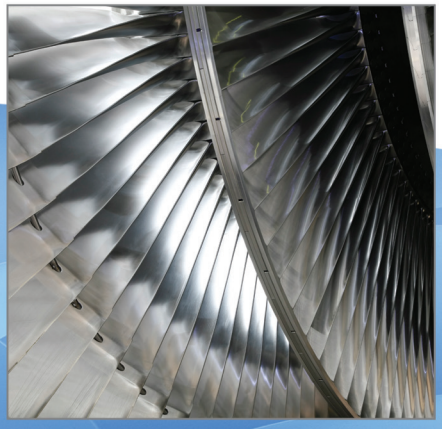
Delcam Advanced Manufacturing Solutions

Powering your productivity



PowerMILL

Training Course

Important Notice

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

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

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

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

Copyright © 2013 – Delcam plc. All rights reserved.

PowerMILL 2013 Contents

	Chapters	Page Number
Day 1	1. Getting Started	1.1 - 1.26
	2. Machining Set Up in Detail	2.1 - 2.22
	3. Area Clearance Strategies	3.1 - 3.44
Day 2	4. Finish Machining Strategies	4.1 - 4.70
	5. Leads and Links	5.1 - 5.14
	6. Boundaries	6.1 - 6.26
Day 3-4	7. Levels and Sets	7.1 - 7.10
	8. Editing Toolpaths	8.1 - 8.22
	9. Collision Checking	9.1 - 9.14
	10. Patterns	10.1 - 10.24
Day 4-5	11. Curve Editor	11.1 - 11.12
	12. 2D Machining	12.1 - 12.42
	13. NC Programs	13.1 - 13.10
	14. Setup Sheets	14.1 - 14.12
Day 5	15. Specialist Finishing	15.1 - 15.38
	16. Component Thickness	16.1 - 16.22
	17. Stock Models	17.1 - 17.18

Copyright © Delcam plc

Copyright © Delcam plc

1. Getting Started

This course covers the 3-Axis functionality available in **PowerMILL**. Additional **Five Axis** license options are covered on a separate course.

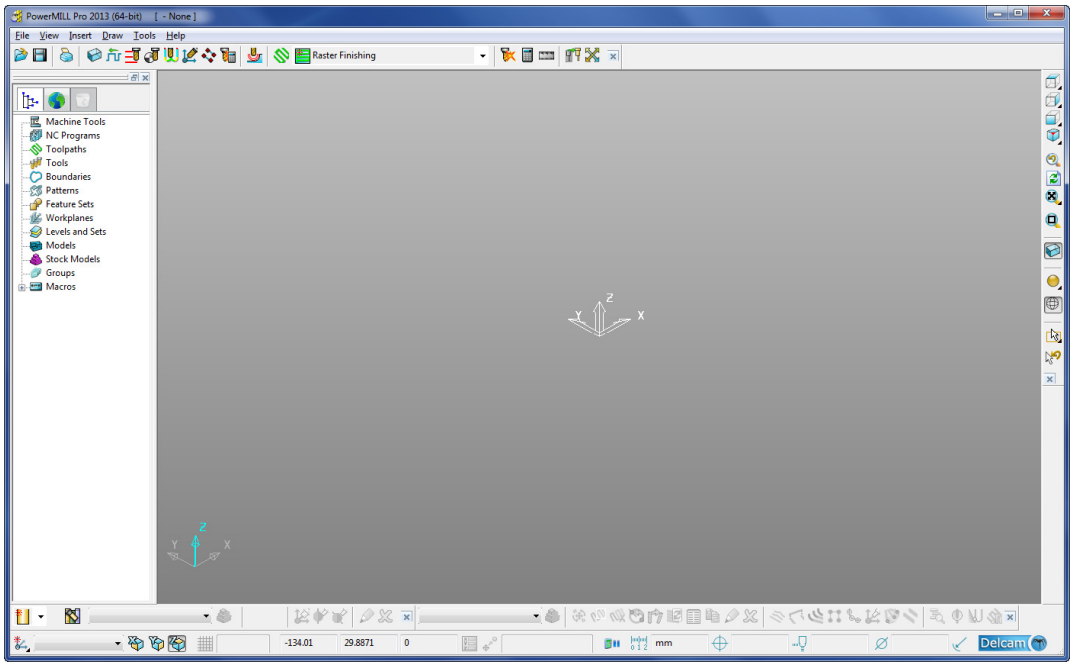
PowerMILL can quickly create gouge-free cutter paths on imported component data. It supports Wireframe, Triangle, Surface, and Solid models created by other Delcam products or from neutral formats such as IGES. If the relevant PS-Exchange translators are purchased, PowerMILL will also directly import data created by other of proprietary CAD packages.

Start PowerMILL

1 Double-click the **PowerMILL** shortcut on the desktop:

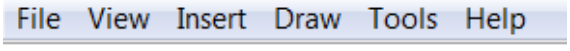
*On the training PC, the icon will be displayed as **PowerMILL**.*

The **PowerMILL** screen is displayed:



The screen is divided into the following main areas:

a **Main menu**

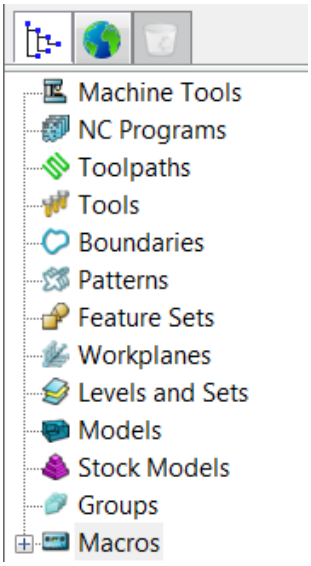


Click one of the menu names on this bar (for example, **File**) to open a pull-down menu list of associated commands and sub-menus. A sub-menu is indicated by a small arrow to the right of the text (for example **File - Recent Projects >**). Highlighting this arrow generates a list of commands/names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

b Main Toolbar

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

c Explorer



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

d Graphics Window

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

e View Toolbar

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

f Information Toolbar

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

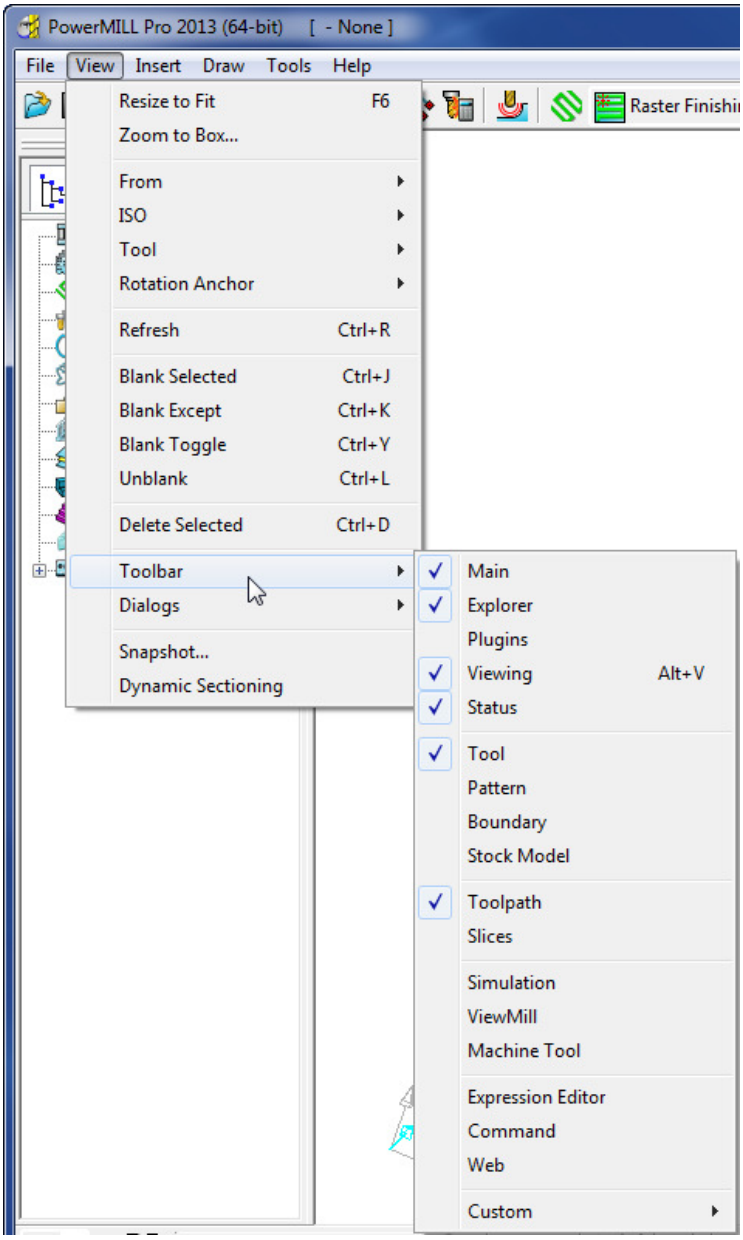
g Tool Toolbar

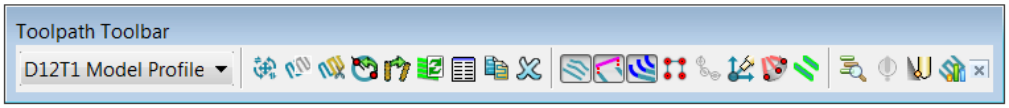
Facilitates the rapid creation of tools in **PowerMILL**.

Copyright © Delcam plc

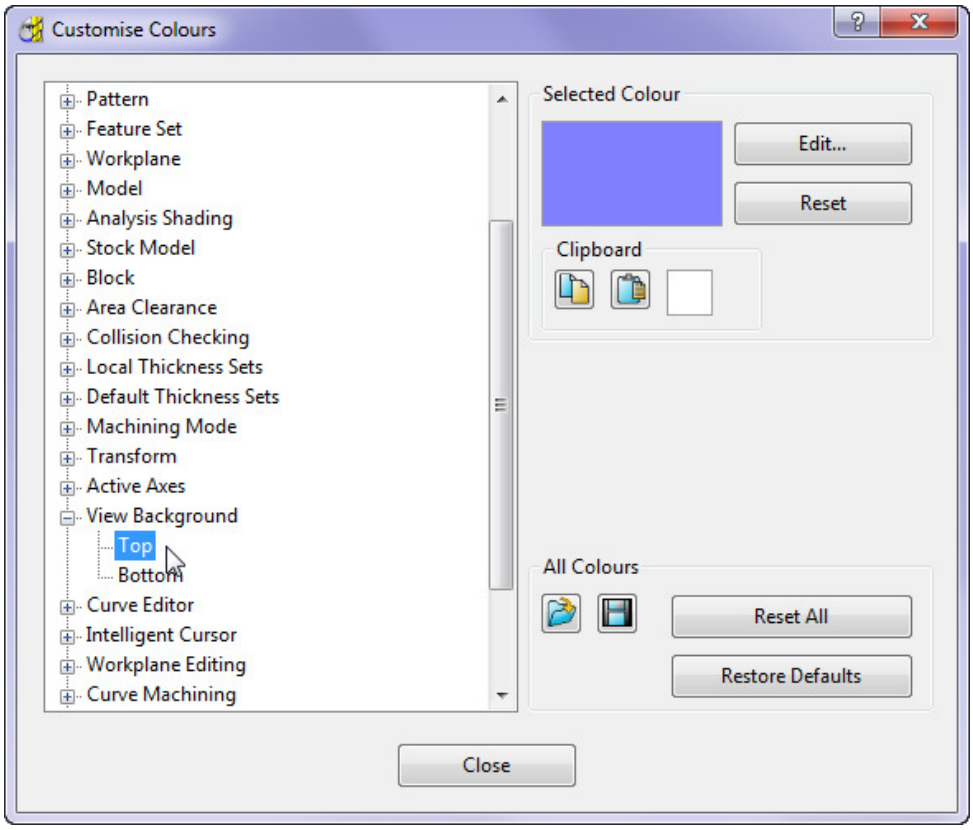
The other toolbars are not factory defaults, and are therefore not shown at initial startup. To display any of these, select them under **View > Toolbar**.

For example, select **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 **PowerMILL** is opened.

Copyright © Delcam plc

Mouse buttons

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

Mouse button 1: Pick and Select

This button is used for selecting items on menus, options on dialogs, 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 if the mouse is clicked, then the item turns yellow indicating that it is selected.

If you select another surface, all currently selected items are de-selected.

- If you want to select more surfaces to add to the currently selected surface, then, while making selections, hold down the **Shift** key on your keyboard.
- If you want to remove a surface from your current selections, hold down the **Ctrl** key on your keyboard and click on the surface that you want to remove.

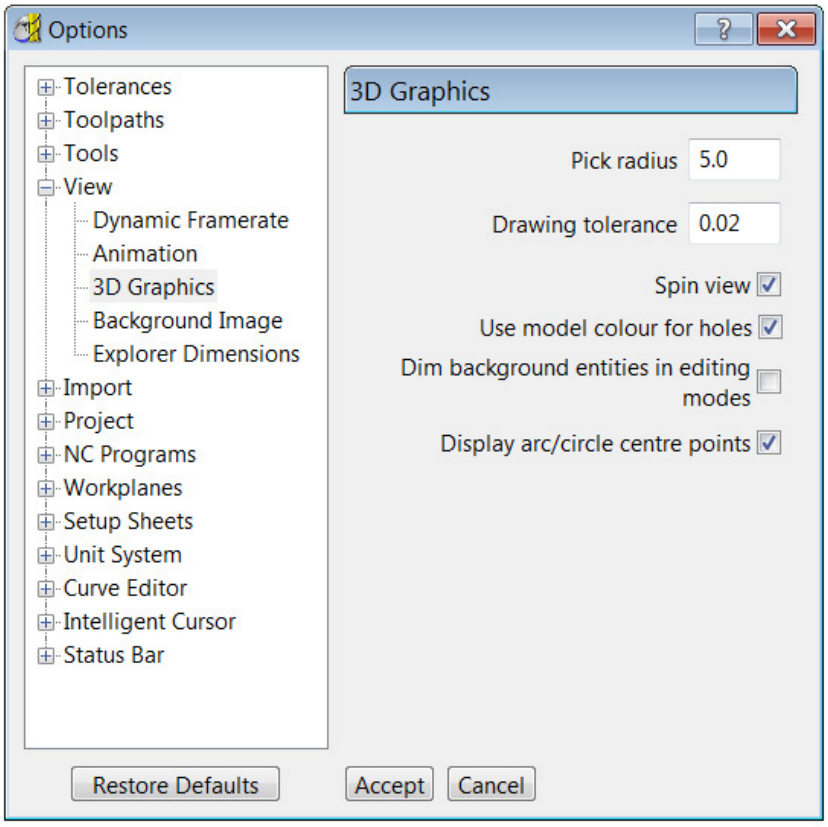
Select by dragging the cursor

Use this option to select multiple entities by dragging the cursor across the required items. This is ideal for quickly selecting multiple surfaces on a model. To deselect entities, press the **Ctrl** key when 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 centred 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 select the **Spin View** option to enable it.



Mouse button 3: Special Menus & PowerMILL Explorer Options

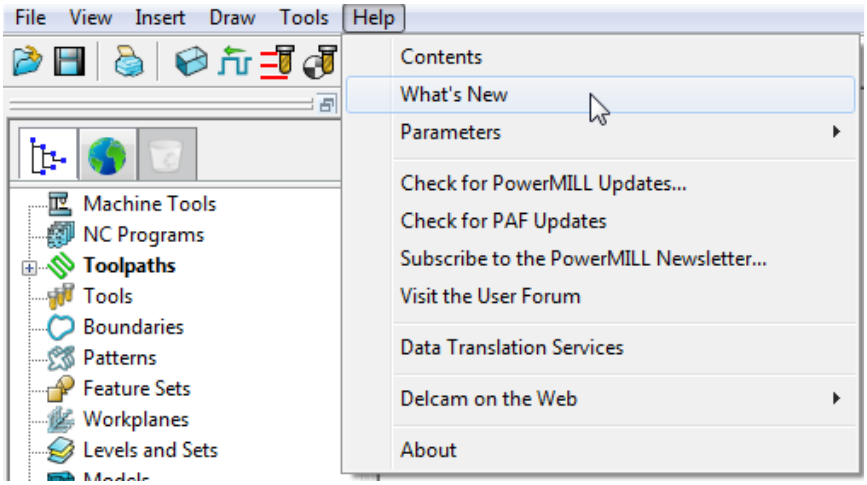
When this button is pressed it brings up a local menu relating to whatever the mouse is over, such as a named item in the **PowerMILL explorer** or a physical entity in the graphics area. If nothing specific is selected, the View menu appears.

Copyright © Delcam plc

HELP!

PowerMILL provides you with help in a number of ways:

- **Tooltips**: Place the mouse cursor over a menu button. A description is of what the button does is displayed.
- **Online Help**: Select Help > Contents from the Main Toolbar to access the online help documents. There is a full index and search facility provided.
- To see a summary of all new functionality available in the current version of **PowerMILL**, select Help > What's New.



- **Context Sensitive Help**: Press the **F1** key to display the help page for the currently active dialog. Click the button in the top right hand corner of the dialog, followed by a left click in any of the input fields to focus on the help topic for that part of the dialog.
- **PowerMILL User Forum**: On the **Main** toolbar, select **Help** > **Visit the User Forum**, to participate in web-based user discussions on **PowerMILL** issues.
- The link to access the forum from any other internet connection is: **http://forum.delcam.com/**
- **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.

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:

a Start up **PowerMILL**.

b Import a **Model**.

c Define the **Block** from which the part will be cut.

d Define the cutting **Tools** to be used.

e Define set up options (**Rapid Move Heights – Start and End Point**).

f Create a **Roughing** Strategy.

g Create a **Finishing** Strategy.

h **Simulate** the toolpaths.

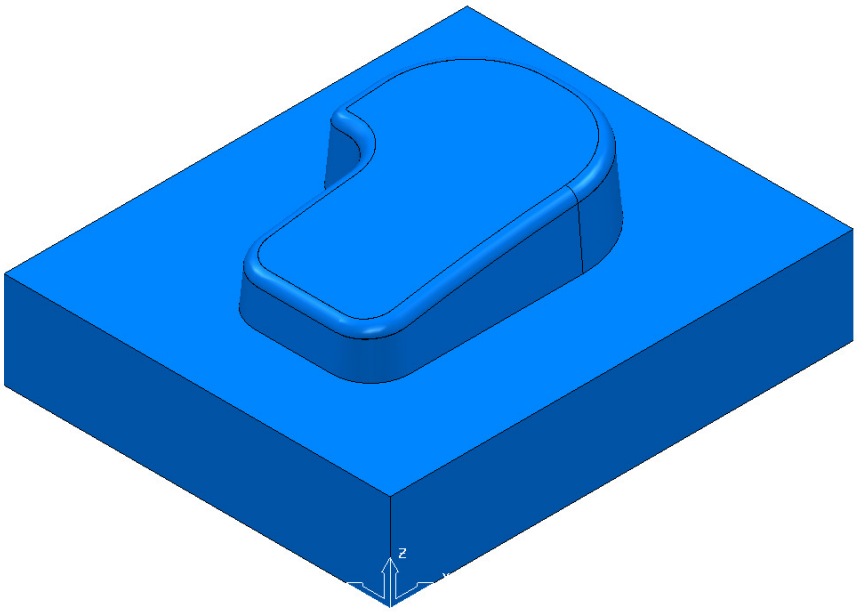
i Create an **NC Program** and output as a post-processed NC data file.

j **Save** the **PowerMILL Project** to an external directory.

Import a Model

2 From the **Main** menu select **File > Import Model** and browse for the model file:

.....\PowerMILL_Data\Models\PmillGettingStarted.dgk

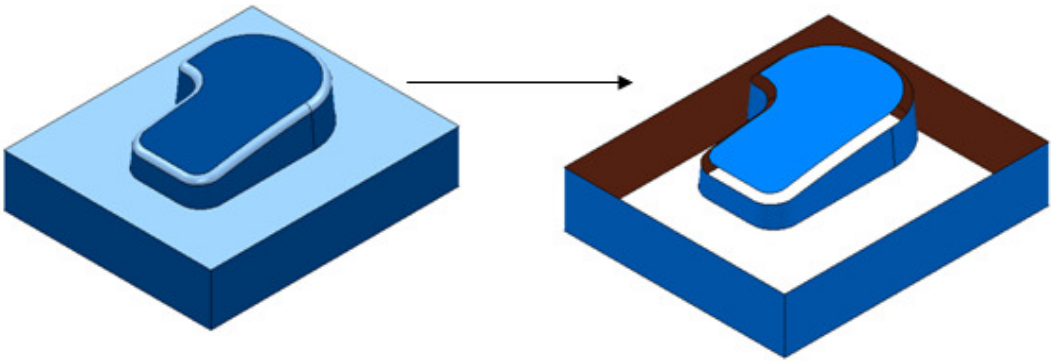


 Copyright © Delcam plc

Blanking model entities

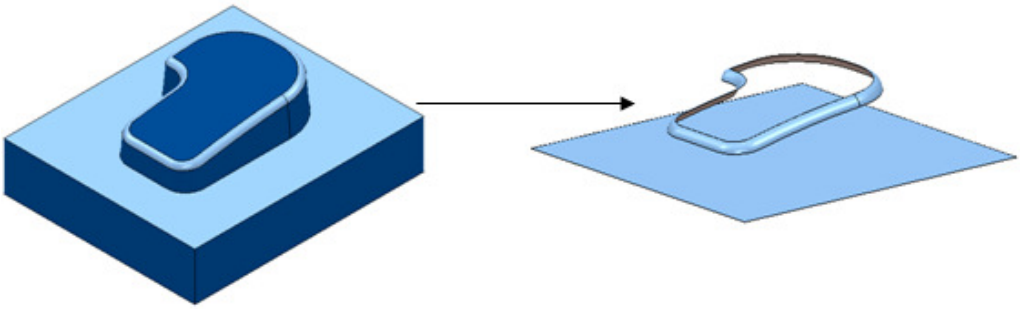
Use the **Blank Selected** (**Ctrl + J**) option available on the **Model** context menu to temporarily hide the selected model components.

In the illustrations below, the ***light blue*** surfaces are selected.



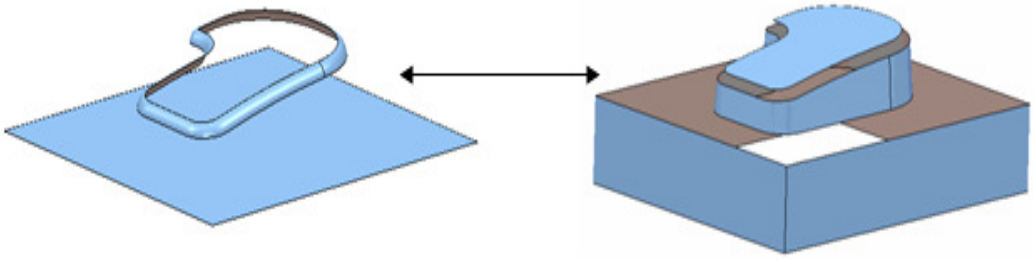
Use the **Blank Except** (**Ctrl K**) option available on the **Model** context menu to temporarily hide all model components except the selected ones.

The 2 selected items are light blue.



To re-display all model components, right-click in the graphics area and select **Unblank (Ctrl L)**. The blanked model components are displayed and selected (back to the left hand image above).

You can also toggle between the blanked and unblanked status of model components. Select the **Blank Toggle** (**Ctrl Y**) option to switch between the ***Blanked*** and ***Unblanked*** components of the model.



Copyright © Delcam plc

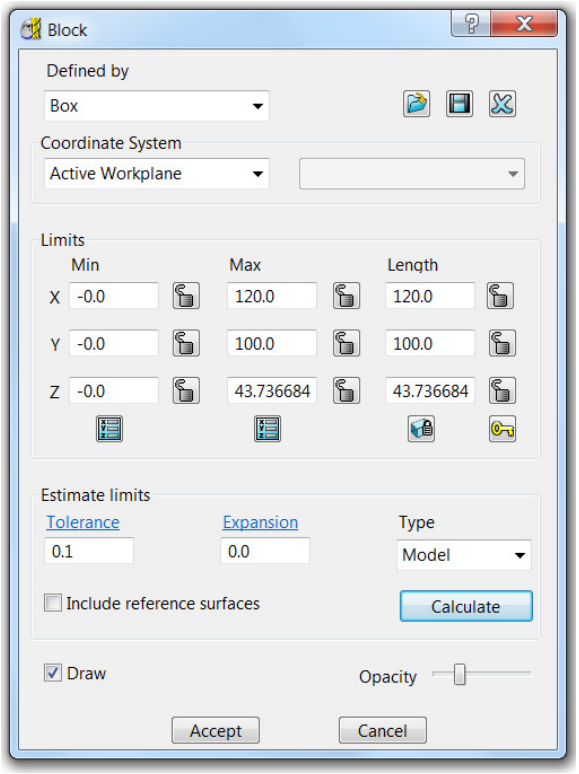
Defining a Block

The **Block** dialog 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 settings on the **Block** dialog are set by default to **Defined by – Box**. Individual values in the dialog can be edited or locked (greyed out) as required. Enter a value in the **Expansion** field to include an offset in the block calculation.

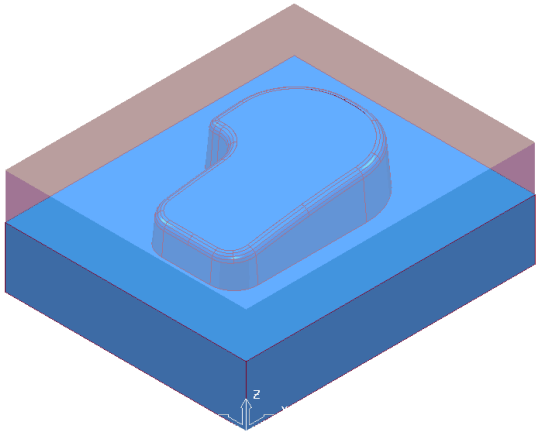
Click the **Calculate** button to define a block around the model dimensions.

1 Click the **Block** button on the **Main** toolbar to display the **Block** dialog.



2 Click the **Calculate** button.

3 Click **Accept**.



 Copyright © Delcam plc

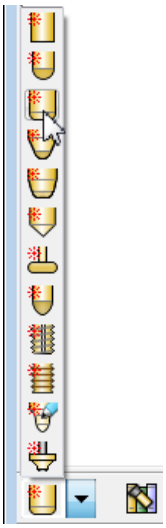
Defining a Cutting Tool

The **Tool** dialogs are accessed using the various tool buttons on the **Tool** toolbar. This is located to the bottom left corner of the graphics area.

For this example, 2 tools will be created, A **Tip Radiused** tool for roughing out and a **Ball Nosed** tool for finishing.

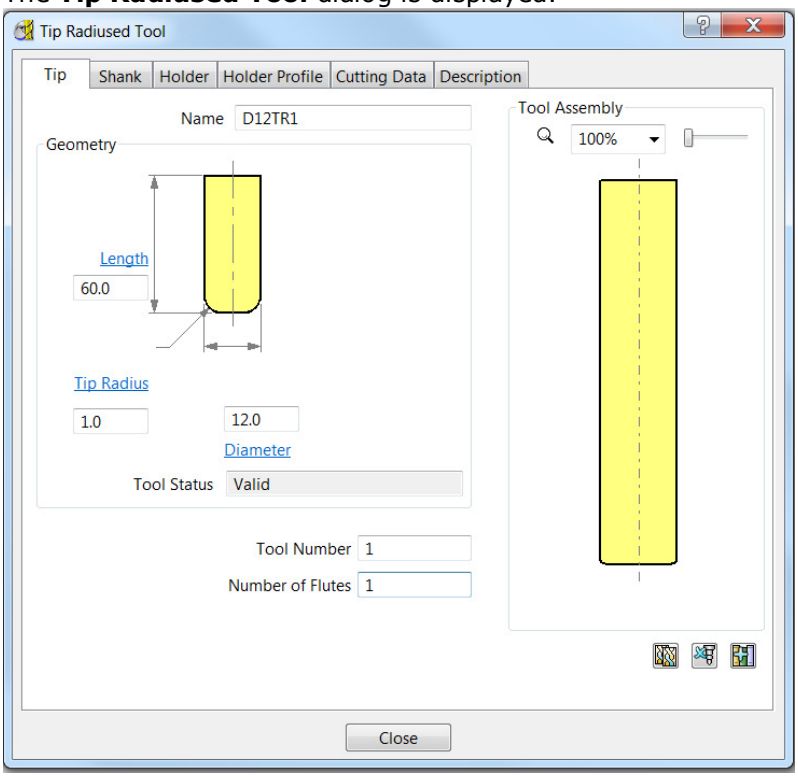
1 Click the down arrow to display all the **Tool** options .

2 Place the cursor over a tool to view the description of the tool type (Tool tips).



3 Select the Create a **Tip Radiused** tool button .

The **Tip Radiused Tool** dialog is displayed.



4 In the **Tip Radiused Tool** dialog, enter:

a **Name** – **D12TR1** It is recommended that the tool is renamed with a more relevant identification.

b **Diameter** – **12** When you enter a value for the **Diameter**, the tool Length automatically defaults to five times this value. You can edit this value if needed.

c **Tip Radi**us – **1**

d **Tool Number** – **1**, A **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.

e Click **Close** on the **Tip Radiused Tool** dialog.

5 Define another tool, this time, select the **Create a Ball Nosed tool** button and in the **Ball Nosed Tool** dialog, enter:

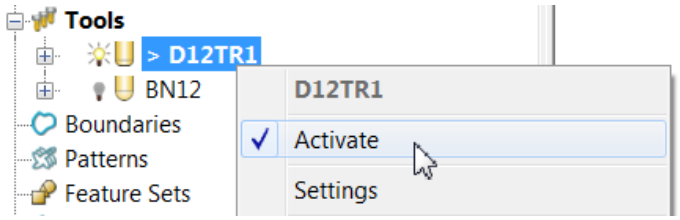
a **Name** – **BN12**

b **Diameter** – **12**

c **Tool Number** – **2**

d Click **Close** on the **Ball Nosed Tool** dialog.

6 In the **PowerMILL *explorer*** panel on the left of the screen, open **Tools** and right mouse click **D12TR1** tool to raise the local menu and select **Activate**.



*Only one tool can be active at any one time and the word **Activate** in the tool context menu is prefixed with a tick* ✔ Activate*. The active tool is automatically included in a strategy dialog when the strategy is created.*

*In the **PowerMILL explorer**, the **Active** tool is displayed in **bold text** and prefixed with '**>**'.*

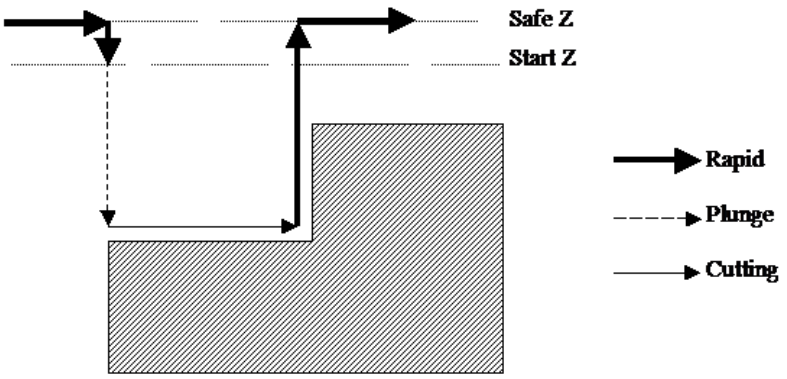
Rapid Move Heights

The **Rapid move heights** options provide safe control rapid tool movements across the component.

- **Safe Z** is the height above the job at which the tool can move at a rapid feedrate, clear of any obstructions such as the work piece or clamps.
- **Start Z** is the height to which the tool will descend, at rapid feed rate prior to applying the plunge feed rate.

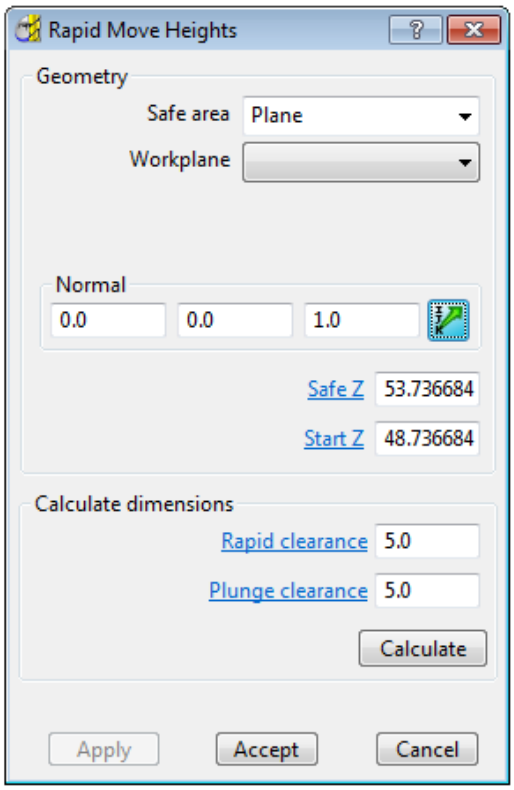
PowerMILL displays rapid moves as dotted red lines, plunge moves as pale blue, and cutting moves as green.

 Copyright © Delcam plc



Setting up Rapid Move Heights

1 On the **Main** toolbar, click the **Rapid Move Heights** button to open the **Rapid Move Heights** dialog.



2 Click **Calculate**.

3 Click **Accept**.

This automatically sets the absolute **Safe Z** and **Start Z** values to be above the block by the distance in the incremental height fields shown at the bottom of the dialog.

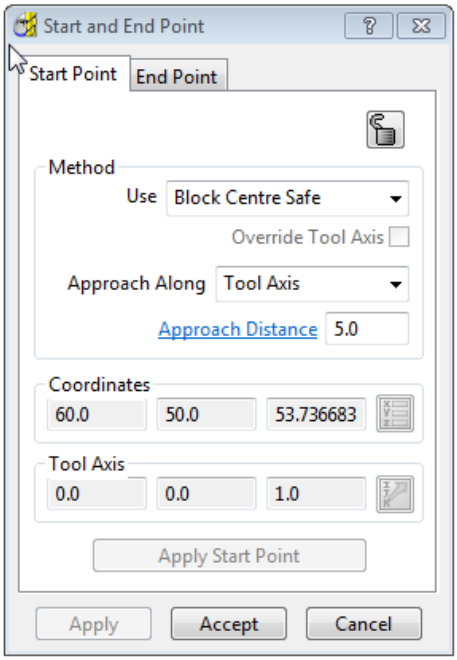
An **Absolute** setting will always cause the tool to feed down from the same height.

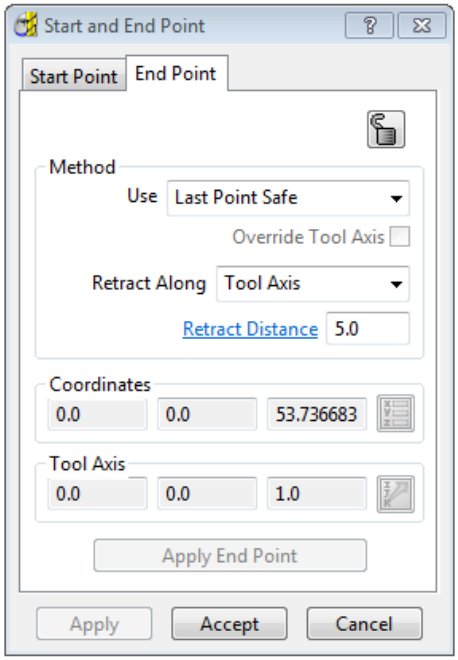
Tool Start and End Point

The **Start and End Point** dialog allows you to define a position where the tool travels to before and after a machining strategy.

Click the tool **Start and End Point** button on the **Main** toolbar to display the **Start and End Point** dialog.

By default, the tool **Start Point** is set to **Block Centre Safe** and the **End Point** is set to **Last Point Safe**.

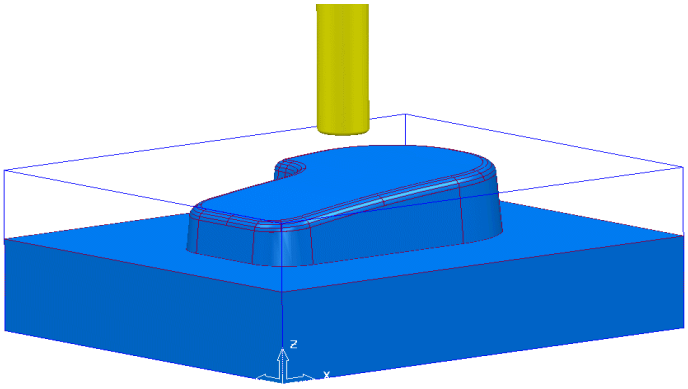




Other **Start and End Point** settings are configured by selecting the different options in the **Method** area on the dialog.

These include **Block Centre Safe**, **First/Last Point Safe**, **First/Last Point**, and **Absolute**.

For now, click **Accept** to use with the default settings.



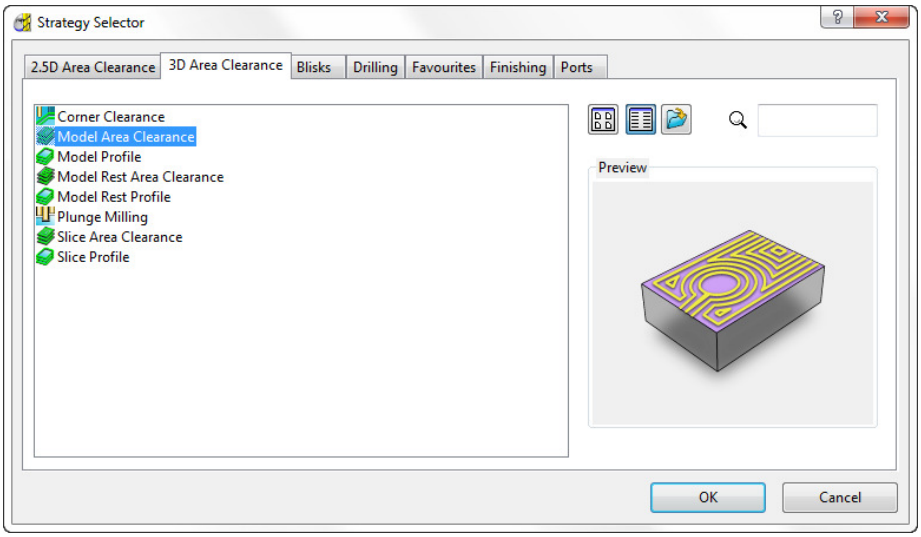
The **D12TR1** tool is placed at the **Block Centre Safe** position ready for you to create the first toolpath.

Copyright © Delcam plc

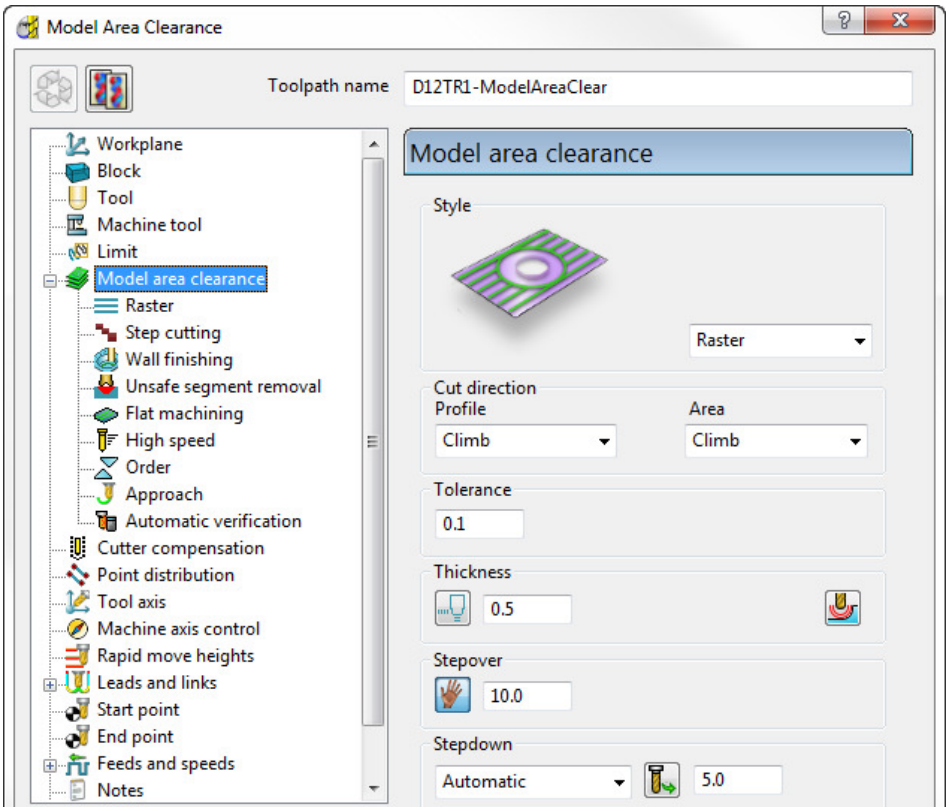
Creating a Roughing Strategy

1 From the **Main** toolbar select the **Toolpath Strategies** button.

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



3 Select the **Model Area Clearance** strategy. This displays the **Model Area Clearance** strategy dialog.



4 In the **Model Area Clearance** strategy dialog:

a In the **Toolpath** name field, enter **D12TR1-ModelAreaClear**.

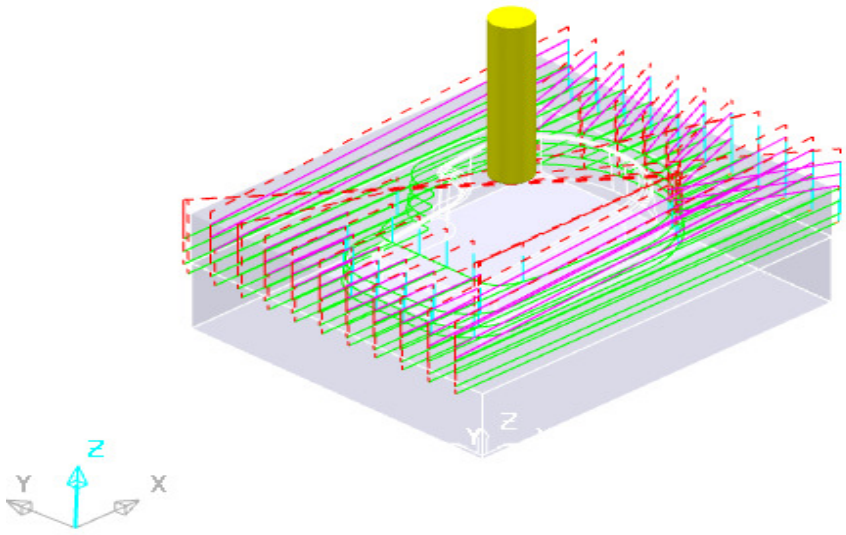
b Select **Style** - **Raster**.

c Edit the **Thickness** value to be **0.5** (This is the amount of material that will be left on the job).

d Edit the **Stepover** value to be **10** (This is the distance between each raster pass (the width of cut)).

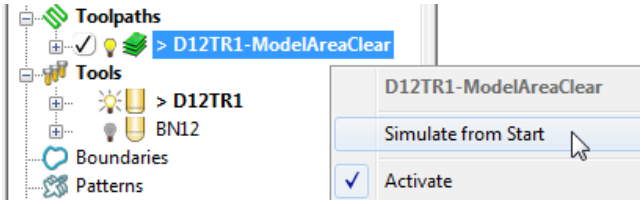
e The **Stepdown** value (depth of cut) is left at the default value of **5**.

f Click **Calculate** to process the machining strategy.



Simulating the toolpath

5 From the **PowerMILL *explorer*** right mouse click on the **Toolpath D12TR1-ModelAreaClear** and from the local menu select, **Simulate** from Start.



The **Simulation** toolbar will appear just above the graphics area.

6 From the **Simulation** toolbar click the **Play** button to run the **Simulation**.

*The other buttons in the **Simulation** toolbar allow you to perform other actions such as **Pause**, **Step**, **Search**, **Goto**.*

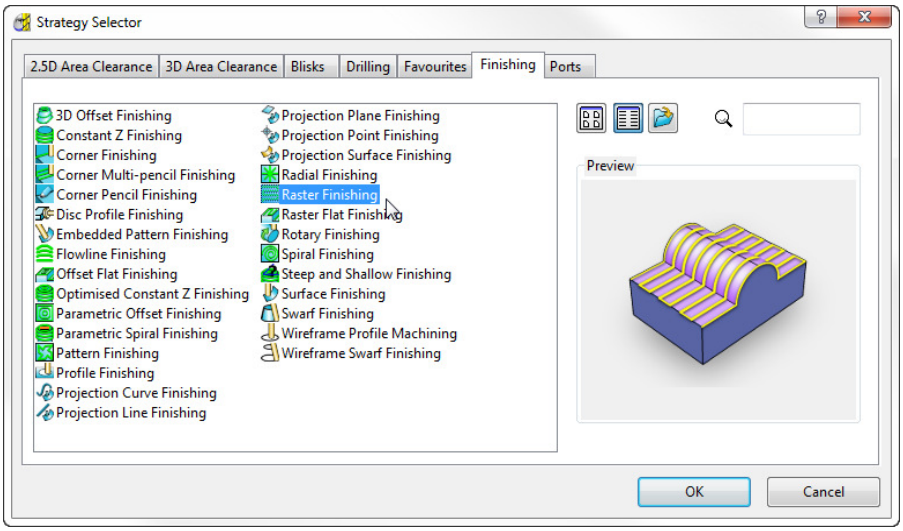
 Copyright © Delcam plc

Creating a Finishing Strategy

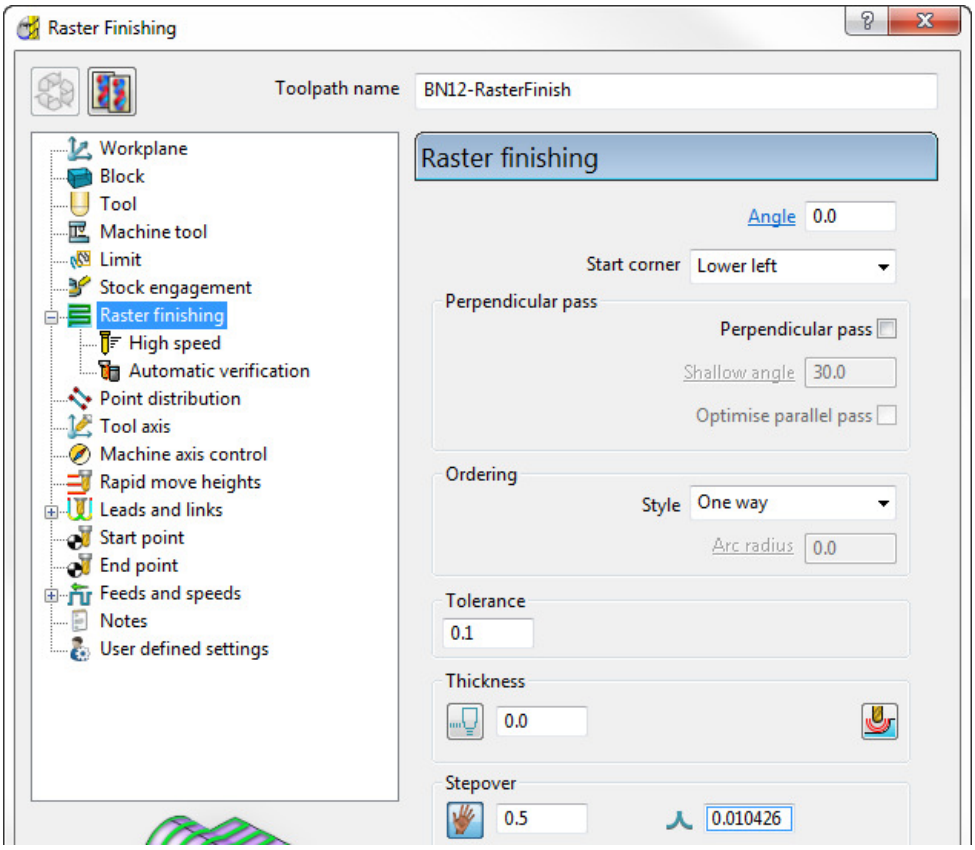
7 In the **PowerMILL *explorer***, right click the **BN12** tool and select **Activate** (or double left click on the tool's, **PowerMILL *explorer***, icon).

8 From the **Main** toolbar select the **Toolpath Strategies** button.

9 Select the **Finishing** tab.



10 Select the **Raster Finishing** strategy and click **OK**.

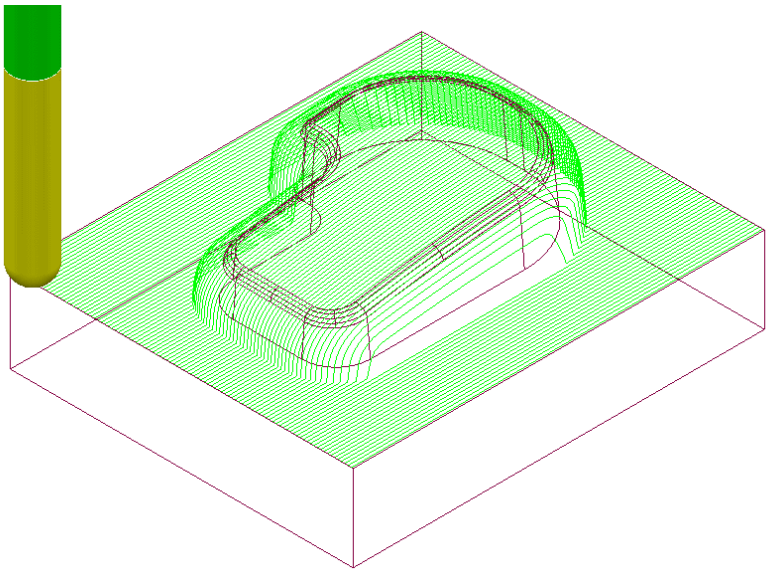


11 In the **Raster Finishing** strategy dialog:

a For the **Toolpath Name**, enter **BN12-RasterFinish**.

b Edit the **Stepover** value to **0.5**

c Click **Calculate** to process the machining strategy.



The **Raster Finishing** pattern is projected down Z onto the component, taking into account tool geometry and machining settings.

The toolpath link moves, clear of the job, are not displayed in this illustration for clarity.

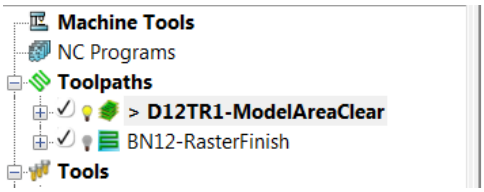
Toolpath Simulation and ViewMILL

PowerMILL provides two levels of options for simulating toolpaths:

- The first level **Simulates** the cutting tool movement along the toolpath.
- The second level (**ViewMILL**) includes a shaded image of the stock material being progressively reduced as the tool moves along the toolpath.

Toolpath Simulation

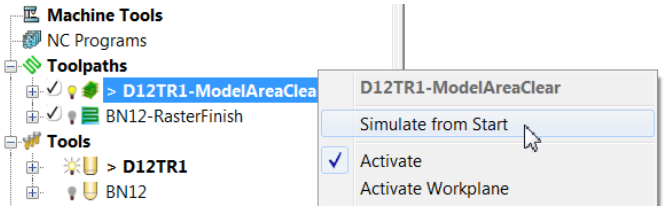
12 In the **PowerMILL *explorer***, right-click on the toolpath **D12TR1-ModelAreaClear** and from the local menu select **Activate**.



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

13 In the **PowerMILL *explorer***, right-click on the roughing toolpath **D12TR1-ModelAreaClear** and select **Simulate from Start**.

 Copyright © Delcam plc



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

Using the simulation control buttons, you can:

- **Play -** start the simulation and play it in continuous mode.
- **Pause -** pause the simulation.
- **Step Back -** step the simulation back by tool moves. Click the **Play** button to resume continuous mode.
- **Step Forward –** step the simulation by tool moves. The faster the speed (defined using **Speed Control**) the bigger the step. Click the **Step Forward** button again to see the next move or click the **Play** button to resume continuous mode.
- **Search Backward -** step the simulation back to the previous toolpath segment.
- **Search Forward -** step the simulation to the next toolpath segment. Click the **Search Forward** button again to see the next component or click the **Play** button to resume continuous mode.
- **Go to Beginning -** moves to the start of the toolpath.
- **Go to End -** moves to the end of the toolpath.
- **Speed Control -** controls the speed of the simulation. The fastest setting is by having the slider at the right, the slowest at the left.
- Select **Point to Point** (toolpath points per second) or **Feedrate** (a multiple of the stored federate) as the basis for the **simulation** feedrate.

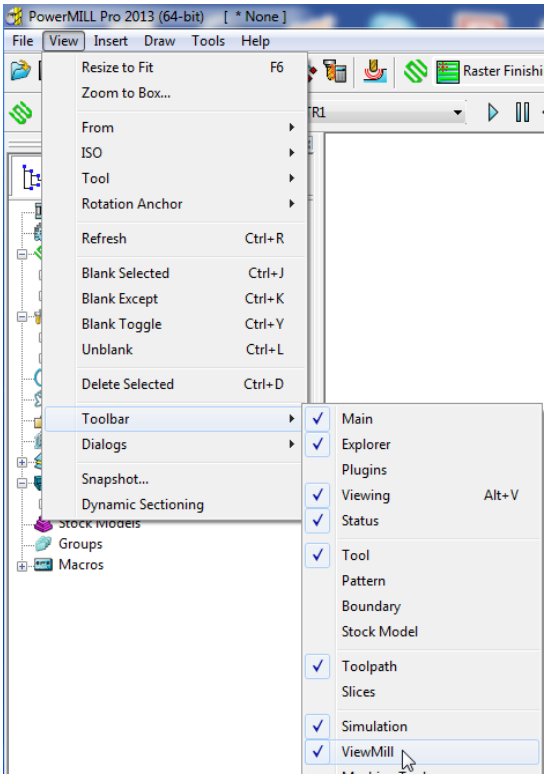
15 Click the **Play** button and **Simulate** the **toolpath**.

16 **Activate** the finishing toolpath **BN12-RasterFinish** and repeat the simulation process.

ViewMILL

1 **Activate** the roughing toolpath **D12TR1-ModelAreaClear** and select it in the **Simulation** Toolbar.

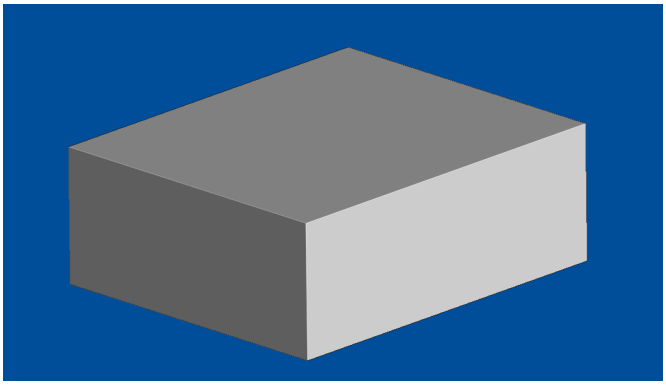
2 From the **Main** pull down menus, select **View > Toolbars > ViewMILL** to raise the **ViewMILL** Toolbar.



The **ViewMILL** toolbar is displayed. Initially, all the buttons are greyed out.

3 Click the button to toggle the graphics window into the **ViewMILL** mode (The **ViewMILL** toolbar buttons are now active).

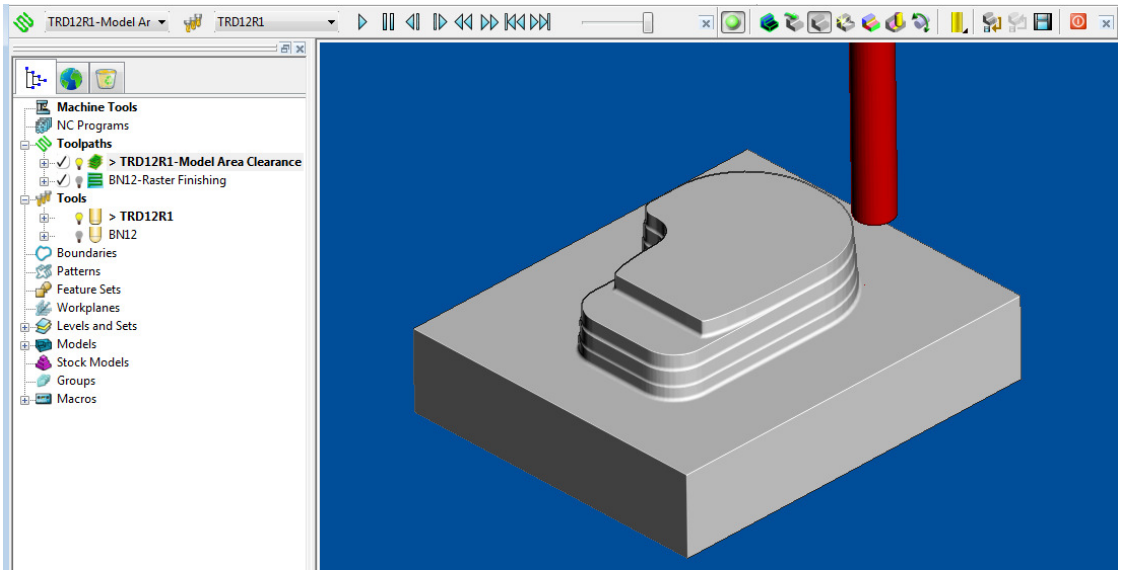
4 Click the button to select a **Plain Shaded Image**.



Copyright © Delcam plc

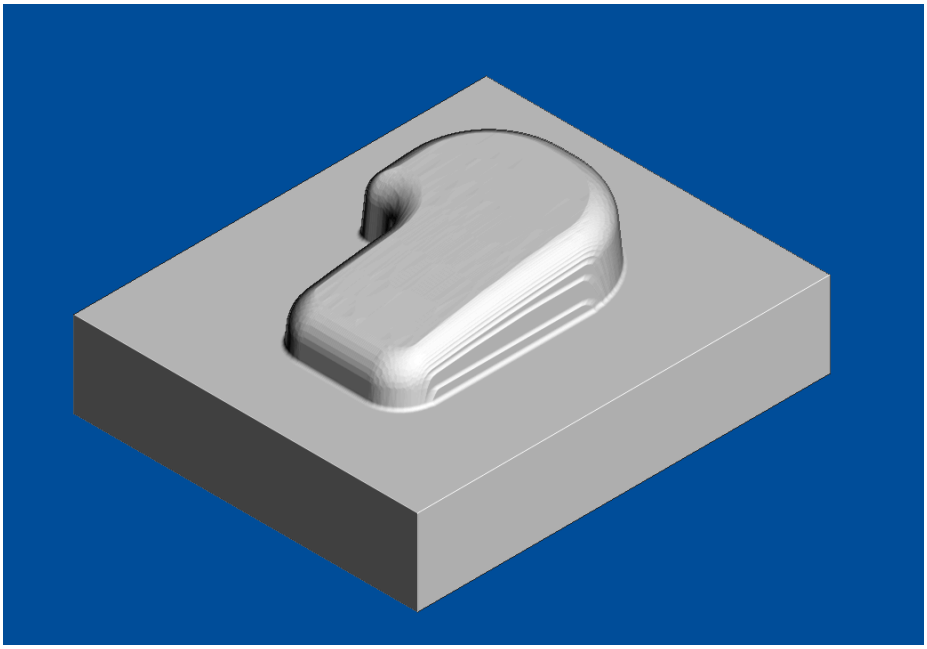
5 Click the **Tool** button to display the tool.

6 Click the **Play** button to start the **ViewMILL** simulation.



7 After the ***simulation*** has finished, select the toolpath **BN12-RasterFinish** on the **Simulation** toolbar, followed by the **tool** button .

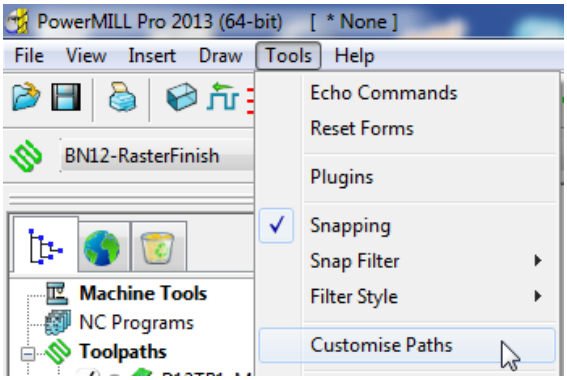
8 Click the **Play** button again to view the continued simulation of material removal by the finishing toolpath.



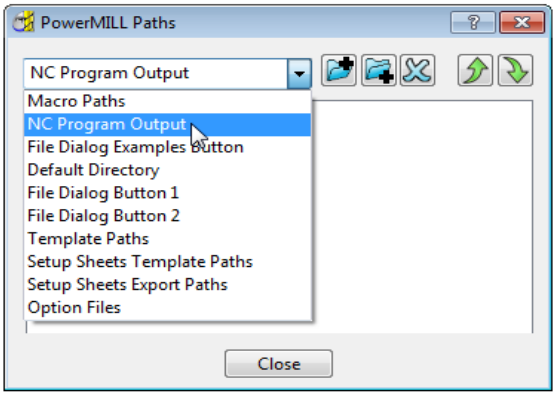
9 In the **Simulation** toolbar, select the **Close** button to delete the **ViewMILL** session.

NC Programs (Post-Processing and NC data Output)

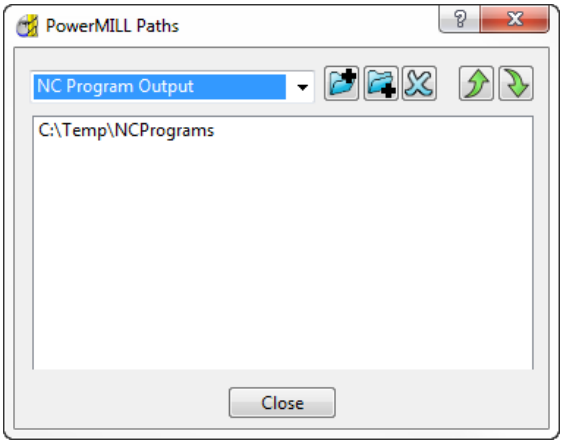
1 From the **Main** menu, select **Tools > Customise Paths** to open the **PowerMILL Paths** dialog.



This displays the **PowerMILL Paths** dialog.



2 In the **PowerMILL Paths** dialog, select the option **NC Programs Output**.



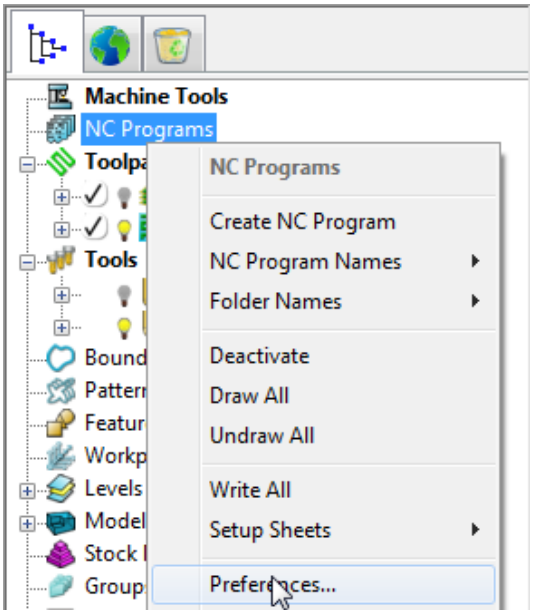
This controls where the post-processed, NC data files are output ready to download to a machine tool controller.

3 Click the ***Add path*** to top of list button.

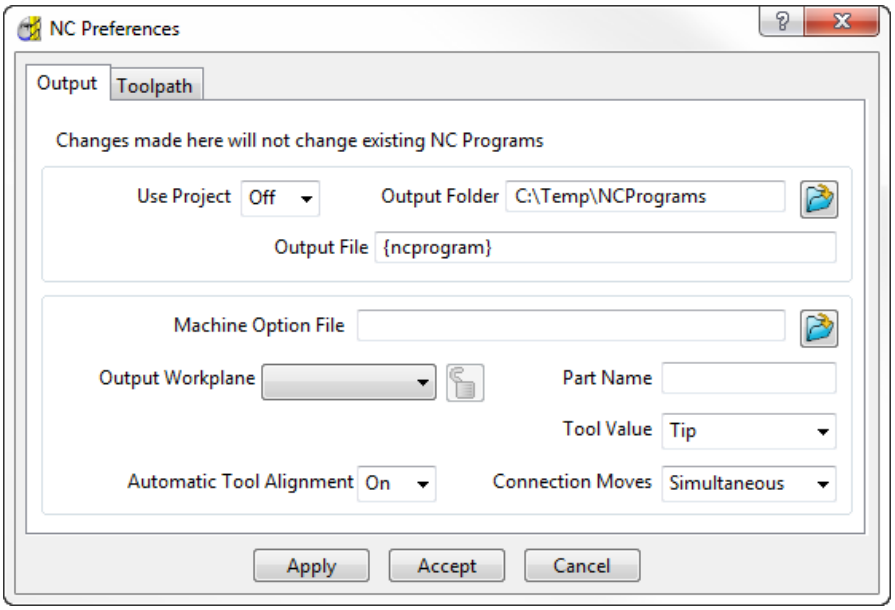
4 In the **Select Path** dialog which appears, browse to the required location **C:\Temp\NCPrograms** and click **OK**.

5 In the **PowerMILL *explorer***, right-click **NC Programs** and select **Preferences** to display the **NC Preferences** dialog.

Copyright © Delcam plc

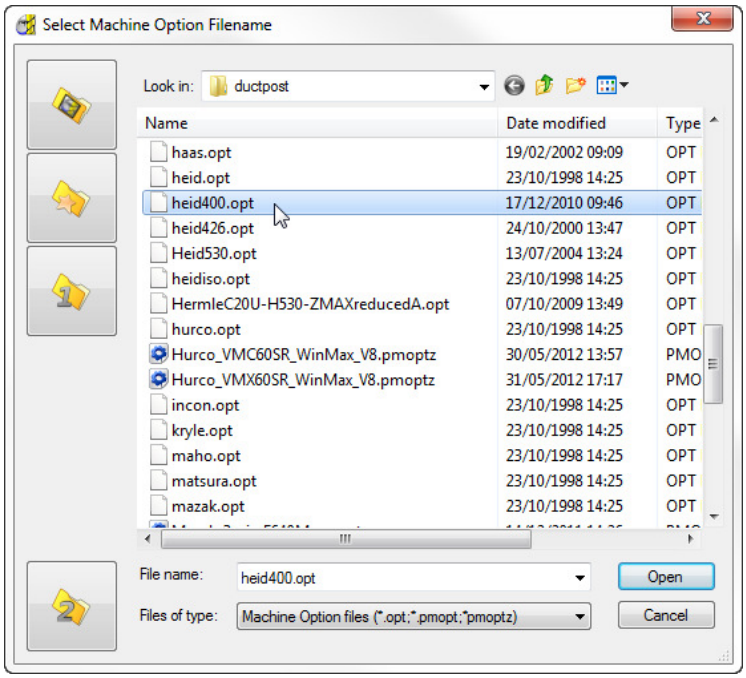


NC Preferences enables you to control the content of output files to download to a machine tool.



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

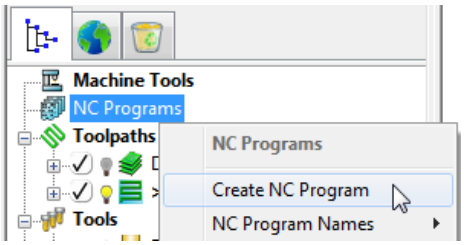
6 In the **NC Preferences** dialog, click **Machine Option File** button. This displays the **Select Machine Option Filename** dialog.



7 Select **heid400.opt** and click **Open**. This selects the **heid400.opt** file and returns you back to the **NC Preferences** dialog.

8 Click **Apply** and then **Accept** on the **NC Preferences** dialog.

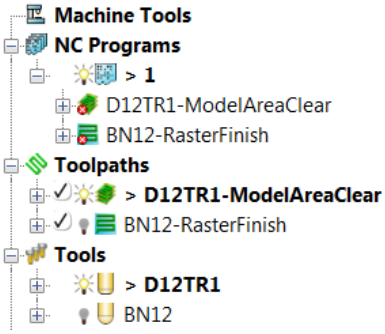
9 In the **PowerMILL *explorer***, right-click **NC Programs** and select **Create NC Program**.



An empty **NC Program** appears in the **PowerMILL *explorer*** and the **NC Program** dialog is displayed.

10 In the **PowerMILL *explorer***, use the left mouse to select and drag the toolpath **D12TR1- ModelAreaClear** onto **NC Program** 1.

*Alternatively, right-click on the toolpath **D12TR1-ModelAreaClear** and select **Add to > NC Program.***

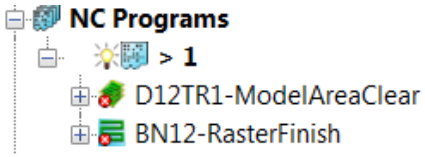


A copy of the ***toolpath name*** will appear in the **NC Program** indicating that it has been assigned as part of the output file.

 Copyright © Delcam plc

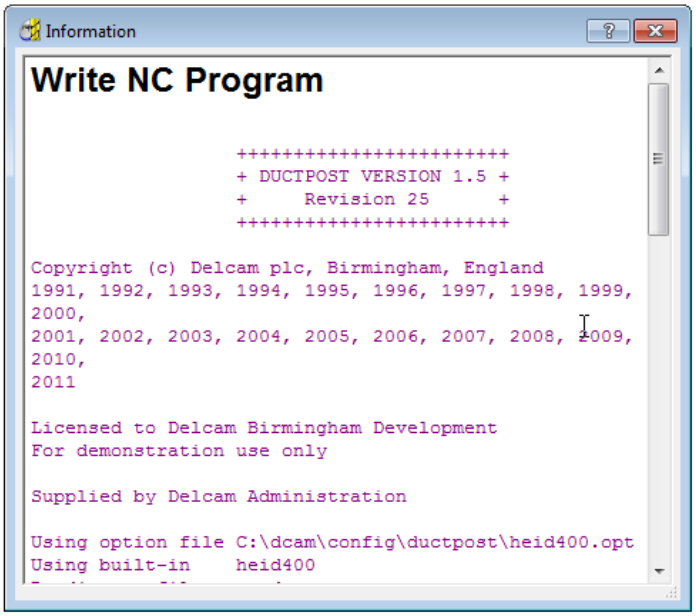
11 In the **PowerMILL *explorer***, click and drag the toolpath **BN12-RasterFinish** onto **NC Program** 1.

Click the small + box to expand and view the contents of the **NC Program**.



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

12 In the **NC Program** dialog displayed in the graphics area, click the **Write** button to start the post processing operation. An Information dialog is displayed with a progress and confirmation summary.



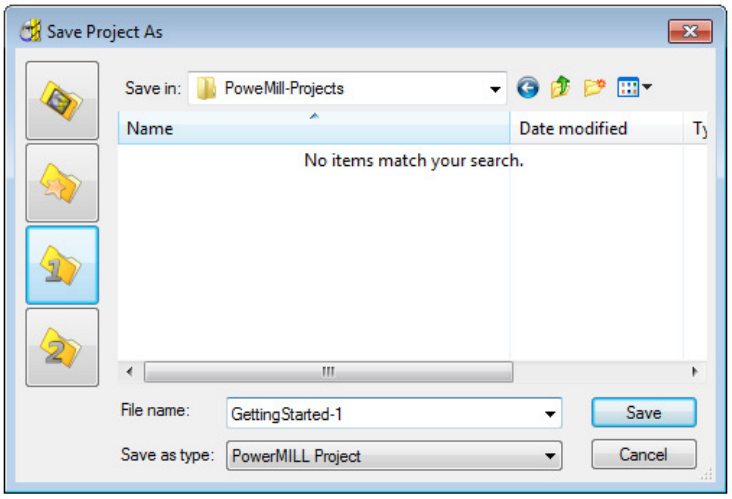
13 **Close** both the **NC Program** and **Information** dialogs.

14 You can find the created NC data file **1.tap** under **C:\temp\NCPrograms** on your computer.

Saving the Project

1 On the **Main** toolbar, click the **Save** button.

This displays the **Save Project As** dialog.



If the **Project** has been **Saved** before, then it will simply be updated.

2 From the **Main** menu , select **File** > **Save Project As**:-

.....\COURSEWORK\PowerMILL-Projects\PmillGettingStarted

3 From the **Main** menu, select **File** > **Delete All**, followed by **Tools** > **Reset Forms**.

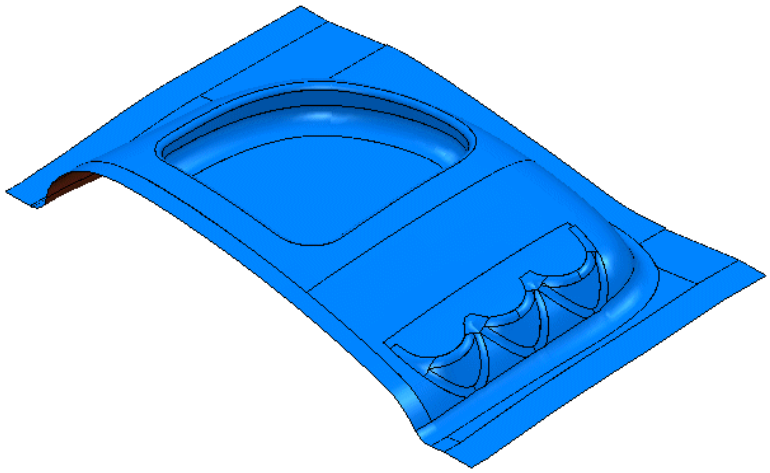
The content of **PowerMILL *explorer*** is deleted and all dialogs are reset to their default settings.

The externally stored copy of the **Project** (**PmillGettingStarted**) can be reopened as required.

Additional Exercise

4 **Import** the **Model**:-

.....\PowerMILL_data\Models\facia.dgk



5 **Save the Project as**:

.....\COURSEWORK\PowerMILL_Projects\Facia-1

6 Use the same **Tools** and **machining strategies** to those in the previous **Project** (PmillGettingStarted).

7 Once you have completed the exercise and saved the project, from the **Main** menu, select **File** > **Delete All** followed by **Tools** > **Reset Forms**.

Copyright © Delcam plc

2. Machining Setup

In the previous section, the toolpaths were created using the default values wherever possible.

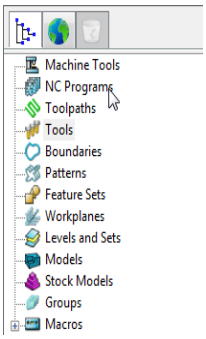
We will now look at the machining Setup options in more detail.

In particular:

- Setting up direct access to regularly used files.
- Use of Workplanes (Positional datums) to create appropriate machining set ups.
- Different methods of creating Workplanes and alignment using the workplane Editor.
- Workplane Aligned to Geometry.
- Tool and Holder definition.
- Material Block Definition.
- Safe Z Heights.

PowerMILL Panes

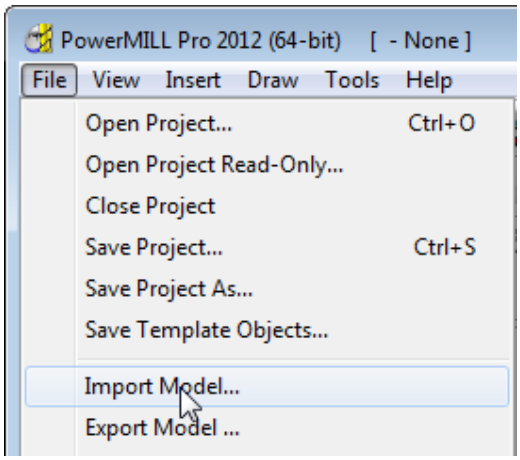
On the left hand side of the screen above the **explorer** are the **PowerMILL** panes.



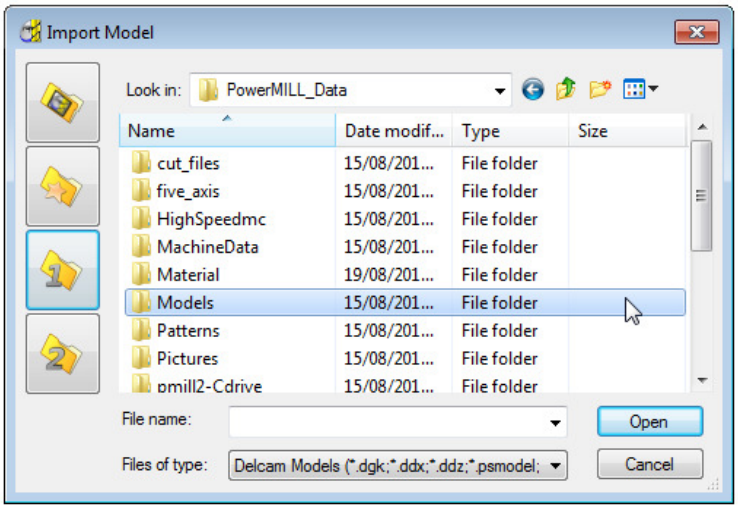
- The first pane displays the **PowerMILL explorer** (as shown above).
- The second pane displays the HTML browser used for viewing HTML files including Help files.
- The third pane opens the **PowerMILL** Recycle Bin.

Setting up direct access to regularly used folders

1 To Import a Model via the **Main** pull down menus, select **File** > **Import Model**.



The **Import Model** dialog is displayed.



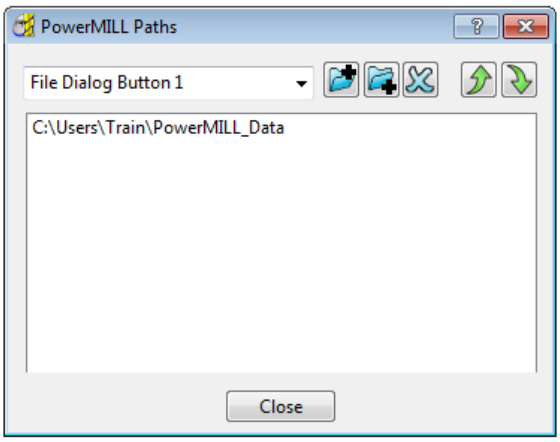
- - changes the directory to the current Project directory.
- - changes the directory to the **Examples** directory.

  This folder contains a selection of sample model files which are installed within **PowerMILL** in a directory called **Examples**.
- - changes the directory to the first user-defined directory.
- - changes the directory to the second user-defined directory.

To customise the paths for user-defined directory folders:

Copyright © Delcam plc

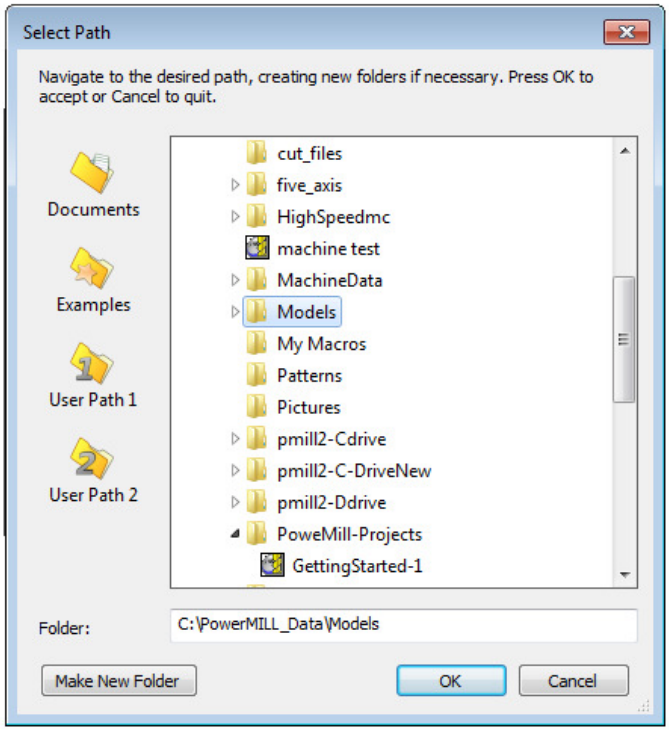
2 From the **Main** menu select **Tools** > **Customise Paths**. The **PowerMILL Paths** dialog is displayed.



3 Select the **File Dialog Button 1** option from the drop-down list.

4 Click the **Add path to top of list** button and browse to:

.....\PowerMILL_Data\Models



5 Click **OK**.

6 Repeat steps 2 and 3, but this time select the **File Dialog Button 2** option to link directly to:

.....\PowerMILL_Data

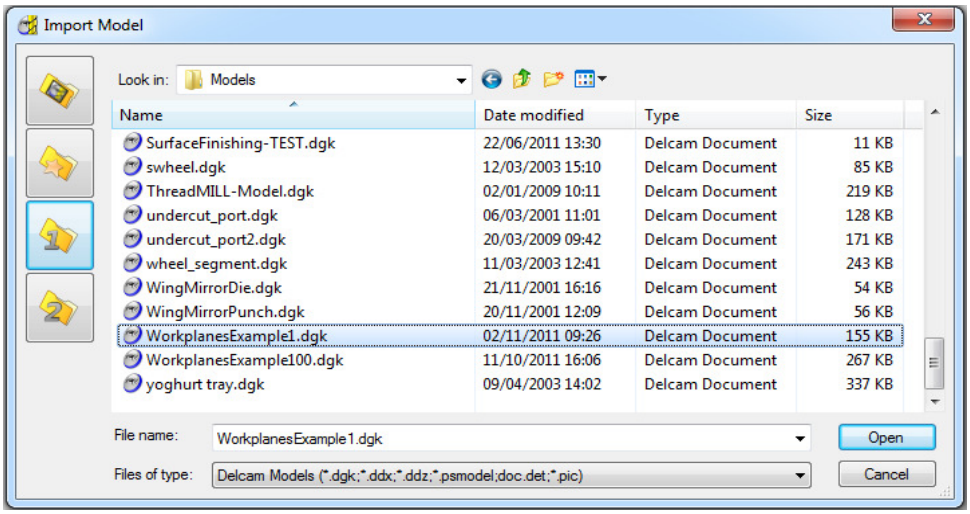
*Outside the training environment, the location of the **PowerMILL_Data** directory will depend on where you have installed it!*

Loading a model into PowerMILL

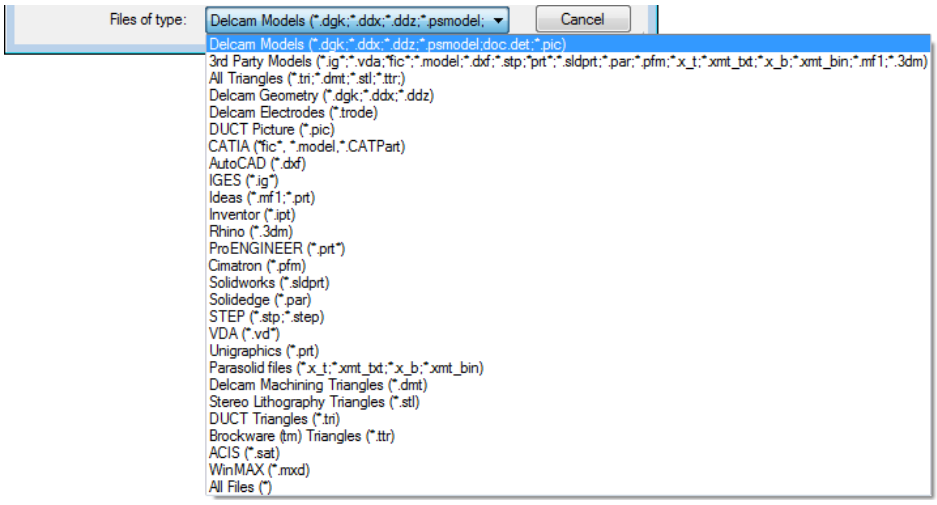
1 From the Main menu, select **File** > **Import Model**.

2 Use **Shortcut Button 1** or browse to:

.....\PowerMILL_Data\Models



*A variety of **Model** formats can be **imported** into **PowerMILL**. Use the **Files of type** filter to widen or narrow your choice.*



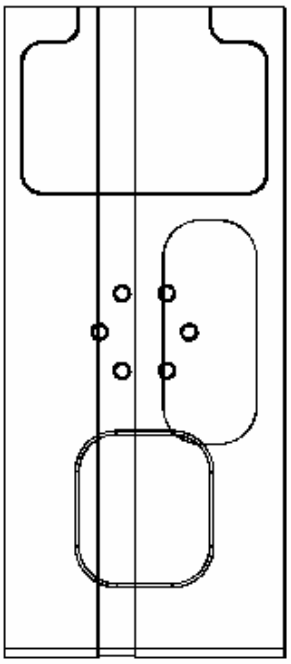
*Please note: **direct transfer - file formats** (eg: CATIA, ProENGINEER, etc) are only available as a cost option.*

3 Back in the main form, select the file **WorkplanesExample1.dgk** and then click **Open**.

4 From the **Viewing** toolbar (Right of graphics area) select **View from top (Z)** followed by **Resize to fit** .

The **model** is displayed in the **PowerMILL** graphics as viewed down the Z-axis with X aligned from left to right, and Y from bottom to top.

Copyright © Delcam plc



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 exceed of the travel limits in Y.

If this is the case, it will be required to align the component with the longest side along X to ensure that it can be positioned within the travel limits.

Saving the Project

It is good practice to create a **Project** as soon as possible and then to **Save** it at regular intervals during the session.

1 On the **Main** menu, select **File** > **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\WorkplaneExample1-EX1

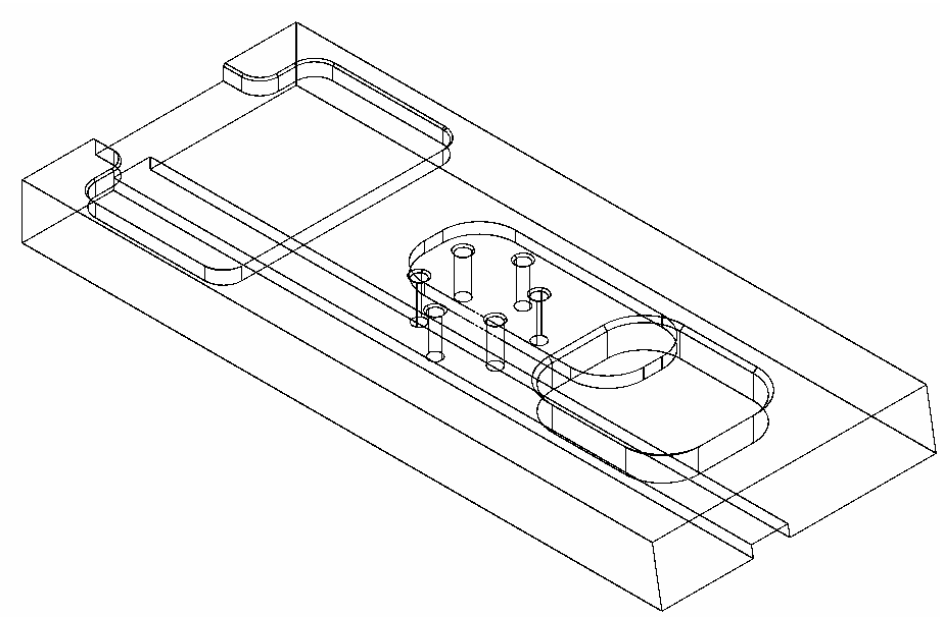
Viewing the Model

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

Above is the **Views** toolbar (shown aligned horizontally).

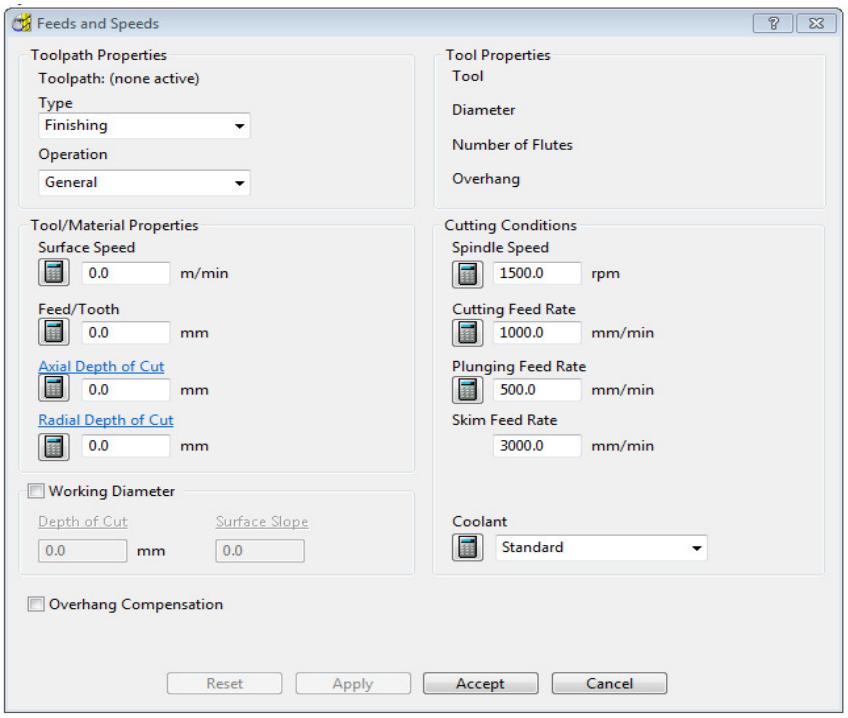
By default, the **Viewing** toolbar is aligned to the right hand side of the graphics area. These options mainly include basic visual orientations along with several shading modes, many of which provide specialist analysis of the model.

1 From the **View** toolbar, select an isometric **ISO 1** view.



Setting Feed rates

The current **Feed rate** values are applied directly to a newly created **toolpath**. The **Feed rate** values can be stored in a **Tool** definition to update the **Feed rate** dialog accordingly on **Tool** activation. At this stage we will look at manually setting the feed rate individually. Click the **Feeds and Speeds** button on the **main** toolbar to display the **Feeds and Speeds** dialog.



The default values (as shown above) can be modified as required and **Accept** selected for them to be the default for future toolpaths. **Apply** is used to update the **Feeds and Speeds** of an ***active*** **toolpath**. Note **Apply** is 'greyed out' (as above) if there is currently no ***active*** **toolpath**.

 Copyright © Delcam plc

Creation and Application of Workplanes.

When a model is imported into **PowerMILL** it is more than likely that the original global datum (**Transform**) will not provide a suitable position/orientation for tool setting or applying machining strategies.

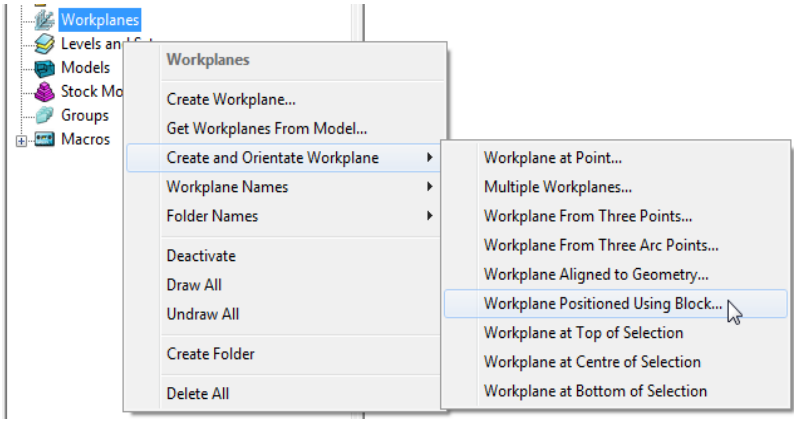
Workplanes provide the ideal method of creating suitable machining datums and alignments without having to resort to physically moving the component model.

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

By deactivating a **workplane**, the original coordinate set-up can easily be re-activated for tasks such as checking original dimensions or for importing a 'new issue' updated model.

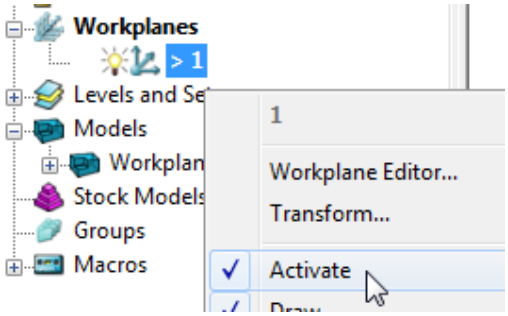
Creating a new Workplane at top of Selection

1. Press the left mouse button and drag a box over the entire **model** to select.
2. In **PowerMILL explorer**, right-click **Workplanes** and select **Create and Orientate Workplane** > **Workplane at top of Selection**.



A **Workplane** is created and appears at the top-centre of the selected model. The new **Workplane** 1 is also registered in **PowerMILL *explorer***.

3. Right-click the newly renamed **workplane - 1** and select **Activate**.

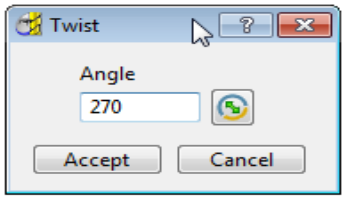


The next step will be to rotate the new **Active - Workplane - 1** to re-orientate the model such that the longer side is aligned to the **X-Axis**.

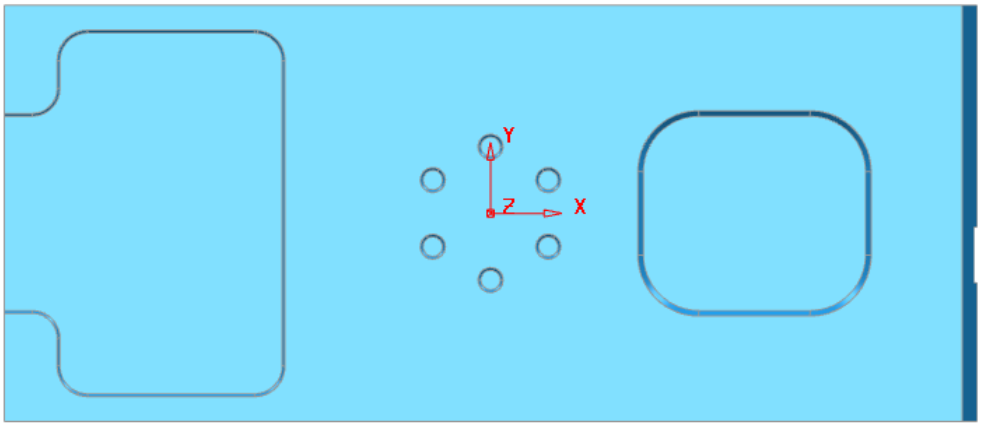
1. From the same local menu, select **Workplane Editor** to display the **Workplane** editing toolbar.

2 Double click the **Twist about Z** button. This displays the **Twist** dialog.

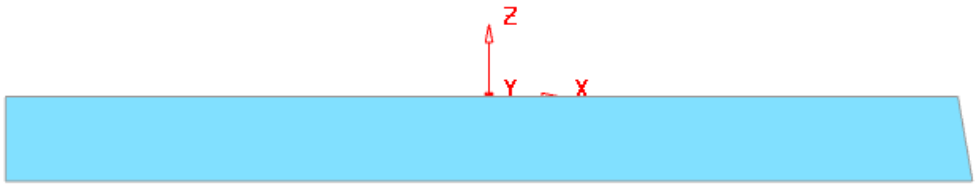
3 Enter **Angle 270** and click **Accept**.



4 Click the **Green Tick** on the **Workplane** toolbar to accept the changes.



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



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

Workplane Positioned using the Block

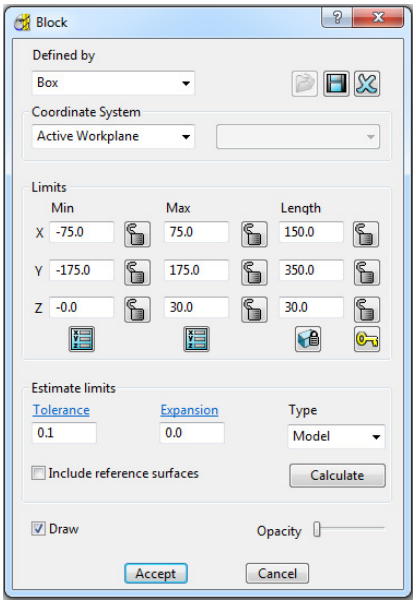
Next, the same **Workplane** alignment as above will be demonstrated using a different method.

1 Right click on the **Workplane** 1 (either in the **PowerMILL *explorer*** or directly in the graphics area) and from the local menu select **Delete Workplane**.

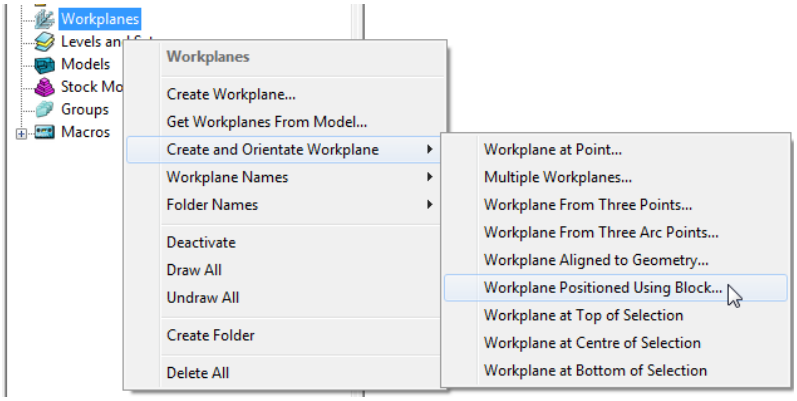
Another method of creating a **Workplane** is from *key points* on the defined **Block**.

 Copyright © Delcam plc

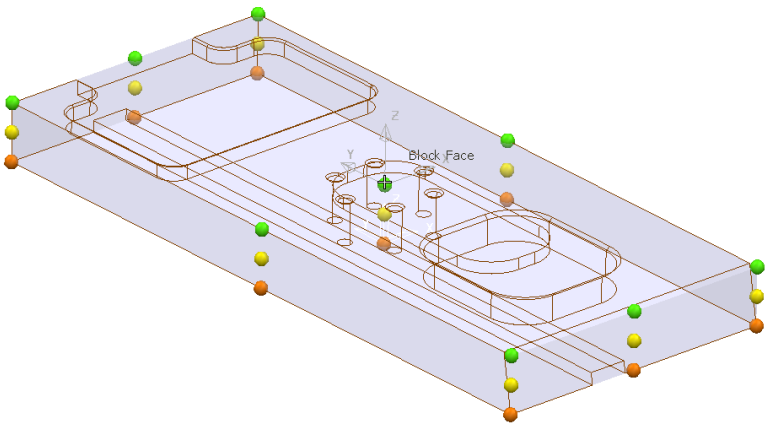
2 From the **main** menu, select the **Block** Icon the following form will appear select **Active Workplane** press **Calculate** followed by **Accept**.



3 In the **PowerMILL explorer**, Right mouse click on **Workplanes** to access the following menus.



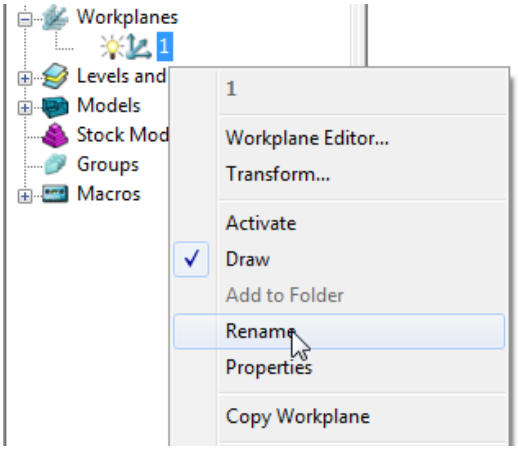
4 Select **Workplane Positioned using Block** from the above menu.



5 Click the new **workplane** on to the green *sphere* at the *centre* of the *top face* of the **block**.

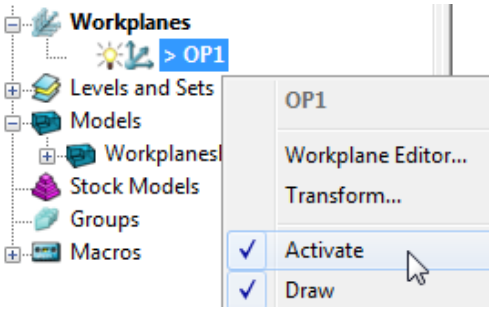
To enable easy identification, it is recommended to rename, created entities with a name relevant to their application.

6 In the **PowerMILL *explorer***, right-click the newly created **Workplane** and select **Rename**.



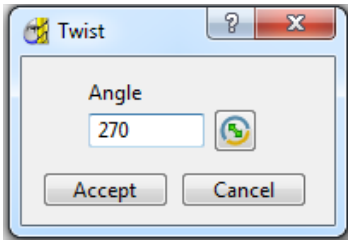
7 Change the name to **OP1**.

8 Right click on the **workplane** and from the local menu, select **Activate**.



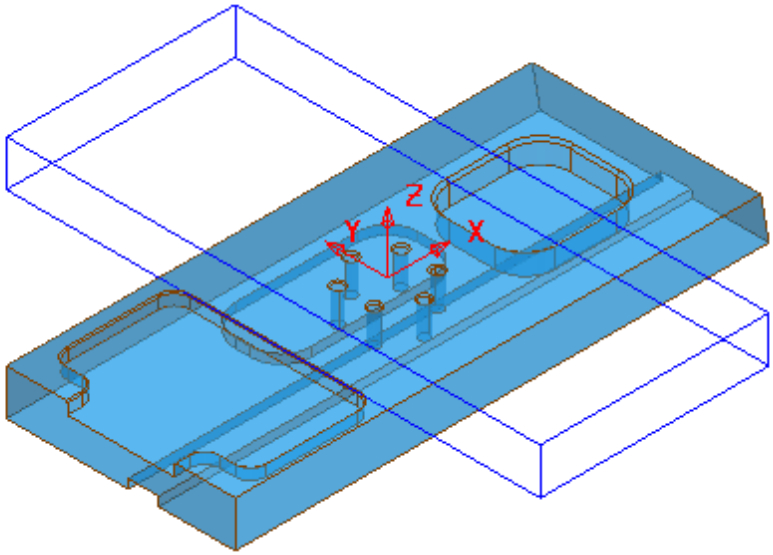
9 Then from the local menu select **Workplane Editor**.

10 Double click on **Z Axis Twist** and enter **270** to orientate the **workplane** as shown below.

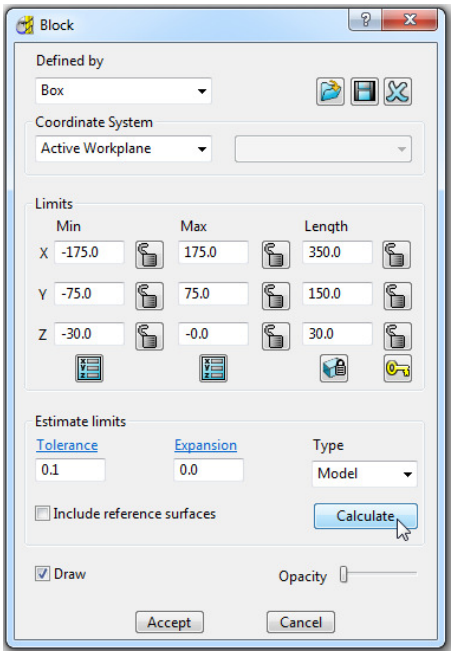


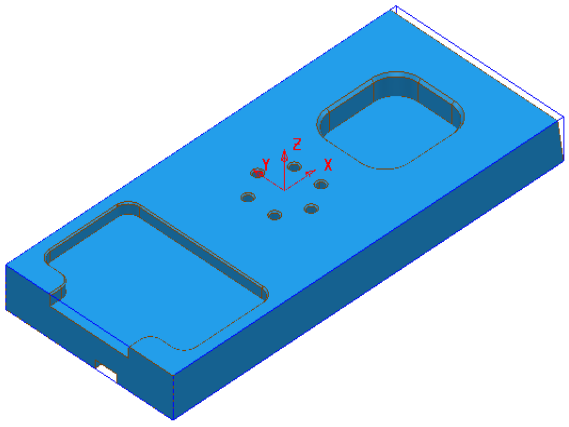
 Copyright © Delcam plc

This action will cause the **Block** to be aligned 90 degrees out of position, relative to the **model** (as shown on the next page). The safest way of correcting this is to **Calculate** the **Block** again while the new **workplane** is *active*. If the user de-activates the **workplan**e then the **Block** will again move out of position. This is an ideal warning to prevent the user inadvertently machining relative to the wrong datum.



1 From the **main** menu, select the **Block** Icon the following form will appear select **Active Workplane** press **Calculate** followed by **Accept**.

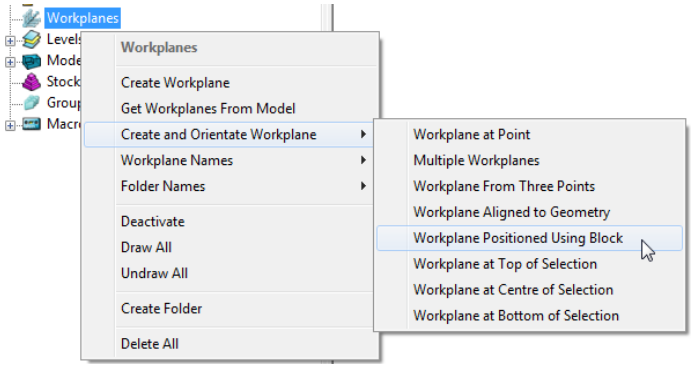




The **Block** is now re-aligned to the model relative to the active **Workplane**.

Create second workplane for OP2

We will now create a second workplane for the opposite side for OP2. You can only have one workplane active at one time. So the machining Strategies require a suitable naming convention to identify which active workplane they relate to.

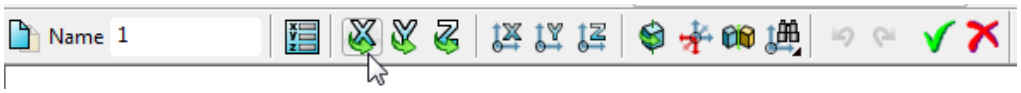


2 With the left mouse key depressed, drag a box over the entire **model** to select.

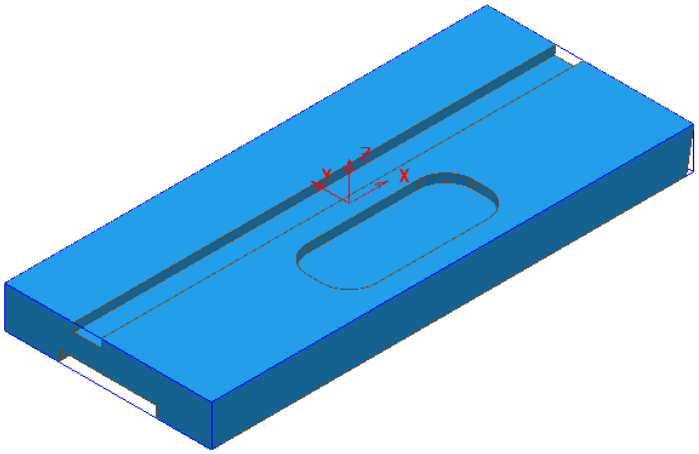
3 In the **PowerMILL *explorer***, right-click **Workplanes** and select **Create and Orientate Workplane** > **Workplane at bottom of Selection**.

4 **Rename** the new **Workplane** as **OP2**.

5 Hold down the centre wheel on the mouse and rotate the view around the **model**.



6 Double click on the **X Twist** and enter **180** degrees so that the new **workplane** is normal to the underside of the model.



7 Select **Accept** for the new **workplane** **OP2** to be updated.

8 Select **File** - **Save Project**:-

The contents of the Project saved earlier will be updated.

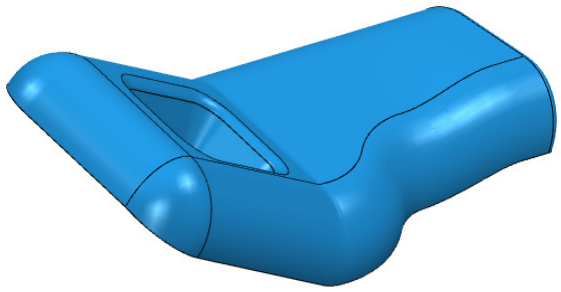
9 Select **File** - **Delete all** followed by **Tools** - **Reset forms**.

The contents of the **PowerMILL *explorer*** will be deleted and all forms will be returned to the default, factory settings.

Copyright © Delcam plc

Workplane Aligned to Geometry

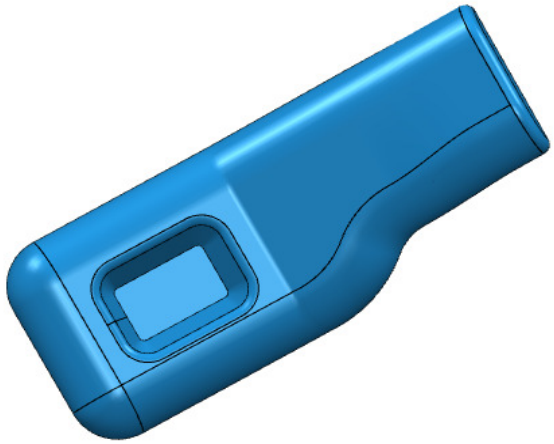
A very powerful tool for aligning components is to create a **Workplane** perpendicular to a face using **Workplane aligned to Geometry**. The next example shows this in detail. (See original model orientation)



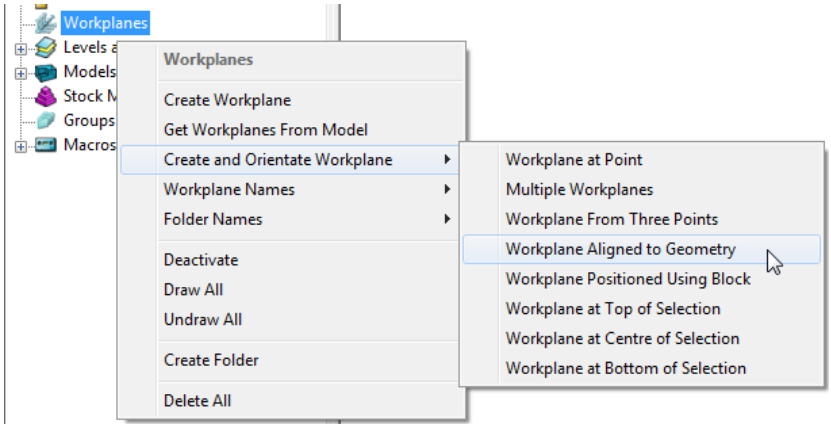
1 **Import** the **Model**:-

.....\PowerMILL_Data\Models\InteriorTrim.dgk

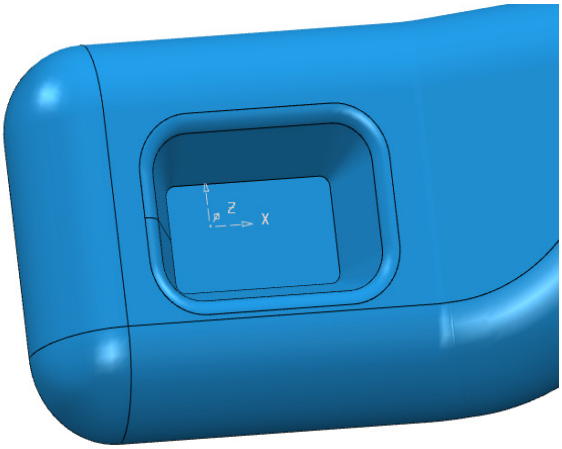
2 While the centre mouse wheel is depressed, move the mouse to rotate the view of the model so that the base of the pocket is fully visible.



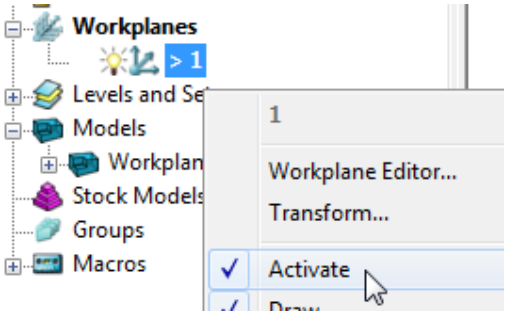
3 Right click on **Workplanes** and select **Create and Orientate Workplane** followed by **Workplane aligned to Geometry**.



4 Left mouse click to locate a **Workplane** on the base of the pocket as shown.

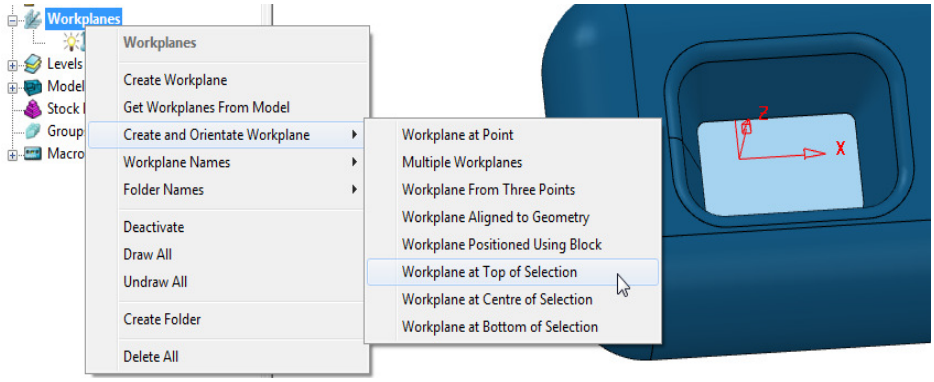


5 Right click on the new **workplane** and select **Activate**.



6 With the left mouse key depressed, click on the bottom face of the pocket to make it selected (shaded light blue below).

7 Then right click on **Workplanes** to open the local menus below.



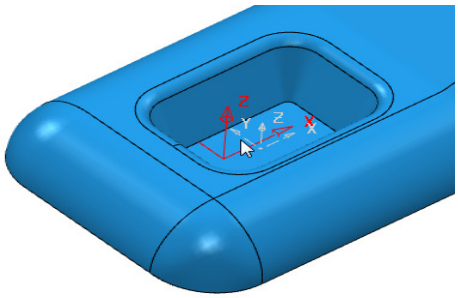
8 From **Create and Orientate Workplane**, select **Workplane at Top of Selection**.

9 The new **workplane** will be centrally located at the base of the pocket.

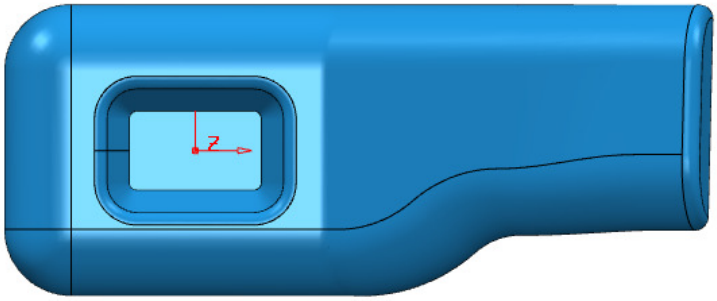
10 **Rename** the new **workplane** as **Datum** and make it **Active**.

11 **Delete** the original **workplane**.

 Copyright © Delcam plc



The new datum is from the centre of the pocket as shown below.



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

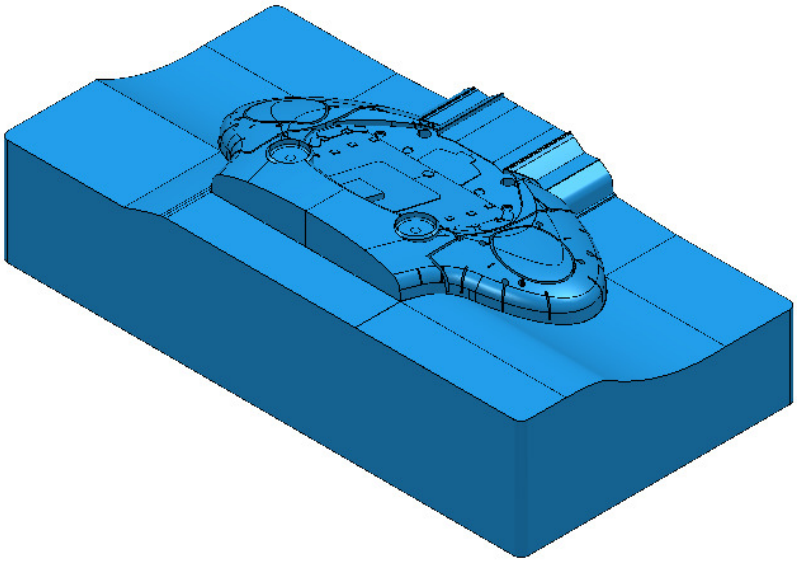
.....\COURSEWORK\PowerMILL_Projects\Workplane-Example2

13 Select **File** – **Delete all** followed by **Tools** – **Reset forms**.

Speaker Core Example - Set Up

1 **Import** the **Model**:- **.....\PowerMILL_Data\Models\speaker_core.dgk**

*Use **Button 1** in the **Import Model** dialog for quick access!*

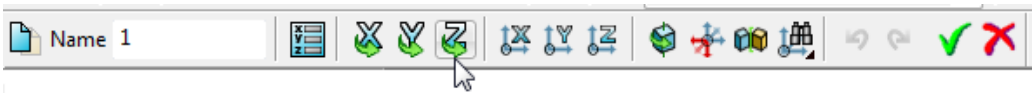


For this example it is required for the long side of the component to be aligned to the **X** direction and not **Y** as shown above.

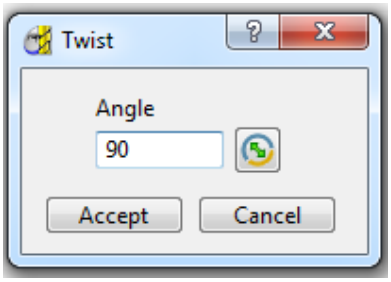
2 Depress the left mouse key and drag a box across the whole **model** to select all **Surfaces**.

3 Select **Create and Orientate Workplane** – **Workplane at Top of Selection**.

4 In the **PowerMILL *explorer*** right mouse click on the new **Workplane** and from the local menu select **Workplane Editor**.



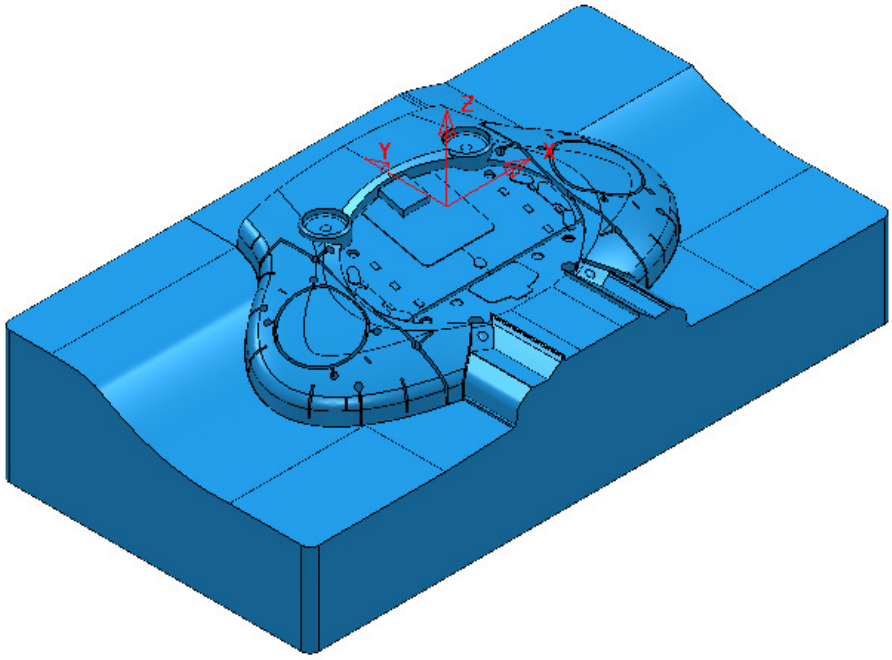
5 In the toolbar, click on the **Z-Twist** option and enter **90** in the form for the **Workplane** rotation angle.



6 **Accept** the ***Twist*** form and then click on the **Green** tick on the toolbar to exit the **Workplane Editor** (and finally accept the changes).

7 **Activate** the new **Workplane** 1.

8 Select an **ISO 1** view.



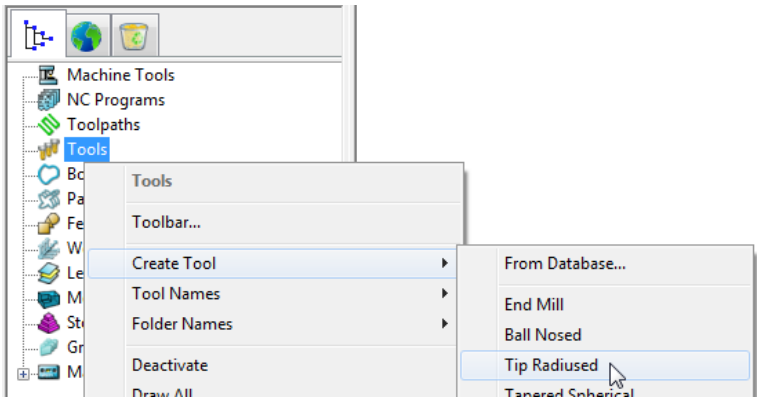
The long side of the component is now running along the **X** direction.

Now that the component model is suitably aligned, the next stage is to create some suitable cutting **Tools** ready to carry out the machining process.

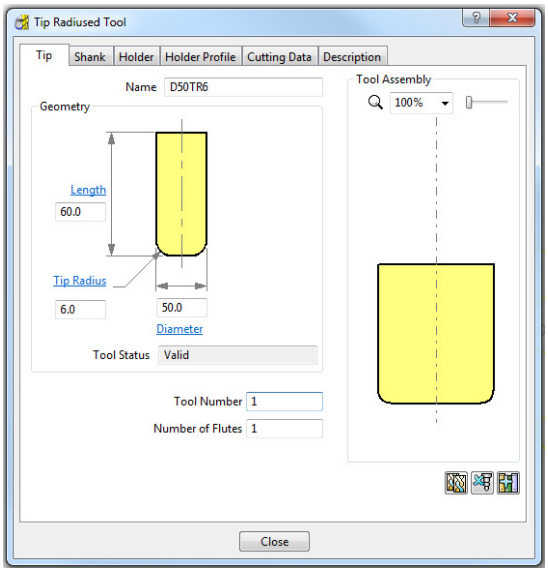
 Copyright © Delcam plc

Tool and Holder Definition

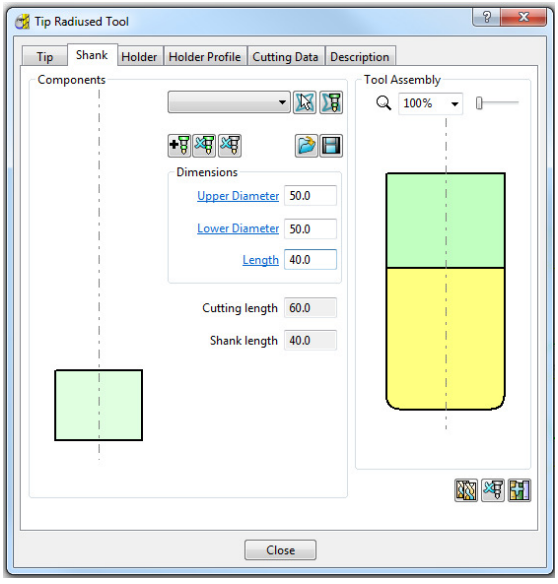
9 From the **PowerMILL *explorer*** select **Tools** - **Create Tool** - **Tip Radiused**.



10 In the **Tip Radiused Tool** dialog, define a **Dia 50 Tiprad 6** - Length **60** - Named **D50TR6** - Tool Number **1**.



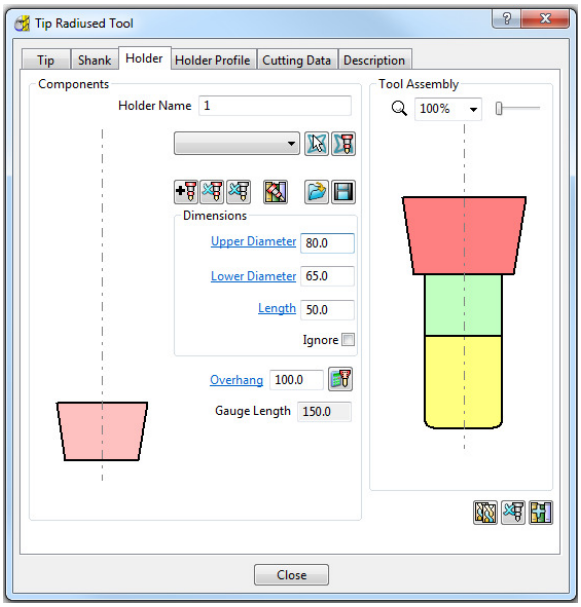
11 In the **Shank** dialog, click the ***Add a shank component*** icon and enter: Upper/Lower Diameter **50** and Length **40**.



12 In the **Holder** dialog, click the ***Add holder component*** button and enter:

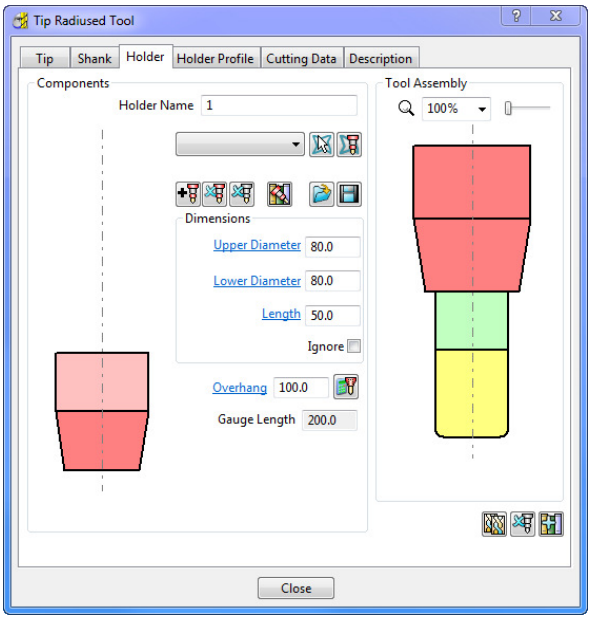
- Upper Diameter – **80**
- Lower Diameter – **65**
- Length – **50**
- Overhang - **100**

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



13 In the **Holder** dialog, click the **Add holder component** button. Enter:

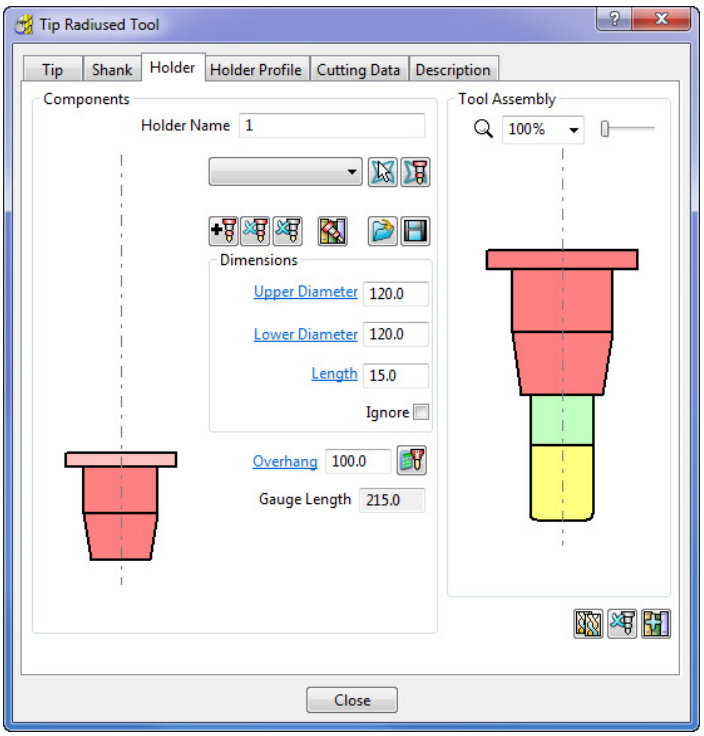
- Upper Diameter – **80**
- Lower Diameter – **80**
- Length – **50**
- Overhang –**100**



14 In the **Holder** dialog, click the **Add holder component** button Enter:

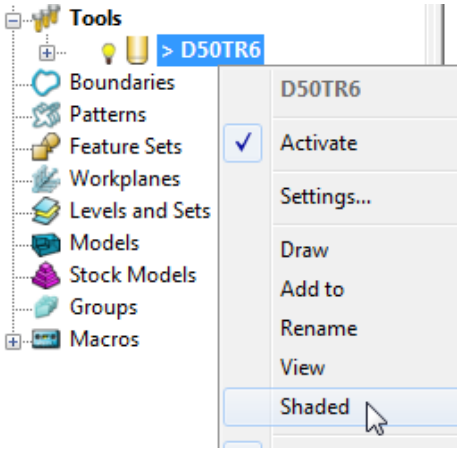
 Copyright © Delcam plc

- Upper Diameter – **120**
- Lower Diameter – **120**
- Length – **15**
- Overhang - **100**



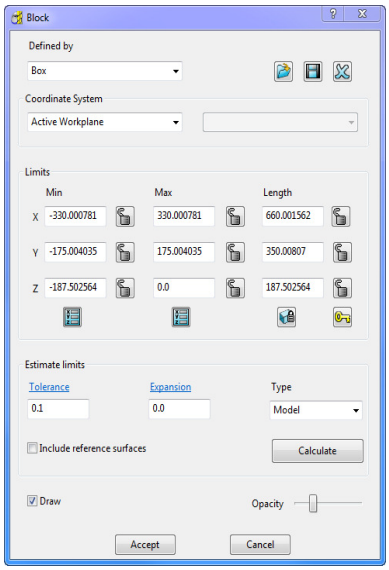
15 **Close** the dialog.

*You can control the style in which the tool is displayed in the graphics window. In the **PowerMILL explorer**, right-click on the **Tool** - **D50TR6** and from the local menu, select **Shaded**.*



Block Definition

16 **Calculate** a **Block** using **Defined by** - **Box** and **Type** - **Model**.



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

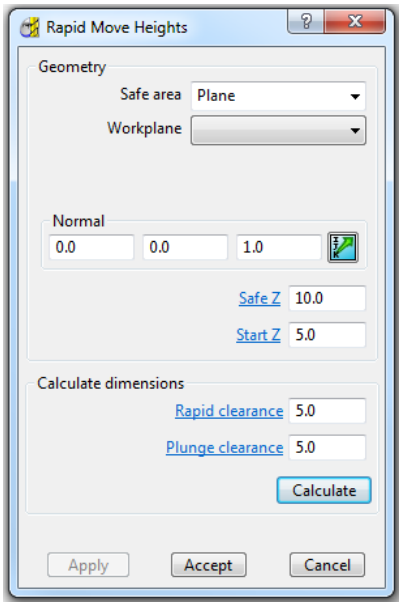
The dimensions of the **Block** can be entered manually or calculated directly to the Type of entity: **Model**, **Boundary**, **Pattern**, or **Feature**.

The **Opacity** slider controls the degree of shading (from the left, initially wireframe and then graduating from clear to dense).

Rapid Move Heights

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

17 In the **Rapid Move Heights** dialog, click the **Calculate** tab to set **Safe Z** and **Start Z** to the default values of **10** and **5** respectively above the **Model** or **Block**.



 Copyright © Delcam plc

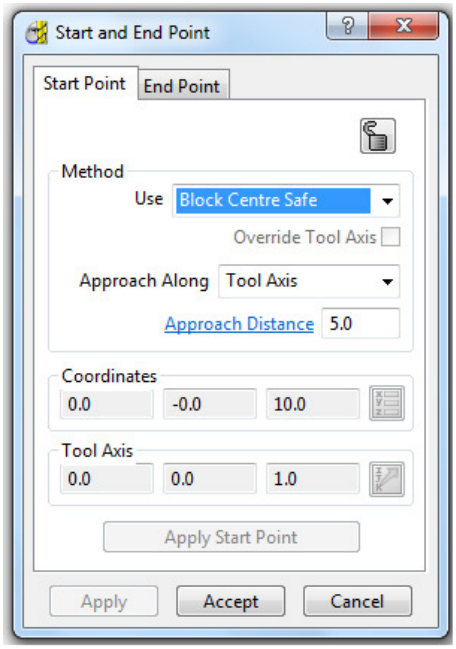
Start and End Point

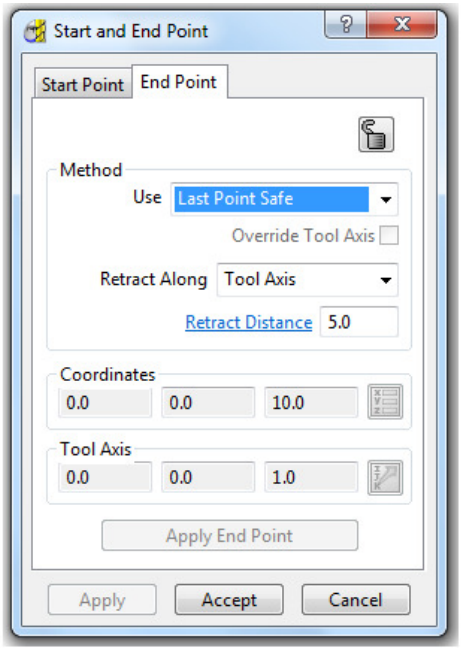
The **Start Point** and **End Point** define the first and last points on a toolpath. These can be controlled using 4 different positional options.

18 In the **Start and End Point** dialog, **Accept** the default settings :

a **Start Point - Block Centre Safe**

b **End Point - Last Point Safe**





Save Project

19 It is essential to regularly **Save** your work outside the **PowerMILL** session. By saving to a **Project** the contents of the **PowerMILL *explorer*** are saved into an external folder.

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

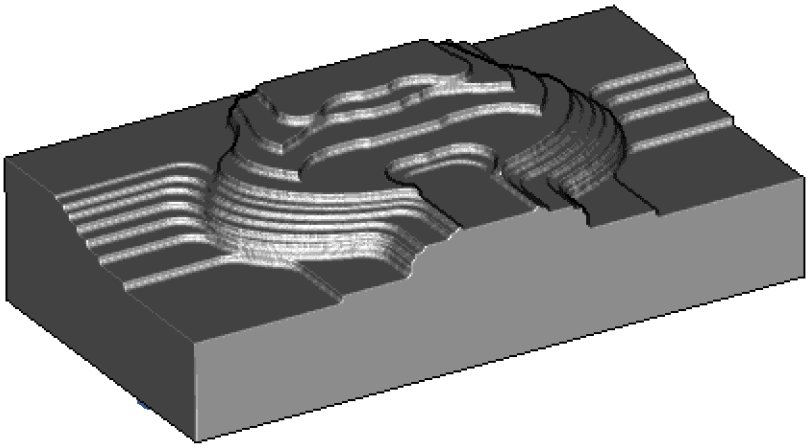
.....\COURSEWORK\PowerMILL-Projects\Spkr-Core

*Do Not Close the **Project** as it will be continued in the following chapter.*

 Copyright © Delcam plc

3. 3D Area Clearance

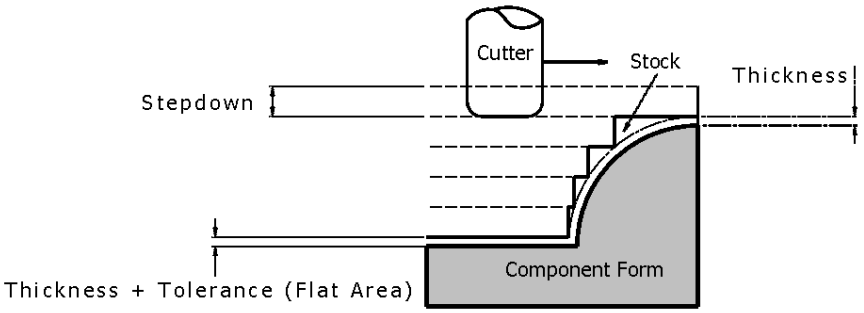
The main strategies for roughing a 3D component model are called **3D Area Clearance**. These provide a choice of 2D material removal methods, which progressively machine the area (**Slice**), up to the component contour, down a sequence of user-defined **Z Heights**.



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 – Model Area Clearance example

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

1 Ensure that the **Project** saved at the end of the last chapter is still active.

2 **Activate** the **Workplane** 1.

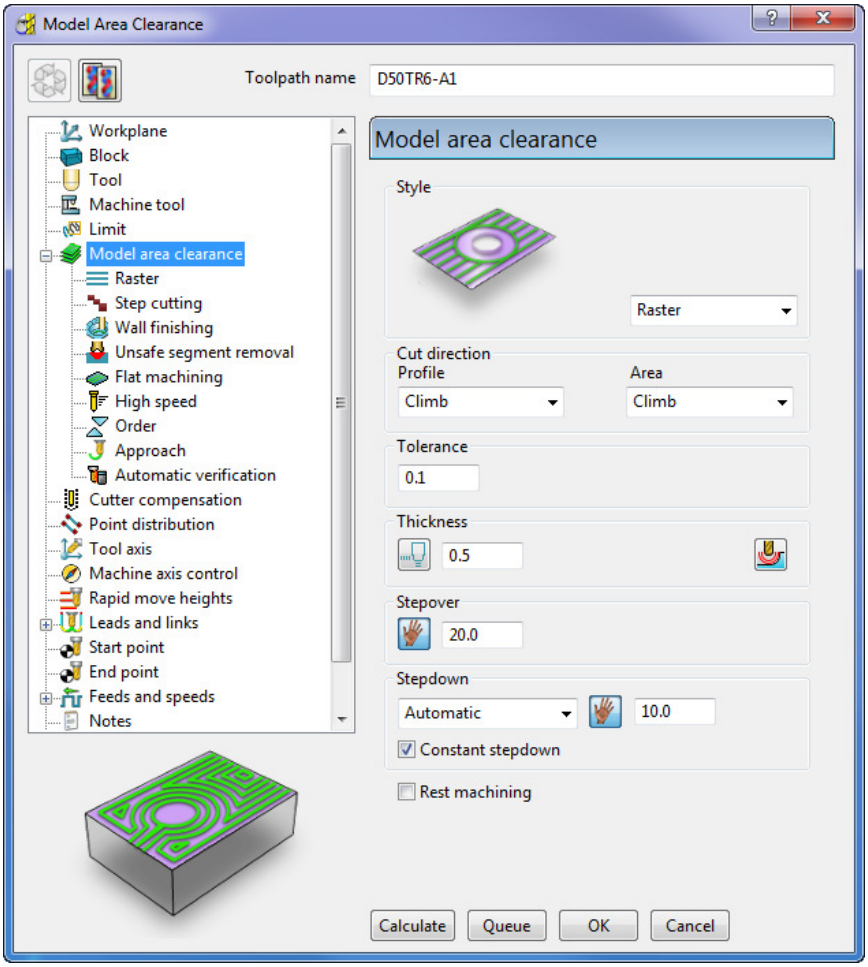
3 From the **main** toolbar select **Block** and using, ***Defined by*** **Box** – ***Type*** **Model** select **Calculate** followed by **accept**.

4 From the **main** toolbar select **Rapid Move Heights** and in the form select **Calculate** followed by **Accept**.

5 From the **Main** toolbar select **Toolpath Strategies** .

6 Select the **3D Area Clearance** tab followed by **Model Area Clearance**.

7 Select **OK** to open the **Model Area Clearance** strategy form.

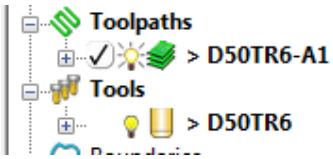


Copyright © Delcam plc

*As soon as the **Model Area Clearance** dialog is opened, the unprocessed **Toolpath** appears in the **PowerMILL explorer**.*

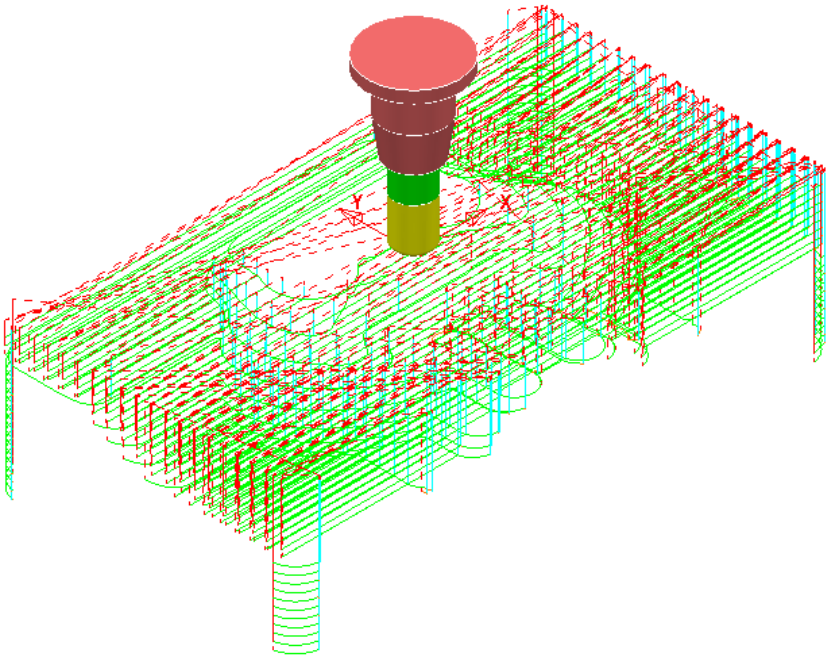
8 In the **Model Area Clearance** strategy dialog:

- **a** Enter the **Toolpath name** as **D50TR6-A1**
- **b** Select **Style** as **Raster**.
- **c** Set **Thickness** to **0.5**
- **d** Set **Stepover** to **20**
- **e** Set **Stepdown** to **10**
- **f** Keep all other values as default, **Calculate** the strategy.
- **g** After the **Toolpath** has been created, close the dialog.



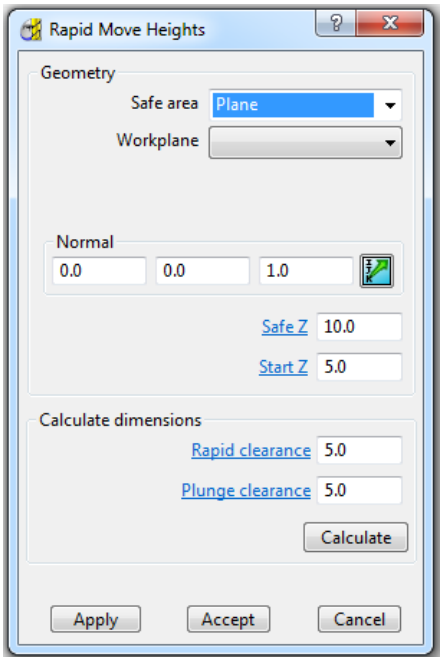
Double-click individual toolpaths to activate or de-activate them.

Click the small + box to expand and view data used to create the toolpath.



Rapid Move Heights in detail

You can modify the **Safe Z** and **Start Z** values using the **Rapid Move Heights** dialog.

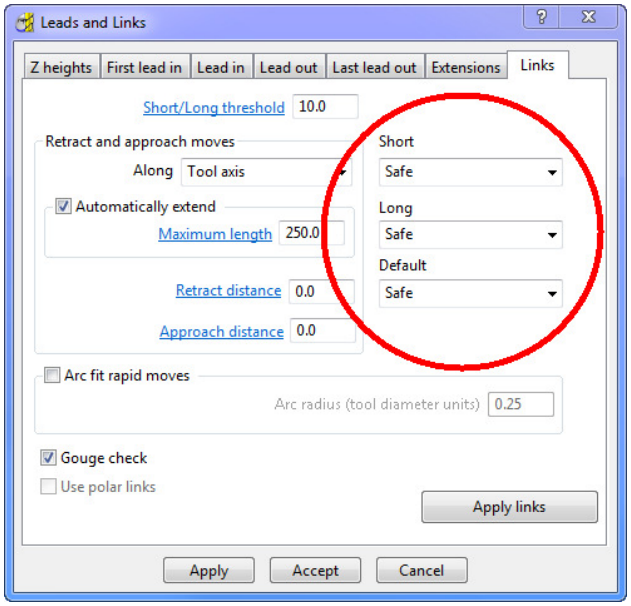


1 Click the **Calculate** button to set the **Safe Z** and **Start Z** to be at safe distances above the top of the **Model** (whole or selected) or **Block** (whichever is the highest).

Safe Z is always above Start Z.

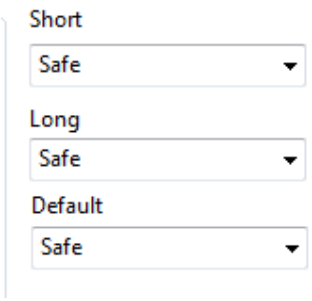
- **Safe Z** specifies a fixed height at which rapid moves across the work piece can safely occur.
- **Start Z** specifies a fixed height at which the slower, plunge feed rate is applied before the tool cuts into the work piece.

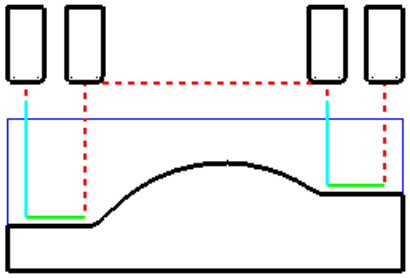
Rapid Move Heights works in conjunction with the **Leads and Links** dialog, which in turn, provides a more flexible choice of toolpath link moves.



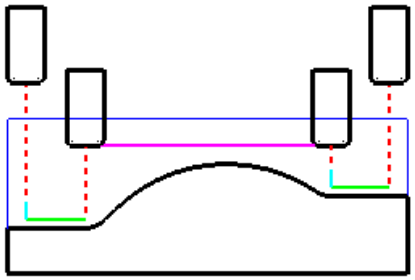
Copyright © Delcam plc

- **Safe** – is selected by default, and applies the plunge feed rate at a specific height above the job. This is more predictable and reassuring, but the non-cutting (air) moves takes more time and is inefficient, especially in the case of large, deep components.

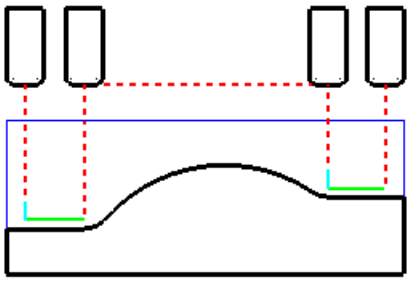




- **Skim -** enables the downward, rapid feed rate to continue to a specified **Incremental Start Z** above the full plunge depth before the slow plunge feed rate 'cuts in'. **Skim** then applies a rapid retract to an **Incremental Safe Z** above the highest point on the component 'in line' with a linear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



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

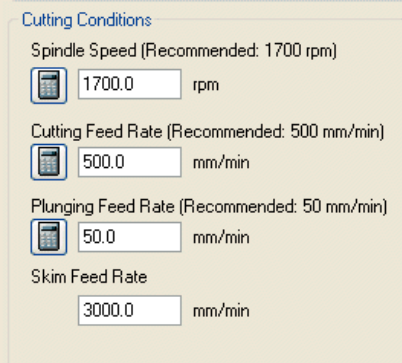


Feed Rates

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

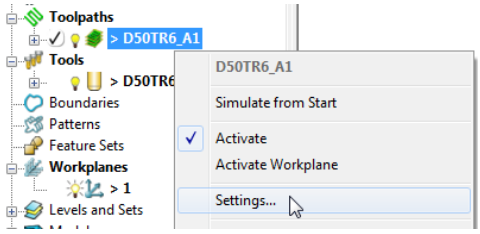
- Fixed **(G0) Rapid** moves = Red Dashed, **Toolpath** elements
- Variable **(G1) Feed Rate** moves = Green/Orange, **Toolpath** elements.
- Plunge **(G1) Feed Rate** moves = Pale Blue, **Toolpath** elements
- Skim **(G1) Feed Rate** moves = Purple, **Toolpath** elements.

1 From the **Main** toolbar, select the **Feeds and Speeds** button and enter the values (shown below) in the **Cutting Conditons** area of the dialog.



2 Click **Apply** to assign these values in the active **Toolpath** (**D50TR6-A1**) and then select **Accept** to close the dialog.

3 In the **PowerMILL *explorer***, right-click the toolpath and select **Settings...**.

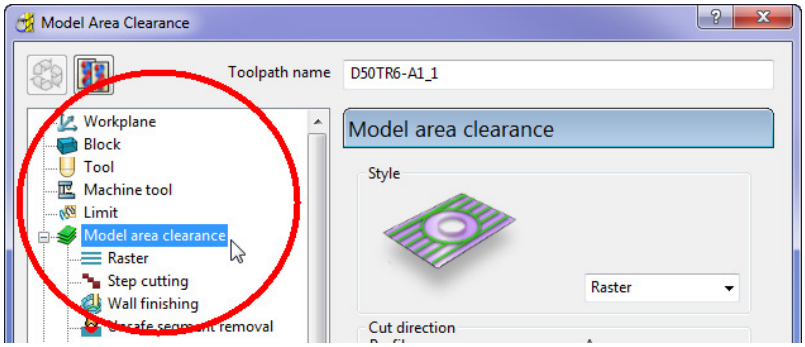


*The toolpath can also be activated or deactivated using the **Activate** option in the context menu.*

4 This reopens the **Model Area Clearance** dialog.

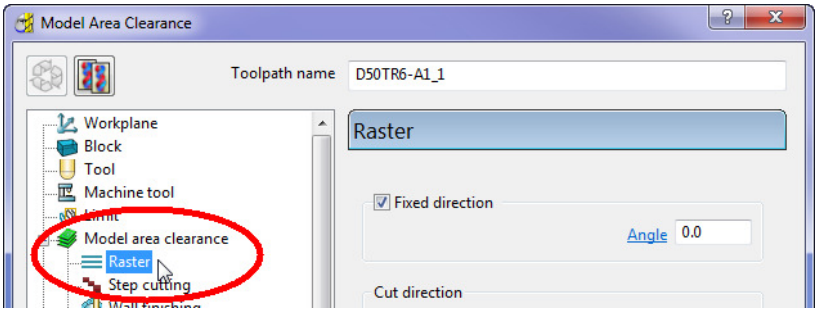
5 Click the make a copy of the **Toolpath** button.

6 On the left of the **Model Area Clearance** dialog is a local ***explorer*** window.

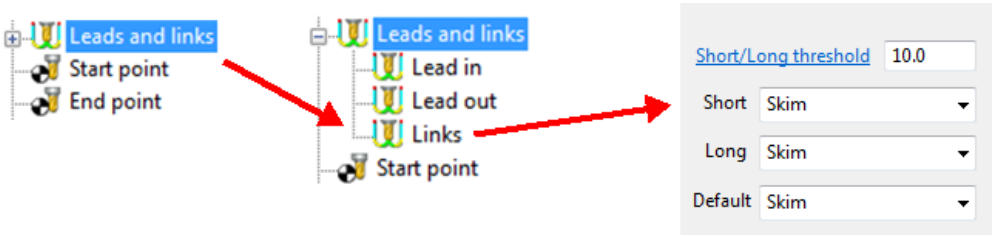


Copyright © Delcam plc

When a local ***explorer*** option is selected the individual page is displayed in the main dialog. This makes it easier to navigate specifically through the pages for the required strategy.

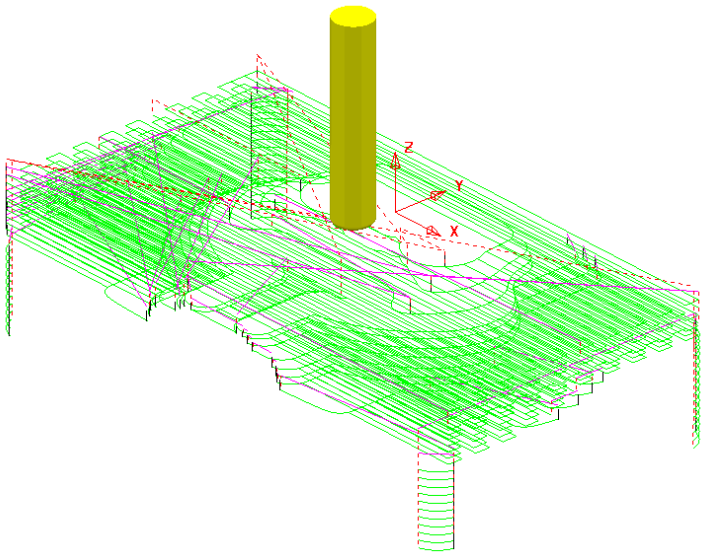


7 From the local ***explorer***, select the ⊞ to the left of **Leads and links** to access the further options **Lead in**, **Lead out**, and **Links**.



8 Select the **Links** page and change the **Short**, **Long**, and **Default Links** to **Skim**.

9 Click **Calculate** to create a new **Toolpath** with the modified settings. Once processing is complete, close the dialog.



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

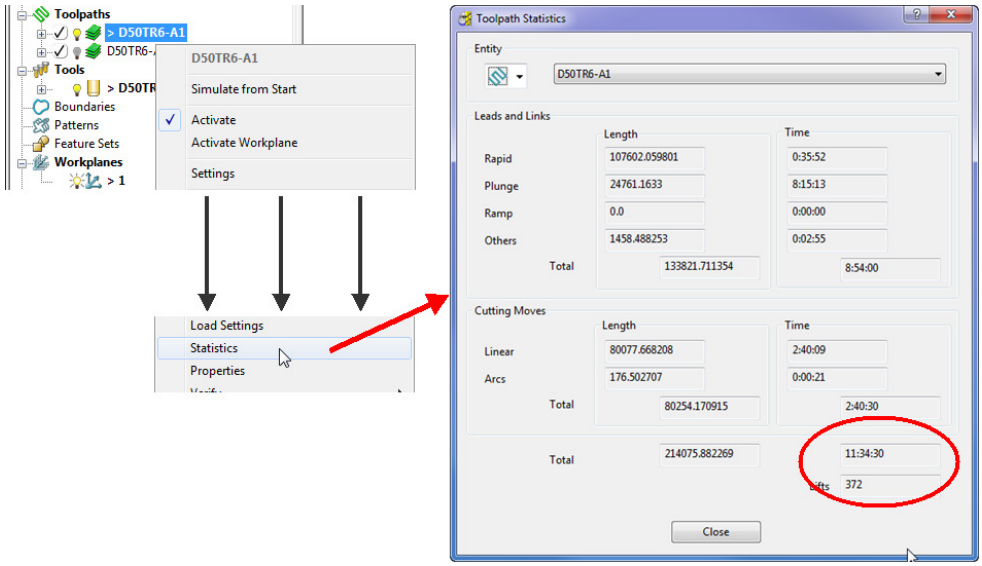
Statistics

Provides you with essential information about the active toolpath and associated parameters.

1 In the **PowerMILL *explorer***, right-click the original **Toolpath D50TR6-A1** and from the local menu select **Activate**.

2 In the same menu, select **Statistics**.

The dialog with information about the toolpath and its associated settings is displayed.



In this case, the total machining time is around **11hrs 34mins**.

3 Right mouse click on the the second **Toolpath D50TR6-A1_1** and from the local menu select and view the **Statistics**.

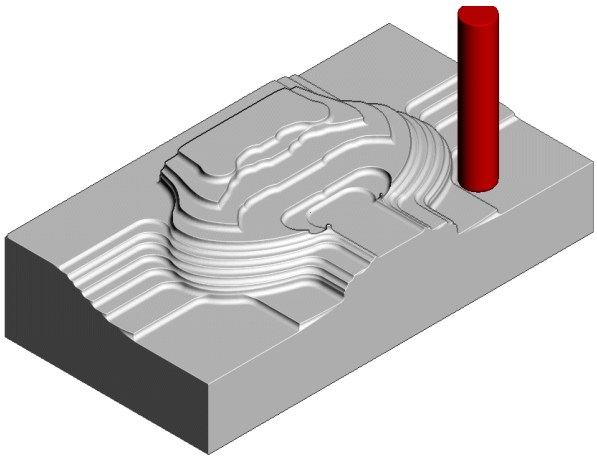
In this case, the total machining time is around **4hrs 59mins**.

This large saving is achieved simply by using **Skim** in the **Links** dialog. In the original toolpath, **Safe** was used, resulting in an excessive amount of **Toolpath link** moves being in the air.

Copyright © Delcam plc

Simulating the toolpath'

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



2 Switch the **ViewMILL** to **No Image** to return to the **PowerMILL** session and disconnect **ViewMILL** from the simulation process.

When you toggle back to **PowerMILL**, the **ViewMILL** session still exists in the background, so that any subsequent toolpaths can later be used to continue the simulation.

If the **ViewMILL** session is still connected, then, even though it is set to **No Image**, the **ViewMILL** simulation will continue to update in parallel with any further toolpath simulations.

Saving the Project

1 From the **Main** menu, select **File** > **Save Project**.

The **Project** (.....\COURSEWORK\PowerMILL-Projects\Spkr-Core) is now updated.

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

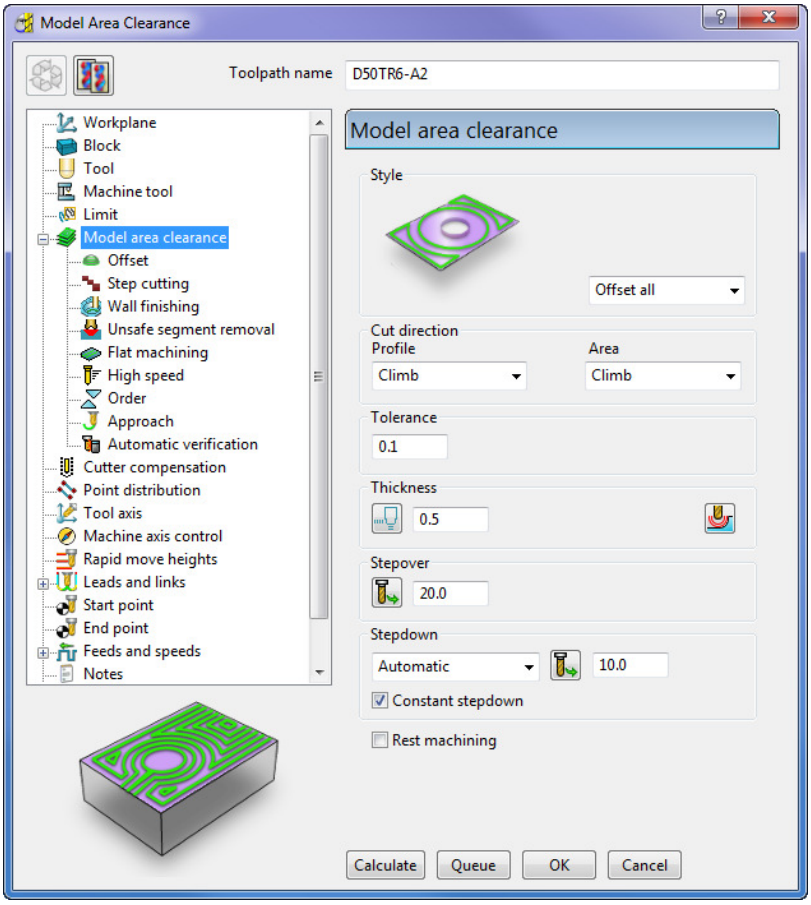
Offset All – Model Area Clearance example

The **Offset All** strategy starts by following the contour of the both the ***Block*** and ***Component form*** at the active Z height. It then progressively offsets into the remaining areas of stock. The process will tend to include plunge moves within the defined **Block**.

1 From the **Main** toolbar click the **Toolpath Strategies** button .

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

3 Select **Model Area Clearance** to display the strategy dialog.



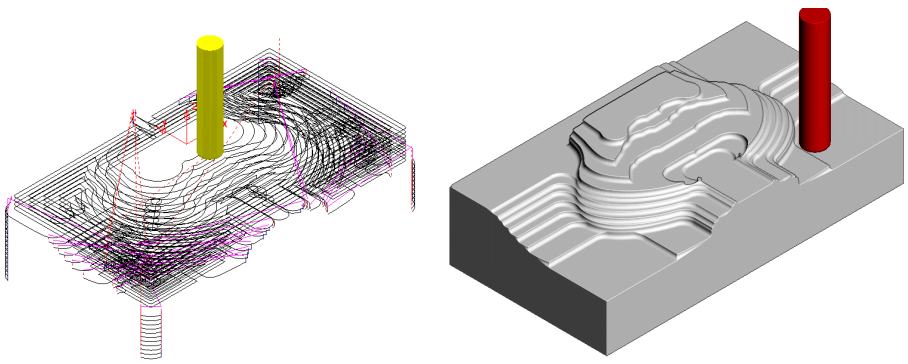
4 In the strategy dialog:

- **a** Enter **Toolpath Name** as **D50TR6-A2**
- **b** Select **Style** – **Offset all**.
- **c** Set **Thickness** to **0.5**
- **d** Set **Stepover** to **20**
- **e** Set **Stepdown** to **10**
- **f** Keep all other values as default.

5 Select **Calculate** and when the **Toolpath** is processed, close the dialog.

6 Close the current **Viewmill** simulation .

7 Perform both a **Toolpath** and **ViewMILL** simulation on the final toolpath.



Copyright © Delcam plc

8 Click the **ViewMILL** suspend button and **No Image** button to return to the **PowerMILL** session.

9 From the **Main** menu, select **File** > **Save Project**.

The **Project** *(.....\COURSEWORK\PowerMILL-Projects\Spkr-Core)* is now updated to include the **3D Area Clearance** toolpaths.

10 From the **Main** menu, select **File** > **Delete All**.

11 From the **Main** menu, select **Tools** > **Reset Forms**.

Offset Model – Model Area Clearance example

This variant of the **Offset Area Clearance** is primarily designed for ***High Speed Machining***. It provides very consistent tool loading at the expense of increased number of rapid air moves across the component. This strategy, if applied correctly, can dramatically help minimise wear to both tooling and machine.

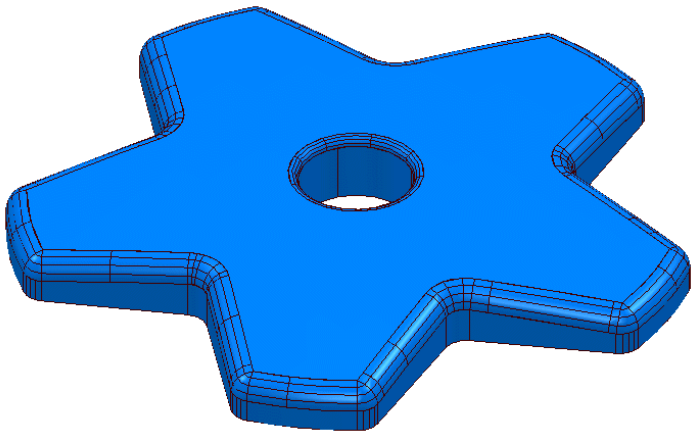
The strategy is based on the profile around the component at each Z Height, being continually offset outwards, to be trimmed to the limits of the material **Block**.

1 **Import** the **Model**:

.....\PowerMILL_Data\models\Handle.dgk

2 **Save Project As**:

.....\COURSEWORK\PowerMILL_Projects\Handle

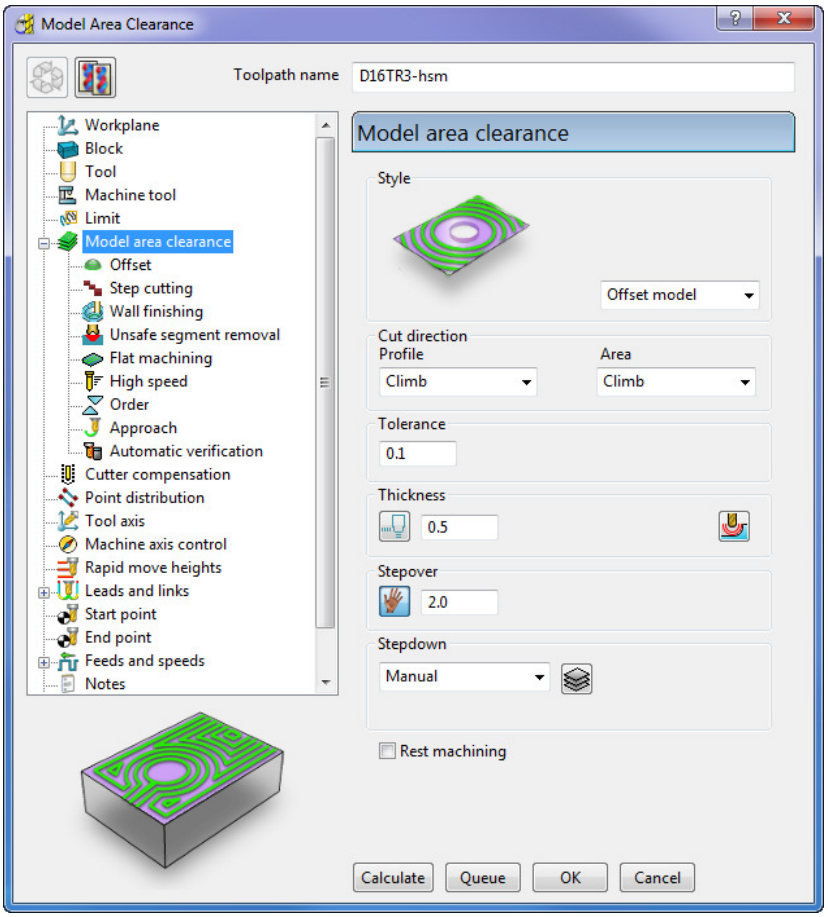


3 Create a ***Dia*** 16, ***Tiprad*** 3, **Tool** named **D16TR3**.

4 **Calculate** a **Block** to the full model dimensions. Lock the **Z max** and **Z min** values, enter an **Expansion** value of 10 and **Calculate**.

