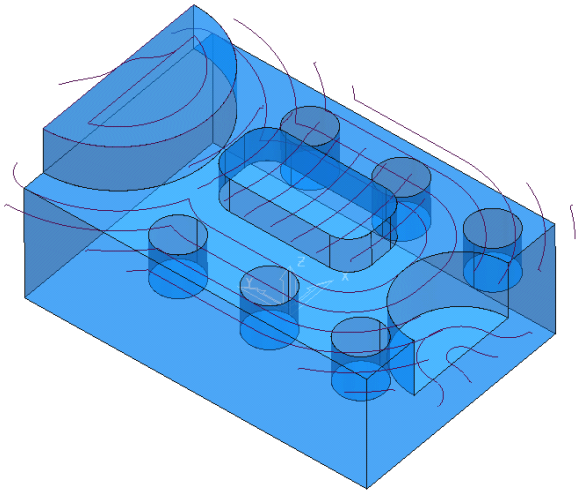
- **Delete All** and **Reset Forms**
- **Import** the **model**:- **D:\users\training\PowerMILL_Data\Models\Flats.dgk**.
- Create a **20mm** diameter **End Mill** tool named **EM20**.
- **Calculate** the **Block** and set the **Safe** heights.
- Open the **Offset Flat Finishing** form and fill in as below.

It should be noted that where a flat area is adjacent to a vertical face **Flat finishing** the tool will rub against the vertical face which is likely to result in deflection. It is recommended in these situations that a different **Radial Thickness** be applied to prevent contact with the walls.



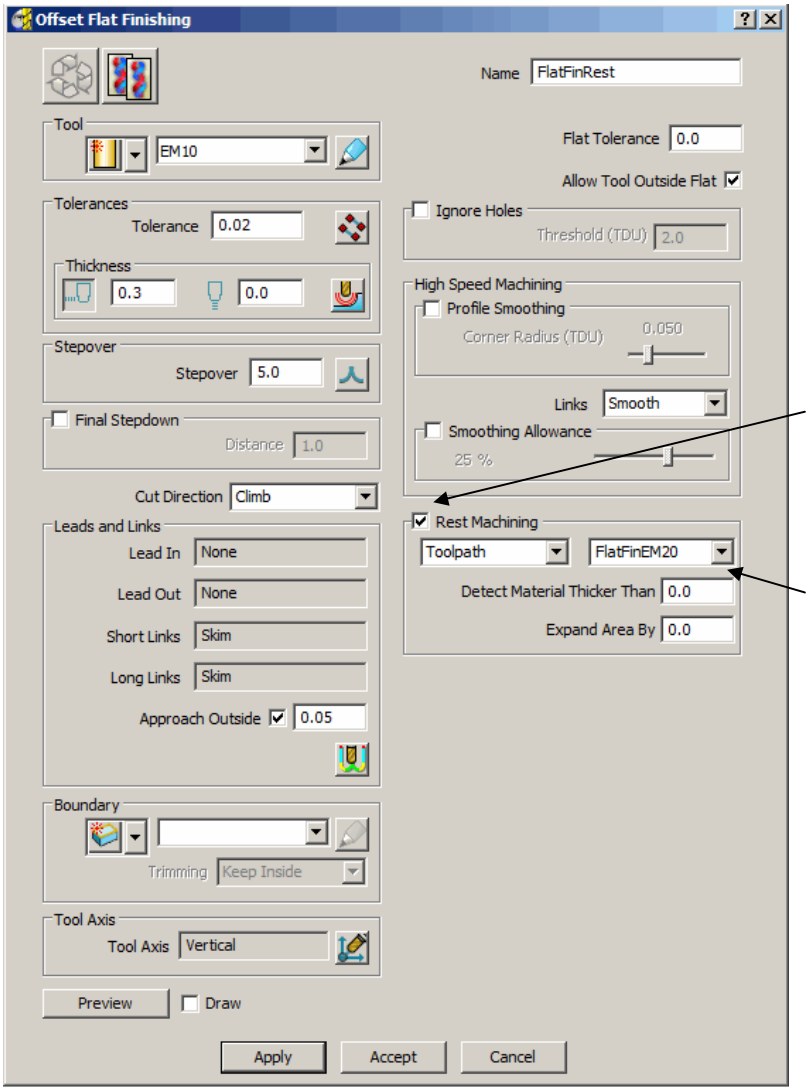
- ***Name*** the toolpath **FlatFinEM20**.
- Select the **Use Axial Thickness** icon.
- Enter separate **Thickness** values for **Radial 0.3** and **Axial 0**.

- **Apply** 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**.
- **Copy** the toolpath **Flat Fin EM20** and fill in the form as below.

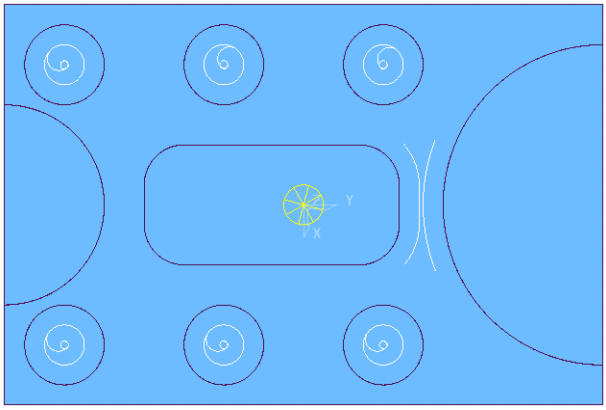


***Name* FlatFinRest**

Tick **Rest Machining**.

Reference Toolpath **FlatFinEM20**

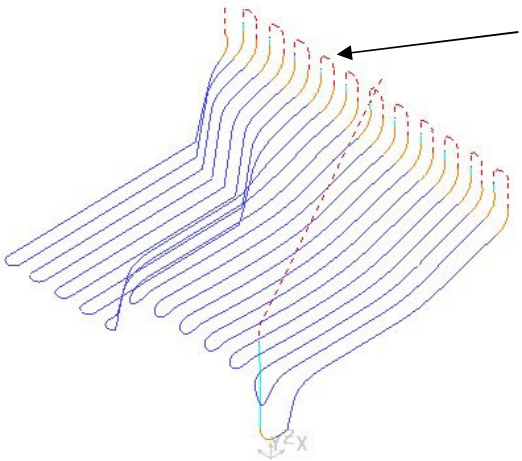
- **Apply** and **Cancel** the form.



The bottoms of the holes and the area behind the boss have now been finished as shown in the Z view below.

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.

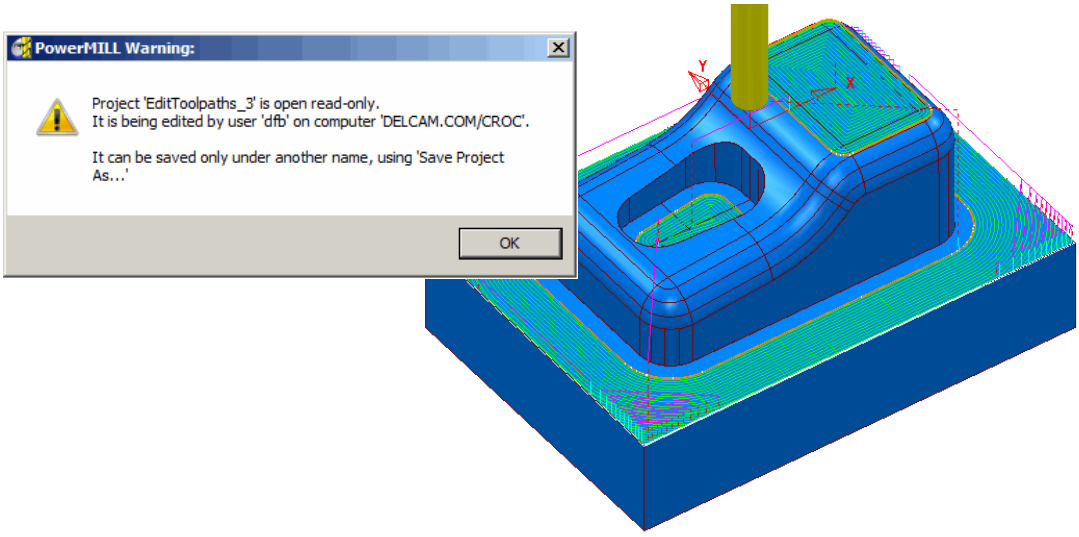
This option is ideal for applications where it is desirable to avoid sudden sharp changes in direction; for example (High Speed Machining).

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

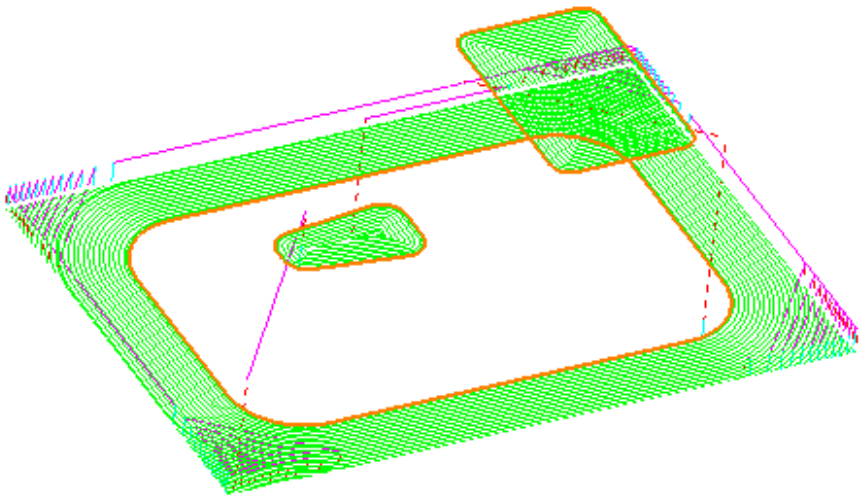
- **Delete All** and **Reset forms**.
- **Open** the read-only **Project**:-
  D:\users\training\PowerMILL_Data\Projects\EditToolpaths_3



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

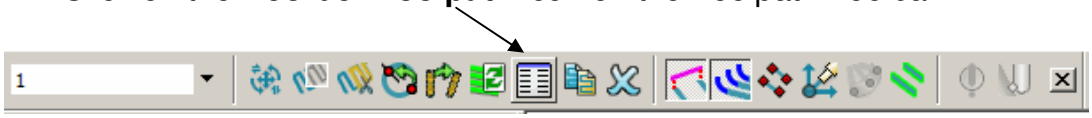
The **Project** contains a **Dia 10 Rad 3 Tipped Tool**, and a **3D Offset** finishing toolpath controlled by both **Boundary** and **Pattern** segments.

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

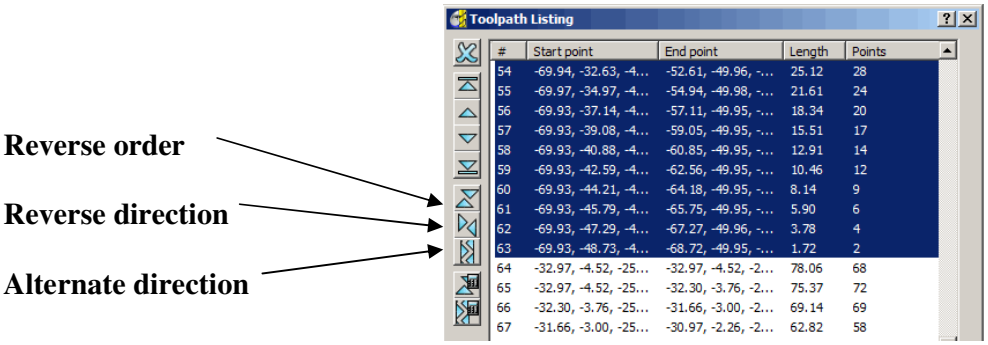


Observe the current **direction** and **ordering** of the strategy considering potential improvements. For example, the lower area machining would benefit from climb milling inwards towards the component form and the pocket machining could start central and climb mill outwards towards the sidewall.

- 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.



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.
- 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.

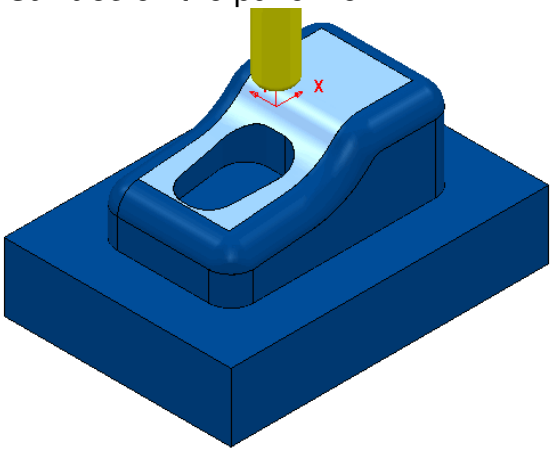
Note:- Extra care is required when using the **Spiral** option in the selected finishing strategy as these are continuous tooltracks and can only be reversed and not internally reordered (If a spiral track is created to **Upcut** outwards then it can only be modified to **Upcut** inwards).

- Do not close the **Project** down yet as it will now be used to demonstrate both the **Parametric Offset Finishing** and **Surface Finishing** strategies.

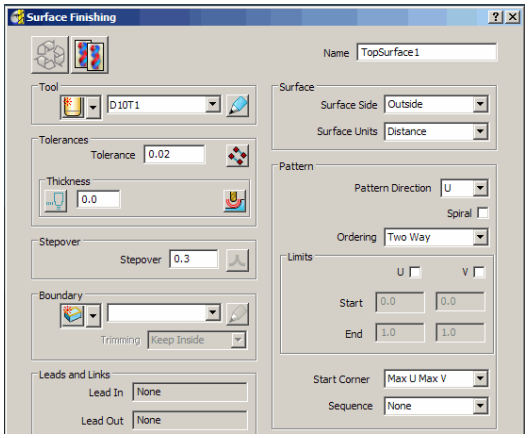
Surface Finishing

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***.

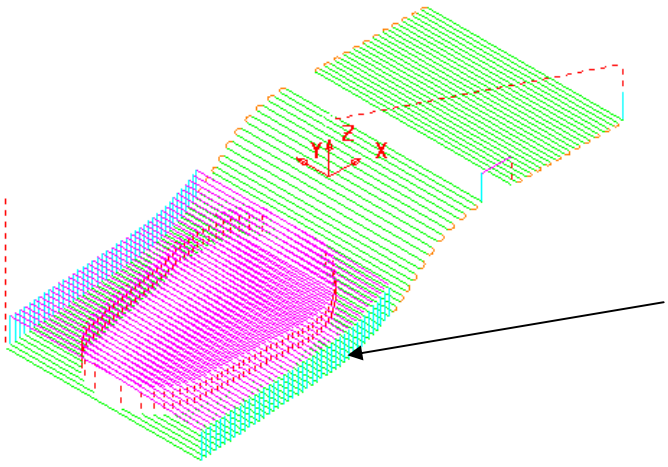
- Create and **Activate** a **Dia 10**, **Rad 1**, **Tiprad tool** named **D10t1**.
- Select the ***upper surface*** on the punch form.



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

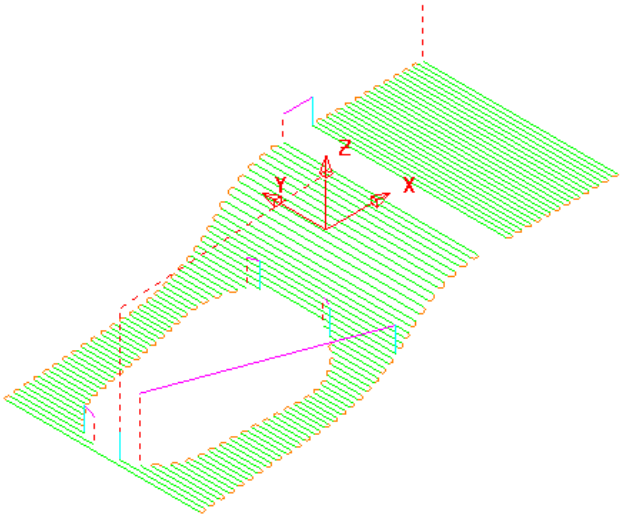


- **Apply** and **Close** the form.



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.

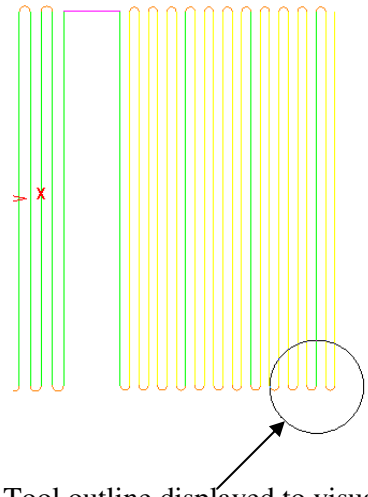
- From the local **toolpath menu** (Right mouse click), select **Edit – Reorder** and in the form, click the **Automatic reorder and reverse** icon. 

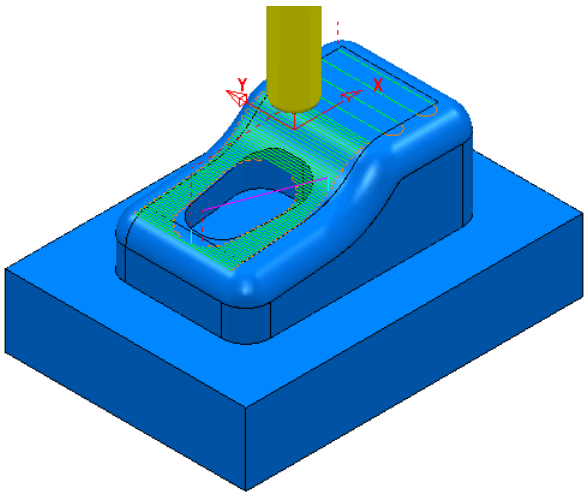


The **Toolpath** follows the curves of the selected **Surface**.
The toolpath illustrated has been created with **1mm Stepover** to help with visualisation.
It has also been edited using the **Automatic Reorder and Reverse** option to eliminate as many air moves as possible.

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).





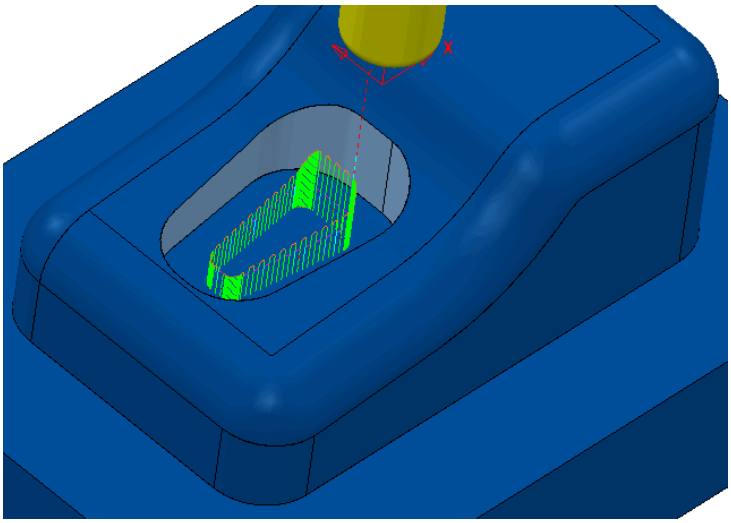
Tool outline displayed to visualise suitable **Stepover** (**Draw - Cursor - Tool**).

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

Surface Finishing Exercise

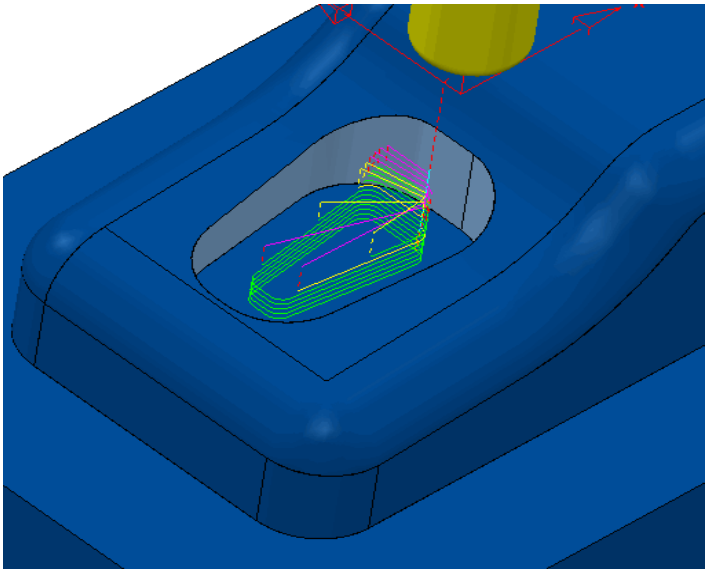
The direction of the curves on the ***selected*** **surface** will dictate whether the tool tracks are aligned **along** or **across** the model form.

- **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 **Apply** the form.



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

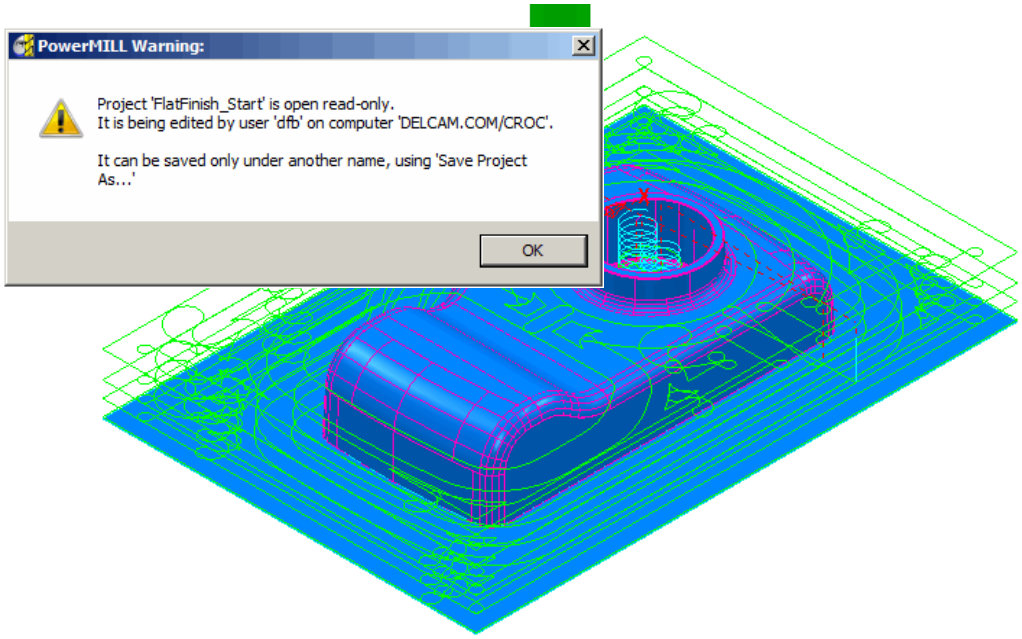
Note that the **tool tracks** do not run parallel to the ***trimmed*** top edge of the sidewall **surface**.

Note:- A **spiral** option in the form of a ***tick box*** is available in the **Surface Finishing** form to apply a continuous **tool track** down closed **pocket** or **upstand** areas.

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.

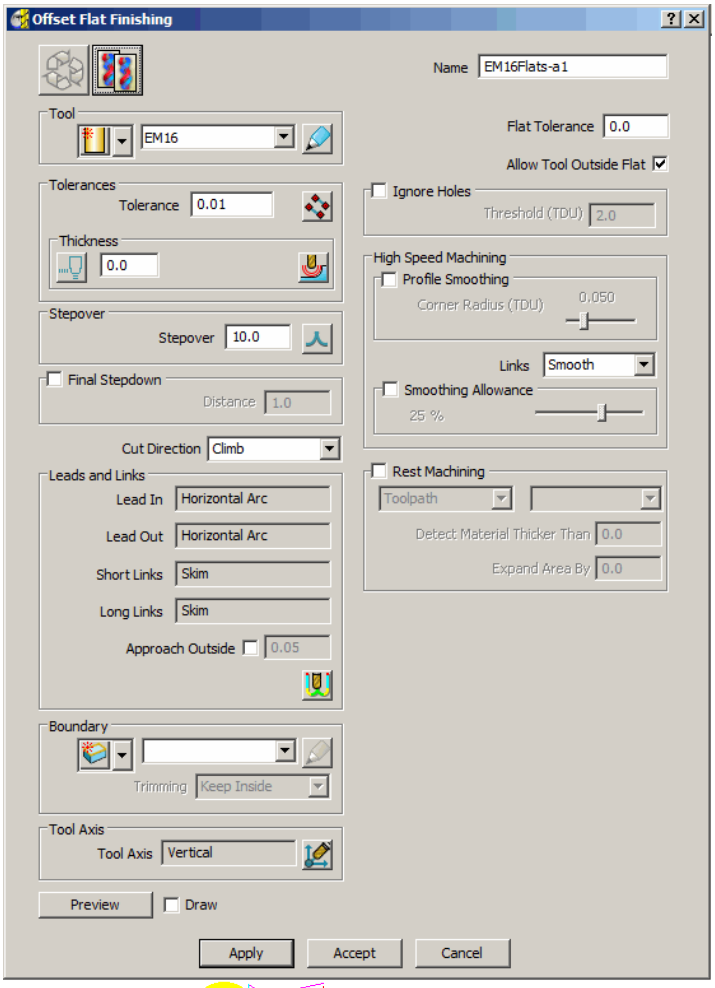
- **Delete All** and **Reset forms**.
- From **File - Open Project** select the read-only **Project**:- **D:\users\training\PowerMILL_Data\Projects\FlatFinish_Start**

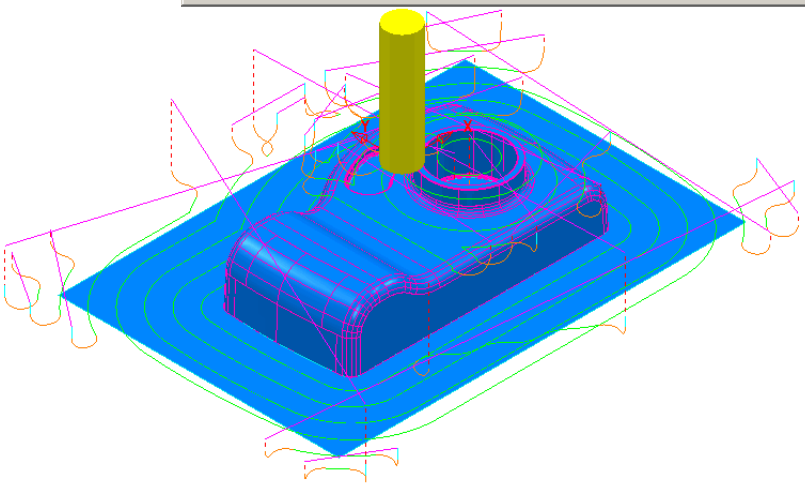


- Select **OK** in the ***PowerMILL Warning*** form.
- **Save Project As**:- **D:\users\training\COURSEWORK\PowerMILL-Projects\3AxisSwarfExample**

Offset Flat Finishing

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

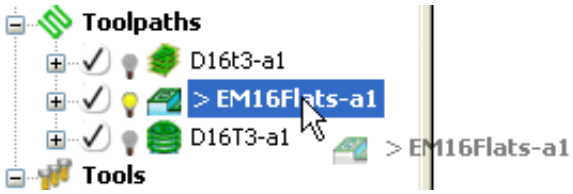




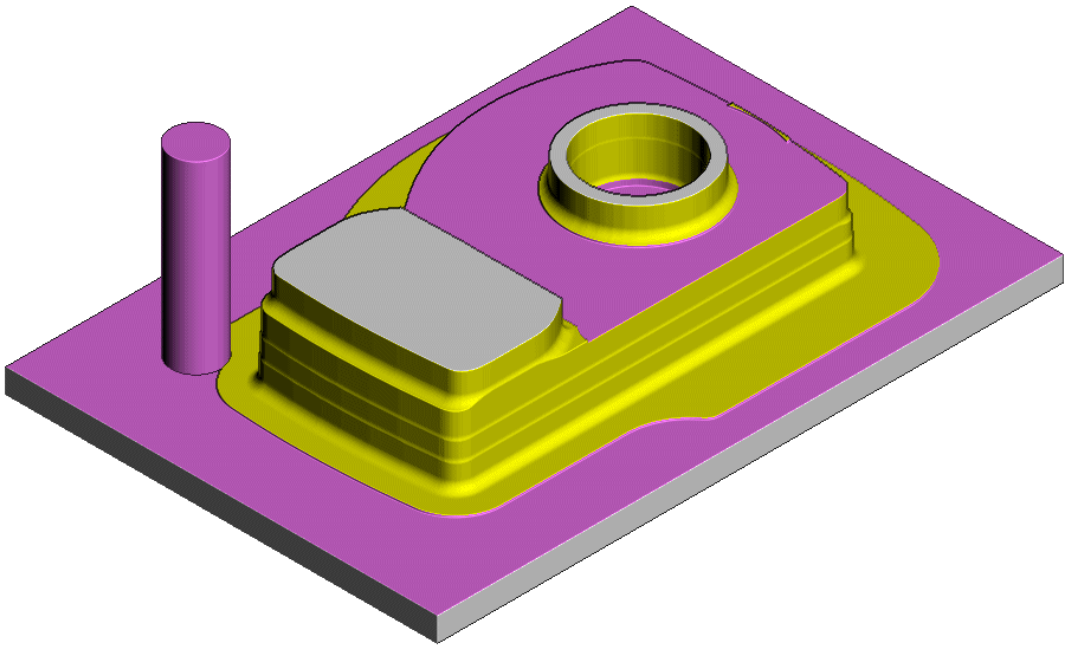
The **Offset Flats** finishing strategy will appear as shown above. In this case suitable **Leads** have been applied retrospectively:- **Lead In/Out - (1st choice) Horizontal Arc - (2nd Choice) Pocket Centre** and **Extensions – In/Out - Vertical Arc**

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.



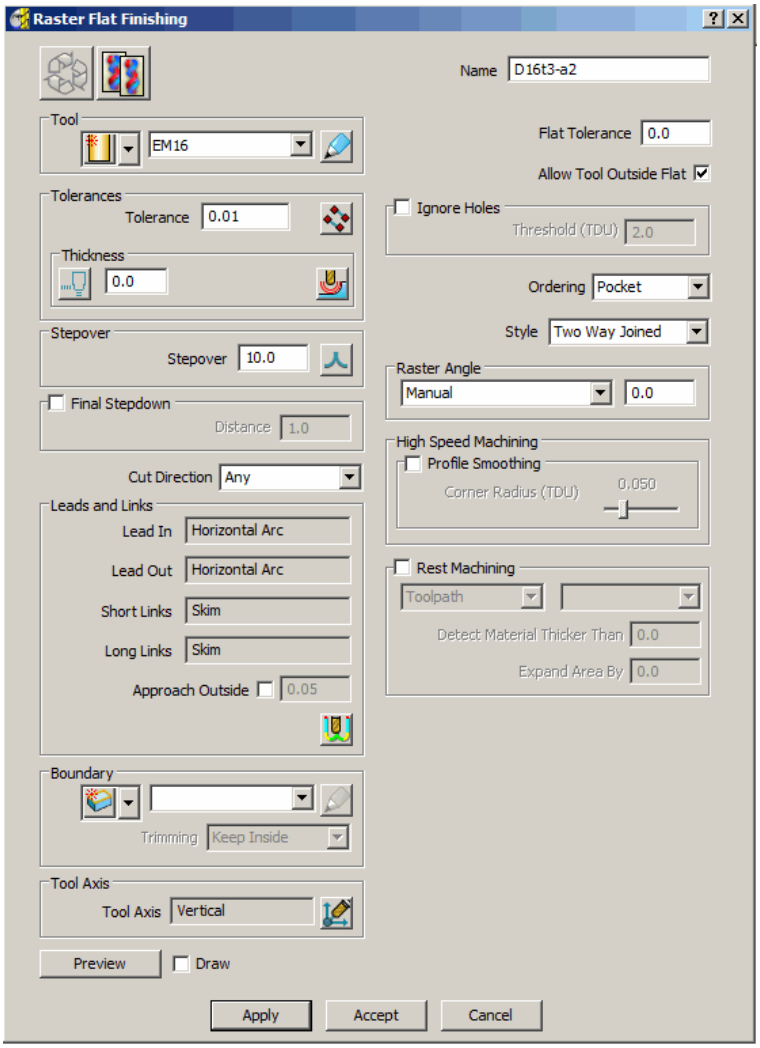
- Perform a **ViewMILL simulation** on all 3 toolpaths to check that method and result of material removal is acceptable.

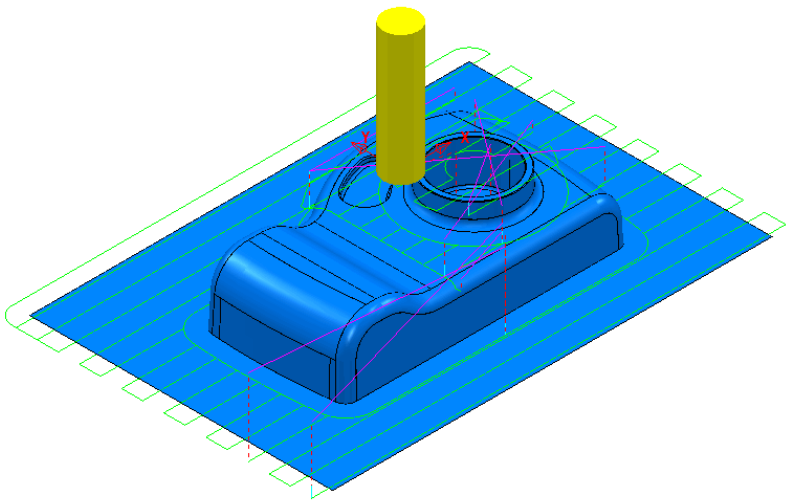


The **Offset Flats** strategy is shown above nearing the end of it's **ViewMILL** simulation.

Raster Flat Finishing

- From the **Toolpath strategies** form, select **Raster Flat Finishing**.
- Fill in the form exactly as shown below and select **Apply**.



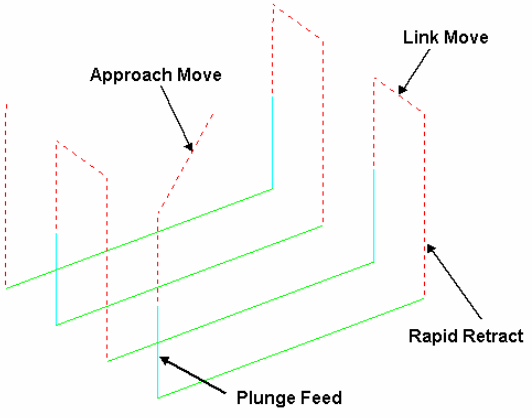


Raster Flat strategy

5. Toolpath Leads and Links

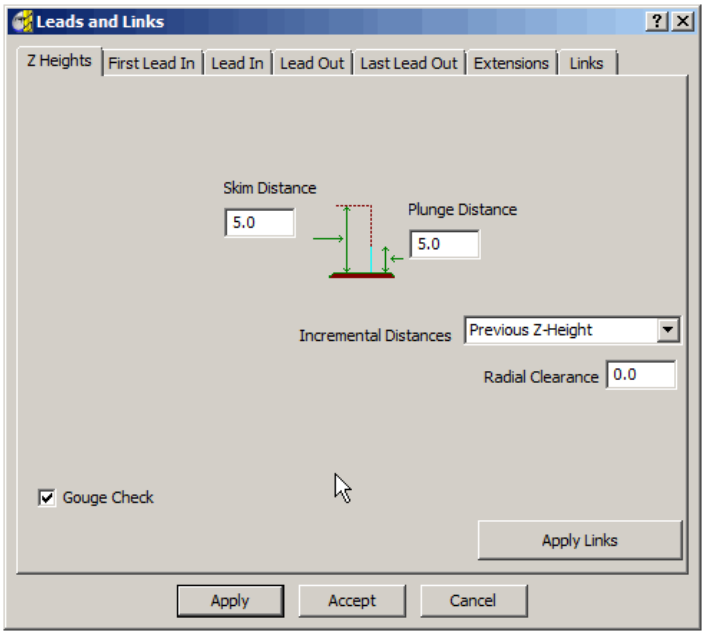
Introduction

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 a 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 rapids 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**.

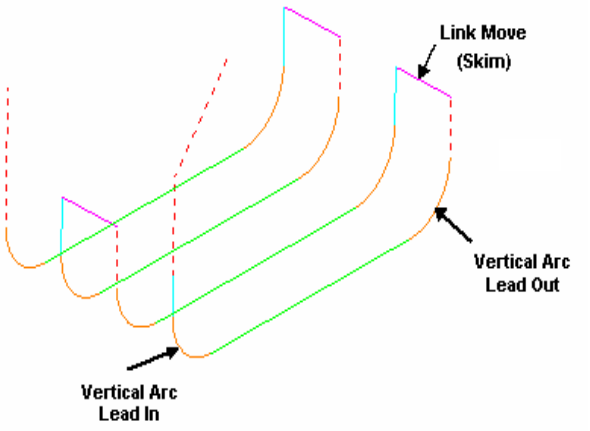


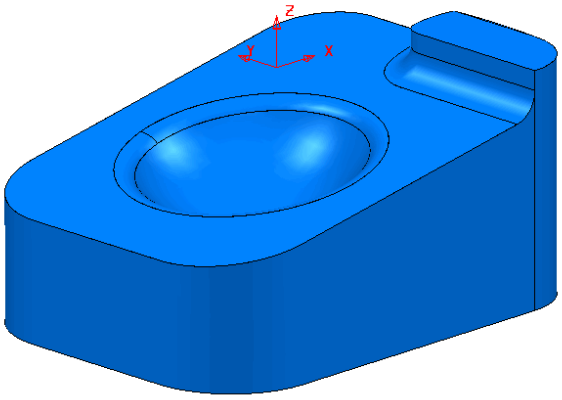
Illustration shows:-
Lead In/Out - **Vertical Arc** and
Incremental - **Skim** - ***Link*** moves.

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

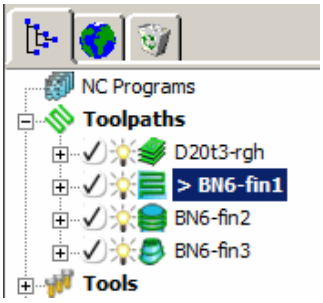
- **Delete All** and **Reset forms**.
- From **File** select **Open Project** and **select** the readonly ***Project:-*** **D:\users\train\PowerMILL_Data\Projects\LeadsLinks-Start**



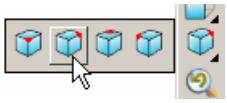
A message informing the user that the **Project** is **ReadOnly** will appear.

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**:- **D:\users\training\COURSEWORK\PowerMILL-Projects\LeadsLinks-example**



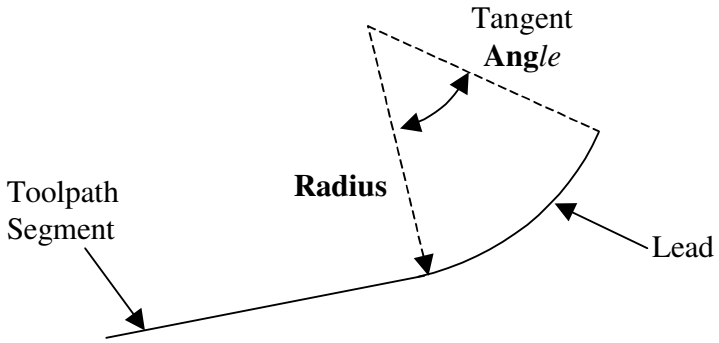
The **Project** contains 1 **Roughing toolpath** and 3 **Finishing toolpaths**. The following example will illustrate the retrospective addition of suitable **Leads and Links** to the 3 finishing toolpaths.

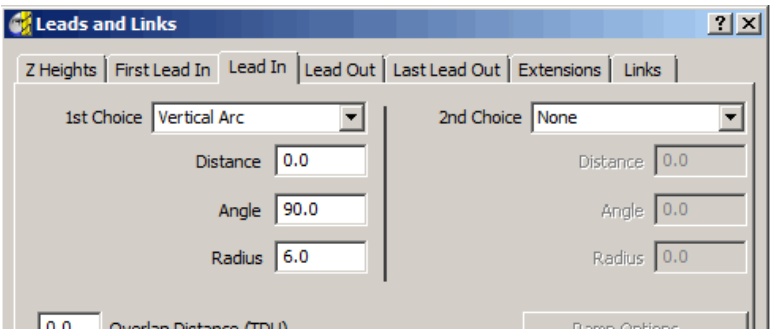


- Select an **ISO2** view.
- Right click on the ***Toolpath*** **BN6-fin1** and select **Activate**.
- Open the **Leads and Links** form. 

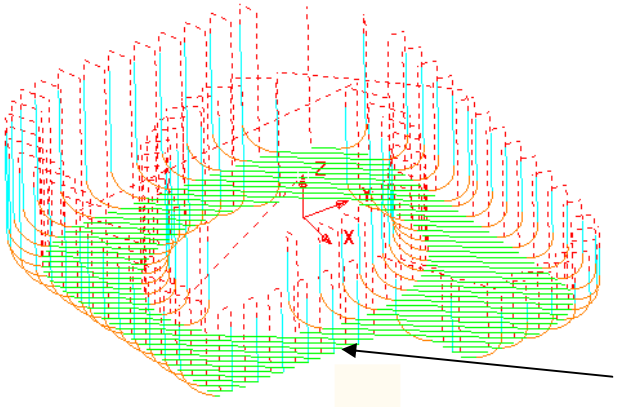
Vertical Arc Leads

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).



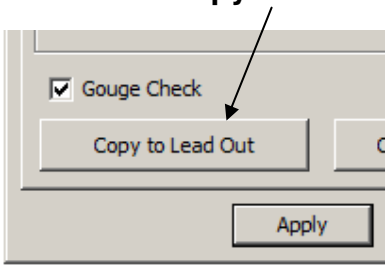


- Select the **Lead In** tab and for ***1st Choice*** input an ***Angle*** **90.0** and ***Radius*** **6.0** before clicking **Apply** at the bottom of the form.

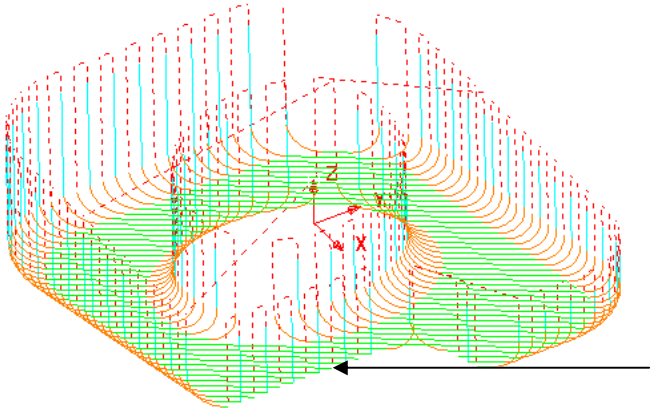


There is now a **Vertical Arc** leading onto most of the tool tracks. If the required **Lead Out** is to be exactly the same as the **Lead In** and vice versa the **Copy to Lead Out** option allows easy copying of the settings between the two.
The **Leads** in this area have remained unchanged as any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

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

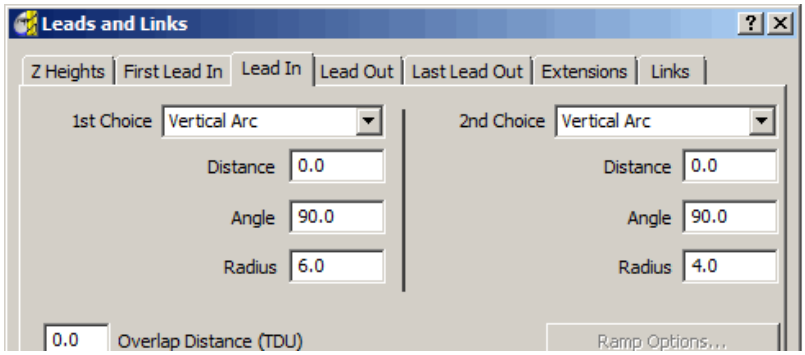


- **Apply** the form to update the **Active** toolpath.

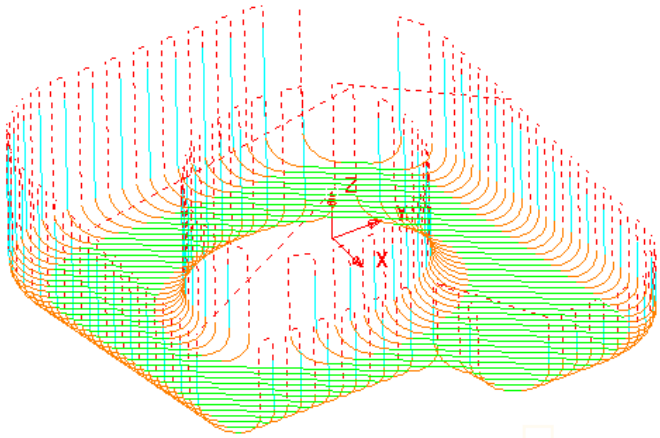


Where the operation is gouge free a **Lead Out** move, similar to the **Lead In** is applied at the end of each tool track.

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.



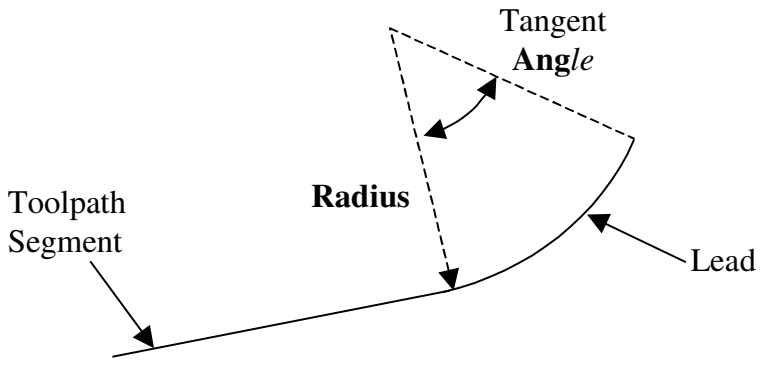
Wherever it is ***gouge free*** the ***1st Choice*** of a **Radius 6**, **Vertical Arc**, **Lead In/Out** is applied.

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

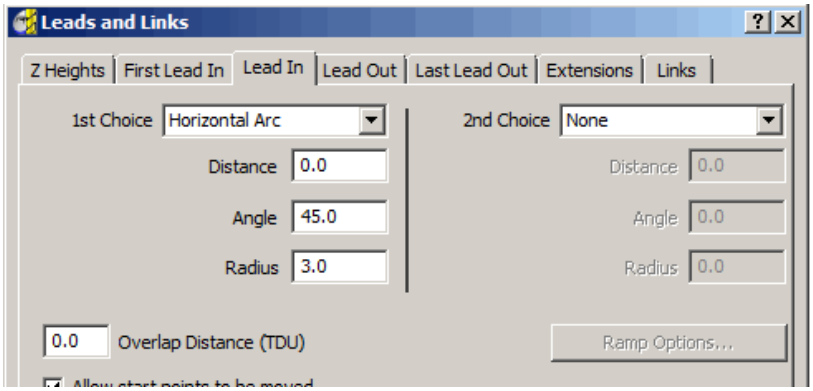
If neither the 1st or 2nd **Choice** is possible without a **gouge** occurring, then no **Lead In/Out** will be applied to that part of the toolpath.

Horizontal Arc Leads

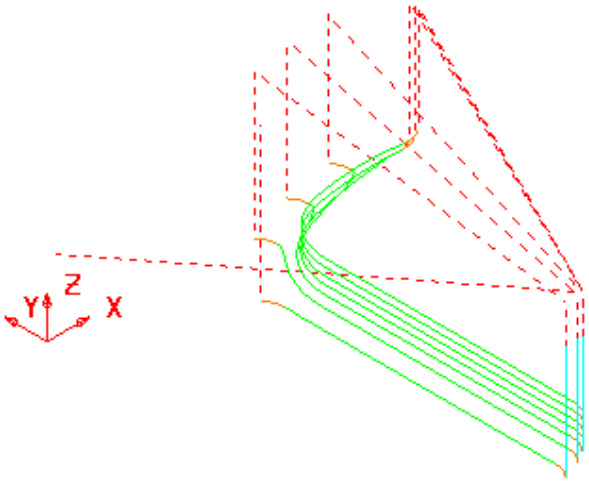
This produces circular leads on the horizontal plane, and the **Radius** and **Angle** value is similar to that for Vertical Arc. This type of lead is frequently used for toolpaths running at constant Z, or with only small changes in Z height.



- 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.

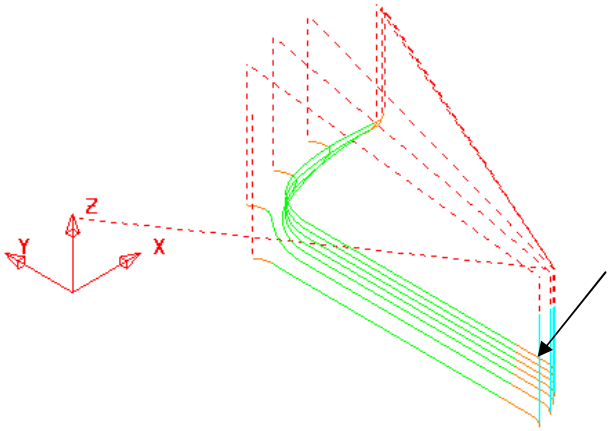


- Change the ***1st Choice*** **Lead In** to be a **Horizontal Arc** with ***Angle 45*** and ***Radius*** **3.0** then select the **Copy to Lead Out** button before selecting **Apply**.



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.

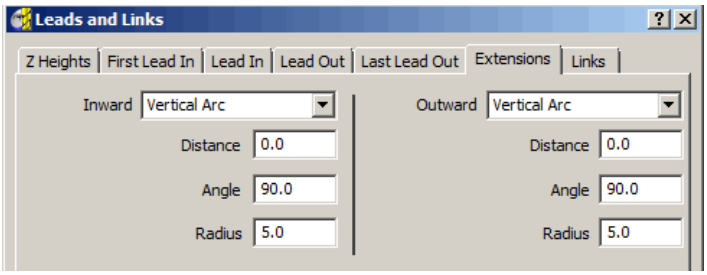


An additional (**Distance**) extension move is added to the ***tool track*** before the original **Lead In** option 'kicks in'.

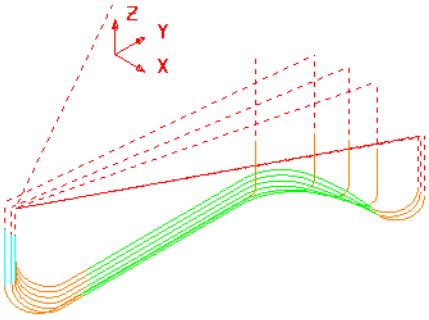
Extensions

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.

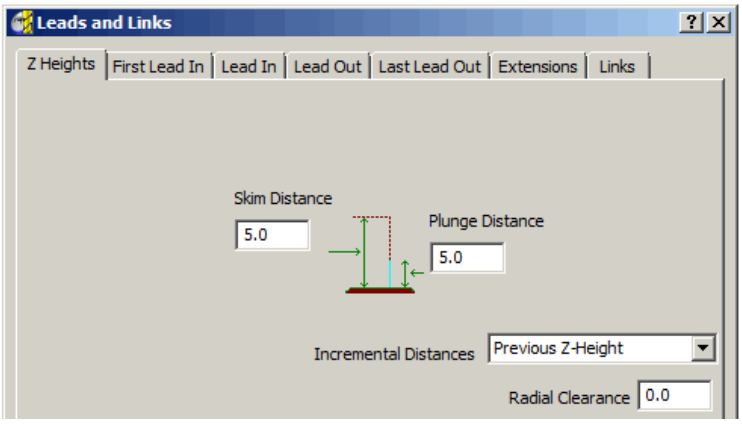
For **Short** links the available options include **Safe Z**, **Incremental**, **Skim**, **On Surface**, **Stepdown**, **Straight**, and **Circular Arc**.

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 the **Z Heights** tab to access the **Skim** and **Plunge Distance** boxes.

