

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 © 2014 – Delcam. All rights reserved.

## **PowerMILL 2016 Contents**

	<b>Chapters</b>	Page Number
<u>Day 1</u>	1. Getting Started	1.1 - 1.26
	2. Machining Set Up in Detail	2.1 - 2.22
	3. Area Clearance Strategies	3.1 - 3.54
<u>Day 2</u>	4. Finish Machining Strategies	4.1 - 4.54
	5. Leads and Links	5.1 - 5.14
	6. Boundaries	6.1 - 6.32
<u>Day 3</u>	7. Levels and Sets	7.1 - 7.10
	8. Editing Toolpaths	8.1 - 8.24
	9. Collision Checking	9.1 - 9.14
	10. Patterns	10.1 - 10.18
<u>Day 4</u>	11. Curve Editor	11.1 - 11.12
	12. 2D Machining	12.1 - 12.48
	13. NC Programs	13.1 - 13.10
	14. Setup Sheets	14.1 - 14.8
<u>Day 5</u>	15. Specialist Finishing	15.1 - 15.38
<u> </u>	16. Component Thickness	16.1 - 16.22
	17. Stock Models	17.1 - 17.18
	18. Macros & User Menus	18.1 - 18.10
	19. Tutorials	19.1 - 19.36
		10.1 10.00

#### Contents

## **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 **Delcam Exchange** translators are purchased, **PowerMILL** will also directly import data created by other proprietary CAD packages.

## Start PowerMILL



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

On the training

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

The **PowerMILL** screen is displayed:

3		PowerMILL Pro 2016	DEMONSTRATI	ON licence [ * No	ine ]				×
File View Insert Draw Tools Help									
🔌 🖪 💊 🖗 ភិះ 🗐 🖉 💆	🔅 🖬 🞍 🚫	No favourites set	V K 2	, 🖬 🚥 🖪 💥					
Active     Machine Tools     Active     Machine Tools     Soundaries     Patterns     Festure Sets     Wodels     Active Sets     Models     Groups     Macros				**** ×					
	Y Z X								
🍇 🚽 🥸 🍄 🏢	242.798	240.057 0	4 <sup>0</sup>	<b>511</b> [mm] mm	$\ominus$	Q	Ø	🥑 Delcam 🥐	0

The screen is divided into the following main areas:

a Main menu

File View Insert Draw Tools Help

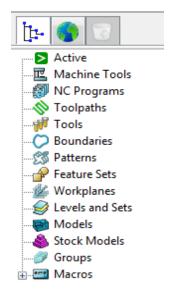
Click one of the menu names on this bar (for example, **File**) to open a pull-down menu list of associated commands and sub-menus. A sub-menu is indicated by a small arrow to the right of the text (for example **File - Recent Projects >**). Highlighting this arrow generates a list of commands/names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

**b Main Toolbar** (shown un-docked)



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

c Explorer



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

#### d Graphics Window

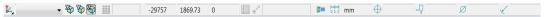
This is the large, visual display area to the right of the **Explorer** (Look at the illustration on previous page).

e View Toolbar (shown as horizontal)



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

#### f Information Toolbar



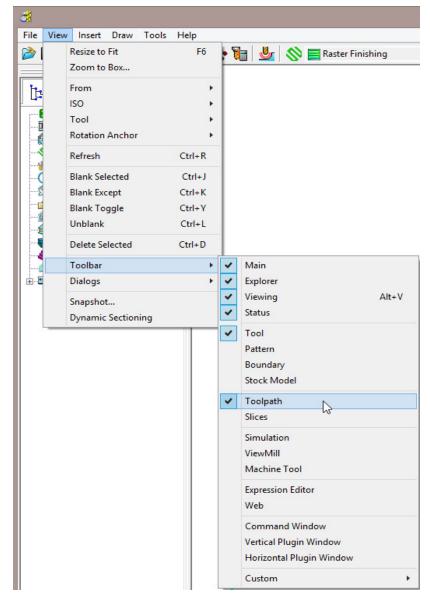
This area provides a reminder of some of the active setup options.

g Tool Toolbar



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

Most available toolbars are not factory defaults, and are therefore not shown at initial start-up. To display any of these, select them under **View > Toolbar**.



For example, select **View > Toolbar > Toolpath** to display the **Toolpath Toolbar**.

The **Toolpath** toolbar appears (as shown below).

D12TR1-RGH1 💿 🔌 🔅 🐶 % 🤭 🐊 😰 🗐 🍋 🏵 🎗 🛛 🔂 🖏 🗱 🖉 🖉 🖉 🖉 🖉 🖉

#### Changing the Background Colour.

To change the background colour of the graphics area, select **Tools > Customise Colours** and from the dialog select **View Background**.

	Tool	ls Help				
		Echo Commands				
		Reset Forms				
		Mirror Project				
1		Plugins				
	•	Snapping				
		Snap Filter	•			
		Filter Style				
		Customise Paths				
		Release Licences	Þ			
		Request Licences	Þ			
		Customise Keyboard Shortcuts				
		Customise Colours				
		Options				
	_		-			

The Top and/or Bottom colours can be changed independently and reset using **Restore Defaults** to restore to the original settings:

Ci	ustomise Colours	?	×
Area Clearance     Collision Checking     Collision Checking	All Colours	Edit Reset Reset All store Defau	
	ciose		

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

### **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 tracker ball.
- 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.

Tolerances     Toolpaths     Tools     Boundaries     View     Dynamic framerate     Animation     Spin view	<del>3</del>	Options ? ×
	<ul> <li>Toolpaths</li> <li>Tools</li> <li>Boundaries</li> <li>View</li> <li>Dynamic framerate</li> <li>Animation</li> <li>Bo Graphics</li> <li>Background image</li> <li>Explorer dimensions</li> <li>Plugin Windows</li> <li>Import</li> <li>Export</li> <li>Project</li> </ul>	Pick radius 5.0 Drawing tolerance 0.1 Spin view □ Use model colour for holes ✔ Dim background entities in editing □ modes □ Display arc/circle centre points ✔

### Mouse button 3: Special Menus & PowerMILL Explorer Options



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

#### **HELP!**

**PowerMILL** provides 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.

File View Insert Draw Tools	Help
🎽 🖥 🧉 🕼 🖉	Contents
	What's New
12. 00 00	Parameters
	Check for PowerMILL Updates
Active	Check for PAF Updates
NC Programs	Subscribe to the PowerMILL Newsletter
No Toolpaths	Visit the User Forum
	Data Translation Services
	Delcam on the Web
- Workplanes	About
🥪 🅪 Levels and Sets	

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: <u>http://forum.delcam.com/</u>
- Telephone and Email Support: UK customers with up-to-date software maintenance can call 01216831010 or mail <u>support@delcam.com</u> 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.

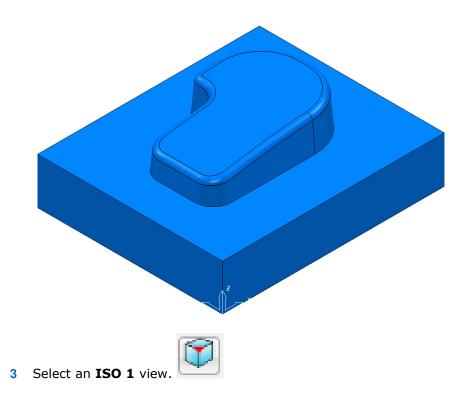
The basic procedure is:

- a Start up **PowerMILL**.
- b Import a Model.
- c Define the **Block** that defines the material size (or 3D machining limits).
- 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:

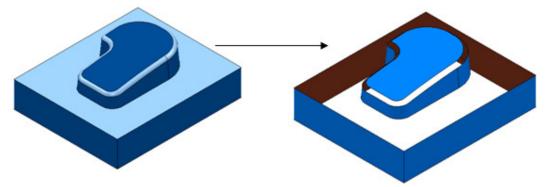
.....\PowerMILL\_Data\Models\PmillGettingStarted.dgk



#### **Blanking model entities**

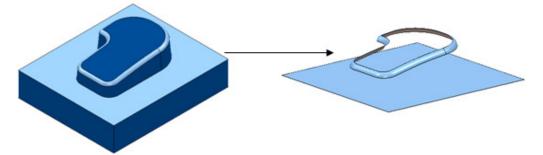
Use the **Blank Selected** (**Ctrl + J**) option available on the **Model** context menu to temporarily hide the selected model components.

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



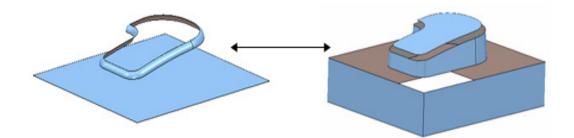
Use the **Blank Except** (**Ctrl K**) option available on the **Model** context menu to temporarily hide all model components except the selected ones.

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.



### **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.

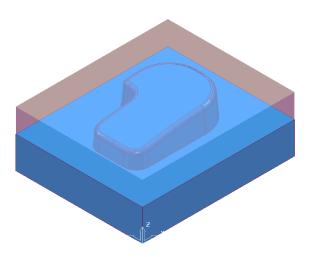
The settings on the **Block** dialog are set by default to **Defined by – Box**. Individual values in the dialog can be edited or locked (greyed out) as required. Enter a value in the **Expansion** field to include an offset in the block calculation.

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.

<b>3</b>	Block	? ×
Defined by		
Box	~	📄 🖪 💥
Coordinate System		
Active Workplane	¥	¥
Limits		
Min	Max	Length
X -0.0	120.0	120.0
Y -0.0	100.0	100.0
z -0.0	43.736684	43.736684
Y Z	X Z	<b>C</b>
Estimate limits		
Tolerance	Expansion	Туре
0.1	0.0	Model 🗸
Include reference	e surfaces	Calculate
✓ Draw		Opacity
A	Accept	Cancel

- 2 Click the **Calculate** button.
- 3 Click Accept.



#### **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.

Tip radiused tool	? ×
Tip Shank Holder Holder profile Cutting data Description	
Tip Shank Holder Holder profile Cutting data Description          Name       D12TR1         Geometry       Image: Comparison of the second	
	X 🗚 👪
Close	

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 Radius 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**.

🖃 样 Tools		
👜 🔆 🕌 > D12TF	21	
🗄 🛛 🤋 📙 BN12		D12TR1
C Boundaries	$\checkmark$	Activate
		ليک Settings

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**

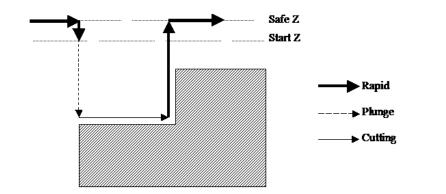
The **Rapid move heights** across the component.



options provide safe control rapid tool movements

- **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.



#### **Setting up Rapid Move Heights**

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

to open the

🥳 Rapid Move Heights ? 🗙				
Geometry				
Safe area Plane 🗸				
Workplane 🗸 🗸				
Normal				
0.0 0.0 1.0				
Rapid height 53.736684				
Plunge height 48.736684				
Calculate dimensions				
Rapid clearance 5.0				
Plunge clearance 5.0				
Calculate				
Drawing options				
Opacity 50				
Draw plunge surface				
Opacity 47				
Apply Accept Cancel				

- 2 Click Calculate.
- 3 Click Accept.

This sets the absolute **Safe Z** and **Start Z** values to be above the **Model** or **Block** by the incremental distances, displayed in the **Calculate dimensions** area of the dialog.

#### **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 **Start and End Point** dialog.



on the Main toolbar to display the

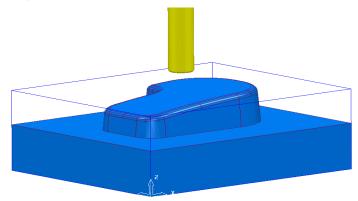
By default, the tool **Start Point** is set to **Block Centre Safe** and the **End Point** is set to **Last Point Safe**.

🥳 Start and End Point 🛛 ? 💌	🥳 Start and End Point ? 🗙
Start Point End Point	Start Point End Point
Method Use Block Centre Safe Override Tool Axis Approach Along Tool Axis Approach Distance 5.0	Method Use Last Point Safe v Override Tool Axis Retract Along Tool Axis v <u>Retract Distance</u> 5.0
Coordinates 60.0 50.0 53.736683	Coordinates 0.0 0.0 53.736683
Tool Axis           0.0         0.0         1.0	Tool Axis           0.0         0.0         1.0
Apply Start Point	Apply End Point
Apply Accept Cancel	Apply Accept Cancel

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.



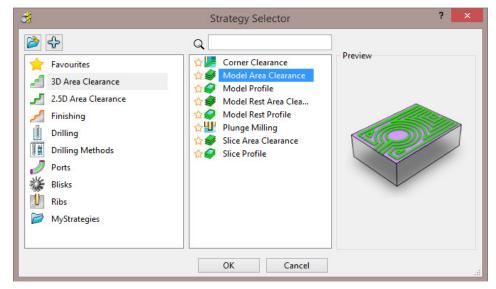
The **D12TR1** tool is placed at the **Block Centre Safe** position ready for you to create the first toolpath.

#### **Creating a Roughing Strategy**

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



2 From the left hand list select **3D Area Clearance** 

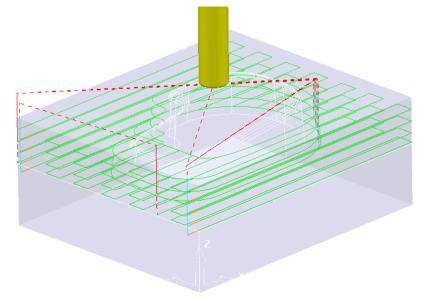


**3** From the right hand list select the **Model Area Clearance** strategy. This displays the **Model Area Clearance** strategy dialog.

Toolpath name D12TR1-RGH1     Workplane   Block   Tool   Model area clearance   Model area clearance   Machine tool   Limit   Machine tool   Limit   Model area clearance   Raster   Step cutting   Wall finishing   Unsafe segment removal   Flat machining   High speed   Order   Approach   Automatic verification   Cutter compensation   Point distribution	Mo	del Area Clearance ? ×
Block Tool Machine tool Machine tool Machine tool Multiple Model area clearance Raster Step cutting Wall finishing Wall finishing Musafe segment removal Flat machining Cut direction Profile Area Climb Any Cut direction Profile Area Climb Nodel area clearance Cut direction Profile Area Climb Nodel area clearance Cut direction Profile Area Climb Nodel area Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Climb Nodel area Climb Nodel area Climb	Toolpath name	D12TR1-RGH1
Machine axis control Machine axis control Rapid move heights General Stepover Stepover Stepover Stepdown Automatic $\sim$ 5.0 Constant stepdown	Block J Tool Machine tool Machine tool Machine tool Machine tool Machine area clearance Raster Raster Raster Model area clearance Raster Model area clearance Raster	Style          Raster <ul> <li>Raster</li> <li>Raster</li> <li>Cut direction</li> <li>Profile</li> <li>Area</li> <li>Climb</li> <li>Any</li> <li>Tolerance</li> <li>0.1</li> </ul> Tolerance       0.1         Thickness       Image: Comparison of the second

4 In the Model Area Clearance strategy dialog:

- a In the **Toolpath** name field, enter **D12TR1-RGH1**.
- **b** Select *Style* **Raster** with *Profile* Climb and *Area* Any.
- 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 **8** (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-RGH1 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**.





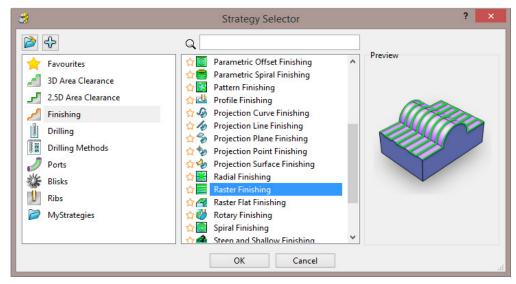
The buttons beyond **Play** enable other actions such as **Pause**, **Step**, **Search**, and **Go to**.

#### **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.



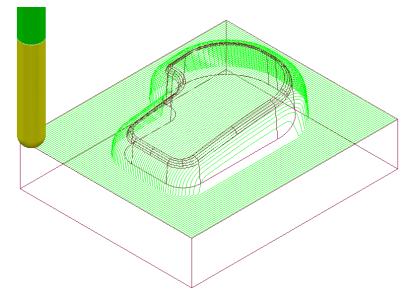
9 From the left hand list select **Finishing**.



10 From the right hand list select the **Raster Finishing** strategy and click **OK**.

<del>3</del> 1	Raster Finishing ? 🗙
Toolpath name	BN12-FIN1
🔛 Workplane	Raster finishing
U Tool 	✓ Fixed direction Angle 0.0
¥ Stock engagement ¥ Raster finishing ¶ High speed	Start corner Lower left 🗸 🗸
Point distribution	Perpendicular pass Perpendicular pass Shallow angle 30.0
Machine axis control	Optimise parallel pass
<ul> <li>↓ Leads and links</li> <li>↓ Leads and links</li> <li>↓ Start point</li> <li>↓ End point</li> <li>↓ Freeds and speeds</li> </ul>	Ordering Style One way  Arc radius 0.0
	Tolerance 0.1
	Thickness
	Stepover 0.3  0.003751

- 11 In the **Raster Finishing** strategy dialog:
  - a For the Toolpath Name, enter BN12-FIN1.
  - **b** Edit the **Stepover** value to **0.3**
  - 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**.



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:

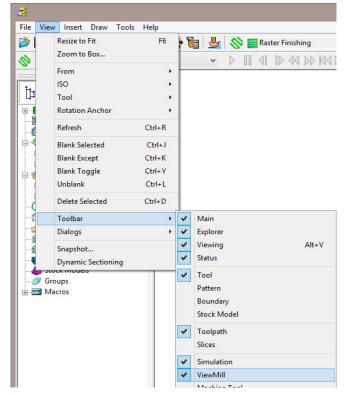
- **Play -** start the simulation and play it in continuous mode.
- Pause pause the simulation.
- Step Back step the simulation back by tool moves. Click the Play button to resume continuous mode.
- Step Forward step the simulation by tool moves. The faster the speed

(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.
  - $\wedge$
- 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 **Play** and **Simulate** the **toolpath**.
- **16 Activate** the finishing toolpath **BN12-FIN1** and repeat the simulation process.

### **ViewMILL**

- 1 Activate the roughing toolpath D12TR1-RGH1 and select it in the Simulation Toolbar.
- 2 From the Main pull down menus, select View > Toolbars > ViewMILL to raise the ViewMILL Toolbar.

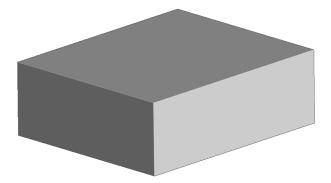


The **ViewMILL** toolbar is displayed. Initially, all the buttons are greyed out.

- 3 Click the <u>use</u> button to toggle the graphics window into the **ViewMILL** mode (The **ViewMILL** toolbar buttons are now active).



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



Click the **Tool** button to display the tool. 5 6 Click the Play button to start the ViewMILL simulation. 🚫 BN12-Raster Finishi 🔻 y di **BN12** • х D12TR1-RGH1 ~ 🙀 D12TR1 ~ **Q** 0 × 5 × [t- 🤇 D 🔽 H-> Active Machine Tools NC Programs NooPrograms
 Organis
 Organis

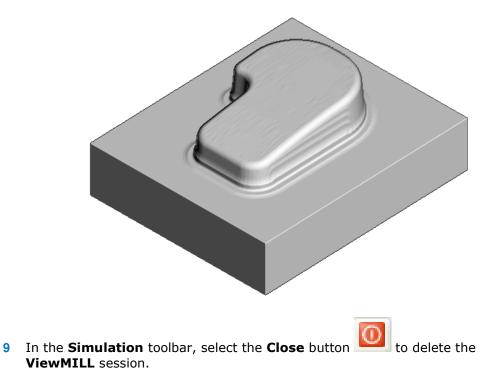
7 After the *simulation* has finished, select the toolpath BN12-FIN1 on the

Simulation toolbar, followed by the tool button

♀ ↓ > D12TR1 ♥ ↓ BN12

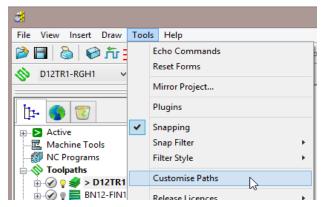
÷... . C Boundaries Batterns Peature Sets Workplanes Levels and Sets Models Stock Models Groups Macros

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

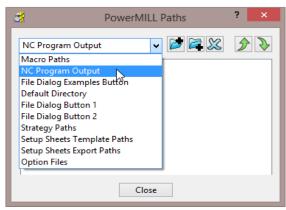


### NC Programs (Post-Processing and NC data Output)

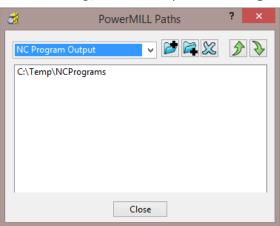
 From the Main menu, select Tools > Customise Paths to open the PowerMILL Paths dialog.



This displays the **PowerMILL Paths** dialog.

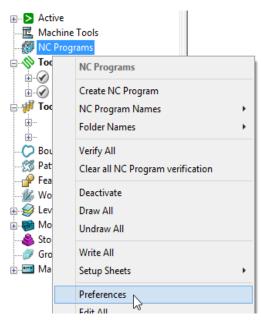


2 In the **PowerMILL Paths** dialog, select the option **NC Programs Output**.



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.



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

- 25	NC Preferences	?	×		
Output	Output Toolpath Fixture offset Verification				
Chang	Changes made here will not change existing NC Programs				
	Use Project Off v Output Folder C:\Temp\NCPrograms				
	Output File {ncprogram}				
	Machine Option File C:\dcam\config\ductpost\heid400.opt		2		
	Machine Tool v Model Location		~		
0	Output Workplane V Part Name				
	Tool Value Tip		~		
	Automatic Tool Alignment On v Connection Moves Move,Ro	tate	~		
	Close				

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.

6	Look in: 🌗	ductpost 🗸	G 🤌 📂 🛄 -	
~	Name		Date modified	Туре
~	A155.opt	t	23/10/1998 14:25	OPT File
2	🔼 haas.opt		19/02/2002 09:09	OPT File
	📇 heid.opt		23/10/1998 14:25	OPT File
~	🔼 heid400.	opt	17/12/2010 09:46	OPT File
1)	🔼 heid426.	op Type: OPT File	24/10/2000 13:47	OPT File
	Moid520	on Size: 50 bytes	13/07/2004 13:24	OPT File
~	📇 heidiso.	Date modified: 17/12/2010 09:46	23/10/1998 14:25	OPT File
2	📇 Hermle(	20U-H530-ZMAXreducedA.opt	07/10/2009 13:49	OPT File
	📇 hurco.oj	ot	23/10/1998 14:25	OPT File
	📇 incon.op	ot	23/10/1998 14:25	OPT File
	📇 kryle.op	t i i i i i i i i i i i i i i i i i i i	23/10/1998 14:25	OPT File
	📇 maho.oj	ot	23/10/1998 14:25	OPT File
	🔼 matsura	opt	23/10/1998 14:25	OPT File
	📇 mazak.o	pt	23/10/1998 14:25	OPT File
	📇 mitsu.op	ot	23/10/1998 14:25	OPT File
	<			>
	File name:	heid400.opt	~	Open

- 7 Select **heid400.opt** and click **Open**. This selects the **heid400.opt** file and returns you back to the **NC Preferences** dialog.
- 8 Close the NC Preferences dialog.
- 9 In the PowerMILL explorer, right-click NC Programs and select Create NC Program.

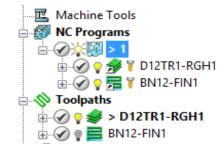
tr- 🌍 🔞			
🐵 > Active			
NC Programs			
🖶 🚸 Toolpaths	NC Programs		
📄 🖶 🗸 💡 🥩 👝			
🗈 🗸 🛊 🚍 Bl	Create NC Program		
🖃 🕡 Tools	NC Program Names		

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

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



Alternatively, right-click on the toolpath **D12TR1-RGH1** 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.

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.

	Information ?	×			
Write NC Program					
	++++++++++++++++++++++++++++++++++++++				
1991, 1992, 1993	clcam plc, Birmingham, England , 1994, 1995, 1996, 1997, 1998, 1999, 2000 , 2004, 2005, 2006, 2007, 2008, 2009, 2010				
Licensed to Delcam Birmingham Development For demonstration use only					
Supplied by Delcam Administration					
Using option file C:\dcam\config\ductpost\heid400.opt Using built-in heid400 Reading cutfile records					
	d	<b>v</b>			

- **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**

15 On the Main toolbar, click the Save button.





As this is the first time that this **Project** is saved, the **Save Project As** dialog will open.

<b>3</b>		Sav	e Project As			x
	Save in: 🌗	PowerMILL_Projects	~	G 🤌 📂 🛄 -		
	Name	*		Date modified	Туре	
<u>♦</u>		Νο	items match y	our search.		
	<					>
	File name:				✓ Save	
	Save as type:	PowerMILL Project			✓ Cancel	

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

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

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

17 From the *pulldown* menus, select File > Delete All, followed by Tools > Reset Forms.

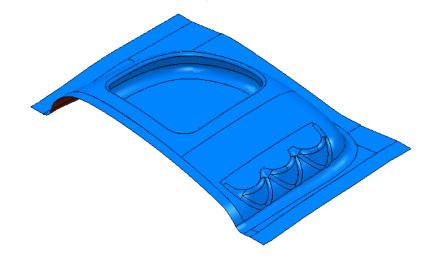
The content of **PowerMILL** *explorer* is deleted and all dialogs are reset to their default settings.

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

#### **Additional Exercise**

1 Import the Model:-

.....\PowerMILL\_Data\Models\facia.dgk



#### 2 Save the Project as:

- .....\COURSEWORK\PowerMILL\_Projects\Facia-1
- **3** Use the same **Tools** and **machining strategies** to those in the previous **Project** (PmillGettingStarted).
- 4 Once this exercise is completed (and saved as a **Project**) locate the *pulldown* menus, and select **File** > **Delete All** followed by **Tools** > **Reset Forms**.

## 2. Machining Setup

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

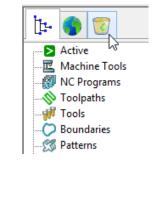
We will now look at the machining Setup options in more detail.

In particular:

- Setting up direct access to regularly used files using **Customise Paths**.
- Setting Feed rates.
- Workplanes (Positional datums) to create appropriate machining set ups.
- Tool with Shank and Holder definition.

#### **PowerMILL Panes**

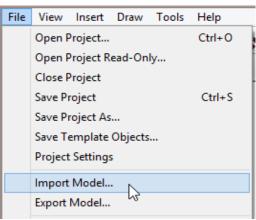
The **PowerMILL** panes are located above the **PowerMILL** explorer area.



- The first pane displays the PowerMILL explorer (as shown above).
- The second pane <u>the second pane</u> displays the **HTML** browser used for viewing **Help** files.
- The third pane opens the PowerMILL Recycle Bin. Entities deleted from the PowerMILL explorer can be recovered from the Recycle Bin.

#### Setting up direct access to regularly used folders

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



The **Import Model** dialog is displayed.

A	Look in: 🌗 PowerMILL_Data	v 🗿 🌶 📂 🖽 -	
~	Name	Date modified	Туре
~	five_axis	10/09/2014 12:07	File folder
	InteractiveMacroTraining	08/07/2014 14:18	File folder
	MachineData	05/02/2015 08:50	File folder
~	Models	11/05/2015 14:27	File folder
	Patterns	08/07/2014 14:18	File folder
_	Pictures	08/07/2014 14:18	File folder
~	\mu pmill2-2016	28/07/2015 09:15	File folder
2	Projects	22/06/2015 09:33	File folder
_	SampleMacros	08/07/201/ 1/19	File folder
	<		>
	File name:	~	Open

- 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 the **PowerMILL** installation 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:

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

💏 PowerMILL Paths	? 🔀
File Dialog Button 1 🗸 📝 🛃	⋧ 🌶 📎
C:\Users\Train\PowerMILL_Data	
Close	

- 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

Select Path           Navigate to the desired path, creating new folders if necessary. Press OK to accept or Cancel to quit.			
Documents Examples User Path 1	<ul> <li>PowerMILL_Data</li> <li>Five_axis</li> <li>Five_axis</li> <li>InteractiveMacroT</li> <li>MachineData</li> <li>Models</li> <li>Patterns</li> <li>Pictures</li> <li>pinill2_C</li> <li>Projects</li> <li>SampleMacros</li> <li>setup-sheets</li> <li>Triangles</li> </ul>	-	^
>       PowerMILL_Data-FiveAxisOnly         >       PowerMILL_Data-NoFiveAxis         >       PowerMILL_Data-NoFiveAxis         >       PowerMILL_Data-NoFiveAxis         >       PowerMILL_Data-NoFiveAxis         >       PowerMILL_Data-NoFiveAxis         >       PowerMILL_Data-NoFiveAxis         >       PowerMILL_Data-NoFiveAxis			
Make New Folde	r	ОК	Cancel

- 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

6	Look in: リ	Models	v 🌀 🤌 📂 🗔 🗸				
<b>V</b>	Name	^	Date modified	Туре	Size		
	wheel_s	egment.dgk	11/03/2003 12:41	DGK File		243 KB	
2	WingMi	irrorDie.dgk	21/11/2001 16:16	DGK File		54 KB	
*	WingMi	irrorPunch.dgk	20/11/2001 12:09	DGK File		56 KB	
3	Workpla	anesExample1.dgk	02/11/2011 09:26	DGK File		155 KB	
>	Y-Branc	hDuct.dgk	03/01/2014 10:53	DGK File		125 KB	
	Yoghur	t tray.dgk	09/04/2003 14:02	DGK File		337 KB	
5	Zlimits.	dgk	26/06/2012 09:37	DGK File		86 KB	
<b>V</b>	File name:	WorkplanesExample1.dgk			v	Open	

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

Files of type:	Delcam Models (*.dgk;*.ddx;*.ddz;*.psmodel;  Cancel
	Delcam Models (*.dgk;*.ddz;*.psmodel;doc.det;*.pic)
	] 3rd Party Models (*ig*;*.vda;*fic*;*.model;*.dxf;*.stp;*prt*;*.sldprt;*.par;*.pfm;*x_t;*.xmt_txt;*.x_b;*.xmt_bin;*.mf1;*.3dm)
	All Triangles (*.tri;*.dmt;*.stl;*.ttr;)
	Delcam Geometry (*.dgk;*.ddx;*.ddz)
	Delcam Electrodes (*trode)
	DUCT Picture (*.pic)
	CATIA ("fic*, *.model,*.CATPart)
	AutoCAD (*.dxf)
	IGES (*ig*)
	Ideas (*.mf1;*.prt)
	Inventor (* ipt)
	Rhino (* 3dm)
	ProENGINEER (*.prt*)
	Cimatron (*.pfm)
	Solidworks (*.sldprt)
	Solidedge (*.par)
	STEP (*.stp;*.step) VDA (*.vd*)
	Unigraphics (*.prt)
	Parasolid files (*x_t;*xmt_txt;*x_b;*xmt_bin)
	Delcam Machining Triangles (*.dmt)
	Stereo Lithography Triangles (".stl)
	DUCT Triangles (*.tri)
	Brockware (tm) Triangles (*ttr)
	ACIS (*.sat)
	WinMAX (* mxd)
	All Files (*)
	e anna C

*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 Resize to fit

The **model** is re-scaled to fit within the **PowerMILL** graphic area.

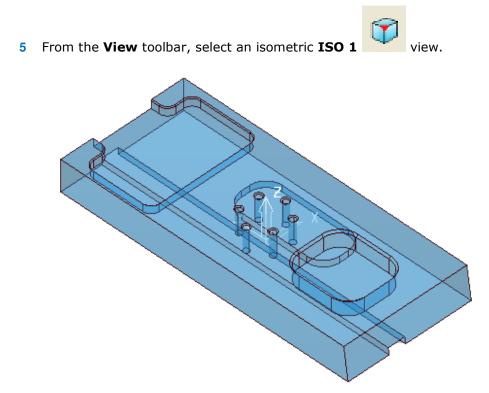
#### Viewing the Model

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.



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.

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

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

#### **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 **I** on the **main** toolbar to display the **Feeds and Speeds** dialog.

💰 Feeds and	d Speeds ? ×
Toolpath Properties Toolpath: (none active) Type Finishing Operation General	Tool Properties Tool Diameter Number of Flutes Overhang
Tool/Material Properties         Stock Material:         Surface Speed         Image: 0.0       m/min         Feed/Tooth         Image: 0.0       mm         Axial Depth of Cut         Image: 0.0       mm         Radial Depth of Cut         Image: 0.0       mm         Working Diameter	Cutting Conditions Spindle Speed 1500.0 rpm Cutting Feed Rate 1000.0 mm/min Plunging Feed Rate 500.0 mm/min Skim Feed Rate 3000.0 mm/min
Depth of Cut     Surface Slope       0.0     mm     0.0       Overhang Compensation	Standard V
Reset Apply	Accept Cancel

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' if there is currently no *active* **toolpath**.

## **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 physically move 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 no **workplane** is active, then the original global **Transform** is the datum.

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

# Workplane (1) - At Top of Selection

- 7 With the left mouse button depressed, drag a box over the entire **mode**l to select it.
- 8 In **PowerMILL explorer**, right-click **Workplanes** and select **Create and Orientate Workplane > Workplane at top of Selection**.

- 🎉 Workplane	s			
		Workplanes		
Stock Mod		Create Workplane		
Ø Groups		Get Workplanes From Model	_	
ia		Create and Orientate Workplane	÷	Workplane at Point
		Workplane Names	•	Multiple Workplanes
		Folder Names	•	Workplane From Three Points
	Draw All Undraw Al	Deactivate Draw All Undraw All		Workplane From Three Arc Points Workplane Aligned to Geometry Workplane Positioned Using Block
		Create Folder	l	Workplane at Top of Selection Workplane at Centre of Selection
	_	Delete All		Workplane at Bottom of Selection

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

9 Right-click the newly renamed Workplane 1 and from the local menu 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**.

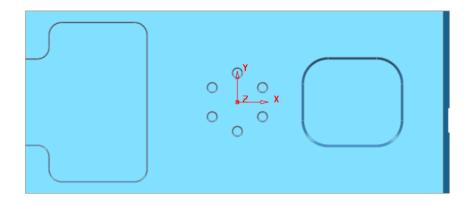
 From the same local menu, select Workplane Editor to display the Workplane editing toolbar.

Name 1	1	X 🛛 🕅	ţ¥ ţZ	6	*	<b>通</b> り	6	
		1						

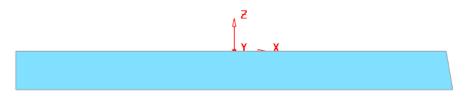
- 11 Double click the **Twist about Z** button. This displays the **Twist** dialog.
- 12 Enter Angle 270 and click Accept.

- 33	Twist	?	×
	Angle 270	5	
A	ccept	Cance	el

13 Click the **Green Tick** located to the right of the **Workplane** toolbar to accept the changes.



14 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.

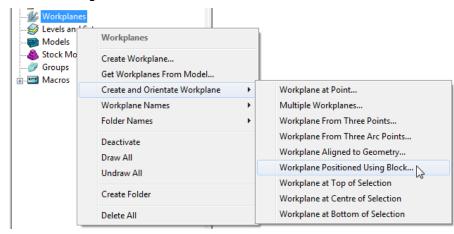
# Workplane (2) - Positioned using the Block

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

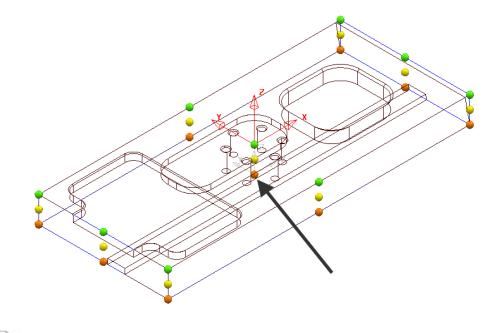
15 From the **main** menu, select the **Block** Icon **W** the following form will appear select **Active Workplane** press **Calculate** followed by **Accept**.

<del>3</del>	Block	?	×
Defined by			
Box	~	۵ 🖪 🖉	ζ
Coordinate System			
Active Workplane	~		<u>×</u>
Limits			
Min	Max	Length	
Х -175.0	175.0	350.0	
γ -75.0	75.0	<b>150.0</b>	
Z -30.0	-0.0	30.0	
X Y Z	X X Z	6	<b>v</b>
Estimate limits			_
Tolerance	Expansion	Туре	
0.1	0.0	Model	×
Include reference	surfaces	Calculate	
✓ Draw		Opacity	_
Ac	cept	Cancel	

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



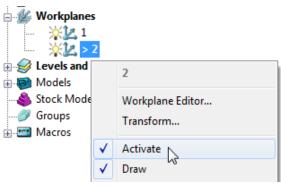
- 17 Select Workplane Positioned using Block from the above menu.
- **18** Click the new **Workplane** on to the orange *sphere* located at the *centre* of the *bottom face* of the **Block**.

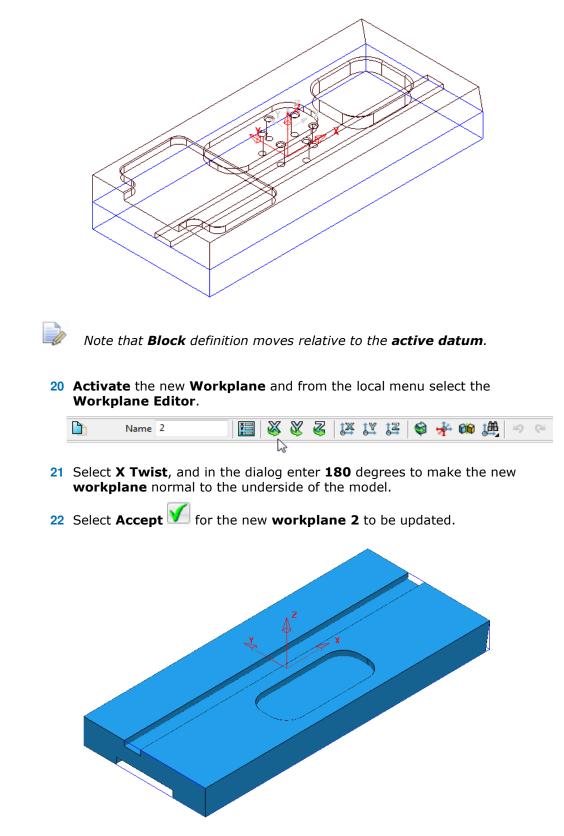




The new workplane will inherit the default name 2.

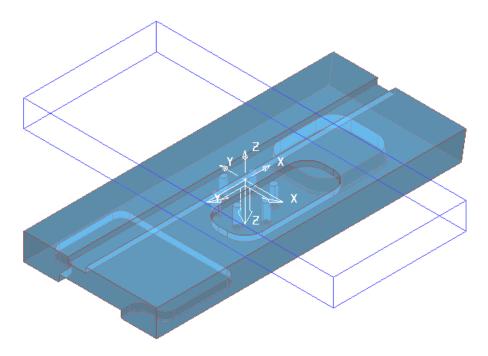
**19** Right click on **workplane 2** and from the local menu, select **Activate**.





The **Block** returns to the correct position relative to the **model** which in this case is dimensionally the same for both **Workplanes 1** and **2** when either is **Active**.

23 De-activate Workplane 2.



The **Block** is now dimensionally positioned relative to the global **Transform**.

24 Select File - Save Project:-

The contents of the **Project** saved earlier will be updated.

# **Block Coordinate System**

The above type of **Block** Co-ordinate, definition (**Active Workplane**) is ideal for 3 Axis applications. The physical re-positioning of the **Block** will provide a useful visual warning if the machining datum, **Workplane** is de-activated.

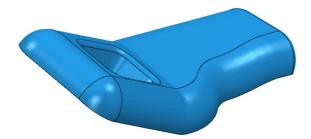
📆 Block	? ×
Defined by	
Box 👻	P 🖪 🐹
Coordinate System	
Active Workplane 🗸	▼
Active Workplane	
Global Transform	
Named Workplane	Length
x −0.0 🖳 350.0	S 350.0 S

Other **Coordinate System** options become essential for advanced applications such as multi-axis, *3Plus2* set ups. In this case the **Block** definition can be fixed to either the **Global Transform** or a **Named Workplane**.

# Workplane Aligned to Geometry

This option enables the user 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 Select File - Delete all followed by Tools - Reset forms.



2 Import the Model:-

#### .....\PowerMILL\_Data\Models\InteriorTrim.dgk

3 With the centre mouse wheel held down, move the mouse to rotate the view of the **Model** so that the base of the pocket is fully visible.



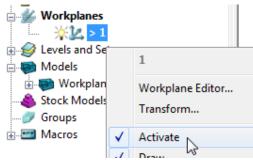
4 Right click on Workplanes and select Create and Orientate Workplane followed by Workplane aligned to Geometry.

- 🕍 Workpla		_
🥩 Levels ar	Workplanes	
🔏 Stock M	Create Workplane	
- Ø Groups	Get Workplanes From Model	
⊕ Macros	Create and Orientate Workplane	Workplane at Point
	Workplane Names	Multiple Workplanes
	Folder Names	Workplane From Three Points
	Deactivate	Workplane From Three Arc Points
	Draw All	Workplane Aligned to Geometry
	Undraw All	Workplane Positioned Using Block
		Workplane at Top of Selection
	Create Folder	Workplane at Centre of Selection
	Delete All	Workplane at Bottom of Selection

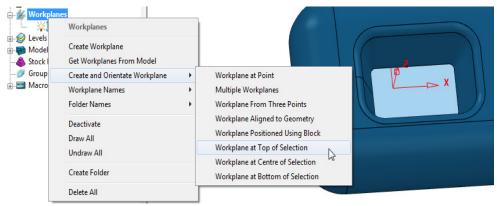
5 Left mouse click on the base of the pocket (as shown below) to locate a new Workplane (with its Z-axis normal to the surface).



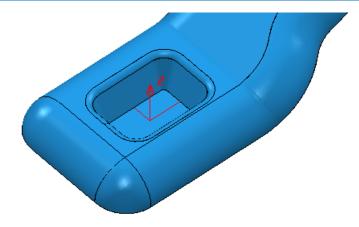
6 In the **PowerMILL** *explorer*, right click on the new **workplane** and select **Activate**.



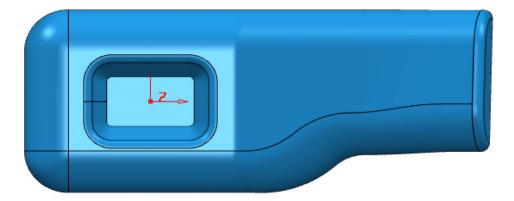
- 7 Use the left mouse and click on the bottom face of the pocket to make it selected (shaded light blue below right).
- 8 Then right click on **Workplanes** to open the local menus below.



- 9 From Create and Orientate Workplane, select Workplane at Top of Selection.
- 10 The new **Workplane** will be centrally located at the base of the pocket.
- **11 Rename** the new **Workplane** as **Datum** and make it **Active**.
- **12 Delete** the original **Workplane**.



The new **Workplane** is central to the pocket base as shown below.



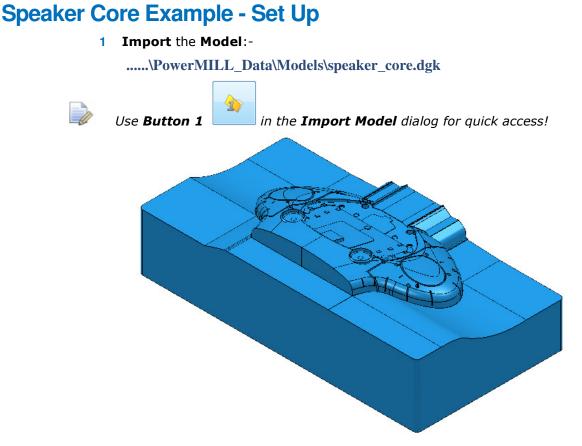
13 Save the Project as:-

- .....\COURSEWORK\PowerMILL\_Projects\Workplane-Example2
- 14 Select File Delete all followed by Tools Reset forms.

# Other local Workplane menu, creation options

- a **Create Workplane** produces a single **Workplane** on the current active datum and opens the **Workplane Editor** context toolbar.
- **b Create and Orientate Workplane** opens the following form, that provides a selection of specialised **Workplane** creation options.

V	Vorkplane at Point
N	Aultiple Workplanes
v	Vorkplane From Three Points
v	Vorkplane From Three Arc Points
v	Vorkplane Aligned to Geometry
v	Vorkplane Positioned Using Block
v	Vorkplane at Top of Selection
v	Vorkplane at Centre of Selection
v	Vorkplane at Bottom of Selection



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

- 2 Depress the left mouse key and drag a box across the whole model to select all Surfaces.
- **3** From the **PowerMILL** *explorer*, select **Workplanes 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**.

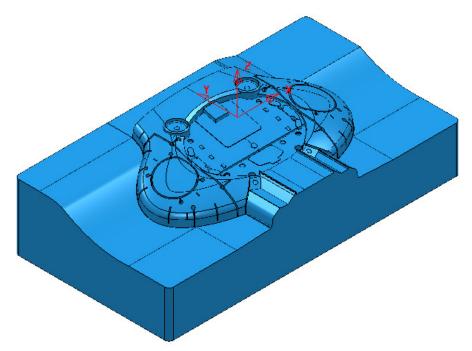


5 In the above toolbar, click on the **Z-Twist** option and enter **90** in the form for the **Workplane** rotation angle.

📆 Twist	8 ×
Angle 90	<u>S</u>
Accept	Cancel

- 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.

# **Tool and Holder Definition**

For the cutting **Tools** in this example the **Shank** and **Holder** will be included as part of the overall definition.

Note it is also possible to store specific **feeds** and **speed** values in a Tool definition. Not a default setting, but **PowerMILL** can be set to recognise these and automatically update the **Feeds and Speed** dialog on activation of the **Tool**.

Bou	Tools		
🔐 🌈 Fea	Toolbar		
⊡ ∰ Wo	Create Tool	•	From Database
j	Tool Names	•	End Mill
⊕ 🥪 Lev ⊕ 📾 Mo	Folder Names	•	Ball Nosed
A Sto	Deactivate		Tip Radiused
🧭 Gro	Draw All		Tapered Spherical
Ма	Undraw All		Tapered Tipped

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

	Tip radiused tool		? ×
Tip Shank Holder Ho	Ider profile Cutting data Us	ser Defined Settings	Description
Geometry Length 160.0 Tip radius 6.0 Tool status Vali	neter	Tool assembly Q 100%	v
		×	🕅 🗚 🚮
	Close		

10 In the **Tip Radiused Tool** dialog, input a **Name D50TR6** - **Diameter 50** - **Tip Radius 6** - **Length 160** - **Tool Number 1**.

11 In the Shank dialog, click Add a shank component and enter: Upper/Lower Diameter 50 and Length 100.

	Helder metile Cutting data User Defined Settin	ne Description
Tip Shank Holde	r       Holder profile       Cutting data       User Defined Settin         Tool assemi       Image: Color of the setting o	bly
	Connection	

12 In the Holder dialog, click Add holder component and input:

- Upper Diameter 120
- Lower Diameter 90
- Length 50
- Overhang 200



The **Overhang** is the vertical distance from the bottom of the **Holder** to the bottom of the tool **Tip**.

<b>3</b>	Tip radiused tool ?	×
Tip Shank Holder	Holder profile Cutting data User Defined Settings Description	on
Components Holder n	name 1 Q 100% V	_
	- XX X	
	•\$\$\$\$\$ 🔯 🖻 🗌	7
	Dimensions Upper diameter 120.0	
	Lower diameter 90.0	
	Length 50.0 Ignore	
	Overhang 200.0	
	Gauge length 260.0	
	Connection	
		3 🚱
	Close	

**13** In the **Holder** dialog, click the **Add holder component** button<sub>#</sub>. Enter:

- Upper Diameter 120
- Lower Diameter 120
- Length 50

<del>3</del>	Tip radiused tool	? ×
Tip Shank Hold	er Holder profile Cutting data User	Defined Settings Description
Components		Tool assembly
Holde	r name 1	Q 100% v
	- 🕅 📜	
	Dimensions	
	Upper diameter 120.0	
	Lower diameter 120.0	
	Length 50.0	
	Ignore 🗌	
	Overhang 200.0	
1	Gauge length 300.0	
	Connection	
	~	
		x 🕅 🥸 🕅
	Close	

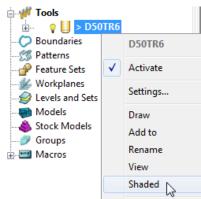
14 In the Holder dialog, click the Add holder component button Enter:

- Upper Diameter 150
- Lower Diameter 150
- Length 25

8			Tip	radiused to	ol	? ×
Tip	Shank	Holder	Holder profile	Cutting data	User Defined Setting	s Description
	Shank ponents	Holder n	ame 1	<ul> <li>Image: Second state</li> <li>Image: Second state&lt;</li></ul>	Tool assemble	
				Close	×	XX × 1

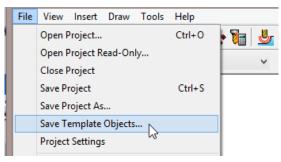
**15 Close** the dialog.

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**.





A **Tool** can be saved outside **PowerMILL** as a permanently stored template (.ptf).





The tool must be the only item in the **PowerMILL explorer** as all saveable entities will also be added to the template file.

# **Block Definition**

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).

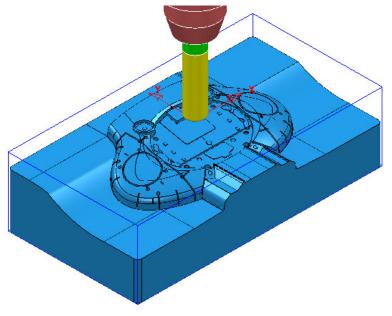
The dimensions of the **Block** can be entered manually or calculated directly to the Type of entity: **Model**, **Boundary**, **Pattern**, or **Feature**.

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

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

<del>3</del>	Block	? ×
Defined by		
Box	~	🔌 🔳 🗶
Coordinate System		
Active Workplane	¥	~
Limits		
Min	Max	Length
X -330.0007	330.00078	660.00156
Y -175.0040	175.00403	350.00807
Z -187.5025	-	187.50256
X Y	¥	<b>C</b>
Estimate limits		
Tolerance	<b>Expansion</b>	Туре
0.1	0.0	Model 🗸
Include reference	e surfaces	Calculate
✔ Draw		Opacity
A	ccept	Cancel

17 Move the **Opacity** slider fully to the left for a wireframe **Block**.



# **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).

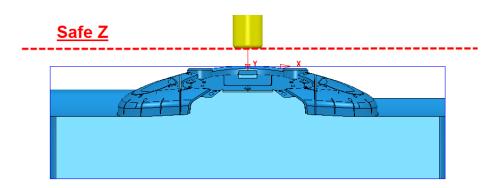
18 In the Rapid Move Heights dialog, click the Calculate tab to set Safe Z and Start Z to the default values of 10 and 5 respectively above the Model or Block.

😚 Rapid Move Heights ? 🗙
Geometry
Safe area Plane 🗸
Workplane 🗸 🗸
Normal
0.0 0.0 1.0
Rapid height 10.0
Plunge height 5.0
Calculate dimensions Rapid clearance 5.0
Plunge clearance 5.0
Calculate
Drawing options
Draw rapid surface
Opacity 20
Draw plunge surface
Opacity 20
Apply Accept Cancel



The **Rapid Move Heights** are calculated relative to the highest point on the **Block** or **Model**.

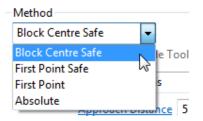
If parts of the **model** are selected, then the de-selected parts will be ignored when the **Rapid Move Heights** are calculated!



By default the **Tool** will traverse across **Safe Z** and then drop to **Start Z** using a **Rapid** move (eg **G0**). It will then continue downwards at the **Plunging Feed Rate** (eg **G1**).

# 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.



- 19 In the Start and End Point dialog, Accept the default settings :
- c Start Point Block Centre Safe
- d End Point Last Point Safe

Start Point End Point
Use Block Centre Safe 🗸 🗸
Override Tool Axis Approach Along Tool Axis Approach Distance 5.0
Coordinates 0.0 -0.0 10.0
Tool Axis         0.0         1.0         1.0
Apply Start Point Apply Accept Cancel

🥶 Start and End Point ? 🗙
Start Point End Point
Method
Use Last Point Safe  V Override Tool Axis
Retract Along Tool Axis V
Retract Distance 5.0
Coordinates 0.0 0.0 10.0
Tool Axis         1.0
Apply End Point
Apply Accept Cancel

# **Save Project**

- 20 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.
- 21 Select File Save Project As:-

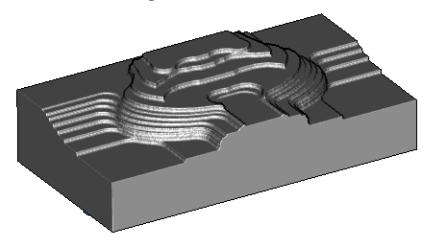
....\COURSEWORK\PowerMILL-Projects\Spkr-Core



<u>Do Not Close</u> the **Project** as it will be continued in the following chapter.

# 3. 3D Area Clearance

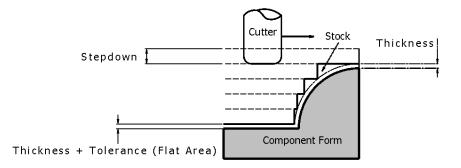
The main strategies for roughing a 3D component model are called **3D Area Clearance**. These provide a choice of 2D material removal methods, which progressively machine the area (**Slice**), up to the component contour, down a sequence of user-defined **Z Heights**.



# **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**.

- 1 Ensure that the recently saved **Project** (*Spkr-Core*) is still active.
- 2 Activate both the Workplane and the tool D50TR6.
- 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 and from 3D Area Clearance open a Model Area Clearance strategy.

<b>3</b>	Strategy Selector		?	×
Favourites     JD Area Clearance	Q       Corner Clearance       Image: Second S	Preview		

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

Workplane	
Block	Model area clearance
Tool	Style
Machine tool	<u>All</u>
🕅 Limit	
🖶 🥩 Model area clearance	
Raster	
Step cutting	Raster
	Cut direction
	Profile Area
	Climb v Any
 ↓ High speed	
	Tolerance
Automatic verification	0.1
Cutter compensation	
Point distribution	Thickness
Tool axis	
Machine axis control	
	Stepover
🗄 🗍 Leads and links	20.0
	Stepdown
+ Treeds and speeds	Automatic 🗸 🦞 10.0
History	✓ Constant stepdown
Notes and Description	
	Rest machining

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

- 7 In the Model Area Clearance dialog:
  - a Enter the Toolpath name as D50TR6-A1
  - **b** Select **Style** as **Raster**.
  - c For Cut direction Area select Any
  - d Set Thickness to 0.5
  - e Set Stepover to 20
  - f Set Stepdown to 10
- 8 In the Model Area Clearance local *explorer* select Limit.

Block	<u>6</u>	~
l l		_

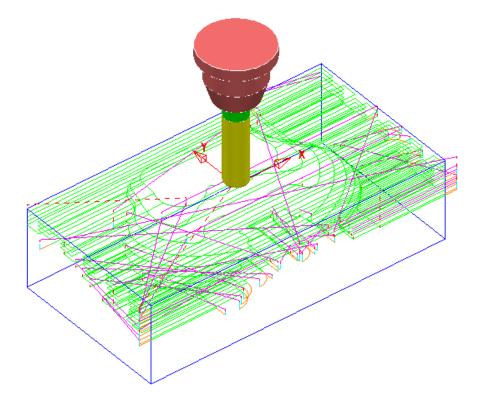
- 9 In the **Limit** options, **Block** category, switch to *limit tool center to block edge*.
- **10** Keeping all other values as default, **Calculate** the strategy.
- 11 After the **Toolpath** has been created, close the dialog.





Double left mouse click on the explorer symbol (in this case  $\circledast$ ) to activate or de-activate the **Toolpath**.

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



# **Rapid Move Heights combined with Leads and Links**

You can modify the basic **Safe Z** and **Start Z** from the **Rapid Move Heights** dialog.

😚 Rapid Move Heights ? 🗙
Geometry Safe area Plane
Workplane 🗸 🗸
Normal
Rapid height 10.0
Plunge height 5.0
Calculate dimensions
Rapid clearance 5.0
Plunge clearance 5.0
Calculate
Drawing options Draw rapid surface
Opacity 20
Draw plunge surface
Opacity 20
Apply Accept Cancel

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).



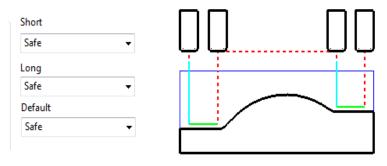
Safe Z is always above Start Z.

- **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.

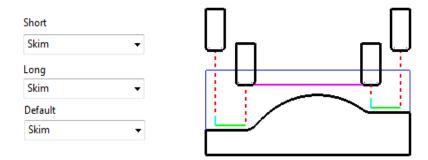
**Rapid Move Heights** works in conjunction with the **Leads and Links** dialog with providing a more comprehensive choice of **Toolpath** link options.

×	Ŀ	eads and Lin	ks	? ×
Z heights	First lead in	Lead in	Lead out	Last lead out
Extensio	ns	Links	Point	distribution
	Short/Long thresh	nold 10.0		
Retract and a	pproach moves		Short	
	Along Tool axis	~	Safe	~
- 🗸 Automat	ically extend		Long	
	Maximum leng	th 250.0	Safe	~
			Default	
	Retract dista	nce 0.0	Safe	~
	Approach dista	nce 0.0		
Arc fit rapi	d moves			
		Arc radius (to	ol diameter unit	s) 0.25
Gouge chee	k			
Use polar li				
ose polar in				Apply links
	Apply	Accept	Cancel	

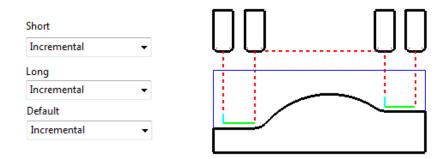
 Safe – Applies the plunge feed rate from a fixed height above the job. This is more predictable and reassuring, but the non-cutting (air) moves takes more time and is inefficient, especially in the case of large, deep components.



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.



The default **Link** move settings in the **Model Area Clearance** strategy template are **Short/Long = Skim** and **Default = Incremental**.

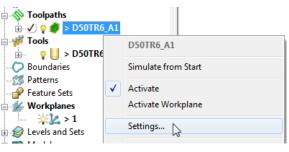
### **Feed Rates**

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 **III** and enter the values (shown below) in the **Cutting Conditions** area of the dialog.

Cutting Conditions
Spindle Speed (Recommended: 1200 rpm)
Cutting Feed Rate (Recommended: 400 mm/min)
Plunging Feed Rate (Recommended: 40 mm/min)
Skim Feed Rate 3000.0 mm/min

- 2 Click Apply to assign these values to the active Toolpath (D50TR6-A1) and then select Accept to close the dialog.
- 3 In the **PowerMILL** explorer, right-click on the toolpath and select Settings....

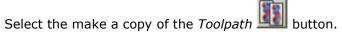




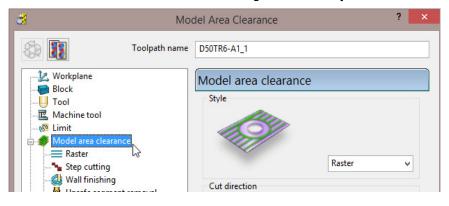
5

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

4 This reopens the Model Area Clearance dialog.



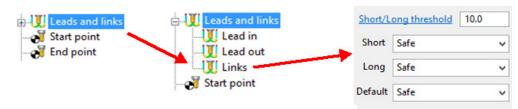
6 On the left of the **Model Area Clearance** dialog is a local **explorer** window.



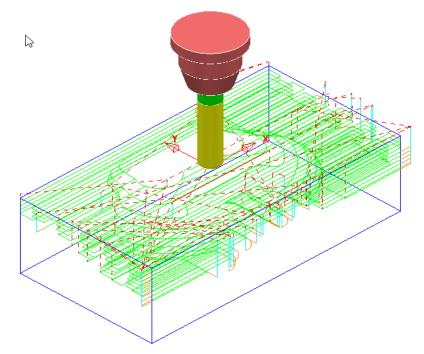
When a local **explorer** option is selected the individual page is displayed in the main dialog. This makes it easier to navigate specifically through the pages for the required strategy.

di Ma	odel Area Clearance ? ×
Toolpath name	D50TR6-A1_1
Workplane Slock Jool Machine tool Machine tool Model area clearance	Raster Fixed direction Angle 0.0
Raster Step cutting	Cut direction

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 In the Links page change the Short, Long, and Default link moves to Safe.
- 9 Click **Calculate** to create a new **Toolpath** with the modified settings. Once processing is complete, close the dialog.



The **Tool** now performs rapid moves (dashed red) at a fixed height (**Safe Z**) above the part (Default height is **10**).

All **Plunge** feedrate (pale blue) moves start from a height between the **Safe Z** and the top of the part (default height is **5**)

# **Statistics**

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

1 In the **PowerMILL** *explorer*, right-click the latest **Toolpath D50TR6-A1\_1** and from the local menu select **Statistics**.

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

( <b>1</b> )		
🥳 Toolpath Statisti	CS	? ×
Entity Solution D501	[R6-A1_1	~
Leads and Links		
	Length	Time
Rapid	14001.073784	0:04:40
Plunge	2577.074961	0:51:32
Ramp	0.0	0:00:00
Others	1383.180954	0:02:45
Total	17961.329699	0:58:58
Cutting Moves		
	Length	Time
Linear	50474.336296	1:40:56
Arcs	177.797471	0:00:21
Total	50652.133767	1:41:18
Total	68613.463466	2:40:16
		Lifts 47
	Close	

In this case, the total machining time is around 2hrs 40mins

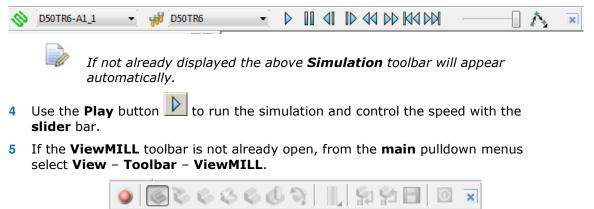
2 Right mouse click on the original **Toolpath D50TR6-A1** and from the local menu select and view the **Statistics**.

In this case, the total machining time is around **2hrs**.

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

# Simulating the toolpath

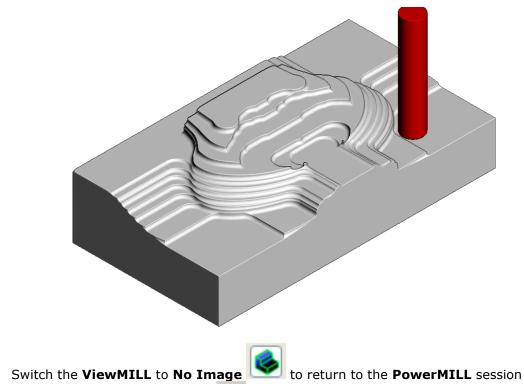
3 Right mouse click on the Toolpath D50TR6-A1 and from the local menu select Simulate from Start.



6 In the **ViewMILL** toolbar, select the **Red** sphere to connect **ViewMILL** to the simulation controls (The sphere will now be **Green** and the current **Block** will be loaded into **ViewMILL**).



7 Select the Plain Shaded Image option and run the simulation again



and then disconnect **ViewMILL I** from the **simulation** process.

When you toggle back to **PowerMILL**, the **ViewMILL** session still exists in the background, ready to include further toolpaths as required.

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

# **Saving the Project**

8

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

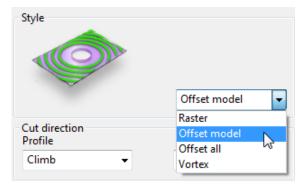
The **Project** (....\COURSEWORK\PowerMILL-Projects\Spkr-Core) is now updated.

We will now *delete* the contents of the **PowerMILL** *explorer* and return the **PowerMILL** session back to the original default settings.

- 10 From the Main menu, select File > Delete All.
- 11 From the Main menu, select Tools > Reset Forms.

# **Other Model Area Clearance - Style examples**

There are 4 different *Style* options in the **Model Area Clearance** strategy.



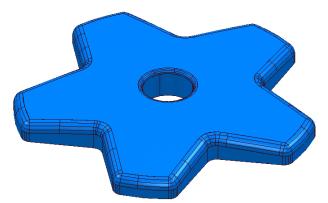
We have already used **Style** – **Raster** in the previous example. So for the next section we will simply show the basics of other 3 options (**Offset Model**, **Offset All**, and **Vortex**) on a single Z level.

# **Offset Model – Basic 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 go a long way to 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

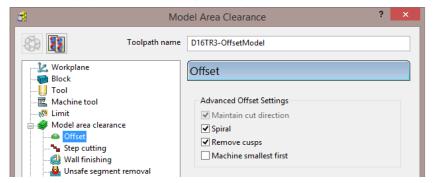


- 3 Create a *Dia* 16, *Tiprad* 3, Tool named D16TR3.
- 4 Calculate a Block <sup>1</sup> to the full model dimensions.
- 5 Lock the Z max and Z min values, enter an Expansion value of 10 and Calculate.
- 6 In the Rapid Move Heights 🔜 dialog, select Calculate.

- 7 Select Toolpath Strategies , and from 3D Area Clearance open a Model Area Clearance strategy.
- 8 Input data into the pages of the form exactly as shown below.

	WIO	del Area Clearance ?
💱 🚺 Toolp	ath name	D16TR3-OffsetModel
Workplane Block Tool	^	Model area clearance Style
		Offset model V
الله Wall finishing He Unsafe segment remova Plat machining Tr High speed Order J Approach	al .	Cut direction Profile Area Climb V Climb V Tolerance 0.1
Automatic verification     Cutter compensation     Point distribution     Tool axis     Machine axis control     Apid move heights		Thickness 
U Leads and links     Start point     End point     Feeds and speeds     History     Notes and Description		Y   2.0   Stepdown     Automatic   Y   20.0     Y   Constant stepdown

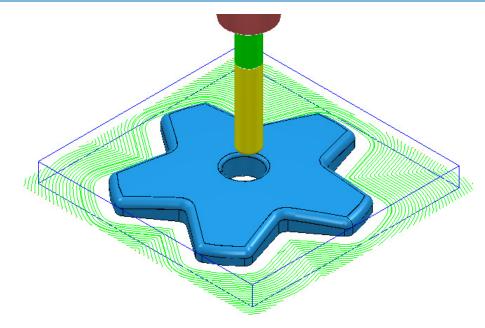
9 Select the **Offset** page and tick the **Spiral** option.



**10** Select the **Flat machining** page and select **Machine flats** - **Off**.

Workplane	Flat machining
Tool	Machine flats
	Off
🔞 Limit	
🚔 🛷 Model area clearance	Multiple cuts
Offset	Number of cuts Stepdown
Step cutting	1 1.0
	Final stepdown
Flat machining	1.0

- 11 At the bottom of the dialog, click **Calculate** to create the **Toolpath**.
- **12** Close the dialog and select an **ISO 1** view.



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

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

Machine Tool     More Programs     More Pr		
	D16TR3-hsm	
Patterns	Simulate from Start	
⊕ Peature S ⊕ 🖗 Workplan	Activate	10

14 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. The material removal rate in this type of strategy is generally very consistent. The strategy can be further improved using advanced settings to progressively smooth away sharp corners and straighten the toolpath geometry as it offsets further away from the component form.

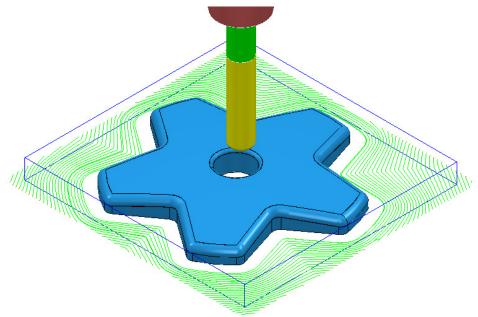
**15** In the **PowerMILL** *explorer*, right-click the active **D16TR3-OffsetModel** toolpath and select **Settings** to reopen the **Model Area Clearance** dialog.



- **16** Click **III** to create an active copy of the **toolpath**.
- 17 In the Model Area Clearance dialog, select the High speed page.

di Mo	del Area Clearance ? ×
Toolpath name	D16TR3-OffsetModel_HSM
Workplane     Block	High speed
U Tool 	✓ Profile smoothing Radius ( tool diameter units )
<ul> <li>Model area clearance</li> <li>Offset</li> <li>Step cutting</li> <li>Wall finishing</li> </ul>	■ 0.2
Unsafe segment removal	25.0 %
<b>↓</b> High speed <b>↓</b> Order <b>↓</b> Approach	Trochoidal moves
Lutomatic verification      Luter compensation      Point distribution	10.0 %
Fond axis     Tool axis     Machine axis control	Links Smooth

- 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\_OffsetModel\_HSM** with the improved smoothing characteristics.



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

2

# **Offset All – Basic example**

This variant of the **Offset Area Clearance** is better suited to softer materials such as modelling foam, resins etc. It will remove material with a larger **stepover** and the minimum amount of lifts, but this is more than likely to result in the tooltracks containing sharp changes of direction and an inconsistent removal rate.

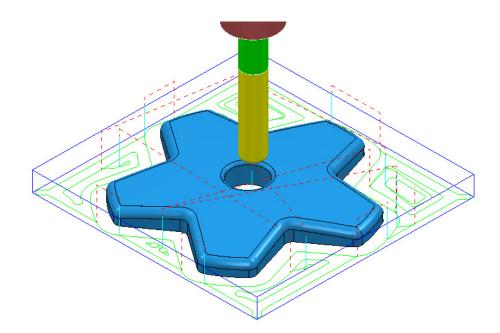
- 1 In the **PowerMILL** *explorer*, right-click the active **D16TR3-OffsetModel Toolpath** and select **Settings** to reopen the **Model Area Clearance** dialog.
  - Click to create a copy of the **Toolpath**.
- 3 Rename the new toolpath as D16TR3-OffsetAll.
- 4 In the **Model Area Clearance** dialog, main page select **Style Offset All** and then input data as shown below.