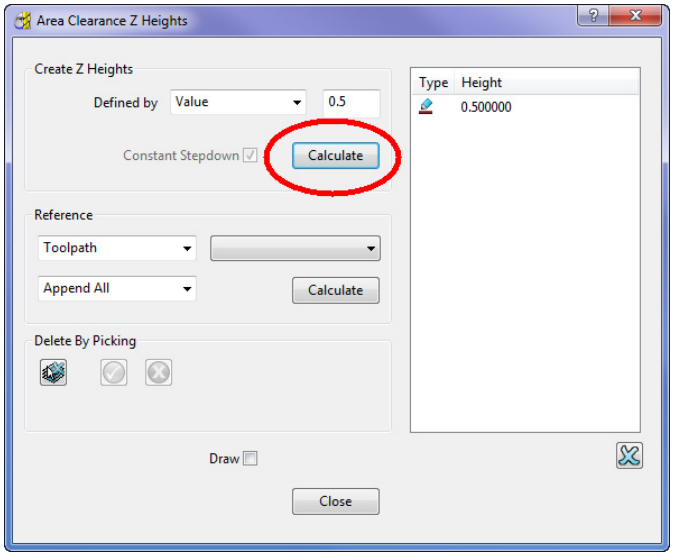
5 In the **Rapid Move Heights** dialog, select **Calculate**.

6 Select **Toolpath Strategies**, and from **3D Area Clearance**, open a **Model Area Clearance** strategy.



7 Select **Stepdown > Manual** and click the **Z Heights** button to display the **Area Clearance Z Heights** dialog.

8 In the **Area Clearance Z Heights** dialog, select **Defined by** - **Value** and enter **0.5**.



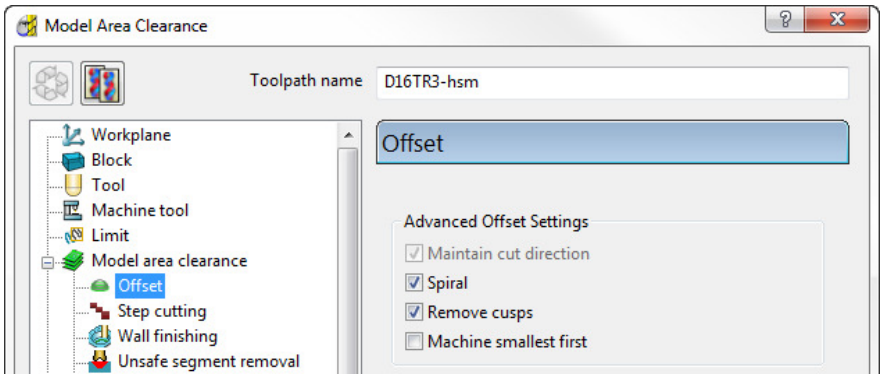
a Click **Calculate**.

*If **Z Heights** already exist, click to delete them.*

b Close the **Area Clearance Z Heights** dialog.

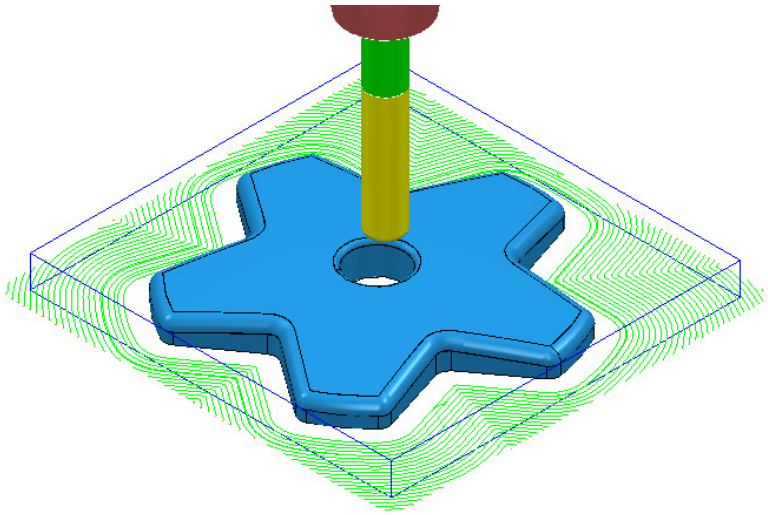
Copyright © Delcam plc

9 Select the **Offset** page and tick the **Spiral** option.



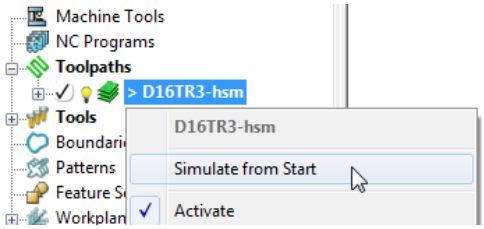
10 At the bottom of the dialog, click **Calculate** to create the **Toolpath**.

11 Close the dialog and select an **ISO 1** view.



*The toolpath is displayed above with the **Leads** and **Links** undrawn.*

12 In **PowerMILL *explorer***, right-click the **Toolpath** and select **Simulate from Start** to display the **Simulation** toolbar.



13 On the **Simulation** toolbar, select **Play** to view the toolpath simulation.

*To stop the simulation, press the **Esc** key on your keyboard.*

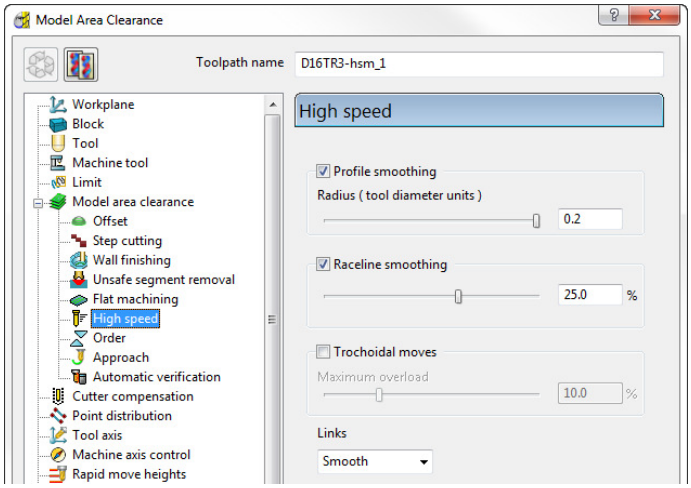
The tool continuously, climb mills, starting each tool track at a distance from the material block to allow the tool to reach optimum **Feed Rate** before contact. Each pass will benefit from having a consistent material removal rate. The strategy can be further improved using advanced settings to progressively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

Copyright © Delcam plc

14 In the **PowerMILL *explorer***, right-click the active **D16TR3-hsm** toolpath and select **Settings** to reopen the **Model Area Clearance** dialog.

15 Click and create a copy of the toolpath.

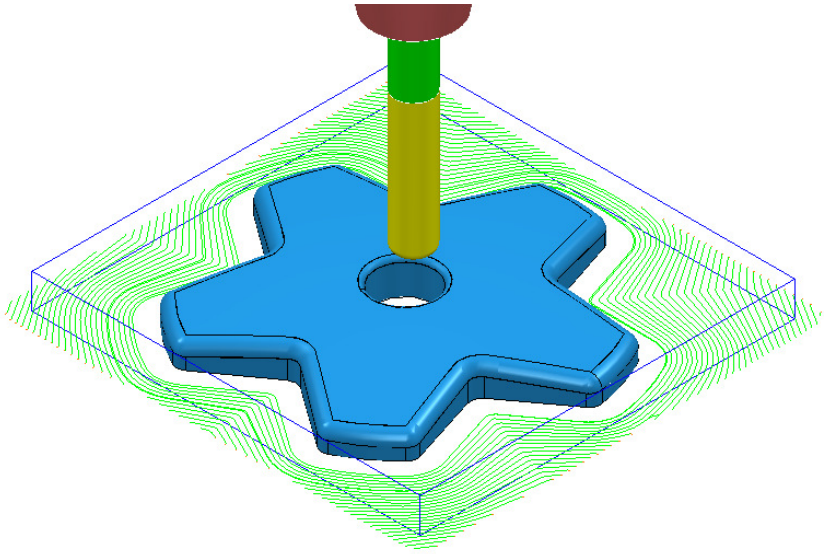
16 In the **Model Area Clearance** dialog, select the **High speed** page.



a Select **Profile smoothing** and set **Radius** (**TDU**) to the maximum slider value of **0.2**

b Select **Raceline** Smoothing and set the slider value to **25** (%).

c Click **Calculate** to create a new toolpath **D16TR3_hsm1_1** with the improved smoothing characteristics.



Compare the **Model Area Clearance** toolpaths. Note the progressive straightening of tool tracks on the second strategy with the 25% **Raceline Smoothing** applied.

17 Select **File** - **Save Project** to update the contents of:-

.....\COURSEWORK\PowerMILL_Projects\Handle

18 From the **Main** pull down menus, select **File** > **Delete All**.

19 From the **Main** pull down menus, select **Tools** > **Reset Forms**.

Copyright © Delcam plc

Area Clearance – Unsafe Segment Removal

This option is designed to filter out machining of confined areas that involves a 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.

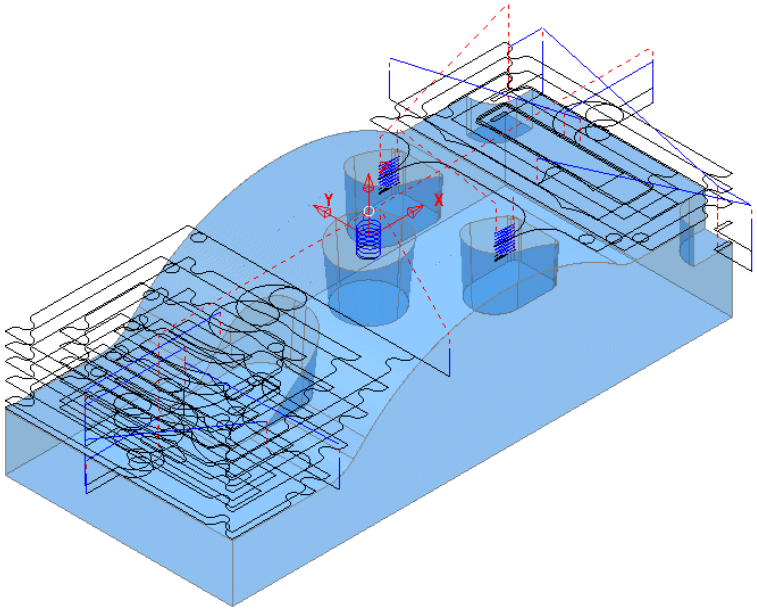
1 **Open** the read-only **Project**:

.....\PowerMILL_Data\Projects\MountingBlock-Start

2 **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\MountingBlock

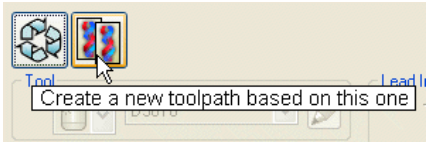
3 **Activate** the **Toolpath** named **No-AreaFilter**.



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

4 Right-click on the toolpath named **No-AreaFilter** and from the local menu, select **Settings** to open the original **Offset Area Clearance** dialog.

5 Click the **Create a new toolpath based on this one** button, to create a copy of the toolpath.

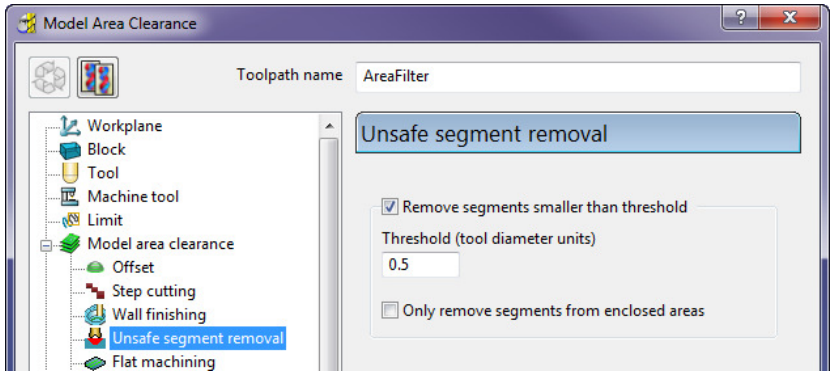


6 Rename the copy of the toolpath as **AreaFilter**.

7 In the toolpath dialog, select the **Unsafe segment removal** page.

8 Input a ***Threshold (tool diameter units)*** value of **0.5**

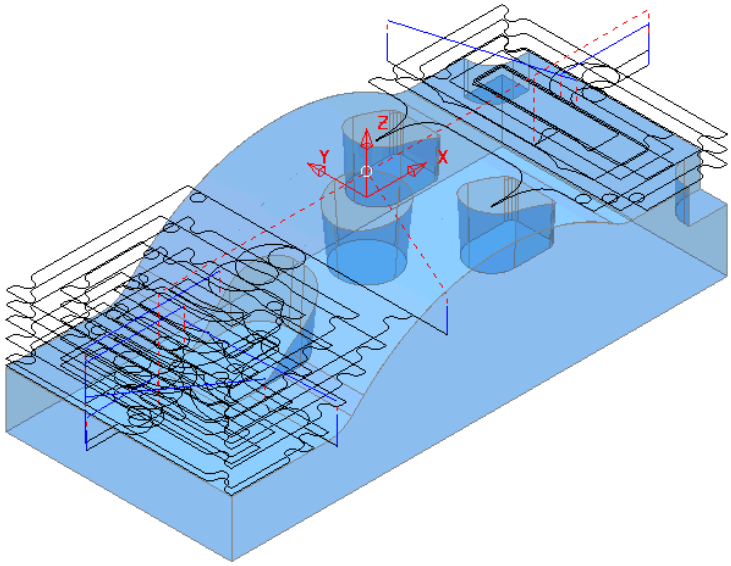
9 Un-tick the ***Only remove segments from enclosed areas*** option.



All tool tracks spanning a distance less than the Tool Diameter (TDU) are filtered out and will not appear in the final toolpath.

10 **Calculate** the **Offset Area Clear Model** dialog and once processing is complete, close the dialog.

Unsafe Segment removal has been applied to prevent the cutter

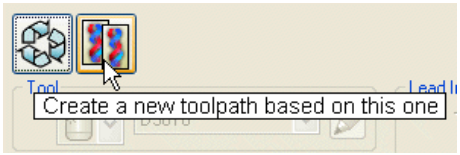


attemp ting to plunge into and machine confined areas.

However, there are a couple of undesirable spikes on the remaining toolpath. These can be reduced by applying the **High Speed** - **Profile Smoothing** option.

1 In the **PowerMILL *explorer***, right-click the toolpath named **AreaFilter** and select **Settings** to open the original **Offset Area Clearance** dialog.

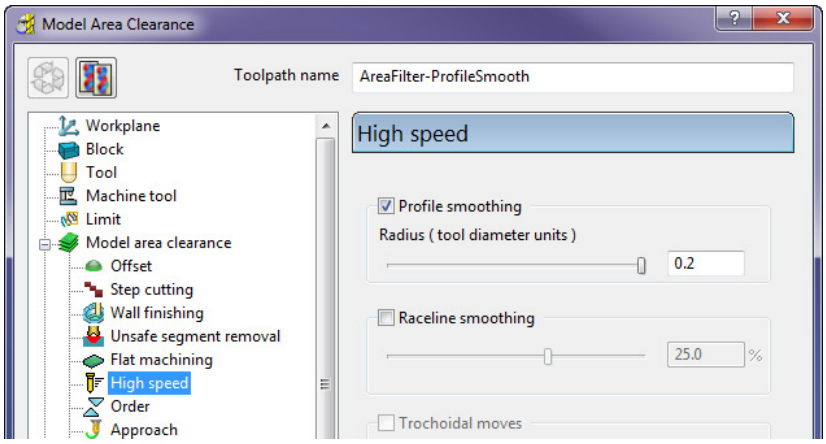
2 Click **Create a new toolpath based on this one** button, to create a copy of the toolpath.



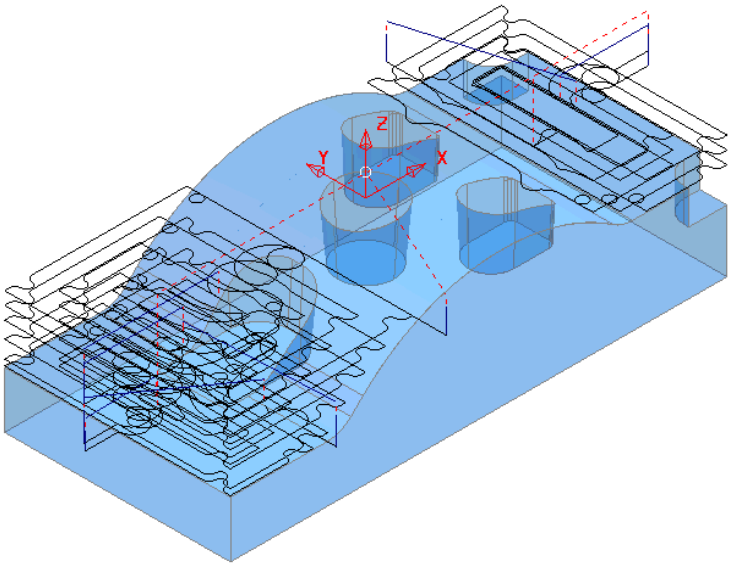
3 **Rename** the copy of the toolpath as **AreaFilter-ProfileSmooth**.

4 Keep the same **Unsafe Segment Removal** settings switched on, and in **High Speed**, select the **Profile Smoothing** option with the **Radius (Tool Diameter Units)** value set to **0.2**

 Copyright © Delcam plc



5 **Calculate** the **Model Area Clearance** toolpath and once processing is complete, close the dialog.



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

6 From the **Main** pull down menus, select **File** > **Save Project** to update the stored data (**.....\COURSEWORK\PowerMILL-Projects\MountingBlock**).

7 From the **Main** pull down menus select **File** > **Delete All**.

8 From the **Main** pull down menus select **Tools** > **Reset Forms**.

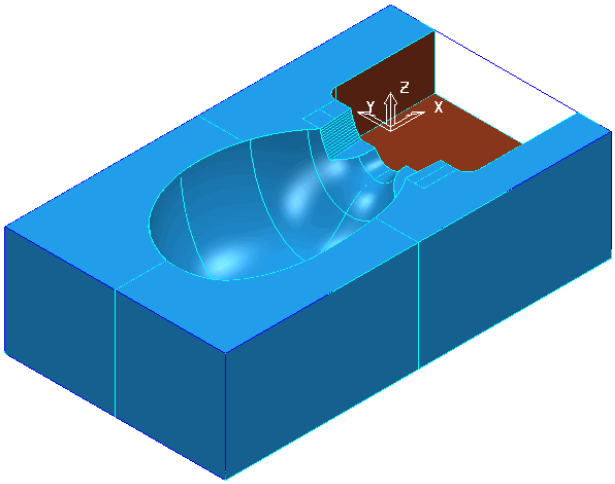
Model Rest Area Clearance example

It is generally good practice to use as larger diameter tool as possible for the initial **Area Clearance** operation. This ensures that the maximum amount of material is removed as quickly as possible. However, the large 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 tools before running the **Finishing** strategies.

In the **Model Rest Area Clearance** options, a smaller **Tool** is referenced either to an existing **Toolpath** or a **Stock Model**, such that tool tracks will only be produced locally within the remaining material (stock).

9 **Import** the **model**:

.....\PowerMILL_Data\models\WingMirrorDie.dgk



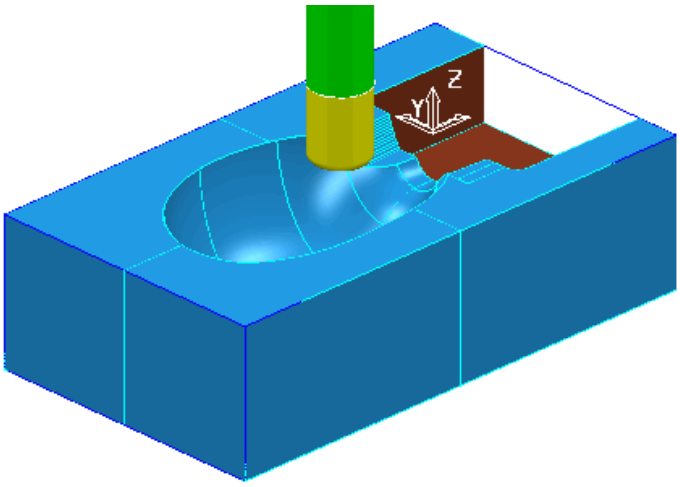
10 **Save Project** as:

.....\COURSEWORK\PowerMILL-Projects\Wing_Mirror_Die

11 Create a **Tip Radiused** tool of Dia **40** tiprad **6** and Name **D40TR6**.

12 From the **Main** toolbar open the **Block** dialog and **Calculate** using the Defined by - **Box** to and Type - **Model**.

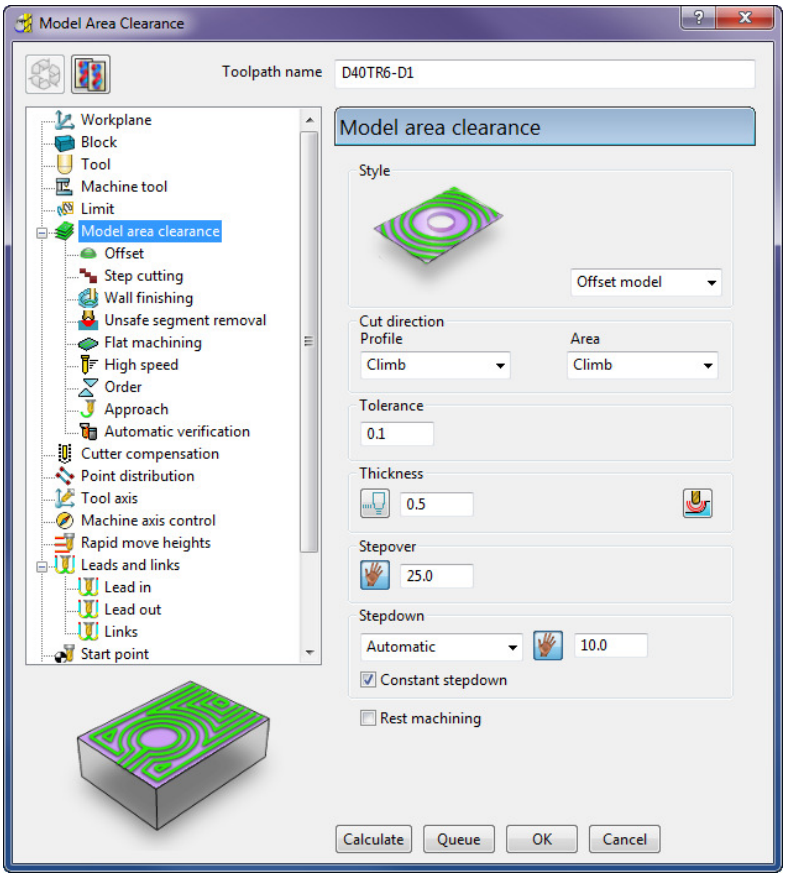
13 Using the default settings, select **Calculate** in the **Rapid Move Heights**.



Copyright © Delcam plc

14 From the **Main** toolbar, select **Toolpath Strategies** .

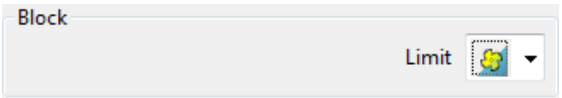
15 In **3D Area Clearance** tab, select the **Model Area Clearance** strategy.



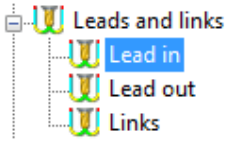
16 In the **Model Area Clearance** strategy dialog:

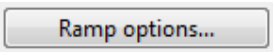
- **a** Enter **Toolpath name** - **D40TR6-D1**
- **b** Select **Style** - **Offset Model**
- **c** Input **Thickness** 0.5
- **d** Input **Stepover** 25
- **e** Input **Stepdown** 10

17 In the local ***explorer***, select the **Limits** page, and set the **Block - Limit** to **Limit tool centre to block edge**.

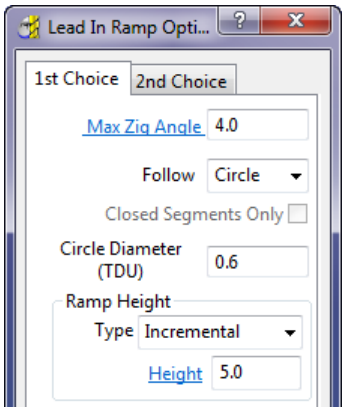


18 In the local ***explorer***, expand the **Leads and Links** options, and then select **Lead In**.

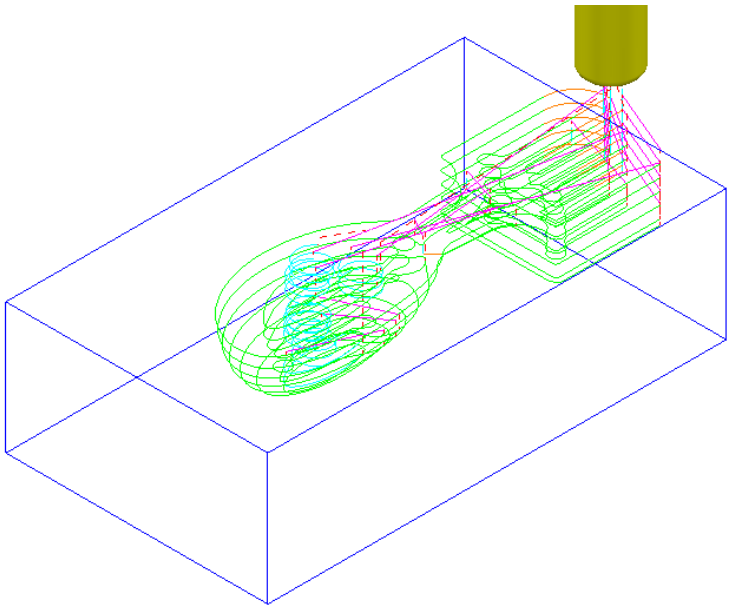


19 In the **Lead in** window, select **1st Choice** - **Ramp** - 

20 Input the following **Ramp Options**:

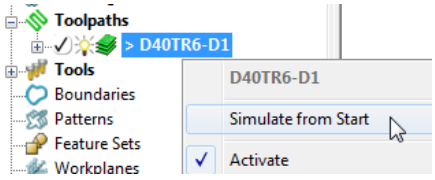


21 Click **Calculate** to process the **Model Area Clearance** toolpath.



The **Offset Model** - **Area Clearance** strategy follows the model contours while gradually offsetting into the remaining material at each Z Height.

22 Right-click the toolpath and select **Simulate from Start** to display the **Simulation** toolbar.



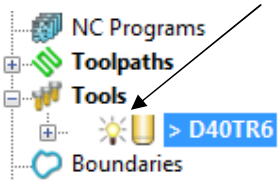
23 Ensure that the **ViewMILL** toolbar is also active.

Copyright © Delcam plc

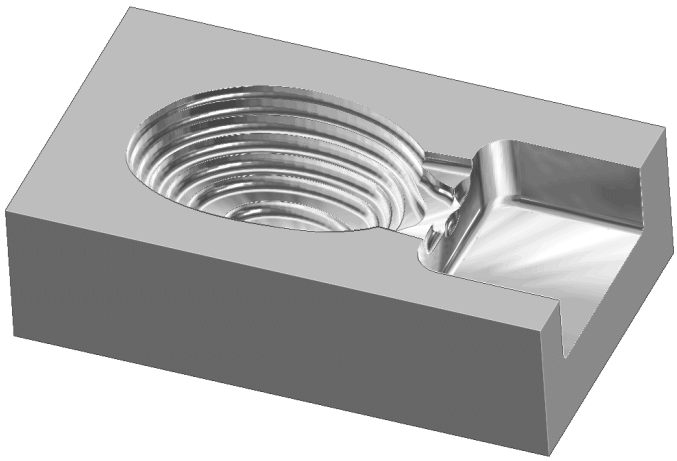
24 Switch the **ViewMILL On/Suspend** to a green sphere , and then select the ***shiny*** shaded image button .

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

Undrawing the tool will speed up the simulation. You can control the display of the tool using the light bulb on the tool entity in explorer.



The finished result indicates that the current tool has not been able to remove enough material from some areas of the model.



As a result, a further area clearance strategy is required using a smaller tool to continue locally into the remaining areas. This technique is known as **Rest Machining**.

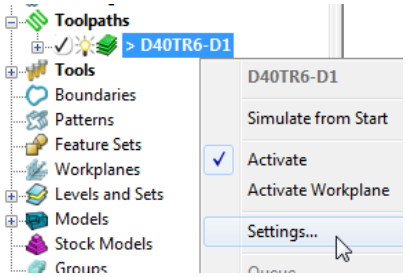
26 Click to suspend **ViewMILL**.

27 Click to return to the **PowerMILL** window.

Rest Roughing using a Reference Toolpath

1 Create a **Tip Radiused** tool of Diameter 16, Tip Radius 3 and Name it as **D16TR3**.

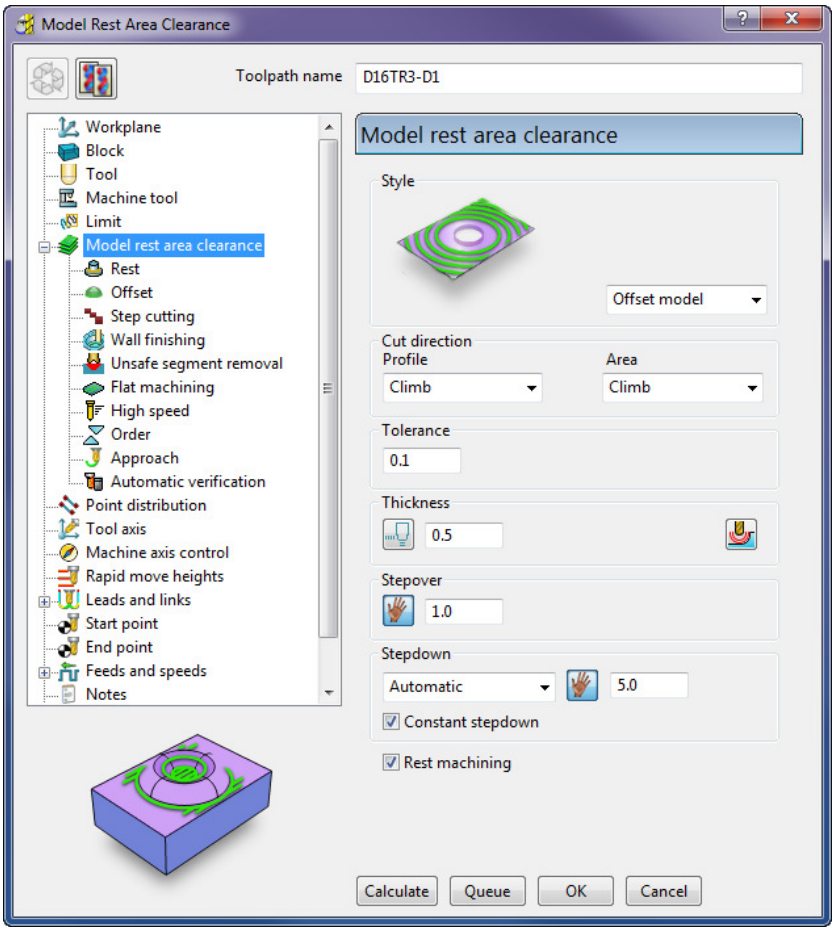
2 In the **PowerMILL *explorer***, right-click on the toolpath **D40TR6-D1** and select **Settings** to open the **Model Area Clearance** dialog.



3 Click **Create a new toolpath based on this one** button, to create a copy of the existing toolpath.

All settings originally used to create the toolpath are re-activated.

4 In the **Model Rest Area Clearance** dialog:



a Enter a new ***Toolpath name*** **D16TR3-D1**

b Enter **Thickness** as **0.5**

c Enter **Stepover** as **1.0**

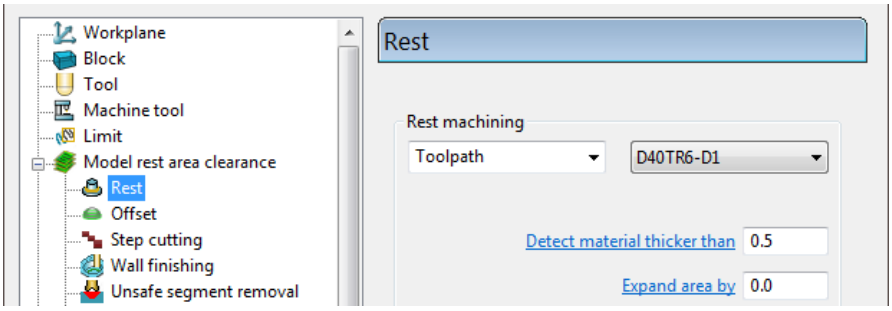
d Enter **Stepdown** as **5.0**

e Tick the **Rest machining** option to display the **Rest** page

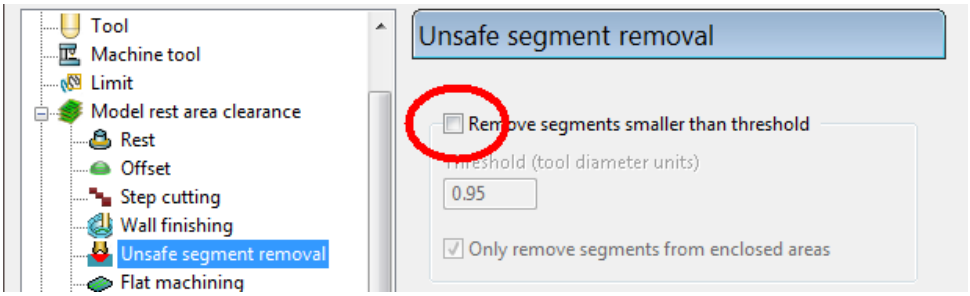
in the toolpath dialog.

 Copyright © Delcam plc

f Select the **Rest machining** type as **Toolpath**, enter **D40TR6-D1** as the reference toolpath, and input ***Detect material thicker than*** as **0.5**

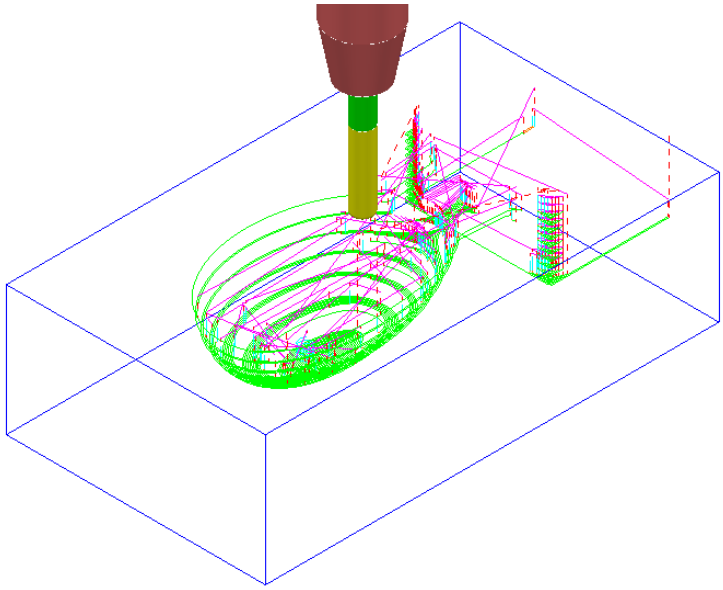


g In the local ***explorer***, select **Unsafe segment removal** page and un-tick the **Remove segments smaller than threshold** option.



h **Activate** the **D16TR3** tool to be used in the new toolpath.

i Click **Calculate** to process and create the additional **Offset Model - Area Clearance** toolpath.



j Close the dialog.

5 From the **Main** pull down menus select **File - Save Project** to update:

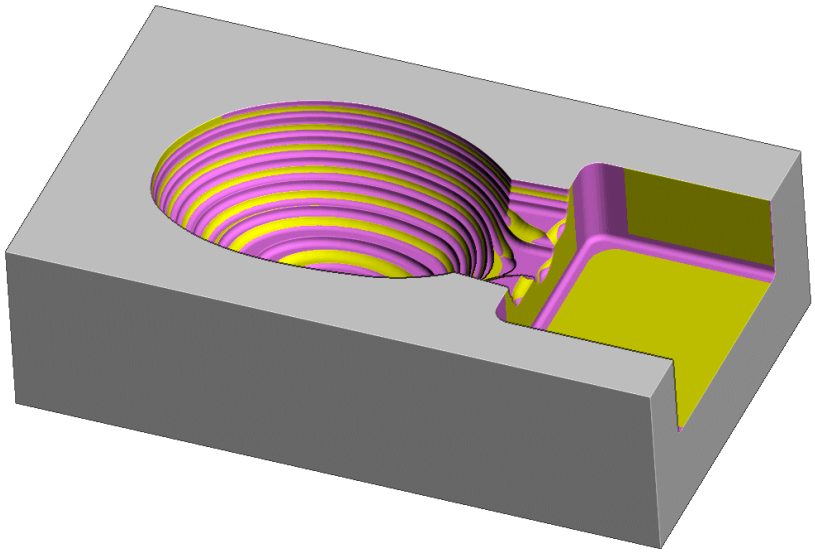
.....\COURSEWORK\PowerMILL-Projects\Wing_Mirror_Die

6 Switch the **ViewMILL On/Suspend** button to a green sphere .

7 Select the **Rainbow Shaded Image** .

8 Select the toolpath **D16TR3-D1** and **Play** the **simulation**.

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



The **Reference Toolpath** has roughed out material closer to the component form.

This will reduce the risk of excessive wear or damage to tools used for the subsequent finishing operations.

9 Click the **ViewMILL On/Suspend** button to a red sphere and then click to return to the **PowerMILL** window.

Rest Machining using a Stock Model

An alternative method for rest machining is the use of a **Stock Model** instead of a **reference toolpath**.

The benefits of using a **Stock Model** compared to a reference toolpath are:

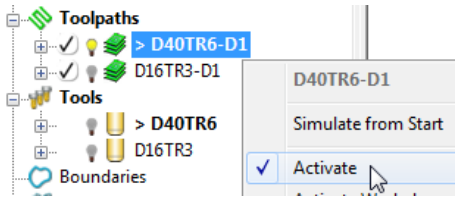
- The remaining surplus material can be visualised and registered along with the associated toolpaths.
- A rest machining strategy referenced to a **Stock Model** will account for all previous operations currently registered to that **Stock Model**.
- A reference machining strategy created to a different workplane (for example, multi-axis) can be referenced to a **Stock Model**.

It is not possible to apply a rest machining strategy referenced to a toolpath that was created to a different workplane.

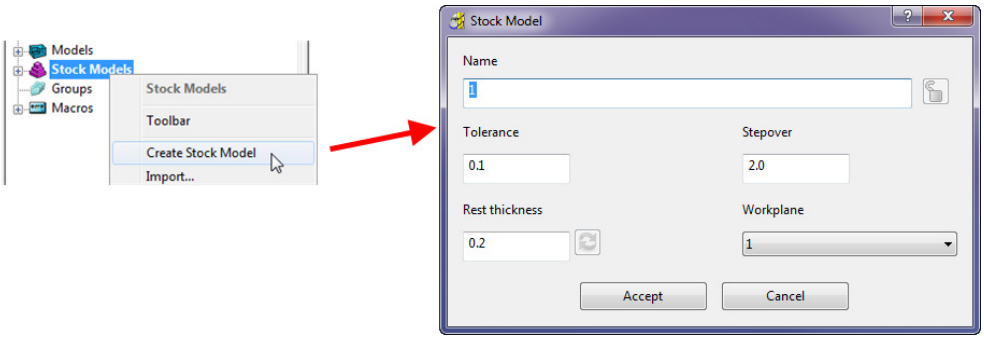
This example first creates and displays a **Stock Model** for the **D40TR6-D1** toolpath. Then, a copy of the **D16TR3-D1** toolpath is created to use the **Stock Model** as the rest machining reference.

Copyright © Delcam plc

1 In the **PowerMILL *explorer***, **Activate** the toolpath **D40TR6-D1**.

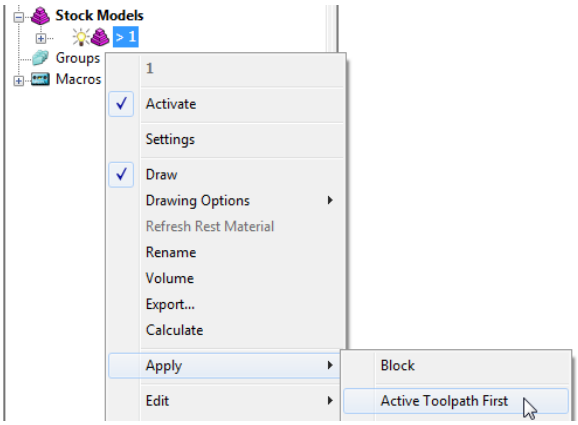


2 In the **PowerMILL *explorer***, right-click **Stock Models** and select **Create Stock Model**.

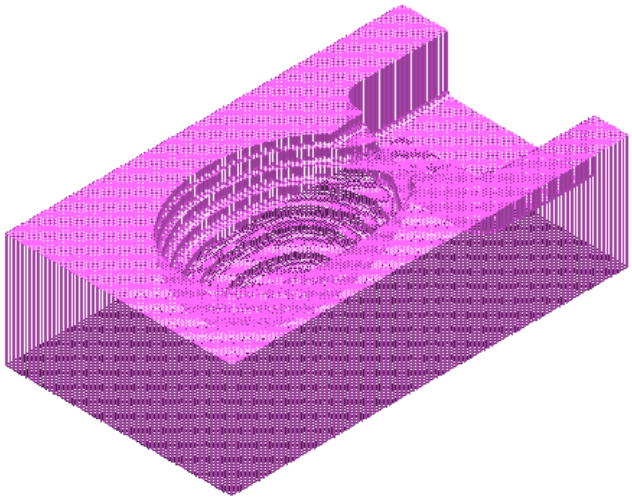


3 **Accept** the above default values for the ***triangle mesh*** creation.

4 In the **PowerMILL *explorer***, right-click the newly created **Stock Model** 1 and select **Apply** > **Active Toolpath First**.



5 From the same menu, select **Calculate** to create the **Stock Model**.



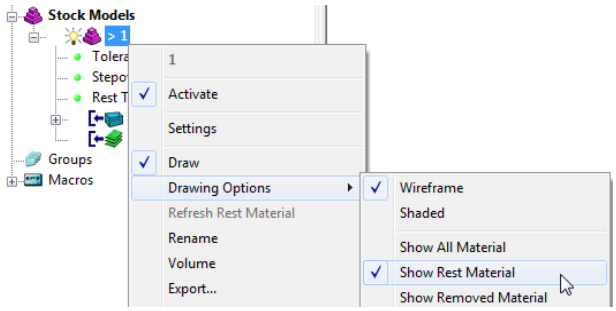
A triangular mesh will appear representing the state of the material after the effect of the first machining strategy, **D40TR6-D1**.

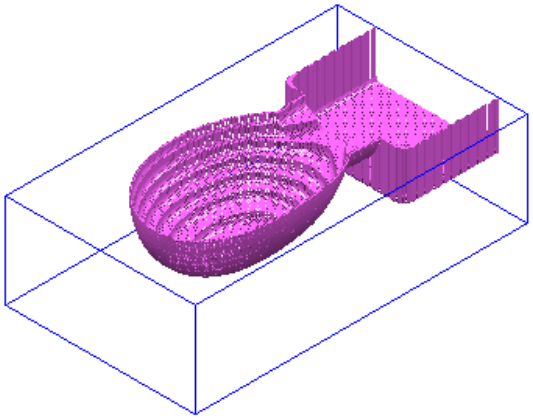
In the next two actions, The **Stock Model** will be changed to:-

a Visually remove the component form (**Show Rest Material**).

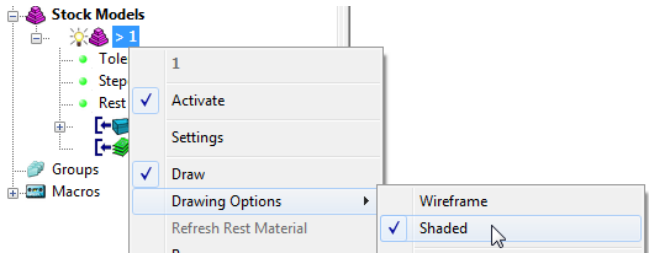
b Visually change the style from triangle mesh to a shaded form (**Shaded**).

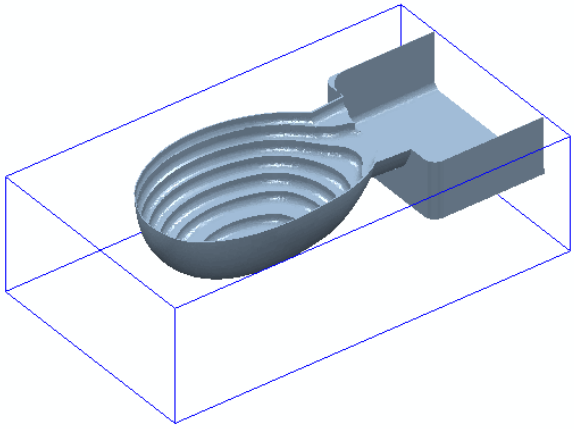
6 From the same menu, select **Drawing Options** > **Show Rest Material**.





7 From the same menu, select **Drawing Options** > **Shaded** to display the shaded view of the **Stock Model**.





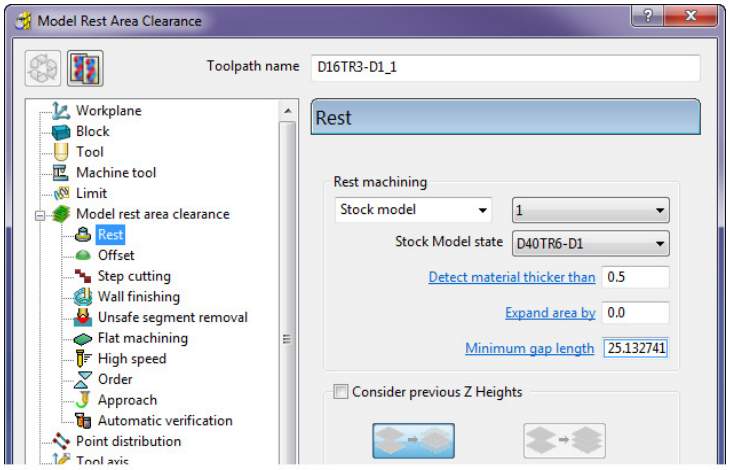
 Copyright © Delcam plc

8 In the **PowerMILL *explorer***, **Activate** the toolpath **D16TR3_D1**.

9 Right-click on the toolpath **D16TR3_D1** and select **Settings** to open the **Model Rest Area Clearance** strategy dialog.

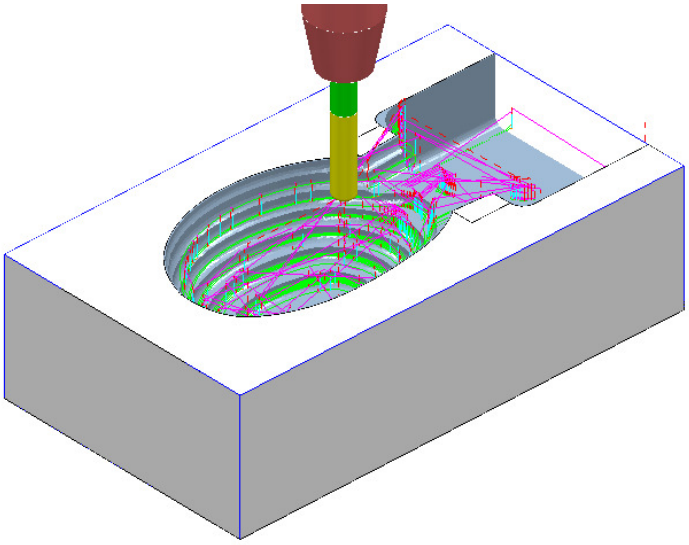
10 Click to create a copy of the toolpath.

11 Select the **Rest** page and modify the settings as below using **Stock model** 1.

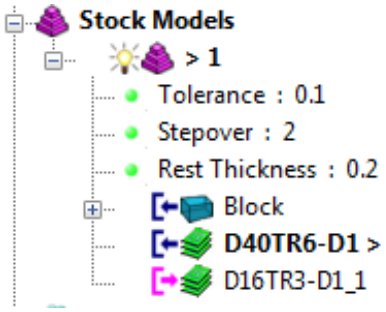


12 Click **Calculate** to create an alternative and more efficient **Rest Machining** toolpath named **D16TR3-D1_1**.

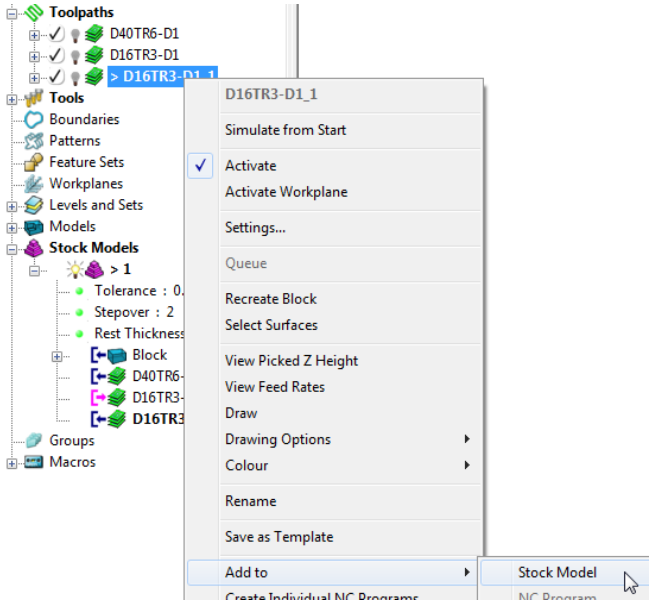
13 C**lose** the machining strategy dialog.



Although the toolpath **D16TR-D1_1** appears as a referenced entity in the **Stock Model** list, the resultant machining effect is not automatically added.

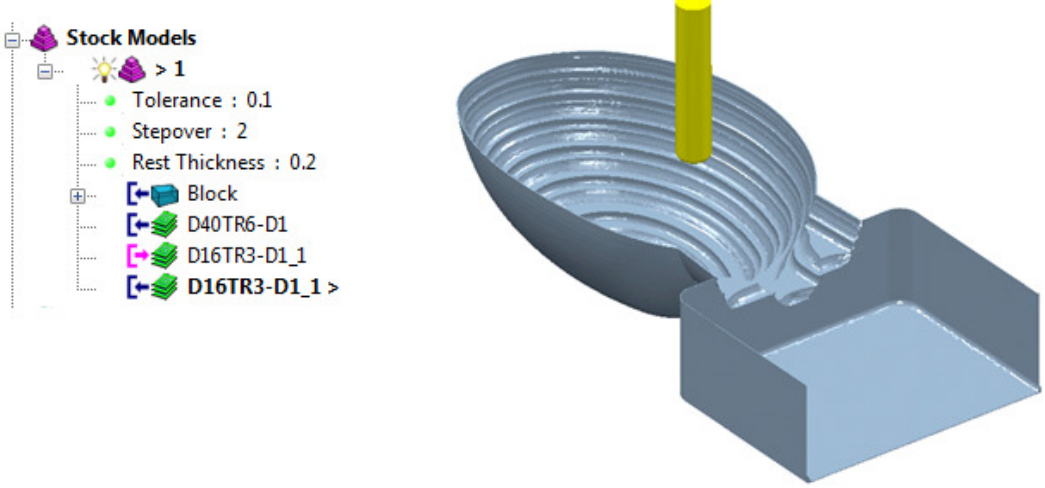


14 In the **PowerMILL *explorer***, right-click the toolpath, **D16TR3-D1_1** and select **Add to** > **Stock Model**.



*At this stage, the **Stock Model** will disappear from the screen.*

15 Right-click the active **Stock Model** and select **Calculate** to update it.



16 From the **Main** pull down menus, select **File** > **Save Project** to update:

.....\COURSEWORK\PowerMILL-Projects\WingMirrorDie

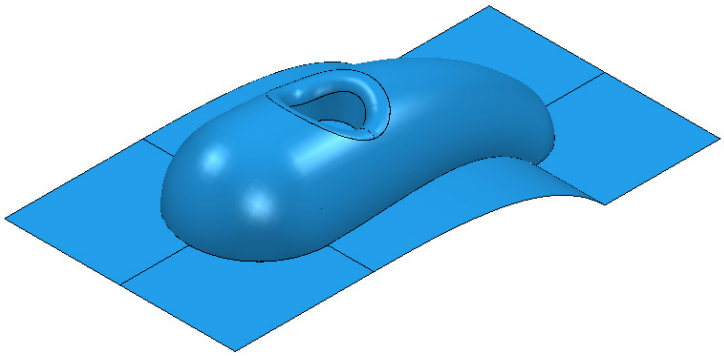
Copyright © Delcam plc

Exercise

Area clearance with both unsafe segment removal and rest roughing

1 **Import** the **Model**:

.....\PowerMILL_Data\Models\Cowling



Machining setup

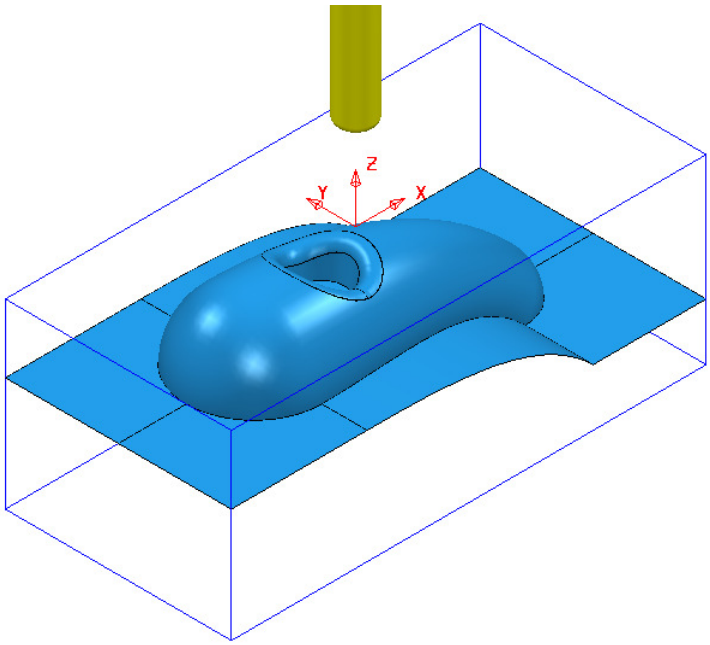
2 Create the following tools:

- **Tip Radiused** tool – ***Name*** **D16TR3**, Diameter 16, Tip Radius 3.
- **Tip Radiused** tool – ***Name*** **D12TR1**, Diameter 12, Tip Radius 1.

3 Create a **Workplane** and centrally position it to the ***top centre*** of the **Model** as a more suitable machining datum.

4 **Calculate** a **Block** using ***Defined by*** – **Box** and **Type** - **Model**.

5 **Calculate** the **Rapid Move Heights** using the default settings.



Area Clearance Strategies

1 Create a **Model Area Clearance** strategy using:

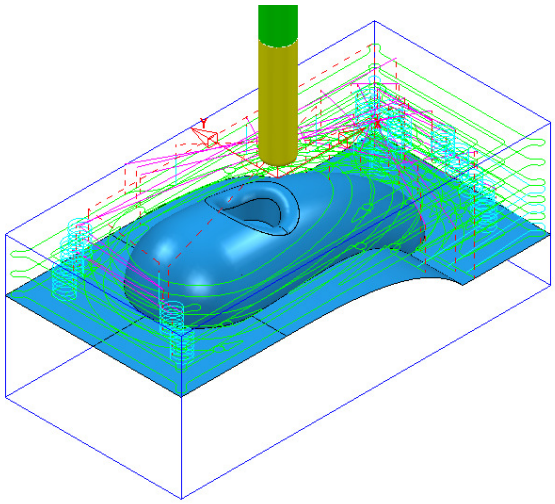
a The **D16TR3** tool.

b The ***Style*** as **Offset All**.

c ***Stepover*** 8 and ***Stepdown*** 12

d A ***Thickness*** of 0.5

e The **Unsafe segment removal** option to prevent the **D16TR3** tool from attempting to machine into the central pocket.



2 Create another **Model Area Clearance** strategy using:

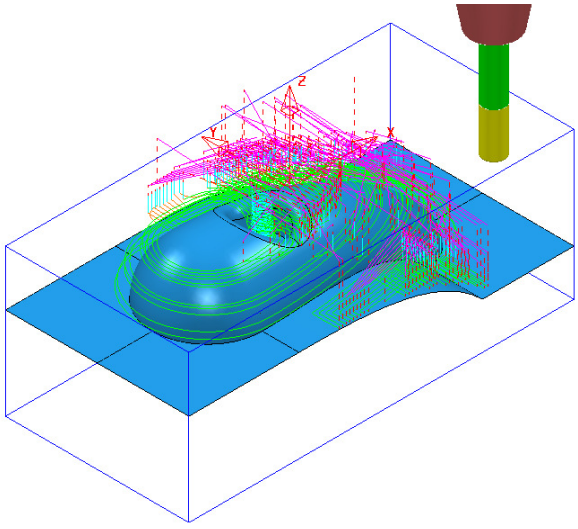
a The **D12TR1** tool.

b The ***Style*** as **Offset Model**.

c Stepover 2 and Stepdown 3

d Enable the **Rest machining** page and reference to the previous toolpath.

e Disable the **Unsafe segment removal** option.



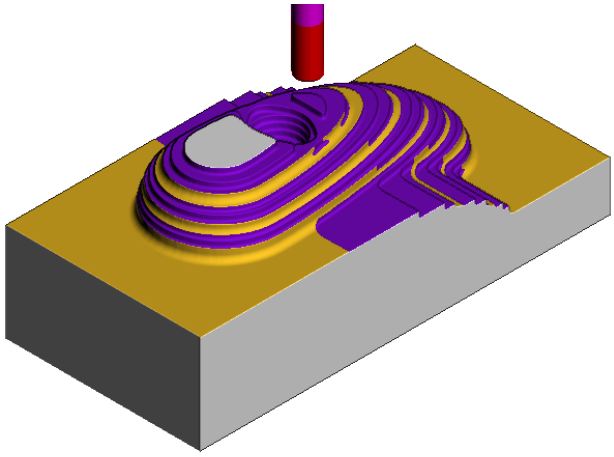
3 **Save Project As**:

.....\COURSEWORK\PowerMILL_Projects\ModelAreaClear-Exercise

Copyright © Delcam plc

ViewMILL Simulation

1 Perform a **ViewMILL** simulation with the **Rainbow Shaded Image** option active on both of the **Model Area Clearance** toolpaths.



2 From the **Main** pull down menus, select **File** > **Delete All**.

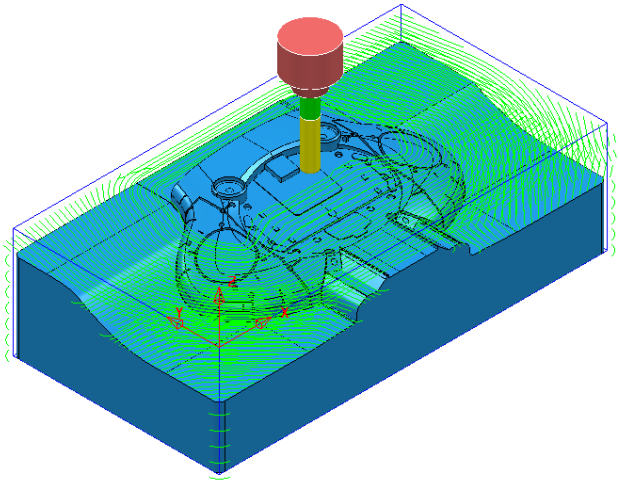
3 From the **Main** pull down menus, select **Tools** > **Reset Forms**.

Area Clearance using the Step option

If a large stepdown is used in the **Area Clearance** strategies then the bulk of the waste material will be removed as quickly as possible. This will inevitably leave large steps on the remaining stock that will need to be reduced using a separate **rest** roughing operation. If it is acceptable to use the same **Tool** then the **Step** option provides a form of rest machining within the same **Area Clearance** strategy. Additional passes are applied (where required) that step upwards between the original Z Heights

1 Select **File** – **Open Project** (Read only):-

...\PowerMILL_Data\Projects\StepRoughing_Start



2 **Save Project As**:

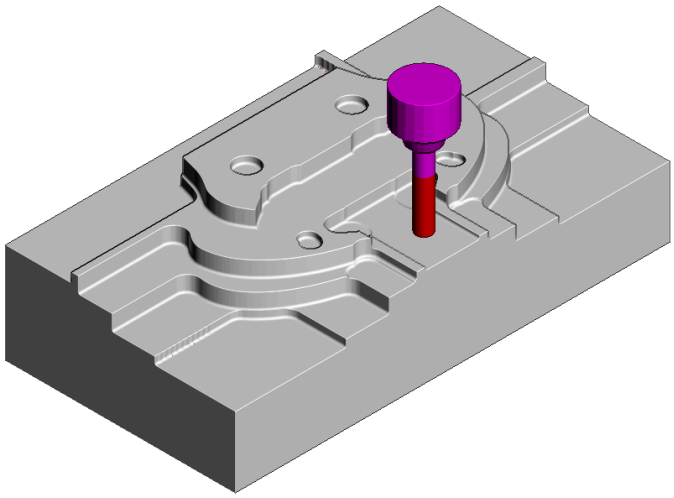
...\COURSEWORK\PowerMILL_Projects\StepRoughing-EX1

The **Project** contains the ***Speaker_Core*** model and a **Model Area Clearance** strategy (**D25TR3-RGH1**) with ***Machine flats*** – **Area** active.

3 **Activate** the toolpath **D25TR3-RGH1** to restore the settings used.

4 **Calculate** the **Block** to the **Model** dimensions.

5 Run a **ViewMILL** simulation on the toolpath **D25TR3-RGH1**.



The original strategy has used a large stepdown and ignored the flat areas so that the bulk of the material is removed as quickly as possible. This has left a large amount of stock on the both the steps and flat areas as shown in **ViewMILL**. To accommodate this, the same strategy will be re-cycled and appropriate **Step** options applied to produce a form that is closer to the model shape.

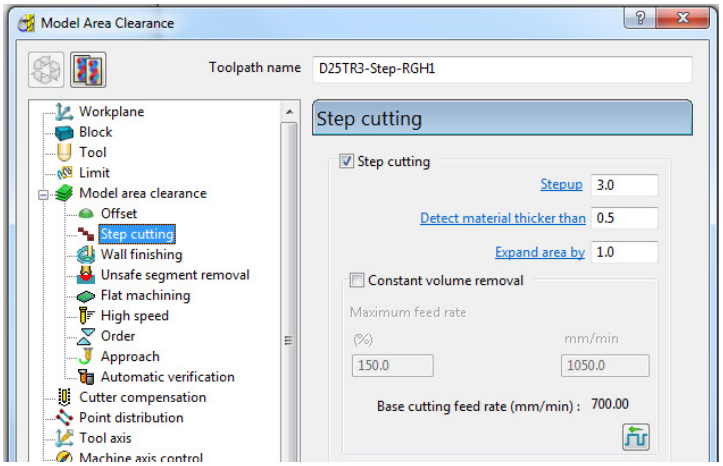
6 Select **Exit ViewMILL** to close the session.

7 Right mouse click on the active toolpath **D25TR3-RGH1** and from the local menu select **Settings**.

8 Select the 'Re-cycle' toolpath option .

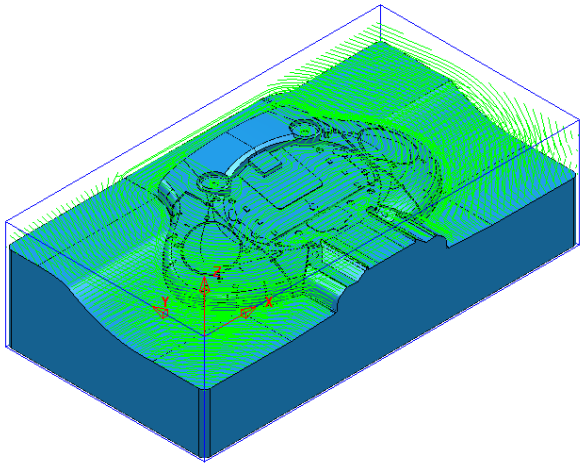
9 In the **Model Area Clearance** dialog, local ***explorer***, select the **Step** option.

10 Tick the **Step cutting** box and enter values exactly as shown below.

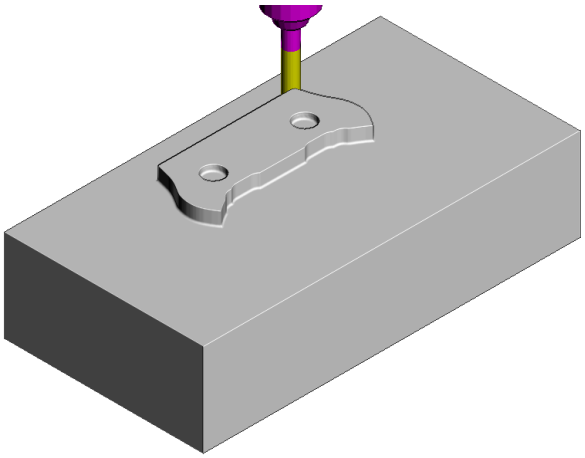


 Copyright © Delcam plc

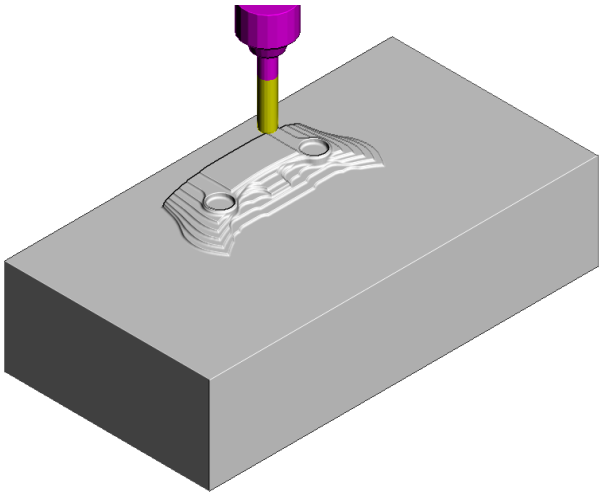
11 Select **Calculate** to process the modified strategy.



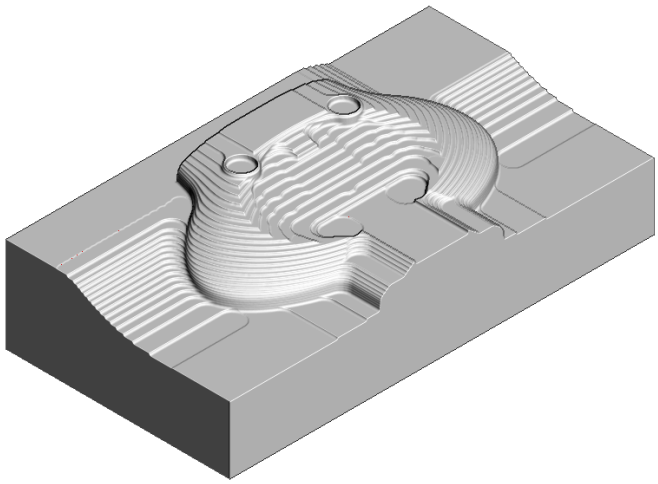
12 Run a new **ViewMILL** simulation on the toolpath **D25TR3-RGH1**.



The above image shows the first **slice** being fully machined.



The above image shows the effect of the **Step cutting** option tracking upwards from the first slice to get closer to the model form.



The Component has been 'Roughed Out' with maximum efficiency, close to the final shape within the same toolpath.

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

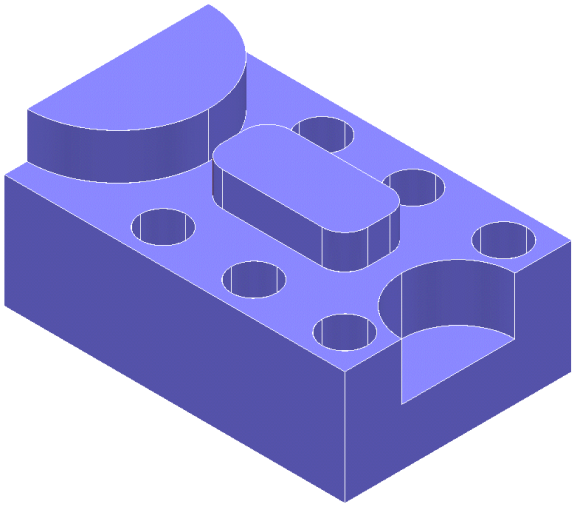
14 From the **main** pulldown menus select **File** - **Delete All** and **Tools** – **Reset Forms**.

Machining Flats within Model Area Clearance

The **Model Area Clearance** strategies in **PowerMILL** include options to control the way in which flat areas of the component model are machined. These are accessed from the **Machine Flats** page on the dialog.

1 **Import** the **Model**:

.....\PowerMILL_Data\Models\Flats.dgk



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

.....\COURSEWORK\PowerMILL-Projects\AreaClearFlats

3 Create a 12mm diameter **End Mill** tool named **EM12**.

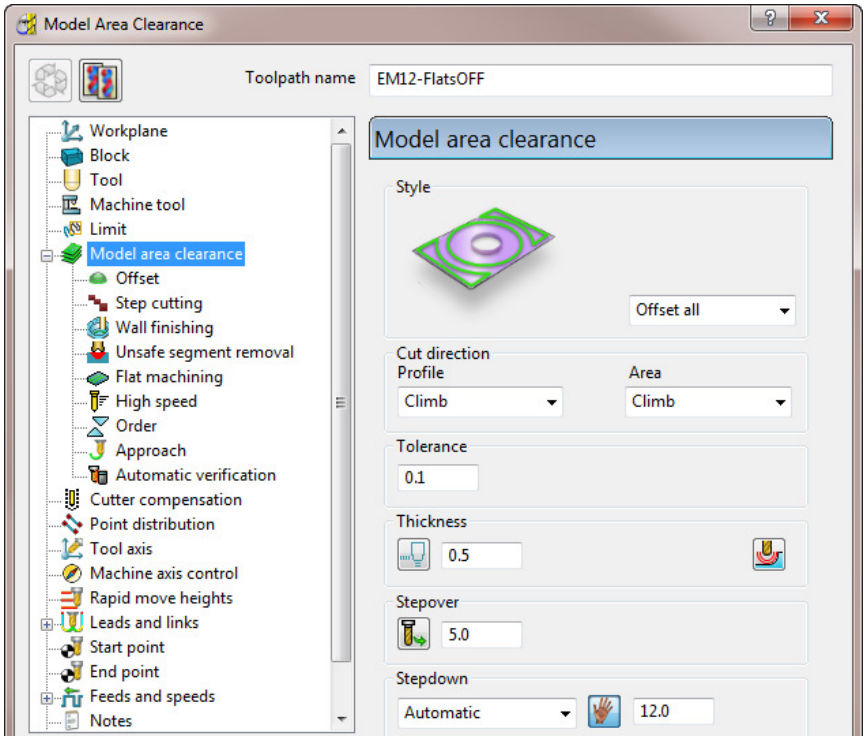
4 **Calculate** the **Block** using the default settings.

Copyright © Delcam plc

5 **Calculate** default **Rapid Move Heights**.

6 Keep the **Start and End Point** settings as default.

7 In the **Strategy Selector**, select the **3D Area Clearance** tab, and then open the **Model Area Clearance** strategy form.



8 In the **Model Area Clearance** strategy dialog:

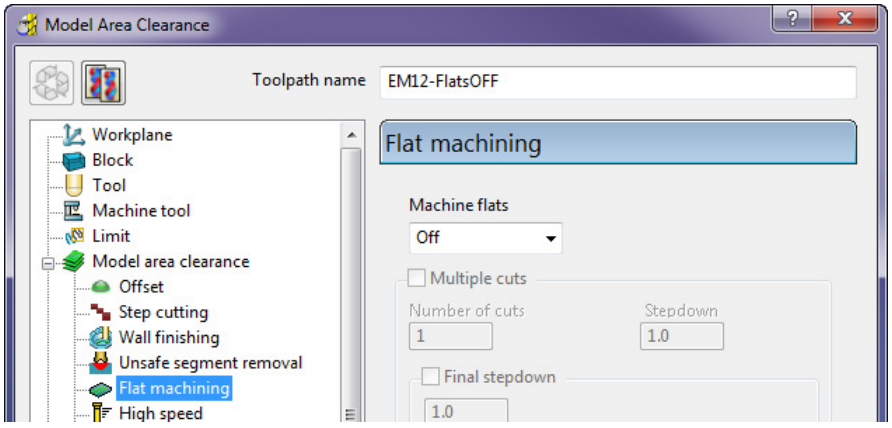
a Enter the ***Name*** as **EM12-FlatsOFF**.

b Set **Thickness** to **0.5**

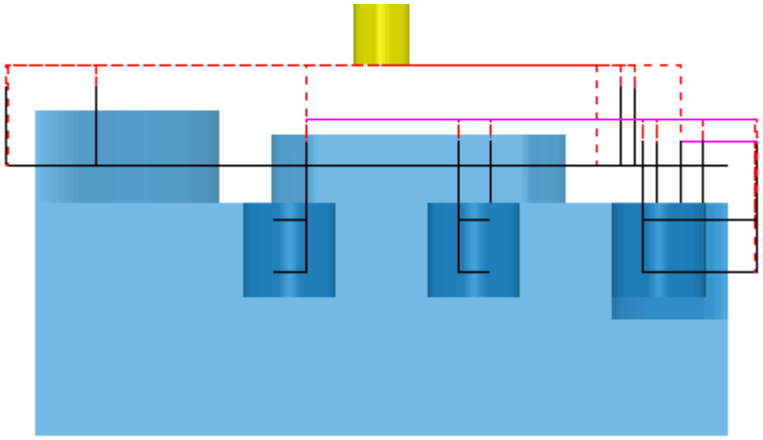
c Set **Stepover** to **5**

d Set **Stepdown** to **12**

e Select the **Flat** machining page and set **Machine flats** to **Off**.



9 Click **Calculate** to process the toolpath and then **Close** the dialog.



When the **Machine flats** option is set to **Off**, the toolpath totally ignores the flat surfaces of the model. It maintains the defined **Stepdown** value and performs area clearance across the material **Block** at each **Z Height**.

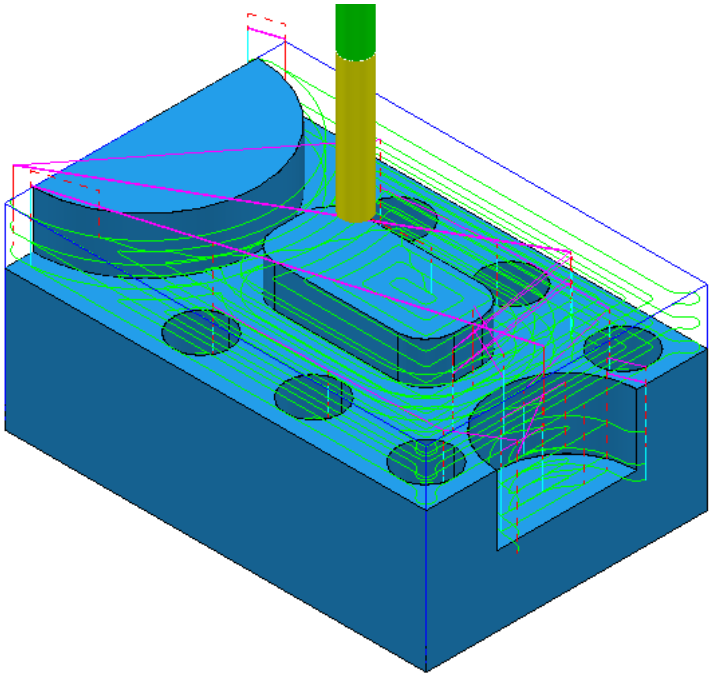
1 Right-click the active toolpath and select Settings.

2 Click to create a copy of the toolpath.

3 Change the new toolpath name to **EM12-FlatsLEVEL**.

4 On the **Flat machining** page, change ***Machine flats*** to **Level**.

5 **Calculate** to process the toolpath and then **Close** the dialog.



At the Flat area heights now removes material across the whole block area leaving 0.6mm stock above each Flat face (The **Thickness** plus the **Tolerance** values).

6 Right-click the active toolpath and select **Settings**.

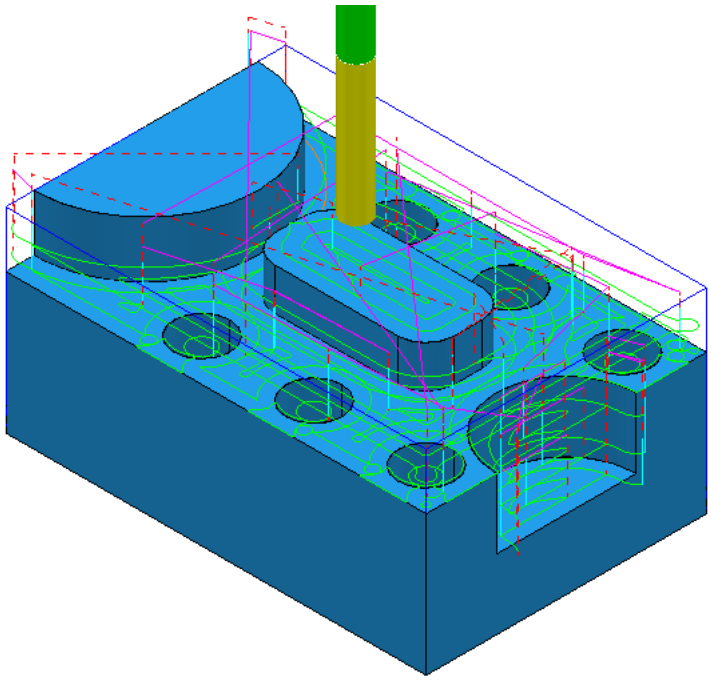
7 Click to create a copy of the toolpath.

8 Change the toolpath name to **EM12-FlatsAREA**.

Copyright © Delcam plc

9 On the **Flat machining** page, change ***Machine flats*** to **Area**.

10 **Calculate** to process the toolpath and then **Close** the dialog.



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

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

.....\COURSEWORK\PowerMILL-Projects\AreaClearFlats

Machining Flats separately using Slice Area Clearance

In **PowerMILL Pro** it is possible to use **Slice Area Clearance** to separately machine the flat areas of a model to their exact height. This could be used:

a As a final finishing strategy, applied to the flat areas after a **Model Area Clearance** where machine flats was set to **Level** or **Area**.

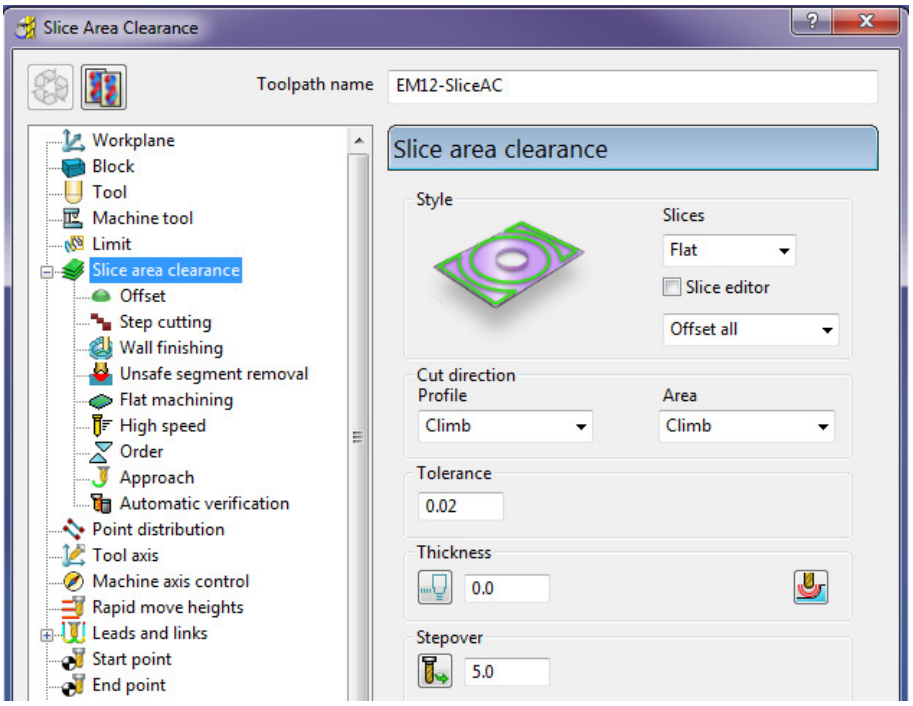
b As a secondary roughing strategy, applied to the flat areas only, that follows on from a **Model Area Clearance** where ***Machine flats*** was set to **Off**.

12 Right-click on the **EM12-FlatsArea** toolpath and select **Activate** to reinstate all settings and parameters used.

13 On the **Main** toolbar, click **Toolpath Strategies** .

14 Select **3D Area Clearance** tab and from the options, select **Slice Area Clearance** to open the following dialog:

Copyright © Delcam plc



a Enter ***Name*** as **EM12-SliceAC**.

b Set ***Slices*** as **Flat**.

c Set ***Style*** as **Offset all**.

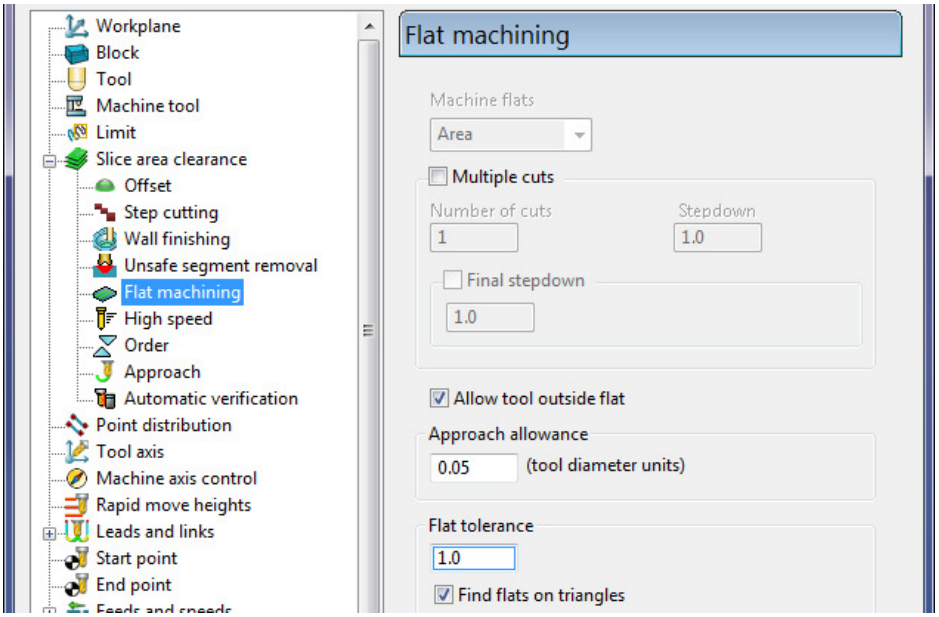
d Set ***Tolerance*** 0.02

e Set ***Thickness*** 0

f Set ***Stepover*** 5

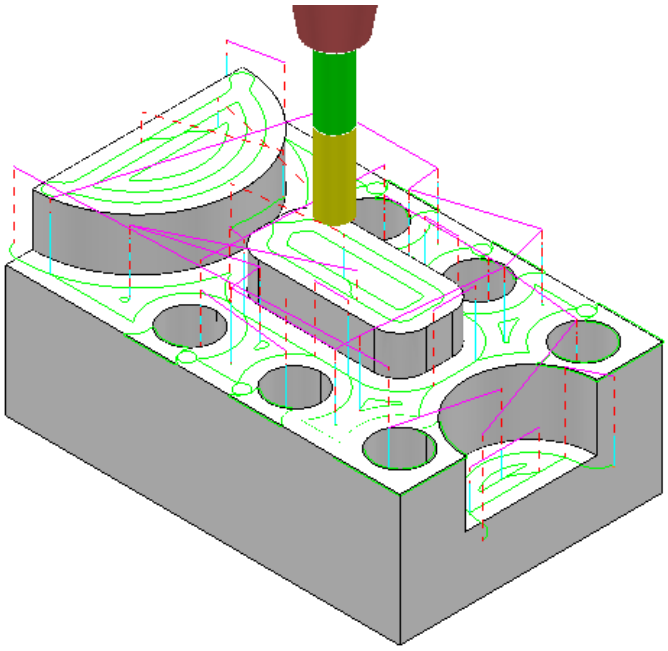
15 Select the **Flat machining** page and:

a Select a tick in the **Allow tool outside flat** box.



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

 Copyright © Delcam plc



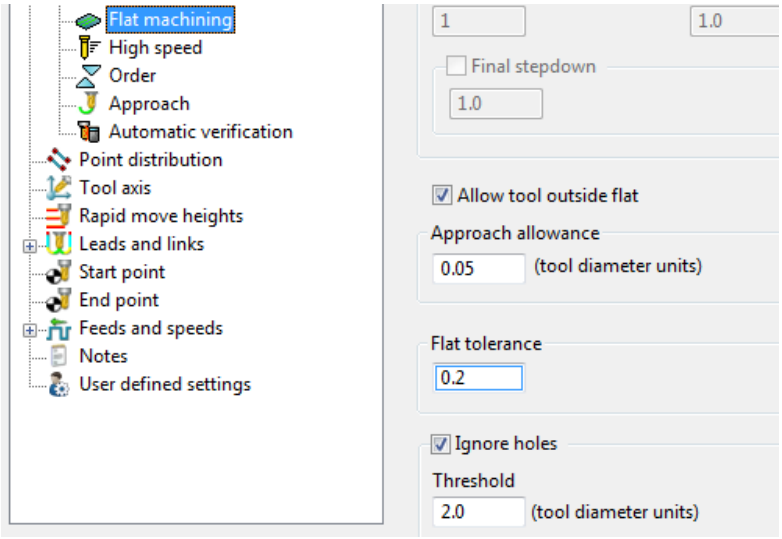
The **Area Clearance** now creates only **Flat** areas.

At the moment, the cutter profiles around the edge of the holes located on a flat area. If preferred, the strategy can be changed to **Ignore Holes** causing the tool to pass straight across them.

17 Right-click on the **Active** toolpath and select **Settings** to re-open the **Slice Area Clearance** dialog.

18 Click on to create a copy of the toolpath.

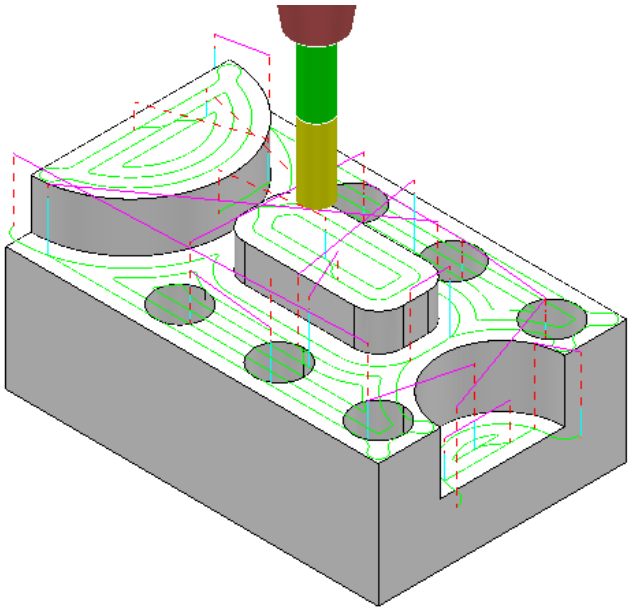
19 Rename the toolpath as **EM12-SliceAC-IgnoreHoles** and then select the **Flat machining** page.



20 Select a **Tick** in the **Ignore holes** box.

The **Threshold** (tool diameter units) defines the maximum size of holes to ignore. With the tool **EM12** selected and the **Threshold** set to 2, holes that are greater than **Dia 24mm** will not be ingored.

21 Select **Calculate** to process the toolpath and then **Cancel** the dialog.



The cutter now passes across the top of the holes without lifting, producing a smoother toolpath.

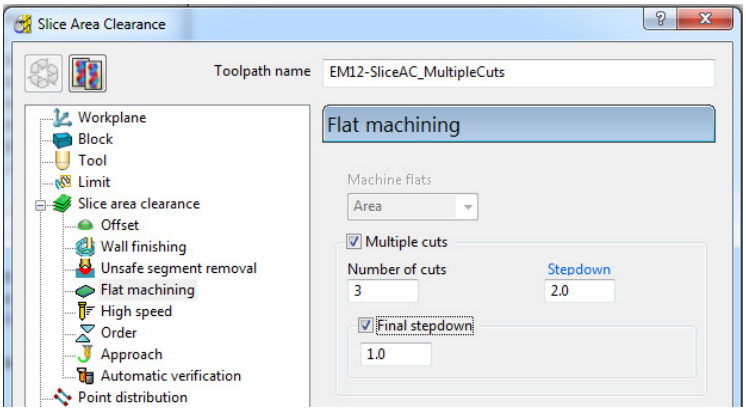
Ignore Holes *refers to the slice definition at the top of the holes. The flat base of any hole will still be machined.*

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

1 Right-click the **Active** toolpath and select **Settings** to re-open the **Slice Area Clearance** dialog.

2 Click on to create a copy of the toolpath.

3 Rename the toolpath as **EM12-SliceAC-MultipleCuts** and then select the **Flat** machining page.



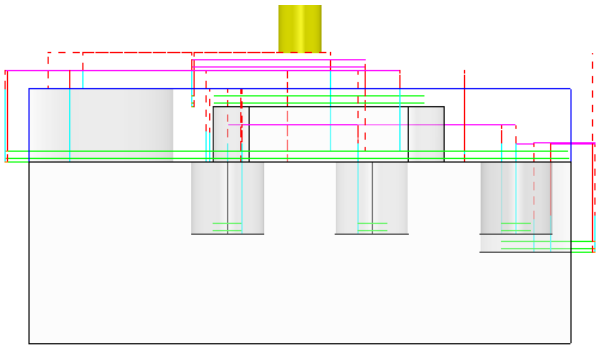
a Select the **Multiple cuts** option and set the Number of cuts as 3.

b Input the **Stepdown** as **2.0**

c Select the **Final stepdown** option and input **1.0**

 Copyright © Delcam plc

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



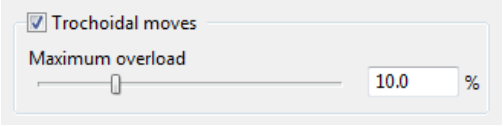
*The 'D' shaped **Flat** area at the very top level does not use **Multiple Cuts** as the top of the **Block** is the overall **Z limit** for the strategy.*

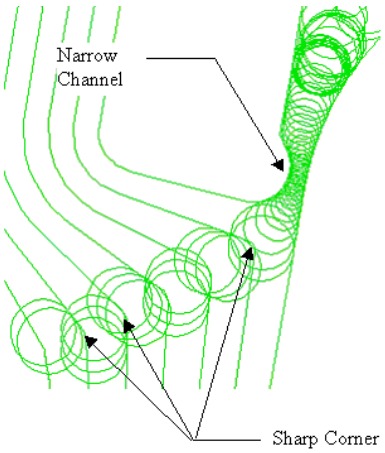
Further information on Area Clearance Machining

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

Restrict Tool Overload – Trochoidal moves

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

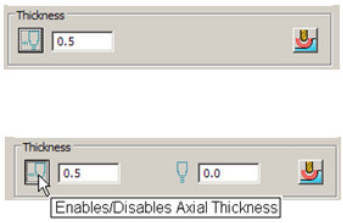


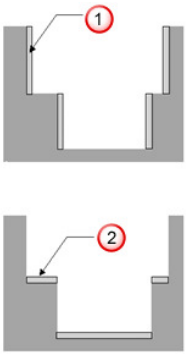


If the tool reaches a specified overload situation, **PowerMILL** automatically inputs a trochoidal path to eliminate full width cuts. This will occur in corners, narrow channels, slots, etc. The degree of movement is controlled with the slider as a percentage value in the form.

Thickness

Click the **Thickness** button in strategy dialogs to enter separate **Radial** and **Axial** thicknesses.





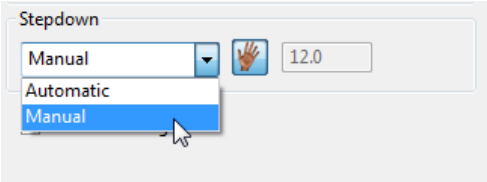
-

1 - Radial Thickness

2 - Axial Thickness

Z Heights – Manual entry

To manually enter Z heights in a strategy, select **Manual** from the **Stepdown** list.



Click to display the **Area Clearance Z Heights** dialog.

Defined by defines how Z heights are specified. You can define a Z height by:

- **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 top of the block and then steps down a defined height in Z. The final Z Height will be at the bottom of the block.

  If the **Constant Stepdown** option is selected, the distance between all Z height levels remain constant. To achieve this, the actual stepdown value will more than likely have to be automatically modified to be less than the nominal value.
- **Value** - creates a single **Z Height** at the defined value.
- **Flat** - Identifies flat areas of the model and creates a **Z height** (+ thickness) at these values. It can also be set to machine the specific flat areas instead of the whole of the slice.
- **Intermediate** - adds a specified number of **Z Heights** between the current ones.

Append Z Heights - **Z Heights** can be also be appended from an existing **Model Area Clearance** toolpath using the **Reference** section of the **Area Clearance Z Height** dialog.

Copyright © Delcam plc

Profiling

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

When

The **When** settings on a **Raster** page in an **Area Clearance** strategy dialog allow you to determine when the profiling pass occurs during machining.

None – No profiling pass is performed

Before – The profiling pass is performed before the Raster path.

During – The profiling is performed at suitable intervals during the Raster path.

After – The profiling pass is performed last.

Cut Direction

The **Cut Direction** settings control the direction of the tool along the toolpath:

Any – This instructs the cutter to machine in both directions.

Climb – This instructs the cutter to profile such that the material is to the right of the direction of travel (Climb milling).

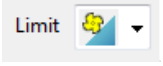
Conventional – This instructs the cutter to profile such that the material is to the left of the direction of travel (Conventional milling).

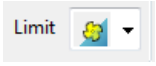
Wall Finishing

The **Wall Finishing** page in the **Model Area Clearance** strategy dialog allows you to make an additional, final profiling pass to further reduce tool wear.

Allow tool outside block

The **Allow tool outside block** setting is available on the **Limit** page of the **Model Area Clearance** dialog.

This option enables the first pass of an **Area Clearance** strategy to be performed to the specified **Stepover** 

rather than the full radius of the tool 

Flat Machining

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

Level - **Flat Machining** occurs across the whole area defined by the **Block**.

The nominal **Stepdown** value is locally reduced as required to accommodate each **Flat area** as part of an equally spaced group.

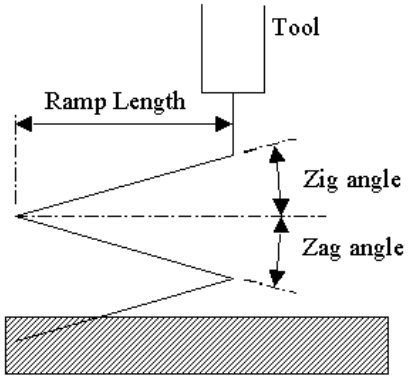
Area - **Flat Machining** is confined to the individual **Flat area**.

The nominal **Stepdown** value remains constant (independent of the **Flat areas**).

Ramping

The **Ramp** option is available on the **Leads and Links** dialog to use where it is impossible to approach from outside the **Block** at the full machining depth (for example, within a pocket).

- There are 3 different options for the type of ramp move - **Toolpath**, **Circle**, and **Line**. These can be used to lead on to the start of a tool track.
- The **Zig** angle is the angle of descent along the machining direction as the tool ramps into the material. If the length of the **Zig** angle is limited to a finite distance, a ramp move in the opposite direction (**Zag** angle) can also be applied.



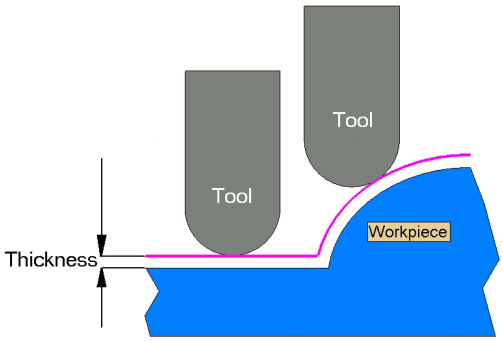
- The **Ramp Length** is defined as **Tool Diameter Units** (**TDU**). For example, A **Ramp Length** of **2 TDU** would equal **20 mm** for a **Dia 10 mm** tool.
- If **Approach outside allowance** is set on the **Flat machining** page of strategy dialog, it will take priority over **Ramping**.
- If the defined geometry for a **Ramp** move is such that it would cause a gouge, then the tool approaches using a default **Plunge** move.

Copyright © Delcam plc

4. Finishing Strategies

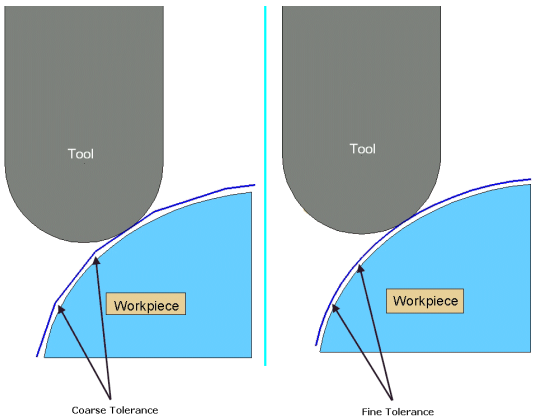
Finishing strategies machine the actual component form, and where applicable, follow on from the **Area Clearance** operation. Suitable values are required to control the accuracy and amount of excess material to be left on a component by a toolpath. The parameters used for this purpose are called **Thickness** and **Tolerance**.

Thickness is the amount of extra material specified to remain on the work-piece after machining. This can be applied generally as an offset from the model (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 **model** (Component Thickness).*

Tolerance controls the accuracy to which the cutter path follows the shape of the work-piece. For roughing, use a Coarse tolerance, and for finishing use a Fine tolerance.



*A positive **Thickness** value should always be greater than the **Tolerance** value.*

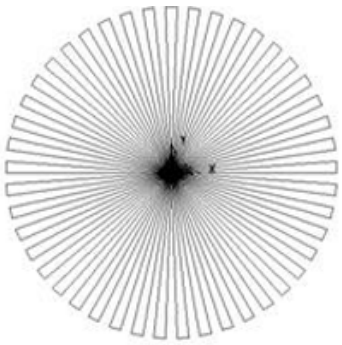
Raster, Radial, Spiral, and Pattern Finishing

These four types of strategy generate toolpaths by projecting a wireframe, pattern 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** dialog. Select Preview to view the resultant pattern. If you are satisfied with the preview, click Calculate to process the toolpath.

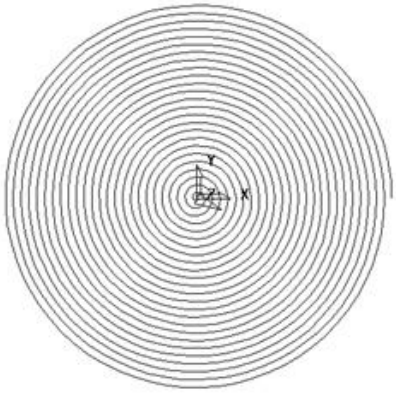
The **Pattern** strategy uses an existing, pre-defined **Pattern** or **Toolpath**, which is either projected down Z onto the model or used in situ to generate a new toolpath.

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

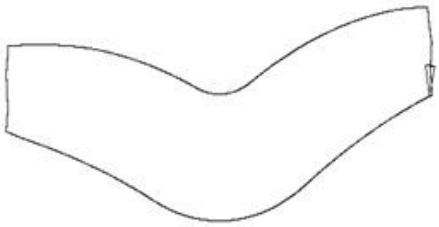
Raster



Radial



Spiral



User-defined

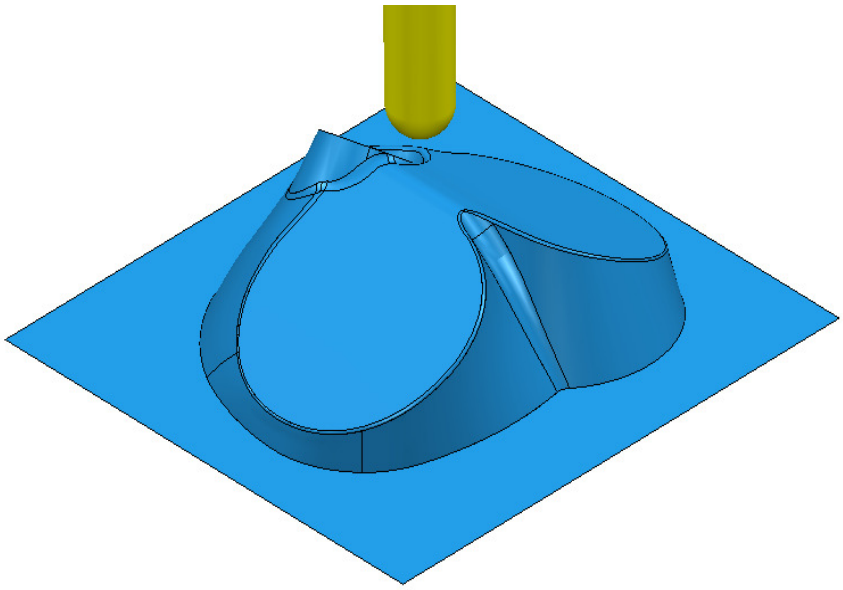
The **Raster finishing** strategy is used as an example in the next section.

Raster Finishing

1. From the **Main** pull down menus select **File - Delete All** and **Tools** - **Reset Forms**.
2. Then select **File** > **Open Project**:

   .....\PowerMILL_Data\Projects\chamber_start

Copyright © Delcam plc



The **Read-Only Project** contains a component **model** and a Dia 12 **Ball Nose** tool.

3 From the **Main** menu , select **File** >**Save Project As**:

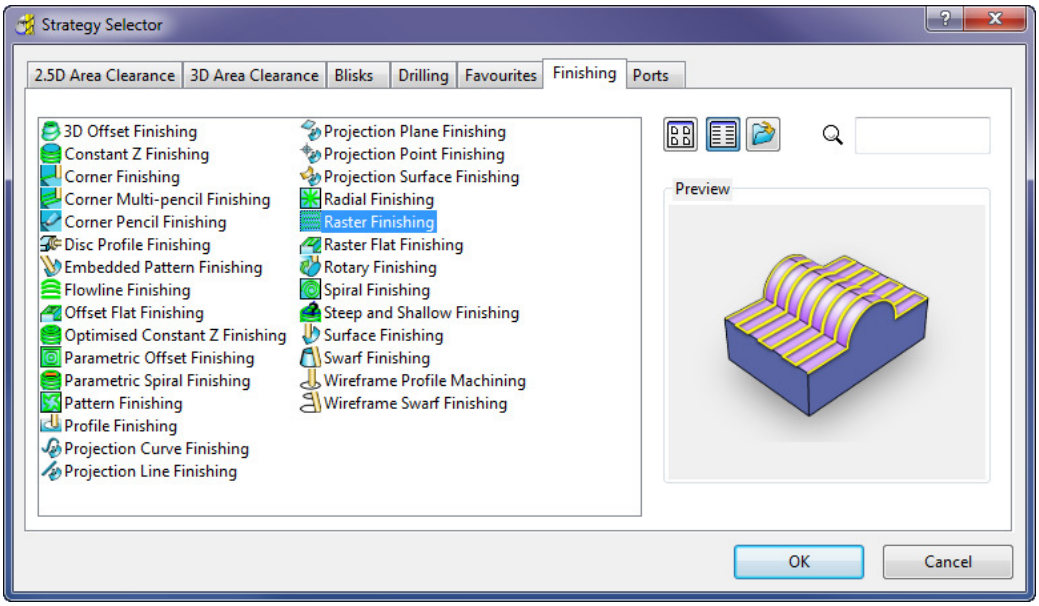
.....\COURSEWORK\PowerMILL-Projects\chamber

4 **Calculate** a **Block** using the default settings.

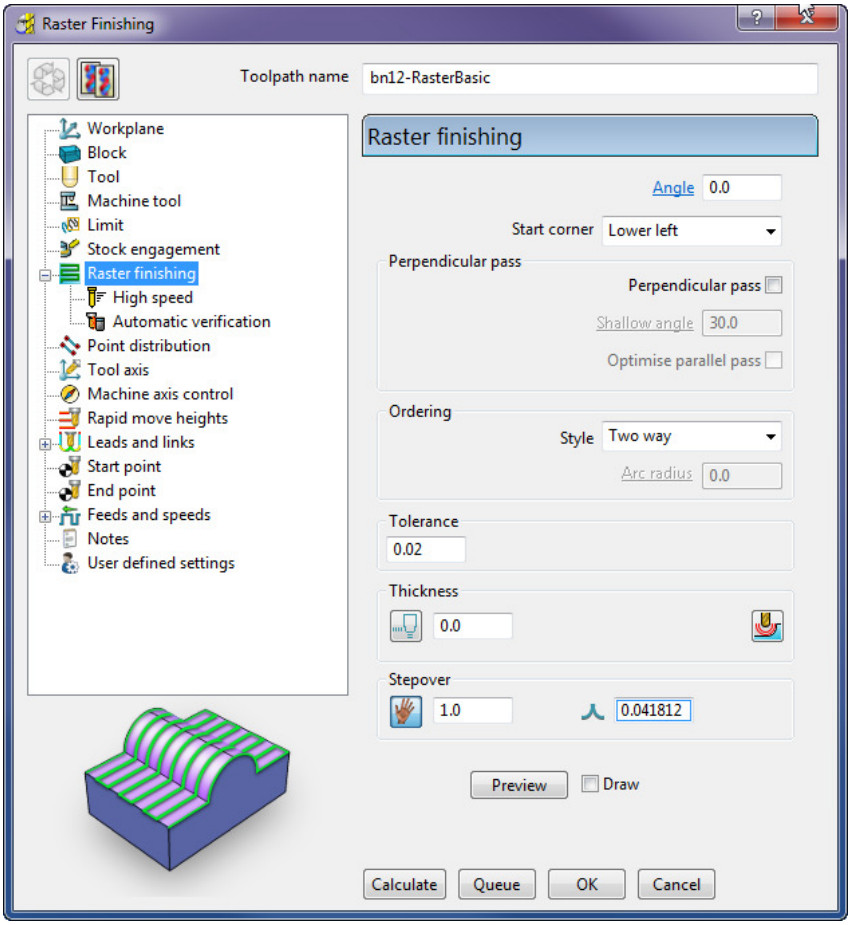
5 **Calculate** the **Rapid Move Heights** using the default settings to ensure link moves occur at a safe height above the model.

6 On the **Main** toolbar, click to open the **Toolpath Strategies** dialog.

7 On the **Strategy Selector**, click the **Finishing** tab and then the **Raster Finishing** strategy.



8 Click **OK** to display the **Raster Finishing** strategy dialog and enter the following settings and values.



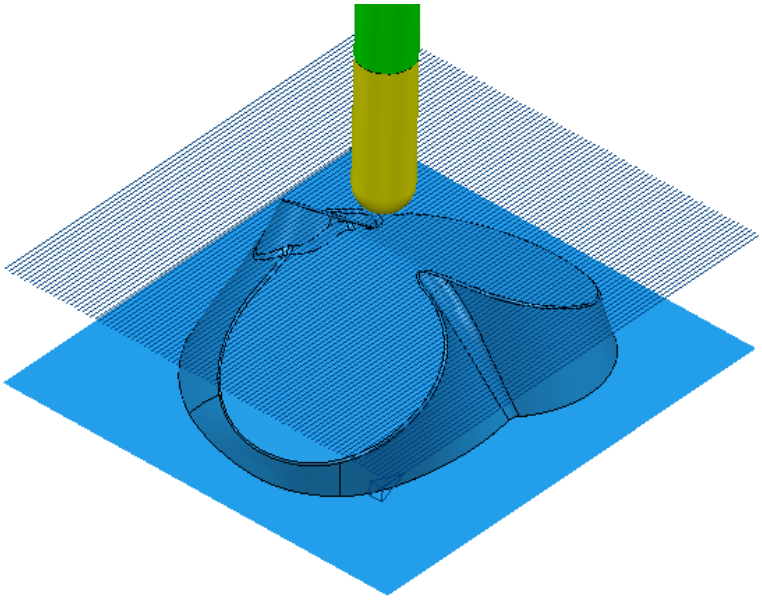
a Name the toolpath **bn12-RasterBasic**.

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

c Select **Ordering** - **Style** as **Two Way**.

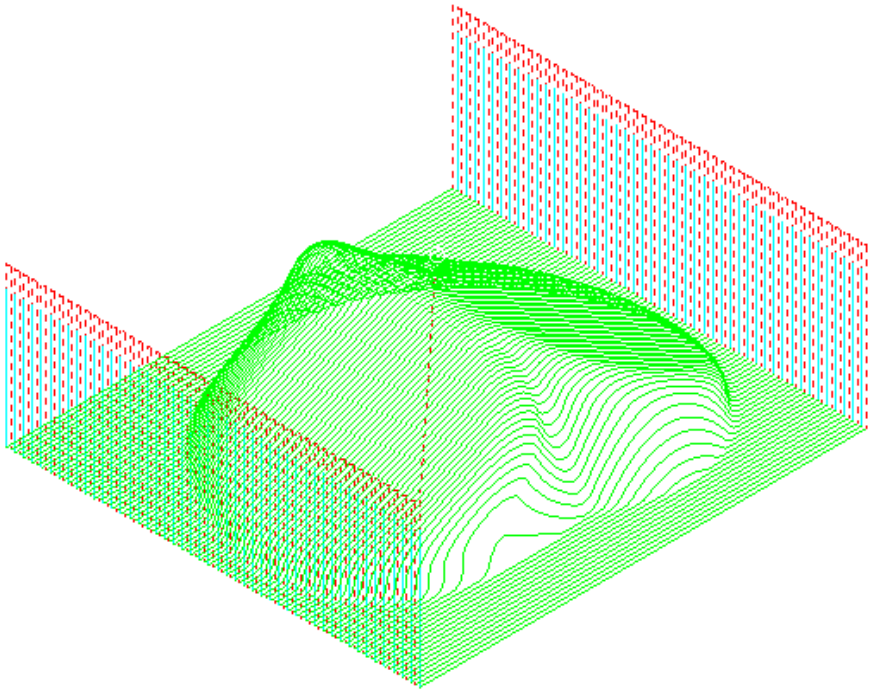
d Enter a **Stepover** of 1

e Click the **Preview** tab to display the pattern (as shown below).



Copyright © Delcam plc

9 Click **Calculate** to process the toolpath and then **Close** the dialog.



On closer inspection, the toolpath contains sharp corners along the vertical plane direction. These can be arc fitted within the strategy form.

In the **Raster Finishing**, **High speed** page the **Arc fit corners** option in the dialog *arc fits* these to a maximum radius of **0.2** TDU (Tool Diameter Units).

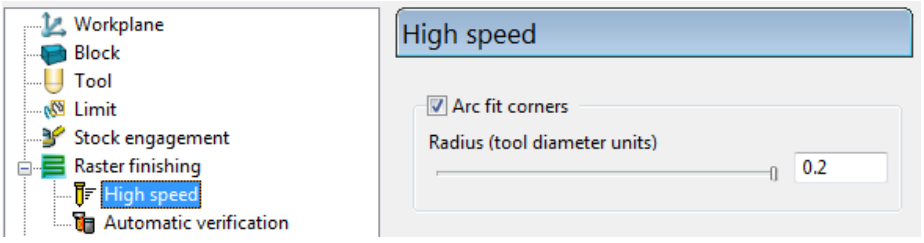
10 In **PowerMILL *explorer***, right-click the **bn12-RasterBasic** toolpath and select Settings.

11 Click to create a copy of the toolpath.

12 On the strategy dialog:

a Rename the new toolpath as **bn12-RasterArcfit**.

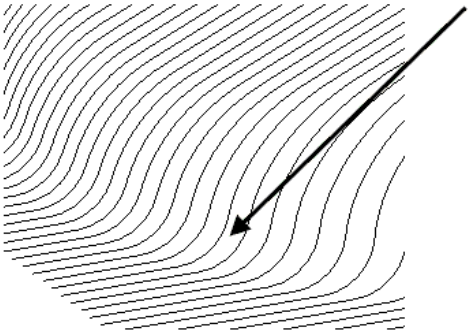
b Open the **High speed** page.



c Select the **Arc Fit** option and edit the **Arc Radius** to the maximum value **0.2**

d Click **Calculate** to process the toolpath and then **Close** the dialog.

13 Zoom into the toolpath to view the ***arc fitted*** tooltracks:

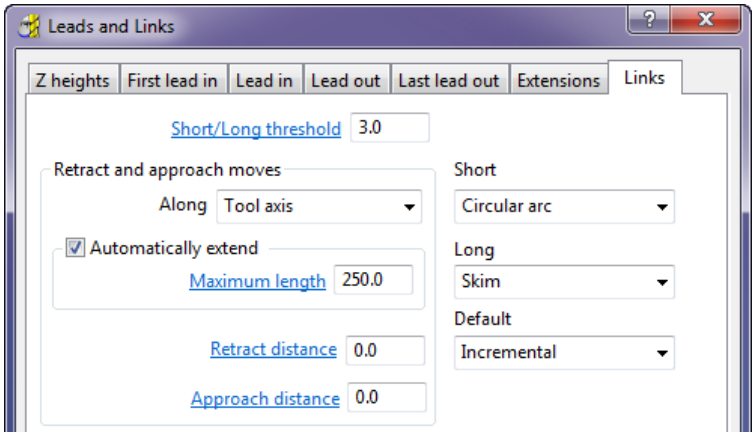


The value **0.2** (**TDU**), multiplied by the tool diameter (**12 mm**) produces a radius of **2.4 mm**.

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

1 From the **Main** toolbar, select **Leads and Links** .

2 Select the **Links** tab on the **Leads and Links** dialog.



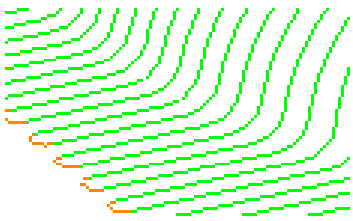
a Set the **Short**/**Long threshold** to **3.0**

b Set the **Short** links to **Circular Arc**.

c Set the **Long** links to **Skim** and **Default** links to **Incremental**.

*If the distance from the end of a tool track is within 3 mm from the start of the next, then the **Short** links option (**Circular Arc**) applies.*

3 **Apply** and then **Accept** the dialog.



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

The next stage is to simulate the latest, active toolpath, **bn12**-**RasterArcfit**.

Copyright © Delcam plc

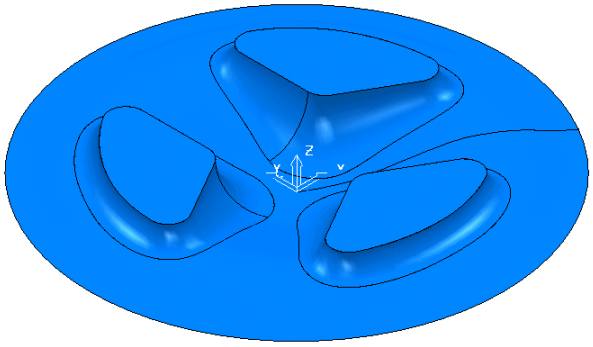
Pattern Finishing

This strategy requires a user-defined **Pattern** for projection onto the **model** as a toolpath. This option can be used for applications such as scribe lines, text, and custom strategies. A **Pattern** is created either from within **PowerMILL** or from imported wireframe data.

1 Select **File** - **Delete All** followed by **Tools** - **Reset Forms**.

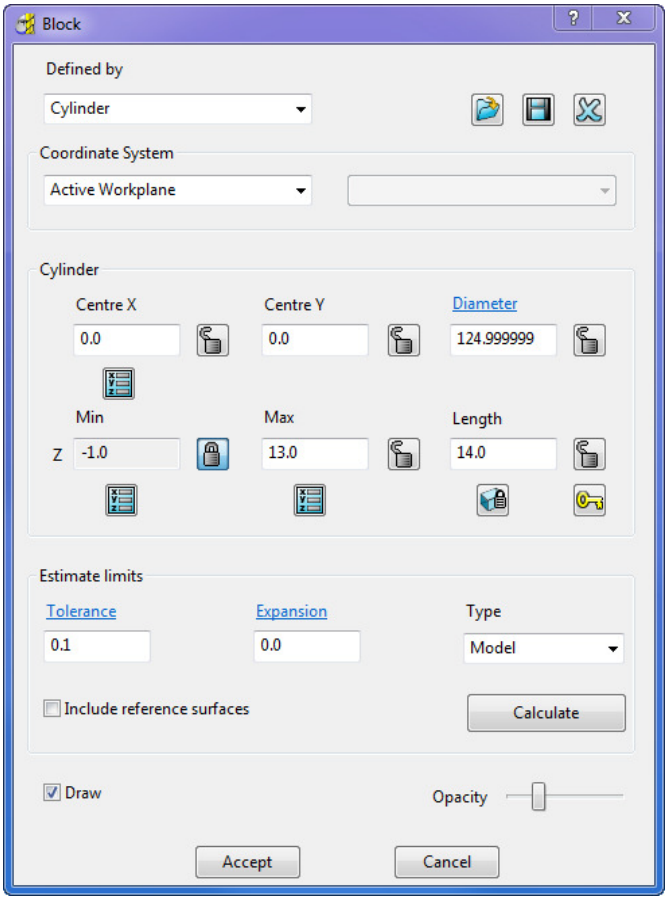
2 **Import** the **Model**:

.....\PowerMILL_Data\Models\swheel.dgk



Since the component is circular, a **Block *Defined by*** **Cylinder** will be used.

3 Click the **Block** button on the **Main** toolbar to open the **Block** dialog.

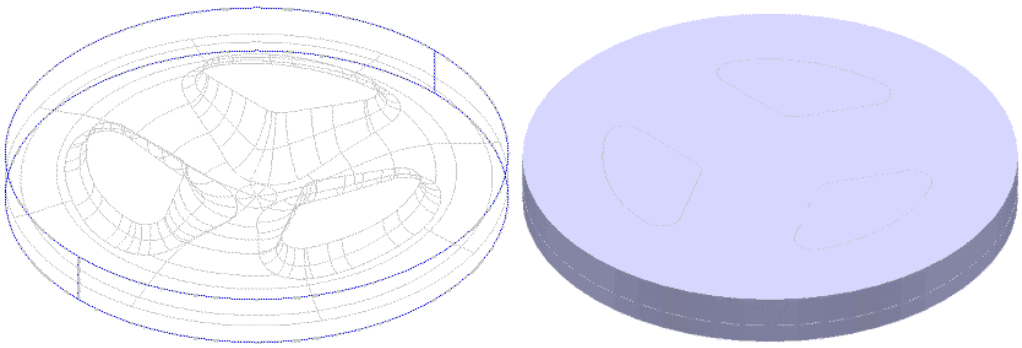


4 Calculate a **Block**, ***Defined by*** **Cylinder** using ***Type*** **Model**.

5 To provide more stock on the base, modify the **Min Z** value to **–1**, Lock the value, and click **Accept**.

Move the slider

to change the **Opacity** of the **Block**.



Before the **Pattern finishing** strategy is created, an **Area Clearance** strategy is required to remove the bulk of the material, using a **Thickness** of 0.5, followed by a **Semi-Finishing** strategy with a **Thickness** of 0.2.

6 Define a **Tip Radius** tool:

- a **Name** - **D10TR1**
- b **Diameter** 10
- c **Tip Radius** 1
- d **Tool Number** 1

7 Define a **Ball Nose** tool:

- a **Name** - **BN6**
- b **Diameter** 6
- c **Tool Number** 2

8 **Activate** the **D10TR1** Tip Radius tool.

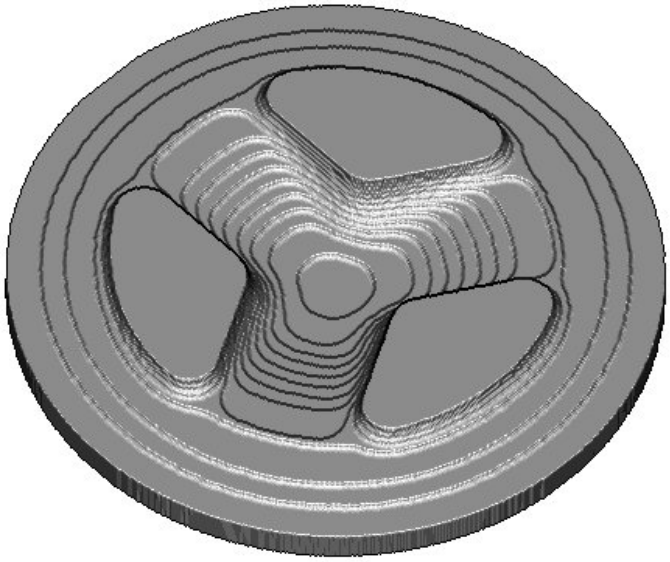
9 **Calculate** the **Rapid Move Heights** .

10 **Calculate** a **Model Area Clearance** toolpath using the following parameters:

- a **Name** – **D10TR1-RoughOP1**
- b **Style** - **Offset all**
- c **Cut Direction** – **Climb** (Both ***Profile*** and ***Area***).
- d **Tolerance** 0.1
- e **Thickness** 0.5
- f **Stepover** 5.0
- g **Stepdown** 1.0

Copyright © Delcam plc

11 **Simulate** the toolpath in **ViewMILL**:



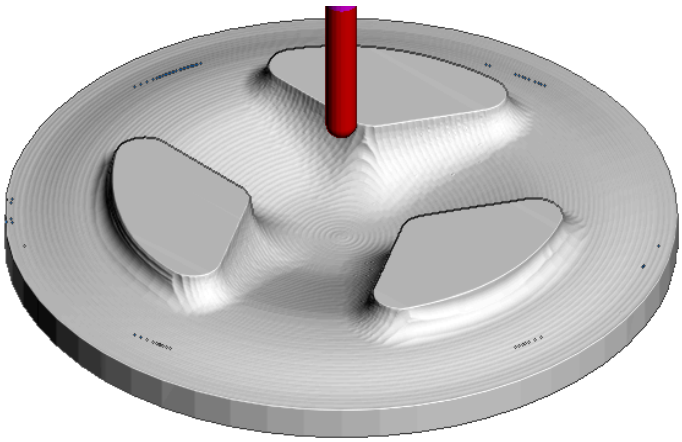
12 Click and return the **ViewMILL** display to **No Image**.

13 In **PowerMILL *explorer***, activate the BN6 Ball Nose tool.

14 Calculate a **Spiral Finishing** toolpath using the following parameters:

- a **Name** - **BN6-SemiFinishOP1**
- b **Centre Point** - **X** 0, **Y** 0
- c **Radius** - **Start** 62.0, **End** 0.0
- d **Direction** - **Clockwise**
- e **Tolerance** 0.1
- f **Thickness** 0.2
- g **Stepover** 1.0

15 **Simulate** the new toolpath in **ViewMILL**:



16 Click and return the **ViewMILL** display to **No Image**.

17 In the **PowerMILL *explorer***, right-click the **SemiFinishOP1** toolpath and select **Settings**.

18 Click and **create a copy of the toolpath**.

19 Enter:-

a **Name - BN6-FinishOP1**

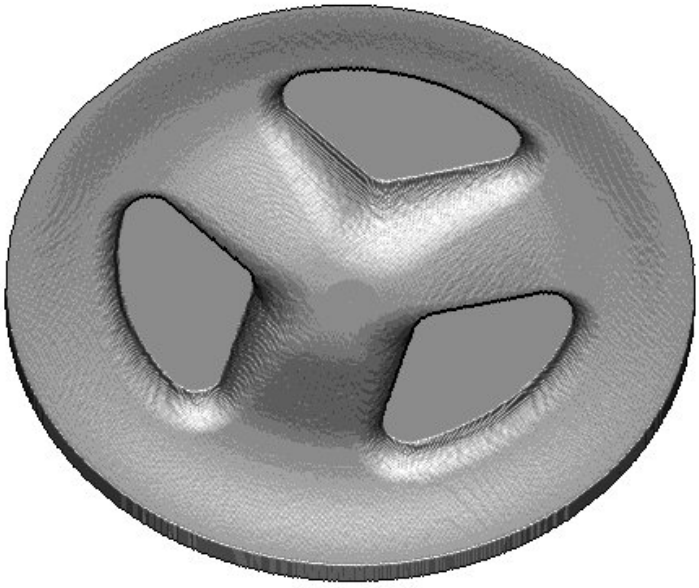
b **Tolerance - 0.01**

c **Thickness - 0.0**

d **Stepover - 0.3**

20 Click **Calculate** and when processed **Close** the dialog.

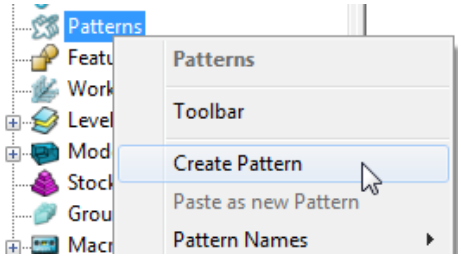
21 **Simulate** the final finishing toolpath in **ViewMILL**:



Next, some text will be engraved into the above finished, wheel form.

1 Click to return the **ViewMILL** display to **No Image**.

2 In the **PowerMILL *explorer***, right-click **Patterns** and select **Create Pattern**.

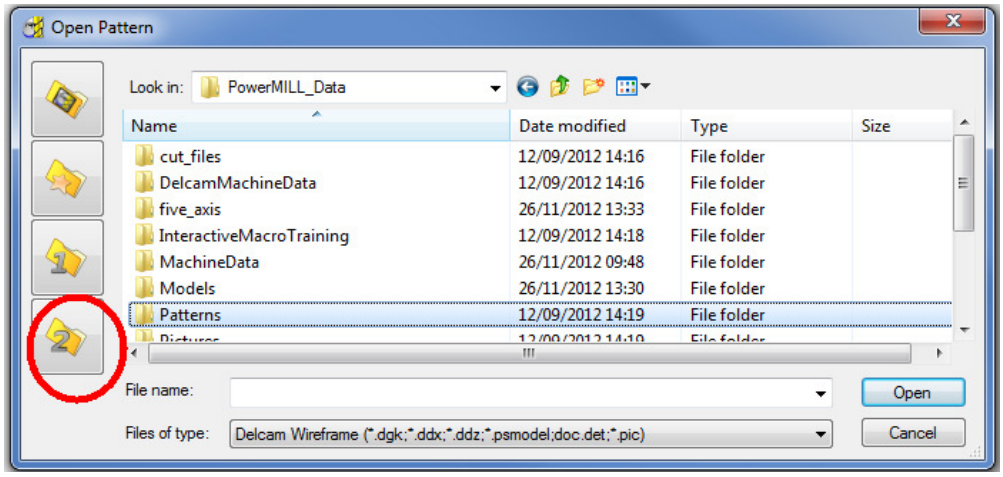


A new, empty **Pattern** is created.

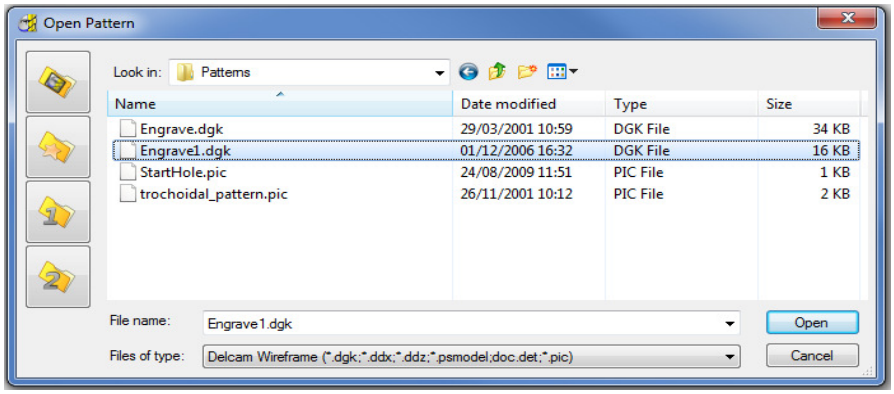
3 Right mouse click on the new empty **Pattern** and from the local menu select **Insert** - **File** to access the Open Pattern dialog.

4 Click ***button 2*** (Shown inside red border below) on the **Open Pattern** dialog:

 Copyright © Delcam plc

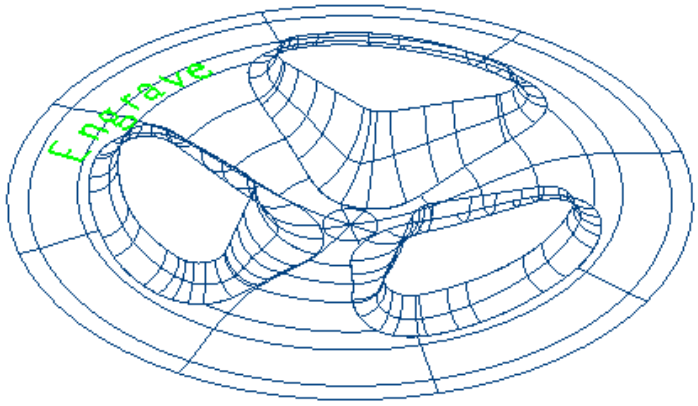


5 Double click the **Patterns** folder and then select the **Engrave1.dgk** file.



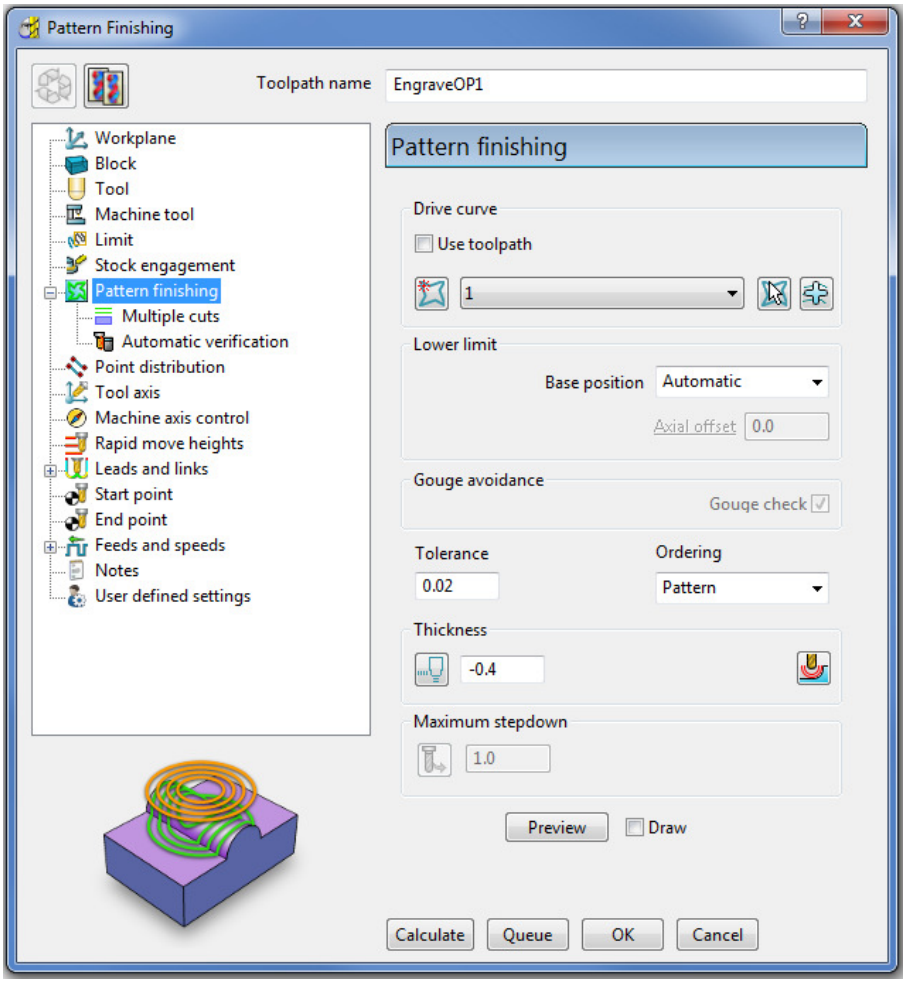
6 Click **Open**.

The imported **Pattern** is made up of 2D geometry and is positioned at **Z** 0. This 2D geometry will be projected down Z onto the 3D model to create a single **toolpath** with a negative **Thickness** value.



7 Define a **Ball Nose** tool of diameter 1, named **BN1**, with **Tool Number 3**.

8 Create a **Pattern Finishing** toolpath.



a Enter the ***Toolpath name*** **BN1-EngraveOp1**

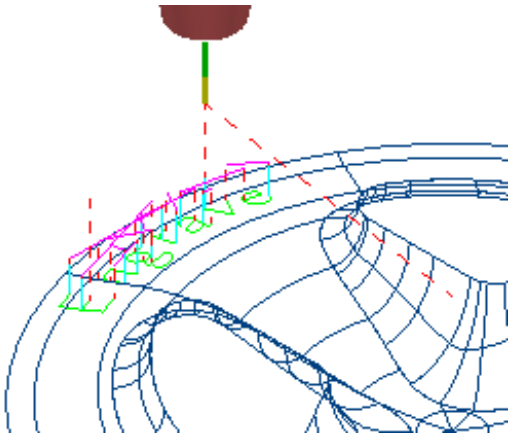
b Input ***Pattern*** 1 (if not already selected).

c With the ***Base position*** set to either **Automatic** or **Vertical**, the **Pattern** form will be projected down **Z** onto the **model**.

d Enter a ***Tolerance*** 0.02 and ***Thickness*** -0.4

9 Select **Calculate** to process the toolpath then **Close** the dialog.

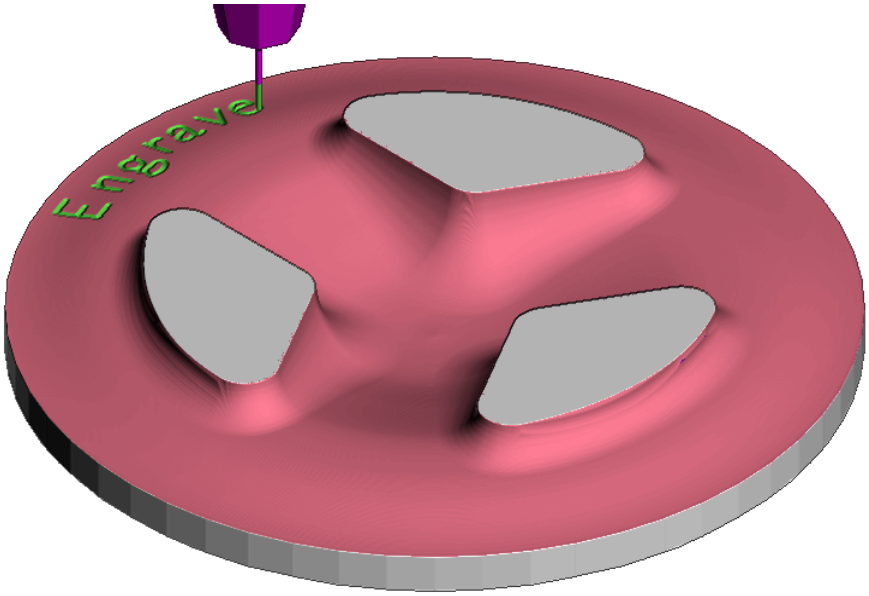
The **Pattern** is machined to an offset depth of **-0.4** into the finished **model** form.



 Copyright © Delcam plc

*It is not possible to use a –ve thickness value that is greater than the Tool Tip Radius. There is, however, an alternative way to achieve this (**Chapter 11 Patterns, page 20 - Deep Pattern Machining**).*

10 **Simulate** the engraving toolpath in **ViewMILL**.

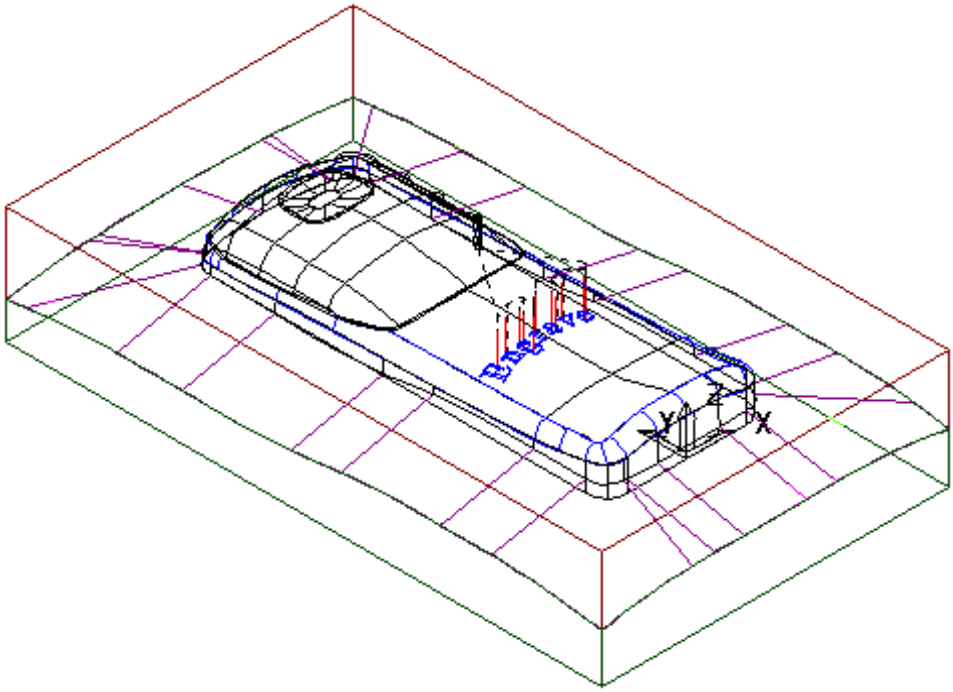


11 Select **File** - **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\PatternMachining

Pattern Exercise

In this exercise you are to use same imported **Pattern** (Engrave1) to machine text into a telephone handset **Model**.



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

.....\COURSEWORK\PowerMILL-Projects\PhoneEngrave

2 **Delete** the 2 toolpaths, **BN6-SemiFinishOP1** and **BN6-FinishOP1**.

3 Right-click **Models** on the **PowerMILL *explorer*** and select **Delete All**.

4 **Import** the **Model**:- **.....\PowerMILL_Data\Models\phone.dgk**

5 **Recycle** the **Model Area Clearance** toolpath **D10TR1-RoughOP1** using a **Block *Defined by* Box**, and with the **Rapid Move Heights** re-calculated.

6 Create a new **Raster finishing** toolpath using the following parameters:

Name - **BN6-SemiFinishOP1**

Tolerance **0.1**

Thickness **0.2**

Stepover **1.0**

7 Create a new **Raster finishing** toolpath using the following parameters:

Name - **BN6-FinishOP1**

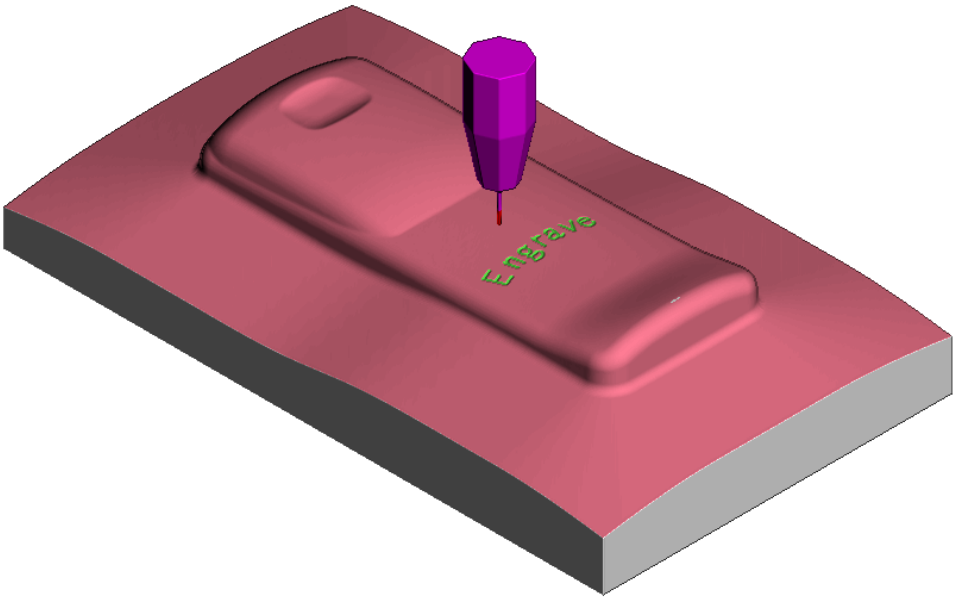
Tolerance **0.01**

Thickness **0**

Stepover **0.3**

8 **Recycle** the **Pattern Finishing** toolpath **EngraveOp1** using a **Block *Defined by* Box**, and with the **Rapid Move Heights** re-calculated.

9 Perform a full **ViewMILL** simulation on all 4 toolpaths.



10 Select **File** - **Save Project**

 Copyright © Delcam plc

3D Offset and Constant Z Finishing

In this section, **3D Offset** and **Constant Z** finishing strategies are applied to a model that consists of a combination of flat and steep areas plus a pocket with vertical walls. The two strategies are limited to where they are most compatible.

A **Boundary** is 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 Finishing creates the tool stepover relative to the 3D **Model** surface providing consistency over both flat areas and steep sidewalls.

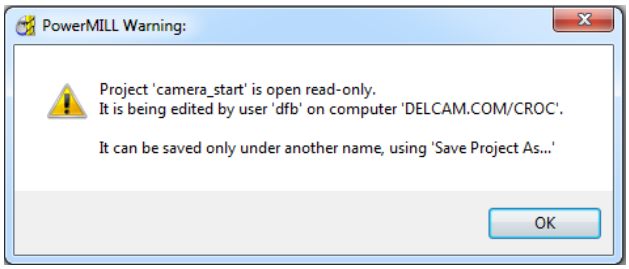
*In this exercise, it is not recommended to apply this toolpath to the complete model without using **Boundaries**. Although this toolpath maintains a constant **Stepover**, there is nothing to prevent the tool plunging with a full width cut into the deep pocket area.*

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

2 Select **File** - **Open Project** and in the dialog browse to:

.....\PowerMILL_Data\Projects\camera_start

The **Project** is locked and subsequent changes cannot be saved unless the included **lock** file is deleted.

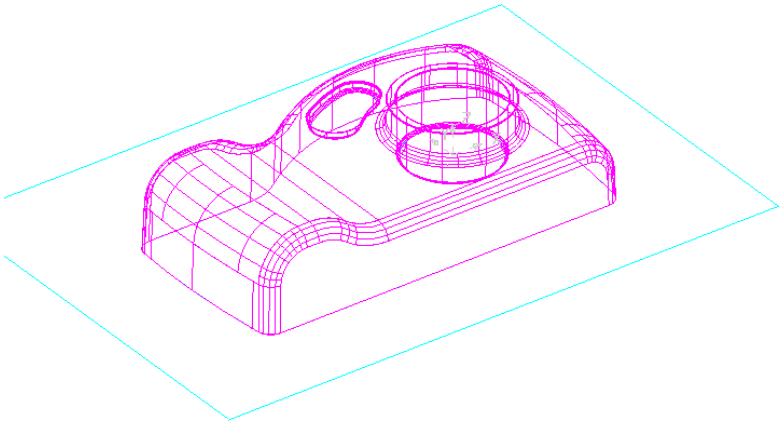


3 Select **OK** to continue loading the **Project** into the **PowerMILL** session.

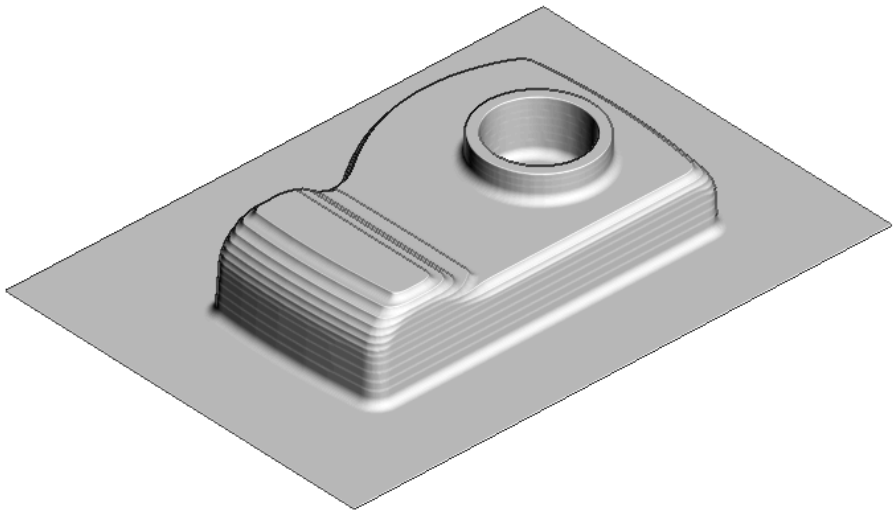
4 From the **Main** pull down menus, select **File** > **Save Project As**:

.....\COURSEWORK\PowerMILL_Projects\Camera_Example

The **Project** already contains a **Tip Radiused** tool and an **Offset Area Clearance** toolpath along with the associated settings.



1 **Activate** the existing **Model Area Clearance**, toolpath, and perform a **ViewMILL** simulation.



Use of Shallow Boundary to define specific machining areas

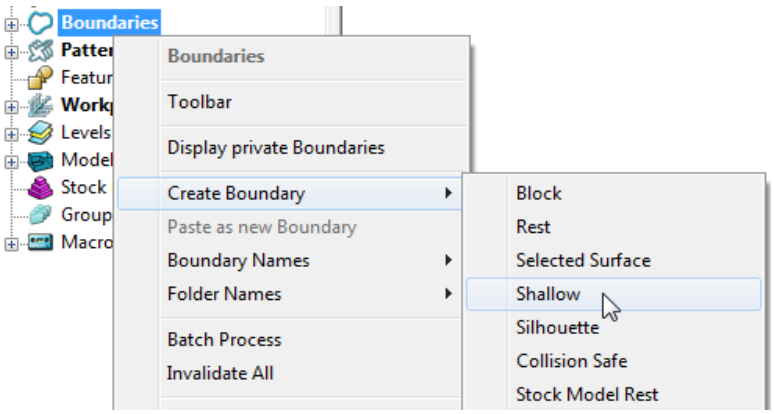
Specific finishing strategies are more effective on either ***Steep*** or ***Shallow*** parts of a 3D component **model**. For example, **Constant Z** finishing is most effective on steep sidewalls while **Raster** Finishing is most effective in shallow areas.

A **Shallow Boundary** creates segments around areas of 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.

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

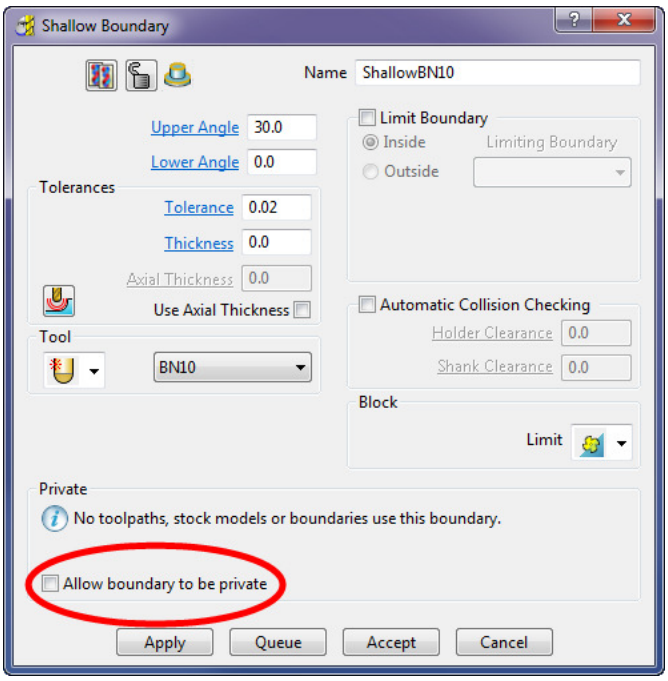
In the below example, a **Shallow Boundary** is used to discriminate areas most suited to the individual machining strategies. This type of **Boundary** is calculated, taking the **active Tool** into consideration.

1. Define a **Ball Nose** tool of **Diameter 10** called **BN10**.
2. In the **PowerMILL *explorer***, right-click **Boundaries** and select **Create Boundary** > **Shallow**.



 Copyright © Delcam plc

3 In the displayed **Shallow Boundary** dialog:



a Enter the **Name** as **ShallowBN10**

b Enter ***Upper Angle*** 30 and ***Lower Angle*** 0

c Enter ***Tolerance*** as 0.02

d Enter ***Thickness*** as 0

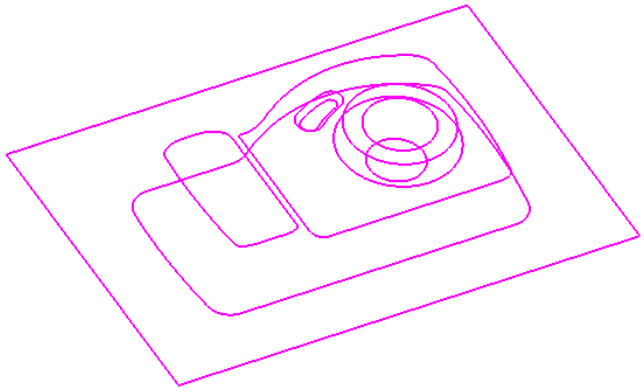
e Ensure the correct tool **BN10** is **Active**.

f Un-tick the box, ***Allow boundary to be private***.

g **Apply** and **Accept** the dialog.

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

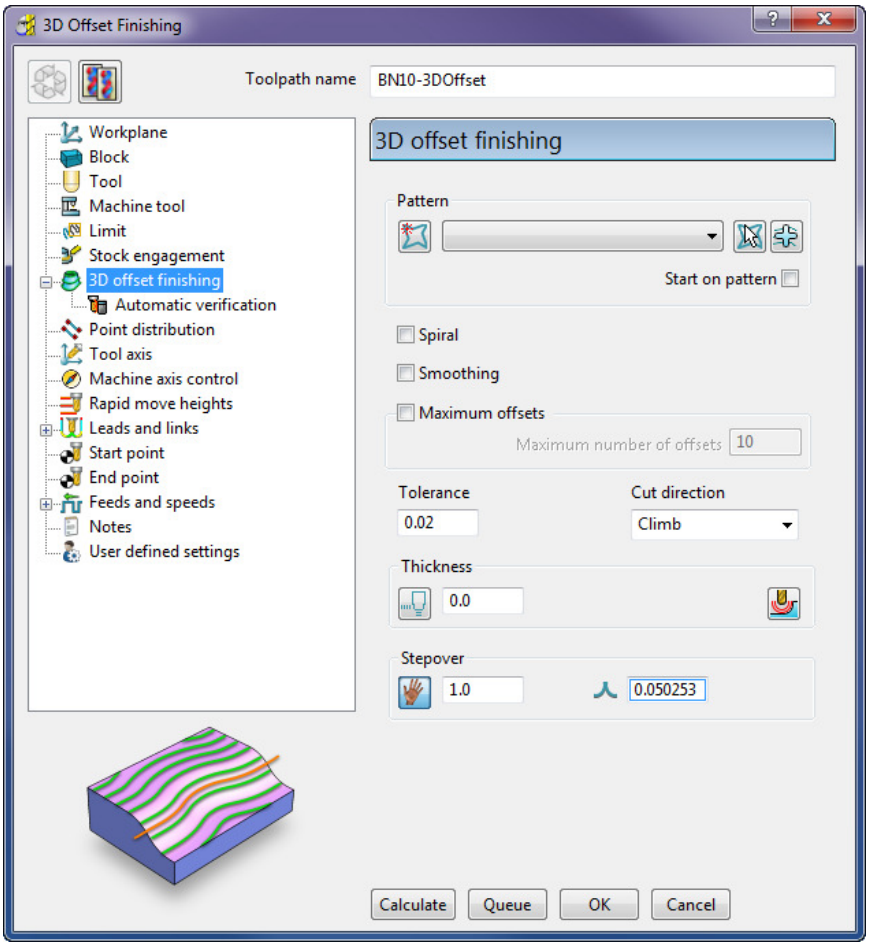
With the **model**, **Tool**, and **toolpaths** undrawn, the **Boundary** can be viewed as shown:



This **boundary** is made up of numerous segments; each one dividing the model into **steep** and **shallow** areas. Any of these segments can be selected and individually deleted at any time, except when it is assigned to a toolpath and locked.

1 From the **Main** toolbar, click the **Toolpath Strategies** button.

2 From the **Finishing** tab, select **3D Offset Finishing** strategy and click **OK**.



3 In the **3D Offset** Finishing strategy dialog:

a Enter ***Toolpath name*** as **BN10-3DOffset**

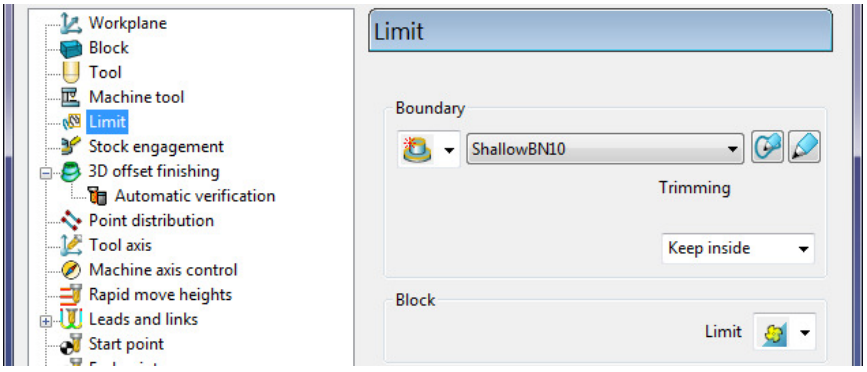
b Set the ***Cut Direction*** to **Climb**

c Enter a ***Tolerance*** of **0.02**

d Enter a ***Thickness*** of **0**

e Enter a ***Stepover*** of **1.0**

4 On the **3D Offset** finishing dialog, select the **Limit** page

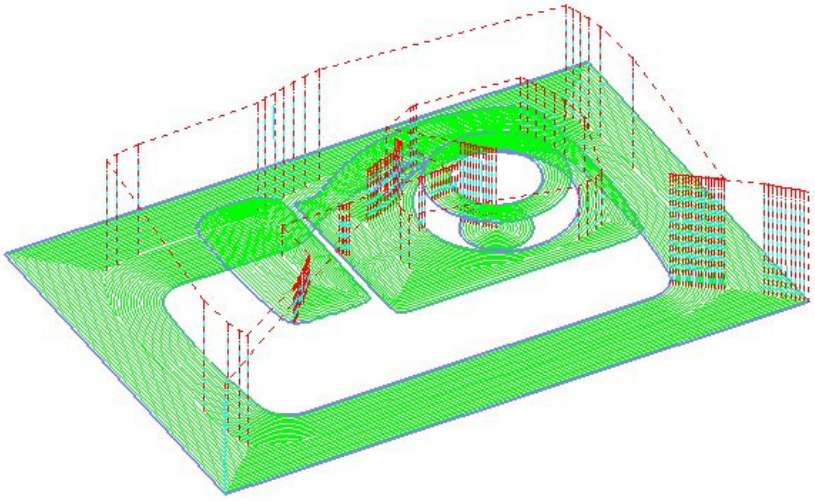


5 Select the **Boundary - ShallowBN10**.

 Copyright © Delcam plc

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

The **Toolpath** follows the contours of the **Boundary** segments and is produced only within the ***shallow*** areas of the **model** (***Inside*** the **Boundary**).



7 Click the **Leads and Links** button from the top of the screen.

8 Select the **Links** tab and:

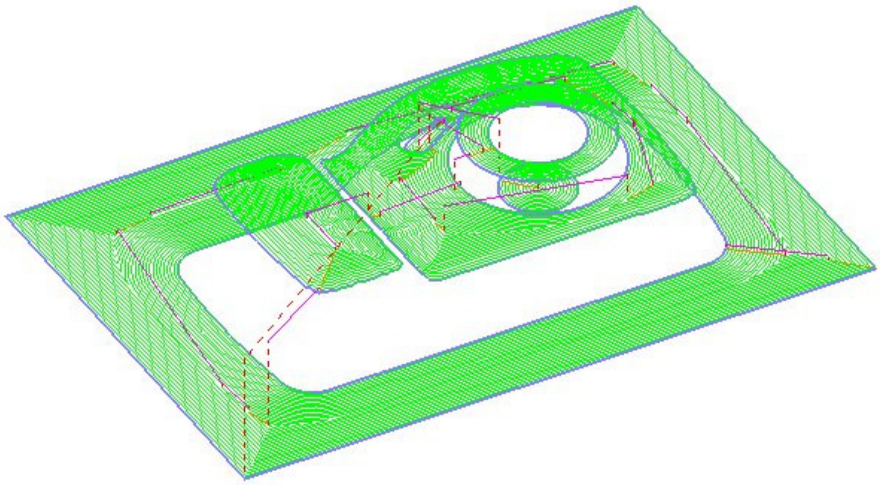
a In **Lead in**, Un-tick **Add leads to short links**, change the **1st Choice** to **Vertical Arc**, **Angle** 90 and **Radius** 5.

b Click the **Lead Out the same as lead in** button.

a In **Links** input 4 in Short/Long threshold, change the **Short** to **On Surface**, **Long** to **Skim**, and default to **Incremental**.

9 **Apply** and **Accept** the dialog.

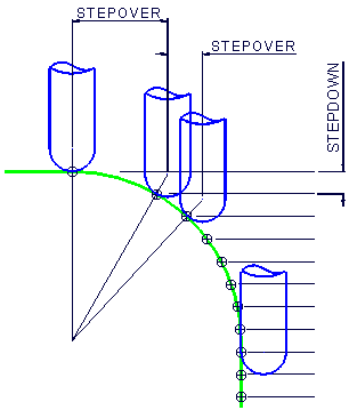
The **toolpath** now is more efficient with the rapid moves at ***skim height*** and the ***short links*** being forced onto the surface.



10 **Simulate** the **3D Offset Finishing** toolpath in **ViewMILL**.

Constant Z Finishing

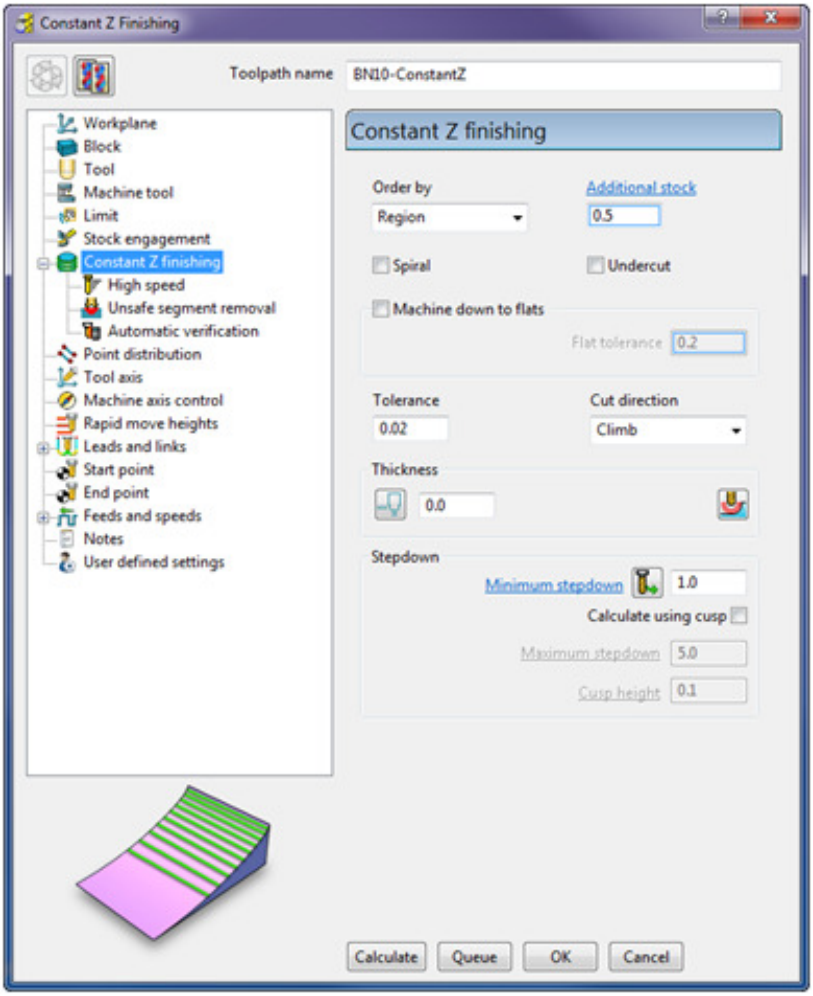
Constant Z Finishing projects each tool track horizontally onto the component at fixed heights defined by the **Stepdown**.



As the component surface becomes shallow, the actual tool stepover increases until it becomes non-existent on flat areas.

1 Click the **Strategy Selector** button on the **Main** toolbar.

2 From the **Finishing** tab, select the **Constant Z Finishing** strategy and click **OK**.



 Copyright © Delcam plc

3 In the **Constant Z Finishing** dialog:

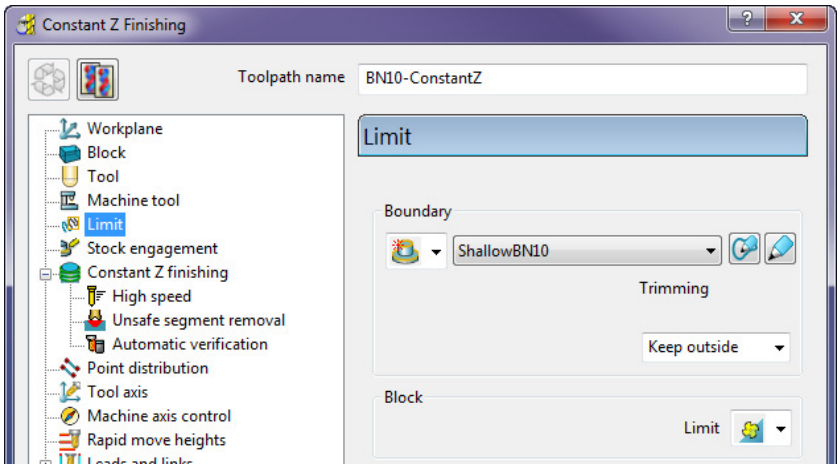
a Enter a **Toolpath name** - **BN10-ConstantZ**

b Enter a **Tolerance** of **0.02**

c Set the **Cut Direction** to **Climb**.

d Enter a **Stepdown** of **1**

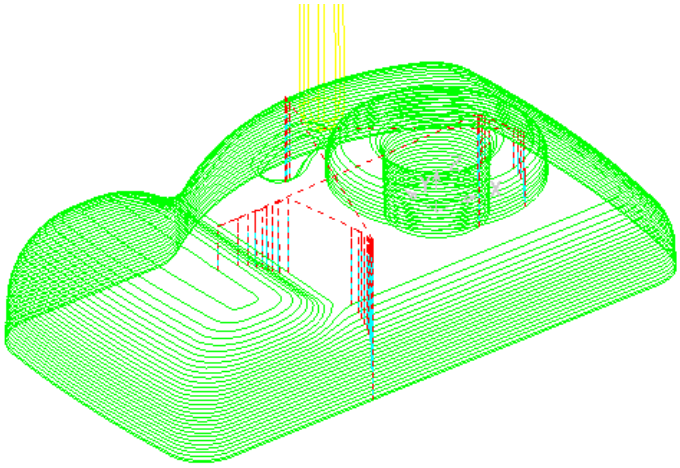
4 On **Constant Z finishing** dialog, select the **Limit** page and set **Trimming** to **Keep outside**.



5 Click **Calculate** to process the toolpath and then **Close** the dialog.

By using the **Boundary - Trimming** option as **Keep outside**, the toolpath is limited to the steep areas of the **model**.

