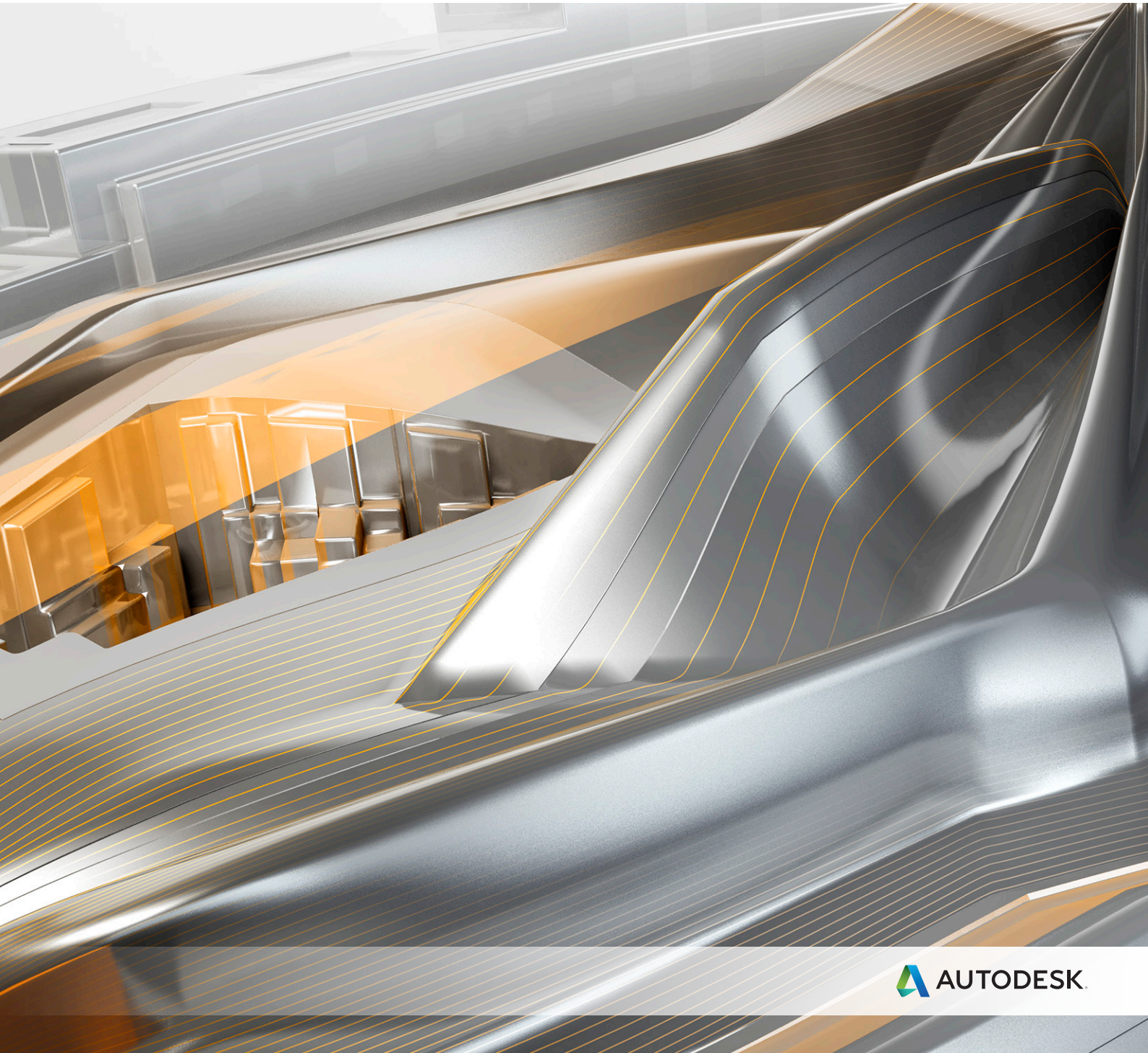
AUTODESK® **POWERMILL**®

# The expert high-speed and multi-axis solution

## Training Course



AUTODESK.

Copyright © Autodesk

# PowerMill 2017 Contents


| | Chapters | Page Number |
|---|---|---|
| **Day 1** | 1. Getting Started | 1.1 - 1.26 |
| | 2. Machining Set Up in Detail | 2.1 - 2.22 |
| | 3. Area Clearance Strategies | 3.1 - 3.54 |
| **Day 2** | 4. Finish Machining Strategies | 4.1 - 4.54 |
| | 5. Leads and Links | 5.1 - 5.14 |
| | 6. Boundaries | 6.1 - 6.32 |
| | 7. Levels and Sets | 7.1 - 7.12 |
| **Day 3** | 8. Editing Toolpaths | 8.1 - 8.22 |
| | 9. Tool Collision Checking | 9.1 - 9.14 |
| | 10. Patterns | 10.1 - 10.18 |
| | 11. Curve Editor | 11.1 - 11.12 |
| **Day 4** | 12. 2D Machining | 12.1 - 12.48 |
| | 13. NC Programs | 13.1 - 13.10 |
| | 14. Setup Sheets | 14.1 - 14.8 |
| | 15. Specialist Finishing | 15.1 - 15.32 |
| **Day 5** | 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 |


Copyright © Autodesk

Copyright © Autodesk

# 1. Getting Started

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

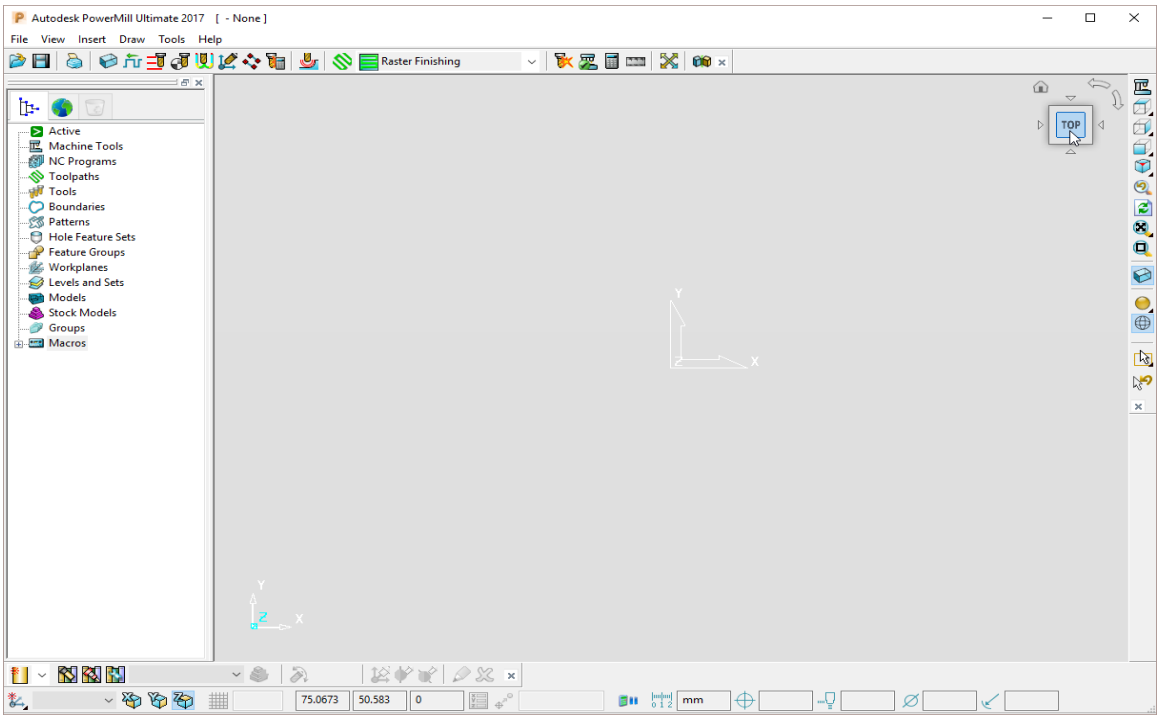
**PowerMill** can quickly create gouge-free cutter paths on imported component data. It supports Wireframe, Triangle, Surface, and Solid models. Translators can be purchased for **PowerMill** to directly import data created by other proprietary CAD packages.

## Start PowerMill

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

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

The **PowerMill** screen is displayed:



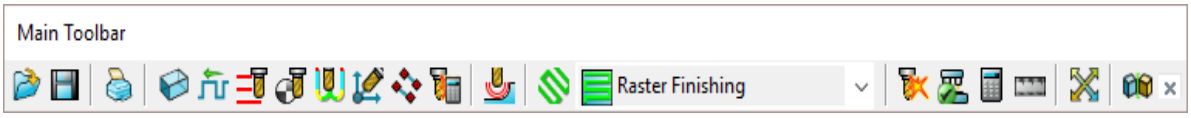

The screen is divided into the following main areas:

a **Main pulldown menus** (Located top left above the graphics area).

File View Insert Draw Tools Help

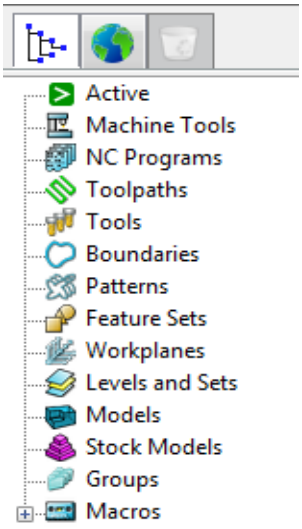
Click one of the menu names on this bar (for example, **File**) to open a pull-down menu list of associated commands and sub-menus. A sub-menu is indicated by a small arrow to the right of the text (for example **File - Recent Projects >**). Highlighting this arrow generates a list of commands/names 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** **PowerMill Explorer** (Left of the Graphics area).

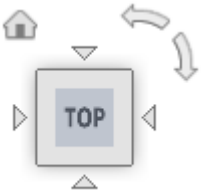



The **PowerMill *explorer*** provides access and storage of **PowerMill** entities created during the session.

**d** **Graphics area**

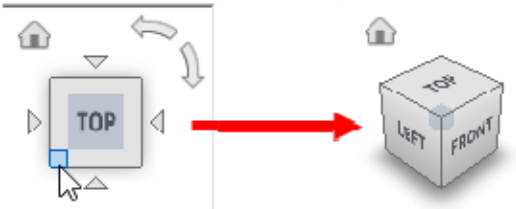
This is the large, visual display area to the right of the **Explorer** (Look at the illustration on previous page).

In the top right corner of the Graphics area is the **ViewCube**.




This is used to quickly select a range of standard viewing orientations.

For an **ISO1** view click the lower left corner of the **ViewCube** (arrowed ).





Copyright © Autodesk

To return to the default **Top** view select the **Home** symbol to the top left of the ViewCube (arrowed ).

Another method would have been to select the central Top area (arrowed ).

To enable 90 incremental rotation on the planer views select one of the circular arrows (arrowed ).

**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.

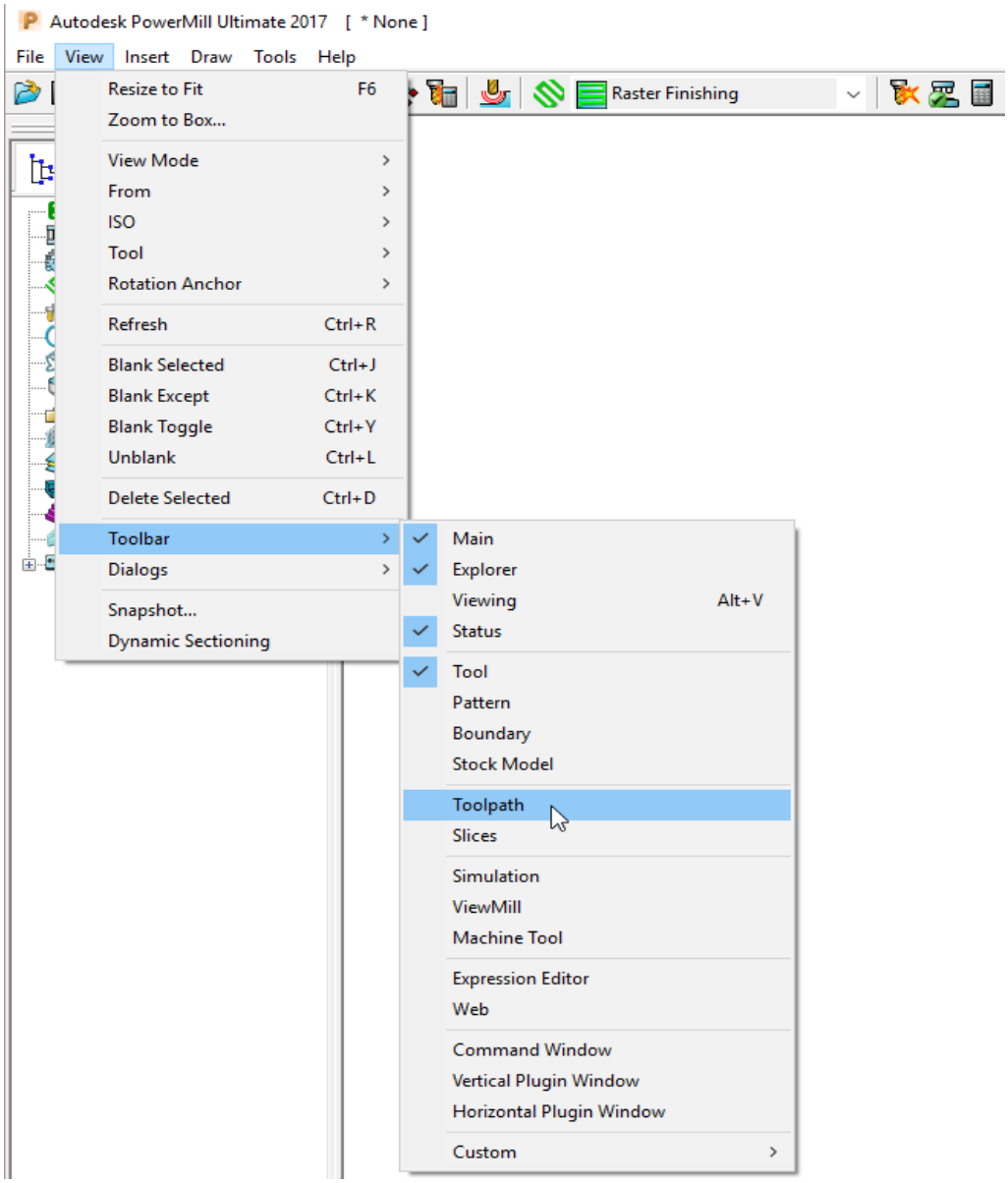
**g** **Tool Toolbar**

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**.

Copyright © Autodesk



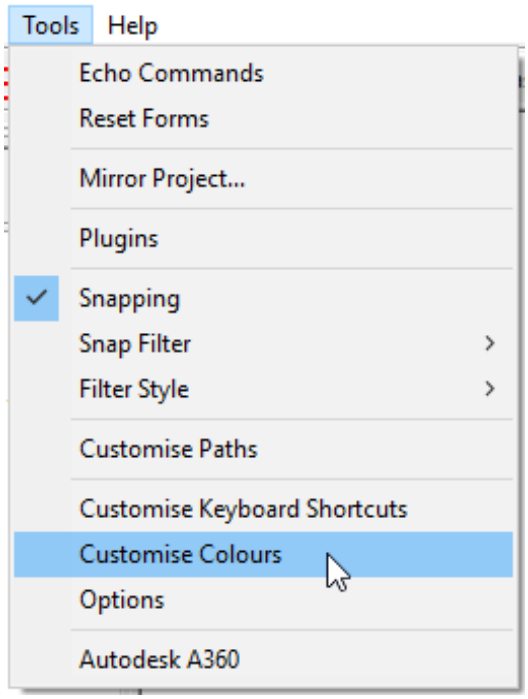

The **Toolpath** toolbar appears (as shown below).

The **Toolpath** toolbar provides access to a comprehensive selection of essential **Toolpath *editing*** and ***visualisation*** options.

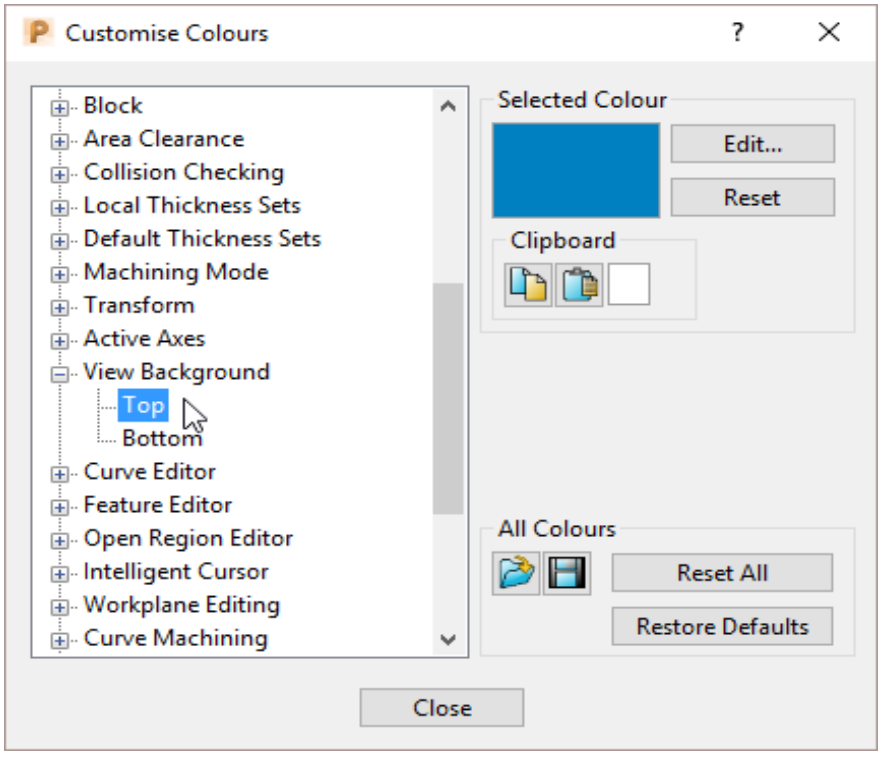

Copyright © Autodesk

# Changing the Background Colour.

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




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




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

Copyright © Autodesk

# 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.

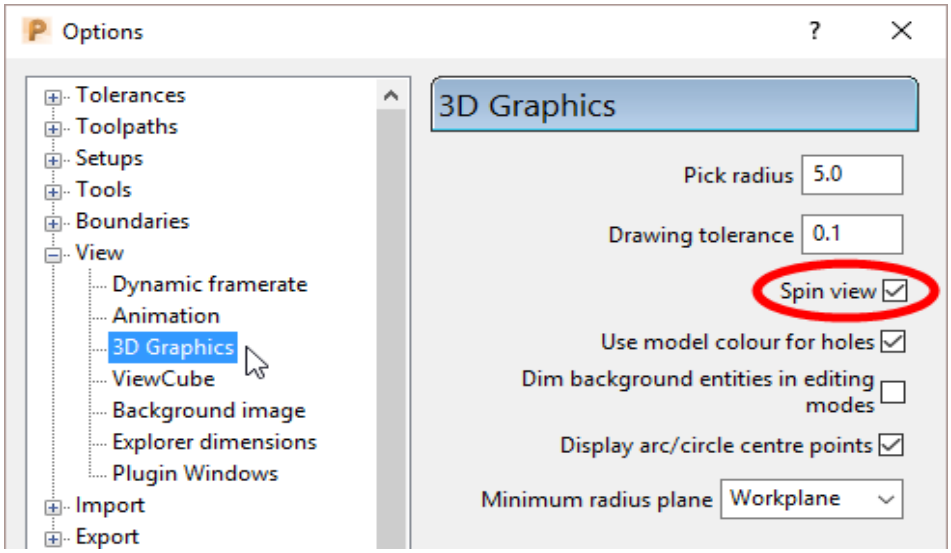
## Select by dragging the cursor

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.


Copyright © Autodesk

# Mouse button 2 (Middle) : 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.




# Mouse button 3 : Special Menus & PowerMill Explorer Options

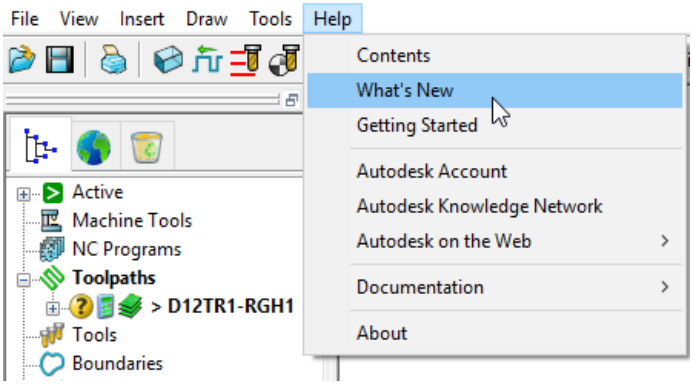
When this button is pressed it brings up a local menu relating to whatever the mouse is over, such as a named item in the **PowerMill explorer** or a physical entity in the graphics area. If nothing specific is selected, the View menu appears.

Copyright © Autodesk

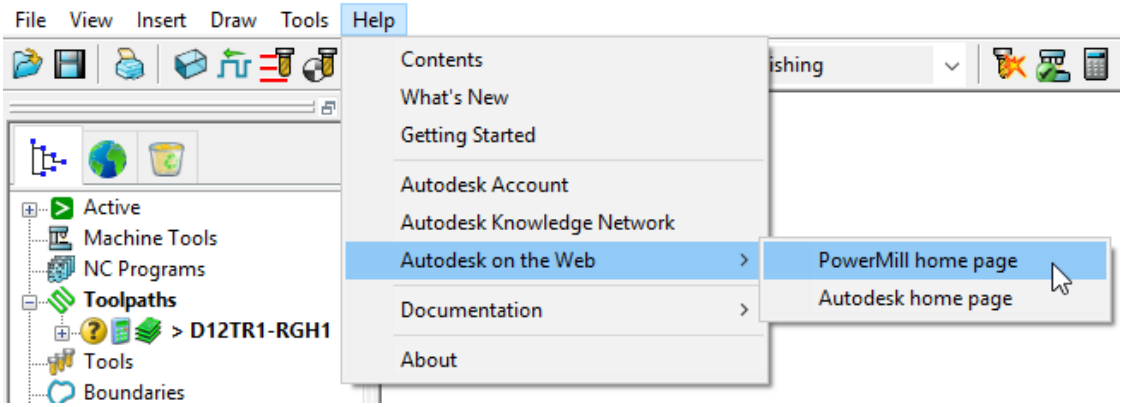
# HELP!

**PowerMill** provides you with help in a number of ways:

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




- **Context Sensitive Help**: Press the **F1** key to display the help page for the currently active dialog. Click the ? button in the top right hand corner of a 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** > **Autodesk on the Web** > **PowerMill home page** and then select **Support/Services – Forums** to participate in web-based user discussions on **PowerMill** issues.




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


Copyright © Autodesk

# 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.

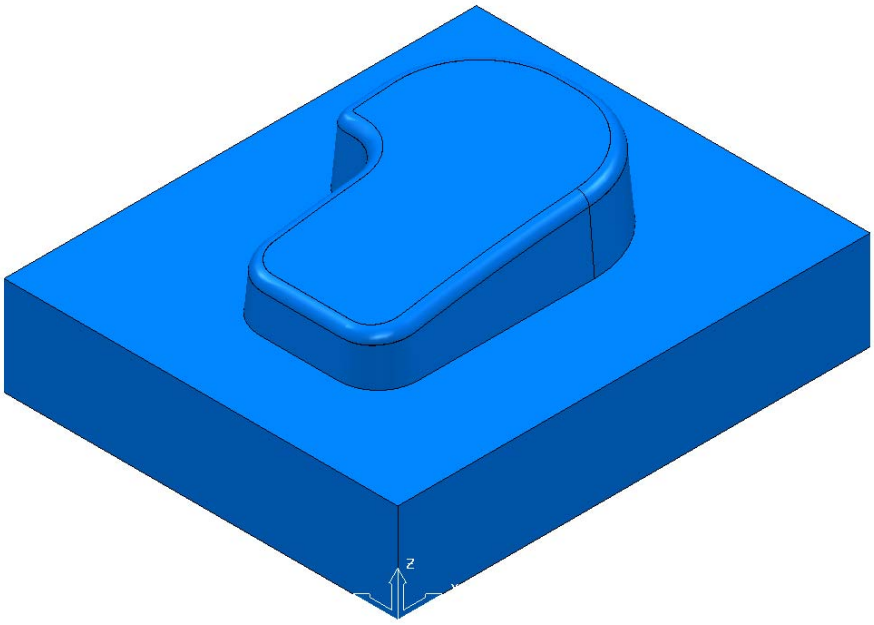
i Create an **NC Program** and output as a post-processed NC data file.

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**

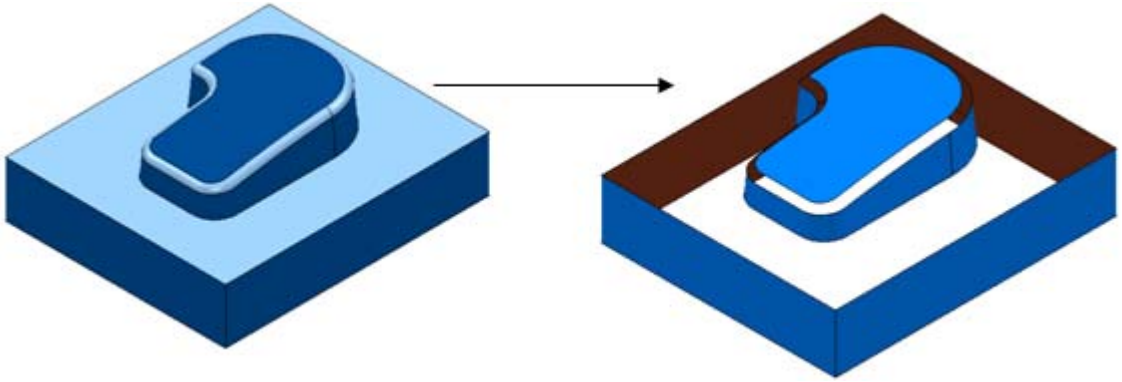


3 Select an **ISO 1** view.

# Blanking model entities

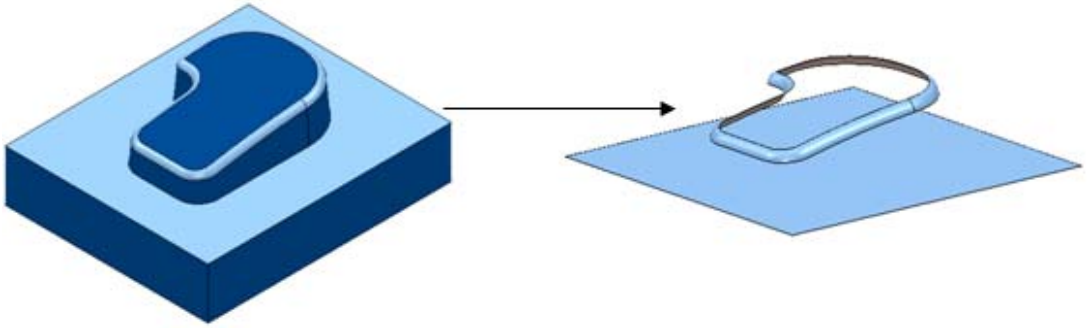
Use the **Blank Selected** (**Ctrl + J**) option available on the **Model** context menu to temporarily hide the selected model components.

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



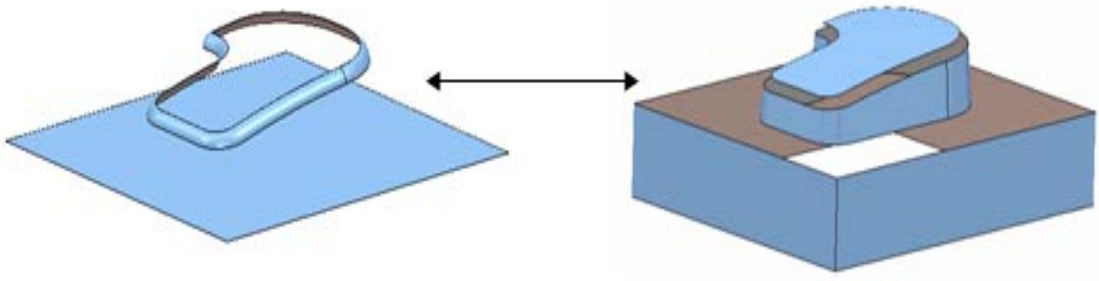
Use the **Blank Except** (**Ctrl K**) option available on the **Model** context menu to temporarily hide all model components except the selected ones.

The 2 selected items are light blue.



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

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




Copyright © Autodesk

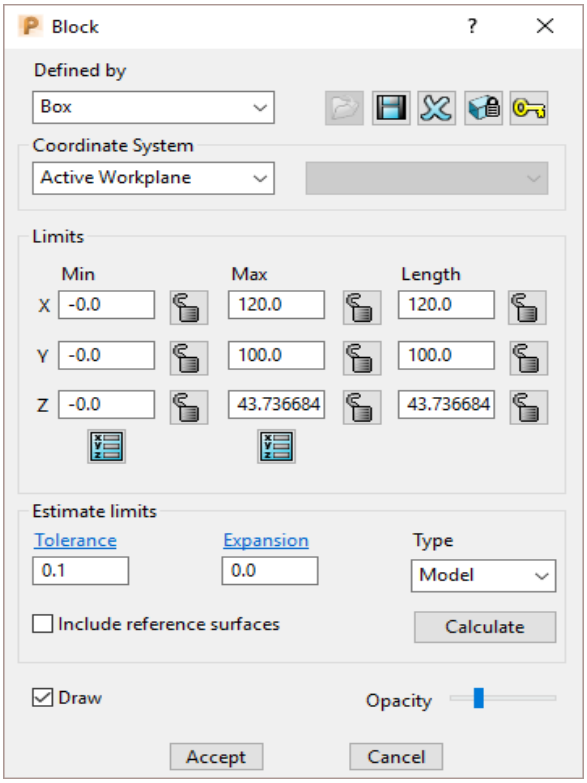
# Defining a Block

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

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

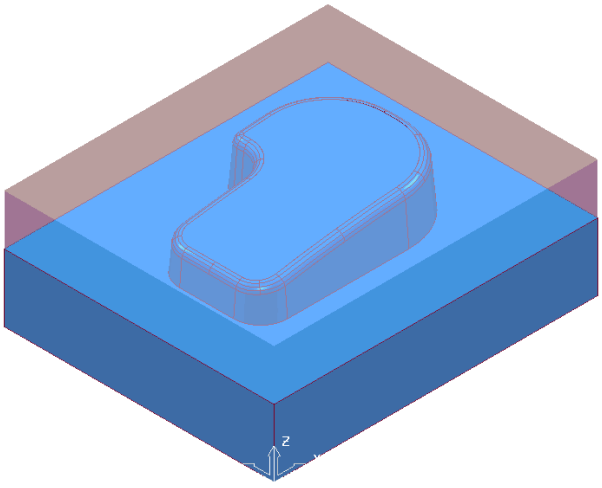
Click the **Calculate** button to define a block around the model dimensions.

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




2 Click the **Calculate** button.

3 Click **Accept**.



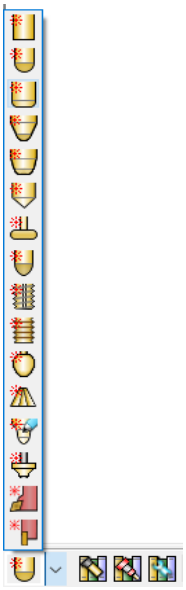
# Defining a Cutting Tool

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.

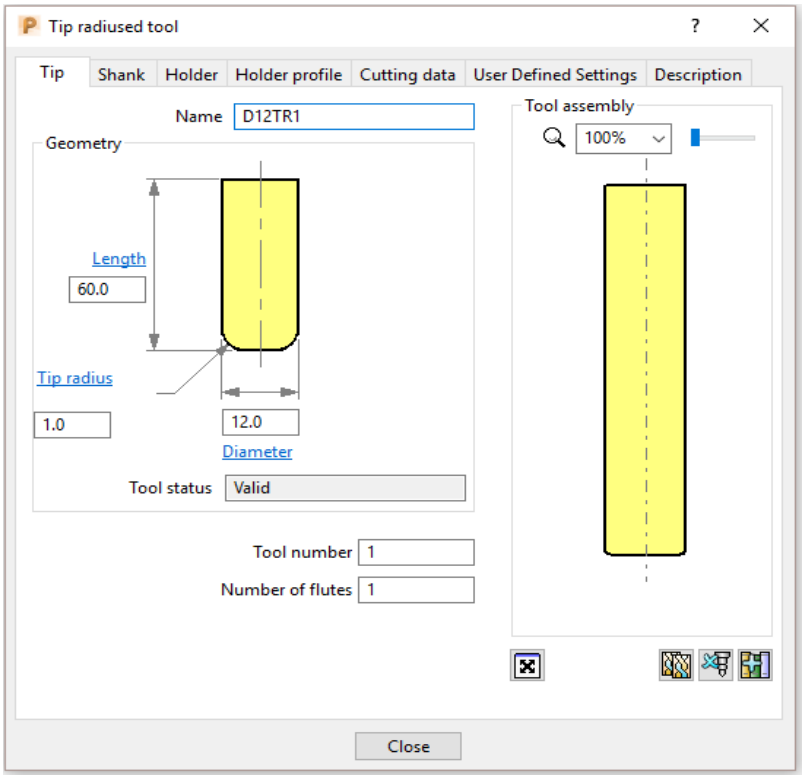
1 Click the down arrow to display all the **Tool** options .

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 .

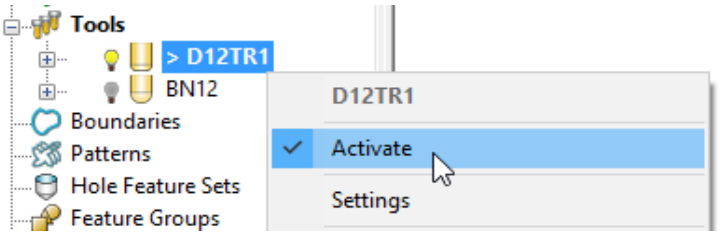
The **Tip Radiused Tool** dialog is displayed.





Copyright © Autodesk

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**.




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

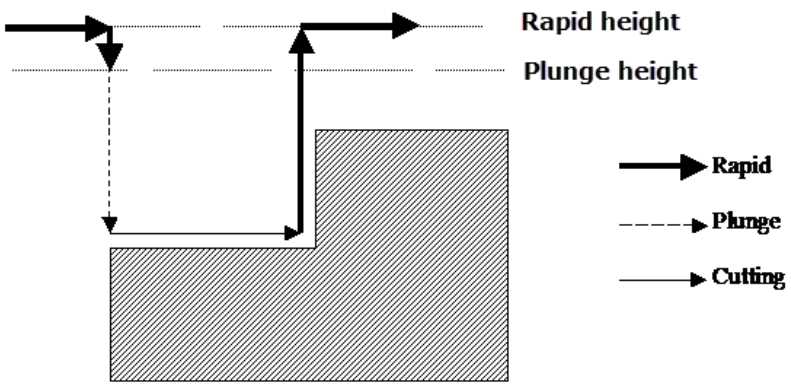
*In the **PowerMill explorer**, the **Active** tool is displayed in **bold text** and prefixed with '**>**'.*

# Rapid Move Heights

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

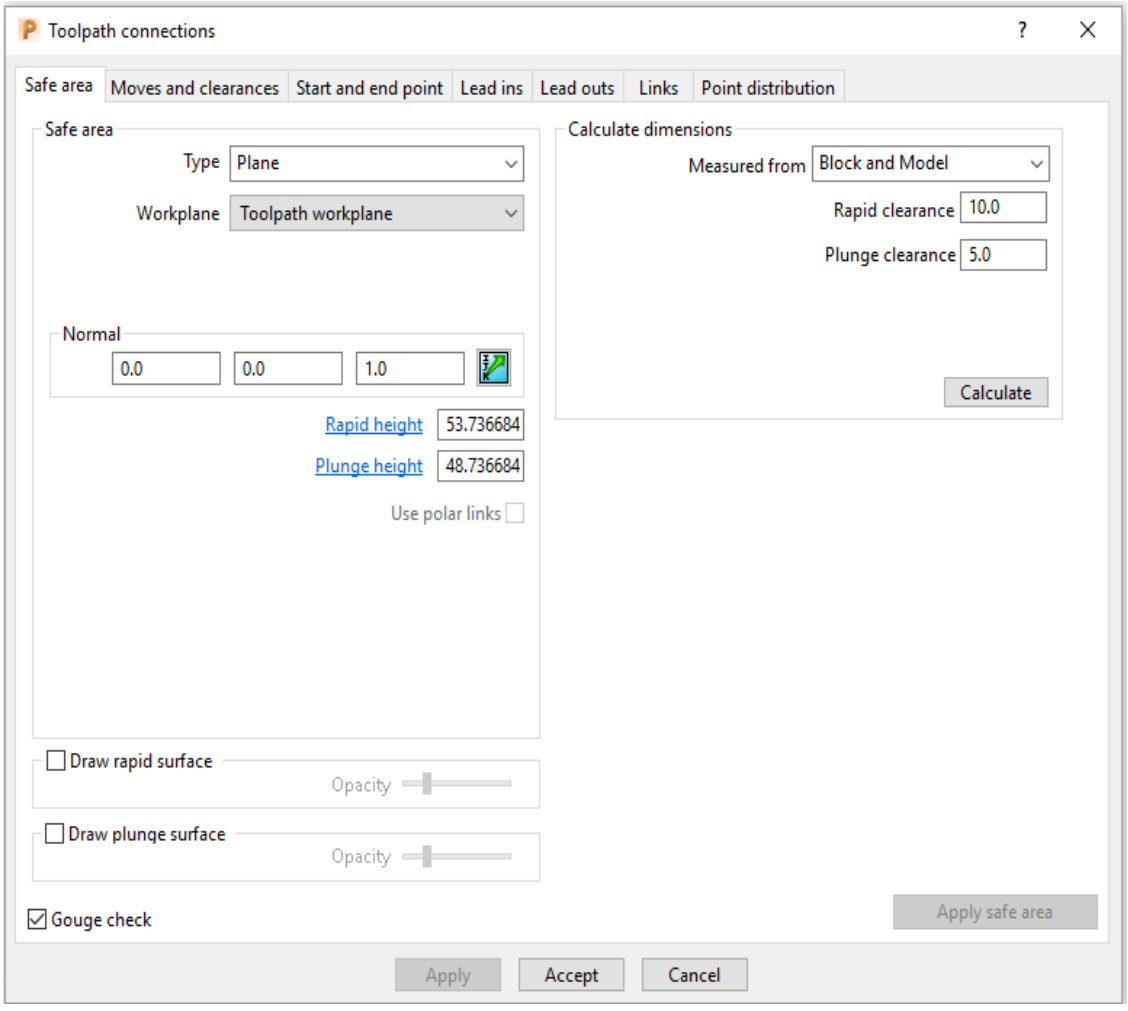
- **Rapid height** is the height above the job at which the tool can move at a rapid feedrate, clear of any obstructions such as the work piece or clamps.
- **Plunge height** is the height to which the tool will descend, at rapid feed rate prior to applying the plunge feed rate.

**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 open the **Toolpath connections** dialog.




2 Click **Calculate**.

3 Click **Accept**.

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

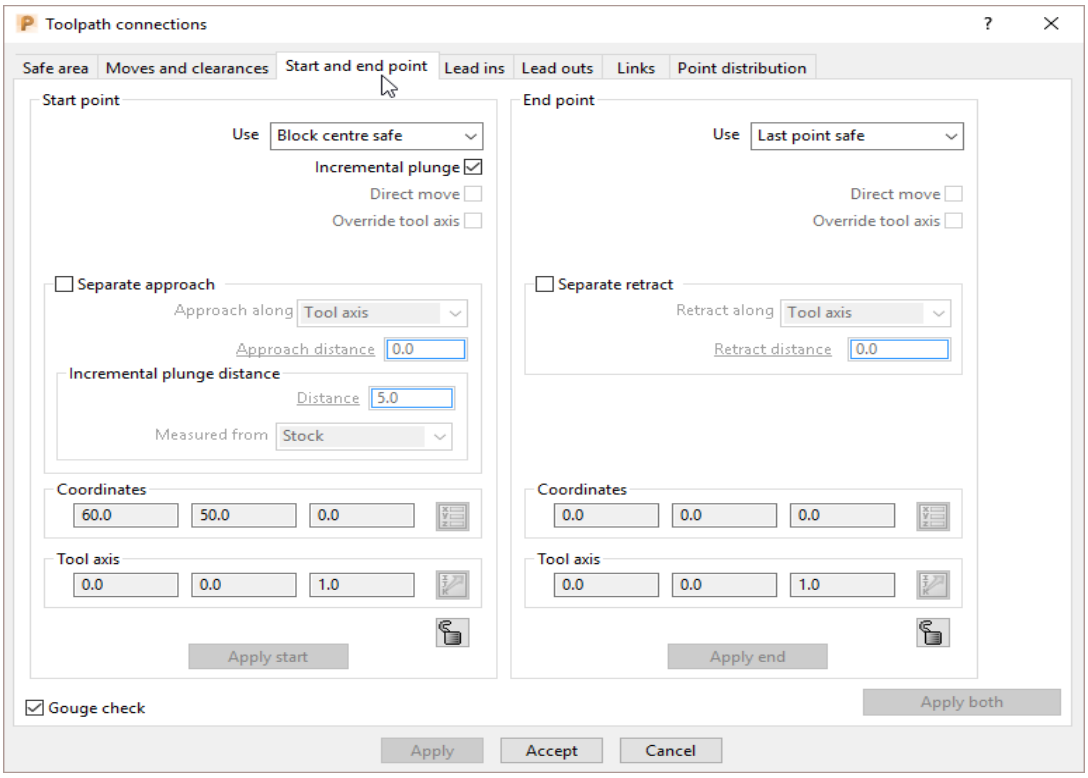

Copyright © Autodesk

# Toolpath - 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.

1 Click the tool **Start and end point** button on the **Main** toolbar to display the **Toolpath connections** dialog.

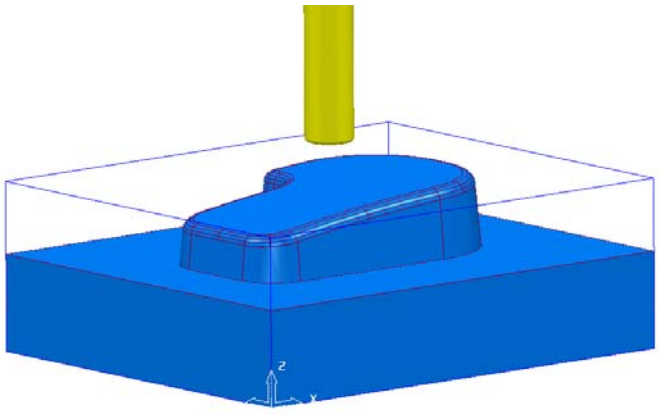
By default, the tool **Start point** is set to **Block centre safe** and the **End point** is set to **Last point safe**.




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

These include **Block centre safe**, **First / Last point safe**, **First / Last point**, and **Absolute**.

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

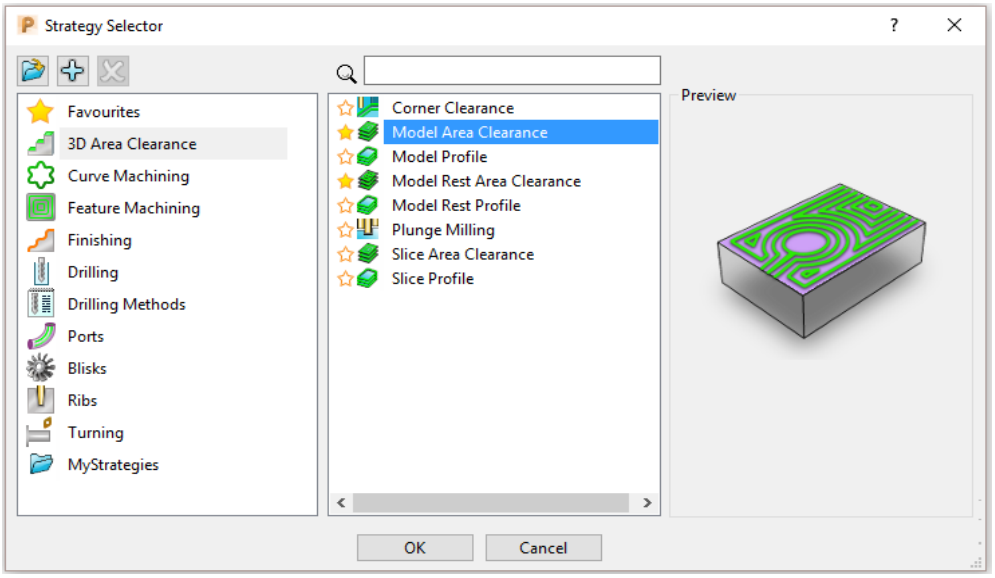


The **D12TR1** tool is placed at the **Block centre safe** position ready for you to create the first toolpath.

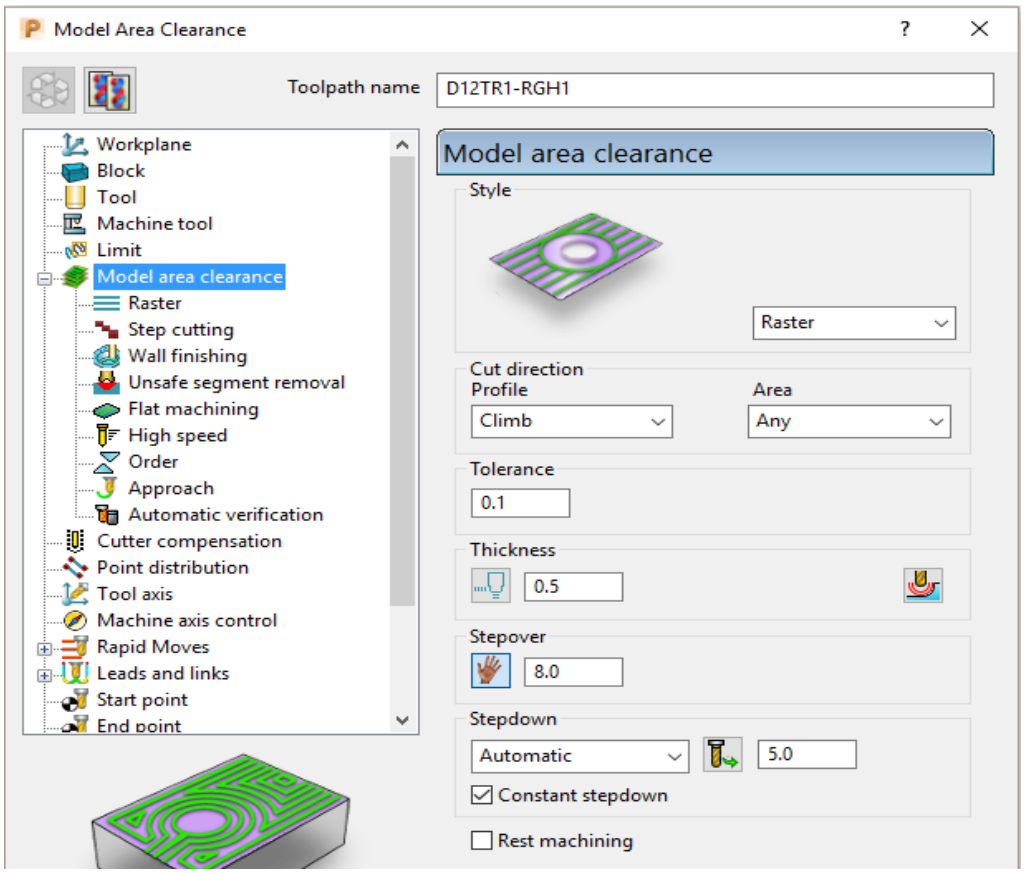
# Creating a Roughing Strategy

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.





Copyright © Autodesk

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

a In the **Toolpath** name field, enter **D12TR1-RGH1**.

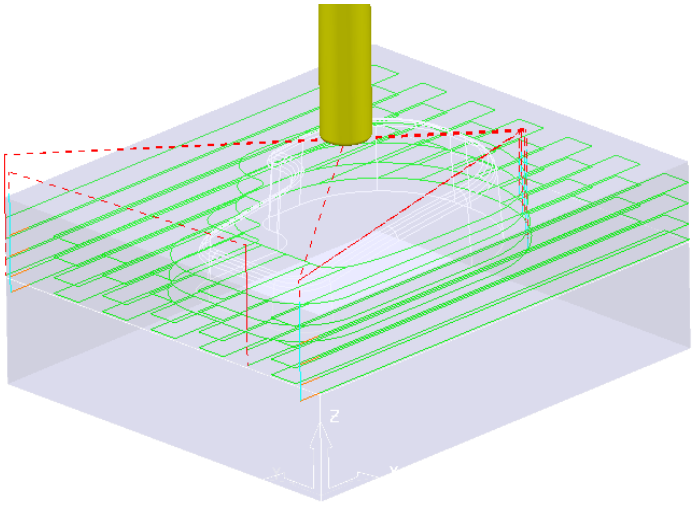
b Select ***Style*** – **Raster** with ***Profile*** **- Climb** and ***Area*** **- Any**.

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 and then **Close** the dialogue.

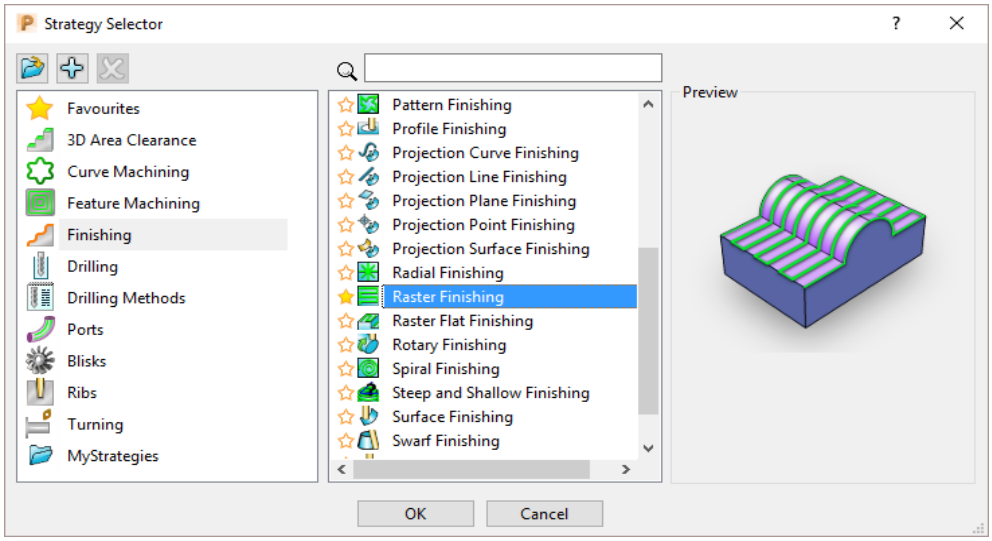


# Creating a Finishing Strategy

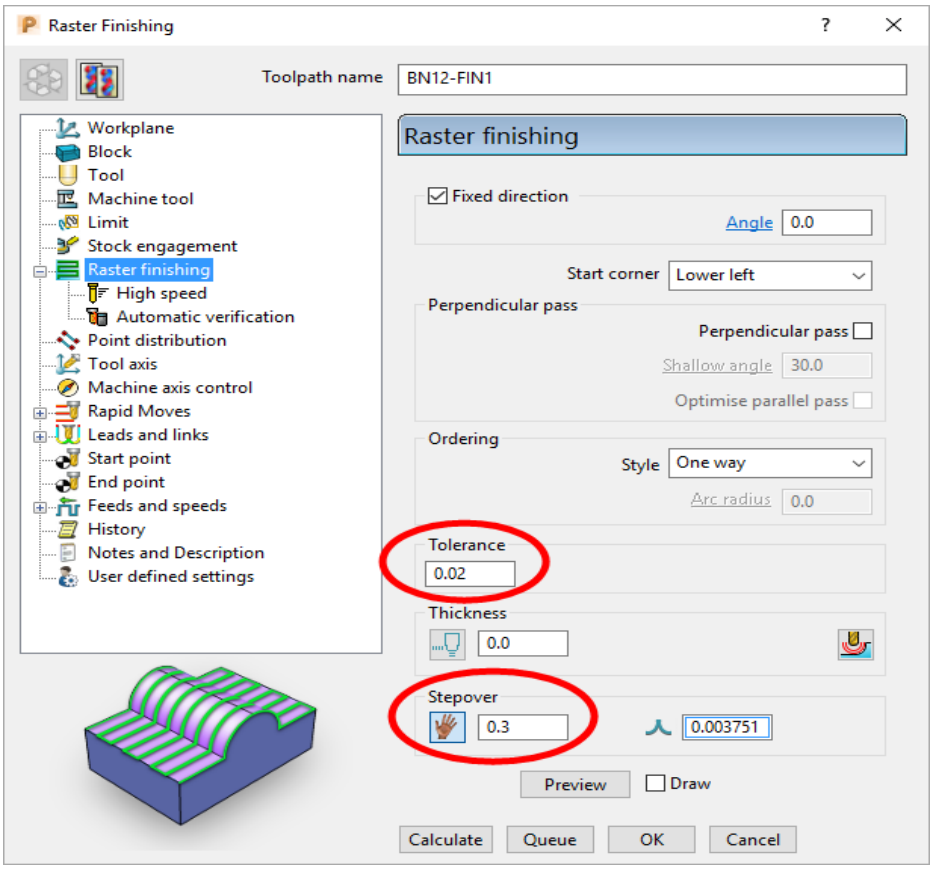
5 In the **PowerMill *explorer***, right click the **BN12** tool and select **Activate** (or double left click on the tool's, **PowerMill *explorer***, icon).

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

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



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

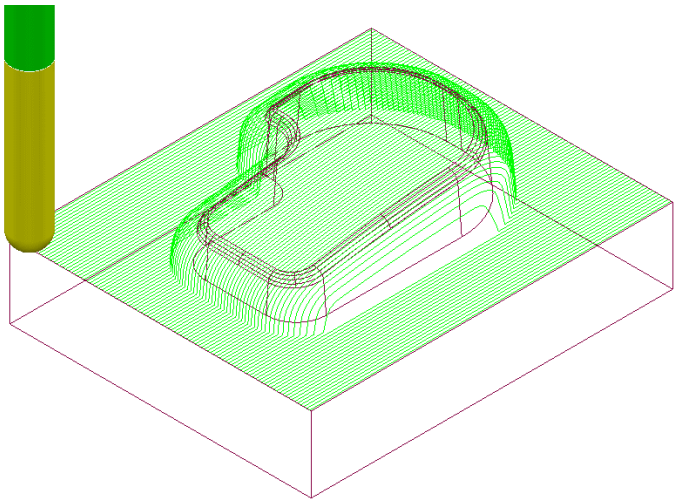



9 In the **Raster Finishing** strategy dialog:

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.

10 **Close** the dialogue.

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


Copyright © Autodesk

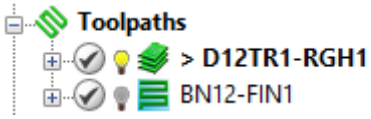
# Toolpath Simulation and ViewMILL

**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.

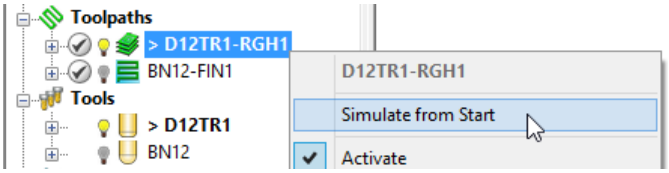
## Toolpath Simulation

11 In the **PowerMill *explorer***, right-click on the toolpath **D12TR1-RGH1** and from the local menu select **Activate**.




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

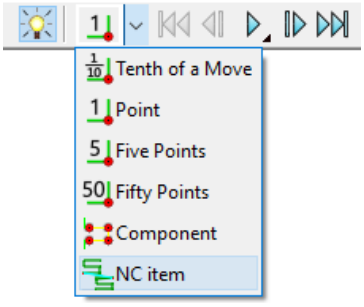
12 From the **PowerMill *explorer*** right mouse click on the **Toolpath D12TR1-RGH1** and from the local menu select, **Simulate from Start**.




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.

The 5 main buttons are **Rewind**, **Step backward**, **Play**/(**Pause**), **Step forward**, and **Go to end**. While playing the **Start** button changes to **Pause** (or simply left mouse click in the graphics area).

The options below allow further control of the **Step** simulation.



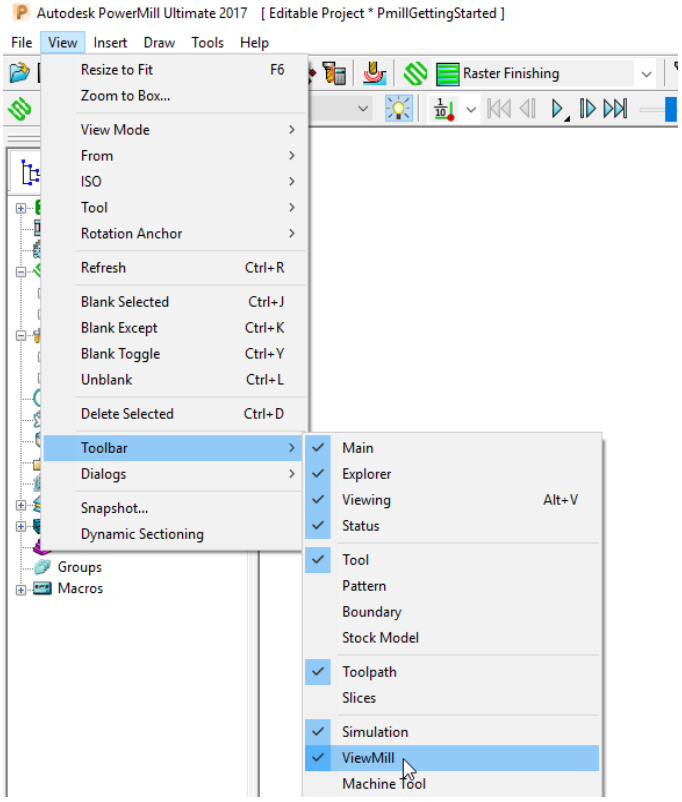

13 From the **Simulation** toolbar click the **Play** button to run the **Simulation**.

14 **Activate** the finishing toolpath **BN12-FIN1** and repeat the simulation process.

# ViewMILL

1 **Activate** the roughing toolpath **D12TR1-RGH1** and from the local menu select **Simulate form Start**.

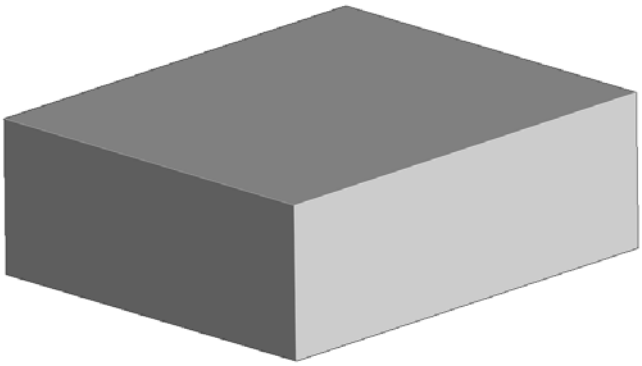
2 From the **Main** pull down menus, select **View > Toolbar > ViewMILL** to raise the **ViewMILL** Toolbar.




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

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

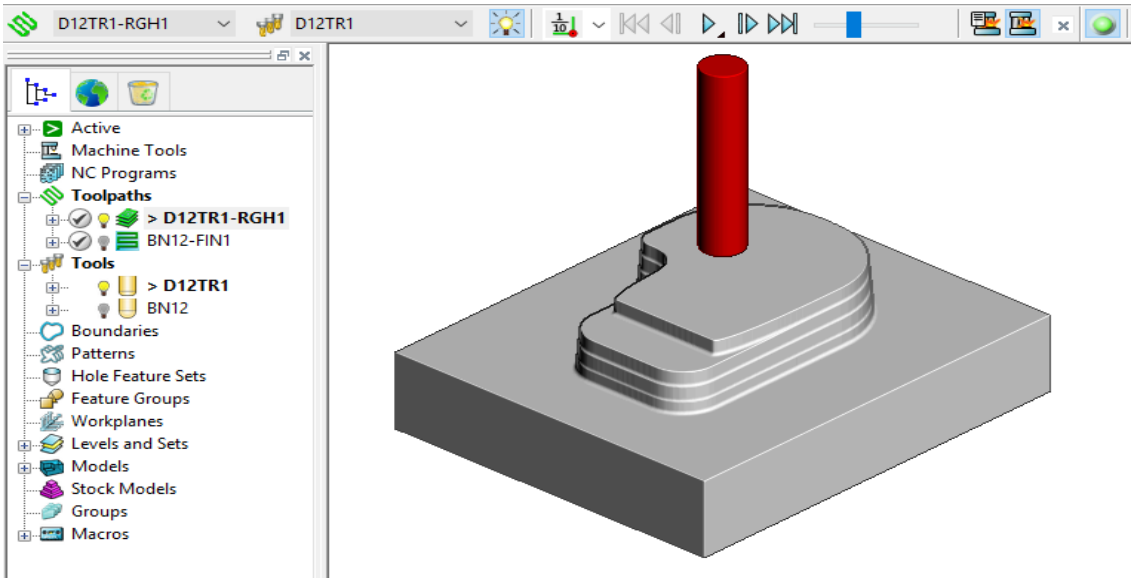
4 Click the button to select a **Plain Shaded Image**.




Copyright © Autodesk

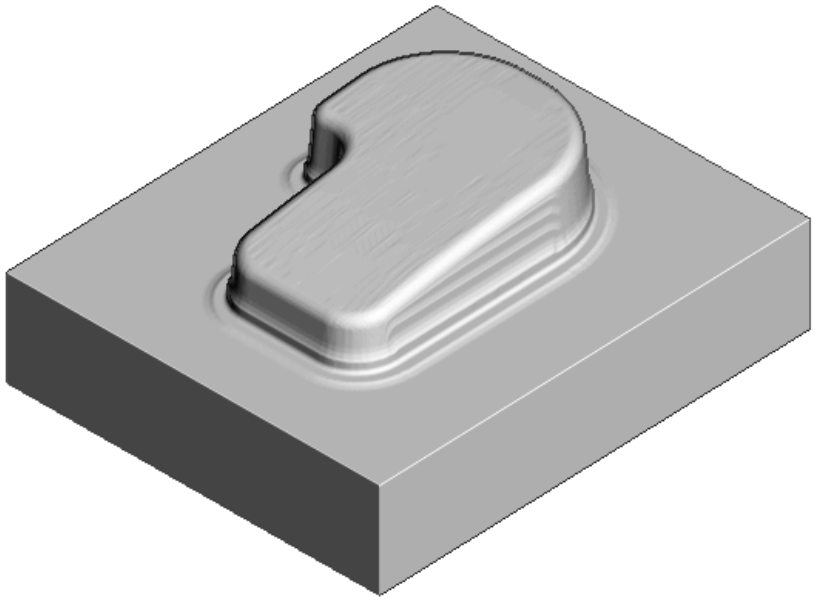
5 Click the **Tool** button to display the tool.

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




7 After the ***simulation*** has finished, select the toolpath **BN12-FIN1** on the **Simulation** toolbar, followed by the **tool** button .

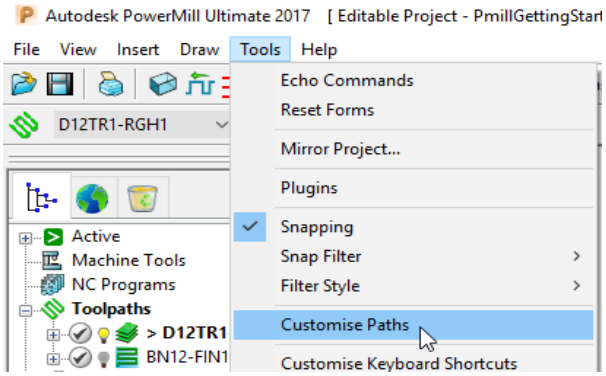
8 Click the **Play** button again to view the continued simulation of material removal by the finishing toolpath.



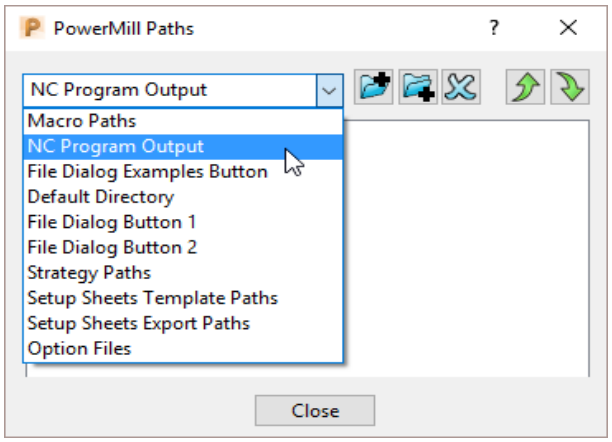
9 In the **Simulation** toolbar, select the **Close** button to delete the **ViewMILL** session.

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

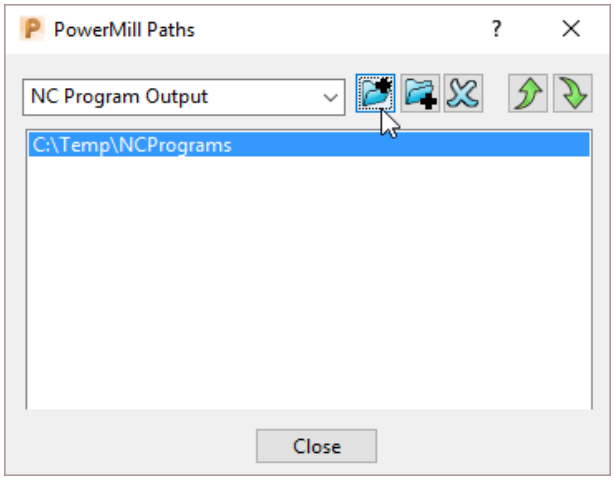
1 From the **Main** menu, select **Tools > Customise Paths** to open the **PowerMill Paths** dialog.




This displays the **PowerMill Paths** dialog.




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



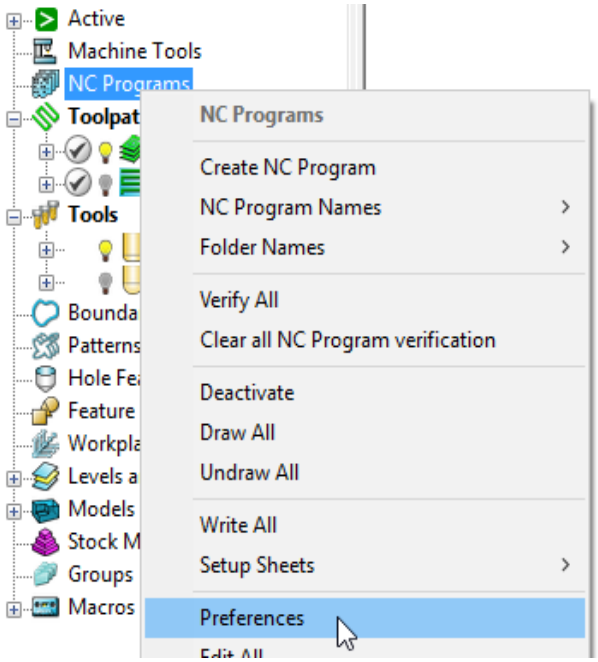

This controls where the post-processed, NC data files are output ready to download to a machine tool controller.

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

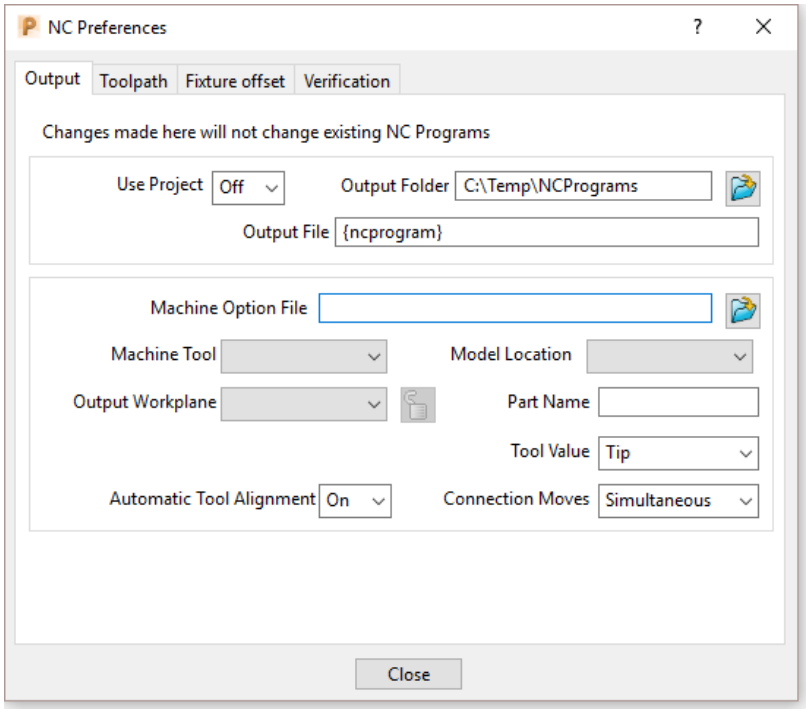
4 In the **Select Path** dialog which appears, browse to the required location **C:\Temp\NCPrograms** and click **OK**.

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


Copyright © Autodesk

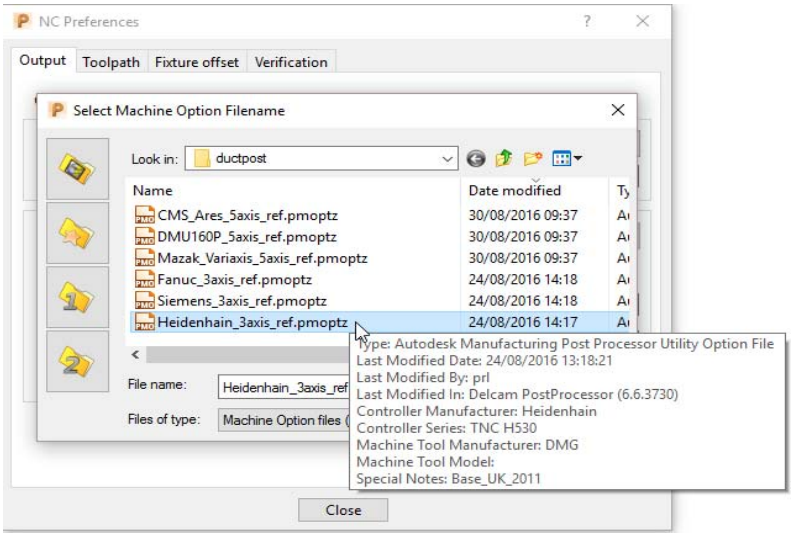



**NC Preferences** enables you to control the content of output files to download to a machine tool.




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

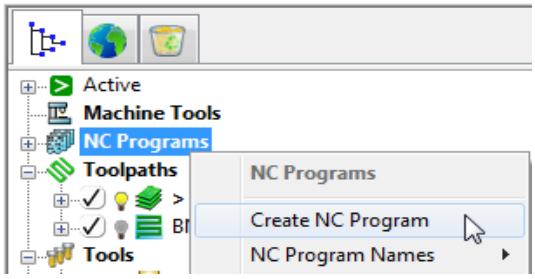
6 In the **NC Preferences** dialog, click **Machine Option File** button. This displays the **Select Machine Option Filename** dialog.




7 Select **Heidenhain_3axis_ref.opt** and click **Open**. This selects the option file and returns you back to the **NC Preferences** dialog.

8 **Close** the **NC Preferences** dialog.

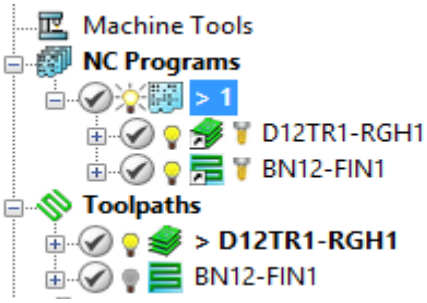
9 In the **PowerMill *explorer***, right-click **NC Programs** and select **Create NC Program**.




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**.*

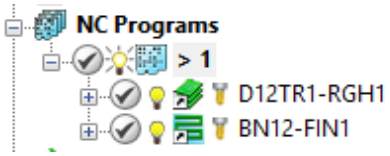



A copy of the ***toolpath name*** will appear in the **NC Program** indicating that it has been assigned as part of the output file.

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

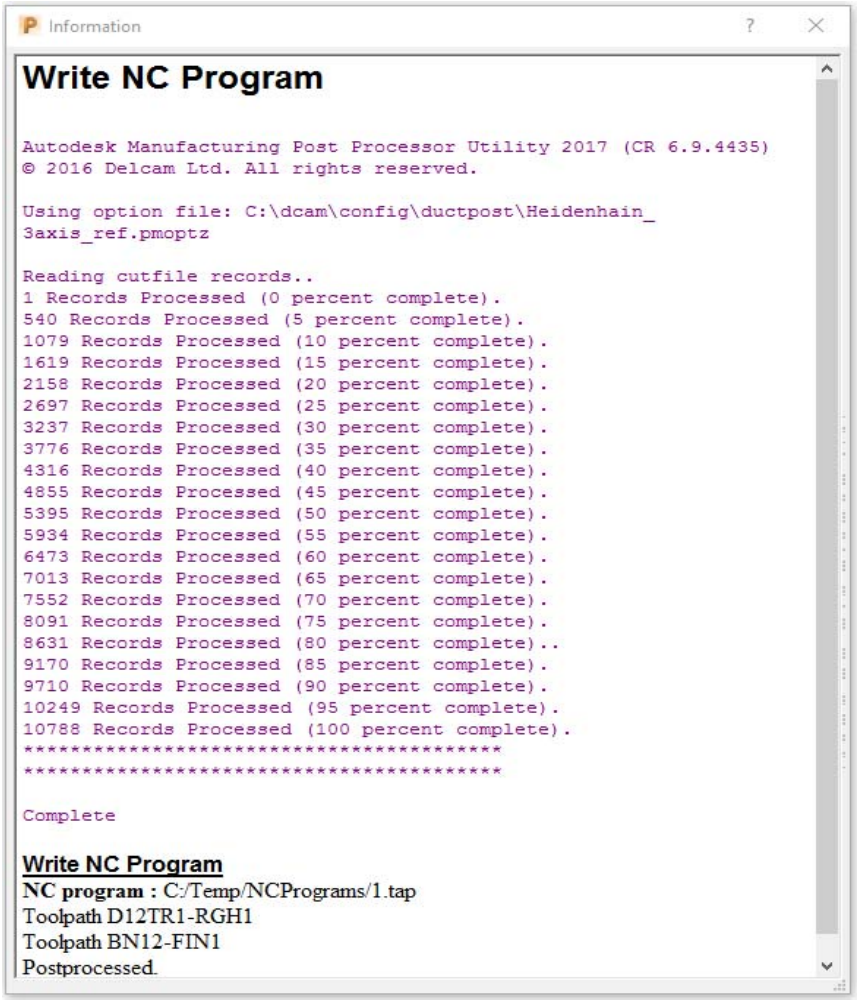
Click the small **+** box to expand and view the contents of the **NC Program**.


Copyright © Autodesk




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

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



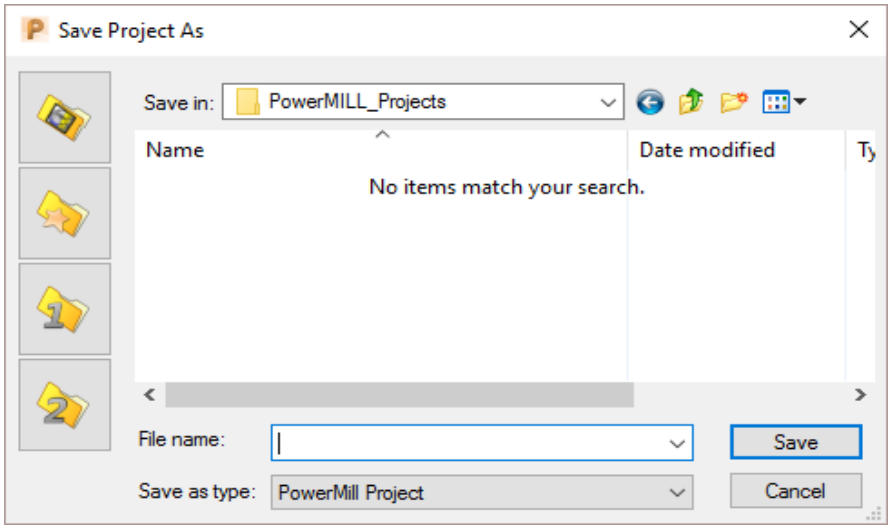

13 **Close** both the **NC Program** and **Information** dialogs.

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

# Saving the Project

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.*




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**.

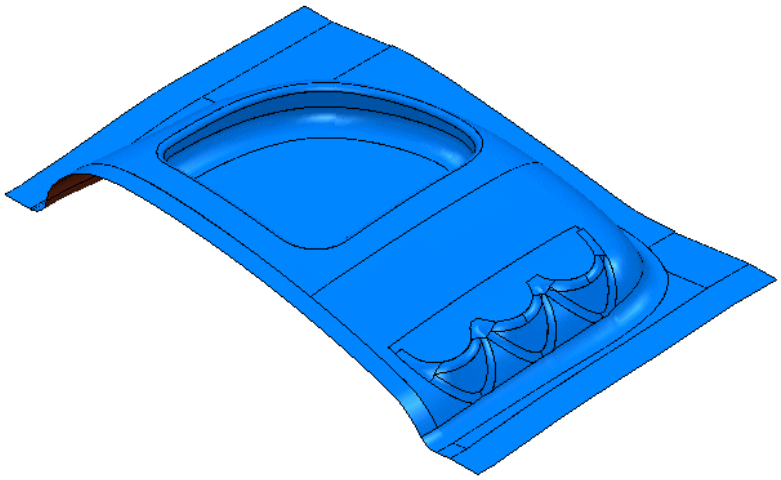
The content of **PowerMill *explorer*** is deleted and all dialogs are reset to their default settings.

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**.


Copyright © Autodesk

# 2. Machining Setup

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

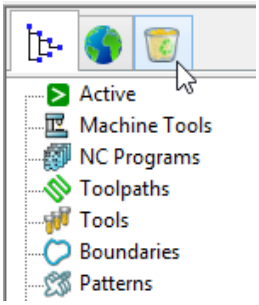
We will now look at the machining Setup options in more detail.

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.

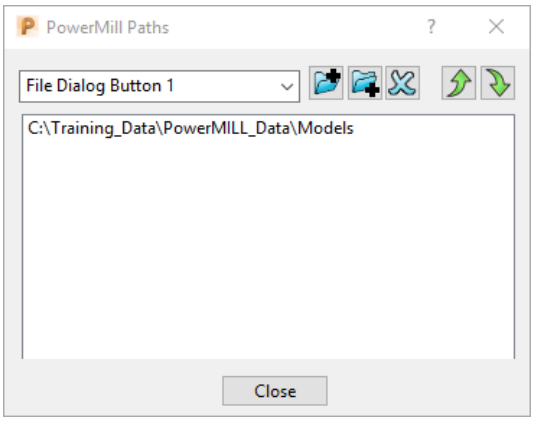



- The first pane displays the **PowerMill explorer** (as shown above).
- The second pane displays the **HTML** browser used for viewing **Help** files.
- The third pane opens the **PowerMill *Recycle Bin***. Entities deleted from the ***PowerMill explorer*** can be recovered from the **Recycle Bin**.

Copyright © Autodesk

# Setting up direct access to regularly used folders

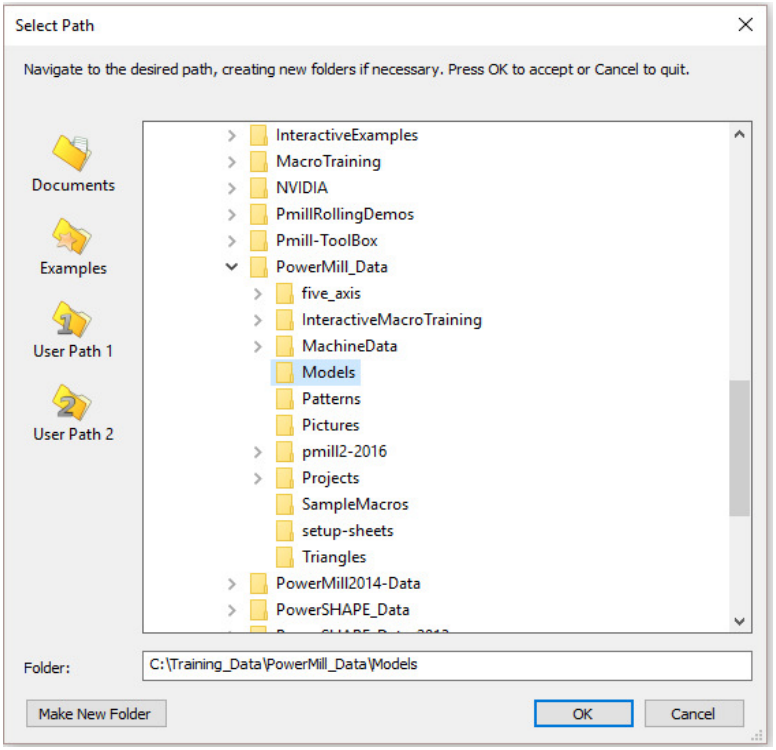
1 From the **Main** menu select **Tools** > **Customise Paths**. The **PowerMill Paths** dialog is displayed.




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

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

**.....\PowerMill_Data\Models**




4 Click **OK**.

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

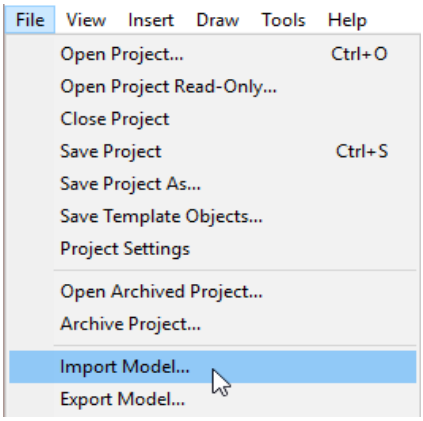
**....\PowerMill_Data**

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

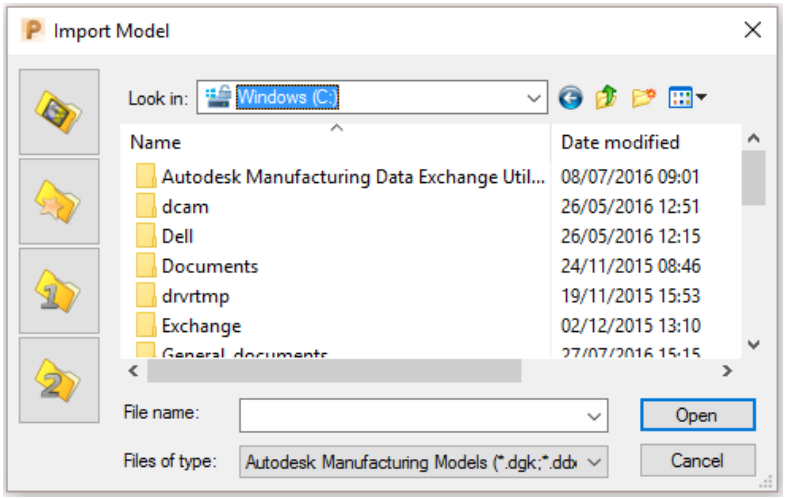

Copyright © Autodesk

# Loading a model into PowerMill

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




The **Import Model** dialog is displayed.




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

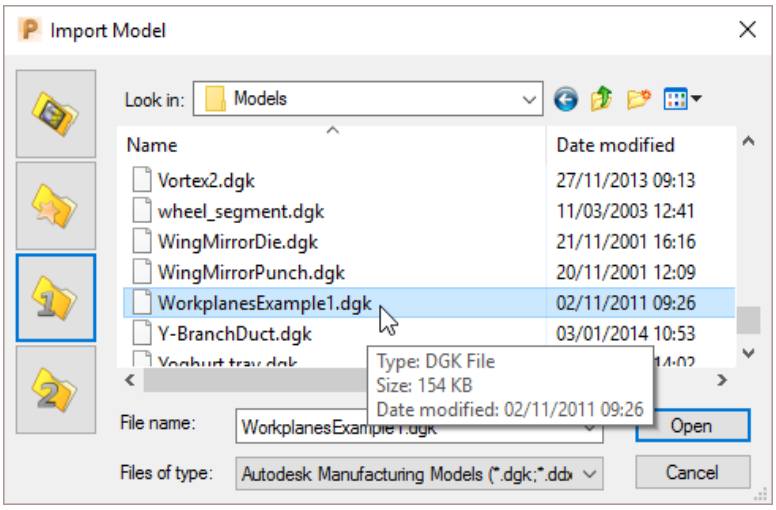
This folder contains a selection of sample model files which are installed within the **PowerMill** installation in a directory called **Examples**.

- - changes the directory to the first ***user-defined*** directory.
- - changes the directory to the second ***user-defined*** directory.

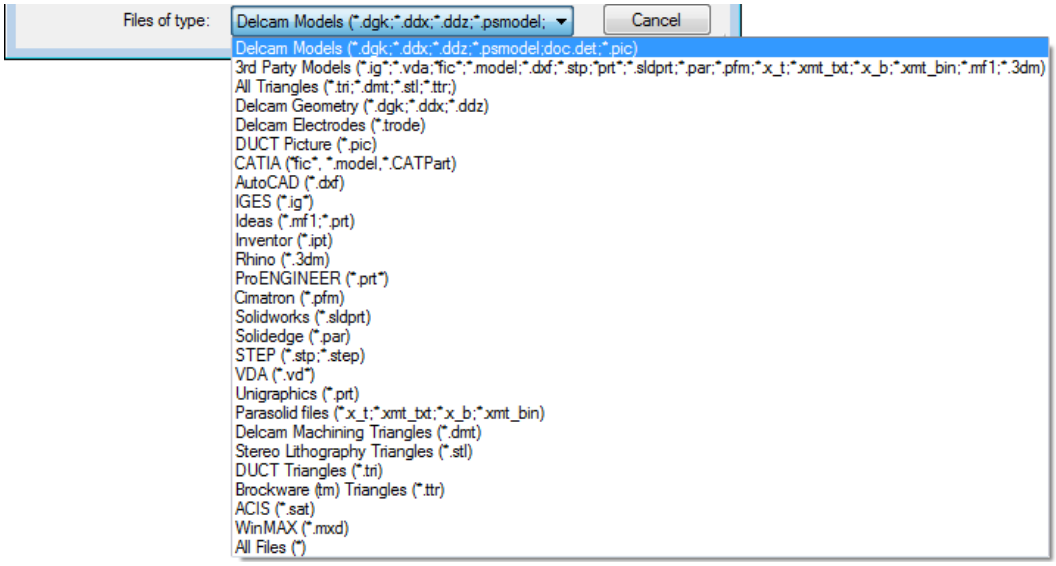
1 From the **Main** menu, select **File** > **Import Model**.

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

**....\PowerMill_Data\Models**

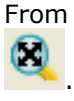



*A variety of **Model** formats can be imported into **PowerMill**. Use the **Files of type** filter to widen or narrow your choice.*




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

3 Back in the main **Import Model** dialogue, 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.


Copyright © Autodesk

# Viewing the Model

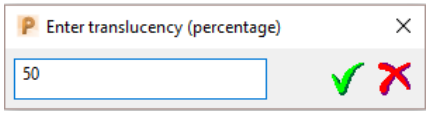
Although the **model** is displayed, it is a good idea to look at it from all angles to fully understand its size and features.

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

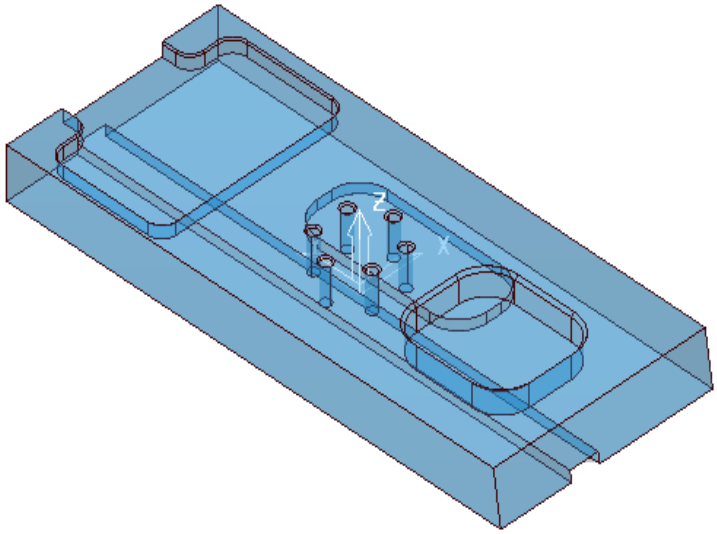
By default, the **Viewing** toolbar is aligned to the right hand side of the graphics area. These options mainly include basic visual orientations along with several shading modes, many of which provide specialist analysis of the model.

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

6 Right mouse click on the **Model** and from the local menu select **Transluceny**.




7 Enter **50** into the data input box and select the ***green tick*** to accept.



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, the component will have to be aligned with the longest side along X to ensure that it can be positioned within the travel limits.

# Saving the Project

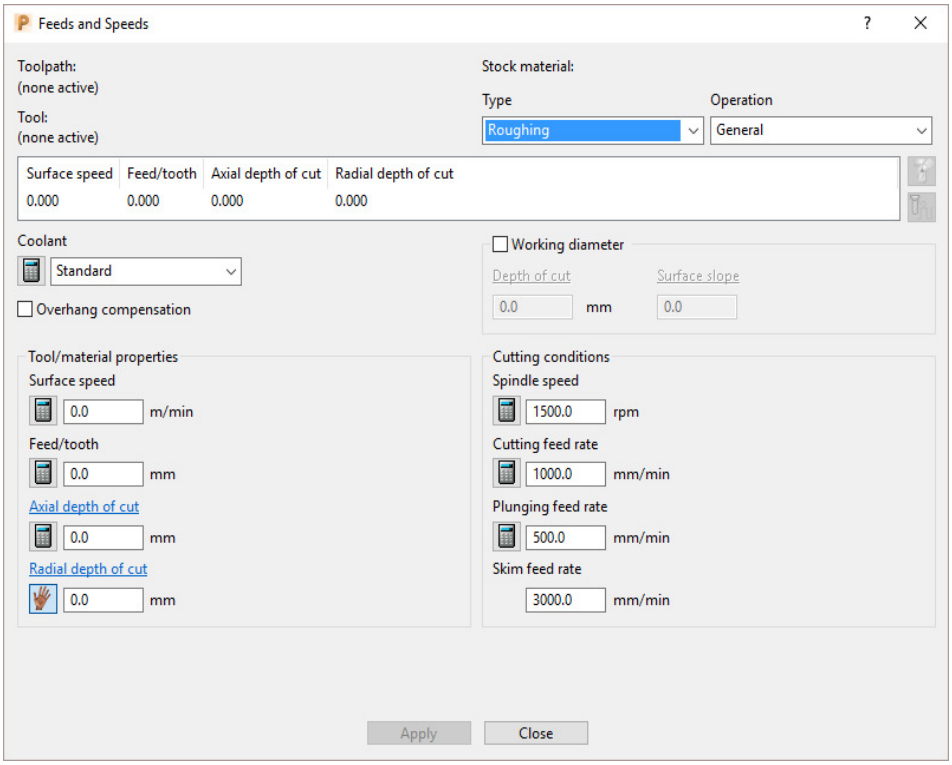
It is good practice to create a **Project** as soon as possible and then to **Save** it at regular intervals during the session.

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




The default values (as shown above) can be modified as required and **Accept** selected for them to be the default for future toolpaths. **Apply** is used to update the **Feeds and Speeds** of an ***active*** **toolpath**. Note **Apply** is 'greyed out' 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.

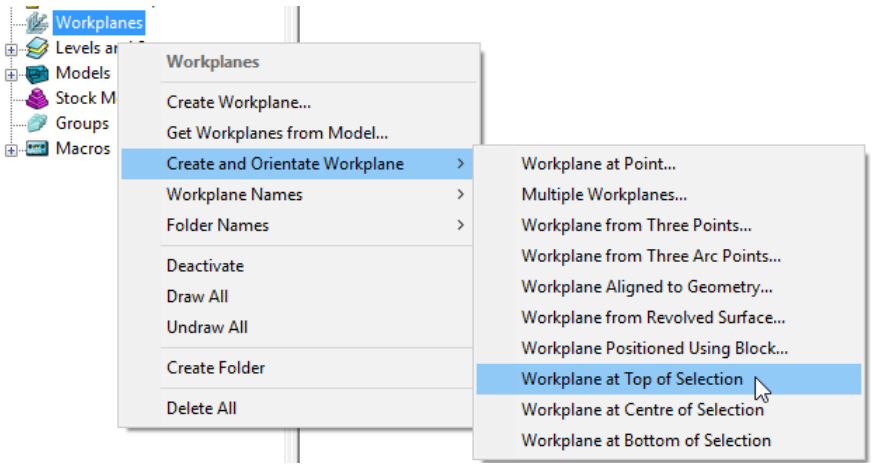
By deactivating a **workplane**, the original coordinate set-up can easily be re-activated for tasks such as checking original dimensions or for importing a 'new issue' updated model.


Copyright © Autodesk

# Workplane (1) - At Top of Selection

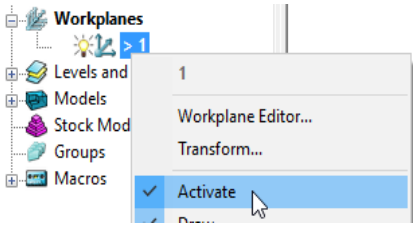
9 With the left mouse button depressed, drag a box over the entire **model** to select it.

10 In **PowerMill explorer**, right-click **Workplanes** and select **Create and Orientate Workplane** > **Workplane at top of Selection**.




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

11 Right-click the newly renamed **Workplane 1** and from the local menu select **Activate**.

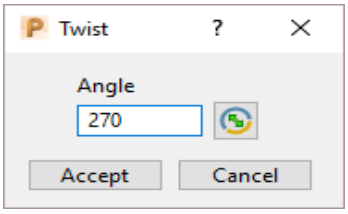



The next step will be to rotate the new **Active** - **Workplane** - **1** to re-orientate the model such that the longer side is aligned to the **X-Axis**.

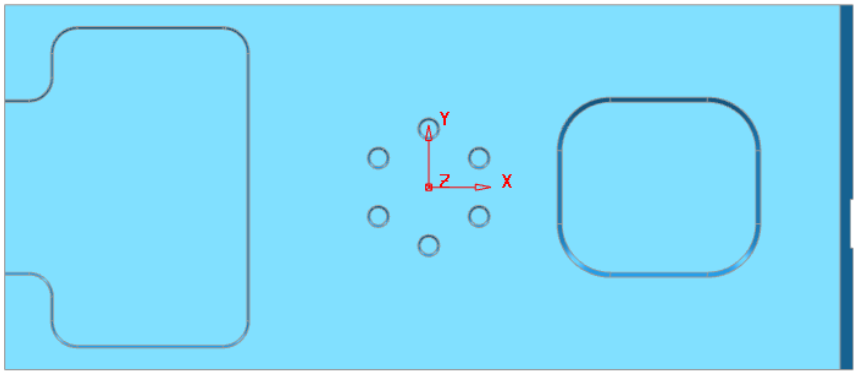
12 From the same local menu, select **Workplane Editor** to display the **Workplane** editing toolbar.

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

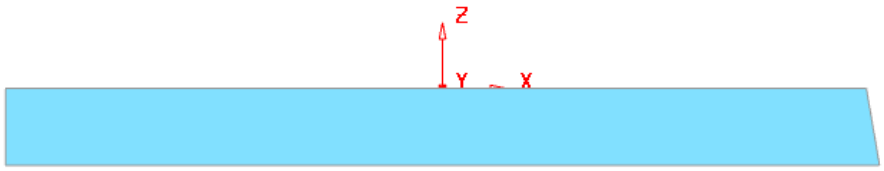
14 Enter **Angle 270** and click **Accept**.




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




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

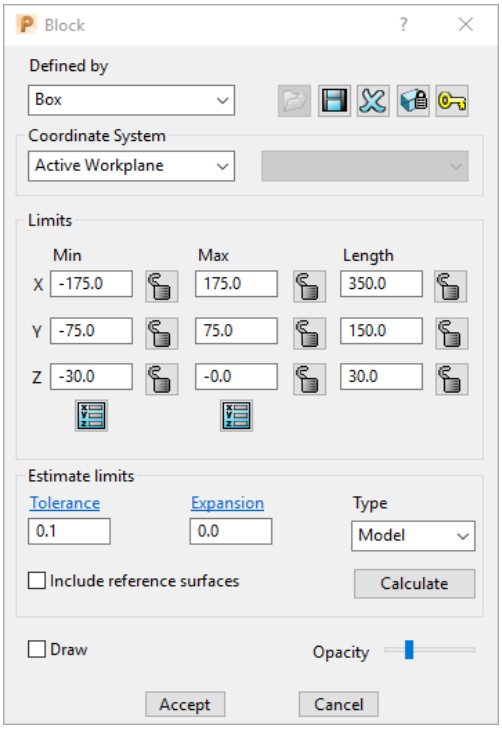



The **Workplane** is shown positioned on the top of the model, which has the longer edge aligned to X.

# Workplane (2) - Positioned using the Block

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

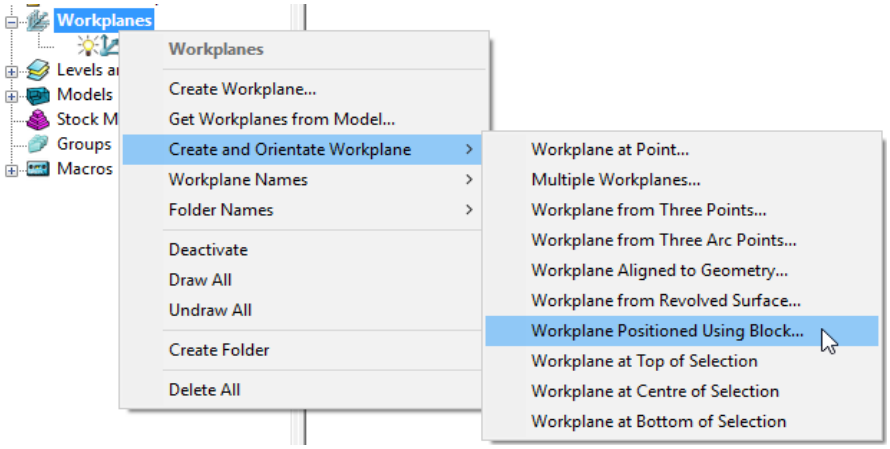
17 From the **main** menu, select the **Block** Icon the following form will appear select **Active Workplane** press **Calculate** followed by **Accept**.





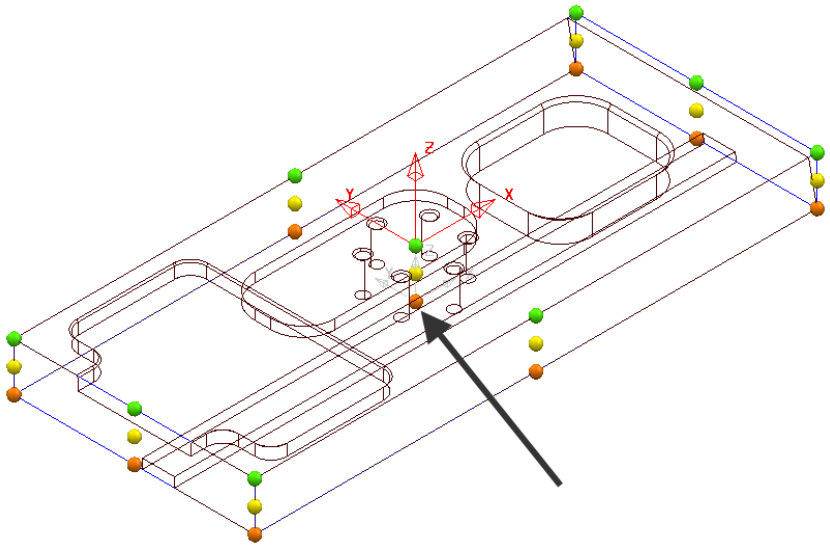
Copyright © Autodesk

18 In the **PowerMill explorer**, Right mouse click on **Workplanes** to access the following menus.



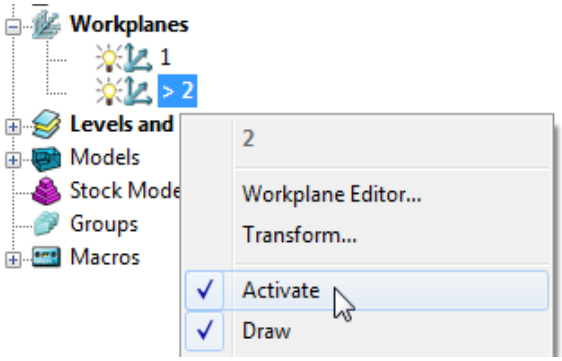

19 Select **Workplane Positioned Using Block** from the above menu.

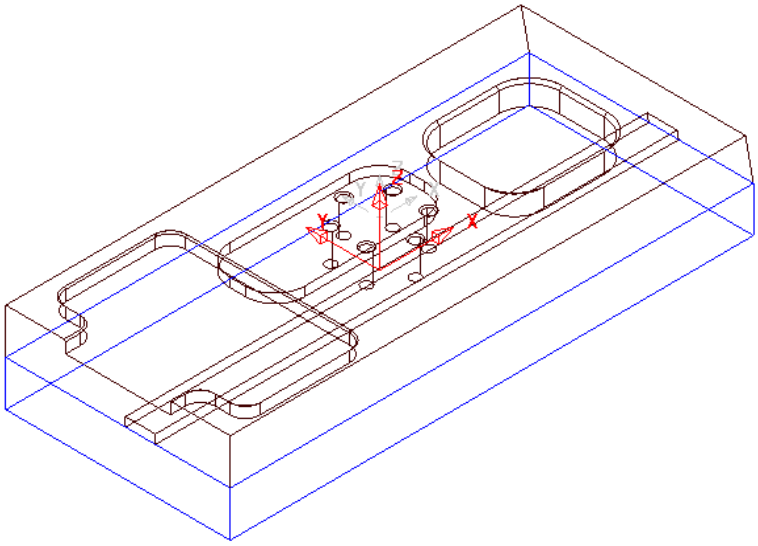
20 Click the new **Workplane** on to the orange *sphere* located at the *centre* of the *bottom face* of the **Block**.




*The new **workplane** will inherit the default name **2**.*

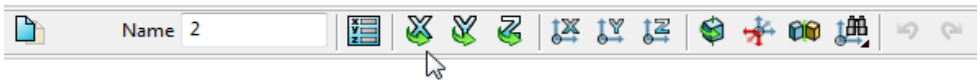
21 Right click on **workplane 2** and from the local menu, select **Activate**.





*Note that **Block** definition moves relative to the **active datum**.*

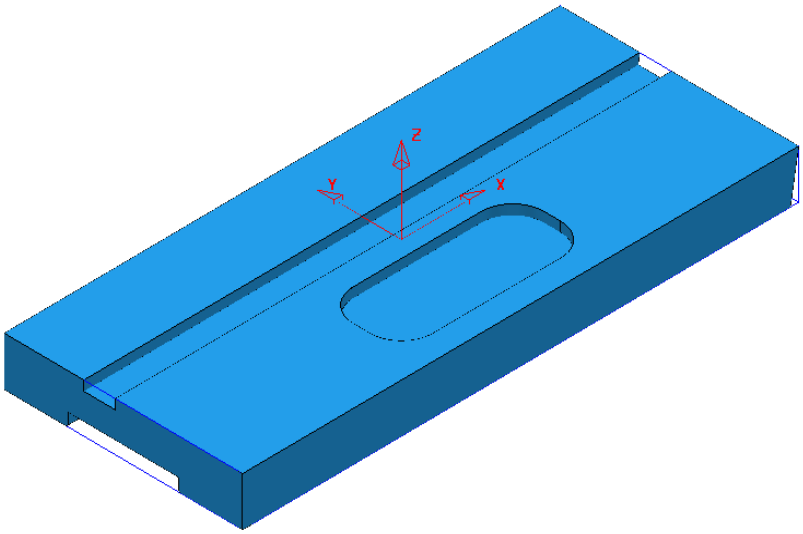
22 **Activate** the new **Workplane** and from the local menu select the **Workplane Editor**.




23 Select **X Twist**, and in the dialog enter **180** degrees to make the new **workplane** normal to the underside of the model.

24 Select **Accept** for the new **workplane 2** to be updated.

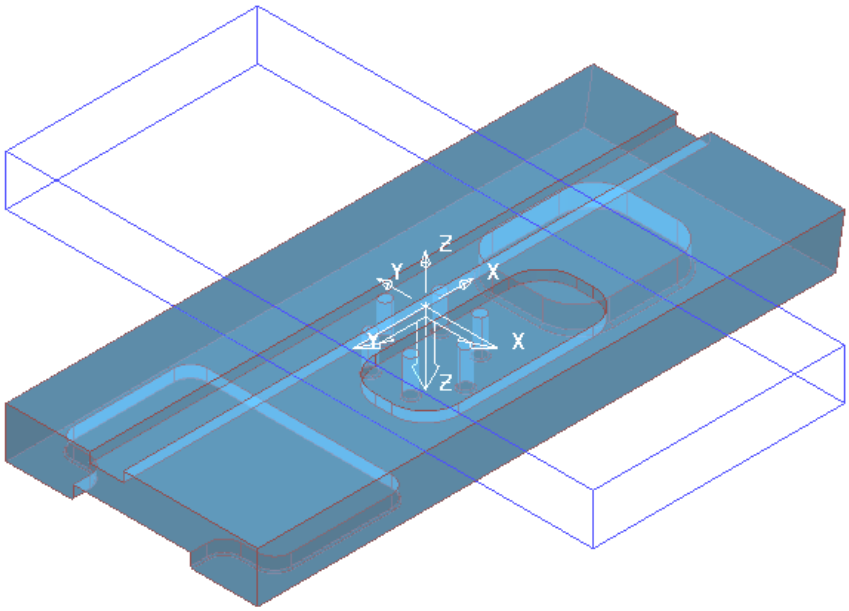
25 Select an **ISO 1** view.




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**.


Copyright © Autodesk

26 De-activate **Workplane 2**.



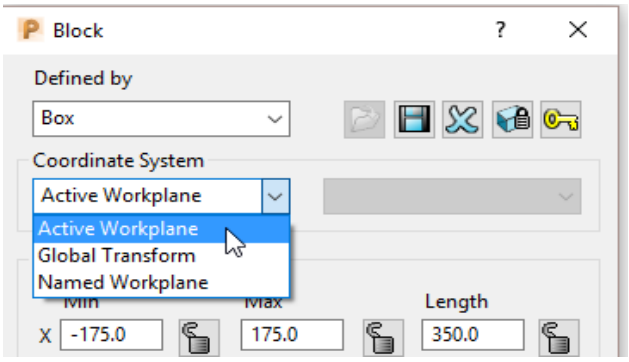

The **Block** is now dimensionally positioned relative to the global **Transform**.

27 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.

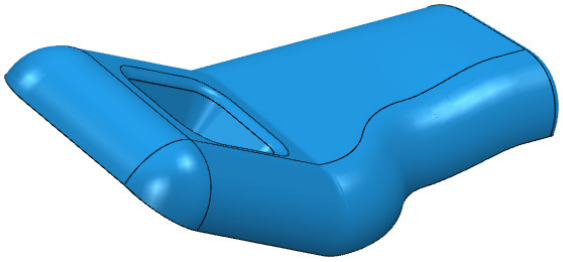



Other **Coordinate System** options become essential for advanced applications such as multi-axis, ***3Plus2*** set ups. In this case the **Block** definition can be fixed to either the **Global Transform** or a **Named Workplane**.

# 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).

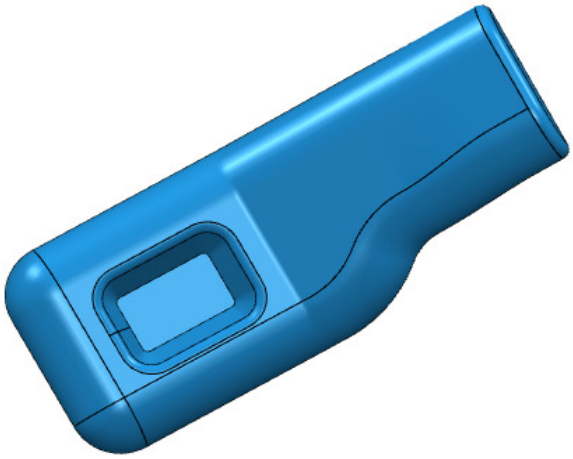
1 Select **File - Delete all** followed by **Tools - Reset forms**.



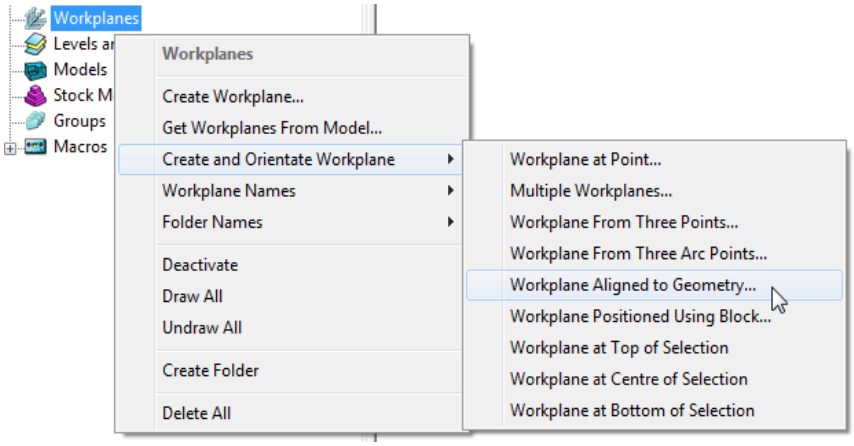
2 **Import** the **Model**:-

**.....\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.



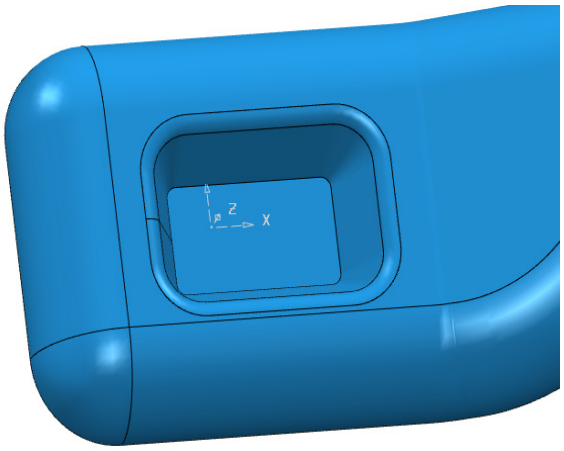
4 Right click on **Workplanes** and select **Create and Orientate Workplane** followed by **Workplane aligned to Geometry**.



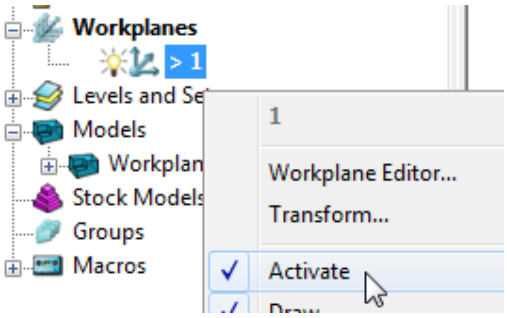


Copyright © Autodesk

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).

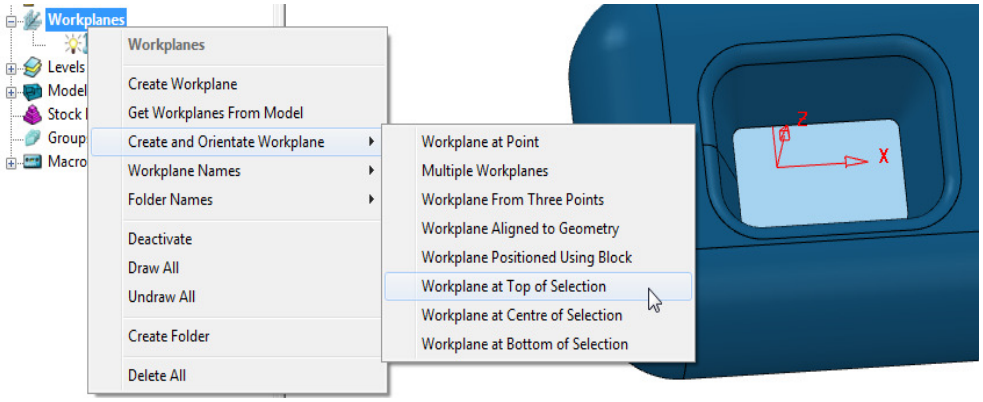



6 In the **PowerMill *explorer***, right click on the new **workplane** and select **Activate**.




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.

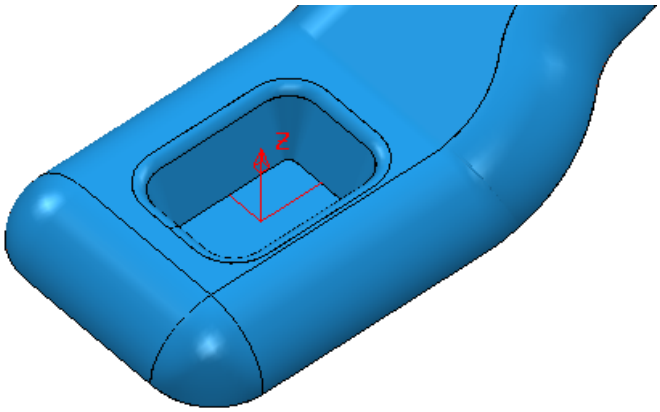



9 From **Create and Orientate Workplane**, select **Workplane at Top of Selection**.

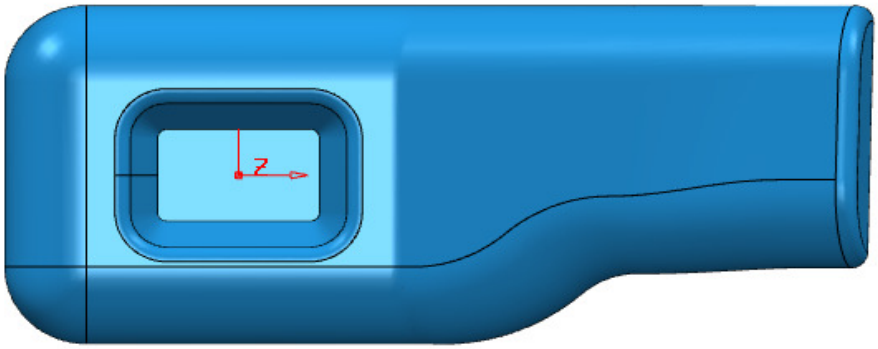
10 The new **Workplane** will be centrally located at the base of the pocket.

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.



2 **Save** the **Project** as:-

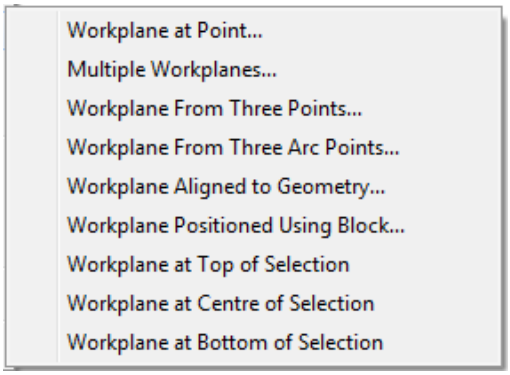
**.....\COURSEWORK\PowerMill_Projects\Workplane-Example2**

3 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.





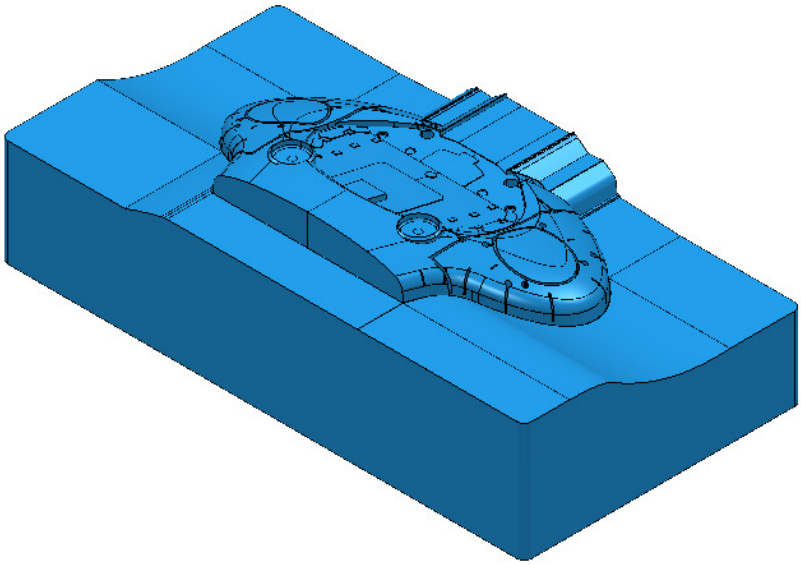
Copyright © Autodesk

# Speaker Core Example - Set Up

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

**.....\PowerMill_Data\Models\speaker_core.dgk**

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



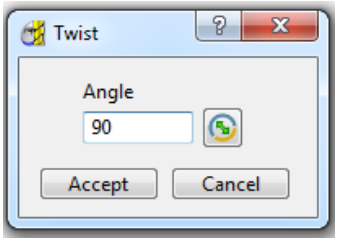
For this example it is required for the long side of the component to be aligned to the **X** direction and not **Y** as shown above.

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

3 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**.

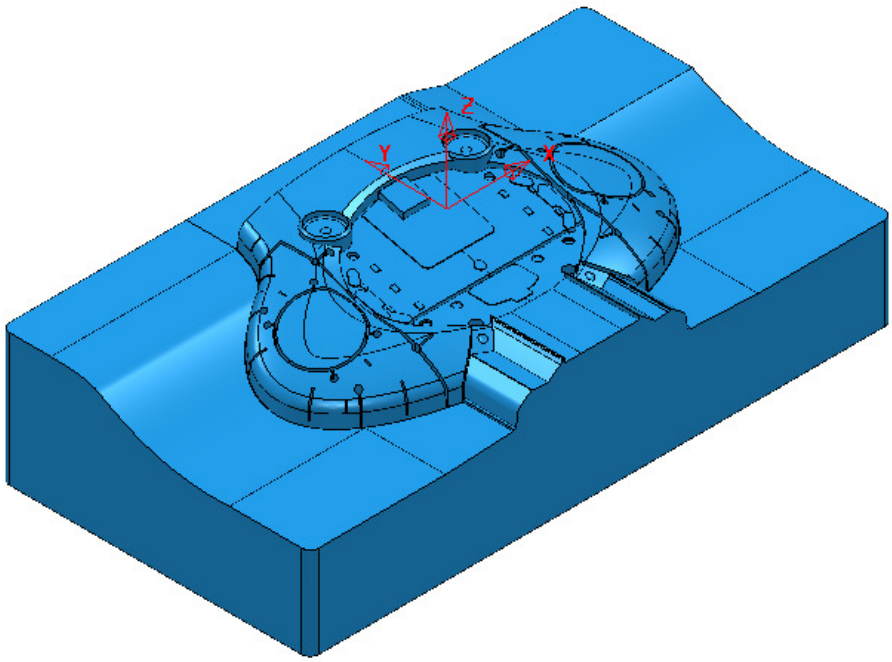
5 In the above toolbar, click on the **Z-Twist** option and enter **90** in the form for the **Workplane** rotation angle.




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.

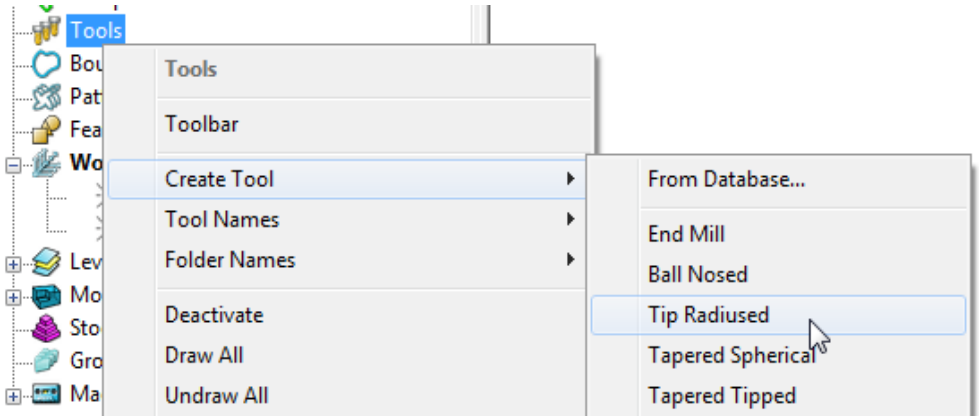
Now that the component model is suitably aligned, the next stage is to create some suitable cutting **Tools** ready to carry out the machining process.

# Tool and Holder Definition

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

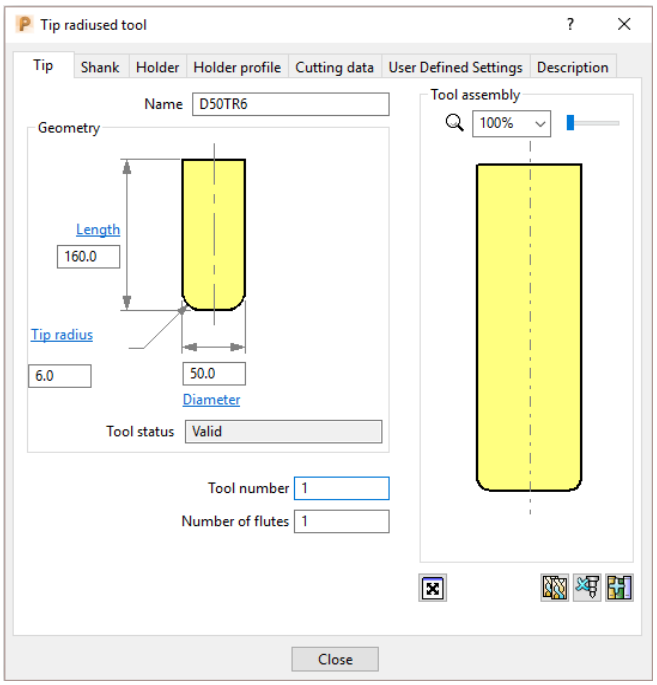
Note it is also possible to store specific **feeds** and **speed** values in a Tool definition. Not a default setting, but **PowerMill** can be set to recognise these and automatically update the **Feeds and Speed** dialog on activation of the **Tool**.

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

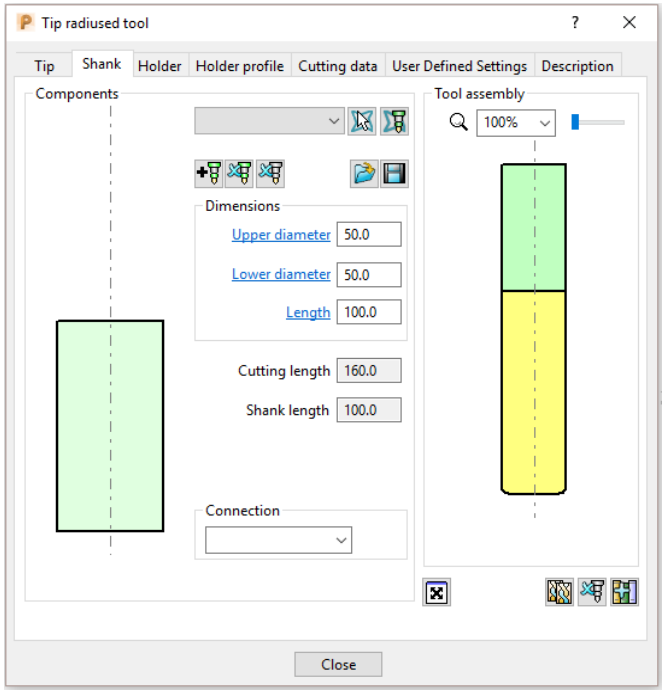



10 In the Tip Radiused Tool dialog, input a ***Name*** **D50TR6** - ***Diameter*** **50** - ***Tip Radius*** **6** - ***Length*** **160** - ***Tool Number*** **1**.


Copyright © Autodesk



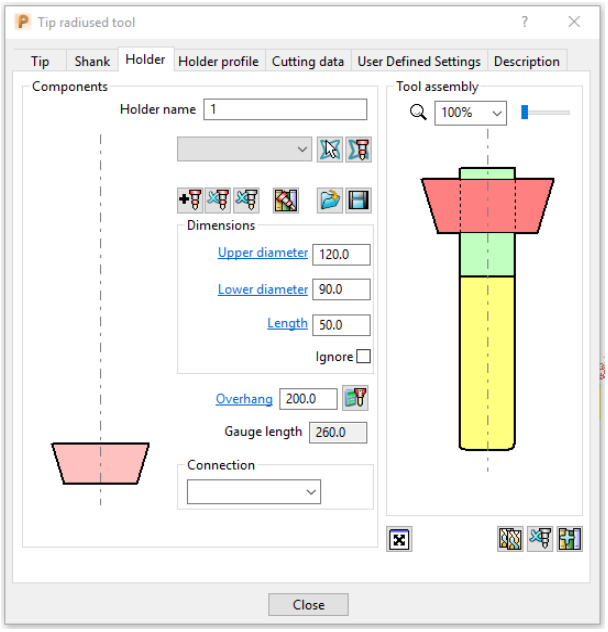

11 In the **Shank** dialog, click **Add a shank component** and enter: **Upper/Lower *Diameter* 50** and ***Length* 100**.




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

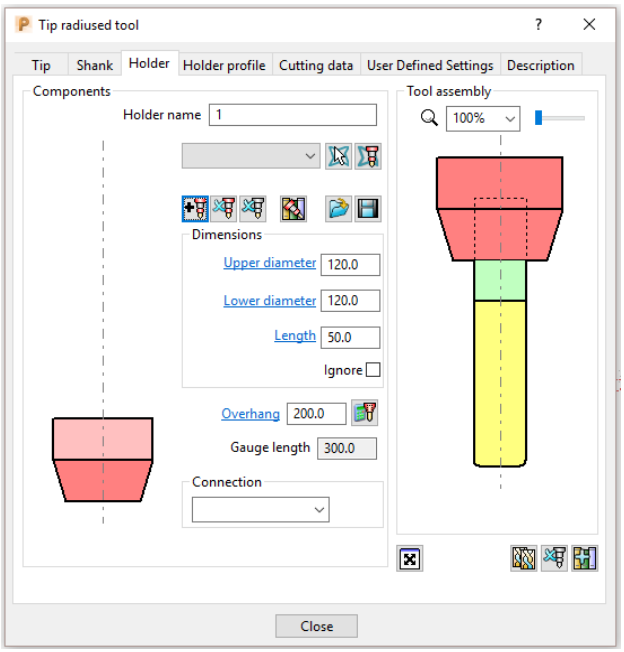
- **Upper Diameter – 120**
- **Lower Diameter – 90**
- **Length – 50**
- **Overhang - 200**

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




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

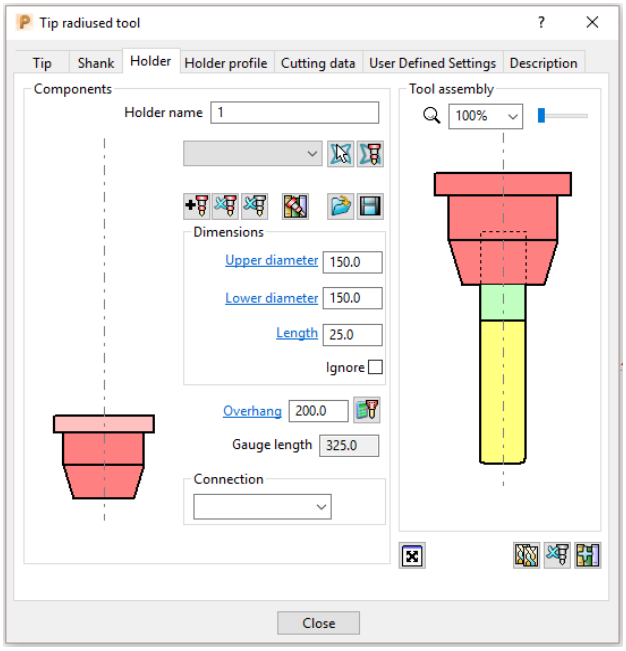
- **Upper Diameter – 120**
- **Lower Diameter – 120**
- **Length – 50**




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

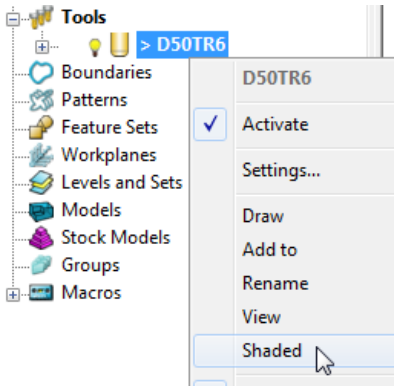
- **Upper Diameter – 150**
- **Lower Diameter – 150**
- **Length – 25**


Copyright © Autodesk

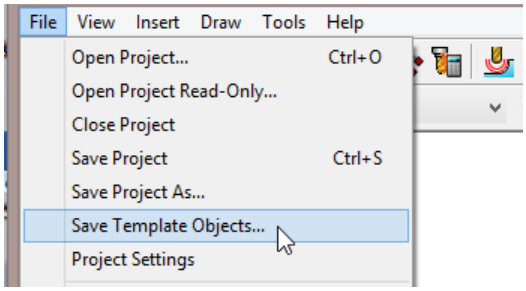



**15** **Close** the dialog.

*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.*

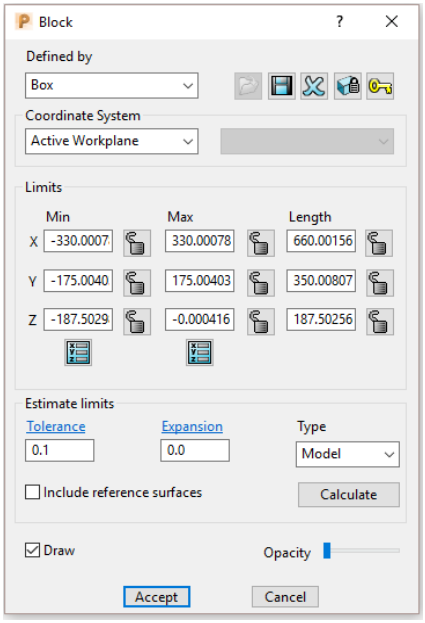
# Block Definition

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).

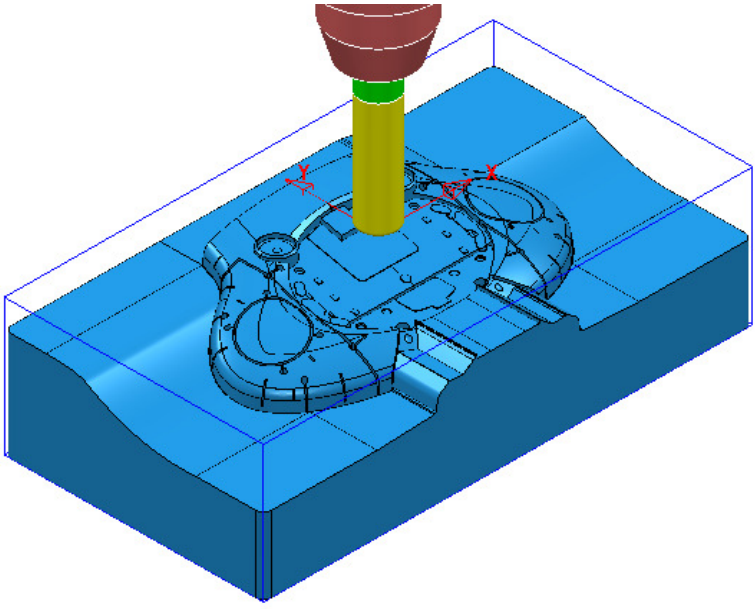
The dimensions of the **Block** can be entered manually or calculated directly to the Type of entity: **Model**, **Boundary**, **Pattern**, or **Feature**.

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**.




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

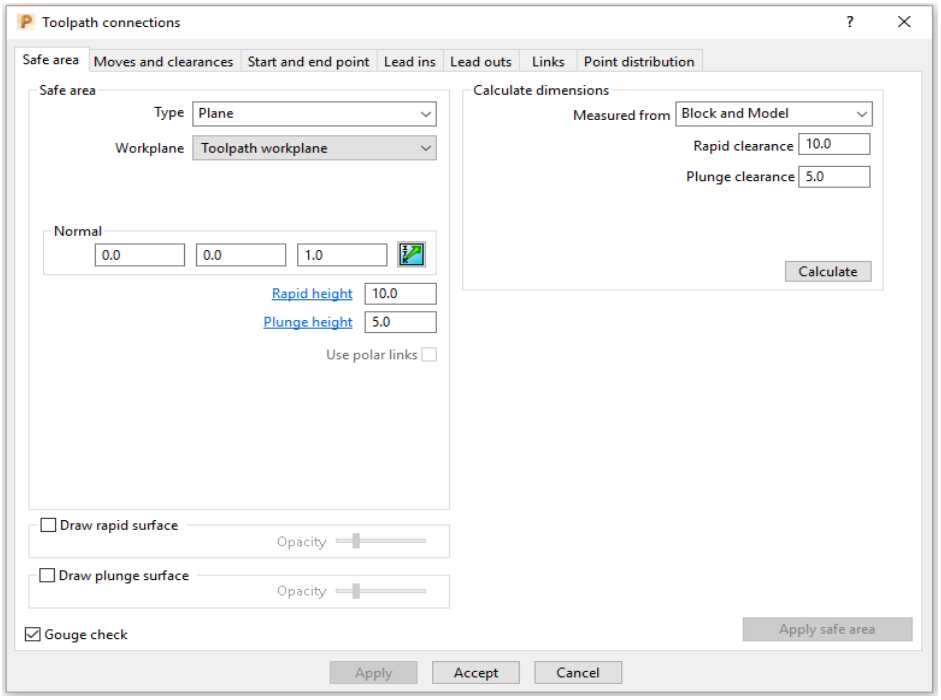



Copyright © Autodesk

# Rapid Move Heights

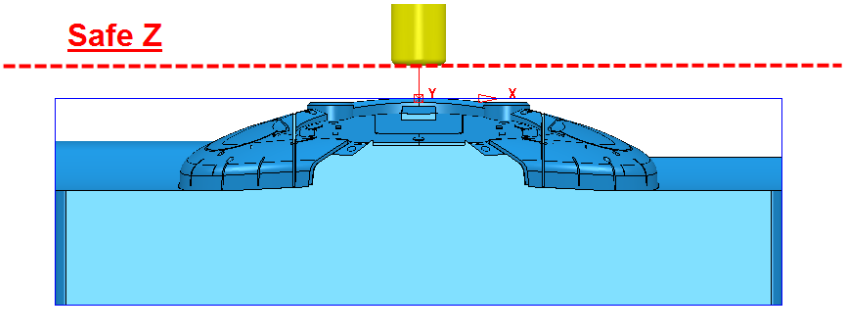
The **Rapid Move Heights** icon provides access to the **Toolpath connections – Safe area** dialog. This enables the control of rapid link moves between tool tracks that clear of the top of the **Model** or **Block** (whichever is the higher).

18 In the **Toolpath connections** dialog, click the **Calculate** tab to set **Rapid Height** and **Plunge height** to the default values of **10** and **5** respectively above the **Model** or **Block**.




*The **Rapid Height** is calculated relative to the highest point on the **Block** or **Model**.*

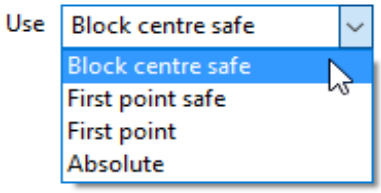
*If parts of the **model** are selected, then the de-selected parts will be ignored when the **Rapid Height** is calculated!*




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

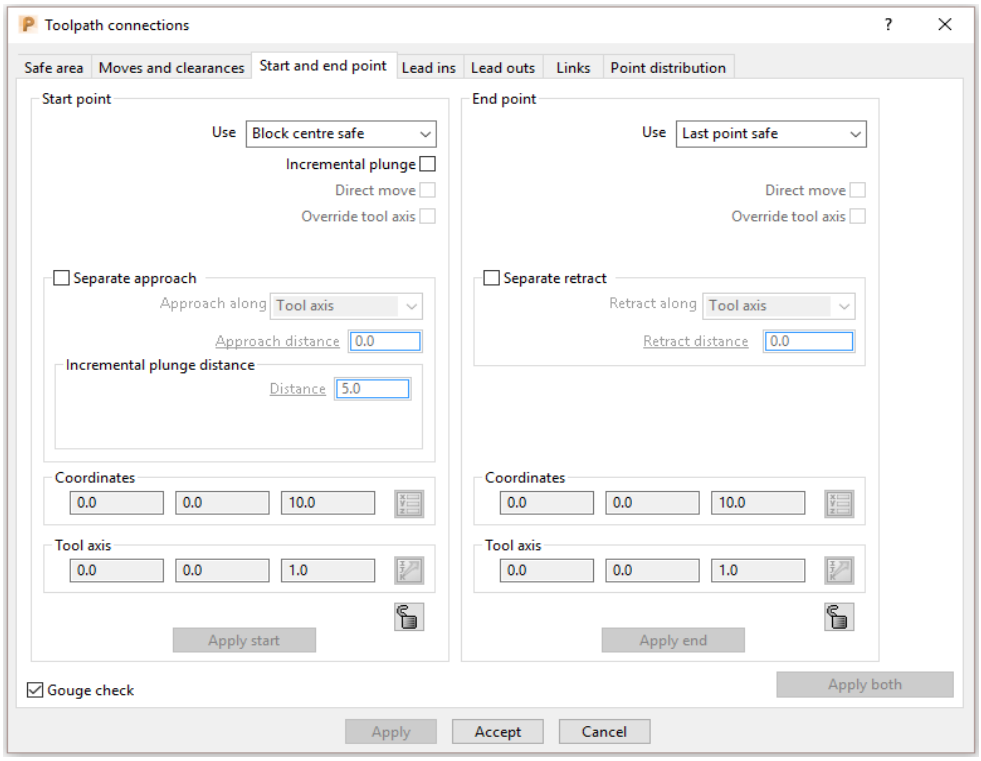
The **Start Point** and **End Point** icon provides access to the **Toolpath connections – Start and End** point dialog. This enables control of the ***first*** and ***last points*** on a toolpath from a choice of 4 different positional options.




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

a **Start Point - Block Centre Safe**

b **End Point - Last Point Safe**




# Save Project

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**:-

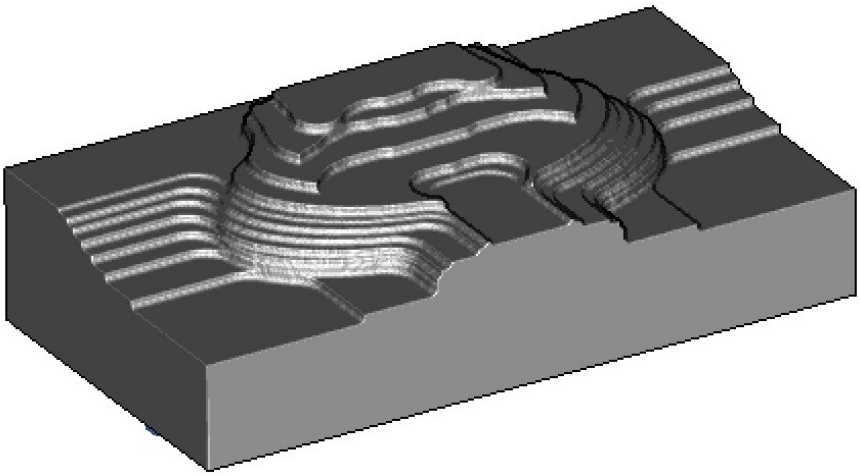
**....\COURSEWORK\PowerMill-Projects\Spkr-Core**

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


Copyright © Autodesk

# 3. 3D Area Clearance

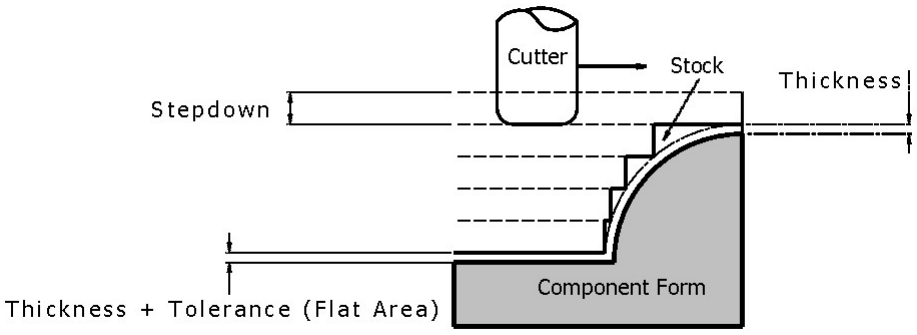
The main strategies for roughing a 3D component model are called 3D Area Clearance. These provide a choice of 2D material removal methods, which progressively machine the area (Slice), up to the component contour, down a sequence of user-defined Z Heights.



## 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.

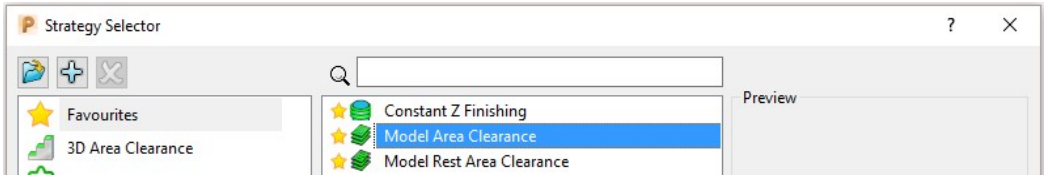
Thickness is the amount of extra material specified to remain on the work-piece after machining. This can be applied generally (as shown), or independently as separate Radial and Axial values within the machining options.



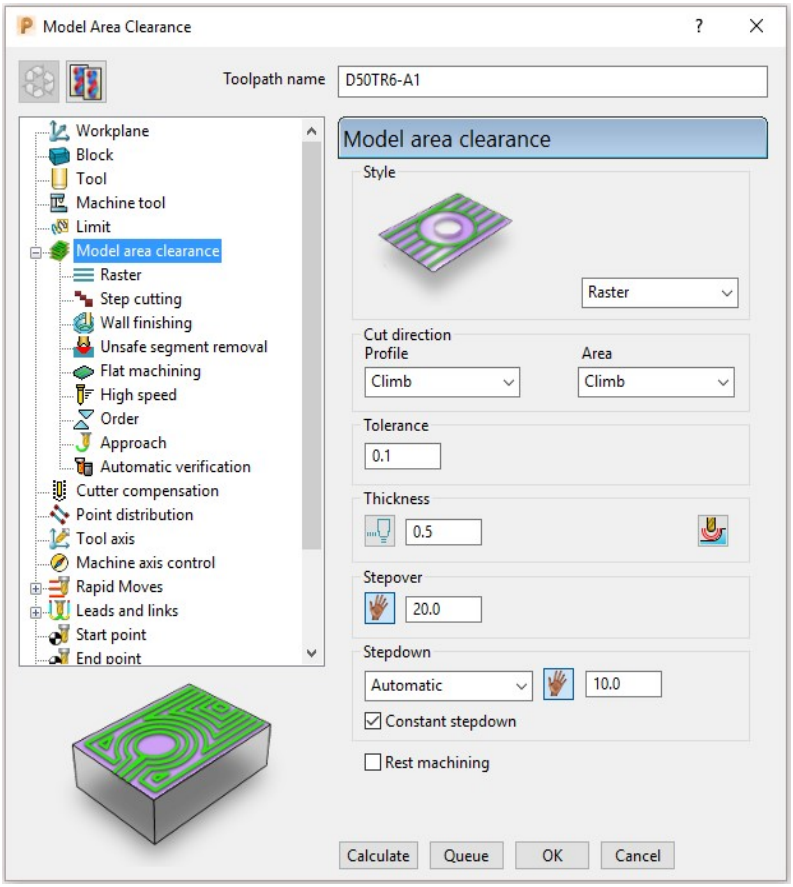
# Raster – Model Area Clearance example

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 dialog select **Calculate** followed by **Accept**.
5. From the **main** toolbar select **Toolpath strategies** and from **3D Area Clearance** open a **Model Area Clearance strategy**.




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



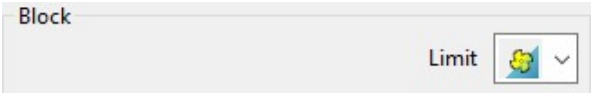

*As soon as the **Model Area Clearance** dialog is opened, the unprocessed **Toolpath** appears in the **PowerMill explorer**.*


Copyright © Autodesk

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**.



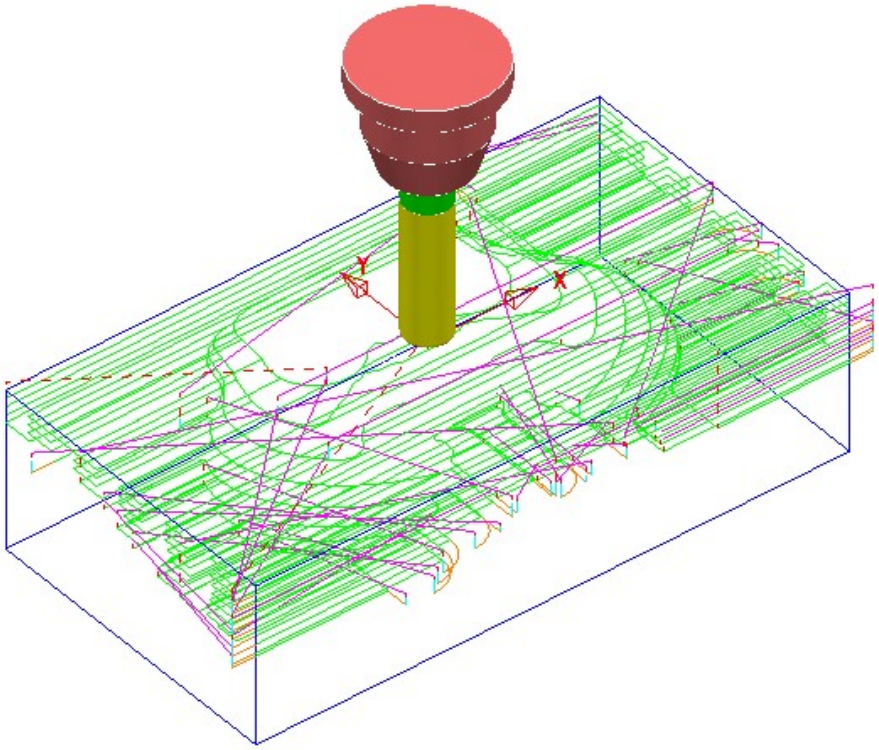

9 In the **Limit** options, **Block** category, switch to ***limit tool centre to block edge***.

10 Keeping all other values as default, **Calculate** the strategy.

11 After the **Toolpath** has been created, **Close** the dialog.

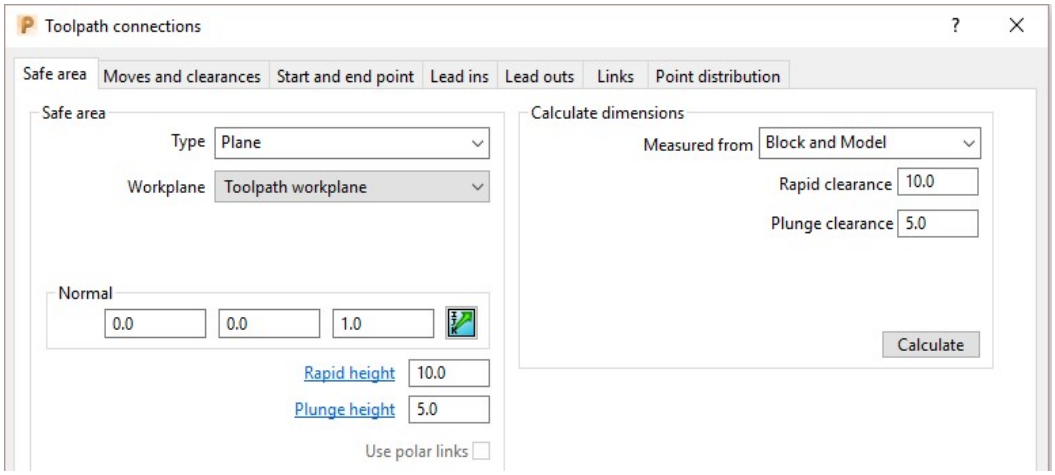
*Double left mouse click on the explorer symbol (in this case ) to activate or de-activate the **Toolpath**.*

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



# Safe rapid move heights combined with Leads and Links

**Rapid height** and **Plunge height** values are set from the **Safe Area** dialog.

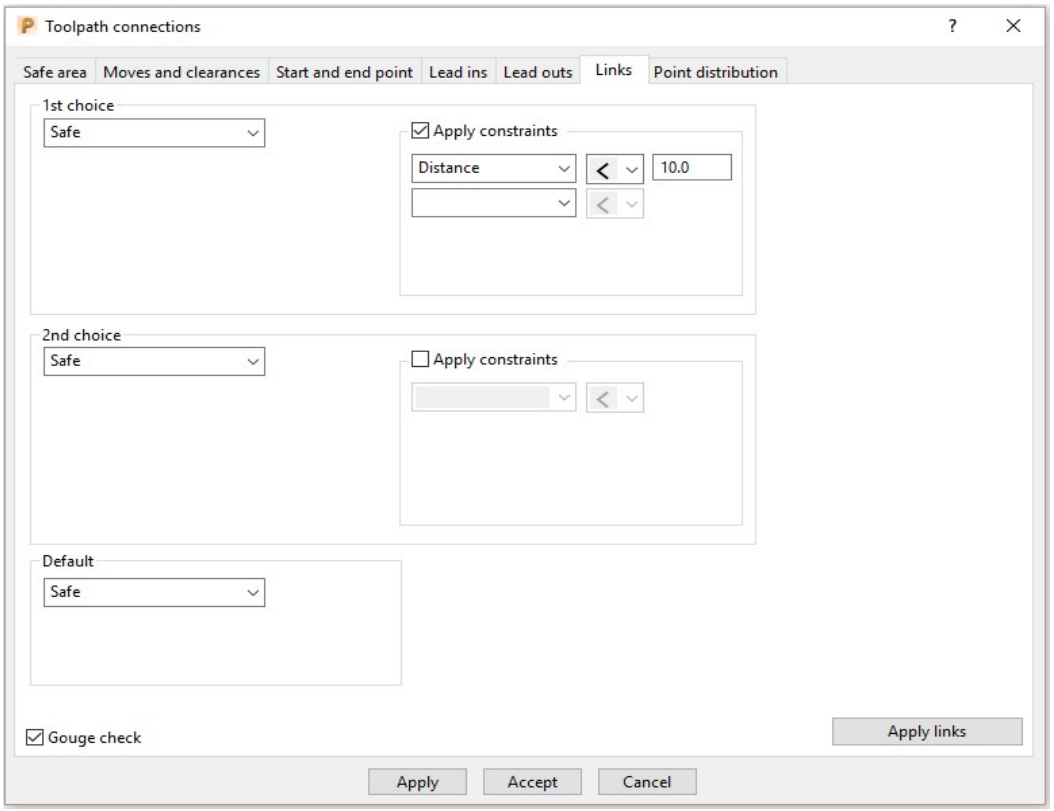



12 Click the **Calculate** button to set the **Rapid height** and **Plunge height** to be at safe distances above the top of the **Model** (whole or selected) or **Block** (whichever is the highest).

***Rapid height** is always above **Plunge height**.*

- **Rapid height** specifies a fixed height at which safe rapid moves occur across the work piece.
- **Plunge height** specifies a fixed height at which the slower, plunge feed rate is applied before the tool finally drops onto a cutting move.

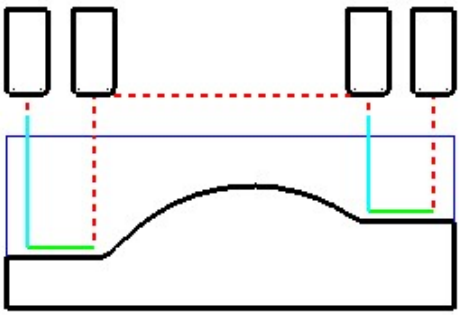
13 **Rapid** and **Plunge heights** work in conjunction with the **Leads and Links** options providing a more comprehensive choice of **Toolpath** link options.



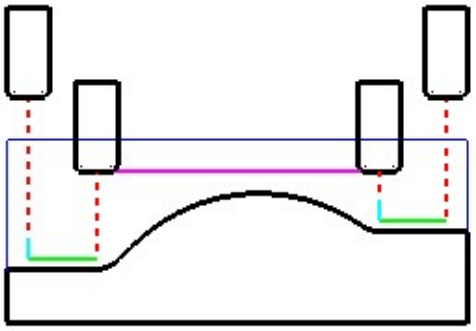


Copyright © Autodesk

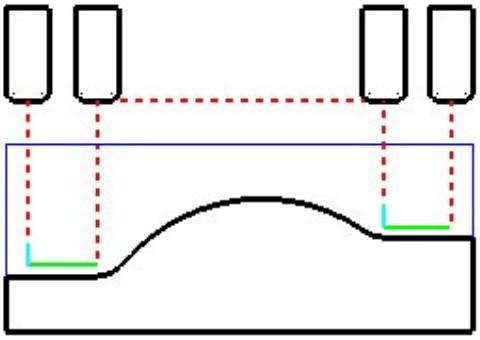
- **Safe** – Applies the **Plunging feed rate** from a fixed height above the job. This is more predictable and reassuring, but the non-cutting (air) moves takes more time and is inefficient, especially in the case of large, deep components.



- **Skim** - enables the downward, rapid feed rate to continue to a specified incremental **Plunge distance** above the full plunge depth before the slow plunge feed rate 'cuts in'. **Skim** then applies a rapid retract to an incremental **Axial clearance** 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 **Plunge distance** measured from the full plunge depth, at which point, the slow **Plunging feed rate** 'cuts in'. The **Plunge** option differs from **Skim** in that all rapid link moves occur at the absolute **Rapid height**.



*The default **Link** move settings in the **Model Area Clearance** strategy template are **1st choice/2nd choice** = **Skim** and **Default** = **Incremental**.*

# Feed Rates

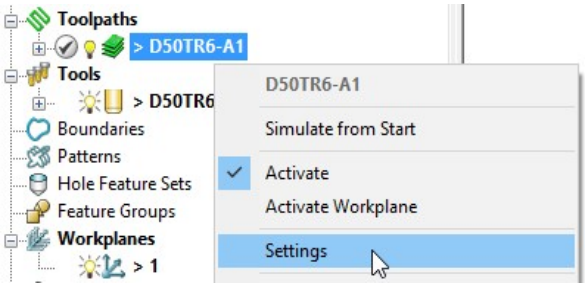
The **Feeds and Speeds** dialog uses the **Style** and **Colour** of elements along a toolpath to register the correct **Rapid Move** or **Feed Rate** values.

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

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

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...**.

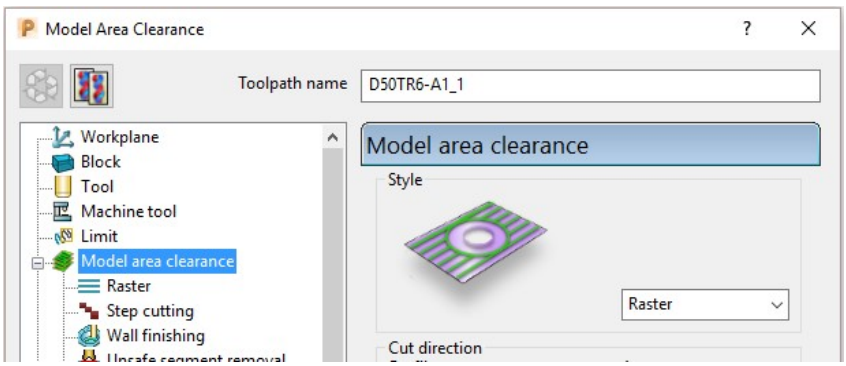



*The toolpath can also be activated or deactivated using the **Activate** option in the context menu.*

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

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

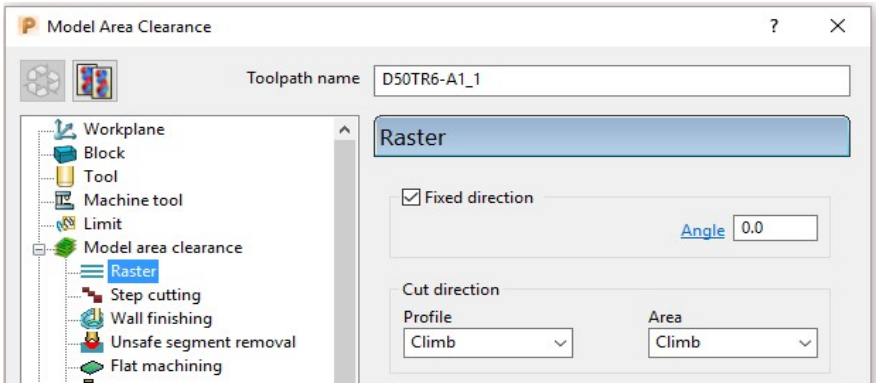
6 On the left of the **Model Area Clearance** dialog is a local ***explorer*** window.



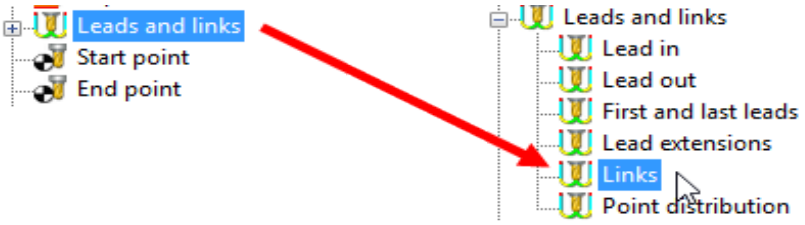


Copyright © Autodesk

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

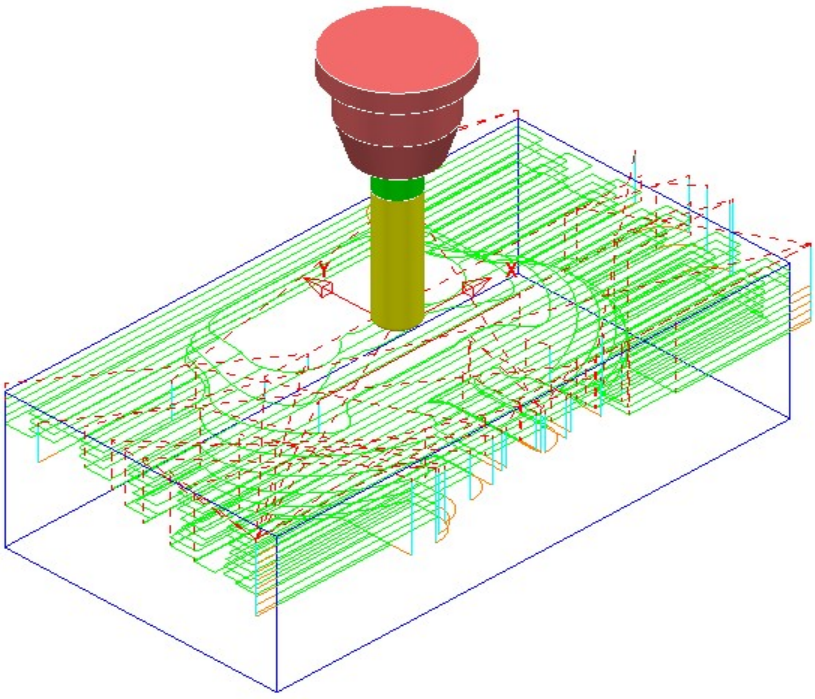



7 From the local ***explorer***, select the ⊞ to the left of **Leads and links** to access the further options **Lead in**, **Lead out**, **Links** etc.




8 In the **Links** page change the **1st choice**, **2nd choice**, and **Default** link moves to **Safe**.

9 At the bottom of the main dialog, select **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).

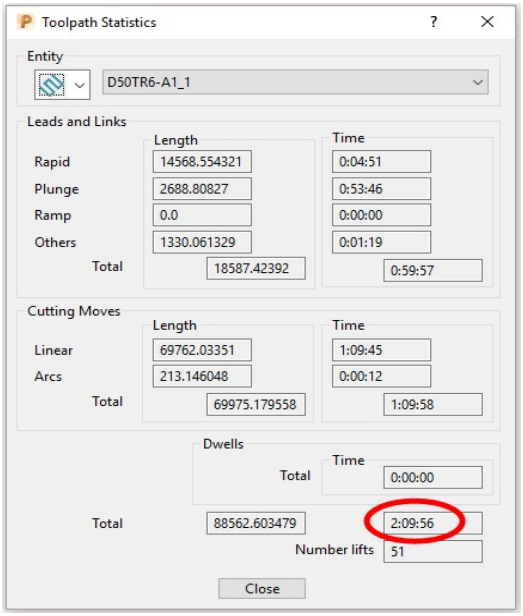
All **Plunge** feedrate (pale blue) moves start from a height between the **Safe Z** and the top of the part (default height is **5**).

# 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.




In this case, the total machining time is around **2hrs 10mins.**

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 **1hrs 30mins**.

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**.

*If not already displayed the above **Simulation** toolbar will appear automatically.*

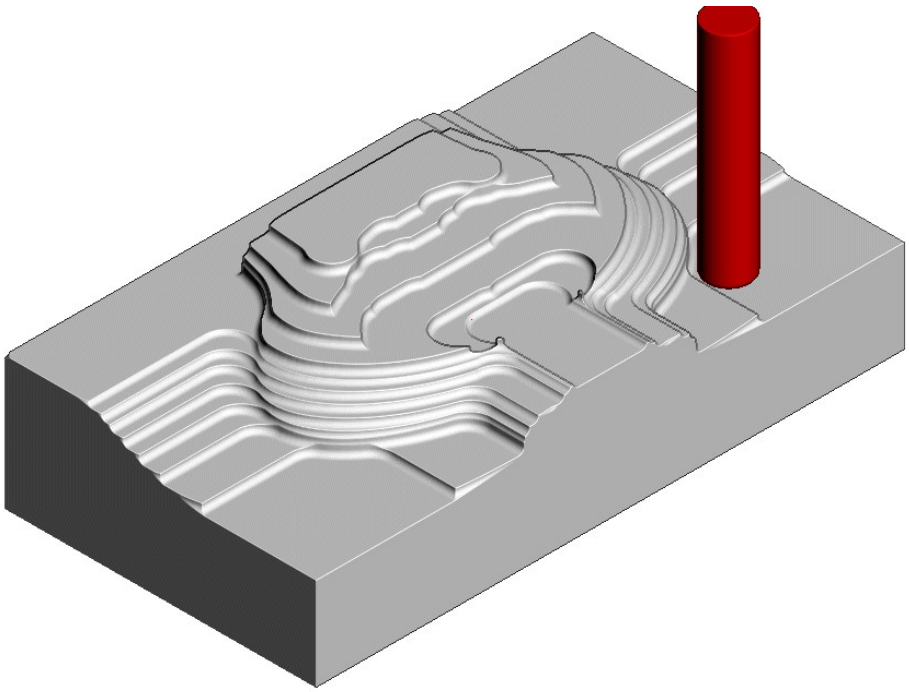
4 Use the **Play** button to run the simulation and control the speed with the slider bar.