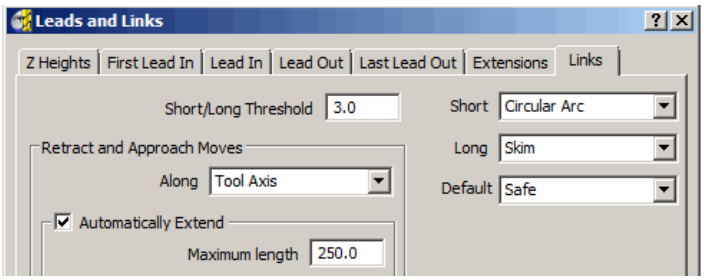


The values will be left as default for the time being.

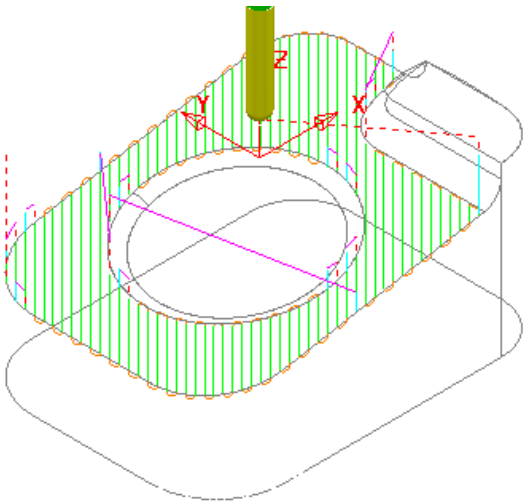
The **Skim Distance** is the ***Incremental height*** at which the **tool** clears the **model** form between tool tracks.

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.

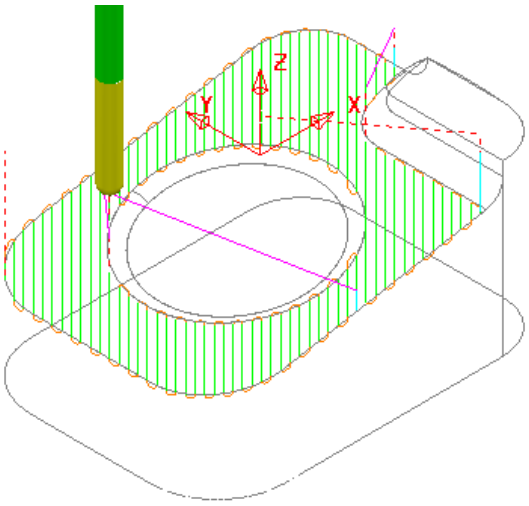


- Input the **Short/Long Threshold** as **3** (**Link** move distances ***less*** than this value will use the **Short** ***link*** option and those ***greater*** will use the **Long** ***link*** option).
- Select ***Short*** as **Circular** ad ***Long*** as **Skim** and leave ***Default*** as **Safe**.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

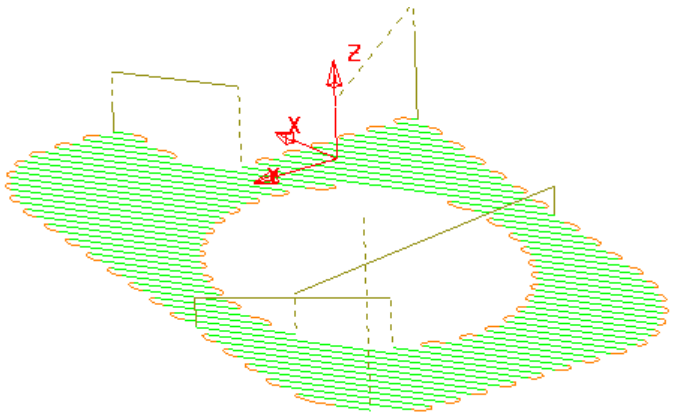


Note that by increasing the value of the **Short/Long Threshold** the number of ***Short Link*** **Cicular Arcs** has greatly increased.

The remaining **lift and plunge** moves would benefit from **Vertical Arc** ***Lead In/Out*** moves but not the existing **Circular Arc** ***Links***. This can be achieved locally by selecting the individual lift or plunge moves and then applying the **Leads and Links** form.

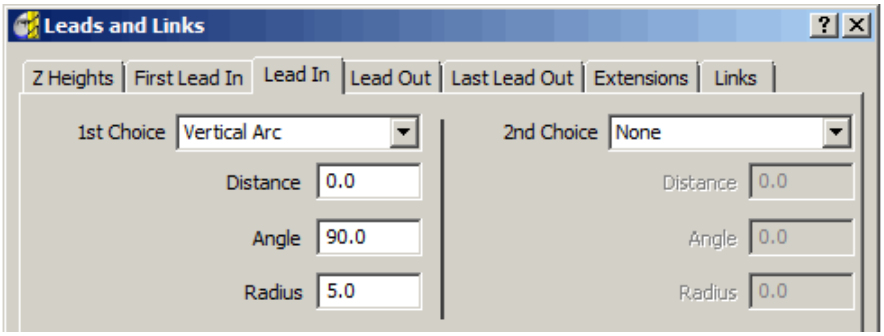
Applying Leads and Links locally to individual tool tracks

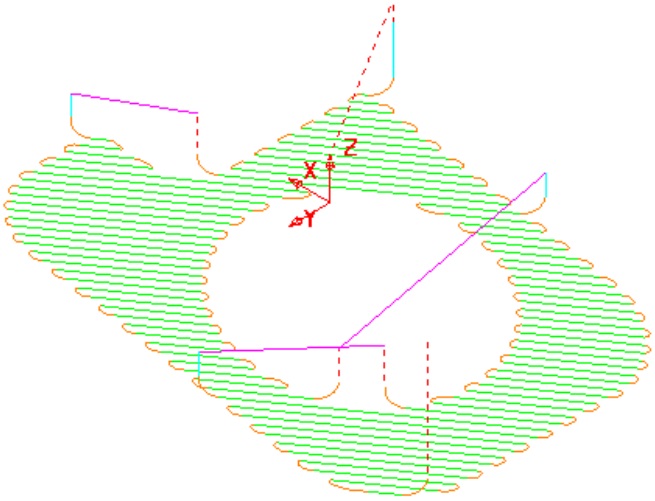
- With the **Shift key** depressed, select the individual ***vertical moves*** where the **Long** (**Skim**) and **Default** ***Links*** (**Safe**) have been applied.



The selected link moves to be modified are coloured **dark yellow** in the illustration to assist with visualisation. In reality, any selected item in the graphics area will be coloured **bright yellow**.

- In the **Leads and Links** form select the **Lead In** tab and select ***1st Choice*** as **Vertical Arc** with ***Angle*** **90** and ***Radius*** **5**.
- Select the **Copy to Lead Out** tab and then **Apply** the form.





The existing **Circular Arc - Links** remain unchanged while all of the selected **vertical link** moves are applied with a **Vertical Arc - Lead**.

- 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

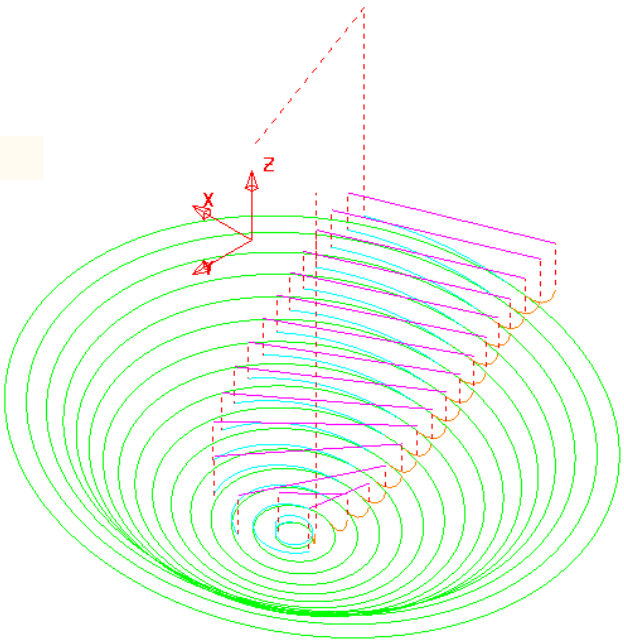
- **Activate** the ***toolpath*** **BN6-fin3** and **Apply** the **Leads and Links** as follows:-

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***).

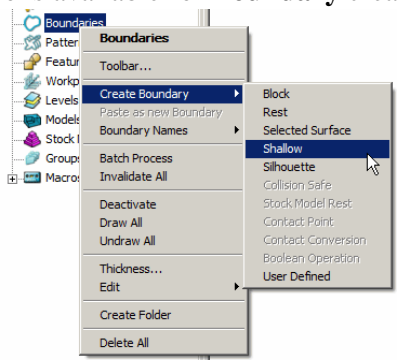


The finished **Leads and Links** will be as shown above.

6. Boundaries

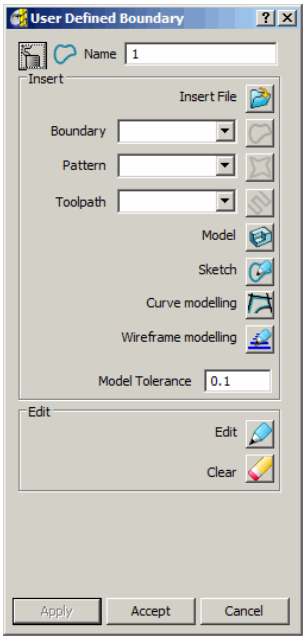
Introduction.

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

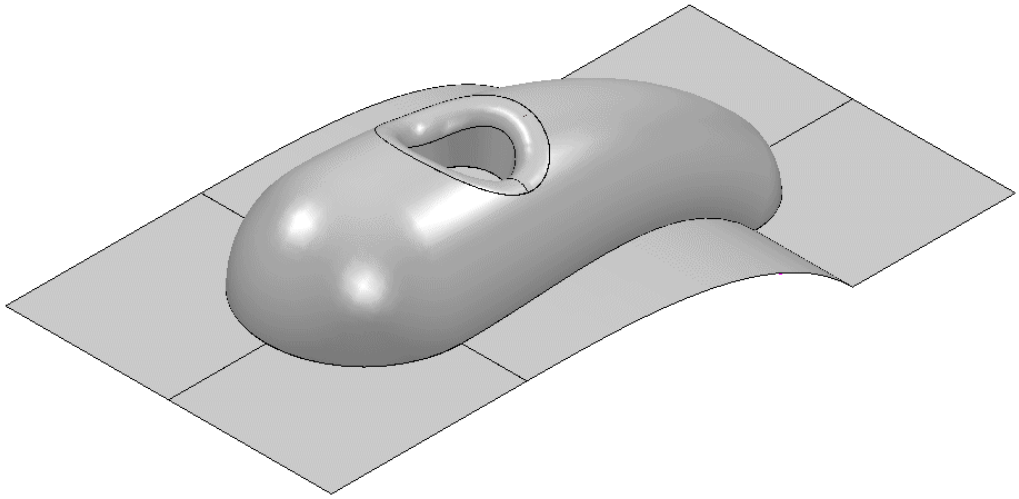
This type of **Boundary** is created from several options via an additional sub-menu. While all other main **Boundary** options involve interaction with other PowerMILL entities the **User Defined** options generally involve direct conversion of existing wireframes.



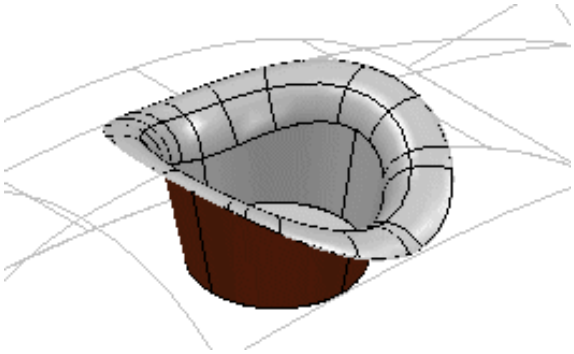
Boundary, Pattern, and Toolpath icons are greyed out but become active on the selection of an entity.

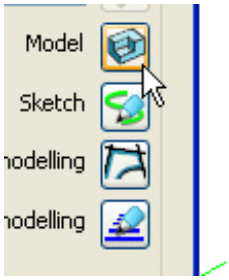
Inserts the edge of the selected model.
Enables free form or coordinate segment input.
Opens Composite Curve generator.
Opens Wireframe Modelling.

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

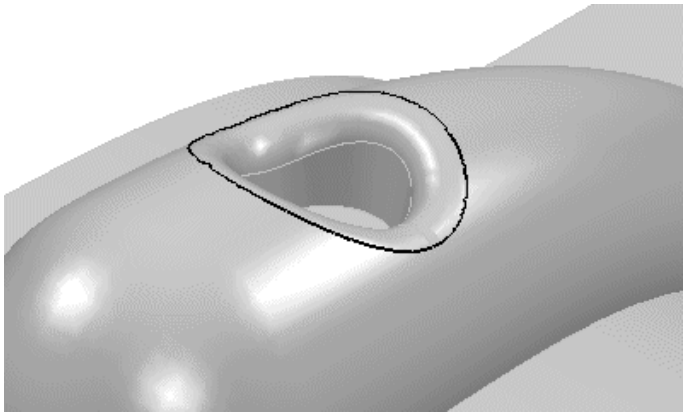


- Select the **surfaces** defining the central pocket and fillet.





- 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).



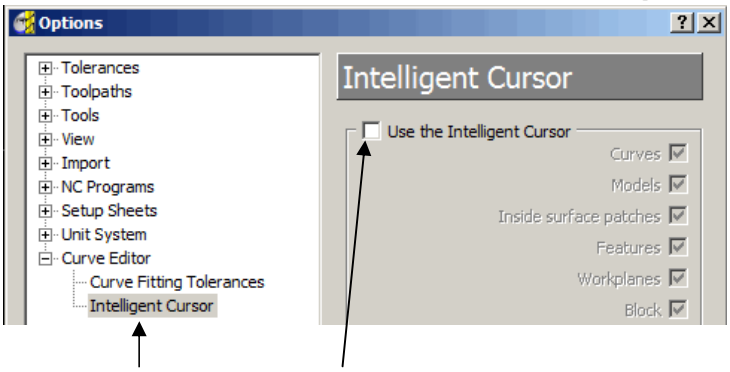
A **Boundary** segment is created around the edge of the selected part of the model.

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

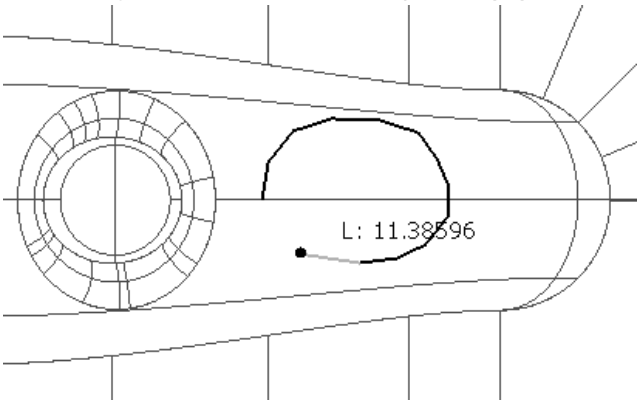
- In the **explorer** right click over **Boundaries** - **Create Boundary** - **User Defined** and in the **User Defined Boundary** form left click the **Sketch** icon. 

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

- In the **Curve Editor** toolbar select the **Curve editor options** icon. 



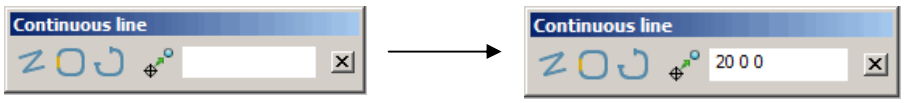
- Select **Intelligent Cursor** and **untick** the **Use of Intelligent Cursor** box to enable full 'free form' sketching to occur.
- **Accept** the form.
- In the **Curve Editor** toolbar, select the create **Continuous Lines** option. 
- Using the ***left mouse key***, snap points to create a sketched **Boundary**.



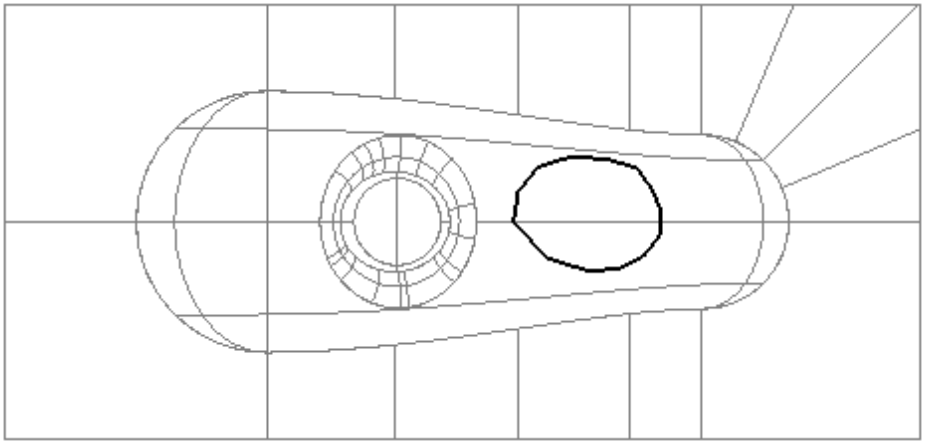
Note: Do not attempt to close the segment by trying to snap the 'final span' back on to the start point. Use the Close segment icon to achieve an exact match. 

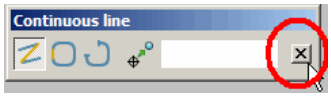
- For any invalid points, sequentially remove using the **Undo** icon  and to insert the final span select the **Close segment** icon. 

Points can also be input as exact ***Incremental*** values, using ***X Y Z*** coordinates in the data input box, located in the **Continuous Line** toolbar.



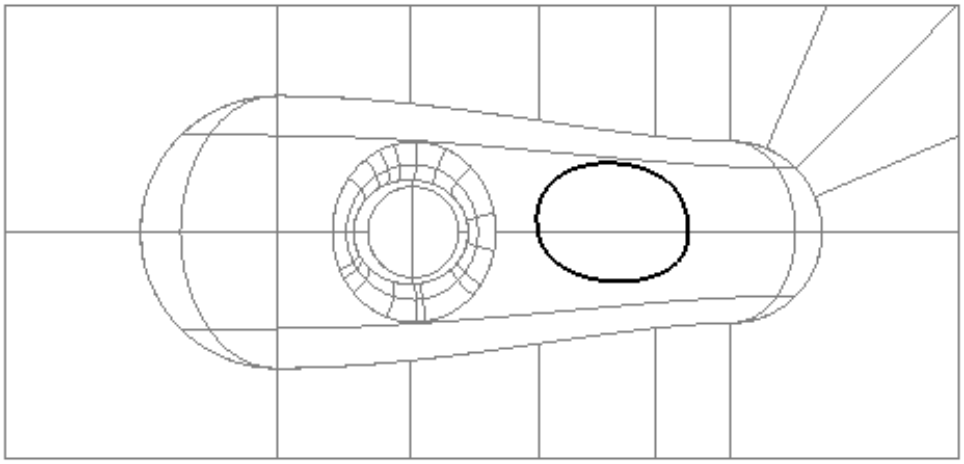
To insert as ***absolute*** coordinate values toggle the icon  to  .



- Once the segment is closed, exit the **Continuous line** 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** to open the following form.

- Enter a value of **0.5** before selecting the **green tick** to apply and close.

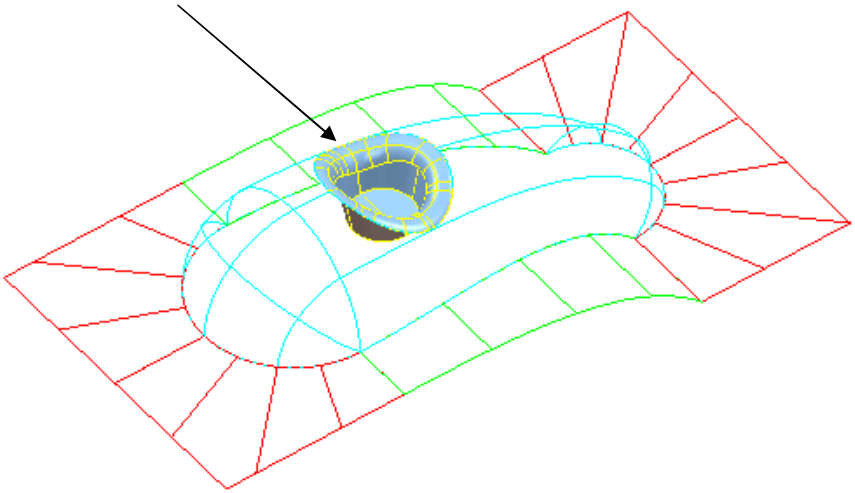


The **Boundary** segment is ***splined*** through the ***polyline points*** where doing so maintains the original form to within a ***maximum deviation*** of **0.5**. In instances where this is not possible a tangency will remain either side of the point.

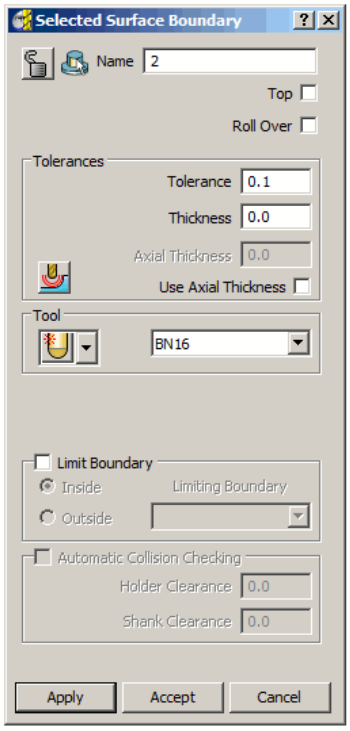
Selected Surface Boundary

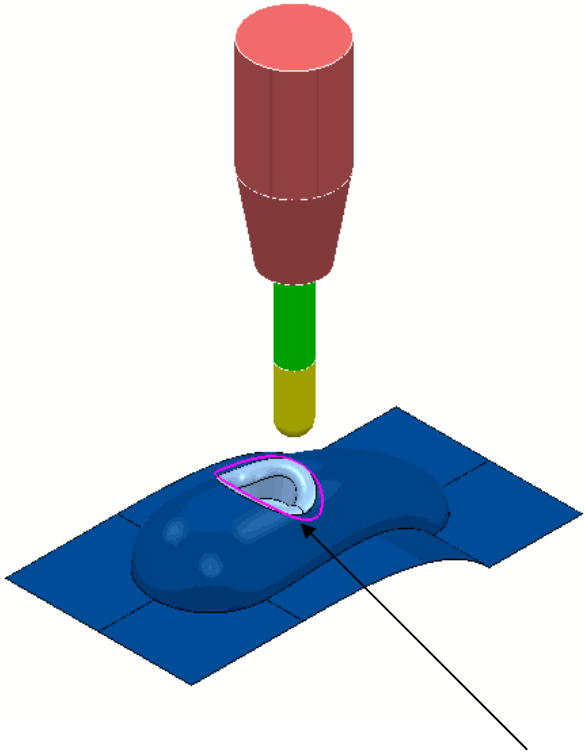
A **Selected Surface Boundary** defines one or more segments where the active tool looses contact with the **selected surfaces**. The segments represent the tip of the active tool.

- Create a **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 **Selected Surface**.



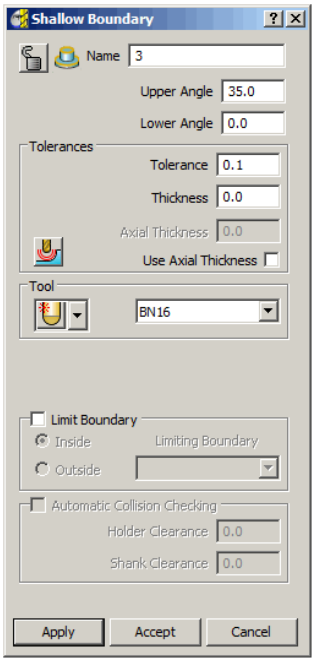


- Using the above settings **Apply** the form to create the above **Boundary** segment.

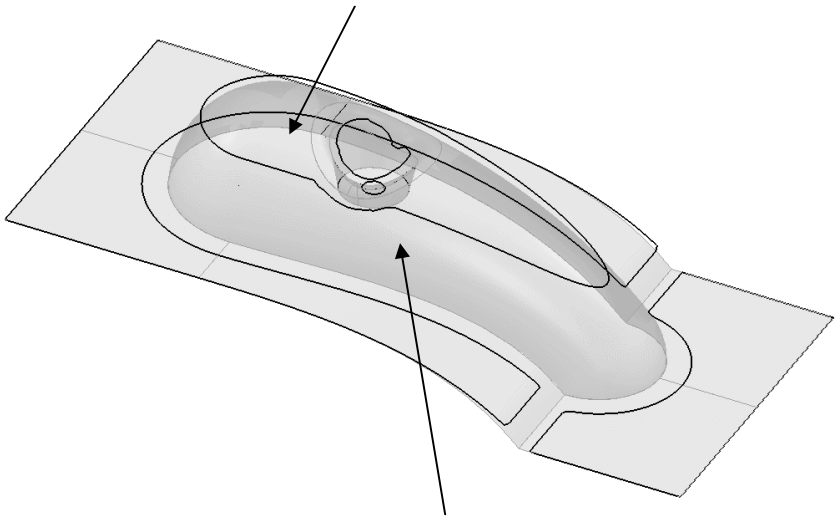
Shallow Boundary

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).



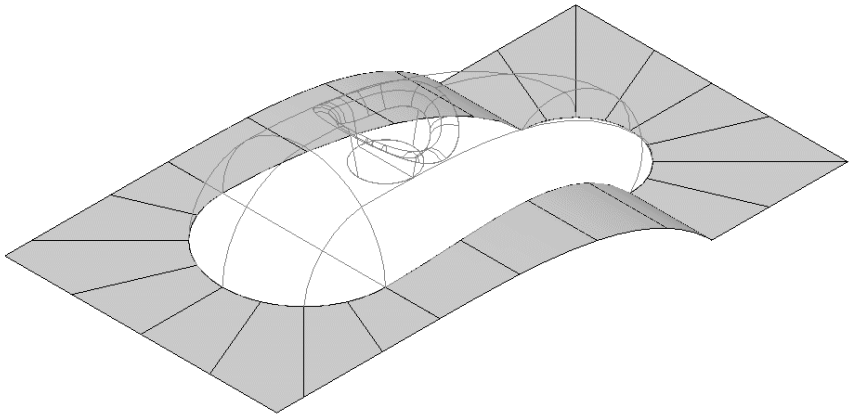
Typical area more suited to a **Constant Z** strategy (below 35 degrees).

- Using the above settings **Apply** the form to create the above **Boundary** segments.

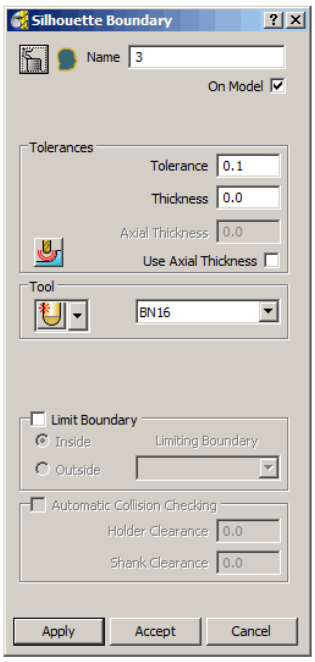
Silhouette Boundary

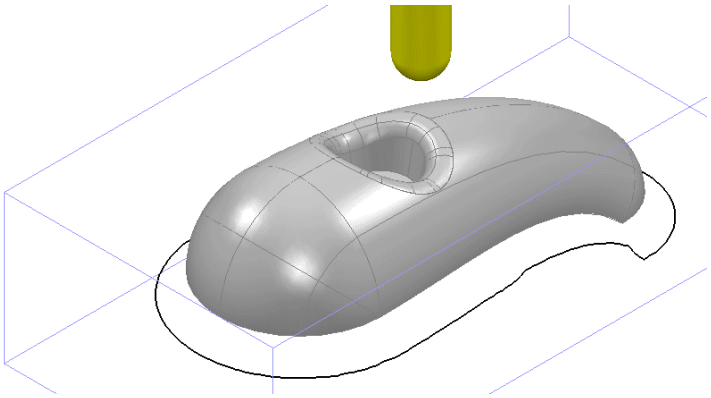
This type of **Boundary** defines the 2D the outline around the selected model adjusted to the contact point of the tool along Z.

- 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 **Silhouette**.

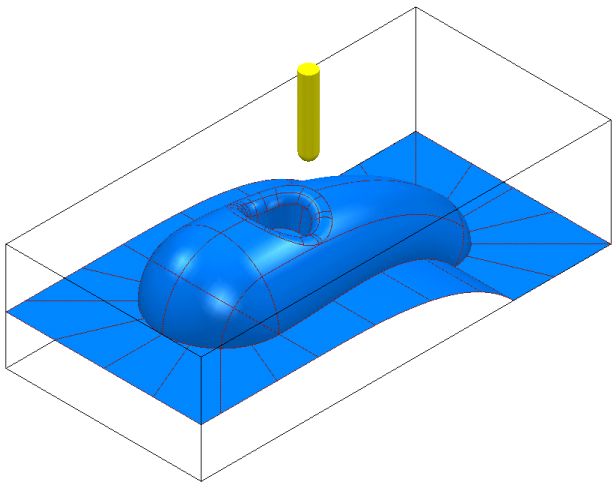




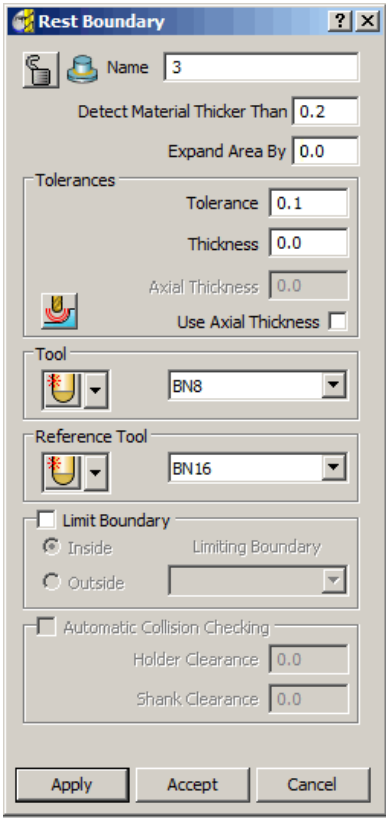
Rest Boundary

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.

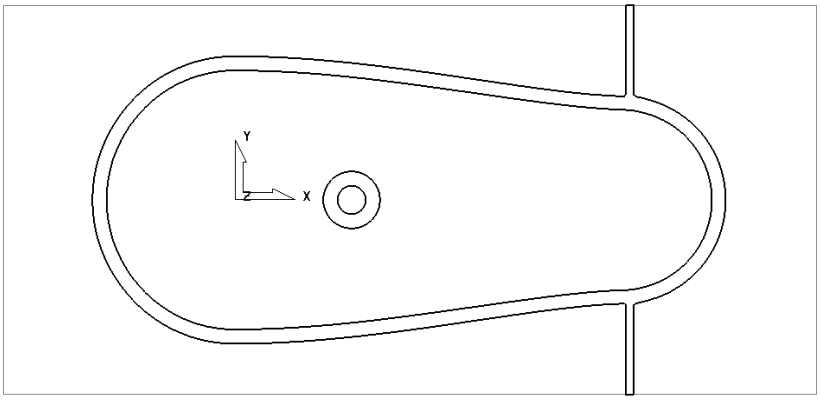
- **Delete** all existing **Boundaries** (In the **explorer**, ***Right click*** on **Boundaries** and from the local menu select **Delete All**).
- Create a **Ball Nosed** tool of ***Diameter*** **8** with the ***Name*** **bn8**.
- In the **explorer** right mouse click over **Models** and select **Delete All** to remove the now (incomplete) component.
- **Import** the original (complete) **Model** back into the **Project** (**D:\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.

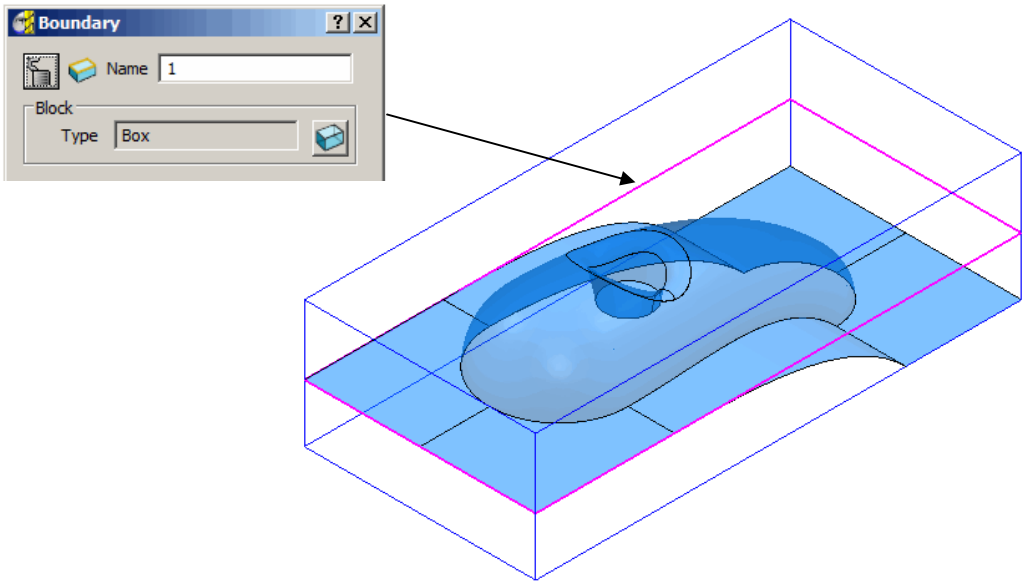


The above **Boundary** identifies areas on the component that are inaccessible to the **bn16** tool geometry to be locally machined with the **bn8** tool. On closer inspection the central pocket area would be more effectively machined with a flat bottomed tool and a smoother more continuous toolpath 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.

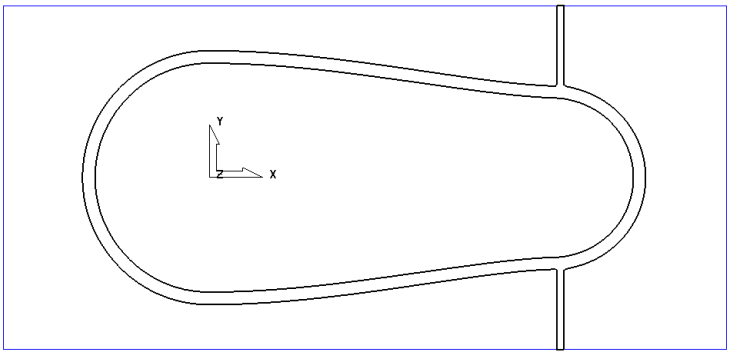
- In **Rapid Move Heights** - **Reset to safe heights**.
- Use the default **Start and End Point** settings.
- **Calculate** a **Block**, ***Defined by*** **Box** with ***Type*** **Model**.
- In the **explorer** right click over **Boundaries** and select the option **Block**.



The resultant **2D Boundary** is defined around the outside edge of the **Block** at **Z0**.

Editing Boundaries

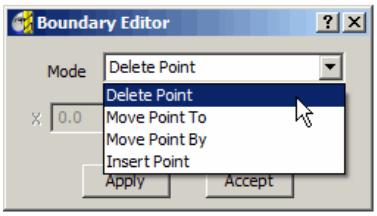
- In the **explorer** right click the ***Boundary*** (**1**) icon to open the local pull down menu and select **Edit – Copy Boundary** to create a copy named (**1_1**).
- In the **explorer** right click over the new **Boundary** and **Rename** as **master**.
- In the **explorer** click (off) the light bulb adjacent to the copy (**master**) to remove it from view in the graphics area.
- Delete the 2 inner segments on **Boundary 1** 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 Editor Form and Pull Down menu

Once created, a **Boundary** can be modified through a series of editing options accessed from the **Boundary Toolbar** via the **Edits** icon. Alternatively the form can be accessed by right clicking the actual **Boundary** in the **Explorer** and selecting **Edit – points** in the local pull down menu.



The **Mode** options provide various **Point** editing functions as described below.

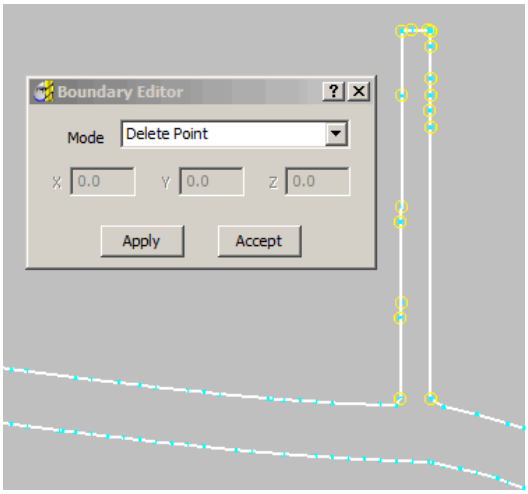
Delete Point - Selects individual points on a boundary segment to be deleted.

Move Point To – Selects a point to move to absolute X, Y and Z co-ordinates or indicate the position with the **Left** mouse button.

Move Point By - Selects points to move incrementally by X, Y and Z values.

Insert Point - Selects two points on the boundary between which to insert a point.

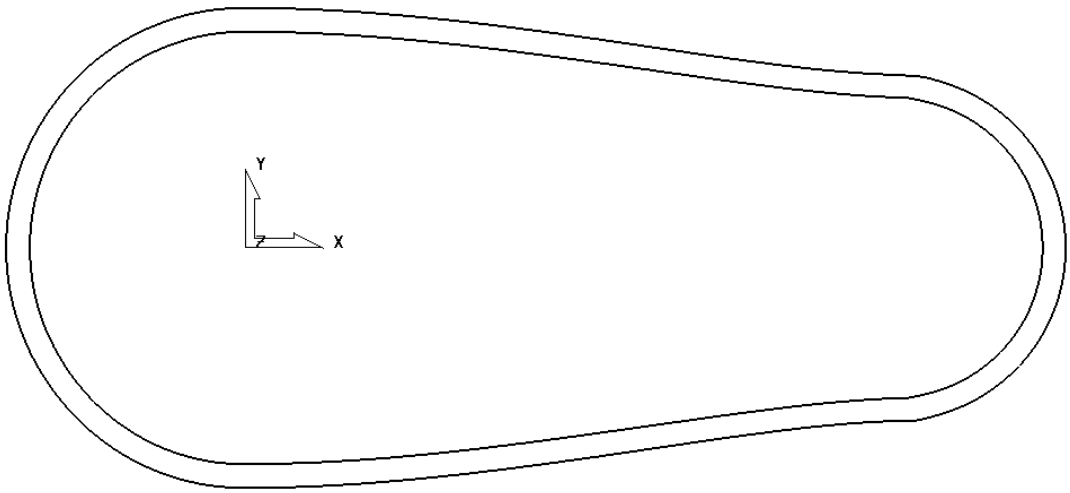
- Right click over the original **Boundary icon** (**1**) in the **Explorer** and select **Edit – points** in the local pull down menu.
- Select the **Mode** – **Delete Point** and Zoom into one of the spurs to provide easier access.



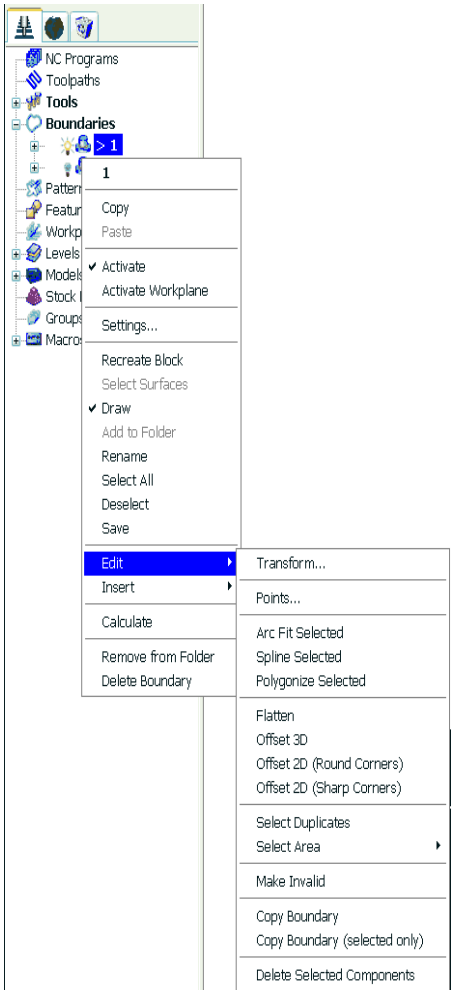
For improved access to close proximity, points **Zoom** in closer as required.

- Box around the main area of the spur and **Apply** the form to delete the selected points (Individual points can be ***selected*** or ***deselected*** if left clicked with the **Ctrl** key depressed or for ***select*** only using the **Shift** key).

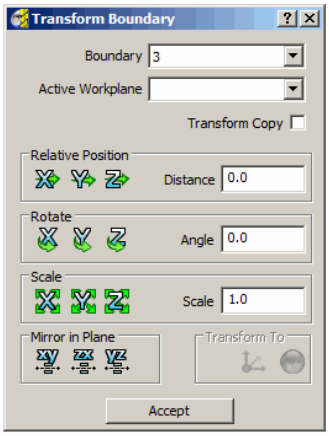
- Repeat the procedure on the other spur (as shown on the following page).



Further editing options can be found by right clicking over a **Boundary** either in the graphics area or in the **explorer** then selecting **Edit**.



Transform:-



***Move* –** moves the selected boundary or segment by a specific distance in X, Y or Z.

***Rotate* –** rotates the selected boundary or segment around X, Y or Z.

***Scale* –** scales the selected boundary or segment by a specific amount.

***Mirror* –** mirrors the boundary in the XY, YZ or XZ plane.

***Workplane Transform* –** moves the boundary so that it is in the same place relative to the workplane as it was to the global transform.

Points – displays the **Boundary Editor Form**.

Arc Fit Selected – Arc fits along selected boundary segments.

Spline Selected – Splines a curve through points on the selected boundary segments.

Polygonize Selected – Converts a curved boundary segment into a series of straight lines.

Flatten - This option flattens the boundary making it 2D.

Offset 3D - This offsets the boundary by the distance specified along the component surfaces.

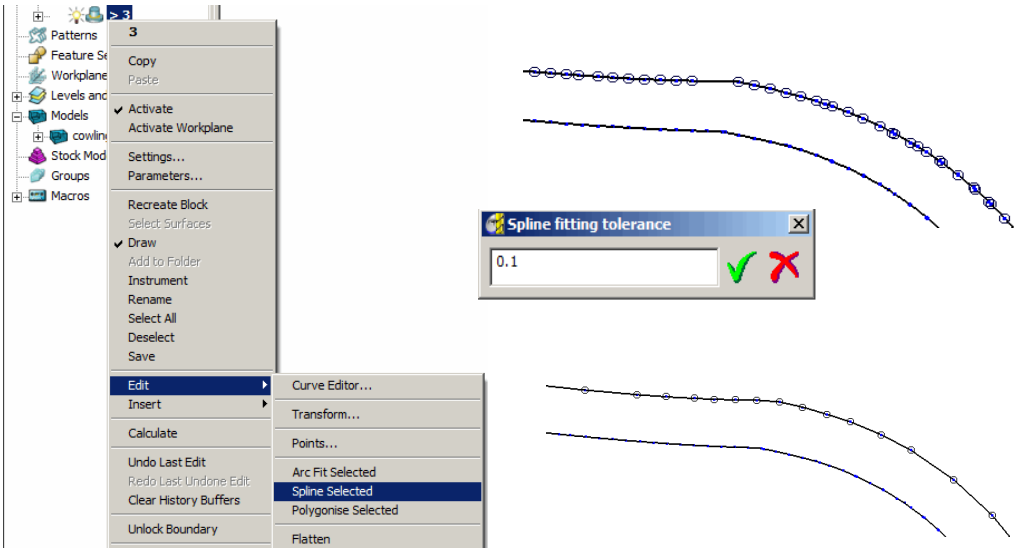
Offset 2D - This offsets the boundary by the given distance whilst flattening it automatically.

Select Duplicates – selects all duplicate segments of the boundary and can be used effectively in conjunction with the **Delete Selected** option.

Select Area – Selects segments **Greater Than**, or **Less Than** the **Ratio of the Tool Area**.

Make Invalid – De-calculates an existing **Boundary**.

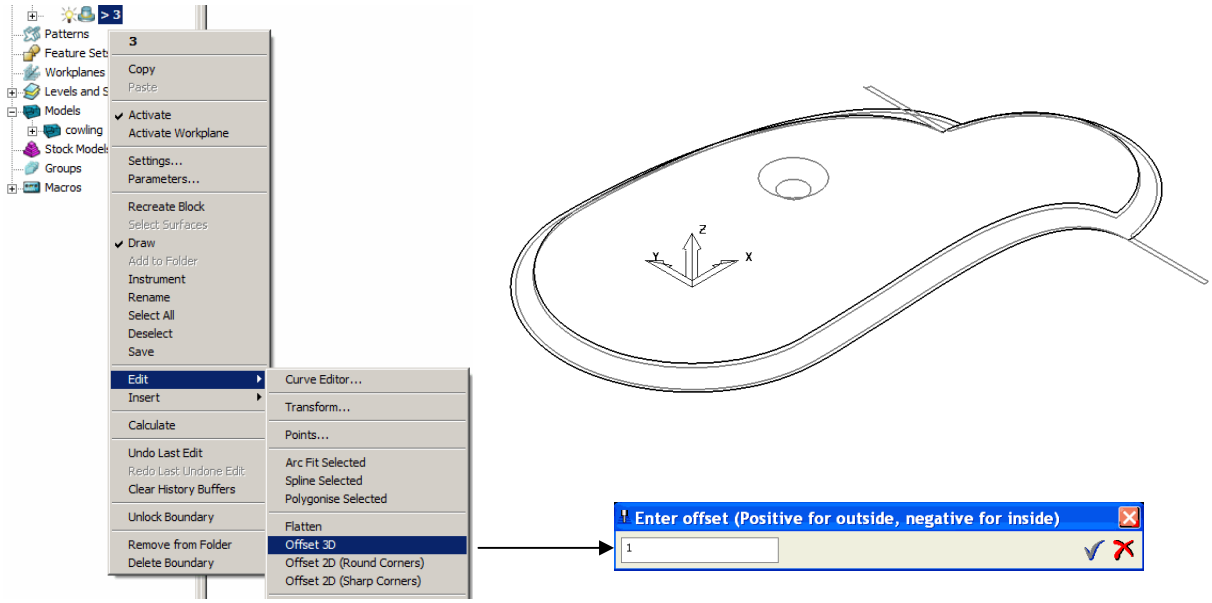
- Select the outer segment of the **Boundary** (**1**) and Right click on it's **Boundary Icon** in the **explorer** to open the local pull down menu.



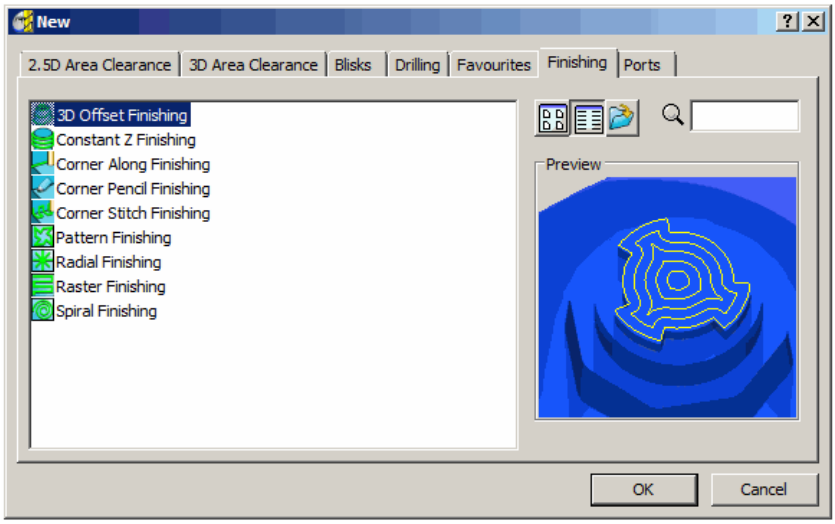
- Select **Edit** – **Spline Selected** entering a tolerance value of **0.1** in the form before clicking the **Green Tick** to apply.
- Re-select the outer segment to visualise the effect of the **Spline** option (as shown above right).

The **Selected Segment** has now been **Splined** (curve fitted and repointed 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.

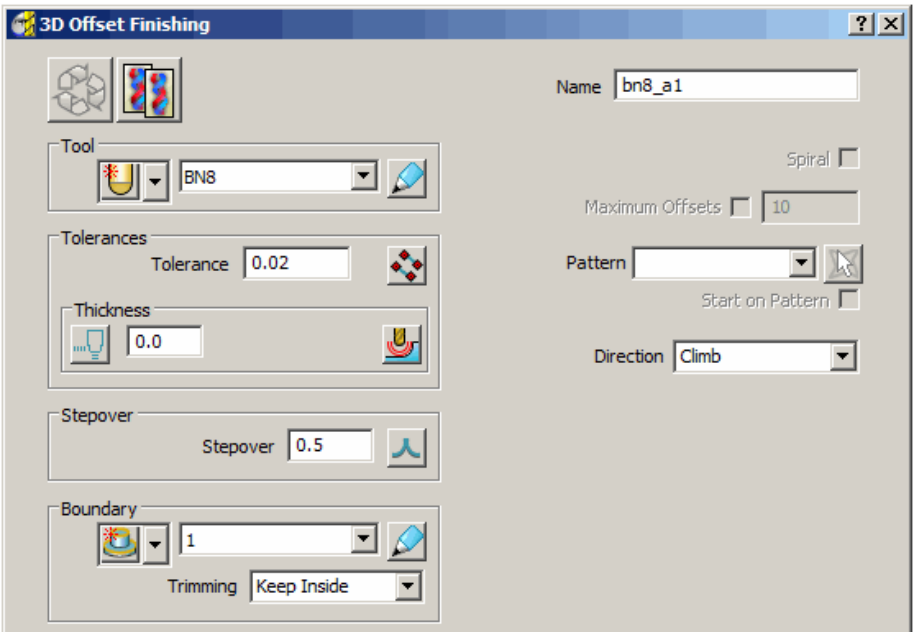
- Right click on the **Boundary** icon in the **explorer** to open the local pull down menu again.
- Select **Edit - Offset 3D** entering a value of **1** in the form before clicking the **Green Tick** to apply (All **Boundary *segments*** are **Offset** relative to the surface normals of the component as shown below).



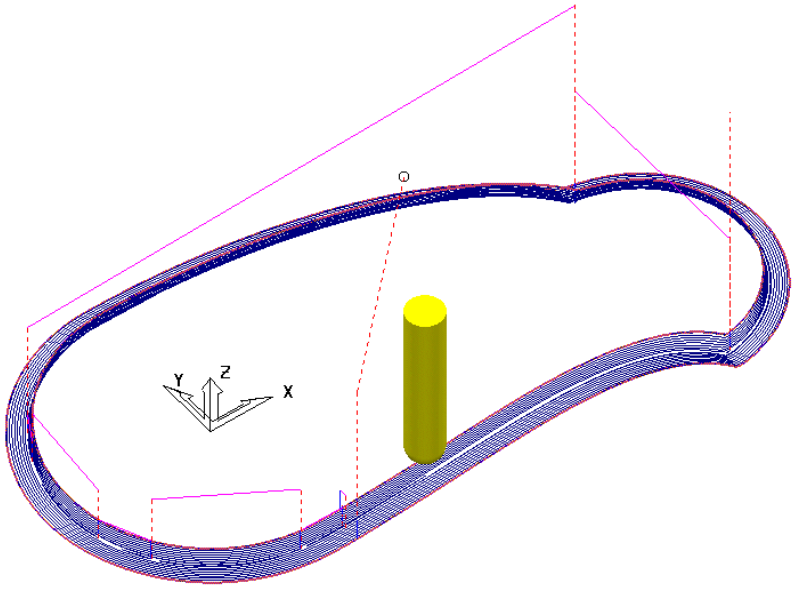
- In the **explorer** click (on) the light bulb adjacent to the copy of the original **Boundary** (**master**) created earlier (It will be displayed alongside the edited **Boundary** (**1**) as shown above illustrating the effect of the **3D Offset**).
- Select the **Toolpath strategies** icon  to open the following form and select the **Finishing** options.