*If a **Boundary** had not been used, parts of the toolpath on the shallow areas would have an excessive **Stepover** as shown below:*



1 Click the **Leads and Links** button on the **Main** toolbar.

2 Select the **Lead In** tab and:

e In **Lead in**, change the **1st Choice** to **Horizontal Arc**, **Angle** **90** and **Radius** **5**

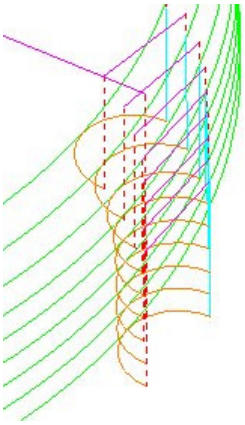
f Click the **Lead Out the same as lead in** button.

3 **Apply** and **Accept** the dialog.

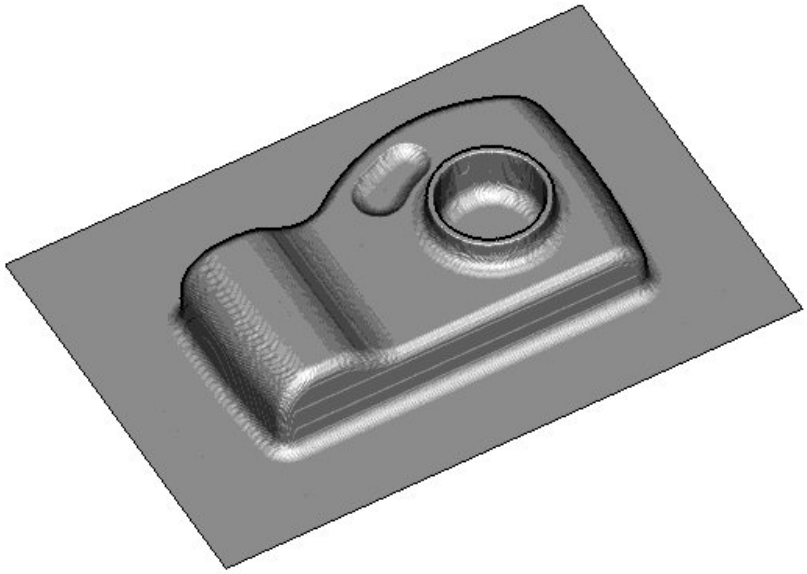
The tool now leads in and out of the toolpath with a horizontal arc.

If the tool lifts, it will only lift by the skim distance taken from the first page of the **Leads and Links** dialog (Z Heights tab).

The ***rapid feed*** moves at ***skim height*** are purple in colour while the ***plunge feed*** moves are light blue.



4 **Simulate** the toolpath in **ViewMILL**.



5 Select **File** - **Save Project**:

To update:- **.....\COURSEWORK\PowerMILL-Projects\Camera_Example**

*Note: The above saved **Project** will be re-opened later so a copy will now be saved for the next example!*

6 Select **File** - **Save Project**:

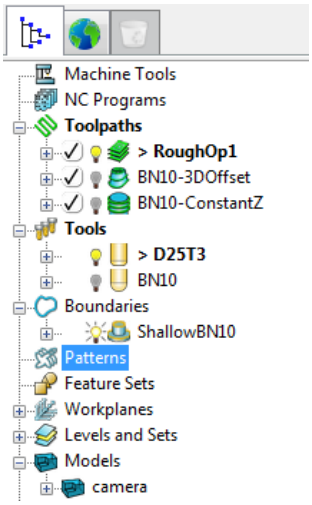
.....\COURSEWORK\PowerMILL-Projects\New-Template

Copyright © Delcam plc

Saving a Project as a re-useable Template

If the ***Settings*** and ***Strategies*** used in a particular **Project** could be regularly used over a range of components the process can be saved as a re-useable **Template**.

7 Look at the contents of the current **Project** in the **PowerMILL *explorer***.



Entities such as **NC Programs**, un-processed **Toolpaths**, **Tools, Boundaries** could be re-used on a range of different jobs, hence would be ideal for storing in a **Template.**

Entities such as **Workplanes**, **Models** would be unique to different components hence should not be stored in a **Template**.

8 In the **PowerMILL *explorer***, right mouse click on **Models** and from the local menu select **Delete All**.

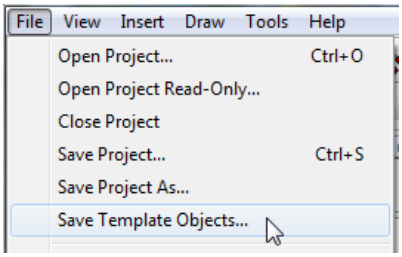
9 In the **PowerMILL *explorer***, right mouse click on **Toolpaths** and from the local menu select **Invalidate All**.

10 In the **PowerMILL *explorer***, right mouse click on **Levels** and from the local menu select **Delete Empty Levels and Sets**.

11 In the **PowerMILL *explorer***, right mouse click on **Workplanes** and from the local menu select **Delete All**.

12 In the **PowerMILL *explorer***, right mouse click on the **Boundary *ShallowBN10*** and from the local menu select **Edit - Make Invalid**.

13 From the **main** pull down menus select, **File** – **Save Template Objects**.



14 In the browser, **Save** the **Template** as:-

.....\COURSEWORK\PowerMILL_Projects\MasterProject-SteepShallow

The required components of the **Project** are stored in a **Template**.

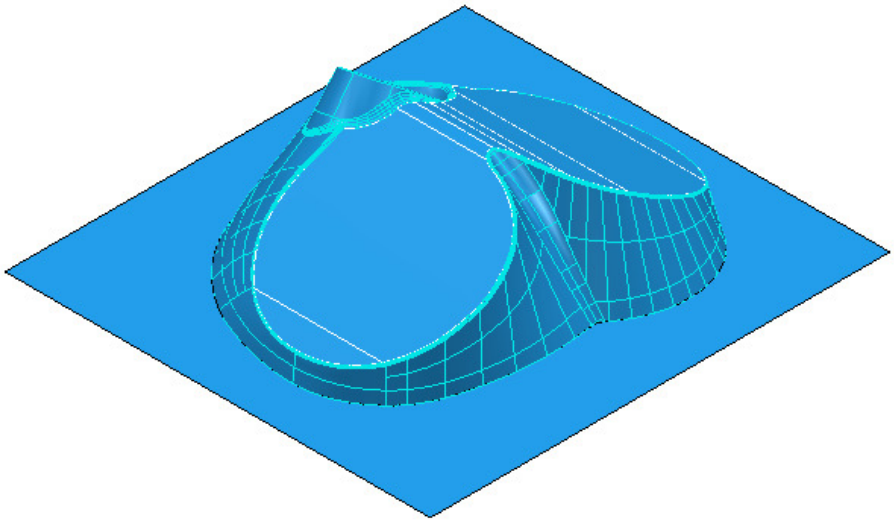
15 Select **File** - **Save Project** to update:

.....\COURSEWORK\PowerMILL-Projects\New-Template

Exercise – Apply the Template to a different Model

1 From the **Main** menu, select **File** – **Delete All** and **Tools** – **Reset Forms**.

2 **Import** the **Model**:- **.....\PowerMILL_Data\Models\Chamber.dgk**



3 From the **Main** menu, select **Insert** – **Template Object**.

4 In the browser, **Open** the **Template**:-

.....\COURSEWORK\PowerMILL_Projects\MasterProject-SteepShallow.ptf

5 Create a **Block** around the new **model** using the default settings.

6 **Calculate** the **Rapid Move Heights**.

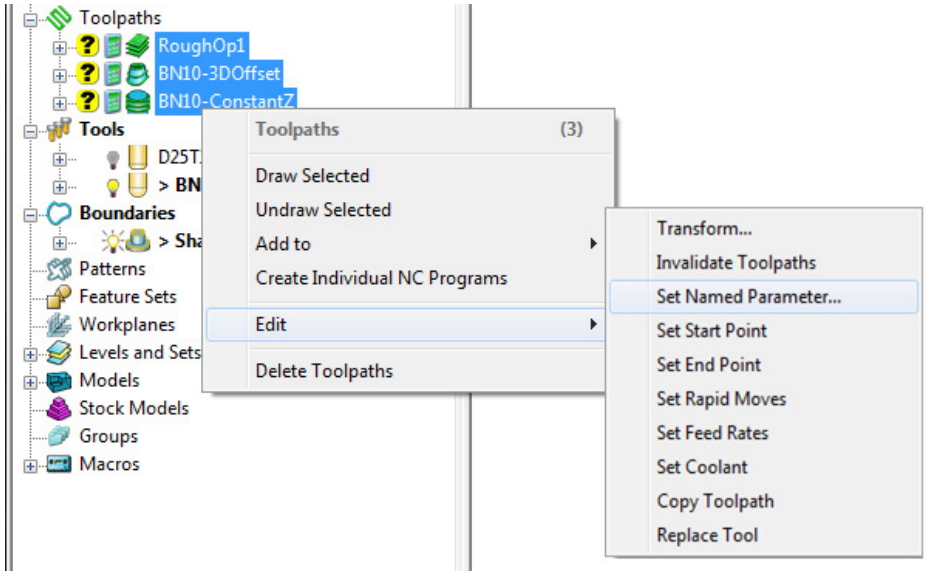
7 **Activate** the **BN10** tool.

8 Right mouse click on the **Boundary - ShallowBN10** and select **Settings**.

9 In the dialog select **Apply** to create the correct **Shallow Boundary** for the new **model** and then **Accept** to close.

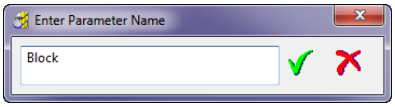
10 **Activate** the new **Boundary**.

11 In the **PowerMILL *explorer*** select all 3 unprocessed **toolpaths** and right mouse click on them.



 Copyright © Delcam plc

12 From the local menu select **Edit** – **Set Named Parameter** and in the dialog input **Block**.

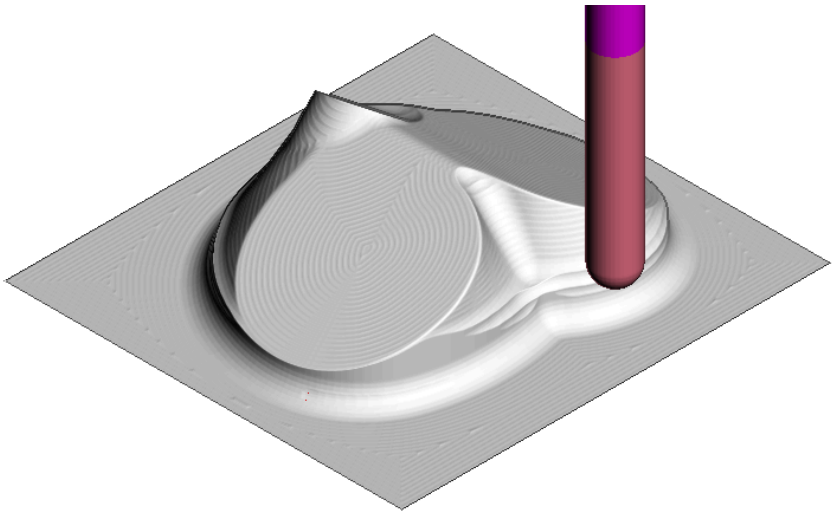


13 From the same local menu select **Edit** – **Set Rapid Moves**.

*The **Block**, **Rapid Move Heights**, and **Active boundary** will be now embedded in the 3 unprocessed toolpaths.*

14 In the **PowerMILL *explorer*** right mouse click on **Toolpaths** and from the local menu select **Batch Process**.

15 Run a **ViewMILL simulation** on all of the 3 **toolpaths**.



16 **Select File - Save Project** to update:-

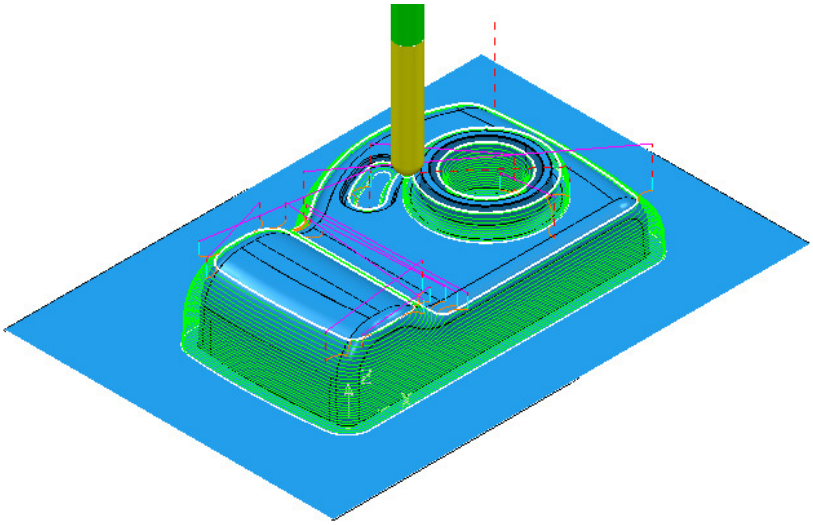
.....\COURSEWORK\PowerMILL_Projects\ChamberFromTemplate

3D Offset Finishing – Spiral and Smoothing Options

17 Select **File – Delete All** and **Tools – Reset forms**.

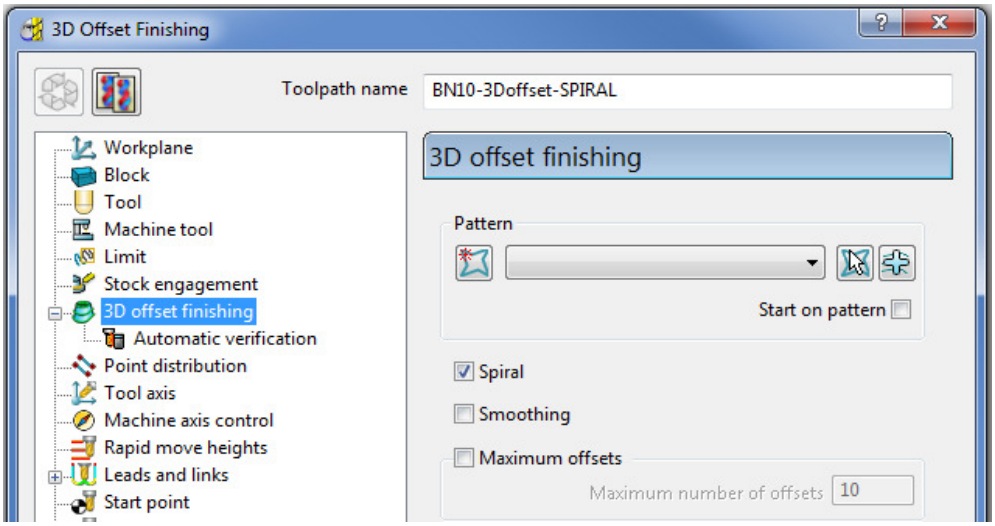
18 **Open** the (recently saved) **Project**:-

.....\COURSEWORK\PowerMILL-Projects\Camera_Example



19 In the **PowerMILL *explorer***, right-click the **BN10-3Doffset** toolpath and select **Settings**.

20 Click to create a copy of the toolpath.



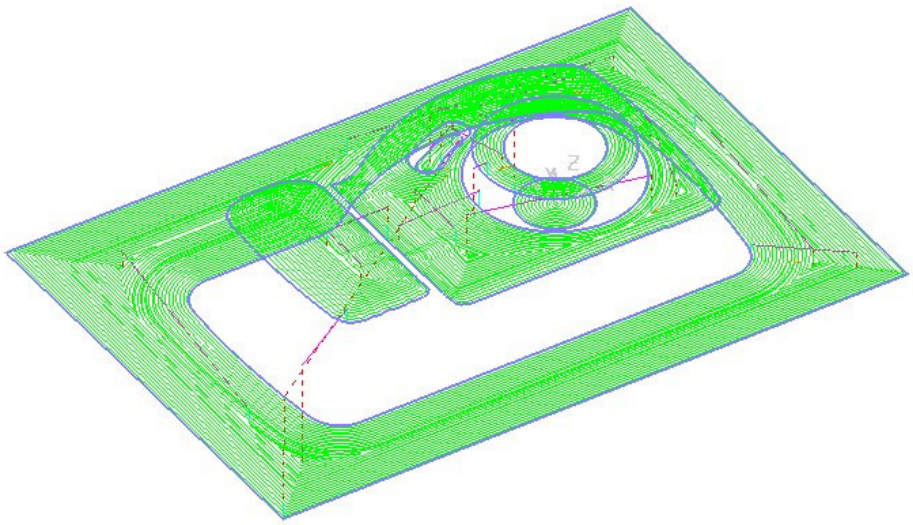
a **Rename** the toolpath as **BN10-3Doffset-Spiral**.

b Tick the **Spiral** option.

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

21 Select **Calculate** and when the toolpath is created, **Close** the dialog.

Copyright © Delcam plc



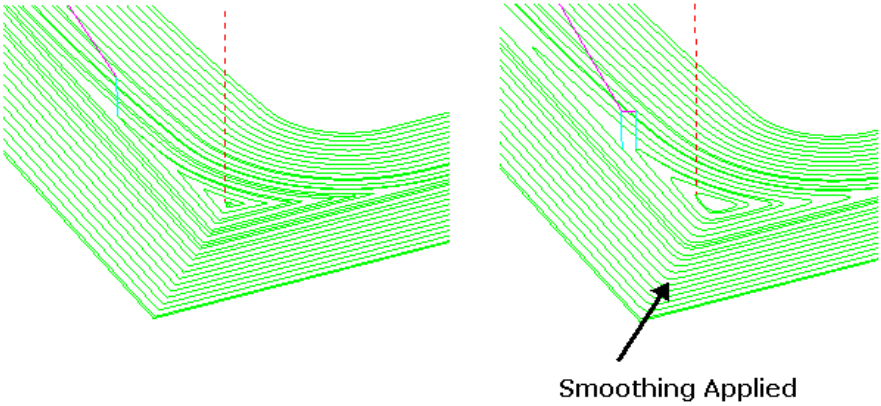
To further improve the compatibility of this strategy for HSM, you can select the **Smoothing** option. This results in sharp corners being replaced with a smooth blend wherever possible.

22 In the **PowerMILL *explorer***, right mouse click on the **BN10-3Doffset_Spiral** toolpath and select **Settings**.

23 Click to create a copy of the toolpath.

24 In the **3D Offset Finishing** strategy dialog:

- **a** Rename the new toolpath as **BN10-3Doffset-Spiral_Smooth**.
- **b** Tick the **Smoothing** option.
- **c** Click **Calculate** and when the toolpath is created, **Close** the dialog.



25 Select **File** - **Save Project** to update (but do not close!):-

.....\COURSEWORK\PowerMILL-Projects\Camera_Example

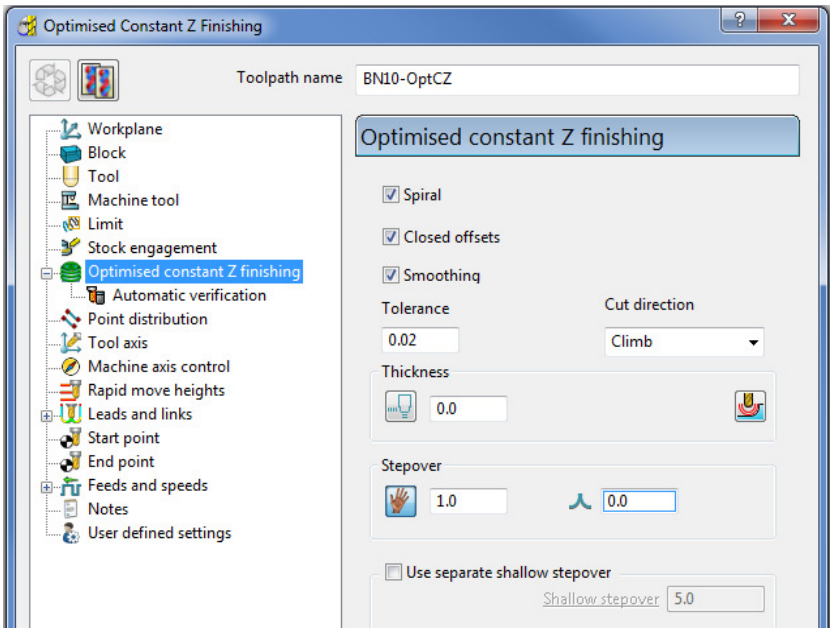
Optimised Constant Z Finishing

This strategy is a mixture of **Constant Z** and **3D Offset** machining; **Constant Z** is applied to steep areas and, **3D Offset** is applied to other areas.

1 Deactivate the **Boundary**, ***ShallowBN10*** in the **PowerMILL *explorer***.

2 Select the **Toolpath Strategies** dialog and from **Finishing** tab, select **Optimised constant Z finishing** and then OK.

3 In the **Optimised constant Z finishing** dialog:



a Enter **Toolpath name** as **BN10-OptCZ**

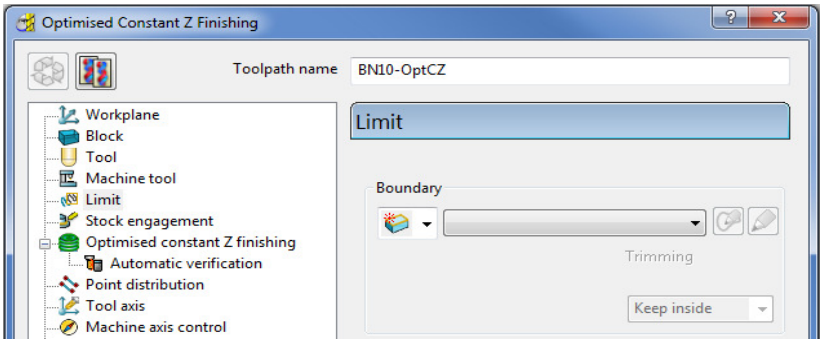
b Select the **Spiral**, **Closed Offsets**, and **Smoothing** options.

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

d Set the **Cut direction** to **Climb**.

e Enter a **Stepover** value of **1**

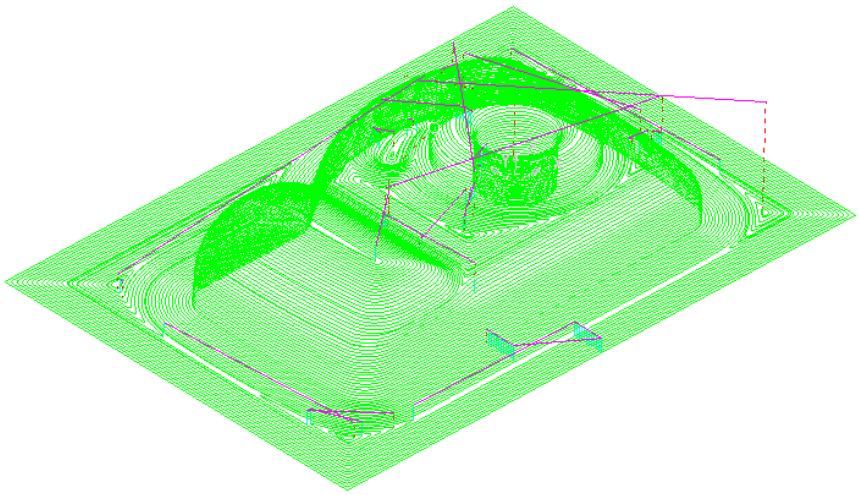
f On the **Limit** page, make sure that the **Boundary** is not selected.



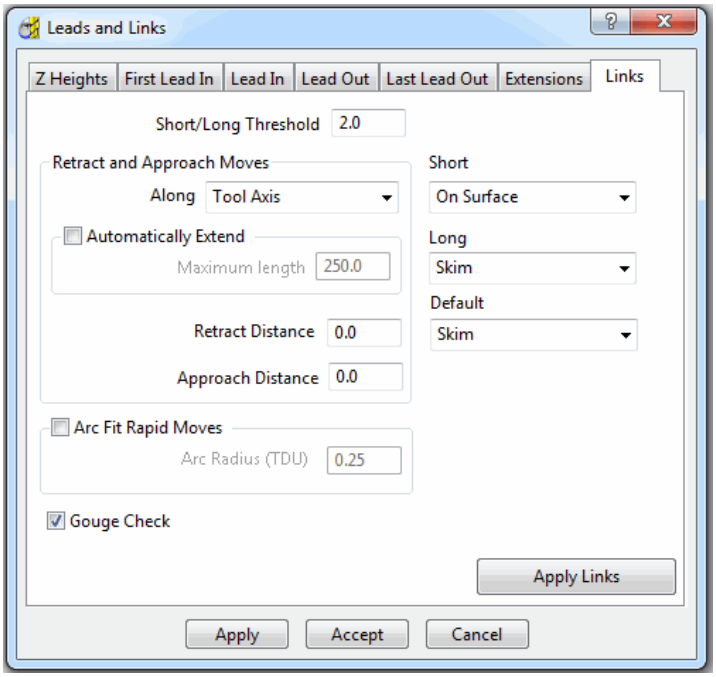
4 Select **Calculate** and when the toolpath is created, **Close** the dialog.

Copyright © Delcam plc

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



5 From the **Main** menu, click the **Leads and Links** button to display the **Leads and Links** dialog and then, select the **Links** tab.



a Change the **Short/Long Threshold** to 2.

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

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

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

7 Select **File - Save Project** to update (but do not close):-

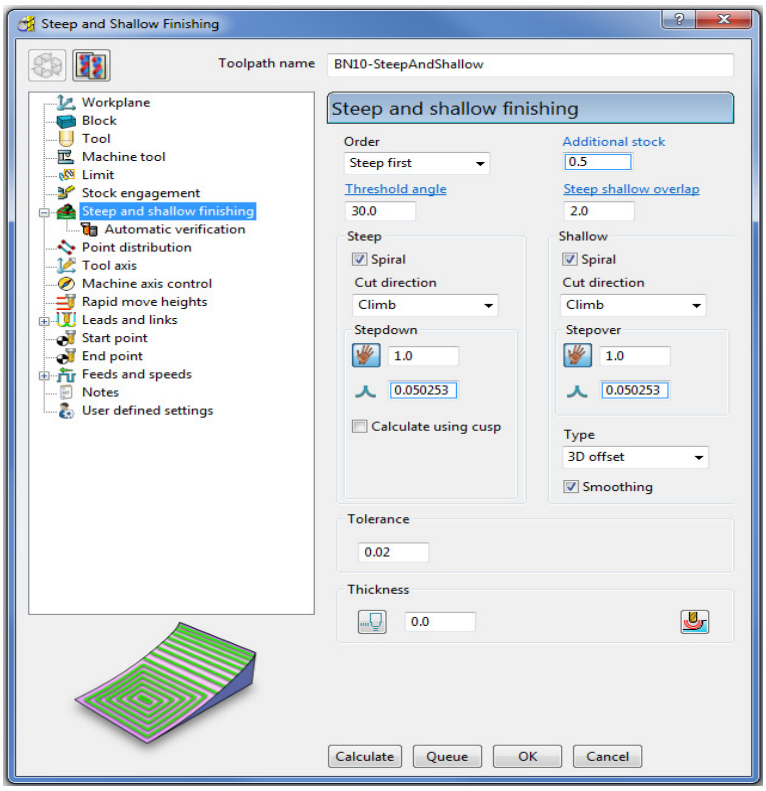
.....\COURSEWORK\PowerMILL-Projects\Camera_Example

Steep and Shallow Finishing

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

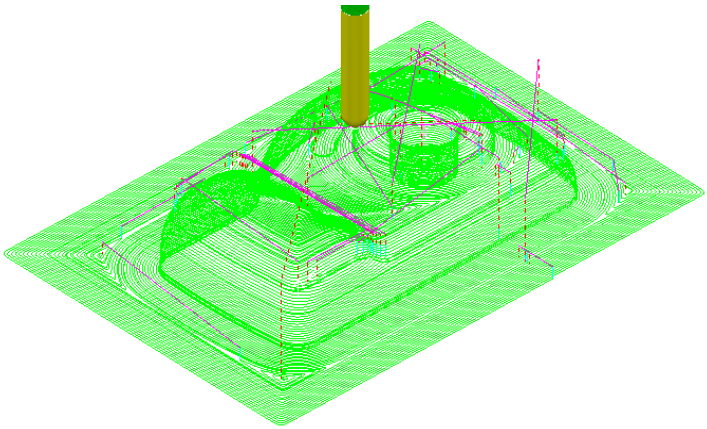
1 From the **Finishing** strategies tab in the **Strategy Selector** dialog, select **Steep and Shallow** finishing.

2 Enter the values exactly as shown:



a Enter **Toolpath name** as **BN10-SteepAndShallow**

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



You can control the angular threshold between the **3D Offset** and **Constant Z** components of the hybrid **Steep and Shallow** finishing strategy.

4 Select **File** - **Save Project** to update:-

.....\COURSEWORK\PowerMILL-Projects\Camera_Example

Copyright © Delcam plc

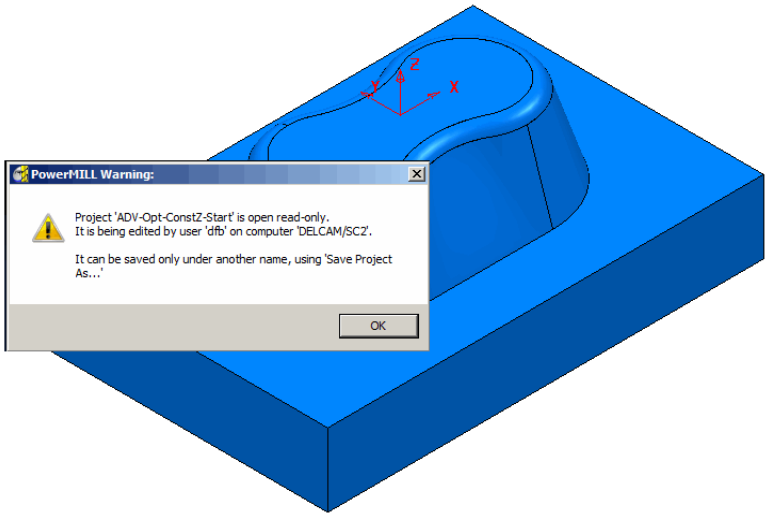
Optimised Constant Z and Steep and Shallow finishing with separate Shallow Stepover

Both **Optimised Constant Z** and **Steep and Shallow** strategies have an option to apply a separate, larger **Stepover** across shallow areas. This option is designed to enable a more efficient machining of shallow areas with a **Tip Radiused** tool.

1 **Delete All** and **Reset Forms**.

2 Select **File** - **Open Project** and in the dialog browse to:

.....\PowerMILL_Data\Projects\ADV-Opt-ConstZ-Start



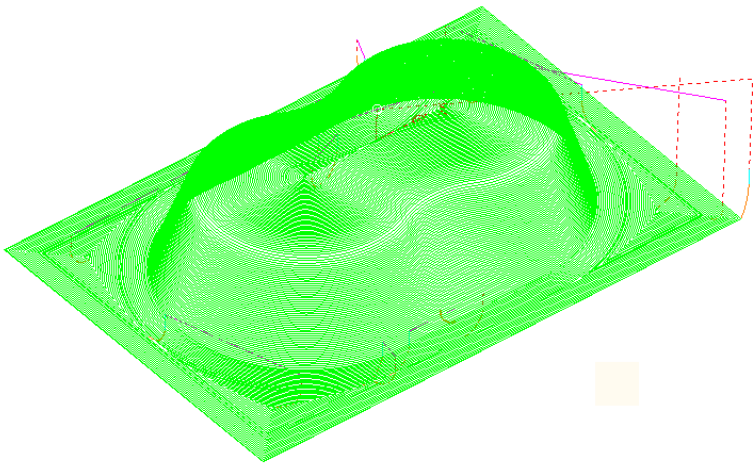
3 Click **OK** to load the **Project** into **PowerMILL**.

4 **Save Project As**:

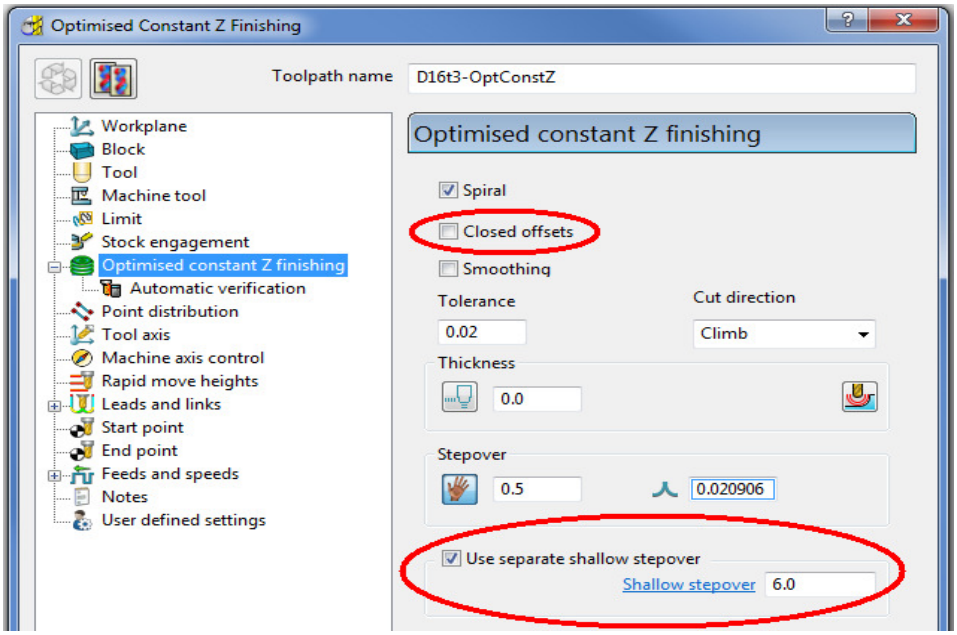
.....\COURSEWORK\PowerMILL_Projects\ADV-CZ-example

5 In the ***PowerMILL*** **explorer**, right-click toolpath **D16t3-OptConstZ** and select **Settings**.

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



6 Click Recycle to enable edits to the existing toolpath.

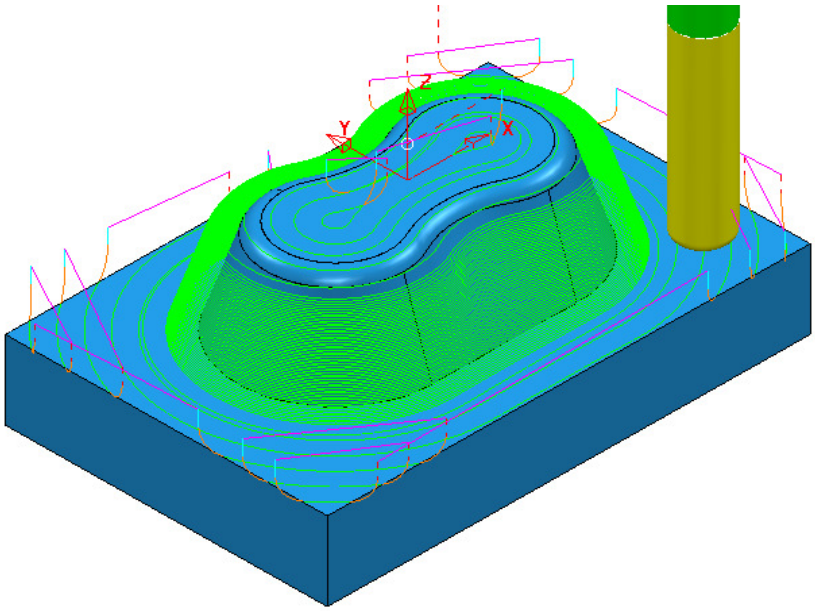


a Un-tick **Closed offsets**.

b Select the **Use separate shallow stepover** option.

c Enter a **Shallow stepover** of **6.0**

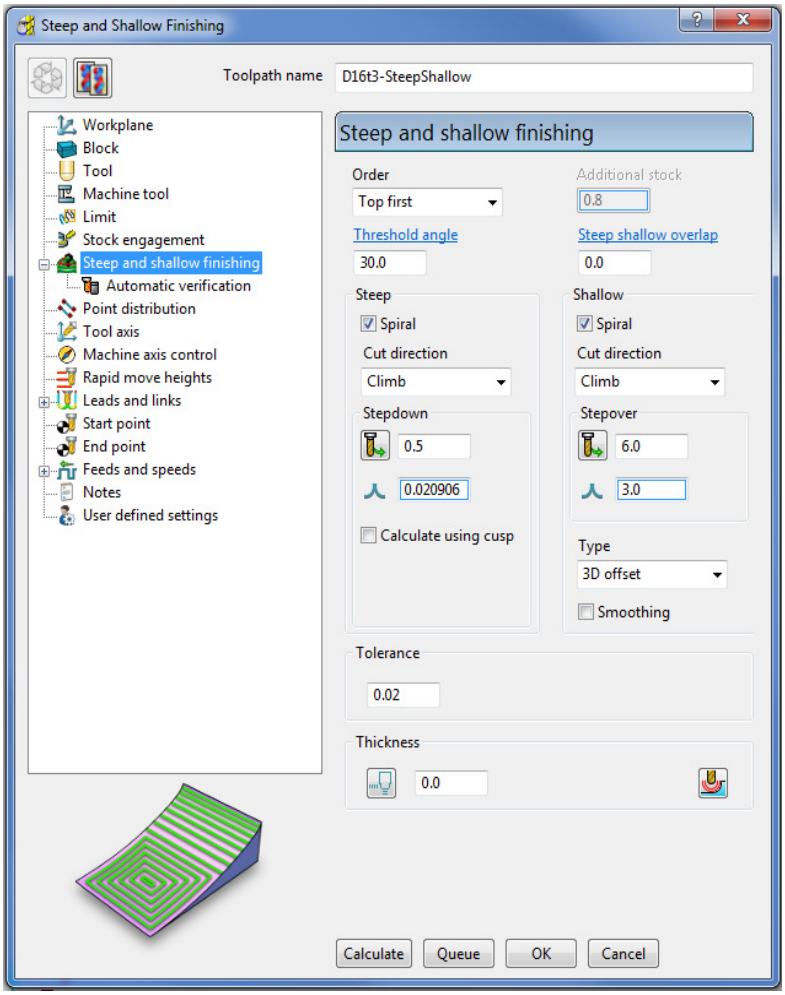
7 Click **Calculate** to process the toolpath and then **Close** the dialog.



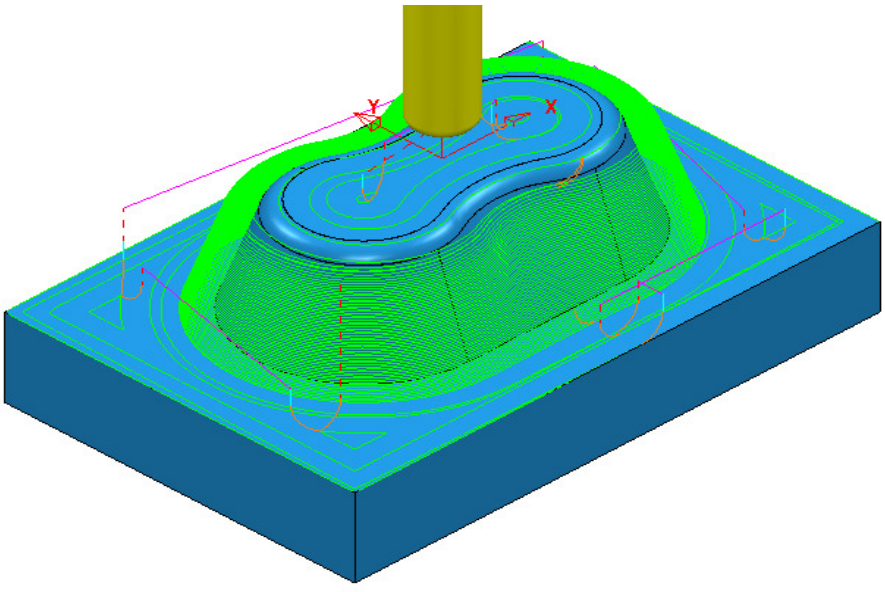
The flat areas are machined using a large stepover while the steep areas use a small stepover.

8 Open a new **Steep and Shallow** finishing strategy and enter data exactly as shown below.

Copyright © Delcam plc



9 Click **Calculate** to process the toolpath, and then **Close** the dialog.



10 Select **File - Save Project**, to update:-

.....\COURSEWORK\PowerMILL_Projects\ADV-CZ-example

Constant Z Finishing – Additional Stock option

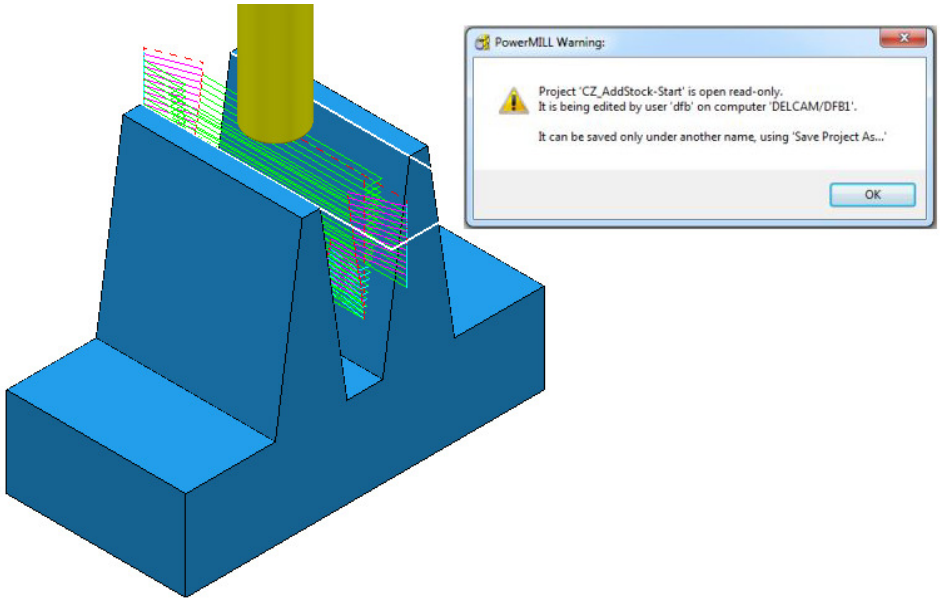
The **Additional Stock** option causes a **Constant Z** strategy to locally switch the ***Order by*** from **Region** to **Level** below the point where the width of a deep, angled slot reaches a specified width.

Above this point, the tool will continuously machine down each individual, angled face. Below this point the tool will alternate machining from one angled face to the other as the strategy steps downwards.

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

12 Select **File** - **Open Project** and in the dialog browse to:

.....\PowerMILL_Data\Projects\CZ_AddStock-Start



13 Select OK in the **PowerMILL Warning** dialog and **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\CZ_AddStock-EX1

14 Right-click the **Roughing** toolpath and from the local menu select **Activate** to reinstate the original **Block**.

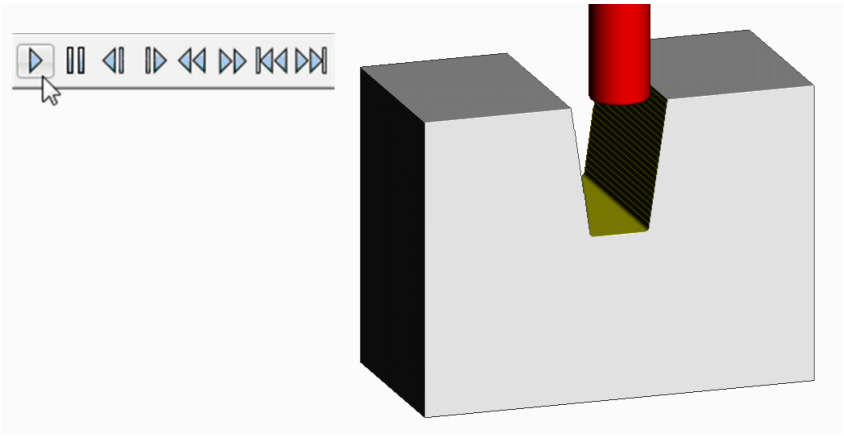
15 Right-click the **Roughing** toolpath again and select **Simulate from Start**.

16 From the **Main** toolbar, select **View** > **Toolbar** > **ViewMILL** to access the **ViewMILL** toolbar (if not already displayed).

17 On the **ViewMILL** toolbar click the **Red** sphere to switch **ViewMILL** on (**Green** sphere).

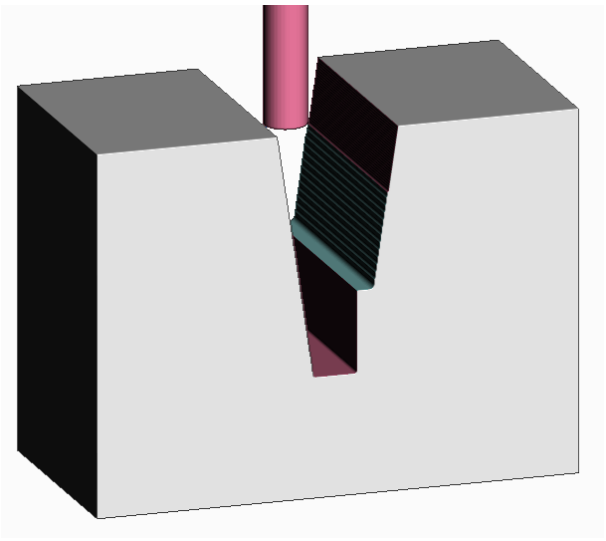
18 Click **Rainbow Shading** button on the **ViewMILL** toolbar and then click **Play** on the **Simulation** Toolbar.

Copyright © Delcam plc



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.

19 Right mouse click on the **0_AdditionalStock** toolpath and select **Simulate from Start** followed by **Play** button on the **Simulation** Toolbar.



You can see that the existing **Constant Z** finishing strategy ploughs 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, re-calculate the **Constant Z** strategy to include suitable settings in the **Add Stock** option.

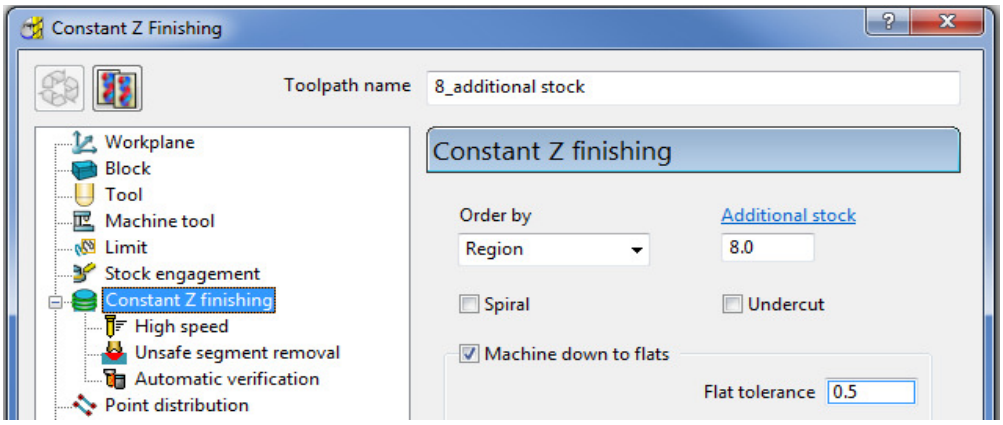
20 In the **ViewMILL** toolbar select the **No Image** button and switch **ViewMILL On/Suspend** back to red .

21 **Activate** the **0_AdditionalStock** toolpath.

22 Right mouse click on the toolpath, and from the local menu select **Settings** to open the original **Constant Z Finishing** dialog.

23 Click to create a copy of the toolpath.

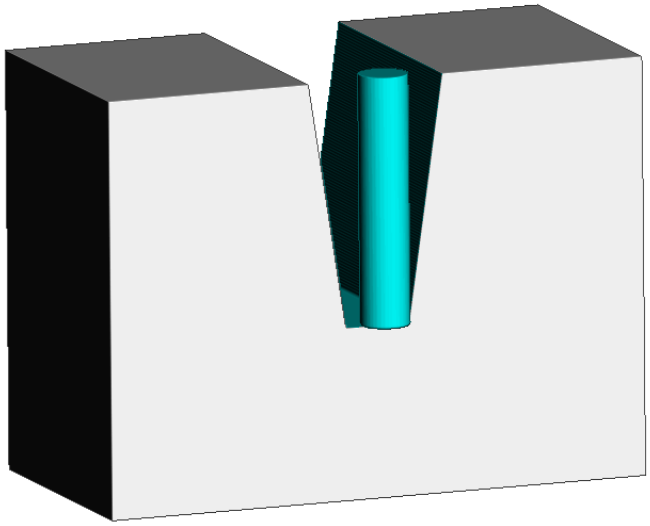
24 Rename the new toolpath as **8_AdditionalStock** and in the **Additional Stock** box input the value **8.0**



25 Click **Calculate** toolpath and then **Close** the dialog.

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

26 Repeat the **ViewMILL** simulation, starting with the **Roughing** strategy as before, but then following on with the toolpath **8_AdditionalStock**.



The new **Constant Z** finishing strategy orders the tool tracks to machine continuously down each individual sidewall to a height just above the remaining stock. Below this point, the tool tracks alternate from wall to wall into the remaining stock.

27 Select **File - Save Project** to update:-

.....\COURSEWORK\PowerMILL-Projects\CZ_AddStock-EX1

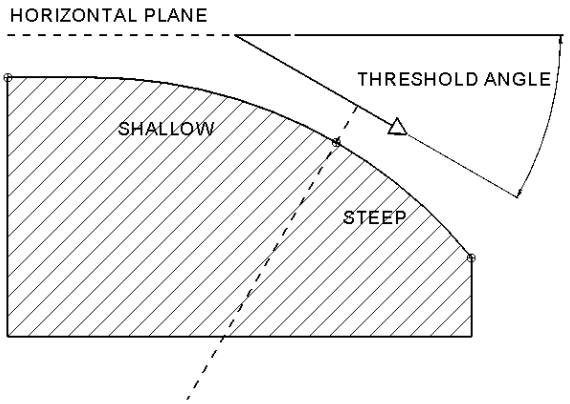
Copyright © Delcam plc

Corner Finishing

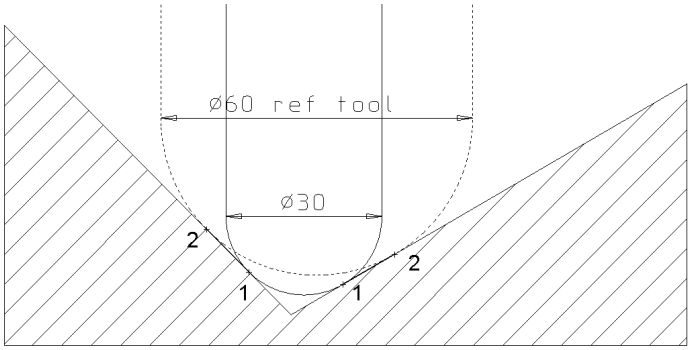
Corner Finishing strategies identify and remove material in corners that are inaccessible to previously used, larger tools. There are **3** independent types of strategy: **Along, Stitch**, and **Automatic**.

All **Corner Finishing** strategies include a **Threshold Angle** option. When you select the **Output** type either as **Steep** or as **Shallow**, the **Threshold Angle** determines the angle from the horizontal at which steep and shallow portions are split.

You can modify the **Threshold Angle** to prevent the tool running up or down steep slopes. For example, you could apply a **Stitch** strategy to track across the steep areas, while using the **Along** strategy in shallow areas.



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



In the following example, the reference tool is a Dia 10 Ball-nose and the **Corner** finishing is continues with a diameter Dia 5 Ball-nose.

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

2 From the **Main** menu, Select **File - Open Project** and select:

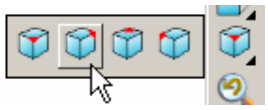
.....\PowerMILL_Data\Projects\CornerFinish_Start

3 From the **Main** menu, select **File - Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\CornerFinish

Corner Along Finishing

4 Select an **ISO2** view.

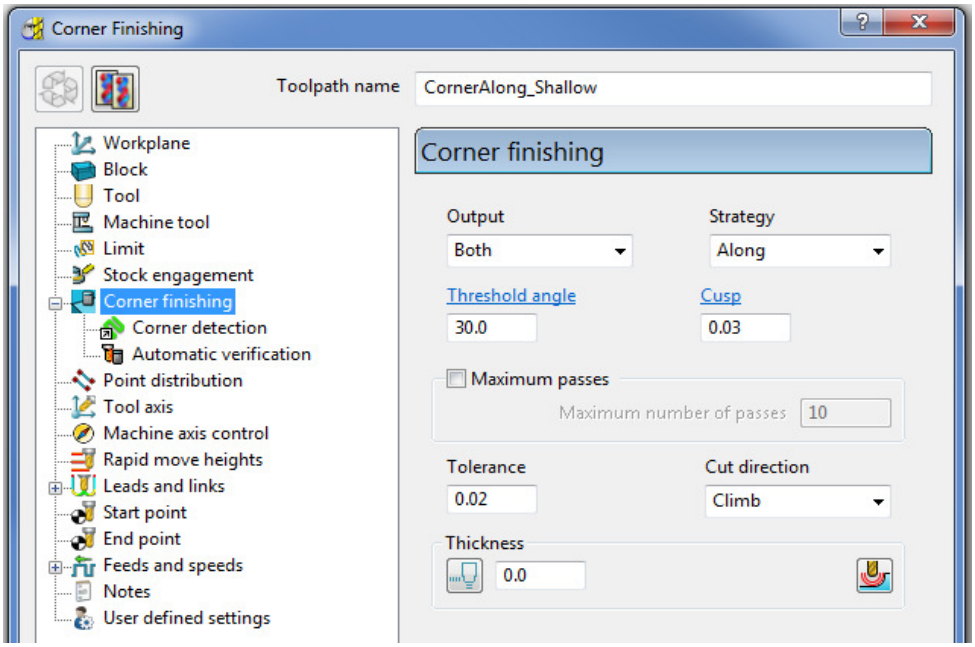


5 **Activate** the **toolpath**, **D16T3-rghA1** to restore the settings.

6 Define a diameter 5 ball nose tool named **BN5**

7 Click and open the **Toolpath** strategies dialog, and from the **Finishing** tab, select **Corner Finishing**.

8 In the **Corner Finishing** strategy dialog:



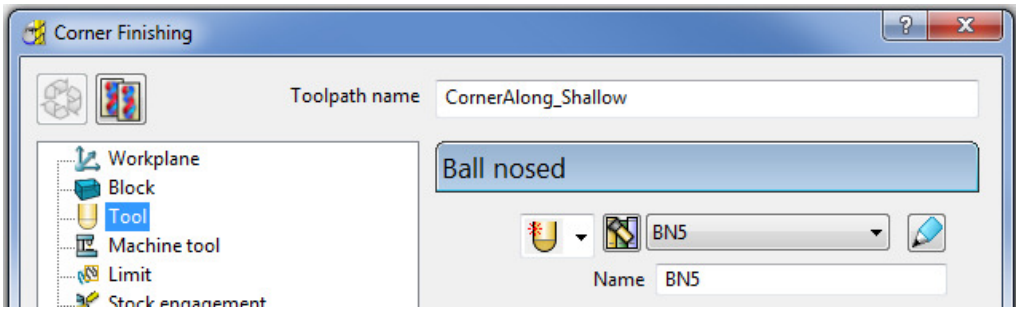
a Enter **Toolpath** name as **CornerAlong_Shallow**

b Change **Output** to **Shallow** and **Strategy** to **Along**.

c Set **Cusp** to **0.03**

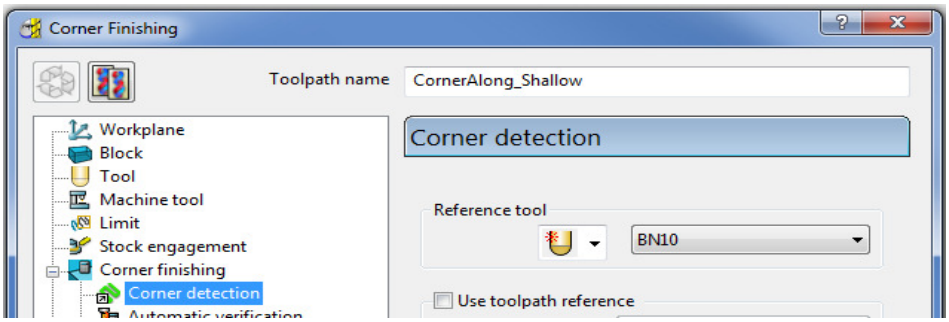
d Set **Cut direction** to **Climb**.

e On the **Tool** page, check that the **BN5** tool is selected.



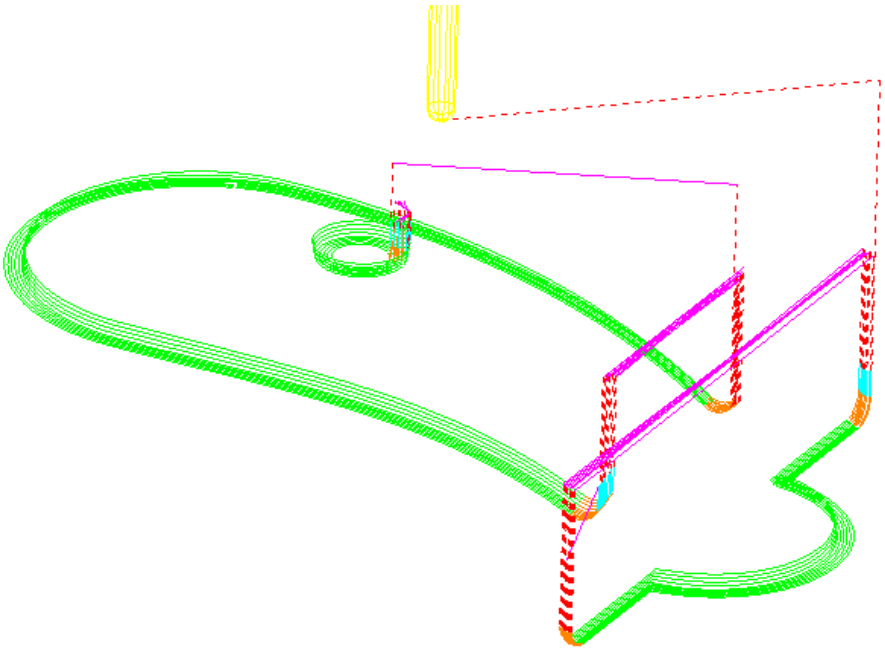
f Select the **Corner** detection page and select the **Reference tool** as **BN10**.

Copyright © Delcam plc



9 **Calculate** to process the toolpath and then **Close** the dialog.

10 **Simulate** the toolpath and observe the order of the machining:



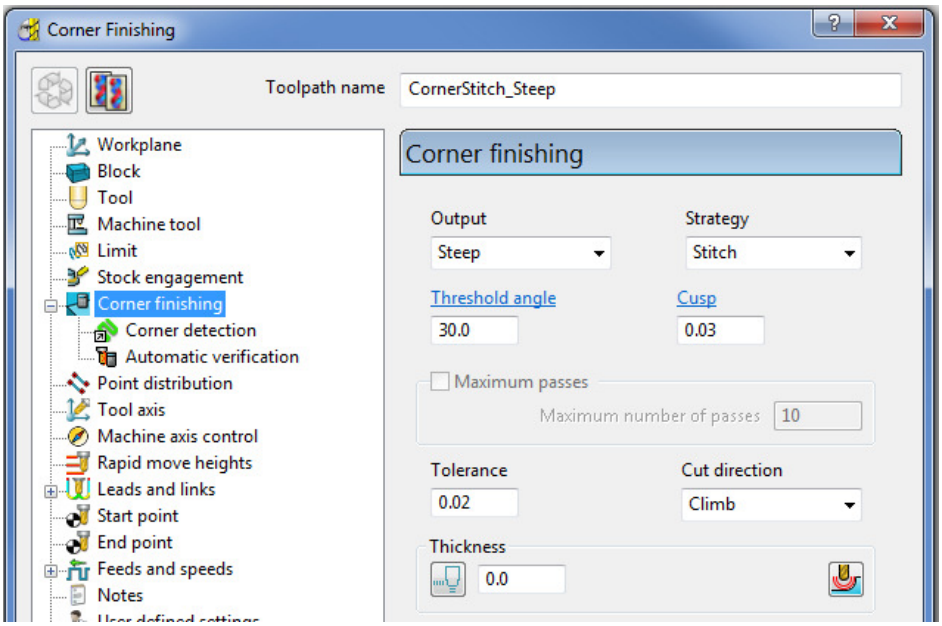
Note: With **Output** set to **Shallow**, the toolpath has stayed within the shallow areas of the model (between horizontal and a downward angle of 30 Degrees).

*Use a **Threshold angle** of **90** for the **Shallow** option to apply in all valid rest areas. This would also result in the **Steep** option area to be non-existent.*

Corner Stitch Finishing

1 From the **Finishing** strategies tab, select **Corner Finishing**.

2 In the **Corner Finishing** strategy dialog:



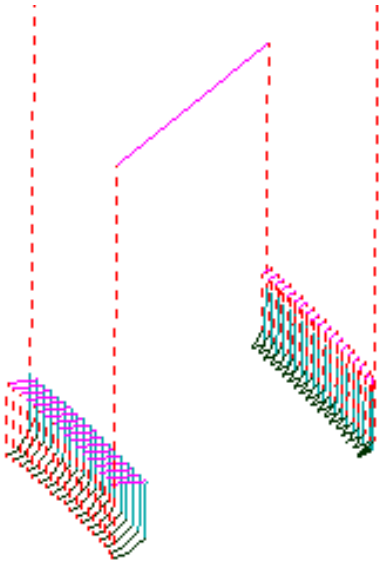
a Enter **Toolpath name** as **CornerStitch_Steep**

b Change ***Output*** to **Steep** and ***Strategy*** to **Stitch**.

c Check that the **BN5** tool is active on the **Tool** page.

d Check that the **Reference tool** is **BN10** on the **Corner detection** page.

3 **Calculate** to process the toolpath and then **Close** the dialog.



This time, only the steep areas are machined.

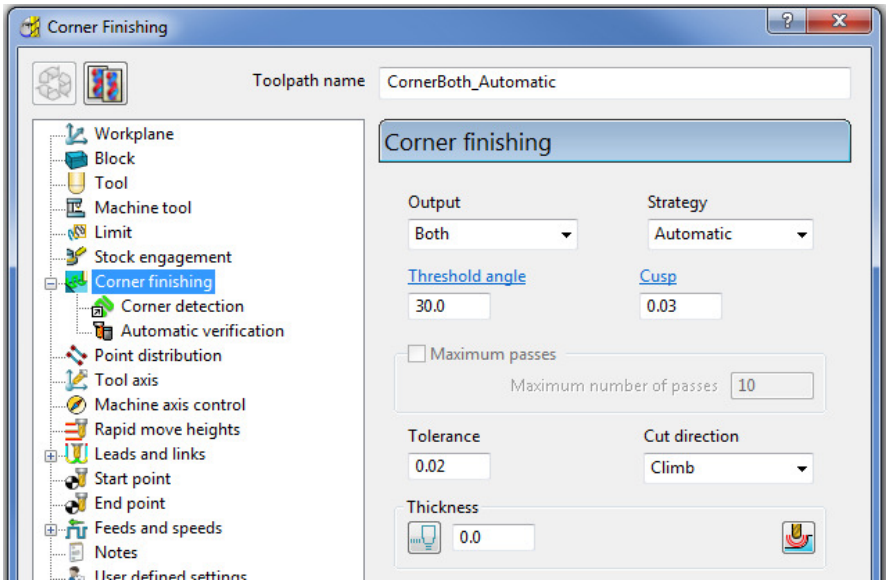
4 From the **Main** menu select **File > Save Project** to update:-

.....\COURSEWORK\Projects\CornerFinish

Copyright © Delcam plc

Corner Automatic Finishing

1 From the **Toolpath strategies** dialog, select the **Corner Finishing** strategy and enter the data into the dialog exactly as shown below:

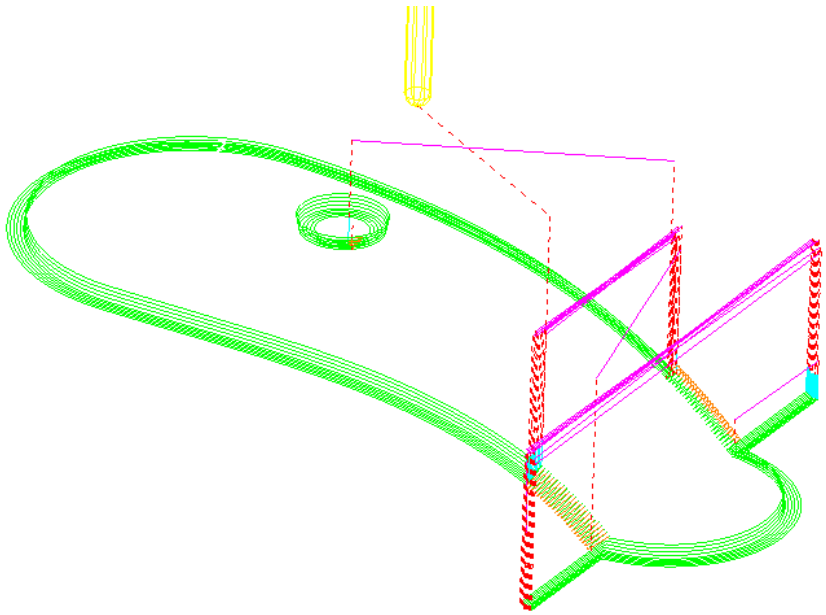


a Use **Strategy** - **Automatic**.

b Change **Output** to **Both**.

c Change the **Threshold** angle back to **30.0**

2 Click **Calculate** to process the toolpath and then **Close** the dialog.



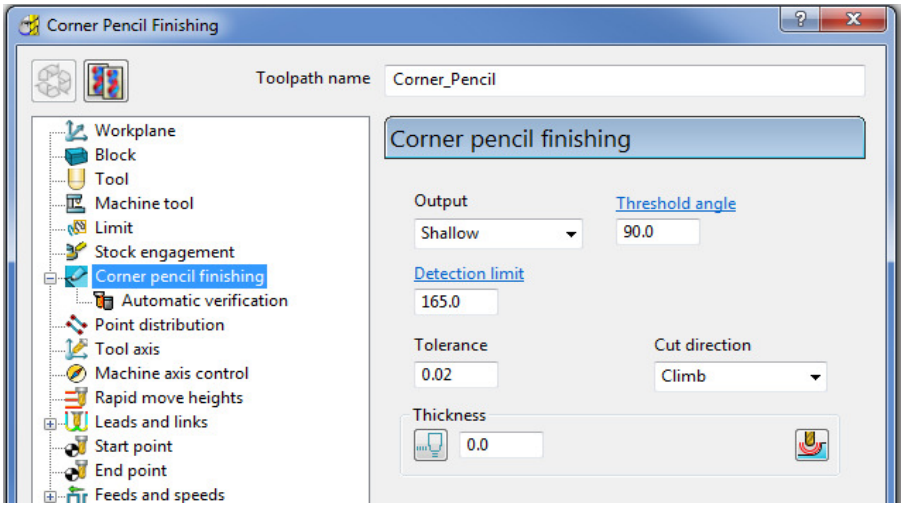
This strategy automatically uses **Along** for the **Shallow** areas and **Stitch** for the **Steep** areas.

Corner Pencil Finishing

Corner Pencil creates a single toolpath along the actual intersection between non-tangential faces on the model.

1 From the **Toolpath Strategies** dialog, select and open the **Corner Pencil Finishing** strategy.

2 In the **Corner Pencil Finishing** strategy dialog:



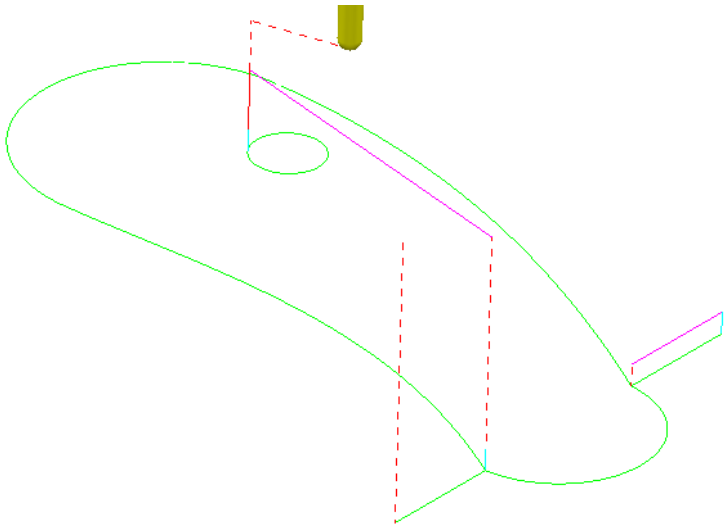
a Change **Output** to **Shallow**.

b Change the **Threshold Angle** to 90

*Note: With a **Threshold Angle** of **90** the **Steep** angle will not exist, so only **Output** with **Shallow** or **Both** selected will enable a toolpath to be created.*

c On the **Tool** page, make the **BN5** tool active.

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



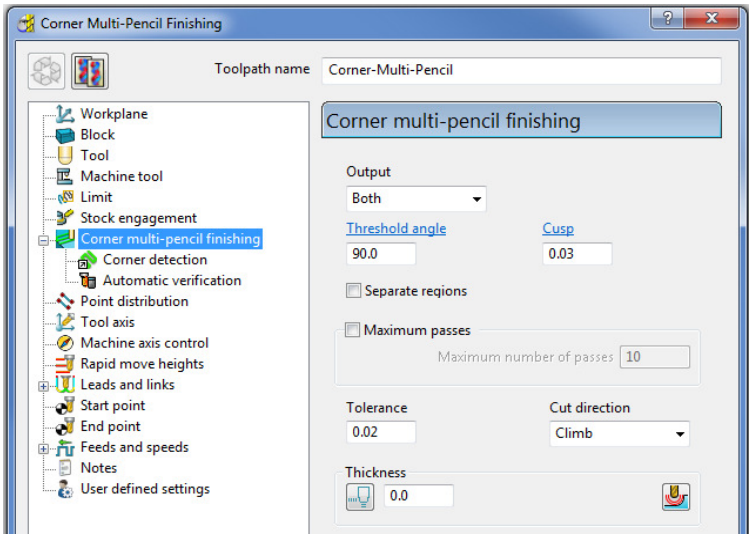
Copyright © Delcam plc

Corner Multi-pencil Finishing

Corner Multi-pencil Finishing creates a multi-track, rest finishing, toolpath based on the **Pencil Finishing** strategy. On creation it visually looks similar to a **Corner Finishing - Along** toolpath, but on closer inspection there are subtle differences to the toolpath order and the fact it includes tool tracks along the actual model intersections.

1 From the **Toolpath Strategies** dialog, select and open the **Corner Multi-Pencil Finishing** strategy.

2 In the **Corner Multi-Pencil Finishing** strategy dialog:



d Change **Output** to **Both**.

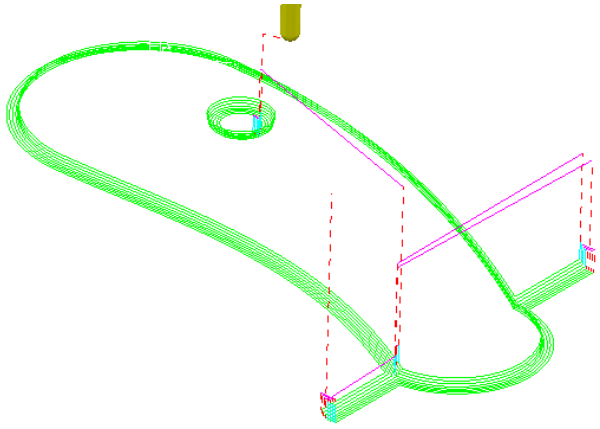
*Note: With a **Threshold Angle** of **90** the **Steep** angle will not exist, so only **Output** with **Shallow** or **Both** selected will enable a toolpath to be created.*

e Change the **Threshold Angle** to **90**

f On the **Tool** page, make the **BN5** tool active.

g In the **Corner detection** page check that **BN10** is selected.

3 **Calculate** to process the toolpath and then **Close** the dialog.



4 From the **Main** menu select **File > Save Project** to update:-

.....\COURSEWORK\Projects\CornerFinish

Flat Finishing

These strategies apply to purely flat surfaces only. Flat areas within a variable height surface are not recognised. The minor discrepancies in the flatness of a surface is controlled by a **Flat Tolerance** value.

1 **Delete All** and **Reset Forms**.

2 From the **Main** menu, select **File - Open Project**:

.....\PowerMILL_Data\Projects\FlatFinish_Start

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

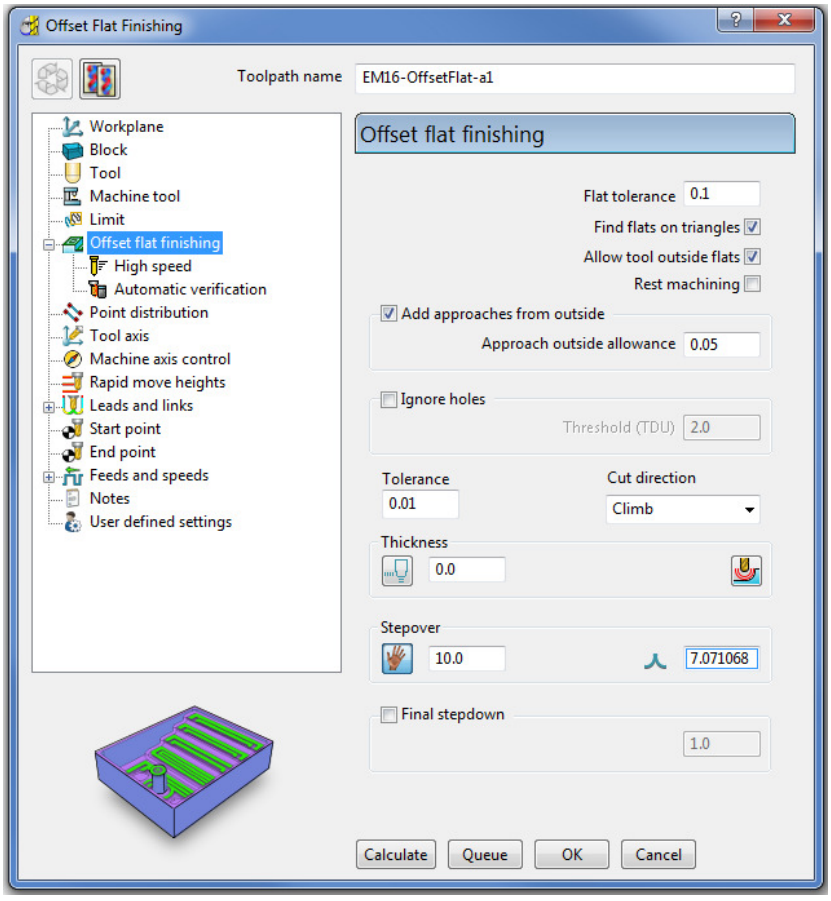
.....\COURSEWORK\PowerMILL-Projects\FlatFinish-Example1

Offset Flat Finishing

1 In the **PowerMILL *explorer***, **Activate** the Dia 16 End Mill (**EM16**) tool.

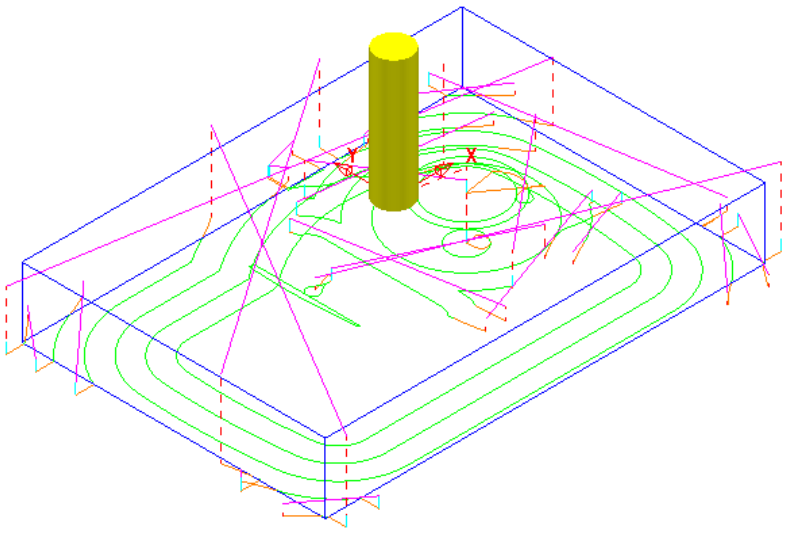
2 In the **Strategy Selector** dialog, select the **Finishing** tab, and select **Offset Flat Finishing**.

3 Fill in the **Offset Flat Finishing** dialog exactly as shown.



4 Select **Calculate** and when completed, **Close** the dialog.

Copyright © Delcam plc



In this case, suitable Leads have been applied retrospectively:

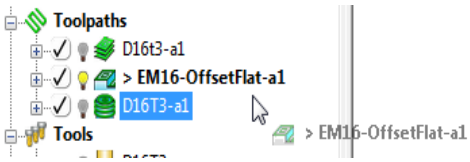
Lead In/Out:

1st choice - Pocket Centre

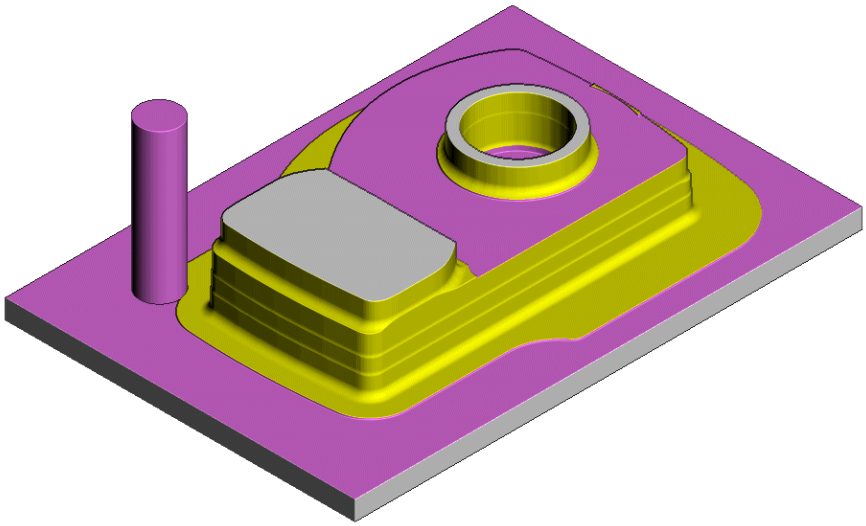
2nd Choice - Horizontal Arc - Angle 90 - Radius 8

The 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 make good sense to move it up the toolpath order in the **PowerMILL *explorer***.

1 In the **PowerMILL *explorer***, left click and drag the **Offset Flats Finishing** toolpath (**EM16Flats-a1**) to the new position.



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

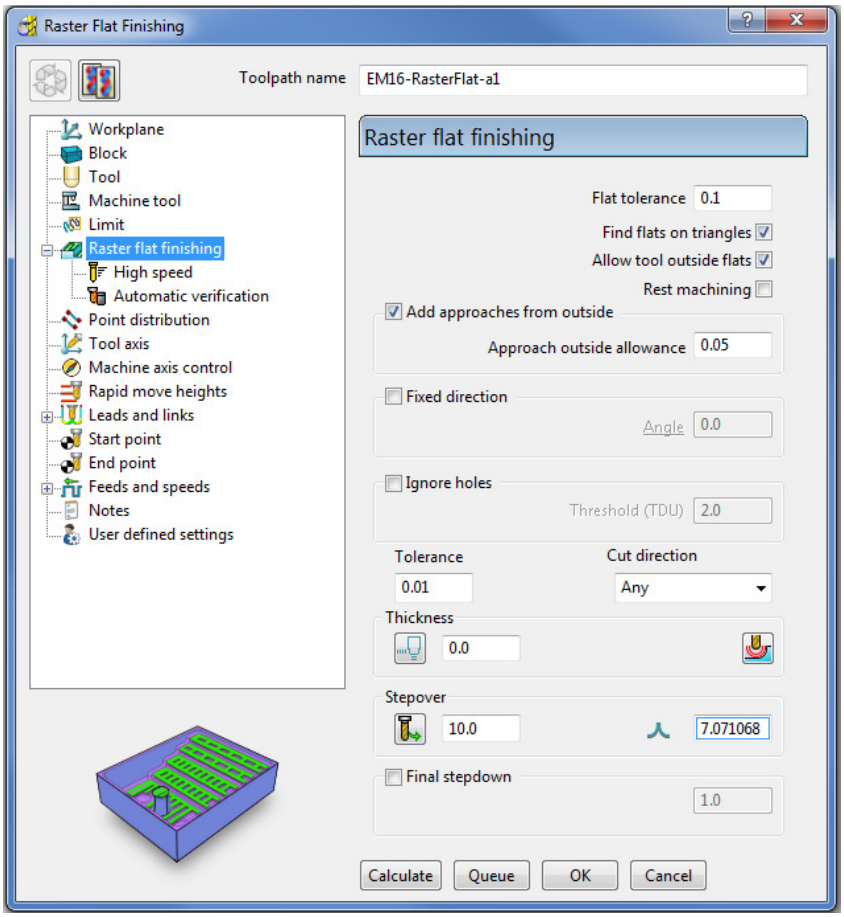


The **ViewMILL** simulation after the **Offset Flat Finishing** strategy is shown above.

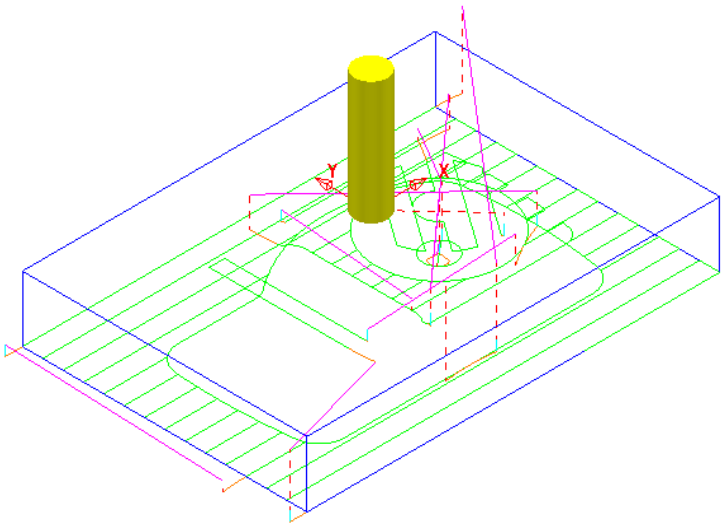
Raster Flat Finishing

1 In the **Strategy Selector**, from the **Finishing** tab, select **Raster Flat Finishing**.

2 Fill in the dialog exactly as shown and click **Calculate**.



3 Select **Calculate** and when completed, **Close** the dialog.



4 From the **Main** menu select **File > Save Project** to update:-

.....\COURSEWORK\PowerMILL-Projects\FlatFinish-Example1

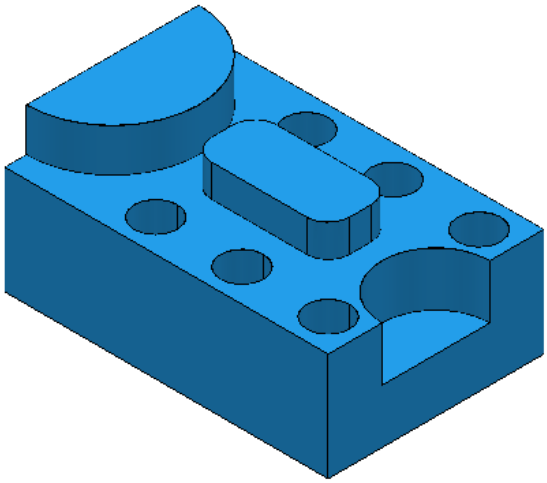
Copyright © Delcam plc

Flat Machining example

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

2 Select **File - Import Model**:

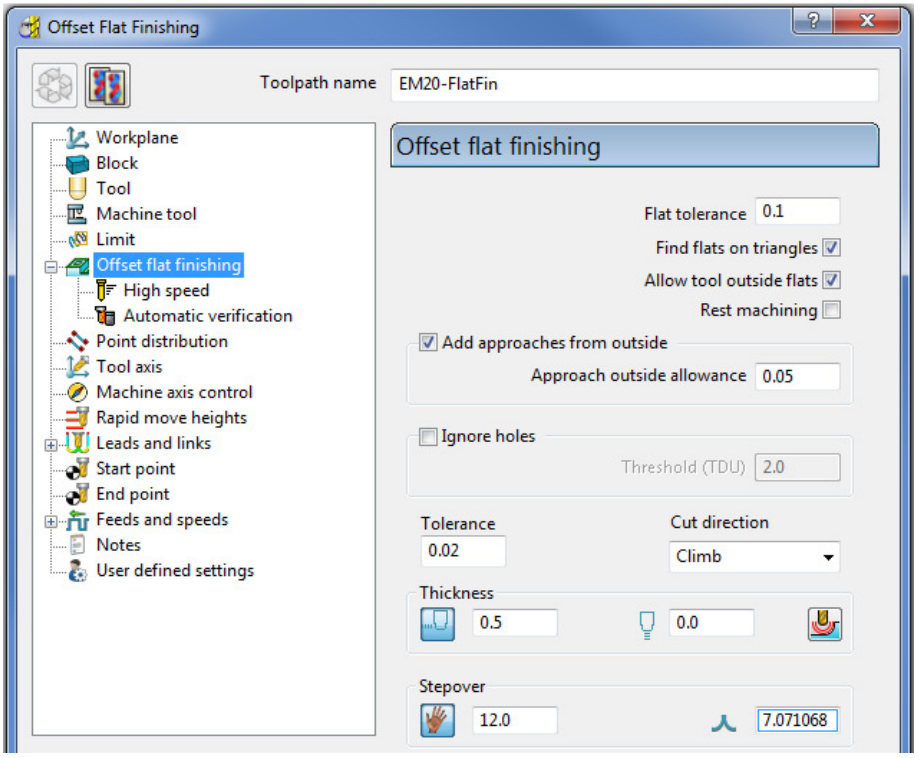
.....\PowerMILL_Data\Models\Flats.dgk



3 Create a **20mm** diameter **End Mill** tool named **EM20**.

4 **Calculate** the (default) **Block** and then reset the **Safe** Heights.

5 Open an **Offset Flat Finishing** strategy and:

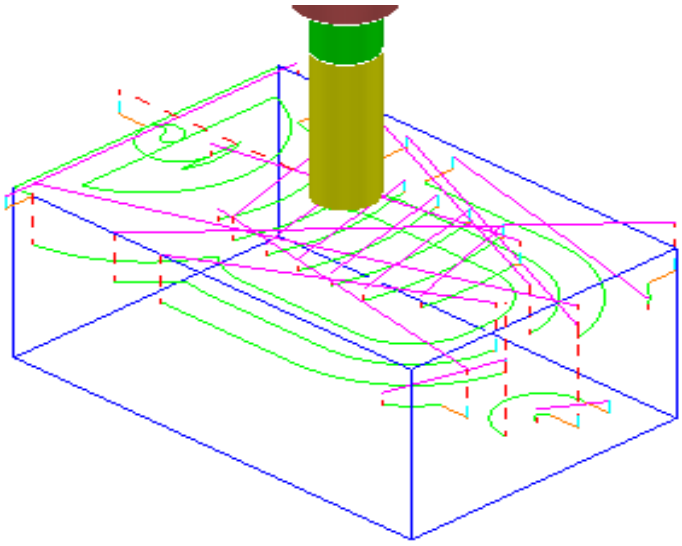


a Name the toolpath **EM20-FlatFin**.

b Click the use **Axial Thickness** button and enter separate **Thickness** values for **Radial 0.5** and **Axial 0**

c **Calculate** to process the toolpath and then **Close** the dialog.

The **Flat** areas are 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.

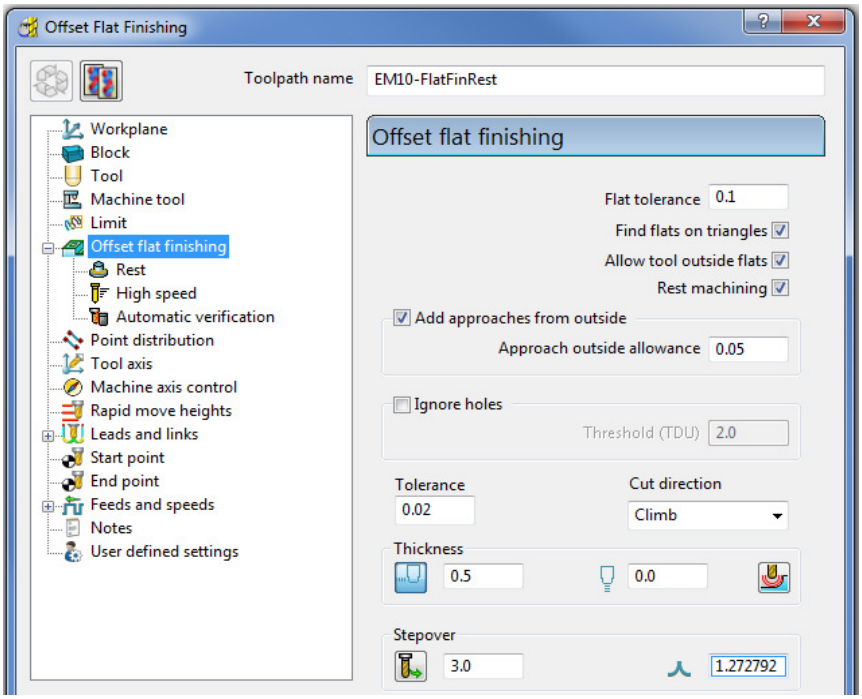


6 Create a **Dia 10 End Mill**, named **EM10**.

7 **Activate** the **Toolpath**, **EM20-FlatFin** and then select **Settings**.

1 Click and create a copy of the toolpath.

2 In the **main** page of the strategy dialog:



a Enter **Toolpath name** as **EM10-FlatFinRest**.

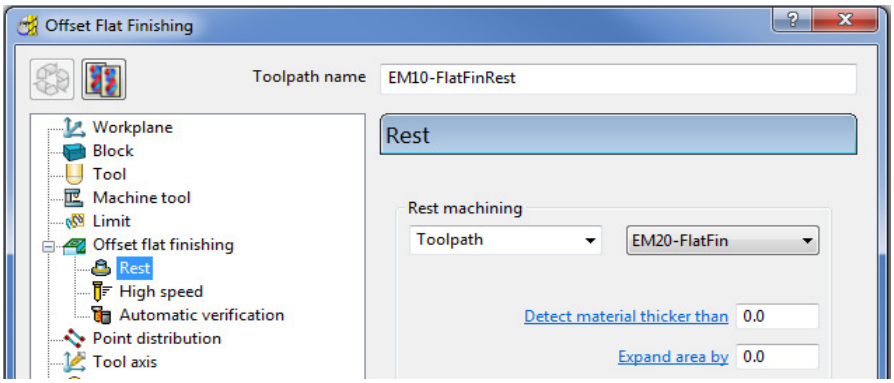
b Enter **Stepover** as **3.0**

c **Activate** the **EM10** tool.

d Tick the **Rest machining** box and then select the **Rest** page.

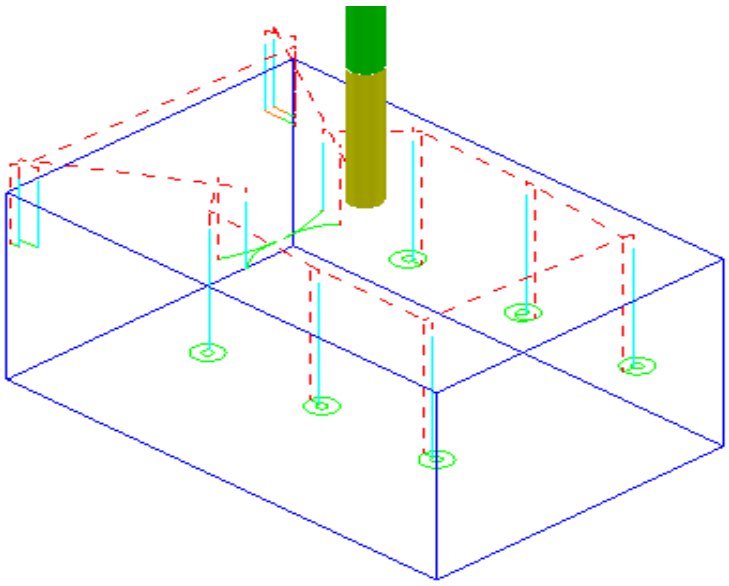
 Copyright © Delcam plc

e On the **Rest** page, select the reference **Toolpath** as **EM20-FlatFin**.



3 **Calculate** the toolpath and when created, **Close** the dialog.

The final finishing passes at the base of the holes and the area behind the boss are now created:



4 From the **Main** menu select **File > Save Project** to update:-

.....\COURSEWORK\FlatFinishing-Example2

Surface Finishing Example 1

The **Surface Finishing** strategy creates tool tracks that follow the natural, ***surface curves*** on a single, selected, **Surface**.

The tool tracks in a Surface Finishing strategy will not run parallel to the edges of a trimmed, surface.

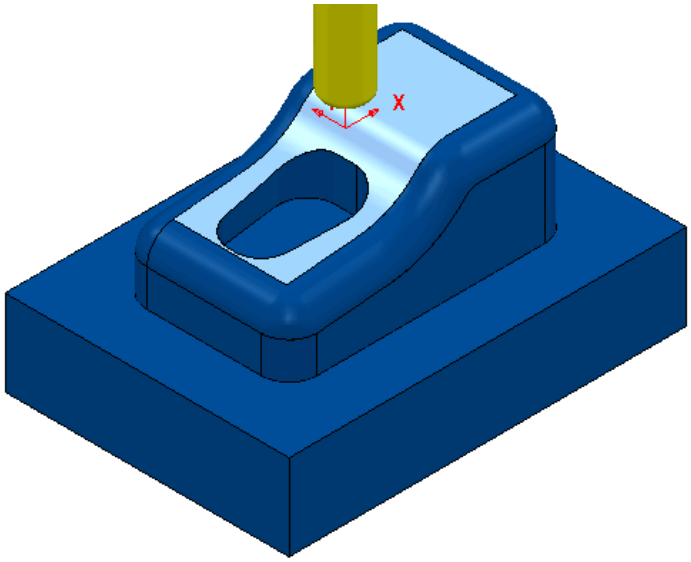
1 From the **Main** menu, select **File - Delete All** and **Tools** - **Reset Forms**.

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

.....\PowerMILL_Data\Projects\EditToolpaths_3

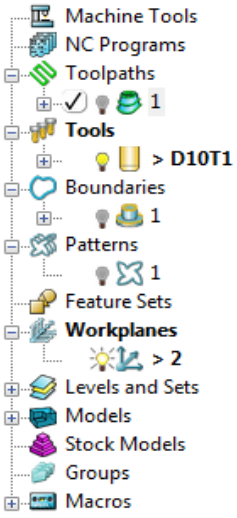
3 From the **Main** menu , select **File - Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\PunchForm_3



4 **Activate** the existing **toolpath** 1 to reinstate all the original settings

All you need from the original settings is the original; **Active Workplane**, **Block**, **Rapid Move Heights,** and the ***Tiprad tool*** named **D10t1**.

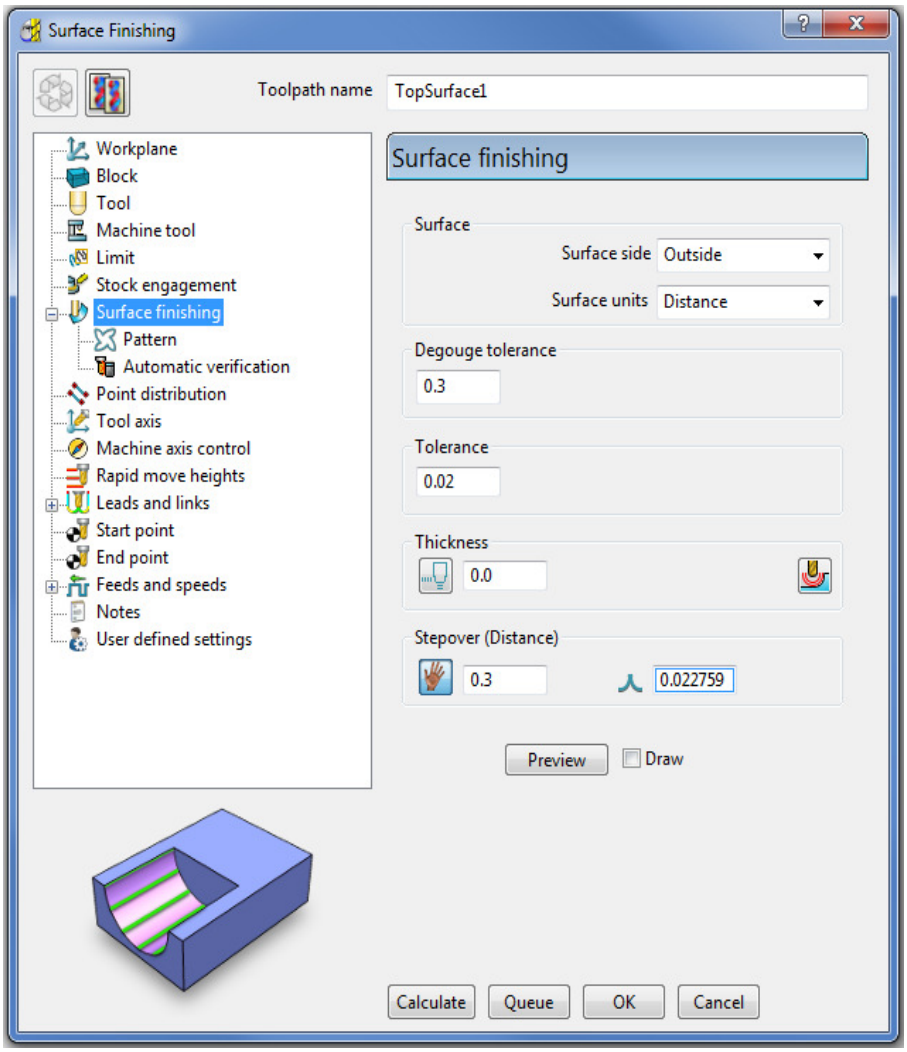


5 De-Activate **toolpath** 1, **Boundary** 1, and **Pattern** 1.

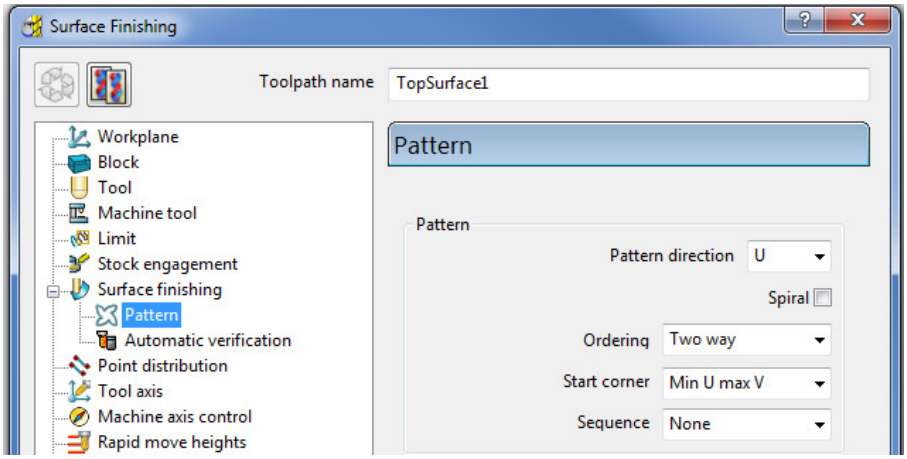
6 Select the ***upper surface*** on the punch model (as displayed above).

Copyright © Delcam plc

7 Open the **Surface Finishing** strategy and enter values as shown below.



8 Select the **Pattern** page:

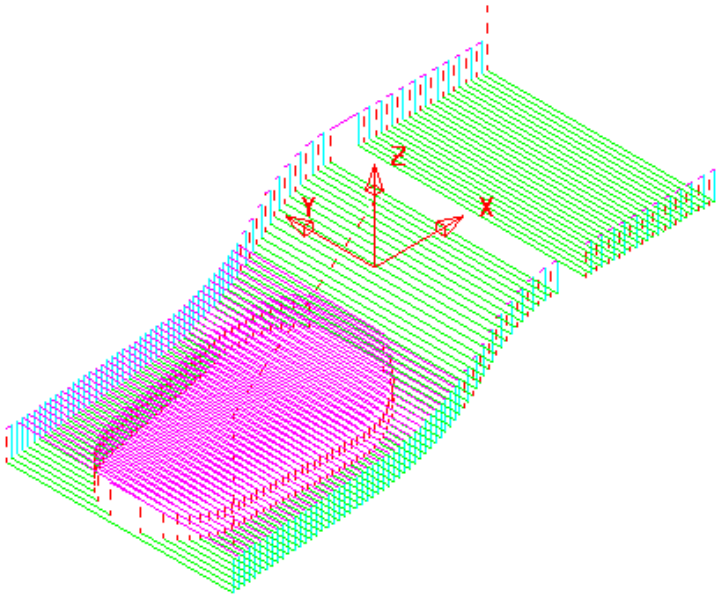


a Set **Pattern direction – U**

b Select - **Two way**

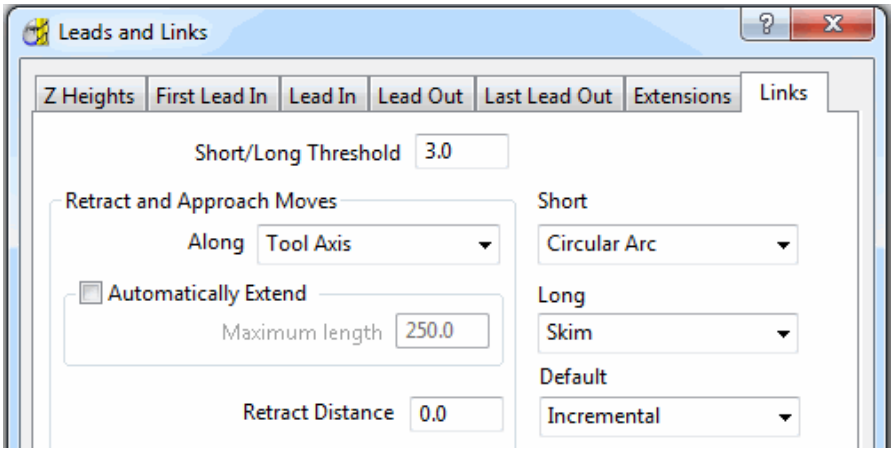
c Select - **Min U max V**

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



10 From the **Main** toolbar, click the **Leads and Links** button.

11 On the **Links** page:



a Enter **Short/Long** - **Threshold** 3.0

b Select **Short** - **Circular Arc**

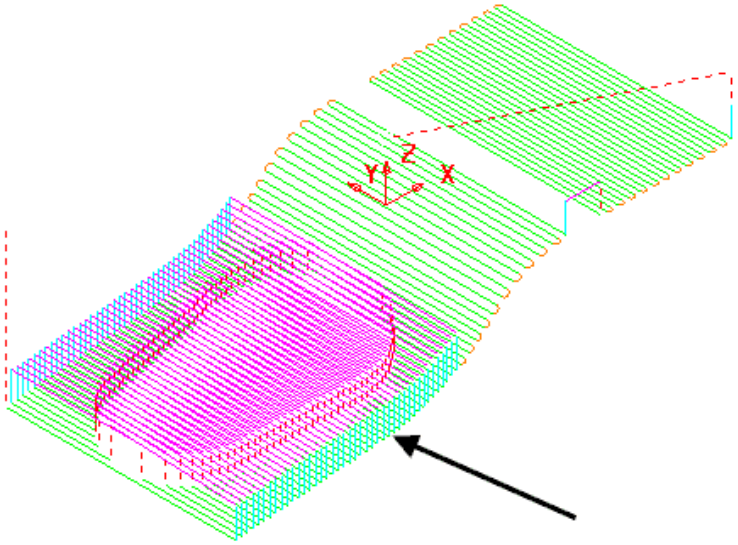
c Select **Long** - **Skim**

d Select **Default** – **Incremental**

12 **Apply** the dialog to update the ***active*** **toolpath**.

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

 Copyright © Delcam plc

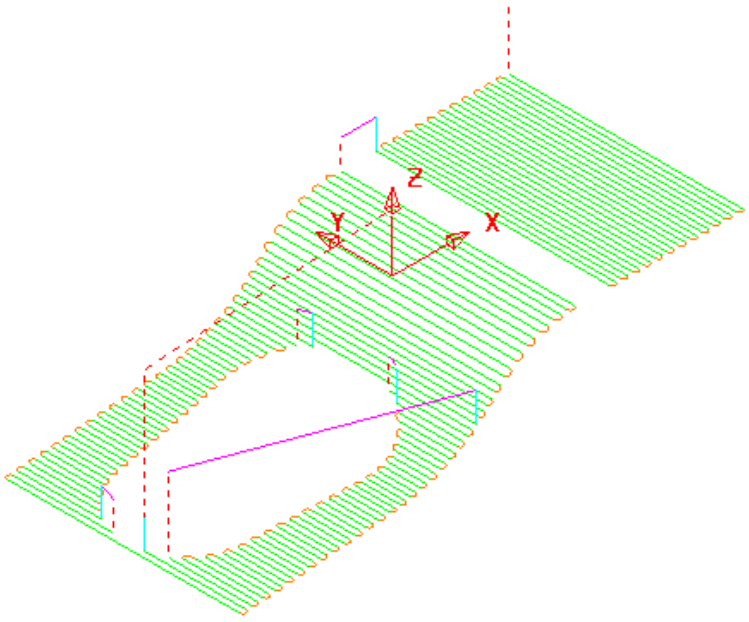


In the resultant toolpath, the natural order of the tool tracks and link moves across the 2 narrow areas is not very efficient.

This can be fixed by applying suitable toolpath **Edit - Reorder** options.

13 Right-click on the **toolpath** and from the local menu, select **Edit > Reorder**.

14 In the **Toolpath Listing** dialog, click the **Automatic reorder and reverse** button.



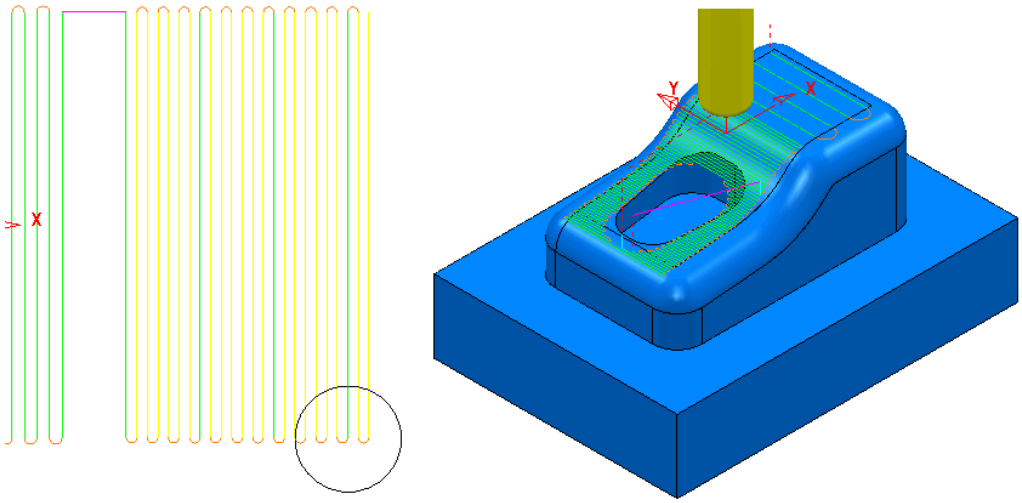
The toolpath follows the curves of the selected surface.

The illustrated toolpath is created with a **1 mm 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 10 mm** flat area of the tool used.

1 From the **Main** menu, select **Draw > Cursor > Tool** to help with the visualisation for the next action.

2 Select groups of 6 tool tracks leaving the single tool track outside each set as shown below:



The **Tool** outline is displayed above to visualise suitable **Stepover** (**Draw > Cursor > Tool**).

3 Right-click the toolpath and select **Edit > Delete Selected Components** to remove the selected tool tracks.

4 From the **Main** menu select **File > Save Project** to update:-

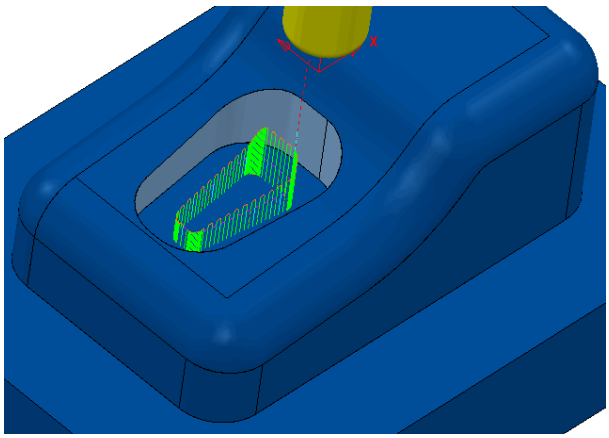
.....\COURSEWORK\PowerMILL-Projects\PunchForm_3

Surface Finishing - Example 2

The direction of the curves on the selected surface dictates whether the tool tracks are aligned along or across the model dialog.

1 **Activate** the **Tip Radiused** tool, **D10t1**

2 Create a **Surface Finishing** strategy with the pocket sidewall surface selected and using a **1 mm Stepover** as shown:

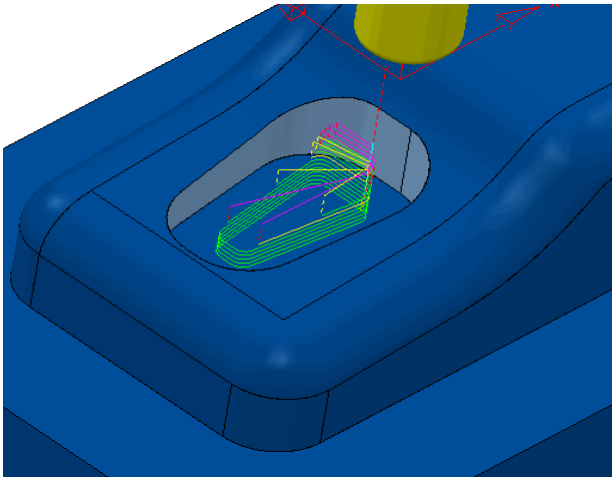


Due to the natural direction based on along or across the surface, the tool tracks run up and down the sidewalls.

This is not the correct choice as it is required for the tool to contour down the sidewalls.

Copyright © Delcam plc

3 **Recycle** the above **Surface Finishing** strategy and change the **Pattern Direction** option from **U** to **V** and then select **Calculate**.



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.

*A **Spiral** option is available on the **Pattern** page of the **Surface Finishing** strategy dialog. This can be applied for a continuous tool track down closed pocket or upstand areas.*

4 From the **Main** menu select **File > Save Project** to update:-

.....\COURSEWORK\PowerMILL-Projects\PunchForm_3

Geometric input of data values into a Strategy dialog

Where applicable, the data input boxes on the **PowerMILL** dialogs can be filled automatically with values measured interactively on the model geometry. These are labelled in **blue text** which when selected will cause the **Measure** toolbar to appear.

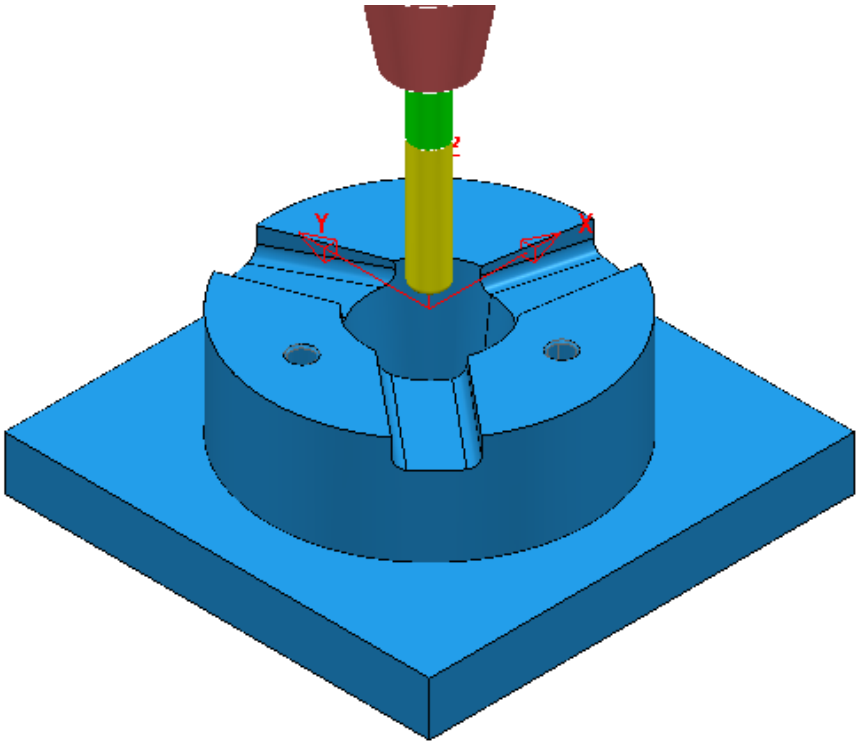
The **Measure** toolbar contains a range of geometric options enabling the user to dynamically trap the relevant dimensional values.

*If the imported **Project** was created in versions earlier than **PowerMILL 2012,** the dialogs for the existing entities will not support geometric input.*

5 In the **main** menu select **File** – **Delete All** and **Tools** – **Reset Forms**.

6 **Open** the read-only **Projec**t:-

.....\PowerMILL_Data\Projects\TriSlot-Billet-Start



7 **Save Project As**:-

.....\COURSEWORK\PowerMILL_Projects\TriSlotBillet-EX1

The **Project** contains all the necessary toolpaths to fully machine the component apart from a **0.5 Axial Thickness** remaining in the 3 Angled slots.

This final operation will be applied using 3 separate **Raster** finishing strategies, each aligned parallel to the slot, sidewalls.

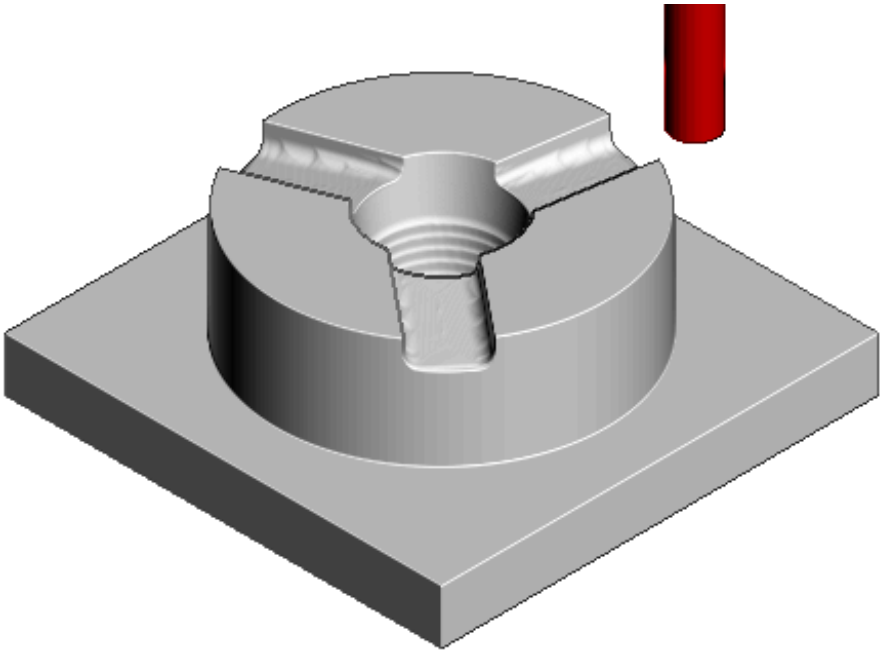
Copyright © Delcam plc

8 Select an **ISO 1** view.

9 Right mouse click on the ***NC Program*** 1 and from the local menu select **Simulate from Start**.

10 Open the **Block** form and **Calculate** the **Block** as ***Defined by*** **- Box** and ***Type*** - **Model**.

11 Run a full **ViewMILL simulation** on the ***NC Program*** 1.

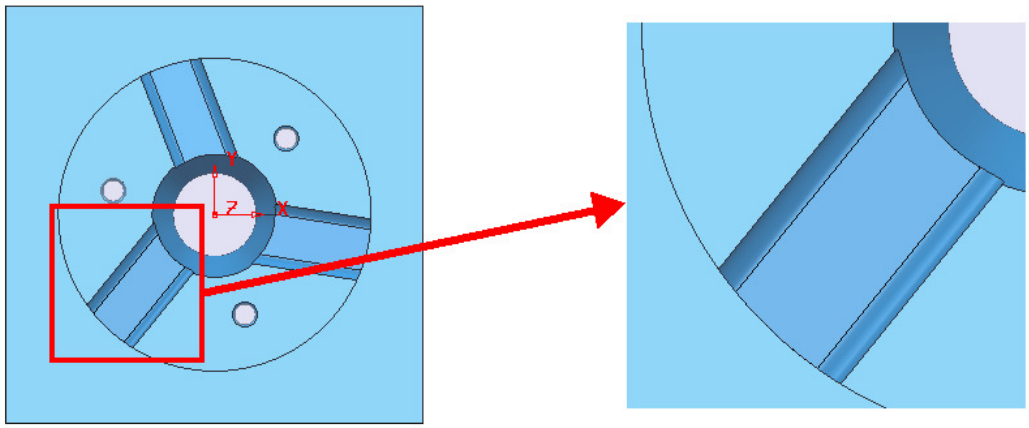


12 **Save** the **ViewMILL** model at this stage.

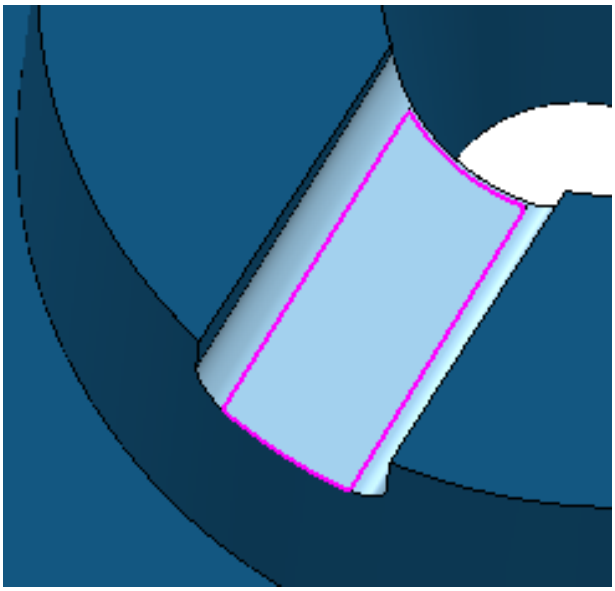
13 Return to the **PowerMILL** (***No Image***) view and disconnect **ViewMILL** from the simulation (Switch the Green Sphere to Red).

14 **Activate** the **BN10** tool.

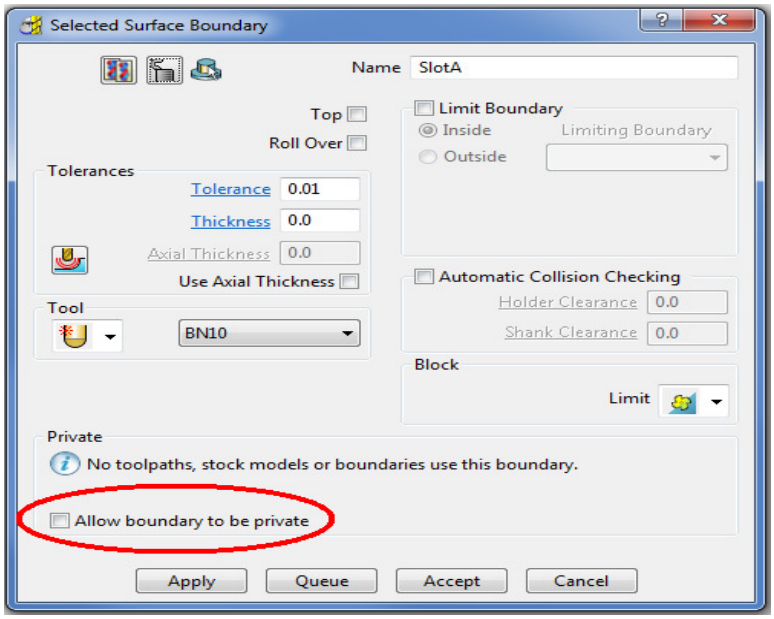
15 Select a **View down Z** and zoom into the area around the lower left slot.



16 Select the **3 surfaces** in the slot defining the base and fillets.



17 In the **PowerMILL *explorer*** right mouse click on **Boundaries** and from the local menu left mouse click **Create Boundary - Selected Surface**.



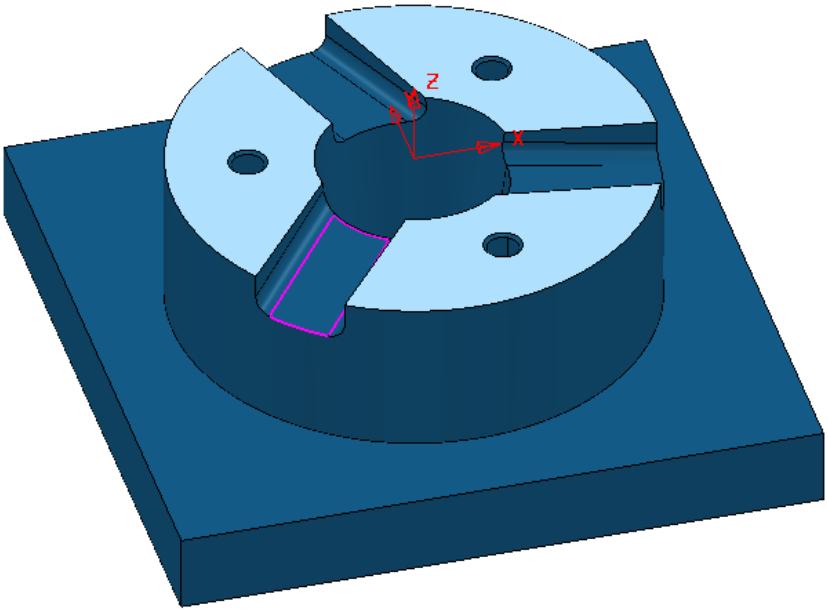
18 Input the **Name** SlotA, input the **Tolerance** 0.01, **Thickness** 0 and un-tick **Allow boundary to be private**, before clicking on **Apply**.

This **Boundary *SlotA*** (as shown in diagram at the top of the page) will be used to limit the final **Raster Finishing** strategy to the required area within the Slot.

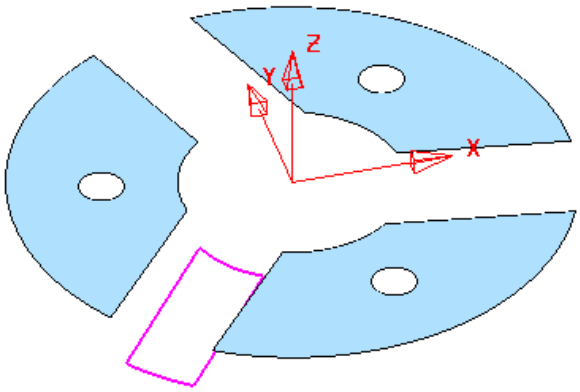
The **Raster Finishing** strategy will be arranged to run parallel with the slot the exact angle being obtained dynamically using the ***Geometric Input*** option.

19 Select the 3 flat upper faces of the **model** to provide an accurate reference, on a fixed Z height for dynamically extracting ***Angular*** geometric values.

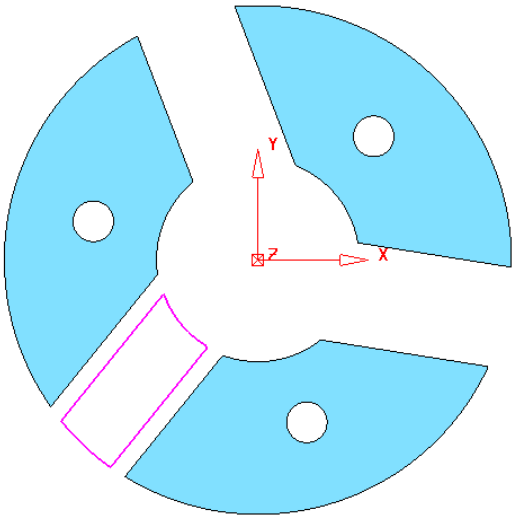
Copyright © Delcam plc



20 Use the keys, **Ctrl K** (Blank Except) to remove all but the selected surfaces from the view.



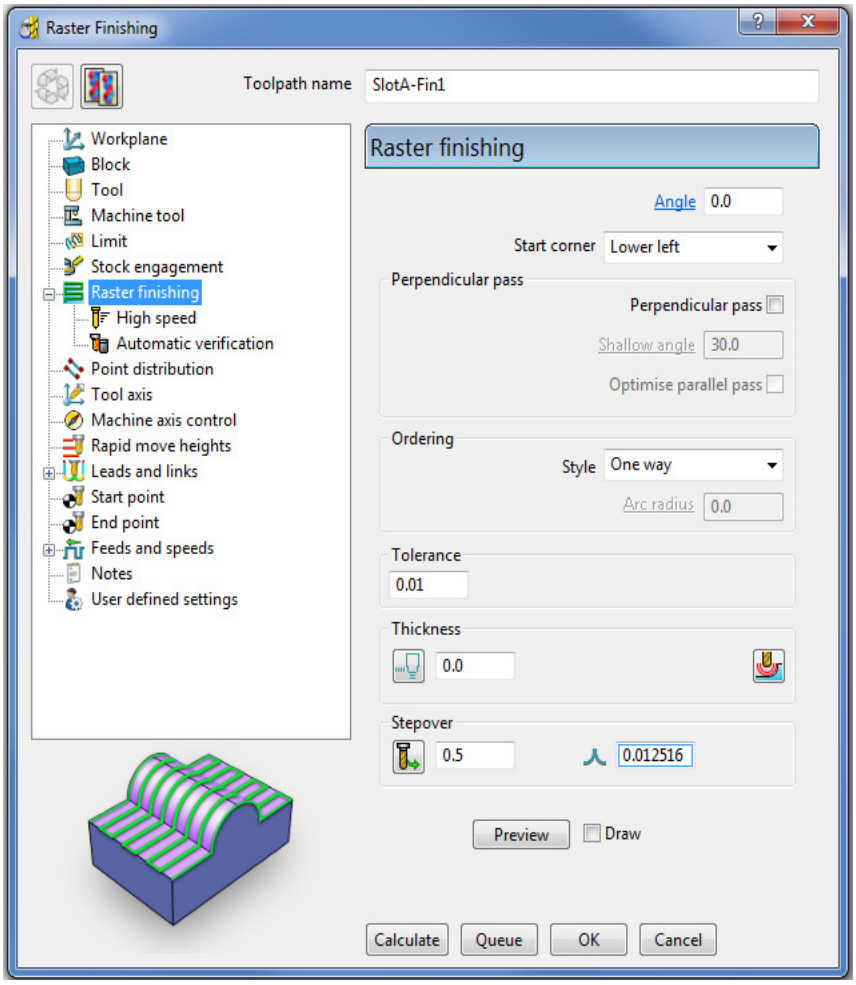
21 Select a **View down Z**.

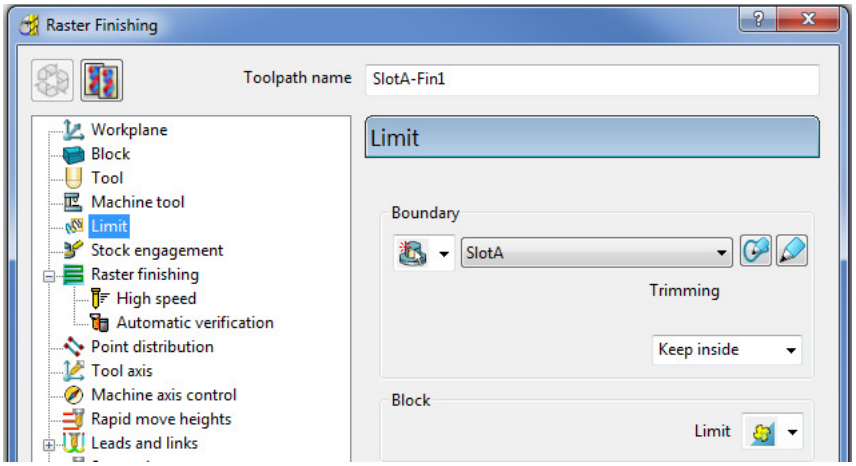


22 On the **Main Toolbar**, click to display the **Toolpath Strategies** dialog.

23 On the **Strategy Selector**, click the **Finishing** tab and select the **Raster Finishing** strategy.

24 Enter data into the displayed pages of the form exactly as shown below.

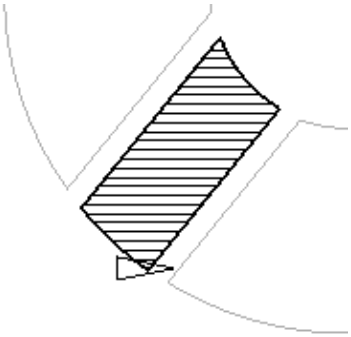




25 Select a **Wireframe** view for the displayed **Model**.

Copyright © Delcam plc

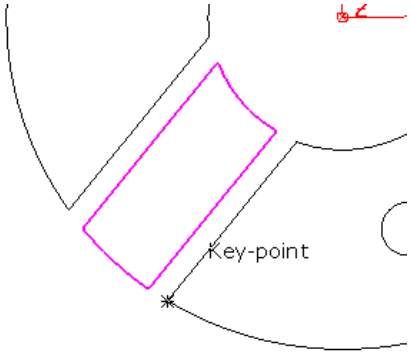
26 Select **Preview** to display the current ***Pattern*** for the **Raster Strategy**.



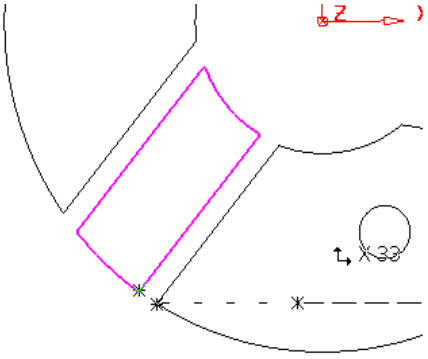
27 Back on the **Raster Finishing** main page click on the blue text, **Angle** beside the associated input box (Top right of dialog).

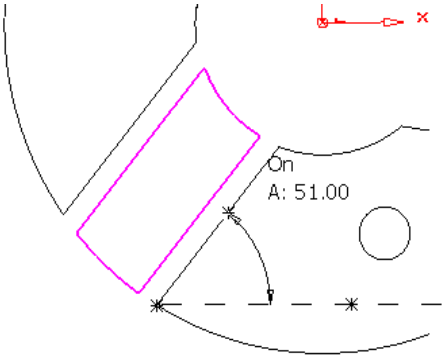
The **Measure** mode toolbar will appear.

28 In the toolbar select the **Minor Angle** option.



29 Use the ***cursor*** to locate the **Key-point** at the base of the angle and 'snap' with the left mouse key to define the 1st point of the Angle.



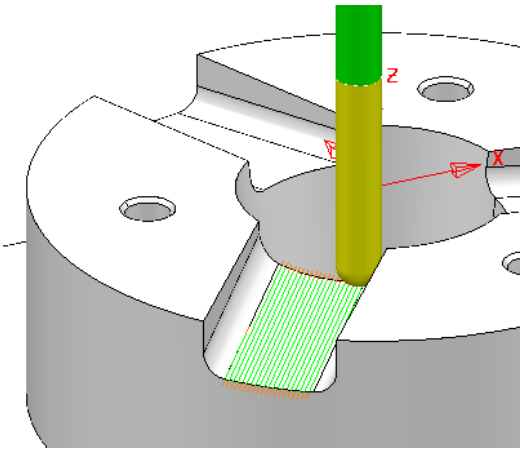


30 Hover the ***cursor*** on the **key-point** again and drag the cursor to the right to pick up the ***horizontal angle***, and 'snap' a 2nd point along this at a suitable distance.

31 Move the ***cursor*** to a suitable position along the line running at an angle from the original **key-point** and 'snap' the 3rd point to obtain the required ***Angle*** value.

The ***Geometric Input*** value obtained dynamically is automatically inserted into the **Angle** input box.

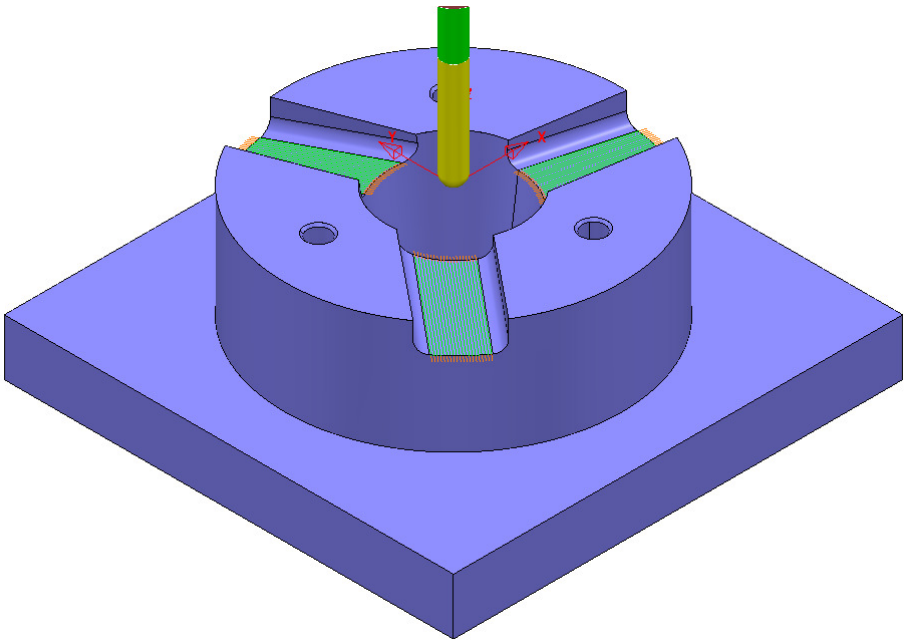
32 **Calculate** the ***Raster Finishing*** strategy and then **Close** the dialog.



*The **Measure - Mode Toolbar** will close automatically when the toolpath starts to process.*

EXERCISE

33 Create similar ***Selected Surface*** **Boundaries** (named **SlotB** and **SlotC**) and **Raster Finishing** strategies (named **SlotB-FIN1** and **SlotC-FIN1)** for the other 2 Slots.



34 Select **File - Save Project** to update:-

.....\COURSEWORK\PowerMILL_Projects\TriSlotBillet-EX1

 Copyright © Delcam plc

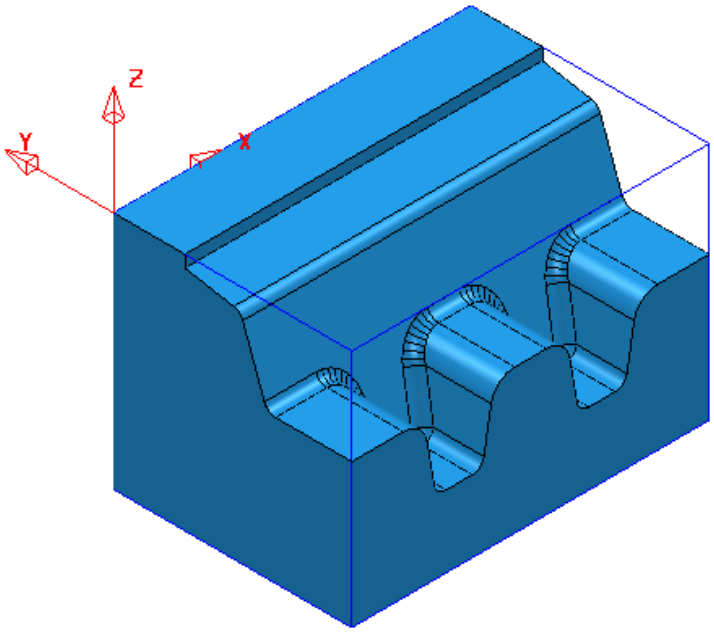
Toolpath Z-Limits

All **Toolpath** types, except drilling, can now be limited to applied maximum and minimum **Z** values within the defined **Block**.

1 In the main menu select File – Delete All and Tools – Reset Forms.

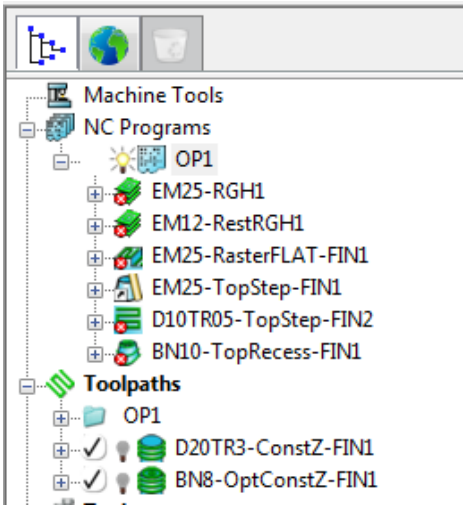
2 Open the read-only Project:-

.....\PowerMILL_Data\Projects\Zlimits-Start



3 **Save Project As**:-

.....\COURSEWORK\Zlimits-EX1



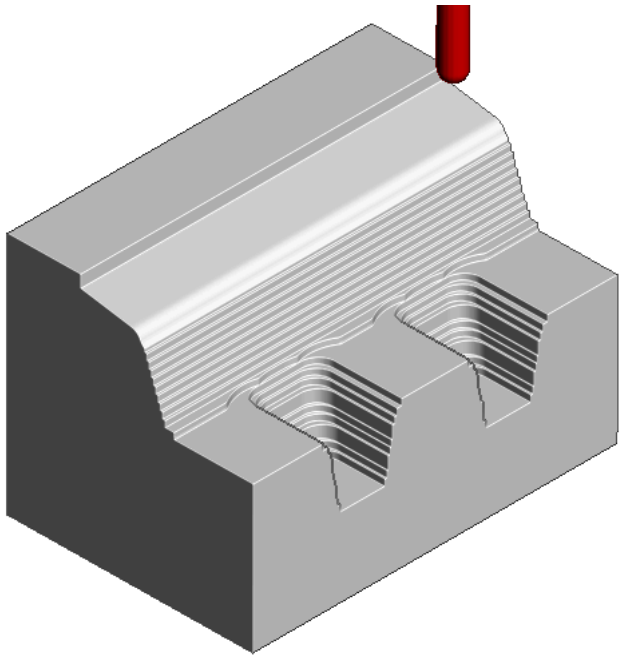
The **Project** already contains 7 **Toolpaths**, 5 of which have been put in a sub-folder named **OP1**. The 2 remaining **Toolpaths** require modification as they are both duplicating areas already included in other strategies. This will become evident during a ***ViewMILL* simulation**.

4 **Calculate** a **Block** to the **Model** dimensions.

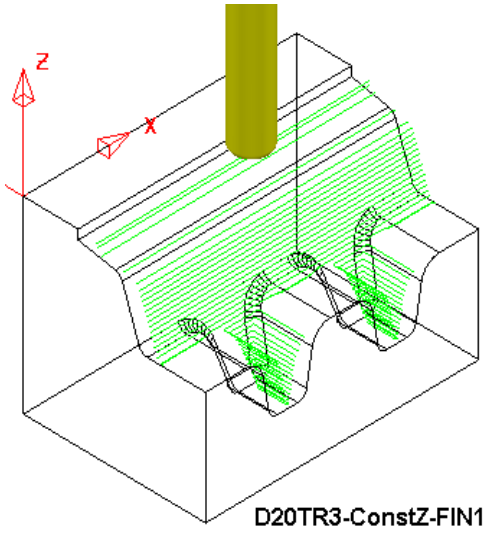
5 **Calculate** the **Rapid Move Heights** using default settings.

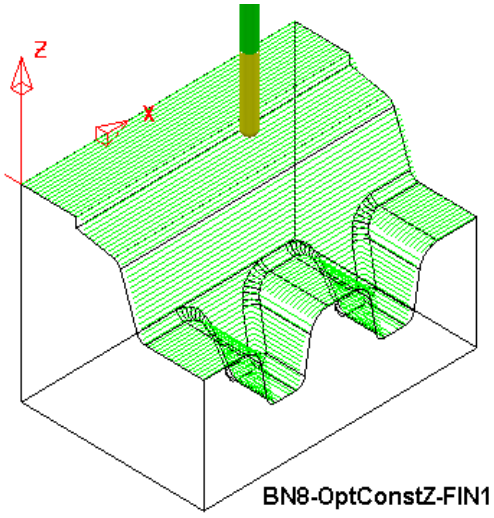
6 Run a ***ViewMILL* simulation** on the existing **NC Program** OP1.

7 Once completed save the **ViewMILL** model at this stage.



The ***angled face*** and ***lower recesses*** have yet to be finish machined. The 2 remaining toolpaths both operate over the full extent of the **Block**.





D20TR3-ConstZ-FIN1 needs to be localised to operate down the **angled face** to the top of the lower recesses.

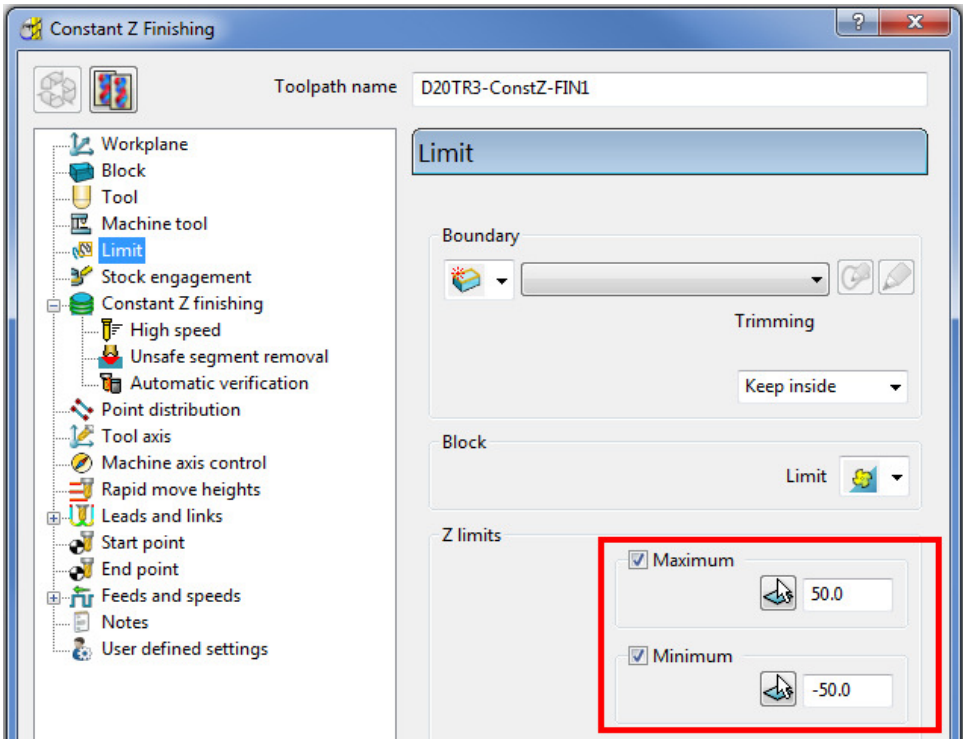
BN10-OptConstZ-FIN1 needs to be localised to operate inside the 2 **lower recesses**.

This will easily be achieved for both toolpaths within the strategy dialog using the **Z-Limits** option.

 Copyright © Delcam plc

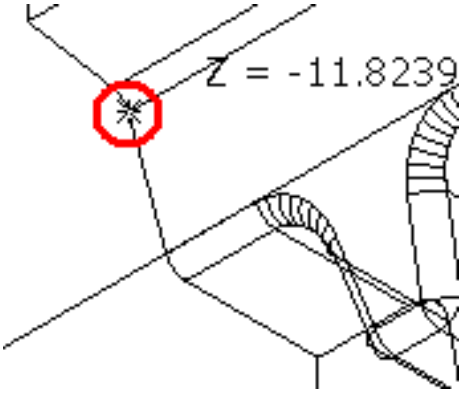
8 Return to the **PowerMILL** (***No Image***) view and disconnect from the simulation (Switch the Green Sphere to Red).

9 In the **PowerMILL** ***explorer*** Right mouse click on the toolpath **D20TR3-ConstZ-FIN1** and from the local menu, select **Activate** followed by **Settings** to open the strategy dialog.



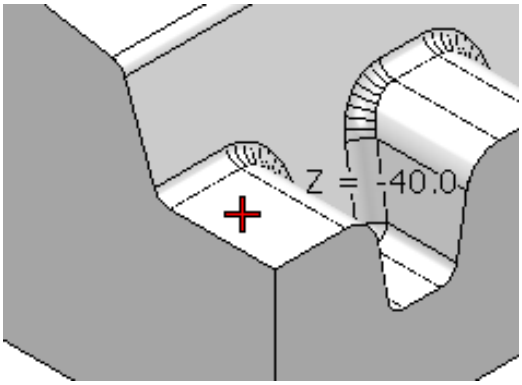
10 Select the ***Recycle*** option followed by the **Limit** page and ***tick*** both the **Maximum** and **Minimum** boxes.

11 For The **Maximum** select ***Pick the highest Z Value*** option.



12 Snap the ***key-point*** (shown above) at the top corner of the angled face to collect and insert the value **-11.82394**.

13 For The **Minimum** select ***Pick the lowest Z Value*** option.

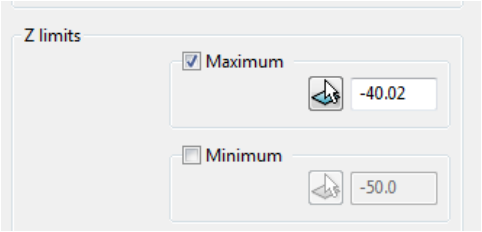


14 Snap the ***key-point*** (shown above) on the top of the flat area to collect and insert the value **-40**.

15 Back in the dialog select **Calculate** to re-process the updated **Constant Z** finishing strategy and then **Close** the dialog.

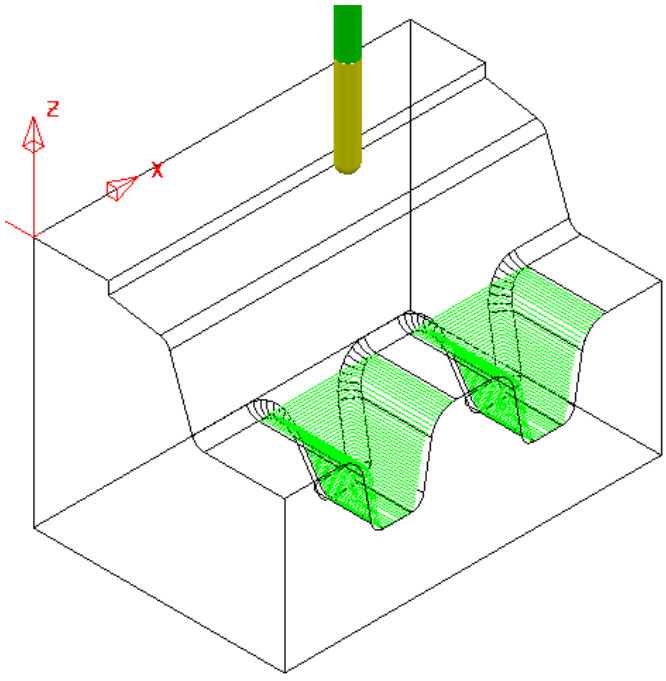
16 In the **PowerMILL *explorer*** Right mouse click on the toolpath **BN8-OptConstZ-FIN1** and from the local menu, select **Activate** followed by **Settings** to open the strategy dialog.

17 Select the ***Recycle*** option followed by the **Limit** page and this time ***tick*** just the **Maximum** box.



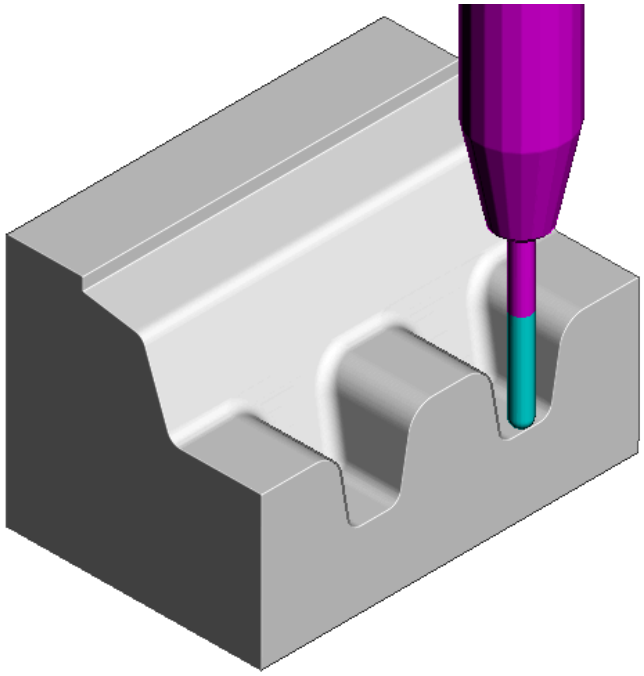
18 Input the value **-40.02** in the **Maximum** box.

19 Back in the main form select **Calculate** to re-process the updated **Optimised Constant Z** finishing strategy and then **Close** the dialog.



 Copyright © Delcam plc

20 Re-connect **ViewMILL** and continue the existing simulation with the 2 updated toolpaths; **D20tR3-ConstZ-FIN1** followed by **BN8-OptConstZ-FIN1**.



The Component is now fully machined.

21 Select **File** – **Save Project** to update:-

.....\COURSEWORK\PowerMILL_Projects\Zlimits-EX1

Toolpath Processing Options

Calculate

When all the settings and values are input ready to create a toolpath, the **Calculate** button can be clicked to start the toolpath calculation process.

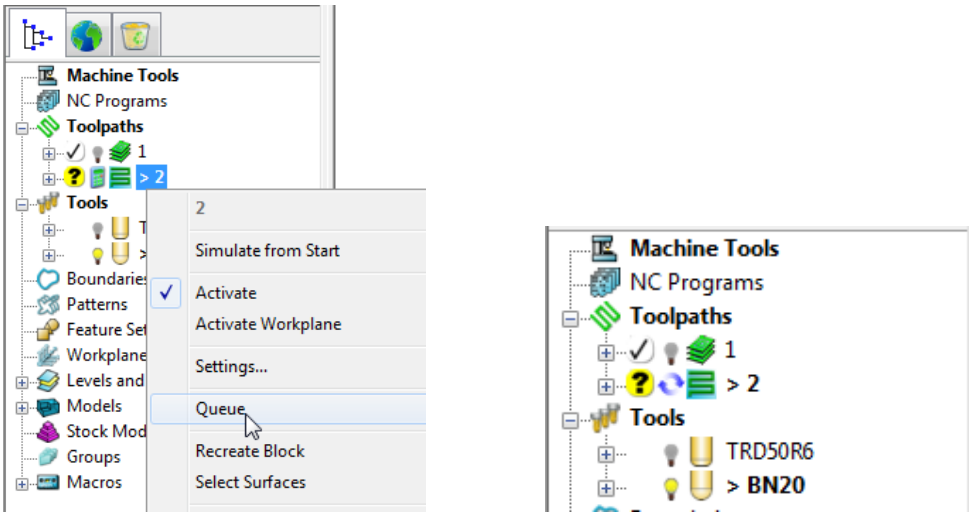
*You cannot use other **PowerMILL** options while the toolpaths are being calculated.*

Queue

If the **Queue** option is applied the user can continue working in **PowerMILL** while the toolpath is processed in the background. Several strategies can be put in the queue to be calculated in the background, in sequential order.

To **Queue** a toolpath for calculation, you can, either:

- Click the **Queue** button in a strategy dialog, or,
- Right-click on a toolpath and select **Queue** from the local menu.



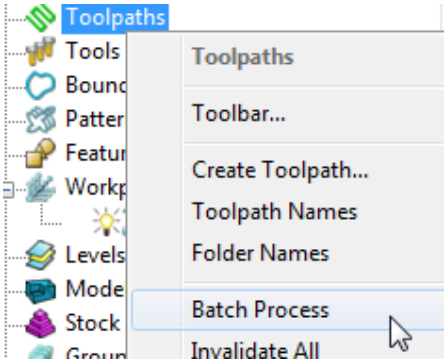
*The '**Calculator**' symbol changes to the '**Queue**' symbol when the toolpath is queued.*

Batch Process

Once all data has been entered into a strategy dialog, click the **OK** button to save the settings and close as an un-processed toolpath.

You can save one or more un-processed, machining strategies to be sequentially calculated at a later time.

To **Batch Process** toolpaths, right-click a toolpath and select **Batch Process**.



Toolpath Processing Queue

The **Toolpath** - **Queue** option allows you to continue working in **PowerMILL** while the toolpath is being processed in the background. One or more strategies can be added to the **Queue** and they are calculated in the background, in sequential order.

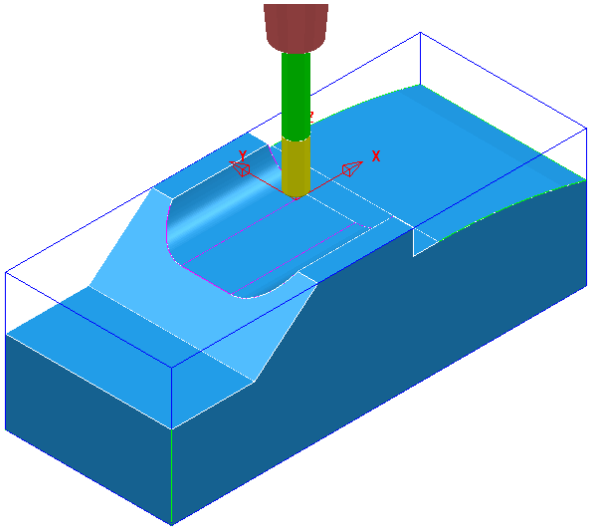
The following 'start up' **Project** contains two unprocessed toolpaths. The dialog for each one will be opened in turn, but **Queue** will be used instead of **Calculate**. This enables you to continue work on a new **Corner finishing - Along** strategy while the original toolpaths are processed in the background.

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

2 Select **File – Open Project**:

.....\PowerMILL_Data\Projects\ToolpathProcessing-Start

 Copyright © Delcam plc

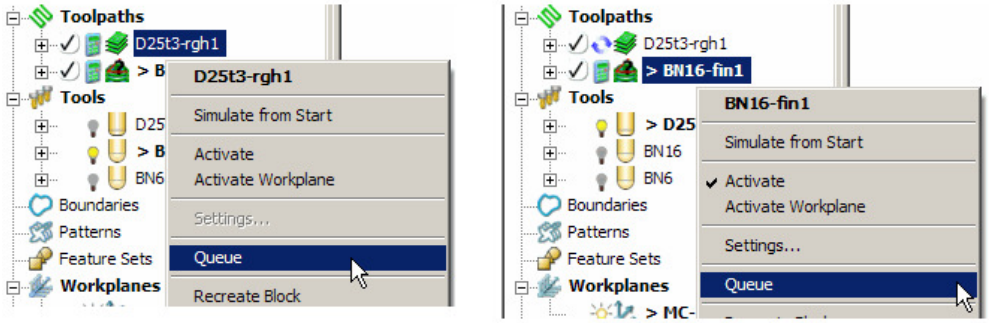


3 Select **File - Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\ToolQueue-ex1

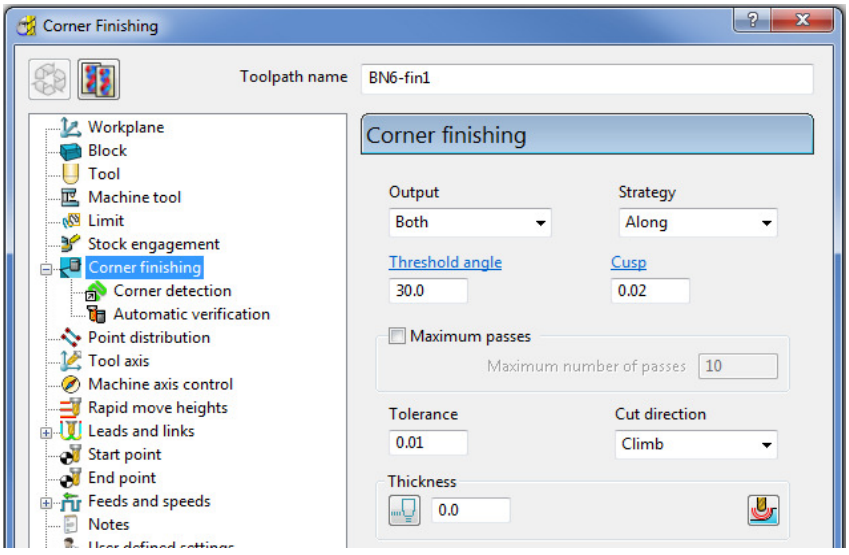
4 In the **PowerMILL *explorer***, right-click the **toolpath** named **D25t3-rgh** and select **Queue**.

5 Repeat this on the **Toolpath** named **BN16-fin1**

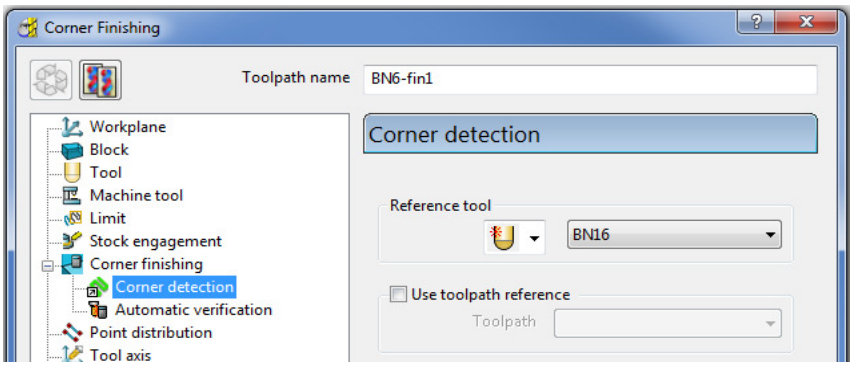


6 While the original toolpaths are being processed in the background, **Activate** the tool **BN6**.

7 Open the **Corner Finishing** strategy and set the ***Strategy*** to **Along**. Enter the other values exactly as shown below.



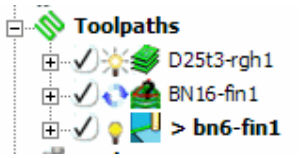
8 Select the **Corner detection** page and select the **Reference tool** as **BN16**

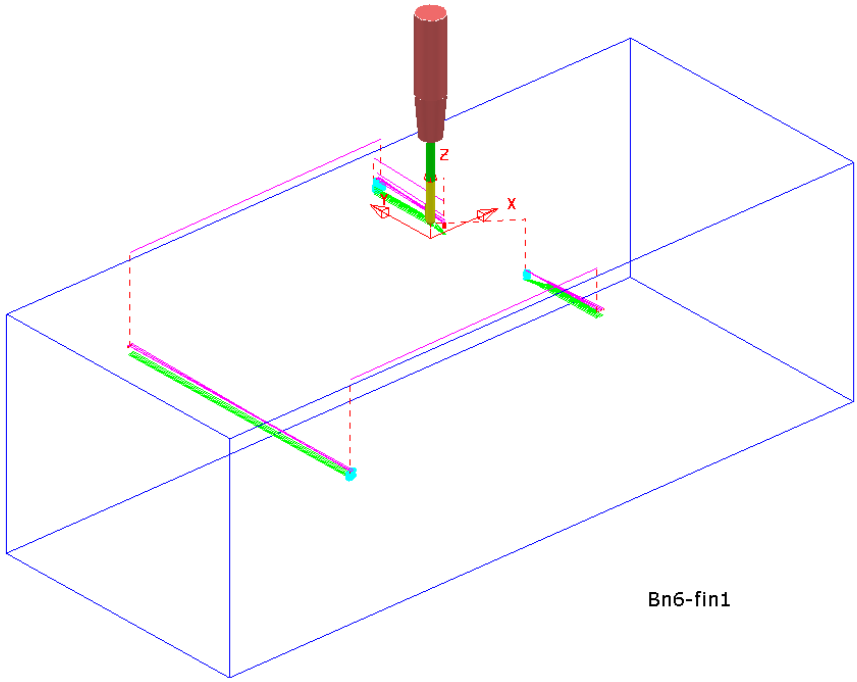


9 Click **Calculate** to process the toolpath and then **Close** the dialog.

*The latest **Corner Finishing** strategy starts processing immediately and is likely to be completed before the **Steep and Shallow finishing** strategy currently pending in the **Queue**.*

When a toolpath is added to the **Queue**, and is in an unprocessed status, the ***Calculator symbol*** on the Toolpath in the **PowerMILL *explorer*** changes to the **Queue** symbol .





10 Select **File - Save Project** to update:

.....\COURSEWORK\PowerMILL-Projects\ToolQueue-ex1

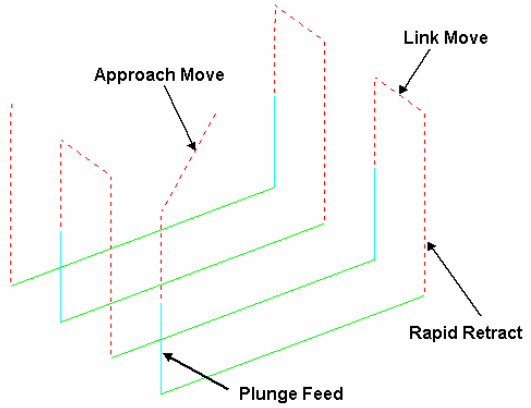
 Copyright © Delcam plc

5. Toolpath Leads & Links

If a tool is allowed to start machining directly at the start of a tool track, it first plunges down into the remaining stock, and then suddenly changes direction to run along the tool track. This is likely to result in machining marks, vibration, and excess wear on both the tool and machinery.

To avoid sudden loading on the tool, appropriate **Lead** moves (at cutting feedrate) on and off tool tracks can be applied.

Air moves (**Link**) between individual tool tracks can add significant amount of extra time to a machining operation. This can be greatly reduced by applying alternative, Link move options.



Default Leads and Links

Click on the **Main** toolbar to display the **Leads and Links** dialog and options. You can also access the **Leads and Links** dialog from within a strategy dialog. Note that **Leads and Links** can be retrospectively applied to an existing toolpath.

Z Heights

Skim and Plunge distance - Provides variable control of rapid move heights within a component. These operate in conjunction with Safe Z and Start Z to minimise slow and unnecessary air moves while machining the component form.

Skim distance – An incremental distance above the model where rapid feed moves occur from the end of one tool track to the start of the next. The tool traverses at a high feed rate 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 feed move.

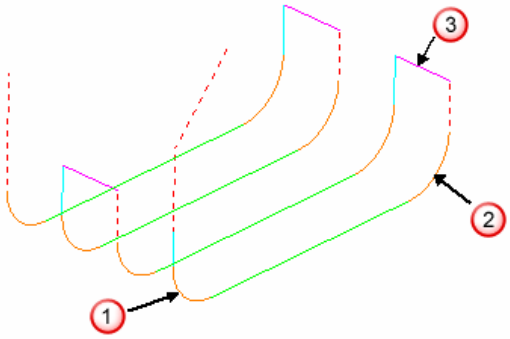
Copyright © Delcam plc

Lead In/Lead out Moves

Lead In controls the tool movement onto the start of a tool track and **Lead Out** controls the movement away from the end of a tool track.

Available **Lead In** and **Lead Out** moves are:

- None
- Surface normal arc
- Vertical Arc
- Horizontal Arc
- Horizontal Arc Left
- Horizontal Arc Right
- Extended Move
- Boxed
- Straight
- Ramp
- Pocket centre



In the above illustration:-

1 - **Lead In** = **Vertical Arc**

2 - **Lead Out** = **Vertical Arc**

3 - **Link move** = **Skim**

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 must be gouge protected. **Gouge check** is set as default and any **Lead** that would result in a gouge is not created.

On tooltracks where the **1st Choice** cannot be applied, then the **2nd Choice** is used. If it is not possible to apply either then the **Lead** options are applied as **None**.

The current **Leads and Links** settings are automatically applied when creating new machining strategies.

Copyright © Delcam plc

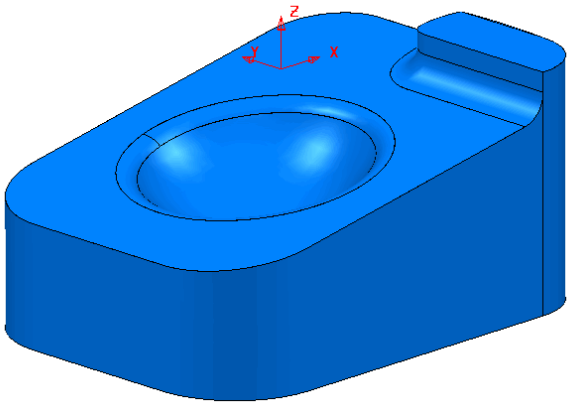
Example

The following example illustrates the retrospective addition of suitable Leads and Links to finishing toolpaths. The project contains one Roughing toolpath and three Finishing toolpaths.