💏 Mo	del Area Clearance	? ×
Toolpath name	D16TR3-OffsetAll	
Workplane Block Tool Tool Machine tool Machine tool Machine tool Multinishing Model area clearance Multinishing Multinishing Multinishing Multinishing Multinishing Flat machining Flat machining Flat machining Flat machining Flat machining Flat machining Multinishing Multinis	Model area clearance   Style   Offset all   Cut direction   Profile   Area   Climb   Tolerance   0.1   Thickness   0.5   Stepover   6.0   Stepdown   Automatic   Yee   Offset all	
$\checkmark$	Calculate Queue OK Cancel	

5 In the **Model Area Clearance** dialog, select the **High speed** page and update the data as shown in the following illustration.

💰 M	odel Area Clearance ? 🗙
Toolpath name	e D16TR3-OffsetAll
Workplane     Block     Tool     Machine tool     Model area clearance	High speed ✓ Profile smoothing Radius ( tool diameter units )
Offset     Step cutting     Wall finishing     Wall finishing     Flat machining     Flat machining     Flat machining     Flat machining	Raceline smoothing
Order     Approach     Guter compensation     Order     Order	Trochoidal moves Maximum overload 10.0 % Links Smooth

- 6 At the bottom of the dialog, click **Calculate** to create the **Toolpath**.
- 7 Close the dialog and select an **ISO 1** view.
- 8 Perform a toolpath **Simulation** to observe the tool movement in this strategy.



In the above **Offset-All** style toolpath, the tool plunges into the individual areas and profiles it's way outwards following both the component **model** and **Block** forms.

# **Vortex - Basic example**

This specialist, variant of **Model Area Clearance** is primarily designed for use with solid carbide tools on hard materials. Wherever necessary the **Vortex** strategy applies complex, optimised trochoidal, movement to achieve the ultimate control over the loading on a cutting tool.

- 1 In the **PowerMILL** *explorer*, right-click the active **D16TR3-hsm\_1 Toolpath** and select **Settings** to reopen the **Model Area Clearance** dialog.
- 2 Click and create a copy of the **Toolpath**.
- 3 In the Model Area Clearance dialog, main page select Style Vortex.

		del Area Clearance	
-22 III	Toolpath name	D16TR3-Vortex	
Workplane Block B	removal cation n	Model area clearance         Style         Vortex         Cut direction         Profile       Area         Climb       Climb         Tolerance       0.1         Thickness       0.5         Stepover       1.0         Stepover       1.0         Stepdown       Automatic         Automatic       1.0         Rest machining	<ul> <li>✓</li> <li>✓</li> <li>✓</li> </ul>
C.			
<b></b>		Calculate Queue OK Cancel	

- a Input Thickness 0.5
- **b** Input **Stepover 1**
- c Input Stepdown 20
- 4 Select the Flat machining page and set *Machine flats* to Off.

Model Area Clearance			×
Toolpath name	D16TR3-Vortex		
Workplane	Flat machining		
<mark>U</mark> Tool <b>E</b> Machine tool	Machine flats		
Imit ⊡	Multiple cuts		
Vortex Step cutting Wall finishing	Number of cuts     Stepdown       1     1.0		
Unsafe segment removal	Final stepdown		

5 Select the Vortex Page and set Minimum Radius (r) to 3.0

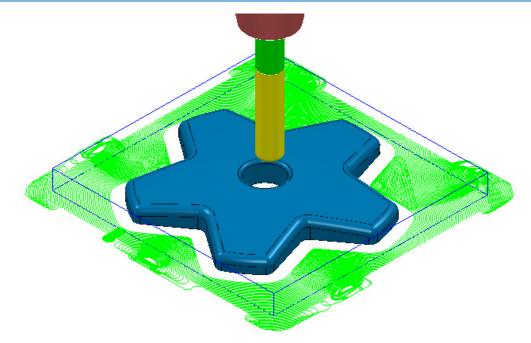
Mo	del Area Clearance	?	x
Toolpath name	D16TR3-Vortex		
Workplane	Vortex		
U Tool 	Cutting feed rate 500.0 mm/min	ភិ	
	Minimum radius (r)	1	
- 🙀 Unsafe segment removal 	Minimum point spacing           Image: Output spacing           Image: Output spacing	1	
Order	Lift on return moves		

Increasing the **Minimum Radius** value will enlarge the inherent **Trochoidal** moves which should in turn enable a higher **Feedrate** to be used with the **Vortex** strategy.

6 Select the *Unsafe segment removal* page, tick **Remove segments smaller than threshold** and input a *Threshold* (TDU) value of **0.5** 

Mo	del Area Clearance	?	×
Toolpath name	D16TR3-Vortex		
Image: Second system         Image: Second system	Unsafe segment removal         ✓ Remove segments smaller than threshold         Threshold (tool diameter units)         0.5         Only remove segments from enclosed areas		

- 7 At the bottom of the dialog, click **Calculate** to create the **Toolpath**.
- 8 Close the dialog and select an **ISO 1** view.



9 Select File - Save Project to update the contents of:-.....\COURSEWORK\PowerMILL\_Projects\Handle

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

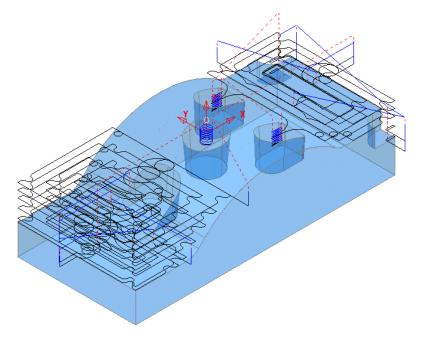
# Area Clearance – Unsafe Segment Removal

This option is designed to filter out machining of confined areas that 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 **Model Area Clearance** dialog.
- 5 Click the **Create a new toolpath based on this one** button, to create a copy of the toolpath.

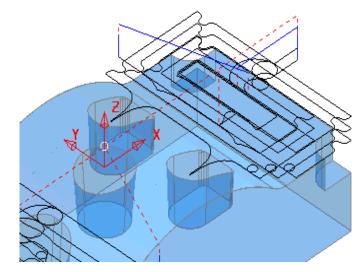


- 6 **Rename** the copy of the toolpath as **AreaFilter**.
- 7 In the Model Area Clearance dialog, select the Unsafe segment removal page.
- 8 Tick the option, *Remove segments smaller than threshold*.
- 9 Input a Threshold (tool diameter units) value of 0.5
- 10 Un-tick the **Only remove segments from enclosed areas** option.

🛃 Mo	del Area Clearance ? ×
Toolpath name	AreaFilter
Workplane     Block     Tool     Tool     Machine tool     Model area clearance     Offset     Step cutting     Wall finishing     Wall finishing     Flat machining	Unsafe segment removal  Remove segments smaller than threshold  Threshold (tool diameter units)  0.5  Only remove segments from enclosed areas

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

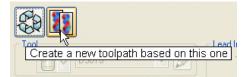
11 **Calculate** the **Model Area Clearance** dialog and once processing is complete, close the dialog.



**Unsafe Segment removal** has been applied to prevent the cutter attempting to plunge into and machine confined areas (Pockets).

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

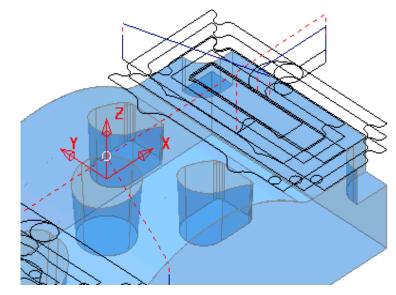
- 12 In the **PowerMILL** *explorer*, right-click the toolpath named **AreaFilter** and select **Settings** to open the original **Model Area Clearance** dialog.
- **13** Click **Create a new toolpath based on this one** button, to create a copy of the toolpath.



- **14 Rename** the copy of the toolpath as **AreaFilter-ProfileSmooth**.
- 15 In the Model Area Clearance dialog, select the High Speed page.
- 16 Select the **Profile Smoothing** option with the **Radius (Tool Diameter Units)** value set to **0.2**

di Mo	odel Area Clearance	?	×
Toolpath name	AreaFilter_ProfileSmooth		
Workplane Block Machine tool Machine tool	High speed         ✓ Profile smoothing         Radius ( tool diameter units )         0.2         Raceline smoothing		
Flat machining	25.0		%

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



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

- 18 From the Main pull down menus, select File > Save Project to update the stored data (.....\COURSEWORK\PowerMILL-Projects\MountingBlock).
- 19 From the Main pull down menus select File > Delete All.
- 20 From the Main pull down menus select Tools > Reset Forms.

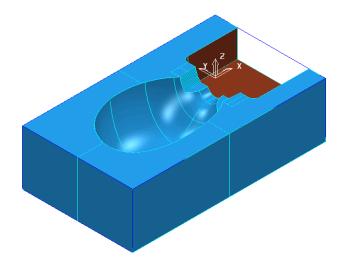
## Model Rest Area Clearance example

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).

**1 Import** the **model**:

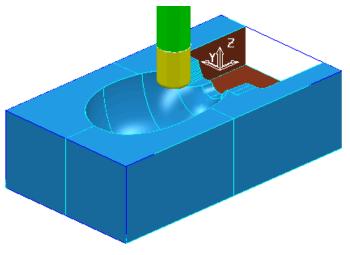
.....\PowerMILL\_Data\models\WingMirrorDie.dgk



2 Save Project as:

### .....\COURSEWORK\PowerMILL-Projects\WingMirrorDie-Ex1

- 3 Create a Tip Radiused tool of *Dia* 40 *tiprad* 6 and Name D40TR6.
- 4 From the Main toolbar open the Block dialog and Calculate using the default settings, Defined by Box to and Type Model.
- 5 Using the default settings, select **Calculate** in the **Rapid Move Heights**.



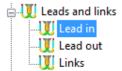
6 Select Toolpath Strategies , and from 3D Area Clearance open a Model Area Clearance strategy.

<b>3</b>	Mo	del Area Clearance	?	×
£2 🚺	Toolpath name	D40TR6-D1		
Workplane Block Block Glock G	clearance ting shing egment removal hining sed h tic verification pensation ution s control heights nks	Model area clearance         Style         Offset mod         Cut direction         Profile       Area         Climb       Climb         Tolerance       0.1         Thickness       0.5         Stepover       20.0         Stepdown       Automatic         Automatic           Constant stepdown       Rest machining	lel v	
		Calculate Queue OK Cancel		

- 7 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 20
  - e Input Stepdown 10
- 8 In the local *explorer*, select the Limits page, and set the **Block Limit** to Limit tool centre to block edge.

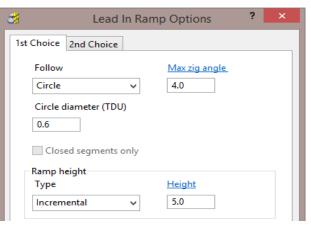


9 In the local *explorer*, expand the Leads and Links options, and then select Lead In.

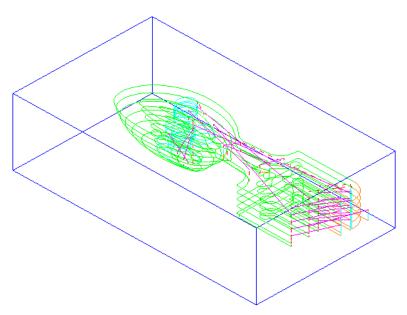


10 In the Lead in window, select 1st Choice - Ramp - Ramp options...

**11** Input the following **Ramp Options**:



- 12 Click Calculate to process the Model Area Clearance toolpath.
- 13 Select an **ISO 2** view.



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

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

	Colparts Colpa	Image: TR6-D1       D40TR6-D1       Simulate from Start       Activate		
∆40TR6-D1      ✓     √	D40TR6 V P		©,	×

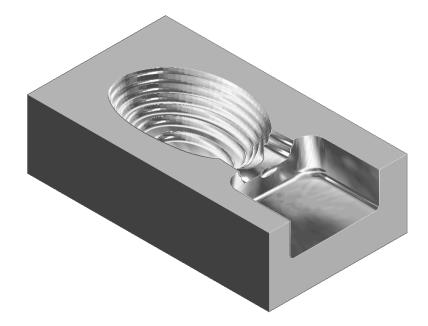
**15** Ensure that the **ViewMILL** toolbar is also active.

D Boundaries

🚳 🗞 🕹 🕹 🍪 🍕 📙 💱 😂 🗃 **16** Switch the **ViewMILL On/Suspend** setting to a green sphere **W**, and then select the *shiny* shaded image icon button to start the simulation. 17 Press the Play D40TR6-D1 ✓ d D40TR6 × 0 ۵ 0 × 0 Undrawing the tool will speed up the simulation. You can control the display of the tool by left clicking on the tool entity, light bulb in explorer. Toolpaths Toolpaths 🗄 🐗 Tools 🖮 💓 Tools 🔮 📙 > D40TR6 🔆 📙 > D40TR6 ÷---÷...

The finished result indicates that in some areas of the model there is still too much material for the finishing strategies to cope with.

🔿 Boundaries



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.

18 Click to suspend ViewMILL.
19 Click storeturn to the PowerMILL window.

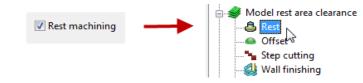
### **Rest Roughing using a Reference Toolpath**

- 1 Create a **Tip Radiused** tool of Diameter **16**, Tip Radius **3** and name it as **D16TR3**.
- 2 Select Toolpath Strategies , and from 3D Area Clearance open a Model Rest Area Clearance strategy.

💰 Mode	I Rest Area Clearance ? ×
Toolpath name	D16TR3-D1
Workplane Block Tool Machine tool Minit Model rest area clearance Rest Offset Step cutting Wall finishing Unsafe segment removal Flat machining Tool axis Automatic verification Point distribution Approach Automatic verification Notel axis Machine axis control Rapid move heights Col axis Automatic verification Flat machinel Automatic verification Notes and Jacks History Notes and Description	Style   Offset model   Cut direction   Profile   Area   Climb   Climb   Tolerance   0.1   Thickness   0.5   Stepover   1.0   Stepover   1.0   Stepdown   Automatic   V   S.0   Constant stepdown   Rest machining
	Calculate Queue OK Cancel

By directly selecting **Model Rest Area Clearance**, the **Rest machining** options and the most suitable style, **Offset Model** will already be active in dialog.

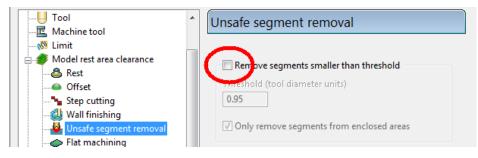
- a Enter a new Toolpath name D16TR3-D1
- b Keep the Thickness as 0.5
- c Enter Stepover as 1
- d Enter Stepdown as 5
- e Select the **Rest** page from the strategy **explorer**.



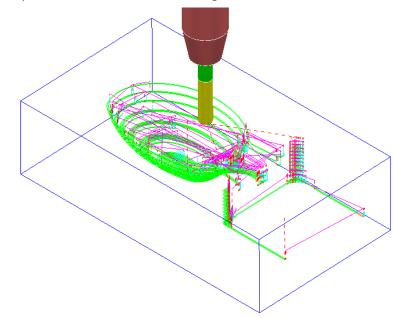
f Set the **Rest machining** type as **Toolpath**, enter **D40TR6-D1** as the reference toolpath, and input *Detect material thicker than* as **0.6** 

Workplane	^	Rest
U Tool 		Rest machining Toolpath V D40TR6-D1 V
		Detect material thicker than 0.6
		Expand area by 1.2

g In the local *explorer*, select **Unsafe segment removal** page and if set, *untick* the **Remove segments smaller than threshold** option.



- h Activate the D16TR3 tool to be used in the new toolpath.
- 3 Click **Calculate** to process and create the additional **Offset Model Area Clearance** toolpath and then **Close** the dialog.

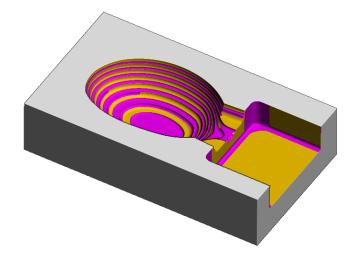


4 From the Main pull down menus select File - Save Project to update: .....\COURSEWORK\PowerMILL-Projects\Wing\_Mirror\_Die

- 5 Switch the ViewMILL On/Suspend button to a green sphere
- 6 Select the Rainbow Shaded Image
- 7 Right mouse click on the toolpath **D16TR3-D1** and select **Simulate from Start**.
- 8 Press the **Play button** to start 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 successfully accessed 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 storeturn to the PowerMILL window.

### **Rest Machining using a Stock Model**

An alternative method for rest roughing 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 include the effect of all previous toolpaths currently registered to that Stock Model.
- A toolpath that was 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**.

In this example a **Stock Model** is created to initially include the **D40TR6-D1** toolpath. Then, a copy of the toolpath, **D16TR3-D1** is created using the **Stock Model** for the *rest machining* reference.

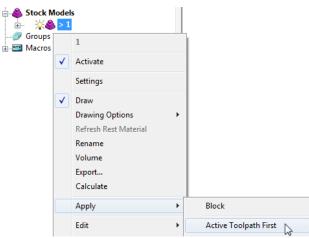
1 In the **PowerMILL** explorer, Activate the toolpath **D40TR6-D1**.



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

Macros Toolbar Tolerance Stepover 0.1 2.0 Import Rest thickness Workplane	2.0
Impart	
	117.1 MILLER
0.2	

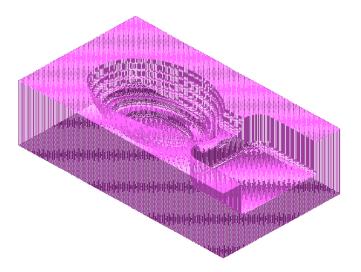
- 3 Accept the above default values for the *triangle mesh* creation.
- 4 In the PowerMILL explorer, right-click the newly created Stock Model 1 and select Apply > Active Toolpath First.



Both the currently active Toolpath and Block are added to Stock Model.



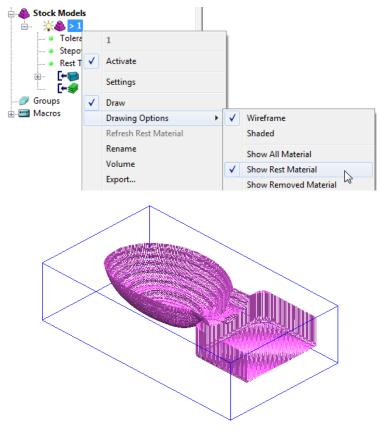
5 From the same menu, select **Calculate** to physically update the **Stock Model**.



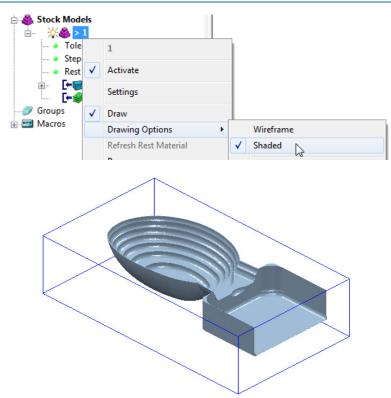
A triangular mesh will appear representing the state of the material after the effect of the first machining strategy, **D40TR6-D1**.

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**.



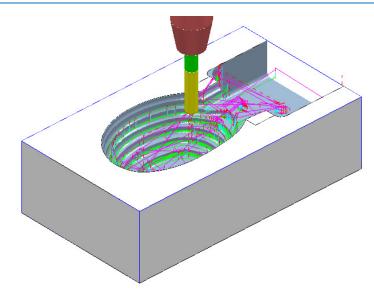
7 From the same menu, select **Drawing Options** > **Shaded** to display the shaded view of the **Stock Model**.



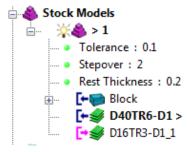
- 8 In the **PowerMILL** explorer, Activate the toolpath D16TR3\_D1.
- 9 Right-click on the toolpath D16TR3\_D1 and select Settings to open the Model Rest Area Clearance strategy dialog.
- **10** Click **11** to create a copy of the toolpath.
- 11 Select the **Rest** page and modify the settings as below using **Stock model 1**.

🛃 Mod	lel Rest Area Clearance	?	×
Toolpath nam	e D16TR3-D1_1		
Workplane     Block     Jool     Tool     Machine tool     Well rest area clearance     Model rest area clearance     Gfset     Gfset     Step cutting     Wall finishing     Wall finishing     Flat machining     Flat machining     Flat machining     Flat machining     Grder     Approach     Automatic verification     Point distribution	Expand area by	0.5 0.0 25.132741	]

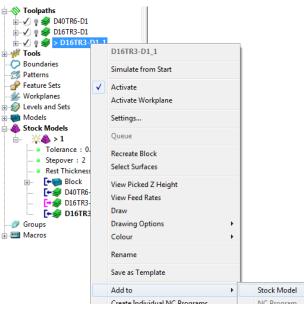
- 12 Click **Calculate** to create an alternative and more efficient **Rest Machining** toolpath named **D16TR3-D1\_1**.
- **13 Close** the machining strategy dialog.



Although the toolpath **D16TR-D1\_1** appears (coloured purple) as a referenced entity in the **Stock Model** list, the resultant machining effect is not included.



14 In the **PowerMILL** *explorer*, right-click the toolpath, **D16TR3-D1\_1** and select Add to > Stock Model.

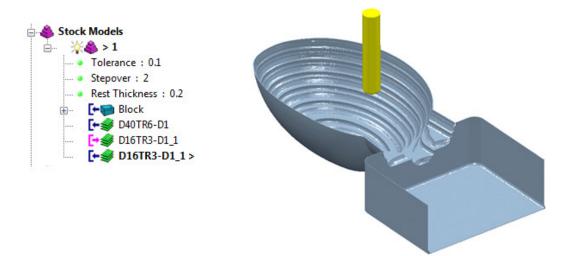




At this stage, the **Stock Model** will disappear from the screen.

6

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



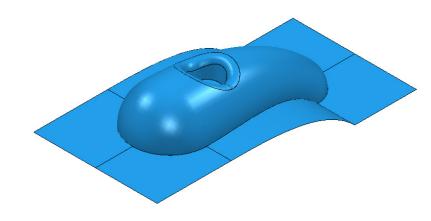
- 16 From the Main pull down menus, select File > Save Project to update: .....\COURSEWORK\PowerMILL-Projects\WingMirrorDie
- 17 From the Main pull down menus select File > Delete All.
- 18 From the Main pull down menus select Tools > Reset Forms.

## **Exercise**

# Area clearance with both unsafe segment removal and rest roughing

1 Import the Model:

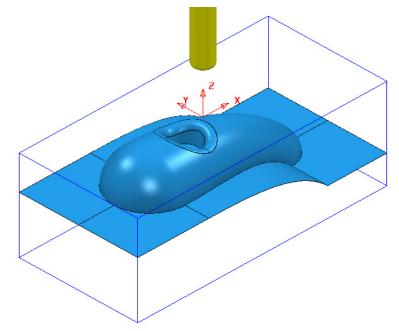
.....\PowerMILL\_Data\Models\Cowling.dgk



2 Select an **ISO 1** view.

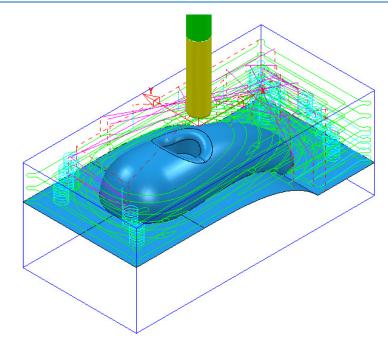
### **Machining setup**

- 3 Create the following tools:
  - Tip Radiused tool Name D16TR3, Diameter 16 Tip Radius 3
  - Tip Radiused tool Name D12TR1, Diameter 12 Tip Radius 1
- 4 Create a Workplane centrally position at the *top centre* of the Model, and then Activate it.
- 5 Calculate a Block using *Defined by* Box and Type Model, then change the Min Z value to -80 and lock it by selecting the adjacent padlock.
- 6 **Calculate** the **Rapid Move Heights** using the default settings.

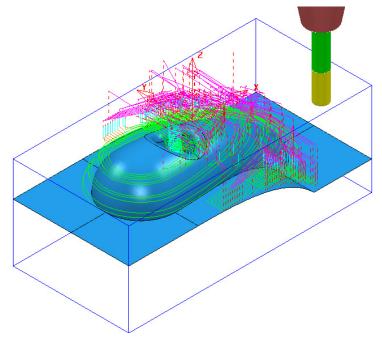


### **Area Clearance Strategies**

- 1 Create a Model Area Clearance strategy named D16TR3-RGH1 using:
  - a The D16TR3 tool.
  - **b** The *Style* as **Offset All**.
  - c A *Thickness* of **0.5**
  - d Stepover 8 and Stepdown 12
  - e In the Flat Machining page use Level.
  - f Select the Unsafe segment removal page.
  - g **Tick** the box **Remove segments smaller than threshold** and leave the other settings as default.



- 2 Create another Model Area Clearance strategy named D12TR1-RGH1 using:
  - a The D12TR1 tool.
  - **b** The *Style* as **Offset Model**.
  - c Stepover 2 and Stepdown 3
  - d In the Flat Machining page use Area.
  - e Enable the **Rest machining** page and reference to the previous toolpath.
  - f Disable the **Unsafe segment removal** option.

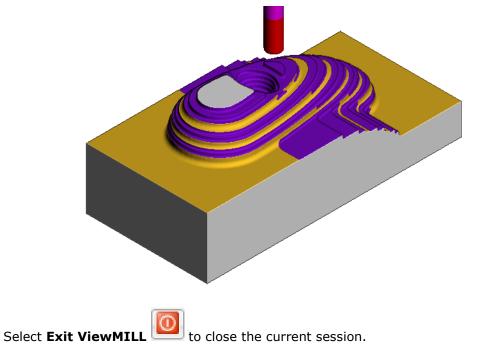


3 Save Project As:

.....\COURSEWORK\PowerMILL\_Projects\ModelAreaClear-Exercise

### **ViewMILL Simulation**

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



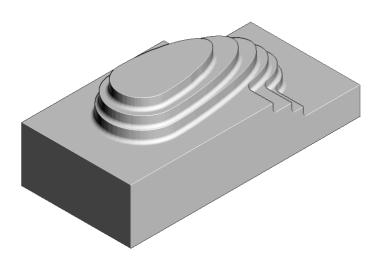


5

Do not close the **Project** as it will be continued in the following section.

### 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 would normally be reduced using a separate **rest** roughing operation. However, if it is acceptable to use the same **Tool** then the **Step** option will provide rest machining within the same **Area Clearance** strategy. In this case additional passes are applied (where required) that step upwards between the original **Z Heights**.



3

- 1 Activate the existing toolpath D16TR3-RGH1 to restore the settings used.
- 2 Run a new ViewMILL simulation (as shown above) on the toolpath D16TR3-RGH1.
  - $\odot$

Select **Exit ViewMILL** <sup>LEE</sup> to close the current session.

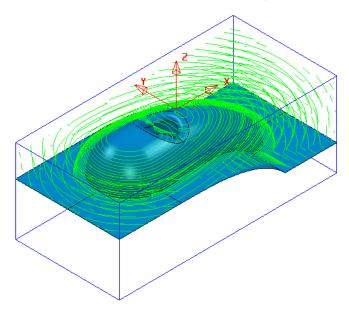
- 4 Right mouse click on the active toolpath D16TR3-RGH1 and from the local menu select Settings.
- 5 Select the 'Make a Copy' of the toolpath option



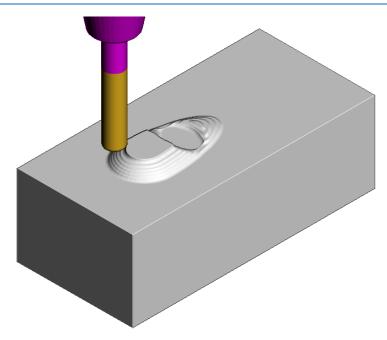
- 6 Rename the new toolpath as D16TR3\_Step-RGH1
- 7 In the main page of the dialog change the *Style* to **Offset model**.
- 8 In the Model Area Clearance, local *explorer*, select Step cutting.
- 9 Tick the Step cutting box and enter values exactly as shown below.

<b>3</b>	Model Area Clearance ?		
Toolpath nam	D16TR3_Step-RGH1		
Workplane	Step cutting		
	✓ Step cutting		
	Stepup 1.0		
☐	Detect material thicker than 0.2		
Step cutting	Expand area by 0.4		
	Constant volume removal		

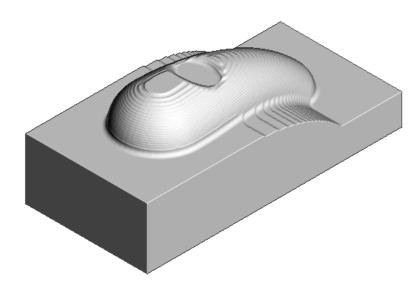
#### 10 Select Calculate to create the toolpath D16TR3\_Step-RGH1.



11 Run a new ViewMILL simulation on the toolpath D16TR3\_Step-RGH1.



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 strategy.

### **12 Save** the **Project**.

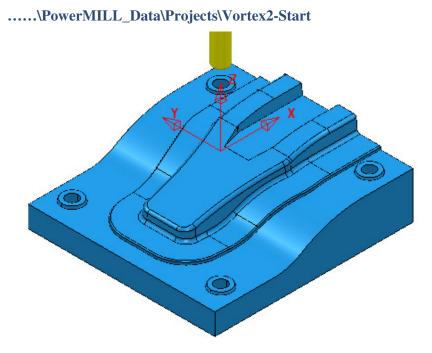
- **13** From the **Main** pull down menus select **File** > **Delete All**.
- 14 From the **Main** pull down menus select **Tools** > **Reset Forms**.

### Model Area Clearance – Vortex example

**Vortex** machining is an **area clearance** strategy which rapidly removes material from a 3D part while controlling tool load. **Vortex** is best suited to solid carbide tools and is frequently used in combination with **step** cutting.

The engagement angle never exceeds that produced by a straight line cut with a given *stepover*. This eliminates excessive tool load and all full-width cuts. This enables you to increase feed rates. As the tool approaches the maximum engagement angle, the strategy automatically changes to use a trochoidal path to avoid tool overload.

1 Open the (read-only) Project:-



2 Save Project As:-

.....\COURSEWORK\PowerMILL\_Projects\Vortex2

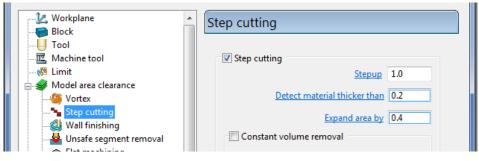
- 3 Activate the workplane, TopCentre-datum.
- 4 From the main toolbar select Block <sup>▶</sup> and using, *Defined by* Box *Type* Model select Calculate followed by accept.
- 5 From the **main** toolbar select **Rapid Move Heights** and in the form select **Calculate** followed by **Accept**.
- 6 Select Toolpath Strategies , and from 3D Area Clearance open a Model Area Clearance strategy.

💏 Mc	odel Area Clearance ? 🗙
Toolpath name	D16TR3-Vortex-RGH1
Workplane	Model area clearance
Block	Style
□ 🌀 Model area clearance	
Vortex	
Step cutting	Vortex 🗸
	Cut direction
	Profile Area
Flat machining	Climb v Climb v
<b>F</b> High speed	
	Tolerance
J Approach	0.1
Automatic verification     Cutter compensation	
Point distribution	Thickness
Tool axis	
Machine axis control	
Rapid move heights	Stepover
	3.0
Start point	
- J End point	Stepdown
🗄 👘 Feeds and speeds	Automatic 🗸 🐇 12.0
	Constant stepdown
Notes and Description	Constant stepdown
	Rest machining
	Calculate Queue OK Cancel

- 7 In the **Model area clearance** page of the strategy dialog:
  - a Enter Toolpath name D16TR3-Vortex-RGH1
  - **b** Select **Style Vortex**
  - c Set Cut direction Both Profile and Area Climb
  - d Input Thickness 0.5
  - e Input Stepover 3.0
  - f Input Stepdown 12.0
- 8 Open the **Vortex** page of the strategy dialog:

	Vortex	
	Cutting feed rate	
	1000.0 mm/min	<b>ii</b>
Model area clearance	Minimum radius (r)	
Step cutting	3.0	6
		•
Unsafe segment removal	Minimum point spacing	
Flat machining ¶₹ High speed	= 0.05	1
Order		
J Approach	✓ Lift on air moves	
Automatic verification	Z dista	ance (z) 0.1

**9** Open the **Step cutting** page of the strategy dialog:



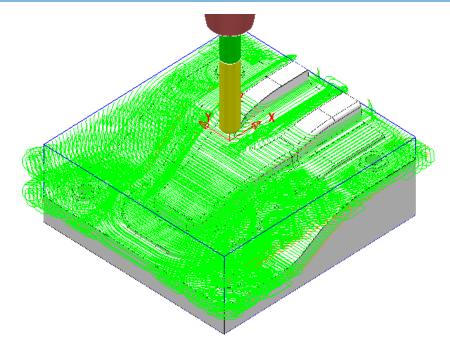
- a Tick the Step cutting box and then input a Stepup value of 1.0
- **10** Open the **Flat machining** page of the strategy dialog:

Workplane	Flat machining
	Machine flats
Model area clearance	□ Multiple cuts
Step cutting	Number of cuts     Stepdown       1     1.0
Unsafe segment removal      Flat machining      If High speed	E Final stepdown

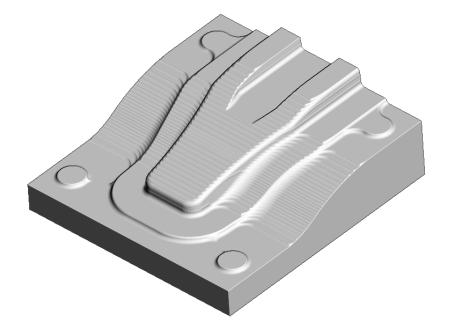
**11** Open the **Approach** page of the strategy dialog:

Workplane	Approach
U Tool 	Helical ramp
	Max zig angle 4.0 Circle diameter (TDU) 0.65
Step cutting	Height 5.0
	Add approaches from outside
Order J Approach	

- 12 In the Helical ramp options window, select *Max zig angle* 4 with a *Circle diameter* 0.65 and *Height* 5.
- 13 At the bottom of the dialogue select **Calculate** to process the **Vortex Model Area Clearance** toolpath.



14 Perform a **ViewMILL simulation** on the toolpath to observe the **Vortex** strategy.

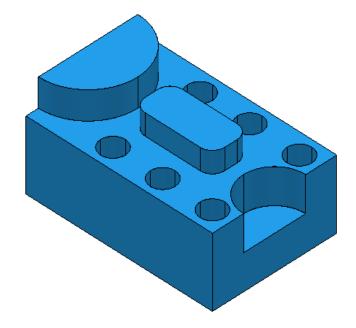


15 Save the Project.

### Machining Flat features 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.

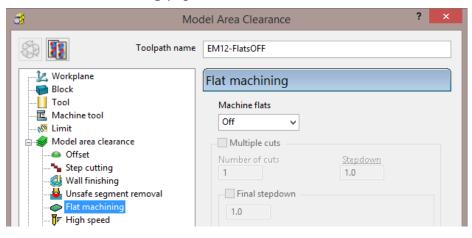
- 1 Select File Delete All and Tools Reset Forms.
- 2 Import the Model:
  - .....\PowerMILL\_Data\Models\Flats.dgk



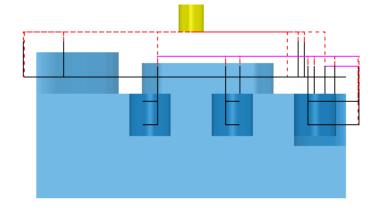
- 3 Select File Save Project As: .....\COURSEWORK\PowerMILL-Projects\AreaClearFlats
- 4 Create a 12mm diameter End Mill tool named EM12.
- 5 **Calculate** the **Block** using the default settings.
- 6 Calculate default Rapid Move Heights.
- 7 Keep the **Start and End Point** settings as default.
- 8 Select **Toolpath Strategies**, and from **3D Area Clearance** open a **Model Area Clearance** strategy.

<del>3</del>	Mo	del Area Clearance	?	×
Toolpath n	name	EM12-FlatsOFF		
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Model area clearance</li> <li>Offset</li> <li>Step cutting</li> <li>Wall finishing</li> <li>Wall finishing</li> <li>Wall finishing</li> <li>Flat machining</li> <li>F High speed</li> <li>Order</li> <li>Approach</li> <li>Automatic verification</li> <li>Cutter compensation</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> </ul>		Model area clearance         Style         Offset all         Offset all         Cut direction         Profile         Area         Climb         Tolerance         0.1         Tolerance         0.1         Thickness         0.5         Stepover         5.0         Stepdown         Automatic         V         On the stepdown	~	
		Rest machining		

- 9 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.

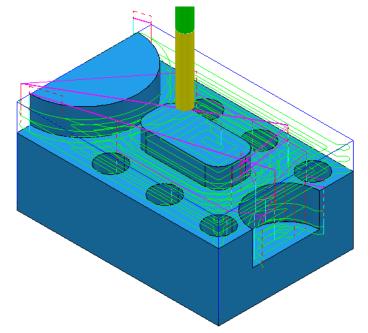


10 Click **Calculate** to process the toolpath and then **Close** the dialog.



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**.

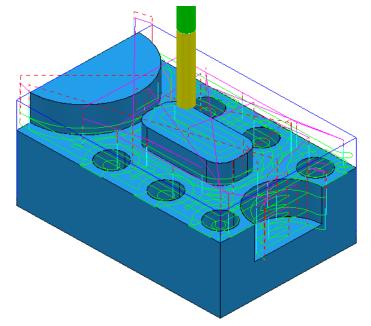
- 1 Right-click the active toolpath, EM12-FlatsOFF and select Settings.
- 2 Click to create a copy of the toolpath.
- 3 Change the new toolpath name to **EM12-FlatsLEVEL**.
- 4 On the **Flat machining** page, change *Machine flats* to Level.
- **5 Calculate** to process the toolpath and then **Close** the dialog.



When the **Machine Flats** option is set to **Level**, the toolpath machines the whole **Block** area for each flat area. The **Stepdowns** are reduced as required to be constant between individual flat areas.

- 6 Right-click the active toolpath and select **Settings**.
  - Click 🗾 to crea
- 7 Click <sup>11</sup> to create a copy of the toolpath.
- 8 Change the toolpath name to EM12-FlatsAREA.

- 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 on each individual flat area leaving a total of 0.6mm stock (The **Thickness** plus the **Tolerance**). The nominal stepdown remains constant with flat area machining occurring separately between these levels.

11 Select File - Save Project to update:-

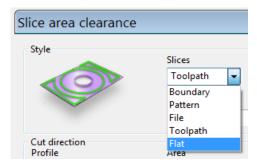
.....\COURSEWORK\PowerMILL-Projects\AreaClearFlats

### 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:

- As a finishing strategy, applied to the flat areas after a Model Area
   Clearance roughing strategy 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**.

There are several types of 2D entity that are supported in the Slices options:-



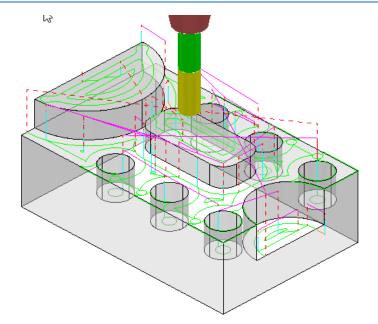
- 12 Right-click on the **EM12-FlatsArea** toolpath and select **Activate** to reinstate all settings and parameters used.
- 13 Select Toolpath Strategies , and from 3D Area Clearance open a Slice Area Clearance strategy.

III Sli	ce Area Clearance	? ×
Toolpath name	EM12-SliceAC	
Workplane Block Block Constraints Block Constraints Block Block Constraints Block Constraints Constra	Slice area clearance Style Cut direction Profile Climb Tolerance 0.02 Thickness 0.0 Stepover 5.0	Slices Flat v Slice editor Offset all v Area Climb v

- 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
- 14 Select the Flat machining page (Available when Slices Flat is active) and:

Hat machining  F High speed  Order	E
Approach     Approach     Automatic verification     Point distribution     Y Tool axis     O Machine axis control	Allow tool outside flat Approach allowance 0.05 (tool diameter units)

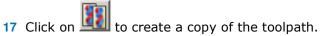
- a Check that the Allow tool outside flat box is unticked.
- 15 Select **Calculate** to process the toolpath and then **Close** the dialog.



This Slice Area Clearance strategy is confined to the Flat areas only.

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

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



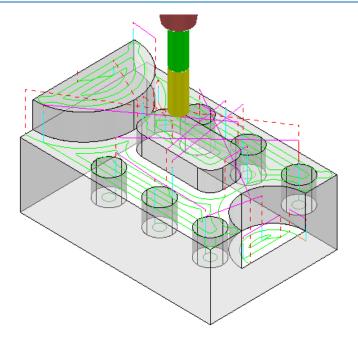
18 Rename the toolpath as EM12-SliceAC-IgnoreHoles and then select the Flat machining page.

Flat machining	1.0
Order     Approach     Automatic verification     Point distribution     Tool axis	Allow tool outside flat Approach allowance
Machine axis control     Machine axis control     Hapid move heights     Leads and links     Start point     Fnd point	Flat tolerance 0.2 ✓ Find flats on triangles
Freeds and speeds      Feeds and speeds      Fistory      For Notes and Description      Subset defined settings	Ignore holes Threshold 2.0 (tool diameter units)

**19** 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 ignored.

20 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, more efficient, toolpath.

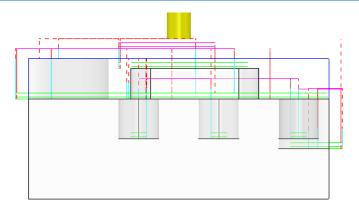


**Ignore Holes** refers to the area defined by the slice definition at the top of the holes. The flat base of any hole will still be machined.

- 21 Right-click the Active toolpath and select Settings to re-open the Slice Area Clearance dialog.
- 22 Click on 10 to create a copy of the toolpath.
- 23 Rename the toolpath as EM12-SliceAC-MultipleCuts and then select the Flat machining page.

Sli	Slice Area Clearance		
Toolpath name	EM12-SliceAC_MultipleCuts		
Workplane  Block  Machine tool  Kachine too	Flat machining         Machine flats         Area         Multiple cuts         Number of cuts         3         2.0         Final stepdown         1.0	odown	

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



If there is a large amount of material remaining above the flat areas, then the **Multiple Cuts** option is applied to reduce the tool loading during **Area Clearance**.



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.



Select File - Save Project to update:-

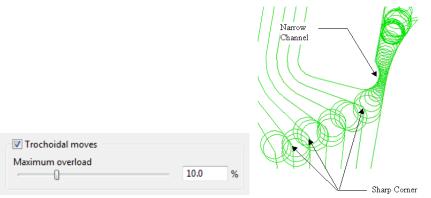
(.....\COURSEWORK\PowerMILL-Projects\AreaClearFlats)

# 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 of the specified stepover.

### **Thickness**

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

### Z Heights – Manual entry

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

Stepdown			
Manual	- 1	12.0	
Automatic		-	
Manual			
	-43-		

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

### 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 enables the strategy to exist outside the **Block** by the **tool radius** value.



This option limits the strategy with the **tool centre** flush with the **Block**.

Limit 🗕 🕤 🚽

### **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.

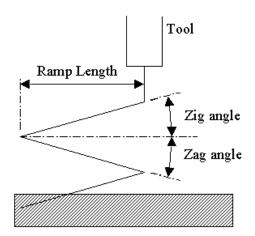
Area - Flat Machining is confined to the individual Flat areas.

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.

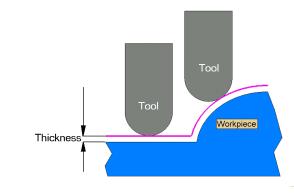


- The Ramp Length is defined as Tool Diameter Units (TDU). For example, A Ramp Length of 2 TDU would equal 20 mm for a Dia 10 mm tool.
- If Add approaches from outside is set, it will take priority over Ramping.
- If the defined geometry for a **Ramp** move is such that it would cause a gouge, then the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

# 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**.

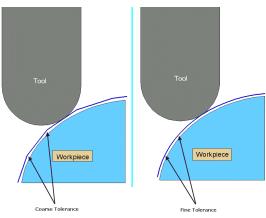
**Thickness** is the amount of extra material specified to remain on the work-piece after machining. This can be applied generally as an offset from the model (as shown), or independently as separate **Radial** and **Axial** values within the machining options.



Ò

It is also possible to assign additional **Thickness** values to selected 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.





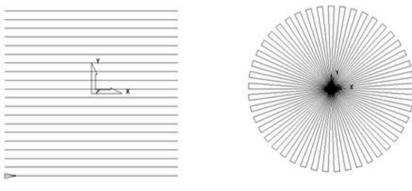
*A positive* **Thickness** value should always be greater than the **Tolerance** value.

## **Raster Finishing and Pattern Finishing**

**Raster Finishing** works by projecting a wireframe, Pattern down the Z-axis onto the model. The standard pattern applied in **Raster Finishing** is obtained by entering values directly into the **Finishing** dialog. Two other types of standard pattern strategy (not covered in this course) include **Radial** and **Spiral**.

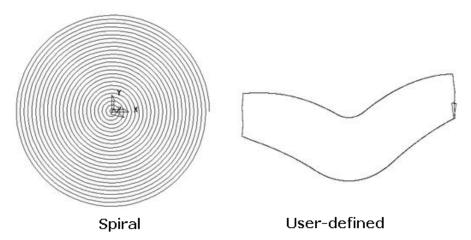
The **Pattern Finishing** strategy uses an existing, pre-defined **Pattern** or **Toolpath**, which is either projected down Z onto the model or used in situ as a **Drive Curve** to generate the toolpath.

Typical previews of all 4 available pattern strategies are shown below as viewed down Z.



Raster



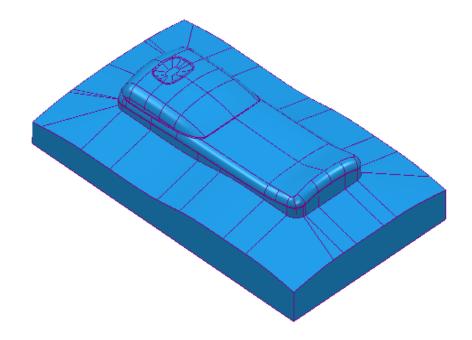


The Raster finishing strategy is used as an example in the next section.

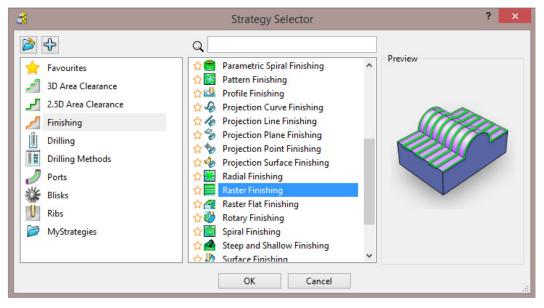
### **Raster Finishing**

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

.....\PowerMILL\_Data\Projects\Handset\_Start



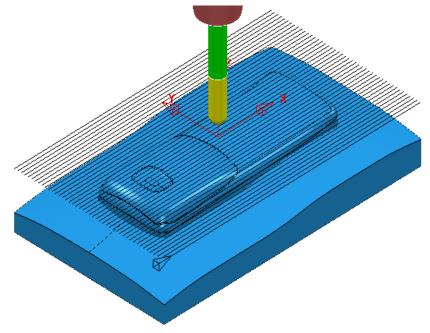
- 3 From the Main menu , select File >Save Project As: .....\COURSEWORK\PowerMILL-Projects\Handset\_PatternMC
- 4 Activate the existing toolpath D25TR3-RGH1 to restore all the set-up settings.
- 5 Activate the tool BN12.
- 6 From the main toolbar, select Toolpath Strategies and from Finishing open a Raster Finishing strategy.



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

<b>3</b>	Raster Finishing ? ×
Toolpath name	BN12-Raster-SEM1
Vorkplane	Raster finishing
	Fixed direction
→ Stock engagement → Raster finishing → Figh speed → Automatic verification	Start corner Lower left 🗸 🗸
Point distribution     Point     Rapid move heights     Point     Leads and links     Point     Feeds and speeds     Pittory     Point     Notes and Description     Viser defined settings	Perpendicular pass  Shallow angle 30.0
	Optimise parallel pass
	Style Two way  Arc radius 0.0
	Tolerance 0.1
	Thickness
	Stepover 2.0
	Preview Draw
	Calculate Queue OK Cancel

- a Name the toolpath **BN12**-**Raster-SEM1**.
- b Enter the Tolerance as 0.1 and Thickness as 0.4
- c Select Ordering Style as Two Way.
- d Enter a **Stepover** of **2**
- e Click the **Preview** tab to display the pattern (as shown below).

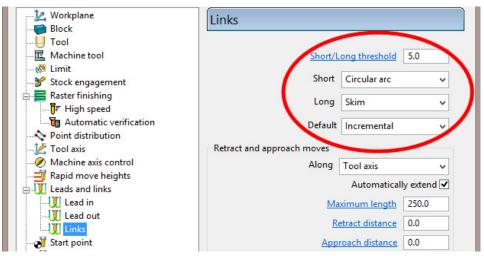


Several options exist to add **lead and link** moves at the start and end of each individual tooltrack. The aim is to provide a smoother transition for the tool to approach tangentially onto the component form instead of simply plunging directly down the Z-axis.

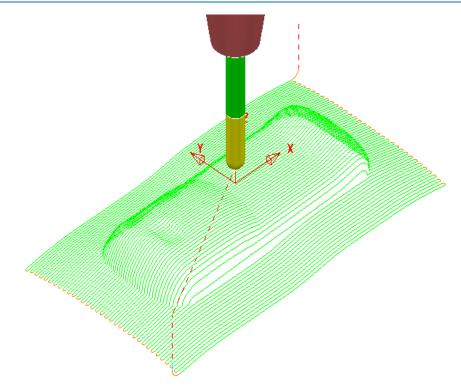
- 8 In the **Raster Finishing** dialog open the **Leads and Links** options (left click on the small box containing +).
- 9 Select the Lead in page and select 1st choice as Vertical arc, input Angle 90, Radius 6.0, and un-tick the option Add leads to short links.

Workplane Block	Lead in
U Tool	
	1st choice Vertical arc 🗸 🗸
→ Stock engagement	Distance 0.0
🚊 🚍 Raster finishing	Distance 0.0
🗍 🖅 High speed	Angle 90.0
Automatic verification	
	Radius 6.0
🔣 Tool axis	
Machine axis control	Ramp options
🖃 🔰 Leads and links	Overlap distance (tool diameter units) 0.0
	Allow start points to be moved 🗸
	Anow state points to be moved
	Add leads to short links
🚽 😽 Start point	
	Add leads at tool axis discontinuities
Feeds and speeds	Angular threshold 90.0
History	
Notes and Description	
💫 User defined settings	Gouge check 🗹
<ul> <li></li></ul>	¢€

- 10 Select Lead out the same as lead in.
- 11 Select the **Links** page and input the value and options exactly as shown in the following image.



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



The above toolpath contains sharp corners along the vertical plane. These can be **arc fitted** within the **Raster Finishing** dialog.

In the **Raster Finishing**, **High speed** page the **Arc fit corners** option inserts a radius in the vertical plane (maximum **0.2** Tool Diameter Units).

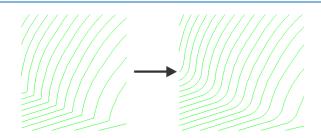
13 In **PowerMILL** *explorer*, right-click the **BN12-Raster-SEM1** toolpath and select **Settings**.



- **14** Click **W** to enable the toolpath **BN12**-**Raster-SEM1** to be edited.
- 15 On the strategy dialog:
  - a Open the **High speed** page.

	High speed
U Tool Machine tool Limit ≩ Stock engagement	☑ Arc fit corners Radius (tool diameter units)
→ ■     Raster finishing       → ■     Raster finishing       → ■     High speed       → ■     Automatic verification	î 0.2

- b Select the Arc Fit option and edit the Arc Radius to the maximum value 0.2
- c Click **Calculate** to process the toolpath and then **Close** the dialog.
- **16** Zoom into the toolpath to view the **arc fitted** tooltracks:



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

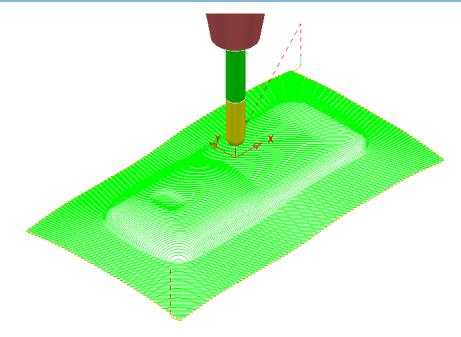
17 In **PowerMILL** *explorer*, right-click the **BN12-Raster-SEM1** toolpath and select **Settings**.



- **18** Click **Description** to create a copy of the toolpath named **BN12-Raster-SEM1\_1**
- **19** Rename the new toolpath as **BN12**-**Raster-FIN1** and input suitable finishing values as shown in the following image.

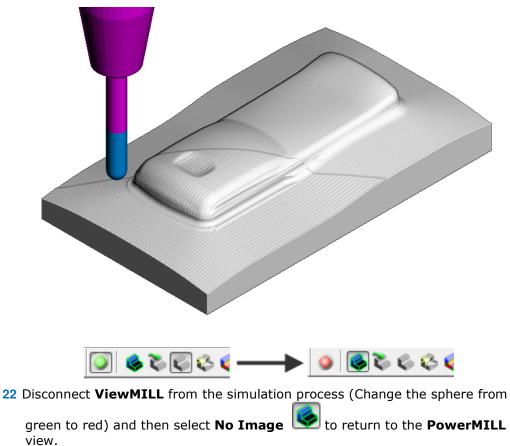
- 💏	Raster Finishing ? ×
Toolpath name	e BN12-Raster-FIN1
Workplane Block	Raster finishing
	Fixed direction Angle 135.0
Raster finishing	Start corner Lower left 🗸 🗸
Automatic verification Point distribution Machine axis control Apid move heights Leads and links Start point Feeds and speeds History Notes and Description User defined settings	Perpendicular pass Perpendicular pass Shallow angle 30.0
	Optimise parallel pass
	Style Two way  Arc radius 0.0
	Tolerance 0.02
	Thickness 0.0
	Stepover (0.5 0.010426)
	Preview Draw
	Calculate Queue OK Cancel

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



If the *raster pattern* is projected downwards parallel to a steep wall then the actual distance between the tooltracks is likely to become unacceptably large.

For this reason the *raster pattern* in **BN12-Raster-FIN1** has been created at a **135 Angle** to minimise where the tooltracks run parallel to steep walls.

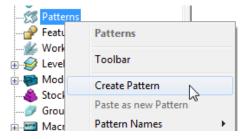


21 Run a full ViewMILL simulation on all 3 existing toolpaths.

### **Pattern Finishing**

This strategy requires a user-defined **Pattern** to be projected 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 CAD data.

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



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



- 24 Right mouse click on the new empty **Pattern** and from the local menu select **Insert File** to access the Open Pattern dialog.
- 25 Click *button* 2 (Shown inside red border below) on the **Open Pattern** dialog:

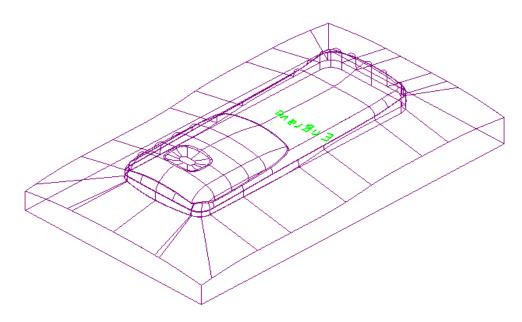
Look in: 🍑 PowerMILL_Data	V G Ø 📁 🛄 V			
Name	Date modified	Туре	Size	
Ive_axis	08/07/2014 14:18	File folder		
InteractiveMacroTraining	08/07/2014 14:18	File folder		
📙 MachineData	08/07/2014 14:18	File folder		
li Models	14/07/2014 13:40	File folder		
퉬 Patterns	08/07/2014 14:18	File folder		
Pictures	08/07/2014 14:18	File folder		
🌗 pmill2_C	17/07/2014 14:13	File folder		
📗 Projects	08/07/2014 14:18	File folder		
📕 📗 SampleMacros	08/07/2014 14:18	File folder		
setup-sheets	08/07/2014 14:18	File folder		
🍌 Triangles	08/07/2014 14:18	File folder		
File name:			~	Оре

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

Look in: 🌗	Patterns	v 🌀 🤌 📴 v			
Name	^	Date modified	Туре	Size	
Engrave.d	lgk	29/03/2001 10:59	DGK File		34 KB
Engrave1.	dgk	01/12/2006 16:32	DGK File		16 KB
StartHole	pic	24/08/2009 11:51	PIC File		1 KB
trochoida	I_pattern.pic	26/11/2001 10:12	PIC File		2 KB
File name:	Engrave1.dgk			v	Oper
Files of type:	Delease Wienferme /* delet	ldx;*.ddz;*.psmodel;doc.det;*.pic)		~	Cance

27 Click Open.

The imported **Pattern** is made up of 2D geometry and is positioned relative to the **Transform** (World co-ordinates).



Next this 2D geometry will be projected along Z onto the 3D model to create a single **Pattern Finishing** toolpath with a negative **Thickness** value.

#### 28 Activate the tool BN1.

29 From the main toolbar, select Toolpath Strategies and from Finishing open a Pattern Finishing strategy.

<b>3</b>	Pattern Finishing ? ×
Toolpath name	BN1-Engrave
Workplane Block Uol Kool K	Pattern finishing         Drive curve         Use toolpath         1       Image: Constraint of the second s
Rapid move heights Leads and links Start point Find point Feeds and speeds History Source and Description User defined settings	Gouge avoidance       Gouge check I         Tolerance       Ordering         0.02       Pattern         Thickness       Image: 10.4         Image: 10.4       Image: 10.4
	I.0       Preview       Draw       Calculate       Queue       OK       Cancel

- a Enter the Toolpath name BN1-Engrave
- b Input **Pattern 1**.
- In 3-Axis applications the *Base position* can be set to either Automatic or Vertical, for the Pattern form to be projected down Z onto the model.

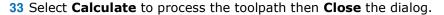
٠.

d Enter a Tolerance 0.02 and Thickness -0.4

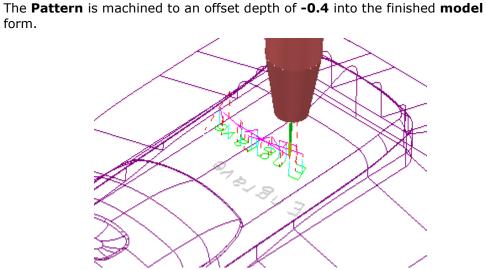
30 Select the Lead in page and select 1st choice as None.

- 31 Select the option Lead out the same as lead in.
- 32 Select the Links page and input the value and options exactly as shown in the following image.

Workplane	Links
Tool	
🖳 🔟 Machine tool	
🔞 Limit	Short/Long threshold 5.0
Stock engagement	Short Skim -
🚊 🔣 Pattern finishing	June 1
	Long Skim 👻
🔚 Automatic verification	
	Default Incremental 👻
	Retract and approach movies



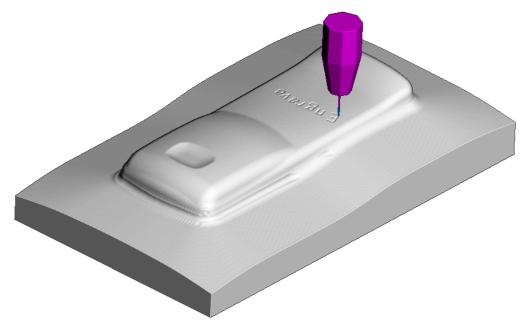
form.





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 10 Patterns, page 20 - Deep Pattern Machining).

34 Simulate the engraving toolpath in ViewMILL.



35 Select File - Save Project to update the contents of: .....\COURSEWORK\PowerMILL-Projects\Handset\_PatternMC

### **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. In this example a **Boundary** will be created and used to limit the two strategies to where they are most effective.

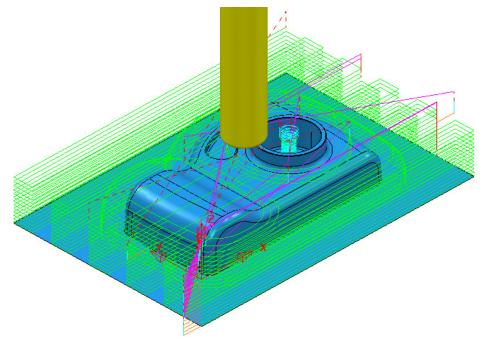
- 1 Select File Delete All and File Reset Forms.
- 2 Select File Open Project and in the dialog browse to:
  - .....\PowerMILL\_Data\Projects\camera\_start

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 a **Model Area Clearance** toolpath along with the associated settings.



# Creation of a Shallow Boundary to define specific machining areas

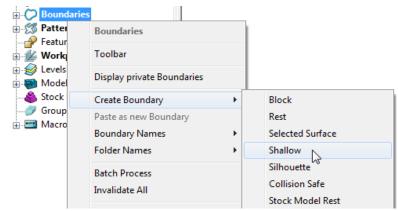
A **Shallow Boundary** creates segments around areas of the **model** that are defined by an **Upper** and **Lower** threshold angle. It is therefore specifically suited to steep walled and shallow surface machining techniques.

-0	$\sim$
-0	
	1

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**, **Thickness**, and **Tolerance** 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**.

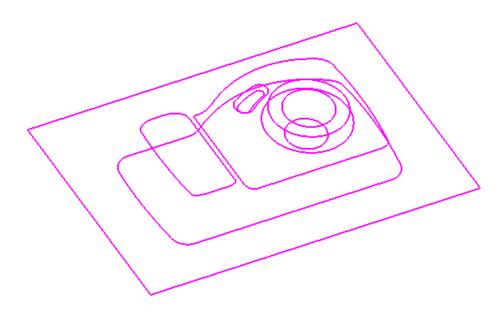


3 In the displayed **Shallow Boundary** dialog:

¥ 1111 1		Ilow Boundary ? ×
Tolerances	Upper Angle 30.0 Lower Angle 0.0 Tolerance 0.02 Thickness 0.0 Axial Thickness 0.0 Use Axial Thickness	Limit Boundary  Inside Limiting Boundary  Outside  Automatic Collision Checking Holder Clearance 0.0 Shank Clearance 0.0 Block
Edit History	ndary to be private	Limit 💆 🗸
[	Apply Queu	ue Accept Cancel

- 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.

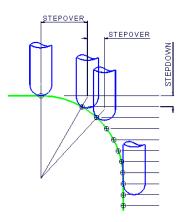
With the **model**, **Tool**, and **toolpaths** undrawn, the **Boundary** can be viewed as shown:



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.

### **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 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open a **Constant Z Finishing** strategy.

Co	nstant Z Finishing		?	x
Toolpath name	BN10-ConstantZ			
Workplane Block Uron Stock Constant Z finishing Unsafe segment removal Unsafe segment removal Definit distribution Uron Machine axis control	Constant Z finishing Order by Region V Spiral Machine down to flats	Additional stock 0.5 Undercut Flat tolerance 0.0		
Rapid move heights Cleads and links Cleads and links Cleads and links Cleads and speeds Cleads and speeds Cleads and pescription Cleads and Description Cleads and Description Cleads and settings	Tolerance 0.02 Thickness 0.0 Stepdown <u>Minimum</u>	Cut direction Climb stepdown 💓 0.5 Calculate using c	usp [	

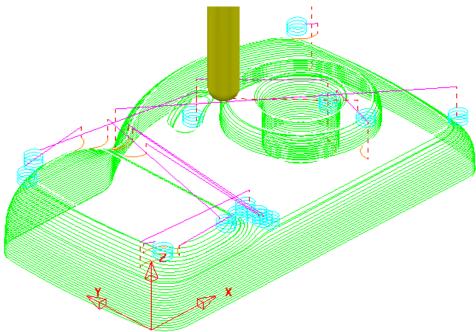
- 2 In the **Constant Z Finishing** dialog:
  - a Enter a Toolpath name BN10-ConstantZ
  - **b** *Tick* the **Spiral** option.
  - c Enter a **Tolerance** of **0.02**
  - d Set the **Cut Direction** to **Climb**.
  - e Enter a Stepdown of 0.5

3 On **Constant Z finishing** dialog, select the **Limit** page and with the **Boundary ShallowBN10** active, set **Trimming** to **Keep outside**.

💰 Co	onstant Z Finishing ? ×
Toolpath name	BN10-ConstantZ
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Constant Z finishing</li> <li>Unsafe segment removal</li> <li>Automatic verification</li> <li>Point distribution</li> </ul>	Limit Boundary ShallowBN10 V V V Trimming Keep outside V
<u>I</u> Tool axis Ø Machine axis control <del></del> I Rapid move heights ⊕Ū Leads and links	Block

4 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**.



### Introduction to Leads and Links

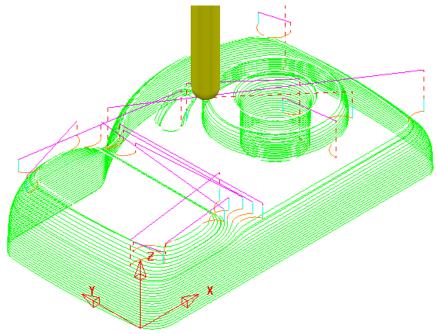
The current toolpath (BN10-ConstantZ) has inherited circular Ramp lead moves (pale blue) from the previously activated toolpath RoughOp1. From the main toolbar we will open the Leads and Links dialogue and apply Horizontal Arcs to both the start and end of the Long link moves and On Surface to the Short link moves (< 4).

- 1 Click the Leads and Links button on the **Main** toolbar.
- 2 Select the Lead In tab and:
  - a In Lead in, change the 1st Choice to Horizontal Arc, Angle 90 and Radius 5
  - Tick the Add leads to short links box. b
  - c 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 active 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.





### **3D Offset Finishing**

**3D Offset** Finishing creates the toolpath **stepover** relative to the **3D Model** providing consistency over both flat and steep areas. To produce an improved tool movement over the model a **Shallow Boundary** will be used to limit this strategy to where it is most effective.

1 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open a **3D Offset Finishing** strategy.

30	O Offset Finishing ? 🗙
Toolpath name	BN10-3Doffset
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Freeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	3D offset finishing         Pattern         Image: Start on pattern         Image: Start on pattern         Image: Spiral         Image: Smoothing         Image: Maximum offsets         Image: Maximum number of offsets         Image: Maximum number of offsets         Image: Tolerance         Image: Climb         Image: Output         Image: Output
	Calculate Queue OK Cancel

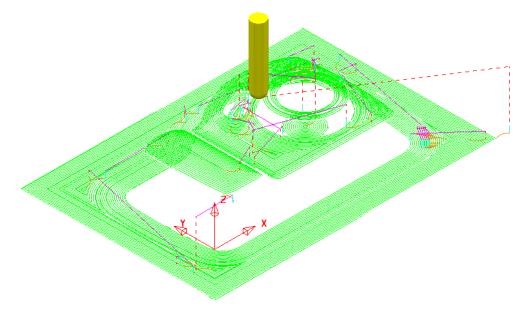
- 2 In the **3D Offset** Finishing strategy dialog:
  - a Enter Toolpath name as BN10-3Doffset
  - **b** *Tick* the **Spiral** option
  - c Set the Cut Direction to Climb
  - d Enter a *Tolerance* of 0.02
  - e Enter a **Thickness** of **0**
  - f Enter a *Stepover* of **0.5**

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

Workplane 	Limit
	Boundary       Boundary       ShallowBN10       Trimming
Automatic verification  Point distribution  Control  Machine axis control	Keep inside 🗸
	Block Limit 💆 🗸

- 4 Select the **Boundary ShallowBN10**.
- 5 Select Calculate to create the toolpath and then Close the dialog.

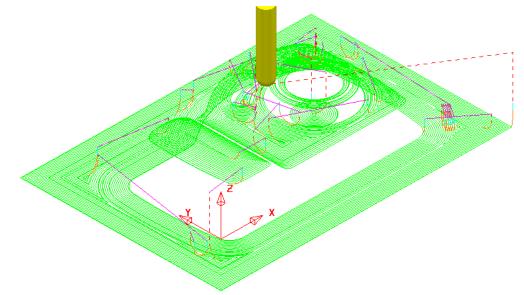
The **Toolpath** follows the contours of the **Boundary** segments and is produced only within the *shallow* areas of the **model** (*Inside* the **Boundary**).



This **toolpath** has inherited the current settings in the **Leads and Links** dialog, as applied to the previous **toolpath BN10-ConstantZ**. A more suitable option for bot **Lead in** and **Lead out** would be **Vertical arc**.

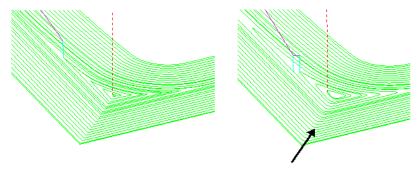
- 6 Click the **Leads and Links** witton from the top of the screen.
- 7 Select the Lead in tab and:
  - a Un-tick Add leads to short links, change the 1st Choice to Vertical Arc, Angle 90 and Radius 5.
  - **b** Click the **Lead Out the same as lead in** button.
- 8 Apply and Accept the dialog.

The **toolpath** now is more efficient with the rapid moves at *skim height*, the *long links* using **Vertical Arc** Leads, and the *short links* being linked along the surface.