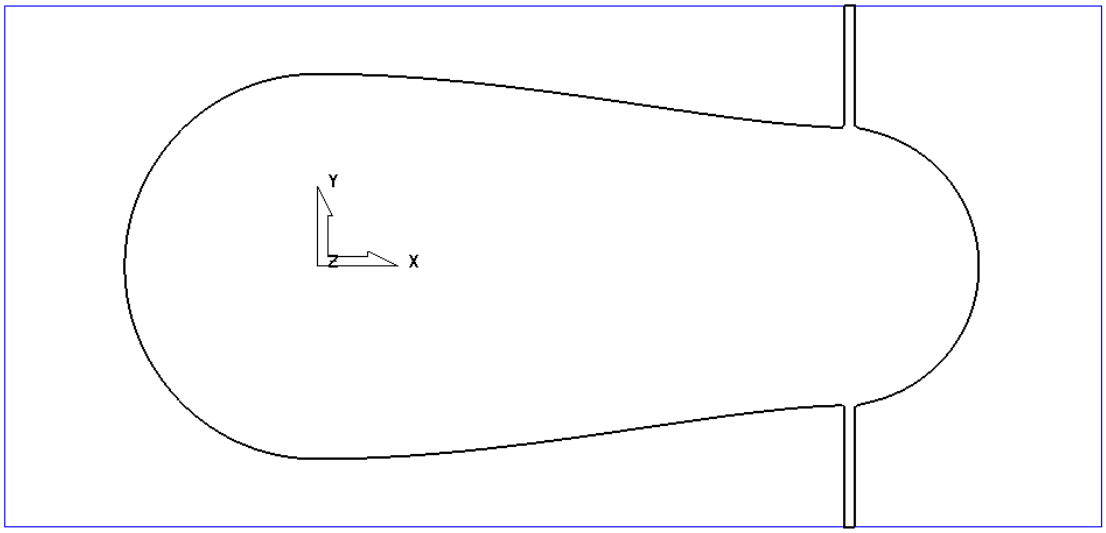
- Select **3D Offset Finishing** to open the following form and enter data exactly as shown below.



- **Apply** the form to create an alternative form of **Rest** machining using the modified **Rest Boundary** (**1**) as the limit for the **3D Offset** strategy **bn8_a1**.
- As soon as the toolpath has been created **Cancel** the above form.

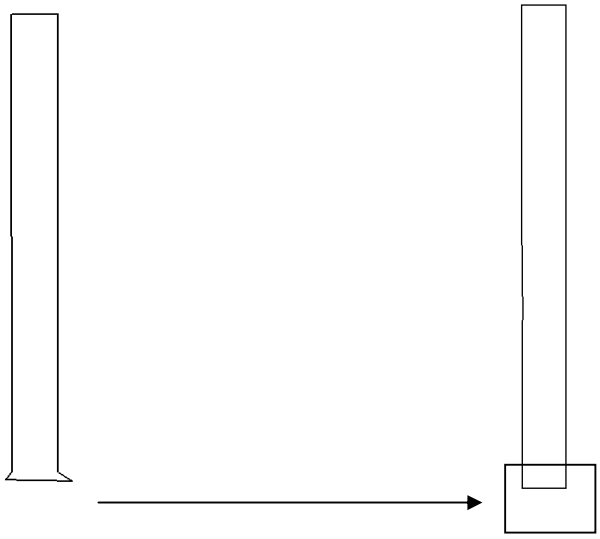


- Undraw the **Toolpath** (**bn8_a1**) and **Boundary** (**1**) by switching off the respective light bulb symbols in the **explorer**.
- **Select** the outer segment of the displayed **Boundary** (**master**).
- Right click the **Boundary** icon (**master**) in the **explorer** to open the local pull 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 (**2**).
- In the **explorer** click (off) the light bulb adjacent to the **Boundary** (**master**) to remove it from view.

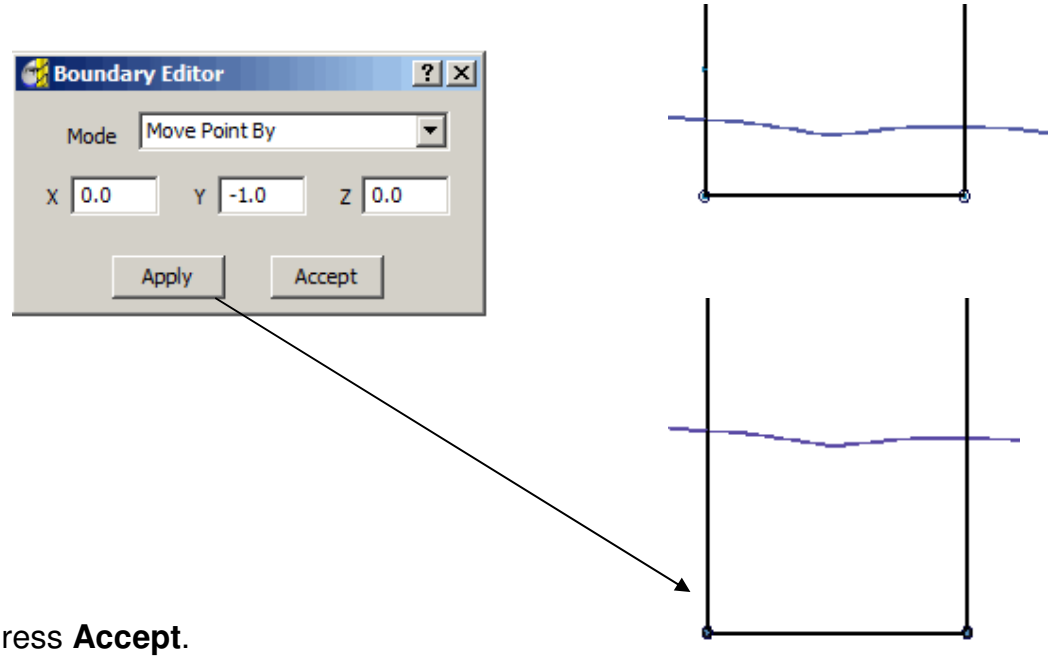


- Right click over the **Boundary icon** (**2**) in the **Explorer** and select **Activate** followed by **Edit – Points** in the local pull down menu.

- Select the **Mode** – **Delete Point** and select all points on the remaining segments apart from the upper spur.
- **Apply** the form to delete selected points as required (in stages) gradually zooming into the upper spur for easier access (as shown below).

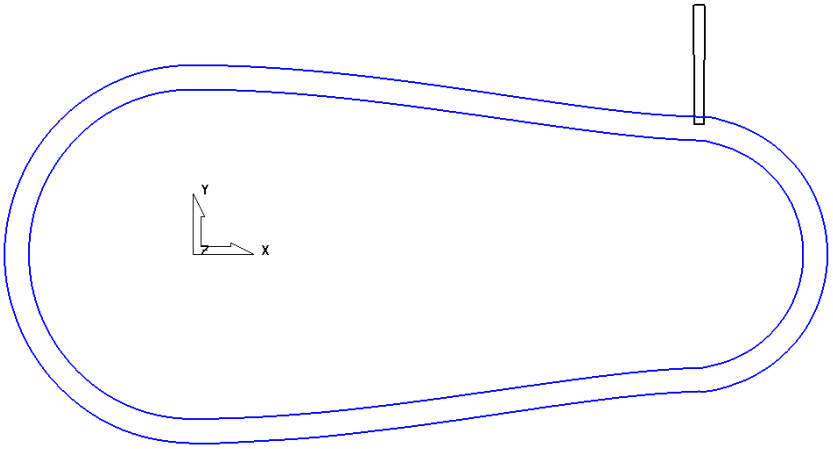


- Once the above rectangular form remains **Zoom** onto the lower end.
- In the **Explorer** right click the **Boundary icon** (**2**) to open the local pull down menu and select **Edit - Points**.
- Select the **Mode** – **Move Point By** and select both points at the base of the upper spur and enter **X0 Y–1 Z0** in the form (as shown below).
- **Apply** the form to incrementally move the selected points a distance of **Y-1** to provide additional overlap into the **Boundary** (**1**).

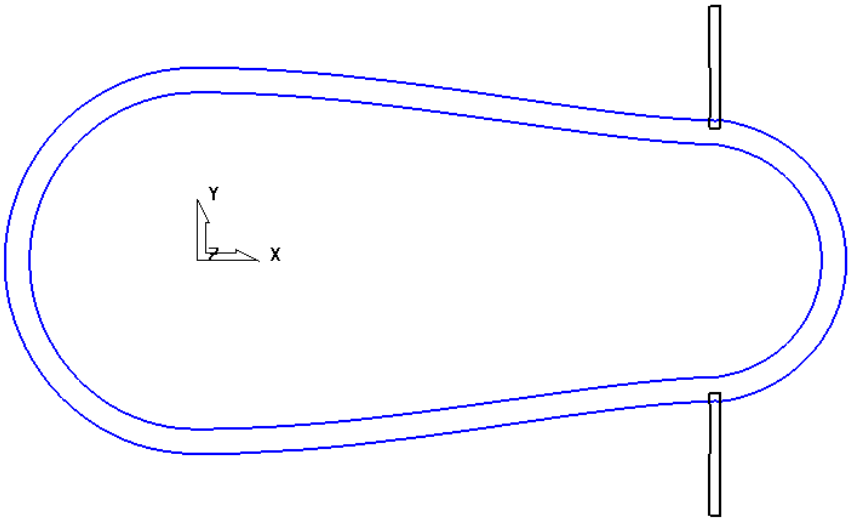


- Press **Accept**.

- **Resize** the View. 



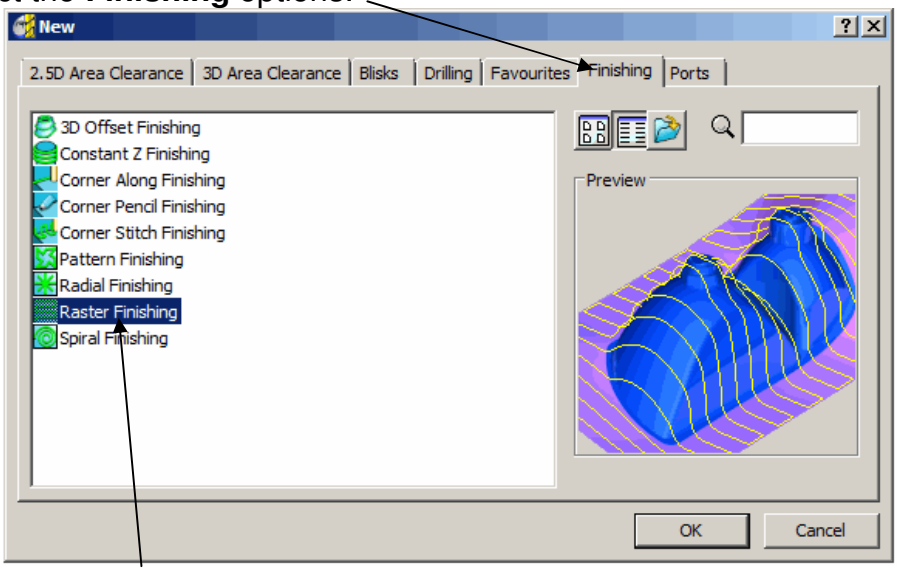
- In the **explorer** right click the **Boundary** (**2**) icon to open the local pull down menu and select **Edit – Copy Boundary** to create a copy (**2_1**).
- Using the local pull down menu again, **Rename** the copy as (**3**).
- In the **explorer** right click the **Boundary icon** (**3**) to open the local pull down menu and select **Edit – Transform – Mirror in Plane ZX**.



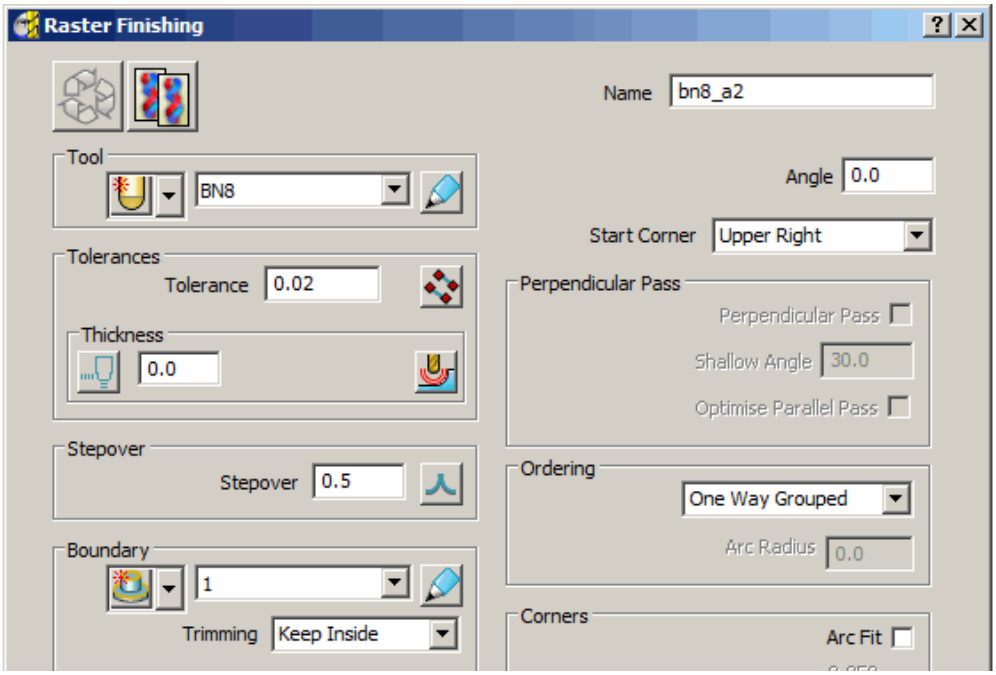
The 2 separate spur boundaries can be **Appended** together either dynamically within the **Explorer** or by using the **Insert** option in the local pull down menu. This example will use the first method.

- Move the cursor over the **Boundary** (**3**) icon in the **Explorer**, depress the **left mouse key** and keep it down.
- Depress the **Ctrl key** on the keyboard and also keep it down while the cursor is moved to the right (Observe the small circular symbol with a diagonal line across it along with a ghosted image of the **Boundary** icon).

- Still keeping the **left mouse key** and **Ctrl key** depressed, move the cursor up and across to the **Boundary** (**2**) icon.
- Release the **left mouse key** first followed by the **Ctrl key** (a copy of **Boundary** (**3**) will have been appended into **Boundary** (**2**)).
- In the **explorer** right click the **Boundary icon** (**3**) to open the local pull down menu and select **Delete Boundary**.
- Select the **Toolpath strategies** icon  to open the following form and select the **Finishing** options.

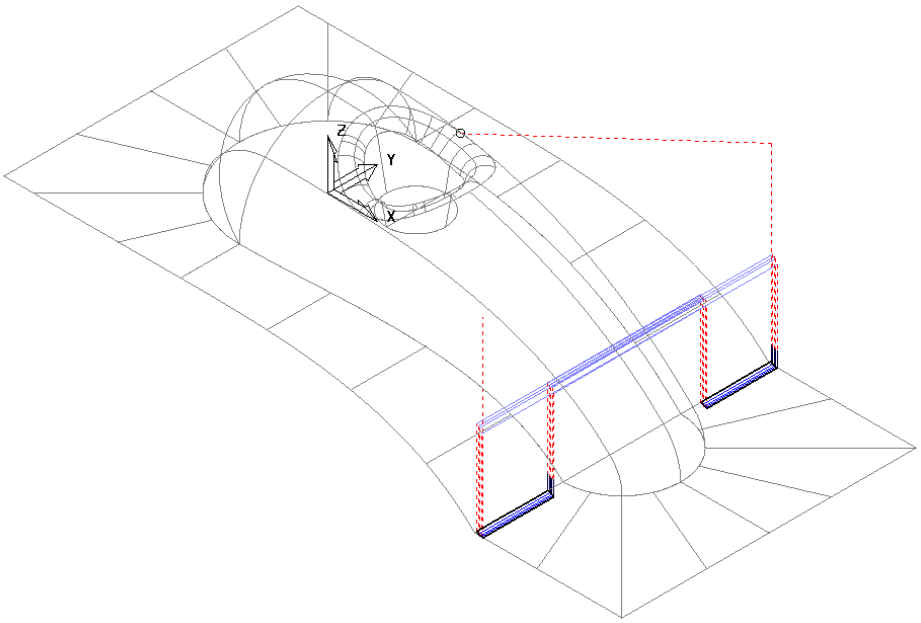


- Select **Raster Finishing** to open the following form and enter data exactly as shown below.

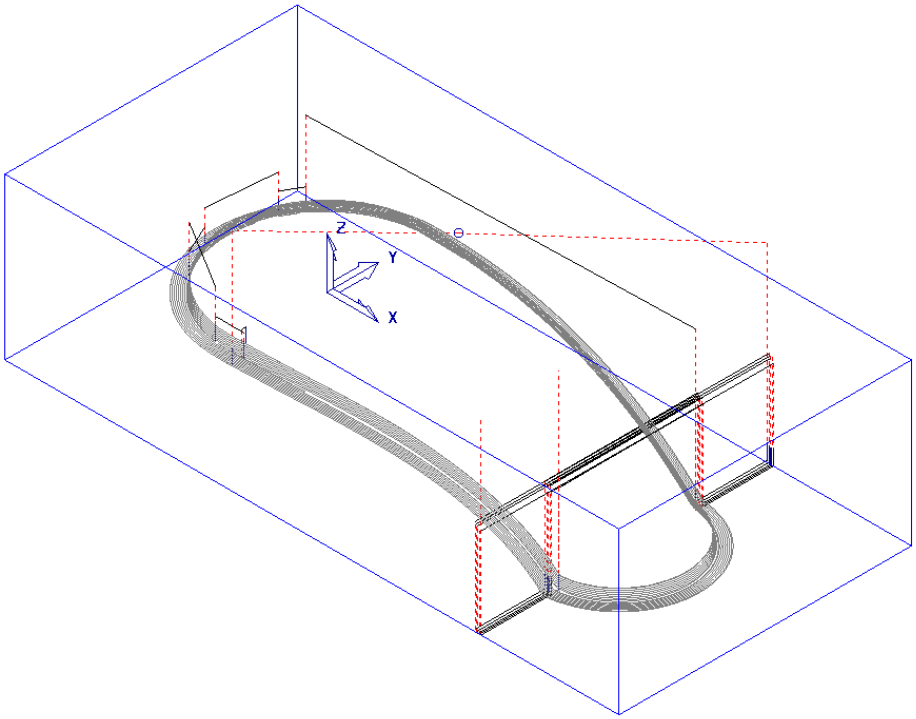


- **Apply** the form to create a local finishing strategy using the modified **Rest Boundary** (**2**) as the limit for the **Raster** strategy **bn8_a2**.

- As soon as the toolpath has been created press **Cancel**.



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**:-

D:\users\training\COURSEWORK\PowerMILL-Projects\Boundary-tests

Creating Blanking Surfaces using Boundaries

Blanking planes are used to cap holes and pockets in a model to prevent the tool from machining those areas. There are three types of **Plane** than can be created in PowerMILL.

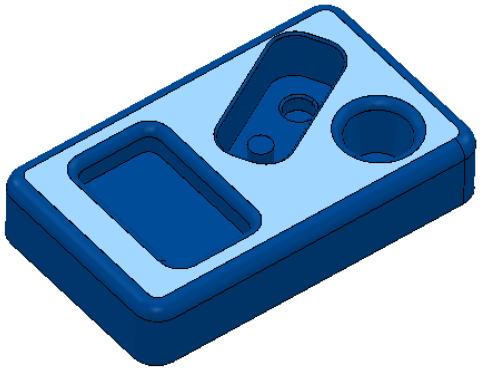
1. **Block –** allows the creation of a plane at a specific Z Height after the material **Block** had been defined.
2. **Best Fit –** creates two planes, one touching the boundary at its highest point and the other through its lowest point. The planes are parallel to the best fit plane (the plane that 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

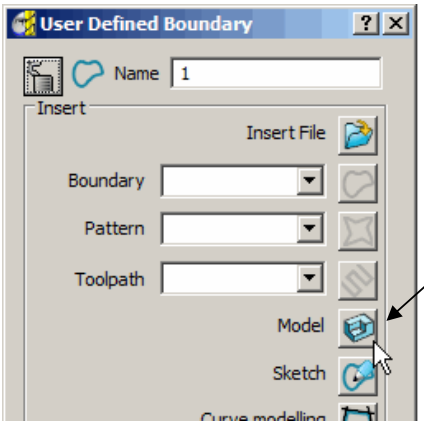
- **Delete** the **current model, toolpaths and boundaries.**
- **Import** the **model**:-

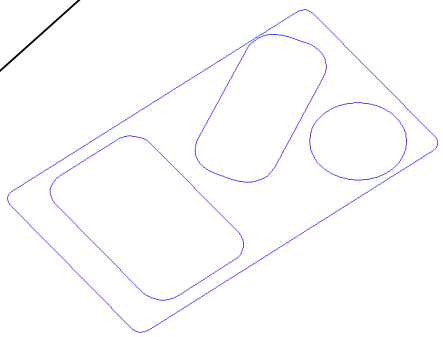
D:\users\training\PowerMILL_Data\Models\pockets.dgk

- Define the **Block** to **Min/Max limits**.
- Select the top **surface** of the model (shown shaded).



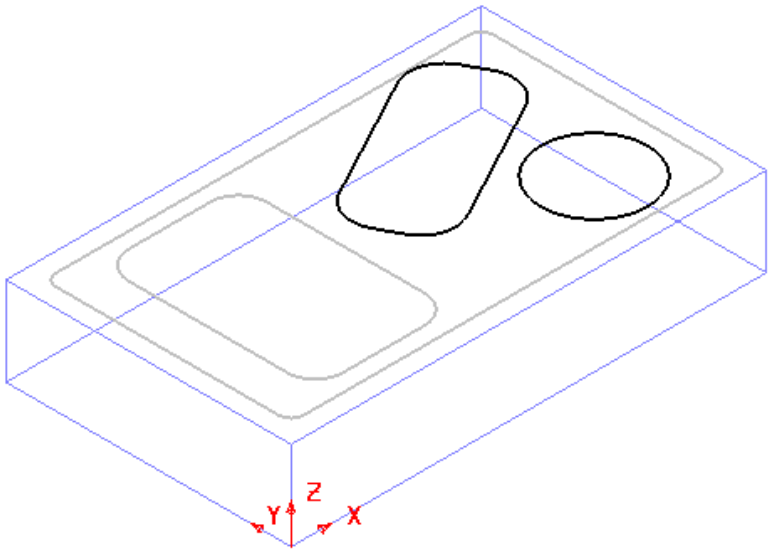
- Create a **User Defined Boundary**, select the icon **Model** and **Accept** the form.



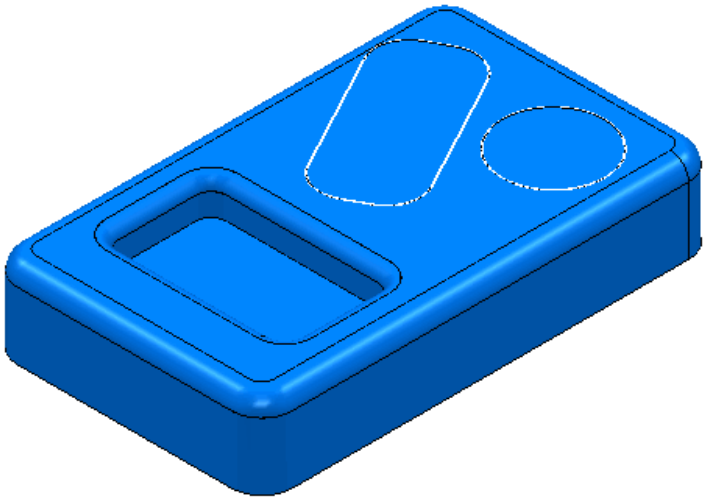


The **boundary** produced has 3 inner segments. For this example only the two smallest segments are required.

- Select the two highlighted segments by dragging a window over them.



- Right click over **Models** in the **explorer**, and select **Create Plane - Projected**.



Two blanking planes are created as shown.

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

In the **Models** area of the **explorer**, a new model called **Planes** has been created.
To delete the planes simply right click over **Planes** and select **Delete Model**.

Exercise

- From **Examples**, open the model **bucket.dgk** and generate 4 surface planes to cover the foot recesses on the upper base surface.

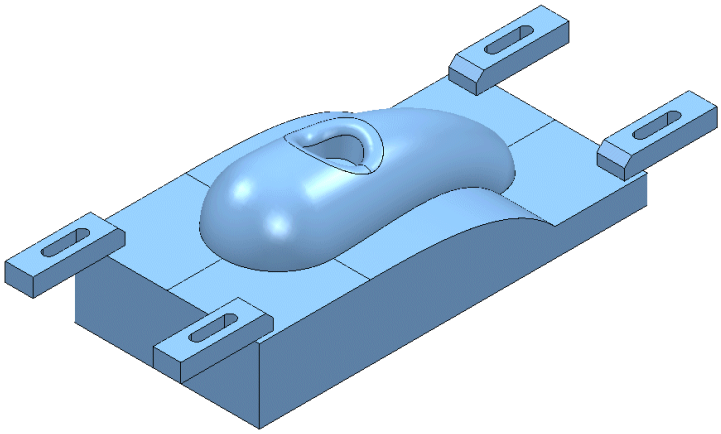
7. Levels and Sets

Introduction

Levels and **Sets** provide a more specific method for controlling whether groups of model components are ***selected*** or ***displayed*** within the ***PowerMILL***, graphics area.
Any selection of component parts in the imported **CAD Model** can be **Acquired** to a **Level** or **Set**. Once assigned to a **Level** or **Set,** model components can be collectively ***selected***, ***undisplayed***, ***deselected***, or ***displayed*** using the local menu options. The subtle difference between **Levels** and **Sets** is illustrated during the following worked examples.

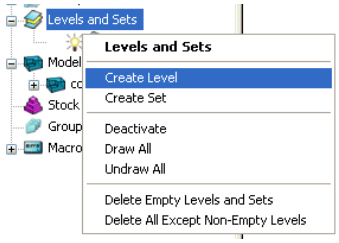
Level Creation

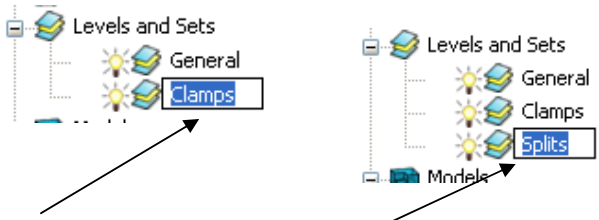
- From the main **pulldown** menus select **File - Delete All**.
- From ***File*** **- Import Model** select:-
  D:\users\training\PowerMILL_Data\Models\CowlingWithClamps.dgk



The imported **model** includes **clamps** which must be avoided by all **machining** strategies created in the **Project**. Also, the clamps are not to be included during the calculation of the **Block**.
To make the **model** selection process easier, a new **Level**, ***Named*** **Clamps** will be created and all **clamp Surfaces** will be acquired to it. Another **Level**, ***Named*** **Splits** will be created and all 6 of the split **Surfaces** will be acquired to it.

- Create a **Dia 16 + 3 Tip Radiused** tool ***named*** **D16t3**.
- In the **explorer** right mouse click on **Levels and Sets** to access the local menu and from it select **Create Level**.



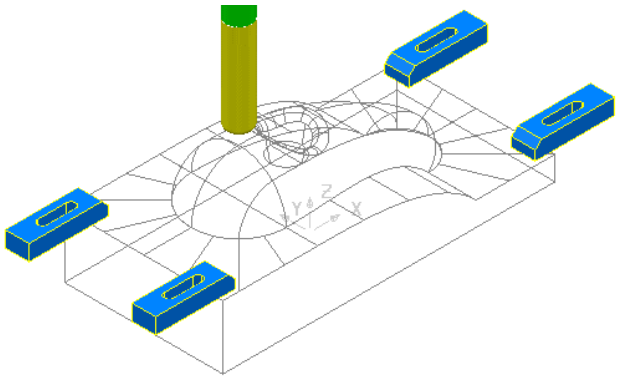


- **Rename** the new **Level** as ***Clamps***.
- Create another new **level** and **Rename** it as ***Splits***.

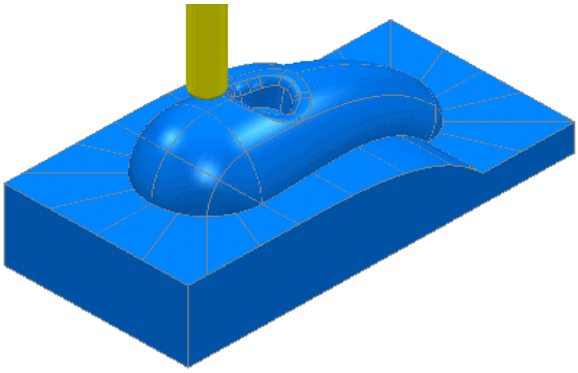
All ***model components*** are currently **Acquired** to the default ***General*** **Level**.
If the original **CAD model** has components already assigned to different **Levels** then these will be recreated in the **PowerMILL Project** on **Import**.

Acquiring model entities to a Level

- In the **explorer** left click on the **light bulb** on **Level *Clamps*** to switch it off (If any items are **acquired** to this **level** they will be currently **undisplayed**).
- In the ***Graphics Area*** use the **Shift key** and **Left mouse button** to accumulate the **selection** of all the **surfaces** that define the clamps.

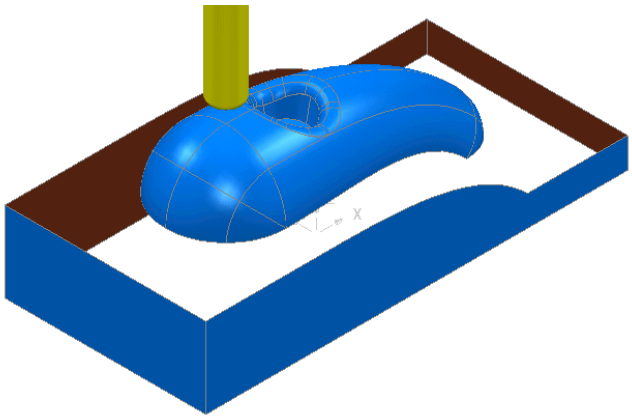


- In the **explorer** right click on the **Level *Clamps*** and from the local menu select **Acquire Selected Model Geometry** and observe that the clamps disappear from the view (The **Level** is switched off).



- Select the **6 Surfaces** defining the split face and acquire them to the **level** *named* ***Splits***.

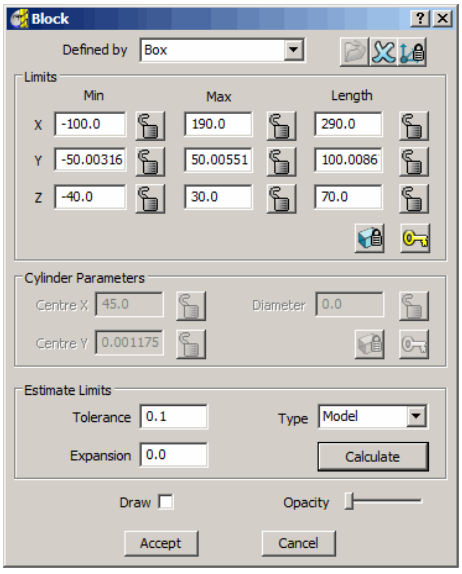
- ***Switch off*** the **level** named ***Splits*** to prove that the correct **surfaces** have been acquired (as shown below).

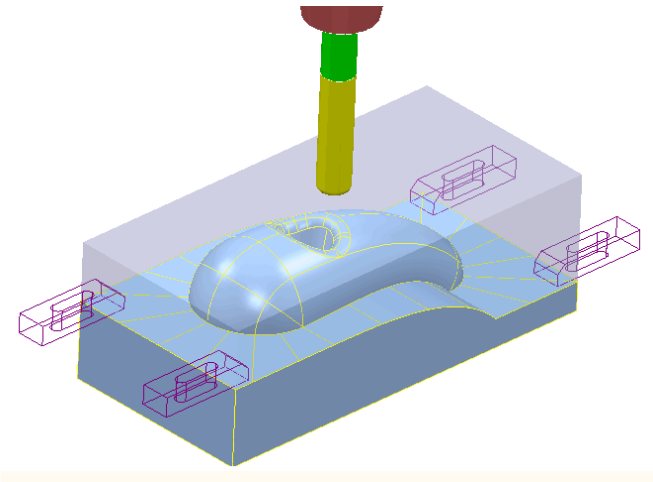


- ***Switch on*** *all* **levels** (The ***clamps*** and ***split faces*** will reappear).

A **Selected Surface** - **Boundary** will be created to protect the clamps from being inadvertently machined.

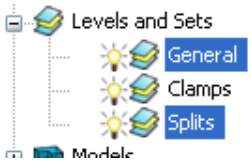
- In the **explorer** right mouse click on the **level** named ***General*** and from the local menu click on **Select Surfaces** (The ***main component*** **surfaces** excluding the ***split faces*** will be **selected**).
- While still in the **explorer** right mouse click on the **level** named ***splits*** and from the local menu click on **Select Surfaces**. (The ***split face*** **surfaces** will be added to the current selection).
- **Calculate** a **Block** using the default **Box** and **Model** options.
- Use default **Rapid Move Heights** and **Start and End Point**.





The **Block** will be created to the dimensions of the ***selected*** **surfaces**.

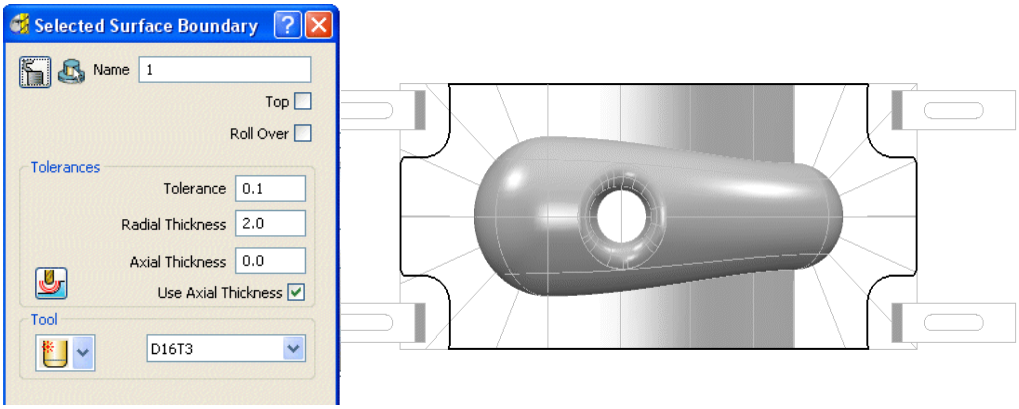
- **Left mouse click** in the ***graphics area*** away from the **model** to deselect all entities.
- In the **explorer** ***right mouse click*** on the **level** named ***General*** and with the **Ctrl Key** held down ***left mouse click*** on the **level** named ***Splits***.



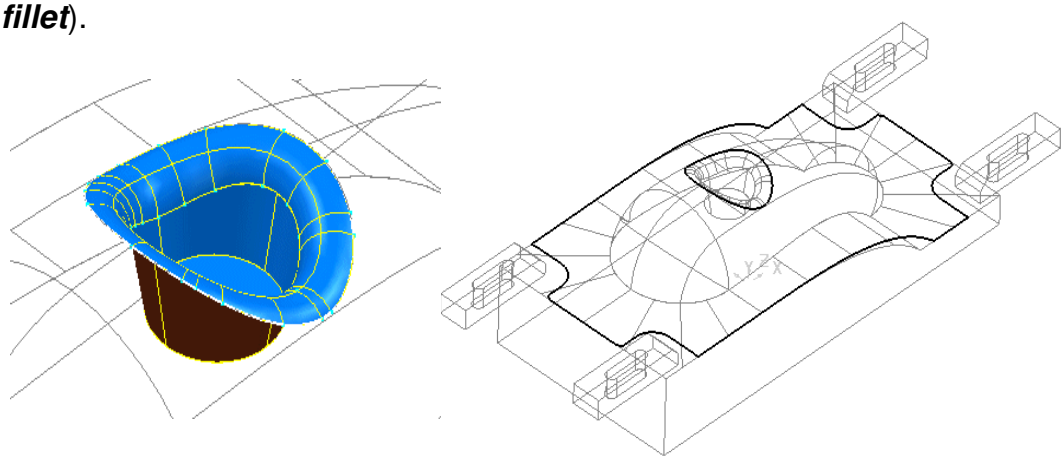
Both the ***selected levels*** will be highlighted in the **explorer**.

- Right mouse click on one of the ***highlighted levels*** and in the local menu click on **Select Surfaces** (All **surfaces** excluding the ***clamps*** will be **selected**).

- Create a **Selected Surface Boundary** with **Radial Thickness 2** ,using the current selection to produce the following segment.

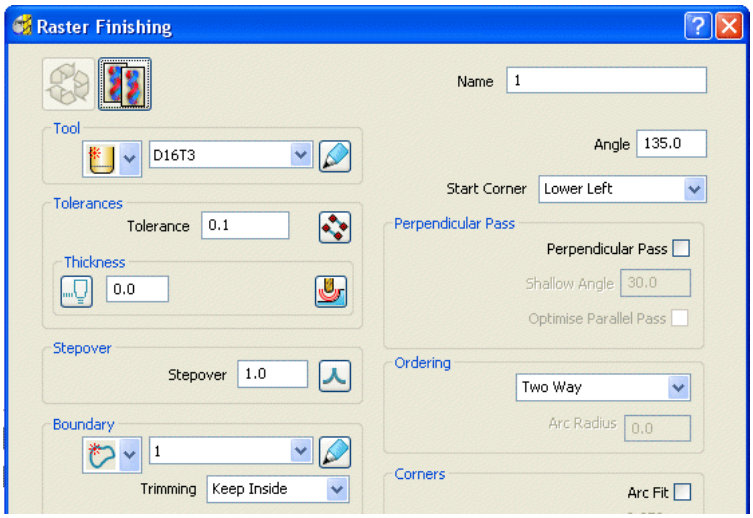


- Manually select the **surfaces** defining the ***central pocket*** (including the ***fillet***).

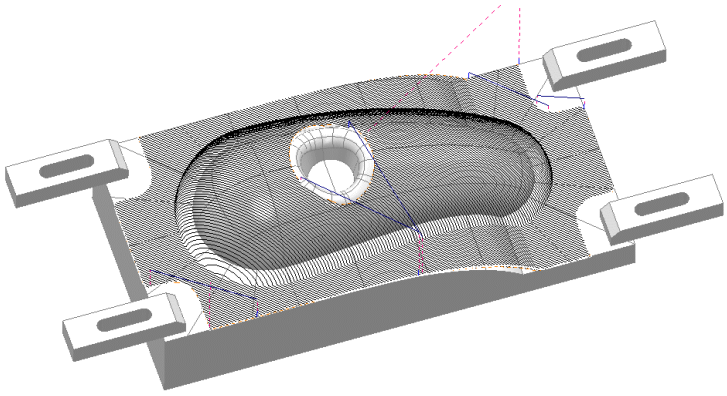


- ***Right mouse click*** on the existing **Boundary** and from the local menu select **Insert – Model** to add a new ***segment*** (Shown above right).

- In **Toolpath Strategies** - **Finishing** select **Raster Finishing** and input the data into the form exactly as shown below.



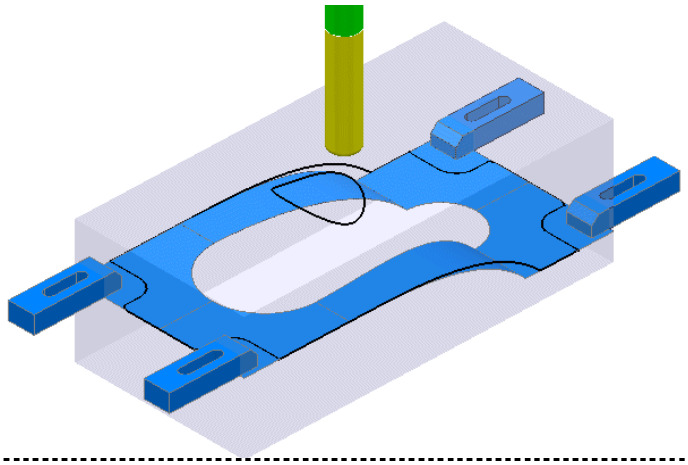
- **Apply** and **Accept** the form to create the following finishing toolpath.



The outer **boundary segment** provides a 2mm clearance around the clamps.

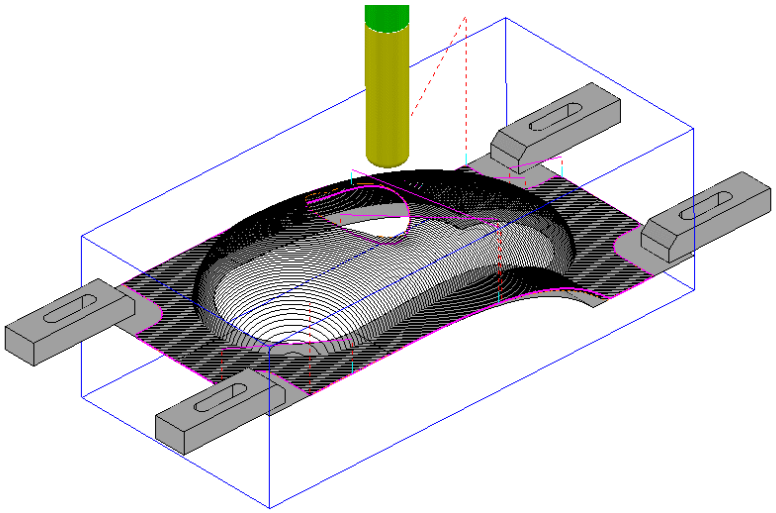
Switching Off a Level

- In the **explorer** switch off the ***light bulb*** on the **level** named ***General***.



Only the ***Clamp*** and ***Split*** **Surfaces** will be displayed in the graphics area.

- Right mouse click on the **toolpath** and from the local menu, select **Settings** to re-open the form.
- Select **Make a Copy** before selecting **Apply** and **Accept** to generate a copy of the same **Finishing** strategy.

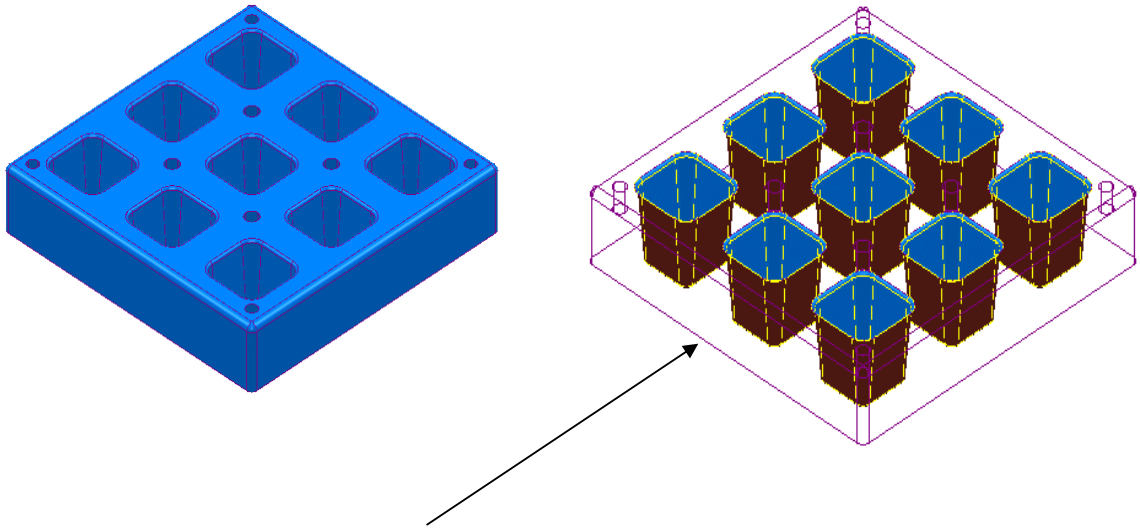


If a **Level** is switched off, ***PowerMILL*** will still create **toolpaths** over the ***acquired***, un-displayed part of the model.

Sets

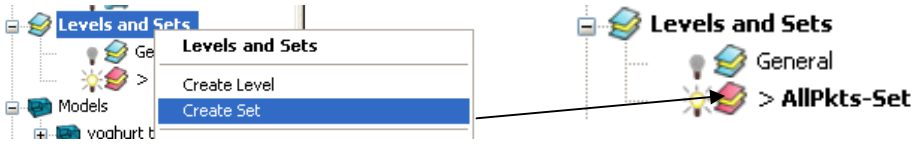
A ***model entity*** must always be acquired to a **level** but can only exist on one specific **Level** at the same time.
Sets differ from **Levels** in that the same ***model entities*** can exist on more than one **Set** at the same time or none at all.

- From the main **pulldown** menus select **File - Delete All**.
- From ***File*** **- Import Model** select:- **D:\users\training\PowerMILL_Data\Models\YogurtTray.dgk**

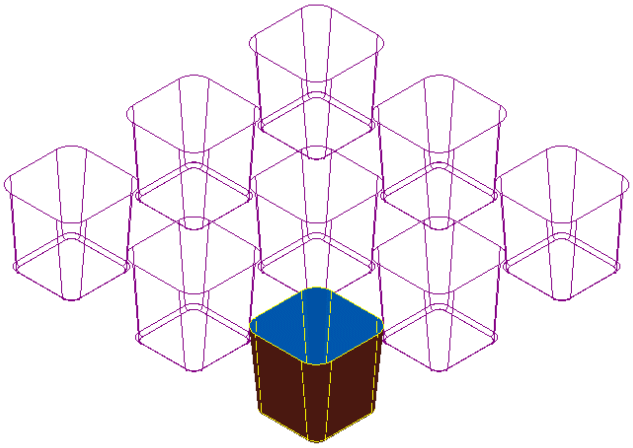


- Manually select all **surfaces** defining the ***9 pockets***.
- Create a **Set** and **rename** as ***AllPkts-Set***.

- ***Right mouse click*** on the **Set**, ***AllPkts-Set*** and from the local menu select **Acquire Selected Model Geometry**.



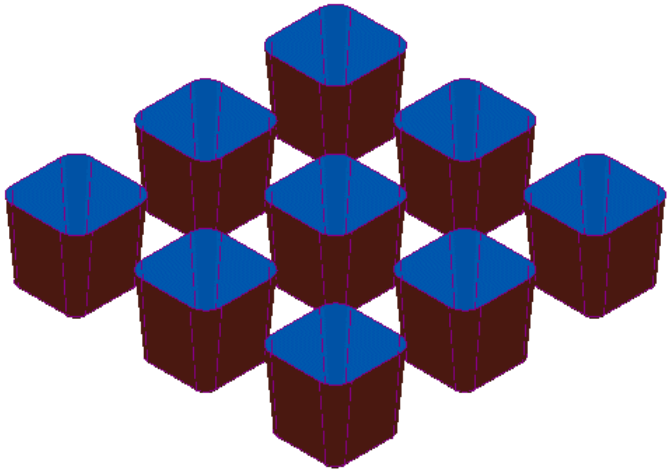
- Manually select all **surfaces** defining the bottom left ***pocket***.



The selected **Surfaces** will soon be **acquired** to two **Sets** at the same time.

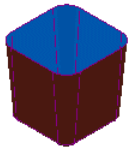
- Create a **Set** and **rename** as ***Pkt1-Set***.
- ***Right mouse click*** on the ***Set***, ***Pkt1-Set*** and from the local menu select **Acquire Selected Model Geometry**.

- ***Switch off*** both **Level** - **General** and **Set**, ***Pkt1-Set***.
- ***Switch on*** the **Set**, ***AllPkts-Set***.



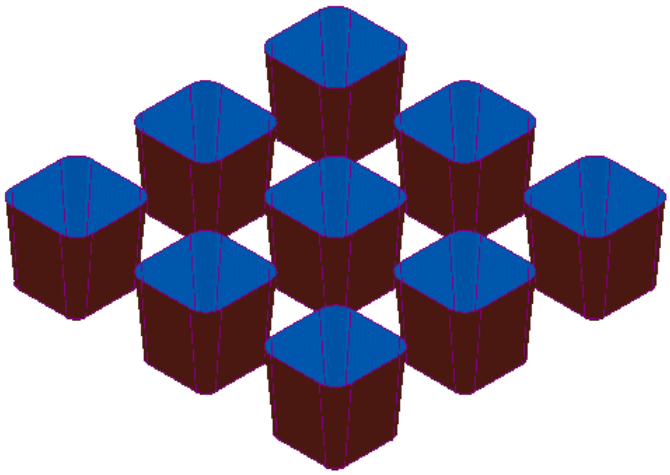
All **surfaces** acquired to **AllPkts-Set** will be displayed.

- ***Switch off*** the **Set** ***AllPkts-Set*** and then ***Switch on*** the **Set**, ***Pkt1-Set***.



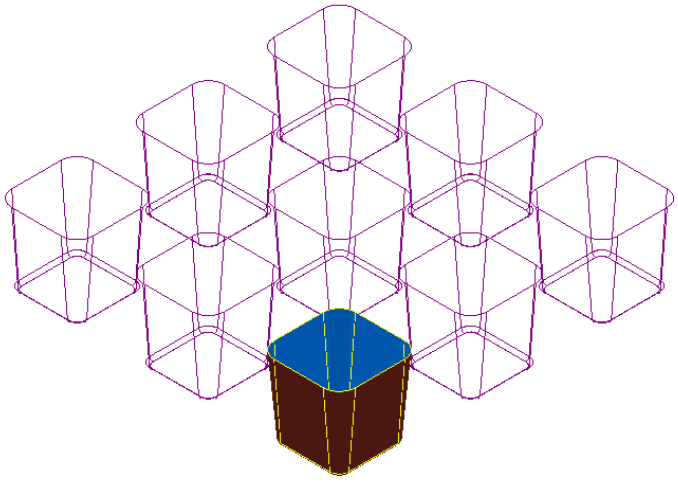
All **surfaces** ***acquired*** to **Pkt1-Set** are displayed. These are also included as displayed when the **Set**, **AllPkts-Set** is ***switched on*** and **Pkt1-Set** is ***switched off***. This illustrates that ***model entities*** can be ***acquired to*** different **Sets** at the same time.

- ***Switch off*** the **Set**, ***Pkt1-Set*** and then ***Switch on*** the **Set**, ***AllPkts-Set***.
- ***Right mouse click*** on the **Set**, ***AllPkts-Set*** and from the local menu click on **Select Surfaces**.
- **Create** a new **level** and ***rename*** it as ***AllPkts-Lev***.
- In the **explorer** right click on the ***Level AllPkts-Lev*** and from the local menu select **Acquire Selected Model Geometry**.
- **Switch off** all **Levels** and **Sets** except **AllPkts-Lev**.

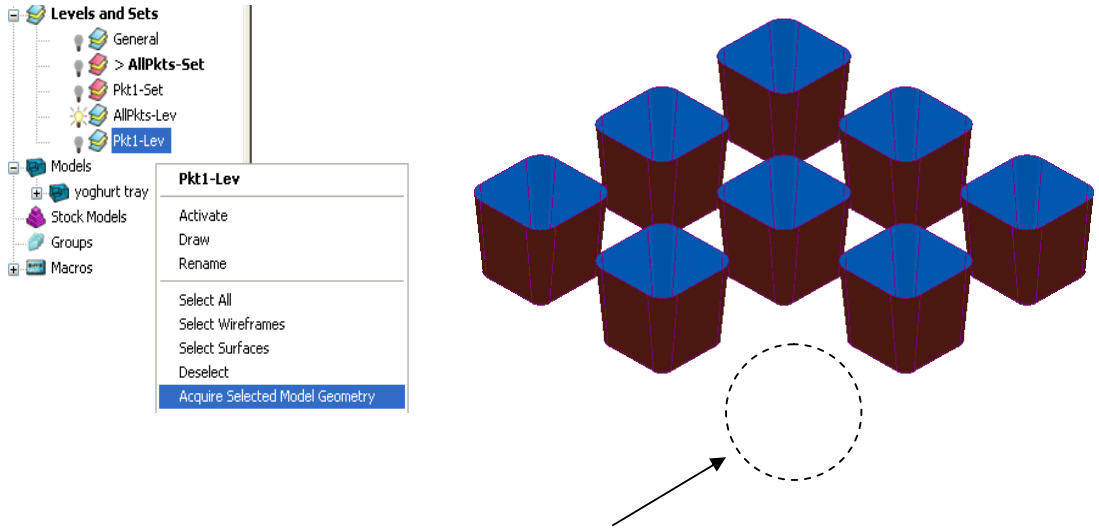


All **surfaces** acquired to **AllPkts-Lev** will be displayed.