5 If the **ViewMILL** toolbar is not already open, from the **main** pulldown menus select **View – Toolbar – ViewMILL**.

Copyright © Autodesk

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.



8 Switch the **ViewMILL** to **No Image** to return to the **PowerMill** session and then disconnect **ViewMILL** 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**, the **ViewMILL** session will continue to update in parallel with any further toolpath simulations.

# Saving the Project

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

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

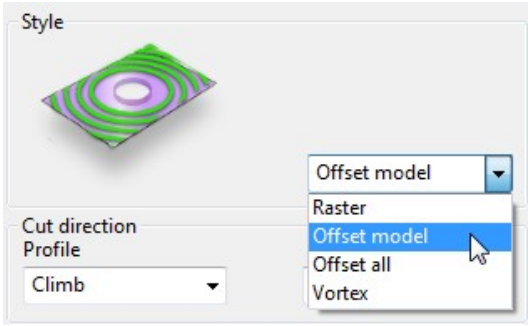
We will now ***delete*** the contents of the **PowerMill *explorer*** and return the **PowerMill** session back to the original default settings.

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.

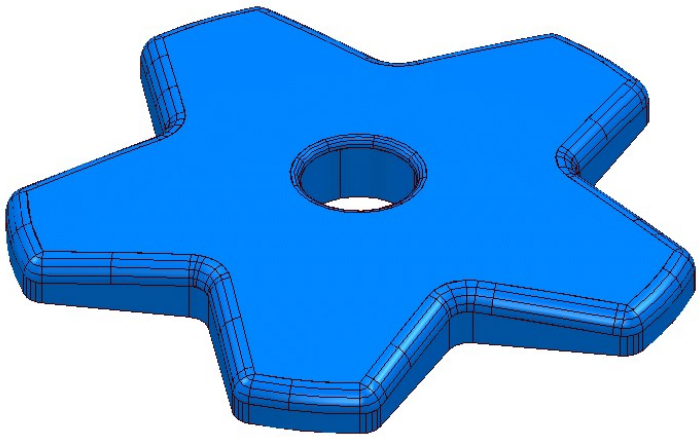
The strategy is based on the profile around the component at each Z Height, being continually offset outwards, to be trimmed to the limits of the material **Block**.

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** to the full model dimensions.

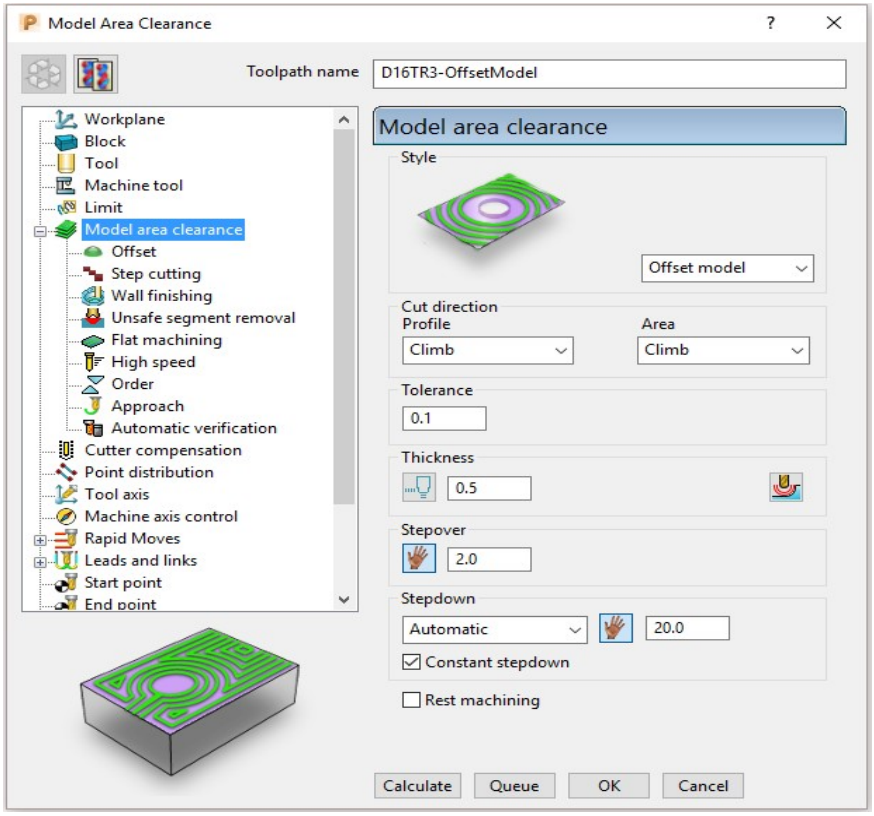
5 **Lock** the **Z max** and **Z min** values, enter an **Expansion** value of **10** and **Calculate**.


Copyright © Autodesk

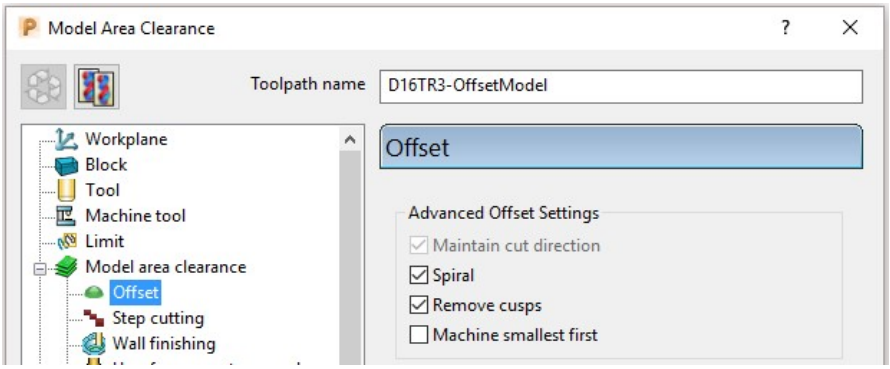
6 In the **Rapid move heights** dialog, select **Calculate**.

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

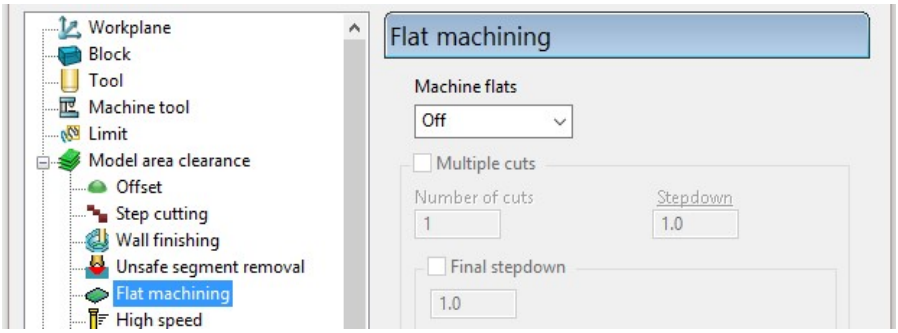
8 Input data into the pages of the form exactly as shown below.




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



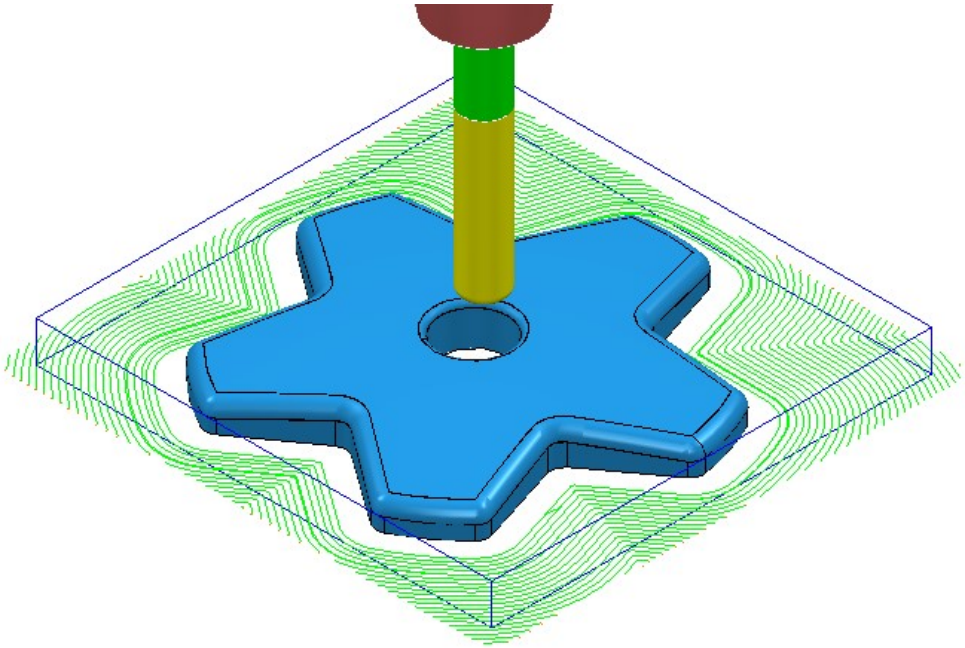

10 Select the **Flat machining** page and select ***Machine flats*** - **Off**.



11 Select **Unsafe segment removal** and in the dialog ***tick* - Remove segments smaller than threshold**.

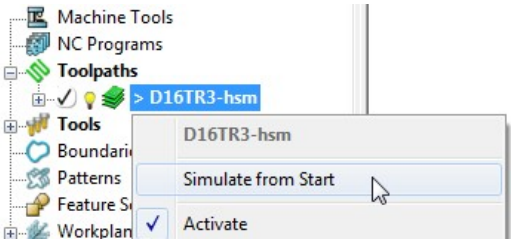
12 At the bottom of the dialog, click **Calculate** to create the Toolpath.

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



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

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




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

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

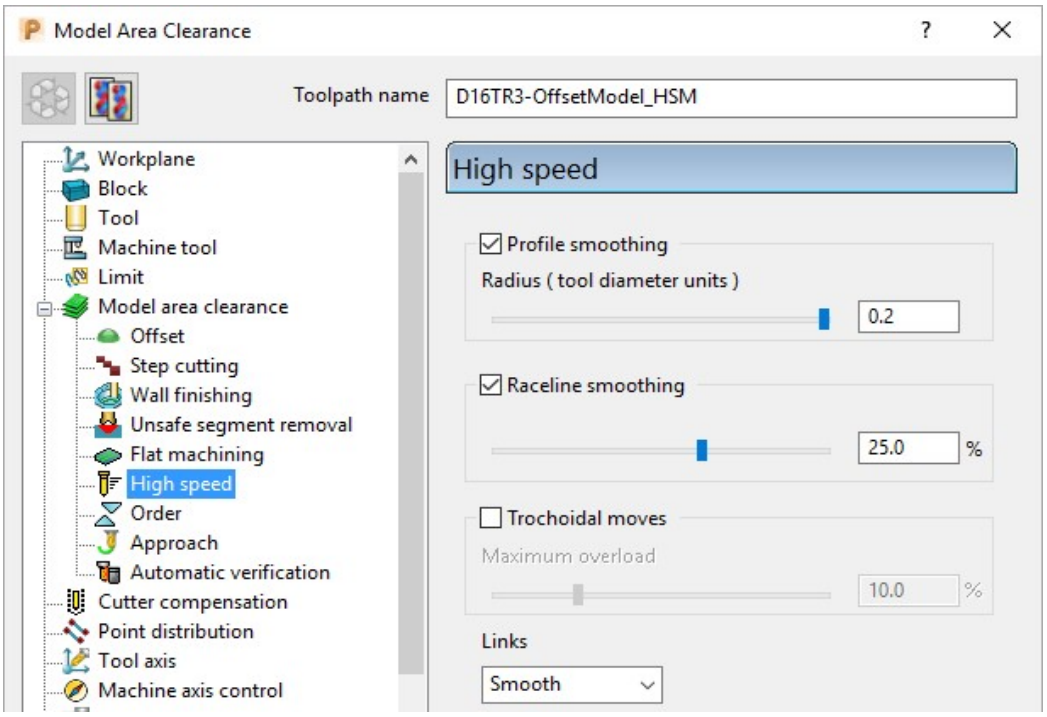
The tool continuously, climb mills, starting each tool track at a distance from the material **Block** to allow the tool to reach optimum **Feed Rate** before contact. The material removal rate in this type of strategy is generally very consistent. The strategy can be further improved using advanced settings to progressively smooth away sharp corners and straighten the toolpath geometry as it offsets further away from the component form.

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

17 Click to create an active copy of the **toolpath**.

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

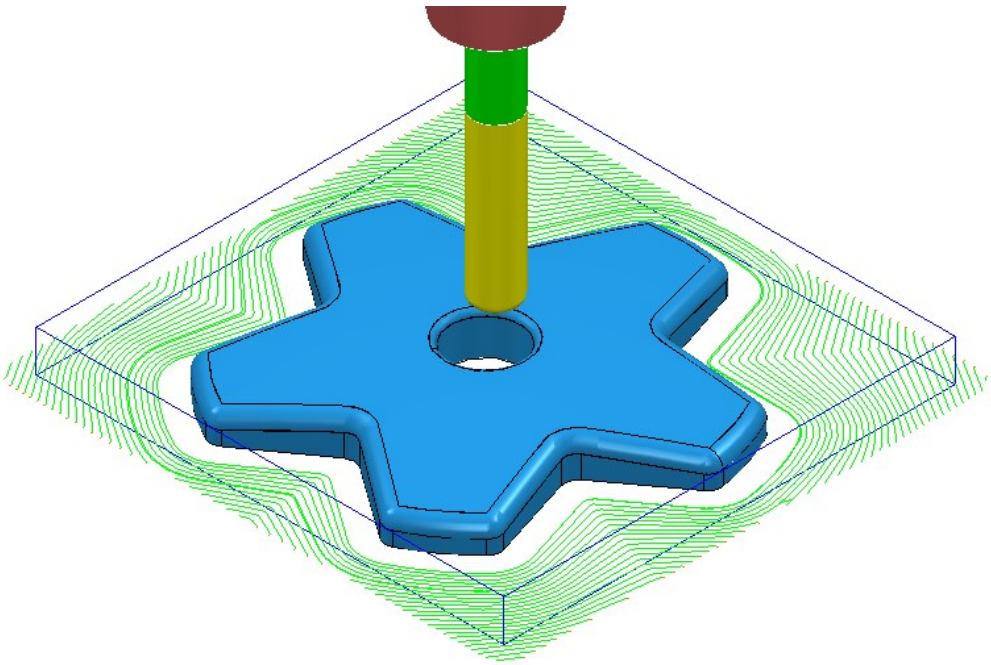

Copyright © Autodesk




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

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

i Click **Calculate** to create a new toolpath **D16TR3_OffsetModel_HSM** with the improved smoothing characteristics.

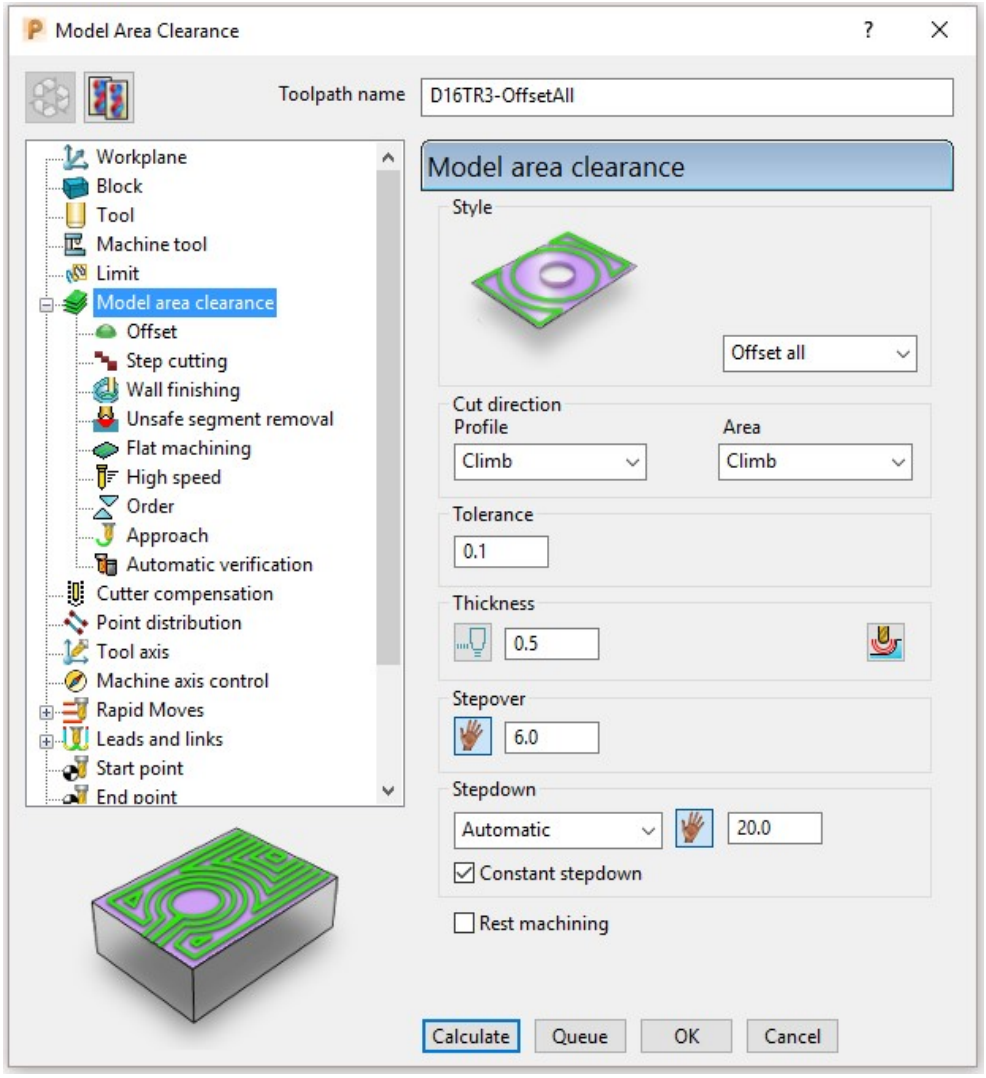


Compare the **Model Area Clearance** toolpaths. Note the progressive straightening of tool tracks on the second strategy with **Raceline smoothing** being applied.

# Offset All – Basic example

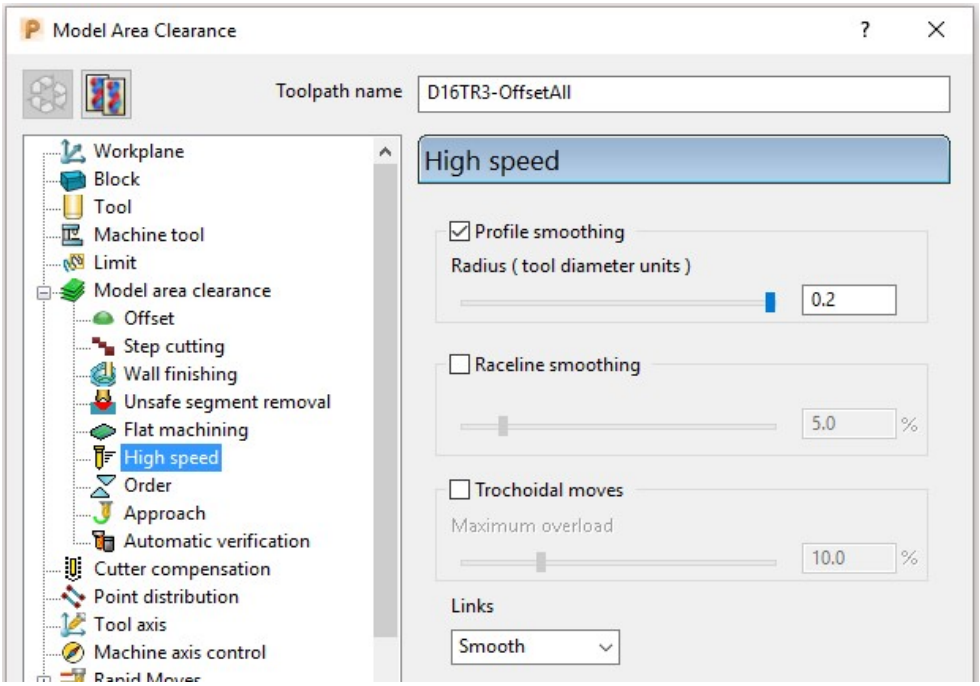
This variant of the **Offset Area Clearance** is better suited to softer materials such as modelling foam, resins etc. It will remove material with a larger **stepover** and the minimum amount of lifts, but this is more than likely to result in the tool tracks containing sharp changes of direction and 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**.
2. 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.




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

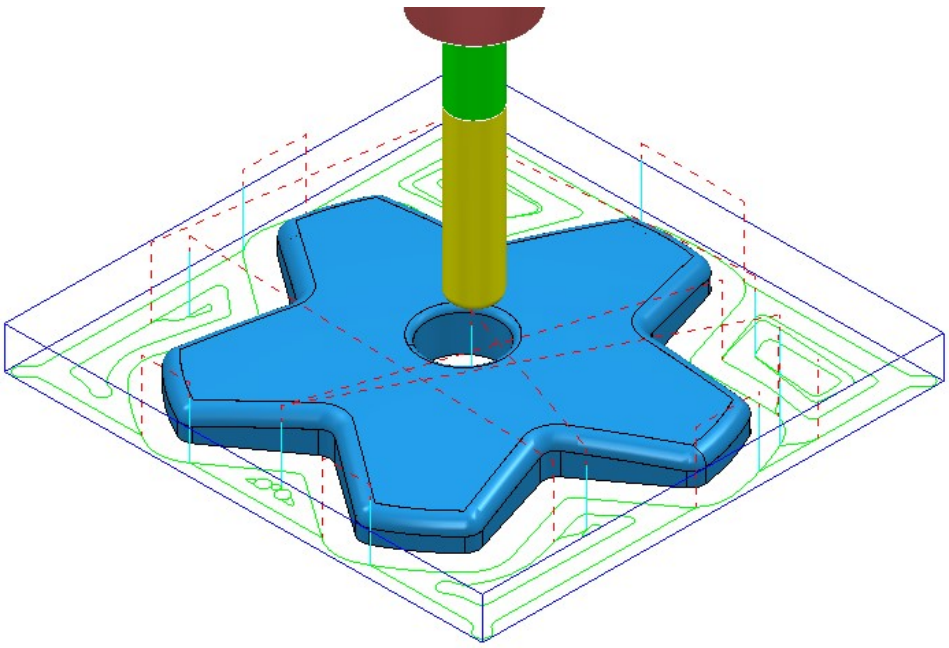

Copyright © Autodesk




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.

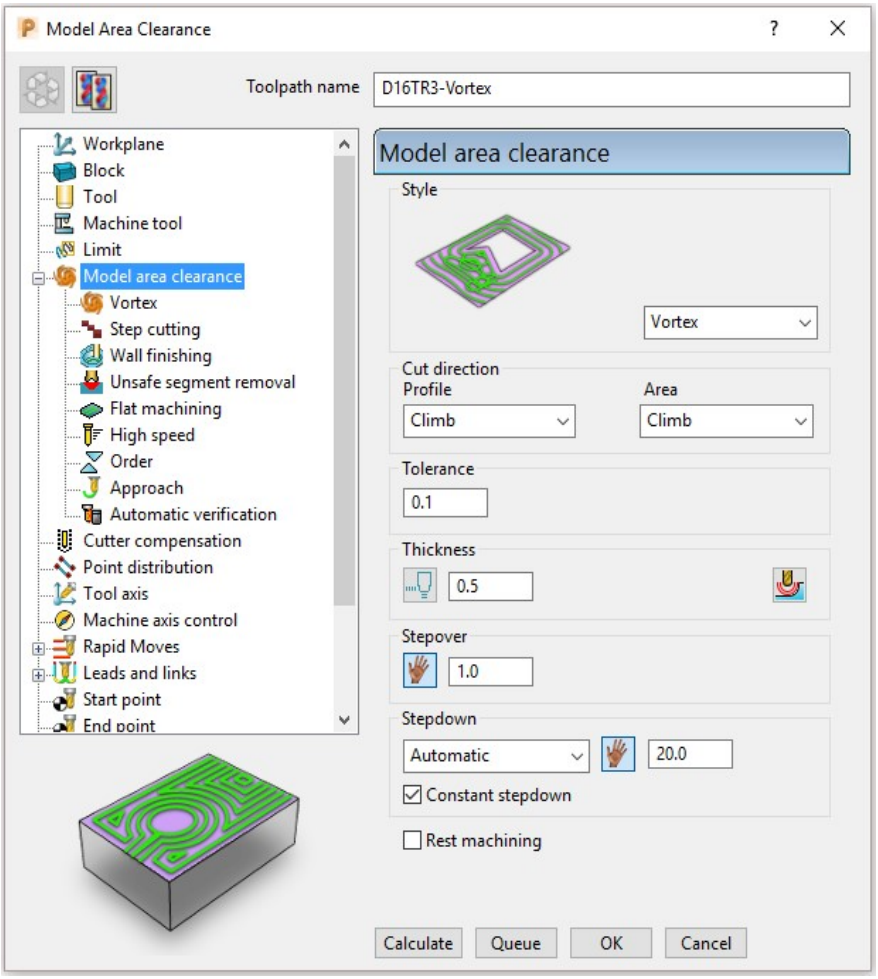


In the above **Offset-All** style toolpath, the tool plunges into the individual areas and profiles it's way outwards following both the component **model** and **Block** forms.

# Vortex - Basic example

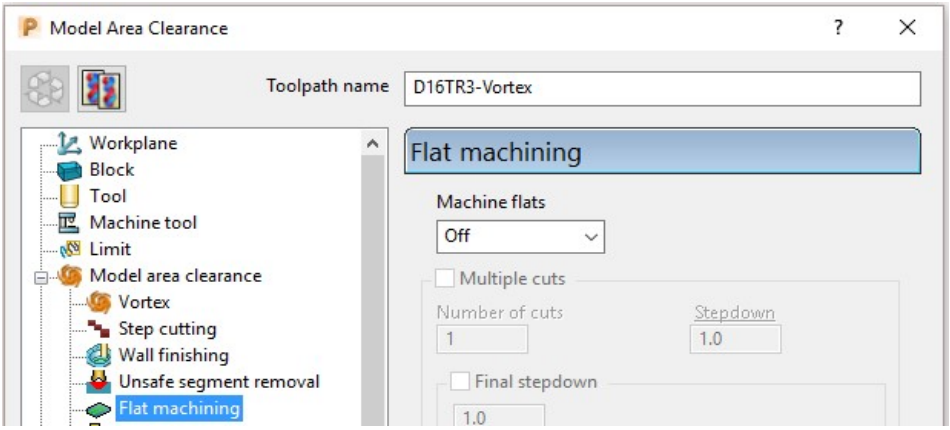
This specialist, variant of **Model Area Clearance** is primarily designed for use with solid carbide tools on hard materials. Wherever necessary the **Vortex** strategy applies complex, optimised trochoidal, movement to achieve the ultimate control over the loading on a cutting tool.

1. In the **PowerMill *explorer***, right-click the active **D16TR3-HSM 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**.

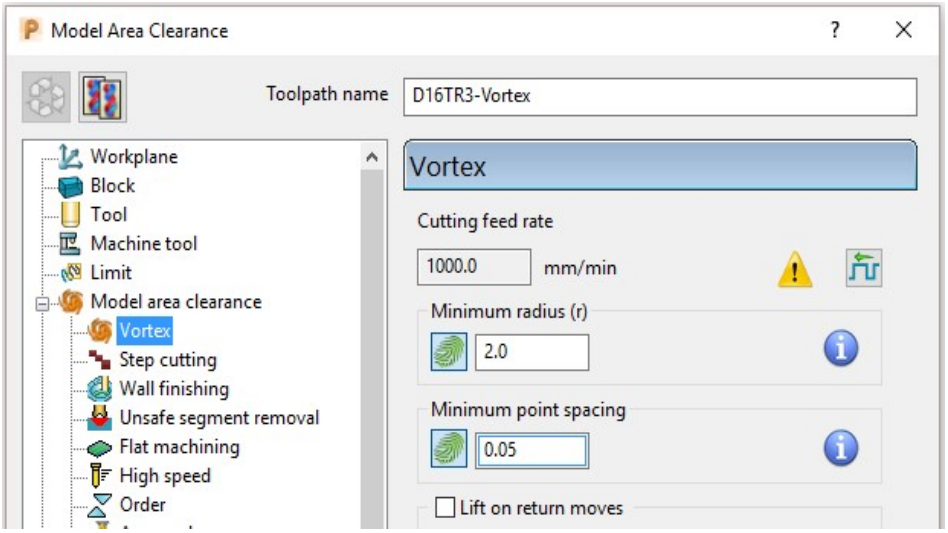



   a. Input **Thickness 0.5**
   b. Input **Stepover 1**
   c. Input **Stepdown 20**
4. Select the **Flat machining** page and set *Machine flats* to **Off**.


Copyright © Autodesk

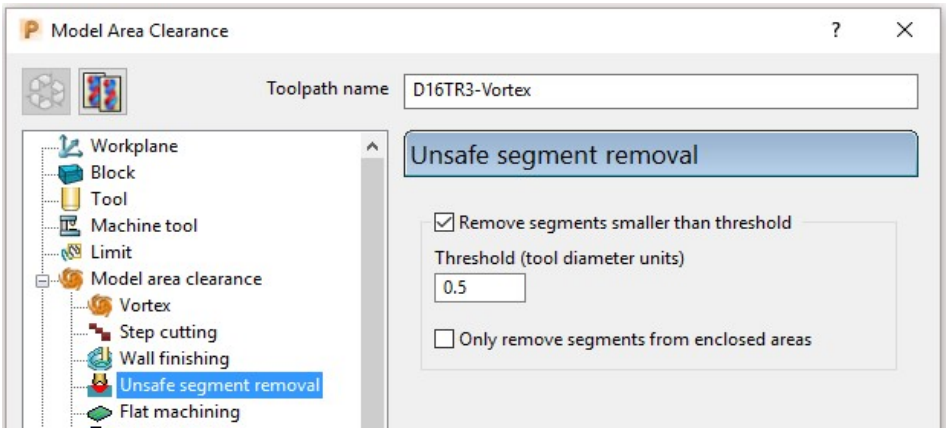



5 Select the **Vortex** Page and set **Minimum Radius (r)** to **2.0**



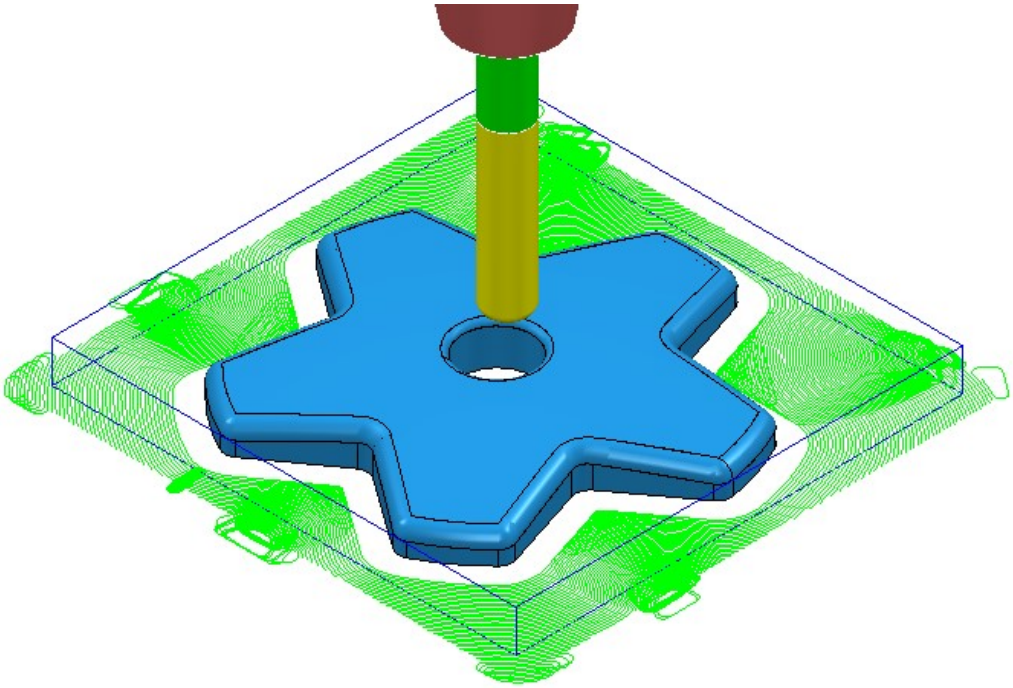

Increasing the **Minimum Radius** value will enlarge the inherent ***Trochoidal*** moves which should in turn enable a higher **Feedrate** to be used with the **Vortex** strategy.

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




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**.


Copyright © Autodesk

# 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.

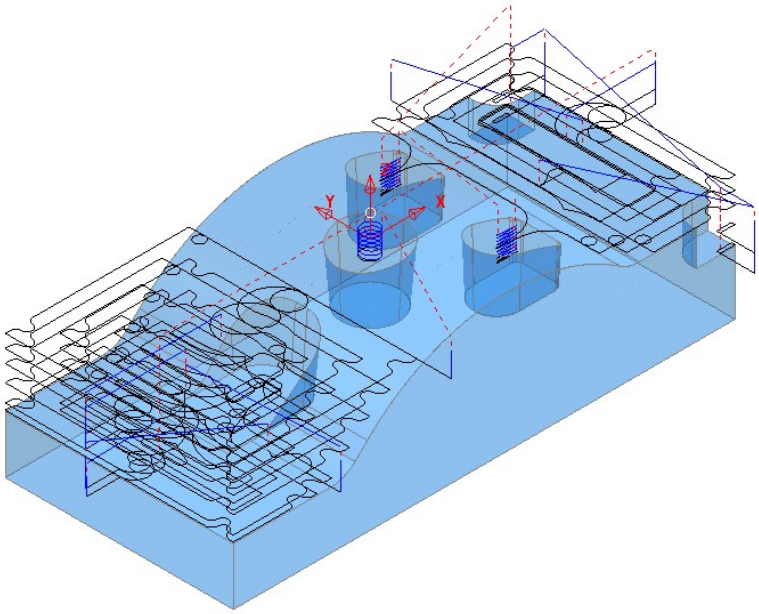
12 **Open** the read-only **Project**:

**.....\PowerMill_Data\Projects\MountingBlock-Start**

13 **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\MountingBlock**

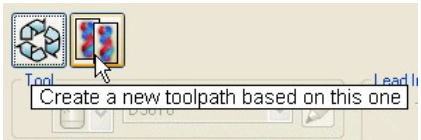
14 **Activate** the **Toolpath** named **No-AreaFilter**.



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.

15 Right-click on the **Toolpath** named **No-AreaFilter** and from the local menu, select **Settings** to open the original **Model Area Clearance** dialog.

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




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

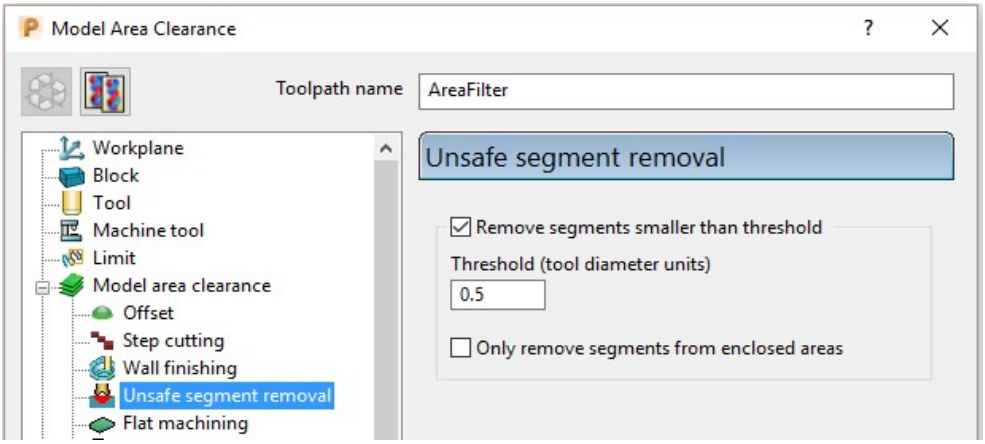
18 In the **Model Area Clearance** dialog, select the **Unsafe segment removal** page.

19 **Tick** the option, **Remove segments smaller than threshold**.

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

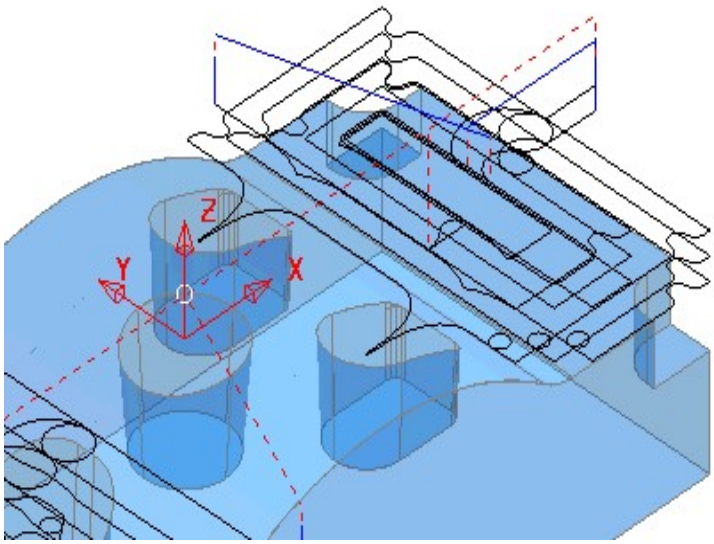
21 **Un-tick** the **Only remove segments from enclosed areas** option.

Copyright © Autodesk




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

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

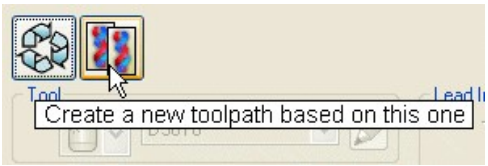



**Unsafe Segment** removal has been applied to prevent the cutter attempting to plunge into and machine confined areas (Pockets).

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

23 In the **PowerMill *explorer***, right-click the toolpath named **AreaFilter** and select **Settings** to open the original **Model Area Clearance** dialog.

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

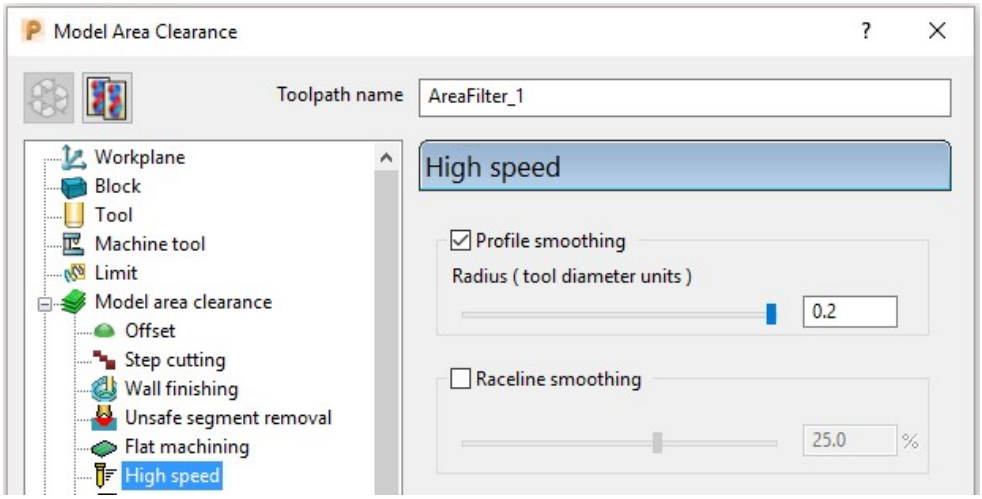



25 Rename the copy of the toolpath as **AreaFilter-ProfileSmooth**.

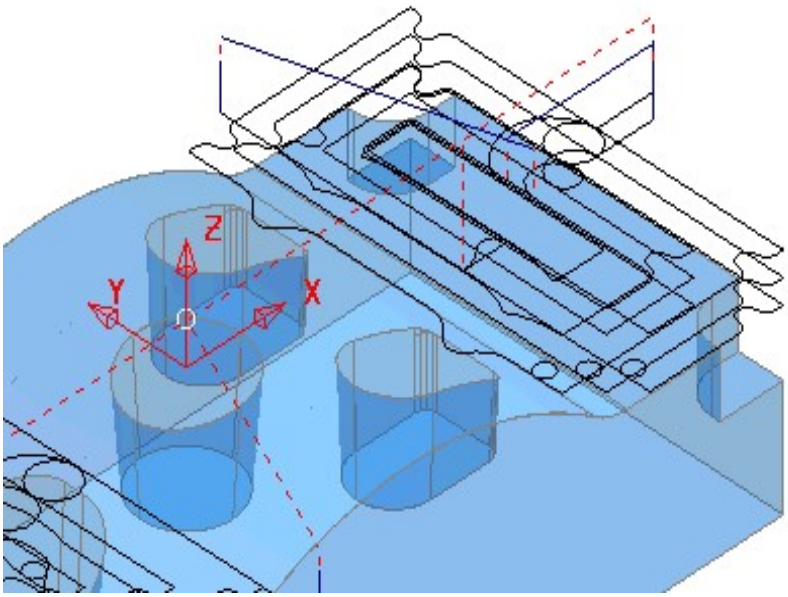
26 In the **Model Area Clearance** dialog, select the **High Speed** page.


Copyright © Autodesk

27 Select the **Profile Smoothing** option with the Radius (Tool Diameter Units) value set to **0.2**




28 **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.

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

30 From the **main** pull down menus select **File > Delete All**.

31 From the **main** pull down menus select **Tools > Reset Forms**.

Copyright © Autodesk

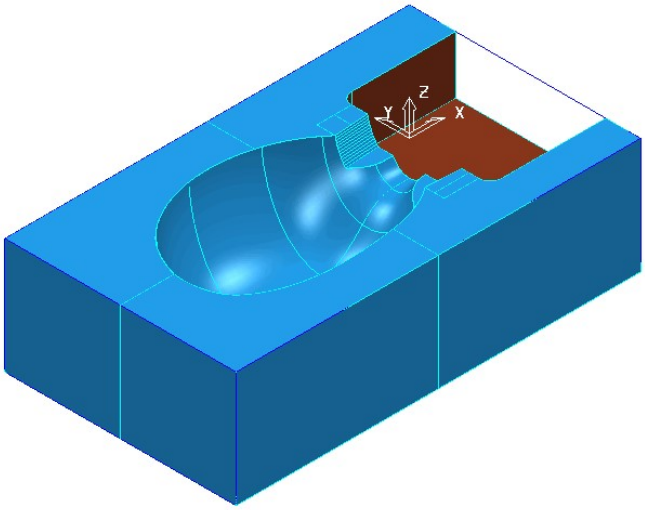
# Model Rest Area Clearance example

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**



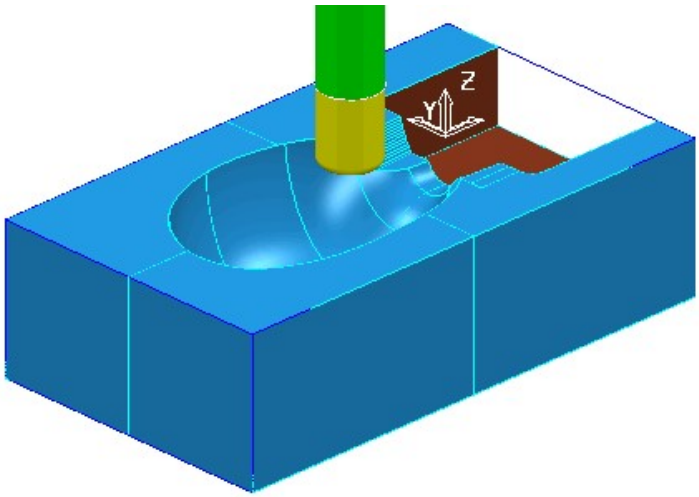

2 **Save Project** as:

**.....\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**.

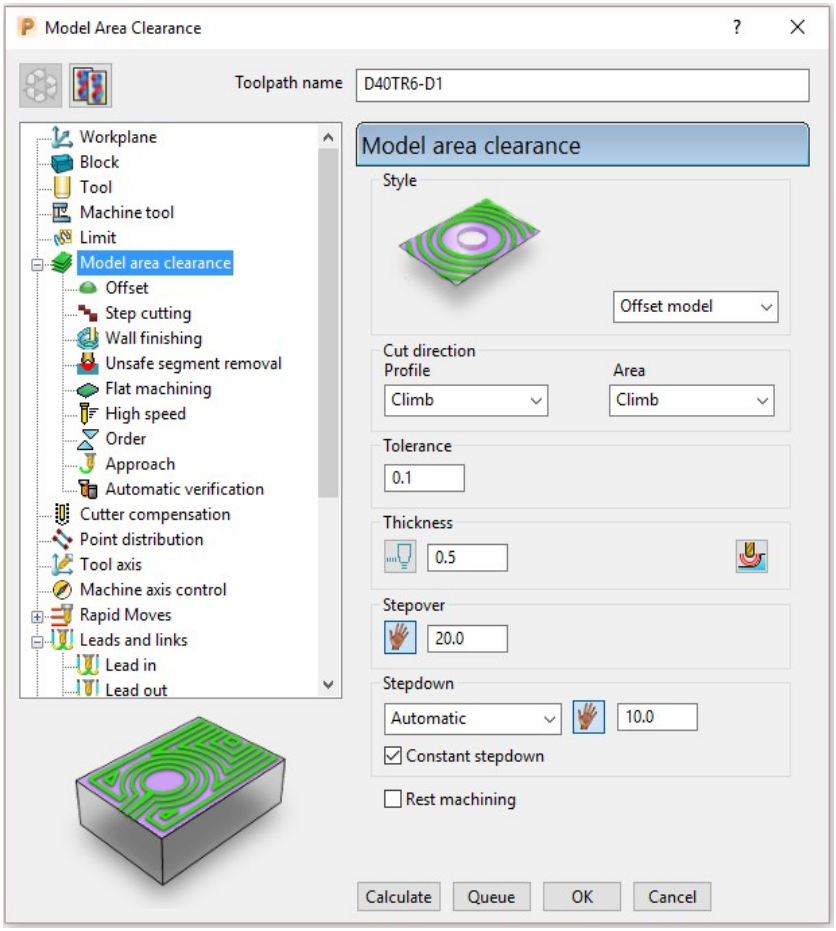
5 in the **Rapid move heights** dialog select **Calculate**.





Copyright © Autodesk

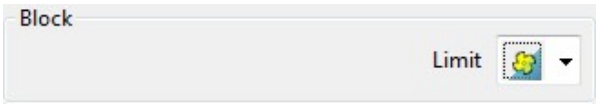
6 Select **Toolpath Strategies** , and from **3D Area Clearance** open a **Model Area Clearance** strategy.



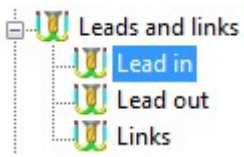

7 In the **Model Area Clearance** strategy dialog:

- 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**.

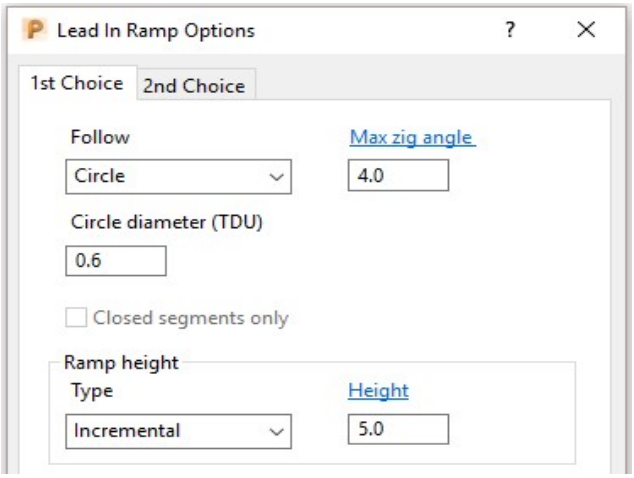



9 In the local explorer, expand the **Leads and Links** options, and then select **Lead In**.



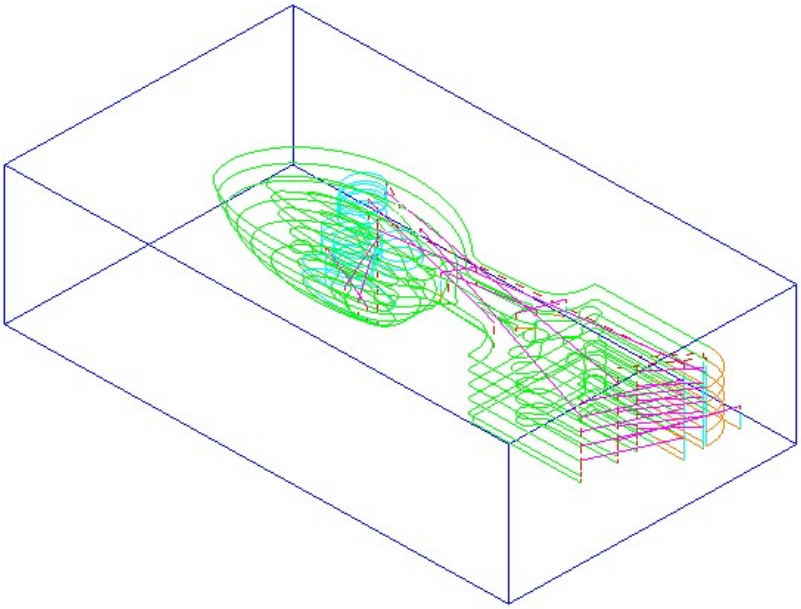
10 In the **Lead in** page, select **1st Choice** - **Ramp** - (Open the ramp options form).

11 Input the following **Ramp Options**:



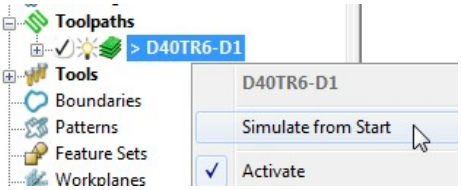

12 Click **Calculate** to process the **Model Area Clearance** toolpath.

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.





Copyright © Autodesk

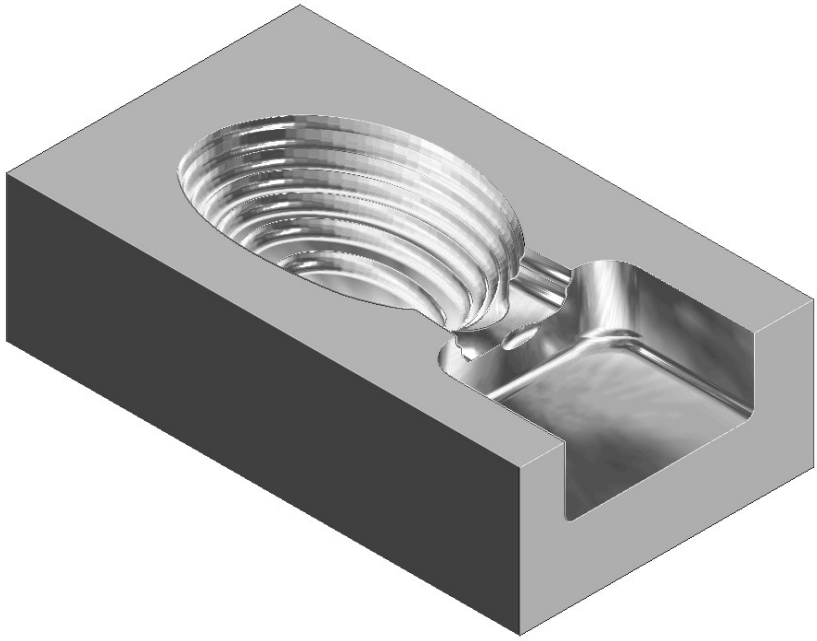
15 Ensure that the **ViewMILL** toolbar is also active.

16 Switch the **ViewMILL On/Suspend** setting to a green sphere , and then select the shiny shaded image icon .

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

*Undrawing the tool will help to speed up the simulation. You can control the display of the tool by left clicking on the tool entity, light bulb in explorer.*

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



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.

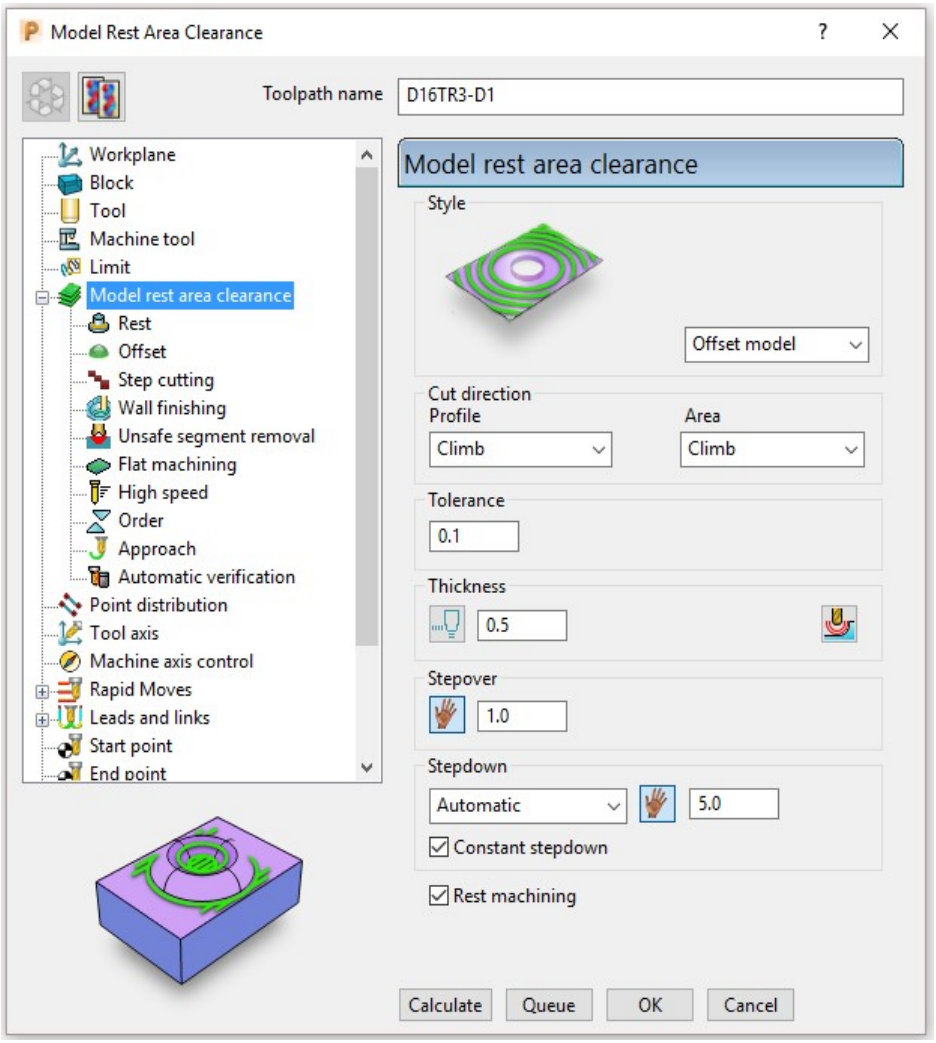
18 Click to suspend **ViewMILL**.

19 Click to return to the **PowerMill** window.

# Rest Roughing using a Reference Toolpath

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

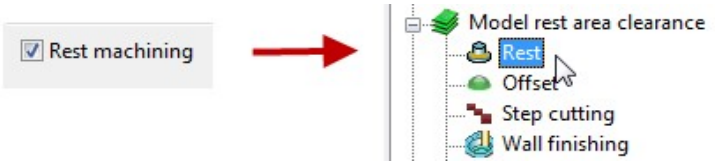
2 Select **Toolpath Strategies** , and from **3D Area Clearance** open a **Model Rest Area Clearance** strategy.




3 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**

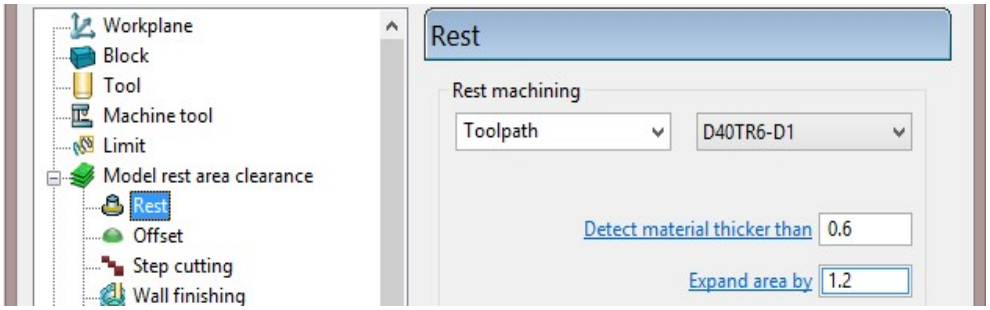
4 Tick **Rest machining** box to offer the **Rest** page in the strategy explorer.



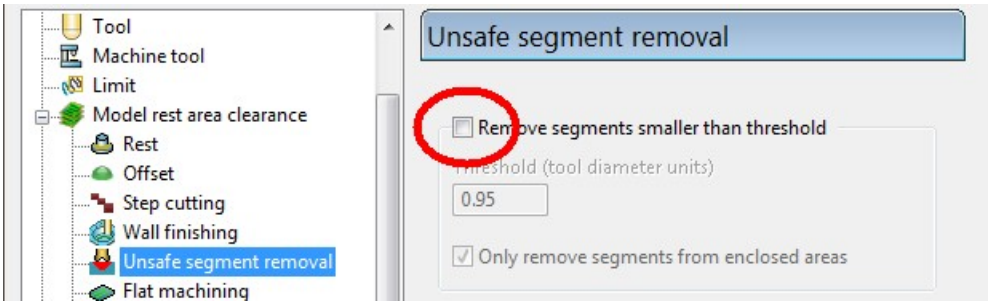


Copyright © Autodesk

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

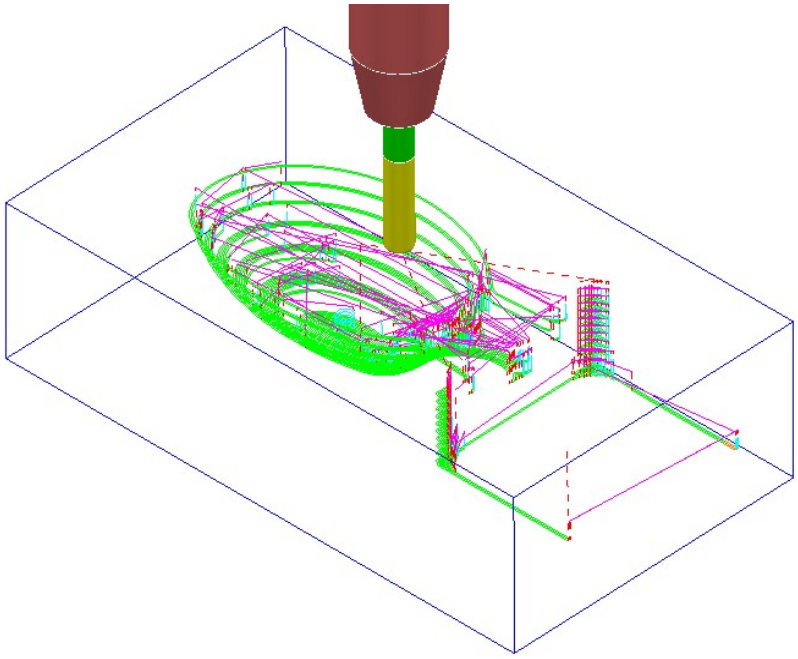



6 In the local explorer, select **Unsafe segment removal** page and if set, un-tick the **Remove segments smaller than threshold** option.




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

8 Click **Calculate** to process and create the additional **Offset Model - Area Clearance** toolpath and then **Close** the dialog.



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

**.....\COURSEWORK\PowerMill-Projects\Wing_Mirror_Die**

Copyright © Autodesk

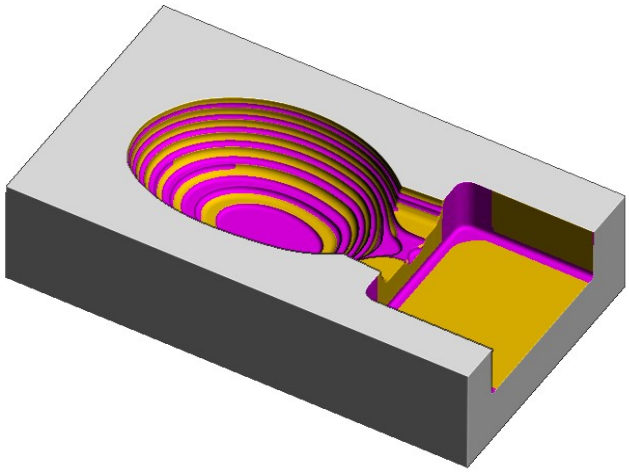
10 Switch the **ViewMILL On/Suspend** button to a green sphere .

11 Select the **Rainbow Shaded** Image .

12 Right mouse click on the **toolpath D16TR3-D1** and select **Simulate from Start**.

13 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.

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

## Rest Machining using a Stock Model

An alternative method for rest 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**.

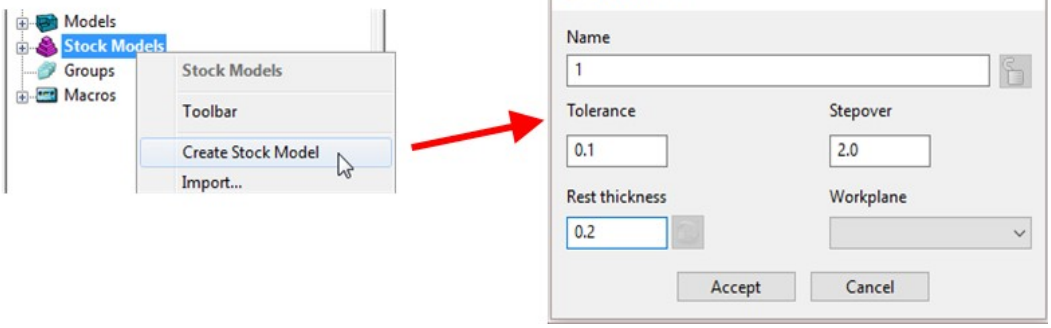
*It is not possible to apply a rest machining strategy referenced to a toolpath that was created to a different workplane.*


Copyright © Autodesk

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.

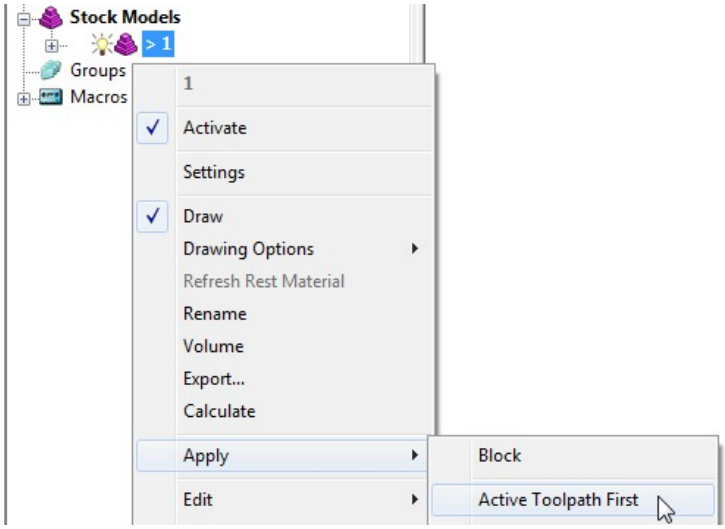
1 In the **PowerMill *explorer***, activate the toolpath **D40TR6-D1**.

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



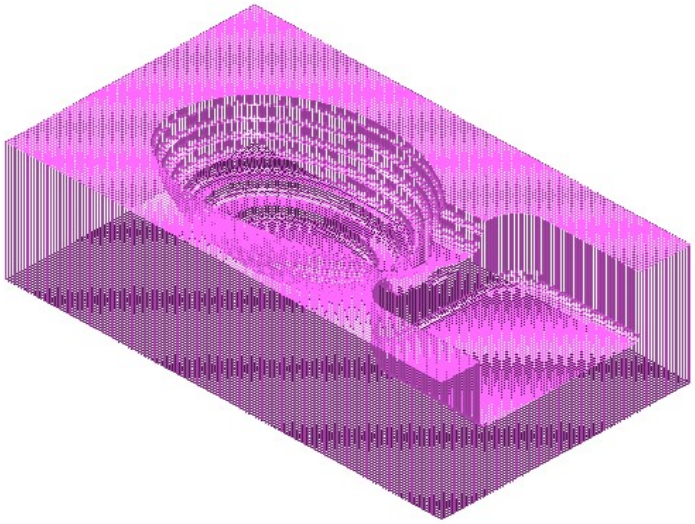

3 **Accept** the above default values for the **stock model** triangle mesh creation.

4 In the **PowerMill *explorer***, right-click the newly created **Stock Model 1** and from the local menu 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**.

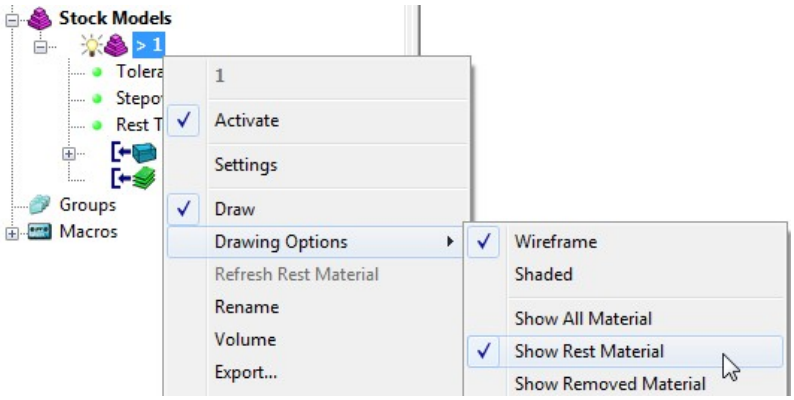


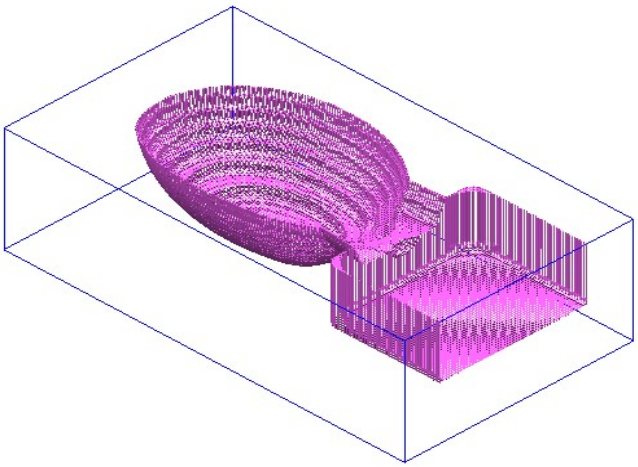
A triangular mesh will appear representing the state of the material after the effect of the first machining strategy, **D40TR6-D1**.

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**).

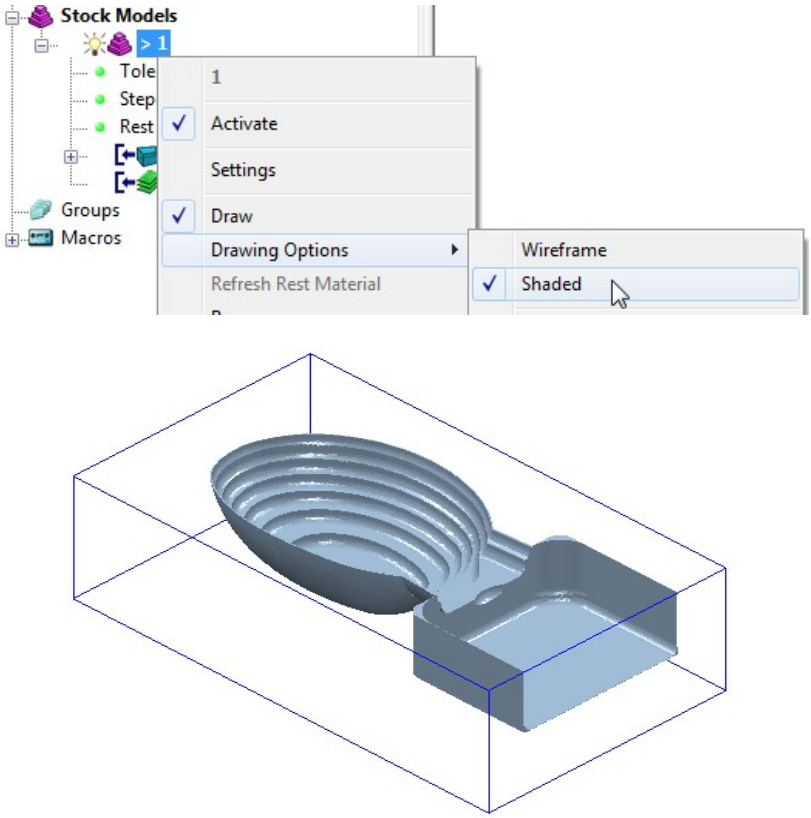
**6** From the same menu, select **Drawing Options** > **Show Rest Material.**



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


Copyright © Autodesk

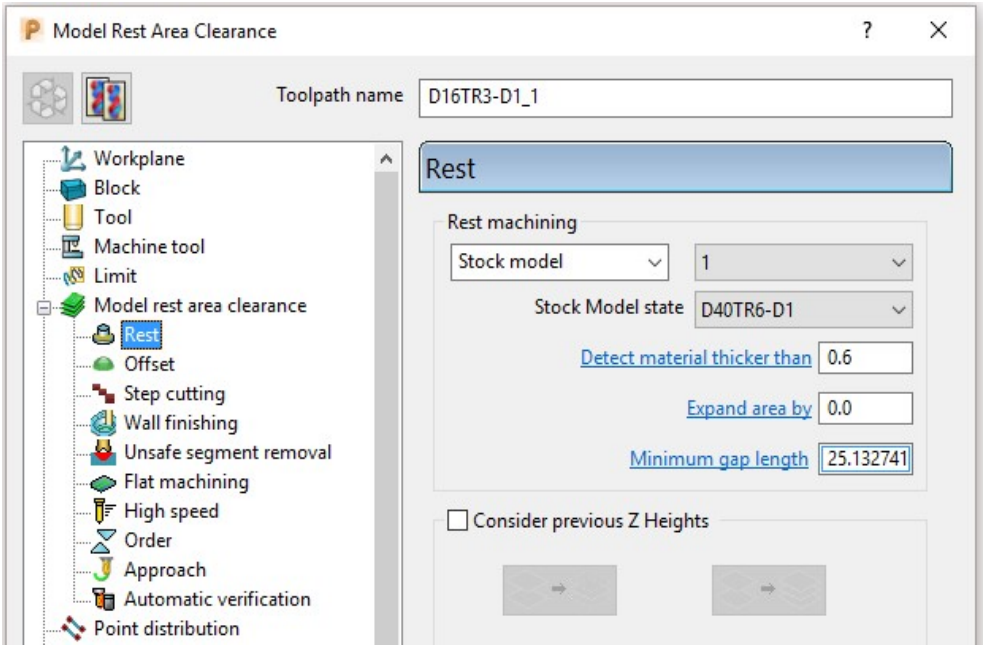



8 In the **PowerMill *explorer***, activate the toolpath **D16TR3_D1**.

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

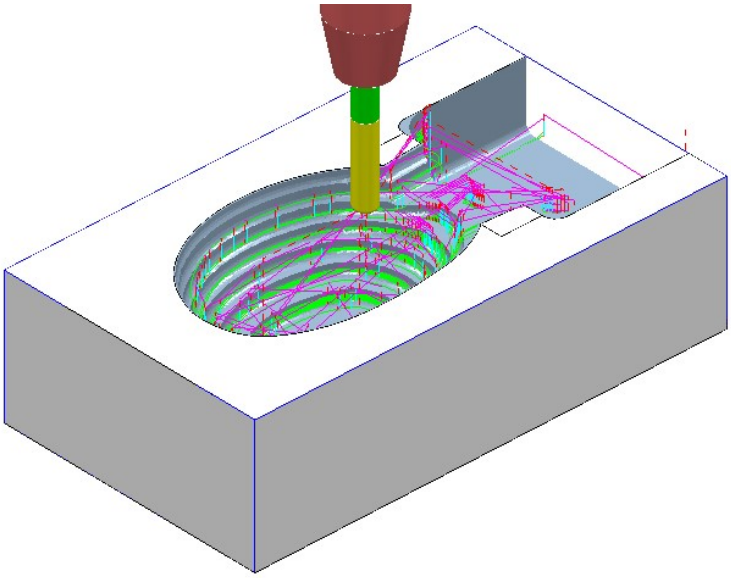
10 Click to create a copy of the toolpath.

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

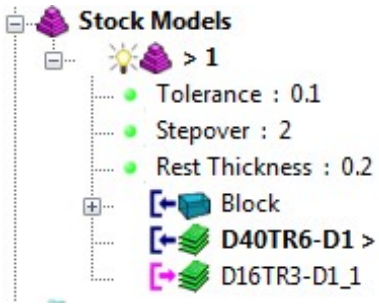



12 Click **Calculate** to create an alternative and more efficient rest machining toolpath named **D16TR3-D1_1**.

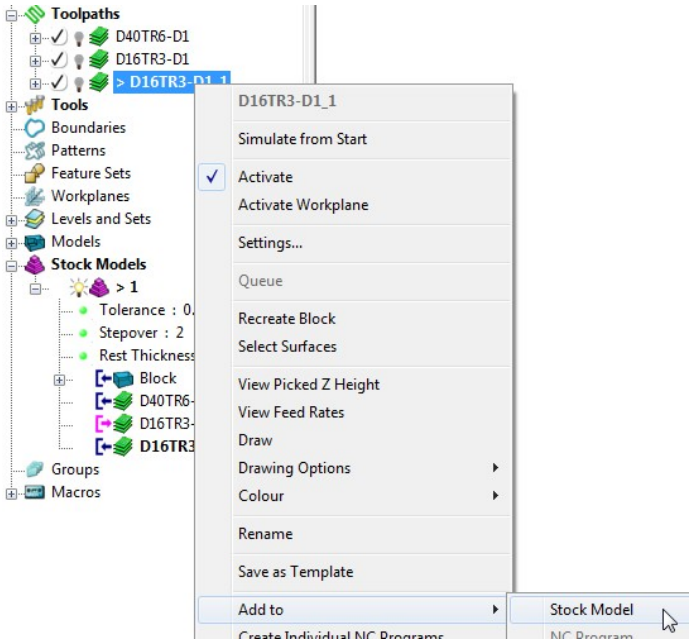
13 Close the machining strategy dialog.



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.



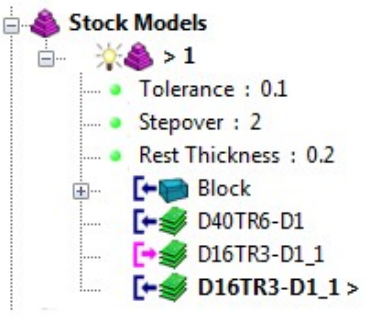

14 In the **PowerMill *explorer***, right-click the toolpath, **D16TR3-D1_1** and select **Add to** > **Stock Model**.

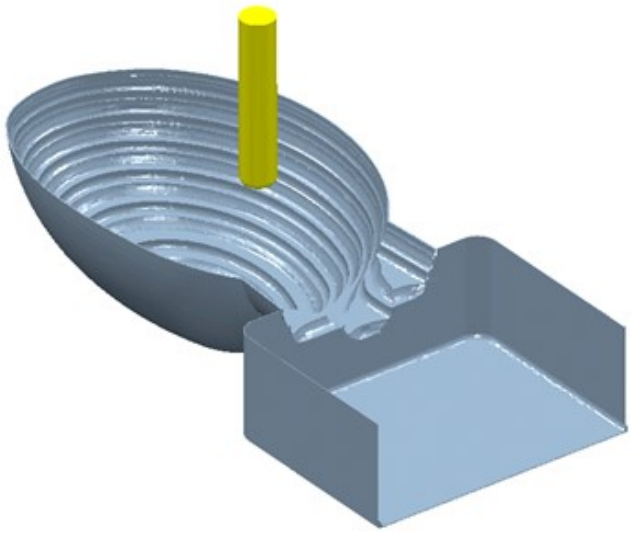



*At this stage, the **Stock Model** will disappear from the screen.*


Copyright © Autodesk

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**

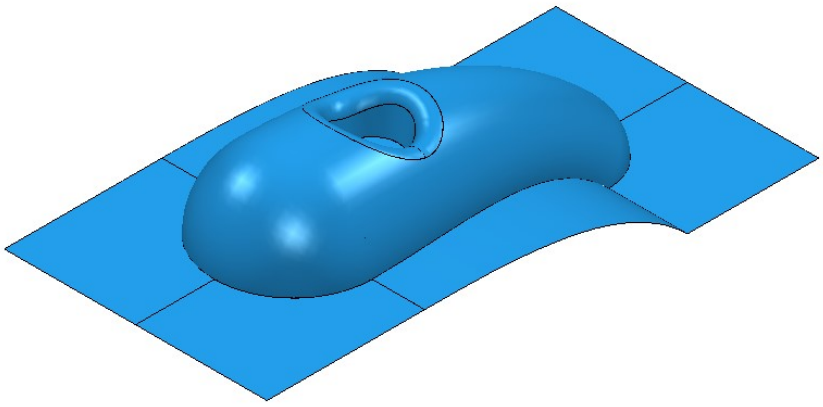
17 From the **main** pull down menus select **File** > **Delete All**.

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

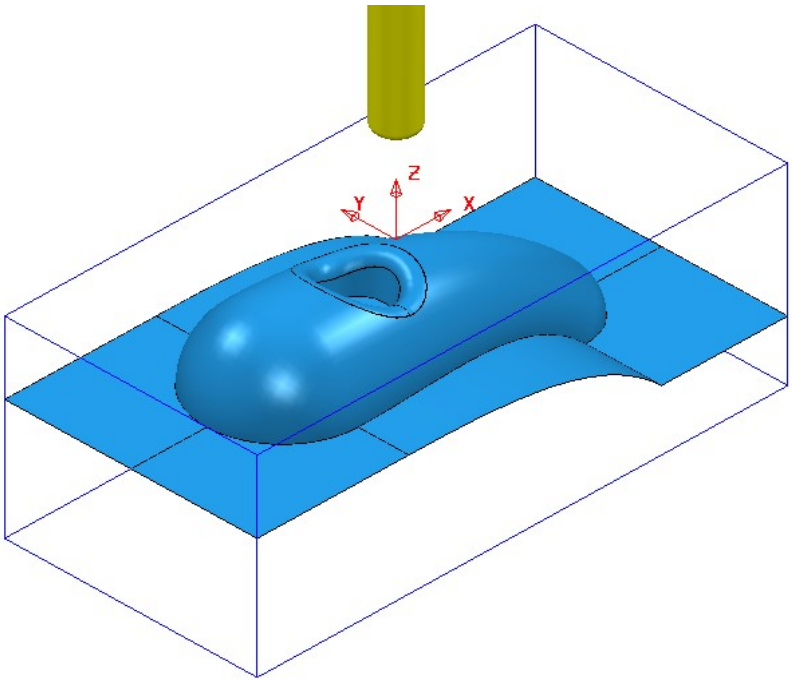
3 Create the following tools:

- **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**

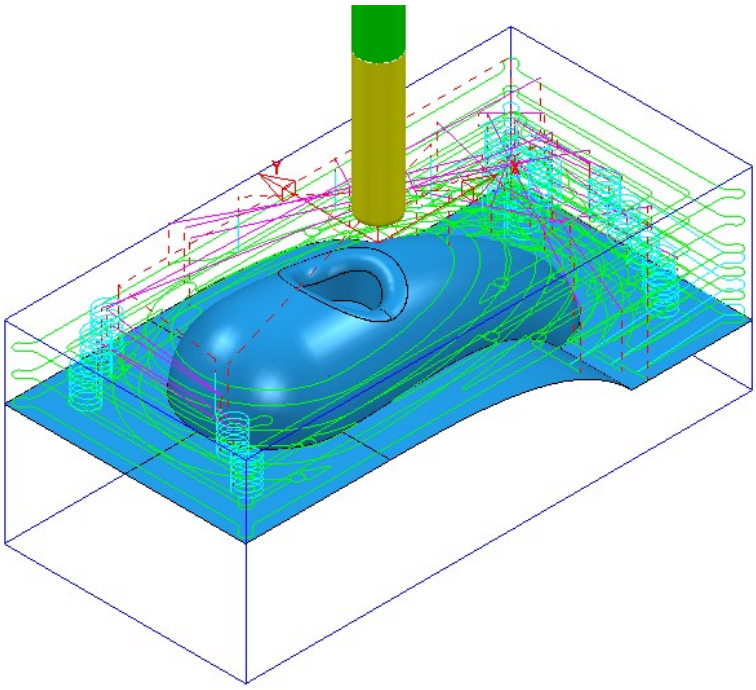
e In the **Flat Machining** page use **Level**.

f Select the **Unsafe segment removal page**.

g **Tick** the box **Remove segments smaller than threshold** and leave the other settings as default.

h In the **Leads and links – Lead in** set **1st choice** as **Ramp** and in ramp options use **Follow circle – Max zig Angle 4.0 – Circle diameter 0.65**


Copyright © Autodesk



2 Create another **Model Area Clearance** strategy named **D12TR1-RGH1** using:

a The **D12TR1** tool.

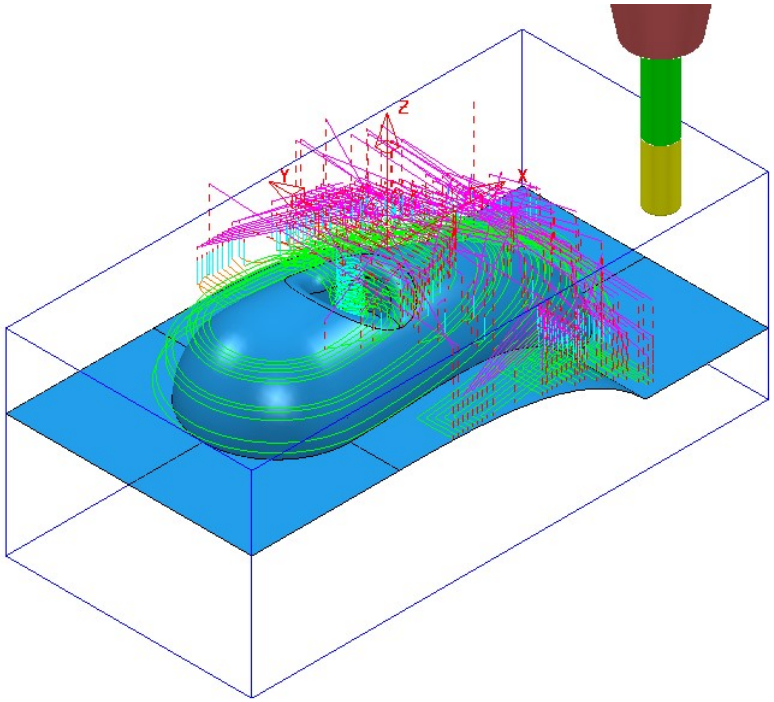
b The **Style** as **Offset Model**.

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**.

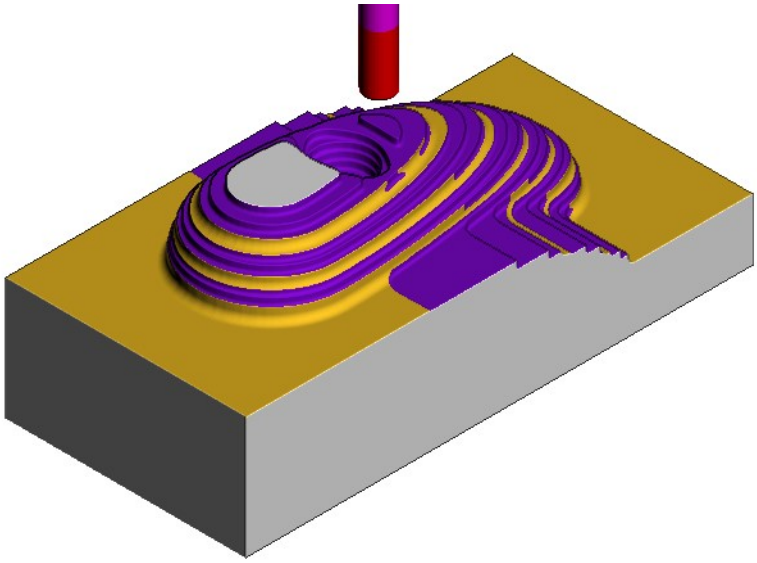


3 **Save Project As**:

**.....\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.

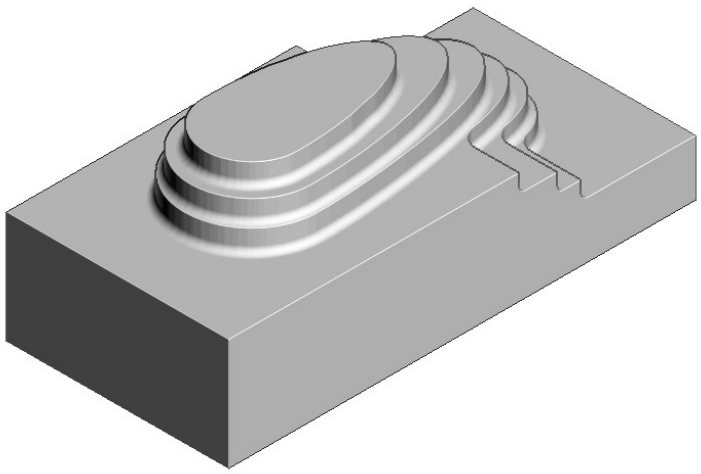


5 Select **Exit ViewMILL** to close the current session.

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

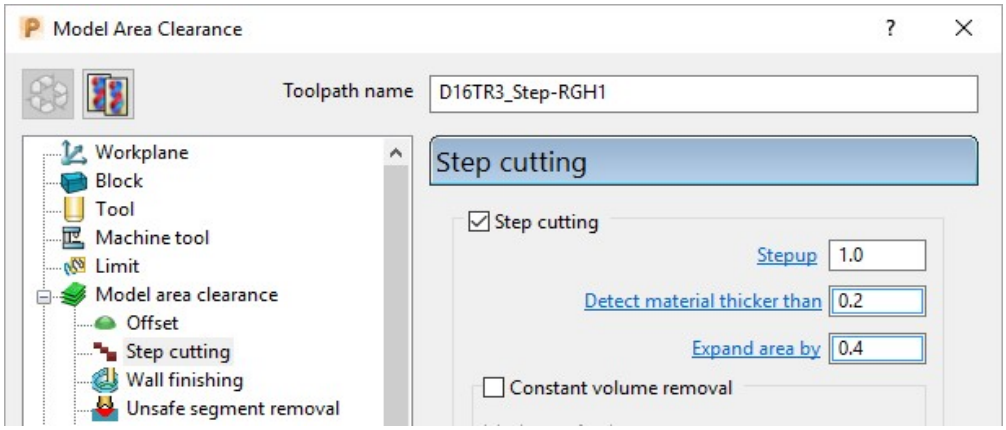
# Area Clearance using the Step option

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**.

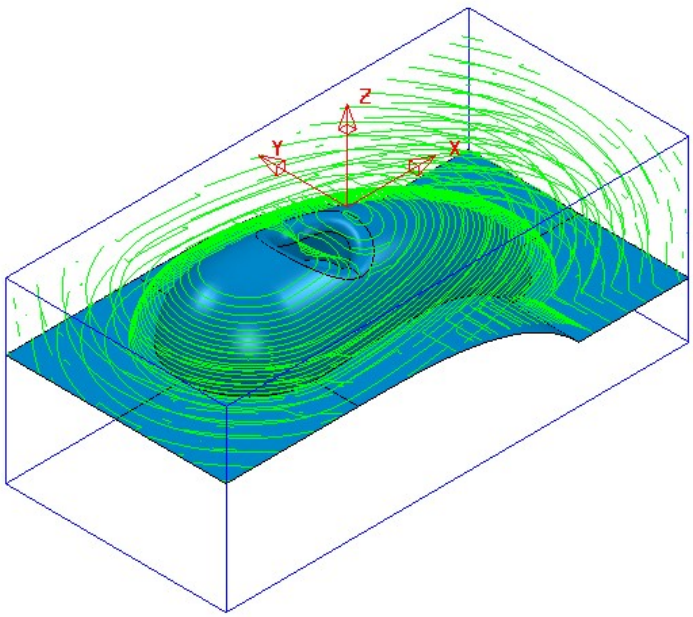



Copyright © Autodesk

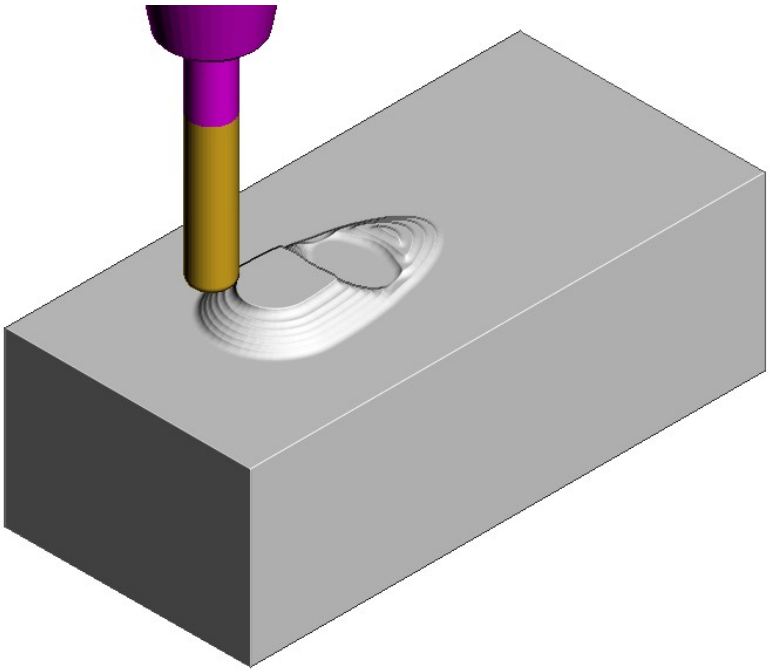
1. Activate the existing toolpath **D16TR3-RGH1** to restore the settings used.
2. Run a new **ViewMILL** simulation (as shown above) on the toolpath **D16TR3-RGH1**.
3. Select **Exit ViewMILL** to close the current session.
4. Right mouse click on the active toolpath **D16TR3-RGH1** and from the local 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.



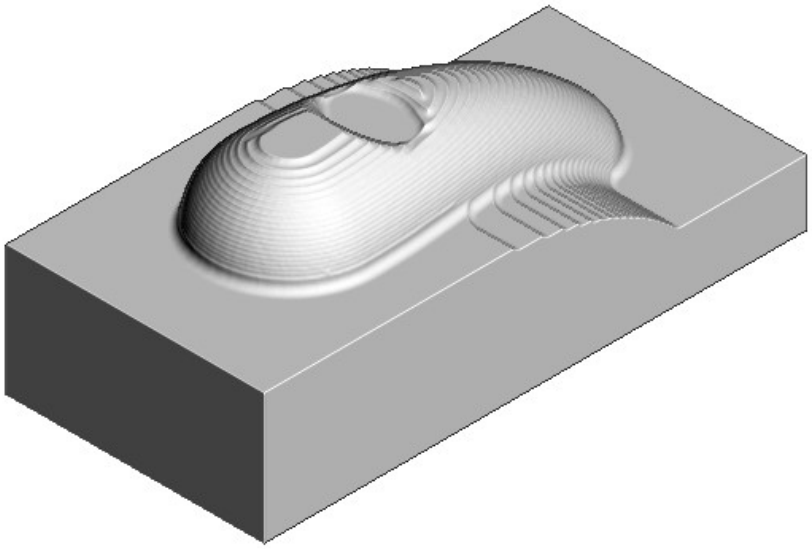

6. Select **Calculate** to create the toolpath **D16TR3_Step-RGH1**.




7. 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.

8 **Save** the **Project**.

9 From the **main** pull down menus select **File > Delete All**.

10 From the **main** pull down menus select **Tools > Reset Forms**.


Copyright © Autodesk

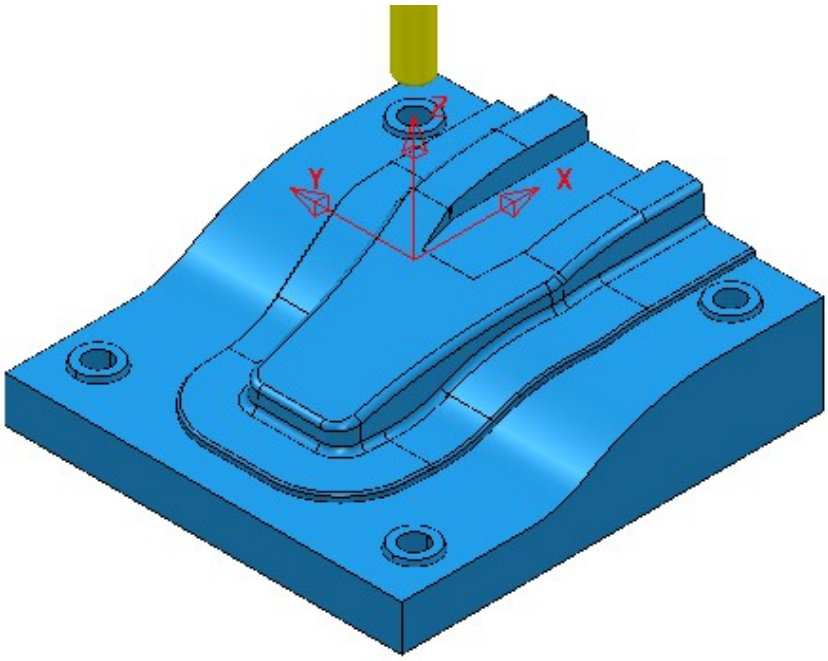
# Model Area Clearance – Vortex example

**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.

10 **Open** the (read-only) **Project**:-

**……\PowerMill_Data\Projects\Vortex2-Start**




11 **Save Project As**:-

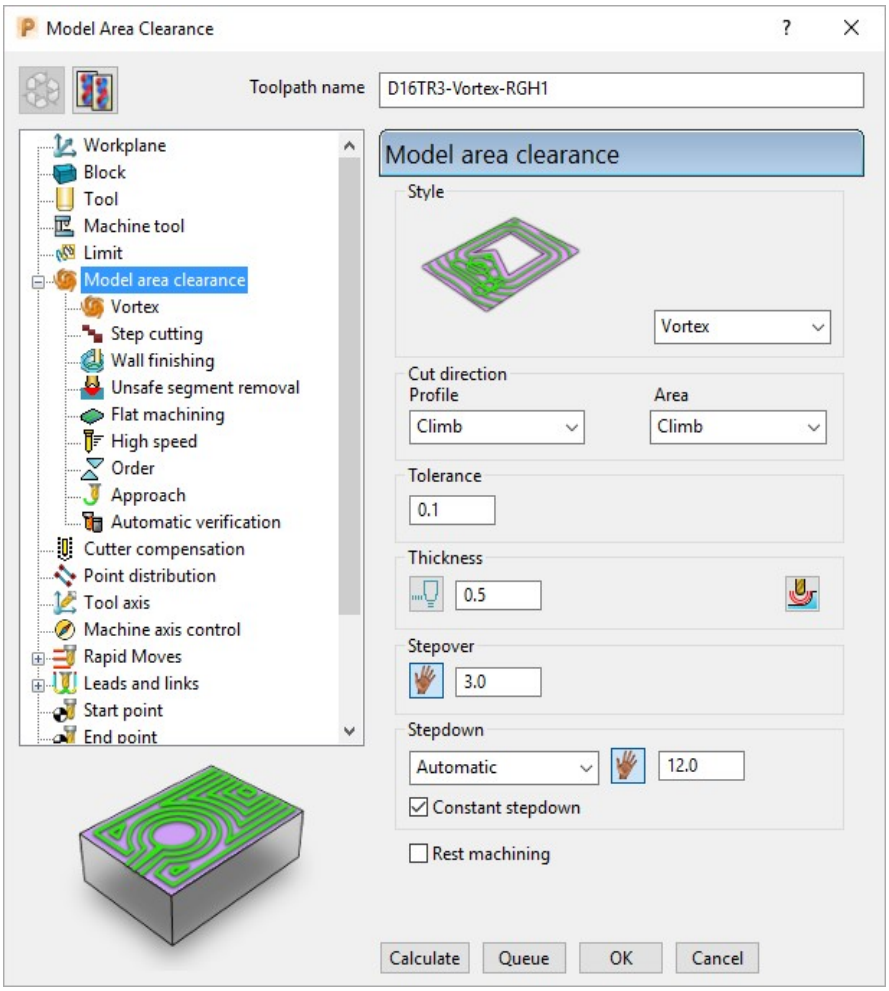
**……\COURSEWORK\PowerMill_Projects\Vortex2**

12 Activate the **workplane**, **TopCentre-datum**.

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

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

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




16 In the **Model area clearance** page of the strategy dialog:

g Enter Toolpath name – **D16TR3-Vortex-RGH1**

h Select **Style** – **Vortex**

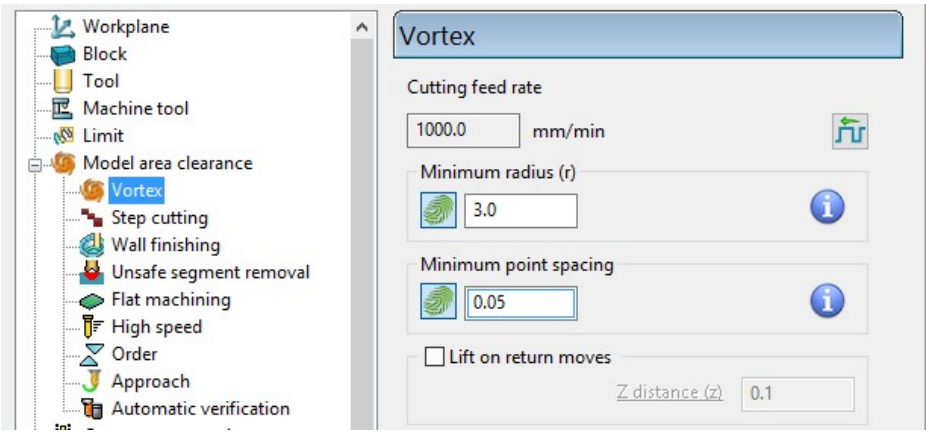
i Set ***Cut direction*** with both ***Profile*** and ***Area*** as **Climb**.

j Input **Thickness 0.5**

k Input **Stepover 3.0**

l Input **Stepdown 12.0**

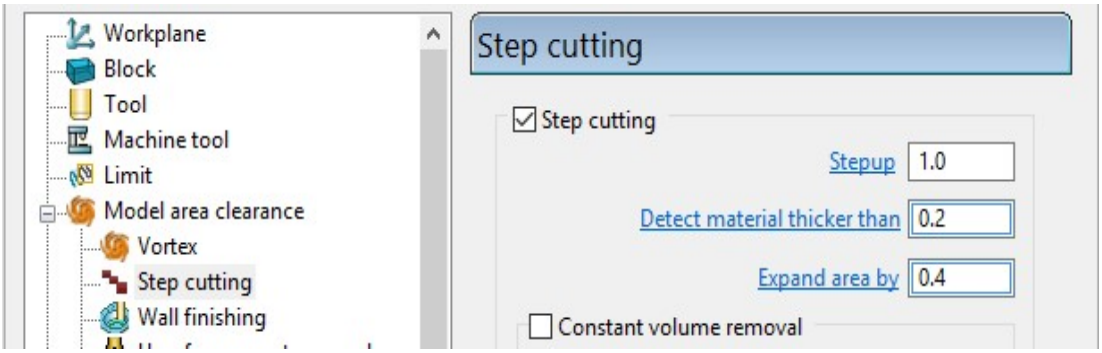
17 Open the **Vortex** page of the strategy dialog:




a Input **Minimum radius (r)** as **3.0**

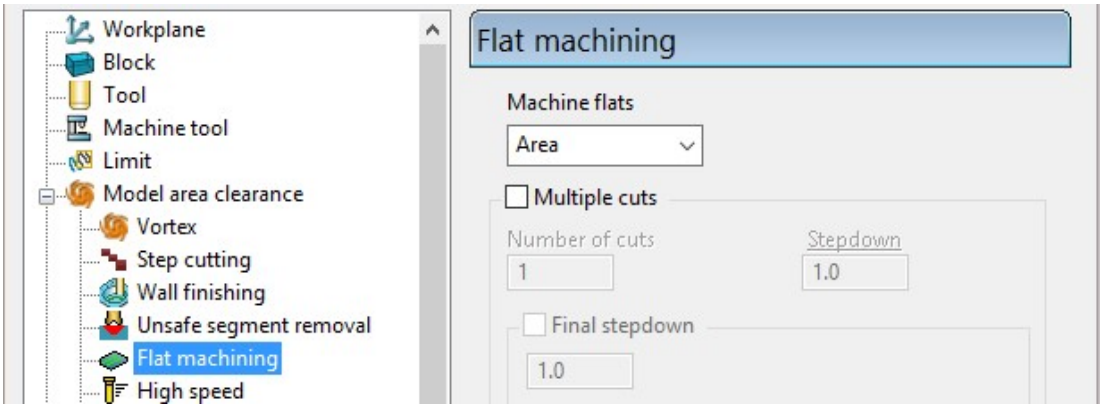

Copyright © Autodesk

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



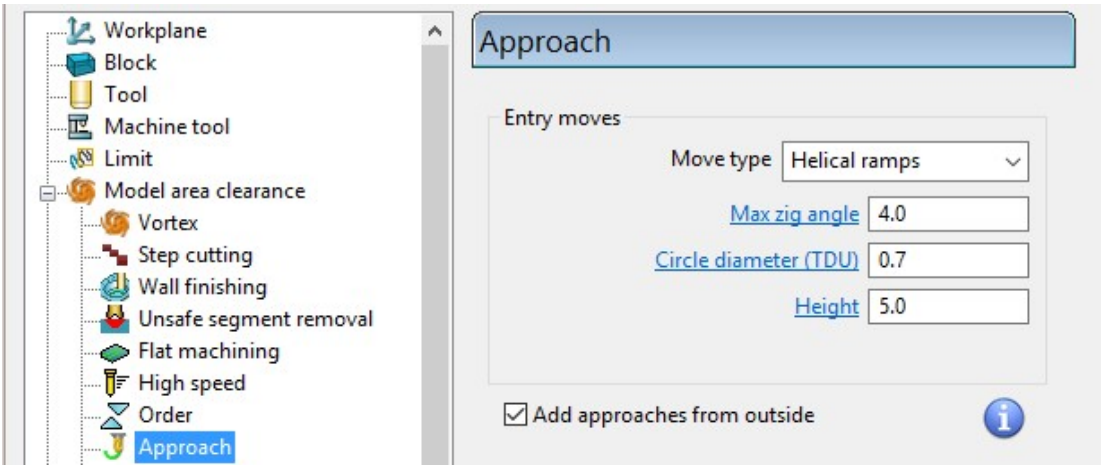

a **Tick** the **Step cutting** box and then input a **Stepup value** of **1.0**

19 Open the **Flat machining** page of the strategy dialog:




a Set **Machine flats** to **Level**.

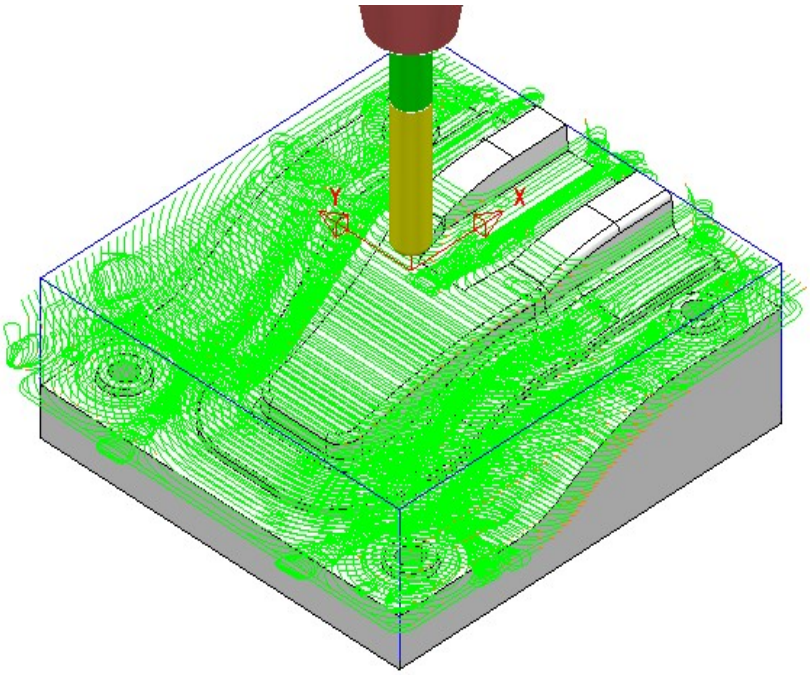
20 Open the **Approach** page of the strategy dialog:



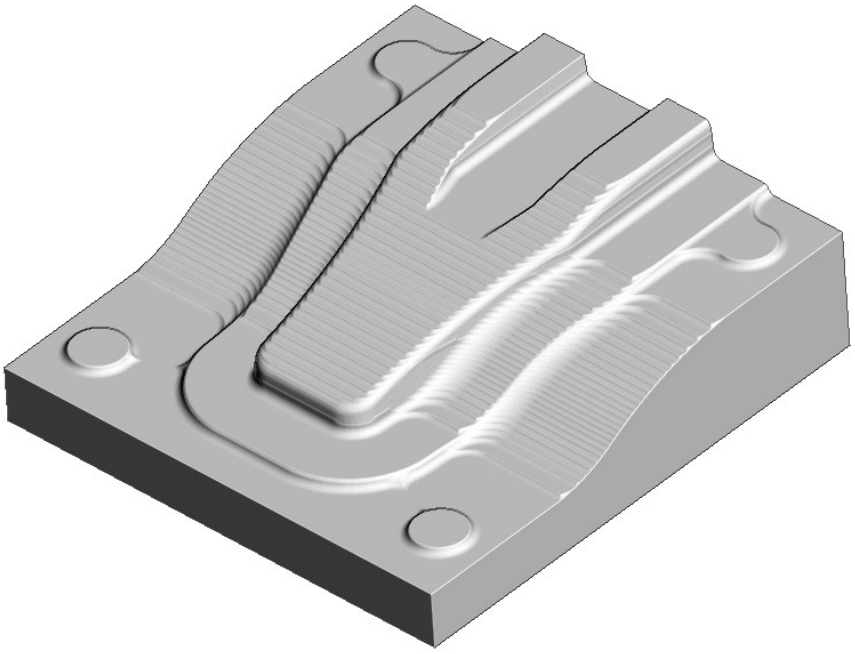

21 In the **Helical ramp** options window, select **Max zig angle 4** with a **Circle diameter 0.7** and **Height 5**.

22 At the bottom of the dialogue select **Calculate** to process the **Vortex** - **Model Area Clearance** toolpath.

Copyright © Autodesk



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



24 **Save** the **Project**.


Copyright © Autodesk

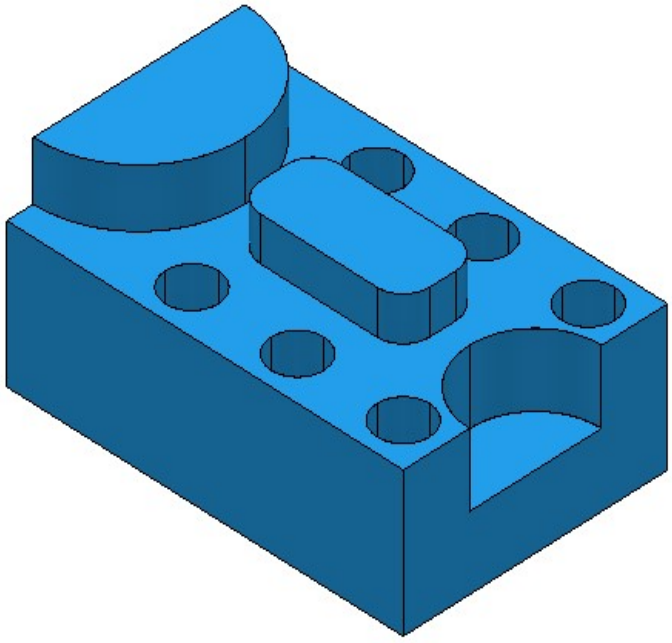
# Machining Flat features within Model Area Clearance

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**

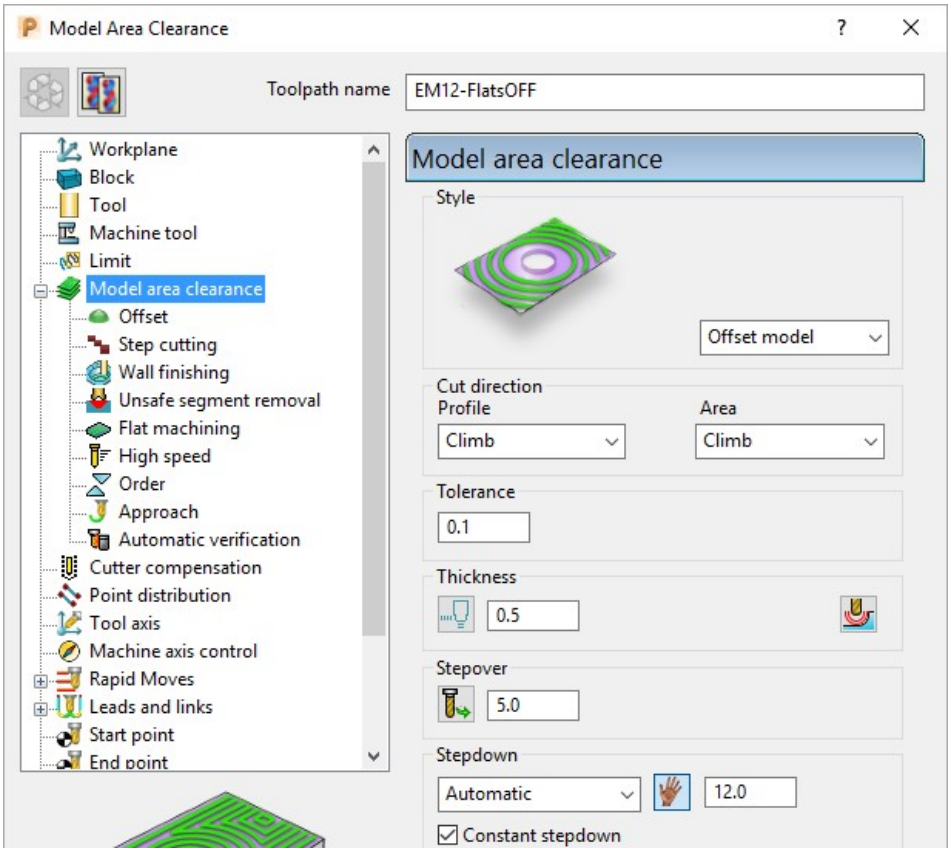
4 Create a 12mm diameter **End Mill** tool named **EM12**.

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

6 Calculate default **Rapid Move Heights**.

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

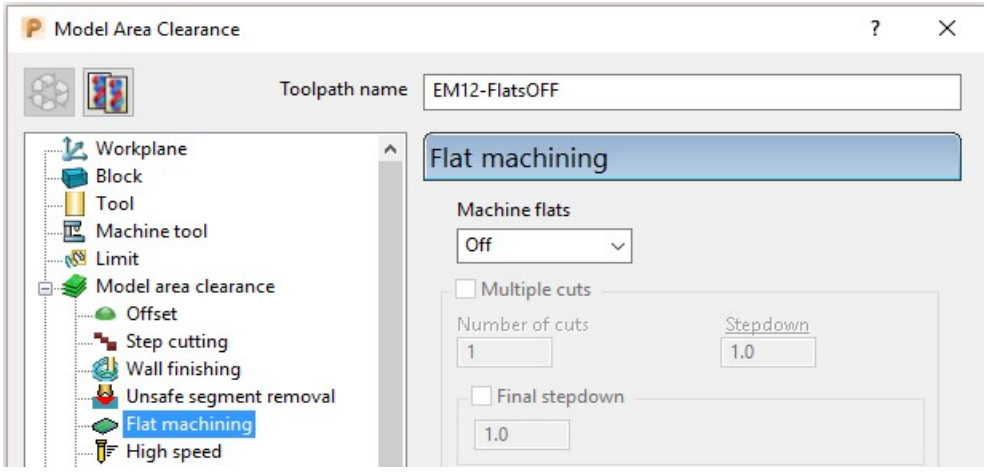
8 Select **Toolpath Strategies** , and from **3D Area Clearance** open a **Model Area Clearance strategy**.




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

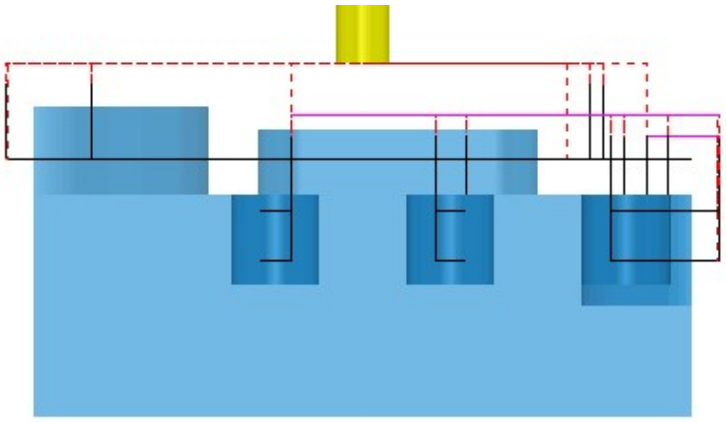
- a Enter the *Name* as **EM12-FlatsOFF**.
- b Set Thickness to **0.5**
- c Set Stepover to **5**
- d Set Stepdown to **12**

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




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


Copyright © Autodesk



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**.

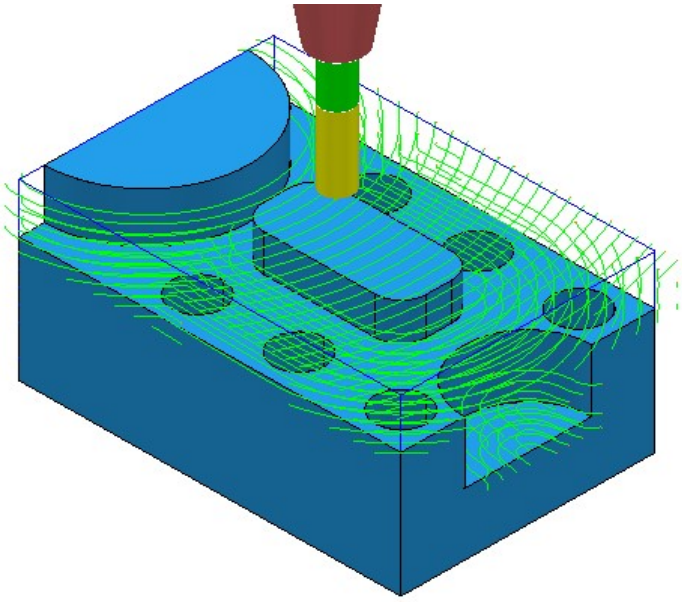
12 Right-click the active toolpath, **EM12-FlatsOFF** and select **Settings**.

13 Click to create a copy of the toolpath.

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

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

16 **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.

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

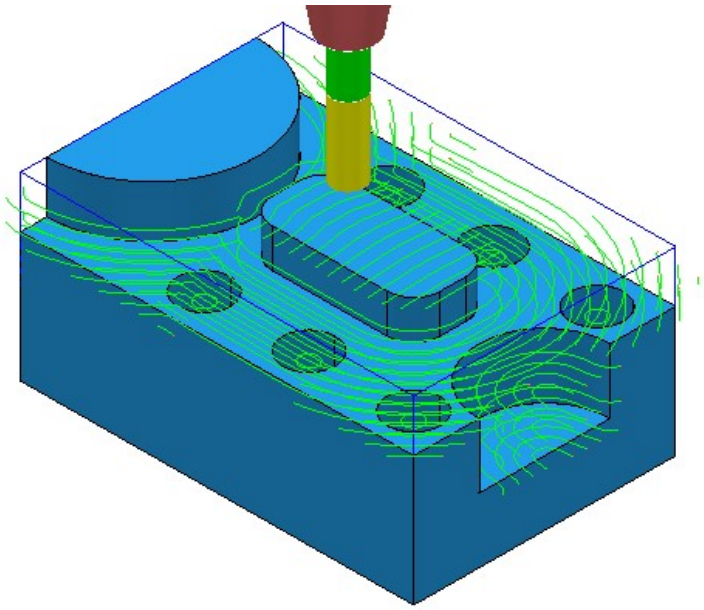
18 Click to create a copy of the toolpath.

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

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

Copyright © Autodesk

21 **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.

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

**.....\COURSEWORK\PowerMill-Projects\AreaClearFlats**

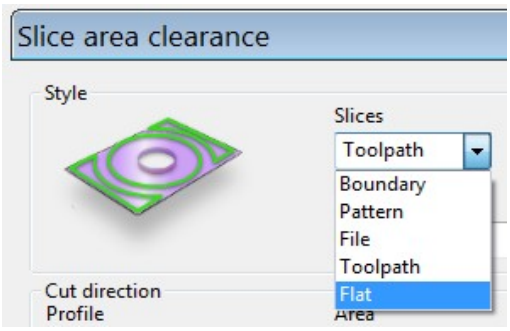
# Machining Flats separately using Slice Area Clearance

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:

a 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**.

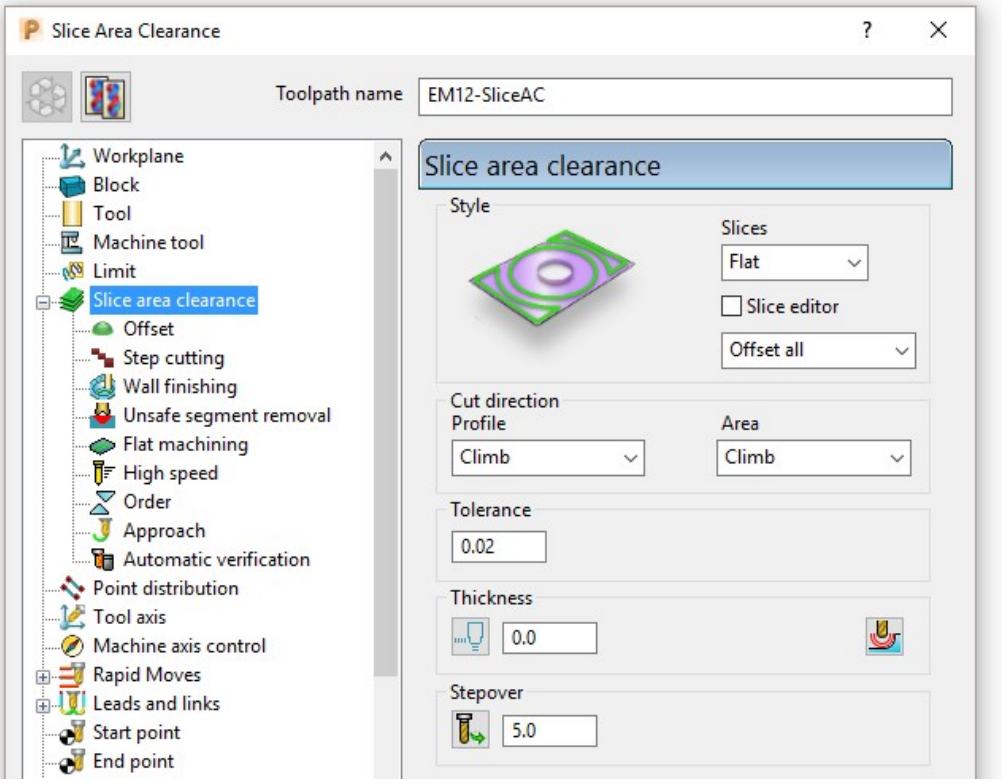
There are several types of 2D entity that are supported in the **Slices** options:-




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


Copyright © Autodesk

12 Select **Toolpath Strategies** , and from **3D Area Clearance** open a **Slice Area Clearance** strategy.




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

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

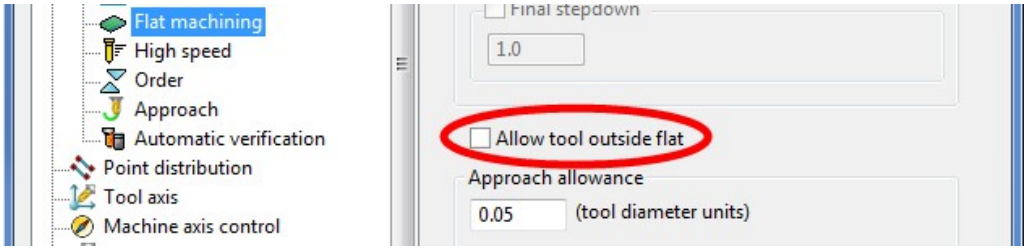
c Set ***Style*** as **Offset All**.

d Set **Tolerance 0.02**

e Set **Thickness 0**

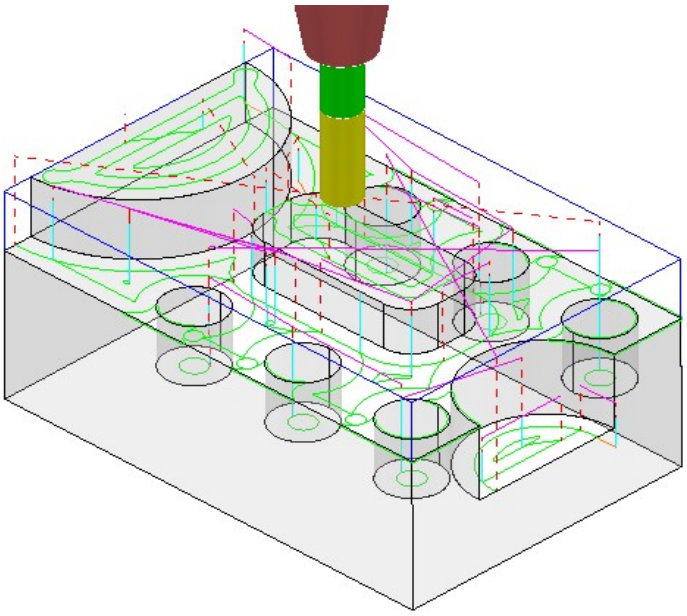
f Set **Stepover 5**

13 Select the **Flat machining** page (Available when ***Slices*** - **Flat** is active).




14 Check that the **Allow tool outside flat box** is unticked.

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



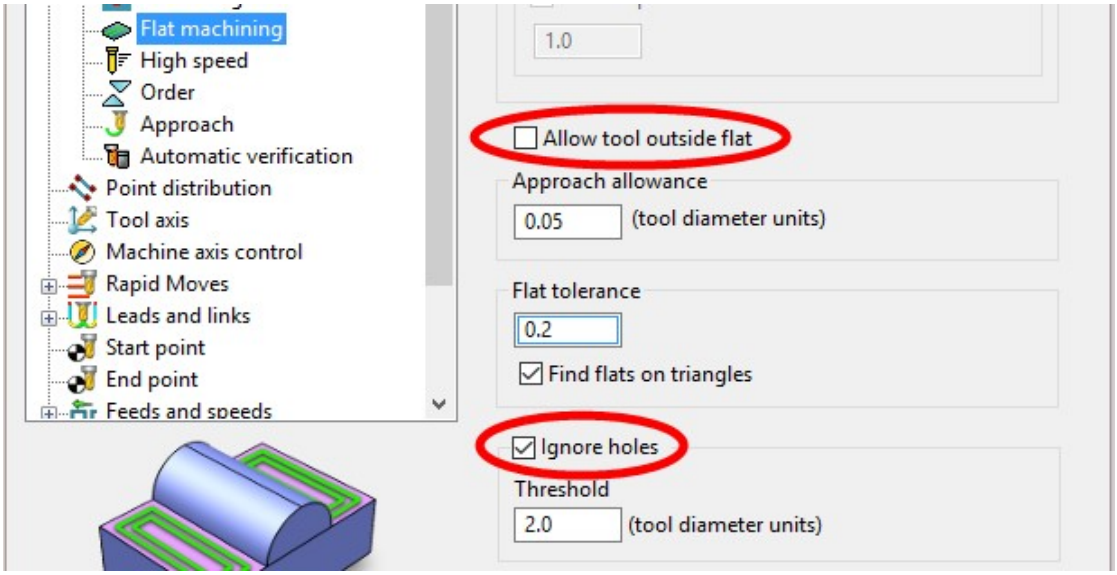
This **Slice Area Clearance** strategy is confined to the **Flat** areas only.

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.

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

24 Click on to create a copy of the toolpath.

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

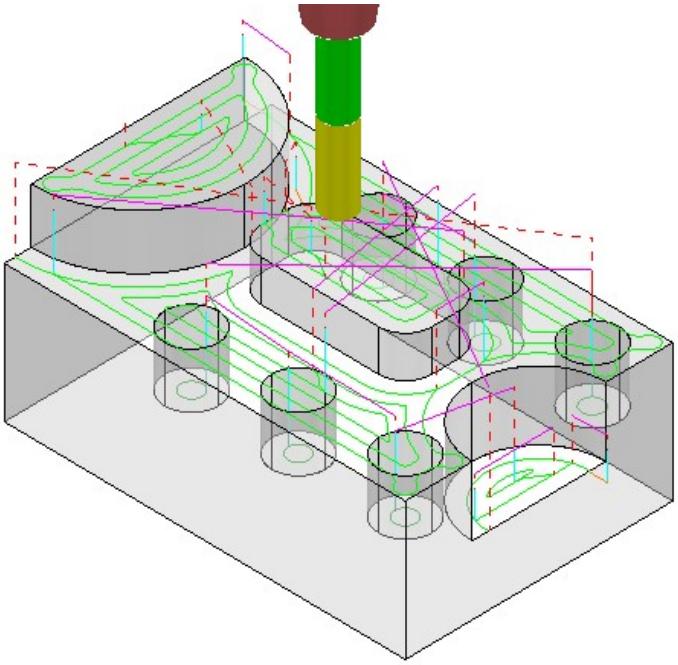



26 Select **Tick** in the **Ignore holes** box.

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.

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


Copyright © Autodesk



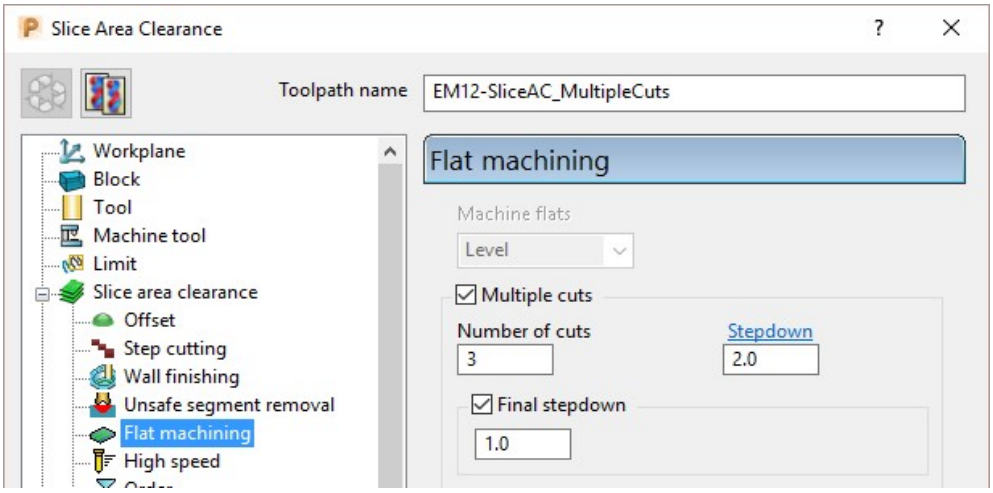
The cutter now passes across the top of the holes without lifting, producing a smoother, more efficient, toolpath.

*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.*

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

29 Click on to create a copy of the toolpath.

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

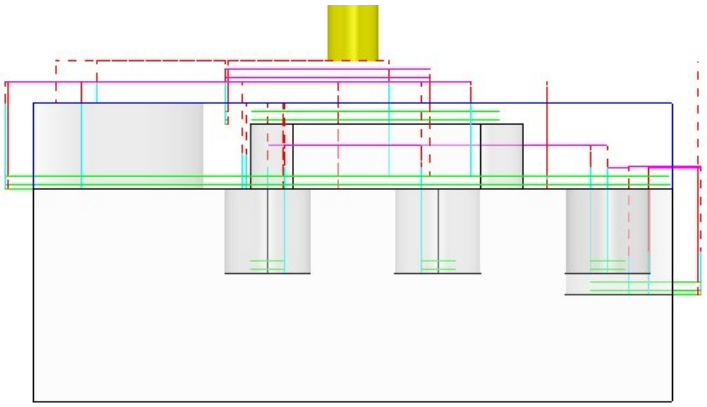



g Select the **Multiple cuts** option and set the **Number of cuts** as **3**.

h Input the **Stepdown** as **2.0**

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

31 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.*

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

**(.....\COURSEWORK\PowerMill-Projects\AreaClearFlats)**

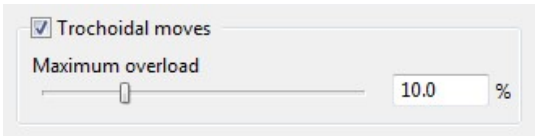

Copyright © Autodesk

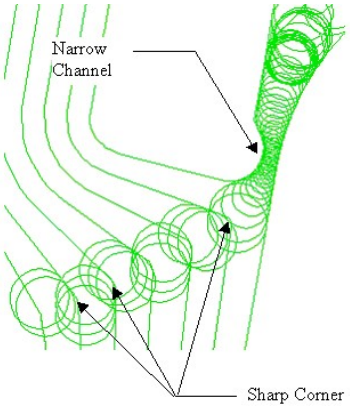
# Further information on Area Clearance Machining

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.

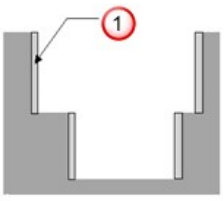


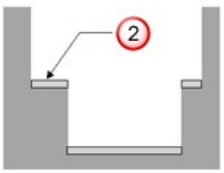




If the tool reaches a specified overload situation, **PowerMill** automatically inputs a trochoidal path to eliminate full width cuts. This will occur in corners, narrow channels, slots, etc. The degree of movement is controlled with the slider as a percentage value of the specified stepover.

## Thickness

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



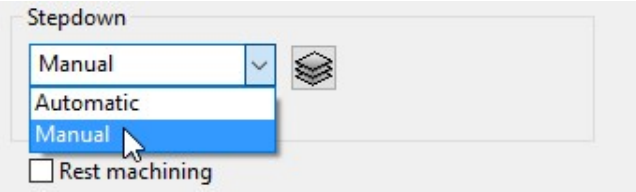




1 - Radial Thickness

2 - Axial Thickness

# Z Heights – Manual entry

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




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

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

- **Number** - divides the **Block** equally into the defined number of Z heights, the lowest of which will be at the base 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 after the Raster path.


Copyright © Autodesk

## 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).

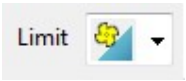
## Wall Finishing

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.

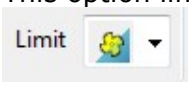
## Allow tool outside block

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**.




## 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.

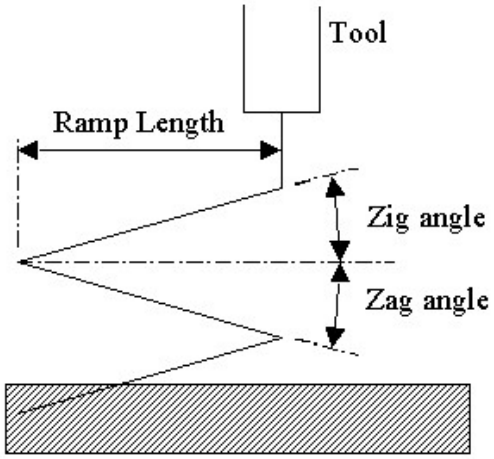
The nominal Stepdown value remains constant (independent of the Flat areas).

## 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.

Copyright © Autodesk




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 the **Ramping**.

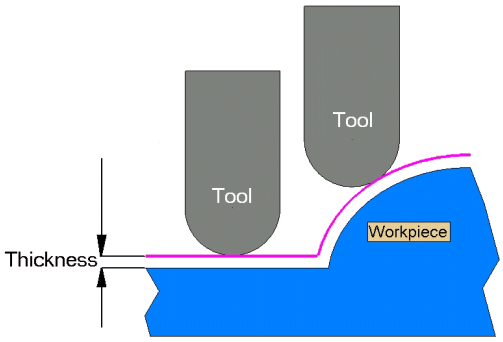
If the defined geometry for a **Ramp** move is such that it would cause a gouge, then the tool approaches using a default **Plunge** move.


Copyright © Autodesk

# 4. Finishing Strategies

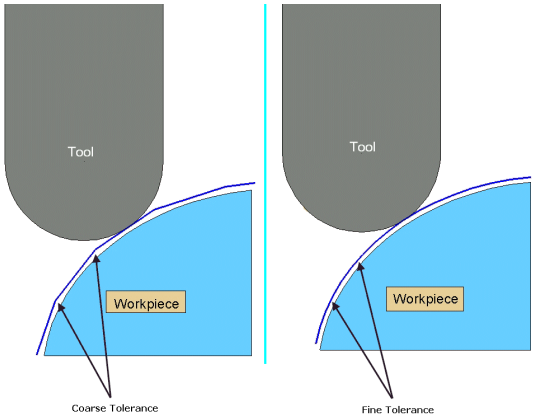
**Finishing strategies** machine the actual component form, and where applicable, follow on from the **Area Clearance** operation. Suitable values are required to control the accuracy and amount of excess material to be left on a component by a toolpath. The parameters used for this purpose are called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

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




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

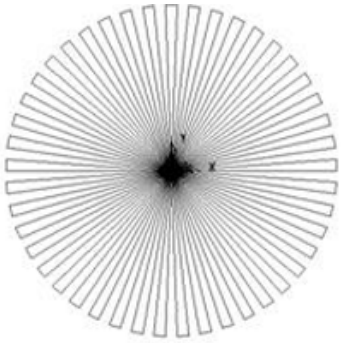
# Raster Finishing and Pattern Finishing

**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**.

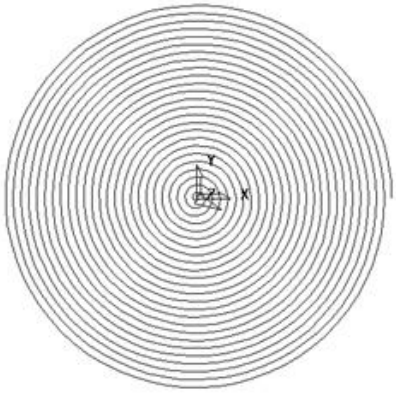
The **Pattern Finishing** strategy uses an existing, pre-defined **Pattern** or **Toolpath**, which is either projected down Z onto the model or used in situ as a **Drive Curve** to generate the toolpath.

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

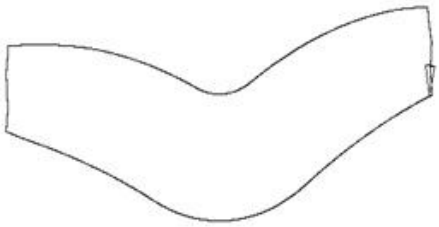
Raster



Radial



Spiral



User-defined

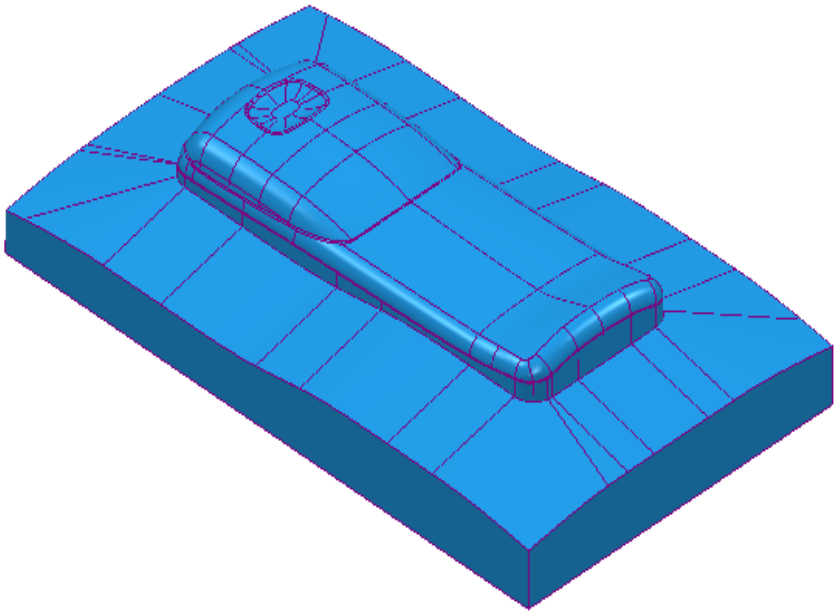
The **Raster finishing** strategy is used as an example in the next section.

## Raster Finishing

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

**…...\PowerMill_Data\Projects\Handset_Start**


Copyright © Autodesk



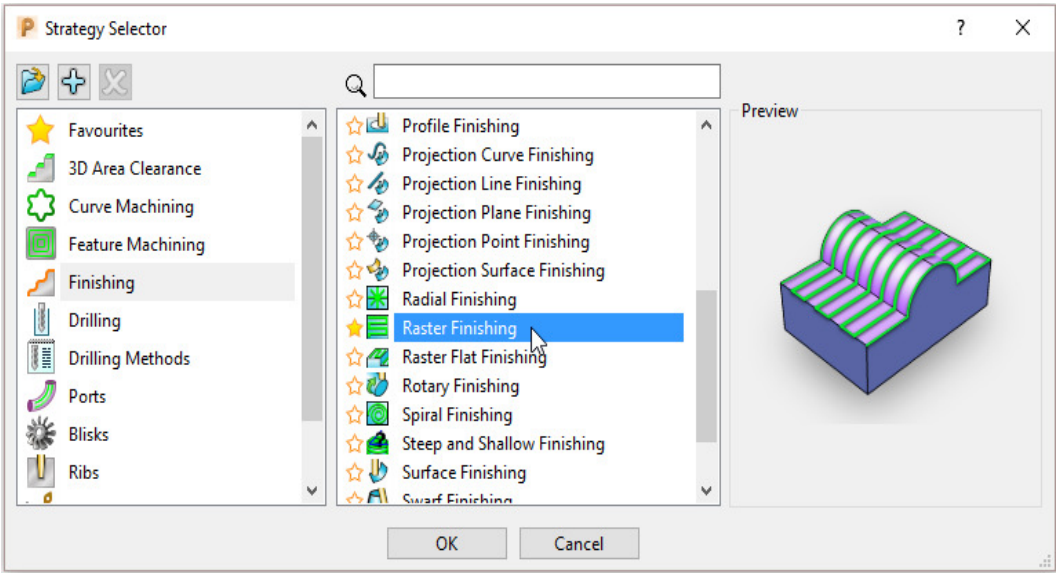
3 From the main menu , select **File** - **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\Handset_PatternMC**

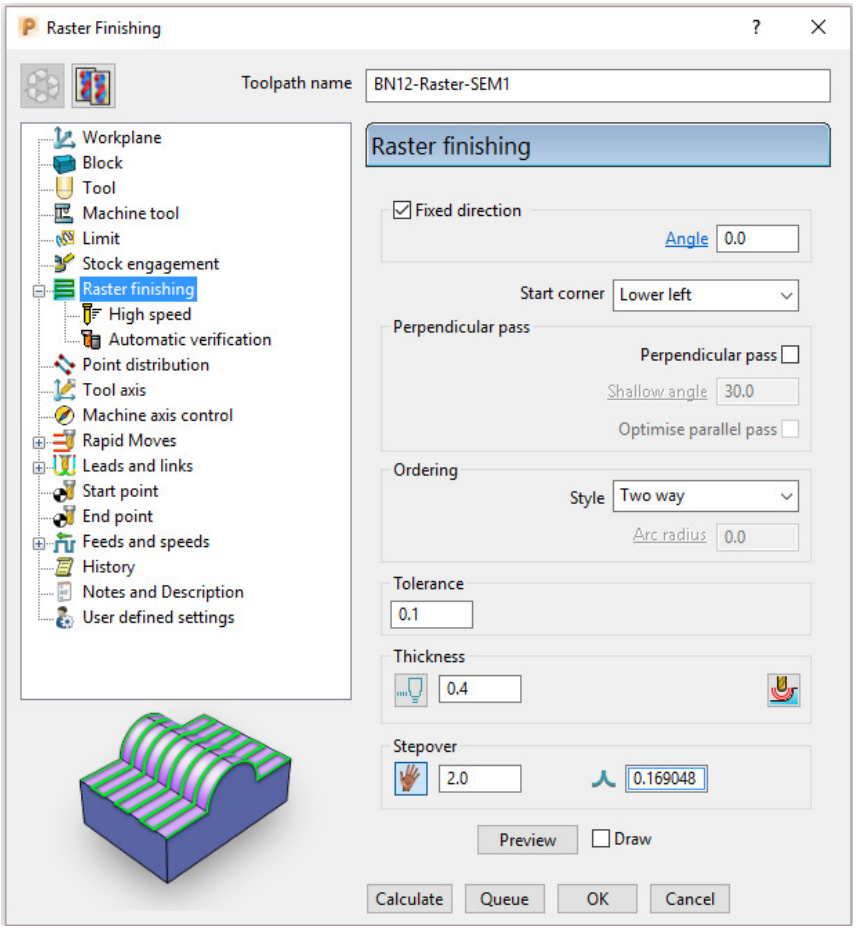
4 Activate the existing toolpath **D25TR3-RGH1** to restore all the **Block** and settings.

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.



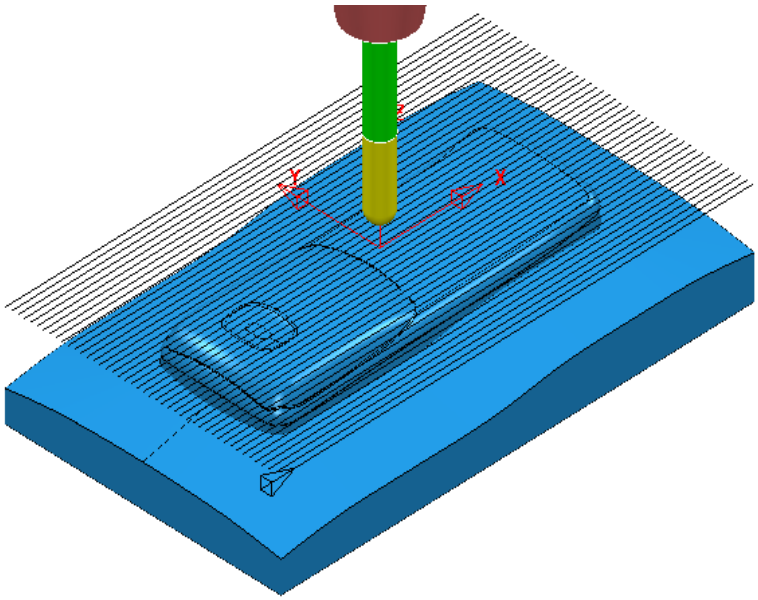

a Name the toolpath **BN12-Raster-SEM1**.

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).



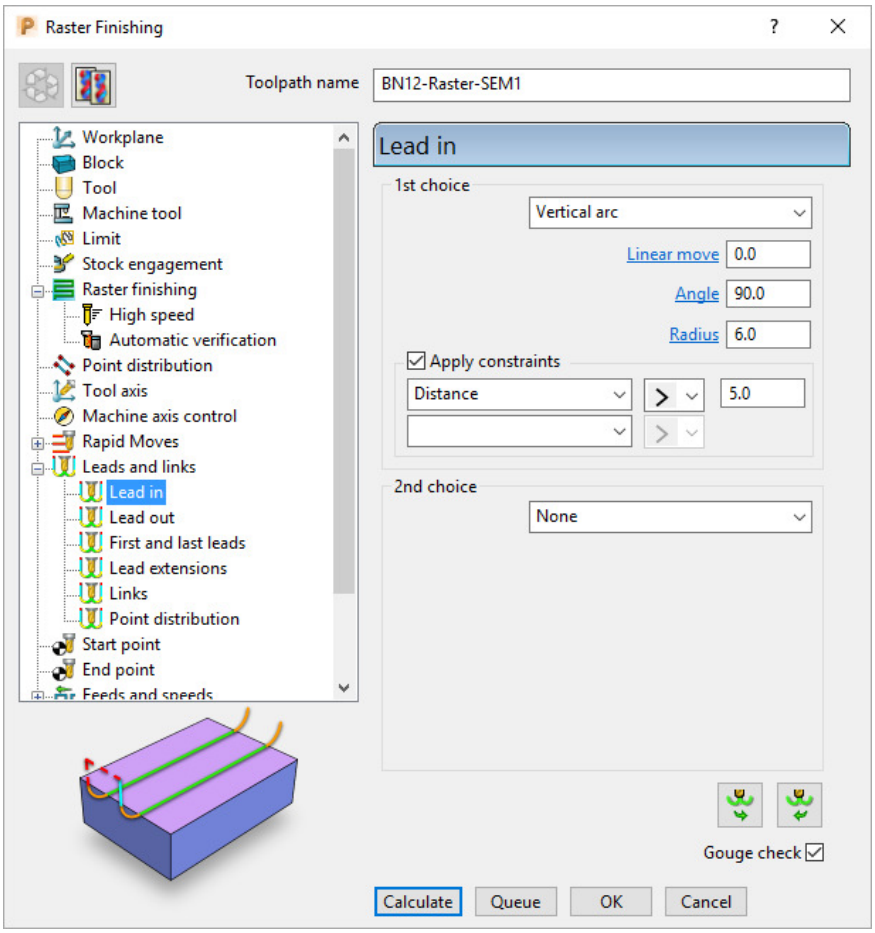

Copyright © Autodesk

Several options exist to add **lead and link** moves at the start and end of each individual tooltrack. The aim is to reduce unnecessary lifta and to provide a smoother transition of the tool 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**

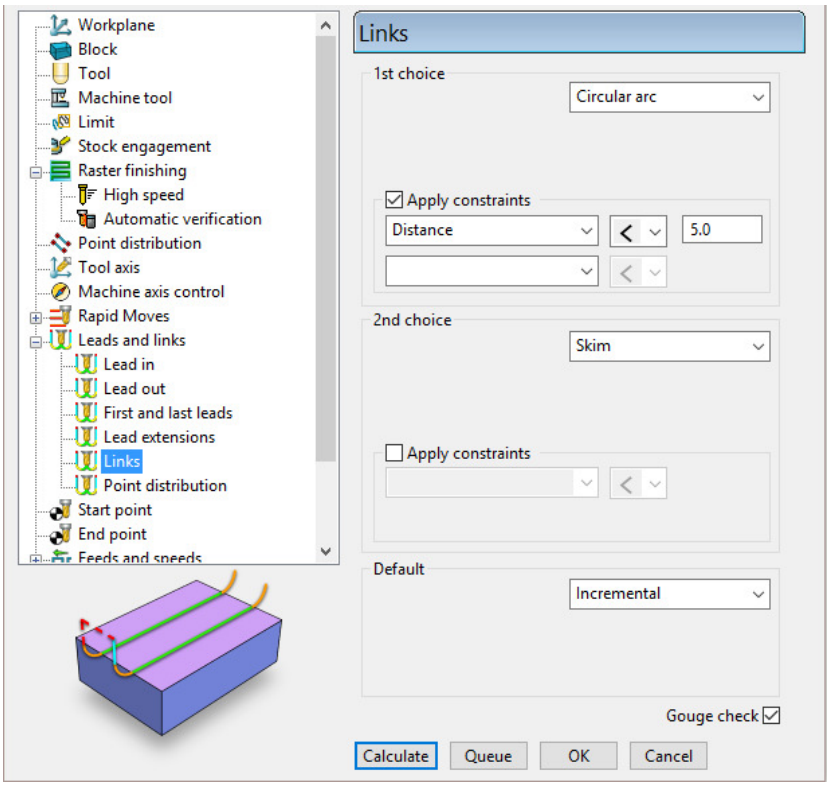
10 Tick the **Apply constraints** box and set the **Distance > 5.0** (Greater than).



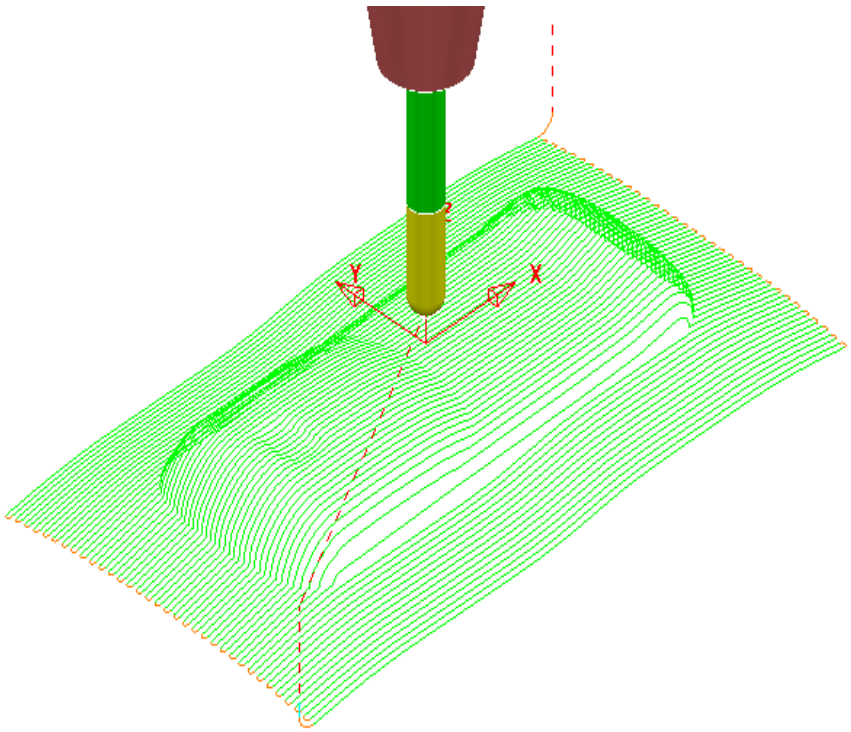

11 At the bottom right of the same dialog, select **Lead out the same as lead in**.

12 Select the **Links** page (shown on next page) and input the **1st choice** as **Circular arc**.

13 Tick the **Apply constraints** box and set the **Distance < 5.0** (less than).




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

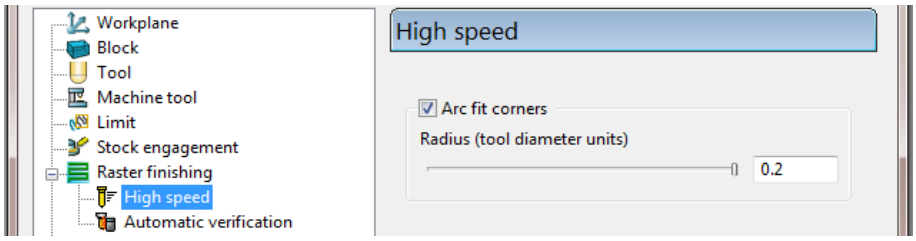



The above toolpath contains sharp corners along the vertical plane. These can be arc fitted within the **Raster Finishing** dialog using the **Arc fit corners** option.


Copyright © Autodesk

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

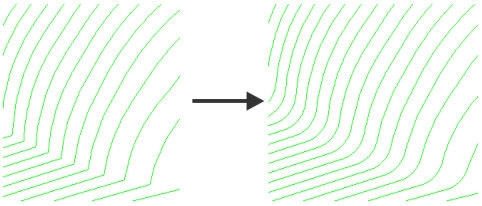
16 Click to enable the toolpath **BN12-Raster-SEM1** to be edited and open the **High speed** page.




a Select the **Arc fit corners** option and edit the arc radius to the maximum value **0.2**

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

17 Zoom into the toolpath to view the arc fitted tooltracks:



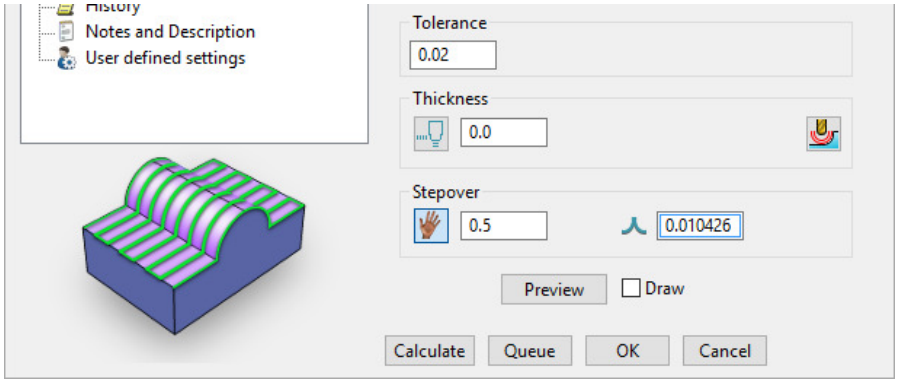
The value **0.2** (**TDU**), multiplied by the tool diameter (**12**) produces a radius of **2.4**

18 In **PowerMill *explorer***, right-click on the toolpath **BN12-Raster-SEM1** and select **Settings**.

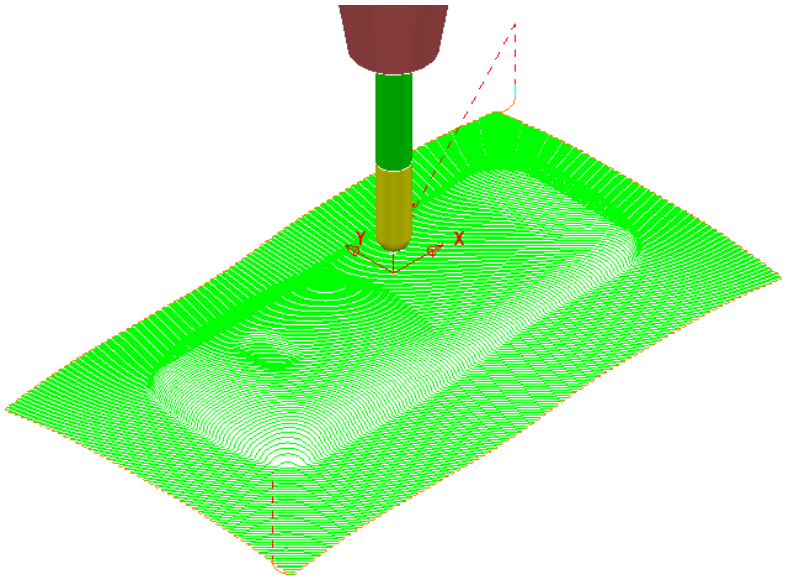
19 Click to create a copy of the toolpath and rename it as **BN12-Raster-FIN1**.

20 Change the raster finishing **Angle** from 0 to **135**.

21 input more suitable finishing values as shown in the following image.



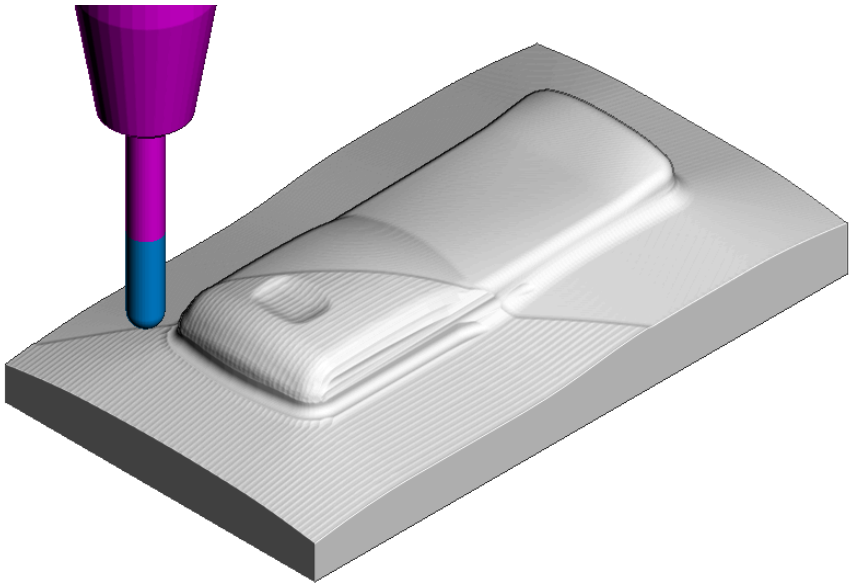

22 Click **Calculate** to process the toolpath and then **Close** the dialog.



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.

23 Run a full **ViewMILL** simulation on all 3 existing toolpaths.



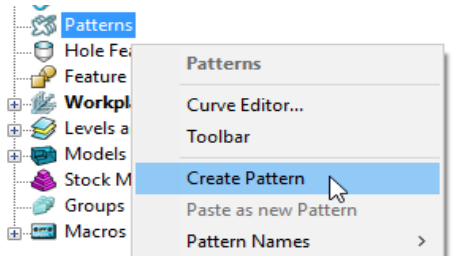
24 Disconnect **ViewMILL** from the simulation process (Change the sphere from green to red) and then select **No Image** to return to the **PowerMill** view.


Copyright © Autodesk

# 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.

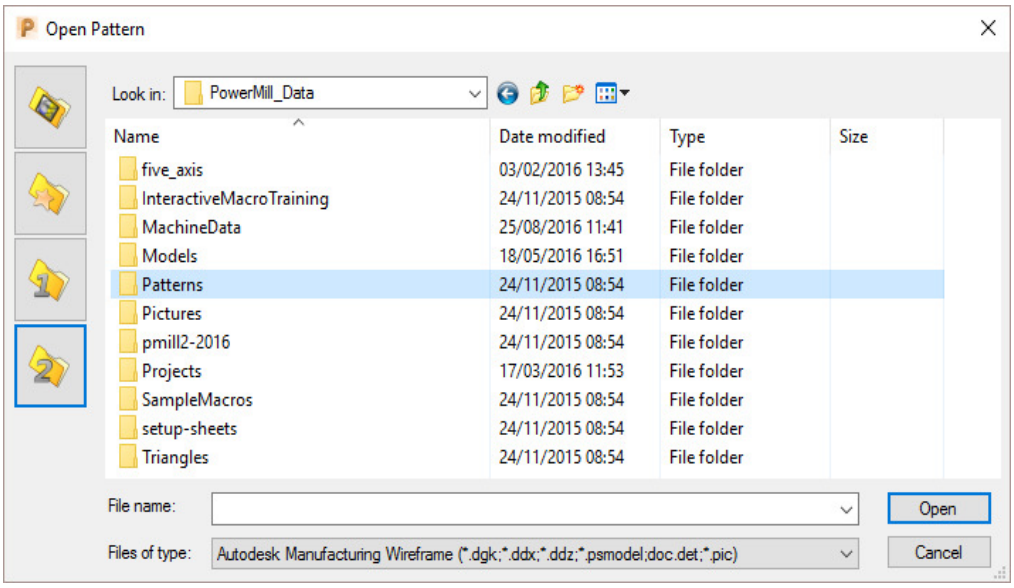
25 In the **PowerMill** ***explorer***, right-click **Patterns** and select **Create Pattern**.



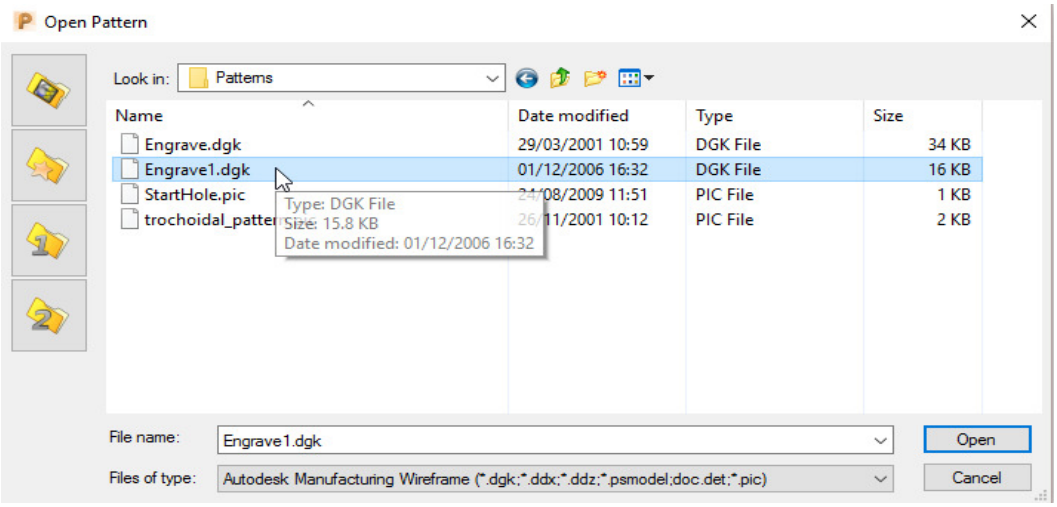

A new, empty **Pattern** is created.