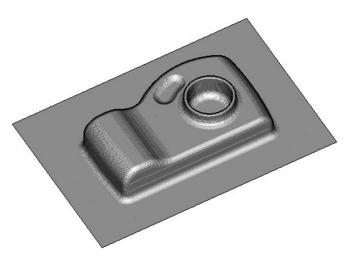


To further improve this **3D Offset** strategy, you can tick the **Smoothing** option. This results in sharp corners being replaced with a smooth blend wherever possible.



Smoothing Applied

9 Simulate all of the toolpaths in ViewMILL.



10 Select File - Save Project to update (but do not close):-To update:- ......\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.

- 11 De-activate the Boundary, ShallowBN10 in the PowerMILL explorer.
- 12 From the main toolbar, select Toolpath Strategies and from Finishing open an Optimised constant Z Finishing strategy.

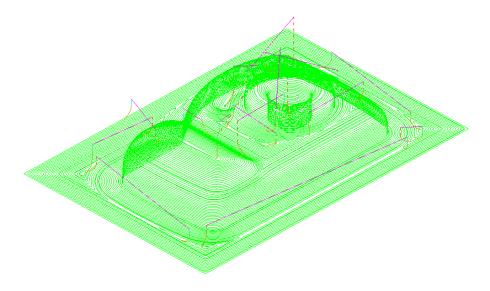
Optimised Constant Z Finishing ?			×
Toolpath name	BN10-OptCZ		
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Optimised constant Z finishing</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>U Leads and links</li> <li>Start Point</li> <li>End point</li> <li>Feeds and speeds</li> <li>Notes</li> <li>User defined settings</li> </ul>	Optimised constant Z finishing         ✓ Spiral         ✓ Closed offsets         ✓ Smoothing         Tolerance       Cut direction         0.02       Climb         Thickness         ✓ 0.0         Stepover         ✓ 1.0       ▲ 0.050253         Use separate shallow stepover         Shallow stepover         Shallow stepover		

- a Enter Toolpath name as BN10-OptCZ
- **b** *Tick* 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.

🥳 Optimis	ed Constant Z Finishing	?	×
Toolpath name	BN10-OptCZ		
Workplane     Block     Tool     Tool     Machine tool     Stock engagement     Stock engagement     Goptimised constant Z finishing     Automatic verification     Point distribution     Y Tool axis	Limit Boundary Image: Second S	_	

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

Note the consistent 3D **Stepover** across the whole component.



14 From the Main menu, click the Leads and Links button to display the Leads and Links dialog and then, select the Links tab.

	Leads and Lin	ks	?
Z heights First lead in	Lead in	Lead out	Last lead ou
Extensions	Links	Point	distribution
Short/Long thre	shold 2.0		
Retract and approach moves		Short	
Along Tool axis	~	On surface	¥
Automatically extend		Long	
<u>Maximum len</u>	<u>igth</u> 250.0	Skim	~
		Default	
Retract dist	ance 0.0	Skim	~
Approach dis	tance 0.0		
Arc fit rapid moves			
	Arc radius (to	ol diameter unit:	s) 0.25
✔ Gouge check			
Use polar links			
			Apply links
Apply	Accept	Cancel	

- 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.
- 15 Apply and Accept the dialog.

16 Select File - Save Project to update (but do not close):.....\COURSEWORK\PowerMILL-Projects\Camera\_Example

### **Steep and Shallow Finishing**

This strategy includes the following additional options:-

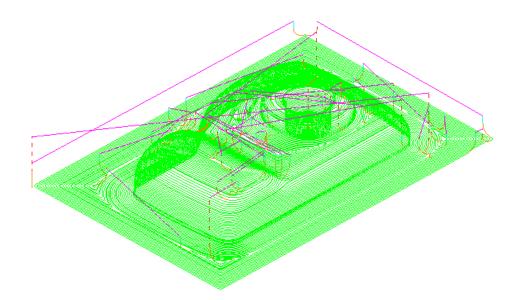
The option to input a **Threshold Angle** to control the extent of the **Constant Z** and **3D Offset** areas of the toolpath.

The option to input a **steep shallow overlap** value between the **Constant Z** and **3D Offset** areas.

- 1 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open a **Steep and Shallow Finishing** strategy.
- 2 Enter the values exactly as shown:

🛃 Steep	and Shallow Finishing	? ×
Toolpath name	BN10-SteepAndShallow	
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Machine shallow finishing</li> <li>Point distribution</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Steep and shallow fini Order Steep first V Threshold angle 30.0 Steep Spiral Cut direction Climb V Stepdown W 1.0 Closo253 Calculate using cusp	ishing Additional stock 0.5 Steep shallow overlap 2.0 Shallow ✓ Spiral Cut direction Climb v Stepover ✓ 1.0 ↓ 0.050253 Type 3D offset v Smoothing
	Tolerance 0.02 Thickness 	
	Calculate Queue O	K Cancel

- a Enter Toolpath name as BN10-SteepAndShallow.
- **b** For the **Order** select **Steep first**.
- c For the **Threshold angle** input **30**.
- d Tick both of the Spiral boxes.
- e For both the **Stepdown** and **Stepover** input **1**.
- f For **Type** (shallow) select **3D offset**.
- g Tick the Smoothing box.
- **3** Select **Calculate** to process the toolpath and then **Close** the dialog.



4 Select File - Save Project to update:-.....\COURSEWORK\PowerMILL-Projects\Camera\_Example

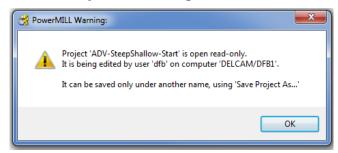
# Steep and Shallow finishing with separate Shallow Stepover

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.

The main advantage with the **Steep and Shallow** strategy is that the **Threshold Angle** is user defined. This can be set to **0** making the large stepover apply over flat areas only.

- 1 Delete All and Reset Forms.
- 2 Select File Open Project and in the dialog browse to:

.....\PowerMILL\_Data\Projects\ADV-SteepShallow-Start

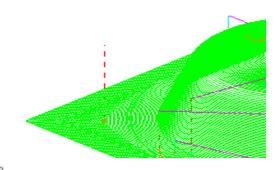


- 3 Click **OK** to load the read-only **Project** into **PowerMILL**.
- 4 Save Project As:

.....\COURSEWORK\PowerMILL\_Projects\ADV-SteepShallow-EX1

5 In the **PowerMILL** *explorer*, right-click on the existing toolpath **D16t3-SteepShallow-FIN** and select **Settings**.

The component form includes 2 flat areas which are currently finished 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.



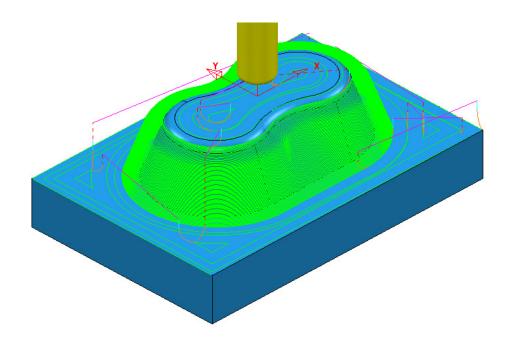
- 6 Click **Recycle** to enable edits to the existing toolpath.
- 7 Update the main page of the dialog exactly as shown below.

Steep	and Shallow Finishing	? ×
Toolpath name	D16t3-SteepShallow-FIN	
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Steep and shallow fini Order Top first v Threshold angle 0.0 Steep Spiral Cut direction Climb v Stepdown W 0.01 Calculate using cusp Maximum stepdown 0.3 Tolerance 0.02	Additional stock 0.8 Steep shallow overlap 0.0 Shallow Shallow Spiral Cut direction Climb Stepover 6.0 3.0 Type 3D offset Smoothing
	Thickness 	K Cancel

a Select Order as Steep first.

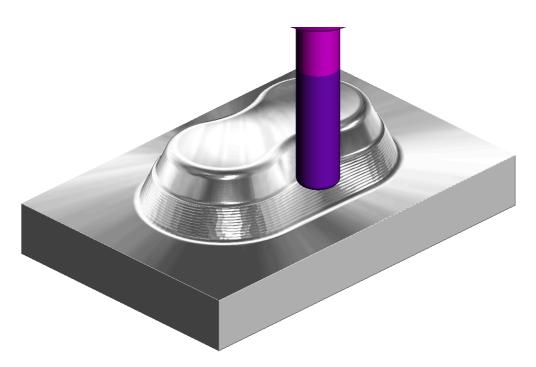
- **b** Input the *Threshold angle* as **0**
- c Tick the Calculate using cusp box and input
- d Input a Stepdown of 0.01 and a Maximum stepdown of 0.3
- e For the shallow regions, enter a Stepover of 6

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



The *flat areas* are machined more efficiently using a large **stepover** while the *steep areas* use a small, variable **stepdown** based on the **cusp height**. This provides an efficiently produced, consistent finish over the whole form.

9 Run a ViewMILL simulation on both the *Roughing* and *Finishing* toolpaths.



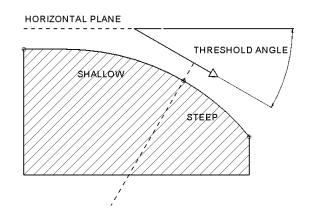
10 Select File - Save Project, to update:-.....\COURSEWORK\PowerMILL\_Projects\ADV-SteepShallow-EX1

## **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. This is the angle down from the horizontal, where the *steep* and *shallow* portions meet.

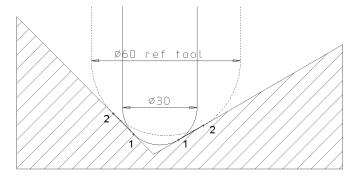
The **Threshold Angle** can be defined to prevent the tool running up or down steep slopes. For example, you could apply a **Stitch** strategy to track across the **steep** areas, while using the **Along** strategy in **shallow** areas.





If the Threshold Angle is 90 then the Steep area will not exist.

The **Corner Finishing** and **Multi-Pencil** strategies machine the areas that are inaccessible to the Reference tool (2) but are accessible to the active tool, as in the diagram.



In the following example, the **Corner Finishing** strategy uses a Dia 10 Ball Nose for the reference tool, and a diameter Dia 5 Ball-nose to perform the rest machining.

- 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 Create a Dia 5 ball nose tool named **BN5**.
- 7 From the main toolbar, select **Toolpath Strategies** and from **Finishing** open a **Corner Finishing** strategy.

<b>3</b>	Corner Finishing	? ×
Toolpath name	BN5-CornerAlong_Shallow	
Workplane Block Block Machine tool Stock engagement Stock engagement Corner detection Automatic verification Point distribution Tool axis Machine axis control	Corner finishing Output Shallow Threshold angle 30.0 Maximum passes Maximum num	Strategy Along V Cusp 0.03 ber of passes 10
Rapid move heights 	Tolerance 0.02 Thickness 	Cut direction

- a Enter Toolpath name as BN5-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 *Active*.

<b>₩</b>	Corner Finishing	? ×
	Toolpath name BN5-CornerAlong_Shallow	
Workplane	Ball nosed	
Block	1 V 🕅 BN5	<b>v</b>
Machine tool		· · ·
	Name BN5	

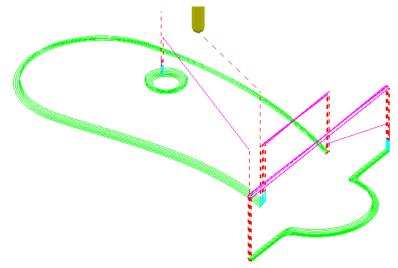
f Select the **Corner** detection page and select the **Reference tool** as **BN10**.

đ	Corner Finishing	? ×
Toolp	ath name BN5-CornerAlong_Shallow	
<ul> <li></li></ul>	Corner detection Reference tool BN10 BN10	~
Corner finishing	Use toolpath reference Toolpath	~

8 Select the **Lead in** page and set the **1**<sup>st</sup> **Choice** to **None**.

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

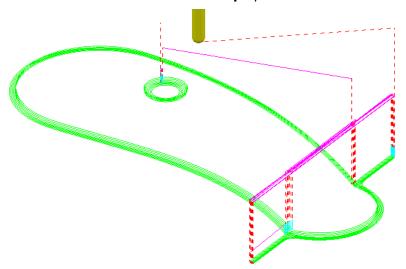
**10 Simulate** the toolpath and observe the order of the machining:



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).



If a **Threshold angle** of **90** is used, the **Shallow** option will apply to all valid rest areas. In this case the **Steep** option is non-existent.

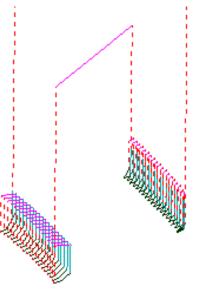


### **Corner Stitch Finishing**

1 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open a **Corner Finishing** strategy.

<u> <del>3</del></u>	Corner Finishing	? ×
Toolpath name	BN5-CornerStitch_Steep	
	Corner finishing	
	Output	Strategy
Machine tool	Steep 🗸	Stitch 🗸
Stock engagement     Corner finishing     Corner detection     Automatic verification     Point distribution     Corner detection     Tool axis     Machine axis control	Threshold angle         30.0         Maximum passes         Maximum nur	Cusp 0.03 mber of passes 10
Trapid move heights	Tolerance	Cut direction
E-U Leads and links	0.02	Climb 🗸
	Thickness	

- a Enter Toolpath name as BN5-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.
- 2 **Calculate** to process the toolpath and then **Close** the dialog.



This time, only the steep areas are machined.

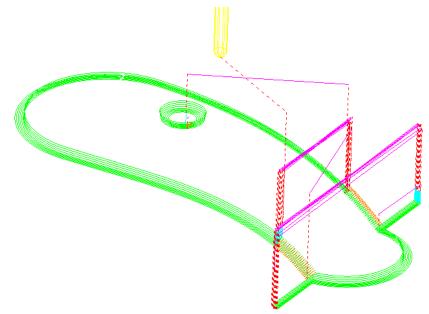
3 From the Main menu select File > Save Project to update:-.....\COURSEWORK\Projects\CornerFinish

### **Corner Automatic Finishing**

1 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open a **Corner Finishing** strategy.

<u> </u>	Corner Finishing		?	×
Toolpath name	BN5-CornerBoth_Automatic			
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Machine tool</li> <li>Stock engagement</li> <li>Corner finishing</li> <li>Corner detection</li> <li>Automatic verification</li> <li>Point distribution</li> </ul>	Corner finishing Output Both Threshold angle 30.0 Maximum passes	Strategy Automatic Cusp 0.03	~	
Tool axis Machine axis control Machine axis control Rapid move heights U Leads and links Start Point Ind point Feeds and speeds Motes	Maximum nu Tolerance 0.02 Thickness 	mber of passes 10 Cut direction Climb	~	

- a Enter Toolpath name as BN5-CornerBoth\_Automatic.
- **b** Use Strategy Automatic.
- c Change **Output** to **Both**.
- d 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 Finishing** creates a single toolpath along the actual intersection between non-tangential faces on the model.

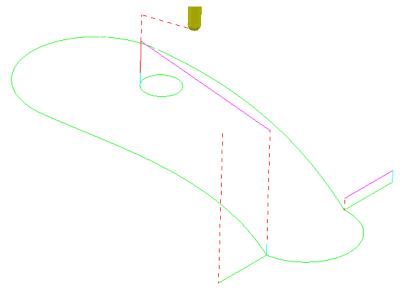
1 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open a **Corner Pencil Finishing** strategy.

Cor	mer Pencil Finishing	?	×
Toolpath name	BN5-CornerPencil		
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Stock engagement</li> <li>Corner pencil finishing</li> <li>Automatic verification</li> <li>Point distribution</li> </ul>	Corner pencil finishing         Output       Threshold angle         Shallow       90.0         Detection limit       165.0		
- Tool axis Machine axis control Rapid move heights Leads and links W Start Point M Fnd point	Tolerance   Cut direction     0.02   Climb		_

- a Enter Toolpath name as BN5-CornerPencil.
- **b** Change **Output** to **Shallow**.
- c 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.* 

- d On the **Tool** page, make the **BN5** tool active.
- 2 Select **Calculate** to process the toolpath, and then **Close** the dialog.



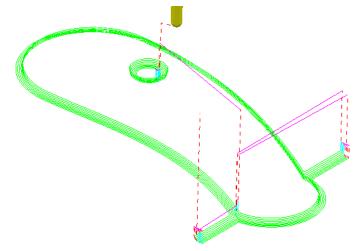
### **Corner Multi-pencil Finishing**

**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** toolpath, but on closer inspection there are subtle differences to the toolpath order and that it includes tool tracks along the model intersections.

1 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open a **Multi-Pencil Finishing** strategy.

Corner	r Multi-Pencil Finishing	?	×
Toolpath name	BN5-CornerMultiPencil		
Workplane     Block     Tool     Tool     Machine tool     Stock engagement     Corner multi-pencil finishing     Corner detection     Automatic verification     Point distribution     Yool axis     Machine axis control     Rapid move heights     Leads and links     Start point	Corner multi-pencil finishing         Output         Shallow         Threshold angle         90.0         0.03         Separate regions         Maximum passes         Maximum number of passes         Tolerance         Cut direction		
	0.02 Climb	<u> </u>	' r

- e Enter Toolpath name as BN5-CornerMultiPencil.
- f Change **Output** to **Both**.
- g Change the Threshold Angle to 90
- h On the **Tool** page, make the **BN5** tool active.
- i In the **Corner detection** page check that **BN10** is selected.
- 2 **Calculate** to process the toolpath and then **Close** the dialog.



3 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:

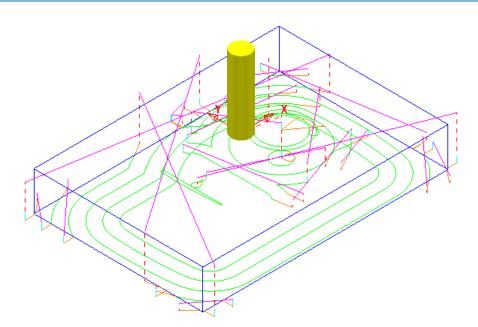
.....\COURSEWORK\PowerMILL-Projects\FlatFinish-Example1

### **Offset Flat Finishing**

- 1 In the **PowerMILL** explorer, Activate the Toolpath D16TR3-RGH1.
- 2 Activate the Dia 16 End Mill (EM16) Tool.
- 3 De-Activate Boundary 1.
- 4 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open an **Offset Flat Finishing** strategy.

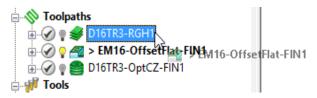
<b>3</b> 0	ffset Flat Finishing	? ×
Toolpath name	EM16-OffsetFlat-FIN1	
	Offset flat finishing	
<mark>  </mark> Tool <b>E</b> Machine tool		Flat tolerance 0.0
🔀 Limit		Find flats on triangles ✔ Allow tool outside flats ✔
<mark>]</mark> ₹ High speed		Rest machining
Point distribution	Add approaches from	
	Approach	outside allowance 0.05
Rapid move heights	Ignore holes	
Start point		Threshold (TDU) 2.0
Feeds and speeds	Tolerance 0.01	Cut direction
	Thickness	Climb
User defined settings		<u></u>
	Stepover	
	10.0	★ 7.071068
	Final stepdown	
		1.0
↓ ↓	Calculate Queue	OK Cancel

5 Select **Calculate** and when completed, **Close** the dialog.

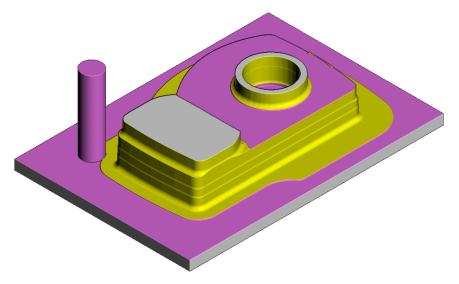


The imported **Project** contains both an **Offset Area Clear** roughing strategy and an **Optimised Constant Z** finishing strategy. It is recommended that **Offset Flats** (**EM16-OffsetFlats-FIN1**) is run directly after the **3D Area Clearance** (**D16t3-RGH1**). For this reason, it would make good sense to move it up the toolpath order in the **PowerMILL** *explorer*.

1 In the **PowerMILL** *explorer*, hold the left mouse key down on the toolpath **EM16-OffsetFlats-FIN1** while dragging and releasing it onto **D16TR3-RGH1**.



2 Perform a **ViewMILL** simulation on all 3 toolpaths to check that the **Toolpath** order is acceptable.



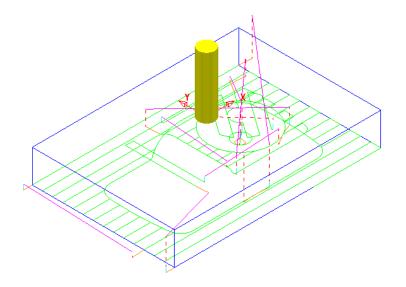
The ViewMILL simulation after the Offset Flat Finishing strategy is shown above.

### **Raster Flat Finishing**

1 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open a **Raster Flat Finishing** strategy.

<ul> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Stock engagement</li> <li>Raster flat finishing</li> <li>F High speed</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>U Leads and links</li> <li>Start point</li> <li>Freeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Raster flat finishing Flat tolerance Find flats on Allow tool out Rest m △ Add approaches from outside Approach outside allowance ○ Fixed direction Angle	triangles tside flats bachining 0.05
Automatic verification  Point distribution  Machine axis control  Rapid move heights  Class and links  Start point  Faceds and speeds  Faceds and speeds  History  Notes and Description	Approach outside allowance	
Start point     Total point     Freeds and speeds     History     Notes and Description		0.0
Ser defined settings	Threshold (TDU) Tolerance Cut directio 0.01 Any	
	Thickness 0.0 Stepover 10.0	7.071068
R	Final stepdown	1.0

2 Select **Calculate** and when completed, **Close** the dialog.

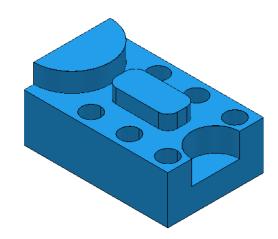


3 From the Main menu select File > Save Project to update:-.....\COURSEWORK\PowerMILL-Projects\FlatFinish-Example1

### Flat Machining example

- 1 Select File Delete All and Tools Reset Forms.
- 2 Select File Import Model:-

.....\PowerMILL\_Data\Models\Flats.dgk



3 Save the Project as:-

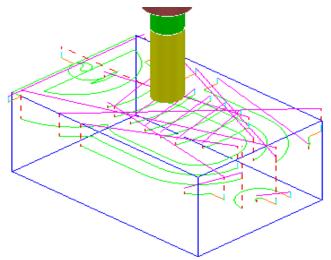
.....\COURSEWORK\PowerMILL\_Projects\FlatFinishing-Example2

- 4 Create a 20mm diameter End Mill tool named EM20.
- 5 Calculate the (default) Block and then reset the Rapid Move Heights.
- 6 Open an Offset Flat Finishing strategy and:-

<b>3</b> 0	ffset Flat Finishing 🛛 🗧 🗙
Toolpath name	EM20-FlatFin
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Offset flat finishing</li> <li>High speed</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Offset flat finishing   Flat tolerance 0.2 Find flats on triangles I Allow tool outside flats I Rest machining Ignore holes Threshold (TDU) 2.0 Tolerance Cut direction 0.02 Climb Thickness 0.15 0.0 Stepover 12.0 Add 8,485281

- a Name the toolpath EM20-FlatFin.
- **b Untick** the **Rest machining** box.
- c Click the use **Axial Thickness** button and enter separate **Thickness** values for **Radial 0.5** and **Axial 0**
- **d Calculate** to process the toolpath and then **Close** the dialog.

Most of the **Flat** areas are finish machined, apart from the gap between the Upstands and inside the holes due to the **20mm** diameter tool being too large. We will now use a smaller tool to **Rest** machine these areas.



- 7 Create a Dia 10 End Mill, named EM10.
- 8 Activate the Toolpath, EM20-FlatFin and then select Settings.

88

- 1 Click to create a copy of the strategy.
- 2 In the main **Offset flat finishing** page of the strategy dialog:

Of Of	fset Flat Finishing ? ×
Toolpath name	EM10-FlatFinRest
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Offset flat finishing</li> <li>Rest</li> <li>F High speed</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Offset flat finishing   Flat tolerance   0.2   Find flats on triangles   Allow tool outside flats   Allow tool outside flats   Rest machining   Add approaches from outside   Approach outside allowance   0.05   Ignore holes   Threshold (TDU)   2.0   Tolerance   Out direction   0.02   Cut direction   0.03   Stepover   3.0   2.12132

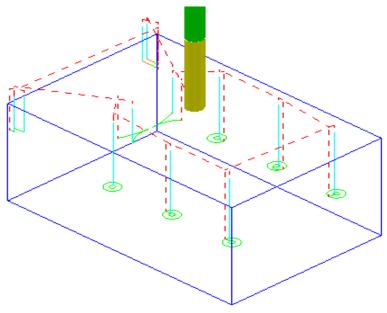
- a Enter Toolpath name as EM10-FlatFinRest.
- b Enter Stepover as 3.0
- c Activate the EM10 tool.
- **d** *Tick* the **Rest machining** box and then select the **Rest** page.

e On the **Rest** page, select the reference **Toolpath** as **EM20-FlatFin**.

<u>3</u> 0	ffset Flat Finishing	?	×
Toolpath name	EM10-FlatFinRest		
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Offset flat finishing</li> <li>Rest</li> <li>F High speed</li> <li>Automatic verification</li> <li>Point distribution</li> </ul>	Rest Rest machining Toolpath V EM20-FlatFin Detect material thicker than 0.5 Expand area by 0.0	~	

**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**.



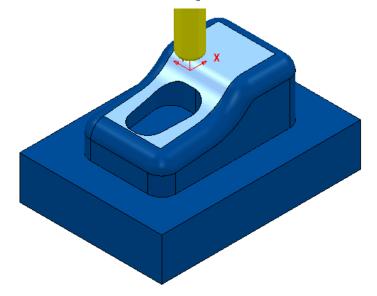
The tool tracks in a Surface Finishing strategy will not run parallel to the edges of a <u>trimmed</u>, surface.

- 1 From the Main menu, select File Delete All and Tools Reset Forms.
- 2 Open the read-only Project:-

.....\PowerMILL\_Data\Projects\EditToolpaths\_3

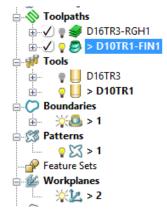
3 From the Main menu , select File - Save Project As:

.....\COURSEWORK\PowerMILL-Projects\PunchForm\_3



4 Activate the toolpath D10TR1-FIN1 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 **D10TR1**.



- 5 De-Activate both the *Toolpath* D10TR1-FIN1, and *Boundary* 1.
- 6 Select the *upper surface* on the punch model (as displayed above).

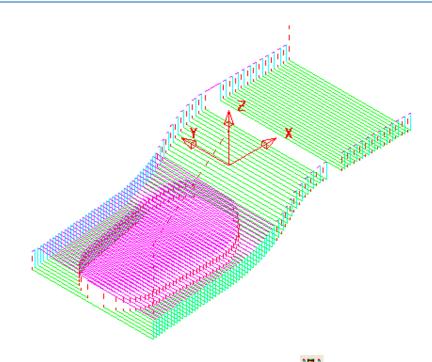
7 Open a **Surface Finishing** strategy and enter values as shown below.

🛪 S	urface Finishing	? ×
Toolpath name	D10TR1-TopSurface1	
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Kachine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Surface finishing</li> <li>Pattern</li> <li>Pattern</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Freeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Surface finishing Surface Surface side Outside Surface units Distance Degouge tolerance 0.3 Tolerance 0.01 Thickness 0.0 Stepover (Distance) 0.3 Output Draw	
	Calculate Queue OK Cancel	

8 Select the **Pattern** page:

<del>d</del> i	Surface Finishing		?
Toolpath nan	D10TR1-TopSurface1		
Workplane Block	Pattern		
U Tool 	Pattern Patter	n direction U	~
Surface finishing		Sp	iral 🗌
Pattern Automatic verification	Ordering	Two way	~
Point distribution	Start corner	Min U max V	~
Tool axis     Machine axis control     Appid move heights	Sequence	None	~

- a Set Pattern direction U
- **b** Un-tick **Spiral**, and then select **Ordering Two way**
- c Select Start corner Min U max V
- 9 Select the **Preview** tab to display the **Pattern** along its *start point* (arrow)
- 10 Select **Calculate** to create the toolpath and then **Close** the dialog.



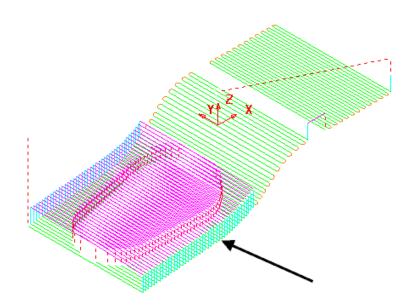
- 11 From the **Main** toolbar, click the **Leads and Links** 関 button.
- 12 On the Links page:

🖞 Leads and Links 🛛 ? 🗙						
Z heights	First lead in	Lead out	Last lead out			
Extensio	ins	Links	Point	distribution		
	Short/Long threshold 3.0					
Retract and a	pproach moves		Short			
	Along Tool axis	~	Circular arc	~		
- Automat	Automatically extend					
	<u>Maximum leng</u>	Skim	~			
	Default					
	Retract distance 0.0			~		
	Approach dist	ance 0.0				

- a Enter Short/Long Threshold 3.0
- **b** Select **Short Circular Arc**
- c Select Long Skim
- d Select Default Incremental

**13 Apply** the dialog to update the *active* toolpath.

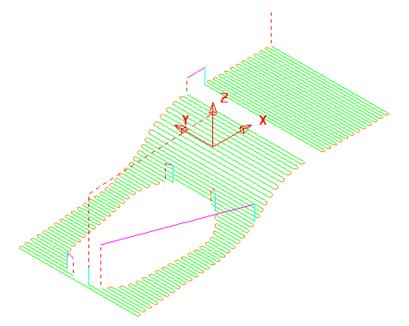
Where the **Link** move from the end of a tool track to the start of the next is within 3 mm, an (orange) arc move will be applied.



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.

- **14** Right-click on the **toolpath** and from the local menu, select **Edit > Reorder**.
- **15** In the **Toolpath Listing** dialog, click the **Automatic reorder and reverse** button.

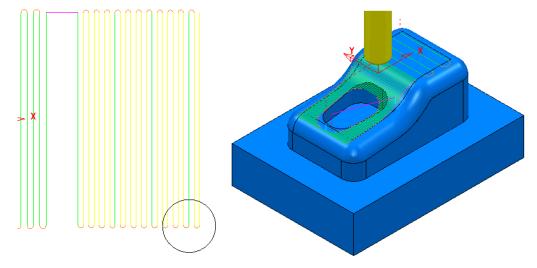


The toolpath follows the curves of the selected surface.

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 active **Tool**.

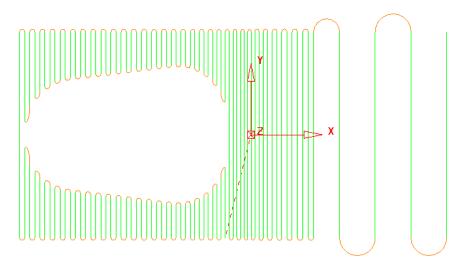
- 1 From the **Main** menu, select **Draw > Cursor > Tool** to help with the visualisation for the next action.
- 2 Using the **Shift key** and **left mouse key** select groups tool tracks leaving a single pass within the tool diameter range, outside each set as shown below:



The **Tool** outline is displayed above to visualise suitable **Stepover** (**Draw** > **Cursor** > **Tool**).

If it is necessary to de-select tool tracks form the current selection use the **Ctrl Key** and **left mouse key**.

- 3 Right-click the toolpath and select Edit > Delete Selected Components to remove the selected tool tracks.
- 4 In the **Leads and Links** dialog select the **Links** options and change the **Short/Long threshold** to **15** before selecting **Apply**.



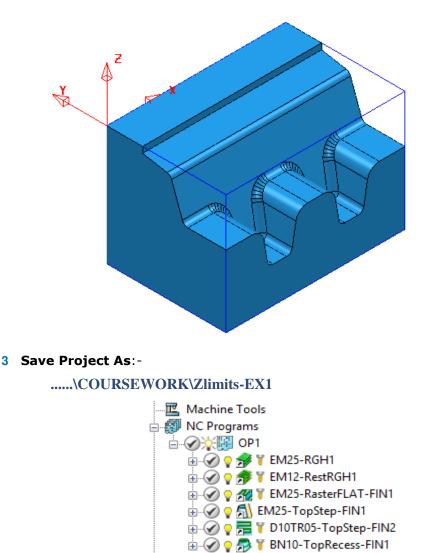
5 From the Main menu select File > Save Project to update:-.....\COURSEWORK\PowerMILL-Projects\PunchForm\_3

### **Toolpath Z-Limits**

All **Toolpath** types, except drilling, can now be limited to applied maximum and minimum **Z** values within the defined **Block**.

- 1 From the main menus, select File Delete All and Tools Reset Forms.
- 2 Open the read-only Project:-

.....\PowerMILL\_Data\Projects\Zlimits-Start



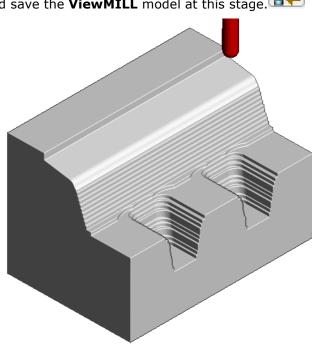
🖶 📣 Toolpaths

🗄 🝿 Tools

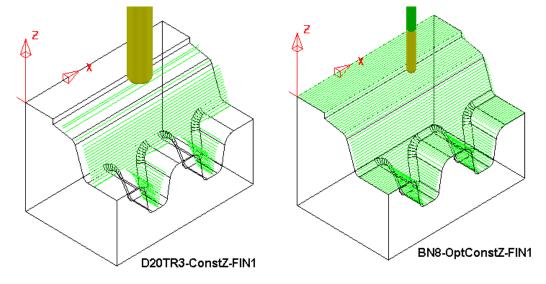
become evident during a *ViewMILL* simulation.

The **Project** already contains 8 **Toolpaths**, 6 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

- 4 Activate the toolpath D20TR3-ConstantZ-FIN1 to reinstate the original Block and settings.
- 5 Run a ViewMILL simulation on the existing NC Program OP1.
- 6 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.

**BN8-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.

7 Return to the **PowerMILL** (*No Image*) view

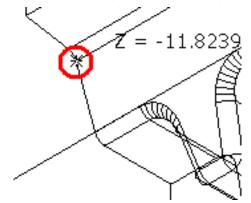
and disconnect ViewMILL

from the simulation (Switch the Green Sphere to Red).

8 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.

🥳 Co	onstant Z Finishing ? ×
Toolpath name	D20TR3-ConstZ-FIN1
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Machine tool</li> <li>Machine tool</li> <li>Stock engagement</li> <li>Constant Z finishing</li> <li>F High speed</li> <li>Unsafe segment removal</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Limit Boundary Boundary Trimming Keep inside Block Limit Solo Minimum Solo Minimum Solo

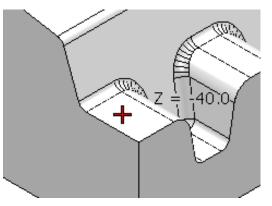
- 9 Select **Recycle** followed by the **Limit** page and **tick** both the **Maximum** and **Minimum** boxes.
- 10 For The Maximum select *Pick the highest Z Value* option.



11 Snap the *key-point* (shown above) at the top corner of the angled face to collect and insert the value **-11.82394**.



12 For The Minimum select *Pick the lowest Z Value* option.



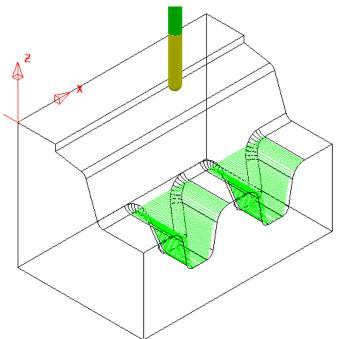
- 13 Snap the *key-point* (shown above) on the top of the flat area to collect and insert the value **-40**.
- 14 Back in the dialog select **Calculate** to re-process the updated **Constant Z** finishing strategy and then **Close** the dialog.
- 15 In the PowerMILL explorer Right mouse click on the toolpath BN8-OptConstZ-FIN1 and from the local menu, select Activate followed by Settings to open the strategy dialog.

r		0	٩.		1
L	C	Þ	Ł	2	1
14	$\sim$	λ.	.F	۲	l
H	9	۲	3	<	I

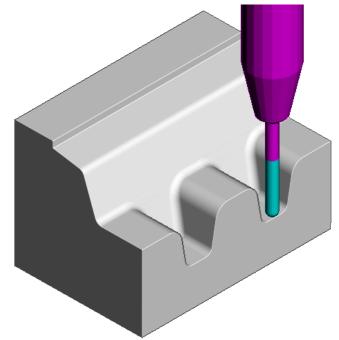
- 3
- **16** Select **Recycle** followed by the **Limit** page and this time **tick** just the **Maximum** box.



- 17 Input the value -40.02 in the Maximum box.
- 18 Back in the main form select Calculate to re-process the updated Optimised Constant Z finishing strategy and then Close the dialog.



 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.

20 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.

<b>;;;; Tools</b> 	2 Simulate from Start	
Boundarie Patterns Peature Set Workplane	Activate Activate Workplane Settings	Machine Tools MC Programs Toolpaths
Evels and     Models     Stock Mod	Queue	
·····Ø Groups	Select Surfaces	

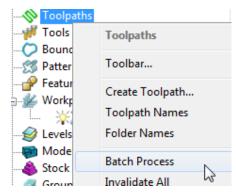
The '**Calculator**' symbol changes to the '**Queue**' symbol when the toolpath is queued.

#### **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.



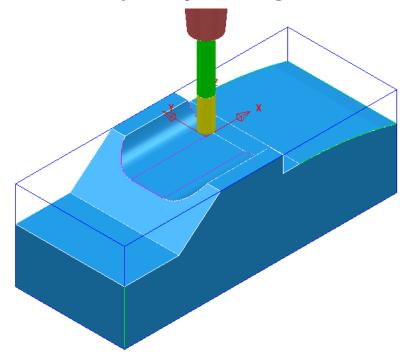
### **Toolpath Processing Queue**

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



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**.

	13-rgh1 D25t3-rgh1	⊡ (Noolpaths ⊕ (?) (Noolpaths ⊕ (?) () () () () () () () () () () () () ()	h1
Tools	Simulate from Start		BN16-fin1
🗄 🔋 <u>Ŭ</u> BN16	✓ Activate		Simulate from Start
🗄 – 💡 📙 BN6 💭 Boundaries	Activate Workplane	Boundaries	Activate
	Settings		Activate Workplane
Workplanes	Queue N	Workplanes	Settings
	4	₩ SMC-D:	Queue

- 6 While the original toolpaths are being processed in the background, Activate the tool BN6.
- 7 Open a Corner Finishing strategy and set the Strategy to Along. Enter the other values exactly as shown below.

<b>đ</b>	Corner Finishing	? ×
Toolpath name	BN6-fin1	
Workplane Block Tool Xachine tool Stock engagement Corner finishing Automatic verification Orner detection Orner detection Orner finishing Automatic verification Tool axis	Corner finishing Output Both v Threshold angle 30.0 Maximum passes Maximum numb	Strategy Along v Cusp 0.02
Machine axis control Rapid move heights Cleads and links Start point Field point Fields and speeds History Notes and Description	Tolerance 0.01 Thickness 	Cut direction Climb

8 Select the **Corner detection** page and select the **Reference tool** as **BN16** 

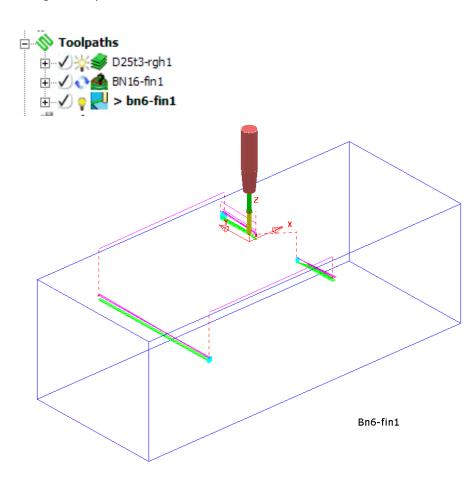
<del>3</del>	Corner Finishing ? ×
Toolpath name	BN6-fin1
Workplane	Corner detection
	Reference tool
	<b>€ • • • • • • • • • •</b>
Corner finishing	Use toolpath reference
Automatic verification	Toolpath

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 😒.



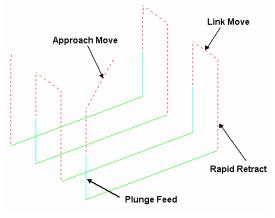
10 Select File - Save Project to update: .....\COURSEWORK\PowerMILL-Projects\ToolQueue-ex1

# 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.

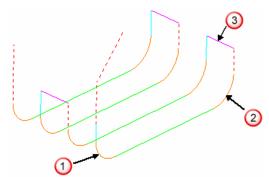
**Plunge distance** – An incremental distance above the local, component surface where a downward rapid movement of a tool changes to plunge feed move.

### 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:-

- 🛈 Lead In = Vertical Arc
- 🕗 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**.

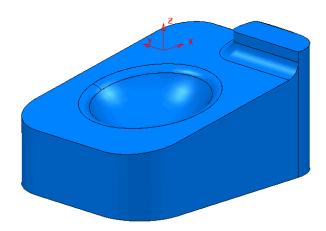
The current **Leads and Links** settings are automatically applied when creating new machining strategies.

### 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



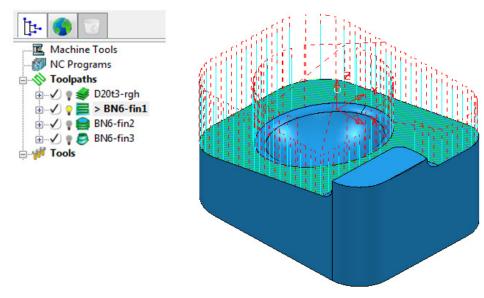
3 Select File - Save Project As:

.....\COURSEWORK\PowerMILL-Projects\LeadsLinks-example

4 Select an **ISO2** view.

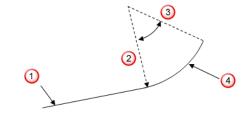


5 Activate the toolpath, BN6-fin1.



### **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.

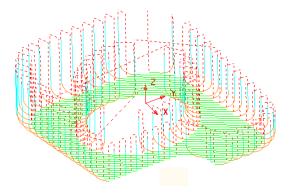


- 1 Toolpath segment
- 2 Radius
- 3 Tangent angle
- 4 Lead
- 6 On the **Main** toolbar, click 💟 to open the **Leads and Links** dialog.
- 7 On the **Leads and Links** dialog, select the **Lead In** tab and:

<del>3</del> 4	I	_eads a	nd Link	s		? ×
Extensions Links Point distribution					on	
Z heights	First lead in	Lea	d in 🛛	Lead out	Last l	ead out
1st choice			2nd cl	hoice		
Vertical arc		~	Non	e		~
	Distance 0.0				Distance [	0.0
	Angle 90.	0			<u>Angle</u>	0.0
	Radius 6.0				<u>Radius</u>	0.0

- 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.



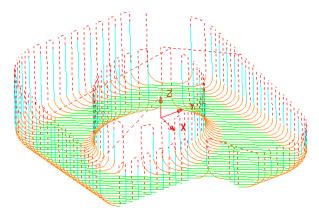


The new (gouge protected) **Vertical** leads have not been created on the tool tracks that stop close to a raised area of the model.

8 Click the Copy to Lead Out button.



9 Click **Apply** to update the active toolpath.

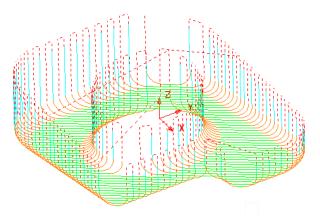


Where gouge free, a **Lead Out** move is applied at the end of each tool track.

0	<b>X</b>		Le	eads ar	nd Linl	cs	?	×
	Extension	ıs		Links		Point	distribution	
	Z heights	First lea	d in	Lead	lin	Lead out	Last lead	out
	1st choice				2nd c	hoice		
	Vertical arc			~	Verti	cal arc		~
		<u>Distanc</u>	<u>e</u> 0.0				Distance 0.0	
		Angl	<u>e</u> 90.0				Angle 90.	0
		<u>Radiu</u>	<u>is</u> 6.0				Radius 4.0	

10 Select Vertical Arc from the 2nd Choice list.

- a Enter an Angle of 90.0 and a reduced value Radius of 4.0
- **b** 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.

## 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.

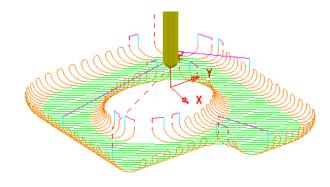
- Short/Long threshold This defines the distance limit up to which the Short -Links apply. Any move from the end of a tool track to the start of the next, which exceeds this distance is defined by the Long - Link move.
- For Short links, the available options include Safe Z, Incremental, Skim, On Surface, 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.
- **11** Activate the toolpath **BN6-fin1**.
- 12 Click and open the Leads and Links dialog.
- 13 Select the Z Heights tab to access the Skim and Plunge distance options.

<del>3</del> 2		Leads and Lir	iks	? ×
Extensio	ns	Links	Point	distribution
Z heights	First lead in	Lead in	Lead out	Last lead out
	5.0	listance → ↓ ↓ ↓ ↓ ← cremental distan		

- **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.
- 14 Select the **Links** tab.

<b>#</b>	L	eads and Lin	ks	? ×
Z heights	First lead in	Lead in	Lead out	Last lead out
Extensio	ins	Links	Point	distribution
	Short/Long thres	hold 3.0		
Retract and a	pproach moves		Short	
	Along Tool axis	~	Circular arc	~
- 🗸 Automat	ically extend		Long	
	Maximum leng	<u>ith</u> 250.0	Skim	~
			Default	
	Retract dista	nce 0.0	Incremental	~
	Approach dista	ance 0.0		

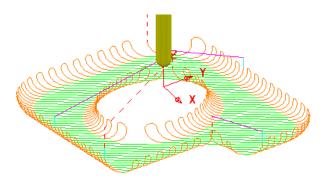
- a Enter the Short/Long threshold as 3
- **b** Link move distances less than this value will use the **Short** options and those greater will use the **Long** options.
- c Select *Short* as Circular arc, *Long* as Skim, and *Default* as Incremental.
- 15 Click **Apply**.





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).

- 16 Increase the value for **Short/Long threshold** to **8**.
- 17 Click Apply.





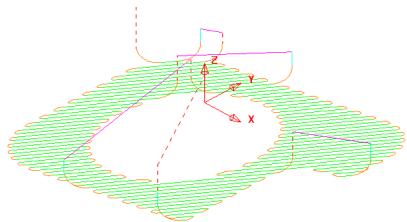
By increasing the value of the **Short/Long threshold**, the number of occurrences of **Short Link - Circular Arcs** has increased.

The remaining **Link** moves would benefit from **Vertical Arc** - **Lead In/Out** moves, but not on the existing **Circular Arc** Links.

- **18** Open either the **Lead in** or **Lead out** page.
- 19 Un-tick the Add leads to short links option.

0.0 Overlap distance (tool diameter units)	r
Allow start points to be moved	
Add leads to short links	
Add leads at tool axis discontinuities	
Angular threshold 90.0	

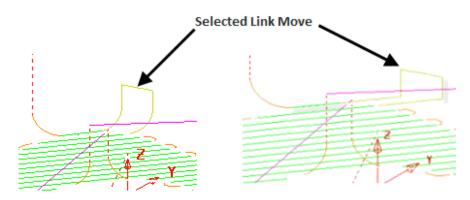
20 Click Apply.



The Lead moves exist on the Long links but not on the Short links.

### Applying Leads and Links locally to individual tool tracks

- 21 Use the *left mouse key* to select one of the individual link moves where the Lead is currently a **Vertical Arc**.
- 22 In the Leads and Links dialog, select the Lead In tab and select 1st Choice as Extended Move with Distance 10.
- 23 Select the Copy to Lead Out tab.
- 24 Click Apply.
- 25 The selected Vertical Arc lead move is changed to an Extended move.

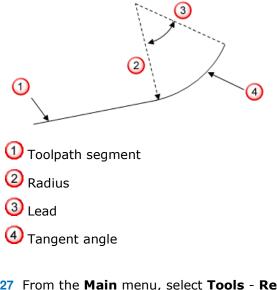


26 Select File - Save Project to update:

.....\COURSEWORK\PowerMILL-Projects\LeadsLinks-example

### **Horizontal Arc Leads**

This produces circular leads on the horizontal plane. This type of lead is frequently used for toolpaths running at constant Z, or with only small changes in Z height.

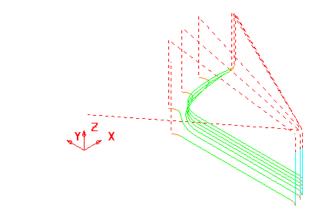


- 27 From the Main menu, select Tools Reset Forms to return the Leads and Links settings to default.
- 28 In the **PowerMILL** *explorer*, right-click the **BN6-fin2** toolpath and select **Activate**.
- 29 Select an **ISO1** View.

30 Click I on the Main toolbar to open the Leads and Links dialog.

×		Leads a	nd Linl	s		? ×
Extension	ıs	Links		Point	distributio	n
Z heights	First lead in	Lead	d in [	Lead out	Last le	ad out
1st choice			2nd cl	noice		
Horizontal arc		~	Non	e		~
	Distance 0	).0			Distance [	0.0
	Angle 4	5.0			Angle 9	90.0
	Radius 3	.0			Radius (	4.0
0.0 Overlap	distance (tool d	diameter u	nits)	F	amp option	ns

- 31 Select the Lead In tab, and change 1st Choice to Horizontal Arc with an Angle of 45 and a Radius of 3.0
- 32 Click Copy to Lead Out and then Apply.



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.

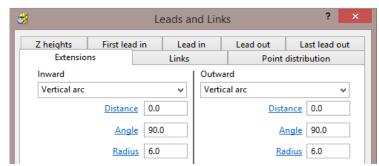
**33** In the existing **1st Choice** - **Lead In** page input a value of **5** for the **Distance** and select **Apply**.

I	ads and Link
Extensions Z heights First lead in choice orizontal arc 5.0 Angle 45. Radius 3.0	Links Lead in 2nd ch V None

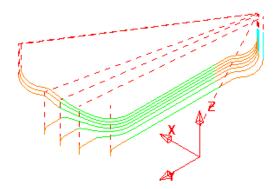
### **Extensions**

**Extensions** are additional lead options that are applied outside the existing **Lead In/Out** moves.

- 34 Select an **ISO 4** view.
- 35 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



36 Click Apply.



A Vertical Arc - Extension is added to all of the Lead In/Out moves.

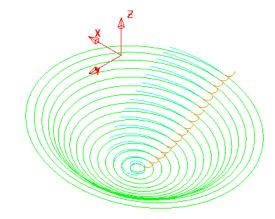
### **Additional Short Links options**

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 gougechecked 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.

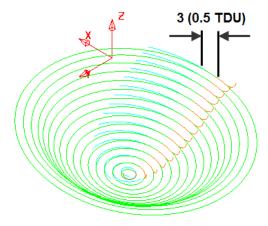
### **Exercise**

- 1 Activate the BN6-fin3 toolpath and Apply the Leads and Links as follows:
  - a Links: Long/Short/Default Skim.
  - 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.

- 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**).

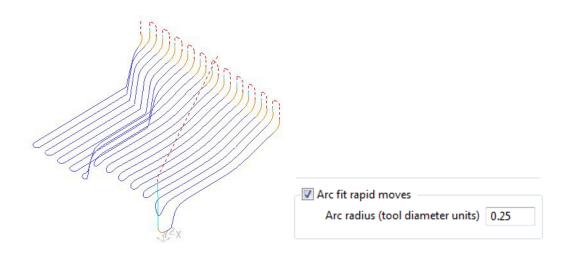
### **Arc Fitting Rapid Moves**

### 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.



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**).

# 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.

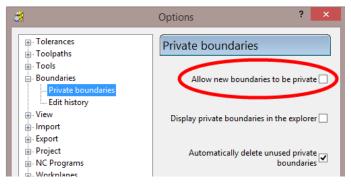
For example, **Constant Z finishing** is more effective on steep sidewalls and **Raster finishing** on shallow areas.

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 existing geometric, entities.

	Boundaries		
Workplar 🥩 Levels an ன Models	Curve Editor Toolbar		
Stock Mc	Display private Boundaries		
Macros	Create Boundary	•	Block
_	Paste as new Boundary		Rest
	Boundary Names	+	Selected Surface
	Folder Names	•	Shallow
	Batch Process Invalidate All		Silhouette
	Deactivate Draw All Undraw All		Stock Model Rest Contact Point Contact Conversion Boolean Operation
	Thickness		User Defined

#### Switch off the Private Boundaries creation option

- 1 From the Main menu, select File Delete All and Tools Reset Forms.
- 2 From the main pulldown menus select Tools Options and in the dialog select Boundaries – Private boundaries.



- 3 Un-tick the box named Allow new boundaries to be private.
- 4 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.

<del>3</del>	User Defined Boundary	? ×
👔 🔓 📿	Name 1	
Wirefr	Insert File	Clear 🟑
Private	to be private	0
Edit History	y on calculation	${ \textcircled{\sc l}}$
Ар	oply Queue Accept Cancel	

**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.



Sketch - Opens the Curve Editor toolbar.

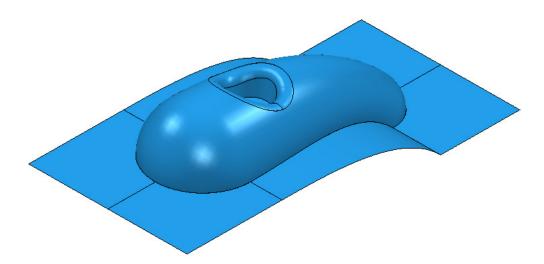
Composite curve - Opens the Create Composite Curve toolbar.

Wireframe Modelling - Provides access to the PowerSHAPE wireframe module.

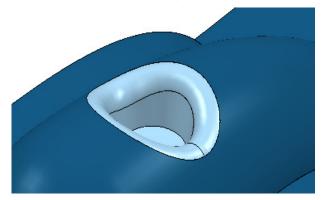
5 Select File - Save Project As: .....\COURSEWORK\PowerMILL\_Projects\BoundaryBasics

Select File - Import Model... 6

.....\PowerMILL\_Data\Models\cowling.dgk



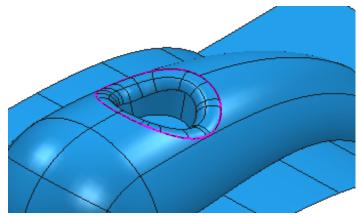
7 Select the **surfaces** defining the central pocket and fillet.



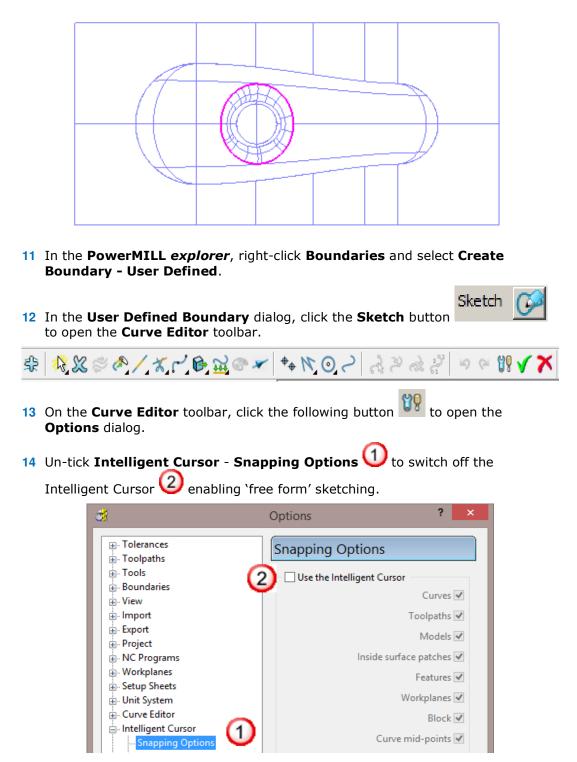
- 8 In the **PowerMILL** *explorer*, right-click **Boundaries** and select **Create User Defined**.
- 9 In the **User Defined Boundary** dialog, click the **Model** button.



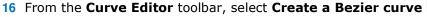
A **Boundary** segment is created around the edge of the selected part of the model.



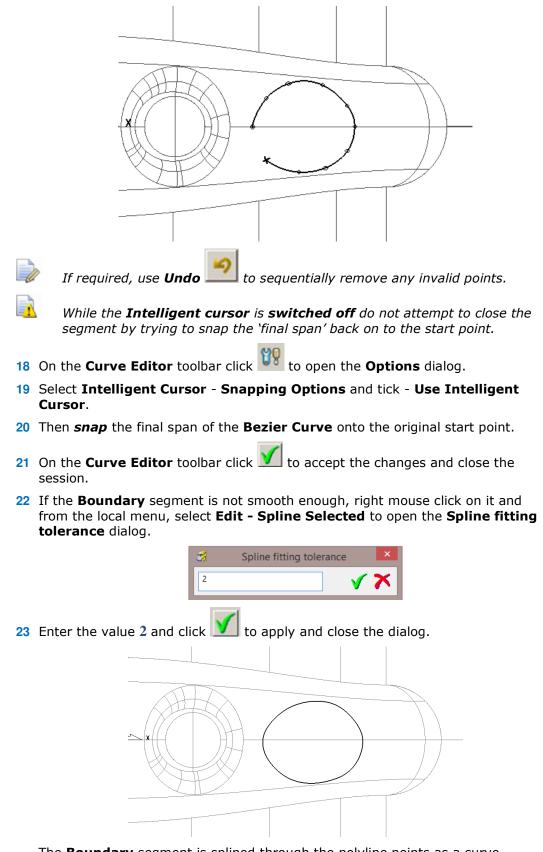
**10** Select a *View from top* (Z) and switch off the **shaded** view but retain the **wireframe** view.



15 Click Accept to keep the changes.



17 Using the left mouse key, click points within the model to create a sketched **Boundary**.



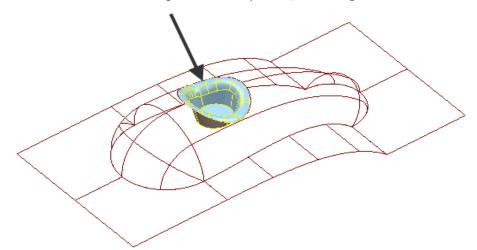
The **Boundary** segment is splined through the polyline points as a curve maintaining the original form to within a maximum deviation of 2.

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 position 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**.

Selected Sur	face Boundary ? ×	
🛐 🔓 💩 Name	3	
Top Roll Over Tolerances0.02	Limit Boundary     Inside     Limiting Boundary     Outside	
Ihickness     0.0       Axial Thickness     0.0       Use Axial Thickness     0       Tool     Image: State S	Automatic Collision Checking Holder Clearance 0.0 Shank Clearance 0.0	
Private	Limit 🔬 🗸	
Edit History  Apply edit history on calculation	<b>S</b>	

5 Using the above settings, click **Apply** to create a single segment, **Selected Surface Boundary**.

## **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**.

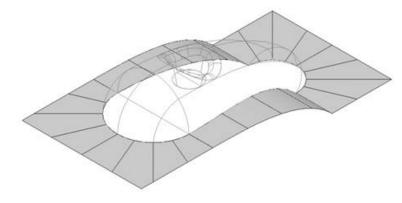
Image: Solution of the second sec	*	Sha	allow Boundary	?
Upper Angle 0.0   Lower Angle 0.0   Tolerances 0.1   Thickness 0.0   Avial Thickness 0.0   Use Avial Thickness 0   Use Avial Thickness 0   Use Avial Thickness 0   Block Limit   Limit v	22 6	<u>&amp;</u>	Name 4	
Iolerance   0.1   Thickness   0.0   Axial Thickness   0.0   Use Axial Thickness   Tool   Holder Clearance   0.0   Shank Clearance   0.0   Block   Limit go v   Private   Allow boundary to be private   Edit History   V Apply edit history on calculation   (2)	Tolerances		Inside	
Axial Thickness   Use Axial Thickness   Tool   Image: Shark Clearance   BN16   Block   Limit Image I   Allow boundary to be private   Image: Clearance   Allow boundary to be private   Image: Clearance   Image:	Tolerances	Tolerance 0.1		
Use Axial Thickness Use Axial Thickness Holder Clearance 0.0 Use Axial Thickness Holder Clearance 0.0 Shank Clearance 0.0 Block Limit Sov Edit History Apply edit history on calculation (2)				
Image: BN16   Block   Limit Image: Clearance 0.0   Block   Limit	<u></u>		s 🗌 🗌 Automatic	Collision Checking
Limit g ∨         Private         Allow boundary to be private         Edit History         Apply edit history on calculation		8N16		
Private Allow boundary to be private Edit History Apply edit history on calculation			Block	
□ Allow boundary to be private     ①       Edit History     ②       ☑ Apply edit history on calculation     ③				Limit 👷
Apply edit history on calculation	22332	ndary to be private		
Apply Queue Accept Cancel		t history on calculation		
		Apply Que	ue Accept	Cancel

- 1 Shallow area more suited to Raster Finishing (above 30 degrees).
- 2 Steep area more suited to a Constant Z Finishing (below 30 degrees).
- 2 **Apply** the form 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**.

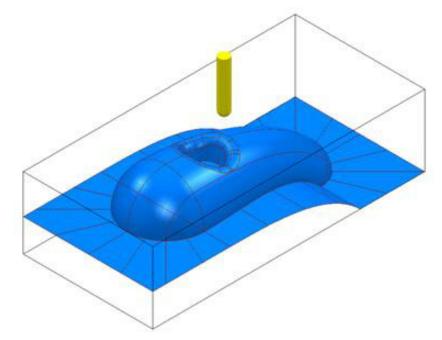
3	Silhouette	Boundary ? ×
22 6	🔋 🥬 Name	5
Tolerances	On Model 🗸	Limit Boundary     Inside     Limiting Boundary     Outside
	Tolerance 0.01 Vertical Tolerance 0.01 Thickness 0.0	Automatic Collision Checking
Tool	Axial Thickness 0.0 Use Axial Thickness D	Holder Clearance 0.0 Shank Clearance 0.0 Block
	oundary to be private	()
Edit History	/ dit history on calculation	0
	Apply Queue	Accept Cancel

# **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



4 In the **PowerMILL** *explorer*, right-click **Boundaries** and select **Create Boundary - Rest** to open the **Rest Boundary** dialog.

; II 6 &	Rest Boundary ? × Name 6
Detect Material Thicker Expand Arr Tolerances Toleranc Thicknee	Outside Limiting Boundary     Outside
Axial Thickness Use Axial T Tool BN8	
Reference Tool U BN16	Block
Private	vate 🚺
Edit History Apply edit history on calc	ulation
Apply	Queue Accept Cancel

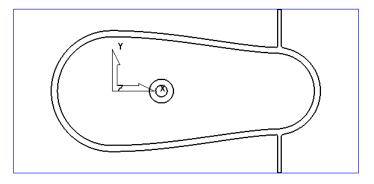
- 5 Input *Expand Area* 0, select *Tool* BN8, and *Reference Tool* BN16.
- 6 Set with *Limit tool centre to block edge*.
- 7 Click **Apply** to create the following **Rest Boundary**.
- 8 Click **Accept** to close the dialog.
- 9 Save the Project to update:-

.....\COURSEWORK\PowerMILL\_Projects\BoundaryBasics

10 Select a View down the Z Axis an

and **Un-draw** the model.

Limit



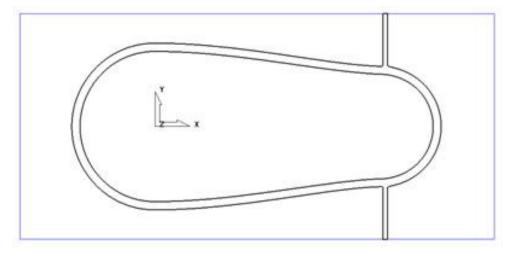
The above **Boundary** identifies areas on the component that are inaccessible to the **BN16** tool geometry to be locally machined with the **BN8** tool.

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 are 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**.

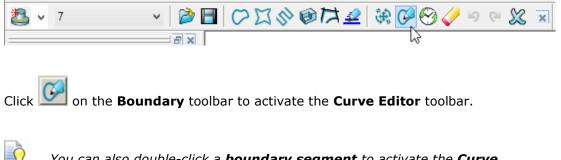
- 1 In the **PowerMILL** *explorer*, right-click the **Rest Boundary** (6) entity and in the local menu select **Edit Copy Boundary** to create a copy named (**6\_1**).
- 2 In the **PowerMILL** *explorer*, right-click the original **Rest Boundary** (6) and *Rename* it as **Master**.
- 3 Rename the new copy of the Rest Boundary (6\_1) as Rest.
- 4 Activate the Boundary named Rest.
- 5 Select the **2** *inner segments* on the **Rest boundary** (**Rest**) by dragging a box over them, and then press the **Delete** key on your keyboard to delete them.



The next stage will be to Offset the Boundary and then 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.



You can also double-click a **boundary segment** to activate the **Curve Editor** Toolbar.

# Edit > Transform

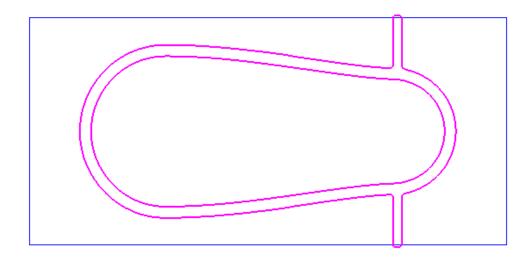
- 1 In the **PowerMILL** *explorer*, right-click on the boundary **Rest** and from the local menu, select **Curve Editor**.
- 2 In the Curve Editor toolbar select the Offset option



3 In the **Offset** toolbar, select the **3D Smooth** button, input a **Distance** of 1 and then press the **Return** key.

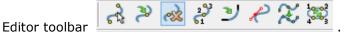


4 Click Main the **Curve Editor** toolbar to accept the changes and exit.

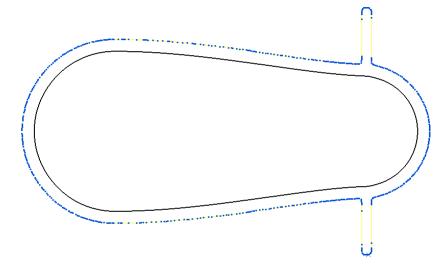


# **Curve Point - editing**

The **Point** editing options are located towards the right hand side of the Curve

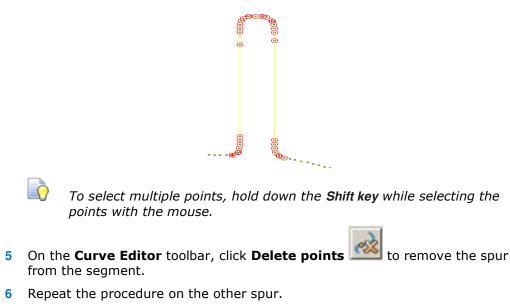


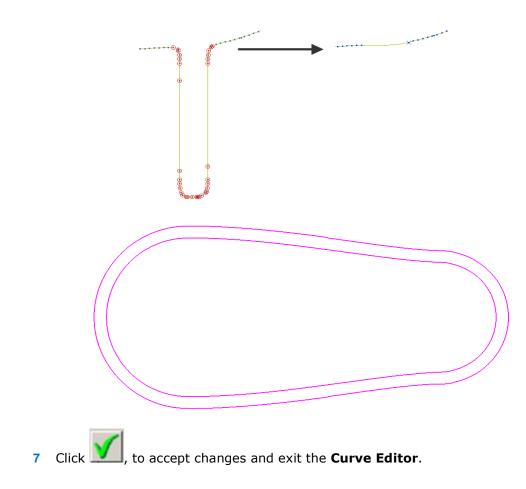
- 1 In the **PowerMILL explorer**, right-click **Boundaries** and from the local menu select **Toolbar**.
- 2 On the **Boundary** toolbar, click the **Curve Editor** button is to display the **Curve Editor** toolbar.
- 3 Select (Left click) the outer **boundary segment** in the graphics area.



The points are displayed on the selected boundary segment.

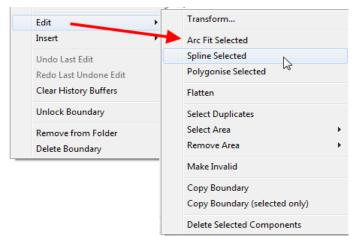
**Zoom** into the area of the selected segment and select all the points that define the spur.





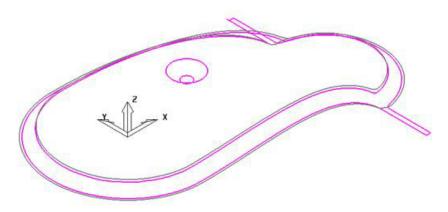
# Other boundary editing options

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.



### **Review the modified Boundary**

8 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.

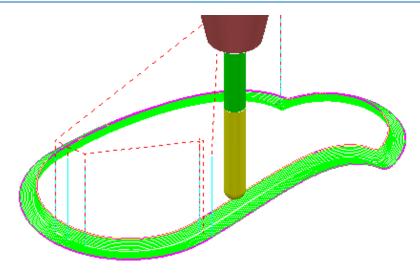


## Create a 3D Offset strategy limited to the modified Boundary

- 9 Activate the BN8 tool and Boundary named Rest.
- 10 On the **Main** toolbar, click to display the **Strategy Selector**.
- 11 From the Finishing options, select the **3D Offset Finishing** strategy.
- 12 In the **3D Offset Finishing** strategy dialog, enter data as below.