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

2 Select **File - Open Project**:

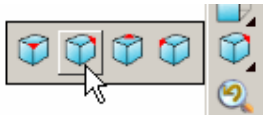
…\PowerMILL_Data\Projects\LeadsLinks-Start



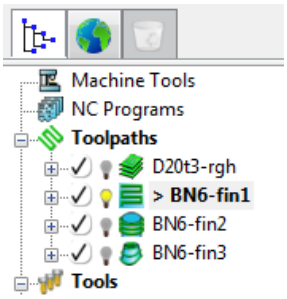
3 Select **File - Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\LeadsLinks-example

4 Select an **ISO2** view.

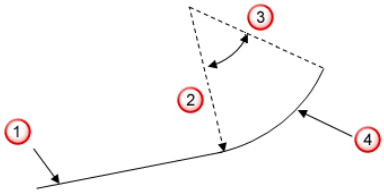


5 **Activate** the toolpath, **BN6-fin1**.



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, with the **Gouge check** option selected, the additional moves are not made if a gouge might occur.



1 - Toolpath segment

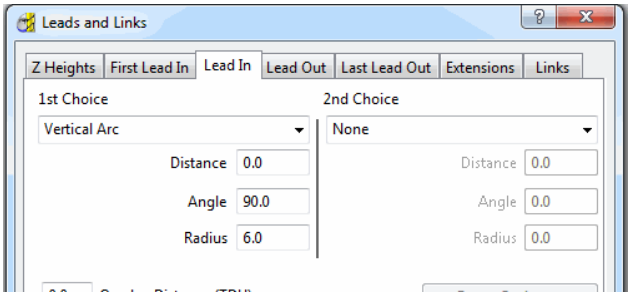
2 - Radius

3 - Lead

4 - Tangent angle

1 On the **Main** toolbar, click to open the **Leads and Links** dialog.

2 On the **Leads and Links** dialog, select the **Lead In** tab and:

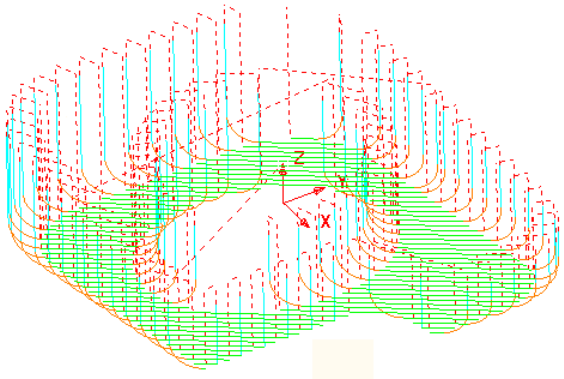


a Select **Vertical Arc** from the **1st Choice** list.

b Enter an **Angle** of 90.0 and a **Radius** of 6.0

c Click **Apply**.

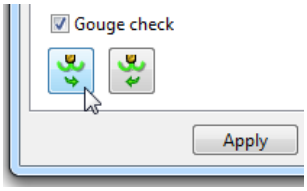
There is now a **Vertical Arc** leading onto most of the tool tracks.



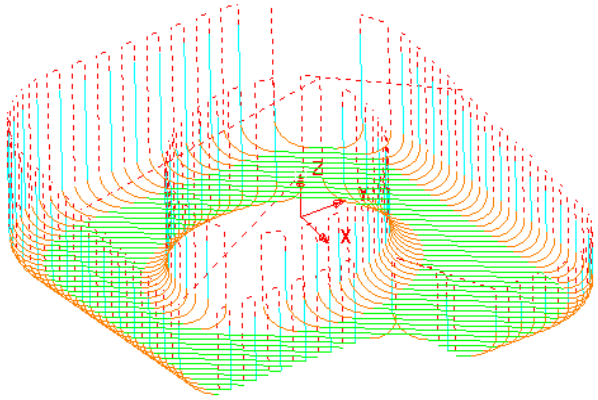
*The new (gouge protected) **Vertical** leads have not been created on the tool tracks that stop close to a raised area of the model.*

Copyright © Delcam plc

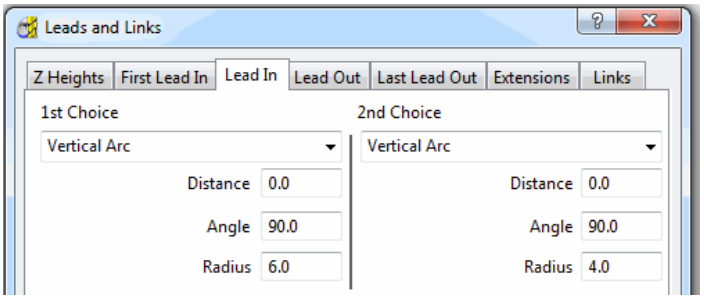
3 Click the **Copy to Lead Out** button.



4 Click **Apply** to update the active toolpath.



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

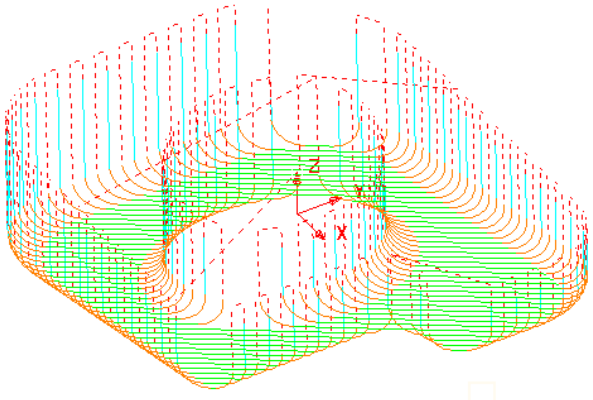


5 Select **Vertical Arc** from the **2nd Choice** list.

a Enter an **Angle** of 90.0 and a reduced value **Radius** of 4.0.

b As before, click the **Copy to Lead Out** button.

c Click **Apply**.



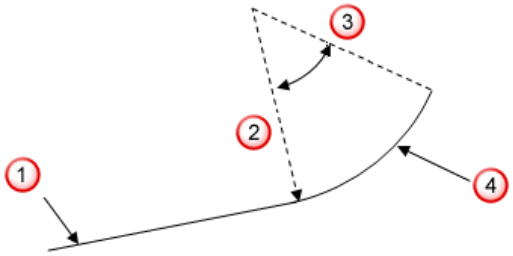
Wherever it is gouge free on the above toolpath a **1st Choice** of **Lead In/Out** with a **Vertical Arc** is applied.

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 is 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 the Vertical Arc. This type of lead is frequently used for toolpaths running at constant Z, or with only small changes in Z height.



1 Toolpath segment

2 Radius

3 Lead

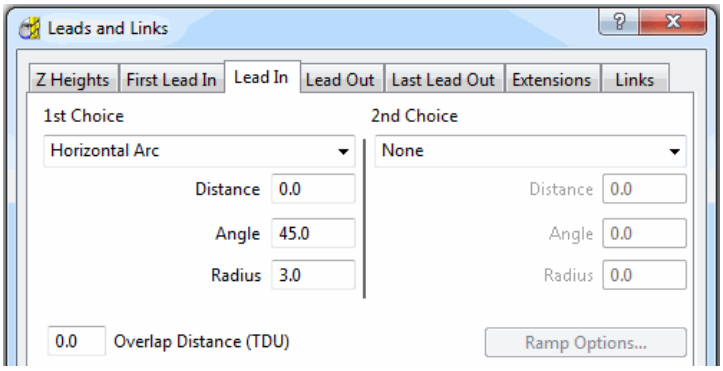
4 Tangent angle

1 From the **Main** menu, select **Tools - Reset Forms** to return the Leads and Links settings to default.

2 In the **PowerMILL *explorer***, right-click the **BN6-fin2** toolpath and select **Activate**.

3 Select an **ISO1** View.

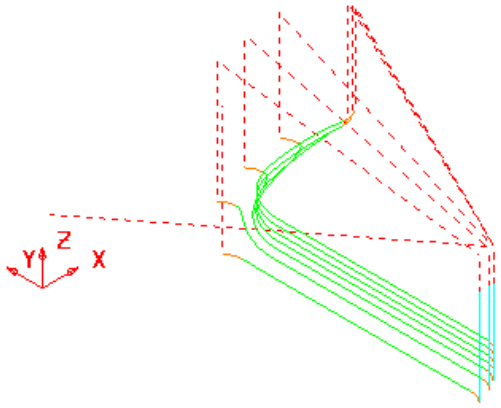
4 Click on the **Main** toolbar to open the **Leads and Links** dialog.



5 Select the **Lead In** tab, and change **1st Choice** to **Horizontal Arc** with an **Angle** of 45 and a **Radius** of 3.0

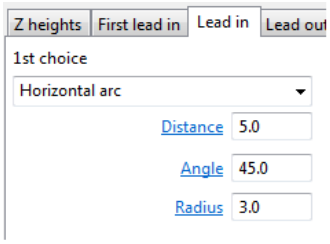
6 Click **Copy to Lead Out** and then **Apply**.

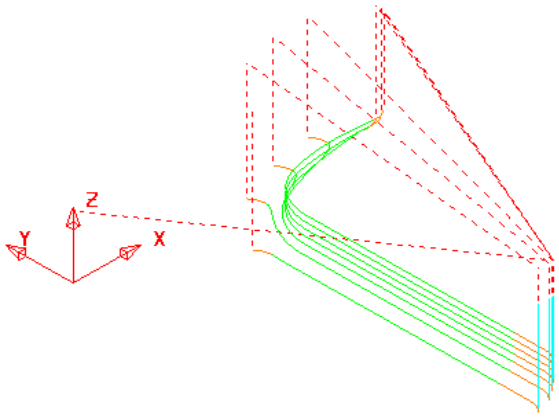
 Copyright © Delcam plc



*There is no need to utilise the **2nd choice** options as the **1st Choice** of **Horizontal Arc** - **Leads In/Out** have appeared on all the tool tracks.*

7 In the existing **1st Choice** - **Lead In** page input a value of **5** for the **Distance** and select **Apply**.





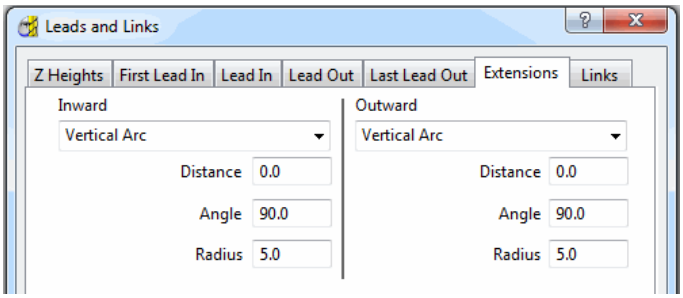
*An additional (**Distance**) extension move is inserted before the start of the tool track and after the original **Lead in** - **Vertical Arc**.*

Extensions

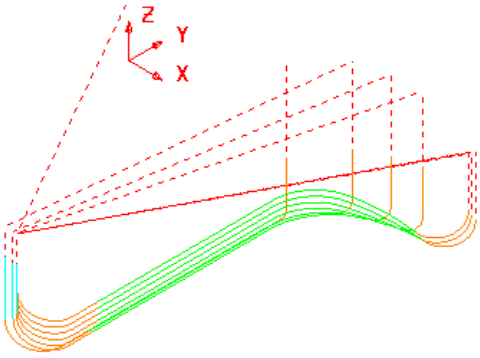
Extensions are additional lead options that are applied outside the existing **Lead In/Out** moves.

1 Select an **ISO2** view.

2 In the **Leads and Links** dialog, select the **Extensions** tab and for both the **Inward** and **Outward** options, enter a **Vertical Arc** with **Angle** 90.0 and **Radius** 5.0



3 Click **Apply**.



A **Vertical Arc - Extension** is added to all of the **Lead In/Out** moves.

Links

A **Link** move is the movement of the tool from the end of a tool track to the start of the next. For efficient tool moves across the model, you can reduce the height of the **link** moves 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 by the **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**.
- For **Default** (first and last) links the available options include **Safe Z**, **Incremental**, and **Skim**.
- **Arc fit rapid moves** can be applied, if compatible with a machine control system.

Copyright © Delcam plc

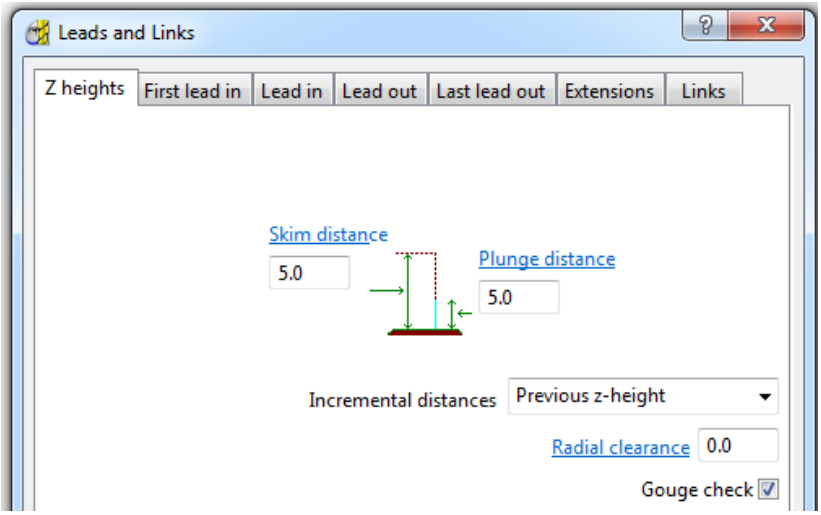
1 From the **Main** menu, select **Tools** - **Reset Forms** to reset the **Leads and Links** settings to the factory defaults.

2 **Activate** the toolpath bn6-fin1.

3 Select an **ISO 2** view.

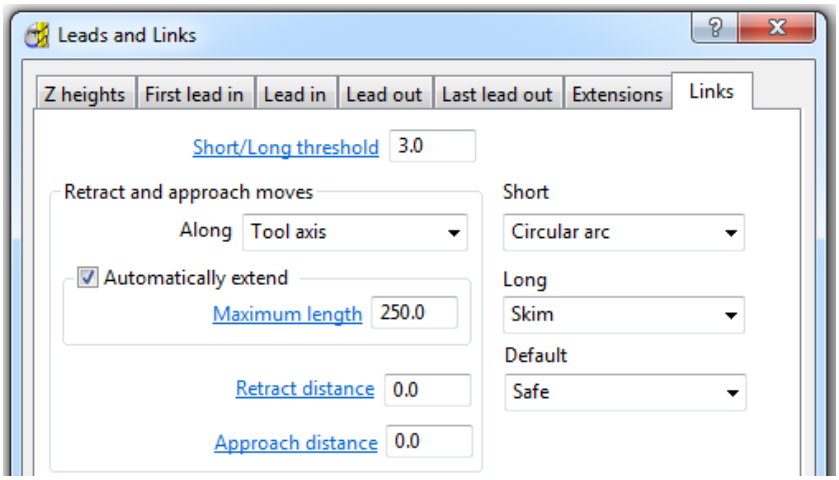
4 Click and open the **Leads and Links** dialog.

5 Select the **Z Heights** tab to access the ***Skim*** and ***Plunge*** **distance** options.



- **Skim Distance** - is the incremental height at which the tool clears the highest point on the **model** between tool tracks.
- **Plunge Distance** - is the Incremental height above the **model** at which the tool rapids down to before using the **Plunge Feed Rate**.

6 Select the **Links** tab.

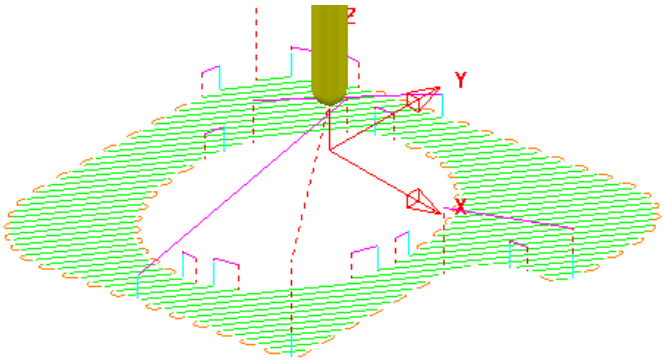


a Enter the **Short/Long threshold** as 3

b Link move distances less than this value will use the **Short** options and those greater will use the **Long** options.

c Select ***Short*** as **Circular** and ***Long*** and ***Default*** as **Skim**.

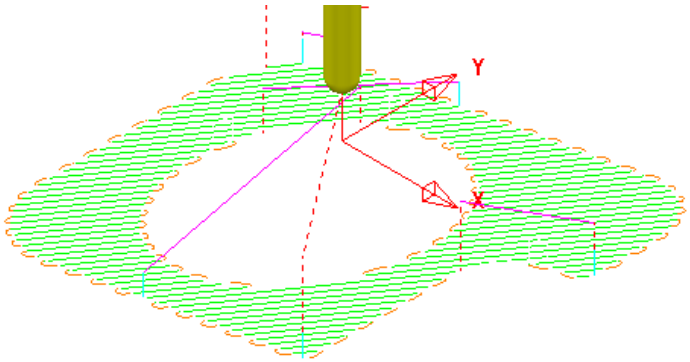
7 Click **Apply**.



*The distance between the ends of adjacent tool tracks is greater than the **Short/Long threshold** value in several areas of the toolpath (In these instances **Skim** moves apply).*

8 Change the **Short/Long threshold** to 5

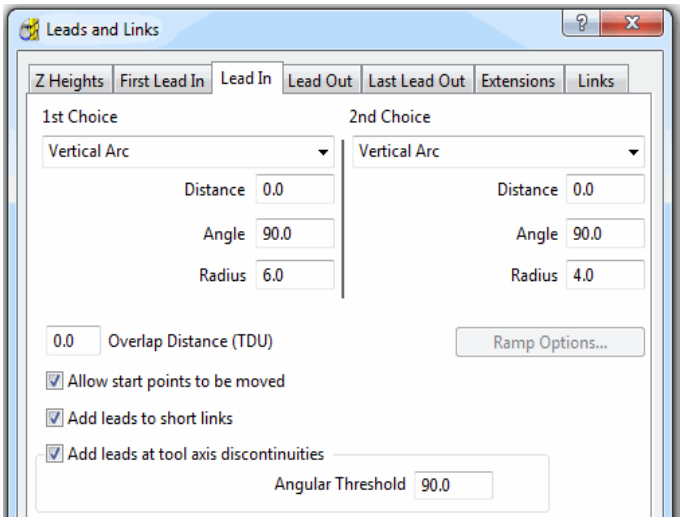
9 Click **Apply**.



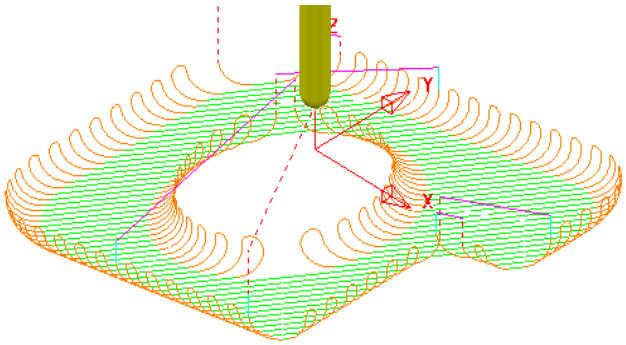
*By increasing the value of the **Short/Long threshold**, the number of occurances of **Short Link - Circular Arcs** has increased.*

The remaining **Link** moves would benefit from **Vertical Arc** - **Lead In/Out** moves, but not on the existing **Circular Arc** Links.

10 **Apply** a **Vertical Arc** to both **Lead In** and **Lead Out** as shown:

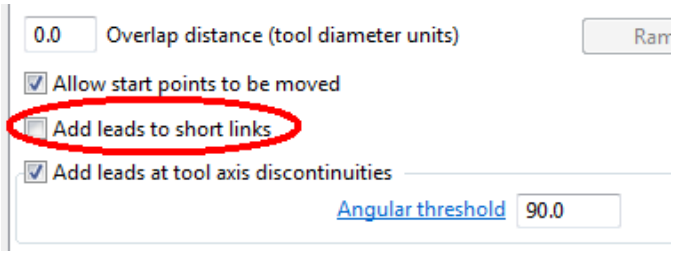


 Copyright © Delcam plc

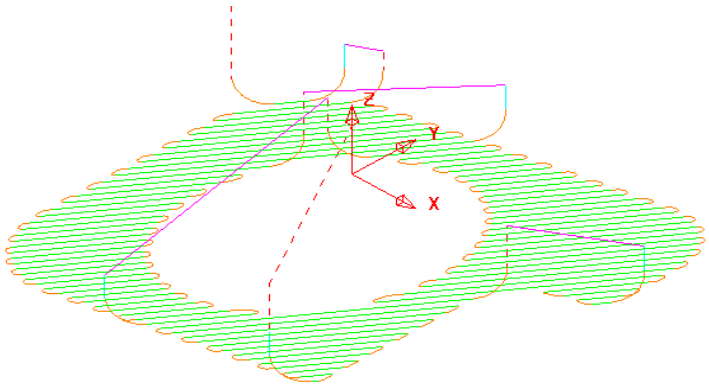


*The **Vertical Arc - Leads** are also added to the **Circular Arc - Short** link moves. This can be prevented by un-ticking the **Add leads to short links** option.*

11 Un-tick the **Add leads to short links** option.



12 Click **Apply**.



The **Lead** moves exist on the **Long links** but not on the **Short links**.

Applying Leads and Links locally to individual tool tracks

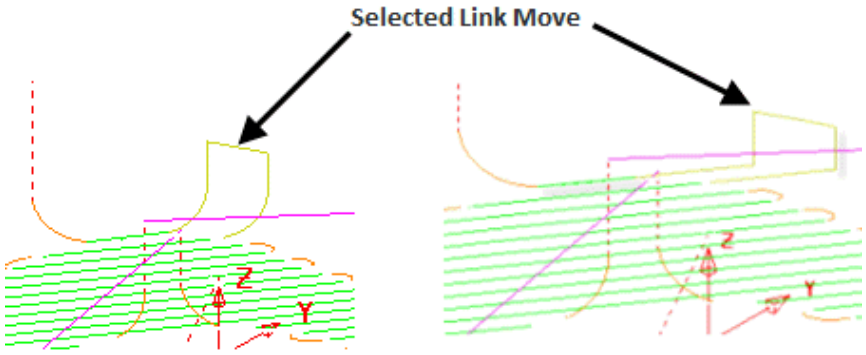
13 Use the ***left mouse key*** to select one of the individual link moves where the Lead is currently a **Vertical Arc**.

14 In the **Leads and Links** dialog, select the **Lead In** tab and select **1st Choice** as **Extended Move** with **Distance 10**.

15 Select the **Copy to Lead Out** tab.

16 Click **Apply**.

17 The selected **Vertical Arc** lead move is changed to an **Extended** move.



18 Select **File** - **Save Project** to update:

.....\COURSEWORK\PowerMILL-Projects\LeadsLinks-example

Additional Short Links options

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

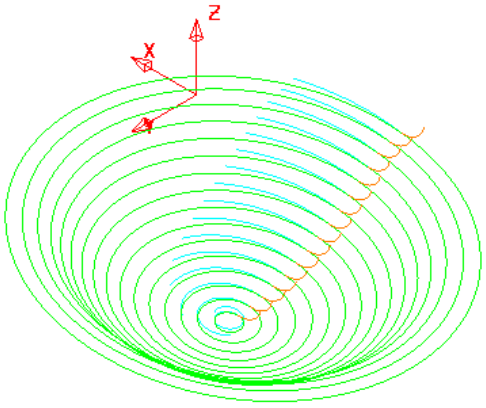
Exercise

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

a **Links**: **Long/Short/Default** – **Skim**.

b **Lead In**: 1st Choice - **Ramp** - ***Options*** - **Max Zig Angle** 4, ***Follow*** **Toolpath**, and **Height** 1

c **Lead Out**: 1st Choice - **Vertical Arc** - **Angle** 90 - **Radius** 3

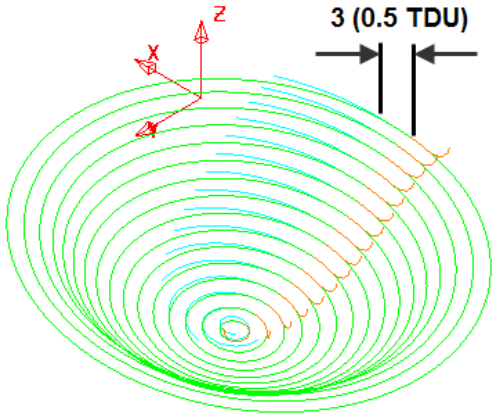


The **Links** have been visually removed to enable a better view of the **Leads**.

 Copyright © Delcam plc

2 In the **Leads and Links** dialog select the **Lead in** page and input **0.5** in the box named **Overlap distance (tool diameter units)**.

3 Apply the **Leads and Links** dialog.



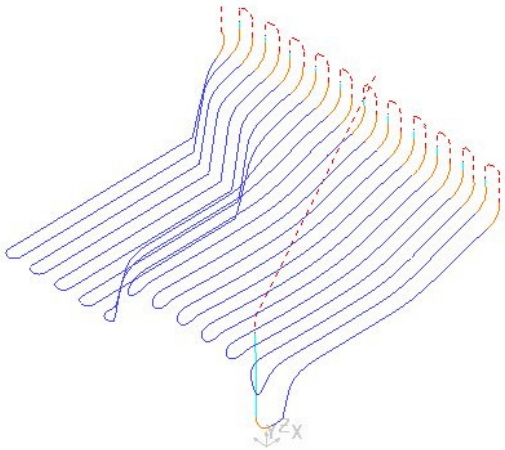
Overlap Distance causes the cutter to continue beyond the natural end point along the original toolpath by a specified distance based on the Tool Diameter (**TDU**). In this case the distance is **3mm** (**0.5 TDU**).

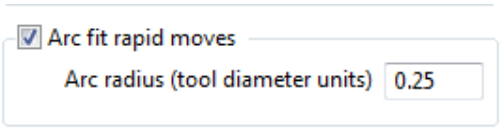
Arc Fitting Rapid Moves

Leads and Links

Arc Fitting works on all ***Rapid Moves*** including **Skim** moves. This option is ideal for applications where you want to avoid sharp changes in direction; for example, High Speed Machining.

Note; not all machine tool controllers will support this option.





- The **Arc Fit Rapid Moves** option is located in the **Links** dialog.
- The **Arc fit rapid moves** box must be **ticked**.
- The **Arc radius** is defined as a proportion of the active, Tool Diameter (**TDU**).

Copyright © Delcam plc

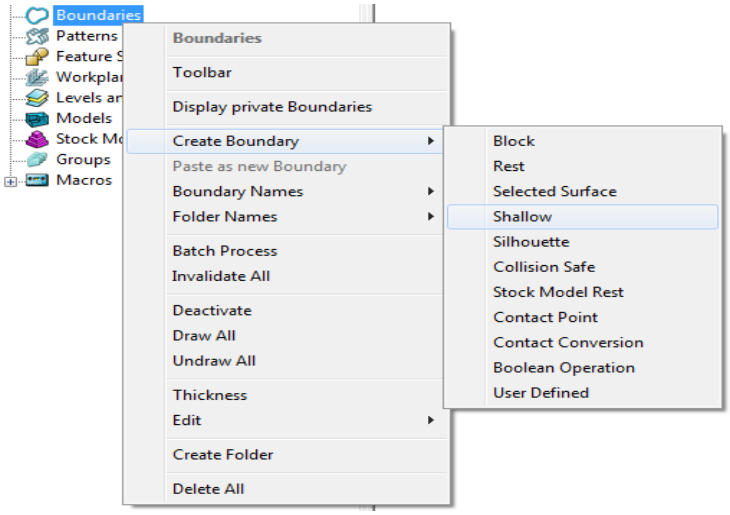
6. Boundaries

A **Boundary** consists of one or more closed wireframe segments. Boundaries are mainly used to limit machining strategies to specific areas of the component.

Earlier in the course, **Boundaries** were utilised to limit machining strategies, so that they occur in more compatible areas of the component.

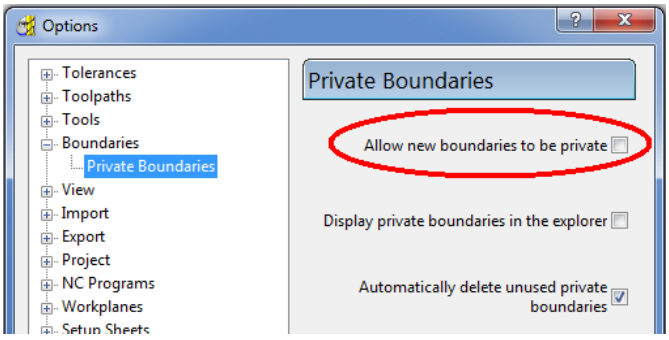
For example, **Constant Z finishing** is more effective on steep sidewalls and **Raster finishing** on shallow areas.

There are several options available for creating boundaries. Most involve interaction with other **PowerMILL** entities (eg **Tools**, **Thickness**, **Stock Model**, etc), while **User Defined** will access a separate dialog for direct input of exisiting geometric, entities.



Switch off the Private Boundaries creation option

1 From the **main** pulldown menus select **Tools** – **Options** and in the dialog select **Boundaries** – **Private boundaries**.



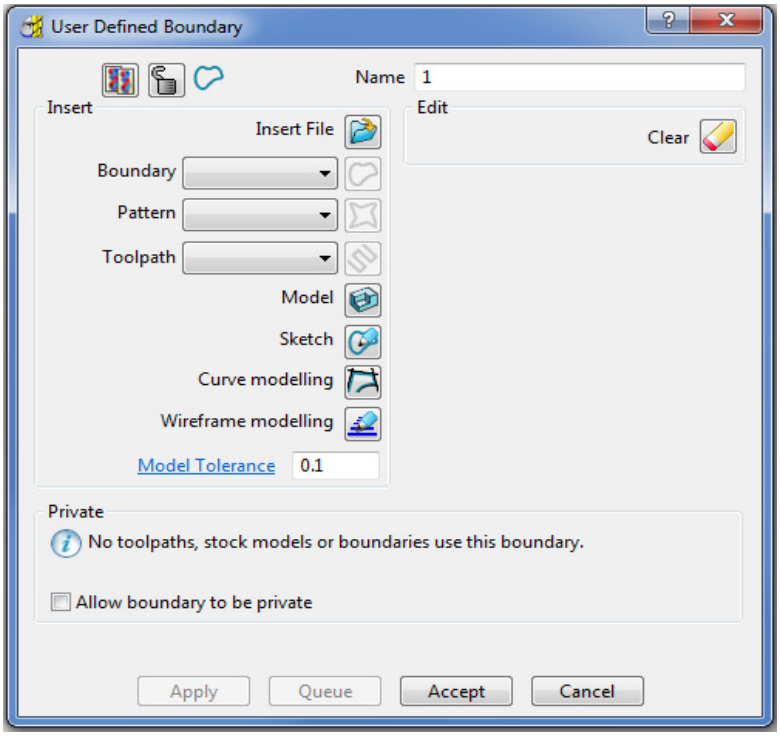
2 **Un-tick** the box named **Allow new boundaries to be private**.

3 Select **Accept** to close the **Options** dialog**.**

User Defined Boundary

To access the **User Defined** – ***Boundary*** options, go to the **PowerMILL *explorer***, and select **Boundaries** > **Create Boundary** > **User Defined**.

This will access a separate sub-menu in which the range of options, generally involve direct input of existing wireframe entities.



Insert File - Opens the browser to locate an externally stored geometry file.

The **Boundary**, **Pattern**, and **Toolpath** buttons are disabled unless a compatible entity is selected in the adjacent input box.

Model - Inserts the edge of the selected **model**.

Sketch - Opens the **Curve Editor** mode toolbar.

Curve Modelling - Opens the **composite curve** generator.

Wireframe Modelling - Provides access to the PowerSHAPE wireframe module.

4 From the **Main** menu, select **File - Delete All**.

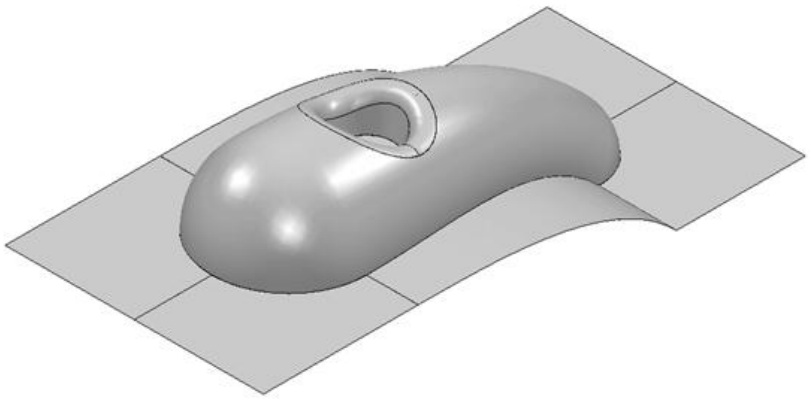
5 Select **File** - **Save Project As**:

.....\COURSEWORK\PowerMILL_Projects\BoundaryBasics

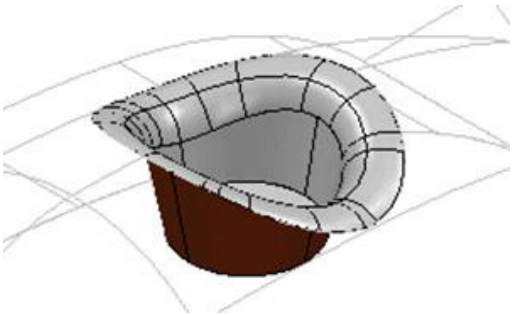
6 Select **File** - **Import Model...**

.....\PowerMILL_Data\Models\cowling

Copyright © Delcam plc

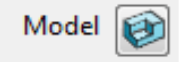


7 Select the **surfaces** defining the central pocket and fillet.

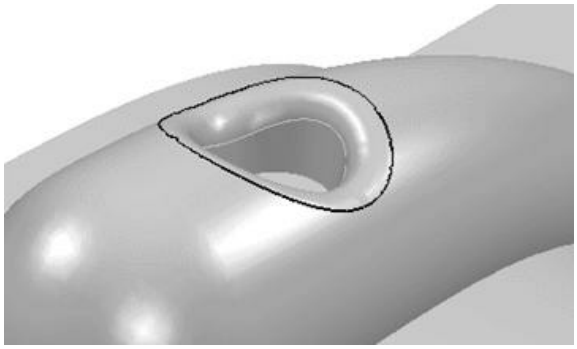


8 In the **PowerMILL *explorer***, right-click **Boundaries** and select **Create** - **User Defined**.

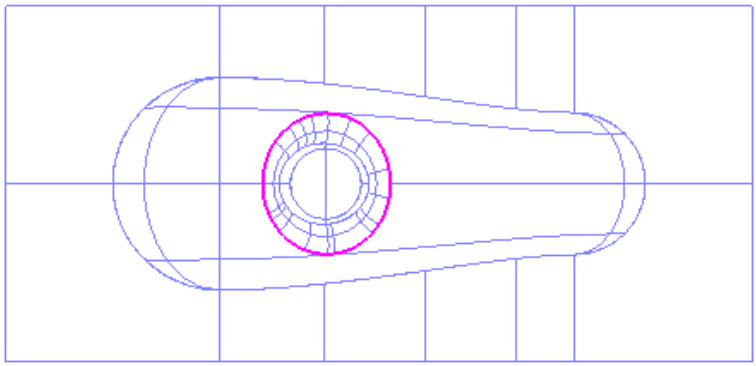
9 In the **User Defined Boundary** dialog, click the **Model** button.



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



10 Select a ***View from top*** (Z) and switch off the **shaded** view but retain the **wireframe** view.



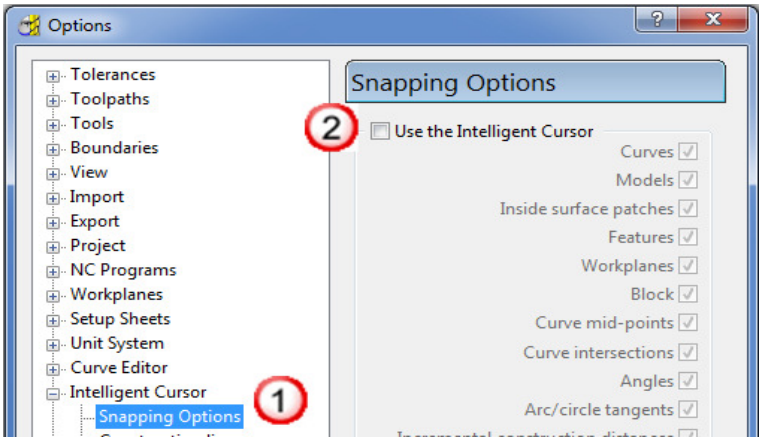
11 In the **PowerMILL *explorer***, right-click **Boundaries** and select **Create Boundary - User Defined**.

12 In the **User Defined Boundary** dialog, click the **Sketch** button

to open the **Curve Editor** mode toolbar.

13 On the **Curve Editor** mode toolbar, click the following button to open the **Options** dialog.

14 Un-tick **Intelligent Cursor** - **Snapping Options** (1) to switch off the Intelligent Cursor (2) enabling 'free form' sketching.

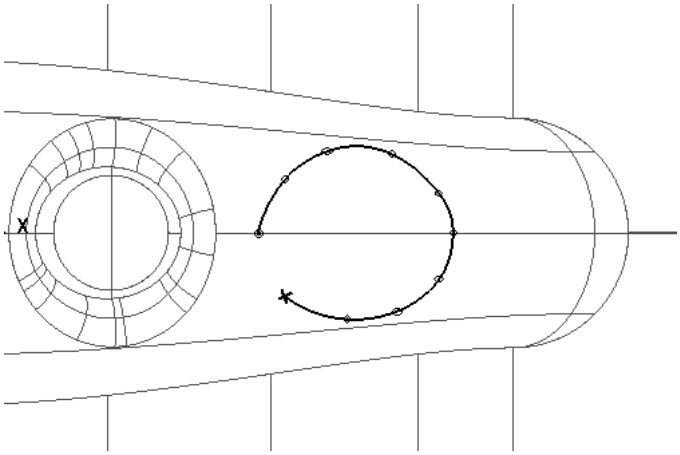


15 Click **Accept** to keep the changes.

16 From the **Curve Editor** toolbar, select **Create a Bezier curve** .

17 Using the left mouse key, click points within the model to create a sketched **Boundary**.

 Copyright © Delcam plc



*If required, use **Undo** to sequentially remove any invalid points.*

*While the **Intelligent cursor** is **switched off** do not attempt to close the segment by trying to snap the 'final span' back on to the start point.*

18 On the **Curve Editor** toolbar, click to open the **Options** dialog.

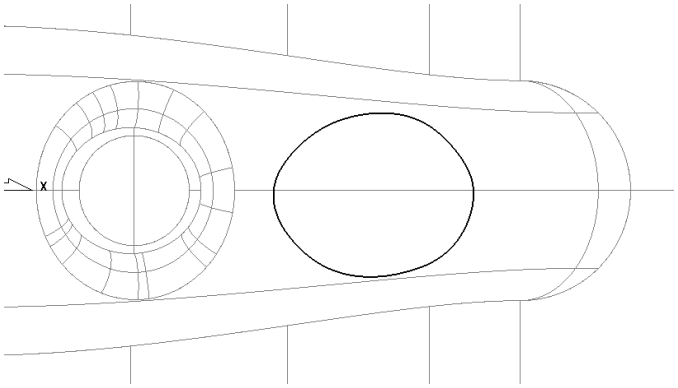
19 Select **Intelligent Cursor** - **Snapping Options** and tick - **Use Intelligent Cursor**.

20 Then ***snap*** the final span of the **Bezier Curve** onto the original start point.

21 On the **Curve Editor** toolbar, click to accept the changes and close the session.

22 If the **Boundary** segment is not smooth enough, right mouse click on it and from the local menu, select **Edit - Spline Selected** to open the **Spline fitting tolerance** dialog.

23 Enter the value 2 and click to apply and close the dialog.



The **Boundary** segment is splined through the polyline points as a curve maintaining the original form to within a maximum deviation of 2.

In instances where the required deviation would be greater than 2, a tangency will remain either side of the affected points.

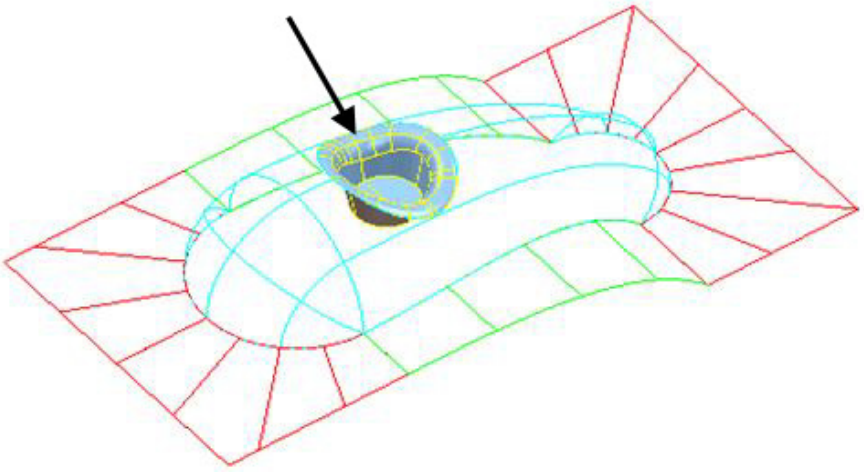
Selected Surface Boundary

A **Selected Surface Boundary** defines one or more segments around where the ***active tool*** would fully machine the selected surfaces. The segments represent the tip positon of the ***active tool***.

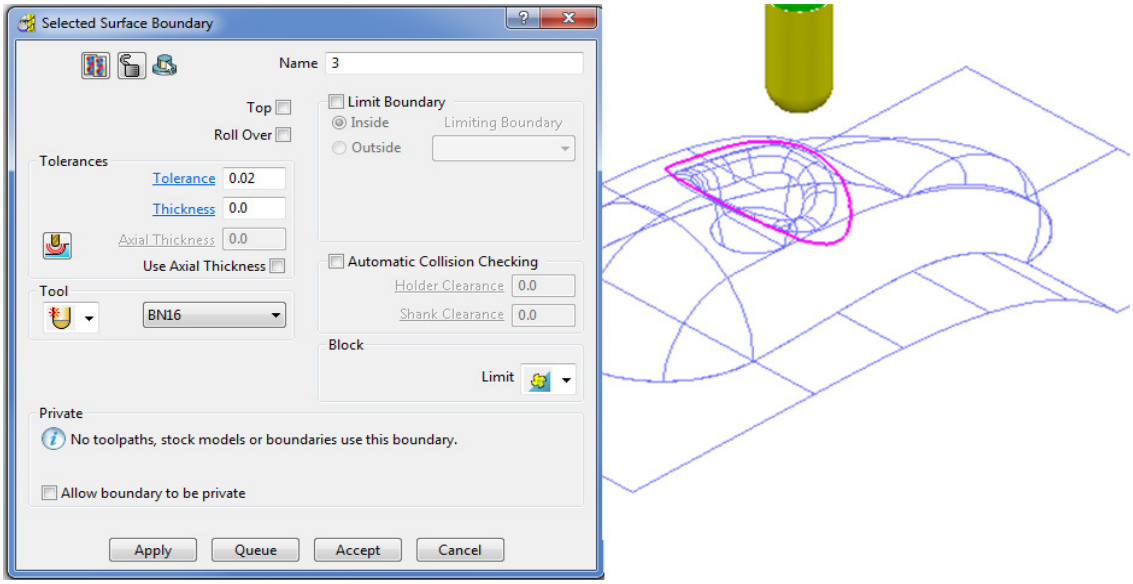
1 Create a **Block** - ***Defined by*** **Box** using ***Type*** **- Model**.

2 Create a **Ball Nosed** tool of **Diameter 16** with the ***Name*** **BN16**.

3 Select the **surfaces** defining the central pocket, including the fillet.



4 In the **PowerMILL *explorer***, right-click **Boundaries** and select **Create Boundary - Selected Surface**.



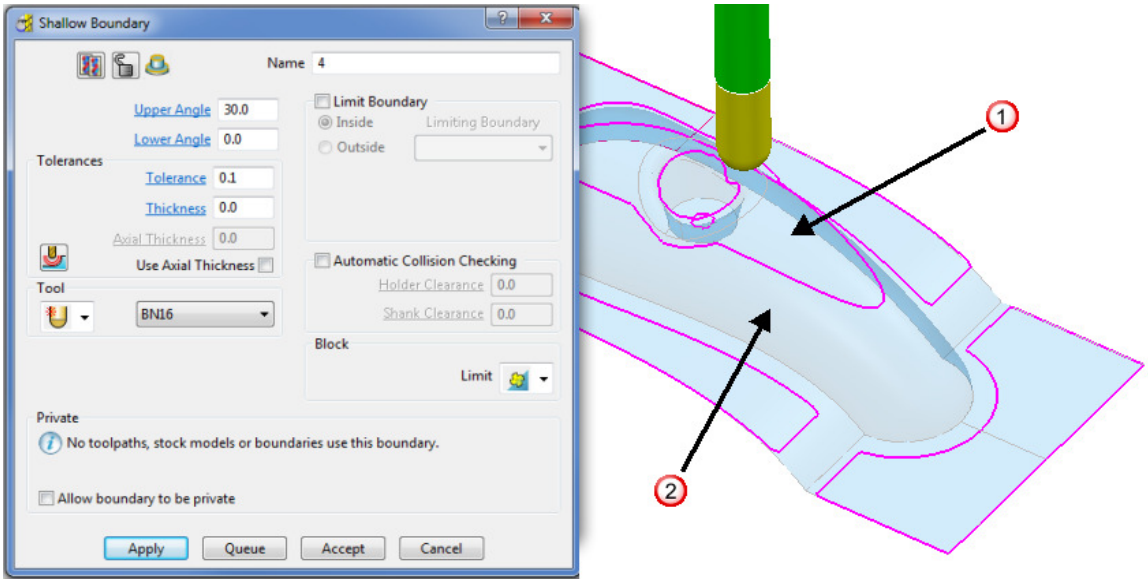
5 Using the above settings, click **Apply** to create a single segment, **Selected Surface Boundary**.

Copyright © Delcam plc

Shallow Boundary

A **Shallow Boundary** defines one or more segments where the model drops through a specified **Upper Angle** downwards from the horizontal plane. It is designed to differentiate ***steep*** and ***shallow*** areas where **Constant Z** and **Pattern** strategies are more effective. The **boundary** is calculated relative to the active tool parameters.

1 In the **PowerMILL *explorer***, right-click **Boundaries** and using the default settings, **Create Boundary - Shallow**.



1 - Typical area more suited to a **Pattern** strategy (above 35 degrees).

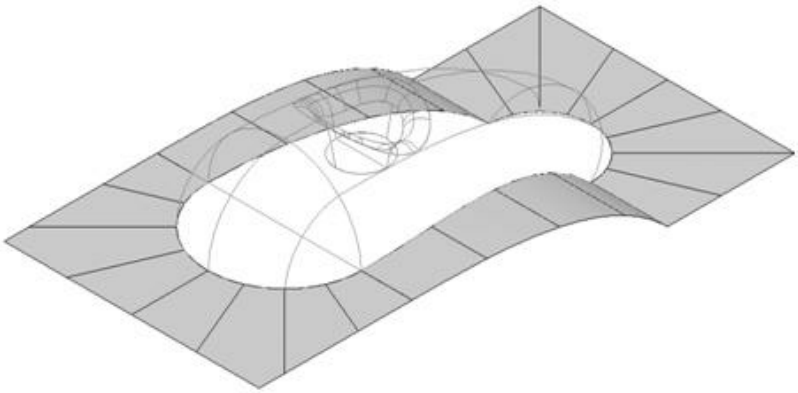
2 - Typical area more suited to a **Constant Z** strategy (below 35 degrees).

2 Using the above settings, click **Apply** to create the above **Boundary** segments.

Silhouette Boundary

A **Silhouette Boundary** defines the outline around the maximum dimensions selected model, offset by the ***tool radius***.

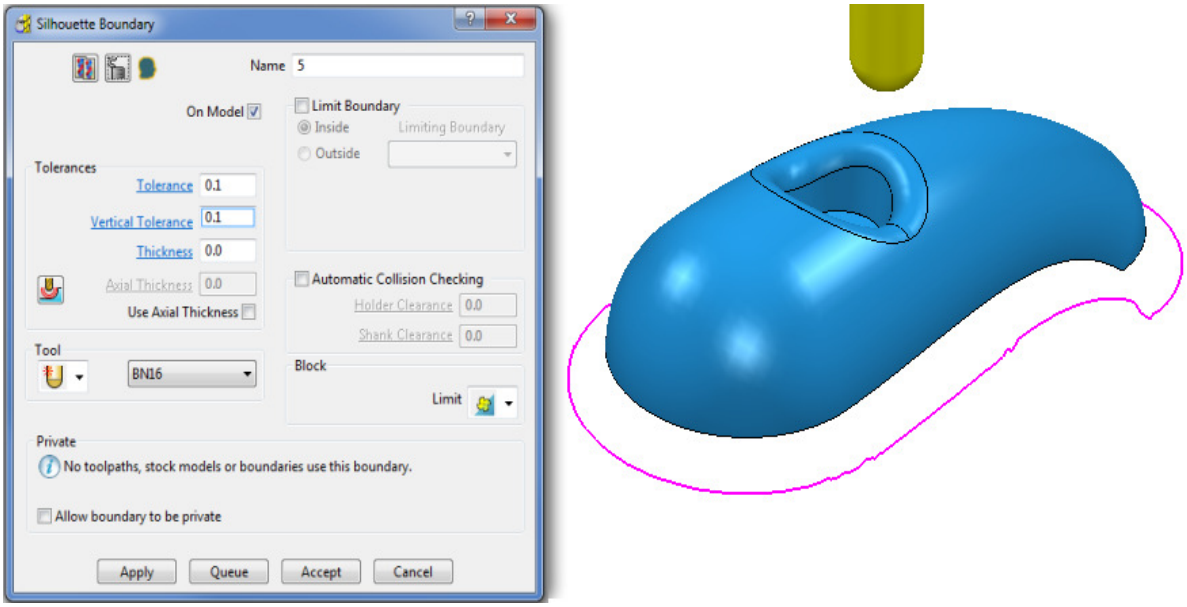
1 Select the following **surfaces** on the model:



*Use the **Shift** key on your keyboard to select multiple surfaces.*

2 Right-click the **Model** and select **Edit - Delete Selected Components**.

3 With the selected areas now deleted, in the **PowerMILL *explorer***, right-click **Boundaries** and select **Create Boundary** - **Silhouette**.



Rest Boundary

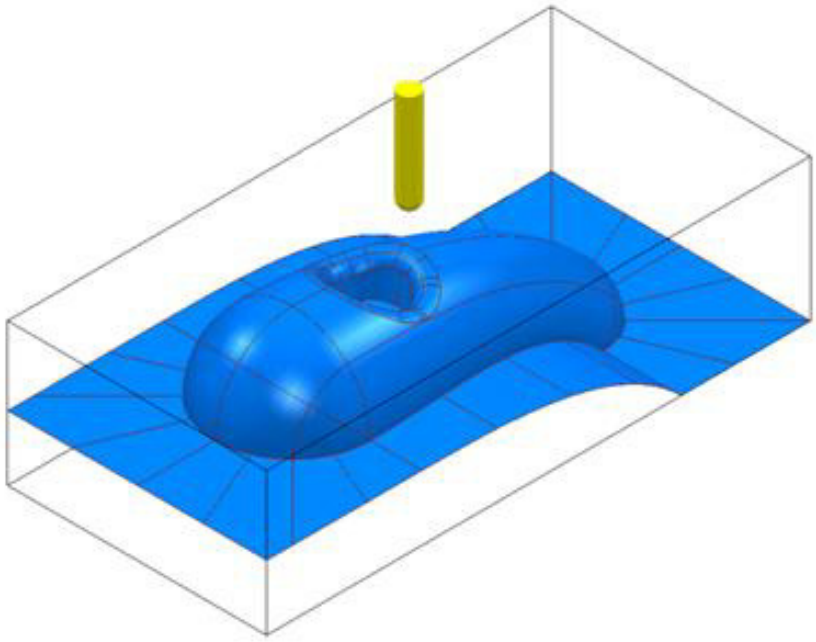
A **Rest Boundary** defines the area inaccessible to a specified reference tool. It also requires a smaller active tool to be specified otherwise it will not generate segments.

1 Create a **Ball Nosed** tool of **Diameter 8** with the **Name BN8**.

2 In the **PowerMILL *explorer***, right-click **Models** and select **Delete All** to remove the incomplete component.

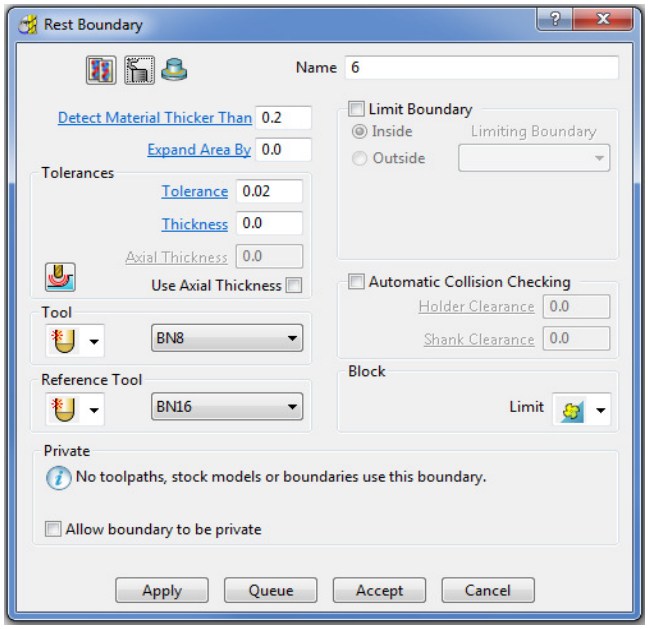
3 **Import** the original **Model** back into the **Project**:

.....\PowerMILL_Data\Models\cowling

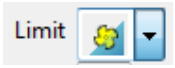


Copyright © Delcam plc

4 In the **PowerMILL *explorer***, right-click **Boundaries** and select **Create Boundary - Rest** to open the **Rest Boundary** dialog.



5 Input **Expand Area** - **0**, select **Tool** - **BN8**, and **Reference Tool** - **BN16**.

6 Ensure that **Limit** is set with ***tool centre to block edge***. 

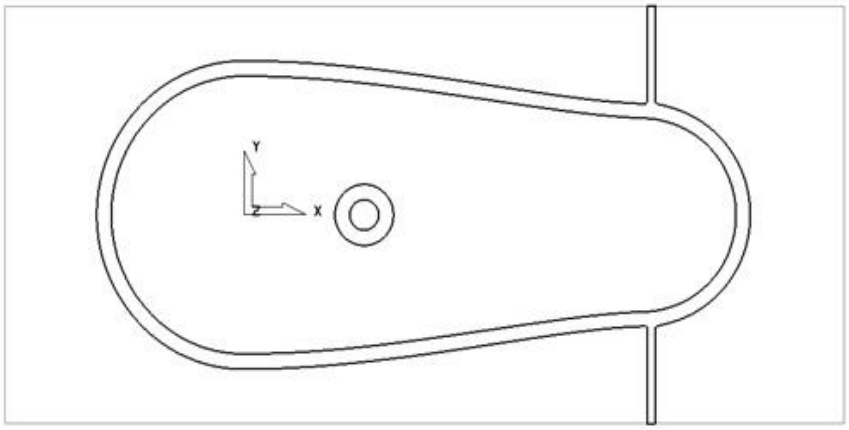
7 Click **Apply** to create the following **Rest Boundary**.

8 Click **Accept** to close the dialog.

9 **Save** the **Project** to update:-

.....\COURSEWORK\PowerMILL_Projects\BoundaryBasics

10 Select a **View** down the **Z Axis** and **Un-draw** the model.



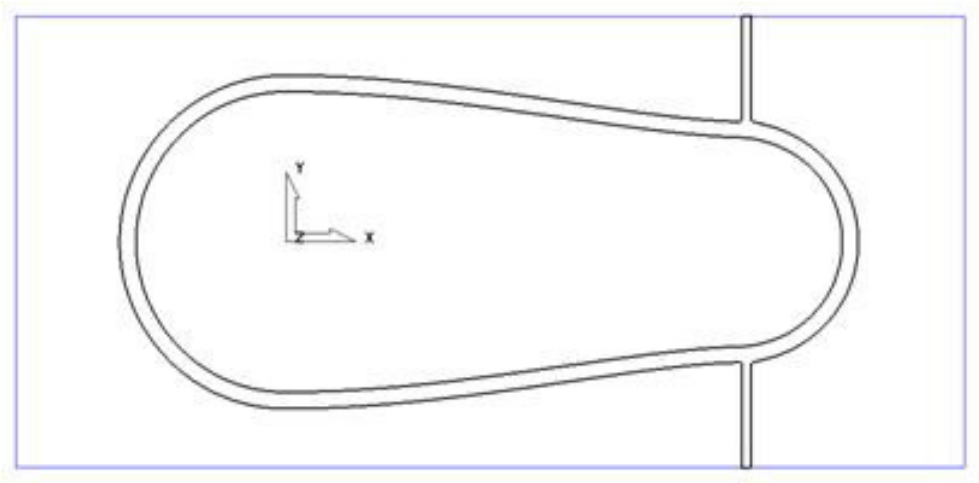
The above **Boundary** identifies areas on the component that are inaccessible to the **BN16** tool geometry to be locally machined with the **BN8** tool.

On closer inspection, the central pocket area would be more effectively machined with a flat bottomed tool and a smoother, more continuous toolpath, can be created if the 2 spurs were removed from the outer segment to be machined later as a separate toolpath.

Editing Boundaries

It is often necessary to edit the original geometry defined by **Boundary** segments. The best way to achieve this in **PowerMILL** is via the dedicated **Curve Editor** tool. As it is not exclusive to **Boundaries**, a more detailed range of **Curve Editor**, options and applications are covered in **Chapter 11**.

1. In the **PowerMILL *explorer***, right-click the **Rest Boundary** (**6**) entity and in the local menu select **Edit** - **Copy Boundary** to create a copy named (**6_1**).
2. In the **PowerMILL *explorer***, right-click the original **Rest Boundary** (**6**) and ***Rename*** it as **Master**.
3. ***Rename*** the new copy of the **Rest Boundary** (**6_1**) as **Rest**.
4. In the **PowerMILL *explorer***, click the light bulb adjacent to **Master** to stop it from being displayed in the graphics area.
5. Select the ***2 inner segments*** on the **Rest boundary** (**Rest**) by dragging a box around, and then press the **Delete** key on your keyboard to delete them.



The next stage is to remove the two spurs, which will be carried out after an explanation of the **Boundary Editor** dialog.

Boundary – Access to the Curve Editor

Once created, you can modify a Boundary segment using the options available on the **Boundary** Toolbar.

Click on the Boundary toolbar to activate the **Curve Editor** toolbar.

*You can also double-click a **boundary segment** to activate the **Curve Editor** Toolbar.*

Copyright © Delcam plc

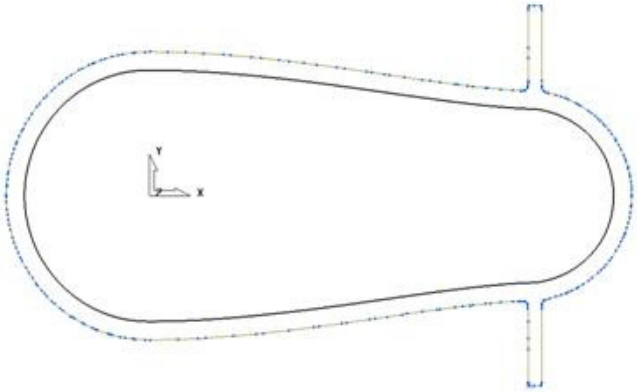
Curve Point - editing

The **Point** editing options can only be applied to ***Continuous Line*** (Polygonised) segments.

1 In the **PowerMILL explorer**, right-click **Boundaries** and from the local menu select **Toolbar**.

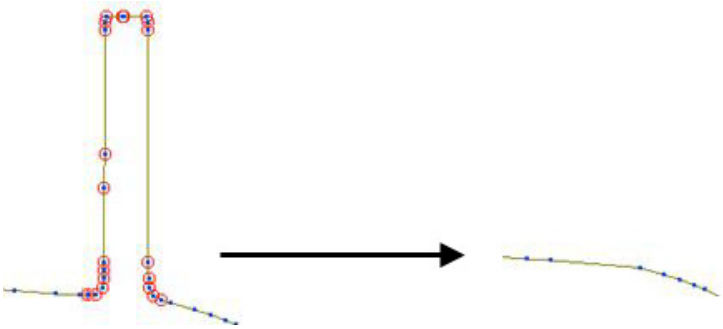
2 On the **Boundary** toolbar, click the **Curve Editor** button to display the **Curve Editor** toolbar.

3 Select the outer **boundary segment** in the graphics area.



The points are displayed on the selected boundary segment.

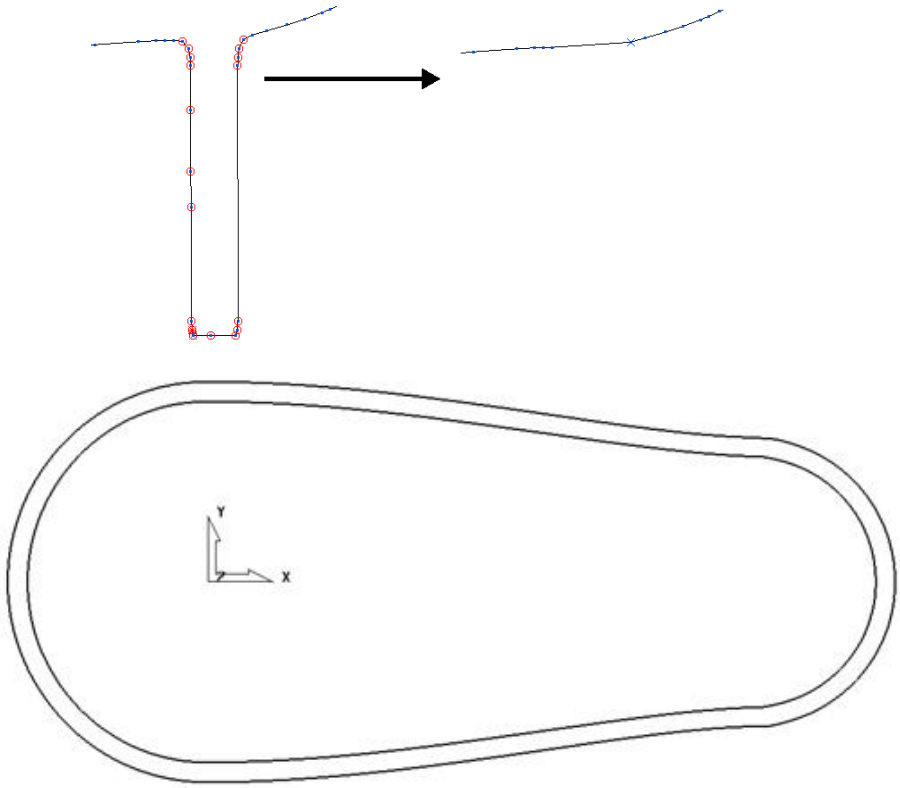
4 **Zoom** into the area of the selected segment and select all the points that define the spur.



*To select multiple points, hold down the **Shift key** while selecting the points with the mouse.*

5 On the **Curve Editor** toolbar, click **Delete points** to remove the spur from the segment.

6 Repeat the procedure on the other spur.

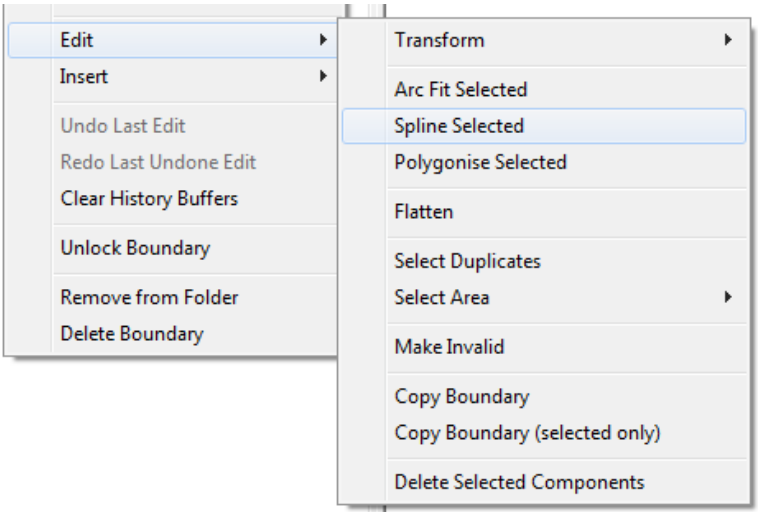


Ensure that all segments are either selected or not selected before leaving the editor (If some segments are selected then it will result in the remainder being discarded when exiting the editor).

7 Click , to accept changes and exit the **Curve Editor**.

Other boundary editing options

Right-click on a **Boundary** in the graphics area or in the **PowerMILL *explorer*** and from the local menu, select **Edit** to view other boundary editing options.



 Copyright © Delcam plc

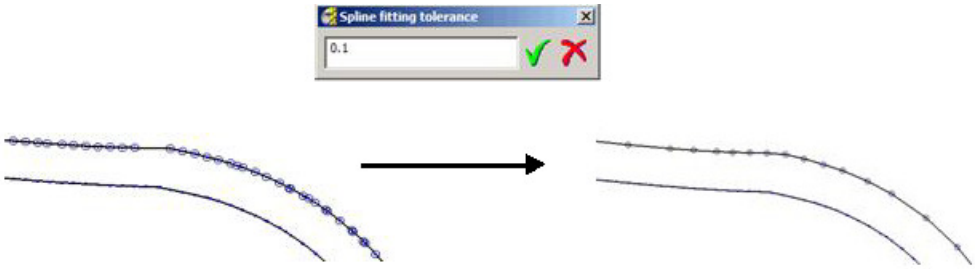
Edit > Spline Selected

1 In the graphics window, select the outer segment of the boundary **Rest**.

2 In the **PowerMILL *explorer***, right-click the Boundary Rest and select Edit > Spline Selected.

3 In the **Spline fitting tolerance** dialog, enter 0.1 and click to apply.

4 Re-select the ***outer segment*** to view the effect of the Spline.

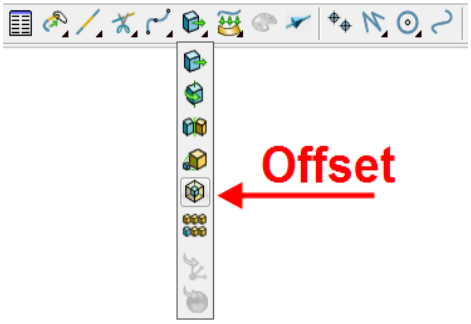


The selected segment is now Splined (Curve fitted and the points re-distributed within the specified tolerance value) to produce a smoother form with unnecessary points removed. This improves the quality of subsequent operations such as offsetting of the boundary.

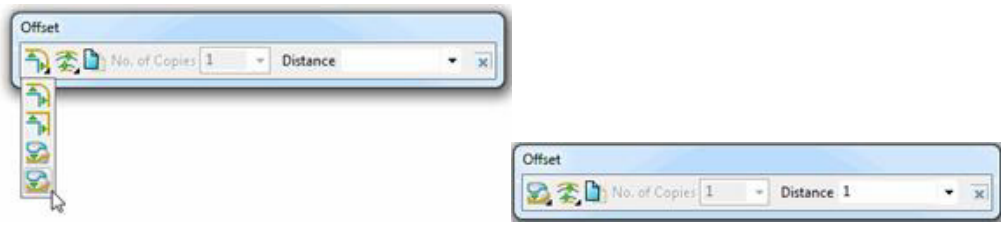
Edit > Transform

1 In the **PowerMILL *explorer***, right-click on the boundary **Rest** and from the local menu, select **Curve Editor**.

2 In the **Curve Editor** toolbar select the **Offset** option

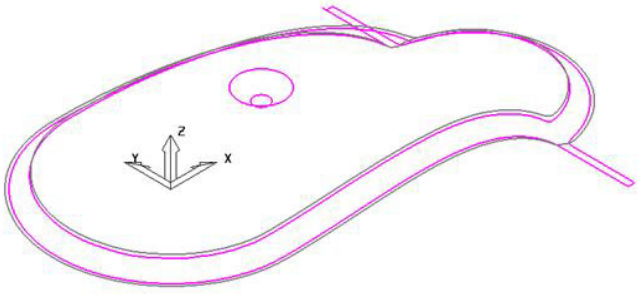


3 In the **Offset** toolbar, select the **3D Smooth** button, input a **Distance** of 1 and then press the **Return** key.



4 Click in the **Curve Editor** toolbar, to accept the changes and exit.

5 In the **PowerMILL explorer**, click the light bulb next to the copy of the original boundary **Master** created earlier. The original **boundary** is displayed alongside the edited **boundary**, **Rest** illustrating the effect of the 1mm 3D Offset.

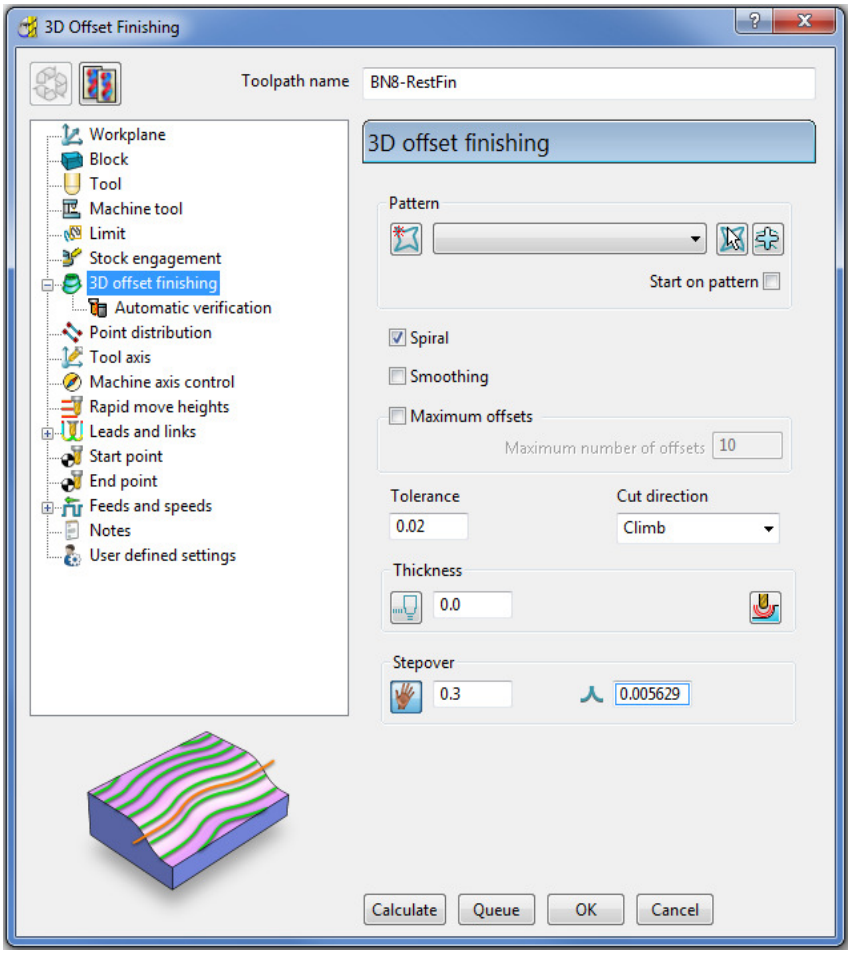


6 **Activate** the **BN8** tool and **Boundary** named **Rest**.

7 On the **Main** toolbar, click to display the Strategy Selector.

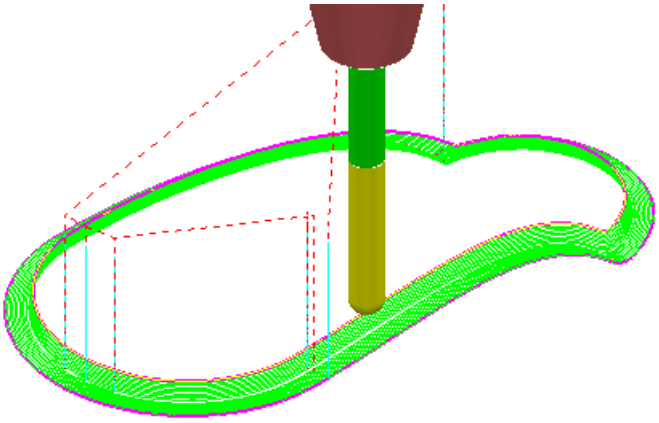
8 From the **Finishing** tab, select the **3D Offset Finishing** strategy.

9 In the **3D Offset Finishing** strategy dialog, enter data as below.



10 Click **Calculate** to process the toolpath and then **Close** to exit the dialog.

Copyright © Delcam plc



11 Undraw the **BN8-RestFin** toolpath and boundary **Rest** by switching off the respective light bulb symbols in the **PowerMILL *explorer***.

12 Switch on the respective light bulb to display the boundary named **Master**.

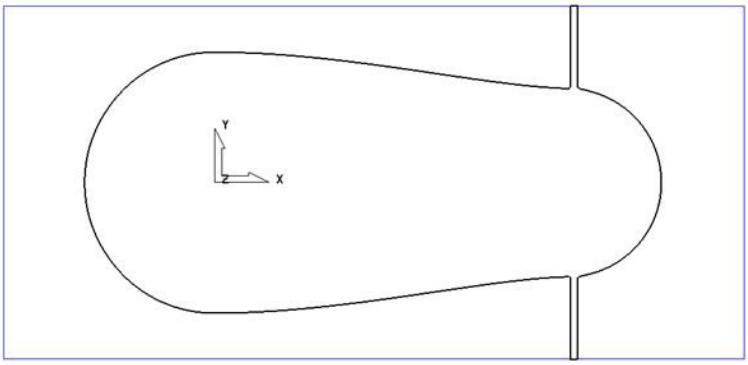
13 In the graphics area, select the ***outer segment*** of the displayed **Boundary** named **Master**.

14 In the **PowerMILL *explorer***, right-click the boundary **Master** and select **Edit** - **Copy Boundary** (**selected only**) to create a new boundary **Master_1** consisting of the outer segment only.

15 **Rename** the boundary **Master_1** as **Spur1**.

16 Select a view from top **Z**.

17 In the **PowerMILL *explorer***, turn off the light bulb adjacent to the boundary **Master** to remove it from view.



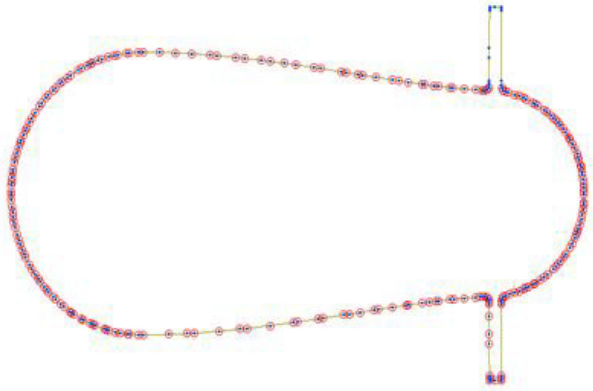
18 In the **PowerMILL *explorer***, right-click the boundary **Spur1** and select **Activate**.

19 In the graphics area, double-click the ***boundary segment*** to activate the **Curve Editor** toolbar.

20 Select all **points** along the ***boundary segment*** except those which define the rectangular part of the upper spur.

*Use the **Shift** key on your keyboard to select multiple points on the boundary segment.*

*Use the **CTRL** key on your keyboard to deselect one or more of the selected points on the boundary segment.*



If you want, you can select the points in stages. This gives you better control when viewing selected points.

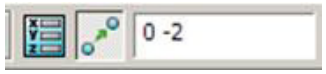
21 On the **Curve Editor** toolbar:

a Click the **Delete Points** button to delete all the selected points.

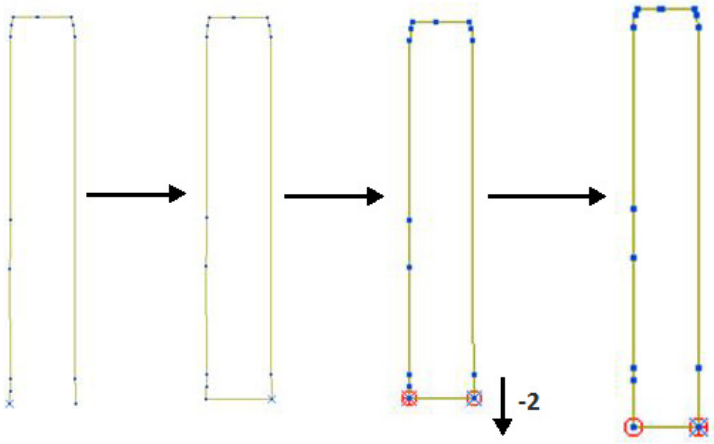
b Click the Join Close button to close the open rectangular segment.

c Press Shift on your keyboard and select the points at the base of the rectangular segment.

d On the status bar below the graphics area, click the Relative Move button and enter **0 -2** in the data box and press ***Enter*** on your keyboard.



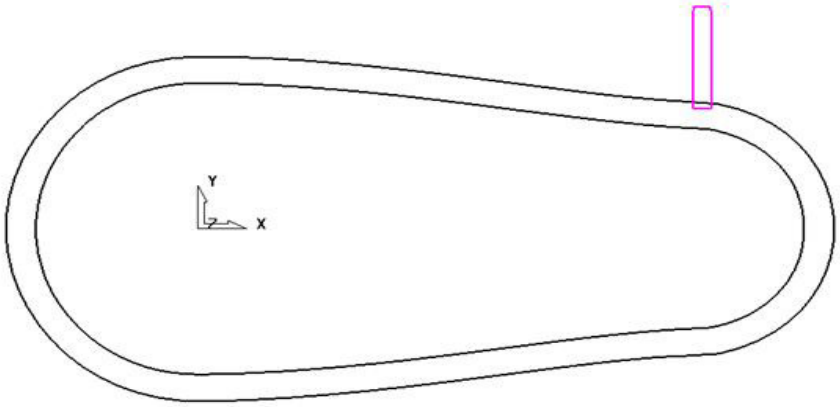
The selected points are moved by a distance of **Y-2**.



22 Click to exit the **Curve Editor** toolbar.

23 Click the light bulbs on to display both ***Boundaries*** **Rest** and **Spur1**, and then **Resize the View** .

 Copyright © Delcam plc

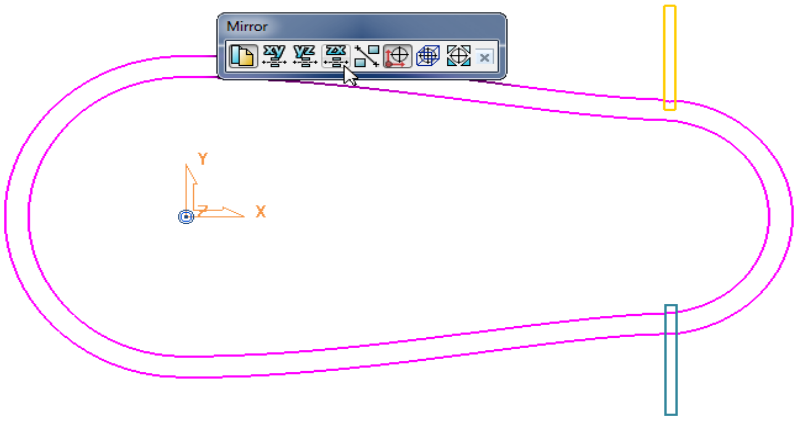


24 In the graphics area, double-click the ***Boundary*** **Spur1** to activate the **Curve Editor** toolbar.

25 Form the **Curve editor** toolbar, select **Transformations** - **Mirror** option .

26 On the **Mirror** toolbar select ***Origin at the active workplane***.

27 Ensure that the **Keep Original** option is set.



28 From the **Mirror** toolbar, click the ***Mirror in ZX*** button .

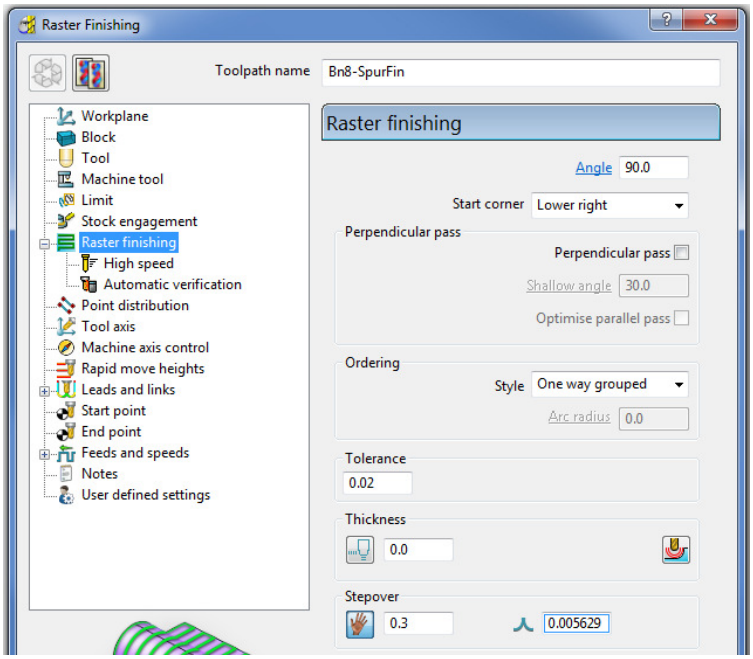
29 Click on the **Mirror** toolbar to accept and close.

30 On the **Boundary Transform** toolbar, click to accept and exit the toolbar.

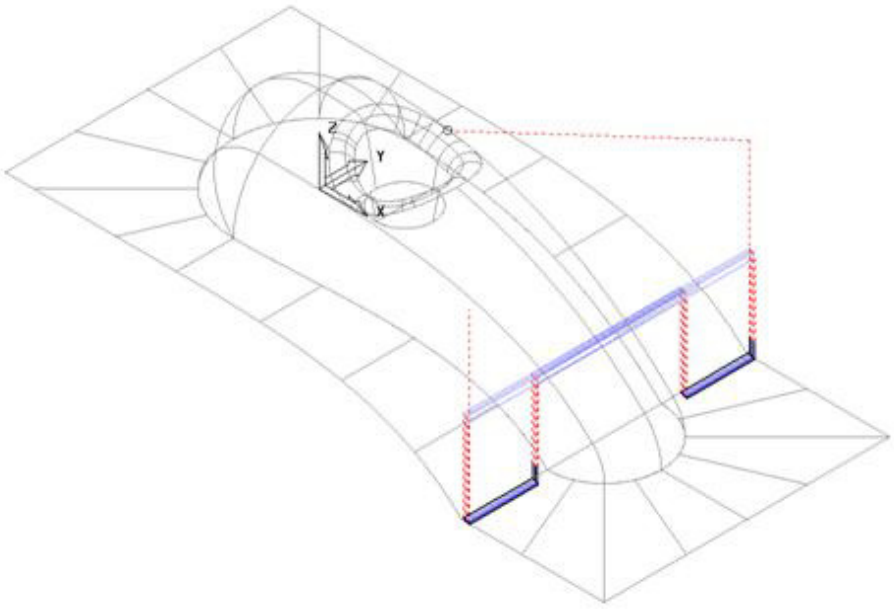
The **boundary** named **Spur1** now contains 2 segments.

1 Make sure that the ***tool*** **BN8** and ***boundary*** **Spur1** are both **Active**.

2 From the **Strategy Selector**, **Finishing** tab, select the **Raster Finishing** strategy and enter data as shown:

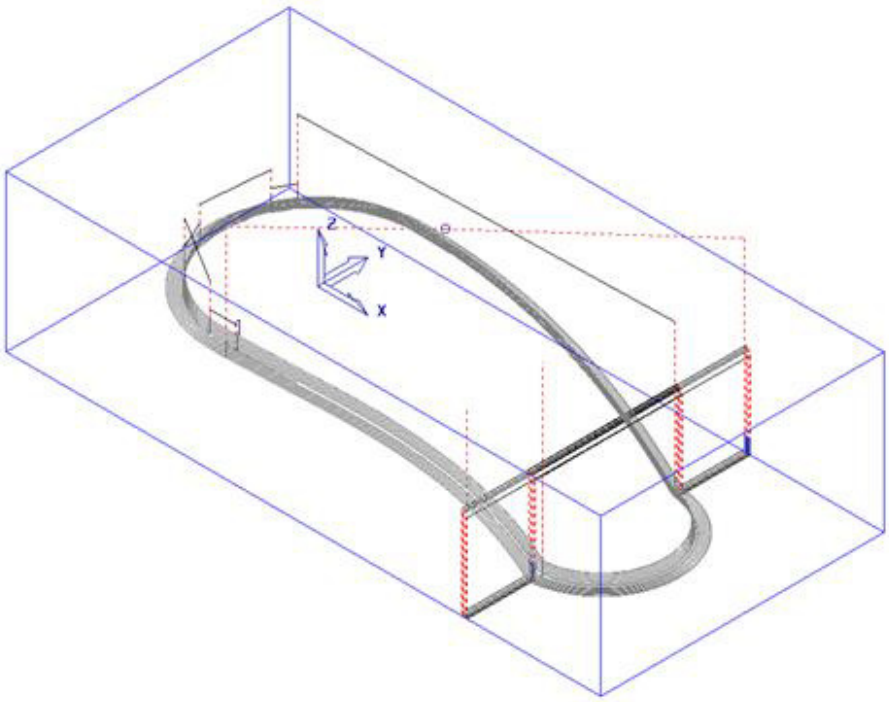


3 Click **Calculate** to process the toolpath, and then **Close** the dialog.



The machining strategies are localised and the tool track shape controlled with the help of edited Boundaries originating from a single Rest Boundary.

 Copyright © Delcam plc



4 **Save** the **Project** to update:-

.....\COURSEWORK\PowerMILL_Projects\BoundaryBasics

Creating Blanking Surfaces using Boundaries

Blanking surfaces are used to cap holes and pockets in a model to prevent the tool from machining into those areas. You can create three types of planes in PowerMILL.

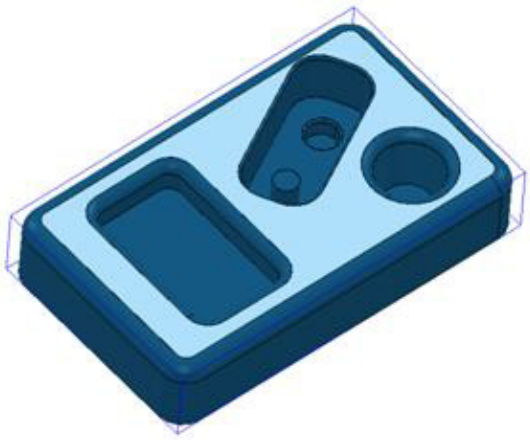
- Block – creates a plane, at a specific Z Height, after the material block is defined.
- Best Fit – creates two planes, one touching the boundary at its highest point, and the other through its lowest point. The planes are parallel to the best fit plane (the plane that has the smallest maximum distance from the boundary).
- Projected – creates a plane parallel to the XY axis of the active workplane. The plane will have the Z Height of the highest point of the boundary.

Example

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

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

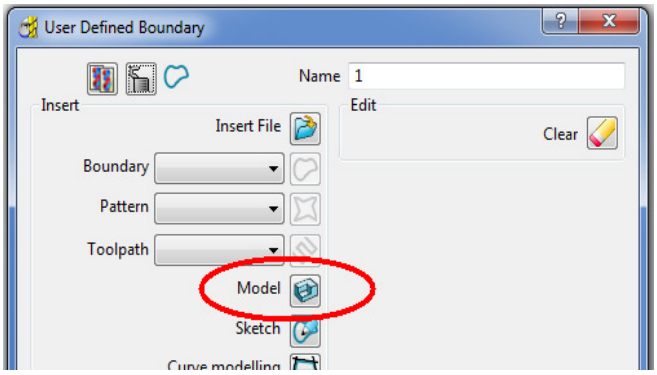
.....\PowerMILL_Data\Models\pockets.dgk



3 **Calculate** the **Block** defined by **Box** to the **Model** Min/Max limits.

4 Select the ***top surface*** of the **model** (shown shaded).

5 In the **PowerMILL *explorer***, right mouse click on **Boundaries** and from the local menu select **Create Boundary – User Defined**.

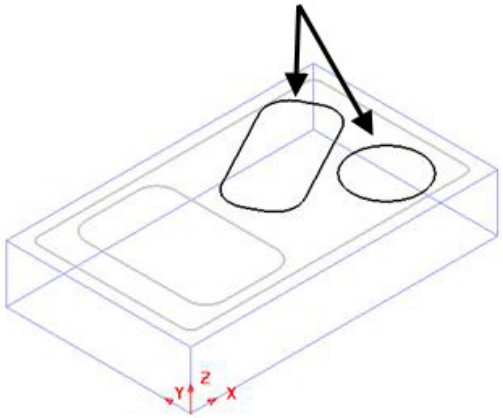


6 Click the **Model** icon and then select **Accept** to close the dialog.

7 Un-draw the **model** to enable a clear view of the **boundary**.

The **Boundary** produced has 4 segments. For this example, only the two smallest segments are required.

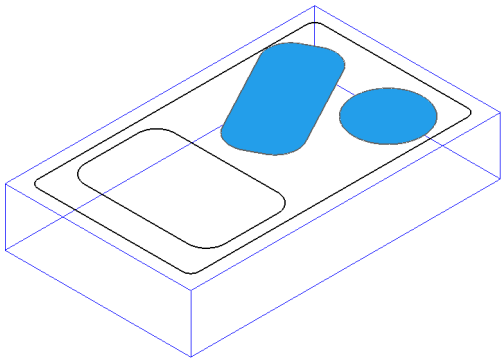
8 Select the two highlighted segments by dragging the cursor over them.



9 In the **PowerMILL *explorer***, right-click **Models** and from the local menu select **Create Plane - Projected**.

 Copyright © Delcam plc

10 Planer **Surfaces** have been created inside the ***boundary segments***.

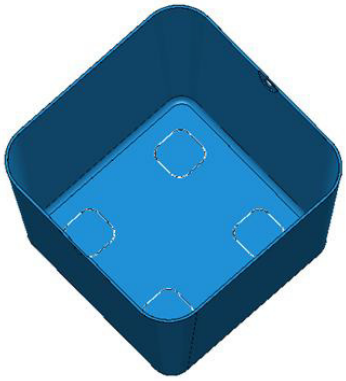


No need to Save the Project this time!

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

Exercise

Import the **Model** …\PowerMILL_Data\Models\bucket.dgk and generate 4 surface planes to cover the foot recesses on the upper base surface.



Private Boundaries

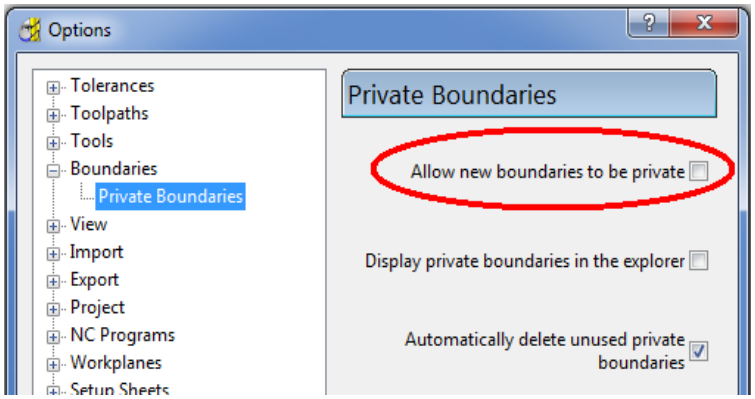
All **Boundary** types will (by default) be created as **private**. This makes the **Boundary** exclusively owned by an individual machining strategy. If the ***parameters*** or ***entities*** used in the strategy are modified, then the associated **private boundary** will automatically adjust on calculation.

Private Boundaries are of particular use in **Templates** as they will automatically regenerate when the host **toolpath** is calculated on a new model.

A **private boundary** is not suitable in cases where the ***segments*** will need to be deleted or geometrically edited once assigned to the host ***strategy***. It is however possible to create a copy on which you can ***switch off*** the **Allow boundary to be private** option.

It is possible to prevent boundaries from originally being created as private:-

From the **main** pull down menus select ***Tools*** – ***Options*** – ***Boundaries*** - ***Private Boundaries***, and un-tick **Allow new boundaries to be private**.

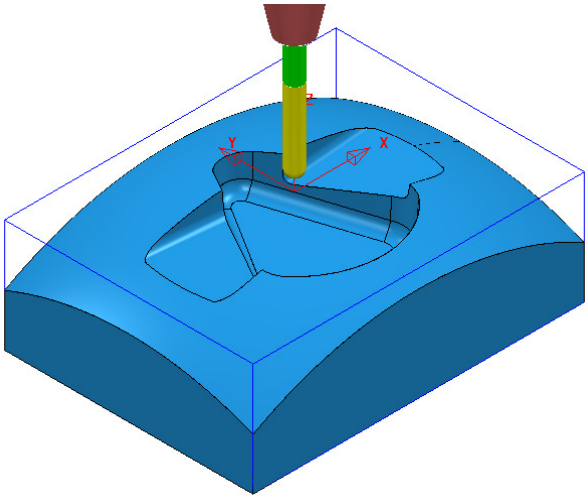


Private Boundaries – Example:-

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

2 **Open** the (readonly) **Project**:-

.....\PowerMILL_Data\Projects\PrivateBoundaries_Start

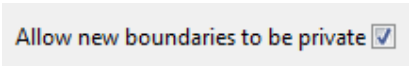


3 **Save Project As**:-

.....\COURSEWORK\PowerMILL_Projects\PrivateBoundaries-EX1

The **Project** includes a **Constant Z** and a **3DOffset *finishing*** toolpaths both of which currently apply over the whole of the **Block** area. To localise the 2 finishing toolpaths to areas of the model where they are most effective both strategies will be recycled to include a **Shallow** – ***private boundary***.

4 From the **main** pulldown menus select **Tools** - **Options** - **Boundaries** - **Private boundaries**:-



5 **Tick** the box named **Allow new boundaries to be private**.

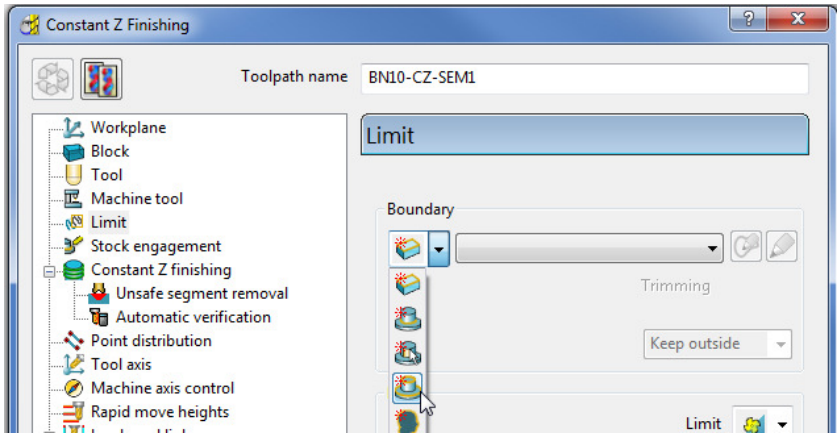
6 Select **Accept** to close the **Options** dialog**.**

7 **Activate** the ***toolpath***, **BN10-CZ-SEM1**.

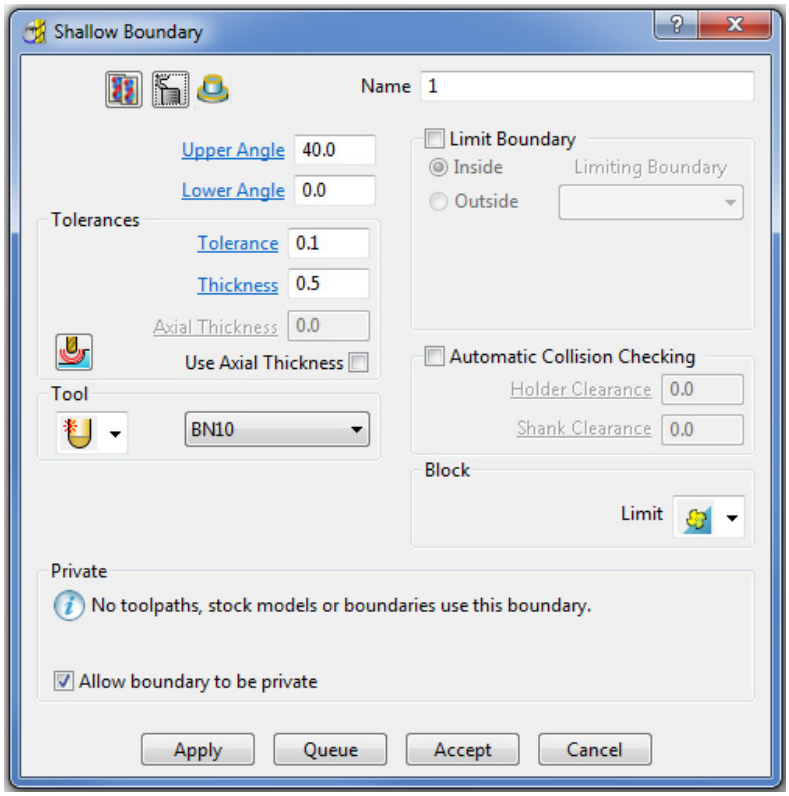
8 From the ***toolpath***, **Bn10-CZ-SEM1** local menu select settings, followed by the ***Recycle*** option.

9 In the **Constant Z** dialog select the **Limit** page.

 Copyright © Delcam plc

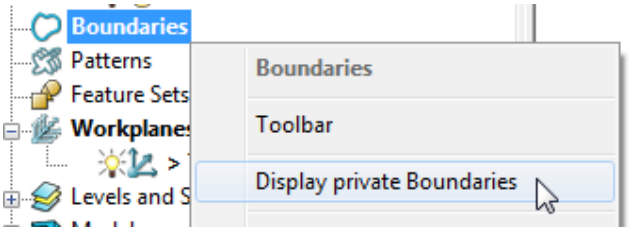


10 From the ***Boundary*** pull down options select **Create a shallow boundary**.



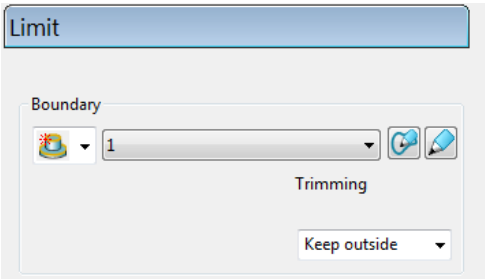
11 Using the default settings **Apply** the **Shallow Boundary** dialog.

Initially the new **Boundary** 1 will exist in the **PowerMILL *explorer*** until the toolpath is calculated. At that point it will disappear (become invisible) from the **Boundary** folder. However, the folder name, **Boundary** will still be displayed in bold text to identify the existence of the hidden ***private*** **Boundary** 1.

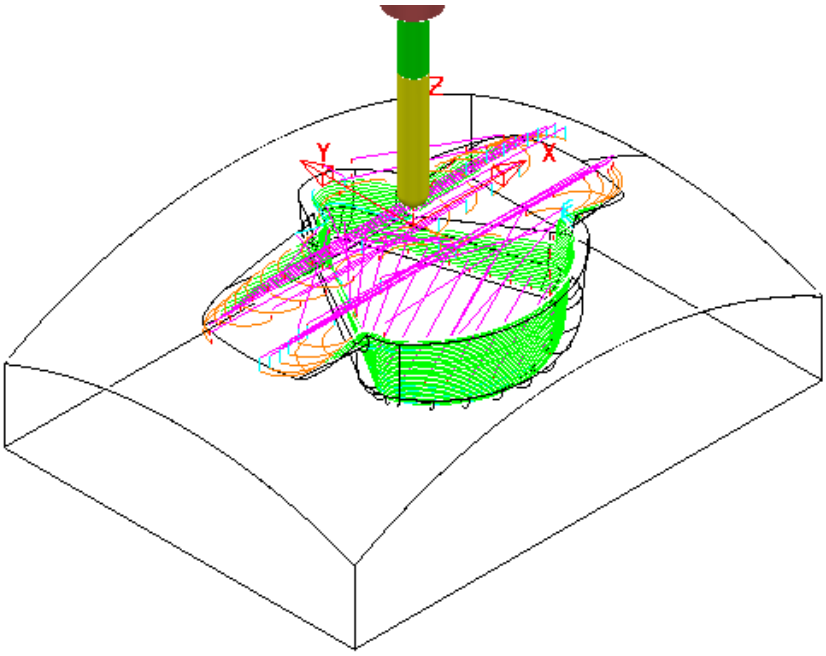


*The **private Boundary** will be displayed in the **PowerMILL explorer** if the above option is **ticked** in the local **Boundaries** menu.*

12 Select the **Limit** page and set ***Trimming*** to **Keep outside**.



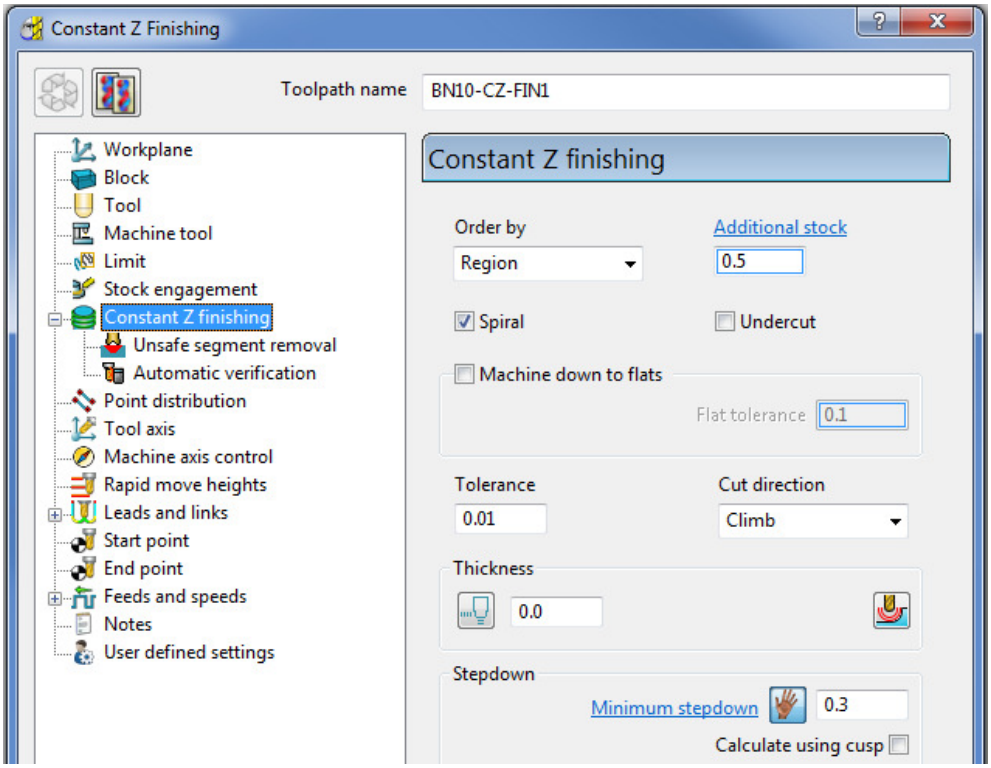
13 **Calculate** (but do not **Close**) the **Constant Z Finishing** strategy.



The **Constant Z** strategy **BN10-CZ-SEM1** is now limited to be ***outside*** the segments of ***private*** **Boundary** 1.

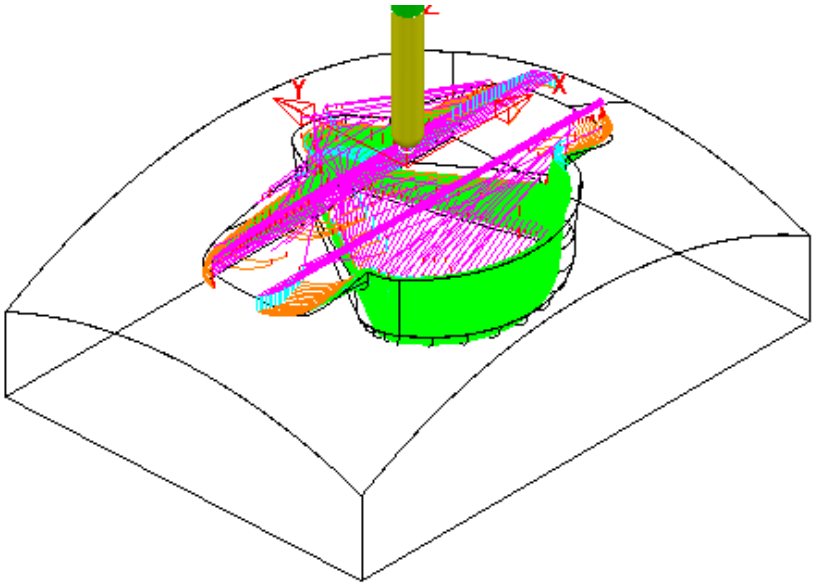
14 Select the 'create a copy' option in the **Constant Z Finishing** dialog.

Copyright © Delcam plc

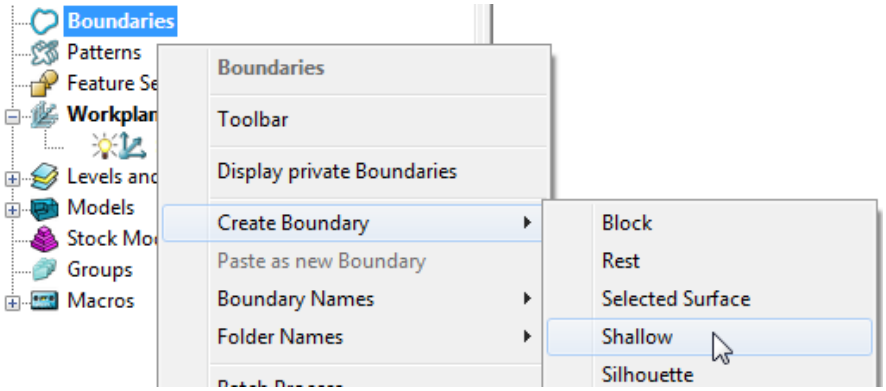


15 Edit the ***Toolpath Name*** to **BN10-CZ-FIN1**, input ***Tolerance*** **0.01**, ***Thickness*** **0**, and **Minimum stepdown** **0.3**

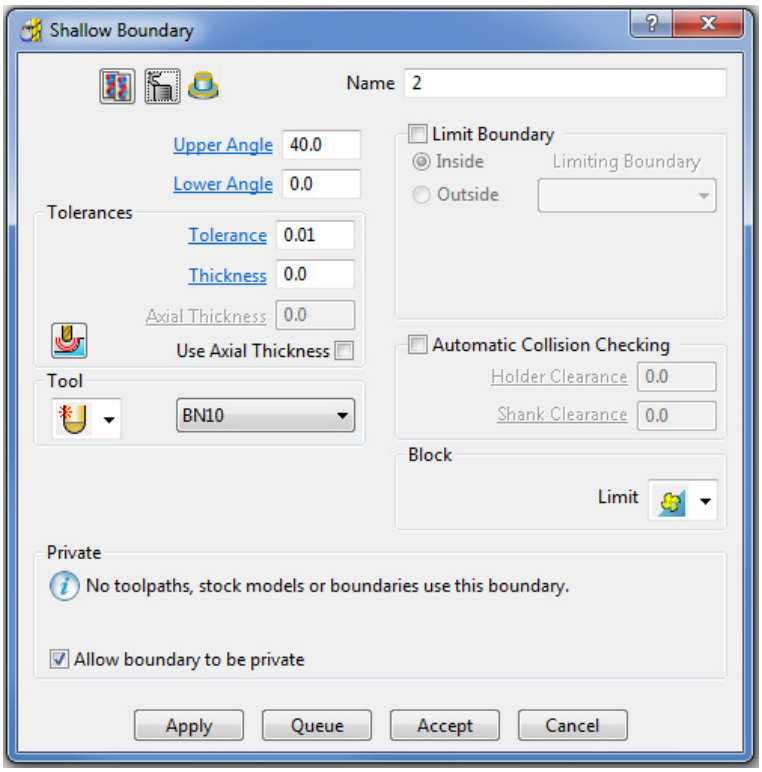
16 **Calculate** the new toolpath and **Close** the dialog.



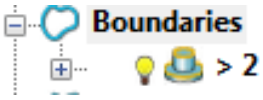
17 A new ***private*** **Boundary** will automatically, be created with the same parameters (**Tolerance**, **Thickness**, etc) as used in the ***main*** page of the **Constant Z Finishing** dialog.



18 In the **PowerMILL *explorer***, right mouse click on **Boundaries** and from the local menu select **Create Boundary > Shallow**.



19 Using the above default settings **Apply** the dialog to create a new, (un-assigned) ***private* Boundary**.



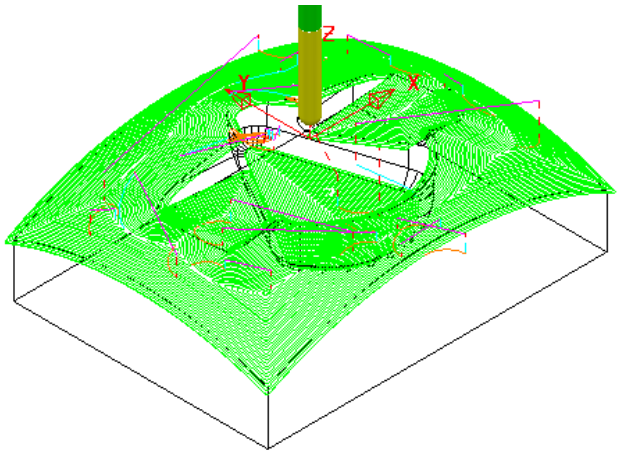
Initially this new ***private* Boundary** will be displayed in the **PowerMILL *explorer*** but will become hidden once it has been applied in a new toolpath calculation.

 Copyright © Delcam plc

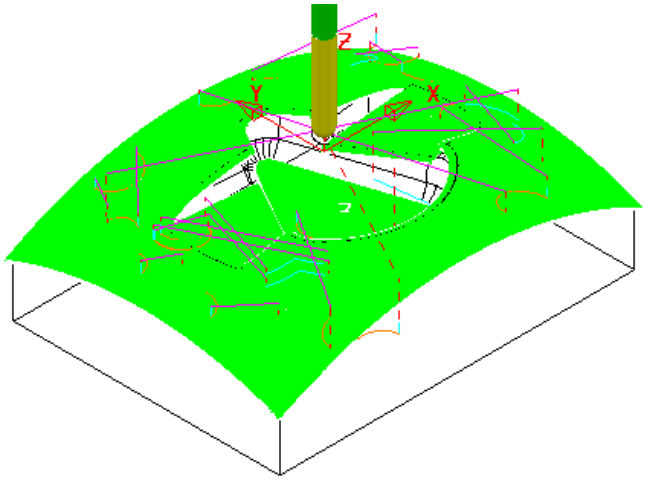
Exercise

20 In the **PowerMILL *explorer***, **Activate** the toolpath and then select **Settings** to open the dialog.

21 ***Recycle*** with the new ***private*** **Boundary** 2, **Active** and with ***Trimming*** set to **Keep inside**.

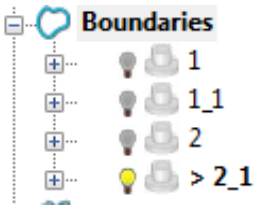


22 ***Make a Copy*** of the above toolpath with the new ***Name*** **BN10-3Doff-FIN1** and with **Tolerance 0.01**, **Thickness 0** and **Stepover 0.3**



23 From the **main** pulldown menus select **Tools** – **Options** and in the dialog select **Boundaries** – **Private boundaries**.

24 **Tick** the box named ***Display private*** **boundaries *in the explorer***.



*This demonstrates that if a **toolpath** containing a **Private Boundary** is copied then the new one will contain a separate **Private Boundary**.*

 Copyright © Delcam plc

7. Levels and Sets

Levels and **Sets** allow you to control the selection and display of model components in the **PowerMILL** graphics area.

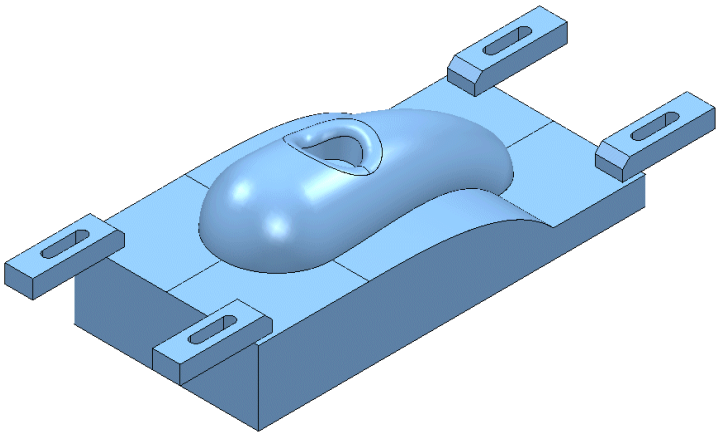
You can ***Acquire*** components from the currently imported **Model** to a **Level** or **Set**. Once acquired to a **Level** or **Set**, you can select, de-select, display, or hide ***Model components*** from the individual **Level** or **Set** context menus.

The subtle differences between Levels and Sets are illustrated using the following examples.

Creating Levels

1. From the **Main** menu, select **File** > **Delete All** and **Tools** > **Reset Forms**.
2. Select **File** > **Import Model**:

   .....\PowerMILL_Data\Models\CowlingWithClamps.dgk



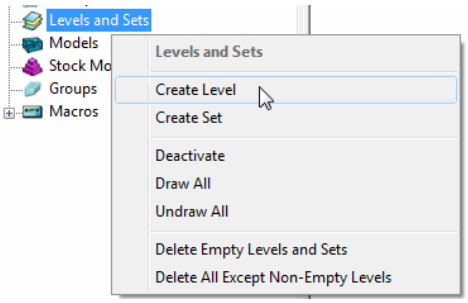
The imported **Model** includes clamps, which:

- must be avoided by all machining strategies created in the **Project**.
- are not included during the calculation of the **Block**.

To make the model, component selection easier, a new **Level** named **Clamps** is created and all clamp surfaces are assigned to it. Another **Level**, named **Splits** is created and all 6 split surfaces are assigned to it.

- Create a Dia 16 tool with a 3 Tip Radius named **D16tr3**.
- In the **PowerMILL *explorer***, right-click **Levels and Sets** and select **Create Level**.

Copyright © Delcam plc



3 **Rename** the new **Level** as **Clamps**.

4 Create another new **Level** and **Rename** it as **Splits**.

All **model** components are currently acquired to the default ***General*** **Level**.

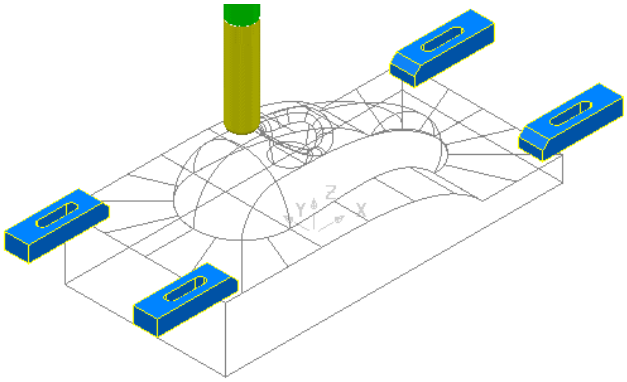
If the original CAD model has components already assigned to different **Levels**, then these will be recreated when the **model** is imported into the **PowerMILL project**.

Acquiring model entities to a Level

5 In the **PowerMILL *explorer***, click on the bulb to switch off the **Level** named ***Clamps***.

Any items acquired to this **level** will no longer be displayed.

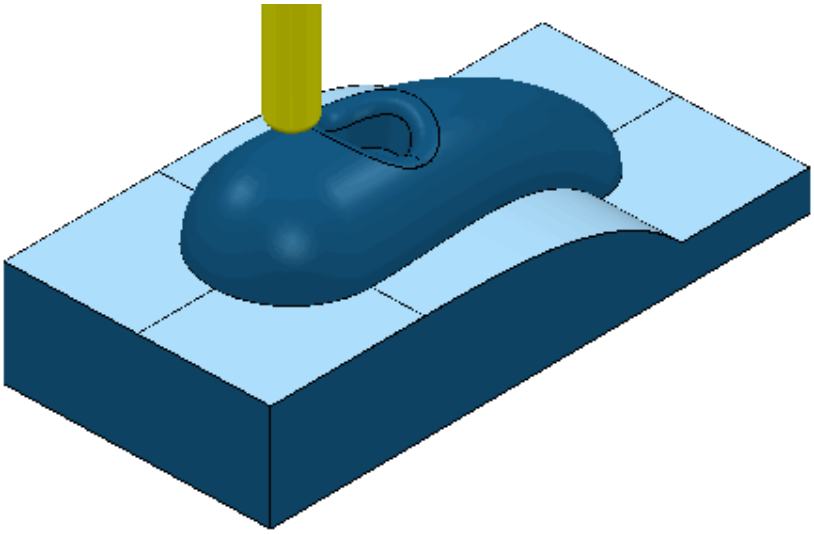
6 In the graphics area, hold down the ***left mouse*** and ***Shift*** keys while selecting the **surfaces** that define the clamps.



7 In the **PowerMILL *explorer***, right-click on the **level** named ***Clamps*** and select ***Acquire Selected Model Geometry***.

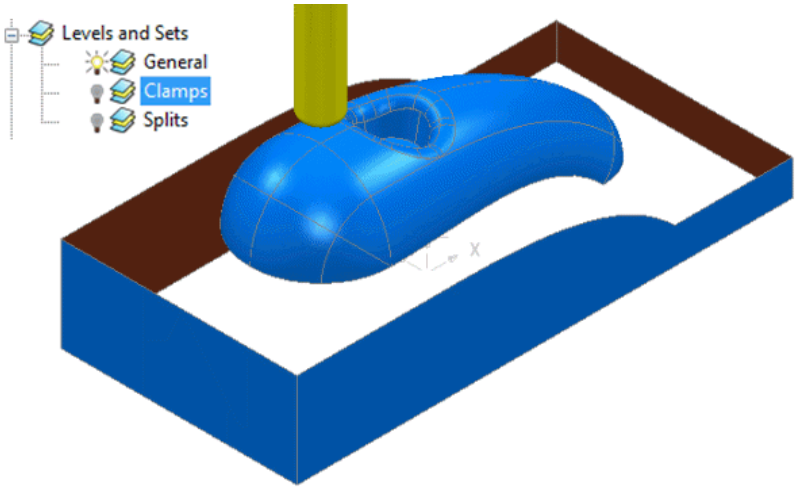
The clamps disappear from the view (The **Level** is switched off).

 Copyright © Delcam plc



8 Select the 6 **Surfaces** defining the split face and ***Acquire*** them to the level named ***Splits***.

9 Switch off the **level** named ***Splits*** to visually check that the correct **surfaces** have been acquired.



10 Switch on all **levels** (The clamps and split **surfaces** reappear).

Now, create a **Selected Surface Boundary** to protect the clamps from being machined.

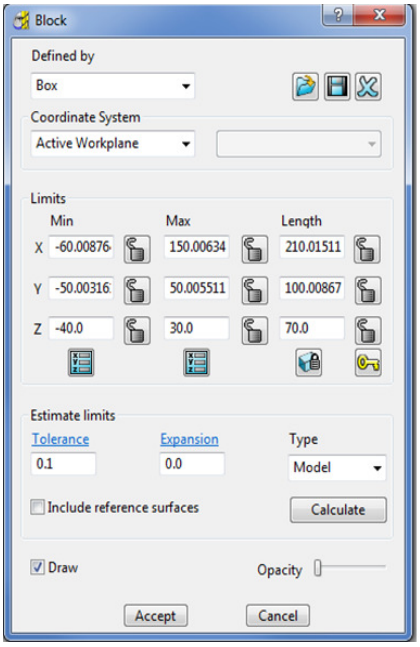
1 In the **PowerMILL *explorer***, right-click the **Level**, ***General*** and choose **Select Surfaces**. The main component **surfaces** excluding the split faces are selected.

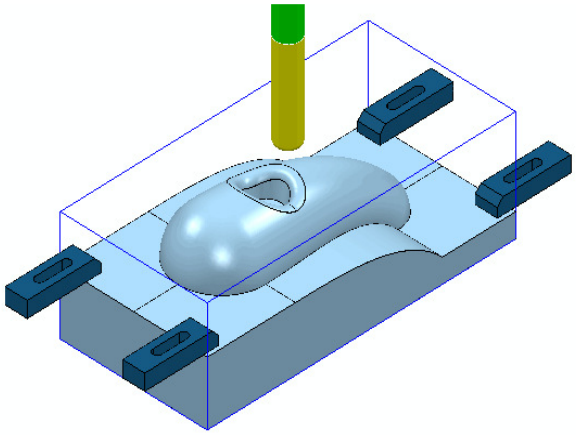
2 While still in the **PowerMILL *explorer***, right-click the ***Splits*** **level** and choose **Select Surfaces**. The split face surfaces are added to the current selection.

3 Calculate a **Block** using the default **Box** and **Model** options.

4 Use the default **Rapid Move Heights** and **Start and End Point** options.

Copyright © Delcam plc



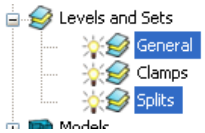


The **Block** is created to the dimensions of the selected surfaces.

5 Left mouse click in the graphics area away from the model to deselect all entities.

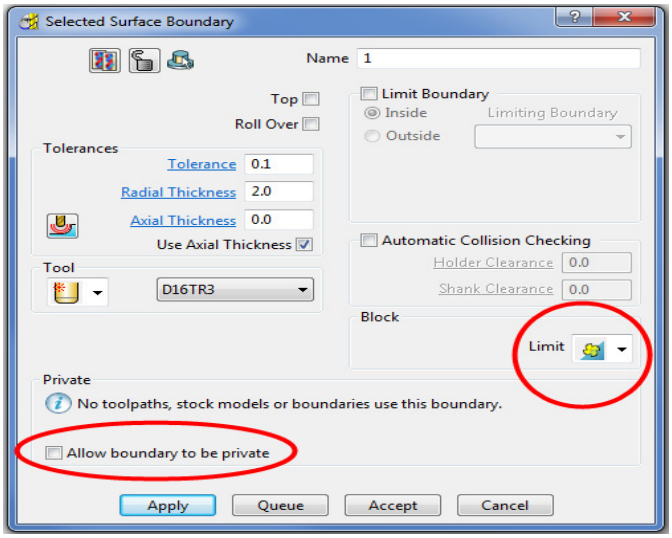
6 In the **PowerMILL *explorer***, click on the **Level**, **General**, and with the **Ctrl** key held down, click on the ***Splits*** **level**.

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

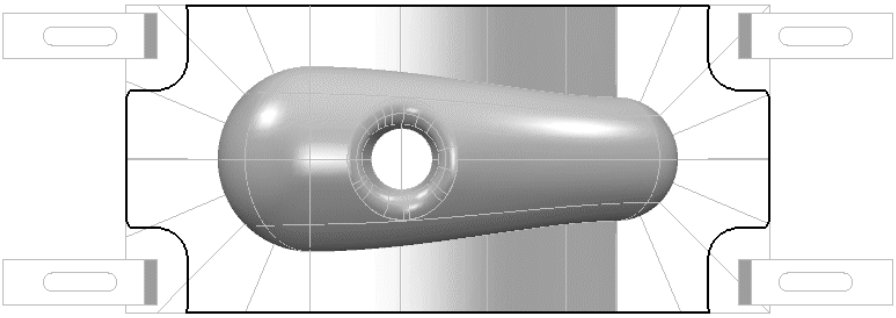


7 Right-click on one of the highlighted levels and choose **Select Surfaces**. All **surfaces** excluding the clamps are selected.

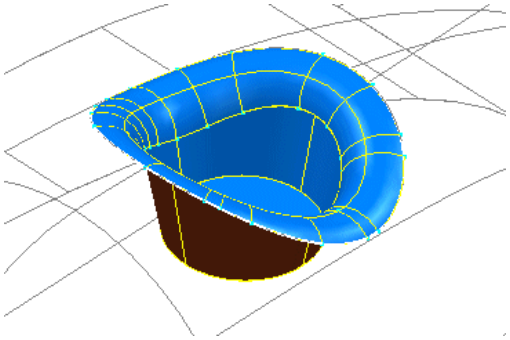
8 **Apply** a **Selected Surface Boundary** with **Radial Thickness** 2, **Limit** set to ***Block Edge***, with ***Allow boundary to be private*** un-ticked.



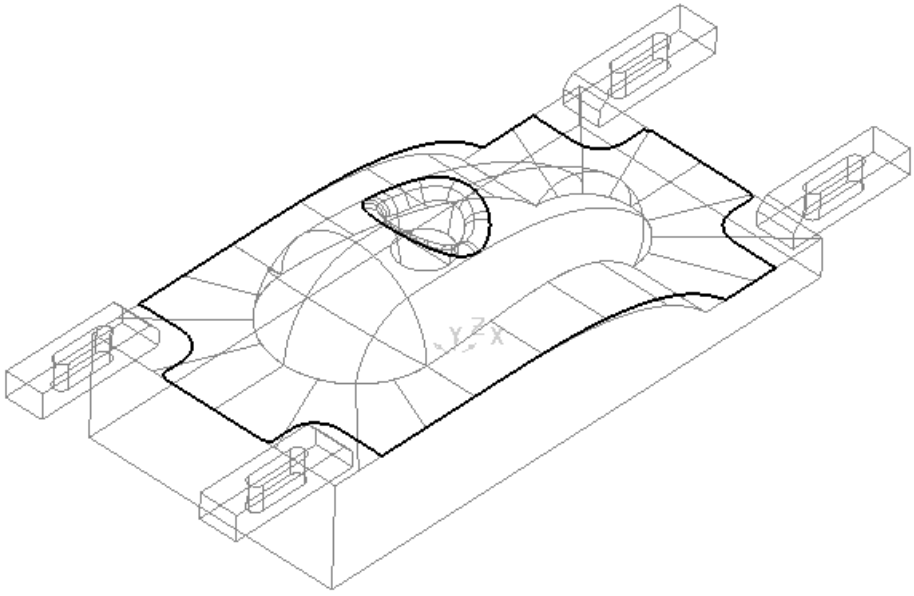
 Copyright © Delcam plc



9 Manually select the **surfaces** defining the central pocket, including the fillet.



10 Right-click the existing **Boundary** (1) and from the local menu, select **Insert** > **Model** to add a new segment.

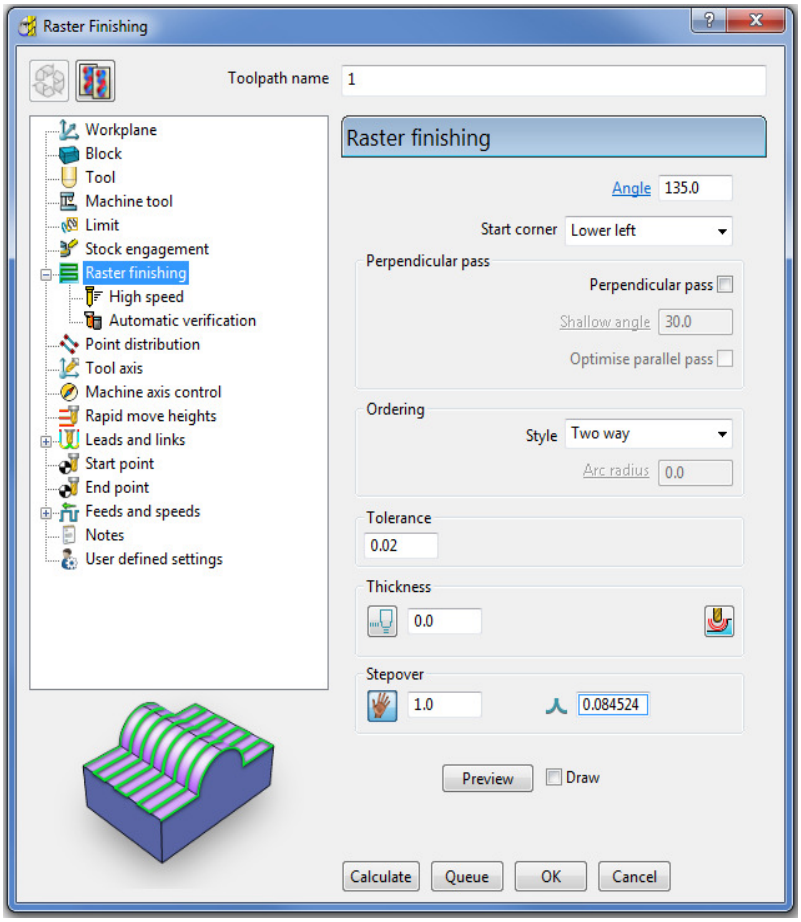


11 Check that both the **Tool** and **Boundary** are **Active** so that they will be automatically included when a new machining strategy is created.

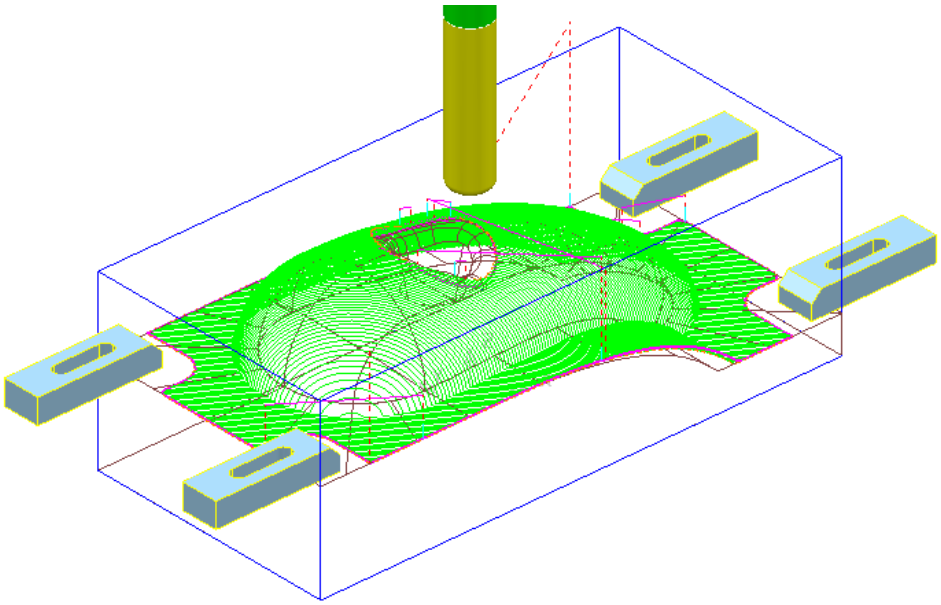
12 From the **Strategy Selector** select the **Finishing** tab and in the dialog select, **Raster Finishing**.

13 In the **Raster Finishing** dialog, enter data exactly as shown in the following illustration.

Copyright © Delcam plc



14 Click **Calculate** to process the **toolpath** and then **Close** the dialog.

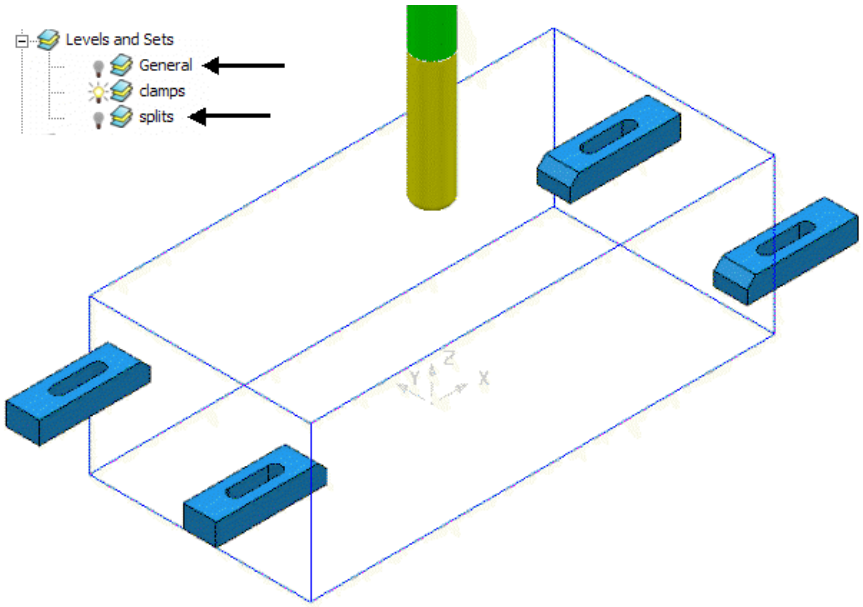


The **Radial Thickness** of 2 used in the outer **boundary *segment*** provides suitable clearance around the clamps.

 Copyright © Delcam plc

Switching Off a Level

1 In the **PowerMILL *explorer***, switch off the light bulbs on both the **General** and **Splits** levels.

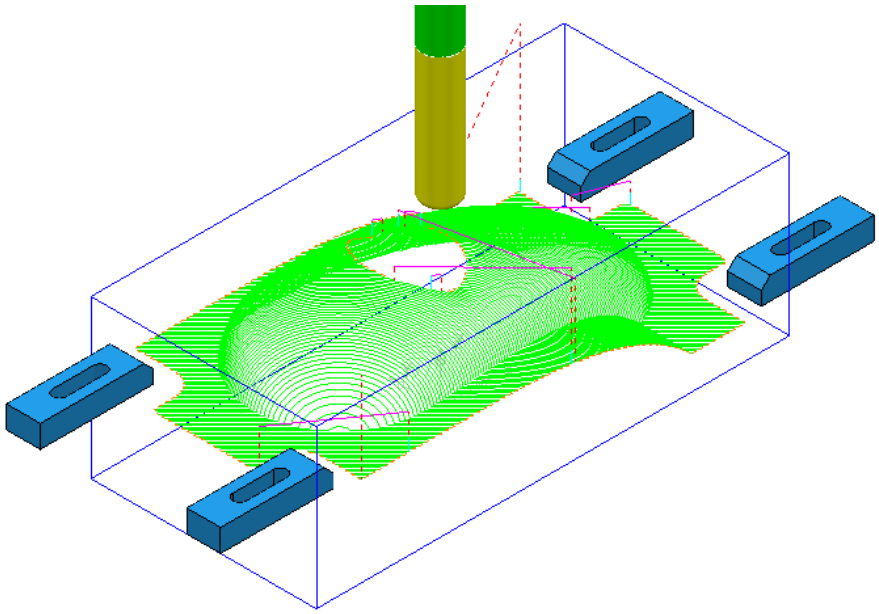


Only the ***Clamp*** **Surfaces** are displayed in the graphics area.

2 Right-click the **toolpath** and select **Settings** to re-open the dialog.

3 Select create a ***Copy*** and click **Calculate** to process a new **toolpath**.

4 **Close** the **Raster Finishing** dialog.



*Even if a **level** is switched off, **PowerMILL** will still create a toolpath over all (including un-displayed) parts of the model.*

Sets

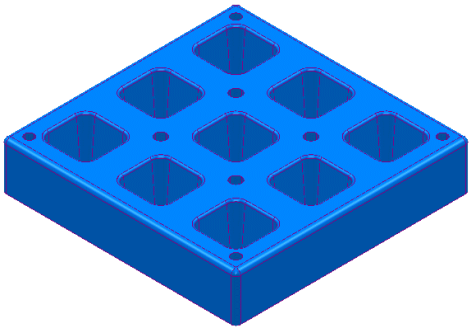
A **Model** entity will always exist on a **Level** but not on more than one at the same time. You cannot delete a **Level** while **Model** entities are acquired to it.

A **Model** entity can co-exist on more than one **Set** at the same time. You can delete a **Set** while **Model** entities are acquired to it.

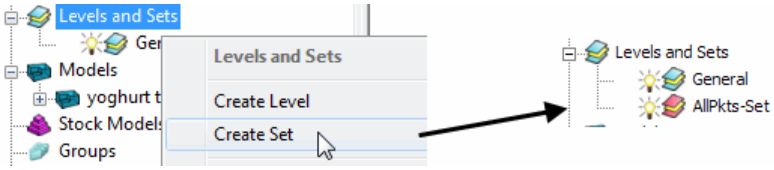
1 From the **Main** menu, click **File** > **Delete All** and **Tools Reset Forms**.

2 From the **Main** pulldown menus **File** > **Import Model** and select:

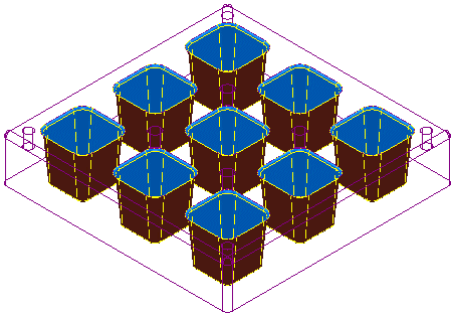
.....\PowerMILL_Data\Models\YogurtTray.dgk



3 Create a **Set** and rename it as **AllPkts-Set**.

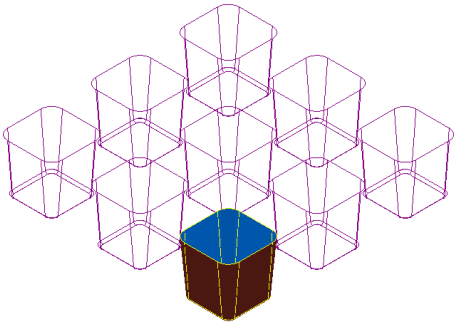


4 Manually select all **surfaces** defining the 9 pockets.



5 With the 9 pockets still selected, right-click **AllPkts-Set** and select **Acquire Selected Model Geometry**.

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



Copyright © Delcam plc

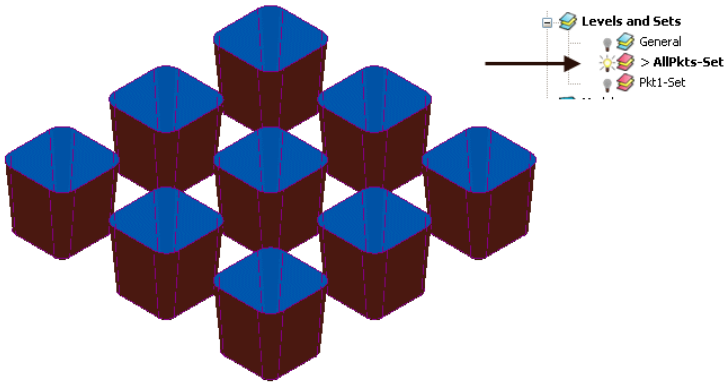
The selected **surfaces** will soon be acquired to two **Sets** simultaneously.

7 Create a new **Set** and rename it as **Pkt1-Set**.

8 Right-click **Pkt1-Set** and select **Acquire Selected Model Geometry**.

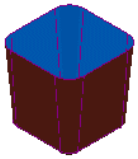
9 Switch off both the **level *General*** and the **Set *Pkt1-Set***.

10 Switch on **AllPkts-Set**.



All **surfaces** currently acquired to **AllPkts-Set** are displayed, including those that exist on the visually 'switched off' **Set**.

11 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 displayed when the **AllPkts-Set** is switched on and **Pkt1-Set** is switched off. This illustrates that model entities can be acquired to more than one **Set** at the same time.

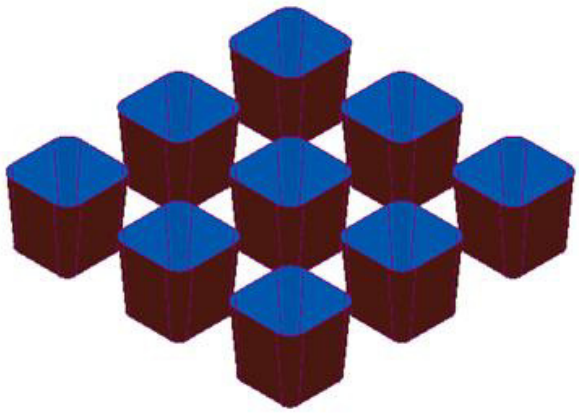
1 Switch off **Pkt1-Set** and then switch on **AllPkts-Set**.

2 Right-click **AllPkts-Set** and from the local menu click **Select Surfaces**.

3 Create a new **Level** and rename it as **AllPkts-Lev**.

4 In the **PowerMILL *explorer***, right-click **Level *AllPkts-Lev*** and select **Acquire Selected Model Geometry**.

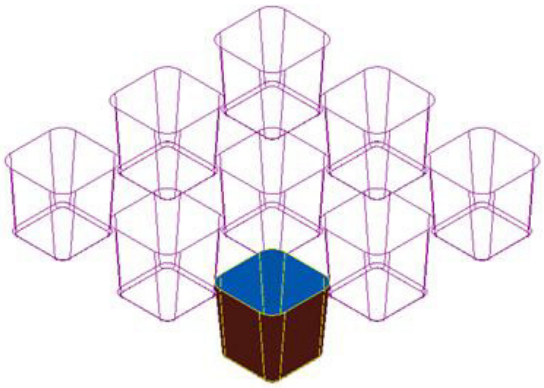
5 Switch off all **Levels** and **Sets** except **AllPkts-Lev**.



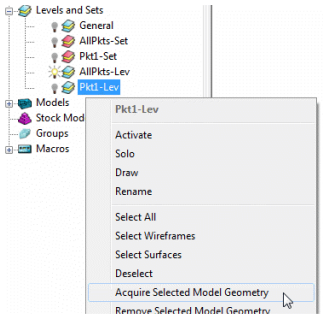
All surfaces acquired to **AllPkts-Lev** are displayed.

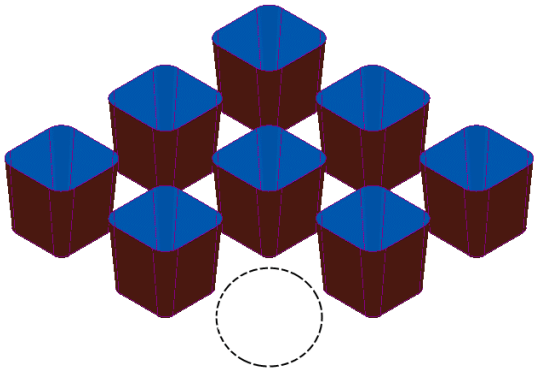
6 Create a new **Level**, rename it as **Pkt1-Lev**, and then switch it off.

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



8 In the **PowerMILL *explorer***, right-click **Pkt1-Lev** and from the pop-up 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 exist in more than one **Level** at any one time.

Switch on the **level *Pkt1-Lev*** and the bottom left pocket surfaces are displayed again.

*When using a combination of **Levels** and **Sets** it is good practice to keep the **Sets** visually switched off. If you want to select items from a particular **Set**, then, right-click on it and use one of the **Select** options from the context menu.*

*This prevents confusion if you switch a **Level** off only to find the included items are still visible due to a **Set** containing the same items being switched on.*

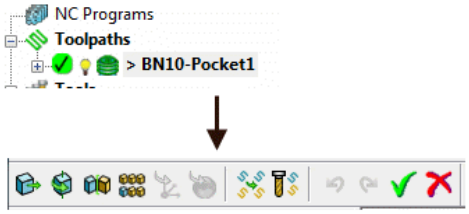
 Copyright © Delcam plc

8. Editing Toolpaths

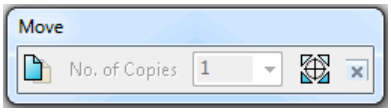
Edit > Transform

The **Transform** option allows you to **Move**, **Rotate**, and **Mirror** toolpaths relative to an active **workplane**.

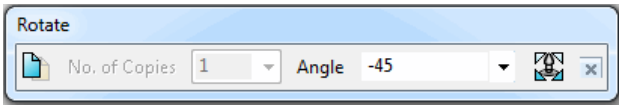
To access the options right-click on a **toolpath** and select **Edit** > **Transform**.



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

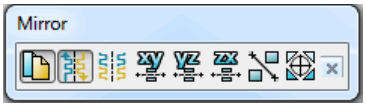


- **Transform > Rotate** - The **Rotate** function allows the rotation of a selected toolpath by a user-defined ***Angle*** around a selected **X**, **Y**, or **Z** axis.



- **Transform > Mirror** - This provides 3 options to ***Mirror*** a selected toolpath across the **XY**, **XZ**, or **YZ** planes. When a toolpath is mirrored, PowerMILL identifies the gouge status to be unknown, so it is advisable to check for gouges.

It is important to note that when a toolpath is mirrored, the cutting direction effectively becomes reversed.

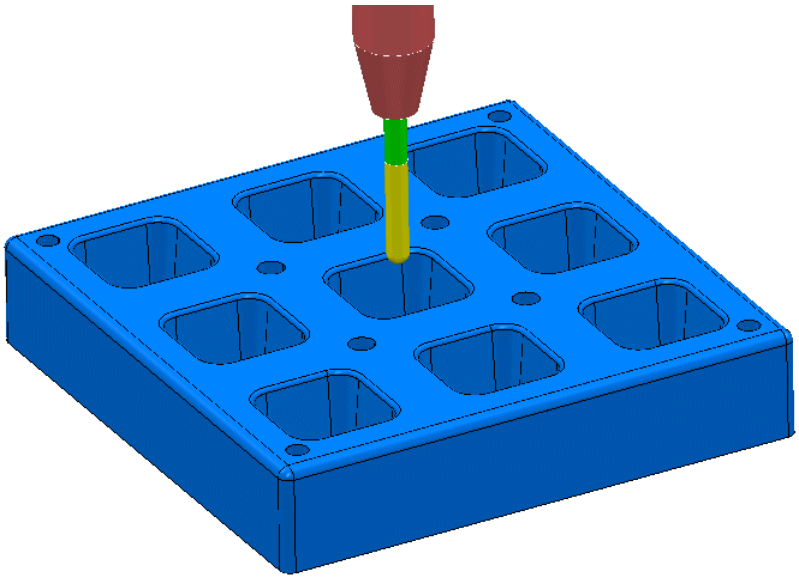


In the following example a single toolpath is created and then the **Transform > Move** option applied to copy toolpaths into the remaining cavities.

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

2 Select **File > Open Project**:-

.....\PowerMILL_Data\Projects\EditToolpaths_1



The **Project** contains a multi-cavity die and a **Dia 10 Ball Nosed** tool.

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

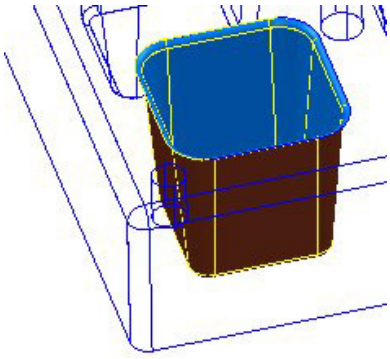
3 **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\Transform

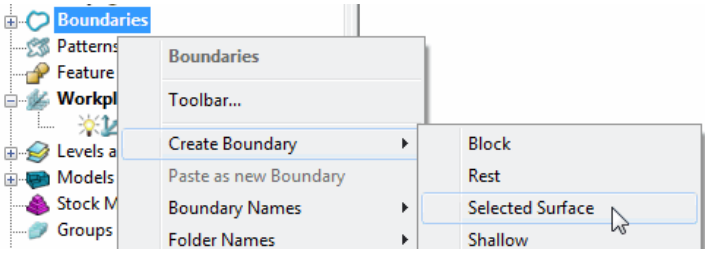
4 Create a **Block** using **Defined by > Box** and **Type > Model**.

5 Reset the **Rapid Move Heights**.

6 Select all of the **surfaces** (including the fillet) that define the bottom left pocket.

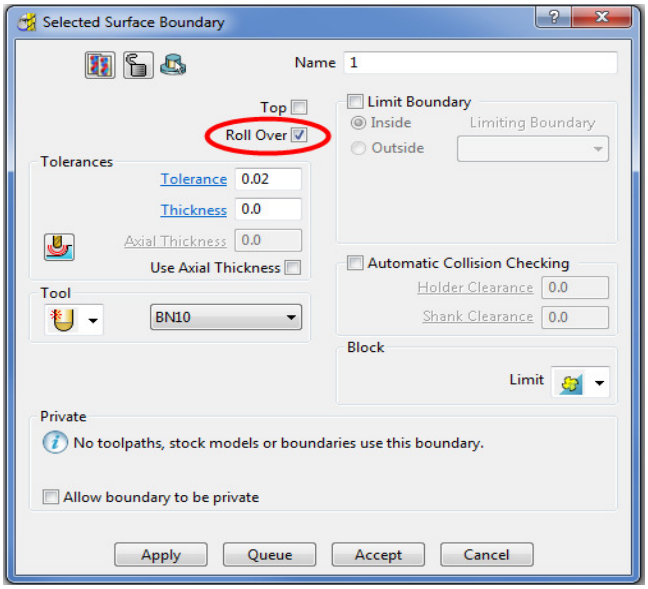


7 In the **PowerMILL *explorer***, right-click **Boundaries** and select **Create Boundary > Selected Surfaces**.



8 In the **Selected Surface Boundary** dialog, tick **Roll Over** and enter a **Tolerance** 0.02

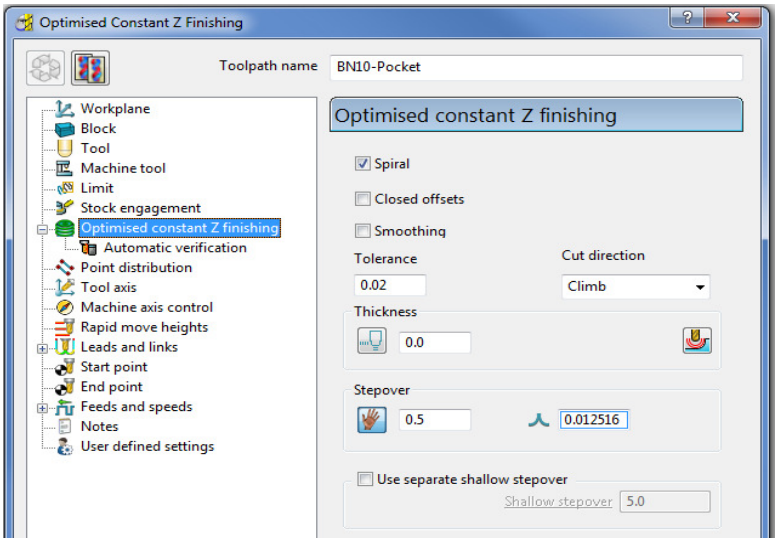
 Copyright © Delcam plc

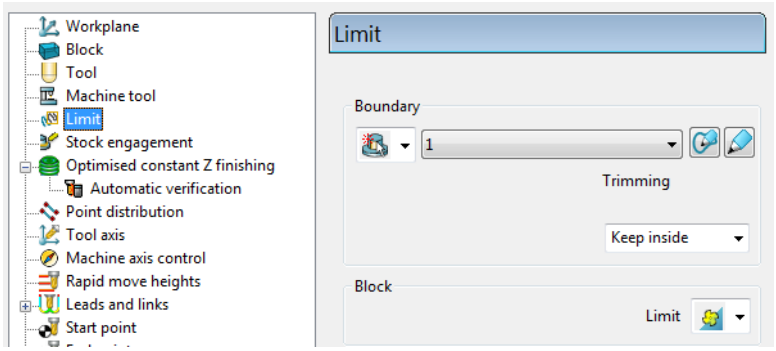


9 Click **Apply** and **Accept** the dialog.

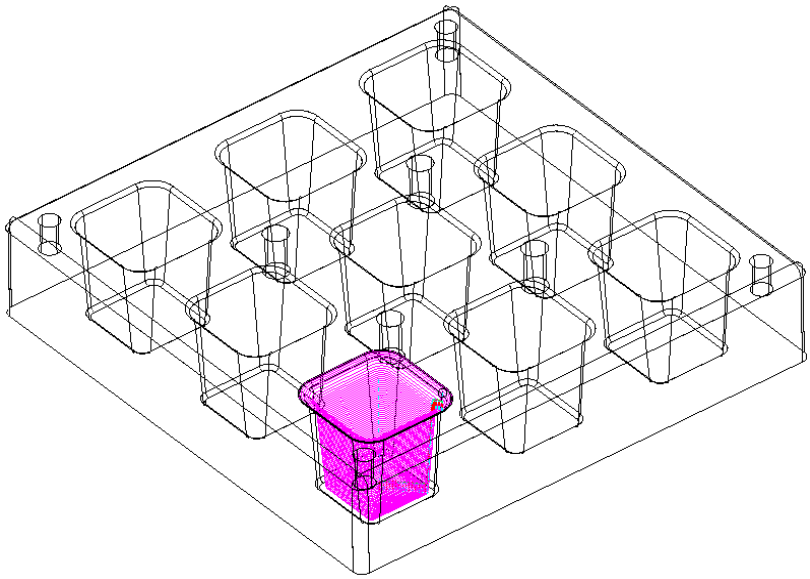
A **Selected Surfaces Boundary** defines the limit where the selected tool and associated parameters would fully machine the selected surfaces.

10 Open the **Optimised Constant Z Finishing** strategy and enter the values and settings exactly as shown in the 2 pages below:





11 Click **Calculate** to process the toolpath and then **Close** the dialog.



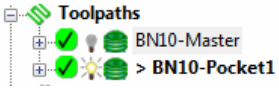
12 In the **PowerMILL *explorer***, right-click the **toolpath BN10-Pocket** and select **Simulate from Start**, and click **Play** in the **Simulation** toolbar.

Transforming toolpaths

Before transforming toolpaths, it is advisable to create a master copy of the original.

1 Right-click the **BN10-Pocket** toolpath and select **Edit > Copy Toolpath** and **Rename** the copy **BN10-Pocket_1** as **BN10-Master**.

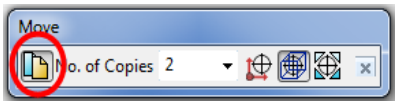
2 In the **PowerMILL *explorer*** select the toolpath **BN10-Master** and with the left mouse key depressed, drag it onto **Toolpaths** to move it to the top of the list.



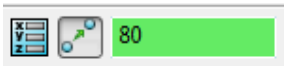
3 In the **PowerMILL *explorer***, right-click on the **toolpath BN10-Pocket** and select **Edit > Transform** to display the **Toolpath Transform** toolbar.

4 On the **Toolpath Transform** toolbar, click **Move** .

5 On the **Move** toolbar, click **Keep original** with **No. of Copies** as **2**.



6 In the data entry box located below the graphics area, enter the ***relative distance*** along X as **80**, and press ***Enter*** on your keyboard.

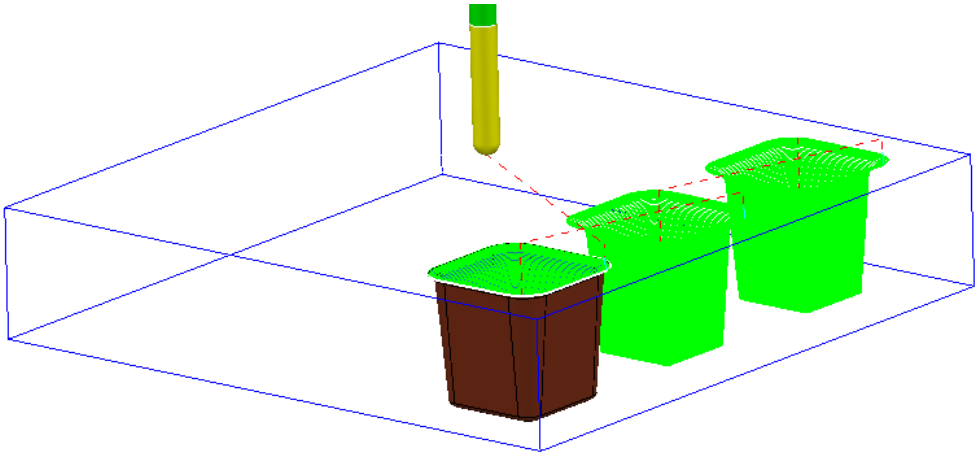


X Y Z values (separated by a space) are entered into this box as required.

7 In the **Transform** toolbar, click to close and accept the changes.

Copyright © Delcam plc

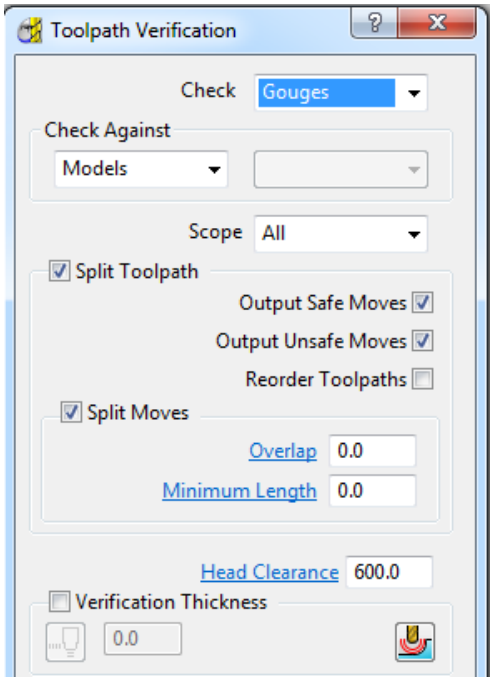
A new **toolpath** named **Pocket_1** has been created that includes 3 copies including the original, transformed along X by a distance of **80**.



Gouge checking a toolpath

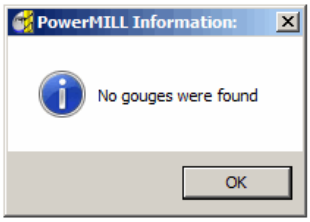
A ? next to the toolpath indicates that it has not been gouge checked directly to the model.

1. In the **PowerMILL *explorer***, activate the **toolpath *BN10-Pocket_1***.
2. Right-click on the **toolpath *BN10-Pocket_1*** and from the local menu select **Verify > Toolpath**.
3. In **Check** select the **Gouges** option.

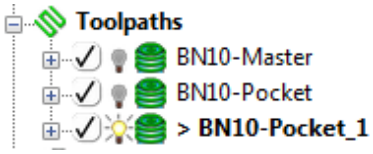


4. **Apply** the **Toolpath Verification** dialog.

A **PowerMILL information** box is displayed to inform of the gouge check status. In this instance, ***No gouges were found***.



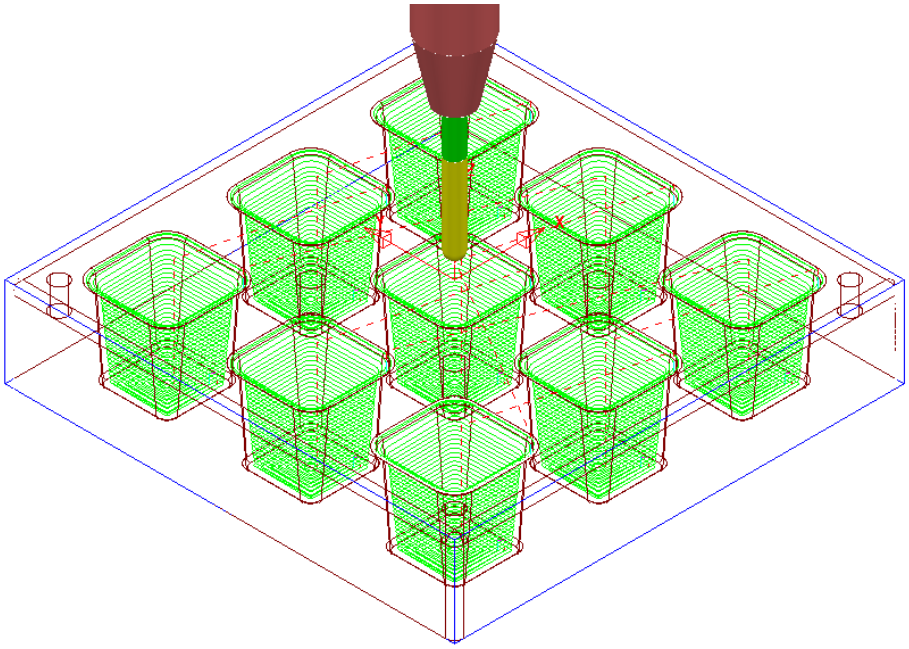
In the **PowerMILL *explorer***, the style of the icon also updates indicating that the **toolpath** is gouge free.



Exercise

5 Delete the toolpath **BN10-Pocket_1**.

6 Use the **Multiple Transform** option to create **9** copies of the toolpath **BN10-Pocket1** as a **3**x**3** matrix along X and Y with an incremental value of **80**.



7 Apply **Verify** on the new toolpath **BN10-Pocket_1** to check for for gouges.

8 **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\Transform

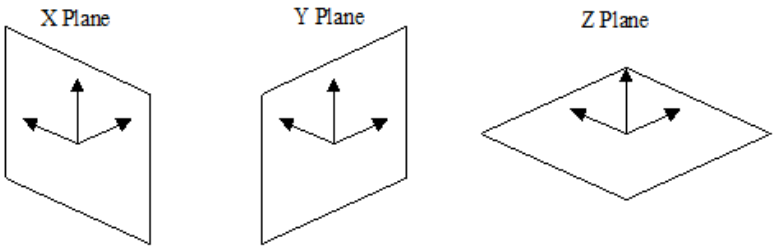
Copyright © Delcam plc

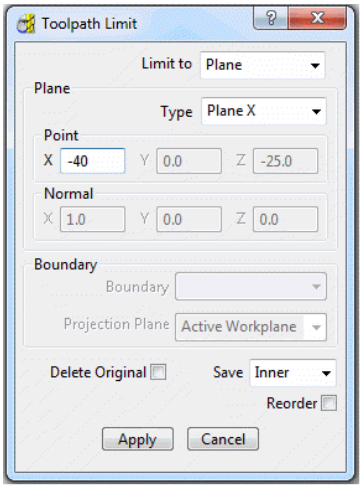
Edit > Limit...

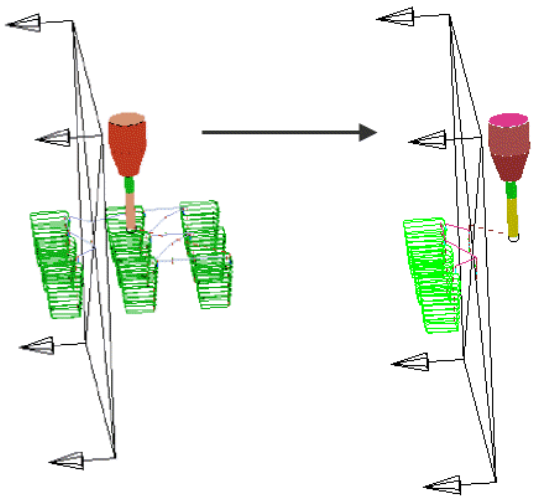
Edit > Limit provides a series of options to retrospectively, trim a toolpath to a **Plane**, **Polygon**, or **Boundary**.

Edit > Limit > To a Plane

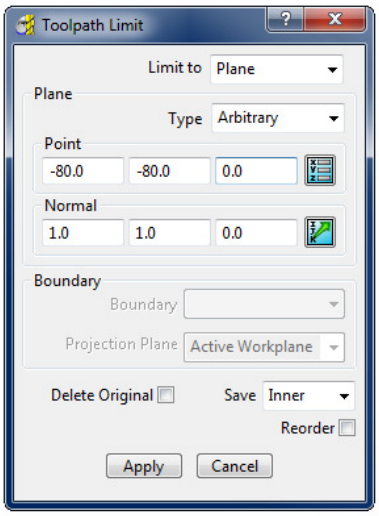
Edit > Limit to - ***Plane*** allows the user to select a plane at specified distance along the **X**, **Y**, or **Z** axis.

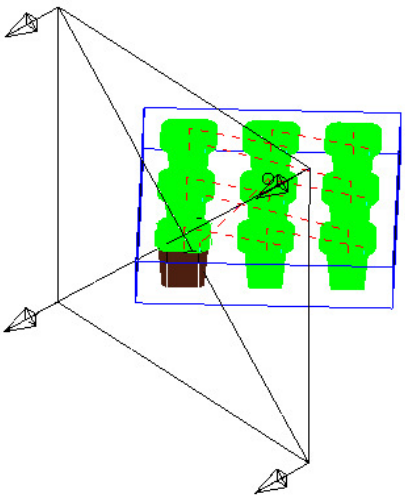






The **Edit > Limit to** - **Arbitrary** option allows the user to specify an X,Y,Z ***Point*** and a user defined, ***Normal*** vector for the limiting plane.



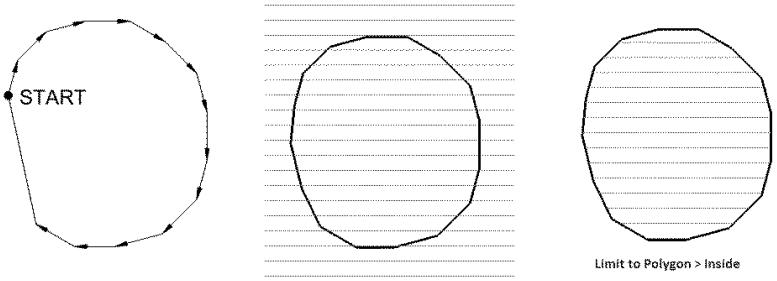


Edit > Limit > to a Polygon

Using the mouse, you can sketch polygons with any number of sides.