- **Create** a new **level**, ***rename*** it as ***Pkt1-Lev***, *and* ***switch it off***.
- Manually select all **surfaces** defining the bottom left ***pocket***.



- In the **explorer** right click on the ***Level Pkt1-Lev*** and from the local menu select **Acquire Selected Model Geometry**.



All **surfaces** defining the bottom left ***pocket*** disappear from the view on being acquired to the (**switched off**) **level** ***Pkt1-Lev***. This illustrates that unlike **Sets**, it is not possible for model entities to be **acquired** to more than one **Level** at any one time.

- ***Switch on*** **Level** ***Pkt1-Lev*** and the bottom left ***pocket*** **surfaces** will be displayed again.

Note:-
When using a combination of **Levels** and **Sets** on the **model** it is good policy to keep the **Sets** ***visually, switched off***. If a particular set needs to be selected then this is achieved by right mouse clicking on its icon in the **explorer** and using one of the **Select** options in the local menu. This will prevent confusion if the user switches a **Level** ***off*** only to find the included items are still ***visible*** due to a set containing the same items being ***switched on***.

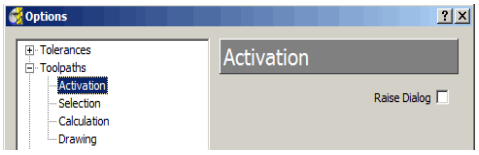
8. Editing Toolpaths

Toolpaths

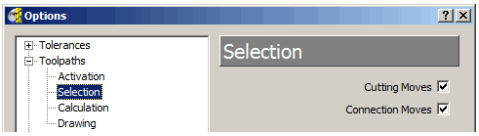
Any information stored in the **PowerMILL explorer** including toolpaths will be lost upon exiting the program, unless saved in a **Project**.

Options Form

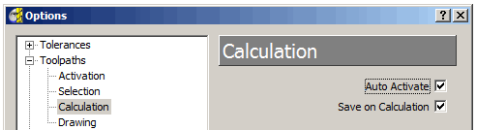
The ***default options*** for **Toolpaths** can be tailored to suit the user application. They are found in the ***main pull down*** menus under **Tools - Options** followed by selecting the ⊞ next to **Toolpaths** to access the relevant options. The main options applicable to **Toolpaths** are as illustrated below:-



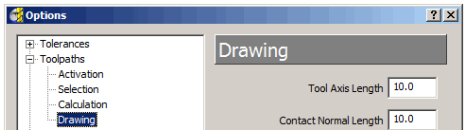
Raise Dialogue - Opens the machining form complete with settings on activation of a toolpath.



Cutting/Connection Moves - enables the selection of cutting or connection moves in the currently active toolpath.

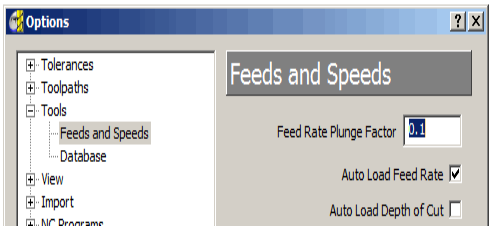


Auto Activate - The most recent toolpath is automatically made active.
Save on Calculation - Automatically **Saves** the active **Project** on the creation of a toolpath.



These options control the visual size of the **Tool Axis Length** and **Contact Normal Length** displayed.

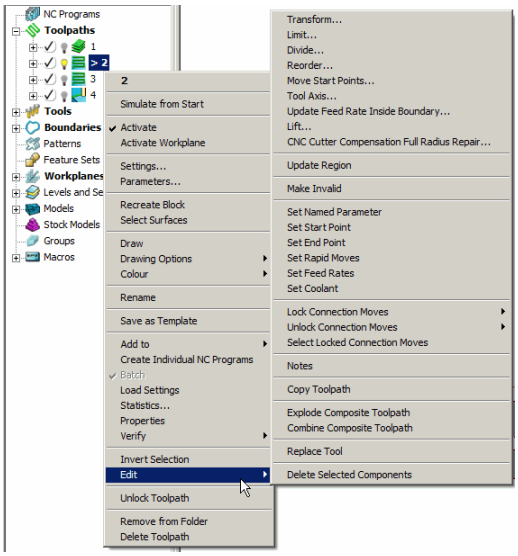
Some of the options located in the **Tools** options also relate to **Toolpaths**:-



Feed Rate Plunge Factor -When loading the cutting feed rate from a tool, the plunge rate is set as the defined Factor of the cutting feed.
Auto Load Feed Rate – Sets the cutting Feed Rate defined with the tool as well as the percentage Plunge Factor on tool activation.

Editing Toolpaths

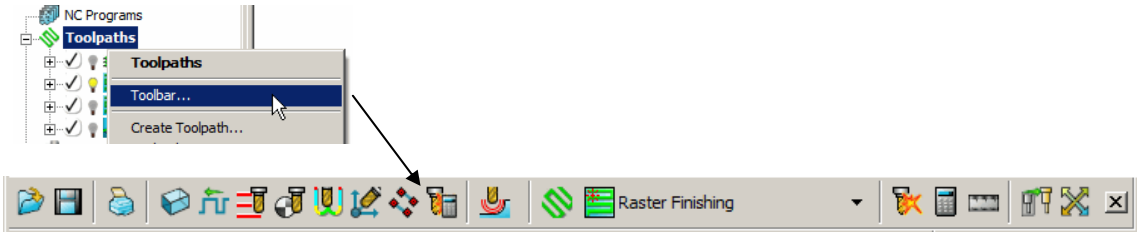
The toolpath editing options are accessed in the **PowerMILL explorer** by right clicking on the **Active Toolpath** and selecting the **Edit** pull down menu, as shown below.



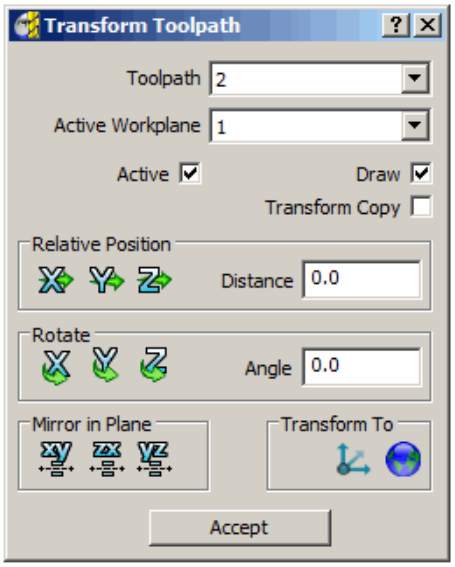
Note; It is also possible to access the menu by Left mouse clicking over the **Active Toolpath** displayed in the graphics area.

Alternatively a **toolbar** is available containing icons to access editing functions for the **active toolpath**.

Right click on **Toolpaths** in the **explorer** and from the menu, select **Toolbar**.

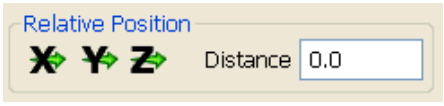


The ***toolpath toolbar*** will appear at the top area of the screen. The options will only be available for use while a **Toolpath** is **Active**.

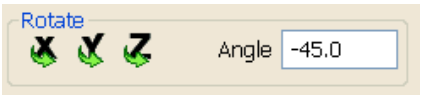


Edit >Transform

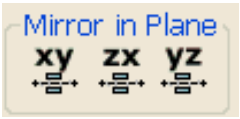
The transform option allows the user to **Move**, **Rotate**, and **Mirror** toolpaths relative to an active workplane.



Transform – Move
The **Move** function allows a selected toolpath to be moved a user defined **Distance** along a selected axis.



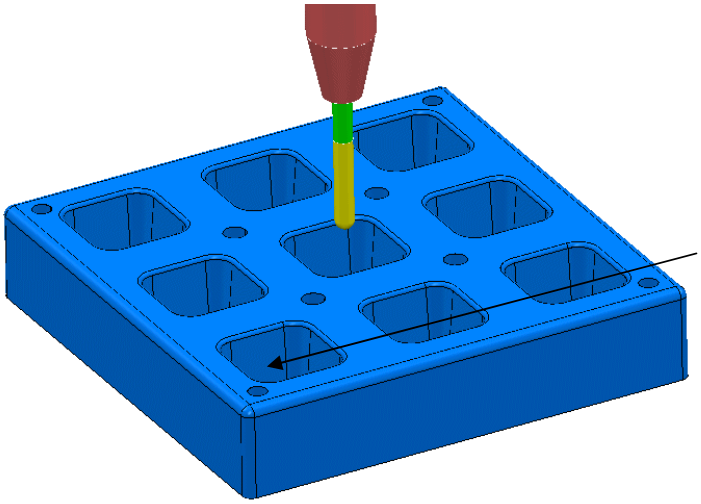
Transform – Rotate
The **Rotate** function allows the rotation of a selected toolpath by a user defined **Angle** around a selected X Y or Z, axis**.**



Transform – Mirror
This provides 3 options to **Mirror** a selected toolpath across the **XY**, **XZ** or **YZ** planes. This is useful for creating toolpaths for a right-handed component from a **toolpath** calculated for a left-handed component, or vice-versa. It will also be necessary to mirror the model and re-apply leads and links if the new **toolpath** is to be **verified** to check for **gouges**.
It is important to note that when a toolpath is mirrored the **cutting direction** effectively becomes **reversed**.

For the following example a single toolpath will be created and the **Transform - Move** option applied to produce toolpaths for the remaining cavities.

- **Delete All** and **Reset forms**.
- **From File – Open Project** browse the form and select the ***Project***:- **D:\users\training\PowerMILL_Data\Projects\EditToolpath_1**

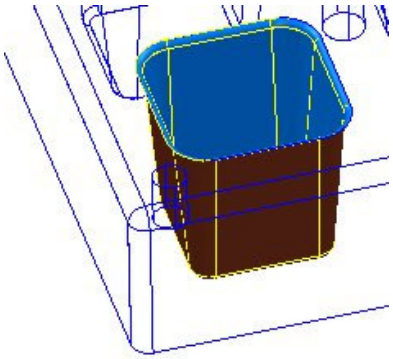


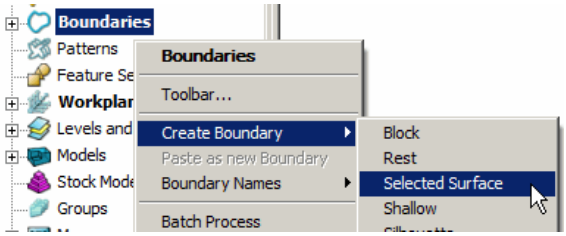
The **Project** contains a multi cavity and a **Dia 8 Ball Nosed** tool.

A **toolpath** will be created in the lower left cavity that will be limited to **a Selected Surface Boundary**.

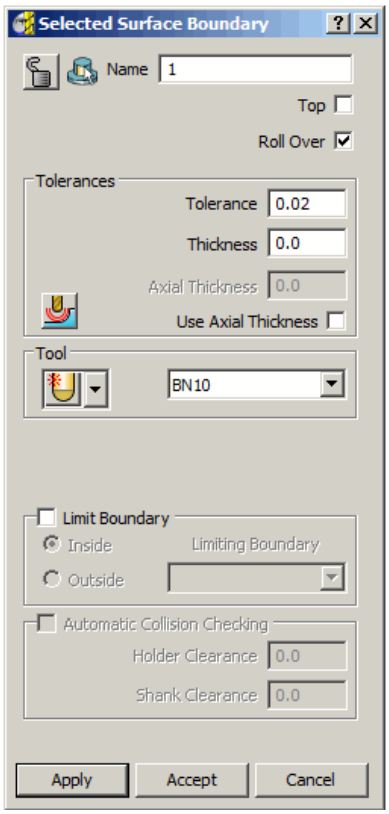
- **Save Project As**:-
- **D:\users\training\COURSEWORK\PowerMILL-Projects\Transform**
- Calculate a **Block** to **Min/Max Limits** and **Lock**  to the global coordinate system.

- **Select** the ***surfaces*** defining the bottom left pocket shown below.





- Right click over **Boundaries** in the **explorer** and select **Create Boundary - Selected Surfaces**.



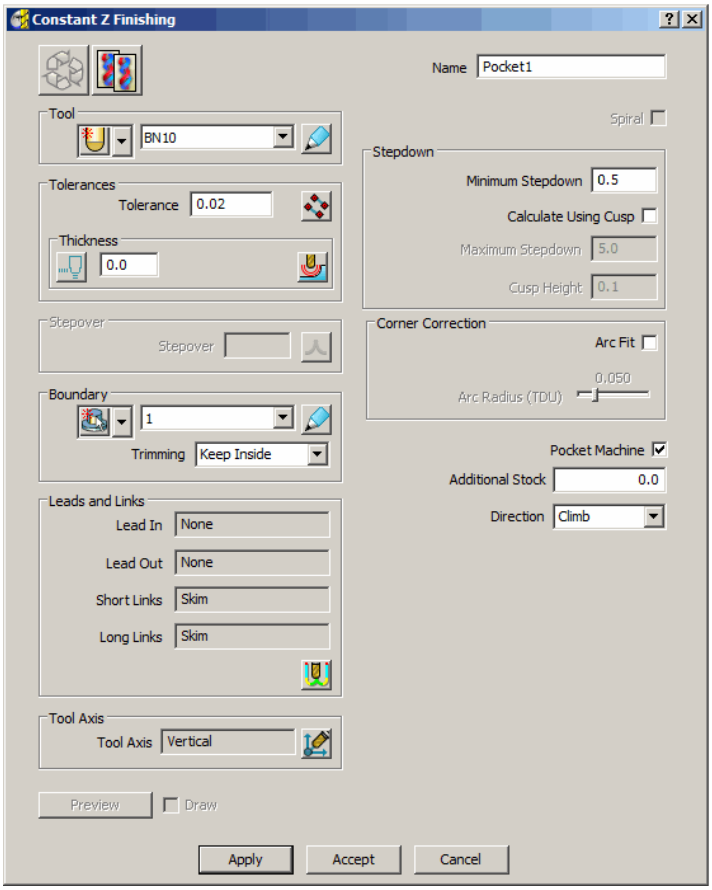
- Select the option **Roll Over**.

- Enter a **Tolerance 0.02**.

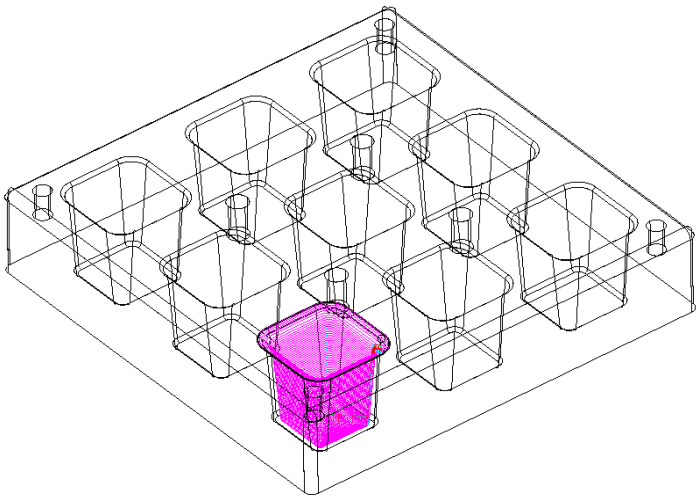
A **Selected Surfaces - Boundary** defines the limit where the selected tool and associated parameters would lose contact with the selected surfaces while simultaneously compensating for any adjacent unselected surfaces.
The option creates **Boundary** segments based on the total selection and not individual surfaces.

- **Apply** and **Accept** the form.

- Select a **Constant Z Finishing** strategy.



- Enter ***Name*** – **Pocket1**.
- Set ***Tolerance*** **0.02**.
- Set ***Direction*** – **Climb**.
- Set ***all Links*** to **Skim**.
- **Apply** and **Cancel** the form.



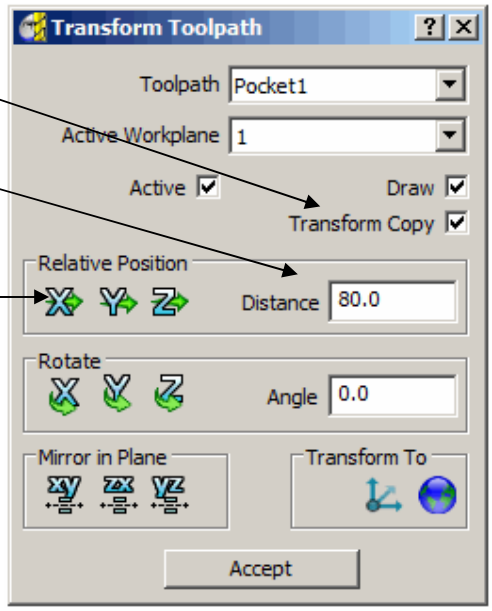
This toolpath will be transformed to create toolpaths for the remaining pockets.

- From the **PowerMILL explorer** – **Toolpaths** - **Pocket1** local menu select **Simulate from Start**, and click the **Play** *icon*  in the **Simulation** toolbar.

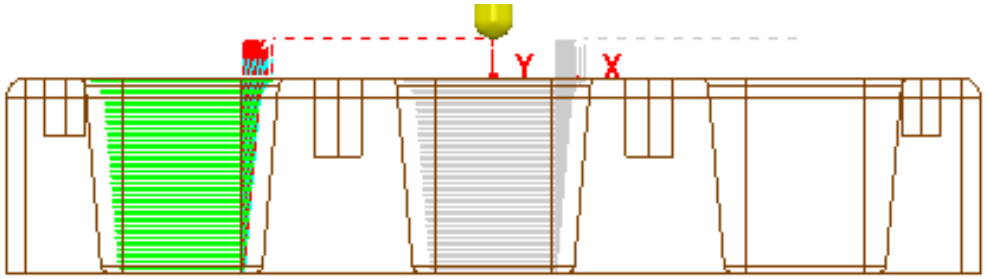
Note that the tool is travelling in a **climb mill** direction and the sequence of cutting is also from the top of the job to the bottom. It is advisable, before transforming toolpaths, to create a **Master** copy of the original.

- Right click over the toolpath **'Pocket1'** in the **explorer** and select **Edit - Copy** toolpath '**Pocket1**' and **Rename** the copy as '**Master**'.
- Right click over the toolpath **'Pocket1'** in the **explorer** and select **Edit – Transform** from the menu to open the **Transform Toolpath** form.

- **Tick** the **Transform Copy** option.
- Set **Relative Position** - **Distance 80**.
- Select move **Along X**.
- **Accept** the form.

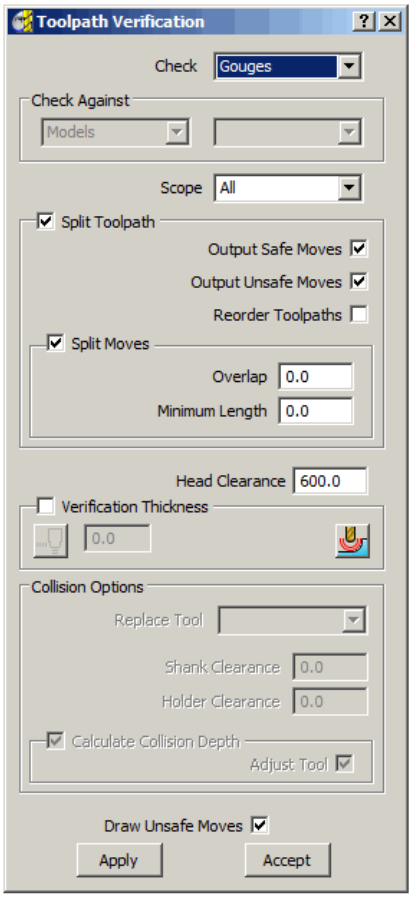


A copy of the ***original toolpath*** has been created and ***transformed*** along X by a distance of **80**, with the name **'Pocket1_1'**.



The yellow icon next to the toolpath indicates that it has not been gouge checked directly to the model.

- In the **explorer Activate** toolpath **Pocket1_1**.
- Right click over the toolpath **Pocket1_1** and select **Verify > Toolpath**.



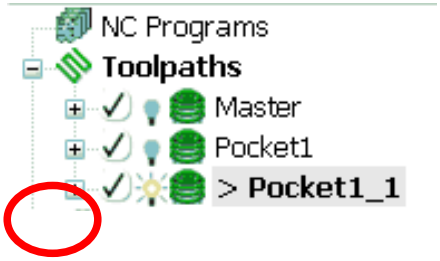
- Select **Gouges** for the ***Check*** option.

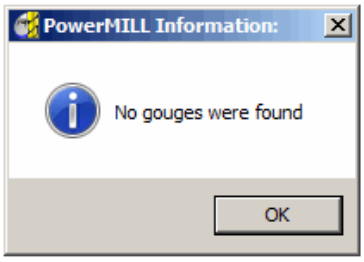
There are two options in the ***Check*** area, **Collisions** or **Gouges**.
The **Collisions** option will only perform a collision check if a **shank** and **holder** have been defined along with the **tool**.
The **Gouge** option performs a collision check on the basic tool (the **shank** and **holder** are not taken into account).

In the ***Scope*** area the default option **All** can be changed if required to focus on a particular type of feature on the toolpath from; **Cutting Moves**, **Connection Moves**, **Leads**, or **Links**.

- **Apply** the form.

An information box appears informing the user, that in this instance, no gouges have been found. The icon also updates indicating the toolpath is gouge checked.

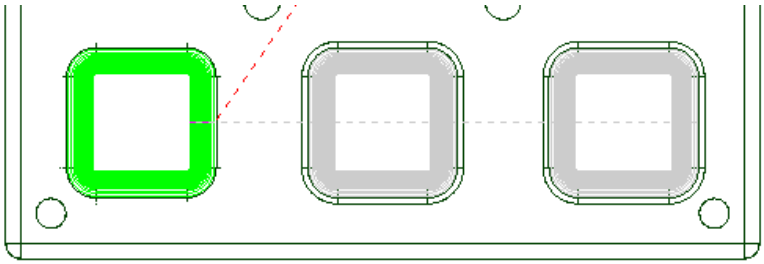




- **OK** the information window and **Accept** the toolpath verification form.
- Right click over the toolpath **'Pocket1_1'** in the **explorer** and select **Edit – Transform** from the menu to open the **Transform Toolpath** form**.**

- Using the previous settings with **Transform Copy** ticked, enter a **Distance 80** followed by move **Along X**.

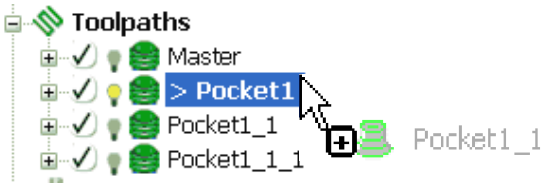
- **Activate** the new toolpath, **'Pocket1_1_1'**, and check for gouges.



The two new toolpaths can be **Appended** to the original toolpath if required.

Note – It is important to **Append** toolpaths in the order that they are required to run, and toolpaths can only be **Appended** if they share the same **Tool** and **Tool Axis**.

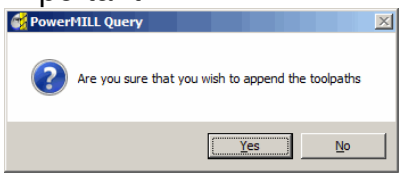
- **Activate** toolpath **Pocket1** and undraw (switch off lightbulbs) on toolpaths **Pocket1_1** and **Pocket1_1_1**.



- To **Append** click with the **left** mouse keeping it held down over toolpath **Pocket1_1**, then press the **Ctrl key** also keeping it held down while dragging the mouse over toolpath **Pocket1**.

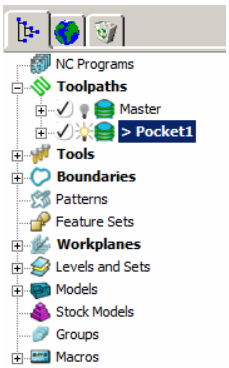
A plus sign (as above) will appear if the toolpaths can be **Appended**.

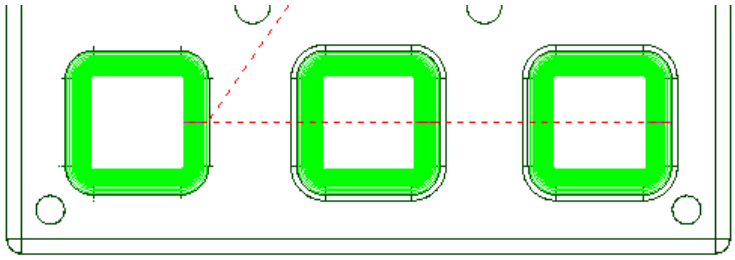
- Release the **left mouse** button first followed by the **Ctrl** key (The order is important.



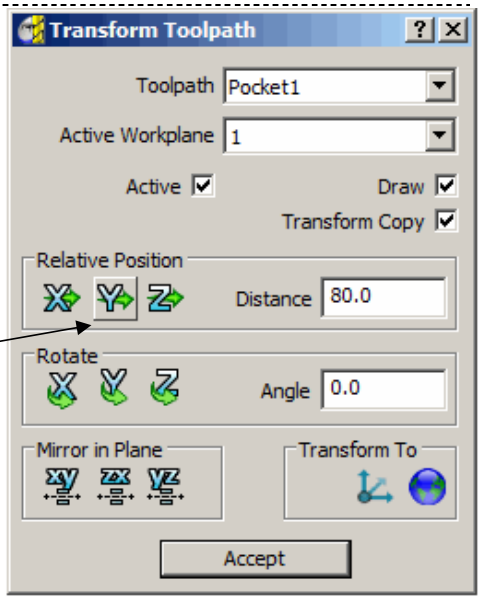
This form will appear requesting confirmation for the **Append** action.

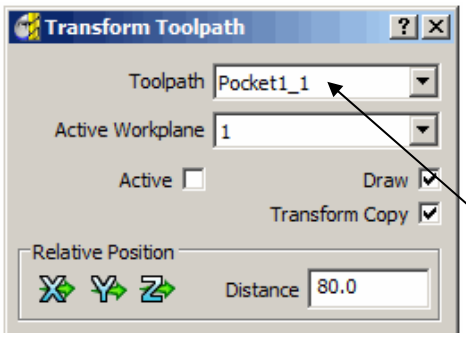
- Select **Yes** from the above form.
- **Append** toolpath '**Pocket1_1_1**' to the first toolpath '**Pocket1**'.
- **Delete** toolpaths '**Pocket1_1_1**' and '**Pocket1_1**'.
- Draw (light bulb) toolpath **Pocket1** to view all three toolpaths.



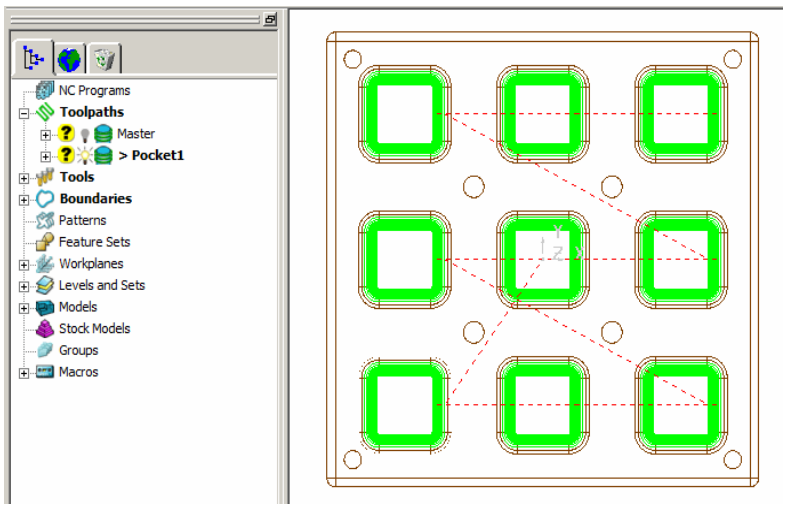


- Right click over the toolpath **Pocket1** in the **explorer** and select **Edit - Transform** from the menu to open the **Transform Toolpath** form.
- **Tick** the **Transform Copy** option.
- Set a **Relative Position** of **Distance 80**.
- Select move **Along Y**.





- Select the newly transformed ***toolpath*** **Pocket1_1** in the **Toolpath** drop down menu and move a **Transform Copy** by a **Distance** of **80** along **Y**.
- **Accept** the form.



- **Append** these new toolpaths into the original **toolpath** (**Pocket1**) as before and **Verify** to gouge check.
- **Delete** toolpaths '**Pocket1_1_1**' and '**Pocket1_1**'.

- **Save Project As** :-

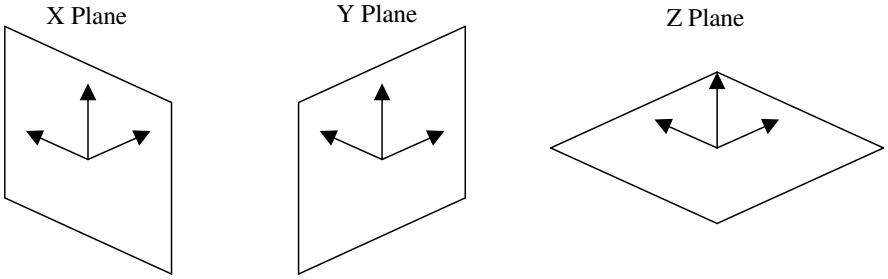
D:\users\training\COURSEWORK\PowerMILL-Projects\Transform

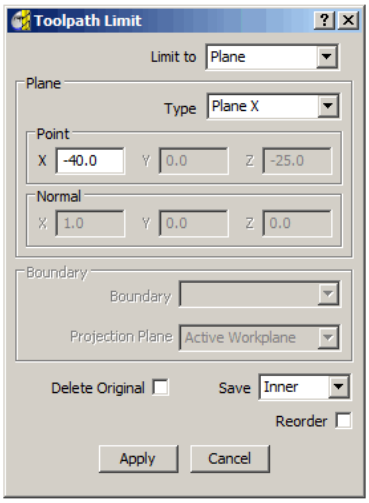
Edit > Limit...

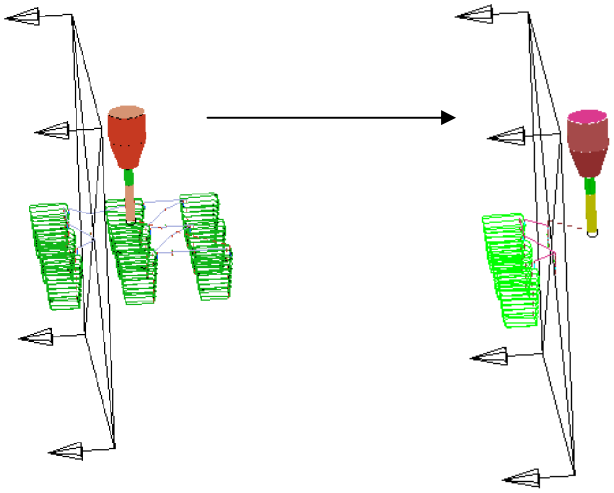
Edit - Limit provides a series of options to retrospectively trim a **toolpath** to a **Plane**, **Polygon** or **Boundary**.

Edit - Limiting - *to a* Plane

Edit - Limit - *To a* Plane allows the user to select a plane at specified distance across the X, Y, or Z Axis. Alternatively there is an **Arbitrary** option which allows the user to specify an origin point aligned with a direction vector in terms of X, Y, Z for the **Limiting** plane.

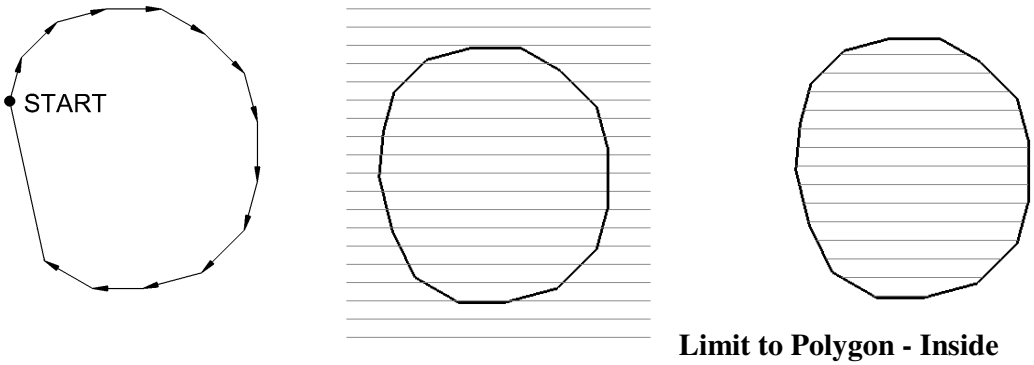






Edit - Limiting - *to a* Polygon

Polygons with any number of sides can be sketched using the left mouse button. This allows complex areas to be defined, with the option to save the **Inside**, **Outside**, or **Both** sides of the **polygon.** N.B. before making a polygon ensure that the snap filter is set to pick **Anywhere**.

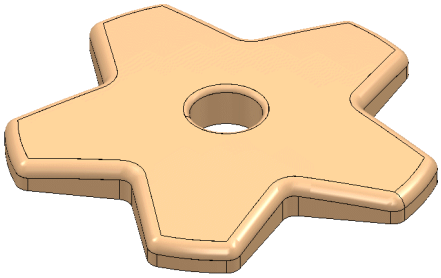


Limit to Polygon - Inside

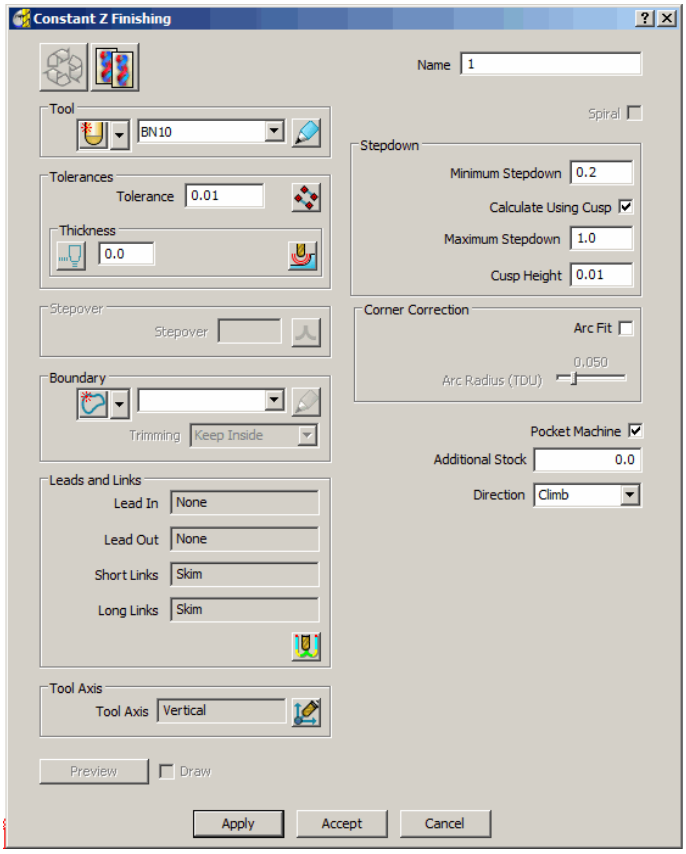
Edit - Limit - *to a* Boundary

This provides the option to limit the toolpath to areas defined by an existing **Boundary**. Again the option exists to save the **Inside**, **Outside**, or **Both** toolpath areas.

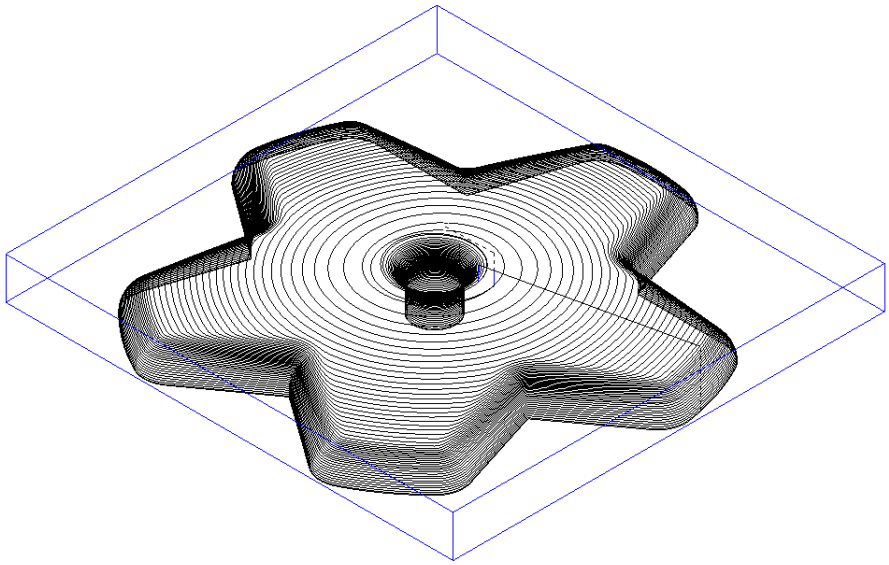
- **Delete All** and **Reset Forms**.
- **Open** the **model**:- **D:\users\training\PowerMILL_Data\Models\handle.dgk**.



- **Calculate** a **Block** to the **Max/Min Limits** of the **model** and **lock** the **Z Min** and **Z Max**, then input **expansion 10** before applying **Calculate** again.
- Define a **Dia 10mm Ball Nose** named **BN10**.
- In the **Rapid Move Heights Form** click **Reset to Safe Heights**.
- Set the **Start Point** to **Block Centre Safe** and **End Point** to **Last Point Safe**.
- Select a **Constant Z Finishing** strategy and enter data exactly as shown below before selecting **Apply**.

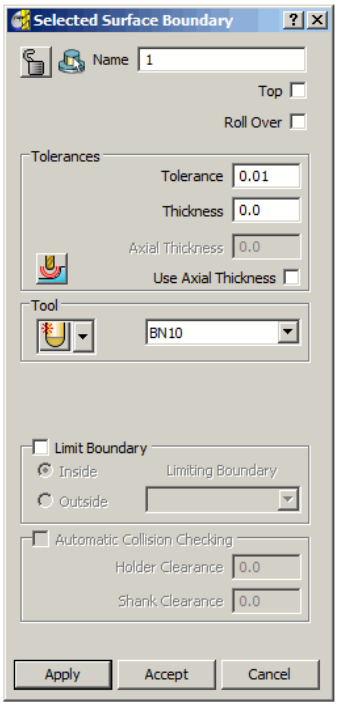


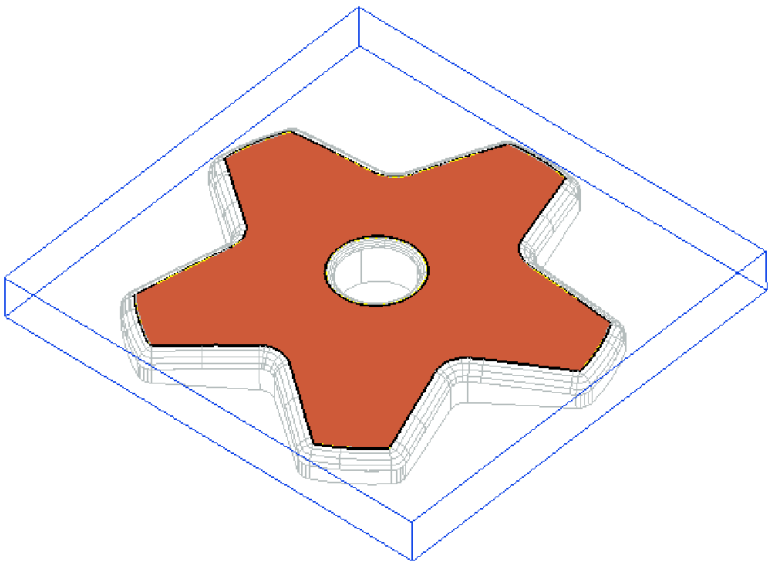
The **Stepdown** will be varied within the **Maximum** and **Minimum** values in an attempt to achieve the specified **Cusp Height**. This enhancement will still not be effective on flat or nearly flat areas.



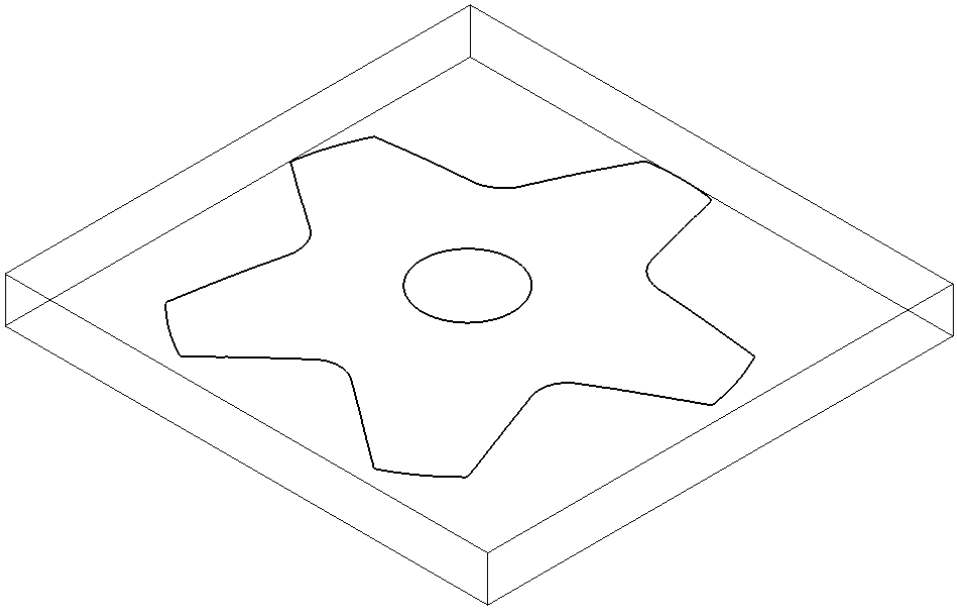
As is clear in the above illustration the **Constant Z** finishing strategy created is not suitable for shallow areas of the component. As a result a **Boundary** will be created and the toolpath will be retrospectively limited to the steep areas.

- Select the main upper surface on the model, right click on **Boundaries** in the **explorer** and from the local menu click the option **Selected Surface** to open the following form.

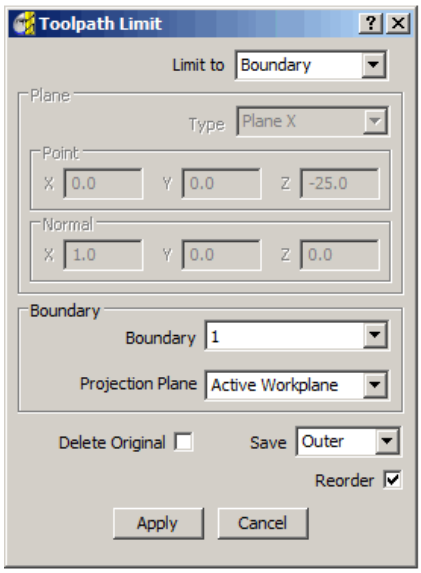


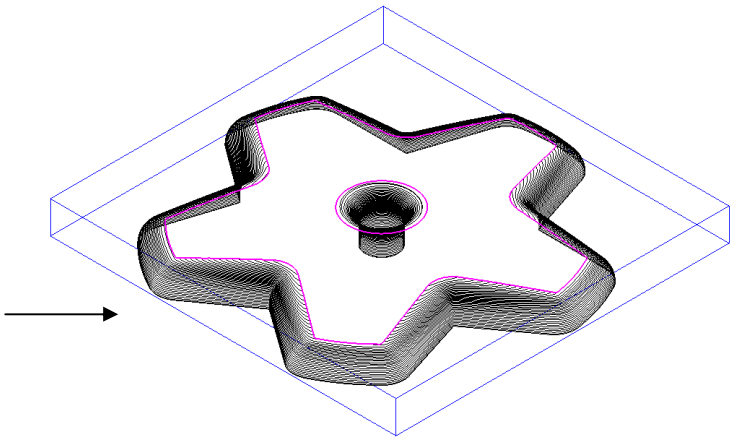


- Enter data exactly as shown above before selecting **Apply** and then **Accept** the form.



- In the explorer **Right mouse click** over the **Active** toolpath and in the local menu select **Edit - Limit - *limit to* Boundary**.
- Enter data into the form exactly as shown below before selecting **Apply** to create a copy of the toolpath limited to outside the **Boundary**.





The **toolpath** has been retrospectively ***limited*** to the outside of the **Boundary**.

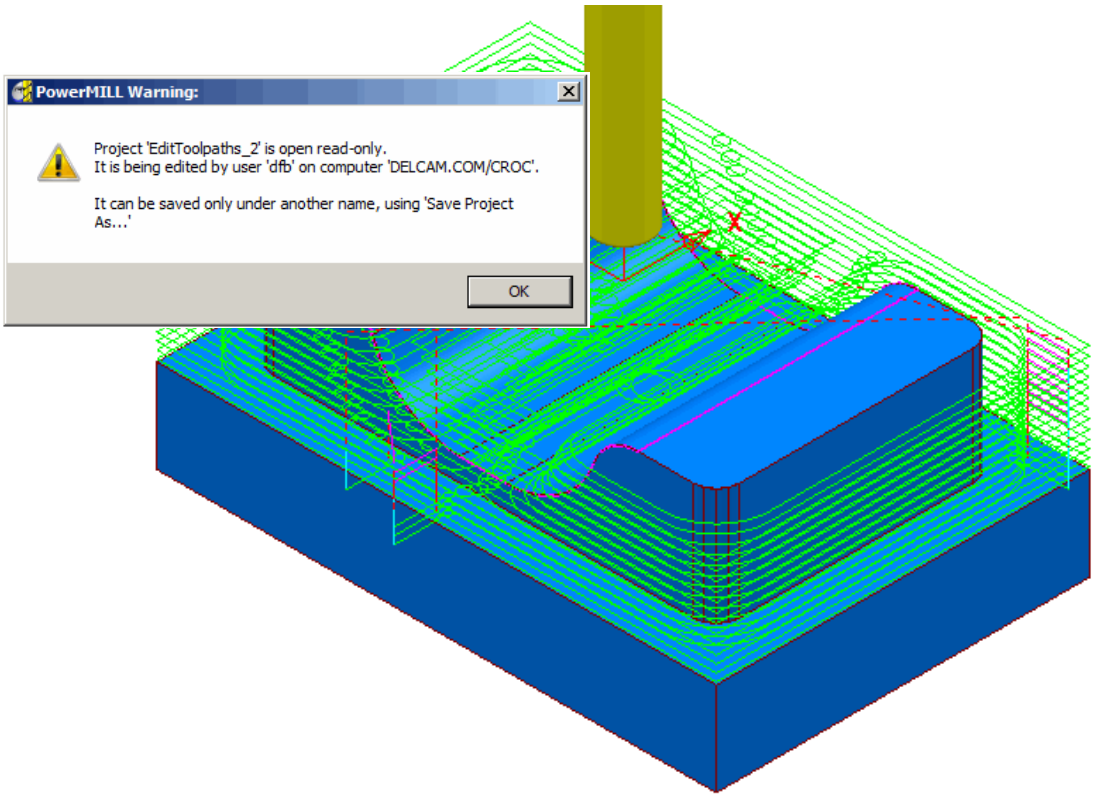
Delete Original

The **Delete Original** flag, if ticked causes the deletion of the ***original toolpath*** once the new toolpath has been created.

Changing the Order and Direction within Toolpaths

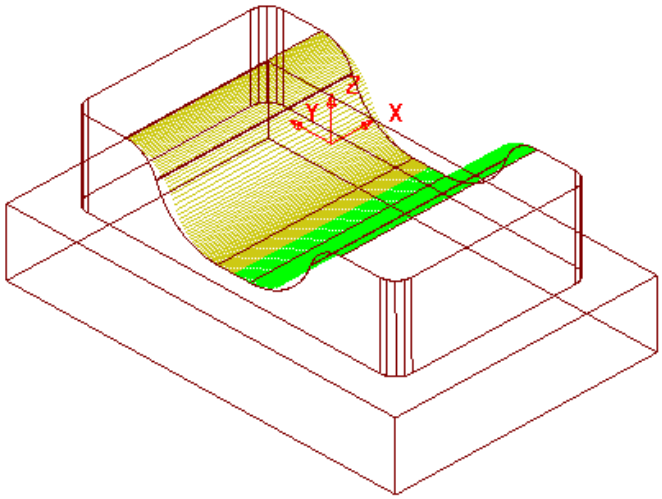
For toolpaths containing internal link moves, the **order** and **direction** of the tool tracks can be changed. For example, if a machining sequence starts at the bottom of part, progressing upwards, reversing the order will change the tool track sequence to start at the top of the part and progress 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.

- **Delete All** and **Reset forms**.
- From **File – Open Project** select the read-only ***Project***:- **D:\users\training\PowerMILL_Data\Projects\EditToolpath_2**



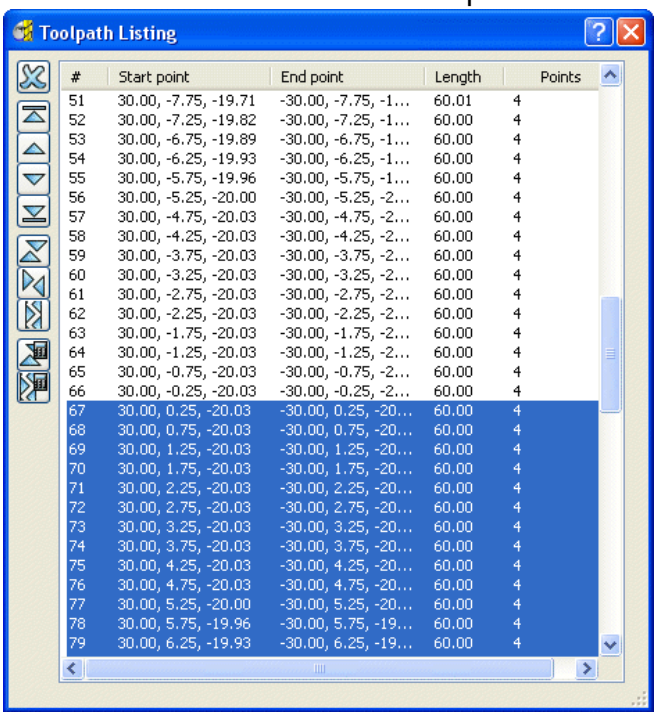
- Click **OK** to close the ***PowerMILL Warning*** form.
- In **File - Save Project As**:- **D:\users\training\COURSEWORK\PowerMILL-Projects\PunchInsert_1**

- In the **explorer - Toolpaths** area right mouse click over the **Active** toolpath **bn12-finish-a1** and from the menu select **Simulate from Start**.



The Tool **Climb Mills** across the form stepping downwards to the base before stepping up the other side. The toolpath will be **Reordered** so that the tool **Climb Mills** down both sides of the form.

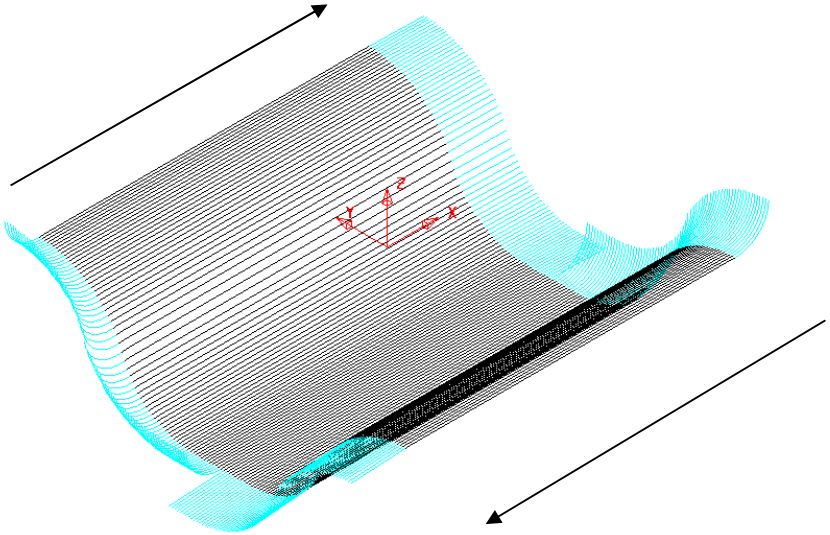
- Using the left mouse key, select all **tool tracks** on **bn12-finish-a1** beyond the centre of the form (Light Grey in the above illustration).
- In the **explorer** right mouse click over the ***Active*** toolpath **bn12-finish-a1** and select **Edit - Reorder** to open the following form.



All existing **tool tracks** (identified by number) are listed in the current order of machining.