- 31	3D Offset Finishing ? ×
Toolpath nam	BN8-FIN1
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Point distribution</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>Fistory</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	3D offset finishing     Pattern   Image: Start on patt

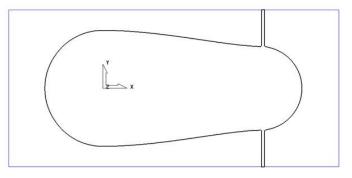
13 Click **Calculate** to process the toolpath and then **Close** to exit the dialog.



### Create a Boundary for the original side branches.

- 14 Undraw the *Toolpath* BN8-FIN1 and *Boundary* Rest by switching off the respective light bulb symbols in the **PowerMILL** *explorer*.
- 15 Switch on the respective light bulb to display the **Boundary** named **Master**.
- 16 In the graphics area, select the *outer segment* of the displayed **Boundary** named **Master**.
- 17 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.
- **18 Rename** the boundary **Master\_1** as **Spur1**.
- 19 Select a view from top Z.
- 20 In the **PowerMILL** *explorer*, turn off the light bulb adjacent to the boundary **Master** to remove it from view.

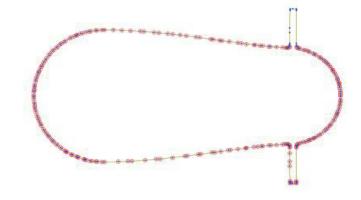


- 21 In the **PowerMILL** *explorer*, right-click the boundary **Spur1** and select **Activate**.
- 22 In the graphics area, double-click the **boundary segment** to activate the **Curve Editor** toolbar.
- 23 Select all **points** along the **boundary segment** except those which define the rectangular 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 preferred it is okay to select and delete the points in stages.

24 On the Curve Editor toolbar:



a Click the **Delete Points** button points.

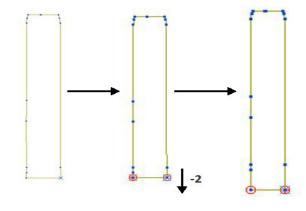
to delete all the selected

- **b** Select the **points** at the base of the rectangular segment.
- **c** On the **status bar** (below the graphics area), click the **Relative Move**

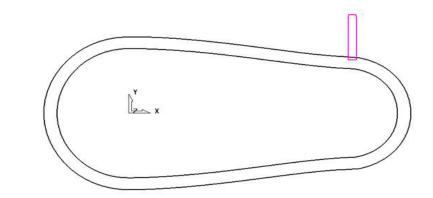
button and enter **0** -2 (for the X and Y move, values) in the data box and press *Enter* on your keyboard.



The selected points are moved by a distance of **Y-2**.



- 25 Click Multiple to exit the **Curve Editor** toolbar.
- 26 Click the light bulbs on to display both *Boundaries* Rest and Spur1, and then Resize the View .



- 27 In the graphics area, double-click the *Boundary* Spur1 to activate the Curve Editor toolbar.
- 28 Form the Curve editor toolbar, select Transformations Mirror option

29 On the **Mirror** toolbar select **Origin at the active workplane**.

30 Ensure that the **Keep Original** option is set.

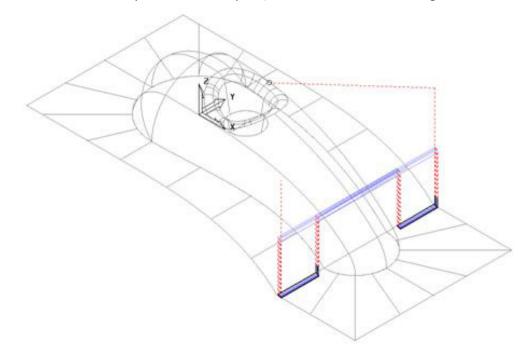
31 From the Mirror toolbar, click the Mirror in ZX button
32 Click I on the Mirror toolbar to accept and close.
33 On the Boundary Transform toolbar, click I on accept and exit the toolbar.

The **Boundary** named **Spur1** now contains 2 segments.

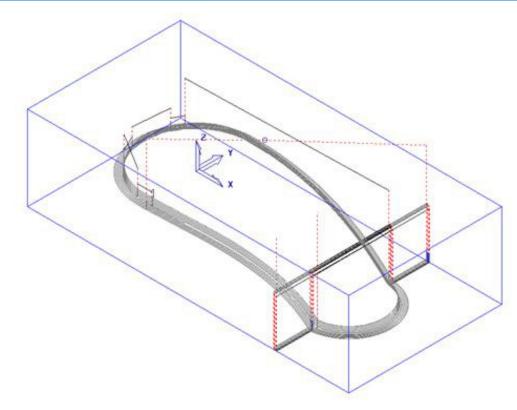
- 1 Make sure that the *tool* BN8 and *boundary* Spur1 are both Active.
- 2 From the **Strategy Selector**, **Finishing** options, select the **Raster Finishing** strategy and enter data as shown on the following page:

<b>3</b>	Raster Finishing ? ×
Toolpath name	BN8-Spur-FIN
	Raster finishing
U Tool Machine tool Limit Stock engagement	Fixed direction Angle 90.0
E	Start corner Lower right ¥
Automatic verification     Point distribution     Yool axis     Machine axis control	Perpendicular pass
- I Rapid move heights I Leads and links J Leads and links J Leads and links J Leads and links I Leads and links I Leads and speeds	Ordering Style One way grouped Arc radius 0.0
History     Solution     S	Tolerance 0.02
	Thickness
	Stepover 0.3 0.0005629

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



The machining strategies are localised with the help of edited **Boundaries** originating from the original **Rest Boundary** (**Master**).



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.



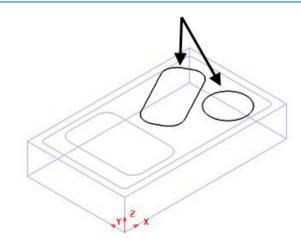
- 4 Select the *top surface* of the **model** (shown shaded).
- 5 In the **PowerMILL** *explorer*, right mouse click on **Boundaries** and from the local menu select **Create Boundary User Defined**.

<b>₫</b>	User Defined Boundary	? ×
1	♥ Name 1	
Insert	Edit	Clear 🏑
Boundary	✓ Ø	
Pattern	✓ X	
Toolpath	Y 🔊	
	Model	
	Sketch	
	Composite curve 🎦	

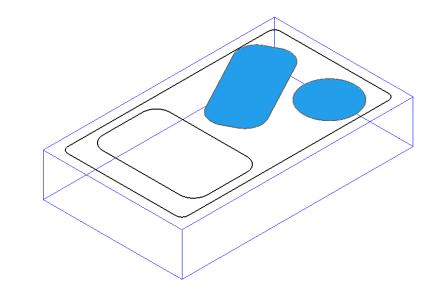
- 6 Click the **Model** icon and then select **Accept** to close the dialog.
- 7 Un-draw the **model** to enable a clear view of the **boundary**.

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**.
- 10 Switch on the Plain Shade model option.
- 11 Planer **Surfaces** have been created inside the selected **boundary segments**.



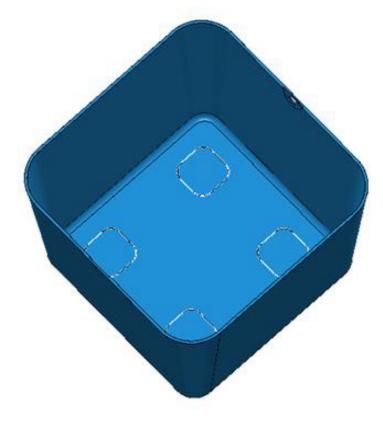


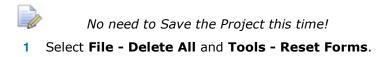
No need to Save the Project this time!

12 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.

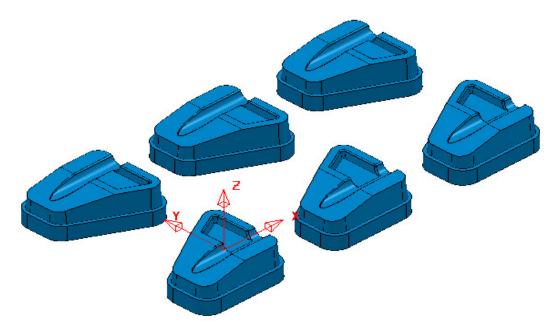




# **Block defined by Boundary**

An existing **Project** will be opened that contains multiple components that will be roughed out using a complex **Block** arrangement defined by **Boundary** segments.

- 2 Select File Delete All and Tools Reset Forms.
- 3 Open the (read-only) Project:-
- ...\PowerMILL\_Data\Projects\MultiBlock\_Start



- 4 Save the Project as:-
  - ....\COURSEWORK\PowerMILL\_Projects\MultiPart\_Block
- 5 Select all of the component models (Drag a box over all **surfaces**).
- 6 In the **PowerMILL** *explorer* right mouse click on **Boundaries** and from the local menu select **Create Boundary User Defined**.
- 7 In the User Defined Boundary dialog ensure that the Allow Boundary to be

private box is un-ticked and then click on the Model icon

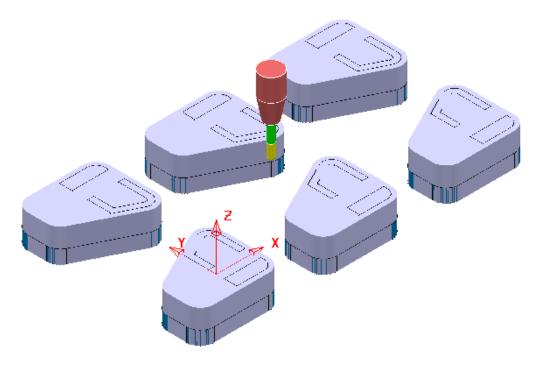
- 8 Select Accept to close the User Defined Boundary dialog.
- 9 Activate the new Boundary.



- 10 From the main toolbar select the **Block** options
- 11 In the dialog select **Defined by Boundary** and **Type Model** followed by **Calculate**.

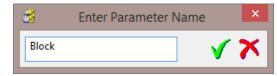
<b>3</b>	Block		?	×
Defined by				
Boundary	~		D	X
Coordinate System				
Active Workplane	~			~
Limits				
Min	Max		Length	
× -75.0	525.0		600.0	S
Y -45.48848 ₽	295.48848		340.97697	5
Z -60.0	0.00003	S	60.00003	S
X Y Z	¥ Z		<b>A</b>	<del>0-1</del>
Estimate limits				
Tolerance	Expansion		Туре	
0.1	0.0		Model	~
Include reference	surfaces		Calcul	ate
✔ Draw		Ора	city	
Ad	cept	Can	cel	

12 Select Accept to close the **Block** dialog.



The complex **Block** is as shown above consisting of localised triangle models around each individual component.

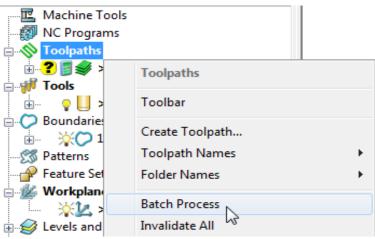
- **13** De-activate the **Boundary**.
- 14 In the PowerMILL explorer right mouse click on the existing, un-processed toolpath D12TR1-RGH1 and from the local menu select Edit Set Named Parameter.



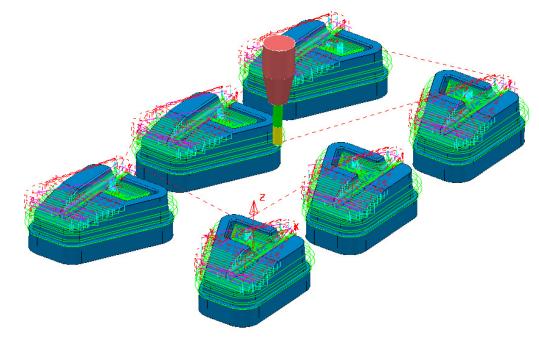
15 Input **Block** and select the *green tick* to accept.

The current **Block** definition will now be active within the unprocessed **Model Area Clearance** strategy.

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



**Batch Process** will sequentially run down the **toolpath list** (in this case 1) and **Calculate** all valid, un-processed strategies.



17 Select File - Save Project to update:-...\COURSEWORK\PowerMILL\_Projects\MultiPart\_Block

# **Private Boundaries**

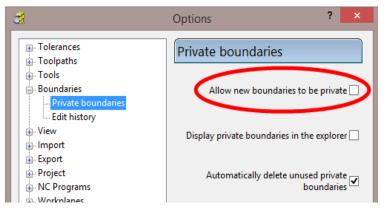
All **Boundary** types will (by default) be created as **private**. This makes the **Boundary** exclusively owned by an individual machining strategy. If the **parameters** or **entities** used in the strategy are modified, then the associated **private boundary** will automatically adjust on calculation.

**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 within the dialog.

It is also possible to prevent Boundaries from being created as private:-

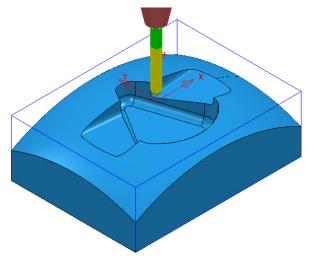
From the **main** pull down menus select **Tools – Options – Boundaries – Private Boundaries**, and un-tick **Allow new boundaries to be private**.



# **Private Boundaries – Example:-**

- 1 Select File Delete All and Tools Reset Forms.
- 2 Open the (readonly) Project:-

.....\PowerMILL\_Data\Projects\PrivateBoundaries\_Start



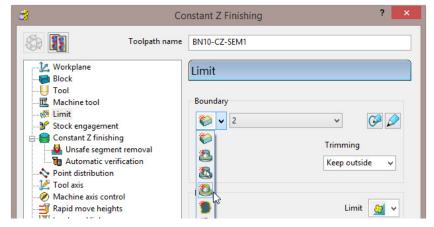
- 3 Save Project As:-
  - .....\COURSEWORK\PowerMILL\_Projects\PrivateBoundaries-EX1

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:-

Allow new boundaries to be private 📝

- 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.



10 From the *Boundary* pull down options select Create a shallow boundary.

<del>3</del> ă	Shallow	Boundary ? ×
1	🚨 Name	e 1
Tolerances	Upper Angle     40.0       Lower Angle     0.0       Tolerance     0.1       Thickness     0.5       Axial Thickness     0.0	<ul> <li>Limit Boundary</li> <li>Inside Limiting Boundary</li> <li>Outside </li> </ul>
Tool	Use Axial Thickness	Automatic Collision Checking Holder Clearance 0.0 Shank Clearance 0.0
		Block
Private ✓ Allow boun	dary to be private	0
Edit History Apply edit I	nistory on calculation	$\odot$
	Apply Queue	Accept Cancel

11 Using the default settings **Apply** and **Accept** 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** 

	Boundaries
Feature Sets     Workplane:	Toolbar
∐ <u>×k</u> > —	Display private Payradarias
Levels and S	Display private Boundaries

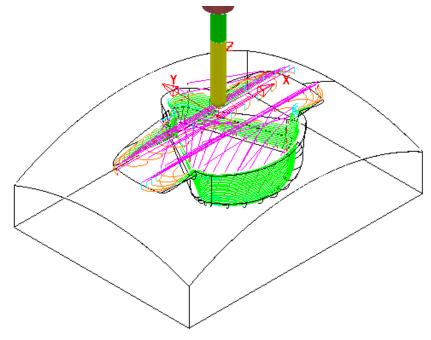


The **private Boundary** will be displayed in the **PowerMILL explorer** if the above option is **ticked** in the local **Boundaries** menu.

12 Select the **Limit** page and set *Trimming* to **Keep outside**.

Limit	
Devendent	
Boundary	
۵ 🗸 🚺	- 🖉 🖉
	Trimming
	Keep outside 🗸 🗸

**13** Calculate (but <u>do not</u> Close) the Constant Z Finishing strategy.

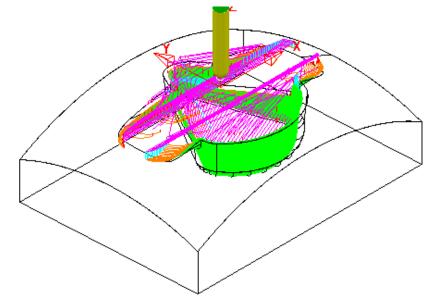


The **Constant Z** strategy **BN10-CZ-SEM1** is now limited to be *outside* the segments of *private* **Boundary 1** 

**14** Select the `create a copy' option **in the Constant Z Finishing** dialog.

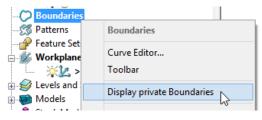
💰 Co	nstant Z Finishing		?	×
Toolpath name	BN10-CZ-FIN1			
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Constant Z finishing</li> <li>Unsafe segment removal</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> </ul>	Constant Z finishing Order by Region v Spiral	Additional stock 0.5 Undercut Flat tolerance 0.1		
Machine axis control Rapid move heights Leads and links Start point End point Feeds and speeds History Notes and Description User defined settings	Tolerance 0.01 Thickness 0.0 Stepdown <u>Minimum</u>	Cut direction Climb	v Usp	

- 15 Modify 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.

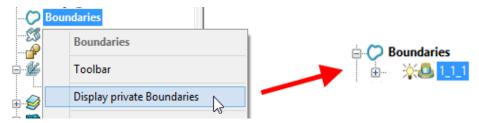
**18** In the **PowerMILL** *explorer*, right mouse click on **Boundaries** and from the local menu **tick** the option **Display private Boundaries**.



19 With the 2 existing *private* **Boundaries** displayed, right mouse click on either and from the local menu select **Edit** – **Copy Boundary**.



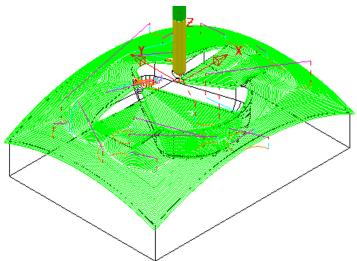
20 Right mouse click on **Boundaries** and from the local menu **untick** the option **Display private Boundaries**.



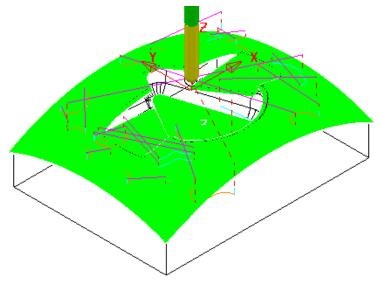
Only the new **private Boundary** will be displayed in the **PowerMILL** explorer but will later become hidden once it has been applied in a new toolpath calculation.

# **Exercise**

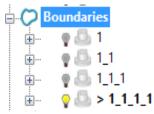
- 21 In the **PowerMILL** *explorer*, **Activate** the toolpath **BN10-3Doff-SEM1** and then select **Settings** to open the dialog.
- 22 *Recycle* with the new *private* Boundary 1\_1\_1, Active and with *Trimming* set to Keep inside.



23 Calculate a *Copy* of the above toolpath with the new *Name* BN10-3Doff-FIN1 and with Tolerance 0.01, Thickness 0, and Stepover 0.3



- 24 From the **main** pulldown menus select **Tools Options** and in the dialog select **Boundaries Private boundaries**.
- 25 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**.

# 7. Levels and Sets

**Levels** and **Sets** allow you to control the selection and display of model components in the **PowerMILL** graphics area.

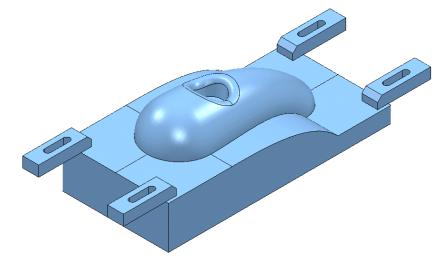
You can *Acquire* components from the currently imported **Model** to a **Level** or **Set**. Once acquired to a **Level** or **Set**, you can select, de-select, display, or hide *Model components* from the individual **Level** or **Set** context menus.

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**.
- Must not be included in the calculation of the **Block**.

To make the model selection easier, a new **Level** named **Clamps** will be created and all clamp surfaces assigned to it. Another **Level**, named **Splits** will be created and all 6 split surfaces assigned to it.

3 In the **PowerMILL** *explorer*, right-click **Levels and Sets** and select **Create Level**.

	Sets
	Levels and Sets
	Create Level
- Macros	Create Set
	Deactivate
	Draw All
	Undraw All
	Delete Empty Levels and Sets
	Delete All Except Non-Empty Levels

4 Rename the new Level as Clamps.



5 Create another new Level and Rename it as Splits.



All model components are currently acquired to the default General Level.

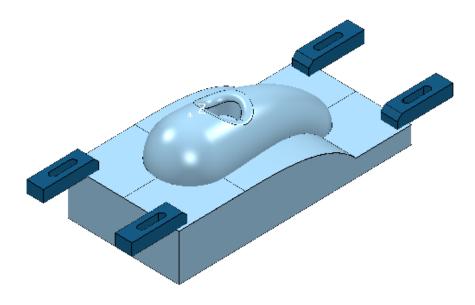
If the original CAD model has components already assigned to different **Levels**, then these will be recreated when the **model** is imported into the **PowerMILL project**.

# Acquiring model entities to a Level

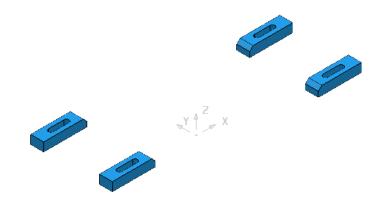
6 In the **PowerMILL** *explorer*, click the 'light bulb' off next to the **Level** named *Clamps*.

Any items acquired to this **level** will no longer be displayed.

7 In the graphics area, use the *left mouse* and *Shift* keys to select all **surfaces** except the clamps.



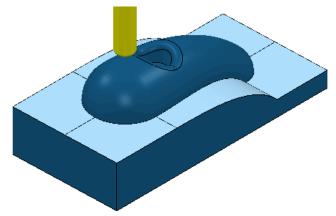
8 On the keyboard select **Ctrl J** to *blank* the selected **Surfaces**.



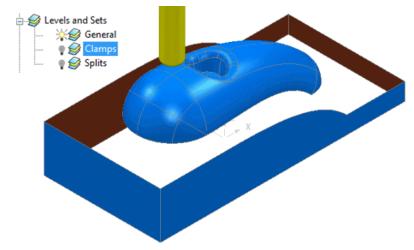
- 9 Select the *Clamp* surfaces (Hold down the left hand mouse key while dragging a box over them).
- 10 In the **PowerMILL** *explorer*, right-click on the **level** named *Clamps* and select *Acquire Selected Model Geometry*.

The clamps disappear from the view (The Level is switched off).

11 On the keyboard select **Ctrl L** to *unblank* all eligible **Surfaces**.



- 12 Select the 6 **Surfaces** defining the split face and **Acquire** them to the level named **Splits**.
- **13** Switch off the **level** named *Splits* to visually check that the correct **surfaces** have been acquired.



**14** Switch on all **levels** (The clamps and split **surfaces** reappear).

A **Selected Surface Boundary** will now be created to **limit** 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 Create a Dia 12 Ball Nose tool named BN12.
- 4 Calculate a Block using the default Box and Model options.
- 5 Use the default **Rapid Move Heights** and **Start and End Point** options.

<del>3</del> 1	Block	? ×
Defined by		
Box	¥	🔌 🖪 🗶
Coordinate System		
Active Workplane	¥	~
Limits Min X -60.00876 Y -50.00316 Z -40.0 E	Max 150.00634 50.005511 30.0	100.00867
Estimate limits		
Tolerance 0.1	Expansion 0.0	Type Model V
0.1	0.0	Model 🗸
Include reference s	urfaces	Calculate
✓ Draw		Opacity
Acc	ept	Cancel

The **Block** is created to the dimensions of the selected surfaces.

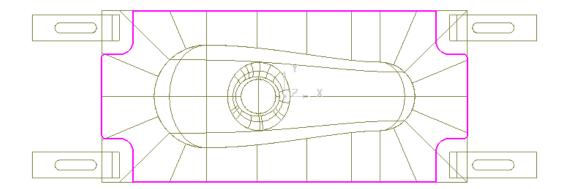
- 6 Left mouse click in the graphics area away from the **model** to deselect all entities.
- 7 In the **PowerMILL** *explorer*, click on the **Level**, **General**, and with the **Ctrl** key held down, click on the **Splits level**.

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



- 8 Right-click on one of the highlighted levels and choose **Select Surfaces**. All **surfaces** excluding the clamps are selected.
- 9 Right mouse click on **Boundaries** in the **PowerMILL** explorer.
- 10 From the local menu select Create Boundary Selected Surface Boundary with Radial Thickness 2, Limit set to Block Edge, with Allow boundary to be private un-ticked.

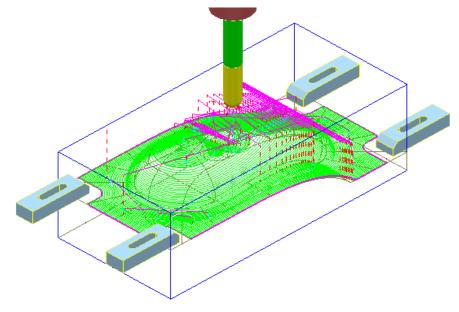
Selected Surface Boundary ? ×						
🚺 🔓 💩 Name	1					
Top  Roll Over Tolerances Tolerance 0.02 Radial Thickness 0.0 Axial Thickness 0.0 Use Axial Thickness	Limit Boundary  Inside Limiting Boundary  Outside  Automatic Collision Checking Holder Clearance 0.0					
Tool N12 V	Shank Clearance 0.0					
	Block					
Private Allow boundary to be private						
Edit History P Apply edit history on calculation	<b>©</b>					
Apply Queue	Accept Cancel					



- 11 Check that both the **Tool BN12** and **Boundary 1** are **Active** so that they will be automatically included when a new machining strategy is created.
- 12 From the main toolbar, select Toolpath Strategies and from Finishing open a Steep and Shallow Finishing strategy.
- **13** In the **Steep and Shallow Finishing** dialog, enter data exactly as shown in the following illustration.

😚 Steep and Shallow Finishing ? 🗙						
Toolpath name	BN12-FIN1					
Workplane     Block     Tool     Tool     Machine tool     We Limit     Stock engagement     Steep and shallow finishing     Point distribution     Point distribution     Automatic verification     Point distribution     Rapid move heights     Leads and links     Start point     Feeds and speeds     History     Notes and Description     Sure defined settings	Steep and shallow fini Order Top first v Threshold angle 30.0 Steep Spiral Cut direction Climb v Stepdown $rac{100}{100}$ 1.0 1.0 Calculate using cusp	Additional stock 0.6 Steep shallow overlap 0.0 Shallow Stepover Stepover Stepover Stepover Stepover Stepover State Stepover Stepover State Stepover Stepover Stepover Stepover Stepover Stepover Stepover Stepover Shallow Stepover				
	Tolerance       0.1       Thickness	K Cancel				

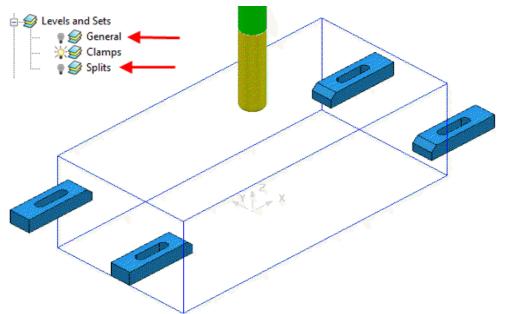
14 Click **Calculate** to process the **toolpath** and then **Close** the dialog.



The **Radial Thickness** of **2** used in the outer **Boundary** *segment* provides a suitable clearance around the clamps.

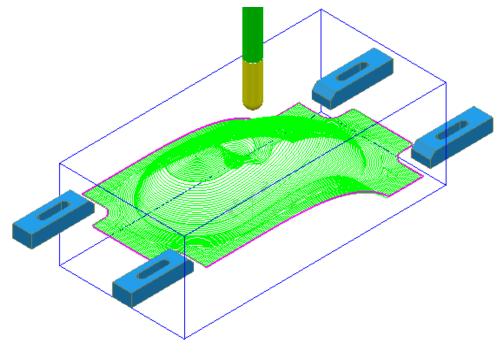
# **Switching Off a Level**

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.

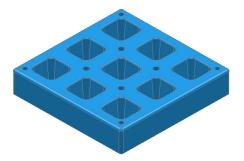
# Sets

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:

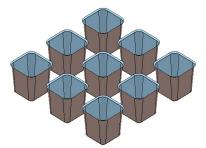
.....\PowerMILL\_Data\Models\YogurtTray.dgk



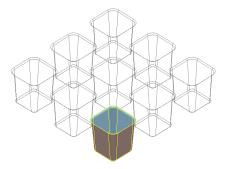
- 3 In the **PowerMILL** explorer right mouse click on Levels and Sets.
- 4 From the local menu, select **Create Set** and rename it as **AllPkts-Set**.



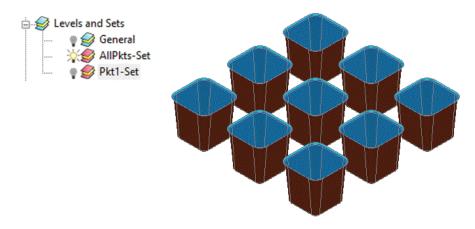
5 Manually select all **surfaces** defining the 9 pockets.



- 6 With the 9 pockets still selected, right-click **AllPkts-Set** and select **Acquire Selected Model Geometry**.
- 7 Manually select all **surfaces** defining the bottom left pocket.



- 8 Create a new **Set** and rename it as **Pkt1-Set**.
- 9 Right-click **Pkt1-Set** and select **Acquire Selected Model Geometry**.
- 10 On the keyboard select **Ctrl L** to *unblank* all invisible **Surfaces**.
- 11 Switch off both the level *General* and the Set *Pkt1-Set*.
- 12 Switch on **AllPkts-Set**.



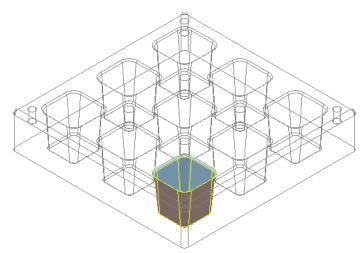
The **surfaces** currently acquired to **AllPkts-Set** are displayed, including those that exist on the visually 'switched off' **Set**.

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



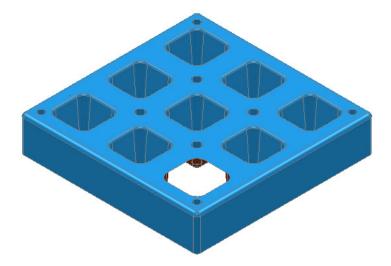
The **surfaces** acquired to **Pkt1-Set** are displayed. They are also displayed when the **AllPkts-Set** is switched on and **Pkt1-Set** is switched off. This illustrates that model entities can be acquired to more than one **Set** at the same time.

- 1 Switch off both **Sets** and switch on the **Level General**.
- 2 In the **PowerMILL** *explorer* right-click on **Pkt1-Set** and from the local menu click **Select Surfaces**.



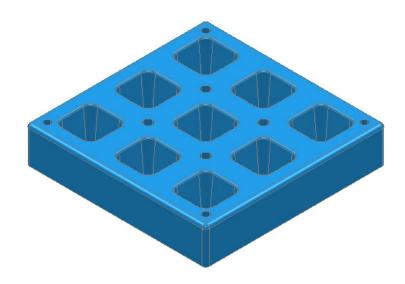
3 Create a new Level and rename it as Pkt1-Level.

- 4 Switch of the light bulb symbol next to the new Level Pkt1-Level.
- 5 In the **PowerMILL** *explorer*, right-click *Pkt1-Level* and select **Acquire Selected Model Geometry**.



The **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 co-exist on more than one **Level** at any one time.

6 Switch on the **level** *Pkt1-Lev* and the bottom left pocket surfaces are displayed again.





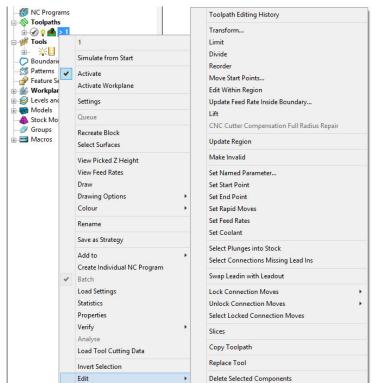
When using a combination of **Levels** and **Sets** it is good practice to keep the **Sets** visually switched off. If you want to select items from a particular **Set**, then, right-click on it and use one of the **Select** options from the context menu.



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.

# 8. Editing Toolpaths

There is a large selection of **Toolpath** editing options that can be applied retrospectively.



Many of the **Toolpath** editing options are accessed from the local menus (as shown above). Most of these edits will no longer apply if a **Toolpath** is subsequently recalculated.

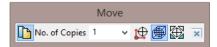
### **Edit > Transform**

The **Transform** option allows you to **Move**, **Rotate**, and **Mirror** toolpaths relative to the **Transform** (Global datum) or an active **workplane**.

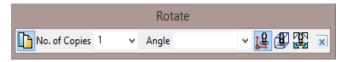
To access the options right-click on a **toolpath** and select **Edit** > **Transform**.



 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.

			6	
		1		×
	2	1	/	
L	-0			

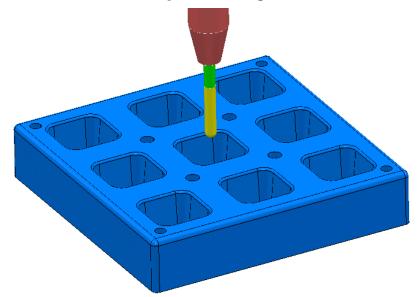
It is important to note that when a toolpath is mirrored, the cutting direction effectively becomes reversed.

Mirror		
┣┇╬╔╔╔╝		

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.

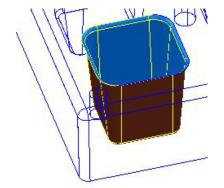
A **toolpath** will be created in the lower left cavity that is limited to be inside a **Selected Surface** Boundary.

3 Save Project As:

.....\COURSEWORK\PowerMILL-Projects\Transform

- 4 Create a **Block** using **Defined by > Box** and **Type > Model**.
- 5 **Calculate** the **Rapid Move Heights** using the default settings.

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**.

🗈 📿 Boundari	es			
Patterns	Boundaries		1	
workpla ⊕ 🥩 Levels a	Curve Editor Toolbar			
⊕ 👘 Models & Stock M	Display private Boundaries			
Groups	Create Boundary	•	Block	
🗄 📟 Macros	Paste as new Boundary		Rest	
	Boundary Names	•	Selected Surfac	
	Folder Names	•	Shallow	63

8 In the Selected Surface Boundary dialog, tick Roll Over and enter a Tolerance 0.02 and *un-tick*, Allow boundary to be private.

Name 1 Top Limit Bounda Roll Over Control Outside Outside	ary Limiting Boundary
Roll Over	
	ollision Checking older Clearance 0.0 hank Clearance 0.0
Block	Limit 👩 🗸
rivate	
alculation	8
	al Thickness

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

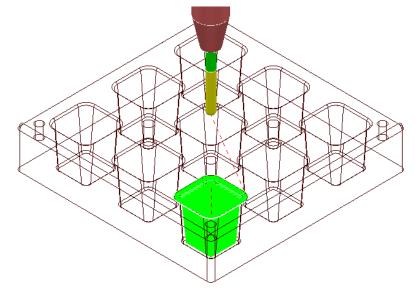
A **Selected Surfaces Boundary** defines the limit where the *active* **Tool** and associated parameters would fully machine the selected surfaces.

10 Open the **Optimised Constant Z Finishing** strategy and enter the values and settings exactly as shown in the 2 dialog pages below:

<del>3</del> 4	Optimis	ed Constant Z Finishing	9	?	×
£2 II	Toolpath name	BN10-Pocket			
Workplane     Block     Tool     Tool     Machine tool     With Stock engager     Optimised coo     Automatic     Automatic     Automatic     Rapid move h     Leads and link     Start point     Start p	istant Z finishing verification ion control eights s eds ecription	Optimised constant	Cut direction Climb	~	
			Shallow stepover 5.0		

Optim	ised Constant Z Finishing 🛛 📍 🗙
Toolpath nam	e BN10-Pocket
Workplane Block U Tool Block	Limit Boundary Boundary       Image: Comparison of the second secon
Point distribution	Trimming Keep inside
⊘ Machine axis control Rapid move heights ⊕ U Leads and links Start point	Block

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 then 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.



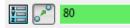
3 In the **PowerMILL** *explorer*, right-click on the **toolpath BN10-Pocket** and select **Edit > Transform** to display the **Toolpath Transform** toolbar.



- 4 Switch off the 'append toolpaths' option
- 5 On the Toolpath Transform toolbar, click Move  ${}^{ullet}$
- 6 On the Move toolbar, click Keep original with No. of Copies as 2.



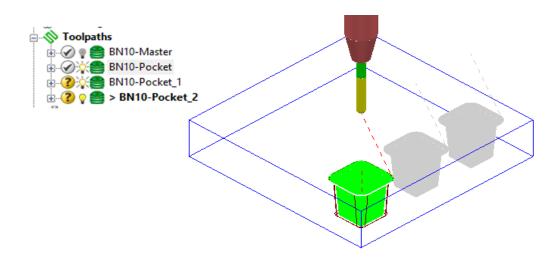
7 In the data entry box located below the graphics area, enter the *relative distance* along X as **80**, and press *Enter* on your keyboard.



X Y Z values (separated by a space) are entered into this box as required.

8 In the **Transform** toolbar, click  $\checkmark$  to close and accept the changes.

Two new copies of the original **toolpath** named **BN10-Pocket\_1** and **BN10-Pocket\_2** are created and transformed along **X** by a relative distance of **80**.

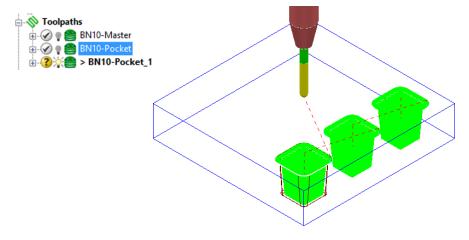


Note: It is possible to directly append all of tool tracks into a new, single toolpath **BN10-Pocket\_1**.



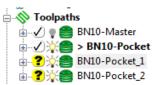
This is achieved by selecting the **Automatically append transformed toolpaths** 

option 🖼 before applying the transform values.



## Gouge checking a toolpath

A ? next to the toolpath indicates that it has not been gouge checked directly to the model.

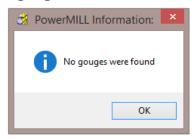


- 1 Activate the toolpath BN10-Pocket\_1.
- 2 Right-click on the toolpath BN10-Pocket\_1 and from the local menu select Verify > Toolpath.
- 3 In **Check** select the **Gouges** option.

🛃 Toolpath Verification ? 🗙
Check Gouges V
Check Against
Models V
Scope All 🗸
✓ Split Toolpath
Output Safe Moves 🗸
Output Unsafe Moves 🗹
Reorder Toolpaths
Split Moves
Overlap 0.0
Minimum Length 0.0
Head Clearance 600.0
Verification Thickness

4 Apply the Toolpath Verification dialog.

A **PowerMILL information** box is displayed to inform of the gouge checking status. In this instance, *No gouges were found*.



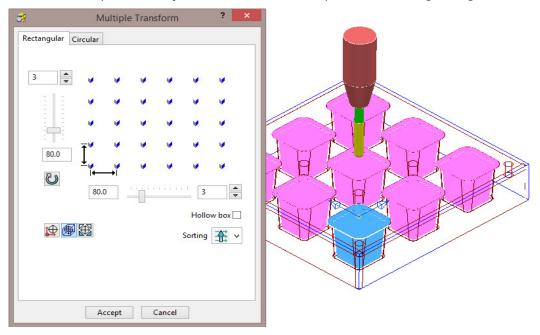
In the **PowerMILL** *explorer*, the style of the icon also updates indicating that the **toolpath** is gouge free.



5 Repeat the Verify > Toolpath process on BN10-Pocket\_2

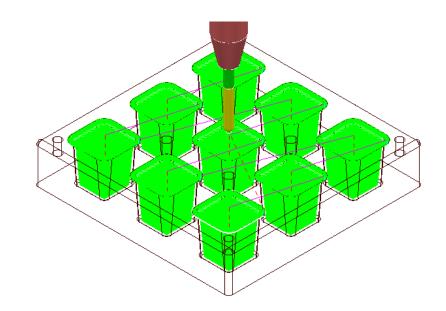
### **Multiple Transform**

- 6 Delete both of the toolpaths BN10-Pocket\_1 and BN10-Pocket\_2.
- 7 Right mouse click on toolpath BN10-Pocket and from the local menu select Edit > Transform.
- 8 Select the option Automatically append transformed toolpaths.
- 9 Select the option **Multiple Transform** to open the following dialog.



10 Enter **3** for number of copies and a pitch of **80** for both in **X** and **Y** before selecting **Accept**.

11 In the **Transform** toolbar, click  $\checkmark$  to close and accept the changes.





A new **toolpath** (**BN10-Pocket\_1**) is created that machines all 9 pockets.



- 12 Activate and Verify the new toolpath BN10-Pocket\_1 to check for gouges.
- 13 Select File Save Project to update:

.....\COURSEWORK\PowerMILL-Projects\Transform

## Edit > Limit...

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



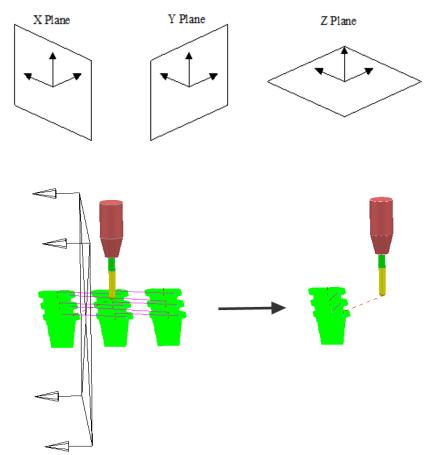
These types of retrospective limiting will be undone if the **Strategy** is recalculated.

For more permanent toolpath limiting use a **Boundary** as part of the original **Strategy** calculation.

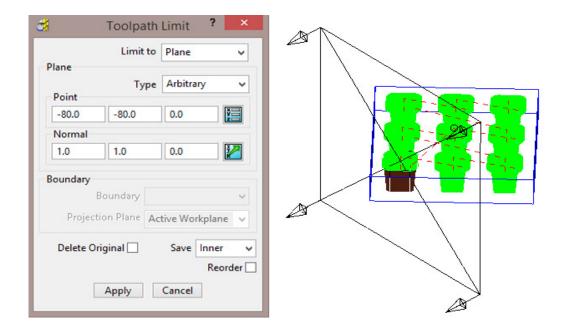
A typical application for retrospective **Toolpath** limiting would be the (one off) remachining of a local area that has been welded.

## 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.



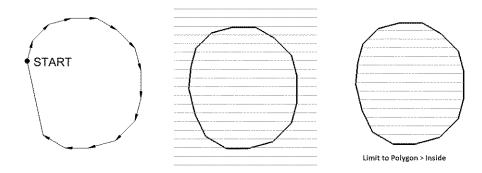
## Edit > Limit > to a Polygon

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.

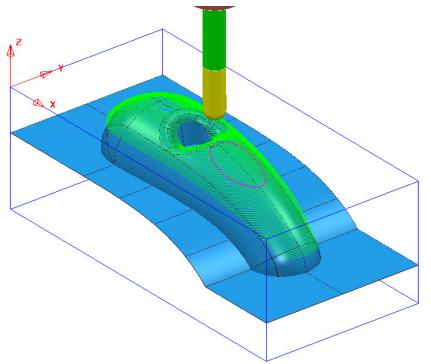


Before making a polygon ensure that the snap filter is set to pick Anywhere.



## Edit > Limit > to a Boundary

This provides the option to retrospectively limit a **toolpath** to an existing **Boundary**. This is okay if a temporary, quick solution is acceptable, otherwise the **Boundary** should ultimately be included within the strategy calculation for a permanent solution. A typical application for this would be the 'one off' machining of an area that has been welded due to a previous gouging error.



â	Toolpa	ath Limit	? ×
Dises	Limit	to Boundar	у ~
Plane	Ту	pe Arbitraŋ	· ·
-80.0	-80.0	0.0	X Y Z
Normal			
1.0	1.0	0.0	HIM
Boundary			
	Boundary		~
Projec	ction Plane	Active Work	plane 🗸
Delete O	Original 🗸	Save 1	nner 🗸
			Reorder
	Apply	Cancel	



U

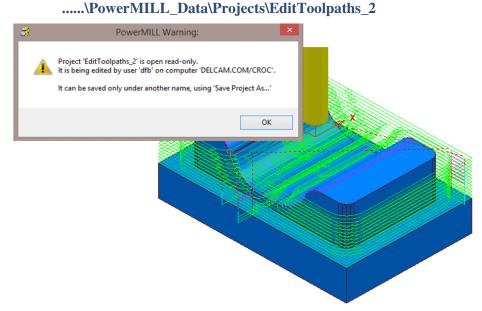
If the **Delete Original** option is selected, it causes the deletion of the original toolpath once the new toolpath is created.

If the toolpath is re-Calculated any retrospective limiting will not apply to the new or edited strategy.

## 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.

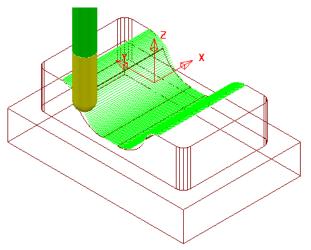
- 1 Select File Delete All and Tools Reset Forms.
- 2 From File > Open Project select the read-only Project:



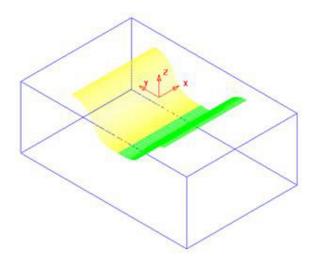
**3** Select **File > Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\EditToolpaths-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.



- 5 Left-click to select all tool tracks on **bn12-finish-a1** beyond the centre of the form (Yellow in the illustration above).
- 6 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.

	-		Reorder Tool	path Segments	3	×
	X	#	Start point	End point	Length	^
	$\square$	54	30.00, -6.25, -19.93	-30.00, -6.25, -19.95	60.00	
		55	30.00, -5.75, -19.96	-30.00, -5.75, -19.99	60.00	
		56	30.00, -5.25, -20.00	-30.00, -5.25, -20.02	60.00	
	$\overline{}$	57	30.00, -4.75, -20.03	-30.00, -4.75, -20.05	60.00	
Reverse order	$\mathbf{\Sigma}$	58	30.00, -4.25, -20.03	-30.00, -4.25, -20.05	60.00	
Reverse order		59	30.00, -3.75, -20.03	-30.00, -3.75, -20.05	60.00	•
Devenue dimention		60	30.00, -3.25, -20.03	-30.00, -3.25, -20.05	60.00	
Reverse direction ——	10	61	30.00, -2.75, -20.03	-30.00, -2.75, -20.05	60.00	
		62	30.00, -2.25, -20.03	-30.00, -2.25, -20.05	60.00	
Alternate directions	M	63	30.00, -1.75, -20.03	-30.00, -1.75, -20.05	60.00	.83
		64	30.00, -1.25, -20.03	-30.00, -1.25, -20.05	60.00	•
		65	30.00, -0.75, -20.03	-30.00, -0.75, -20.05	60.00	
		66	30.00, -0.25, -20.03	-30.00, -0.25, -20.05	60.00	
		67	30.00, 0.25, -20.03	-30.00, 0.25, -20.05	60.00	
		68	30.00, 0.75, -20.03	-30.00, 0.75, -20.05	60.00	
		69	30.00, 1.25, -20.03	-30.00, 1.25, -20.05	60.00	
		70	30.00, 1.75, -20.03	-30.00, 1.75, -20.05	60.00	
		71	30.00, 2.25, -20.03	-30.00, 2.25, -20.05	60.00	
		72	30.00, 2.75, -20.03	-30.00, 2.75, -20.05	60.00	
		73	30.00, 3.25, -20.03	-30.00, 3.25, -20.05	60.00	4
		74	30.00, 3.75, -20.03	-30.00, 3.75, -20.05	60.00	<b>v</b>
		<				>

The selected segments are shown highlighted (blue) 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 of the toolpath will be edited.

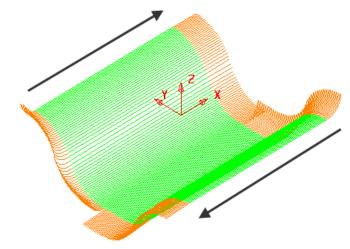
The 2 buttons to the lower left of the dialog are Automatic Reorder

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 unnecessary air moves.

- 7 Click on the **Reverse Order** button to reverse the sequence of the selected tool tracks.
- 8 Click on the **Reverse Direction** button it to reverse the direction of each selected tool track.
- 9 Open the Leads and Links dialog and Apply:-

Vertical arc:- Lead in with Angle 90 and Radius 6 and

Lead out with Extended move with Distance 10



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* has been applied to the above toolpath to identify the directional differences.

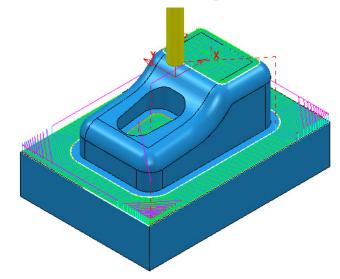
**10** Save Project As:

.....\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 Project:-

.....\PowerMILL\_Data\Projects\EditToolpaths\_3

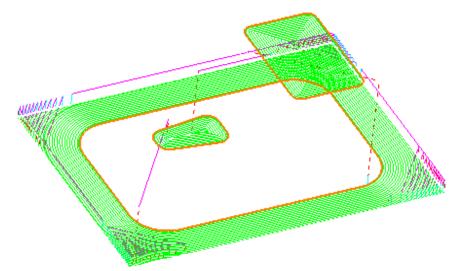


#### **3** Save Project As:

.....\COURSEWORK\PowerMILL-Projects\EditToolpaths-EX2

The **Project** includes a **3D Offset** finishing toolpath **D10TR1-FIN1** controlled by both **Boundary** and **Pattern** segments.

4 **Simulate** the **Toolpath** using a slow setting.



Observe the current direction and ordering of the strategy considering potential improvements.

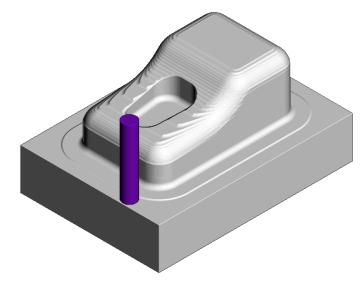
For example, the lower area machining would benefit from climb milling inwards towards the component form and the pocket machining could start central and climb mill outwards towards the sidewall.

5 From the Main menu, select View > Toolbar > Toolpath to display the Toolpath Toolbar.

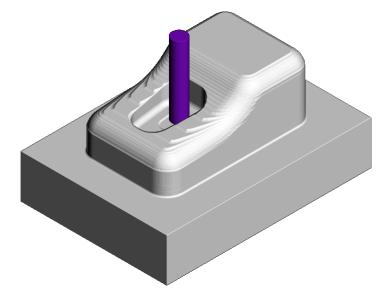


6 Select all tool tracks in the lower area and select **Reverse Order** A. This part of the strategy should now climb mill inwards, towards the main component

dialog. If not 'climb milling', then apply **Reverse Direction M**.



7 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.

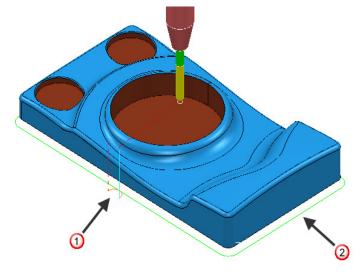
- 1 Select File Delete All and Tools Reset forms.
- 2 Select File Open Project:

.....\PowerMILL\_Data\Projects\limiting-example

3 Save Project As:

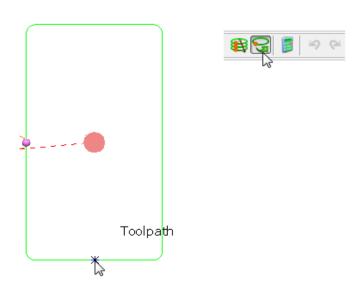
.....\COURSEWORK\PowerMILL-Projects\MoveStartPoint-EX1

4 Select an **ISO1** view.



The toolpath start point 1 needs to be moved to position 2.

- 5 Select a **View from top (Z)**.
- 6 Right-click toolpath 1 and select Edit > Move Start Points and from the toolbar select Move start point by clicking a toolpath segment .



7 Click a point along the toolpath segment and then  $\checkmark$  to accept the change.



This has moved the toolpath start point.

8 Save the **Project** to update:-

```
.....\COURSEWORK\PowerMILL-Projects\MoveStartPoint-EX1
```

## Moving multiple points

It is possible to apply the **Move start points** option on the toolpath to multiple positions.

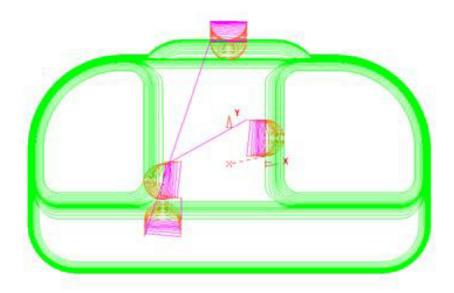
- 1 Select File Delete All and Tools Reset Forms.
- 2 Open the **Project**:

.....\PowerMILL\_Data\Projects\PunchForm10-Start

3 Save Project As:

.....\COURSEWORK\PowerMILL\_Projects\PunchForm10

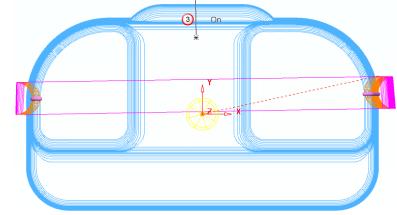
4 Select a **View from top (Z)**.



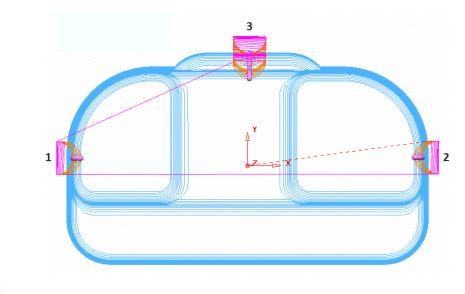
- 5 Right-click the *active* toolpath and from the local menu select Edit > Move Start Points (The *Move Start Points* toolbar is displayed).
- 6 If not already selected, on the *toolbar*, click the **Move Start Points by** drawing a line button.



- 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.
     (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 v on the **Move Start Point** toolbar to accept the changes.
- 9 Save the Project to update:-

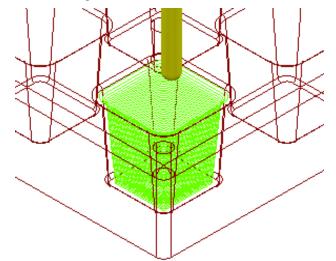
.....\COURSEWORK\PowerMILL-Projects\ PunchForm10

## **Feeds and Speeds**

You can modify Feed Rates using the **Feeds and Speeds** dialog to retrospectively **Apply** them to the **Active toolpath**.

-	Feeds an	nd Speeds ? ×
<u>11</u>	Toolpath Properties Toolpath: D12t1-fin1 Type Finishing v Operation General v	Tool Properties Tool: D12T1 Diameter: 12.0 mm Number of Flutes: 1 Overhang: 60.0 mm
	Tool/Material Properties         Stock Material:         Surface Speed (Recommended: 67.86 m/min)	Cutting Conditions Spindle Speed (Recommended: 1800 rpm) 1800.0 rpm Cutting Feed Rate (Recommended: 400 mm/min) 400.0 mm/min Plunging Feed Rate (Recommended: 40 mm/min) 200.0 mm/min Skim Feed Rate 3000.0 mm/min
	Depth of Cut Surface Slope 0.0 mm 0.0 Overhang Compensation Reset Apply	Coolant       Image: Coolant       Accept       Cancel

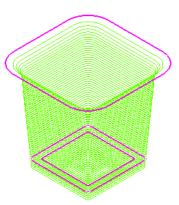
- 1 Select File Delete All and Tools Reset Forms.
- 2 Select File > Open Project:
  - .....\PowerMILL\_Data\Projects\ExtraFeedrates-Start



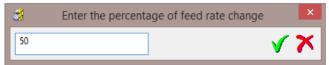
3 Select File > Save Project As:

.....\COURSEWORK\PowerMILL-Projects\ExtraFeedrates-Example

4 Undraw the Model, Block, Workplane, and Tool to display the Toolpath and 3 Boundaries.

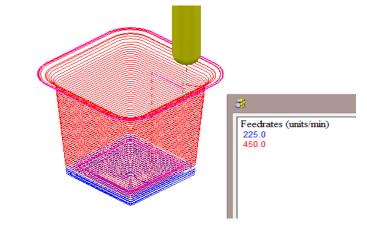


- 5 Activate both the Toolpath and Boundary 2 (The middle one of the 3).
- 6 Right-click the **Toolpath** and select **Edit** > **Update Feed Rate Inside Boundary**.
- 7 Enter value as **50** and click **V** to apply the percentage difference to the **Feed Rate** within the **Active Boundary**.



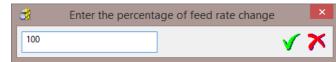
The **Toolpath** is split into two different coloured areas that are allocated a separate **Feed Rate** as a percentage of the original. The colour coded **Feedrates** are displayed in an **Information** dialog that appears automatically.

- 8 The Feedrates Information dialog can also be opened by:
  - a Selecting View Feedrates from the local toolpath (BN10-A1) menu.
  - **b** By double-clicking on a *Light bulb* adjacent to a toolpath (**BN10-A1**) in the **PowerMILL explorer**.

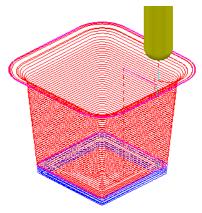


The original Cutting Feedrate assigned to the toolpath was **450 mm/min**.

- 9 Activate both the Toolpath and Boundary 3.
- Right-click the Toolpath and from the local menu select Edit > Update Feed Rate Inside Boundary.
- 11 Enter the value **100** and click **V** 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.



12 The *Feedrates* Information dialog can also be opened by double-clicking on a *Light bulb* adjacent to a **Toolpath** in the **PowerMILL explorer**.

In this case, the nominal **Feed Rate** is **450 mm/min**, and the modified section (near the intersection of the Base and Sidewall of the pocket) 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.

## Safe Z and Start Z Positions

The tool **Rapid Move Heights** can be modified in the dialog and applied retrospectively to the **Active toolpath**.

	🥳 Rapid Move Heights ?
	Geometry
	Safe area Plane 🗸 🗸
	Workplane
	Normal
=0	0.0 0.0 1.0
	Rapid height 10.0
	Plunge height 5.0
	Calculate dimensions
	Rapid clearance 5.0
	Plunge clearance 5.0
	Calculate
	Drawing options
	Draw rapid surface Opacity 20
	Draw plunge surface Opacity 20
	opacity 20
	Apply Accept Cancel

## **Editing Tool Start and End Point Positions**

The tool **Start Point/End Point** can be modified in the dialog and applied retrospectively to the **Active toolpath**.

	Start and End Point ?
	Start Point End Point
<mark>∢∄</mark> ——→	Method Use First Point Safe Override Tool Axis Approach Along Tool Axis Approach Distance 5.0
	Coordinates -70.701416 -98.404835 10.0
	Tool Axis           0.0         0.0
	Apply Start Point
	Apply Accept Cancel

# 9. Tool Holder Collision Checking

## **Collision Checking**

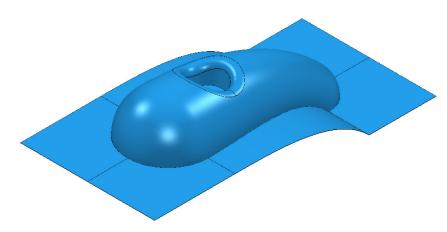
Collision Checking can either be applied retrospectively or directly during toolpath creation, as long as a tool **Shank** and **Holder** are defined with the **Active Tool**.

## **Retrospective Collision Checking (Verify -Toolpath)**

If the **toolpath** menu option, **Verify > Toolpath -** (*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-Short with a length of 10 and tool number of 1.

<b>3</b>	Ball nosed tool ? 🗙
Tip	Shank Holder Holder profile Cutting data Description
Geor	Name BN5 Coll assembly
	Length 10.0 5.0 Diameter Tool status Valid Tool number 1 Number of flutes 1
	Close

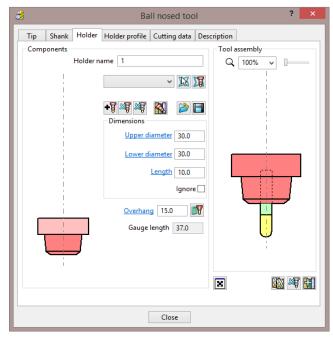
- 6 Select the **Shank** tab.
- 7 Select Add a **shank** component **+**.
- 8 Fill in the dialog exactly as shown in the image below.

Ball nosed tool	?	×		
Tip Shank Holder Holder profile Cutting data Description				
Components Tool assembly				
✓ XX XX Q 100% ✓		-		
+₹ ¥₹ ≥ ⊡				
Upper diameter 5.0				
Lower diameter 5.0				
Length 20.0				
Cutting length 10.0				
Shank length 20.0				
	r			
× 🕷 🤻 👪				
Close				

- 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 following image.

±	Ball nosed tool	? ×
Tip Shank Holder	Holder profile Cutting data Descri	ption
Components		Tool assembly
Holder r	ame 1	Q 100% ¥ 🛛 🗕
	- 🕅 📜	
	+ 🗑 💐 💐 🔯 📄 📄	
	Upper diameter 20.0	
	Lower diameter 15.0	
į.	Length 2.0	
1	Ignore 🗌	
	Overhang 15.0	
	Gauge length 30.0	1
i i	Gauge length 30.0	$\bigcirc$
		1
î.		
	[	x 🕅 🕅 🕅
	Close	

- 12 Select Add holder component 🖽.
- 13 Enter an Upper Diameter of 22, Lower Diameter of 20, and Length of 10.
- 14 Select Add holder component 📆.
- 15 Enter an Upper Diameter of 30, Lower Diameter of 30, 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 From the main toolbar, select Toolpath Strategies and from Finishing open a Raster Finishing strategy.

Pre	sview
	v

- 18 Enter Toolpath name as BN5-RasterFinish, Angle 45, and Stepover 1.
- 19 Select Start Corner Lower Left.
- 20 Select Style Two way.
- 21 In the Leads and Links 22, set both Lead in and Lead out with 1st choice as Surface normal arc with Angle 90 and Radius 3.
- 22 In the *Leads and links* Links page, enter a Short/Long Threshold of 4, set Short to Circular Arc, Long to Skim, and Default to Incremental.
- 23 In Lead in and Lead out pages, untick the option Add leads to short links.

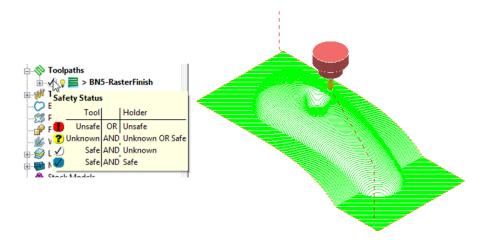
<del>3</del>	1	Raster Finishing	? ×
<ul> <li>3</li> <li>4</li> <li>5</li> <li>5</li> <li>6</li> <li>7</li> <li>7</li></ul>	Toolpath name	BN5-RasterFinish	
Workplane		Raster finishing	
·····Ū Tool ·····ಔ Machine tool ·····ಔ Limit		✓ Fixed direction	Angle 45.0
Stock engageme Raster finishing		Start corner Perpendicular pass	Lower left 🗸 🗸
Point distributio			Perpendicular pass
Tool axis			Shallow angle 30.0
→ Ø Machine axis co → ③ Rapid move heig			Optimise parallel pass
		Ordering Style	Two way 🗸
← 🔊 End point ⊕ - 🎢 Feeds and speed	is		Arc radius 0.0
History Notes and Description		Tolerance 0.02	
		Thickness	<u></u>
		Stepover	0.102084
		Preview	Draw
		Calculate Queue OK	Cancel

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

25 In the **PowerMILL** *explorer* hold the cursor over the small box to the left of the toolpath lightbulb symbol.



26 The Safety Status window will appear.



The new **toolpath** is *gouge free* as far as the **Tip** (cutting part) of the tool is concerned, but up to this stage no *collision checking* has been applied in relation to the **Shank** and **Holder**.

27 From the Main toolbar select Toolpath Verification 🕅

🥳 Toolpath Verification ? 🗙				
Check Collisions 🗸				
Check against				
Models V				
Scope All v				
Scope An V				
Output safe moves				
Output unsafe moves 🗸				
Reorder toolpaths				
✓ Split moves				
Overlap 0.0				
Minimum length 0.0				
Head clearance 600.0				
Verification thickness				
Collision options				
Replace tool				
Holder clearance 2.0				
Shank clearance 3.0				
Calculate collision depth				
✓ Adjust tool				
Adjust overhang only				
Always create new tool				
Draw unsafe moves 🗸				
Apply Accept				

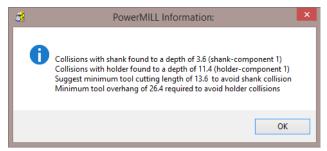
The two available, independent **Check** options are **Collisions** and **Gouges**.

**Shank Clearance** and **Holder Clearance**, allow the user to specify a value for an additional, safe thickness around the tool **Shank** and **Holder** elements.

28 In the Check box select Collisions.

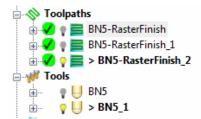
29 Input values in the Shank Clearance 2 and Holder Clearance 3.

30 Click Apply.



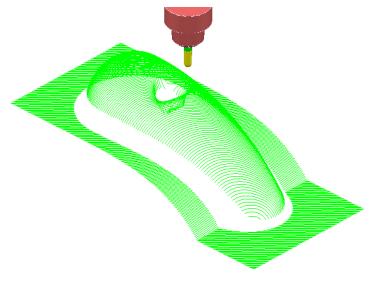
The above **PowerMILL Information** box appears with information on the status of the original tool and required dimensional changes. The default settings will result in a modified **tool** (**BN5\_1**) being created along with 2 new **toolpaths** (The collision and non-collision parts of the original). 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.

- 31 Click OK.
- 32 Click Accept on the Toolpath Verification dialog.



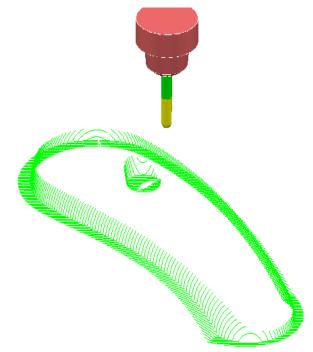
**PowerMILL** has created two new toolpaths, **BN5-Raster Finish\_1** and **BN5-RasterFinish\_2**. A new tool named **BN5\_1** has also been created in the **Tools** area of the **PowerMILL** *explorer*. This new tool has been created with the new valid adjustments made i.e. **Overhang 26.4** 

33 Activate the toolpath RasterFinish\_1 in the Explorer.



This **toolpath** contains the 'collision free' segments of **RasterFinish**, along with the original tool **BN5**.

#### 34 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.

**35 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**.

**36** Select **File > Save Project As**:

.....\COURSEWORK\PowerMILL-Projects\ToolCollisionExample

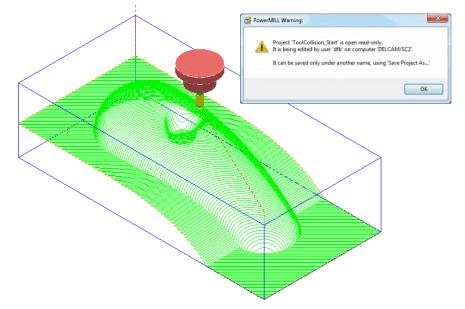
## **Direct Collision Checking (Automatic Verification)**

**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 Select an **ISO 2** view.
- **3 Open** the **Project**:

#### .....\PowerMILL\_Data\Projects\ToolCollision\_Start



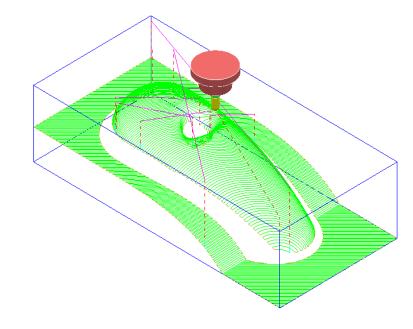
### 4 Save Project As:

#### .....\COURSEWORK\PowerMILL-Projects\ToolCollision-Auto

- 5 Right-click on the **RasterFinish** toolpath in the **PowerMILL** explorer.
- 6 From the local menu, select Activate followed by Settings and in the Raster Finishing dialog select the Copy toolpath icon
- 7 Select the **Automatic Verification** page <sup>1</sup> to open the following dialog:

<b>3</b>	Raster Finishing ? ×
Toolpath name	RasterFinish_1
Workplane	Automatic verification
	Head clearance 600.0
→ Stock engagement	Automatic collision checking     Collision checking against Model
	Holder clearance 3.0 Shank clearance 2.0
	Machine maximum stock

- 8 Tick the Automatic collision checking box, then input the, Holder Clearance as 3 and Shank Clearance as 2.
- 9 Calculate the Raster Finishing strategy to directly create a collision free toolpath, RasterFinish\_1.





This method does not produce any information for the user or automatically create a new collision free tool (**BN5\_1**).