This allows complex areas to be defined, with the option to save the Inside, Outside, or Both sides of the polygon.

*Before making a polygon ensure that the snap filter is set to pick **Anywhere**.*

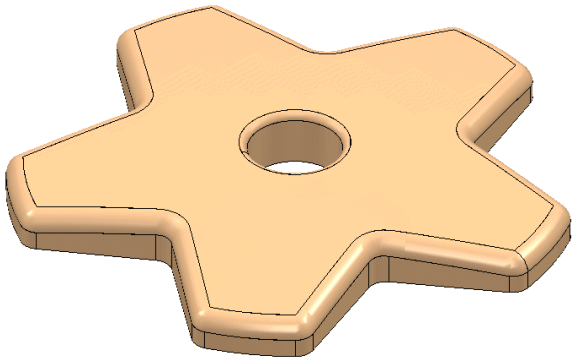


Edit > Limit > to a Boundary

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.

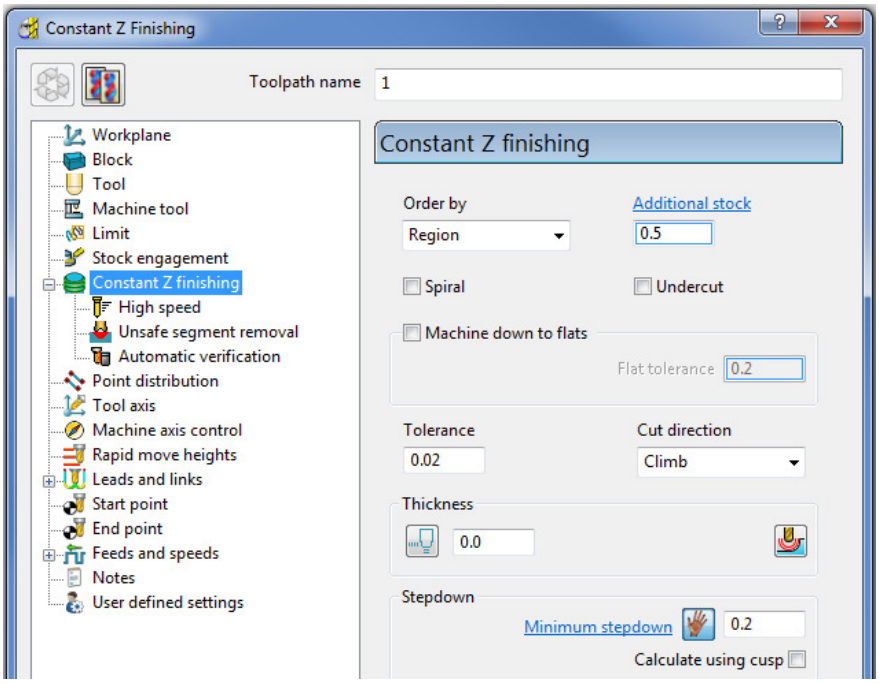
1. Select **File - Delete All** and **Tools -** Reset **Forms**.
2. **Import** the **model**:

   .....\PowerMILL_Data\Models\Handle.dgk



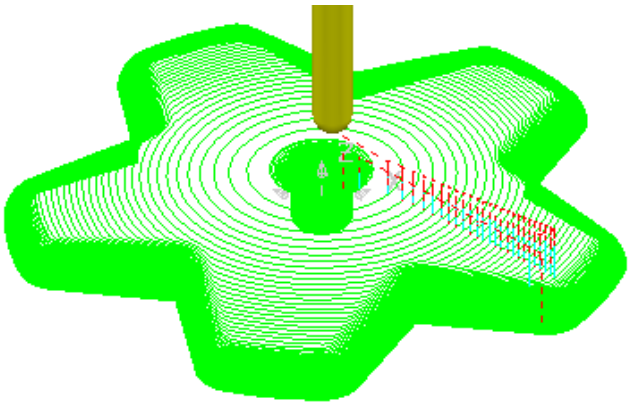
3. **Calculate** a **Block** to the ***Max/Min Limits*** of the **model** and then:
   a. Lock the **Z Min** and **Z Max**.
   b. Enter **Expansion 10**.
   c. Click **Calculate** again.
4. Define a Dia 10mm Ball Nose named **BN10**.
5. In the **Rapid Move Heights** dialog, click **Calculate** to reset the tool to a safe height.
6. From the **Strategy Selector > Finishing** options, open the **Constant Z Finishing** dialog and enter data exactly as shown in the next illustration.

Copyright © Delcam plc



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

The created **Constant Z finishing** strategy is not suitable for shallow areas of the component.

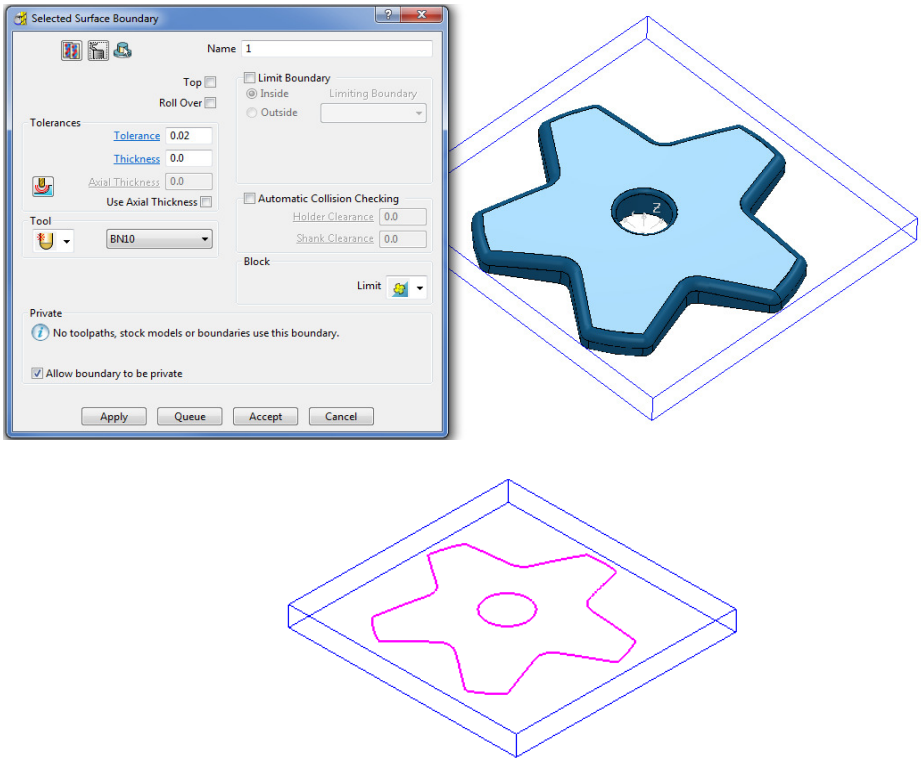


To improve the toolpath, a **Boundary** will be created and the **toolpath** will then be retrospectively limited to the steep areas.

8 Select the main upper surface of the **model**.

9 In the **PowerMILL *explorer***, right-click **Boundaries** and select **Create Boundary - Selected Surface** to open the **Selected Surface Boundary** dialog.

10 Enter the data exactly as shown in the dialog and click **Apply** and then **Accept** to close the dialog.



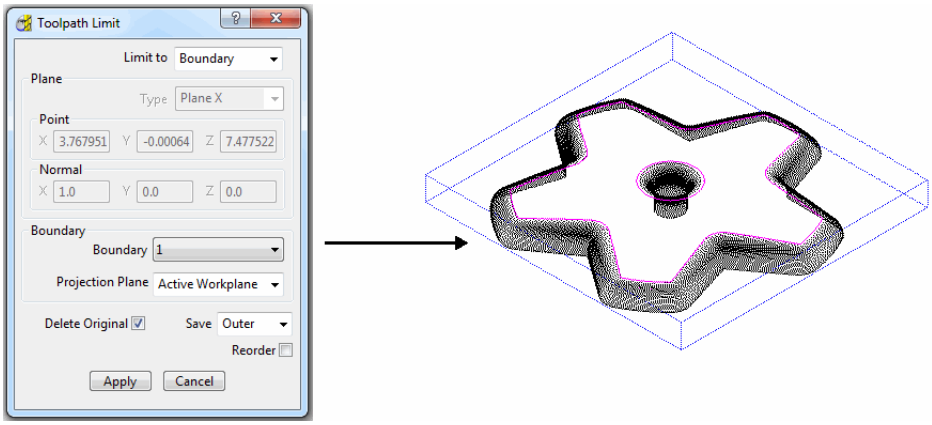
11 In the **PowerMILL *explorer***, right-click on the active **toolpath** and select **Edit > Limit**.

12 In the **Toolpath Limit** dialog which appears:

a From the **Limit to**, select ***Boundary***.

b In the **Boundary** input list select **1**.

c Select **Save** – **Outer** and then click **Apply** to create a copy of the toolpath outside **Boundary** 1.



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

*If the **Delete Original** option is selected, it causes the deletion of the original toolpath once the new toolpath is created.*

If the toolpath is re-Calculated any retrospective limiting will not apply to the new or edited strategy.

 Copyright © Delcam plc

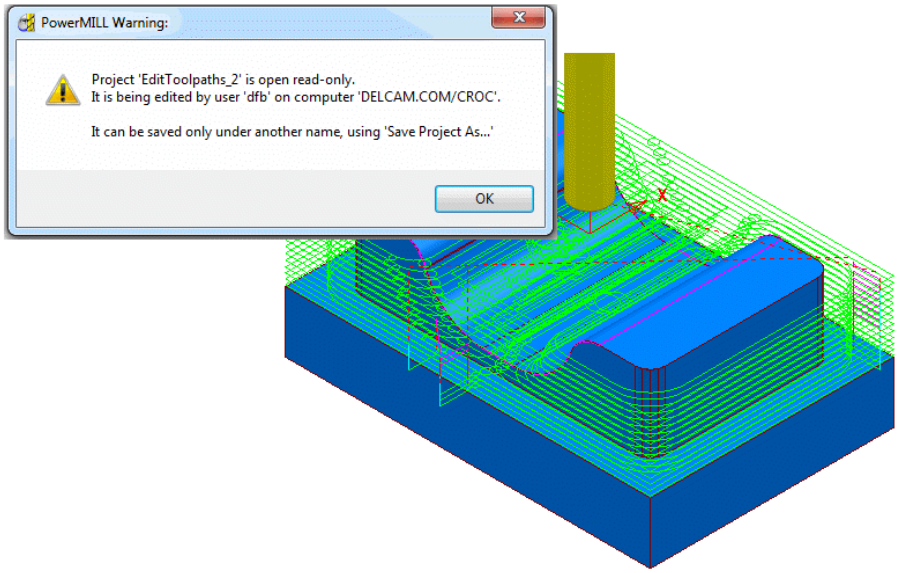
Changing the Order and Direction within Toolpaths – EX1

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 the part, and progresses upwards, reversing the order will change the tool track sequence to start at the top of the part and make it progress downwards.

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

2 From **File > Open Project** select the read-only **Project**:

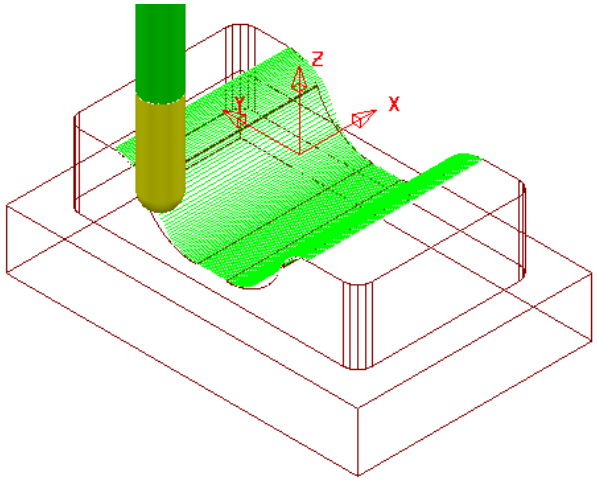
.....\PowerMILL_Data\Projects\EditToolpath_2



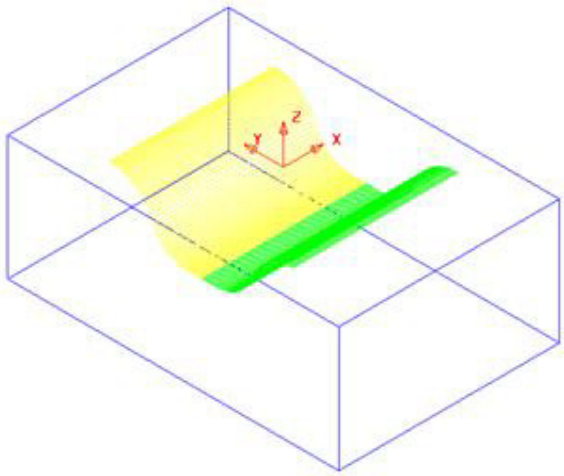
3 Select **File > Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\EditToolpath-EX1

4 In the **PowerMILL *explorer***, right mouse click on the **toolpath bn12-finish-a1** and then from the local menu select **Simulate from Start**.

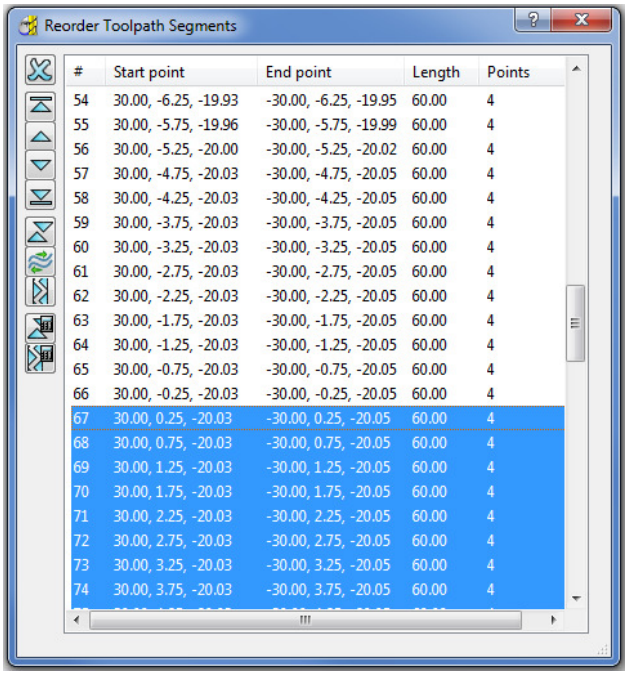


The tool ***Climb Mills*** across the form, stepping downwards to the base before stepping up the other side.



1 Left-click to select all tool tracks on **bn12-finish-a1** beyond the centre of the form (Yellow in the illustration above).

2 In the **PowerMILL *explorer***, right-click the active **toolpath**, ***bn12-finish-a1*** and from the local menu select **Edit > Reorder** to open the following dialog.



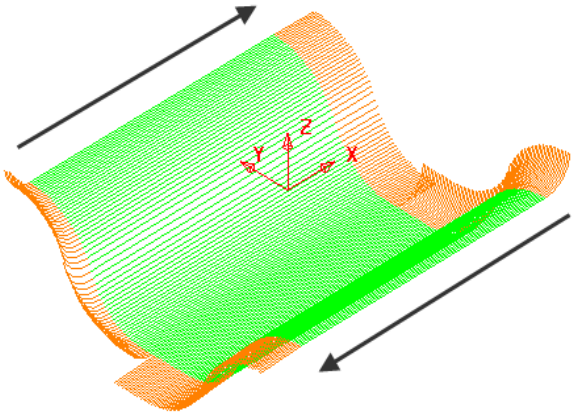
All existing tool tracks (identified by number) are listed in the current order of machining.

The selected tool tracks are highlighted (blue) in the dialog.

3 Click on the **Reverse Order** button to reverse the sequence of the selected tool tracks.

4 Click on the **Reverse Direction** button to reverse the direction of each selected tool track.

 Copyright © Delcam plc



The **Order** and **Direction** of the toolpath has been edited so that it tracks from the outer edges of the form downwards towards the centre of the base.

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

5 **Save Project As**:

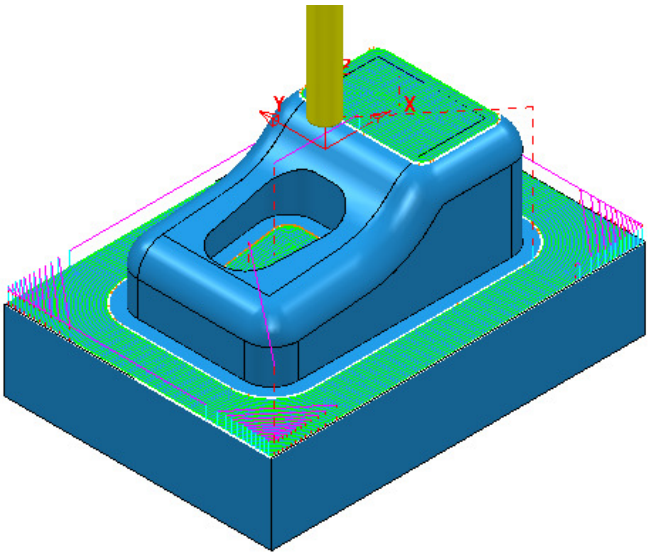
.....\COURSEWORK\powerMILL-Projects\EditToolpath-EX1

Changing the Order and Direction within Toolpaths – EX2

1 From the **main** menu, select **File - Delete All** and **Tools - Reset Forms**.

2 **Open** the read-only **Projec**t:-

.....\PowerMILL_Data\Projects\EditToolpaths_3

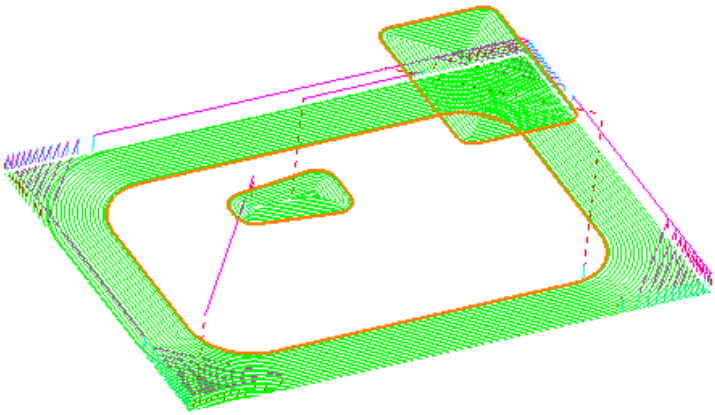


3 **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\EditToolpath-EX2

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

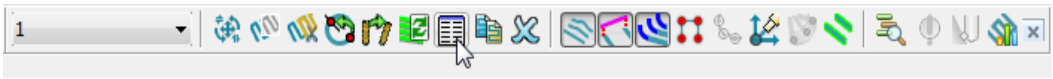
4 **Simulate** the **Toolpath** using the Slow setting.



Observe the current direction and ordering of the strategy considering potential improvements.

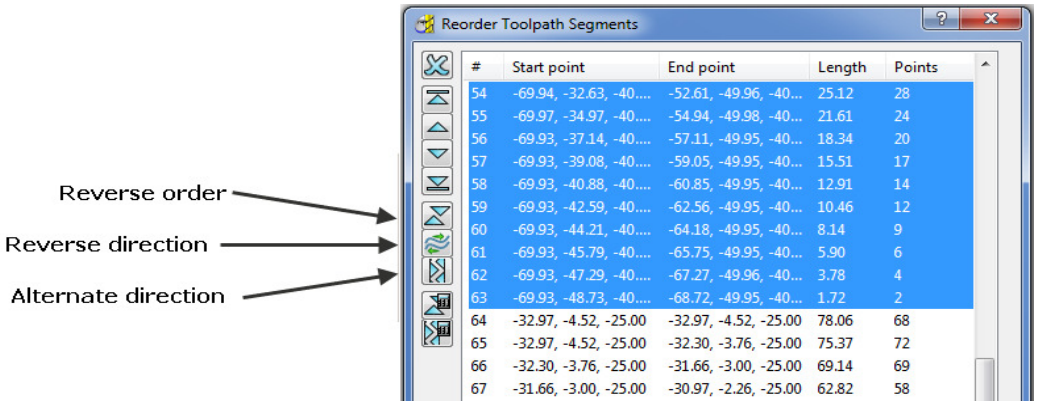
For example, the lower area machining would benefit from climb milling inwards towards the component form and the pocket machining could start central and climb mill outwards towards the sidewall.

5 From the **Main** menu, select **View > Toolbar > Toolpath** to display the **Toolpath** Toolbar.



6 On the **Toolpath Toolbar** Click the **Reorder Toolpath** button. This displays the Toolpath Listing on the Toolpath Toolbar dialog.

The selected segments are shown highlighted in the dialog.



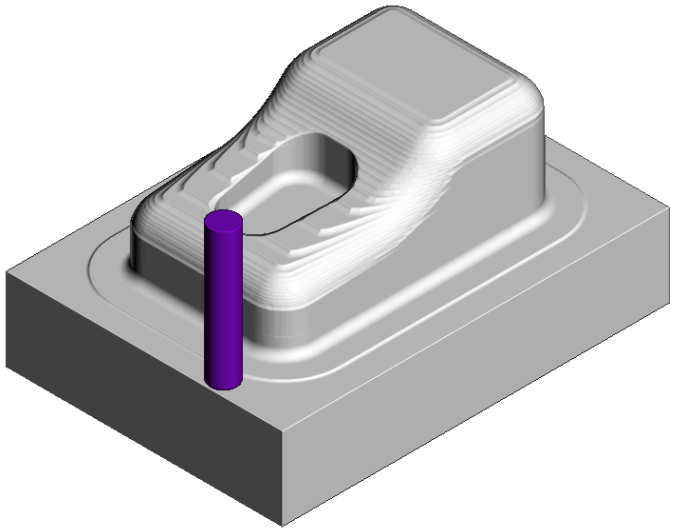
Each segment is listed in the order of execution. If a toolpath segment is selected on the list, the corresponding segment is highlighted in the graphics area (and vice versa). You can modify or move the selected toolpath to another position in the pecking order.

You can use the buttons on the left hand side of the dialog for changing the order and direction of selected segments. If no segments are selected, the whole ot the toolpath will be edited.

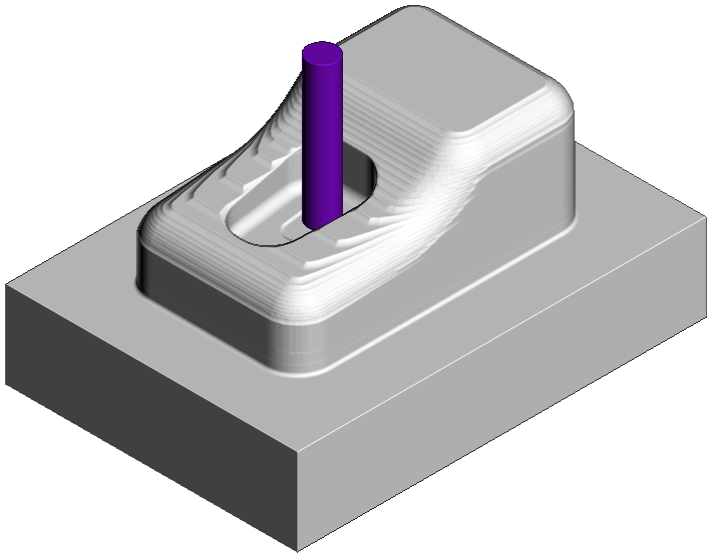
Copyright © Delcam plc

The 2 buttons to the lower left of the dialog are **Automatic Reorder** and **Automatic Reorder and Reverse**. These can only be applied to the whole toolpath and not on selected tool tracks. They are designed purely to minimise air moves, since unnecessary air moves are detrimental to the order or direction of tool tracks.

7 Select all tool tracks in the lower area and select **Reverse Order**. This part of the strategy should now climb mill inwards, towards the main component dialog. If not 'climb milling', then apply **Reverse Direction**.



8 Select the tool tracks 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.



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

Moving Start Points

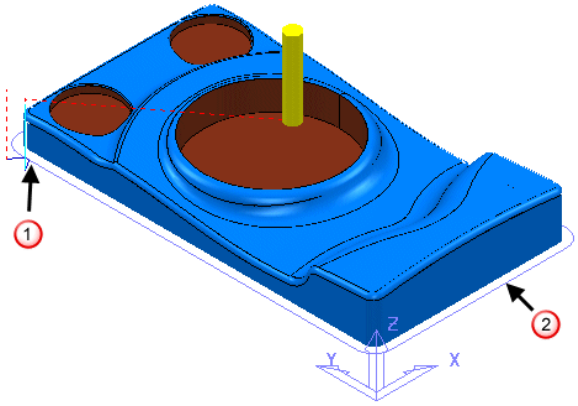
Start points on the tool tracks can be moved to provide a more suitable position for applying leads. This is achieved by defining a line that crosses the new start positions on the toolpath.

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

2 Select **File - Open Project**:

.....\PowerMILL_Data\Projects\limiting-example

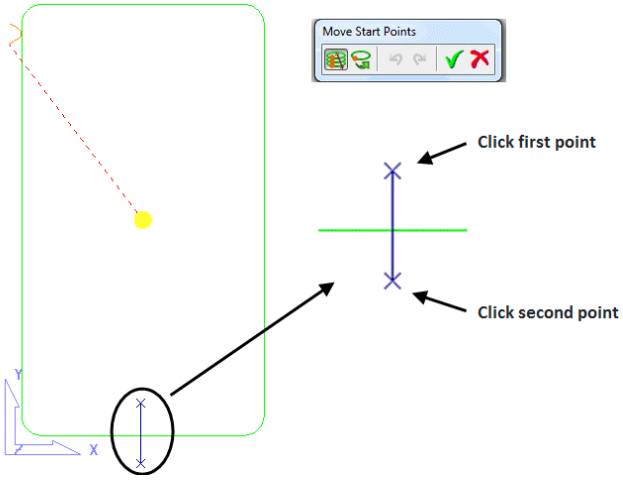
3 Select an **ISO1** view.



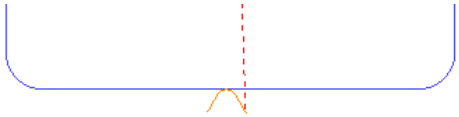
The toolpath start point (1) needs to be moved to (2).

4 Select a **View from top (Z)**.

5 Right-click toolpath 1 and select Edit > Move Start Points. Click and select the two points as shown.



6 Click ✓ to accept the change.



This has moved the start position.

Copyright © Delcam plc

Moving multiple points

It is possible to apply the **Move start points** option on the toolpath to more than one position.

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

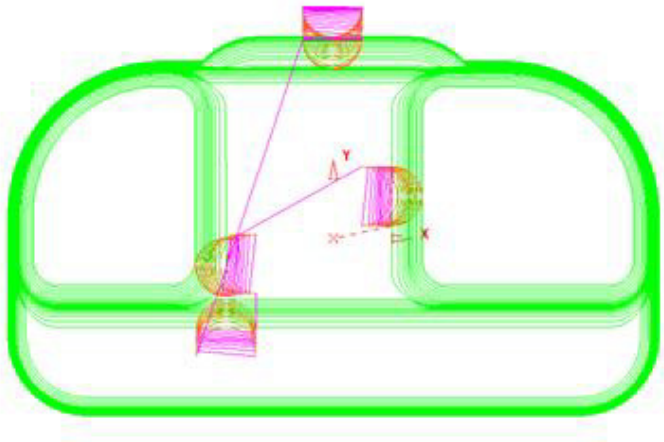
2 Open the **Project**:

.....\PowerMILL_Data\Projects\PunchForm10-Start

3 **Save Project As**:

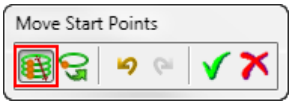
.....\COURSEWORK\PowerMILL_Projects\PunchForm10

4 Select a **View from top (Z)**.



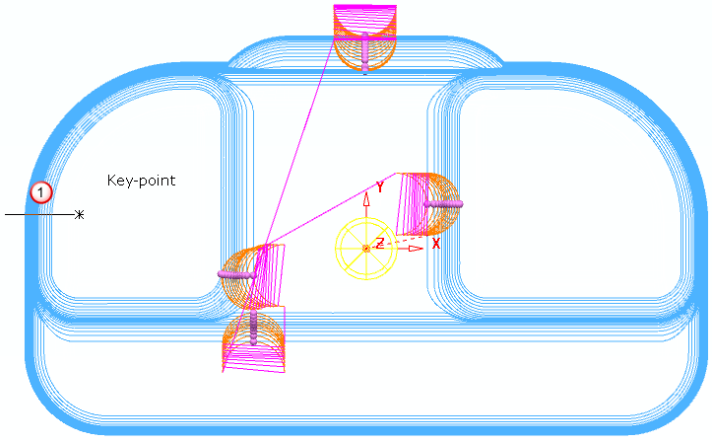
5 Right-click the ***active*** **toolpath** and from the local menu select **Edit > Move Start Points** (The ***Move Start Points*** toolbar is displayed).

6 If not already selected, on the toolbar, click the Move Start Points by drawing a line button.

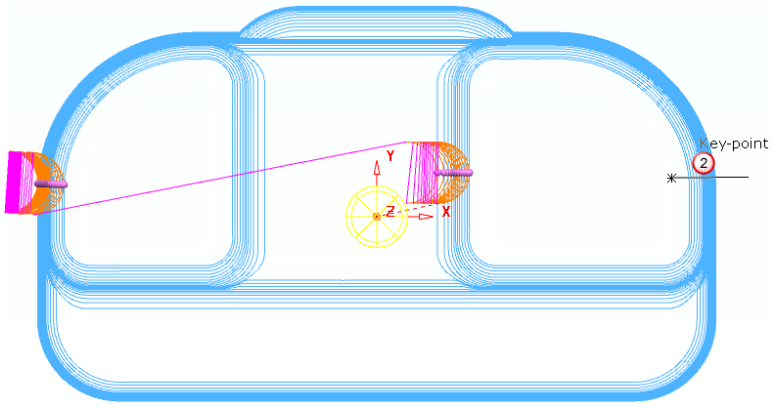


7 On each of the three locations shown below, in the listed order, click and select 2 points on both sides of the tool tracks.

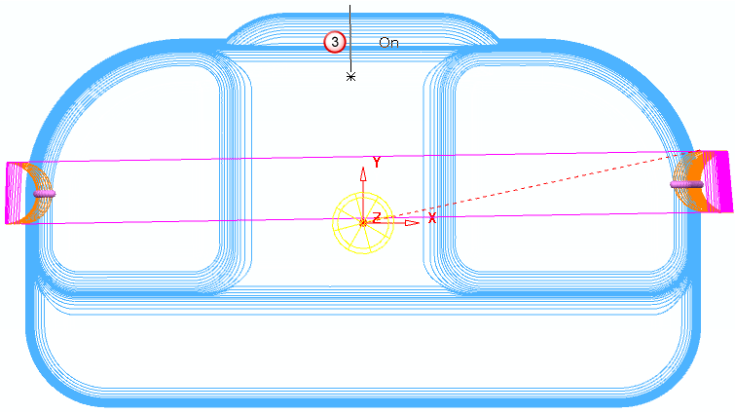
- ① - First move point selection points.



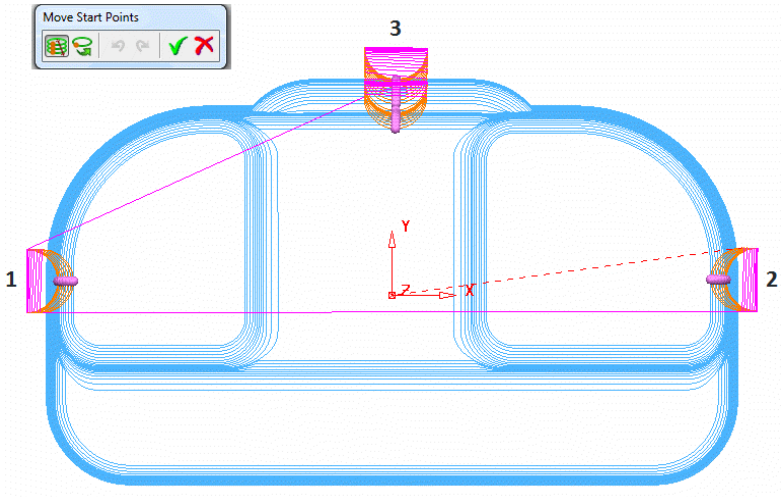
- ② - Second move point selection points.



- 3 - Third move point selection points.



The final output looks similar to this:



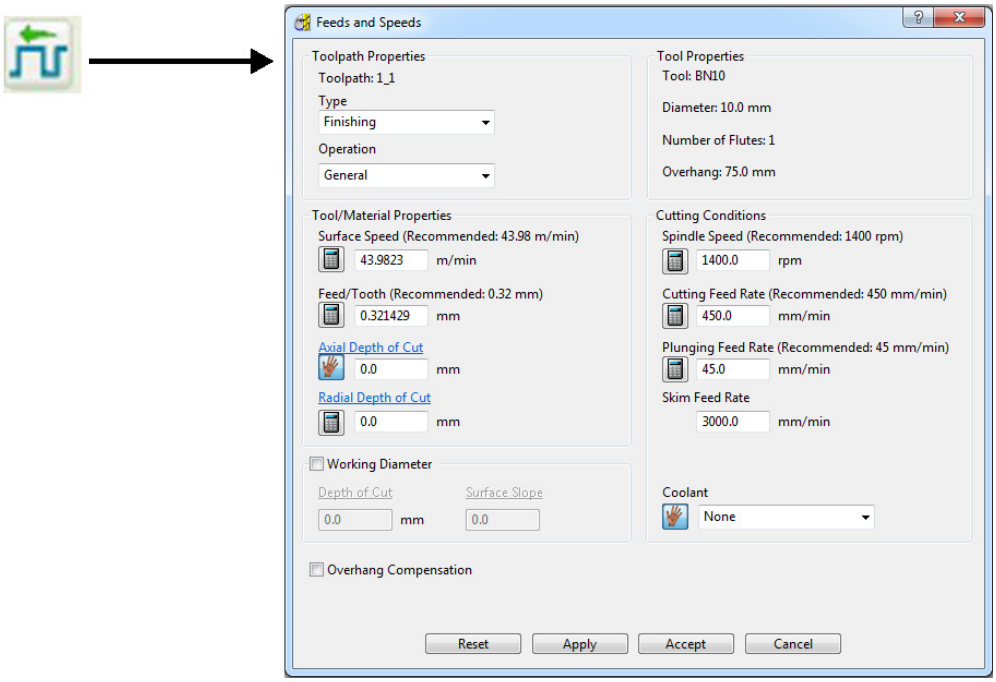
*The tool tracks are coloured blue while the **Move Start Points** toolbar is active.*

8 Click on the **Move Start Point** toolbar to accept the changes.

Copyright © Delcam plc

Feeds and Speeds

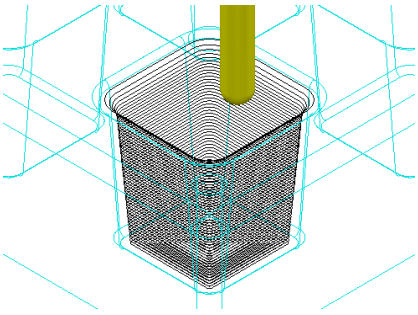
You can modify Feed Rates using the **Feeds and Speeds** dialog to retrospectively **Apply** them to the ***Active*** **toolpath**.



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

2 Select **File > Open Project**:

.....\PowerMILL_Data\Projects\ExtraFeedrates-Start



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

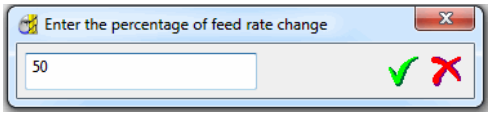
.....\COURSEWORK\PowerMILL-Projects\ExtraFeedrates-example

4 Undraw the **Model**, **Block**, **Workplane**, and **Tool** to display the **Toolpath** and 3 **Boundaries**.

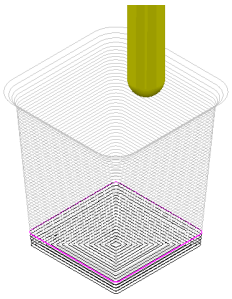
5 **Activate** both the **Toolpath** and **Boundary 2**.

6 Right-click the **Toolpath** and select **Edit > Update Feed Rate Inside Boundary**.

7 Enter value as **50** and click to apply the percentage difference to the **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.

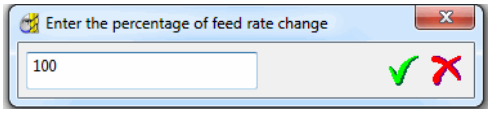


*The original Cutting Feedrate assigned to the toolpath was **450 mm/min**.*

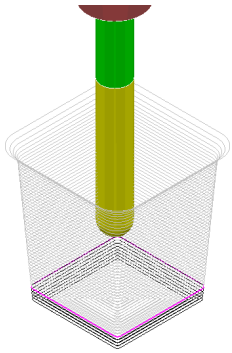
8 **Activate** both the **Toolpath** and **Boundary 3**.

9 Right-click the **Toolpath** and from the local menu select **Edit > Update Feed Rate Inside Boundary**.

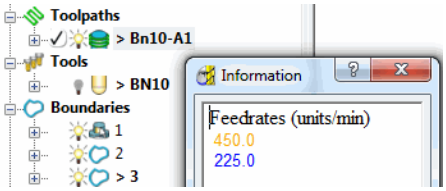
10 Enter the value **100** and click ✓ to apply the percentage difference to the **Feed Rate** within the **Active Boundary**.



The central part of the **Toolpath** area, clear of the side wall, has changed back to the nominal **Feed Rate** value.



11 To check the actual **Feed Rate** values assigned to the coloured areas of the toolpath, in the **PowerMILL *explorer***, double-click on a ***Light bulb*** adjacent to the **Toolpath**.

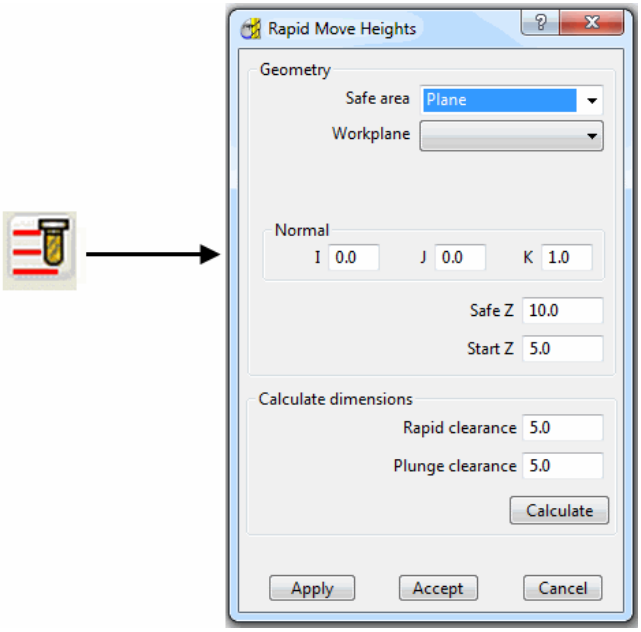


In this case, the nominal **Feed Rate** is **450 mm/min** and the modified section near the intersection of the Base and Sidewall of the pocket is reduced by 50% to **225 mm/min**. The colour of the values displayed in the Information dialog is the same as the colours displayed on the toolpath.

 Copyright © Delcam plc

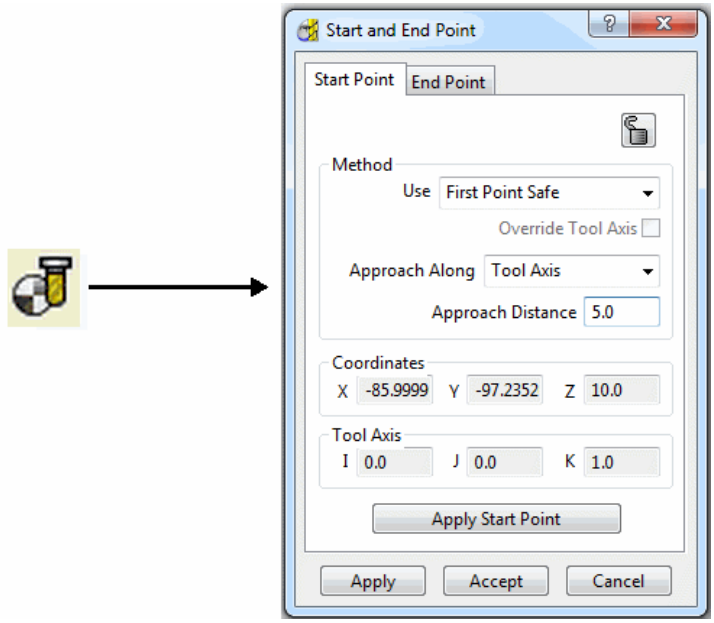
Safe Z and Start Z Positions

The tool **Rapid Move Heights** can be changed in the dialog and applied retrospectively to the **Active toolpath**.



Editing Tool Start and End Point Positions

The tool **Start Point**/**End Point** can be changed in the dialog and applied retrospectively to the **Active toolpath**.



 Copyright © Delcam plc

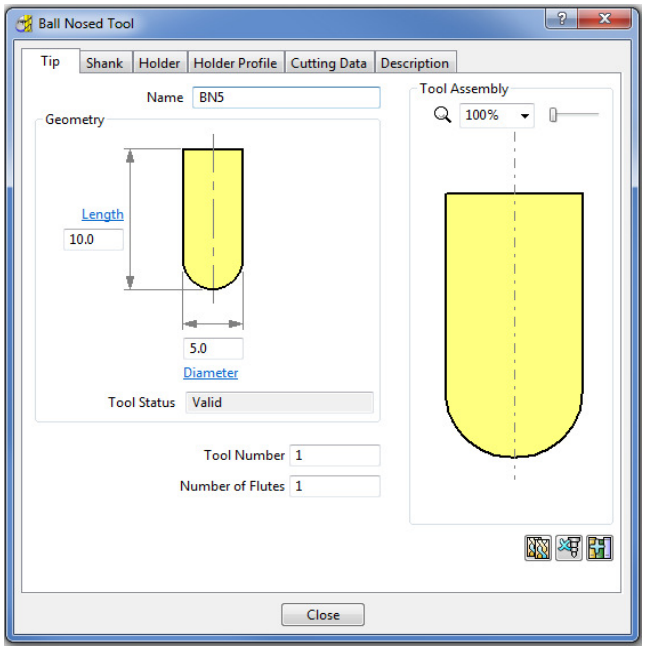
9. Tool Holder Collision Checking

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** complete with appropriate modifications will be substituted into the ***Original*** and ***Collision*** status toolpaths.

1. Select **File** > **Delete all** and **Tools** > **Reset** Forms.
2. **Import** the **Model**:-

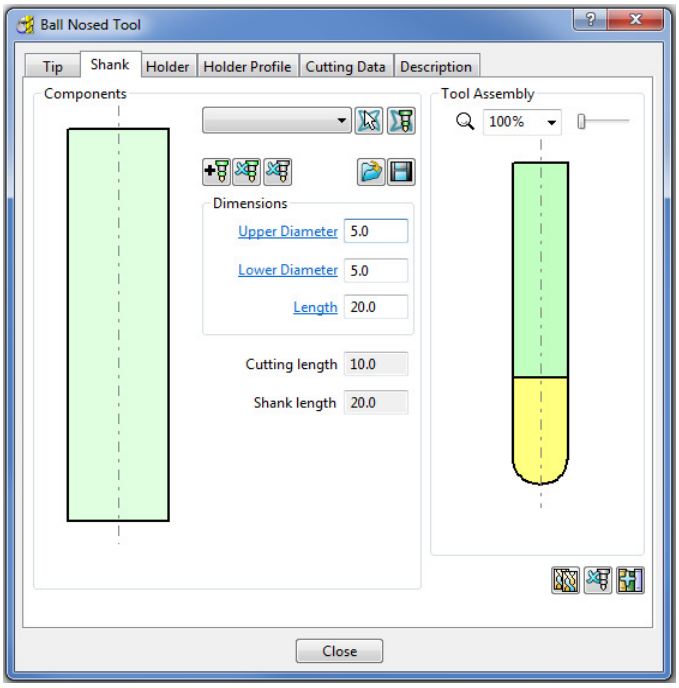
   ...\PowerMILL_Data\Models\cowling.dgk
3. **Calculate** the **Block** to **Min/Max** limits.
4. **Calculate** both the **Rapid Move Heights** and the tool **Start** and **End Points**.
5. Create a Dia 5mm Ball Nose tool named **BN5** with a length of **10** and tool number of **1**.



6 Select the **Shank** tab.

7 Select Add a **shank** component .

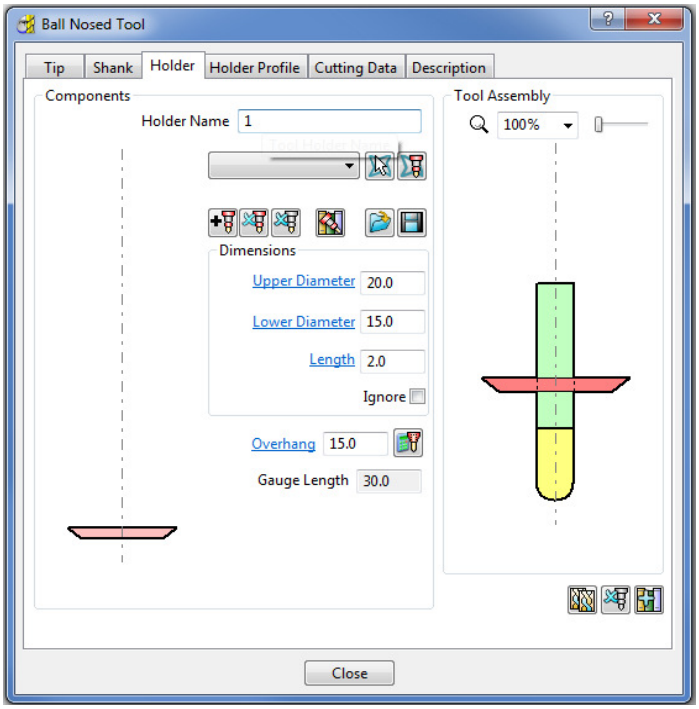
8 Fill in the dialog exactly as shown in the image below.



9 Select the **Holder** tab.

10 Select ***Add holder component*** .

11 Fill in the form for the first **Holder** element exactly as shown in the image below.



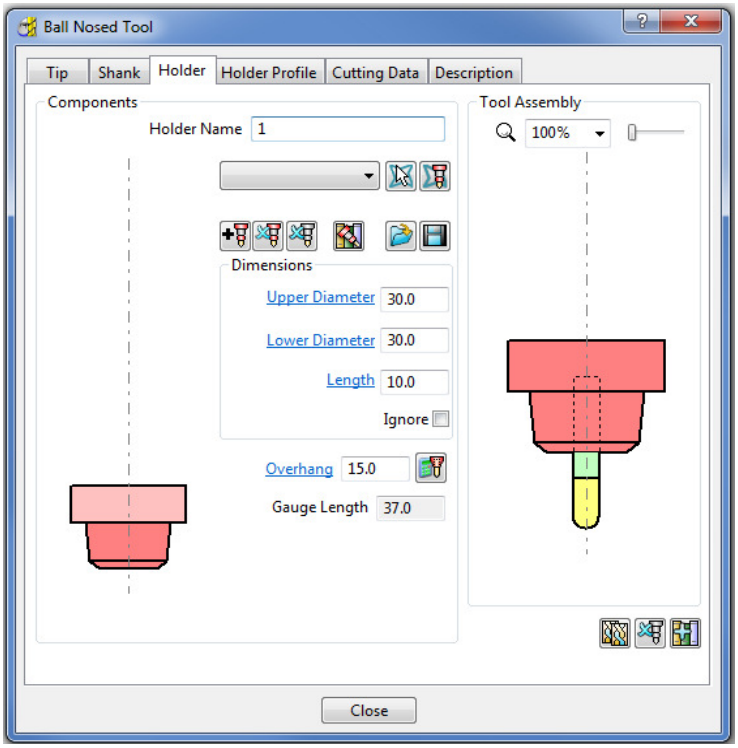
12 Select ***Add holder component*** .

Copyright © Delcam plc

13 Enter an **Upper Diameter** of **22**, a **Lower Diameter** of **20** and a **Length** of **10**.

14 Select ***Add holder component*** .

15 Enter an **Upper Diameter** of **30**, a **Lower Diameter** of **30**, a **Length** of **10** and an **Overhang** of **15**.



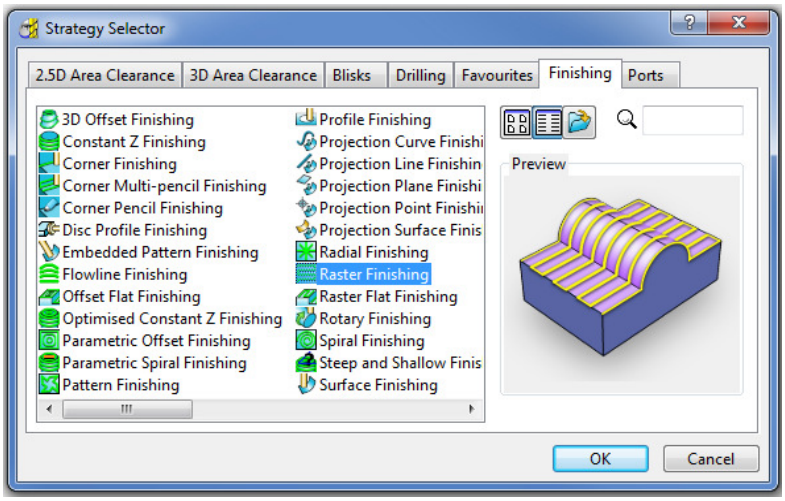
*If it is required to modify or delete a **Holder** element, select (left click) on one of them in the assembly illustration and it will turn pale. The current values for that element will be displayed and available for modification.*

*The same procedure applies to modify a **Shank** element.*

16 Click **Close**.

17 Select **Toolpath Strategies** on the **main** toolbar.

18 From the **Finishing** options select a **Raster Finishing** strategy.



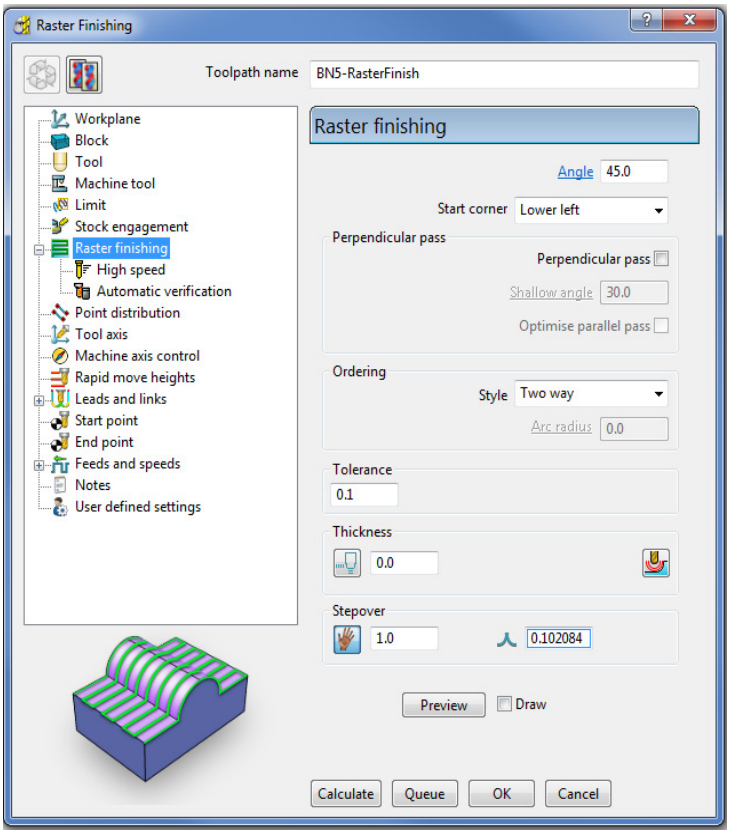
19 Enter the name as **BN5-RasterFinish**, Angle **45**, and **Stepover 1**.

20 Select **Start Corner** - **Lower Left**.

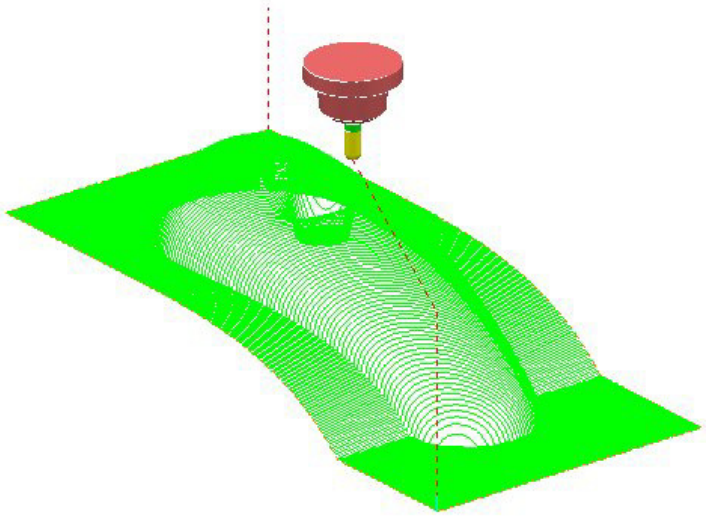
21 Select **Style** - **Two way**.

22 In the **Leads and Links** , **Lead In** and **Lead Out** options, set **1st** and **2nd Choices** to **None**.

23 In the **Leads and Links** - **Links** options, enter a **Short/Long Threshold** of **2**, set **Short** to **Circular Arc** and **Long** to **Skim**.



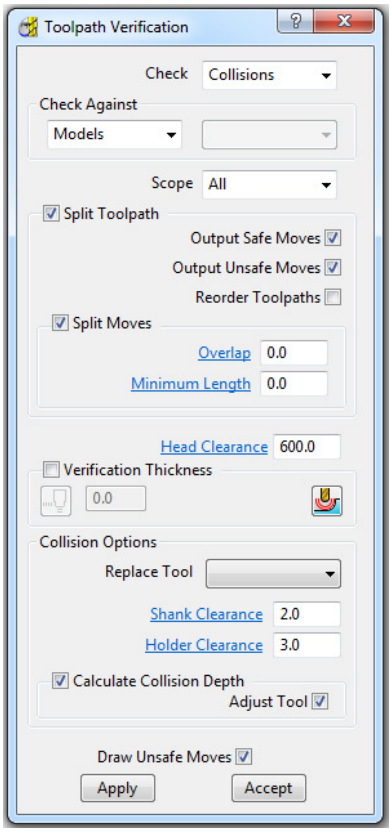
24 Click **Calculate** to process the **toolpath** and then **Close** the dialog.



The new **toolpath** is ***gouge free***, but at this stage no ***collision checking*** has been applied in relation to the **Shank** and **Holder**.

Copyright © Delcam plc

25 From the **Main** toolbar select **Toolpath Verification** .



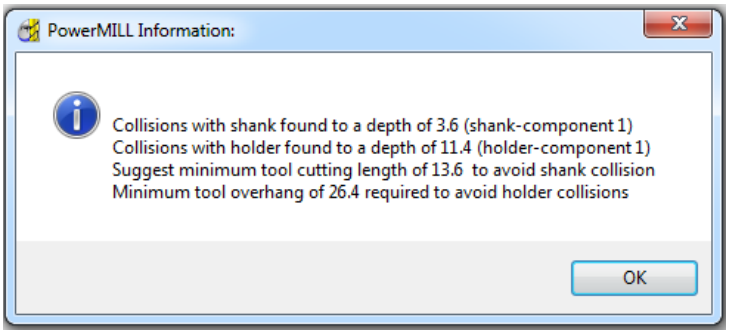
The two available, independent **Check** options are **Collisions** and **Gouges**.

Shank Clearance and **Holder Clearance**, allow the user to specify a value for an additional, safe thickness around the tool **Shank** and **Holder** elements.

26 In the **Check** box select **Collisions**.

27 Input values in the **Shank Clearance 2** and **Holder Clearance 3**.

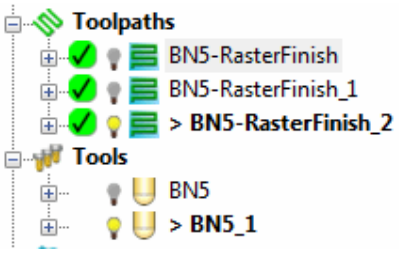
28 Click **Apply**.



The above **PowerMILL Information** box appears with information on the status of the original tool and required dimensional changes. The default settings will result in a modified **tool** (**BN5_1**) being created along with 2 new **toolpaths** (The collision and non-collision parts of the original). The new **tool** (**BN5_1**) will then automatically exist in both the original and new (collision warning area) toolpath. The original **tool** (**BN5**) will remain in the new (collision safe area) toolpath.

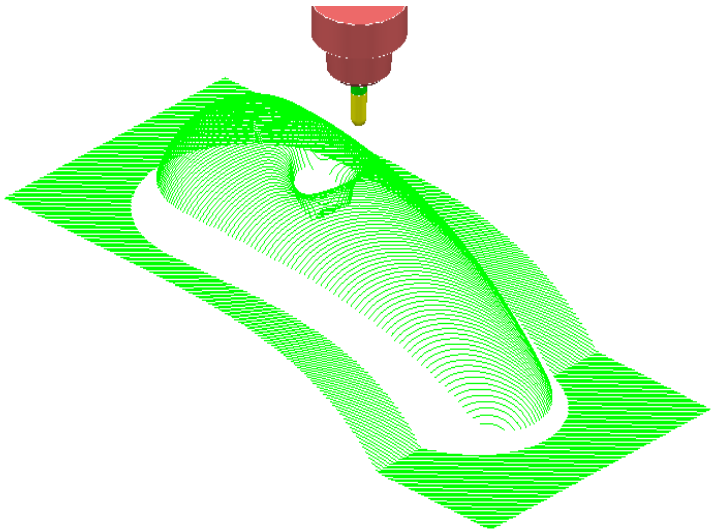
29 Click **OK**.

30 Click **Accept** on the **Toolpath Verification** dialog.



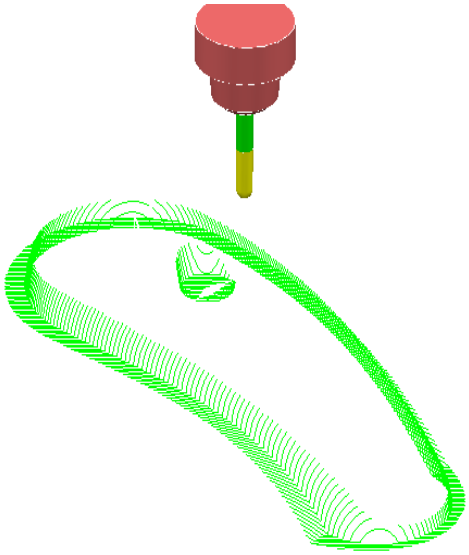
PowerMILL has created two new toolpaths, **BN5-Raster Finish_1** and **BN5-RasterFinish_2**. A new tool named **BN5_1** has also been created in the **Tools** area of the **PowerMILL *explorer***. This new tool has been created with the new valid adjustments made i.e. **Overhang 26.4**

31 **Activate** the toolpath **RasterFinish_1** in the **Explorer**.



This **toolpath** contains the 'collision free' segments of **RasterFinish**, along with the original tool **BN5**.

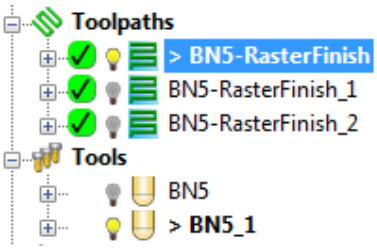
32 **Activate** the toolpath **RasterFinish_2** in the **PowerMILL *explorer***.



This **toolpath** contains the 'collision' segments of the **Raster Finish** if it were machined with the original **BN5** tool.

 Copyright © Delcam plc

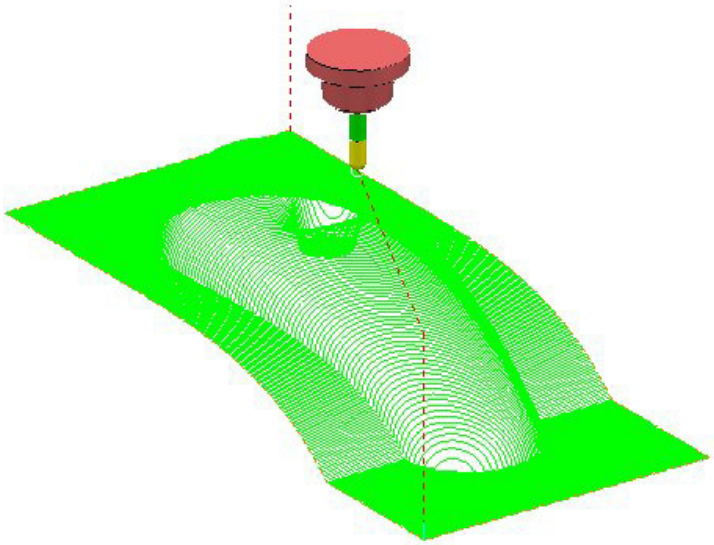
33 **Activate** the original toolpath, **BN5-RasterFinish**, in the **PowerMILL** ***explorer***.



PowerMILL *has substituted a newly created* ***Tool***, ***BN5_1*** *(with an extended overhang) into* ***BN5-RasterFinish*** *and* ***BN5-RasterFinish_2****.*

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

.....\COURSEWORK\PowerMILL-Projects\ToolCollisionExample



Automatic Collision Checking

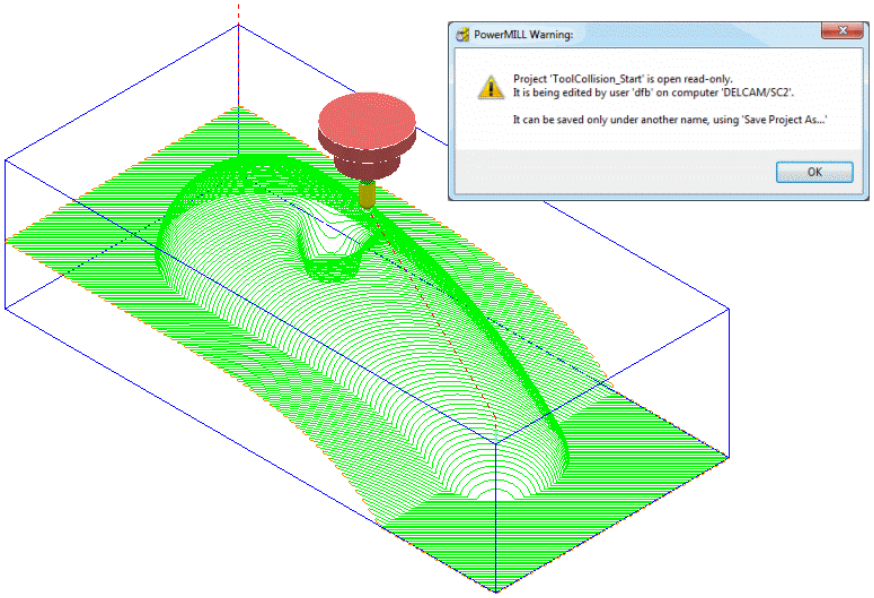
Tool Holder Collision Checking can also be applied during **Toolpath** calculation.

If the **Toolpath** is calculated with a **Tool** definition that includes a **Shank** and **Holder** any part of the toolpath segments that would otherwise be in a collision condition will not appear. These missing segments can then be machined later using a modified **Tool** in conjunction with a **Collision Safe Boundary**.

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

2 **Open** the **Project**:

.....\PowerMILL_Data\Projects\ToolCollision_Start



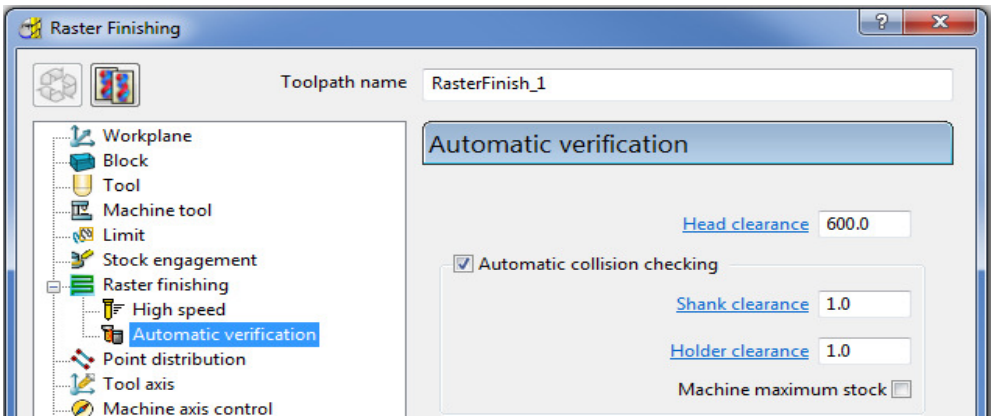
3 **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\ToolCollision-Auto

4 Right-click on the **RasterFinish** toolpath in the **PowerMILL *explorer***.

5 From the local menu, select **Activate** followed by **Settings** and in the **Raster Finishing** dialog select the ***Copy*** toolpath icon .

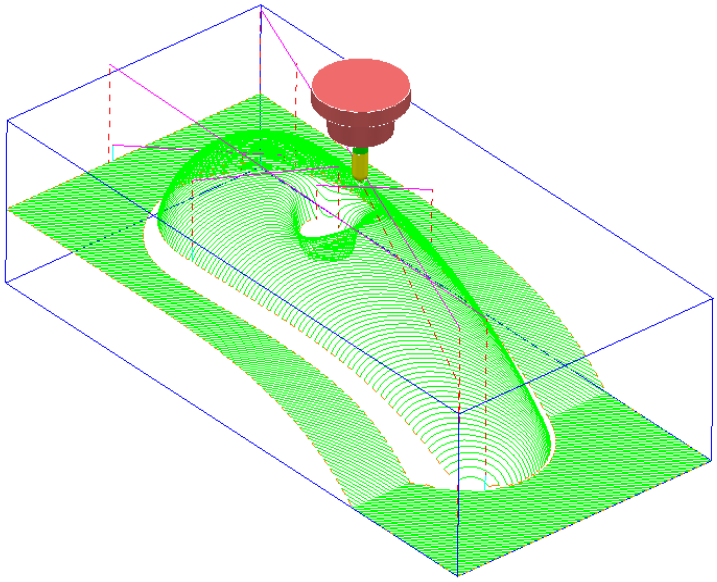
6 Select the **Automatic Verification** page to open the following dialog:



Copyright © Delcam plc

7 Tick the **Automatic collision** checking box, and for both **Shank Clearance** and **Holder Clearance** input **1.0**

8 **Calculate** the **Raster Finishing** strategy to directly create a collision free toolpath, **RasterFinish_1**.



*This method does not produce any advice for the user or automatically create a new compatible tool (**BN5_1**) that would perform collision safe machining over the whole strategy.*

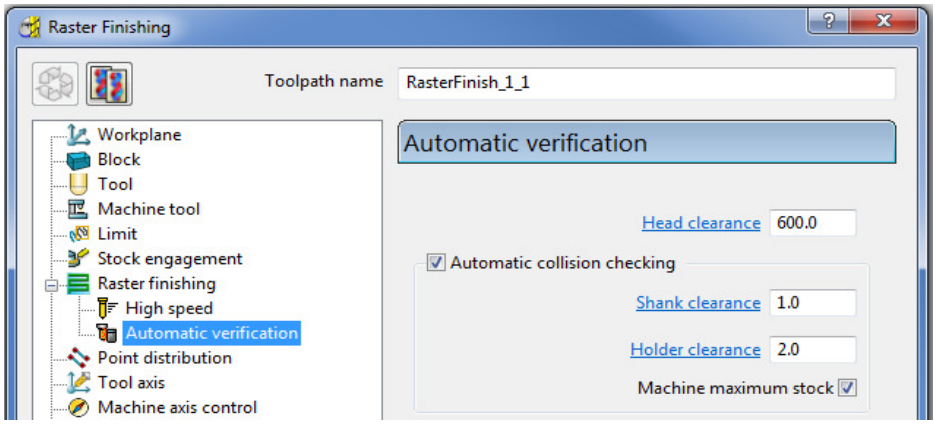
Machine maximum stock

If the additional option **Machine maximum stock** is ticked then a full toolpath will be created with a varying increased **thickness** over the previously inaccessible areas.

9 Right-click on the **toolpath *RasterFinish*** in the **PowerMILL *explorer***.

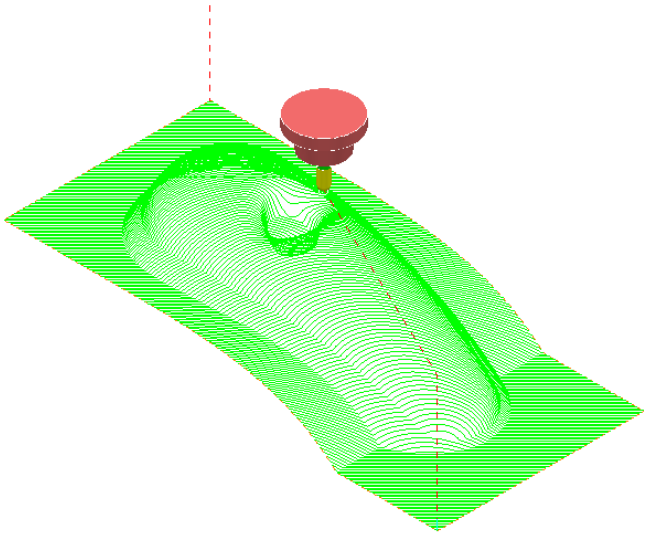
10 From the local menu, select **Activate** followed by **Settings** and in the **Raster Finishing** dialog select the **Copy** toolpath icon .

11 With the **Raster Finishing** dialog open, select the **Automatic Verification** page to access the following dialog:



12 Tick - **Automatic collision checking**, input **Shank Clearance 1.0**, **Holder Clearance 2.0**, and **tick** - **Machine maximum stock**.

13 **Calculate** the **Raster Finishing** dialog to directly create a continous, collision free toolpath, **RasterFinish_1_1**.



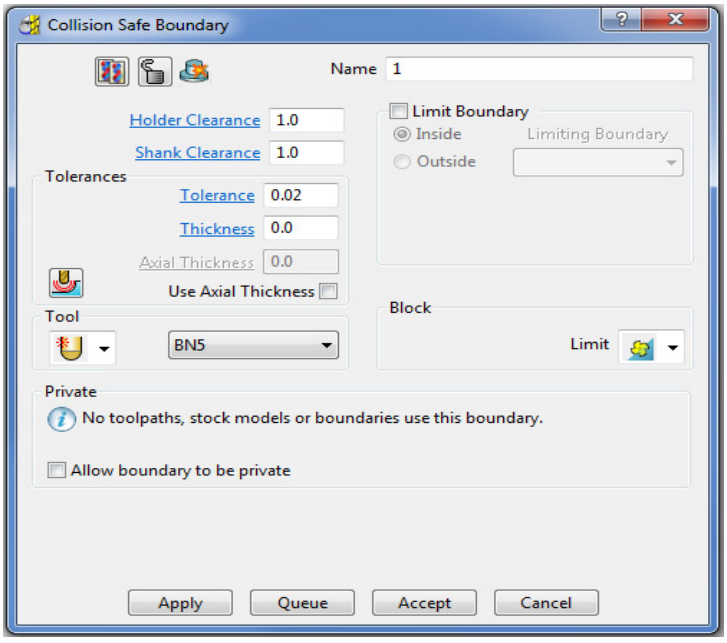
An additional strategy with a compatible (collision free) tool definition is required to remove the remaining stock.

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**).

14 Ensure that the tool **BN5** is **Active**.

15 In the **PowerMILL *explorer***, right-click **Boundaries**, and from the local menu select **Create Boundary > Collision Safe**.



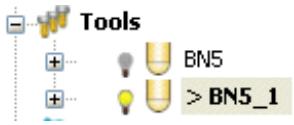
16 **Apply** the dialog.

 Copyright © Delcam plc

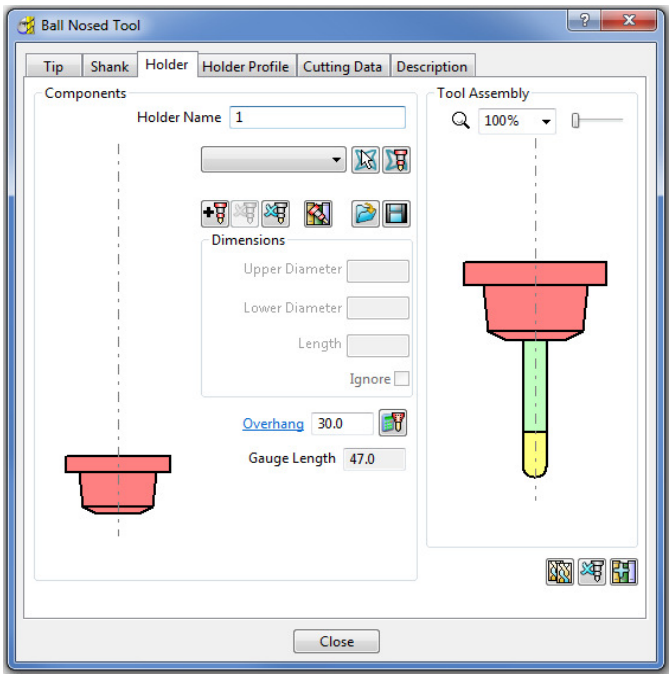
The lower part of the central pocket is more suited to be finish machined using an **End Mill**.

17 Select and **Delete** all segments except the 2 running along the outside of the component form as shown below.

In the **PowerMILL *explorer***, right-click the Tool, **BN5**, and from the local menu select **Edit > Copy Tool** (to create **BN5-1**).



18 Right-click over the new tool **BN5-1** and from the local menu, select **Activate** followed by **Settings**.

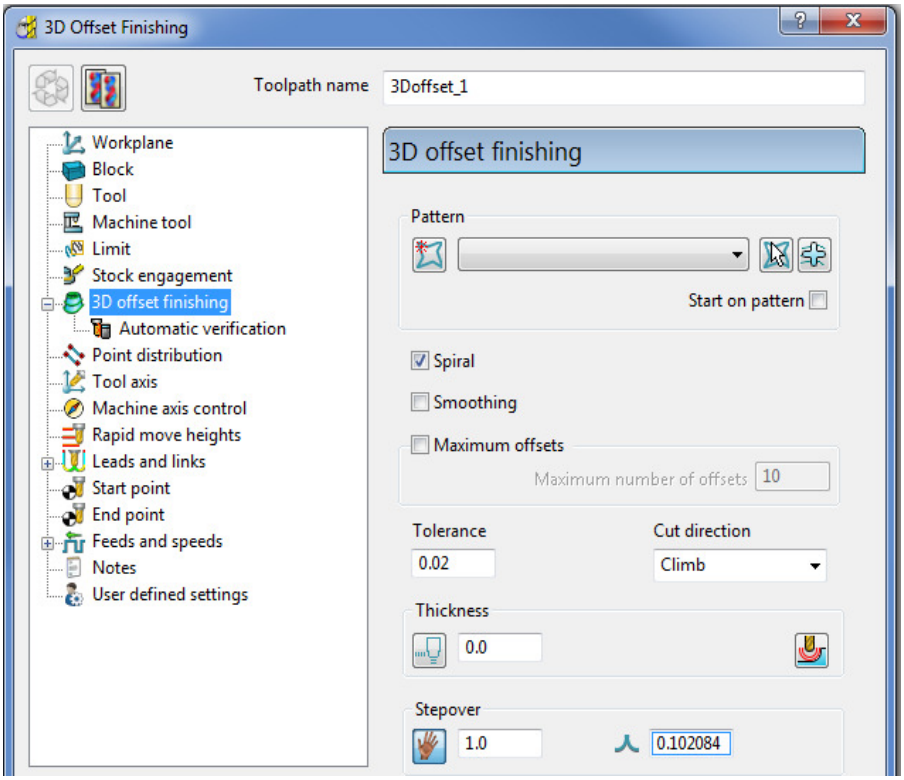


19 In the **Ball Nosed Tool** dialog select the **Holder** tab and change the **Overhang** to **30.0** and then **Close** the form.

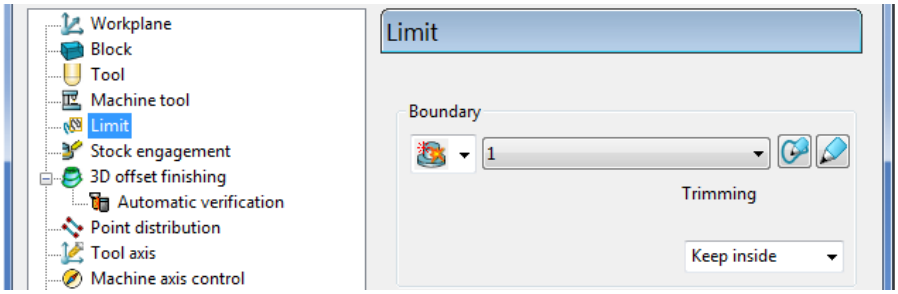
20 Select **Toolpath Strategies** on the **main** toolbar.

21 **Activate** the new 'long reach' **Tool BN5_1**

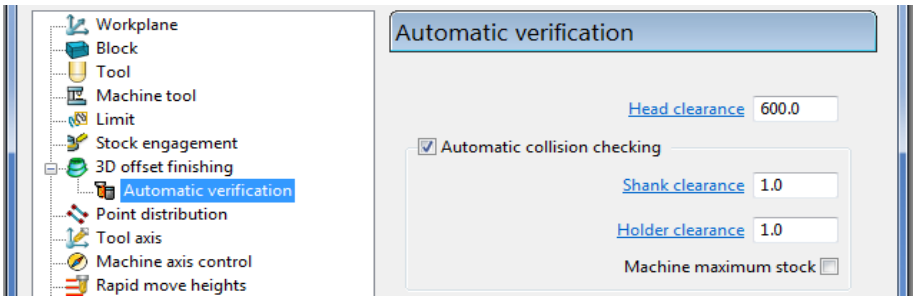
22 From the **Finishing** options select a **3D Offset Finishing** strategy and enter data as shown in the following form.



23 Select the **Limit** page and select **Boundary 1** with **Trimming > Keep inside**.

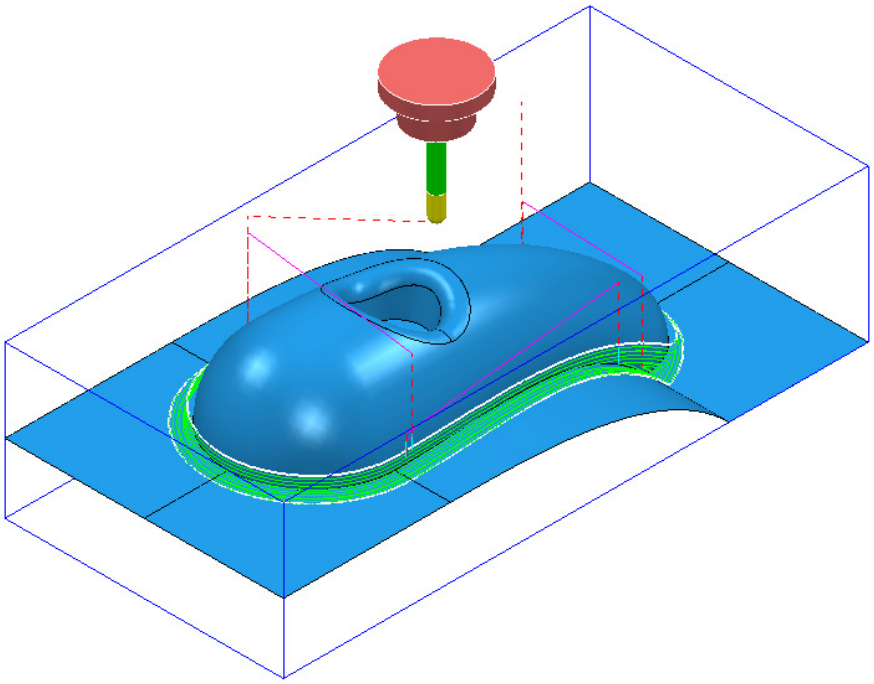


24 Select the **Automatic Verification** page and tick **Automatic collision checking**, input **1** for both **Shank** and **Holder *clearances***, un-tick **Machine maximum stock**.



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

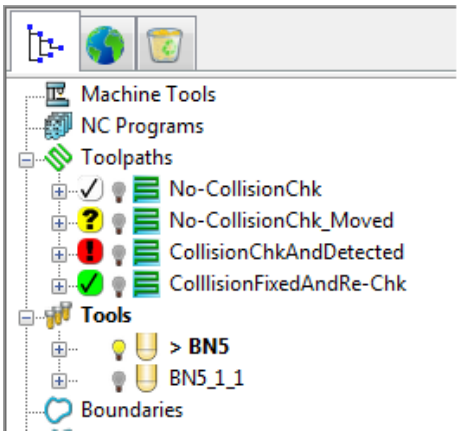
 Copyright © Delcam plc

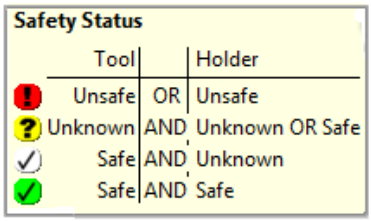


*Note that **Automatic Collision Checking** is still active and as can be seen in the above illustration there is no evidence of the toolpath being fragmented.*

Toolpath Gouge/Collision - Safety Status

When a **Toolpath** is calculated the ***symbol*** to the left of the light bulb in the **PowerMILL *explorer*** indicates the ***Safety Status*** for **Gouges** (Tip/Flutes) or **Collisions** (Shank/Holder).





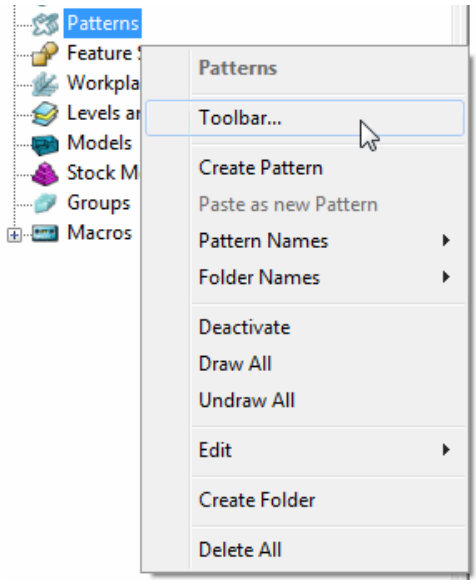
To display the 4 different levels of **Safety Status**, position the mouse over a ***symbol*** in the **PowerMILL *explorer*** and the information form will open (shown above right).

 Copyright © Delcam plc

10. Patterns

Introduction

Patterns are 2D or 3D Wireframe entities used primarily for the control and creation of toolpaths. A selection of **Pattern** creation and manipulation options are provided in a dedicated **Toolbar** (as shown below).



Below is a summary of the **Pattern - toolbar** options:

- **Create Pattern** (empty).

- **Activate the picked pattern**.

- **Pick curves to add to the active Pattern**.

- **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).

Copyright © Delcam plc

- **Insert Boundary into Active Pattern**.

- **Insert Active Toolpath into Active Pattern**.

- **Insert Model into Active Pattern** (Selected Model).

- **Curve modelling**.

- **Wireframe modelling** (PS-Sketcher).

- **Transform pattern**

- **Curve Editor**.

- **Clear**

*The **Pattern** segments can be edited directly from local menu options, or via the **Curve Editor**. Unlike **Boundaries**, **Patterns** can contain open segments.*

When an empty **Pattern** is created it is automatically assigned a number and made **Active** as shown in the **PowerMILL *explorer***.

Automatic Pattern Generation

The **Automatic Pattern Generation** icon opens the **Pattern Maker** dialog. This provides options for the user to construct a range of wireframe layouts that can be used in the **Pattern Finishing** strategy.

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

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

.....\PowerMILL_Data\Models\phone.dgk

3 Select **Block** and **Calculate** using the default settings.

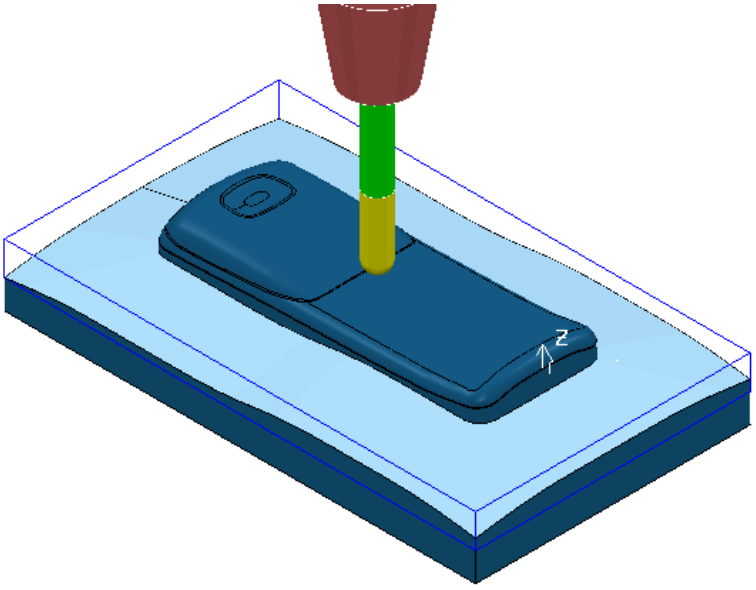
4 Define a dia 12mm Ball Nose tool named **BN12**.

5 Reset the **Rapid Move Heights** .

6 **Calculate** the **Start and End Points** using the default settings.

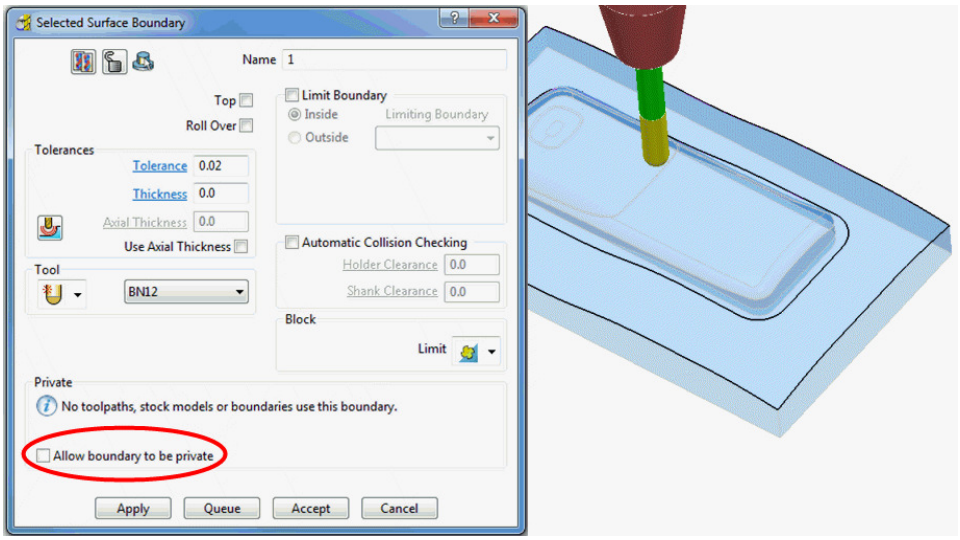
7 Select an **ISO1** view to display the **model**.

 Copyright © Delcam plc



8 Select the 'shutout' **surface** as shown above.

9 Right-click on **Boundaries**, and from the local menu select the option **Selected Surface** and enter the values shown below.



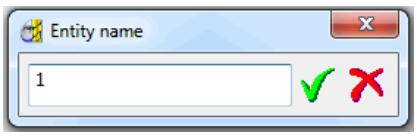
10 **Apply** the dialog.

11 Create a new (empty) **Pattern** .

12 Select the inner **Boundary** segment.

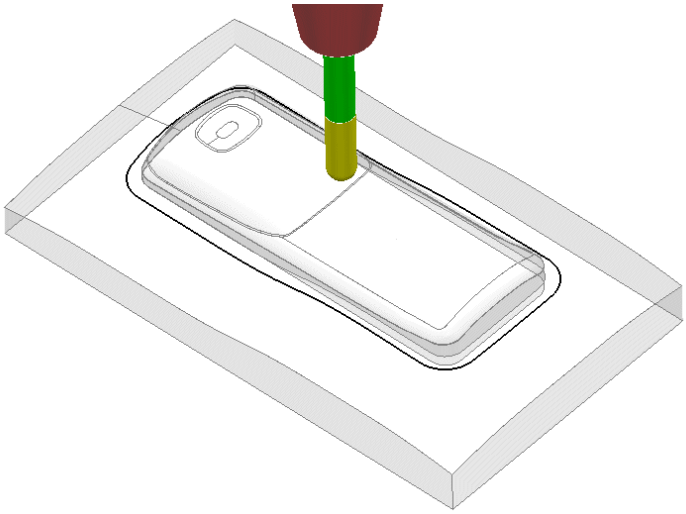
13 In the **Pattern** toolbar select the option to **Insert Boundary into Active Pattern** .

14 Input the **Boundary** name (1) and then click to accept.



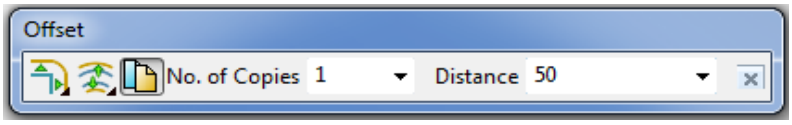
15 Undraw the **Boundary**.

The selected **Boundary** segment will be copied as a **Pattern** segment as shown below.



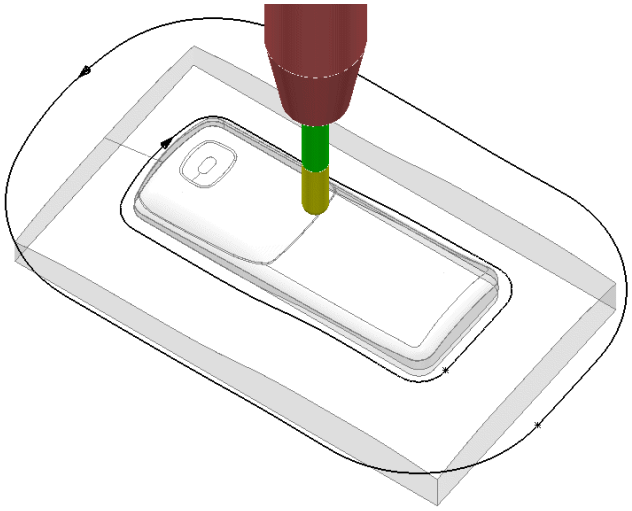
16 Right-click on the **Pattern** segment and from the local menu, select **Curve Editor**.

17 In the **toolbar** move the cursor over **Transformations** and from the exposed vertical toolbar, select the **Offset** option .



18 Select **Keep Original** , the **Round Corners** option, and input **Distance** 50 before clicking the ***Enter*** key.

19 In the **Curve Editor** toolbar, click to accept.



20 Right-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.

Copyright © Delcam plc

21 Select the inner **Pattern** segment and from the local menu select **Edit > Reverse Selected** (Both **Pattern** segments should now run in the same direction).

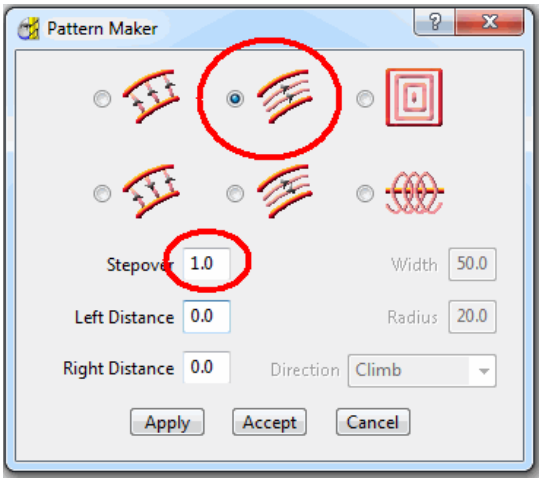
The **Automatic Pattern Generator** will be used to create a new **Pattern** with a series of segments merged between the 2 segments of the original **Pattern**. The resultant **toolpath** will follow the contour of the component form as well as creating corner free, tool tracks with a constant material removal rate.

22 From the **Pattern** toolbar, select **Create Pattern** to create a new **Pattern**.

23 Select **Automatic Pattern Generation** .

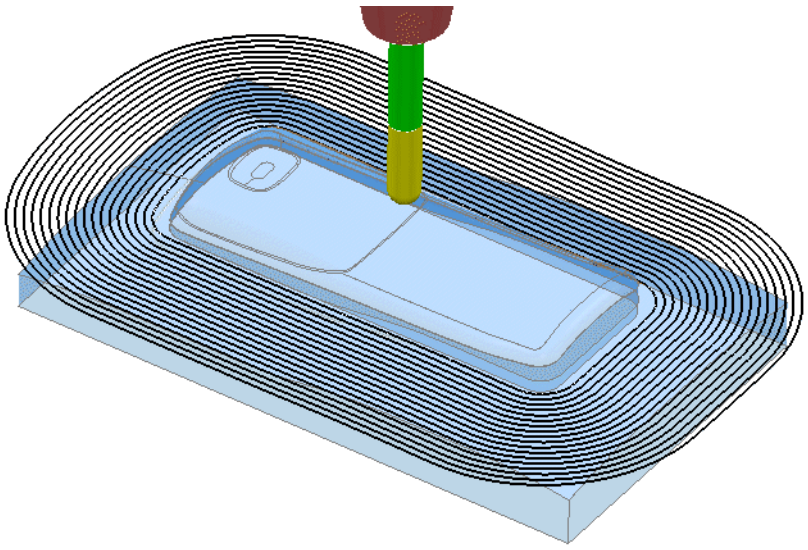
24 With the new (empty) **Pattern** activated and the original **Pattern** displayed, select both visible segments.

25 Select **Create Pattern** along 'one-way curves' with a **Stepover 1.0**



The form allows the user to create a range of multi-segment **Pattern** styles from either 1 or 2 drive curves.

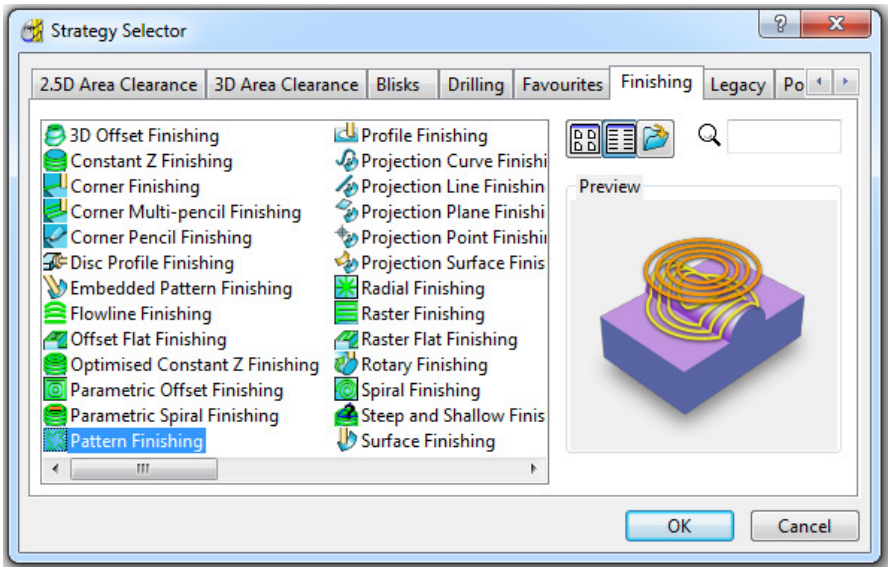
26 Click **Apply**, followed by **Accept**.



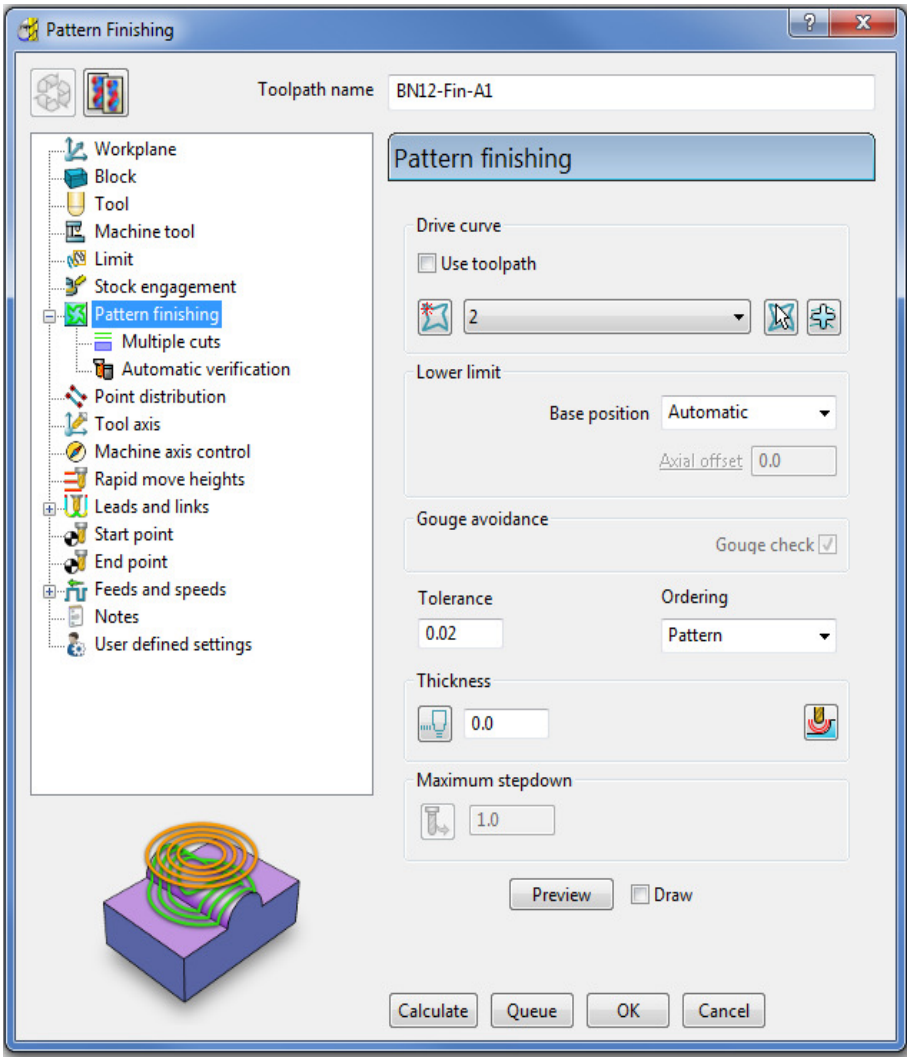
The new **Pattern** has been generated independently; between the two selected segments of the original, de-activated **Pattern** (the previous illustration shows a 2mm Stepover for visual reasons).

The new **Pattern** will now be used in a **Pattern Finishing** strategy.

27 Select a **Pattern Finishing** strategy.



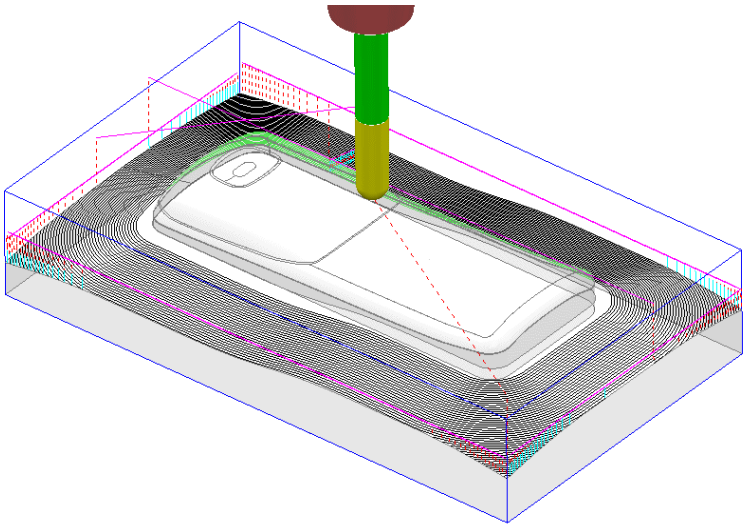
28 Ensure that the **Boundary** is not **Active**.



29 Enter the data into the above form exactly as shown below, click **Calculate**, and then **Close** the form.

Copyright © Delcam plc

30 Undraw the **Model**, **Boundary**, and **Patterns** for a better view of the **toolpath**.



The **Pattern Finishing** toolpath is limited to exist within the defined material **Block**.

Exercise

31 Right mouse click on the new **toolpath** from the local menu, and use the **Edit > Reorder** options to make the above strategy start from the ***outside*** and ***Climb Mill*** towards the centre of the model.

32 **Save** the **Project**.

Trochoidal Pattern Generator

In areas where a full width cut would otherwise be required, the **Trochoidal Pattern** is applied to produce a progression of advancing loops. This in turn will restrict the tool contact area to only a part of its circumference.

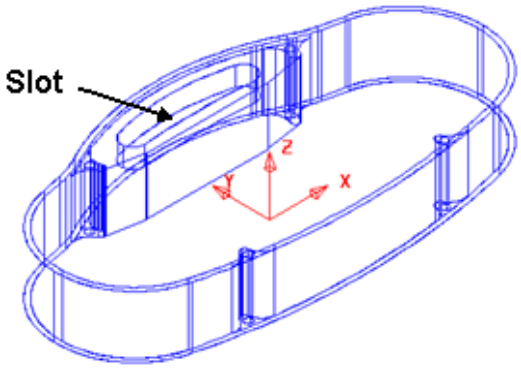
Since the tool is no longer cutting at full width, the problem of ***tool overload*** is greatly reduced. This is essential in applications where coated carbide cutters are used in ***High Speed Machining*** of hardened steel.

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

2 Select **File > Import Model**:

.....\PowerMILL_DATA\Models\dashboard.dgk

3 Select an **ISO1** view.



The above ***Slot*** will be machined using a **Trochoidal Pattern**.

4 Define the **Block** to **Min/Max limits**.

5 In the **PowerMILL *explorer***, right-click on **Patterns** and select **Toolbar**.

6 Select **Create Pattern** from the **Patterns** toolbar.

7 Select **Insert File** into **Active Pattern** from the **Patterns** toolbar and load in the file:

.....\PowerMILL_DATA\Patterns\trochoidal_pattern.pic

8 Select **Create Pattern** (again) from the **Patterns** toolbar.

9 Select **Insert File** into **Active Pattern** from the **Patterns** toolbar and load in the file:

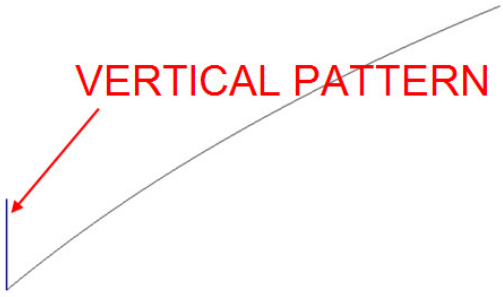
.....\PowerMILL_DATA\Patterns\StartHole.pic

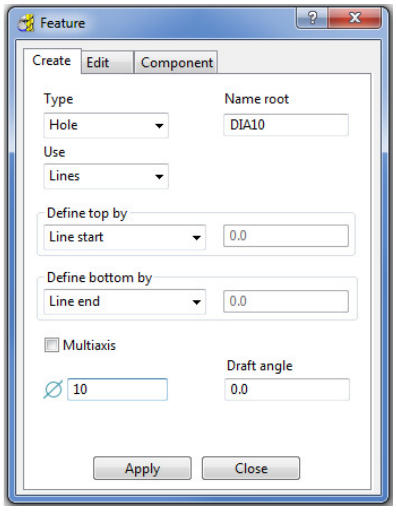
Before producing a Slot, a Helically drilled hole of diameter 8mm must be machined to create clearance for a Slot drill to plunge down to the full Slot depth. This is achieved by using Feature Sets (Covered in more detail in the next chapter).

1 Define a Ball Nose tool of 6 Diameter named **BN6**.

2 In the **PowerMILL *explorer***, right-click over **Feature Sets**, and select **Create Feature Set** to open the new **Feature** dialog.

3 Select the vertical **Pattern**.

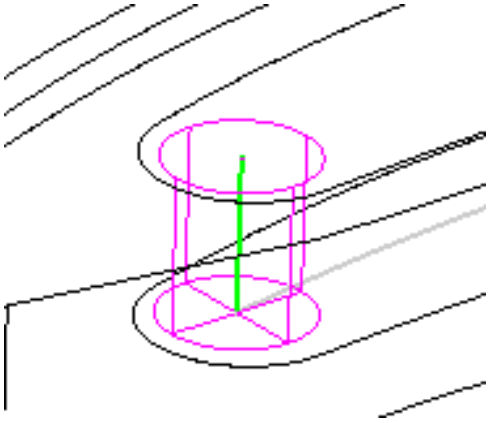




4 On the **Feature** form select the **Create** option and enter the values as shown in the above image, and then select **Apply** followed by **Close**.

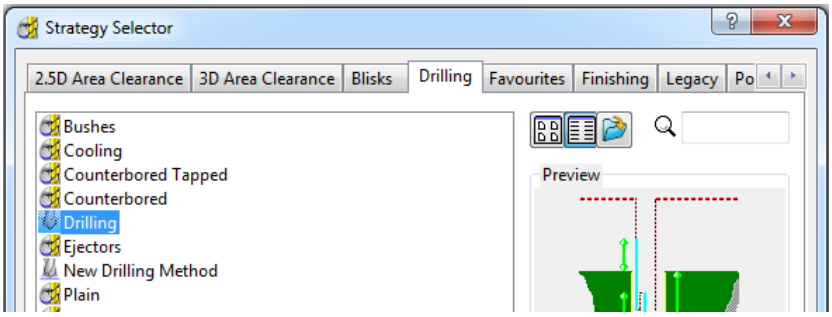
A new **Feature** will be created that includes a **DIA10** the hole ready for helical drilling.

 Copyright © Delcam plc

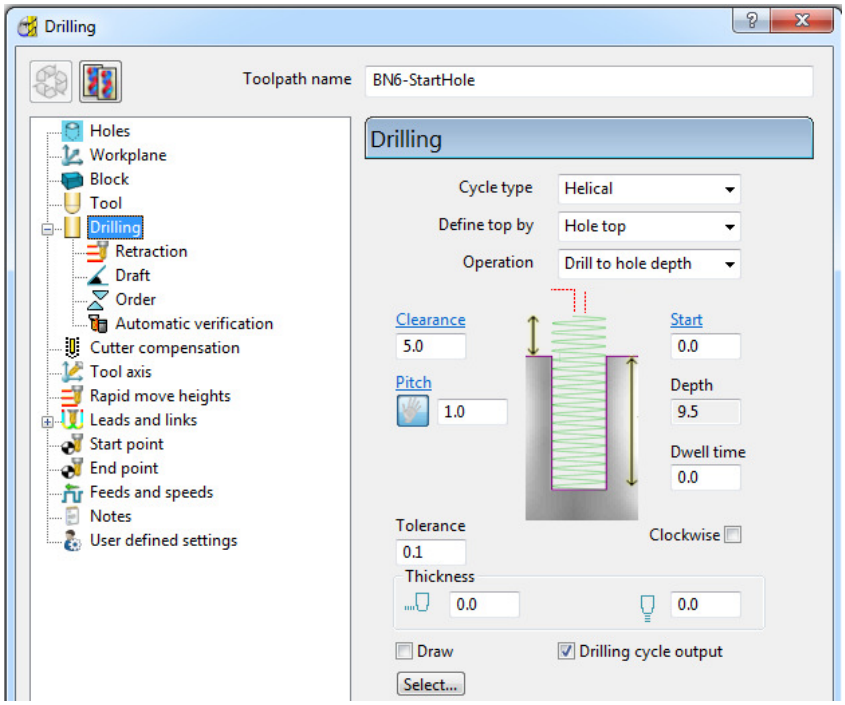


5 Click **Apply**, Click **Toolpath** Strategies on the **main** toolbar and then select the **Drilling** tab.

6 Select the **Drilling** option from the ***Drilling*** dialog.

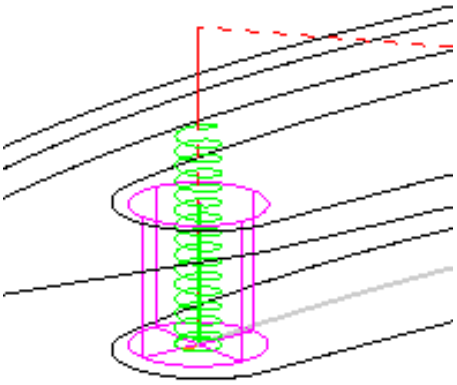


7 In the graphics area, depress the left mouse key, and select the new **Hole Feature**.



8 In the **Drilling** dialog, select **Cycle type - Helical**, input a **Pitch** **1** and then click **Calculate**, followed by **Close**.

9 Run a ***toolpath simulation*** and observe the **Helical *drilling*** strategy.



You can see on the simulation that the tool is **Climb Milling** in an anti-clockwise direction. The 10mm Diameter ***hole*** has now provided some clearance for a smaller tool to plunge into the ***Slot*** before performing a **Trochoidal** milling operation.

Feature Set *machining oprerations are not gouge checked to a* ***Surface*** *model.*

10 Create an **End Mill** of Diameter 6mm called **EM6**.

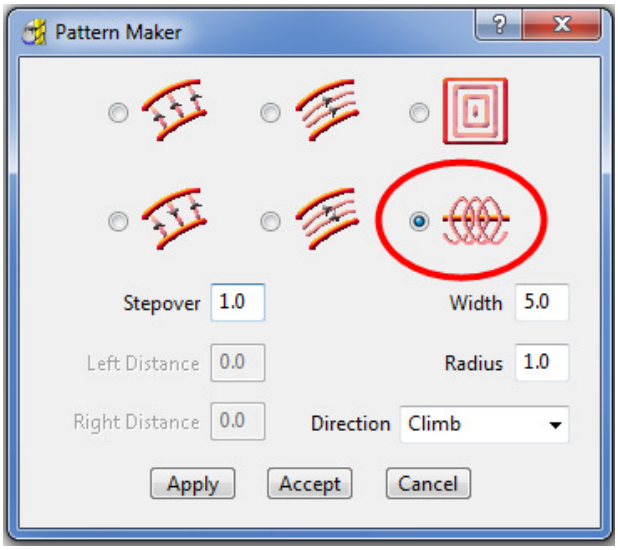
11 De-activate **Pattern 1** (trochoidal_curve), but keep it displayed by toggling the light bulb.

12 Create another empty **Pattern** (3) which will become **Active**.

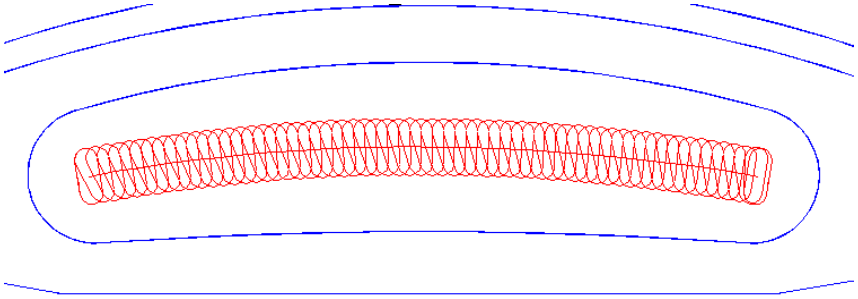
13 Select the (de-activated) curved **Pattern** (1) with the left mouse button.

14 Right mouse click on **Pattern** (3) Select **Automatic Pattern Generation**.

15 In the **Pattern Maker** dialog, select the **Trochoidal** option and enter the data exactly as shown below. Click **Apply**, and then **Accept**.



 Copyright © Delcam plc

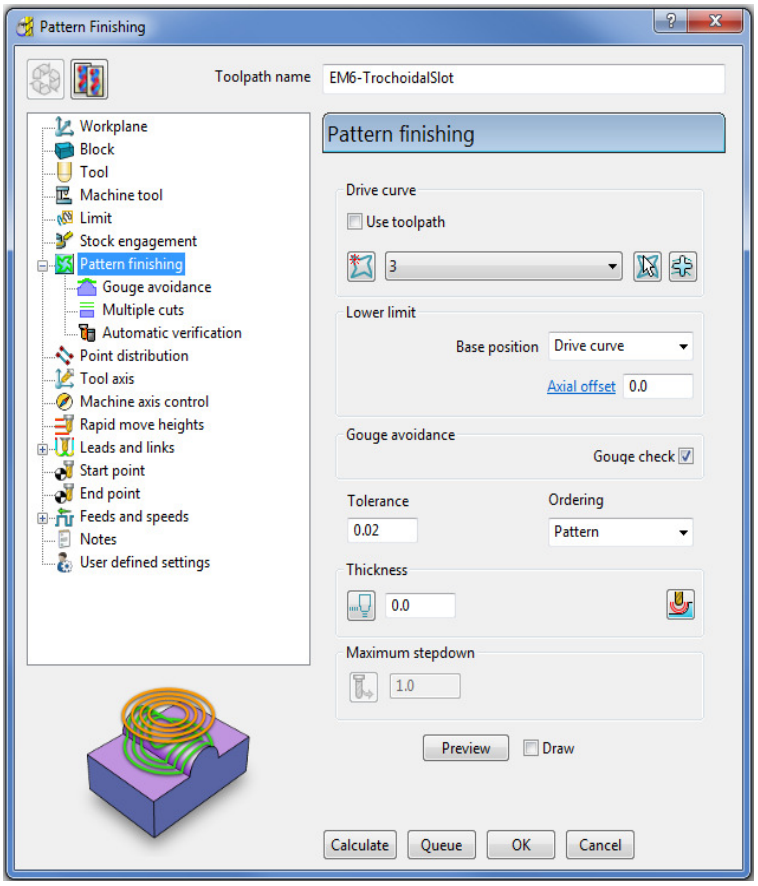


*The **Radius** value must not be greater than half of the **Width** value. A fine Stepover is required when applying to **High Speed Machining** of hardened steel.*

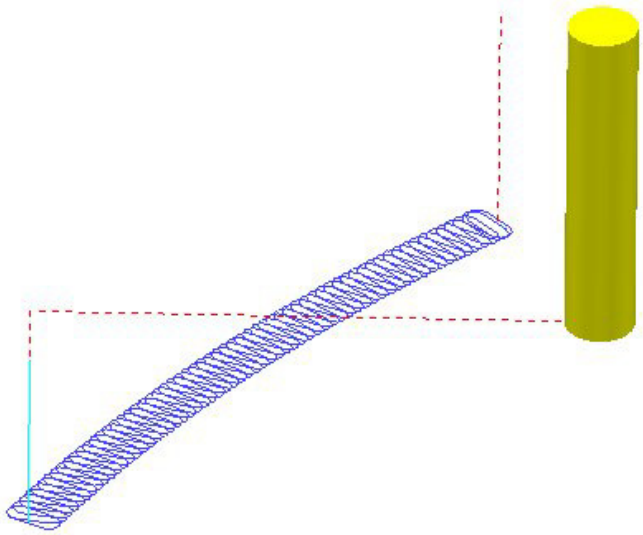
16 Select **Toolpath Strategies** .

17 On the **Finishing** tab, select the **Pattern Finishing** strategy to open the **Pattern Finishing** dialog.

18 Enter the values exactly as shown in the image below.



19 Click **Calculate** to create the toolpath and then **Close** the dialog.

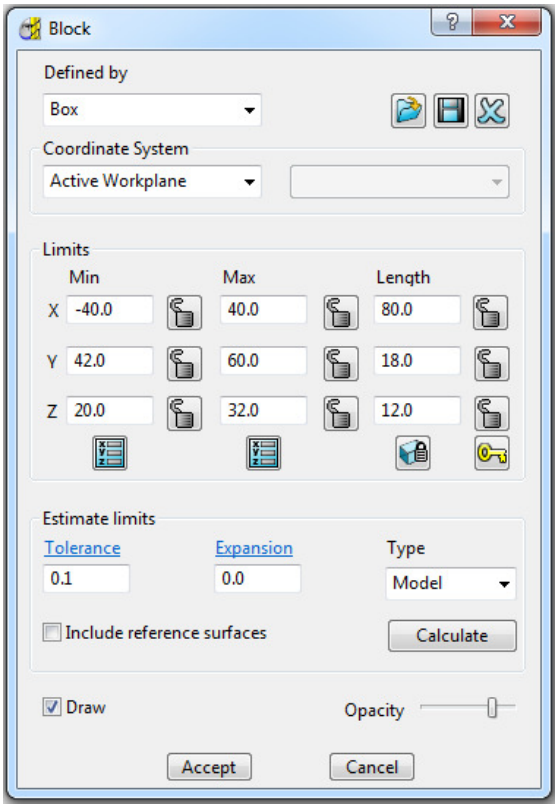


The 6mm End Mill tool will plunge into the previously created 8.5mm clearance Hole.

The Trochoidal toolpath defaults to a Climb Mill direction and is ideally suited to High Speed Machining applications.

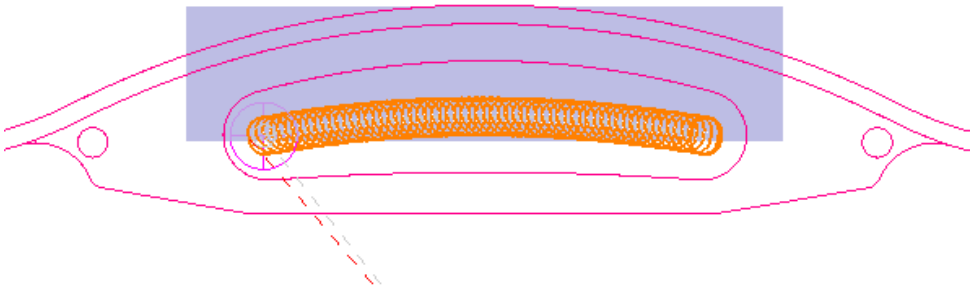
20 **Activate** the **toolpath** 1 (drilling) in the **PowerMILL *explorer***.

21 Open the **Block** form and then manually input values exactly as shown below to create a Block definition locally along the back half the slot width.



22 Click **Accept**.

 Copyright © Delcam plc

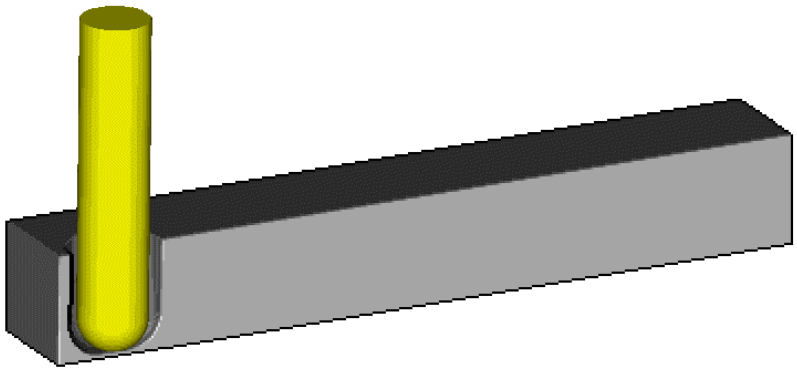


23 Dynamically position the **View** (as shown below) in the graphics area in preparation for a **ViewMILL** simulation.

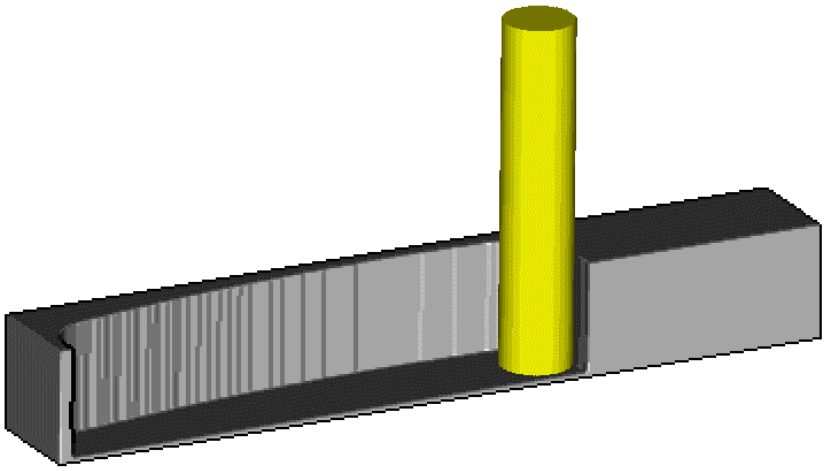
24 Select **ViewMILL On/Suspend** on the **ViewMill** toolbar.

25 Select **Plain Shaded Image** .

26 Visually check both the ***Helical*** and ***Trochoidal*** toolpaths in turn with **ViewMILL**.



Helical drill toolpath.



Trochoidal mill toolpath.

27 **Save** the **Project**.

Patterns applied to 3D Offset machining

A **Pattern** can be used as the form to be offset across the area of a **3D Offset Finishing** strategy.

1 Select **File > Delete All** and **Tools > Reset Forms**

2 **Import** the **model**:

.....\PowerMILL_Data\Models\speaker.dgk

3 **Save Project As**:

.....\COURSEWORK\PowerMILL_Projects\SpeakerPattern-EX1.dgk

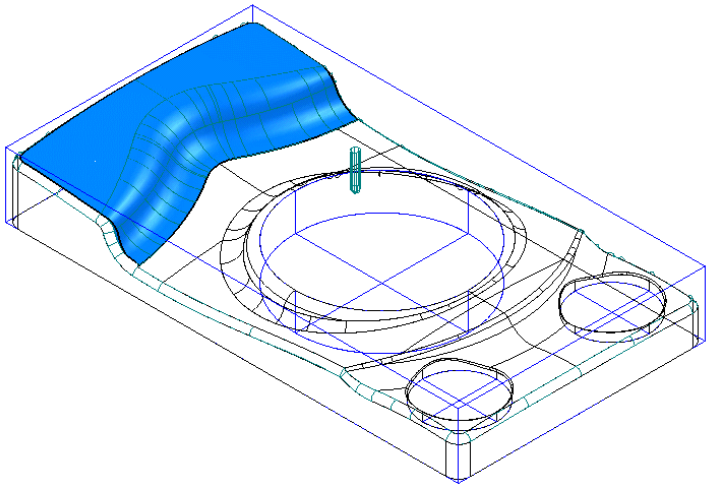
4 Define a Dia 6 Ball Nose tool with the name **BN6**.

5 Define a **Block** to the default Min/Max limits.

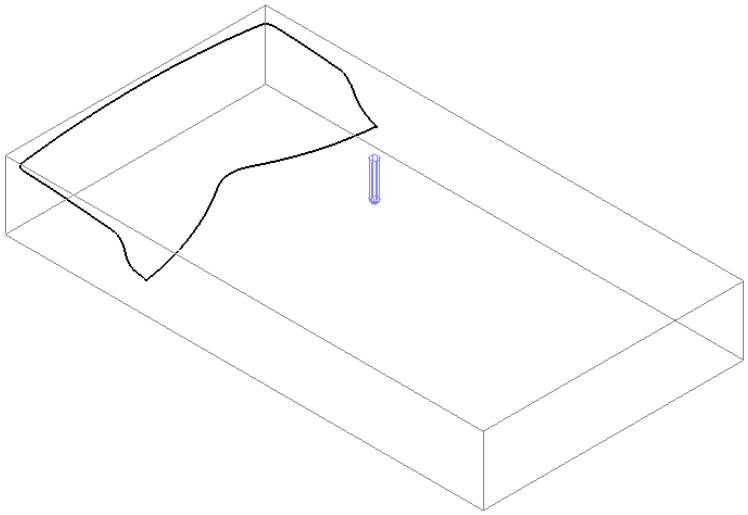
6 Calculate the **Rapid Move Heights** to the default settings.

7 Select an **ISO3** view.

8 Select the 3 shaded **Surfaces** as shown in the following diagram.



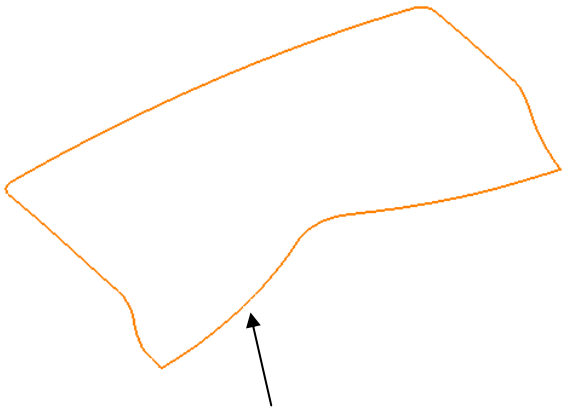
9 Create a **Selected Surface**, **Boundary** with ***allow boundaries to be private*** un-ticked, and with a **Tolerance** value of **0.02**



Copyright © Delcam plc

10 From the **Pattern** toolbar click **Create Pattern** followed by **Insert Boundary into Active Pattern** (Input **Boundary** name into the dialog!).

11 Undraw the **Tool**, **Block**, **Model** and **Boundary** to visually isolate the new **Pattern**.



*Only the leading edge of the **Pattern** is required. The remaining parts are to be removed.*

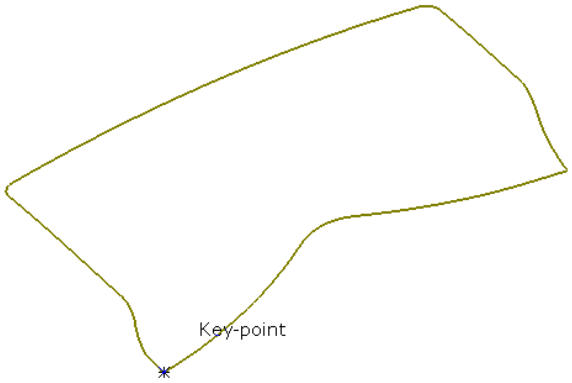
12 Right mouse click on the **Pattern** and from the local menu, select **Curve Editor** to open the following toolbar:

13 Select the option **Cut Item**.

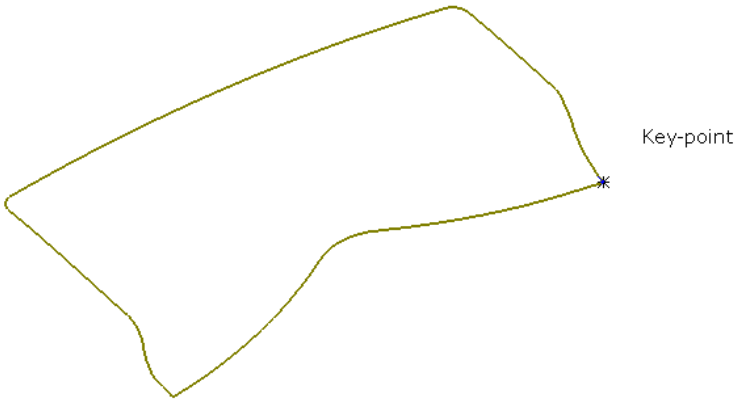
14 Select the **Pattern** segment using the left mouse key.

While performing the next operation it is essential to zoom in as close as possible to the end of the pattern segment!

15 Move the cursor over the left hand end of the leading edge to locate the **Key Point** and then left click to cut the segment at this point.

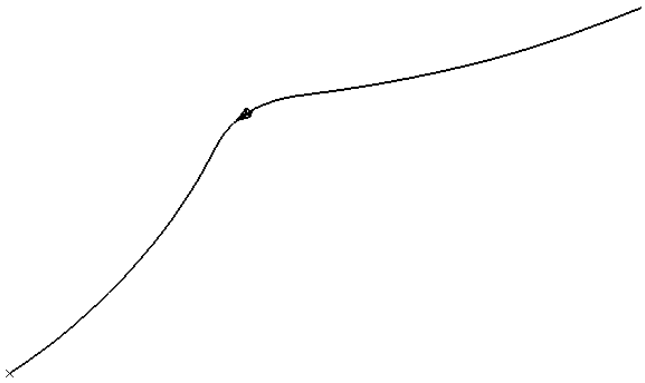


16 Select the leading edge, segment again, locate the **Key Point** and cut exactly at the right hand end of the leading edge.



17 Click on the option **Cut Item** again to de-select it.

18 Select and **Delete** the **Pattern** segments outside the leading edge.

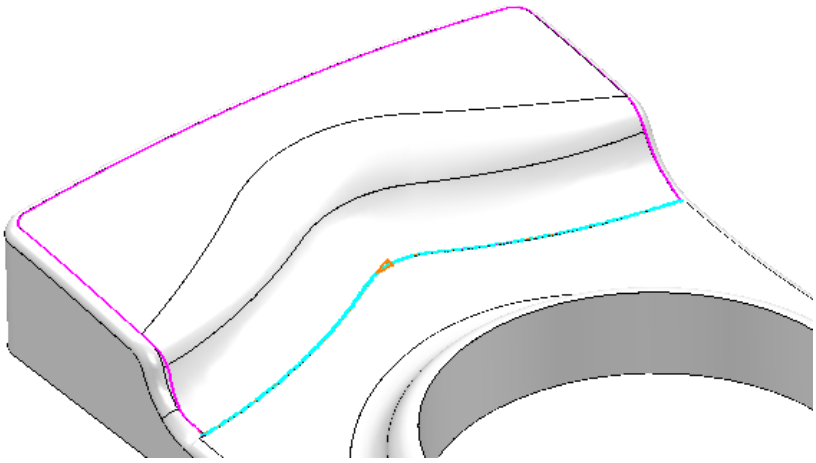


19 Click the green tick to **Save** the new segment and close the **Curve Editor** toolbar.

20 Right mouse click on the **Pattern** segment and from the local menu, select **Instrument** to display the direction of the segment.

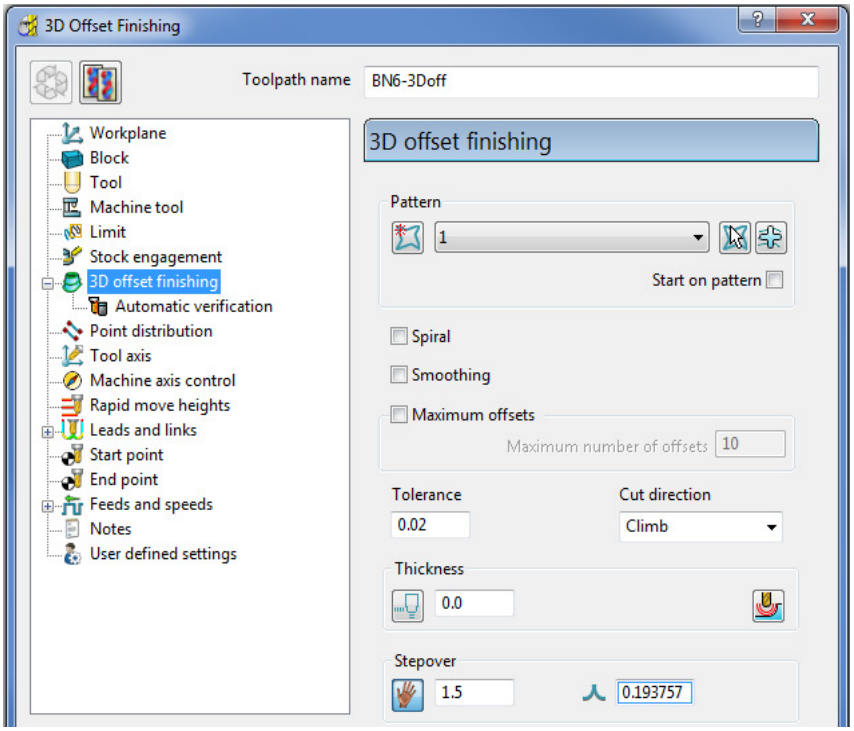
21 If it is not running in the direction (shown above), right mouse click on the **Pattern** segment and from the local menu select **Edit > Reverse Selected**.

22 Re-draw the **Tool**, **Model** and the ***Active*** **Boundary**.

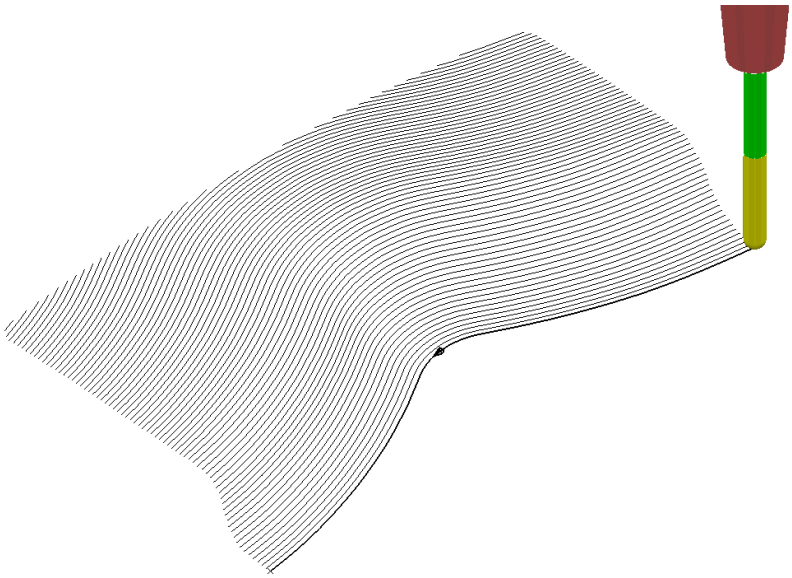


 Copyright © Delcam plc

23 Select a **3D Offset Finishing** strategy and enter data into the dialog exactly as shown below.

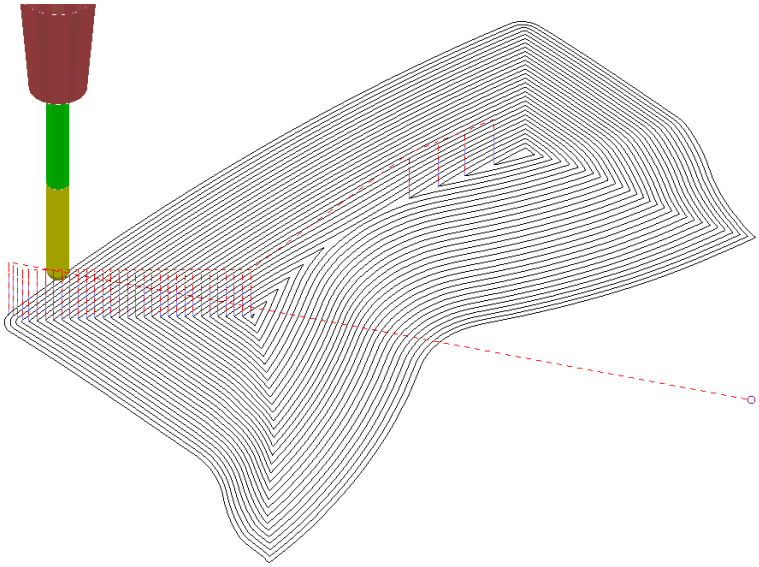


24 Click **Calculate** to create the **toolpath** and then **Close** the dialog.



Leads and Links are un-displayed to provide 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*** applications.



If a **Pattern** is not selected in the **3D Offset** dialog, machining strategy follows the shape of the **Boundary** segment, as shown above.

25 **Save** the **Project**.

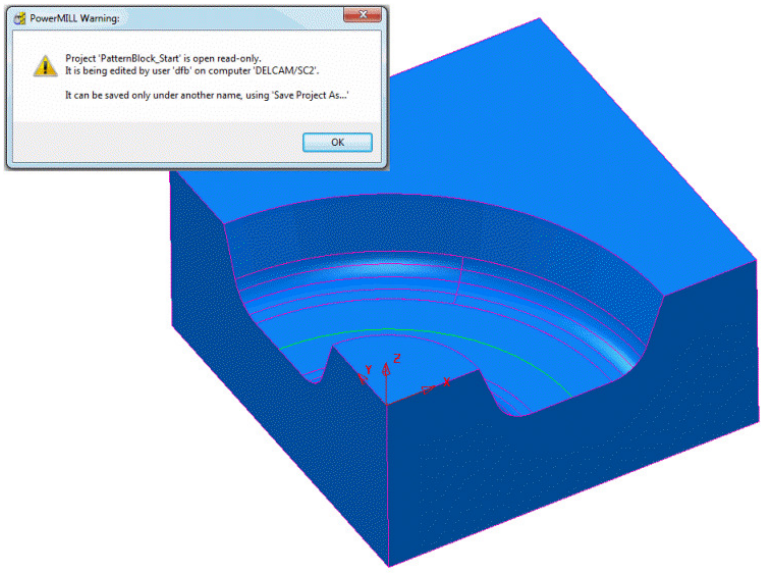
Exercise - 3D Offset Finishing controlled by Pattern

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 that you create a toolpath trimmed inside a **Boundary** with the tool tracks following a **Pattern** that runs centrally along the base of a curved recess:

1 Select **File > Delete All** and **Tools > Reset Forms**

2 **Open** the **Project**:

.....\PowerMILL_Data\Projects\PatternBlock_Start



3 Click **OK** to close the ***PowerMILL Warning*** and **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\PatternBlock_ex1

 Copyright © Delcam plc

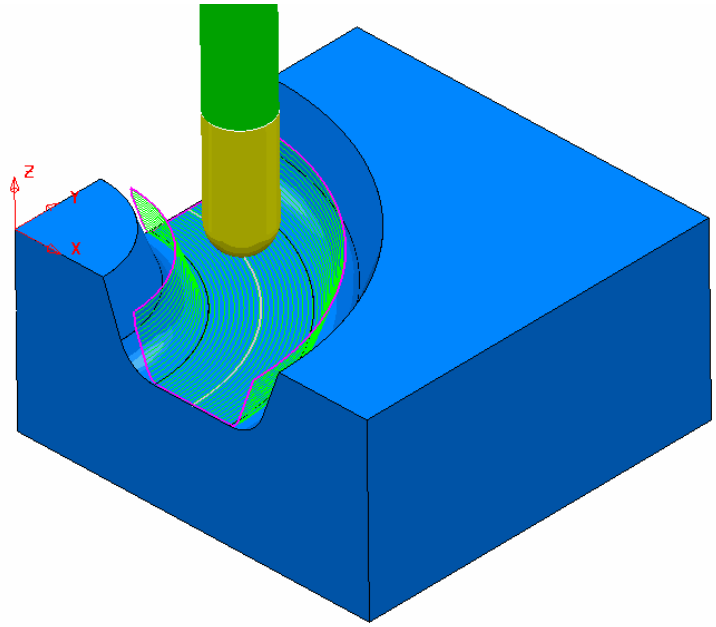
4 **Calculate** the **Block** using the default settings.

5 **Calculate** the **Rapid Move Heights** to the default settings.

6 Create a **Pattern** using **Insert Model** on the ***wireframe arc*** (included with the **Model**) running centrally along the recessed form.

7 Select the 5 **surfaces** that define the recessed form and create a **Selected Surface**, **Boundary** using a **0.01 tolerance**, with the **BN16** tool active.

8 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**.



9 **Save Project As**:

.....\COURSEWORK\PowerMILL_Projects\PatternBlock-EX1

Deep Pattern machining

If **Pattern Finishing** is applied with a –ve **Thickness** it will only create **tool tracks** where the –ve **Thickness** value is less than the **tool**, ***tip radius***. As an example, this makes it impossible to use a –ve **Thickness** value less than zero where an **End Mill** is being used.

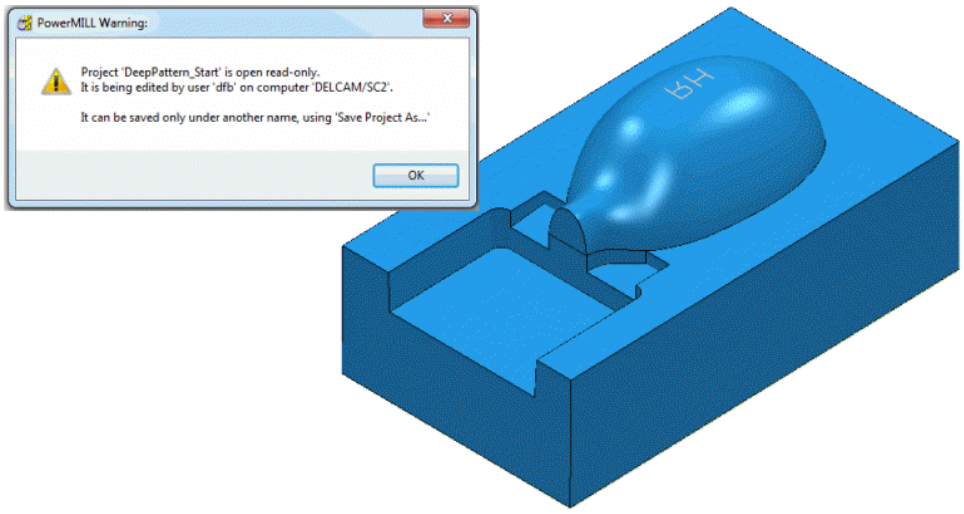
It is, however, possible to produce a **Pattern Finishing** strategy that machines deeper into the model if **Axial Offset** is used in place of a –ve **Thickness** value. This is achieved by first projecting the **Pattern** onto the surface (relative to the tool geometry). **Pattern Finishing** is then applied using the ***Drive Curve*** option with a –ve, **Axial Offset** value while **gouge checking** is unticked.

*The strategy produced will not be based on a 3D offset of the model form but a series of copies of the **Drive Curve** stacked downwards.*

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

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

..... \PowerMILL_Data\Projects\DeepPattern_Start



3 Click **OK** to close the **PowerMILL Warning** and **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\DeepPattern-EX1

The **Project** consists of component surfaces and a **Pattern** on **Z0** representing the text 'RH' ready to be engraved into the component form to a depth of **-3**.

4 **Calculate** a **Block**, ***Defined By*** **- Box**, using ***Type*** **- Model**.

5 Create a Dia 5 Ball Nosed tool with the name **BN5**.

6 **Calculate** the **Rapid Move Heights** to the default settings.

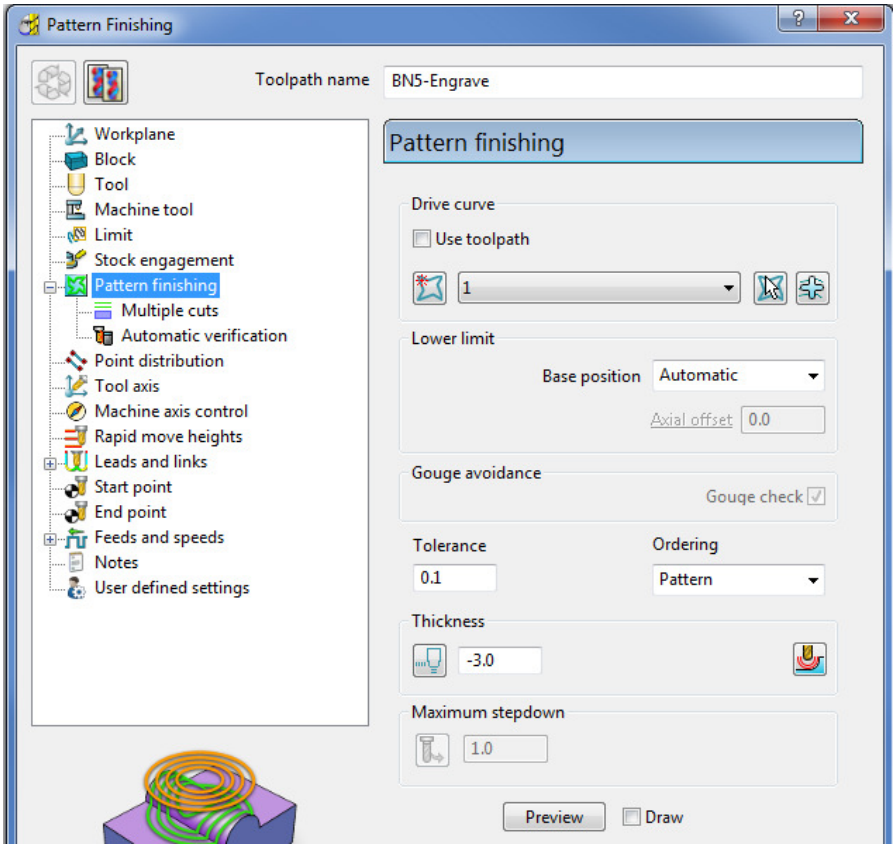
7 In the **PowerMILL *explorer***, right-click over the **Pattern** 1 and from the local menu select, **Edit > Drop** to project it downwards onto the model.

*This projects the **Pattern** down Z, while compensating for the tooling parameters.*

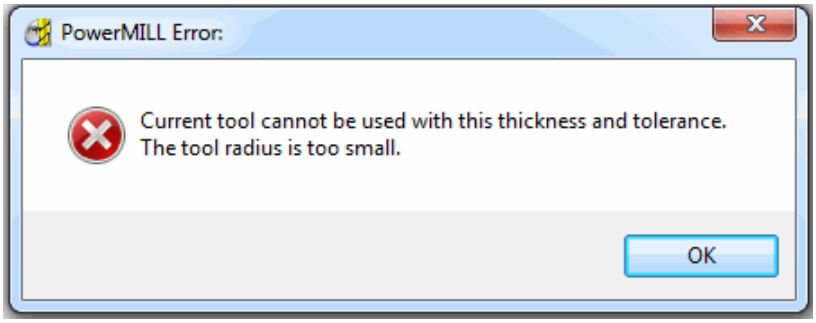
Copyright © Delcam plc

8 From the **main** toolbar, select **Toolpath Strategies** and in the dialog select **Finishing** followed by the **Pattern Finishing** option.

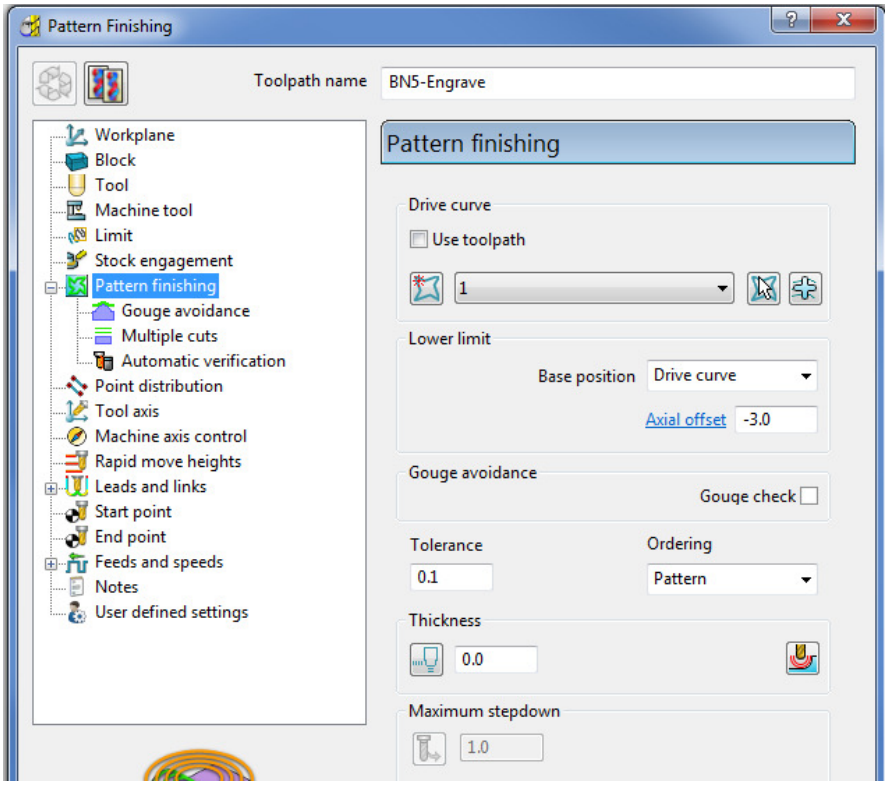
9 Enter the data into the dialog exactly as shown, using ***Base Position*** **Automatic** with ***Thickness*** **-3** and then click **Calculate**.



The **Thickness** of **-3.0** exceeds the Tip Radius of the Dia 5 Ball Nosed tool hence instead of a toolpath appearing, the following ***PowerMILL Error*** form is displayed.



10 Click **OK** to close the ***PowerMILL Error*** message and recycle the **Pattern Finishing** toolpath using the data as shown in the following image.



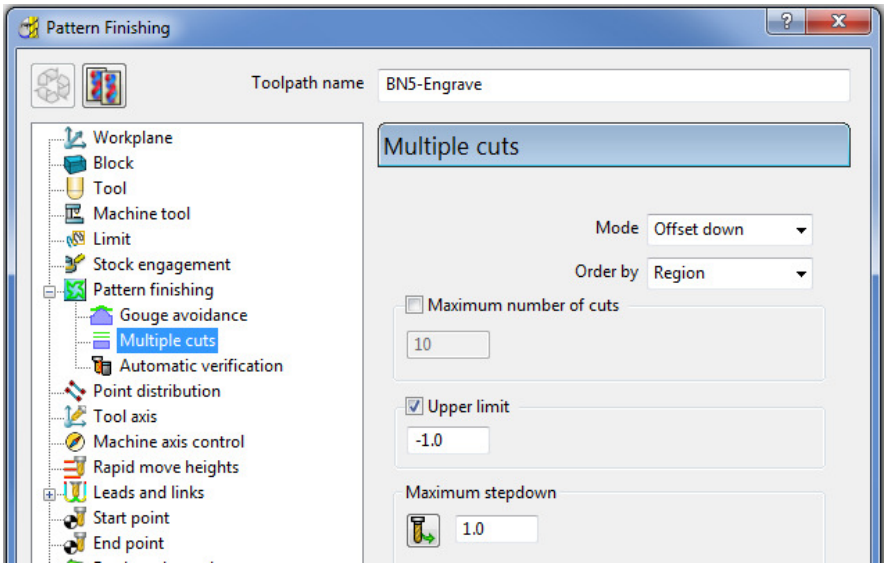
Base Position - **Drive Curve**

Axial Offset **-3.0**

Gouge Check – **un-ticked**

Thickness **0**

11 Select the **Multiple cuts** page and input data exactly as shown below.



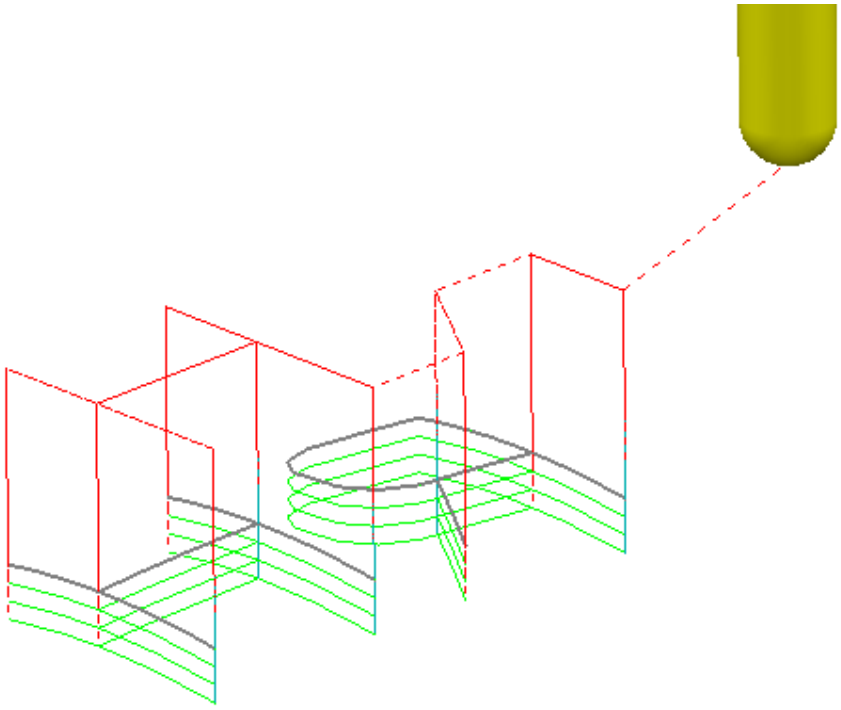
Mode - **Offset down**

Upper limit – **Ticked** with value **-1.0** (First pass at 1mm below the **Pattern**)

Maximum Stepdown **1.0**

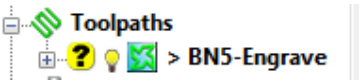
12 Click **Calculate** to create the **toolpath** and then **Close** the dialog.

Copyright © Delcam plc



The Reversed lettering, Pattern 'RH' is now recessed into the Punch form to a depth of 3mm.

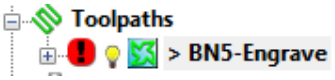
*The **Toolpath** will initially be identified in the **PowerMILL explorer** with a **yellow warning**.*



13 Right mouse click on the **Toolpath** and from the local menu select **Verify** > **Toolpath**.

14 In the **Toolpath Verification** dialog (with the default **Check** – **Gouges** set) select **Apply**.

Once verified for **gouges**, the above ***Toolpath*** **BN5-Engrave** will then be identified in the **PowerMILL** ***explorer*** with a **red warning**.



15 **Save** the **Project**.

 Copyright © Delcam plc

11. Curve Editor

Introduction

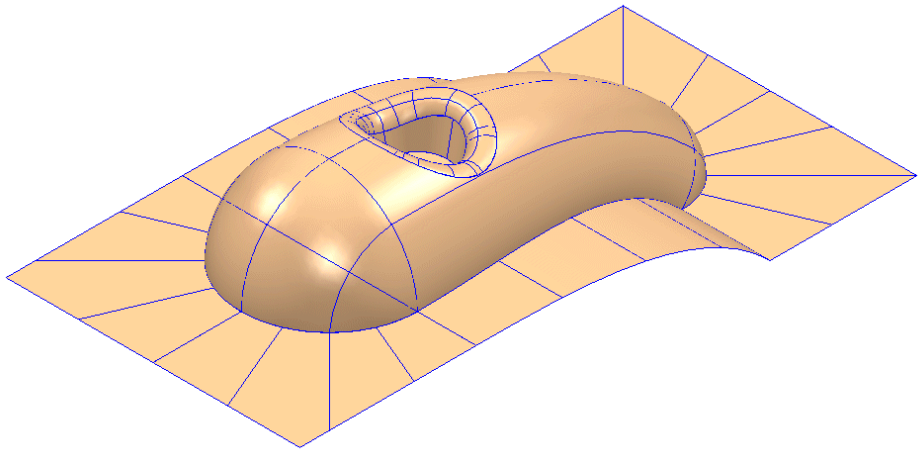
This is a ***wireframe constructor*** within **PowerMILL** that provides the user with a powerful set of tools for the creation and modification of **Boundaries** and **Patterns**. It is accessed via the local **Boundary** or **Pattern** menu option, **Curve Editor...** During construction, the **Intelligent Cursor** option allows geometry items to be snapped onto **key points** on a displayed CAD model.

Boundary modifications using the Curve Editor

A **Shallow Boundary** will be created in the following example and although correct to the settings applied, it will benefit from further modification to provide improved machining strategies. The modifications aim to make the toolpaths, continuously follow the lower, component contour by bridging the gap between the 2 outer segments as well as smoothing any jagged sections of the segments. Due to technical limitations, it is not possible to directly apply **Boundary** editing for these changes. The easiest and most effective method is to make use the powerful tools available in the **Curve Editor**.

1. Select **File > Delete All** and **Tools > Reset Forms**.
2. **Import** the **Model**:

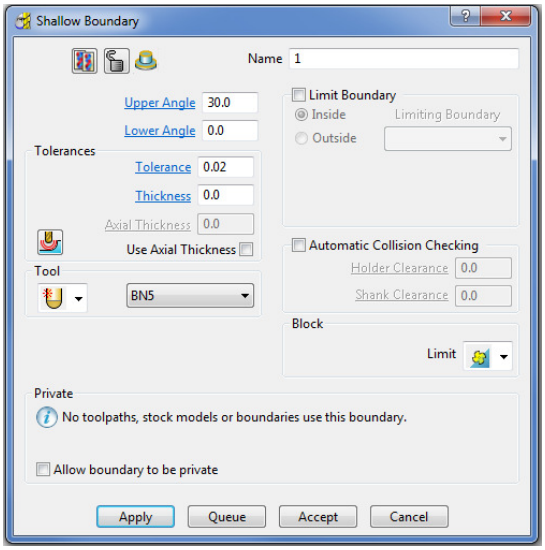
   .....\PowerMILL_Data\models\cowling.dgk



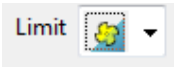
3. **Calculate** a **Block** using ***Defined by*** **- Box** and ***Type*** **Model**.
4. Create a Ball Nose cutter of Diameter 10mm named **BN10**.
5. **Calculate** the **Rapid Move Heights** using the default settings.

6 Use the defalult **Start and End Point** values.

7 In the **PowerMill *explorer***, right-click **Boundaries** and from the local menu select **Create Boundary > Shallow**



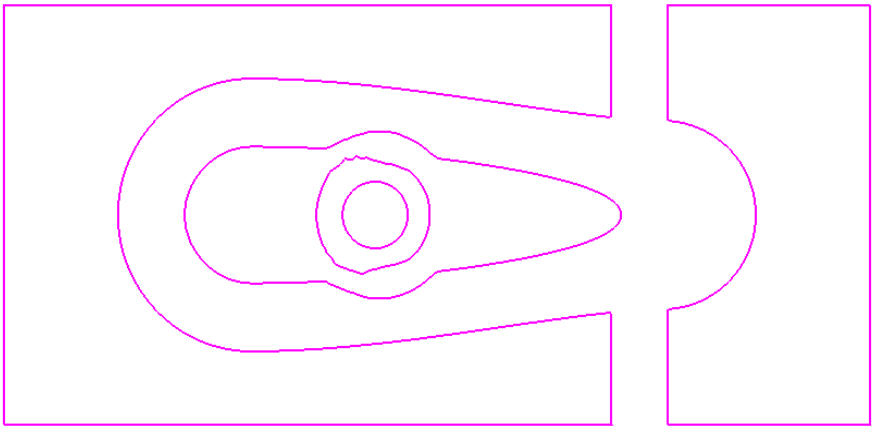
8 Input ***Tolerance* 0.02**

9 Set **limit tool centre to block edge**. 

10 Un-tick **Allow boundary to be private**.

11 Select **Apply** followed by **Accept**.

12 Undraw the **model** to view the new **Boundary**.



The **Boundary** will be modified so that the gap is removed between the 2 outer, closed, segments to form separate inner and outer segments. In addition to this, two of the other segments, which contain jagged sections, will be smoothed.

13 Right-click **Boundary** 1 in the **PowerMILL *explorer*** and from the menu options select **Curve Editor**.

The **Curve Editor** toolbar will appear above the graphics area.

Some licensed icons are temporarily greyed out, only becoming available when a related entity is selected.

Copyright © Delcam plc

14 Select a **view down Z** for a top view of the inserted boundary.

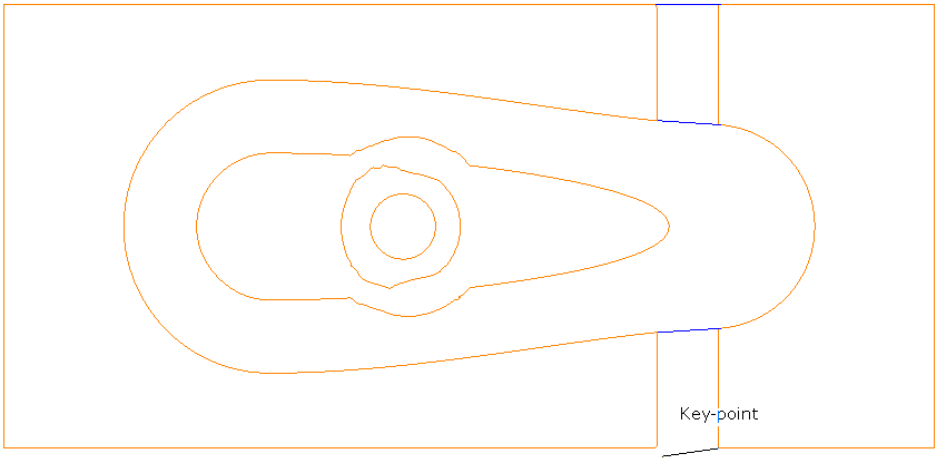
15 Click **Line** from the toolbar then select **Create a single line**.

*Note: It is essential that the **Intelligent Cursor is active** (ticked):-*

If not, this is set in the **Curve Editor** > **Options** menu:-

-- **Intelligent Cursor** -- **Snapping Options** -- (Tick all boxes).

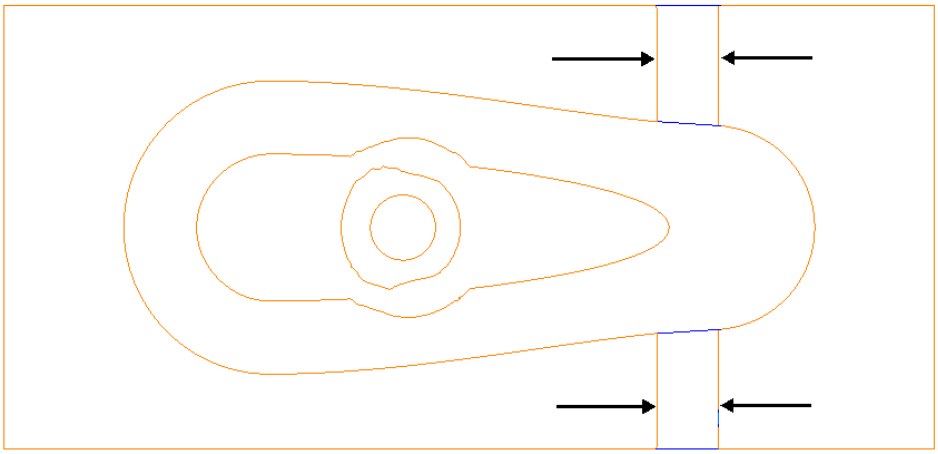
16 Using the left mouse button, snap (**Key points**) to create single lines across the ends of the four gaps, as shown in the image below.



17 Select the **Single Line** option to switch it off.

18 To remove redundant spans on the curves, click the **Limits** option to access **Interactively limit wireframes**.

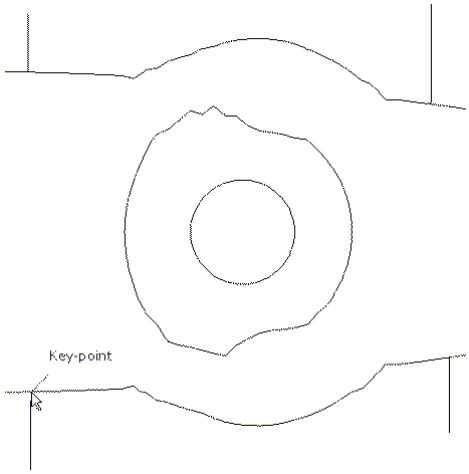
19 Click **Interactively limit wireframes** and using the left mouse button, select the parts of the curve (arrowed) to be removed.



20 Select the **Interactively limit wireframes** option again to switch it off.

21 ***Zoom*** into the middle of the area covered by the 3 inner segments.

Note the jagged effect along these curves.



22 Create 4 **single lines**, snapped to key points and pointing outwards from the curve outside the jagged regions (as shown above).

These will be used as markers to provide a start and end point for new, replacement smoothed curves along the jagged sections. The markers will then be used as the limit to trim away the unwanted, jagged sections of the original curve.

23 Click **Curve** on the **toolbar** then select **Create a Bezier Curve** from the **Curve** toolbar.

24 For the upper section, generate a new, smoother, **curve** by snapping strategically placed key points within this region, starting and ending by snapping to the ends (**Key Point**) of the marker lines.

The selection of suitable key points requires a certain amount of skill on the part of the user. Where it is required to follow the original form closely input several, equispaced points. Where the original shape needs to be smoothed out input the minimum of equispaced points along the form.

Copyright © Delcam plc

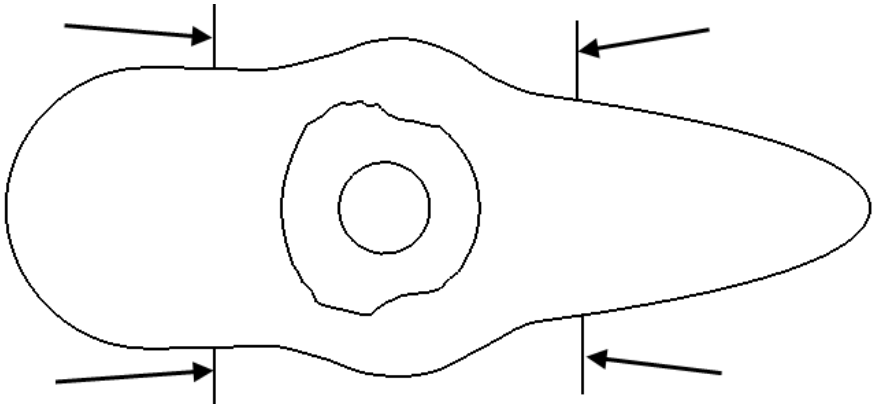
25 Repeat this process for the lower section.