Note: The selected **tool tracks** are highlighted (blue) in the form.

- Click on the **Reverse Order** Icon  to reverse the sequence of the selected tool tracks.
- Click on the **Reverse Direction** Icon  to reverse the direction of each selected tool track.



The **Order** and **Direction** of the toolpath has been edited so that it Climb Mills in 2 groups of sequential tool tracks from the outer edges of the form towards the centre of the base.

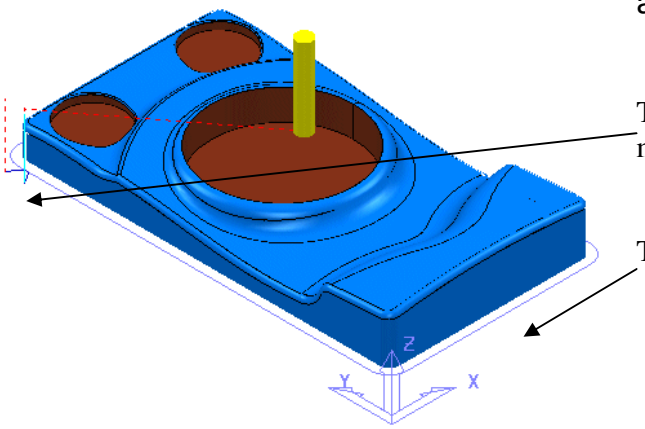
Note: **Vertical Arc** ***Lead In*** and **Extended Move** ***Lead Out*** have been applied to the above toolpath to identify the directional differences.

- **Save** the **Project**:- ***D:\users\training\COURSEWORK\powerMILL-Projects\PunchInsert_1***

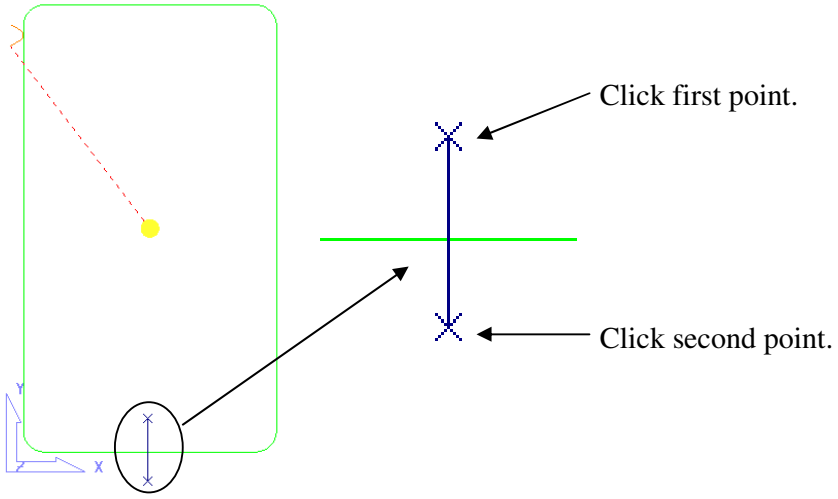
Moving the Start Points

The start points on the tool tracks can be moved to provide a more suitable position for applying leads. This is achieved by defining a line that crosses the new start positions on the toolpath.

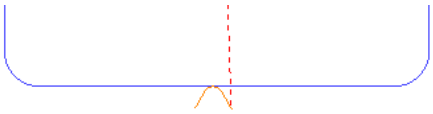
- **Delete all** and **Reset forms**.
- **Open** the **Project:-** **D:\users\training\PowerMILL_Data\Projects\limiting-example** and select an **ISO1** view.



- Select a **View Down Z**.
- Right click over toolpath **1** select **Edit > Move Start Points** and **left click two points** as shown.



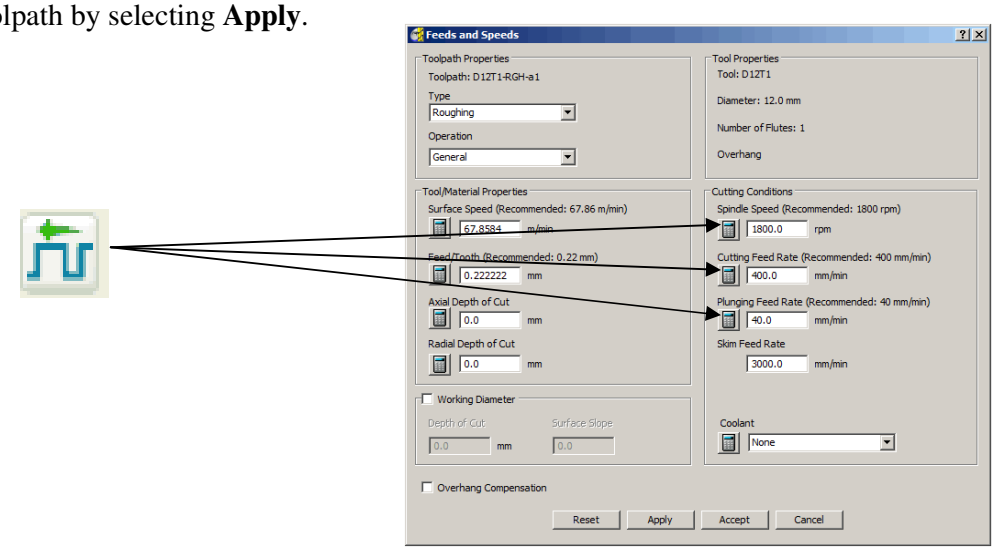
- Press **Return** on the keyboard to accept.



This has moved the start position.

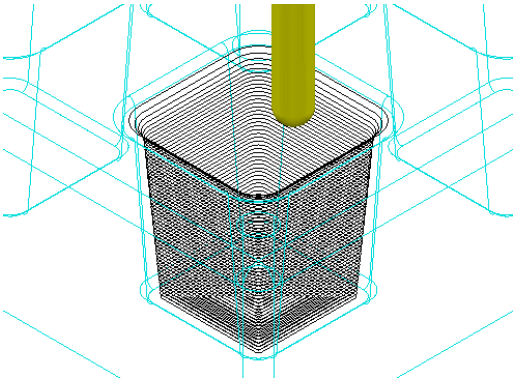
Feeds and Speeds

The contents of the **Feed Rates** form can be modified and retrospectively applied to the **Active** toolpath by selecting **Apply**.



Check the feed rate values assigned to the named toolpath in the **explorer** to verify that they have changed.

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

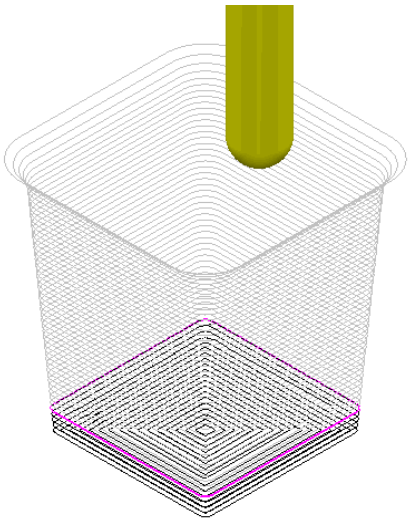


- Click **OK** to close the ***PowerMILL (Read-only) Warning*** form.
- In **File - Save Project As**:-

D:\users\training\COURSEWORK\PowerMILL-Projects\ExtraFeedrates-example

- Undraw the ***Model***, ***Block***, ***Workplan***e, and ***Tool*** to display just the **Toolpath** and 3 **Boundaries**.
- **Activate** both the **Toolpath** and **Boundary 2**.
- Right mouse click on the **Toolpath** and in the ***local menu*** select **Edit – Update Feed Rate Inside Boundary**.

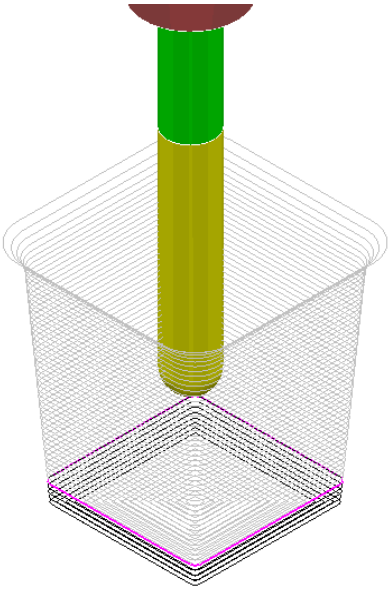
- Input the value **50** and click on the ***Green tick*** to apply the ***percentage*** difference from the nominal **Feed Rate** within the ***Active*** **Boundary**.



The **toolpath** is split into two different coloured areas that are allocated a separate **Feed Rate** as a **percentage** of the original. In the **imported Project** the **Cutting Feedrate** assigned to the **toolpath** is **450 mm/min**.

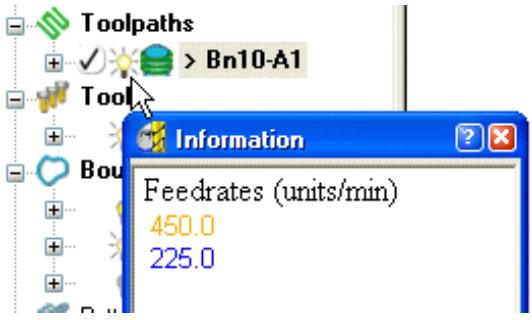
- **Activate** both the **Toolpath** and **Boundary 3**
- Right mouse click on the **Toolpath** and in the ***local menu*** select **Edit – Update Feed Rate Inside Boundary**.

- Input the value **100** and click on the ***Green tick*** to apply the ***percentage*** difference from the nominal **Feed Rate** within the ***Active*** **Boundary**.



The central part of the **Toolpath** area, clear of the side wall has been changed back to the nominal **Feed Rate** value by inputting **100** (%) in the form.

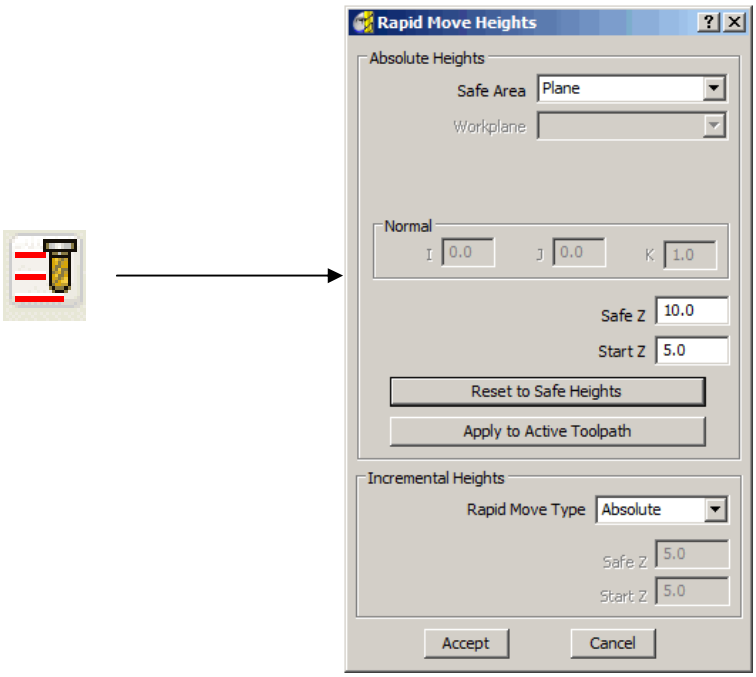
- To see the actual **Feed Rate** values assigned to the ***coloured areas*** of the toolpath ***left mouse click*** - **twice** on the ***Light bulb*** adjacent to the **Toolpath** in the **PowerMILL explorer**.



In this case the nominal **Feed Rate** is **450 mm/min** and the modified section near the intersection of the ***Base*** and ***Sidewall*** of the pocket have been reduced by **50%** to **225 mm/min**. The colour of the values displayed in the **Information** form is the same as the new colours specifying the corresponding areas of the **toolpath**.

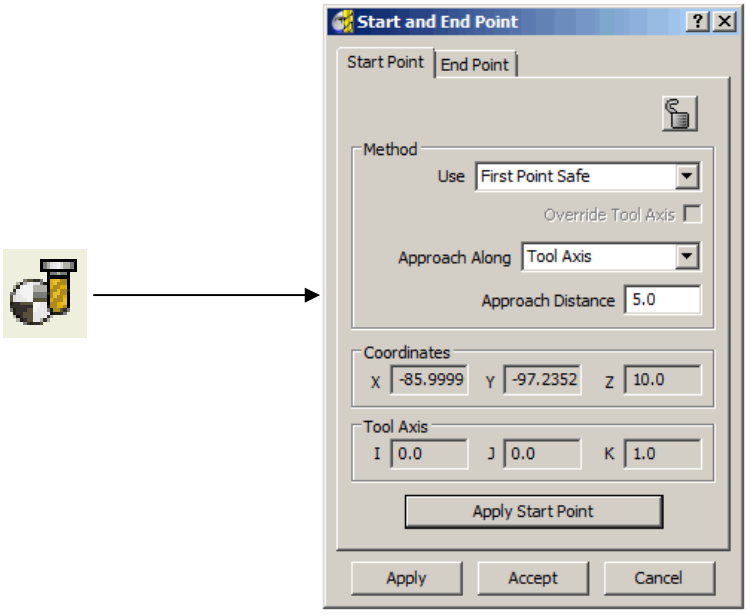
Safe Z and Start Z Positions

The contents of the **Rapid Move Heights** can be modified and retrospectively applied to the **Active** toolpath by selecting **Apply to Active Toolpath**.



Editing Tool Start and End Point Positions

The contents of the **Start and End Point** form can be modified and retrospectively applied to the **Active** toolpath by selecting **Apply Start / End Point**.



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. As a result it is not possible to ***colour shade*** a **Feature**. A ***2D Component*** is built up from a **Feature Set** consisting of one or more related **Features**. Milling will occur 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 (2D machining strategies)

On completion of a **Feature Set**, the **2D Area Clearance** strategies are applied to create all the **2D machining** including 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

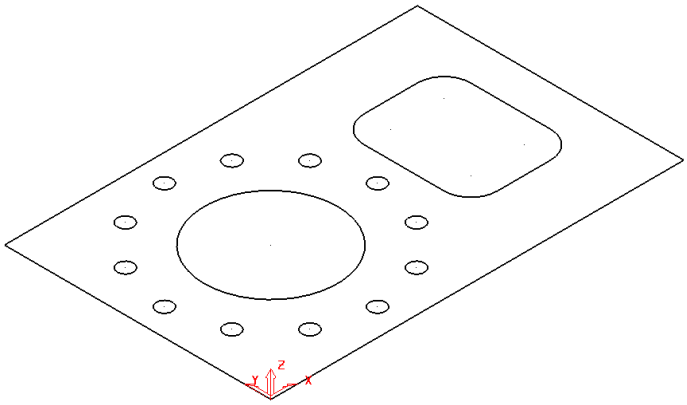
There are six different types of **Feature** which respond very specifically to the 2D machining options:-

1. **Pocket**, defines the area inside a profile, therefore the tool will only machine the inside the **Feature**.
2. **Slot,** is curve based defining the track of the tool (with or without cutter compensation).
3. **Boss**, is an upstand. The tool will only machine around the outside of the **Feature**.
4. **Hole**, this is specifically used with **Drilling** strategies and is 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 data containing pairs of circles/curves or cylindrical surface data can be imported into **PowerMILL** to be directly defined as **Hole** features, thereby removing the need to manually input dimensions (eliminating a possible source of human error).

Creating Features from Pattern segments

- Select **File ➔Delete All** and **Tools ➔ Reset forms**.
- **Open** the ***Project***:-
- **D:\users\training\PowerMILL_Data\Projects\2D-Drawing**.

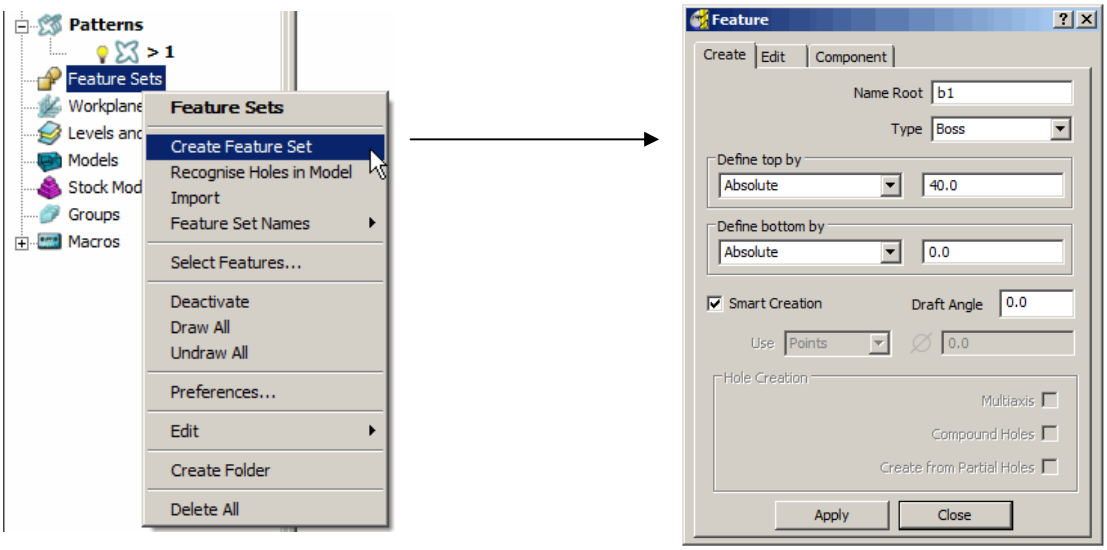


The imported **Project** contains a **Pattern** defining **2D geometry** to be used to create a **Feature Set**. The **Project** is **Locked** to prevent it from being altered hence the first step is to **Save As** a separate **Project** locally with a different name.

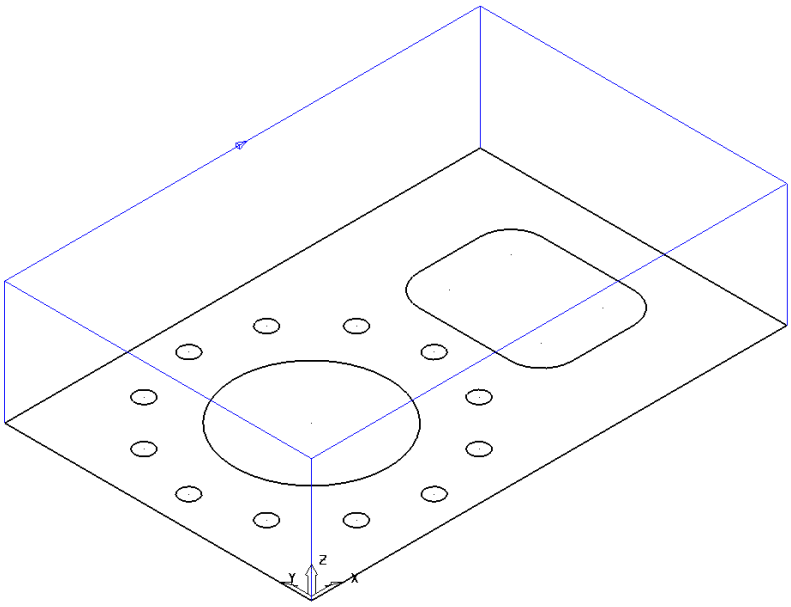
- Select **File** - **Save Project As**:-

D:\users\training\COURSEWORK\PowerMILL-Projects\2DPatternExample

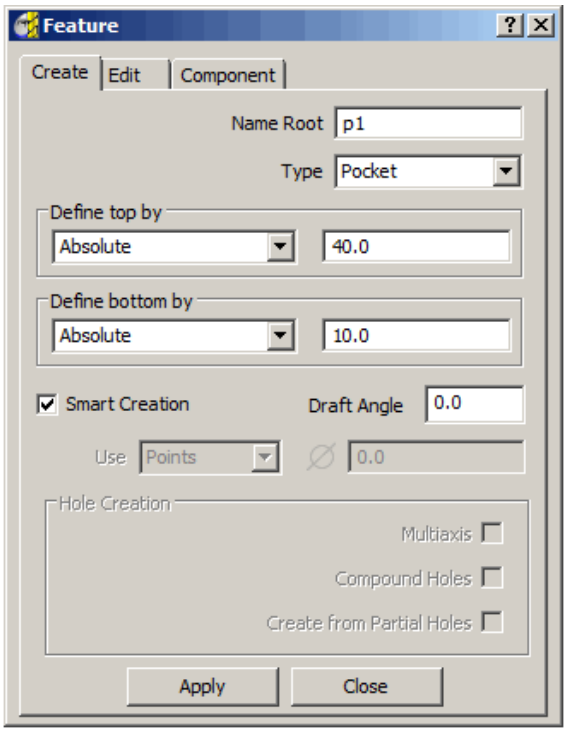
- In the **explorer** right mouse click on **Feature Sets** to access the local Pull Down menu and select **Create Feature Set**.

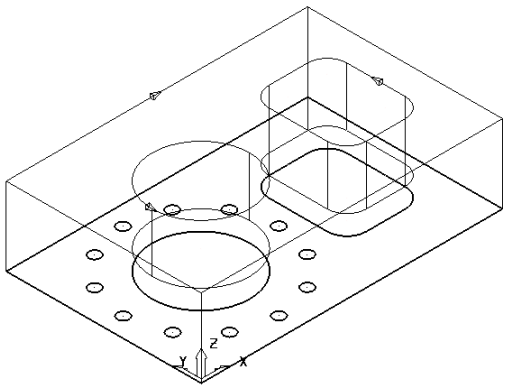


- A new (empty) **Feature Set** - **1** will appear in the **explorer** and the **Feature Form** will open ready to build the 2.5D model.
- **Select** the larger, rectangular **Pattern Segment** and enter values exactly as shown in the above right **Feature Form** before selecting **Apply** (Do not close the form).



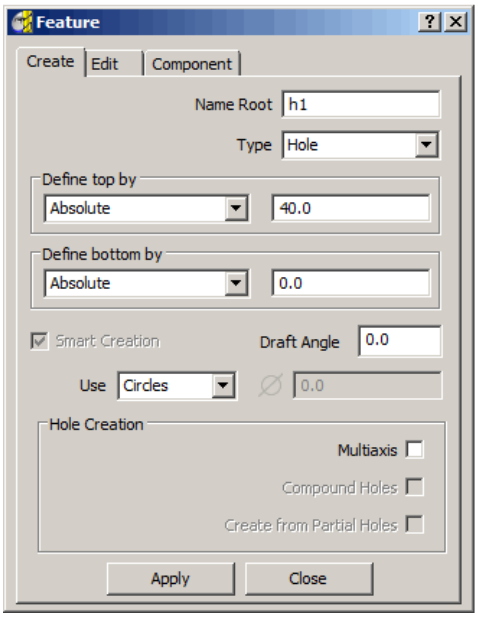
The first **Feature** is the main component body defined as a **Boss** (as shown above).

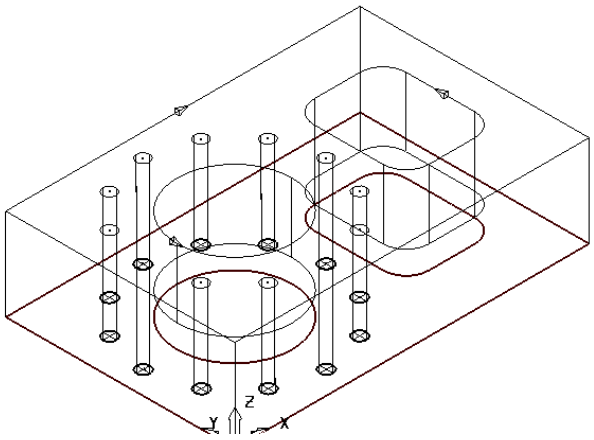




- **Select** both, the filleted rectangle and the large circle **Pattern Segments** and enter values exactly as shown in the above left **Feature Form** before selecting **Apply** (Do not close the form).

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**.

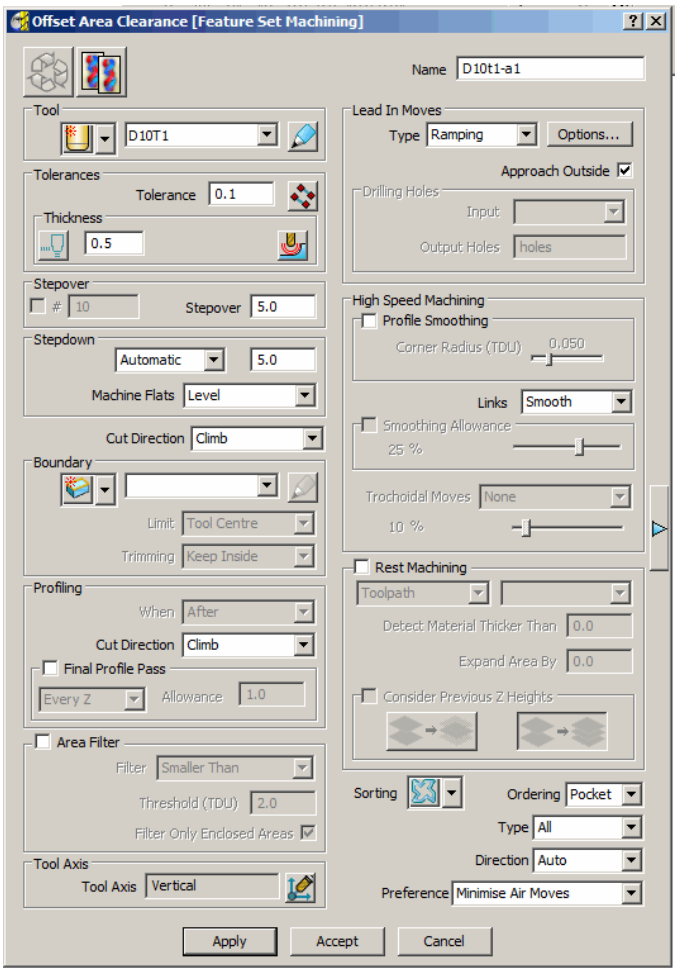
The contents of the **Feature Set** (**1**) should be similar to the illustration on the left (Note: The Author has opted to use a naming system where a **Boss** is prefixed with a **b**, a **Pocket** with a **p**, and a **Hole** with an **h**).

The **Feature Set** is now complete and ready for the creation of suitable **2.5D Machining Strategies**.

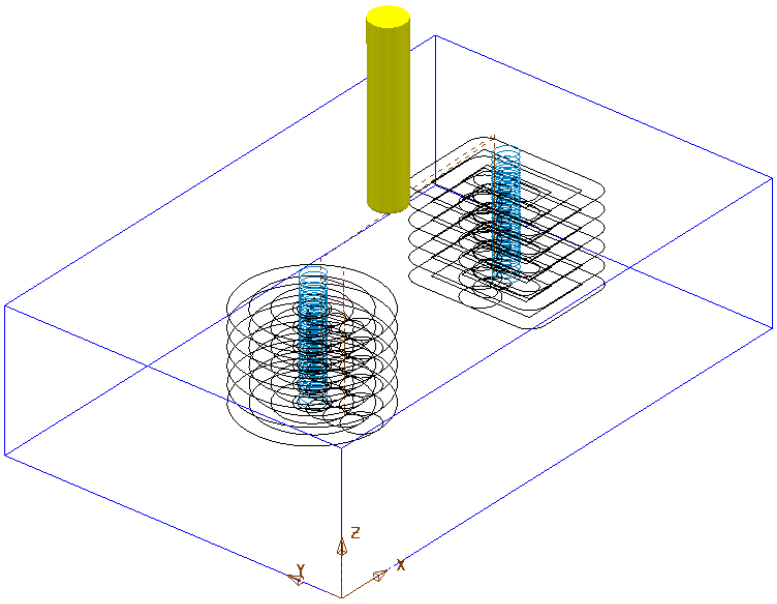
2D Feature Set Machining

- Define a **Dia 10 - tiprad 1** - **Length 50** tool named **D10T1**.
- In the **Material Block** form  ***Type*** - **Feature** before selecting **Calculate**.
- In the **Rapid Move Heights** form  select **Reset to Safe Heights**.
- Open the **Toolpath Strategies** form  and select **2.5D Area Clearance**.

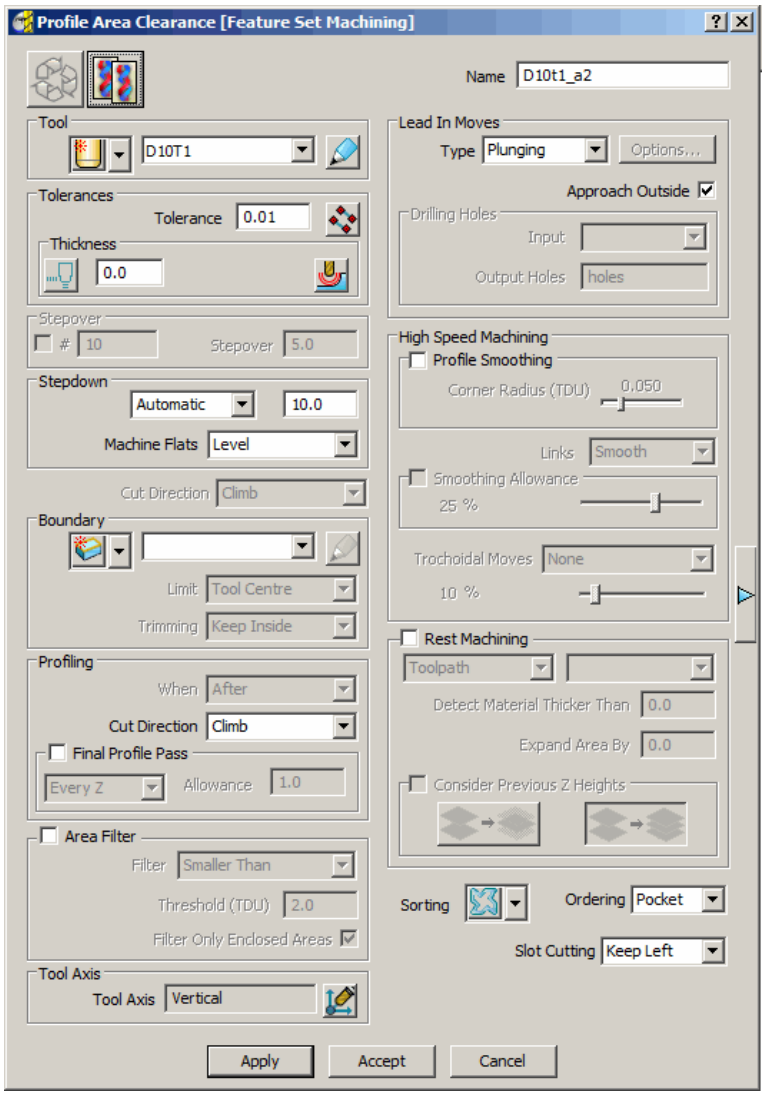
- Select **Offset AreaClear Feature Set** and input data exactly as shown in the following form.



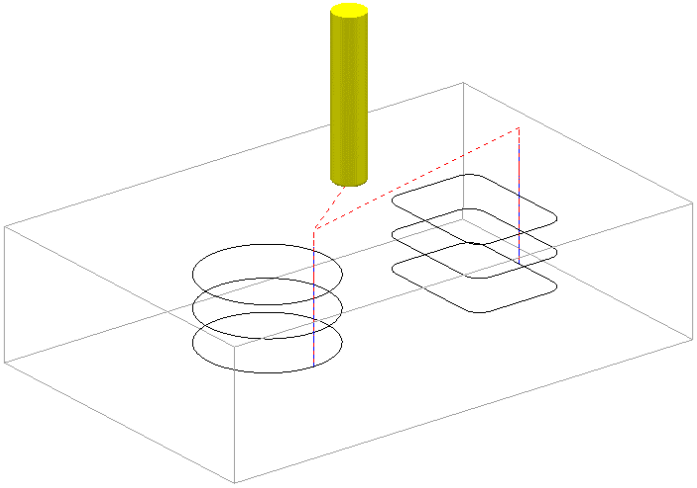
- **Apply** and after processing **Cancel** the form.



- Select **Profile AreaClear Feature Set** and input data exactly as shown in the following form.

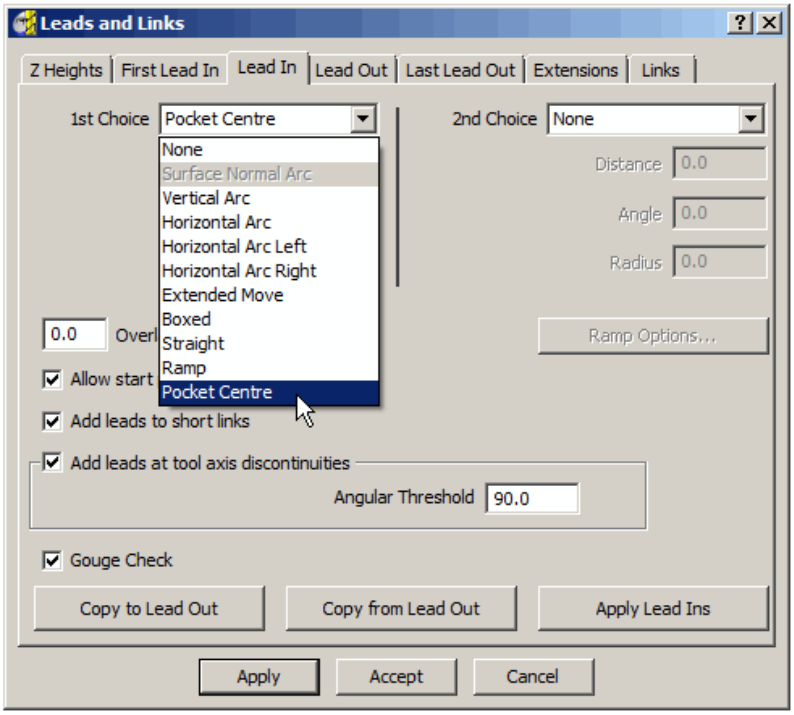


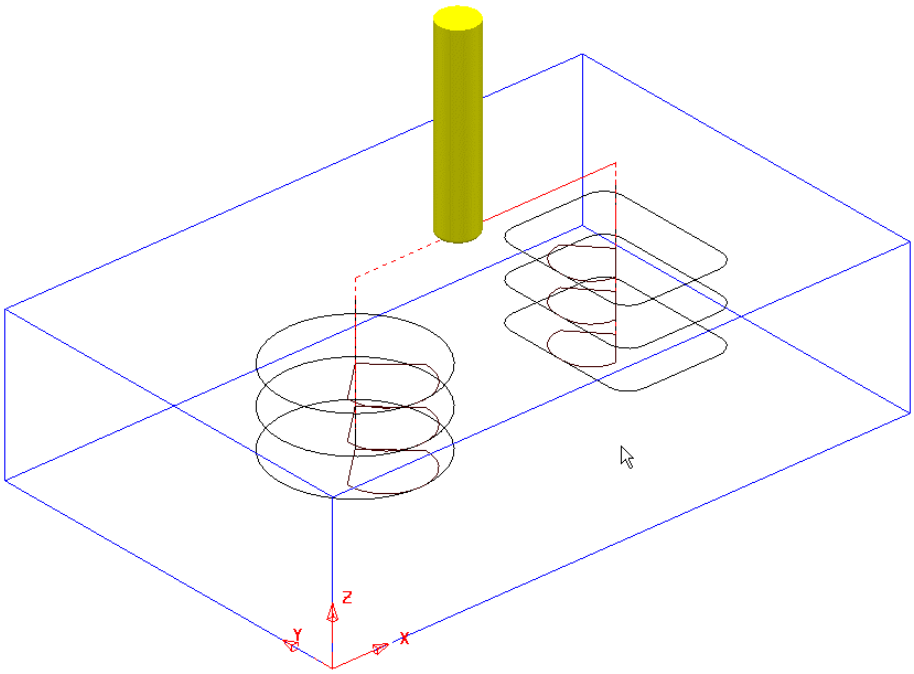
- **Apply** and after processing **Cancel** the form.



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**.

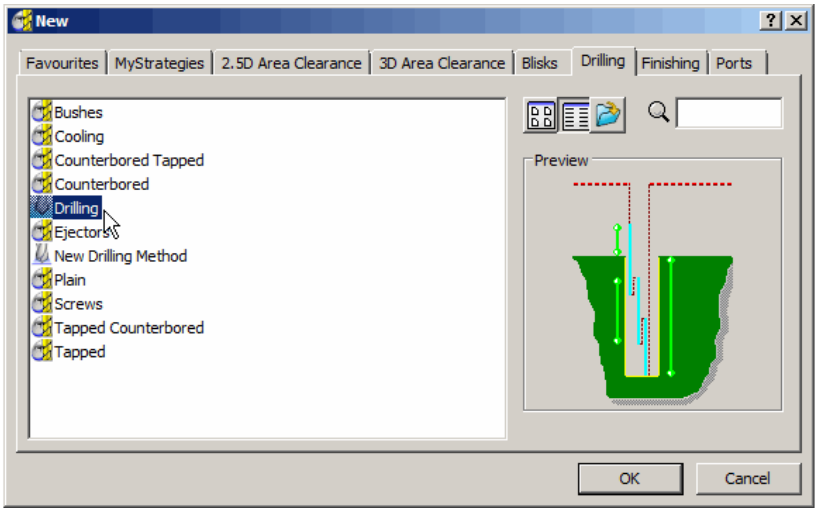




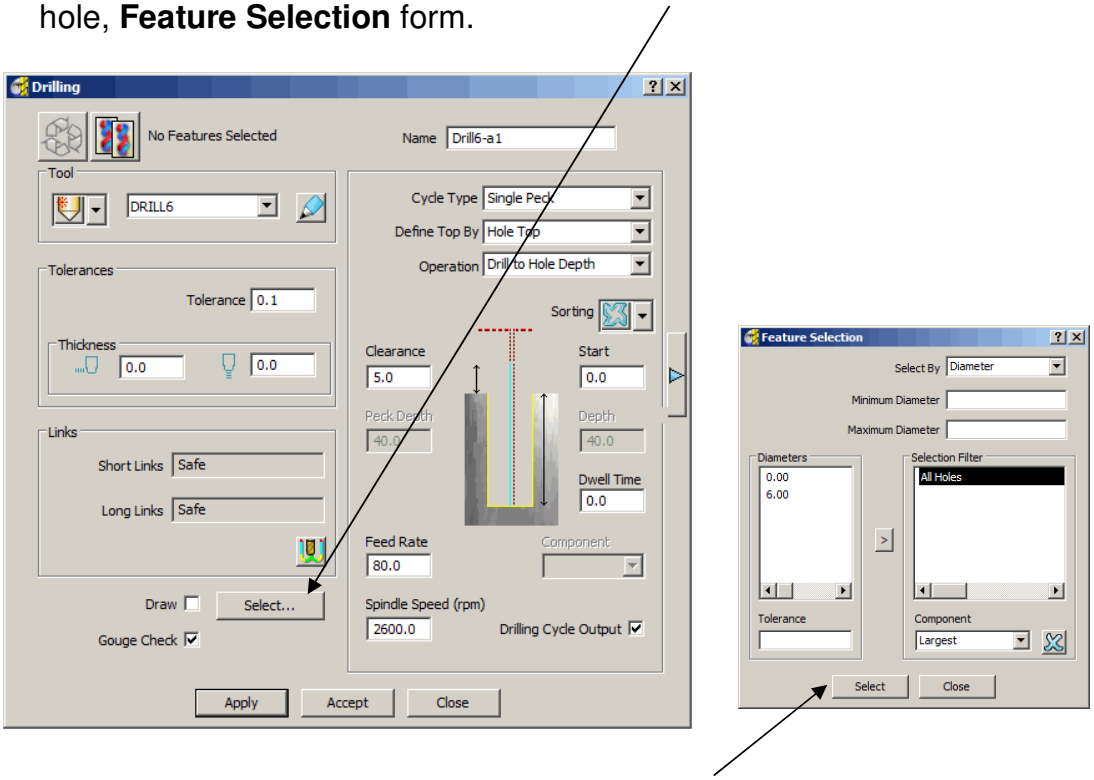
The tool is feeding down, central to the 2 pockets and **leads** on and off with a horizontal arc move.

- Define a **Dia 6 Drill** - **Length 50** and named **Drill6**.

- Open the **Toolpath Strategies** form  and select **Drilling**.
- In the form select **Drilling** as shown in the following form.



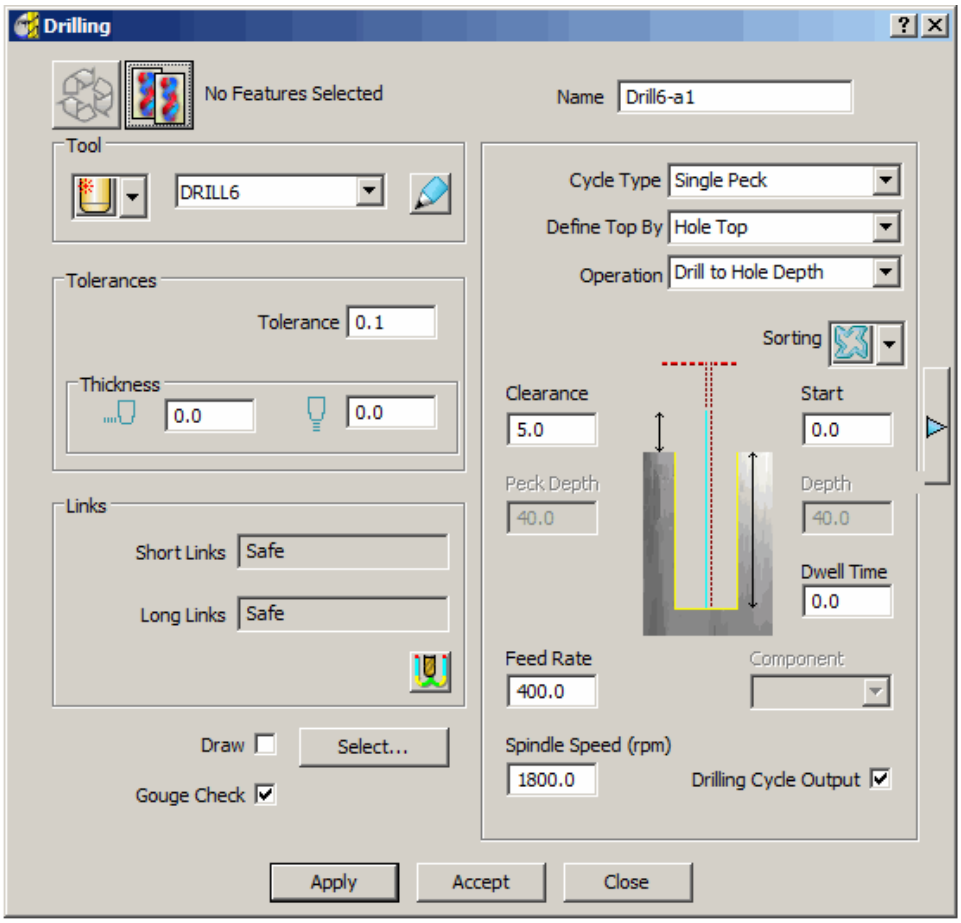
- The **Drilling** form will open in which the **Select** tab is clicked to open the hole, **Feature Selection** form.



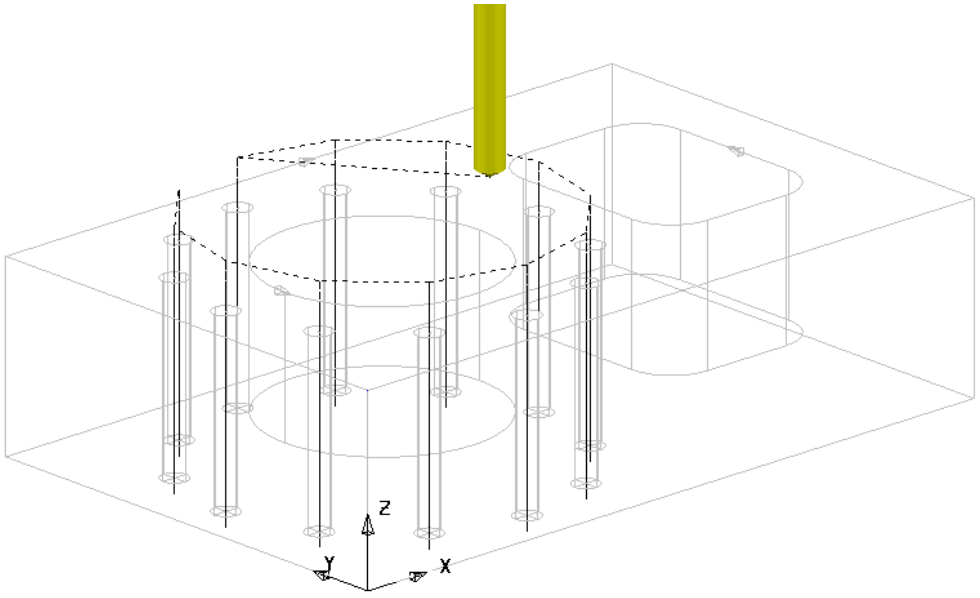
- In the hole, **Feature Selection** form the **Select** tab is clicked to include all defined holes in the **Feature Set**.

The '**No Features Selected**' statement on the **Drilling** form will disappear.

- Enter data into the **Drilling** form exactly as shown below before selecting **Apply**.

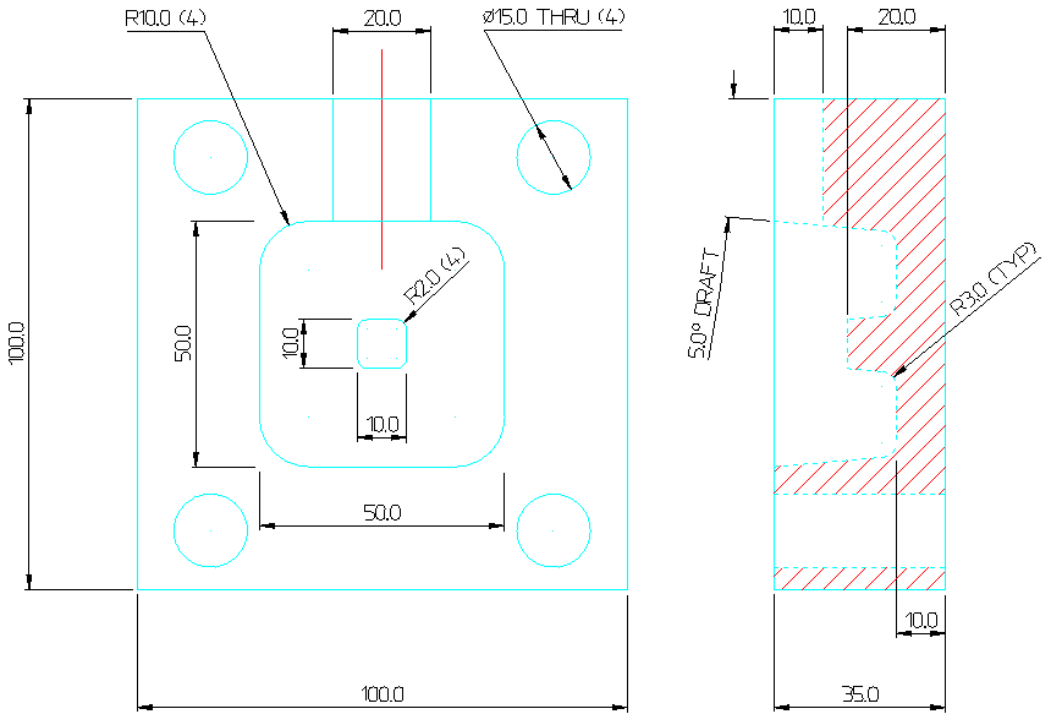


- Close the **Drilling** and **Hole Selection** forms.



2D Machining Exercise

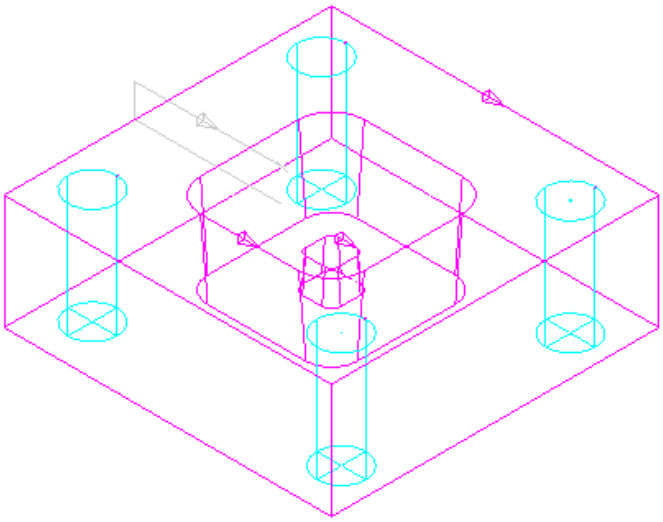
For this exercise the following ***2D drawing*** will be used.



- **Delete All** and **Reset forms**
- **Import** the ***wireframe*** **model** (as shown above):-

D:\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**.

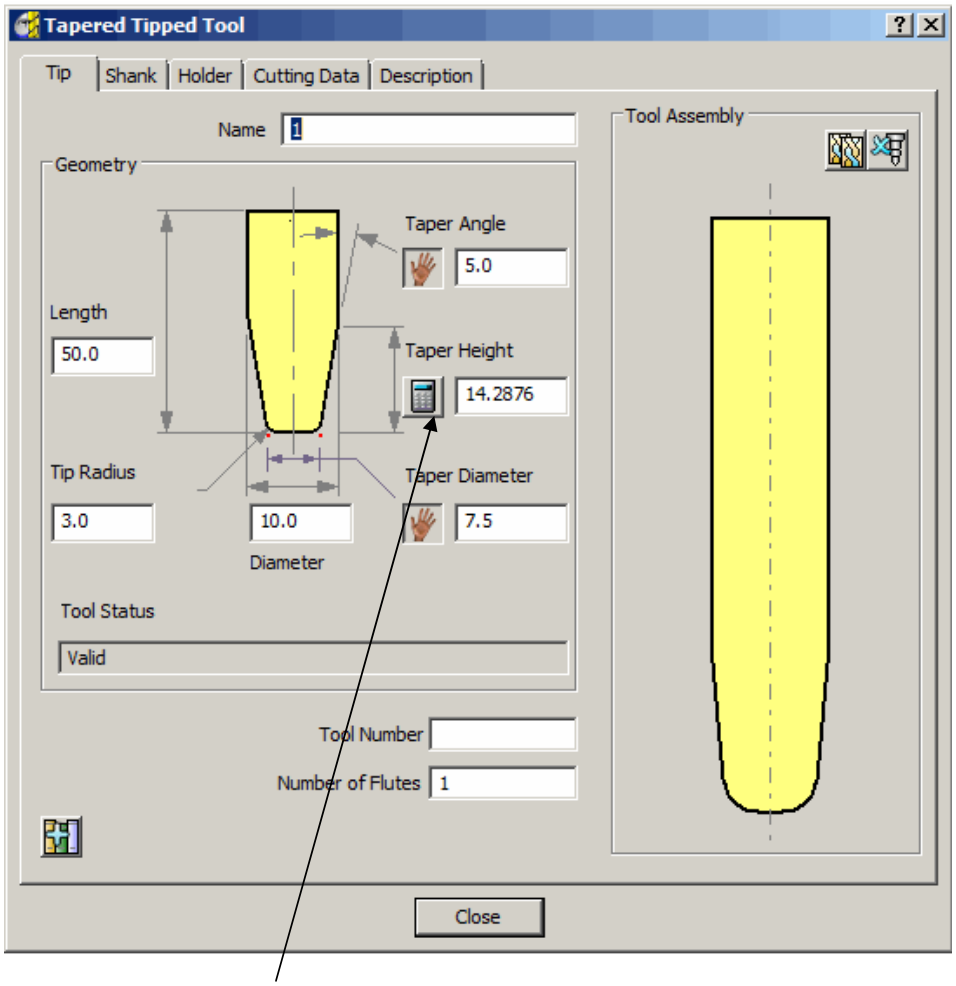
The four ***Dia 15 circles*** are created as **Hole Features**.

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 o**f ***Slot Feature)***.

- Create a **Dia 10** ***Tapered Tipped tool*** named **D10Tr3A5**, with **Tip Radius 3**, **Taper Angle 5Deg**, **Length 50, Taper Diameter 10** (***Final Finishing*** of tapered walls ***Main Pocket*** and ***small Boss***).