## 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.

- 10 Right-click on the **toolpath** *RasterFinish\_1* in the **PowerMILL** *explorer*.
- 11 From the local menu, select Activate followed by Settings and in the Raster

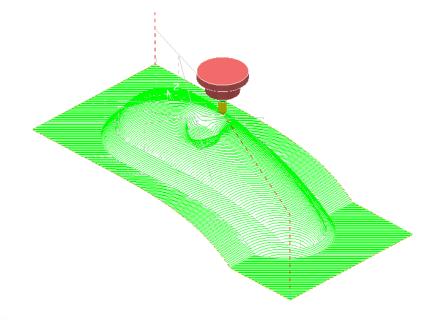
Finishing dialog select the Copy toolpath icon 💹.

12 With the Raster Finishing dialog open, select the Automatic Verification

page 🛅 to access the following dialog:

	Raster Finishing	?	×
Toolpath nam	e RasterFinish_1_1		
Workplane	Automatic verification		
U Tool Ⅲ Machine tool ® Limit Ƴ Stock engagement	Head clearance 600.0	)	
■ Raster finishing	Collision checking against Model		
Automatic verification	Holder clearance 3.0 Shank clearance 2.0		
Machine axis control	Machine maximum sto	ck 🗸	2

- **13** Tick Automatic collision checking, input Shank Clearance 2.0, Holder Clearance 3.0, and tick Machine maximum stock.
- 14 Calculate the Raster Finishing dialog to directly create a continuous, collision free toolpath, RasterFinish\_1\_1.





An additional strategy with a compatible (collision free) tool is required to remove the remaining stock.

## **Collision Safe Boundary**

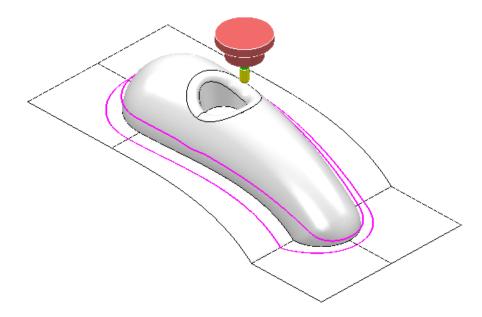
A toolpath for the remaining areas will require a new Dia 5 Ball Nosed tool with increased **Shank** length. The additional **Toolpath** will be calculated to limits set by a **Collision Safe Boundary** based on the current tool (**BN5**).

- 15 Ensure that the tool BN5 is Active.
- **16** In the **PowerMILL** *explorer*, right-click **Boundaries**, and from the local menu select **Create Boundary > Collision Safe**.

<b>3</b>	Collision Saf	e Boundary ? ×
1	🚳 Name	1
Tolerances	Ider Clearance     3.0       ank Clearance     2.0       Tolerance     0.02       Thickness     0.0       xial Thickness     0.0	<ul> <li>Limit Boundary</li> <li>Inside Limiting Boundary</li> <li>Outside </li> </ul>
Tool	Use Axial Thickness	Block
BN:	5 🗸	Limit 👩 🗸
Private Allow bounda	ry to be private	0
	tory on calculation	${}^{\odot}$

**17** Use the above settings and **Apply** the dialog.

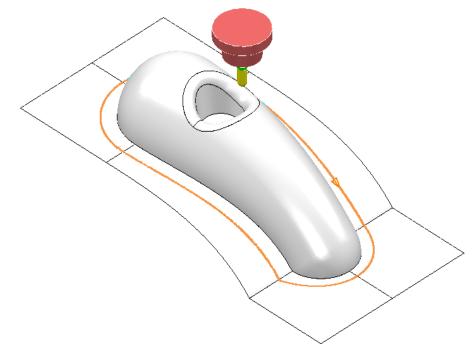
**18** Select and **Delete** all segments except the 2 shown below.



19 Create an empty **Pattern** and copy the outer **Boundary segment** to it.

With the outer **Boundary segment** selected, use **left mouse key** and **CTRL key** to drag and drop **Boundary** (1) into **Pattern** (1) from within the **PowerMILL explorer**.

20 Right mouse click on the new **Pattern** and from the local menu select **Instrument** to display the direction with an **arrow**.



21 Select the **Pattern** segment and if necessary, select **Edit** – **Reverse** selected to set it to comply with a climb milling direction.

ĽÒ.

22 In the PowerMILL explorer, right-click the Tool, BN5, and from the local menu select Edit > Copy Tool (to create BN5-1).



- 23 Right-click over the new tool BN5-1 and from the local menu, select Activate followed by Settings to open the Ball nosed tool dialog.
- 24 For the Tip change the length to 15.
- 25 Select the Holder tab and change the **Overhang** to **30**.
- 26 Close the form.
- 27 Activate the new 'long reach' Tool BN5\_1
- 28 Select Toolpath Strategies 🔊 on the main toolbar.
- **29** From the **Finishing** options select a **3D Offset Finishing** strategy and enter data as shown in the following form.

30	D Offset Finishing ? ×
Toolpath name	BN5LR-3Doffset
Workplane Block Machine tool Machine tool Machine tool Doffset finishing Automatic verification Doint distribution Doint distribution Machine axis control Rapid move heights Machine axis control Start point Leads and links Start point Feeds and speeds History Notes and Description User defined settings	3D offset finishing         Pattern         1       Image: Start on pattern         Start on pattern         Spiral         Smoothing         Maximum offsets         Maximum number of offsets
	Tolerance     Cut direction       0.02     Climb< ✓

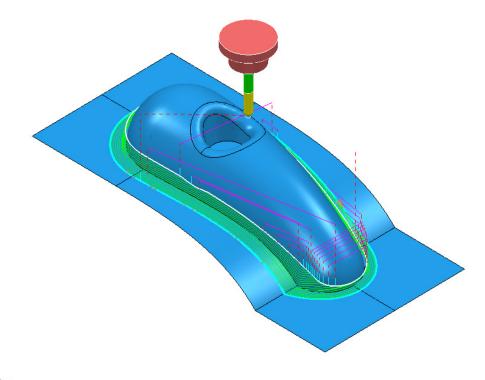
**30** Select the **Limit** page and select **Boundary 1** with **Trimming > Keep inside**.

3	D Offset Finishing	? ×
Toolpath name	BN5LR-3Doffset	
Workplane	Limit	
配 Machine tool 	Boundary	× 🚱 🖉
3D offset finishing     4utomatic verification     Yoint distribution		Trimming Keep inside v
Machine avis control		

31 Select the Automatic Verification page and tick Automatic collision checking, input 1 for both Shank and Holder clearances, un-tick Machine maximum stock.

Toolpath name BN5LR-3Doffset     Workplane   Block   Tool   Tool   Tool   Machine tool   Stock engagement   Stock engagement   Stock engagement   Tool axis   Automatic collision checking   Collision checking against   Model   Holder clearance   Holder clearance   Shank clearance   Collision checking against	3	D Offset Finishing ? ×
Block Tool Machine tool Stock engagement Automatic verification Point distribution Machine axis control Machine axis control	Toolpath name	BN5LR-3Doffset
<ul> <li>Key Limit</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Collision checking</li> <li>Collision checking against Model</li> <li>Holder clearance 3.0</li> <li>Holder clearance 2.0</li> <li>Machine axis control</li> </ul>		Automatic verification
Automatic verification     Point distribution     Tool axis     Machine axis control	🔞 Limit	
Machine axis control	Automatic verification	
D I leads and links		

#### 32 Calculate the toolpath and then Close the dialog.



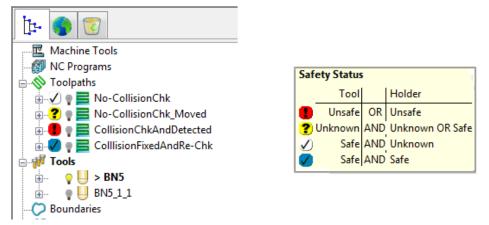
The new toolpath does not exhibit any fragmentation, showing that the active **Automatic Collision Checking** has not detected any problems.

33 Select Save Project to update:

.....\COURSEWORK\PowerMILL-Projects\ToolCollision-Auto

## **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).

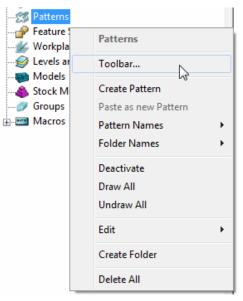


To display the 4 different levels of **Safety Status**, position the mouse over a **symbol**  $\checkmark$  ?  $\checkmark$  in the **PowerMILL** *explorer* and the information form will open (shown above right).

## **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). This will also make the other toolbar options active.

- 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).
- 🖌 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 epens 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:

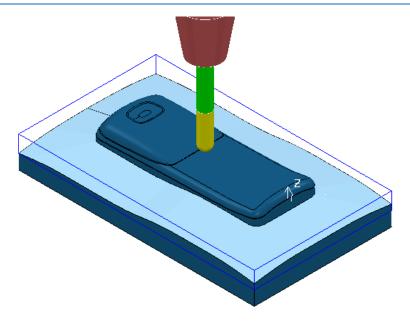
.....\PowerMILL\_Data\Models\phone.dgk



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

- 4 Define a dia 12mm Ball Nose tool named BN12.
- 5 Reset the Rapid Move Heights
- 6 Select an **ISO1** view to display the **model**.

3



- 7 Select the 'shutout' **surface** as shown above.
- 8 In the **PowerMILL** *explorer*, right-click on **Boundaries**, and from the local menu select the option **Selected Surface** and enter the values shown below.

Selected Surf	ace Boundary ?	
👔 🔓 💰 Name		
Top Roll Over Tolerances <u>Tolerance</u> 0.02 <u>Thickness</u> 0.0	Limit Boundary Inside Outside	
Axial Thickness 0.0 Use Axial Thickness Tool	Automatic Collision Checking Holder Clearance 0.0 Shank Clearance 0.0 Block	
Private Allow boundary to be private		
Apply edit history on calculation	Accept Cancel	

**9 Apply** the dialog to create 2 **Boundary** segments.

in jatterns 1	
	Patterns
🖅 🥪 Levels	Curve Editor
🗄 👰 Mode	Toolbar
- 🍓 Stock - 🥟 Group	Create Pattern

**10** In the **PowerMILL** *explorer*, right click on **Patterns** and from the local menu select **Toolbar**.

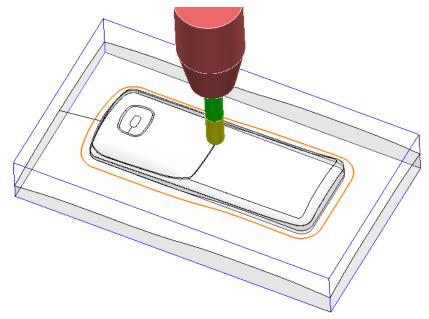
灯 1 🚽 🛛 🖞 🖹 🔪 🔲 📿 🔗 🖉 🖉 🖉 🖉 🖉 🖉	🗴 🗵
---------------------------------	-----

11 In the Patterns toolbar select, Create a new (empty) Pattern



- **12** In the graphics area, select the inner **Boundary** segment.
- 13 In the Patterns toolbar select the option to Insert Boundary into ActivePattern
- 14 Delete the **Boundary**.

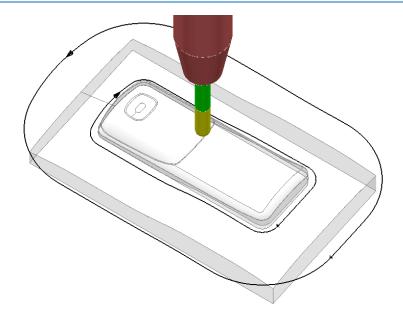
Once the selected **Boundary** segment is copied as a **Pattern** segment the **Boundary** is no longer of use.



15 Right-click on the **Pattern** segment and from the local menu, select **Curve Editor**.

\$\$ 🕺 🗶 🖈 🗐 🔦 🖊 🐔 🕐	】 ■ ッ e   > ≒ X S E S & S &   S ⊙ N +   × ⊗ ≝
----------------------	---

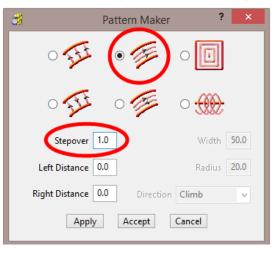
- **16** In the **toolbar** move the cursor over **Transformations** and from the attached vertical toolbar, select the **Offset** option .
  - Offset
- 17 Select **Keep Original** (1), the **Round Corners** option, and input **Distance 50** before clicking the **Enter** key.
- 18 In the **Curve Editor** toolbar, click **V** to accept.



- 19 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.
- 20 Select the <u>inner</u> Pattern segment and from the local menu select Edit > Reverse Selected (Both Pattern segments should now run in the same direction).

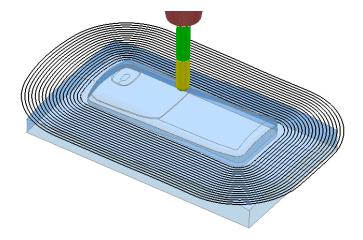
The **Automatic Pattern Generator** will be used to create a new **Pattern** with a series of segments merged between the 2 segments from the original **Pattern**. The resultant **toolpath** will follow the contour of the component form as well as creating corner free, tool tracks with a constant material removal rate.

- 21 From the **Pattern** toolbar, select **Create Pattern** to create a new **Pattern**.
- 22 Select Automatic Pattern Generation 🖻
- 23 With the new (empty) Pattern activated and the original Pattern displayed, select both visible segments.
- 24 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.

25 Click Apply, followed by Accept.



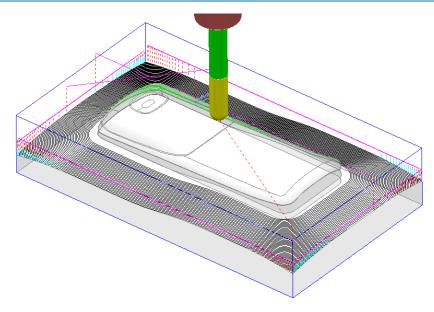
The new **Pattern** has been generated independently; between the two selected segments of the original, de-activated **Pattern** (the above illustration shows a larger 2mm Stepover for visual reasons).

The new **Pattern** will now be used in a **Pattern Finishing** strategy.

26 In the **Strategy Selector**, **Finishing** options, select a **Pattern Finishing** strategy.

₩ P	Pattern Finishing ?
Toolpath name	BN12-Fin-A1
	Pattern finishing
	Drive curve Use toolpath 2  V  K  K
Automatic verification     Point distribution     Tool axis	Lower limit Base position Automatic
Machine axis control Rapid move heights Leads and links Start point Freds and speeds History Notes and Description User defined settings	Axial offset 0.0 Gouge avoidance Gouge check
	Tolerance Ordering     0.02   Pattern     Thickness
	Maximum stepdown
	Preview Draw
	Calculate Queue OK Cancel

27 Enter the data into the above form exactly as shown above, click **Calculate**, and then **Close** the form.



The **Pattern Finishing** toolpath is limited to exist within the defined material **Block**.

## **Exercise**

- 28 Right mouse click on the new toolpath and from the local menu select Edit > Reorder.
- 29 Use both the *Reverse Order* and *Reverse Direction* editing options for the **Toolpath** to start from the outside, *Climb Milling* towards the centre of the model.
- **30** Save the Project.

## Patterns applied to 3D Offset machining

A **Pattern** can be used as a curve to be offset across the area of a **3D Offset Finishing** strategy.

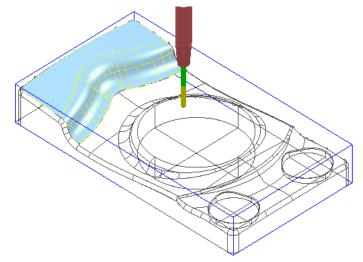
- 1 Select File > Delete All and Tools > Reset Forms
- 2 Import the model:

.....\PowerMILL\_Data\Models\speaker.dgk

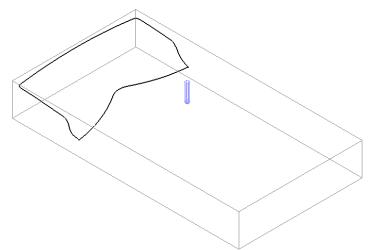
**3** Save Project As:

.....\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

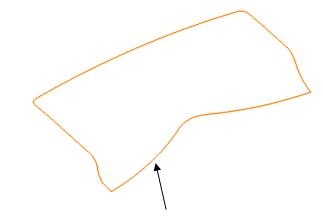




10 From the Pattern toolbar click Create Pattern X followed by Insert

```
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.* 

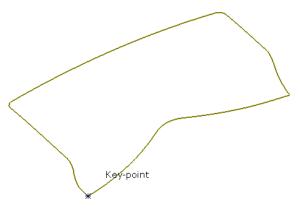
- **12** Select the **Pattern** segment using the left mouse key.
- **13** Right mouse click on the **Pattern** and from the local menu, select **Curve Editor** to open the following toolbar:



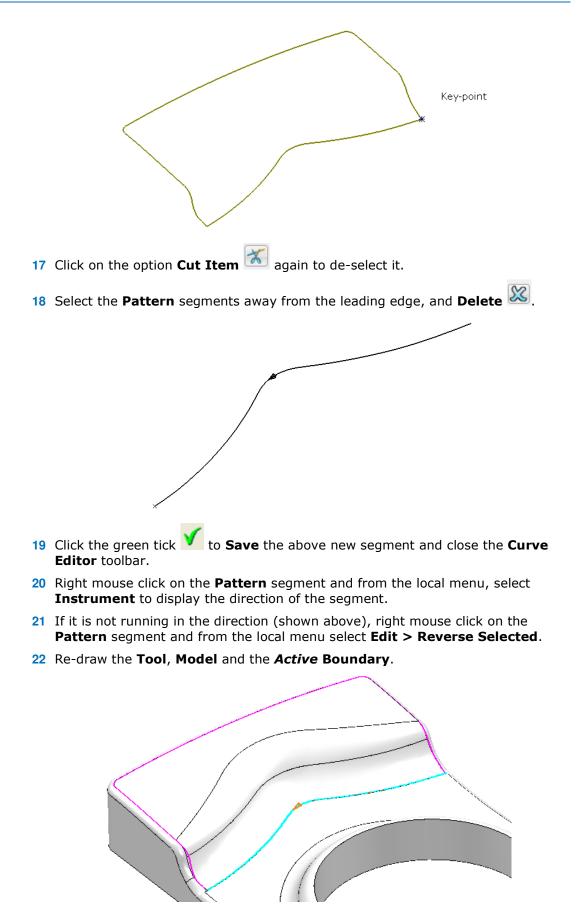
- 14 Select the option Cut Item.

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.



23 Select a **3D Offset Finishing** strategy and enter data into the dialog exactly as shown below.

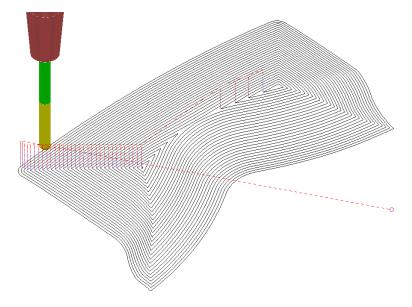
30	D Offset Finishing ? ×
Toolpath name	BN6-3Doff
Workplane Block Jool Kachine tool Kachine tool Kachine tool Stock engagement Stock engagement Stock engagement Automatic verification Automatic verification Machine axis control Machine axis control Machine axis control Kapid move heights Leads and links Start point Leads and links Kather and point Start point History Notes and Description Start defined settings	3D offset finishing         Pattern         1       Image: Start on pattern         Start on pattern         Spiral         Smoothing         Maximum offsets         Maximum number of offsets
	Tolerance     Cut direction       0.02     Climb< ✓       Thickness     ✓       ●.0     ●       Stepover     ●       ●     1.5

24 Click **Calculate** to create the **toolpath** and then **Close** the dialog.



Leads and Links are not 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.

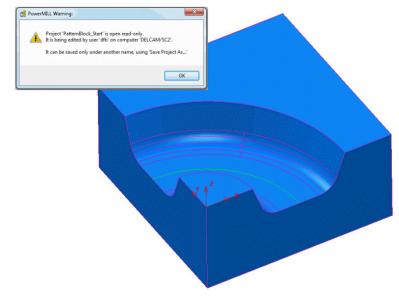
25 Save the Project.

## **Exercise - 3D Offset Finishing controlled by Pattern**

It is not compulsory for the **Pattern** for controlling the toolpath to be coincident with the limiting **Boundary**. In the following exercise a **3D Offset Finishing** will be controlled by a **Pattern** that runs centrally along the base of a curved recess:

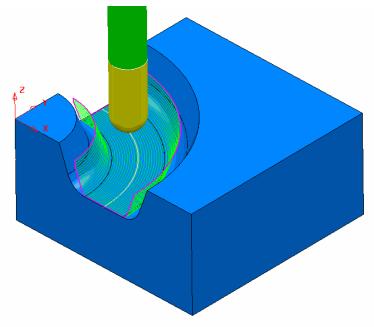
- 1 Select File > Delete All and Tools > Reset Forms
- 2 Open the Project:

#### .....\PowerMILL\_Data\Projects\PatternBlock\_Start



- 3 Click OK to close the *PowerMILL Warning* and Save Project As: .....\COURSEWORK\PowerMILL-Projects\PatternBlock\_ex1
- 4 **Calculate** the **Block** using the default settings.

- 5 **Calculate** the **Rapid Move Heights** to the default settings.
- 6 In the **Viewing toolbar** switch on display **Wireframe**
- 7 Create a **Pattern** using **Insert Model** on the *wireframe arc* (included with the **Model**) running centrally along the recessed form.
- 8 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.
- 9 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**.



**10** 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.

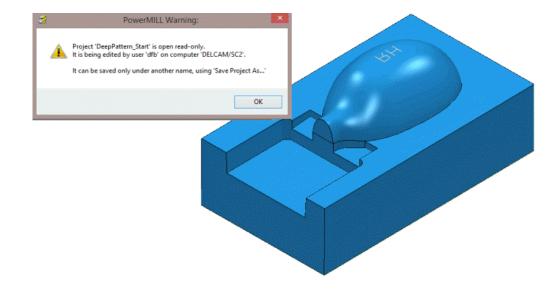
It is, however, possible to produce a **Pattern Finishing** strategy that machines deeper into the model if **Axial Offset** is used in place of a -ve **Thickness** value. This is achieved by first projecting the **Pattern** onto the surface (relative to the tool geometry). **Pattern Finishing** is then 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.



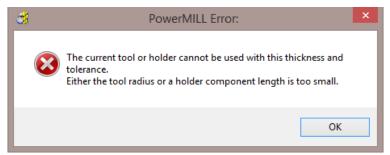
*This projects the* **Pattern** *down Z, while compensating for the tooling parameters.* 

- 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**.

Toolpath name BN5-Engrave	
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Pattern finishing</li> <li>Multiple cuts</li> <li>Multiple cuts</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>End point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>Wiser defined settings</li> <li>Pattern finishing</li> <li>Init</li> <li>Init<!--</th--><th></th></li></ul>	



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.



 Click OK to close the *PowerMILL Error* message and recycle the Pattern Finishing toolpath using the data as shown in the following image.

đ	Pattern Finishing ? 🗙
Toolpath name	BN5-Engrave
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Stock engagement</li> <li>Pattern finishing</li> <li>Multiple cuts</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Pattern finishing   Drive curve   Use toolpath   Image: Second

Base Position - Drive Curve Axial Offset -3.0 Gouge Check - un-ticked Thickness 0

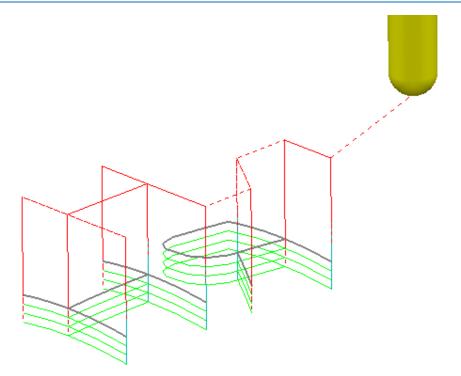
**11** Select the **Multiple cuts** page and input data exactly as shown below.

<del>3</del>	Pattern Finishing		? ×
Toolpath nam	BN5-Engrave		
	Multiple cuts		
U Tool Ⅲ Machine tool ↓ Limit	Mode	Offset down	~
Stock engagement	Order by	Region	~
Pattern finishing     Multiple cuts     Automatic verification	Maximum number of cuts		
Point distribution	✓ Upper limit		
Machine axis control Rapid move heights Leads and links	-1.0		
	Maximum stepdown		
Feeds and speeds			

#### 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.



The Reversed lettering, Pattern `RH' is now recessed into the Punch form to a depth of 3mm.

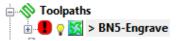


The **Toolpath** will initially be identified in the **PowerMILL explorer** with a **yellow warning**.



- 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**.

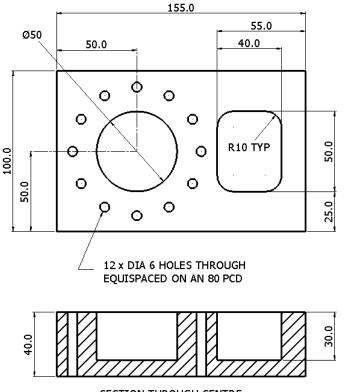
# **11. Curve Editor**

## 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.

# 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.



SECTION THROUGH CENTRE

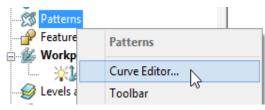
- 1 Select File Delete All and Tools Reset Forms.
- 2 In the **PowerMILL** explorer select Workplanes Create Workplane.

- 📈 Workplan	nes and a second s
🥩 Levels	Workplanes
- 👰 Model	Tomplanes
- 🍓 Stock	Create Workplane
- Ø Group	Get Workplanes From Model

3 The Workplane editor toolbar will become active as soon as a new workplane is created on the global datum.

🗋 Name 1 🔚 🖉 🎖 🎖 🎖 🖾 🖾 🖉 🏟 👫 📖 🛱	6i
----------------------------------	----

- 4 Select the green tick to exit the Workplane Editor toolbar.
- 5 Activate the Workplane.
- 6 Select a view down Z.
- 7 From the **PowerMILL** *explorer*, right mouse click on **Patterns** and from the local menu select **Curve Editor**.



8 From the **Curve Editor** select **Rectangle** option.



9 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.

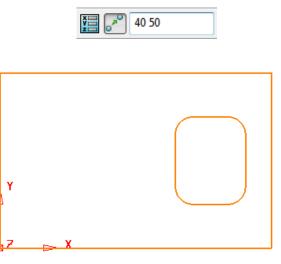
10 Input **155 100** in the **Command Input** box and press the return key to accept the X and Y dimensions of the rectangle.



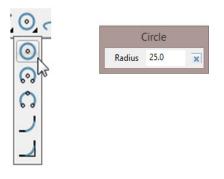
11 Select the corner Radius option and input 10.



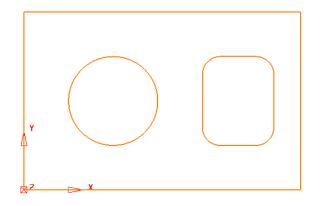
- 12 Create a new **Rectangle of lines** using **100 25** to define the start corner.
- 13 In the data input box, switch from **Absolute b** to **Relative c** input and enter **40 50** for the X and Y dimensions.



14 From the **Create a circle** options select the **full Circle** option.



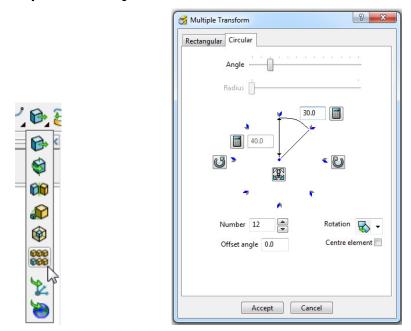
- 15 Input a Radius value of 25.
- **16** Input **50 50** in the **Command Input** box followed by the return key.



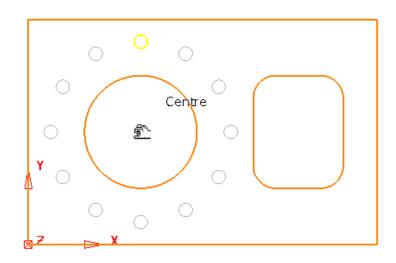
- 17 Create another **Circle** with a centre at **50 90** and a radius value of **3**.
- **18** Click on **10** to close the **Circle creation** options.
- **19** Select the Dia 6 **Circle**.

20 With the new Dia 6 Circle selected, open the Transforms options and select

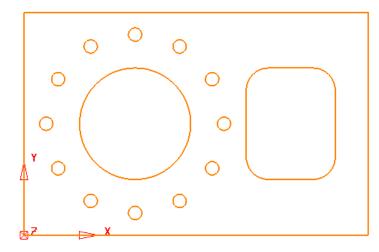
Create pattern of objects



21 On the dialog, select the **Circular** tab and enter **Angle 30**, **Number 12** and select **Reposition rotation axis**, and drag it to the centre of the large circle.



22 Select Accept on the Multiple Transform dialog.



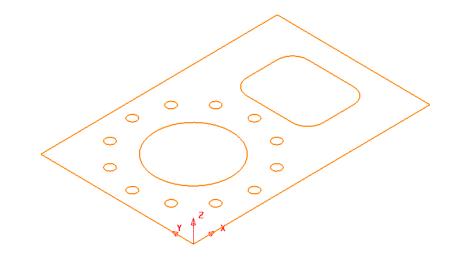
The geometry creation is now complete and is ready to be returned to the **PowerMILL Project** as the contents of the **Pattern**.

**23** 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.

24 Select to accept changes and export the wireframe back to **PowerMILL** to replace the original boundary.



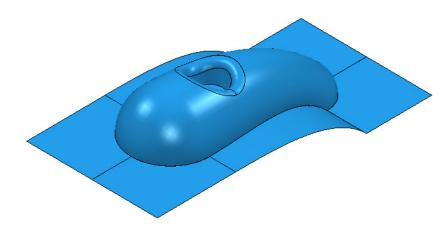
25 Select File > Save Project As:

.....\COURSEWORK\PowerMILL-Projects\2DModel

## **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. The easiest and most effective method is to make use 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 default **Start and End Point** values.
- 7 In the **PowerMILL** *explorer*, right-click **Boundaries** and from the local menu select **Create Boundary > Shallow**

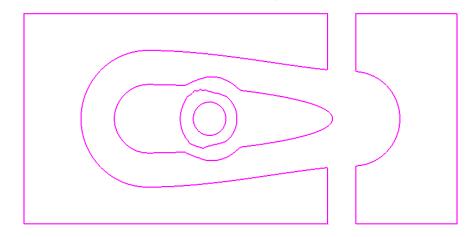
*	Shallov	v Boundary ? 🗙
1	🚨 Nam	1 1
Tolerances	Upper Angle 30.0 Lower Angle 0.0 Tolerance 0.02 Thickness 0.0 Axial Thickness 0.0	Limit Boundary  Inside Limiting Boundary  Outside
<u></u>	Use Axial Thickness	Automatic Collision Checking
Tool		Holder Clearance 0.0
<b>U</b> -	BN10 Y	Shank Clearance 0.0
		Block Limit 😏 🗸
	undary to be private	0
Edit History Apply edi	it history on calculation	$\bigcirc$
	Apply Queue	Accept Cancel

8 Input *Tolerance* 0.02

9 Set limit tool centre to block edge.



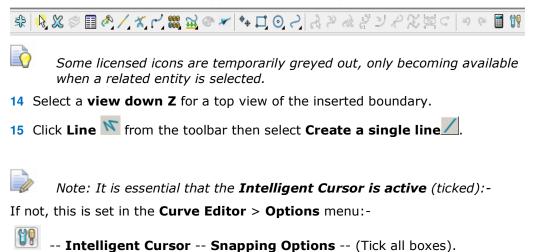
- 10 Un-tick Allow boundary to be private.
- **11** Select **Apply** followed by **Accept**.
- 12 Undraw the **model** to view the new **Boundary**.



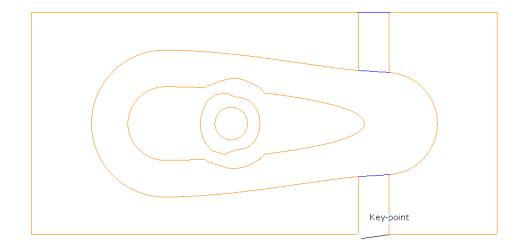
The **Boundary** will be modified so that the gap is removed between the 2 outer, closed, segments to form separate inner and outer segments. In addition to this, two of the other segments, which contain jagged sections, will be smoothed.

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.



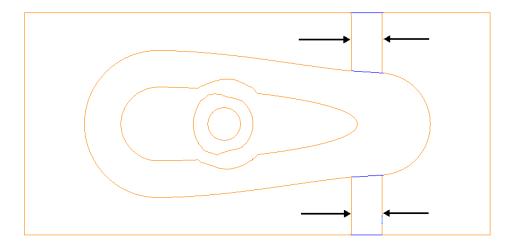
- 16 Using the left mouse button, snap (Key points) to create single lines across the ends of the four gaps, as shown in the image below.



- 17 Select the **Single Line** option  $\checkmark$  to switch it off.
- **18** To remove redundant spans on the curves, click the **Limits** option to access **Interactively limit wireframes**.



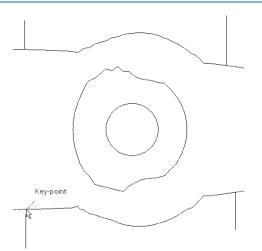
19 Click **Interactively limit wireframes** and using the left mouse button, select the parts of the curve (arrowed) to be removed.



- 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.



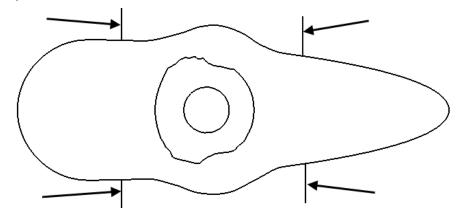
- 25 Repeat this process for the lower section.
- 26 Select the upper, original curve and select the **Cut Item** option.



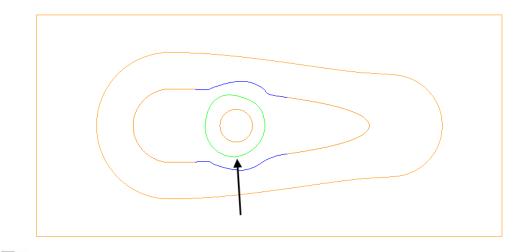
27 Click the **key points** at the lower end of each line to cut the original curve.



- 28 Repeat the above process on the lower, original curve then select and Deleteall the unwanted segments between the 2 pairs of lines.
- 29 Then, select and **Delete** the 4 vertical lines.



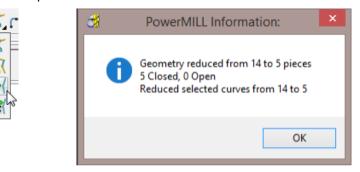
**30** Create a new smooth **Bezier Curve** to replace the one indicated below.



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**.

**31** Select all **boundary** segments by dragging a box over them with the left hand mouse key depressed.

32 Select the option **Merge** selected segments to join together all groups of **curves** that butt up to each other.



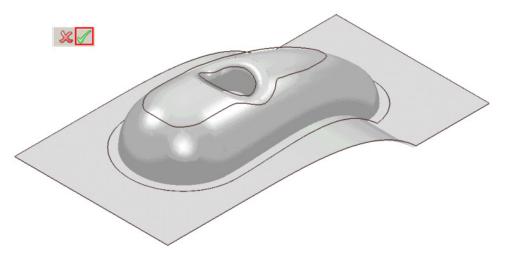
~			
6	1		L

The wireframe will now consist of five single segments as reported in the **PowerMILL Information box** (above right).

- **33** 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.

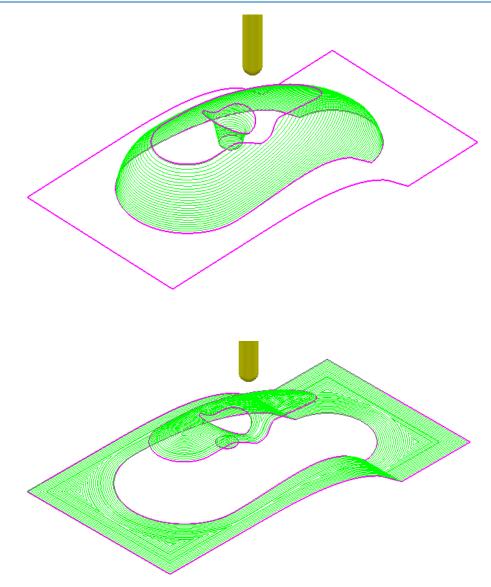
34 Select **V** to **Accept** changes and export the wireframe back to **PowerMILL** to replace the original boundary.



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.

- **35** Generate a **Constant Z Finishing** toolpath with **Stepdown** 1 and in the **Limit** page set the **Boundary Trimming** option to **Keep outside** (Steep areas).
- **36** Generate a **3D Offset Finishing** toolpath with **Stepover** 1 and in the **Limit** page set the **Boundary Trimming** option to **Keep inside** (Shallow areas).
- **37** 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.

# 12.2D Machining

## Introduction

**PowerMILL** offers a range of 2D machining strategies that include **2.5D Feature** machining (including **Drilling**), **2D Curve** machining (including **Chamfer** Milling), and **Face Milling**.

## **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. **Hole Features** can also be created during **3D Area Clearance** strategies to enable the pre-drilling of plunge move, positions. It is essential that a **Feature Set** includes all **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** strategies can only be applied to **Hole Features**. A comprehensive, list of cycle types are supported that include; Standard Drilling, Boring, Helical/Profile milling, Tapping, and Thread Milling.

The six different types of Feature include:-

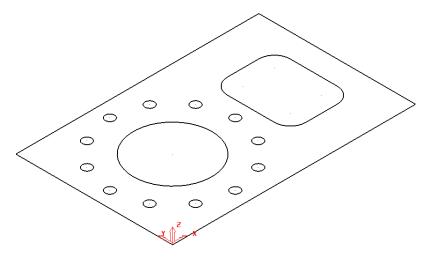
- **Pocket**:- Machining will only occur fully inside this type of closed **Feature**.
- Slot:- Machining will occur with the tool running central, left, or right of this type of open/closed Feature.
- Boss: Machining will only occur fully outside this type of closed Feature.
- Hole, this type of Feature 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:

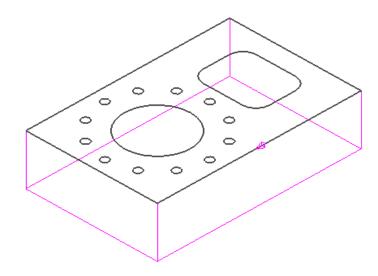
.....\COURSEWORK\PowerMILL-Projects\2DPatternExample

4 In the **PowerMILL** *explorer*, right-click on **Feature Sets** and from the local menu, select **Create Features**.

The **Feature** dialogue will appear in the graphics area.

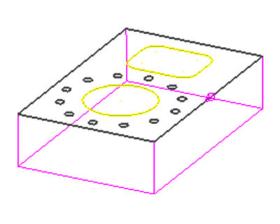
<del>3</del>	Feat	ure	?	×
Create Edit				
Туре		Name root		
Boss 🗸	]	В		
	-	Tolerance		
Smart creation		0.1		
Define top by				
Absolute	~	0.0		
	•			
Define bottom by				
Absolute	~	-40.0		
		Draft angle		
Ø 0.0		0.0		
Apply		Close		

5 Select the large, outer, rectangular **Pattern** segment and enter values exactly as shown (using **Type Boss**) in the above **Feature** dialog before clicking **Apply** (Do not close the dialog).



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

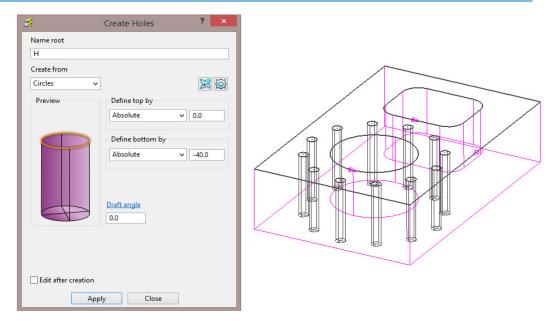
8	Fea	ature	?
Create Edit			
Туре		Name root	
Pocket	Pocket v		
		Tolerance	
<ul> <li>Smart creation</li> </ul>	n	0.1	
Define top by			
Absolute	~	0.0	
Define bottom Absolute	y v	-30.0	
Ø 0.0		Draft angle 0.0	
A	ply	Close	



- 6 Select both, the *rectangular segment* (with corner fillets) and the large *circular* segment.
- 7 Enter values exactly as shown in the above **Feature** dialog (using **Type Pocket**) 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 **Create Features**.

- 8 Close the *Feature* creation dialog.
- 9 Right mouse click on the new Feature Set 1 and from the local menu select Create Holes.



10 Select all of the Dia 6 **circles** (12) and enter values exactly as shown in the above left **Feature** dialog (using *Type* **Hole**) before selecting **Apply**.

```
🛓 🔐 Feature Sets
  🖮 🚽 🖓 🔒 > 1
     <u>ы</u>⊸<sup>©</sup> Н_1
      .
⊞... 🖰 Н_2
      .
⊞... <sup>С</sup> Н_З
      .
⊞.- <sup>©</sup> Н_4
      🗄 – 🖰 Н_5
      .
⊞... 😁 Н_6
      .
⊞⊸<sup>©</sup> Н_7
     🗄 🖰 Н 8
     📩 🕾 H 12
```

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**).

11 Close the *Hole* creation dialog.

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 2 to open the **Block** dialog, set the **Type** as **Feature** and the select **Calculate**.
- 3 Click **I** 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.

Toolpath name   Untransport     Block   Tool   Machine tool   Step cuting   Wall finishing   Unsafe segment removal   High speed   Order   Approach   Cutter compensation   Point distribution   Tool axis   Order dasks and links   Approach   End point   End point   Feeds and speeds   Itiony   Notes and Description   User defined settings     Step courtion   Constant stepdown   Rapid move heights   End point   Start point   End point	💰 Featu	ire Set Area Clearance ? ×
Block Tool Machine tool Timit Feature set area clearance Offset Step cutting Wall finishing Unsafe segment removal F High speed Order Automatic verification Cutter compensation Point distribution Tolerance Offset model V Cut direction Profile Cut direction Cut direction Profile Cut direction Cut direction Profile Cut direction Profile Cut direction Cut direction Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Profile Cut direction Cut direction Cut direction Cut direction Cut direction Cut direction Cut direction Cut direction Cut direction Cut di	Toolpath name	D10TR1-A1
Calculate Queue OK Cancel	Block Tool Machine tool Machine tool Machine tool Machine tool Machine tool Machine tool Machine axis control Machine axis control	Style Offset model v Cut direction Profile Climb Climb Climb Climb V Climb V Climb V Climb V Climb V V Climb V V Climb V V Climb V V Climb V V Climb V V Climb V V V Climb V V V Climb V V V V Climb V V V V Climb V V V V V V V V V V V V V

6 In the **Offset** page tick **Spiral**.

Feature Set Area Clearance ?				×
Toolpath name	D10TR1-A1			
Workplane Block Block Unit Kachine tool Constraint Feature set area clearance Offset Constraint Step cutting Wall finishing	Advanced Offset Settings Maintain cut direction Spiral Remove cusps Machine smallest first			
····· ····· ····· ····· ····· ····· ····· ····· ····· ····· ····· ····· ······	Cut direction Profile Climb v	Area Climb	~	]

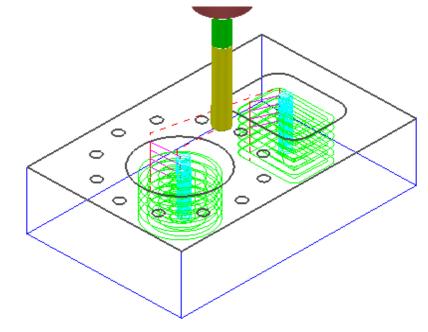
7 In the Leads and Links - Lead in page select:-