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

27 Click button 2 on the lower left of the ***Open Pattern*** dialog:

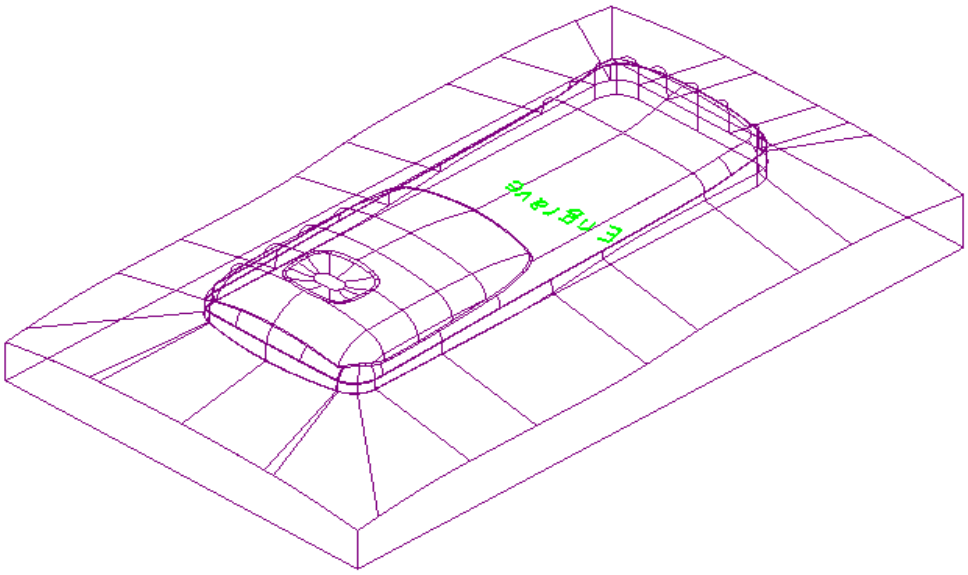



28 Double click the **Patterns** folder to access it and then select the file **Engrave1.dgk**.




29 Click **Open**.

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

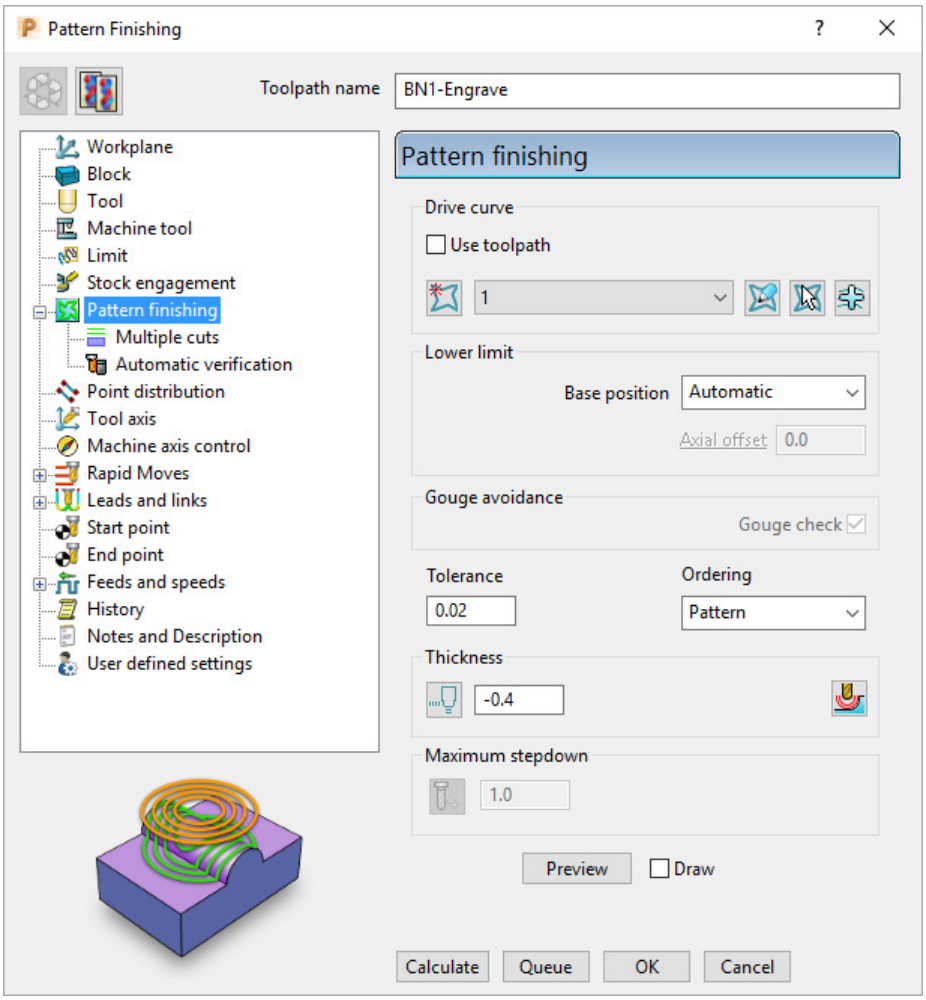


Next this 2D geometry will be projected along Z onto the 3D model to create a single **Pattern Finishing** toolpath with a negative Thickness value.

30 Activate the tool **BN1**.

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


Copyright © Autodesk




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

b Input **Pattern 1**.

c 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.

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

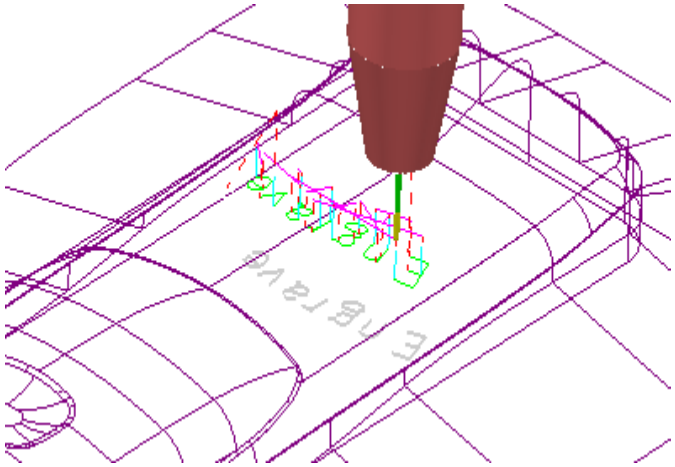
32 Select the **Lead in** page and select *1st choice* as **None**.

33 Select the option **Lead out the same as lead in**.

34 Select the **Links** page set both ***1st choice*** and ***2nd choice*** to **Skim**, and ***Default*** to **Incremental**.

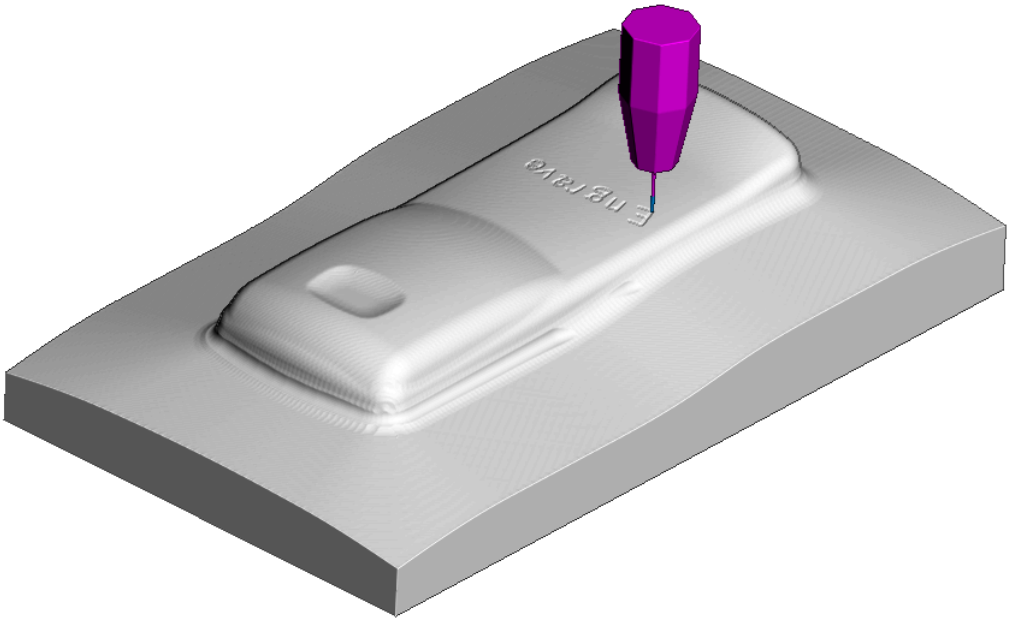
35 Select **Calculate** to process the toolpath then **Close t**he dialog.

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

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



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

**.....\COURSEWORK\PowerMill-Projects\Handset_PatternMC**


Copyright © Autodesk

# 3D Offset and Constant Z Finishing

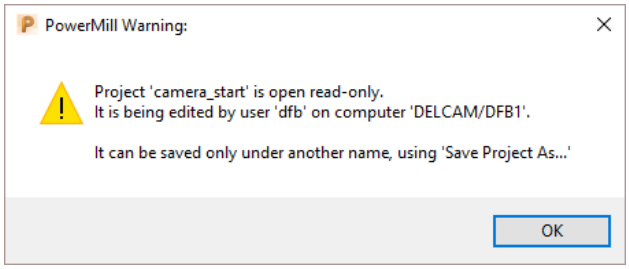
In this section, **3D Offset** and **Constant Z** finishing strategies are applied to a model that consists of a combination of flat and steep areas plus a pocket with vertical walls. In this example a **Boundary** will be created and used to limit the two strategies to where they are most effective.

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

39 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.

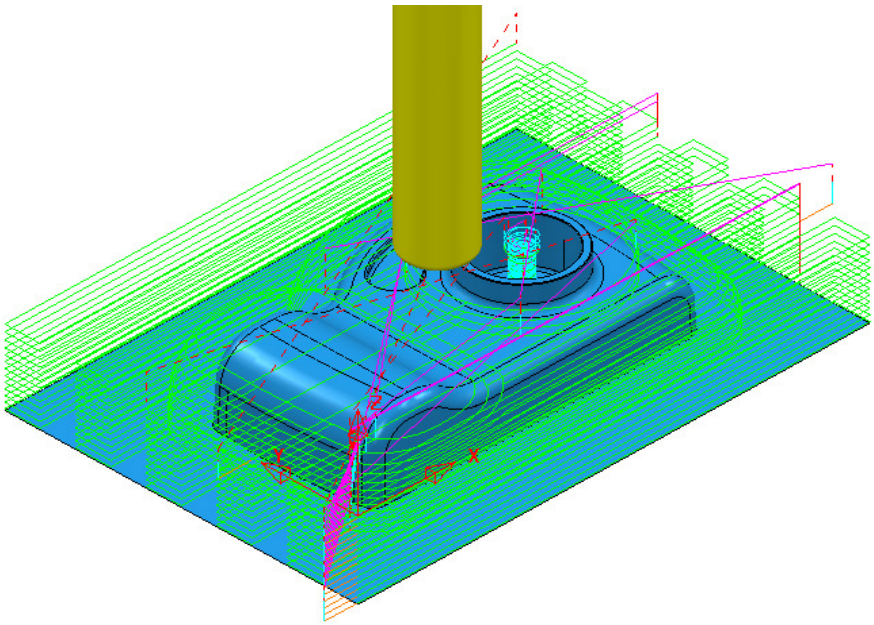



40 Select **OK** to continue loading the Project into the **PowerMill** session.

41 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.



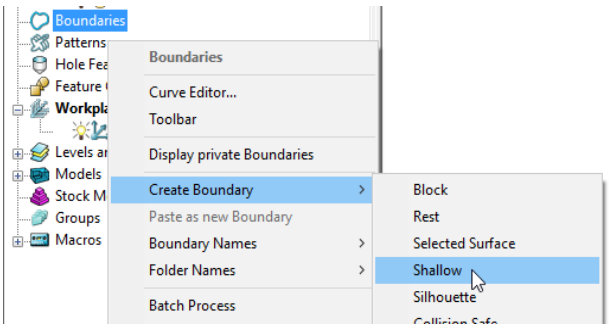
# Creation of a Shallow Boundary to define specific machining areas

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.

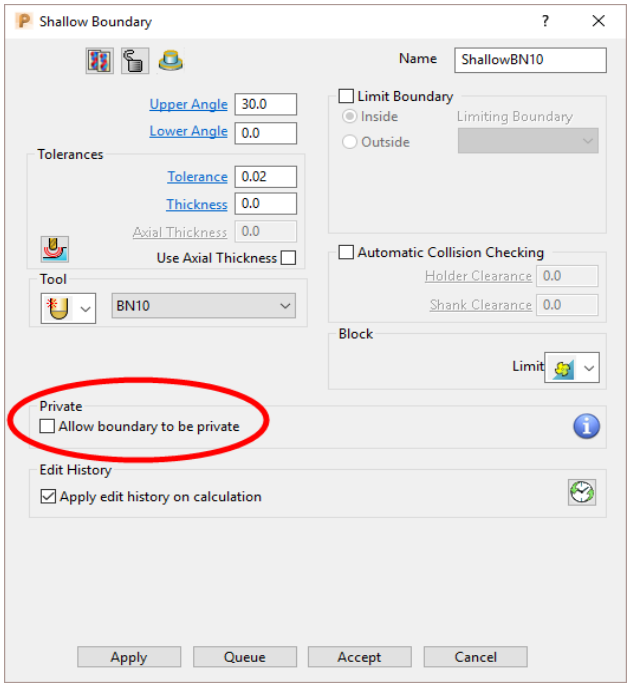
*There are several other types of **Boundary** options available to suit various applications and these are covered later in Chapter 6.*

In the following 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. Activate the existing toolpath **RoughOP1** to restore the original settings.
2. Define a **Ball Nose** tool of diameter **10** named **BN10**.
3. In the **PowerMill *explorer***, right-click **Boundaries** and select **Create Boundary** - **Shallow**.




4. In the displayed **Shallow Boundary** dialog:





Copyright © Autodesk

a Enter the Name as **ShallowBN10**

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

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

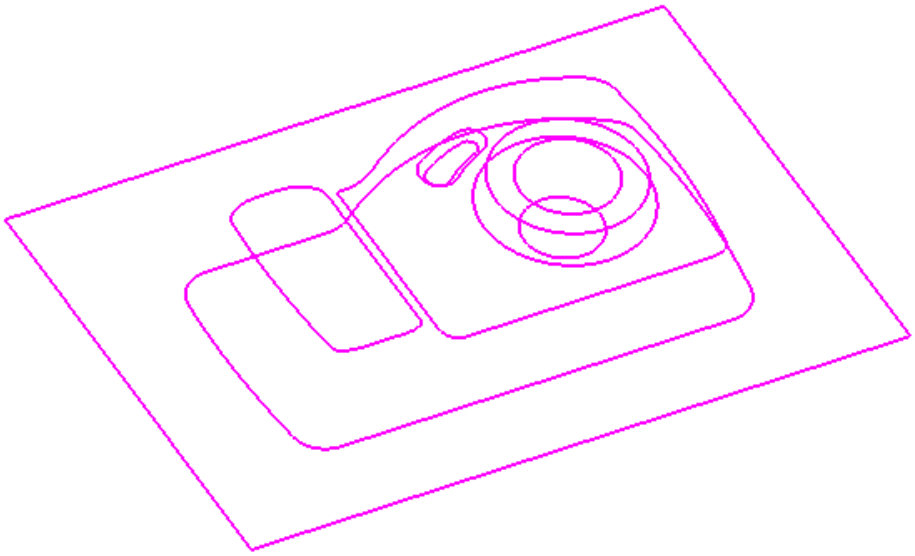
d Enter **Thickness** as **0**

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:

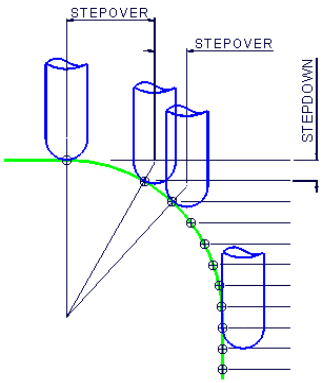


This **boundary** is made up of numerous segments; each one dividing the model into **steep** and **shallow** areas. Any of these segments can be selected and individually deleted at any time, except when it is assigned to a toolpath and locked.

Copyright © Autodesk

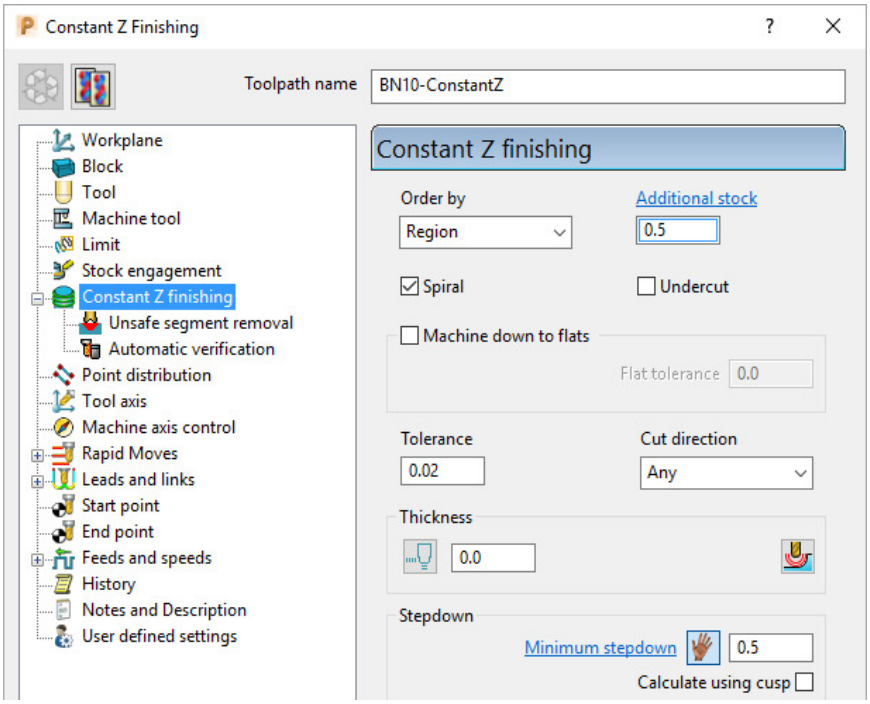
# Constant Z Finishing

**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.

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

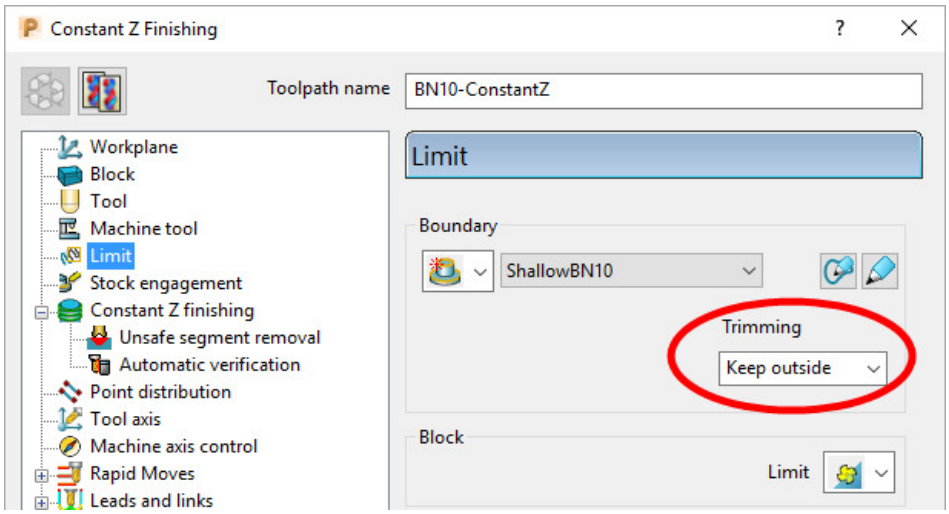



6 In the **Constant Z Finishing** dialog:

- **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**

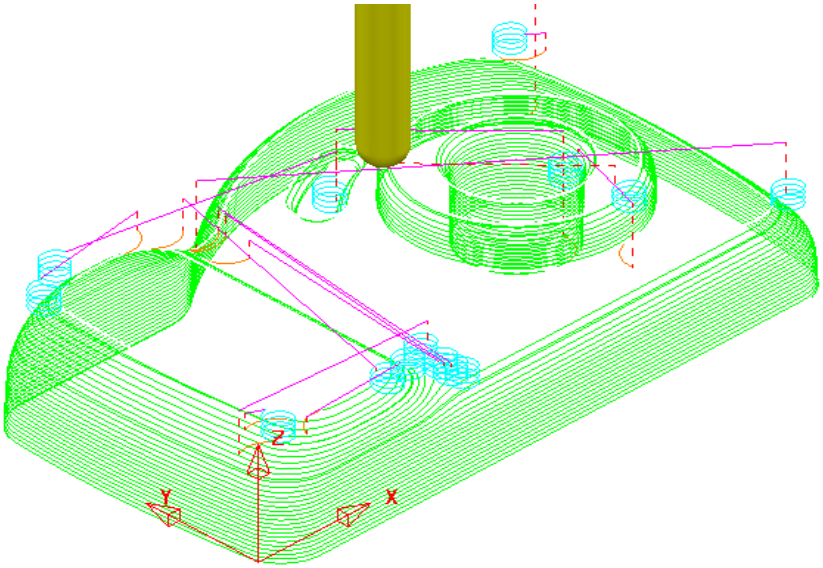

Copyright © Autodesk

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




8 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.

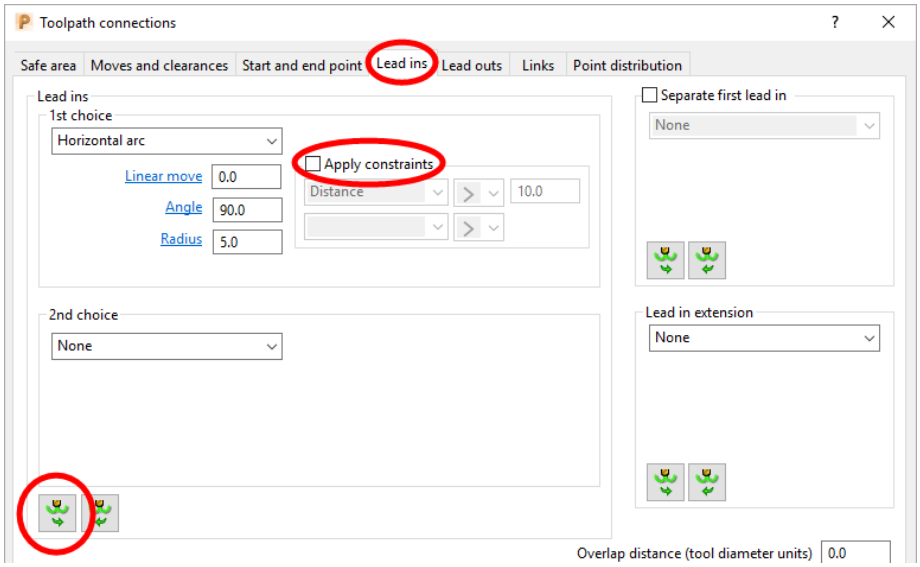



# Introduction to Leads and Links

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.

Copyright © Autodesk

9 Click the **Leads and Links** button on the **main** toolbar.



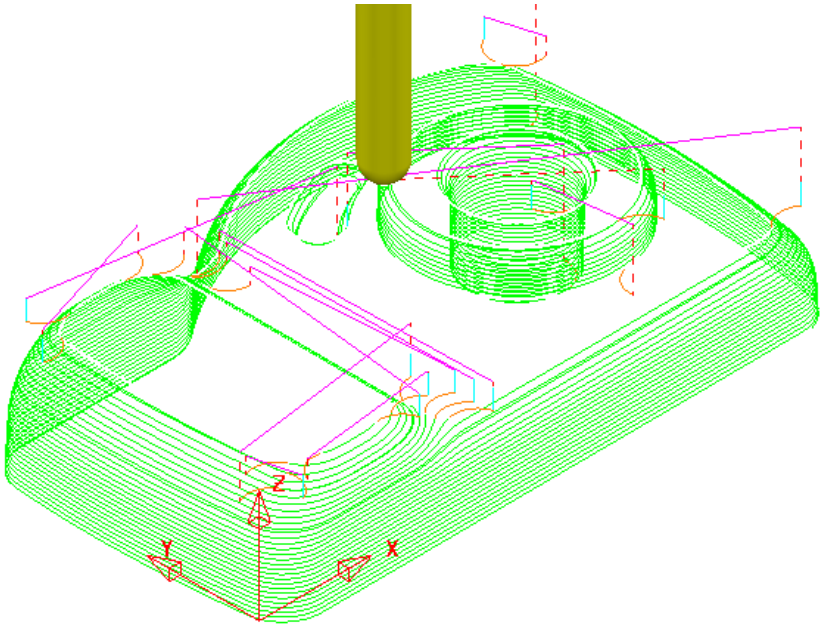

10 Select the **Lead In** tab in the above dialog and:

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

b Un-tick the **Apply constraints** box.

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

11 **Apply** and **Accept** the dialog.



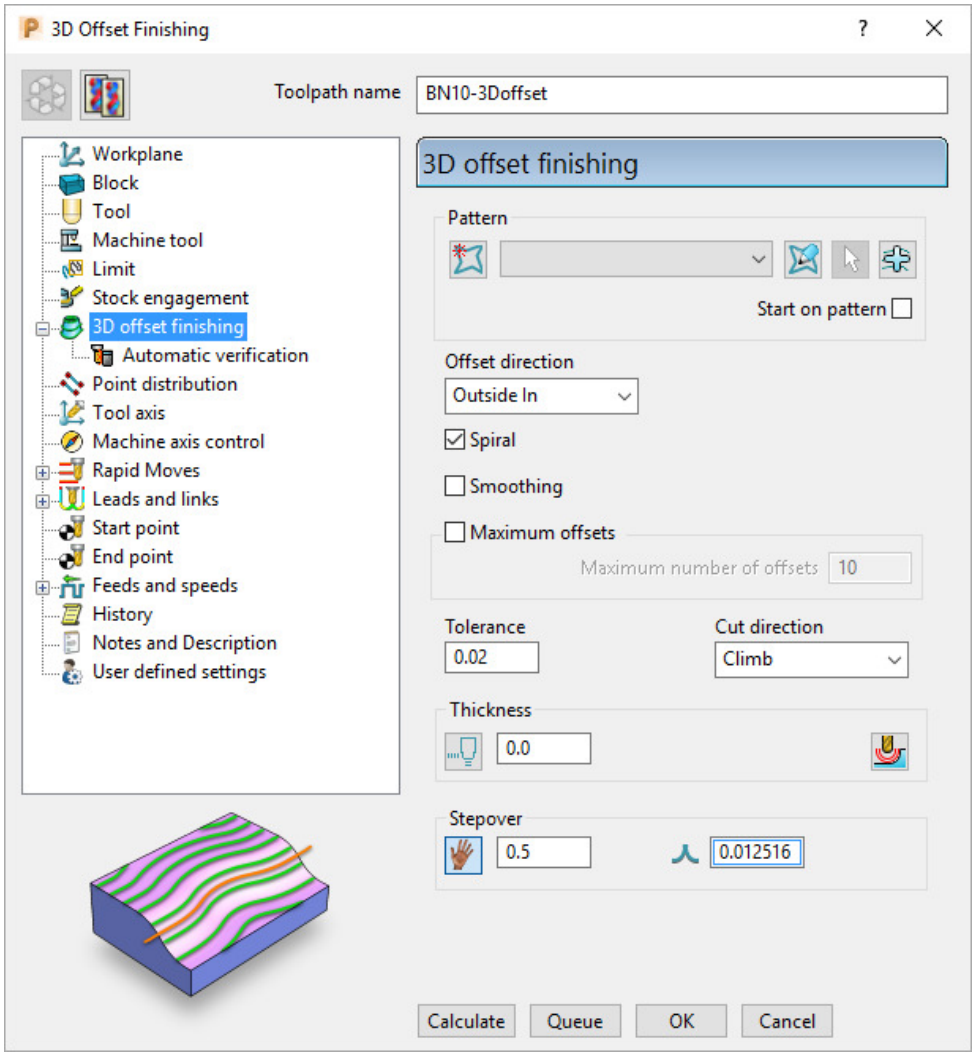

The tool now **leads in** and **out** of the active toolpath with a **Horizontal arc** move.


Copyright © Autodesk

# 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.

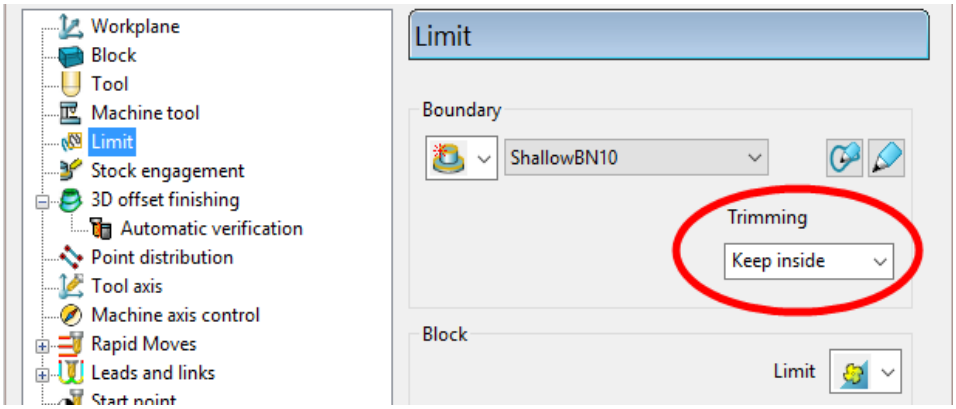
1 From the main toolbar, select **Toolpath Strategies** and from **Finishing** open a **3D Offset Finishing** strategy.




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**

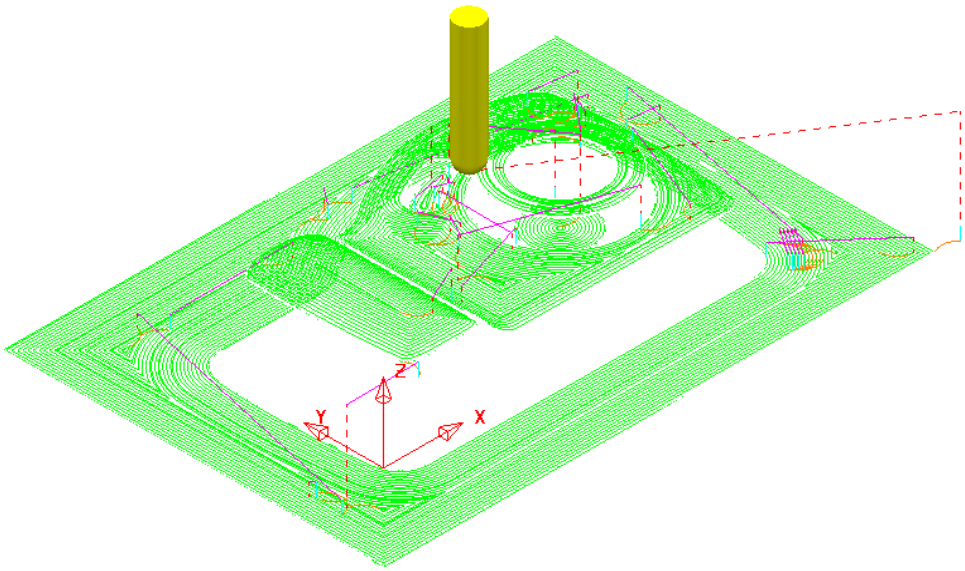
3 On the **3D Offset Finishing** dialog, select the **Limit** page.




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 button from the top of the screen.

7 Select the Lead in tab and:

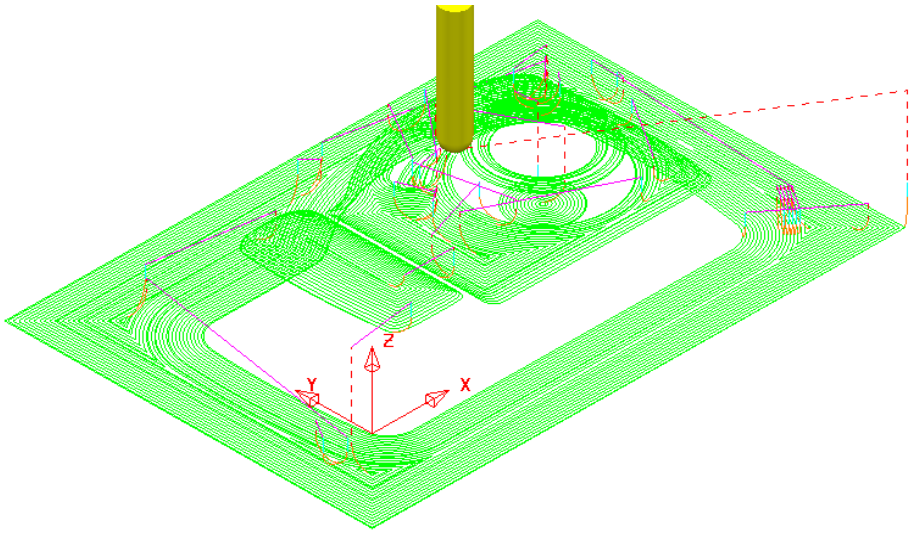
a Un-tick the **Apply** constraints box and change the **1st Choice** to **Vertical arc**, with ***Angle*** **90** and ***Radius*** **5**.

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

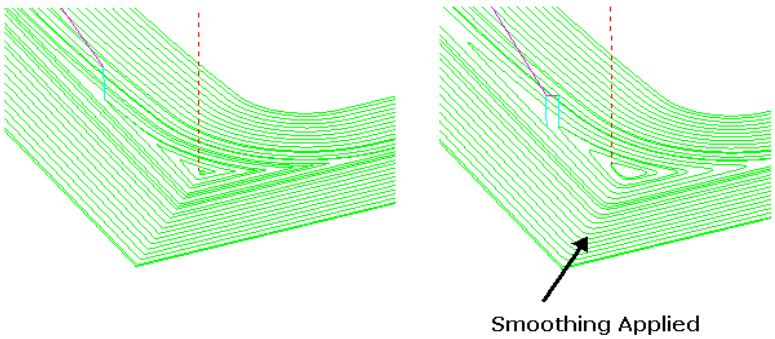
8 **Apply** and **Accept** the dialog.


Copyright © Autodesk

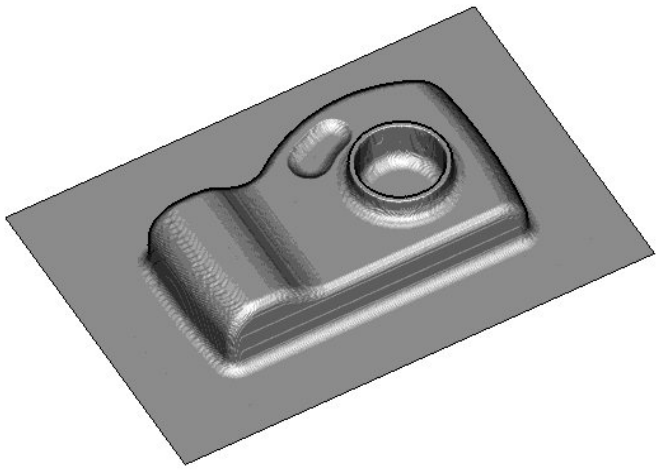
The **toolpath** now is more efficient with the rapid moves at ***skim height***, the ***long links*** using **Vertical arc** Leads, and the ***short links*** being linked along the surface.




*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.*




9 Simulate all of the toolpaths in **ViewMILL**.



10 Select **File** – **Save Project** to update (but do not close):-

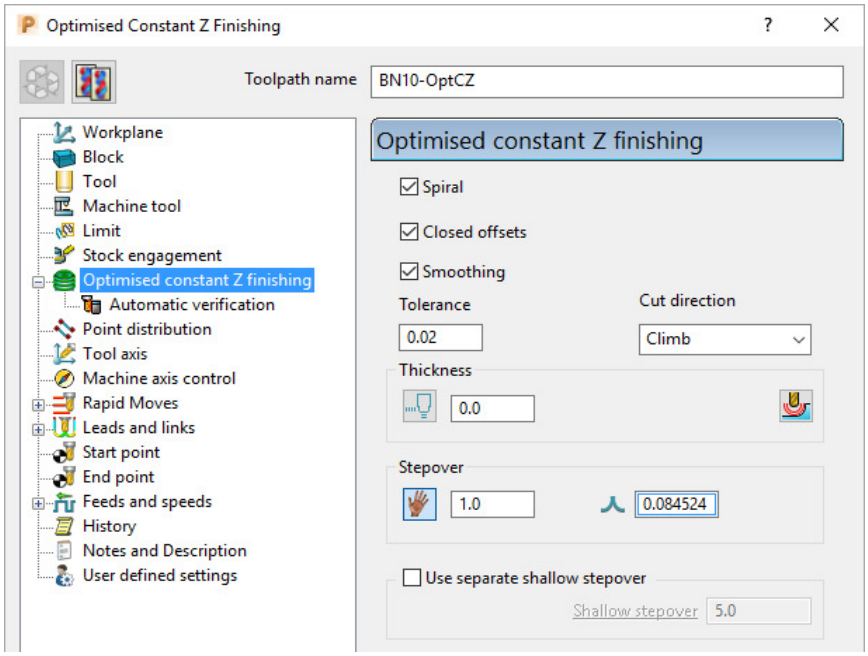
**.....\COURSEWORK\PowerMill-Projects\Camera_Example**

# Optimised Constant Z Finishing

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




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

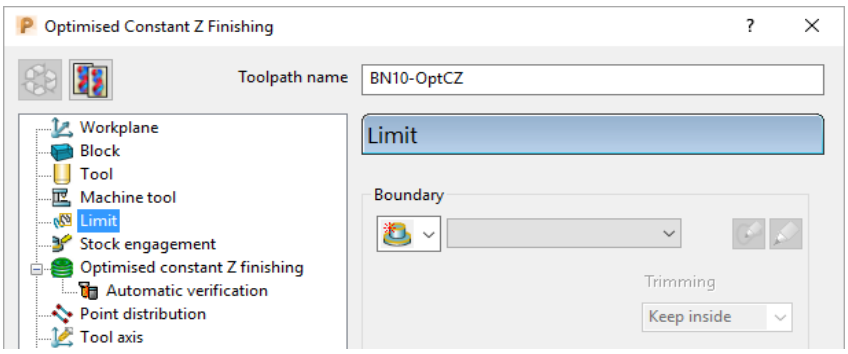
b Tick the **Spiral**, **Closed Offsets**, and **Smoothing options**.

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

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

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

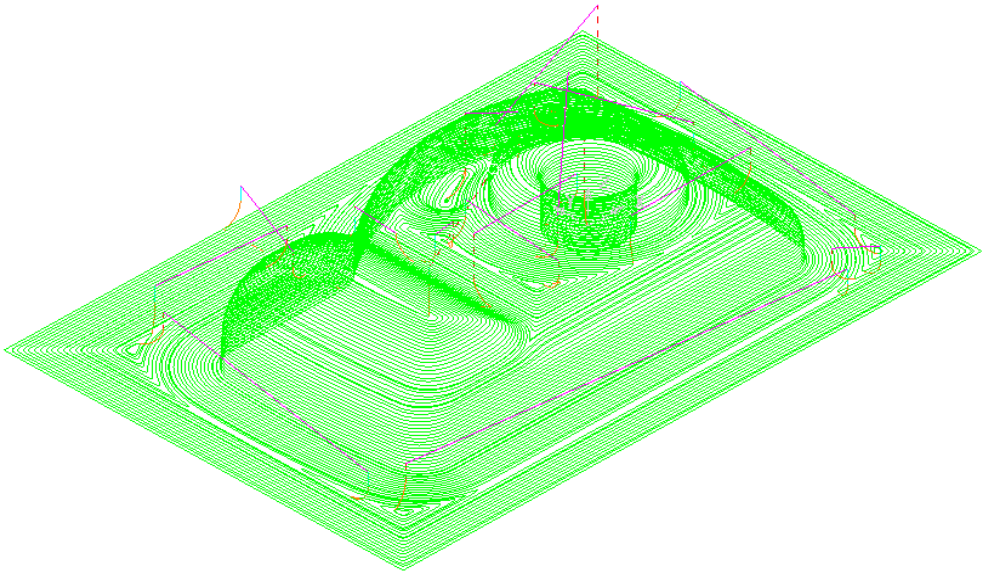
f On the **Limit** page, make sure that the **Boundary** is not selected.




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

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


Copyright © Autodesk



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

**.....\COURSEWORK\PowerMill-Projects\Camera_Example**

# Steep and Shallow Finishing

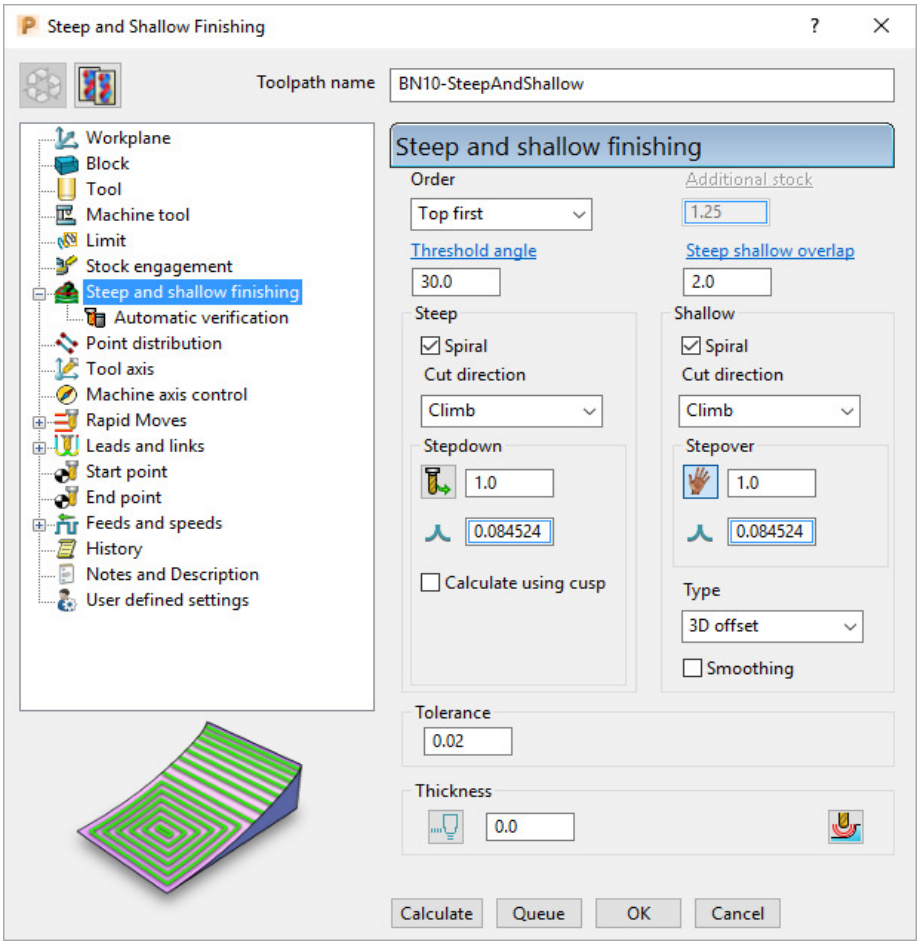
This strategy includes the following additional options:-

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:




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.

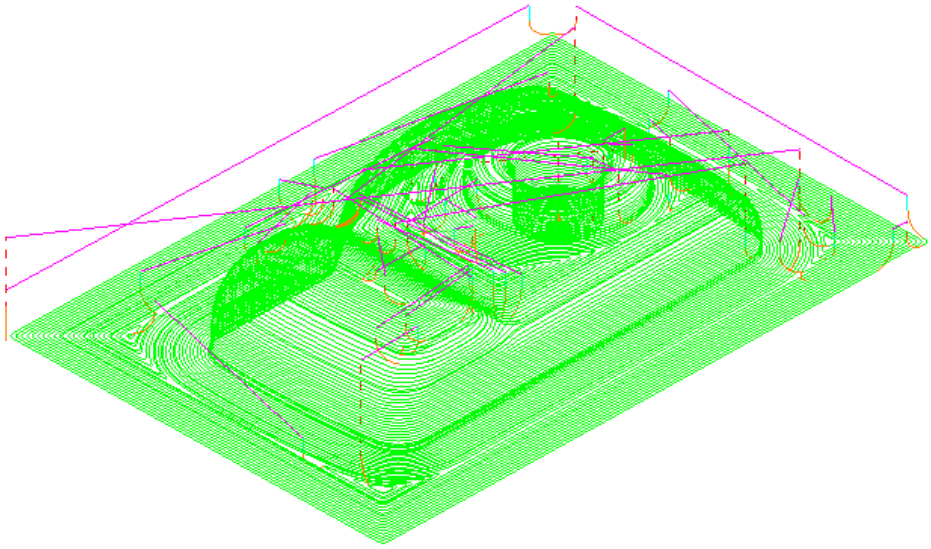
e For both the **Stepdown** and **Stepover** input **1**.

f For **Type** (shallow) select **3D offset**.

g ***Tick*** the **Smoothing** box.

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


Copyright © Autodesk



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

**.....\COURSEWORK\PowerMill-Projects\Camera_Example**

# Steep and Shallow finishing with separate Shallow Stepover

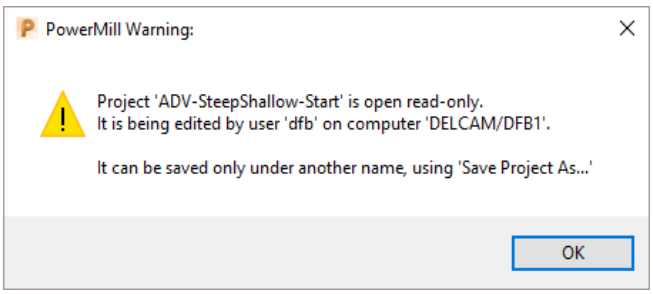
Both **Optimised Constant Z** and **Steep and Shallow** strategies have an option to apply a separate, larger **Stepover** across shallow areas. This option is designed to enable a more efficient machining of shallow areas with a **Tip Radiused** tool.

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

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

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

**.....\PowerMill_Data\Projects\ADV-SteepShallow-Start**



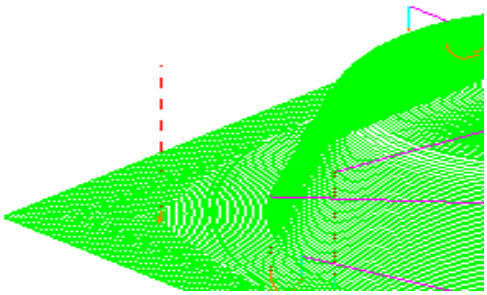

3 Click **OK** to load the read-only **Project** into **PowerMill**.

4 **Save Project As**:

**.....\COURSEWORK\PowerMill_Projects\ADV-SteepShallow-EX1**

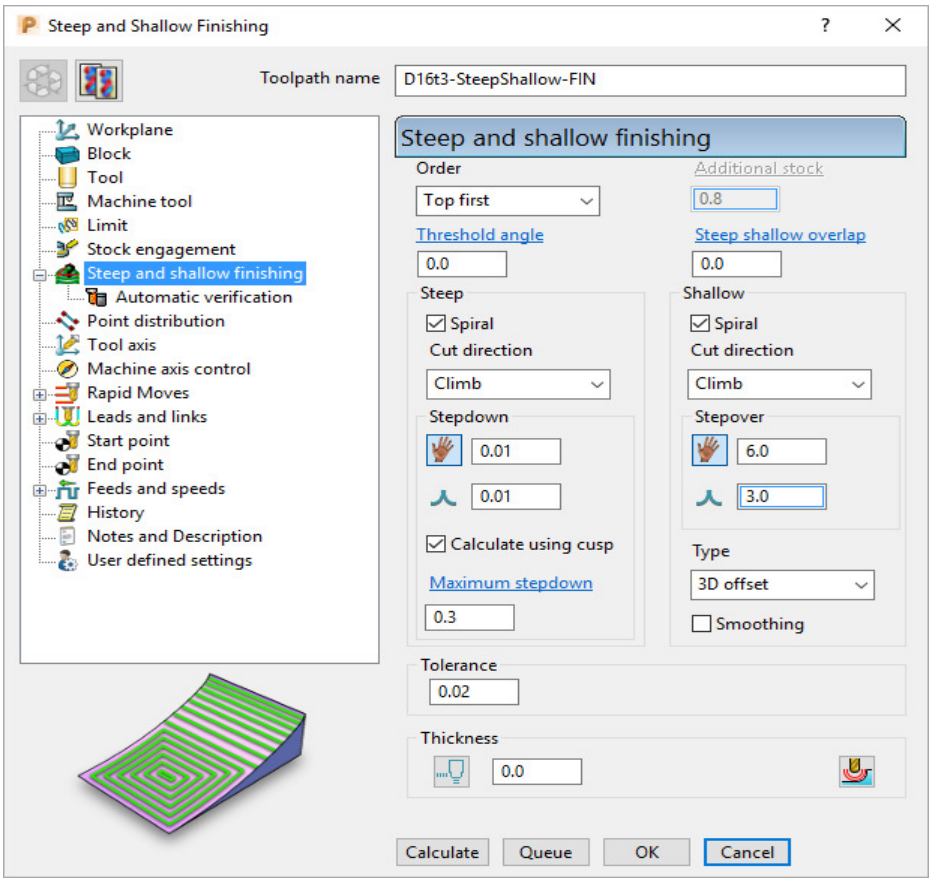
5 In the **PowerMill *explorer***, right-click on the existing toolpath **D16t3-SteepShallow-FIN** and select **Settings**.

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.




a Select **Order** as **Top first**.

b Input the **Threshold angle** as **0**

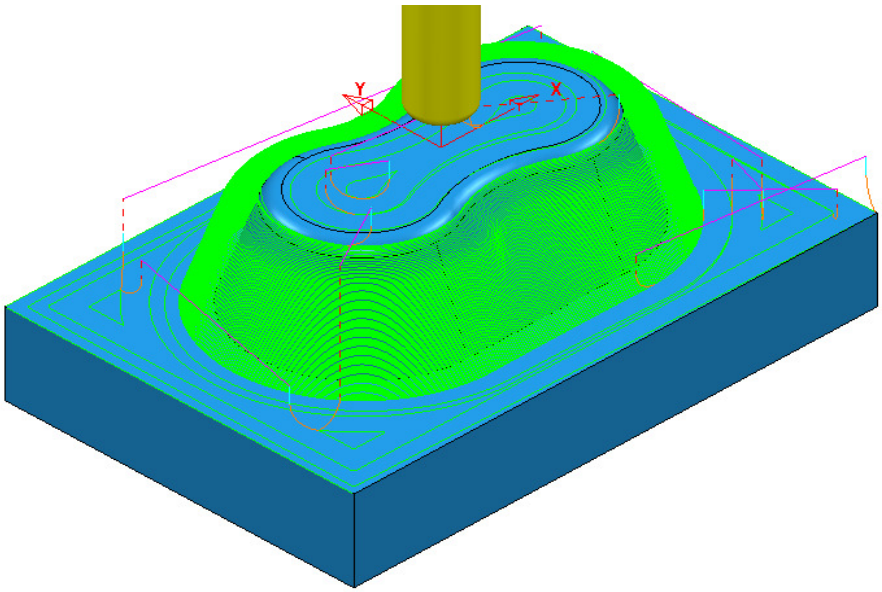
c Tick the **Calculate using cusp** box and input .

d Input a **Stepdown** of **0.01** and a **Maximum stepdown** of **0.3**

e For the shallow regions, enter a **Stepover** of **6**

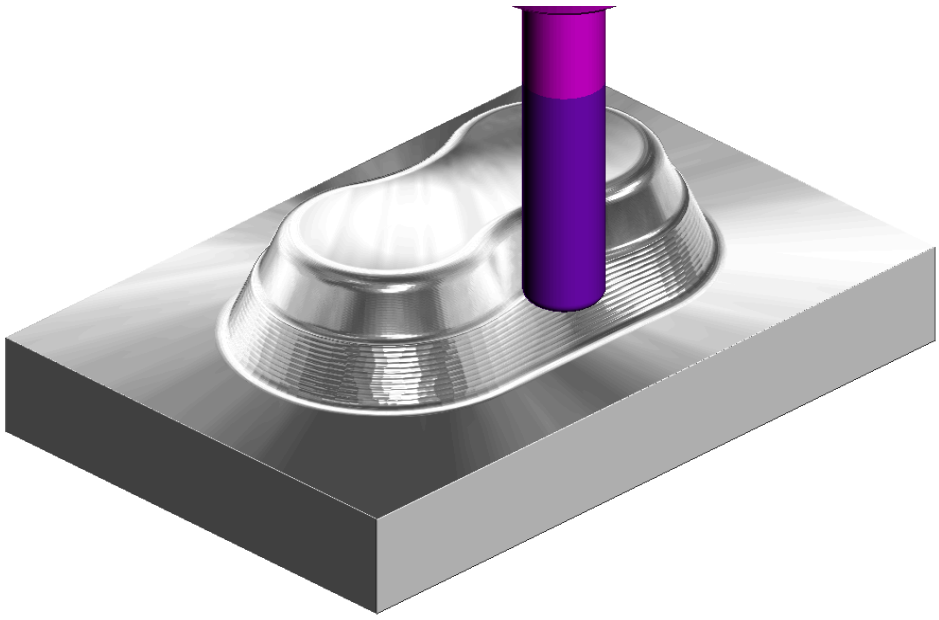

Copyright © Autodesk

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:-

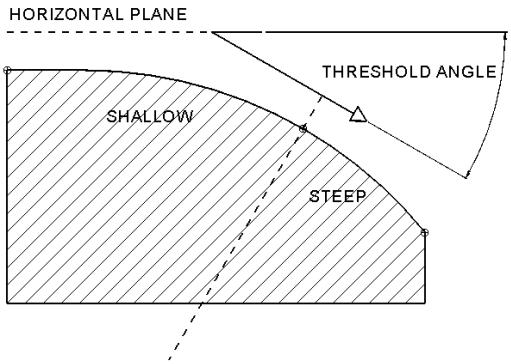
**.....\COURSEWORK\PowerMill_Projects\ADV-SteepShallow-EX1**

# 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**.

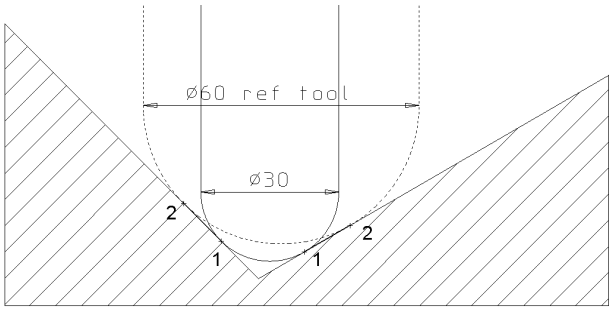
All **Corner Finishing** strategies include a **Threshold Angle** option. This is the downward angle from the horizontal, where the steep and shallow portions meet.

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.

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

12 From the main menu, Select **File** - **Open Project** and select:

**.....\PowerMill_Data\Projects\CornerFinish_start**

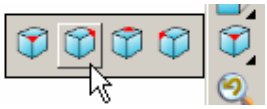
13 From the main menu, select **File** - **Save Project As:**

**.....\COURSEWORK\PowerMill-Projects\CornerFinish**


Copyright © Autodesk

# Corner Along Finishing

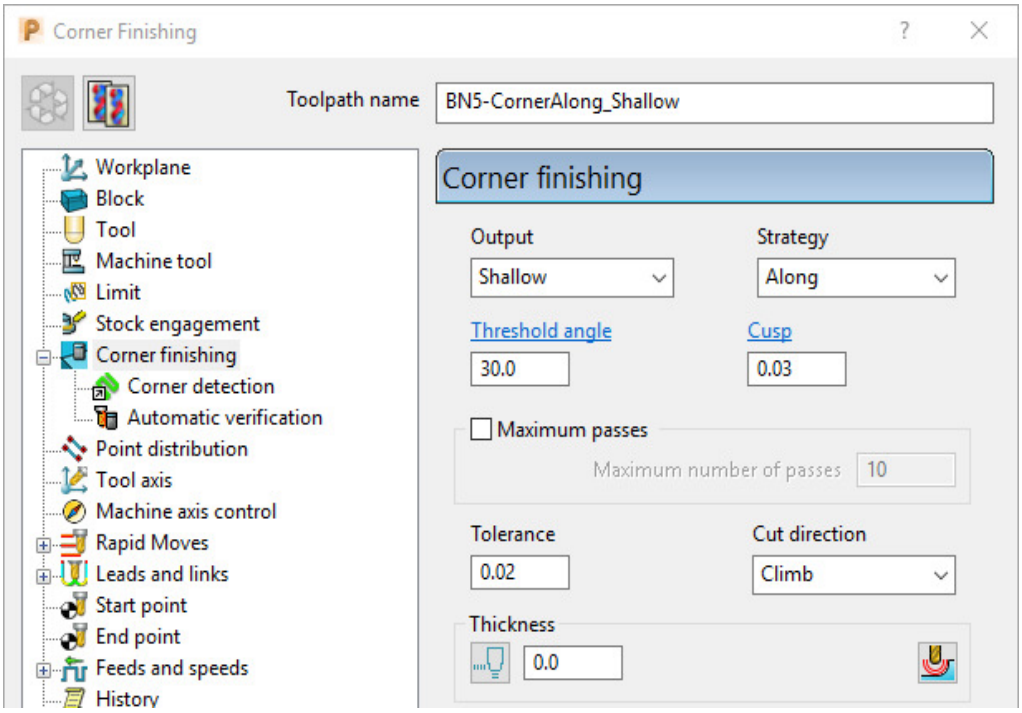
14 Select an **ISO 2** view.



15 Activate the **toolpath**, **D16T3-rghA1** to restore the settings.

16 Create a Dia 5 ball nose tool named **BN5**.

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



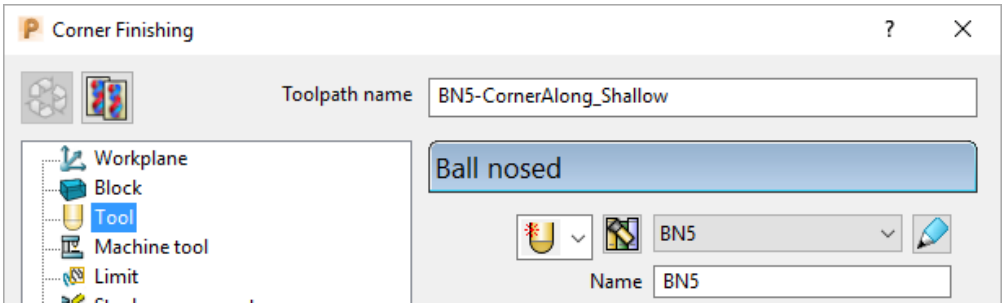

a Enter Toolpath name as **BN5-CornerAlong_Shallow**.

b Change ***Output*** to **Shallow** and ***Strategy*** to **Along**.

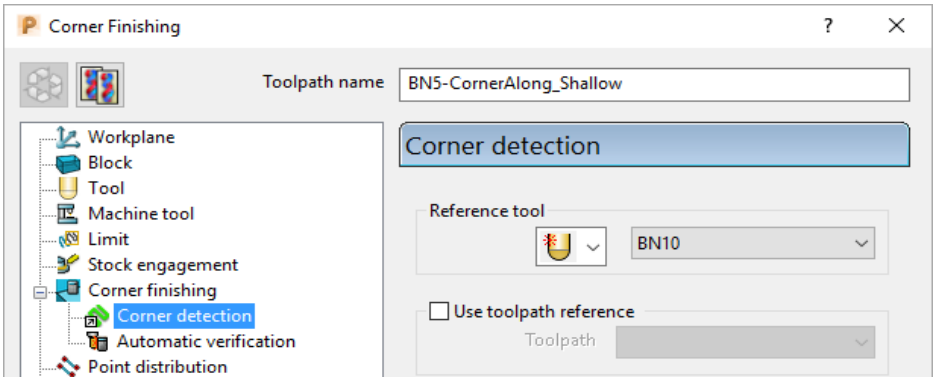
c Set **Cusp** to **0.03**

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

18 On the **Tool** page, check that the **BN5** tool is active.



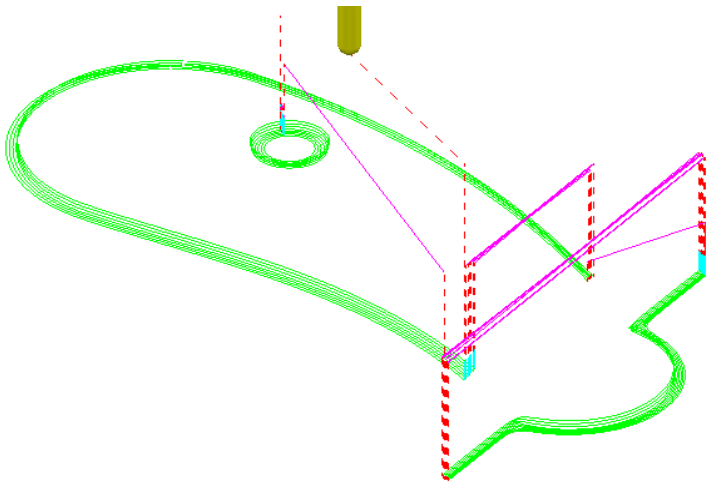
19 Select the **Corner detection** page and select the ***Reference tool*** as **BN10**.




20 Select the **Lead in** page and set the ***1st Choice*** to **None**.

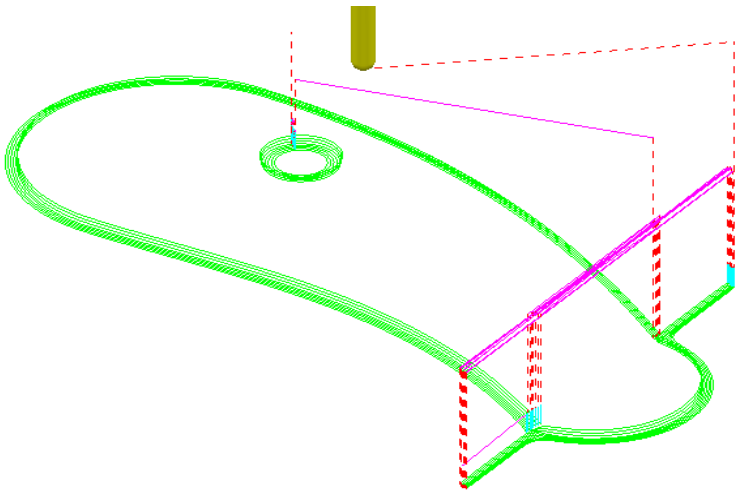
21 **Calculate** to process the **toolpath** and then **Close** the dialog.

22 **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).

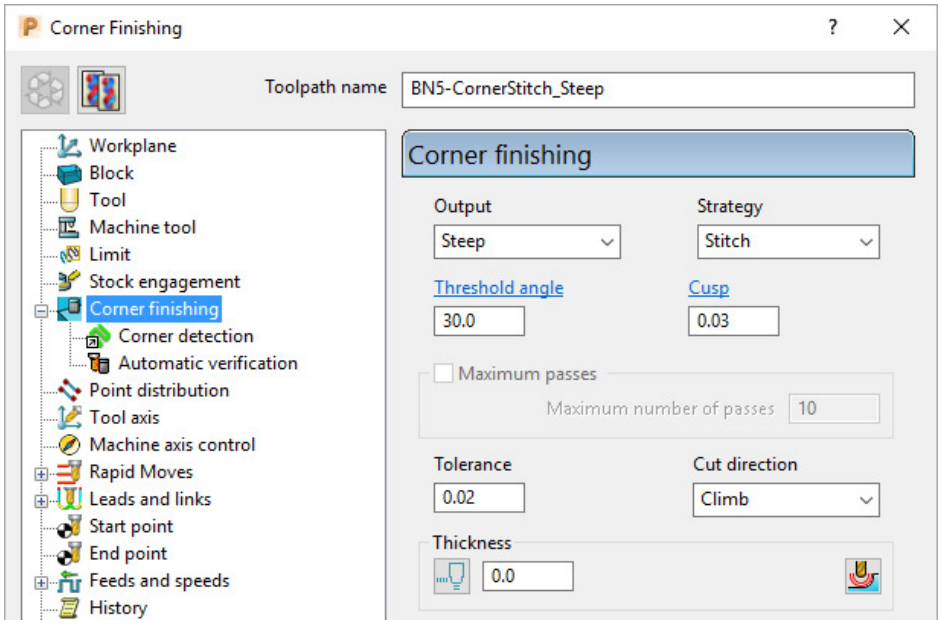
*If the **Threshold Angle** is **90** then the **steep** area will not exist.*




Copyright © Autodesk

# Corner Stitch Finishing

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



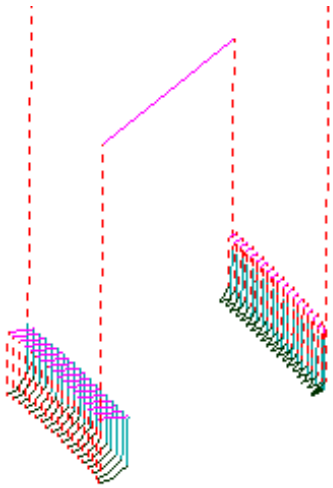

a Enter ***Toolpath name*** as **BN5-CornerStitch_Steep**.

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.

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



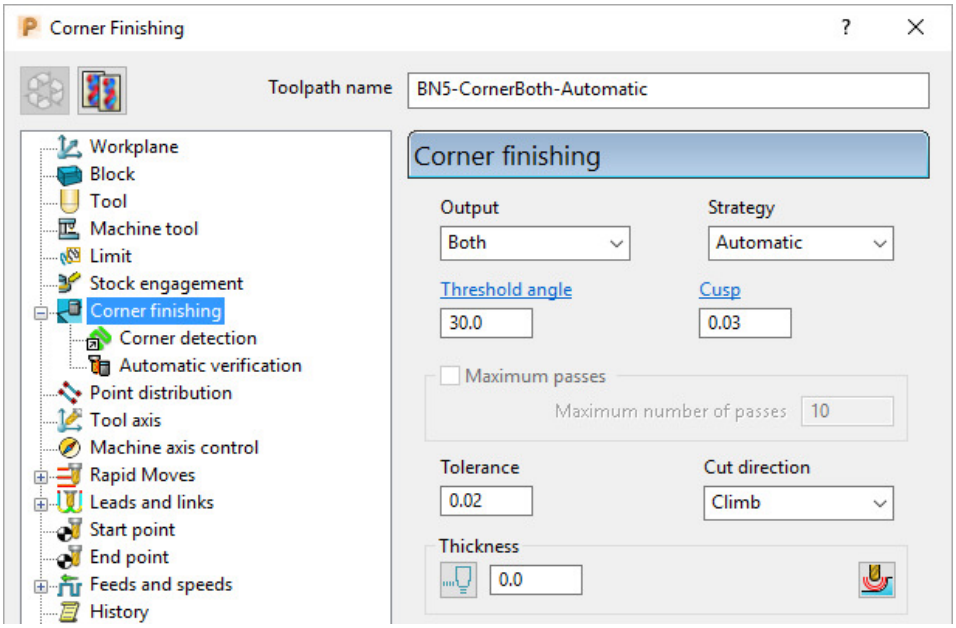
This time, only the steep areas are machined.

25 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.



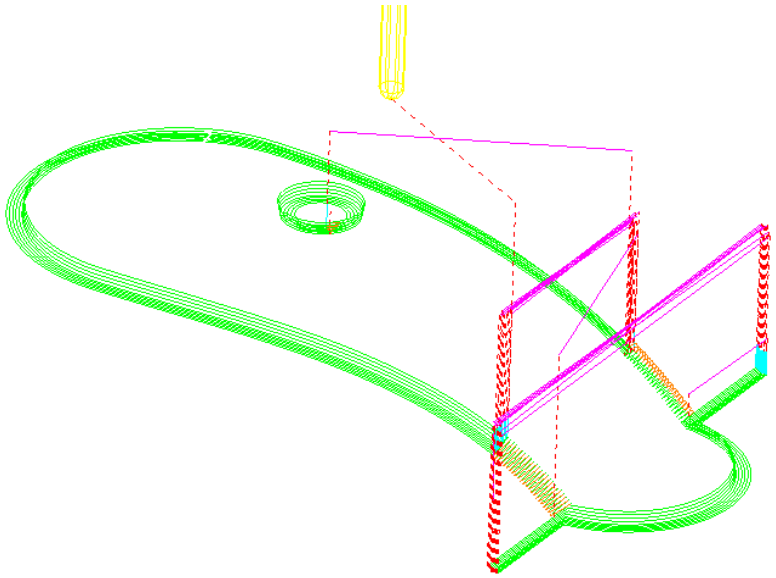

a Enter ***Toolpath name*** as **BN5-CornerBoth_Automatic**.

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.



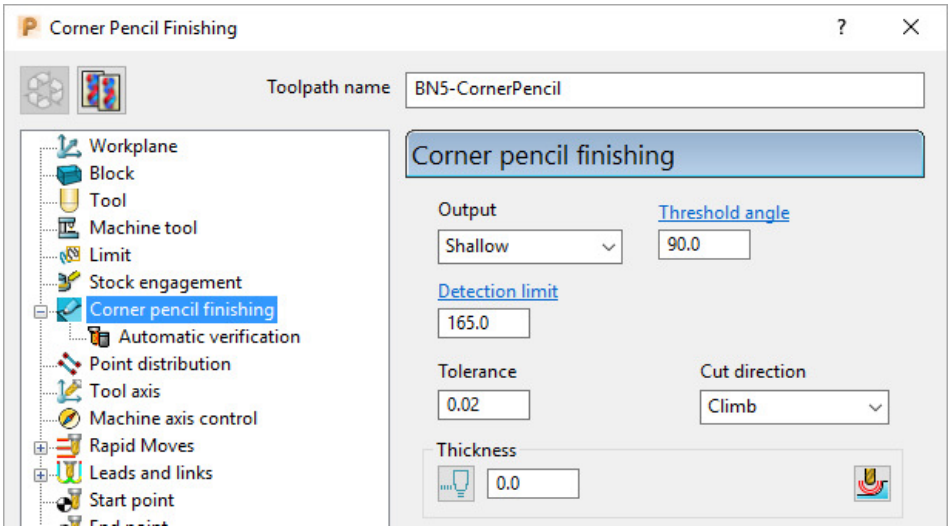
This strategy automatically uses **Along** for the **Shallow** areas and **Stitch** for the **Steep** areas.


Copyright © Autodesk

# Corner Pencil Finishing

**Corner Pencil Finishing** creates a single toolpath along the actual intersection between non-tangential faces on the model.

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




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

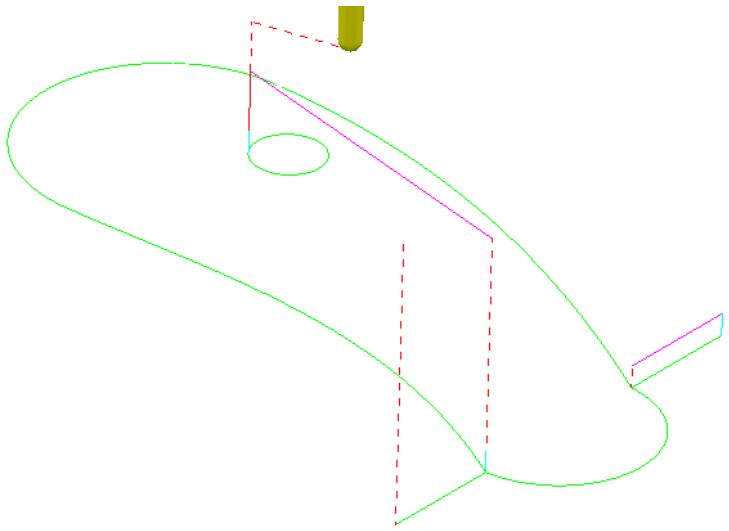
b Change **Output** to **Shallow**.

c Change the **Threshold Angle** to **90**

Note: With a **Threshold Angle** of **90** a **Steep** angle will not exist, so only the **Output** options **Shallow** or **Both** will enable a toolpath to be created.

d On the **Tool** page, make the **BN5** tool active.

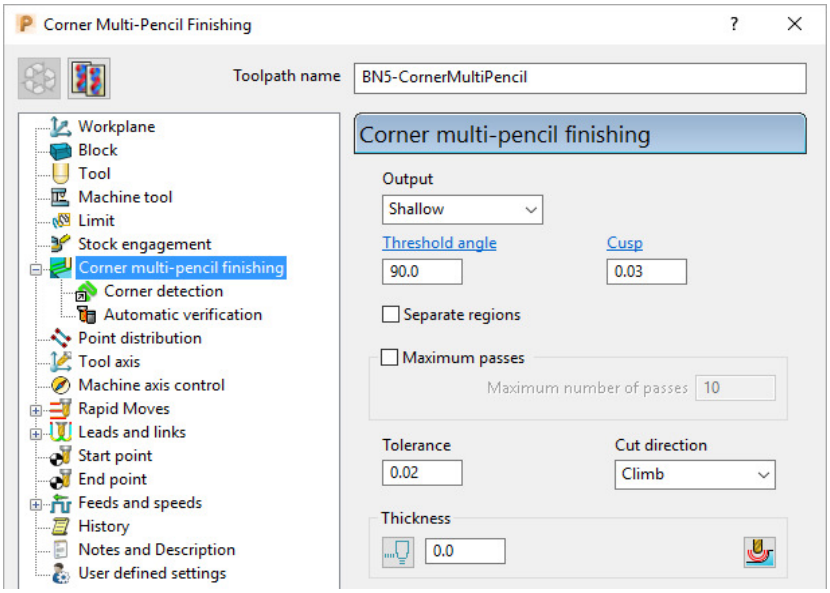
4 Select **Calculate** to process the toolpath, and then **Close** the dialog.



# 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.




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

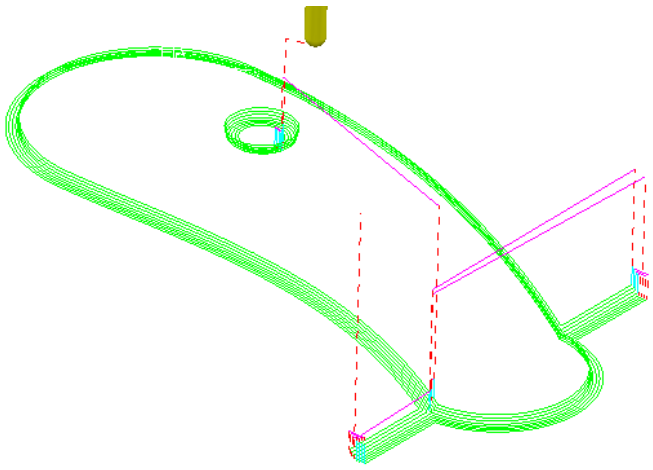
b Change **Output** to **Both**.

c Change the **Threshold Angle** to **90**

d On the **Tool** page, make the **BN5** tool active.

e 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**


Copyright © Autodesk

# 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.

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

5 From the **Main** menu, select **File** - **Open Project:**

**.....\PowerMill_Data\Projects\FlatFinish_Start**

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

**.....\COURSEWORK\PowerMill-Projects\FlatFinish-Example1**

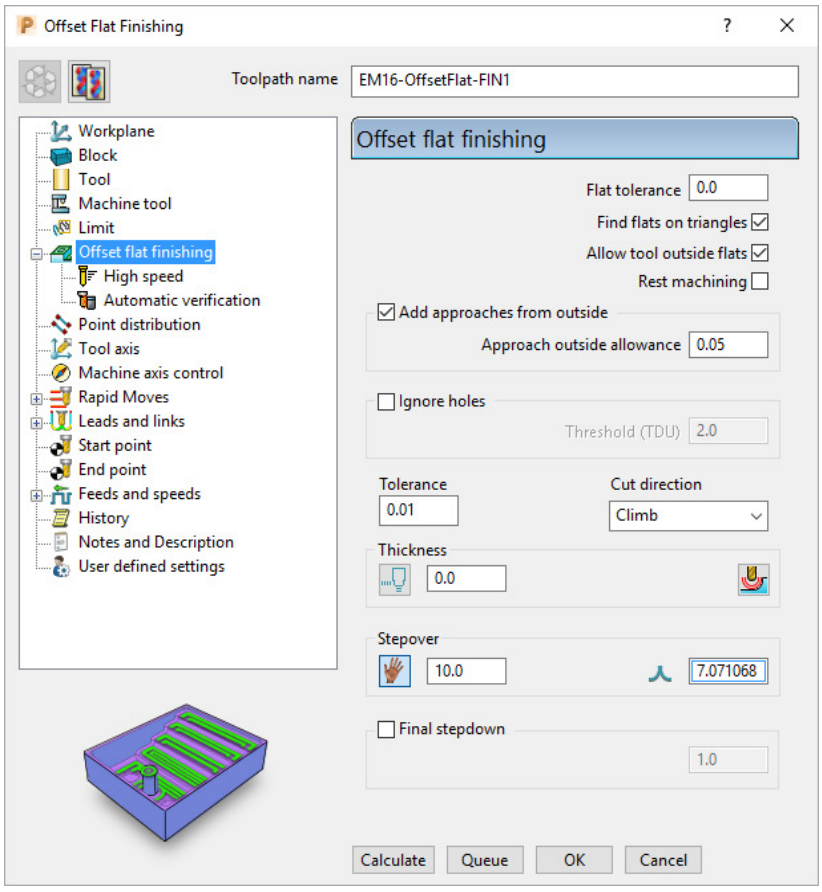
## Offset Flat Finishing

7 In the **PowerMill *explorer***, **Activate** the ***Toolpath*** **D16TR3-RGH1**.

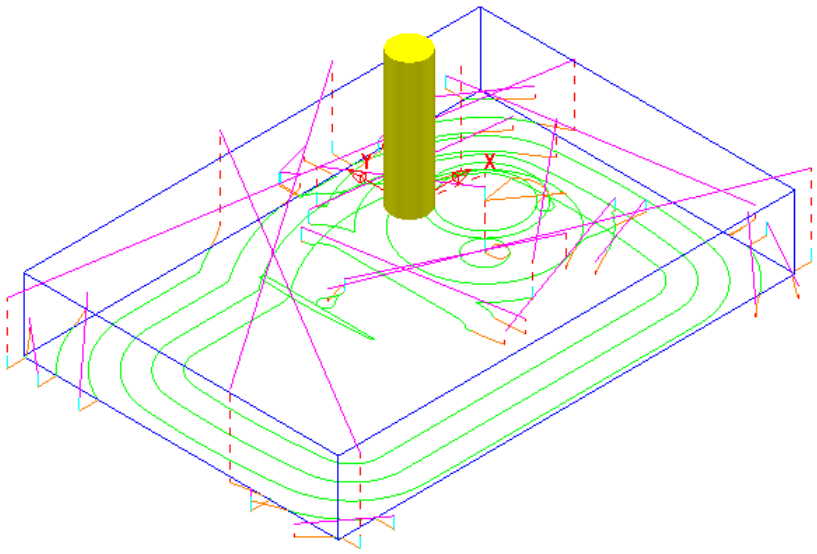
8 Activate the Dia 16 End Mill (**EM16**) **Tool**.

9 De-Activate **Boundary 1**.

10 From the **main** toolbar, select **Toolpath Strategies** and from **Finishing** open an **Offset Flat Finishing** strategy.

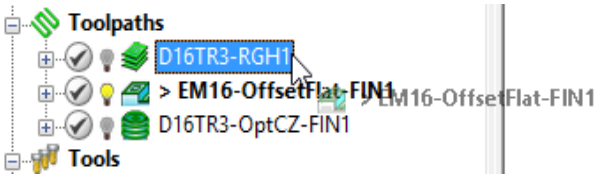



11 Select **Calculate** and when completed, **Close** the dialog.

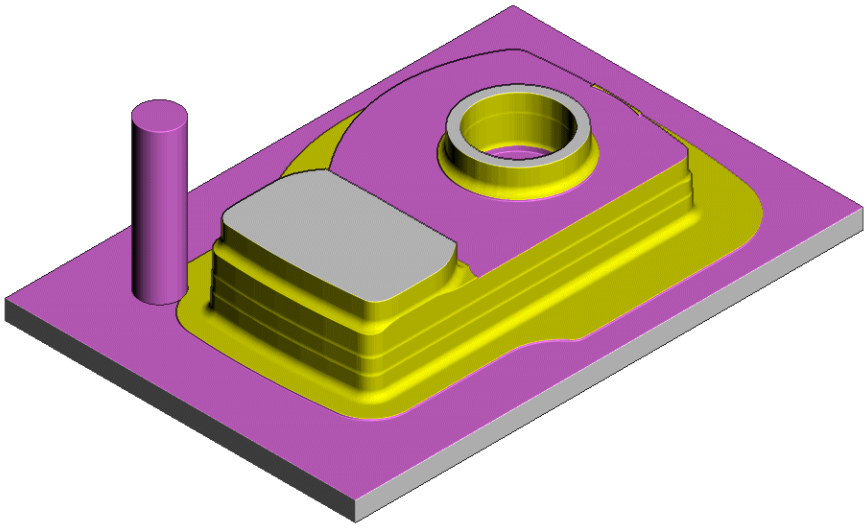


The imported **Project** contains both an **Offset Area Clear** roughing strategy and an **Optimised Constant Z** finishing strategy. It is recommended that **Offset Flats** (**EM16-OffsetFlats-FIN1**) is run directly after the **3D Area Clearance** (**D16t3-RGH1**). For this reason, it is necessary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

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




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

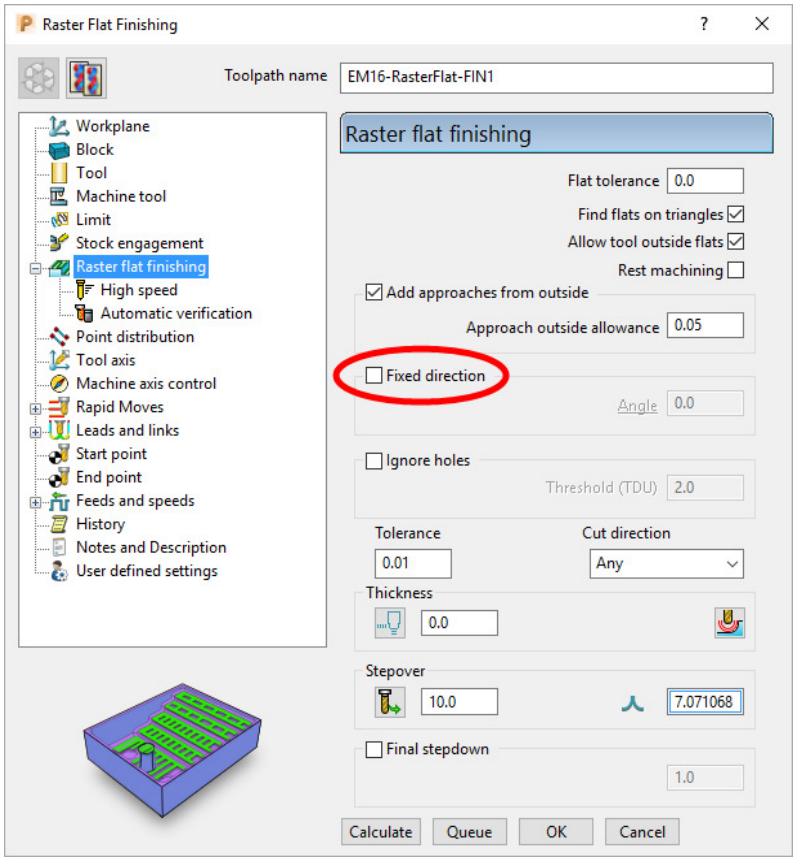


The **ViewMILL** simulation after the **Offset Flat Finishing** strategy is shown above.

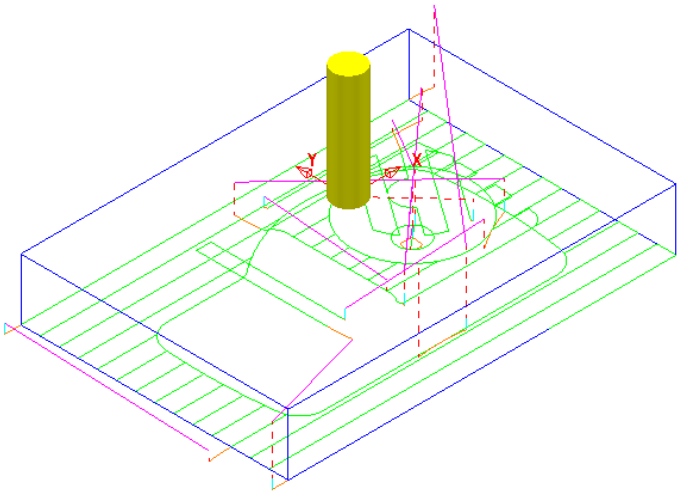

Copyright © Autodesk

# Raster Flat Finishing

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




2 Using the above settings, select **calculate** and when completed, **Close** the dialog.



3 From the **main** menu select **File** - **Save Project** to update:-

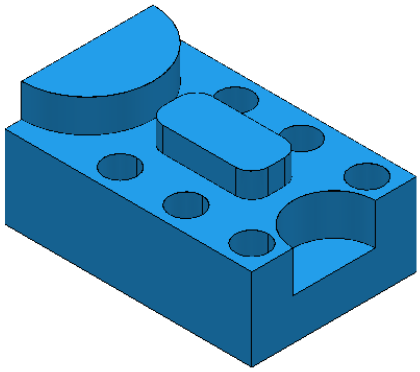
**.....\COURSEWORK\PowerMill-Projects\FlatFinish-Example1**

# Flat Machining example

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

5 Select **File** - **Import Model**:-

**.....\PowerMill_Data\Models\Flats.dgk**



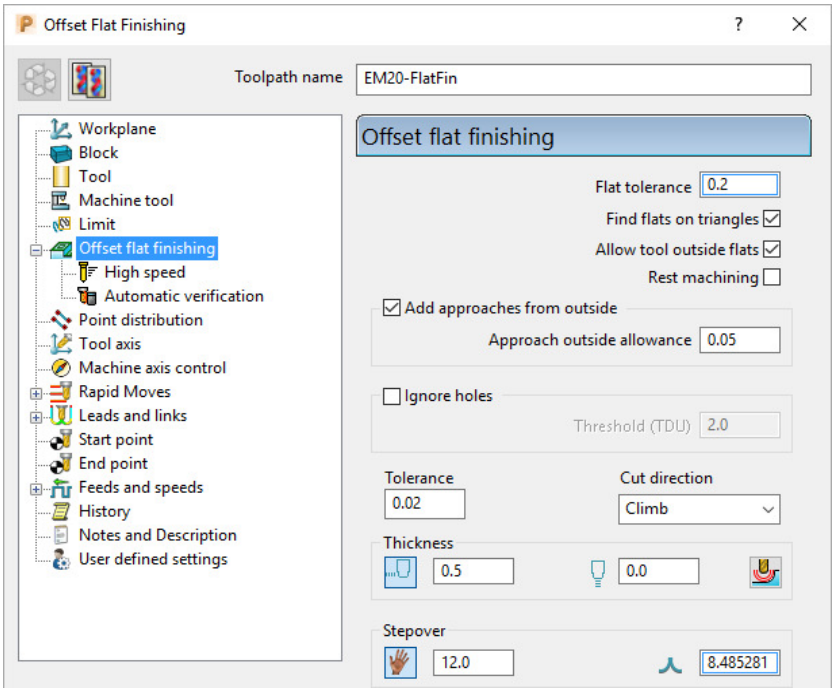
6 **Save the Project as**:-

**.....\COURSEWORK\PowerMill_Projects\FlatFinishing-Example2**

7 Create a 20mm diameter End Mill tool named **EM20**.

8 Calculate the (default) **Block** and then **Calculate** the **Rapid Move Heights**.

9 Open an **Offset Flat Finishing** strategy and:-




a Name the toolpath **EM20-FlatFin**.

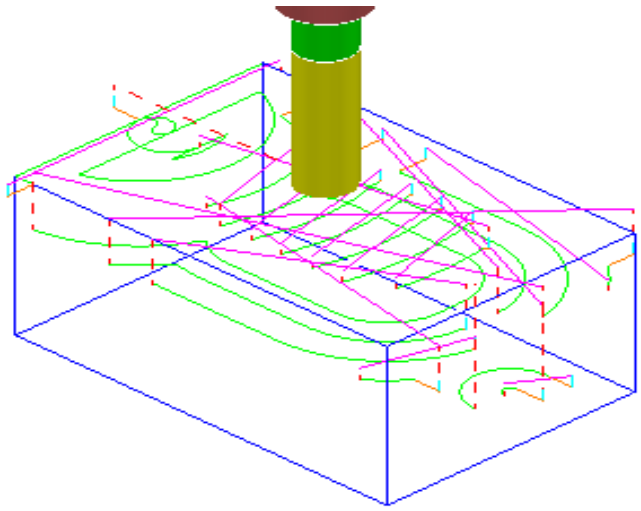
b Untick the **Rest machining** box.

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.


Copyright © Autodesk

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.

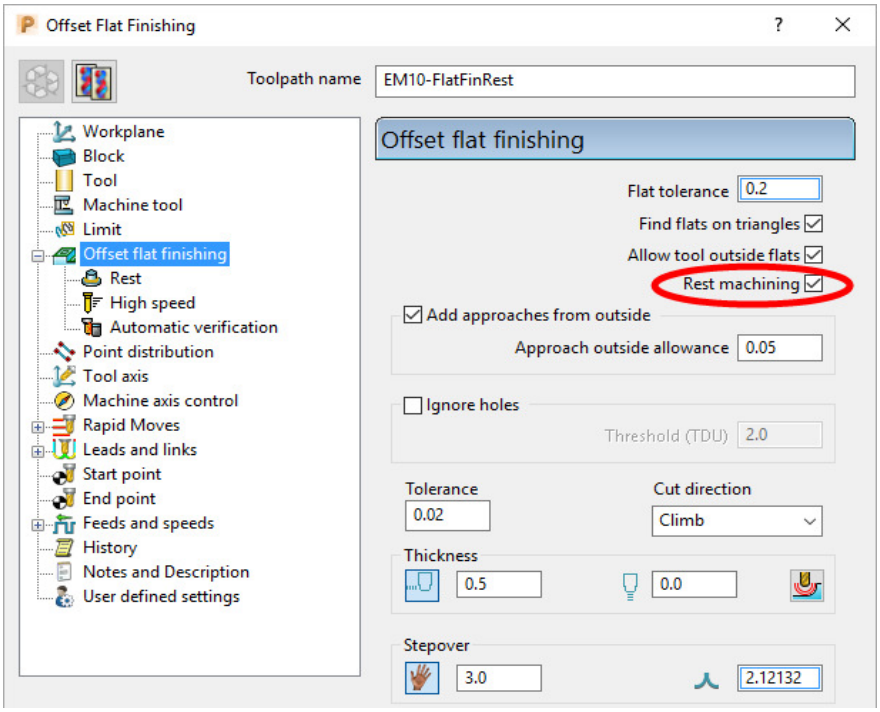


10 Create a **Dia 10 End Mill**, named **EM10**.

11 Activate the **Toolpath**, **EM20-FlatFin** and then select **Settings**.

12 Click to create a copy of the strategy.

13 In the main **Offset flat finishing** page of the strategy dialog:



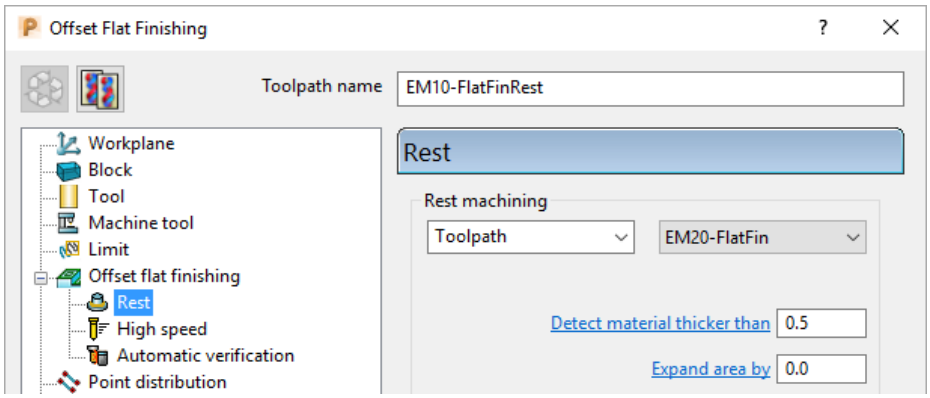

a Enter **Toolpath name** as **EM10-FlatFinRest**.

b Enter **Stepover** as **3.0**

c Activate the **EM10** tool.

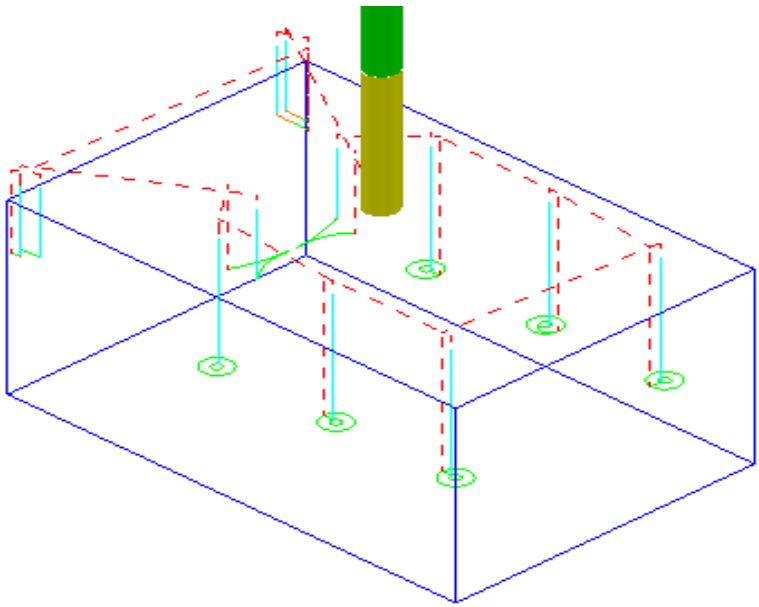
d Tick the **Rest machining** box.

14 Select the **Rest** page, and input the reference Toolpath as **EM20-FlatFin**.




15 **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:



16 From the **main** menu select **File** - **Save Project** to update:-

**.....\COURSEWORK\FlatFinishing-Example2**


Copyright © Autodesk

# Surface Finishing Example 1

The **Surface Finishing** strategy creates tool tracks that follow the natural, ***surface curves*** on a single, selected, **Surface**.

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

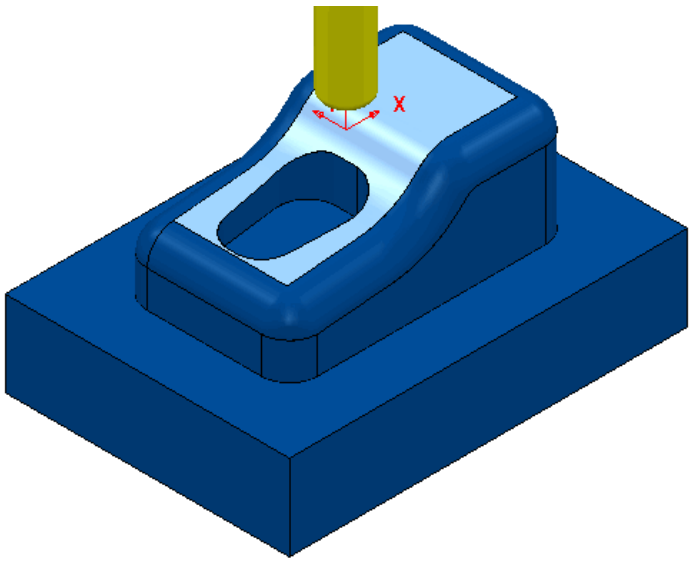
1 From the main menu, select **File** - **Delete All** and **Tools** - **Reset Forms**.

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

**.....\PowerMill_Data\Projects\EditToolpaths_3**

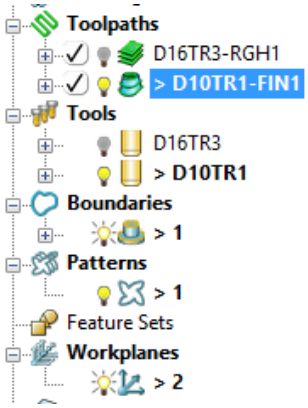
3 From the **main** menu , select **File** - **Save Project As:**

**.....\COURSEWORK\PowerMill-Projects\PunchForm_3**




4 Activate the toolpath **D10TR1-FIN1** to reinstate all the original settings.

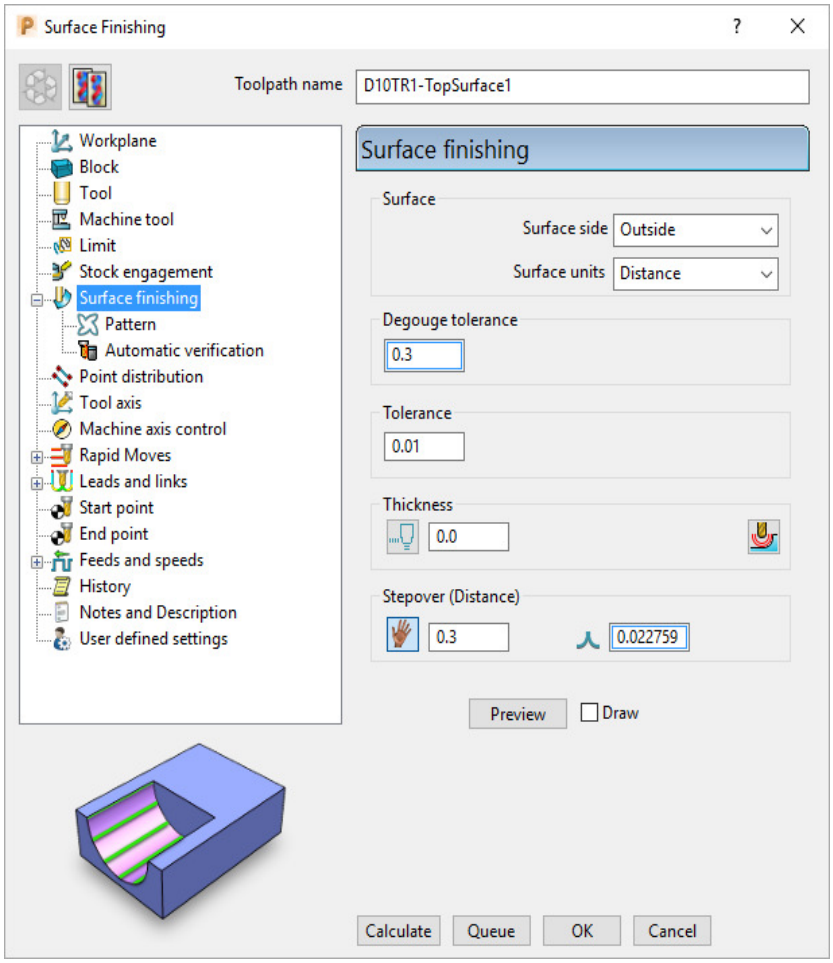
All you need from the original settings is the original; **Active Workplane**, **Block**, **Rapid Move Heights**, and the ***Tiprad tool*** named **D10TR1**.



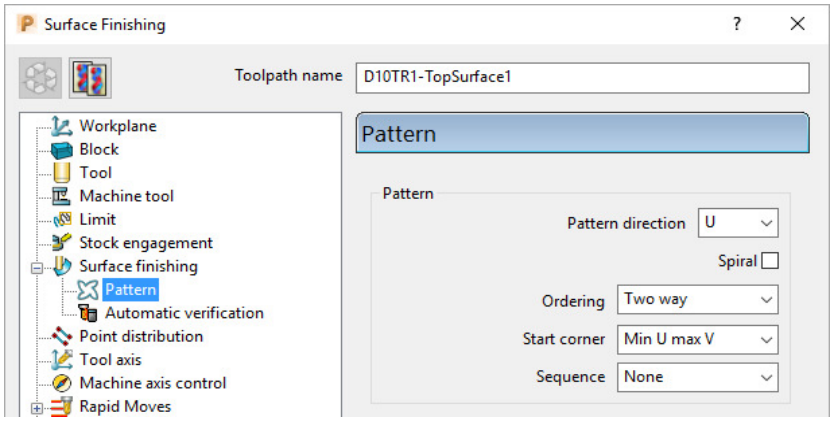

5 De-Activate both the ***Toolpath*** **D10TR1-FIN1**, and ***Boundary*** **1**.

6 Select the ***upper surface*** on the punch model (as displayed above).

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




8 Select the **Pattern** page:




a Set **Pattern direction** – **U**

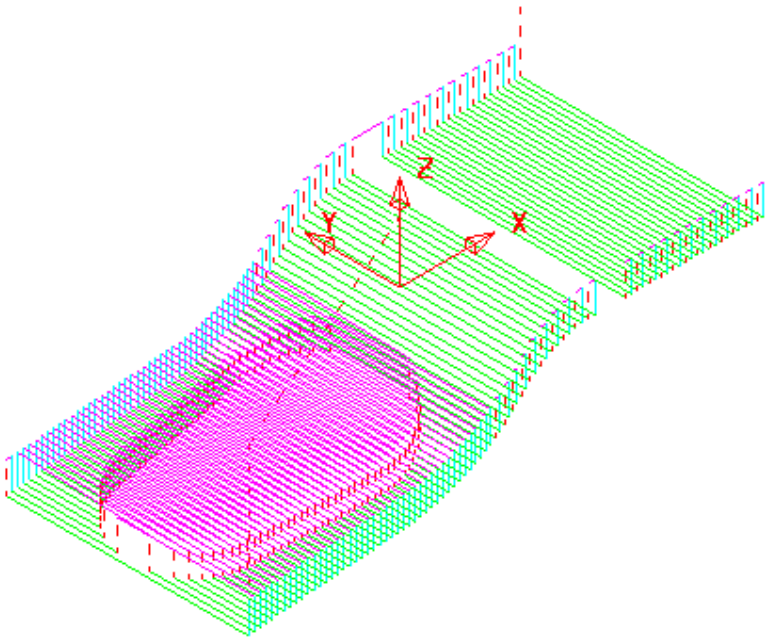
b Un-tick **Spiral**, and then select **Ordering** - **Two way**

c Select **Start corner** - **Min U max V**

9 Select the **Preview** tab to display the **Pattern** along its start point (arrow)

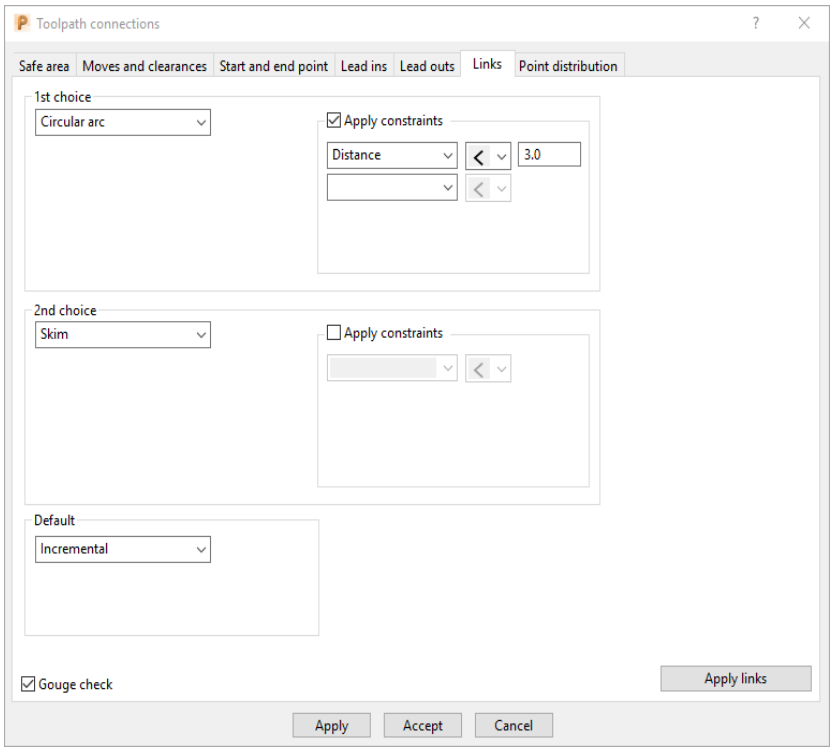
10 Select **Calculate** to create the toolpath and then **Close** the dialog.


Copyright © Autodesk



11 From the main toolbar, click the **Leads and Links** button.

12 On the **Links** page:



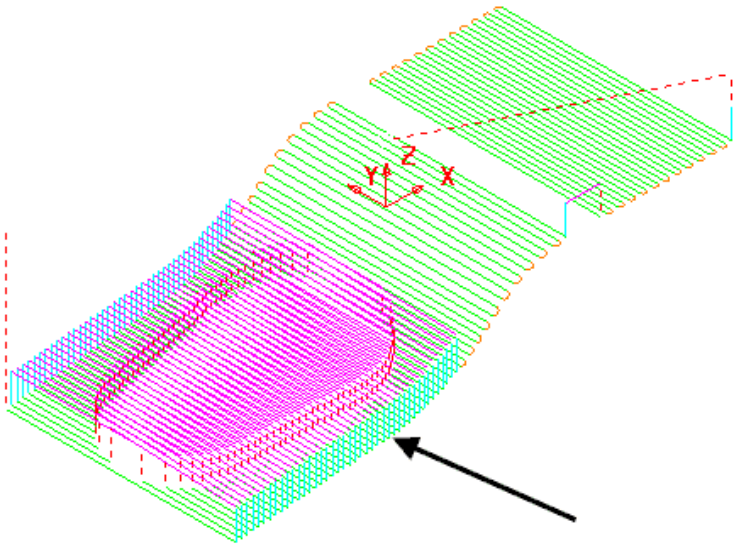

a In ***1st choice*** select **Circular arc** and then tick **Apply constraints** with a **Distance < 3.0**

b Select ***2nd choice*** - **Skim**

c 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.*

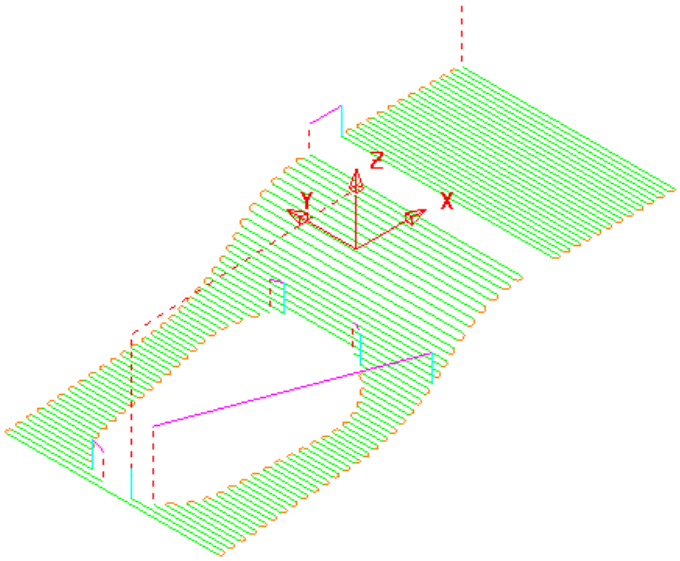



In the resultant toolpath, the natural order of the tool tracks and link moves across the 2 narrow areas is not very efficient.

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.

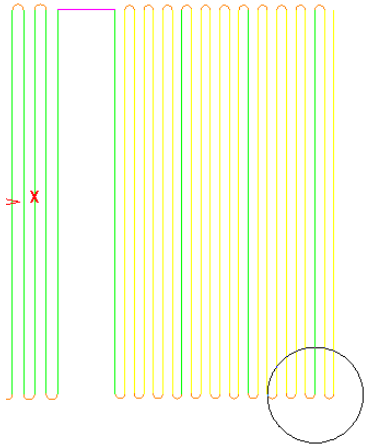

Copyright © Autodesk

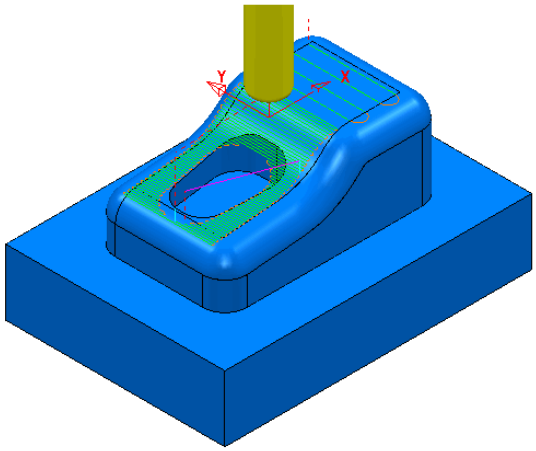
The illustrated toolpath is created with a **1 mm Stepover** to help with visualisation. It has also been edited using the **Automatic Reorder and Reverse** option to eliminate as many air moves as possible.

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**.

16 From the **main** menu, select **Draw** > **Cursor** > **Tool** to provide a visual guide when selecting a suitable number of tool tracks.

17 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:



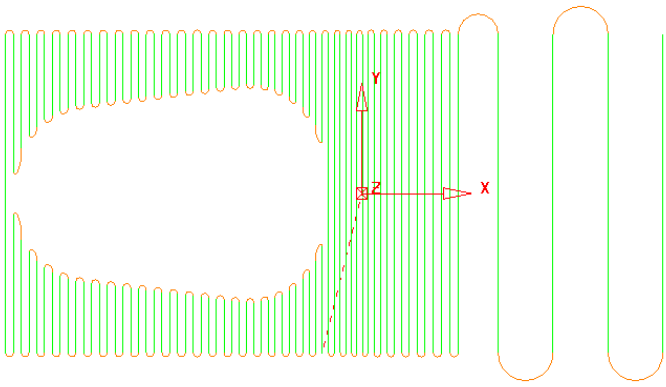




The **Tool** outline is displayed above to visualise suitable **Stepover** (**Draw** > **Cursor** > **Tool**).

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

18 Right-click the toolpath and select **Edit** > **Delete Selected Components** to remove the selected tool tracks.

19 In the **Links** dialog change the ***1st choice*** (with **Apply constraints** still ticked) to **Distance < 15** before selecting **Apply**.




20 From the **main** menu select **File** > **Save Project** to update:-

**.....\COURSEWORK\PowerMill-Projects\PunchForm_3**

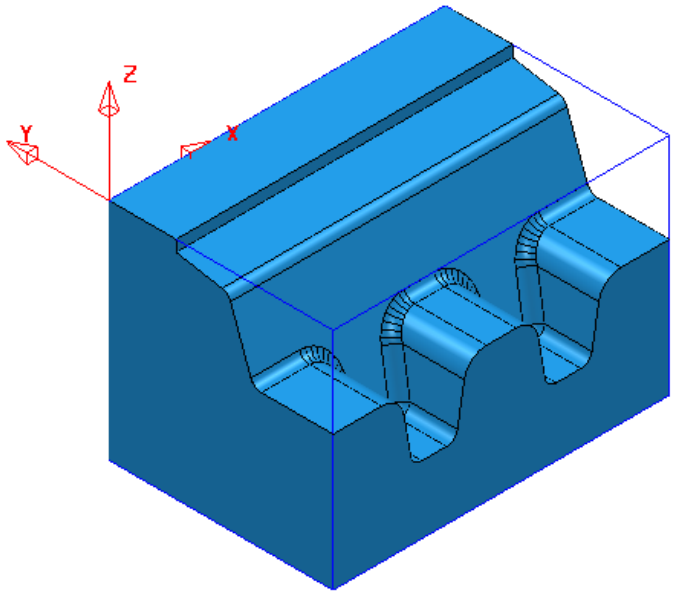
# Toolpath Z-Limits

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**.

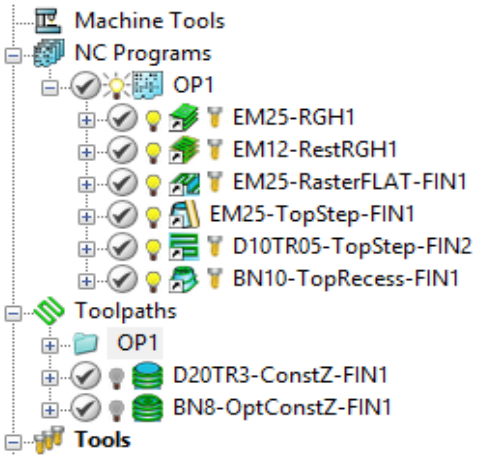
2 **Open** the read-only **Project**:-

**.....\PowerMill_Data\Projects\Zlimits-Start**




3 **Save Project As**:-

**.....\COURSEWORK\Zlimits-EX1**



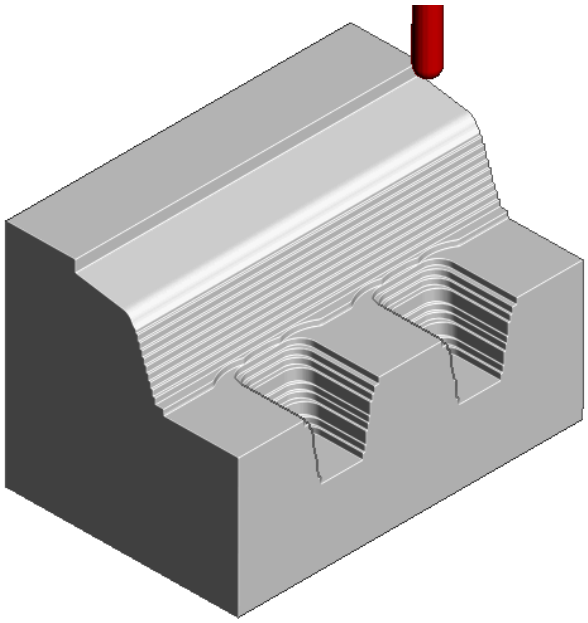

The **Project** already contains **8 Toolpaths**, 6 of which have been put in a **sub-folder** named **OP1**. The 2 remaining **Toolpaths** require modification as they are both duplicating areas already included in other strategies. This will become evident during a **ViewMILL** simulation.


Copyright © Autodesk

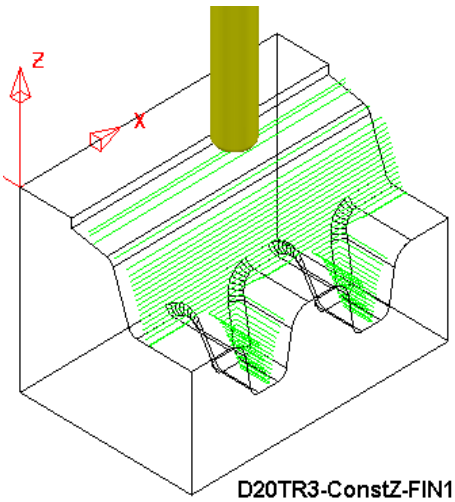
4 Activate the toolpath **D20TR3-ConstantZ-FIN1** to reinstate the original **Block** and settings.

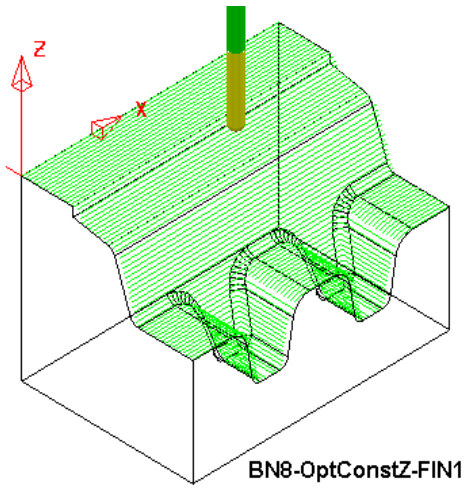
5 Run a **ViewMILL** simulation on the existing ***NC Program*** **OP1**.

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**.



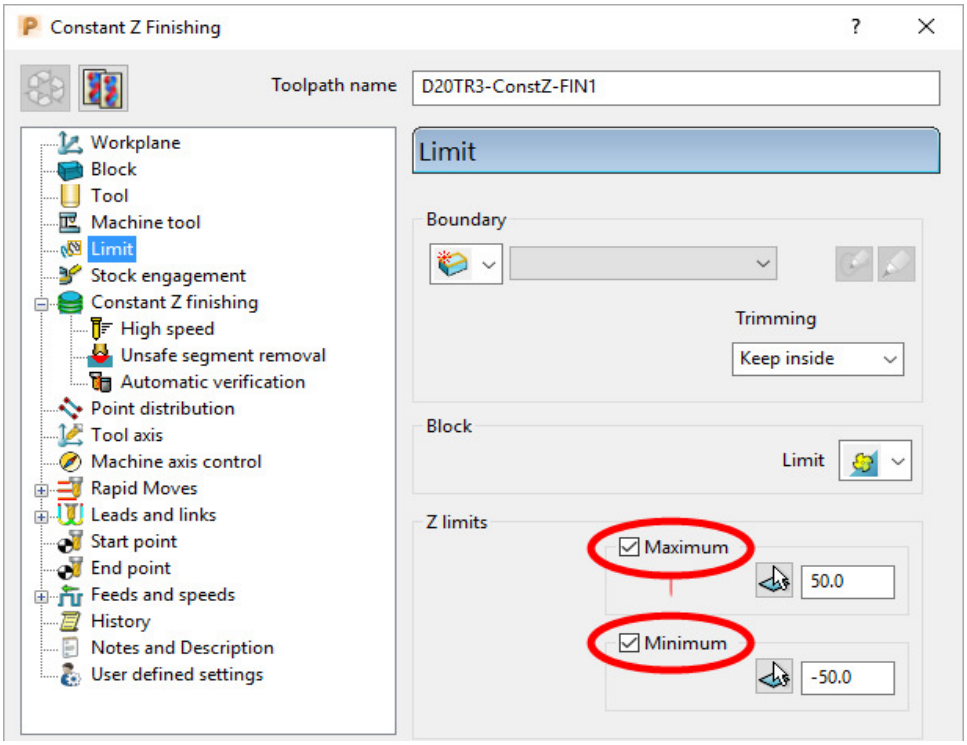

**D20TR3-ConstZ-FIN1** needs to be localised to operate down the **angled face** to the top of the lower recesses.

**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.

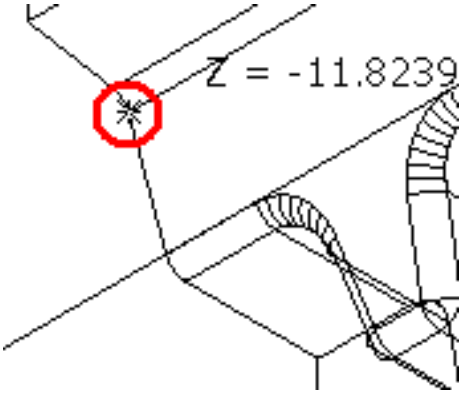
7 Return to the **PowerMill** (***No Image***) view and disconnect **ViewMILL** from the simulation (Switch the ***Green*** Sphere to ***Red***).

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.




9 Select ***Recycle*** followed by the **Limit** page and ***tick*** both the **Maximum** and **Minimum** boxes.

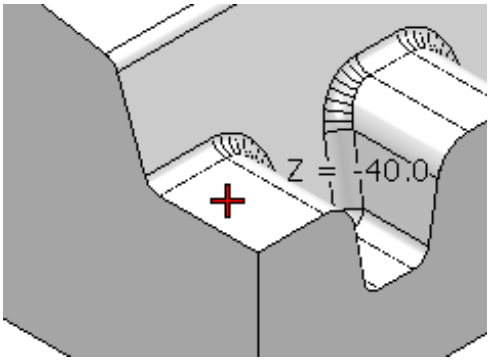
10 For The **Maximum** select ***Pick the highest Z Value*** option.




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


Copyright © Autodesk

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

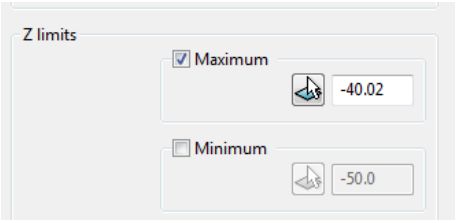



13 Snap the ***key-point*** (shown above) on the top of the flat area to collect and insert the value **-40**.

14 Back in the dialog select **Calculate** to re-process the updated **Constant Z** finishing strategy and then **Close** the dialog.

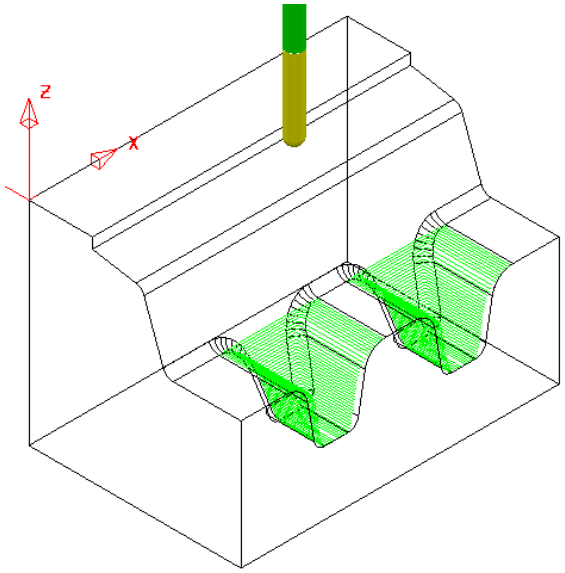
15 In the **PowerMill *explorer***, right mouse click on the toolpath **BN8-OptConstZ-FIN1** and from the local menu, select **Activate** followed by **Settings** to open the strategy dialog.

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

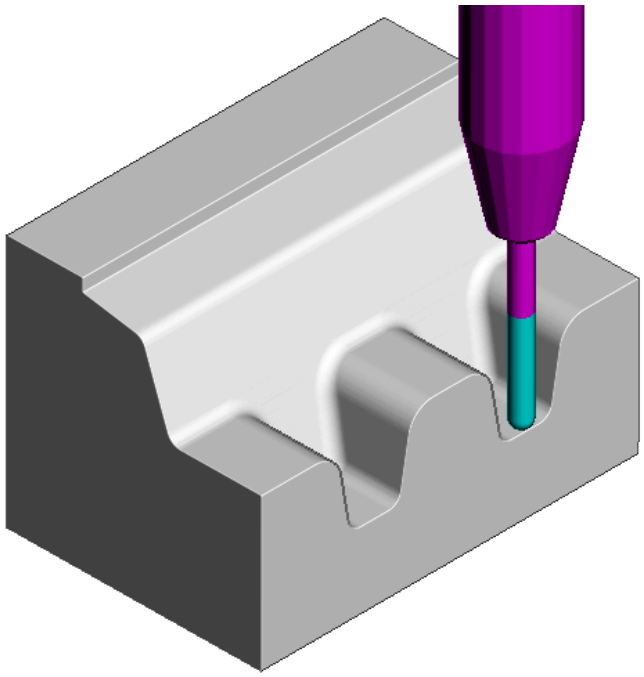



17 Input the value **-40.02** in the **Maximum** box.

18 Back in the main form select **Calculate** to re-process the updated **Optimised Constant Z** finishing strategy and then **Close** the dialog.



19 Re-connect **ViewMILL** and continue the 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**


Copyright © Autodesk

# 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.

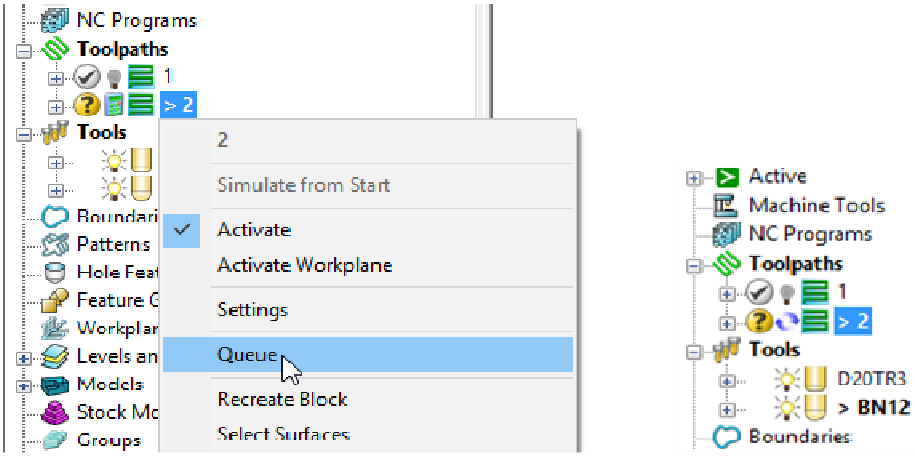
*You cannot use other PowerMill options while the toolpaths are being calculated.*

## 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 sequentially in the background.

To **Queue** a toolpath for calculation, you can, either:

- Click the **Queue** button in a strategy dialog, or,
- Right-click on a toolpath and select **Queue** from the local menu.



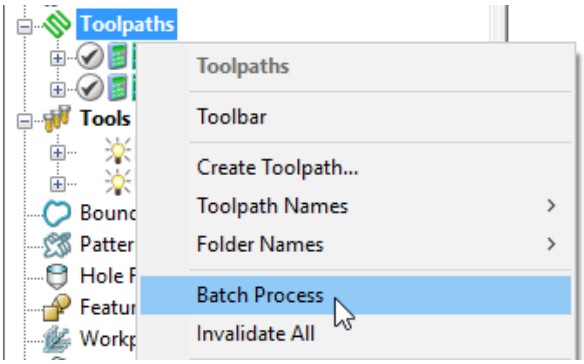

*The '**Calculator**' symbol changes to the '**Queue**' symbol while the toolpath is being processed.*

## 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**.



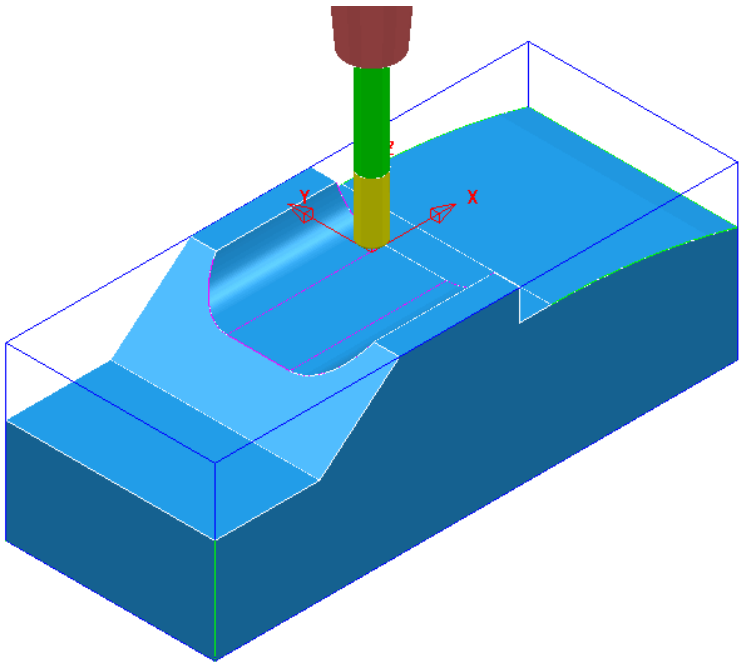

# Toolpath Processing Queue

The **Toolpath** - **Queue** option allows you to continue working in **PowerMill** while one or more toolpaths are processed 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 to work on a new **Corner finishing** strategy while the original toolpaths are being processed in the background.

1. Select **File** - **Delete All** and **Tools** - **Reset Forms**.
2. Select **File** – **Open Project**:

**.....\PowerMill_Data\Projects\ToolpathProcessing-start**

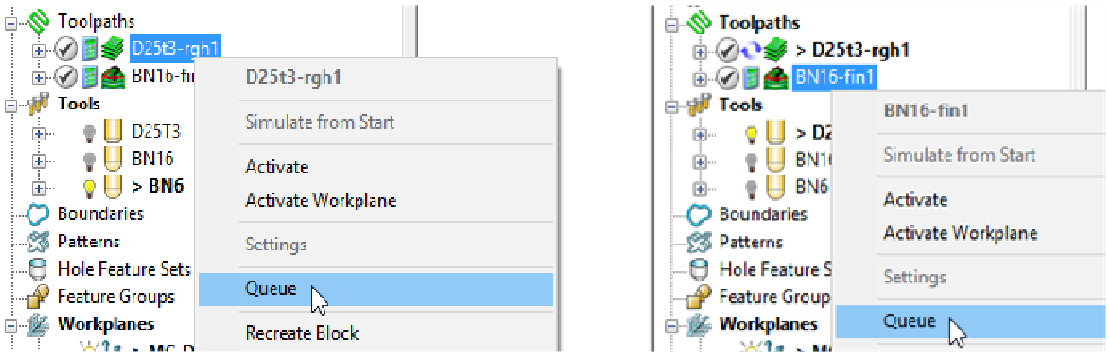


3. Select **File** - **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\ToolQueue-ex1**

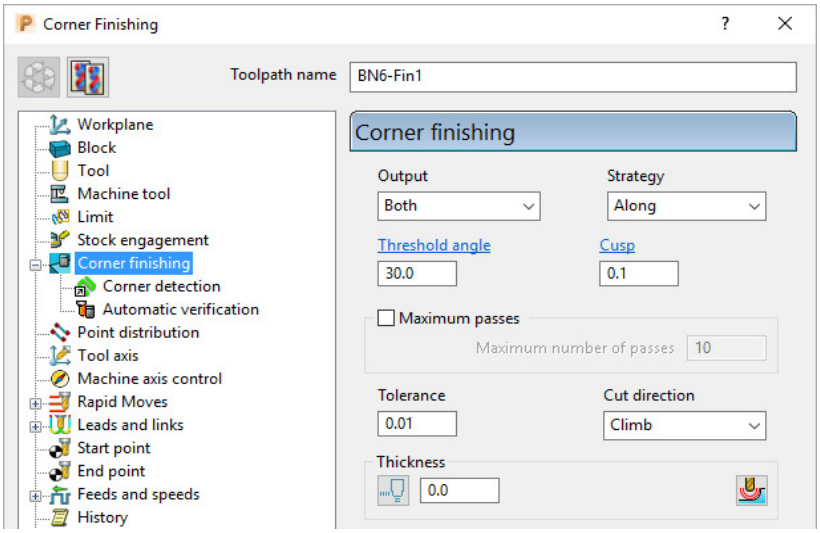
4. In the **PowerMill *explorer***, right-click the toolpath named **D25t3-rgh** and select **Queue**.
5. Repeat this on the toolpath named **BN16-fin1**.


Copyright © Autodesk

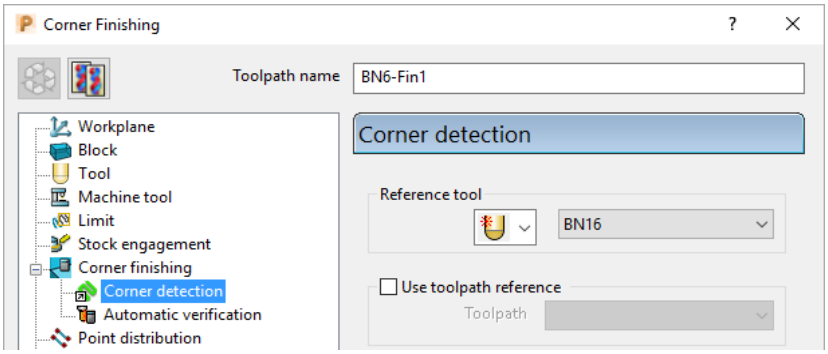



6 While the original toolpaths are being processed in the background, activate the tool **BN6**.

7 Open a **Corner Finishing** strategy and set the ***Strategy*** to **Along**. Enter the other values exactly as shown below.



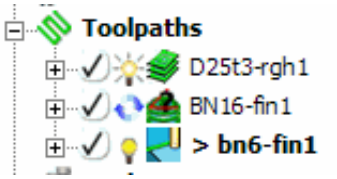

8 Select the **Corner detection** page and select the Reference tool as **BN16**

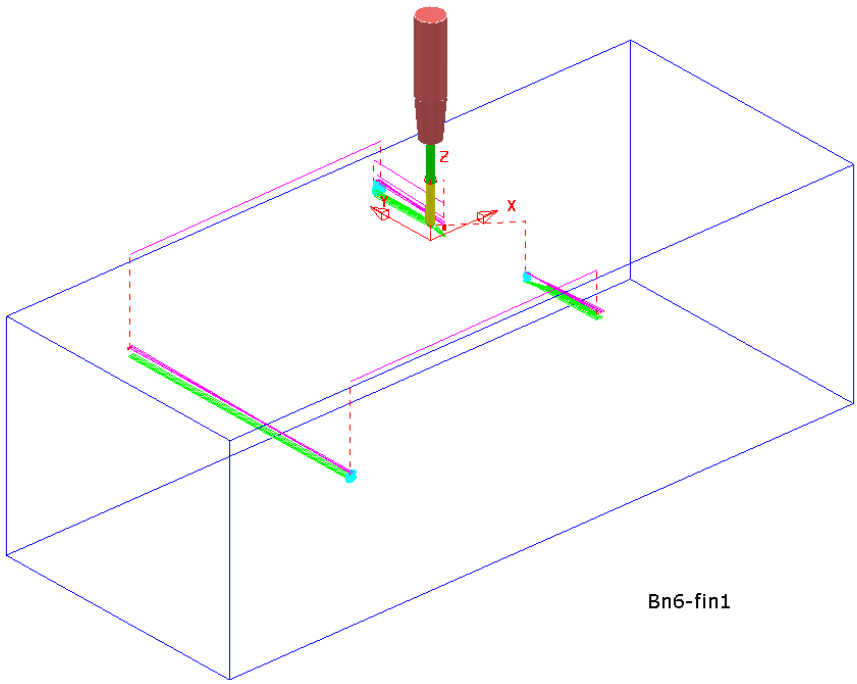



9 Click **Calculate** to process the toolpath and then **Close** the dialog.

*The latest **Corner Finishing** strategy starts processing immediately and is likely to be completed before the **Steep and Shallow finishing** strategy currently pending in the **Queue**.*

When a toolpath is added to the **Queue**, and is in an unprocessed status, the ***Calculator symbol*** on the toolpath in the **PowerMill *explorer*** changes to the **Queue** symbol .




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

**.....\COURSEWORK\PowerMill-Projects\ToolQueue-ex1**

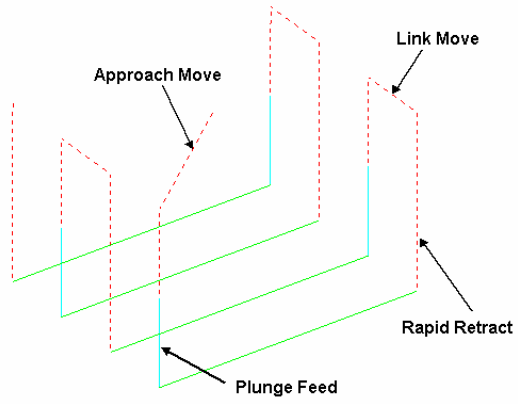

Copyright © Autodesk

# 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.

## Links using Skim

**Skim** - Provides variable control of rapid move heights within a component. These operate in conjunction with **Moves and clearances** settings to minimise slow and unnecessary air moves while machining the component form.

**Moves and clearances** – In this dialog incremental **Clearances** can be input 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 defined **Axial clearance** value (Note **Radial clearance** is also available).

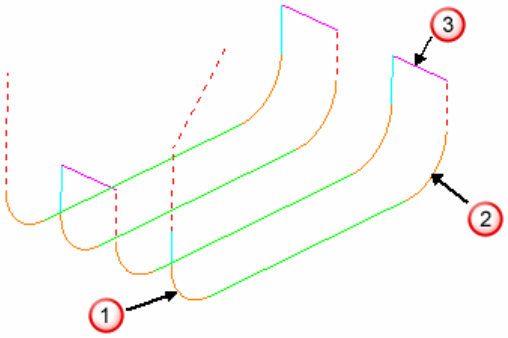
**Incremental Plunge distance** – An incremental distance above the component form where a downward ***rapid*** movement of a tool switches to a ***plunge feed*** move.

# 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:-

1 - **Lead In** = **Vertical Arc**

2 - **Lead Out** = **Vertical Arc**

3 - **Link move** = **Skim**

Toolpath colour codes:

- **Purple** - Rapid Skim Feed Rate G1
- **Pale Blue** - Plunging Feed Rate G1
- **Green/Orange** – Cutting and Lead Move Feed Rates 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 otherwise 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**.


Copyright © Autodesk

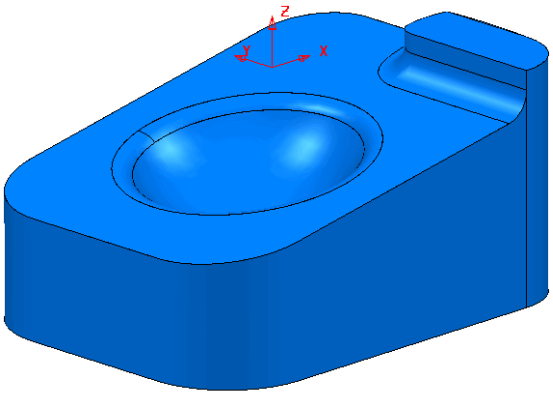
# Example

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**:

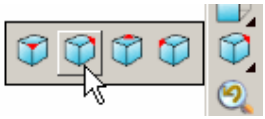
…\PowerMill_Data\Projects\LeadsLinks-Start



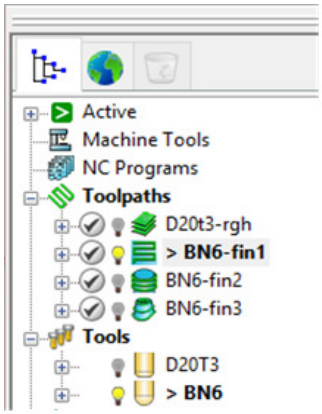
3 Select **File** - **Save Project As:**

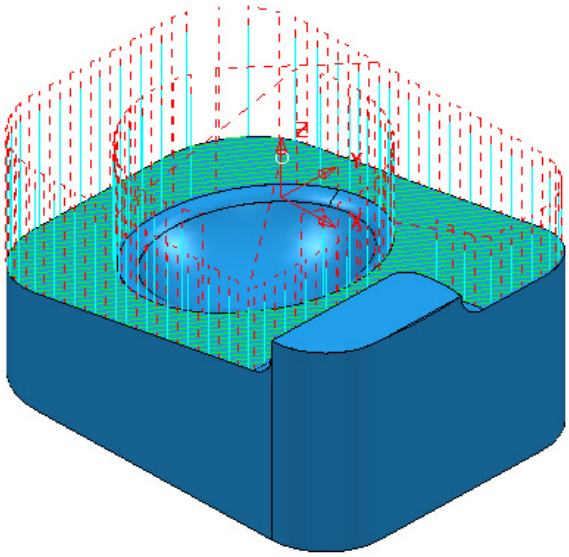
.....\COURSEWORK\PowerMill-Projects\LeadsLinks-example

4 Select an **ISO2** view.



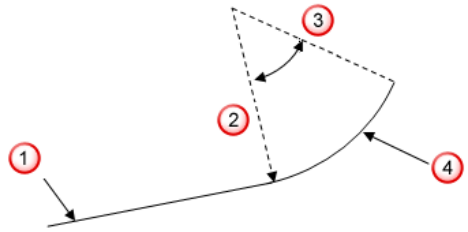
5 Activate the toolpath, **BN6-fin1**.



# 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.



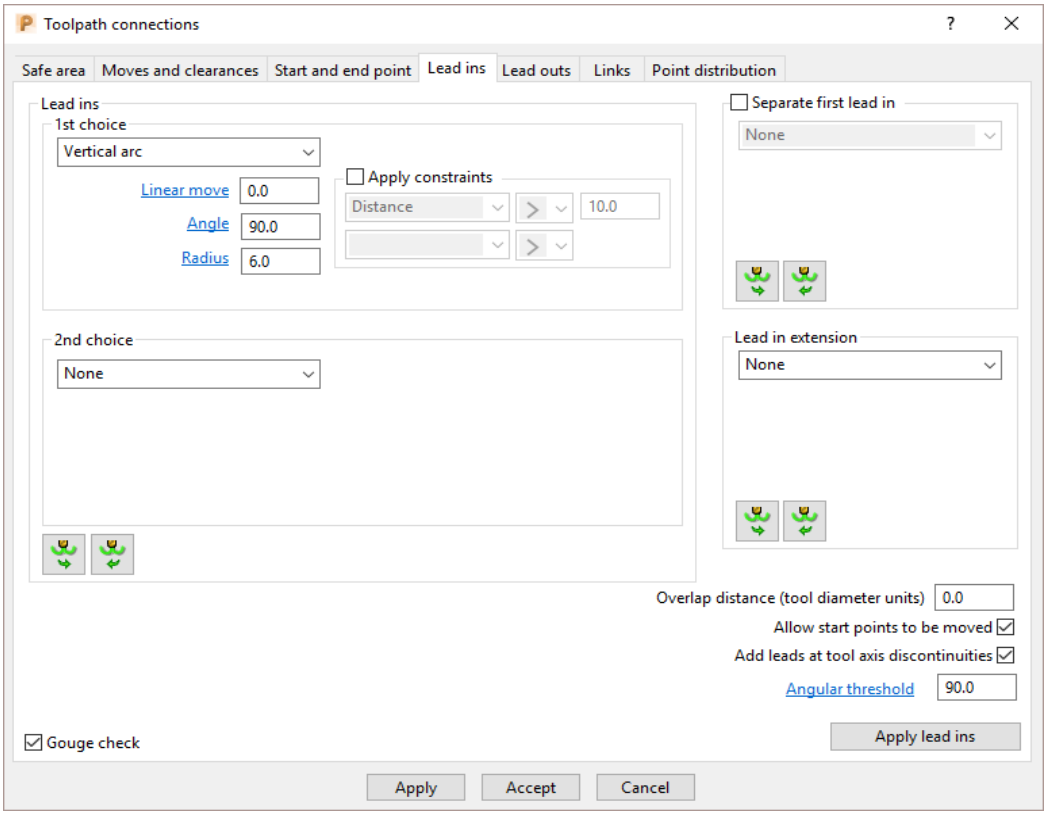

1 - Toolpath segment

2 - Radius

3 - Tangent angle

4 - Lead

6 On the main toolbar, click to open the **Toolpath connections** dialog and the **Lead In** page will be open as the default.

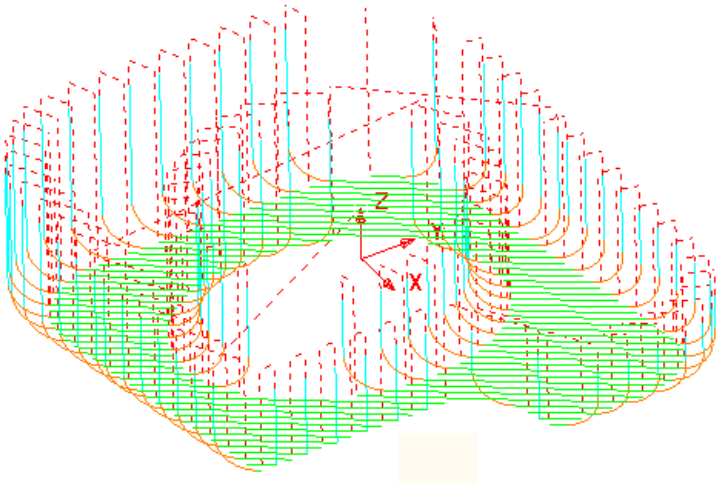



- From the ***1st choice*** list select **Vertical arc**.
- Enter an **Angle** of **90.0** and a **Radius** of **6.0**

7 Click **Apply**.


Copyright © Autodesk

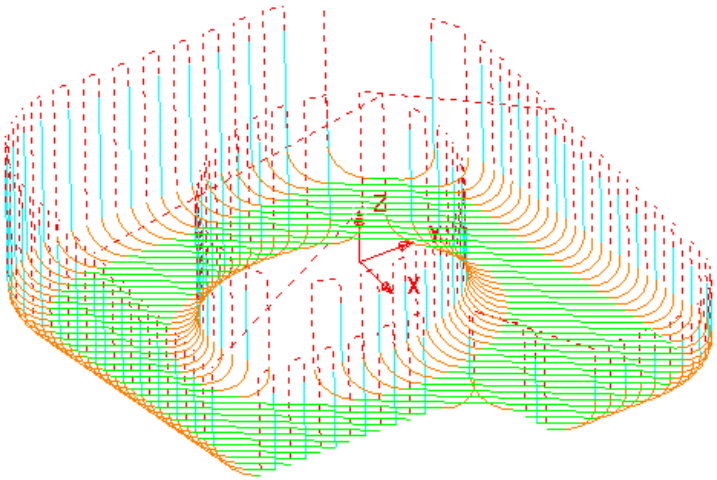
There is now a **Vertical arc** leading onto most of the tool tracks.



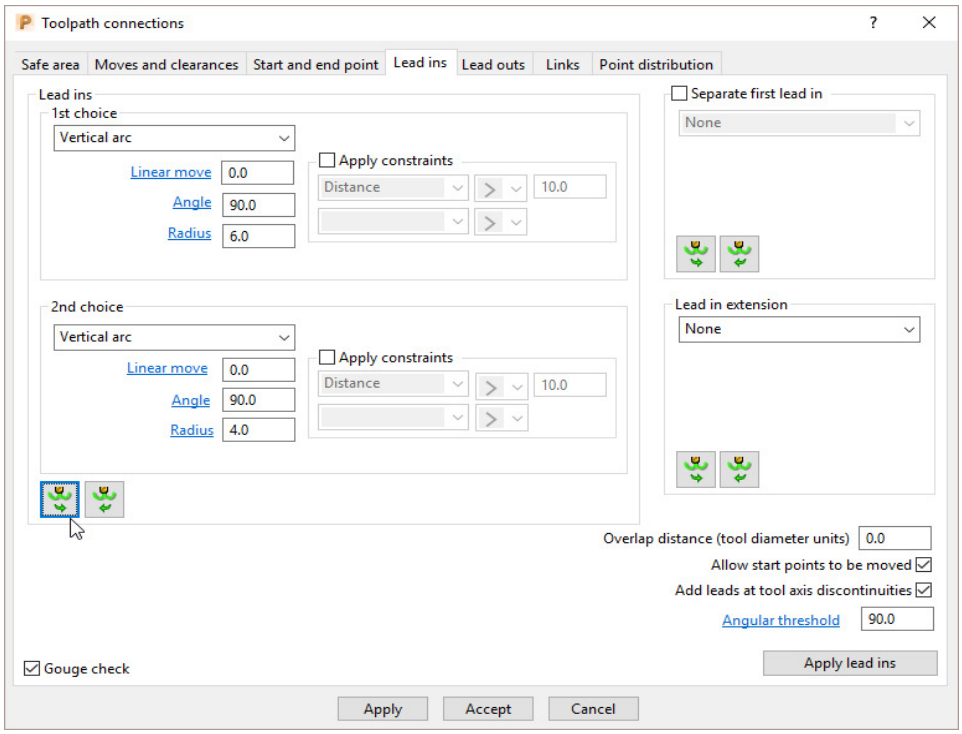

The new (gouge protected) **Vertical** leads have not been created on the tool tracks that stop close to a raised area of the model.

8 From below the ***2nd choice*** options click the ***Copy to Lead Out*** button.

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




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

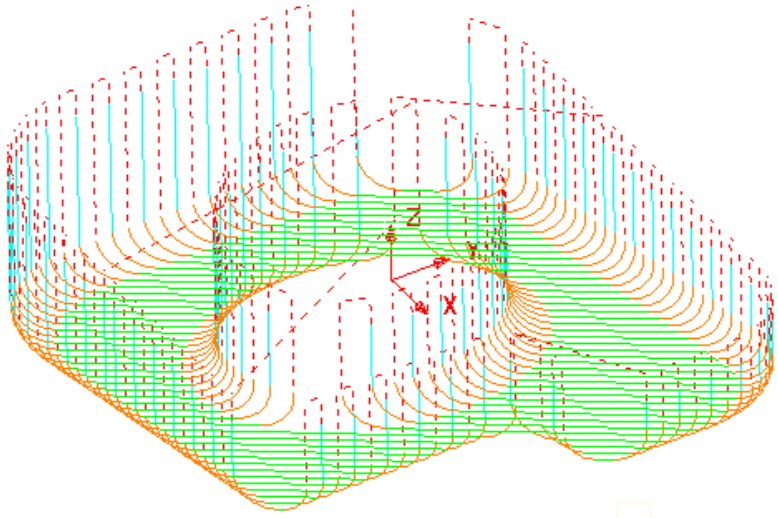



10 Select **Vertical arc** from the **2nd choice** list.

a Enter an **Angle** of **90.0** and this time a reduced value **Radius** of **4.0**

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

11 Click **Apply**.




Wherever it is gouge free on the above toolpath the **1st choice** of **Lead In/Out** with a **Vertical arc** is attempted.

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.

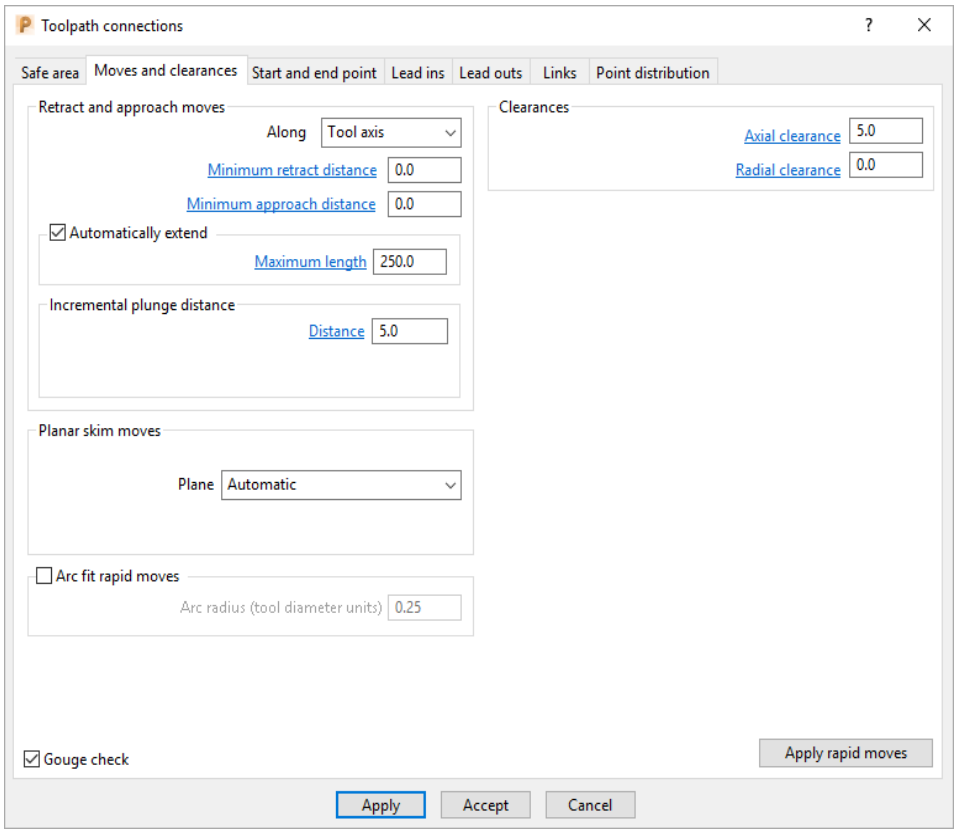

Copyright © Autodesk

# 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.

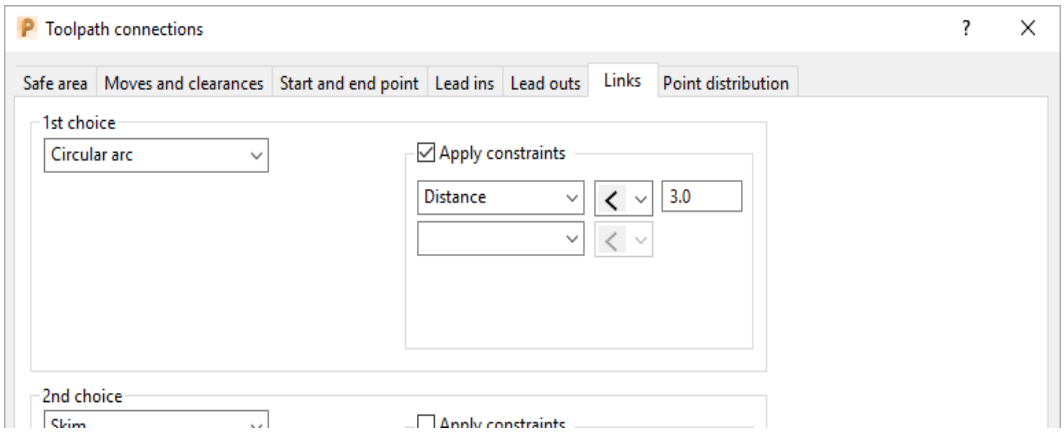
- **Apply constraints** - This allows the user to define the distance within which the **1st** or **2nd choice** toolpath **Links** apply.
- For **1st choice**, the available options include **Safe**, **Incremental**, **Skim**, **On Surface**, **Stepdown**, **Straight**, and **Circular arc**.
- For **2nd choice**, the available options include **Safe**, **Incremental**, and **Skim**.
- For **Default** (first and last links) the available options include **Safe**, **Incremental**, and **Skim**.

1. **Activate** the toolpath **BN6-fin1**.
2. Click and open the **Toolpath connections** dialog.
3. Select the **Moves and clearances** tab to view the default settings for the **Axial** and **Incremental plunge distance** options.




- **Skim** occurs at an incremental height **Axial clearance** (default **5**) for the tool to locally clear the highest point on the model between tool tracks.
- **Rapid** plunge moves change to **Plunge Feed Rate** at the **incremental plunge distance** (default **5**) above the local, model surface.

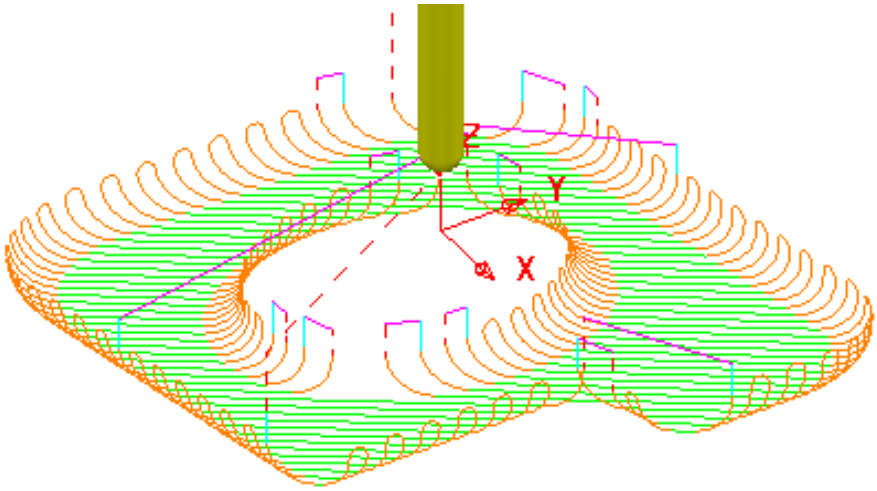
4 Select the **Links** tab.




5 Select ***1st choice*** as **Circular arc**, tick the **Apply constraints** box, and input **Distance < 3**

**Link** moves with a distance of less than the **3** threshold will use the ***1st choice*** options and those greater will use the ***2nd choice*** options.

6 Click **Apply**.

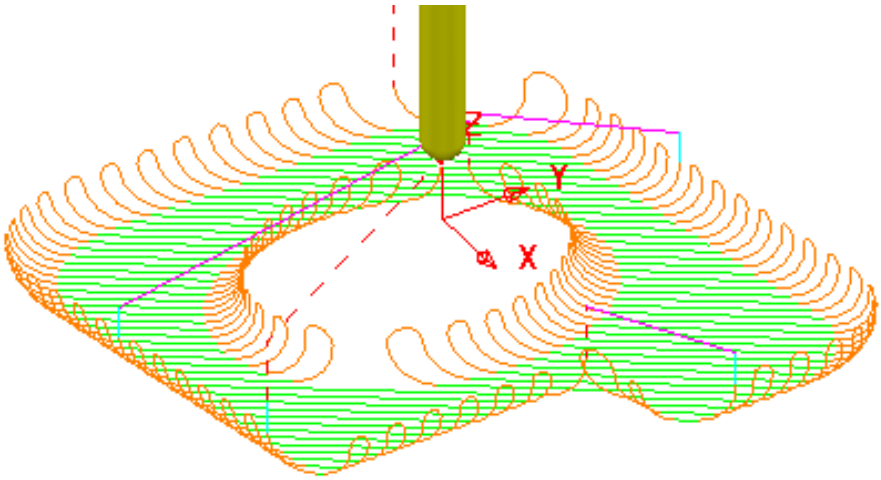



In some areas of the toolpath the distance between the ends of adjacent tool track **leads** are is greater than the **3** threshold value (In these instances **Skim** moves apply instead of **Circular arcs**).

7 For the ***1st choice*** constraints increase the threshold value to **8**.

8 Click **Apply**.


Copyright © Autodesk

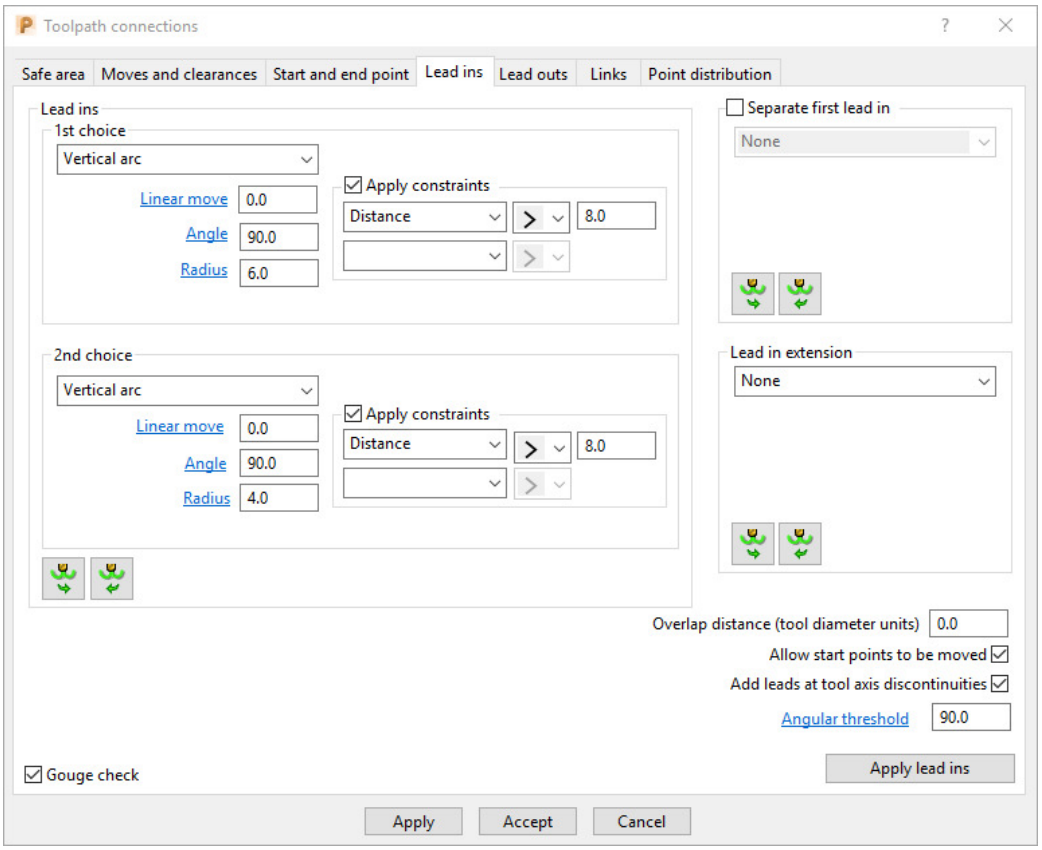



*By increasing the value of the constraints **threshold distance** to **8**, the number of occurrences of 1st choice - Circular arc links has increased.*

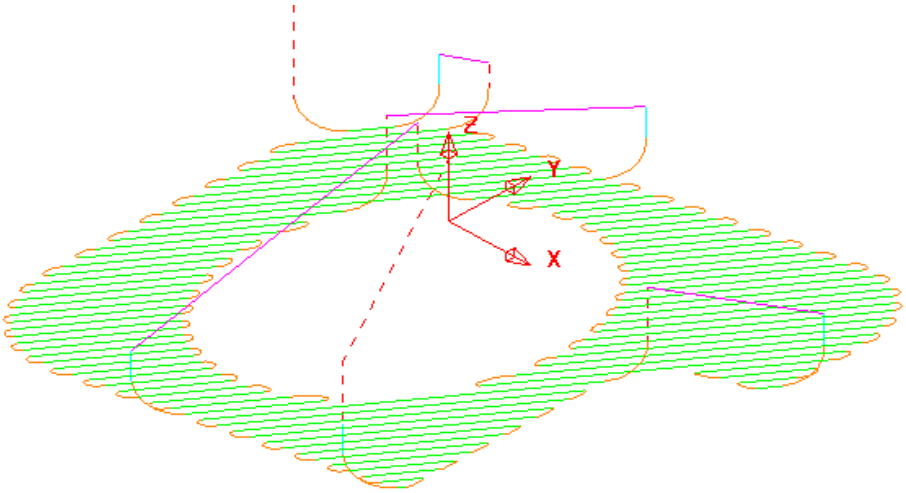
The above ***1st choice*** circular arc **Links** do not need to occur at the ends of the existing **Vertical Lead** moves. For the ***1st choice*** **Links** it will be far more efficient if the **Vertical arc**, **Lead** moves are completely removed.

9 Open the **Lead in** page.

10 For both ***1st choice*** and ***2nd choice*** tick **Apply constraints** with **Distance > 8**




11 Click **Apply**.




The ***Vertical arc*** **Lead** moves exist on the ***2nd choice*** (long) **links** but not on the ***1st choice*** (short) **links**.

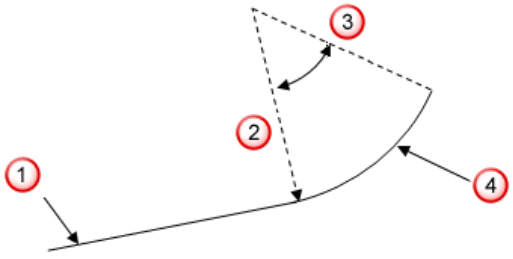
12 Select **File** - **Save Project** to update.

**.....\COURSEWORK\PowerMill-Projects\LeadsLinks-example**

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

# Horizontal Arc Leads

This type of **Lead** is frequently applied where the tool tracks run parallel along steep areas of a component.




1. Toolpath segment
2. Radius
3. Lead
4. Tangent angle

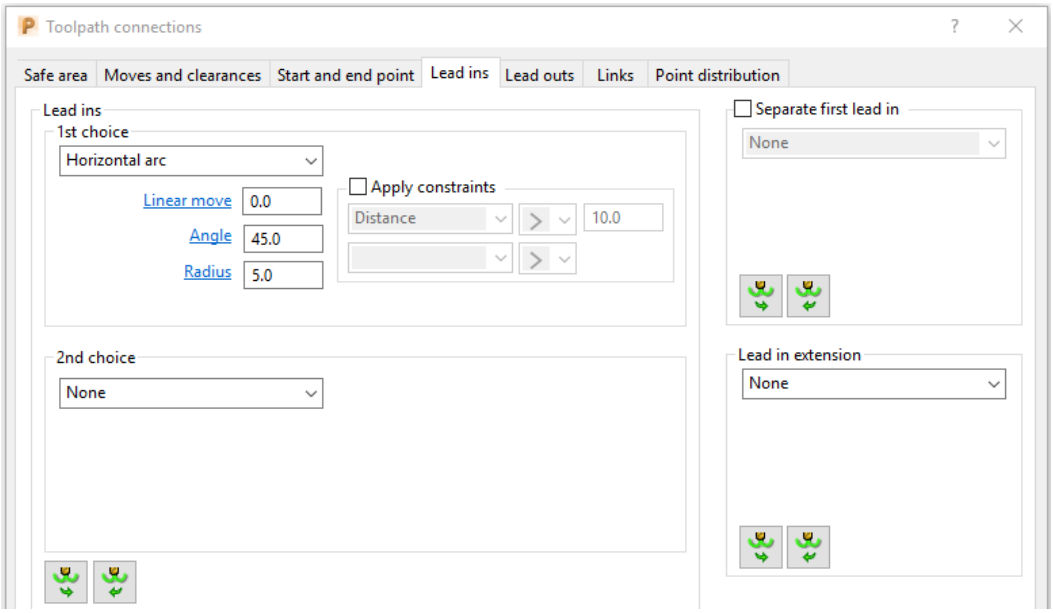

Copyright © Autodesk

13 From the main menu, select **Tools** - **Reset Forms** to return the **Leads and Links** settings to default.

14 In the **PowerMill *explorer***, right-click the **BN6-fin2 toolpath** and select **Activate**.

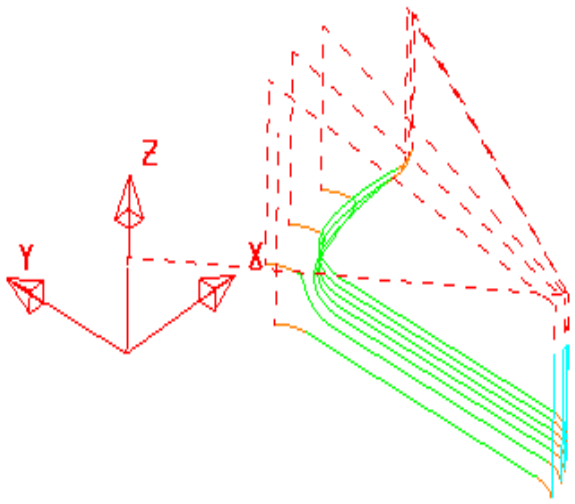
15 Select an **ISO1** view.