26 Select the upper, original curve and select the **Cut Item** option.

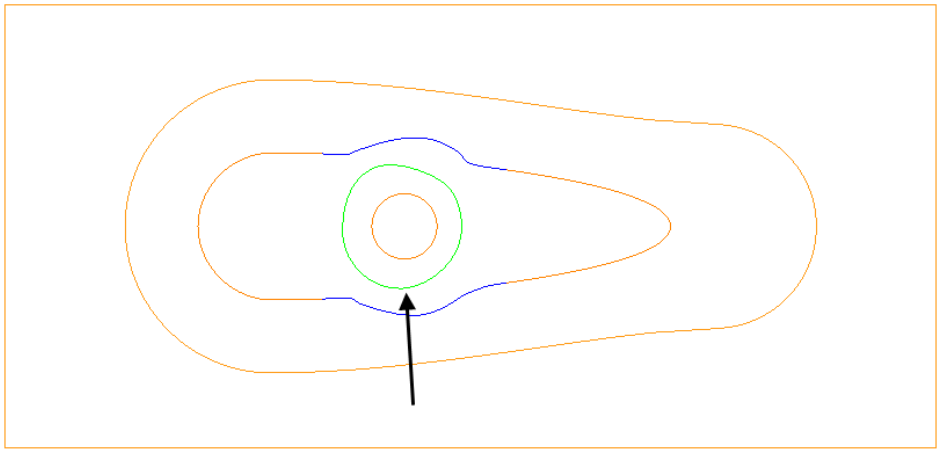
27 Click the **key points** at the lower end of each line to cut the original curve.

28 Repeat the above process on the lower, original curve then select and **Delete** all the unwanted segments between the 2 pairs of **lines**.

29 Then, select and **Delete** the 4 vertical lines.



30 Right-click away from the wireframe and select **Unblank** from the flyout menu.

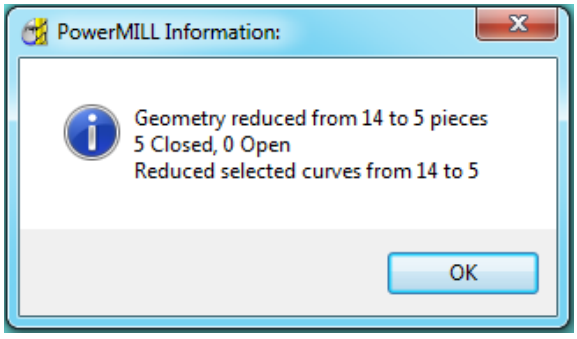


31 Create a new smooth **Bezier Curve** to replace the one indicated above.

*The **Curves** used in the above wireframes will consist of open or closed segments, but if they are to be used to recreate a **boundary** they must form single entity, closed loops. Only closed forms will appear as **boundary** segments when returned to **PowerMILL**.*

1 Select all **boundary** segments by dragging a box over them with the left hand mouse key depressed.

2 Select the option **Merge** selected segments to join together all groups of **curves** that butt up to each other.



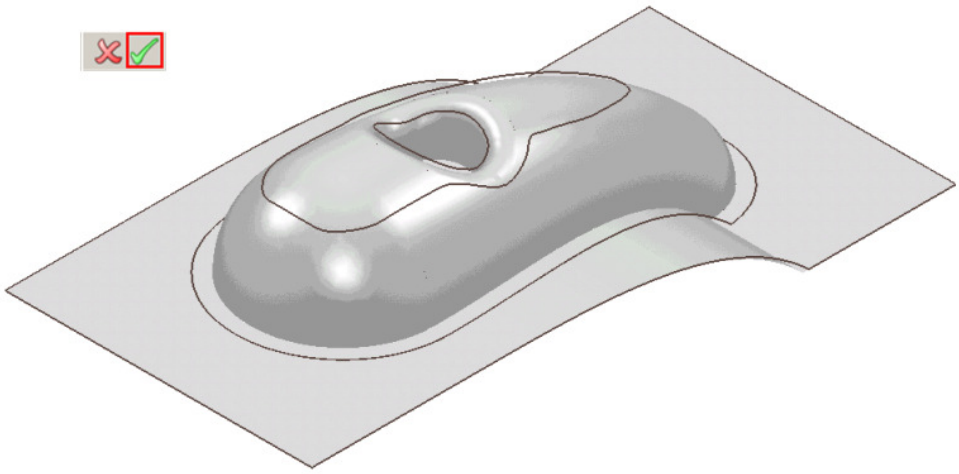
*The wireframe will now consist of five single segments as reported in the **PowerMILL Information box** (above right).*

3 Left click in the graphics area away from the segments to deselect them.

*If any segments are selected then the remainder will no longer exist on return to the main **PowerMILL** window.*

Copyright © Delcam plc

4 Select to **Accept** changes and export the wireframe back to **PowerMILL** to replace the original boundary.



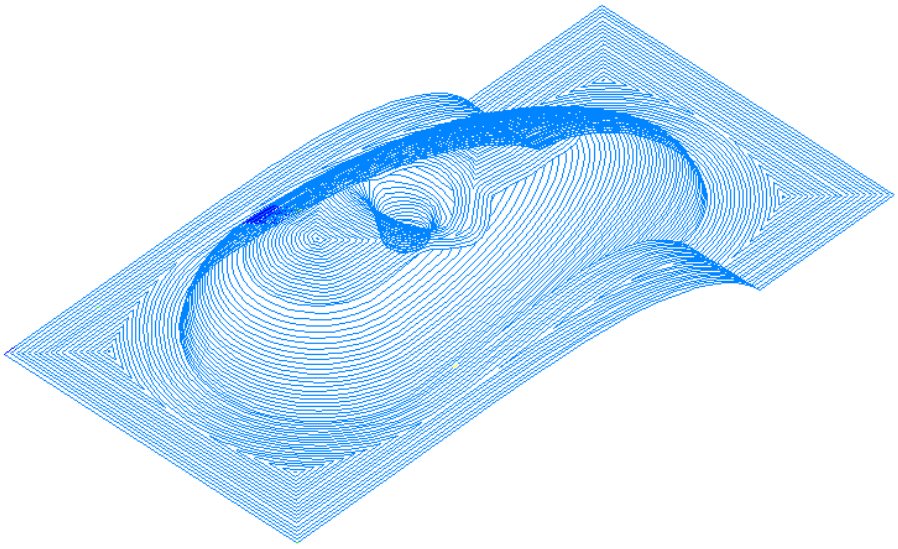
The modified **boundary** is positioned back over the model close to its original 3D form. The new smoothed contours are now a more suitable shape for use in the **3D Offset Finishing** strategy.

5 Generate a **Constant Z Finishing** toolpath with **Stepdown** 1 and in the **Limit** page set the **Boundary** - **Trimming** option to **Keep outside** (Steep areas).

6 Generate a **3D Offset Finishing** toolpath with **Stepover** 1 and in the **Limit** page set the **Boundary** - **Trimming** option to **Keep inside** (Shallow areas).

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

.....\COURSEWORK\PowerMILL-Projects\Boundary-mod



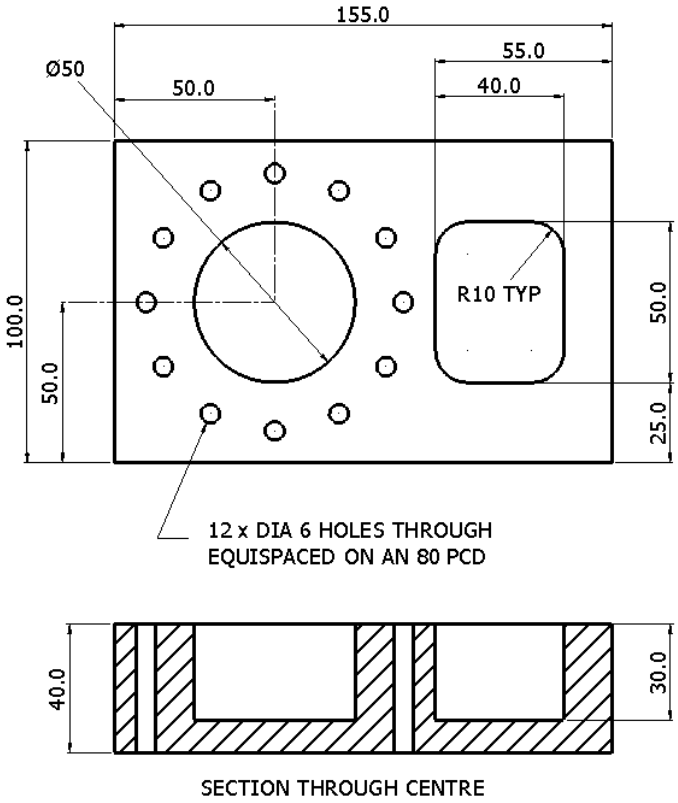
These are the resultant toolpaths with the **Leads** and **Links** undrawn.

*The techniques used in this example can equally be applied to **Patterns**, which also have the added advantage on being able to exist both as open or closed segments (**Boundaries** cannot contain open segments).*

*The same **Curve Editor** toolbar is accessed via the local **Pattern** menu.*

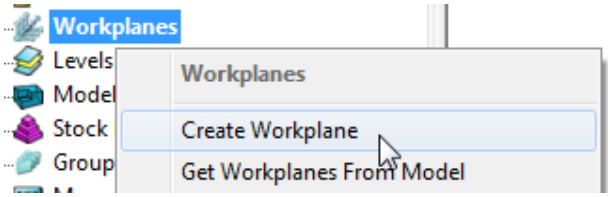
Input of Drawing detail as a Pattern using the Curve Editor

An empty **Pattern** can be inserted into the **Curve** editor thereby providing the user with an effective way to input ***Drawing Detail*** which could in turn be used in the creation of 2.5D **Feature** Sets.

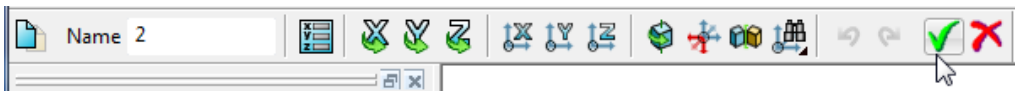


1 Select **File** - **Delete All** and **Tools** > **Reset Forms**.

2 In the **PowerMILL *explorer*** select **Workplanes** - **Create Workplane**.



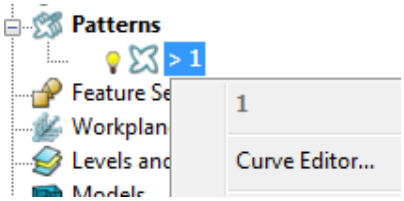
3 The **Workplane** editor toolbar will become active as soon as a new **workplane** is created on the global datum.



4 Select the ***green*** tick to exit the **Workplane Editor** toolbar.

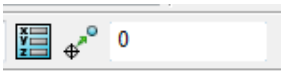
5 From the **PowerMILL *explorer***, create an empty **pattern**, right mouse click on it and from the local menu select **Curve Editor**.

Copyright © Delcam plc



6 From the **Curve Editor** select **Rectangle** option.

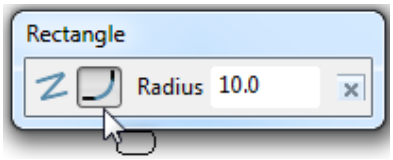
7 Input **0** in the ***Command Input*** box (located to the right and below the graphics area) and press the return key to accept.



This will fix the corner of the **Rectangle of Lines** to the active XYZ datum.

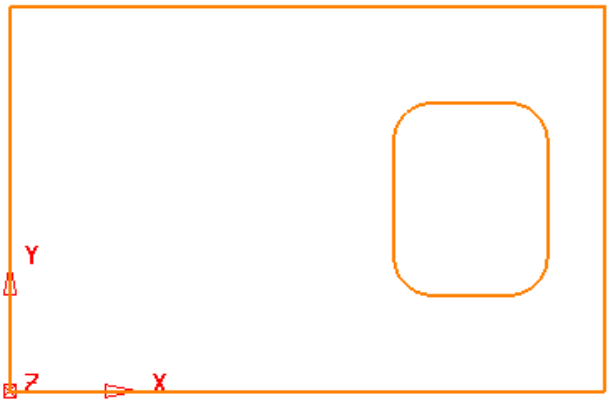
8 Input **155 100** in the **Command Input** box and press the return key to accept the X and Y dimensions of the rectangle.

9 Select the corner **Radius** option and input **10**.

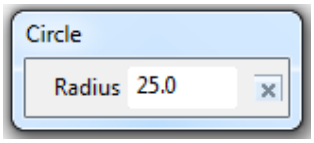


10 Create a new **Rectangle of lines** using **100 25** to define the start corner.

11 In the data input box, switch from **Absolute** to **Relative** input and enter **40 50** for the X and Y dimensions.

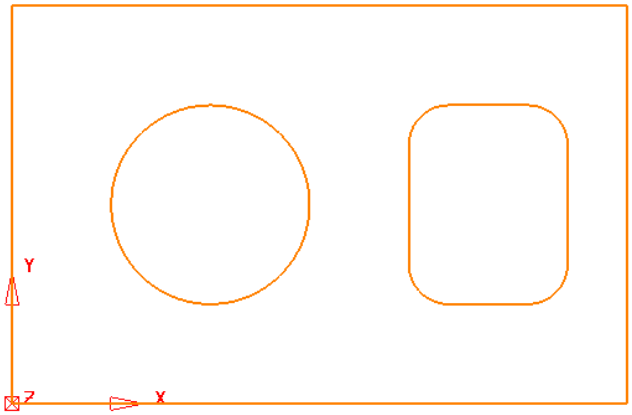


12 From the **Create a circle** options select the **full Circle** option.



13 Input a **Radius** value of **25**.

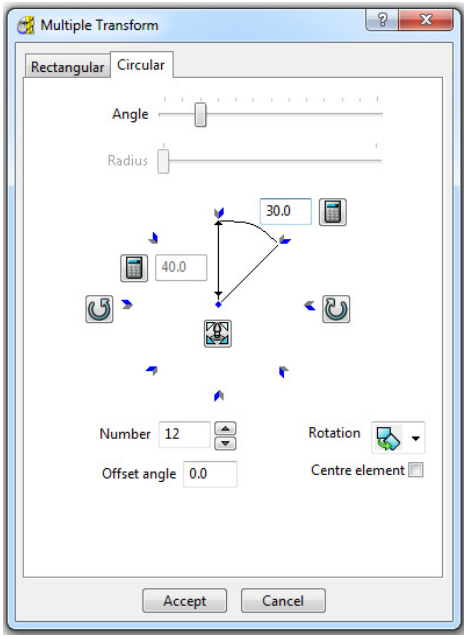
14 Input **50 50** in the **Command Input** box followed by the return key.



15 Create another **Circle** with a centre at **50 90** and a radius value of **3**.

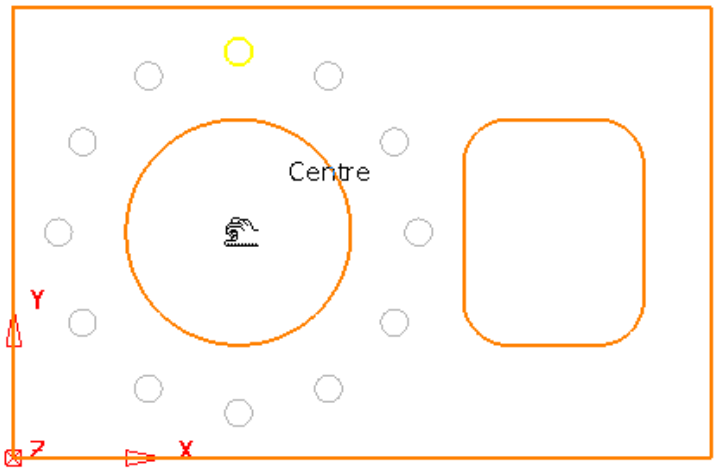
16 Click on to close the **Circle creation** options.

17 With the new Dia 6 circle selected, open the **Transforms** options and select **Create pattern of objects** .

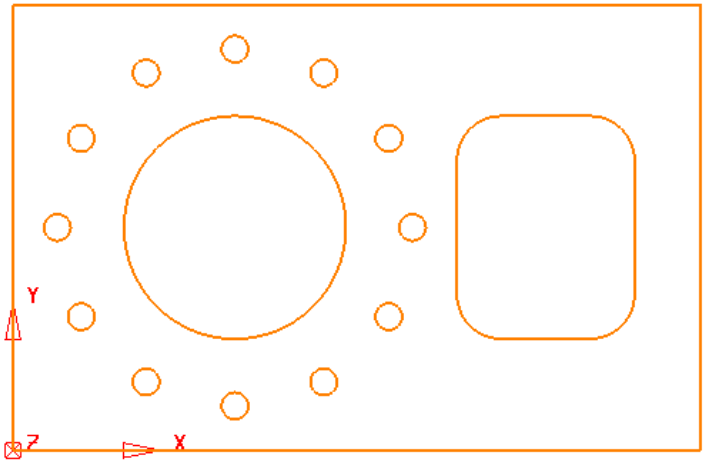


 Copyright © Delcam plc

18 On the dialog, select the **Circular** tab and enter **Angle 30**, **Number 12** and select **Reposition rotation axis** , and snap to the centre of the large circle.



19 Select **Accept** on the **Multiple Transform** dialog.

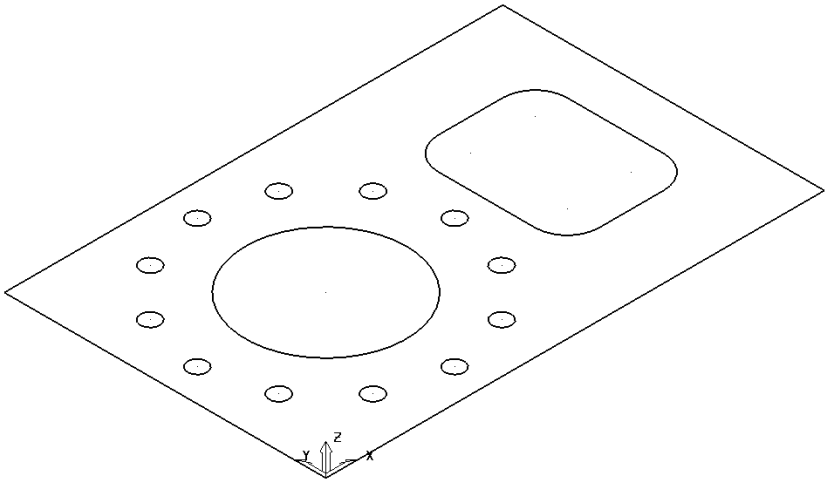


The geometry creation is now complete and is ready to be returned to the **PowerMILL Project** as the contents of the **Pattern**.

20 Left click in the graphics area away from the segments to deselect them.

If any segments are selected then the remainder will no longer exist on return to the main PowerMILL window.

21 Select to accept changes and export the wireframe back to **PowerMILL** to replace the original boundary.



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

.....\COURSEWORK\PowerMILL-Projects\2DModel

Copyright © Delcam plc

12. 2D Machining

Introduction

PowerMILL offers a range of 2D strategies that include ***2D* Curve** machining, **Chamfer** Milling, **Face Milling**, and ***2.5D* Feature** machining. The original **wireframe** can either be **Pattern** or **Model** entities.

Feature Machining

Features are wireframe entities that are extruded along Z. During creation they are specifically defined as a **Pocket**, **Slot**, **Boss,** Hole, **Circular pocket**, or **Circular boss**. It is also possible to extract **Hole** features directly from the 3D model. They can also be created during **3D Area Clearance** strategies to enable the pre-drilling of plunge move, positions. Groups of **Features** are created within a **Feature Set**. It is essential that a **Feature Set** includes all separate **Features** that need to work together to create a **2.5D Feature** machining strategy.

On completion of a **Feature Set**, the 4 **Feature Set** options in the **2.5D Area Clearance** strategies will create roughing, semi finishing, and finishing strategies.

Drilling stategies can only be applied to **Hole Features**. A comprehensive, list of cycle types are supported that include; Standard drilling, Boring, Helical milling, Tapping and Thread Milling.

The six different types of **Feature** include:-

- **Pocket**:- Machining will only occur fully inside this type of closed **Feature**.
- **Slot**:- Machining will occur with the tool running central, left, or right of this type of open/closed **Feature**.
- **Boss**:- Machining will only occur fully outside this type of closed **Feature**.
- **Hole**, this type of **Feature** is specifically used by the **Drilling** strategies.
- **Circular Pocket** - a circular pocket is defined from points, circles or curves.
- **Circular Boss** - a circular boss is defined from points, circles or curves.

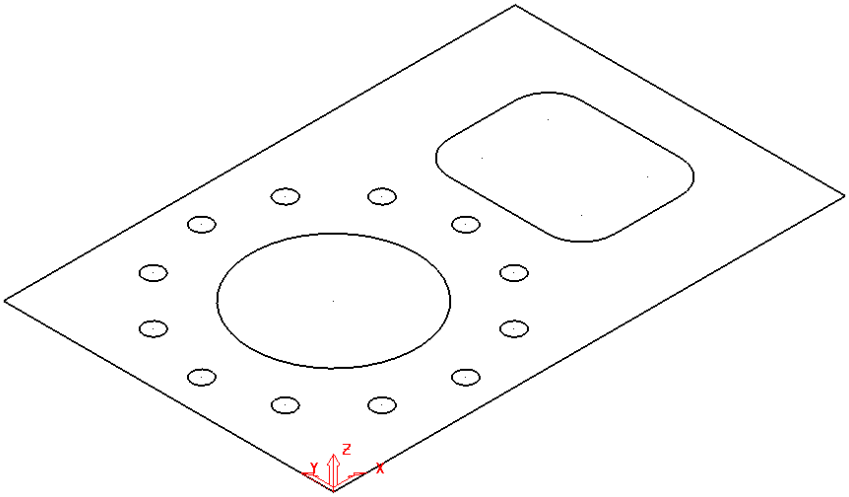
It is not possible to change an existing a Feature to one of a different type.

Creating Features from Pattern segments

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

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

.....\PowerMILL_Data\Projects\2D-Drawing

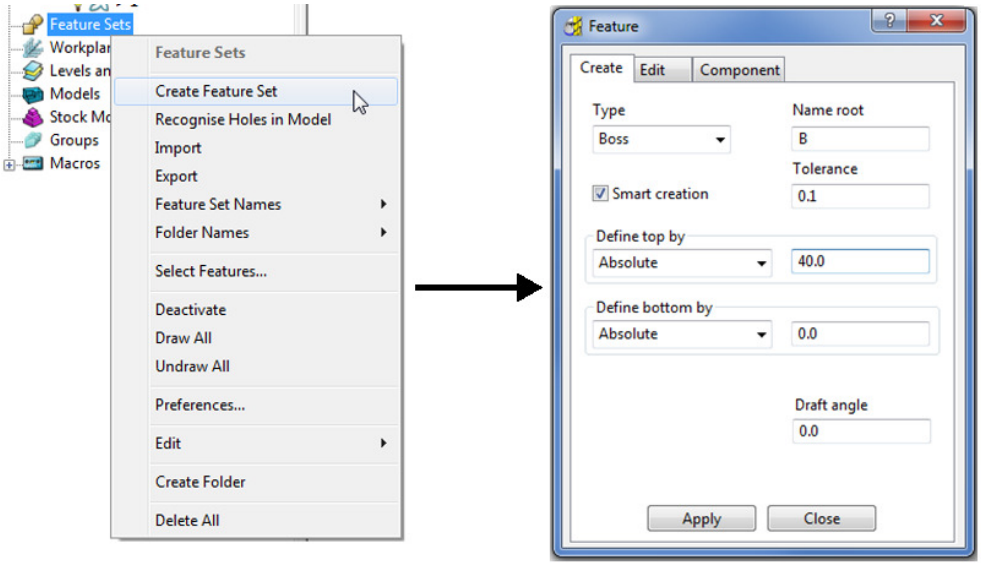


The imported **Project** contains a **Pattern** defining 2D geometry ready to be used to create a **Feature Set**.

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

.....\COURSEWORK\PowerMILL-Projects\2DPatternExample

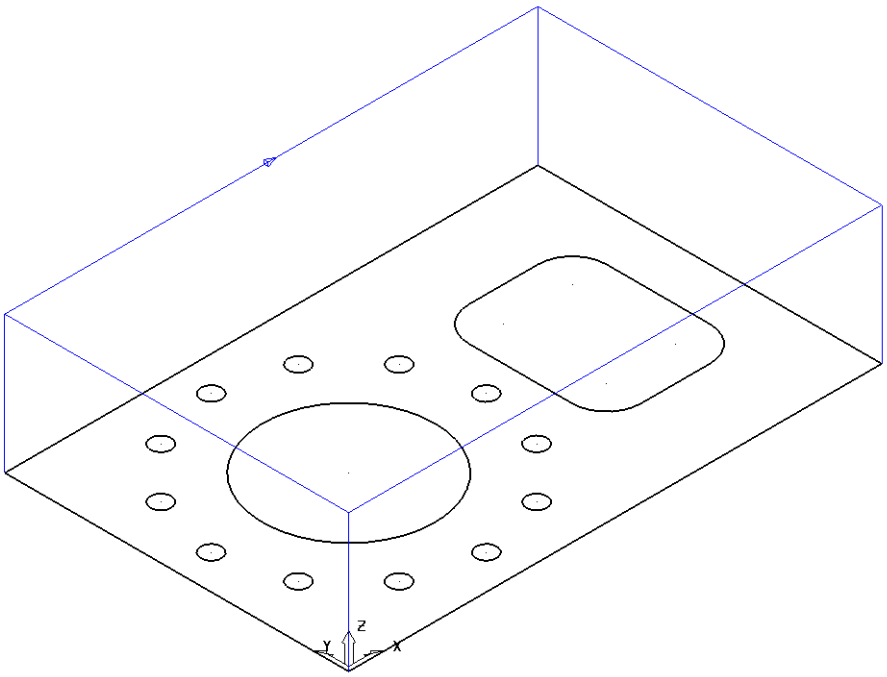
4 In the **PowerMILL *explorer***, right-click on **Feature Sets** and from the local menu, select **Create Feature Set**.



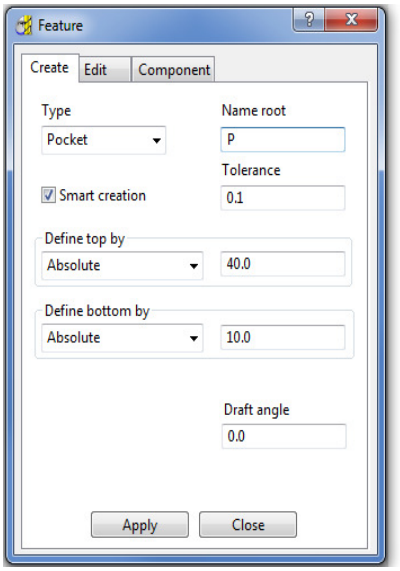
A new (empty) **Feature Set** - **1** will appear in the **PowerMILL *explorer*** and the **Feature** dialog will open ready to build the 2.5D form.

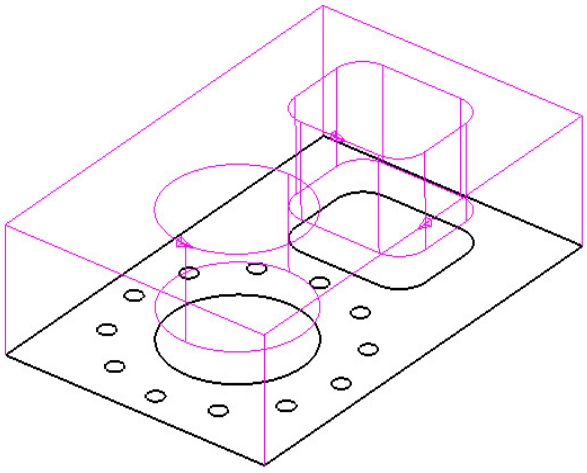
5 Select the larger, rectangular **Pattern *segment*** and enter values exactly as shown in the above right **Feature** dialog before clicking **Apply** (Do not close the dialog).

Copyright © Delcam plc



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

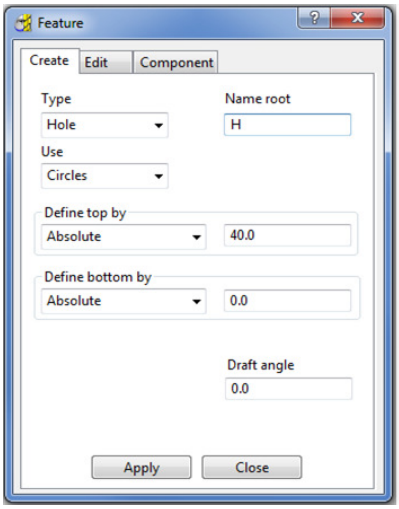


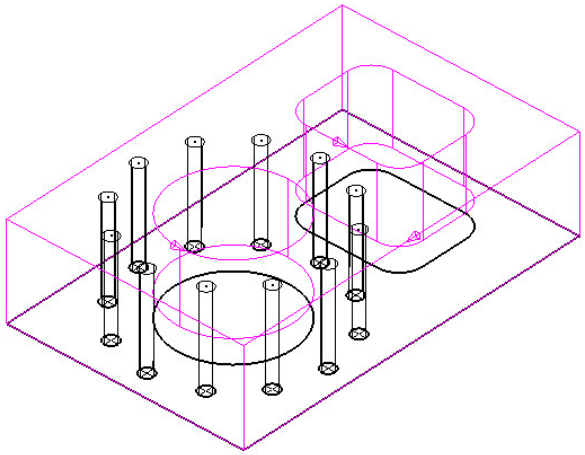


6 Select both, the ***rectangular segment*** (with corner fillets) and the large ***circular*** segment.

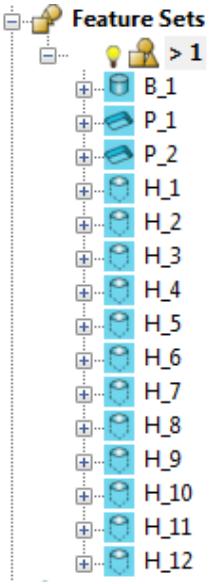
7 Enter values exactly as shown in the above left **Feature** dialog before selecting **Apply** (Do not close the form).

If you close the **Feature** *dialog it can be re-opened in the* ***PowerMILL explorer*** *by a right click on the* ***Feature*** *and then from the local menu select* ***Settings****.*





8 Select all of the Dia 6 **circles** (12) and enter values exactly as shown in the above left **Feature** dialog before selecting **Apply**.



The contents of the **Feature Set** (**1**) should be similar to the above illustration (Note: A naming system has been deployed 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 and **Drilling** strategies.

2D Feature Set Machining

1 Define a Dia 10 - tiprad 1 - Length 50 tool named **D10TR1**.

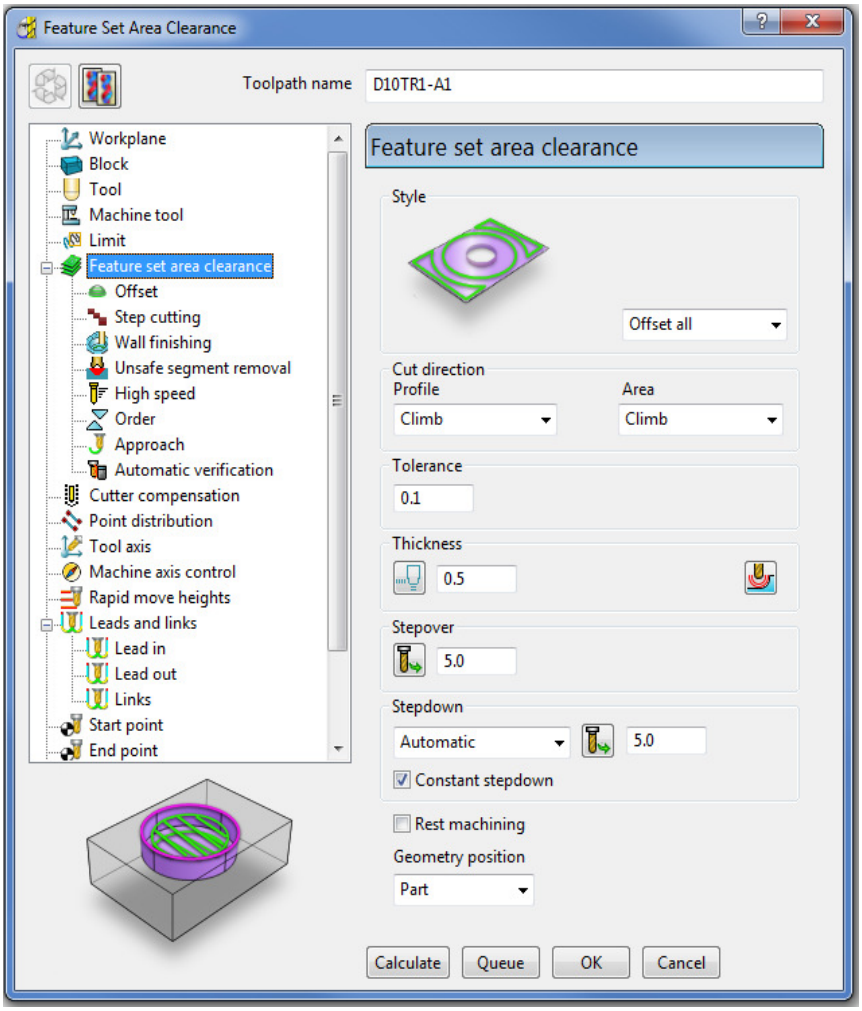
2 Click to open the **Block** dialog, set the **Type** as **Feature** and the select **Calculate**.

3 Click to open the **Rapid Move Heights** dialog and select **Calculate**.

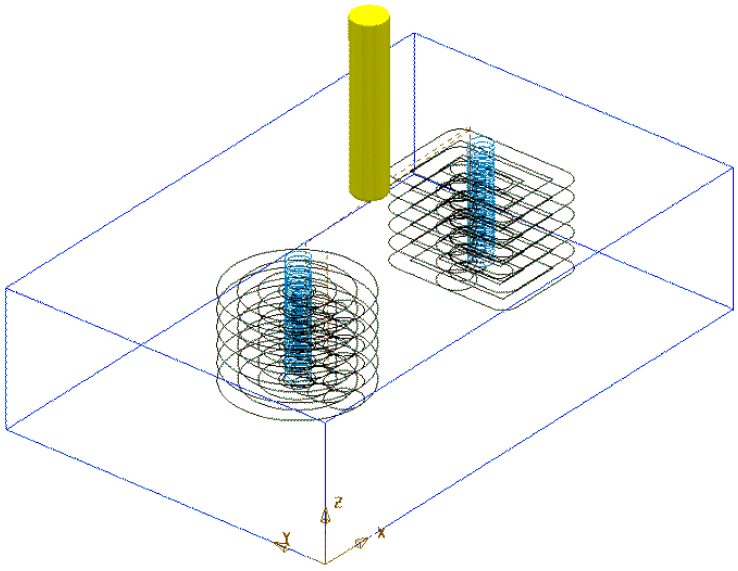
4 Click to open the **Toolpath Strategies** dialog and select **2.5D Area Clearance**.

5 Select **Feature Set Area Clearance** and input the data exactly as shown in the following dialog.

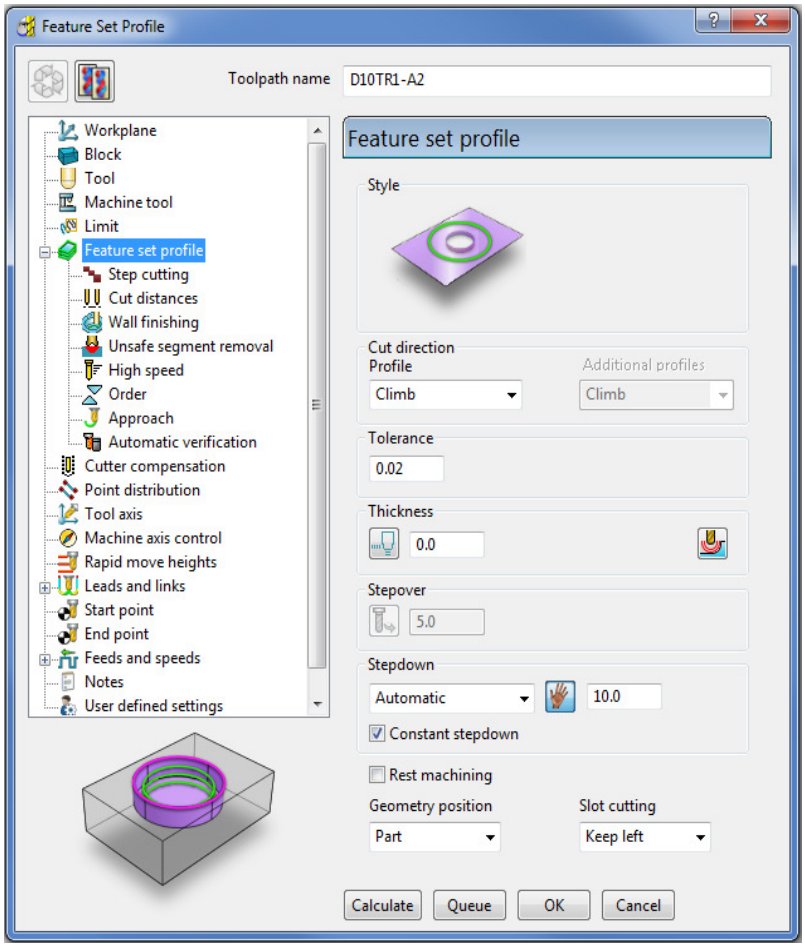
Copyright © Delcam plc



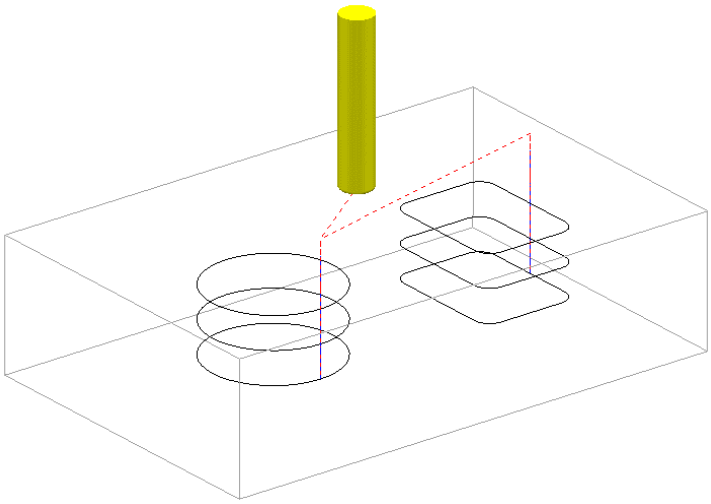
6 Click **Calculate** to process the toolpath and then click **Cancel** to close the dialog.



7 Select **Feature Set Profile** and input data exactly as shown in the following dialog.



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

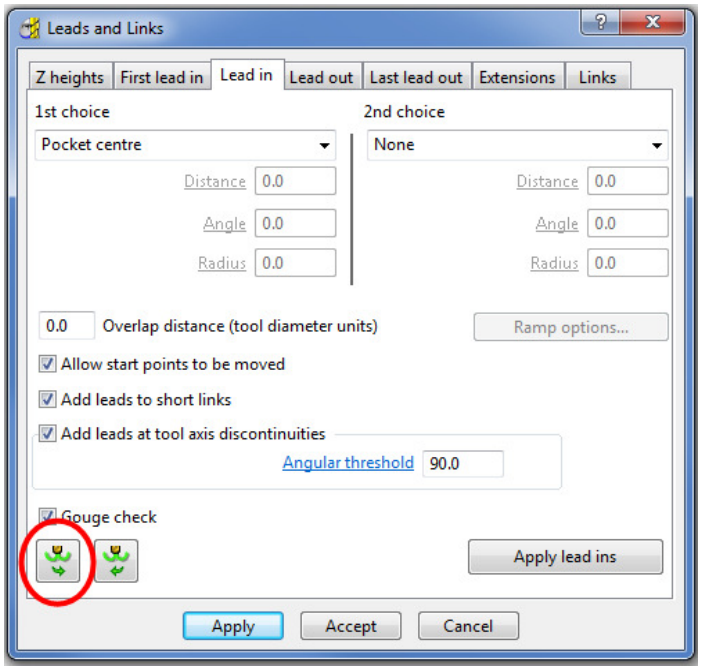


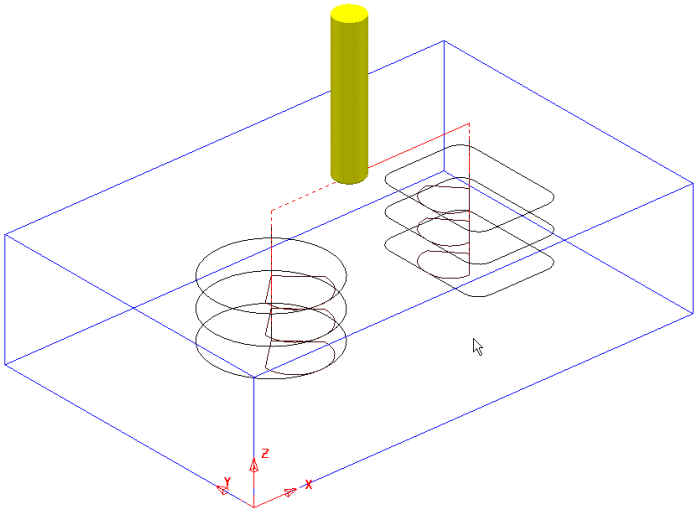
Note the tool is plunging directly onto the form down Z. This situation will be changed retrospectively by applying appropriate options in the Leads and Links dialog.

9 In the **main** toolbar select to open the **Leads and Links** dialog.

10 For **Lead In** - **1st Choice** select **Pocket Centre** and then select the ***Copy to Lead Out*** option followed by **Apply**.

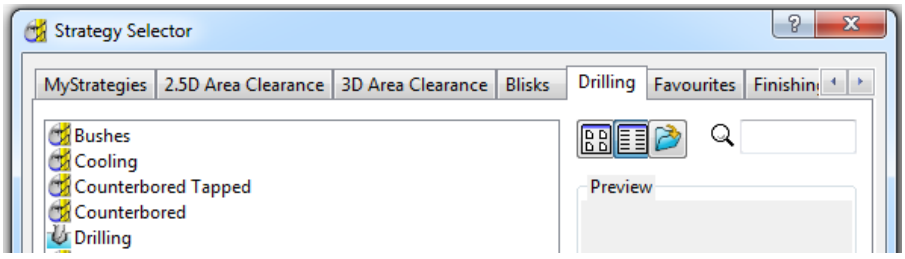
Copyright © Delcam plc



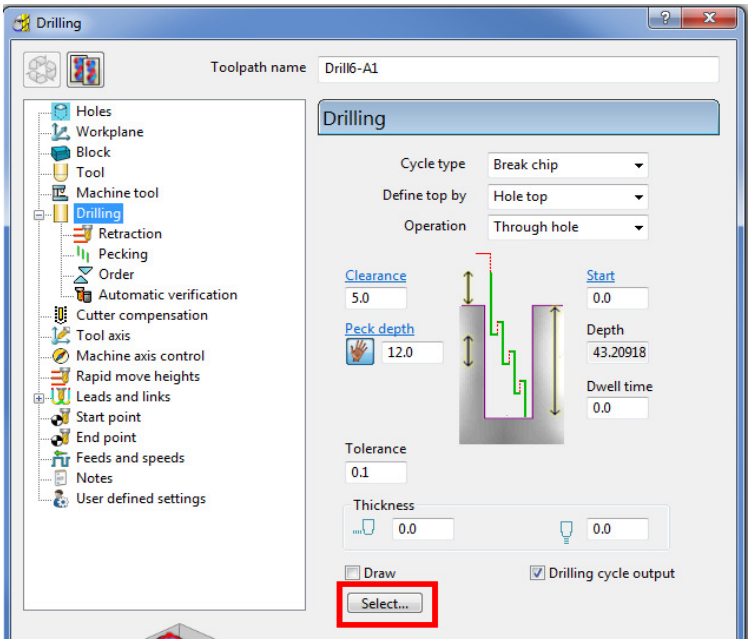


11 Define a Dia 6 **Drill** of Length 50 and named **Drill6**.

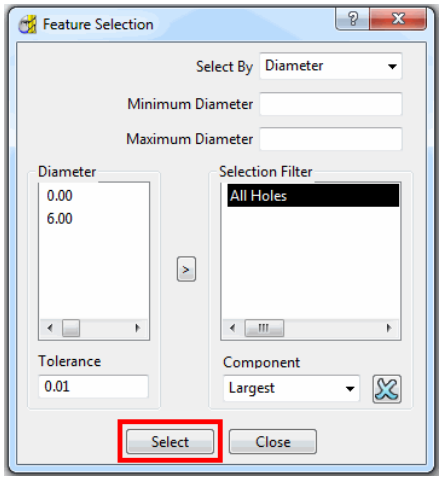
12 Click to open the **Strategy selector** dialog and select the **Drilling** tab.



13 From the list, select **Drilling** to open the **Drilling** dialog (below) and then click **Select** to open the **Feature Selection** dialog.

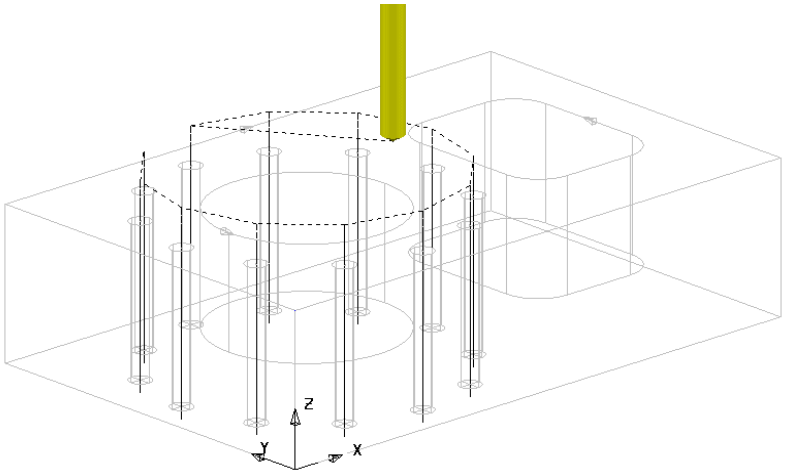


14 In the **Feature Selection** dialog, click **Select** to include all of the defined holes in the **Feature Set**.



15 Enter other data into the **Drilling** dialog exactly as shown above before clicking **Calculate**.

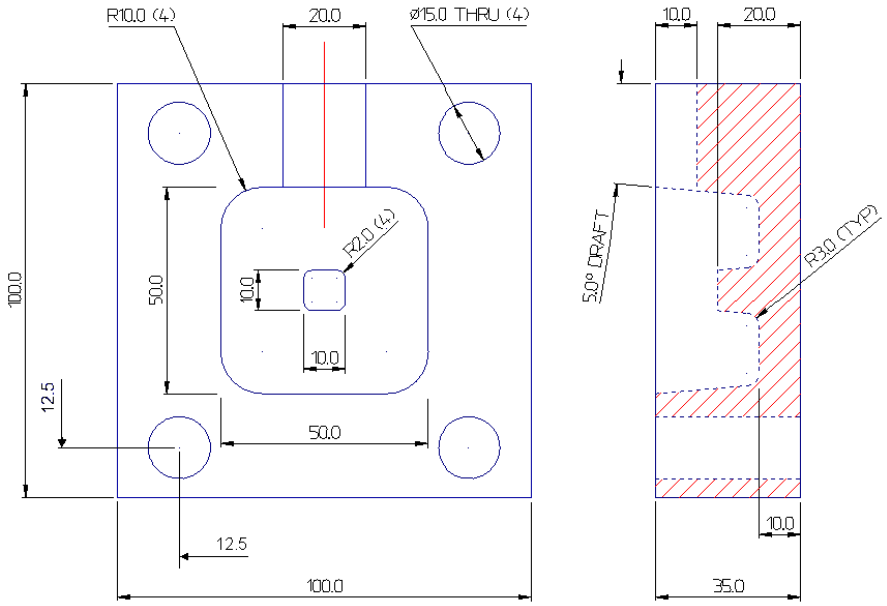
16 **Close** the **Feature Selection** and **Drilling** dialogs.



 Copyright © Delcam plc

2D Machining Exercise

For this exercise the following 2D drawing will be used:



1 Select **File > Delete All** and **Tools > Reset Forms**

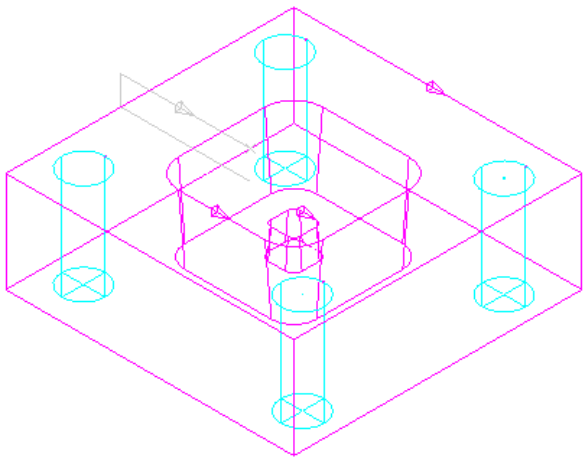
2 **Import** the 2D wireframe **model** (as shown above):

.....\Powermill_Data\Models\2d_Wizard_Example.dgk

3 **Save Project As**:-

.....\COURSEWORK\PowerMILL_Projects\2D_Wizard_EX1

4 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 Feature**.

The four Dia 15 circles are created as **Hole Features**.

2nd Feature Set

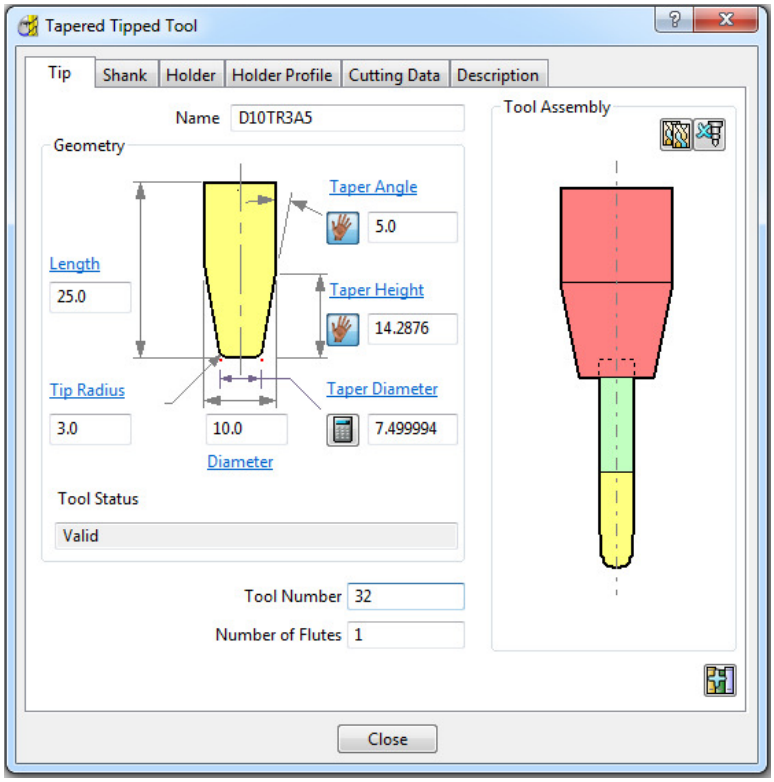
The **Slot Feature** is best created in a second **Feature Set**.

Copyright © Delcam plc

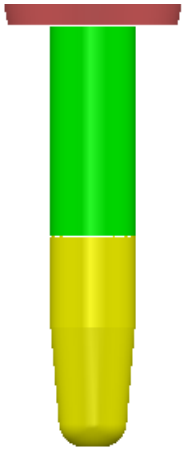
5 Create a Dia10 **End Mill** named **EM10** (Roughing, Semi-finishing of main Pocket and Finishing of Slot Feature).

6 Create a Dia 10 **Tapered Tipped** tool named **D10Tr3A5**, with Tip Radius 3, Taper Angle 5Deg, Length 25, Taper Diameter 7.5 (Final Finishing of tapered walls Main Pocket and small Boss).

7 Create a **Shank** of length 30 and 2 **Holder** entities (Lower Dia 20 and Upper Dia 30) both of length 25.



*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.*



8 Create a Dia 15 **Drill** named **Drill15**.

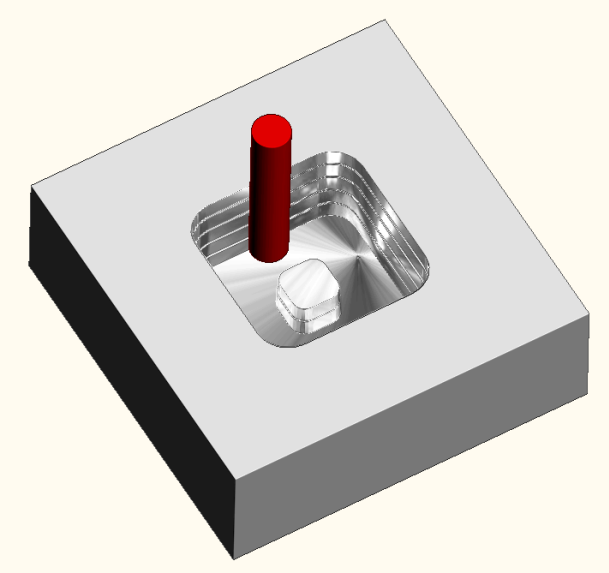
9 Create suitable **2D machining** strategies to produce an end result similar to that shown in the following **ViewMill** simulation.

Copyright © Delcam plc

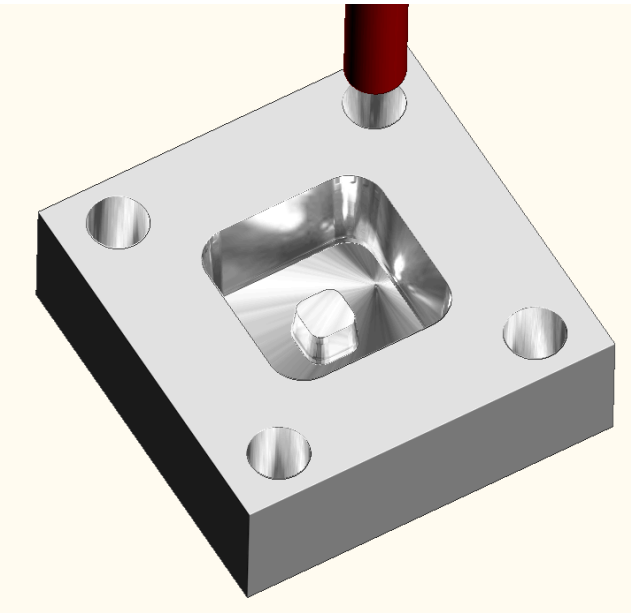
Suggested Strategies:

Main Block and Central Pocket

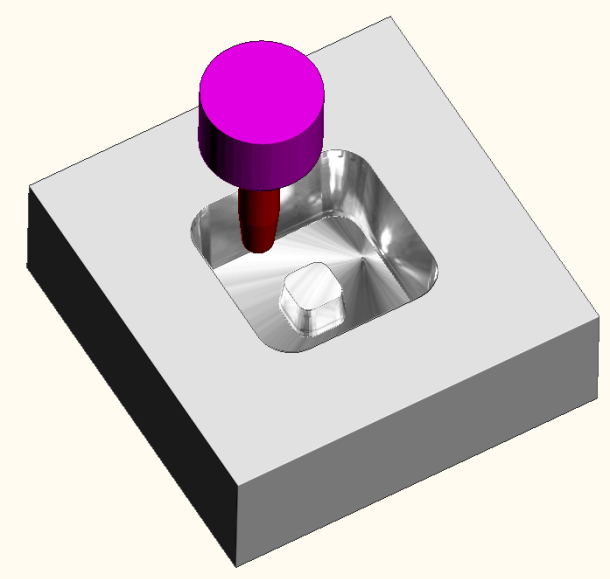
EM10 - Feature Set Area Clear – Offset all



D10Tr3A5 - Feature set Profile



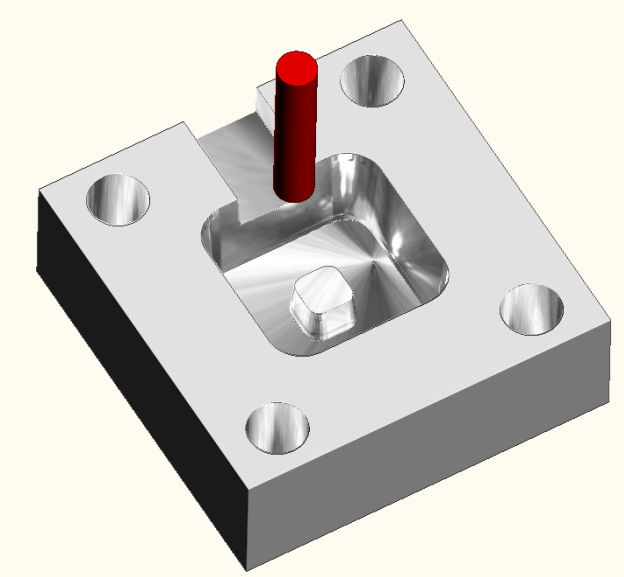
Drill15 - Drilling

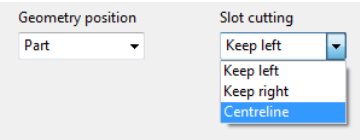


The **Slot** will be machined independently from the main pocket and for this reason it will be created it in a separate **Feature Set**.

The Y dimension of the existing Block will have to be adjusted to fully, include the Slot Feature.

EM10 - Feature Set Profile





The required **Slot** width is achieved by creating 3 toolpaths using the 3 different **Slot cutting** options, starting with ***Centreline***, followed by ***Keep left***, and then ***Keep right***.

Copyright © Delcam plc

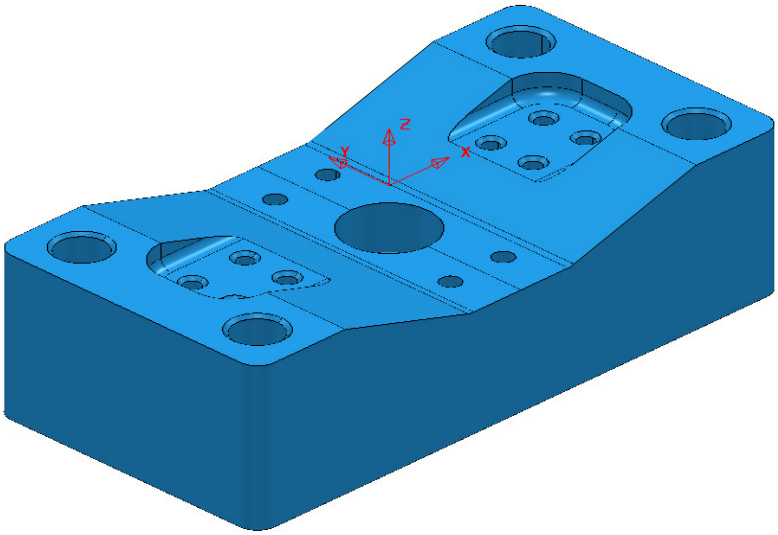
Creating Features directly from holes in the Model

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**.

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

2 Select **File > Import Model**:

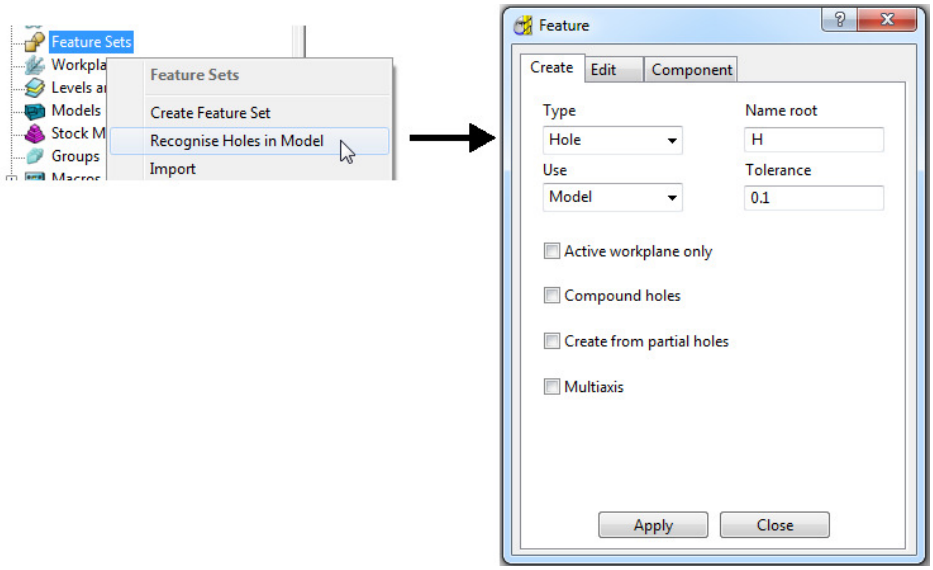
.....\PowerMILL_Data\Models\3Axis-drilling



3 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 more than likely to result in some **holes** being created the wrong way up (It is however possible to reverse a **Hole Feature**).

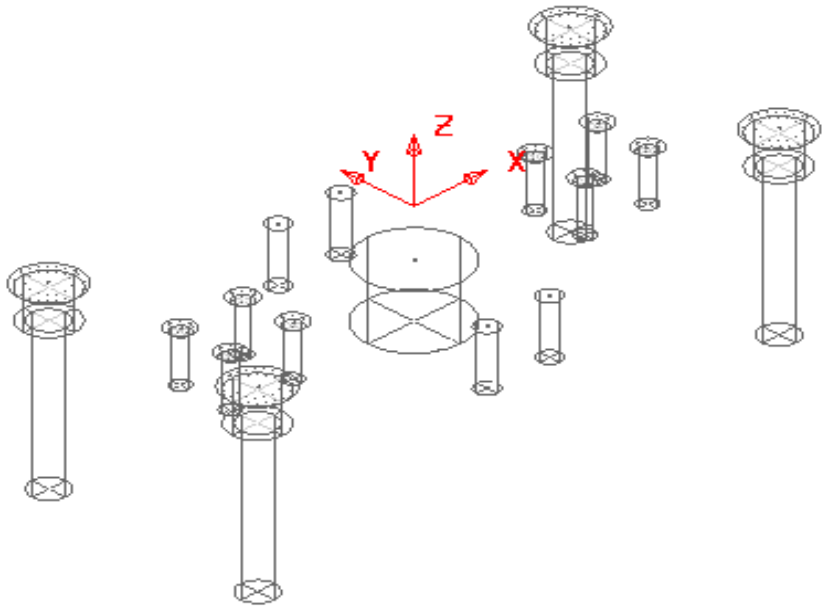
4 Select the whole **model**.

5 In the **PowerMILL *explorer***, right-click **Feature Sets** and from the local menu select **Recognise Holes in Model**.



6 Input data into the **Feature** dialog exactly as shown above and click **Apply**.

7 Undraw the **model**.



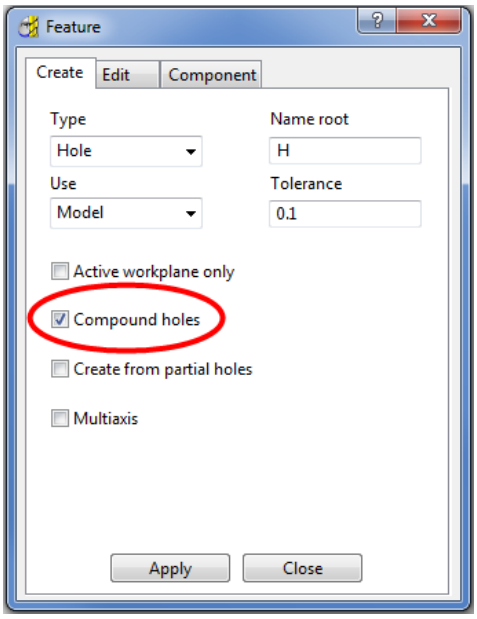
A **Hole Feature** is created for each of the 33 selected, ***Hole*** elements within the **model** (including the Chamfers and Counter-bores as separate **Features**).

Hole Feature creation using Compound Holes

8 De-Activate and un-draw the **Feature** (**1**).

9 Draw and select the whole **Model** again.

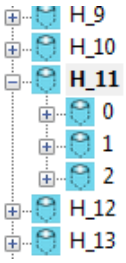
10 In the **PowerMILL *explorer***, right-click **Feature Sets** and from the local menu select **Recognise Holes in Model**.



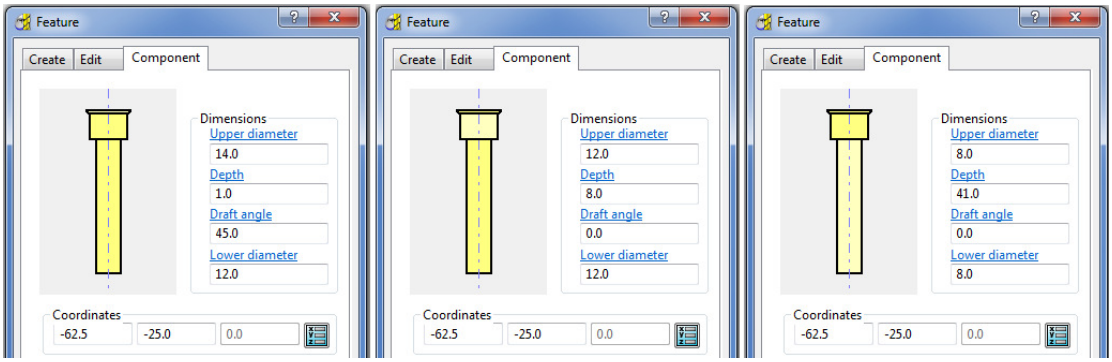
11 Input data into the **Feature** dialog exactly as shown above and click **Apply**.

This time 17 items are created in **Feature Set** (**2**), with the Chamfers and Counter-bores being contained as compound items within the relevant **Hole Features**.

Copyright © Delcam plc



For example; The ***Compound Hole Feature*** **H_11** consists of a Chamfer, Counterbore, and the main Hole. If required all of these sub-items can be dimensionally modified independently of each other.

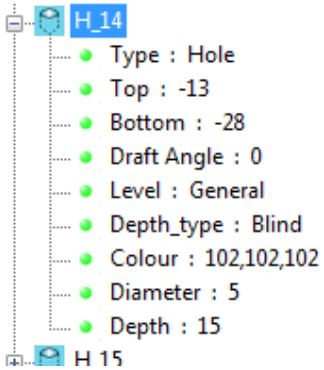


Left click on one of the sub-items on the image in the above **Feature** dialog and the specific dimensions can be edited as required.

12 **Calculate** a **Block** using ***Defined by*** - **Box** and ***Type*** - **model**.

13 **Calculate** the (default) **Rapid Move Heights**.

Before suitable tooling can be selected to perform appropriate Drilling operations it is necessary to obtain dimensional information from the **Hole Features**. This can be obtained by opening up the individual **Features** in the **PowerMILL *explorer***.

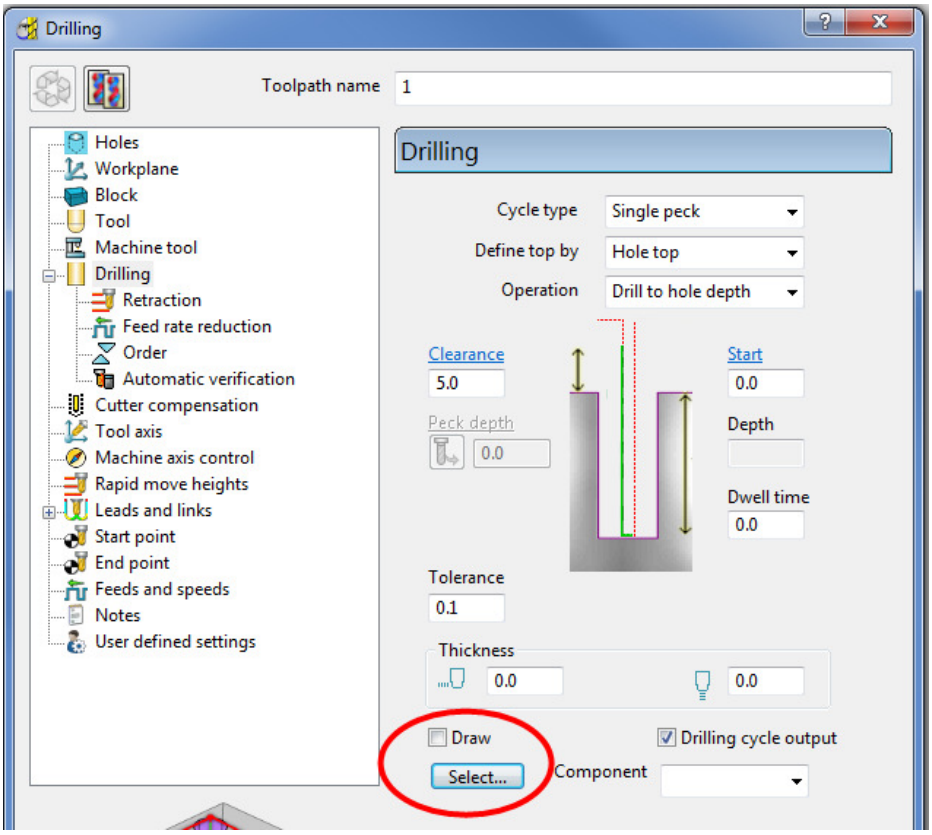


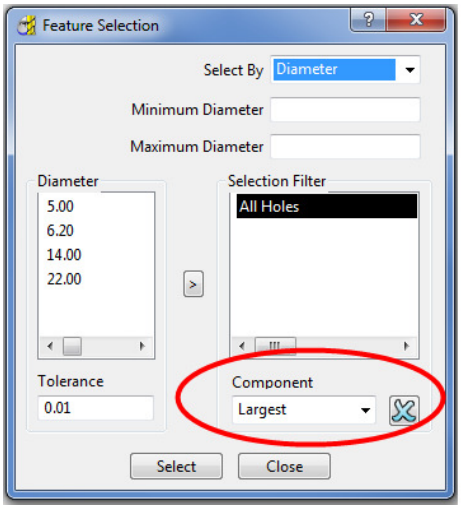
Alternatively, if it is just the different Diameters that are required then this can be obtained from the **Select** filter in the **Drilling** strategy dialog.

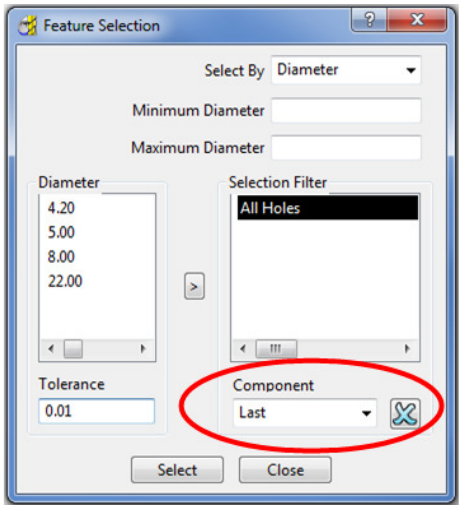
14 Click to open the **Toolpath Strategies** dialog and select **Drilling**.

15 From the ***Drilling*** options select **Drilling**.

16 In the **Drillling** dialog left click on the **Select** tab to open the hole selection filter.







In the **Feature Selection** dialog an individual **Component** of a ***compound hole*** **Feature** can be selected. The full list of **Component** options include:- **Largest**, **Last**, **First**, **Second**, **Third**, **Forth**, **Fifth**. This selection is in addition to the ***Select by:-*** **Diameter**, **Depth**, etc, options.

EXERCISE

17 Define suitable tools, then **drill** and **counter bore** the **Holes** using the **Drilling** dialog options.

Copyright © Delcam plc

Hole Capping

Hole Features can be used as the basis for capping holes in the 3D **model**. If the intersection of the hole and the top face is non-planar, **PowerMILL** will cap the hole with a **surface** that follows the tangency of the surrounding 3D **model**.

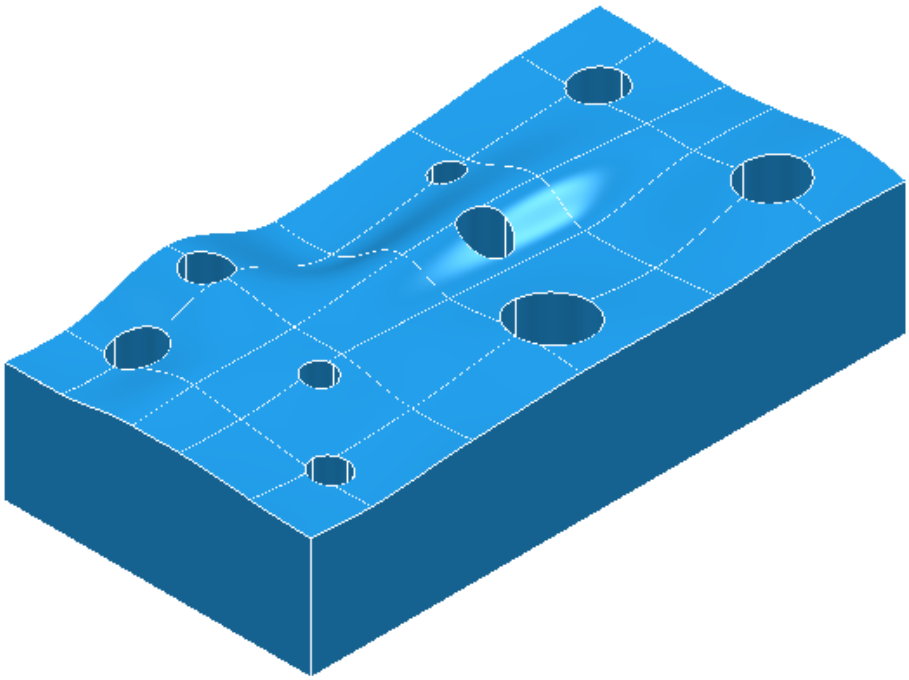
Example

The following example illustrates the process of capping cylindrical **Hole Features** defined from a **3D model**.

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

2 **Import** the **model**:

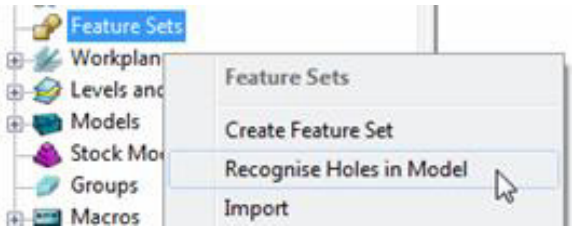
.....\PowerMILL_Data\ Block_with_holes.dgk

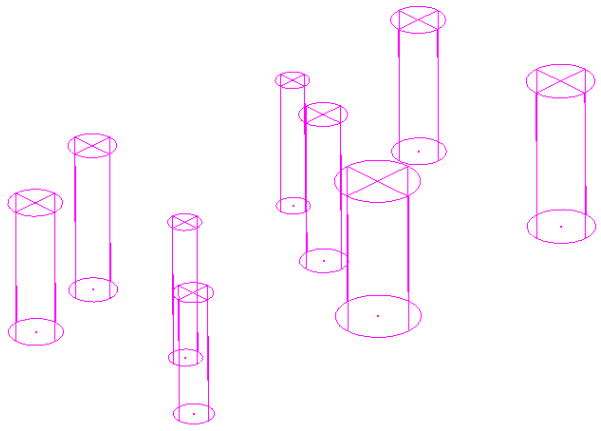


3 Calculate a **Block** using ***Defined by*** - **Box** and ***Type*** - **Model** (In this case the **Hole Features** will be created upside down).

4 Select the whole **model** by by dragging a box over it while the left mouse button is depressed.

5 Select **Recognise Holes in Model**.

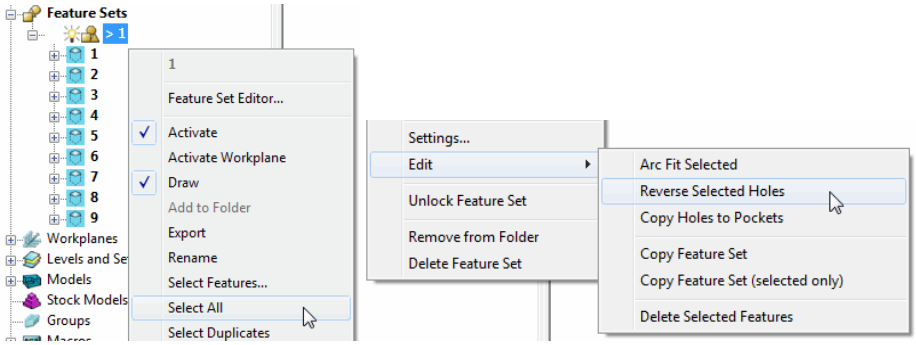




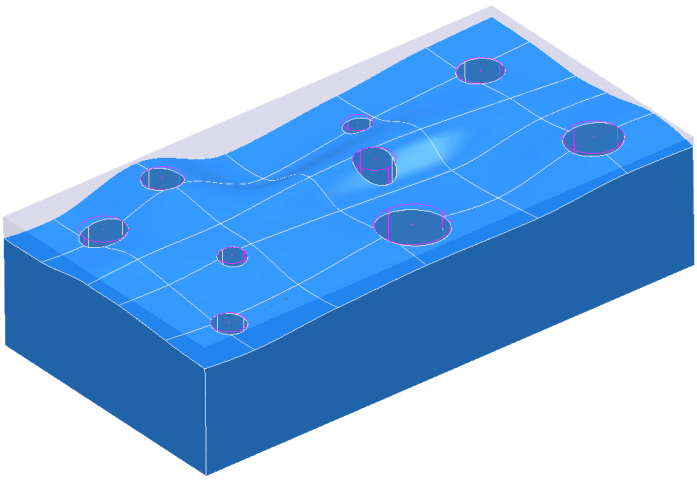
In this case the tops of the **Hole Features** are positioned at the shortest distance from the face of the **Block**. In this case all **Hole Features** are created upside down. If the **Block** had not been defined prior to this, then the **Hole Features** would have been created the 'right way up' relative to the Z Axis.

Any **Hole Features** that are created to a different orientation from the original Z Axis, will automatically be assigned to a separate **Feature Set** and assigned **Workplane**.

In this case the holes were deliberately created upside down to illustrate the **Reverse Holes** editing option.



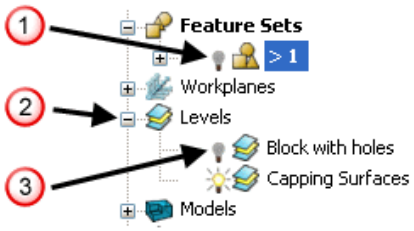
6 Right-click **Feature Set 1** and in the menu click **Select All** followed by **Edit** > **Reverse Selected Holes**.



7 Finally, select the top surface of the **Model** and in the **PowerMILL *explorer***, right-click **Feature Set 1** and select the **Cap Holes** option.

Copyright © Delcam plc

The new **Capping 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.

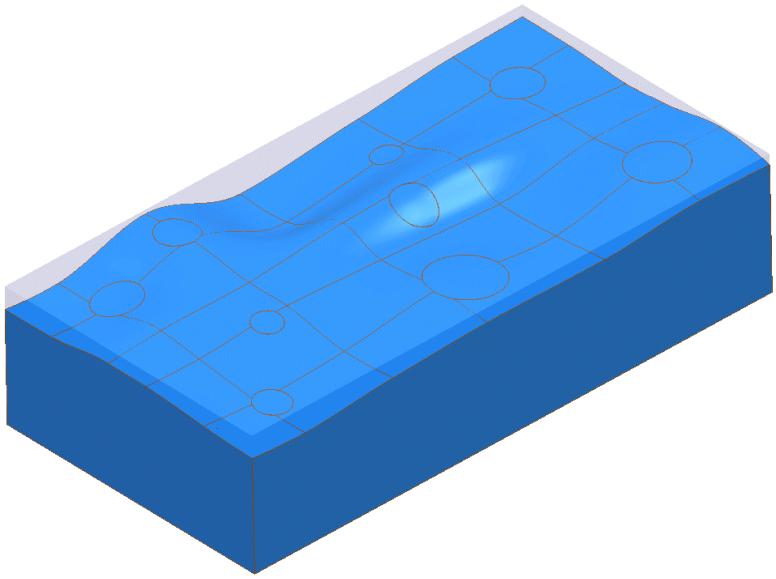


8 Click the ***light bulb*** to undraw **Feature Set 1** (1).

9 Expand **Levels** by clicking on the **+** symbol (2).

10 Click the ***light bulb*** to undraw main **surfaces** (3).

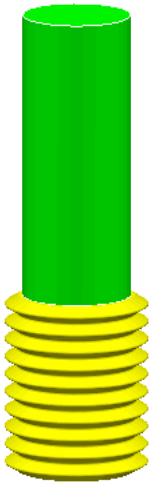
The new **Capping Surfaces** may be created inside out (Surface Normals reversed). If required they can be selected and from the local **Surfaces** menu, **Reverse Selected** can be applied.



*The **Capping Surfaces** are automatically assigned to the new **Level** for easy selection.*

11 Click on the ***light bulb*** on the ***Block with holes*** level to re-display the main model.

Thread Milling



Thread Milling is included as one of the options in the **Drilling** dialog.

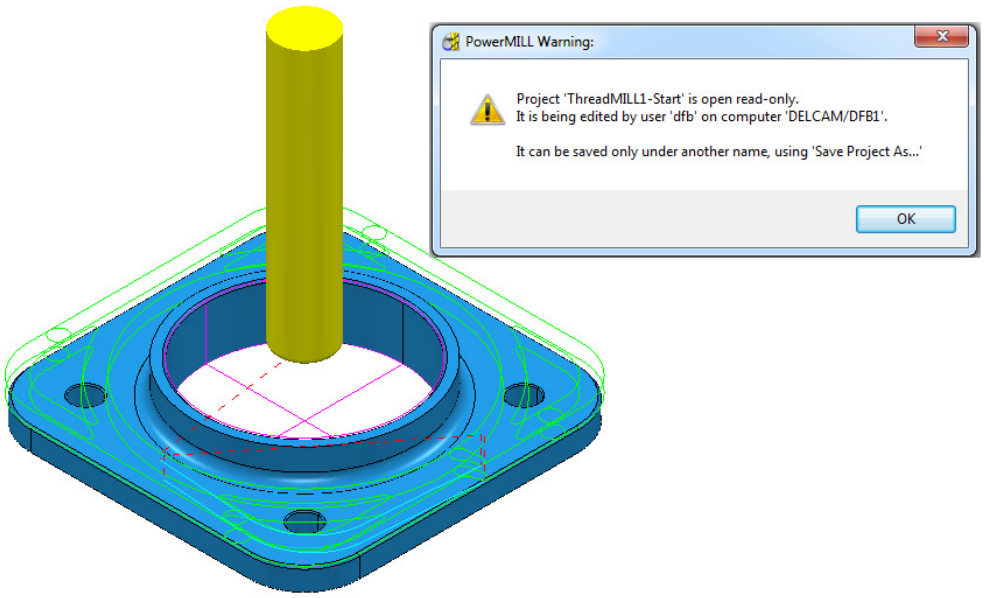
If the **Cycle Type** - **Thread Milling** option is selected, the tool plunges centrally to a specified depth, leads on to the sidewall, and then pitches upwards before leading off.

Within the options it is possible to apply a number of passes (stepping outwards by an allowance value) to avoid tool overload. It is also possible to choose either a Right Hand or Left Hand thread.

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

2 **Open** the **Project**:

.....\PowerMILL_Data\Projects\ThreadMILL1-Start



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

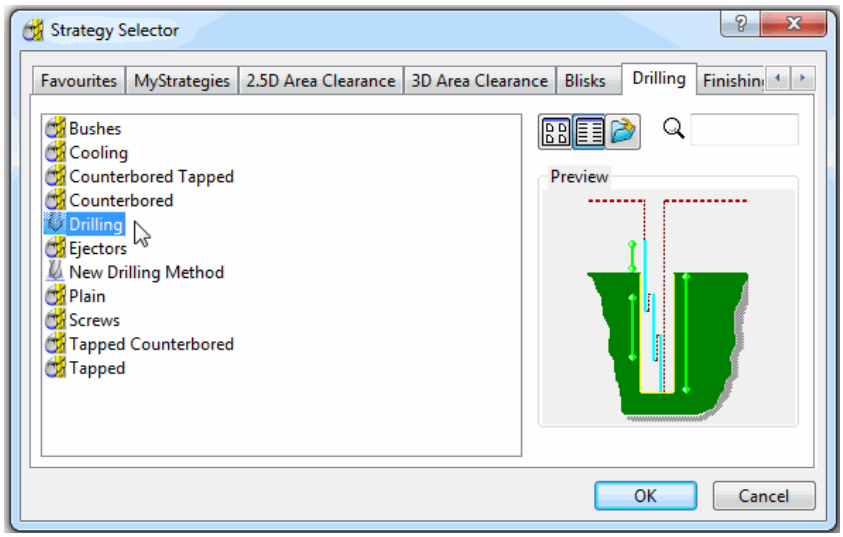
.....\COURSEWORK\PowerMILL-Projects\ThreadMILL-Example1

4 Activate the ***Thread Milling*** Tool named **M40-Coarse-Pitch-5**.

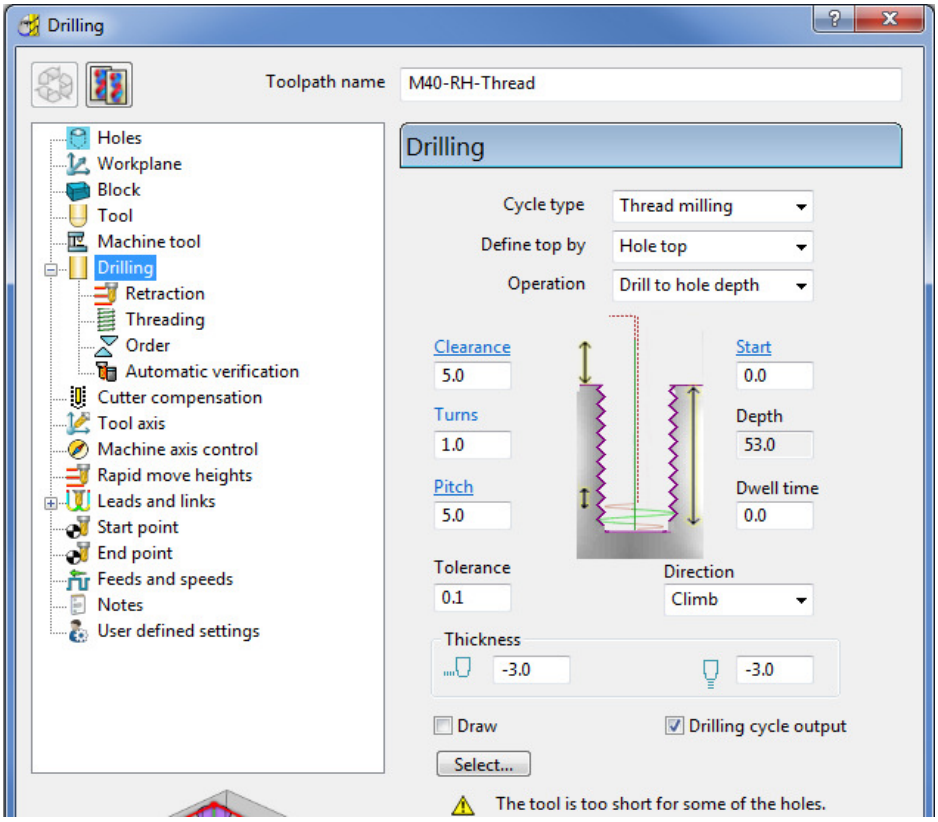
 Copyright © Delcam plc

5 Activate the **Pocket Feature** around the centre wall of the component.

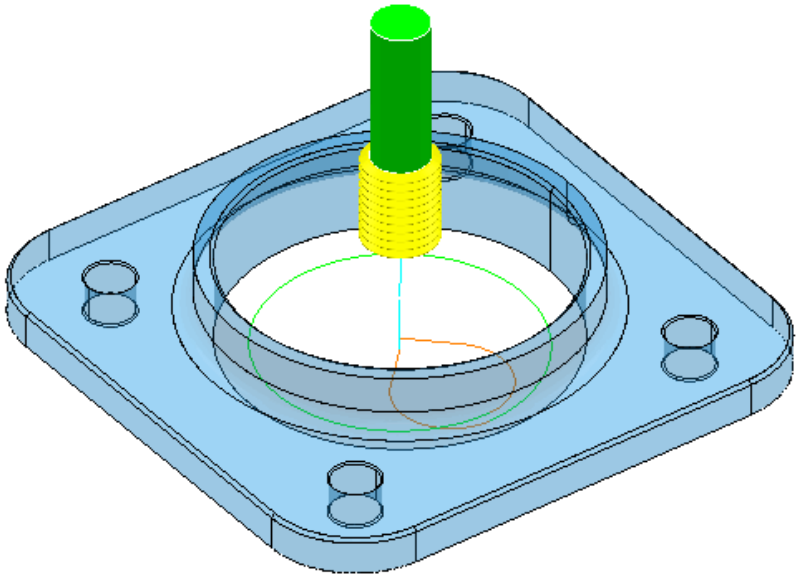
6 Select **Toolpath Strategies** and in the **Strategy Selector** dialog, select **Drilling**.



7 In The **Drilling** dialog, 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).



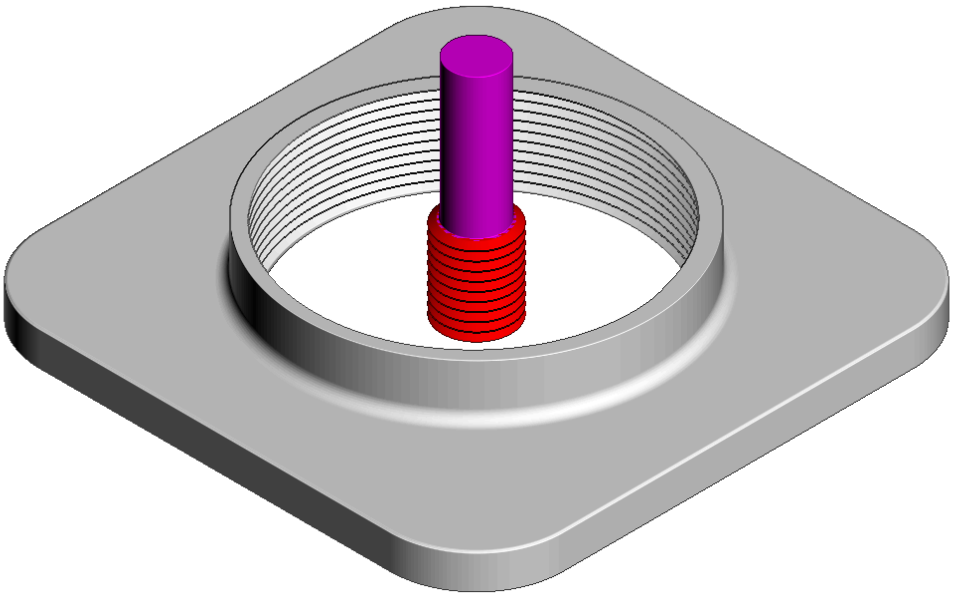
8 **Calculate** the form and then **Close** the dialog.



9 **Calculate** a new **Block** using **Defined by** - **Box** and **Type** - **Model**.

10 Right-click the **NC Program** ***MC-All*** (Do not activate it) and in the local menu select **Simulate from Start**.

11 Open a new **ViewMILL** session and run a simulation of all toolpaths. (Do not activate the toolpath, otherwise the current **Block** definition will change).



The completed **ViewMILL** simulation is as shown above.

12 Select **File** - **Save Project**.

Copyright © Delcam plc

2D Curve Machining

PowerMILL contains 4 strategies that directly operate on 2D **Pattern** segments. These include:

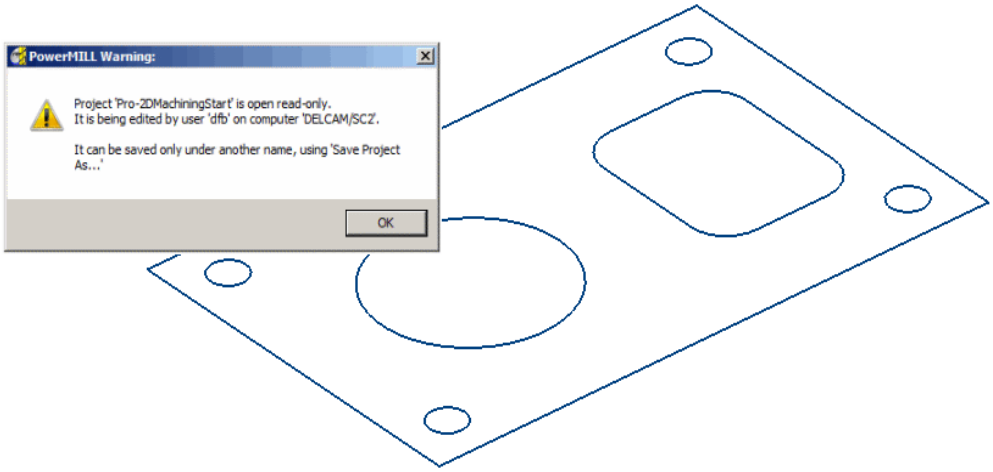
- **Face Milling**.
- **2D Curve Area Clearance**.
- **2D Curve Profile**.
- **Chamfer Milling**.

Example 1

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

2 **Open** the **Project**:

.....\PowerMILL_Data\Projects\Pro-2DMachiningStart



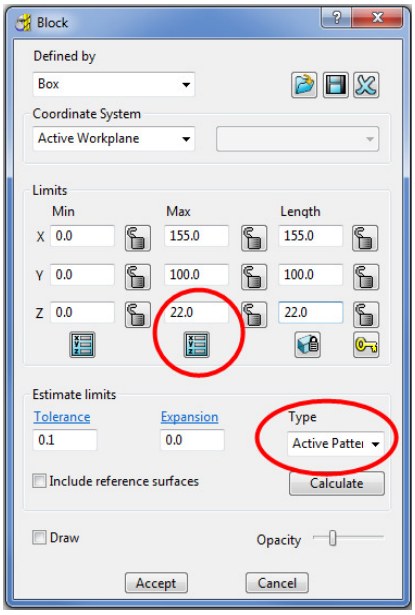
3 Select **File** > **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\PRO-2DMachining

4 **Activate** the **Pattern** named **CurveAll**.

5 **Calculate** a **Block** using **Defined by - Box** and **Type** - **Active Pattern** (Do not close the dialog).

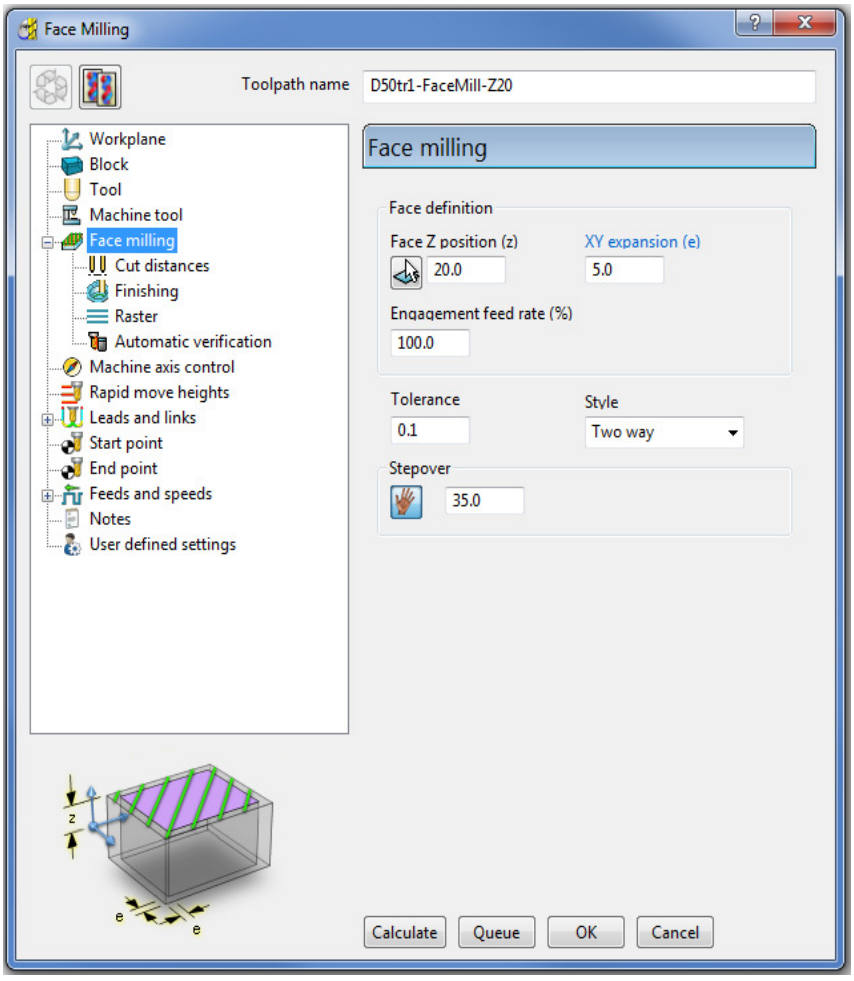
6 In the **Maximum Z** co-ordinate box input **22** before selecting **Accept** to close the dialog.



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

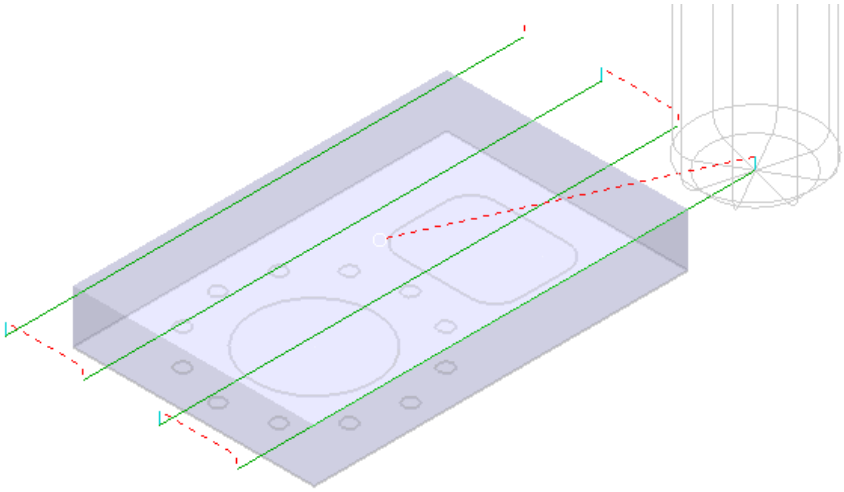
8 **Activate** the Face Milling tool named **D50T6**.

9 From the **2.5D Area Clearance** options select **Face Milling** and enter data into the dialog exactly as shown below.



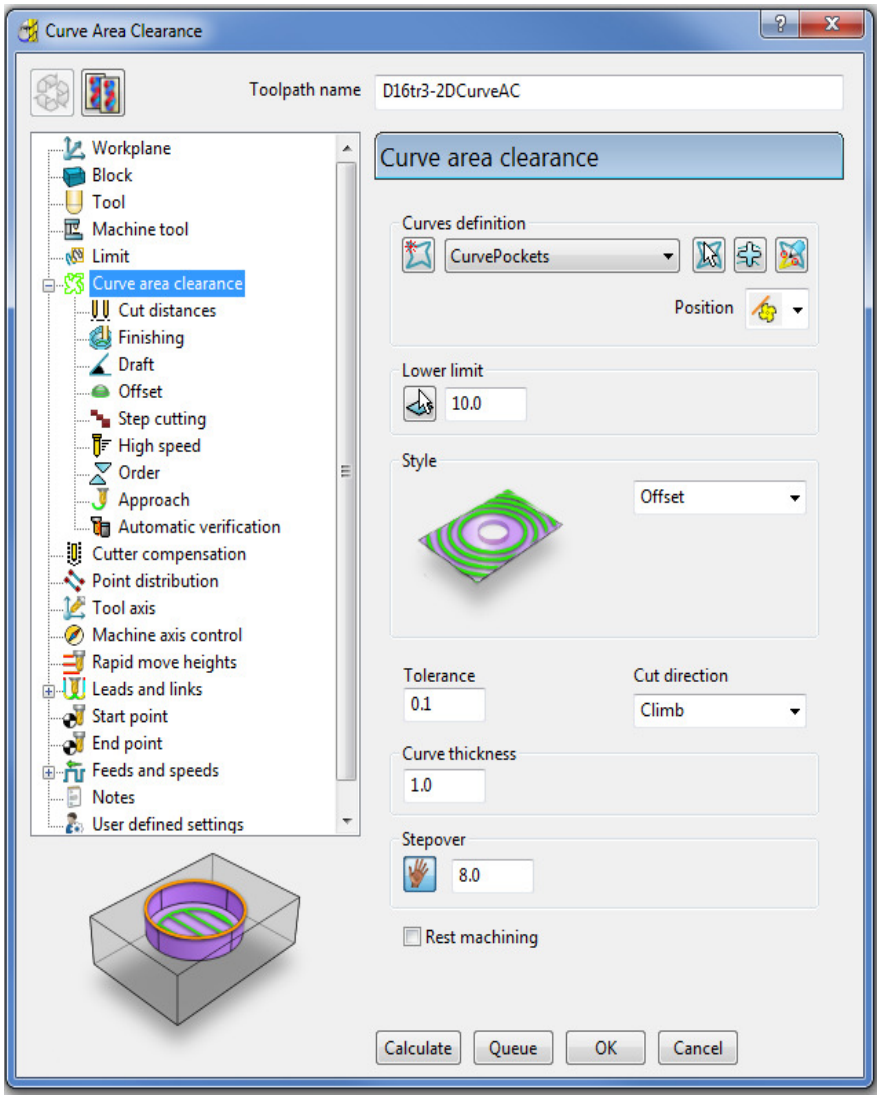
10 Click **Calculate** to create the toolpath and then **Close** the dialog.

Copyright © Delcam plc

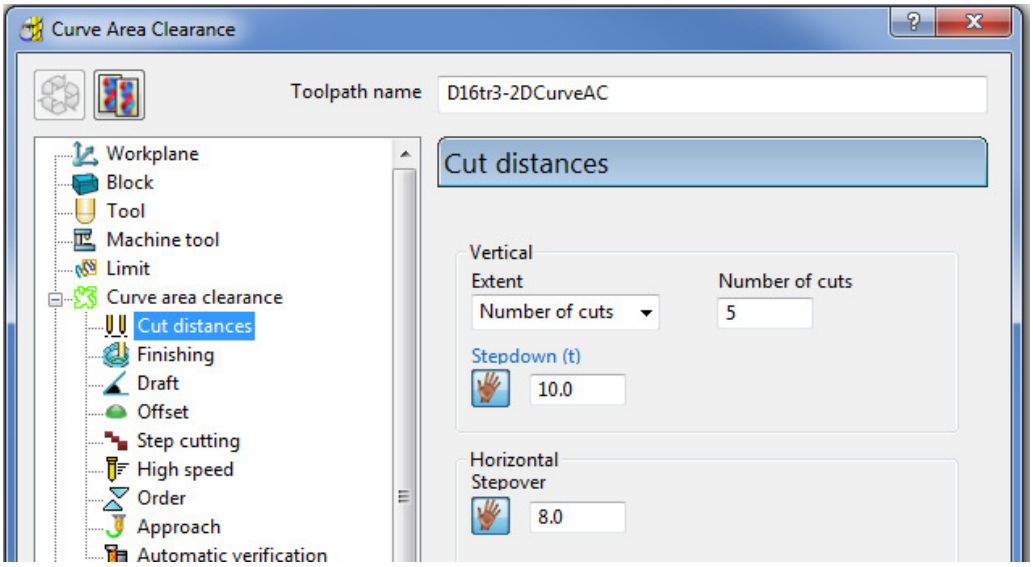


11 **Activate** the Tip Radiused tool named **D16T3**.

12 In the **2.5D Area Clearance** options select **2D Curve Area Clearance** and enter data into the form exactly as shown below.

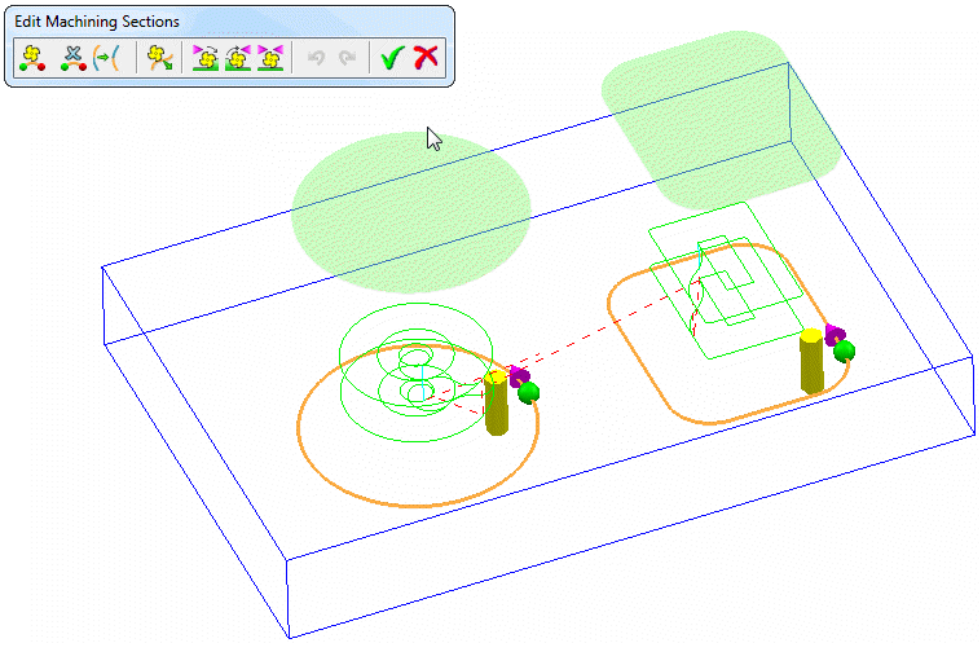


13 Select the **Cut distances** page and input **Stepdown** 10.



14 From the main **Curve area clearance** page, ***Curves definition***, select ***Interactively modify machinable sections*** .

These options enable the user to have full control of direction of cut and the areas to be machined.



*While the **Curve Area Clearance** form is open, the areas to be machined are displayed as a shaded preview.*

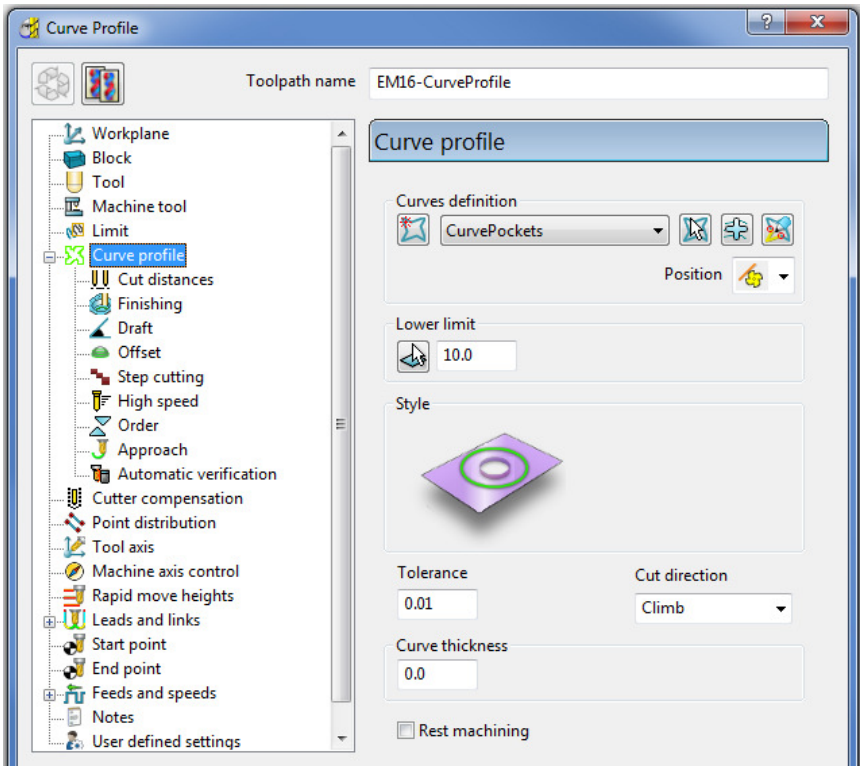
15 Select to exit the **Edit Machininable sections** toolbar.

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

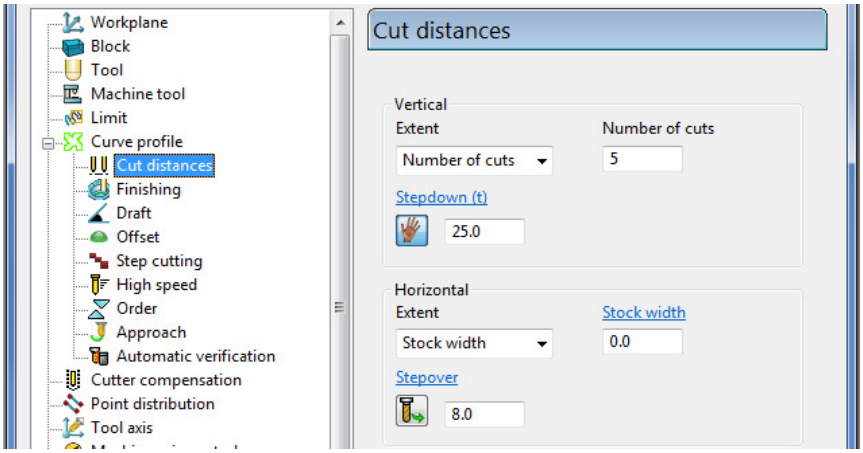
17 Activate the End Mill named **EM16**.

18 From the **2.5D Area Clearance** options select **2D Curve Profile** and enter data into the dialog exactly as shown below.

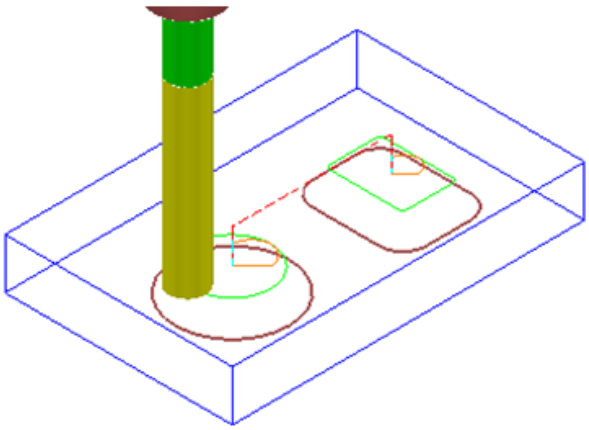
Copyright © Delcam plc



19 Select the **Cut distances** page and input **Stepdown** 25.

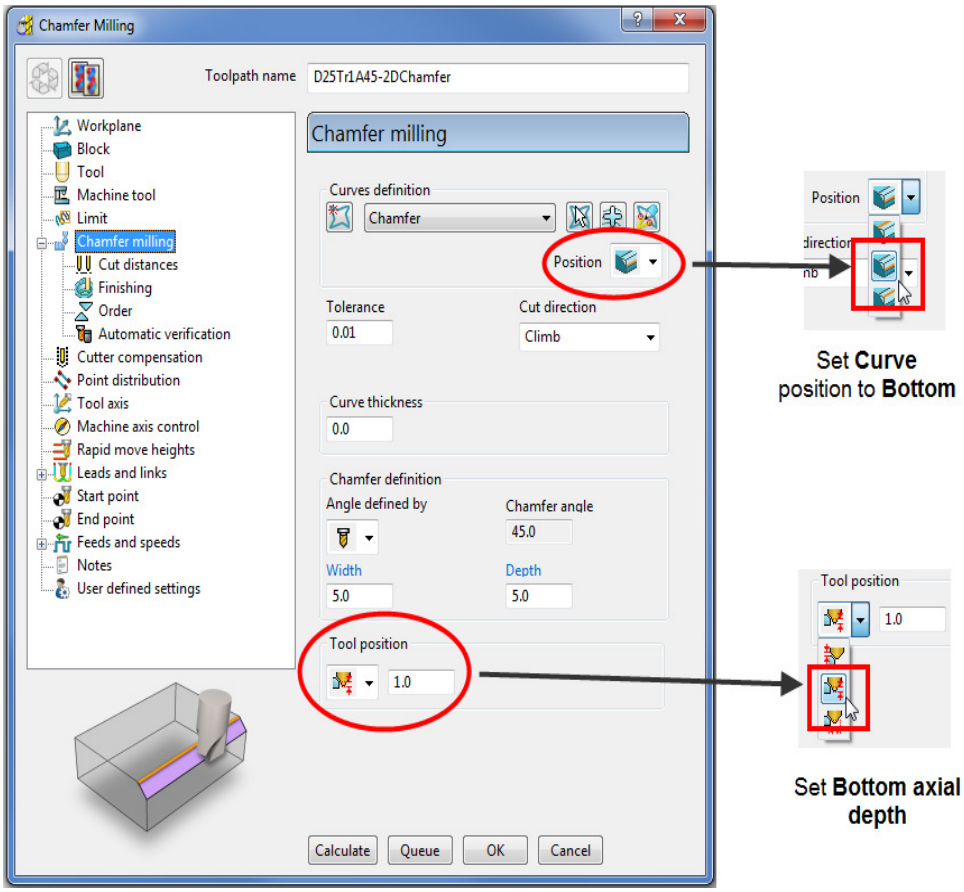


20 Click **Calculate** to process the toolpath and then **Close** the dialog.

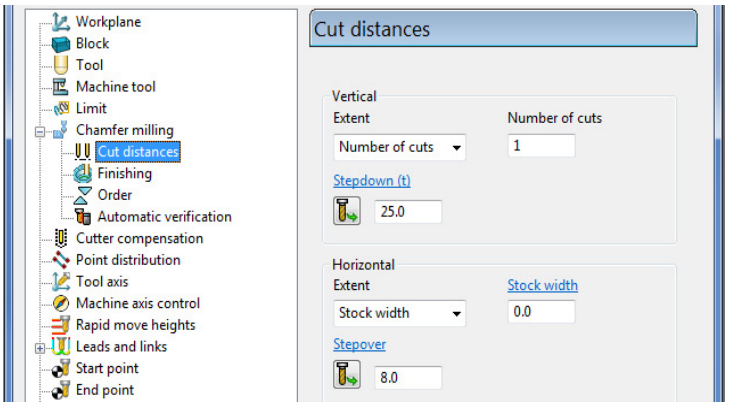


21 **Activate** the Chamfer Tool named **D25TrA45**.

22 From the **2.5D Area Clearance** options select **Chamfer milling** and enter data into the dialog exactly as shown below.

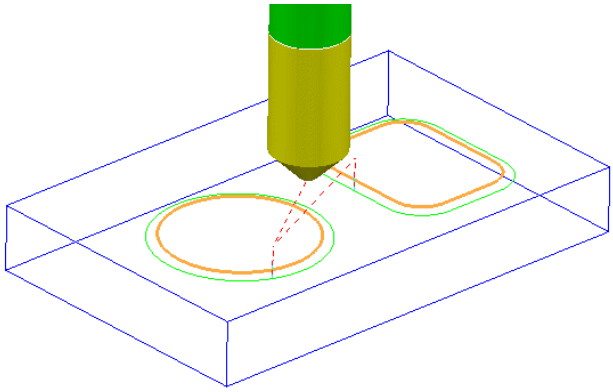


23 Select the **Cut distances** page and input **Number of cuts** as 1.



24 Click **Calculate** to process the toolpath (Do not close the dialog).

Copyright © Delcam plc



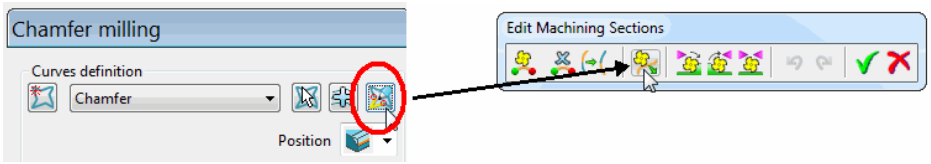
The resultant **Chamfer milling** strategy has appeared (incorrectly) on the outside of the **Pattern** segments.

The toolpath needs to be recycled with the Chamfer Milling tool tracks changed to appear on the inside of the **Pattern** segments.

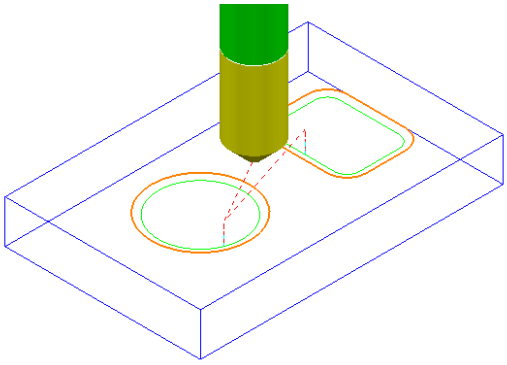
25 In the **Chamfer milling** dialog, select the Recycle icon to enable changes to be made to the above strategy.

26 Select **Interactively modify machinable sections** to access the Edit Machinable Sections toolbar.

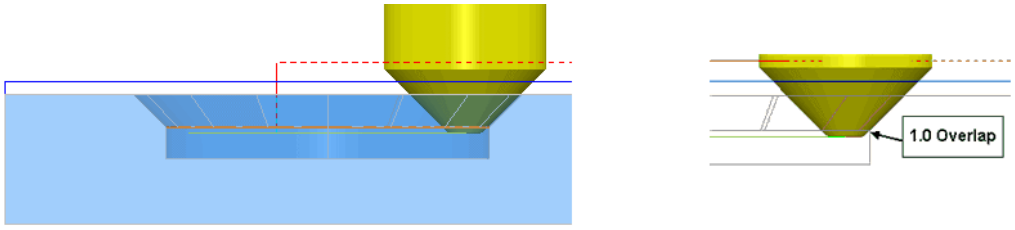
27 Select the **Reverse machining side** option, followed by to accept the changes and close the toolbar.



28 In the main **Chamfer milling** dialog, select Calculate to create the **toolpath** and then **Close** the dialog.

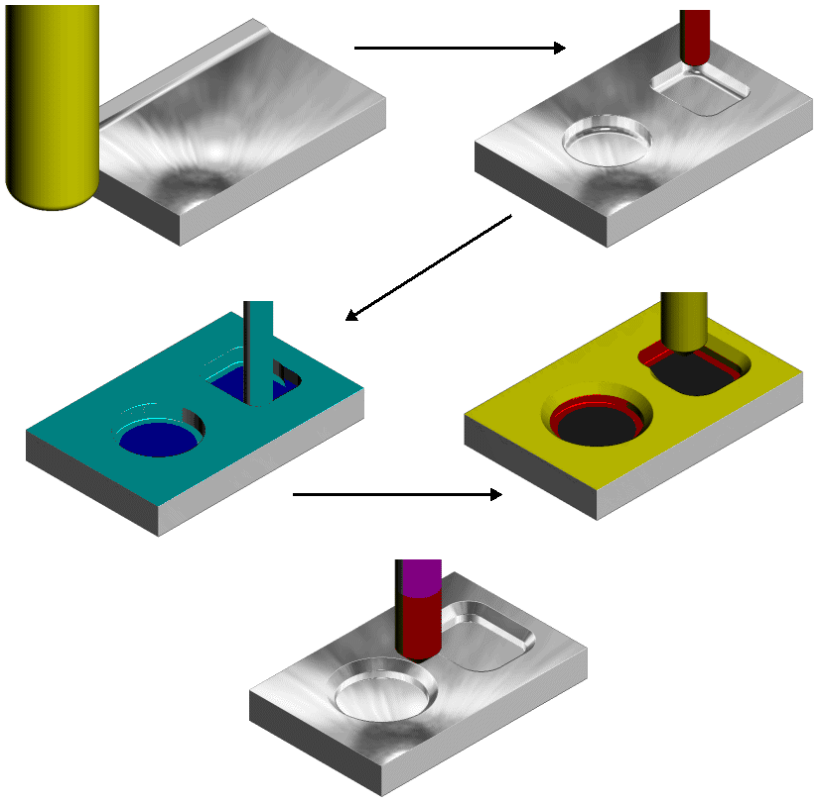


The **Chamfer milling** options selected result in the angled part of the tool overlapping the base of the chamfer by 1mm.



A 3D **model** of the finished component has been imported to provide a visual check of the **Chamfer machining**.

29 Run a **ViewMILL** simulation of the whole machining process.



Cutter Compensation Example

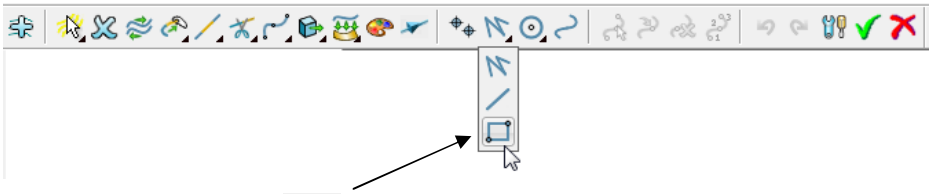
Where applicable a **Cutter Compensation** page is available in a **2.5D Machining Strategy** dialog ***explorer*** options.

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

31 In the **PowerMILL *explorer*** right click on **Patterns** and from the local menu select **Create Pattern**.

32 Rename the **Pattern** as **Profile1**.

33 Right click on the new **Pattern** and from the local menu select **Curve Editor**.



34 Select **Rectangle** from the lines options.

35 In the data input box (below the graphics area) input **0** to set the 1st corner of the rectangle, followed by **100 50** to define the X Y dimensions.

 Copyright © Delcam plc

36 Select **Rectangle** from the lines options again to deselect it.

37 Select the ***green tick*** to close the **Curve Editor** and accept the rectangular **Pattern**.

38 In the **PowerMILL *explorer*** right click on **Workplanes** and from the local menu select **Create Workplane**.

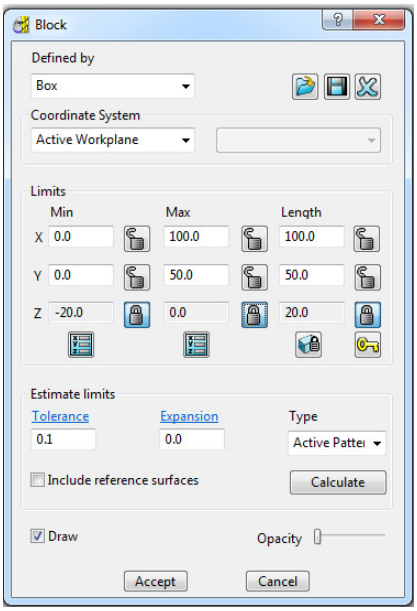
39 In the **Workplane Editor** toolbar, select the ***green tick*** save the new **Workplane** the default **0** coordinate position.

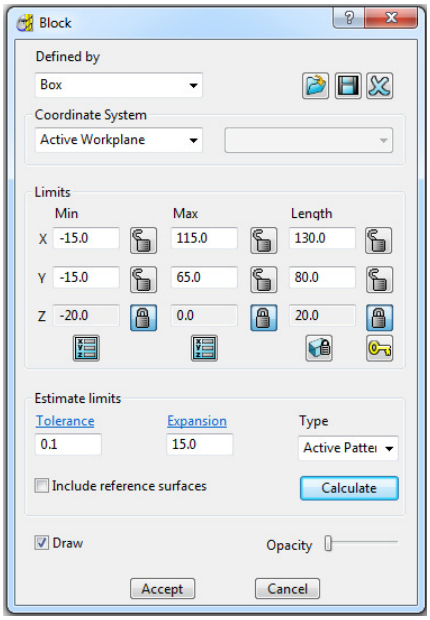
40 **Activate** the new **Workplane**.

41 Create a Dia 10 **End Mill** named **EM10**.

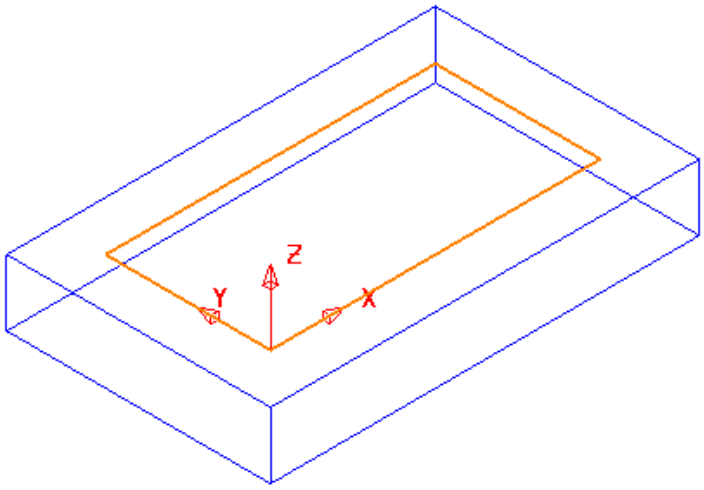
42 **Calculate** a **Block** using **Defined by** – **Box** and **Type** – **Active Pattern**.

43 Input the value ***Min Z*** as **-20** and then lock both **Min Z** and **Max Z** values.





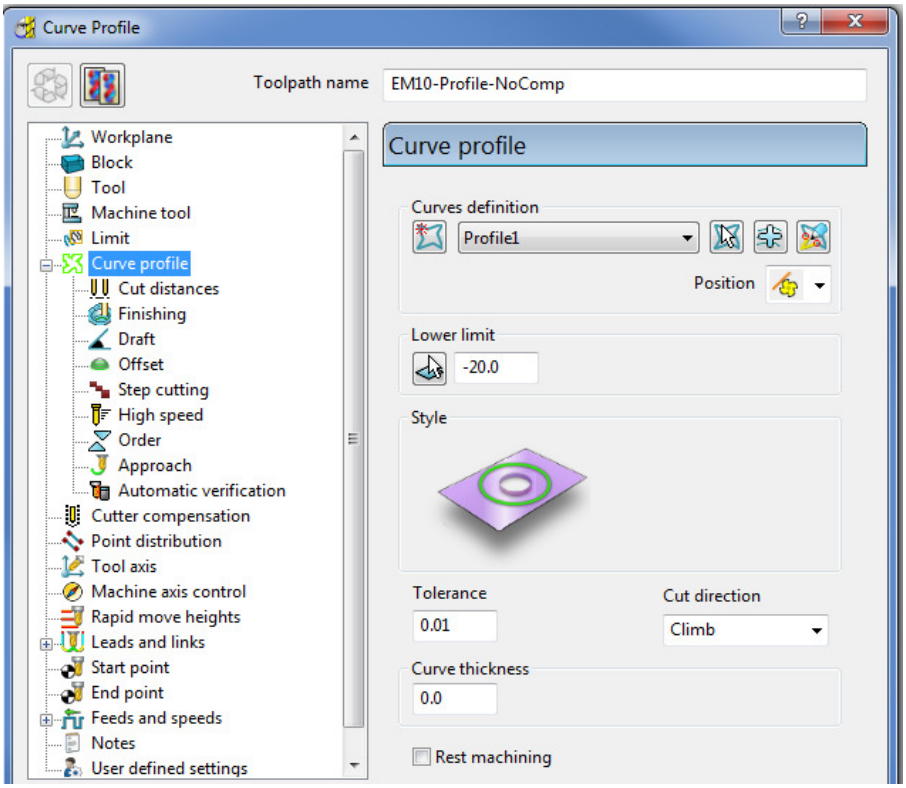
44 Input **Expansion** **15** and then select **Calculate** to add stock to the X and Y dimensions of the **Block**.



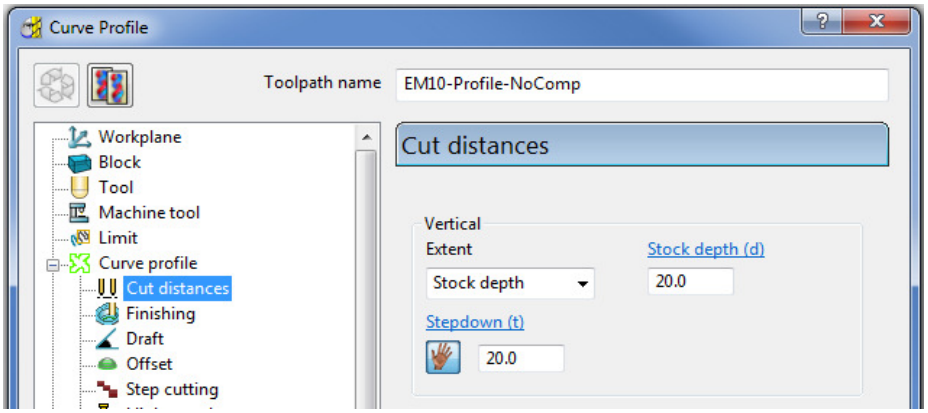
45 In the **Rapid Move Heights** dialog select **Calculate**.

46 In the **Strategy Selector** form select the **2.5D Area Clearance** options.

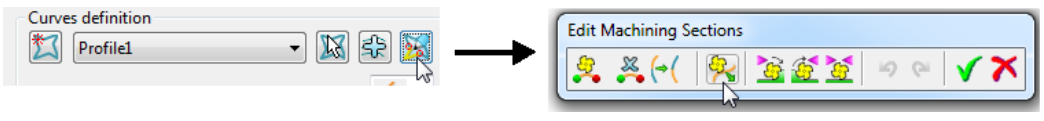
47 Then select the **2D Curve Profile** strategy and enter data as shown in the following illustration.



48 Select the **Cut Distances** page and input **20** for both the **Stock depth** and the **Stepdown**.

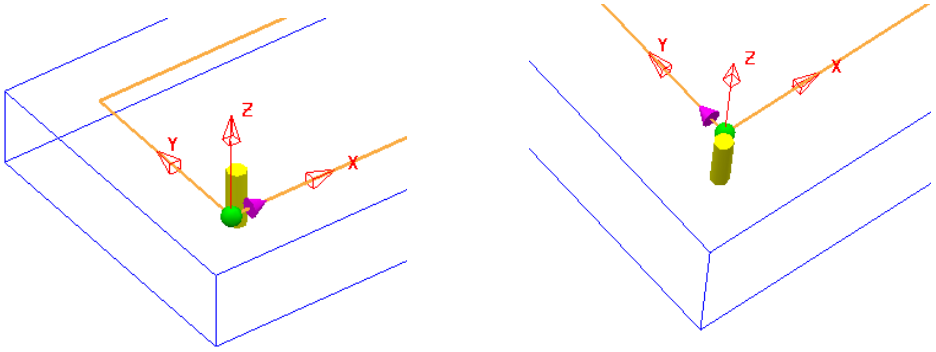


49 Return to the **Curve Profile** page and select the ***Edit Machining Sections options.***



50 If required, select **Reverse machining side** to move the profiling pass to the outside of the **Pattern**.

 Copyright © Delcam plc

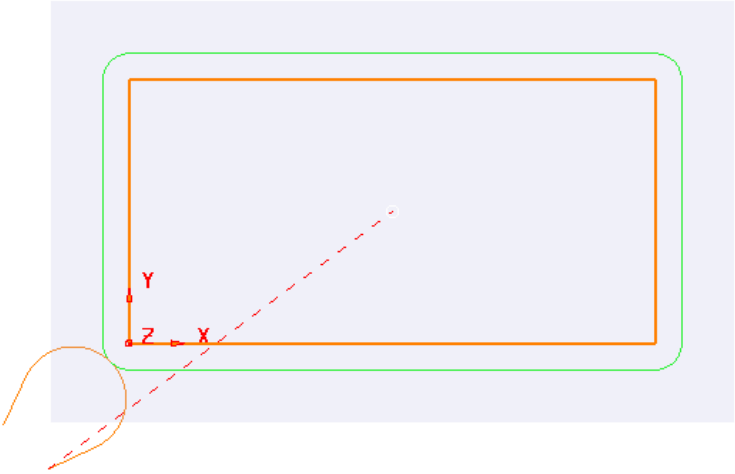


The above preview shows the cutter being switched from the inside to the outside of the **Pattern** prior to calculating the toolpath.

This toolpath will not contain any data that relates to Tool Compensation output.

51 Select the ***green tick*** to accept the changes and exit **Edit Machining Sections**.

52 **Calculate** the **Curve Profile** toolpath.



53 Open the **Leads and Links** form and select **Lead in** – **Horizontal Arc** – **Angle** 110 and **Rad** 10.

54 Copy the **Lead In** settings to **Lead Out**.

55 Select **Extensions** and input **Extended move** 10 for both **Inward** and **Outward**.

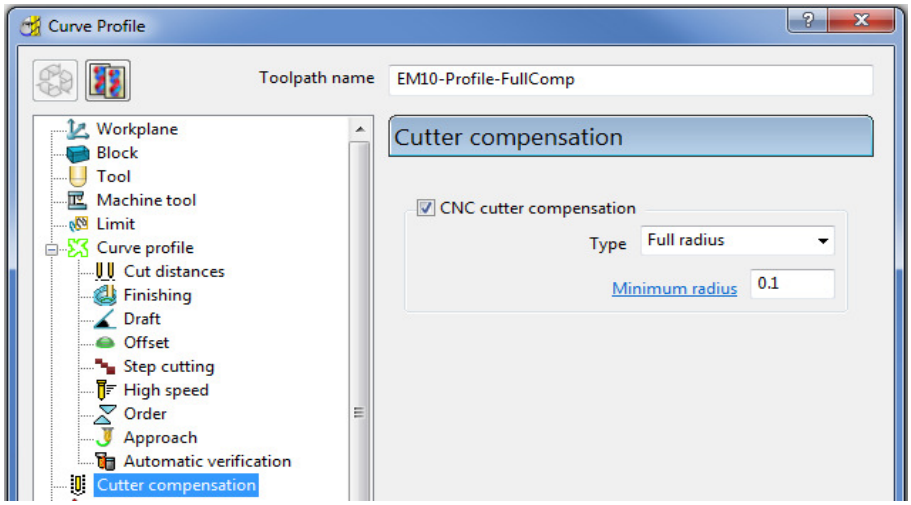
56 **Apply** the **Leads and Links** settings to the active toolpath.

57 Right click on the active toolpath (**EM10-Profile-NoComp**) and from the local menu select **Settings** to open the **Curve Profile** dialog.

58 Once the **Curve Profile** dialog is open select the ***Make a Copy*** option.

59 **Rename** the new toolpath as **EM10-Profile-FullComp**.

60 Select the **Cutter Compensation** page, tick the **CNC cutter compensation** box, and select **Type** as **Full radius**.

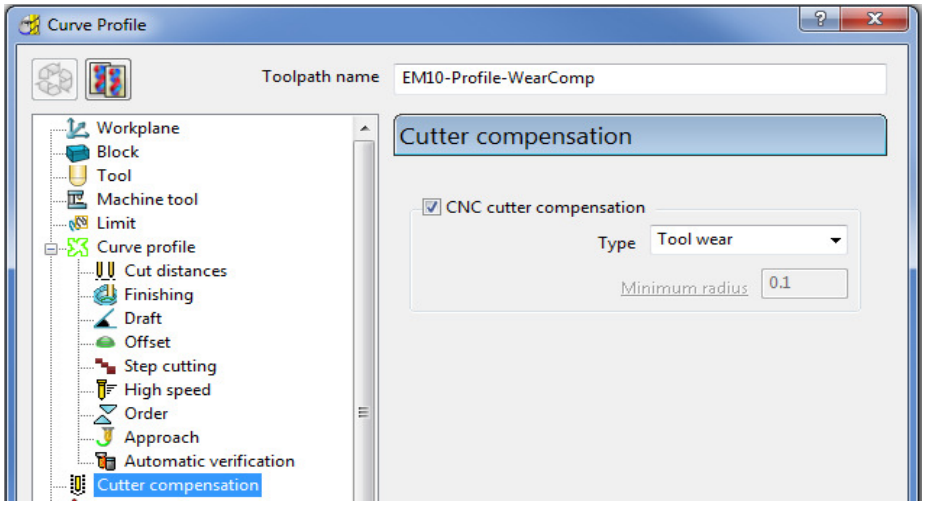


61 **Calculate** the new **Curve Profile** toolpath, but do not close the form.

62 Select the **Make a Copy** option again.

63 **Rename** the new toolpath as **EM10-Profile-WearComp**.

64 Select the **Cutter Compensation** page, tick the **CNC cutter compensation** box, and select **Type** as **Tool wear**.



65 **Calculate** the new **Curve Profile** toolpath, and then **Close** the dialog.

The cutting moves for EM10-Profile-FullComp using the Full Radius compensation are output dimensionally on the actual 2D Curve.

*The cutting moves for **EM10-Profile-WearComp** and **EM10-Profile-NoComp** are dimensionally output to include the tool radius offset from the original curve.*

*For the 2 toolpaths with compensation applied, the **heid** NCData output file will include an **RL** code. This allows the operator to input and apply compensation values into the machine tool controller, registry as required.*

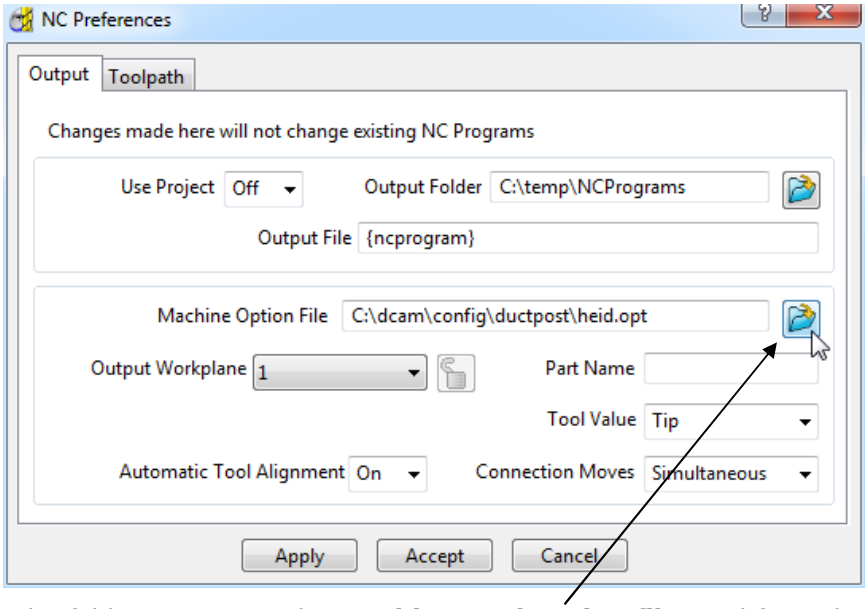
66 **Save Project As**:-

.....\COURSEWORK\PowerMILL-Projects\ToolCompExample

 Copyright © Delcam plc

Viewing the NCData output files for a heid controller

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



68 Open the folder containing the ***Machine Tool Option files*** and from the list select the file **heid.opt** and click the **Open** tab.

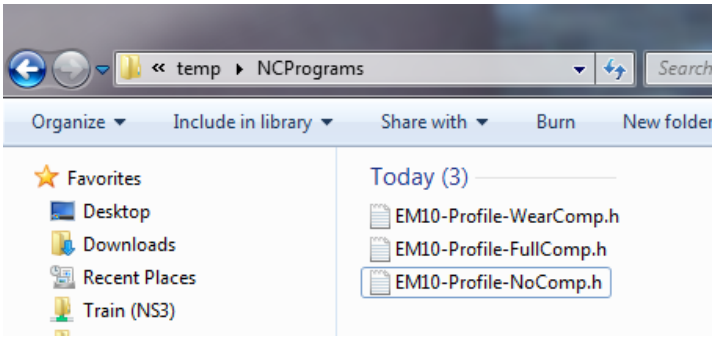
69 **Apply** and **Accept** the **NC Preferences** dialog.

70 In the **PowerMILL *explorer*** right mouse click on **Toolpaths** and from the local menu select **Create Individual NC Programs**.

71 Right mouse click on **NC Programs** and from the local menu select **Write All**.

72 Minimise the **PowerMILL** window to access the ***Desktop*** options and select ***Windows explorer***.

73 Once ***Windows explorer*** has opened browse to **C:\temp\NCPrograms**.



74 In **Windows *explorer***, right mouse click on an ***NCdata*** output file and from the local menu select **Open with** – **Wordpad**.

75 Repeat stage 73-74 for all 3 ***NCdata*** output files and compare the output.

Full Compensation

```
0 BEGIN PGM  1 MM
1 TOOL DEF 23 L+0,000 R+0,000
2 TOOL CALL 23 Z S1350,000
3 L Z+10,000 R0 F9999 M
4 L X-15,443 Y-23,896 R0 F9999 M03
5 L Z+5,000 R0 F M
6 L Z-20,000 R0 F40 M90
7 L X-4,267 Y-24,201 RL F400 M
8 CC X-10,607 Y-10,607
9 C X+0,000 Y+0,000 DR+ RL F M
10 L Y+50,000 RL F M
11 L X+100,000 RL F M
12 L Y+0,000 RL F M
13 L X+0,000 RL F M
14 CC X-10,607 Y-10,607
15 C X-24,201 Y-4,267 DR+ RL F M
16 L X-23,896 Y-15,443 R0 F M
17 L Z+10,000 R0 F9999 M
18 STOP M02
19 END PGM  1 MM
```

Wear Compensation

```
0 BEGIN PGM  1 MM
1 TOOL DEF 23 L+0,000 R+0,000
2 TOOL CALL 23 Z S1350,000
3 L Z+10,000 R0 F9999 M
4 L X-15,443 Y-23,896 R0 F9999 M03
5 L Z+5,000 R0 F M
6 L Z-20,000 R0 F40 M90
7 L X-6,380 Y-19,670 RL F400 M
8 CC X-10,607 Y-10,607
9 C X-3,536 Y-3,536 DR+ RL F M
10 CC X+0,000 Y+0,000
11 C X-5,000 Y+0,000 DR- RL F M
12 L Y+50,000 RL F M
13 CC X+0,000 Y+50,000
14 C X+0,000 Y+55,000 DR- RL F M
15 L X+100,000 RL F M
16 CC X+100,000 Y+50,000
17 C X+105,000 Y+50,000 DR- RL F M
18 L Y+0,000 RL F M
19 CC X+100,000 Y+0,000
20 C X+100,000 Y-5,000 DR- RL F M
21 L X+0,000 RL F M
22 CC X+0,000 Y+0,000
23 C X-3,536 Y-3,536 DR- RL F M
24 CC X-10,607 Y-10,607
25 C X-19,670 Y-6,380 DR+ RL F M
26 L X-23,896 Y-15,443 R0 F M
27 L Z+10,000 R0 F9999 M
28 STOP M02
29 END PGM  1 MM
```

*The output file using **Full Compensation** is exactly matching the geometry of the 100 X 50 rectangular Pattern.*

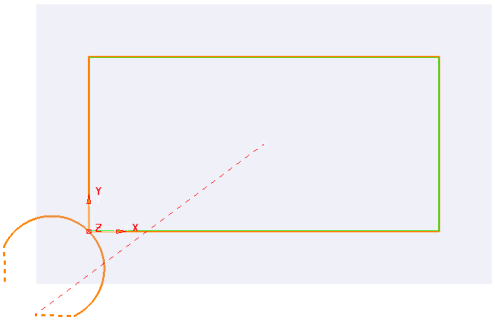
*The output file using **Wear Compensation** is offset by 10mm (the cutter radius) outside the 100 X 50 rectangular Pattern.*

Displaying the compensated toolpath for Full radius

The cutting moves for **EM10-Profile-FullComp** with the **Full Radius** compensation can be displayed in **PowerMILL** as would be output to an ***NCdata*** *output* file.

76 **Activate** the toolpath **EM10-Profile-FullComp**.

77 Select on the **Toolpath** toolbar.



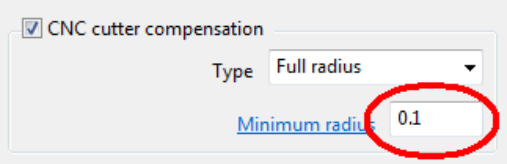
The fully compensated toolpath, **EM10-Profile-FullComp** is visually displayed as shown above.

78 **Save Project As**:-

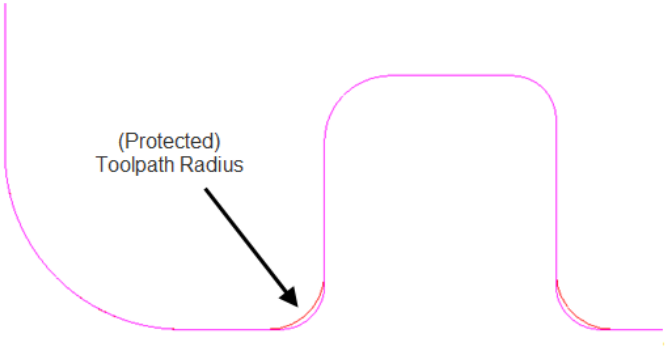
.....\COURSEWORK\PowerMILL-Projects\ToolCompExample

Copyright © Delcam plc

Protected Cutter Compensation



With **Type** – **Full radius** selected any internal (concave) corners will require a suitable Minimum radius value included to allow for corner radii that are less than the required full offset value.

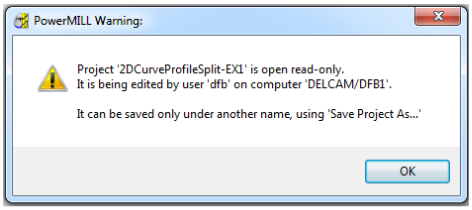


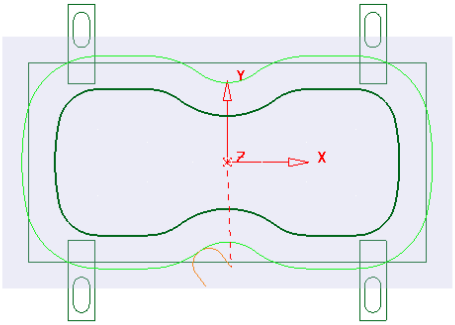
2D Curve Machining Example – Inc. Clamp Avoidance

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

2 **Open Project**:

.....\PowerMILL_Data\Projects\2DCurveProfileSplit-Start

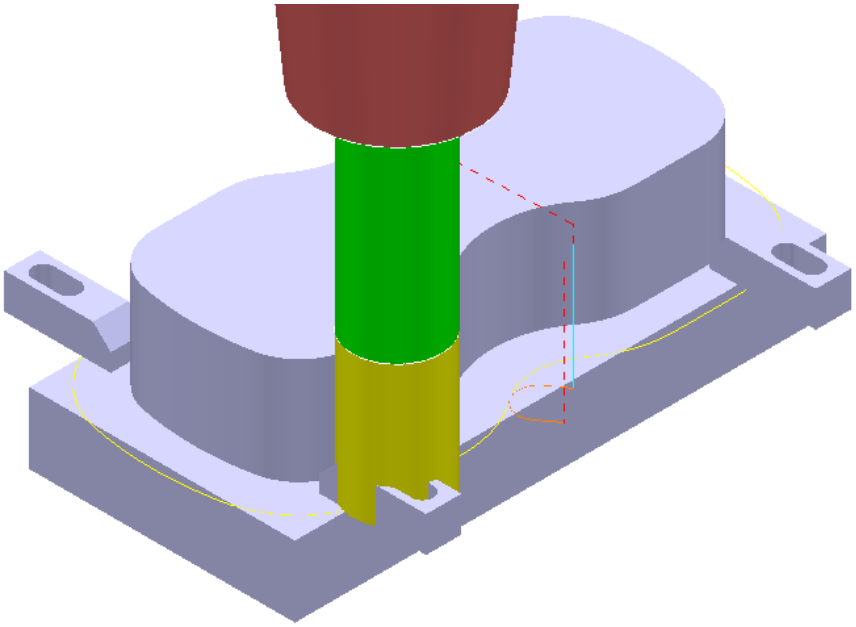




The imported **Project** also contains a Dia25 **End Mill** and a **Pattern** defining the 2D geometry used in the existing 2D Machining Strategy.

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

.....\COURSEWORK\PowerMILL-Projects\2DCurveProfileSplit-EX1



The existing 2D Curve profile toolpath is clearly passing straight through the defined clamp positions. This will be fixed by applying the Split Section option that is accessed from the 2D Curve Profile form.

4 Select a **View down Z**.

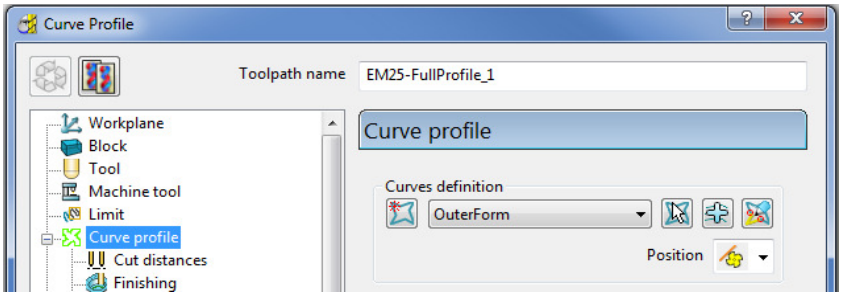
5 Ensure that the **Pattern** - **OuterForm** is displayed (**PowerMILL *explorer***, light bulb on).

 Copyright © Delcam plc

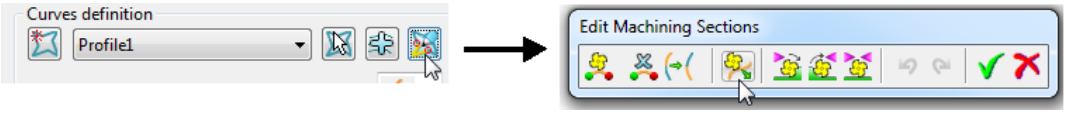
6 Right click on the toolpath **EM25-FullProfile** and from the local menu ensure **Activate** is ticked, and then select **Settings**.

7 Once the **Curve Profile** form is open select the ***Make a Copy*** option.

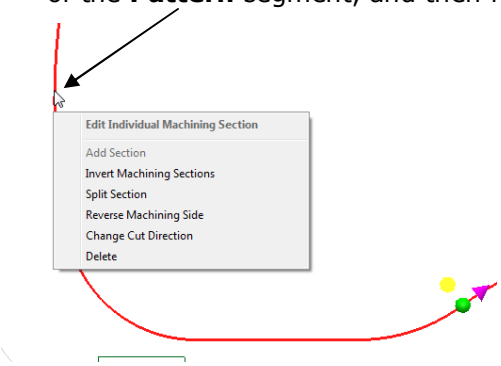
8 **Rename** the new toolpath as **EM25-SplitProfile**.



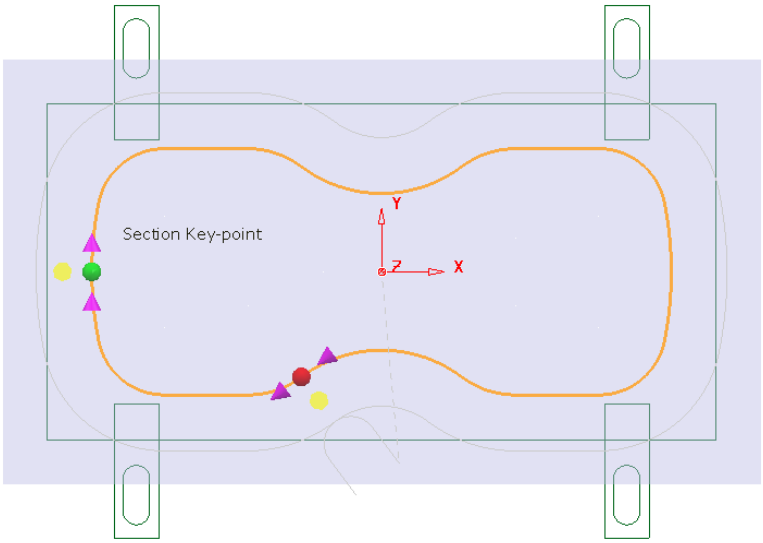
9 On the **Curve Profile** page select **Interactively modify machinable** sections.



10 While the above toolbar is open, move the cursor to a point on the left hand side of the **Pattern** segment, and then right click to open the local menu.



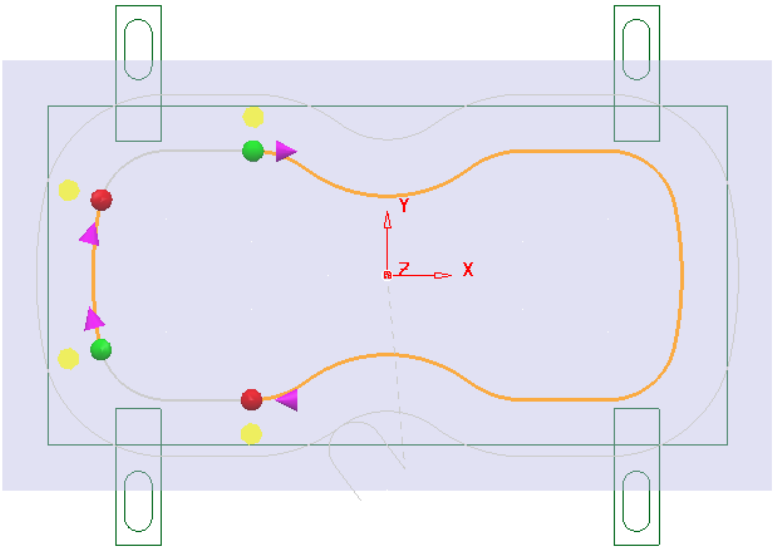
11 From the local menu select **Split Section** and an extra ***start and end*** point will be created on the **Pattern** at this point.



12 From the main pull down menus select **Draw** – **Cursor** – **Cross Hair** (Ctrl H).

The cross hair will be displayed on the cursor to assist with alignment of start and end point.

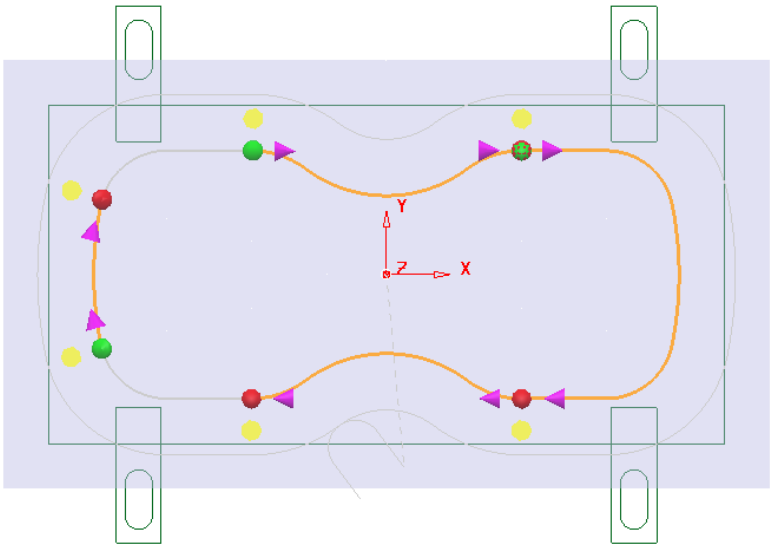
13 Using the left mouse key, drag the new ***green*** - start point along the **Pattern** to a position to the right of the upper left clamp (as shown below).



14 Locate and drag the corresponding ***red*** - end point to a position to the right of the lower left clamp (as shown above).

15 Repeat the process on the other span and drag the start and end points to the left hand side of the **Pattern** (as shown above).

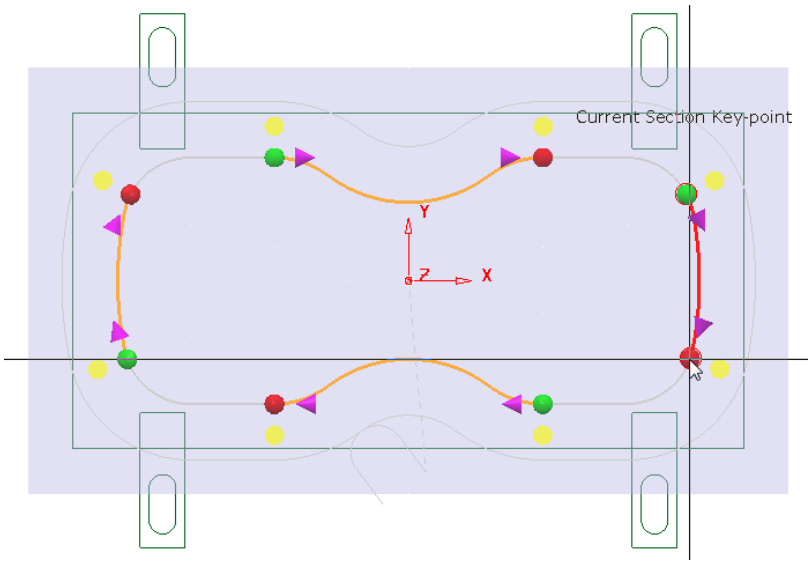
16 Split the toolpath again twice, to the left of both the upper and lower right clamps (as shown below).



17 Move the new start and end points to form 4 distinct spans along the **Pattern** as shown in the following diagram.

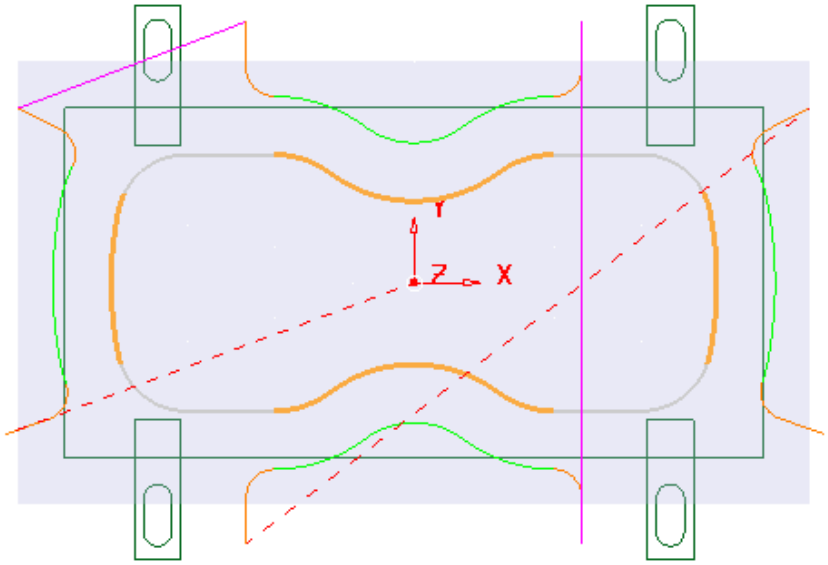
For accurate alignment use the displayed cross hairs and key points.

 Copyright © Delcam plc



18 Once the 4 new spans have been aligned, select the green tick in the **Edit Machining Sections** toolbar to accept the modifications.

19 Back in the **Curve Profile machining** form select **Calculate** to process the new EM25-SplitProfile toolpath and then **Close** the dialog.



To check visually check the toolpath for collisions, a triangle model of the stock along with the clamps will be imported and used as the **ViewMILL** block.

20 Select an **ISO1** view.

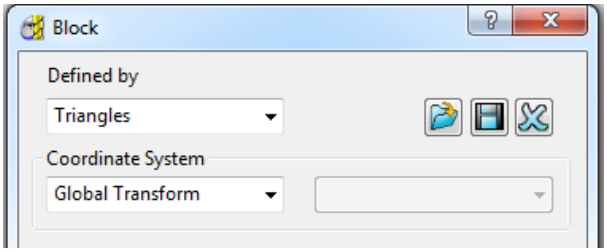
21 Right mouse click on the toolpath **EM25-SplitProfile** in the **PowerMILL *explorer*** and from the local menu select **Simulate from start**.

22 Open the **Block** form and select **Defined by – Triangles** and **Coordinate System - Global Transform** (See diagram on next page).

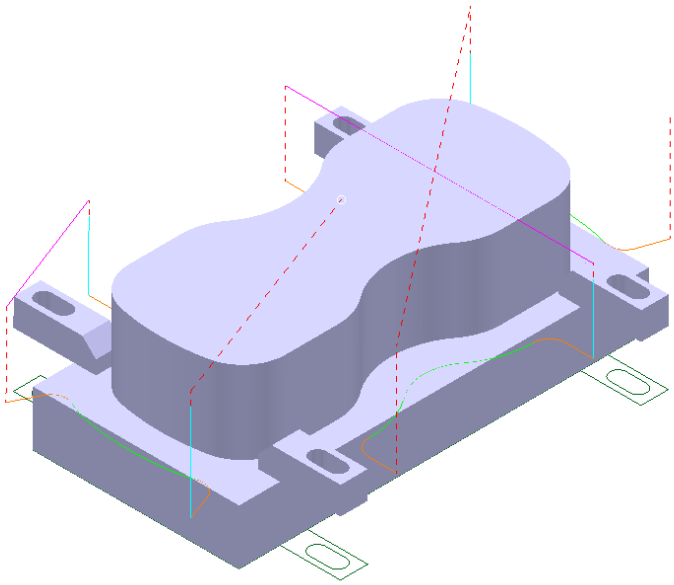
23 Select the **Load block from file** option and browse to the model:-

.....\PowerMILL_Data\Triangles\2DCurveprofileSplit.dmt

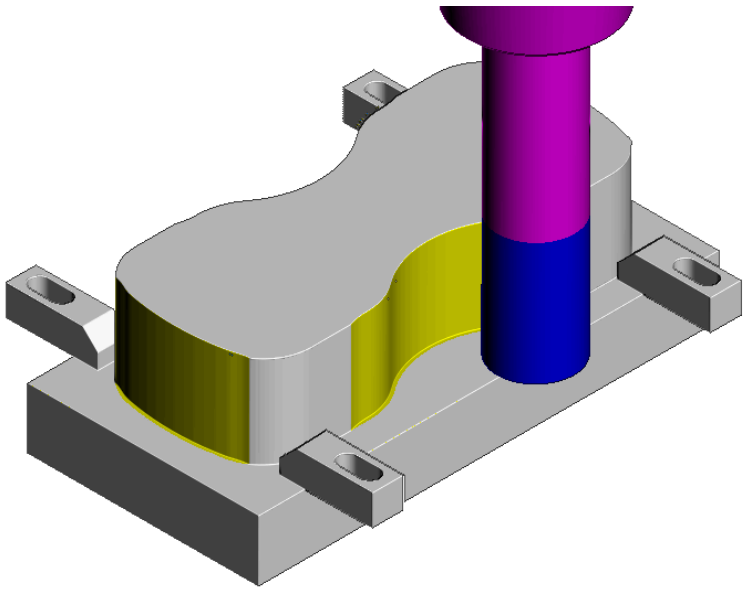
24 Accept the **Block** form.



The **Block** will appear as shown below (Opacity slider set to full))



25 Connect **ViewMILL** to the toolpath simulation and run with the **Rainbow Shaded Image** option set.



The tool is no longer crashing through the clamp positions.

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

Copyright © Delcam plc

13. NC Programs

Introduction

Once a series of **toolpaths** have been generated their names can be listed in an **NC Program** file to be post processed as an output file for a specific **NC machine control**. Any number of **toolpaths** can be included and reordered as required depending on the limitations of the particular **NC machine** along with the associated **post processor**.

If the **NC machine** has a **tool change** facility then **toolpaths** created with **different cutter sizes** can be included in the same ***NC Program***.

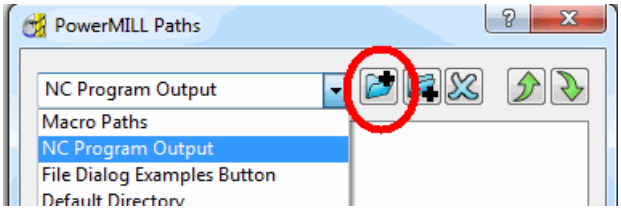
If the **NC machine** does not have a tool change facility then only **toolpaths** created with the same cutter can be included in the same **NC Program**.

1. Select **File > Delete All** and **Tools > Reset Forms**.
2. **Open** the 'read only' **Project**:

   .....\PowerMILL_Data\Projects\NCProgram-Start
3. Select **File > Save Project As**:-

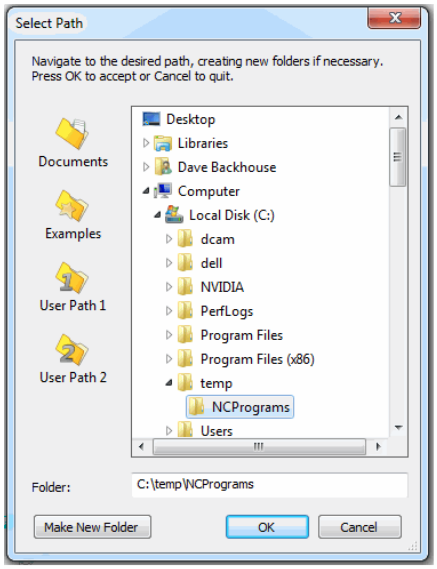
   .....\COURSEWORK\PowerMILL-Projects\NCProgram-ex1
4. From the top menu bar, select **Tools > Customise Paths**.
5. In the **PowerMILL Paths** dialog, select **NC Program Output** followed by **Add path to top of list** .

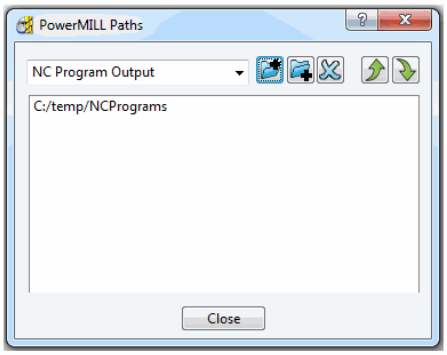
   It is now required to define a path to where the **Post Processed** files (NCdata) are to be stored.