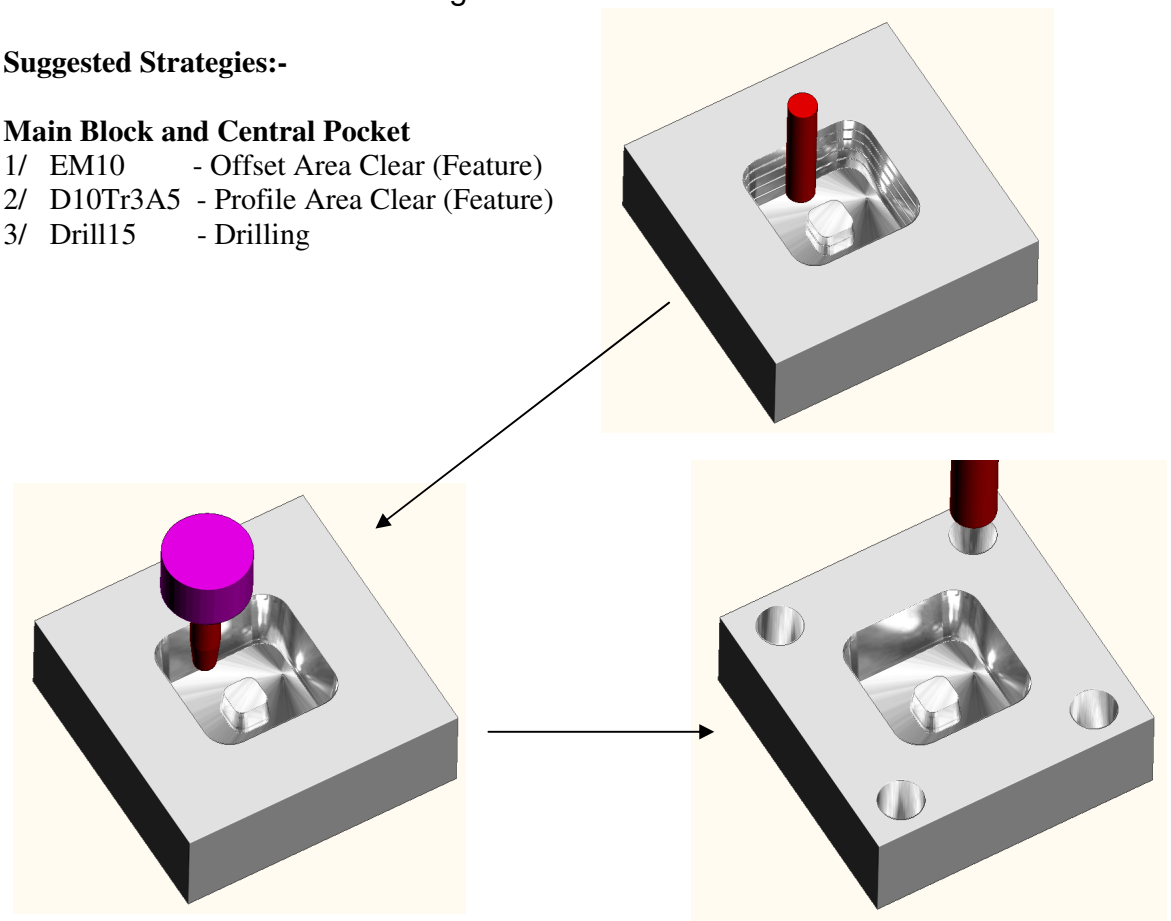
Note:- To calculate the **Taper Height** based on the **Taper Angle** and base **Taper Diameter** the ***Manual Input***, **icon**  adjacent to the **Taper Height** is **clicked** to apply the **Calculate** option  to this parameter.

- 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.

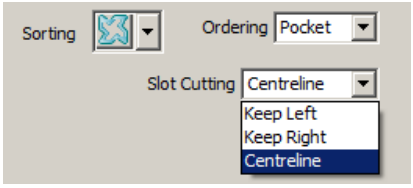
Suggested Strategies:-

Main Block and Central Pocket
1/ EM10 - Offset Area Clear (Feature)
2/ D10Tr3A5 - Profile Area Clear (Feature)
3/ Drill15 - Drilling

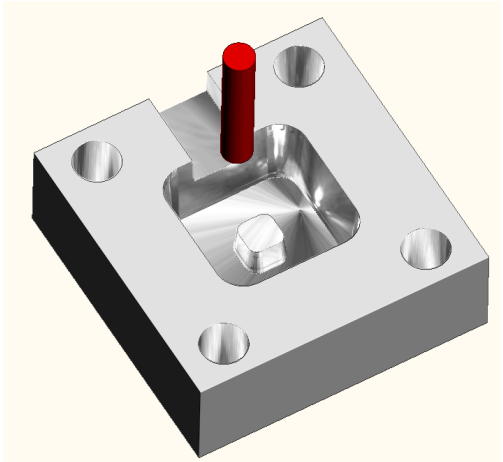


The **Slot** will be machined independently from the main pocket and for this reason it is better to define it in a separate **Feature Set**. Note; The Y dimension of the existing **Block** will have to be adjusted to fully, include the **Slot** Feature.

Slot
1/ EM10 - Profile Area Clear (Feature)



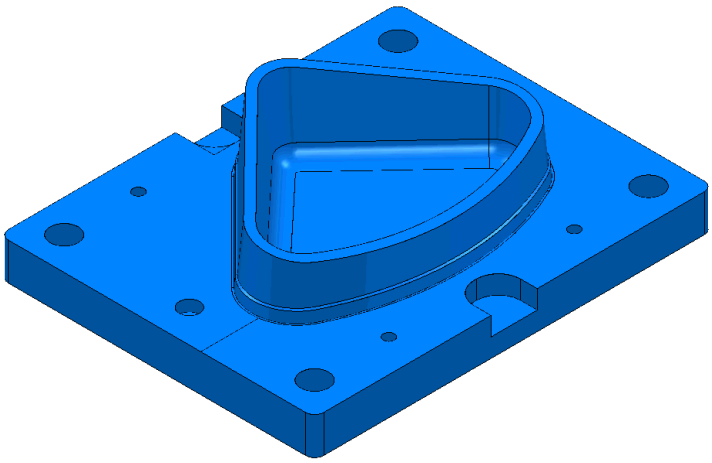
Width of Slot is achieved by creating ***multiple toolpaths*** that use different **Slot Cutting** options.



Creating Features from Model holes

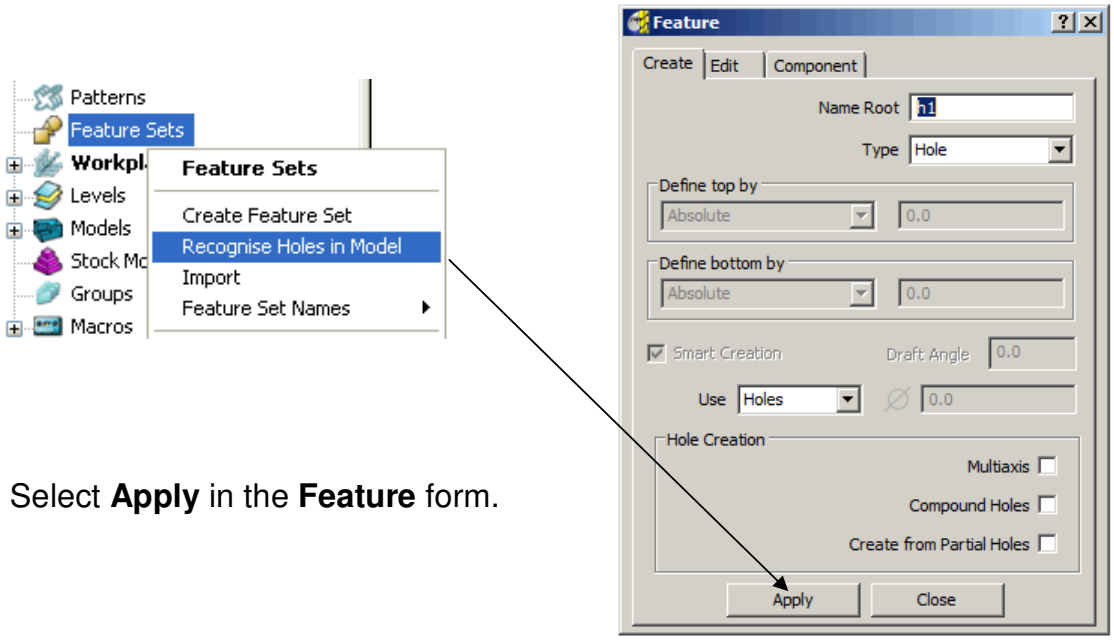
As well as being able to use imported ***2D geometry***, **Hole Features** can be automatically defined from selected **Holes** in the ***3D Surface or Solid*** **model**.

- **Delete All** and **Reset forms**.
- Select **File – Import Model:-**
D:\users\training\PowerMILL_Data\Models\corner_bowl.dgk.



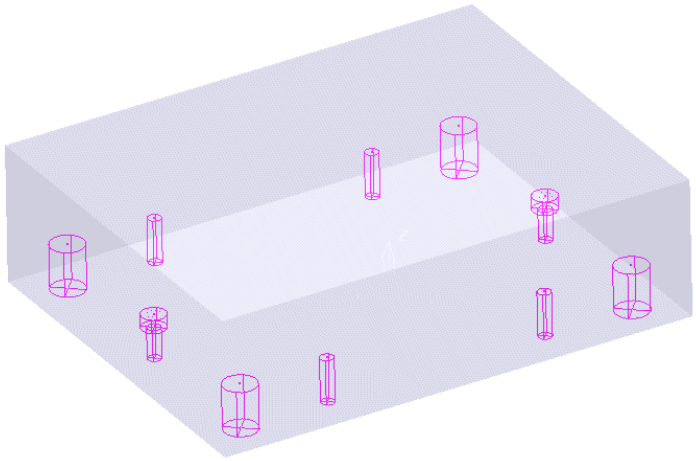
- Make sure the **Block** is not defined at this stage, as the top of the **Hole** features will be created at the nearest **Z** dimension (Max or Min) of the **Block**. This is likely to result in some holes being created the wrong way up (It is however possible to reverse a **Hole** feature).

- Select the whole model.
- Right click over **Feature Sets** and select **Recognise Holes in Model**.



- Select **Apply** in the **Feature** form.

- 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 these holes

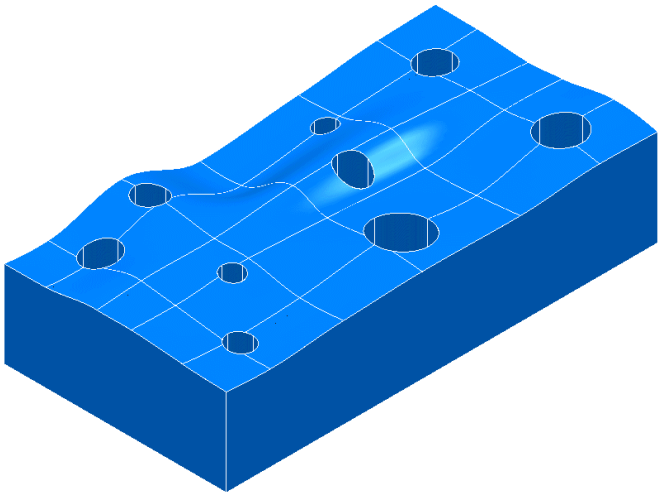
Hole Capping

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**.

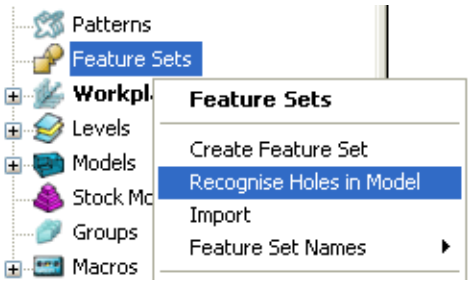
- **Delete All** and **Reset Forms**.
- **Import** the model **Block_with_holes.dgk** from **PowerMILL_Data**.

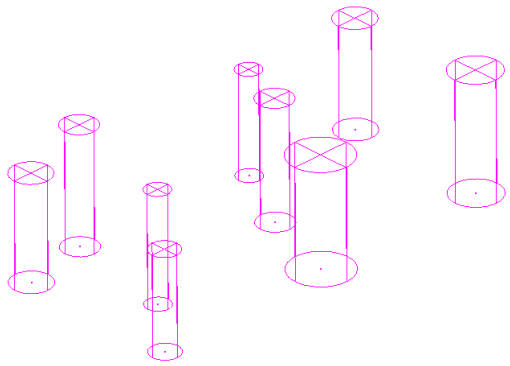


- Calculate a **Block** defined by a **Box** around the **model** (In this case the **Holes** will be created upside down).
- Select the whole **model** by holding down the left hand mouse button and dragging a box over it.

Next **Hole Features** will be created from the model.

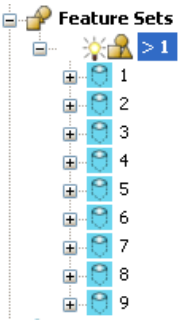
- Select **Recognise Holes in Model**.





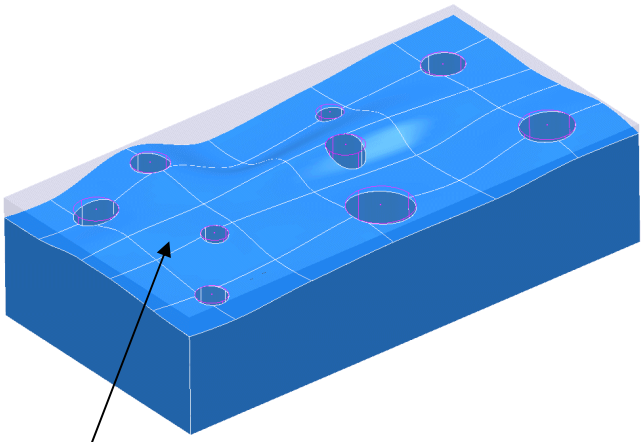
The **tops of the holes** are defined as the shortest distance from the **top** or **bottom** of the material **Block**.
If **Hole Features** are created to a different orientation from the natural **Z Axis**, They are assigned to a new **Feature Set**. A new **Workplane** is also created with the **Z Axis** aligned to the new **Feature Set**, **Hole** direction.

Feature Set, **1** will be created containing 9 **Hole** features extracted from the model.



In this case the holes will be created upside down to illustrate the **Reverse Holes** editing option.

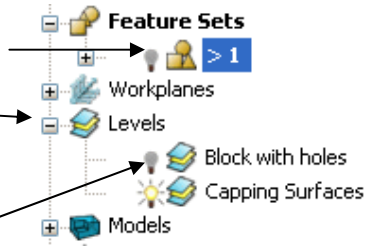
- Right click on **Feature Set 1** and click **Select All** followed by **Edit > Reverse Holes**.



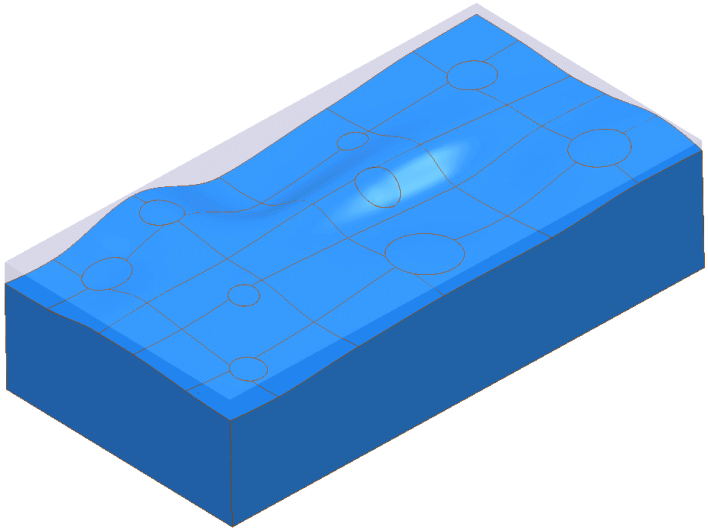
- Finally, select the **top surface**, then in the **explorer**, right click over **Feature Set 1** and select the **Cap Holes** option.

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.

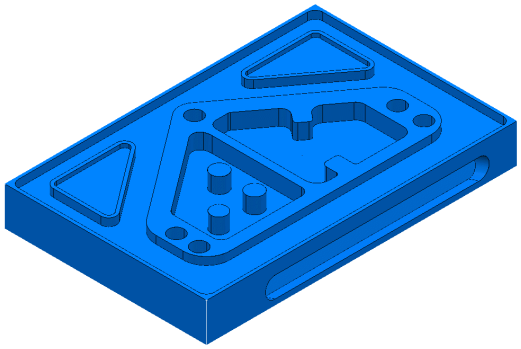


Note; the **Capping Surfaces** are automatically assigned to the new **Level** for easy selection.

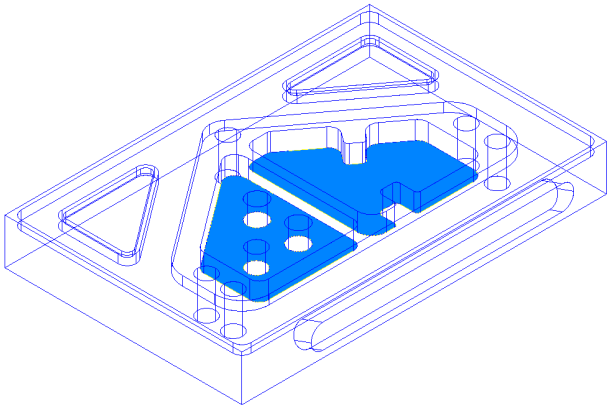
- Click on the ***light bulb*** to draw the **Block with holes** level back on.

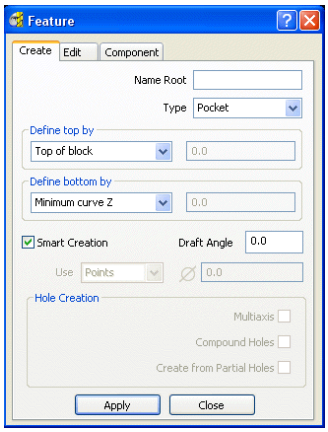
Creating Features from a 3D CAD Model

- **Delete All** and **Reset Forms**.
- **Import** the model **2DExample.dgk** from **examples**.

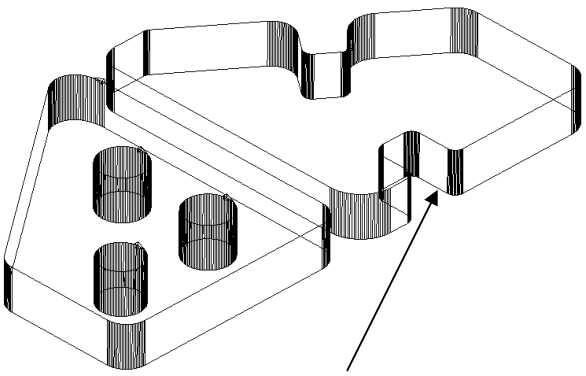


- **Create** a **New** feature with the ***Name*** **Example2**.
- ***Calculate*** a **Block** defined by a **Box** around the **model**.
- Select the **Two** surfaces shown and in the **Feature** form select ***Type*** **Pocket**.





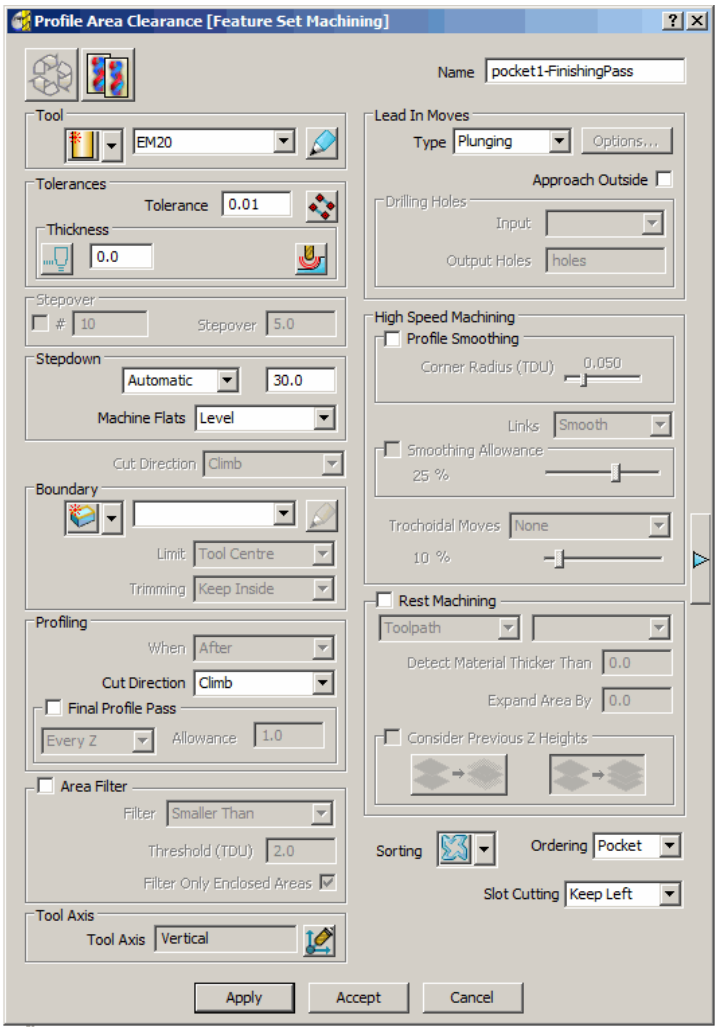
- Input ***Define top by*** **– Top of Block** and **Define Bottom by – Minimum curve Z** shown and create a **Pocket Feature**.
- Create a **Dia 20 End Mill** named **EM20**.



From the **2** selected **Surfaces** a total of **5 Features** are created (**2** large **Pockets** and **3** circular **Bosses**).

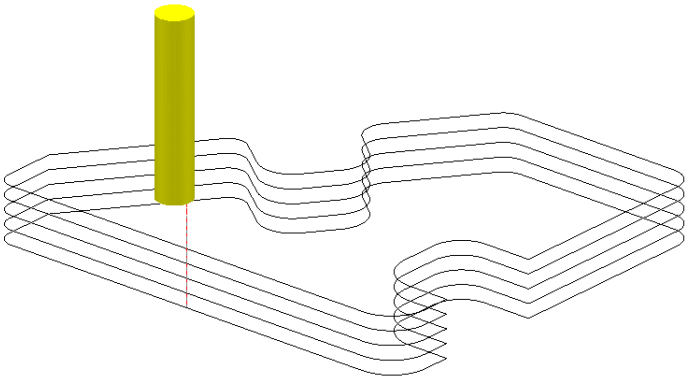
- Select the ***Pocket*** **Feature** which does not have any ***Boss*** **Features** inside.

- From **Machining Strategies** select the **2.5D Area Clearance** tab.
- Select **Profile AreaClear Feature Set** and input data **exactly** as shown in the following form.



- **Untick** – ***Approach Outside***.
- ***Stepdown* 30**.
- ***Cut Direction*** – **Climb**.

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



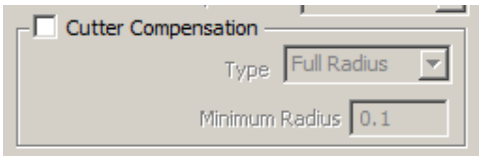
- **Save** the **Project** as:-

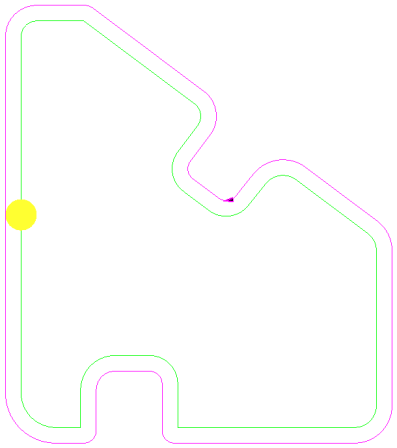
D:\users\training\COURSEWORK\PowerMILL-Projects\2Dtest

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**.

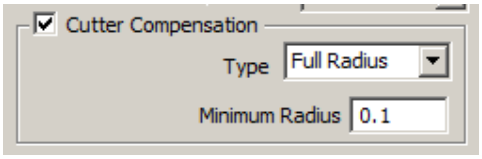
- Activate Toolpath '**Pocket Finish**'.
- Select the **Settings** of the toolpath and make a **Copy**.
- Select the option to toggle the '**Expert Area Clearance**' form.

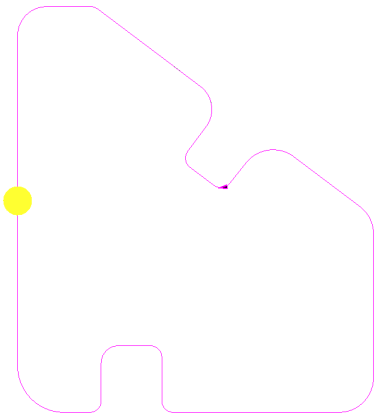




Cutter Compensation un-ticked is the default setting when creating a toolpath. **PowerMILL** compensates for the entire tool radius, and in doing so assumes the tool used is an accurate tool size, meaning the user is not required to define compensation at the machine tool.

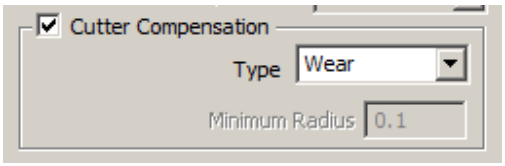
- Tick the ***Cutter Compensation*** box and change the **Type** to **Full Radius** and **Recycle** the toolpath.

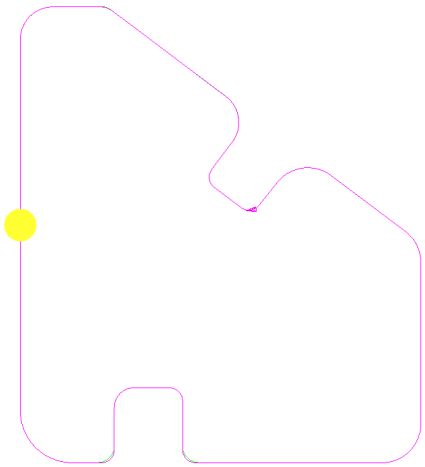




With **Off** selected **PowerMILL** will not compensate for the radius of the tool. In this instance it is required to define the tool radius at the machine tool. This process is often referred to as **Part Edge Programming** but is of limited value as most machine tools will not accurately represent the contour in concave corners, or when an arc exists that is less than the tool radius.

- Select the **Settings** of the toolpath and make a **Copy**.
- Set the ***Cutter Compensation*** - **Type** to **Wear** and **Recycle** the toolpath.





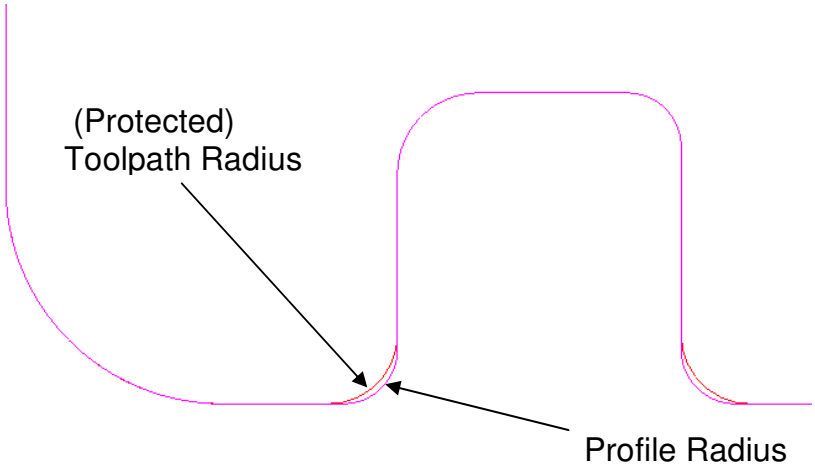
Protected Cutter Compensation

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.

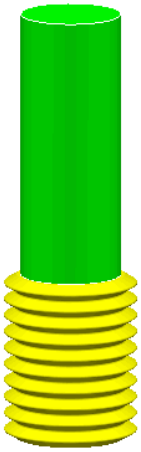
.

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.



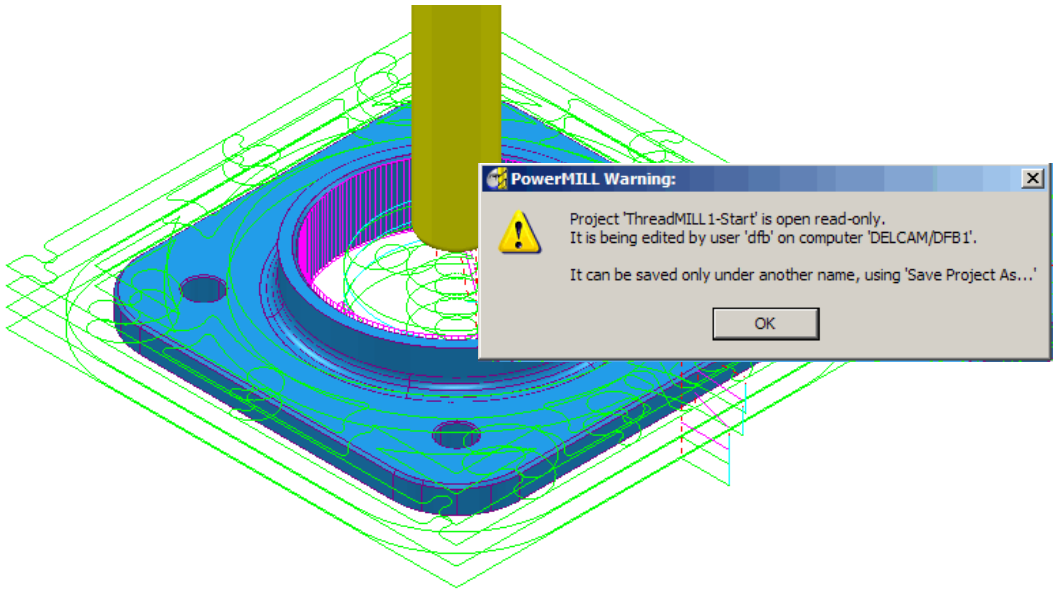
Thread Milling



This specialised **Drilling** Strategy is designed to use a **Thread Milling** tool to ***Mill*** a **Thread** into the walls of circular **Hole Feature**. The tool plunges central 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***:-

D:\users\training\PowerMILL_Data\Projects\ThreadMILL1-Start



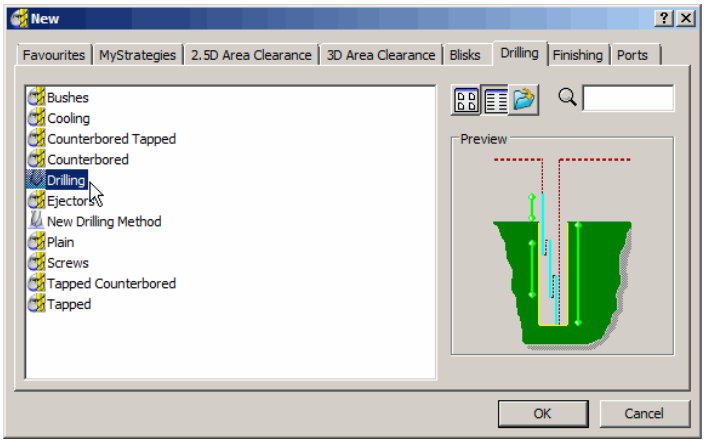
The imported **Project** basically contains a **Cad model**, a predefined **Hole Feature**, a **3D Offset Area Clearance** strategy using a **Tip Raduised** cutter, and a **Thread Milling** tool. The original **Project** is **Locked** to prevent it from being altered, hence the first step is to **Save As** a separate **Project** locally with a different name.

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

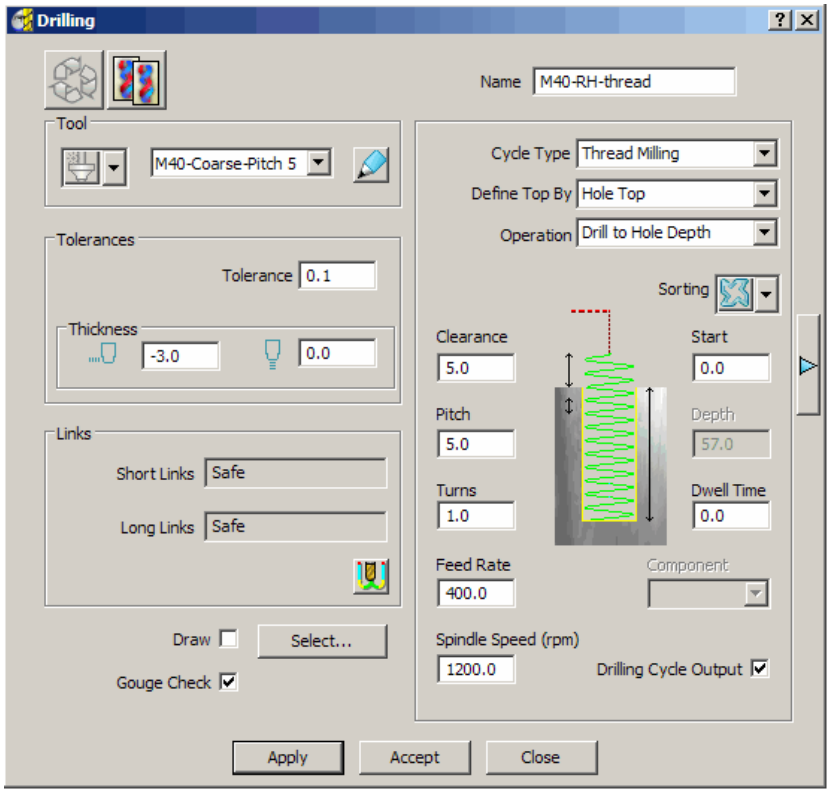
D:\users\training\COURSEWORK\PowerMILL-Projects\ThreadMILL-Example1

- Activate the ***Thread Milling*** **Tool** named **M40-Coarse-Pitch-5**.
- Select the **Hole *Feature*** around the centre wall of the component.

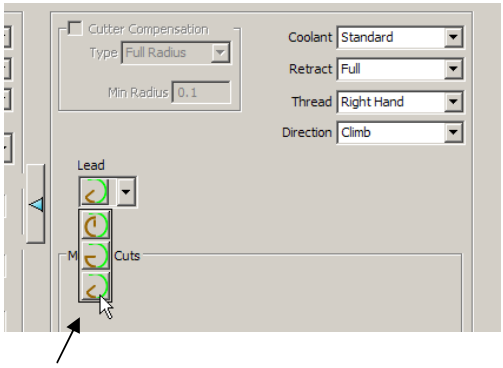
- Open the **Toolpath Strategies** form  and select **Drilling**.
- In the form select **Drilling** as shown in the following form.



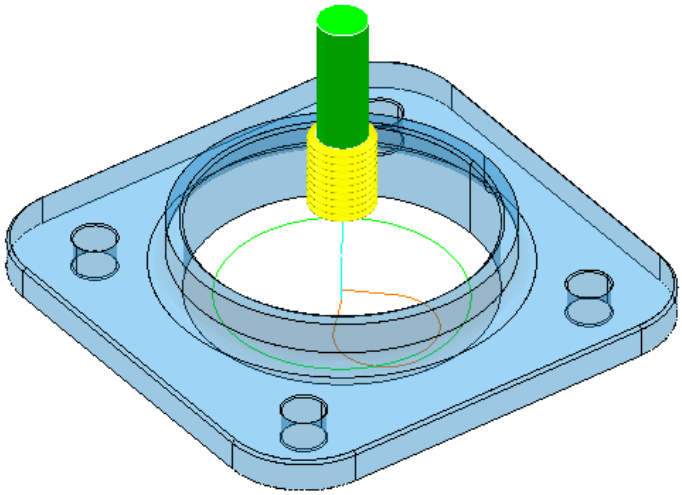
- In The **Drilling** form set the **Cycle Type** to **Thread Milling**, **Radial Thickness -3**, **Pitch 5**, ***Name*** **M40-RH-Thread**, leaving all other parameters as default (as shown below).



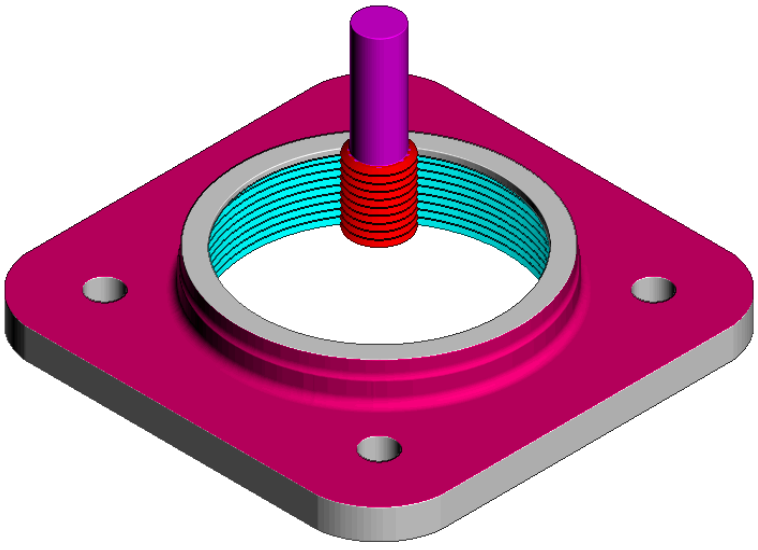
- Select **Expert** settings  to expand the form.



- Select the use a **45 degree arc lead** option and return to the main form.
- **Apply** the form main to ***mill*** a **Right Hand** thread ***3mm deep*** into the wall.



- 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**.
- Perform a **ViewMILL simulation** of both **toolpaths** (Do not Activate either toolpath, so that the current **Block** definition is maintained).



PRO - 2D Machining Options

Introduction

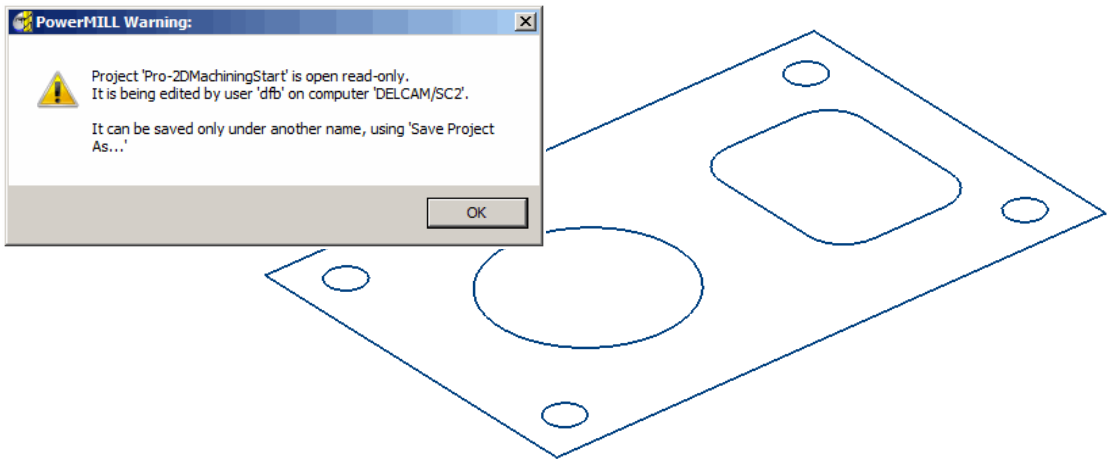
PowerMILL Pro contains ***4 new strategies*** designed for more efficient **2D Machining** applications. These include:-

1. **Face Milling**.
2. **2D Curve Area Clearance**.
3. **2D Curve Profile**.
4. **Chamfer Milling**.

2D Machining Example using Patterns

- Select **File ➔Delete All** and **Tools ➔ Reset forms**.
- **Open** the ***Project***:-

D:\users\training\PowerMILL_Data\Projects\Pro-2DMachiningStart.

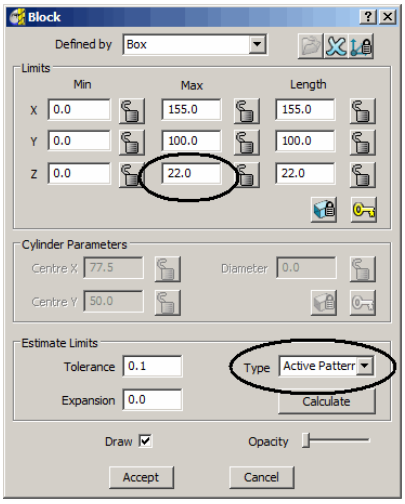


The imported **Project** contains **4 Tools** and **3 Patterns** defining the **2D geometry** to be used in the **2D Machining** options. The **Project** is **Locked** to prevent it from being altered hence the first step is to **Save As** a separate **Project** locally with a different name.

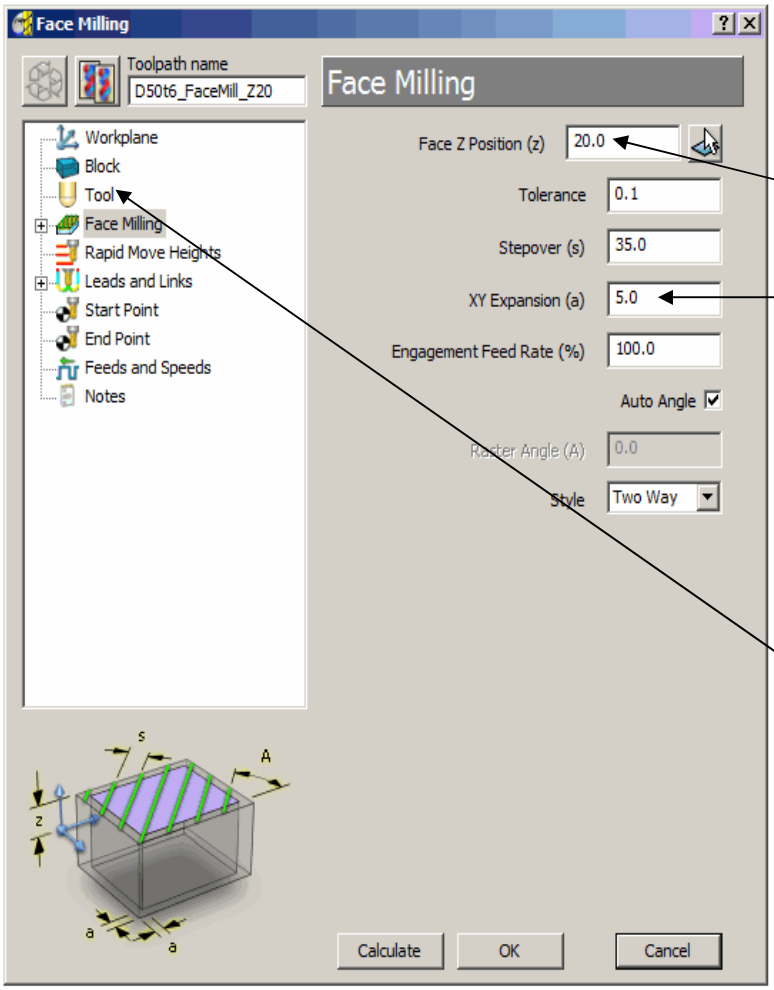
- Select **File** - **Save Project As**:-

D:\users\training\COURSEWORK\PowerMILL-Projects\PRO-2DMachining

- **Activate** the **Pattern** named **CurveAC**.
- Open the **Block** form and set ***Defined by*** to **Box** followed by ***Type*** as **Active Pattern** and select **Calculate** (**Do not close the form**).
- In the **Max Z *Box*** input the value **22** before selecting **Accept** to close the form.



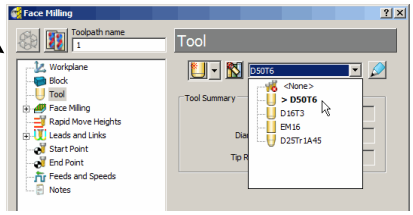
- In **Rapid Move Heights** click on **Reset to Safe Heights**.
- **Activate** the **Face Milling** tool named **D50T6**.
- From the **2.5D Area Clearance** options select **Face Milling** and enter data into the form exactly as shown below.



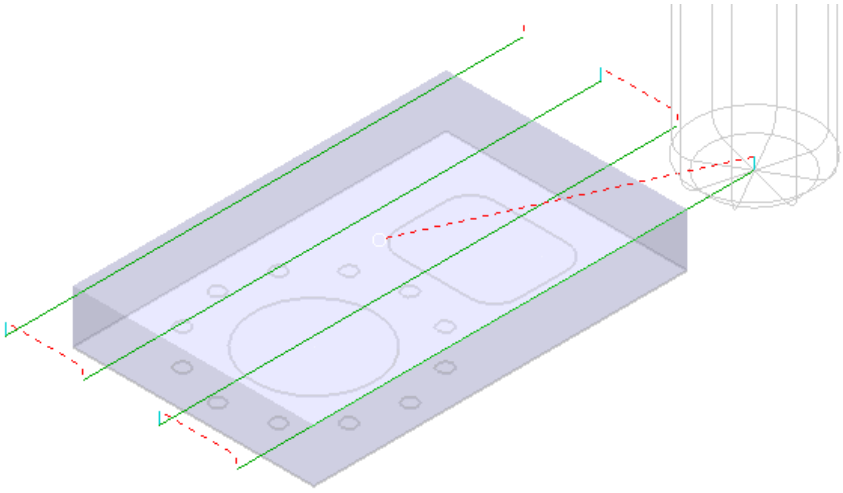
Height of finished component.

Extra Tool 'run off'.

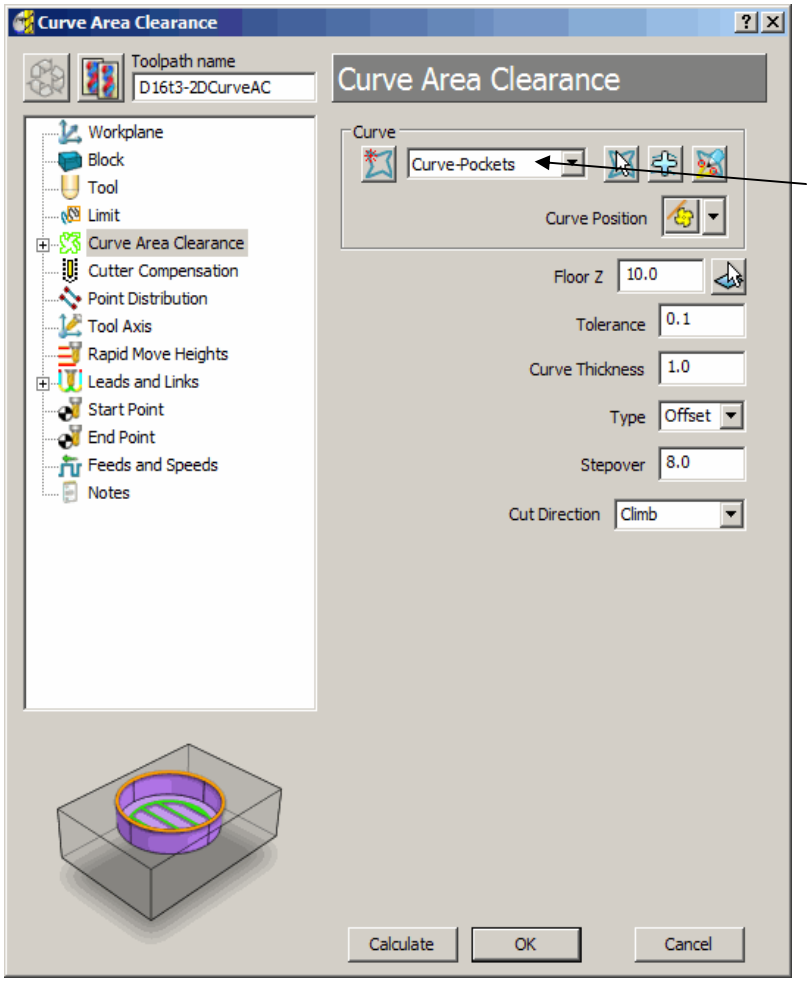
The **explorer** items can be directly selected to access associated selections and settings, such as **Tools**, **Rapid Move Heights**, etc.



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

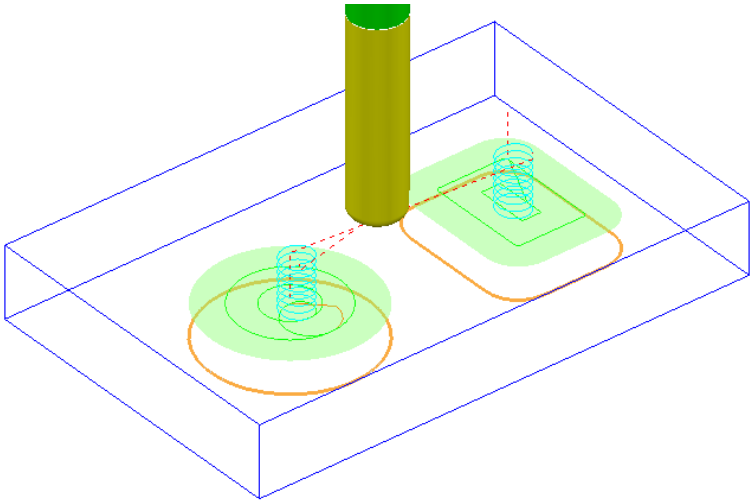


- **Activate** the **Tip Radiused** tool named **D16T3**.
- From the **2.5D Area Clearance** options select **2D Curve Area Clearance** and enter data into the form exactly as shown below.



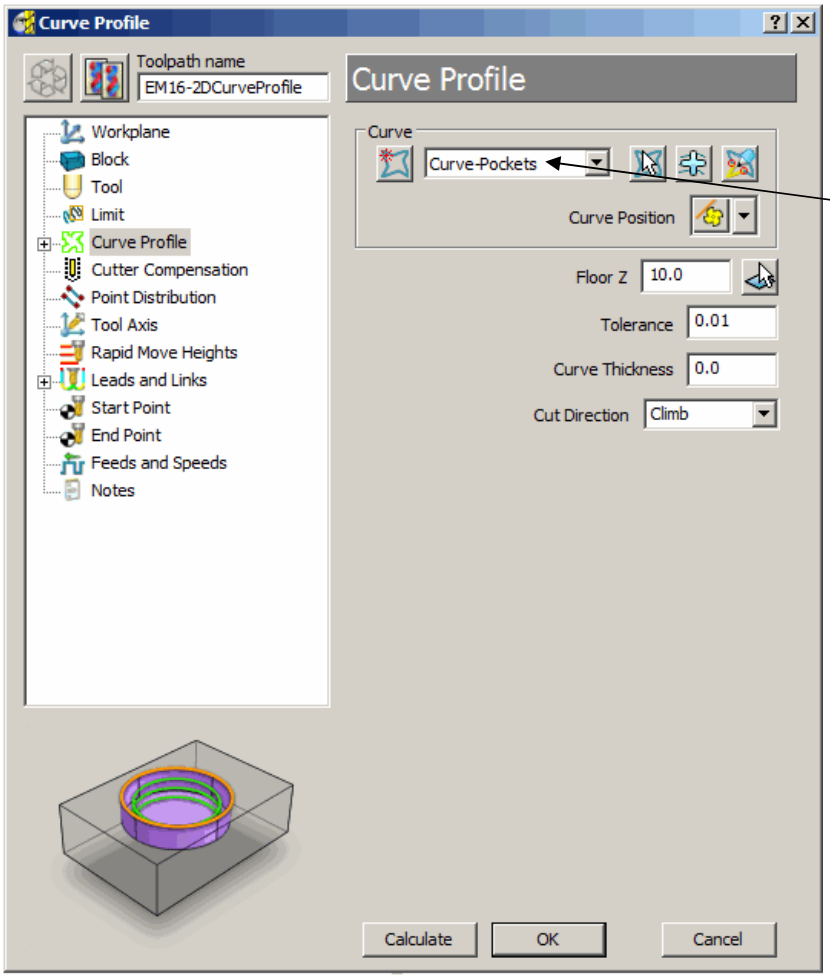
Select the Pattern:-
Curve-Pockets

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



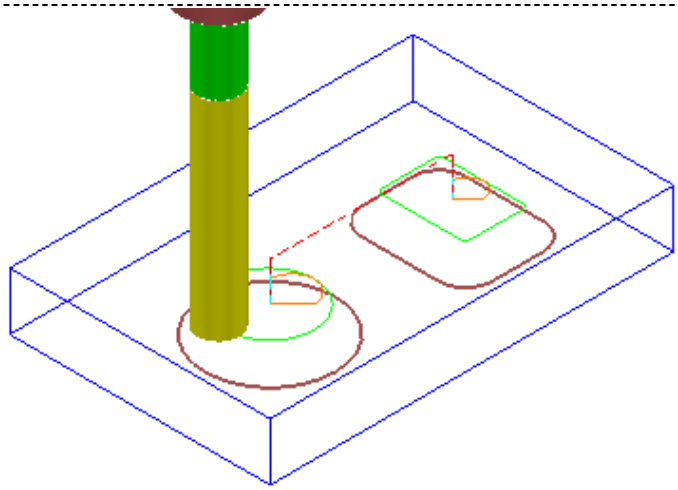
Note: While the **Curve Area Clearance** form is open, the areas to be machined are displayed as a shaded preview.