16 Click and open the **Toolpath connections** dialog.



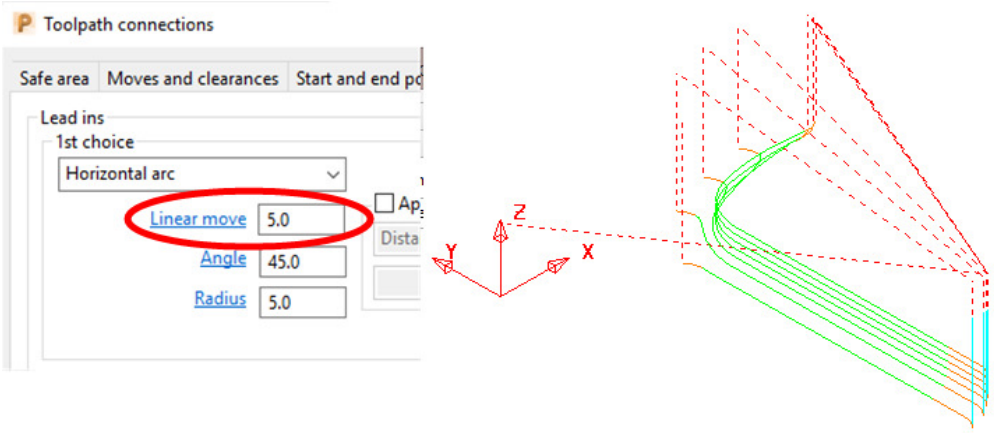

17 Select the **Lead In** tab, and set the **1st Choice** to **Horizontal arc** with an **Angle** of **45** and a **Radius** of **5**.

18 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 of the tool tracks.*

19 In the **Lead in** page ***1st Choice***, input a **Linear move** of **5** and select **Apply**.



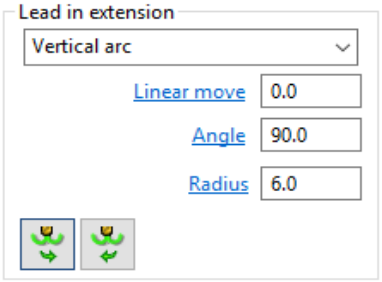

# Extensions

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

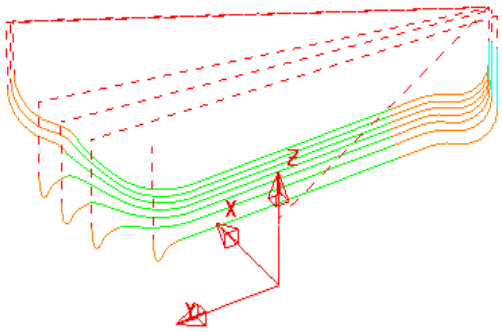
20 Select an **ISO 4** view.

21 In the **Toolpath connections** dialog, select the **Lead in** page, and locate the **Lead in extension** area.

22 For the **Lead in extension** use a **Vertical arc** with **Angle 90.0** and **Radius 6.0**




23 Click on the ***Copy to outward extension*** icon and Click **Apply**.

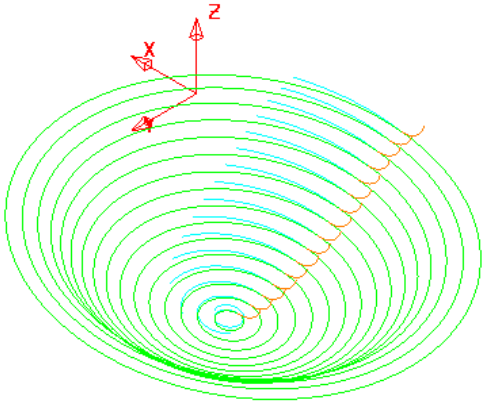



A **Vertical Arc** - **Extension** is added to all of the **Lead In/Out** moves.


Copyright © Autodesk

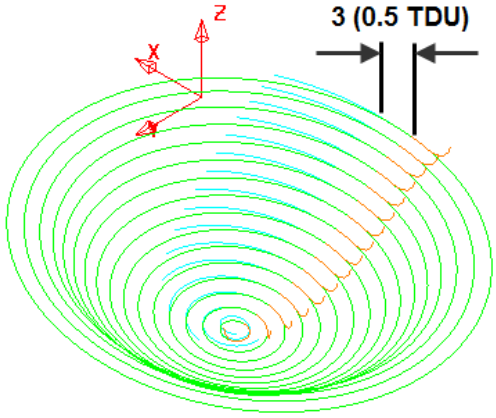
# Exercise

1. From the main menu, select **Tools** - **Reset Forms** to return the **Leads and Links** settings to default.
2. Activate the **BN6-fin3** toolpath and **Apply** the following **Leads and Links**:
   - c **Links**: **1st choice**/**2nd choice**/**Default** – **Skim**.
   - d **Lead in**: **1st Choice** - **Ramp** using the **Ramp Options** , ***Follow*** **Toolpath**, ***Max Zig Angle*** **4**, and ***Ramp height*** using ***Type*** **- Incremental** at a **Height** of **1**.
   - e **Lead out**: **1st Choice** - **Vertical Arc** with **Angle 90** and **Radius 3**




The **Links** have been visually removed to enable a clearer view of the **Leads**.

3. In the **Leads and Links** dialog select the **Lead in** page and input **0.5** in the box named **Overlap distance** (Located in the lower right corner of the dialog).
4. **Apply** the **Toolpath connections** 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).

5. Select **File** - **Save Project** to update.

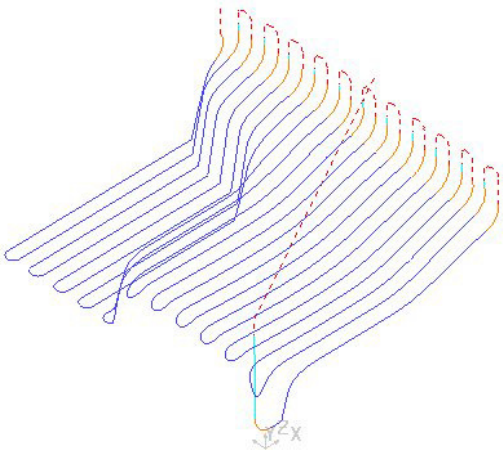
**.....\COURSEWORK\PowerMill-Projects\LeadsLinks-example**

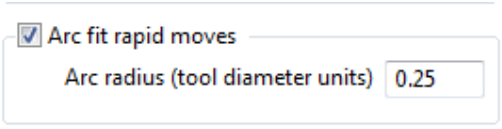
# Arc fit rapid moves

## Leads and Links

This is ideal for applications where you want to avoid all sharp changes of direction including the none machining part of the link moves. **Arc fit rapid moves** applies to both ***Rapid*** and ***Skim*** moves

*Not all machine tool controllers will support this option.*






- The **Arc Fit Rapid Moves** option is located in the **Toolpath connections** dialog – **Moves and clearances** page.
- The **Arc fit rapid moves** box must be ticked.
- The **Arc radius** is a proportion of the active, Tool Diameter (TDU).

## Other Information for Toolpath connections

***Toolpath connections*** *options such as* ***Leads*** *and* ***Links*** *can be applied independently to one or more selected* ***Link*** *moves.*

*The very first downward move onto a toolpath will default to use a* ***Plunge feed rate*** *all the way from the absolute* ***Plunge Height****. To make this work incrementally relative to the model open the* ***Start and end point*** *page and tick the* ***Incremental plunge*** *box.*


Copyright © Autodesk

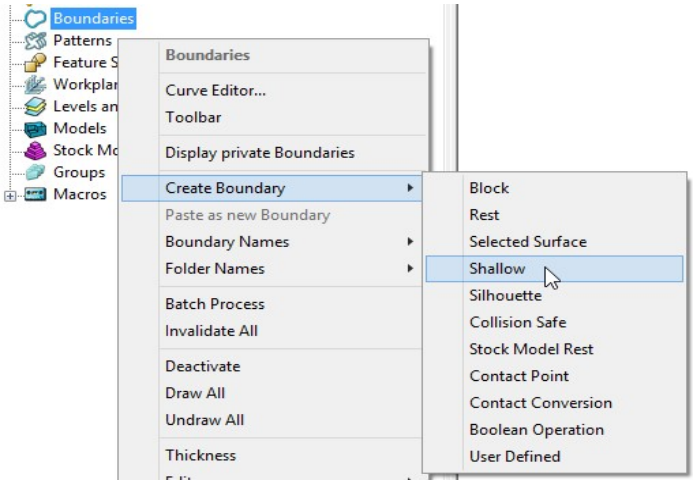
# 6. Boundaries

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.

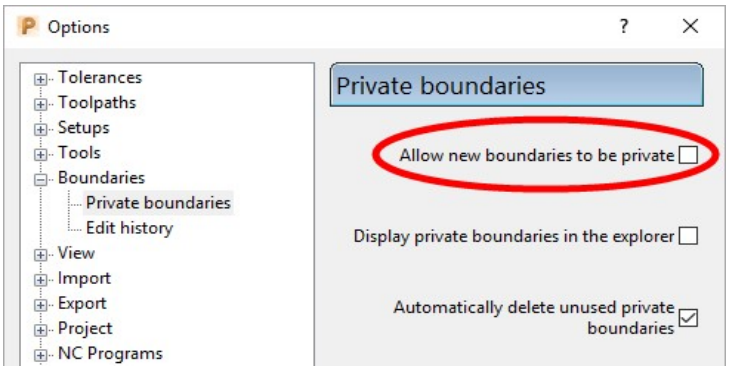
For example, **Constant Z finishing** is more effective on steep sidewalls and Raster finishing on shallow areas.

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.




## 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 browse to **Boundaries** – **Private boundaries**.

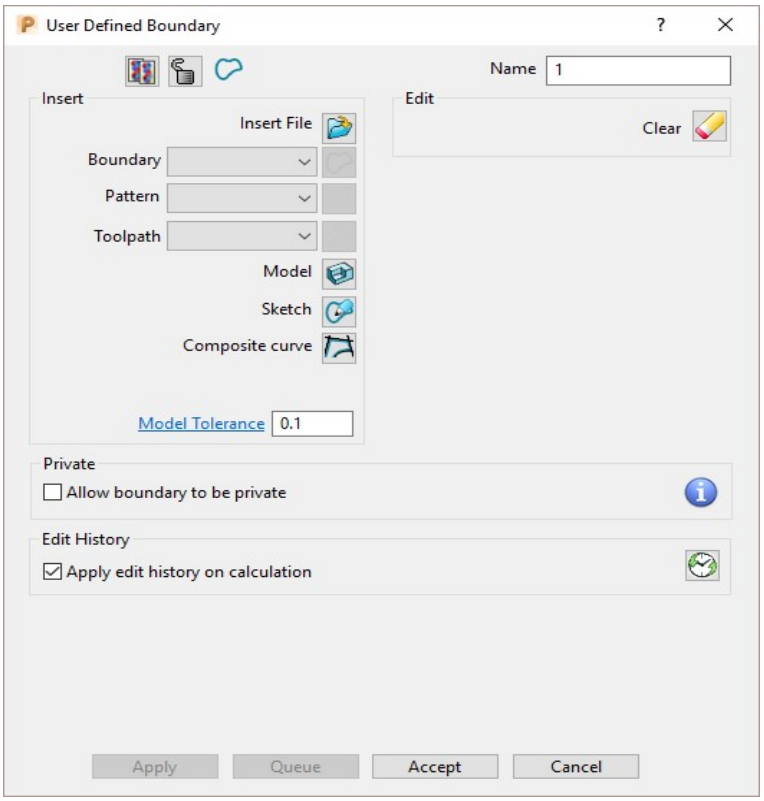



3. Un-tick the box named **Allow new boundaries to be private**.
4. Select **Accept** to close the **Options** dialog.

# 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.




**Insert File** - Opens the browser to locate an externally stored geometry file.

The Boundary, Pattern, and Toolpath buttons are disabled unless a compatible entity is selected in the adjacent input box.

**Model** - Inserts the edge of the selected **model**.

**Sketch** - Opens the **Curve Editor** toolbar.

**Composite curve** - Opens the **Create Composite Curve** toolbar.

**Wireframe Modelling** - Provides access to the **PowerShape** wireframe constructor.

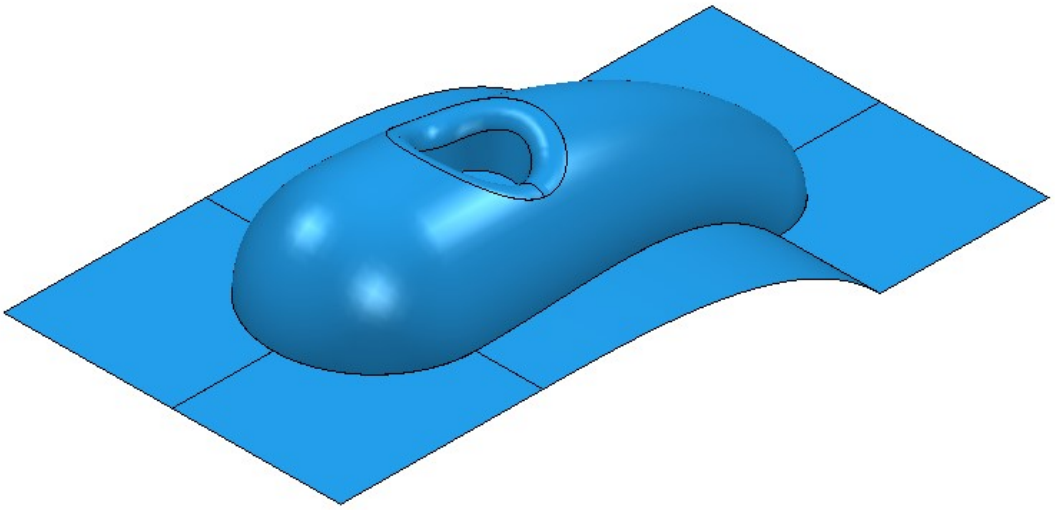
5 Select **File** - **Save Project As:**

**.....\COURSEWORK\PowerMill_Projects\BoundaryBasics**

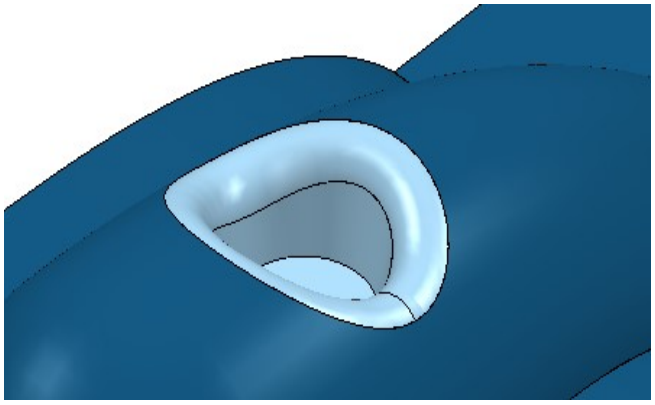
6 Select **File** - **Import Model...**

**.....\PowerMill_Data\Models\cowling.dgk**


Copyright © Autodesk



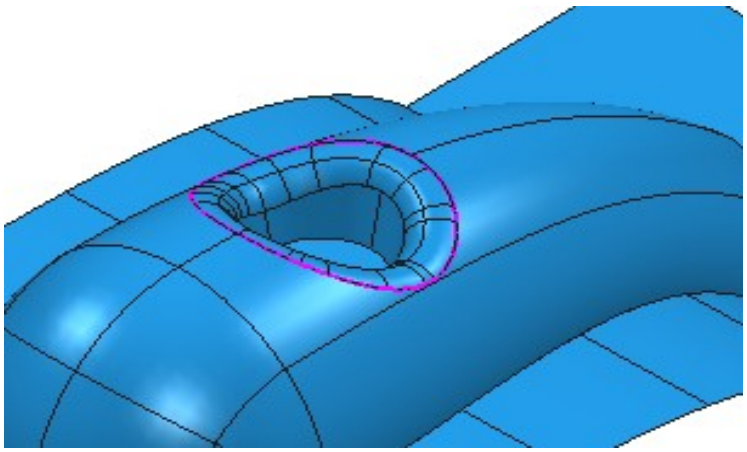
7 Select the surfaces defining the central pocket and fillet.



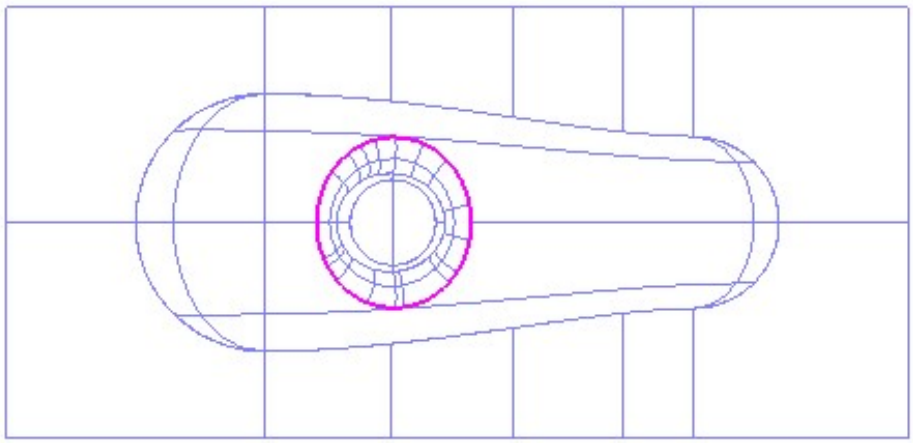
8 In the **PowerMill *explorer***, right-click **Boundaries** and select **Create** - **User Defined**.

9 In the User **Defined Boundary** dialog, click the **Model** button.

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



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



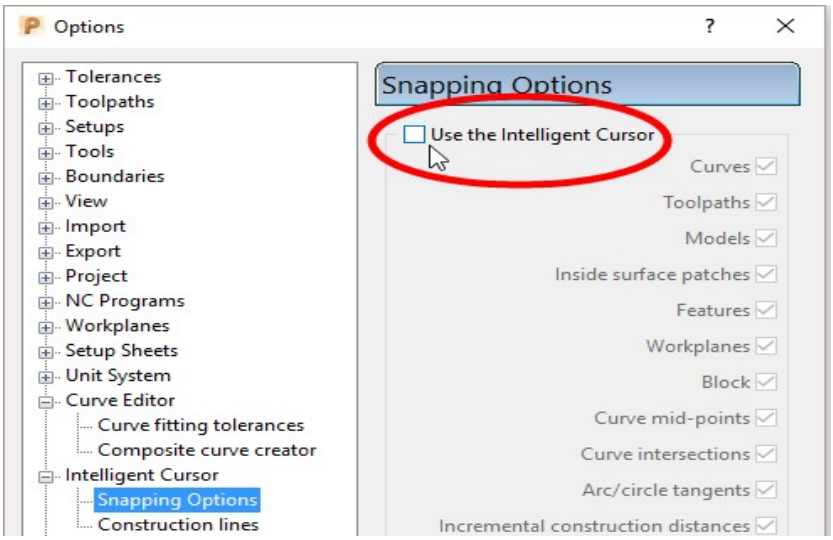
11 In the **PowerMill *explorer***, right-click **Boundaries** and select **Create Boundary** - **User Defined**.

12 In the **User Defined Boundary** dialog, click the **Sketch** button

to open the **Curve Editor** toolbar.

13 On the **Curve Editor** toolbar, click the following button to open the **Options** dialog.

14 Un-tick **Intelligent Cursor** - **Snapping Options** - ***Use the Intelligent Cursor*** to enable 'free form' sketching.

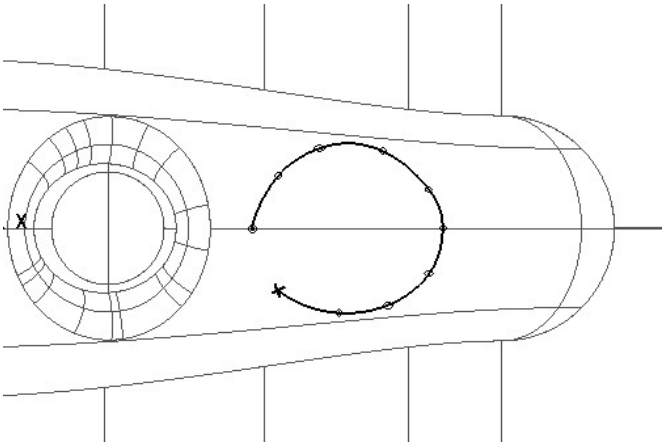



15 Click **Accept** to keep the changes.

16 From the **Curve Editor** toolbar, select **Create a Bezier curve** .

17 Using the left mouse key, click points within the model to sketch an oval shaped **Boundary**.


Copyright © Autodesk



*If required, use **Undo** to sequentially remove any invalid points.*

*While the **Intelligent cursor** is switched off do not attempt to close the segment by trying to snap the 'final span' back on to the start point.*

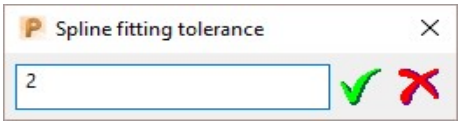
18 On the **Curve Editor** toolbar click to open the **Options** dialog.

19 Select **Intelligent Cursor** - **Snapping Options** and tick - **Use Intelligent Cursor**.

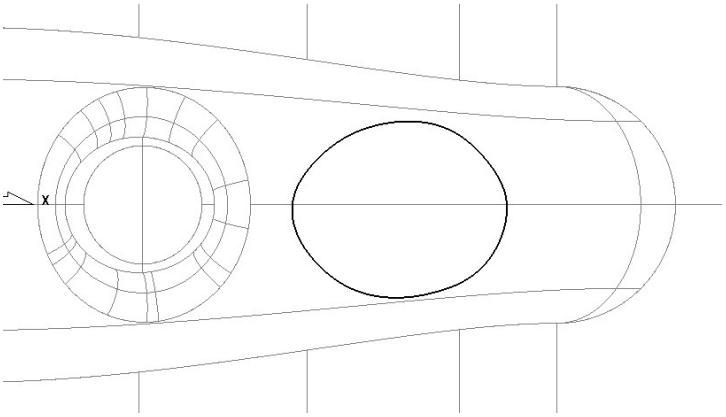
20 Then snap the final span of the **Bezier Curve** onto the original start point.

21 On the **Curve Editor** toolbar click to accept the changes and close the session.

22 If the **Boundary** segment is not smooth enough, right mouse click on it and from the local menu, select **Edit** - **Spline Selected** to open the **Spline** fitting tolerance dialog.




23 Enter the value **2** and click to apply and close the dialog.



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

In instances where the required deviation would be greater than 2, a tangency will remain either side of the affected points.

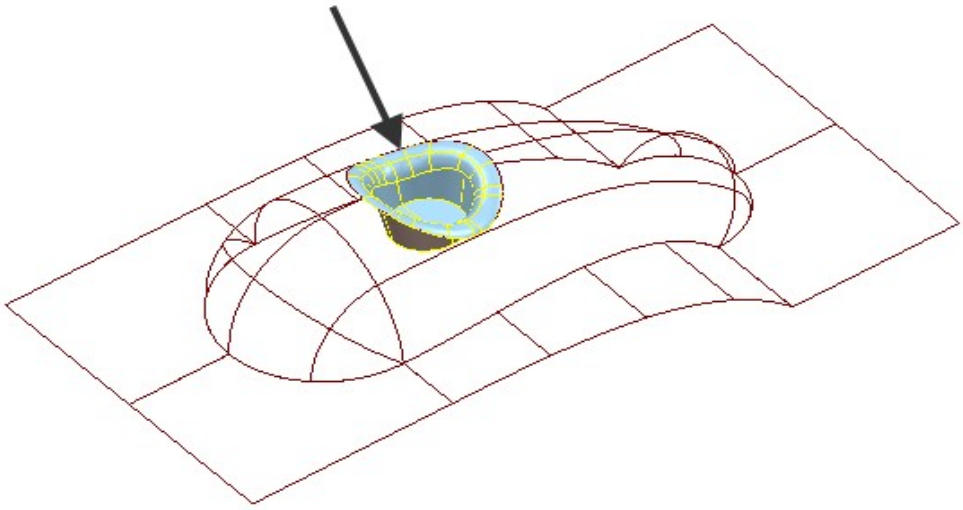
# Selected Surface Boundary

A ***Selected Surface*** **Boundary** defines one or more segments around where the active tool would fully machine the selected surfaces. The segments represent the tip position of the active tool.

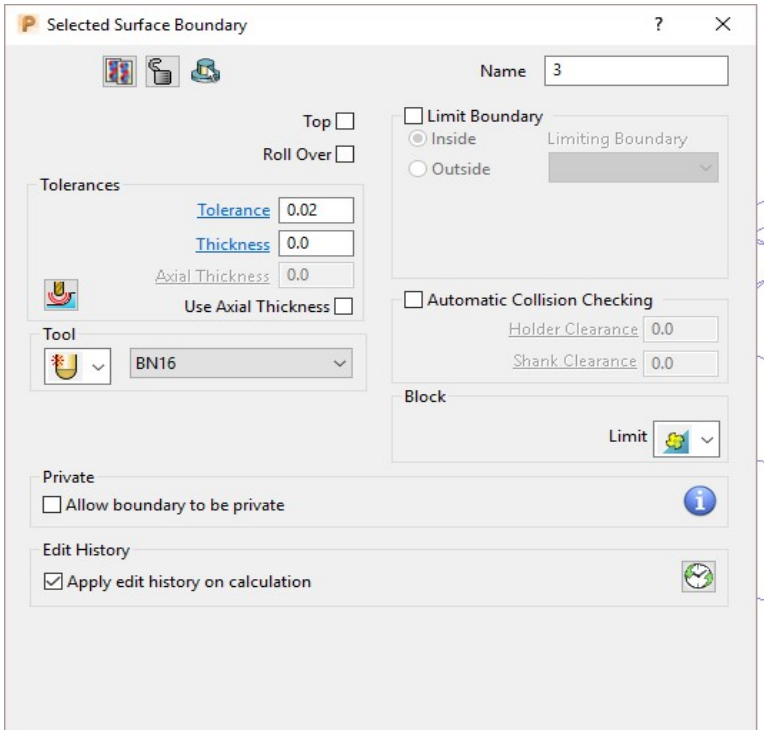
1 Create a **Block** - **Defined by** - **Box** using **Type** - **Model**.

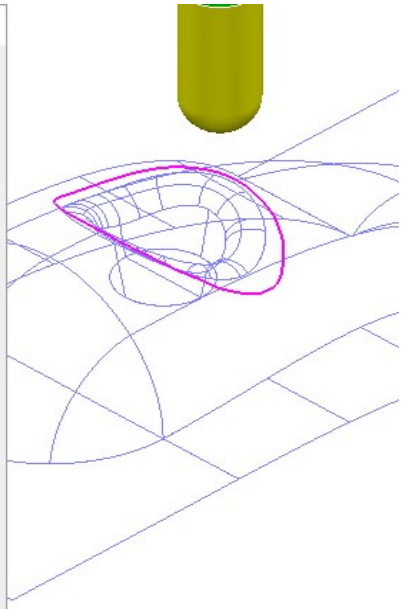
2 Create a **Ball Nosed** tool of diameter **16** with the Name **BN16**.

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**.



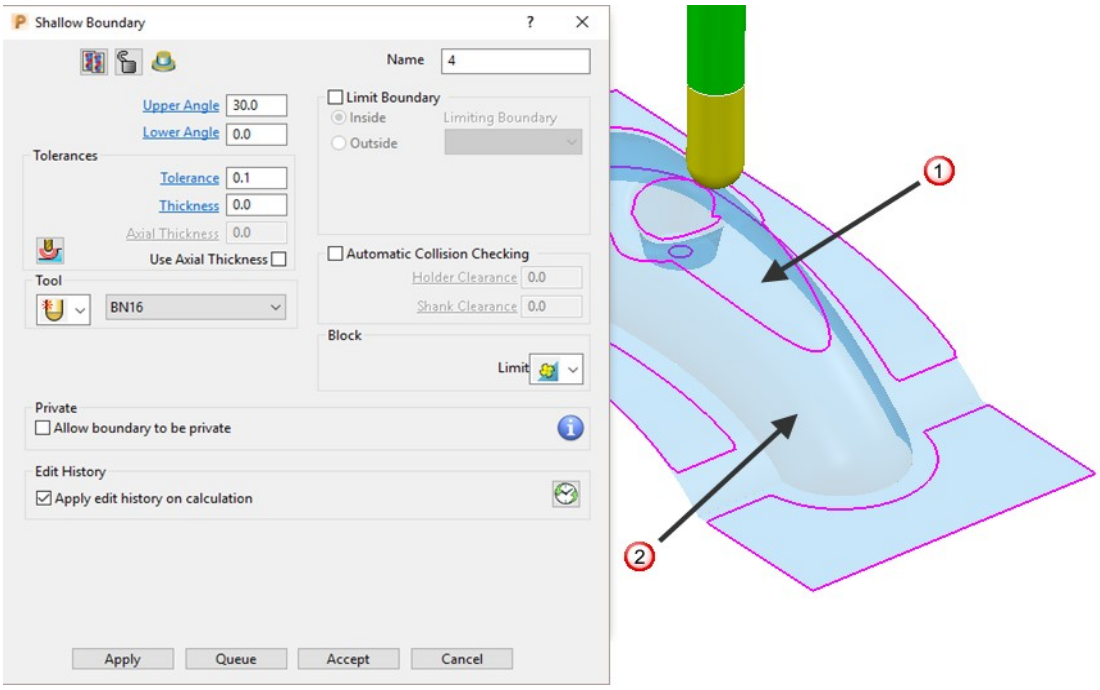
5 Using the above settings, click **Apply** to create a single segment, **Selected Surface Boundary**.


Copyright © Autodesk

# 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 allowing appropriate machining strategies to be applied where they are most effective . The **boundary** is calculated relative to the active **Tool** and **Thickness** parameters.

6 In the **PowerMill *explorer***, right-click on **Boundaries** and using the default settings, **Create Boundary** - **Shallow**.




1 - ***Shallow*** area more suited to **Raster Finishing** (above 30 degrees).

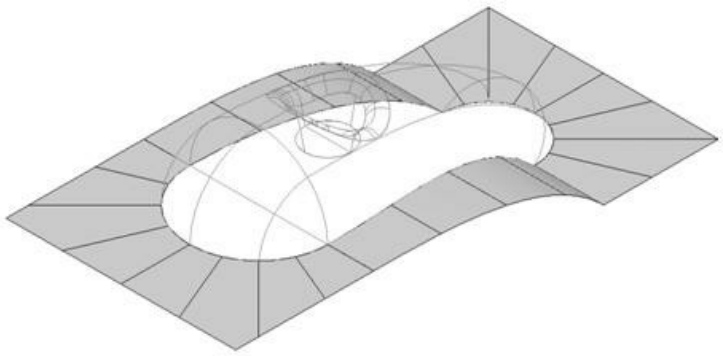
2 - ***Steep*** area more suited to a **Constant Z** Finishing (below 30 degrees).

7 **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.

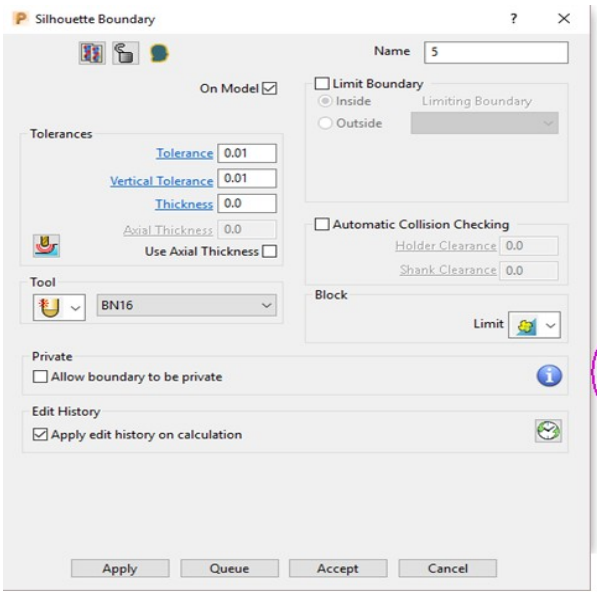
1 Select the following surfaces on the model:

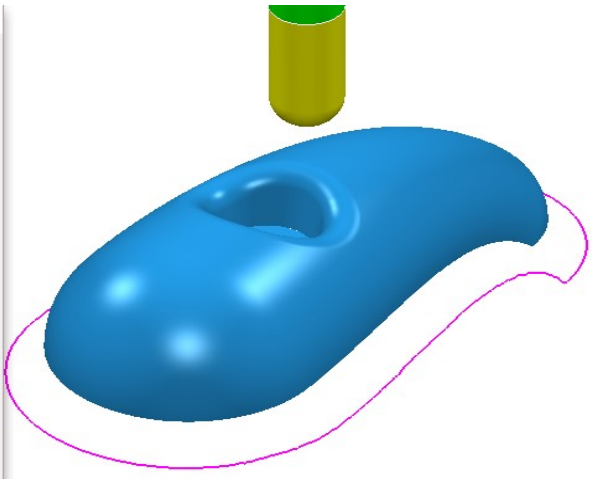


*Use the Shift key on your keyboard to select multiple surfaces.*

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**.



# Rest Boundary

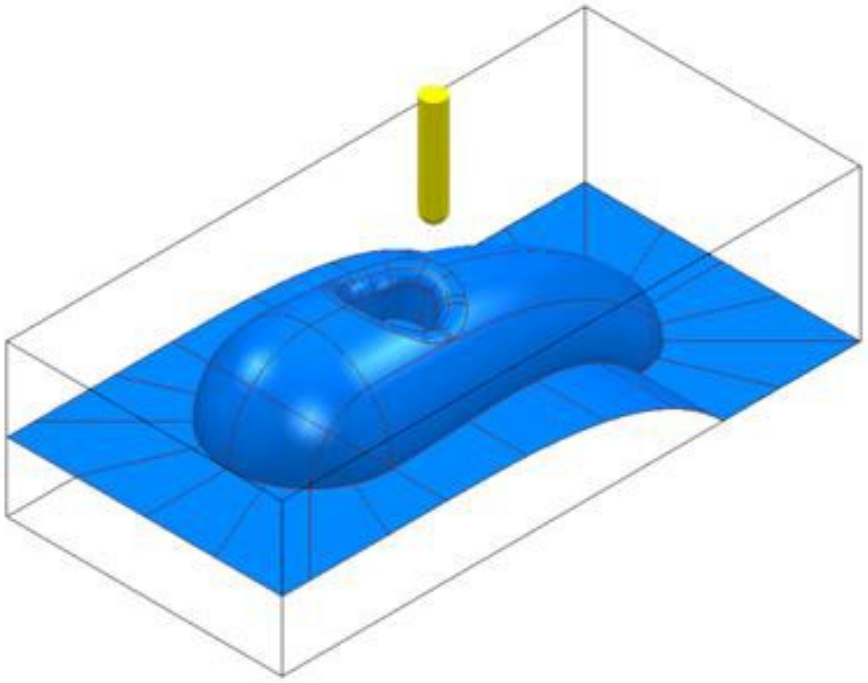
A **Rest Boundary** defines the area inaccessible to a specified reference tool. It also requires a smaller active tool to be specified otherwise it will not generate segments.

1 Create a **Ball Nosed** tool of diameter **8** with the ***Name*** **BN8**.

2 In the **PowerMill *explorer***, right-click **Models** and select **Delete All** to remove the incomplete component.

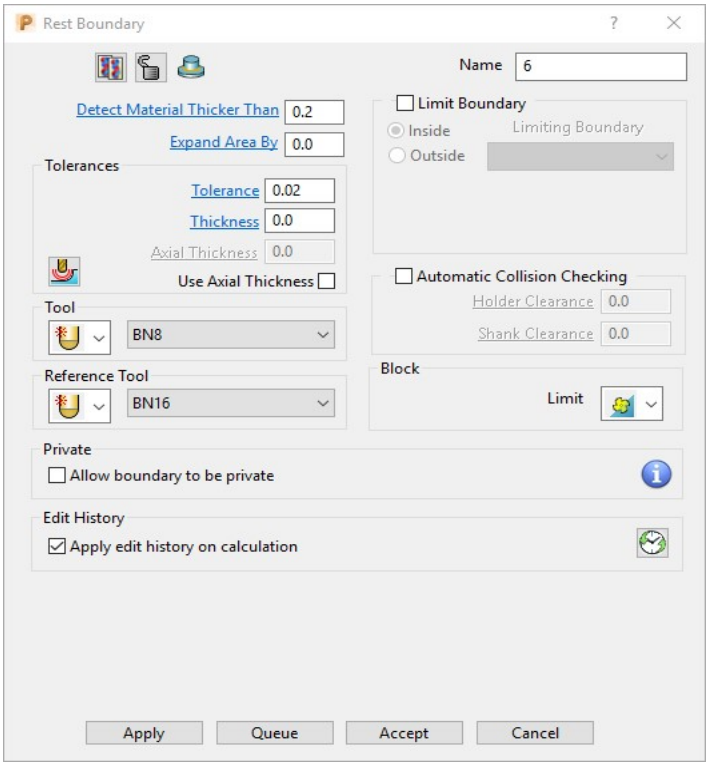
3 **Import** the original **Model** back into the **Project**:

**.....\PowerMill_Data\Models\cowling**




Copyright © Autodesk

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




5 Input **Expand Area 0**, select **Tool BN8**, and **Reference Tool BN16**.

6 Set with **Limit tool centre to block edge**.

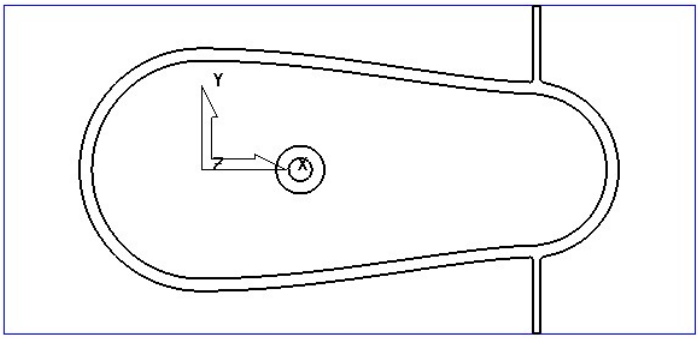

7 Click **Apply** to create the following **Rest Boundary**.

8 Click **Accept** to close the dialog.

9 **Save** the **Project** to update:-

**.....\COURSEWORK\PowerMill_Projects\BoundaryBasics**

10 Select a **View from top** and un-draw the model.




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

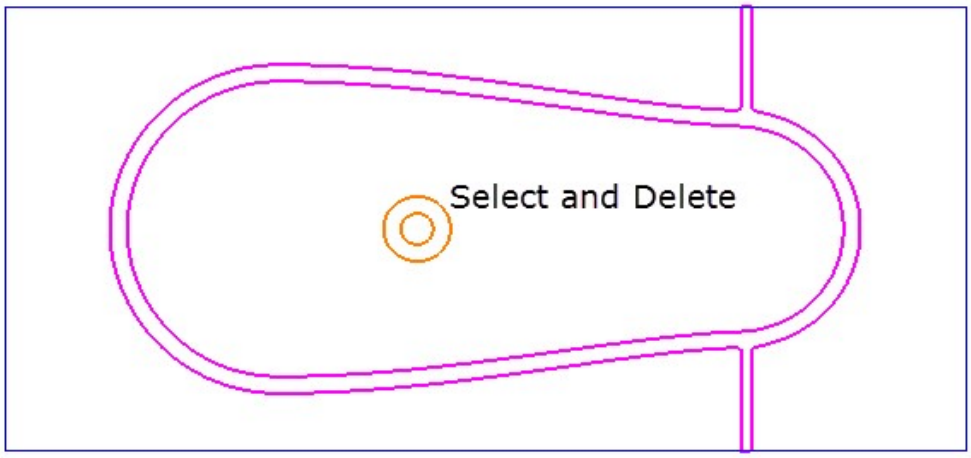
On closer inspection, the central pocket area would be more effectively machined with a flat bottomed tool and a smoother, more continuous toolpath, can be created if the 2 spurs are removed from the outer segment (to be machined later as a separate toolpath).

Copyright © Autodesk

# 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.

6. Click on the **Boundary** toolbar to activate the **Curve Editor** toolbar.

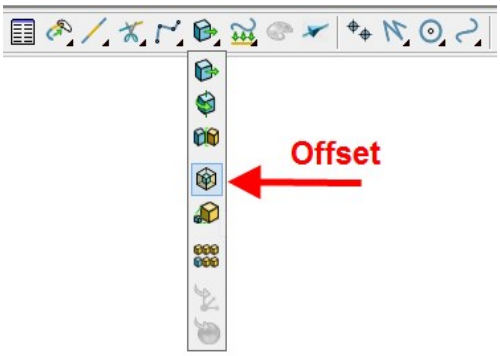
*You can also double-click a **boundary** segment to activate the **Curve Editor** Toolbar.*


Copyright © Autodesk

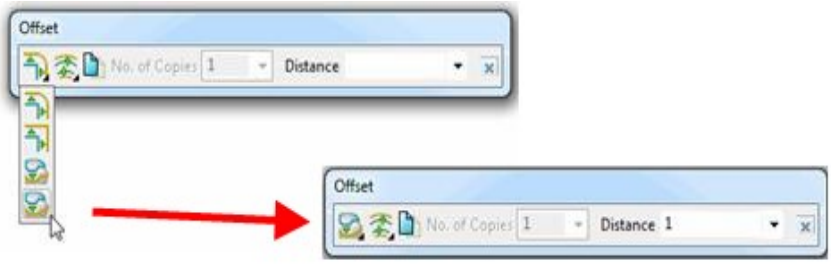
# Edit > Transform

7 In the **PowerMill explorer**, right-click on the ***boundary*** **Rest** and from the local menu, select **Curve Editor**.

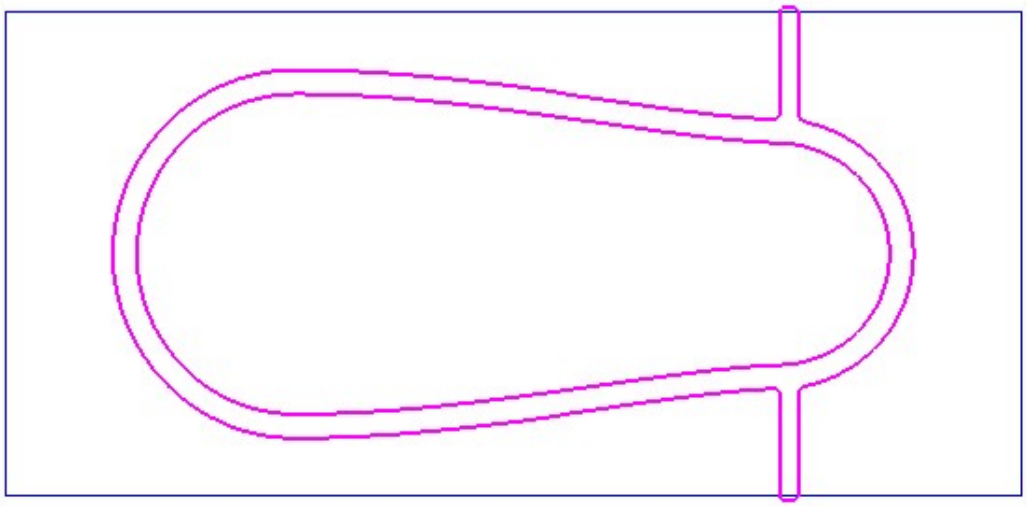
8 In the **Curve Editor** toolbar select the **Offset** option




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




10 Click in 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 Editor toolbar .

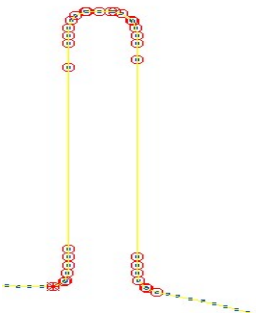
1 In the PowerMill explorer, right-click **Boundaries** and from the local menu select **Toolbar**.

2 On the **Boundary** toolbar, click the **Curve Editor** button to display the **Curve Editor** toolbar.

3 Select (Left click) the outer **boundary segment** in the graphics area.

The points are displayed on the selected boundary segment.

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

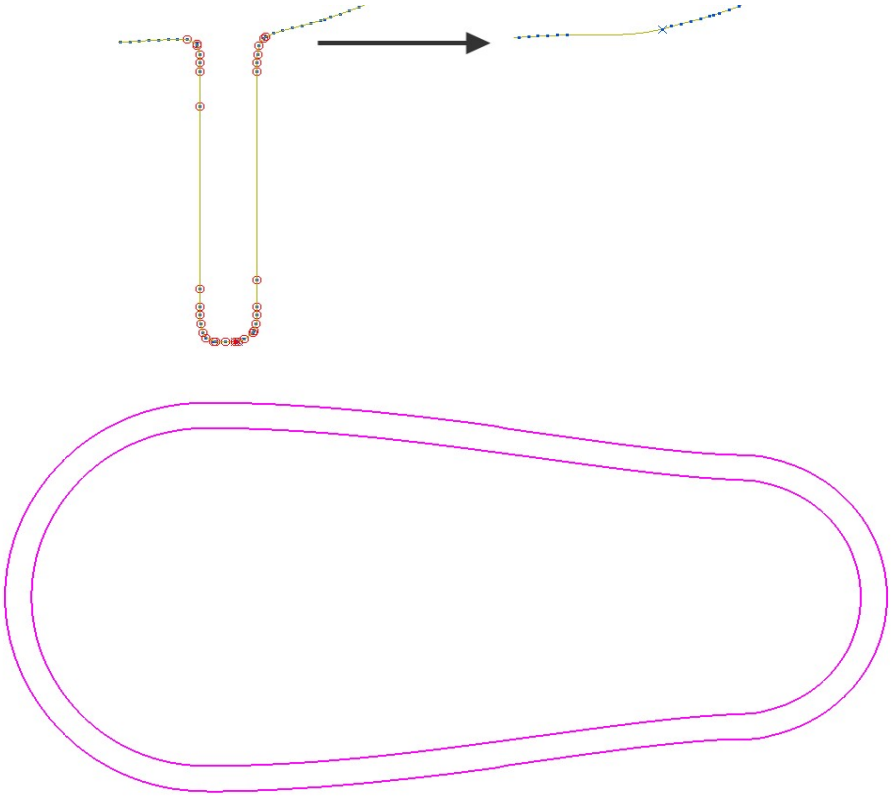


*To select multiple points, hold down the **Shift key** while individually selecting or dragging across the points with the mouse.*

5 On the **Curve Editor** toolbar, click **Delete** points to remove the spur from the segment.

6 Repeat the procedure on the other spur.

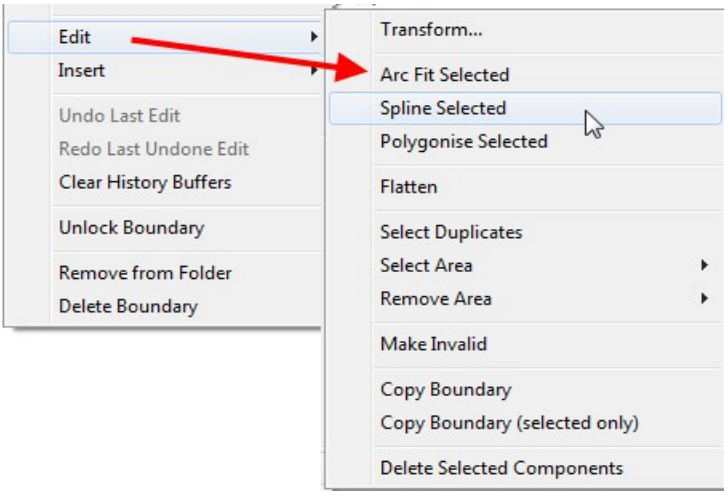

Copyright © Autodesk



7 Click , to accept changes and exit the **Curve Editor**.

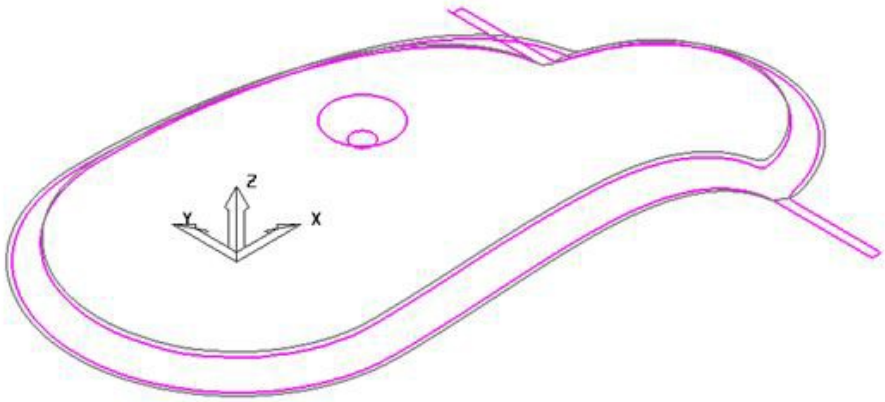
## Other boundary editing options

Right-click on a **Boundary** in the graphics area or in the **PowerMill e*xplorer*** 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.

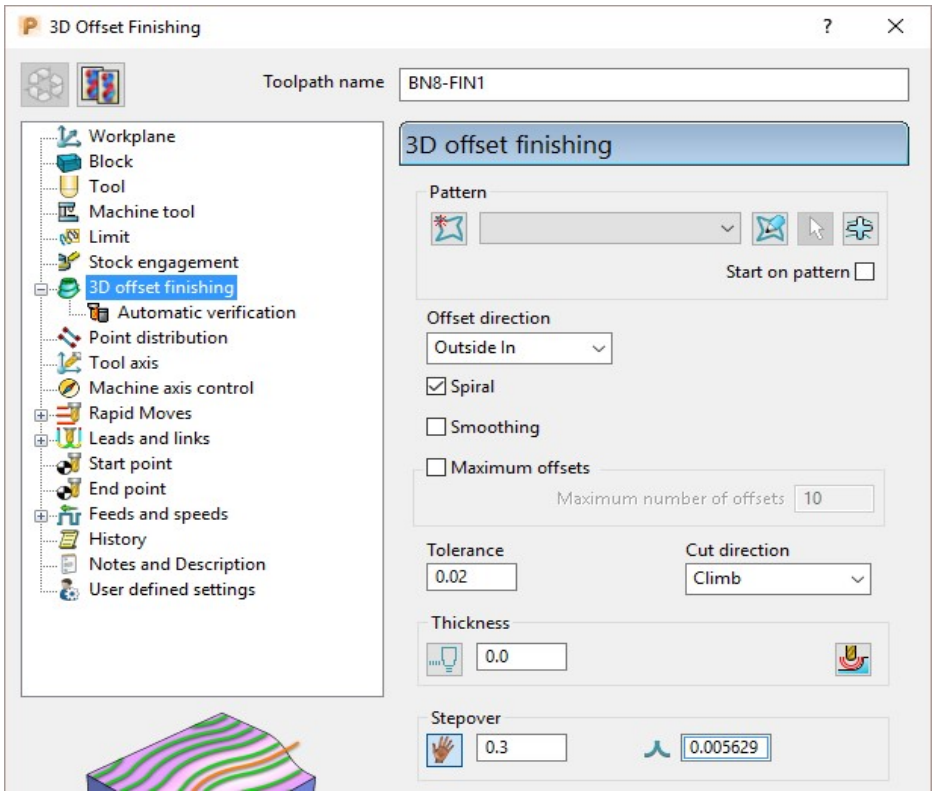



## Create a 3D Offset strategy limited to the modified Boundary

9 **Activate** the **BN8** tool and **Boundary** named **Rest**.

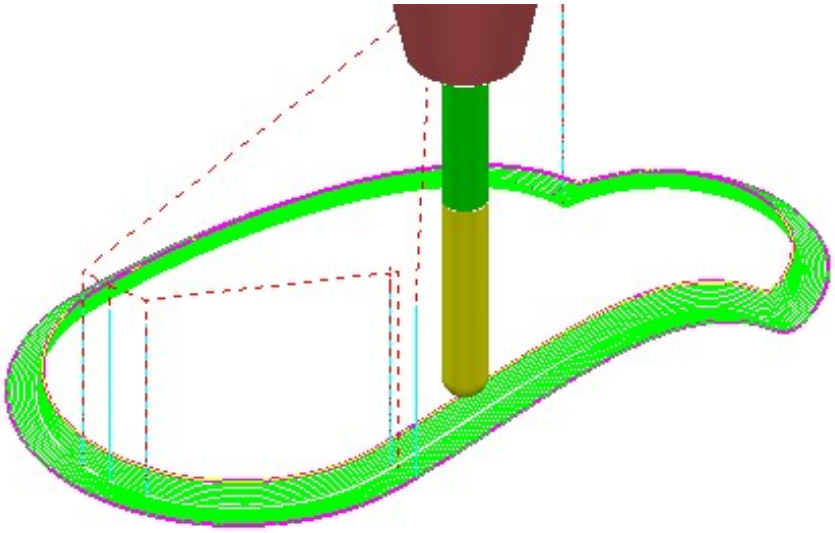
10 On the main toolbar, click to display the **Strategy Selector**.

11 From the **Finishing** options, select the **3D Offset Finishing** strategy dialog and enter the data as shown below.




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


Copyright © Autodesk



## Create a Boundary for the original side branches.

13 Undraw the ***Toolpath*** **BN8-FIN1** and ***Boundary*** **Rest** by switching off the respective light bulb symbols in the **PowerMill *explorer***.

14 Switch on the respective light bulb to display the **Boundary** named **Master**.

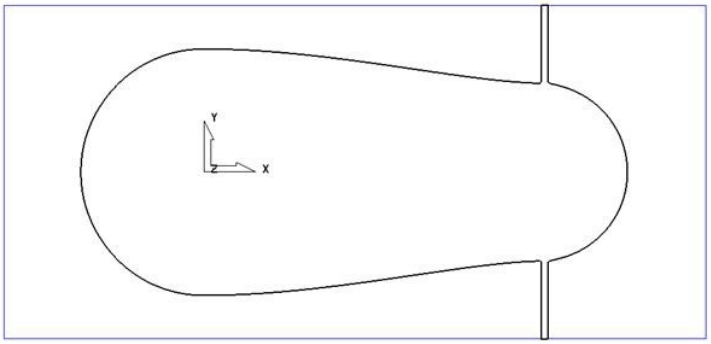
15 In the graphics area, select the **outer segment** of the displayed **Boundary**.

16 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.

17 Rename the boundary **Master_1** as **Spur1**.

18 Select a **view from top Z**.

19 In the **PowerMill *explorer***, turn off the light bulb adjacent to the ***boundary*** **Master** to remove it from view.




20 In the **PowerMill explorer**, right-click the boundary **Spur1** and select **Activate**.

21 In the graphics area, double-click the ***boundary segment*** to activate the **Curve Editor** toolbar.

22 Select all **points** along the ***boundary segment*** except those which define the rectangular part of the upper spur.

Copyright © Autodesk

*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.*

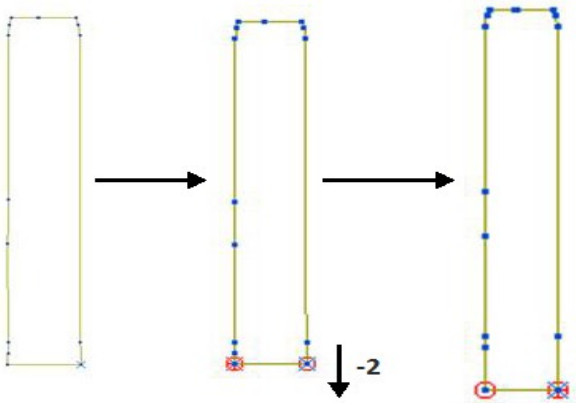
23 On the **Curve Editor** toolbar:

24 Click the **Delete Points** button to delete all the selected points.

25 Select the **points** along the base of the rectangular segment.

26 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 **Return** on your keyboard.

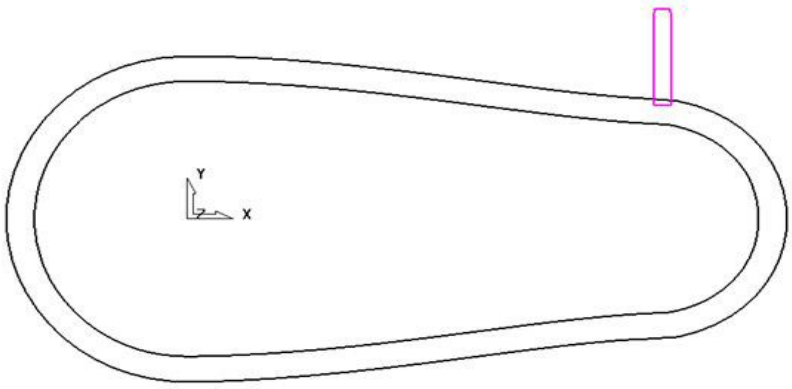
The selected points are moved by a distance of **Y-2**.




27 Click to exit the **Curve Editor** toolbar.

28 Click the light bulbs on to display both ***boundaries*** **Rest** and **Spur1**, and then **Resize the View** .


Copyright © Autodesk

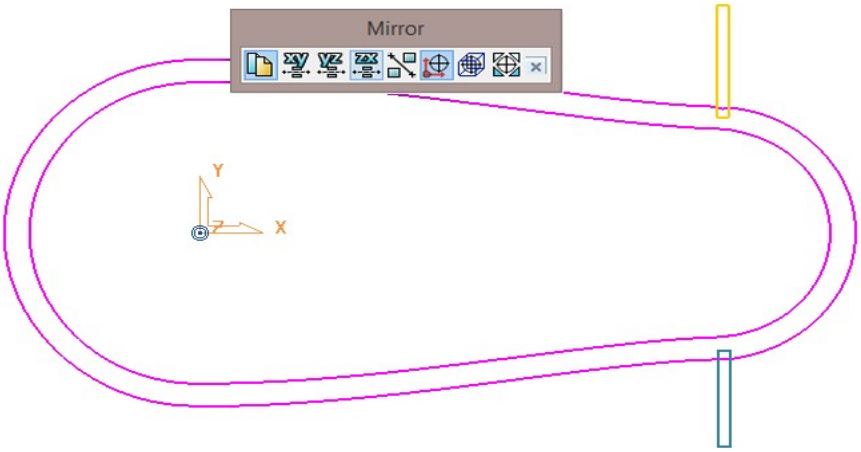


29 In the graphics area, double-click the ***boundary*** **Spur1** to activate the **Curve Editor** toolbar.

30 From the **Curve editor** toolbar, select the **Transformations** - **Mirror** option .

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

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




33 From the **Mirror** toolbar, click the **Mirror in ZX** button .

34 Click on the **Mirror** toolbar to accept and close.

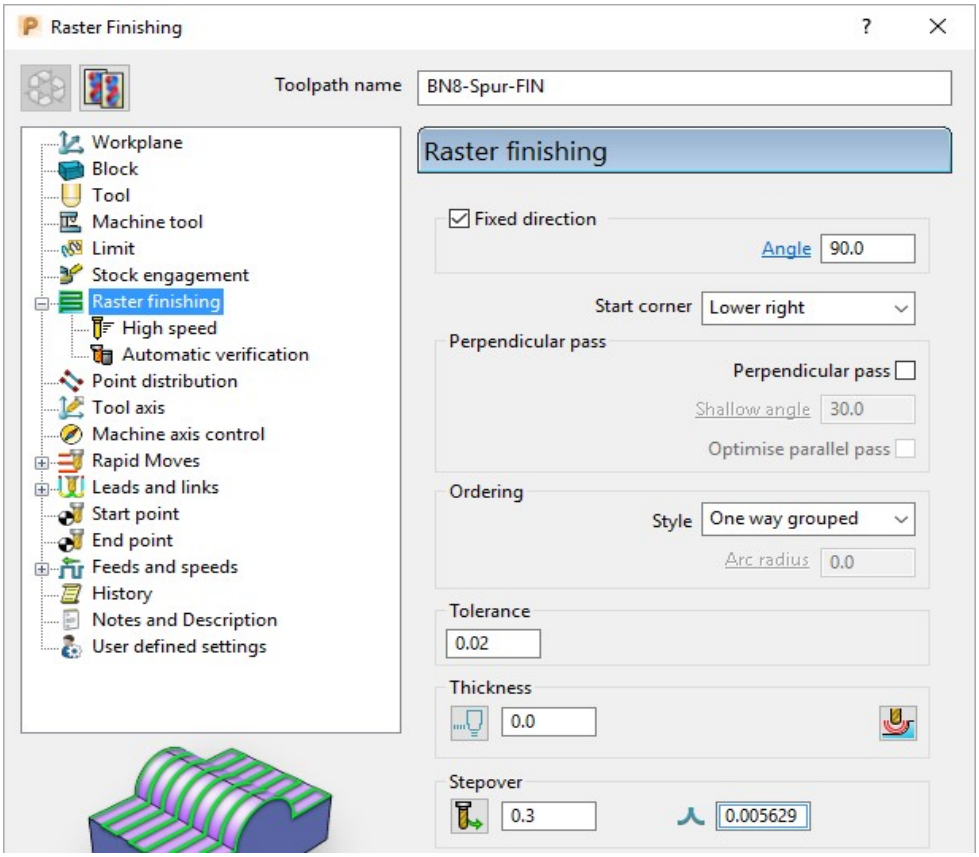
35 On the **Boundary Transform** toolbar, click to accept and exit the toolbar.

*The Boundary named Spur1 now contains 2 segments.*

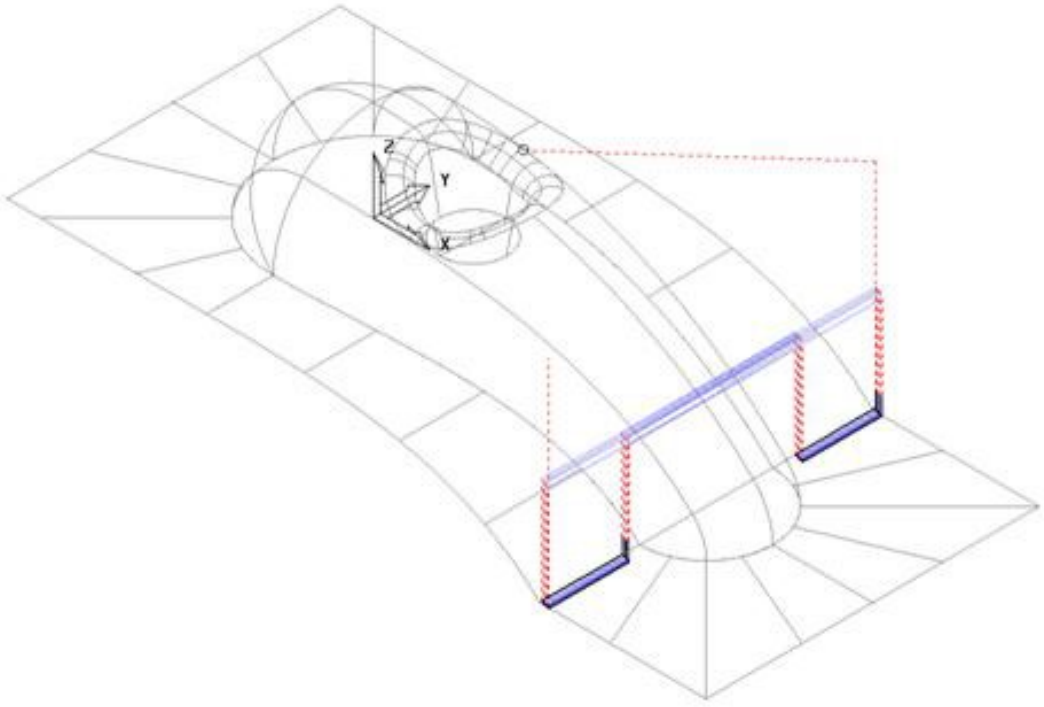
36 Make sure that the ***tool*** **BN8** and ***boundary*** **Spur1** are both **Active**.

37 From the **Strategy Selector**, **Finishing** options, select the **Raster Finishing** strategy and enter data as shown on the following page:

Copyright © Autodesk

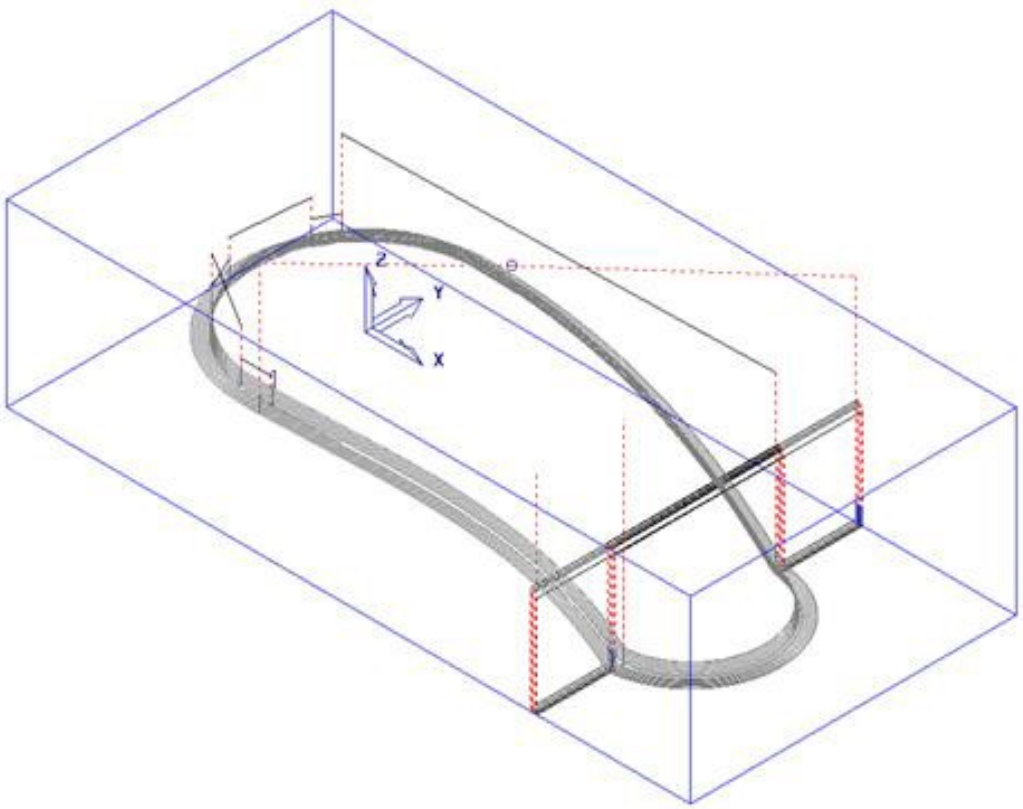



**38** Click Calculate to process the toolpath, and then Close the dialog.




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


Copyright © Autodesk




39 **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** – Provides the options to create 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.

Copyright © Autodesk

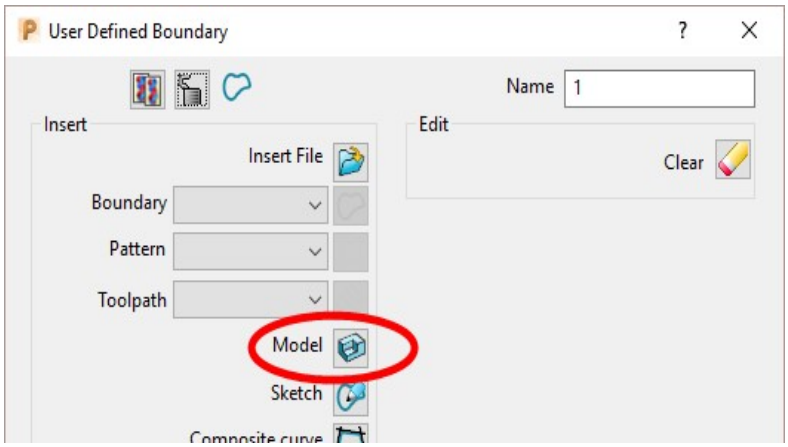
# Example

1. Select **File** – **Delete All** and **Tools** – **Reset Forms**.
2. **Import** the **Model**:

   **.....\PowerMill_Data\Models\pockets.dgk**

3. **Calculate** the **Block** defined by **Box** to the Model **Min/Max limits**.

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

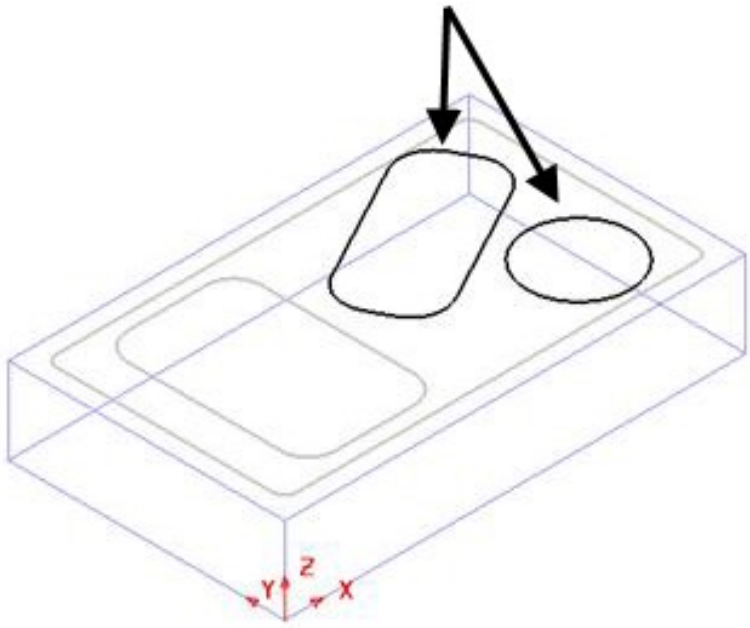



6. Click the **Model** icon and then select **Accept** to close the dialog.
7. Un-draw the **model** to provide a clear view of the **boundary**.

   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.

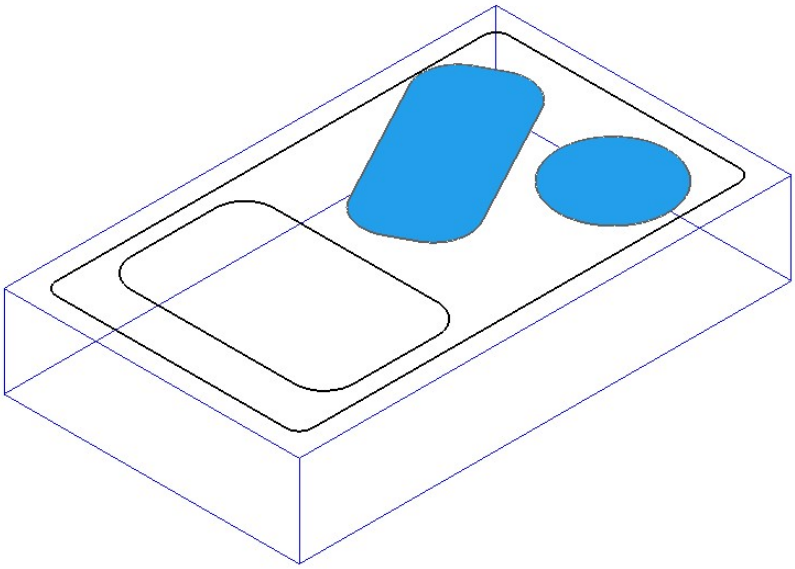

Copyright © Autodesk




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!*

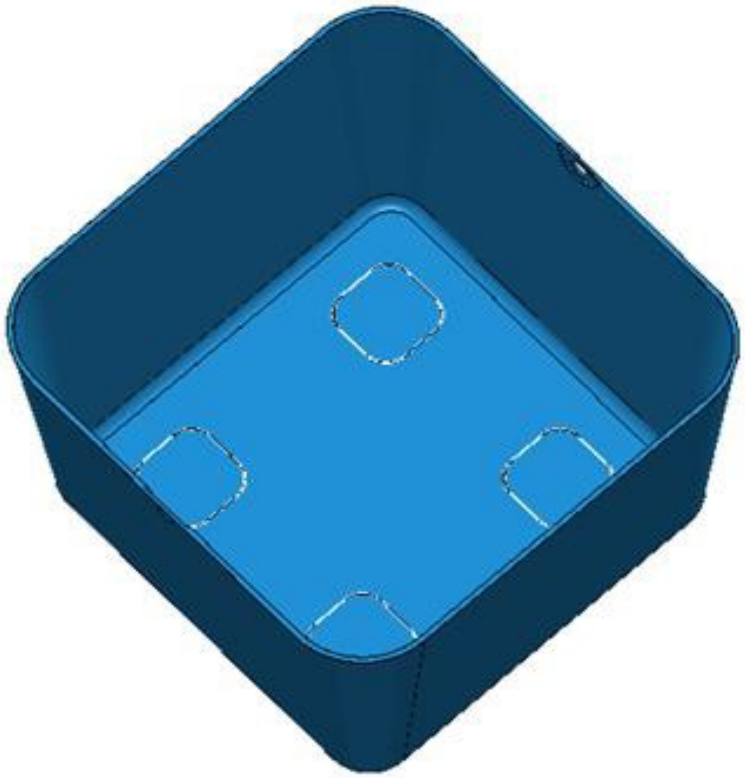
12 Select **File - Delete All** and **Tools** - **Reset Forms**.

# Exercise

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

**....\PowerMill_Data\Models\bucket.dgk**

2 Create 4 surface planes to cover the foot recesses on the upper base surface.



*No need to Save the Project this time!*


Copyright © Autodesk

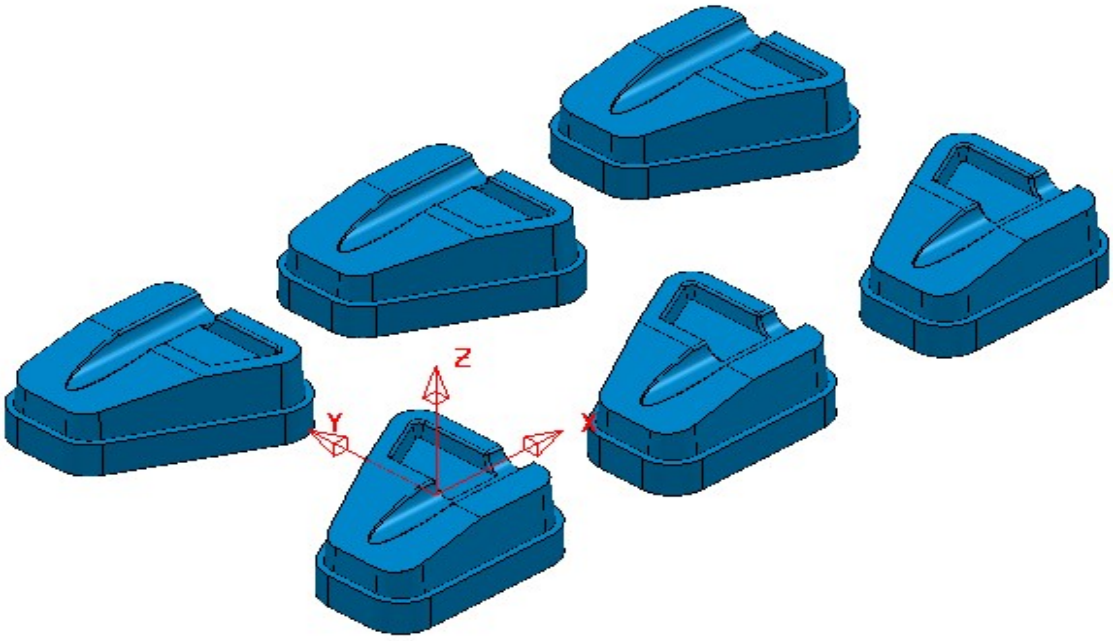
# Block defined by Boundary

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

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

2 Open the (read-only) **Project**:-

**….\PowerMill_Data\Projects\MultiBlock_Start**




3 **Save** the **Project as**:-

**….\COURSEWORK\PowerMill_Projects\MultiPart_Block**

4 Select all of the component models (Drag a box over all surfaces).

5 In the **PowerMill *explorer*** right mouse click on **Boundaries** and from the local menu select **Create Boundary** - **User Defined**.

6 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 .

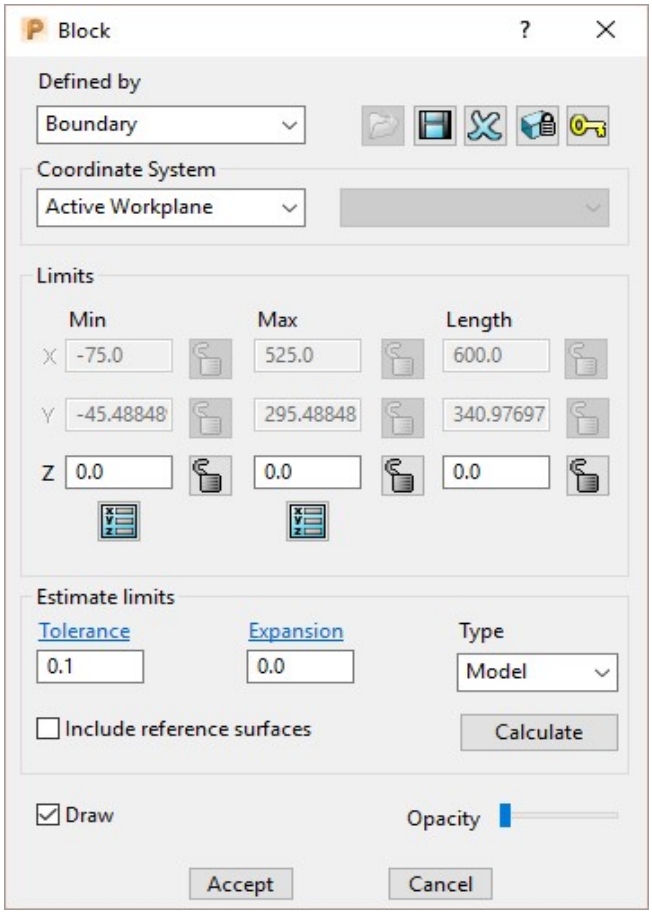
7 Select **Accept** to close the **User Defined Boundary** dialog.

8 **Activate** the new **Boundary**.

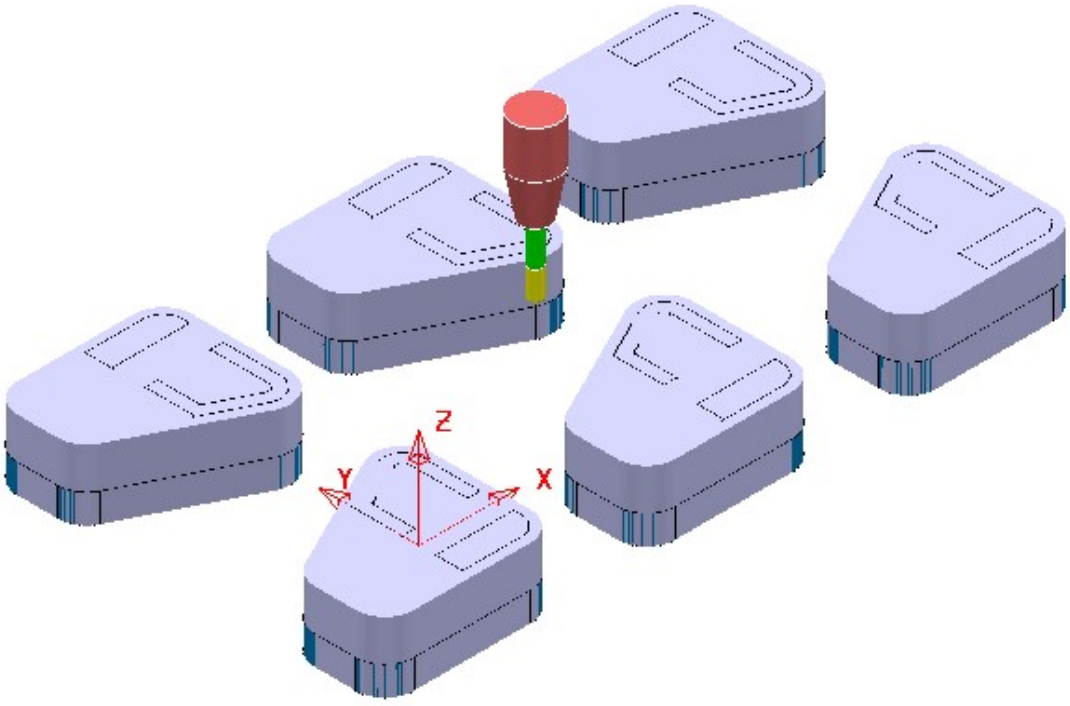
9 From the main toolbar select the **Block** options .

10 In the dialog select ***Defined by*** - **Boundary** and ***Type*** – **Model** followed by **Calculate**.

Copyright © Autodesk




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

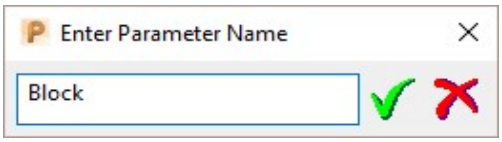



The complex **Block** is as shown above consisting of localised triangle models around each individual component.


Copyright © Autodesk

12 De-activate the **Boundary**.

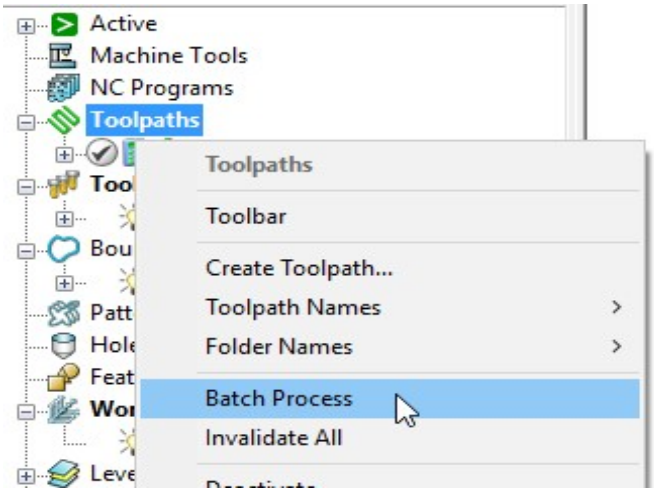
13 In the **PowerMill *explorer*** right mouse click on the existing, un-processed toolpath **D12TR1-RGH1** and from the local menu select **Edit** – **Set Named Parameter**.



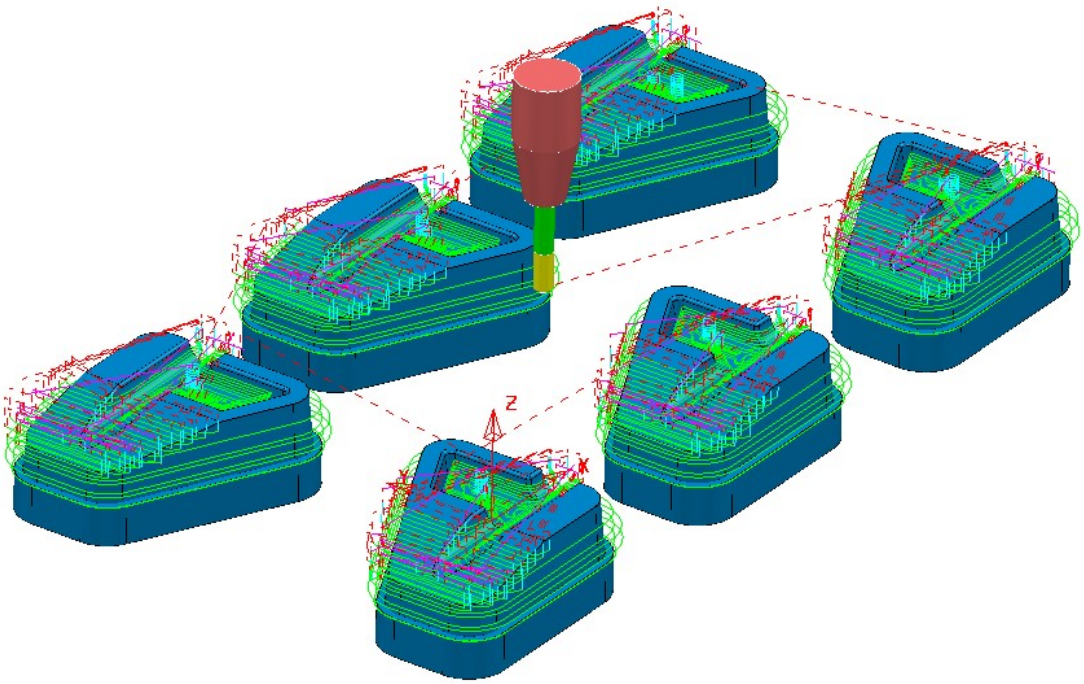

Input **Block** and select the ***green tick*** to accept.

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

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




**Batch Process** will sequentially calculate all valid, un-processed strategies.




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

**…\COURSEWORK\PowerMill_Projects\MultiPart_Block**

# Private Boundaries

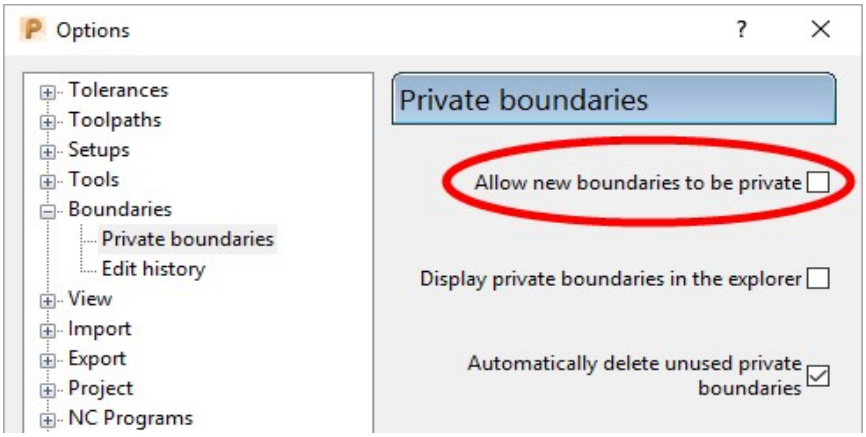
All **Boundary** types will (by default) be created as **private**. This makes the **Boundary** exclusively owned by the machining strategy that it is first applied to. If the parameters or entities used in the strategy are modified, then the associated **private boundary** will automatically match these 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:-

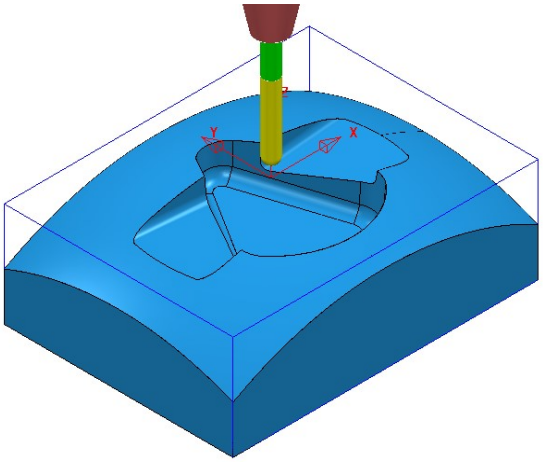
From the main pull down menus select **Tools** – **Options** – **Boundaries** - **Private Boundaries**, and un-tick **Allow new boundaries to be private**.




# 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**


Copyright © Autodesk

The **Project** includes a **Constant Z** and a **3DOffset *finishing*** toolpaths both of which currently apply over the whole of the **Block** area. To localise the 2 finishing toolpaths to areas of the model where they are most effective both strategies will be recycled to include a **Shallow** – ***private boundary***.

4 From the main pulldown menus select **Tools** - **Options** - **Boundaries** - **Private boundaries**:-

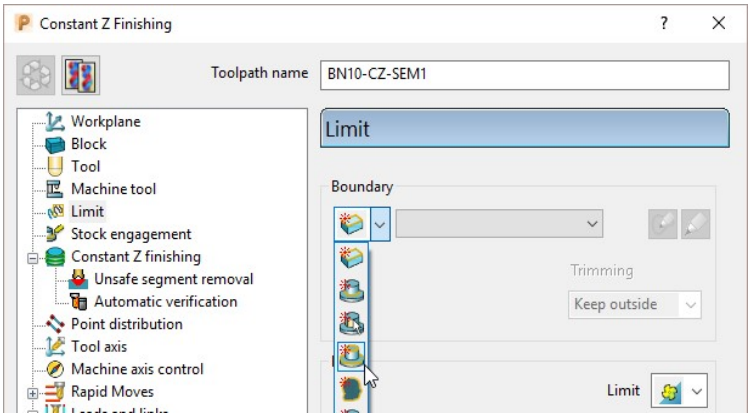
5 Tick the box named **Allow new boundaries to be private**.

6 Select **Accept** to close the **Options** dialog.

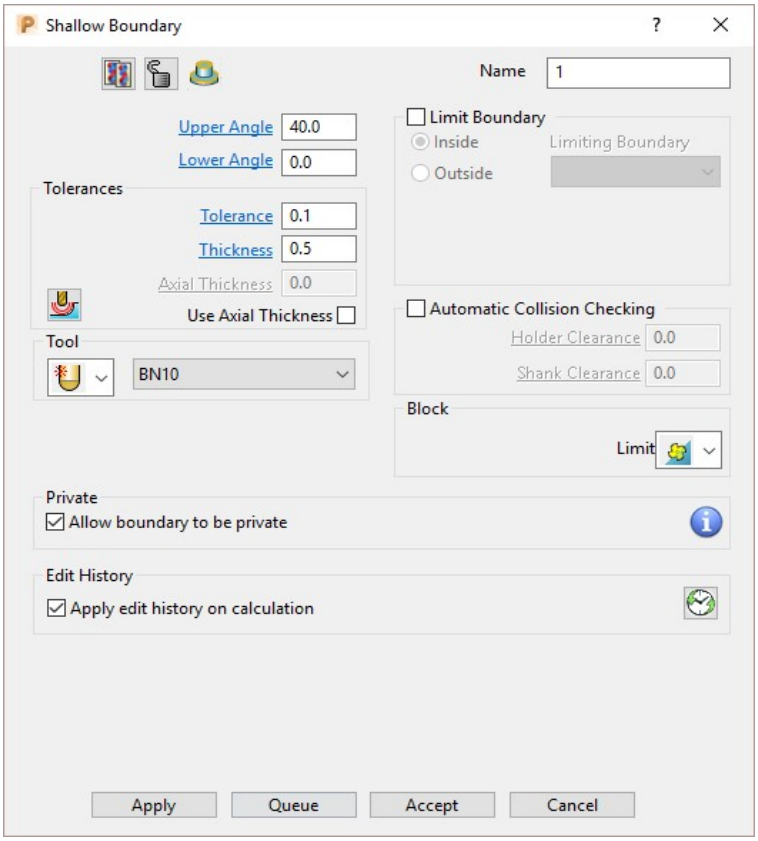
7 **Activate** the **toolpath**, **BN10-CZ-SEM1**.

8 From the **toolpath**, **BN10-CZ-SEM1** local menu select **Settings**, followed by the ***Recycle*** option.

9 In the **Constant Z** dialog select the **Limit** page.

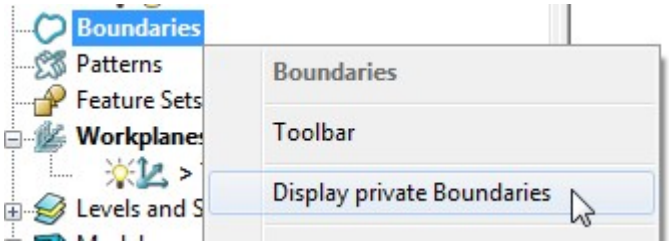



10 From the **Boundary** pull down options select **Create a shallow boundary**.



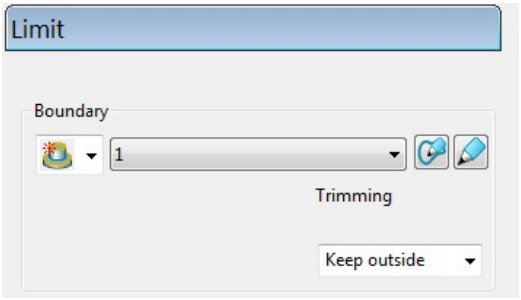
11 Using the default settings **Apply** and **Accept** the **Shallow Boundary** dialog.

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**

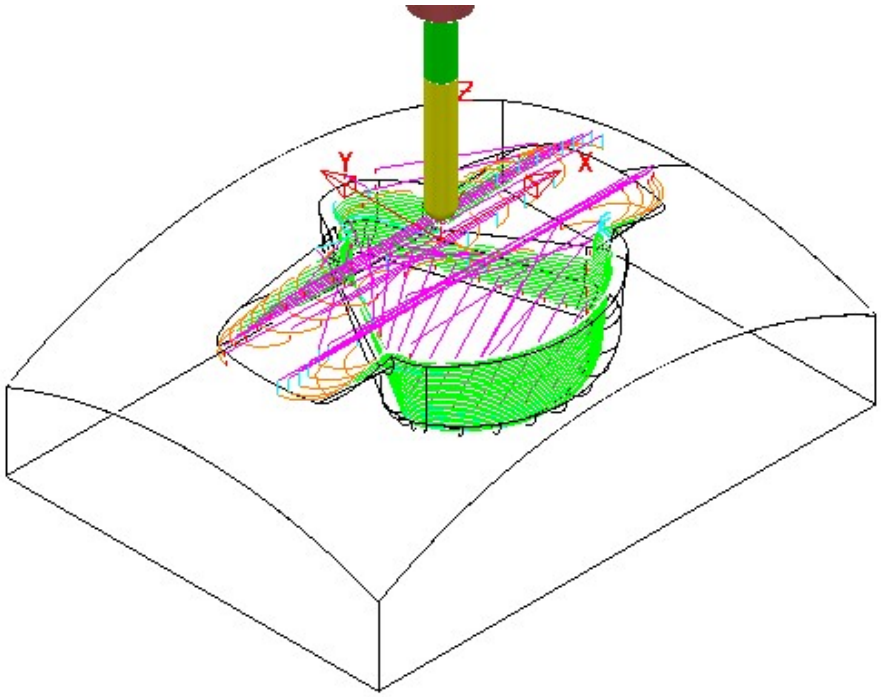



The ***private*** **Boundary** will be displayed in the **PowerMill *explorer*** if the above option is ticked in the local **Boundaries** menu.

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



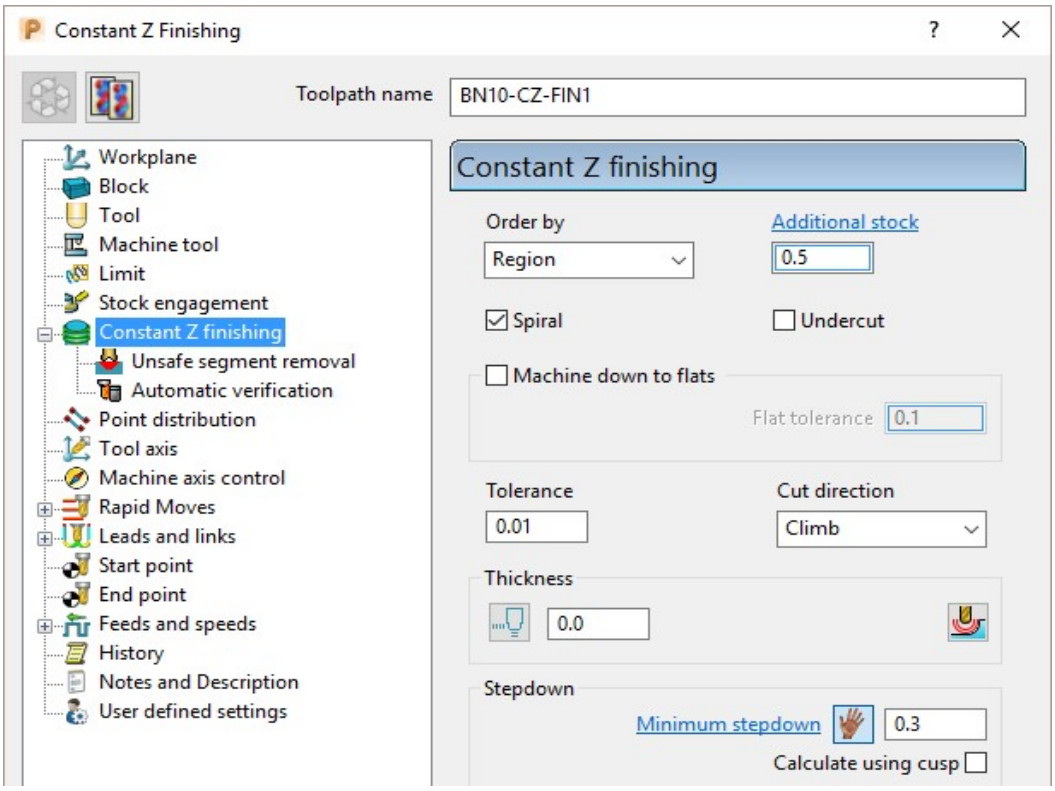

13 **Calculate** (but do not Close) the **Constant Z Finishing** strategy.




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

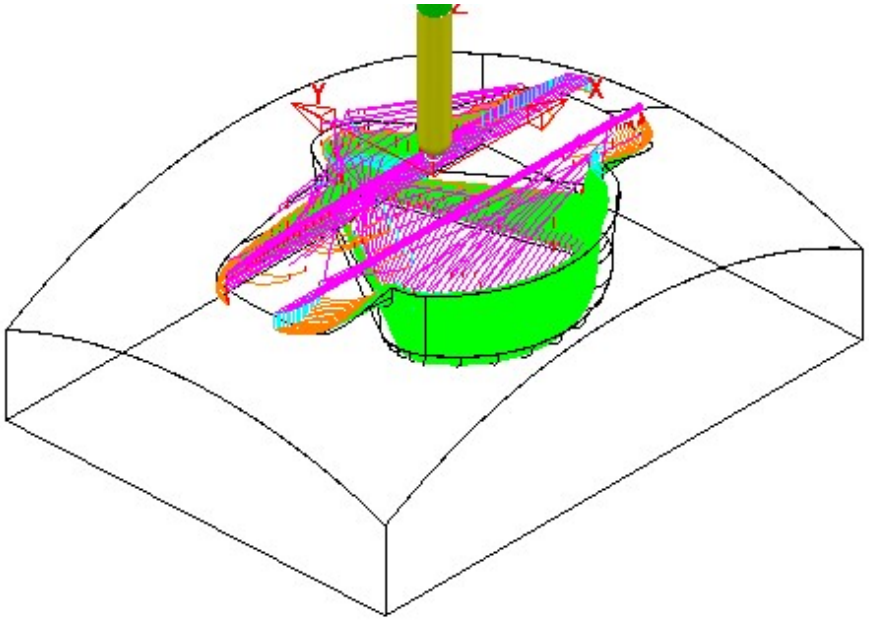

Copyright © Autodesk

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




15 Modify the **Toolpath Name** to **BN10-CZ-FIN1**, input **Tolerance 0.01**, **Thickness 0**, and **Minimum stepdown 0.3**

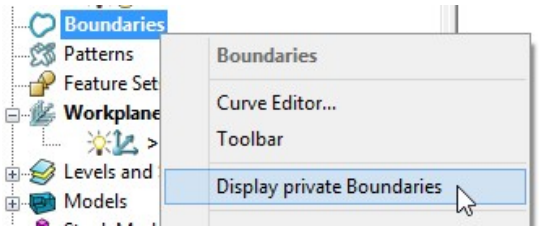
16 **Calculate** the new toolpath and **Close** the dialog.



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.

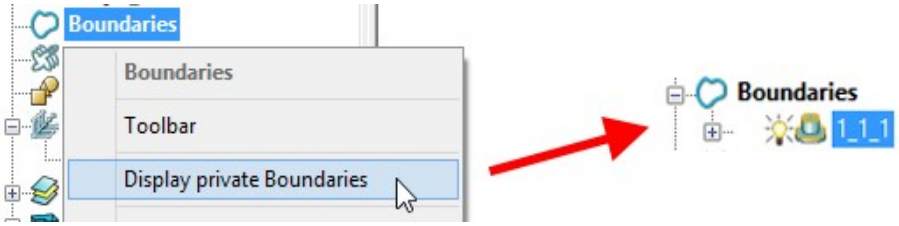
Copyright © Autodesk

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**.

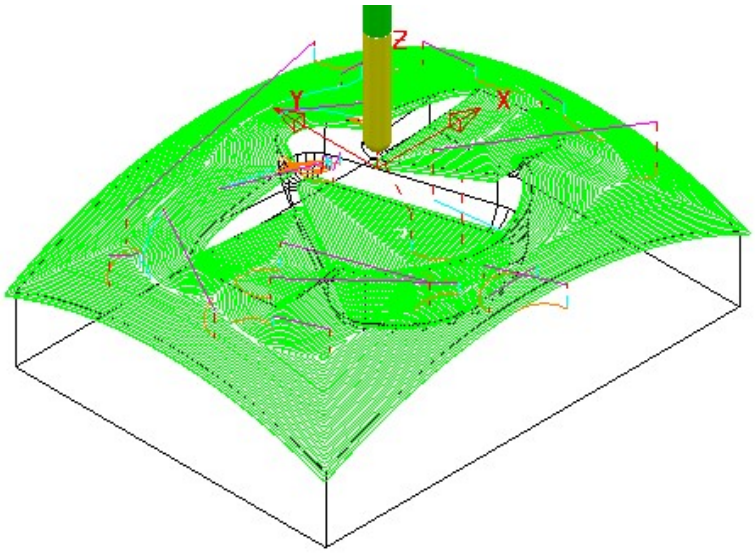



Only the new ***private*** **Boundary** will be displayed in the **PowerMill *explorer*** but will later become hidden once it has been applied in a new toolpath calculation.

# Exercise

21 In the **PowerMill *explorer***, activate the toolpath **BN10-3Doff-SEM1** and then select **Settings** to open the dialog.

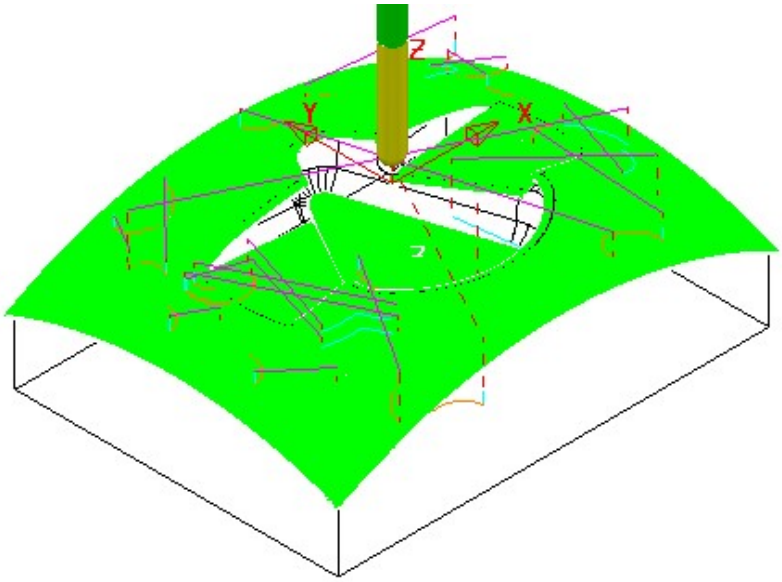
22 **Recycle** with the new ***private*** **Boundary 1_1_1** active, and with the ***Trimming*** set to **Keep inside**.





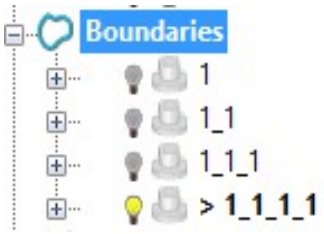
Copyright © Autodesk

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**.

Copyright © Autodesk

# 7. Levels and Sets

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

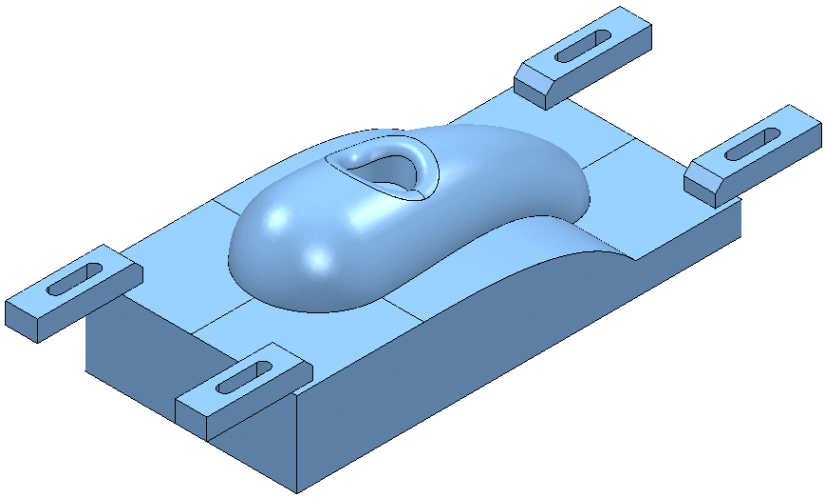
You can **Acquire** components from the currently imported **Model** to a **Level** or **Set**. Once acquired you can select, de-select, display, or hide **Model** components by using the individual **Level** or **Set** context menus.

The subtle differences between **Levels** and **Sets** are illustrated in 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**



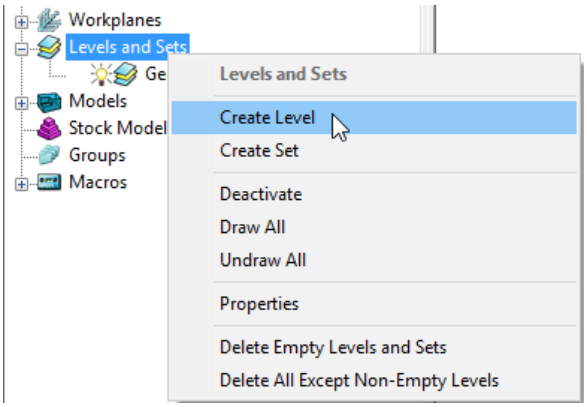
The imported **Model** includes clamps, which:

- 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**.

Copyright © Autodesk




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**.

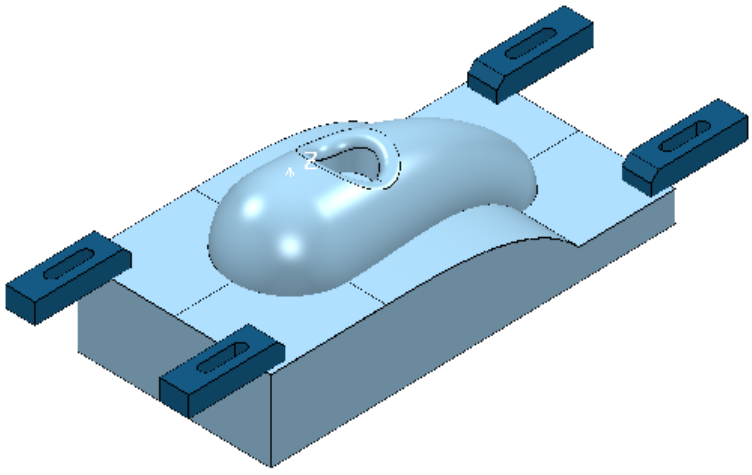
If the original CAD model has components already assigned to different **Levels**, then this will still be the case when the **model** is imported into **PowerMill**.

## Acquiring model entities to a Level

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

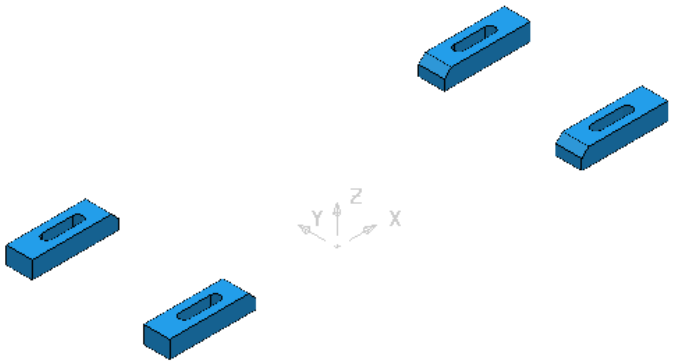
Any items acquired to this **level** will no longer be displayed.

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**.


Copyright © Autodesk

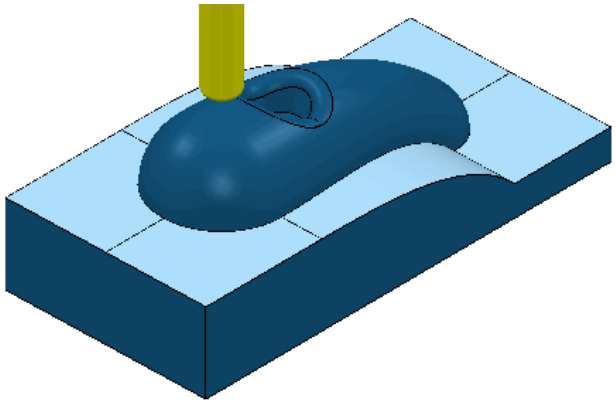


9 Select the ***Clamp*** surfaces (Hold down the left hand mouse key while dragging a box over them).

10 In the **PowerMill *explorer***, right-click on the **level** named ***Clamps*** and select ***Acquire Selected Model Geometry***.

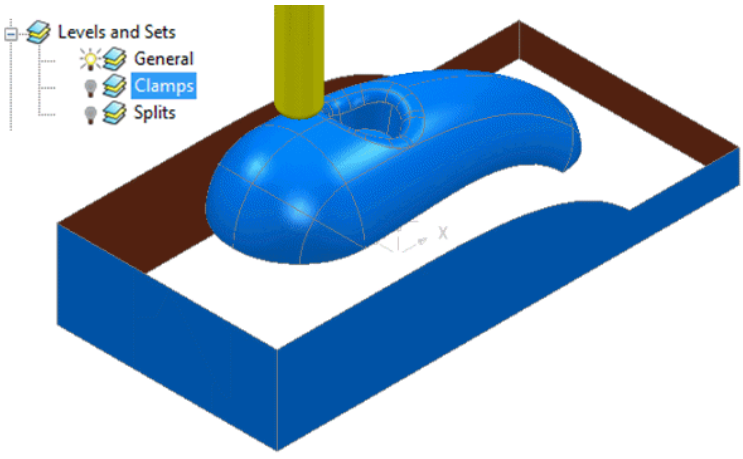
The clamps disappear from the view (The **Level *Clamps*** is switched off).

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.

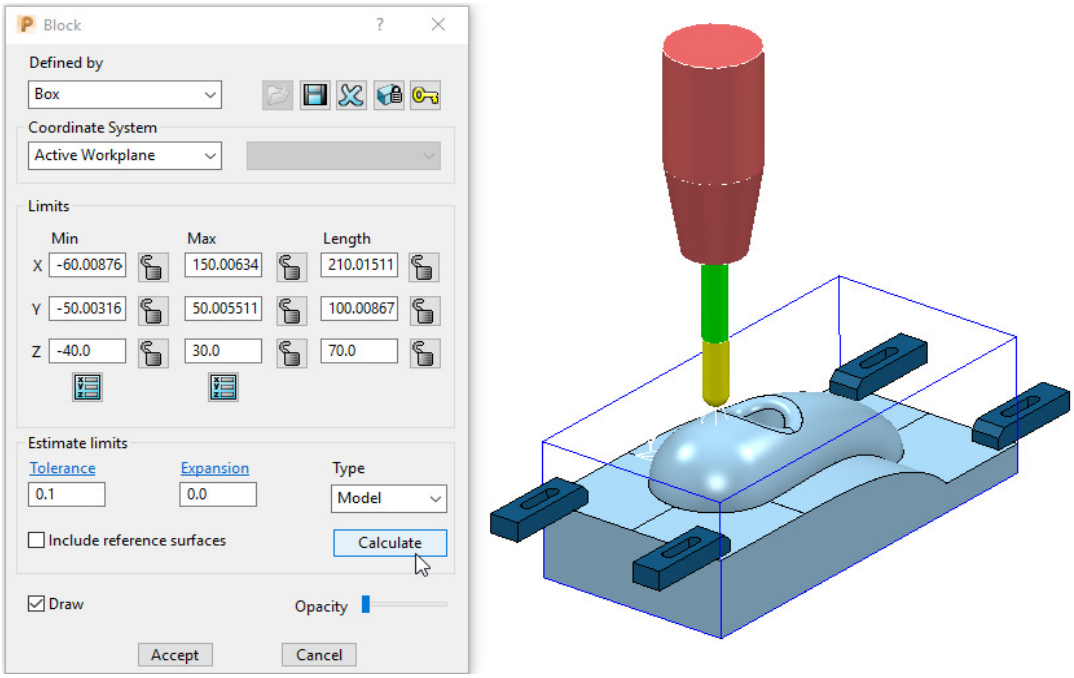
1 Create a Dia 12 Ball Nose tool named **BN12**.

2 In the **PowerMill *explorer***, right-click the **Level**, ***General*** and choose **Select Surfaces**. The main component **surfaces** excluding the split faces are selected.

3 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.

4 **Calculate** a **Block** using the default **Box** and **Model** options.

5 Use the default **Rapid Move Heights** and **Start and End Point** options.

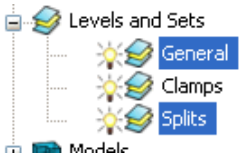



The **Block** is created to the dimensions of the selected surfaces.

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***.

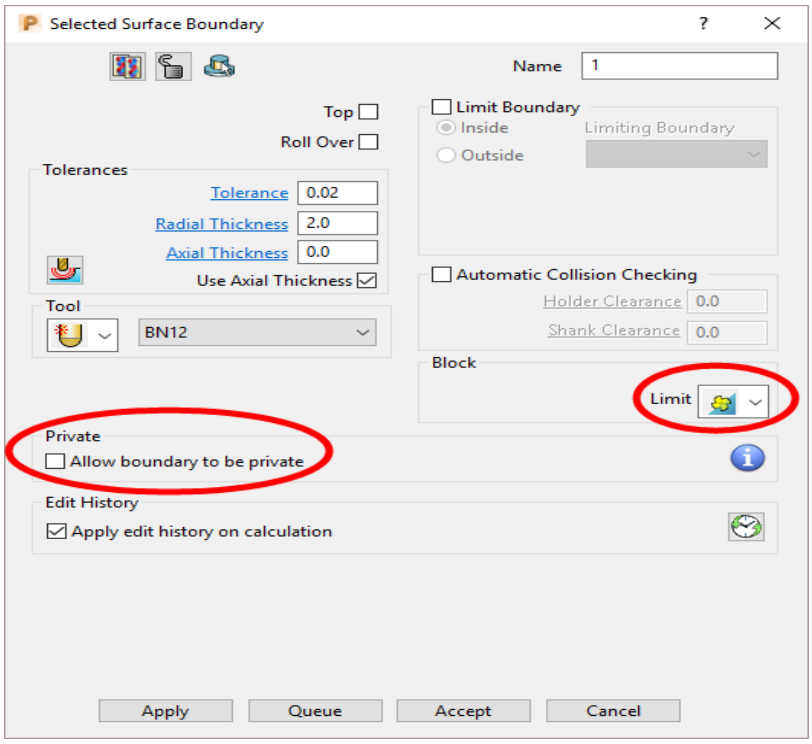


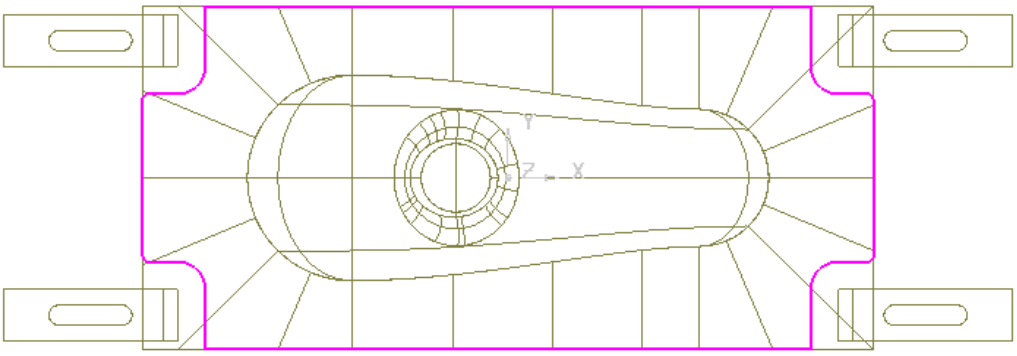

8 Right-click on one of the highlighted levels and choose **Select Surfaces**. All **surfaces** excluding the clamps are selected.

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.


Copyright © Autodesk

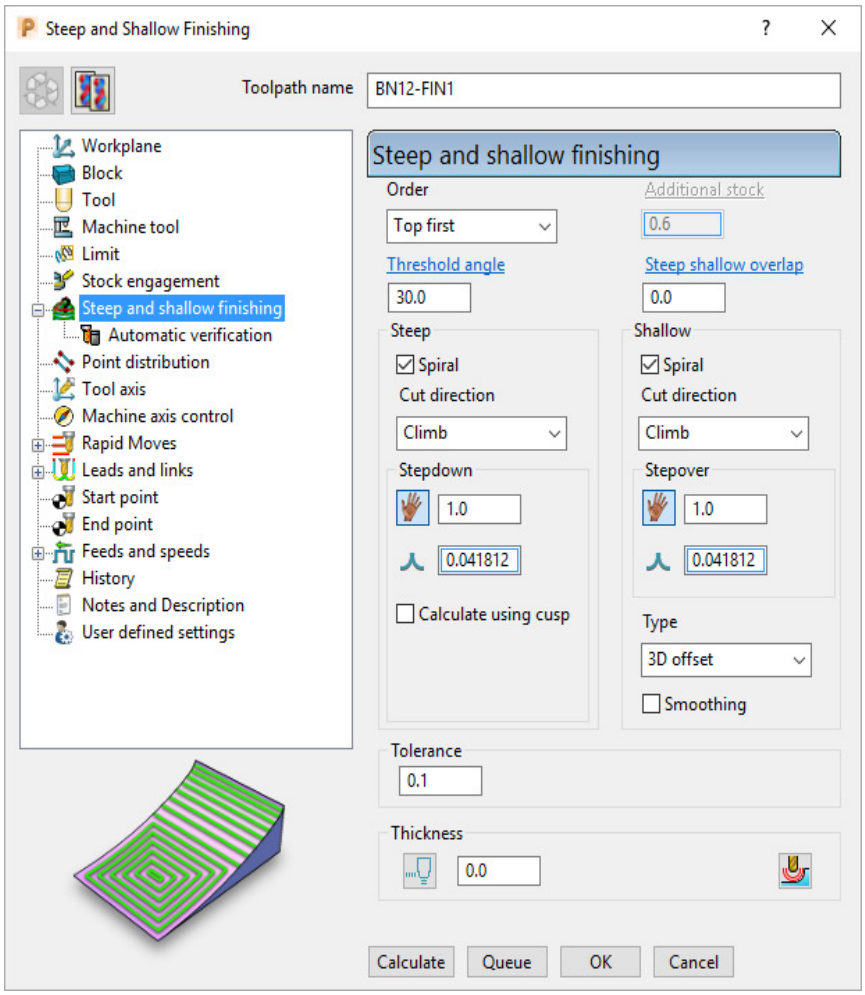


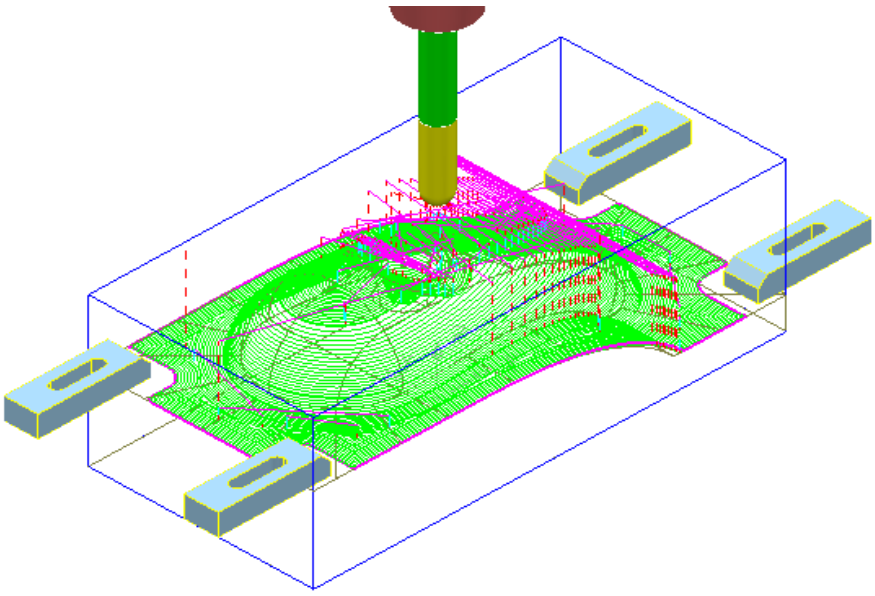
11 Check that both the **Tool *BN12*** and **Boundary *1*** are **Active** so that they will be automatically included when a new machining strategy is created.

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.




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

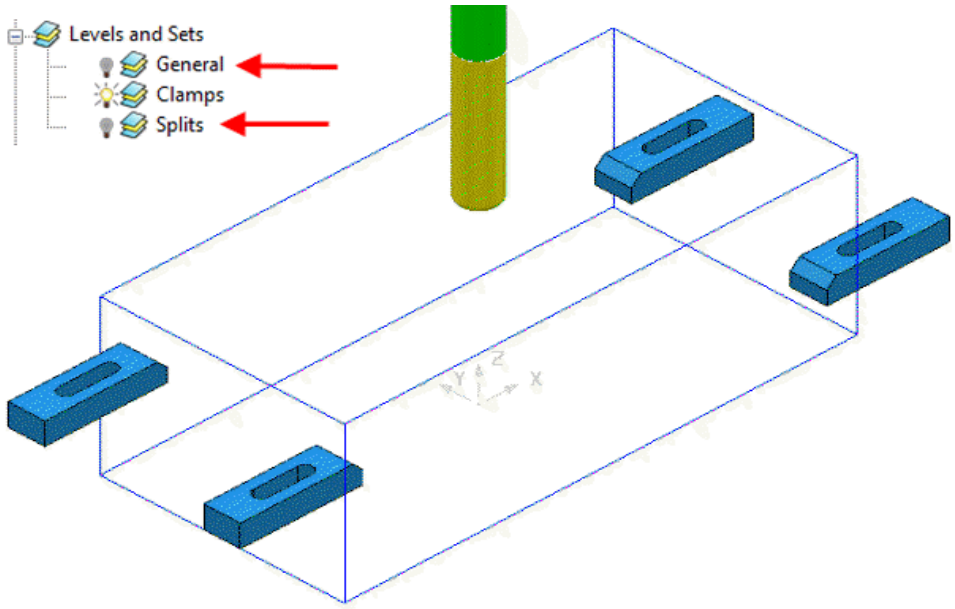


The **Radial Thickness** of **2** used in the outer **Boundary *segment*** provides a suitable clearance around the clamps.


Copyright © Autodesk

# Switching Off a Level

15 In the **PowerMill *explorer***, switch off the light bulbs on both the **General** and **Splits** levels.

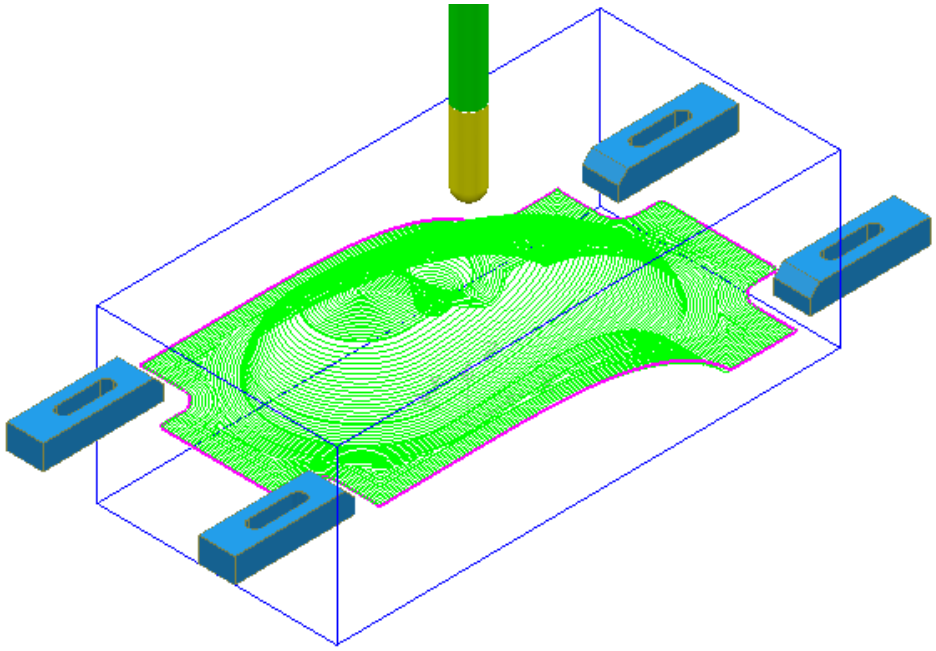



Only the ***Clamp*** **Surfaces** are displayed in the graphics area.

16 Right-click the **toolpath** and select **Settings** to re-open the dialog.

17 Select create a ***Copy*** and click **Calculate** to process a new **toolpath**.

18 **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

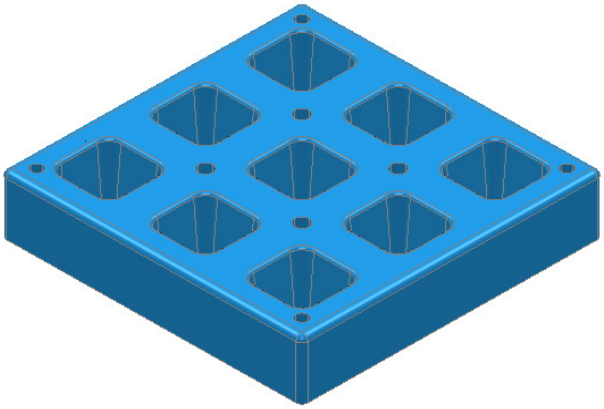
A **Model** entity will always exist on a **Level** but not on more than one at the same time. You cannot delete a **Level** while **Model** entities are acquired to it.

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.

19 From the **Main** menu, click **File** > **Delete All** and **Tools Reset Forms**.

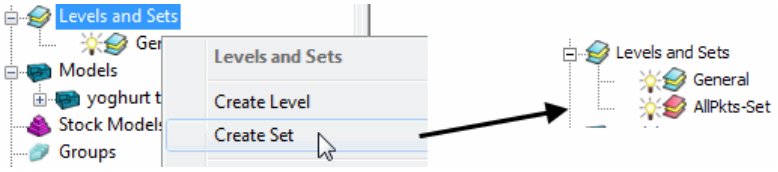
20 From the **Main** pulldown menus **File** > **Import Model** and select:

**.....\PowerMill_Data\Models\YogurtTray.dgk**

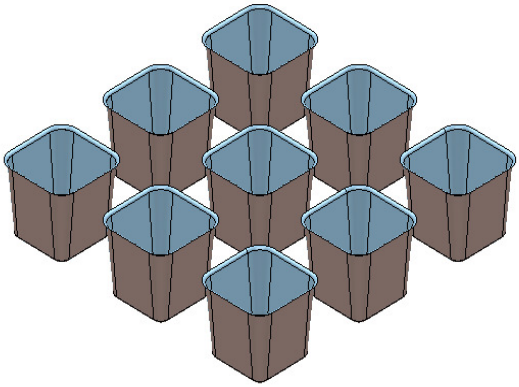


21 In the **PowerMill *explorer*** right mouse click on **Levels and Sets**.

22 From the local menu, select **Create Set** and rename it as **AllPkts-Set**.



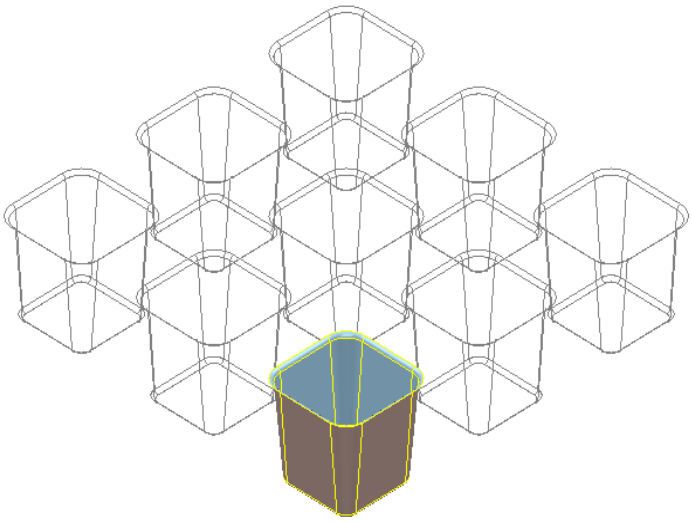

23 Manually select all **surfaces** defining the 9 pockets.



24 With the 9 pockets still selected, right-click **AllPkts-Set** and select **Acquire Selected Model Geometry**.


Copyright © Autodesk

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



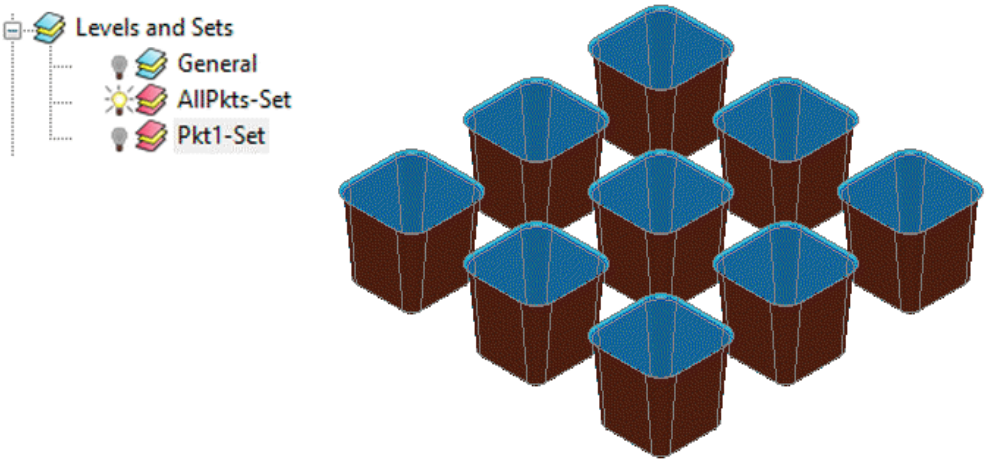
26 Create a new **Set** and rename it as **Pkt1-Set**.

27 Right-click **Pkt1-Set** and select **Acquire Selected Model Geometry**.

28 On the keyboard select **Ctrl L** to ***unblank*** all invisible **Surfaces**.

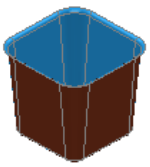
29 Switch off both the **level *General*** and the **Set *Pkt1-Set***.

30 Switch on **AllPkts-Set**.




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

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

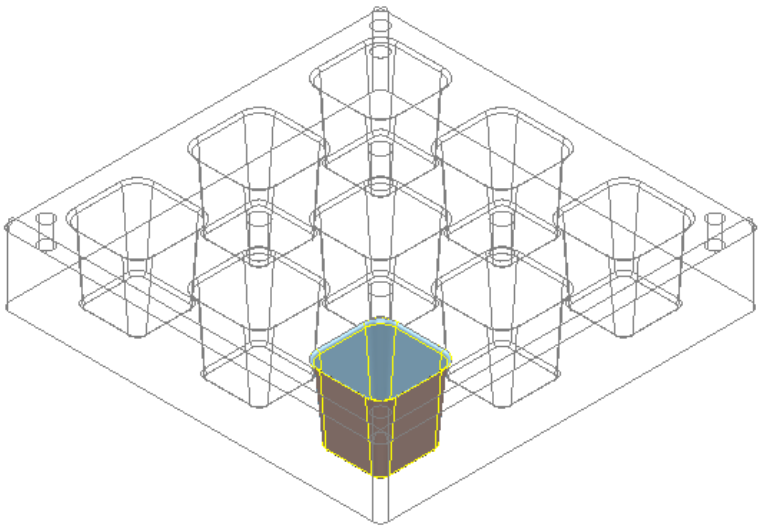


The **surfaces** acquired to **Pkt1-Set** are displayed. They are also displayed when the **AllPkts-Set** is switched on and **Pkt1-Set** is switched off. This illustrates that model entities can be acquired to more than one **Set** at the same time.

Copyright © Autodesk

32 Switch off both **Sets** and switch on the **Level General**.

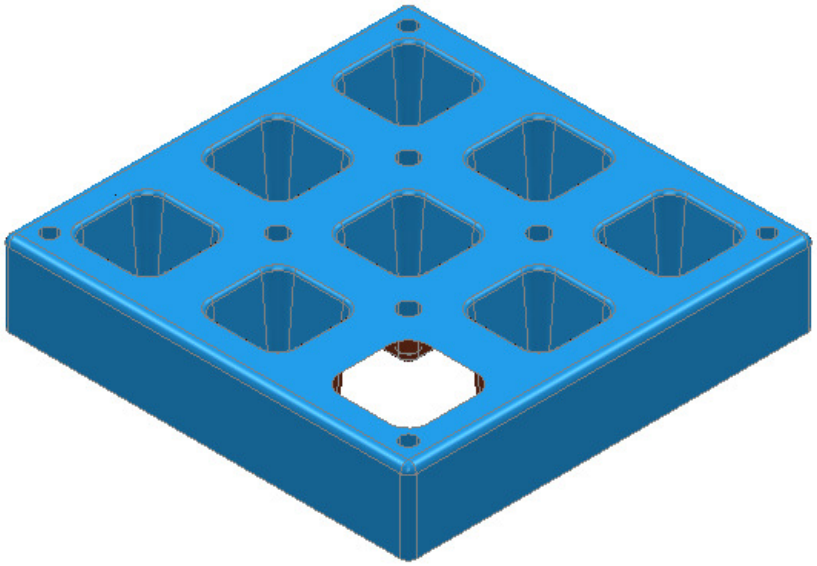
33 In the **PowerMill *explorer*** right-click on **Pkt1-Set** and from the local menu click **Select Surfaces**.



34 Create a new **Level** and rename it as **Pkt1-Level**.

35 Switch of the light bulb symbol next to the new **Level *Pkt1-Level***.

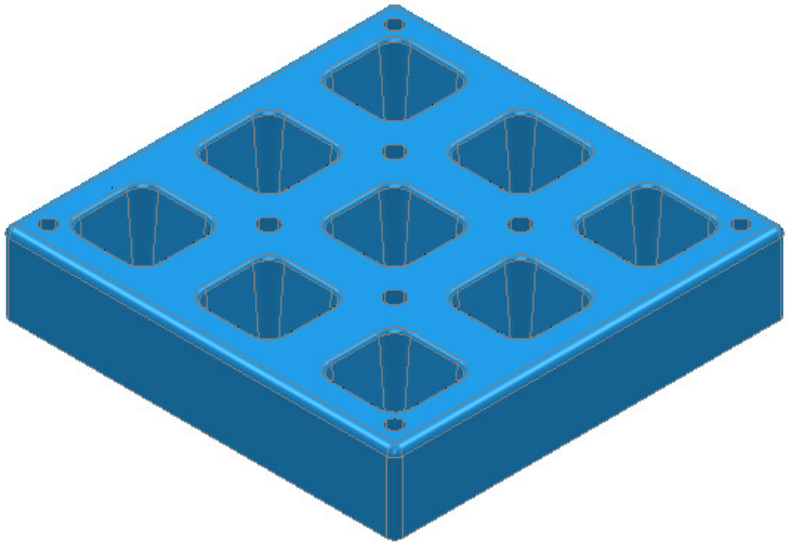
36 In the **PowerMill *explorer***, right-click ***Pkt1-Level*** and select **Acquire Selected Model Geometry**.



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

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


Copyright © Autodesk



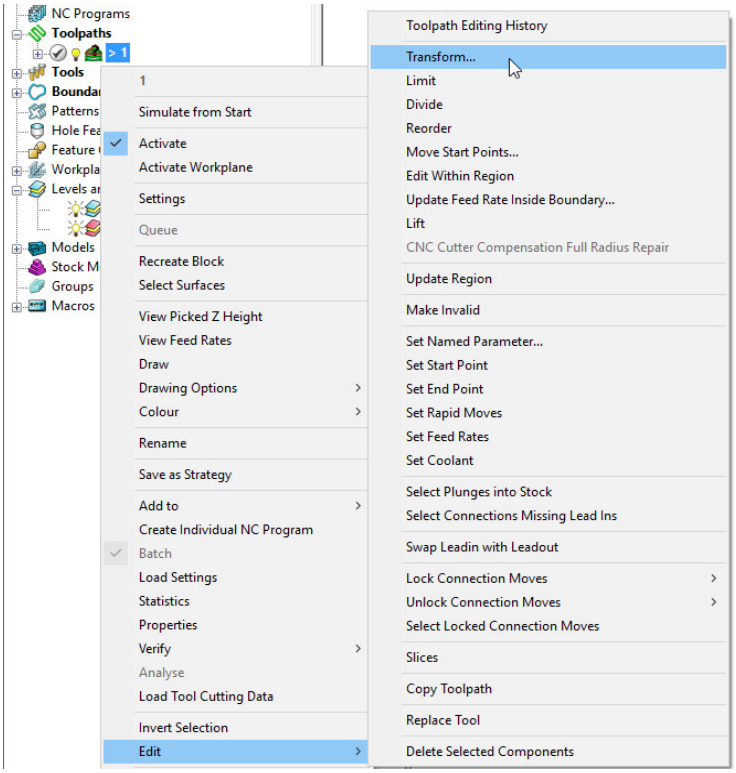
*When using a combination of **Levels** and **Sets** it is good practice to keep the **Sets** visually switched off. If you want to select items from a particular **Set**, then, right-click on it and use one of the **Select** options from the context menu.*

*This prevents confusion if you switch a **Level** off only to find the included items are still visible due to a **Set** containing the same items being switched on.*

Copyright © Autodesk

# 8. Editing Toolpaths

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



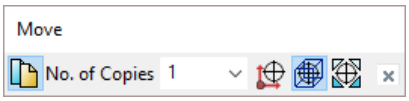

Many of the **Toolpath** editing options are accessed from the local menus (as shown above). Most of these edits will no longer apply if a **Toolpath** is subsequently re-calculated.

## Edit > Transform

The **Transform** option allows you to **Move**, **Rotate**, and **Mirror** toolpaths relative to the active datum.

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

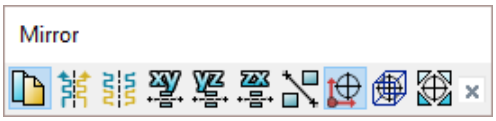
**Transform** > **Move** - The **Move** function allows a selected toolpath to be moved by a user-defined ***Distance*** along a selected axis.




**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 as unknown, so it is advisable to check for gouges.

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

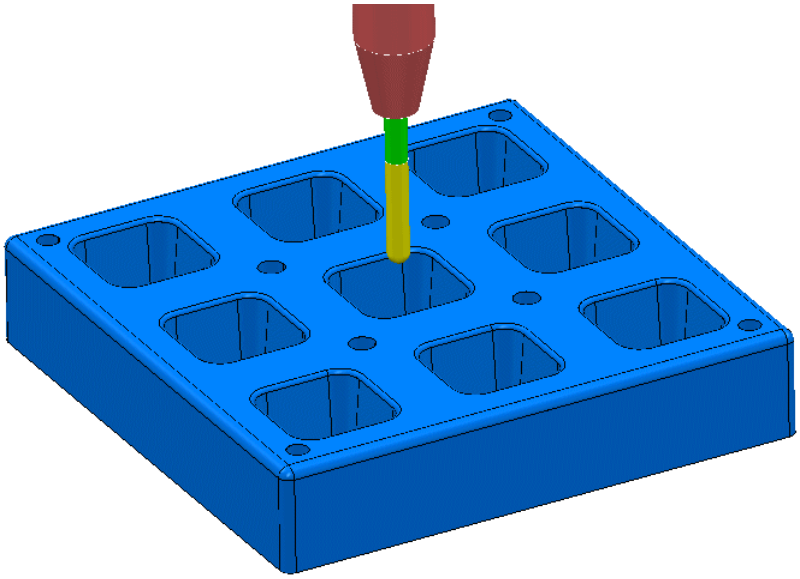



In the following example a single toolpath is created and then the **Transform** > **Move** option applied to copy toolpaths into the remaining cavities.

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

2 **Open** the read-only **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**:

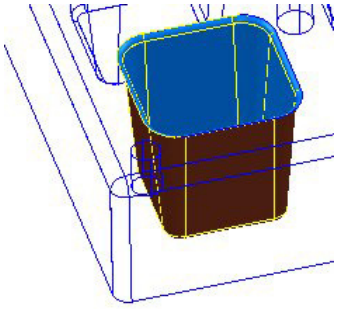
**.....\COURSEWORK\PowerMill-Projects\Transform**

4 Create a **Block** using ***Defined by*** > **Box** and ***Type*** > **Model**.

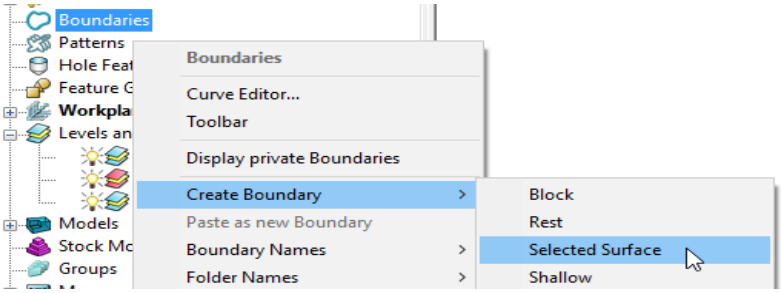
5 **Calculate** the **Rapid Move Heights** using the default settings.


Copyright © Autodesk

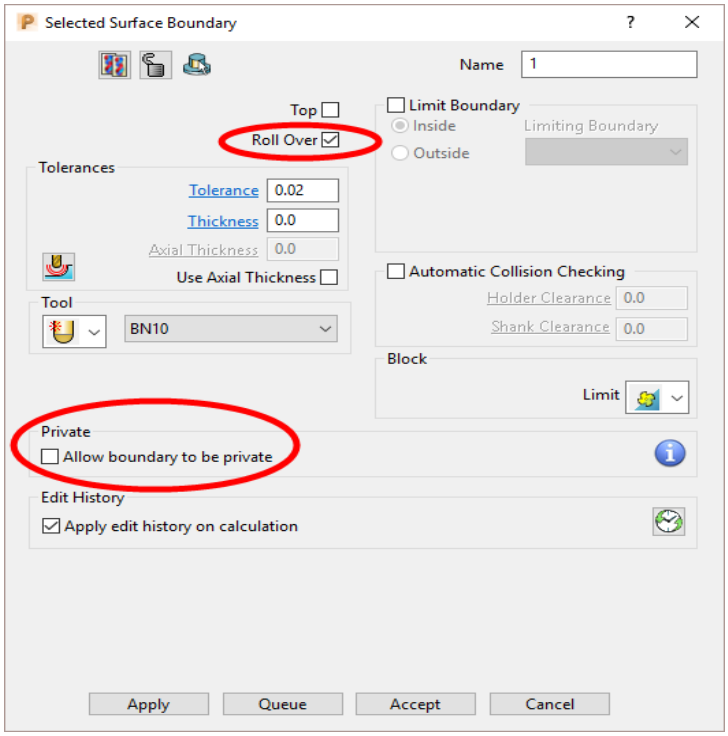
6 Select all of the **surfaces** (including the fillet) that define the bottom left pocket.



7 In the **PowerMill *explorer***, right-click **Boundaries** and select **Create Boundary** > **Selected Surface**.



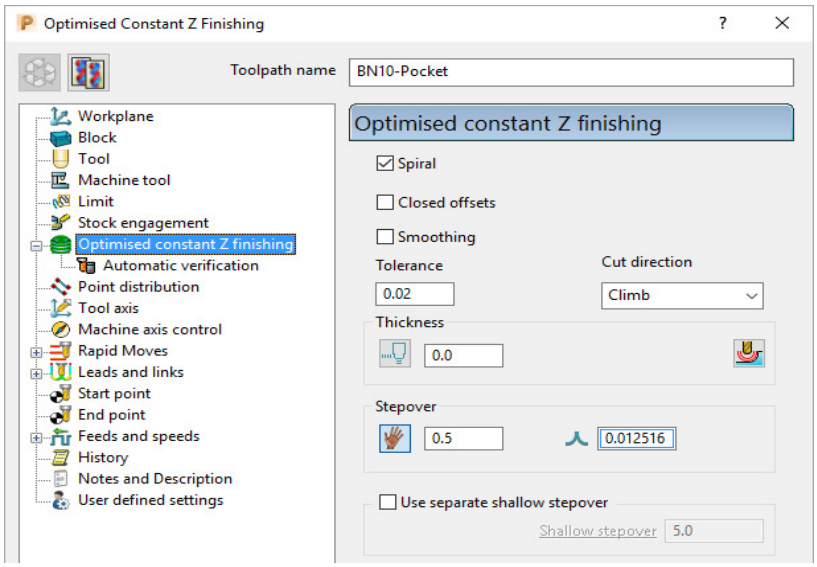

8 In the **Selected Surface Boundary** dialog, tick **Roll Over** and enter a ***Tolerance* 0.02** and un-tick, **Allow boundary to be private**.

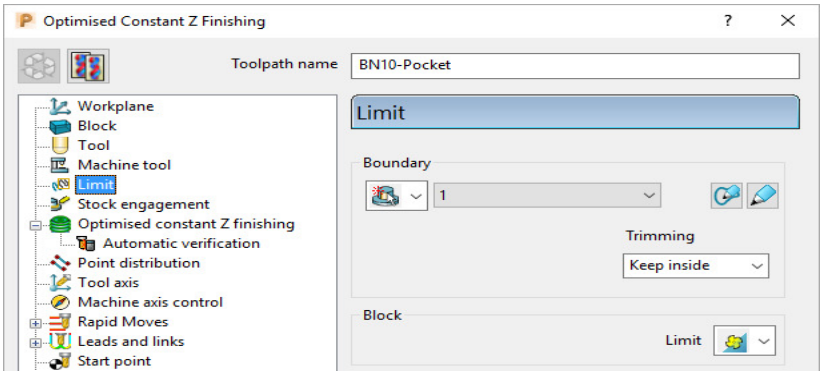



9 Click **Apply** and **Accept** the dialog.

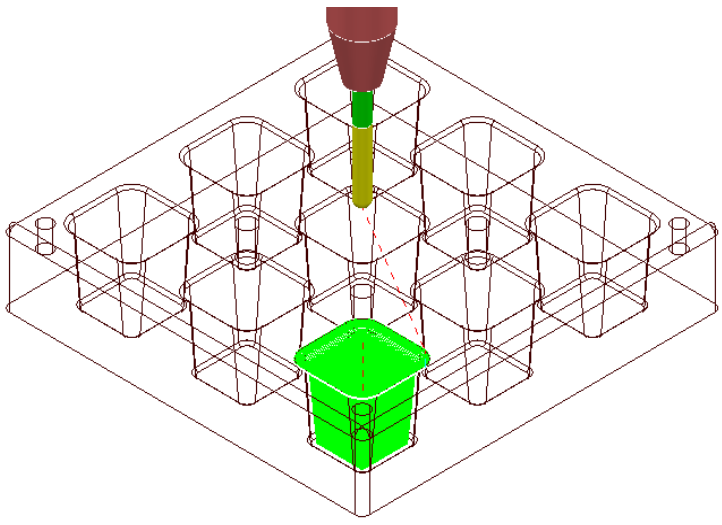
A ***Selected Surface* Boundary** defines the limit where the active **Tool** and associated parameters would fully machine the selected surfaces.

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




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.


Copyright © Autodesk

# 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.

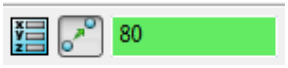
3 In the **PowerMill *explorer***, right-click on the toolpath **BN10-Pocket** and select **Edit** > **Transform** to display the **Toolpath Transform** toolbar.

4 Check that the ***append toolpaths*** option is switched off .

5 On the **Toolpath Transform** toolbar, click **Move** .

6 On the **Move** toolbar, click **Keep original** with **No. of Copies** as **2**.

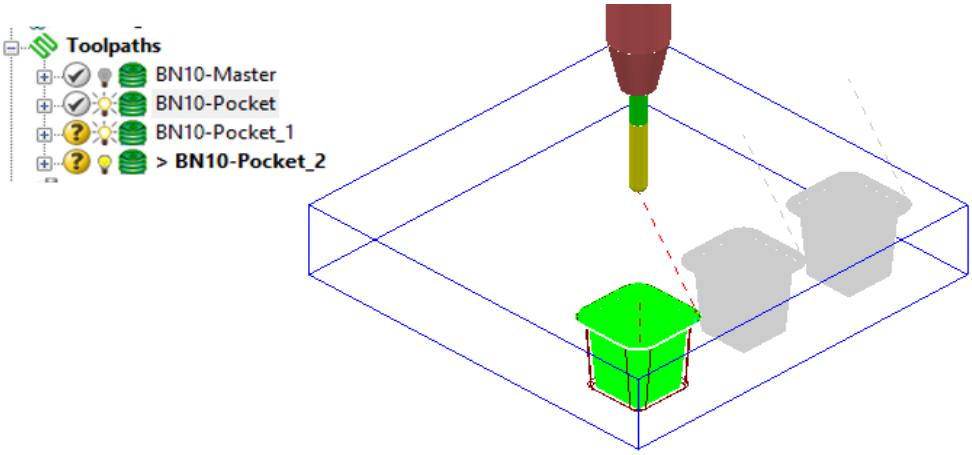
7 In the data entry box located below the graphics area, enter the relative **distance** along X as **80**, and press ***Enter*** on your keyboard.




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

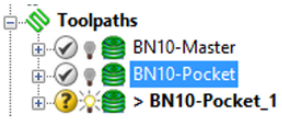
8 In the **Transform** toolbar, click to close and accept the changes.

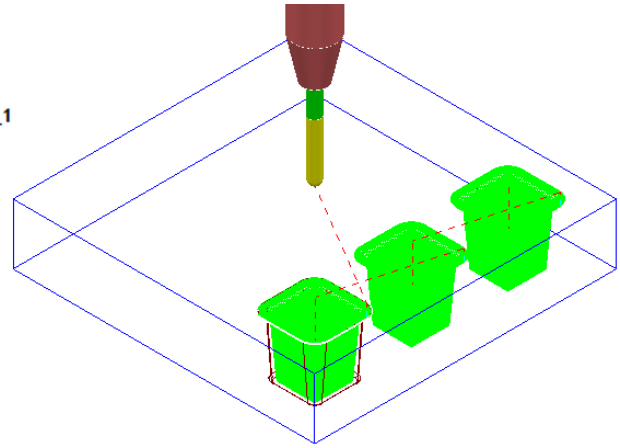
Two new copies of the original **toolpath** named **BN10-Pocket_1** and **BN10-Pocket_2** are created and transformed along **X** by a relative distance of **80**.



Note: It is possible to directly append all of tool tracks into a new, single toolpath **BN10-Pocket_1**.

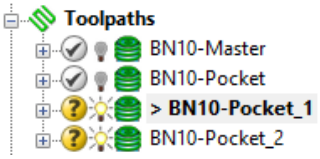
This is achieved by selecting the **Automatically append transformed toolpaths** option before applying the transform values.



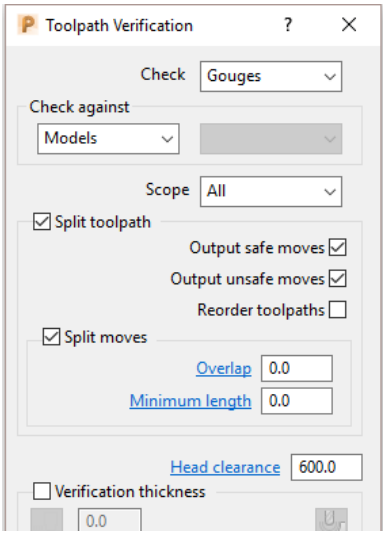



# Gouge checking a toolpath

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.

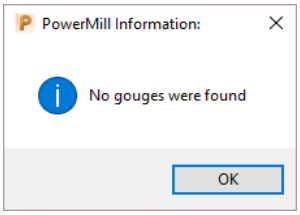




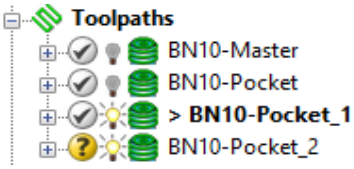
Copyright © Autodesk

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**

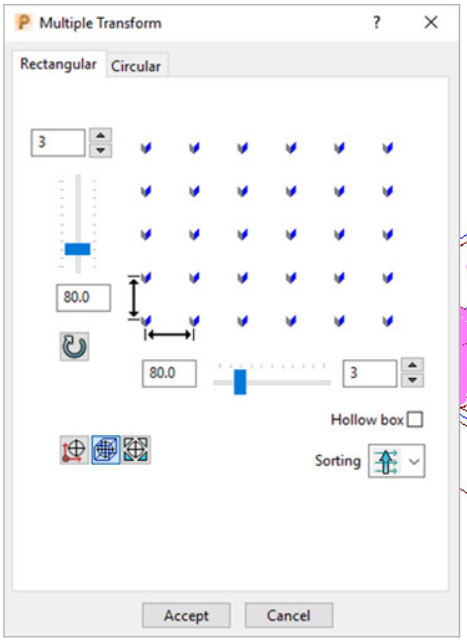
# Multiple Transform

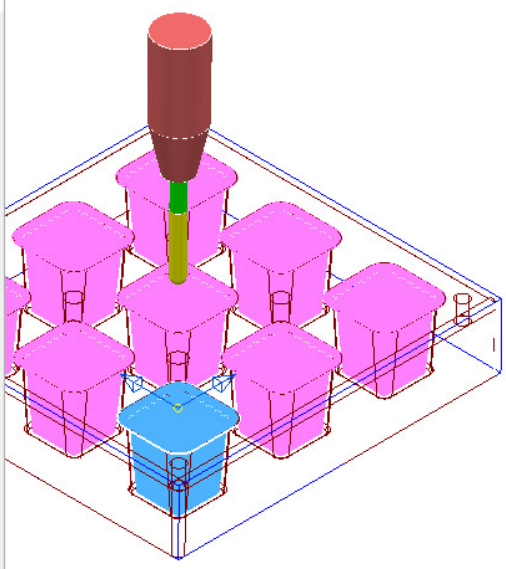
6 Delete both toolpaths **BN10-Pocket_1** and **BN10-Pocket_2**.

7 Right mouse click on toolpath **BN10-Pocket** and from the local menu select **Edit** > **Transform**.

8 Select the option **Automatically append transformed toolpaths**.

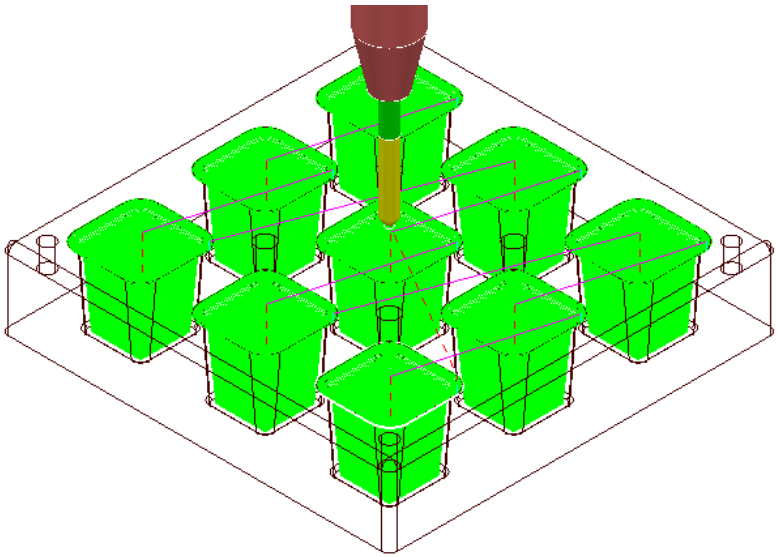
9 Select the option **Multiple Transform** to open the following dialog.



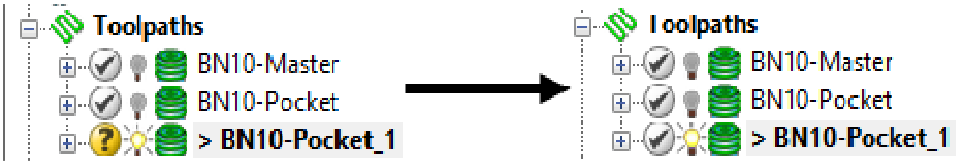



10 Enter **3** for number of copies and a pitch of **80** for both in **X** and **Y** before selecting **Accept**.

11 In the **Transform** toolbar, click ✓ 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 re-calculated.

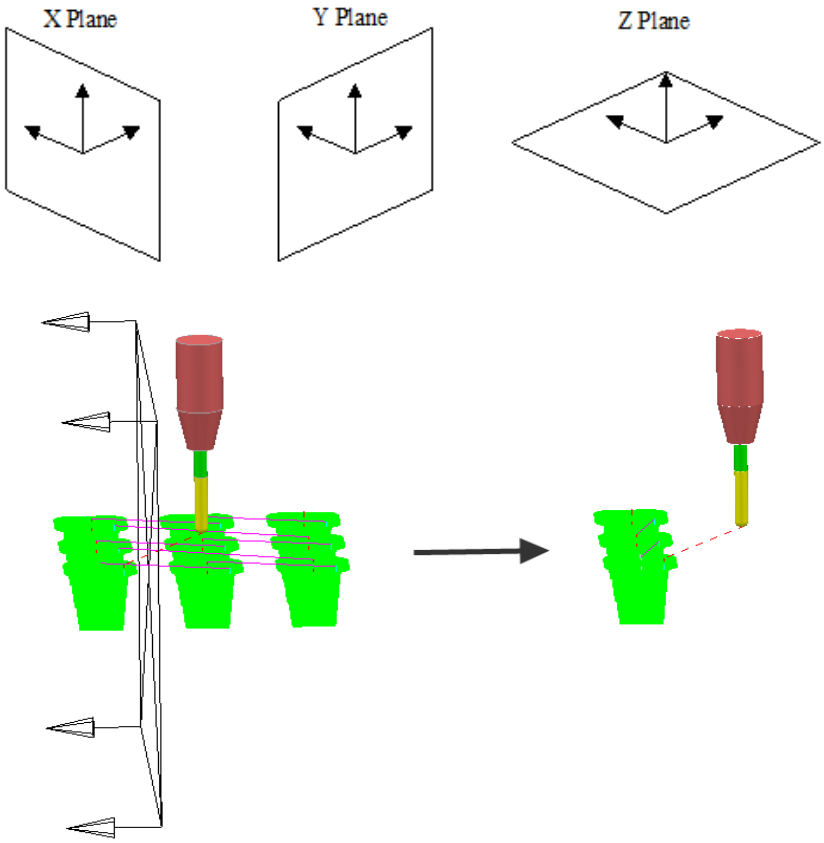
For more permanent toolpath limiting use a **Boundary** within the original **Strategy** calculation.

A typical application for retrospective **Toolpath** limiting would be the (one off) re-machining of a local area that has been welded.

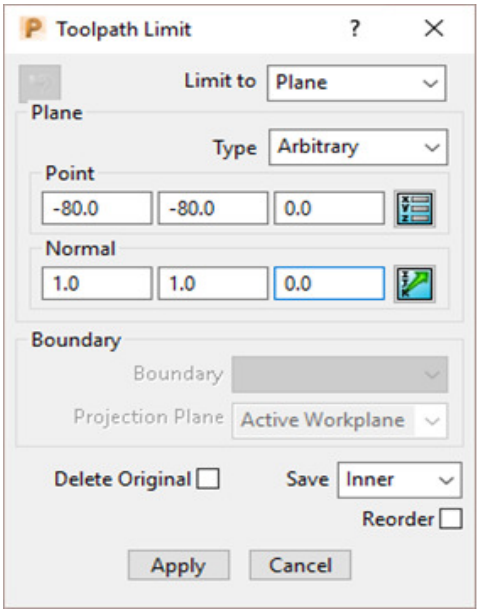

Copyright © Autodesk

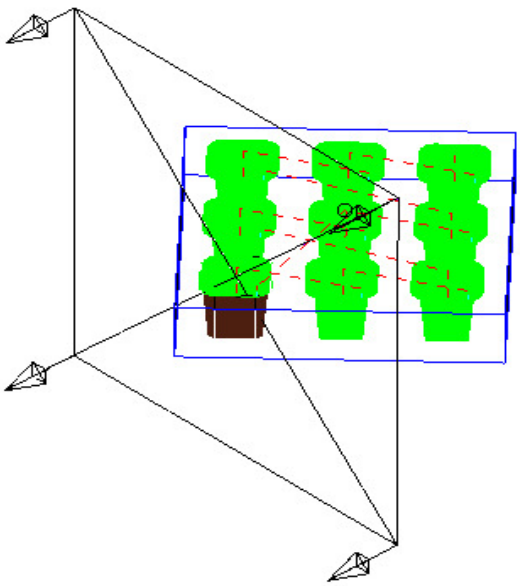
# Edit > Limit > To a Plane

**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.



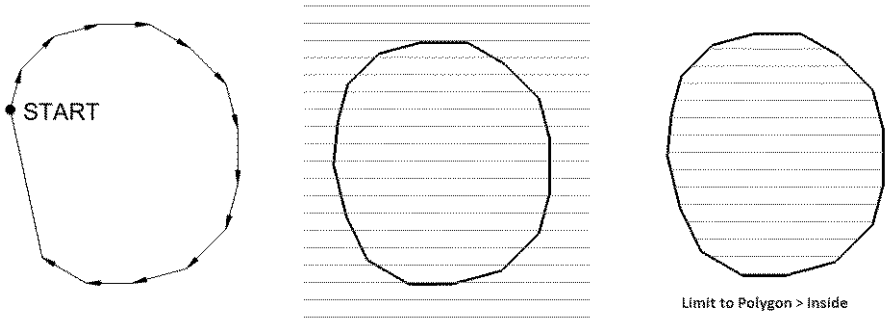



# Edit > Limit > to a Polygon

Using the mouse, you can sketch polygons with any number of sides.