6. In the **PowerMILL Paths** dialog, Select ***NC Program Output*** and add the new path **C:\temp\NCPrograms**.

*The destination folder for **NCPrograms** must exist on your pc otherwise it will not be possible to output the **NC Program** files.*

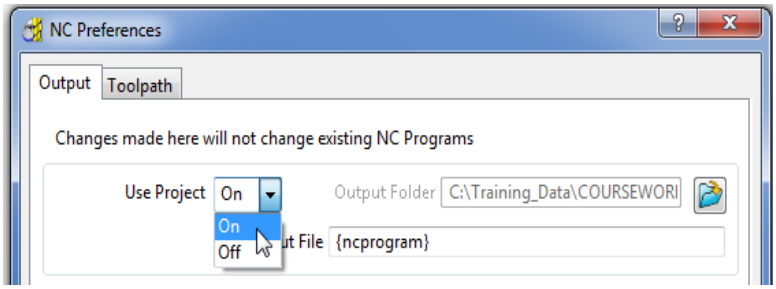




7 Click **OK**.

The Use Project - On option

An alternative method to control the destination for **NCdata** output is to set the ***active* Project** as the default, destination for outputting **NC Data** files. In the **NCPrograms - Preferences** form, switch ***Use Project*** to **ON**.

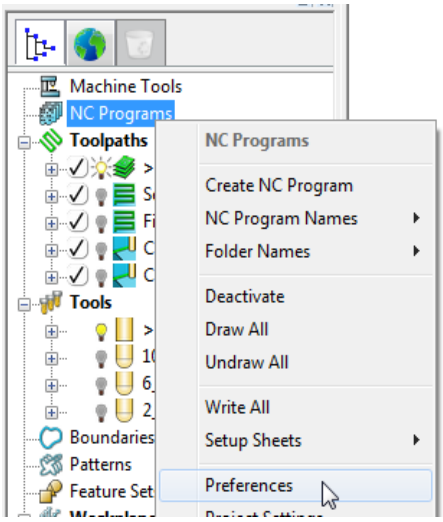


On applying the **Write** command on an ***NC Program*** a folder named **NC Programs** will be created in the individual **Project** folder containing the associated NCdata output file.

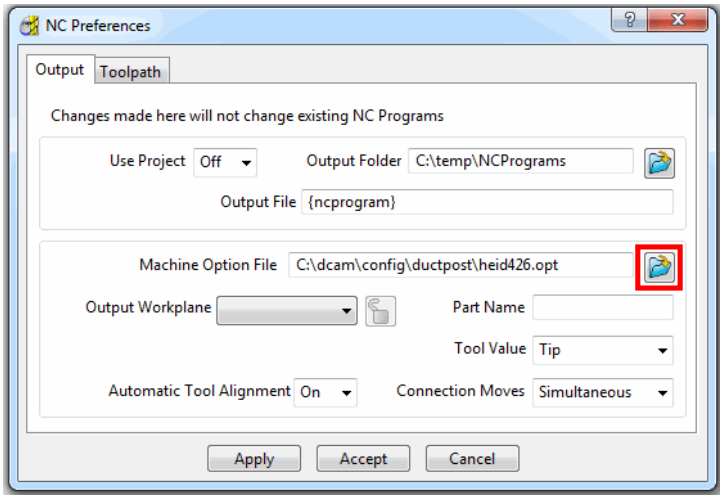
Copyright © Delcam plc

Setting the NC Preferences

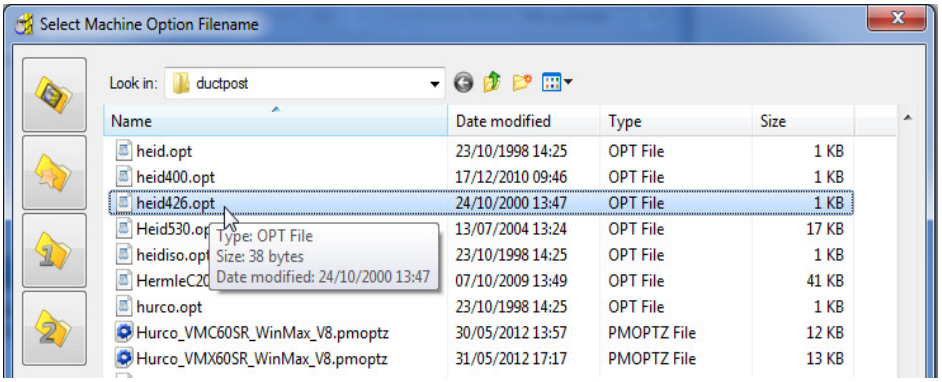
1 In the **PowerMILL *explorer***, right-click **NC Programs** and select **Preferences**.



2 In the **NC Preferences** dialog, select the folder icon to access the machine tool, **Option File** list.



3 From the ***Select Machine Option Filename*** dialog (below), select the **heid426.opt** and click the **Open** tab.



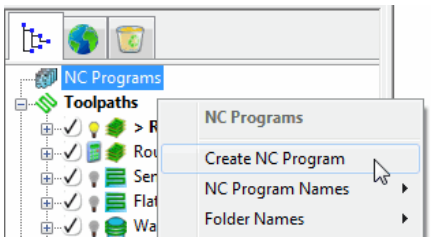
The selected option file determines which **Machine Tool** output language is applied to the ***NC Program***.

*It is not unusual to have more than one **Option File** for the same machine tool to allow for different operational requirements.*

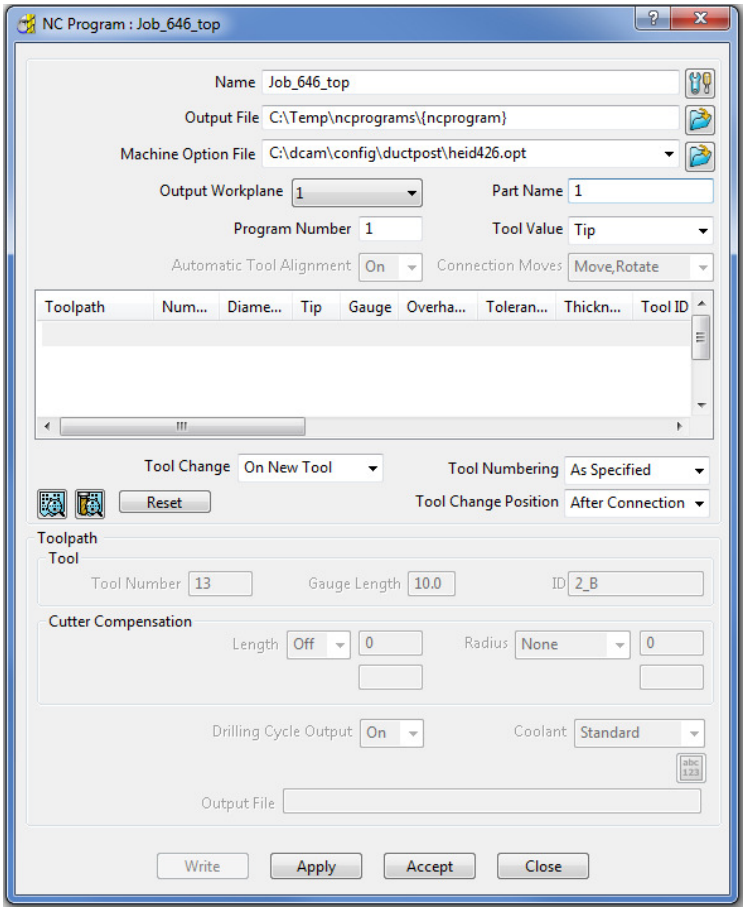
4 Click **Accept** on the **NC Preferences** dialog.

Creating the NC Program

1 In the **PowerMILL *explorer***, right-click on **NC Program** and from the local menu, select **Create NC Program**.

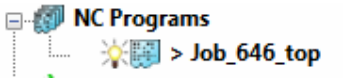


2 Enter the name **Job_646_top** and click **Accept**.



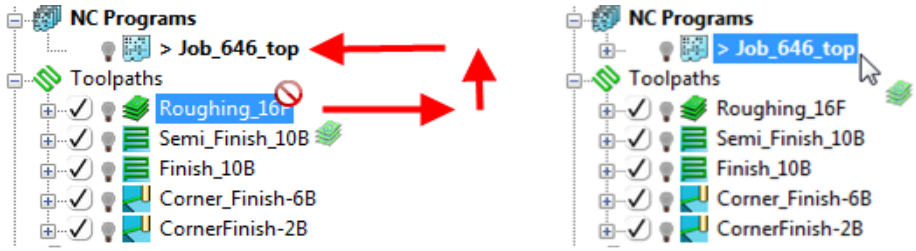
3 In the **PowerMILL *explorer*** select the **+/-** box by **NC Programs** to toggle a display of the contents (**Job_646_top**) on or off.

Copyright © Delcam plc



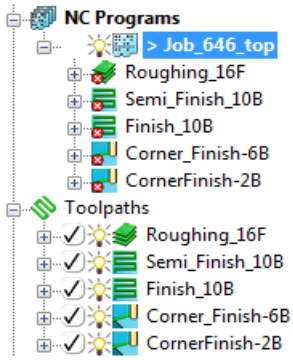
One or more **Toolpaths** can be added to the active, ***NC Program***.

4 Select the **Area Clearance** icon with the left mouse button and keeping it held down drag the mouse to the right (The cursor will display ghosted icons).



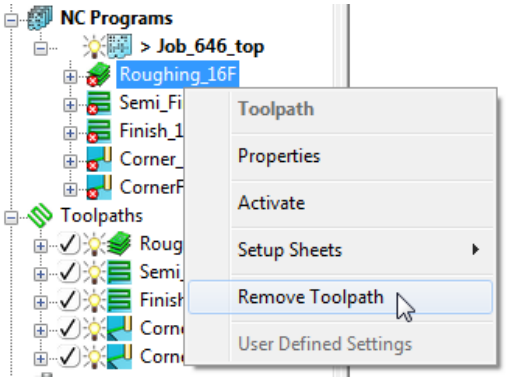
5 Continue to drag the toolpath onto the active **NC Program**, **Job_646_top** and release the mouse button. In the **PowerMILL *explorer***, **NC Programs** area select the +/- box by **Job_646_top** to toggle a display of the contents (**Roughing_16**).

6 Drag the other toolpaths into the last file in the active **NC program** area to maintain the required machining order.

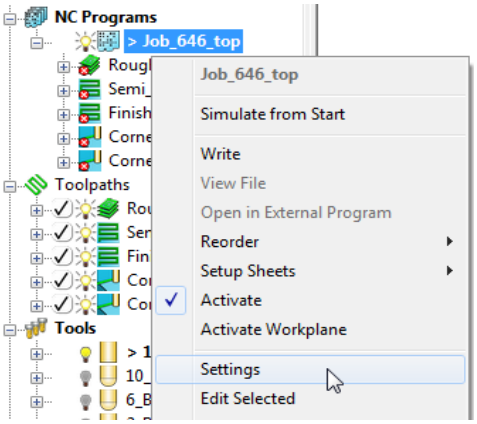


*You can remove individual toolpaths from the **NC Program** list.*

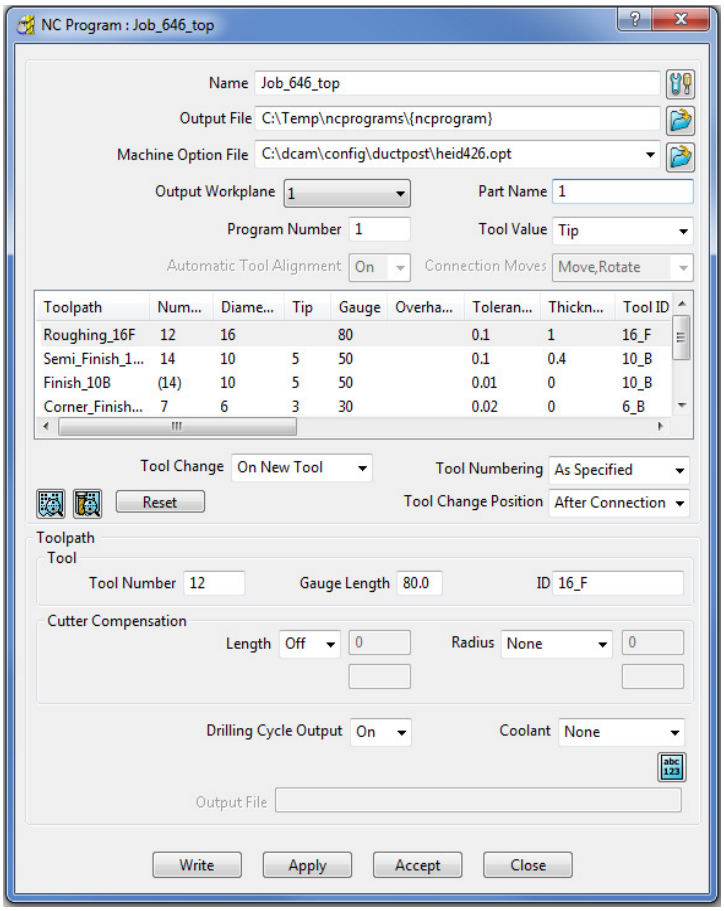
7 Right-click any of the toolpaths in the current **NC Program** to access the local **Toolpath** options, but do not apply any of the options at this stage.



8 Right-click on the **NC Program** (**Job_646_top**) and select **Settings**.



*You can open (and edit) the **NC Program** form current by selecting **Settings** from the local right click menu.*



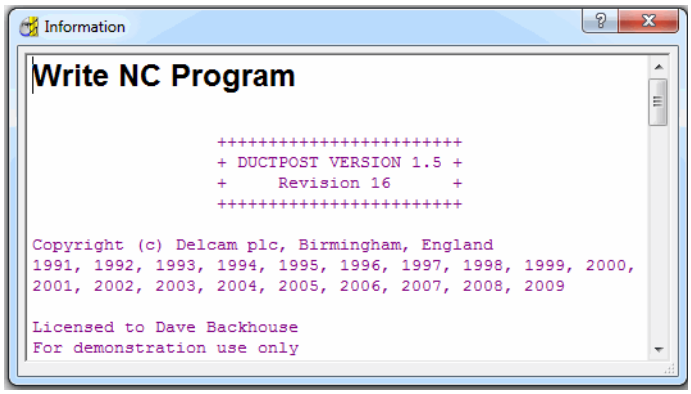
*Several of the settings assigned to a specific toolpath can be changed in the above dialog. The toolpath row is selected and relevant values (Tool Number, Coolant, etc) are then modified before clicking **Apply**. These local eidts do not affect the original toolpath.*

9 Click **Accept**.

Copyright © Delcam plc

10 Right-click on the **NC Program**, **Job_646_top** and select **Write** to create the **Post-Processed** output file.

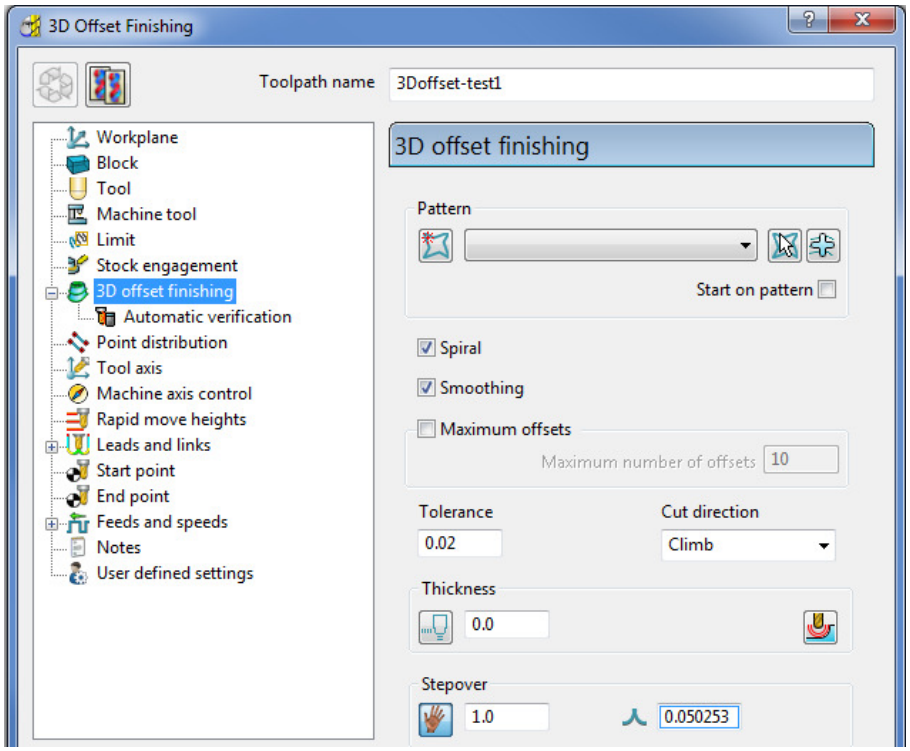
During the translation an **Information** window appears containing a progress report.

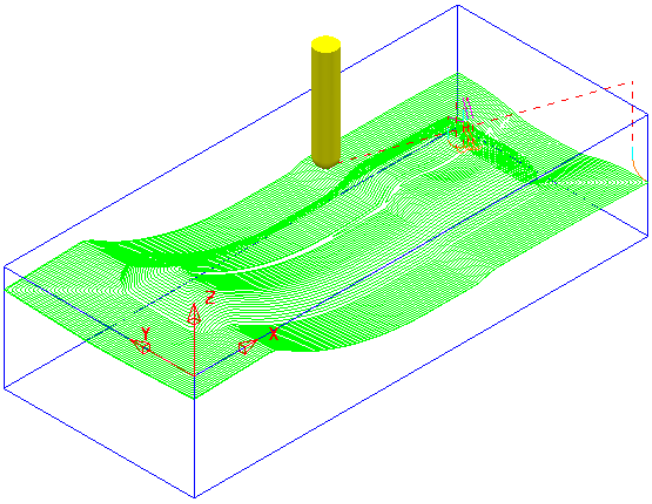


Newly created **Toolpaths** are automatically added to the ***Active* NC program**. It can be useful to **Create** and **Activate** an **NC Program** prior to toolpath creation to save having to drag them in later as a separate operation.

11 De-activate the ***NC Program***, **Job_646_top** in the **PowerMILL *explorer***.

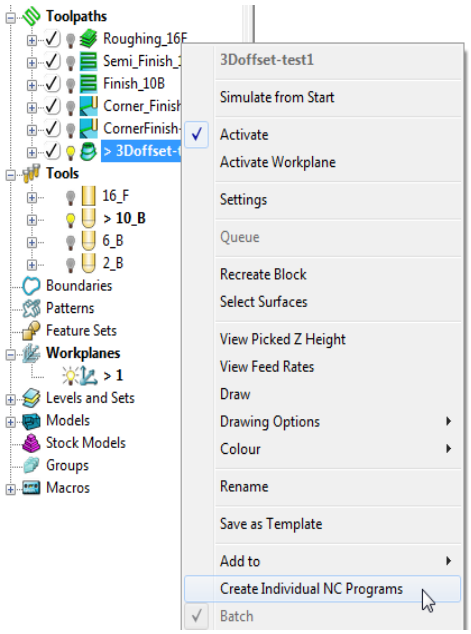
12 **Calculate** a **3D Offset Finishing** toolpath, named **3Doffset-test1** using the settings and values as shown below.



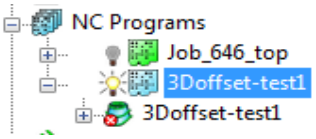


*It is not always required to output all **toolpaths** in one **NC Program**. As a result it is possible to create an **NC Program** directly from an individual **toolpath** entity.*

13 Right-click on the toolpath named **3Doffset-test1** and select **Create Individual NC Programs**.



14 A new ***NC Program*** is created. Select the **+** symbol to the left of ***NC Program*** **1** to access its contents (the **+** symbol will change to a **-** symbol).



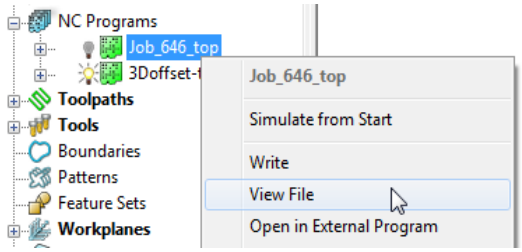
15 Right-click on the ***NC Program***, **3Doffset-test1** and select **Write** to output the post-processed file.

Copyright © Delcam plc

Viewing the NC-Data output

Once Post-processed, the **NC Data** output file can be viewed directly from the local menu.

- Right-click on the ***NC Program***, **Job_646_top** and from the local menu, select **View File**.



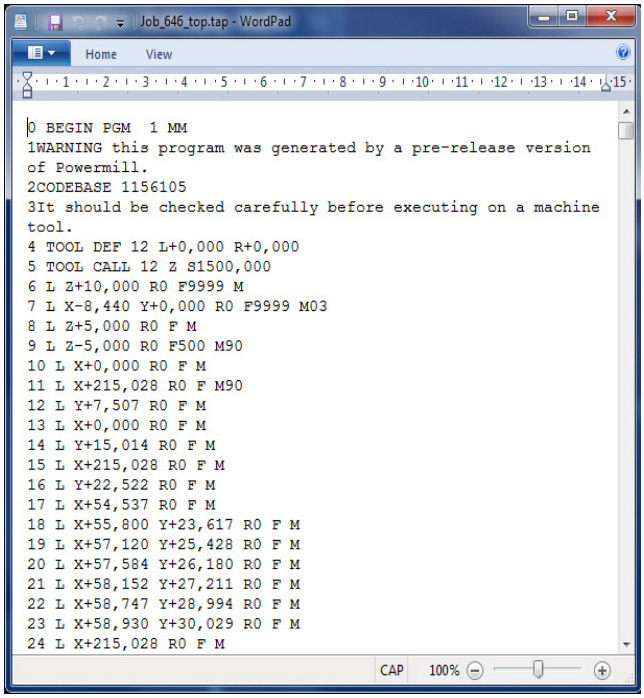
An **Information** dialog will open to enable the user to view (but not edit) the contents of the NC Data output file.

Manually Editing the NC-Data output

If the user needs to edit the code manually, then the NC-Data output file will have to be edited in Wordpad.

1. From **Windows Explorer**, browse to **C:\temp\NCPrograms** to where the **NC-Data** files are downloaded.

2. Right-click on one of the **NC-Data** files and from the local menu, select **Open With > Wordpad**.



3. From the **main** (top) toolbar select **File > Save** to update the **Project** followed by **Close**.

 Copyright © Delcam plc

14. Setup Sheets

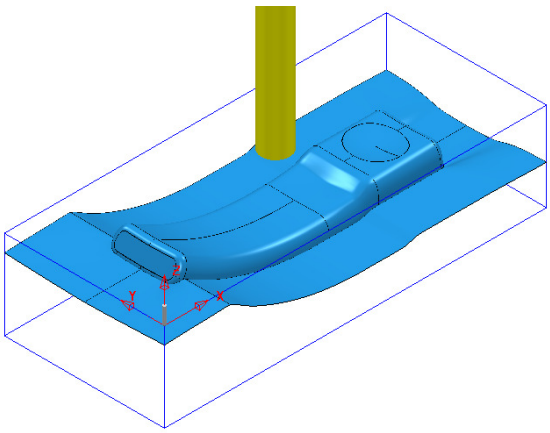
Introduction

All the Setup data required from a ***PowerMILL* Project** such as ***cutter definitions***, ***images*** and ***information*** relating to **toolpaths** can be downloaded as **HTML files** and then printed out as required. This is essential for the CNC Operators who will be running the programs.

1 Select **Delete > All** and **Tools > Reset forms**.

2 **Open** the **Project**:

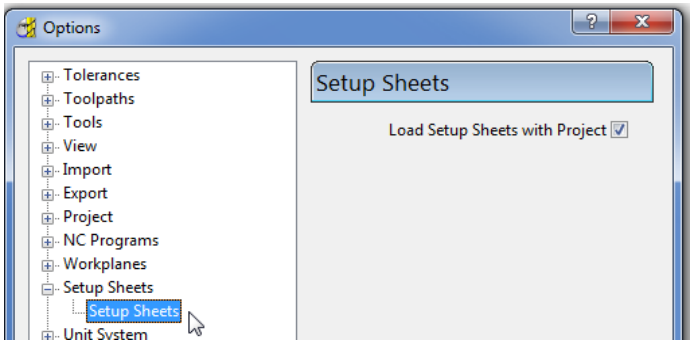
.....\PowerMILL_Data\Projects\SetupSheet-Start



3 Select **Save Project As**:

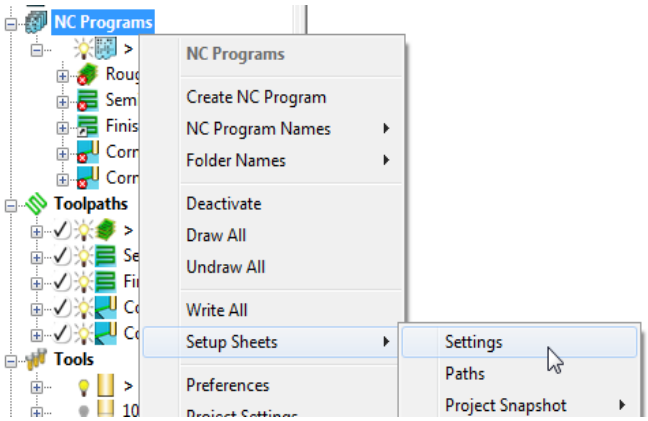
.....\COURSEWORK\PowerMILL_Projects\SetupSheet-ex1

4 From the **main** pulldown menus select **Tools > Options > Setup Sheets > Setup Sheets**, and **tick** the box **Load Setup Sheets with Project**.

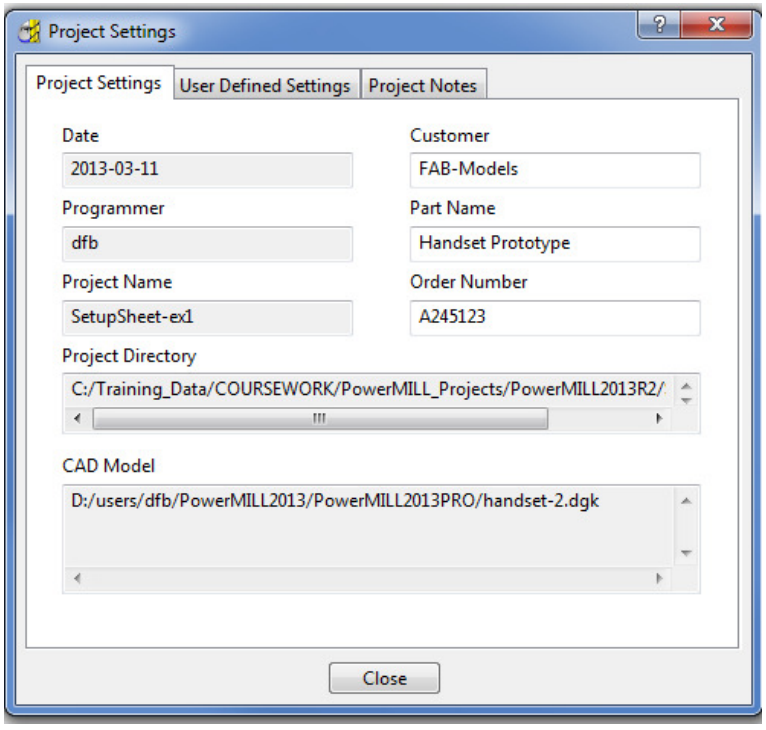


5 **Accept** the form.

6 In the **PowerMILL *explorer*** right-click on **NC Programs** and select **Setup Sheets > Settings...**



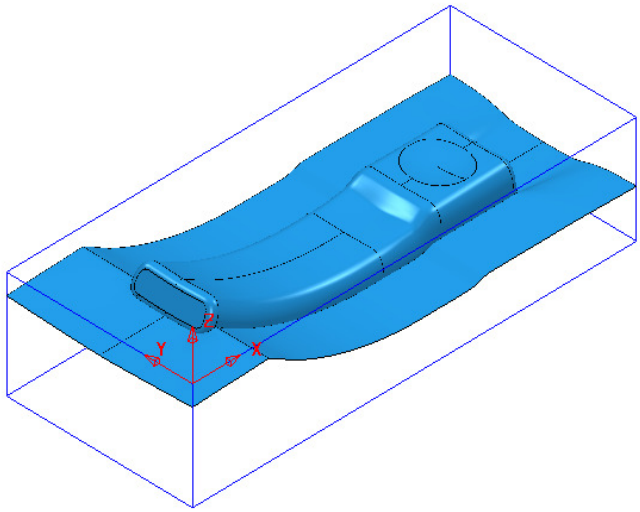
7 In the **Project Settings** dialog, enter the information as shown below and select **Close**.



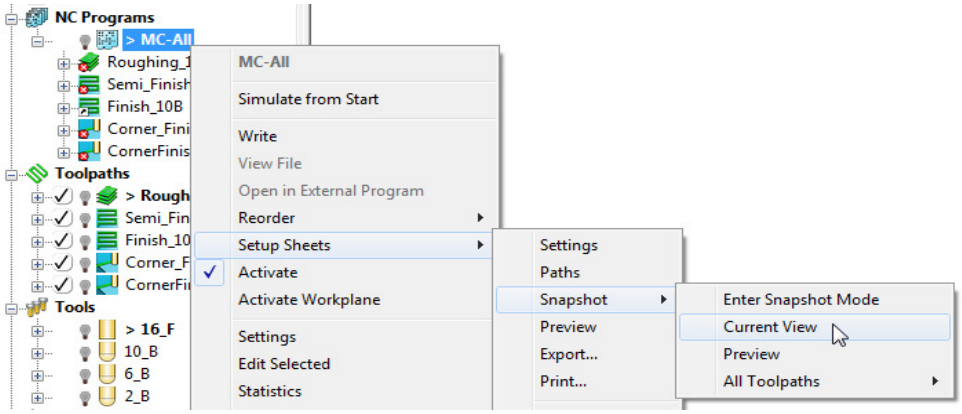
*There are 2 other tabs for the user to input further information as **User Defined Settings** and **Project Notes**.*

Copyright © Delcam plc

8 Ensure that no **tools** or **toolpaths** are visible and only the **shaded model** and ***wireframe* Block** (minimum **Opacity**) are displayed in an **ISO 1** view.



9 Right-click on the **NC Program** named **MC-All** and from the local menu select **Setup Sheets > Snapshot > Current View**.

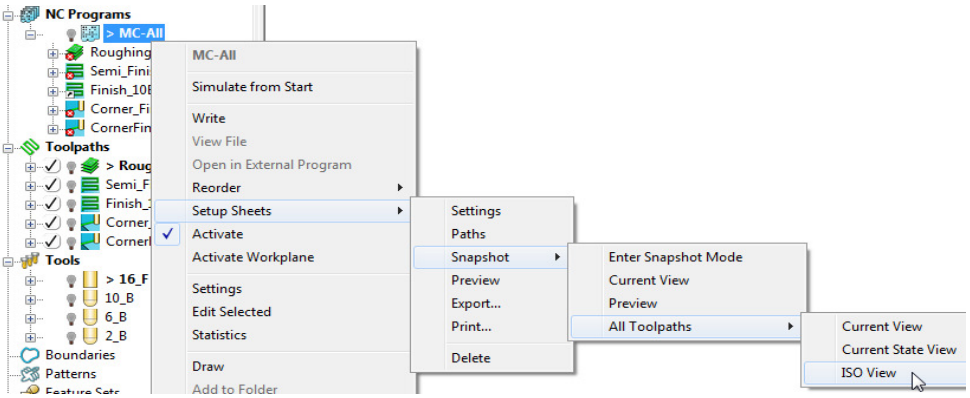


An image of the model has now been captured and will be automatically used on the front page of the setup sheet.

10 Undraw the **model** leaving the ***wireframe* block** as the only displayed item.

11 Ensure that the **NCProgram** - **MC-All** is displayed (light bulb icon on).

12 Right-click on the **NC Program** - **MC-All** again and select **Setup Sheets > Snapshot > All Toolpaths > Iso View**.



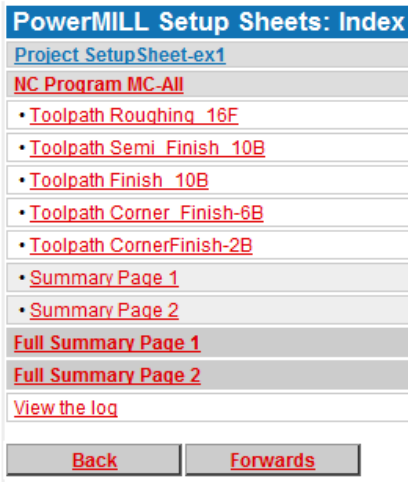
Images of all the **toolpaths** in the selected **NC program** have now been captured.

Each **toolpath** will have a separate page on the **Setup Sheet**.

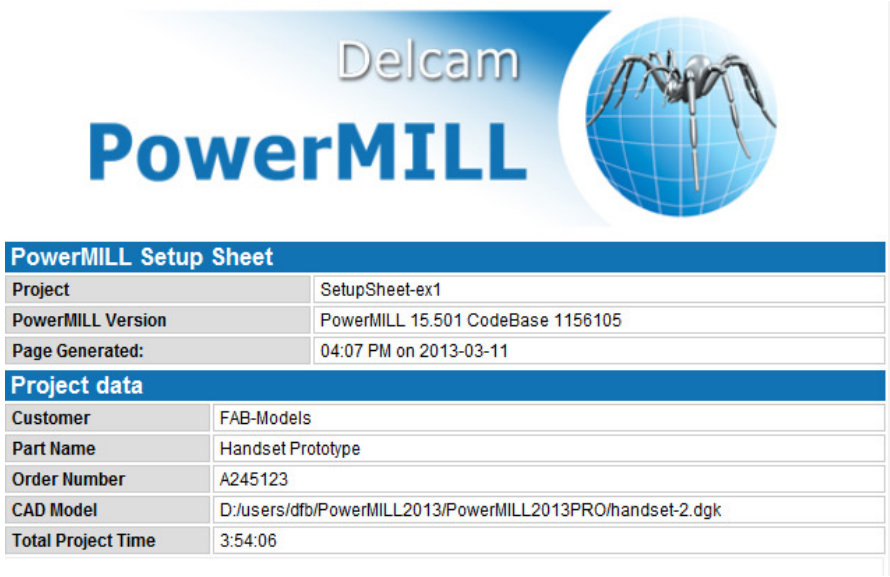
13 Right-click on the **NC Program** - **MC-All** again and select **Setup Sheets - Preview**.

This action will create the **Setup Sheet**, **HTML** files.

The **PowerMILL Setup Sheets: Index** is displayed on the left side of the screen.



14 In the above ***Index*** dialog left click on **Project Setup Sheet-ex1** to display the following sheet.



15 From the **Index** page select the **Toolpath** and **Summary** pages in turn and view the content.

Copyright © Delcam plc

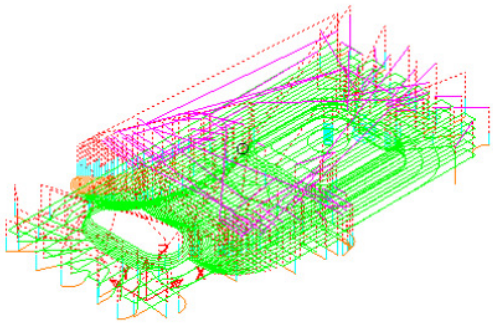
Delcam PowerMILL

PowerMILL Setup Sheet

Project	SetupSheet-ex1
NC Program	MC-All
PowerMILL Version	PowerMILL 15.501 CodeBase 1156105
Page Generated:	04:22 PM on 2013-03-11
Order Number	A245123
CAD Model	D:/users/dfb/PowerMILL2013/PowerMILL2013PRO/handset-2.dgk

Toolpath

Name	Roughing_16F		
Description			
Strategy	Raster Area Clearance		
Tolerance	0.1	Cutting Feed Rate	500
Global Thickness	1	Plunge Feed Rate	500
Radial Thickness	1	Rapid Feed Rate	3000
Axial Thickness		Spindle	1500
Stepover	8	Cutting Time	0:30:57
Stepdown	5	Total Time	0:38:38
Block	215.028 × 90.087 × 50		

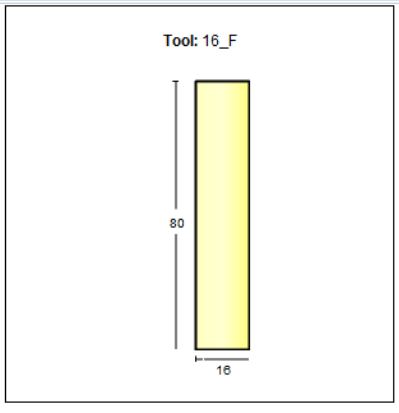


Toolpath Limits

	X	Y	Z
Start Point	107.514	45.043	10
Minimum	-11.258	-10.544	-31
Maximum	226.375	101.331	10

Tool

Name	16_F
Number	12
Type	End Mill
Diameter	16
Tip radius	
Length	80



Toolpath Notes

*Each HTML page can be printed out as individual items. Right-click on the page and from the local menu, select **Print Picture**.*

The 1st sheet of the **Summary Page** is as shown below.

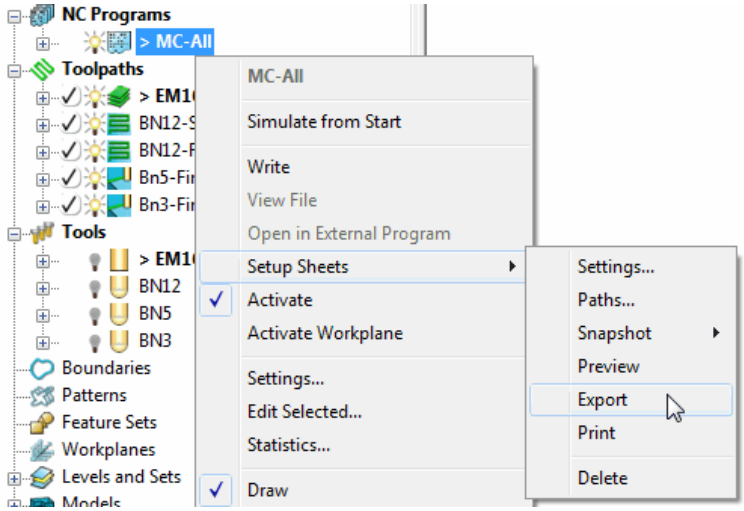
Delcam PowerMILL

PowerMILL Setup Sheet	
Project	SetupSheet-ex1
NC Program	MC-All
PowerMILL Version	PowerMILL 15.501 CodeBase 1156105
Page Generated:	04:22 PM on 2013-03-11

Summary sheet 1 of 2

Toolpath / TAP File	Strategy	Tool	
Roughing_16F C:/Temp/NCPrograms/MC-All.tap	Raster Area Clearance	Type	End Mill
		Diameter	16
		Tip radius	
		Length	80
		Number	12
Semi_Finish_10B C:/Temp/NCPrograms/MC-All.tap	Raster	Type	Ball Nosed
		Diameter	10
		Tip radius	5
		Length	50
		Number	14
Finish_10B C:/Temp/NCPrograms/MC-All.tap	Raster	Type	Ball Nosed
		Diameter	10
		Tip radius	5
		Length	50
		Number	(14)
Corner_Finish-6B C:/Temp/NCPrograms/MC-All.tap	Along Corner	Type	Ball Nosed
		Diameter	6
		Tip radius	3
		Length	30
		Number	7

16 Right-click on the **NC Program** - **MC-All** and select **Setup Sheets > Export**.



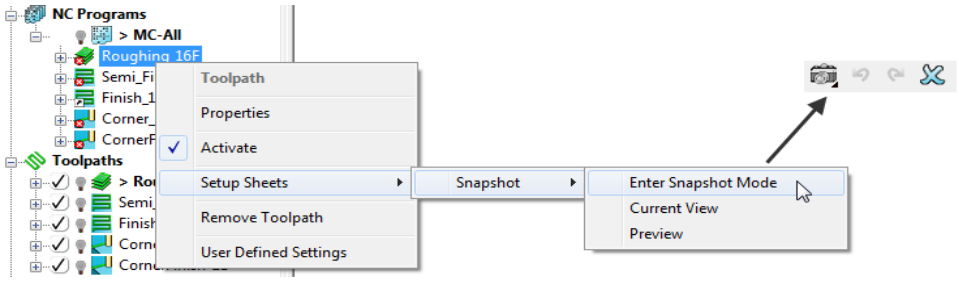
Export stores all the Setup Sheets as HTML documents into the current Project Folder.

Setup Sheets can be printed internally from PowerMILL or externally from the Exported HTML documents.

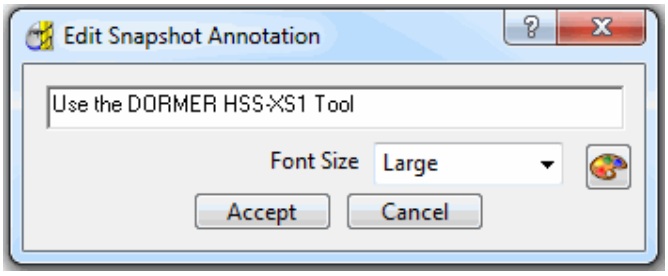
Copyright © Delcam plc

Inputting of Text onto a Setup Sheet > Toolpath Page

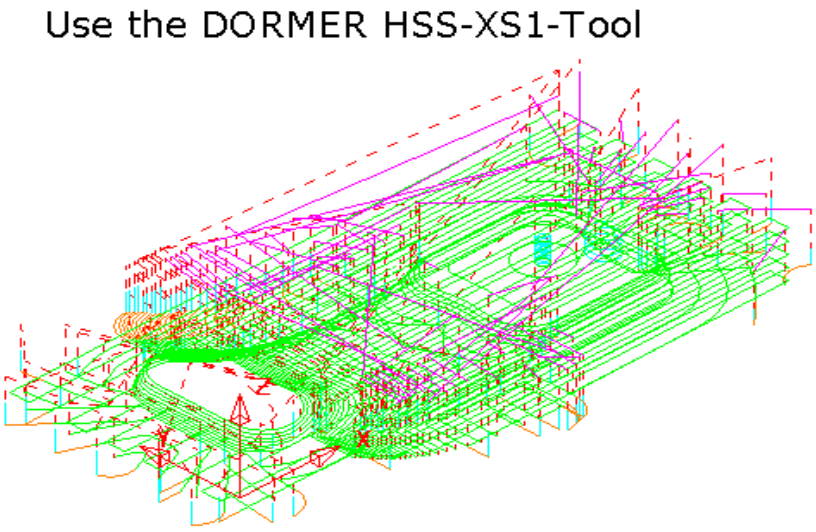
1 In the **NC Program**, **MC-All**, right-click on **EM16-Rgh-A1** and from the local menu, select **Setup Sheets** > **Snapshot** >**Enter Snapshot Mode**.



2 Once the **Snapshot** context toolbar appears left mouse click in the graphics area to open the **Edit Snapshot Annotation**.



3 Change the ***Font Size*** to **Large**, input the above ***text***, and in the Graphics area, left mouse click at the required position before clicking **Accept**.

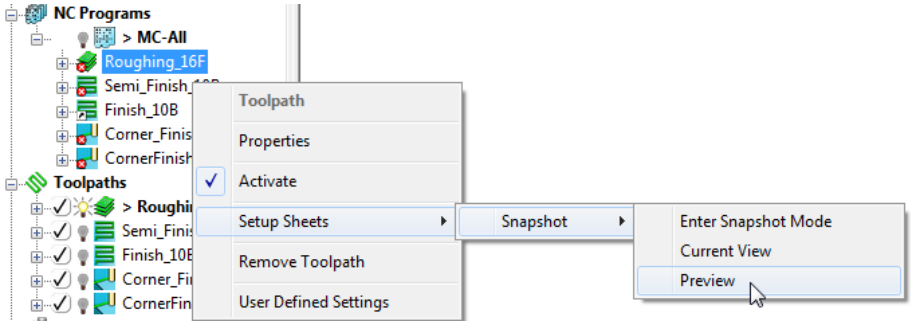


4 Select the ***Save snapshot as current view*** option.

5 Select the ***Green tick*** to accept and close the **Annotation** toolbar.

The setup sheet will be re-processed to include the above text.

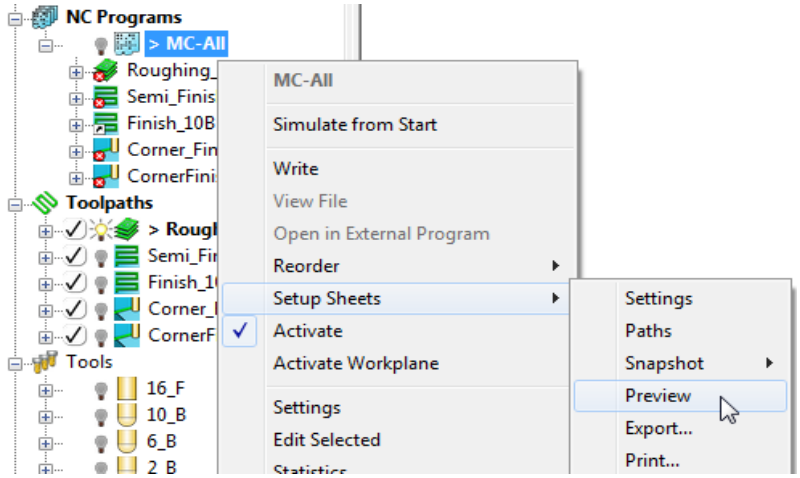
6 To view the modified snapshot, right-click on **Roughing_16F** in the **NC Program** and select **Setup Sheets** > **Snapshot** > **Preview**.



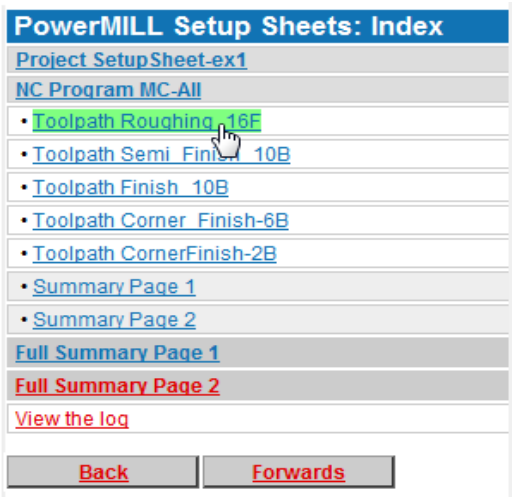
The toolpath and text image shown on the previous page will be displayed in the Preview.

7 Switch from the **Preview** window back to the **PowerMILL *explorer*** .

8 From **NC Programs**, right-click on **MC-All** and from the menus select **Setup Sheets** > **Preview**.



9 From the **Index**, select the Toolpath **EM16-Rgh-A1** to display the relevant **HTML** setup sheet page.

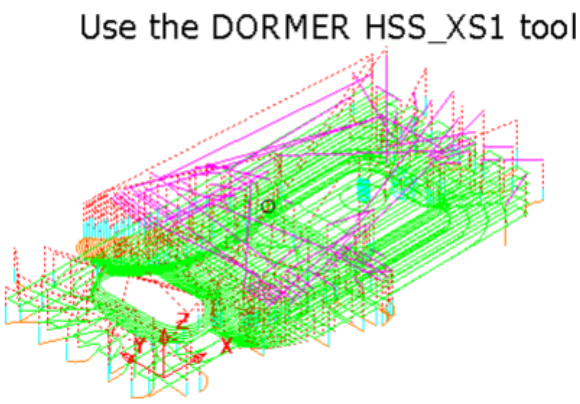


 Copyright © Delcam plc

Delcam PowerMILL

PowerMILL Setup Sheet	
Project	SetupSheet-ex1
NC Program	MC-All
PowerMILL Version	PowerMILL 15.501 CodeBase 1156105
Page Generated:	10:20 AM on 2013-03-12
Order Number	A245123
CAD Model	D:/users/dfb/PowerMILL2013/PowerMILL2013PRO/handset-2.dgk

Toolpath			
Name	Roughing_16F		
Description			
Strategy	Raster Area Clearance		
Tolerance	0.1	Cutting Feed Rate	500
Global Thickness	1	Plunge Feed Rate	500
Radial Thickness	1	Rapid Feed Rate	3000
Axial Thickness		Spindle	1500
Stepover	8	Cutting Time	0:30:57
Stepdown	5	Total Time	0:38:38
Block	215.028 × 90.087 × 50		



Customising Setup Sheets

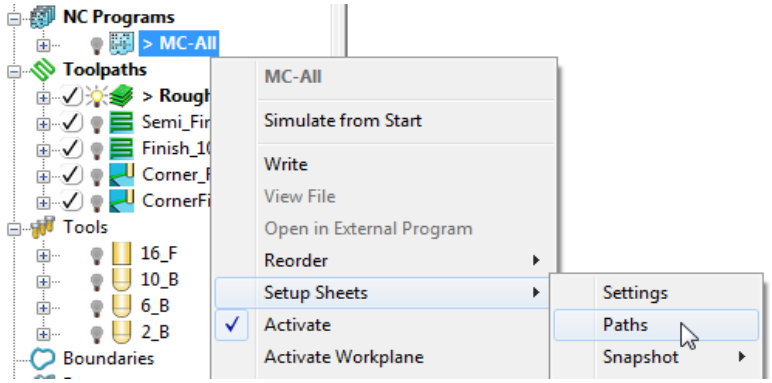
By editing the **HTML** templates it is possible to customise the output of the **Setup Sheets**.

The use of **HTML** editing software is the recommended method to implement changes to the **Setup Sheet** output. There are free downloads available from the web. **Microsoft Word** can also be used but this can give unpredictable results.

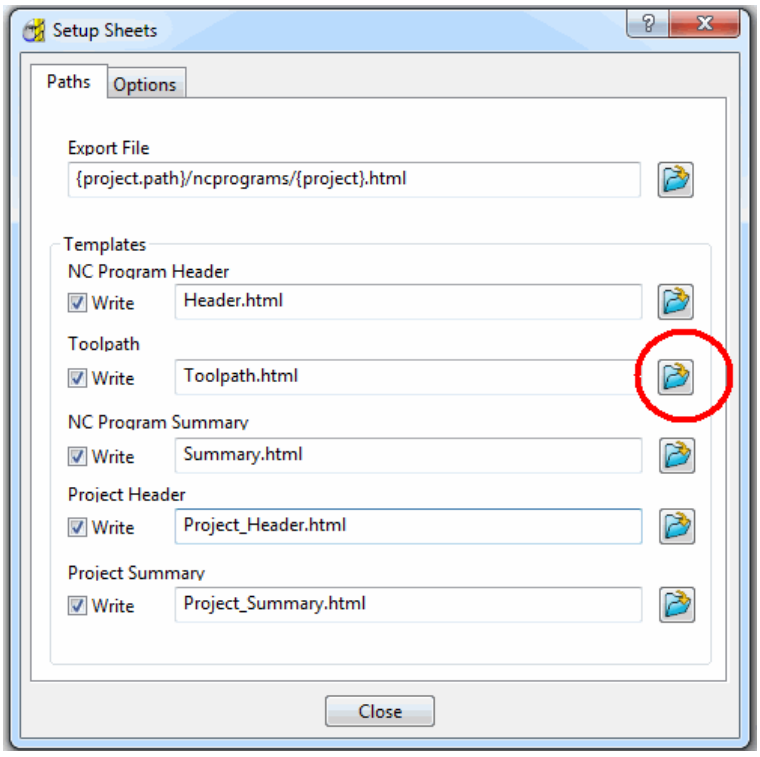
Always create a copy of the template document so that the original is still available if things go wrong.

In the following example an alternative template will be substituted into the **Setup Sheet Toolpath Template** search paths.

1 Right-click on the **NC Program** named **MC-All** and select the option **Setup Sheets** > **Paths**.

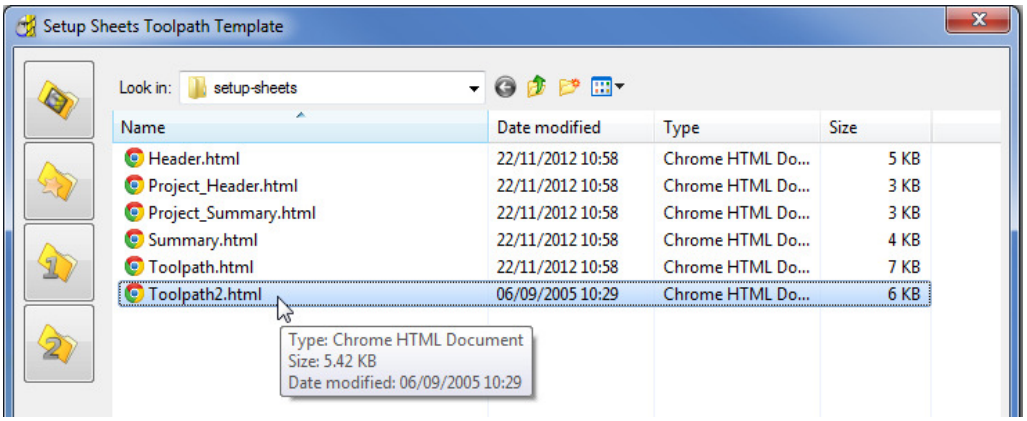


2 On the **Paths** tab select the folder icon in the **Toolpath** > **Write** field.



 Copyright © Delcam plc

3 Browse to **.....\PowerMILL_Data\setup-sheets** and **Open** the **Toolpath2.html** file (customised template).

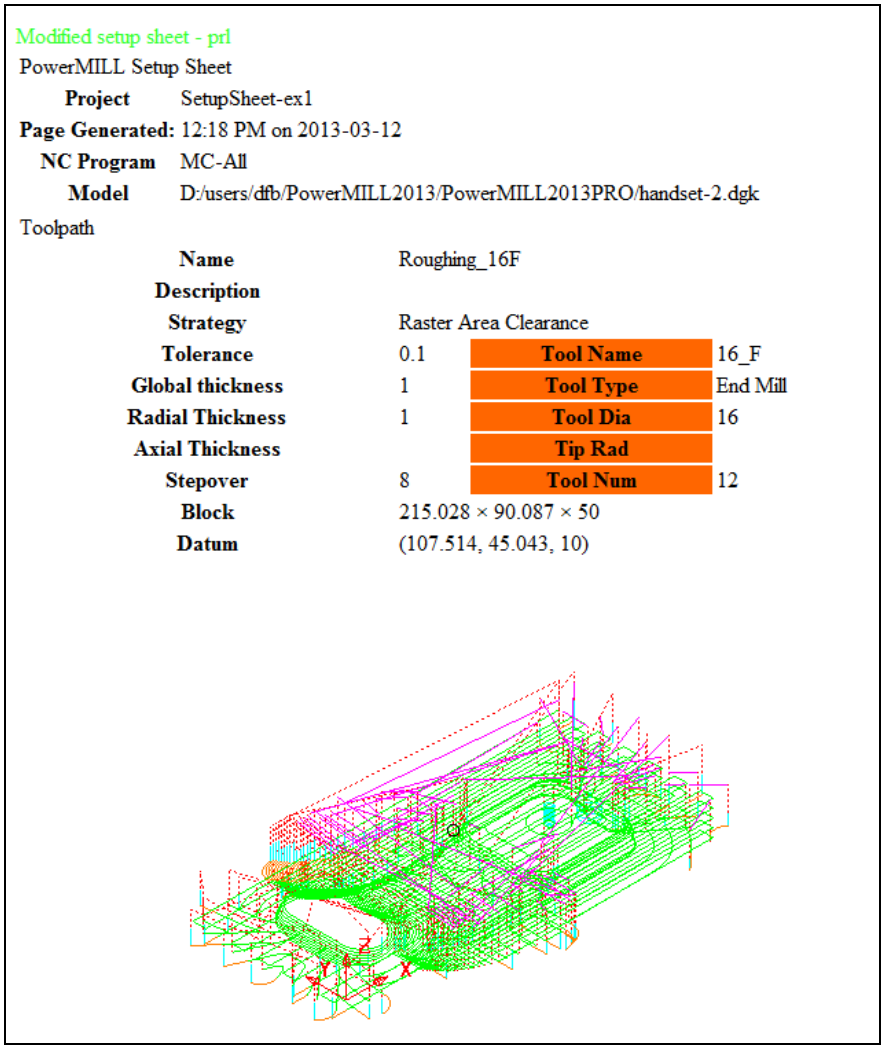


4 Close the form.

5 Click Close on the **Setup Sheets** dialog.

6 Right-click on the **NC Program** named **MC-All** select the option, **Setup Sheets** > **Preview** and re-run the **Setup Sheets** using the new template.

7 On the **Index** page select the new page for the toolpath **Roughing_16F** (Shown below)



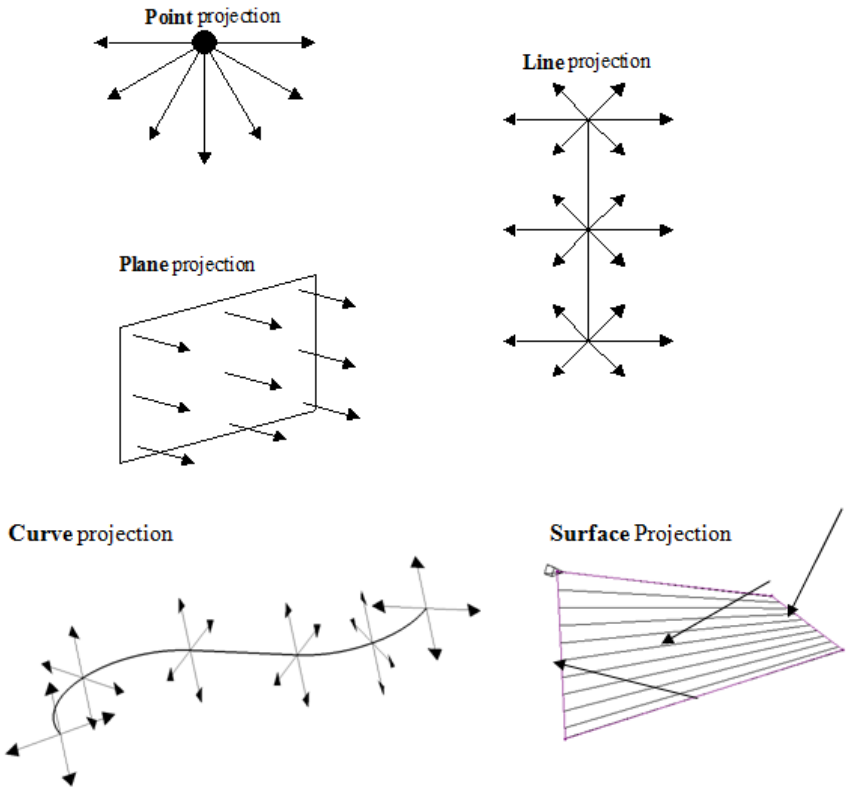
 Copyright © Delcam plc

15. Specialist Finishing

Projection Machining.

Projection Finishing provides greater control of the projection direction of machining strategies onto a component. Applications include the machining of undercut features using specialist tooling and a more accurate finish for features positioned at complex angles to the tool axis. **Projection Finishing** is also an essential strategy for use with **Five Axis** tool alignment applications.

There are five different strategies **Point**, **Line**, **Plane**, **Curve**, and **Surface Projection**.



The simplest way to understand the principle of projection machining is to regard the first four strategies comparable to beams of light radiating into or away from a defined source.

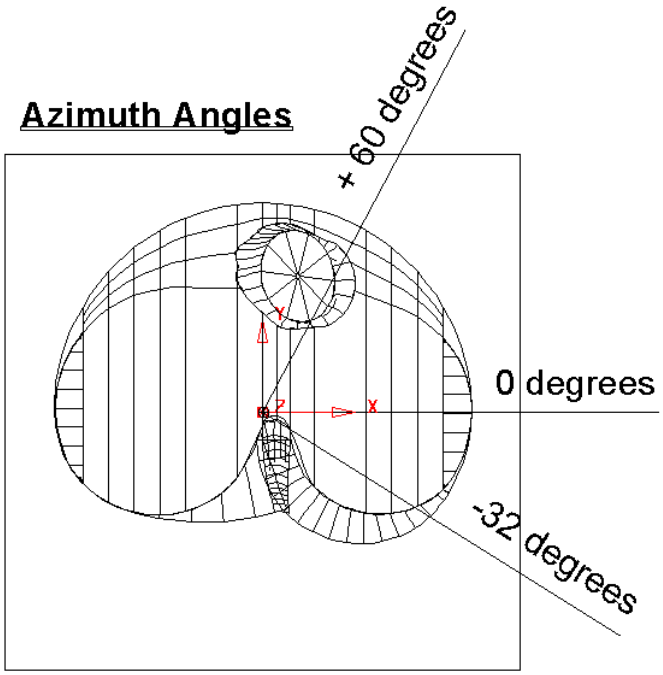
- A **Point** strategy is similar to light radiating from a single light bulb.
- A **Plane** strategy is similar to light radiating from a rectangular floodlight.
- A **Line** strategy is similar to light radiating from a fluorescent tube.

- A **Curve** projection strategy is similar to light radiating from a curved neon tube.
- A **Surface** projection projects the tool Inwards or Outwards along the normals of a selected surface.

In the **Finishing** dialog, the projection **Direction** can be toggled between **Outwards** and **Inwards** from the user defined plane.

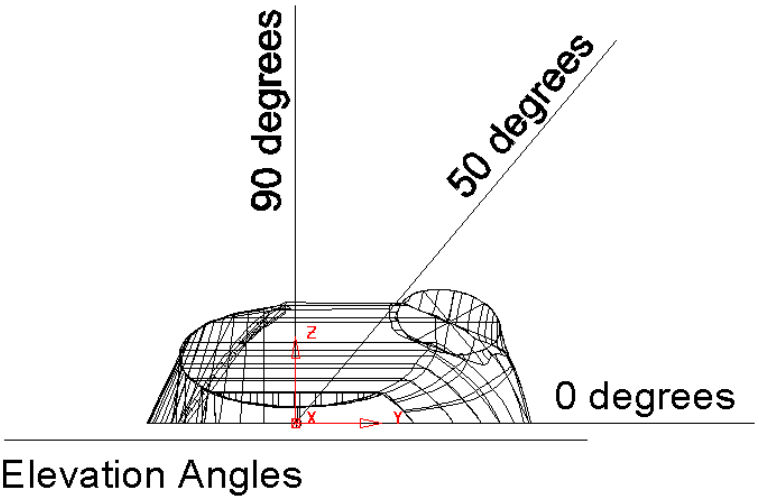
In the **Projection Finishing** forms, angular definitions are input using **Azimuth** and **Elevation**. By using a combination of the two any complex 3D angle can be defined.

- **Azimuth Angle**:- Starting from **+X = 0** this is the angle anticlockwise around the **Z** axis.



This is shown on the chamber model, with a view looking down the Z-axis.

- **Elevation Angle** - this is the upward angle from the **XY** plane



This is shown on the chamber model, with a view looking down the X-axis.

Copyright © Delcam plc

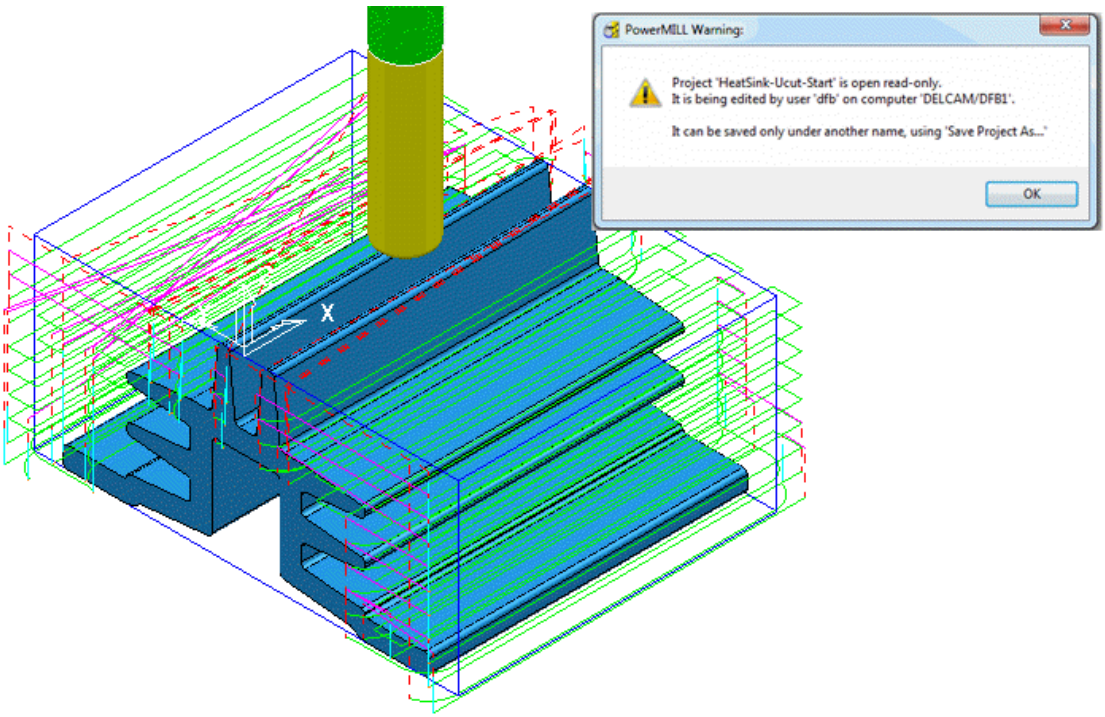
Plane Projection Example

All five **Projection Finishing** strategies can be used to machine undercut features if used with suitable tooling such as Disc or Spherical. The following example shows an undercut application using the **Plane Projection** strategy.

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

2 Select **File** > **Open Project**:

.....\PowerMILL_Data\Projects\Heatsink-Ucut-Start

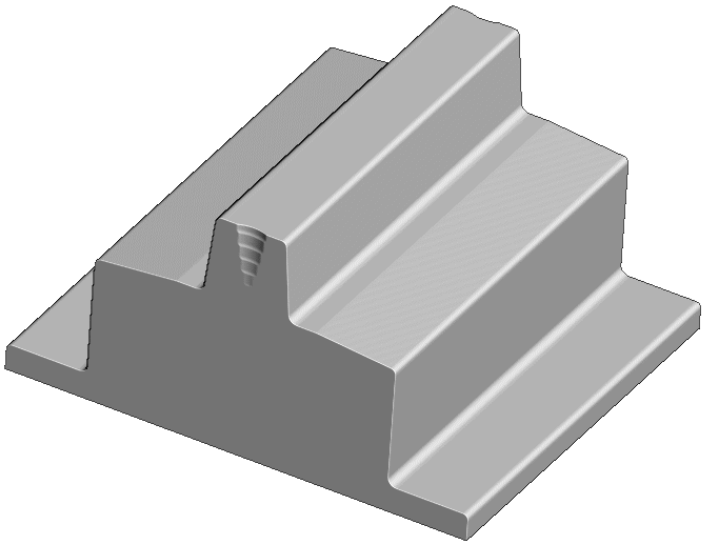


3 **Save** the **Project As**:

.....\COURSEWORK\PowerMILL_Projects\ Heatsink-Ucut-EX1

4 Open the **Block** form and with **Defined by - Box** and **Type - Model** set select **Calculate**.

5 **Simulate** both of the existing toolpaths in **ViewMILL**.



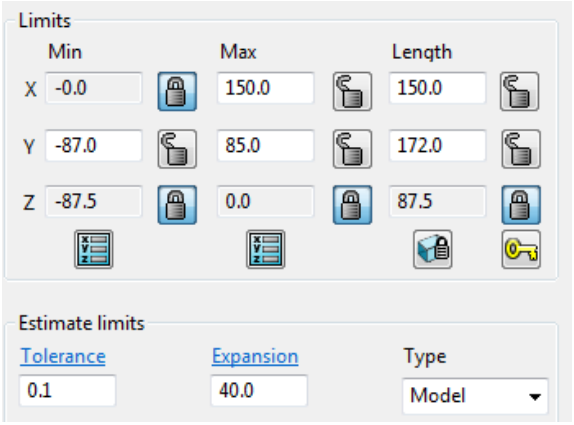
6 Return the **ViewMILL** simulation to the **No-Image** view .

7 Disconnect **ViewMILL** from the simulation .

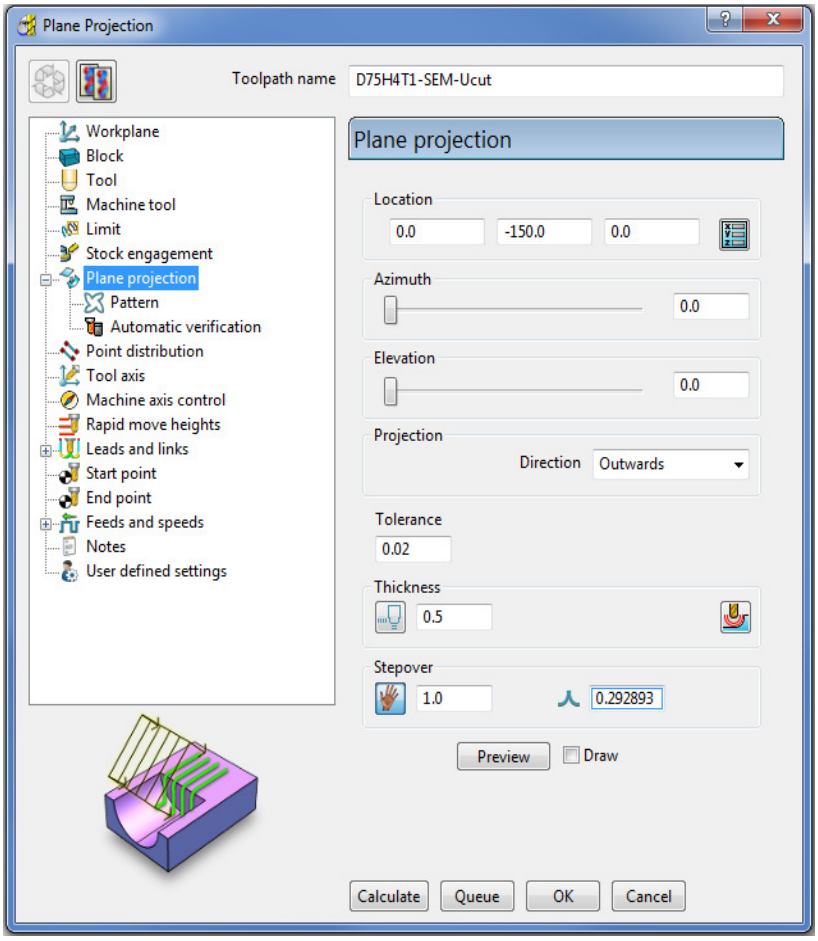
8 **Activate** the **D75-H4-T1** Disc Cutter.

9 In the **Block** dialog ***lock*** the **X** and **Z** values.

10 Calculate the **Y** values of the **Block** with an **Expansion** value of **40**.

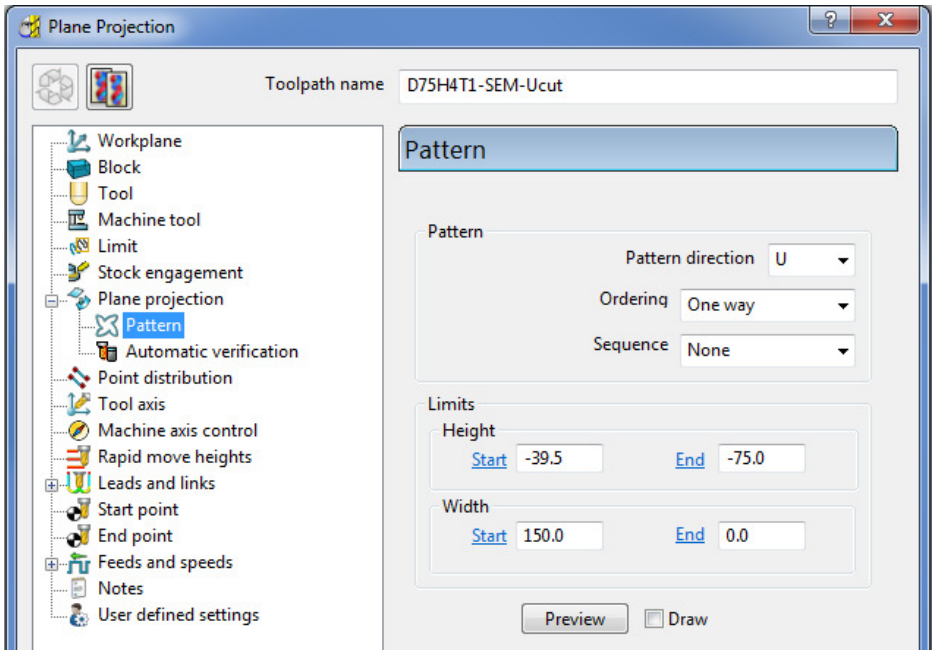


11 In the **Strategy Selector** - **Finishing** dialog, select **Projection Plane Finishing** and fill in the pages of the dialog exactly as shown below.



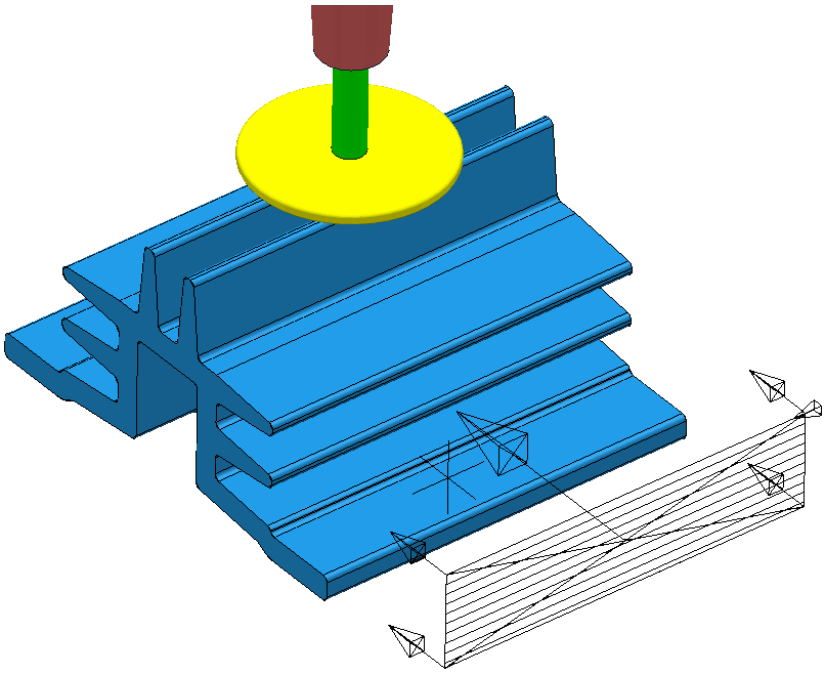
The Location is the local datum for the projection strategy. All other coordinates entered in the dialog are relative to this point.

Copyright © Delcam plc



The values for the **Start** and **End** heights can be obtained by viewing the model in the **X** direction and using the cursor position to display **Z** values at the bottom right of the screen.

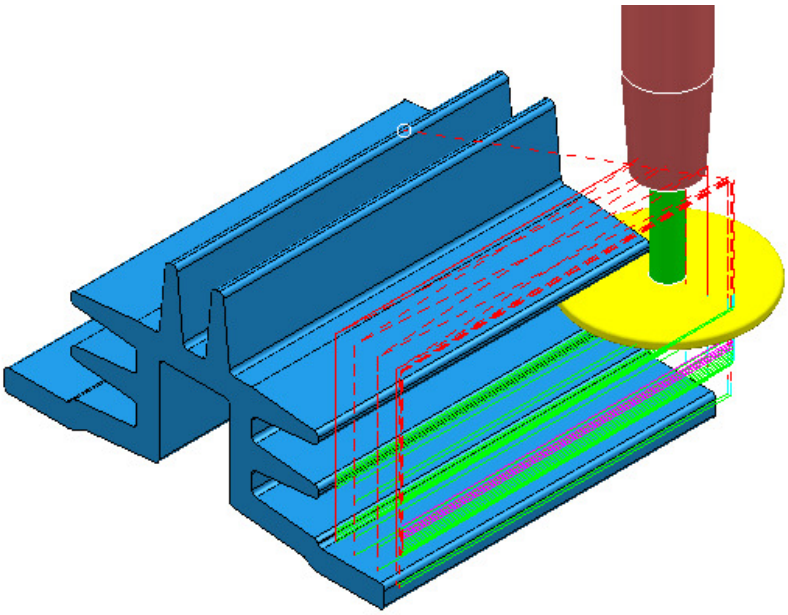
12 Select **Preview** to view the **Plane Projection** pattern.



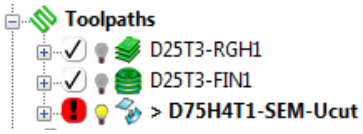
The **Projection Plane > Preview** is displayed as shown and the user can dynamically alter the view to visually check for potential adjustments.

When using projection finishing, (**Point**, **Line** or **Plane**), the **Anchor Point** must be positioned (along the projection direction) away from the model by at least the cutter radius.

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

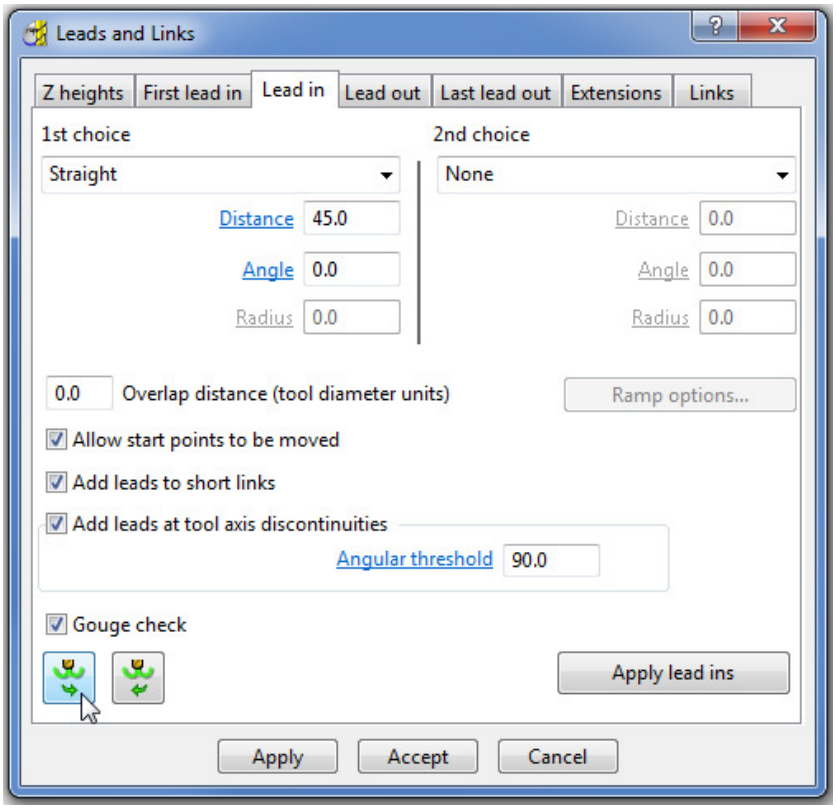


Observe that the tool is **gouging** through the fins when it retracts at the end of each pass. This is confirmed by the ***red*** toolpath, icon displaying an exclamation mark in the explorer.



In this case suitable Leads and Links will be applied to easily rectify the problem.

1 Open the Leads and Links dialog and select the Lead In tab.



 Copyright © Delcam plc

2 Input the following information:

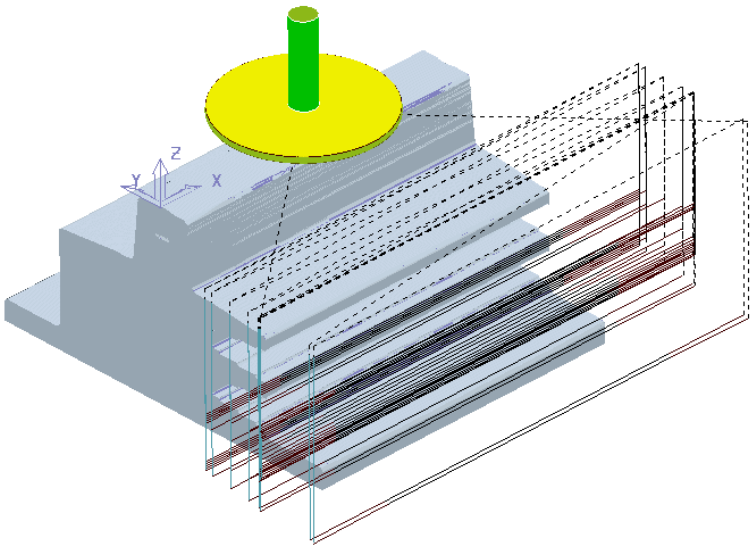
1st Choice - Straight

Distance 45

Angle 0

3 Select the **Lead Out the same as Lead in** tab.

4 **Apply** and **Accept** the dialog.



The toolpath **Lead** moves are now updated and the ***red warning*** icon in the **PowerMILL *explorer*** has automatically updated to ***white***.

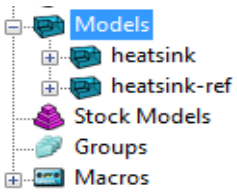
5 **Save**, but do not close, the **Project** as it is to be continued in the next section.

Surface Projection Example

Surface Projection will be used for the **finishing** strategy as it will provide a consistent **stepover** within the undercut regions. To allow this a specially created, single, **reference surface** will be imported (To create this surface the user would require the services of a suitable CAD package such as **PowerSHAPE**). The **reference surface** does not necessarily have to represent the true shape of the finished component. In practice it could be positioned inside, outside, or flush with the actual model surfaces to be machined. In this example the **reference surface** is flush with the existing model.

1 **Import** the additional **model**:

.....\PowerMILL_data\models\heatsink-ref.dgk



The new imported surface appears as a separate item in the **Models** section of the explorer. This will assist when selecting it as the **reference surface** to be used with the **Surface Projection** dialog.

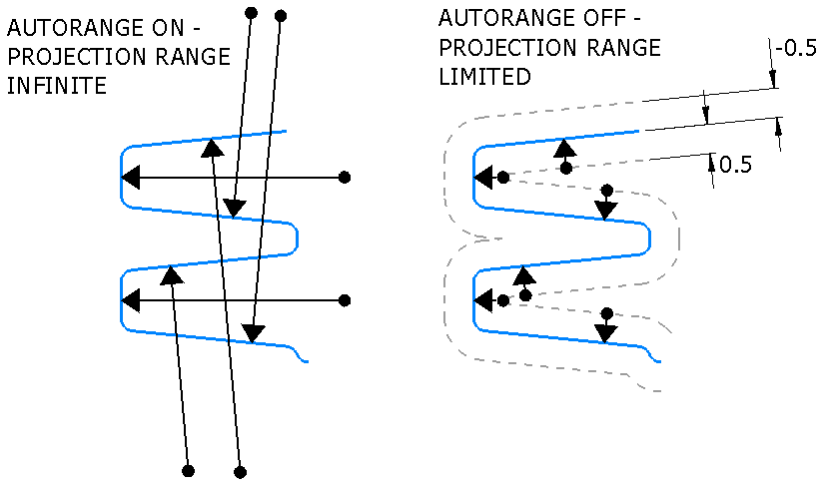
2 From the main pull down tabs select **View** - **Toolbar** - **Command** to open the command input window below the graphics area and type in the following 3 lines:-

EDIT SURFPROJ AUTORANGE OFF

EDIT SURFPROJ RANGEMIN –1

EDIT SURFPROJ RANGEMAX 1

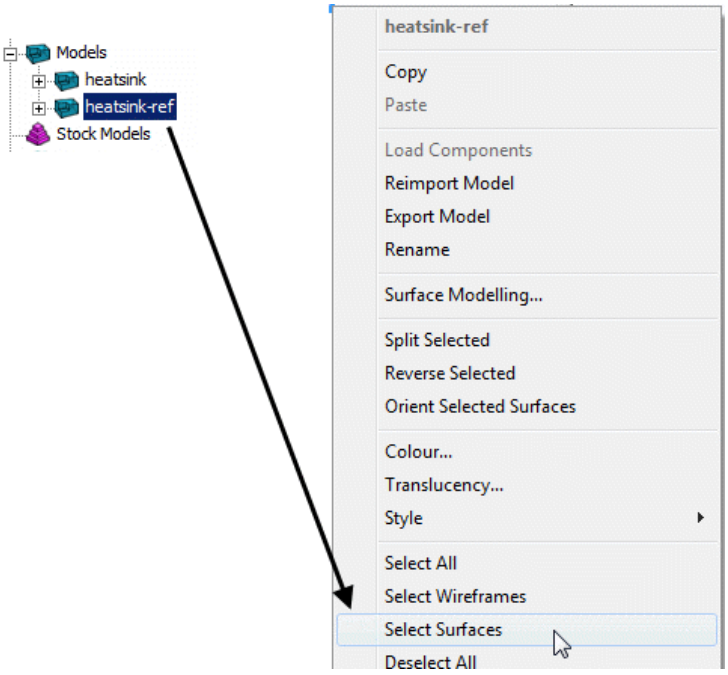
This is to restrict the tool projection distance to within 0.5mm of the reference surface. Otherwise it would be projected from infinity onto non-undercut faces on the outside of the model.



*The default setting is restored by typing **EDIT SURFPROJ AUTORANGE ON**.*

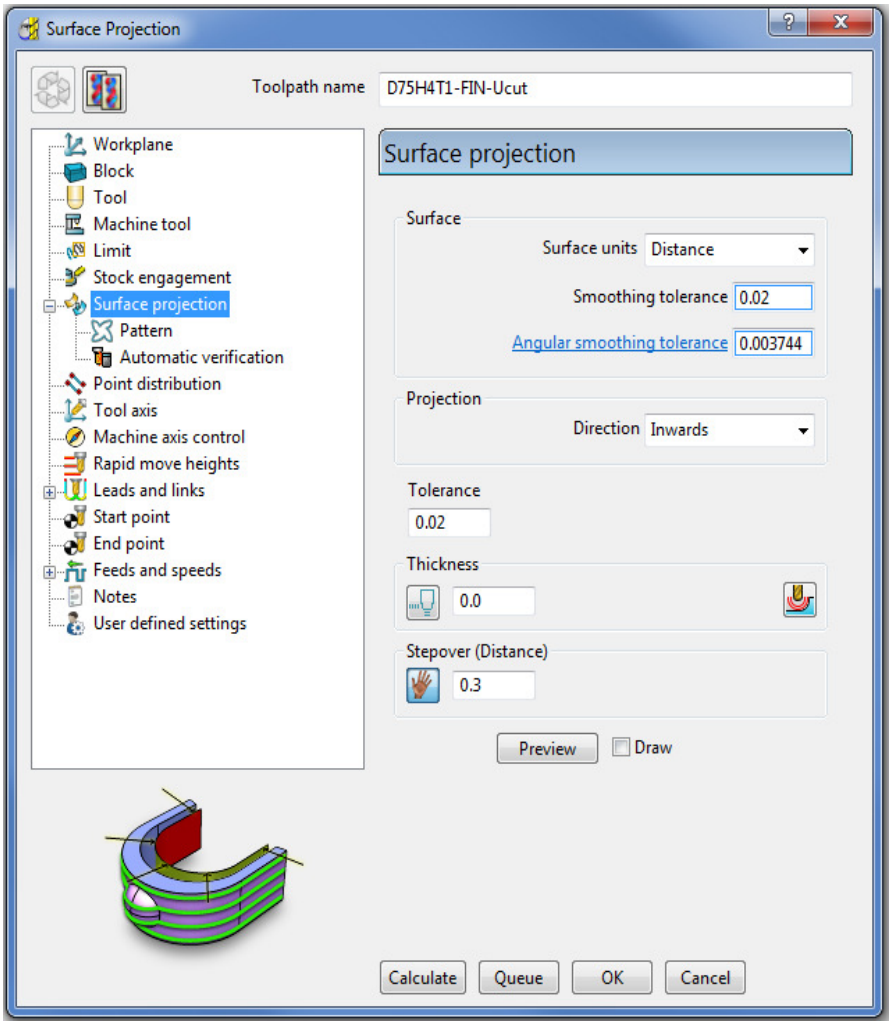
3 From the **Toolpath Strategies** dialog, select **Projection Surface Finishing**.

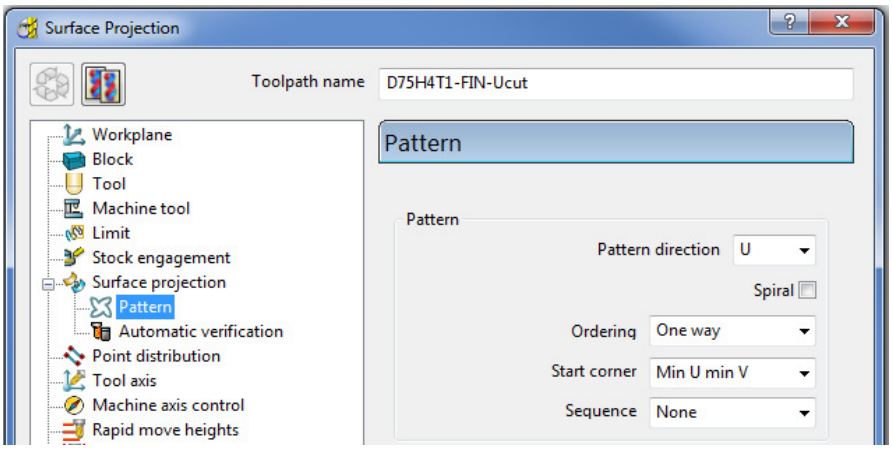
4 In the **PowerMILL *explorer***, right-click the model, **Heatsink-ref** and from the local pull down menu click **Select Surfaces**.



 Copyright © Delcam plc

5 In the **Surface Projection Finishing** dialog input the settings and data exactly as shown below.

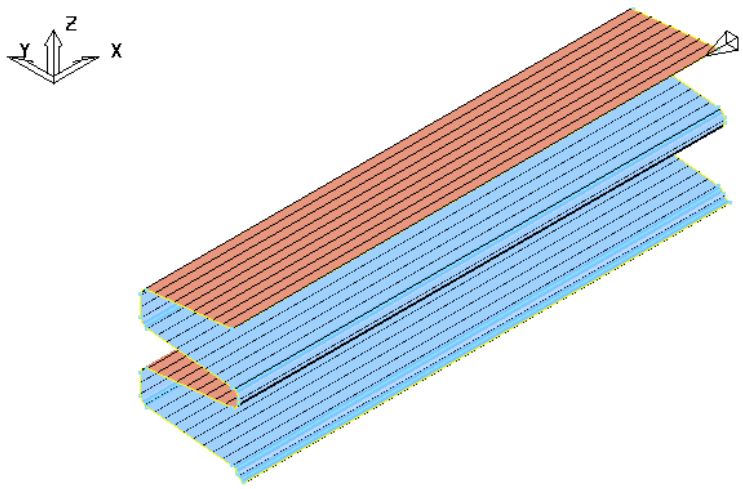




6 Select **Ordering** as **One way**.

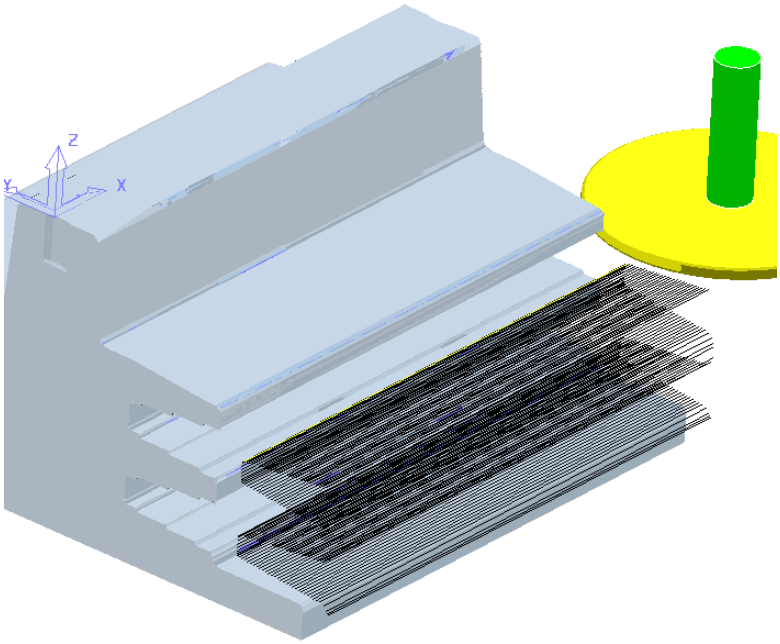
7 Set the **Start Corner** as: **Max U max V**

8 Select **Preview**.



The **Preview** pattern for the strategy is displayed on the ***reference surface*** (for clarity, a stepover value of 3 was used in this illustration).

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



*The unidirectional toolpath is shown here without the defined **Leads and Links** displayed.*

*A suitable **Lead In** and **Lead Out** must be applied to prevent the tool from **gouging** through the overhang.*

The application of **Surface Projection** Finishing has produced a strategy with a consistant ***stepover*** across all of the undercut areas.

10 Select **File** > **Save** to update the stored **Project**.

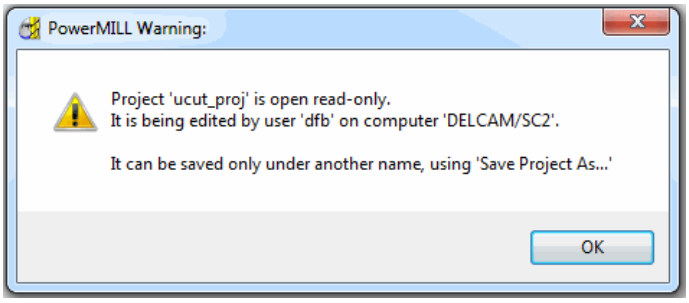
 Copyright © Delcam plc

Line Projection Example

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

2 Select **File** > **Open Project**:

.....\PowerMILL_Data\Projects\ucut_proj

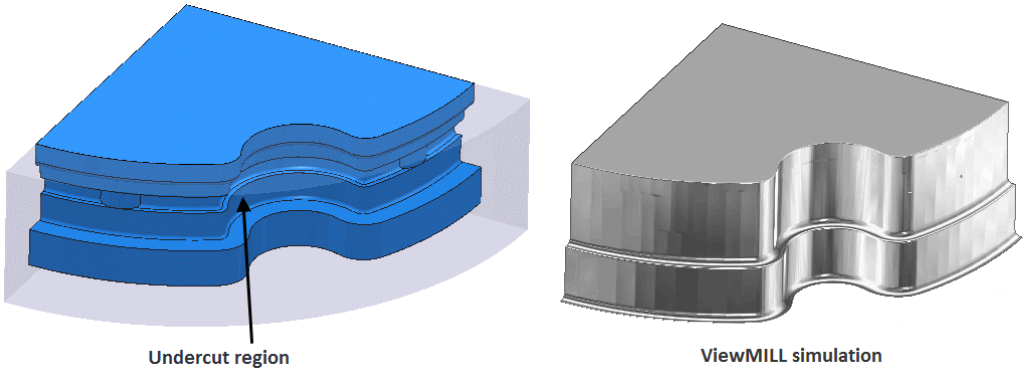


3 Select **File** > **Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\Line-Projection

4 Select an **ISO 3** view.

The imported **Project** (training_ucut_proj) contains a **Model**, **Block**, **Roughing** and **Semi-Finishing** strategies with **1** and **0.3 thickness** values respectively. The undercut regions have not yet been machined.

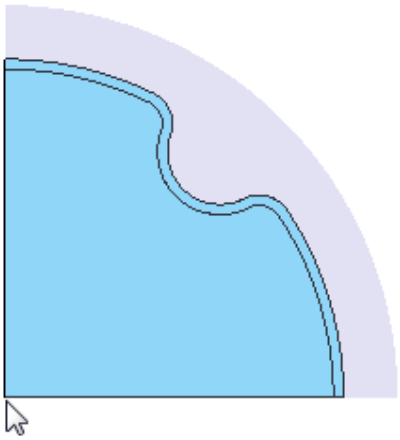


5 **Activate** the **toolpath** (**Rough**) to reinstate the settings and then simulate both existing toolpaths in **ViewMILL**.

Tipped Disc cutters will be used for machining the undercut areas. The toolpath will be created by projecting a suitable strategy inwards, towards a vertical line, located at the lower, left corner.

6 Select a **View from Top (Z)**.

7 Place the cursor over the lower left, corner point without clicking and read the **X** & **Y** values from the information toolbar located below the graphics area.

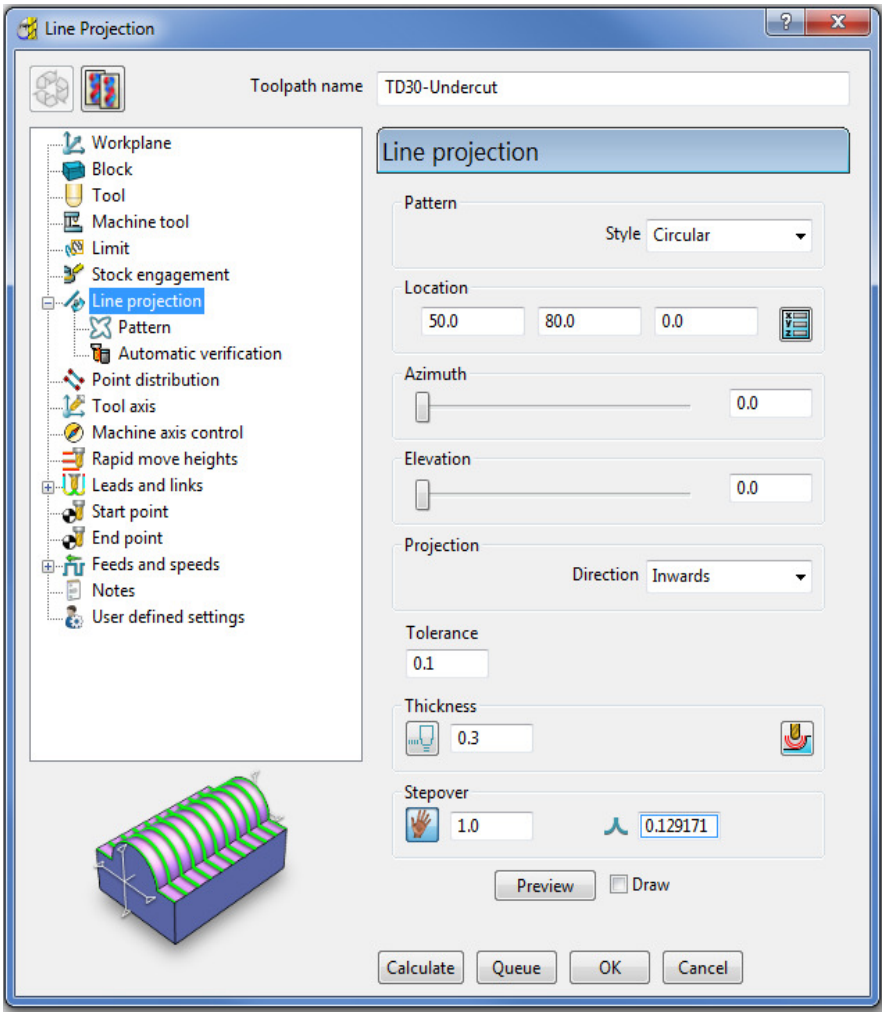


*The **Intelligent Cursor** will accurately lock onto the corner providing accurate coordinate values.*

8 Activate the **TD30** cutting tool.

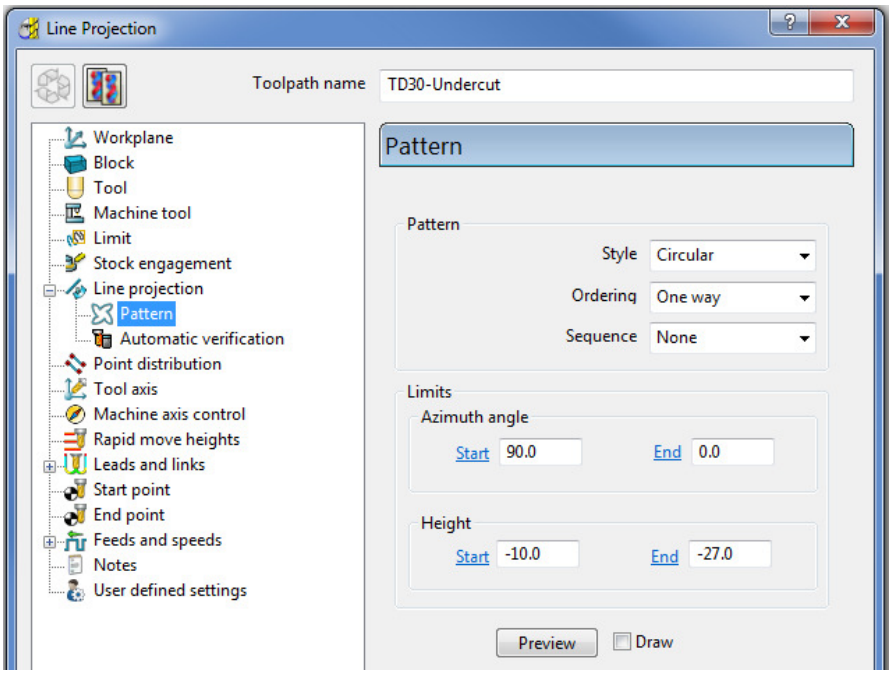
9 From the **Toolpath** strategies dialog, select **Line Projection Finishing**.

10 Fill in the pages of the strategy dialog exactly as shown below.



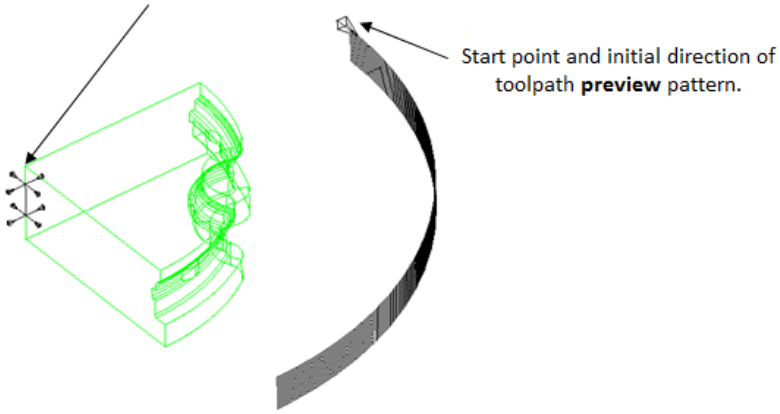
 Copyright © Delcam plc

The values for the **Start** and **End** heights were obtained by dynamically moving the cursor over the model to obtain the **Z** values in the ***cursor coordinate boxes*** at the bottom right of the graphics area.

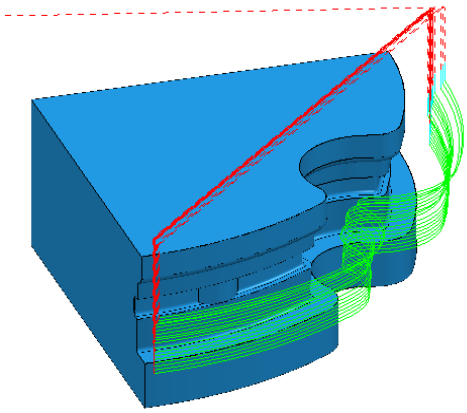


11 Select **Preview**.

The **Preview** shows the location of the ***projection line*** with **arrows** showing an inward projection direction from the **pattern**.



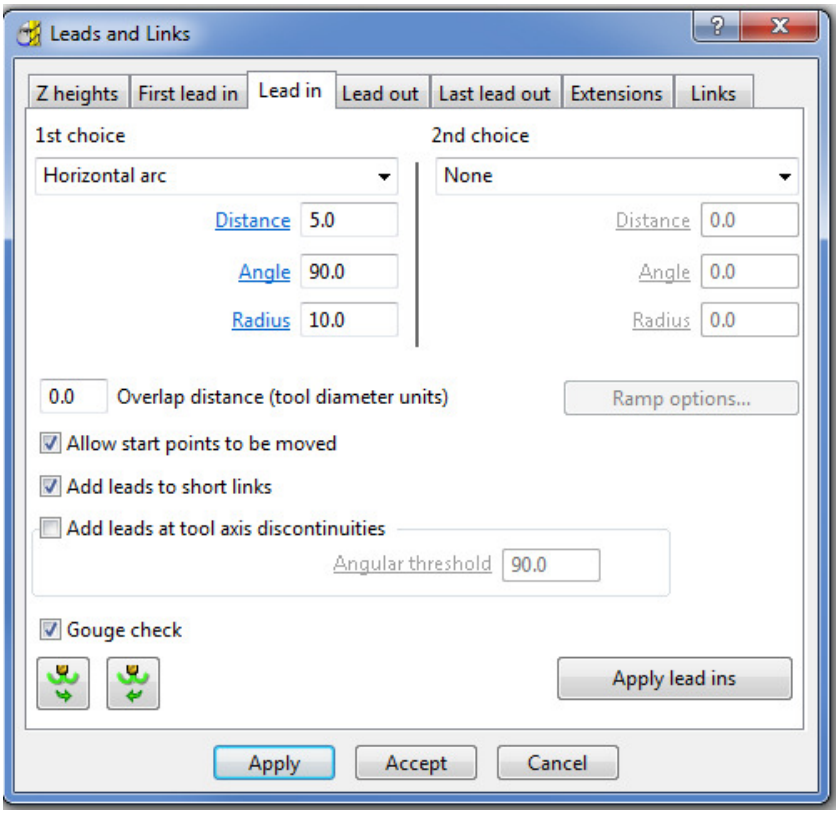
12 **Calculate** the toolpath and then **Close** the dialog.



The tool is currently **gouging** through the overhang at the end of each pass.

This is indicated by the red colour of the toolpath icon as displayed in the **PowerMILL *explorer***.

13 Open the **Leads and Links** dialog.



14 Select the **Lead In** tab.

15 Input the following information:

1st Choice - Horizontal Arc

Distance - 5.0

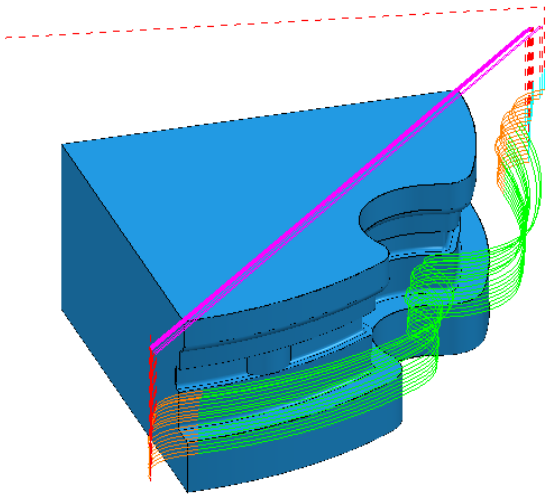
Radius - 2.0

Angle - 90

16 Select the **Lead out the same as lead in** tab.

17 Click **Apply** and then **Accept** to close the dialog.

 Copyright © Delcam plc



*The **extension** value in **Leads and Links** must be more than the tool radius to prevent gouging.*

*The **red** warning icon in the **PowerMILL explorer** will now have changed to **green**.*

18 Select **File** > **Save** to update the stored **Project**.

Point Projection Example

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

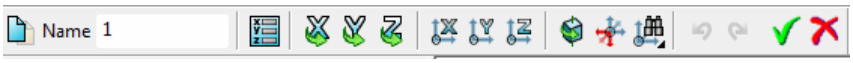
2 Select **File** > **Import Model**:

.....\PowerMILL_Data\Models\radknob.dgk

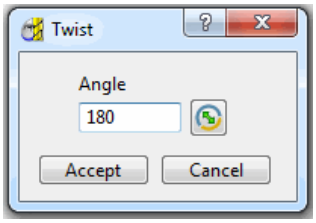
3 Select **File** > **Save Project As**:

.....\COURSEWORK\PowerMILL_Projects\PointProj-ex1

4 In the **PowerMILL *explorer***, right-click **Workplane 1** and from the local menu select **Activate** followed by **Workplane Editor** to display the following toolbar:

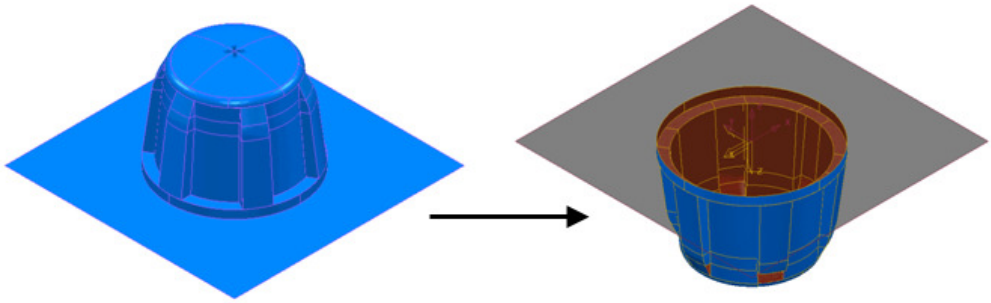


5 From the toolbar select **Twist about Y** to open the the **Twist** dialog.



6 In **Angle**, enter **180** and click **Accept**.

7 Click the **Green** tick to accept the changes and close the toolbar.



The **Workplane** is rotated **180** degrees about **Y** to provide vertical machining access to the inner wall. However, machining is currently impossible due to the pocket being shielded by an untrimmed planer **surface** (shown part shaded). This example will apply **Component Thickness** options to set this **surface** to be ***ignored*** while applying the machining strategy.

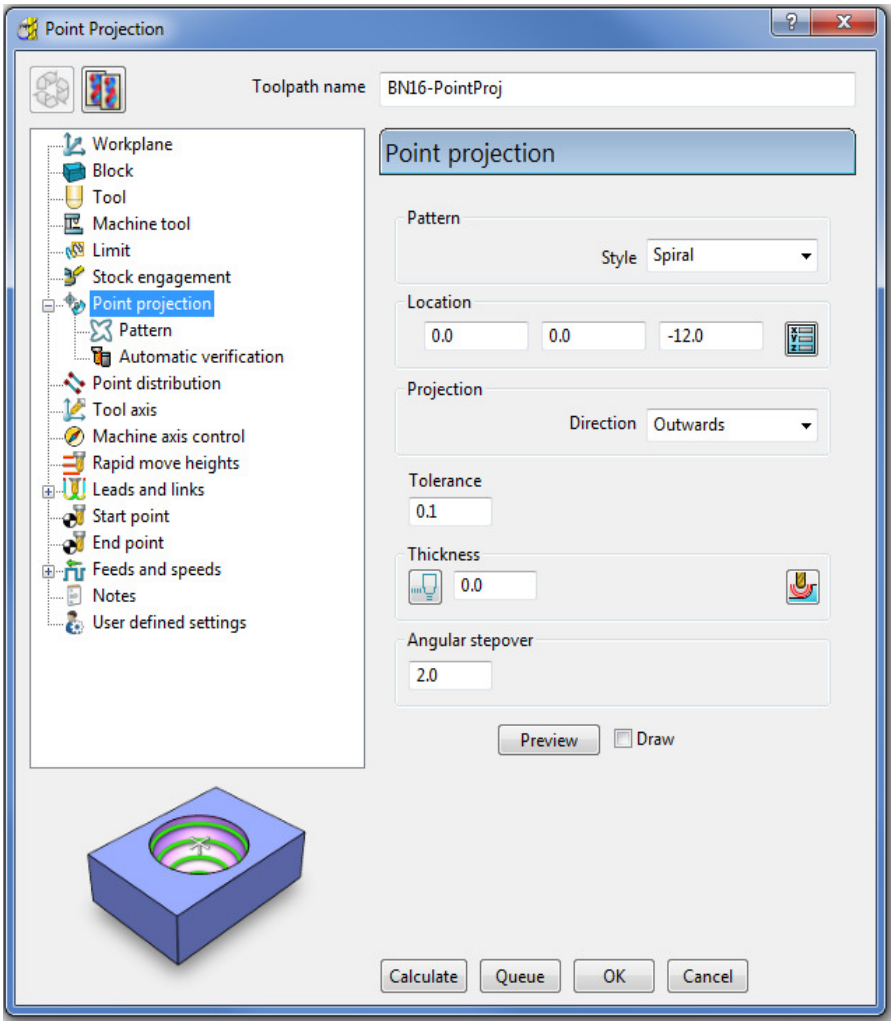
8 Create a Dia **16** ball nosed cutter named **BN16**.

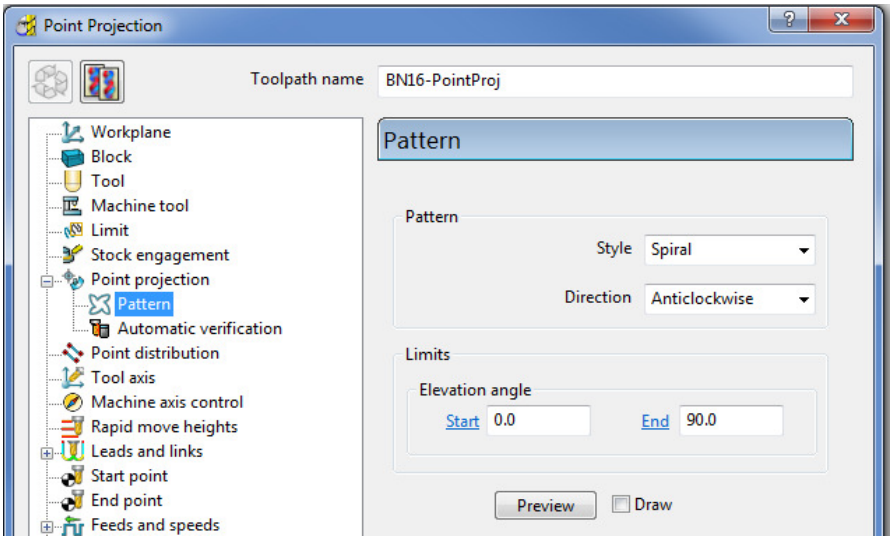
9 **Calculate** a **Block** using **Defined by** - **Box** and **Type** - **Model**.

10 **Calculate** the **Rapid Move Heights** using the default settings.

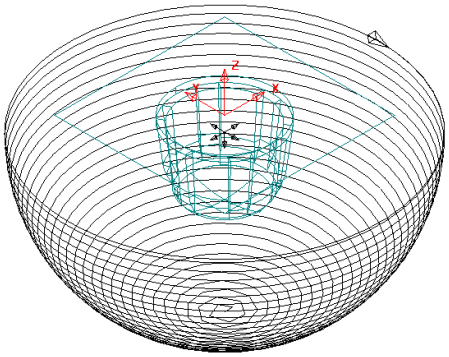
11 From the **Strategy Selector** dialog, select **Projection Point Finishing**.

Copyright © Delcam plc

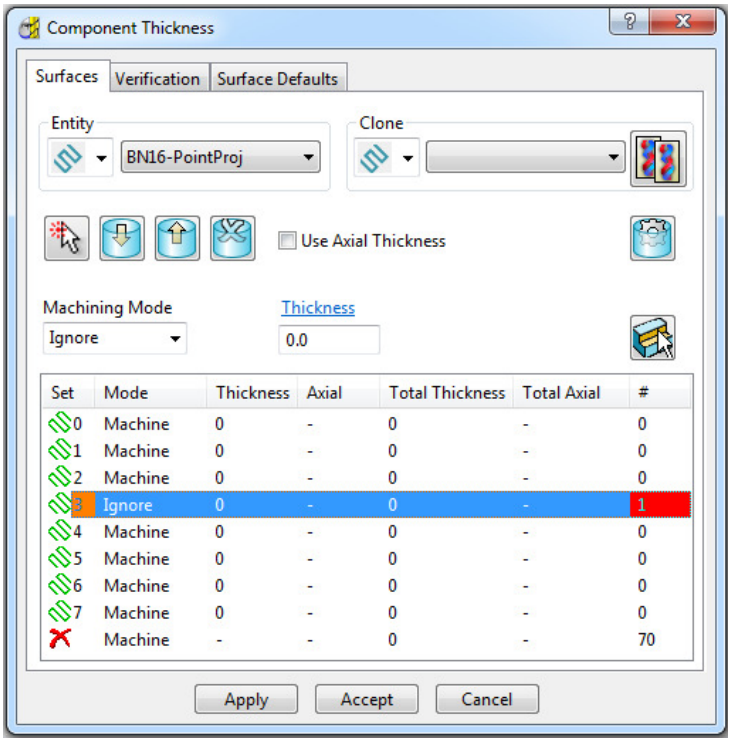




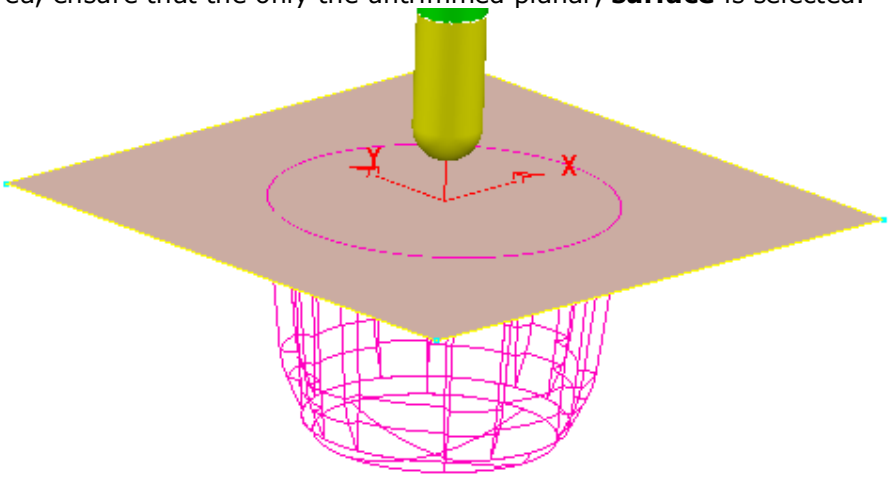
12 Fill in the page of the dialog exactly as shown above and select **Preview** to observe the **Point Projection** setup.



13 In the main page of the **Point Projection Finishing** dialog select the **Component Thickness** icon and I the dialog select the **Surfaces** tab.



14 Click on any of the 8 rows to display a coloured box at the start. In the graphics area, ensure that the only the untrimmed planar, **surface** is selected.



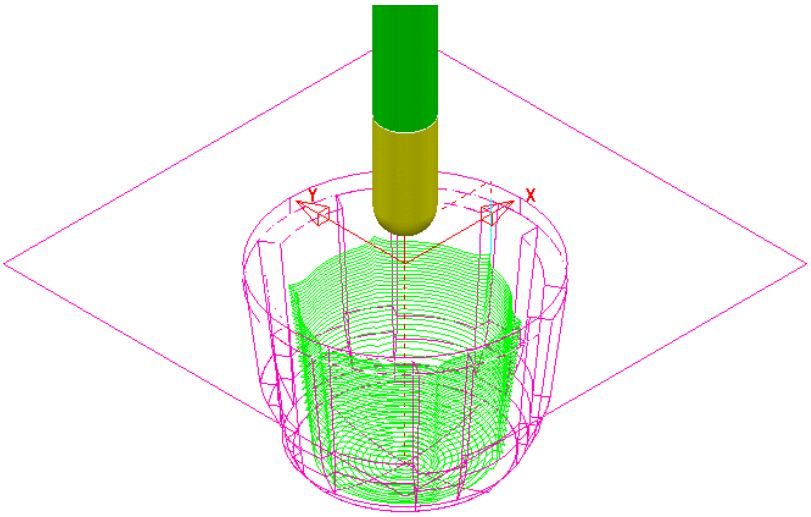
Copyright © Delcam plc

15 Click the **Acquire Components** icon to add the selected untrimmed, planar surface to the row.

16 Select **Ignore** in the **Machining Mode** box.

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

18 **Calculate** the **Projection Point Finishing** toolpath and then **Close** the dialog.



The inside of the pocket has successfully been machined. This would not have happened if the untrimmed surface, blocking access had not been included in a **Component Thickness** list with the **Machining Mode** set to **Ignore**.

19 Select **File** > **Save** to update the stored **Project**.

Curve Projection Example

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

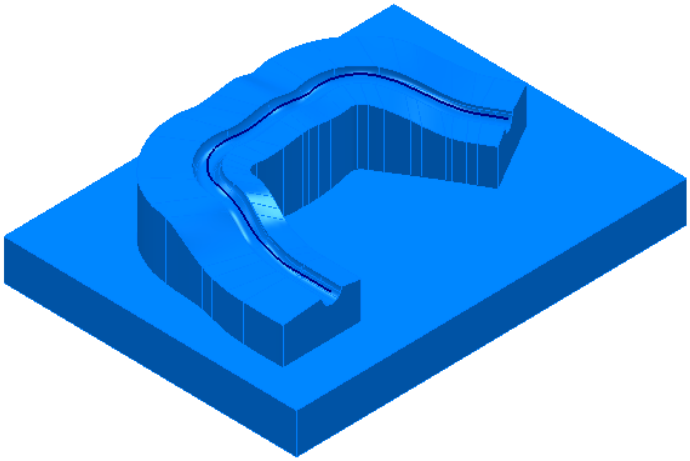
2 Select **File** - **Import Model** and select:

.....\PowerMILL_Data\Models\ crvproj_3D.dgk

(Note the model also includes a wireframe curve running centrally along the slot).

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

.....\COURSEWORK\PowerMILL_Projects\CurveProj-Ex1



4 Calculate the **Block** using **Defined by - Box** and **Type - Model**.

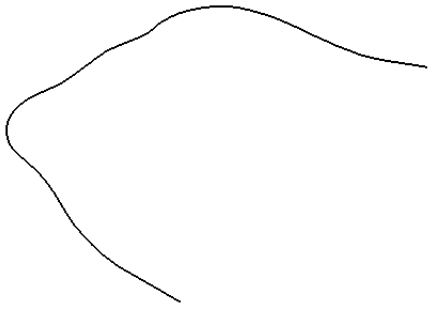
5 Define a Dia **16** Ball Nosed tool called **BN16**.

6 Calculate **Rapid Move Heights** using the default settings.

7 In the **PowerMILL *explorer***, right-click **Pattern** and from the local menu select **Create Pattern**.

8 Select the ***wireframe curve*** running centrally along the slot on the **Model**

(Note; The **Wireframe** view must be active for the curve to be visible).

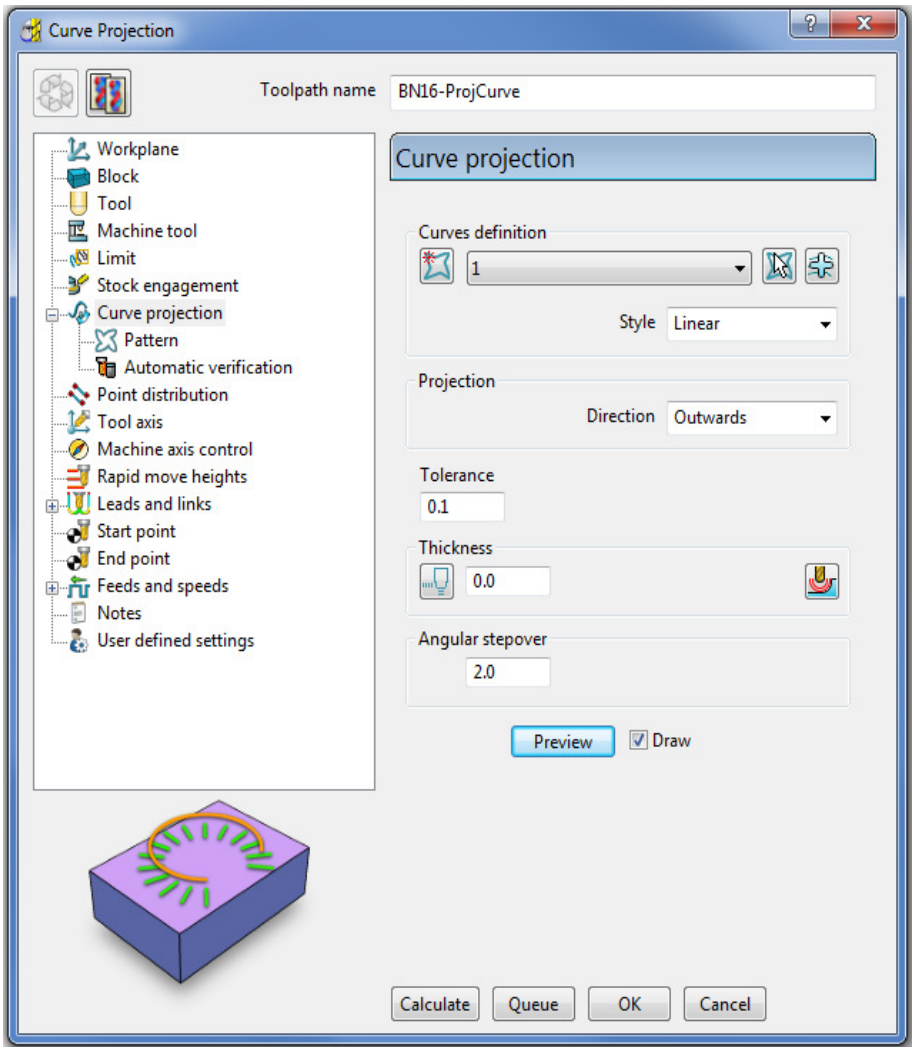


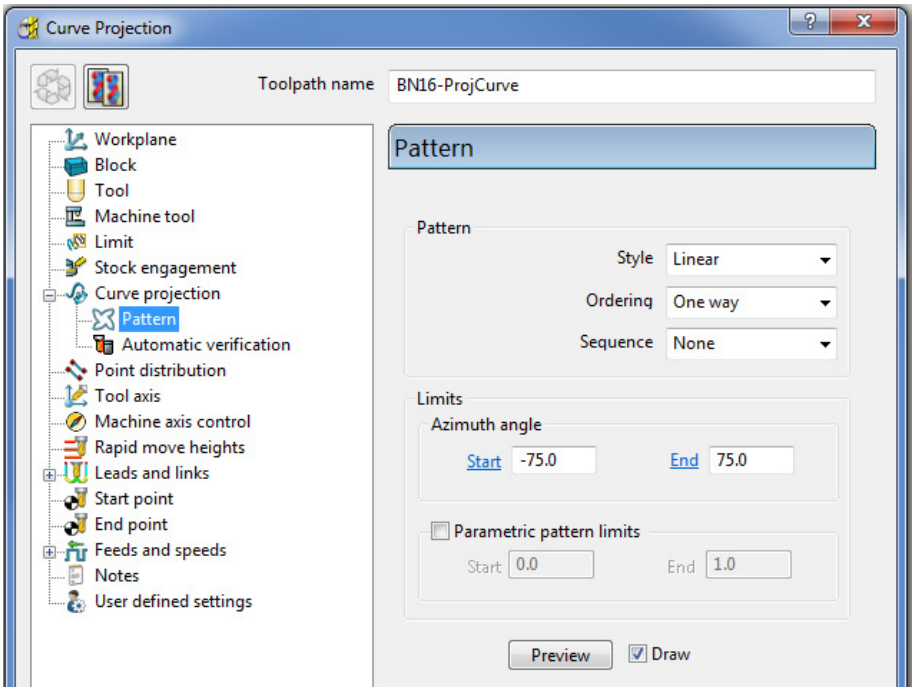
9 In the **PowerMILL *explorer***, right-click on the new **Pattern** (**1**) and in the local menu select **Insert** > **Model**.

10 From the **Strategy Selector** dialog, select **Projection Curve Finishing**.

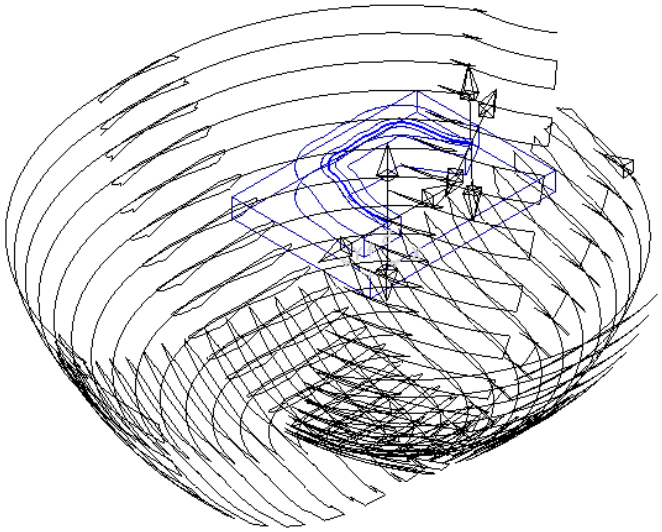
11 Fill in the page of the dialog exactly as shown above and select **Preview** to observe the ***Curve Projection*** setup.

Copyright © Delcam plc

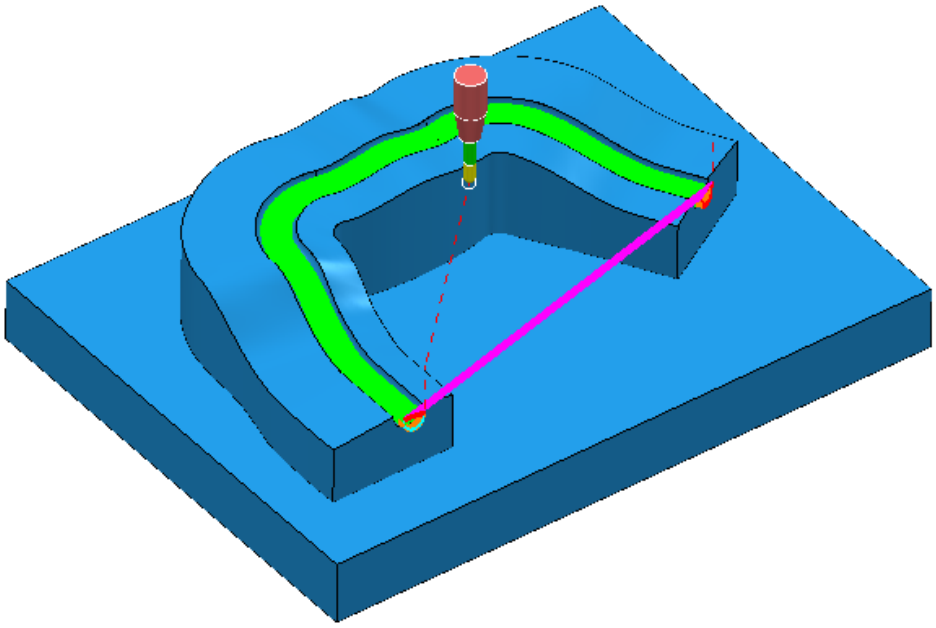

Curve Projection
Toolpath name
BN16-ProjCurve
Workplane
Block
Tool
Machine tool
Limit
Stock engagement
Curve projection
Pattern
Automatic verification
Point distribution
Tool axis
Machine axis control
Rapid move heights
Leads and links
Start point
End point
Feeds and speeds
Notes
User defined settings
Curve projection
Curves definition
1
Style
Linear
Projection
Direction
Outwards
Tolerance
0.1
Thickness
0.0
Angular stepover
2.0
Preview
Draw
Calculate
Queue
OK
Cancel


Curve Projection
Toolpath name
BN16-ProjCurve
Workplane
Block
Tool
Machine tool
Limit
Stock engagement
Curve projection
Pattern
Automatic verification
Point distribution
Tool axis
Machine axis control
Rapid move heights
Leads and links
Start point
End point
Feeds and speeds
Notes
User defined settings
Pattern
Pattern
Style
Linear
Ordering
One way
Sequence
None
Limits
Azimuth angle
Start
-75.0
End
75.0
Parametric pattern limits
Start
0.0
End
1.0
Preview
Draw

Visually the preview strategy is somewhat of a mess, containing crossovers and loops along the path. The actual toolpath however should not suffer the same problem, with the projection distance to the machining areas being a lot shorter (well before loops or crossovers start to appear).



12 **Calculate** the **Projection Curve** toolpath and then **Close** the dialog.



*The **Curve** used as the basis for the strategy will in the majority of cases be created in a CAD package as part of the **Model**. In these cases the imported **wireframe** curve must then be inserted into a **Pattern**.*

13 Select **File** > **Save** to update the stored **Project**.

Copyright © Delcam plc

Swarf Finishing

Swarf Finishing is specifically designed to machine down one or more surfaces where a linear transition exists along the tool alignment direction. It will not work where the transition is concave or convex.

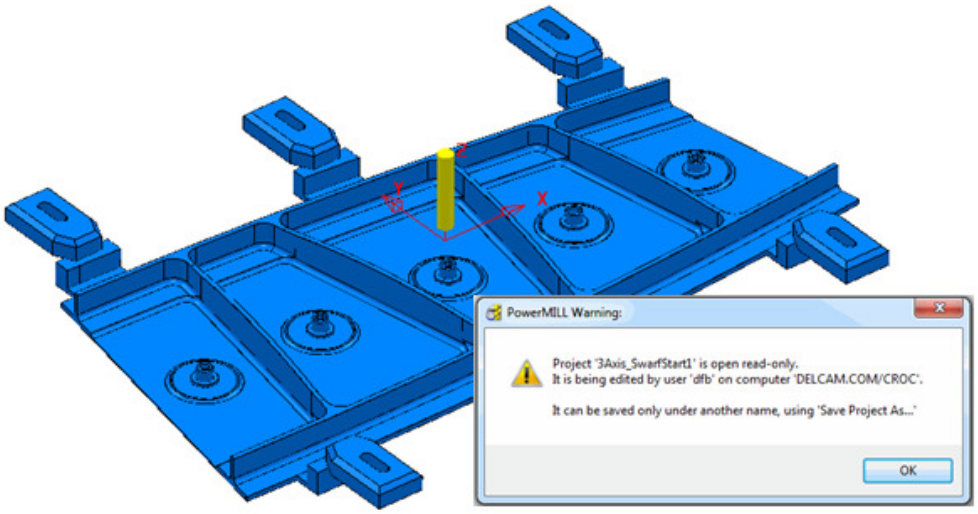
The strategy can be set to create either single (default) or multiple passes, exactly following the upper edge, lower edge, or both (merging).

In the following example the Spar sidewalls will be selected and Swarf machined. This will provide a strategy similar in appearance to Constant Z but applied locally to the selected surfaces and guaranteed to run the final pass exactly along the bottom edge (subject to gouge checking).

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

2 From **File** > **Open Project** select the (read-only) **Project**:

.....\PowerMILL_Data\Projects\3Axis_SwarfStart1



3 Select **OK** in the ***PowerMILL Warning*** dialog.

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

.....\COURSEWORK\PowerMILL-Projects\3AxisSwarfExample

5 In the **PowerMILL *explorer***, activate the **workplane *mc-Datum***.

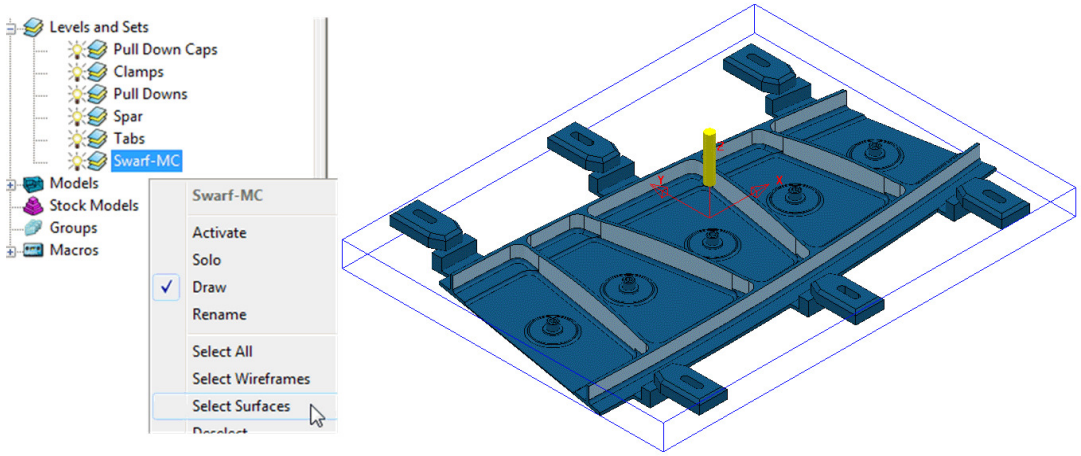
6 In the **PowerMILL *explorer***, activate the Dia 16 End Mill (**EM16**).

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

8 Calculate the **Rapid Move Heights** using the default settings.

9 In the **PowerMILL *explorer***, click the [**+**] adjacent to **Levels** to display all of the named levels.

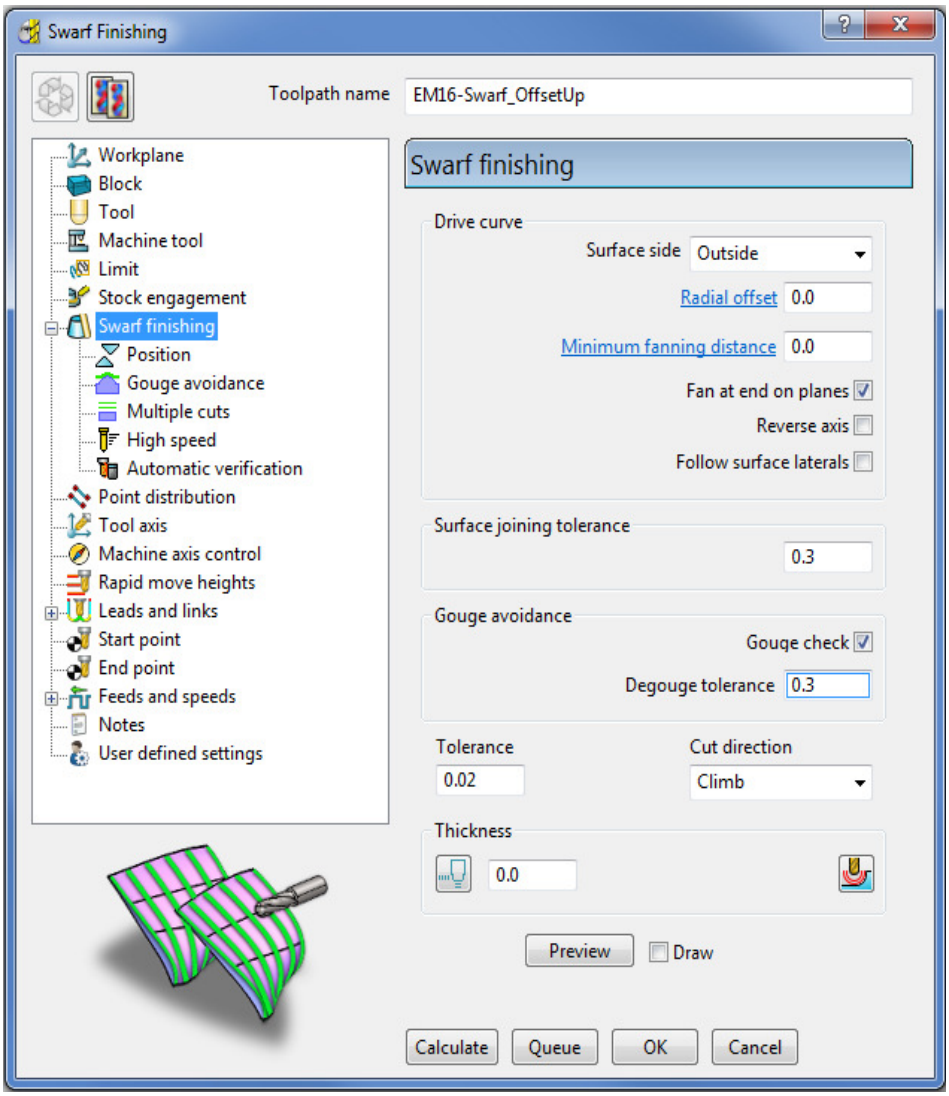
10 Right-click on the **Level *Swarf-MC*** and click **Select Surfaces** from the local menu.



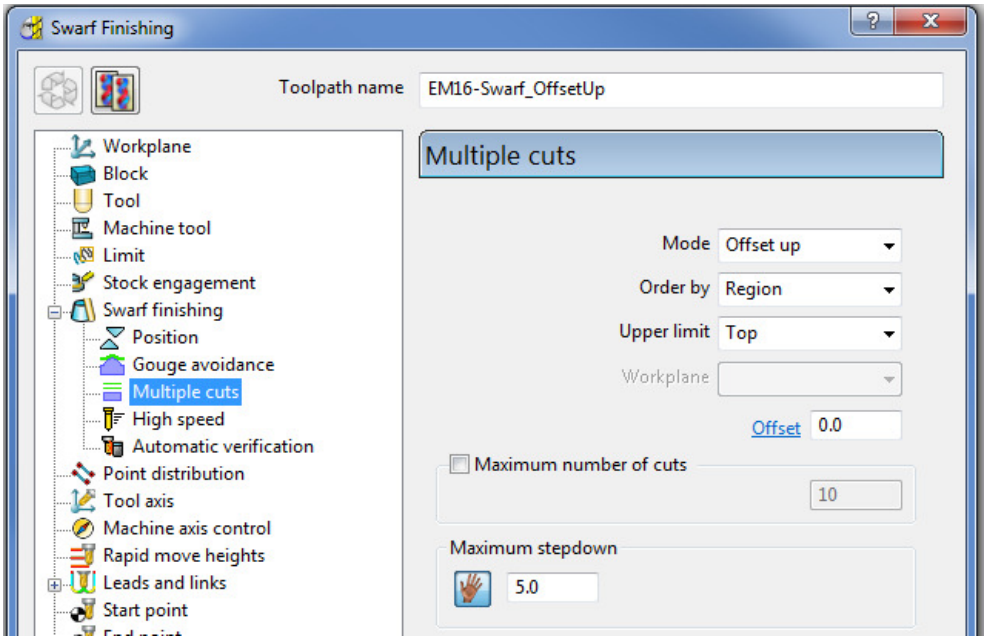
All **Surfaces** on **Level *Swarf-MC*** will be selected (light blue) to be included in the **Swarf** strategy.

11 Select the **Strategy Selector** and select the **Finishing** tab in the dialog.

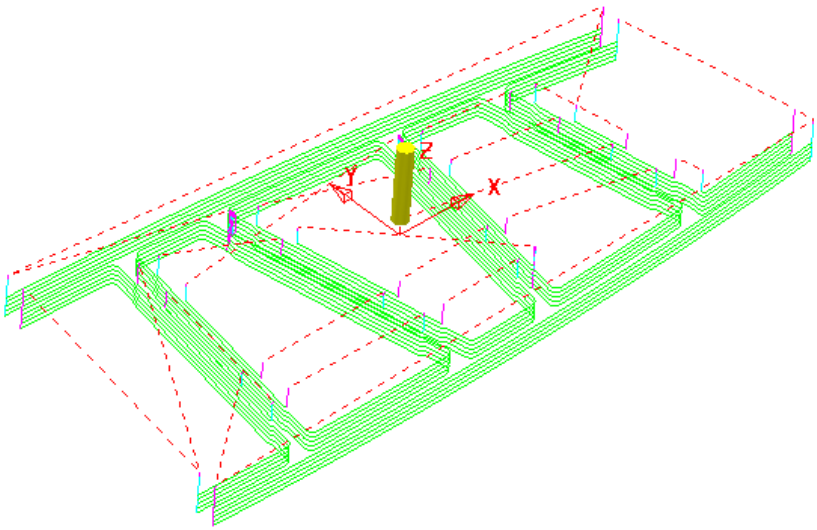
12 Select the **Swarf Finishing** strategy and enter data exactly as shown in the following 2 pages of the dialog.



 Copyright © Delcam plc



13 Select **Calculate** to create the **Swarf Finishing** toolpath and then **Close** the dialog.



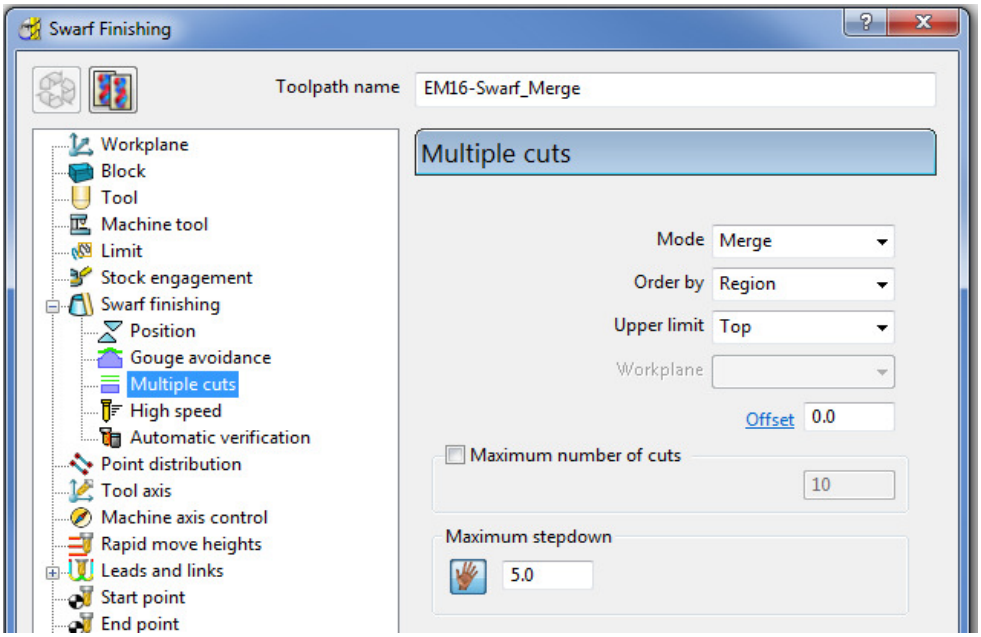
*The tool tracks that machine the base of the **selected surfaces** have progressively offset upwards (by the stepdown value) until fragmentation occurs when the upper edge is exceeded. This can be prevented by using, **Mode Merge** instead of **Offset Up**.*

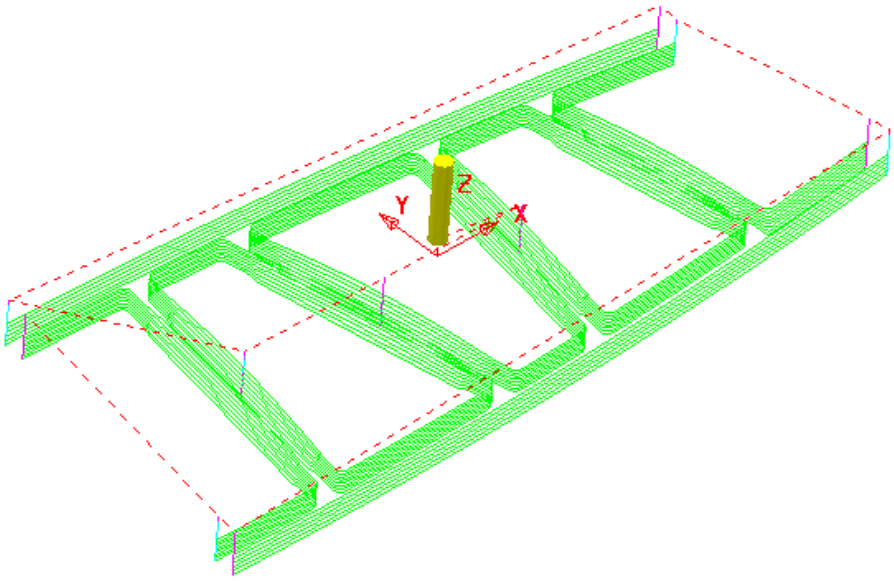
14 In the **PowerMILL *explorer***, right-click the **toolpath *EM16-Swarf_OffsetUp*** and select **Settings**.

15 Select the ***Copy Toolpath*** icon from the dialog .

16 **Rename** the new **Toolpath** as **EM16-Swarf_Merge**.

17 In the page labelled **Multiple Cuts** change the **Mode** to **Merge** and with all **surfaces** on the **Level *Swarf-mc*** selected, **Calculate** the strategy.





__Merge__ will use a variable __stepdown__ to exactly track the __selected surfaces,__ upper and lower edges and minimize the number of lift moves.

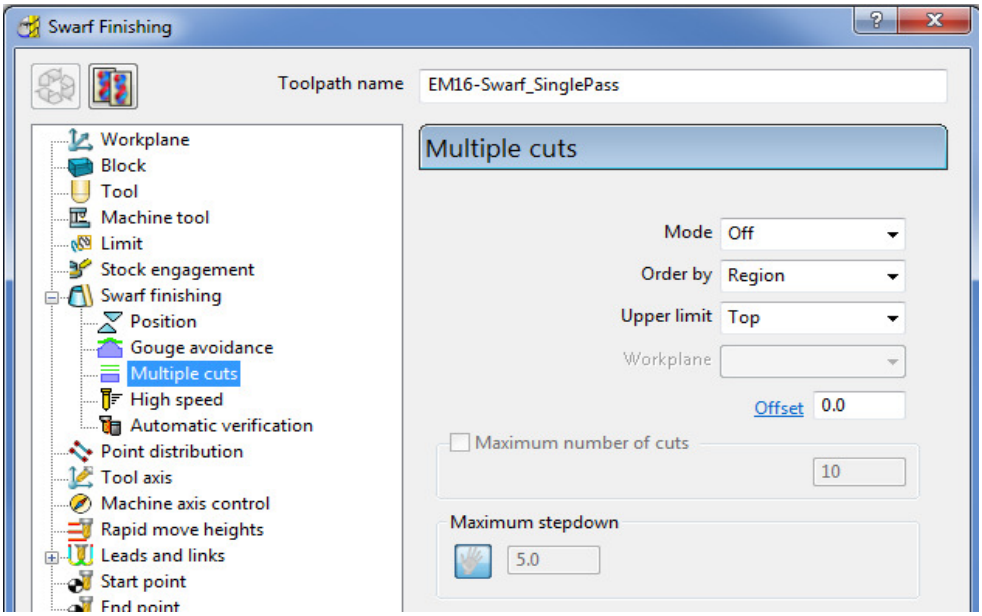
18 Right-click the last **toolpath *EM16-Swarf_Merge***, in the **PowerMILL *explorer*** and select **Settings**.

19 Select the ***Copy Toolpath*** option.

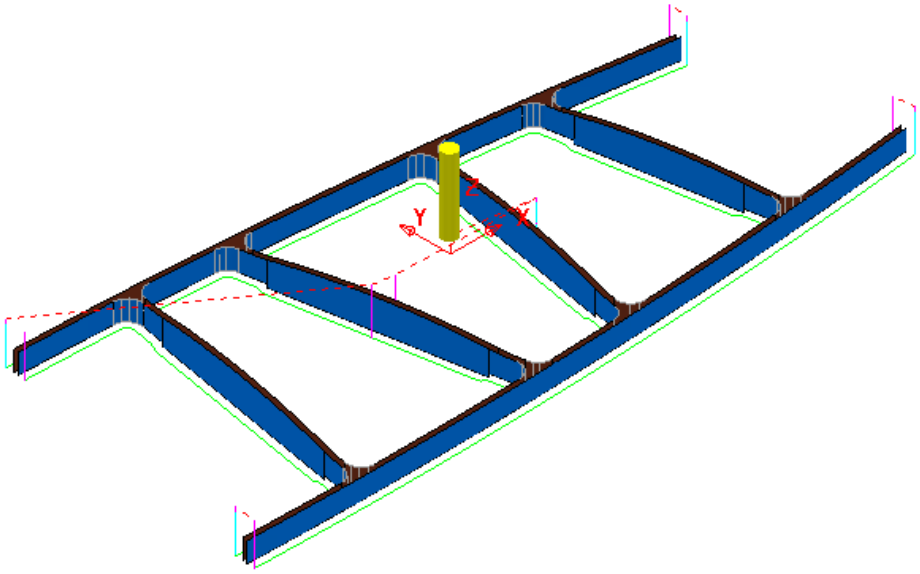
20 **Rename** the new **toolpath** as **EM16-Swarf_SinglePass**.

21 On the **Multiple Cuts** page, change the **Mode** to **Off**.

 Copyright © Delcam plc



22 With all **surfaces** on the **Level *Swarf-mc*** selected, **Calculate** the strategy.



23 Close the **Swarf Finishing** dialog.

24 Select **File** > **Save** to update the stored **Project**.

.....\COURSEWORK\PowerMILL-Projects\3AxisSwarfExample

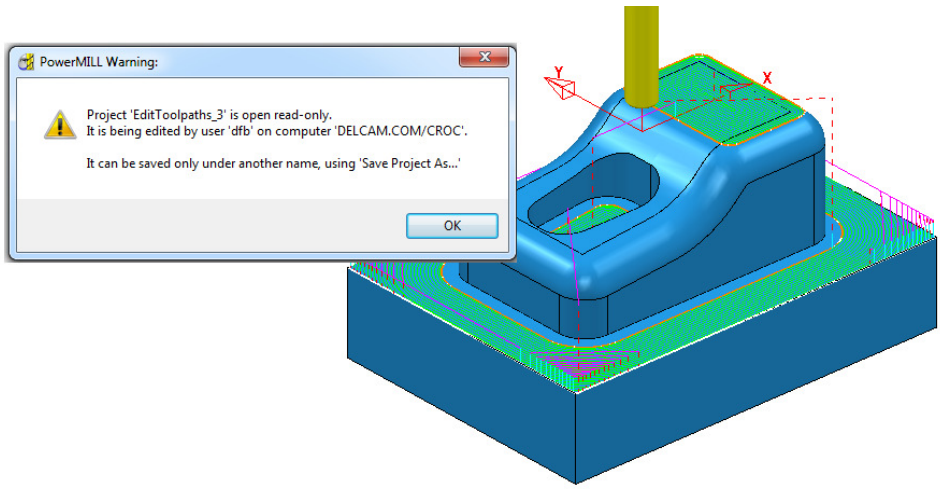
Parametric Offset Finishing

Parametric Offset Finishing creates a strategy that tracks between two individual **Patterns** with a stepover that adjusts to the shape of the ***selected surfaces***. The tool tracks can be created parallel (along) or span between (across) the **Pattern** segments.

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

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

.....\PowerMILL_Data\Projects\EditToolpaths_3

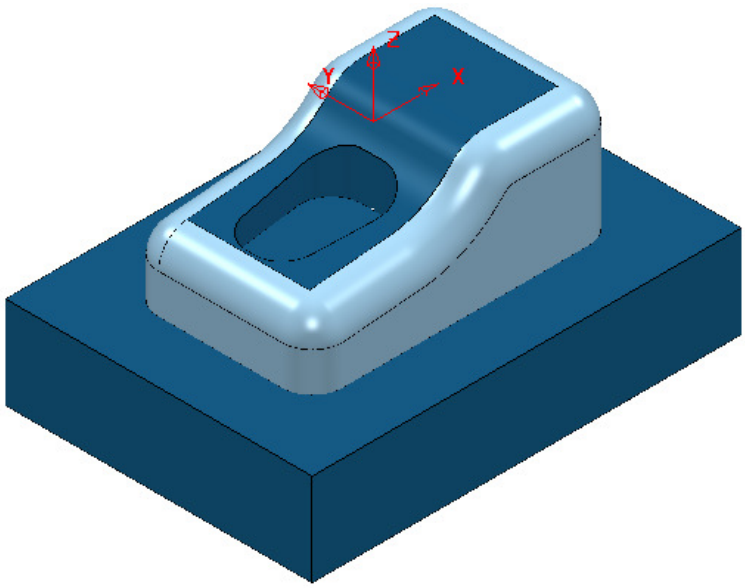


3 Click **OK** in the ***PowerMILL Warning*** dialog and **Save Project As**:

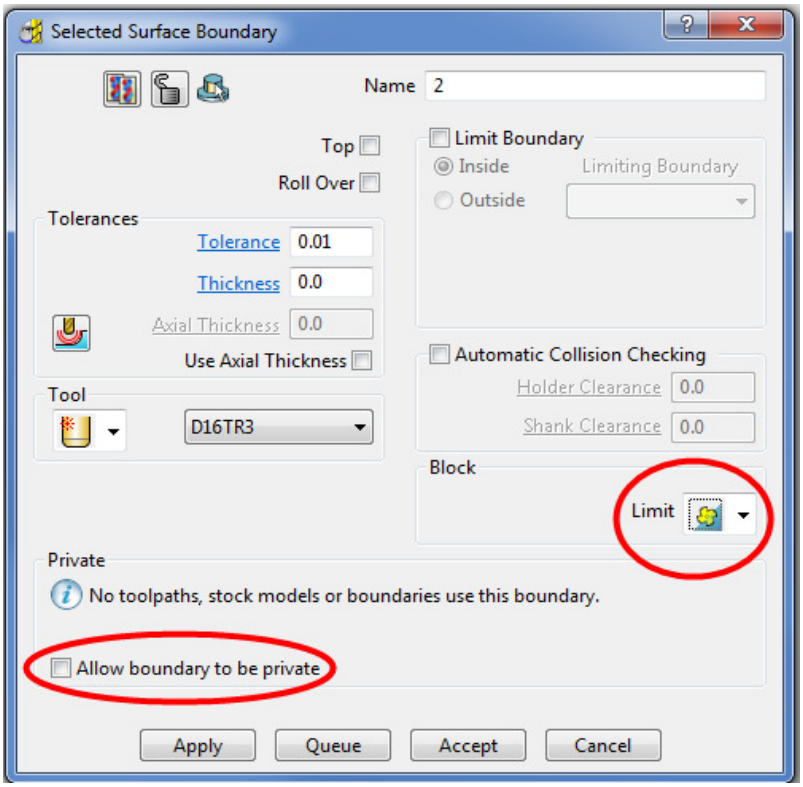
.....\COURSEWORK\PowerMILL-Projects\ParaOffset-ex1

4 Activate the Dia 16 Rad 3 **Tip Radiused** tool named **D16TR3**.

5 With the new **tool *active***, and the **surfaces** shown (below left) selected, create a **Selected Surface Boundary**.

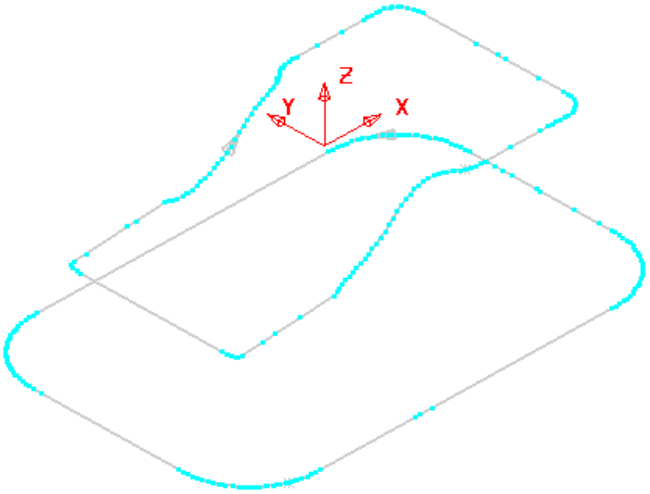


Copyright © Delcam plc



6 Create an empty **Pattern**, named ***Upper*** and **Insert** the upper **Boundary** segment into it.

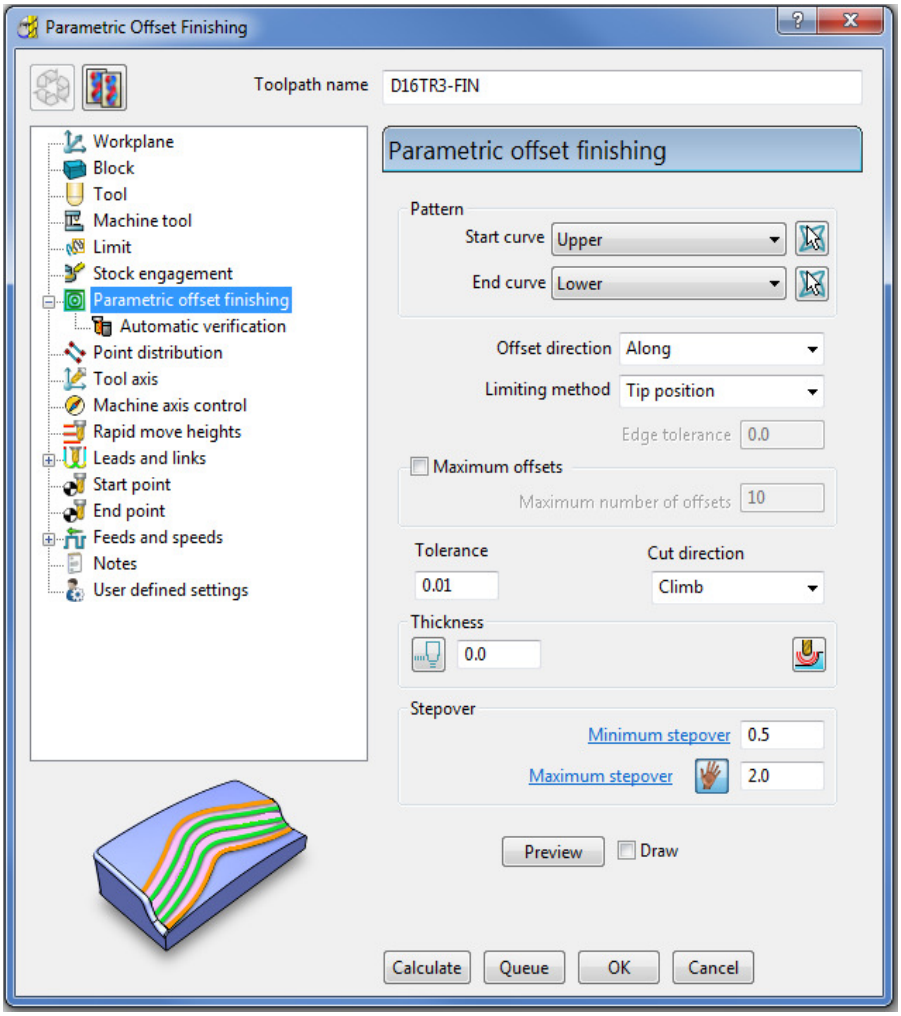
7 Create another empty **Pattern**, named ***Lower*** and **Insert** the lower **Boundary** segment into it.



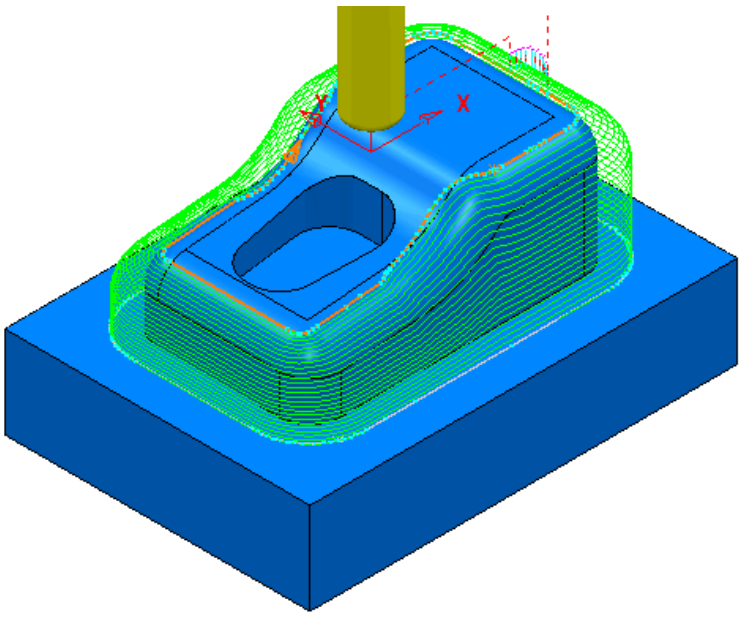
It is not necessary for both of the **Pattern** segments to run in the same direction as the ***Upper*** one will control the toolpath direction.

8 Make sure that no **Boundaries** are **Active**.

9 Open the **Parametric Offset Finishing** dialog and fill in as shown below.



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



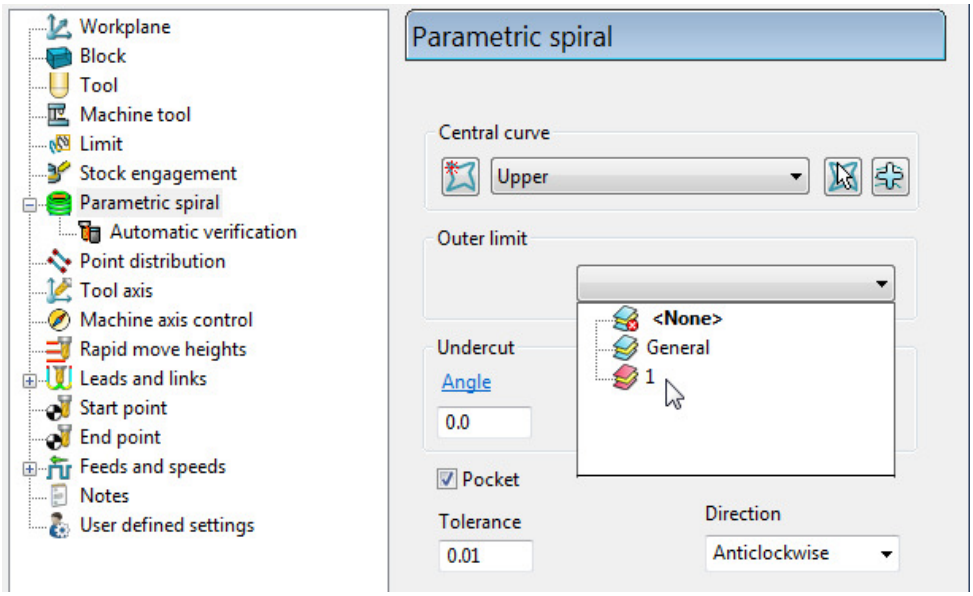
11 Select **File** > **Save** to update the stored **Project**.