- **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.

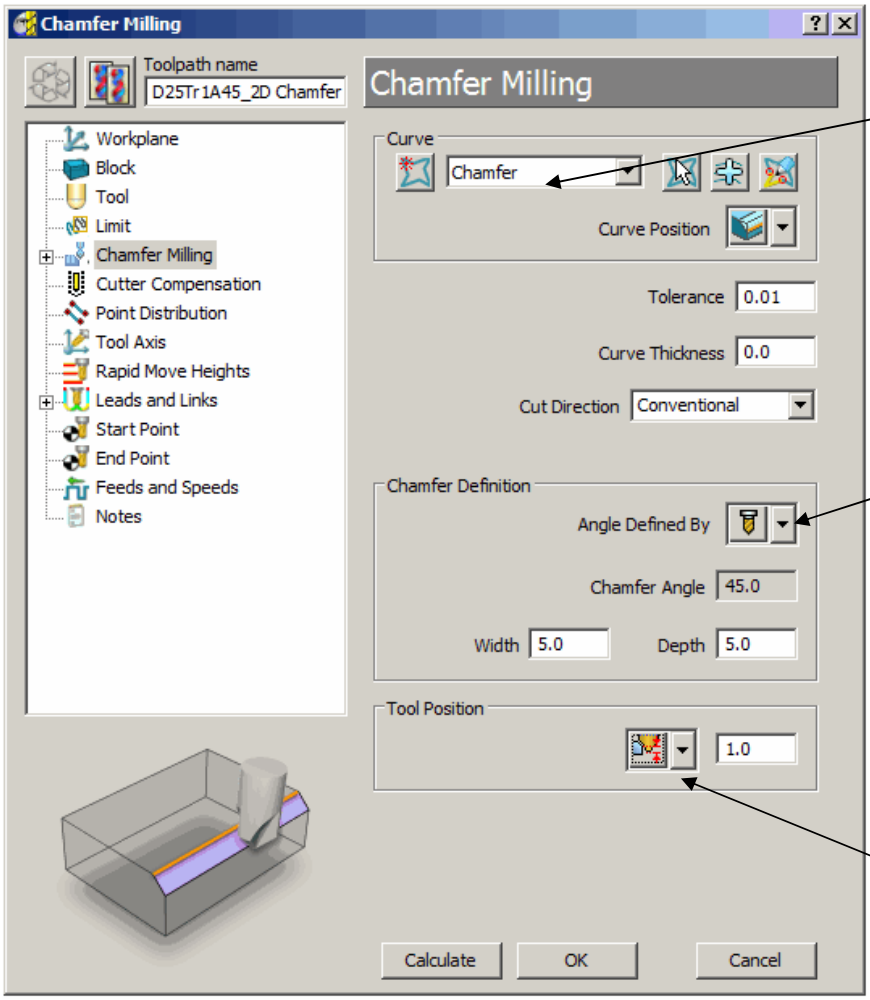


Select the Pattern:- **Curve-Pockets**

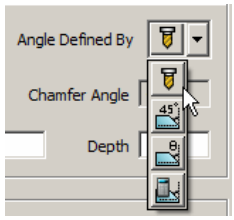
- Select **Calculate** to process the toolpath.



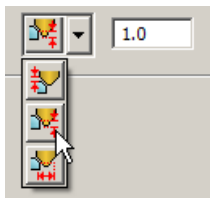
- **Activate** the **Chamfer Tool** named **D25Tr1A45/_2D Chamfer**.
- From the **2.5D Area Clearance** options select **Chamfer Milling** and enter data into the form exactly as shown below.



Select the **Pattern**:- **Chamfer**

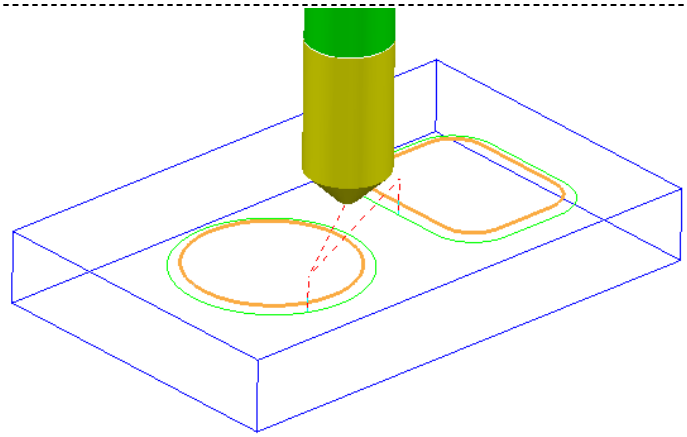


Default Chamfer **Angle** setting used (**Tool** Geometry).



The **Pattern** used as the basis for the chamfer is positioned on the lower edge and a **1.0** overlap is specified.

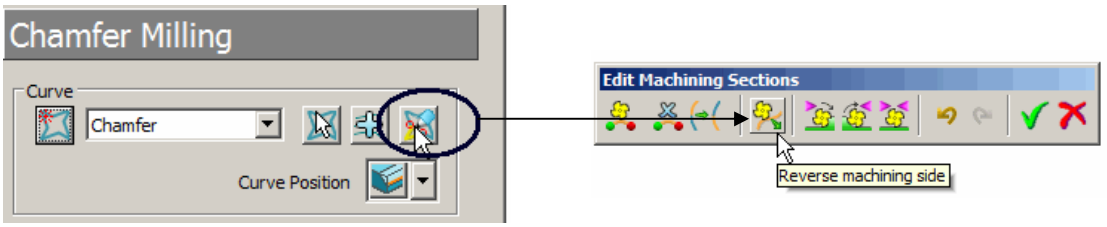
- Select **Calculate** to process the toolpath (Do not close the form).



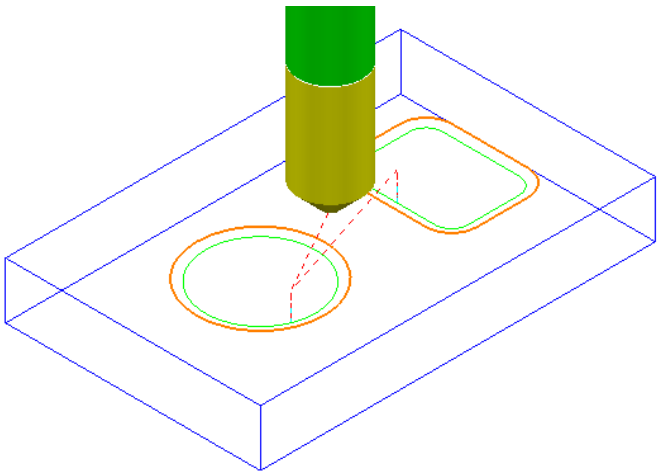
The resultant **Chamfer Milling** strategy has appeared on the wrong side of the **Pattern**.

The **toolpath** needs to be ***recycled*** with the **Chamfer Milling** tool tracks on the inside of the **Pattern** segments.

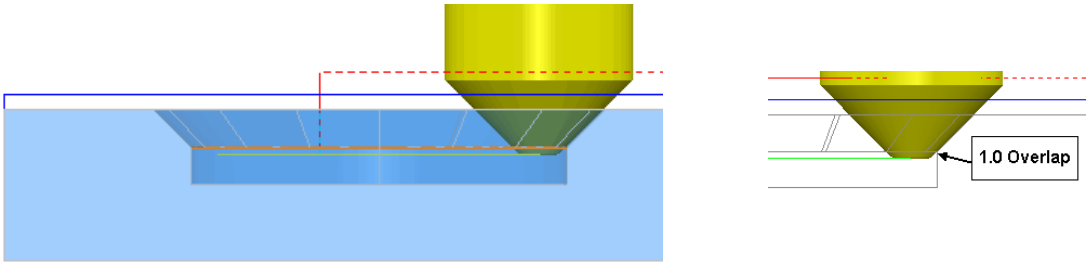
- In the **Chamfer Milling** form, select the ***Recycle*** icon to enable changes to be made to the above strategy.
- Select the ***Interactively modify machinable sections*** icon to access the ***Edit Machinable Sections*** toolbar.
- Select the ***Reverse machining side*** option, followed by the ***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).

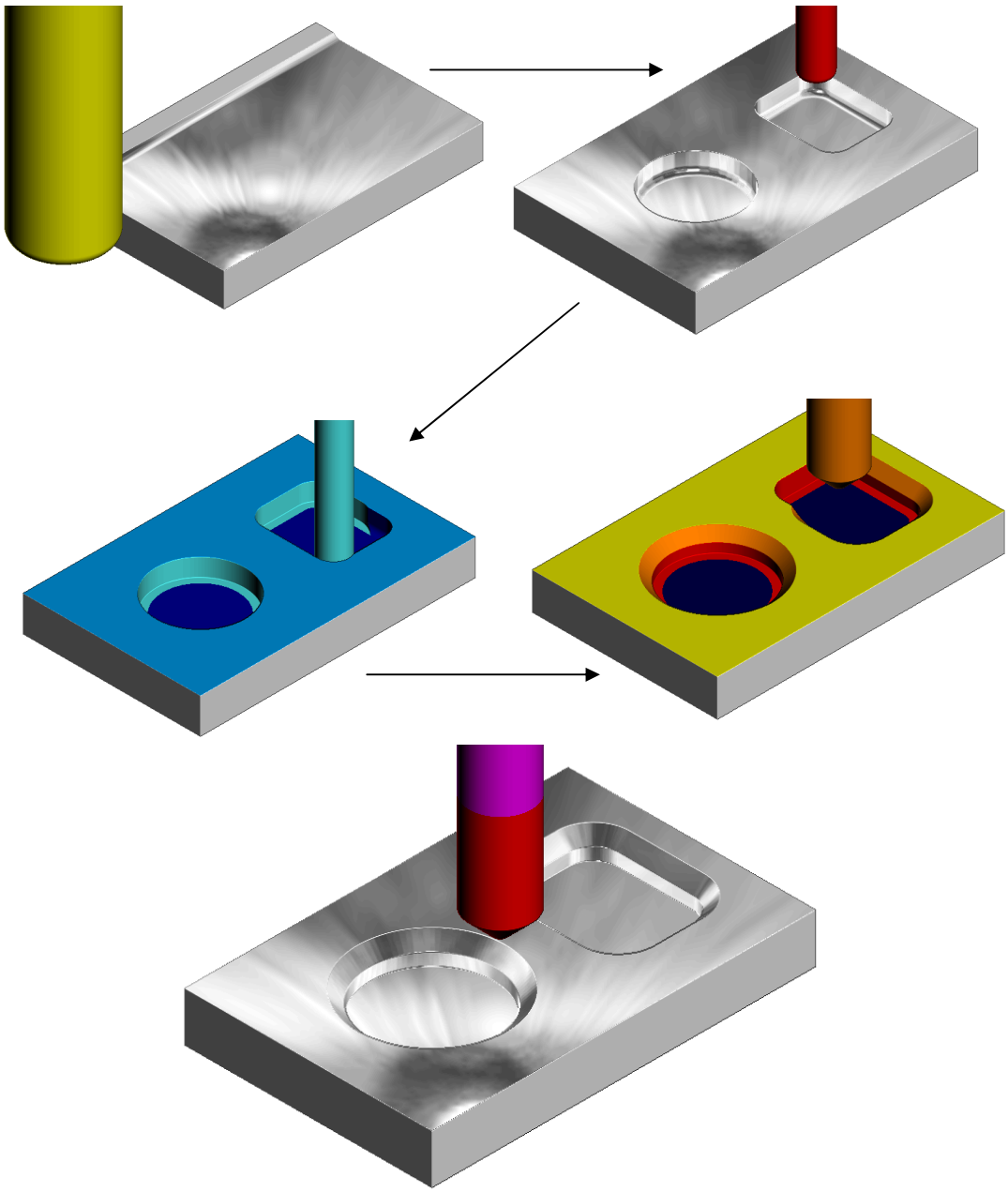


The **Chamfer Milling** options selected result in the angled part of the tool overlapping the base of the Chamfer by **1mm**.



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.

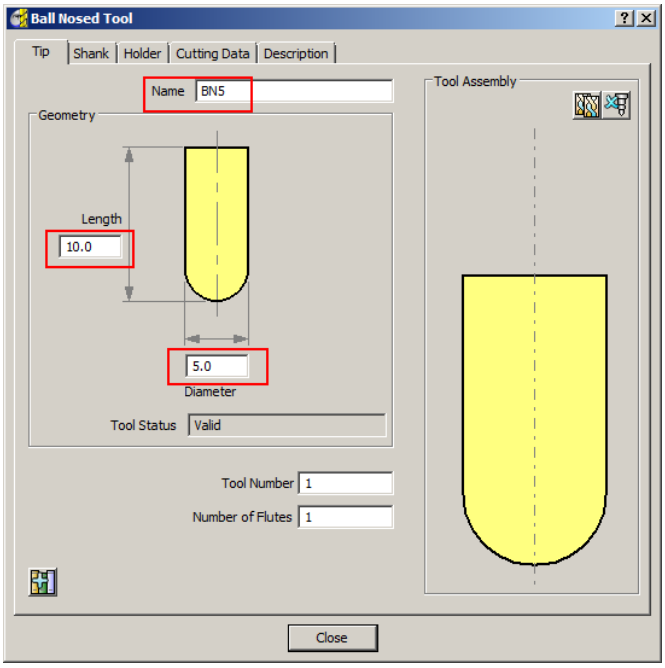


10. Tool Holder Collision Checking

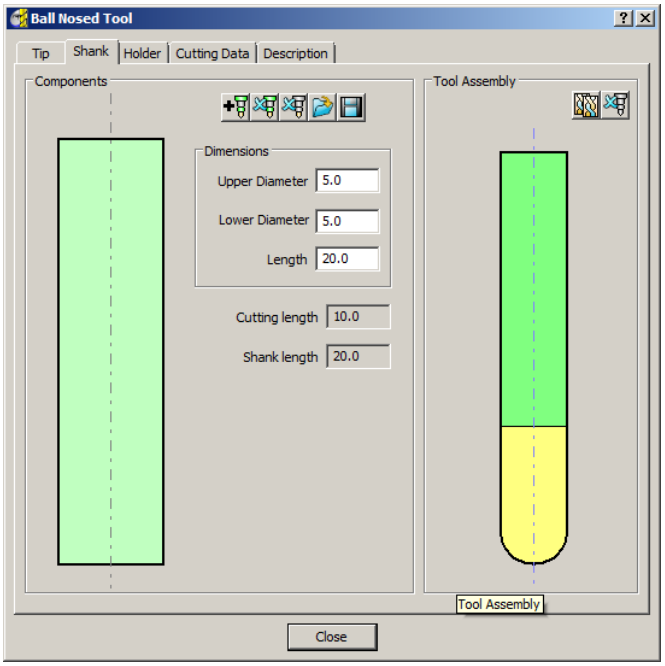
Collision Checking

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.
- From the **Examples** folder open the model **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**.

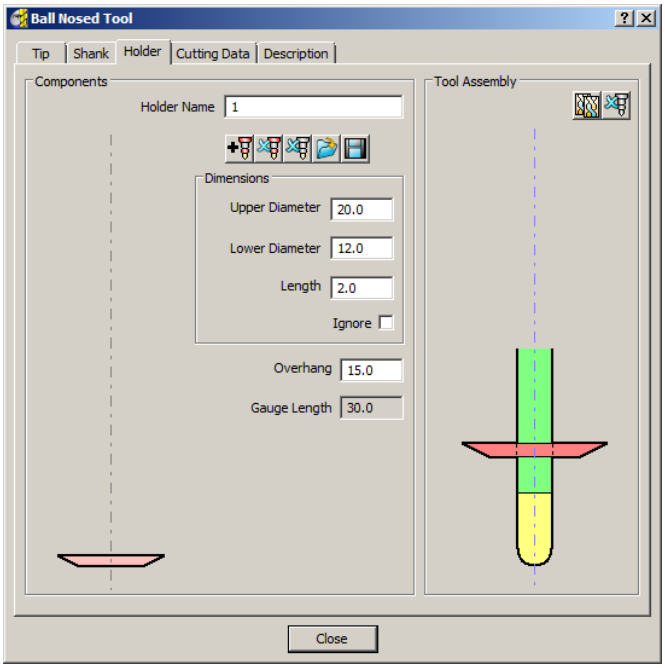


- Select the **Shank** tab.
- Select **Add a shank component** .
- Fill in the form as shown in the image below.



You will now add a **Shank**.

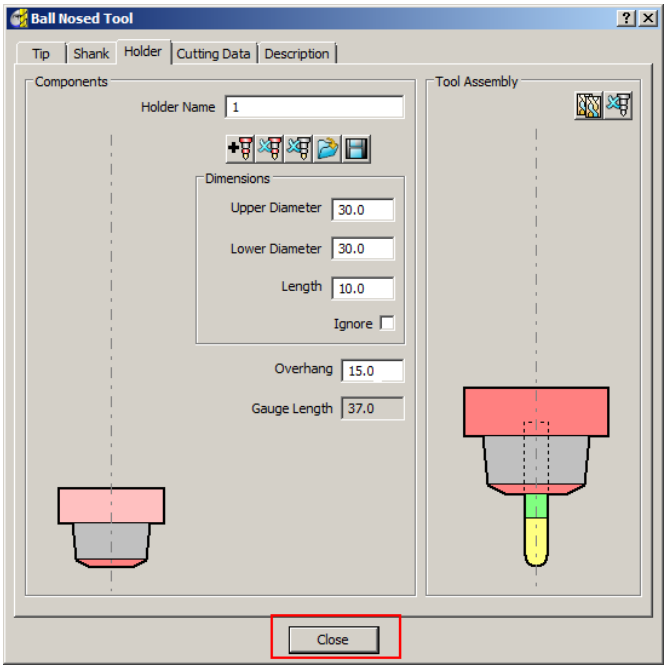
- Select the **Holder** tab.
- Select **Add holder component** .
- Fill in the form as shown in the image below.



A picture of the current active tool is displayed on the right hand side of the dialogue.
Overhang – the length of the cutter protruding from the chuck or collet. If any collisions are found this value is increased to the minimum value to avoid such collisions.
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 section of tool holder.

- Select **Add holder component** .
- Enter an **Upper Diameter** of **22**, a **Lower Diameter** of **20** and a **Length** of **10**.

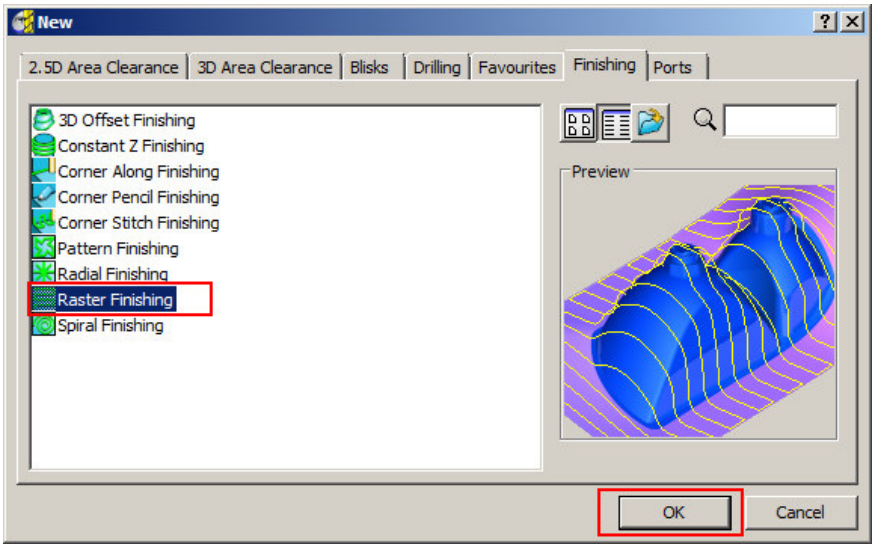
- Select **Add holder component** .
- Enter an **Upper Diameter** of **30**, a **Lower Diameter** of **30**, a **Length** of **10 and** an **Overhang** of **15**.
- Select **Save tool holder** in **C:\Temp** with a **File name** as **ToolHolder 1.pmlth**.
- Select **Close**.



Once you've added the components you can easily modify or delete them by simply clicking on one of them in the form (they turn pale. The current values are displayed and available for modification.
The same applies if it is necessary to modify the **Shank** details.

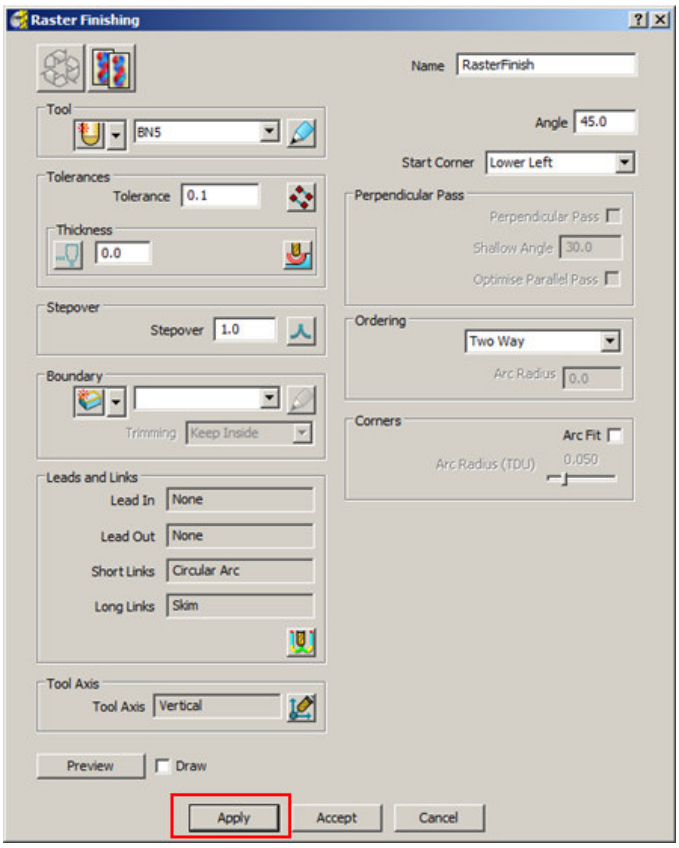
After creating the tool **Holder** you save it. It can then be loaded at a later date to check further toolpaths for collisions if required.

- Select **Toolpath Strategies** on the main toolbar.
- Select a **Raster Finishing** strategy.
- Select **OK**.

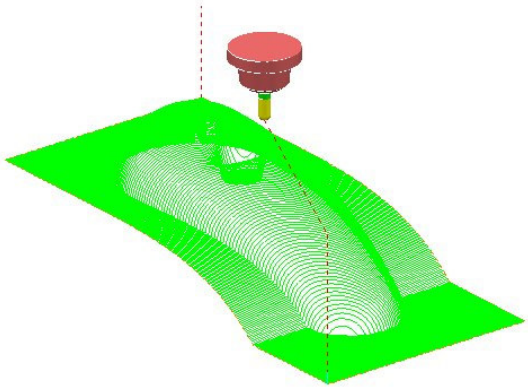


- Enter a **Name** as **RasterFinish**, an **Angle** of **45** and **Stepover** of **1**.
- Select **Lower Left** as the **Start Corner**.
- Select **Leads and links**  .
- In the **Leads and Links** dialog box, select the tab **Lead In** and set the **1st** and **2nd Choices** to **None**.
- Do exactly 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**.

The **Raster Finishing** strategy form should look like in the image below.

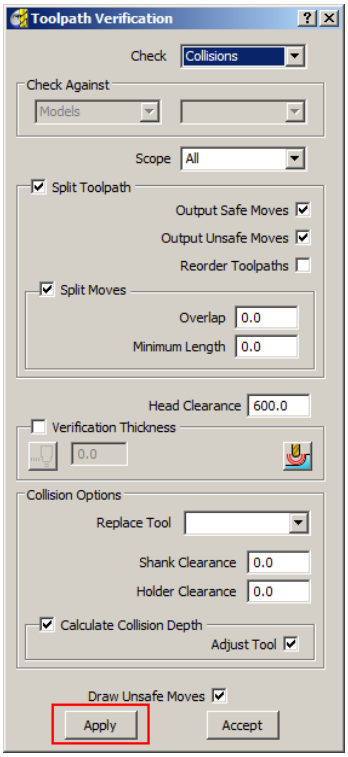


- Select **Apply**, followed by **Cancel**.



The new toolpath is generated but at this stage no collision checking has been applied.

- Select **Toolpath Verification** .



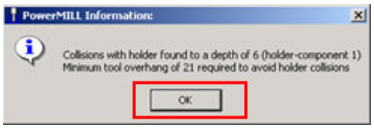
The two **Check** options here are **Collisions** and **Gouges**. **Scope** contains options to control which actual elements of a toolpath are checked.

The amount by which the unsafe move is extended to **Overlap** with an adjacent safe move.

Represents a specific safe area around the tool **Shank** and **Holder** to be taken into account while collision checking.

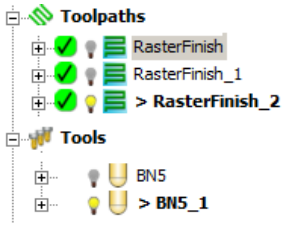
Using the above settings the original toolpath is retained and two new toolpaths are created **Split** to represent the collision and collision safe areas. For the original and collision area toolpaths, the original tool is automatically, replaced by a copy with a suitably extended **Tool Length** for collision free machining.

- Select **Apply**.
- Select **OK**.



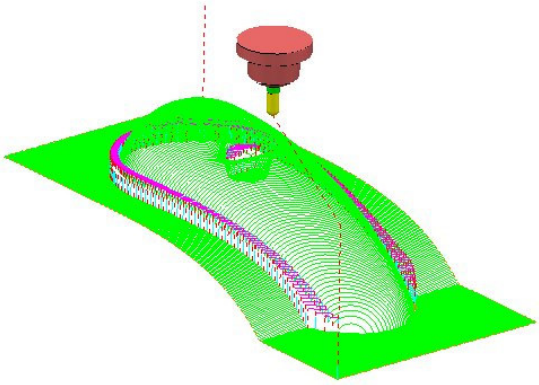
In this case **PowerMILL** has detected that with the current tool **Holder** and **Shank** settings collisions will be present at a **depth** of **6mm**. A suggested tool **Overhang** of **21mm** will be required to avoid this.

- Select **Accept o**n the **Toolpath Verification** form.



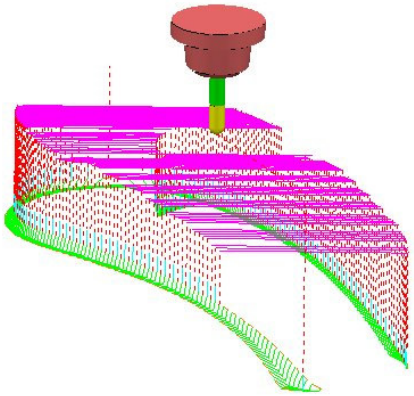
PowerMILL has created two more toolpaths, raster finish_1 and raster finish_2. A new tool called **BN5_1** has also been created in the tools area in the explorer. This new tool has been created with the new adjustments made i.e. **Overhang 21**.

- Activate the toolpath **RasterFinish_1** in the **Explorer**.



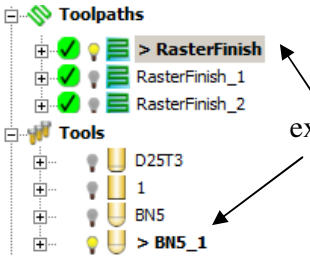
This toolpath only contains segments of the original toolpath, **RasterFinish**, that do not have any collisions so the original tool and tool holder, **BN5** is still associated with it.
If your preference is to run with the shorter overhang then you could run this toolpath on these areas shown.

- **Activate** the toolpath **RasterFinish_2** in the **Explorer**.



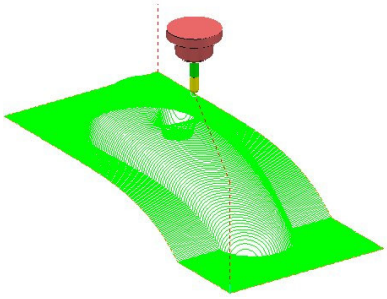
This toolpath contains segments of the original toolpath that can only be machined with the **newly** adjusted tool **Overhang**.

- **Activate** the original toolpath **RasterFinish** in the **Explorer**.



PowerMILL has replaced the newly created tool (with the extended overhang) into the original toolpath.

- Select **File - Save Project As** and enter as **File name** as: **ToolCollisionExample** and save it in D:\users\training\COURSEWORK\PowerMILL-Projects.

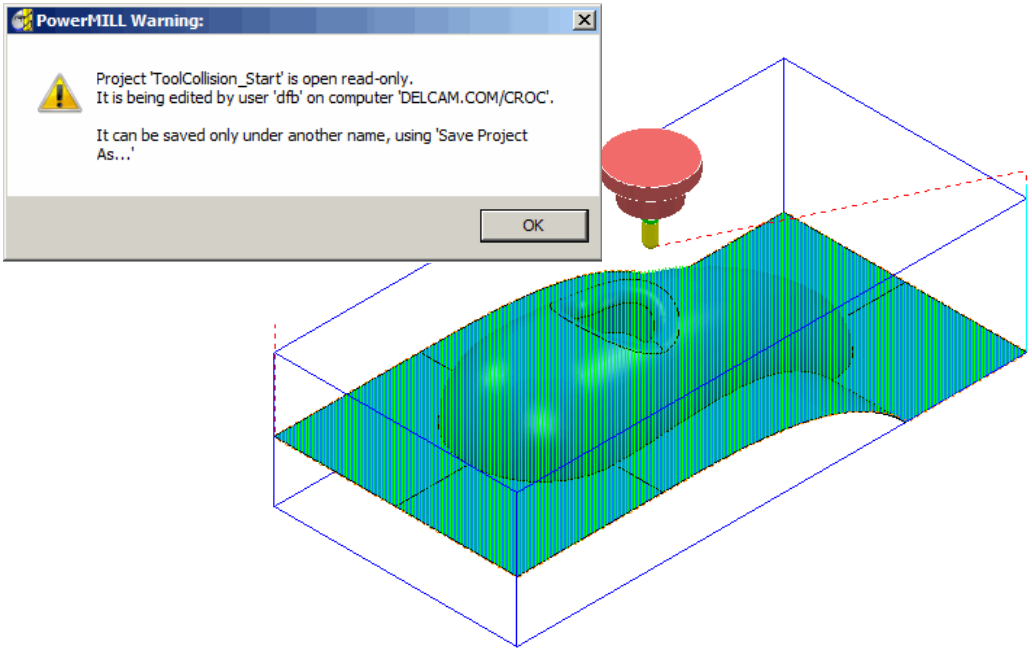


Pro - Tool Holder Collision Checking

Automatic Collision Checking

In ***PowerMILL Pro***, **Tool Holder Collision Checking** can be applied at the time of the toolpath calculation.
If the toolpath is calculated using a tool that includes a shank and tool holder any potential part of the toolpath segments 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**.

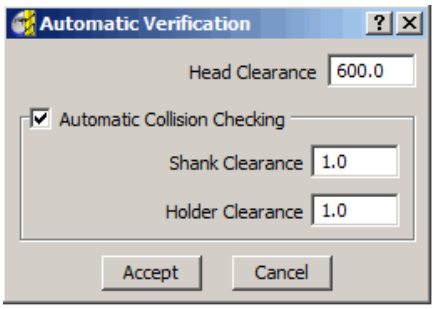
- Select **File - Delete All** and **Tools - Reset Forms**.
- **Open** the **Project**:-
- **D:\users\training\PowerMILL_Data\Projects\ToolCollision_Start**



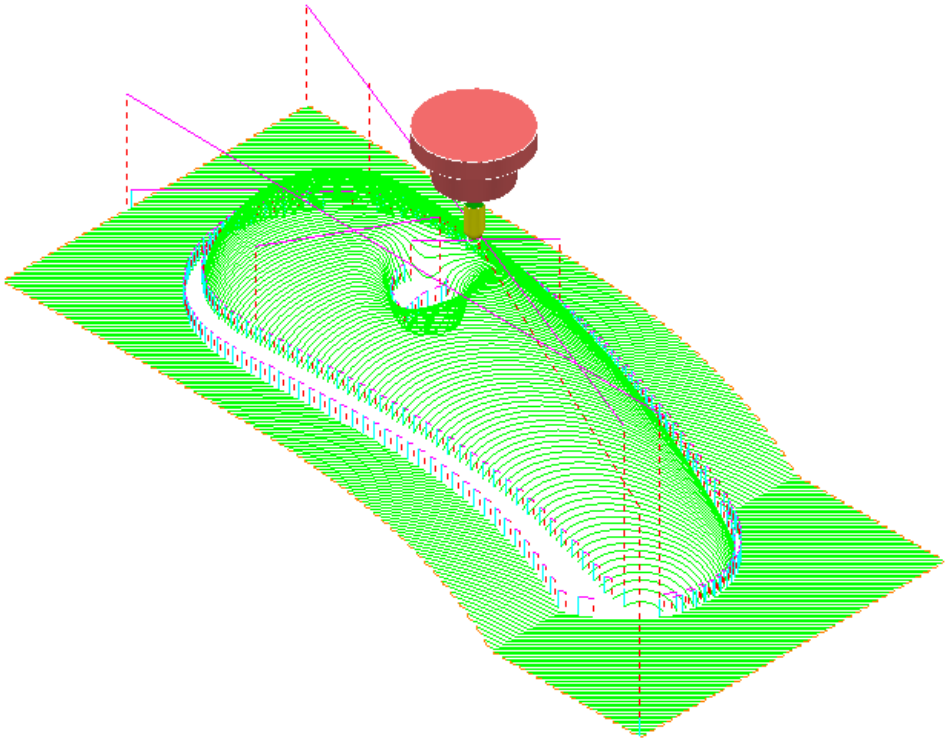
- Select **OK** in the ***PowerMILL Warning*** form.
- **Save Project As**:-
  D:\users\training\COURSEWORK\PowerMILL-Projects\ToolCollisionPro_1

- 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 Finising form open, select the **Automatic Verification** icon from the ***Main Toolbar*** to open the following form.



- Tick the box ***Automatic Collision Checking*** and set both **Shank Clearance** and **Holder Clearance** to **1.0**
- **Accept** the form.
- **Apply** the **Raster Finishing** form to directly create a collision free toolpath, **RasterFinish_1**.

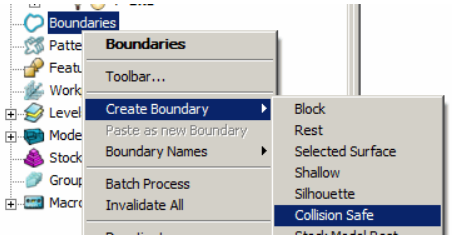


Note:- This method does not display a new **Shank** length for the missing area or automatically create a new compatible tool (**bn5_1**).

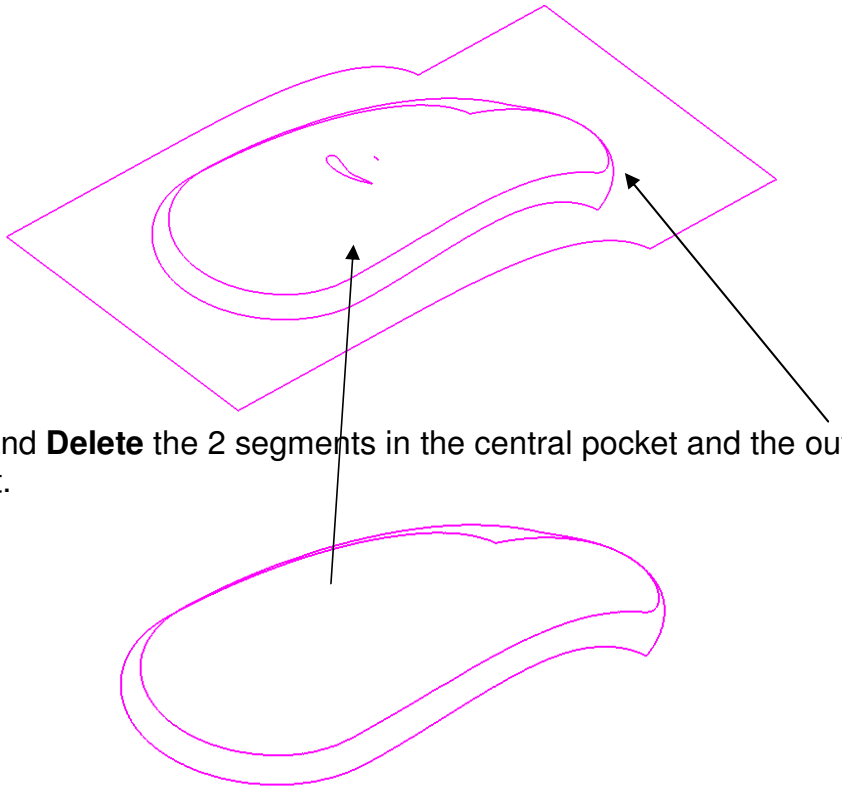
Collision Safe Boundary

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**.



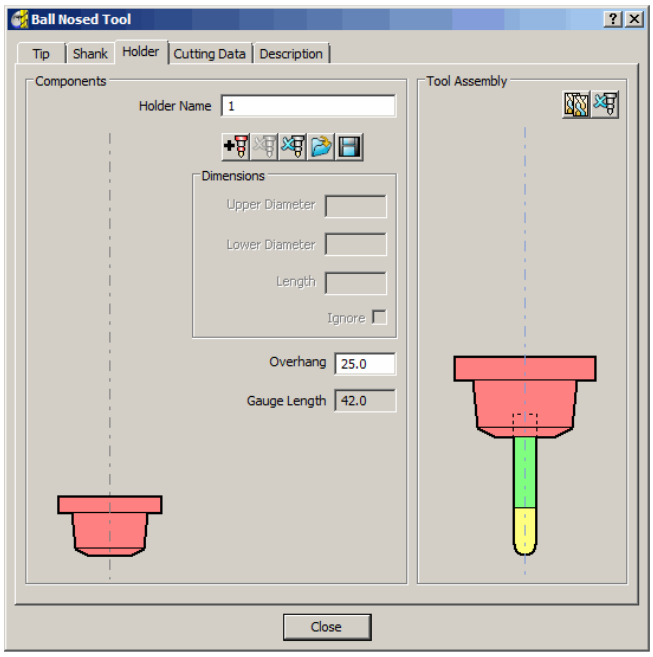
The lower part of the central pocket is more suited to be finish machined using an **End Mill**.



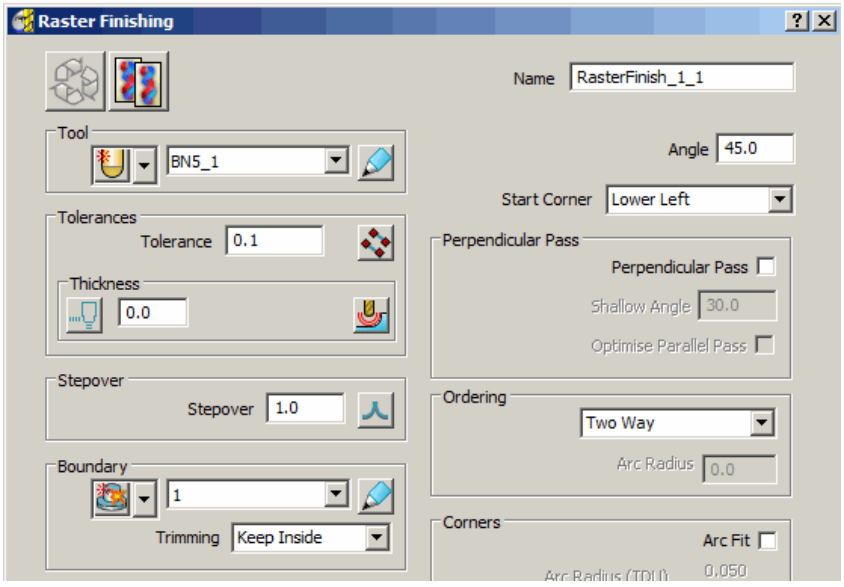
- **Select** and **Delete** the 2 segments in the central pocket and the outer segment.

- In the **explorer** right mouse click over the **Tool** - **BN5** and from the local menu select **Edit - Copy Tool** (to create **BN5-1**).

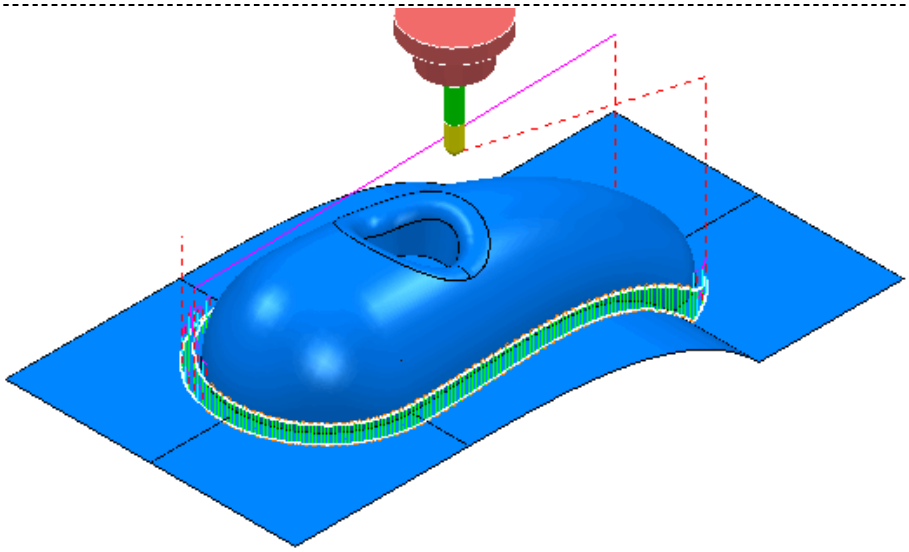
- Right mouse click over the new tool **BN5-1** and from the local menu select **Activate** followed by **Settings**.



- 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 **Copy toolpath** icon.

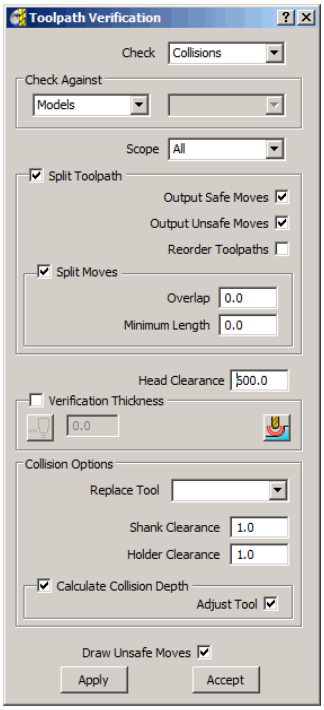


- In the ***Tool*** area of the form select **BN5_1** and in ***Boundary*** select **1** with ***Trimming*** - **Keep Inside**.
- **Apply** the **Raster Finishing** form to create a new toolpath, **RasterFinish_1_1** limited to be ***Inside*** **Boundary 1**.



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. Having said that it would not do any harm to apply retrospective **collision checking** to confirm this.

- Select the retrospective **Toolpath Verification** icon from the top toolbar. 



- Select **Check Collisions** with both **Shank** and **Holder Clearance** set to **1.0** and **Apply** the form.

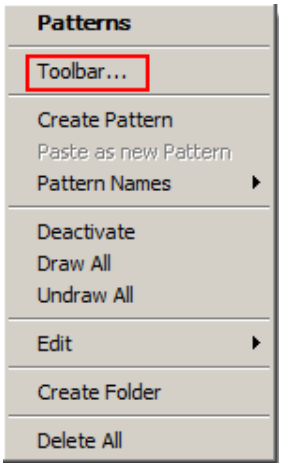
The message confirms the **Active toolpath** to be **collision safe**.

11. Patterns

Introduction

Patterns are **2D** or **3D Wireframe** entities mainly used primarily for toolpath creation. A **Pattern** is either projected onto the model or traced in situ using **Pattern Finishing**. **Patterns** can be used as a medium to take **Wireframes** in and out of the **Wireframe Modeller**. Unlike **Boundaries**, **Patterns** can contain **open segments**.

- Select **File** - **Delete All**.
- Select **Tools** - **Reset Forms**.
- Right click on **Patterns** from the **Explorer** and select **Toolbar**.



- From the top menu select **Create pattern** and this generates an empty **Pattern Named 1** into which you can insert geometry.

When created, the **Pattern** is automatically assigned a number and made **Active** as shown in the **Explorer**.

Pattern segments are created using one or more of the following options:

Automatic Pattern Generation (Open the **Pattern Maker** form).

Insert File into Active Pattern (eg. dgk pic dxf iges -------).

Save Active Pattern as a file (dgk pic dxf).

Insert Boundary into Active Pattern.

Insert Active Toolpath into Active Pattern.

Insert Model into Active Pattern (Selected Model).

Curve modelling.

Wireframe modelling (**PS-Sketcher**).

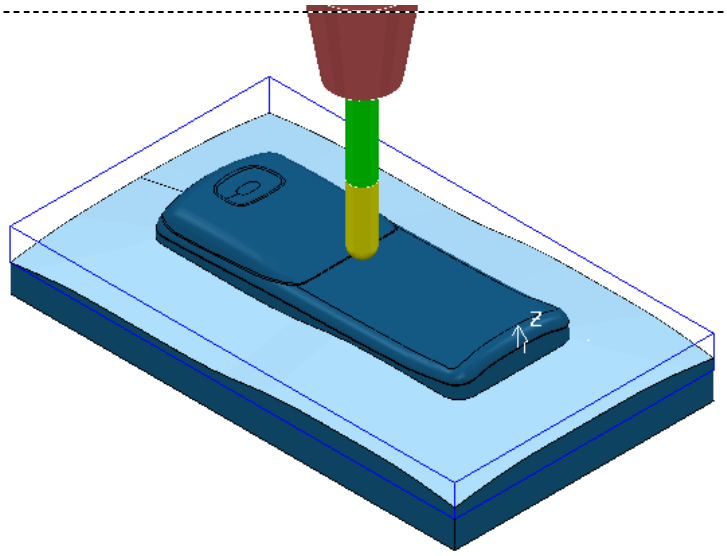
Automatic Pattern Generation

The **Automatic Pattern Generation** icon opens the **Pattern Maker** form. This provides 6 different options allowing the user to create 4 different types of **Offset Patterns** between 2 open segments or to the left and right of a single segment, an **Offset Pattern** inside a closed segment, as well as a **Trochoidal Pattern** across a segment.

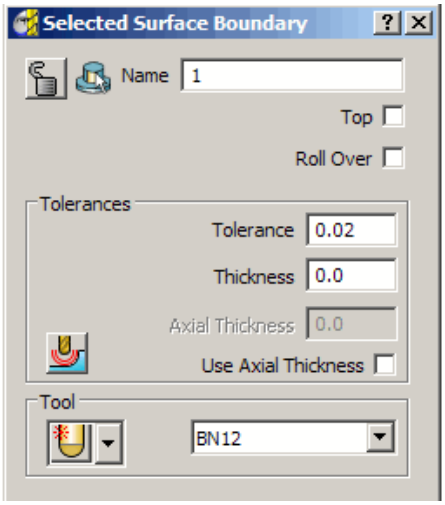
- Select **File - Delete all**.
- Select **Tools - Reset forms**.
- **Import** the **Model**:-
  D:\users\training\PowerMILL_Data\Models\phone.dgk.

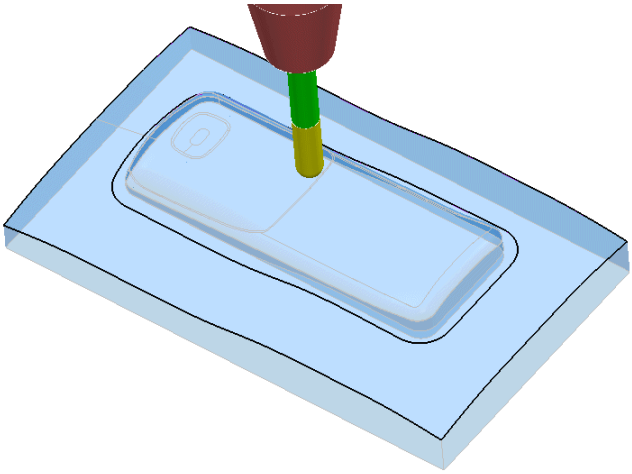
- Select **Block** and **Calculate** a **Block** to model **Limits**.
- Define a **dia 12mm Ball Nose** tool named **bn12**.
- Select **Rapid Move Heights** **and then Reset to Safe Heights**.

- Select **Start and End Points** .
- Select an **Iso1** view to display the **model**.



- Right click on **Boundaries**, select the option **Selected Surface** from the **Create Boundary** flyout menu options and enter the values shown below right.

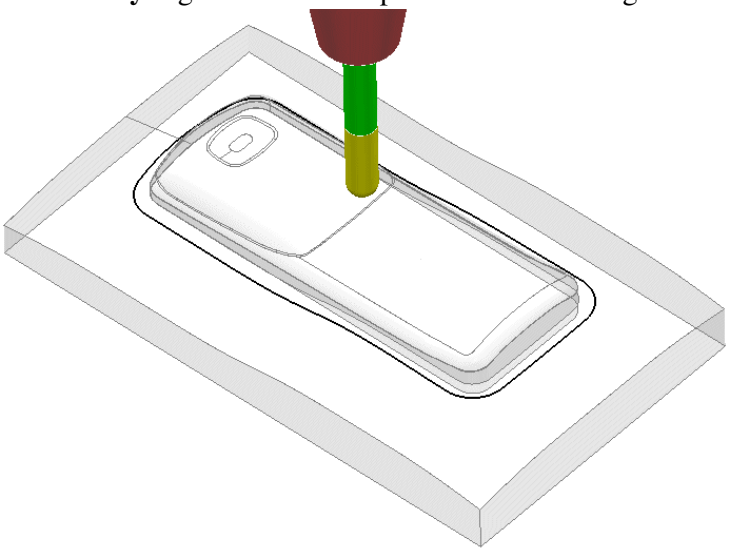




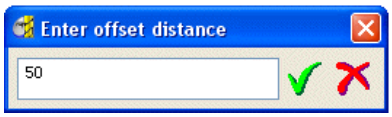
- Create a new (empty) **Pattern**  .
- Select the inner **Boundary segment**.
- In the **Explorer**, right mouse click on the new (empty) **Pattern** and from the local menu select **Insert – Boundary**.
- Input the **Boundary Name** (**1**) and click on the **Green Tick** to accept.

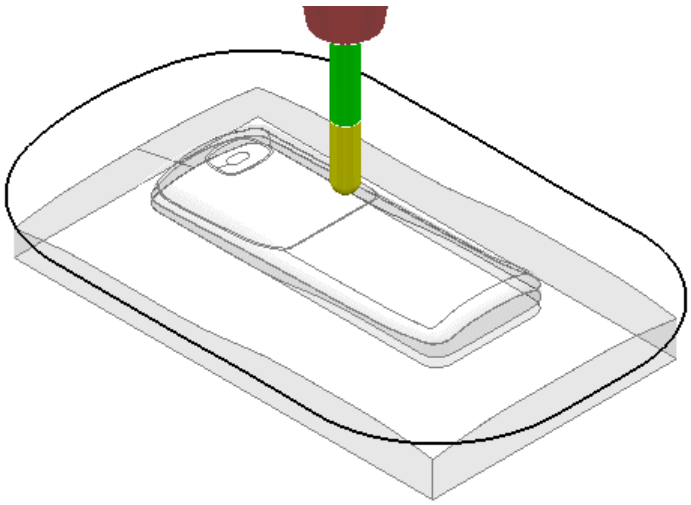
- Undraw the **Boundary**.

The selected **Boundary** segment will be copied as a **Pattern** segment as shown below.



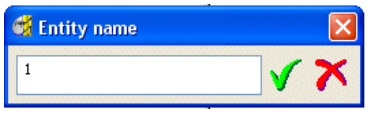
- Select and **right mouse click** on the **Pattern segment** and in the local menu select **Edit – Offset 2D (Round Corners)**.
- Input an **offset distance** value of **50** in the form and click on the **Green Tick** to accept.