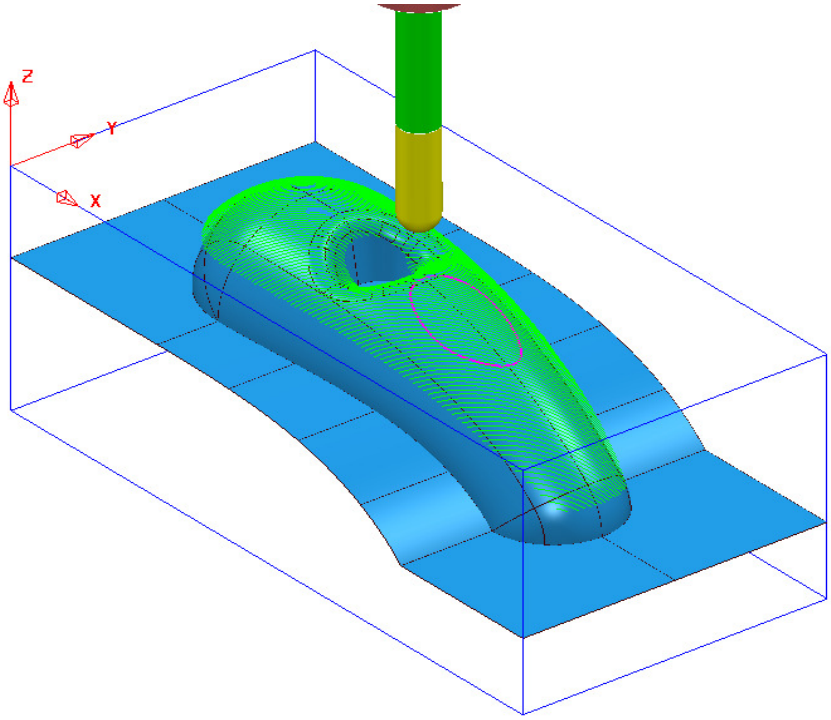
This allows complex areas to be defined, with the option to save the Inside, Outside, or Both sides of the polygon.

*Before making a polygon ensure that the Intelligent cursor is switched off.*



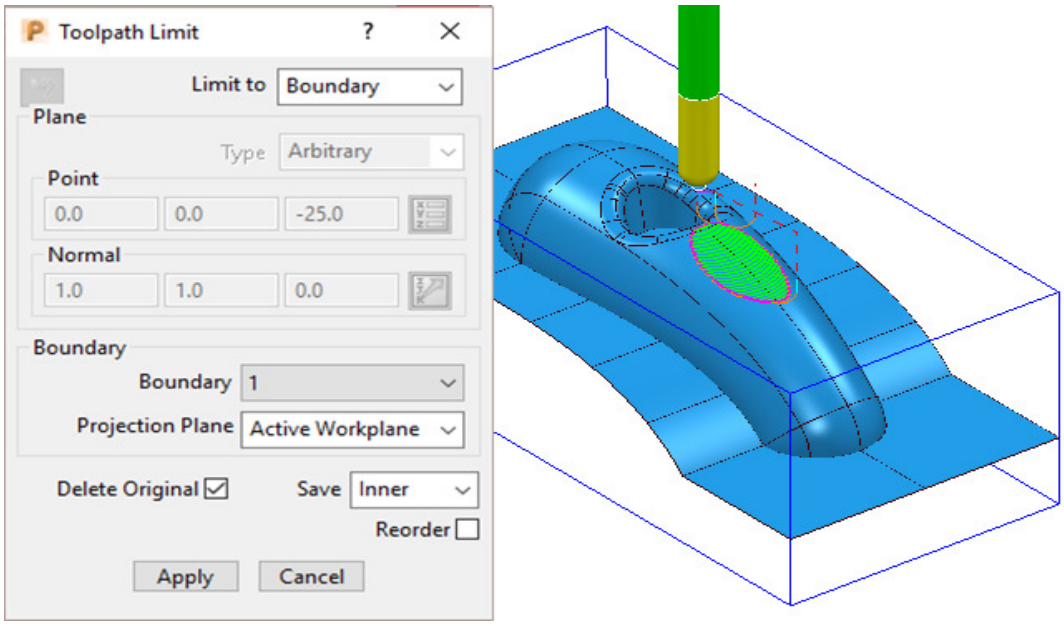

# Edit > Limit > to a Boundary

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.





Copyright © Autodesk




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.

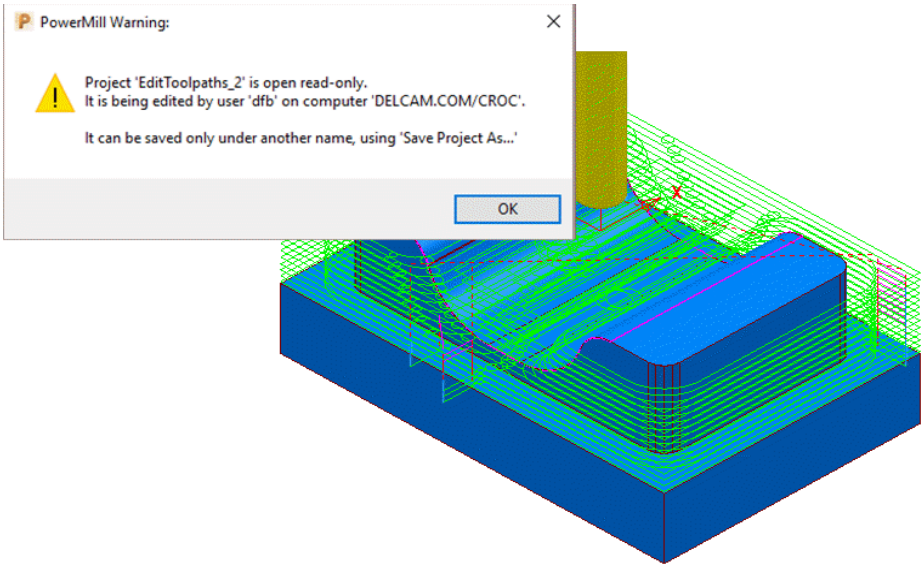
# Changing the Order and Direction within Toolpaths – EX1

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.

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

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

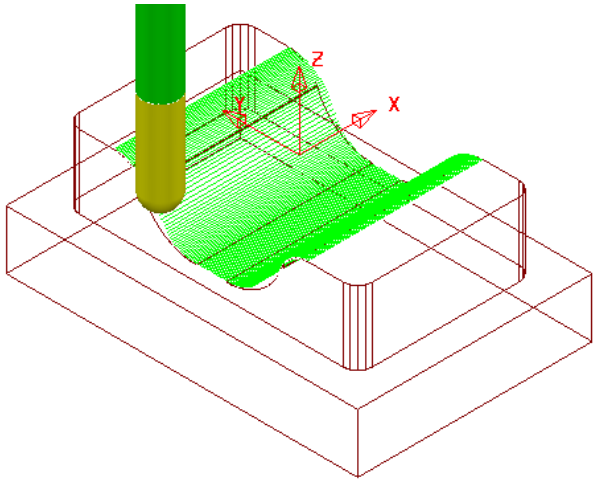
**.....\PowerMill_Data\Projects\EditToolpaths_2**




16 Select **File** > **Save Project As**:

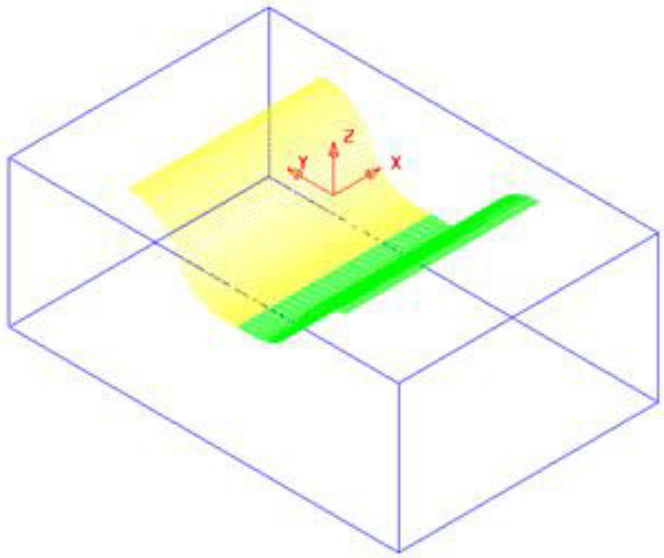
**.....\COURSEWORK\PowerMill-Projects\EditToolpaths-EX1**

17 In the **PowerMill *explorer***, right mouse click on the toolpath **bn12-finish-a1** and then from the local menu select **Simulate from Start**.



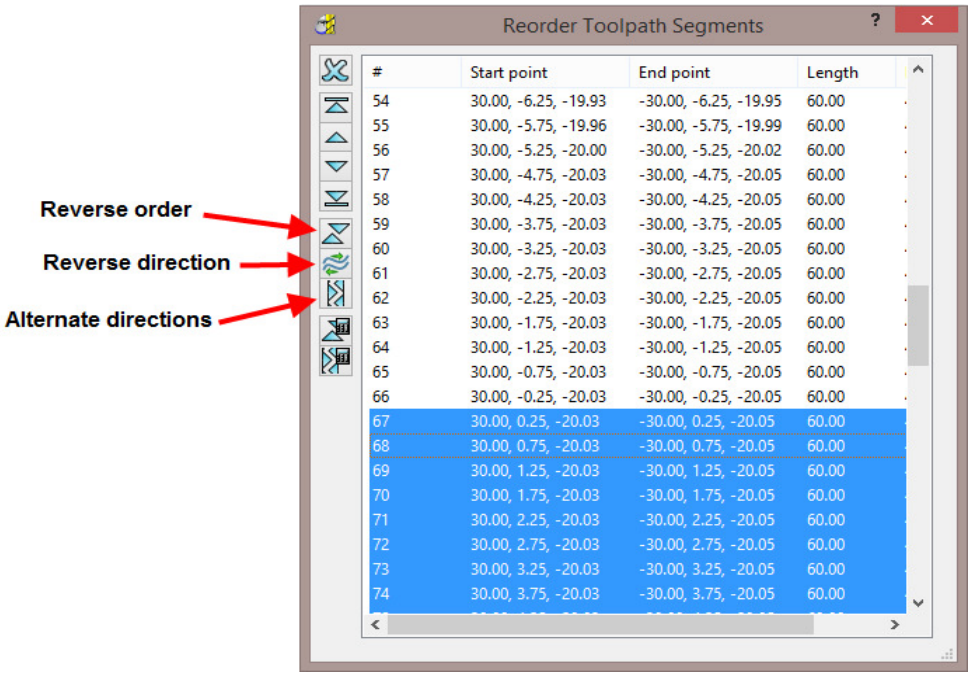

The tool ***Climb Mills*** across the form, stepping downwards to the base before stepping up the other side.


Copyright © Autodesk




18 Left-click to select all tool tracks on **bn12-finish-a1** beyond the centre of the form (Yellow in the illustration above).

19 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.




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.

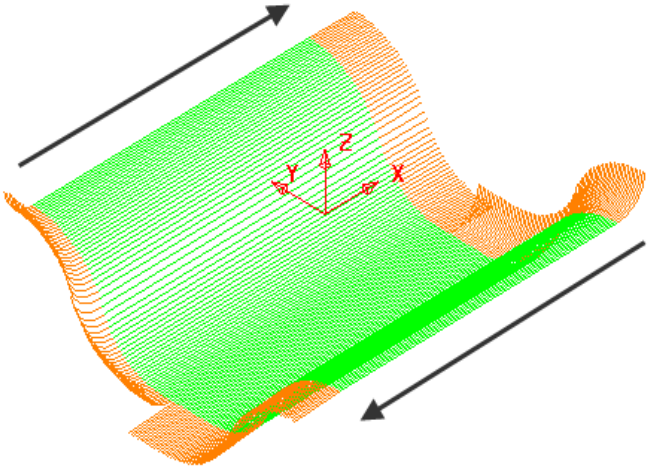
20 Click on the **Reverse Order** button to reverse the sequence of the selected tool tracks.

21 Click on the **Reverse Direction** button to reverse the direction of each selected tool track.

22 Open the **Leads and Links** dialog and **Apply** with the following settings:-

**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.*

23 **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\EditToolpath-EX1**

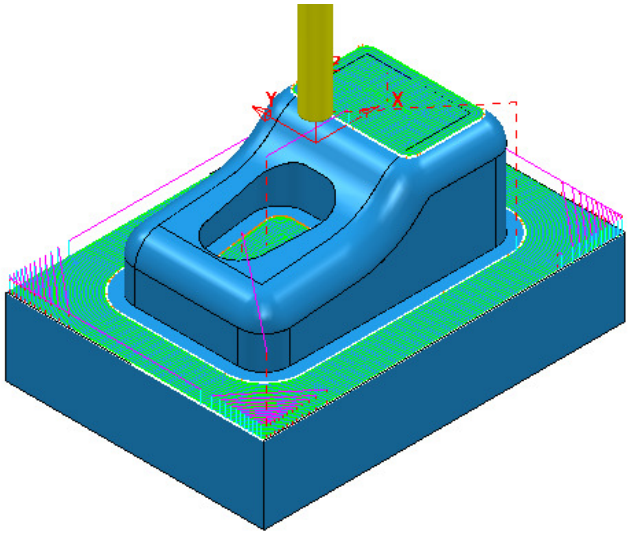

Copyright © Autodesk

# 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**

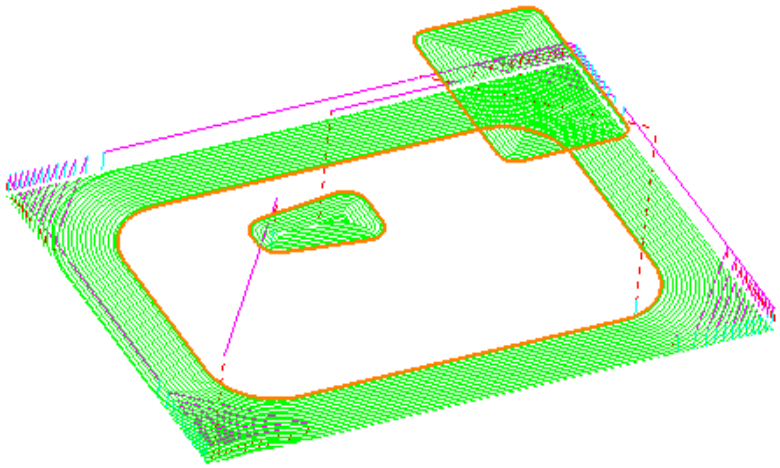


3 **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\EditToolpaths-EX2**

The **Project** includes a **3D Offset** finishing toolpath **D10TR1-FIN1** controlled by both **Boundary** and **Pattern** segments.

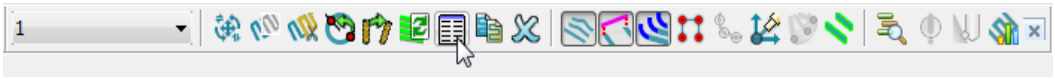
4 **Simulate** the **Toolpath** using a slow setting.



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

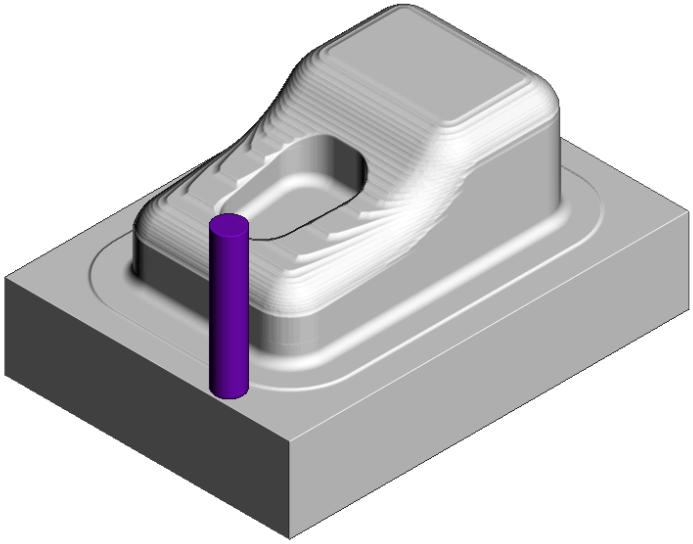
5 From the **main** menu, select **View** > **Toolbar** > **Toolpath** to display the **Toolpath** toolbar.



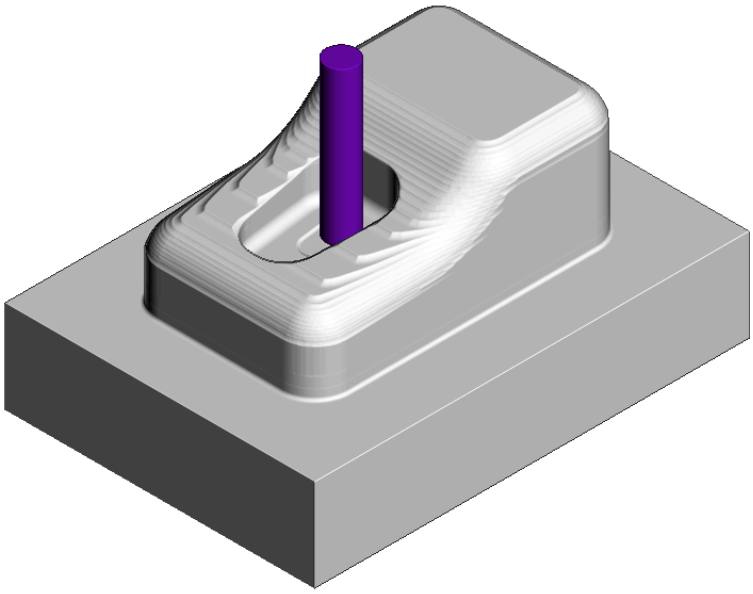
6 Select the **Reorder toolpath** option .

7 Select all tool tracks in the lower area and select **Reverse Order** . This part of the strategy should now climb mill inwards, towards the main component dialog.

8 If it is not climb milling, then apply **Reverse Direction** .



9 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.




Copyright © Autodesk

# 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**.

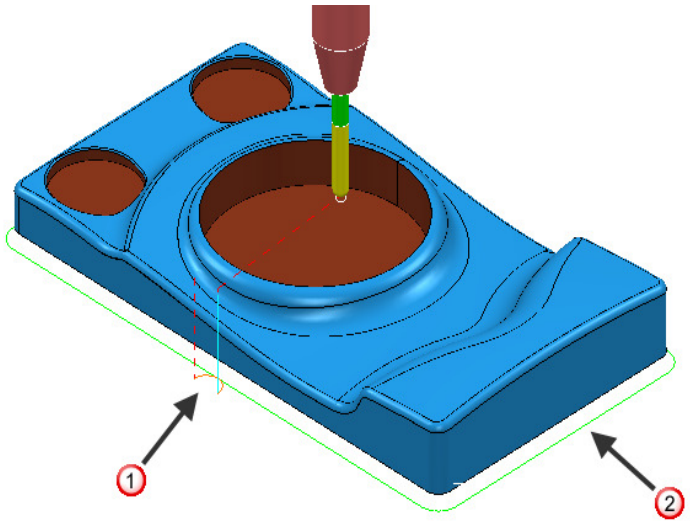
2 Select **File** - **Open Project**:

**.....\PowerMill_Data\Projects\limiting-example**

3 **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\MoveStartPoint-EX1**

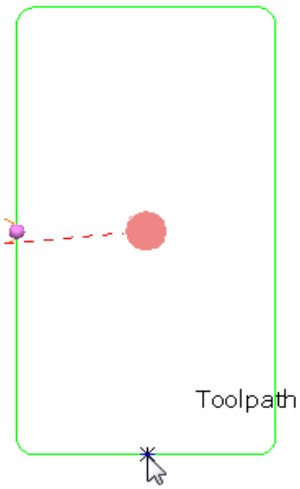
4 Select an **ISO1** view.



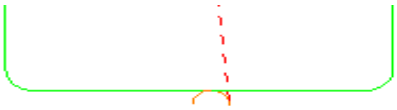

The toolpath start point (1) needs to be moved to position (2).

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 to accept the change.



This has moved the toolpath start point.

8 Select **Save Project** to update:-

**.....\COURSEWORK\PowerMill-Projects\MoveStartPoint-EX1**

# Moving multiple start points

It is possible to apply the **Move start points** options on the toolpath to multiple positions.

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

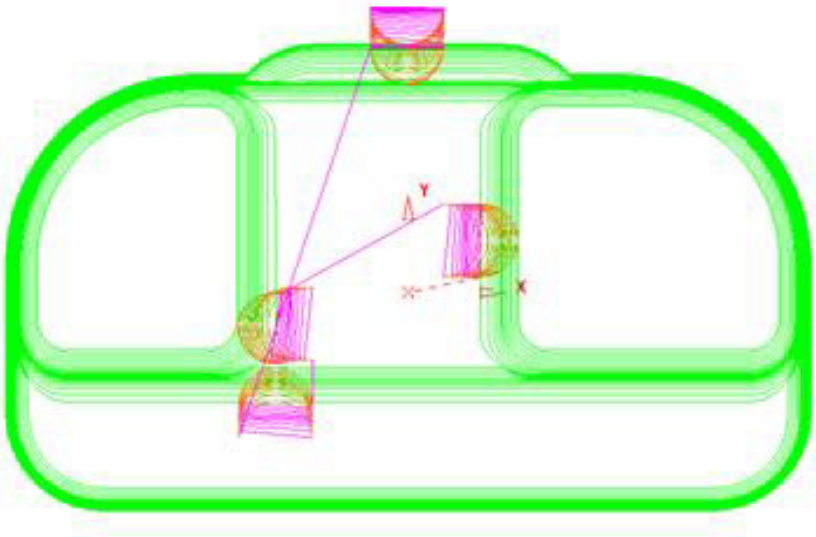
2 Open the **Project**:

**.....\PowerMill_Data\Projects\PunchForm10-Start**

3 **Save Project As**:

**.....\COURSEWORK\PowerMill_Projects\PunchForm10**

4 Select a **View from top (Z)**.



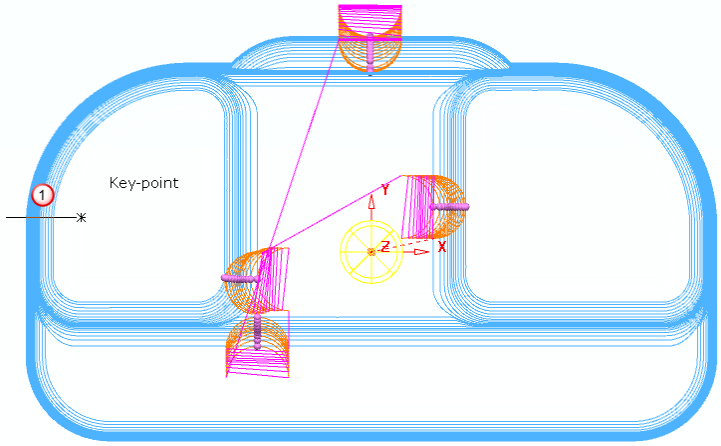

5 Right-click the active **toolpath** and from the local menu select **Edit** > **Move Start Points** (The **Move Start Points** toolbar will appear).

6 If not already active on the toolbar, click the **Move Start Points by drawing a line** button (shown arrowed above).

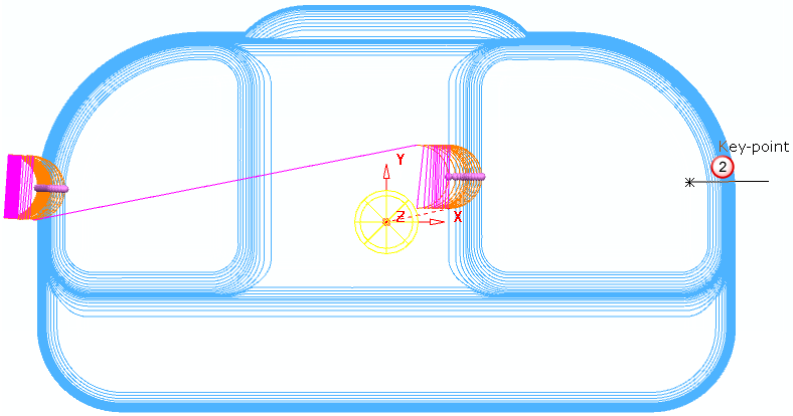

Copyright © Autodesk

7 On each of the three locations shown below, in the listed order, click and select 2 Key-points on both sides of the tool tracks.

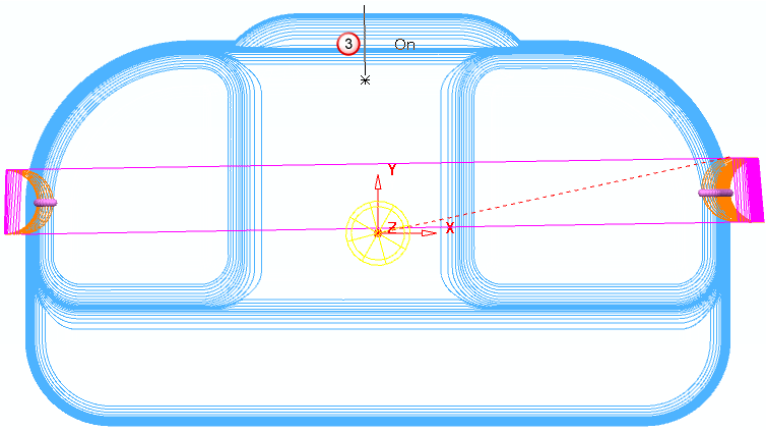
1 - First move point selection points.



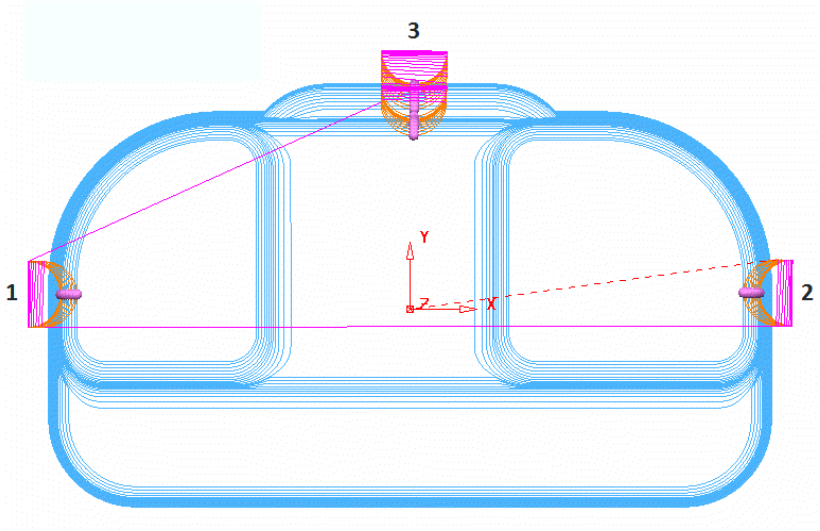

2 - Second move point selection points.




3 - Third move point selection points.



The final output looks similar to this:




*The tool tracks are coloured blue while the Move Start Points toolbar is active.*

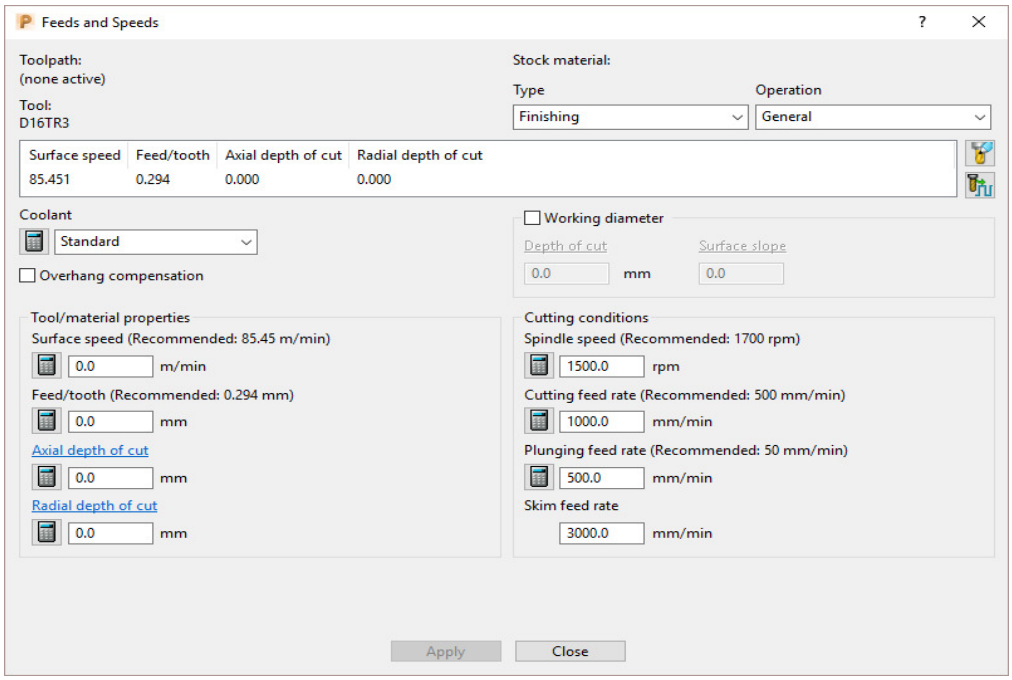
8 Click on the **Move Start Point** toolbar to accept the changes.

9 Save the **Project** to update:-

**.....\COURSEWORK\PowerMill-Projects\ PunchForm10**

# Feeds and Speeds

You can modify the values in the **Feeds and Speeds** dialog and retrospectively **Apply** them to the **Active toolpath**.



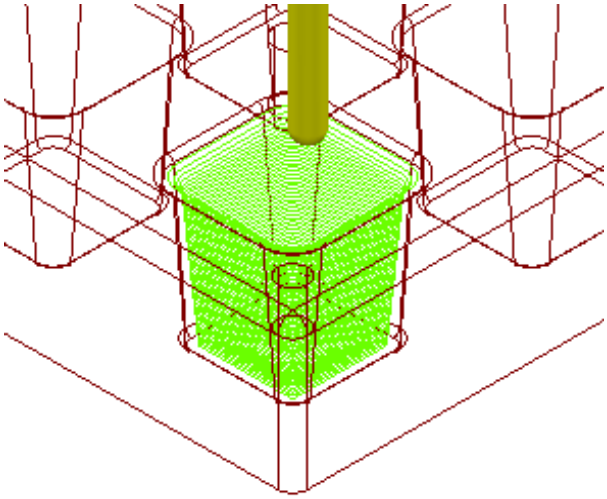


Copyright © Autodesk

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

11 Select **File** > **Open Project**:

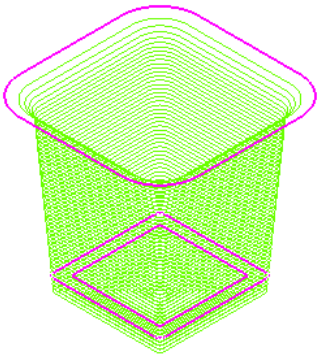
**.....\PowerMill_Data\Projects\ExtraFeedrates-Start**



12 Select **File** > **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\ExtraFeedrates-Example**

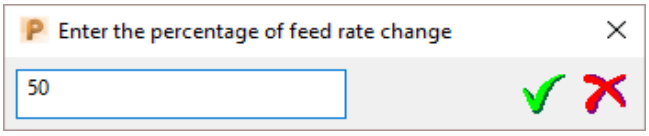
13 Undraw the **Model**, **Block**, **Workplane**, and **Tool** to display just the **Toolpath** and **3 Boundaries**.



14 **Activate** both the **Toolpath** and **Boundary 2** (The middle one of the 3).

15 Right-click the **Toolpath** and select **Edit** > **Update Feed Rate Inside Boundary**.

16 Enter value as **50** and click ✓ to apply the percentage difference to the Feed Rate within the **Active Boundary**.

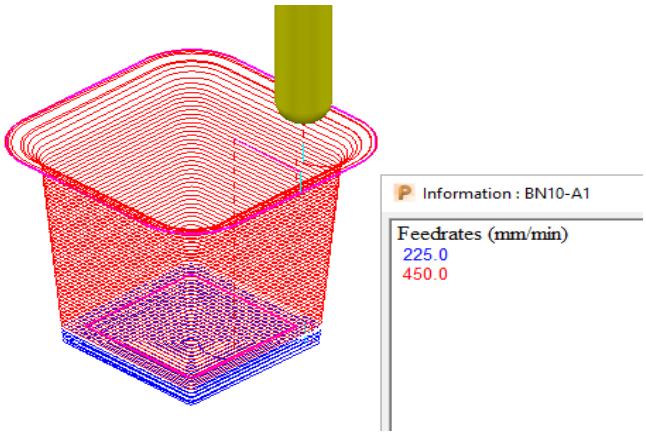



The **Toolpath** is split into two different coloured areas that are allocated a separate **Feed Rate** as a percentage of the original. The colour coded **Feedrates** are displayed in an Information dialog that appears automatically.

*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***.

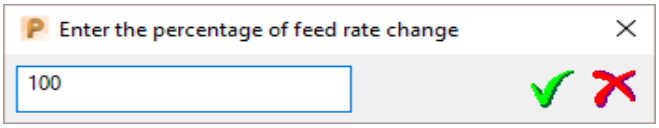



*The original Cutting Feedrate assigned to the toolpath was **450 mm/min.***

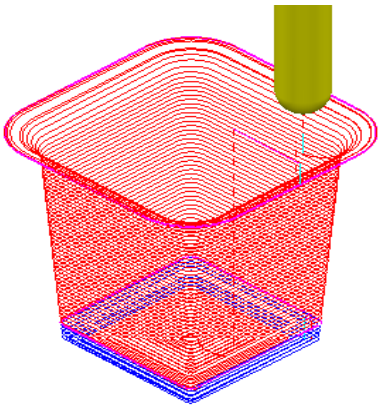
17 **Activate** both the **Toolpath** and **Boundary 3**.

18 Right-click the **Toolpath** and from the local menu select **Edit** > U**pdate Feed Rate Inside Boundary**.

19 Enter the value **100** and click to apply the percentage difference to the **Feed Rate** within the **Active Boundary**.




The central part of the **Toolpath** area, clear of the side wall, has now changed back to the nominal **Feed Rate** value.



In this case, the nominal **Feed Rate** is **450 mm/min**, and the modified section (near the intersection of the Base and Sidewall of the pocket) is reduced by 50% to **225 mm/min**. The colour of the values displayed in the Information dialog is the same as the colours displayed on the toolpath.


Copyright © Autodesk

# 9. Tool Collision Checking

## Collision Checking

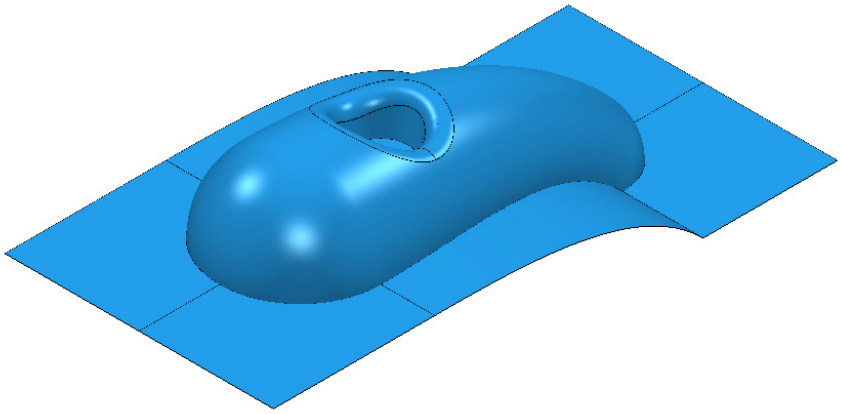
**Collision Checking** can either be applied retrospectively or directly during toolpath creation, as long as a tool Shank and Holder are defined with the ***Active*** **Tool**.

### Retrospective Collision Checking (Verify -Toolpath)

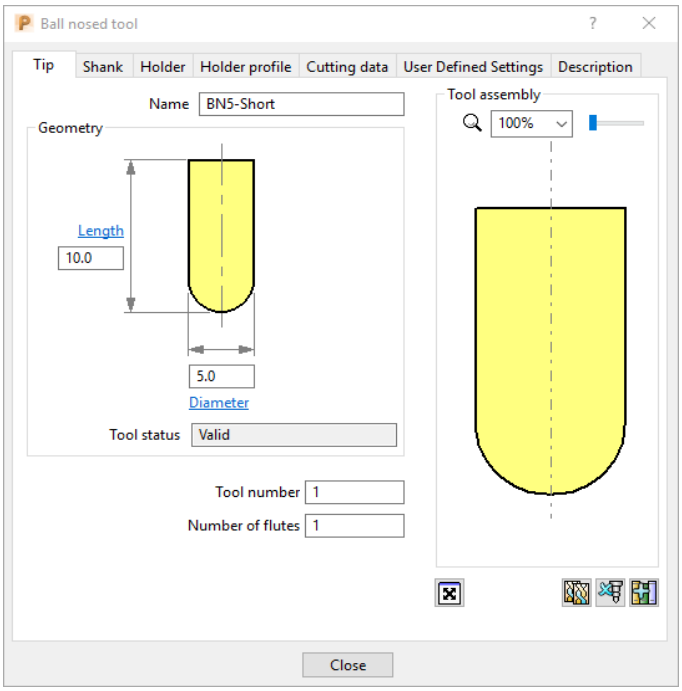
If the **toolpath** menu option, **Verify > Toolpath -** (***Collisions***) is applied, two additional toolpaths will be created from the original, one being collision safe, and the other being in collision. At the same time a copy of the **Tool** complete with appropriate modifications will be substituted 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**



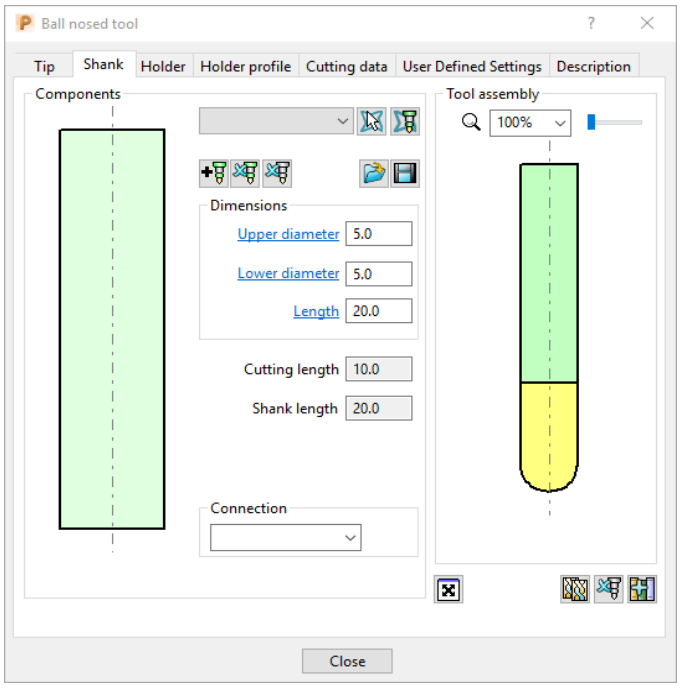
3. Calculate the **Block** to **Min/Max** limits.
4. Calculate both the **Rapid Move Heights** and the tool **Start and End Points**.
5. Create a Dia 5mm Ball Nose tool named **BN5-Short** with a length of **10** and tool number of **1**.




6 Select the **Shank** tab.

7 Select **Add a shank component** .

8 Fill in the dialog exactly as shown in the image below.

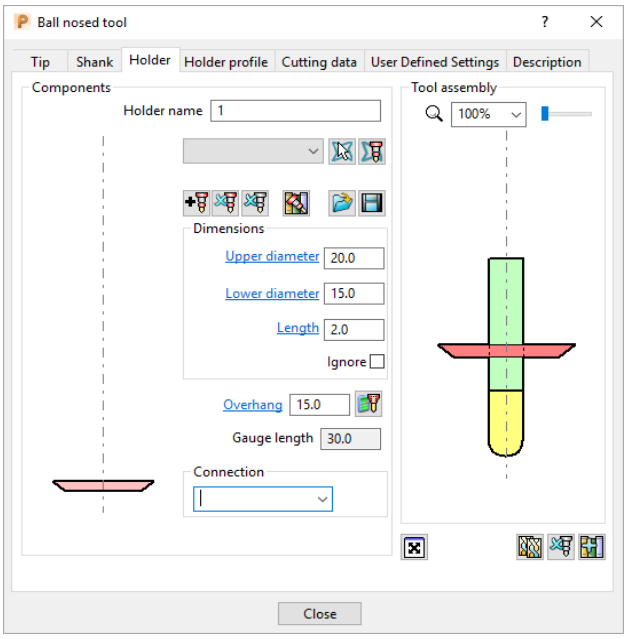



9 Select the **Holder** tab.

10 Select **Add** holder component .

11 Fill in the form for the first **Holder** element exactly as shown in the following image.


Copyright © Autodesk

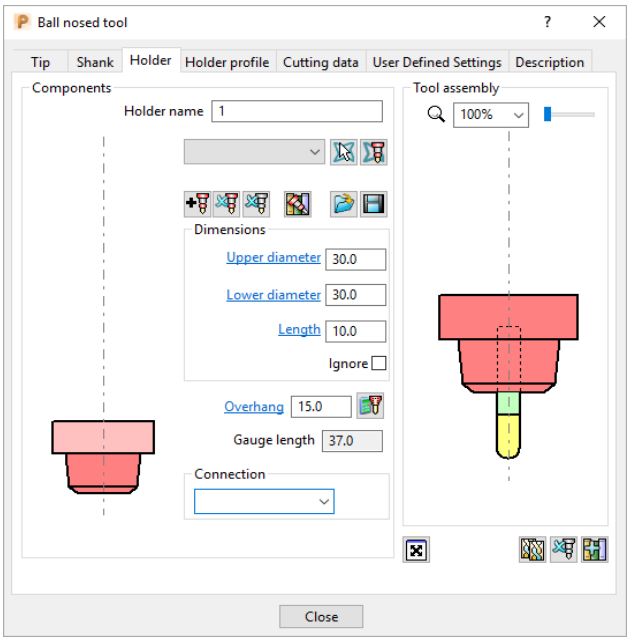



12 Select **Add holder component** .

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**.

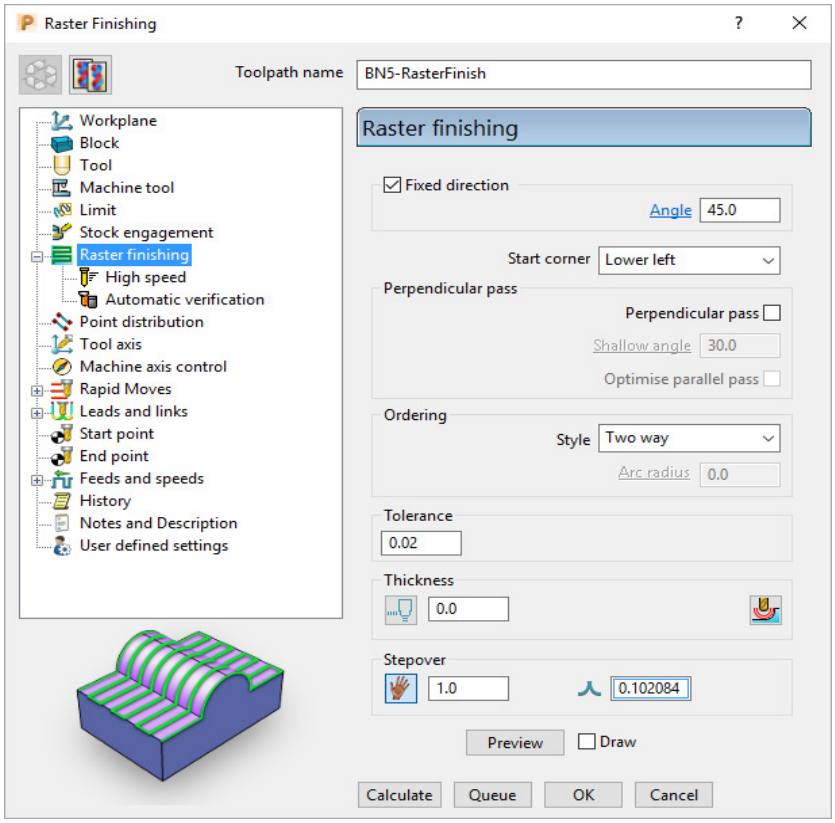



*If it is required to modify or delete a Holder element, select (left click) on one of them in the assembly illustration and it will turn pale. The current values for that element will be displayed and available for modification.*

*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.




18 Enter **Toolpath name** as **BN5-RasterFinish**, **Angle 45**, and **Stepover 1**.

19 Select **Start Corner** - **Lower Left**.

20 Select **Style** - **Two way**.

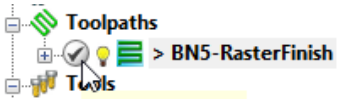
21 In the **Leads and Links** , set both **Lead in** and **Lead out** with ***1st choice*** as **Surface normal arc** with **Angle 90** and **Radius 3**, with ***Apply constraints*** set to **Distance > 4**.

22 In the **Links** page:-

- For ***1st choice*** enter **Circular Arc** with **Apply constraints** set to **Distance < 4**.
- Set ***2nd choice*** to **Skim**, and ***Default*** to **Incremental**.

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

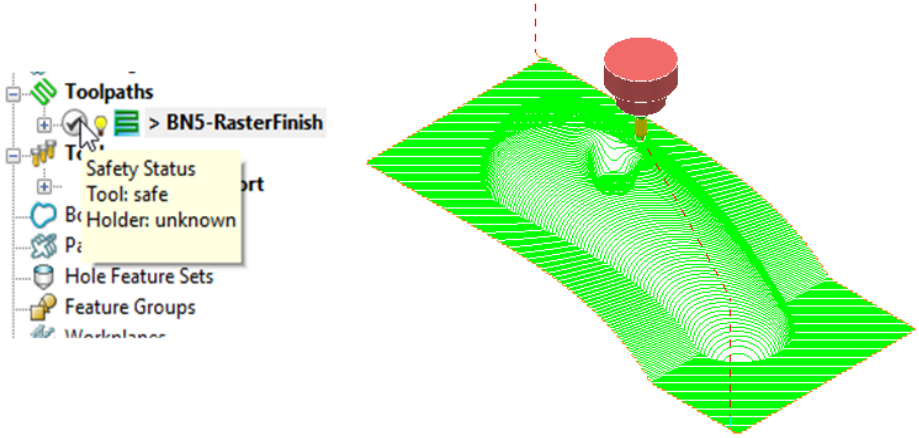
24 In the **PowerMill *explorer*** hold the cursor over the small box to the left of the toolpath lightbulb symbol.





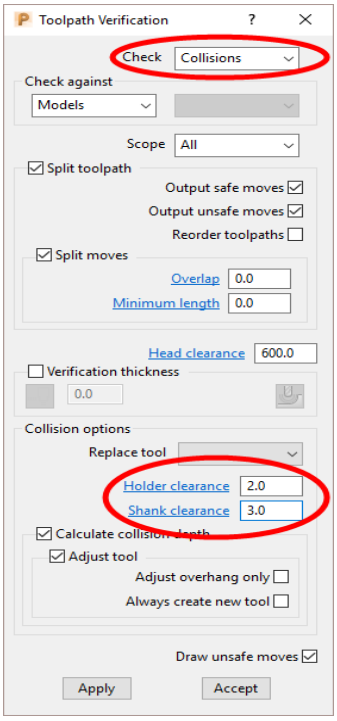
Copyright © Autodesk

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**.

25 From the **main** toolbar select **Toolpath Verification** .



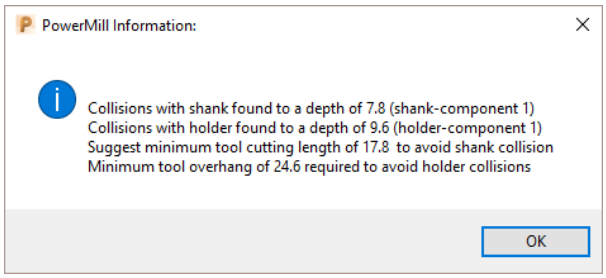

The two available, independent **Check** options are **Collisions** and **Gouges**.

**Shank Clearance** and **Holder Clearance**, allow the user to specify a value for an additional, safe thickness around the tool **Shank** and **Holder** elements.

26 In the ***Check*** box select **Collisions**.

27 Input the values ***Shank Clearance* 2** and ***Holder Clearance* 3**.

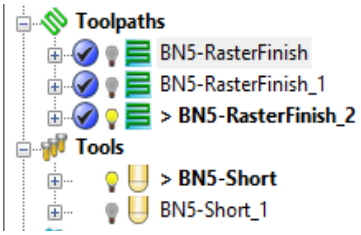
28 Click **Apply**.




The above **PowerMill Information** box appears with information on the status of the original tool and required dimensional changes. The default settings will result in a modified **tool** (**BN5_1**) being created along with 2 new **toolpaths** (The collision and non-collision parts of the original). The new **tool** (**BN5-Short_1**) will then automatically exist in both the original and new (collision warning area) toolpath. The original **tool** (**BN5-Short**) will remain in the new (collision safe area) toolpath.

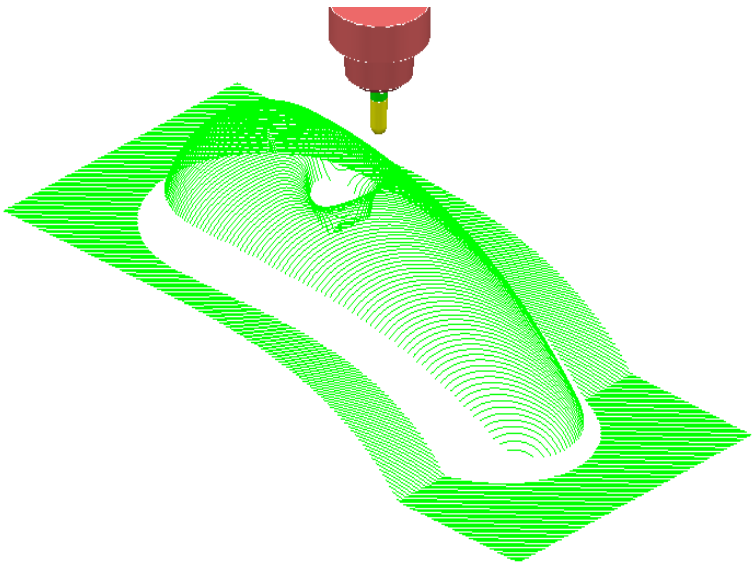
29 Click **OK**.

30 Click **Accept** on the **Toolpath Verification** dialog.




**PowerMill** has created two new toolpaths, **BN5-Raster Finish_1** and **BN5-RasterFinish_2**. A new tool named **BN5_1** has also been created in the **Tools** area of the **PowerMill *explorer***. This new tool has been created with the new valid adjustments made i.e. **Overhang 26.4**

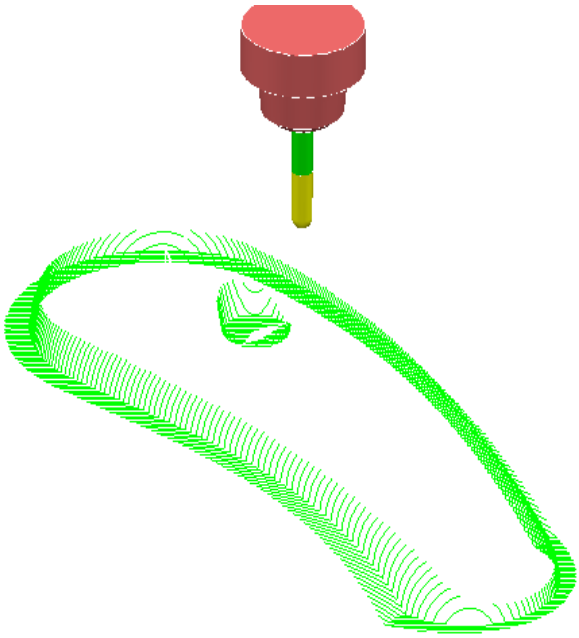
31 Activate the toolpath **RasterFinish_1**.



This **toolpath** contains the 'collision free' segments of **RasterFinish**, along with the original tool **BN5**.

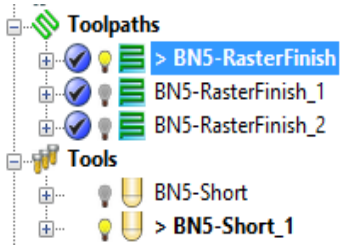

Copyright © Autodesk

32 Activate the toolpath **RasterFinish_2**.



This **toolpath** contains the 'collision' segments of the **Raster Finish** if it were machined with the original **BN5** tool.

33 Activate the original toolpath, **BN5-RasterFinish**.




***PowerMill** has substituted a newly created **Tool**, **BN5-Short_1** (with an extended overhang) into **BN5-RasterFinish** and **BN5-RasterFinish_2**.*

34 Select **File** > **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\ToolCollisionExample**

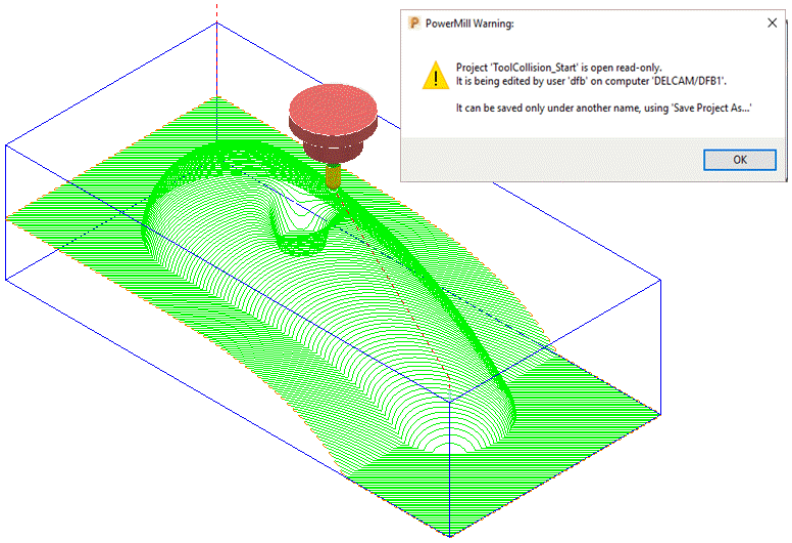
# Direct Collision Checking (Automatic Verification)

***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**:

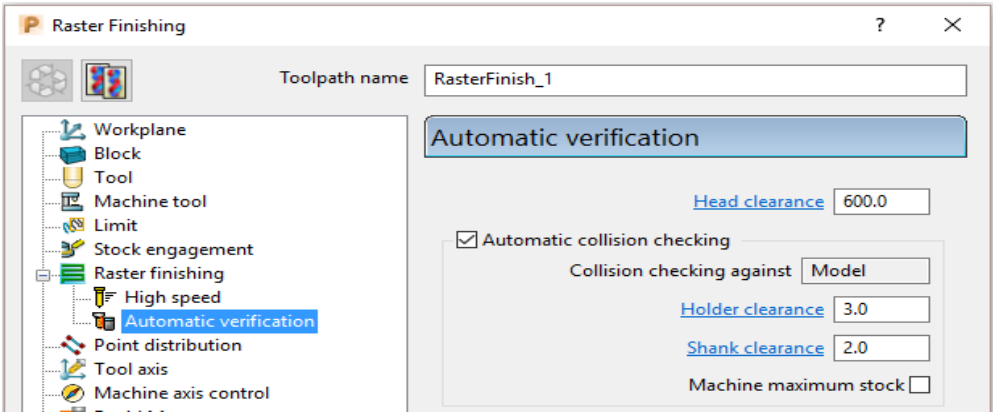
   **.....\PowerMill_Data\Projects\ToolCollision_Start**




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 to open the following dialog:

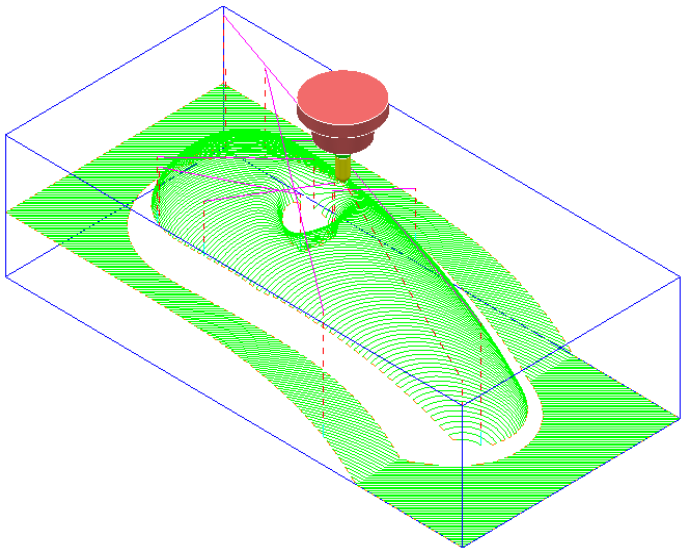




Copyright © Autodesk

8 Tick the **Automatic collision checking** box, then input ***Holder Clearance*** **3** and ***Shank Clearance*** **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**).*

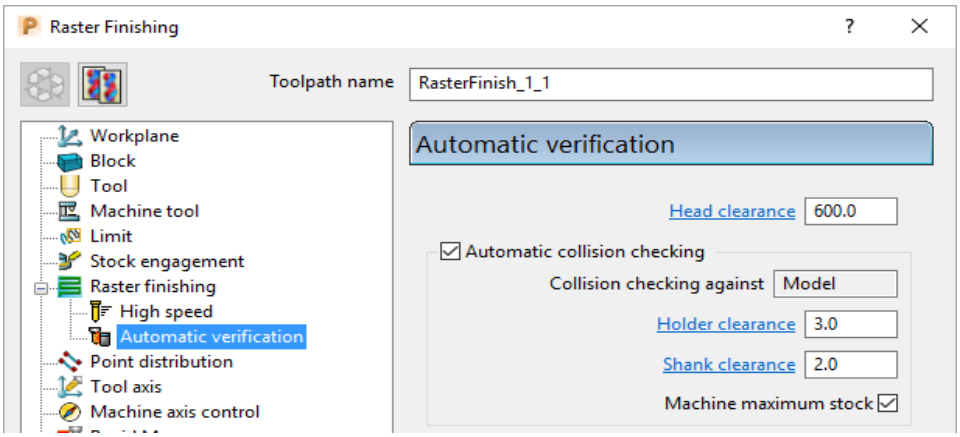
## Machine maximum stock

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***.

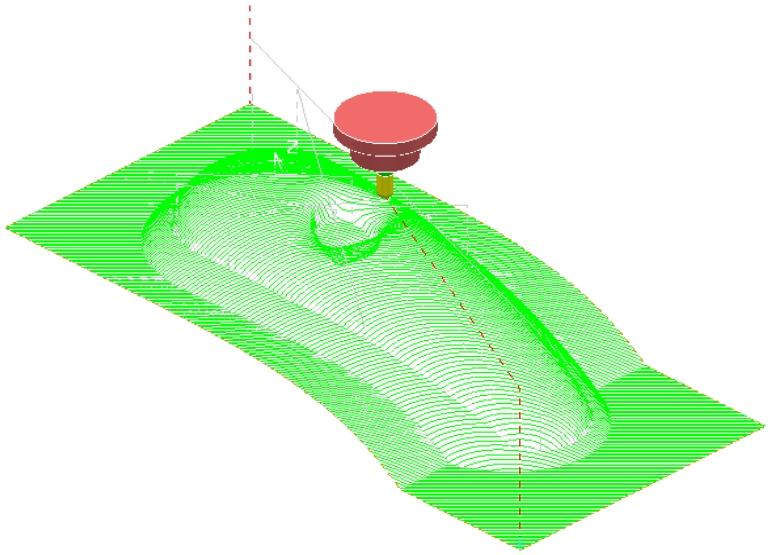
11 From the local menu, select **Activate** followed by **Settings** and in the **Raster Finishing** dialog select **Copy** toolpath .

12 With the **Raster Finishing** dialog open, select the **Automatic Verification** page to access the following dialog:



13 Tick - **Automatic collision checking**, input ***Shank Clearance* 2**, ***Holder Clearance* 3**, and tick - **Machine maximum stock**.

14 **Calculate** the **Raster Finishing** dialog to directly create a continuous, collision free toolpath, **RasterFinish_1_1**.



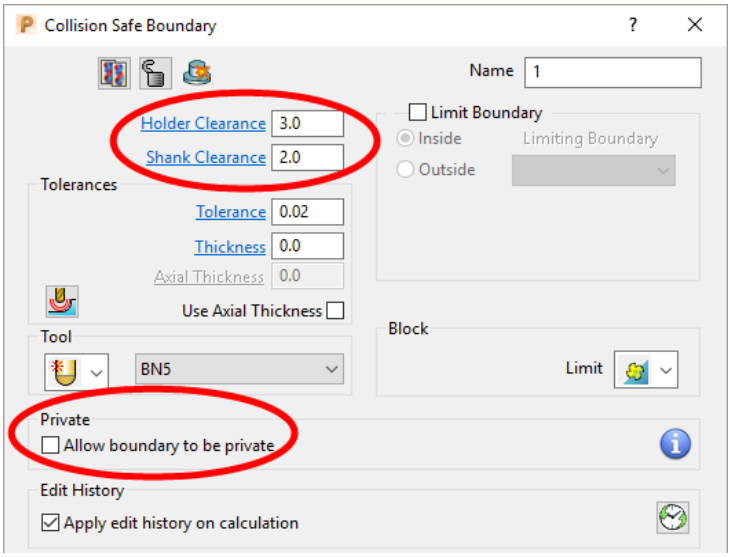
*An additional strategy with a compatible (collision free) tool is required to remove the remaining stock.*

## 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**.

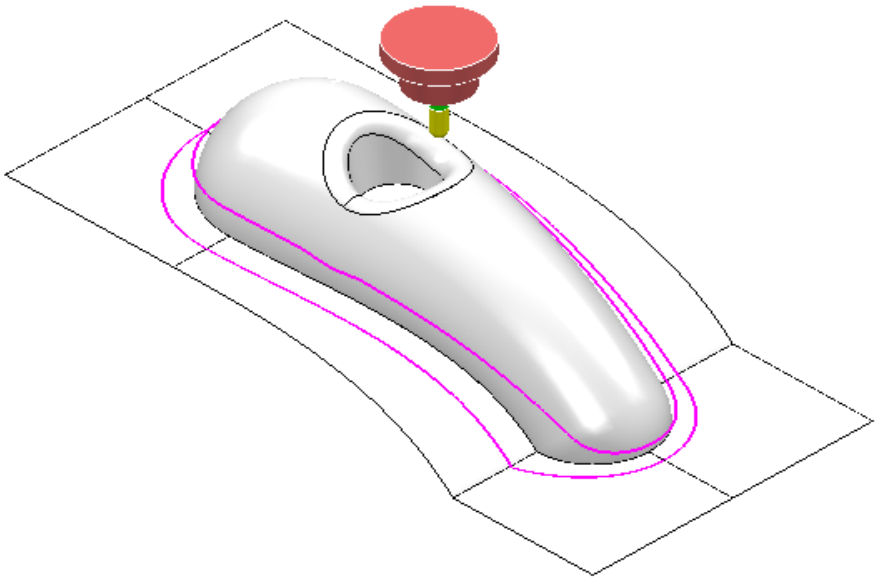
16 In the **PowerMill *explorer***, right-click Boundaries, and from the local menu select Create Boundary > Collision Safe.




17 Use the above settings and **Apply** the dialog.


Copyright © Autodesk

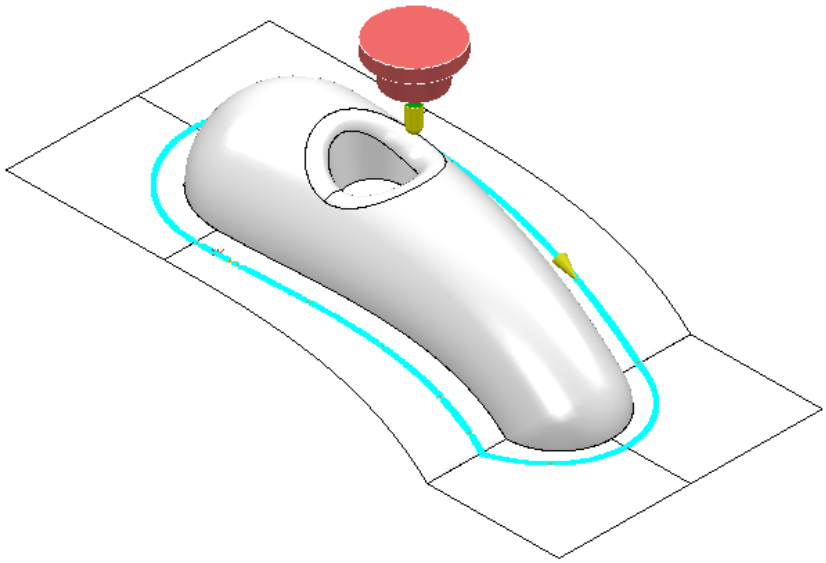
18 Select and **Delete** all segments except the 2 shown below.



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.

Copyright © Autodesk

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**.

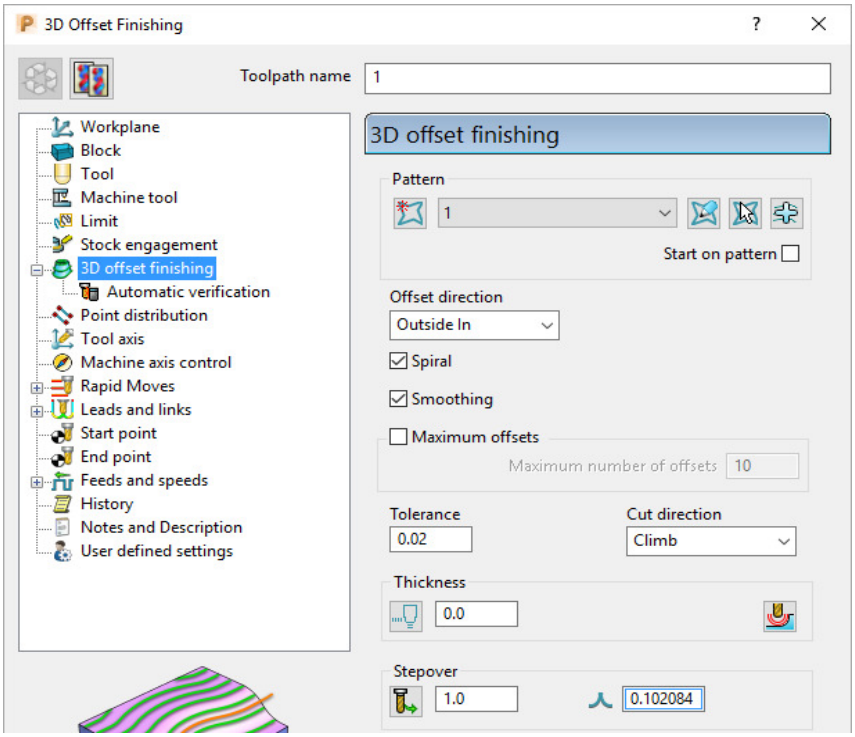
25 Select the **Holder** tab and change the **Overhang** to **30**.

26 Close the form.

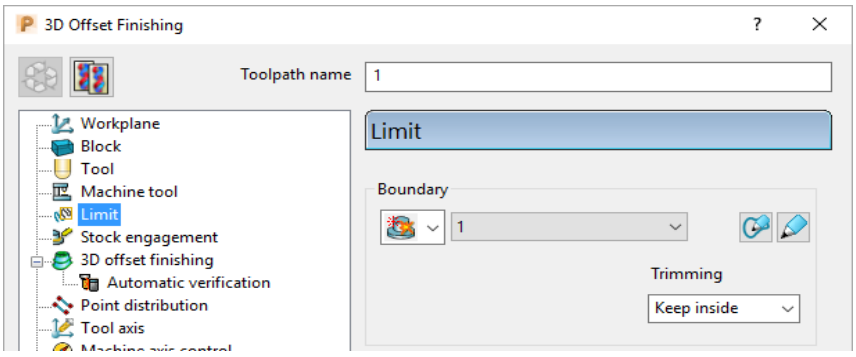
27 **Activate** the new 'long reach' tool, **BN5_1**

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.



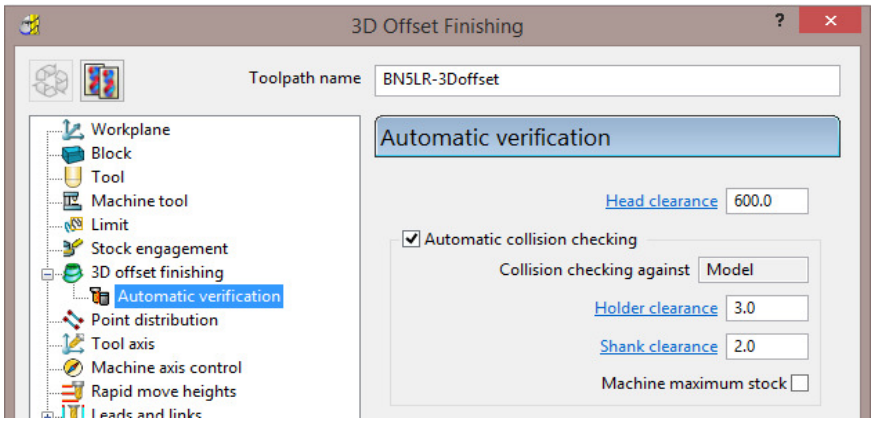

30 Select the **Limit** page and select **Boundary 1** with ***Trimming*** as **Keep inside**.



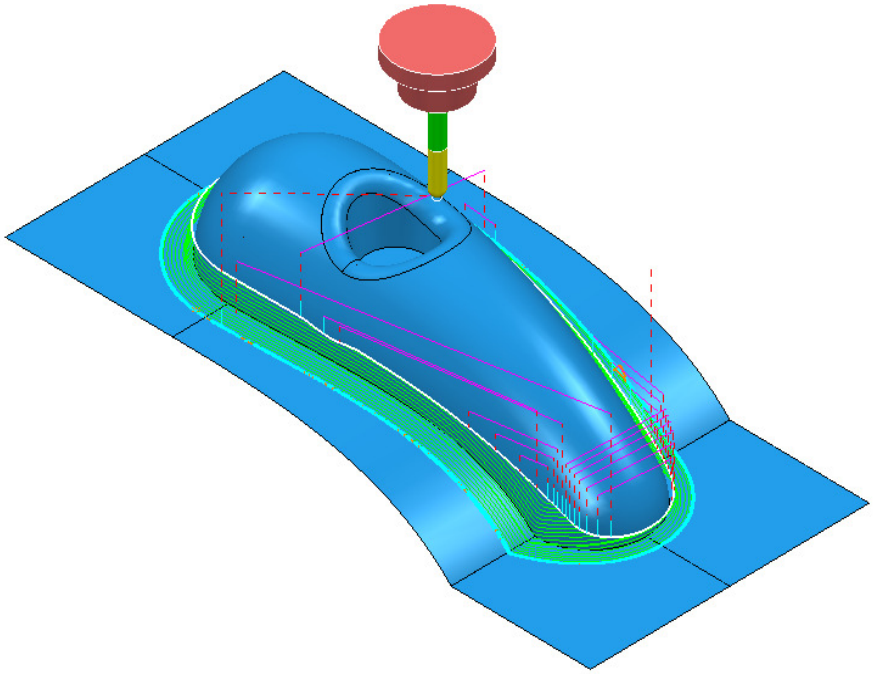


Copyright © Autodesk

31 Select the **Automatic Verification** page and tick **Automatic collision checking**, input **1** for both **Shank** and **Holder** clearances, un-tick **Machine maximum stock**.




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:

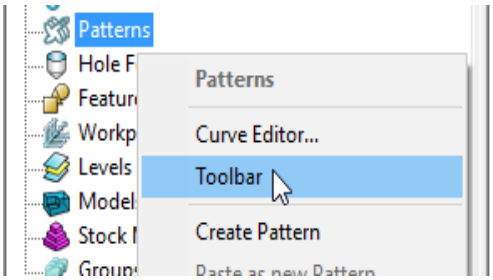
**.....\COURSEWORK\PowerMill-Projects\ToolCollision-Auto**

Copyright © Autodesk

# 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 activate other **toolbar** options.

- **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**.

- **Transform pattern**

- **Curve Editor**.

- **Clear**

*The **Pattern** segments can be edited directly from local menu options, or via the **Curve Editor**. Unlike **Boundaries**, **Patterns** can contain open segments.*

When an empty **Pattern** is created it is automatically assigned a number and made **Active** as shown in the **PowerMill *explorer***.

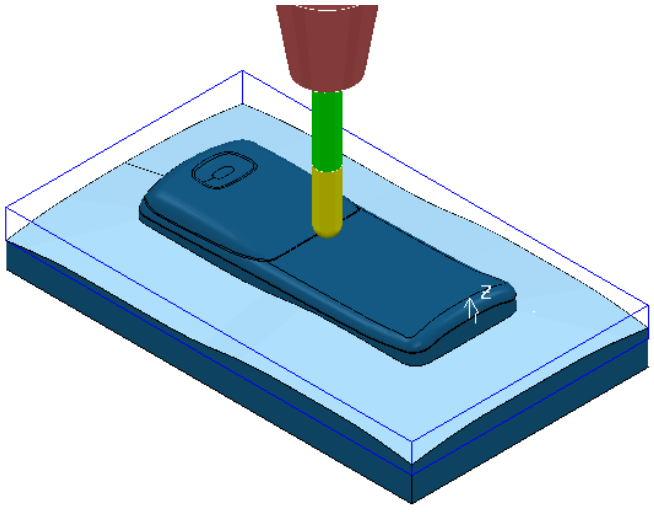
# Automatic Pattern Generation

The **Automatic Pattern Generation** icon opens the **Pattern Maker** dialog. This provides options for the user to construct a range of wireframe layouts that can be used in the **Pattern Finishing** strategy.

1. Select **File** > **Delete All** and **Tools** > **Reset Forms**.
2. **Import** the **Model**:

   **.....\PowerMill_Data\Models\phone.dgk**

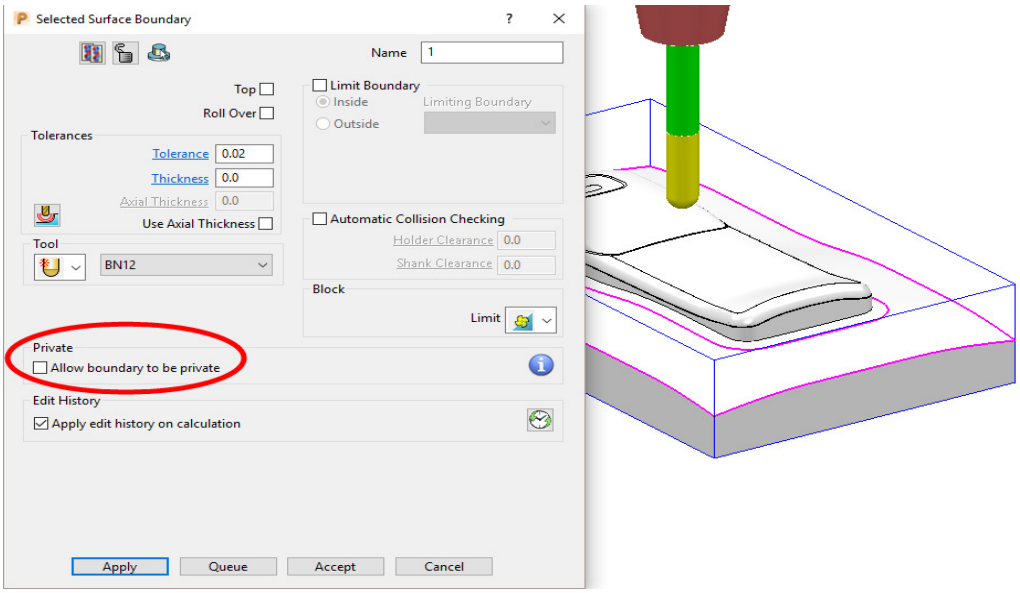
3. Select **Block** and **Calculate** using the default settings.
4. Define a dia 12mm Ball Nose tool named **BN12**.
5. Reset the **Rapid Move Heights**.
6. Select an **ISO1** view to display the **model**.



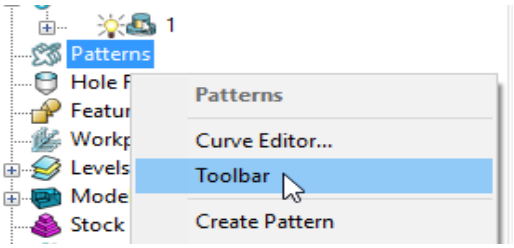
7. Select the 'shutout' surface as shown above (light blue).


Copyright © Autodesk

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.




9 **Apply** the dialog to create 2 **Boundary** segments.




10 In the **PowerMill *explorer***, right click on **Patterns** and from the local menu select **Toolbar**.

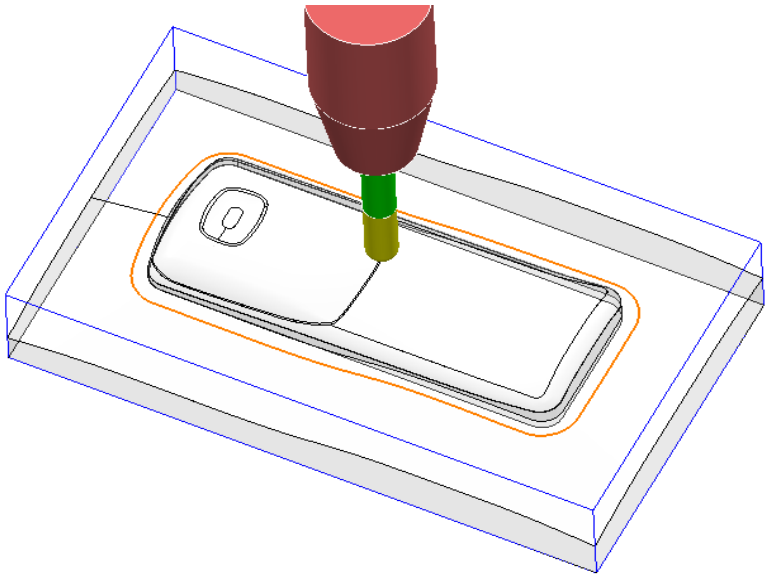
11 In the **Patterns** toolbar select, **Create a new (empty) Pattern** .

12 In the graphics area, select the inner **Boundary** segment.

13 In the **Patterns** toolbar select the option to **Insert Boundary into Active Pattern** .

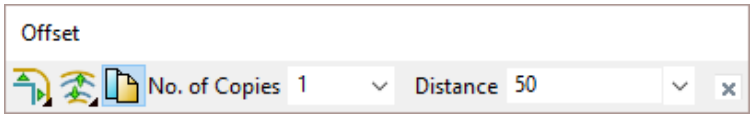
14 Delete the **Boundary**.

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**.

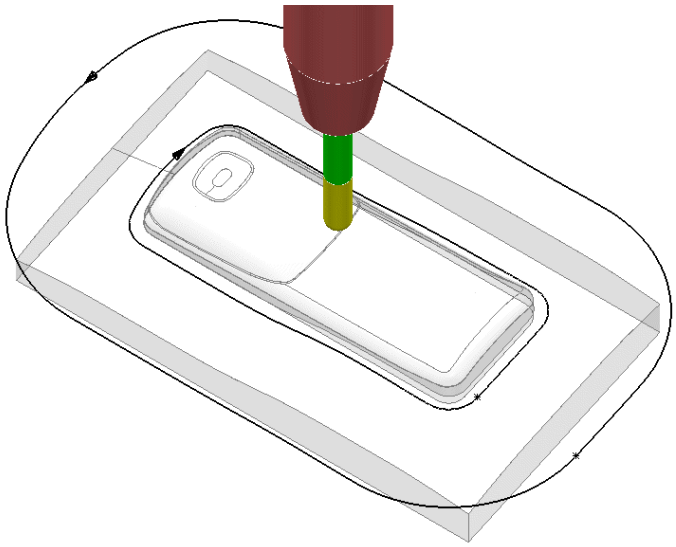
16 In the toolbar move the cursor over **Transformations** and from the attached vertical toolbar, select the **Offset** option .




17 Select **Keep Original** , the **Round Corners** option, and input **Distance 50** before clicking the ***Enter*** key.

18 In the **Curve Editor** toolbar, click to accept.


Copyright © Autodesk



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 inner **Pattern** segment and from the local menu select **Edit** > **Reverse Selected** (Both **Pattern** segments should now run in the same direction).

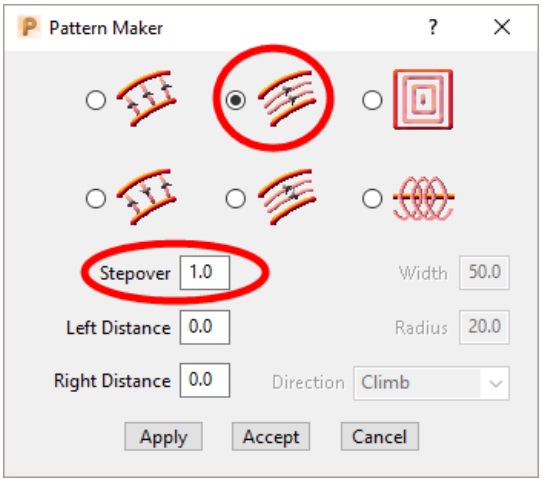
The **Automatic Pattern Generator** will be used to create a new **Pattern** with a series of segments merged between the 2 segments from the original **Pattern**. The resultant **toolpath** will follow the contour of the component form as well as creating corner free, tool tracks with a constant material removal rate.

21 From the **Pattern** toolbar, select **Create Pattern** to create a new **Pattern**.

22 Select **Automatic Pattern Generation** .

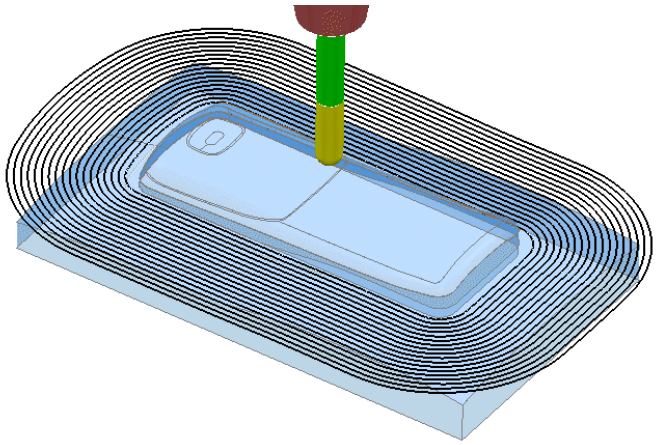
23 With the new (empty) **Pattern** activated and the original **Pattern** displayed, select both visible segments.

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.

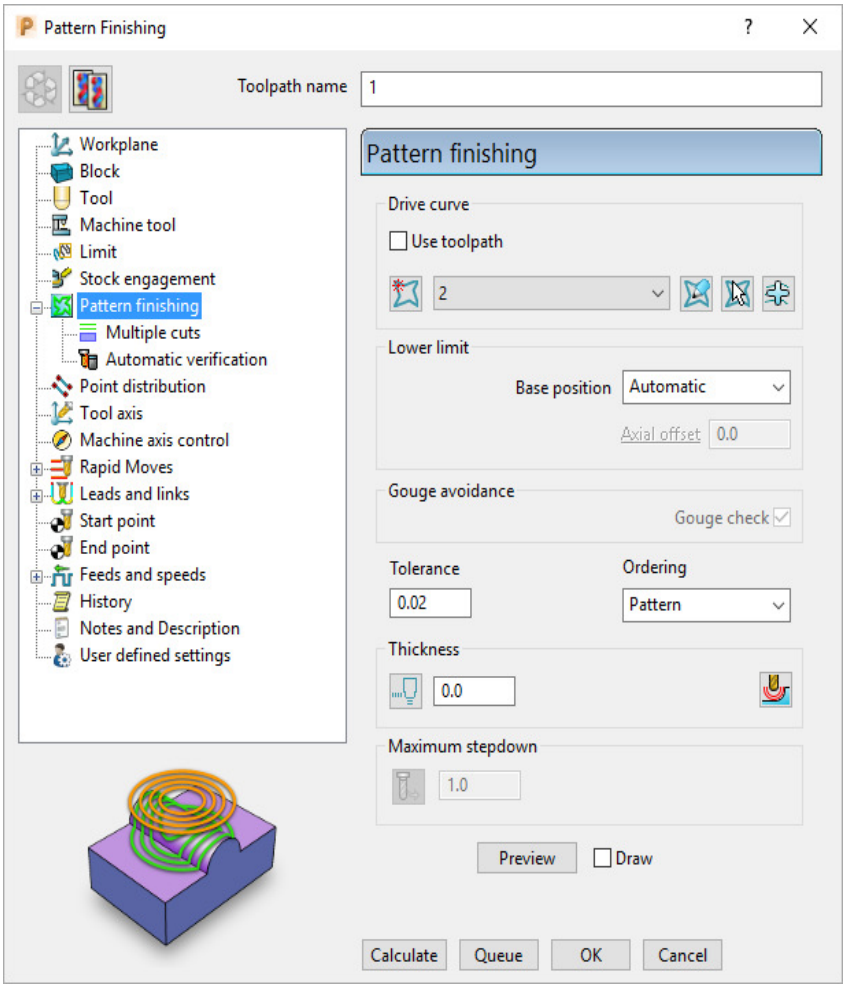
25 Click **Apply**, followed by **Accept**.



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).

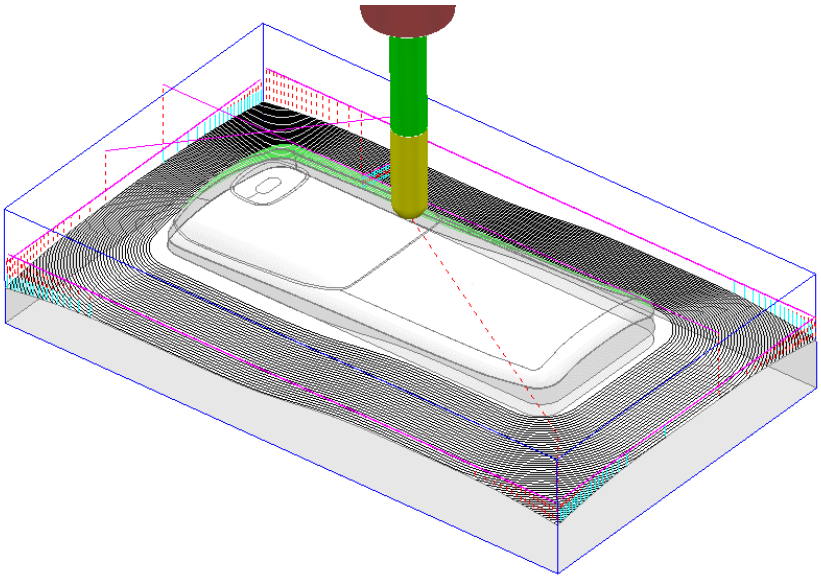
The new **Pattern** will now be used in a **Pattern Finishing** strategy.

26 In the **Strategy Selector**, **Finishing** options, select a **Pattern Finishing** strategy.




27 Enter the data into the above form exactly as shown above, click **Calculate**, and then **Close** the form.


Copyright © Autodesk



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

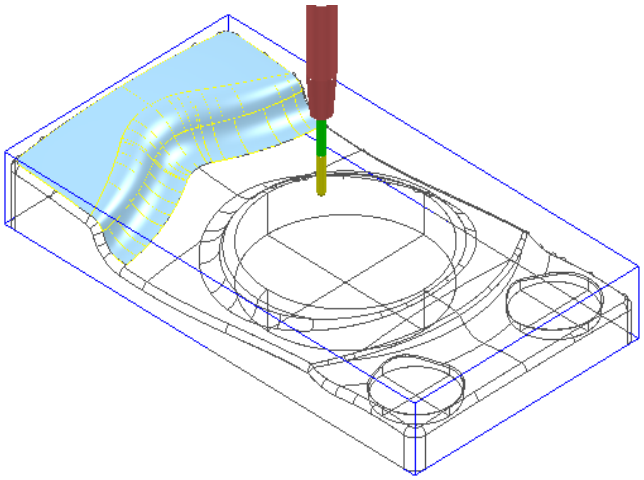
A **Pattern** can be used as a curve to be offset across the area of a **3D Offset Finishing** strategy.

1. Select **File** > **Delete All** and **Tools** > **Reset Forms**
2. **Import** the model:

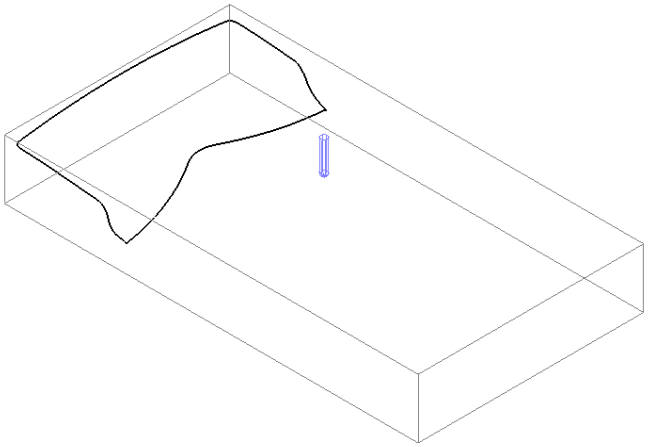
   **.....\PowerMill_Data\Models\speaker.dgk**
3. **Save Project As**:

   **.....\COURSEWORK\PowerMill_Projects\SpeakerPattern-EX1**

4. Define a Dia 6 Ball Nose tool with the name **BN6**.
5. Define a **Block** to the default **Min/Max limits**.
6. **Calculate** the **Rapid Move Heights** to the default settings.
7. Select an **ISO3** view.
8. Select the 3 shaded **Surfaces** as shown in the following diagram.



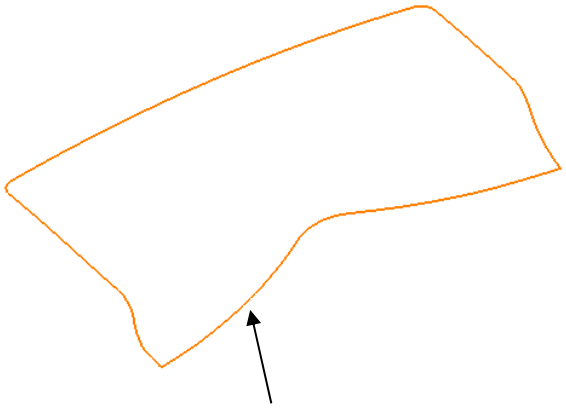
9. Create a ***Selected Surface***, **Boundary** with ***allow boundaries to be private*** un-ticked, and with a **Tolerance** value of **0.02**.




Copyright © Autodesk

10 From the **Pattern** toolbar click **Create Pattern** followed by **Insert Boundary into Active Pattern** (Input **Boundary** name into the dialog).

11 Undraw the **Tool**, **Block**, **Model** and **Boundary** to visually isolate the new **Pattern**.



*Only the leading edge of the **Pattern** is required. The remaining parts are to be removed.*

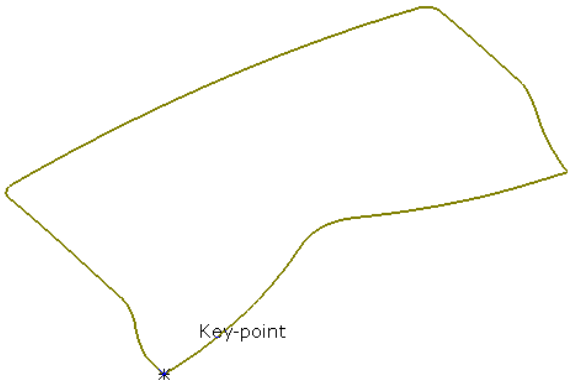
12 Select the **Pattern** segment using the left mouse key.

13 Right mouse click on the **Pattern** and from the local menu, select **Curve Editor** to open the following toolbar:

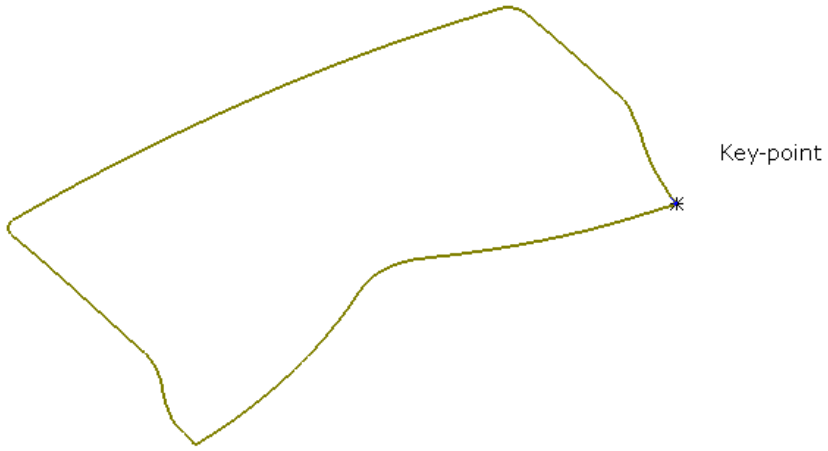
14 Select the option **Cut Item**.

*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.

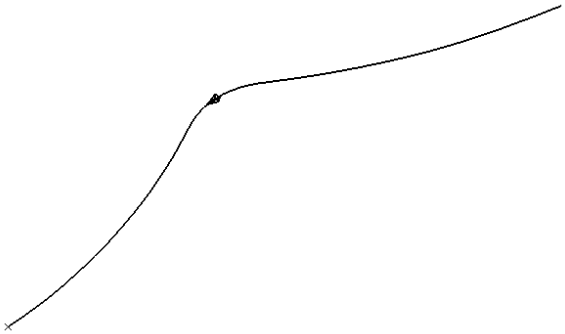


16 Select the leading edge, segment again, locate the **Key Point** and cut exactly at the right hand end of the leading edge.



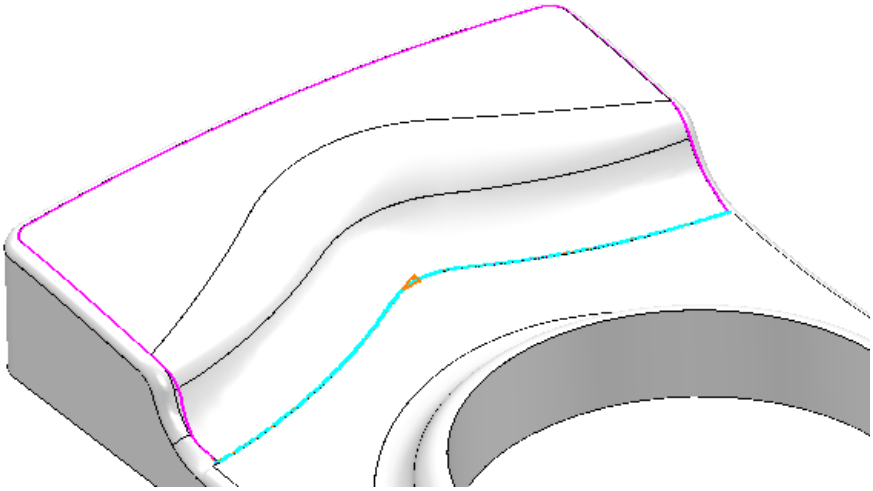

17 Click on the option **Cut Item** again to de-select it.

18 Select the **Pattern** segments away from the leading edge, and **Delete** .



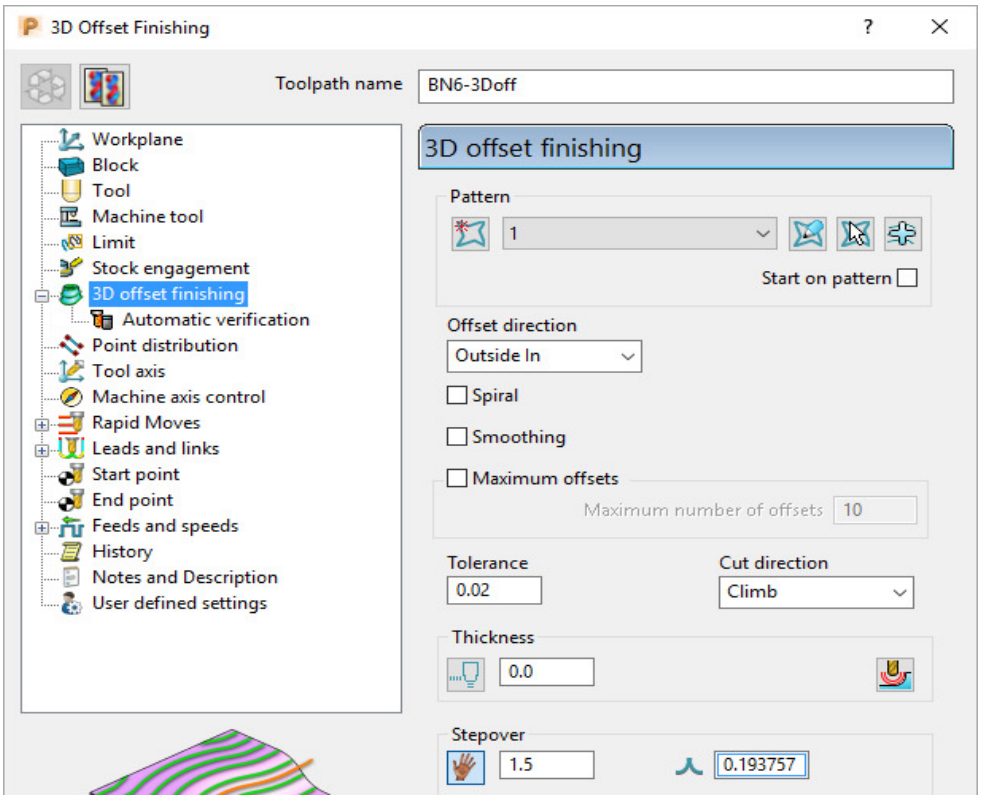
19 Click the green tick to save the above new segment and close the **Curve Editor** toolbar.

20 Re-draw the **Tool**, **Model**, and the **Active Boundary**.

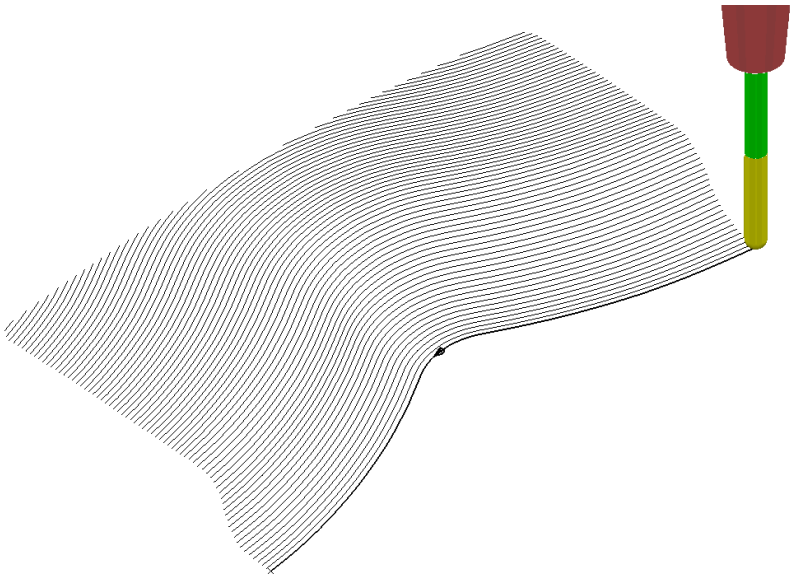



Copyright © Autodesk

21 Select a **3D Offset Finishing** strategy and enter data into the dialog exactly as shown below.

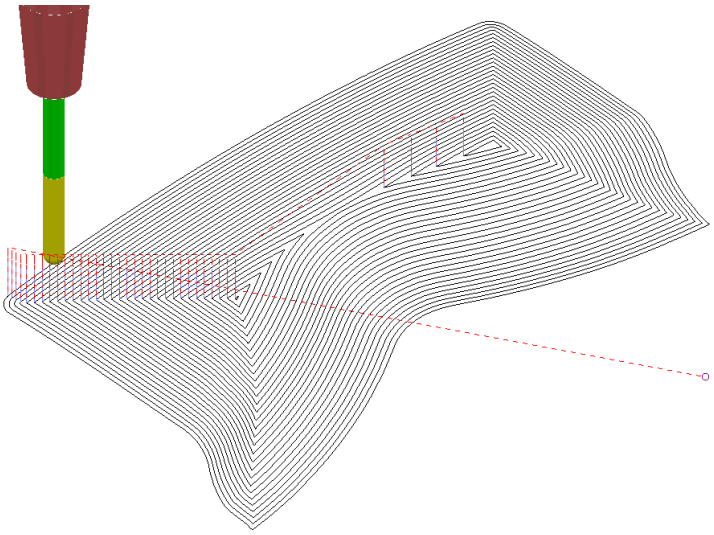



22 Click **Calculate** to create the toolpath and then **Close** the dialog.



**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.

23 **Save** the **Project.**

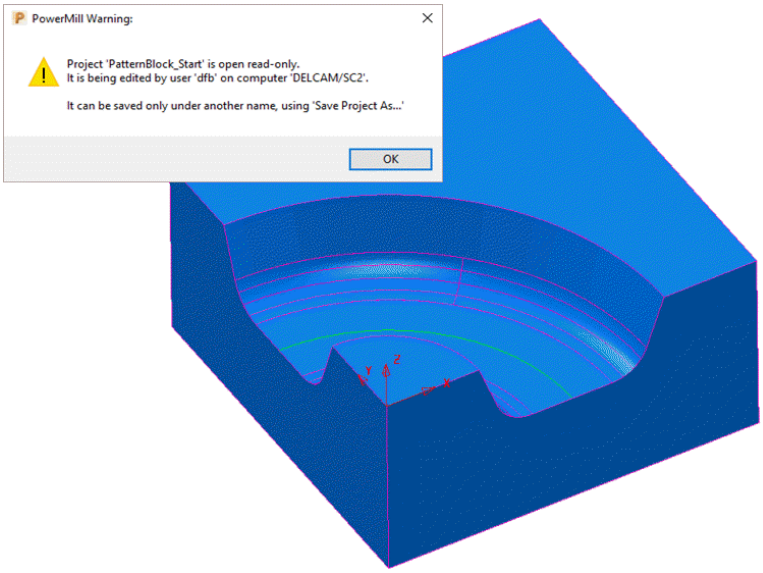
# Exercise - 3D Offset Finishing controlled by Pattern

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.


Copyright © Autodesk

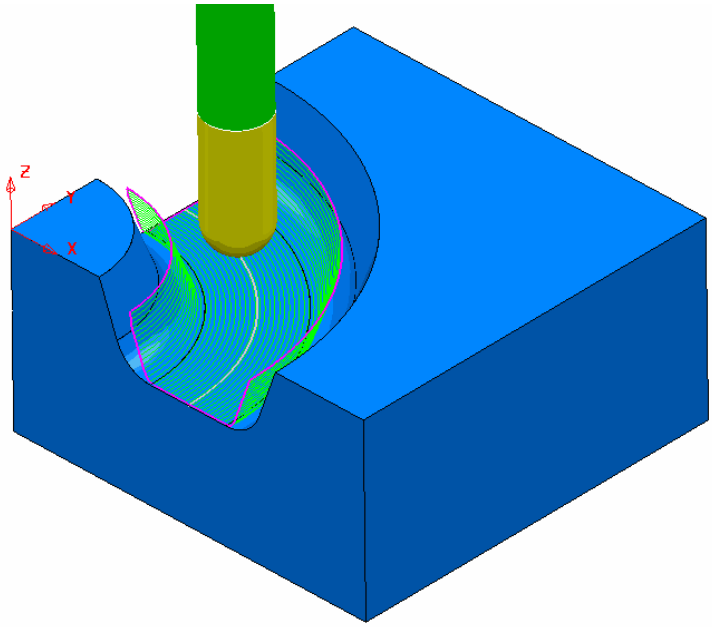
5 **Calculate** the **Rapid Move Heights** to the default settings.

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.

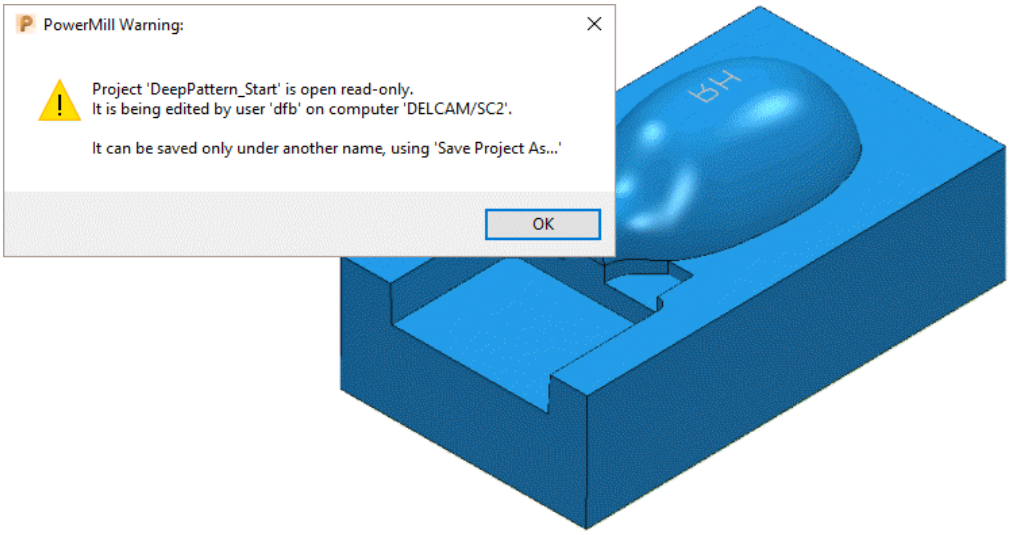
It is, however, possible to produce a **Pattern Finishing** strategy that machines deeper into the model if **Axial Offset** is used in place of a –ve **Thickness** value. This is achieved by first projecting the **Pattern** onto the surface (relative to the tool geometry). **Pattern Finishing** is then 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.

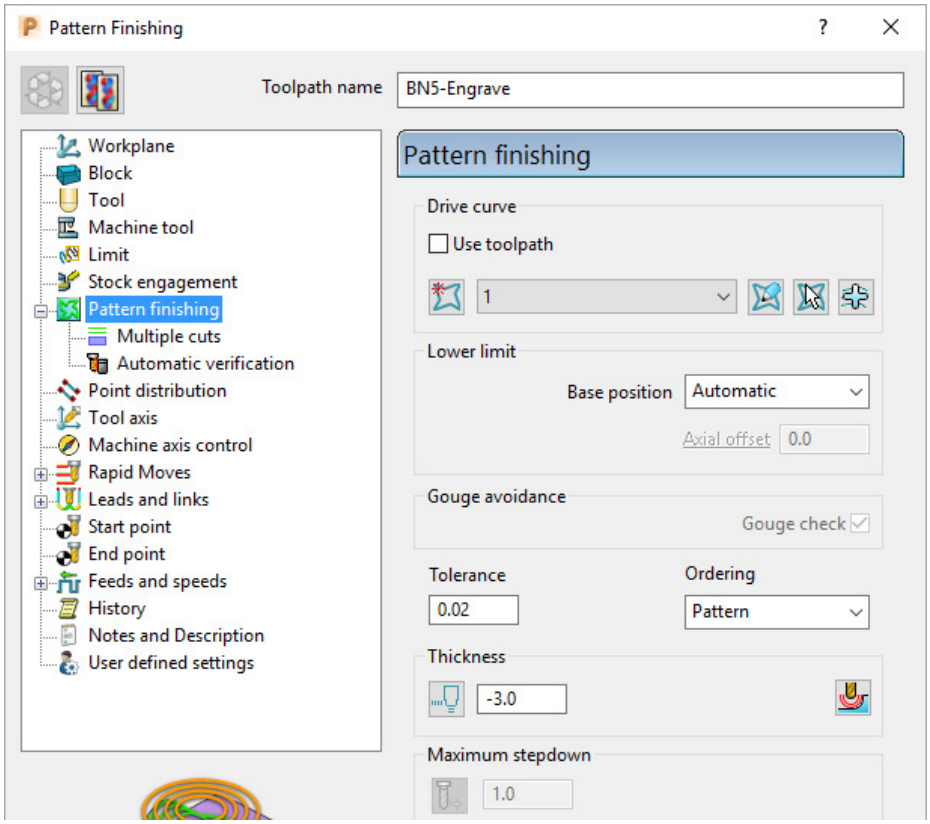
7 In the **PowerMill *explorer***, right-click over the **Pattern 1** and from the local menu select, **Edit** > **Drop** to project it downwards onto the model.

*This projects the **Pattern** down Z, while compensating for the tooling parameters.*

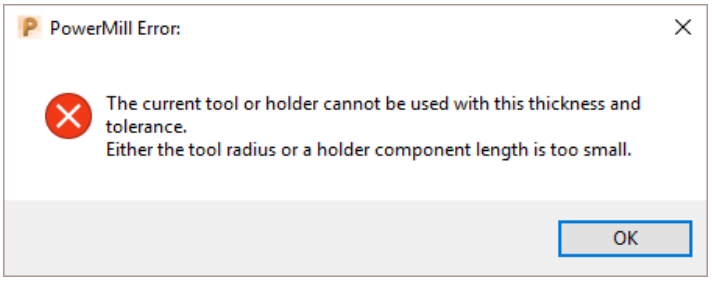

Copyright © Autodesk

8 From the **main** toolbar, select **Toolpath Strategies** and in the dialog select **Finishing** followed by the **Pattern Finishing** option.

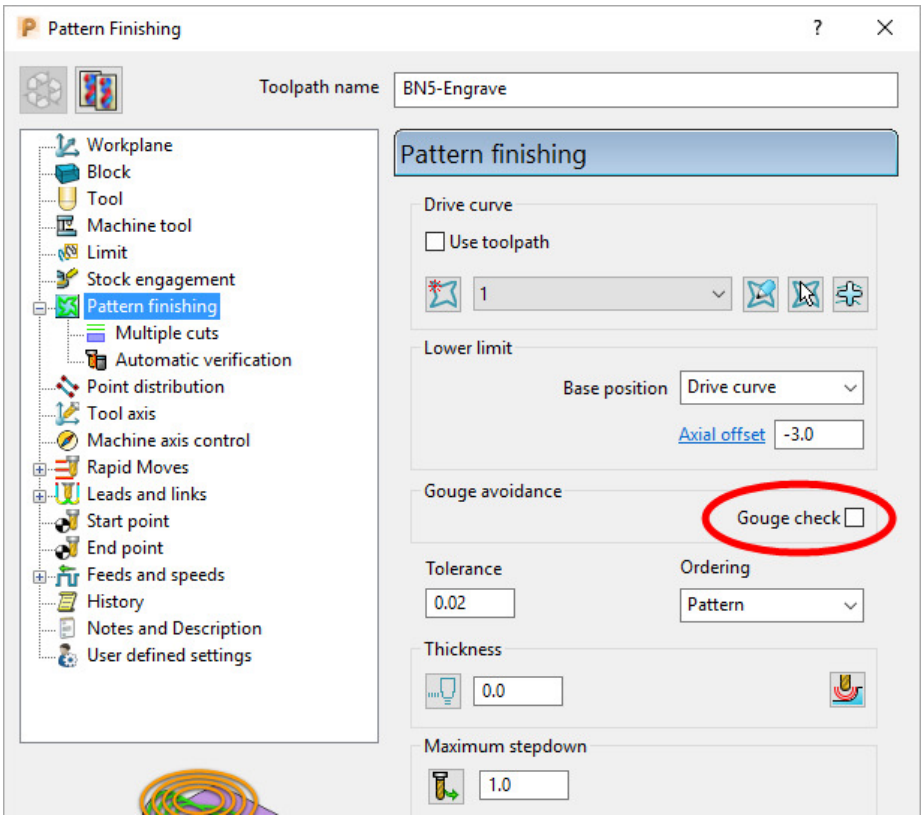
9 Enter the data into the dialog exactly as shown, using ***Base Position*** **Automatic** with ***Thickness*** **-3** and then click **Calculate**.




*The Thickness of -3.0 exceeds the Tip Radius of the Dia 5 Ball Nosed tool hence instead of a toolpath appearing, the following PowerMill Error form is displayed.*




10 Click **OK** to close the ***PowerMill Error*** message and recycle the **Pattern Finishing** toolpath using the data as shown in the following image.



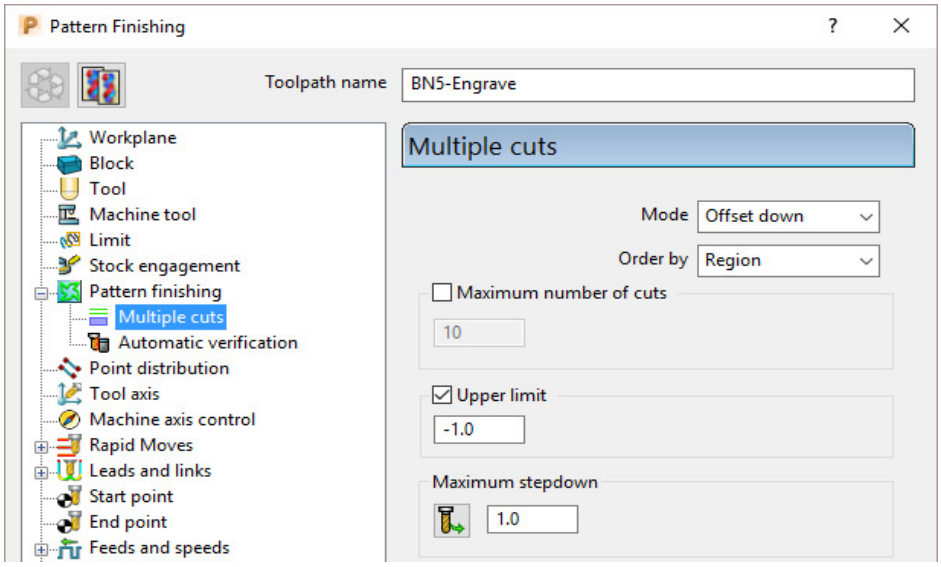

***Base Position*** - **Drive Curve**

***Axial Offset*** **-3.0**

***Gouge Check*** – **un-ticked**

***Thickness*** **0**

11 Select the **Multiple cuts** page and input data exactly as shown below.



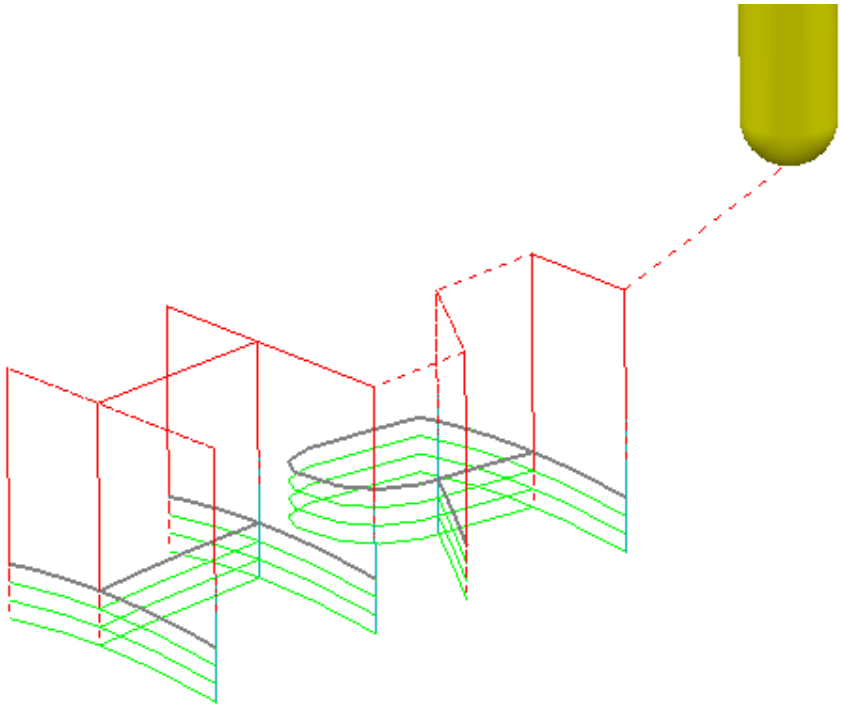

***Mode*** - **Offset down**

***Upper limit*** – **Ticked** with value **-1.0** (First pass at 1mm below the **Pattern**)

***Maximum Stepdown*** **1.0**

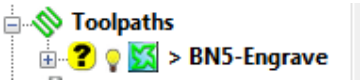

Copyright © Autodesk

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.

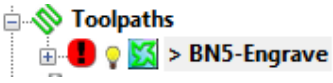
*The Toolpath will initially be identified in the PowerMill explorer with a yellow warning.*




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**.

Copyright © Autodesk

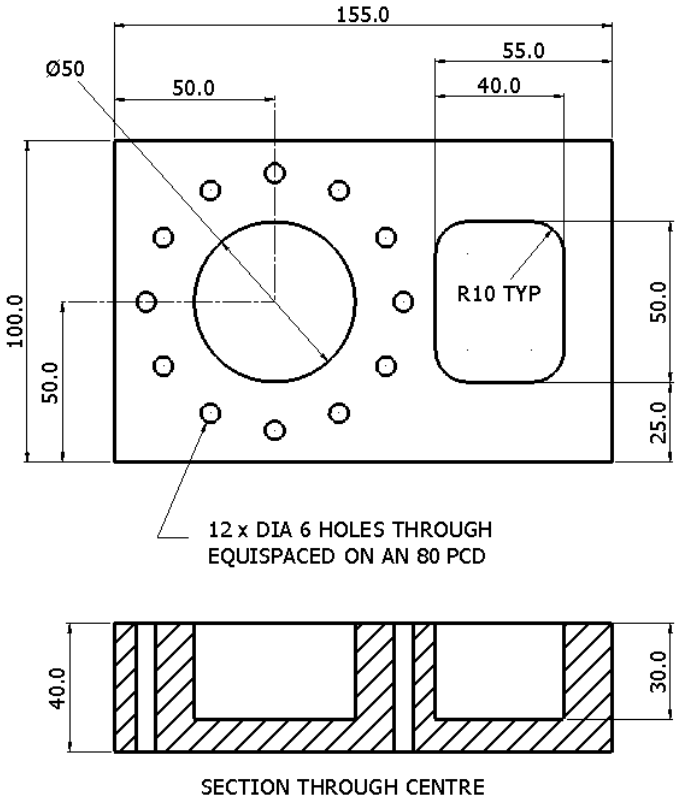
# 11. Curve Editor

## 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.

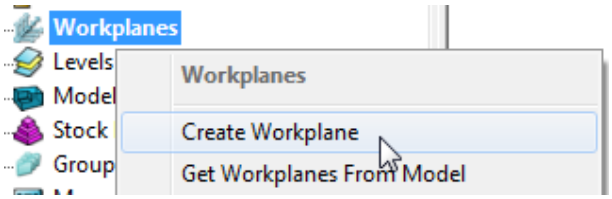
## Input of Drawing detail as a Pattern using the Curve Editor

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.

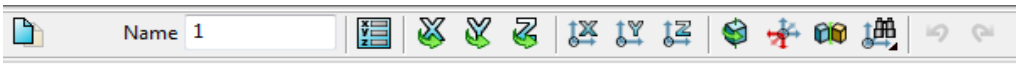


1 Select **File** - **Delete All** and **Tools** - **Reset Forms**.

2 In the **PowerMill *explorer*** select **Workplanes** - **Create Workplane**.




3 The **Workplane** editor toolbar will become active as soon as a new **workplane** is created on the global datum.

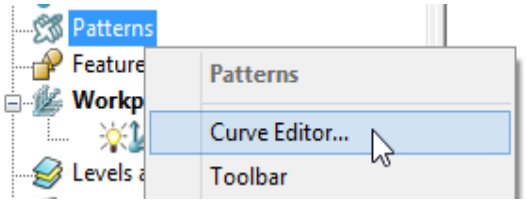



4 Select the ***green*** tick to exit the **Workplane Editor** toolbar.

5 **Activate** the **Workplane**.

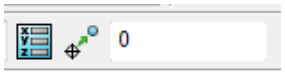
6 Select a **view down Z**.

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.

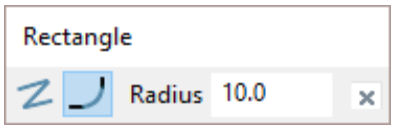



This will fix the corner of the **Rectangle of Lines** to the active XYZ datum.

10 Input **155 100** in the **Command Input** box and press the return key to accept the X and Y dimensions of the rectangle.

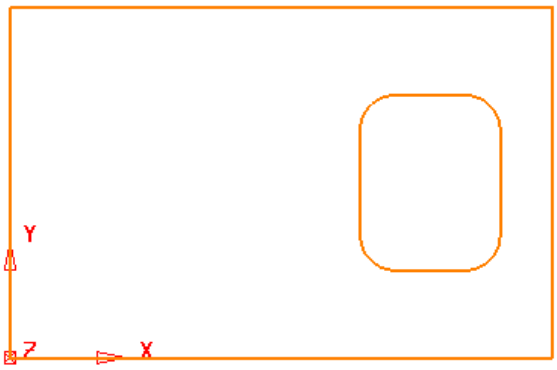
11 Select the corner **Radius** option and input **10**.


Copyright © Autodesk

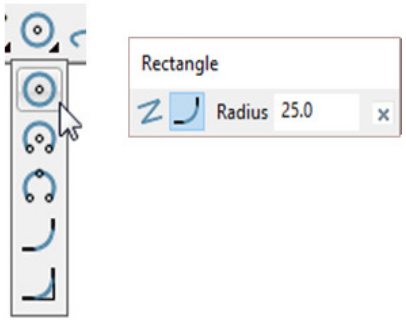



12 Create a new **Rectangle of lines** using **100 25** to define the start corner.

13 In the data input box, switch from **Absolute** to **Relative** input and enter **40 50** for the X and Y dimensions.

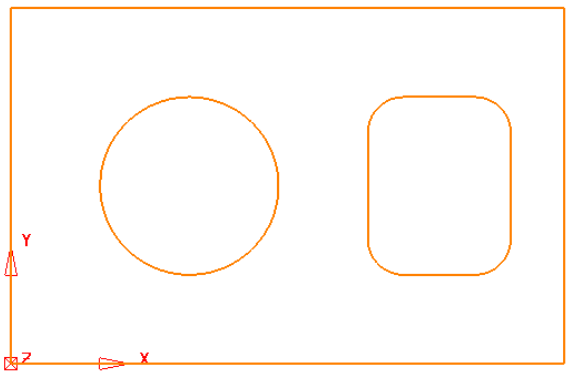



14 From the **Create a circle** options select the **full Circle** option.




15 Input a **Radius** value of **25**.

16 Input **50 50** in the **Command Input** box followed by the return key.

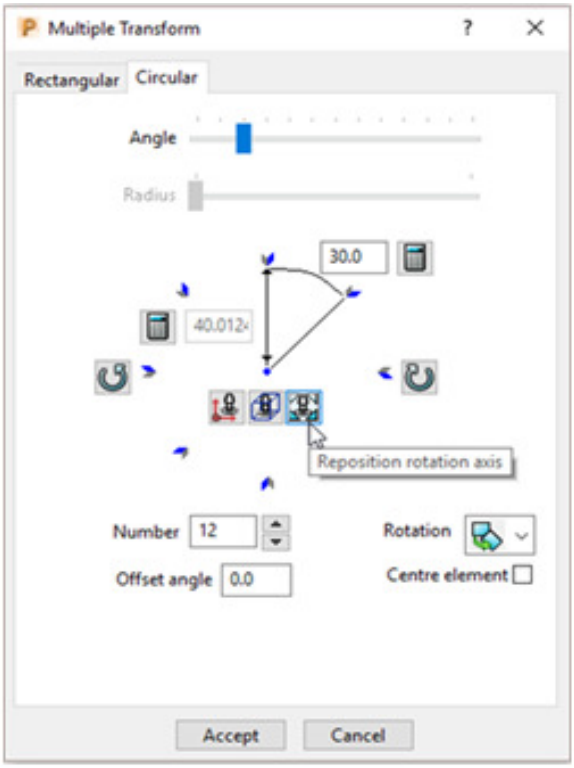


17 Create another **Circle** of radius **3** centred at **50 90**.

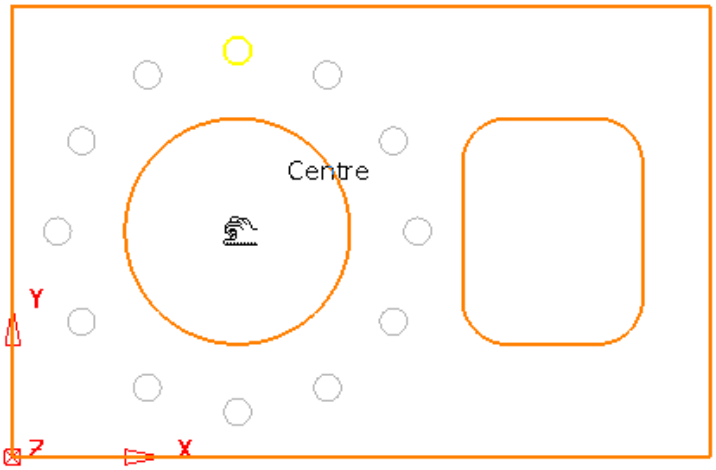
18 Click on to close the **Circle creation** options.

19 Select the Dia 6 **Circle**.

20 With the new Dia 6 **Circle** selected, open the **Transforms** options and select **Create pattern of objects** .

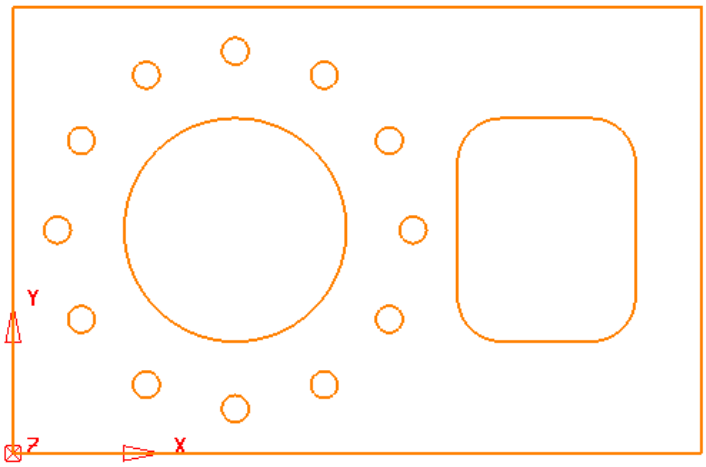



21 On the dialog, select the **Circular** tab and enter **Angle 30**, **Number 12** and select **Reposition rotation axis** , and drag it to the centre of the large circle.




22 Select **Accept** on the **Multiple Transform** dialog.


Copyright © Autodesk

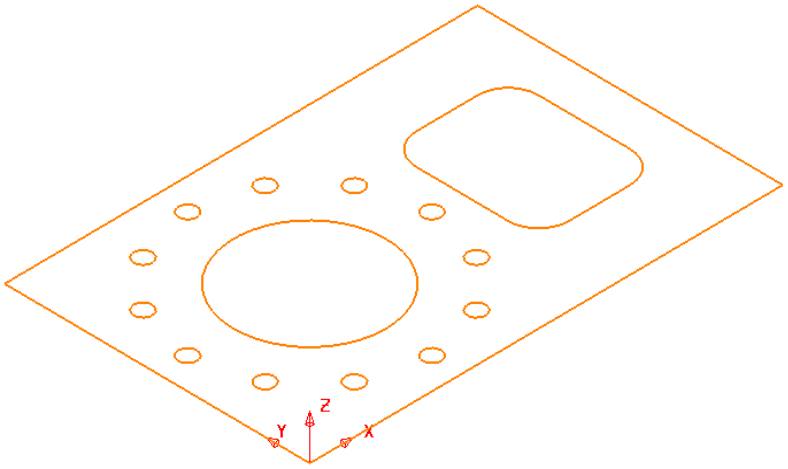



The geometry creation is now complete and is ready to be returned to the **PowerMill Project** as the contents of the **Pattern**.

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**:

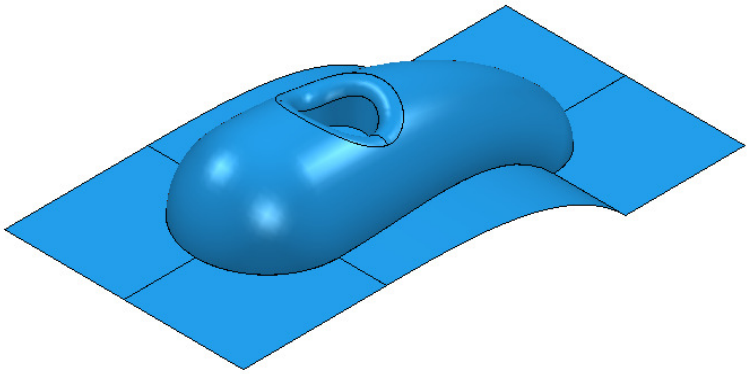
**.....\COURSEWORK\PowerMill-Projects\2DModel**

# Boundary modifications using the Curve Editor

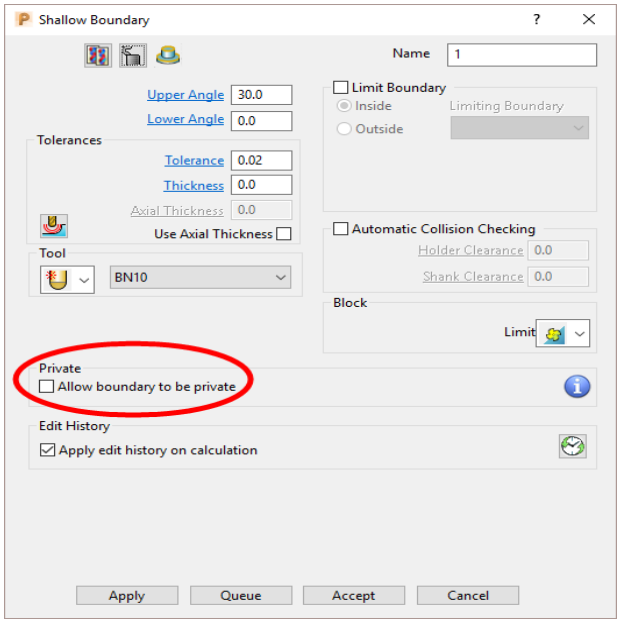
A **Shallow Boundary** will be created in the following example and although correct to the settings applied, it will benefit from further modification to provide improved machining strategies. The modifications aim to make the toolpaths, continuously follow the lower, component contour by bridging the 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**

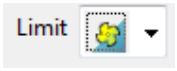


3. **Calculate** a **Block** using ***Defined by*** **- Box** and ***Type*** **Model**.
4. Create a Ball Nose cutter of Diameter 10mm named **BN10**.
5. **Calculate** the **Rapid Move Heights** using the default settings.
6. Use the default **Start and End Point** values.
7. In the **PowerMill *explorer***, right-click **Boundaries** and from the local menu select **Create Boundary > Shallow**




8. Input ***Tolerance*** **0.02**

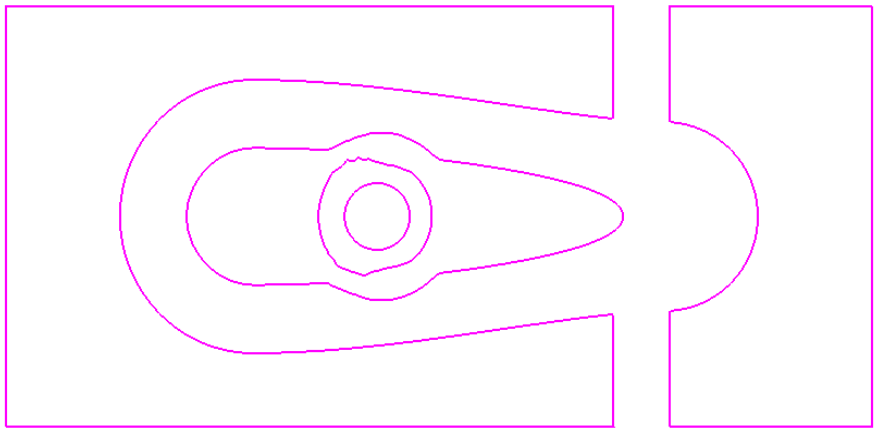

Copyright © Autodesk

9 Set **limit tool centre to block edge**. 


10 Un-tick **Allow boundary to be private**.

11 Select **Apply** followed by **Accept**.

12 Undraw the **model** to view the new **Boundary**.



The **Boundary** will be modified so that the gap is removed between the 2 outer, closed, segments to form separate inner and outer segments. In addition to this, two of the other segments, which contain jagged sections, will be smoothed.

13 Right-click **Boundary 1** in the **PowerMill *explorer*** and from the menu options select **Curve Editor**.

The **Curve Editor** toolbar will appear above the graphics area.

*Some options are currently greyed out, only becoming available when a related entity is selected.*

14 Select a **view down Z** for a top view of the inserted boundary.

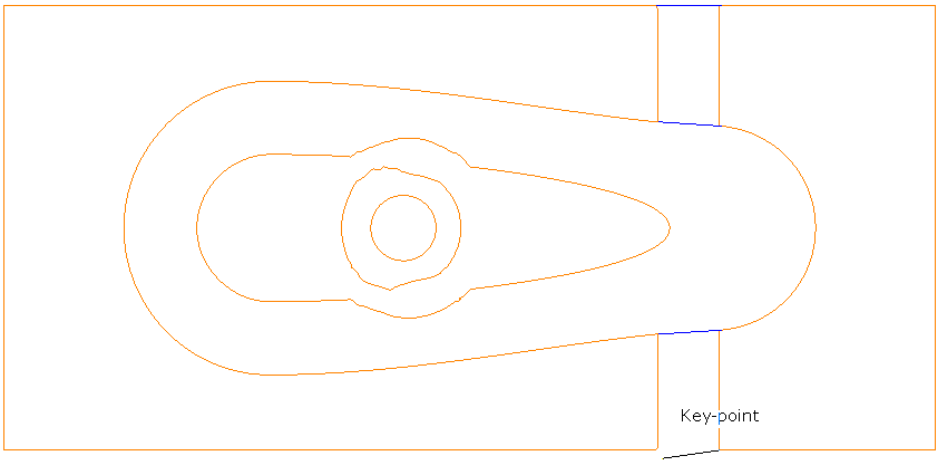
15 Click **Line** from the toolbar then select **Create a single line**.

*Note: It is essential that the **Intelligent Cursor is active** (ticked):-*

If not, this is set in the **Curve Editor** > **Options** menu:-

-- **Intelligent Cursor** -- **Snapping Options** -- (Tick all boxes).

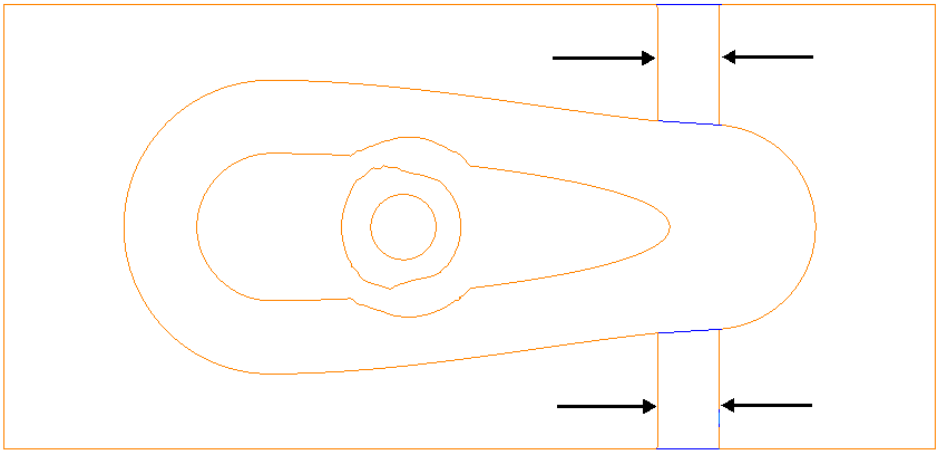
16 Using the left mouse button, snap (**Key points**) to create single lines across the ends of the four gaps, as shown in the image below.




17 Select the **Single Line** option to switch it off.

18 To remove redundant spans on the curves, click the **Limits** option to access **Interactively limit wireframes**.

19 Click **Interactively limit wireframes** and using the left mouse button, select the parts of the curve (arrowed) to be removed.

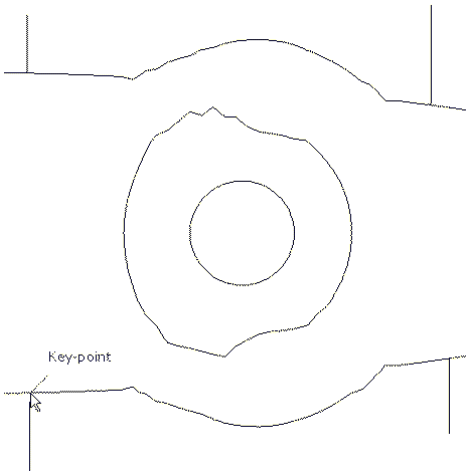


20 Select the **Interactively limit wireframes** option again to switch it off.

21 ***Zoom*** into the middle of the area covered by the 3 inner segments.

*Note the jagged effect along these curves.*


Copyright © Autodesk



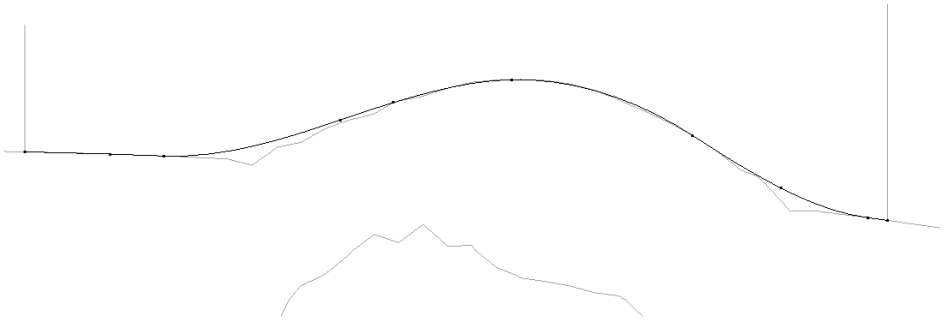

22 Create 4 **single lines**, snapped to key points and pointing outwards from the curve outside the jagged regions (as shown above).

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.

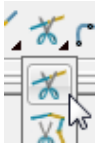
24 For the upper section, generate a new, smoother, **curve** by snapping strategically placed key points within this region, starting and ending by snapping to the ends (**Key Point**) of the marker lines.

*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.

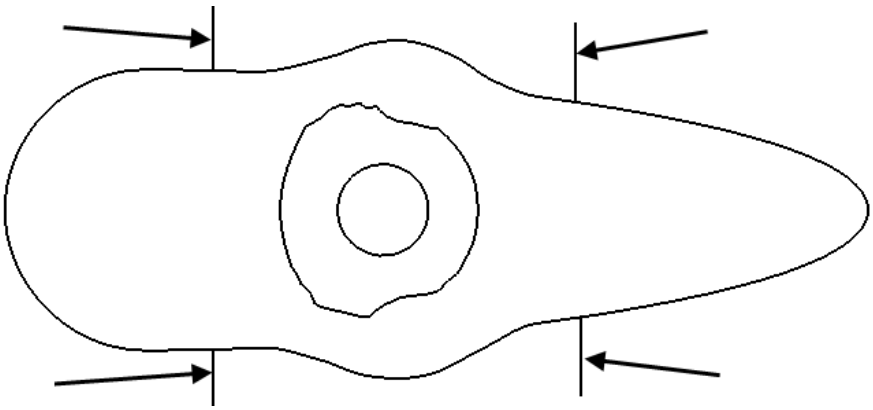
26 Select the upper, original curve and select the **Cut Item** option.



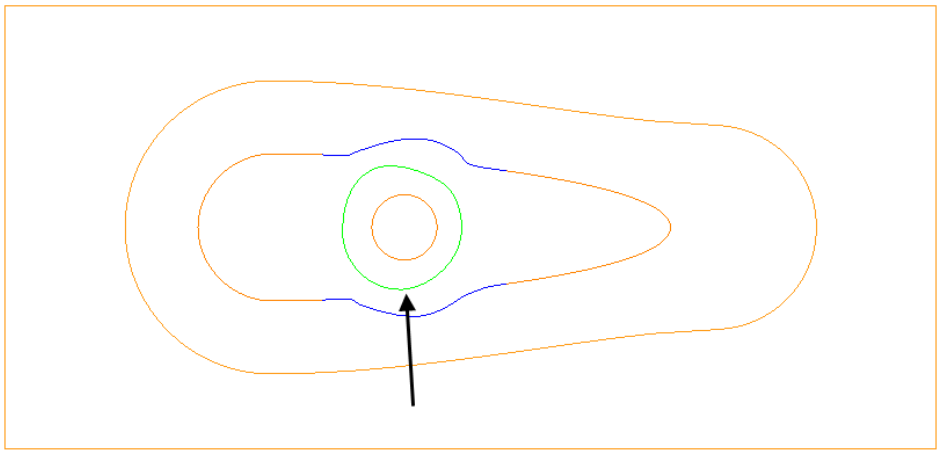
27 Click the **key points** at the lower end of each line to cut the original curve.

28 Repeat the above process on the lower, original curve then select and **Delete** all 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.

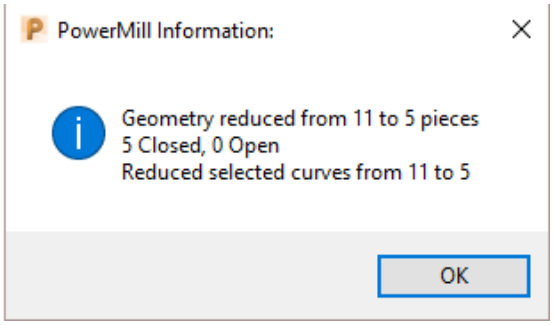


*The **Curves** used in the above wireframes will consist of open or closed segments, but if they are to be used to recreate a **boundary** they must form single entity, closed loops. Only closed forms will appear as **boundary** segments when returned to **PowerMill**.*


Copyright © Autodesk

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.

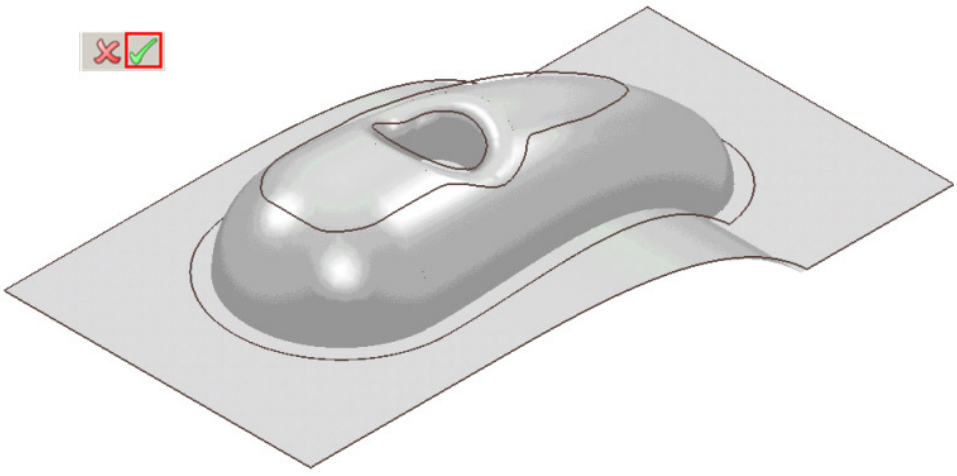


*The wireframe will now consist of five single segments as reported in the **PowerMill Information box** (above right).*

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 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.

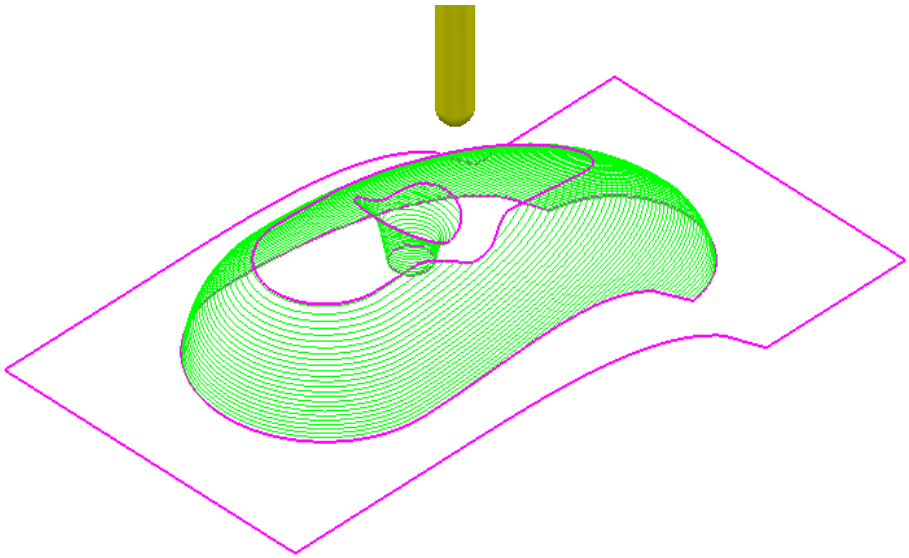
35 Generate a **Constant Z Finishing** toolpath with **Stepdown** 1 and in the **Limit** page set the **Boundary** - **Trimming** option to **Keep outside** (Steep areas).

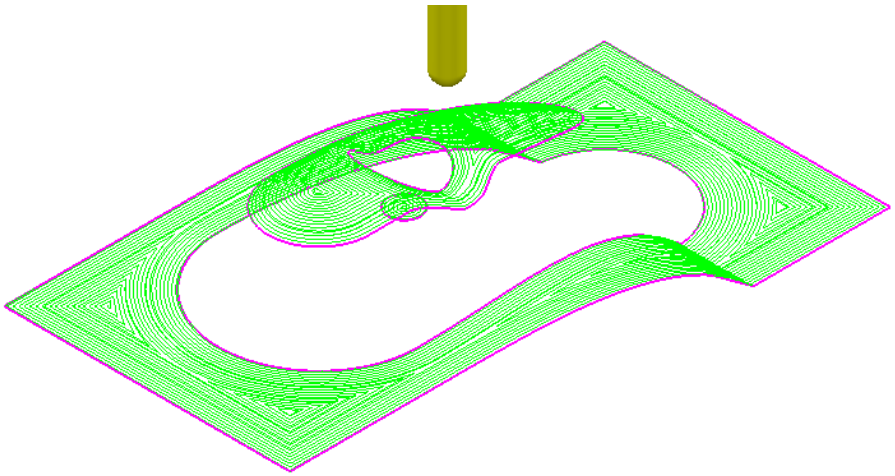
36 Generate a **3D Offset Finishing** toolpath with **Stepover** 1 and in the **Limit** page set the **Boundary** - **Trimming** option to **Keep inside** (Shallow areas).

Copyright © Autodesk

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.*


Copyright © Autodesk

# 12. 2D Machining

## Introduction

**PowerMill** offers a comprehensive range of 2.5D machining strategies created by either **Feature Machining** (***Drilling***, ***Turning,*** and ***Milling***) or direct **Curve Machining** (***Milling***).

### Feature Group Machining

**Features** are wireframe entities that are extruded along Z for ***Milling*** or rotated about the X axis for **Turning**. During creation they are specifically defined in a **Feature Group** that can contain an assortment of different **Feature** types. It is essential that a **Feature Group** includes all **Features** that need to be machined together as a component. ***Hole*** **Features** are contained separately in ***Hole Feature Sets***. It is also possible to extract ***Hole*** **Features** directly from the 3D model. ***Hole Feature Sets*** can also be created during **3D Area Clearance** strategies to enable the pre-drilling of plunge move, positions.

On completion of a **Feature Group**, the range of **Feature Machining** strategies will allow the user to 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.

There are several types of **Feature** that can be included in a **Feature Group** which fall into 2 main groups consisting of 5 for ***Milling*** and 4 for ***Turning***.

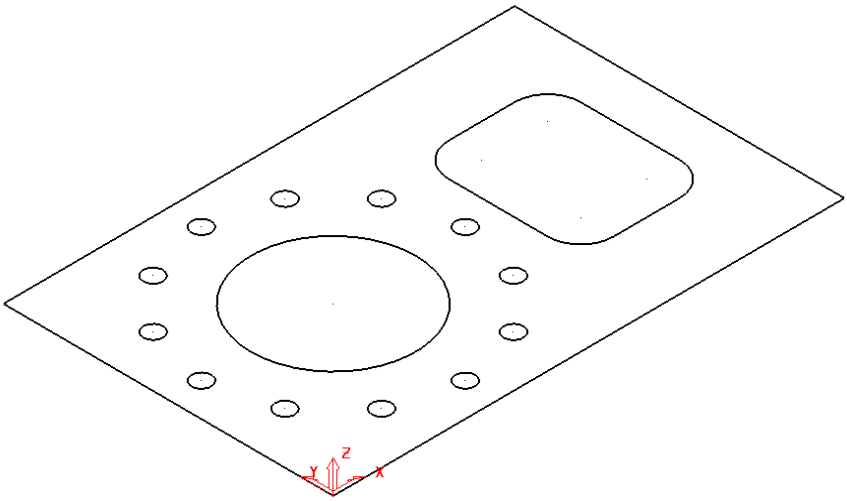
*It is not possible to change an existing a **Feature** into one of a different type.*

# 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**



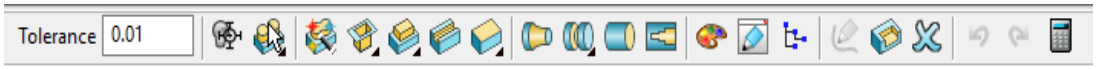

The imported **Project** contains a **Pattern** defining 2D geometry ready to be used to create a **Feature Set**.

3 Select **File** > **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\2DPatternExample**

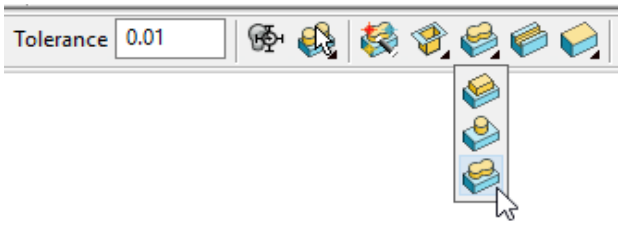
4 In the **PowerMill *explorer***, right-click on **Feature Groups** and from the local menu, select **Feature** editor.

The **Feature** toolbar (shown below) will appear above the graphics area.




5 Back in the graphics area select the outer rectangle on the displayed **Pattern**.

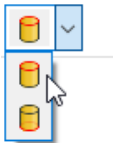
6 From the **Feature** editor toolbar select **Create a freeform boss**.

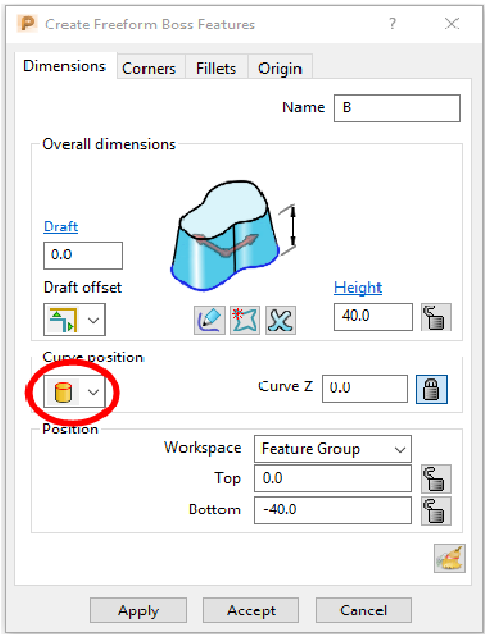


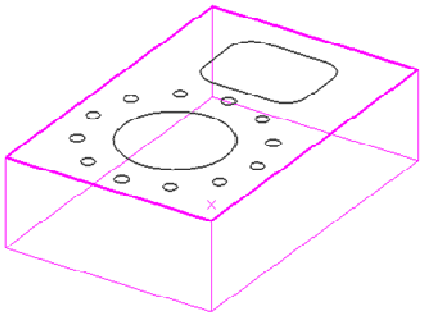

This will open the **Create Freeform Boss Feature** dialogue.


Copyright © Autodesk

7 Fill in the **Create Freeform Boss Feature** dialogue exactly as shown on the following page (Note; use ***Curve position*** as **Top**).

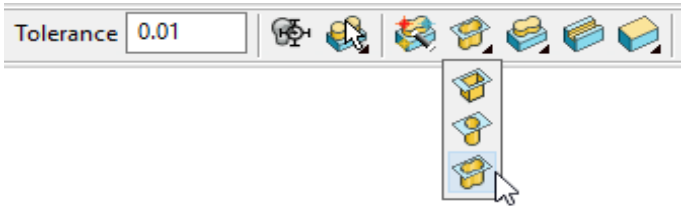




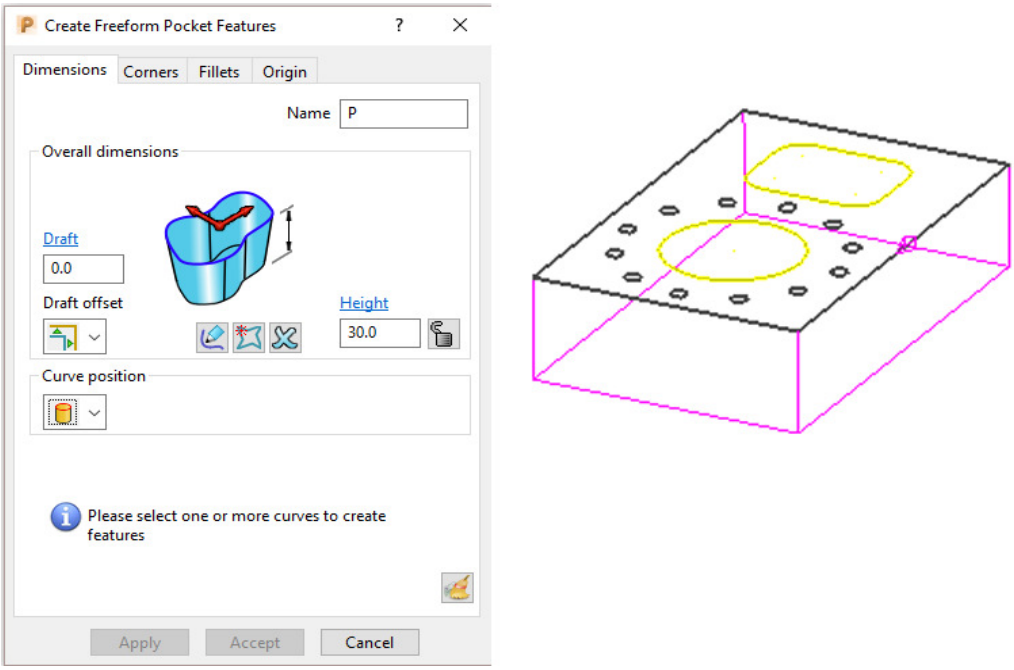



8 Select **Apply** to create the rectangular **Boss Feature** (shown above) and then **Cancel** the dialogue.

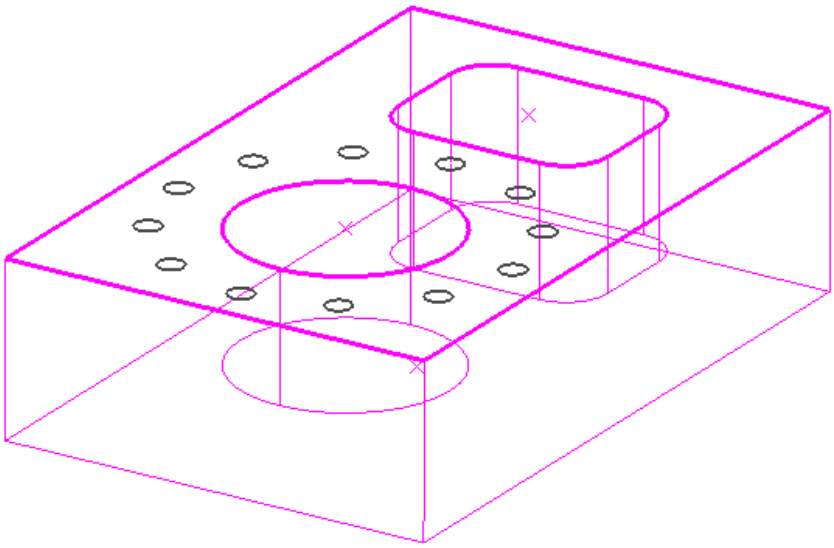
9 From the **Feature Editor** toolbar select the **Create a freeform pocket** option.




10 Select the 2 inner **Pattern** segments (circle and radiused rectangle) and fill in the dialogue exactly as shown below.




11 Select **Apply** to create the 2 **Pocket Features** (shown on the following page).

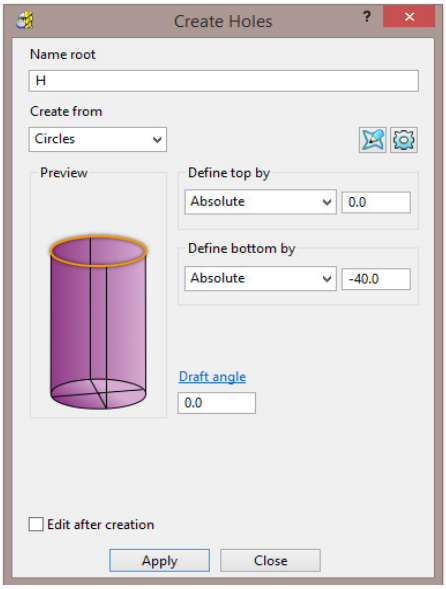


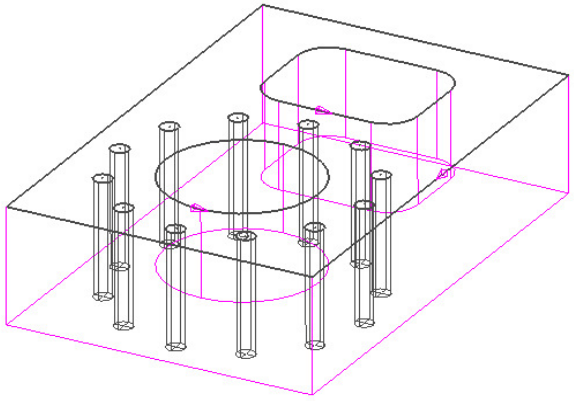
12 Cancel the **Feature** creation dialog and then select the **Green** tick on the **Feature Editor** toolbar to accept all of the created **Features**.

13 In the **PowerMill *explorer*** right click on **Hole Feature Sets** and from the local menu select **Create Holes**.

14 Select all of the Dia 6 circular **Pattern** segments and then enter values exactly as shown in the following **Create Holes** dialog (Note use ***Create from*** - **Circles**).

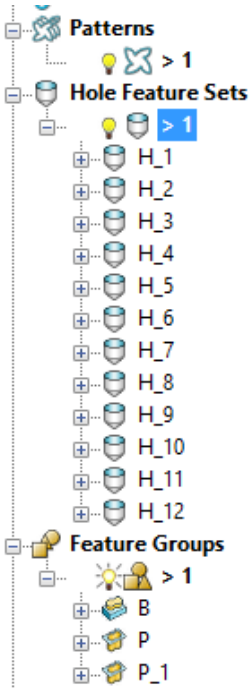

Copyright © Autodesk



15 Select **Apply** to create the **12 Hole Features**.

16 Close the **Create Hole** dialogue.

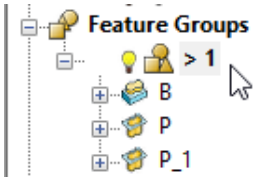



The contents of the **Hole Feature Set** and the **Feature Group** should be similar to the above illustration (Note: A naming system has been chosen where a **Boss** is prefixed with a **B**, a **Pocket** with a **P**, and a **Hole** with an **H**).

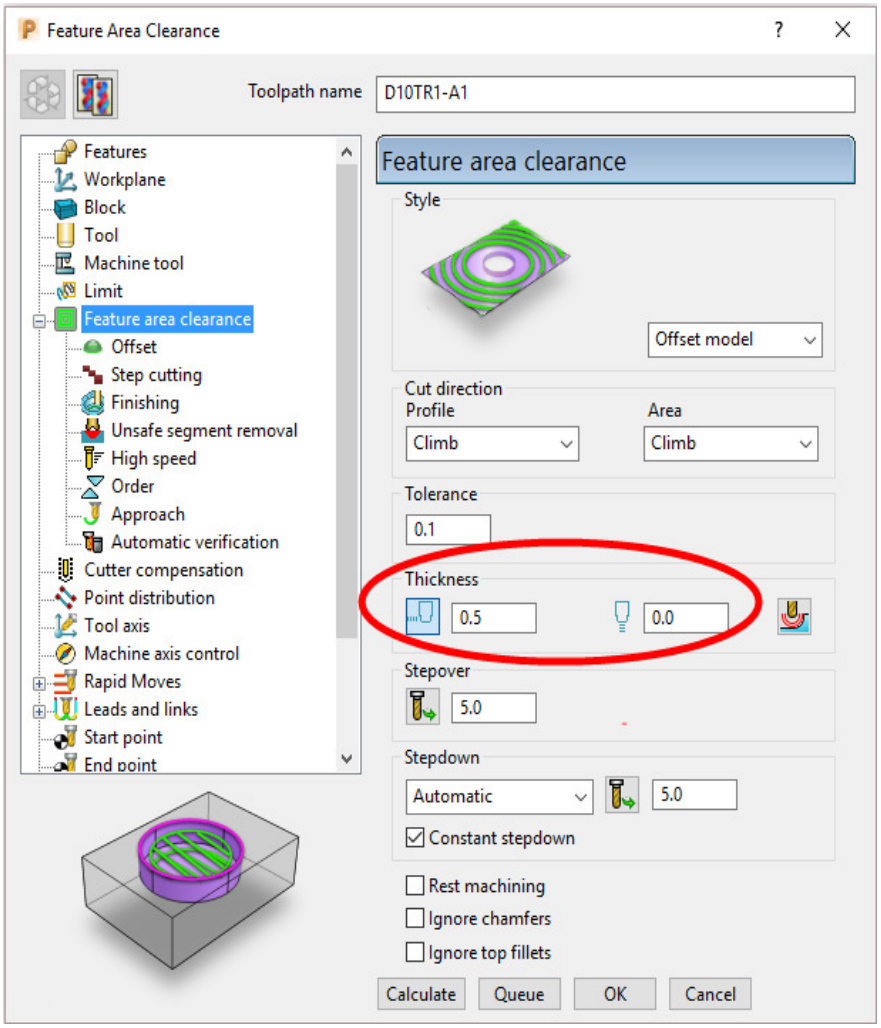
Copyright © Autodesk

# 2D Feature Set Machining

1. Define a Dia 10 - tiprad 1 - Length 50 tool named **D10TR1**.
2. Click to open the **Block** dialog, set the ***Type*** as **Feature** and then select **Calculate**.
3. Click to open the **Rapid Move Heights** dialog and select **Calculate**.
4. Activate the **Feature Group** (**1**).