```
1<sup>st</sup> choice > Ramp
```

Ramp Options > Follow Circle - Circle diameter (TDU) 0.65

Lead in	📸 Lead In Ramp Options 📍 🗙
	1st Choice 2nd Choice
1st choice Ramp 👻	Follow Max zig angle
Distance 0.0	Circle ¥ 4.0
Angle 0.0	Circle diameter (TDU)
Radius 0.0	0.65
Ramp options	Closed segments only
	Ramp height
Overlap distance (tool diameter units) 0.0	Type <u>Height</u>
Allow start points to be moved 🔽	Incremental y 5.0
Add leads to short links 📝	

8 Select **Calculate** to process the toolpath and then select **Cancel** to close the dialog.





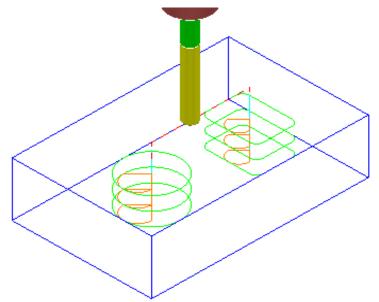
The **Pocket Features** are machined with **0.5** stock left on the sidewalls and **0** stock on the base (Radial thickness 0.5 and Axial thickness 0).

9 Click Stoppen the **Toolpath Strategies** dialog and select **2.5D Area Clearance**.

**10** Select **Feature Set Profile** and input data exactly as shown in the following dialog.

Ċ₫ F	eature Set Profile ?
Toolpath name	D10TR1-A2
Workplane	Feature set profile Style
	Cut direction Profile Additional profiles
	Climb V Climb V
Approach     Automatic verification     Cutter compensation     Point distribution     Tool axis     Machine axis control     Rapid move heights     Leads and links	Tolerance 0.02 Thickness 0.0 Stepover
- J       Start point         - J       End point         ⊕ - I       Freeds and speeds         - I       History         - I       Notes and Description         - J       User defined settings	5.0 Stepdown Automatic V 10.0 Constant stepdown
User denned settings	Rest machining Geometry position Slot cutting Part V Keep left V

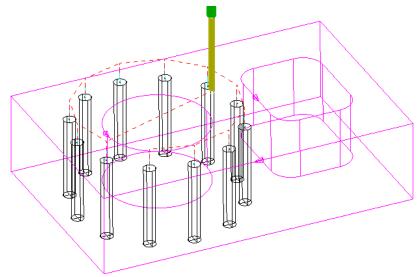
- 11 Select the Lead In page and for 1st Choice select Pocket Centre and then select the Lead out same as lead in option
- 12 Select **Calculate** to process the toolpath and then **Close** the dialog.



- 13 Create a Dia 3 Drill of Length 30, named Drill3.
- 14 Click  $\bigotimes$  to open the **Strategy selector** dialog and select the **Drilling** tab.
- 15 From the list, select **Single Peck** to open the **Drilling** dialog (below).

- 🕉	Drilling ? ×
Toolpath name	Drill3-A1
Holes	Drilling
Block	Cycle type Single peck
Machine tool	Define top by
Drilling	Hole top
Feed rate reduction	Operation
🔚 Automatic verification	Centre drill 🗸
<ul> <li>Yool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Start point</li> <li>For Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Clearance 5.0 Peck Depth 0.0 Depth 2.0 Dwell time 0.0 Tolerance
	0.1 Thickness 
	Drilling cycle output     Draw     Select
	Calculate Queue OK Cancel

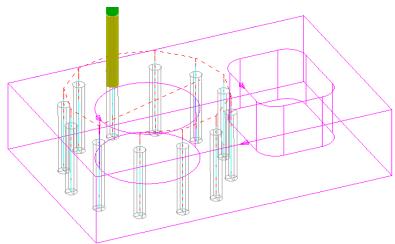
- **16** Select **Calculate** and all **Hole Features** will automatically be selected before the **Drilling** toolpath is processed.
- 17 Select **Cancel** to close the dialog.



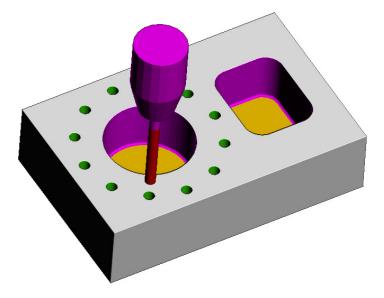
- **18** Create a Dia 6 **Drill** of Length 50 and named **Drill6**.
- 19 Click  $\bigotimes$  to open the **Strategy selector** dialog and select the **Drilling** tab.
- 20 From the list, select **Break Chip** to open the **Drilling** dialog (below) and then click **Select** to open the **Feature Selection** dialog.

3	Drilling	?	×
Toolpath name	Drill6-A1		
Holes Workplane Block Tool Retraction Feed rate reduction Feed rate reduction Feeds and links Feeds and speeds Feed rate reduction Feed rate rate rate rate rate rate rate rate	Drilling Cycle type Break chip v Define top by Hole top v Operation Through hole v Clearance 5.0 Peck Depth 12.0 Depth 2.0 Dwell time 0.0		
	0.1       Thickness		
	Calculate Queue OK Cancel		

- 21 Select **Calculate** and all of the contained **Hole Features** will automatically be selected before the **Drilling** toolpath is processed.
- 22 Select **Cancel** to close the dialog.



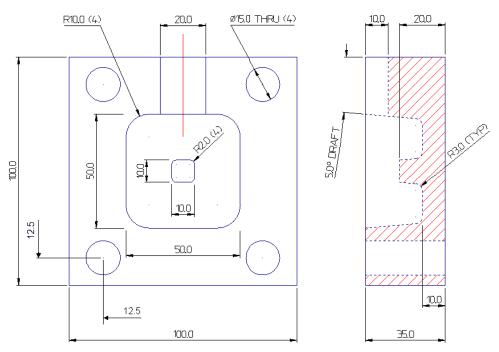
23 Run a full **ViewMILL** simulation on all 4 toolpaths.



24 Save the Project.

## **2D Machining Exercise**

For this exercise the following 2D drawing will be used along with the imported 2D wireframe **Pattern**:



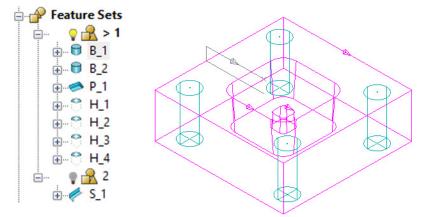
- 1 Select File > Delete All and Tools > Reset Forms
- 2 Create an empty **Pattern** and **Insert File**:

.....\PowerMILL\_Data\Models\2d\_Wizard\_Example.dgk

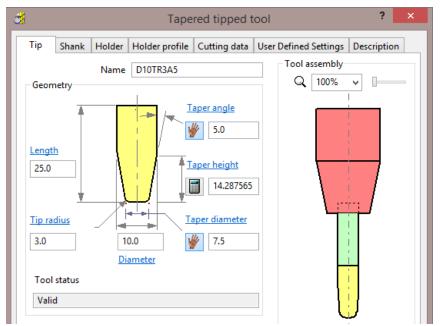
3 Save Project As:-

.....\COURSEWORK\PowerMILL\_Projects\2D\_Wizard\_EX1

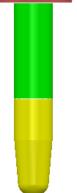
4 Create a **Feature Set** (1) and create appropriate **Features** using the above drawing dimensions.



- a The outer square and small tapered, central upstand are created as **Boss** *Features*.
- **b** The intermediate square is created as a tapered **Pocket** *Feature*.
- c The four Dia 15 circles are created as **Hole** *Features*.
- d The single red line on the drawing is used to define a Slot Feature (This should be created in a separate Feature Set (1\_1)).
- 5 Create a Dia **10** with **3 Tip Radius** tool named **D10TR3** (Roughing, Semifinishing within the main Pocket Feature including the small Boss).
- 6 Create a Dia 10 Tapered Tipped tool named D10TR3A5, with Tip Radius 3, Taper Angle 5 Deg, Length 25, Taper Diameter 7.5 (Final Finishing of tapered walls Main Pocket and small Boss).
- 7 Create a **Shank** of Dia **10** Length **30**.
- 8 Create 2 Holder with Overhang 50 and entities:
  - a Lower Dia 20 Upper Dia 30 and length 25.
  - b Upper/Lower Dia **30** and Length **25**.



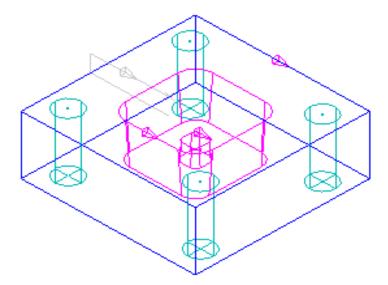
To calculate the Taper Height based on the Taper Angle and base Taper Diameter, the Manual Input icon , adjacent to the Taper Height, is clicked to apply the **Calculate** option to this parameter.



- 9 Create a Dia 15 Drill named Drill15.
- 10 Create an End Mill of Dia 10 named EM10.

🖕 👘 T	ools	
<b></b>	- <b>-</b>	> D10TR3
<b>.</b>	•	D10TR3A5
<b></b>	<b>9</b> U	DRILL15
<b>±</b>		EM10

- 11 Activate the tool **D10TR3**.
- 12 Activate, Feature Set 1 and create a Block (Type Feature).



**13 Calculate** default **Rapid Move Heights** ready to create the 2D **Feature** machining strategies.

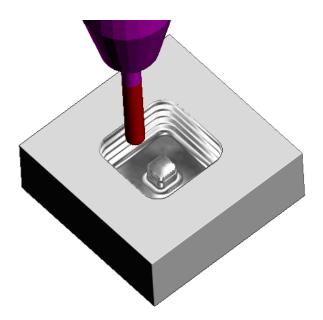
### **Main Block and Central Pocket**

D10TR3 - Feature Set Area Clearance – Offset Model

- 14 Click Stoppen the Toolpath Strategies dialog and select 2.5D Area Clearance.
- **15** Select **Feature Set Area Clearance** and input the data exactly as shown in the following dialog.

😚 Feature Set Area Clearance ? 🗙		
Toolpath name	D10TR3-RGH	
Workplane Block J Tool Kachine tool Kachine teol Kachine set area clearance Cffset	Feature set area clearance Style Offset model	
Step cutting Wall finishing Unsafe segment removal High speed Coder Approach	Cut direction Profile Area Climb V Climb V	
Automatic verification     Cutter compensation     Point distribution     Tool axis     Machine axis control     Rapid move heights	0.1 Thickness 	
<ul> <li></li></ul>	Stepover 3.0 Stepdown	
History	Automatic V 5.0	

**16 Calculate** the toolpath and perform a **ViewMILL** simulation as shown below.



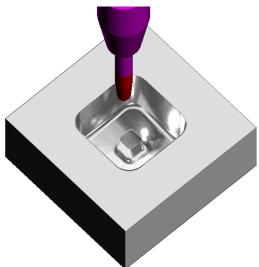
D10Tr3A5 - Feature set Profile

17 Click to open the **Toolpath Strategies** dialog and select **2.5D Area Clearance**.

Select **Feature Set Profile** and input the data exactly as shown in the following dialog.

<b>3</b>	Feature Set Profile ?
Toolpath r	D10TR3A5-FIN
Workplane     Block     Tool     Machine tool     Get     Limit     Feature set profile     Step cutting     UCut distances	Feature set profile Style
Wall finishing Unsafe segment removal Unsafe	Cut direction Profile Additional profiles Climb Climb V Tolerance 0.02 Thickness 0.0 Stepover 3.0 Stepdown Automatic V V 10.0 Constant stepdown Rest machining
	Geometry position Slot cutting Part v Keep left v Calculate Queue OK Cancel

**18 Calculate** the toolpath and continue the **ViewMILL** simulation (as shown below).

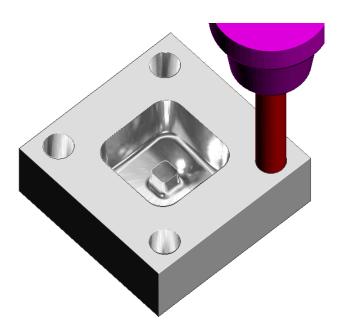


#### Drill15 - Drilling

- 19 Click 🔊 to open the Toolpath Strategies dialog and select Drilling.
- 20 Select **Deep Drill** and input the data exactly as shown in the following dialog.

3	Drilling	? ×
Toolpath name	Drill15	
Holes Workplane Block Tool Machine tool Machine tool Machine tool Herraction Feed rate reduction Pecking Order Automatic verification Machine axis control Rapid move heights Machine axis control Rapid move heights Eads and links Start point Feeds and speeds History Notes and Description User defined settings	Drilling         Cycle type         Deep drill         Define top by         Hole top         Operation         Through hole         V         Clearance         5.0         Peck Depth         V         Tolerance         0.1         Thickness         V         0.0	Start         0.0         Depth         0.0         Dwell time         0.0

**21 Calculate** the toolpath and continue the **ViewMILL** simulation (as shown below).



#### EM10 - Feature Set Profile

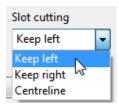


The Y dimension of the existing Block will need to be adjusted to fully, include the Slot Feature.

- 22 Open the **Block** dialog and input **60** for the **Max Y** value before selecting **Accept** (<u>Do not</u> select Calculate in the Block dialog).
- 23 Activate Feature Set 2.
- 24 Click Stoppen the Toolpath Strategies dialog and select 2.5D Area Clearance.
- 25 Select **Feature Set Profile** and input the data exactly as shown in the following dialog.

Toolpath name   Morkplane   Block   Tool   Machine tool   Machine tool   Limit   Feature set profile   Step cutting   Usafe segment removal   Filip speed   Order   EM10-SLOT_Centre	?	×
Block  Tool  Machine tool  Feature set profile  Feature set profile  Unsafe segment removal  Fir High speed		
Unsafe segment removal Profile Additional		
J Approach         J Automatic verification         Outter compensation         Cutter compensation         J Cutter compensation         J Coll axis         J Tool axis         Machine axis control         Rapid move heights         Leads and links         J Leads and links         J Leads and links         J End point	profiles	
Image: Feeds and speeds         Image: History         Image: Notes and Description         Image: User defined settings         Image: User define	•	

- 26 Select the *Slot* Feature only and then Calculate the toolpath.
- 27 Create 2 more copies of this strategy using *Slot Cutting* Keep left and Keep right respectively.



Geometry position Slot cutting Part Keep left • Keep left Keep right Centreline

28 Continue the ViewMILL simulation on the last 3 toolpaths (as shown below).

The required **Slot** width is achieved by creating 3 separate toolpaths using the 3 different **Slot cutting** options, starting with **Centreline**, followed by Keep left, and then Keep right.

29 Save the Project.

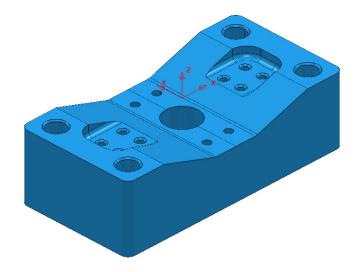


## **Creating Features directly from holes in the Model**

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



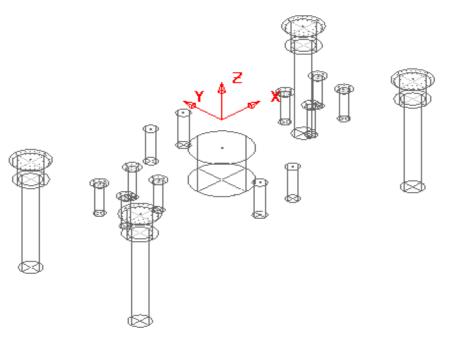
### 3 Save Project As:-

#### .....\COURSEWORK\PowerMILL-Projects\DrilledHoles-Example

- 4 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**).
- 5 Activate the Workplane MC-datum.
- 6 Select the whole **model**.
- 7 In the **PowerMILL** *explorer*, right-click on **Feature Sets** and from the local menu select **Create holes**.

	s	😙 Create Holes ? 🗙
	Feature Sets	Name root
🕀 閿 Models	Create Feature Set	н
	Create Features	Create from Tolerance
Here Macros	Create Holes	Model v 0.1
	Import	Create compound holes
		Use active workplane only
		Find holes in both directions
		Find holes going down
		Find holes going up
		Group holes by axis
		Try to recognise partial holes
		Minimum arc angle for partial holes
		300.0
		Ignore capped cylinders
		Edit after creation
		Apply Close

- 8 Input data into the **Feature** dialog exactly as shown on previous page, and click **Apply**.
- 9 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**).

### **EXERCISE**

- 10 Define suitable tools, then **Drill** and **Counter Bore** the **Holes** using the **Drilling** dialog options.
- **11 Save** the **Project**.

### **Hole Capping**

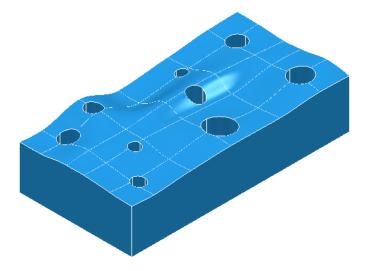
**Hole Features** can be used as the basis for capping holes in the 3D **model**. If the intersection of the hole and the top face is non-planar, **PowerMILL** will cap the hole with a **surface** that follows the tangency of the surrounding 3D **model**.

### 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:

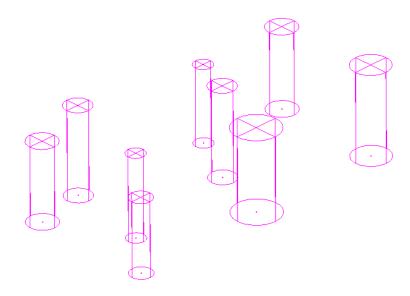
.....\PowerMILL\_Data\Models\Block\_with\_holes.dgk



- 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 dragging a box over it while the left mouse button is depressed.
- 5 Right mouse click on **Feature Sets** and from the local menu select **Create Holes**.

	s	Create Holes
→ 🥵 Workpl ↔ 🥪 Levels a	Feature Sets	Name root
🕀 💓 Models	Create Feature Set	н
	Create Features	Create from Tolerance
Here Macros	Create Holes	
	Import	Create compound holes
		Use active workplane only
		Find holes in both directions
		Find holes going down
		Find holes going up
		Group holes by axis
		Try to recognise partial holes
		Ignore capped cylinders
		Edit after creation
		Apply Close

6 In the **Feature** dialogue enter **Name root H** and select **Apply**.



In this case the tops of the **Hole Features** are positioned at the shortest distance from the face of the **Block**. This results in all of the **Hole Features** being created upside down.



If the **Block** had <u>not</u> been defined prior to this, then the **Hole Features** would have been created the 'right way up' relative to the Z Axis.

It is also possible to control the resultant direction from within the **Create Holes** dialogue. This is achieved if **'Use active workplane only**' is *ticked* along with selecting the option **'Find holes going down**'.



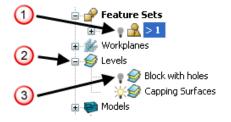
Any **Hole Features** that are created to a different orientation from the original Z Axis, will automatically be assigned to a separate **Feature Set** and associated **Workplane**.

In this case the holes were deliberately created upside down to demonstrate the **Reverse Holes** editing option.

→      →	1	Luic Holes	
			5
🗄 🔁 H_ 🗸 🗸	Activate	Edit	Feature Set Editor
🕀 🔁 H_4	Activate Workplane	Unlock Feature Set	Arc Fit Selected
🗄 🔁 H_ 🗸	Draw		Separate Compound Holes
⊞- <mark>©</mark> H_(	Add to Folder	Remove from folder	
⊞- <mark>©</mark> Н_1		Delete Feature Set Combine Hole	Combine Holes
🗄 🔂 H_(	Export		Cap Holes
⊞⊣ <mark>⊖</mark> H_9	Rename		Reverse Selected Holes
🗄 🎉 Workplane	Select Holes		Reverse Selected Holes
🗄 🥩 Levels and	Select All		
🗄 💓 Models 📃	2		
- 🍓 Stock Moc	Select Duplicates	1	

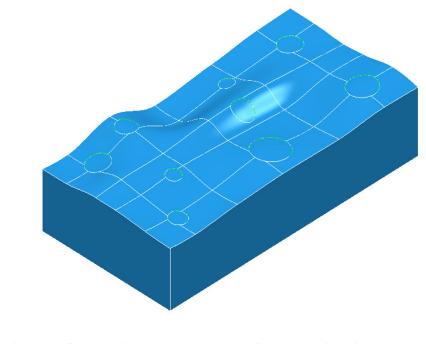
- 7 Right-click **Feature Set 1** and in the menu click **Select All** and then from the same menu, select **Edit** > **Reverse Selected Holes**.
- 8 Finally, select the top surface of the **Model** and in the **PowerMILL** *explorer*, right-click **Feature Set 1** and from the local menu select **Edit Cap Holes**.

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.



- 9 Click the **light bulb** to undraw **Feature Set 1** ((1)).
- 10 Expand **Levels** by clicking on the + symbol (2).
- 11 Click the *light bulb* to undraw main surfaces ( $\Im$ ).

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.

12 Click on the *light bulb* on the *Block with holes* level to re-display the main model.

## **Internal Thread Milling**



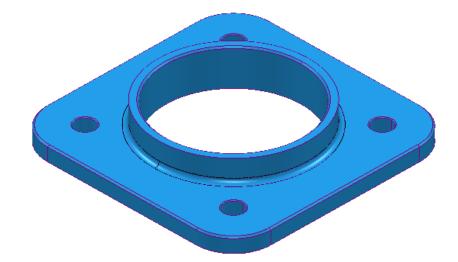
Thread Milling is included as one of the options in the Drilling dialog.

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\ThreadMILL\_2014-Start



- 3 Select File > Save Project As: .....\COURSEWORK\PowerMILL-Projects\ThreadMILL-Example1
- 4 Activate the *Thread Milling* Tool named M40-Coarse-Pitch-5.
- 5 Activate the Feature Set named Pocket (This defines the large central hole).

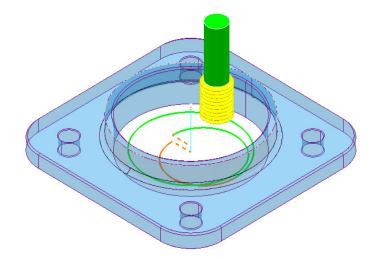
- 6 Select **Toolpath Strategies** and in the **Strategy Selector** dialog, select the **Drilling** tab followed by the **Thread Mill** option.
- 7 In The **Drilling** dialog set both *Radial* and *Axial* **Thickness -3**, **Pitch 5**, **Name M40-RH-IntThread**, leaving all other parameters as default (shown below).

- <del>3</del> 3	Drilling ? ×
Toolpath name	M40-RH-IntThread
Holes Workplane Block Tool Machine tool Filing Retraction Threading Automatic verification Cutter compensation Cutter compens	Drilling   Cycle type   Thread milling   Define top by   Hole top   Hole top   Operation   Drill to hole depth   Direction   1.0   Pitch   5.0   1.0   Pitch   5.0   0.0   Depth   5.0   Direction   0.0   Direction   0.1   Thickness   Image: Constraint of the state of th
	Drilling cycle output Draw
	Calculate Queue OK Cancel

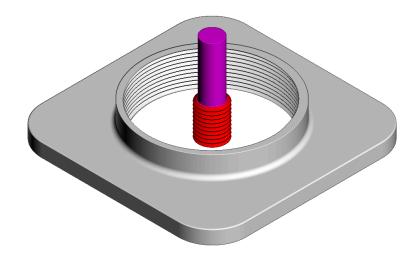
- 8 Select the **Cutter Compensation** page.
- 9 Tick the **CNC cutter compensation** box and select **Type Tool wear**.

- <del>31</del>	Drilling	?	×
Toolpath name	M40-RH-IntThread		
Holes Workplane Block Tool Machine tool Machine tool	Cutter compensation CNC cutter compensation Type Tool wear Minimum radius 0.1	<b>&gt;</b>	

**10 Calculate** the form and then **Close** the dialog.



- 11 Calculate a new Block using Defined by Box and Type Model.
- 12 Right-click the NC Program *MC-All* and in the local menu select Simulate from Start.
- 13 Open a new ViewMILL session and run a simulation of all toolpaths. (Do not activate any of the toolpaths, otherwise the current Block will become oversize).



The completed **ViewMILL** simulation is as shown above.

14 Disconnect ViewMILL from the simulation (but do not apply Exit ViewMILL), and return to the **PowerMILL** window.

## **External Thread Milling**

- **15 Activate** the **Feature Set** named **Boss** (This defines the outside of the central boss).
- 16 Select Toolpath Strategies S and in the Strategy Selector dialog, select the Drilling tab followed by the External Thread option.

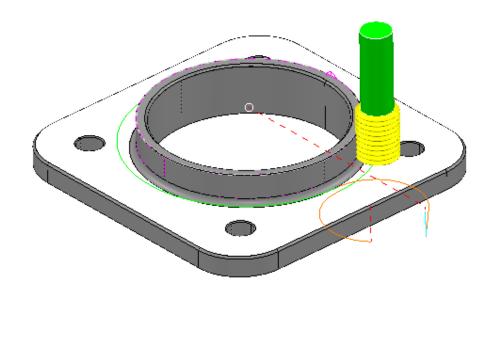
17 In The Drilling dialog, set the Cycle Type to External Thread, both Radial and Axial Thickness -3, Pitch 5, Name M40-RH-IntThread, leaving all other parameters as default (as shown below).

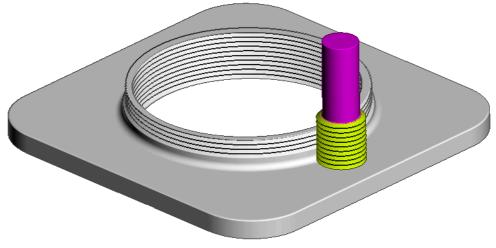
<b>3</b>	External Thread	? ×
Toolpath name	M40-RH-ExtThread	
Bosses	External Thread	
	Cycle type	
	External thread	
External Thread	Depth type Feature depth v	
Threading  Order  Automatic verification  Cutter compensation  Tool axis	Clearance	
	Turns         Depth           1.0         24.0           Pitch         0.0	
Notes and Description	Tolerance Direction	
User defined settings	0.1 Climb	~
	Thickness	
	Drilling cycle output Draw	
	Calculate Queue OK Cancel	

18 Select the **Threading** page and select the **Use a 45 degree arc** lead option.

- <del>3</del> 4	External Thread	? ×
Toolpath name	M40-RH-ExtThred	
Bosses Workplane Block Tool Keternal Thread Automatic verification Cutter compensation Cutter compensation Machine axis control Start Point End point Freeds and speeds	Threading	Number of passes          1         Allowance         1.0

- **19 Calculate** the form and then **Close** the dialog.
- 20 Re-open and re-connect the ViewMILL session and run a simulation of the latest toolpath M40\_RH-ExtThread.





21 Select File - Save Project.

# **2D Curve Machining**

**PowerMILL** contains 3 strategies that directly machine relative to 2D **Pattern** segments. These include:

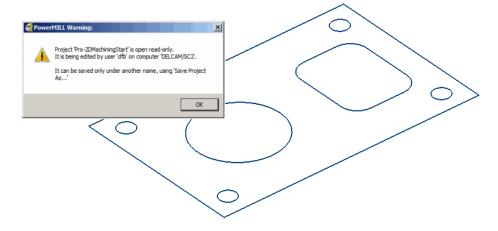
- **2D Curve Area Clearance**.
- 2D Curve Profile.
- Chamfer Milling.

There is also a dedicated **Face Milling** strategy that calculates relative to the **Block** dimensions.

### **Example 1**

- 1 Select File > Delete All and Tools > Reset Forms.
- 2 Open the Project:

.....\PowerMILL\_Data\Projects\2DCurve-2015mc-Start



3 Select File > Save Project As:

.....\COURSEWORK\PowerMILL-Projects\2DCurve-Machining

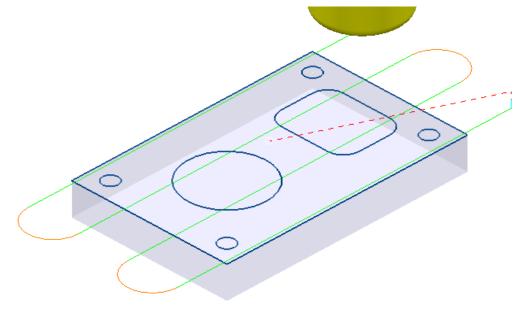
- 4 Activate the Pattern named CurveAll.
- 5 Calculate a Block using Defined by Box and Type Active Pattern (Do not close the dialog).
- 6 In the Minimum Z co-ordinate box input -22 before selecting Accept to close the dialog.

Limits		
Min	Max	Length
X 0.0	155.0	155.0
Y 0.0	<b>100.0</b>	Let 100.0
Z -22.0	0.0	22.0
X	X Y Z	<b>C</b>
Estimate limits		
Tolerance	<b>Expansion</b>	Туре
0.1	0.0	Active Patter 🗸

- 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.

<b>3</b>	Face Milling	? ×
Toolpath name	D50TR6-FaceMill	
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Face milling</li> <li>Cut distances</li> <li>Finishing</li> <li>Raster</li> <li>Automatic verification</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Leads and links</li> <li>Leads and links</li> <li>Lead out</li> <li>Links</li> <li>Start point</li> <li>Freds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Face milling Face definition Face Z position (z) 0.0 Engagement feed rate (%) 100.0 Tolerance 0.1 Stepover $\underbrace{\begin{subarray}{llllllllllllllllllllllllllllllllllll$	XY expansion (e) 5.0 Style Two way
	Calculate Queue	OK Cancel

- 10 In the Leads and Links Links page and set the Short/Long threshold as40 and Short as Circular Arc.
- 11 Click **Calculate** to create the toolpath and then **Close** the dialog.



#### 12 Activate the *Tip Radiused* tool named **D16T3**.

**13** In the **2.5D Area Clearance** options select **2D Curve Area Clearance** and enter data into the form exactly as shown below.

Cu	rve Area Clearance ? ×
Toolpath name	D16TR3-2DCurve-AC
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Curve area clearance</li> <li>Cut distances</li> <li>Finishing</li> <li>Corffset</li> <li>Offset</li> <li>Step cutting</li> <li>F High speed</li> <li>Order</li> <li>Approach</li> <li>Automatic verification</li> <li>Cutter compensation</li> <li>Point distribution</li> </ul>	Curve area clearance
Tool axis Machine axis control Rapid move heights Leads and links Start point Freds and speeds History Notes and Description	Tolerance     Cut direction       0.1     Climb       Curve thickness       1.0       Stepover
	Calculate Queue OK Cancel

14 Select the **Cut distances** page and input **Stepdown 10**.

<b>3</b>	Curve Area Clearance	? ×
	Toolpath name D16TR3-2DCurve-AC	
Workplane Block Block U Tool Curve area clearance	Stepdown (t)     Image: Stepdown state     Horizontal     Stepover     Image: Stepover     Image: Stepover     Image: Stepover     Image: Stepover     Image: Stepover	Stock depth (d) 0.0

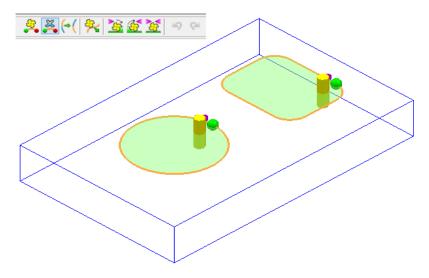
15 In the Leads and Links - Lead in page select:-

1<sup>st</sup> choice > Ramp Ramp Options > Follow Circle - Max zig angle 4 - Circle diameter (TDU) 0.65

**16** From the main **Curve area clearance** page, *Curves definition*, select

Interactively modify machinable sections 🚨

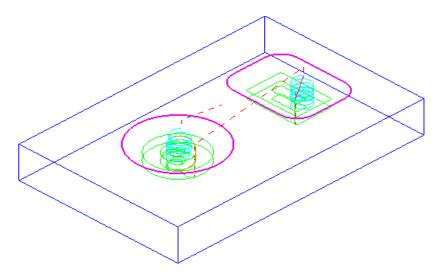
These options enable the user to have full control of direction of cut and the areas to be machined.





While the **Curve Area Clearance** form is open, the areas to be machined are displayed as a shaded preview.

- 17 Select V to exit the **Interactively modify machinable sections** toolbar.
- **18** Select **Calculate** to create the toolpath and then **Close** the dialog.



#### 19 Activate the *End Mill* named EM16.

20 From the **2.5D Area Clearance** options select **2D Curve Profile** and enter data into the dialog exactly as shown below.

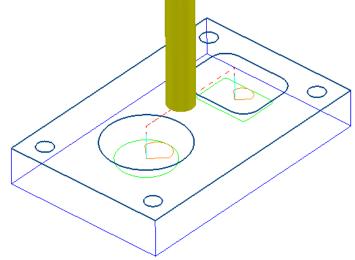
	Curve Profile ? ×
Toolpath name	EM16-CurveProfile
Workplane Block Dol Machine tool Machine	Curve profile Curves definition CurvePockets Position Over limit Over limit Over limit Style Over limit
Tool axis Machine axis control Rapid move heights Leads and links Start point Feeds and speeds History Notes and Description	Tolerance     Cut direction       0.01     Climb       Curve thickness     0.0       Rest machining
	Calculate Queue OK Cancel

21 Select the **Cut distances** page and input **Stepdown 25**.

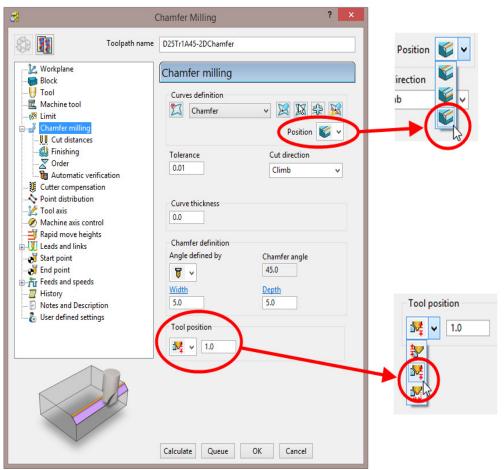
Vorkplane ^	Cut distances
Block 	Vertical Extent Number of cuts Number of cuts v 1 Stepdown (t) 25.0
Step cutting     Step cutting     F High speed     Sorrer     Approach     Cutter compensation     Oint distribution     Yool axis	Horizontal Extent <u>Stock width</u> Stock width Stepover 8.0

- 22 Select the Lead in page and for 1st Choice select Pocket Centre and then select the Lead out same as lead in option
- 23 Click **Calculate** to process the toolpath and then **Close** the dialog.





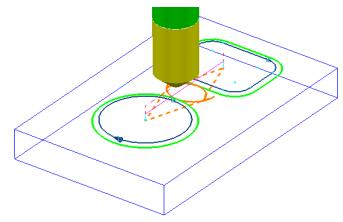
- 24 Activate the Chamfer Tool named D25Tr1A45.
- 25 From the **2.5D Area Clearance** options select **Chamfer milling** and enter data into the dialog exactly as shown below (Do not select Calculate yet!).



₩orkplane → Block	Cut distances	
U Tool Machine tool W Limit Chamfer milling U Cut distances W Finishing ∑ Order ∑ Order ∑ Automatic verification	Vertical Extent Number of cuts v Stepdown (t) 25.0	Number of cuts
Cutter compensation     Point distribution     Cutter compensation     Point distribution     Constant distribution     Achine axis control     Constant distribution     Rapid move heights     Constant dinks     Const	Horizontal Extent Stock width v <u>Stepover</u> 8.0	Stock width 0.0

26 Select the **Cut distances** page and input **Number of cuts** as **1**.

27 Click **Calculate** to process the toolpath (Do not close the dialog).



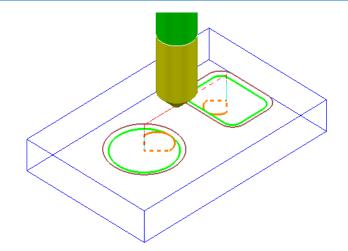
The resultant **Chamfer milling** strategy has appeared (incorrectly) on the outside of the **Pattern** segments.

The toolpath needs to be recycled with the **Chamfer Milling** tool tracks changed to appear on the inside of the **Pattern** segments.

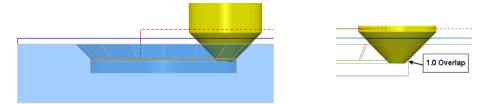
- 28 In the Chamfer milling dialog, select the *Recycle* icon to enable changes to be made to the above strategy.
- 29 Select Interactively modify machinable sections is to access the *Edit Machinable Sections* toolbar.
- 30 Select the **Reverse machining side** option, followed by V to accept the changes and close the toolbar.

Chamfer milling	· · · · · · · · · · · · · · · · · · ·
Curves definition Chamfer Position	

**31** In the main **Chamfer milling** dialog, select **Calculate** to create the **toolpath** and then **Close** the dialog.

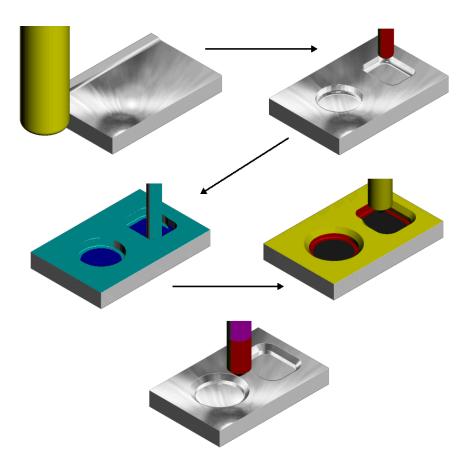


The **Chamfer milling** options selected result in the angled part of the tool overlapping the base of the chamfer by 1mm.



A 3D **model** of the finished component has been imported to provide a visual check of the **Chamfer machining**.

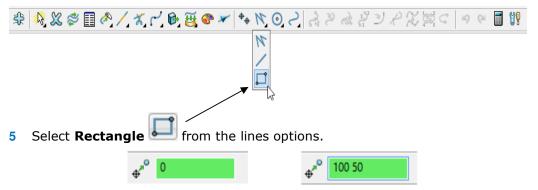
32 Run a **ViewMILL** simulation of the whole machining process.



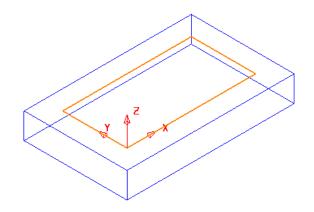
### **Cutter Compensation Example**

Where applicable a **Cutter Compensation** page is available in a **2.5D Machining Strategy** dialog **explorer** options.

- 1 Select File Delete All and Tools Reset forms.
- 2 In the **PowerMILL** *explorer* right click on **Patterns** and from the local menu select **Create Pattern**.
- 3 Rename the **Pattern** as **Profile1**.
- 4 Right click on the new **Pattern** and from the local menu select **Curve Editor**.



- 6 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.
- 7 Select **Rectangle I** from the lines options again to deselect it.
- 8 Select the *green tick* to close the **Curve Editor** and accept the rectangular **Pattern**.
- 9 In the **PowerMILL** *explorer* right click on **Workplanes** and from the local menu select **Create Workplane**.
- 10 In the **Workplane Editor** toolbar, select the *green tick* save.
- 11 Activate the new Workplane.
- 12 Create a Dia 10 End Mill named EM10.
- **13** Calculate a Block using Defined by Box and Type Active Pattern.
- 14 Input the value *Min Z* as -20 and then lock both **Min Z** and **Max Z** values.
- 15 Input **Expansion 15** and then select **Calculate** to add stock to the X and Y dimensions of the **Block**.



- **16** In the **Rapid Move Heights** dialog select **Calculate**.
- 17 In the **Strategy Selector** form select the **2.5D Area Clearance** options.
- 18 Then select the **2D Curve Profile** strategy and enter data as shown in the following illustration.

<b>3</b>	Curve Profile ? ×
Toolpath name	EM10-Profile-NoComp
Workplane     Block     Tool     Tool     Machine tool     Workplane     Machine tool     With the tool     With to	Curve profile  Curves definition  Profile1  Position  Cover limit  Cover limit  Style  Cover limit  Cover limit Cover limit  Cover limit Cover limit
	Tolerance     Cut direction       0.01     Climb       Curve thickness       0.0

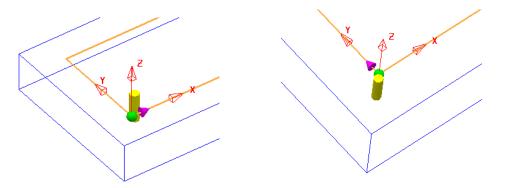
19 Select the **Cut Distances** page and input **20** for both the **Stock depth** and the Stepdown.

<b>3</b>		Curve Profile	?	×
	Toolpath name	EM10-Profile-NoComp		
Workplane Block Gamma Block Tool Machine tool Machine tool Curve profile Machines Machine tool Machine to	^	Vertical       Extent     Stock depth (d)       Stock depth     20.0       Stepdown (t)     20.0		

20 Return to the Curve Profile page and select the Edit Machining Sections options.

Curves definition           Profile1         Image: Curves definition	

21 If required, select **Reverse machining side** to move the profiling pass to the outside of the Pattern.

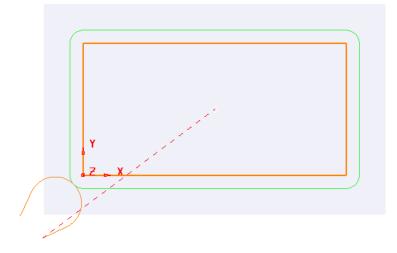


The above preview shows the cutter being switched from the inside to the outside of the **Pattern** prior to calculating the toolpath.



*This toolpath will not contain any data that relates to* **Tool Compensation** *output.* 

- 22 Select the *green tick* to accept the changes and exit **Edit Machining Sections**.
- 23 Calculate the Curve Profile toolpath.



24 Open the Leads and Links form and select Lead in – Horizontal Arc – Angle 110 and Rad 10.

2

- **25** Copy the **Lead In** settings to **Lead Out**.
- 26 Select Extensions and input an Extended move of 10 for both Inward and Outward.
- 27 Apply the Leads and Links settings to the active toolpath.
- 28 Right click on the active toolpath (EM10-Profile-NoComp) and from the local menu select Settings to open the Curve Profile dialog.



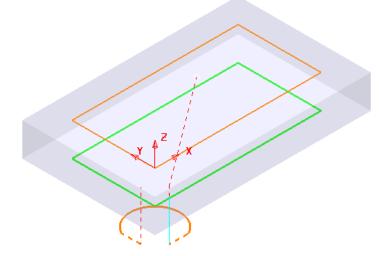
- 29 Once the Curve Profile dialog is open select the Make a Copy option.
- **30 Rename** the new toolpath as **EM10-Profile-FullComp**.
- 31 Select the **Cutter Compensation** page, tick the **CNC cutter compensation** box, and select **Type** as **Full radius**.

đ		Curve Profile	?	×
Тоо	lpath name	EM10-Profile-FullComp		
Workplane Block Machine tool Curve profile Curve profile Curv	^	Cutter compensation Type Full radius Minimum radius 0.	1	

- 32 Calculate the new Curve Profile toolpath, but do not close the form.
- 33 To test the validity of the applied tool compensation the **Toolpath** toolbar needs to be active (**Main** pulldown menu:- **View Toolbar Toolpath**).

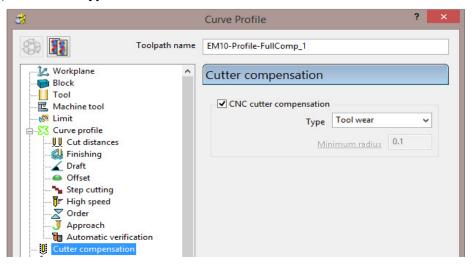


34 In the **Toolpath** toolbar select the **Draw compensated toolpath** option.

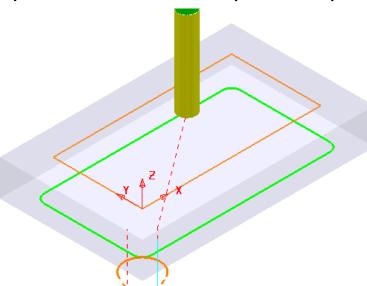


If **Full Compensation** is successfully applied it is displayed as a **thicker** green toolpath but offset inwards to the same **XY** dimensions as the **Pattern** (If the toolpath is displayed in red then the compensation is not geometrically valid).

- 35 Select the Make a Copy option again.
- **36 Rename** the new toolpath as **EM10-Profile-WearComp**.
- **37** Select the **Cutter Compensation** page, tick the **CNC cutter compensation** box, and select **Type** as **Tool wear**.



- 38 Calculate the new Curve Profile toolpath, and then Close the dialog.
- 39 In the **Toolpath** toolbar select the **Draw compensated toolpath** option.



If **Wear Compensation** is successfully applied it is displayed as a **thicker** green version of the original toolpath.

#### 40 Save Project As:-

.....\COURSEWORK\PowerMILL-Projects\ToolCompExample

#### Viewing the NCData output files for a heid controller

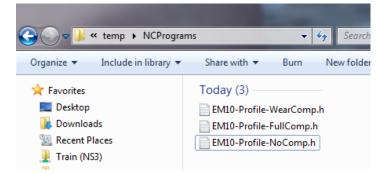
41 In the PowerMILL explorer, right mouse click on NC Programs and from the local menu select Preferences.

MC Preferences	? ×
Output Toolpath Fixture offset	
Changes made here will not change existing NC Programs	
Use Project Off   Output Folder C:\Temp\NCPrograms	
Output File {ncprogram}	
Machine Option File C:\dcam\config\ductpost\heid.opt	
Output Workplane Part Name	
Tool Value Tip	
Automatic Tool Alignment On 👻 Connection Moves Simultar	neous 👻
Close	

- 42 Open the folder containing the *Machine Option File* and from the list select the file heid.opt and click the Open tab.
- 43 Close the NC Preferences dialog.
- 44 In the **PowerMILL** *explorer* right mouse click on **Toolpaths** and from the local menu select **Create Individual NC Programs**.
- 45 Right mouse click on NC Programs and from the local menu select Write All.
- 46 Minimise the PowerMILL window to access the *Desktop* options and select Windows explorer (Or for *Windows 8* use File explorer).



47 Once *Windows explorer* has opened browse to C:\temp\NCPrograms.



- 48 In Windows explorer, right mouse click on an NCdata output file and from the local menu select Open with – WordPad.
- 49 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

7 L X-4,267 Y-24,201 RL F400 M

9 C X+0,000 Y+0,000 DR+ RL F M

15 C X-24,201 Y-4,267 DR+ RL F M 16 L X-23,896 Y-15,443 RO F M

3 L Z+10,000 R0 F9999 M 5 L Z+5,000 R0 F M

6 L Z-20,000 R0 F40 M90

8 CC X-10,607 Y-10,607

10 L Y+50,000 RL F M

12 L Y+0,000 RL F M

13 L X+0,000 RL F M

11 L X+100,000 RL F M

14 CC X-10,607 Y-10,607

17 L Z+10,000 R0 F9999 M

#### **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 3 L 2+10,000 R0 F9999 M 4 L X-15,443 Y-23,896 R0 F9999 M03 5 L 2+5,000 R0 F M 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

			l		
		-	2	0	
		G	0	r	
-	-	3	*		

18 STOP M02

19 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.

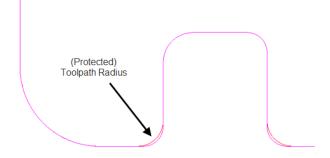
#### 50 Save Project As:-

.....\COURSEWORK\PowerMILL-Projects\ToolCompExample

#### **Protected Cutter Compensation**

CNC cutter compensation	
Туре	Full radius 👻
Min	imum radiu: 0.1
	$\sim$

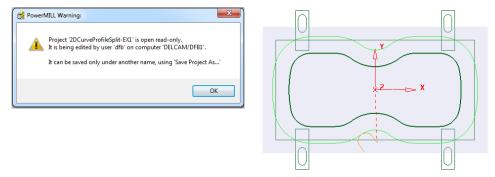
With Type - Full radius selected any internal (concave) corners will require a suitable Minimum radius value included to allow for corner radii that are less than the required full offset value.



#### 2D Curve Machining Example – Inc. Clamp Avoidance

- 1 Select File > Delete All and Tools > Reset forms.
- 2 Open Project:

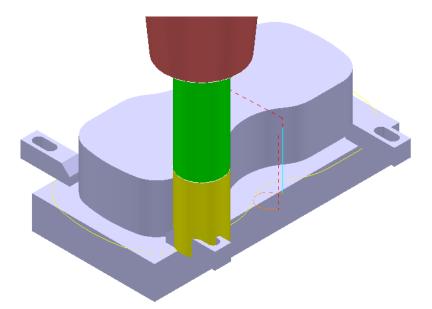
#### .....\PowerMILL\_Data\Projects\2DCurveProfileSplit-Start



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**:

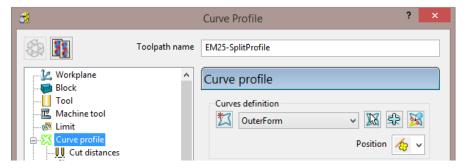
```
.....\COURSEWORK\PowerMILL-Projects\2DCurveProfileSplit-EX1
```



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.
- From the Main pulldown menus select, Draw Cursor Tool and Draw Cursor Cross Hair (Or use the shortcuts Crtl T and Ctrl H).
- 6 Ensure that the **Pattern OuterForm** is displayed (**PowerMILL** *explorer*, light bulb on).

- 7 Right click on the toolpath **EM25-FullProfile** and from the local menu ensure **Activate** is ticked, and then select **Settings**.
- 8 Once the **Curve Profile** form is open select the **Make a Copy** option.
- **9 Rename** the new toolpath as **EM25-SplitProfile**.



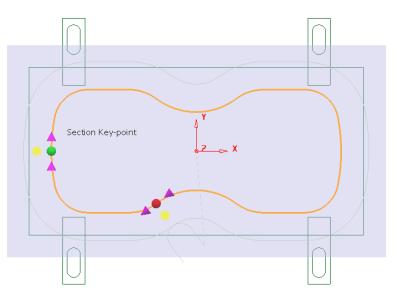
10 On the Curve Profile page select Interactively modify machinable sections.



11 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.

Edit Individual Machining Section	
Add Section	
Invert Machining Sections Split Section	
Reverse Machining Side	
Change Cut Direction	
Delete	

12 From the local menu select **Split Section** and an extra *start and end* point will be created on the **Pattern** at this point.

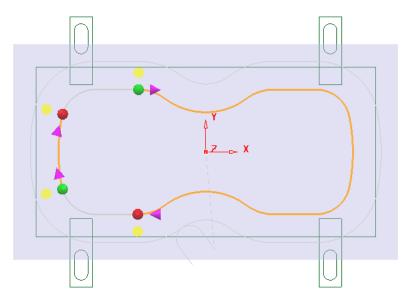


13 From the main pull down menus select Draw – Cursor – Cross Hair (Ctrl H).

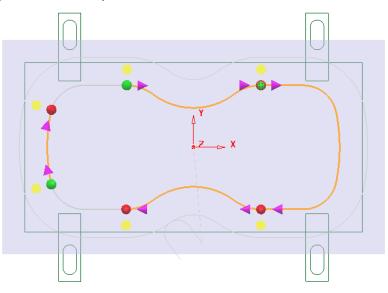


The cross hair will be displayed on the cursor to assist with alignment of start and end point.

14 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).



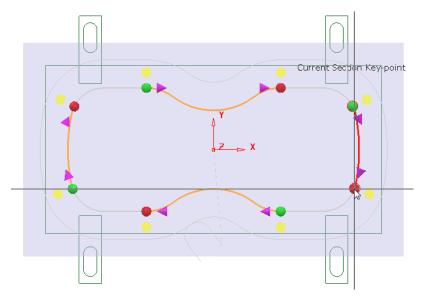
- 15 Locate and drag the corresponding *red* end point to a position to the right of the lower left clamp (as shown above).
- 16 Repeat the process on the other (Left hand) span and drag the start and end points to the left hand side of the **Pattern** (as shown above).
- **17** Split the toolpath again twice, to the left of both the upper and lower right clamps (as shown below).



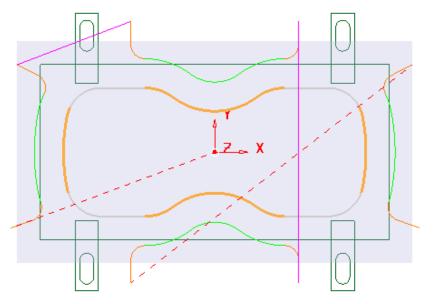
**18** Move the new start and end points to form 4 distinct spans along the **Pattern** as shown in the following diagram.



For accurate alignment use the displayed cross hairs and key points.



- Once the 4 new spans have been aligned, select the green tick in the Edit Machining Sections toolbar to accept the modifications.
- 20 Back in the **Curve Profile machining** form select **Calculate** to process the new EM25-SplitProfile toolpath and then **Close** the dialog.

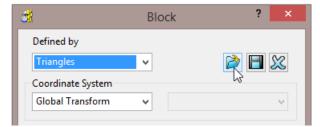


To check visually check the toolpath for collisions, a triangle model of the stock along with the clamps will be imported and used as the **ViewMILL** block.

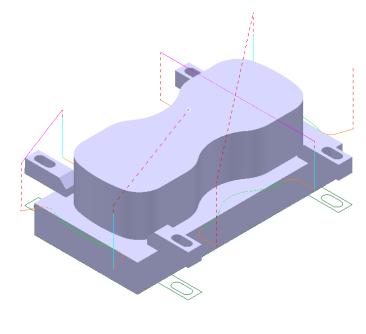
- 21 Select an ISO1 view.
- 22 Right mouse click on the toolpath **EM25-SplitProfile** in the **PowerMILL** *explorer* and from the local menu select **Simulate from start**.
- 23 Open the Block form and select Defined by Triangles and Coordinate System Global Transform (See diagram on next page).
- 24 Select the Load block from file option 24 and browse to the model:-

....\PowerMILL\_Data\Triangles\2DCurveprofileSplit.dmt

25 Accept the Block form.

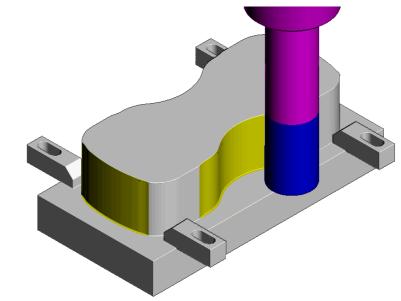


The Block will appear as shown below (Opacity slider set to full))





26 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. 27 Select **File** - **Save** to update the stored **Project**.

# **13. NC Programs**

## Introduction

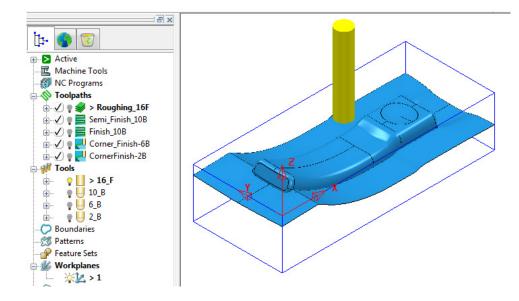
Once a series of **toolpaths** have been generated their names can be listed in an **NC Program** file to be post processed as an output file for a specific **NC machine control**. Any number of **toolpaths** can be included and reordered as required depending on the limitations of the particular **NC 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

#### **Defined PowerMILL Paths – NC Program output**

- 4 From the top pulldown menu bar, select **Tools > Customise Paths**.
- 5 In the **PowerMILL Paths** dialog, select **NC Program Output** followed by **Add** path to top of list

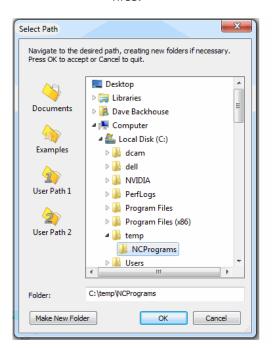
If not already defined, it is now required to set up a path to where the **Post Processed** files (**NCPrograms**) are to be stored.

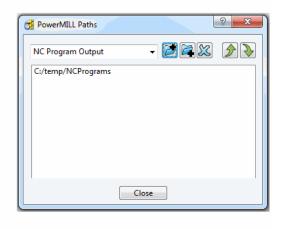
6	🕺 PowerMILL Paths			? ×
	NC Program Output	-		<b>∌ ≥</b>
	Macro Paths		$\smile$	
	NC Program Output			
	File Dialog Examples Button	15		
	Default Directory			
	File Dialog Button 1	I		

6 In the **PowerMILL Paths** dialog, Select *NC Program Output* and (if it is not already displayed) add the new path **C:\Temp\NCPrograms**.



*If this method is used the destination folder for* **NCPrograms** *must exist on your pc otherwise it will not be possible to output the* **NCProgram** *files.* 





- 7 Click **OK** to accept and close the above dialog.
- 8 Close the **PowerMILL Paths** dialogue.

### **Use Project On - NC Program output**

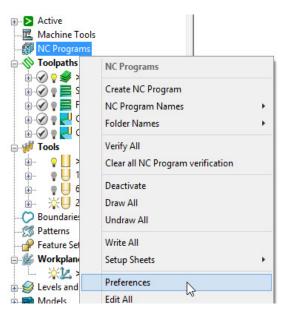
An alternative method to control the destination for **NCdata** output is to use the **active Project** as the default, destination for outputting **NC Data** files. In the **NCPrograms** -**Preferences** form, switch **Use Project** to **ON**.

<b>3</b>		NC Preferences	? ×
Output	Toolpath Fixture offs	et	
Chang	es made here will not o	hange existing NC Programs	
	Use Project Off	Output Folder C:\Training_Data\COURSEWORI	
	On Off	ut File {ncprogram}	

On applying the **Write** command on an *NC Program* a folder named **NC Programs** will be created in the currently active **Project** folder. This will not work if a **PowerMILL Project** has not yet been created (**Save Project**).

## **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.

<del>3</del>	NC Preferences ? ×
Output	Toolpath Fixture offset Verification
Chang	ges made here will not change existing NC Programs
	Use Project Off 🗸 Output Folder C:\Temp\NCPrograms
	Output File {ncprogram}
	Machine Option File C:\dcam\config\ductpost\heid400.opt
	Machine Tool v Model Location 1 v
0	utput Workplane 1 Part Name
	Tool Value Tip 🗸
	Automatic Tool Alignment On v Connection Moves Simultaneous v
	Close

3 From the Select Machine Option Filename dialog (below), select the heid426.opt and click the Open tab.

0	Look in: 🚺 ductpost 🗸	G 🤌 📂 🛄 🗸			
<b>V</b>	Name	Date modified	Туре	Size	
~	🗏 heid.opt	23/10/1998 14:25	OPT File	1 KB	
20	heid400.opt	17/12/2010 09:46	OPT File	1 KB	
	🛋 heid426.opt 📐	24/10/2000 13:47	OPT File	1 KB	
~	Heid530.or Type: OPT File	13/07/2004 13:24	OPT File	17 KB	
1)	heidiso.opt Size: 38 bytes	23/10/1998 14:25	OPT File	1 KB	
	HermleC20 Date modified: 24/10/2000 13:47	07/10/2009 13:49	OPT File	41 KB	
~	hurco.opt	23/10/1998 14:25	OPT File	1 KB	
2	Hurco_VMC60SR_WinMax_V8.pmoptz	30/05/2012 13:57	PMOPTZ File	12 KB	
	Hurco_VMX60SR_WinMax_V8.pmoptz	31/05/2012 17:17	PMOPTZ File	13 KB	

The selected option file translates the output to the correct Machine Tool output format.



It is not unknown 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**.

NC Programs		
⊡ ·· 📎 Toolpaths	NC Programs	
i 🖉 🛊 🔁 s	Create NC Program	
🖶 🗹 🤋 🧮 F	NC Program Names	+
🖻 🗹 🖥 🗖 🤇	Folder Names	+

		NC	Progra	im : Jo	b_646_T	ор		· · ·	>
		Name Job	0_646_To	р					y
	Outp	out File C:\	Temp\N	ICProgra	ams\{ncp	rogram}			P
Ma	achine Optic	on File C:\	dcam\co	onfig\du	uctpost\h	eid400.opt		~	P
	Mad	chine Tool			~	Model Location	n		
	Output \	Workplane	1		~	Part Name	= 1		
		Program	n Numbe	er 1		Tool Value	Tip		۰,
	Autom	natic Tool A	lignmen	nt On	V Cor	nnection Move:	Simulta	neous	
Toolpath	Num	Diame	Tip	Gauge	Overha.	. Toleran	Thickn	Tool II	) /
<	Tool Chan	ge On Ne	w Tool	~	Т	ool Numbering	As Speci	> fied	
<	Tool Chang Reset	ge On Ne	w Tool	~		ool Numbering hange Position		fied	,
		ge On Ne	w Tool	~		-		fied	`
Toolpath Tool			<b>w Tool</b> Gauge		Tool C	hange Position		fied	
Toolpath Tool	Reset		Gauge	: Length	Tool C	hange Position	After Co	fied	`
Toolpath Tool N	Reset umber 12 ensation		Gauge Off ↓	: Length	Tool C	hange Position	After Co	fied nnection	
Toolpath Tool N	Reset 12 umber 12	Length	Gauge Off v	: Length	Tool C 80.0	hange Position ID Radius None	After Co	fied nnection	

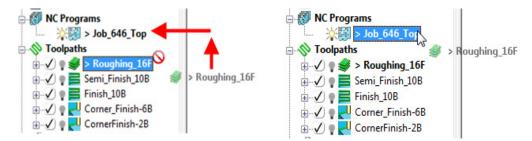
2 Enter the name **Job\_646\_Top** and click **Accept**.

3 In the **PowerMILL** *explorer* select the +/- box by **NC Programs** to toggle the display of the existing **NCPrograms**.

► International State S

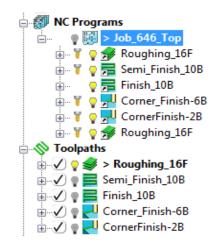
One or more **Toolpaths** can be added to the active, **NC Program**.

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.



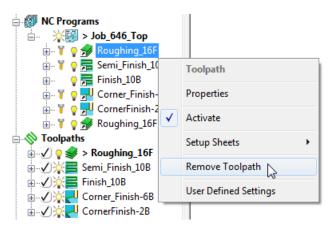
### Add to – NC Program (alternative method)

It is also possible to add several toolpaths to the active **NC Program**. In the **PowerMILL explorer**, select one or more **toolpaths** and then *Right Mouse Click* on one to open the local **Toolpaths** menu.

iand the second	Toolpaths	(4)	
🗄 🗸 🛉 🔁 CornerFin	Draw Selected		
⊡	Undraw Selected		
	Add to	•	Stock Model
in ♥ U 10_B	Create Individual NC Programs		NC Program
Boundaries	Edit	•	Group Folder
Patterns	Delete Toolpaths		Toldel

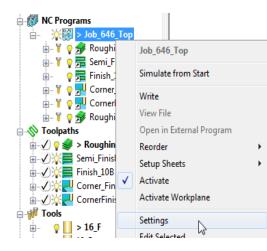
Select **Add to** – **NC Program** and all of the selected toolpaths will be added to the **NC Program** list.

7 Right-click on 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.



Õ

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.

		NC	Prog	ram : Jol	b_64	46_To	р		?	×
		Name Job	o_646_T	ор						89
	Outp	out File C:\	Temp\	NCProgra	ams\	(ncpro	gram}			
Mach	nine Onti	on File C:\	dcam\	confia\du	ictpo	st\heid	400.opt		~	2
		chine Tool		-	~	105	odel Location			
					~	IVI				~
	Output	Workplane	1		~		Part Name	1		
		Program	n Numl	ber 1			Tool Value	Tip		~
	Auton	natic Tool A	lignme	ent On	v	Conn	ection Moves	Simulta	neous	~
Toolpath	Num	Diame	Tip	Gauge	Ove	erha	Toleran	Thickn	Tool IE	
Roughing_16F	12	16	np	80	011			1	16 F	
Semi_Finish_1		10	5	50				0.4	10 B	
Finish_10B	(14)	10	5	50			0.01	0	10_B	
Corner_Finish	7	6	3	30			0.02	0	6_B	~
Т	ool Chan	ge On Ne	w Tool	~		Тоо	I Numbering	As Speci	fied	~
Ka 🐻 🕞	leset				Т	ool Cha	inge Position	After Co	nnection	v
Toolpath										
Tool										200
Tool Nun	nber 12		Gaug	ge Length	80.	0	ID	16_F		
Cutter Compen	sation									
		Length	Off	v 0		R	adius None	v	0	
									_	
		Drilling Cyc		out On			Coolant	None		
					~		Coolant	None		×
		Fixture	offset	None	~	<b>1</b>			1	bc 23
	0	utput File								
	144.55									
	Writ	e	Apply	/	Acc	cept	Close			

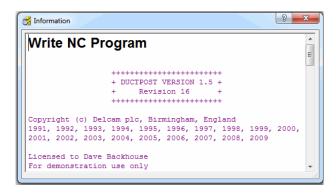
Several of the settings assigned to a specific toolpath can be changed in the above dialog. The toolpath row is selected and relevant values (Tool Number, Coolant, etc) are then modified before clicking **Apply**. These local edits do not affect the original toolpath.

0

#### 9 Click Accept.

10 Right-click on the **NC Program**, **Job\_646\_Top** and select **Write** to create the Post-Processed output file.

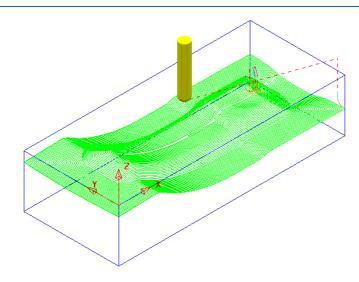
During the translation an **Information** window appears containing a progress report.



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 Activate the existing Dia 10 Ball nose tool named 10\_B.
- **13** Calculate a **3D** Offset Finishing toolpath, named **3Doffset-test1** using the settings and values as shown below.

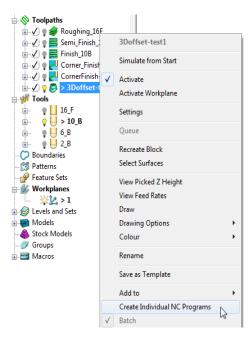
31	D Offset Finishing ? ×
Toolpath name	3Doffset-test1
Workplane         Block         Tool         Wachine axis control         Rapid move heights         Useds and links	3D offset finishing         Pattern         Image: Start on pattern         Image: Spiral         Image: Smoothing         Image: Maximum offsets         Maximum number of offsets         10
- Image: Start point         - Image: End point         - Image: Feeds and speeds         - Image: Feeds and speeds         - Image: History         - Image: Provide the settings         Viser defined settings	Tolerance Cut direction   0.02 Climb   Thickness   Image: Constraint of the original statement of the original statem



Ò

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.

14 Right-click on the toolpath named 3Doffset-test1 and select Create Individual NC Programs.



15 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).





The NCProgram icon for 3Doffset-test1 will remain blue until the output file is processed (Write). The icon will then turn to green.

13. NC Programs

## 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.

⊢	
🖃 🔆 🔯 3Doffse	Job_646_Top
i⊞… ¥ ♀ ऴ 3Do ⊡ <b>⊗ Toolpaths</b>	Simulate from Start
⊕ √ ♥ ≇ Roughi ⊕ √ ♥ ≣ Semi_Fi	Write
$\oplus                                    $	View File
🗄 🗸 🍦 🗾 Corner_	Open in External Program

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.
- Right-click on one of the NC-Data files and from the local menu, select Open
   With > Wordpad.

Job_646_Top.tap - WordPad	_ <b>_</b> X
Home View	
🖬 여 🧐 📼	
	11 12 13
0 BEGIN PGM 1 MM	
1 TOOL DEF 12 L+0,000 R+0,000	
2 TOOL CALL 12 Z S1500,000	
3 L Z+10,000 R0 F9999 M	
4 L X-8,440 Y+0,000 R0 F9999 M03 5 L Z+5,000 R0 F M	
6 L Z-5,000 R0 F500 M90	
7 L X+0,000 R0 F M	
8 L X+215,028 R0 F M90	
9 L Y+7,507 R0 F M	
10 L X+0,000 R0 F M	
11 L Y+15,014 R0 F M	
12 L X+215,028 R0 F M	
13 L Y+22,522 R0 F M	
14 L X+54,537 R0 F M	
15 L X+55,800 Y+23,617 R0 F M	
16 L X+57,120 Y+25,428 R0 F M	
17 L X+57,584 Y+26,180 R0 F M	
18 L X+58,152 Y+27,211 R0 F M	
19 L X+58,747 Y+28,994 RO F M	
20 L X+58,930 Y+30,029 R0 F M	
21 L X+215,028 R0 F M	
22 L Y+37,536 R0 F M 23 L X+58,889 R0 F M	
23 L X+58,889 R0 F M 24 L X+58,634 Y+41,477 R0 F M	
25 L X+58,559 Y+43,260 R0 F M	
26 L X+58,506 Y+45,043 R0 F M	
27 L X+215,028 R0 F M	
•	•
100% (=) -	
100%	~

3 From the main (top) toolbar select File > Save to update the Project followed by Close.

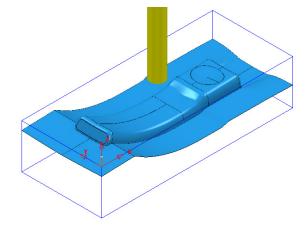
# **14. Setup Sheets**

## Introduction

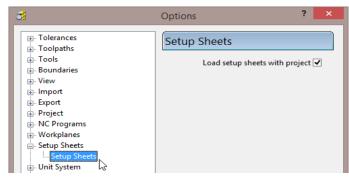
All the Setup data required from a **PowerMILL Project** such as **cutter definitions**, **images** and **information** relating to **toolpaths** can be downloaded as **HTML files** and then printed out as required. This is essential for the CNC Operators who will be running the programs.

- 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 From the main pulldown menus select **File – Project Settings** to open the following dialogue.

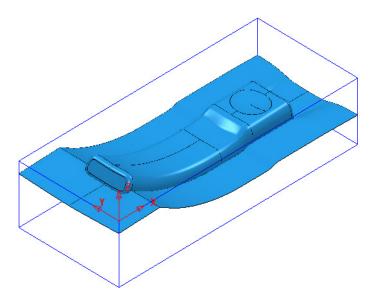
Project Settings	1	t Settings		
Project Settings	User Defined Settings	Notes		
Date		Customer		
2015-09-07		ME		
Programmer		Part Name		_
dfb		IT		
Project Name	2	Order Number		_
SetupSheet-	ex1	A-123789		1
Project Direct	ory			
C:/Training_	Data/COURSEWORK/Pn	nill2016-Projects/SetupSheet-e	x1 🏠	
C:/Training_ <	Data/COURSEWORK/Pn	nill2016-Projects/SetupSheet-e	x1 0	
<	Data/COURSEWORK/Pn	nill2016-Projects/SetupSheet-e		
<				
<		nill2016-Projects/SetupSheet-e MILL2013PRO/handset-2.dgk	>	
< CAD Model D:/users/dfb			>	
<			>	
< CAD Model D:/users/dfb			>	

7 Input **Customer**, **Part Name**, and **Order Number** details as shown above and then **Close** the dialogue.

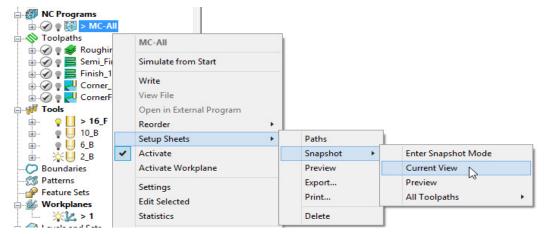


There are 2 other tabs for the user to input additional information as **User Defined Settings** and **Project Notes**.

- 8 **Activate** one of the toolpaths to reinstate the original setup (Block, Workplane, Rapid Move Heights, etc).
- 9 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.



10 Right-click on the *NC Program* named MC-All and from the local menu select Setup Sheets > Project 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.

- 11 Undraw the **model** leaving the **Workplane** and *wireframe* **Block** as the only displayed items.
- 12 Right-click on the NC Program MC-All again and select Setup Sheets > Snapshot > All Toolpaths > Iso View.

NC Programs	(					
	MC-AII					
⊞ (⊘) ♀ == ♥ ⊕ (⊘) ♀ == ♥	Simulate from Start					
Toolpaths	Write View File Open in External Program Reorder	•				
🗄 🕜 🛊 🧮 Finis	Setup Sheets	•	Paths			
E · · · · · · · · · · · · · · · · · · ·	Activate		Snapshot 🕨	Enter Snapshot Mode		
🗄 🛷 🧛 🚽 Corr	Activate Workplane		Preview	Current View		
	Settings		Export	Preview		
	Edit Selected		Print	All Toolpaths	•	Current View
	Statistics		Delete			Current State View
Boundaries	Verify					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.

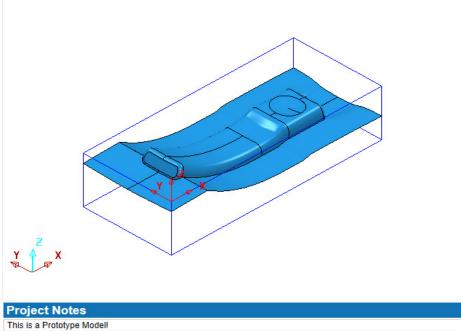
The **PowerMILL Setup Sheets: Index** is displayed on the left side of the screen.

PowerMILL Setup Sheets: Index
Project Setup Sheet-ex1
NC Program MC-All
Toolpath Roughing 16F
Toolpath Semi Finish 10B
Toolpath Finish 10B
Toolpath Corner Finish-6B
<u>Toolpath CornerFinish-2B</u>
<u>Summary Page 1</u>
Summary Page 2
Full Summary Page 1
Full Summary Page 2
View the log
Back Forwards

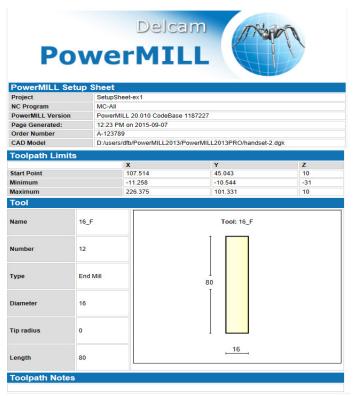
14 In the above *Index* dialog left click on **Project Setup Sheet-ex1** to display the following sheet.

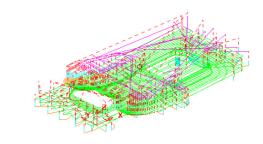
Once a page has been selected the text will change from red to blue.

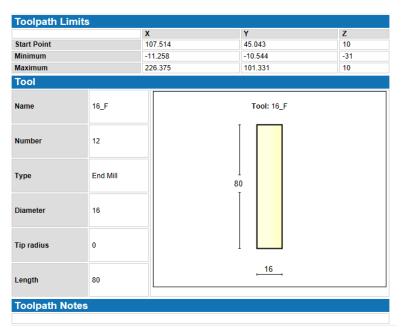
		Delcam MILL
PowerMILL Setu	ip Sheet	
Project		SetupSheet-ex1
PowerMILL Version		PowerMILL 20.010 CodeBase 1187227
Page Generated:		12:23 PM on 2015-09-07
Project data		
Customer	ME	
Part Name	IT	
Order Number	A-123789	
CAD Model	D:/users/dfb	/PowerMILL2013/PowerMILL2013PRO/handset-2.dgk
	3:54:06	



**15** From the **Index** page select the **Toolpath** and **Summary** pages in turn and view the content.









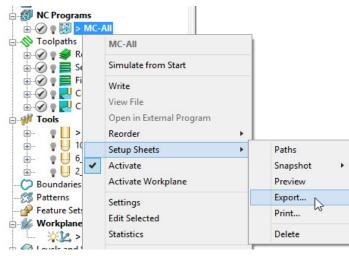
*Each HTML page can be printed out as individual items. Right-click on the page and from the local menu, select Print Picture.* 

The 1<sup>st</sup> sheet of the **Summary Page** is as shown below.

Delcam PowerMILL Setup Sheet					
Project	SetupSheet-	ex1			
PowerMILL Version		0.010 CodeBase 1187227			
Page Generated:	12:23 PM on	2015-09-07			
Summary sheet 1 of 2					
Toolpath / TAP File	NC Program	Strategy	Tool		
	no rrogram	onatogy	Type	End Mill	
		Raster Area Clearance	Diameter	16	
Roughing_16F	MC-All		Tip radius	0	
C:/Temp/NCPrograms/MC-All.h			Length	80	
			Number	12	
			Туре	Ball Nosed	
		Raster	Diameter	10	
Semi_Finish_10B C:/Temp/NCPrograms/MC-All.h	MC-All		Tip radius	5	
C./Temp/NCFTograms/MC-Ait.ii			Length	50	
			Number	14	
			Туре	Ball Nosed	
			Diameter	10	
Finish_10B C:/Temp/NCPrograms/MC-All.h	MC-All	Raster	Tip radius	5	
C./Temp/NorTograms/MC-Ait.in			Length	50	
			Number	(14)	
			Туре	Ball Nosed	
12 12711 22			Diameter	6	
Corner_Finish-6B C:/Temp/NCPrograms/MC-All.h	MC-All	Along Corner	Tip radius	3	
en en prior regramorno All.II			Length	30	
			Number	7	



- **16** Return to the display to show the **PowerMILL** *explorer* again.
- 17 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.

### **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.

=	3
	1
_	9

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.

NC Programs			
🖽 - 🔗 💡 🚀 ¥ Rou	MC-All		
🗄 🕜 💡 📻 Finish	Simulate from Start		
ti () or the second secon	Write View File		
Toolpaths	Open in External Program		
in	Reorder Setup Sheets	Paths N	- 6
BØ ♥ ➡ Corner_F BØ ♥ ➡ CornerFir	Activate	Snapshot	+
E Tools	Activate Workplane	Preview	
	Settings Edit Selected Statistics	Export Print Delete	
🖻 ··· 🤋 📙 2_B	Statistics	Delete	

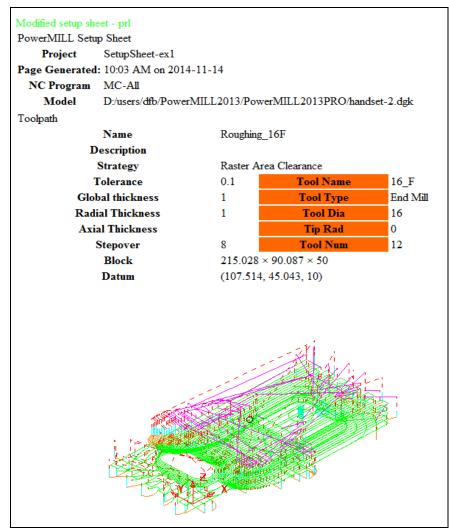
2 On the **Paths** tab select the folder icon in the **Toolpath** > **Write** field.

<del>3</del>	Setup Sheets	?	×
Paths Option	2		
Export File	:h}/ncprograms/{project}.html		3
(projectipat	······································	Ľ	
Templates			
NC Program			
✓ Write	Header.html	Z	2
Toolpath		1	$\sim$
✓ Write	Toolpath.html		≥)
NC Program	Summary		
Vrite	Summary.html		2
Project Head	ler		
✓ Write	Project_Header.html		2
Project Sum	mary		
✓ Write	Project_Summary.html		2
	Close		

3 Browse to ......\PowerMILL\_Data\setup-sheets and Open the Toolpath2.html file (customised template).

Look in:	]] setup-sheets	v 🕝 🤌 📂 🛄 🗸		
Name	^	Date modified	Туре	Size
Head	er.html	22/11/2012 10:58	HTML Document	5 KB
🧧 Projec	t_Header.html	22/11/2012 10:58	HTML Document	3 KB
🧧 🦉 Projec	t_Summary.html	22/11/2012 10:58	HTML Document	3 KB
Sumn	nary.html	22/11/2012 10:58	HTML Document	4 KB
🖉 Toolp	ath.html	22/11/2012 10:58	HTML Document	7 KB
🦲 Toolp	ath2.html	06/09/2005 10:29	HTML Document	6 KB
>	Type: HTML Docu Size: 5.42 KB Date modified: 06/			

- 4 Click **Close** on the **Setup Sheets** dialog.
- 5 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.
- 6 On the **Index** page select the new page for the toolpath **Roughing\_16F** (Shown below)

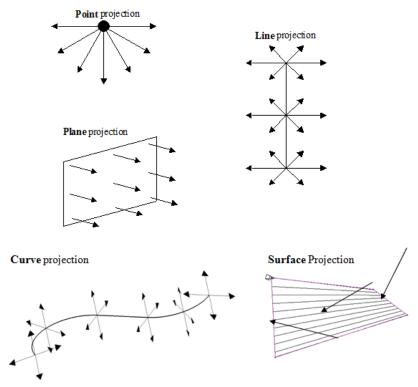


# **15. Specialist Finishing**

## **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** also provides essential strategies 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 (Only the areas visible to the theoretical light source will be machined).

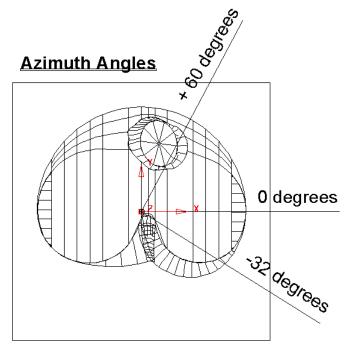
- A **Point** strategy is similar to light radiating from a single light bulb.
- A **Plane** strategy is similar to light radiating from a rectangular floodlight.
- A **Line** strategy is similar to light radiating from a fluorescent tube.

- 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.

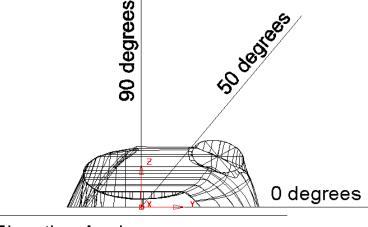
In the **Projection Finishing** forms, angular definitions are input using **Azimuth** and **Elevation**. By using a combination of the two any complex 3D angle can be defined.

 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



#### **Elevation Angles**

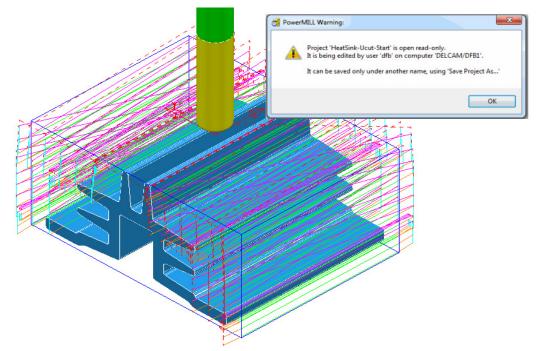
This is shown on the chamber model, with a view looking down the X-axis.

## **Plane Projection Example**

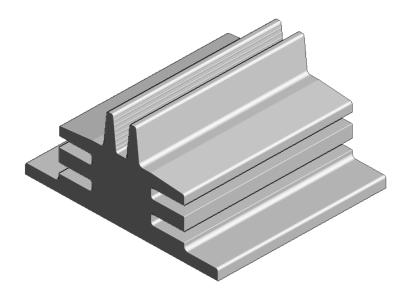
All five **Projection Finishing** strategies can be used to machine undercut features if used with suitable tooling such as Disc or Spherical. The following example shows an undercut application using the **Plane Projection** strategy.

- 1 Select File Delete All and Tools Reset Forms.
- 2 Select File > Open Project:

.....\PowerMILL\_Data\Projects\Heatsink-Ucut2-Start



- **3** Save the Project As:
  - .....\COURSEWORK\PowerMILL\_Projects\ Heatsink-Ucut2-EX1
- 4 Open the **Block** form and with **Defined by Box** and **Type Model** set select **Calculate**.
- 5 Simulate the NCProgram (all of the existing toolpaths) in ViewMILL.



- 6 Return the ViewMILL simulation to the No-Image view
- 7 Disconnect **ViewMILL** from the toolpath simulation



- 8 Activate the D75-H6-T2 Disc Cutter.
- 9 In the **Block** dialog *lock* all 4 of the **X** and **Z** values.
- 10 Calculate the Y values of the Block with an Expansion value of 40.

<del>3</del>	Block	? ×
Defined by		
Box	¥	🔌 🖪 💥
Coordinate System		
Active Workplane	¥	~
Limits		
Min	Max	Length
X -0.0	150.0	<b>150.0</b>
Y -127.0	125.0	252.0
z -87.5	0.0	87.5
<b>X</b>	X X	<b>C</b>
Estimate limits		
Tolerance	<b>Expansion</b>	Туре
0.1	40.0	Model 🗸

11 In the **Strategy Selector** - **Finishing** dialog, select **Projection Plane Finishing** and fill in the pages of the dialog exactly as shown below.

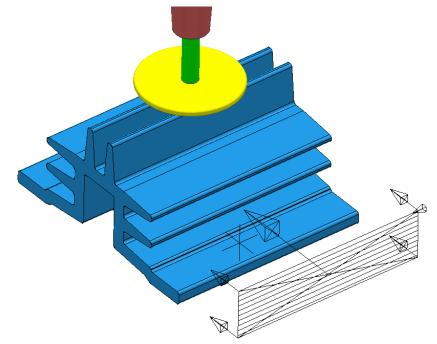
Plane Projection Finishing				
Toolpath name	D75H6T2-SEM1-Ucut			
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Plane projection</li> <li>Pattern</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Plane projection   Location   0.0   -150.0   0.0     Azimuth   0.0   Elevation   0.0   Projection   Direction   Outwards     Tolerance   0.1   Thickness   0.5   Stepover   1.5	~		
Mon	Preview Draw			

The **Location** is the local datum for the projection strategy. All other coordinates entered in the dialog are relative to this point.

<b>3</b>	Plane	Projection Finish	ing			?	×
D 1	Toolpath name	D75H6T2-SEM1-Uco	ut				
Workplane Block Jool E Machine too	I	Pattern					
<sup>®</sup> Limit <sup>®</sup> Stock engage ⊖… <sup>®</sup> Plane project <sup>®</sup> Pattern <sup>®</sup> Automati	ion		Patterr Ordering Sequence	One way None	U	> > >	
Point distribution     Point distribution     Point distribution     Point distribution     Tool axis     Machine axis control     Point     Leads and links     Start point     Feeds and speeds	Limits Height Start -40.2 Width <u>Start</u> 75.0		End -77.0				
History		P	review D	)raw			

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.

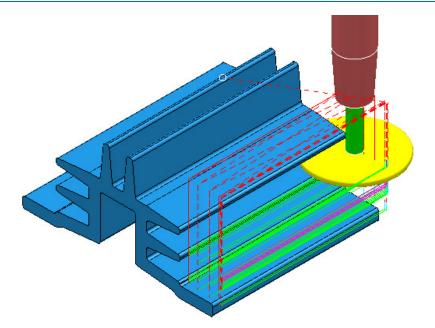
12 Select **Preview** to view the **Plane Projection** pattern.



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.

 Image: Symbol Note
 Toolpaths

 Image: Symbol Note
 Image: Symbol Note

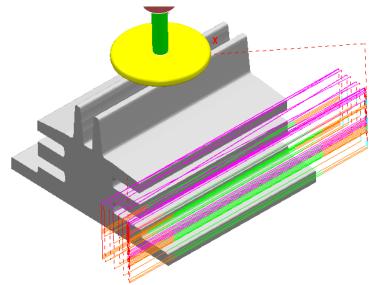
 Image: Symbol Note

In this case suitable  $\ensuremath{\textbf{Leads}}$  and  $\ensuremath{\textbf{Links}}$  will be applied to easily rectify the problem.

1 Open the **Leads and Links** dialog and select the **Lead in** tab.

<del>3</del> 4	Le	ads and Lin	ks		?	×	
Extensions		Links	Poi	Point distribution			
Z heights First lea	id in	Lead in	Lead out	Last	Last lead out		
1st choice		2nd o	hoice				
Straight		✓ Nor	ie			~	
Distan	<u>ce</u> 45.0			<u>Distance</u>	0.0		
Ang	<u>le</u> 0.0			<u>Angle</u>	0.0		
Radii	us <b>0.0</b>			<u>Radius</u>	0.0		
0.0       Overlap distance (tool diameter units)       Ramp options         Image: Allow start points to be moved       Image: Add leads to short links         Image: Add leads at tool axis discontinuities       Angular threshold         Image: Add leads at tool axis discontinuities       Image: Angular threshold							
Gouge check				Apply lead	d ins		
Ар	ply	Accept	Cancel				

- Input the following in the Lead in page:
   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*.

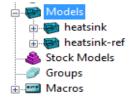
5 Save, but do not close, the **Project** as it is to be continued in the next section.

## **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 **PowerMILL explorer**. This will be useful when selecting it as the **reference surface** to be used with the **Surface Projection** dialog.

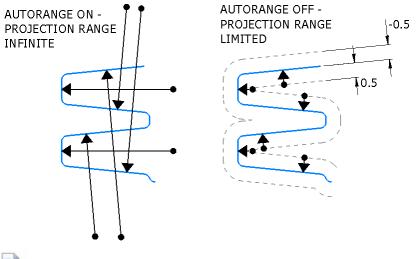
2 From the main pull down menus select View - Toolbar - Command to open the command input window below the graphics area and type in the following 3 lines:-

EDIT SURFPROJ AUTORANGE OFF

#### **EDIT SURFPROJ RANGEMIN –1**

#### **EDIT SURFPROJ RANGEMAX 1**

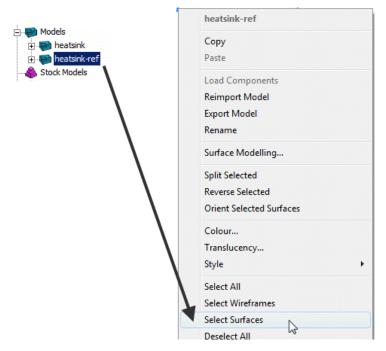
This is to restrict the tool projection distance to within 0.5mm of the reference surface. Otherwise it would be projected from infinity onto non-undercut faces on the outside of the model.



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**.

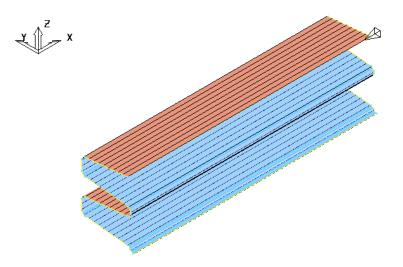


5 In the **Surface Projection Finishing** dialog input the settings and data exactly as shown below.

💰 Surfac	e Projection Finishing ? ×
Toolpath name	D75H6T2-Ucut
Workplane Block Block Kachine tool Kachine	Surface projection          Surface       Surface units       Distance         Smoothing tolerance       0.02         Angular smoothing tolerance       0.003744         Projection       Direction       Inwards         Tolerance       0.02       Inwards         Thickness       0.0       Stepover (Distance)         Image:
	Preview Draw

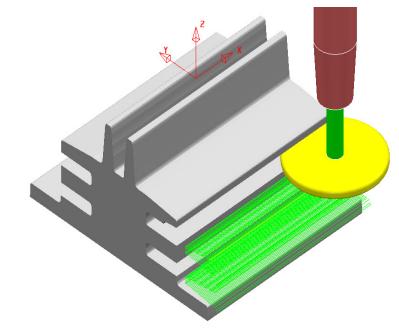
🛃 Surfa	ace Projection Finishing	?	×
Toolpath nam	e D75H6T2-FIN-Ucut		
Workplane	Pattern		
	Pattern Pattern direction	U v	-
Surface projection     Pattern     Automatic verification	Ordering One way	Spiral 🗌	-
Point distribution	Start corner Max U ma	xV √	
Machine axis control	Sequence None	~	<u>'</u>

- 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 consistent **stepover** across all of the undercut areas.

**10** Select **File** > **Save** to update the stored **Project**.

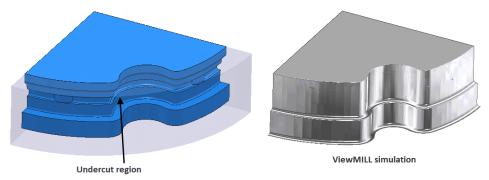
#### **Line Projection Example**

- 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.

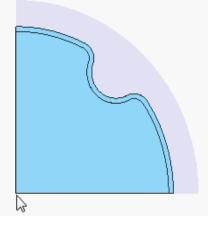
The imported **Project** (training\_ucut\_proj) contains a **Model**, **Tooling**, **Roughing**, and **Semi-Finishing** strategies with **1** and **0.3 thickness** values respectively. The undercut regions have not yet been machined.



5 Activate the toolpath (Rough) to reinstate the settings and then simulate both existing toolpaths in ViewMILL.

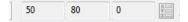
**Tipped Disc** cutters will be used for machining the undercut areas. The toolpath will be created by projecting a suitable strategy inwards, towards a vertical line, located at the lower, left corner.

- 6 Select a **View from Top (Z)**.
- 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 **Projection Line Finishing**.
- **10** Fill in the pages of the strategy dialog exactly as shown below.

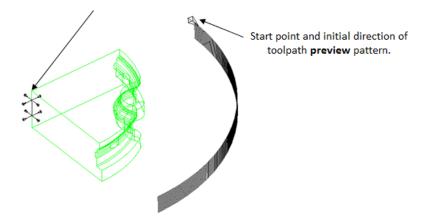
💰 Line	e Projection Finishing	? ×
Toolpath name	TD30-Undercut	
2. Workplane	Line projection	
	Pattern Style Circular	~
Stock engagement	Location	
⊡ <b>∕⁄o Line projection</b> 	50.0 80.0 0.0	X X
Point distribution     Control Action	Azimuth 0.0	
<ul> <li>→ Machine axis control</li> <li>→ Rapid move heights</li> <li>→ U Leads and links</li> </ul>	Elevation 0.0	
	Projection Direction Inwards	~
Notes and Description	Tolerance 0.1	
	Thickness 0.3	<u>U</u>
	Stepover	
Restaurs	Preview Draw	
	Calculate Queue OK Cancel	

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.

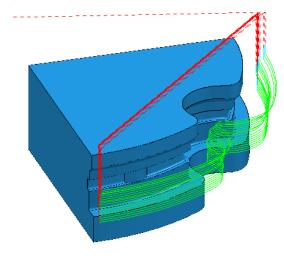
💰 Line I	Projection Finishing	?	×
Toolpath name	TD30-Undercut		
Workplane Block Block Colored Kachine tool Kachine to	Pattern Pattern Pattern Style Ordering Sequence Limits Azimuth angle Start 90.0 Height Start -10.0	Circular v One way v None v	
-27 History Notes and Description 	Preview D	raw	

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.

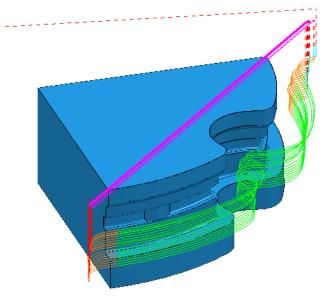
×		Lead	s and Linl	ks	? ×
Extensior	ıs	Lir	nks	Point	distribution
Z heights	First lead in	n	Lead in	Lead out	Last lead out
1st choice			2nd c	hoice	
Horizontal arc			Non	e	~
	Distance	5.0		]	Distance 0.0
	Angle	90.0			Angle 0.0
	Radius	10.0			Radius 0.0

14 Select the **Lead In** tab.

**15** Input the following information:

1st Choice > Horizontal Arc Distance > 5 Angle > 90 Radius > 10

- **16** Select the **Lead out the same as lead in** tab.
- 17 Click **Apply** and then **Accept** to close the dialog.



<u>і</u> т

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

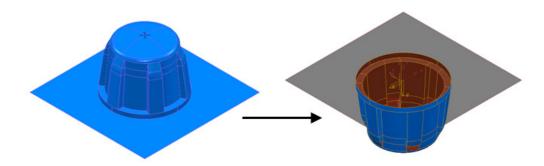
4 In the **PowerMILL** explorer, right-click Workplane 1 and from the local menu select Activate followed by Workplane Editor to display the following toolbar:

|--|

5 From the toolbar select **Twist about Y** to open the **Twist** dialog.

-	Twist	?	×
	Angle 180	<b>S</b>	
	Accept	Canc	el

- 6 In Angle, enter **180** and click Accept.
- 7 Click the **Green** tick to accept the changes and close the toolbar.



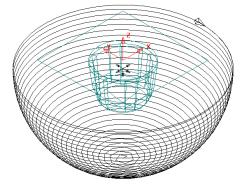
The **Workplane** is rotated **180** degrees about **Y** to provide vertical machining access to the inner wall. However, machining is currently impossible due to the pocket being shielded by an untrimmed planer **surface** (shown part shaded). This example will apply **Component Thickness** options 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.

🥳 Point	Projection Finishing
Toolpath name	BN16-PointProj
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Kachine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Point projection</li> <li>Pattern</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Point projection Pattern Style Spiral  Location 0.0 0.0 -11.0 Frojection Projection Direction Outwards  Tolerance 0.1 Thickness O.0 Angular stepover 2.0 Preview Draw
	Calculate Queue OK Cancel

💰 Point	t Projection Finishing	?	×
Toolpath name	BN16-PointProj		
Workplane Block Tool	Pattern		
Machine tool Machine tool Stock engagement Point projection Automatic verification Point distribution Machine axis control Rapid move heights	Pattern Style Spiral Direction Anticlockwise Limits Elevation angle <u>Start</u> 0.0 <u>End</u> 90.0	~	
<ul> <li>I Leads and links</li> <li>Start Point</li> <li>T End point</li> <li>T Feeds and speeds</li> </ul>	Preview Draw		

12 Fill in the page of the dialog exactly as shown above and select **Preview** to observe the **Point Projection** setup.

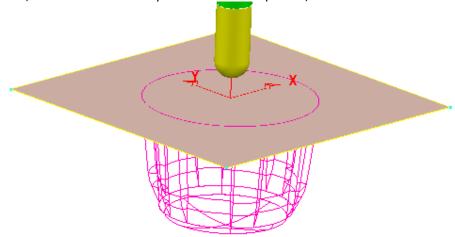


13 In the main page of the **Point Projection Finishing** dialog select the

**Component Thickness** icon and in the dialog select the **Surfaces** tab.

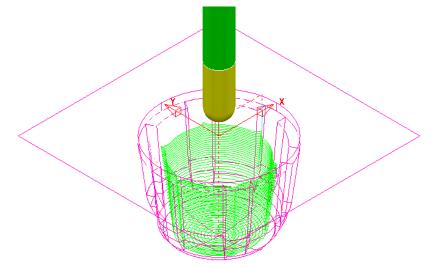
urfaces	Verification	Surface Def	faults			
Entity				Clor	ne	
11	♥ BN16-Poi	ntProj	~	5	• •	~ 🛐
*	1		Use Axial	Thickness		<u> </u>
Machi	ning Mode			Thickness		
Ignore	• •			0.0		E
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#
		0	-	0	-	0
0	Machine					
心0 心1	Machine Machine	0	-	0	1.	0
		0	-	0 0		0 0
10	Machine		-		- - -	-
< €1 €2	Machine Machine	0		0	- - -	0
<ul> <li>№1</li> <li>№2</li> <li>№3</li> </ul>	Machine Machine Ignore	0		0	- - - -	0
©1 ©2 ©3 ©4	Machine Machine Ignore Machine	0 0 0 0		0 0 0	-	0 1 0
©1 ©2 ©4 ©5	Machine Machine Ignore Machine Machine	0 0 0 0		0 0 0 0	-	0 1 0 0
©1 ©2 ©4 ©4 ©5 ©6	Machine Machine Ignore Machine Machine Machine	0 0 0 0 0		0 0 0 0 0	•	0 1 0 0 0

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.





- 15 Click the Acquire Components icon to add the selected, 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**

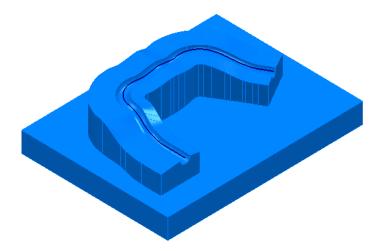
- 1 Select File > Delete All and Tools > Reset forms.
- 2 Select File Import Model and select:

```
.....\PowerMILL_Data\Models\ crvproj_3D.dgk
```

(Note the model also includes a wireframe curve running centrally along the slot).

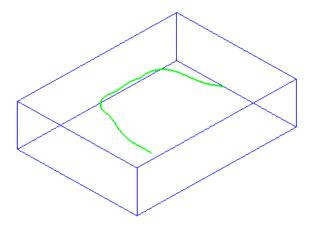
3 Select File > Save Project As:-

.....\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* (coloured green) running centrally along the slot

on the **Model** (Note; The **Wireframe** view must be active for the curve to be visible).



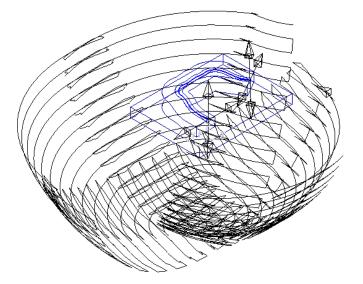
- 9 In the **PowerMILL** *explorer*, right-click on the new **Pattern** (1) and in the local menu select **Insert** > **Model**.
- **10** From the **Strategy Selector** dialog, select **Projection Curve Finishing**.

11 Fill in the page of the dialog exactly as shown below and select **Preview** to observe the *Curve Projection* setup.

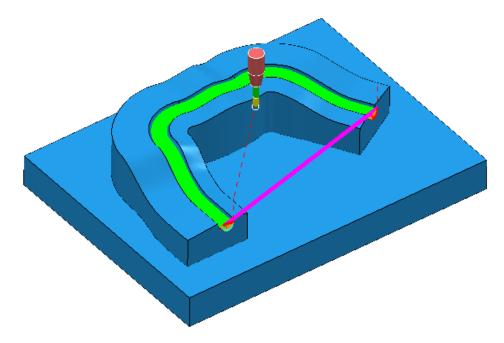
💰 Curve	e Projection Finishing 🛛 📍 🗙
Curve	BN16-ProjCurve Curve projection Curves definition Inclusion Style Linear Projection
<ul> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Direction Outwards
	Calculate Queue OK Cancel

Curve	e Projection Finishing		?	×
Toolpath name	BN16-ProjCurve			
Workplane     Block     Tool     Tool     Machine tool     Winit     Stock engagement     Curve projection <u>Curve projection     Pattern     Automatic verification     Automatic verification     </u>	Pattern Pattern Style Ordering Sequence	Linear One way None	~	
Point distribution 2 Tool axis 2 Machine axis control 2 Rapid move heights 3 Leads and links	Limits Azimuth angle <u>Start</u> -75.0	End 75.0	]	
	Parametric pattern limits Start 0.0	End 1.0		
User defined settings	Preview 🗸 D	raw		

Visually the **preview** 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**.

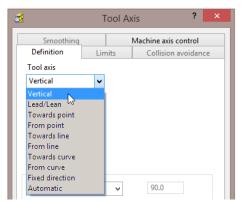
**13** Select **File** > **Save** to update the stored **Project**.

Õ

## **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.

For 3 Axis applications the tool axis **I** must be set to **Vertical**.

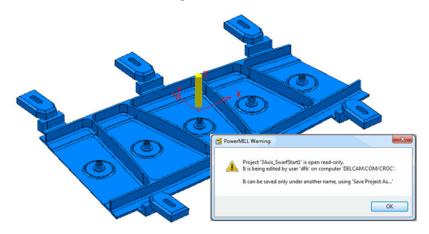


The strategy can be set to create either single (default) or multiple passes, exactly following the upper edge, lower edge, or both (merging).

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

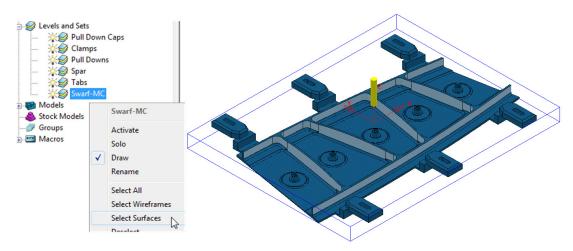


3 From File select Save Project As:

.....\COURSEWORK\PowerMILL-Projects\3AxisSwarfExample

- 4 In the **PowerMILL** *explorer*, activate the **workplane** *mc-Datum*.
- 5 In the **PowerMILL** explorer, activate the Dia 16 End Mill (EM16).
- 6 Calculate a **Block** using **Defined by Box** and **Type Model**.
- 7 Calculate the **Rapid Move Heights** using the default settings.

- 8 In the **PowerMILL** *explorer*, click the [+] adjacent to **Levels and Sets** to access all of the existing **Levels**.
- 9 Right-click on the Level Swarf-MC and click Select Surfaces from the local menu.



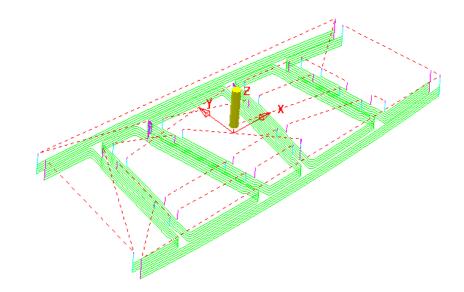
All **Surfaces** on **Level** *Swarf-MC* will be selected (light blue) to be included in the **Swarf** strategy.

- 10 In the Strategy Selector, select the Finishing tab, and then select the Swarf Finishing strategy.
- 11 Enter data exactly as shown in the following 2 pages of the dialog.

Image: Stock engagement   Image: Stoc	<b>3</b>	Swarf Finishing ? ×
Block   Tool   Machine tool   Stock engagement   Swarf finishing   Statiple cuts   Filiph speed   Automatic verification   Statiplic cuts   Sta	Toolpath name	EM16-Swarf_OffsetUp
Thickness	Block	Drive curve Surface side Outside V Radial offset 0.0 Minimum fanning distance 0.0 Fan at end on planes Reverse axis Follow surface laterals Surface joining tolerance 0.3 Gouge avoidance Gouge check Degouge tolerance 0.3 Tolerance Cut direction 0.02 Climb V

1		Swarf Finishing		?	
(2)	Toolpath name	EM16-Swarf_OffsetUp			
Vorkplane		Multiple cuts			
<mark>  </mark> Tool <u>IZ</u> Machine tool <u>K</u> Limit		Mode	Offset up		*
💕 Stock engagemen	t	Order by	Region		~
Swarf finishing		Upper limit	Тор		~
	nce	Workplane			~
<mark> </mark> Multiple cuts ¶∓ High speed			<u>Offset</u>	0.0	
🔚 Automatic ver		Maximum number of cuts			
Point distribution				10	
Machine axis cont → Machine axis cont → Rapid move heigh → ↓↓ Leads and links → ↓↓ Start point		Maximum stepdown			

12 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**.

13 In the **PowerMILL** *explorer*, right-click the **toolpath** *EM16-Swarf\_OffsetUp* and select **Settings**.



- 14 Select the *Copy Toolpath* icon from the dialog
  15 Rename the new Toolpath as EM16-Swarf\_Merge.
- 16 In the page labelled Multiple Cuts change the Mode to Merge and with all surfaces on the Level Swarf-mc selected, Calculate the strategy.

		Swarf Finishing	?
\$2 II	Toolpath name	EM16-Swarf_Merge	
- 💯 Workplane - 🍯 Block		Multiple cuts	
<mark>  </mark> Tool <b>E</b> Machine tool		Mode	Merge 🗸
	t	Order by	Region 🗸
Swarf finishing		Upper limit	Top 🗸
	nce	Workplane	~
Multiple cuts F High speed			Offset 0.0
Automatic ver	ification	Maximum number of cuts	
Point distribution			10
Machine axis cont		Maximum stepdown	
	ts	5.0	
			1
		Y Contraction of the second se	

*Merge* will use a variable *stepdown* to exactly track the *selected surfaces*, upper and lower edges and minimize the number of lift moves.

17 Right-click the last toolpath *EM16-Swarf\_Merge*, in the **PowerMILL** *explorer* and select Settings.

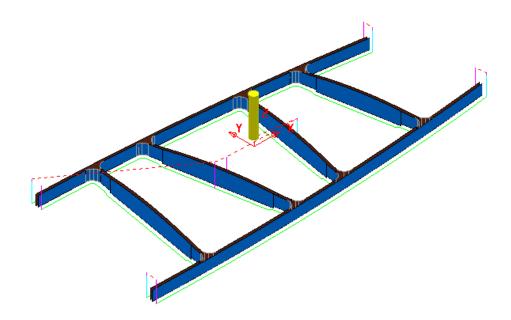


- 18 Select the **Copy Toolpath**
- **19 Rename** the new **toolpath** as **EM16-Swarf\_SinglePass**.
- 20 On the Multiple Cuts page, change the Mode to Off.

-0

<del>3</del>		Swarf Finishing		?		x
	Toolpath name	EM16-SinglePass				
🦢 Workplane 🍯 Block		Multiple cuts				
Tool Machine tool		Mode	Off		*	
	nt	Order by	Region		~	
🖃 🚮 Swarf finishing		Upper limit	Тор		~	
Position	nce	Workplane			~	
<mark>F</mark> High speed			<u>Offset</u>	0.0		
🔚 Automatic ver		Maximum number of cuts				
Point distribution				10		
Machine axis cont Rapid move heigh Leads and links Start point		Maximum stepdown				

21 With all **surfaces** on the **Level** *Swarf-mc* selected, **Calculate** the strategy.



- 22 Close the **Swarf Finishing** dialog.
- 23 Select File > Save to update the stored Project.

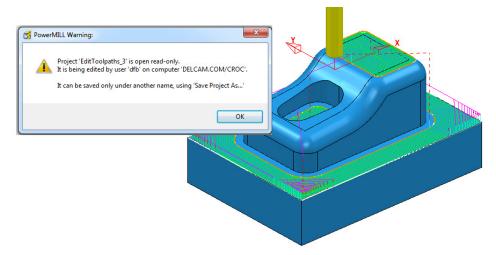
.....\COURSEWORK\PowerMILL-Projects\3AxisSwarfExample

#### **Parametric Offset Finishing**

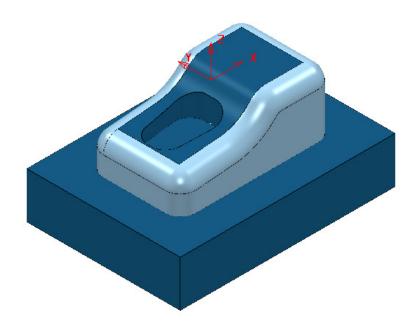
**Parametric Offset Finishing** creates a strategy that tracks between two individual **Patterns** with a stepover that adjusts to the shape of the *selected surfaces*. The tool tracks can be created parallel (along) or span between (across) the **Pattern** segments.

- 1 Select File > Delete All and Tools > Reset forms.
- 2 **Open** the read-only **Project**:

.....\PowerMILL\_Data\Projects\EditToolpaths\_3

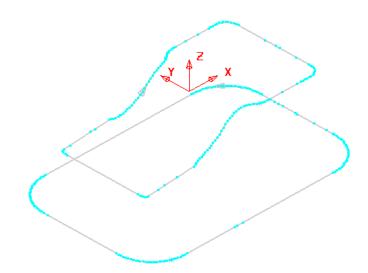


- 3 Click OK in the *PowerMILL Warning* dialog and Save Project As: .....\COURSEWORK\PowerMILL-Projects\ParaOffset-ex1
- 4 Activate the Dia 16 Rad 3 Tip Radiused tool named D16TR3.
- 5 With the new **tool** *active*, and the **surfaces** shown (below left) selected, create a **Selected Surface Boundary**.



🕉 Selected Surf	ace Boundary ? ×
Name Name	2
Top Roll Over Tolerances 0.01 Thickness 0.0 Axial Thickness 0.0	Limit Boundary Inside Limiting Boundary Outside
Use Axial Thickness Use Axial Thickness	Automatic Collision Checking Holder Clearance 0.0 Shank Clearance 0.0 Block
Private Allow boundary to be private Edit History Apply edit history on calculation	() () ()
Apply Queue	Accept Cancel

- 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.

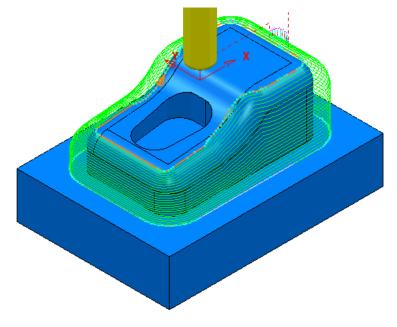


It is not necessary for both of the **Pattern** segments to run in the same direction as the **Upper** one will control the toolpath direction.

- 8 Make sure that no **Boundaries** are **Active**.
- 9 Open the **Parametric Offset Finishing** dialog and fill in as shown below.

di Param	netric Offset Finishing ? ×
Toolpath name	D16TR3-FIN
Workplane     Block     Tool     Machine tool     Limit     Stock engagement     Parametric offset finishing	Parametric offset finishing Start curve Start curve End curve
<ul> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> </ul>	Lower       Image: Construction of the second
	Maximum number of offsets 10 Tolerance Cut direction 0.01 Climb V Thickness
	Maximum Stepover 2.0
	Calculate Queue OK Cancel

**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.

### **Parametric Spiral Machining**

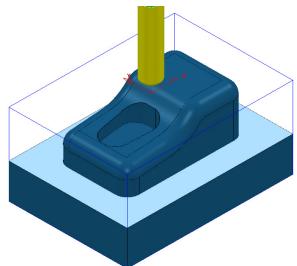
This strategy starts from a **Pattern** and merges a *spiral toolpath* towards an **Outer limit** which is either defined by the **Block** or to one or more **surfaces** acquired to a **Level** or **Set**.

	Parametric spiral	
Tool      Machine tool      Stock engagement      Parametric spiral      Point distribution      Tool axis      Machine axis control      Rapid move heights      Leads and links      Start point      End point      Feeds and speeds	Central curve	▼ I None>
Viser defined settings	V Pocket Tolerance 0.01	Direction Anticlockwise

The direction of the toolpath created in the **Parametric Spiral Finishing** strategy is controlled by the **Cut Direction** *Anticlockwise/Clockwise*. The direction of the central **Pattern** has no control over the toolpath direction.

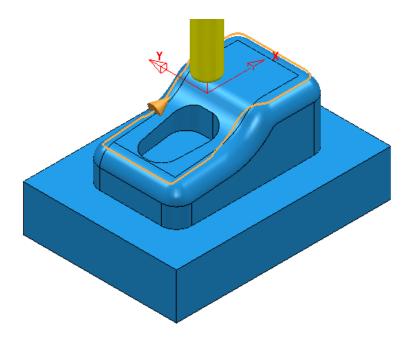
If the **Pocket** option is *ticked* the resultant **toolpath** will start at the **Outer limit**.

1 In the **PowerMILL** *explorer* right mouse click on **Levels and Sets** and from the local menu select **Create Set**.

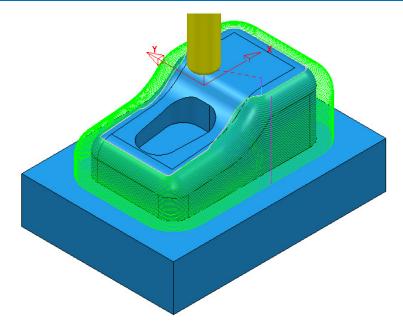


- 2 Select the upper base **Surface** and from the new **Set** (1), local menu select **Acquire Selected Model Geometry**.
- 3 Make sure that no **Boundaries** are **Active** or displayed.
- 4 Open the **Parametric Spiral Finishing** dialog and fill in as shown below.

😚 Param	netric Spiral Finishing ? 🛛 🗙
Toolpath name	D16TR3-Spiral-FIN
Workplane  Block  Glock  Gloc	Parametric spiral Central curve Central curve Upper Vice Limit Undercut Angle 0.0
Start point     End point     Feeds and speeds     History     Notes and Description     User defined settings	0.0       Direction         0.01       Anticlockwise         Thickness       Image: 0.0
	Maximum stepover 2.0 Calculate Queue OK Cancel



5 Select **Calculate** to create the **toolpath** then **Close** the dialog.



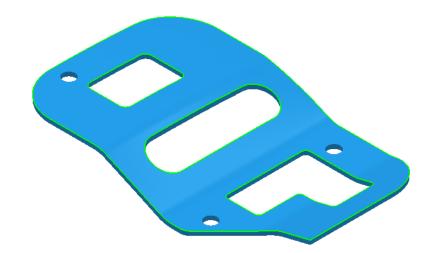
6 Select File > Save to update the stored **Project**.

#### 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

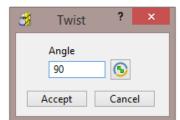
- 4 Create a Dia 5 End Mill named EM5.
- 5 Create an empty **Pattern**, right-click on it, and from the local menu, select **Insert** > **File**:

.....\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.

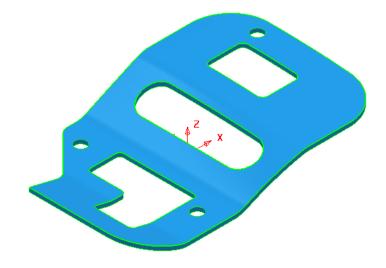
	Name MC-Datum	1	X 🖉 🕅	14 1 <u>2</u>	📌 🕅	6
	2	2				
11 Click	Twist about Z 🗳	칠 to open	the following	g dialog.		

Copyright © Delcam Ltd



**12** Enter the value **90** and click **Accept**.

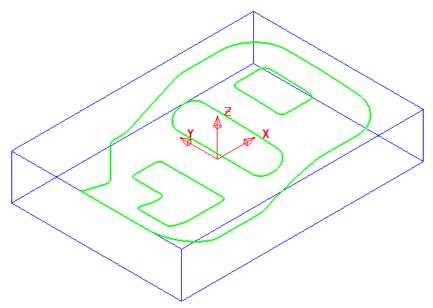
- 13 Click **V** on the **Workplane Editor** toolbar to accept the changes and **Close**.
- 14 Activate the workplane (MC-Datum) and select an ISO 1 view.





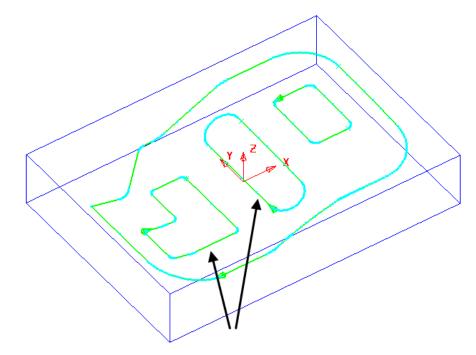
The changes will not be physically active until the **Workplane Editor** toolbar has been **closed**.

- **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.



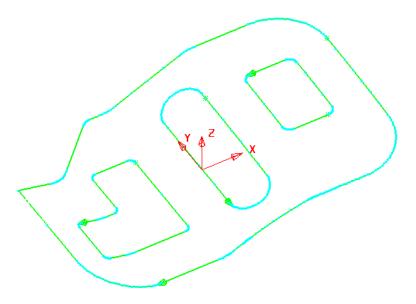
The final task prior to machining is to check the directions of the **Pattern** segments and to **reverse** any that will cause the tool to machine on the wrong side of a profile.

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.

**21** Select the **Wireframe Profile Machining** strategy and enter data exactly as shown on the following 2 pages of the dialog.

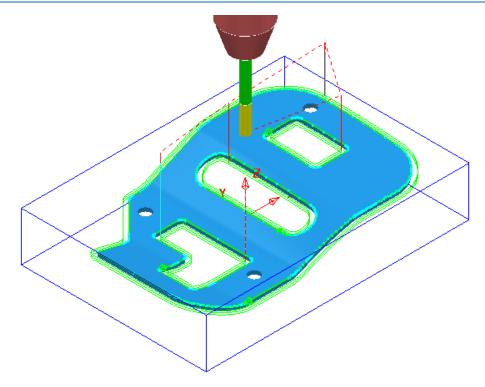
🥶 Wir	reframe Profile Machining ? 🗙
Toolpath na	me EM5-3DProfile
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Multiple cuts</li> <li>High speed</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Wireframe profile machining         Drive curve         Image: Solution curve side         Lower limit         Base position         Curve side         Lower limit         Base position         Curve         Axial offset         -3.0         Gouge avoidance         Gouge check         Degouge tolerance         0.0         Curve thickness         0.0         Preview       Draw

**Curve side** controls which side of the **Pattern** segment direction the tool operates.

Axial offset controls the depth of machining below the wireframe curves.

*	Wirefra	ame Profile Machining		?	>
ي ا	Toolpath name	EM5-3DProfile			
Workpla	ne	Multiple cuts			
Tool I Machine	tool	Mode	Offset down	Ŷ	]
💕 Stock en		Order by	Region	¥	]
T T	ge avoidance		10		
Point dis	matic verification	Upper limit	Type Curve	~	]
	nt links	Maximum stepdown			

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

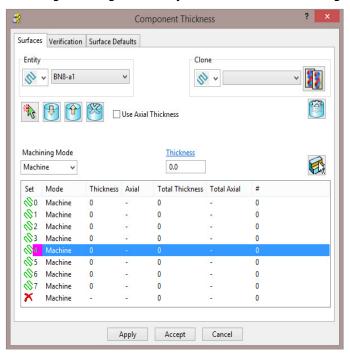


The **Wireframe Profile Machining** performs the profiling of the Pattern segments in 3D.

23 Select File > Save to update the stored Project.

# **16 Component Thickness**

The **Component Thickness** options enable the user to apply varying **Thickness** and **Machining Mode** options to different parts of the component. These can be pre-defined as default settings in the **Thickness Preferences** dialog ready to copy into individual toolpath strategies. They can also be defined independently within individual toolpath strategies using the **Component Thickness** dialog.



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 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.

#### **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
  - PowerMILL Warning: PowerMILL Warning: Project 'CompThic\_Start' is open read-only. It is being edited by user 'dfb' on computer 'DELCAM/DFB1'. It can be saved only under another name, using 'Save Project As...'

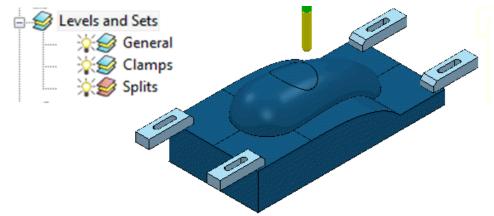
The **Project** contains an unprocessed Interleaved **Constant Z** machining strategy along with a Dia 8 Ball Nosed tool and all essential parameter settings.

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**).

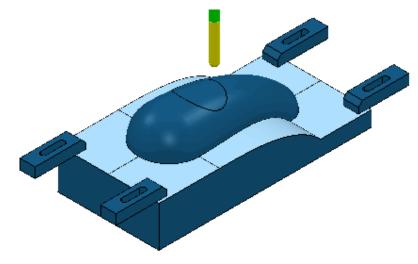
4 In the **PowerMILL** *explorer*, create a new **Level** and rename it as **Clamps** and then create a new **Set** and rename it as **Split**.



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**.

5



- 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.

💰 Steep	and Shallow Finishing	? ×
Toolpath name	BN8-a1 Steep and shallow finis Order Top first v Threshold angle 30.0 Steep V Spiral	shing Additional stock 0.4 Steep shallow overlap 0.0 Shallow Shallow
<ul> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>End point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Cut direction Climb Stepdown () 1.0 () 0.062996 () Calculate using cusp	Cut direction Climb v Stepover 2 1.0 0.062996 Type 3D offset v Smoothing
	Tolerance 0.02 Thickness	
		٩

9 Select **Component Thickness** <sup>U</sup> to open the following form.

	S Verification	n Surface Def	aults				
Entity				Clor	ne		
-			~				90
20	✓ BN8-a1		*	5	×		<b>§§</b>
R.			Lice Avial	Thickness			2
.0	$\bigcirc$		USE AXIdi	THICKNESS			
	in Made			Thisteres			
0.000	ning Mode			Thickness			
Collisi	ion v			2.0			
				2.0			1 AT
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#	
		Thickness 2	Axial		Total Axial -	# 0	
<b>0</b>	Mode			Total Thickness	Total Axial - -	1.55	
Set	Mode Collision	2	828	Total Thickness		0	
< <b>○</b> <b>○</b> 1 <b>○</b> 2	Mode Collision Machine	2 0	828	Total Thickness 2 0		0 0	
< <mark>₿</mark> 0 2</td <td>Mode Collision Machine Machine</td> <td>2 0 0</td> <td>828</td> <td>Total Thickness 2 0 0</td> <td></td> <td>0 0 0</td> <td></td>	Mode Collision Machine Machine	2 0 0	828	Total Thickness 2 0 0		0 0 0	
<sup>0</sup> <sup>0</sup> <sup>0</sup> <sup>1</sup> <sup>2</sup> <sup>2</sup> <sup>3</sup> <sup>4</sup>	Mode Collision Machine Machine Machine	2 0 0 0	828	Total Thickness 2 0 0 0 0		0 0 0 0	
© 0 1 0 2 0 3 0 4 0 5	Mode Collision Machine Machine Machine Machine	2 0 0 0 0	828	Total Thickness 2 0 0 0 0 0 0		0 0 0 0	
© ©1 ©2 ©3 ©4 ©5 ©6	Mode Collision Machine Machine Machine Machine Machine	2 0 0 0 0 0	828	Total Thickness 2 0 0 0 0 0 0 0		0 0 0 0 0 0	
© 0 1 0 2 0 3 0 4 0 5	Mode Collision Machine Machine Machine Machine Machine Machine	2 0 0 0 0 0 0	828	Total Thickness 2 0 0 0 0 0 0 0 0 0 0		0 0 0 0 0 0 0	

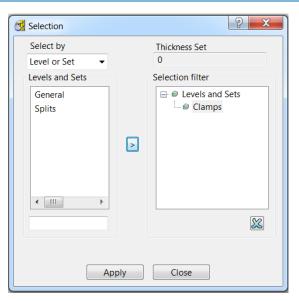
With the **Machining Mode** set to **Collision**, a **Thickness** value of **2** will be applied. At this safe distance the active tool will either *retract* or *profile* around the clamps.

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**.

	Selection	? ×	- 33	Selection	?
Select by Colour Colours	✓ 0 Selection filte		Select by Level or Set v Levels and Sets General Clamps Splits	Thickness Set       0       Selection filter       >	
<	>>		Clamps App	oly Close	2

- 13 Switch Select by from Colour to Level or Set.
- 14 With **Clamps** selected in the **Levels or Sets** box click  $\bowtie$  to acquire the clamp surfaces to the 1st row of the main dialog, and then **Close** the **Selection** dialog.



The 44 *Clamp Surfaces* are acquired to the 1st row.

			Con	nponent Thickne	ess		?	
urfaces	Verification	Surface De	faults					
Entity	♥ BN8-a1		~	Clo	ne		~ 💈	8
*			Use Axial	Thickness				3
Machi	ning Mode			Thickness				2
Collisi	on 🗸			2.0	]		<ul> <li>K</li> </ul>	T
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#		
4.0	Collision	2		2	-	44		
1	Machine	0	8. <del>.</del> .)	0	8558	0		
\$2	Machine	0	-	0		0		
\$3	Machine	0	223	0	222	0		
\$4	Machine	0	-	0	81 <del>-</del> 1	0		
\$5	Machine	0	-	0	1.20	0		
\$6		0	0.70	0		0		
\$7	Machine	0	( <b>1</b> -1)	0	2 <b>-</b> 0	0		
×	Machine	-	<u>121</u>	0	5 <b>2</b> 1	12		

**15 Apply** the **Component Thickness** dialog to update the settings.



*In the above* **Component Thickness** *dialog, it is not necessary to use the 8 available rows in sequential order!* 

**16** Select the 2nd row and a square, colour coded symbol (in this case purple) will appear at the start of the row.

			Cor	nponent Thickne	ess		?	×
Surfaces	Verification	Surface De	aults					
Entity				Clo	ne			
11)	♥ BN8-a1		~	5	> v		~	
*			Use Axia	Thickness			Ľ	3
Machi Machi	ning Mode ne v			Thickness 1.0			<b>F</b>	R
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#		
<b>4 0</b>	Collision	2	-	2	-	44		
<b>4</b> €1	Machine	1		1	- =:	6		
\$2	Machine	0	-	0	-	0		
13	Machine	0	5 <u>1</u> 5	0	5 <u>2</u> 5	0		
\$4	Machine	0	-	0	51 <del></del> 1	0		
\$5	Machine	0	-	0	-	0		
\$6	Machine	0	-	0		0		
\$7	Machine	0	-	0	-	0		
×	Machine	121	-	0	121	6		
			Apply	Accept	Cancel			

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**.

	₫ <mark>8</mark>	Selection ? ×
	Select by Level or Set  Levels and Sets General Clamps	Thickness Set       1       Selection filter       □···· ② Levels and Sets       □··· ③ Splits
	App	Close
18 In Smart Selection	on 🚵 select S	Splits in the Levels or Set

arrow icon  $\square$  to assign them in the second row of the main dialog.

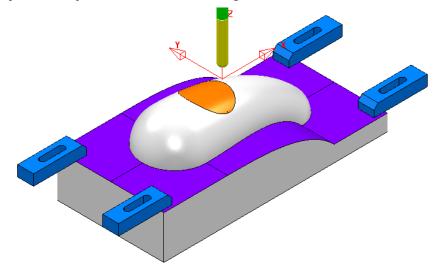
box and click the

**19** Select the 4th row and a square, colour coded symbol (in this case orange) will appear at the start of the row.

infaces	Verification	Surface Det	aults			
Entity				- Clor		
			_	Clot		
12	✓ BN8-a1		~	5	> v	~ 🔰
4		SZ				6-3
R.			Use Axial	Thickness		
Machi	ning Mode			Thickness		
Ignore	×			0.0		
	22					
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#
Set	1.	1.000.000000000000000000000000000000000	Axial		Total Axial	
Set	Collision	2	Axial	Total Thickness 2	Total Axial	44
Set	1.	1.000.000000000000000000000000000000000	Axial - -	2	Total Axial - -	
Set	Collision Machine Machine	2 1	-	2 1	Total Axial - - -	<mark>44</mark> 6
Set	Collision Machine	2 1 0	-	2 1 0	Total Axial	<mark>44</mark> 6 0
Set	Collision Machine Machine Ignore	2 1 0 0	-	2 1 0 0	-	44 6 0
Set	Collision Machine Machine Ignore Machine	2 1 0 0 0	-	2 1 0 0 0	-	44 6 0 1 0
Set	Collision Machine Machine Ignore Machine Machine	2 1 0 0 0 0	-	2 1 0 0 0 0	-	44 6 0 1 0 0
Set Set Set 2 2 3 4 5 6	Collision Machine Machine Ignore Machine Machine Machine	2 1 0 0 0 0 0 0	-	2 1 0 0 0 0 0 0	-	44 6 0 1 0 0 0 0

The *cap surface* must be ignored during the creation of this toolpath.

- 20 With the 4th 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 4<sup>th</sup> row.
- 22 Apply the Component Thickness dialog.



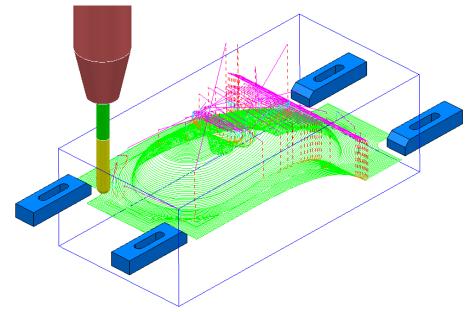
While the **Component Thickness** form is open, **Toolpath Thickness Shade** mode will be active. Acquired **surfaces** are shaded in the same colour as the row.



- **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 <u>Do not</u> close the **Project** as it will be continued in the next section.

# Cloning the Component Thickness settings from an existing Toolpath

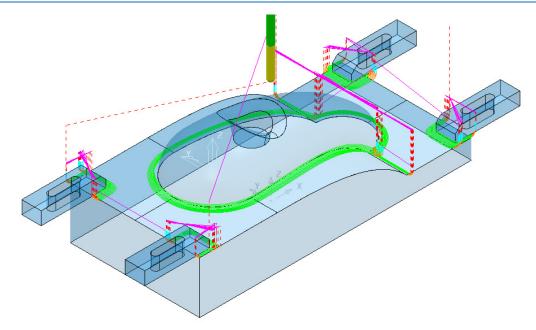
A **Corner Finishing** strategy **BN5-a1** will now be created using 'cloned' **Component Thickness** settings as used in **BN8-a1**.

1 From the **Toolpath Strategies** select the **Finishing** option, **Corner Finishing**, and input settings exactly as shown below:

<del>3</del>	Corner Finishing		? ×
Toolpath name	BN5-a1		
Workplane	Corner finishing		
	Output	Strategy	
	Both v	Along	~
Stock engagement Corner finishing Corner detection	Threshold angle 90.0	Cusp 0.02	
Automatic verification     Point distribution     Yool axis	- 🗌 Maximum passes Maximum num	nber of passes 10	
	Tolerance	Cut direction	
in July Leads and links 	0.02 Thickness	Climb	~

<b>3</b>	Corner Finishing	?	×
Toolpath name	BN5-a1		
Workplane Workplane Block Mochine tool Kachine tool Kachine tool Corner finishing Corner finishing Corner detection Automatic verification Point distribution	Corner detection Reference tool BN8 Use toolpath reference Toolpath	<b>~</b>	
Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of the system       Image: Weight of the system         Image: Weight of th	Overlap 2.0 Detection limit 165 Remove deep		]

2 Select **Calculate** and have a look at the resultant **toolpath** in the graphics area.



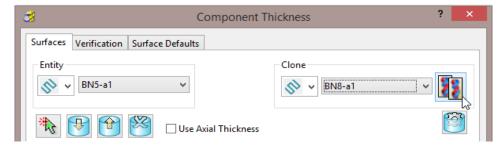
For the moment the **Corner Finishing** strategy is unable to access the base of the central pocket and has also produced unwanted *tool tracks* around the front edge of the clamps. This will be rectified by *cloning* the **Component Thickness** from the toolpath **BN8-a1**.



- Back in the **Corner finishing** page, select **Component Thickness** 3
- 4 In the **Clone** area of the form use the **downward chevron** to display and then select the toolpath Bn8-a1.

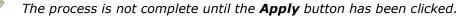
<b>3</b>	Component T	hickness ? ×	
Surfaces	Verification Surface Defaults		
Entity	BN5-a1 Y	Clone SN8-a1 V SN8-a1 V SN8-a1 V	
*	Use Axial Thickness		



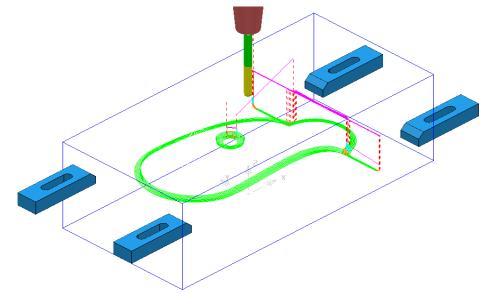


urfaces	s Verification	Surface De				
inace	verification	Surface De	aults			
Entity				Clor	ne	
~	BN5-a1		~		BN8-a1	
110	V DIAD-01		·	5	BING-al	¥
		- <u> </u>				~~~~
×,	$\mathbf{P}$		Ilce Avial	Thickness		123
.0			OJC MAIdi	THICKIIC55		
M	ter Made			Thisteres		
	ning Mode			Thickness	1	
Collisi	ion 🗸			2.0		
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#
2.77	Mode Collision	Thickness 2	Axial -	Total Thickness 2	Total Axial	# 44
<b>1</b> 80	100000					
4 <mark>€</mark> 0 4€1	Collision	2		2		44
4 <mark>€</mark> 0 4€1	Collision Machine	2 1		2 1		<mark>44</mark> 6
<b>4</b> €0 <b>4</b> €1 ≪2 ≪3	Collision Machine Machine	2 1 0		2 1 0		<mark>44</mark> 6 0
	Collision Machine Machine Ignore	2 1 0 0		2 1 0 0		44 6 0 1
	Collision Machine Machine Ignore Machine	2 1 0 0 0		2 1 0 0 0		44 6 0 <b>1</b> 0
<ul> <li>▲</li> <li>↓</li> <li>↓</li></ul>	Collision Machine Machine Ignore Machine Machine	2 1 0 0 0 0		2 1 0 0 0 0		44 6 0 1 0 0
	Collision Machine Machine Ignore Machine Machine Machine	2 1 0 0 0 0 0		2 1 0 0 0 0 0		44 6 0 1 0 0 0
<ul> <li>▲</li> <li>▲</li> <li>↓</li> <li>↓</li></ul>	Collision Machine Machine Ignore Machine Machine Machine Machine	2 1 0 0 0 0 0 0 0		2 1 0 0 0 0 0 0 0		44 6 0 1 0 0 0 0 0

The **Component Thickness** settings originally used in **BN8-a1** are now *cloned* into the strategy dialog for *BN5-a1*.



- 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:-

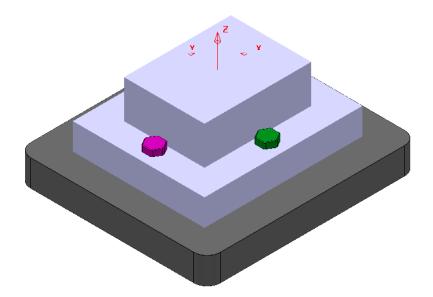
.....\COURSEWORK\PowerMILL-Projects\CompThickness-1

## 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.

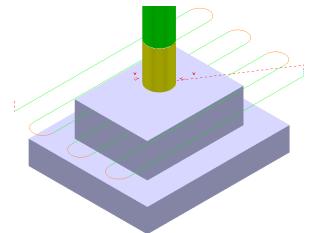
7 From the *main* toolbar select the **Block** dialog.

<b>3</b>	Block	? ×
Defined by		
Triangles	~	🔌 🖪 💥
Coordinate System		
Global Transform	¥	~

8 Select **Defined by** > **Triangles** and **Coordinate System** > **Global** before

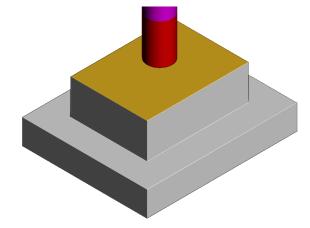
selecting the option Load block from file

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.

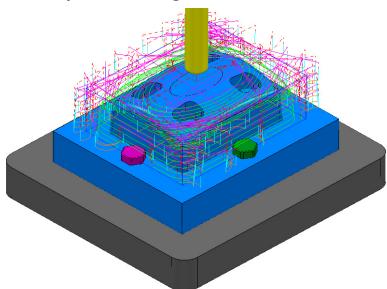


12 Select the **No Image** option **Solution** to return to the **PowerMILL** *window* and for the time being disconnect **ViewMILL** from the **Simulation**.

- 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 bolts 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

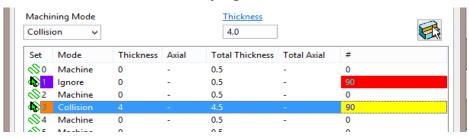
Thickness dialog

19 In the dialog use Smart Selection to acquire the data stored on the Level ClampingBolts-X to one of the colour coded rows.

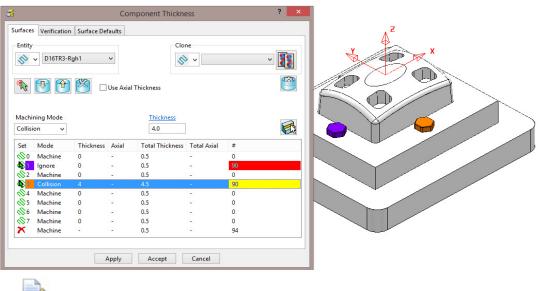
Machi Ignore	ning Mode			<u>Thickness</u> 0.0			
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#	
∞0	Machine	0	-	0.5	-	0	
<b>1</b>	lgnore	0	-	0.5	-	90	
<\\$2	Machine	0	-	0.5	-	0	
€3	Machine	0	-	0.5	-	0	
∕⊗4	Machine	0	-	0.5	-	0	

20 Set the row with *Machining Mode* > Ignore.

21 Select one of the other colour coded rows and use **Smart Selection** to acquire the data stored on the **Level** *ClampingBolts-Y*.

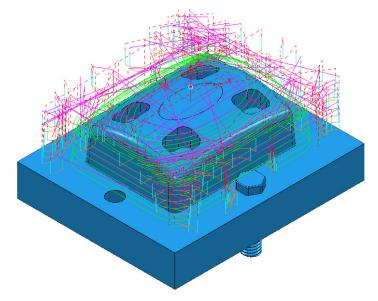


22 Set the row with *Machining Mode* > Collision with a Thickness > 2.0 and then select Apply.



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**.

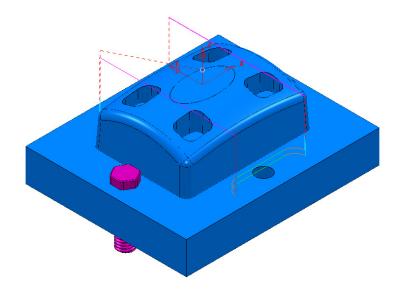
26 In the main page tick the **Rest machining** box and then open the **Rest** page of the dialog.

💰 Mo	del Area Clearance ? ×
Toolpath name	D16TR3-Rgh2
Workplane Block Machine tool Machine tool Model rest area clearance Model rest area clearance Mo	Rest machining Toolpath V D16TR3-Rgh1 V Detect material thicker than 0.5 Expand area by 1.0 Minimum gap length 25.132741
Automatic verification	<b>₹→</b> ∰

- 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).

Machi	ning Mode			Thickness		
Collisi	on 🗸			4.0		
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#
0	Machine	0	-	0.5	-	0
d 🔁 🕹	Collision	4	-	4.5	-	90
<∞2	Machine	0	-	0.5	-	0
<b>4</b> € <mark>3</mark>	Ignore	0	-	0.5	-	90
∕⊗4	Machine	0	-	0.5	-	0
Sec.	A.4. 1.1	•		0.5		•

30 Accept the Component Thickness dialog and then Calculate the toolpath.



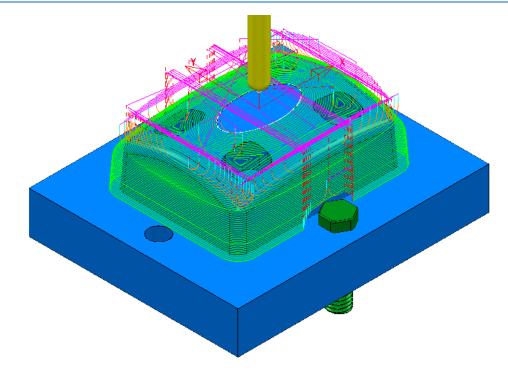
The updated **Rest Roughing**, **toolpath** (*D16TR3-Rgh2*) now runs straight through the *Y-Axis* clamping bolts.

- **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 theComponent Thickness dialog
- 34 In the *Clone* pull down select the **Toolpath** *D16TR3-Rgh1* and the click on the

Copy Thickness Data icon.

			Co	omponent Thickr	iess		?
urfaces	Verification	Surface Def	aults				
Entity	BN10-Fin	1	~		one D16TR3	-Rgh1	~ <b>8</b>
*	1		Use Axi	ial Thickness			
Machi	ning Mode			Thickness			
Machi	ne v			0.0			
Set	Mode	Thickness	Axial	Total Thickness	Total Axial	#	
0	Machine	0	-	0	-	0	
d 🔁 🕹	Ignore	0	0.70	0	0.70	90	
\$2	Machine	0	-	0	-	0	
<b>4€3</b>	Collision	4	22	4	121	90	
\$4	Machine	0	-	0	-	0	
\$5	Machine	0	-	0	(in)	0	
\$6	Machine	0	050	0	171	0	
\$7	Machine	0	-	0	-	0	
×	Machine		121	0		94	
		4	Apply	Accept	Cancel		

**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** all of the current settings will be exactly the same as in **BN10-Fin1**. In this case the required **Component Thickness** settings can be cloned from the **Toolpath D16TR3-Rgh2**.

**39** In the **main** page of the **Steep and Shallow Finishing** dialog, select the

Component Thickness dialog

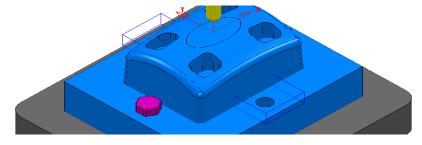
40 In the *Clone* pull down select the **Toolpath** *D16TR3-Rgh2* and the click on the

Copy Thickness Data icon.

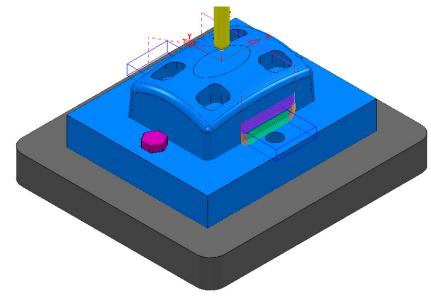
- 41 Accept the Component Thickness dialog.
- 42 In the **Steep and Shallow Finishing** dialog select the **Block** page and *lock* both the **Min Z** value (-35) and **Max Z** value (0). Keep the **Block** page open.
- 43 In the **PowerMILL** *explorer*, right mouse click on the **Level** named *ClampingBolts-Y* and from the local menu click on **Select All**.

Limits		
Min	Max	Length
X -17.79422	17.794229	35.588457
Y -66.19059!	66.190599	132.38119
Z -35.0	-26.0	9.0
X Y Z	X Y Z	<b>C</b>
Estimate limits		
Tolerance	Expansion	Туре
0.1	10.0	Model 👻
🔲 Include refere	nce surfaces	Calculate
Draw		Opacity 🛛

- 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**.



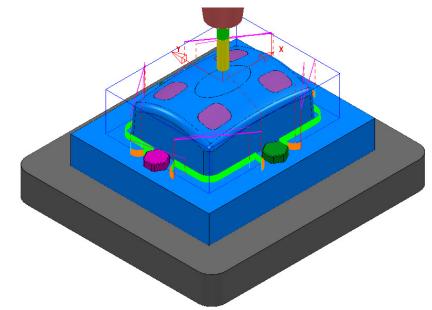
- 46 Finally select **Calculate** at the bottom of the main **Steep and Shallow Finishing** dialog to create the new **toolpath** *BN10-Fin2*.
- 47 Cancel the Steep and Shallow Finishing dialog.



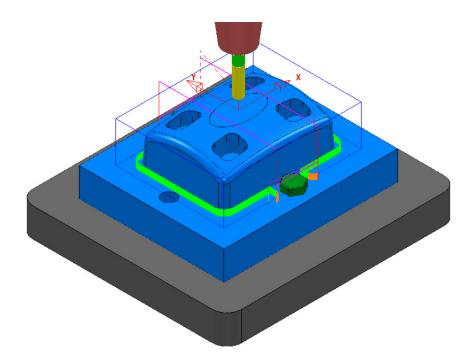
48 Save the Project (but do not close it).

### **Exercise**

**1** Activate the existing Constant Z Finishing toolpath *D8TR1-Fin1*.

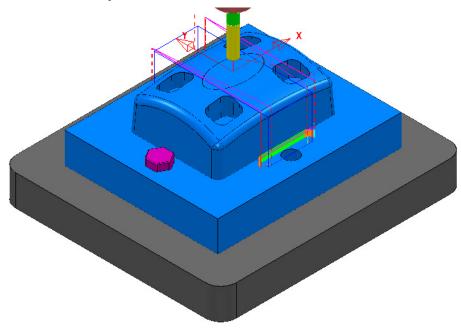


- 2 Select **Settings** and **Recycle** on the above toolpath.
- 3 Use the Component Thickness options to Ignore the surfaces on Level ClampingBolts-X (purple), and Machine with a 4mm thickness around the surfaces on Level ClampingBolts-Y (green).
- 4 Calculate the changes to toolpath D8TR1-Fin1.



- 5 Select **Settings** and **Make a Copy** for the toolpath **D8TR1-Fin1**.
- 6 Rename the copy as **D8TR1-Fin2**.

- 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

# **17. Stock Models**

### Introduction

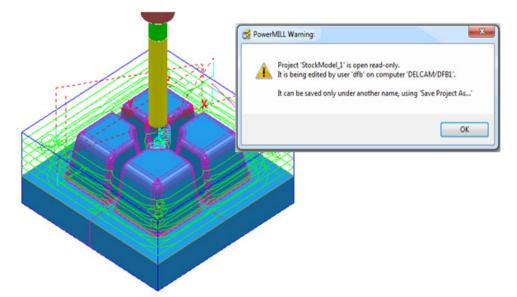
A **Stock Model** provides both a visual and physical record of the remaining material after such entities as the material block and selected toolpaths have been considered.

#### **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.



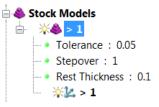
An empty **Stock Model** entity appears in the **PowerMILL** *explorer* and an associated **Stock Model** settings dialog appears in the graphics area.

<b>3</b>	Stock	Model	?	×
Name				
1				S
Tolerance		Stepover		
0.1	]	1.0		
Rest thickness		Workplane		
0.1	2	1		*
	Accept	Cancel		

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.

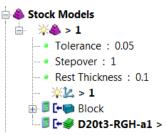
<del>3</del>	Stock Model	? ×
Name		
1		
Tolerance	Stepover	
0.05	1.0	
Rest thickness	Workplane	
0.1	1	~
A	ccept Cancel	

6 Modify the values in the **Stock Model** settings to be exactly as shown above and then **Accept** the dialog.

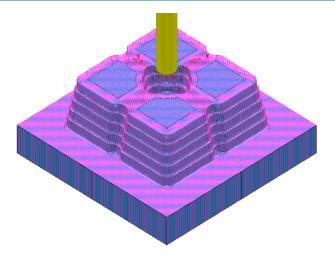


The displayed settings in the Stock Model will change accordingly.

7 Right mouse click on the toolpath D20t3-RGH-a1 and from the local menu select Add to > Stock Model.

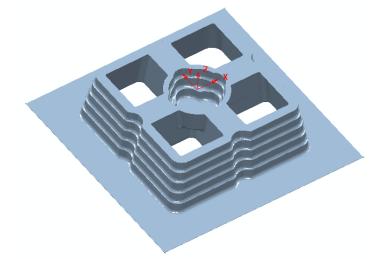


The **toolpath** and associated **Block** are included in the **Stock Model** tree but at this stage are <u>not</u> calculated.



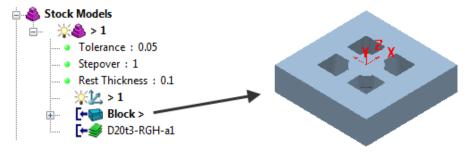
The **Stock Model** will display the visual effect of toolpath **D12t3-RGH-a1** as a purple wireframe mesh.

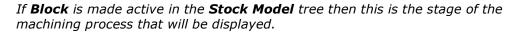
- 9 Right-click on Stock Model 1 and from the local menu, select Drawing Options, and then tick both, Shaded and Show Rest Material.
- 10 From the Viewing toolbar, undraw the model (both Shaded and Wireframe).



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**.





- 11 From the **Strategy Selector** , **Finishing** options select the **Offset Flat Finishing** strategy and enter data in the 3 pages exactly as shown below.

<b>3</b> 0	Offset Flat Finishing ? 🗙
Toolpath name	e D20t3-Flats-a1
Workplane 	Offset flat finishing
	Flat tolerance 0.0 Find flats on triangles 🗸
Gffset flat finishing     Rest	Allow tool outside flats
	Add approaches from outside Approach outside allowance 0.05
	☐ Ignore holes
	Threshold (TDU) 2.0
End point	Tolerance     Cut direction       0.02     Climb
	Thickness
	Stepover 9.0 5.12132
	Final stepdown
	1.0
	Calculate Queue OK Cancel

- Note: Add approaches from outside is *un-ticked* to enable fully а independent Leads to be applied.
- Allow tool outside flats is *un-ticked* to limit the toolpath exactly to the X b Y dimensions of the Block.
- c The Rest Machining box (Rest machining to Stock Model) is ticked to prevent the already finished, upper flat areas from being included in the toolpath.
- d By ticking this option the **Rest** page will become available for selection in the local browser.

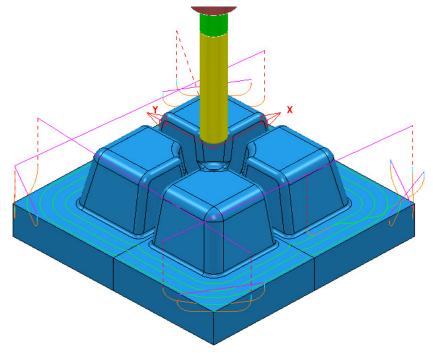
<del>3</del>	Offset Flat Finishing		?	×
Toolpath na	me D20t3-Flats-a1			
Workplane Block Jool Machine tool Sumit	Rest Rest machining Stock model	1	~	
	Stock Model state Detect mater	D20t3-RGH-a1	~	
Automatic verification     Point distribution     Z Tool axis     Machine axis control		xpand area by 0.0 um gap length 31.4	15927	

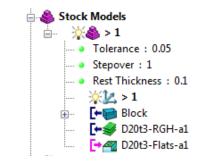
12 In the above dialogue select the **Stock Model** as **1**.

Extension	c	Links	Point	distributi	on
Z heights	First lead in	Lead in	Lead out		ead out
st choice		2nd	hoice		
Vertical arc		✓ No	ne		
	Distance 15.	.0		Distance	0.0
	Angle 90.	.0		Angle	0.0
	Radius 10.	.0		<u>Radius</u>	0.0
✓ Add leads to s ✓ Add leads at t	ool axis discontir	nuities Angular thresho	ld 90.0		
Gouge check				Apply lead	l ins
	Apply	Accept	Cancel		

13 In the **main** toolbar, **Leads and Links** options set the **Lead in** options exactly as shown below.

- 14 Then copy the **Lead in** options to the **Lead out** .
- 15 Accept the Leads and Links dialog.
- **16 Calculate** the **Offset Flat** finishing strategy and then **Close** the dialog.

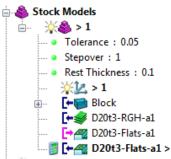




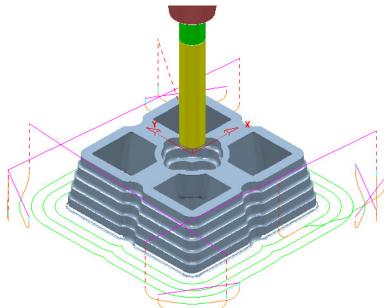
			ς.		
			Ŀ		
		-	0	0	
	-	G	0	r	
-	-	30			

The latest **Toolpath - D20t2-Flats-a1** is automatically added as a reference item in the **Stock Model** list but not as a physical entity.

17 In the PowerMILL explorer, right-click over the Toolpath - D20t3-FLATS-a1 and in the local menu select Add to > Stock Model.



18 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.

19 Activate the toolpath *D20t3-RGH-a1* and in the local menu select Settings... to open the Model Area Clearance dialog.

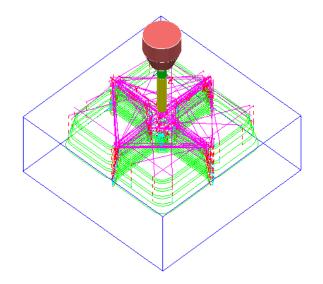
20 Select **Create a new toolpath based on this one** and input new data into the dialog exactly as shown on the following page.

🛪 Mode	el Rest Area Clearance ? 🗙
Toolpath name	D10t1-RGH-a1
Workplane Block Tool Machine tool Minit Model rest area clearance Rest Offset Step cutting Wall finishing Wall finis	Model rest area clearance         Style         Offset model         Profile         Area         Climb         Tolerance         0.1         Thickness         Image: Difference         0.5         Stepover         Image: Difference         Stepdown         Automatic         Image: Difference         Automatic         Image: Difference         Image: Differe         Image: D
	Calculate Queue OK Cancel

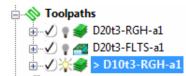
- a Input the **Toolpath** name as **D10t1-RGH-a1**
- 21 Activate the **D10t1** tool.
- 22 Set the *Style* as Offset model.
- 23 Tick the **Rest machining** box.
- 24 Select the **Rest** page and in **Rest Machining** use **Stock Model** 1.

Mod	el Rest Area Clearance	?	×
Toolpath name	D10t1-RGH-a1		
Workplane Block Block Wachine tool Machine tool Machine test area clearance Greet Gree	Rest         Rest machining         Stock model         Stock Model state         D20t3-Flats-a1         Detect material thicker than         0.0         Expand area by         0.0         Minimum gap length         15         Consider previous Z Heights         The second s	)	-

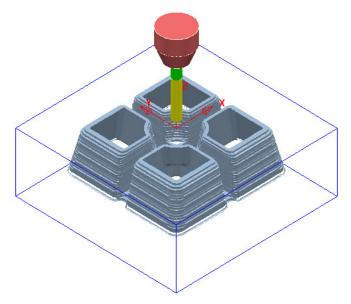
25 Select **Calculate** on the **Model Area Clearance** dialog to machine within the limits of the specified **Stock Model**.



- 26 Close the Model Area Clearance dialog.
- 27 The latest **Toolpath D10t1-RGH-a1** is automatically added as a reference item in the **Stock Model** list but not as a physical entity.
- 28 In the PowerMILL explorer use the left mouse button to drag the Toolpath D10t1-RGH-a1 to the bottom of the list.



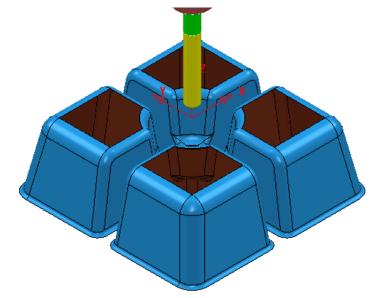
- 29 In the PowerMILL explorer, right-click over the Toolpath D10t1-RGH-a1 and in the local menu select Add to > Stock Model.
- 30 In the **PowerMILL** *explorer*, right-click on the **Stock Model** named **1** and in the local menu select **Calculate**.



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.

- 31 Un-draw the **Stock Model**.
- 32 Select and Blank all except the following surfaces:



- 33 Using the left mouse button, drag a box across the model above to select all displayed surfaces.
- **34** Activate the tool **BN8**.
- 35 In the **PowerMILL** *explorer*, right-click on **Boundaries** and in the local menu, select the **Selected Surface** option.

🕉 Selected Sur	face Boundary ? ×
🚺 🔓 🔊 Name	BN8-SeleSurfs
Top Constraints Top Constraints Tolerances  Tolerance 0.02  Thickness 0.0  Axial Thickness 0.0  Use Axial Thickness  Tool  Tool  Number Of State of	Limit Boundary  Inside Limiting Boundary  Outside  Automatic Collision Checking  Holder Clearance 0.0 Shank Clearance 0.0 Block
Private	Limit 💆 🗸
Allow boundary to be private	1
Edit History Apply edit history on calculation	<b>(</b>
Apply Queue	Accept Cancel

- 36 Enter data exactly as shown above and then **Apply** to create the **Boundary**.
- 37 Check that both Boundary 1 and the BN8 tool are both Active.

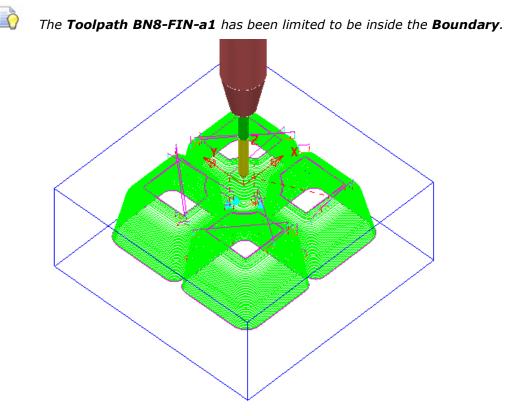
- 38 Select the **Toolpath Strategies** and open the **Finishing** page.
- **39** Select the **Steep and Shallow Finishing** option and enter data into the dialog exactly as shown below:

🛃 Steep	p and Shallow Finishing 🛛 📍 🗙
Toolpath name	BN8-FIN-a1
	Steep and shallow finishing         Order       Additional stock         Top first       0.4         Threshold angle       Steep shallow overlap
<ul> <li>Stock engagement</li> <li>Steep and shallow finishing</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Freeds and speeds</li> <li>History</li> <li>Notes and Description</li> </ul>	30.0       0.0         Steep       Shallow         Spiral       Cut direction         Climb< ▼
User defined settings	Image: Sympletic symplet       Image: Symplet symplet       Image: Symplet symple
	Calculate Queue OK Cancel

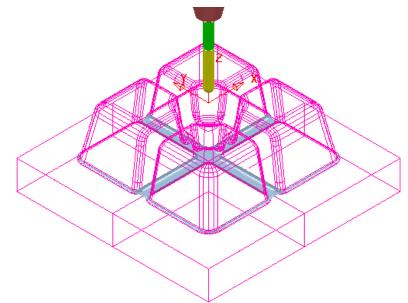
40 Select the **Limit** page and enter data exactly as shown in the following image:

🛪 Steep	and Shallow Finishing 🛛 🗧 🗙
Toolpath name	BN8-FIN-a1
Workplane Block Machine tool Stock engagement Stock engagement Automatic verification Point distribution Tool axis	Limit Boundary BN8-SeleSurfs Trimming Keep inside
@ Machine axis control 	Block Limit 💆 🗸

41 Select **Calculate** to create the **Toolpath** and then **Close** the dialog.



- 42 In the **PowerMILL** *explorer*, right-click the **toolpath** *BN8-FIN-a1* and in the local menu select **Add to** > **Stock Model**.
- 43 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**.

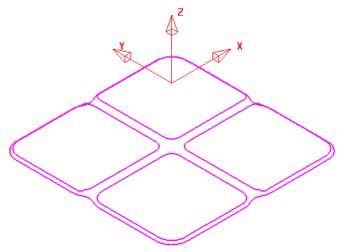
It is however possible to create a **Stock Model Rest Boundary** to define the areas with material left on. This can then be used to limit a finishing strategy such that only the remaining stock is removed.

1 In the **PowerMILL** *explorer*, right-click on **Boundaries** and in the local menu select **Stock Model Rest**.

<del>3</del>	Stock Model	Rest Boundary ? 🗙
8 4	Name Name	BN4-StockModelRest
	1   v     BN8-FIN-a1   v     er Than   0.1     pand Area By   1.0	Limit Boundary     Inside     Limiting Boundary     Outside
لعالم المراجع ا المراجع المراجع	Tolerance     0.02       Thickness     0.0       Thickness     0.0       Jse Axial Thickness	Automatic Collision Checking Holder Clearance 0.0 Shank Clearance 0.0 Block
Private Allow boundary Edit History Ø Apply edit histor		<b>i</b>
Ap	oply Queue	Accept Cancel

2 Enter data exactly as shown in the above image and **Apply** and then **Cancel**.

**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.

Q

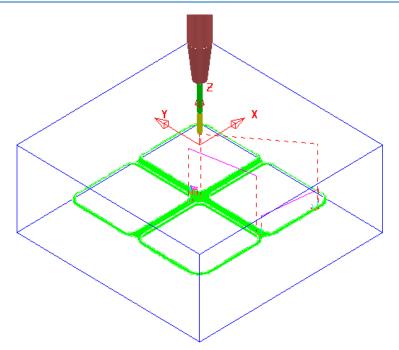
- 3 Select the **Toolpath Strategies**  $\bigotimes$  and select the **Finishing** tab in the dialog.
- 4 Check that the new **Boundary BN4-StockModelRest** and the **BN4** tool are both **Active**.
- 5 Select the **3D Offset Finishing** strategy and enter data exactly as shown below:

- 😹	3D Offset Finishing ? ×
Toolpath nan	ne BN4-FIN-a1
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Machine again and the second se</li></ul>	3D offset finishing         Pattern         Image: Start on pattern         Image: Start on pattern         Image: Spiral         Image: Smoothing         Image: Maximum offsets         Maximum number of offsets         10
	Tolerance     Cut direction       0.02     Climb       Thickness       Imp     0.0       Stepover       Imp     0.3

6 Select the **Limit** page and enter data exactly as shown in the following image:

Toolpath name	BN4-FIN-a1
Workplane         Block         Tool         Machine tool         Stock engagement         Stock engagement         Hattinishing         Automatic verification         Point distribution         Tool axis         Machine axis control	Limit Boundary BN4-StockModelRest Trimming Keep inside
Rapid move heights Deads and links Start point Contraction	Block

- 7 Select **Calculate** to create the toolpath and then **Close** the dialog.
- 8 In the Leads and Links options Apply (1<sup>st</sup> Choice) *Lead in/out* as Surface Normal Arc with *Angle* 90 and *Radius* 3.



- 9 In the **PowerMILL** *explorer*, right-click the **toolpath** *BN4-FIN-a1* and in the local menu select **Add to** > **Stock Model**.
- 10 In the **PowerMILL** *explorer*, right-click on the **Stock Model** and select **Calculate** to update.

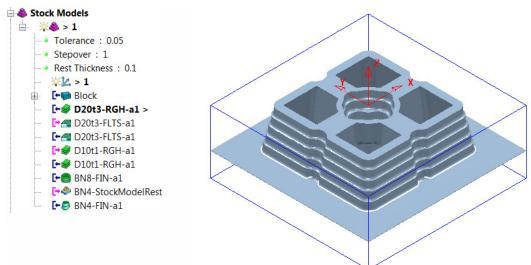


The **Stock Model** displayed with **Show Rest Material** set will have visually disappeared apart from perhaps, a few tiny, insignificant patches.



If a previous toolpath in the **Stock Model** tree is selected then the status of the material at that point will be displayed.

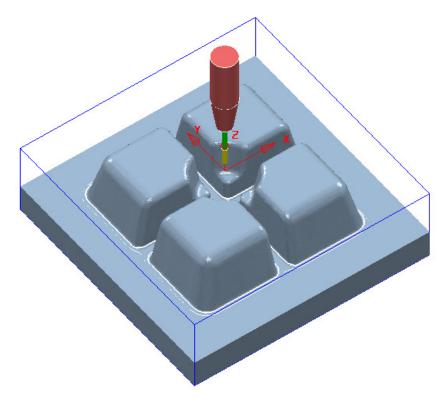
11 In the **PowerMILL** *explorer*, **Stock Model - 1** tree, **Activate** the toolpath named **D20t3-RGH-a1**.



12 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.
- 13 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**.

14 Select File > Save Project to update the stored Project.

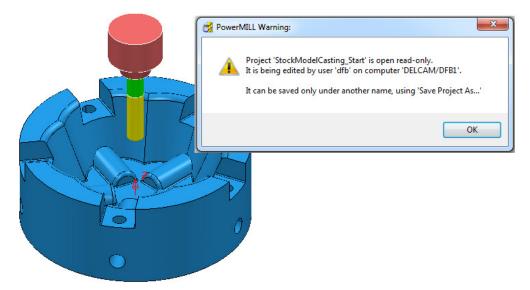
#### Stock Model starting from a complex block.

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 initial **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



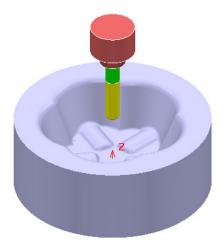
- 3 Select **OK** in the **PowerMILL Warning** dialog.
- 4 Select File Save Project As:

.....\ COURSEWORK\PowerMILL-Projects\StockModelCasting

5 Open the **Block** form and set *Coordinate System* to **Global Transform**.

<del>3</del>	Block	? ×
Defined by		
Triangles	¥	🖄 🖪 🔀
Coordinate System		3
Global Transform	¥	~

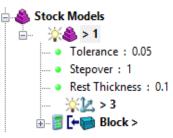
6 Select **Defined by** – **Triangles**, select and browse to the triangle model:-C:\Training Data\PowerMILL Data\Triangles\StockModel-Casting.dmt



- 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 Settings.

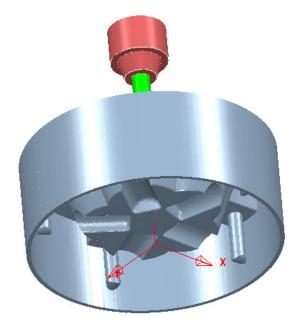
3	Stock Model		×
Name			
1			S
Tolerance	Stepover		
0.05	1.0	]	
Rest thickness	Workplane		
0.1	3		~
	Accept Cancel		

- 11 Edit the dialogue with above values and **Accept**.
- 12 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**.

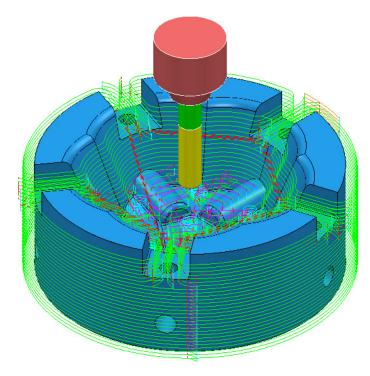
- 13 Right mouse click on the new **Stock Model** and from the local menu select **Calculate**.
- 14 Undraw the **Block**.
- 15 Right mouse click on the new **Stock Model** and from the local menu select **Drawing Options** and tick both **Shaded** and **Show Rest Material**.



The **model** form is visually removed from the **Stock Model**.

#### **Exercise:**

- **16** Calculate a Block of *Type* Cylinder and *Defined by* Model.
- 17 Create a **Model Rest Area Clearance Offset Model** strategy with the **D25TR3** tool, using the **Stock Model** as the reference item.



The **Area Clearance** - **Rest** strategy is created within the defined volume of the **Stock Model**.

## 18. Macros & User Menu

### Introduction

There are various ways in which the user can customise **PowerMILL**. This chapter looks at the creation of:

- Macros
- User Menus

#### **Home Area**

The usual method to enable the **User Menu** to be recognised by **PowerMILL** is to set up and use a **home** area in *Windows*. This can be located to any convenient area on the hard disk. In the following example the **home** area is:-