12 Do not delete the **Project** as it will be continued in the next section.

Copyright © Delcam plc

Parametric Spiral Machining

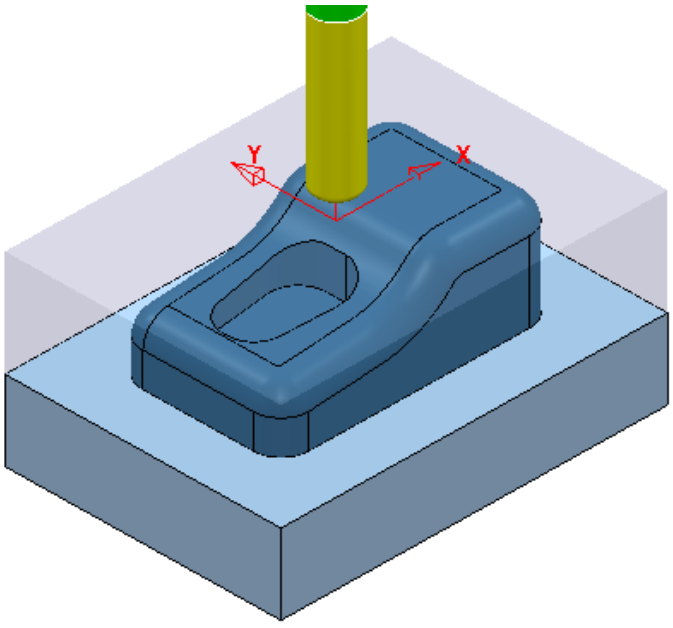
This strategy starts from a **Pattern** and merges a ***spiral toolpath*** towards an **Outer limit** which is either defined by the **Block** or to one or more **surfaces** acquired to a **Level** or **Set**.



The direction of the toolpath created in the **Parametric Spiral Finishing** strategy is contolled by the **Cut Direction** ***Anticlockwise***/***Clockwise***. The direction of the central **Pattern** has no control over the toolpath direction.

If the **Pocket** option is ***ticked*** the resultant **toolpath** will start at the **Outer limit**.

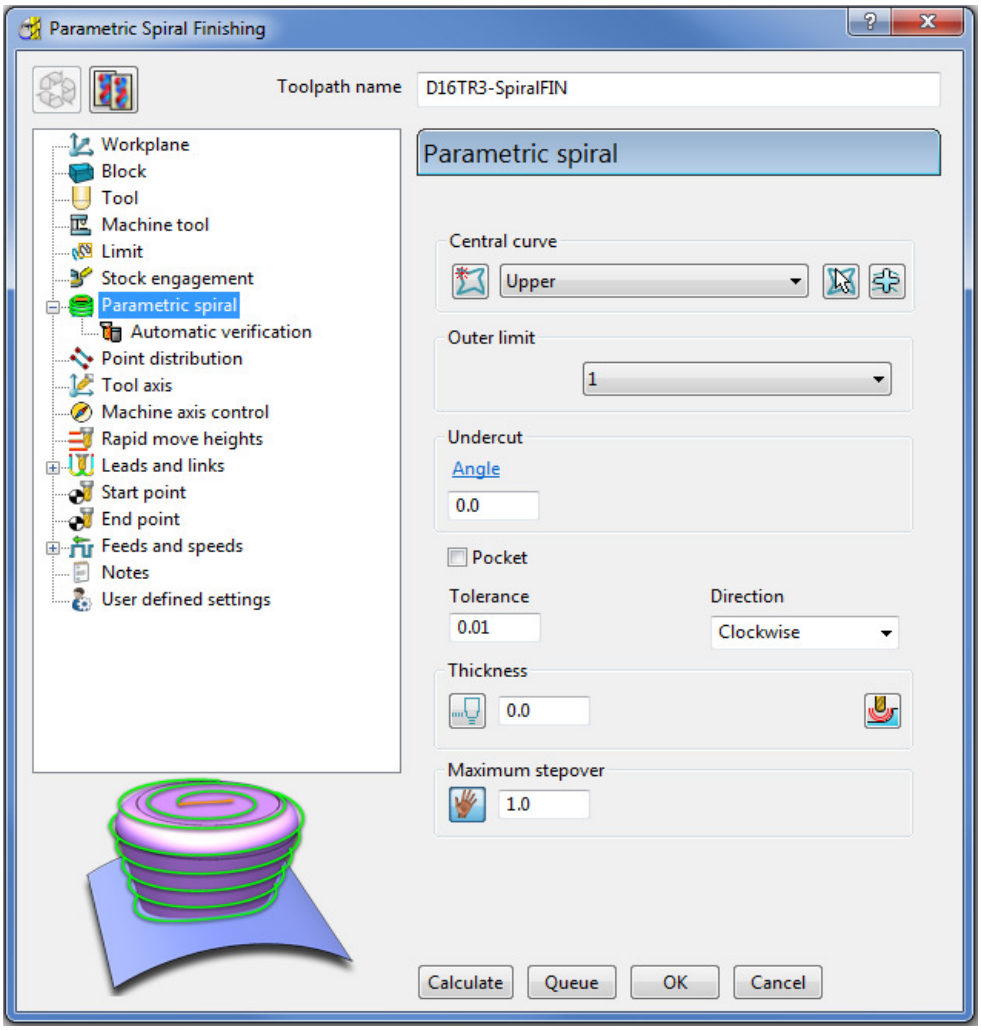
1 In the **PowerMILL** ***explorer*** right mouse click on **Levels and Sets** and from the local menu select **Create Set**.

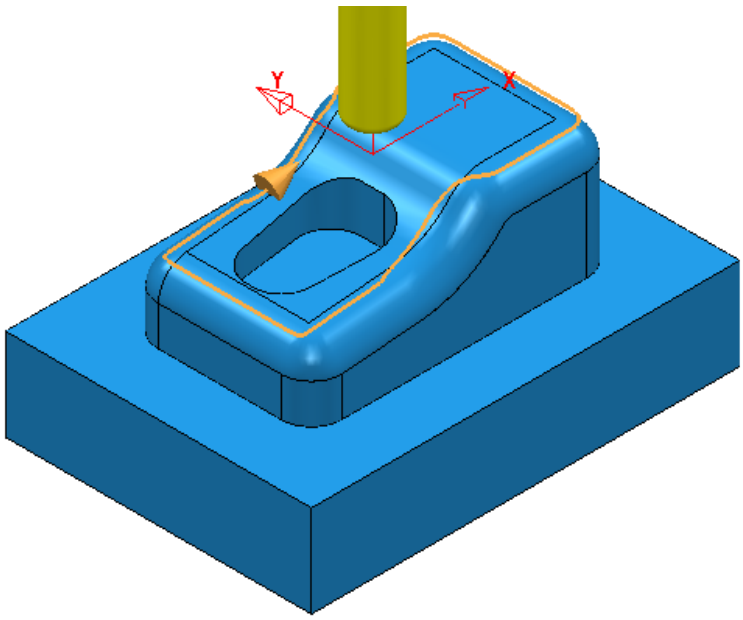


2 Select the base **Surfaces** and from the new **Set** (**1**), local menu select **Acquire Selected Model Geometry**.

3 Make sure that no **Boundaries** are **Active** or displayed.

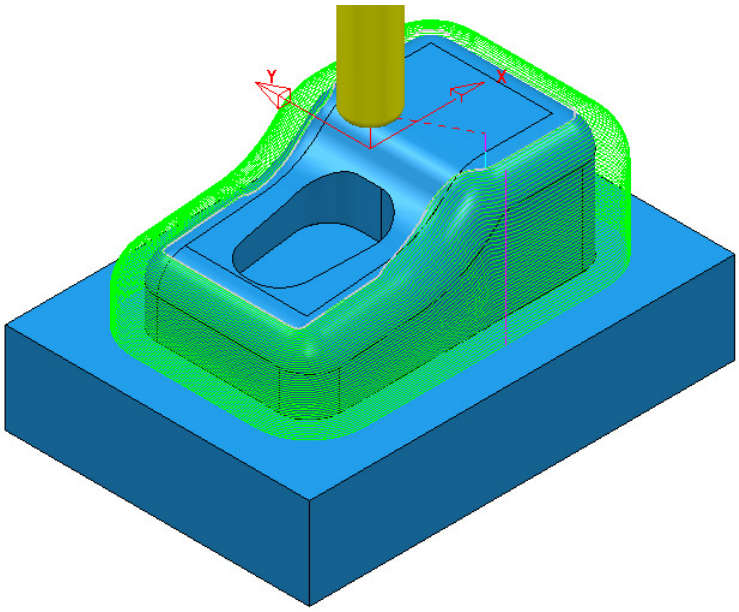
4 Open the **Parametric Spiral Finishing** dialog and fill in as shown below.





5 Select **Calculate** to create the **toolpath** then **Close** the dialog.

Copyright © Delcam plc



6 Select **File** > **Save** to update the stored **Project**.

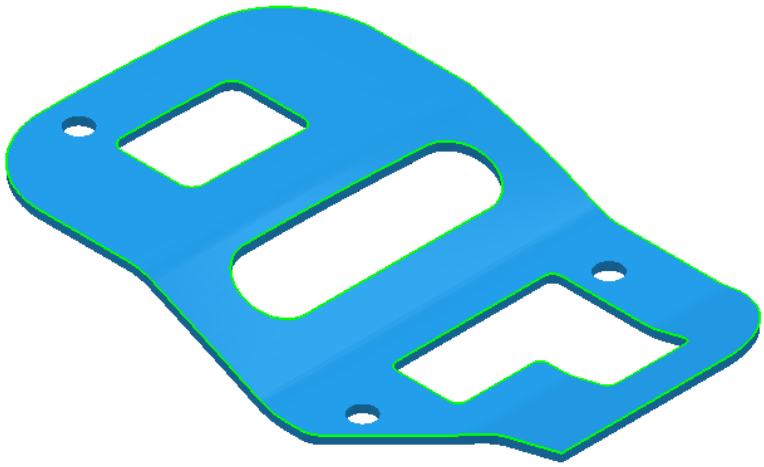
Wireframe Profile Machining

Wireframe Profile machining is designed to machine to the left or right of the selected ***curves*** of a 3D Wireframe. The following example is a pressing that requires the outer form and recesses to be finish profiled. As the main imported model is in STL (triangle) format it is incompatible with **Boundary** options that operate on ***selected surfaces***. It is also impossible to create the necessary 3D Wireframe **curves** direct from the **STL model** in **PowerMILL**. To resolve, the stored 3D Curves will be imported into a pattern to be applied in the **Wireframe Profile Machining** strategy.

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

2 Select **File** and **Import** the **Model**:

.....\PowerMILL_Data\Models\3D-Curve-TriModel.stl



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

.....\COURSEWORK\PowerMILL-Projects\3D-WireframeProfile

4 Create a Dia 5 End Mill named **EM5**.

5 Create an empty **Pattern**, right-click on it, and from the local menu, select **Insert** > **File**:

.....\PowerMILL_Data\Models\3D-Curve-Wire.dxf

6 **Rename** the **Pattern** as **3Dwire**.

7 With the left mouse click on the **model** to select it.

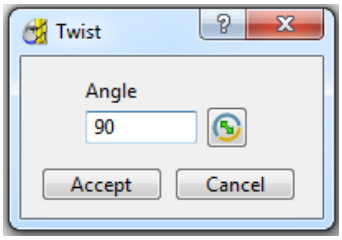
8 In the **PowerMILL *explorer***, right-click on **Workplanes** and from the local menu, select **Create** and **Orientate Workplane** - **Workplane at Bottom of Selection**.

9 **Rename** the **Workplane** as **MC-Datum**.

10 Right-click on the **Workplane** (**MC-Datum**) and from the local menu, select **Workplane Editor** to open the **Workplane Editing** toolbar.

11 Click **Twist about Z** to open the following dialog.

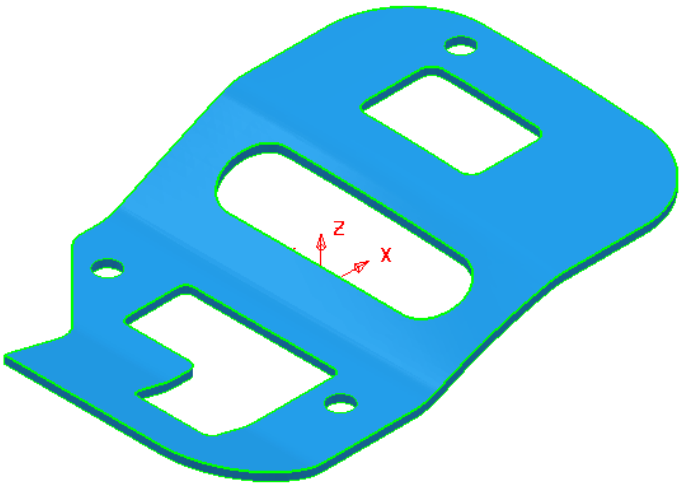
Copyright © Delcam plc



12 Enter the value **90** and click **Accept**.

13 Click on the **Workplane Editor** toolbar to accept the changes and **Close**.

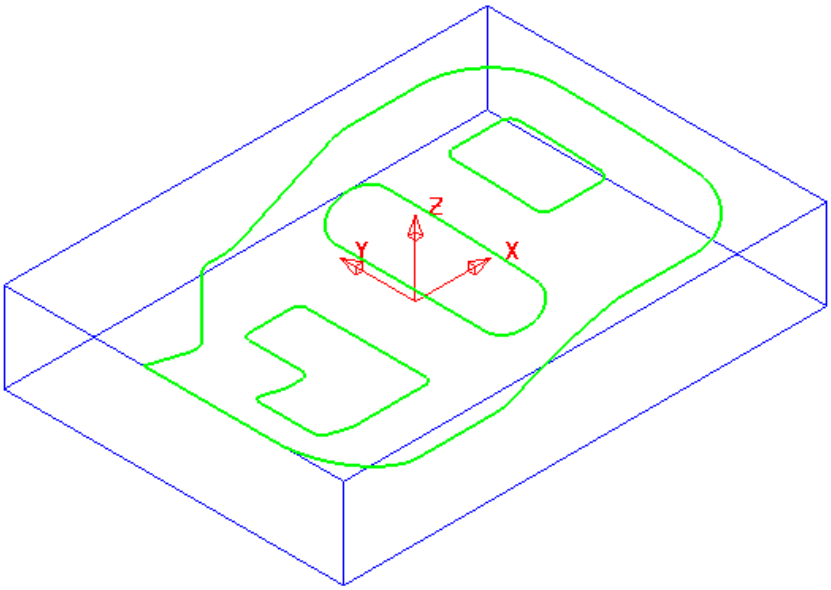
14 **Activate** the **workplane** (**MC-Datum**) and select an **ISO 1** view.



*The changes will not be physically active until the **Workplane Editor** toolbar has been **closed**.*

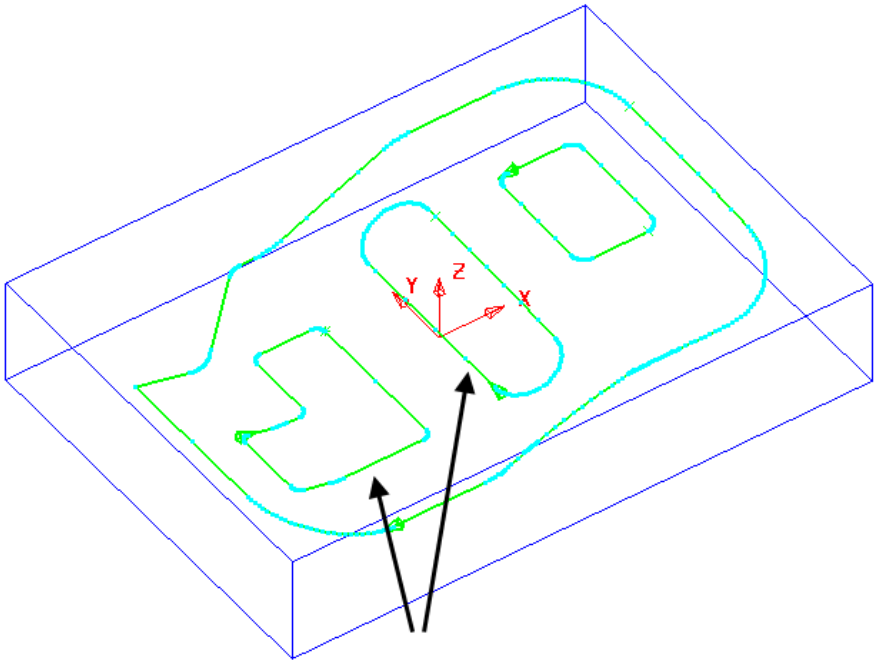
15 **Calculate** a **Block** using ***Defined by*** **Box** and ***Type*** **Model**.

16 ***Lock*** the **Maximum Z** value only and then input an **expansion** value of **10** before re-selecting **Calculate**.



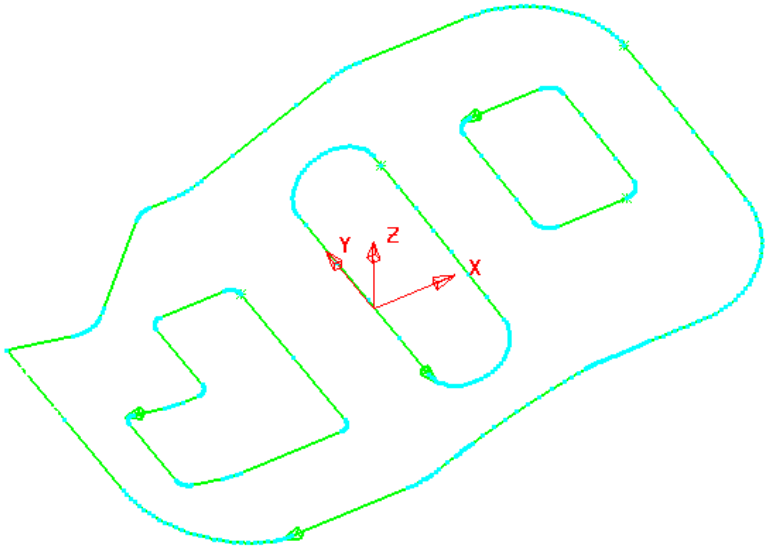
The final task prior to machining is to check the directions of the **Pattern** segments and to **reverse** any that will cause the tool to machine on the wrong side of a profile.

17 Right-click on the **Pattern** (***3Dwire***) and from the local menu, select **Instrument** to display directional arrows on the individual segments.



Two of the segments (indicated above) must be reversed for the **Wireframe Profile Machining** to ***Climb Mill*** to the left of the direction while remaining inside the aperture forms on the **STL model**.

18 Select the 2 **segments** (arrowed on the previous diagram) and from the local **Pattern** menu apply **Edit** > **Reverse Selected**.

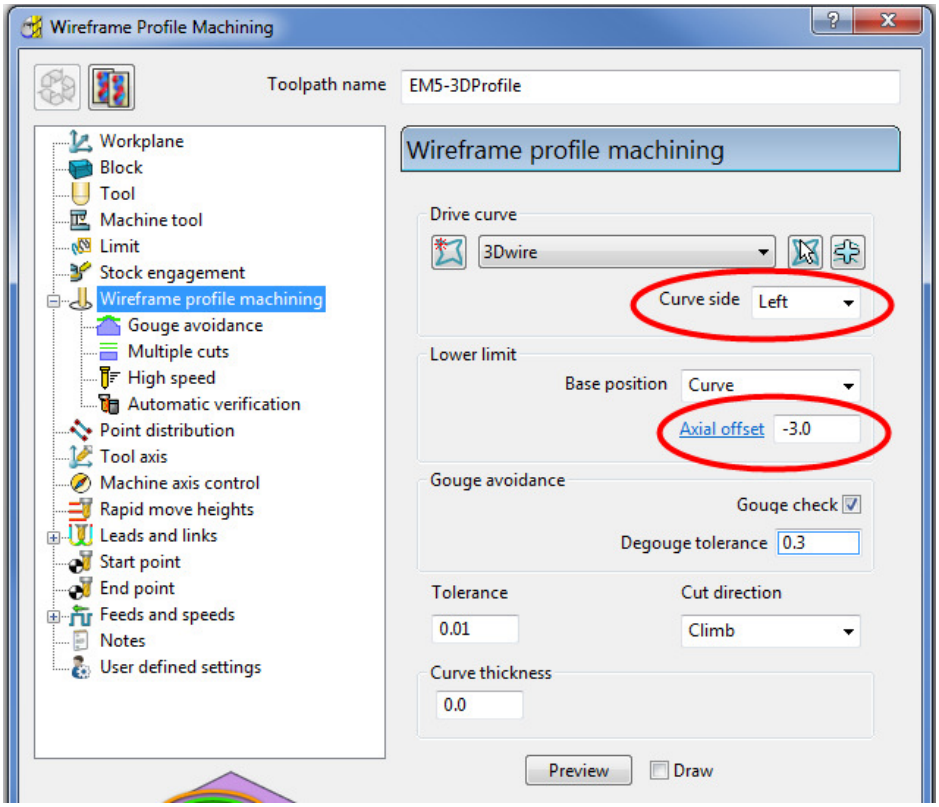


19 **Calculate** the **Rapid Move Heights** using the default settings.

20 Select the **Strategy Selector** and select the **Finishing** tab in the dialog.

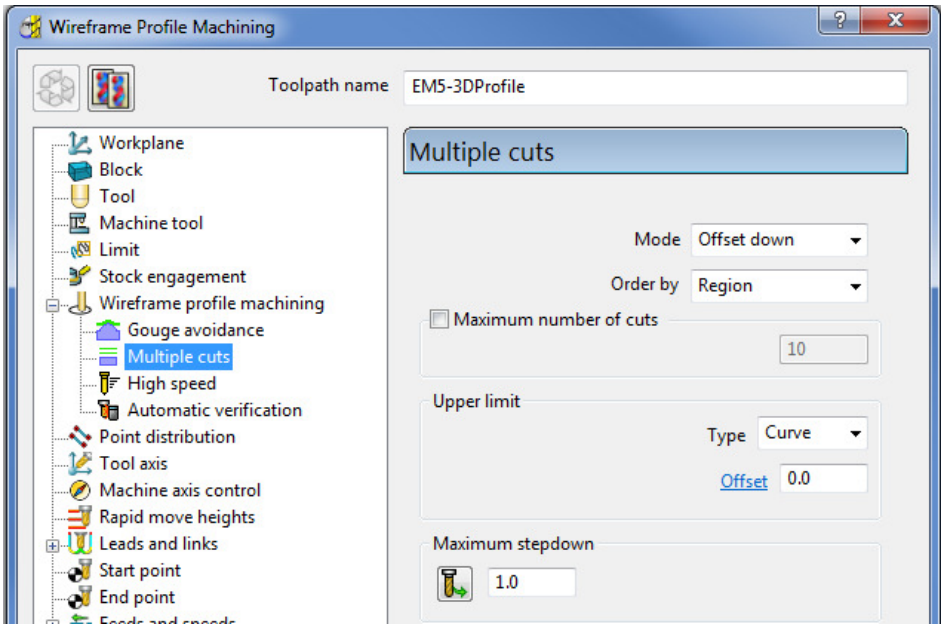
Copyright © Delcam plc

21 Select the **Wireframe Profile Machining** strategy and enter data exactly as shown on the following 2 pages of the dialog.

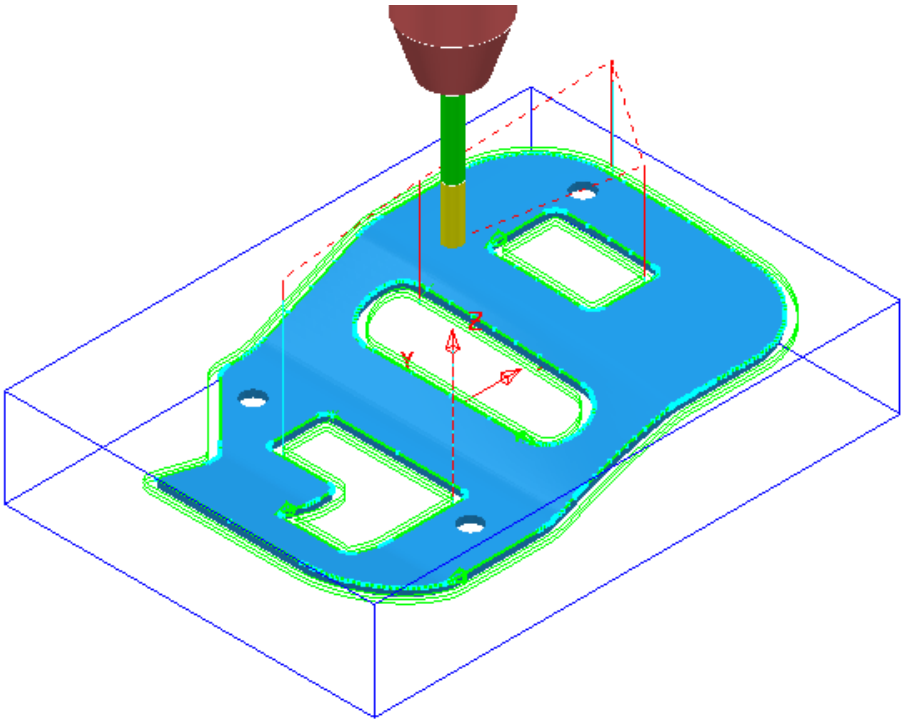


Curve side controls which side of the **Pattern** segment direction the tool operates.

Axial offset controls the depth of machining below the **wireframe curves**.



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



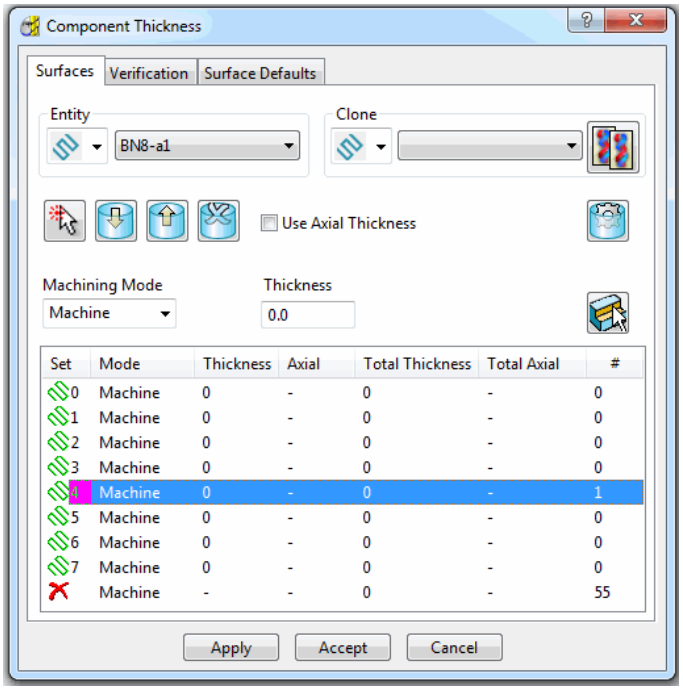
The **Wireframe Profile Machining** performs the profiling of the Pattern segments in 3D.

23 Select **File** > **Save** to update the stored **Project**.

 Copyright © Delcam plc

16. Component Thickness

The **Component Thickness** dialog enables the user to apply varying **Thickness** and **Machining Mode** to different parts of the component. This can either be pre-defined in the **Thickness Preference** dialog or applied independently within an individual toolpath.



The **Component Thickness** tab allows individual **surfaces** or **solid faces** to be assigned with different **thickness** values (***Axial*** and ***Radial***) within a **toolpath**.

Other important features in this dialog include the **Machining Mode** options, **Ignore** or **Collision** to control ***machining status*** on selected parts of the component. This is available from the pull-down list consisting of three options:

- **Machine** (default) is used to define specific areas of the component to be machined with a different **Thickness** to that that in the main strategy dialog.
- **Collision** is used to define areas of the component to be completely avoided by the machining process (such as a clamp).
- **Ignore** is used to define areas of the component to be made invisible to the machining process so that the toolpath will pass through them (such as a blanking surface used to prevent toolpath moves into a hole or pocket during earlier machining strategies).
- The **Component Thickness** options are also available in **Boundary** creation options that include an active **Tool** as part of the calculation.

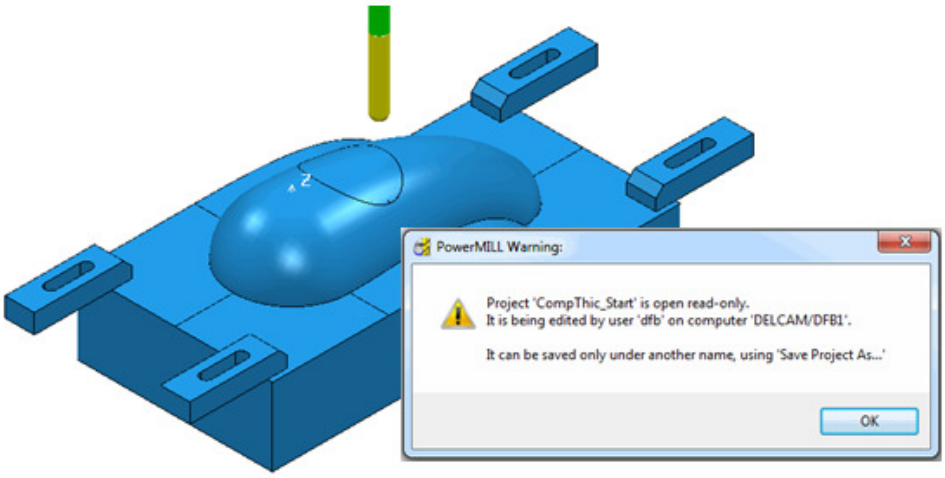
Copyright © Delcam plc

Component Thickness Example

1 Select **File** > **Delete all** and **Tools** > **Reset Forms**.

2 From **File** > **Open Project** select the read-only **Project**:

.....\PowerMILL-Data\Projects\CompThic_Start



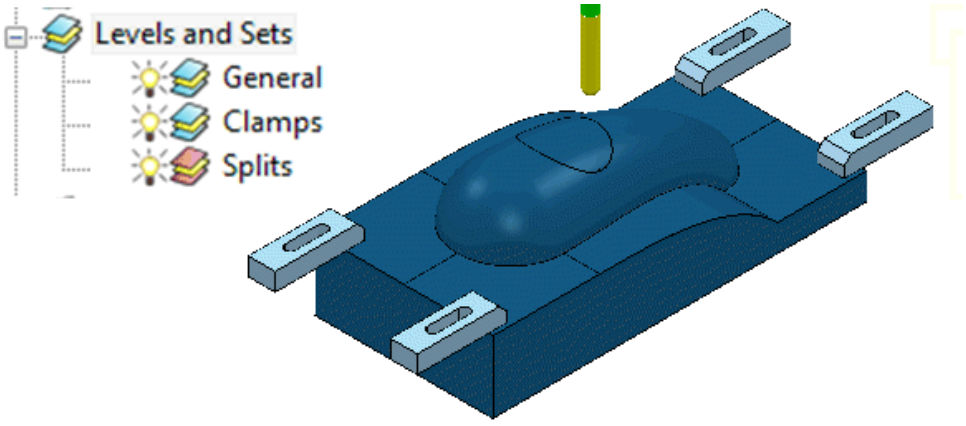
The **Project** contains an unprocessed Interleaved **Constant Z** machining strategy along with a Dia 8 Ball Nosed tool and all essential parameter settings.

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

.....\COURSEWORK\PowerMILL-Projects\CompThickness-1

*To make selection easier the **Clamps** will be transferred to a new **Level** (**Clamps**) while the **split surfaces** will be copied to a new **Set** (**Splits**).*

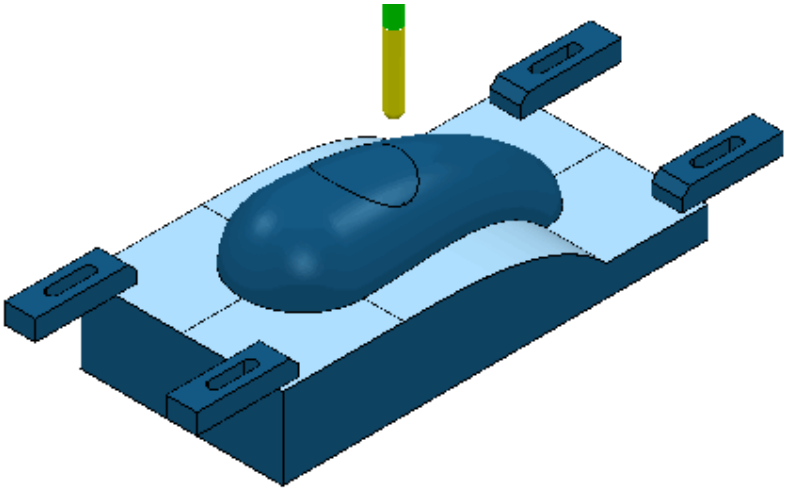
4 In the **PowerMILL *explorer***, create a new **Level** and rename it as **Clamps** and then create a new **Set** and rename it as **Split**.



5 With the clamp surfaces selected, right-click on the **Level** named **Clamps** in the **PowerMILL *explorer*** and from the local menu select, **Acquire Selected Model Geometry**.

*It will be easier to **select** and **Blank** (**Ctrl J**) all the other **surfaces** first before simply dragging a box over all of the remaining **clamp surfaces**.*

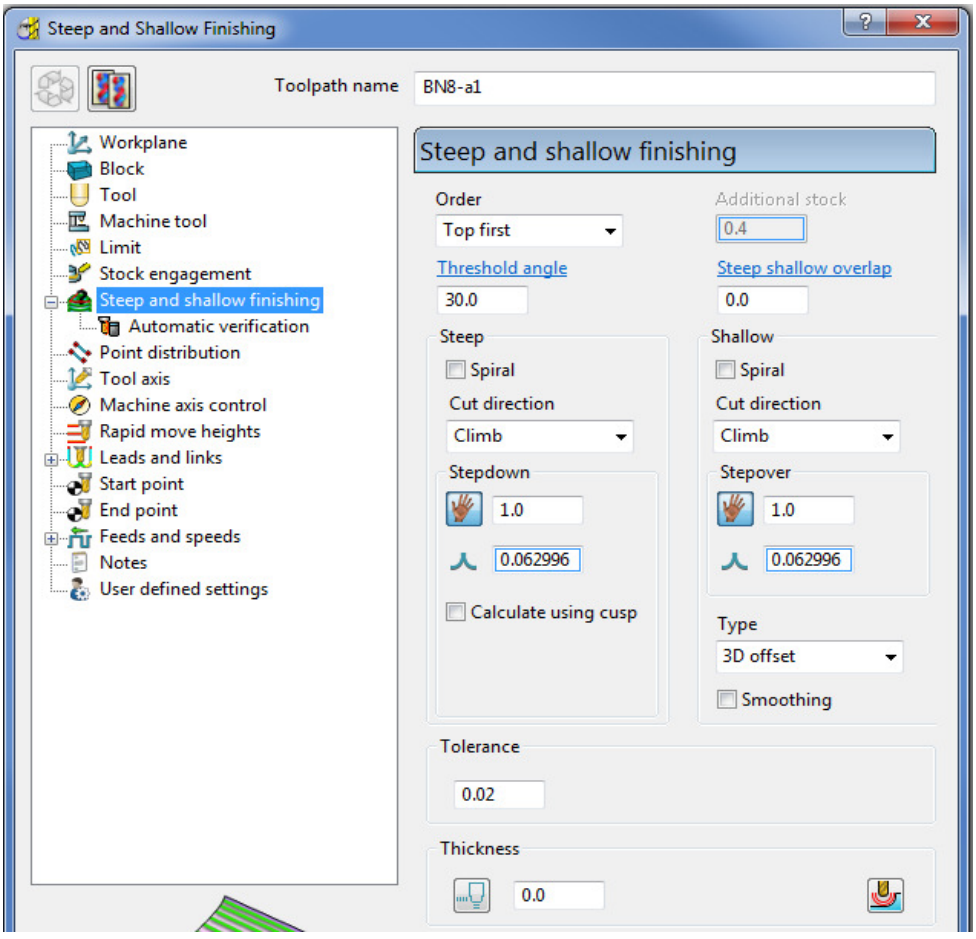
Copyright © Delcam plc



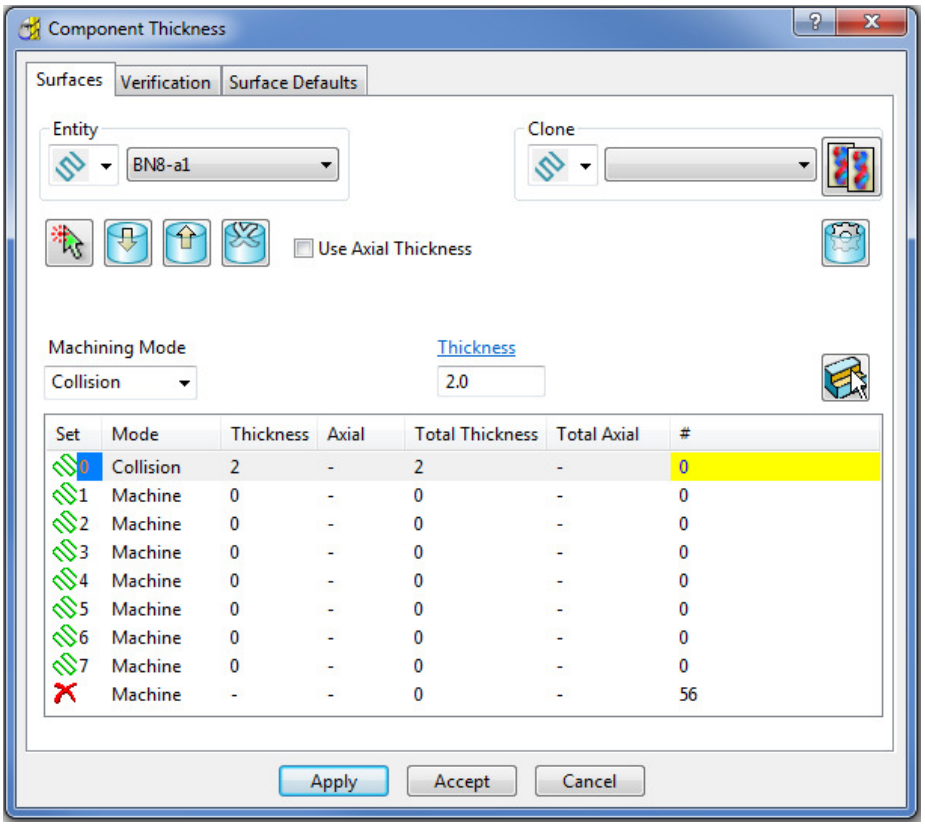
6 With the split surfaces selected, right-click on the **Set** named **Splits** in the **PowerMILL *explorer*** and from the local menu select, **Acquire Selected Model Geometry**.

7 In the **PowerMILL *explorer*** switch off the light bulb next to the **Set** named **Splits** (This is to prevent them still being displayed if the light bulb for the **Level *General*** is still switched on).

8 In the **PowerMILL *explorer***, right-click on the unprocessed, **Toolpath *BN8-a1*** and in the local menu, select **Settings** to open the **Steep and Shallow Finishing** dialog.



9 Select **Component Thickness** to open the following form.

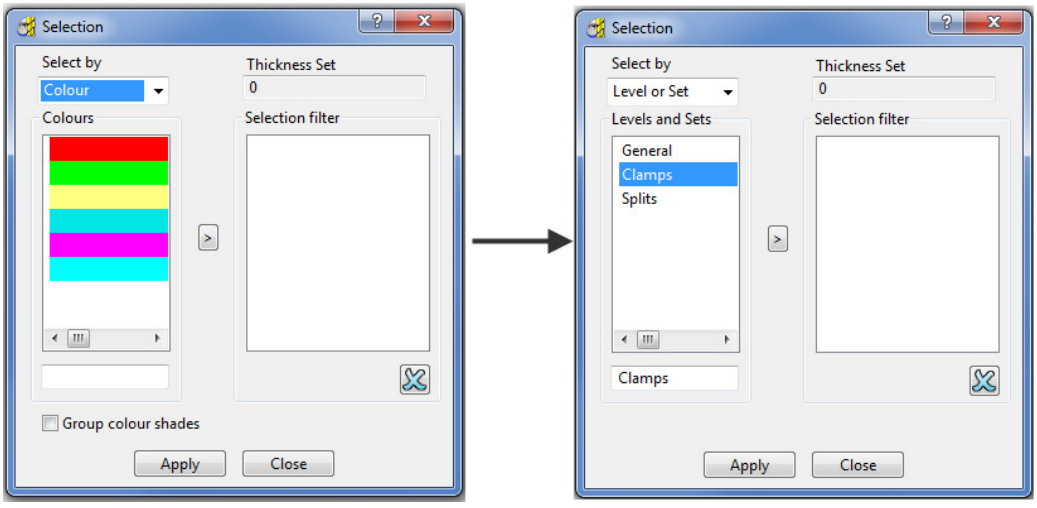


With the **Machining Mode** set to **Collision**, a **Thickness** value of **2** will be applied. At this safe distance the active tool will either ***retract*** or ***profile*** around the clamps.

10 Select the 1st row of the **Component Thickness** table and a square, colour coded symbol (in this case blue) will appear at the start of the row.

11 With the 1st row still selected, change the **Machining Mode** to **Collision** with a **Thickness** value of **2**.

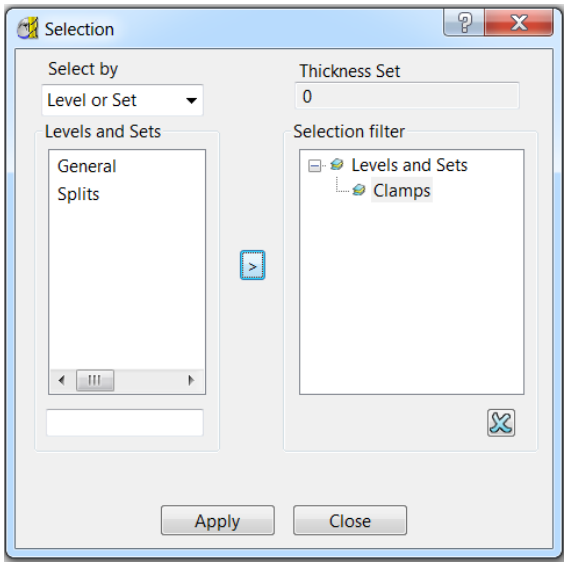
12 Select **Smart Selection** to open the following dialog:



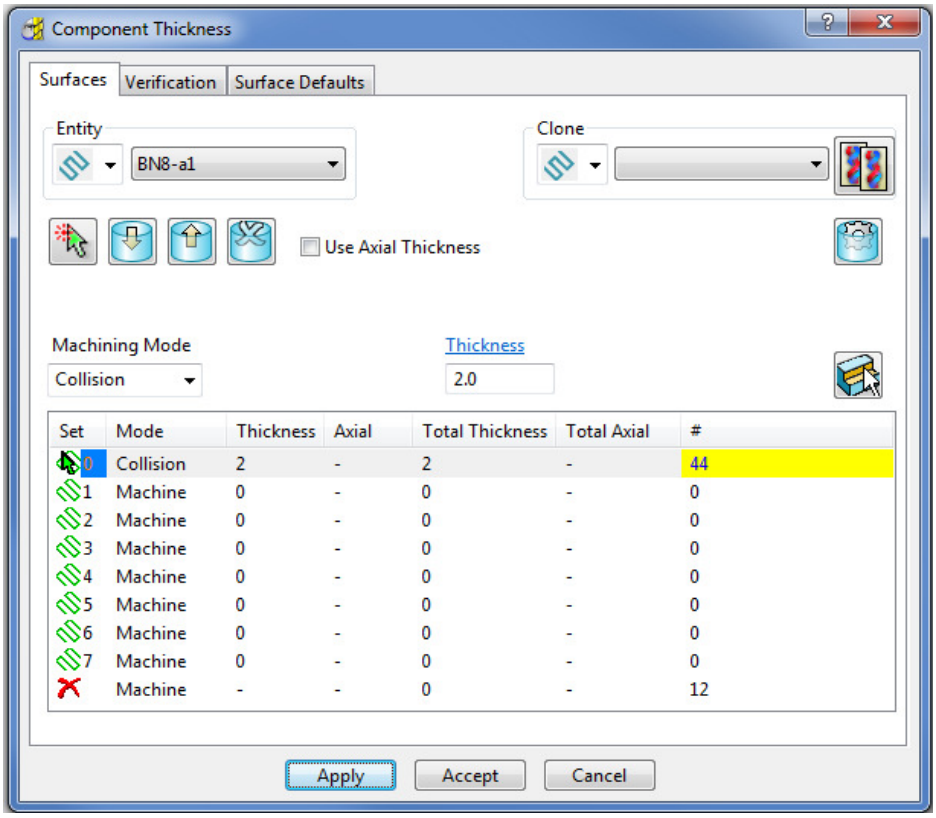
13 Switch **Select by** from **Colour** to **Level or Set**.

14 With **Clamps** selected in the **Levels or Sets** box click to acquire the clamp surfaces to the 1st row of the main dialog, and then **Close** the ***Selection*** dialog.

Copyright © Delcam plc

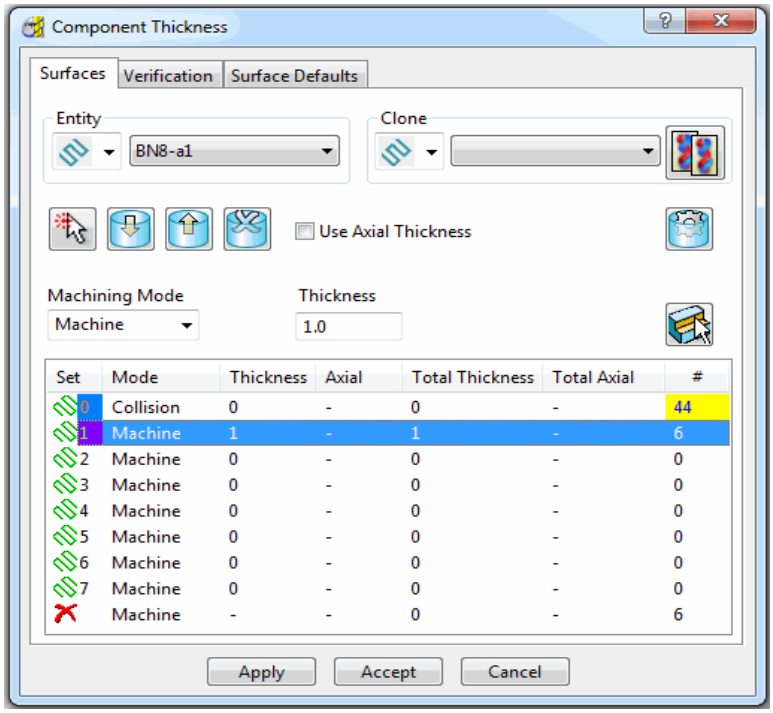


The 44 ***Clamp Surfaces*** are acquired to the 1st row.



15 **Apply** the **Component Thickness** dialog to update the settings.

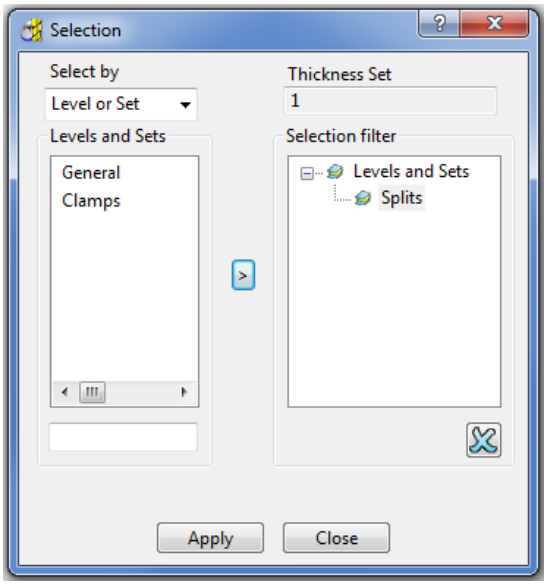
16 Select the 2nd row and a square, colour coded symbol (in this case purple) will appear at the start of the row.



The **Thickness** value of **1.0** will control how much stock is left on the acquired ***split surfaces*** for this strategy.

*This is in addition to a **Thickness** value specified in the main machining strategy dialog. In the above **Component Thickness** dialog the sum of these is displayed in the **Total Thickness** column.*

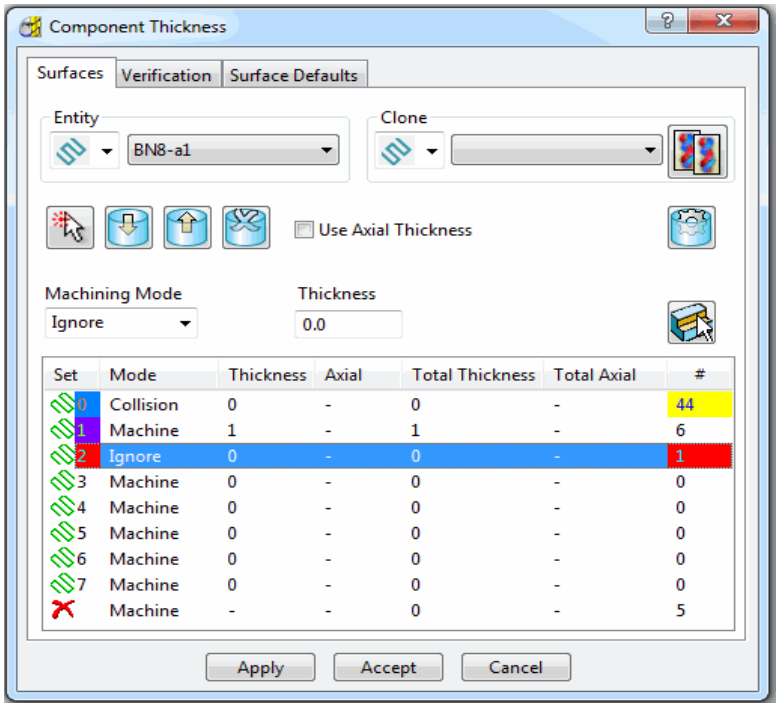
17 With the 2nd row still selected, keep the **Machining Mode** as **Machine** but with a **Thickness** of **1**.



18 In **Smart Selection** select **Splits** in the **Levels or Sets** box and click the arrow icon to assign them in the second row of the main dialog.

Copyright © Delcam plc

19 Select the 3rd row and a square, colour coded symbol (in this case red) will appear at the start of the row.

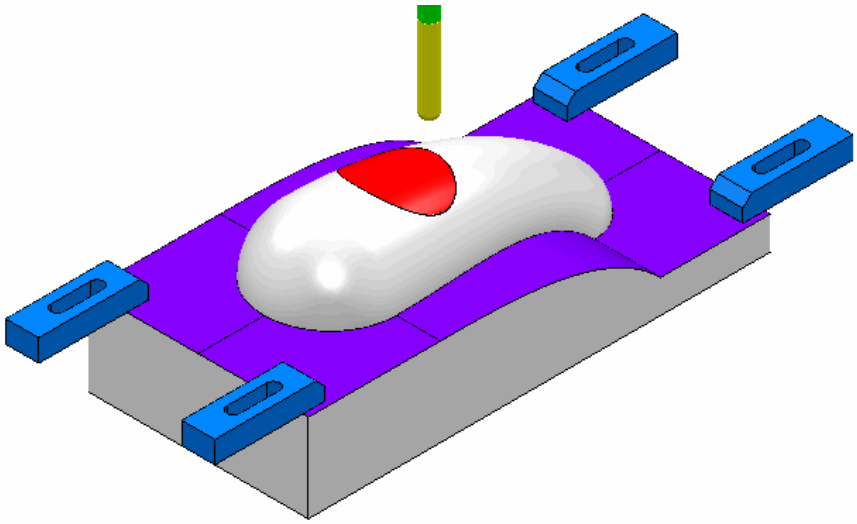


The ***cap surface*** must be ignored during the creation of this toolpath.

20 With the 3rd row still selected, change the **Machining Mode** to **Ignore** with a **Thickness** value of **0**.

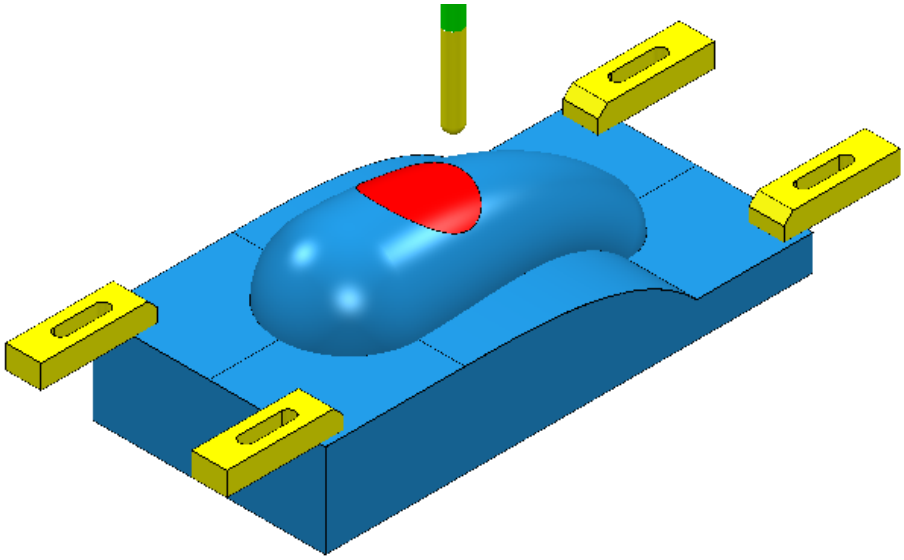
21 Use the left mouse key to manually select the ***cap surface*** and click on **Acquire Components** to include it in the 3rd row.

22 **Apply** the **Component Thickness** dialog.



While the **Component Thickness** form is open, **Toolpath Thickness Shade** mode will be active. Acquired **surfaces** are shaded in the same colour as the row.

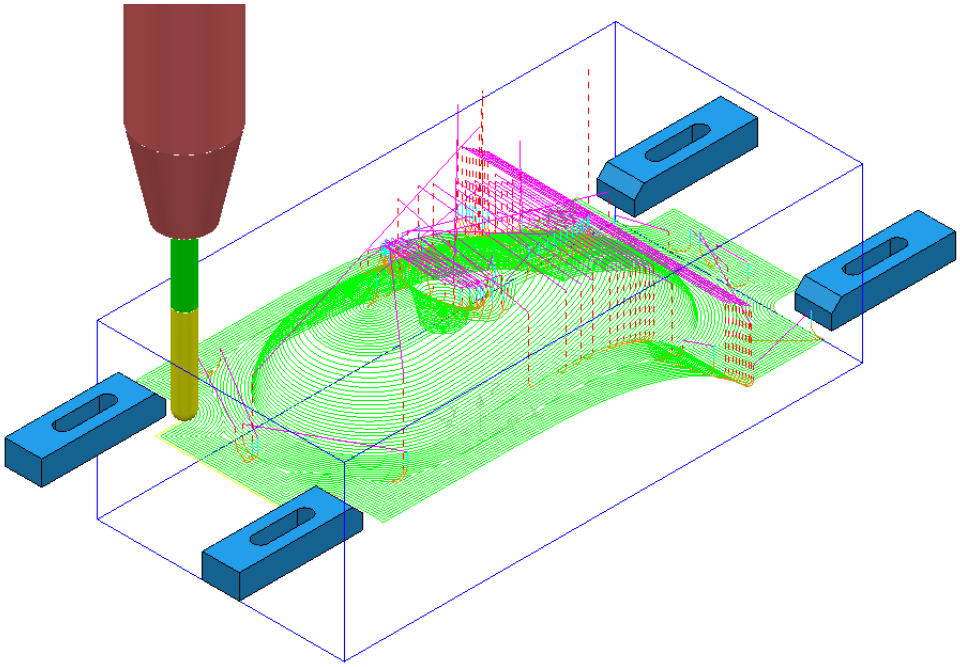
23 In the **Viewing** toolbar select the **Toolpath Machining Mode Shade**.



This time, relevant parts of the **model** are colour coded in ***Red*** (**Ignore**) or ***Yellow*** (**Collision**). Where applicable, these colour codes also appear at the end of a row.

24 **Accept** the **Component Thickness** dialog.

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



26 Select **File** > **Save** to update the stored **Project**.

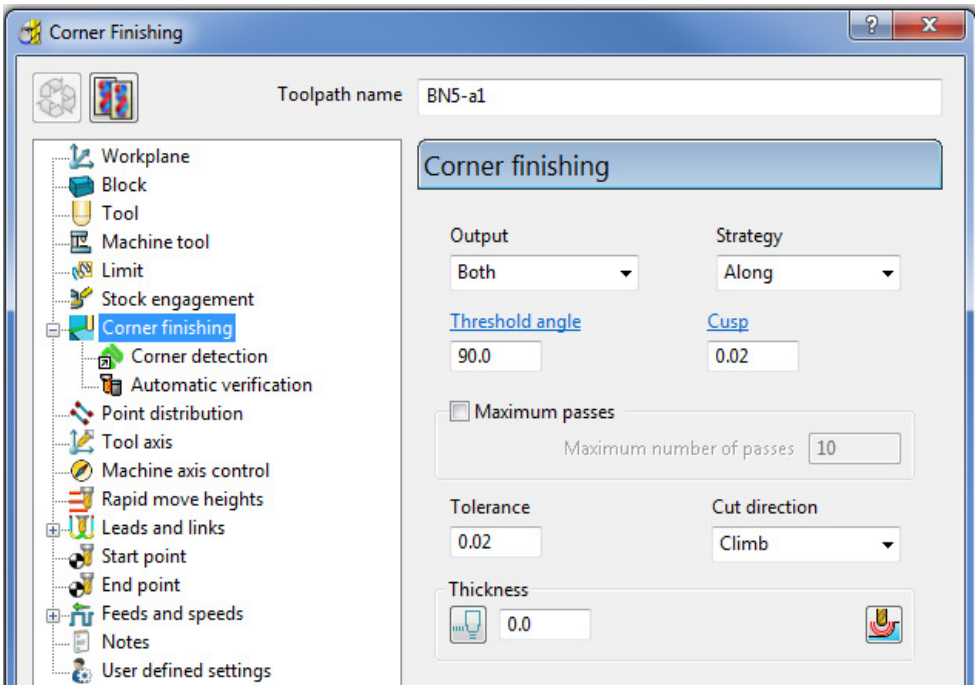
27 Do not close the **Project** as it will be continued in the next section.

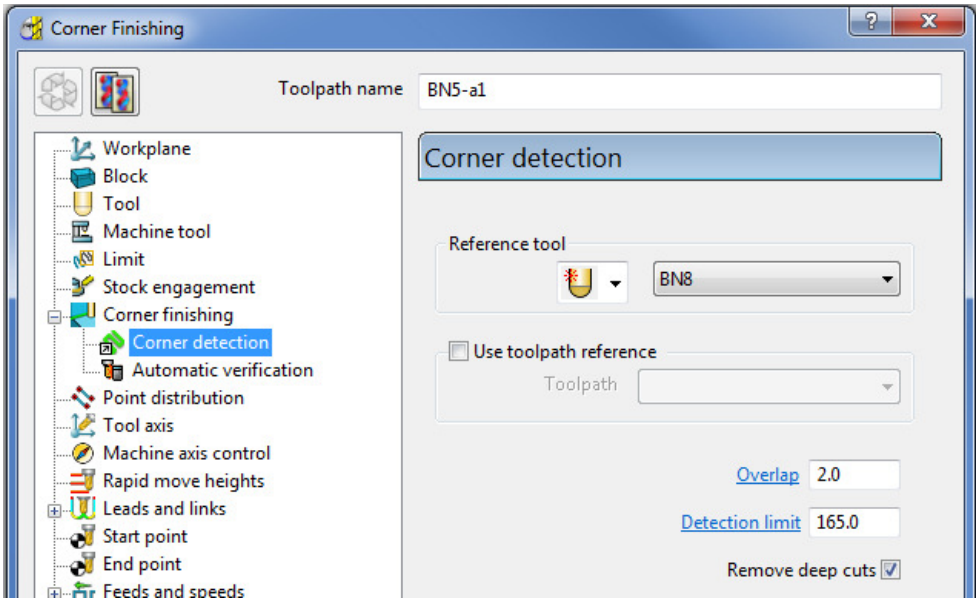
Copyright © Delcam plc

Cloning the Component Thickness settings from an existing Toolpath

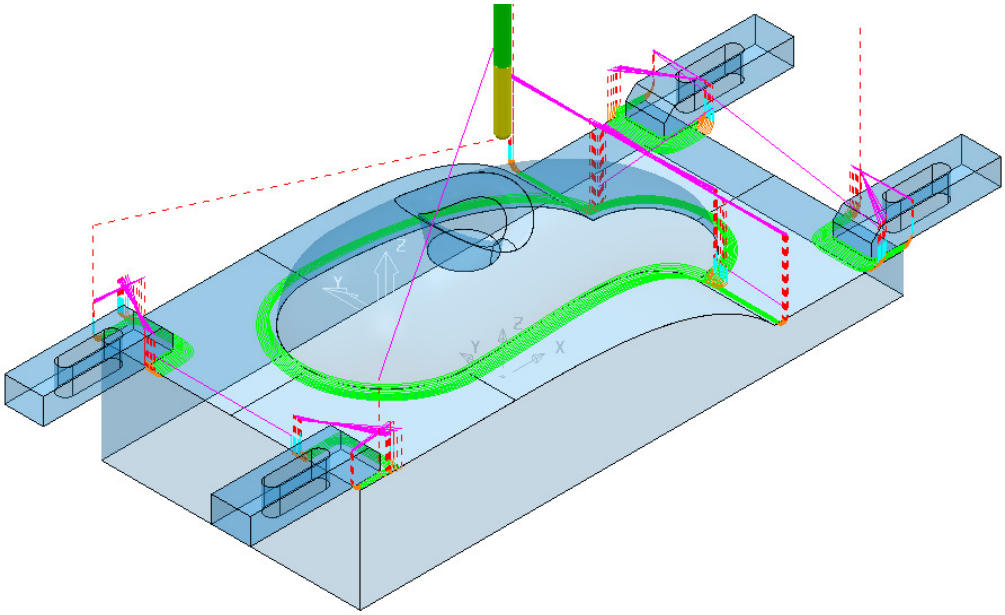
A **Corner Finishing** strategy ***BN5-a1*** will now be created using 'cloned' **Component Thickness** settings as used in ***BN8-a1***.

1 From the **Toolpath Strategies** select the **Finishing** option, **Corner Finishing**, and input settings exactly as shown below:





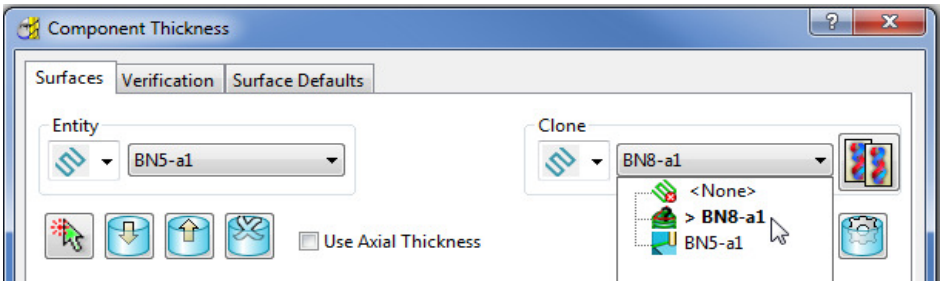
2 Select **Calculate** and have a look at the resultant **toolpath** in the graphics area.



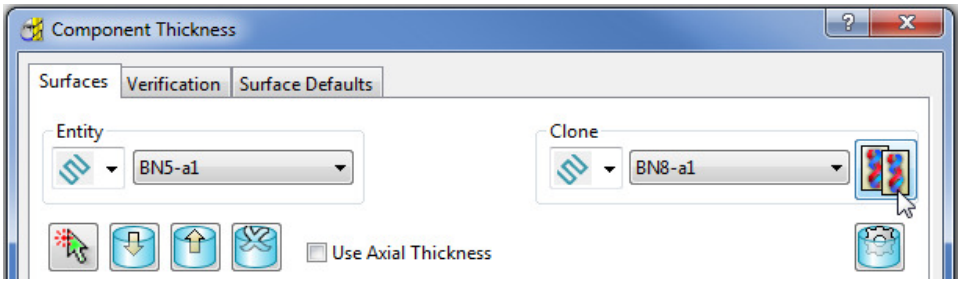
For the moment the **Corner Finishing** strategy is unable to access the base of the central pocket and has also produced unwanted ***tool tracks*** around the front edge of the clamps. This will be rectified by ***cloning*** the **Component Thickness** from the toolpath ***BN8-a1***.

3 Back in the **Corner finishing** page, select **Component Thickness** .

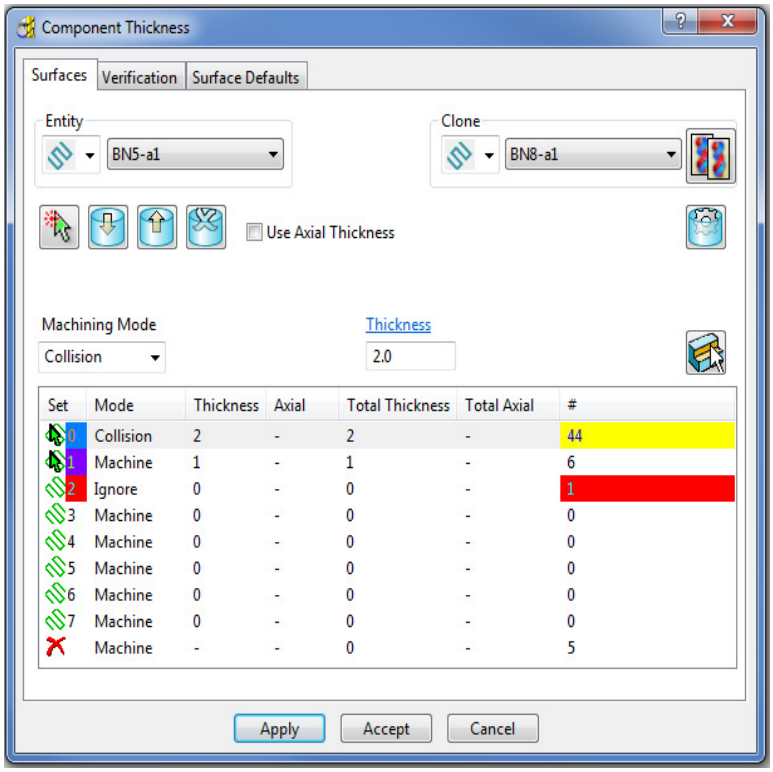
4 In the **Clone** area of the form use the ***downward chevron*** to display and then select the ***toolpath Bn8-a1***.



5 Select the **Copy Thickness Data** icon to ***Clone*** the **Component Thickness** settings into the dialog.



 Copyright © Delcam plc

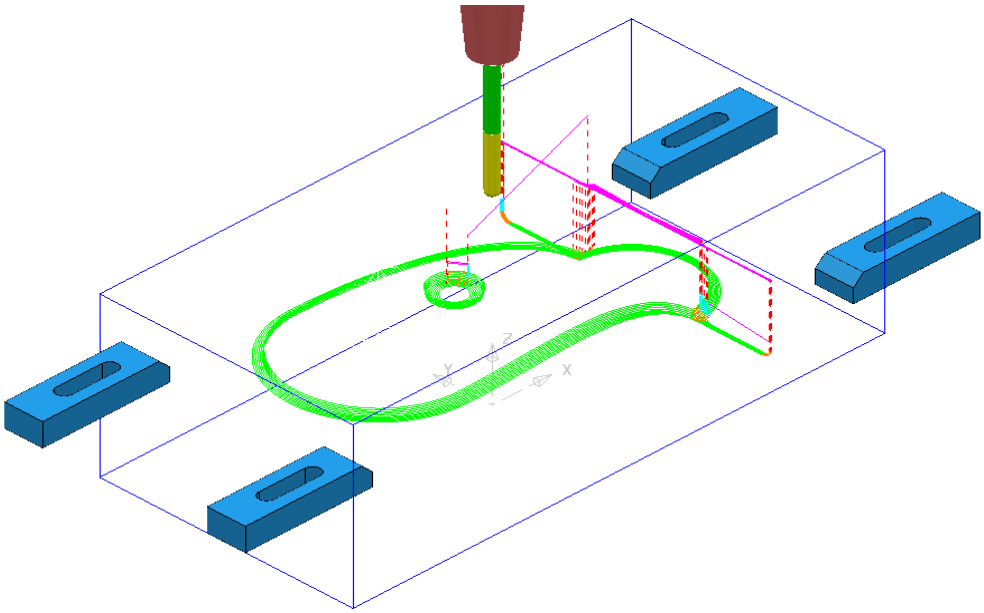


The **Component Thickness** settings originally used in **BN8-a1** are now ***cloned*** into the strategy dialog for ***BN5-a1***.

*The process is not complete until the **Apply** button has been clicked.*

6 Select **Apply** followed by **Accept** to fully update the **Component Thickness** settings in the strategy ***BN5-a1***.

7 **Calculate** the **Corner Finishing** strategy and then Close the dialog.



8 Select **File** > **Save Project**

This will update:-

.....\COURSEWORK\PowerMILL-Projects\CompThickness-1

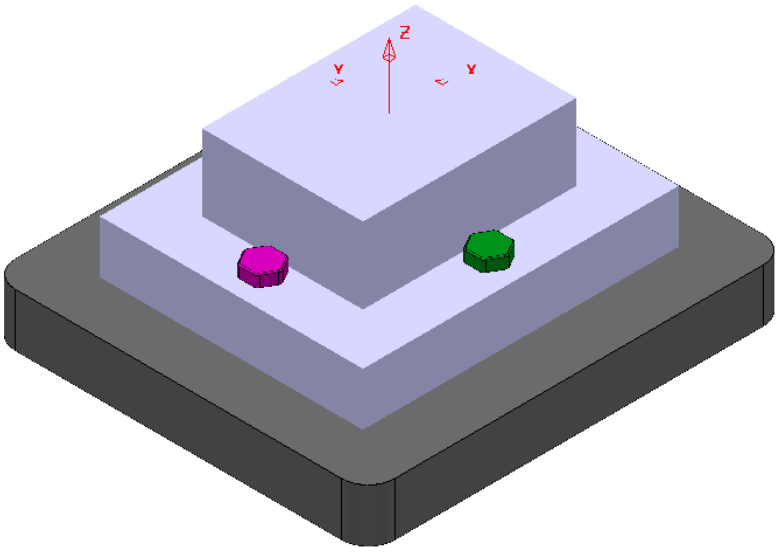
Modification of existing strategies to allow for interchangeable clamping bolts.

The following **Project** already includes some **Strategies** to create the 3D component form on a pre-machined material **Block**. All except the initial ***Face Milling*** strategy need to be modified to accommodate 2 interchangeable pairs of clamping bolts aligned along X and Y respectively.

1 Select **File** > **Delete all** and **Tools** > **Reset Forms**.

2 From **File** > **Open Project** select the read-only **Project**:

.....\PowerMILL-Data\Projects\CompThic-2_Start



The individual parts (Component Model, Capping Surfaces, Fixture Plate, and Clamping Bolts) are already acquired to suitable **Levels**.

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

.....\COURSEWORK\PowerMILL-Projects\CompThickness-2

4 Select an **ISO 1** view (as shown in the above image).

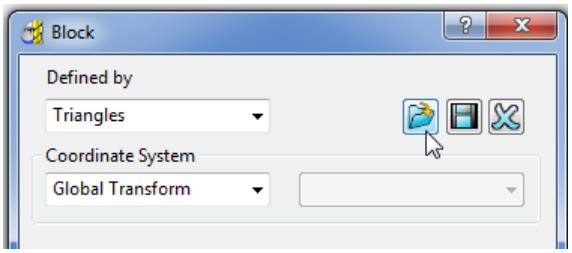
5 In the **PowerMILL explorer**, right mouse click on the **Toolpath** ***EM25-FaceMill-Top*** and from the local menu select **Activate**.

6 In the **PowerMILL explorer**, right mouse click on the **Toolpath** ***EM25-FaceMill-Top*** and from the local menu select **Simulate from Start**.

*Before performing a **ViewMILL** simulation the **Block** will be modified to use a model of the pre-machined material.*

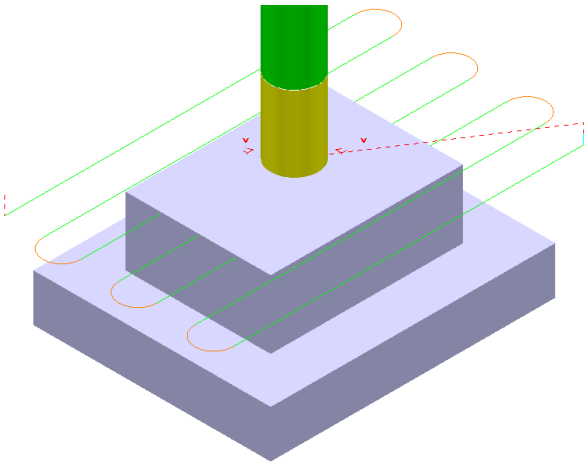
 Copyright © Delcam plc

7 From the ***main*** toolbar select the **Block** dialog.



8 Select **Defined by** > **Triangles** and **Coordinate System** > **Global** before selecting the option **Load block from file**.

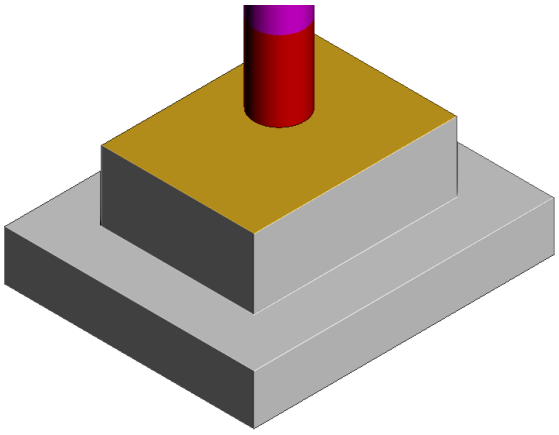
9 In the ***Open Block from Triangles*** dialog browse to:- **C:\Training_Data\PowerMILL_Data\Triangles\CompThic-2_Block.stl** and select **Open**.



This **Block** definition is an accurate model of the pre-machined component.

10 Toggle into **ViewMILL** and select the **Rainbow Shaded Image**.

11 In the **Simulation** toolbar select the ***Play*** button.

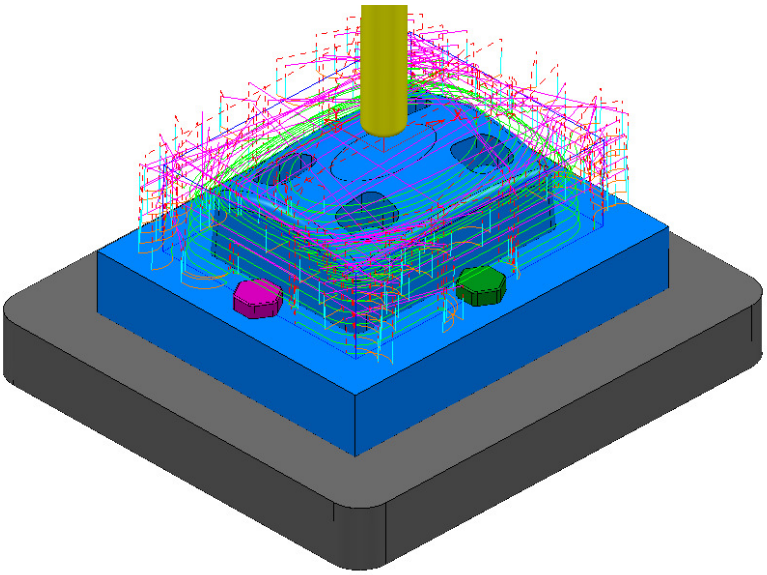


12 Select the **No Image** option to return to the **PowerMILL *window*** and for the time being disconnect **ViewMILL** from the **Simulation**.

13 From the **main** menu select the **Block** option and in the dialog move the **Opacity** slider all the way to the left (wireframe).

14 In the **Viewing** toolbar select the option **Multicolour Shade**.

15 **Activate** the **Toolpath *D16TR3-Rgh1***.



This toolpath will be modified to **Ignore** the ***purple***, clamping blots along the X-Axis (These are to be physically removed when this toolpath is run).

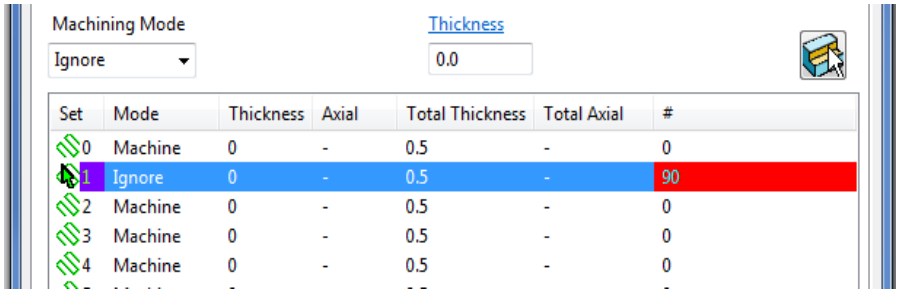
This toolpath will also be modified to apply **Collision** status to the ***green***, clamping bolts from a 2mm distance.

16 Right mouse click on the **Toolpath *D16TR3-Rgh1*** and select **Settings** to open the **Model Area Clearance** dialog.

17 Select the ***Re-cycle*** option to enable editing to occur.

18 In the **main** page of the **Model Area Clearance** dialog, select the **Component Thickness** dialog .

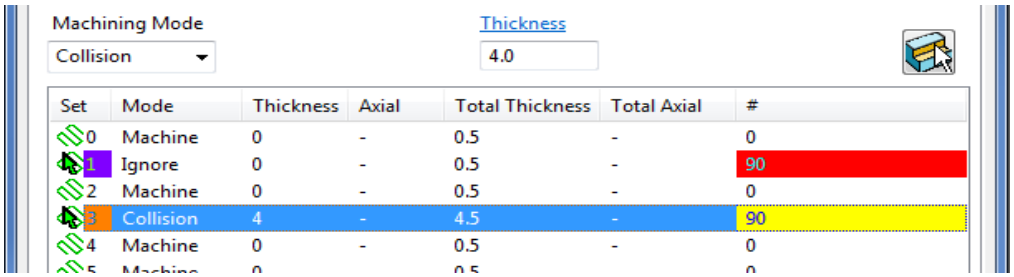
19 In the dialog use **Smart Selection** to acquire the data stored on the **Level *ClampingBolts-X*** to one of the colour coded rows.



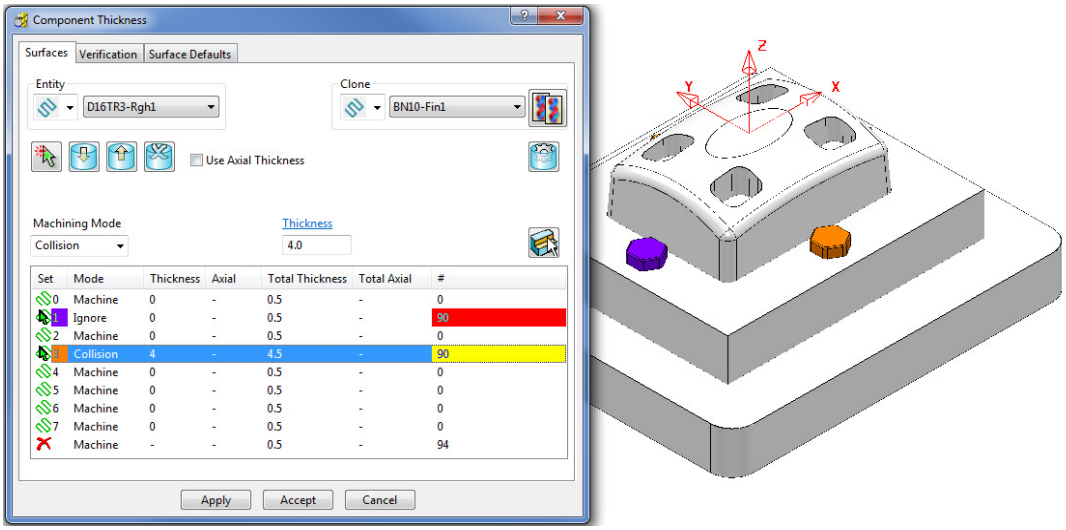
20 Set the row with ***Machining Mode*** > **Ignore**.

Copyright © Delcam plc

21 Select one of the other colour coded rows and use **Smart Selection** to acquire the data stored on the **Level *ClampingBolts-Y***.

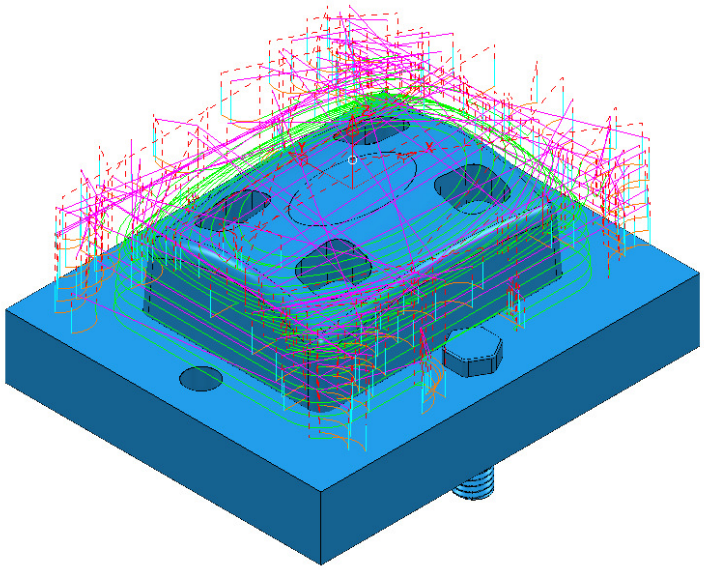


22 Set the row with ***Machining Mode*** > **Collision** with a **Thickness** > **2.0** and the select **Apply**.



The parts of the model acquired to a row will be shaded in the same colour.

23 **Accept** the **Component Thickness** dialog and then **Calculate** the **toolpath**.



The updated **toolpath** (***D16TR3-Rgh1***) now runs straight through the ***X-Axis*** clamping bolt area and clears the **Y-Axis** clamping bolts by a full 2mm.

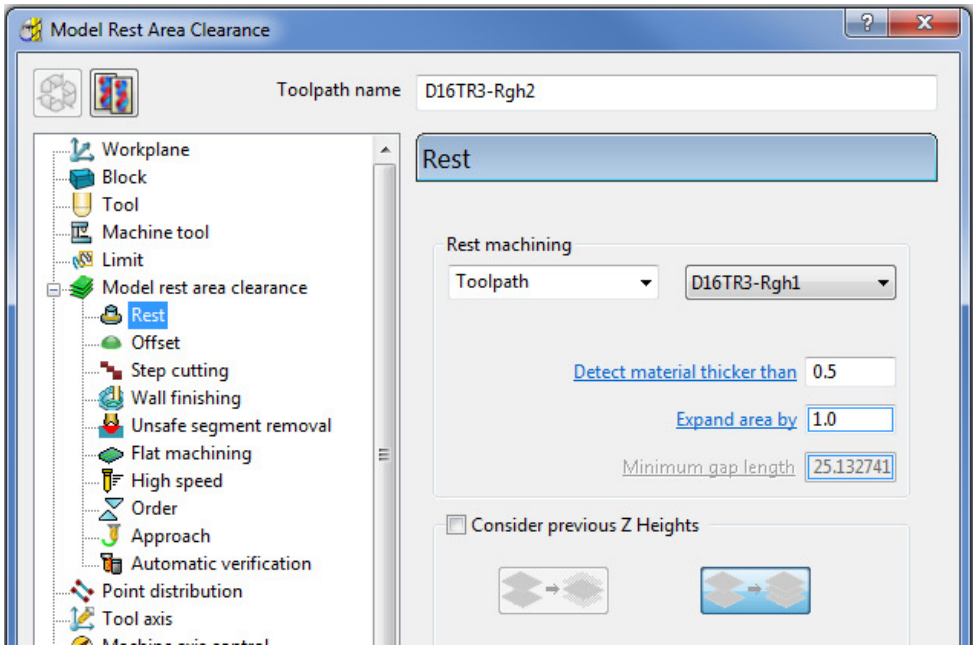
We now require a copy of the **Toolpath** (***D16TR3-Rgh1***) that locally machines the remainder of the component in the clamping bolt area along the Y-Axis.

24 Right mouse click on the **Toolpath *D16TR3-Rgh1*** and select **Settings** to open the **Model Area Clearance** dialog.

25 Select the ***Make a Copy*** option and in the **Model Area Clearance** dialog rename it as ***D16TR3-Rgh2***.

As the new **toolpath** is a direct copy, the **Component Thickness** settings will be exactly the same as in ***D16TR1-Rgh1***.

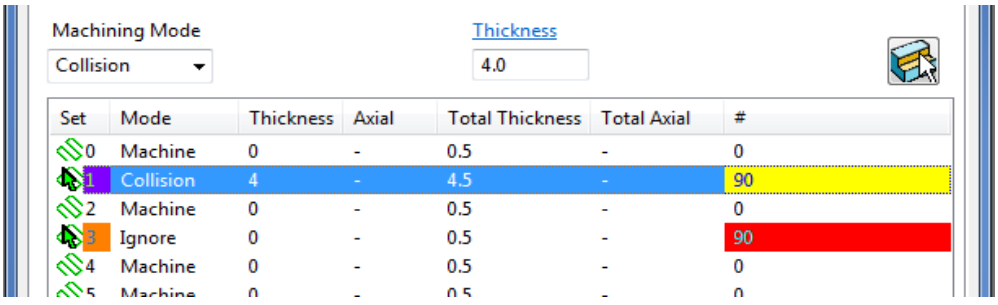
26 In the main page tick the **Rest machining** box and then open the **Rest** page of the dialog.



27 In the above dialog, insert the reference **Toolpath** > **D16TR3-Rgh1** and fill in the other values exactly as shown.

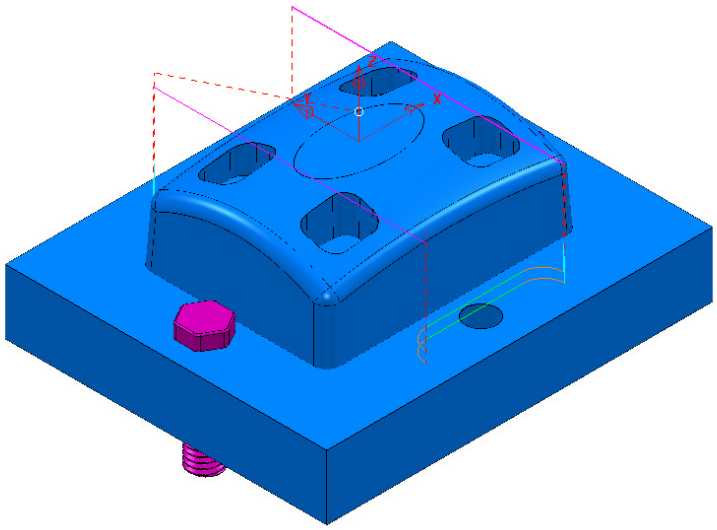
28 In the **main** page of the **Model Area Clearance** dialog, select the **Component Thickness** dialog .

29 Swap over the settings between the 2 existing rows (as shown below).



30 **Accept** the **Component Thickness** dialog and then **Calculate** the **toolpath**.

 Copyright © Delcam plc



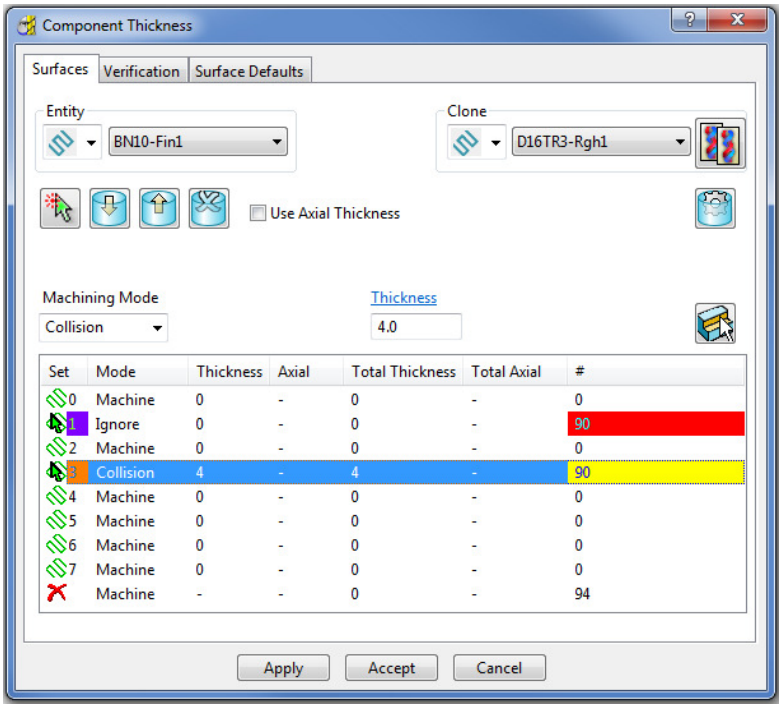
The updated ***Rest Roughing***, **toolpath** (***D16TR3-Rgh2***) now runs straight through the ***Y-Axis*** clamping bolts.

31 **Activate** the **Toolpath *BN10-Fin1*** and then select **Settings** from the local menu to open the **Steep and Shallow Finishing** dialog.

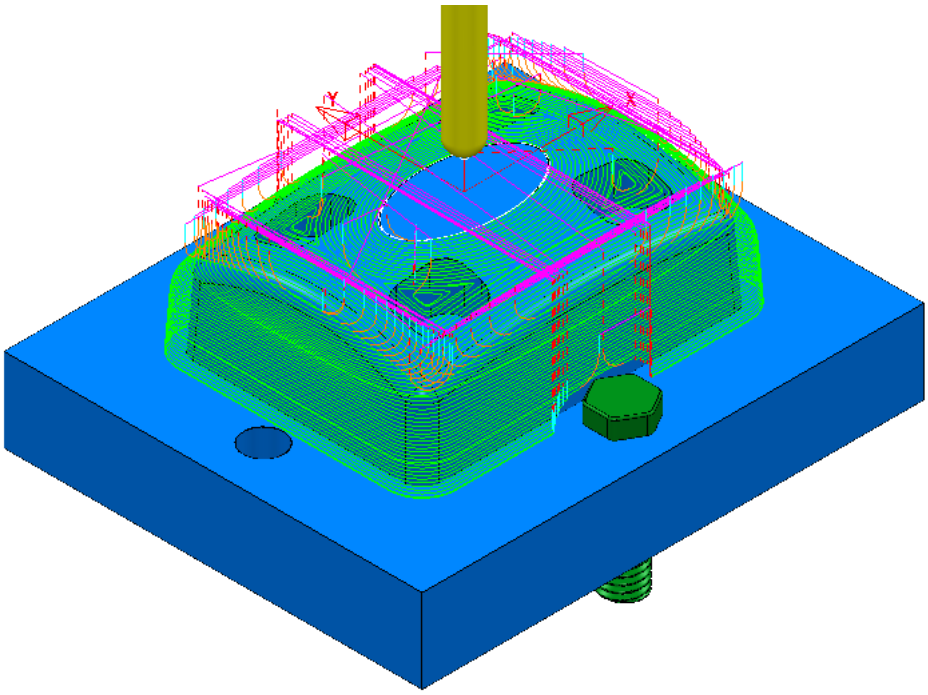
32 Select the ***Re-cycle*** option to enable editing to occur.

33 In the **main** page of the **Steep and Shallow Finishing** dialog, select the **Component Thickness** dialog .

34 In the ***Clone*** pull down select the **Toolpath *D16TR3-Rgh1*** and the click on the **Copy Thickness Data** icon.



35 **Accept** the **Component Thickness** dialog and then **Calculate** the toolpath.



36 The updated **toolpath** (***BN10-Fin1***) now runs straight through the ***X-Axis*** clamping bolt area and clears the **Y-Axis** clamping bolts by a full 2mm.

The next stage is to create a copy of the above **toolpath** (***BN10-Fin1***) that locally machines the remaining areas within the two Y-Axis clamping bolts.

37 Right mouse click on the **Toolpath *BN10-Fin1*** and select **Settings** to open the **Steep and Shallow Finishing** dialog.

38 Select the ***Make a Copy*** option and in the **Model Area Clearance** dialog rename it as ***BN10-Fin2***.

As the new **toolpath** is a direct copy, the **Component Thickness** settings will be exactly the same as in ***BN10-Fin1***. In this case the required **Component Thickness** settings will be cloned from the **Toolpath *D16TR3-Rgh2***.

39 In the **main** page of the **Steep and Shallow Finishing** dialog, select the **Component Thickness** dialog .

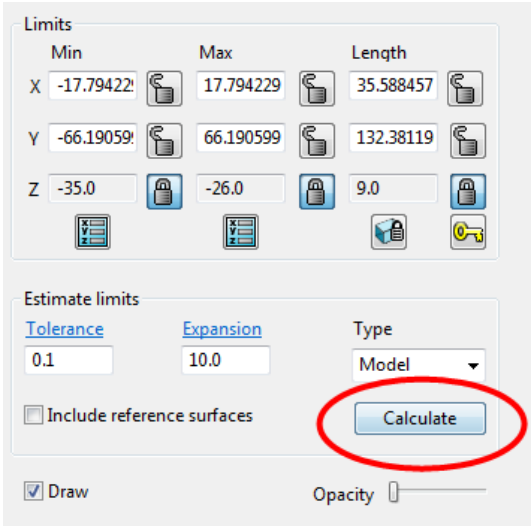
40 In the ***Clone*** pull down select the **Toolpath *D16TR3-Rgh1*** and the click on the **Copy Thickness Data** icon.

41 **Accept** the **Component Thickness** dialog.

42 In the **Steep and Shallow Finishing** dialog select the **Block** page and ***lock*** both the **Min Z** value (-35) and **Max Z** value (0). Keep the **Block** page open.

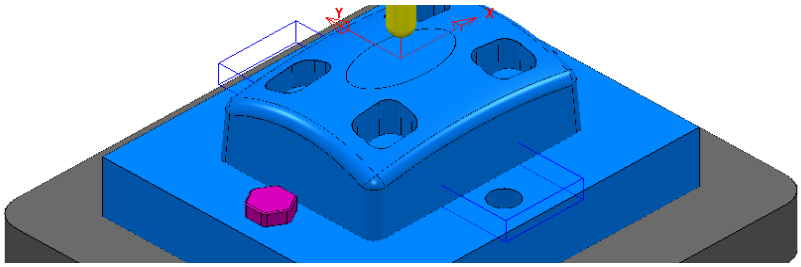
43 In the **PowerMILL *explorer***, right mouse click on the **Level** named ***ClampingBolts-Y*** and from the local menu click on **Select All**.

Copyright © Delcam plc



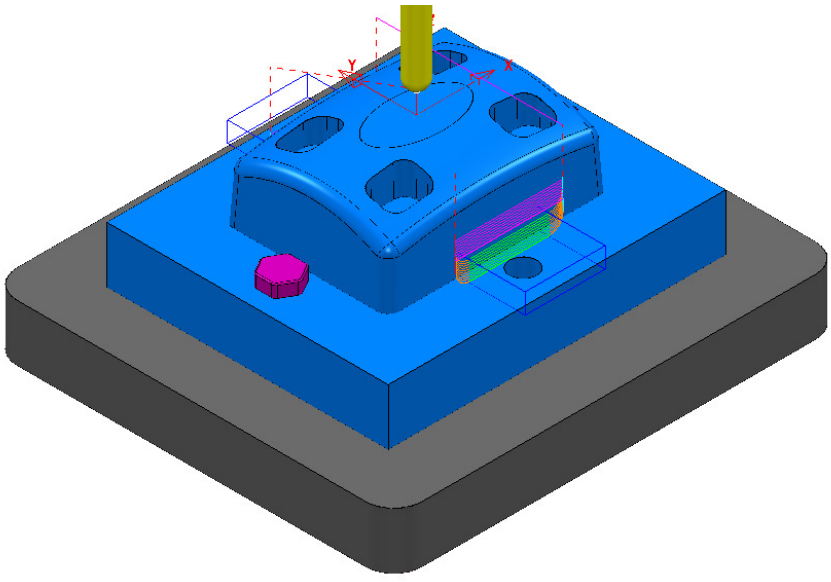
44 Back in the **Block** dialog enter an **Expansion** of **10** and **Calculate**.

45 ***Unlock*** the **Z Max** value and manually input the value **-26**.



46 Finally select **Calculate** at the bottom of the main **Steep and Shallow Finishing** dialog to create the new **toolpath *BN10-Fin2***.

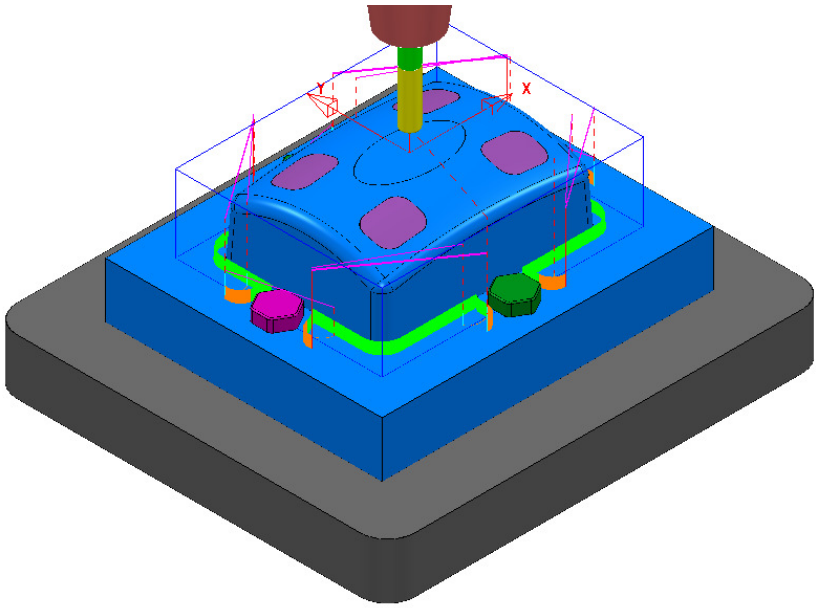
47 **Cancel** the **Steep and Shallow Finishing** dialog.



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

Exercise

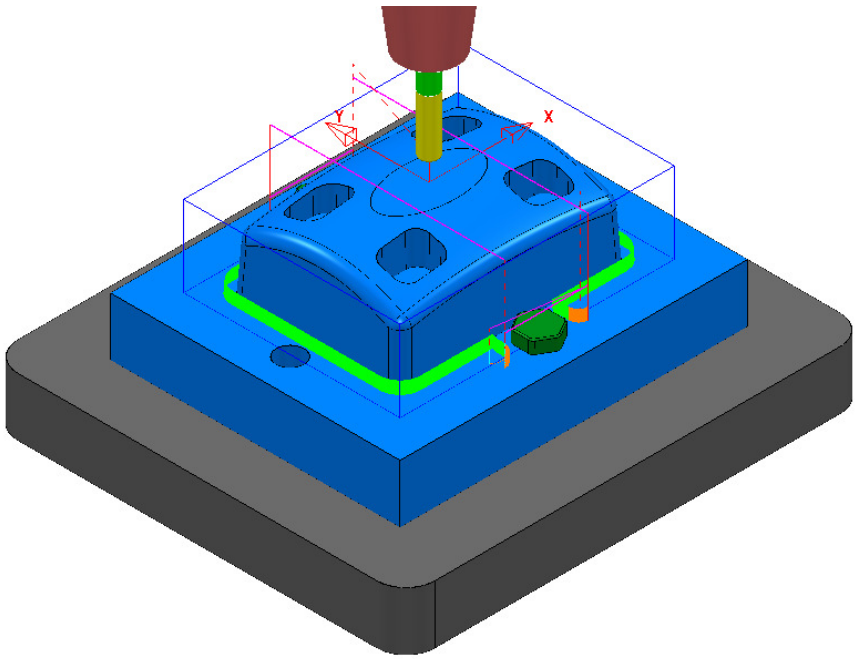
1 **Activate** the existing **Constant Z Finishing** toolpath ***D8TR1-Fin1***.



2 Select **Settings** and **Recycle** on the above toolpath.

3 Use the **Component Thickness** options to **Ignore** the **surfaces** on **Level *ClampingBolts-X*** (purple), and **Machine** with a **4mm thickness** around the **surfaces** on **Level *ClampingBolts-Y*** (green).

4 **Calculate** the changes to **toolpath *D8TR1-Fin1***.



5 Select **Settings** and **Make a Copy** for the toolpath ***D8TR1-Fin1***.

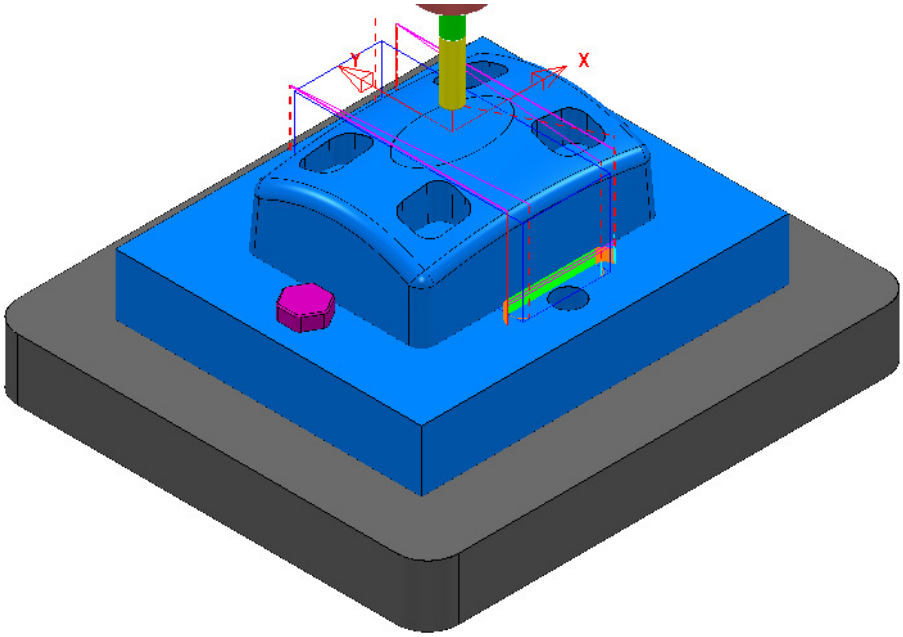
6 Rename the copy as **D8TR1-Fin2**.

Copyright © Delcam plc

7 Use the **Component Thickness** options to **Ignore** the **surfaces** on **Level *ClampingBolts-Y*** (green), and **Machine** with a **4mm thickness** around the **surfaces** on **Level *ClampingBolts-X*** (purple).

8 Modify the **X** dimensions of the **Block** to keep the **Constant Z Finishing *D8TR1-Fin2*** inside the area already machined with ***D8TR1-Fin1***.

9 **Calculate** the **toolpath *D8TR1-Fin2***.



10 In the **PowerMILL explorer** re-order the **Toolpaths** to keep them grouped for the same pair of ***clamping bolts***.

11 **Save** the **Project** to update**:-**

.....\COURSEWORK\PowerMILL-Projects\CompThickness-2

 Copyright © Delcam plc

17. Stock Models

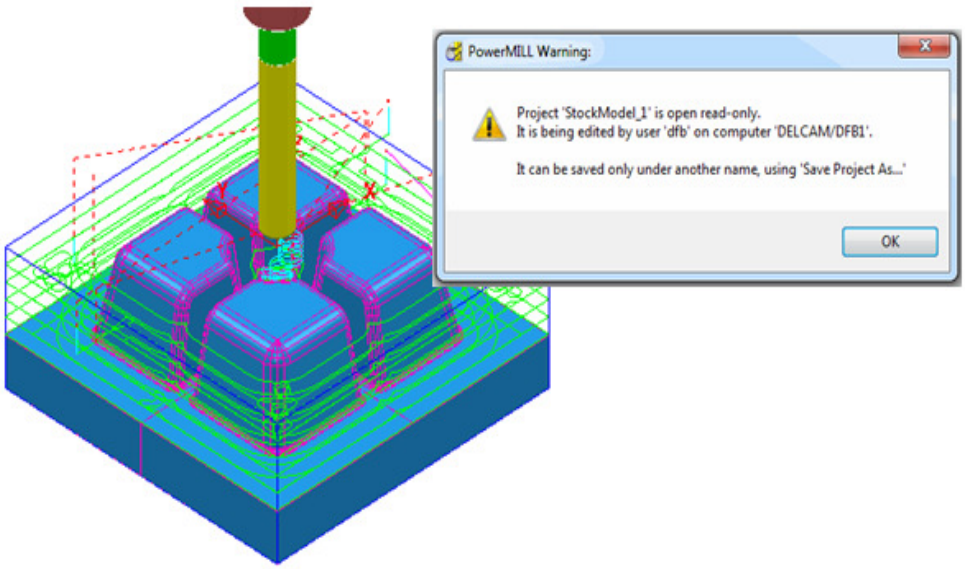
Introduction

A **Stock Model** provides both a visual and physical record of the remaining material after such entities as the material block and selected toolpaths have been considered.

Stock Model Rest Roughing

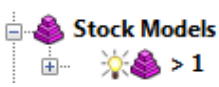
1. Select **File** > **Delete All** and **Tools** > **Reset Forms**.
2. Select **File** > **Open Project**:

   .....\PowerMILL_Data\Projects\StockModel_1
3. Select **OK** in the **PowerMILL Warning** dialog.

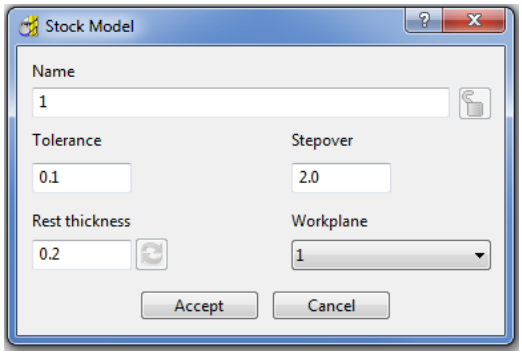


4. Select **File** - **Save Project As**:

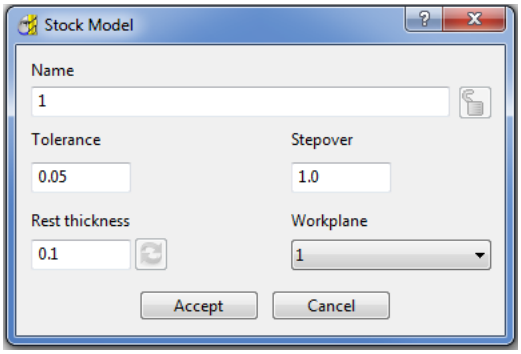
   .....\ COURSEWORK\PowerMILL-Projects\StockModelExample
5. In the **PowerMILL *explorer*** select **Stock Models** > **Create Stock Model**.



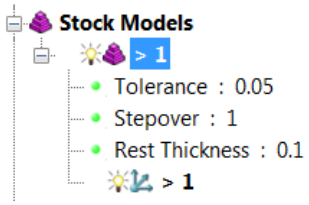
An empty **Stock Model** entity appears in the **PowerMILL *explorer*** and an associated ***Stock Model*** settings dialog appears in the graphics area.



The default settings for triangulation of the **Stock Model** are displayed. In this case we will modify the settings to create a more accurate ***Triangle Mesh*** for the **Stock Model**. Once the **Stock Model** is physically created then these inherent settings cannot be retrospectively altered.

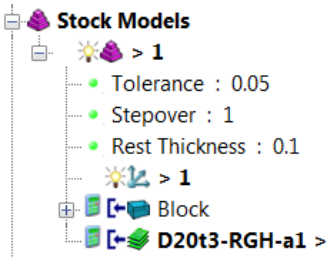


6 Modify the values in the ***Stock Model*** settings to be exactly as shown above and then **Accept** the dialog.

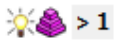


The displayed settings in the **Stock Model** will change accordingly.

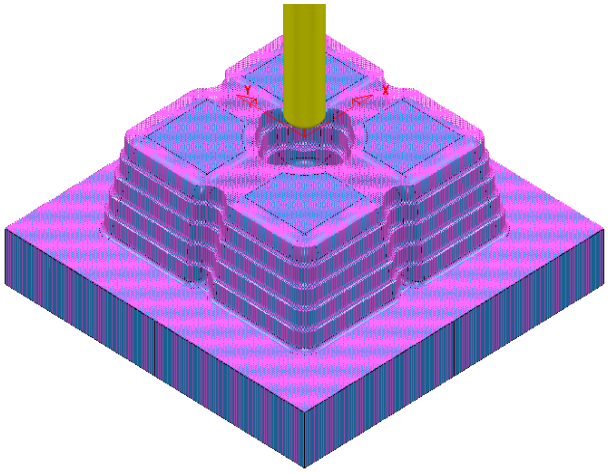
7 Right mouse click on the toolpath **D20t3-RGH-a1** and from the local menu select **Add to** > **Stock Model**.



The **toolpath** and associated **Block** are included in the **Stock Model** tree but at this stage are not calculated.

8 In the **PowerMILL *explorer***, in the **Stock Models** area, right mouse click on  and from the local menu select **Calculate**.

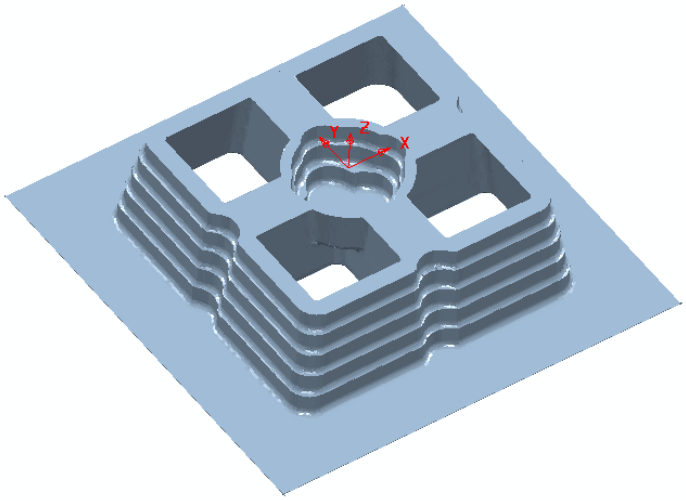
 Copyright © Delcam plc



The **Stock Model** will display the visual effect of toolpath **D12t3-RGH-a1** as a purple wireframe mesh.

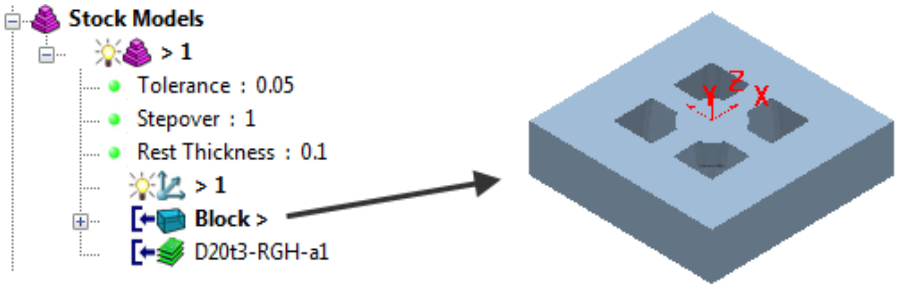
9 Right-click on **Stock Model** - **1** and from the local menu, select **Drawing Options**, and then tick both, **Shaded** and **Show Rest Material**.

10 From the **Viewing** toolbar, undraw the model (both **Shaded** and **Wireframe**).



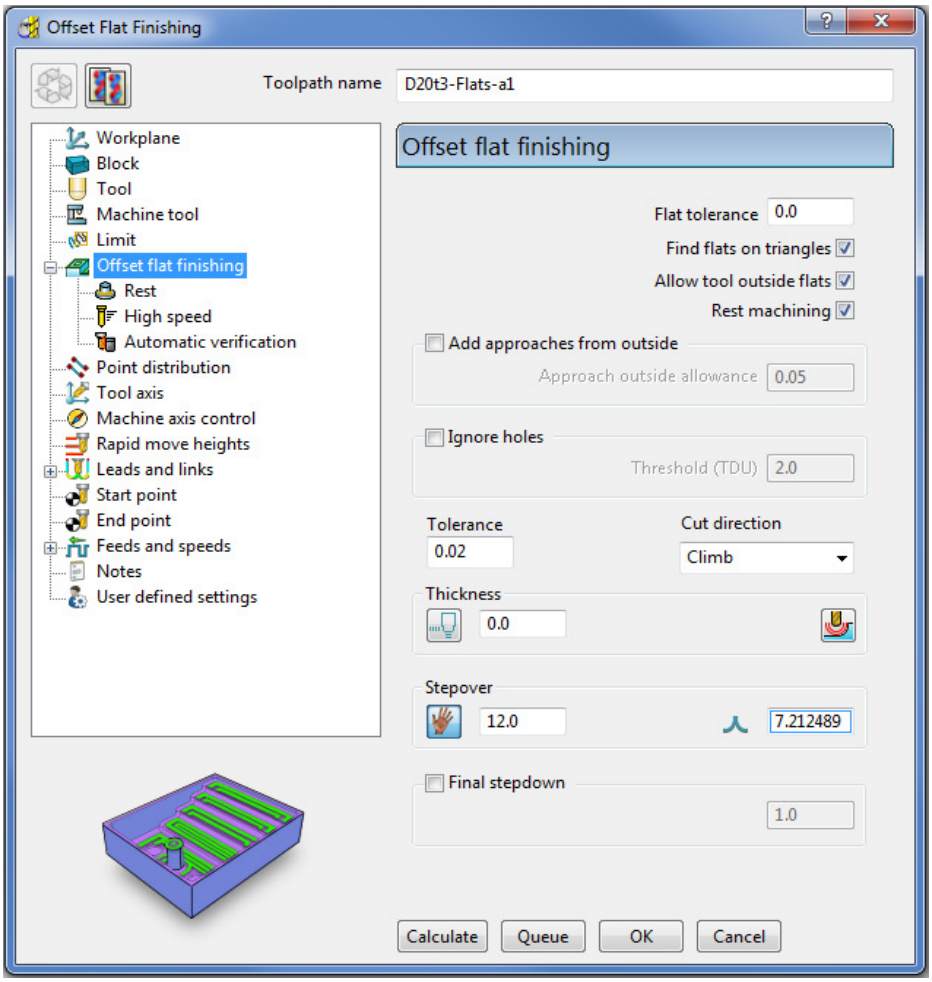
The **Stock Model** is now displayed as shown with the component model and applied toolpath both taken into account.

This represents the remaining material outside the component form after the taking into account the toolpath **D20t3-RGH-a1**.



*If **Block** is made active in the **Stock Model** tree then this is the stage of the machining process that will be displayed.*

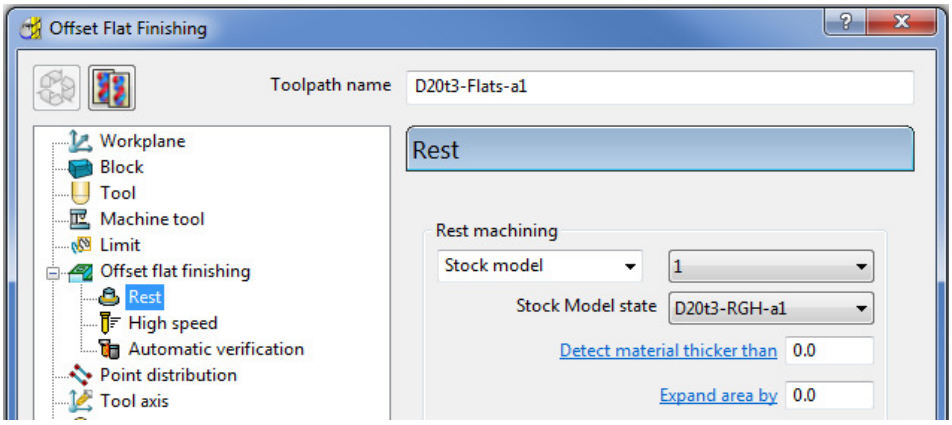
11 From the **Strategy Selector**, **Finishing** options select the **Offset Flat Finishing** strategy and enter data in the 3 pages exactly as shown below.



a Note: **Add approaches from outside** is un-ticked to enable independent **Leads** to be applied.

b The strategy will use **Rest Machining** (ticked) to the **Stock Model** to prevent the 4 upper flat areas from being included.

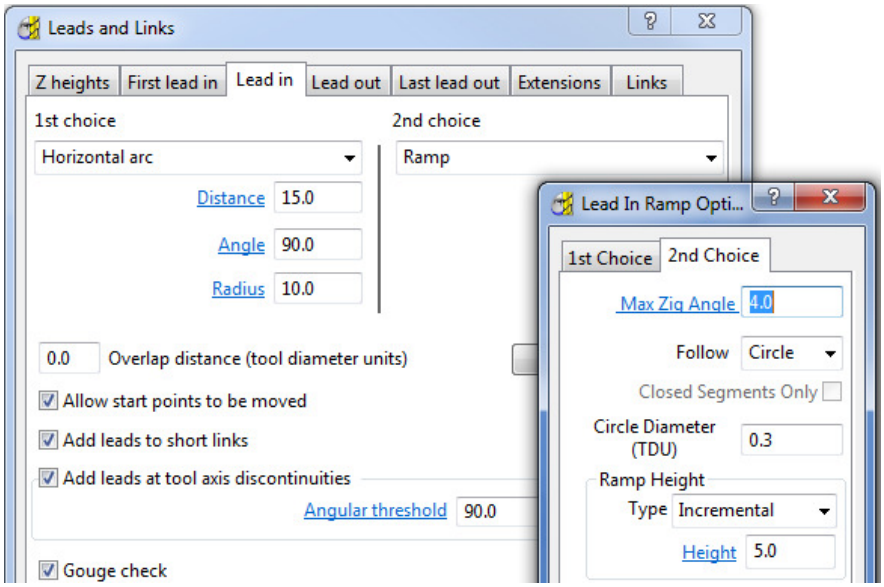
c By ticking this option the **Rest** page will become available for selection in the local browser.



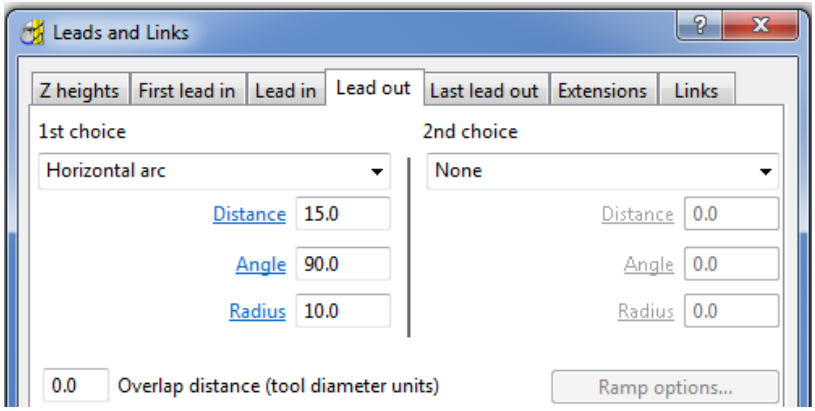
d Select **Stock Model** as **1**.

 Copyright © Delcam plc

12 In the **main** toolbar, **Leads and Links** options set the Lead in options exactly as shown below.

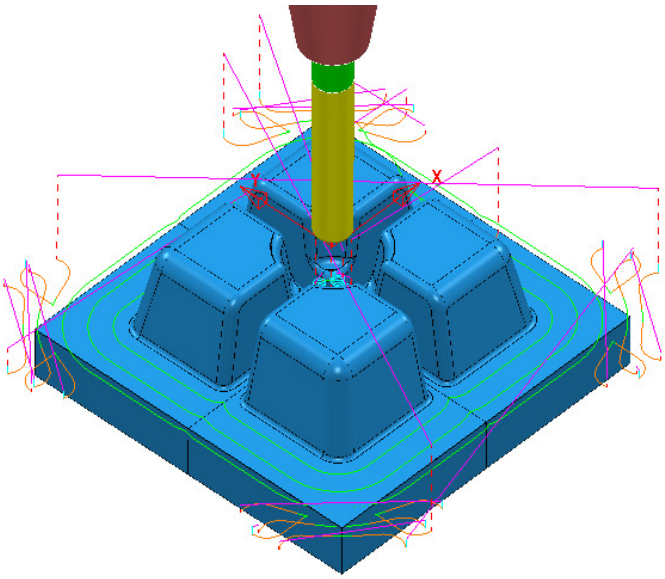


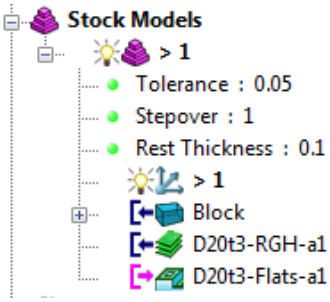
13 Then set the **Lead out** options exactly as shown below.



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

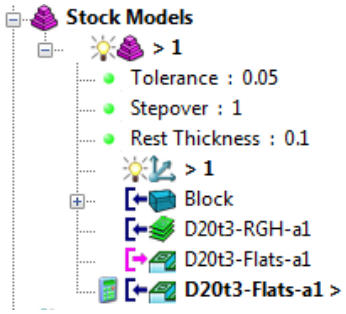
15 **Calculate** the **Offset Flat** finishing strategy and then **Close** the dialog.



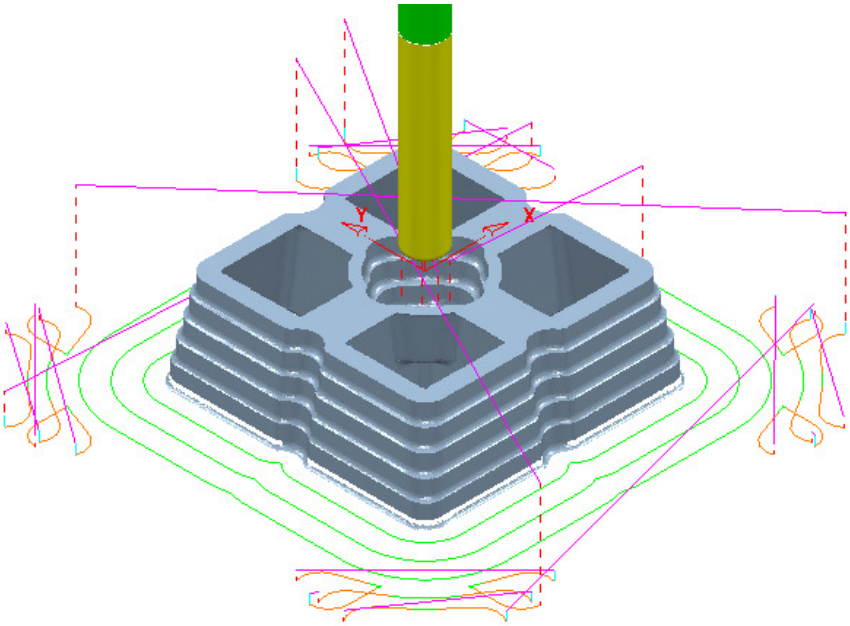


*The latest **Toolpath - D20t2-Flats-a1** is automatically added as a reference item in the **Stock Model** list but not as a physical entity.*

16 In the **PowerMILL *explorer***, right-click over the **Toolpath** - ***D20t3-FLATS-a1*** and in the local menu select **Add to** > **Stock Model**.



17 In the **PowerMILL *explorer***, right-click on the **Stock Model** named **1** and in the local menu select **Calculate**.

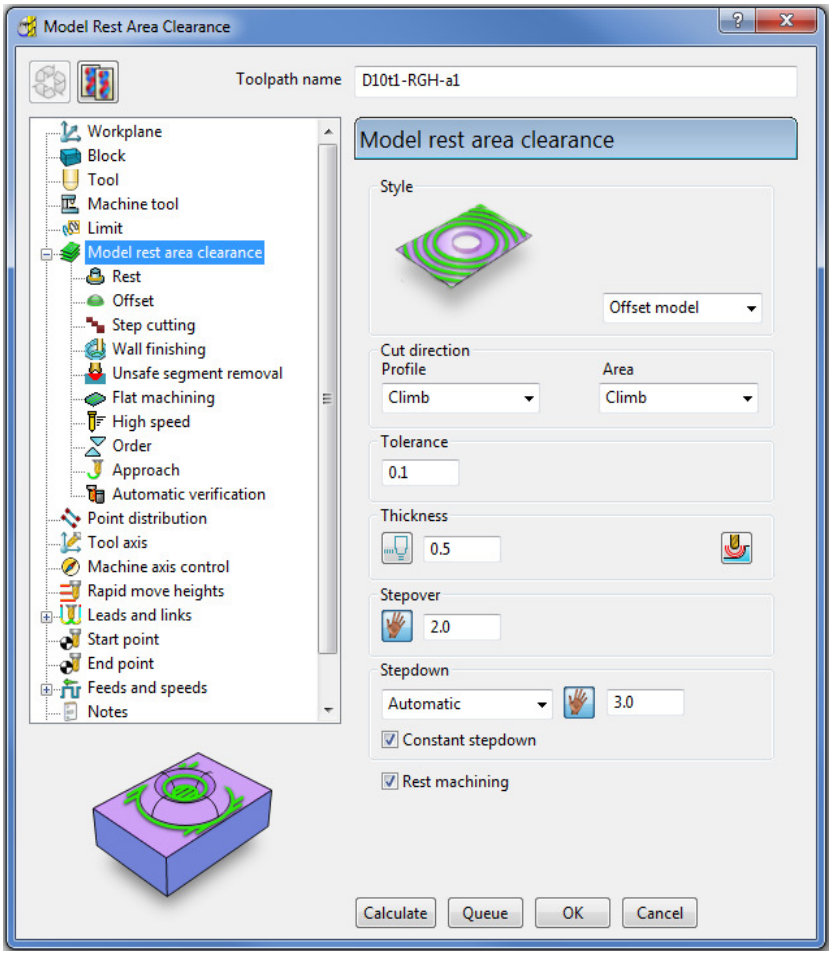


After the calculation, the **Stock Model** will be displayed again only this time with the **Offset Flats Finishing** included.

18 **Activate** the **toolpath *D20t3-RGH-a1*** and in the local menu select **Settings...** to open the **Model Area Clearance** dialog.

 Copyright © Delcam plc

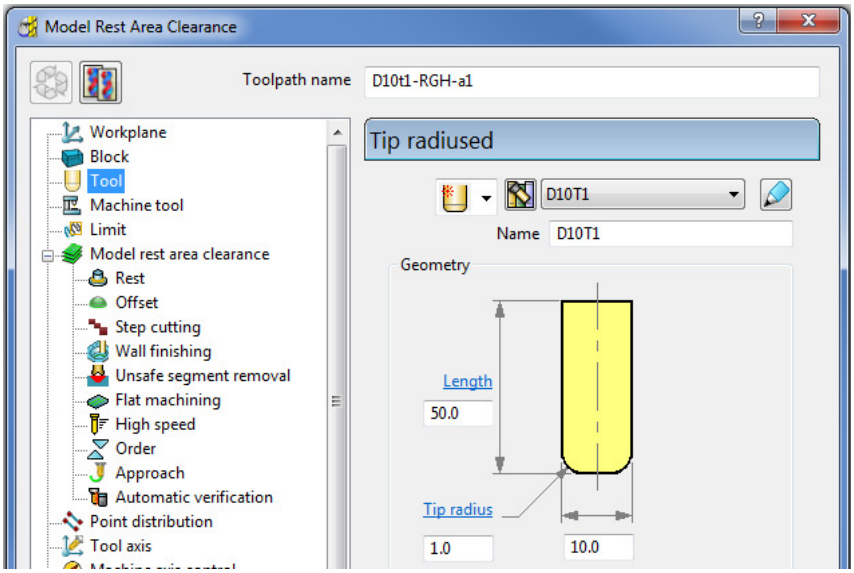
19 Select **Create a new toolpath based on this one** and input new data into the dialog exactly as shown on the following page.



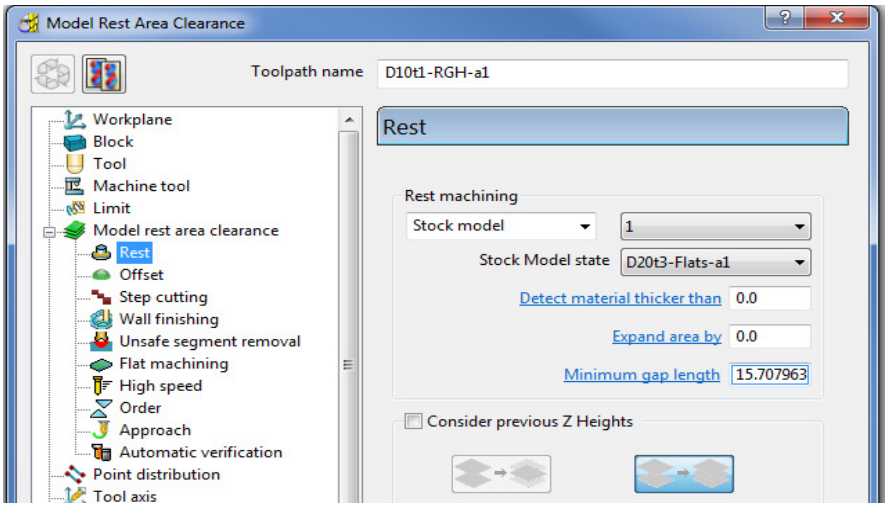
a Input the **Toolpath** name as ***D10t1-RGH-a1***

b Set the ***Style*** as **Offset model**.

c Tick the **Rest machining** box.

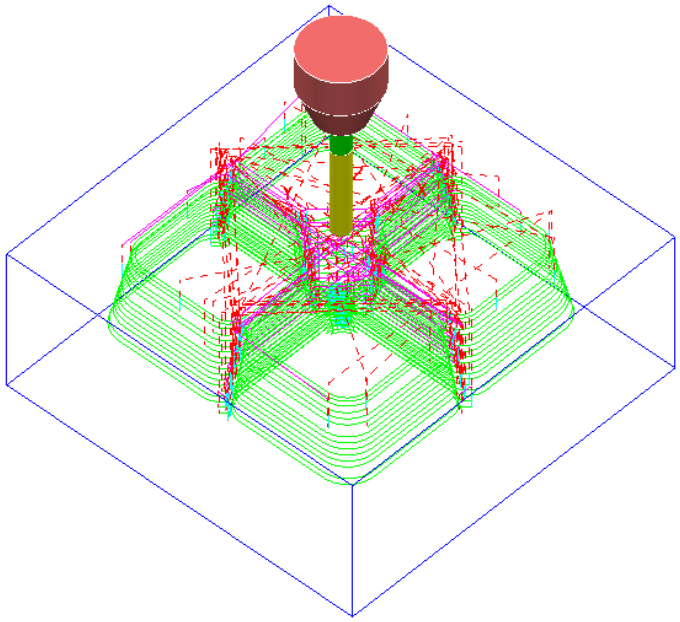


d Select the **D10t1** tool.



e In **Rest Machining** use **Stock Model** - **1**

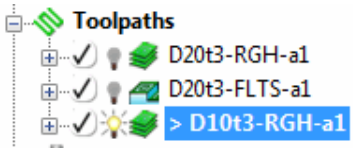
20 Select **Calculate** on the **Model Area Clearance** dialog to machine within the limits of the specified **Stock Model**.



21 Close the **Model Area Clearance** dialog.

22 The latest **Toolpath – D10t1-RGH-a1** is automatically added as a reference item in the **Stock Model** list but not as a physical entity.

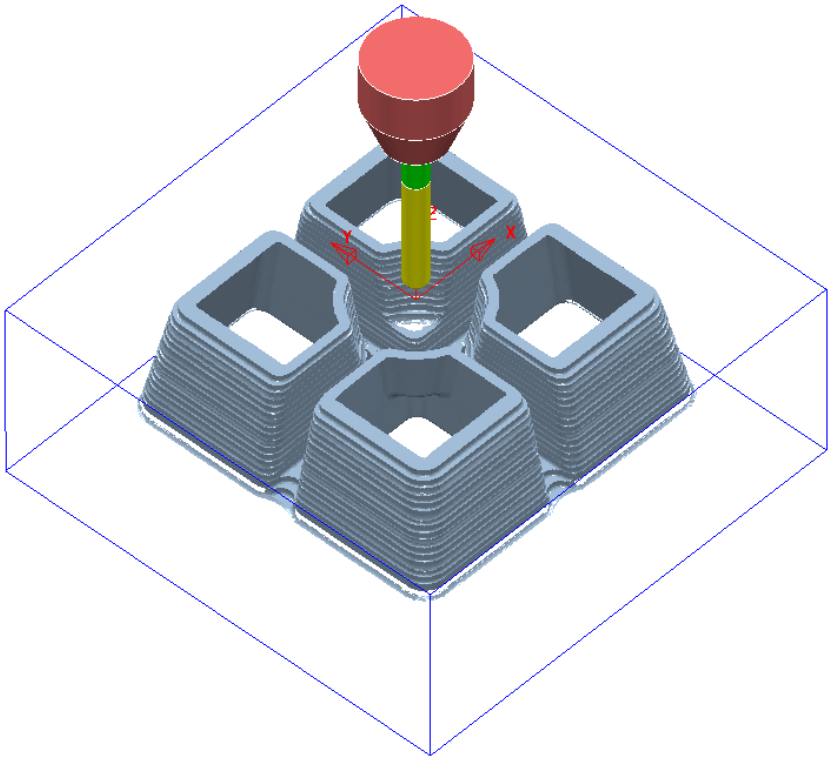
23 In the **PowerMILL *explorer*** use the left mouse button to drag the **Toolpath *D10t1-RGH-a1*** to the bottom of the list.



24 In the **PowerMILL *explorer***, right-click over the **Toolpath** – ***D10t1-RGH-a1*** and in the local menu select **Add to** > **Stock Model**.

25 In the **PowerMILL *explorer***, right-click on the **Stock Model** named **1** and in the local menu select **Calculate**.

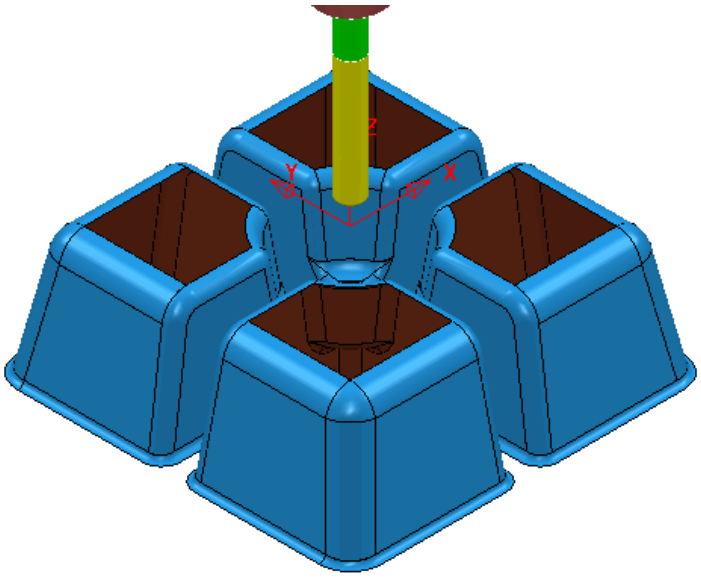
 Copyright © Delcam plc



The **Stock Model** is now displayed as shown with the latest toolpath removed.

Now that the bulk of the material is removed, the main component form is now ready to be ***Finish*** machined.

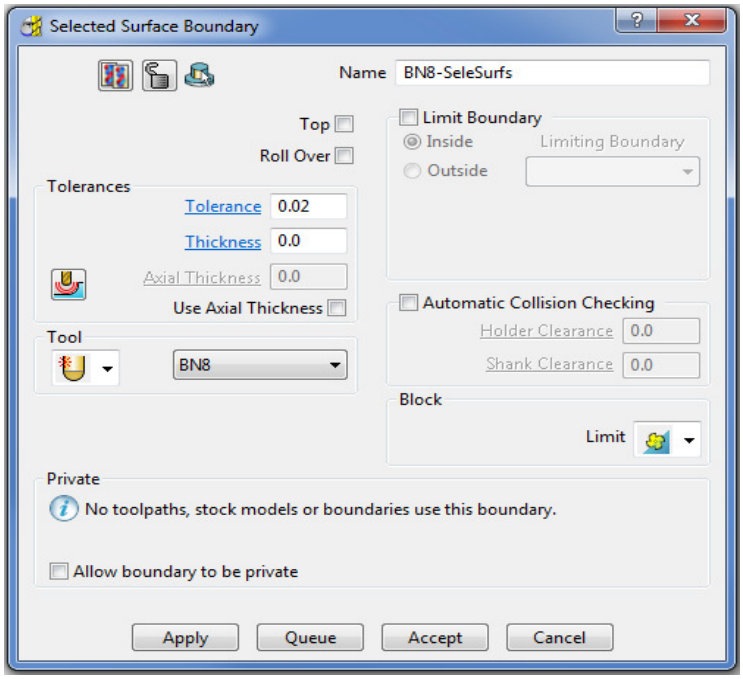
26 **Select** and **Blank** all except the following surfaces:



27 Using the left mouse button, drag a box across the displayed model above to select all **surfaces**.

28 **Activate** the tool **BN8**.

29 In the **PowerMILL *explorer***, right-click on **Boundaries** and in the local menu, select the **Selected Surface** option.

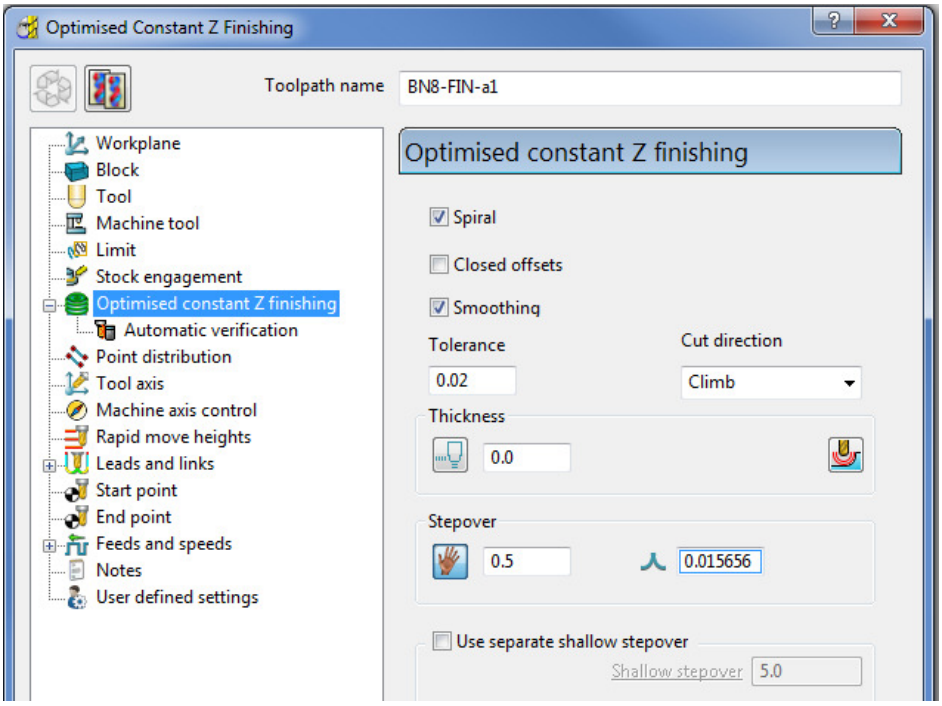


30 Enter data exactly as shown above and then **Apply** to create the **Boundary**.

31 Check that both **Boundary 1** and the **BN8** tool are both **Active**.

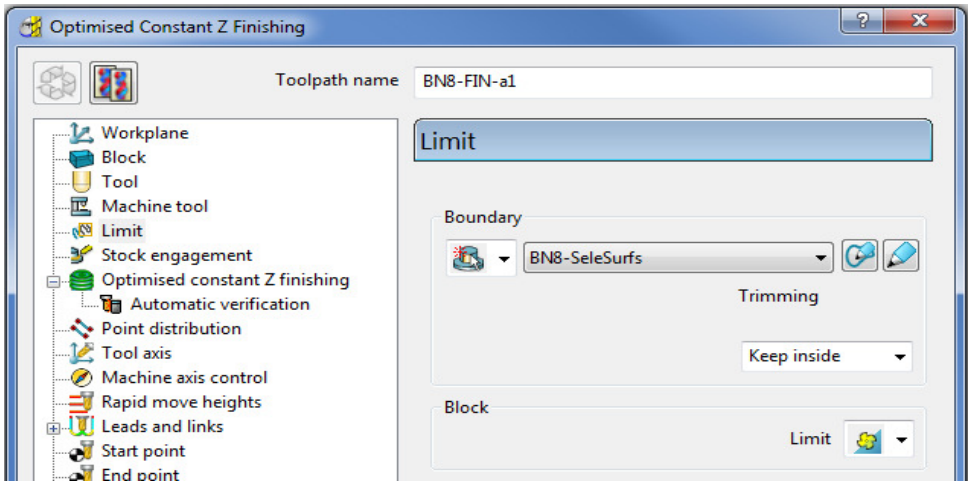
32 Select the **Toolpath Strategies** and select the **Finishing** tab in the dialog.

33 Select the **Optimised Constant Z** Finishing option and enter data into the dialog exactly as shown below:



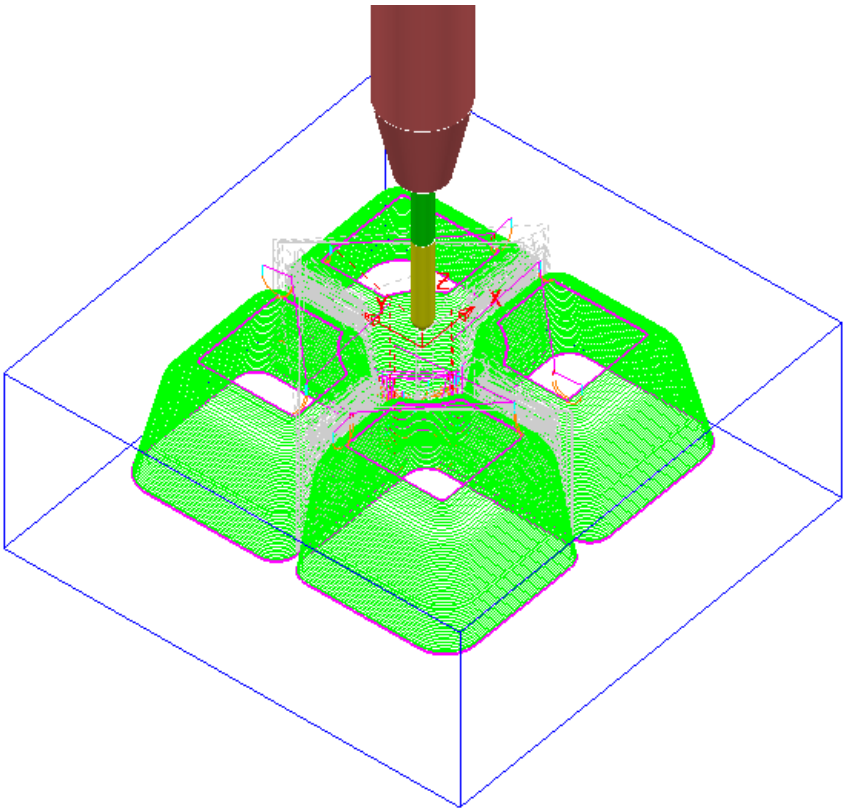
34 Select the **Limit** page and enter data exactly as shown in the following image:

 Copyright © Delcam plc



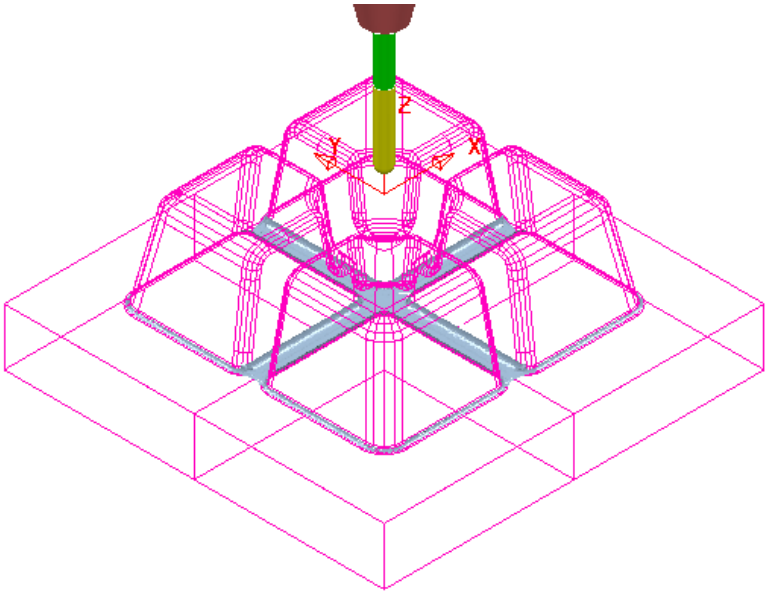
35 Select **Calculate** to create the **Toolpath** and then **Close** the dialog.

*The **Toolpath BN8-FIN-a1** has been limited to be inside the **Boundary**.*



36 In the **PowerMILL *explorer***, right-click the **toolpath *BN8-FIN-a1*** and in the local menu select **Add to** > **Stock Model**.

37 In the **PowerMILL *explorer*** right-click on **Stock Model** – **1** and select **Calculate** to update.



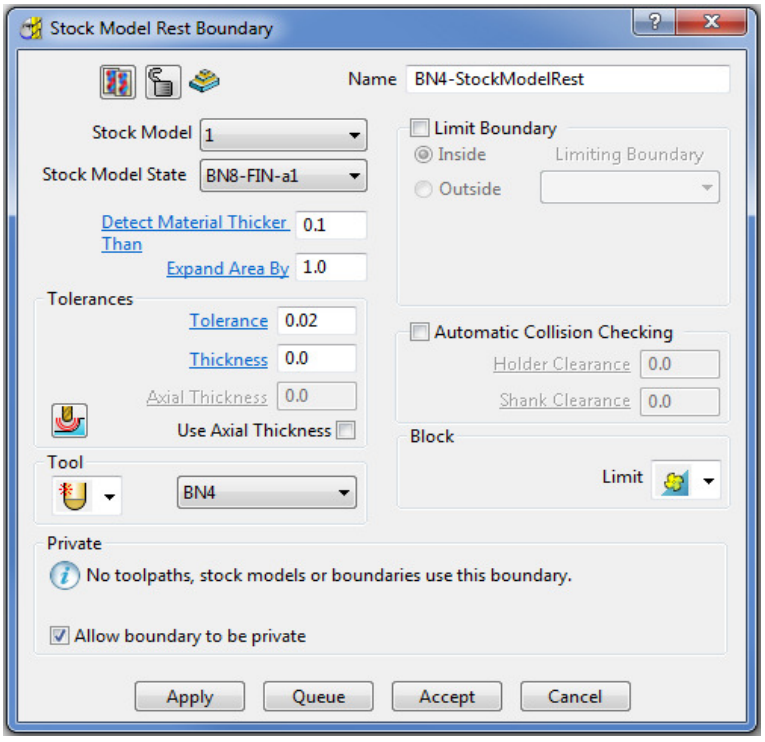
The remaining **Stock Model** is displayed as shown above.

Stock Model Rest Boundary

The **Finishing** strategies do not have a direct ***Rest Machining*** option that relates to the **Stock Model**.

It is however possible to create a ***Stock Model Rest*** **Boundary** to define the areas with material left on. This can then be used to limit a finishing strategy such that only the remaining stock is removed.

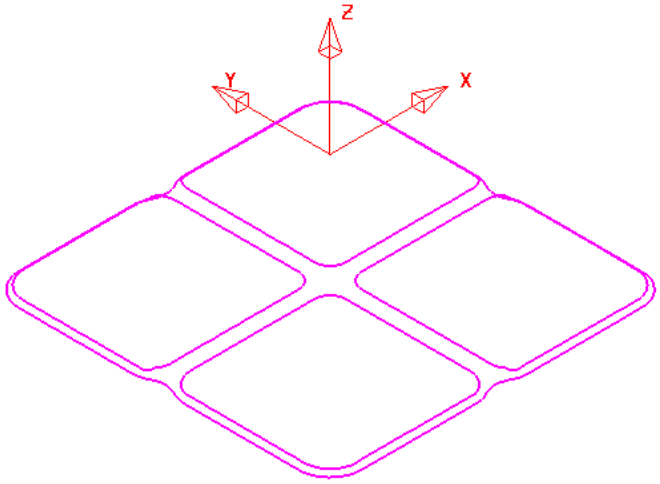
1 In the **PowerMILL** ***explorer***, right-click on **Boundaries** and in the local menu select **Stock Model Rest**.



2 Enter data exactly as shown in the above image and **Apply** and then **Cancel**.

 Copyright © Delcam plc

__Boundary__ segments will only be created where the __BN4__ tool is able to access the model form.

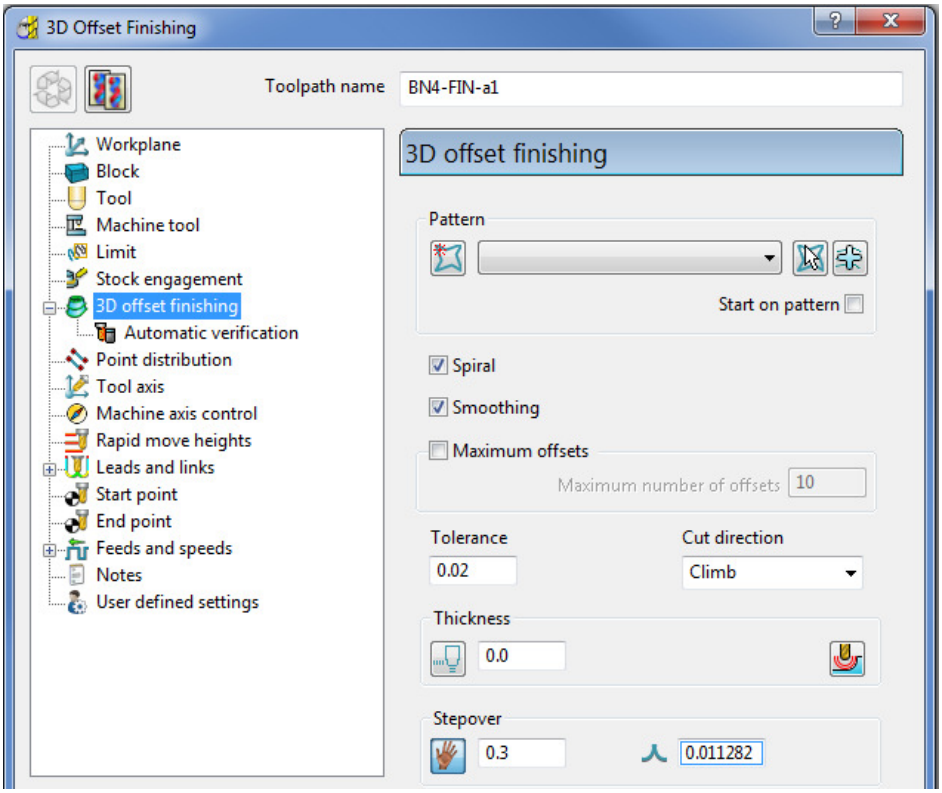


The ***Stock Model Rest*** **Boundary** identifies the area of the remaining **Stock Model**.

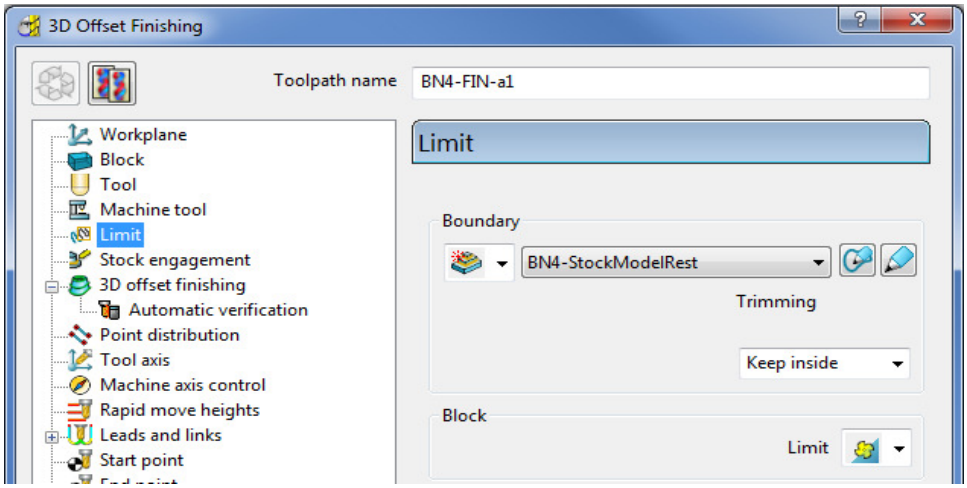
3 Select the **Toolpath Strategies** and select the **Finishing** tab in the dialog.

4 Check that the new **Boundary** ***BN4-StockModelRest*** and the **BN4** tool are both **Active**.

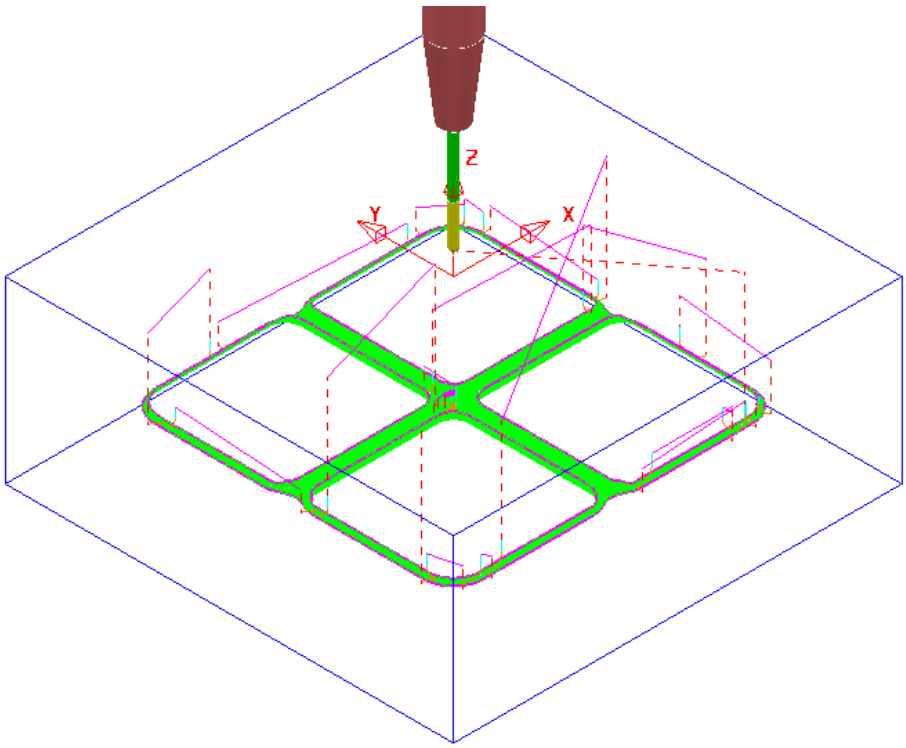
5 Select the **3D Offset Finishing** strategy and enter data exactly as shown below:



6 Select the **Limit** page and enter data exactly as shown in the following image:



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



8 In the **PowerMILL *explorer***, right-click the **toolpath *BN4-FIN-a1*** and in the local menu select **Add to** > **Stock Model**.

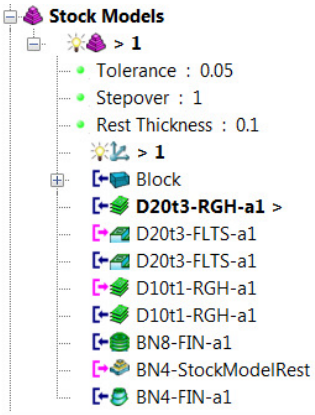
9 In the **PowerMILL *explorer***, right-click on the **Stock Model** and select **Calculate** to update.

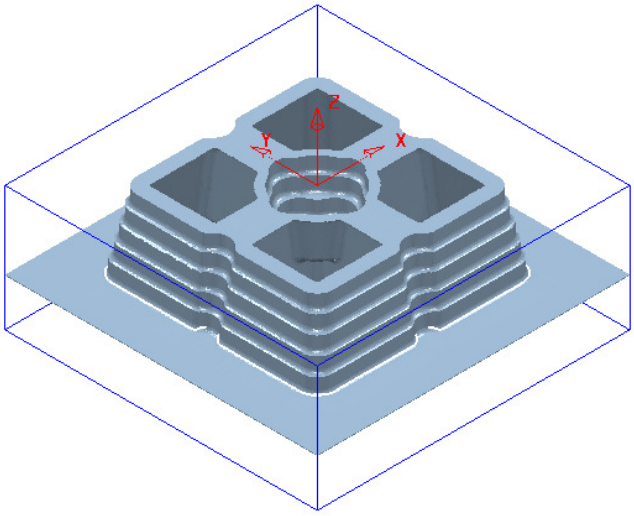
*The **Stock Model** displayed with **Show Rest Material** set will have visually disappeared apart from perhaps, a few tiny, insignificant patches.*

*If a previous toolpath in the **Stock Model** tree is selected then the status of the material at that point will be displayed.*

 Copyright © Delcam plc

10 In the **PowerMILL *explorer*, Stock Model - 1** tree, **Activate** the toolpath named **D20t3-RGH-a1**.

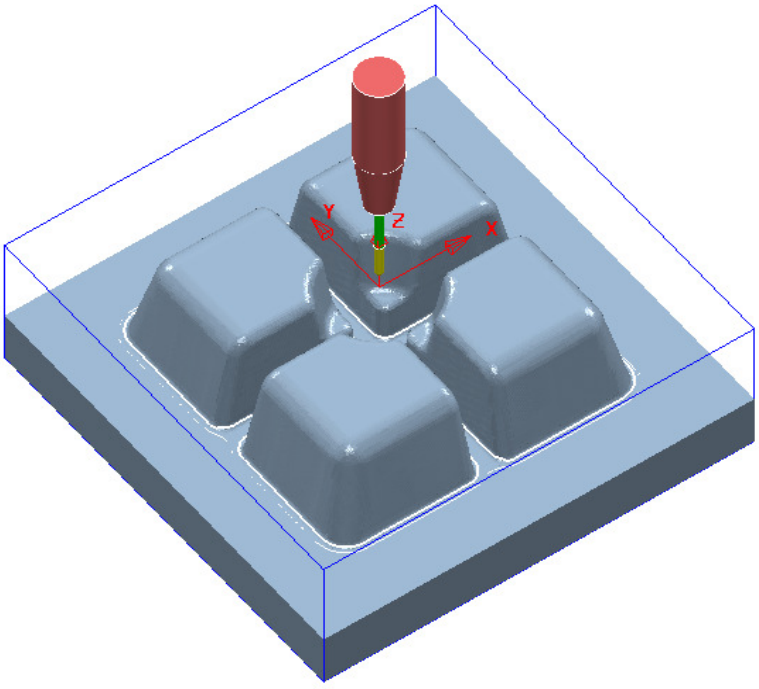




11 In the **PowerMILL *explorer*, Stock Model - 1** tree, **Activate** the toolpath named **BN4-FIN-a1**.

*The **Stock Model** will visually disappear while the **Show Rest Material** option is active.*

12 Right-click on the **Stock Model** and in the local menu, switch on (tick) the option **Show All Material**.



The **Stock Model** is now exactly the same shape and size as the **model**.

13 Select **File** > **Save Project** to update the stored **Project**.

Stock Model starting from a complex block.

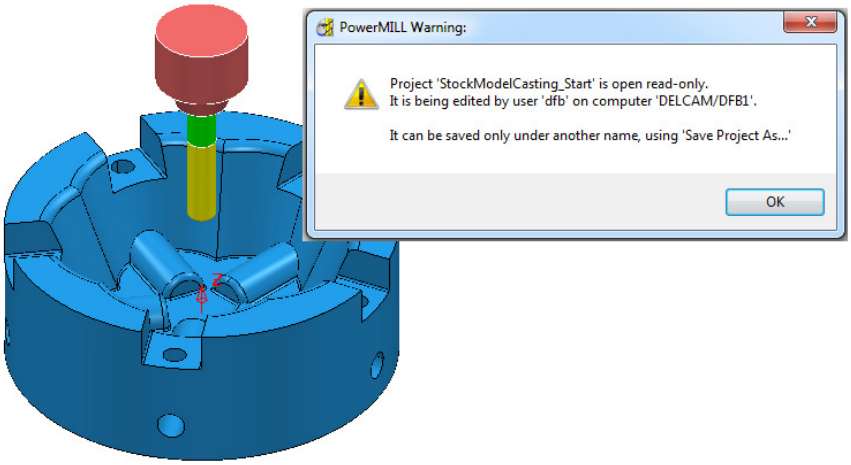
If a casting is to be machined then it is not possible to directly add the complex shape to a **Stock Model**. It is however possible to import a **Triangle** model of the original casting as the initial **Block** and then use it as the first entity in a **Stock Model**. The **Stock Model** can then be used as the referenced entity in an intial **Rest Roughing** strategy.

On import the **Block** must use the option, ***Coordinate System*** as **Global Transform** (Exported **Triangle** models do not support **Workplanes**).

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

2 Select **File** > **Open Project**:

.....\PowerMILL_Data\Projects\StockModelCasting_Start



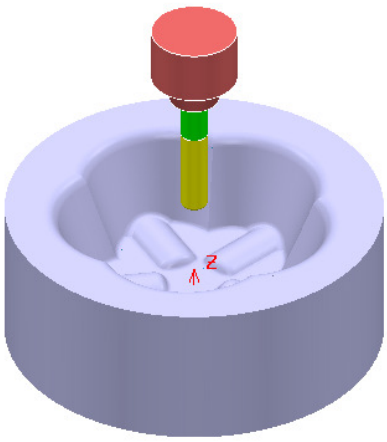
3 Select **OK** in the **PowerMILL Warning** dialog.

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

.....\ COURSEWORK\PowerMILL-Projects\StockModelCasting

5 Open the **Block** form and set ***Coordinate System*** to **Global Transform**.

6 Select **Defined by** – **Triangles** and browse to and **Open**:-
C:\Training_Data\PowerMILL_Data\Triangles\StockModel-Casting.dmt



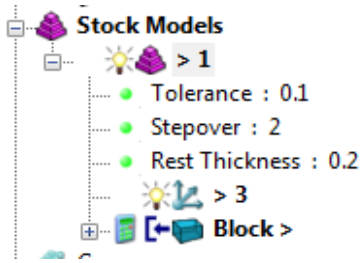
Copyright © Delcam plc

7 Open the **Rapid Move Heights** form and select **Calculate**.

8 **Activate** the tool **D25TR3**.

9 Create a new **Stock Model**.

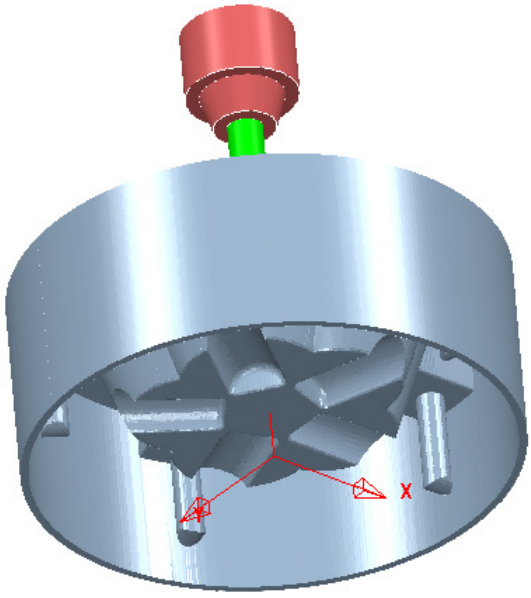
10 Right mouse click on the new **Stock Model** and from the local menu select **Apply Block**.



The complex **Block** is added as the first entity in the **Stock Model**.

11 Right mouse click on the new **Stock Model** and from the local menu select **Calculate**.

12 Right mouse click on the new **Stock Model** and from the local menu select **Drawing Options** and tick both **Shaded** and **Show Rest Material**.



The model form is visually removed from the **Stock Model**.

Exercise:

13 **Calculate** a **Block** of ***Type* - Cylinder** *and* ***Defined by* - Model**.

14 Create a **Model Rest Area Clearance** – **Offset Model** strategy using the above **Stock Model** as the reference item and the **D25TR3** tool.

Copyright © Delcam plc