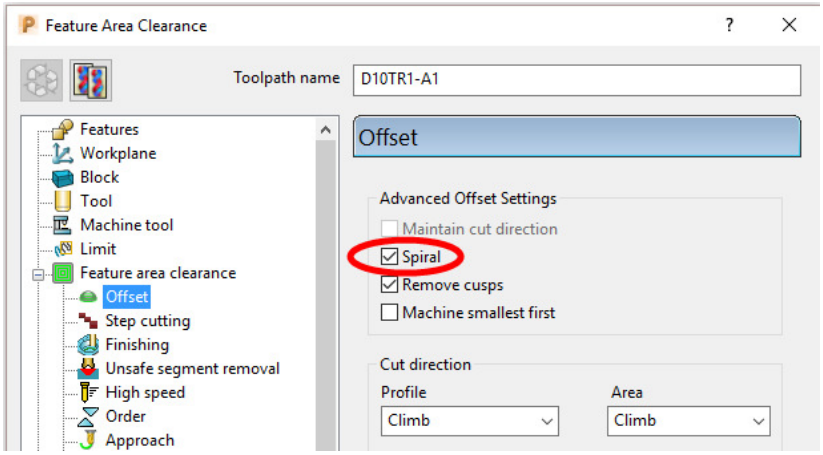

5. Click to open the **Toolpath Strategies** dialog and select **Feature Machining**.
6. Select **Feature Area Clearance** and input the data exactly as shown in the following dialog.





Copyright © Autodesk

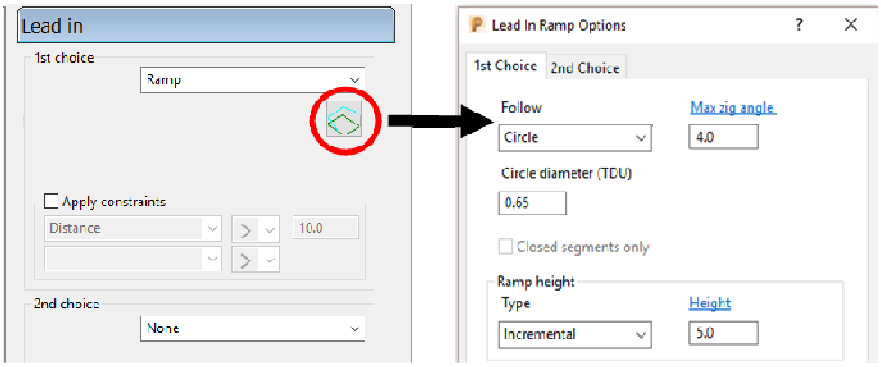
7 In the **Offset** page tick **Spiral**.



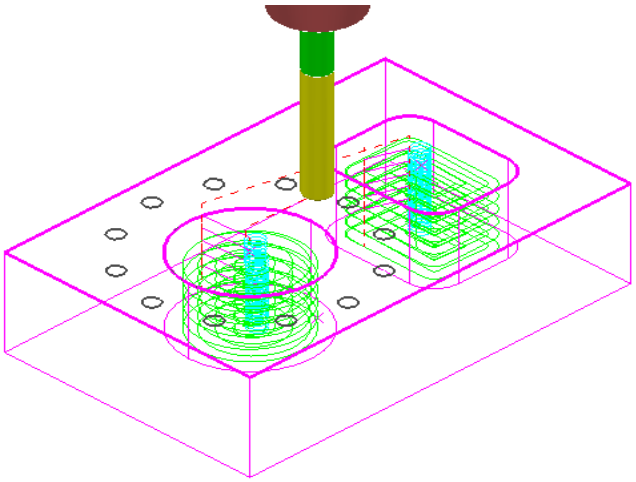

8 In the **Leads and Links** – **Lead in** page select:-

***1st choice*** > **Ramp**

***Ramp Options*** > ***Follow*** **Circle** – **Max zig angle 4.0** - ***Circle diameter (TDU)*** **0.65**



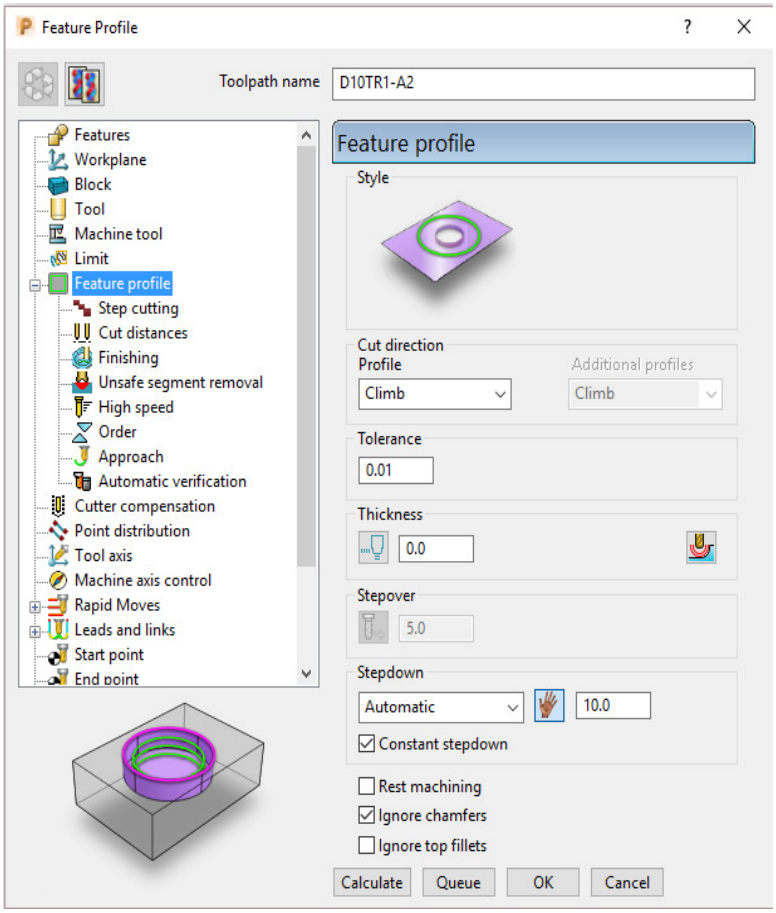

9 Select **Calculate** to process the toolpath and then select **Cancel** to close the dialog.



*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**).*

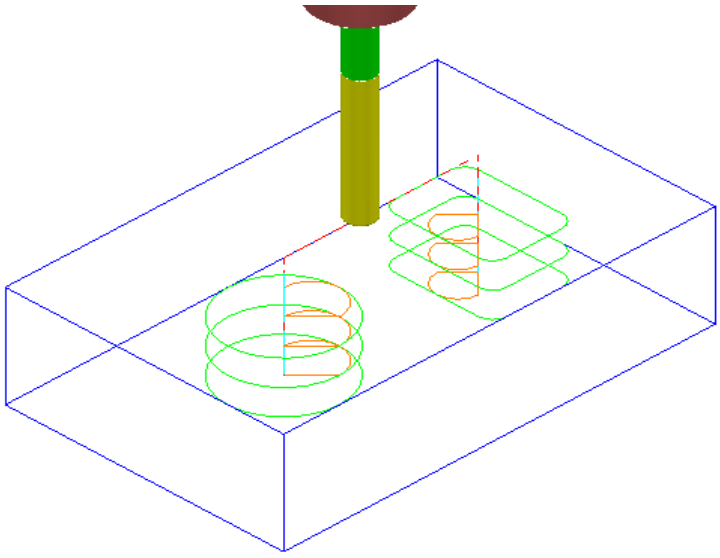
10 Click to open the **Toolpath Strategies** dialog and select **Feature Machining**.

11 Select **Feature Profile** and input data exactly as shown in the following dialog.



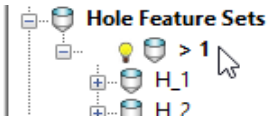

12 Select the **Lead In** page and for **1st Choice** select **Pocket Centre** and then select the **Lead out same as lead in** option .

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




Copyright © Autodesk

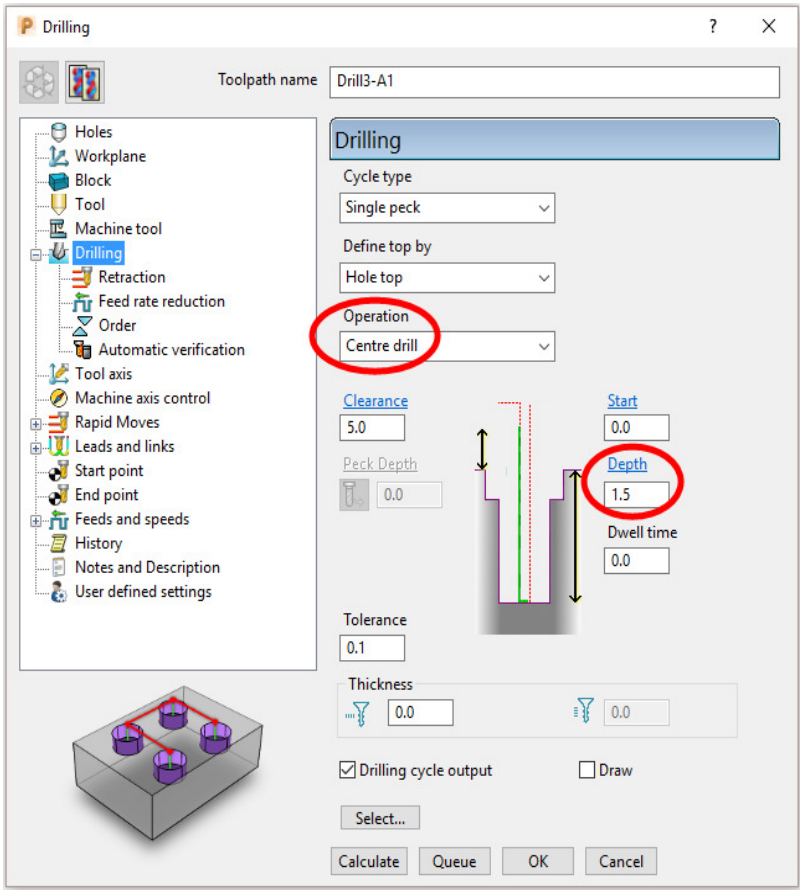
14 Activate the **Hole Feature Set** (**1**).




15 Create a Dia 3 Drill of Length 30, named **Drill3**.

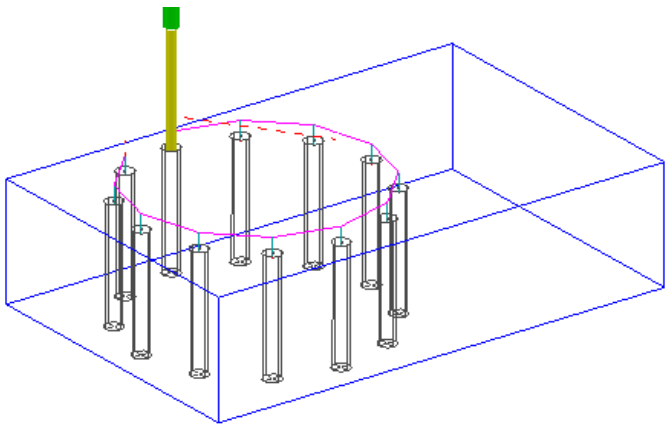
16 Click to open the **Strategy selector** dialog and select the **Drilling** tab.

17 From the list, select **Single Peck** to open the **Drilling** dialog (below).




18 Select **Calculate** and all **Hole Features** will automatically be selected before the **Drilling** toolpath is processed.

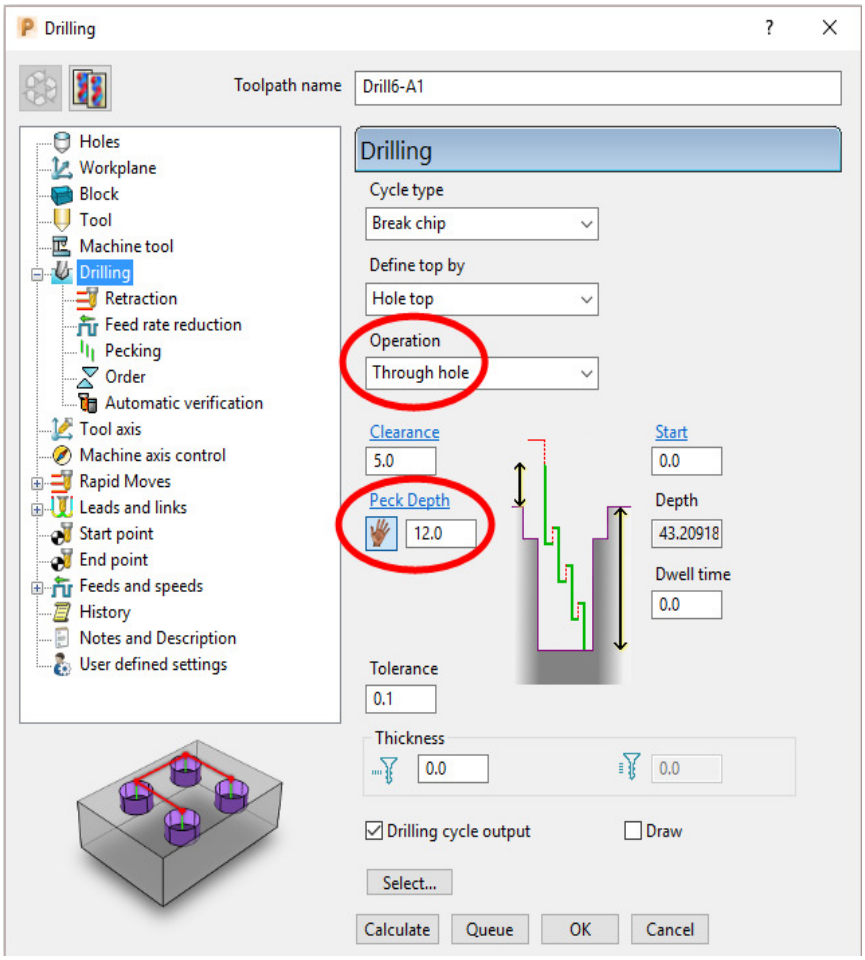
19 Select **Cancel** to close the dialog.



20 Create a Dia 6 Drill of Length 50 and named **Drill6**.

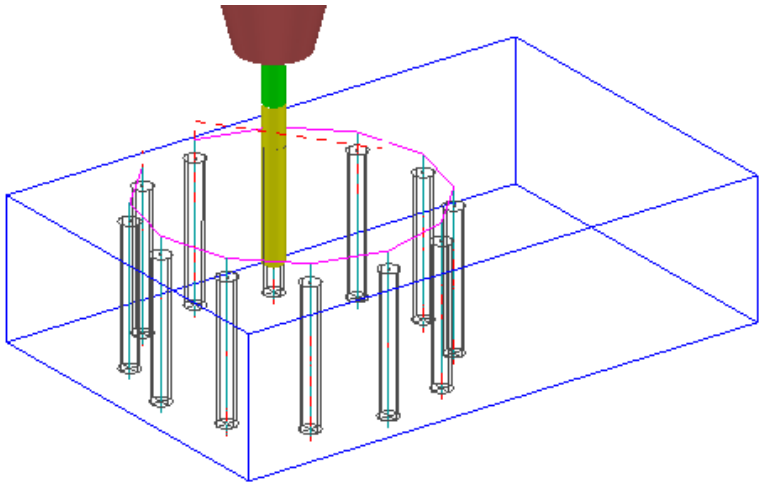
21 Click to open the **Strategy selector** dialog and select the **Drilling** tab.

22 From the list, select **Break Chip** to open the **Drilling** dialog (below).



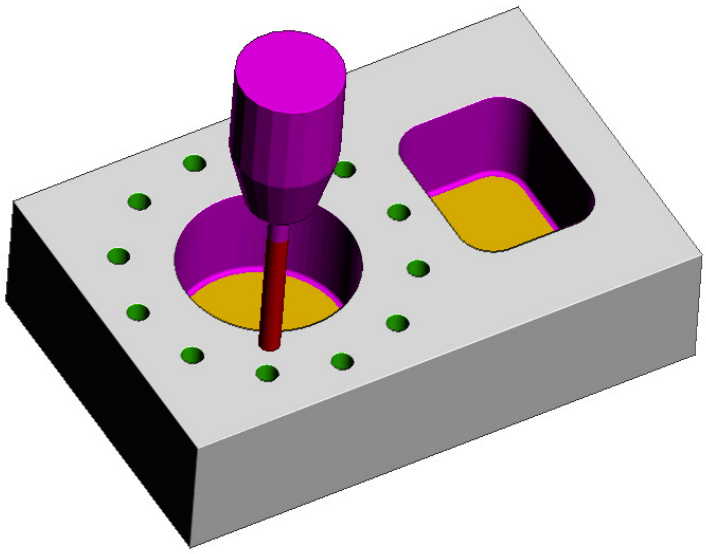

23 Select **Calculate** and all of the contained **Hole Features** will automatically be selected before the **Drilling** toolpath is processed.

24 Select **Cancel** to close the dialog.




Copyright © Autodesk

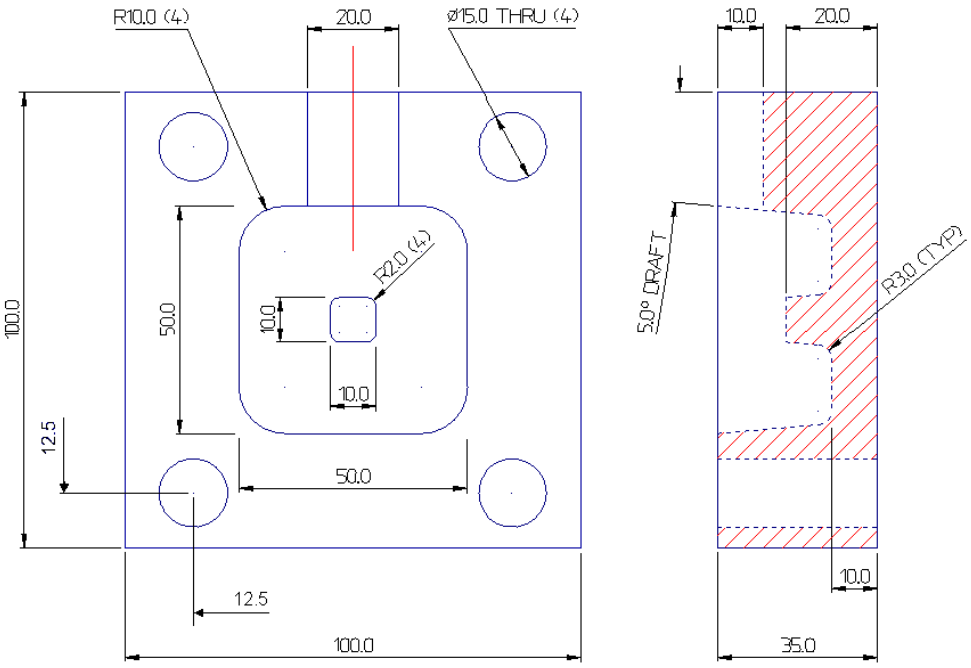
25 Run a full **ViewMILL simulation** on all 4 toolpaths.



26 **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**.

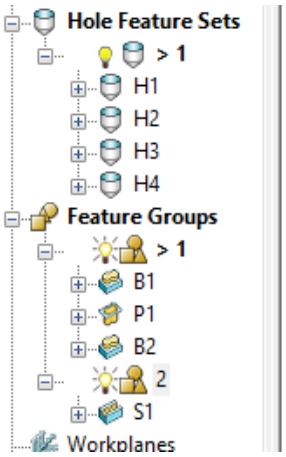
2 Create an empty **Pattern** and **Insert** - **File**:

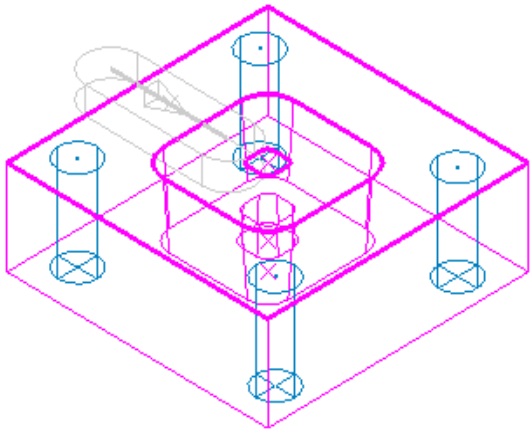
**.....\PowerMill_Data\Models\2d_Wizard_Example.dgk**

3 **Save Project As**:-

**.....\COURSEWORK\PowerMill_Projects\2D_Wizard_EX1**

4 Create two **Feature Groups** (**1** & **2)** using the **Feature Editor** to create the appropriate **Features** using the above drawing dimensions.



a In **Feature Group** (**1**) the outer square and small tapered, central upstand are created as **Freeform Boss Features**.

b The intermediate square is created as a tapered **Freeform Pocket Feature**.

c In **Feature Group** (**2**) the single red line on the drawing is used to define a **Slot Feature** with a slot width of **20**.

d The four Dia **15** circles are created in a separate **Hole Feature Set**.

5 Create a Dia **10** with **3 Tip Radius** tool named **D10TR3** (Roughing, Semi-finishing within the main **Pocket Feature** including the small Boss).

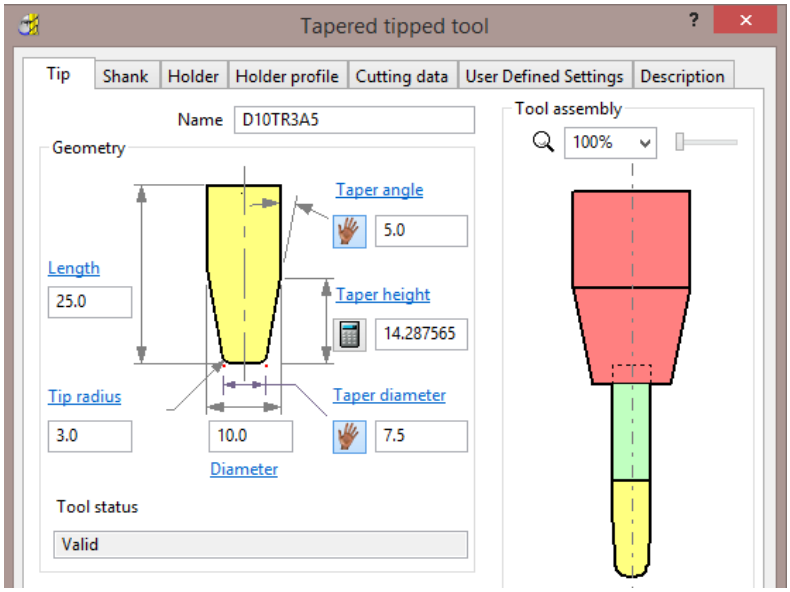
6 Create a Dia **10 Tapered Tipped** tool named **D10TR3A5**, with Tip Radius **3**, Taper Angle **5** Deg, Length **25**, Taper Diameter **7.5** (Final Finishing of tapered walls Main Pocket and small Boss).

7 Create a **Shank** of Dia **10** - length **30**.

8 Create a **Holder** with ***Overhang*** **50** and entities:-

9 Lower Dia **20** - Upper Dia **30** and length **25**.

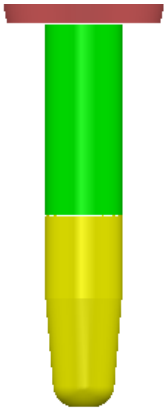
10 Upper/Lower Dia **30** and Length **25**.





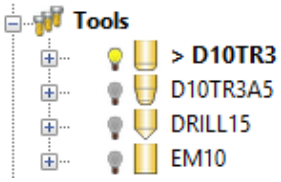
Copyright © Autodesk

*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.*



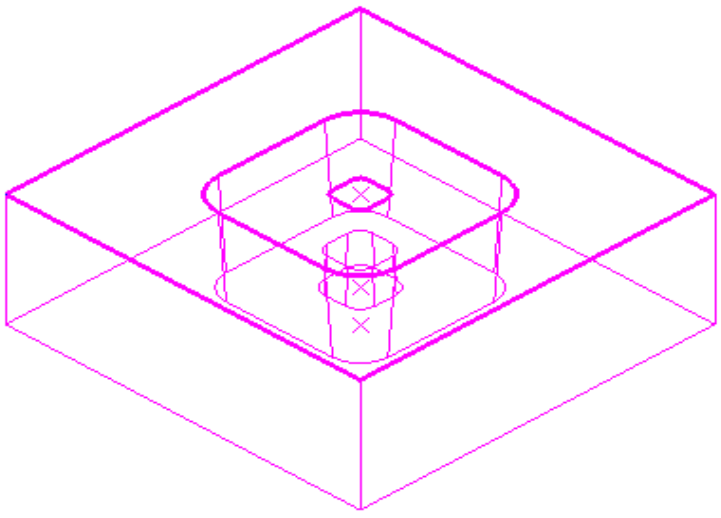
11 Create a Dia 15 **Drill** named **Drill15**.

12 Create an **End Mill** of Dia 10 named **EM10**.




13 Activate the tool **D10TR3**.

14 Activate, **Feature Group 1** and create a **Block** (Type - Feature).



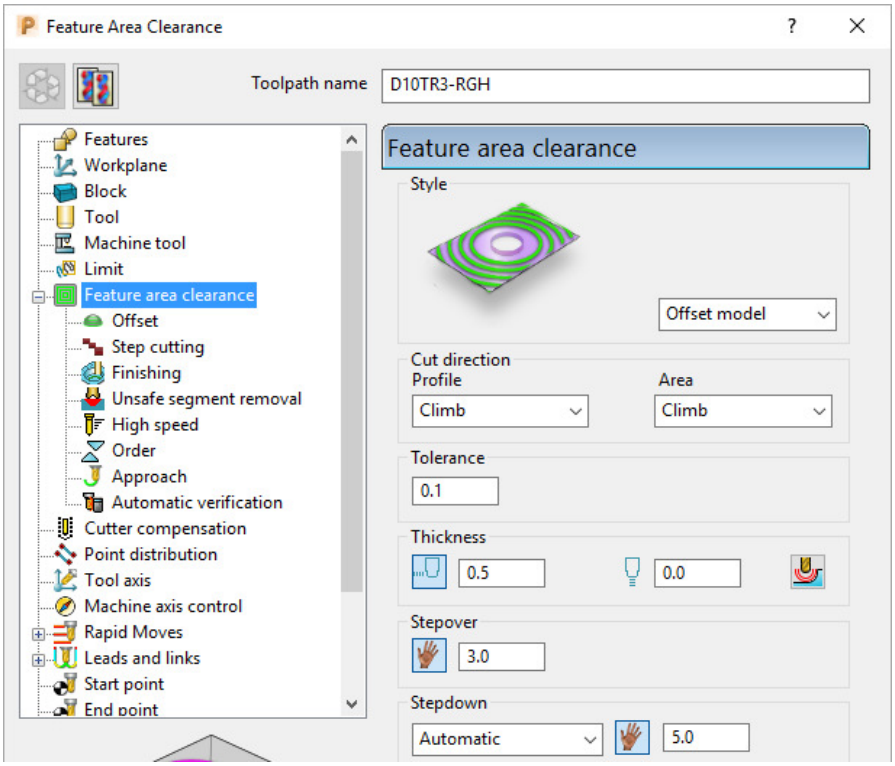
15 **Calculate** default **Rapid Move Heights** ready to create the 2D **Feature** machining strategies.

# Main Block and Central Pocket

**D10TR3** - Feature Area Clearance – Offset Model

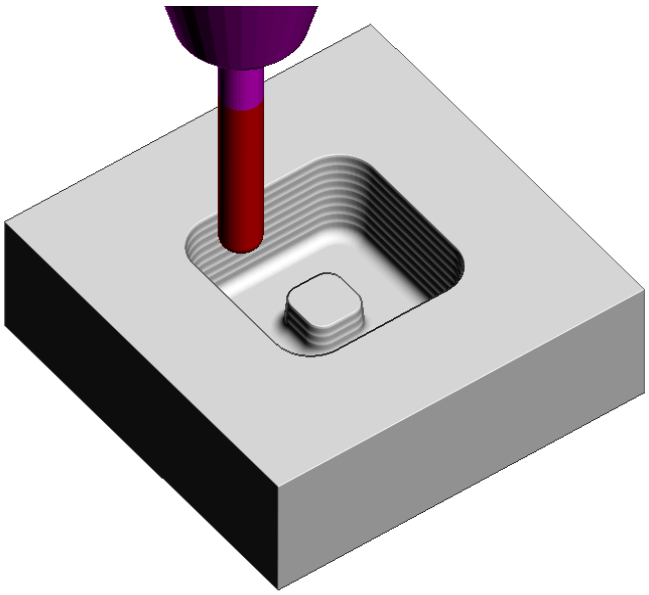
16 Click to open the **Strategy selector** dialog and select **Feature Machining** - **Feature Area Clearance**.

17 In the **Feature Area Clearance** dialog input the data exactly as shown in the following dialog.




18 Open the **Step cutting** page, Tick **Step cutting**, and input a ***Stepup*** of **2.5**.

19 **Calculate** the toolpath and perform a **ViewMILL simulation** as shown below.

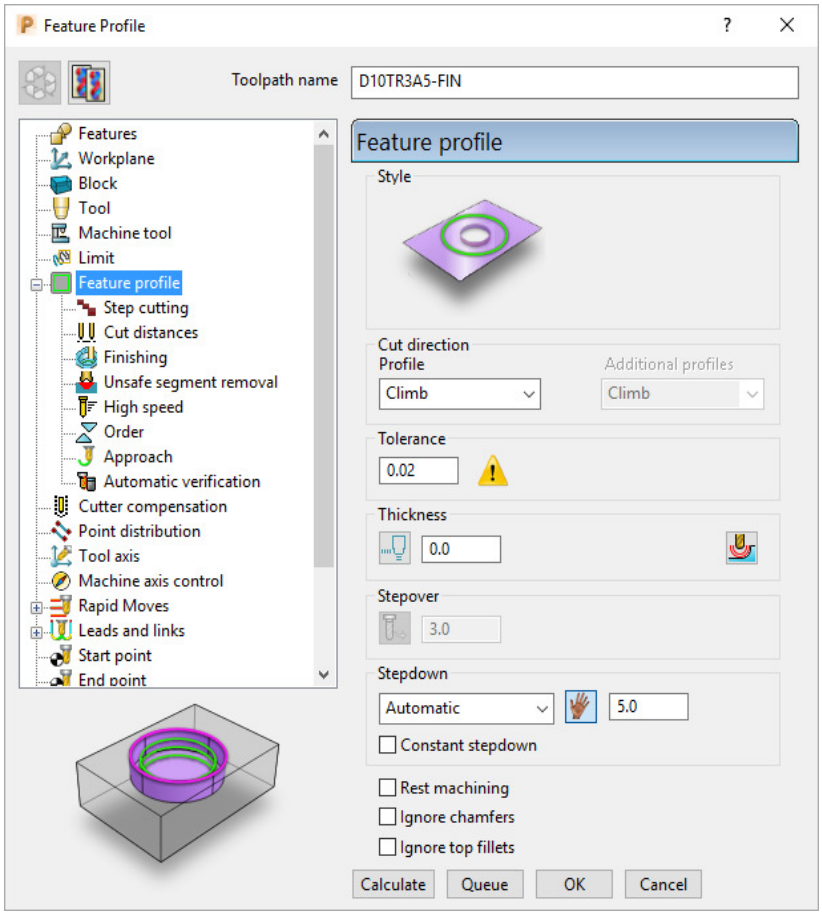



Copyright © Autodesk

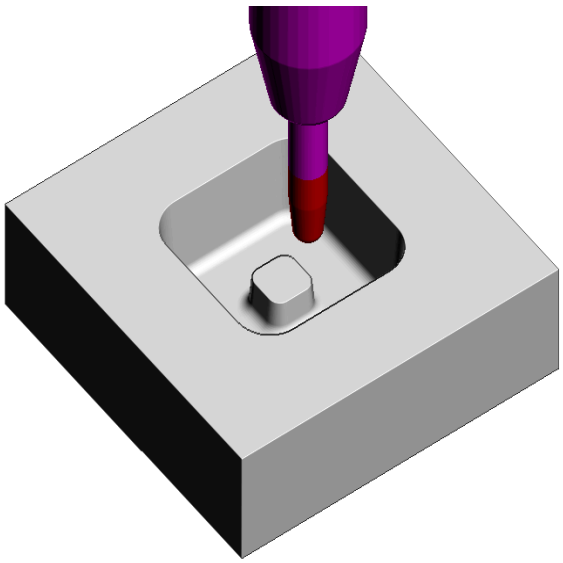
**D10Tr3A5** - Feature set Profile

20 Click to open the **Toolpath Strategies** dialog and select **2.5D Area Clearance**.

Select **Feature Profile** and input the data exactly as shown in the following dialog.



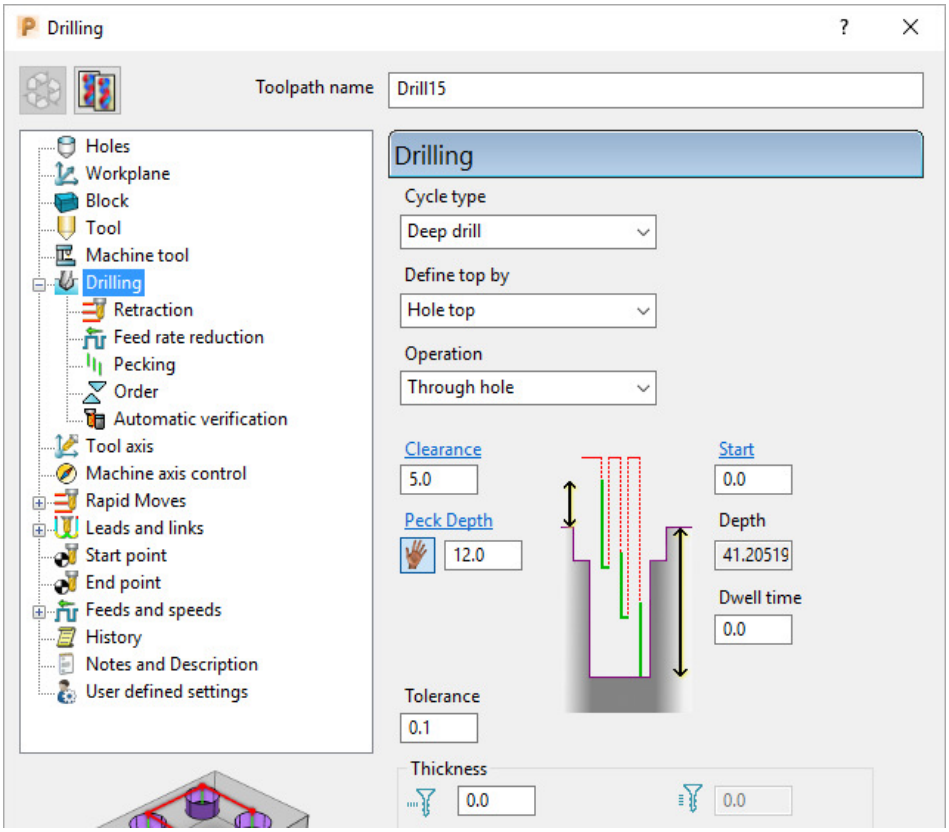

21 **Calculate** the toolpath and continue the **ViewMILL** simulation (as shown below).



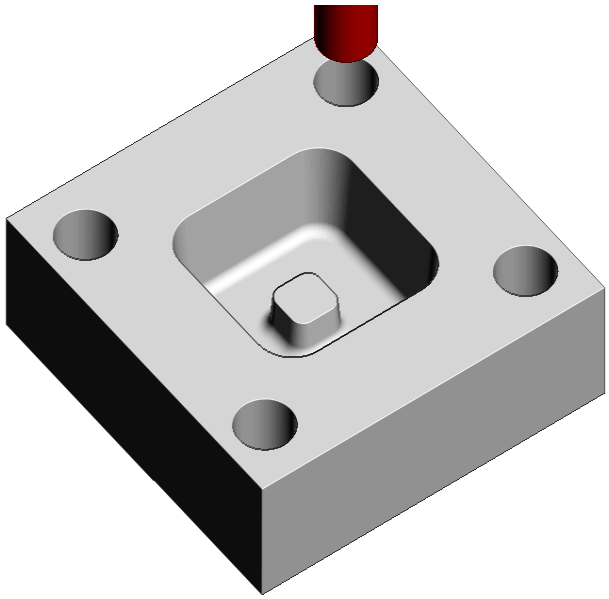
**Drill15** – Drilling

22 Click to open the **Toolpath Strategies** dialog and select **Drilling**.

23 Select **Deep Drill** and input the data exactly as shown in the following dialog.




24 **Calculate** the toolpath and continue the **ViewMILL simulation** (as shown below).




Copyright © Autodesk

**EM10** - Feature Set Slot

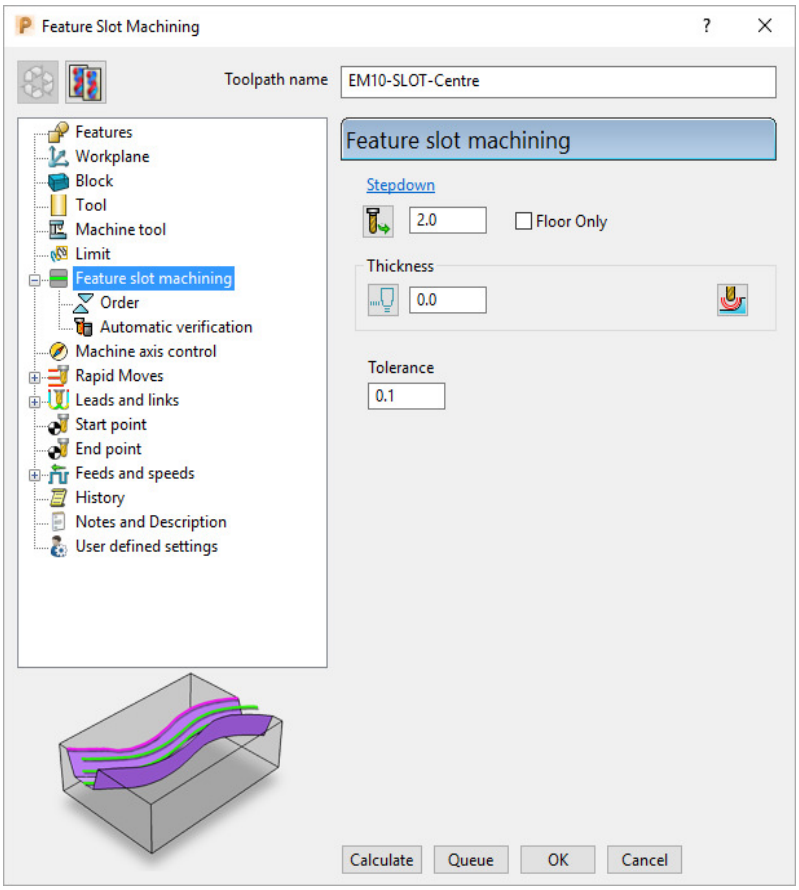
*The Y dimension of the existing Block will need to be adjusted to fully, include the Slot Feature.*

25 Open the **Block** dialog and input **60** for the **Max Y** value before selecting Accept (Do not select Calculate in the Block dialog).

26 Activate **Feature Group 2**.

27 Click to open the **Toolpath Strategies** dialog and select **2.5D Area Clearance**.

28 Select **Feature Set Profile** and input the data exactly as shown in the following dialog.

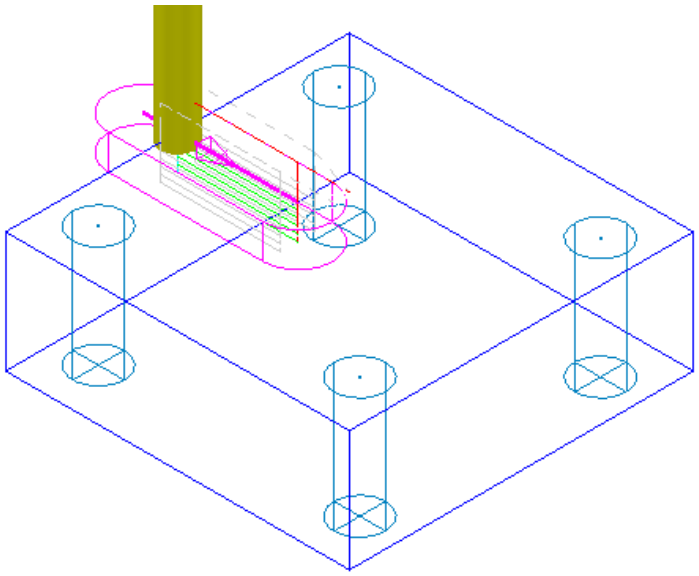



29 Select the **Slot Feature** only and then **Calculate** the toolpath.

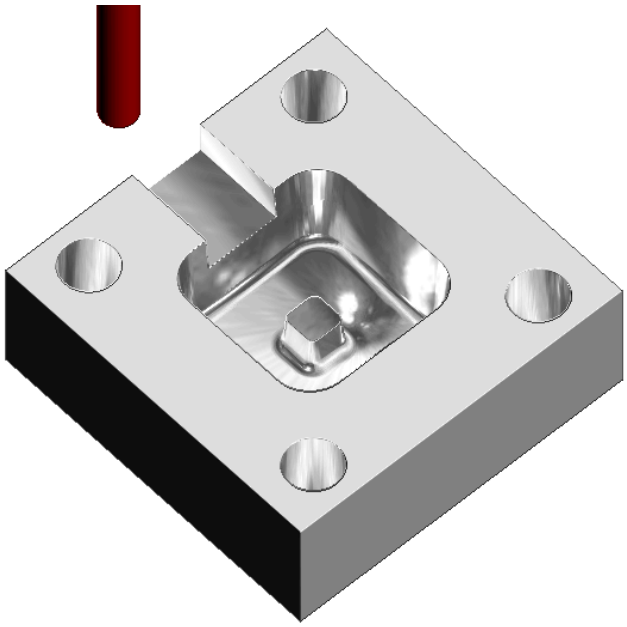
30 From the local toolpath menu use **Edit** –**Transform** – **Move Toolpaths** to make **2** copies of this **Feature Slot** strategy positioned 5mm away on both sides.

31 **Rename** these toolpaths as **EM10-SLOT-RHside** and **EM10-SLOT-LHside**.

32 Where necessary use **edit – Reorder – reverse** to maintain a ***Climb milling*** direction on these 2 toolpaths**.**



33 Continue the **ViewMILL simulation** on the last 3 toolpaths (as shown below).



34 **Save** the **Project**.


Copyright © Autodesk

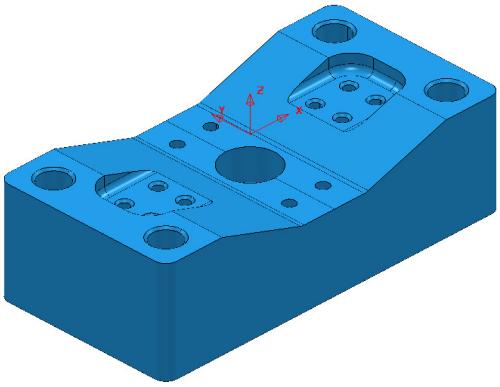
# Creating Features directly from holes in the Model

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**:-

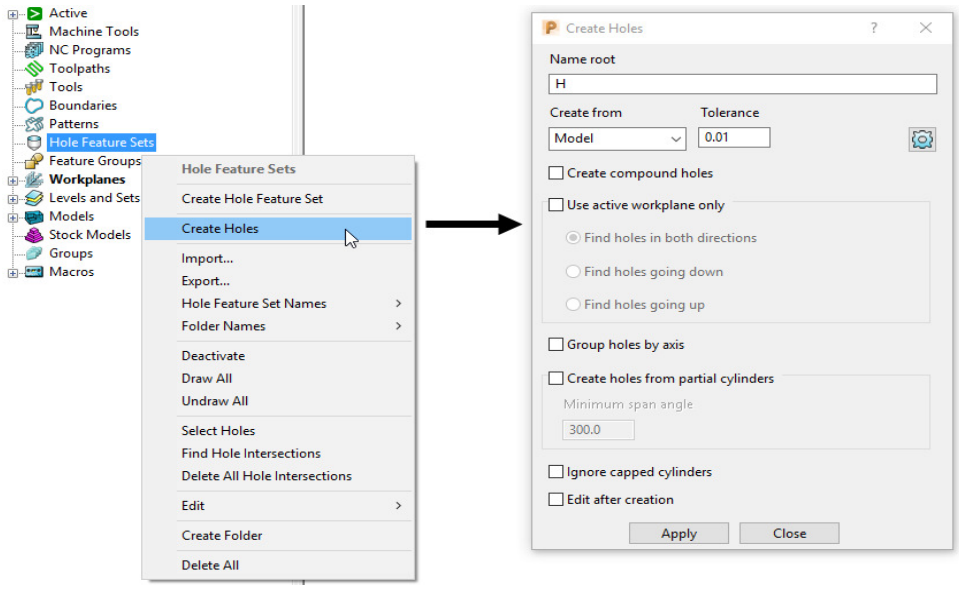
**.....\COURSEWORK\PowerMill-Projects\DrilledHoles-Example**

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**.

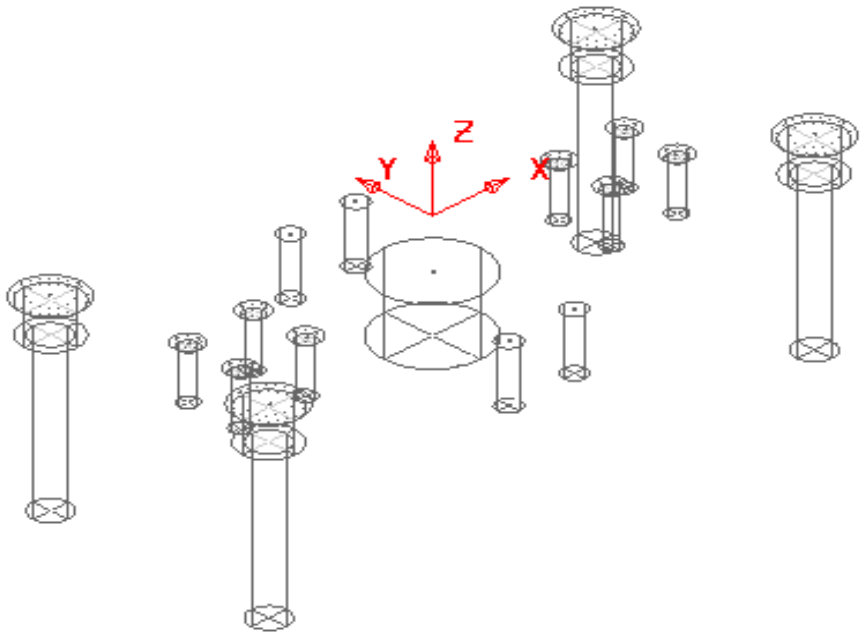
6 Select the whole **model**.

7 In the **PowerMill *explorer***, right-click on **Hole Feature Sets** and from the local menu select **Create Holes**.



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**.


Copyright © Autodesk

# 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**.

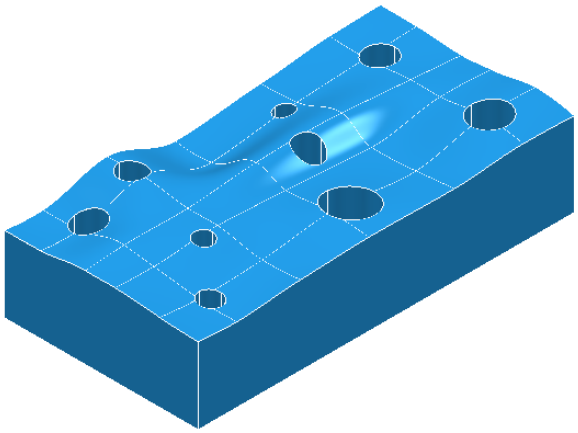
# Example

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**:

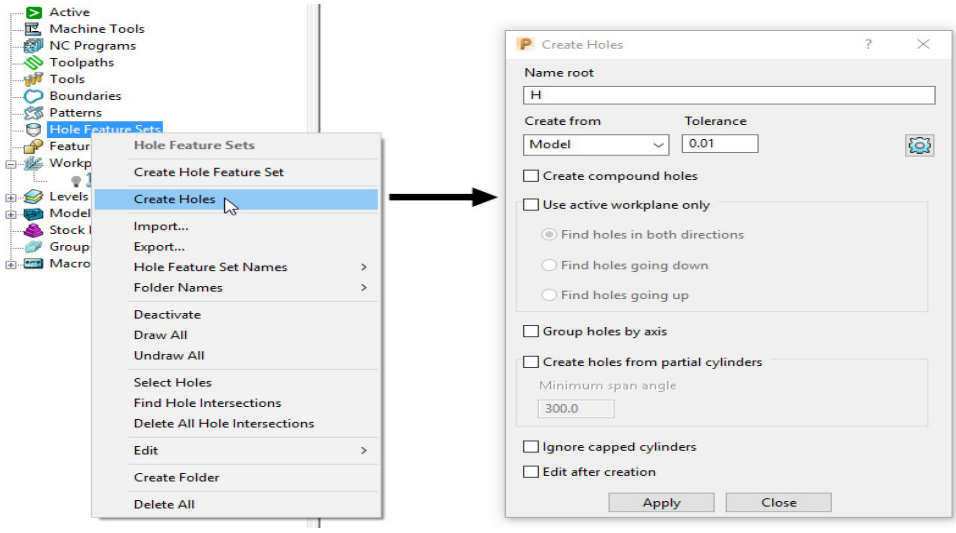
**.....\PowerMill_Data\Models\Block_with_holes.dgk**



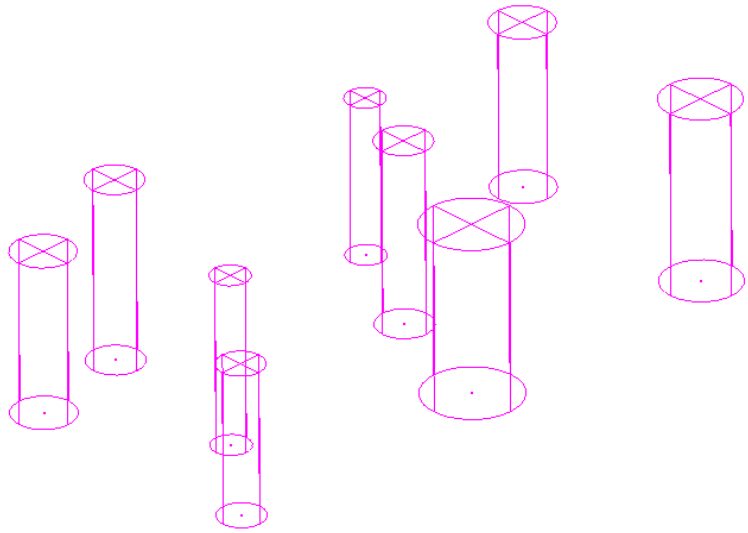
3 **Calculate** a **Block** using **Defined by** - **Box** and **Type** - **Model** (In this case the **Hole Features** will be created upside down).

4 Select and **Blank** the top face, and then select the rest of the **model** by dragging a box over it while the left mouse button is depressed.

5 Right mouse click on **Hole Feature Sets** and from the local menu select **Create Holes**.




6 In the **Feature** dialogue enter ***Name root*** **H** and select **Apply**.



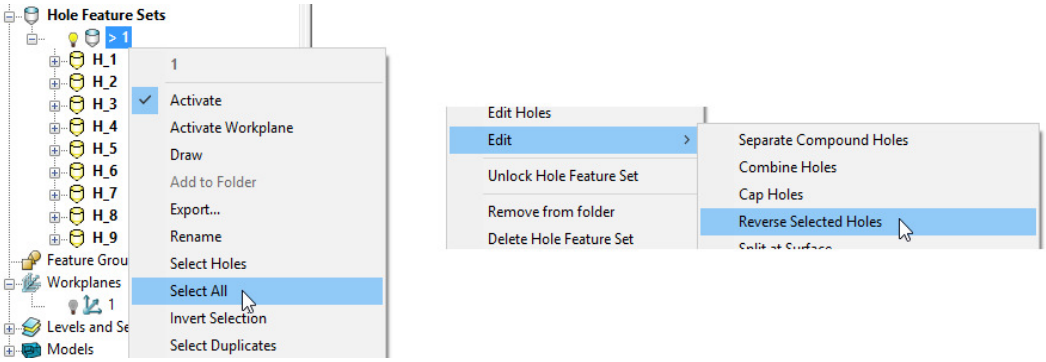
In this case the tops of the **Hole Features** are positioned at the shortest distance from the face of the **Block**. This results in all of the **Hole Features** being created upside down.

*If the **Block** had not 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.

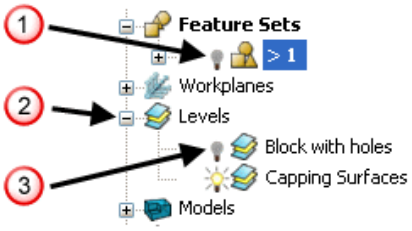



7 Right-click **Feature Set 1** and in the menu click **Select All** and then from the same menu, select **Edit** > **Reverse Selected Holes**.

8 Finally, **Unblank** and 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**.


Copyright © Autodesk

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.

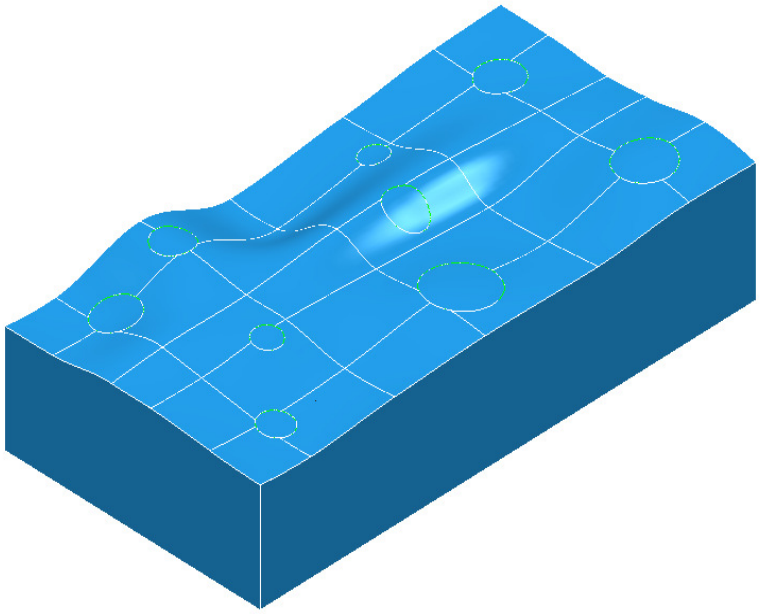



9 Click the ***light bulb*** to undraw **Feature Set 1** (1).

10 Expand **Levels** by clicking on the **+** symbol (2).

11 Click the ***light bulb*** to undraw main **surfaces** (3).

The new **Capping Surfaces** may be created inside out (Surface Normals reversed). If required they can be selected and from the local **Surfaces** menu, **Reverse Selected** can be applied.



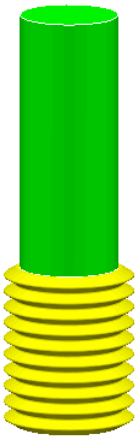
*The **Capping Surfaces** are automatically assigned to the new **Level** for easy selection.*

12 Click on the ***light bulb*** on the **Block with holes** level to re-display the main model.

13 **Save Project As**:-

**.....\COURSEWORK\PowerMill-Projects\CappedHoles-Example**

# Internal Thread Milling



**Thread Milling** is included as one of the options in the **Drilling** dialog.

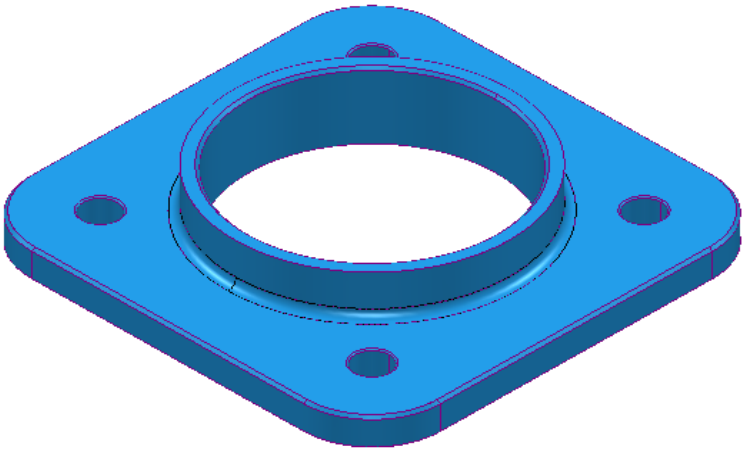
If the **Cycle Type** - **Thread Milling** option is selected, the tool plunges centrally to a specified depth, leads on to the sidewall, and then pitches upwards before leading off.

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**

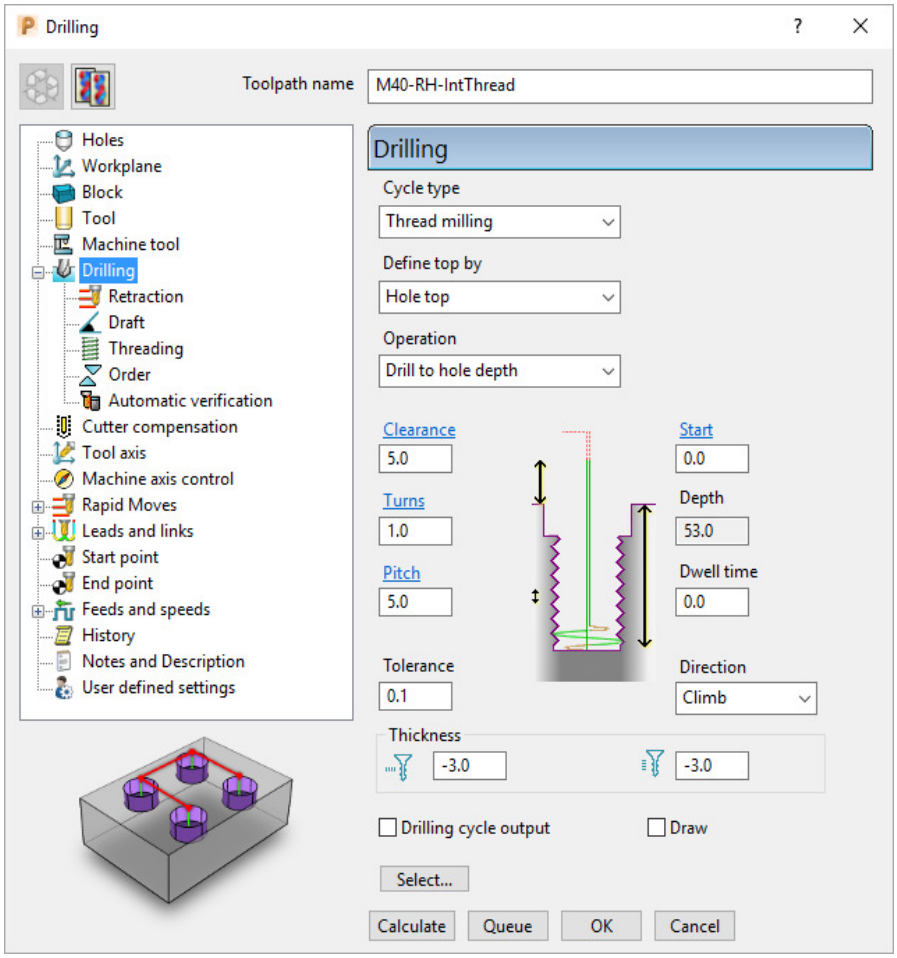
4 Activate the ***Thread Milling*** tool named **M40-Coarse-Pitch-5**.

5 Activate the **Hole Feature Set** named **Pocket** (This defines the large central hole).


Copyright © Autodesk

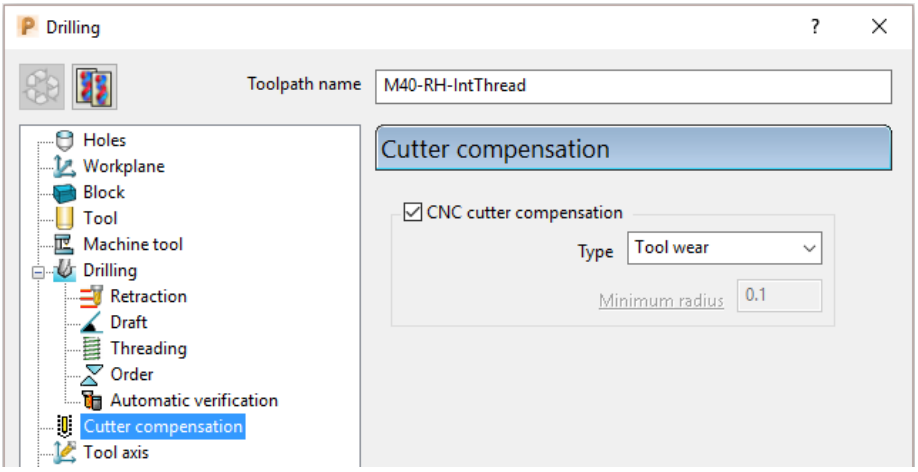
6 Select **Toolpath Strategies** and in the **Strategy Selector** dialog, select the **Drilling** tab followed by the **Thread Mill** option.

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).

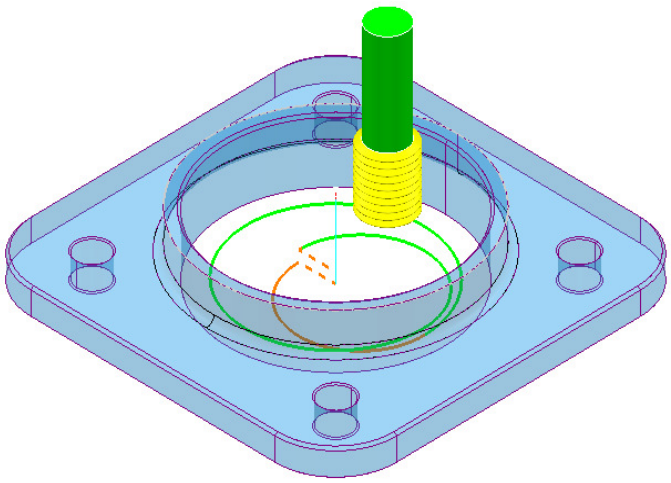



8 Select the **Cutter Compensation** page.

9 Tick the ***CNC cutter compensation*** box and select ***Type*** **Tool wear**.



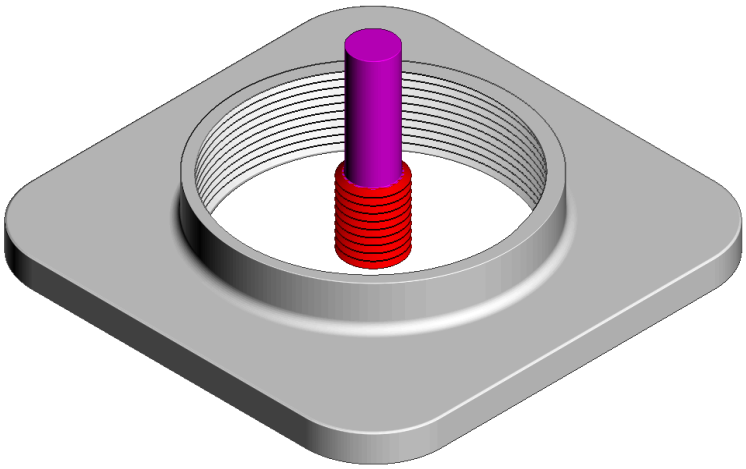

10 **Calculate** the toolpath and then **Close** the dialog.



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.

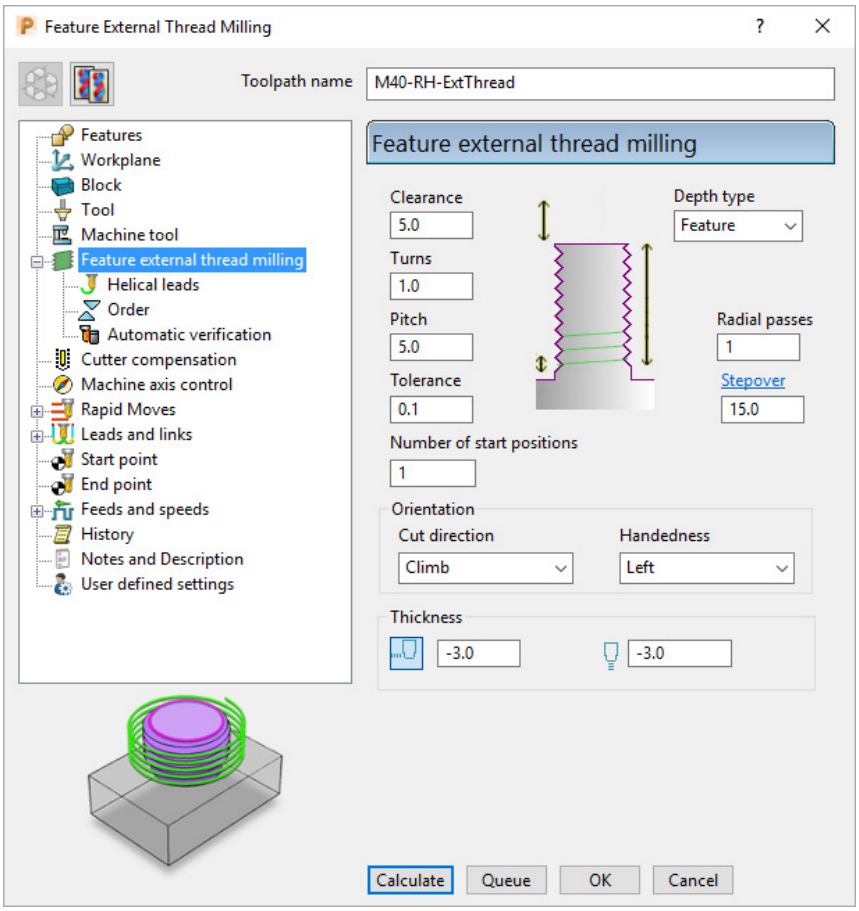
# External Thread Milling

15 Activate the **Feature Set** named **Boss** (This defines the outside of the central boss).

16 Select **Toolpath Strategies** and in the **Strategy Selector** dialog, select the **Feature Machining** tab followed by the **Feature External Thread Milling** option.

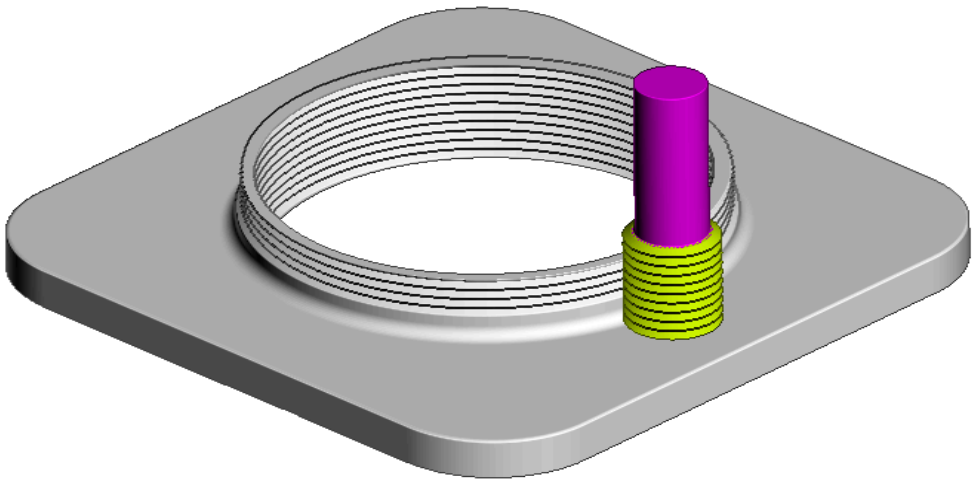

Copyright © Autodesk

17 In the dialog, set both ***Radial*** and ***Axial*** **Thickness -3**, ***Pitch*** **5**, ***Name*** **M40-RH-IntThread**, leaving all other parameters as default (as shown below).




18 **Calculate** the form and then **Close** the dialog.

19 Re-open and re-connect the **ViewMILL** session and run a **simulation** of the latest toolpath **M40_RH-ExtThread**.



20 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**.

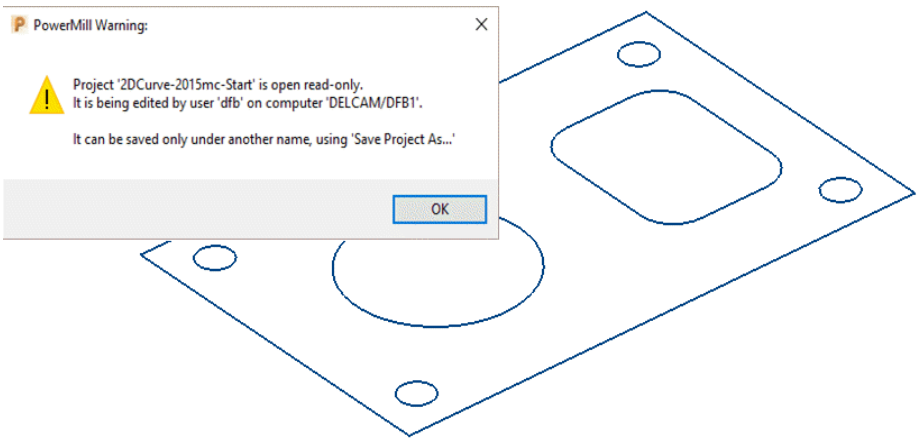
There is also a dedicated **Face Milling** strategy that calculates relative to the **Block** dimensions.

## Example 1

1 Select **File** > **Delete All** and **Tools** > **Reset Forms**.

2 **Open** the **Project**:

**.....\PowerMill_Data\Projects\2DCurve-2015mc-Start**



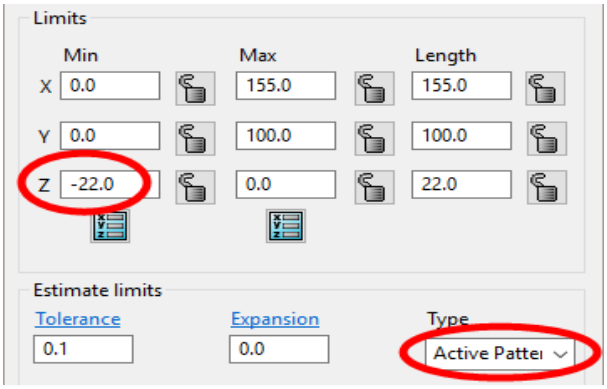

3 Select **File** > **Save Project As**:

**.....\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.



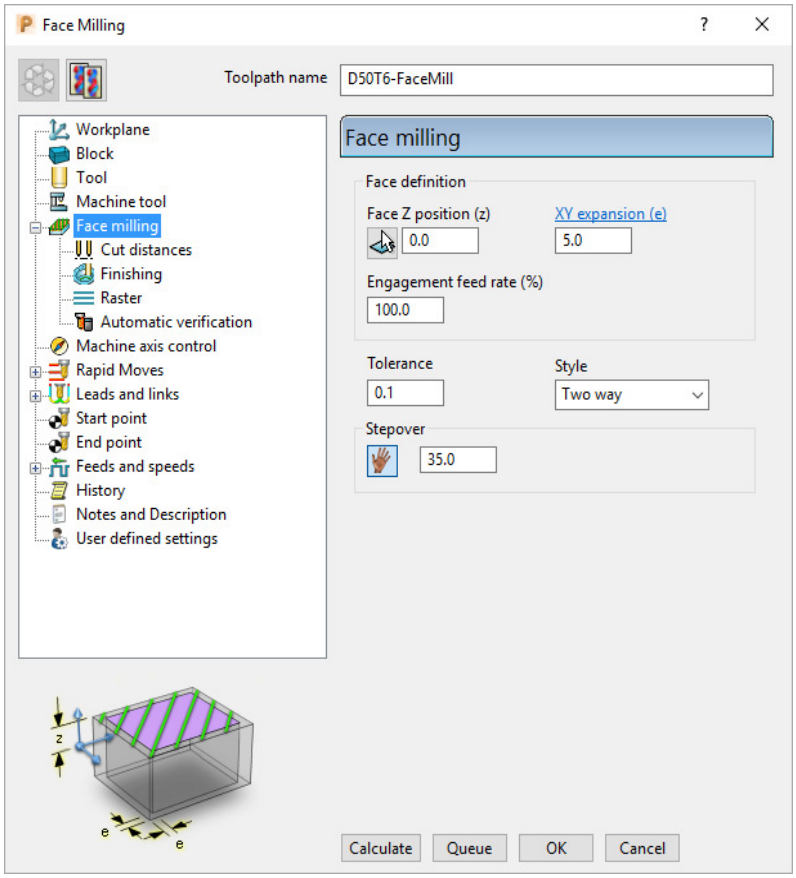


Copyright © Autodesk

7 Select **Rapid move heights** and in the **Safe area** dialog select **Calculate**.

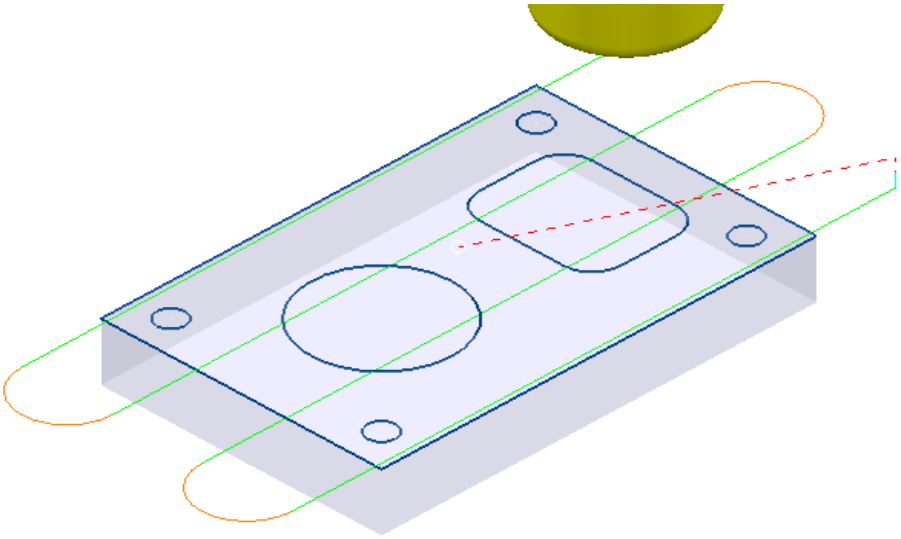
8 Activate the ***Face Milling*** tool named **D50T6**.

9 From the **Curve Machining** options select **Face Milling** and enter data into the dialog exactly as shown below.



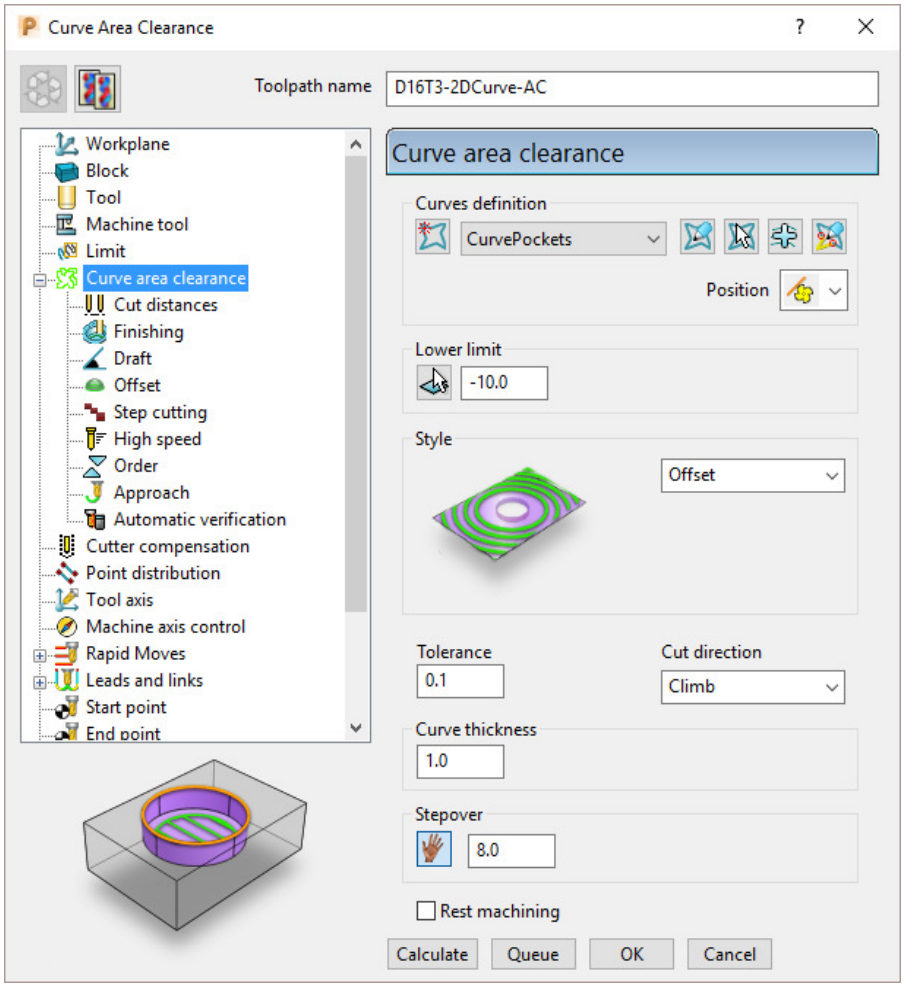

10 In the ***Toolpath connections*** **Links** page, set ***1st choice*** as **Circular Arc** and **Apply constraints** ticked with **Distance < 40**.

11 Click **Calculate** to create the toolpath and then **Close** the dialog.

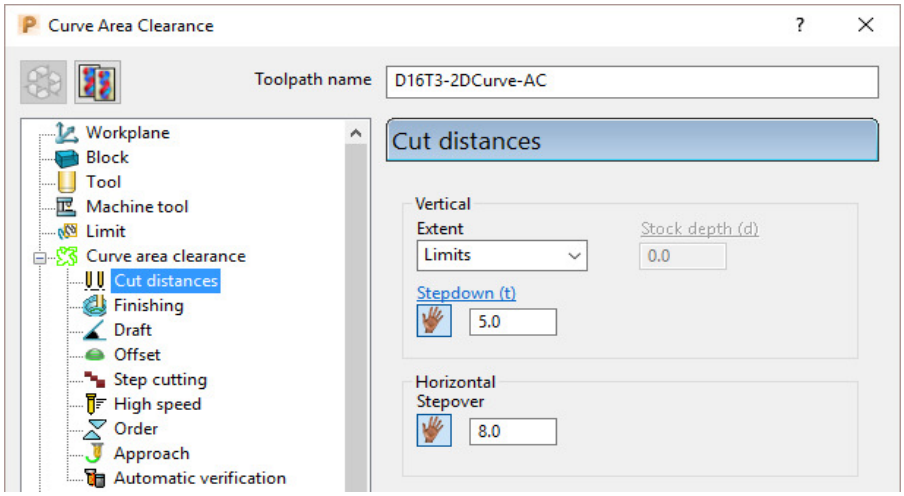


12 Activate the Tip Radiused tool named D16T3.

13 In the **Curve Machining** options select **2D Curve Area Clearance** and enter data into the form exactly as shown below.




14 Select the ***Cut distances*** page and input **Stepdown 5**.





Copyright © Autodesk

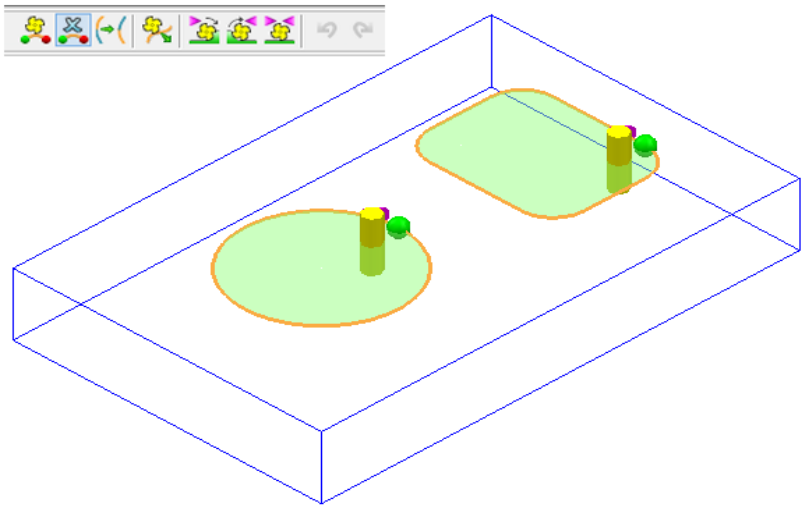
15 In the **Leads and Links** – **Lead in** page select:-

***1st 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** .

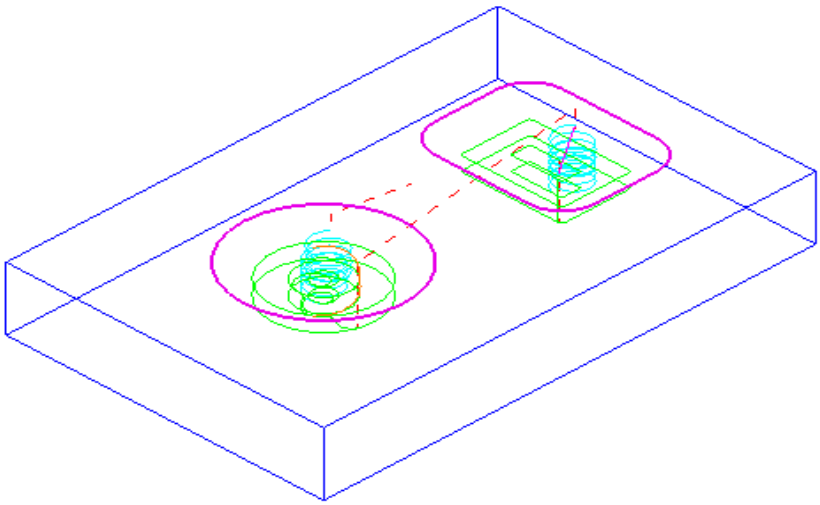
These options enable the user to have full control of direction of cut and the areas to be machined.



*While the **Curve Area Clearance** form is open, the areas to be machined are displayed as a shaded preview.*

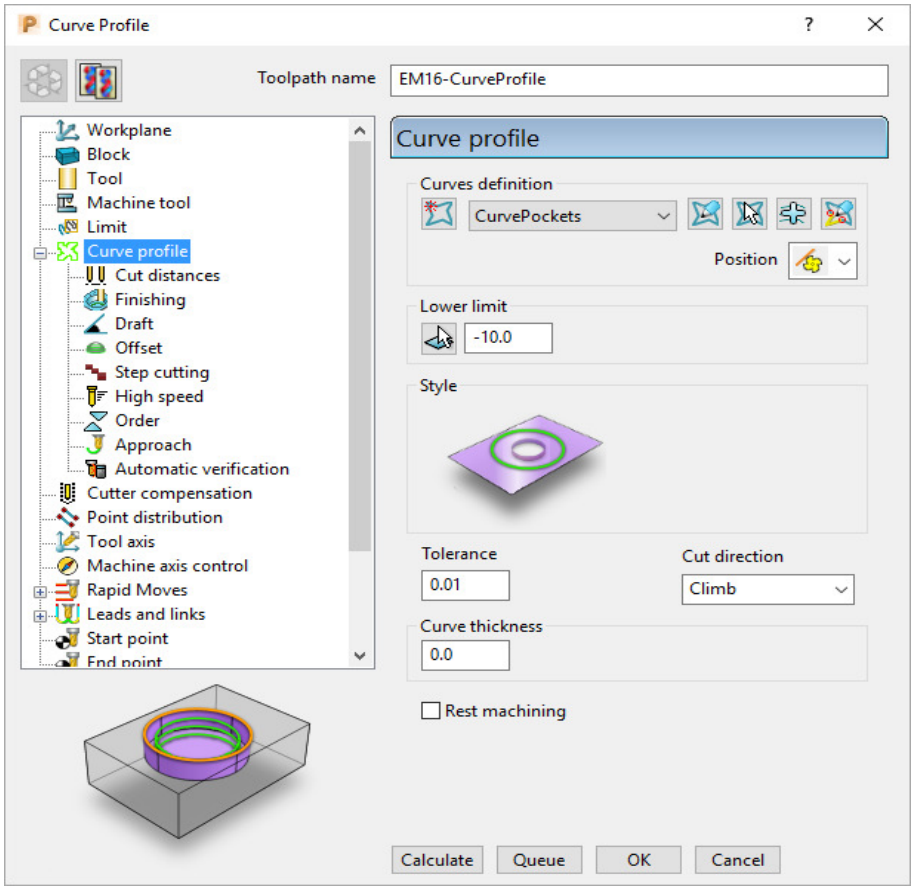
17 Select to exit the **Interactively modify machinable sections** toolbar.

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

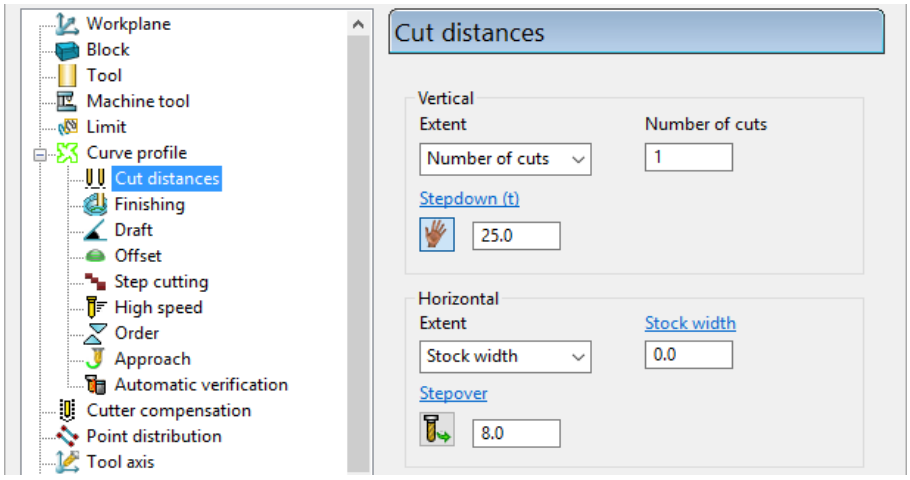


19 Activate the ***End Mill*** named **EM16**.

20 From the **Curve Machining** options select **2D Curve Profile** and enter data into the dialog exactly as shown below.




21 Select the **Cut distances** page and input **Stepdown 25**.

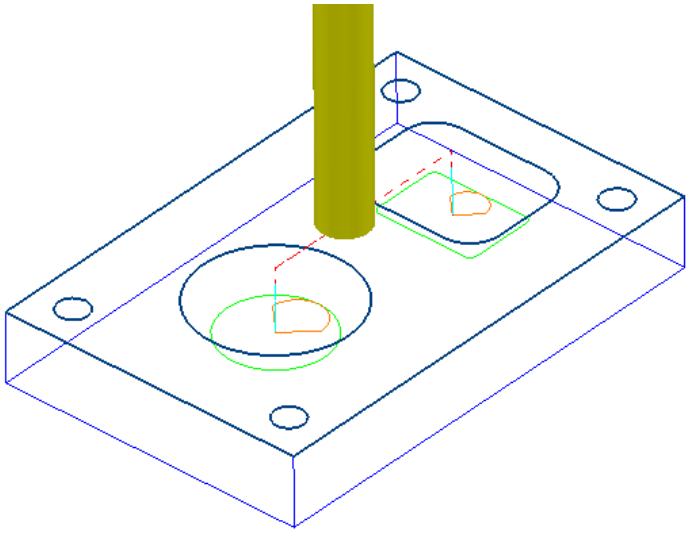



22 Select the **Lead in** page and for **1st Choice** select **Pocket Centre** and then select ***Make the lead outs the same as the lead ins*** option .

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

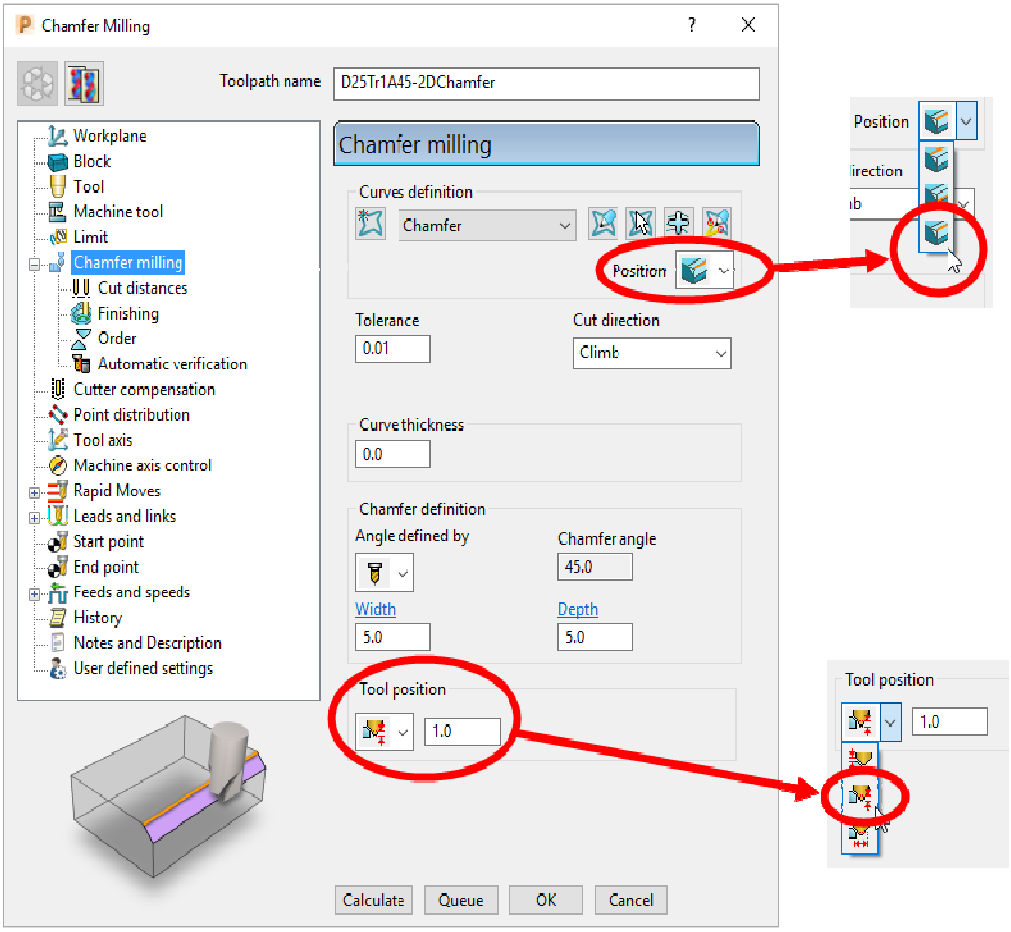

Copyright © Autodesk

*For the leads to work in this case it is necessary to apply **Edit** - **Move Start Points***

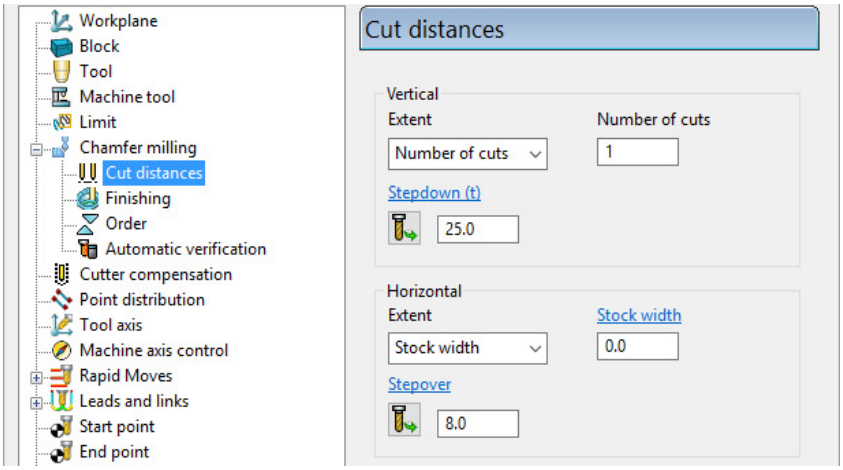


24 Activate the ***Chamfer Tool*** named **D25Tr1A45**.

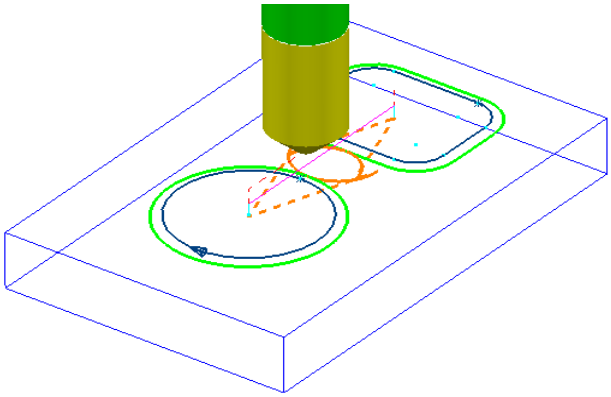
25 From the **Curve Machining** options select **Chamfer Milling** and enter data into the dialog exactly as shown below (Do not select Calculate yet!).



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).



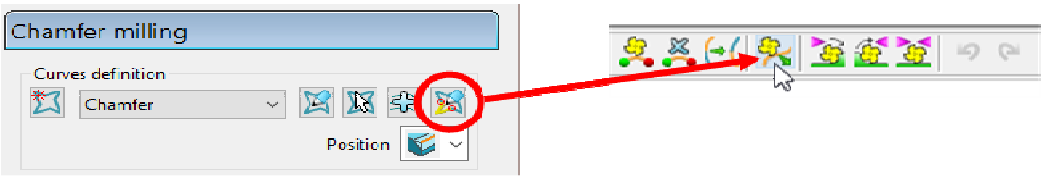
The resultant **Chamfer milling** strategy has appeared (incorrectly) on the outside of the **Pattern** segments.

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.

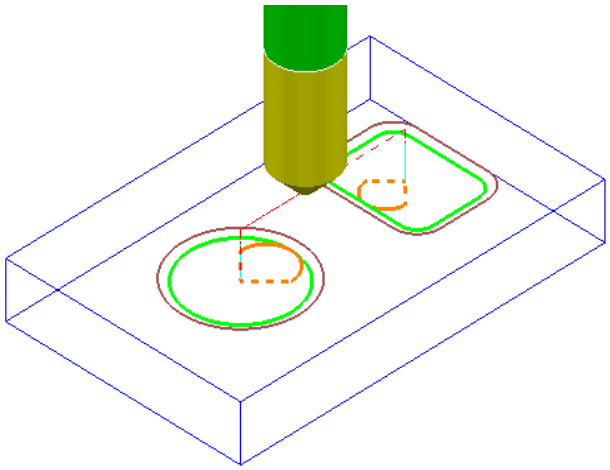
29 Select **Interactively modify machinable sections** to access the ***Edit Machinable*** Sections toolbar.

30 Select the **Reverse machining side** option, followed by to accept the changes and close the toolbar.

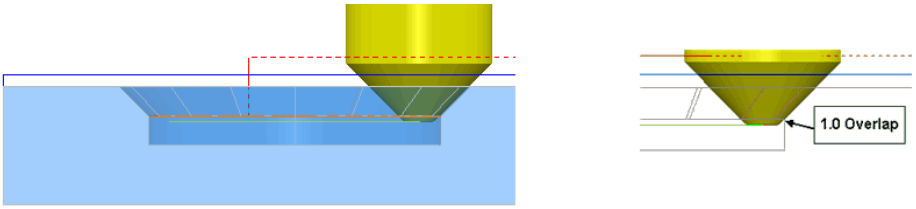



31 In the main **Chamfer milling** dialog, select **Calculate** to create the toolpath and then **Close** the dialog.


Copyright © Autodesk

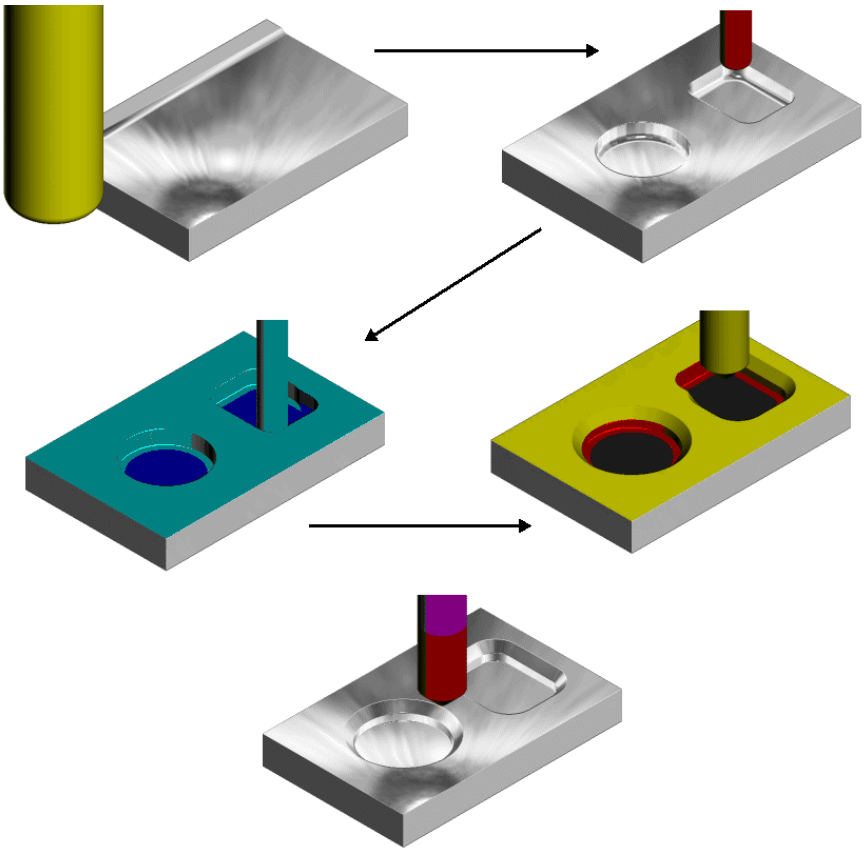


The **Chamfer milling** options selected result in the angled part of the tool overlapping the base of the chamfer by 1mm.




A 3D **model** of the finished component has been imported to provide a visual check of the **Chamfer machining**.

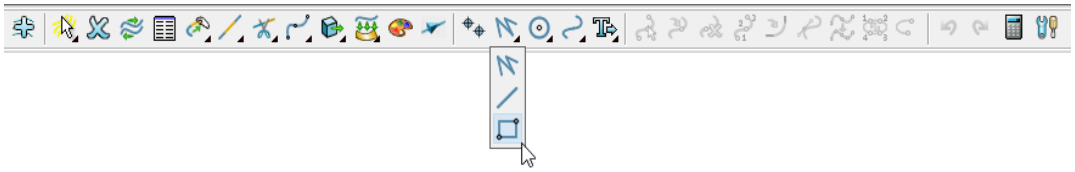
32 Run a **ViewMILL** simulation of the whole machining process.



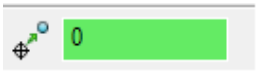
# Cutter Compensation Example

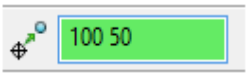
Where applicable a **Cutter Compensation** page is available in a **2.5D Machining Strategy** dialog ***explorer*** options.

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**.



5. Select **Rectangle** from the lines options.

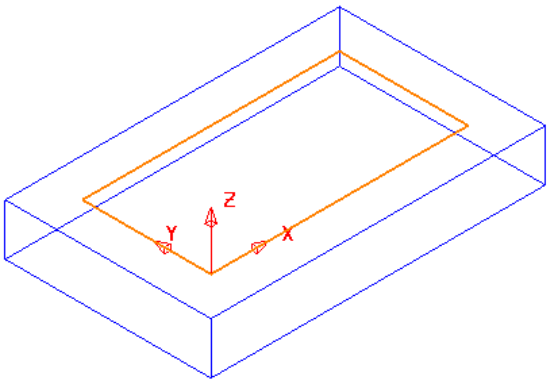



6. In the data input box (below the graphics area) input **0** to set the 1st corner of the rectangle, followed by **100 50** to define the X Y dimensions.

*The numerical input is absolute in this case* *but if it is required to use relative numerical input click on the icon to toggle it to this* *.*

7. Select **Rectangle** from the lines options again to deselect it.
8. Select the ***green tick*** to close the **Curve Editor** and accept the rectangular **Pattern**.
9. Create a Dia 10 **End Mill** named **EM10**.
10. **Calculate** a **Block** using **Defined by** – **Box** and **Type** – **Active Pattern**.
11. Input the value ***Min Z*** as **-20** and then lock both **Min Z** and **Max Z** values.
12. Input **Expansion 15** and then select **Calculate** to add stock to the X and Y dimensions of the **Block**.



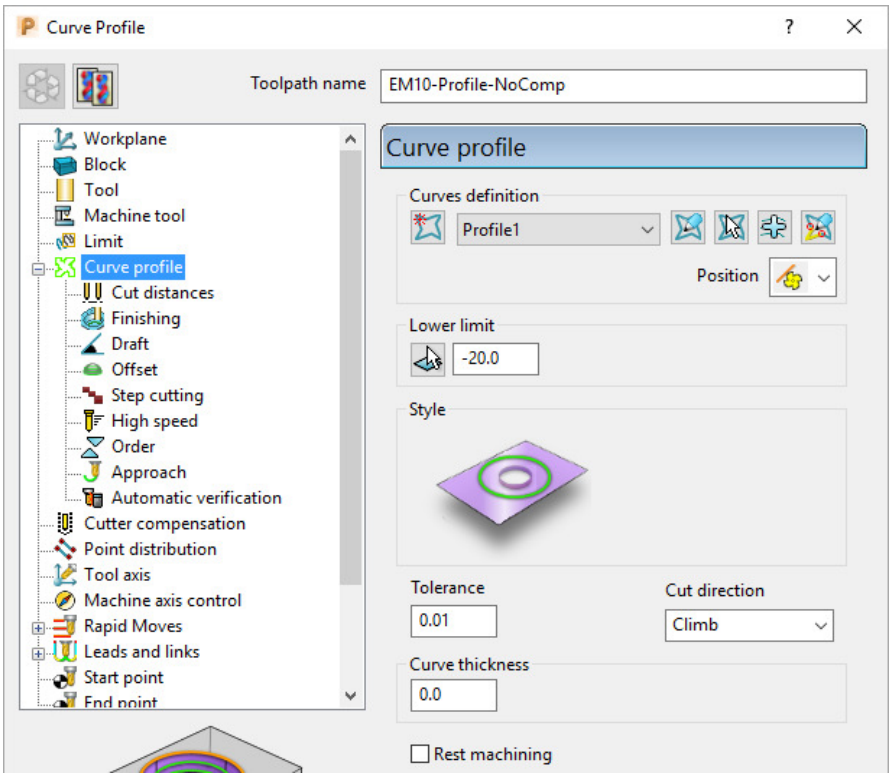


Copyright © Autodesk

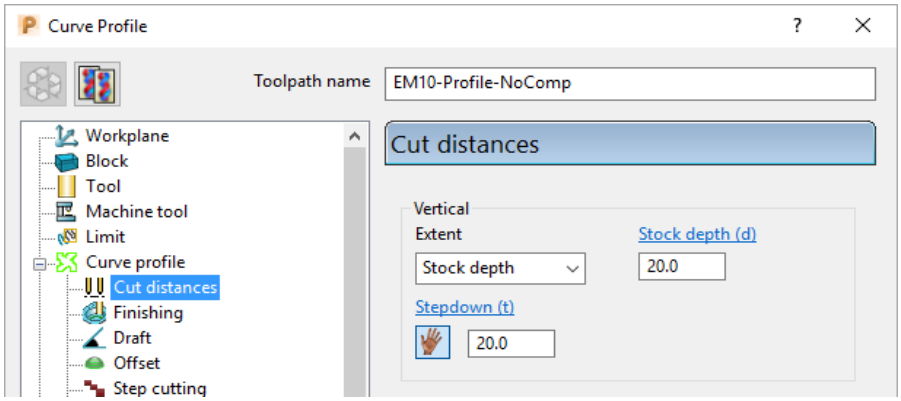
13 Select **Rapid Move Heights** and in the **Safe Area** dialog select **Calculate**.

14 In the **Strategy Selector** form select the **Curve Machining** options.

15 Then select the **2D Curve Profile** strategy and enter data as shown in the following illustration.

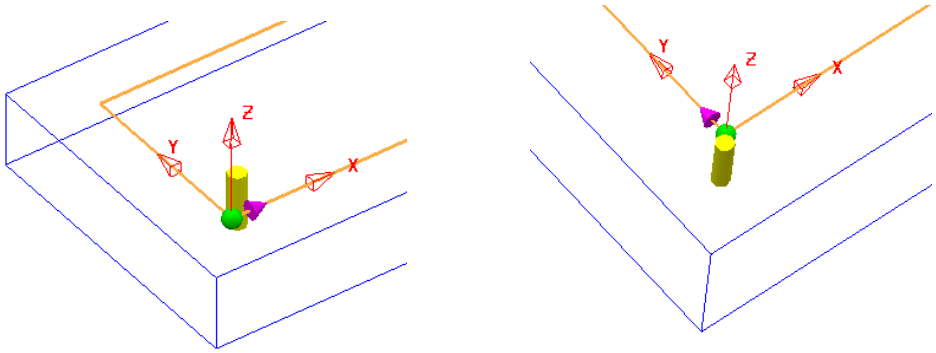



16 Select the **Cut Distances** page and input **20** for both the **Stock depth** and the **Stepdown**.




17 Return to the Curve Profile page and select the *Edit Machining Sections options.*

18 If required, select **Reverse machining side** to move the profiling pass to the outside of the **Pattern**.

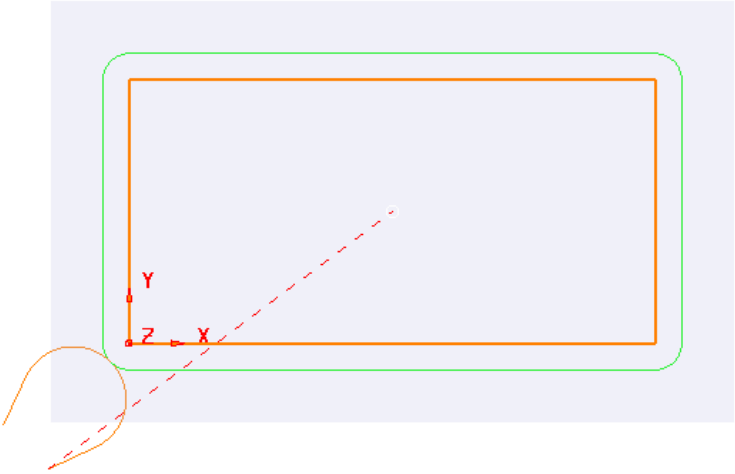



The above preview shows the cutter being switched from the inside to the outside of the **Pattern** prior to calculating the toolpath.

*This toolpath will not contain any data that relates to **Tool Compensation** output.*

19 Select the ***green tick*** to accept the changes and exit **Edit Machining Sections**.

20 **Calculate** the **Curve Profile** toolpath.




21 Open the **Leads and Links** form and select **Lead in** – **Horizontal Arc** – ***Angle*** **110** and ***Rad*** **10**.

22 Copy the **Lead In** settings to **Lead Out**.

23 Select **Lead Extensions** and input an **Extended move** of **10** for both **Extended move in** and **Extended move out**.

24 **Apply** the **Lead ins** and **Lead outs** settings to the active toolpath.

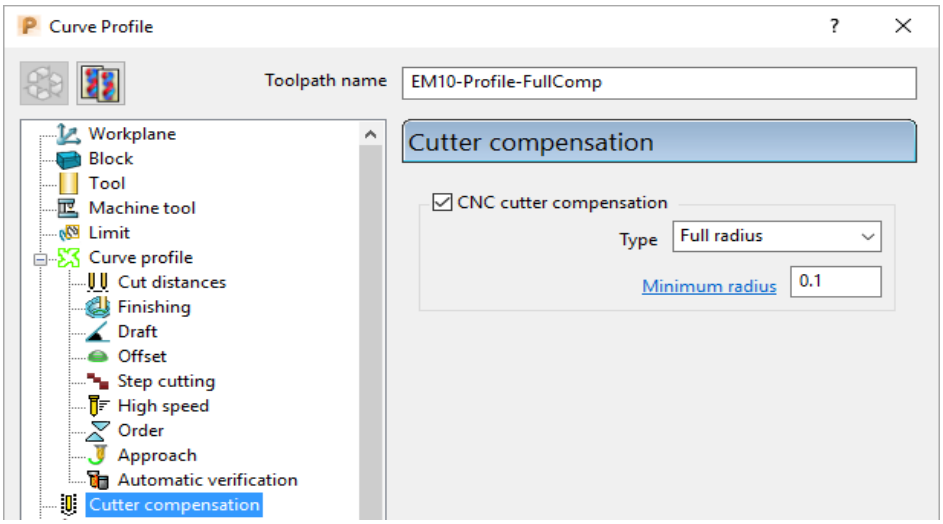
25 Right click on the active toolpath (**EM10-Profile-NoComp**) and from the local menu select **Settings** to open the **Curve Profile** dialog.

26 Once the **Curve Profile** dialog is open select the ***Make a Copy*** option.

27 **Rename** the new toolpath as **EM10-Profile-FullComp**.

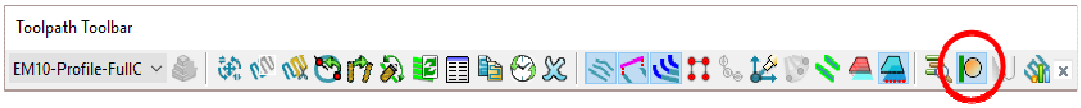

Copyright © Autodesk

28 Select the **Cutter Compensation** page, tick the **CNC cutter compensation** box, and select ***Type*** as **Full radius**.

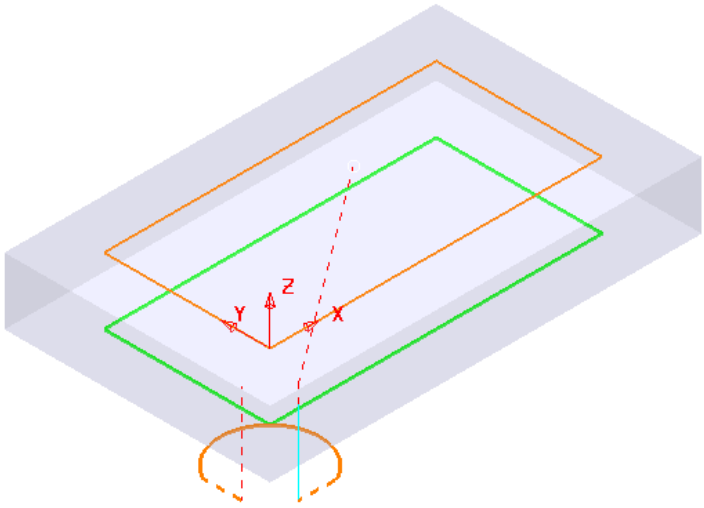



29 **Calculate** the new **Curve Profile** toolpath, but do not close the form.

30 To test the validity of the applied tool compensation the **Toolpath** toolbar needs to be active (**main** pulldown menu:- **View** – **Toolbar** – **Toolpath**).




31 In the **Toolpath** toolbar select the **Draw compensated toolpath** option.

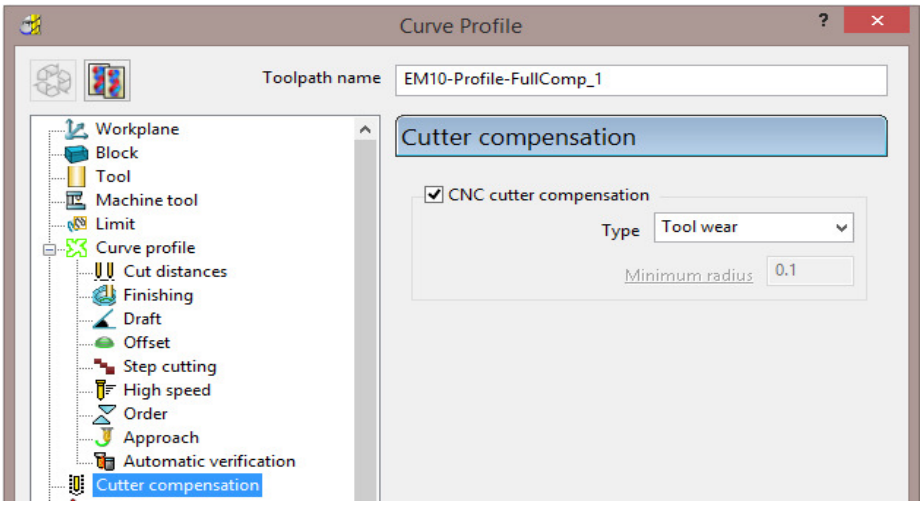



If **Full Compensation** is successfully applied it is displayed as a **thicker** green toolpath but offset inwards to the same **XY** dimensions as the **Pattern** (If the toolpath is displayed in red then the compensation is not geometrically valid).

32 *Sel*ect the **Make a Copy** option again.

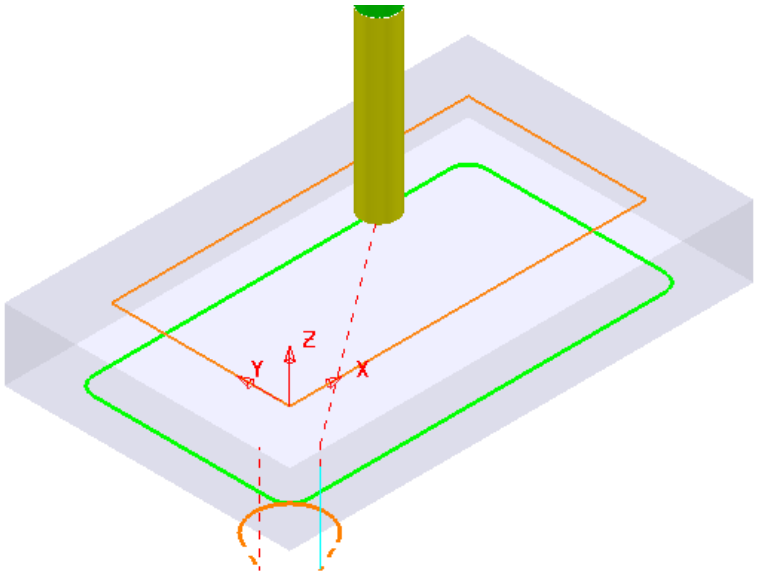
33 Rename the new toolpath as **EM10-Profile-WearComp**.

34 Select the **Cutter Compensation** page, tick the **CNC cutter compensation** box, and select **Type** as **Tool wear**.




35 **Calculate** the new **Curve Profile** toolpath, and then **Close** the dialog.

36 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.

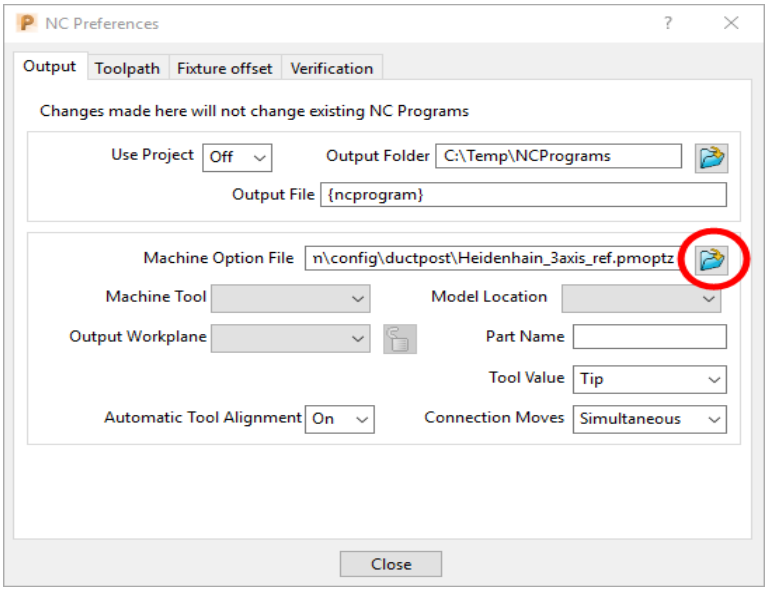
37 **Save Project As**:-

**.....\COURSEWORK\PowerMill-Projects\ToolCompExample**


Copyright © Autodesk

# Viewing the NCData output files for a heid controller

38 In the **PowerMill *explorer***, right mouse click on **NC Programs** and from the local menu select **Preferences**.



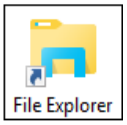

39 Open the folder containing the **Machine Option File** and from the list select the file **Heidenhain_3axis_ref.pmoptz** and click the **Open** tab.

40 Close the **NC Preferences** dialog.

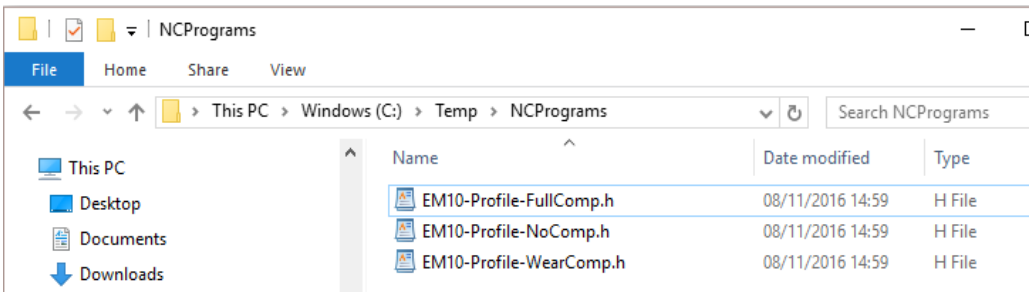
41 In the **PowerMill *explorer*** right mouse click on **Toolpaths** and from the local menu select **Create Individual NC Programs**.

42 Right mouse click on **NC Programs** and from the local menu select **Write All**.

43 Minimise the **PowerMill** window to access the **Desktop** options and select **File explorer**.




44 Once the ***File Explorer*** has opened browse to **C:\temp\NCPrograms**.




45 In **Windows *explorer***, right mouse click on an ***NCdata*** output file and from the local menu select **Open with** – **WordPad**.

46 Repeat for all of the ***NCdata*** output files and compare the output.

### Full Compensation

```
39 L M03
40 ;---------
41 ;TOOLPATH  : EM10-Profile-FullComp
42 ;WORKPLANE : World
43 ;=========
44 CYCL DEF 32.0 TOLERANCE
45 CYCL DEF 32.1 T0.01
46 CYCL DEF 32.2 HSC-MODE:0
47 L M09
48 L X50.0 Y25.0 F MAX
49 L Z10.0 F MAX
50 L X-15.443 Y-23.896 R0 F MAX
51 L Z5.0 F MAX
52 L Z-20.0 FQ1
53 L X-4.267 Y-24.201 RL FQ2
54 CC X-10.607 Y-10.607
55 C X0.0 Y0.0 DR+
56 L Y50.0
57 L X100.0
58 L Y0.0
59 L X0.0
60 CC X-10.607 Y-10.607
61 C X-24.201 Y-4.267 DR+
62 L X-23.896 Y-15.443 R0
63 L Z10.0 F MAX
64 L M09
65 L M05
66 CYCL DEF 32.0 TOLERANCE
67 CYCL DEF 32.1
68 L M30
69 ;
70 END PGM EM10-Profile-FullComp MM
```

### Wear Compensation

```
39 L M03
40 ;=========
41 ;TOOLPATH  : EM10-Profile-WearComp
42 ;WORKPLANE : World
43 ;=========
44 CYCL DEF 32.0 TOLERANCE
45 CYCL DEF 32.1 T0.01
46 CYCL DEF 32.2 HSC-MODE:0
47 L M09
48 L X50.0 Y25.0 F MAX
49 L Z10.0 F MAX
50 L X-15.443 Y-23.896 R0 F MAX
51 L Z5.0 F MAX
52 L Z-20.0 FQ1
53 L X-6.38 Y-19.67 RL FQ2
54 CC X-10.607 Y-10.607
55 C X-3.536 Y-3.536 DR+
56 CC X0.0 Y0.0
57 C X-5.0 Y0.0 DR-
58 L Y50.0
59 CC X0.0 Y50.0
60 C X0.0 Y55.0 DR-
61 L X100.0
62 CC X100.0 Y50.0
63 C X105.0 Y50.0 DR-
64 L Y0.0
65 CC X100.0 Y0.0
66 C X100.0 Y-5.0 DR-
67 L X0.0
68 CC X0.0 Y0.0
69 C X-3.536 Y-3.536 DR-
70 CC X-10.607 Y-10.607
71 C X-19.67 Y-6.38 DR+
72 L X-23.896 Y-15.443 R0
73 L Z10.0 F MAX
74 L M09
75 L M05
76 CYCL DEF 32.0 TOLERANCE
77 CYCL DEF 32.1
70 L M30
79 ;
80 END PGM EM10-Profile-WearComp MM
```

*The output file using Full Compensation is exactly matching the geometry of the 100 X 50 rectangular Pattern.*

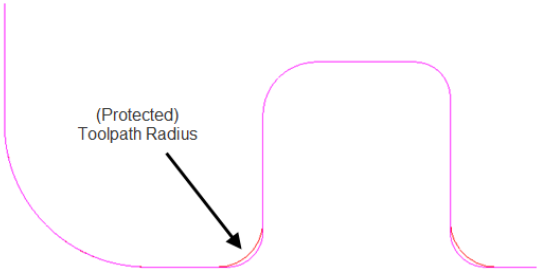
*The output file using Wear Compensation is offset by 10mm (the cutter radius) outside the 100 X 50 rectangular Pattern.*

47 Save Project As:-

**.....\COURSEWORK\PowerMill-Projects\ToolCompExample**

## Protected Cutter Compensation

With **Type** – **Full radius** selected any internal (concave) corners will require a suitable Minimum radius value included to allow for corner radii that are less than the required full offset value.



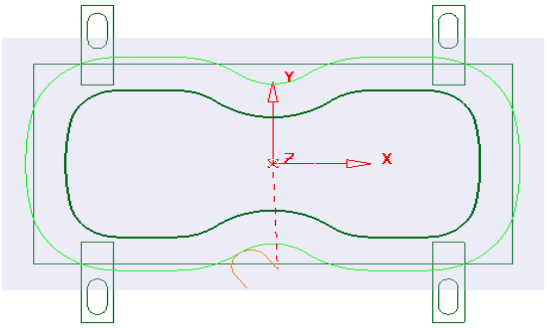


Copyright © Autodesk

# 2D Curve Machining Example – Inc. Clamp Avoidance

1. Select File > Delete All and Tools > Reset forms.
2. Open Project:

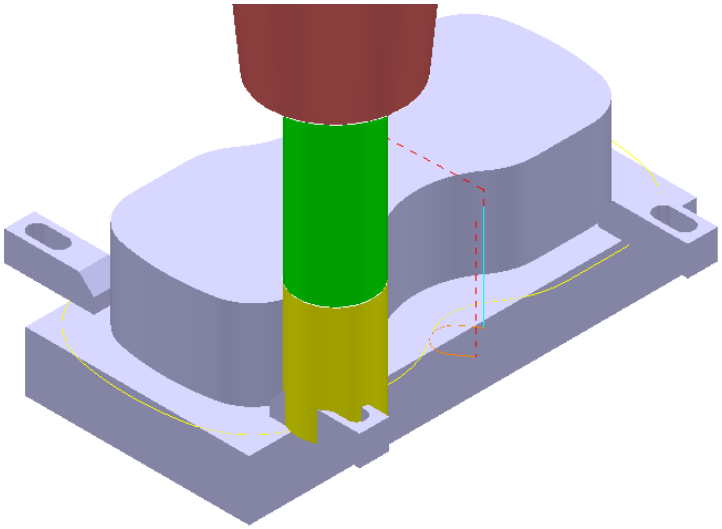
   **.....\PowerMill_Data\Projects\2DCurveProfileSplit-Start**




The imported **Project** also contains a Dia25 **End Mill** and a **Pattern** defining the 2D geometry used in the existing 2D Machining Strategy.

3. Select **File** > **Save Project As**:

   **.....\COURSEWORK\PowerMill-Projects\2DCurveProfileSplit-EX1**



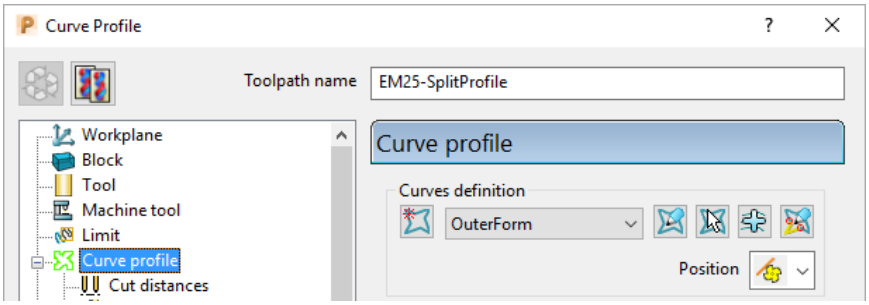
The existing 2D Curve profile toolpath is clearly passing straight through the defined clamp positions. This will be fixed by applying the Split Section option that is accessed from the 2D Curve Profile form.

4. Select a **View down Z**.
5. 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**.

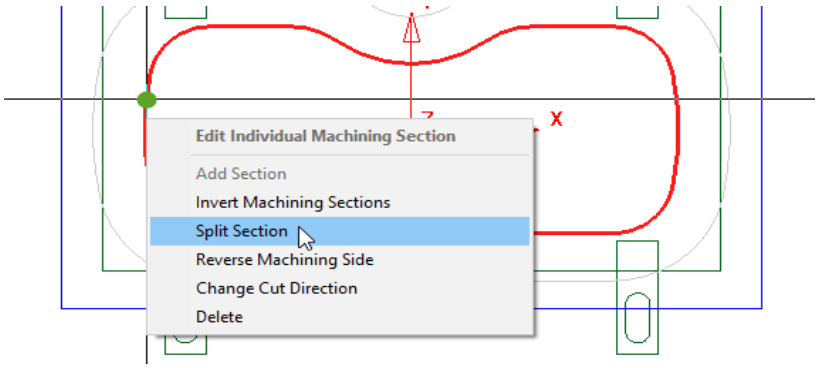
8 Once the **Curve Profile** form is open select the ***Make a Copy*** option.

9 Rename the new toolpath as **EM25-SplitProfile**.

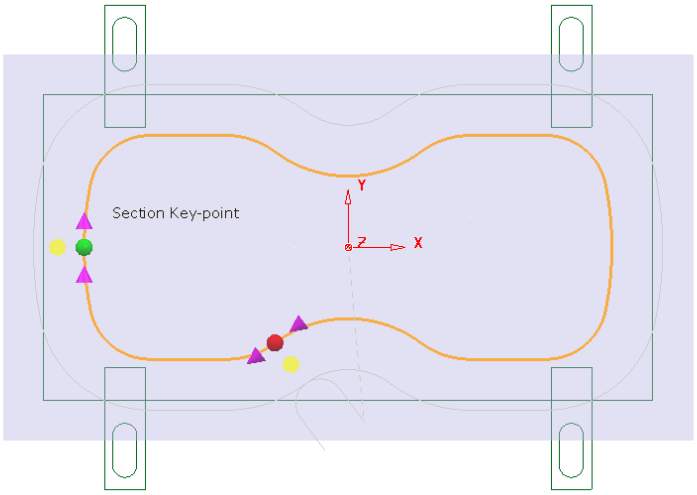



10 On the **Curve Profile** page select **Interactively modify machinable sections**.

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.




12 From the local menu select **Split Section** and an extra **start and end** point will be created on the **Pattern** at this point.

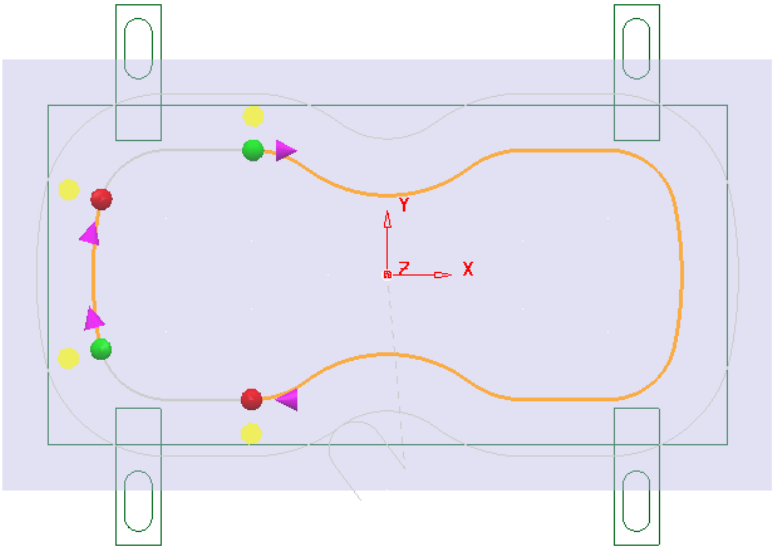



13 From the **main** pull down menus select **Draw** – **Cursor** – **Cross Hair** (***Ctrl*** **H**).


Copyright © Autodesk

*The cross hair will be displayed on the cursor to assist with alignment of start and end point.*

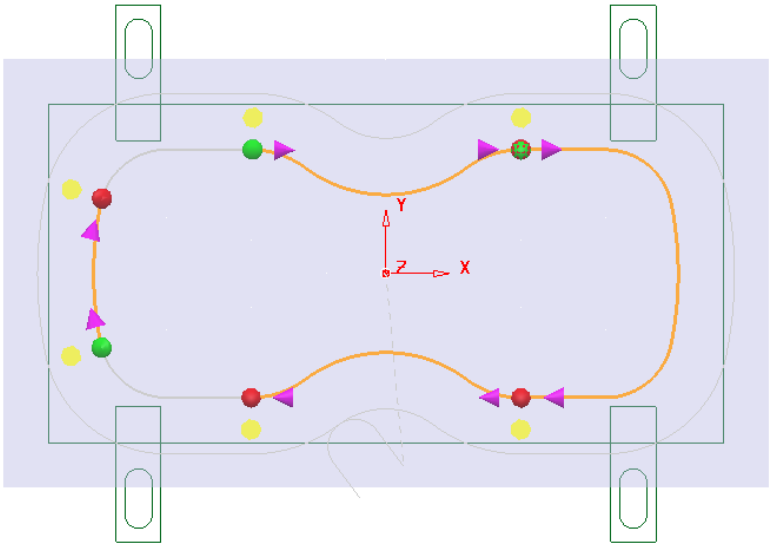
14 Using the left mouse key, drag the new **green** - start point along the **Pattern** to a position to the right of the upper left clamp (as shown below).




15 Locate and drag the corresponding **red** - end point to a position to the right of the lower left clamp (as shown above).

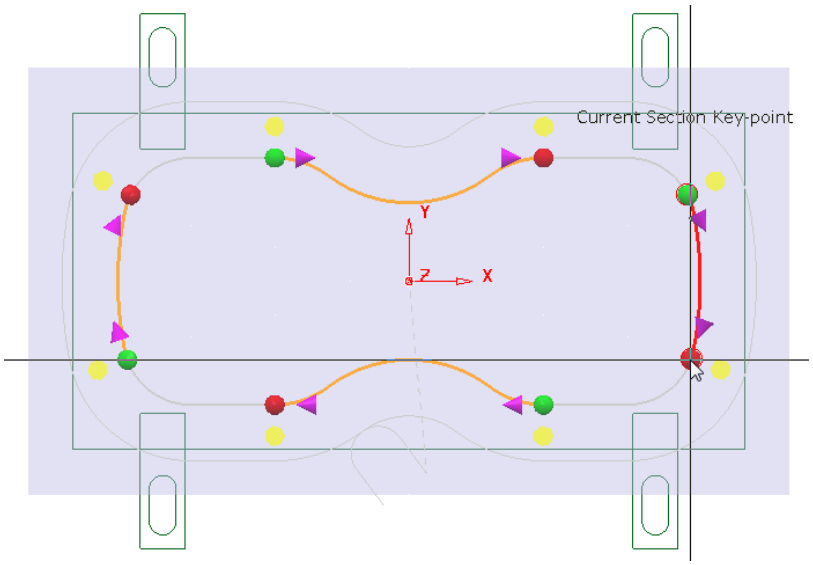
16 Repeat the process on the other (Left hand) span and drag the start and end points to the left hand side of the **Pattern** (as shown above).

17 Split the toolpath again twice, to the left of both the upper and lower right clamps (as shown below).



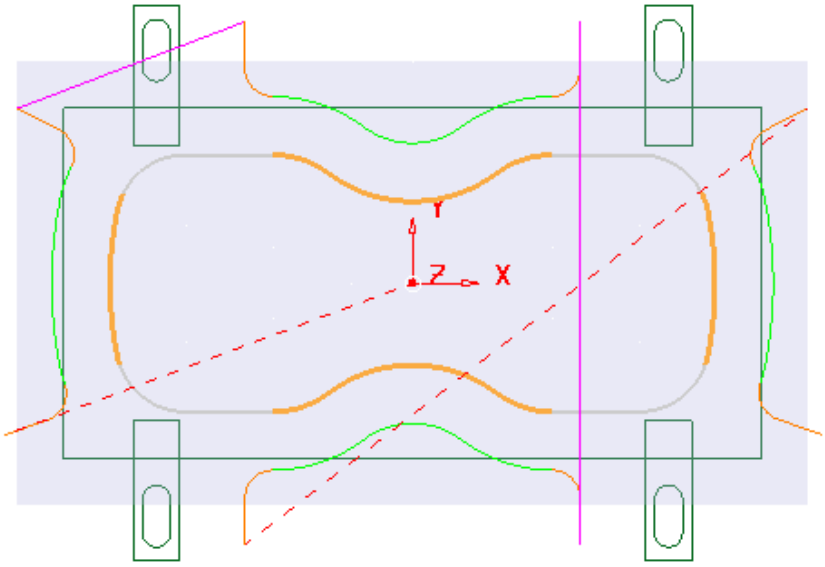

18 Move the new start and end points to form 4 distinct spans along the **Pattern** as shown in the following diagram.

*For accurate alignment use the displayed cross hairs and key points.*




19 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**.

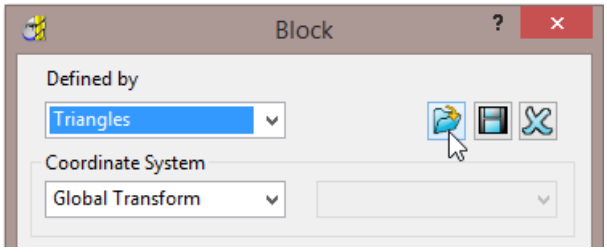
23 Open the **Block** form and select **Defined by – Triangles** and **Coordinate System - Global Transform** (See diagram on next page).

24 Select the **Load block from file** option and browse to the model:-

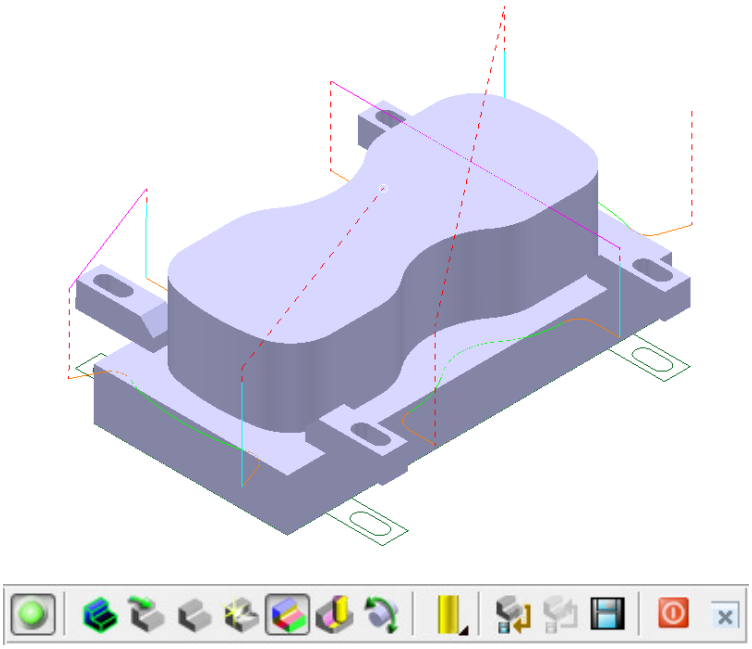
....\PowerMill_Data\Triangles\2DCurveprofileSplit.dmt


Copyright © Autodesk

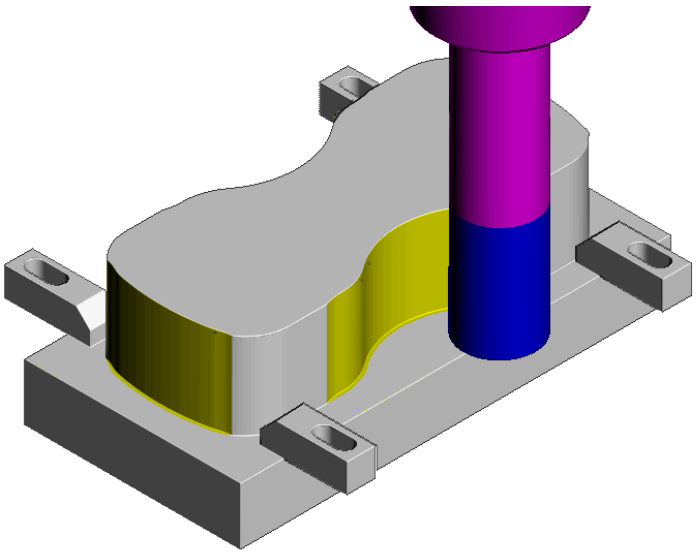
25 **Accept** the **Block** form.




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**.

Copyright © Autodesk

# 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.

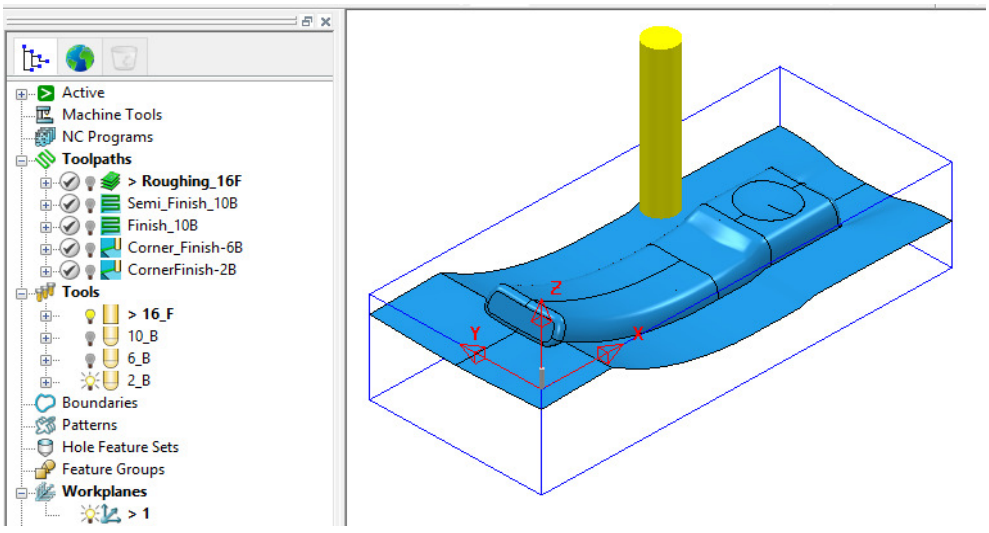
If the **NC machine** has a tool change facility then **toolpaths** created with different cutter sizes can be included in the same **NC Program**.

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**:-

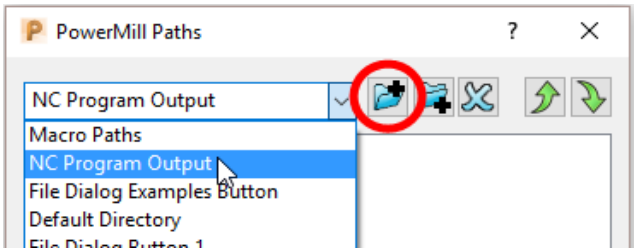
**.....\COURSEWORK\PowerMill-Projects\NCProgram-ex1**

# Defined PowerMill Paths – NC Program output

4 From the top pulldown menu bar, select **Tools > Customise Paths**.

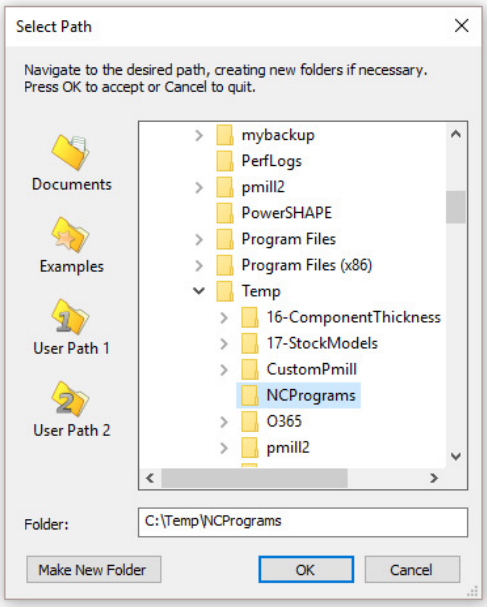
5 In the **PowerMill Paths** dialog, select **NC Program Output** followed by **Add path to top of list**

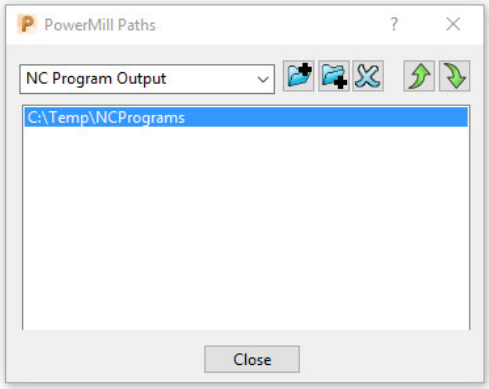
If not already defined, it is now required to set up a path to where the ***Post Processed*** files (**NCPrograms**) are to be stored.




6 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.*



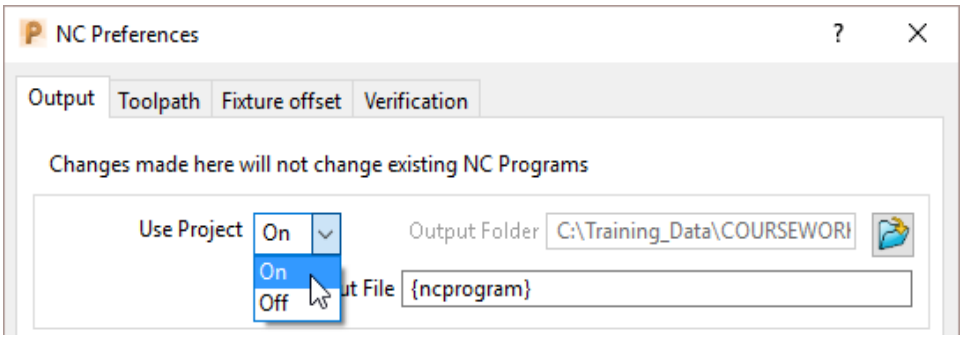

7 Click **OK** to accept and close the above dialog**.**

8 Close the **PowerMill Paths** dialogue.


Copyright © Autodesk

## Use Project On - NC Program output

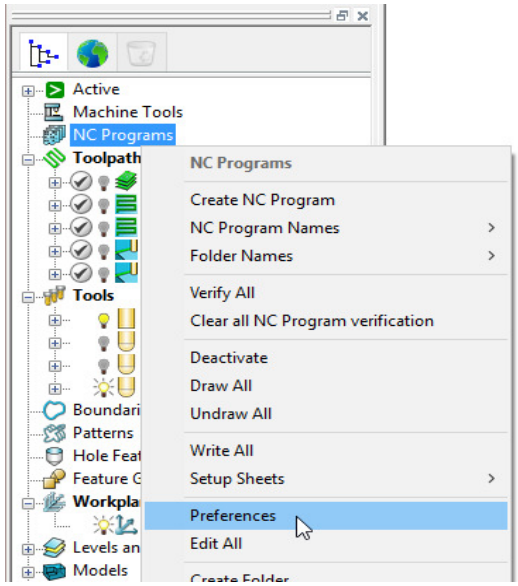
An alternative method to control the destination for **NCdata** output is to use the ***active* Project** as the default, destination for outputting **NC Data** files. In the **NCPrograms - Preferences** form, switch ***Use Project*** to **ON**.



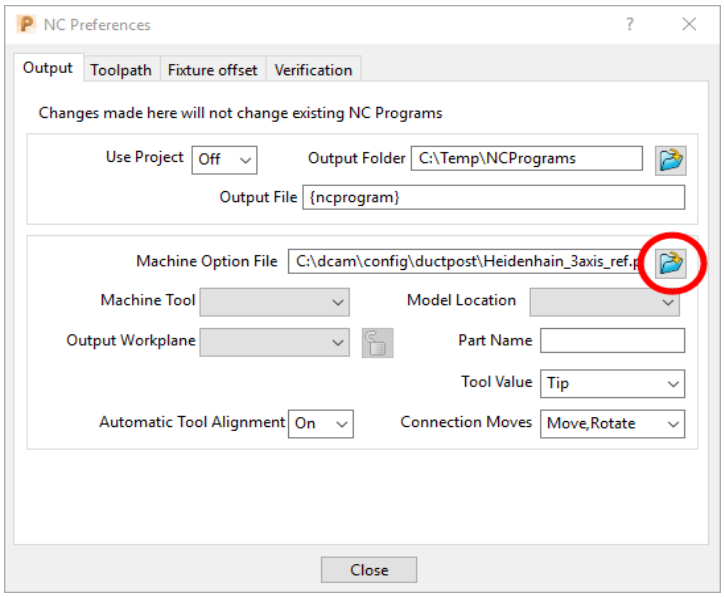

On applying the **Write** command on an ***NC Program*** a folder named **NC Programs** will be created in the currently active **Project** folder. This will not work if a **PowerMill Project** has not yet been created (**Save Project**).

## Setting the NC Preferences

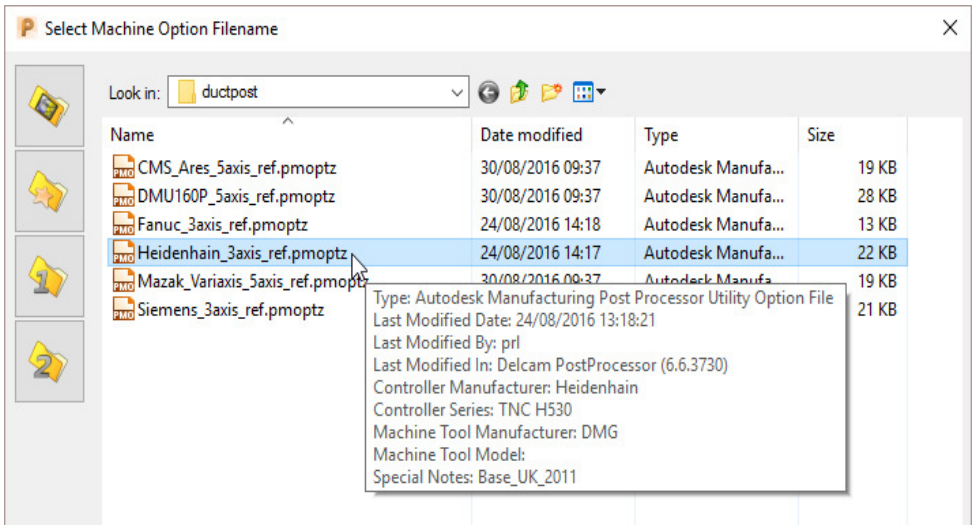
1 In the **PowerMill *explorer***, right-click **NC Programs** and select **Preferences**.




2 In the **references** dialog, select the folder icon to access the machine tool, **Option File** list.




3 From the ***Select Machine Option Filename*** dialog (below), select the **heid426.opt** and click the **Open** tab.




The selected option file translates the output to the correct Machine Tool output format.

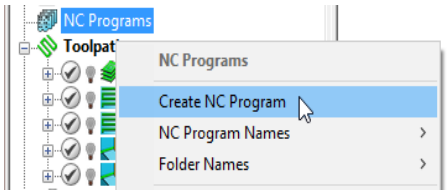
*It is not unknown to have more than one **Option File** for the same machine tool to allow for different operational requirements.*

4 Click **Accept** on the **NC Preferences** dialog.

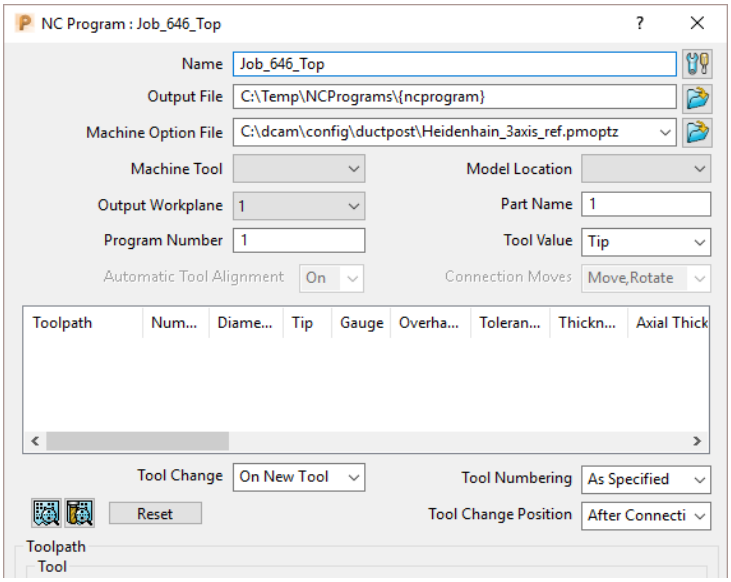

Copyright © Autodesk

# Creating the NC Program

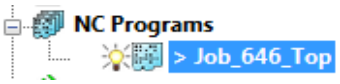
1 In the **PowerMill *explorer***, right-click on **NC Program** and from the local menu, select **Create NC Program**.




2 Enter the name **Job_646_Top** and click **Accept**.

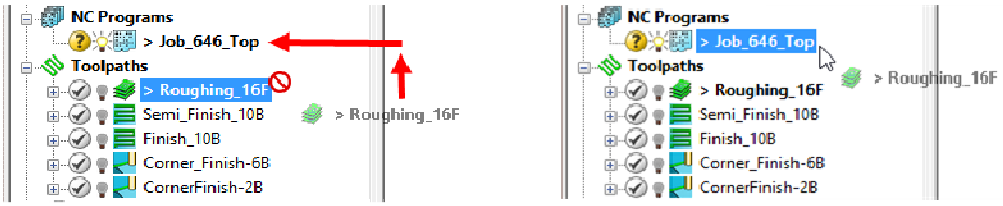



3 In the **PowerMill *explorer*** select the **+/-** box by **NC Programs** to toggle the display of the existing **NCPrograms**.



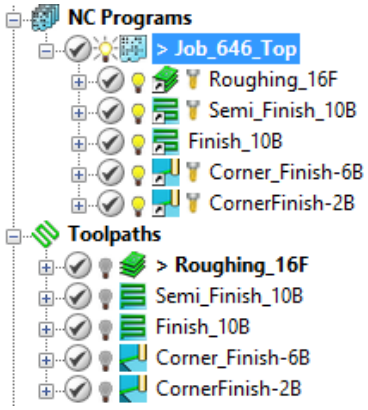

One or more **Toolpaths** can be added to the active, ***NC Program***.

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).



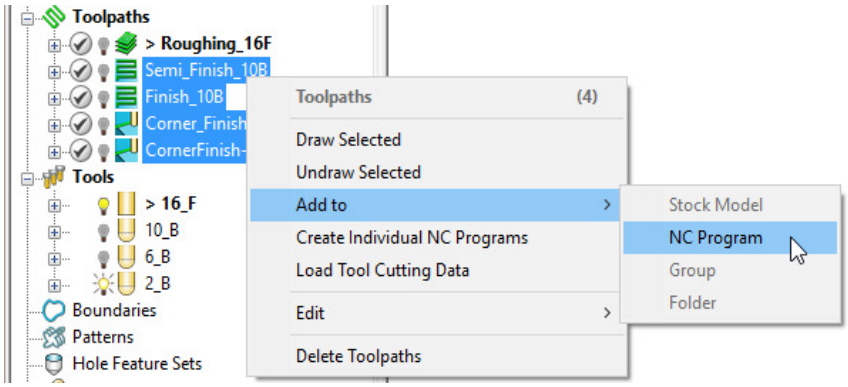

5 Continue to drag the toolpath onto the active **NC Program**, **Job_646_Top** and release the mouse button. In the **PowerMill *explorer***, **NC Programs** area select the +/- box by **Job_646_Top** to toggle a display of the contents (**Roughing_16**).

6 Drag the other toolpaths into the last file in the active **NC program** area to maintain the required machining order.



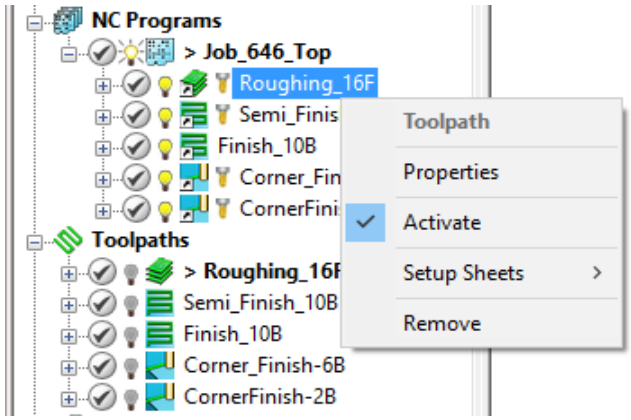

## Add to – NC Program (alternative method)

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.




Select **Add to** – **NC Program** and all of the selected toolpaths will be added to the **NC Program** list.

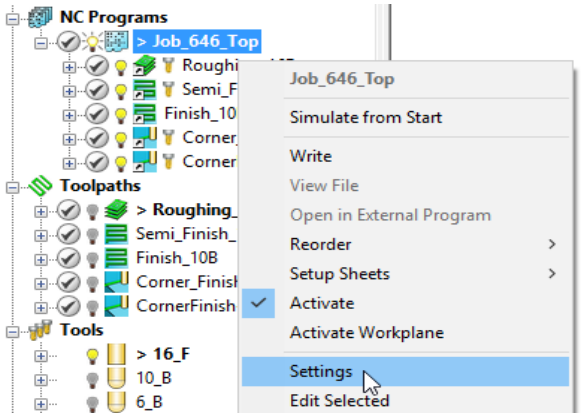
7 Right-click on any of the toolpaths in the current **NC Program** to access the local **Toolpath** options, but do not apply any of the options at this stage.



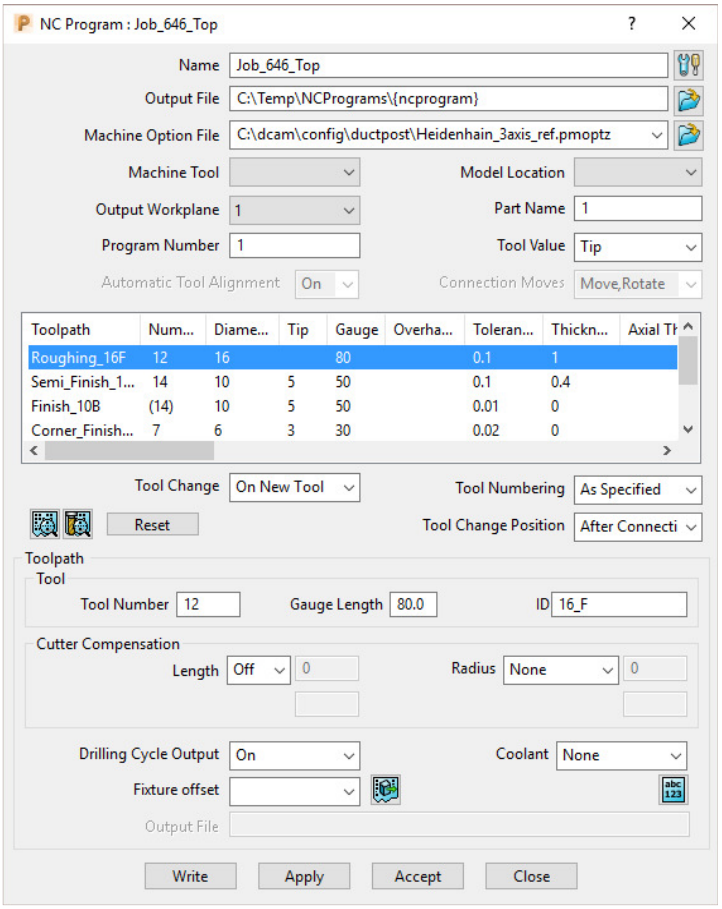


Copyright © Autodesk

8 Right-click on the **NC Program** (**Job_646_top**) and select **Settings**.




*You can open (and edit) the **NC Program** form current by selecting **Settings** from the local right click menu.*

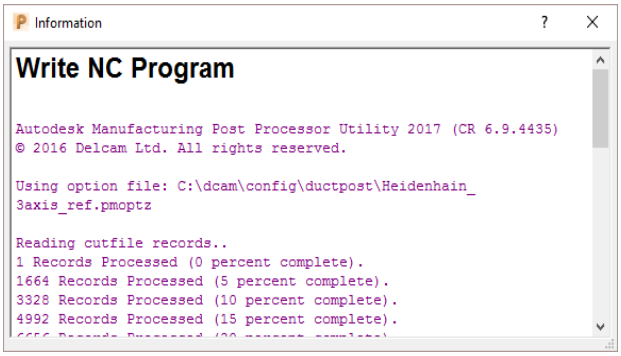



*Several of the settings assigned to a specific toolpath can be changed in the above dialog. The toolpath row is selected and relevant values (Tool Number, Coolant, etc) are then modified before clicking **Apply**. These local edits do not affect the original toolpath.*

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.

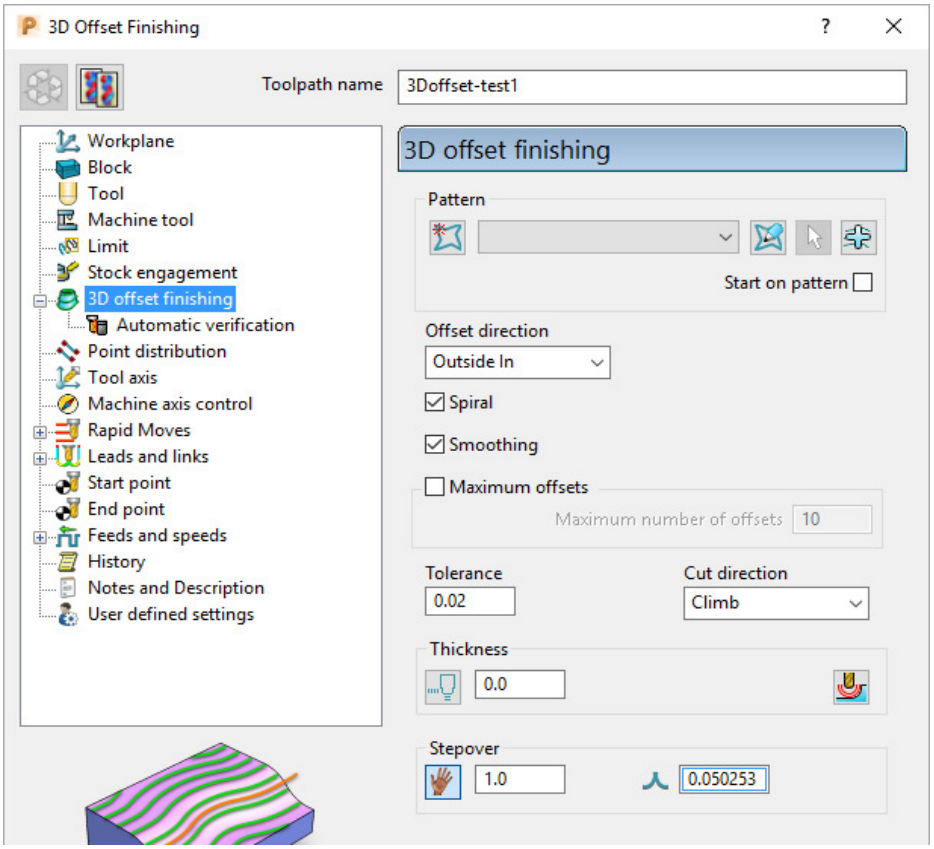



Newly created **Toolpaths** are automatically added to the ***Active*** **NC program**. It can be useful to **Create** and **Activate** an **NC Program** prior to toolpath creation to save having to drag them in later as a separate operation.

11 De-activate the ***NC Program***, **Job_646_Top** in the **PowerMill *explorer***.

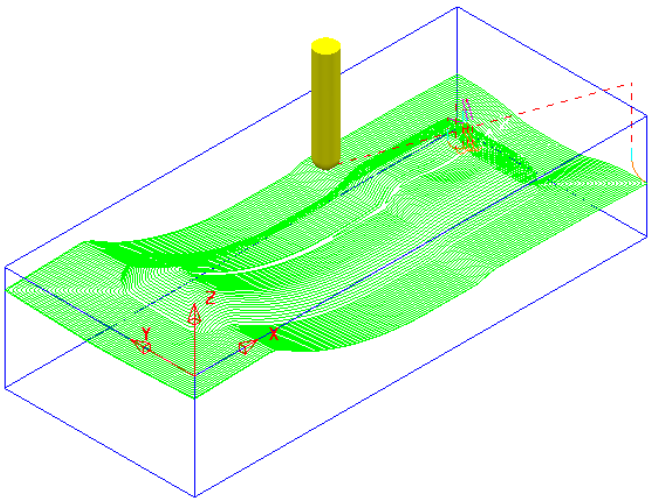
12 **Activate** the existing Dia **10** Ball nose tool named **10_B**.

13 **Calculate** a **3D Offset Finishing** toolpath, named **3Doffset-test1** using the settings and values as shown below.



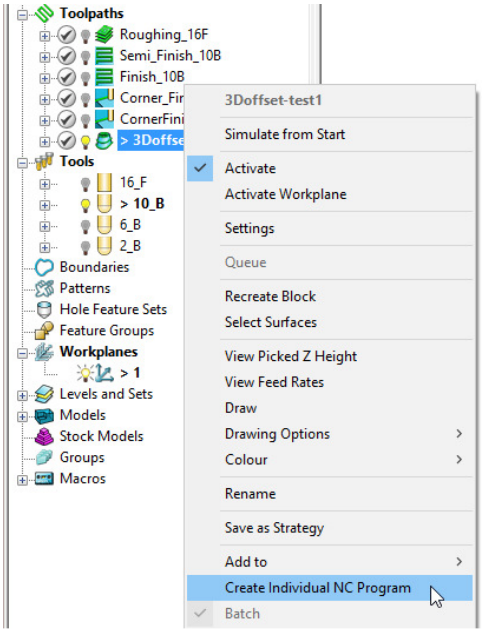


Copyright © Autodesk

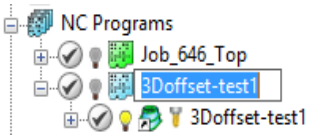


*It is not always required to output all **toolpaths** in one **NC Program**. As a result it is possible to create an **NC Program** directly from an individual **toolpath** entity.*

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).