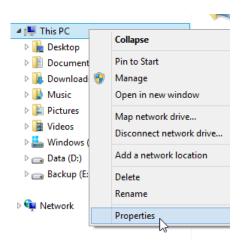
C:

To check if a Home area already exists on a computer;



#### 1 Open File Explorer

In the local explorer, right-click on **This PC** and select **Properties** to open the **System** dialog.



0

Alternatively, at the lower left corner of the screen, right mouse click on

**Start** and from the local menu select **System**.

2 In the **System** dialog select the **Advanced** system settings option and then in the **System** Properties dialog, select **Environment** Variables.

	System	- 🗆 ×
🔄 🎯 🕆 🏦 🕨 🖌 Control Pan	el 🕨 All Control Panel Items 🔺 System 🔍 🖒	Search Control Panel 🔎
Control Panel Home Control Panel Home Control Panel Home Remote Settings Control Settings C	System Properties           View bas         Computer Name         Hardware         Advanced         System Protection         Remote           Window         You must be logged on as an Administrator to make most of these change         Performance         Visual effects, processor scheduling, memory usage, and virtual memory           System         Settings         Settings           Process         User Profiles         Desktop settings related to your sign-in	dowc 8
	System Pen and Settings related to your signed. Computer r Compu Startup and Recovery Compu System startup, system failure, and debugging information Full cor Compu Settings	Change settings
	Windows av Windov Product	

3 If the **home** area is not already defined as shown below, select New...

Environment Variables		
Jser variables for dft		
Variable	Value	^
home	C:	
KEYSHOT_EXTE	C:\Program Files\Delcam\PowerSHAPE1	
TEMP	%USERPROFILE%\AppData\Local\Temp	
TMP	%USERPROFILE%\AppData\Local\Temp	$\sim$
	New Edit Delete	

- 4 In the User Variables dialog, enter home in the Variable name field and C:\ in the Value field.
- 5 Click OK.

C C

You must have Admin rights to create a **New User Variable**.

6 If it does not already exist, create a folder named **pmill2** in the required **home** area (**C:\pmill2**).

If the **pmuser.mac**, **user\_menu** and any other **macro**s are being stored in this folder, then **PowerMILL** will have immediate access through the default **Macro** search paths.

### Macros

A **Macro** is a text file, which contains a sequence of commands to drive **PowerMILL** operations. These can be created by recording each command or by typing in the commands directly. **Macros** (which have a **.mac** extension) can then be run from the **PowerMILL** *explorer* or from a *user defined menu*.

When **PowerMILL** starts up, it will attempt to identify and run the **initialisation macro** (if it exists!) called **pmuser.mac**. By adding customised **PowerMILL** commands to the initialisation macro, the user can set up their own default settings and parameters.

For immediate use a **pmuser.mac** can be placed in the **pmill2** folder, directly below the users **Home** area. The **pmill2** directory is also a convenient location for storing other user-defined **macros** ideally arranged in suitable sub-folders. This will be covered later.

#### Creating a pmuser macro

**Macros** are created in **PowerMILL** by recording the operations as they occur. Only flags or parameters that are physically changed will be recorded into the macro; therefore to record a value that is already set, it must be re-entered or a flag re-switched. For example, if the finishing tolerance is currently set to 0.1mm, and it is required that the macro stores the same value, then it must be re-entered over the existing copy in the form during recording.

- 1 Select File > Delete All.
- 2 To start recording, right-click on Macros in the PowerMILL explorer and select Record.

⊞ <b>⊞</b> Mao	cros	
		Macros
		Run
		Record
	✓	Stop

3 Save **pmuser.mac** by first clicking on the existing folder displayed in Save in.

🥳 Select Re	cord Macro File		<b>X</b>
	Save in: 👔 train 👻	G 🎓 📂 🛄 -	
	Name	Date modified	Туре
	🕌 CopyCAD 2010 Data	02/07/2010 09:45	File folder
2	퉬 CopyCAD 2011 Data	01/12/2010 15:31	File folder
	퉬 CopyCAD Pro 8.0_Data	09/12/2009 12:11	File folder
	COURSEWORK	12/11/2010 15:30	File folder
	퉬 Delcam-Draft Data	08/10/2010 09:28	File folder
	퉬 Exchange	24/06/2008 14:18	File folder
	🔒 pmill2	20/09/2010 16:26	File folder
2	PowerINSPACT Data and a 1 (09 (2000 00:00	00 40 0000 45 57	File folder

4 Then, in the resultant dialog, browse to C:\pmill2 and at the bottom of the dialog enter the file name, **pmuser** before clicking **Save**.

•		•
File name:	pmuser 🔻	Save
Save as type:	Macro (*.mac)	Cancel
	n changes to red while recording is in proc	

The Macro icon changes to red while recording is in progress

The **pmuser.mac** automatically runs whenever **PowerMILL** starts up, thereby providing the user with custom default settings. It can also be run at any time during a session to reinstate the initialisation settings without having to restart **PowerMILL**.

All required data or options must be set or selected if they are to be included in a macro (Even if it is necessary to overwrite or switch them off first).

- 5 From the **Main** pull-down menus select **Tools** > **Options**.
- 6 Select Tools > Feeds and Speeds, change the Feed Rate Plunge Factor to
   0.2, tick the Auto Load Feed Rate box, and Accept the dialog.

<b>1</b>	Options	? ×
Tolerances 	Feeds and speeds	
Feeds and speeds	Feed rate plunge factor	0.2
• Boundaries	Lead in(%)	100.0
in View in Import	Lead out(%)	100.0
Export	Auto load f	feed rate 🗸
Project     NC Programs	Auto load dep	th of cut 🗌

7 Open the Leads and Links dialog and input Short/Long Threshold as 3, both *Short* and *Long* as Skim, and *Default* as Incremental.

<b>3</b>	L	eads and Lin	ks	? ×	
Z heights	First lead in	Lead in	Lead out	Last lead out	
Extensio	ns	Links	Point	distribution	
	Short/Long threst	hold 5.0			
Retract and a	pproach moves		Short		
Along Tool axis 🗸			Skim	~	
Automatically extend			Long		
Maximum length 250.0			Skim	~	
Default					
Retract distance 0.0			Incremental	~	
Approach distance 0.0					

8 Accept the Leads and Links dialog.

9 In the **PowerMILL** *explorer*, right-click on **NC Programs**, and select **Preferences**.

Toolpaths Tools	NC Programs	61	NC Prefere	nces	?	>
Boundaries Boundaries Patterns Peature Set Workplane Evels and Models Stock Mod	Create NC Program NC Program Names Folder Names	Output Toolpath Changes made h	Fixture offset ere will not change existing NC Pr	ograms		
	Deactivate Draw All Undraw All	Use Pro	oject Off v Output Fold	der C:\Temp\NCProg	grams	
Macros	Write All Setup Sheets	Ma	chine Option File			
	Preferences	Output Wor	kplane 🗸 🖌	Part Name		
	Project Settings Edit All			Tool Value		~
	Create Folder	Automa	tic Tool Alignment On 🛛 🗸	Connection Moves	Simultaneous	~
	Text Blocks					
	Statistics					
	Delete All		Close			

- 10 On the NC Preferences dialog Output page, select the Machine Option File as heid400.opt.
- 11 Even if already set, select:

Use Project - On

Tool Value - Tip

Automatic Tool Alignment - On

Connection Moves - Move, Rotate

12 Select the Toolpath page and even if already set, select:-

**Tool Change - On New Tool** 

Tool Numbering - As Specified

**Tool Change Position - After Connection** 

Cutter Compensation: Length - Off

Radius - None

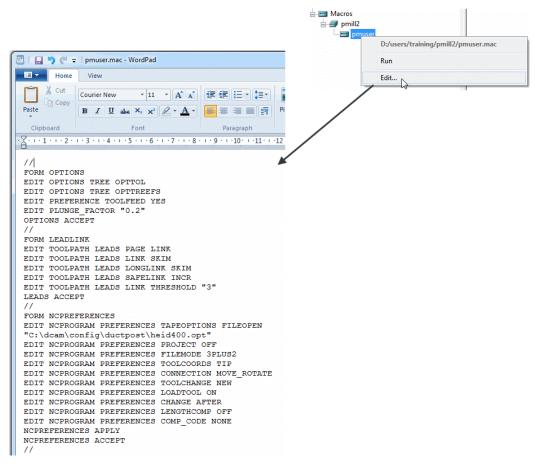
**13** Select **Apply**, followed by **Accept**.

It is important to remember to **Accept** and not to **Apply** forms that are opened while creating an **Initialisation Macro**, the only exception being the creation of **NC Program Preferences** (An initialisation file is for pre-setting forms but at this stage without executing the commands).

14 Select **Stop** from the local **Macros** menu to stop the recording process.

🖶 🛲 Ma	cros	
		Macros
		Run
	$\checkmark$	Record
		Stop
		En Massa

- **15** Select the plus (+) sign next to **Macros** in the **PowerMILL** *explorer* to open the tree.
- 16 Right-click **pmuser**, then select **Edit**.



This is a standard **Wordpad** document that can be edited with new values and then saved.

To try this out you will need to exit and re-open PowerMILL.

17 Select File > Exit and restart PowerMILL to check that the settings from pmuser.mac have been activated.

## **Other Typical Macro Applications**

Apart from tailoring **PowerMILL** by the creation of an initialisation macro other useful applications include macros for un-draw, draw and resetting **Leads and Links**, locating and inserting tooling **Templates**, setting **NC Preferences**, regularly used machining sequences or settings, and input of preview, commands not yet available on the GUI. Note the pmuser macro can be run through the browser at any time during a **PowerMILL** session to restore your settings.

#### **Creating NC Preference macros**

1 From the PowerMILL explorer, record a macro called Fanuc6m\_prefs and save it in:-

#### C:\pmill2\NCpreferences\...

- 2 In the NC Programs > Preferences > Output tab select the Machine Option File as fanuc6m.opt.
- 3 Input the remaining settings exactly as set earlier for the **Heid400** in the pmuser macro.
- 4 Stop the **Macro** recording process.



Even if already set it is essential to re-specify settings if they are to be recorded into the macro.

A **macro** file (**Fanuc6m\_prefs.mac**) is now created that sets up **NC Preferences** for a post-processor option file and the command lines are shown below.

🛎 I 🔚 אופי 🗸	fanuc6m_prefs.mac - WordPad
Home Home	View
Paste	Courier New     ▼     11     ▼     ▲     Image: Ima
Clipboard	Font Paragraph
· Z · · · 1 · · · 2 ·	
EDIT NCPRO "C:\dcam\c EDIT NCPRO EDIT NCPRO EDIT NCPRO EDIT NCPRO EDIT NCPRO EDIT NCPRO EDIT NCPRO EDIT NCPRO EDIT NCPRO	GRAM PREFERENCES PROJECT OFF GRAM PREFERENCES TAPEOPTIONS FILEOPEN onfig/ductpost/fanuc6m.opt" GRAM PREFERENCES NORMAL OFF GRAM PREFERENCES FILEMODE 3PLUS2 GRAM PREFERENCES TOOLCOORDS TIP GRAM PREFERENCES CONNECTION MOVE_ROTATE GRAM PREFERENCES TOOLCHANGE NEW GRAM PREFERENCES LOADTOOL ON GRAM PREFERENCES LENGTHCOMP OFF GRAM PREFERENCES COMP_CODE NONE GRAM PREFERENCES ARCFIT OFF GRAM PREFERENCES CYCLES ON GRAM PREFERENCES COOLANT ON CES APPLY

### **Exercise**

5 Create two more independent NC Preference macros for the Hied400 and Mazak option (Use the filenames h400\_prefs.mac and mazak\_prefs.mac respectively for the macros), and save them to:

C:\pmill2\NCpreferences\...

# **User Menu**

In **PowerMILL** it is possible to create a *user menu* enabling the user to further streamline and customize the command structure. To access the *user menu*, right mouse click within an empty area of the **PowerMILL** *explorer*.

 Right-click in an empty area of the **PowerMILL** explorer to access the current user\_menu (if it exists).

This menu is created from a text file called user\_menu. It must be in the directory **pmill2** in the **Home** area of the computer for the individual user, and saved as a text file (not a word Document) and with no extension. If available it is easier to obtain a copy of an existing user\_menu and modify it as required.

A **user menu** can be setup to provide easy access to existing **PowerMILL commands**, Macros and Templates.

2 Load the in-active **user\_menu** file (which is in the **pmill2** folder) into **Wordpad**.

Below are instructions for the **user\_menu** command line input (these exist in the sample file).

A command line will be added to the above user\_menu to run the **h400\_prefs** macro.

#### I "heid400" 1 "macro C:\pmill2\ncpreferences\h400\_prefs"

3 Add the above command line to run the h400\_prefs macro directly from the user\_menu.

This line locates and runs the file **h400\_prefs.mac** created earlier in this chapter. It needs to include the correct drive and path where the macro is located. All lettering must be in the correct upper or lower case and include the inverted commas and hyphens.

A more complete version of a typical **user\_menu** is as shown below:

📓    (], = ⊃) (C. =    user_menu - WordPad				
Home View	0			
Courier New v 11 v A A F F F = v F v M A A P F A A A F F F F = v F A A A A A A A A A A A A A A A A A A	Find  Control  Find  Control  Find  Control  Find  Find  Control  Find  Find Find			
<pre># ************************************</pre>	on".			
U user_menu S T "WELCOME TO THE USER MENU" S				
<pre>T "[GENERAL]" S I "Unlock Project 1 "project claim" I "Reset My defaults (pmuser)" 1 "macro pmuser" I "Reset Leads-n-links" 1 "macro general\ResetLeads-n-Links" I "drop_bound" 1 "macro general\drop_bound" I "Draw Triangles (Ctrl-Alt-Tab)" 1 "macro general\DrawTriangles" S M "NC PREFERENCE Sub-Menu&gt;&gt;&gt;" I "heid400" 1 "macro NCpreferences\h400_prefs" I "heid426 (default)" 1 "macro NCpreferences\h426_prefs" I "fanuc6m" 1 "macro NCpreferences\mazak_prefs" S </pre>				
<pre>M "[SEPARATE TOOLS - STEEL]" S M "EndMill" I "EM30" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM30" I "EM20" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM20" I "EM10" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM20" I "EM16" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM18" I "EM16" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM18" I "EM10" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM10" I "EM10" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM10" I "EM10" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM10" I "EM6" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM6" I "EM6" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM6" I "EM6" 1 "macro Tool_Libraries\SeparateTools-Steel\EndMills\EM8" I</pre>				
100% 😑				

This is the **user\_menu** file as displayed in the **Wordpad**.

[My Menu]				
[GENERAL]				
Unlock Project				
Reset My defaults (pmuser)				
Reset Leads-n-links				
drop_bound				
Undraw Leads-n-links				
Draw Leads-n-links				
Preview Command list				
Draw Triangles (Ctrl-Alt-Tab)				
BACKGROUND COLOUR Sub-Menu>>>	×			
NC PREFERENCE Sub-Menu>>>	•		heid400	
			heid426 (default)	2
[SEPARATE TOOLS - STEEL]	•		fanuc6m	
			mazak	
		_		

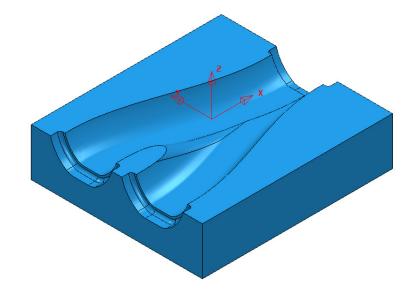
#### Note the **NC-Preferences** section uses a *sub-menu*:

- **M** denotes the start of the sub-menu.
- Z denotes the end of the sub-menu.

# **19. Tutorials**

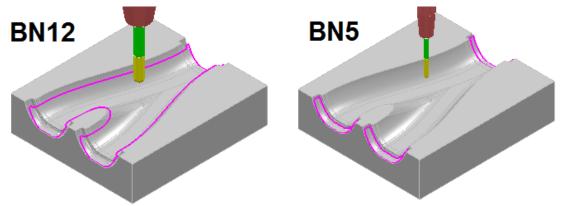
# **Y-Branch**

- 1 Select File > Delete All and Tools > Reset Forms
- 2 Import the Model:-
  - ....\PowerMILL\_Data\Models\Y-BranchDuct.dgk

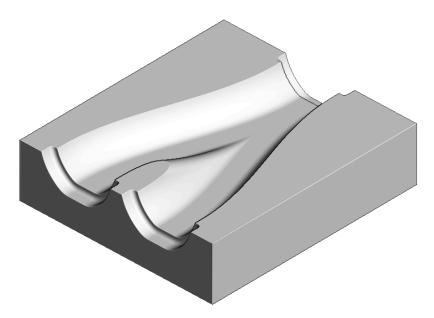


- 3 Save Project As:
  - ....\COURSEWORK\PowerMILL\_Projects\Y-Branch-MC
- 4 Create the following Tools:-
  - Dia 16 with a 3 Tip Radius named D16TR3
  - Dia 12 End Mill named EM12
  - Dia 12 Ball Nose named BN12
  - Dia 5 Ball Nose named BN5
- 5 Create an *Active* NC Program. Newly processed **Toolpaths** are automatically added to an **Active** NC Program.
- 6 Activate the existing Workplane 1, then using the default settings Calculate a suitable Block, and Rapid Move Heights.

- 7 Create a **3D Model Area Clearance Toolpath** using the **Tool D16TR3**.
- 8 Create an **Offset Flat Finishing** *Toolpath* on the base of the Y-form using the *Tool* EM12.
- 9 Create two suitable Boundaries (Selected Surface) to limit the Finishing Toolpaths for the BN12 and BN5 tools. (Note:- For the BN5 tool select all of the end recess surfaces, including the vertical faces, and Tick the box marked Top in the Boundary dialogue).



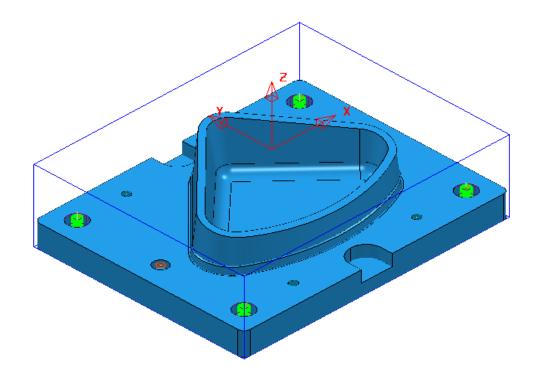
- 10 Activate the Tool BN12.
- 11 Create a Constant Z Finishing Toolpath (limited to be inside the BN12 Boundary) using both a Min (0.01) and Max (0.5) stepdown combined with a suitable Cusp height (0.03) with the Machine down to flats box ticked.
- 12 Activate the Tool BN5.
- 13 Create a suitable 3D Offset Finishing Toolpath (limited to be inside the BN5 Boundary) using a Stepover of 0.25, and with both Spiral and Smoothing ticked.
- 14 Perform a full ViewMILL simulation on the Active NC Program.



Save the Project.

# **Corner Bowl**

- 1 Select File > Delete All and Tools > Reset Forms
- 2 Open the (read only) Project:-
  - ....\PowerMILL\_Data\Projects\CnrBwl-Start



#### **3** Save Project As:

....\COURSEWORK\PowerMILL\_Projects\CornerBowl-Example

The **Project** contains; a **Model**, 2 **Workplanes**, 9 **Tools** (including a *Form* **Tool** along with the **Pattern** used to create it), 7 **Toolpaths** that fully machine the underside (OP1), and an **NC-Program** named **OP1-CnrBwl-All**.

#### Setup

- 4 Activate the Workplane named MC-Datum-OP2.
- 5 Create a new NC Program named OP2-CnrBwI-All.



The New **NC Program** will be **Active** on creation. Newly processed **Toolpaths** are automatically added to an **Active NC Program**.

- 6 Using the default settings, Defined by Box and Type Model, Calculate a Block, change the Max Z value to 0 and lock it before selecting Accept.
- 7 In the **Rapid Move Heights** dialog enter the **Workplane** *MC-Datum-OP2* and **Calculate** before selecting **Accept**.

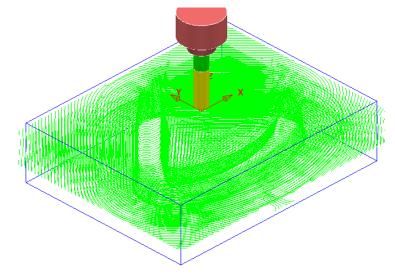
## Area Clearance – Tool D25TR3

- 1 Activate the Tool D25TR3.
- 2 From the Strategy Selector, open the 3D Area Clearance strategy dialog and select a Model Area Clearance strategy.
- 3 Enter the *Toolpath name* OP2-D25TR3-RGH1.
- 4 In the Limit page ensure that no Boundary is Active and then select Allow tool

centre outside the block 🞽

Block	
	Limit 🐓 🗸

- 5 In the Model area clearance page, use *Style* Offset model, with Thickness 0.5, Stepover 5, Stepdown 25.
- 6 In the **Offset** page, tick **Spiral**.
- 7 In the Step cutting page, tick Step Cutting and input <u>Step-up</u> 2, <u>Detect</u> <u>material thicker than</u> 0, and <u>Expand area by</u> 0.
- 8 In the **Unsafe segment removal** page, tick **Remove segments smaller than threshold**, with **Threshold** 0.95, and un-tick **Only remove segments from enclosed areas**.
- 9 In the Flat machining page, set *Machine flats* to Level.
- **10** In the **Approach** page, tick **Add approaches from outside**.
- 11 In the main Leads and Links dialog set:-
  - Lead in/out 1<sup>st</sup> choice set Vertical arc with <u>Distance</u> 15, <u>Angle</u> 90, <u>Radius</u> 8.
  - Lead in 2<sup>nd</sup> choice select Ramp, and in the Ramp options dialog set Follow Circle, <u>Max zig angle</u> 4, Circle Diameter (TDU) 0.65.
- 12 Accept the Leads and Links dialog.
- 13 Select Calculate in the main Area Clearance dialog to create the Toolpath.
- 14 Close the Area Clearance dialog.





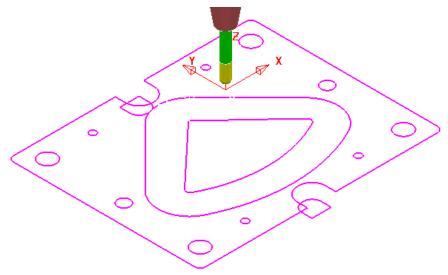
The **toolpath** is automatically added to the **Active NC Program** list.

# Finish Machining – Tool BN16

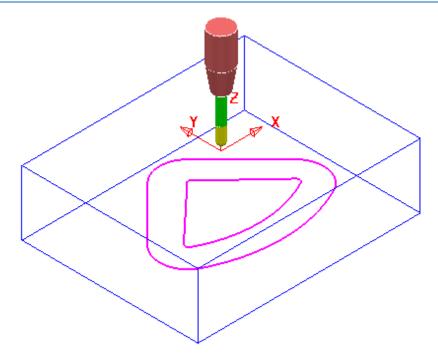
- 1 Activate the Tool BN16.
- 2 In the **PowerMILL** explorer select **Boundaries Create Boundary Shallow**.

- <del>3</del>	Shallow Boundary	? ×
11 6	Name OP2-Flat-BN16	
Upper Ang Lower Ang Tolerances Toleran Inickne Axial Thickne Use Axial Tool EVIC	Lin Olde Lin Outside Lin Outside	niting Boundary  n Checking  Clearance 0.0  Clearance 0.0  Limit
Private Allow boundary to be pri Edit History Apply edit history on cale		el

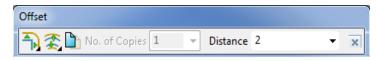
3 Enter data exactly as shown above, and un-tick the box *Allow boundaries to be private* in the **Shallow Boundary** dialog before selecting **Apply**.



4 Select and Delete all except the 2 innermost **Boundary** segments.



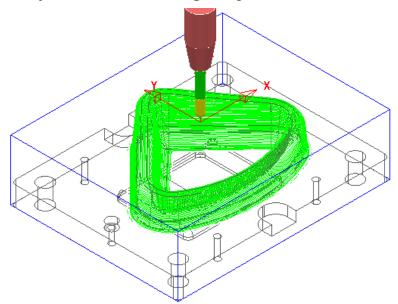
- 5 Right mouse click on the new **Boundary Flat-BN16** and from the local menu select **Curve Editor**.
- 6 In the Curve editor select the outer *Boundary* segment and apply an Offset of 2.



- 7 Then select the inner **Boundary segment** and apply an **Offset** of -6.
- 8 Select the **Green** tick at the far right of the **Curve Editor**, toolbar to accept and return the changes back to **PowerMILL**.
- 9 From the Strategy Selector, open the Finishing dialog and select a Steep and Shallow Finishing strategy.
- 10 In the Shallow Finishing dialog enter the *Toolpath name* OP2-BN16-FIN1.
- 11 In the Limit page, select the *Boundary* Flat-BN16 with *Trimming* set to Keep inside.
- **12** In the **Steep and Shallow finishing** page enter data exactly as shown below.

🕉 Steep	and Shallow Finishing	? ×
Toolpath name	OP2-BN16-FIN1	
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Stock engagement</li> <li>Mathine and shallow finishing</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>End point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Steep and shallow finits Order Top first Threshold angle 30.0 Steep Spiral Cut direction Climb Stepdown Climb Cl	Shing Additional stock 0.8 Steep shallow overlap 0.0 Shallow Spiral Cut direction Climb Stepover 0.3 0.82365 Type 3D offset Smoothing
		<u>.</u>
	Calculate Queue Ok	Cancel

- 13 In the main Leads and Links dialog select Lead in/out 1st choice set Vertical arc with Distance 0, Angle 90, Radius 8.
- 14 Accept the Leads and Links dialog.
- **15** Select **Calculate** in the main **Steep and Shallow finishing** dialog to create the **Toolpath**.
- **16** Close the Steep and shallow finishing dialog.



5

### Finish Machining – Tool EM30

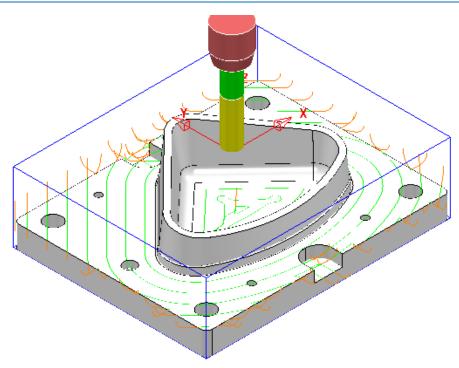
- 1 Ensure that no **Boundary** is **Active**.
- 2 Activate the Tool EM30.
- 3 From the **Strategy Selector**, open the **Finishing** strategy dialog and select an **Offset Flat Finishing** strategy.
- 4 In the Offset Flat Finishing dialog enter *Toolpath name* OP2-EM30-FlatFIN1.

In the <b>Limit</b> pag	e, select <i>Limit tool centre to bl</i>	ock edge 일
	Block	Limit 🐓 🗸

6 In the **Offset flat finishing** page, enter data exactly as shown below (Note the **tick box** settings!).

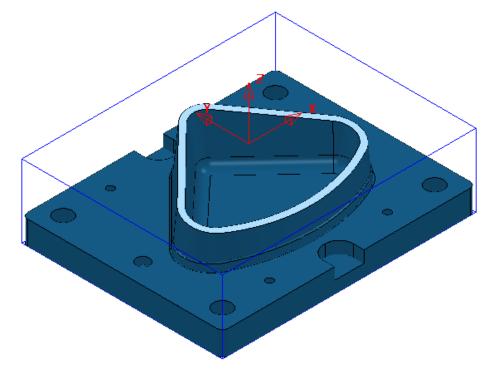
- 💰 – – – – – – – – – – – – – – – – – –	Offset Flat Finishing ? ×				
Toolpath name	OP2-EM30-FlatFIN1				
Workplane Block Tool Machine tool Machine tool Machine tool Minit Grifet flat finishing Fligh speed Automatic verification Point distribution Machine axis control Rapid move heights Leads and links Start point End point Feeds and speeds History Notes and Description Vser defined settings	Offset flat finishing     Flat tolerance   0.2   Find flats on triangles   Allow tool outside flats   Allow tool outside flats   Rest machining   Add approaches from outside   Approach outside allowance   0.05     Threshold (TDU)   2.0     Thickness   0.0     Stepover   25.0     17.67767				

- 7 In the main Leads and Links dialog select Lead in/out 1st choice set Vertical arc with <u>Distance</u> 15, <u>Angle</u> 90, <u>Radius</u> 12.
- 8 Accept the Leads and Links dialog.
- 9 Select Calculate in the main Offset flat finishing dialog to create the Toolpath.
- **10** Close the Offset Flat Finishing dialog.



# Finish Machining – Tool D16TR3

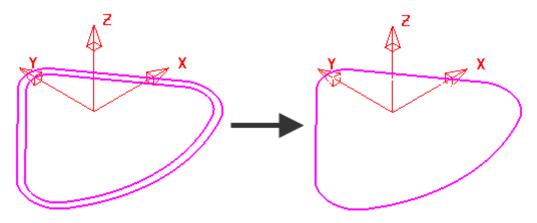
- 1 Activate the *Tool* D16TR3.
- 2 Select the *upper edge* of the **Surface Model** 3D form.



3 In the **PowerMILL** explorer select, **Boundaries – Create Boundary – User Defined**.

🥳 Use	r Defined Boundary	? ×
1 6	Name OP2-Form-TopEdge	
Insert Insert Boundary Pattern Toolpath Mode Sketch Composite curve Wireframe modelling Model Tolerance 0.02		Clear
Private		0
Edit History Apply edit history on calculation	n	$\odot$
Apply	ueue Accept Cancel	

- In the above User defined Boundary dialog *un-tick* the box Allow boundaries
   to be private before selecting the Insert Model option
- 5 Rename the **Boundary** as **OP2-Form-TopEdge** before selecting **Accept** to close the dialog.



- 6 Select and Delete the inner Boundary segment (as shown above).
- 7 From the **Strategy Selector**, open the **Finishing** dialog and select a **Constant Z Finishing** strategy.
- 8 In the **Constant Z** Finishing dialog enter the **Toolpath** name **OP2-D16TR3-FIN1**.

- 9 In the *Limit* page select the *Boundary* Form-TopEdge with *Trimming* set to Keep Outside.
- 10 In the Limit page Z Limits:-

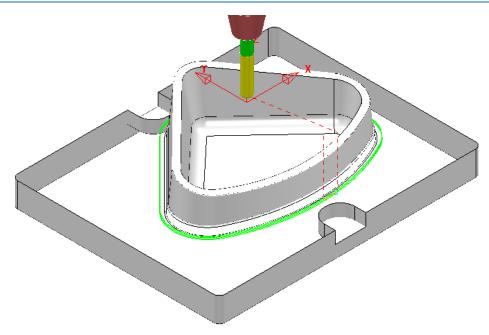


- Tick the Minimum box and use the *picking* option trap and insert the value -85 (left mouse click within the top of the base surface).
- Tick the Maximum box and insert the value -79 (this is 6mm above the top of the base surface).
- 11 In the *Constant Z finishing* page, enter data exactly as shown below (Note the tick box settings!).

Constant Z Finishing ?			
Toolpath name	OP2-D16TR3-FIN1		
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Stock engagement</li> <li>Constant Z finishing</li> <li>Unsafe segment removal</li> <li>Automatic verification</li> <li>Point distribution</li> <li>Tool axis</li> <li>Machine axis control</li> <li>Rapid move heights</li> <li>Leads and links</li> <li>Start point</li> <li>Feeds and speeds</li> <li>History</li> <li>Notes and Description</li> <li>User defined settings</li> </ul>	Constant Z finishing   Order by Additional stock   Region 0.8   I Spiral Undercut   I Machine down to flats Image: Cut direction   Tolerance Cut direction   0.01 Climb   Thickness Image: Cut direction   0.0 </th		

12 In the main Leads and Links dialog:-

- In Lead in 1st choice set Surface normal arc with <u>Distance</u> 0, <u>Angle</u> 90, <u>Radius</u> 8.
- In Lead out 1st choice set Vertical arc with <u>Distance</u> 0, <u>Angle</u> 90, <u>Radius</u> 8.
- 13 Accept the Leads and Links dialog.
- 14 Select Calculate in the main Constant Z finishing dialog to create the Toolpath.
- **15** Close the Constant Z finishing dialog.



# Finish Machining – Tool EM16

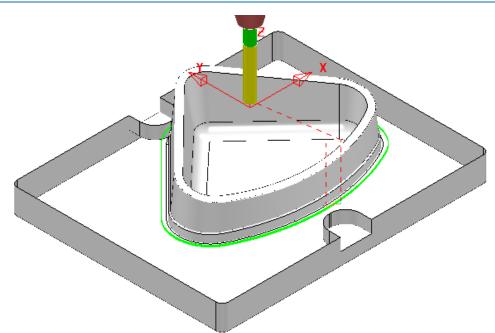
Re-open the OP2-D16TR3-FIN1 toolpath dialog and select Create a new toolpath based on this one

2 Rename the new **toolpath** as **OP2-EM16-FIN1**.

- 3 Activate the Tool EM16.
- 4 In the **Limit** page, **Z Limits** section, update the **Maximum** value to **-82** (this is approx. **3mm** above the top of the base surface).
- 5 In the *Constant Z finishing* page, **Stepdown** section, enter the data exactly as shown below.

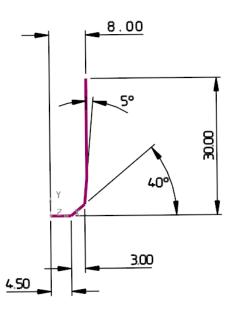
Stepdown		
	Minimum stepdown	0.02
	Calculate us	ing cusp 📝
	Maximum stepdown	0.2
	Cusp height	0.02

6 Select **Calculate** in the main **Constant Z finishing** dialog to create the **Toolpath**.



## Finish Machining – Tool FormTool-1

The 40 Degree angle on top of the step near the bottom of the 3D form will be machined with a **Form Tool**. The profile for the **tool tip** was created from a **Pattern** which in turn was constructed using the **Curve Editor** options.



The **Pattern** must be created in the **+ve X** and **Y** quadrant, relative to the **Transform** (Global Datum) for it to be valid when used in **Form Tool** creation options. Once the **Form Tool** has been created it will no longer be dependent on the **Pattern**.



The above **Tool** and original **Pattern** already exists in the start up **Project**.

- 1 Select a *View down Z* and un-draw all items except the existing **Pattern** *FormTool-1*.
- 2 Zoom into the lower area of the Pattern and from the main toolbar select the Measurer .

	Measure	? ×
	Standard Calculator Scientific Calculator Measure Measure Geometry	
•	X Y Z Direction I J K	
<b>→</b>		

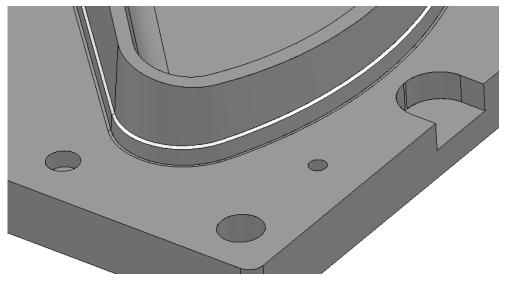
3 With the **Measure** tab selected, click on the **Open the position input** option

	🛪 Position ? 🗙
	Cartesian Polar Circle Between
	Workspace World V
1	0.0 0.0 0.0
	End point
1	7.5 2.517299 0.0
1	Along the line
2	Proportion v 0.5
20	Resultant point
	3.75 1.258649 0.0
	Length 7.911182
	Become origin
	Apply Accept Cancel

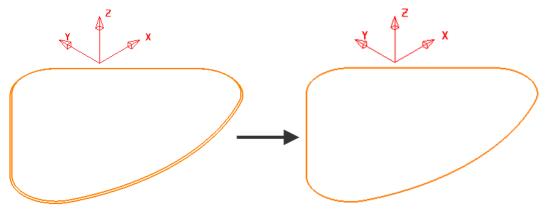
- 4 In the **Position** dialog select the **Between** tab
- 5 On the **Pattern** in the graphics area, left click on key point 1 followed by key point 2 (shown above left).
- 6 Make a note of the displayed **End point** values **7.5** and **2.517299**.

Next a new **Pattern** *segment* will be created along the top, inner edge of the 40 Degree chamfer (corresponding to *key point* **2** above). The above values will be used to offset (**7.5**) and move (**Z-2.517299**) the new **Pattern** *segment* to provide a path for the form tool centre (*key point* **1**).

7 Select all of the **surfaces** that form the top face of the step near the bottom of the 3D form (shown shaded light below).



- 8 In the **PowerMILL** *explorer* right mouse click on **Patterns** and from the local menu select **Create Pattern**.
- 9 Rename the new Pattern as OP2-Chamfer40Deg.
- 10 Right mouse click on the new Pattern and from the local menu select *Insert* -Model (*Pattern segments* will be created along the open edges of the *selected model*).



- 11 Select and Delete the outer Pattern segment (as shown above).
- 12 Right mouse click on the **Pattern** and from the local menu select **Curve Editor**.
- 13 Select the **Pattern** segment.
- 14 In the *Curve Editor* toolbar select **Offset item** (2D Round)



- **15** Input a **Distance** value of **7.5** and press return to accept the action.
- **16** Select the **Pattern** segment.

17 In the *Curve Editor* toolbar select Move item.

Move		
No. of Copies	1	 ×

18 In the Data Input box below the main graphics area enter and return the *relative* XYZ move values 0 0 -2.517299.



- Select the *green tick* to accept the changes and return the **Pattern** to **PowerMILL**.
- **20** Right click on the Pattern, segment and from the local menu select Instrument to display the direction arrow.
- 21 If the arrow is pointing along the upcut direction (anticlockwise if viewed from above) select **Edit Reverse Selected**.

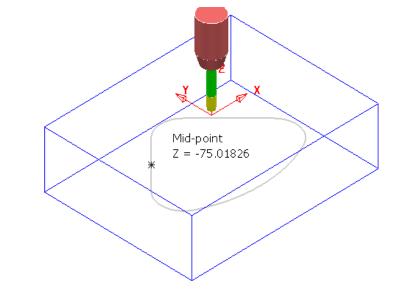
The **Pattern** is now suitably positioned for use as a *tool centre*, **drive curve** to be used in either a **2D Curve Profile** or **Pattern Finishing** strategy.

- 22 Ensure that no **Boundary** is **Active**.
- 23 Activate the Tool FormTool-1.
- 24 From the **Strategy Selector**, open the **2.5D Area Clearance** strategy dialog and select the **2D Curve Profile** strategy.
- 25 In the 2D Curve Profile dialog enter the Toolpath name as:-OP2-FormTool1-Chamfer.

Workplane   Block   Tool   Machine tool   Limit   Curve profile   II. Cut distances   Finishing   Finishing   Draft   Offset   Step cutting   Filiph speed   Order   Approach   Automatic verification   Cutter compensation   Cutter compensation   Point distribution   Tool axis   Machine axis control   Rapid move heights   Leads and links   Start point   Start point	1	Curve Profile	? >	۲
<ul> <li>Block</li> <li>Tool</li> <li>Wachine tool</li> <li>W Limit</li> <li>Curves definition</li> <li>OP2-Chamfer40Deg ∨ № № № № №</li> <li>Position ♥</li> <li>Pos</li></ul>	Toolpath name	OP2-FormTool1-Chamfer		
Freeds and speeds     Best machining     Notes and Description	Block  Tool  Machine tool  Machine tool  Machine tool  Machine tool  Machine tool  Machine arise  Finishing  Machine arise control  Automatic verification  Cutter compensation  Point distribution  Cutter compensation  Machine arise control  Rapid move heights  Machine arise control  Rapid move heights  Cutter point  Find point  Kaine arise point  Find	Curves definition Curves definition OP2-Chamfer40Deg Lower limit .75.01825 Style Tolerance 0.01 Curve thickness 0.0	Cut direction	

26 Set the Position option to The centre of the tool follows the pattern

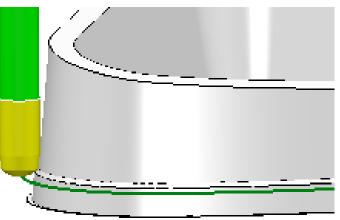
27 Select the **Pick the lowest Z Height** option and snap at any point along the drive curve, Pattern segment.





The snapped **Z height value** is inserted into the **Lower limit** box.

- In the main Leads and Links dialog select Lead in/out 1st choice set Horizontal arc with <u>Distance</u> 0, <u>Angle</u> 90, <u>Radius</u> 12.
- 29 Accept the Leads and Links dialog.
- **30** Select **Calculate** in the main **Offset flat finishing** dialog to create the **Toolpath**.



The Form tool is accurately aligned to the angle across the top of the step.

**31 Close** the **2D Curve Profile** dialog.

# Hole Feature – Drilling

**Drilling** strategies can only be performed on **Hole Features** and not directly on the component **Model**.



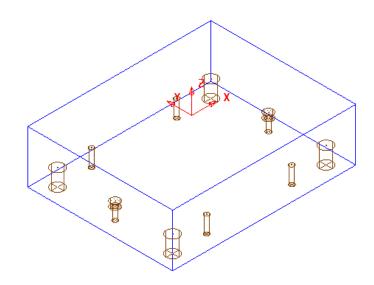
To ensure that the **Hole Features** are created vertical relative to the active **Z** axis the **Block** must first be <u>deleted</u>.

<b>3</b>	Block		?	×
Defined by				
Box	~		D 📔	
Coordinate System				
Active Workplane	~			~
Limits Min	Max		Length	
X 0.0	0.0	S	0.0	S
Y 0.0	0.0	S	0.0	S
Z 0.0	0.0	S	0.0	S
X Y Z	X X Z		<b>6</b>	<mark>0-1</mark>

- 1 From the **main** toolbar select the **Block** dialog and click on the **blue cross** to delete the current **Block** definition.
- 2 Select the whole of the component **model**.
- 3 In the **PowerMILL** *explorer*, right mouse click on **Feature Sets** and from the local menu select **Create Holes**.

Ci Ci	reate Holes	?	×
Name root			
Н			
Create from T	olerance		
Model 🗸	0.02		
Create compound hole	s		
Use active workplane o	nly		
O Find holes in both d	irections		
Find holes going do	wn		
○ Find holes going up			
Group holes by axis			
Try to recognise partial	holes		
Minimum arc angle for	partial holes		
300.0			
Ignore capped cylinder	s		
Edit after creation			
Apply	Close		

4 Enter the data into the Feature dialog exactly as shown above before selecting Apply.

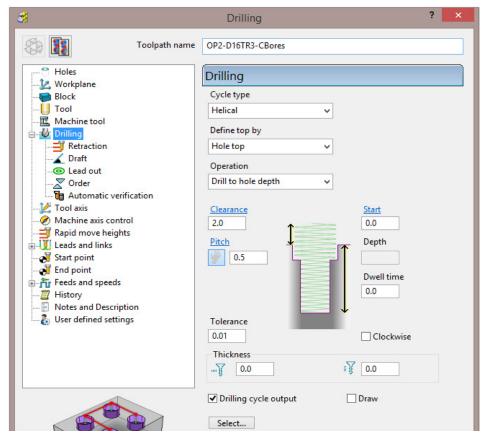


The Feature Set contains a total of 12 defined Hole Features.



Most of the **Hole** detail has been dealt with in **OP1** so the only remaining **Features** that require machining from this side are the 2 Counterbores.

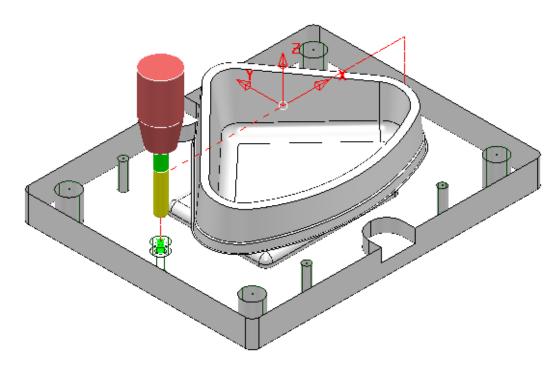
- 5 Rename the new Feature Set as OP2-Holes.
- 6 Activate the toolpath OP2-D16TR3-FIN1 to reinstate the Block and activate the Tool D16TR3.
- 7 From the **Strategy Selector**, open the **Drilling** strategy dialog and select the **Drilling** strategy.



8 Select *Cycle type* Helical and input remaining data into the **Drilling** dialog exactly as shown above and then click on the **Select** tab.

Feature Selection	? ×	<b>.</b>	Feature Selection ? ×
Select By Diameter	~		Select By Diameter 🗸 🗸
Minimum Diameter		Min	imum Diameter
Maximum Diameter		Max	kimum Diameter
Diameter Selection Filter		Diameter 12.00 30.00	Selection Filter ⊡-Ø Diameter Ø 22
30.00	-	▶	> <
< >	×	< >	
Tolerance Component		Tolerance 0.01	Component
0.01 Largest	~	0.01	Largest 🗸
Select Close			Select Close

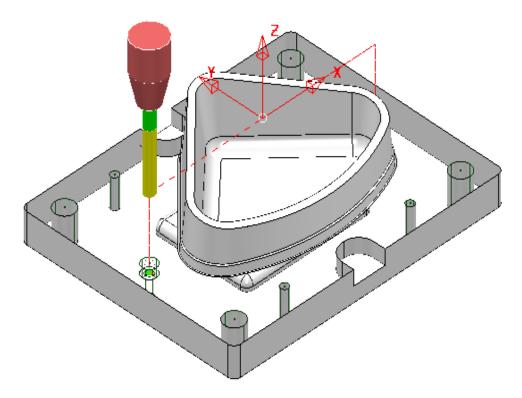
- 9 In the **Diameter** box select **22.00** and click on **b** to add these holes to the **Selection Filter** box.
- **10 Close** the **Feature Selection** dialog and select **Calculate** in the main **Drilling** strategy dialog.



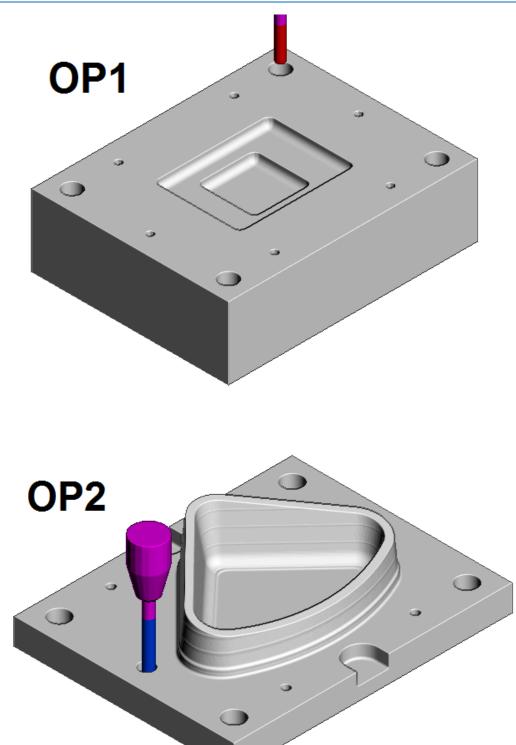
11 Make a Copy of the last Drilling Toolpath named OP2-EM16-CBores and this time use the Tool EM16 and Cycle Type - Profile on both of the counterbore Features.

đ	Drilling ? ×
Toolpath name	e OP2-EM16-CBores
Holes Workplane Block Tool Ketraction Fetraction Fetraction Fetraction Fetraction Forfile Forfile Forfile Forfile Forder Automatic verification Forder	Drilling   Cycle type   Profile   Define top by   Hole top   Operation   Drill to hole depth   Drill to hole depth     Start   9.5   Peck Depth   Image: Clearance   5.0   Peck Depth   Image:
BBB	Drilling cycle output     Draw     Select

**12** Input data into the **Drilling** dialog exactly as shown above and **Calculate**.



**13** Perform **ViewMILL** simulations for both **OP1** and **OP2**.

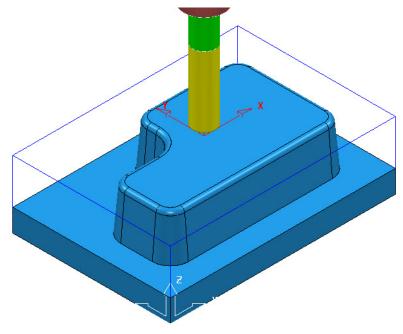


14 Save the Project to update:-

....\COURSEWORK\PowerMILL\_Projects\CornerBowl-Example

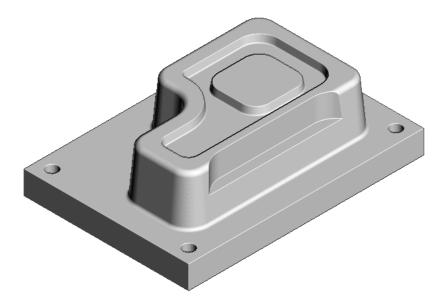
# 2.5D Machining Example

- 1 Select File > Delete All and Tools > Reset Forms
- 2 Open the (read only) Project:-
  - ....\PowerMILL\_Data\Projects\ 2DCurveMachining-2-Start



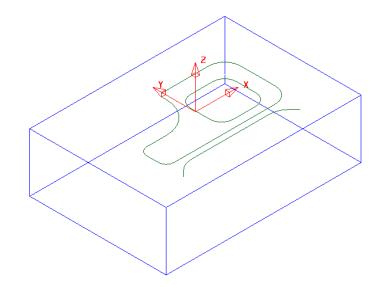
- 3 Save Project As:
  - ....\COURSEWORK\PowerMILL\_Projects\2DCurveMachining-2

The **Project** already includes **Toolpaths** to rough and finish machine the 3D model form. It is however required to create additional recesses into the top face (7.5 deep with 10 Deg Draft) and front wall (25 deep – No draft) and 4 Dia 12 holes near the corners of the base (offset 15 from the outside of the base).

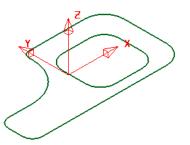


# Create individual Patterns for the Top Recess, Front Recess, and Base Corner Holes:-

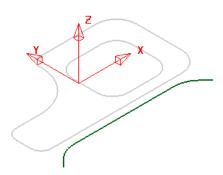
The **Model** also includes **2D** curves which can be made visible by switching on **Wireframe** in the **Views** toolbar.



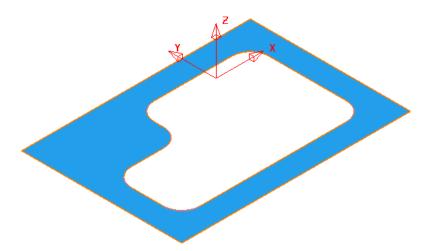
- 4 Create a new **Pattern** (1).
- 5 Select both of the closed *wireframe curves* on the **Model**.
- 6 In the **PowerMILL** *explorer* right mouse click on the new **Pattern** and from the local menu select **Insert Model**.



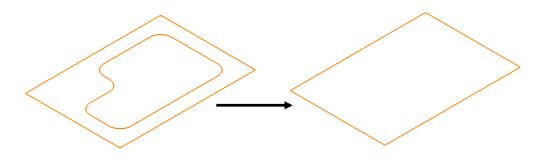
7 Create another new **Pattern** (2) and this time **Insert** the open *wireframe curve* into it.



- 8 Create another new **Pattern** (3).
- 9 Select the upper base surface on the Shaded model and Insert it into the new Pattern.



**10** Select and **Delete** the inner **Pattern** *segment*.



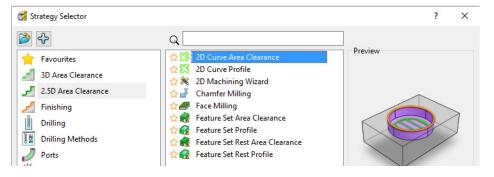
- 11 Double left mouse click on the remaining **Pattern** segment in the graphics area to access the **Curve Editing** toolbar.
- 12 Offset the *selected segment* inwards by 15.

¥, XX	Ŵ		R		*, (			2			+ <sub>4</sub>	7	0	~
Off:	_	N	o, of C	opies	1	~	D	istan	ce -	15		~	×	

**13** To accept the edits, select the *green tick* at the end on the **Curve Editor** toolbar.

## Machine the top recess with a 10 degree draft angle:-

- 14 Activate one of the existing toolpaths to reinstate the settings (Workplane, Block, and Rapid Move Heights).
- 15 Activate the D12TR1 tool.
- 16 From Toolpath strategies open the 2D Curve Area Clearance dialog.



17 Input the data and settings exactly as shown in the following images.

Curve Area Clearance		? ×
Toolpath name	D12TR1-TopRecess-RGH1	
Workplane A	Curve area clearance	
<mark>E</mark> Machine tool <mark>E</mark> Machine tool	Curves definition	- 🔀 🔉 🛠 🕅
Curve area clearance		Position 🍖 🗸
····▲ Draft ····▲ Offset ····▲ Step cutting	Lower limit	
····· J F High speed ····· J Approach ····· J Automatic verification	Style	Offset ~
Cutter compensation  Cutter compensation  Control  Cutter compensation  Cutter compensation		
Rapid move heights	Tolerance 0.1	Cut direction
	Curve thickness 0.5	
Notes and Description	Stepover	
	Rest machining	

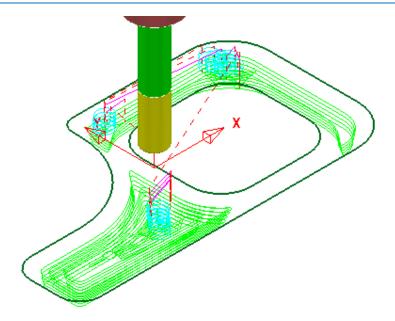
**18** Select to preview and if required apply the **Pattern Decoration** options to modify the machining area and directions.

🥳 Curve Area Clearance		?	×
۵ 🚺	Toolpath name	D12TR1-TopRecess-RGH1	
Workplane Block Glock Gl	^	Cut distances         Vertical         Extent       Stock depth (d)         Stock depth       7.5         Stepdown (t)             1.5	
···· <sup>™</sup> Step cutting ···· <sup>™</sup> High speed ···· <sup>™</sup> Order ···· <sup>™</sup> Approach ···· <sup>™</sup> Automatic verificat	tion	Horizontal Stepover 4.0	

🥳 Curve Area Clearance			2	?	×
ون الله الله الله الله الله الله الله الل	Toolpath name	D12TR1-TopRecess-RGH1			
Workplane	^	Draft			
Block					
U Tool					
Machine tool		✓ Draft			
			Fr	om 🔽 🗸	
📄 🚊 💥 Curve area clearance					
			<u>Angle</u> (Θ)	) 10.0	
- 🛃 Finishing					
Draft		Area filter			
Offset			Threshold (TDU)	0.95	
Step cutting					

🥳 Curve Area Clearance			?	$\times$
الله 🕄	Toolpath name	D12TR1-TopRecess-RGH1		
	•	Offset Advanced Offset Settings Style Remove cusps Spiral		

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



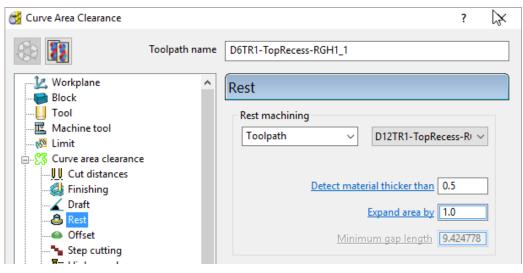
20 Activate the D6TR1 Tool.

#### **21** From **Toolpath strategies** open the **2D Curve Area Clearance** dialog.

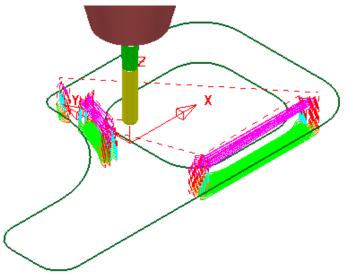
**22** Input the data and settings exactly as shown in the following images.

🥳 Curve Area Clearance		<b>A</b>		?	×
Toolpat	th name	D6TR1-TopRecess-RGH1_1			
	^	Curve area clearance			
····· <mark>U</mark> Tool ····· <b>⊠</b> Machine tool ····· <b>⊠</b> Limit □··· <mark>·S</mark> Curve area clearance		Curves definition	~ 🕅 🕅	\$ \$	
		Lower limit	Position	<b>∕</b> ⊕ ~	
		Style	Offset	~	]
Machine axis control Rapid move heights Leads and links Start point End point Feeds and speeds History	~	Tolerance 0.1 Curve thickness 0.5 Stepover	Cut direction Climb	~	]
		Rest machining			

📆 Curve Area Clearance	2		?	×
٠	Toolpath name	D6TR1-TopRecess-RGH1_1		
Workplane	^	Cut distances		
Tool     Machine tool     Machine tool     More Limit     S Curve area clearance		Vertical Extent <u>Stock depth (d)</u> Stock depth $\checkmark$ 7.5		
		Stepdown (t)		
& Rest & Offset <mark>*</mark> Step cutting		Horizontal Stepover		
High speed Order J Approach		¥ 1.5		



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



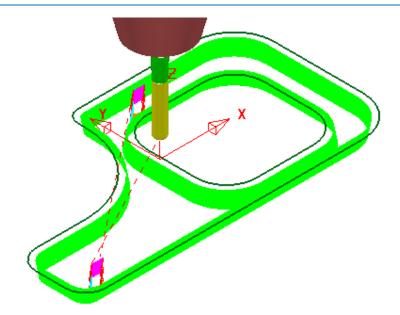
- 24 From Toolpath strategies open the 2D Curve Profile dialog.
- **25** Input the data and settings exactly as shown in the following images.

🥳 Curve Profile	? ×
Toolpath name	D6TR1-TopRecess-FIn1
<ul> <li>Workplane</li> <li>Block</li> <li>Tool</li> <li>Tool</li> <li>Machine tool</li> <li>Limit</li> <li>Curve profile</li> <li>Curve profile</li> <li>Finishing</li> <li>Oraft</li> <li>Offset</li> <li>Step cutting</li> <li>F High speed</li> <li>Order</li> <li>Approach</li> <li>Automatic verification</li> <li>Cutter compensation</li> <li>Point distribution</li> </ul>	Curve profile
Tool axis Machine axis control April move heights Cleads and links Start point Feeds and speeds History Notes and Description	Tolerance     Cut direction       0.01     Climb ✓       Curve thickness     ✓       0.0     ✓

🥳 Curve Profile		?	×
Toolpath name	D6TR1-TopRecess-FIn1		
Workplane Block Machine tool Kachine tool Curve profile Curve profile	Cut distances         Vertical         Extent       Stock depth         Stock depth       7.5         Stepdown (t)             0.25	th (d)	
Step cutting  High speed  Order  Approach  Cutter compensation  Cutter	Horizontal Extent <u>Stock widt</u> Stock width <u>Stepover</u> 1.5	b b b b b b b b b b b b b b b b b b b	

26 Select Calculate to process the toolpath and then Close the dialog

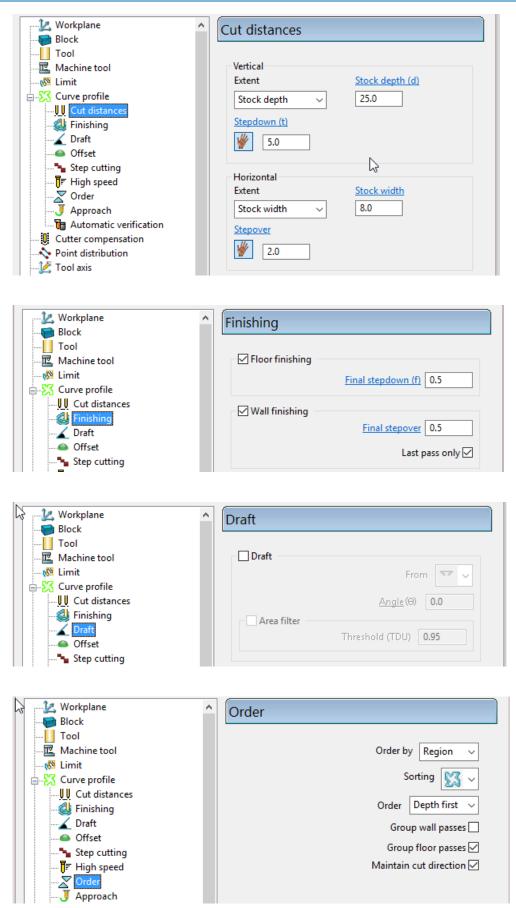
19 - 30



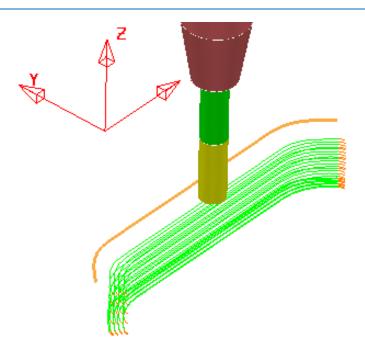
# The next stage is to machine the vertical, front recess:-

- 27 Activate the EM12 Tool.
- 28 From Toolpath strategies open the 2D Curve Profile dialog.
- **29** Input the data and settings exactly as shown in the following images.

🥳 Curve Profile		? ×
Toolpath nam	EM12-FrontRecess-All	
Workplane Block Gamma Block Gamma Block Machine tool Machine tool Curve profile Curve profile Curve profile Curve profile Curve profile Curve profile Finishing Curve profile Finishing Curve profile Finishing Finish	Curve profile Curves definition 2 Lower limit Curves definition 2	✓ ∑ ∑ ♣ ∑ Position  √ ✓
	Tolerance 0.01 Curve thickness 0.0 Rest machining	Cut direction

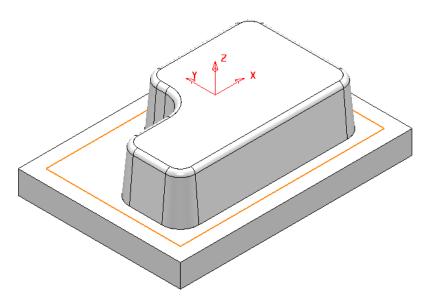


**30** Select **Calculate** to process the toolpath and then **Close** the dialog.



## Drill four Dia 12 holes offset 15mm from the base walls:-

The next stage is to create a **Feature Set** defining four Dia 12 through holes down from the corners of **Pattern 3**.

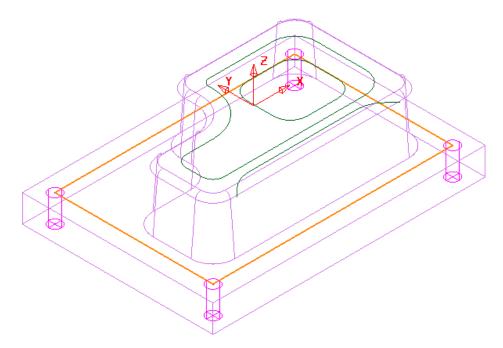


- **31** Select the rectanglar **Pattern 3** located on top of the base surface.
- 32 In the **PowerMILL** *explorer* right mouse click on **Feature Sets** and from the local menu select **Create Holes**.

- **33** In the **Create Holes** dialog select the option *create from* **Points**.
- 34 Input the **Absolute** values **-50** (top) and **-75** (bottom).

Create Holes	? X
Name root	
Create from Tolerance	]
Model v 0.1	
Points Circles oles Model Pairs e only	
Curves Lines Plunges Normals O Find holes going up	
Group holes by axis Create holes from partial cylinders Minimum span angle 300.0	
Ignore capped cylinders Edit after creation	

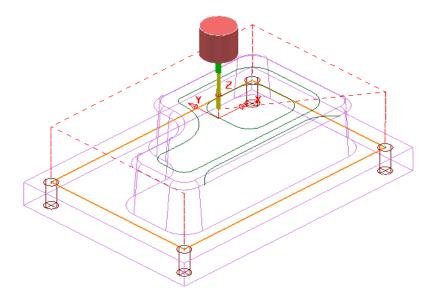
35 Select **Apply** to create four Dia 12 Hole **Features**.



- **36 Activate the Dia 3 Drill**.
- 37 From Toolpath strategies select the Drilling Single Peck option.
- **38** Input the data and settings exactly as shown in the following image.

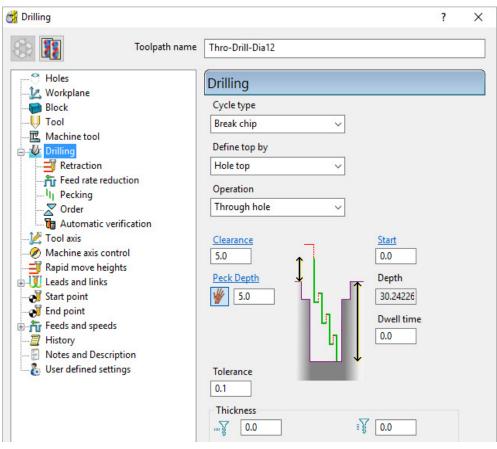
🥳 Drilling		?	×
Toolpath name	C-Drill-Dia3		
Holes Ho	Drilling       Cycle type       Single peck       Define top by       Hole top       Operation       Centre drill		
Tool axis     Machine axis control     Appid move heights     Leads and links     Start point     Feeds and speeds     History     Notes and Description     User defined settings	Start     Start       5.0     0.0       Peck Depth     0.0       Depth     1.5       Dwell time       0.0		
	Tolerance 0.1 Thickness 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0		
	Drilling cycle output Draw		
	Calculate Queue OK Cancel		

**39 Calculate** the Drilling toolpath and then select **Re-cycle I** the strategy.

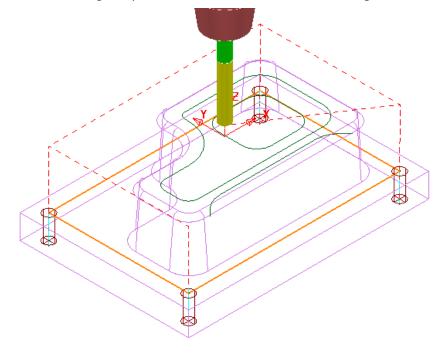


#### 40 Activate the Dia 12 Drill.

41 Input the data and settings exactly as shown in the following image.



**42 Calculate** the Drilling toolpath and then **Close** the above dialog.



- 43 Save the Project to update:-
  - ....\COURSEWORK\PowerMILL\_Projects\2DCurveMachining-2