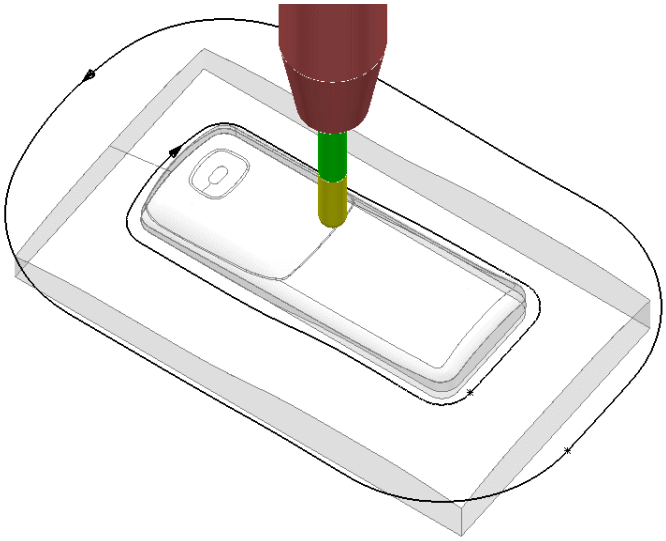


- Select in the **Explorer** the **light bulb** for the **Boundary** to display it again.
- Select the **inner Boundary segment**.

- In the **Explorer**, **right mouse click** on the **Pattern** and from the local menu select **Insert – Boundary**.
- Input the **Boundary Name** (**1**) and click on the **Green Tick** to accept.



- Undraw the **Boundary**.



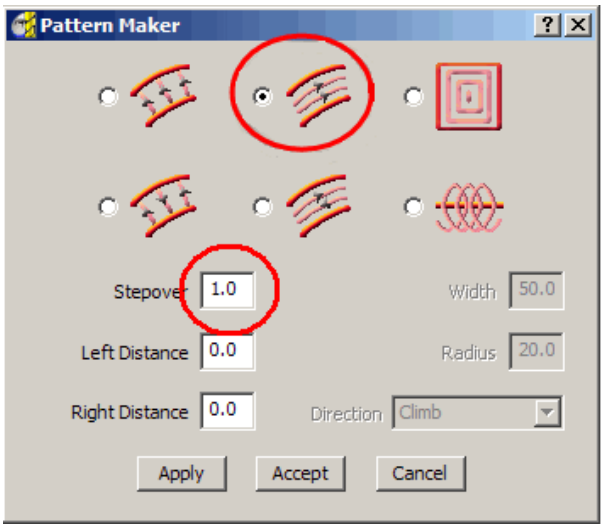
- Right mouse click on the **Pattern** and from the local menu select **Instrument** to display the **direction arrows** on the **segments** as shown in the image above.
- Select the **outer Pattern segment** and from the local menu select **Edit – Reverse Selected** (Both **Pattern segments** should run in the same direction).

The **Automatic Pattern Generator** will be used to create a new **Pattern** with a series of segments merged between the **2 segments** of the original **Pattern**. The resultant **toolpath** will follow the contour of the component form as well as creating corner free, tool tracks with constant material removal rate.

- From the **Pattern toolbar** select **Create Pattern** to create a new **Pattern**.
- Select **Automatic pattern generation** .

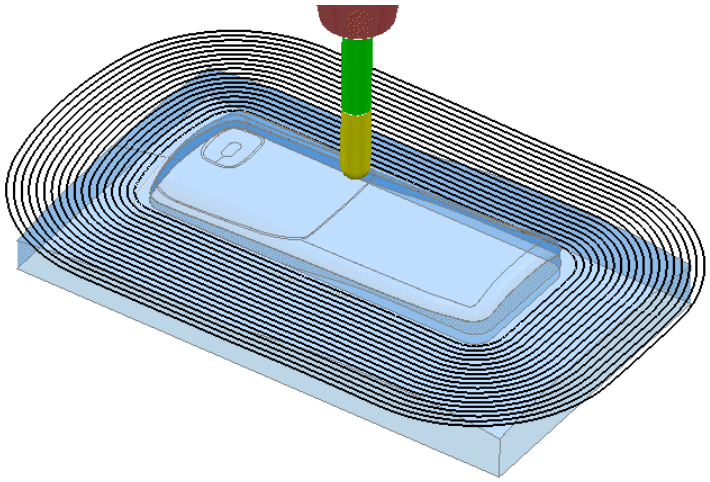
- With the **new Pattern activated** and the **original Pattern** displayed, select **both segments**.

- Select **Create pattern along curves** ***one-way*** and input a **Stepover** value of **1**.



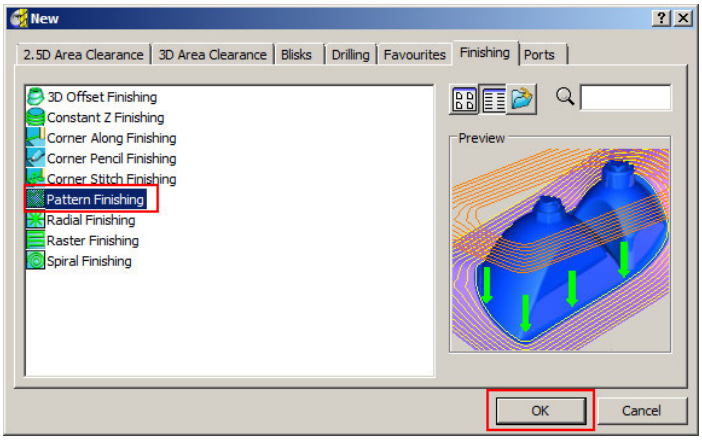
The form allows you to create a range of multi segment **Pattern** styles from drive curves.

- Select **Apply**, followed by **Accept**.

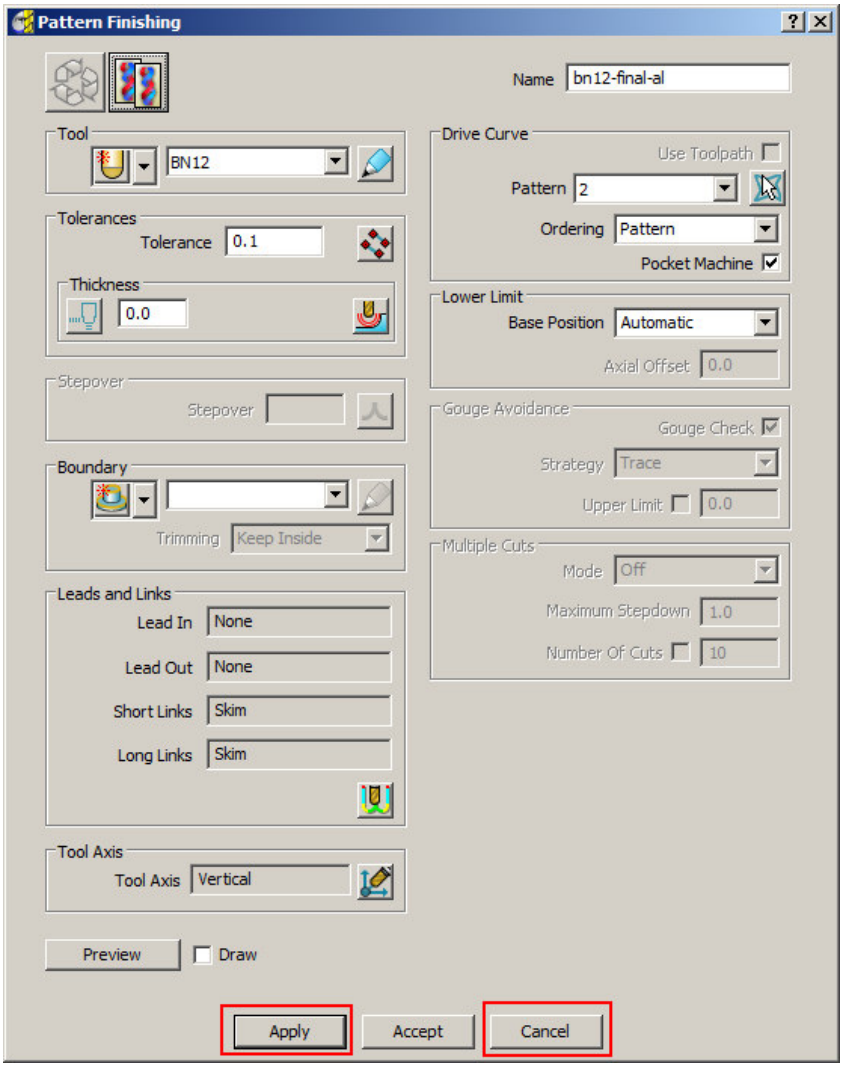


The **new Pattern** has been generated independently; between the **two** selected segments of the **original**, de-activated **Pattern** (the above illustration shows a **2mm Stepover** for visual reasons). The new **Pattern** will now be used in a **Pattern Finishing** strategy.

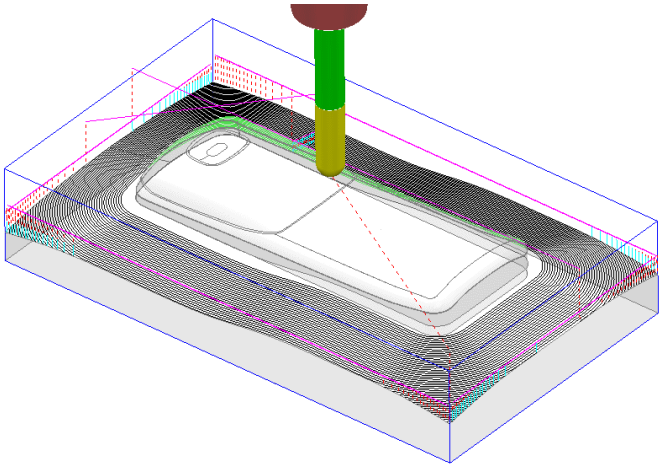
- Select a **Pattern Finishing** strategy.



- Enter the settings in the form exactly as shown below then **Apply** and **Cancel** the form.



- Undraw the **Model**, **Boundary** and **Patterns** to view the **toolpath**.



The **Pattern Finishing**, toolpath is limited to exist within the defined material **Block**.

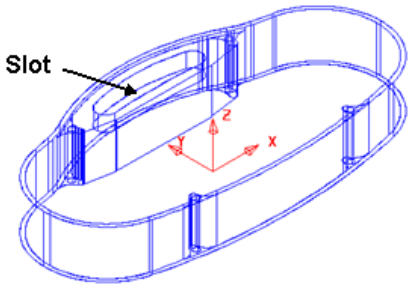
Trochoidal Pattern Generator

The **Trochoidal Pattern** option within the **Automatic Pattern Maker** form is a technique for machining slots especially in **High Speed Machining** applications. The **Trochoidal Pattern** consists of a continuous spiral of advancing loops which have the effect of restricting the tool contact area to only a part of its circumference.

This method requires a tool diameter that is smaller than the slot to be machined. The CNC controller then performs spiral movements to produce the full width slot. Since the tool is no longer cutting at full width the problem of overheating is effectively removed. This is essential where coated carbide cutters are used for **High Speed Machining** of hardened steel.

- Select **File - Delete All**.
- Select **Tools - Reset Forms**.
- Select **File – Import Model** and import the model **dashboard.dgk** from the **Examples** folder.
- Select an **ISO 1** view.

You will machine the **Slot** using the **Trochoidal Pattern** method.

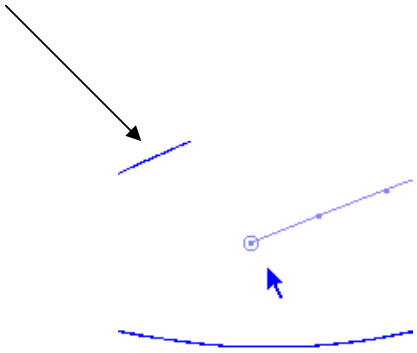


- Define the **Block** to **Min/Max** limits.
- In the **Explorer** right mouse click on **Patterns** and select **Toolbar…**.
- Select **Create Pattern** from the **Patterns** toolbar.
- Select **Insert File into Active Pattern** from the **Patterns** toolbar to load in the file **trochoidal_pattern.pic** from: **D:\users\training\PowerMILL_DATA\Patterns**

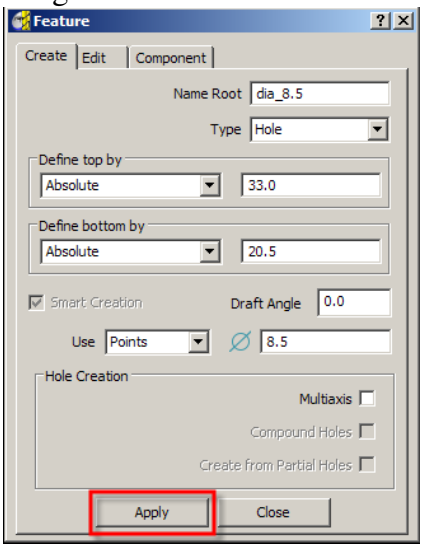
Before producing a **Slot** a **Helically drilled hole** of **diameter 8mm** must be machined to create clearance for a **Slop drill** to **plunge down** to the full **Slot depth**. This is achieved by using **Feature Sets** (Covered in more detail in the next chapter).

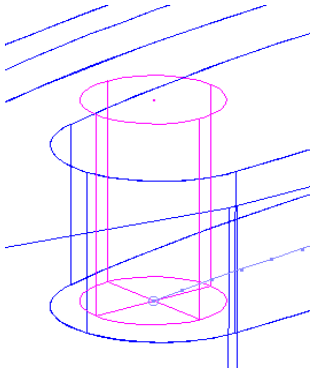
- Define a **Ball Nose** tool of **6mm Diameter**.
- Right mouse click over **Feature Sets** in the **Explorer** and select **Create Feature Set** to open the **Feature Form**.

- **Select** the **curved Pattern** (1st click) and **Shift Select** the **end point** (2nd click) identified by the short arrow.



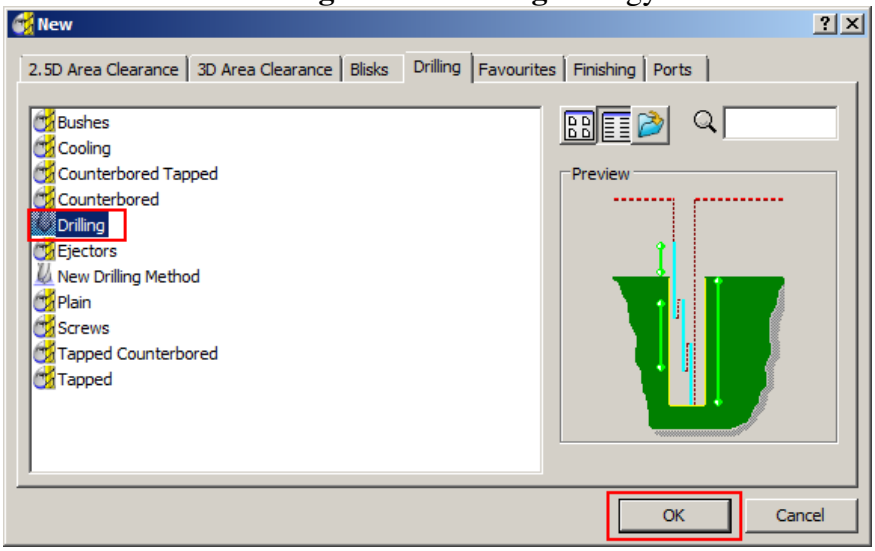
- Select the tab **Create** on the **Feature Form** and enter the values shown in the image below.



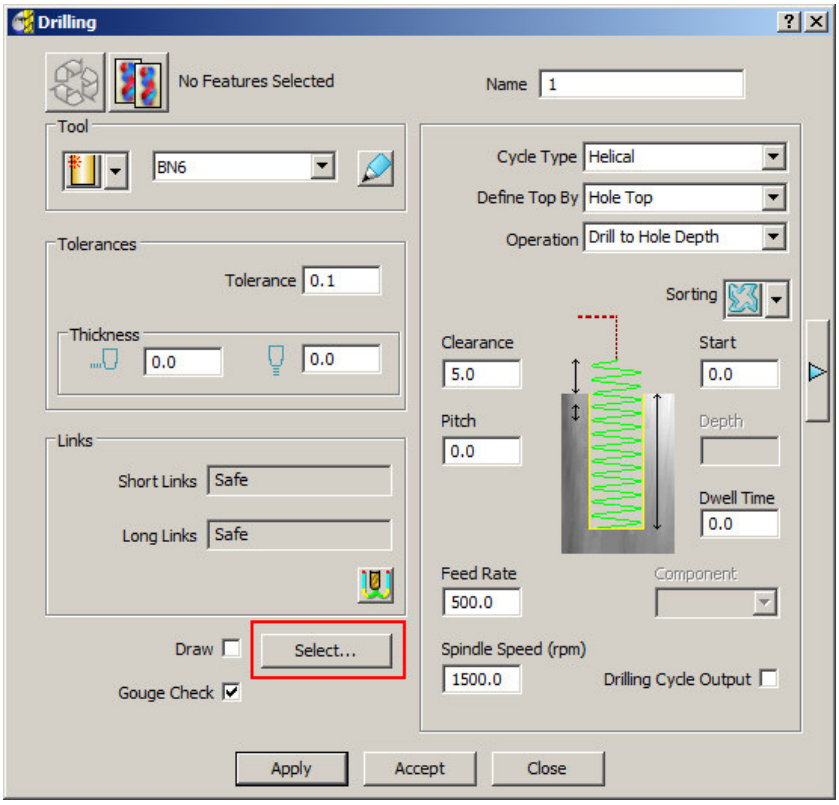


A new **Feature** will be created that represents the hole ready for the **helical drilling**.

- Select **Apply**, followed by **Close**.
- Select **Toolpath Strategies** on the **main** toolbar.
- Select on the **Drilling** tab the **Drilling** strategy.

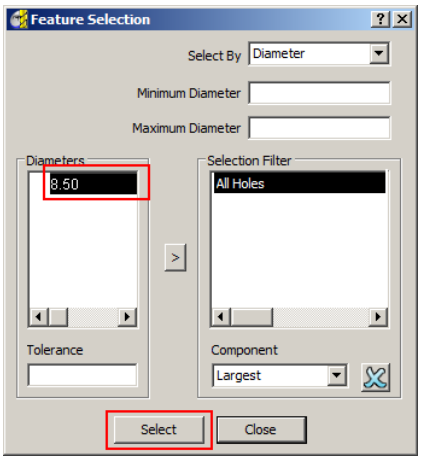


- Select **OK** to open the **Drilling** form.
- Select the **Select…** button.

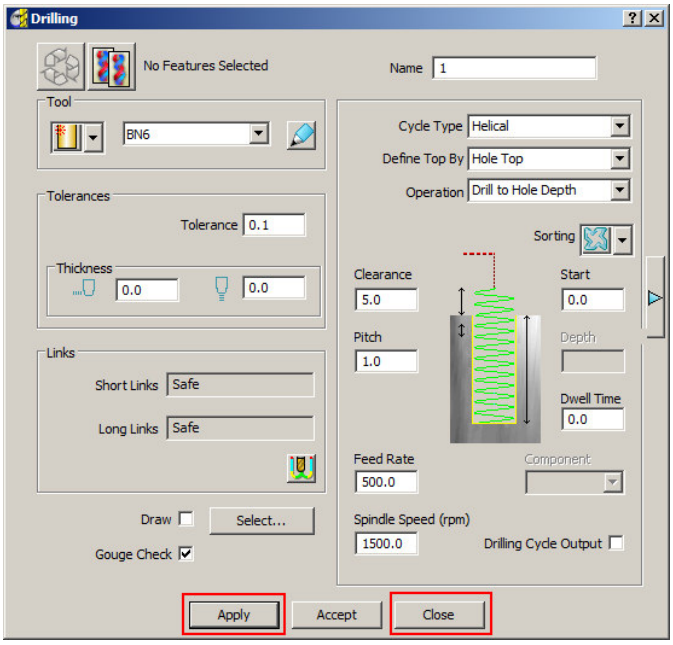


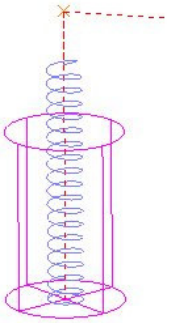
Before the toolpath can be calculated, the **Hole Feature** must first be selected.

- Select the **Diameter 8.5** on the left side of the **Feature Selection** form.
- Select the **Select…** button and then **Close** the form.



- Enter the values as shown below then select **Apply**, followed by **Close**.

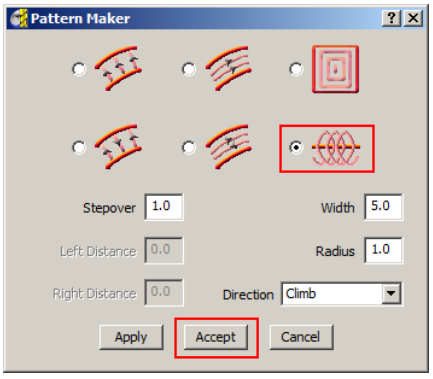


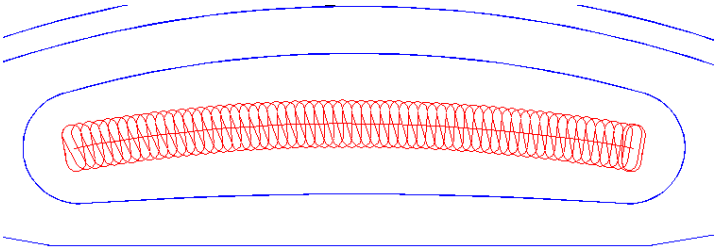


You can see on the **animation** that the tool is **Climb Milling** in an anti-clockwise direction. The **8.5mm Diameter hole** has now provided the clearance necessary for a smaller tool to plunge into the **Slot** prior to the **Trochoidal** milling operation.

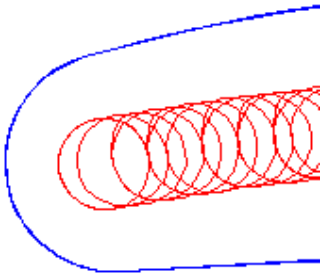
Note: **Feature Sets** are not gouge checked to a **Surface** model so care must be taken when creating them with regard to both size and position.

- Create an **End Mill tool** of **Diameter 6mm** called **em6**.
- Deactivate **Pattern 1** (trochoid_curve), but keep it displayed by toggling the **lightbulb**.
- Create another empty **Pattern** (**2**) which will become **Active**.
- Select the curved **Pattern** (**1**) with the left mouse button.
- Select **Automatic pattern generation** .
- In the **Pattern Maker** form select the **Trochoidal** option and enter the data exactly as shown below. Select **Apply** and **Accept**.



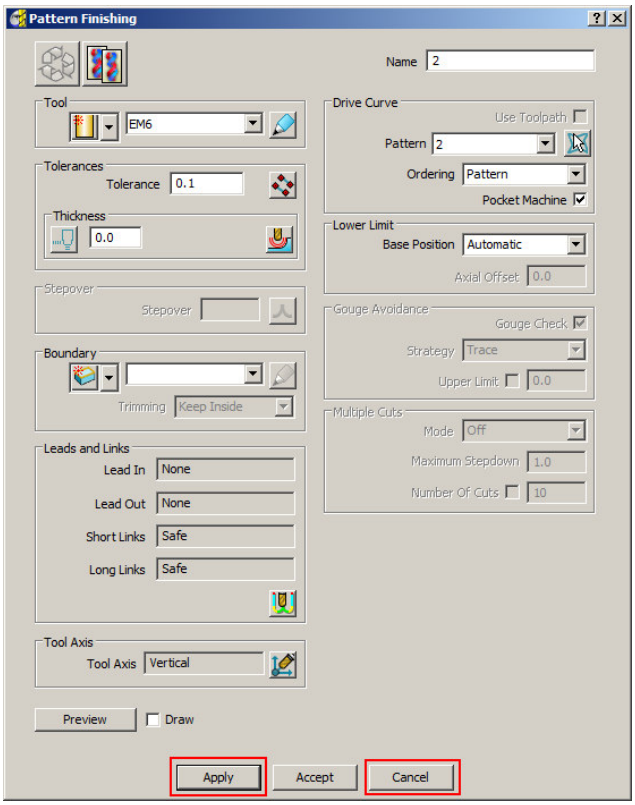


Note: The **Radius** must not be a value greater than half the **Width**. A fine **Stepover** is required when applying to **High Speed Machining** of hardened steel.

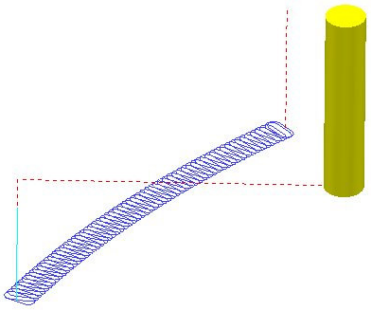


If the **Radius** value used in the **Pattern Maker** form were **2.5** the resultant **Pattern** would appear as shown to the left.

- Select **Toolpath Strategies** .
- On the **Finishing tab**, select the **Pattern Finishing** strategy to open up the **Pattern Finishing** form.
- Enter the values exactly as shown in the image below.



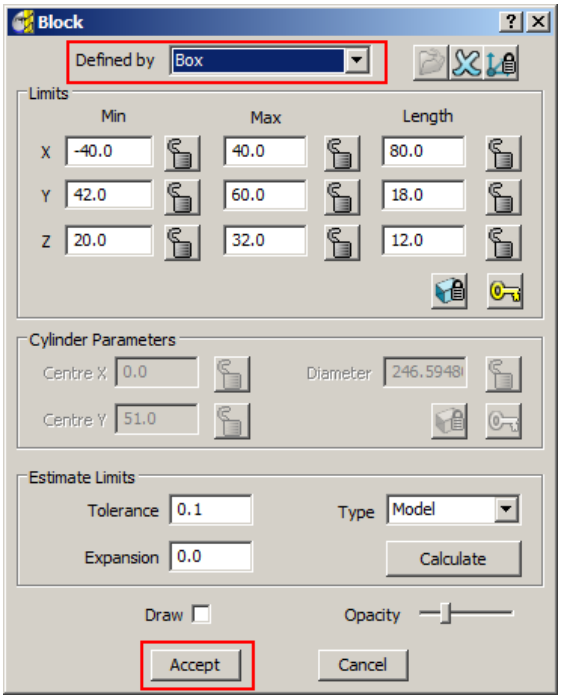
- Select **Apply** and then **Cancel** the form.



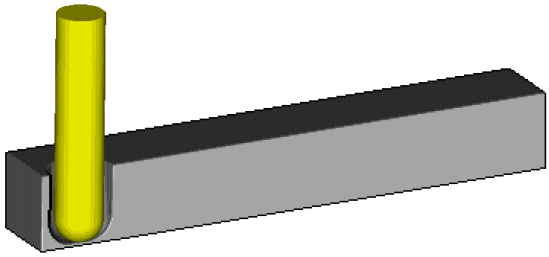
The **6mm End Mill** tool will plunge into the previously created **8.5mm** clearance **Hole**.
The **Trochoidal** toolpath defaults to a **Climb Mill** direction and is ideally suited to **High Speed Machining** applications.

- Activate the **toolpath 1** (drilling) in the **Explorer**.
- Select **Block** .

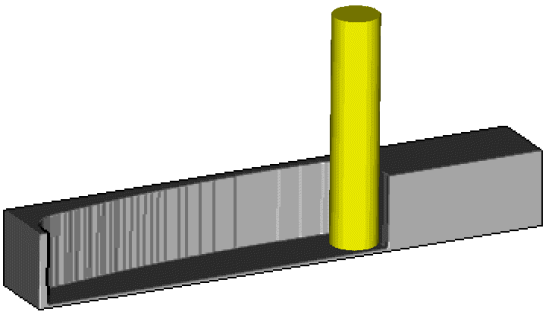
- Select **Defined by** – **Box**.
- Fill in the form exactly as shown for the **Limits Min** and **Max** to create a **Block** definition locally along the back half the slot width.



- Select **Accept**.
- Position the **Block** in the graphics area in preparation for toggling into **Viewmill**.
- Select **ViewMill On/Suspend** on the **ViewMill** toolbar.
- Select **Plain Shaded Image** .
- Check both the **Helical** and **Trochoidal** toolpaths in turn with **ViewMill**.



Helical drill toolpath.



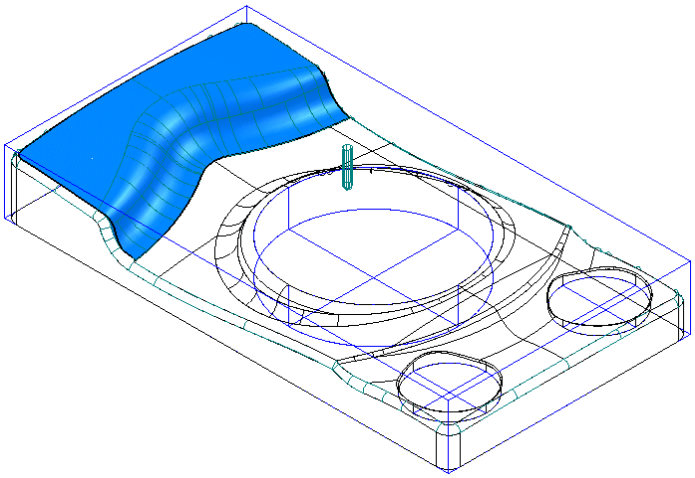
Trochoidal mill toolpath.

Pro - Patterns

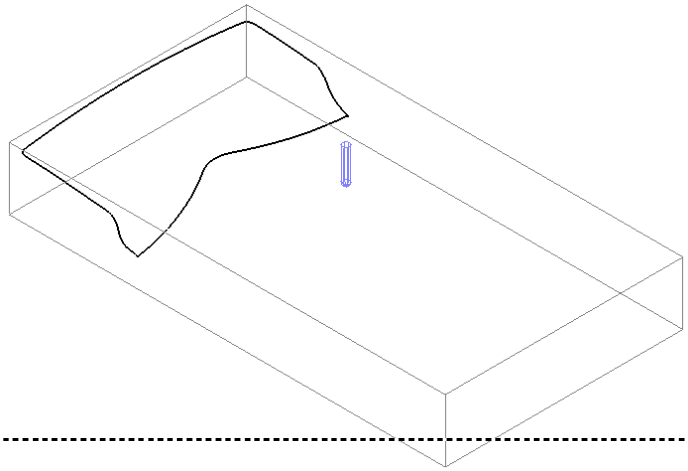
Patterns applied to 3D Offset machining

A **Pattern** can be used as the basic shape to be offset across the machining area of a **3D Offset Finishing** strategy. The following exercise requires the creation of a **Selected Surface Boundary** along with a **Pattern** (created along the one of the sides of the **Boundary**) to control both the toolpath shape and order across a **3D Offset Finishing** strategy.

- From the top ***pull down*** menus select **File** - **Delete All**.
- Import the **model** - **Examples/speaker.dgk** and select an **Iso3** view.
- Define a **∅6 Ball Nose** with the name **bn6**.
- Define a material **Block** to **Min/Max limits**.
- In **Rapid Move Heights** click **Reset to Safe Heights**.
- Set both the tool **Start and End Points** to **Block Centre Safe**.
- Select the **3** shaded **Surfaces** shown in the following diagram.

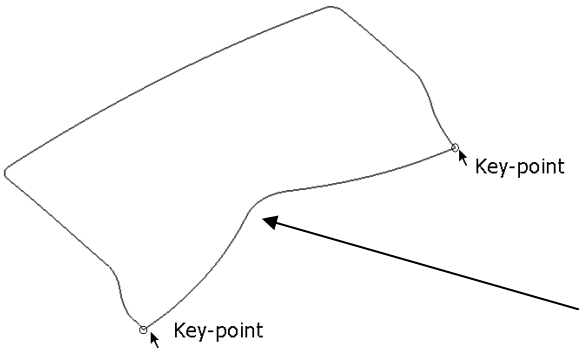


- Create a **Selected Surfaces Boundary** to a **Tolerance** of **0.02**.



- From the **Pattern** toolbar click the **Create Pattern** icon followed by the **Insert Boundary into active pattern** icon.

- Undraw both the **model** and **Boundary** to visually isolate the **Pattern**.



Only the leading edge of the **Pattern** is required. The remaining parts are to be removed.

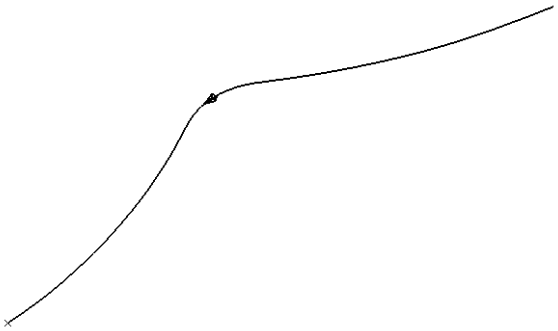
- From the **Pattern** toolbar, select **Curve Modelling** to open the following toolbar.

Define Start Point ***Define End Point***

- Select ***Define Start Point*** and left mouse click at the left of the leading edge as marked in the above diagram with a small arrow and labelled as **Key-point**.
- Select ***Define End Point*** and left mouse click at the right of the leading edge as marked in the above diagram with a small arrow and labelled as **Key-point**.
- Use the left mouse key and click on the lower part of the **Pattern** to define which segment is required (identified with a large arrow in above diagram).

Save ***Exit***

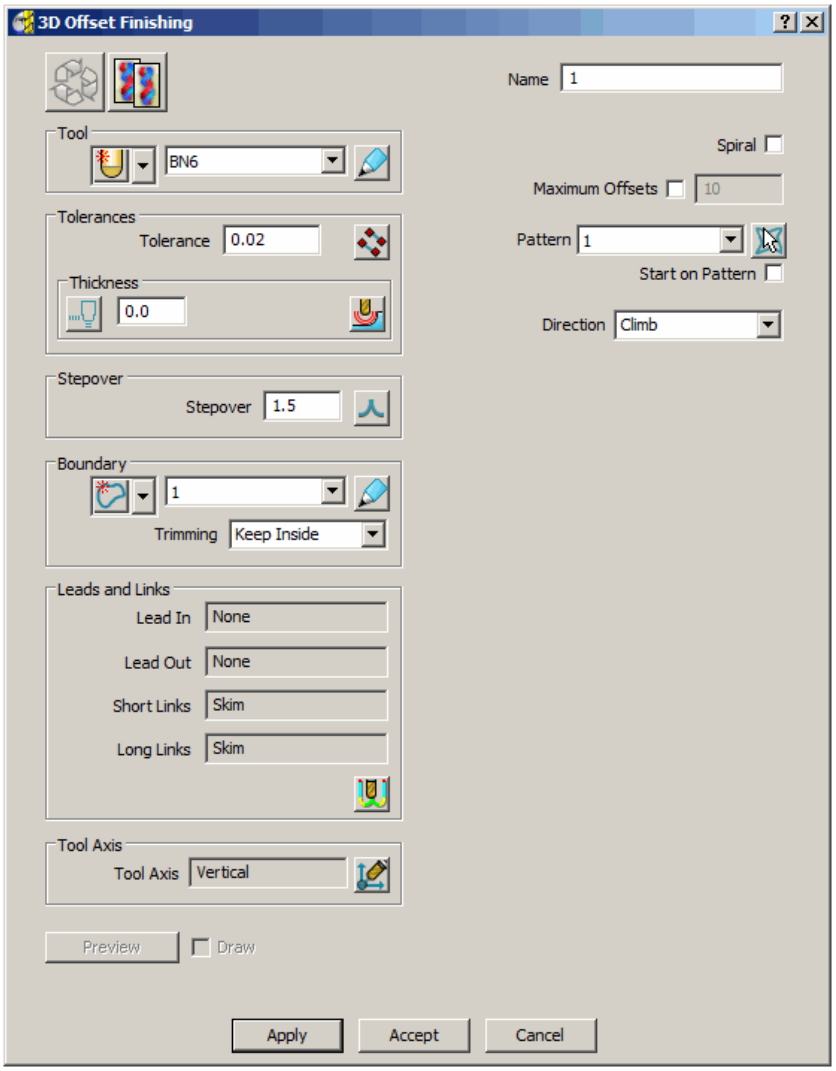
- Select ***Save*** and then ***Exit*** the **Curve Modelling** toolbar.
- At the top of the original segment use the **left mouse key** to **select** and **right mouse key** to access the local **Pattern** menu and select **Edit - Delete Selected Components** leaving the newly defined, bottom leading edge.

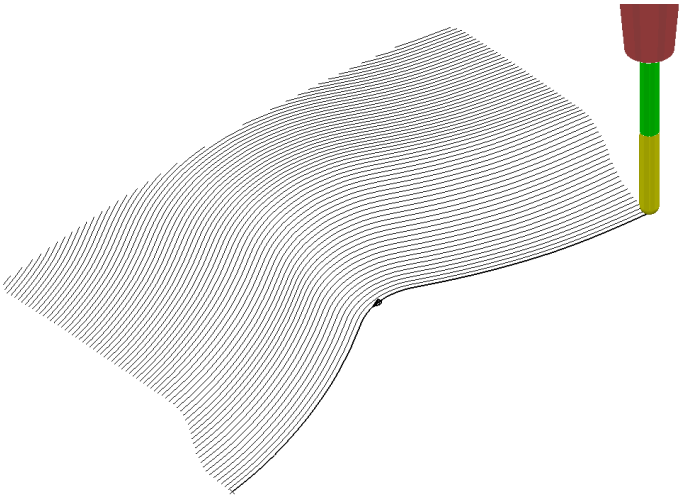


- Also in the local **Pattern** menu select **Instrument** to display the direction of the segment.
- If it is not running in the direction shown select the segment and apply **Edit - Reverse Selected**.

This provides the user with a method of dictating the final machining direction.

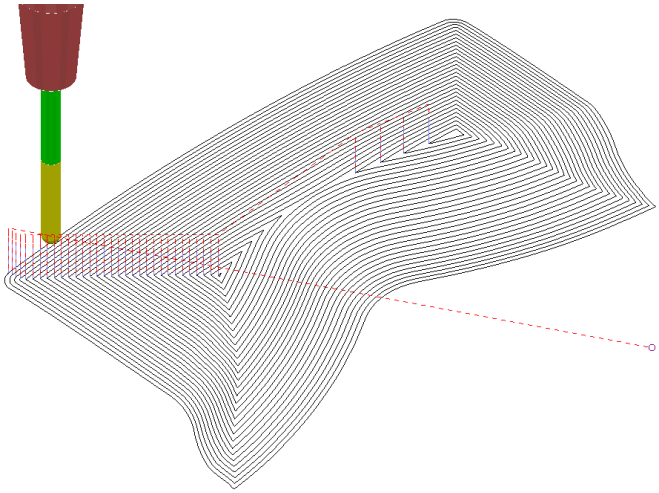
- Select a **3D Offset Finishing** strategy and enter data into the form exactly as shown below.





Draw Leads and **Draw Links** have both been switched off for a clearer view of the toolpath.

The above toolpath starts at the lowest point following the **Pattern** while offsetting across the limiting **Boundary** in a **Climb** milling direction. With the addition of suitable **Lead In** and **Lead Out** moves this is an ideal strategy for ***High Speed Machining***.

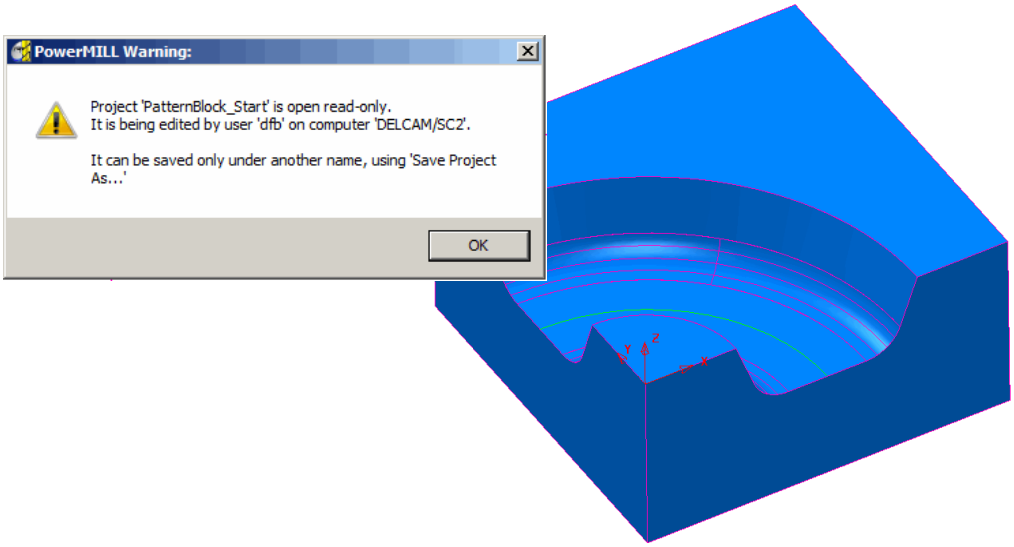


If a **Pattern** is not selected in the **3D Offset** form machining strategy follows the shape of the **Boundary** segment as shown left.

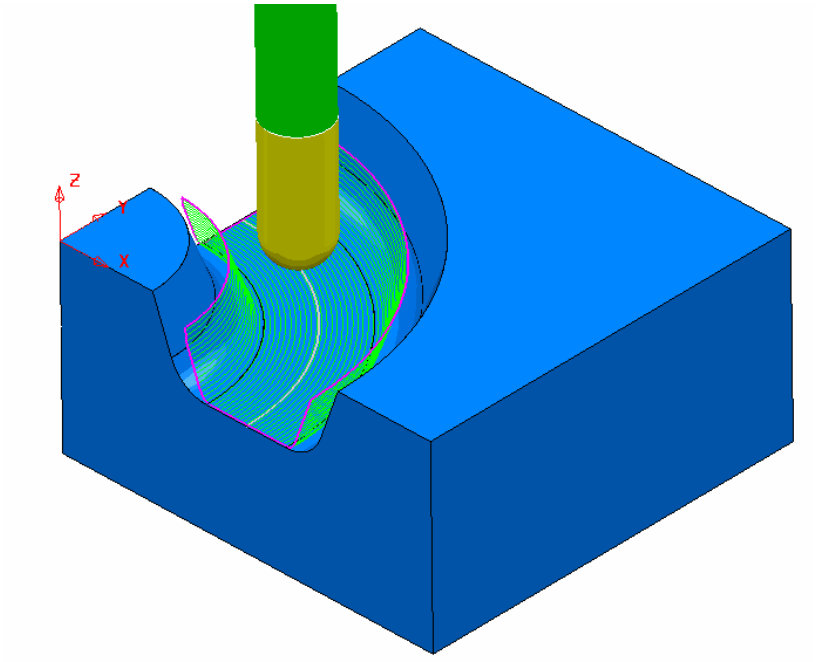
3D Offset Finishing controlled by Pattern Exercise

Note:- It is not compulsory for the **Pattern** used to control the **toolpath** to be coincident with the limiting **Boundary**. In the following exercise it is required to create a **toolpath** trimmed **Inside** a **Boundary** with the tool tracks following a **Pattern** running centrally along the base of a circular recess:-

- Select **Delete All** and **Reset Forms**.
- Open the read only ***Project:- D:\users\training \PowerMILL_data\Projects\PatternBlock_Start***



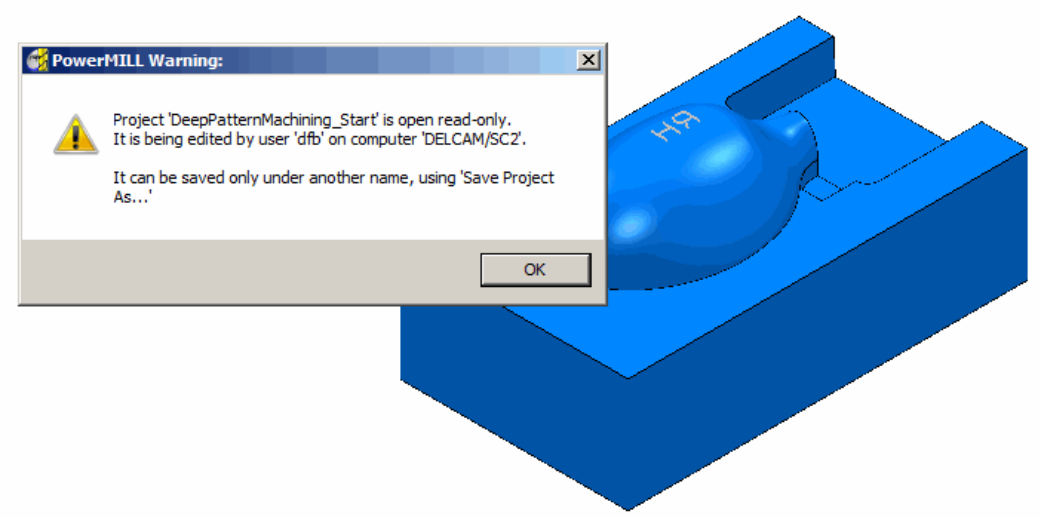
- Select **OK** to close the ***PowerMILL Warning*** and **Save Project As:-** ***D:\users\training\COURSEWORK\PowerMILL-Projects\PatternBlock_ex1***
- Create a material **Block**, ***Defined By*** - **Box** to the **Model** ***limits***.
- In the **Rapid Move Heights** form select **Reset to Safe Heights**.
- Create a **Pattern** using **Insert Model** the ***wireframe arc*** (included with the **Model**) running centrally along the recessed form.
- Select the **5 Surfaces** that define the recessed form and create a ***Selected Surfaces*** **Boundary** using a **0.01 tolerance**, relative to the ***active*** **BN16** tool.
- Create a **3D Offset Finishing** strategy that is limited ***Inside*** the **Boundary** with the order and shape of the ***tool tracks*** controlled by the **Pattern**.



Deep Pattern machining

If a **Pattern Finishing** is applied with a **-ve Thickness** it will only create the strategy if the inserted value is less than the **tip radius** of the defined cutting tool. As an extreme example, this makes it impossible to apply a **thickness** less than **zero** in cases where an **End Mill** is being used.
It is however possible to produce a **Pattern Finishing** strategy that machines deeper into the model by first projecting the **Pattern** to be flush with the surface (relative to the tool geometry) and then by using different options in conjunction with a **-ve Axial Offset**.
Note:- The strategy produced will not be based on an **3D offset** of the ***model form*** but will be a series of ***copies*** of the original **Pattern** stacked downwards. As a result it is not recommended for use on steep areas of the model.

- Select **Delete All** and **Reset Forms**.
- Open the read only ***Project:- D:\users\training \PowerMILL_data\Projects\DeepPattern_Start***

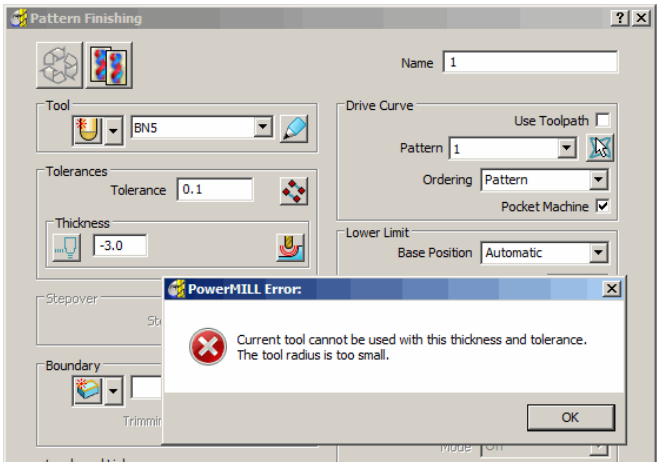


- Select **OK** to close the ***PowerMILL Warning*** and **Save Project As:- *D:\users\training\COURSEWORK\PowerMILL-Projects\DeepPattern_ex1***

The **Project** consists of component **surfaces** and a **Pattern** on **Z0** representing the text **'LH'** ready to be engraved into the component form to a depth of **-3**.

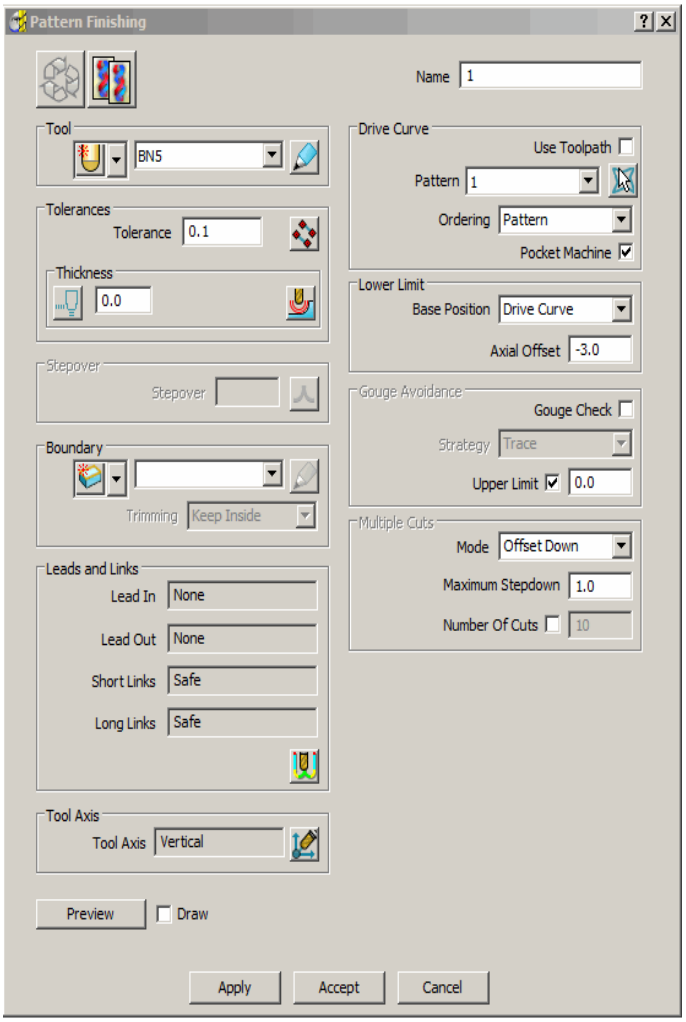
- **Calculate** a material **Block**, ***Defined By-*** **Box**, using ***Type-*** **Model**.
- Create a **Dia 5 Ball Nosed** tool with the ***Name*** **BN5**.
- In the **explorer** right mouse click over the actual ***Pattern*** **1** and from the local menu select **Edit - Drop** to project it downwards onto the model (This projects the **Pattern** down **Z**, adjusted to the geometry of the **Active** tool **BN5** and the current finishing **thickness** value).
- From the **main** toolbar, select **Toolpath Strategies**  and in the form select **Finishing** followed by the **Pattern Finishing** option.

- Enter the data into the form exactly as shown on the following page and **Apply**.



The **Thickness** of **-3.0** exceeds the **Tip Radius** of the **Dia 5 Ball Nosed** tool hence instead of a toolpath appearing, the above **PowerMILL Error** form is displayed.

- Select **OK** to close the form and modify the data in the **Pattern Finishing** form as shown below.

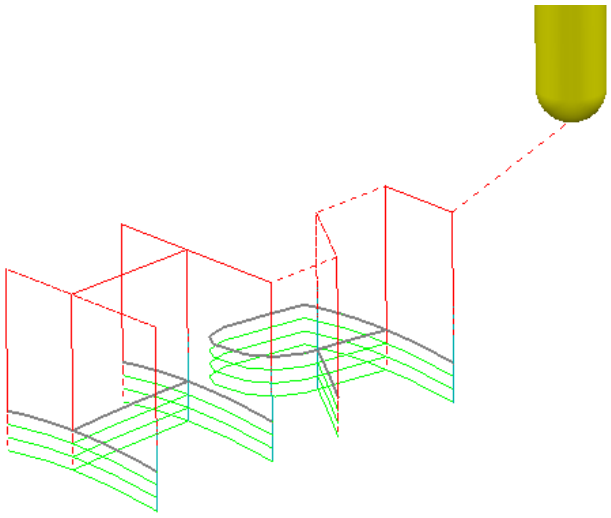


Select ***Base Position*** **Drive Curve** and input ***Axial Offset*** **-3.0**

Switch **Gouge Check** off (untick). Tick **Upper Limit**.

Set **Offset Down** with ***Maximum Stepdown*** **1.0**

- **Apply** the form.



The Reversed lettering, **Pattern** '***RH'*** is now recessed into the Punch form to a depth of 3mm.
Note: The above **toolpath** will be correctly identified in the **explorer** with a **red gouge warning**.