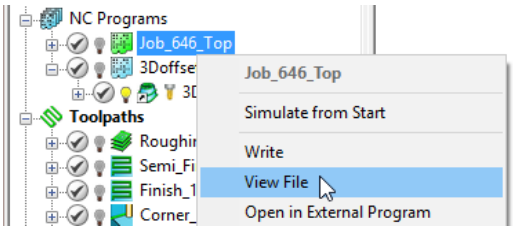



*The **NCProgram** icon for **3Doffset-test1** will remain blue until the output file is processed (**Write**). The icon will then turn to green.*

# Viewing the NC-Data output

Once Post-processed, the **NC Data** output file can be viewed directly from the local menu.

- Right-click on the ***NC Program***, **Job_646_top** and from the local menu, select **View File**.



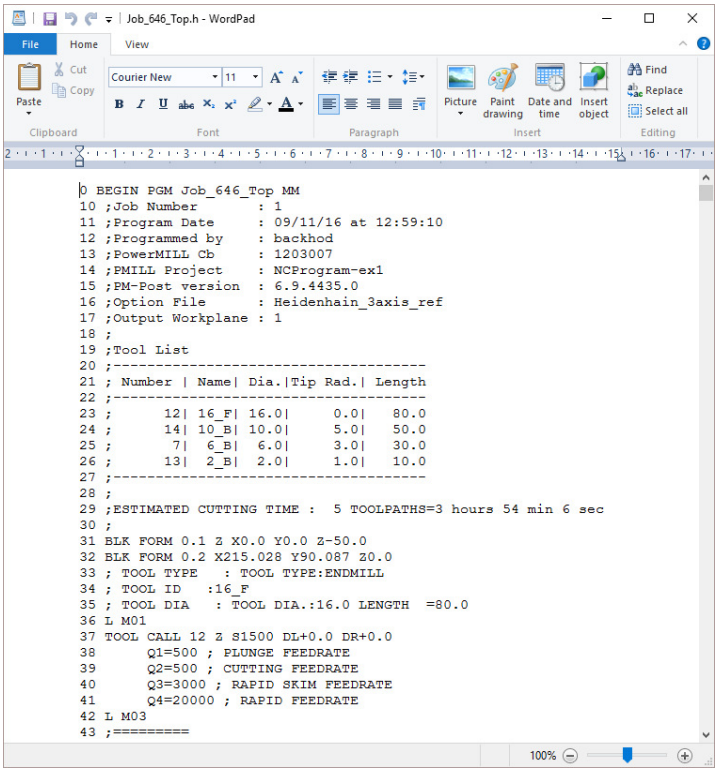

An **Information** dialog will open to enable the user to view (but not edit) the contents of the NC Data output file.

# Manually Editing the NC-Data output

If the user needs to edit the code manually, then the NC-Data output file will have to be edited in Wordpad.

1. From **Windows Explorer**, browse to **C:\temp\NCPrograms** to where the **NC-Data** files are downloaded.

2. Right-click on one of the **NC-Data** files and from the local menu, select **Open With > Wordpad**.




3. From the **main** (top) toolbar select **File > Save** to update the **Project** followed by **Close**.


Copyright © Autodesk

# 14. Setup Sheets

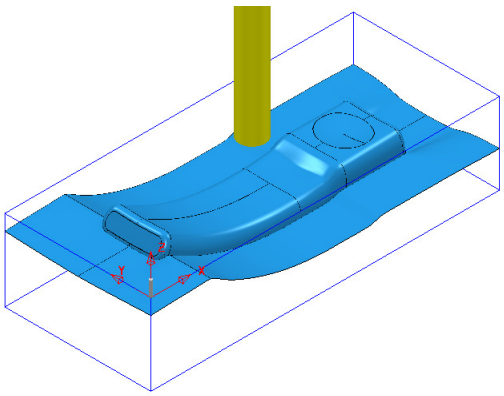
## Introduction

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**:

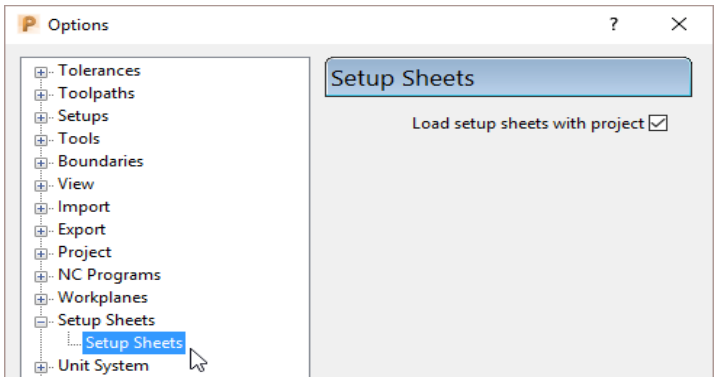
**.....\PowerMill_Data\Projects\SetupSheet-Start**



3 Select **Save Project As**:

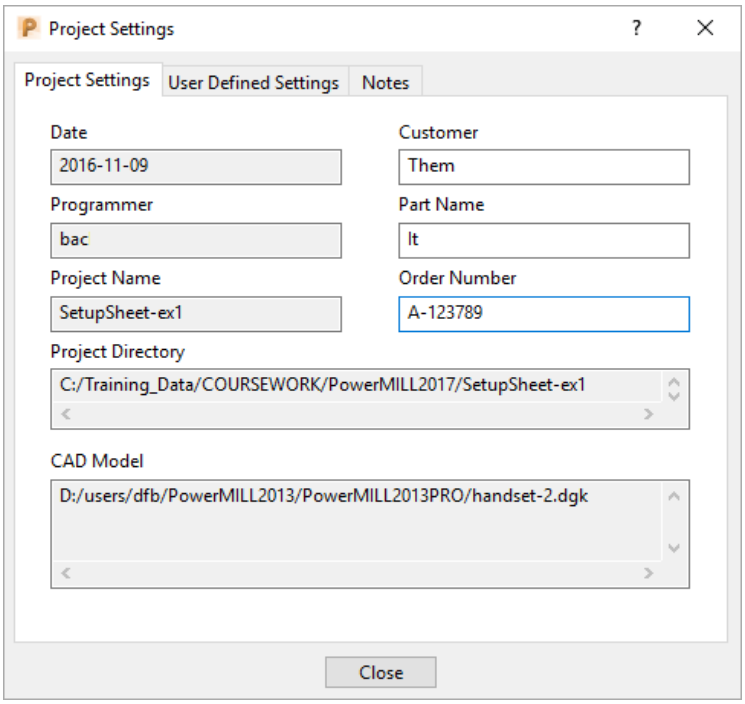
**.....\COURSEWORK\PowerMill_Projects\SetupSheet-ex1**

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.

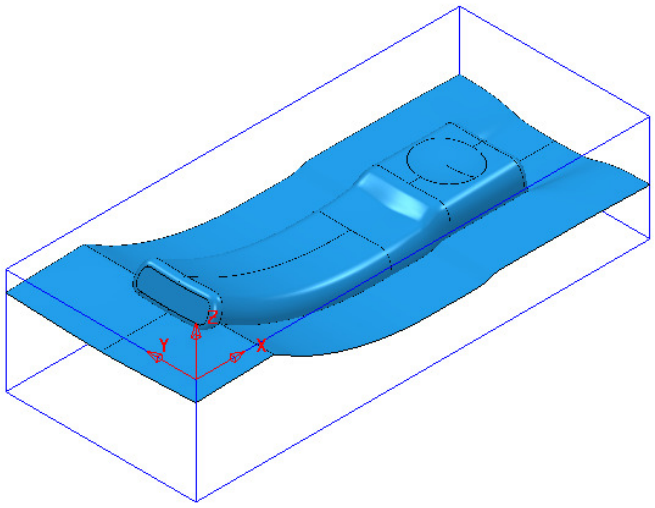



7 Input **Customer**, **Part Name**, and **Order Number** details as shown above and then **Close** the dialogue.

*There are 2 other tabs for the user to input additional information as **User Defined Settings** and **Project Notes**.*

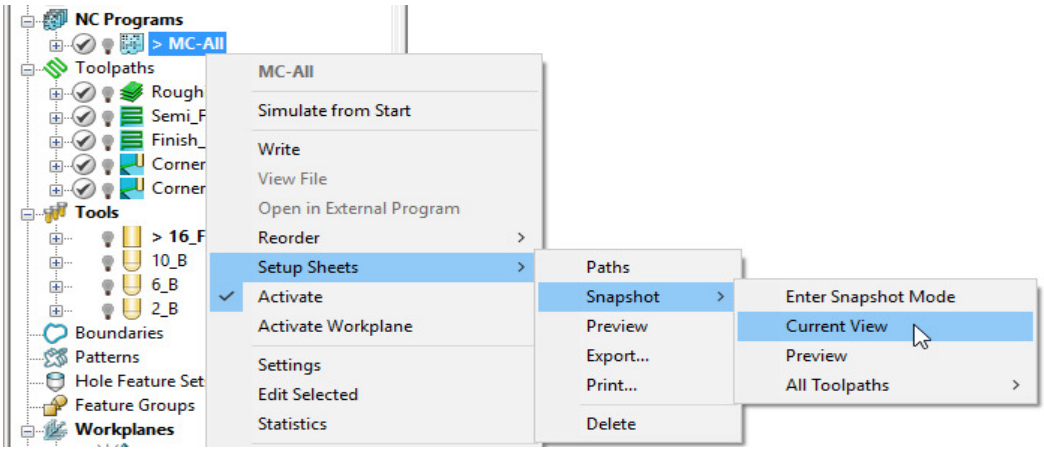
8 **Activate** one of the toolpaths to reinstate the original setup (Block, Workplane, Rapid Move Heights, etc).

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.




Copyright © Autodesk

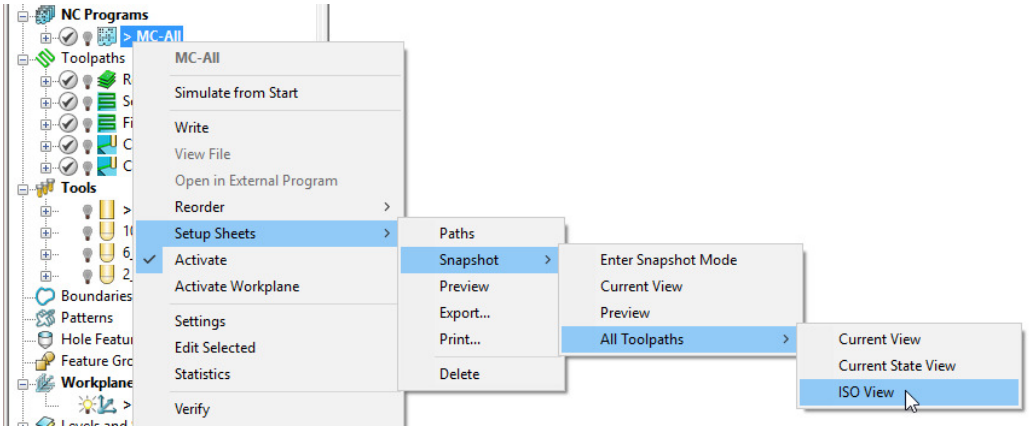
10 Right-click on the ***NC Program*** named **MC-All** and from the local menu select **Setup Sheets > Project Snapshot > Current View**.




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**.



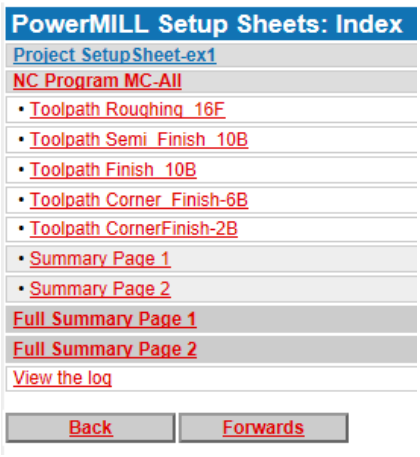

Images of all the **toolpaths** in the selected **NC program** have now been captured.

Each **toolpath** will have a separate page on the **Setup Sheet**.

13 Right-click on the **NC Program** - **MC-All** again and select **Setup Sheets - Preview**.

This action will create the **Setup Sheet**, **HTML** files.

The **PowerMill Setup Sheets: Index** is displayed on the left side of the screen.




14 In the above ***Index*** dialog left click on **Project Setup Sheet-ex1** to display the following sheet.

*Once a page has been selected the text will change from red to blue.*

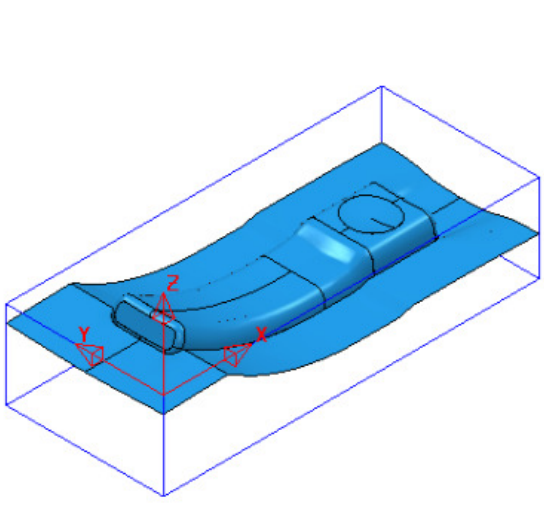
AUTODESK. POWERMILL®

**PowerMill Setup Sheet**

| | |
|---|---|
| Project | SetupSheet-ex1 |
| PowerMill Version | PowerMILL 21.000 CodeBase 1203007 |
| Page Generated: | 02:18 PM on 2016-11-09 |

**NC Program MC-All**

| | |
|---|---|
| Customer | Them |
| Part Name | It |
| Order Number | A-123789 |
| CAD Model | D:/users/dfb/PowerMILL2013/PowerMILL2013PRO/handset-2.dgk |
| Programmer | backhod |
| Option File | C:/dcam/config/ductpost/heid426.opt |
| Tool Datum Setting | Tip |
| Cutting Time For NC Program | 3:38:06 |
| Total Time For NC Program | 3:54:06 |




 Copyright © Autodesk

15 From the **Index** page select the **Toolpath** and **Summary** pages in turn and view the content.

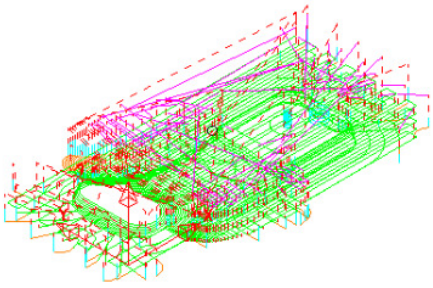
AUTODESK. POWERMILL®

## PowerMill Setup Sheet

| | |
|---|---|
| Project | SetupSheet-ex1 |
| NC Program | MC-All |
| PowerMill Version | PowerMILL 21.000 CodeBase 1203007 |
| Page Generated: | 02:18 PM on 2016-11-09 |
| Order Number | A-123789 |
| CAD Model | D:/users/dfb/PowerMILL2013/PowerMILL2013PRO/handset-2.dgk |

## Toolpath

| | | | |
|---|---|---|---|
| Name | Roughing_16F | | |
| Description | | | |
| Strategy | Raster Area Clearance | | |
| Tolerance | 0.1 | Cutting Feed Rate | 500 |
| Global Thickness | 1 | Plunge Feed Rate | 500 |
| Radial Thickness | 1 | Rapid Feed Rate | 3000 |
| Axial Thickness | | Spindle | 1500 |
| Stepover | 8 | Cutting Time | 0:30:57 |
| Stepdown | 5 | Total Time | 0:38:38 |
| Block | 215.028 × 90.087 × 50 | | |

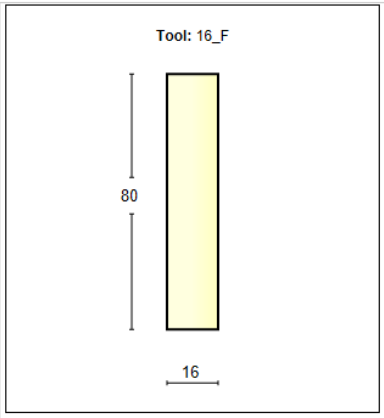


## Toolpath Limits

| | X | Y | Z |
|---|---|---|---|
| Start Point | 107.514 | 45.043 | 10 |
| Minimum | -11.258 | -10.544 | -31 |
| Maximum | 226.375 | 101.331 | 10 |

## Tool

| | |
|---|---|
| Name | 16_F |
| Number | 12 |
| Type | End Mill |
| Diameter | 16 |
| Tip radius | 0 |
| Length | 80 |




## Toolpath Notes

*Each HTML page can be printed out as individual items. Right-click on the page and from the local menu, select **Print Picture**.*

The 1st sheet of the **Summary Page** is as shown below.

AUTODESK. POWERMILL

**PowerMILL Setup Sheet**

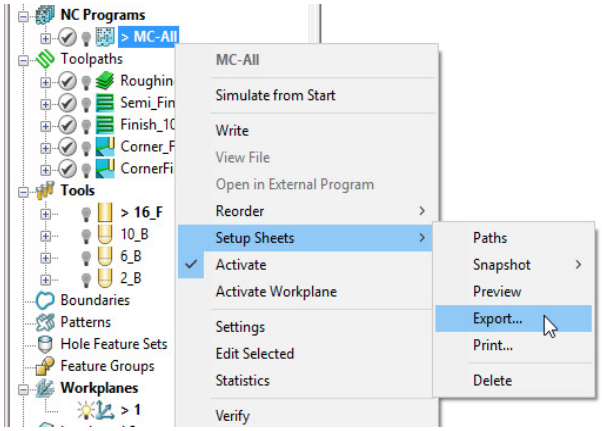
| | |
|---|---|
| Project | SetupSheet-ex1 |
| PowerMILL Version | PowerMILL 21.000 CodeBase 1203007 |
| Page Generated: | 02:18 PM on 2016-11-09 |

**Summary sheet 1 of 2**

| Toolpath / TAP File | NC Program | Strategy | Tool | |
|---|---|---|---|---|
| Roughing_16F<br>C:/Temp/NCPrograms/MC-All.tap | MC-All | Raster Area Clearance | Type | End Mill |
| | | | Diameter | 16 |
| | | | Tip radius | 0 |
| | | | Length | 80 |
| | | | Number | 12 |
| Semi_Finish_10B<br>C:/Temp/NCPrograms/MC-All.tap | MC-All | Raster | Type | Ball Nosed |
| | | | Diameter | 10 |
| | | | Tip radius | 5 |
| | | | Length | 50 |
| | | | Number | 14 |
| Finish_10B<br>C:/Temp/NCPrograms/MC-All.tap | MC-All | Raster | Type | Ball Nosed |
| | | | Diameter | 10 |
| | | | Tip radius | 5 |
| | | | Length | 50 |
| | | | Number | (14) |
| Corner_Finish-6B<br>C:/Temp/NCPrograms/MC-All.tap | MC-All | Along Corner | Type | Ball Nosed |
| | | | Diameter | 6 |
| | | | Tip radius | 3 |
| | | | 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.*


Copyright © Autodesk

# Customising Setup Sheets

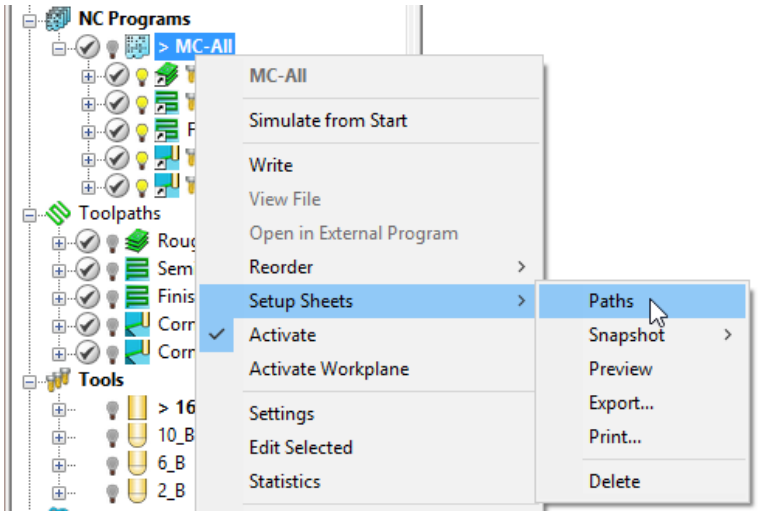
By editing the **HTML** templates it is possible to customise the output of the **Setup Sheets**.

The use of **HTML** editing software is the recommended method to implement changes to the **Setup Sheet** output. There are free downloads available from the web. **Microsoft Word** can also be used but this can give unpredictable results.

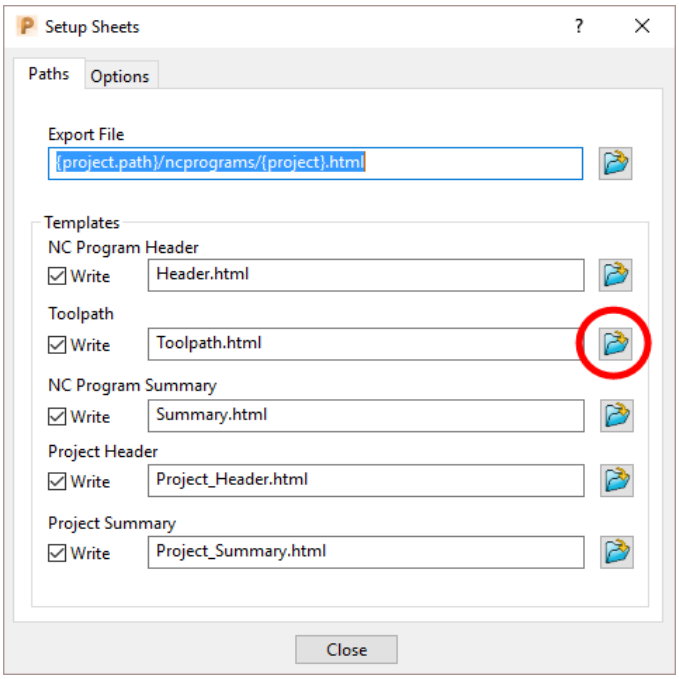
*Always create a copy of the template document so that the original is still available if things go wrong.*

In the following example an alternative template will be substituted into the **Setup Sheet Toolpath Template** search paths.

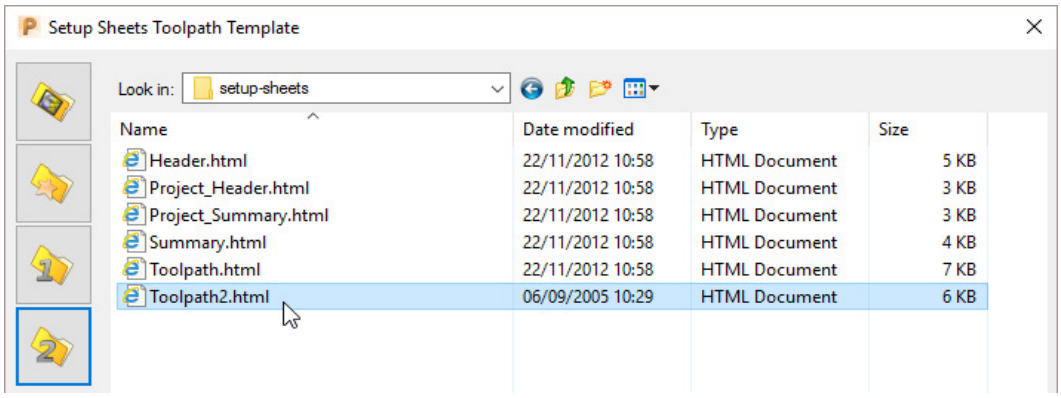
1 Right-click on the **NC Program** named **MC-All** and select the option **Setup Sheets** > **Paths**.




2 On the **Paths** tab select the folder icon in the **Toolpath** > **Write** field.



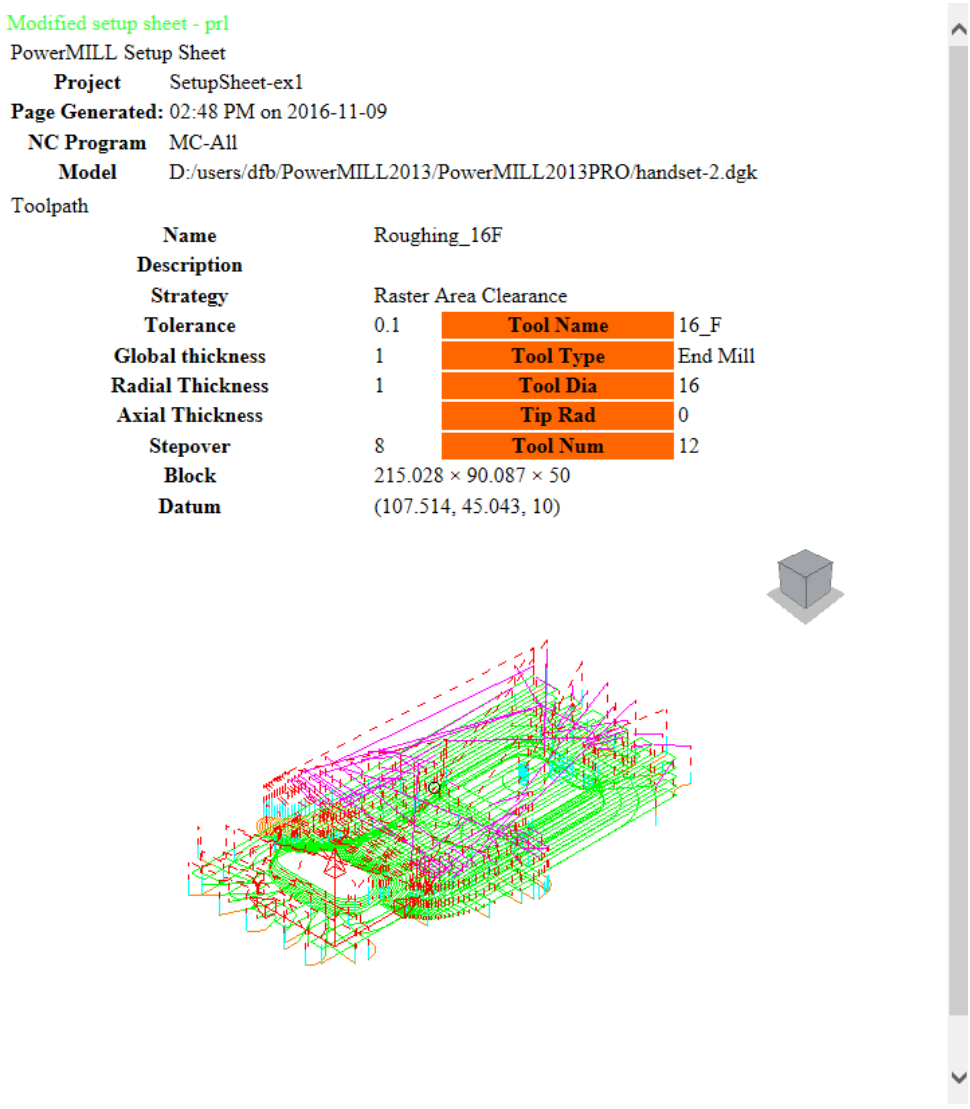
3 Browse to **.....\PowerMill_Data\setup-sheets** and **Open** the **Toolpath2.html** file (customised template).




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)





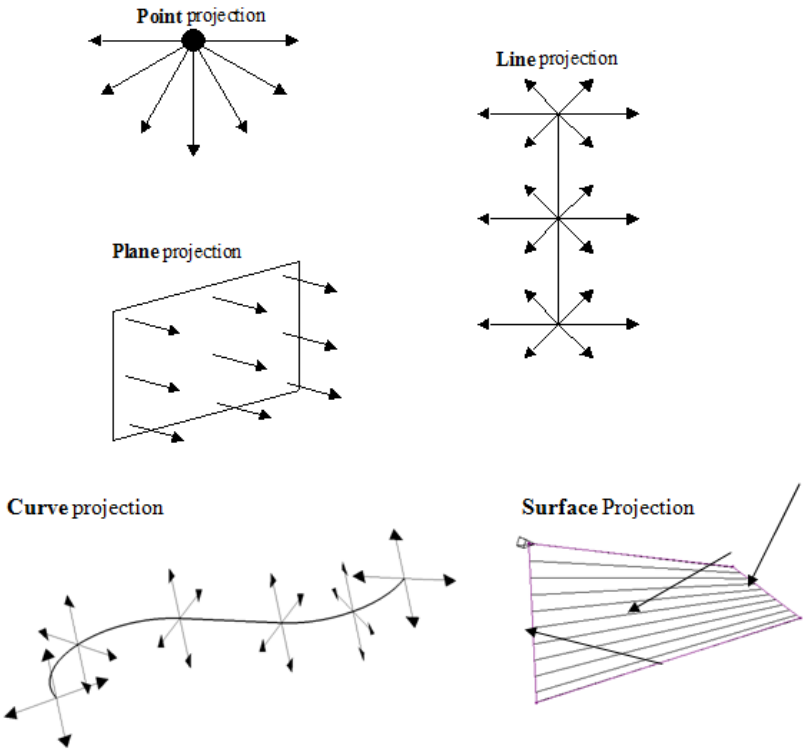
Copyright © Autodesk

# 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).

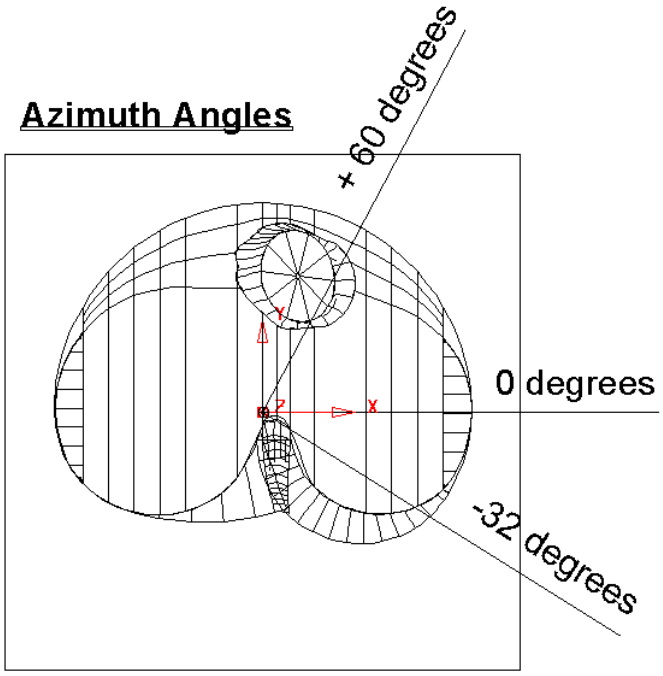
- A **Point** strategy is similar to light radiating from a single light bulb.
- A **Plane** strategy is similar to light radiating from a rectangular floodlight.
- A **Line** strategy is similar to light radiating from a fluorescent tube.

- 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.

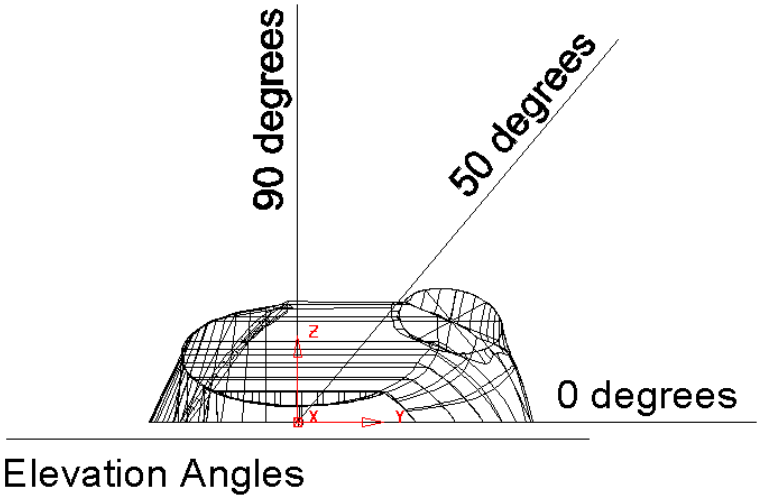
In the **Projection Finishing** forms, angular definitions are input using **Azimuth** and **Elevation**. By using a combination of the two any complex 3D angle can be defined.

- **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




This is shown on the chamber model, with a view looking down the X-axis.

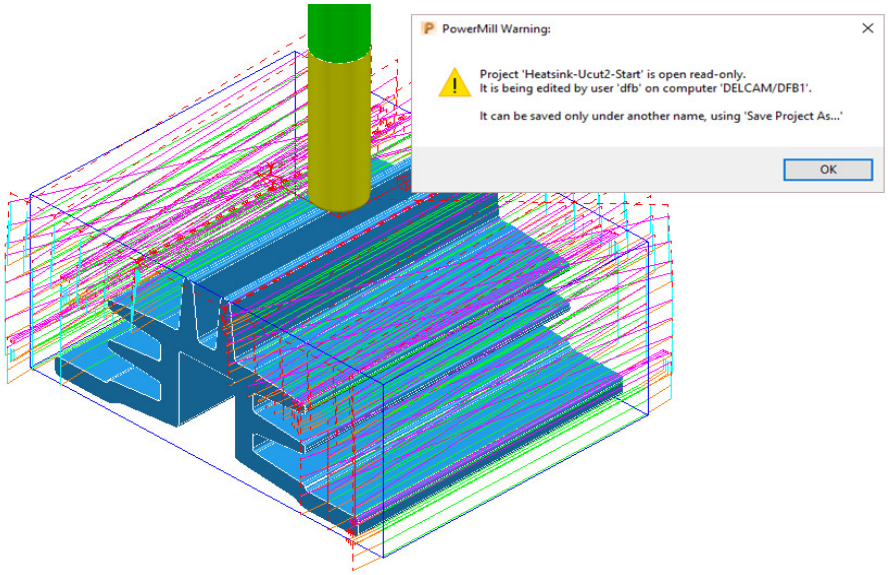

Copyright © Autodesk

# 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**:

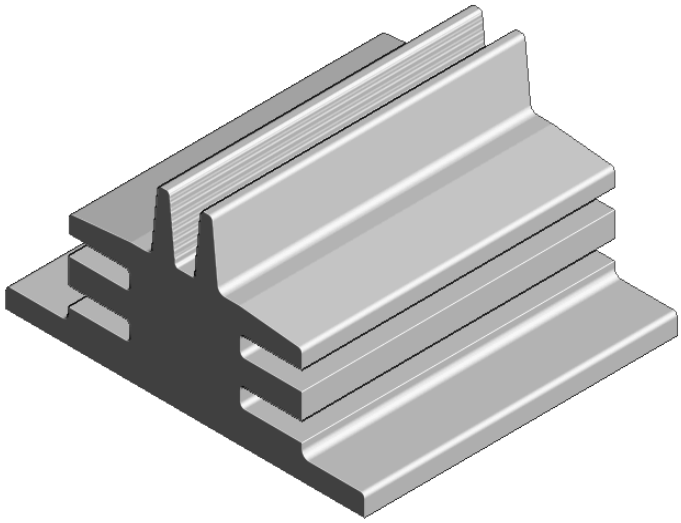
   **.....\PowerMill_Data\Projects\Heatsink-Ucut2-Start**




3. **Save** the **Project As**:

   **.....\COURSEWORK\PowerMill_Projects\ Heatsink-Ucut2-EX1**

4. Activate the toolpath **D25TR3-RGH1** to reinstate the original setup.
5. Open the **Block** form and with ***Defined by*** **- Box** and Type **- Model** set select **Calculate**.
6. **Simulate** the **NCProgram** (all of the existing toolpaths) in **ViewMILL**.



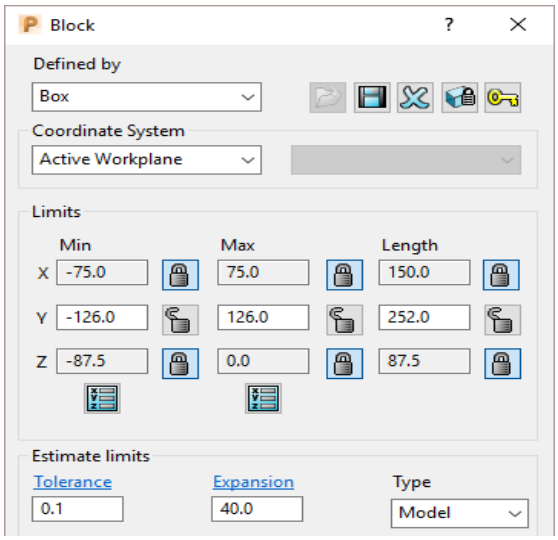
7 Return the **ViewMILL** simulation to the **No-Image** view .

8 Disconnect **ViewMILL** from the toolpath simulation .

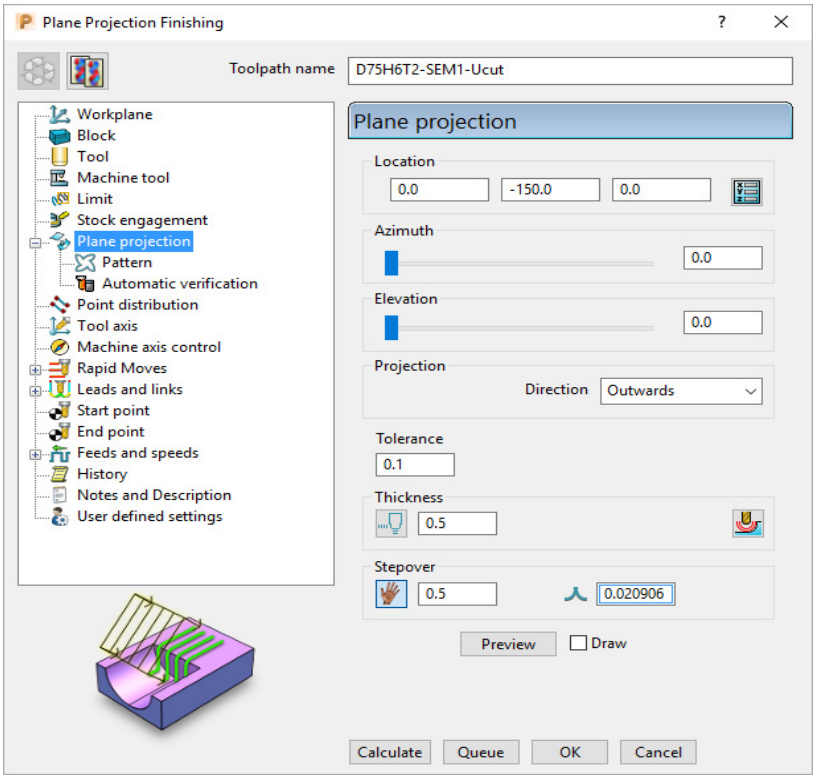
9 **Activate** the **D75-H6-T2** Disc Cutter.

10 In the **Block** dialog ***lock*** all 4 of the **X** and **Z** values.

11 **Calculate** the **Y** values of the **Block** with an **Expansion** value of **40**.

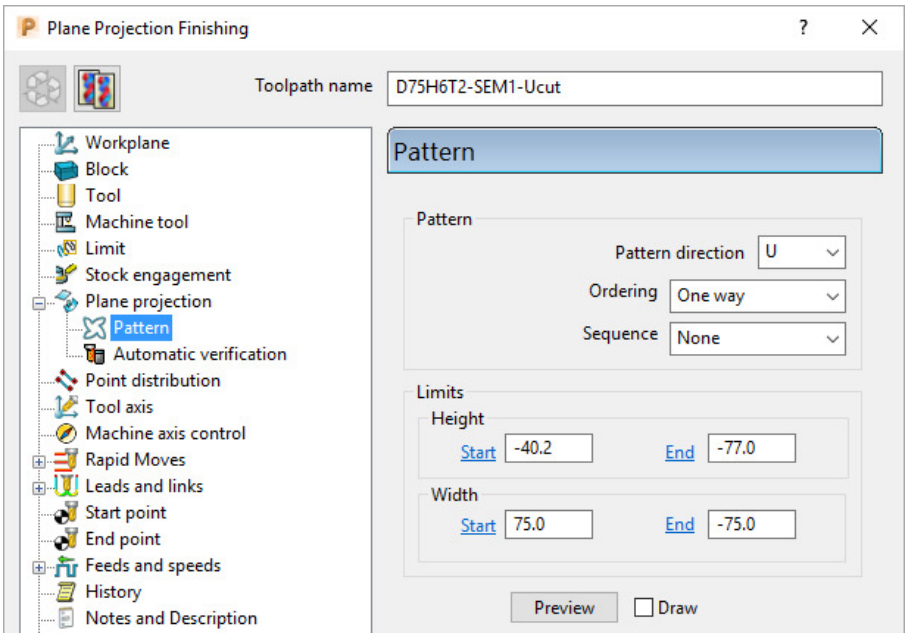



12 In the **Strategy Selector** - **Finishing** dialog, select **Projection Plane Finishing** and fill in the pages of the dialog exactly as shown below.



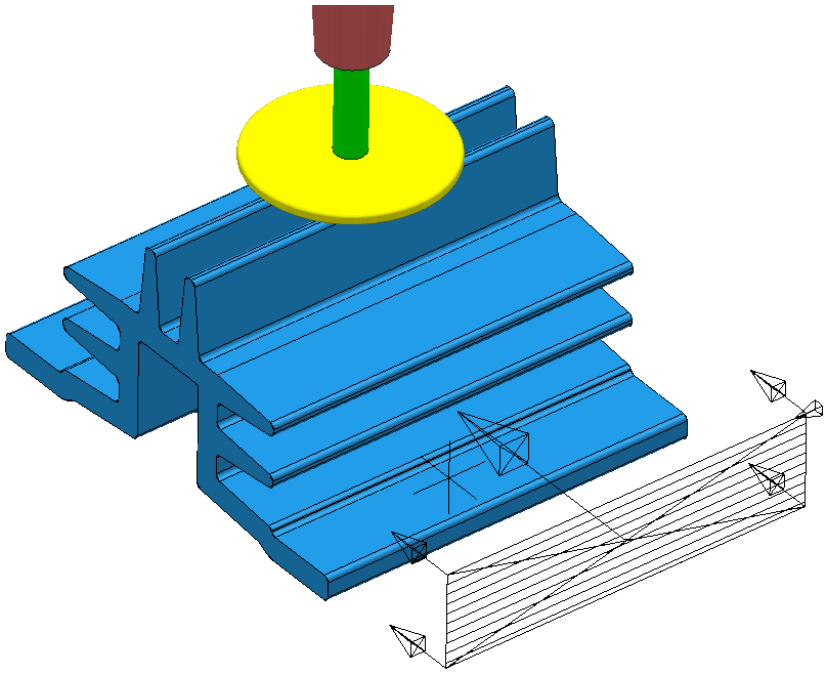

The **Location** is the local datum for the projection strategy. All other coordinates entered in the dialog are relative to this point.


Copyright © Autodesk




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.

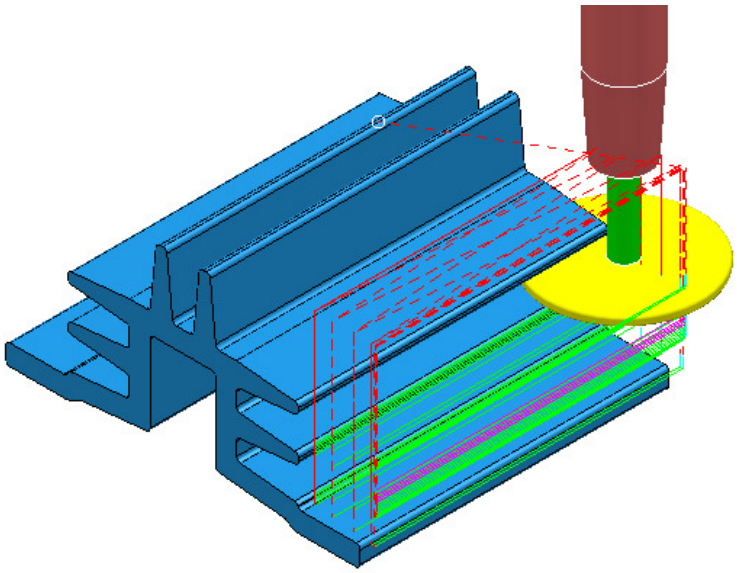
13 Select **Preview** to view the **Plane Projection** pattern.



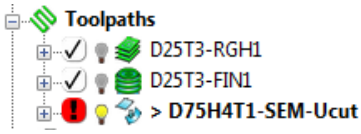
The **Projection Plane > Preview** is displayed as shown and the user can dynamically alter the view to visually check for potential adjustments.

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.

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

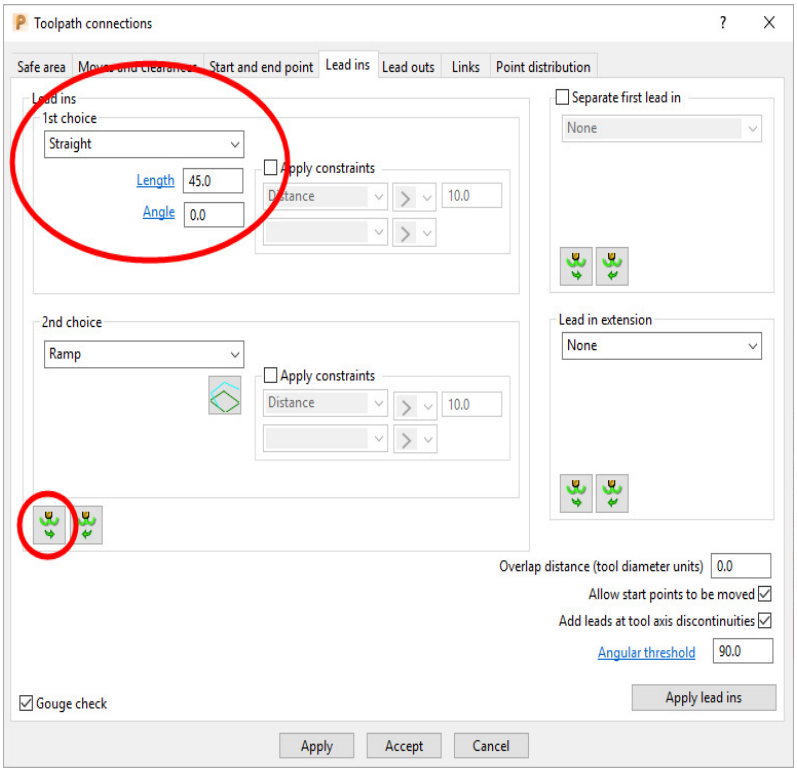


Observe that the tool is **gouging** through the fins when it retracts at the end of each pass. This is confirmed by the ***red*** toolpath, icon displaying an exclamation mark in the explorer.




In this case suitable **Leads and Links** will be applied to easily rectify the problem.

1 Open the **Leads and Links** dialog and select the **Lead in** tab.





Copyright © Autodesk

2 Input the following in the **Lead in** page:

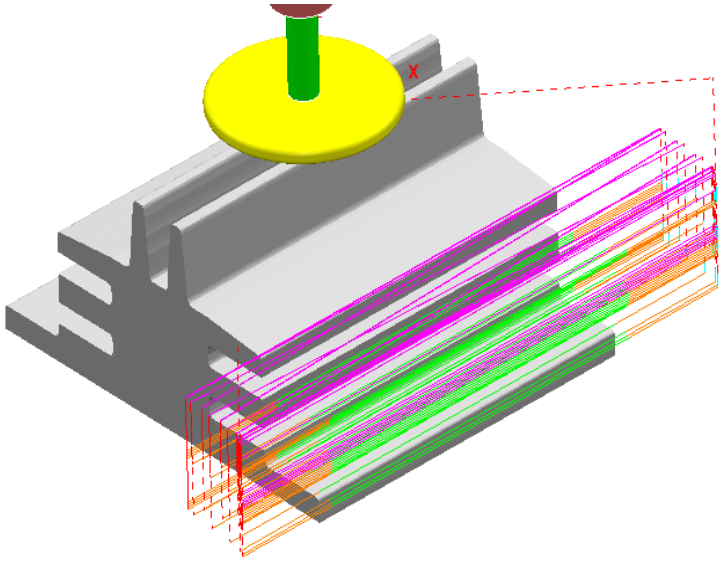
**1st Choice** - **Straight**

**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***.

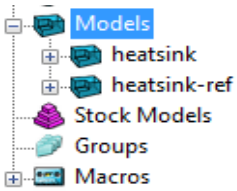
5 **Save**, but do not close, the **Project** as it is to be continued in the next section.

# 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.

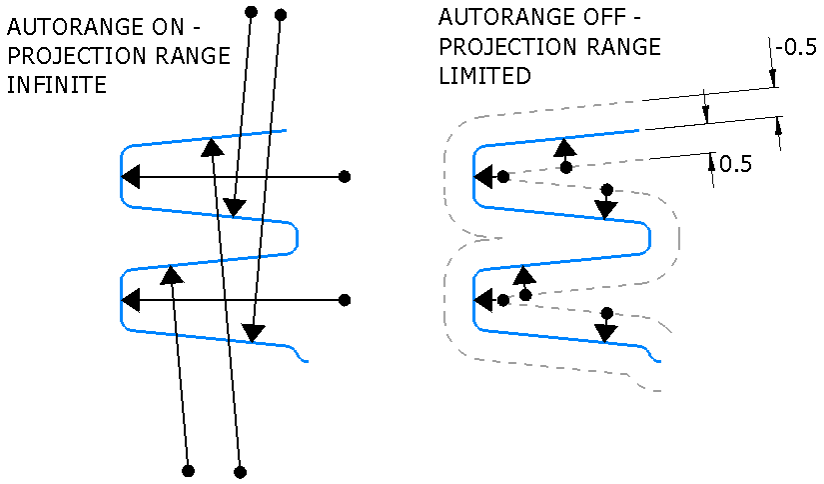
2 From the **main** pull down menus select **View** - **Toolbar** - **Command** to open the **command input** window below the graphics area and type in the following 3 lines:-

**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.

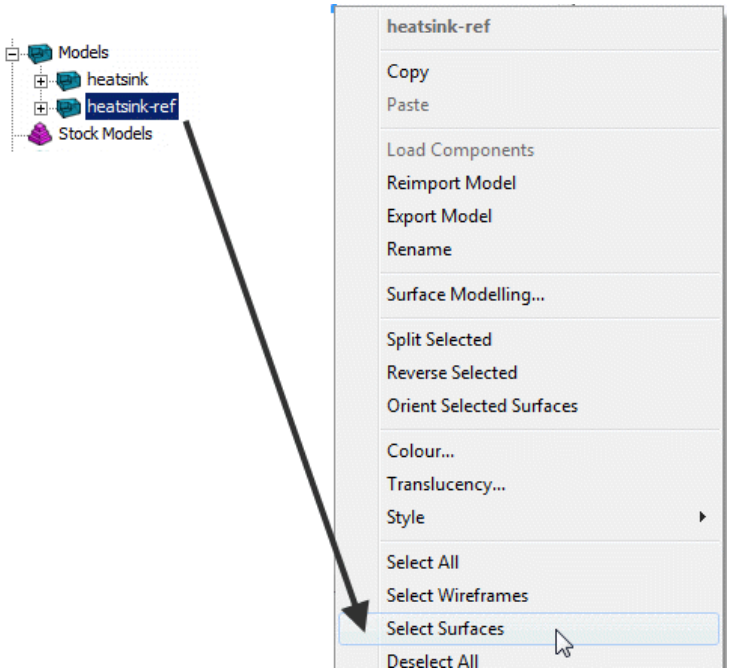



*The default setting is restored by typing:-*

**EDIT SURFPROJ AUTORANGE ON**

3 From the **Toolpath Strategies** dialog, select **Projection Surface Finishing**.

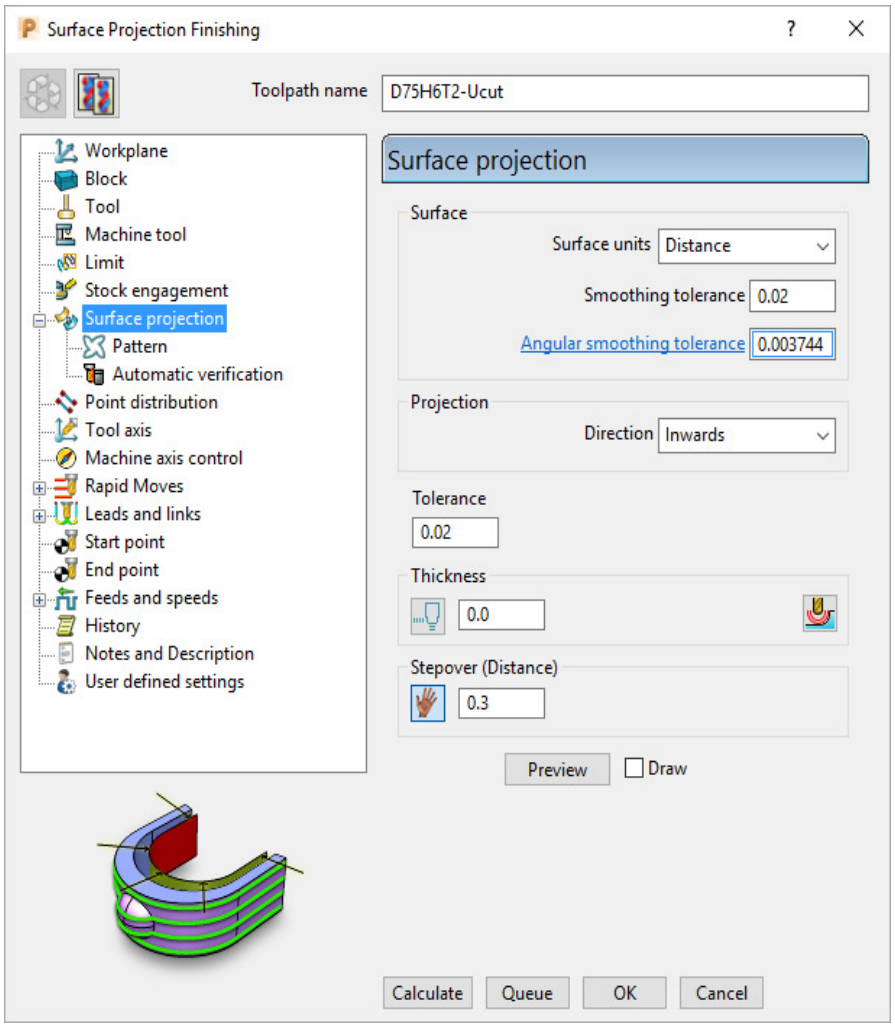
4 In the **PowerMill *explorer***, right-click the model, **Heatsink-ref** and from the local pull down menu click **Select Surfaces**.

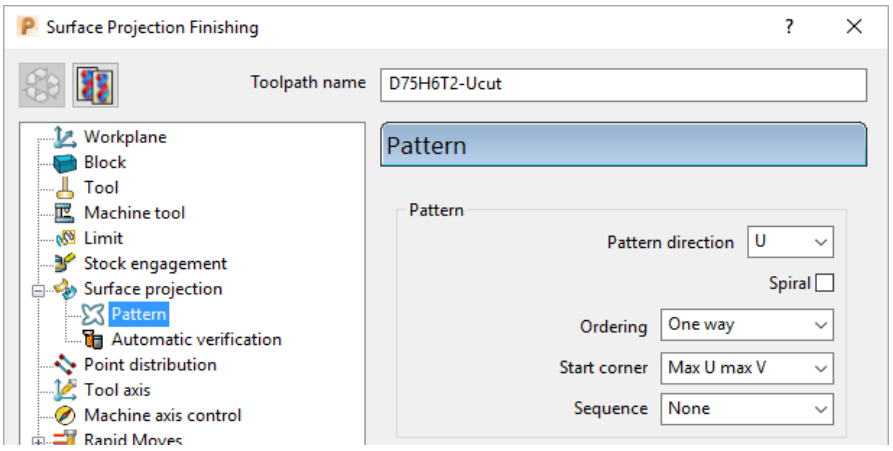




Copyright © Autodesk

5 In the **Surface Projection Finishing** dialog input the settings and data exactly as shown below.

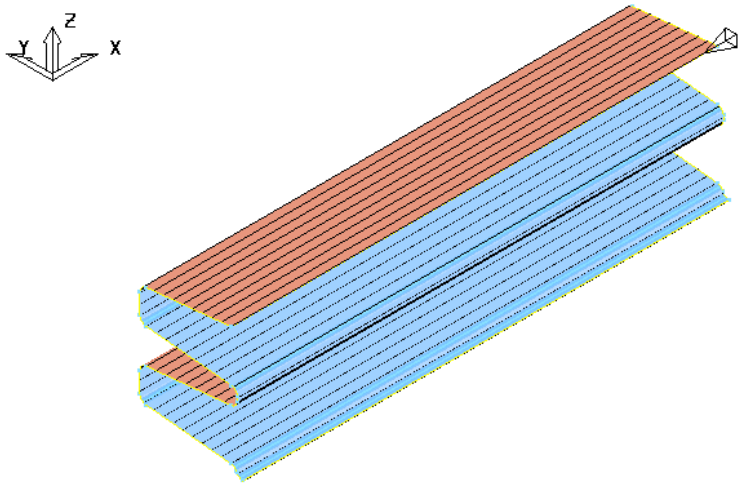



6 Select **Ordering** as **One way**.

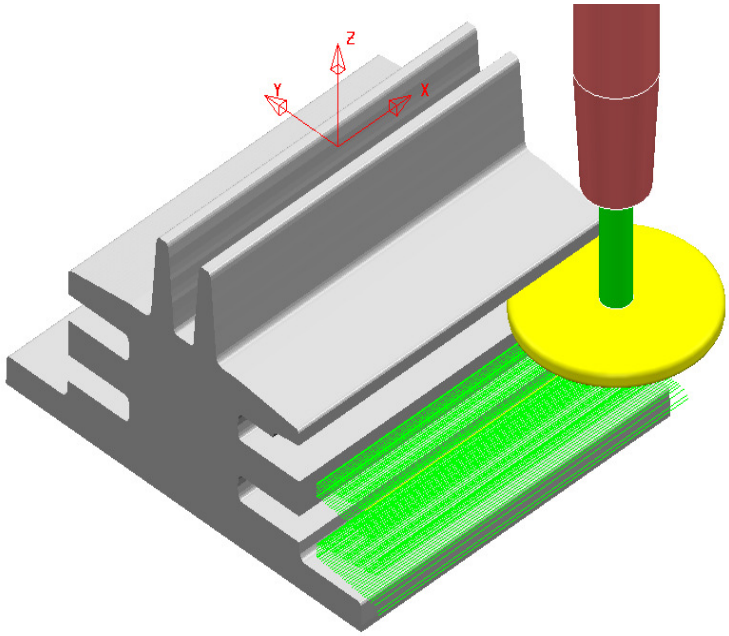
7 Set the **Start Corner** as: **Max U max V**

8 Select **Preview**.




The **Preview** pattern for the strategy is displayed on the ***reference surface*** (for clarity, a stepover value of 3 was used in this illustration).

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




*The unidirectional toolpath is shown here without the defined **Leads and Links** displayed.*

*A suitable **Lead In** and **Lead Out** must be applied to prevent the tool from **gouging** through the overhang.*

The application of **Surface Projection** Finishing has produced a strategy with a consistent ***stepover*** across all of the undercut areas.

10 Select **File** > **Save** to update the stored **Project**.

 Copyright © Autodesk

# Line Projection Example

1. Select **File** > **Delete All** and **Tools** > **Reset Forms**.
2. Select **File** > **Open Project**:

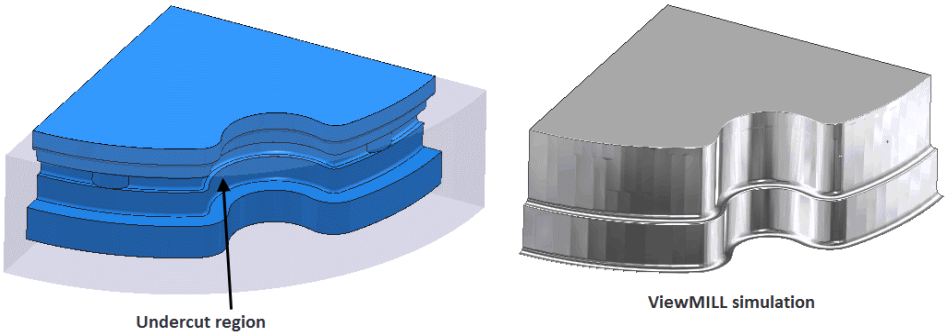
   **.....\PowerMill_Data\Projects\ucut_proj**

3. Select **File** > **Save Project As**:

   **.....\COURSEWORK\PowerMill-Projects\Line-Projection**

4. Select an **ISO 3** view.

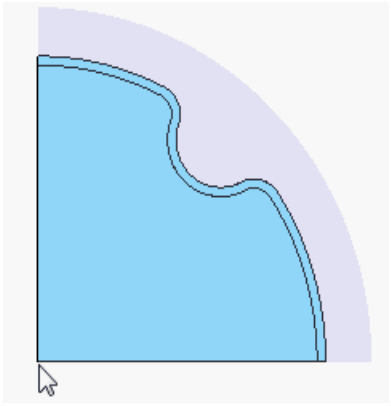
The imported **Project** (training_ucut_proj) contains a **Model**, **Tooling**, **Roughing,** and **Semi-Finishing** strategies with **1** and **0.3 thickness** values respectively. The undercut regions have not yet been machined.



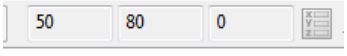

5. **Activate** the **toolpath** (**Rough**) to reinstate the settings and then simulate both existing toolpaths in **ViewMILL**.

**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. Disconnect the **ViewMILL** session and select a **View from Top (Z)**.
7. Place the cursor over the lower left, corner point without clicking and read the **X** & **Y** values from the **information** toolbar located below the graphics area.



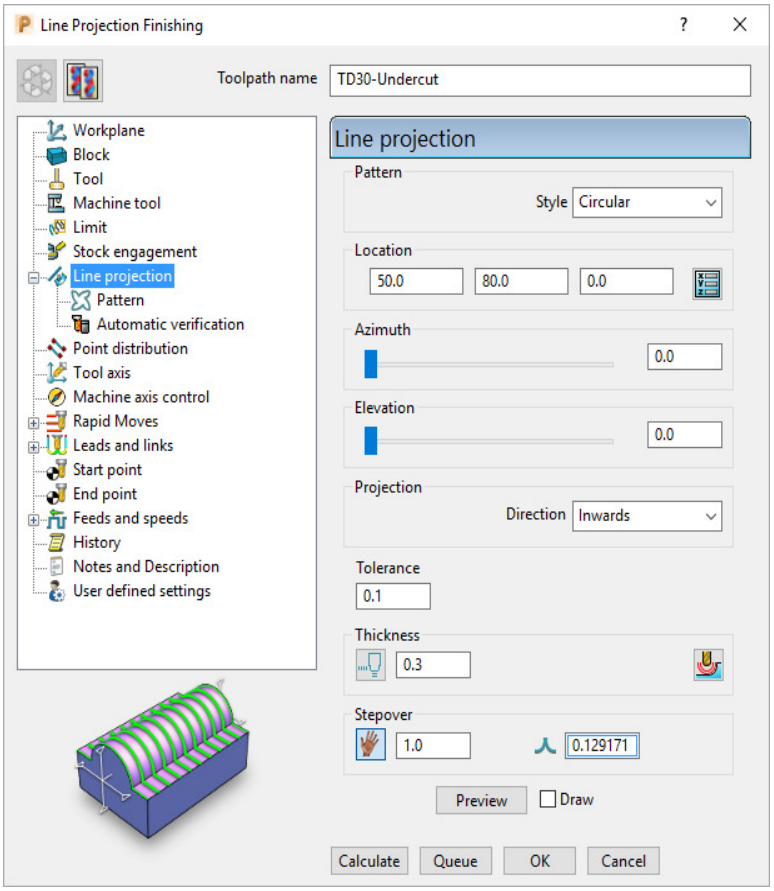
*The **Intelligent Cursor** will accurately lock onto the corner providing accurate coordinate values.*



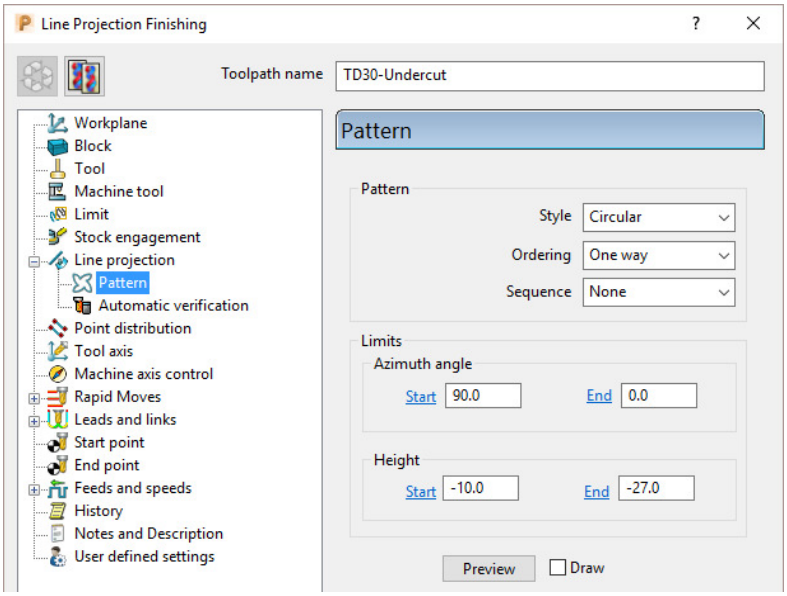
8 **Activate** the **TD30** cutting tool.

9 From the **Toolpath** strategies dialog, select **Projection Line Finishing**.

10 Fill in the pages of the strategy dialog exactly as shown below.



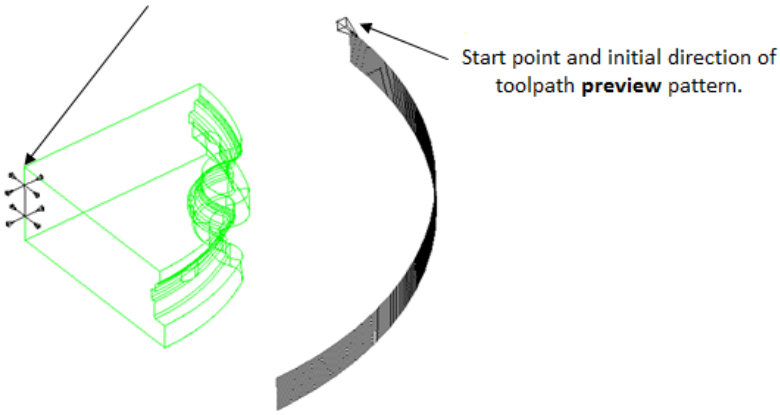

The values for the **Start** and **End** heights were obtained by dynamically moving the cursor over the model to obtain the **Z** values in the ***cursor coordinate boxes*** at the bottom right of the graphics area.



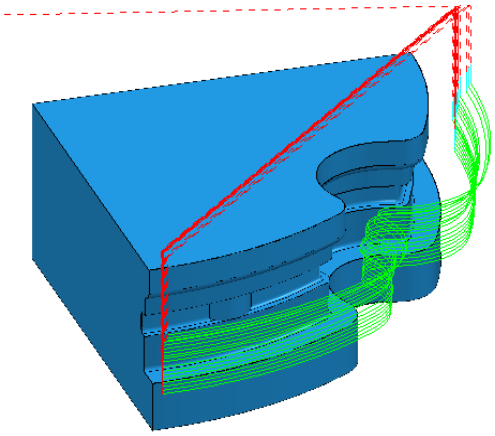

11 Select **Preview**.


Copyright © Autodesk

The **Preview** shows the location of the ***projection line*** with **arrows** showing an inward projection direction from the **pattern**.

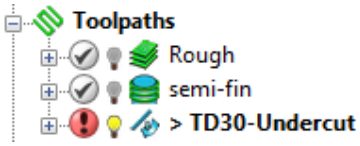



12 **Calculate** the toolpath and then **Close** the dialog.

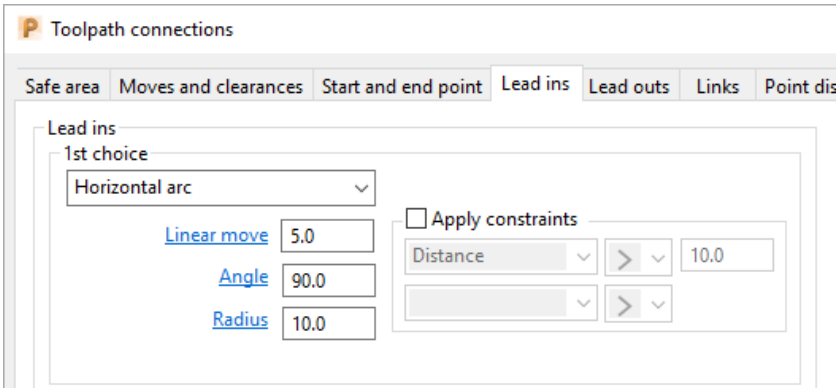


The tool is currently **gouging** through the overhang at the end of each pass.

This is indicated by the red colour of the toolpath icon as displayed in the **PowerMill *explorer***.




13 From the **main** toolbar select **Leads and Links** to access the **Lead in** dialog.



14 Input the following information:

***1st Choice*** > **Horizontal Arc**

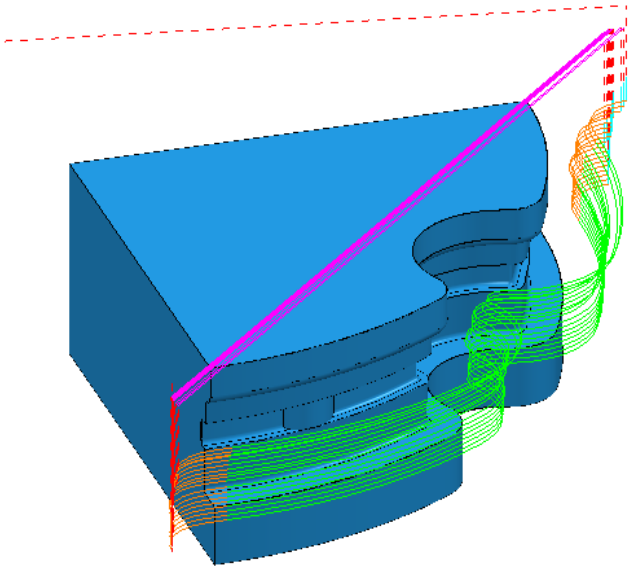
***Distance*** > **5**

***Angle*** > **90**

***Radius*** > **10**

15 Select the **Lead out the same as lead in** tab.

16 Click **Apply** and then **Accept** to close the dialog.



*The **extension** value in **Leads and Links** must be greater than the tool radius to prevent gouging during the link moves.*

*The **red** warning icon in the **PowerMill explorer** will now have changed to **green**.*

17 Select **File** > **Save** to update the stored **Project**.


Copyright © Autodesk

# Point Projection Example

1 Select **File** > **Delete All** and **Tools** > **Reset Forms**.

2 Select **File** > **Import Model**:

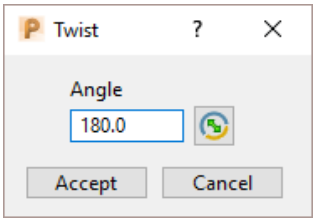
**.....\PowerMill_Data\Models\radknob.dgk**

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:

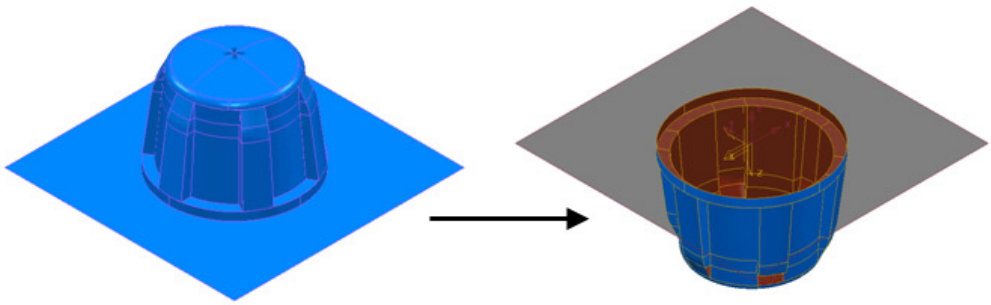
5 From the toolbar select **Twist about Y** to open the **Twist** dialog.




6 In **Angle**, enter **180** and click **Accept**.

7 Click the **Green** tick to accept the changes and close the toolbar.

8 Select an **ISO 1** view.



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.

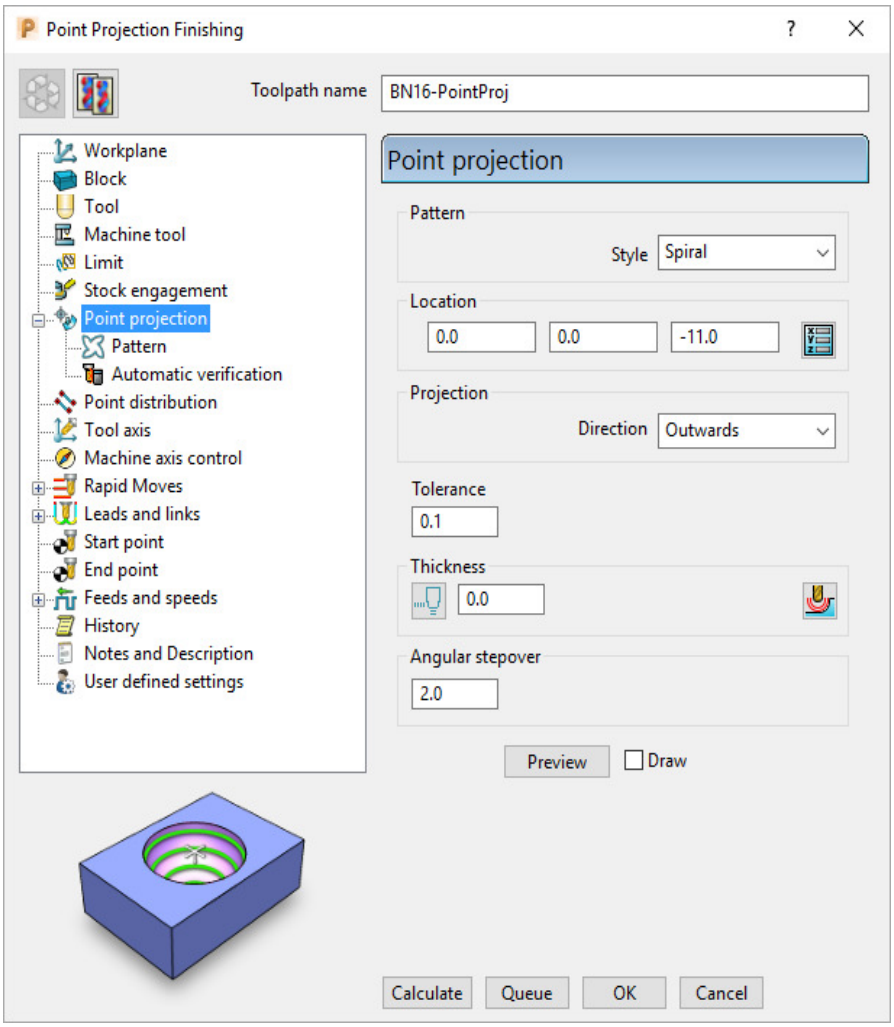
9 Create a Dia **16** ball nosed cutter named **BN16**.

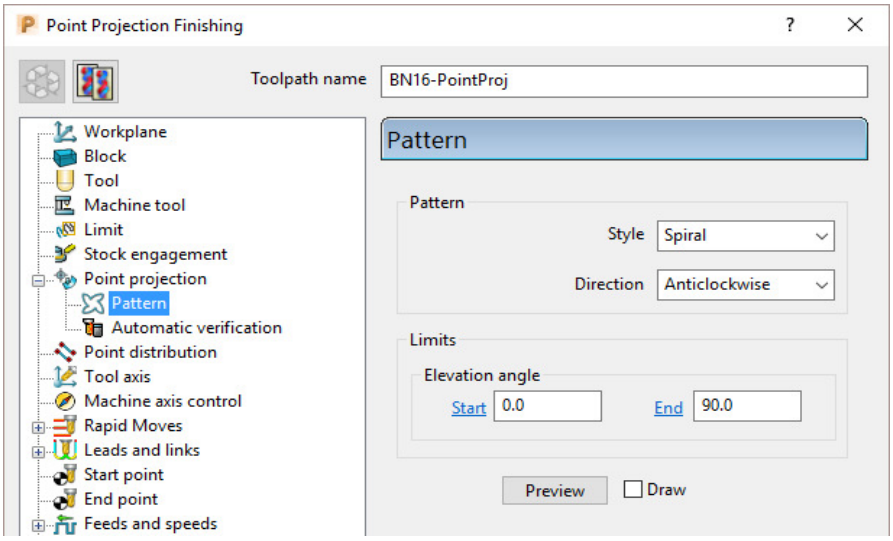
10 **Calculate** a **Block** using **Defined by** - **Box** and **Type** - **Model**.

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

12 From the **Strategy Selector** dialog, select **Projection Point Finishing**.

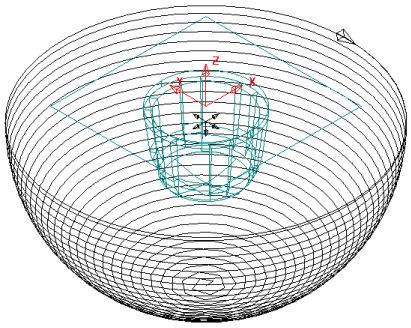
Copyright © Autodesk



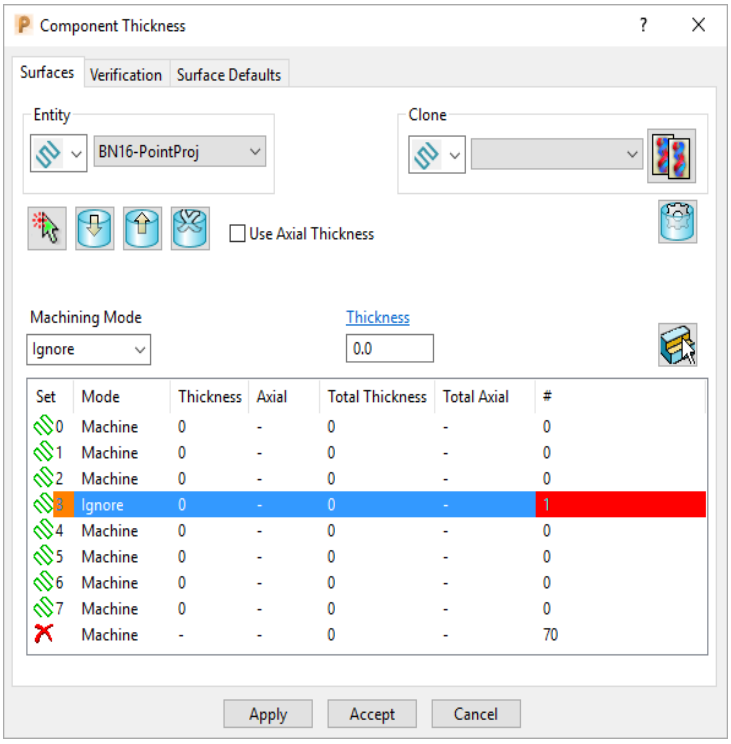




13 Fill in the page of the dialog exactly as shown above and select **Preview** to observe the **Point Projection** setup.

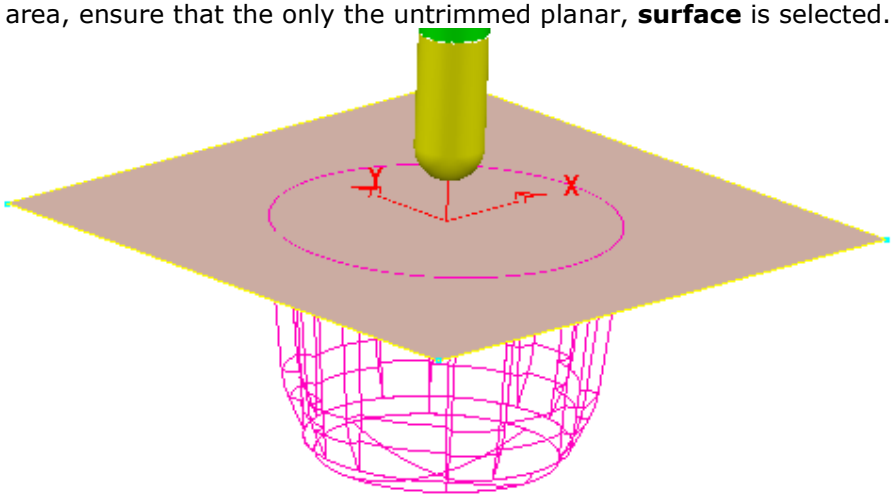

Copyright © Autodesk

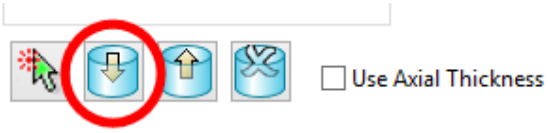


14 In the **main** page of the **Point Projection Finishing** dialog select the **Component Thickness** icon and in the dialog select the **Surfaces** tab.




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.



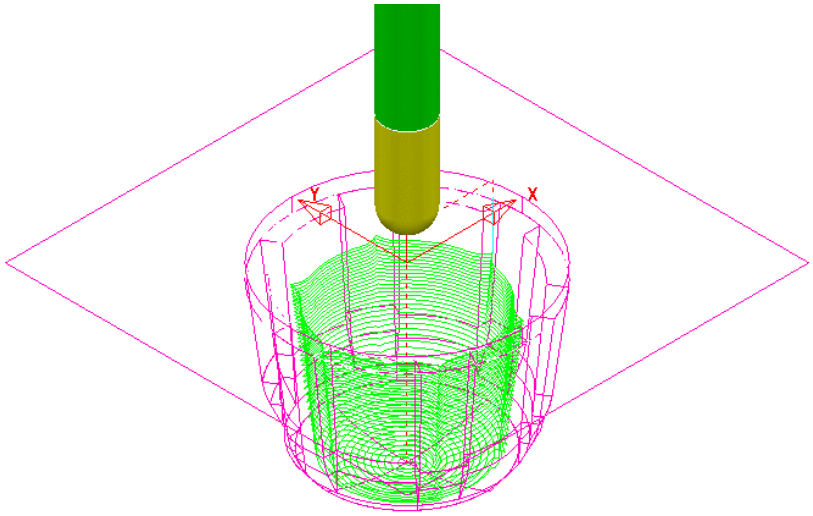



15 Click the **Acquire Components** icon to add the selected, planar surface to the row.

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**.

 Copyright © Autodesk

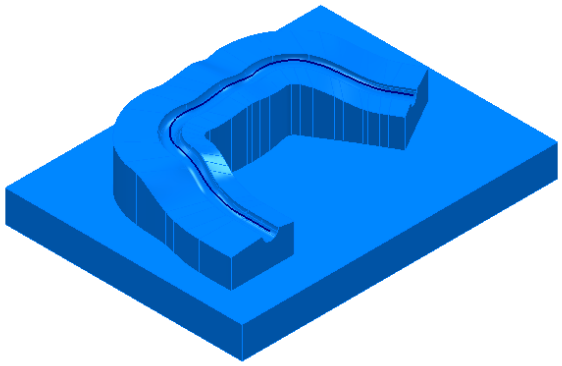
# Curve Projection Example

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**

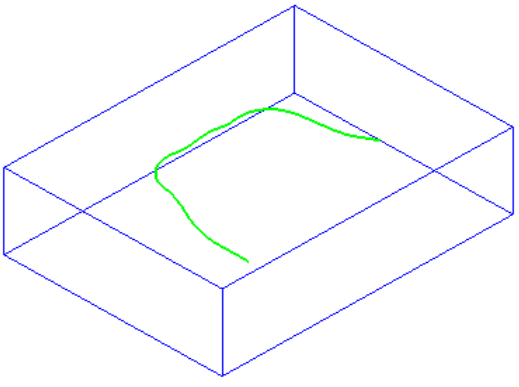
4 Calculate the **Block** using **Defined by - Box** and **Type - Model**.

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**.

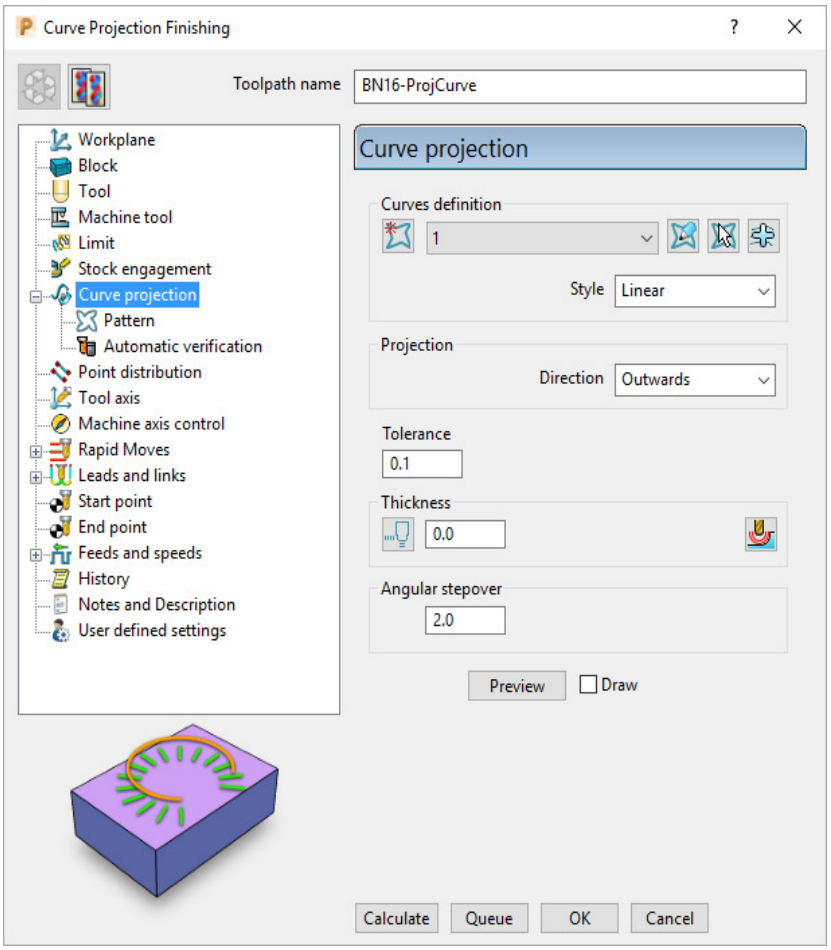
8 Select the ***wireframe curve*** (coloured green) running centrally along the slot on the **Model** (Note; The **Wireframe** view must be active for the curve to be visible).

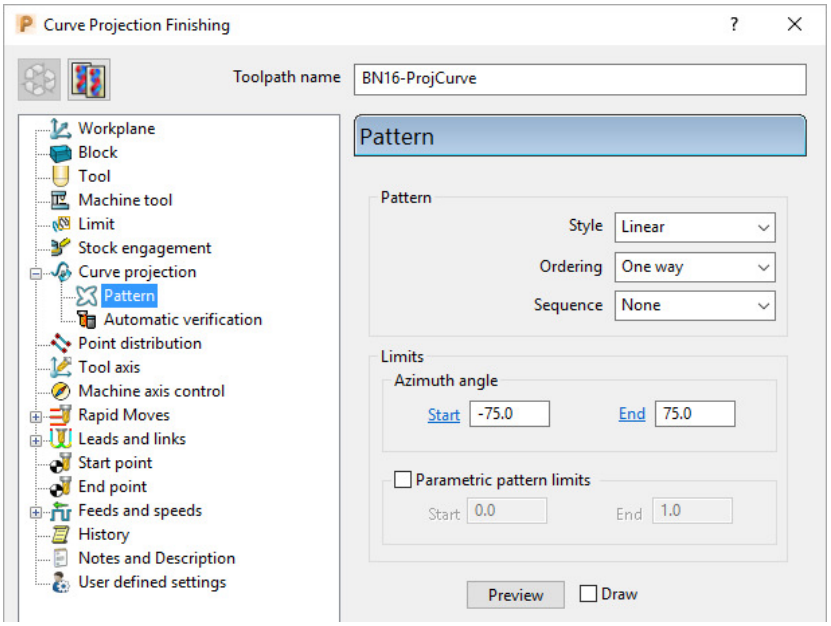


9 In the **PowerMill *explorer***, right-click on the new **Pattern** (**1**) and in the local menu select **Insert** > **Model**.

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.

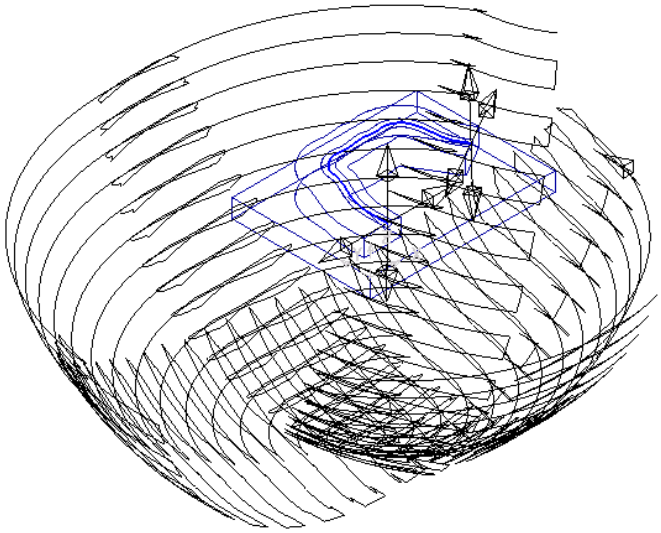


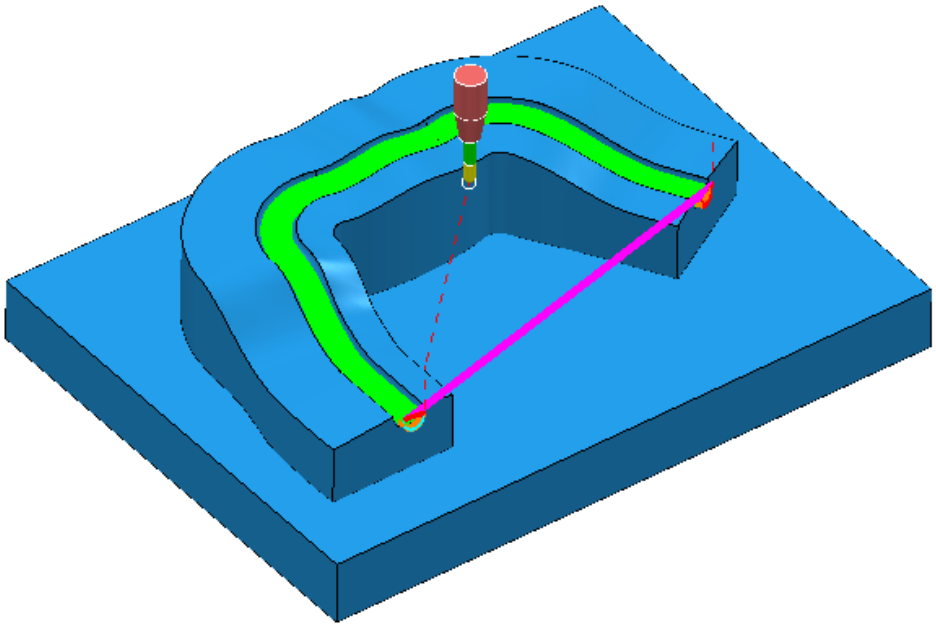


Copyright © Autodesk

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.



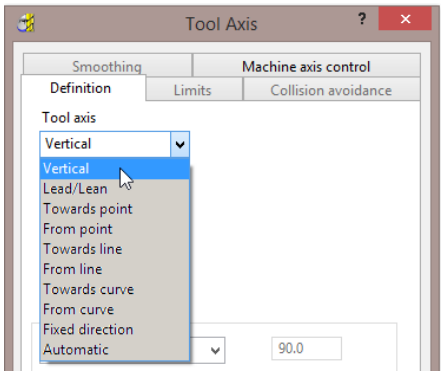
*The **Curve** used as the basis for the strategy will in the majority of cases be created in a CAD package as part of the **Model**. In these cases the imported **wireframe** curve must then be inserted into a **Pattern**.*

13 Select **File** > **Save** to update the stored **Project**.

# Swarf Finishing

Swarf Finishing is specifically designed to machine across one or more surfaces where a linear transition exists along the tool alignment direction. It will not work where the surface transition is concave or convex.

For 3 Axis applications the tool axis must be set to **Vertical**.

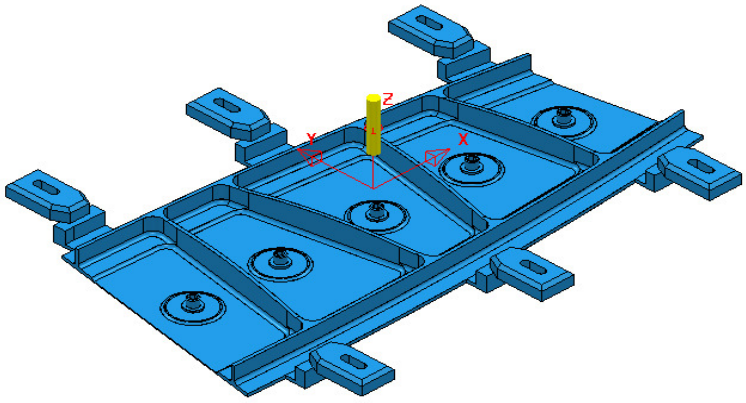



The strategy can be set to create either single (default) or multiple passes, exactly following the upper edge, lower edge, or both (merging).

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**



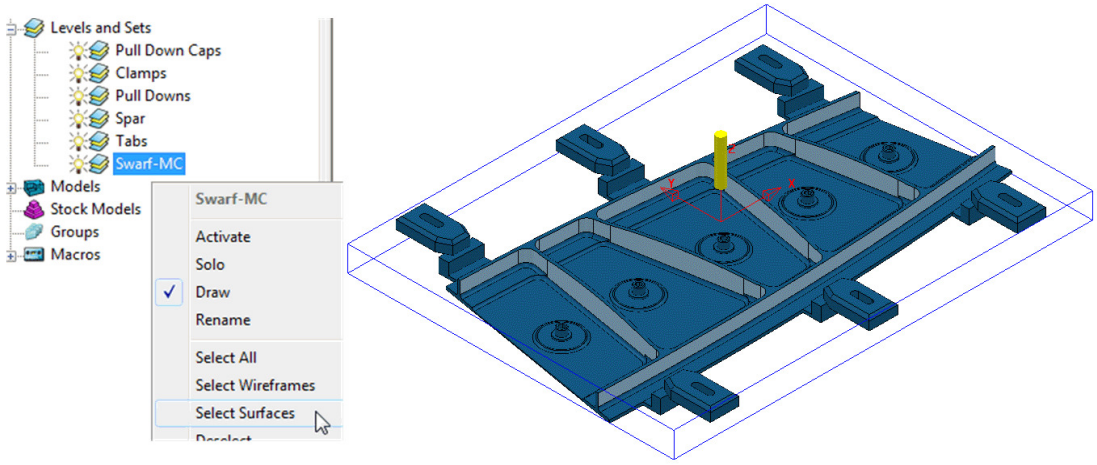

3. From **File** select **Save Project As**:

   **.....\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.


Copyright © Autodesk

8 In the **PowerMill *explorer***, click the [**+**] adjacent to **Levels and Sets** to access all of the existing **Levels**.

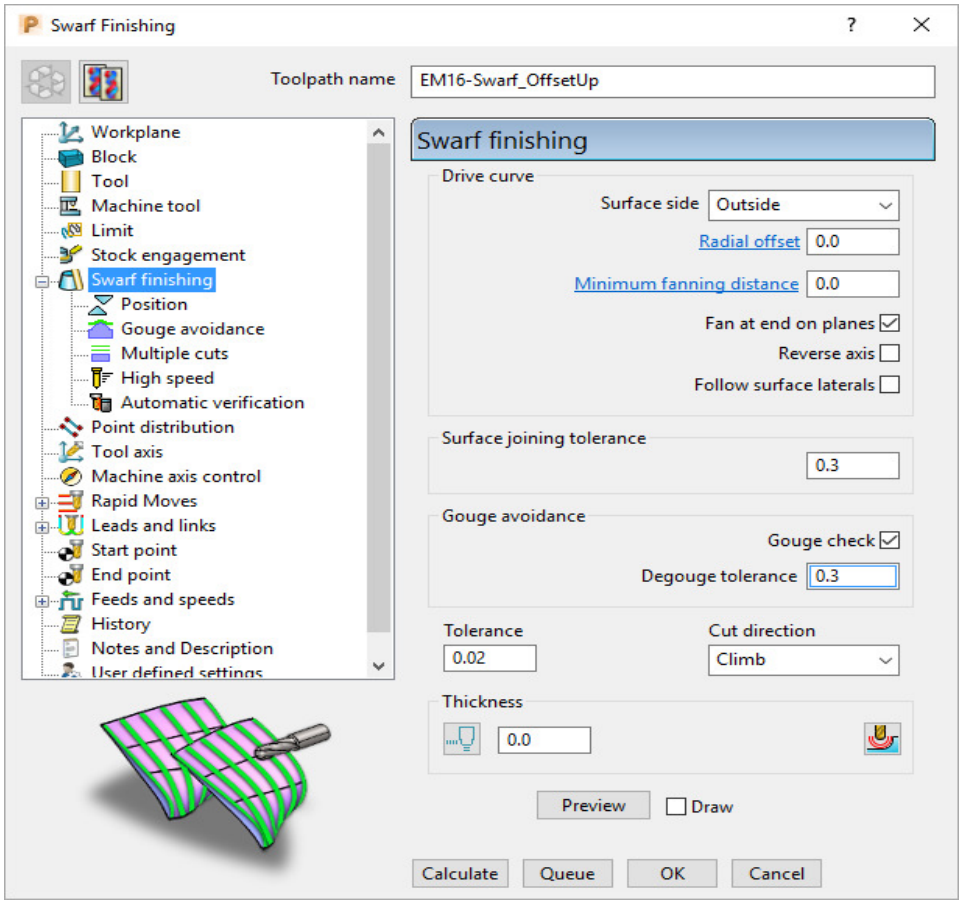
9 Right-click on the **Level *Swarf-MC*** and click **Select Surfaces** from the local menu.

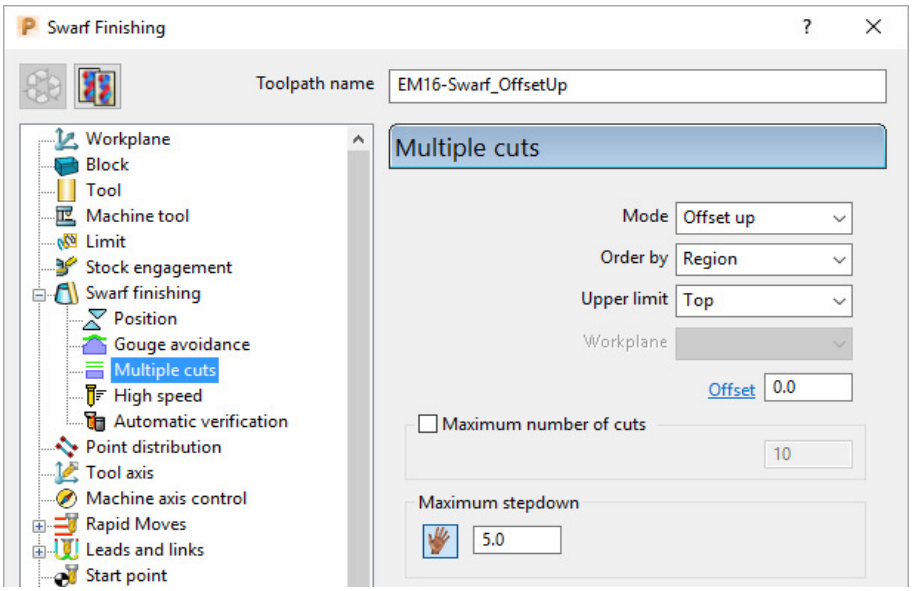



All **Surfaces** on **Level *Swarf-MC*** will be selected (light blue) to be included in the **Swarf** strategy.

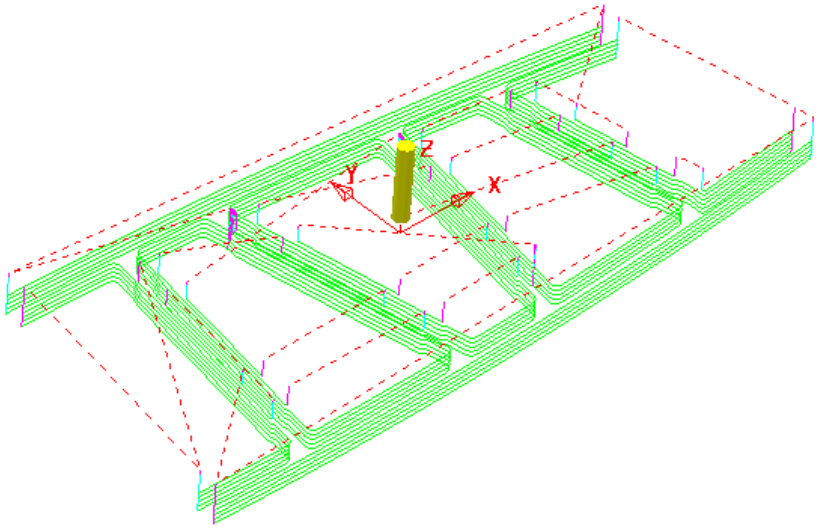
10 In the **Strategy Selector**, select the **Finishing** tab, and then select the **Swarf Finishing** strategy.

11 Enter data exactly as shown in the following 2 pages of the dialog.






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**.*

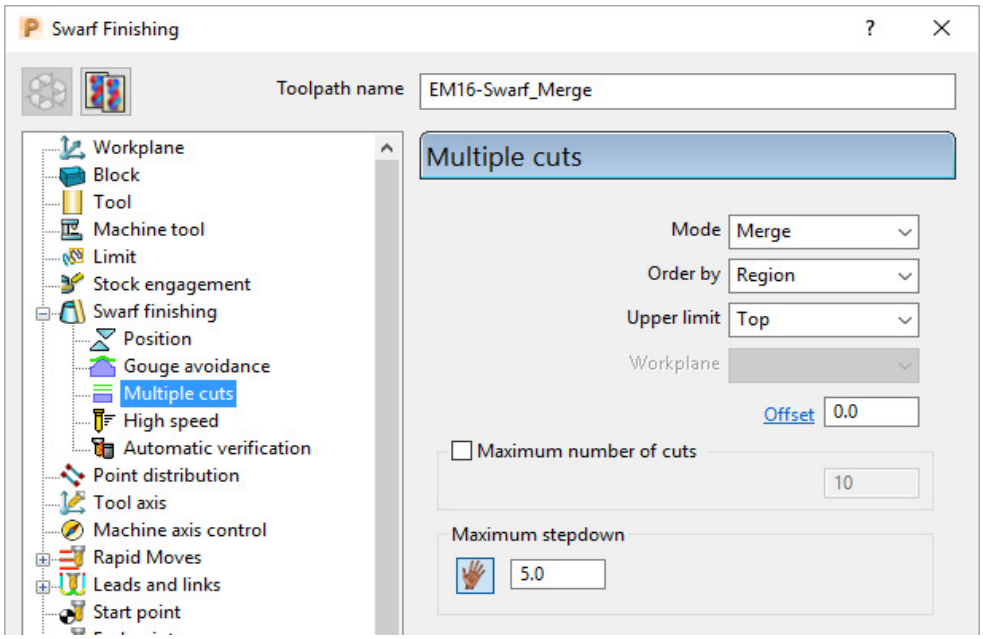
13 In the **PowerMill *explorer***, right-click the **toolpath *EM16-Swarf_OffsetUp*** and select **Settings**.

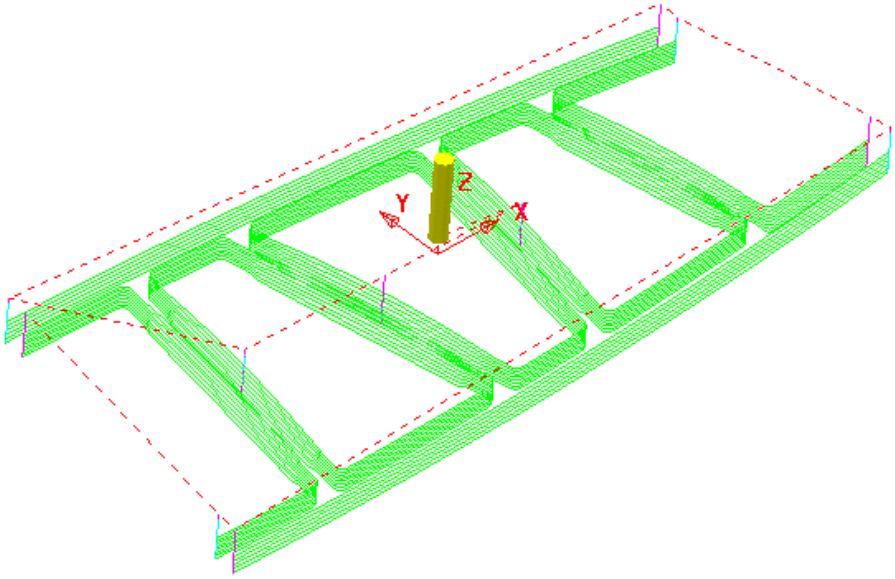
14 Select the ***Copy Toolpath*** icon from the dialog .

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.

 Copyright © Autodesk



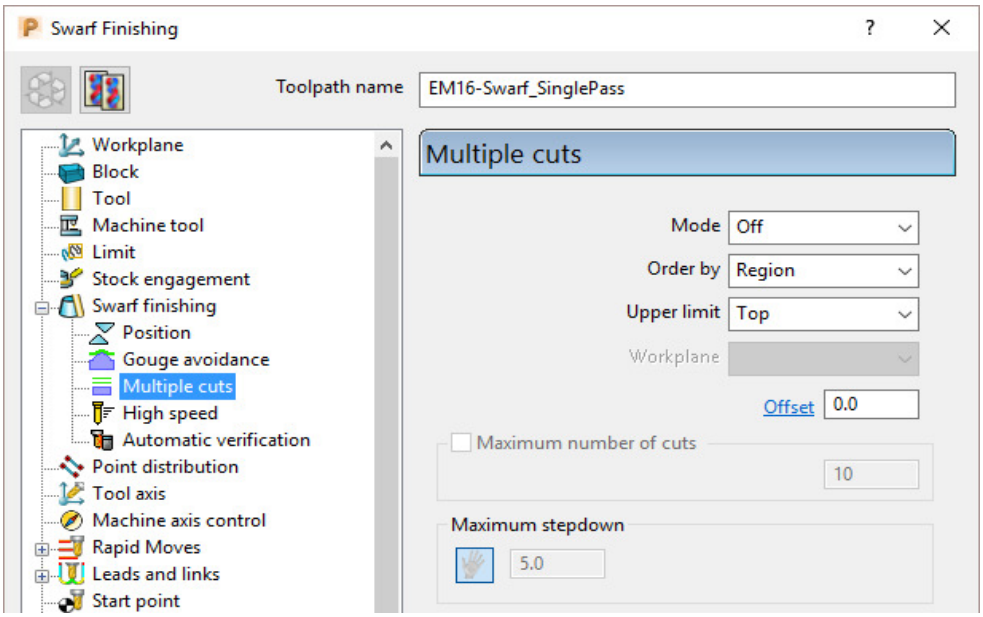

***Merge** will use a variable **stepdown** to exactly track the **selected surfaces,** upper and lower edges and minimize the number of lift moves.*

17 Right-click the last **toolpath *EM16-Swarf_Merge***, in the **PowerMill *explorer*** and select **Settings**.

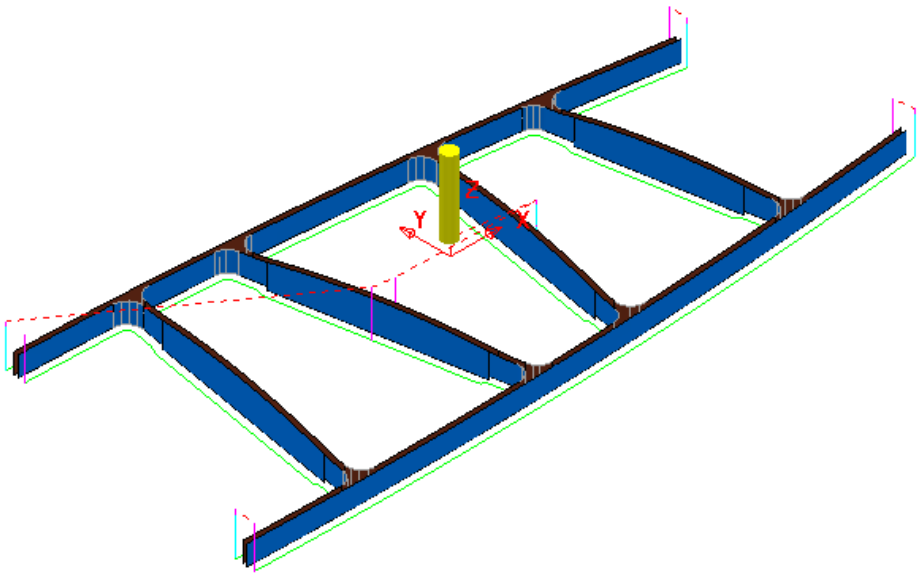
18 Select the ***Copy Toolpath*** option.

19 **Rename** the new **toolpath** as **EM16-Swarf_SinglePass**.

20 On the **Multiple Cuts** page, change the **Mode** to **Off**.




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**

 Copyright © Autodesk

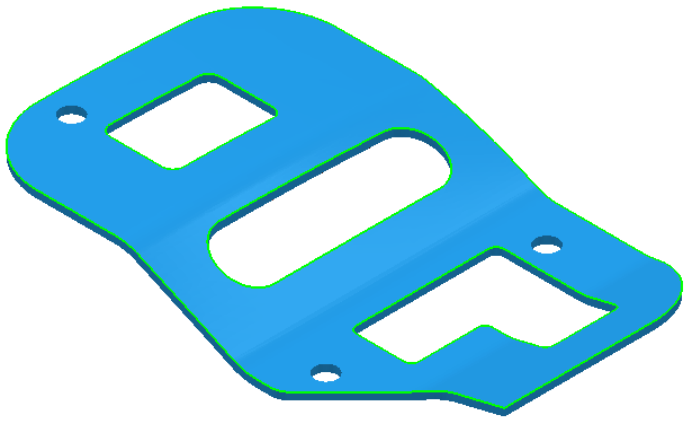
# Wireframe Profile Machining

**Wireframe Profile machining** is designed to machine to the left or right of the selected ***curves*** of a 3D Wireframe. The following example is a pressing that requires the outer form and recesses to be finish profiled. As the main imported model is in STL (triangle) format it is incompatible with **Boundary** options that operate on ***selected surfaces***. It is also impossible to create the necessary 3D Wireframe **curves** direct from the **STL model** in **PowerMill**. To resolve, the stored 3D Curves will be imported into a pattern to be applied in the **Wireframe Profile Machining** strategy.

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

2 Select **File** and **Import** the **Model**:

**.....\PowerMill_Data\Models\3D-Curve-TriModel.stl**



3 Select **File** > **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\3D-WireframeProfile**

4 Create a Dia 5 End Mill named **EM5**.

5 Create an empty **Pattern**, right-click on it, and from the local menu, select **Insert** > **File**:

**.....\PowerMill_Data\Models\3D-Curve-Wire.dxf**

6 **Rename** the **Pattern** as **3Dwire**.

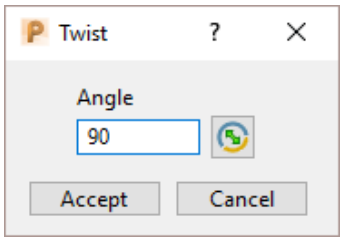
7 With the left mouse click on the **model** to select it.

8 In the **PowerMill *explorer***, right-click on **Workplanes** and from the local menu, select **Create** and **Orientate Workplane** - **Workplane at Bottom of Selection**.

9 **Rename** the **Workplane** as **MC-Datum**.

10 Right-click on the **Workplane** (**MC-Datum**) and from the local menu, select **Workplane Editor** to open the **Workplane Editing** toolbar.

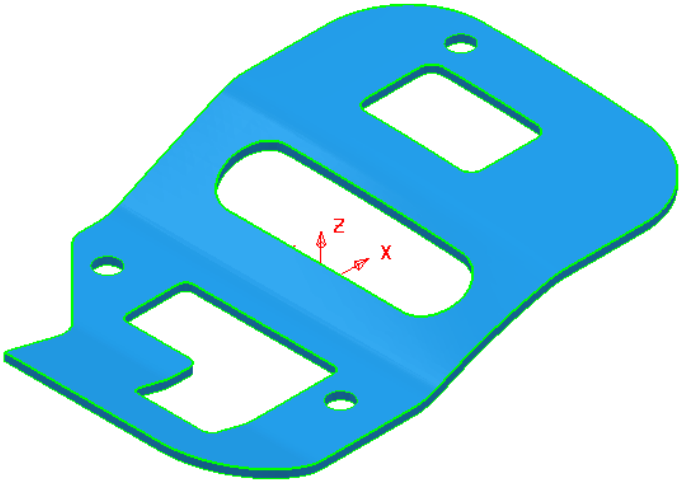
11 Click **Twist about Z** to open the following dialog.




12 Enter the value **90** and click **Accept**.

13 Click on the **Workplane Editor** toolbar to accept the changes and **Close**.

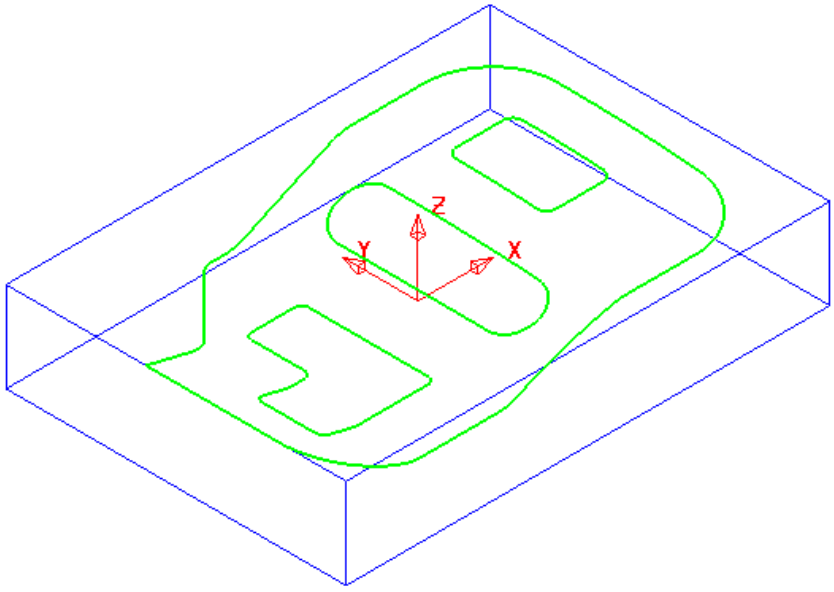
14 **Activate** the **workplane** (**MC-Datum**) and select an **ISO 1** view.



*The changes will not be physically active until the **Workplane Editor** toolbar has been **closed.***

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

16 ***Lock*** the **Maximum Z** value only and then input an **expansion** value of **10** before re-selecting **Calculate**.

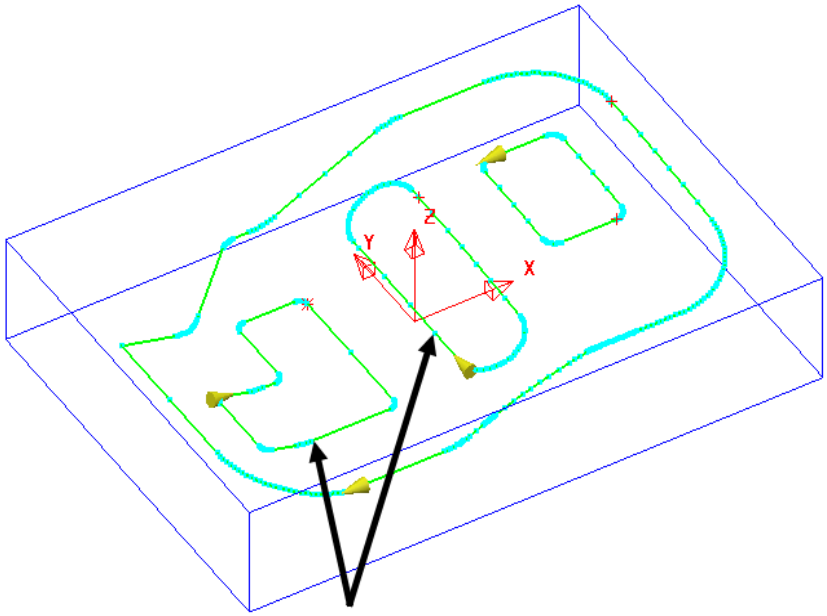




Copyright © Autodesk

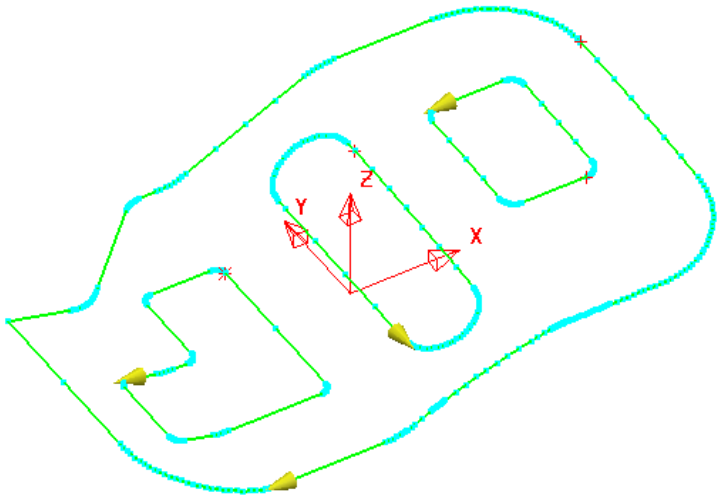
The final task prior to machining is to check the directions of the **Pattern** segments and to **reverse** any that will cause the tool to machine on the wrong side of a profile.

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**.

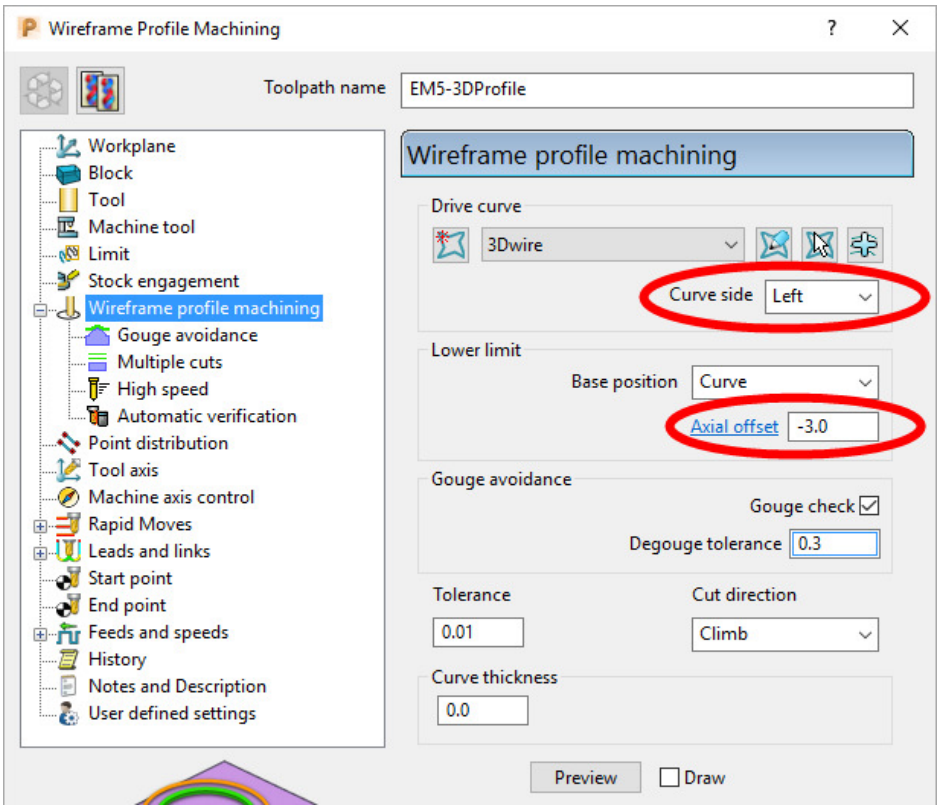
18 Select the 2 **segments** (arrowed on the previous diagram) and from the local **Pattern** menu apply **Edit** > **Reverse Selected**.




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

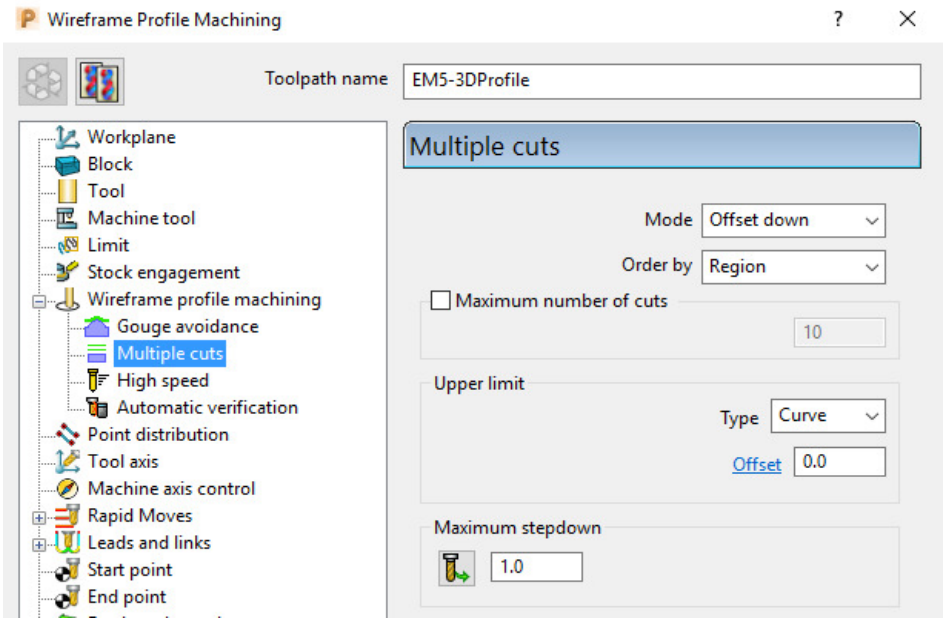
20 Select the **Strategy Selector** and select the **Finishing** tab in the dialog.

21 Select the **Wireframe Profile Machining** strategy and enter data exactly as shown on the following 2 pages of the dialog.



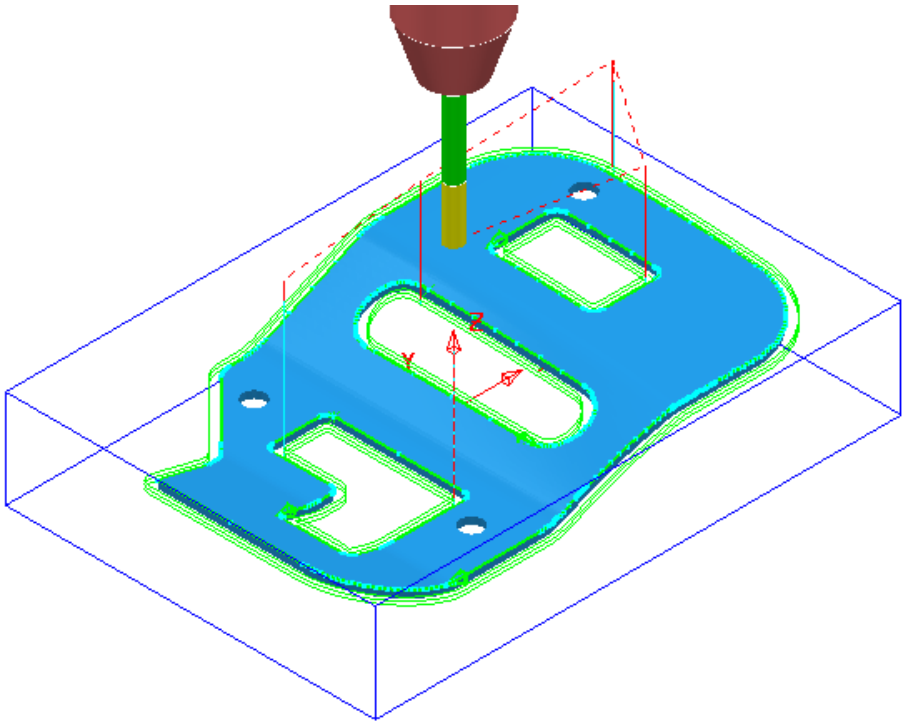

**Curve side** controls which side of the **Pattern** segment direction the tool operates.

**Axial offset** controls the depth of machining below the **wireframe curves**.




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


Copyright © Autodesk



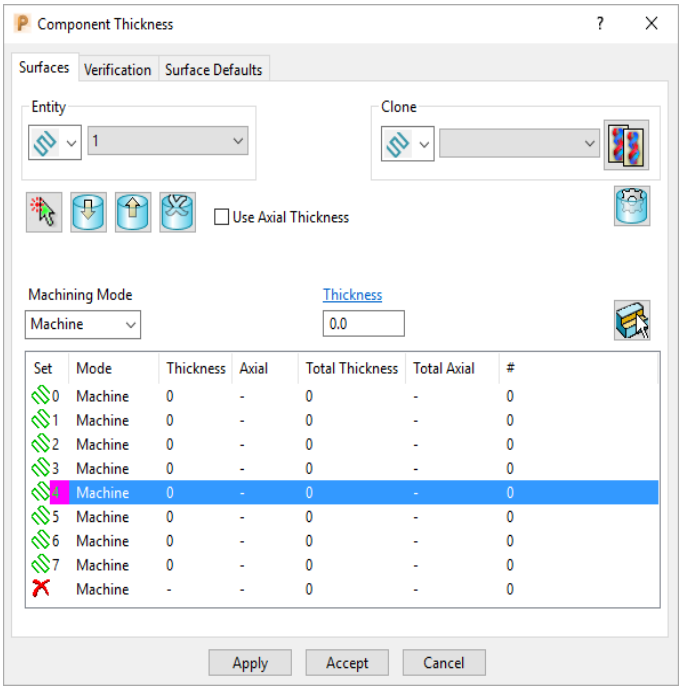
The **Wireframe Profile Machining** performs the profiling of the Pattern segments in 3D.

23 Select **File** > **Save** to update the stored **Project**.

Copyright © Autodesk

# 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 independantly 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.

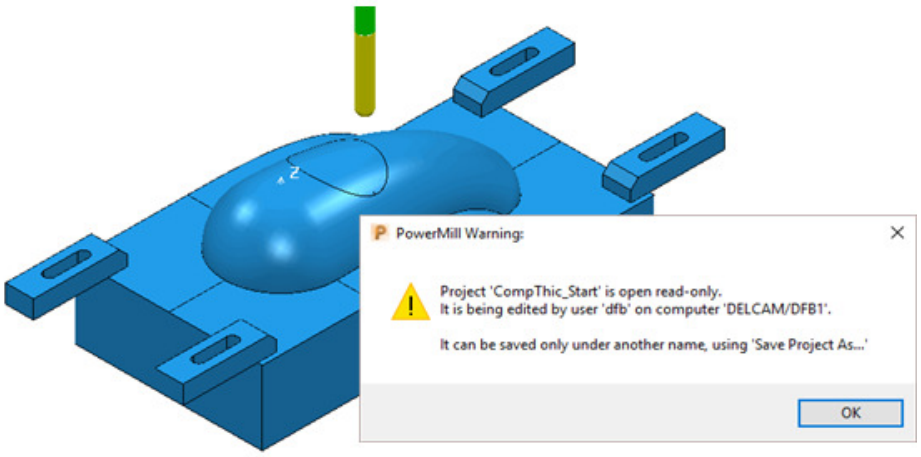
Copyright © Autodesk

# Component Thickness Example

Select **File** > **Delete all** and **Tools** > **Reset Forms**.

From **File** > **Open Project** select the read-only **Project**:

**.....\PowerMill-Data\Projects\CompThic_Start**



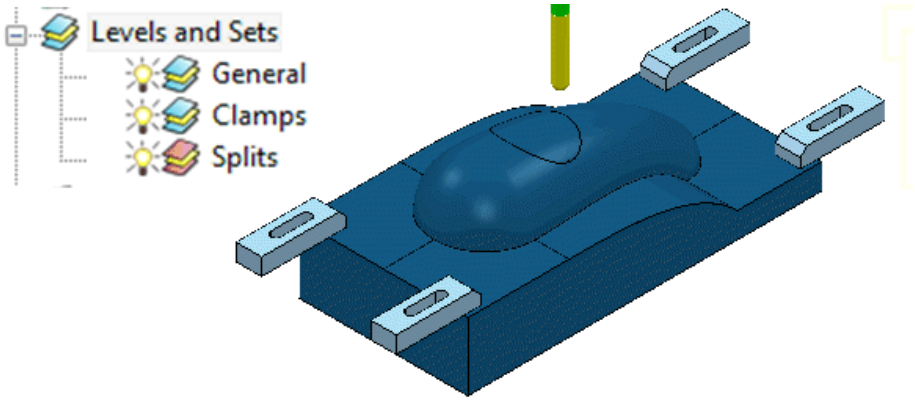

The **Project** contains an unprocessed **Steep and Shallow Finishing** strategy along with a Dia 8 Ball Nosed tool and all essential parameter settings.

Select **File** > **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\CompThickness-1**

*To make selection easier the **clamp surfaces** will be transferred to a new **Level** (**Clamps**) while the **split surfaces** will be copied to a new **Set** (**Splits**).*

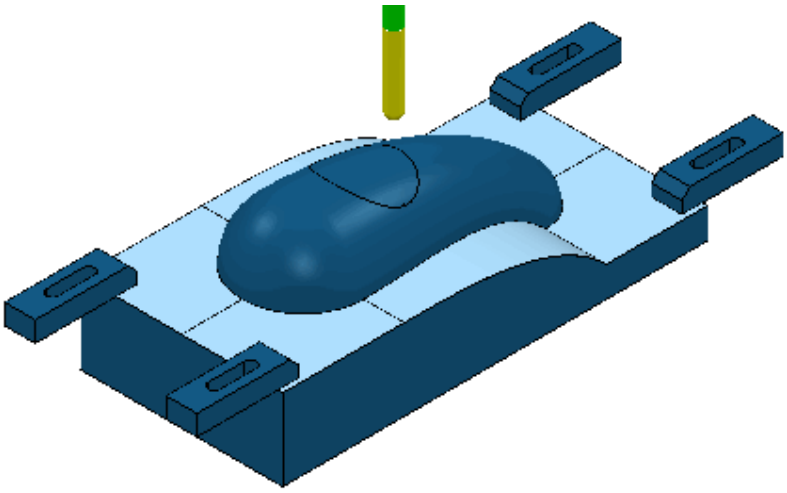
In the **PowerMill *explorer***, create a new **Level** and rename it as **Clamps** and then create a new Set and rename it as **Split**.




With the **clamp surfaces** selected, right-click on the **Level** named **Clamps** in the **PowerMill *explorer*** and from the local menu select, A**cquire Selected Model Geometry**.

*It will be easier to select and **Blank** (**Ctrl J**) all the other surfaces first before simply dragging a box over all of the remaining **clamp surfaces**.*

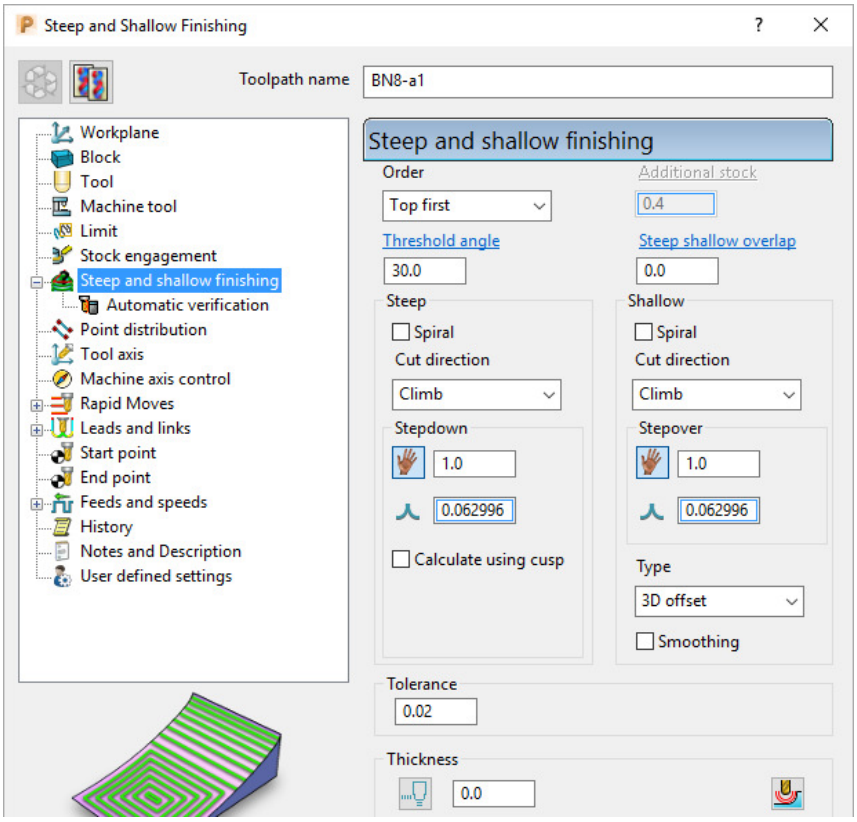

Copyright © Autodesk



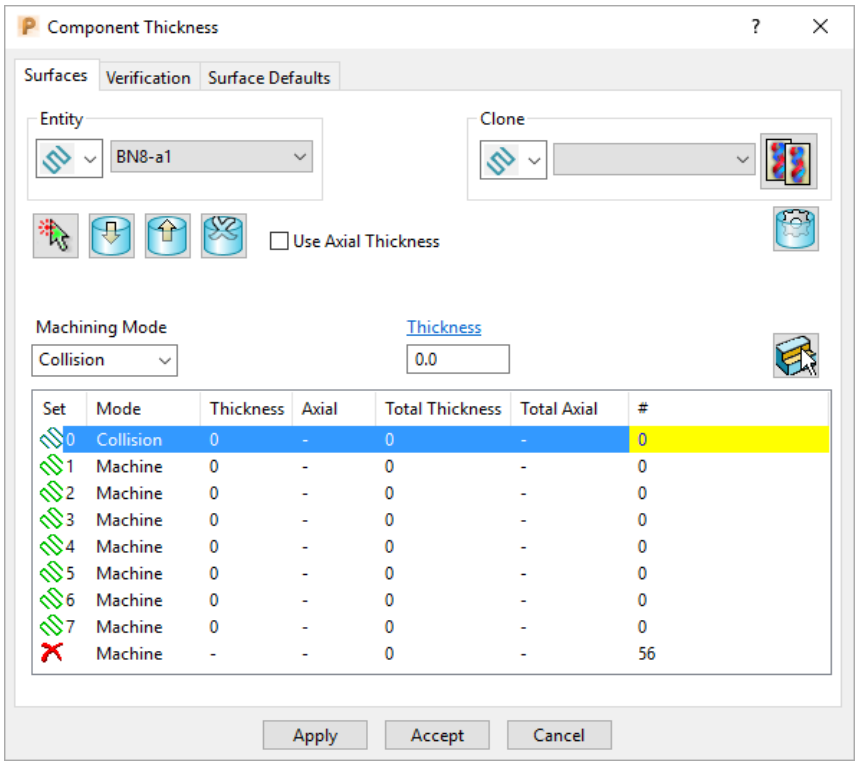
With the **split surfaces** selected, right-click on the **Set** named **Splits** in the **PowerMill *explorer*** and from the local menu select, **Acquire Selected Model Geometry**.

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).

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.




Select **Component Thickness** to open the following form.

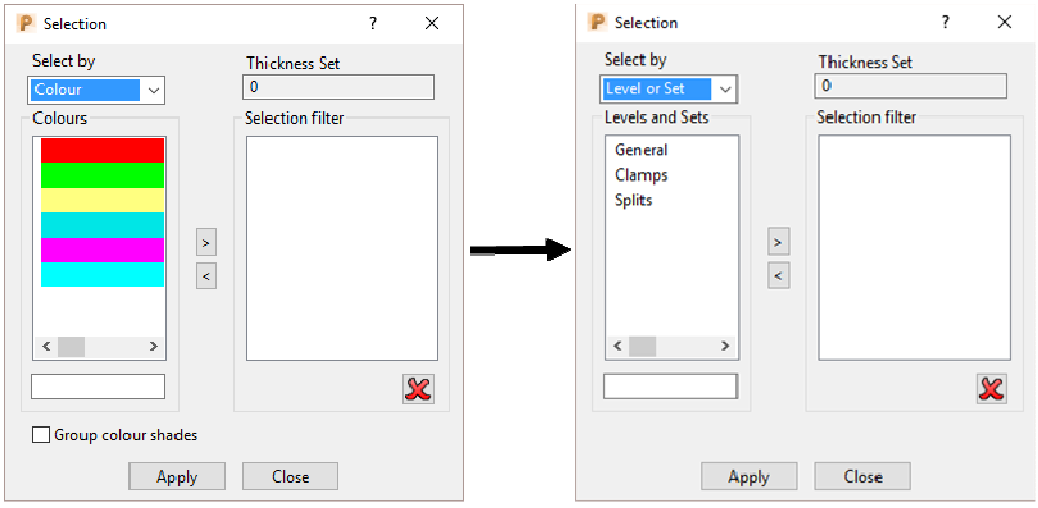



With the **Machining Mode** set to **Collision**, a **Thickness** value of **2** will be applied. At this safe distance the active tool will either ***retract*** or ***profile*** around the clamps.

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.

With the 1st row still selected, change the **Machining Mode** to **Collision** with a **Thickness** value of **2**.

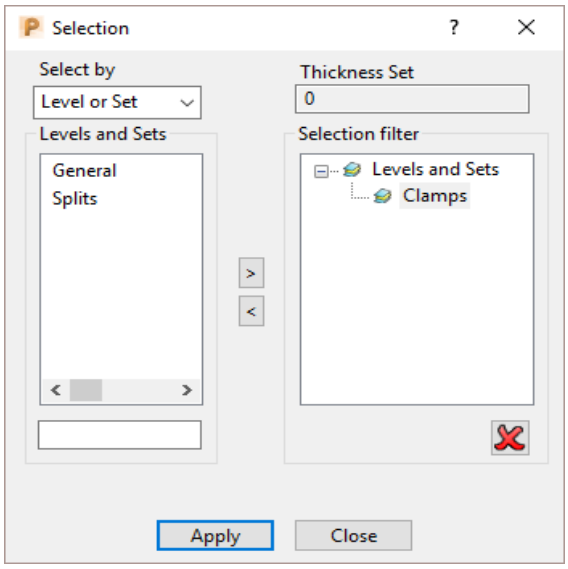
Select **Smart Selection** to open the following dialog:



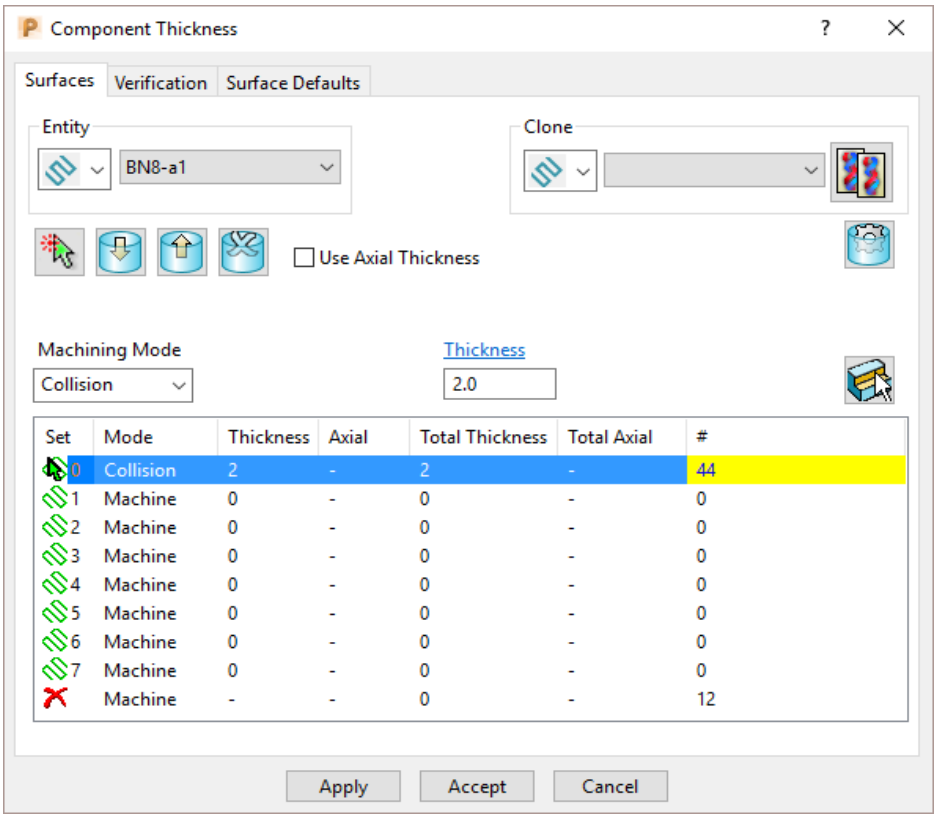

Switch **Select by** from **Colour** to **Level or Set**.

With **Clamps** selected in the **Levels or Sets** box click to acquire the clamp surfaces to the 1st row of the main dialog, and then **Close** the **Selection** dialog.


Copyright © Autodesk




The 44 ***Clamp Surfaces*** are acquired to the 1st row.



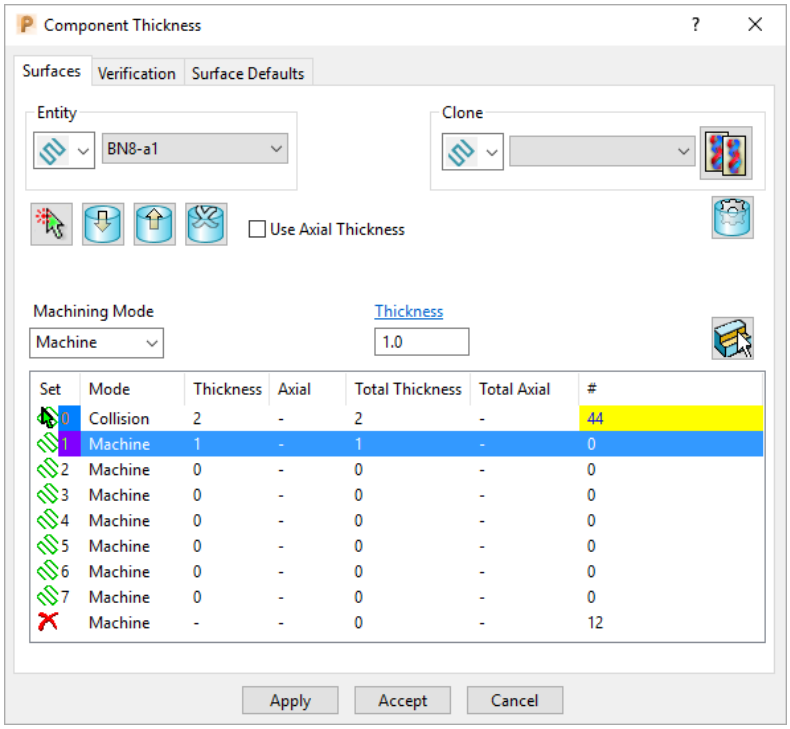

**Apply** the **Component Thickness** dialog to update the settings.

*In the above **Component Thickness** dialog, it is not necessary to use the 8 available rows in sequential order.*

Select the 2nd row and a square, colour coded symbol (in this case purple) will appear at the start of the row.

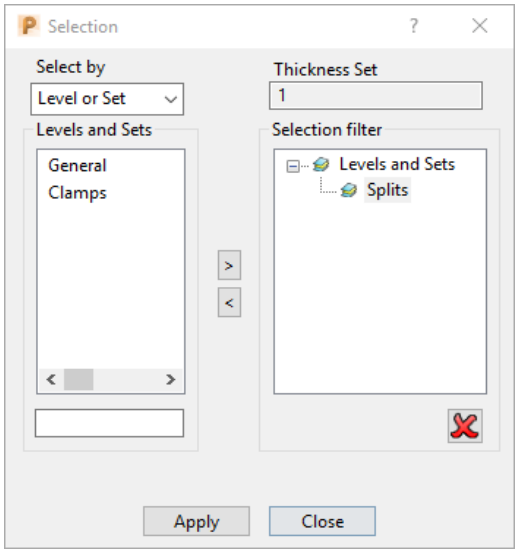
In **Smart Selection** select **Splits** in the **Levels** or **Sets** box and click the arrow icon to Acquire them in the second row of the main dialog.

With the 2nd row still selected, keep the **Machining Mode** as **Machine** but with a **Thickness** of **1**.




The **Thickness** value of **1.0** will control how much stock is left on the acquired ***split surfaces*** for this strategy.

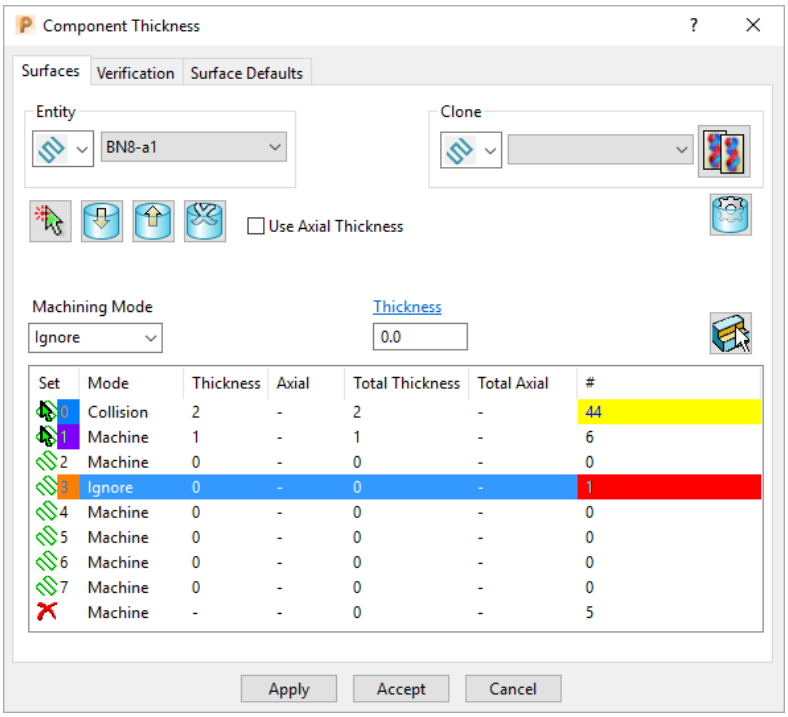
*This is in addition to a Thickness value specified in the main machining strategy dialog. In the above Component Thickness dialog the sum of these is displayed in the Total Thickness column.*





Copyright © Autodesk

Select the 4th row and a square, colour coded symbol (in this case orange) will appear at the start of the row.

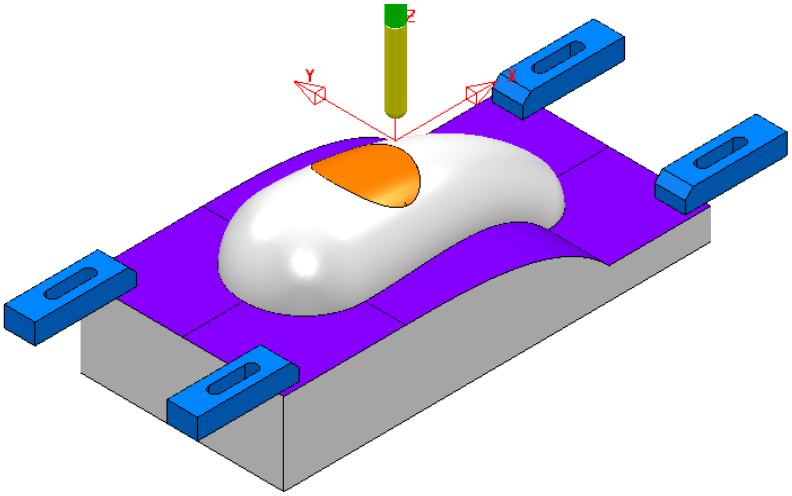



The ***cap surface*** must be ignored during the creation of this toolpath.

With the 4th row still selected, change the **Machining Mode** to **Ignore** with a **Thickness** value of **0**.

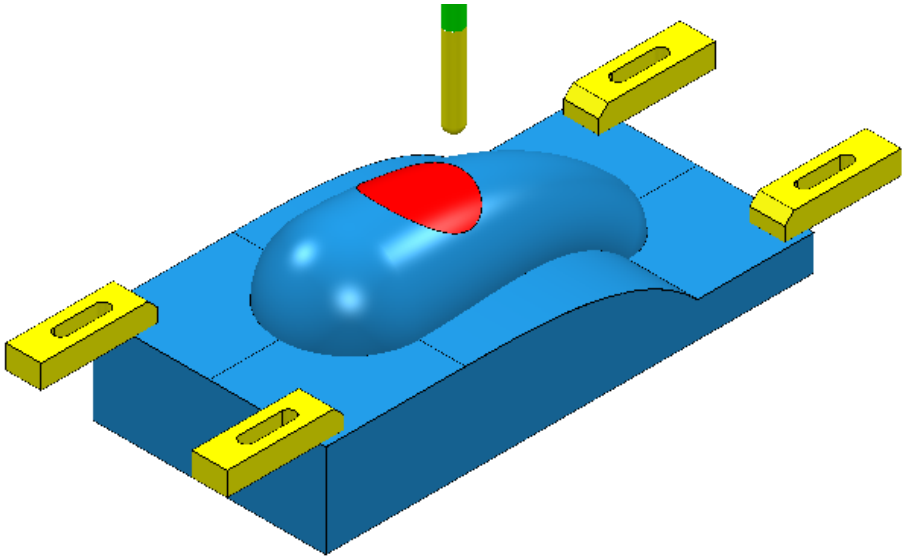
Use the left mouse key to manually select the cap surface and click on **Acquire Components** to include it in the 4th row.

**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.

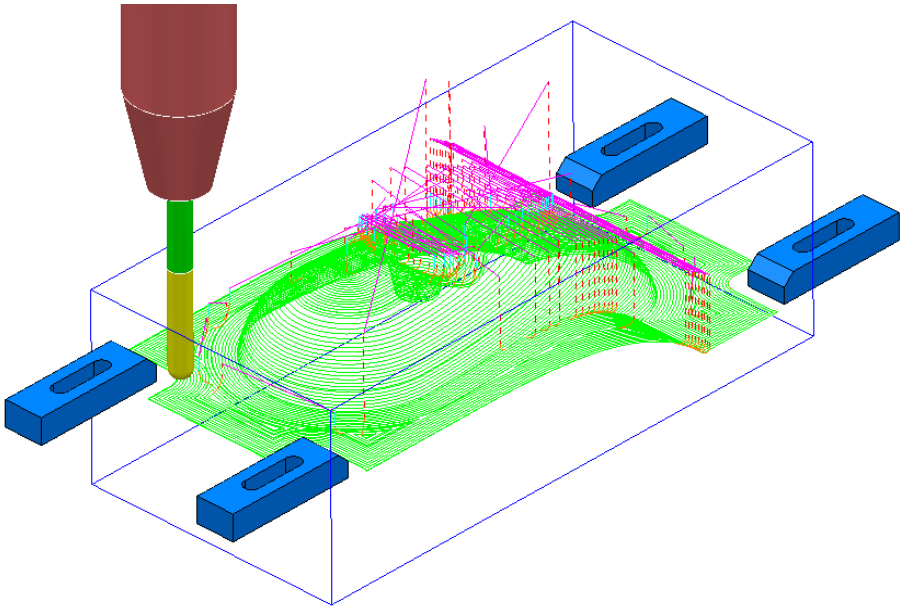
In the **Viewing** toolbar select the **Toolpath Machining Mode** Shade.



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.

**Accept** the **Component Thickness** dialog.

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



Select **File** > **Save** to update the stored Project.

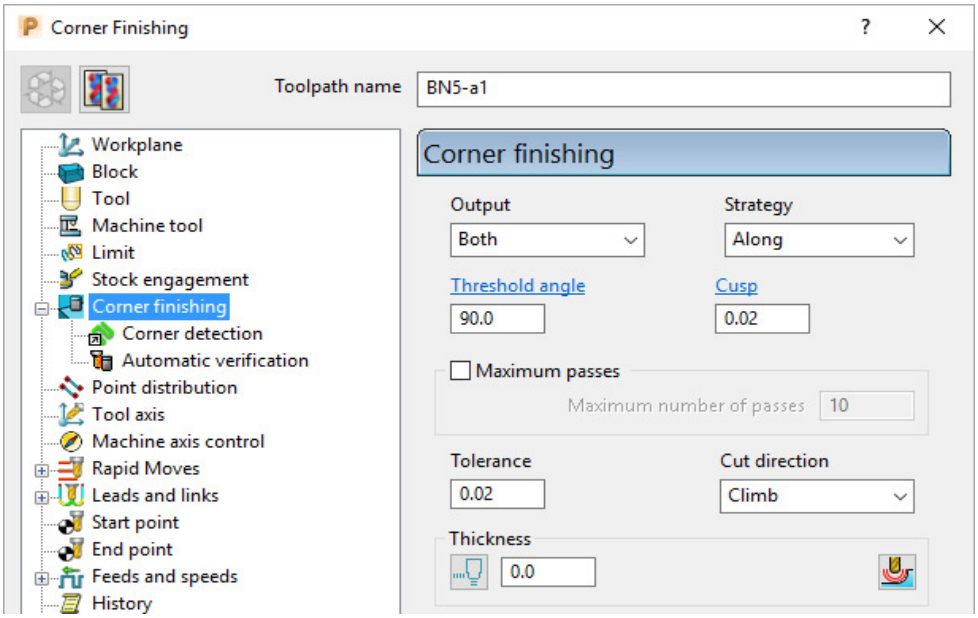
Do not close the **Project** as it will be continued in the next section.

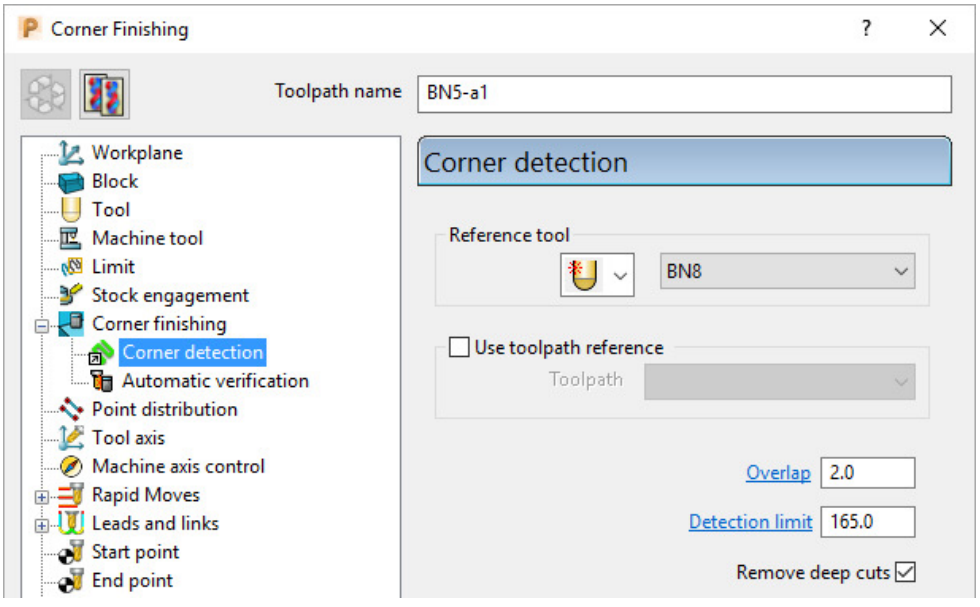

Copyright © Autodesk

# Cloning the Component Thickness settings from an existing Toolpath

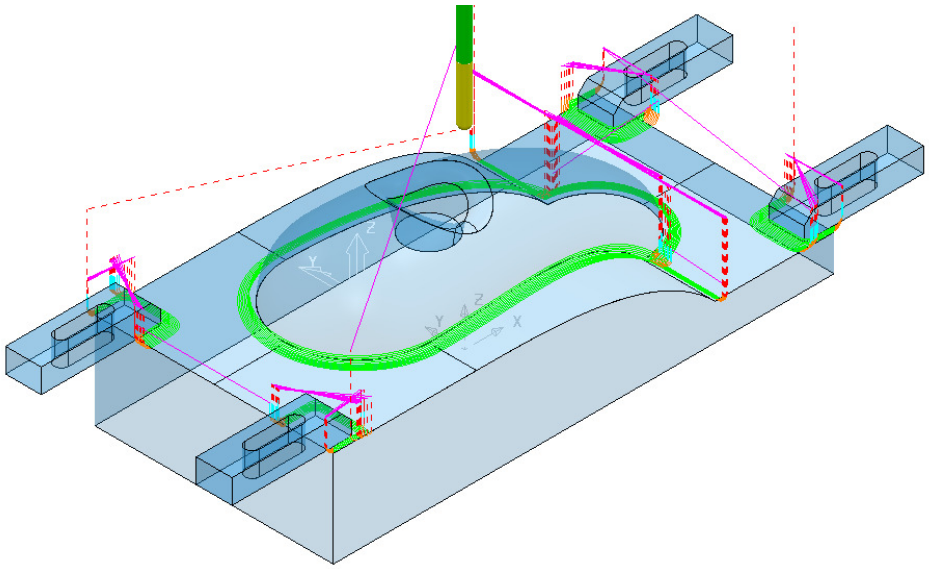
A **Corner Finishing** strategy ***BN5-a1*** will now be created using 'cloned' **Component Thickness** settings as used in ***BN8-a1***.

From the **Toolpath Strategies** select the **Finishing** option, **Corner Finishing**, and input settings exactly as shown below:



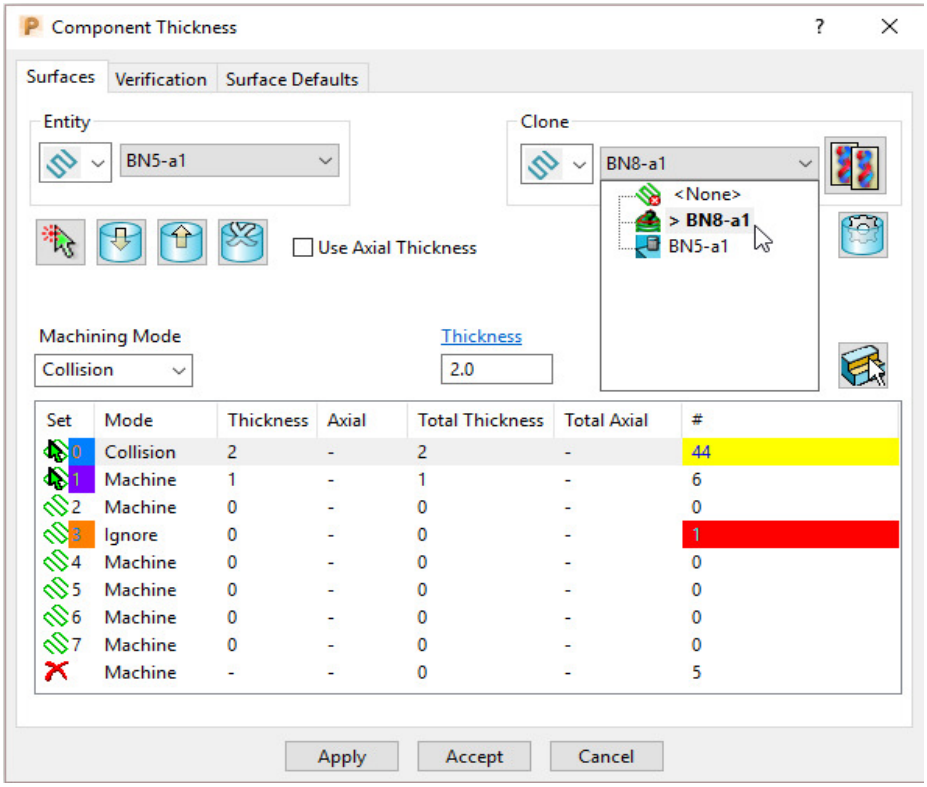

Select **Calculate** and have a look at the resultant toolpath in the graphics area.



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** .

In the ***Clone*** area of the form use the downward chevron to display and then select the ***toolpath*** **Bn8-a1**.




Select the **Copy Thickness Data** icon to **Clone** the **Component Thickness** settings into the dialog.

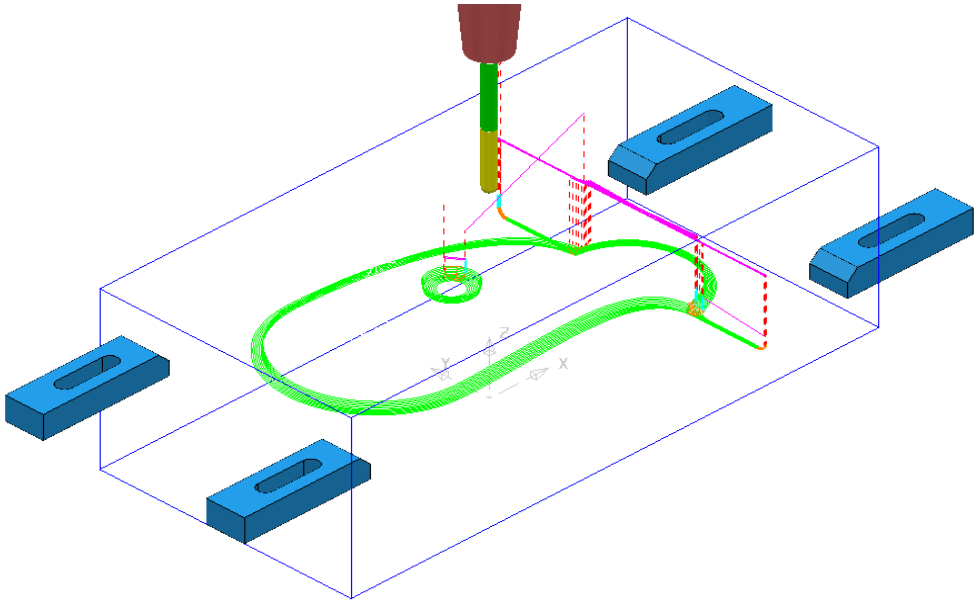

Copyright © Autodesk

The **Component Thickness** settings originally used in **BN8-a1** are now ***cloned*** into the strategy dialog for ***BN5-a1***.

*The process is not complete until the **Apply** button has been clicked.*

Select **Apply** followed by **Accept** to fully update the **Component Thickness** settings in the strategy **BN5-a1**.

Calculate the **Corner Finishing** strategy and then **Close** the dialog.



Select **File** > **Save Project**.

This will update:-

**.....\COURSEWORK\PowerMill-Projects\CompThickness-1**

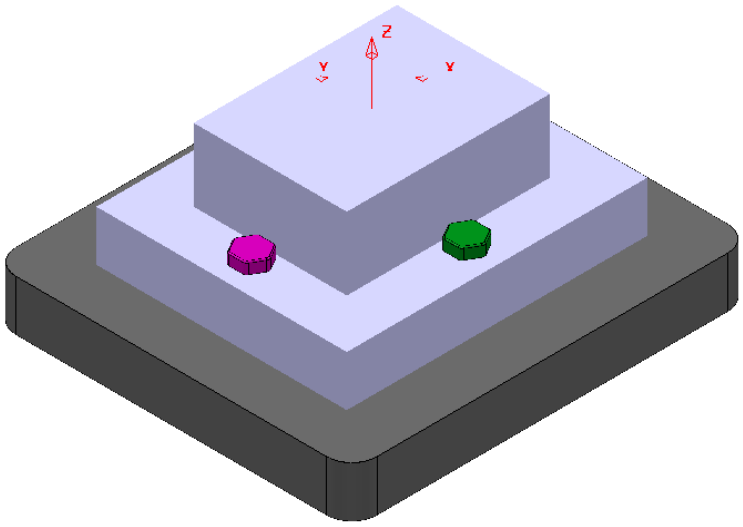
# Modification of existing strategies to allow for interchangeable clamping bolts.

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.

Select **File** > **Delete all** and **Tools** > **Reset Forms**.

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**.

Select **File** > **Save Project As**:

**.....\COURSEWORK\PowerMill-Projects\CompThickness-2**

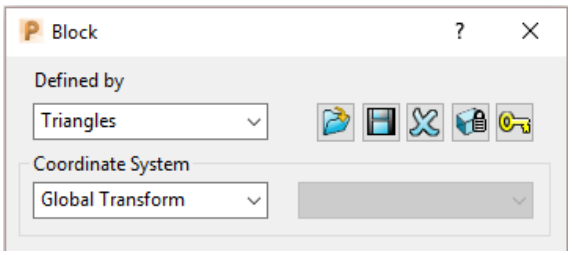
Select an **ISO 1** view (as shown in the above image).

In the **PowerMill *explorer***, right mouse click on the ***Toolpath*** **EM25-FaceMill-Top** and from the local menu select **Activate** followed by **Simulate from Start**.

*Before performing a ViewMILL simulation the Block will be modified to use a model of the pre-machined material.*

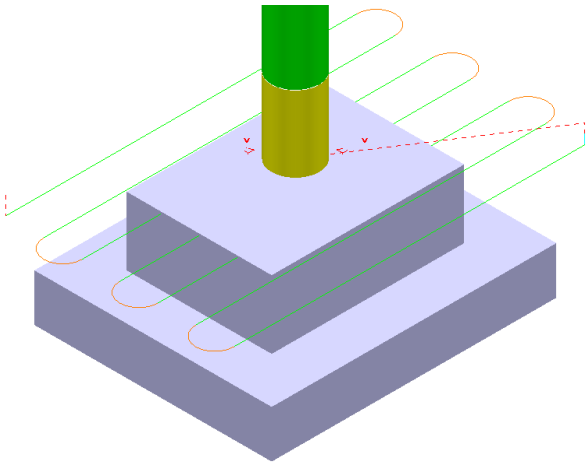
From the ***main*** toolbar select the **Block** dialog.


Copyright © Autodesk




Select **Defined by** > **Triangles** and **Coordinate System** > **Global before** selecting the option **Load block from file**.

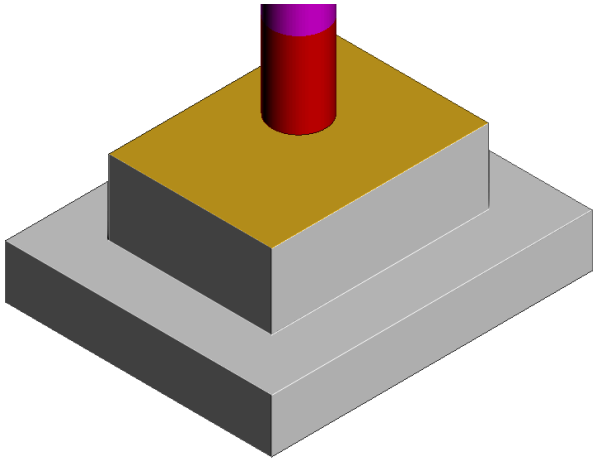
In the **Open Block from Triangles** dialog browse to:-
**....\PowerMill_Data\Triangles\CompThic-2_Block.stl** and select Open.



This **Block** definition is an accurate model of the pre-machined component.

Toggle into **ViewMILL** and select the **Rainbow Shaded Image**.

In the **Simulation** toolbar select the ***Play*** button.

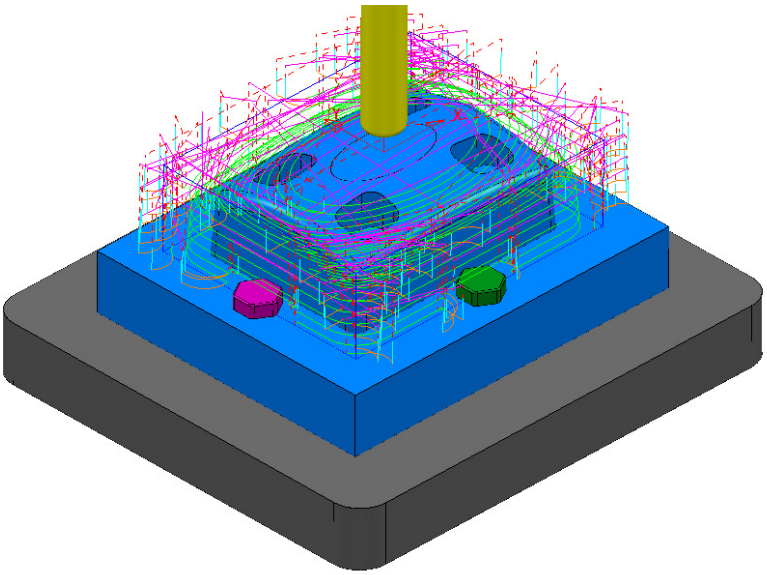


Select the **No Image** option to return to the **PowerMill window** and for the time being disconnect **ViewMILL** from the **Simulation**.

From the **main** menu select the **Block** option and in the dialog move the **Opacity** slider all the way to the left (wireframe).

In the Viewing toolbar select the option **Multicolour Shade**.

**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).

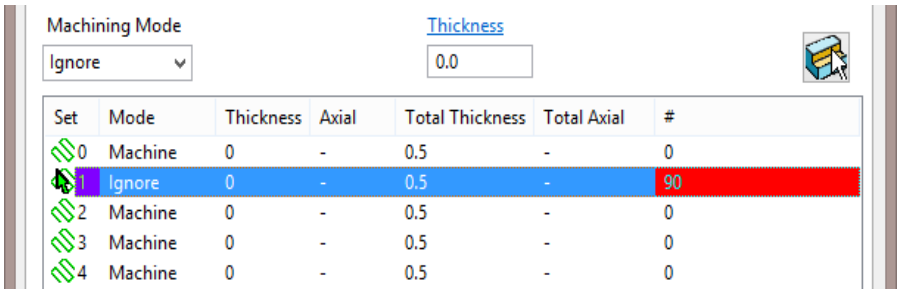
This toolpath will also be modified to apply **Collision** status to the ***green***, clamping bolts from a 2mm distance.

Right mouse click on the **Toolpath D16TR3-Rgh1** and select **Settings** to open the **Model Area Clearance** dialog.

Select the **Re-cycle** option to enable editing to occur.

In the main page of the **Model Area Clearance** dialog, select the **Component Thickness** dialog .

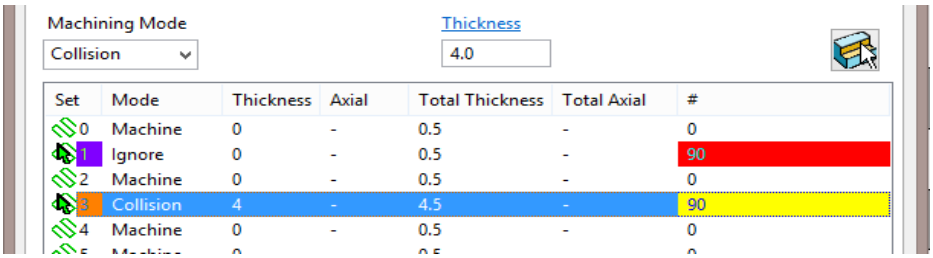
In the dialog use **Smart Selection** to acquire the data stored on the **Level ClampingBolts-X** to one of the colour coded rows.



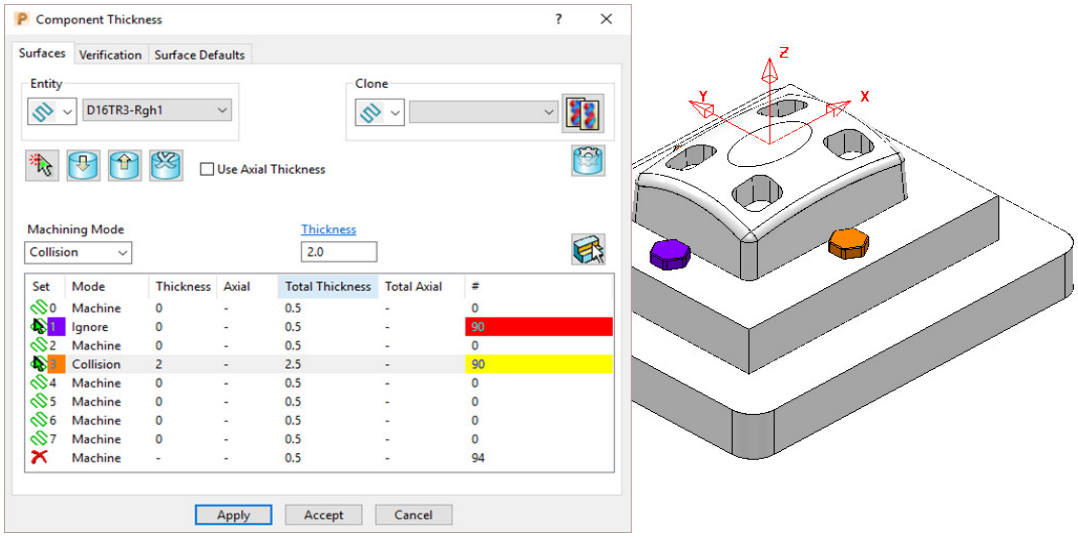

Set the row with **Machining Mode** > **Ignore**.


Copyright © Autodesk

Select one of the other colour coded rows and use **Smart Selection** to acquire the data stored on the **Level *ClampingBolts-Y***.

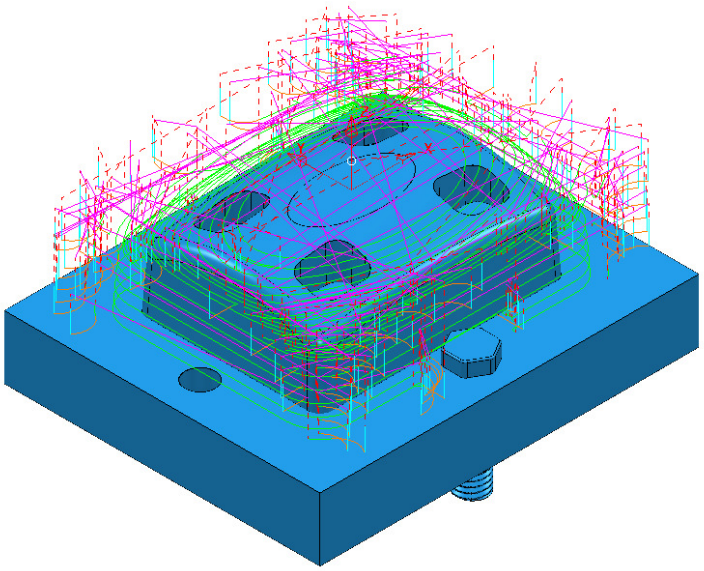



Set the row with ***Machining Mode*** > **Collision** with a **Thickness** > **2.0** and then select **Apply**.




*The parts of the model acquired to a row will be shaded in the same colour.*

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.

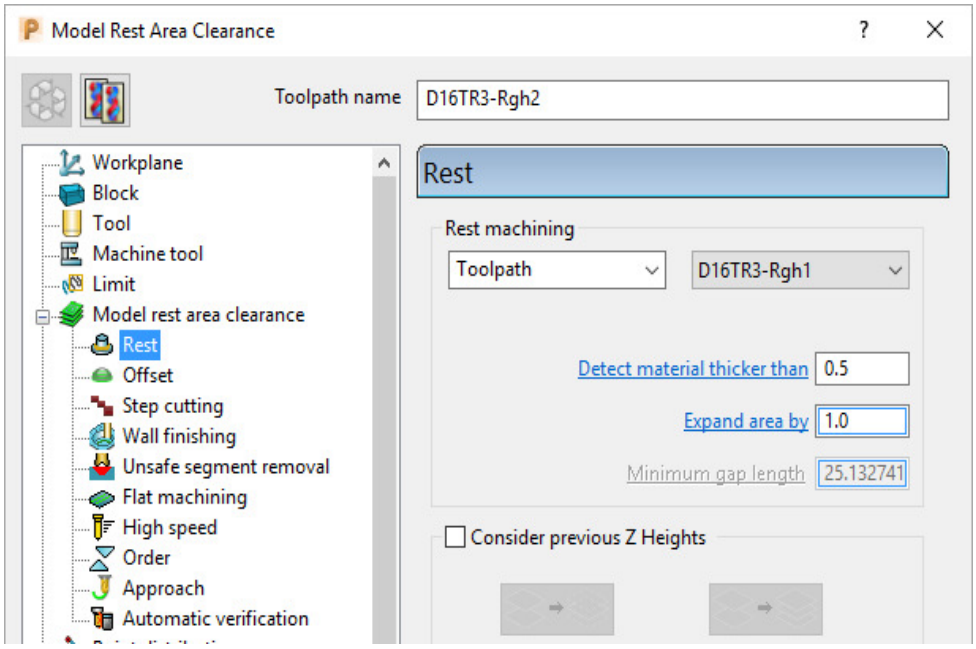
We now require a copy of the ***Toolpath*** (***D16TR3-Rgh1***) that locally machines the remainder of the component in the clamping bolt area along the Y-Axis.

Right mouse click on the ***Toolpath*** **D16TR3-Rgh1** and select **Settings** to open the **Model Area Clearance** dialog.

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.*

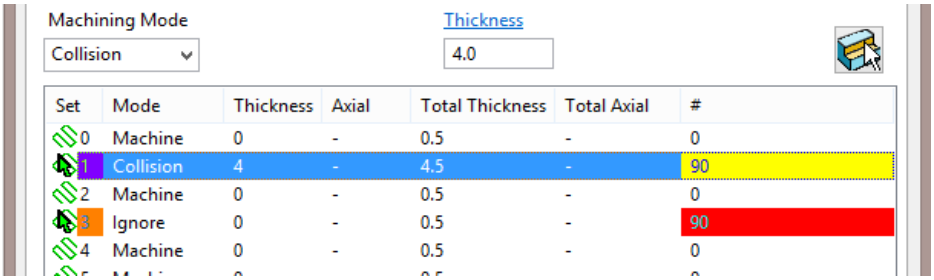
In the main page tick the **Rest machining** box and then open the **Rest** page of the dialog.




In the above dialog, insert the reference **Toolpath** > **D16TR3-Rgh1** and fill in the other values exactly as shown.

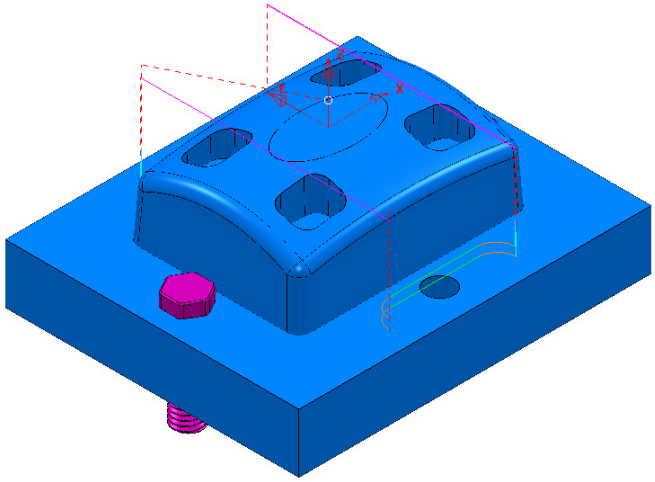
In the main page of the **Model Area Clearance** dialog, select the **Component Thickness** dialog .

Swap over the settings between the 2 existing rows (as shown below).




**Accept** the **Component Thickness** dialog and then **Calculate** the toolpath.


Copyright © Autodesk



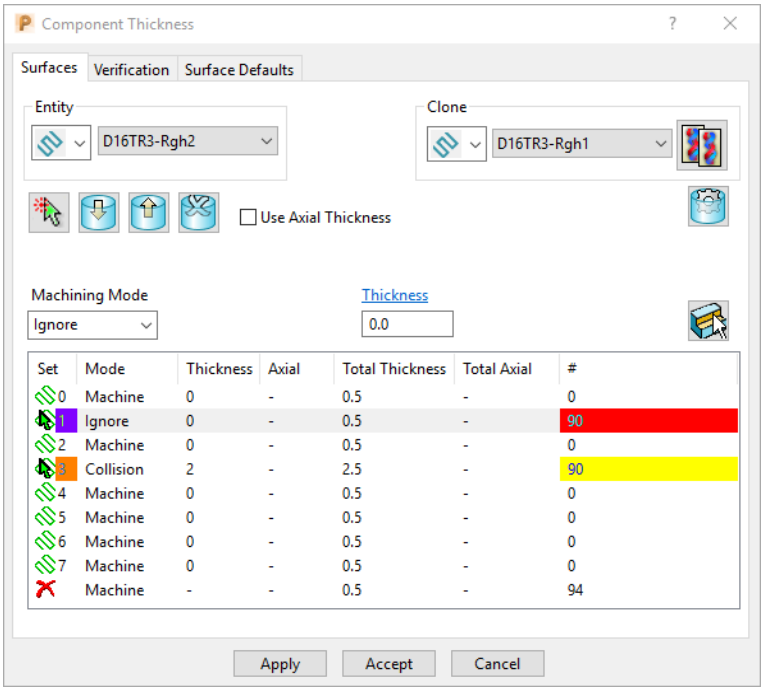
The updated ***Rest Roughing***, **toolpath** (***D16TR3-Rgh2***) now runs straight through the ***Y-Axis*** clamping bolts.

**Activate** the ***Toolpath*** **BN10-Fin1** and then select **Settings** from the local menu to open the **Steep and Shallow Finishing** dialog.

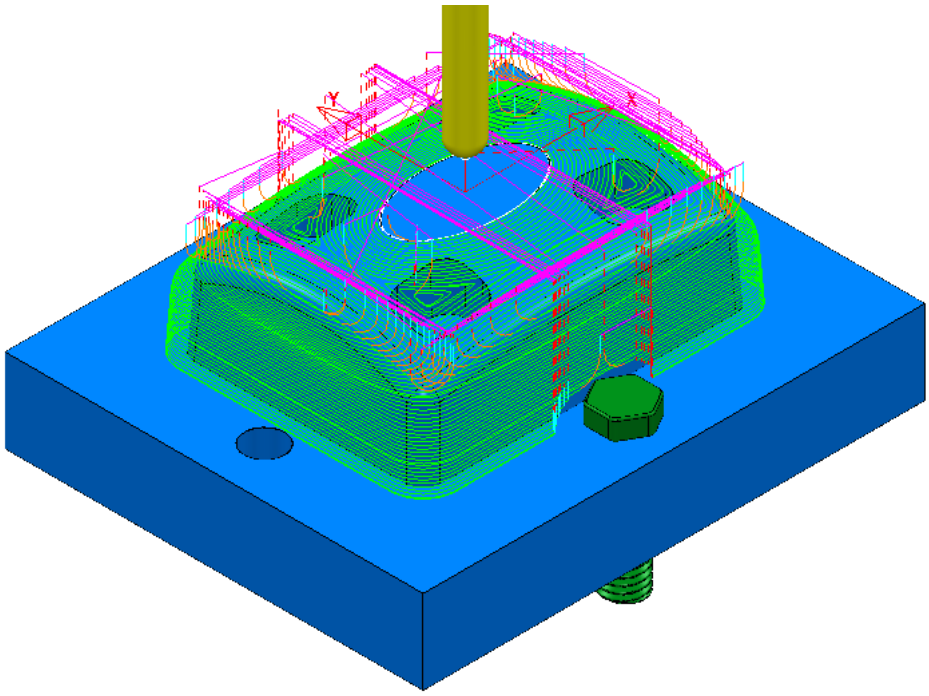
Select the **Re-cycle** option to enable editing to occur.

In the main page of the **Steep and Shallow Finishing** dialog, select the **Component Thickness** dialog .

In the **Clone** pull down select the ***Toolpath*** **D16TR3-Rgh1** and the click on the **Copy Thickness Data** icon.




**Accept** the **Component Thickness** dialog and then **Calculate** the toolpath.



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.

Right mouse click on the **Toolpath BN10-Fin1** and select **Settings** to open the **Steep and Shallow Finishing** dialog.

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**.*

In the main page of the **Steep and Shallow** Finishing dialog, select the **Component Thickness** dialog .

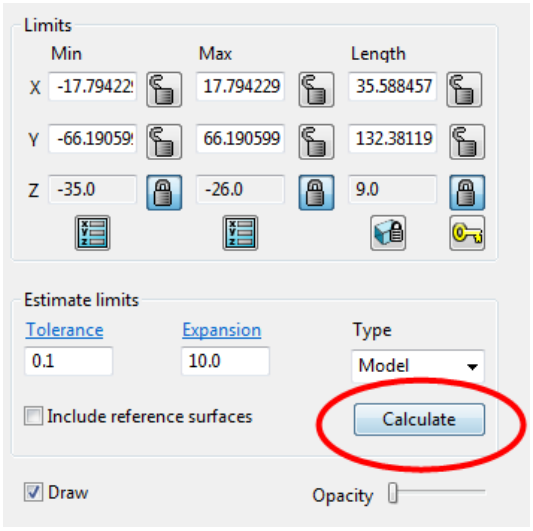
In the **Clone** pull down select the ***Toolpath*** **D16TR3-Rgh2** and then click on the **Copy Thickness Data** icon.

**Accept** the **Component Thickness** dialog.

In the **Steep and Shallow Finishing** dialog select the **Block** page and lock both the **Min Z** value (**-35**) and **Max Z** value (**0**) but do not close the **Block** page.

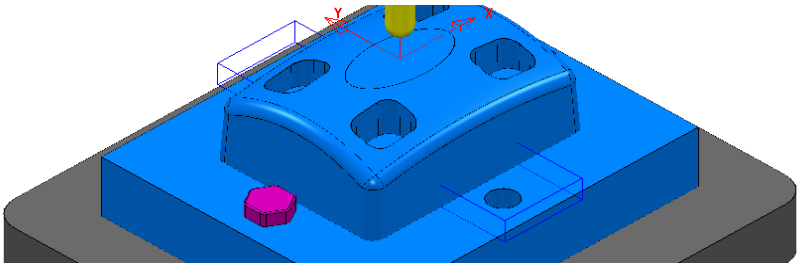
In the **PowerMill *explorer***, right mouse click on the **Level** named ClampingBolts-Y and from the local menu click on Select All.


Copyright © Autodesk



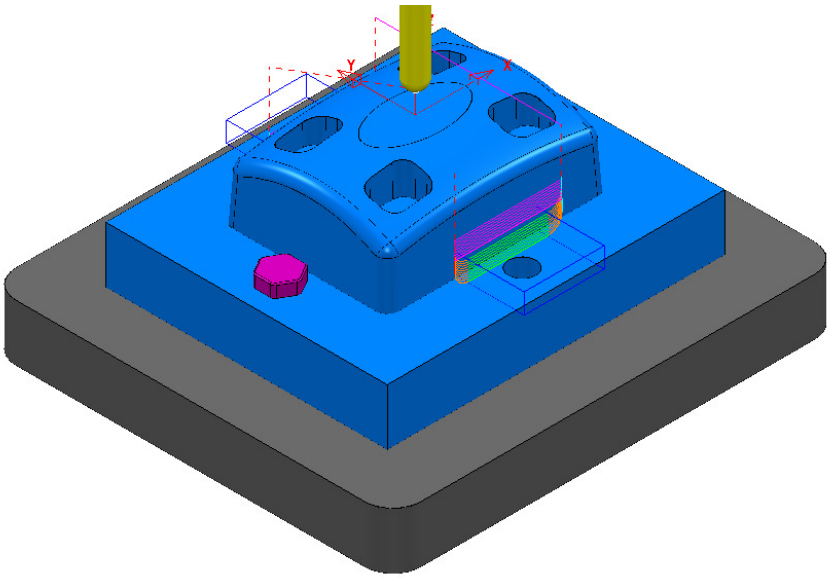

Back in the **Block** dialog enter an **Expansion** of **10** and **Calculate**.

**Unlock** the **Z Max** value and manually input the value **-26**.



Finally select **Calculate** at the bottom of the main **Steep and Shallow Finishing** dialog to create the new **toolpath BN10-Fin2**.

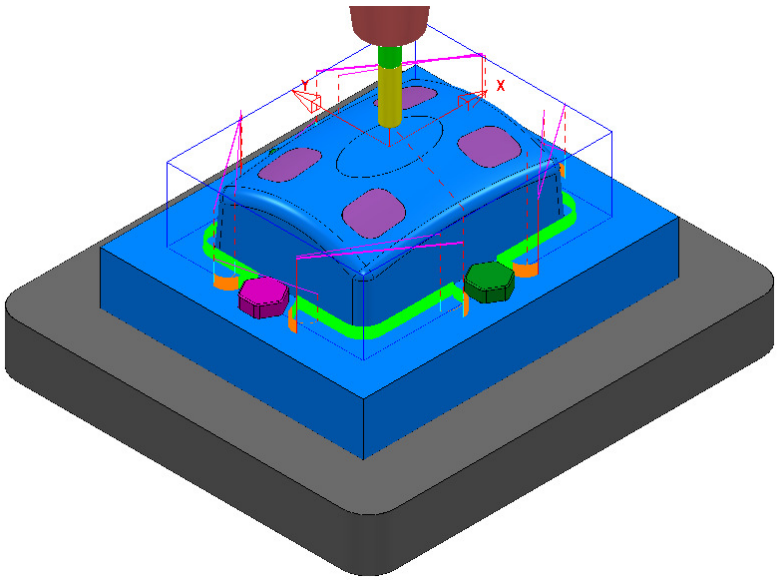
Cancel the **Steep and Shallow Finishing** dialog.



**Save** the **Project** (but do not close it).

# Exercise

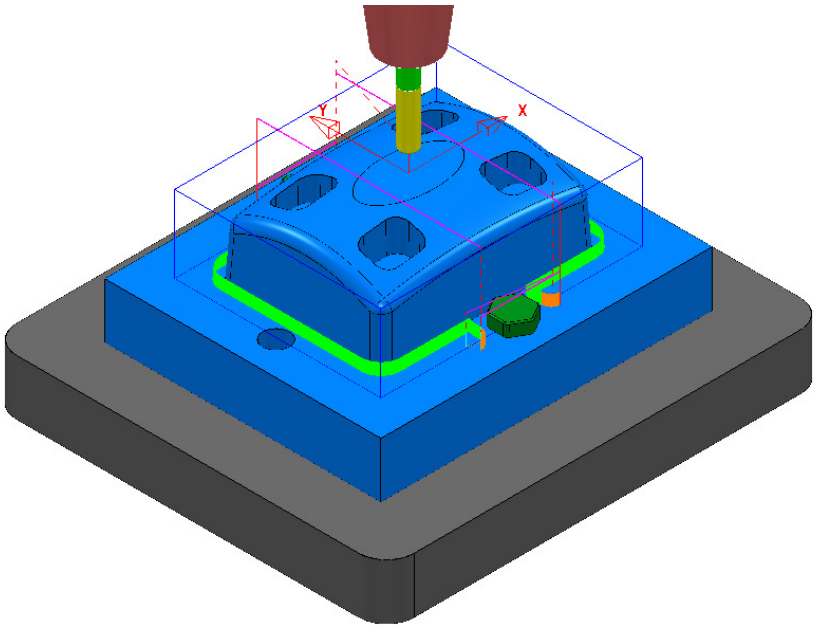
**Activate** the existing **Constant Z Finishing** toolpath **D8TR1-Fin1**.



Select **Settings** and ***Recycle*** on the above toolpath.

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).

**Calculate** the changes to toolpath **D8TR1-Fin1**.



Select **Settings** and ***Make a Copy*** of the toolpath **D8TR1-Fin1**.

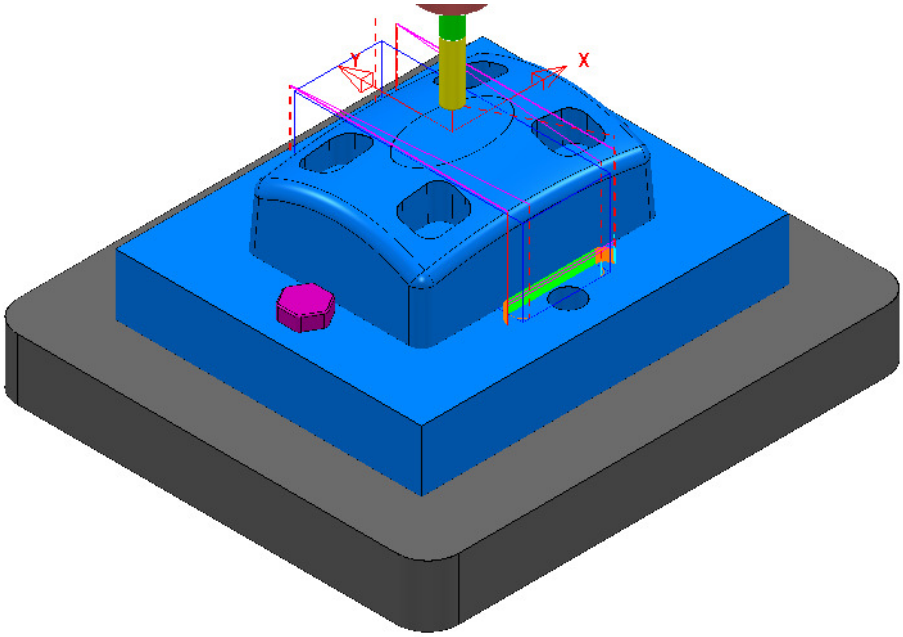

Copyright © Autodesk

**Rename** the copy as **D8TR1-Fin2**.

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).

Modify the X dimensions of the **Block** to keep the **Constant Z Finishing** toolpath **D8TR1-Fin2** inside the area already machined with **D8TR1-Fin1**.

**Calculate** the toolpath **D8TR1-Fin2**.




In the **PowerMill *explorer*** re-order the **Toolpaths** to keep them grouped for the same pair of clamping bolts.

**Save** the **Project** to update:-

**.....\COURSEWORK\PowerMill-Projects\CompThickness-2**

Copyright © Autodesk

# 17. Stock Models

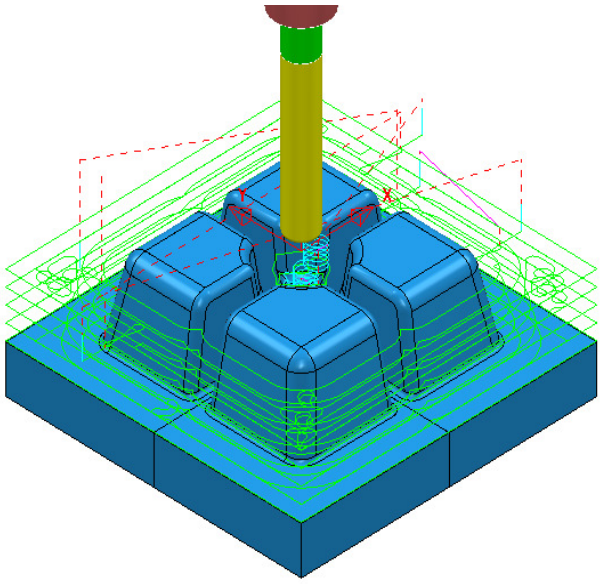
## Introduction

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**:

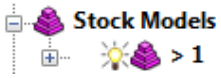
   **.....\PowerMill_Data\Projects\StockModel_1**



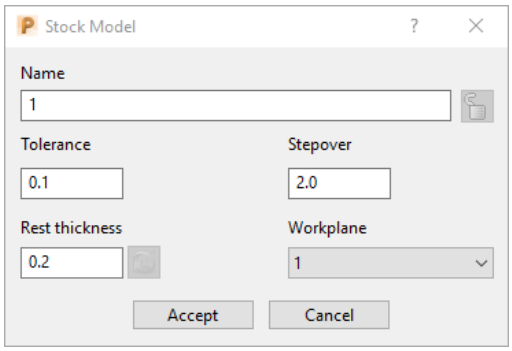
3. Select **File** - **Save Project As:**

   **.....\ COURSEWORK\PowerMill-Projects\StockModelExample**

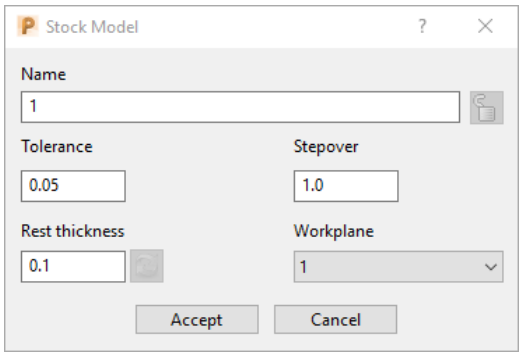
4. In the **PowerMill *explorer*** select **Stock Models** > **Create Stock Model**.



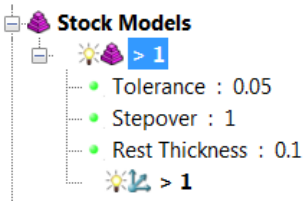

An empty **Stock Model** entity appears in the **PowerMill explorer** and an associated **Stock Model** settings dialog appears in the graphics area.




The default settings for triangulation of the **Stock Model** are displayed. In this case we will modify the settings to create a more accurate **Triangle Mesh** for the **Stock Model**. Once the **Stock Model** is physically created then these inherent settings cannot be retrospectively altered.

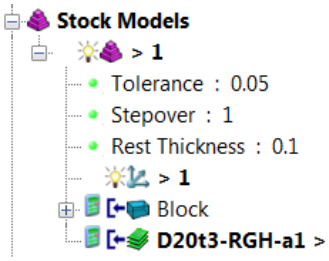



5 Modify the values in the **Stock Model** settings to be exactly as shown above and then **Accept** the dialog.

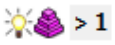



The displayed settings in the **Stock Model** will change accordingly.

6 Right mouse click on the toolpath **D20t3-RGH-a1** and from the local menu select **Add to** > **Stock Model**.

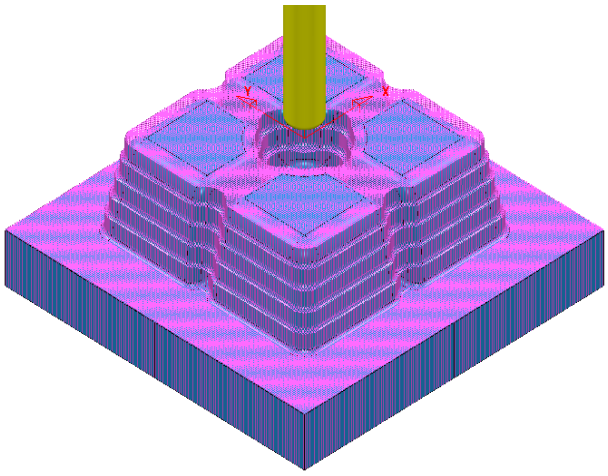



The **toolpath** and associated **Block** are included in the **Stock Model** tree but at this stage are not calculated.

7 In the **PowerMill *explorer***, **Stock Models** area, right mouse click on 

and from the local menu select **Calculate**.

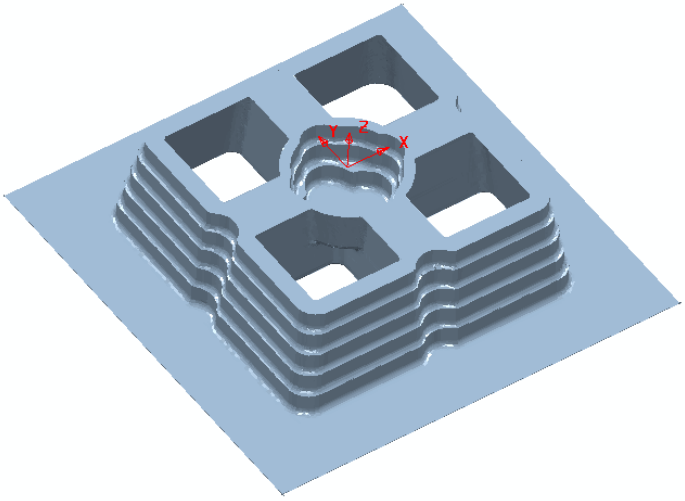

Copyright © Autodesk



The **Stock Model** will display the visual effect of toolpath **D20t3-RGH-a1** as a purple wireframe mesh.

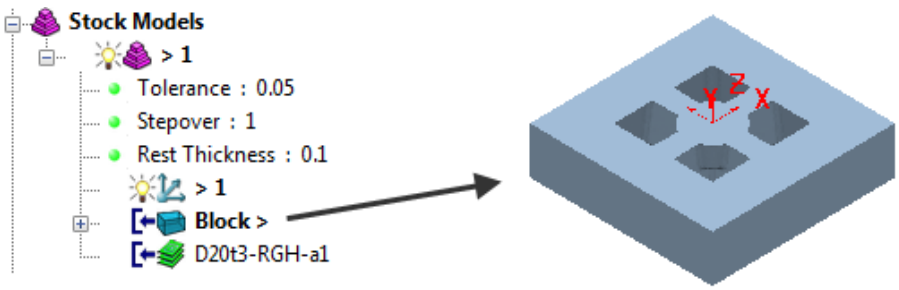
8 Right-click on **Stock Model** - **1** and from the local menu, select **Drawing Options**, and then tick both, **Shaded** and **Show Rest Material**.

9 From the **Viewing** toolbar, undraw the model (both **Shaded** and **Wireframe**).



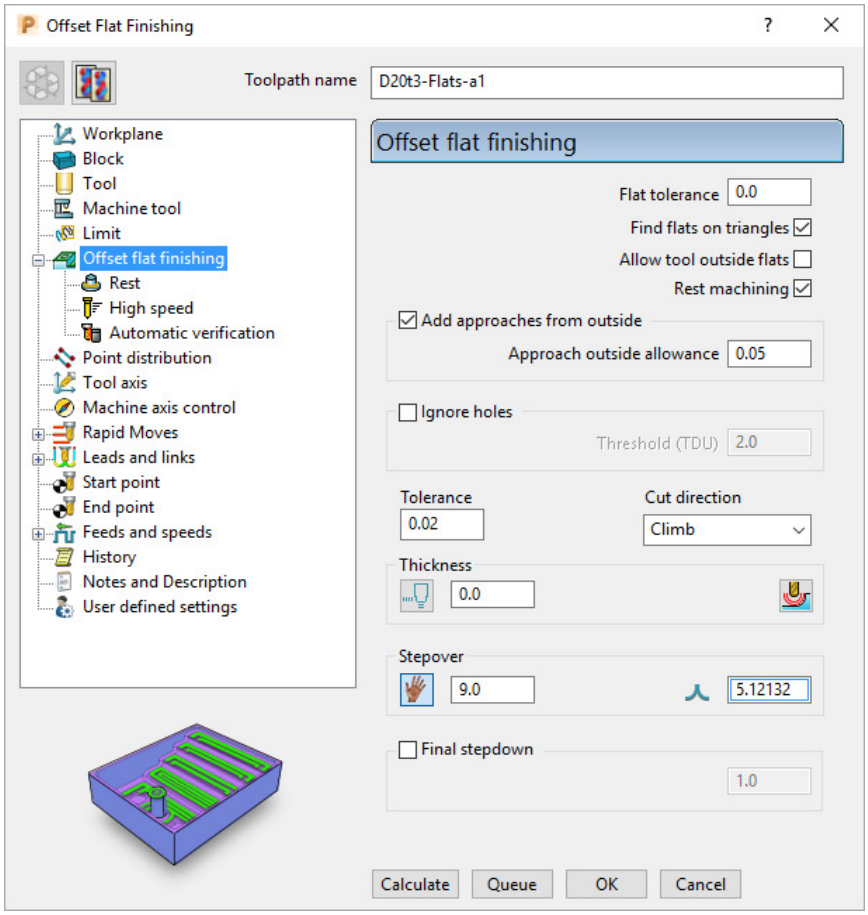
The **Stock Model** is now displayed as shown with the component model and applied toolpath both taken into account.

This represents the remaining material outside the component form after the taking into account the toolpath **D20t3-RGH-a1**.




10 If the **Block** is made active in the **Stock Model** tree then this is the stage of the machining process that will be displayed.

11 From the **Strategy Selector** , **Finishing** options select the **Offset Flat Finishing** strategy and enter data in the 3 pages exactly as shown below.

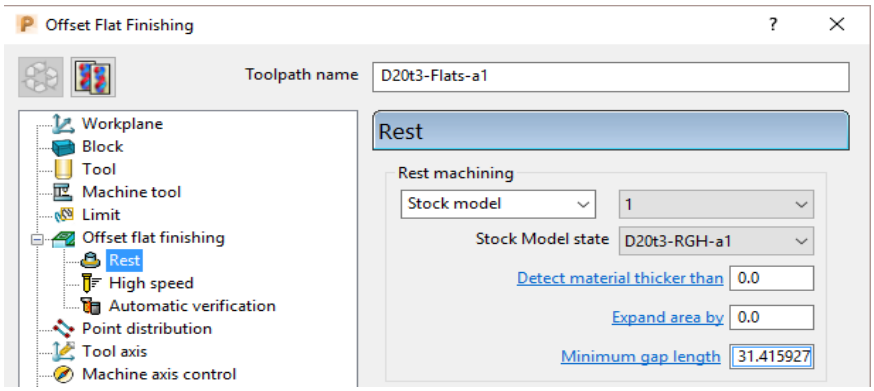



a **Add approaches from outside** is **un-ticked** to enable fully independent **Leads** to be applied.

b **Allow tool outside flats** is **un-ticked** to limit the toolpath exactly to the X 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.

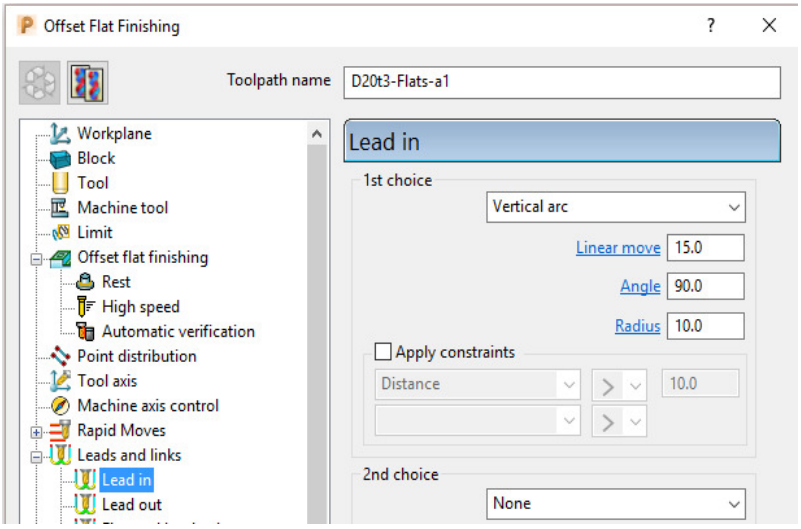
d By ticking this option the **Rest** page will become available for selection in the local browser.




12 In the above dialogue select the **Stock Model** as **1**.


Copyright © Autodesk

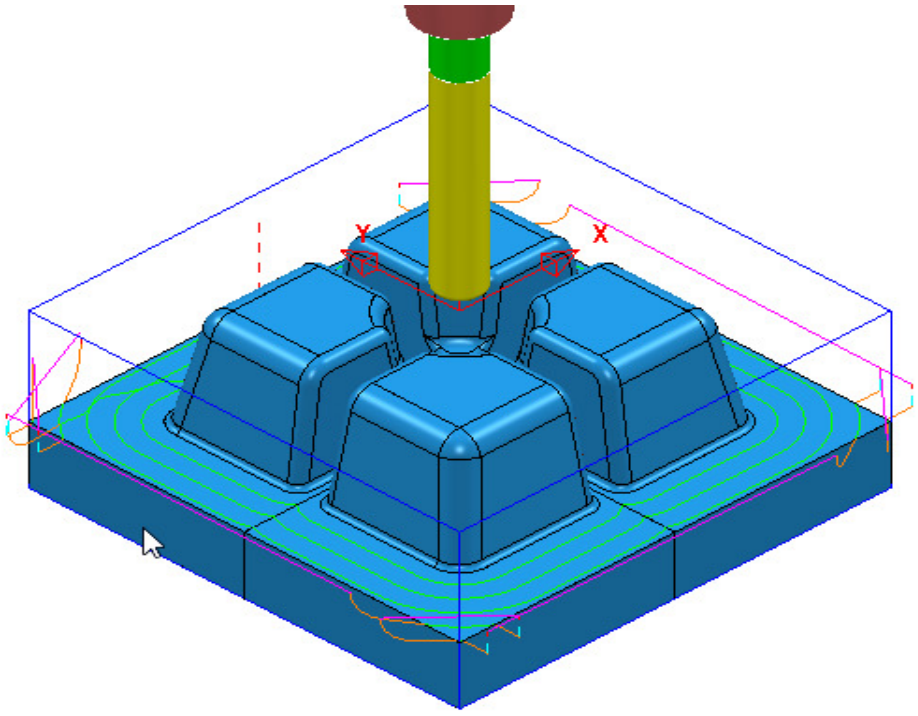
13 From the local explorer select the **Leads and Links** options and in the **Lead in** form enter data exactly as shown below.

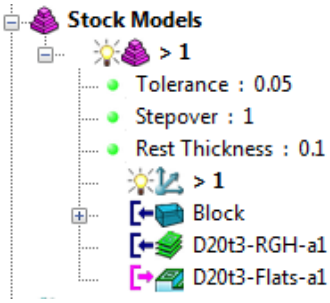



14 Then copy the **Lead in** options to the **Lead out** .

15 **Accept** the dialog.

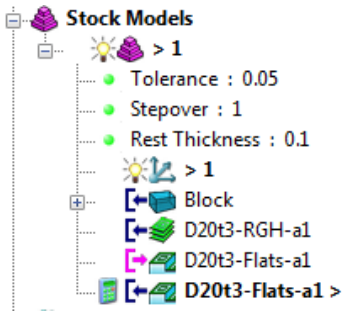
16 **Calculate** the **Offset Flat finishing** strategy and then **Close** the dialog.



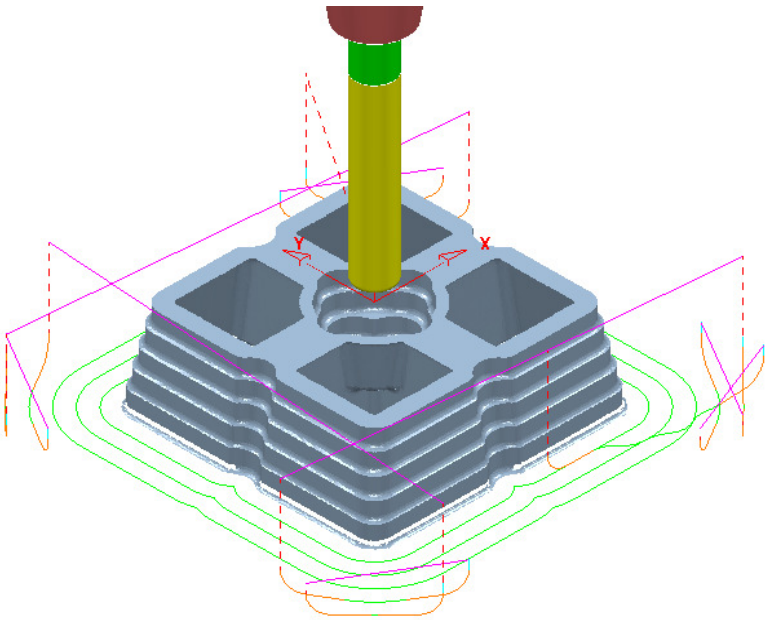



*The latest **Toolpath** - **D20t3-Flats-a1** is automatically added as a reference item in the **Stock Model** list but not as a physical entity.*

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**.

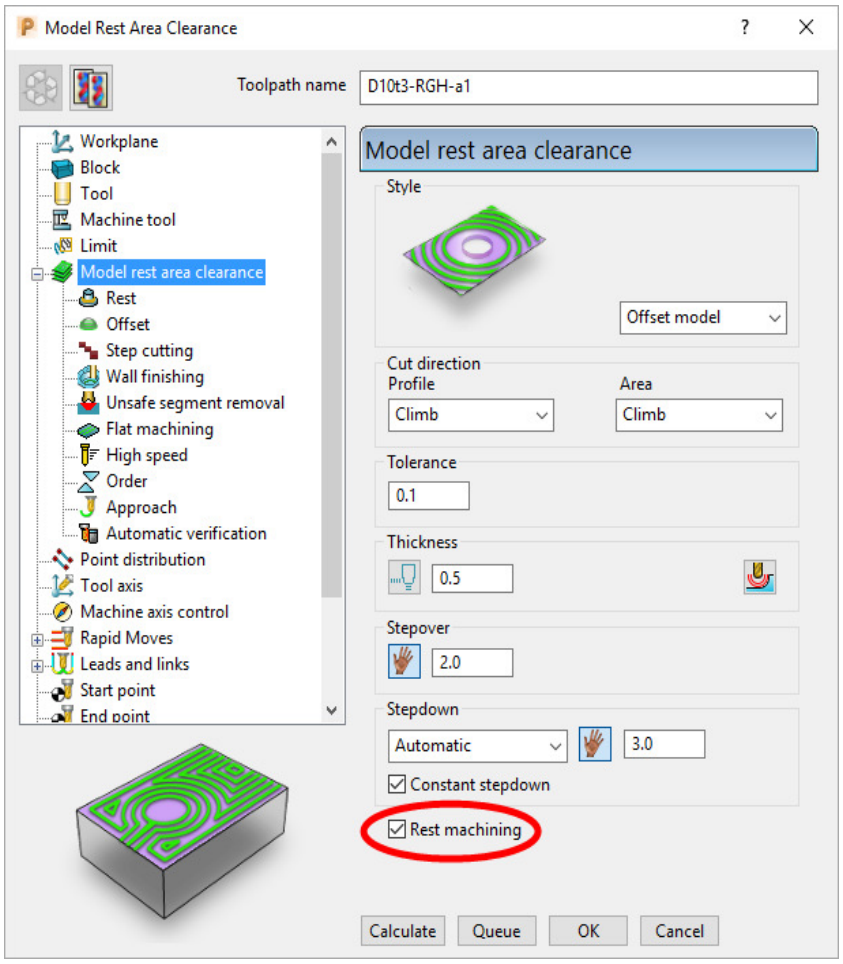



After the calculation, the **Stock Model** will be displayed again only this time with the **Offset Flats Finishing** included.

19 Activate the **toolpath D20t3-RGH-a1** and in the local menu select **Settings...** to open the **Model Area Clearance** dialog.


Copyright © Autodesk

20 Select **Create a new toolpath based on this one** and input new data into the dialog exactly as shown on the following page.

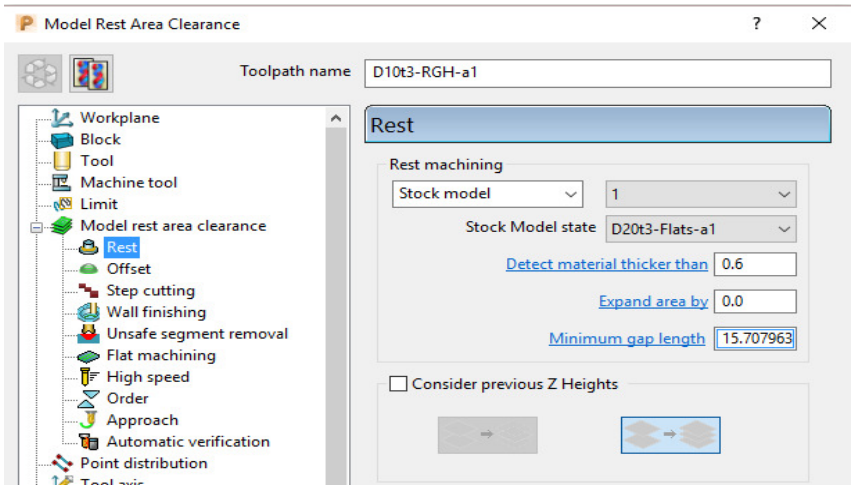



21 Activate the **D10t1** tool.

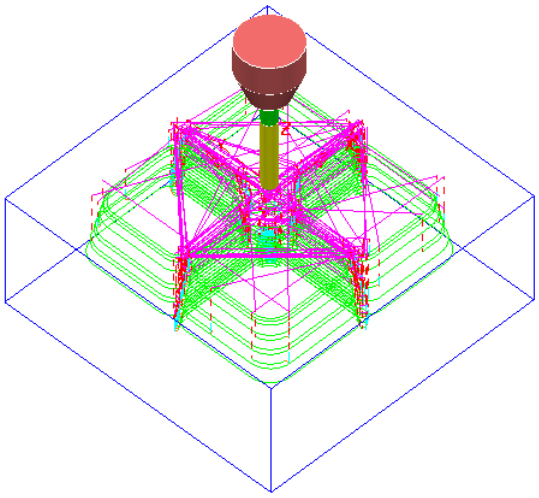
22 Set the ***Style*** as **Offset model**.

Tick the **Rest** machining box.

23 Select the **Rest** page and in ***Rest Machining*** use **Stock Model – 1**.



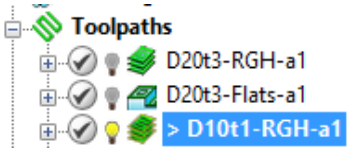
24 Select **Calculate** on the **Model Area Clearance** dialog to machine within the limits of the specified **Stock Model**.



25 Close the **Model Area Clearance** dialog.

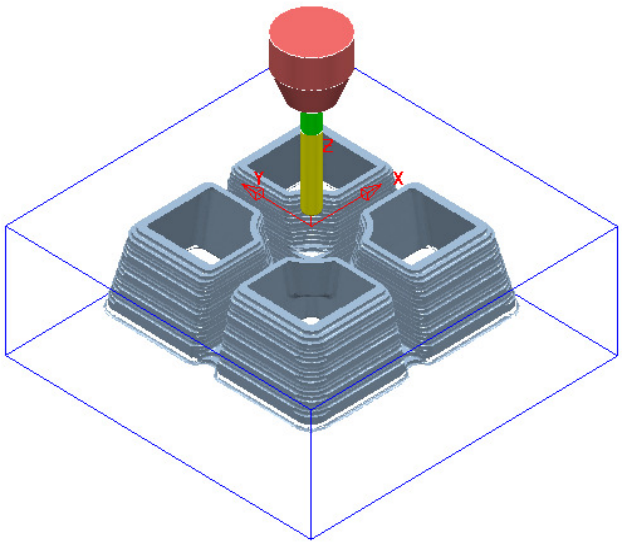
26 The latest **Toolpath** – **D10t1-RGH-a1** is automatically added as a reference item in the **Stock Model** list but not as a physical entity.

27 In the **PowerMill explorer** use the left mouse button to drag the **Toolpath D10t1-RGH-a1** to the bottom of the list.




28 In the **PowerMill *explorer***, right-click over the **Toolpath** – **D10t1-RGH-a1** and in the local menu select **Add to** > **Stock Model**.

29 In the **PowerMill *explorer***, right-click on the **Stock Model** named **1** and in the local menu select **Calculate**.



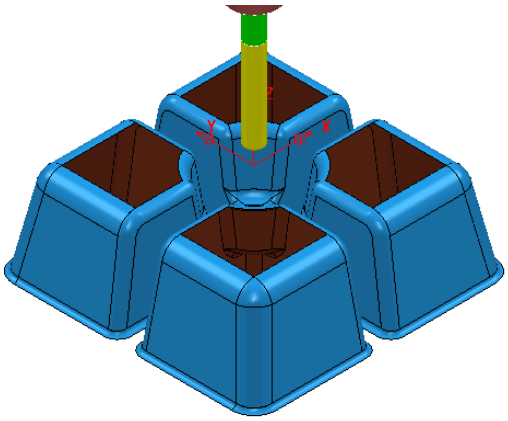
The **Stock Model** is now displayed as shown with the latest toolpath removed.


Copyright © Autodesk

Now that the bulk of the material is removed, the main component form is now ready to be finish machined.

30 Un-draw the **Stock Model**.

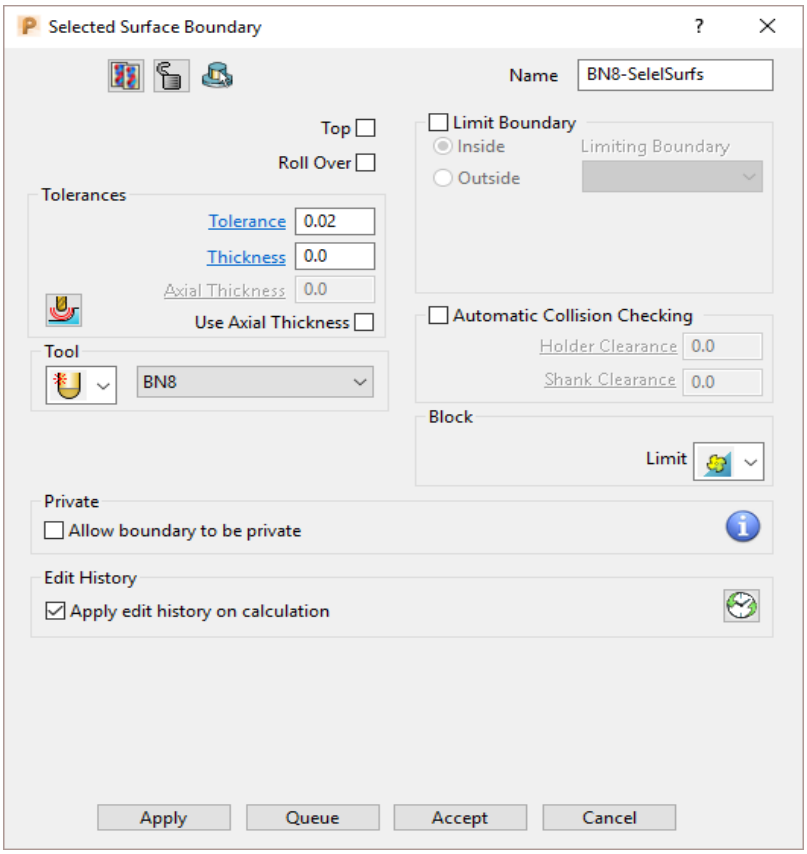
31 **Select** and **Blank** all except the following surfaces:



32 Using the left mouse button, drag a box across the model above to select all displayed surfaces.

33 Activate the tool **BN8**.

34 In the **PowerMill *explorer***, right-click on **Boundaries** and in the local menu, select the **Selected Surface** option.

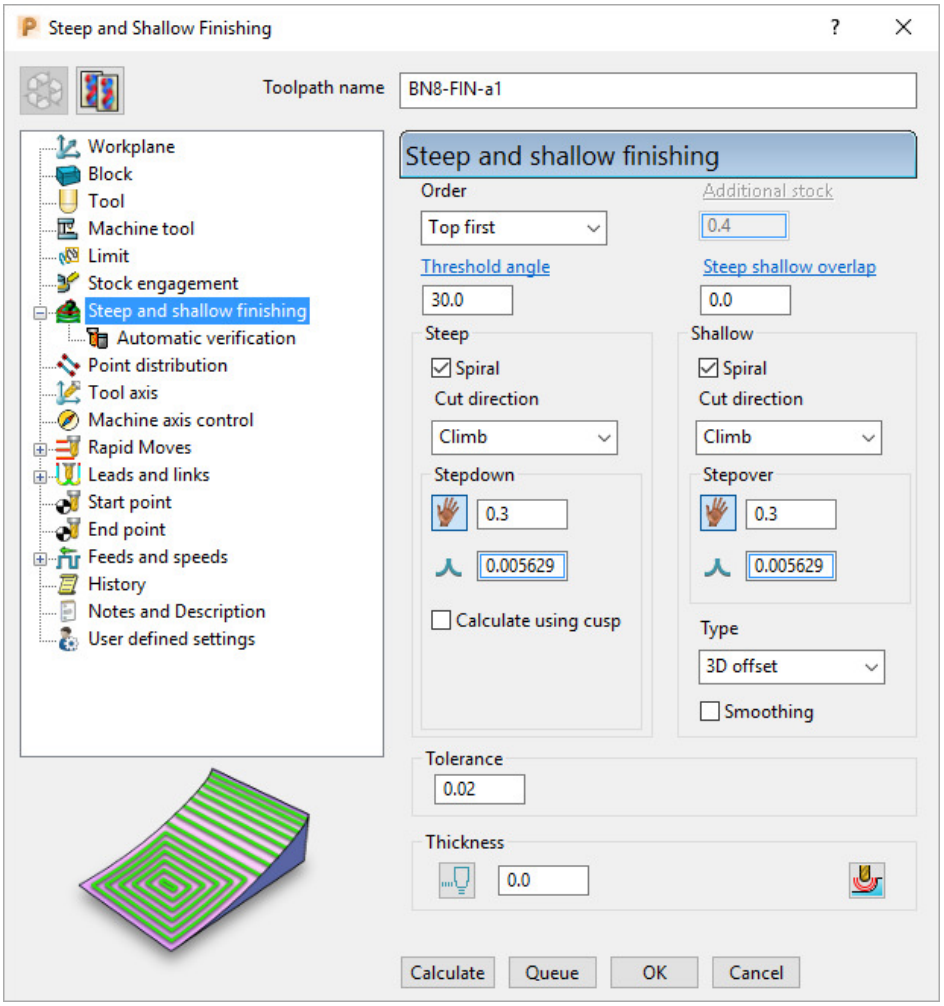



35 Enter data exactly as shown above and then **Apply** to create the **Boundary**.

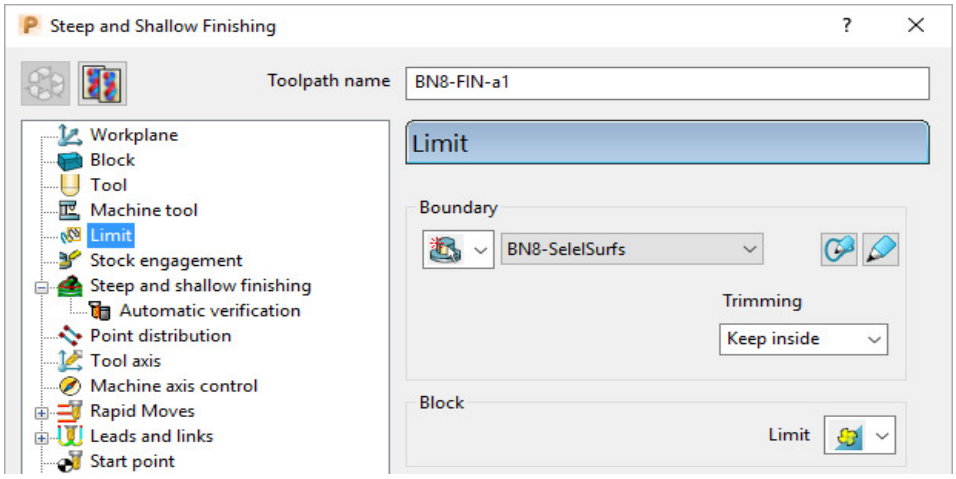
36 Check that both **Boundary 1** and the **BN8** tool are both **Active**.

37 Select the **Toolpath Strategies** and open the **Finishing** page.

38 Select the **Steep and Shallow Finishing** option and enter data into the dialog exactly as shown below:




39 Select the **Limit** page and enter data exactly as shown in the following image:

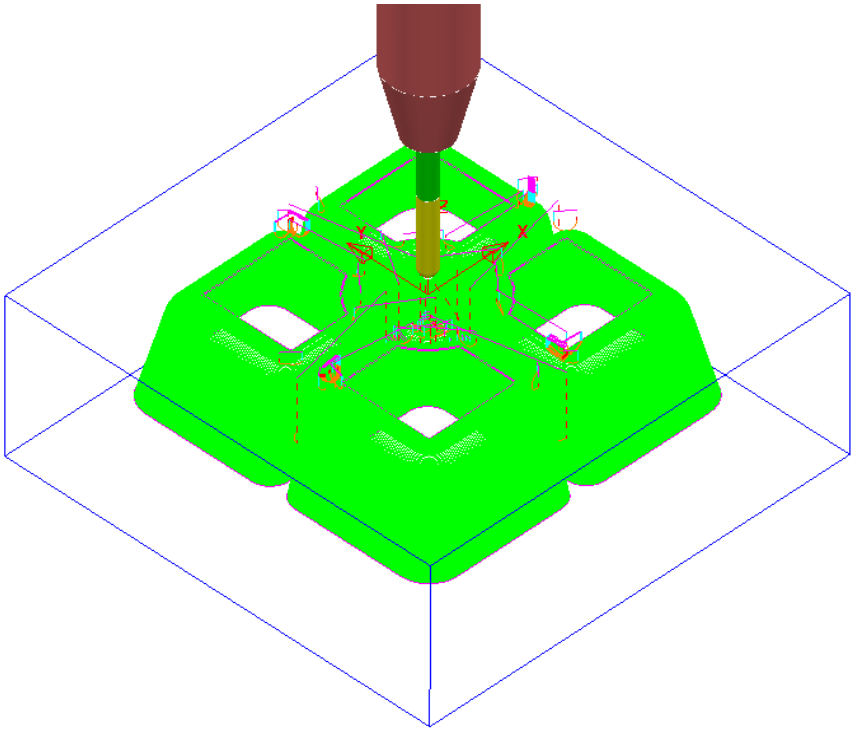



40 Select **Calculate** to create the **Toolpath** and then **Close** the dialog.


Copyright © Autodesk

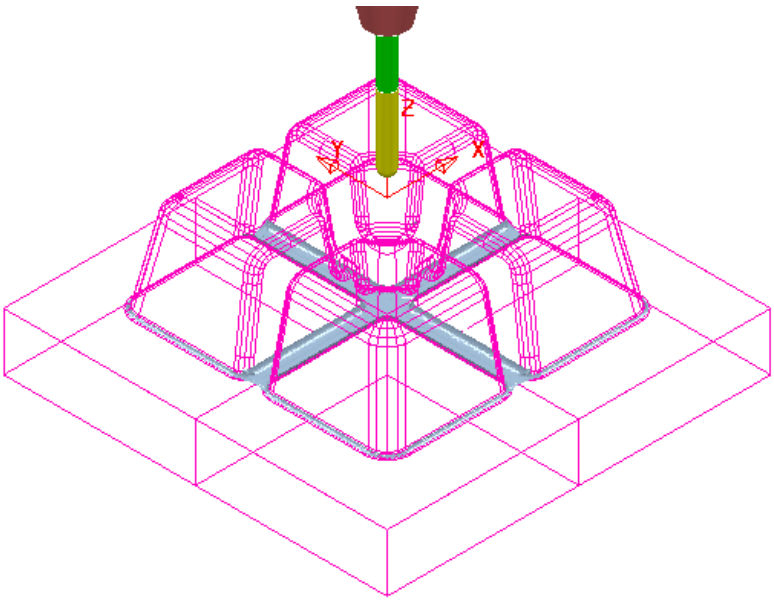
41 In the **Leads and Links** options **Apply** (1st Choice) **Lead in/out** as **Surface Normal Arc** with **Angle 90** and **Radius 4**.

*The **Toolpath BN8-FIN-a1** has been limited to be inside the **Boundary**.*



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.



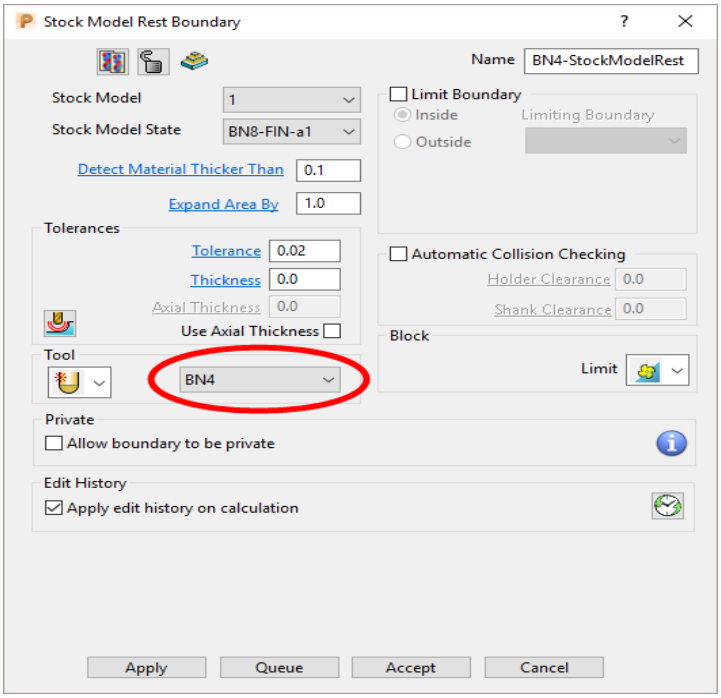
The remaining **Stock Model** is displayed as shown above.

# Stock Model Rest Boundary

Most of the **Finishing** strategies do not have a direct **Rest Machining** option that relates to the **Stock Model**.

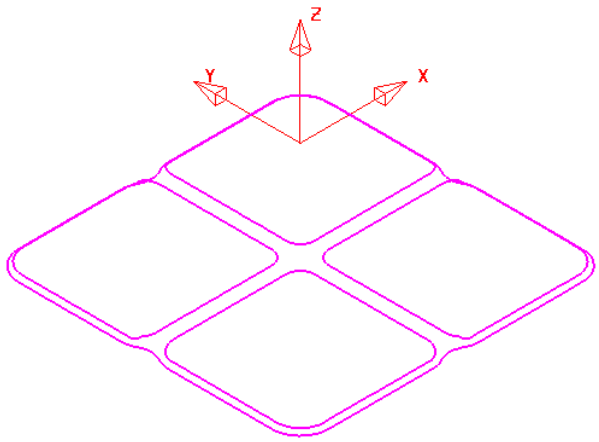
It is however possible to create a **Stock Model Rest Boundary** to define the areas with material left on. This can then be used to limit a finishing strategy such that only the remaining stock is removed.

1 In the **PowerMill *explorer***, right-click on **Boundaries** and in the local menu select **Stock Model Rest**.




2 Enter data exactly as shown in the above image and **Apply** and then **Cancel**.

***Boundary*** *segments will only be created where the* ***BN4*** *tool is able to access the model form.*



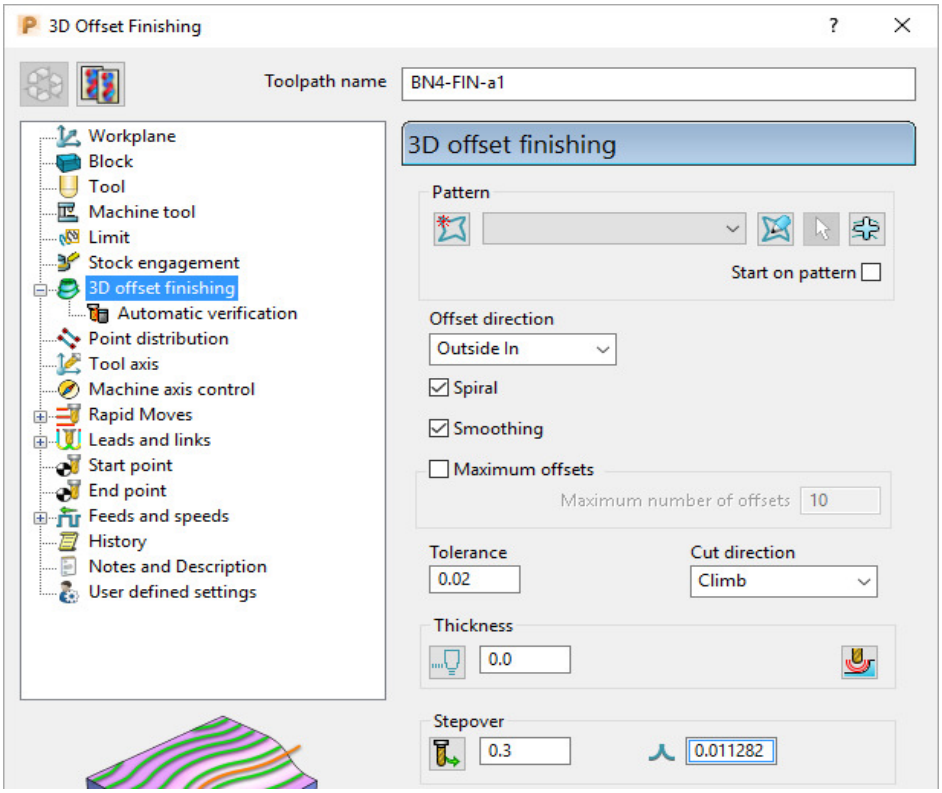

The **Stock Model Rest Boundary** identifies the area of the remaining **Stock Model**.


Copyright © Autodesk

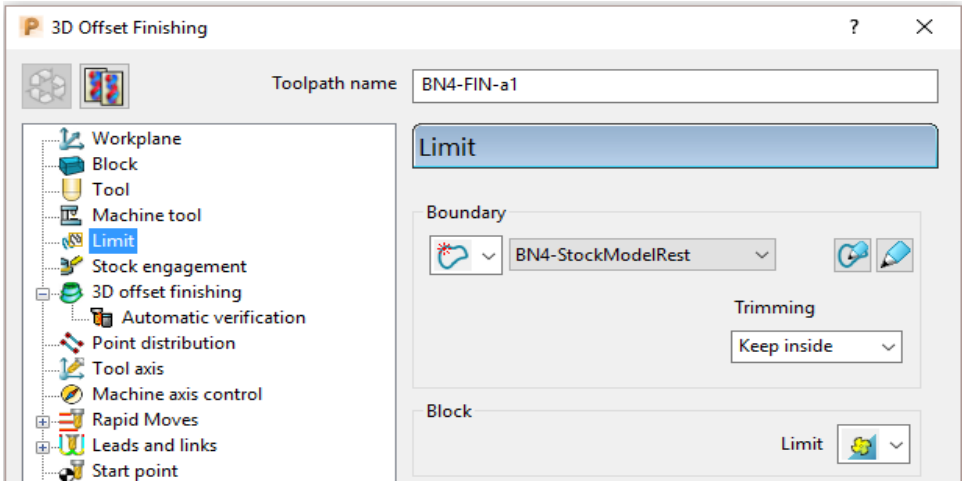
3 Select the **Toolpath Strategies** and select the **Finishing** tab in the dialog.

4 Check that the new **Boundary BN4-StockModelRest** and the **BN4** tool are both **Active**.

5 Select the **3D Offset Finishing** strategy and enter data exactly as shown below:

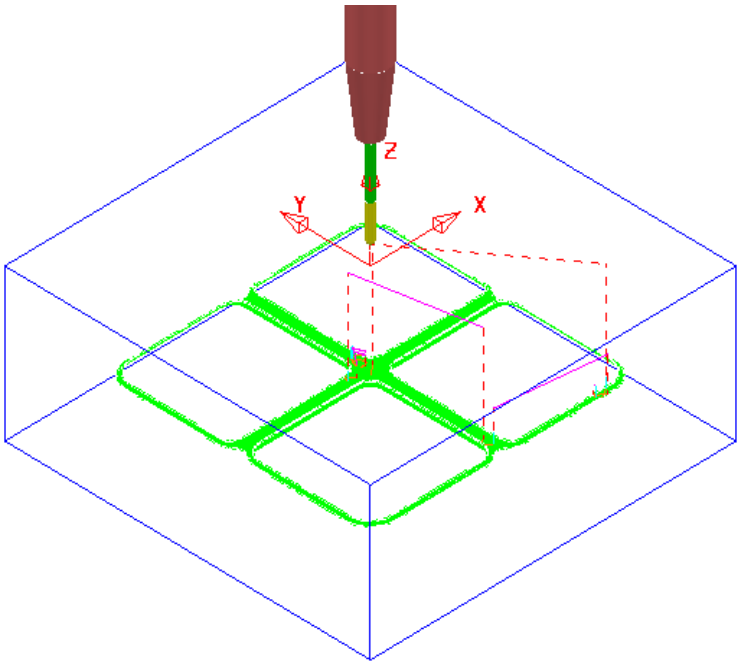



6 Select the **Limit** page and enter data exactly as shown in the following image:




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

8 In the **Leads and Links** options **Apply** (1st Choice) **Lead in/out** as **Surface Normal Arc** with **Angle 90** and **Radius 3**.



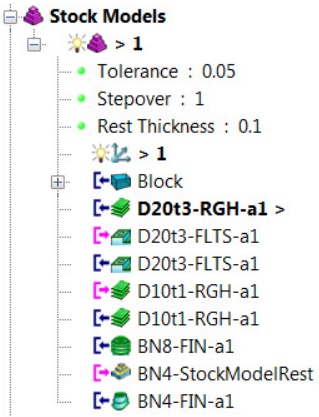

9 In the **PowerMill *explorer***, right-click the **toolpath BN4-FIN-a1** and in the local menu select **Add to** > **Stock Model**.

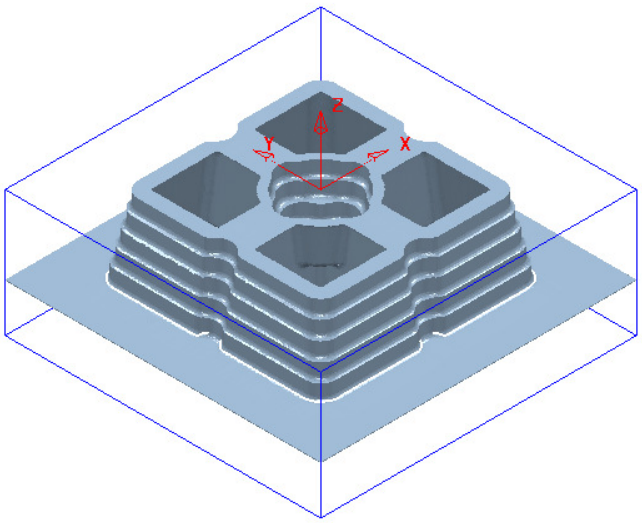
10 In the **PowerMill *explorer***, right-click on the **Stock Model** and select **Calculate** to update.

*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**.



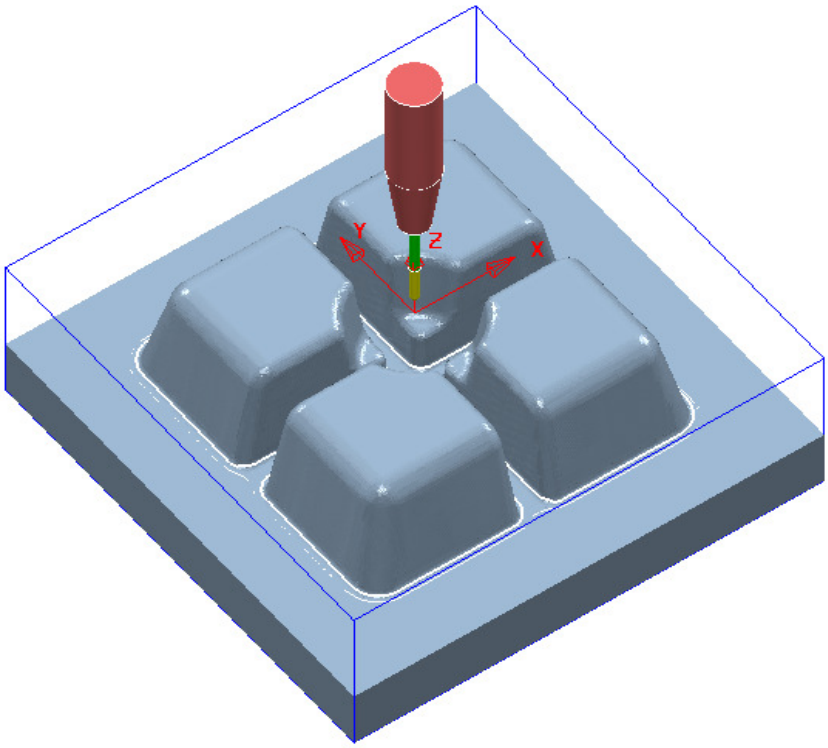


Copyright © Autodesk

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.

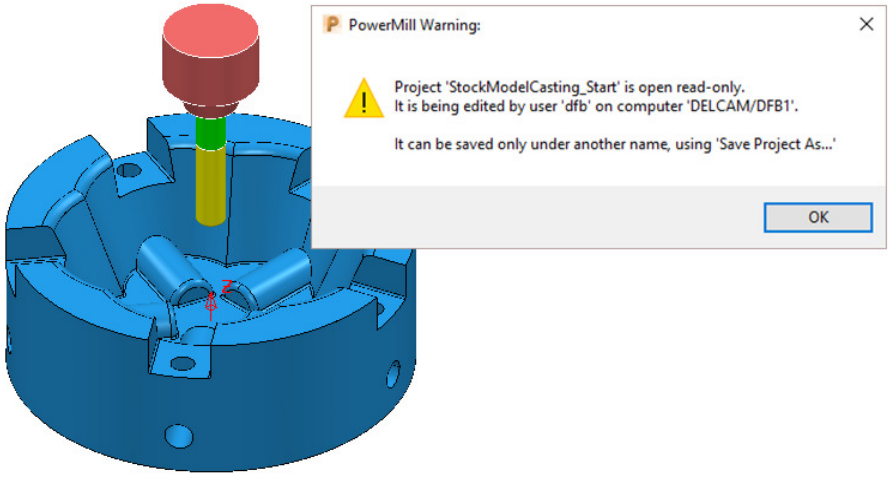
If a casting is to be machined then it is not possible to directly add the complex shape to a **Stock Model**. It is however possible to import a **Triangle** model of the original casting as the initial **Block** and then use it as the first entity in a **Stock Model**. The **Stock Model** can then 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**:

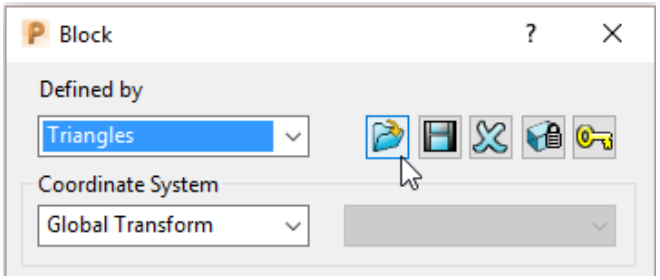
**.....\PowerMill_Data\Projects\StockModelCasting_Start**




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

**.....\ COURSEWORK\PowerMill-Projects\StockModelCasting**

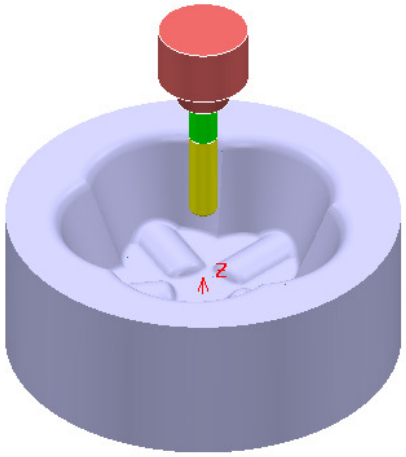
4 Open the **Block** form and set ***Coordinate System*** to **Global Transform**.




5 Select ***Defined by*** – **Triangles**, select and browse to the triangle model:-

**C:\Training_Data\PowerMill_Data\Triangles\StockModel-Casting.dmt**


Copyright © Autodesk

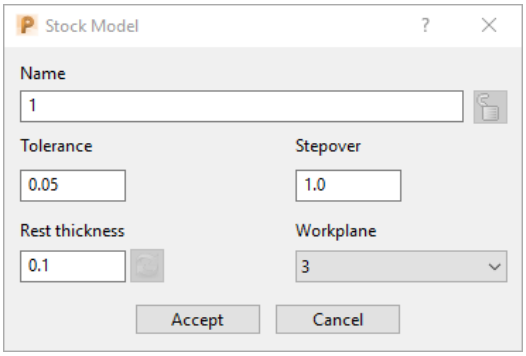



6 Open the **Rapid Move Heights** form and select **Calculate**.

7 **Activate** the tool **D25TR3**.

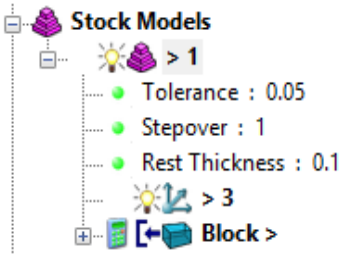
8 Create a new **Stock Model**.

9 Right mouse click on the new **Stock Model** and from the local menu select **Settings**.




10 Edit the dialogue with above values and **Accept**.

11 Right mouse click on the new **Stock Model** and from the local menu select **Apply Block**.



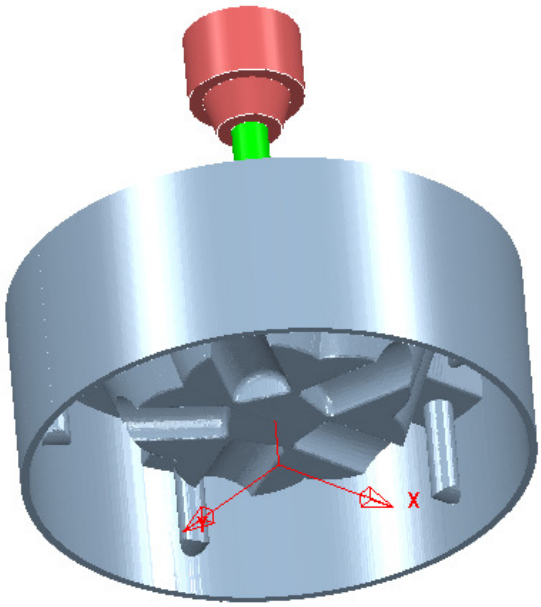

The complex **Block** is added as the first entity in the **Stock Model**.

12 Right mouse click on the new **Stock Model** and from the local menu select **Calculate**.

13 Undraw the Block.

14 Right mouse click on the new **Stock Model** and from the local menu select **Drawing Options** and tick both **Shaded** and **Show Rest Material**.

Copyright © Autodesk

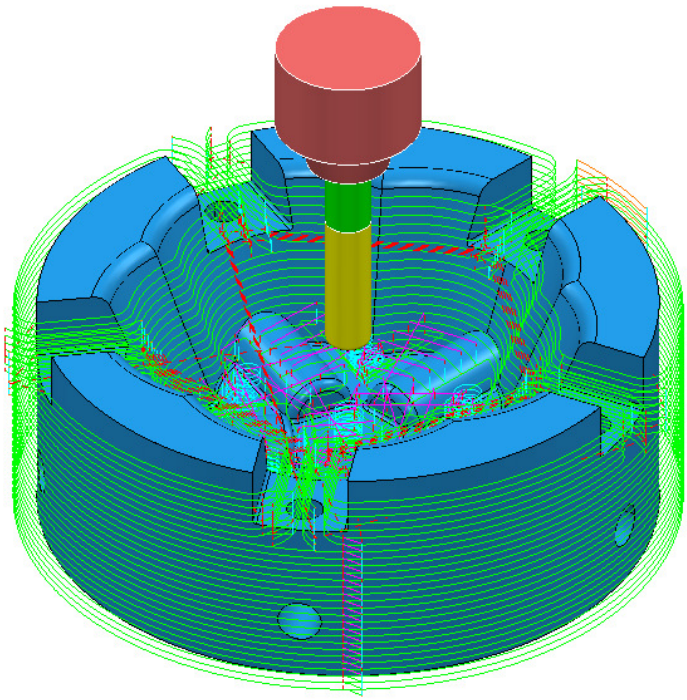



The **model** form is visually removed from the **Stock Model**.

## Exercise:

15 **Calculate a Block** of ***Type*** - **Cylinder** and ***Defined by*** - **Model**.

16 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**.


Copyright © Autodesk

# 18. Macros & User Menu

## Introduction

There are various ways in which the user can customise **PowerMill**. This chapter looks at the creation of:

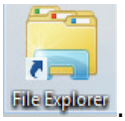
- **Macros**
- **User Menus**

## 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:-

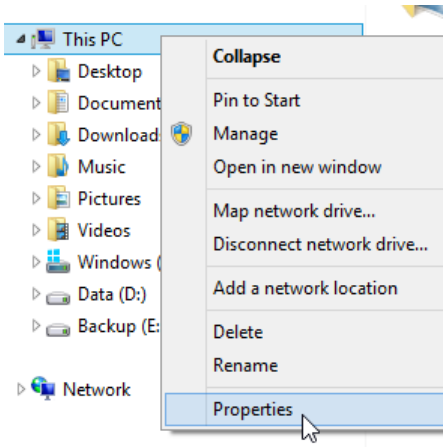
**C:**

To check if a **Home** area already exists on a computer;

1 Open the windows **File Explorer** 

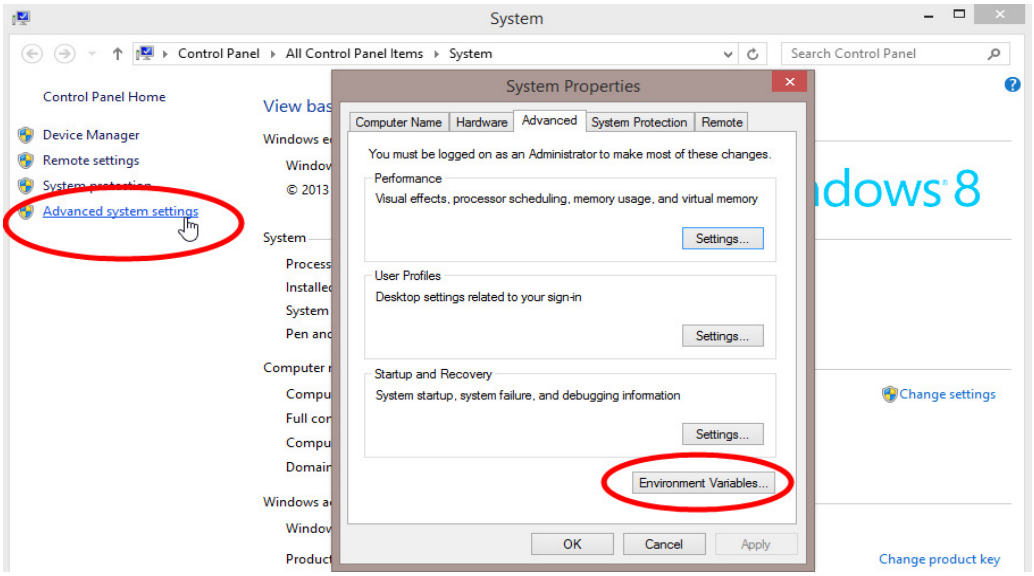
.

2 In the local explorer, right-click on This **PC** and select **Properties** to open the **System** dialog.

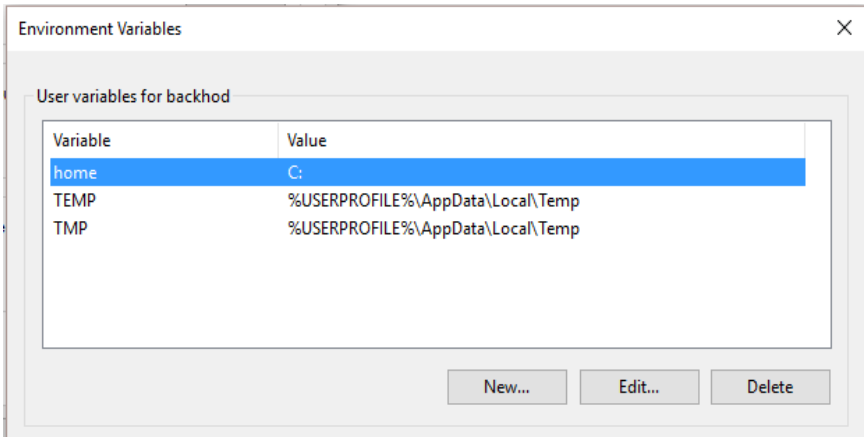


*Alternatively, at the lower left corner of the screen, right mouse click on* ***Start*** *and from the local menu select* ***System****.*

3 In the **System** dialog select the **Advanced system settings** option and then in the **System Properties** dialog, select **Environment Variables**.




4 If the **home** area is not already defined as shown below, select New…




5 In the **User Variables** dialog, enter **home** in the **Variable** name field and **C:** in the **Value** field.

6 Click **OK**.

*You must have Admin rights to create a* ***New User Variable****.*

7 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* ***macros*** *are being stored in this folder, then* ***PowerMill*** *will have immediate access through the default* ***Macro*** *search paths.*


Copyright © Autodesk

# 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**.

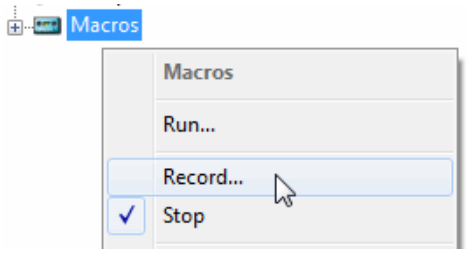
When **PowerMill** starts up, it will attempt to identify and run the **initialisation macro** (if it exists!) called **pmuser.mac**. By adding customised **PowerMill** commands to the initialisation macro, the user can set up their own default settings and parameters.

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.

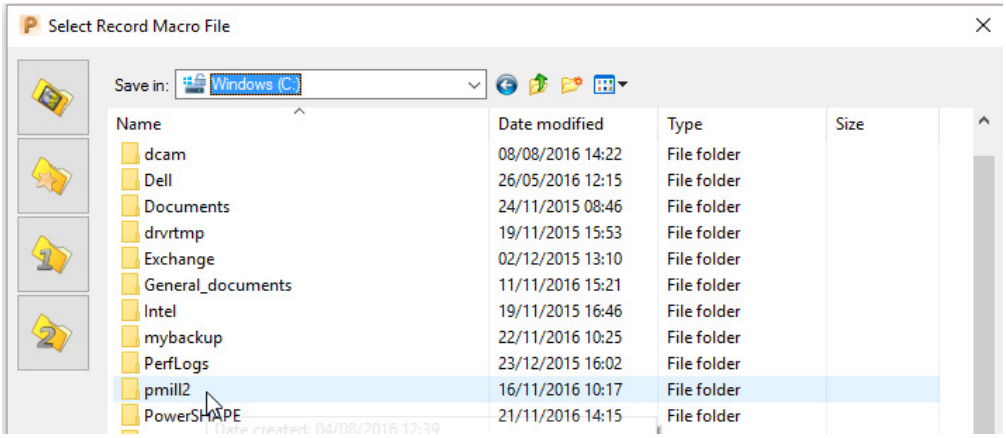
## Creating a pmuser macro

**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.

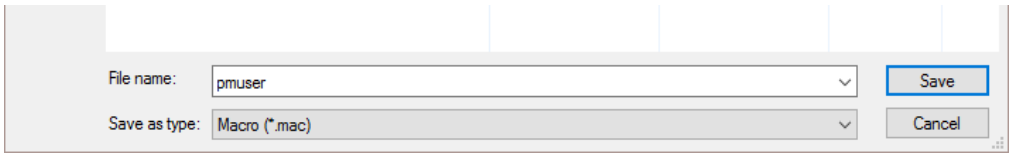
1. Select **File** > **Delete All**.
2. To start recording, right-click on Macros in the **PowerMill *explorer*** and select **Record**.




3. Save **pmuser.mac** by first clicking on the existing folder displayed in **Save in**.




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**.



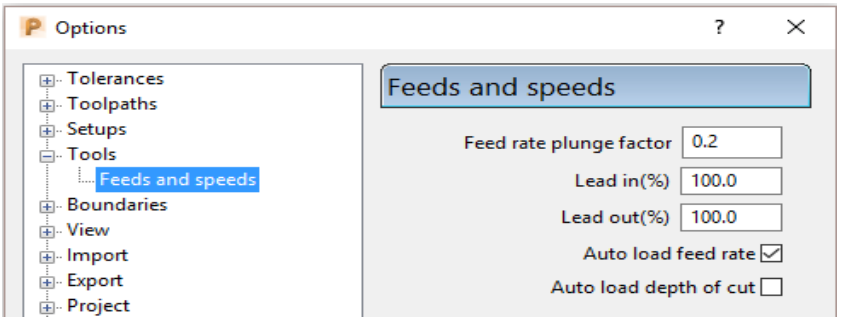

*The Macro icon changes to red while recording is in progress*

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**.

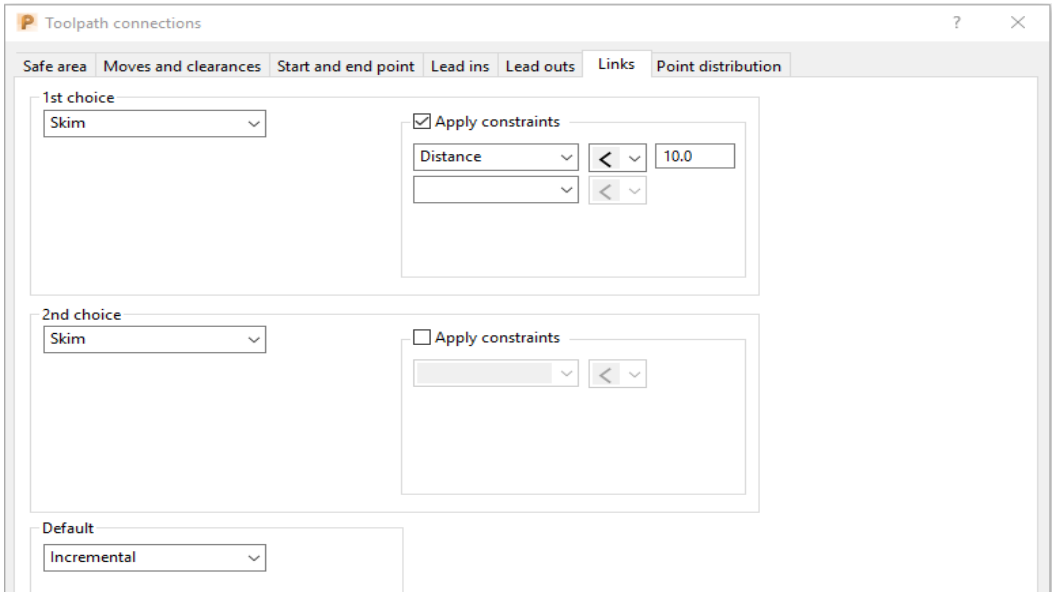
*All required data or options must be set or selected if they are to be included in a macro (Even if it is necessary to overwrite or switch them off first).*

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.



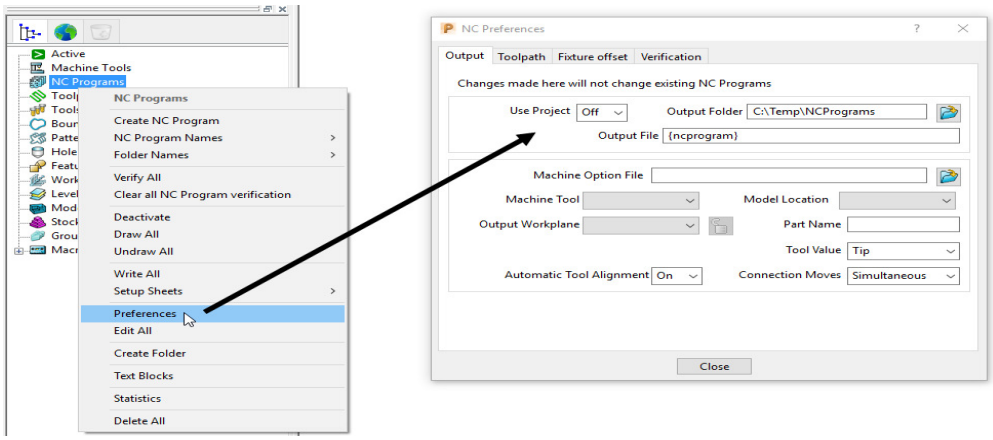

7 Open the **Leads and Links** dialog and input both Short and Long as **Skim**, and Default as **Incremental**.




8 Accept the **Leads and Links** dialog.


Copyright © Autodesk

9 In the **PowerMill *explorer***, right-click on **NC Programs**, and select **Preferences**.




10 On the **NC Preferences** dialog **Output** page, select the **Machine Option File** as **Heidenhain_3axis_ref.pmoptz**.

11 Even if already set, select:

- Use Project - **On**
- Tool Value - **Tip**
- Automatic Tool Alignment - **On**
- Connection Moves - **Move**, **Rotate**

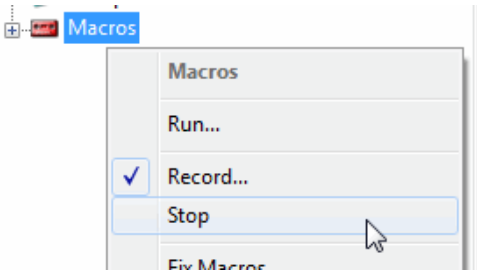
12 Select the **Toolpath** page and even if already set, select:-

- Tool Change - **On New Tool**
- Tool Numbering - **As Specified**
- Tool Change Position - **After Connection**
- Cutter Compensation: Length - **Off**
- Radius - **None**

13 To save and exit the dialogue select **Close**.

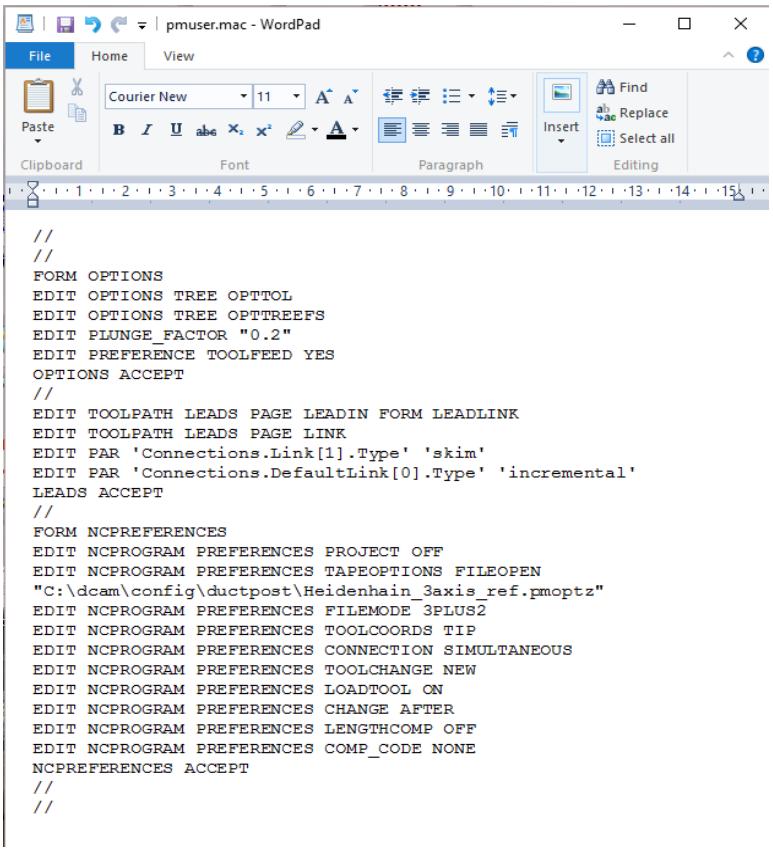
*It is important to remember to Accept and not to Apply forms that are opened while creating an Initialisation Macro, the only exception being the creation of NC Program Preferences (An initialisation file is for pre-setting forms but at this stage without executing the commands).*

14 Select **Stop** from the local **Macros** menu to stop the recording process.




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 prove this is running during **PowerMIll** start up 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 **Fanuc3Axis_prefs** and save it in:-

**C:\pmill2\NCpreferences\...**

2 In the **NC Programs** > **Preferences** > **Output** tab select the **Machine Option File** as **Fanuc_3axis_ref.pmoptz**.

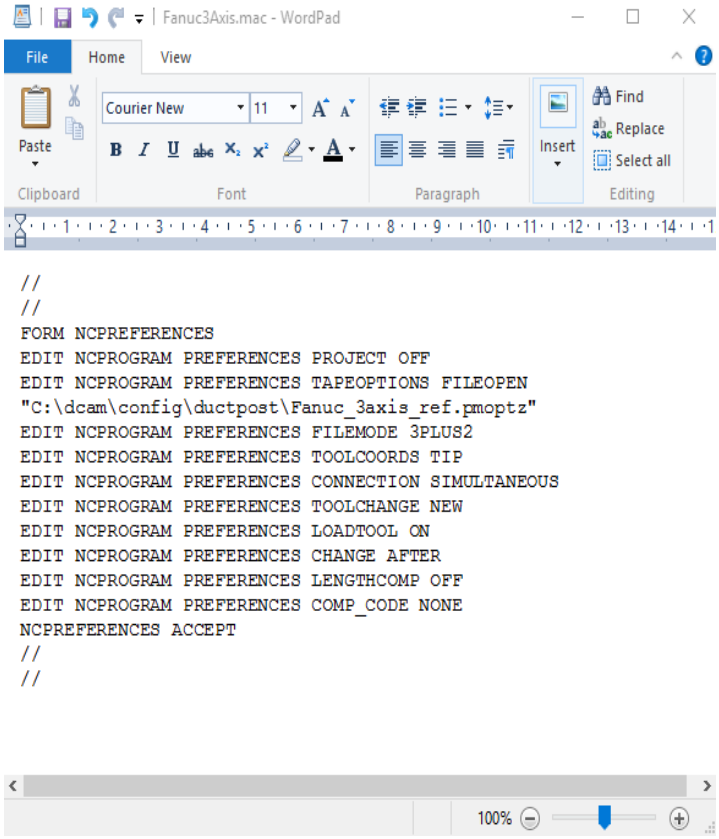
3 Input the remaining settings exactly as applied earlier for the **Heidenhain_3axis_ref.pmoptz** in the **pmuser** macro.

4 Stop the **Macro** recording process.


Copyright © Autodesk

*Even if already set it is essential to re-specify settings if they are to be recorded into the macro.*

A macro file (**Fanuc3Axis_prefs.mac**) is now created that sets up **NC Preferences** for a post-processor option file and the command lines are shown below.




```
//
//
FORM NCPREFERENCES
EDIT NCPROGRAM PREFERENCES PROJECT OFF
EDIT NCPROGRAM PREFERENCES TAPEOPTIONS FILEOPEN
"C:\dcam\config\ductpost\Fanuc_3axis_ref.pmoptz"
EDIT NCPROGRAM PREFERENCES FILEMODE 3PLUS2
EDIT NCPROGRAM PREFERENCES TOOLCOORDS TIP
EDIT NCPROGRAM PREFERENCES CONNECTION SIMULTANEOUS
EDIT NCPROGRAM PREFERENCES TOOLCHANGE NEW
EDIT NCPROGRAM PREFERENCES LOADTOOL ON
EDIT NCPROGRAM PREFERENCES CHANGE AFTER
EDIT NCPROGRAM PREFERENCES LENGTHCOMP OFF
EDIT NCPROGRAM PREFERENCES COMP_CODE NONE
NCPREFERENCES ACCEPT
//
//
```

# Exercise

1. Create two more independent **NC Preference** macros for the Heidenhain 3 Axis and Mazak 3 Axis option files (Use the filenames **Heid3Axis_prefs.mac** and **Mazak3Axis_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.

1 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).

```
# ************** USER MENU ***************
# comment lines
# this file must be called user_menu and be in your HOME area

# This file must be called user_menu and must be located in a directory named
# pmill2 below the user's HOME area.
# T is for Title and cannot be selected.
# S is for separator or line on the menu.
# I is a command line containing a "menu instruction" - code no. - "action".
# M is start new menu/sub-menu.
# Z is end menu/sub-menu.
# comment lines.
# format is I "Label String" 1 "Command string"

U user_menu
T "[My Menu]"
S
```

A command line will be added to the above user_menu to run the **Heid3Axis_prefs** macro.

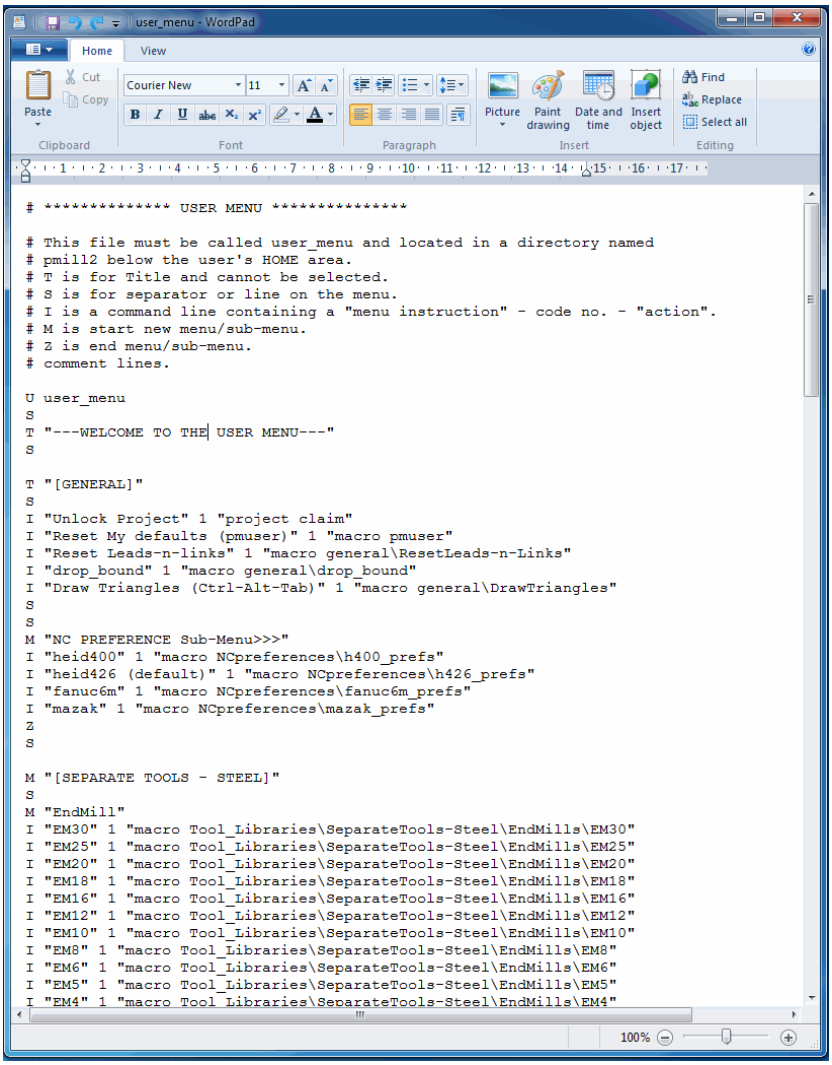
```
I "Heid3Axis" 1 "macro NCpreferences\Heid3Axis_prefs"
```

3 Add the above command line to run the **Heid3Axis_prefs** macro directly from the **user_menu**.

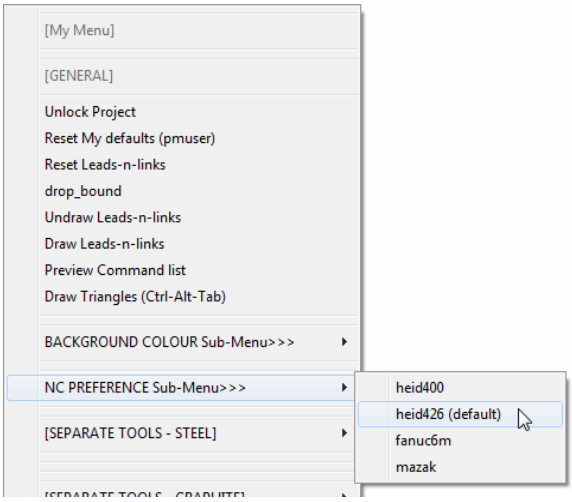
This line locates and runs the file **Heid3Axis_prefs.mac** created earlier in this chapter. It needs to include the correct drive and path where the macro is located. All lettering must be in the correct upper or lower case and include the inverted commas and hyphens.

A more complete version of a typical **user_menu** is as shown below:


Copyright © Autodesk




This is the **user_menu** file as displayed in the **Wordpad**.




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.

Copyright © Autodesk

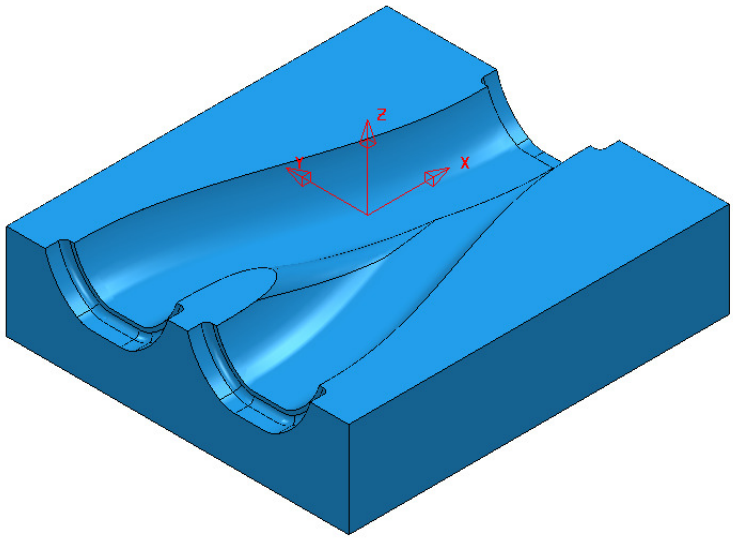
# 19. Tutorials

## 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 **NC Program**.

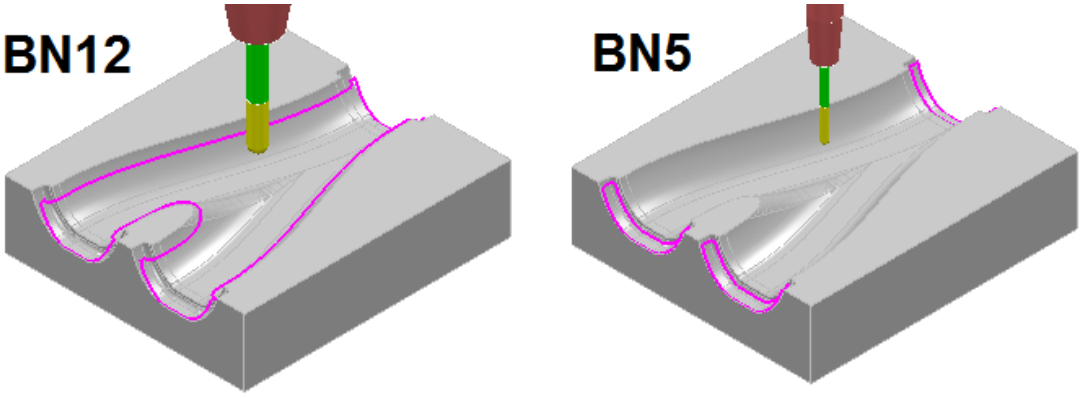
*Newly processed Toolpaths are automatically added to an Active NC Program.*

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).



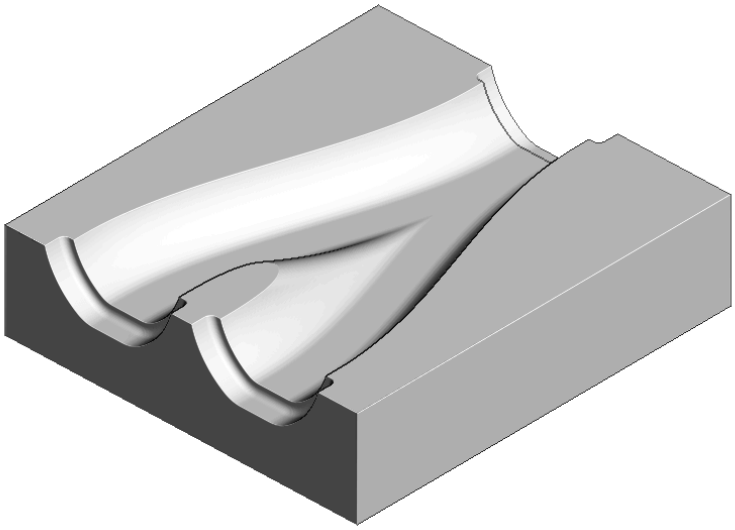

10 Activate the **Tool BN12**.

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**.

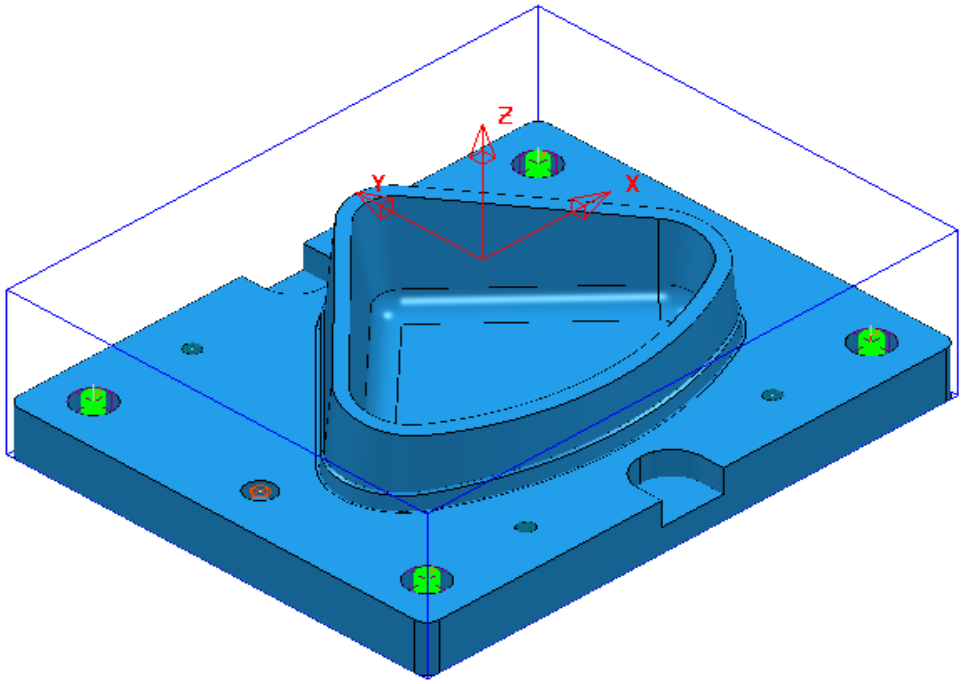

Copyright © Autodesk

# 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-CnrBwl-All**.

*The New **NC Program** will be **Active** on creation. Newly processed toolpaths are automatically added to an **Active NC Program**.*

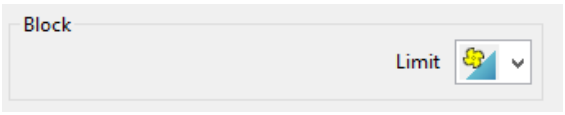
6 Open the **Block** dialog and with **Defined by *Box*** and **Type *Model*** set select **Calculate**.

7 In the **Rapid Move Heights** dialog enter the **Workplane MC-Datum-OP2** and **Calculate** before selecting **Accept**.

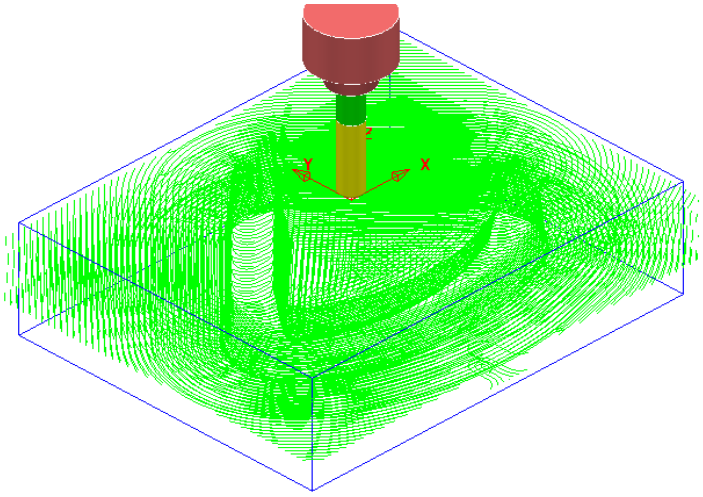
Copyright © Autodesk

# 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** .




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 **Step-up 2**, **Detect material thicker than 0**, and **Expand area by 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 both the **Lead in/out** pages set ***1st choice*** as **Vertical arc** with **Distance 15**, **Angle 90**, and **Radius 8**.
12. For **Lead in**, ***2nd choice*** select **Ramp**, and in the **Ramp options** dialog set Follow **Circle**, **Max zig angle 4**, and **Circle Diameter (TDU) 0.65**.
13. **Accept** the **Leads and Links** dialog.
14. Select **Calculate** in the main **Area Clearance** dialog to create the toolpath.
15. Close the **Area Clearance** dialog.



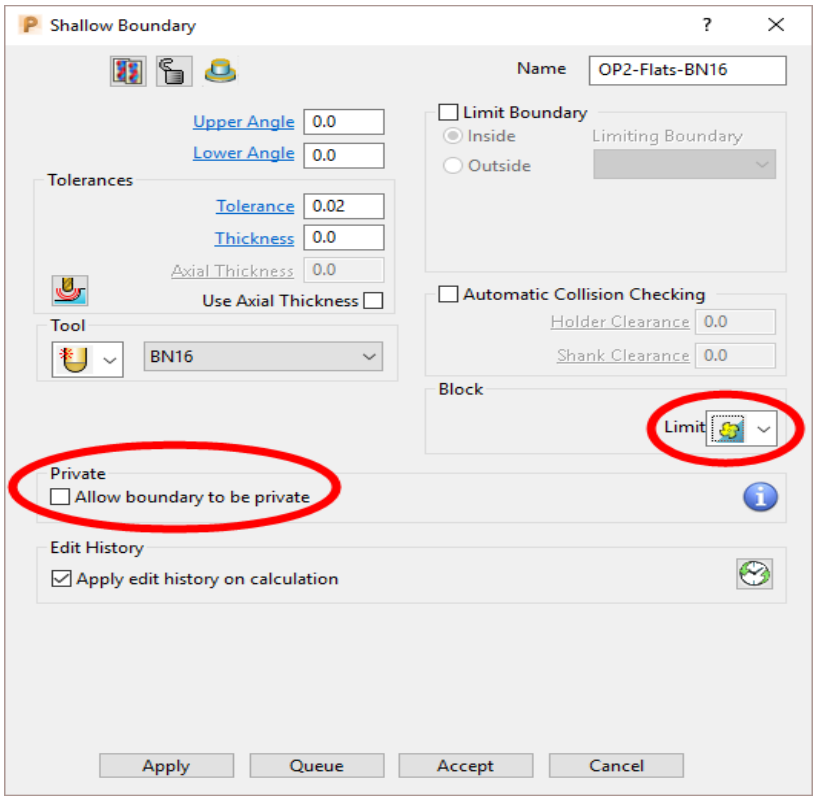
*The **toolpath** is automatically added to the **Active NC Program** list.*


Copyright © Autodesk

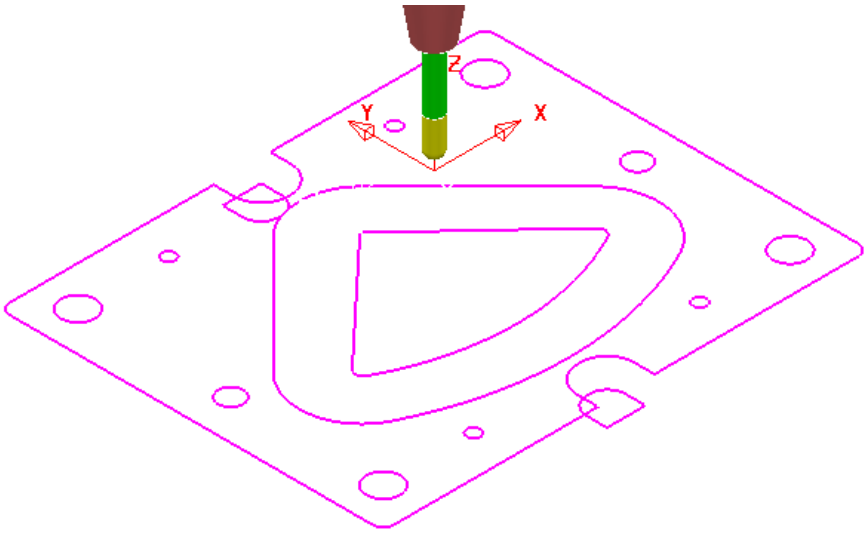
# Finish Machining – Tool BN16

1 **Activate** the ***Tool*** **BN16**.

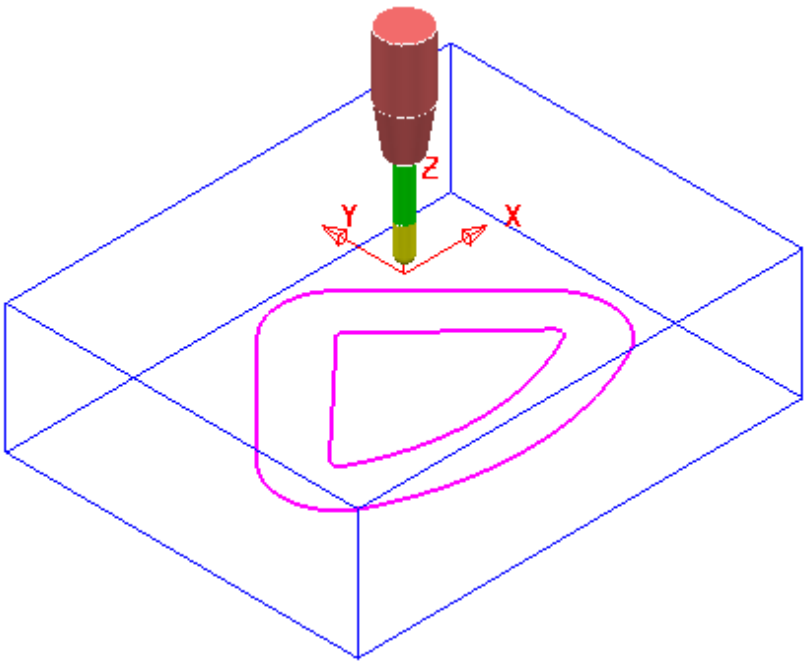
2 In the **PowerMill *explorer*** select **Boundaries** – **Create Boundary** – **Shallow**.




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**.

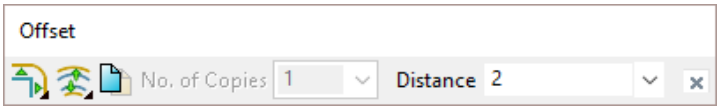



4 Select and **Delete** all except the 2 innermost **Boundary** segments.




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** (2D round) of **2**.




7 Select the inner **Boundary segment** and apply an **Offset** (2D round) 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**.

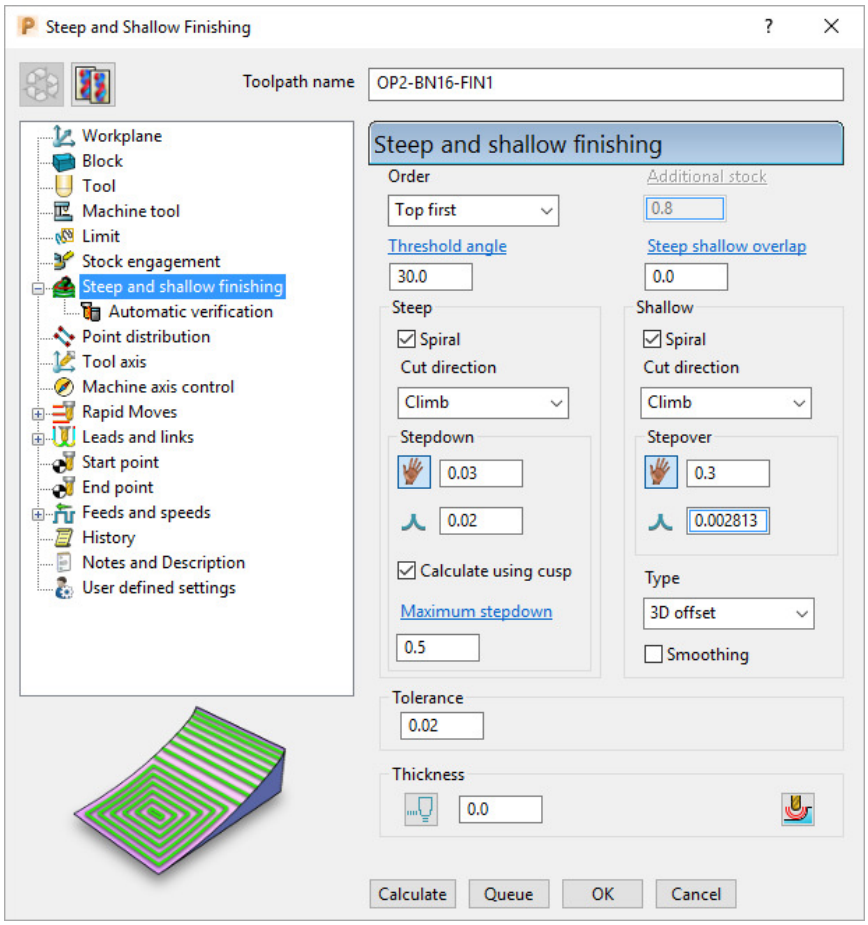
9 From the **Strategy Selector**, open the **Finishing** dialog and select a **Steep and Shallow Finishing** strategy.

10 In the **Steep and 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.

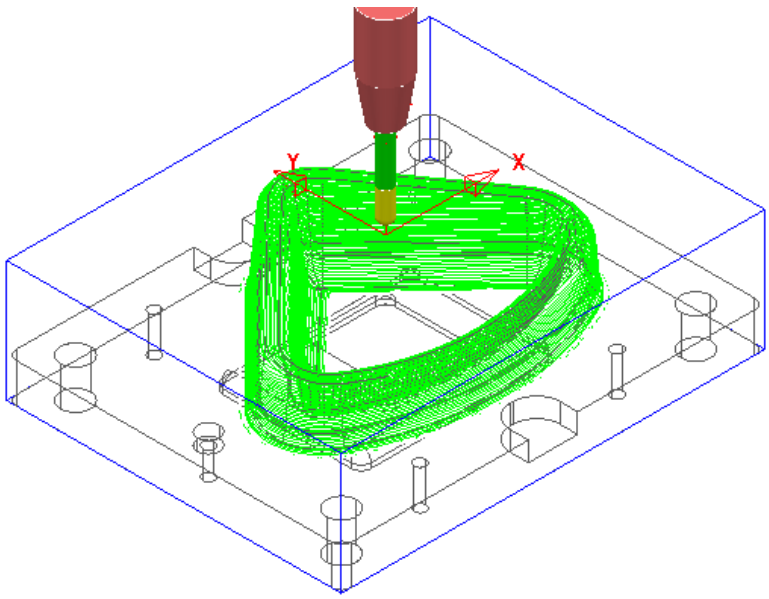

Copyright © Autodesk




13 For the **Leads** select **Lead in/out *1st choice*** set **Vertical arc** with **Distance 0**, **Angle 90**, and **Radius 8**.

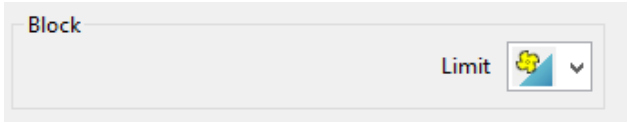
14 Select **Calculate** in the main **Steep and Shallow finishing** dialog to create the toolpath.

15 **Close** the **Steep and shallow finishing** dialog.

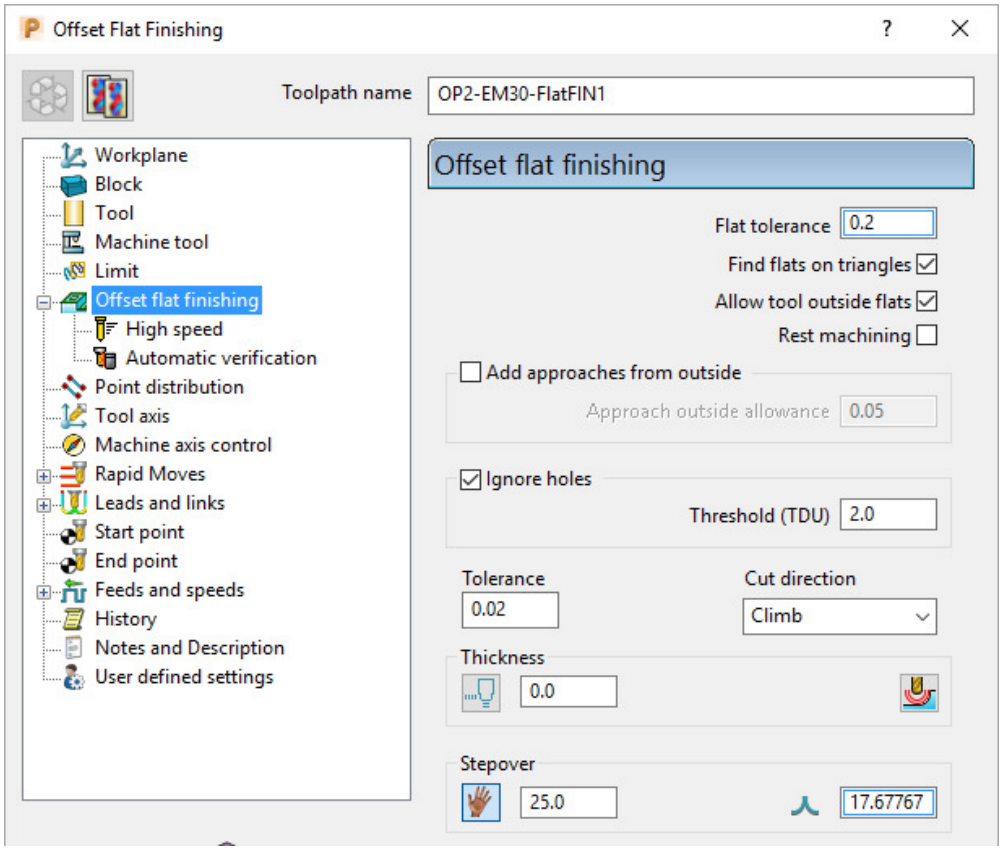


# Finish Machining – Tool EM30

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**.
5. In the **Limit** page, select ***Limit tool centre to block edge*** .

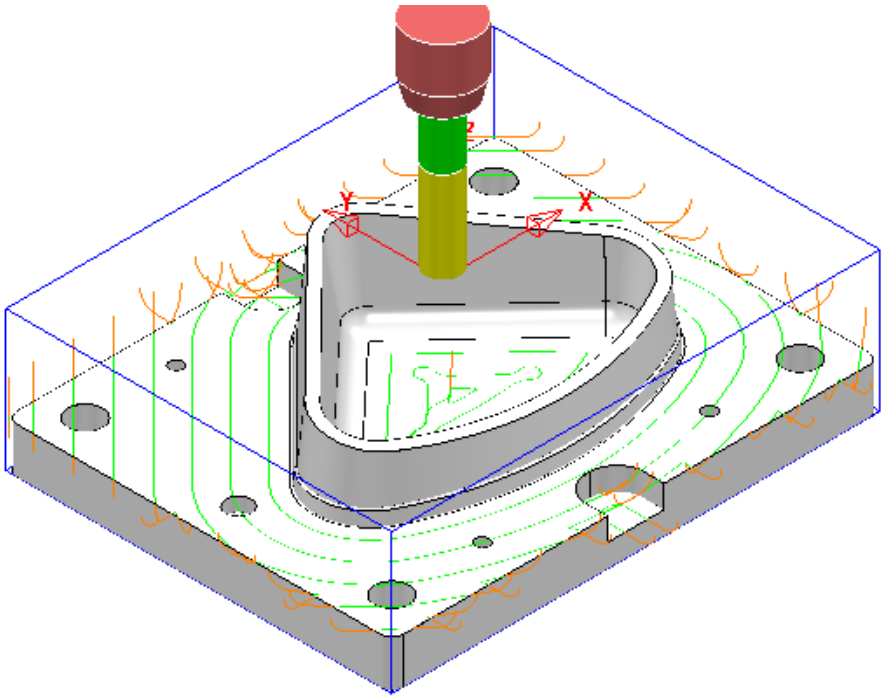



6. In the ***Offset flat finishing*** page, enter data exactly as shown below (Note the ***tick box*** settings!).




7. In the main **Leads and Links** dialog select **Lead in/out** ***1st choice*** set **Vertical arc** with **Distance 15**, **Angle 90**, and **Radius 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.

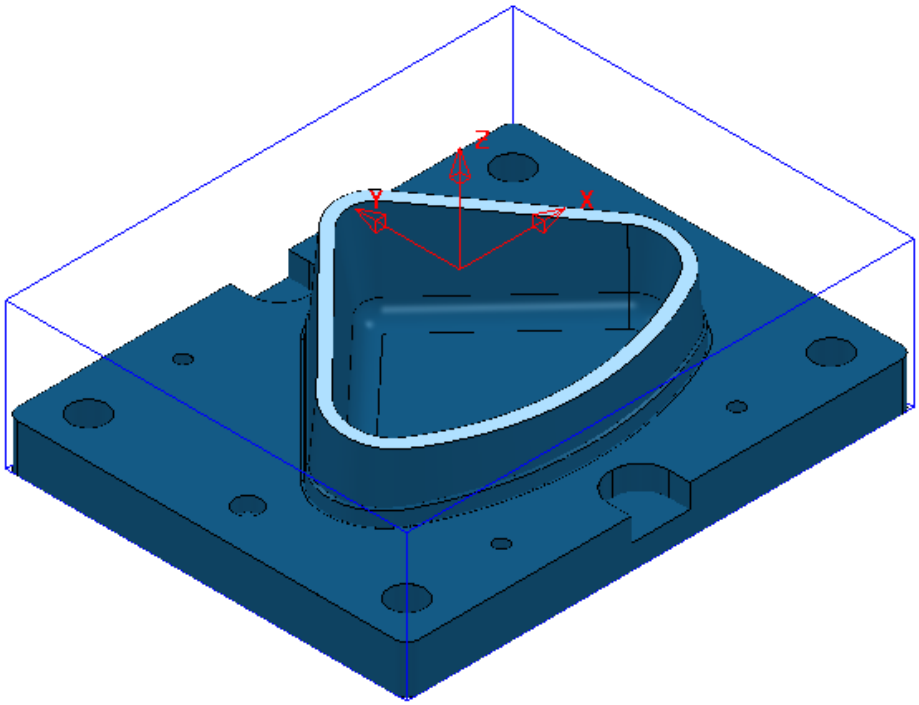

Copyright © Autodesk



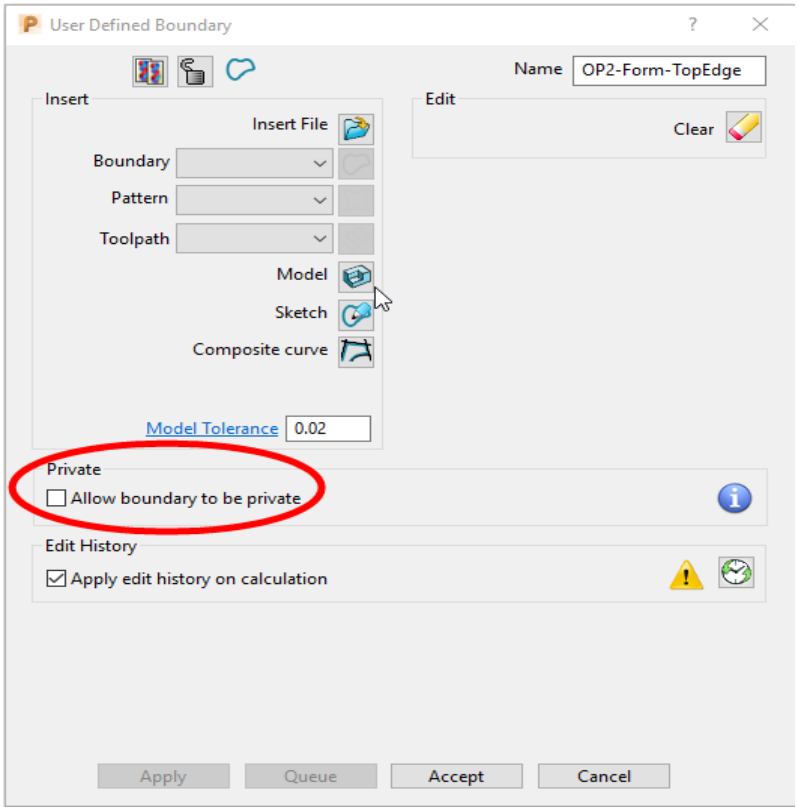
## Finish Machining – Tool D16TR3

1 **Activate** the tool **D16TR3**.

2 Select the upper edge of the **Surface Model** 3D form.

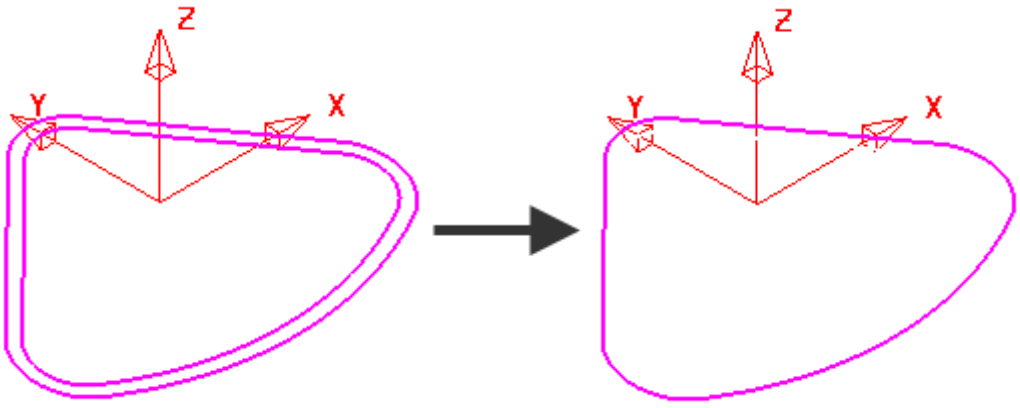


3 In the **PowerMill *explorer*** select, **Boundaries** – **Create Boundary** – **User Defined**.




4 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).

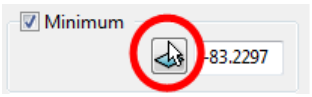
7 From the **Strategy Selector**, open the **Finishing** dialog and select a **Constant Z Finishing** strategy.

8 In the **Constant Z** Finishing dialog enter the **Toolpath** name **OP2-D16TR3-FIN1**.


Copyright © Autodesk

9 In the ***Limit*** page select the ***Boundary*** **OP2-Form-TopEdge** with ***Trimming*** set to **Keep outside**.

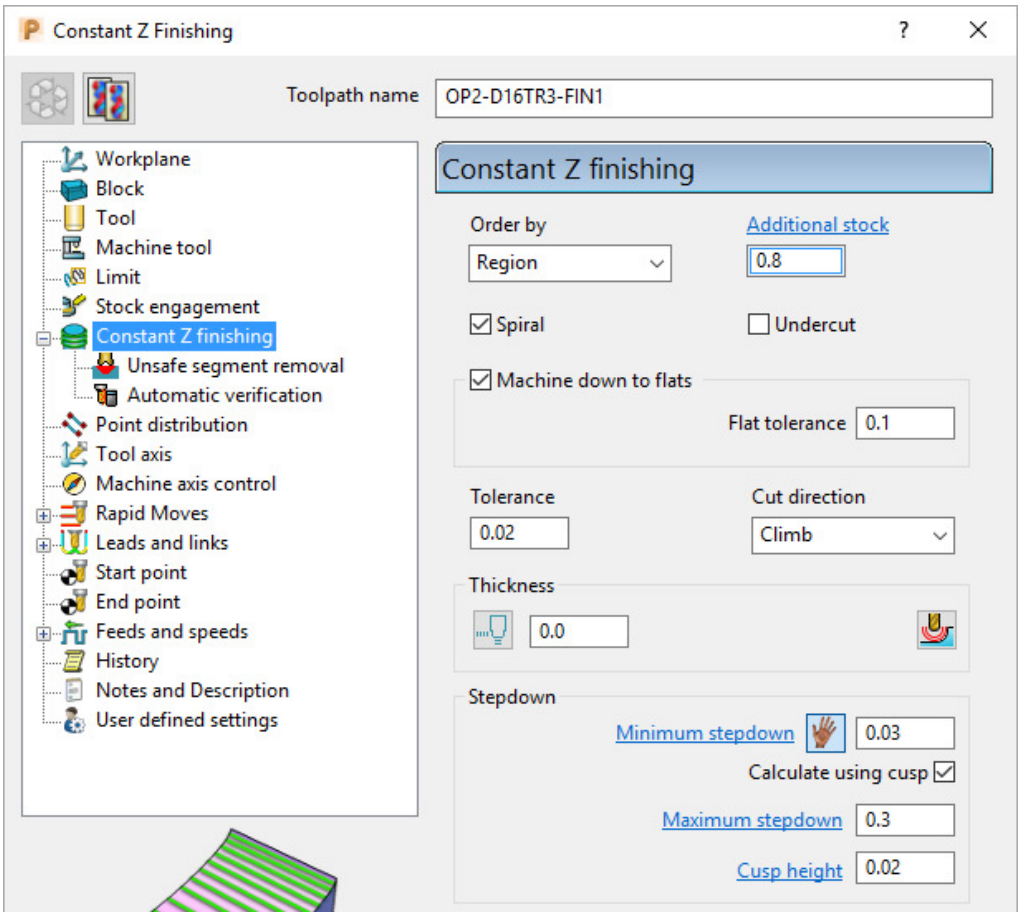
10 In the ***Limit*** page **Z Limits**:-

11 Tick the **Minimum** box and use the ***picking*** option 

to trap and insert the value **-85** (left mouse click within the top of the base surface).

12 Tick the **Maximum** box and insert the value **-79** (this is **6mm** above the top of the base surface).

13 In the ***Constant Z finishing*** page, enter data exactly as shown below (Note the tick box settings!).




14 In the main **Leads and Links** dialog:-

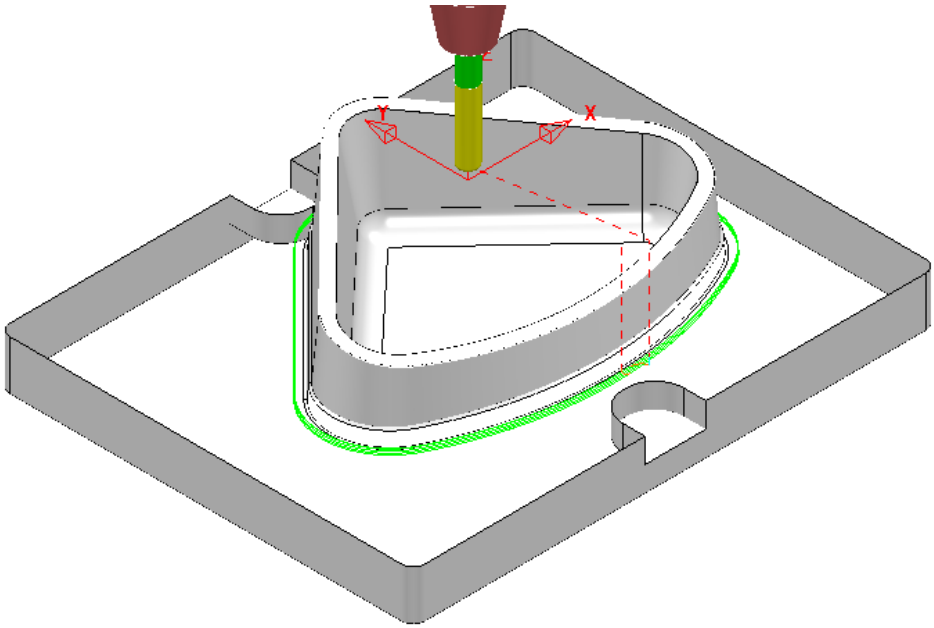
- For the **Lead in *1st choice*** set **Surface normal arc** with **Distance 0**, **Angle 90**, and **Radius 8**.
- For the **Lead out *1st choice*** set **Vertical arc** with **Distance 0**, **Angle 90**, and **Radius 8**.

15 **Accept** the **Leads and Links** dialog.

16 Select **Calculate** in the main **Constant Z finishing** dialog to create the **Toolpath**.

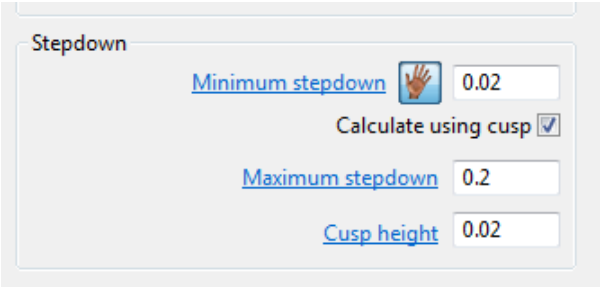
17 **Close** the **Constant Z finishing** dialog.

Copyright © Autodesk



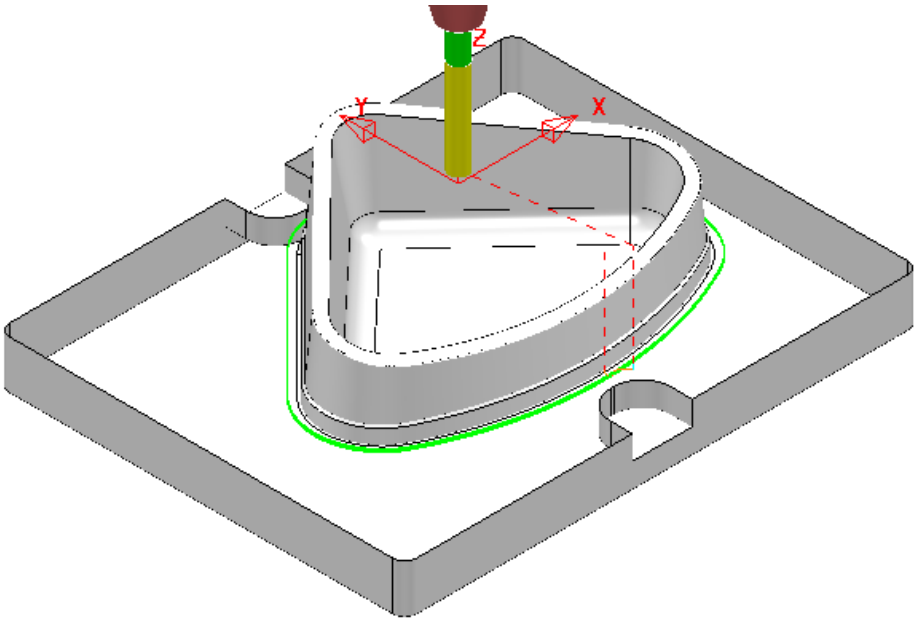

## Finish Machining – Tool EM16

1. 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, tick **Maximum** and input the value **-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.



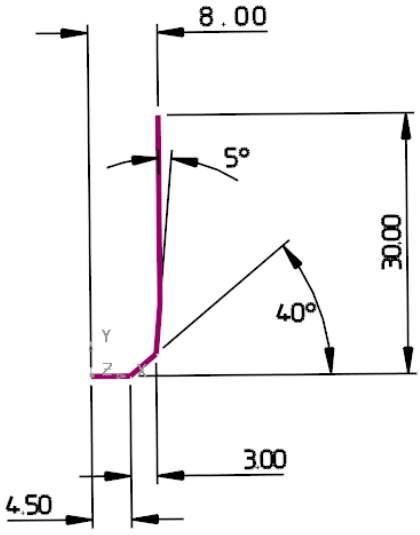

6. Select **Calculate** in the main **Constant Z finishing** dialog to create the **Toolpath**.


Copyright © Autodesk




## 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.

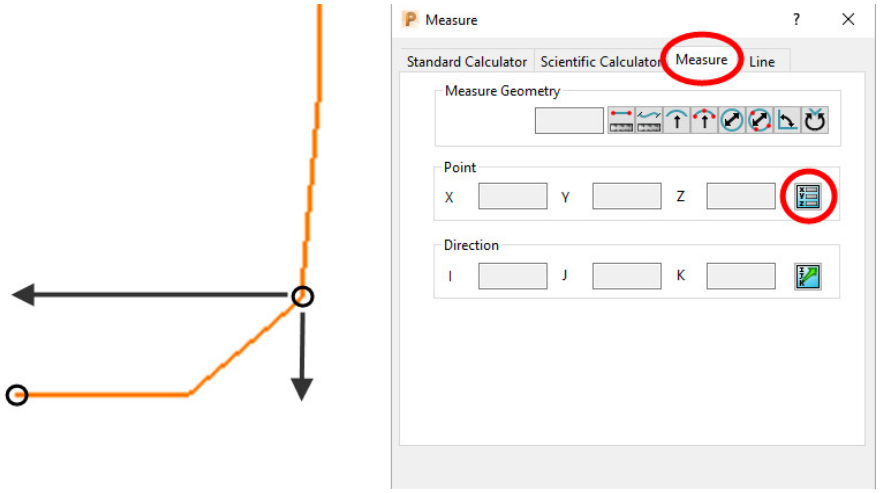



The **Pattern** must be created in the **+ve X** and **Y** quadrant, relative to the **Transform** (Global Datum) for it to be valid when used in **Form Tool** creation options. Once the **Form Tool** has been created it will no longer be dependent on the **Pattern**.

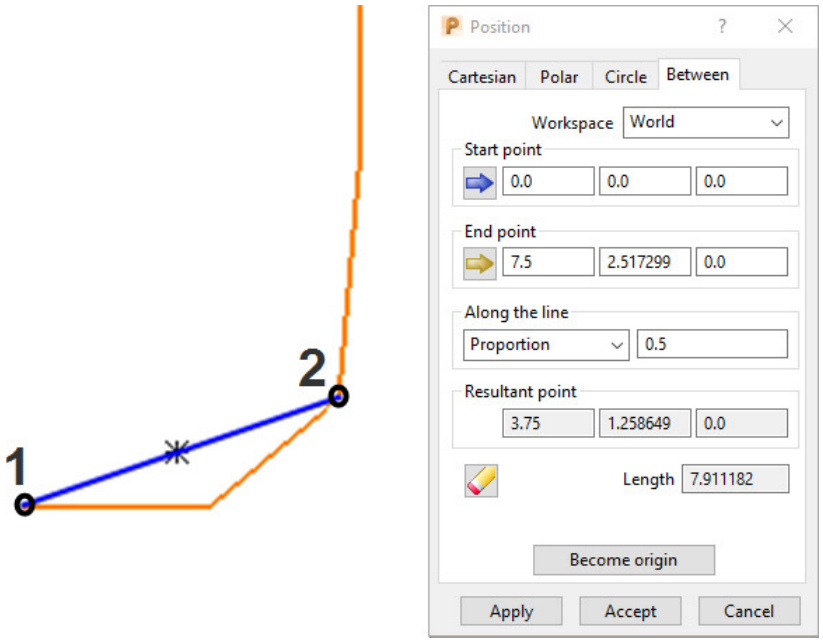
*The above **Tool** and original **Pattern** already exists in the start up **Project**.*

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** .




3 With the **Measure** tab selected, click on the **Open the position input** option .




4 In the **Position** dialog select the **Between** tab

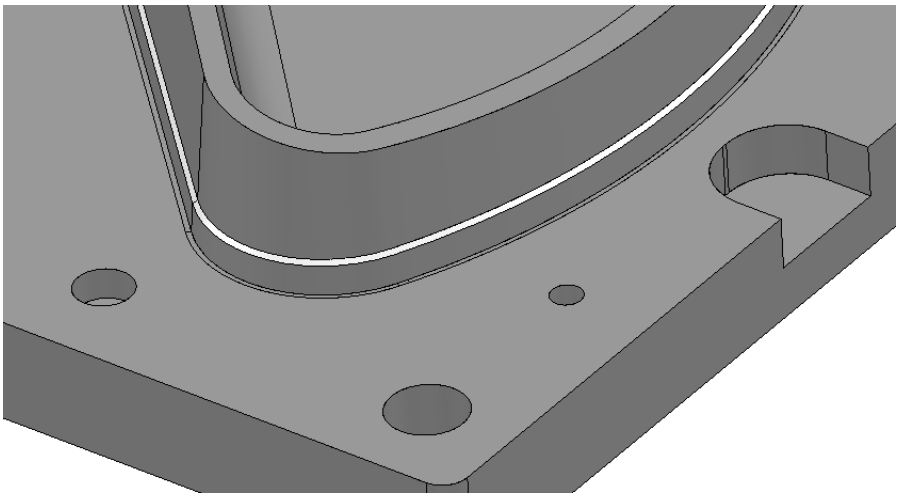
5 On the **Pattern** in the graphics area, left click on ***key point* 1** followed by ***key point* 2** (shown above left).

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**).


Copyright © Autodesk

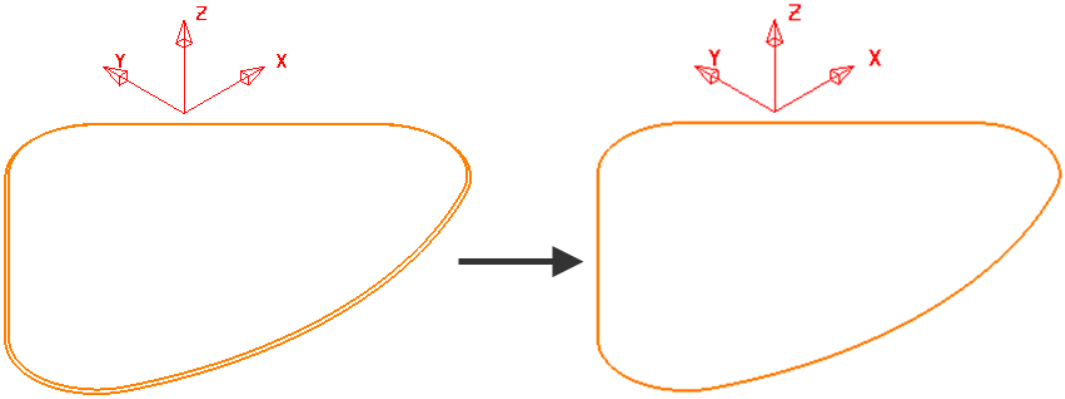
7 Select all of the **surfaces** that form the top face of the step near the bottom of the 3D form (shown shaded light below).



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***).

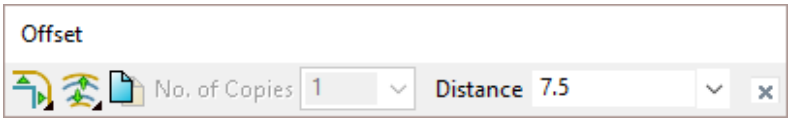



11 Select and **Delete** the outer **Pattern** segment (as shown above).

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)

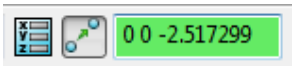



15 Input a **Distance** value of **7.5** and press return to accept the action.

16 Select the **Pattern** segment.

17 In the ***Curve Editor*** toolbar select **Move item**.

18 In the Data Input box below the main graphics area enter and return the relative XYZ move values 0 0 -2.517299.




19 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**.

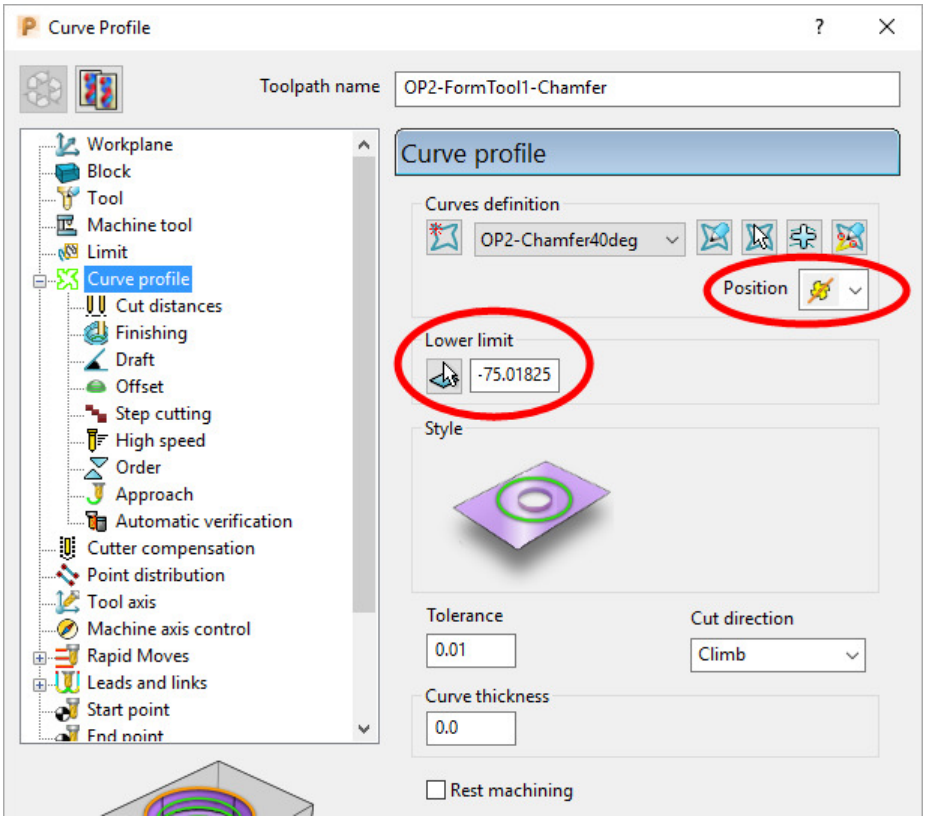
The **Pattern** is now suitably positioned for use as a ***tool centre***, **drive curve** to be used in either a **2D Curve Profile** or **Pattern Finishin**g strategy.

22 Ensure that no **Boundary** is **Active**.

23 **Activate** the tool **FormTool-1**.

24 From the **Strategy Selector**, open the **Curve Machining** dialog and select the **2D Curve Profile** strategy.

25 In the **2D Curve Profile** dialog enter the toolpath name as:- **OP2-FormTool1-Chamfer**.



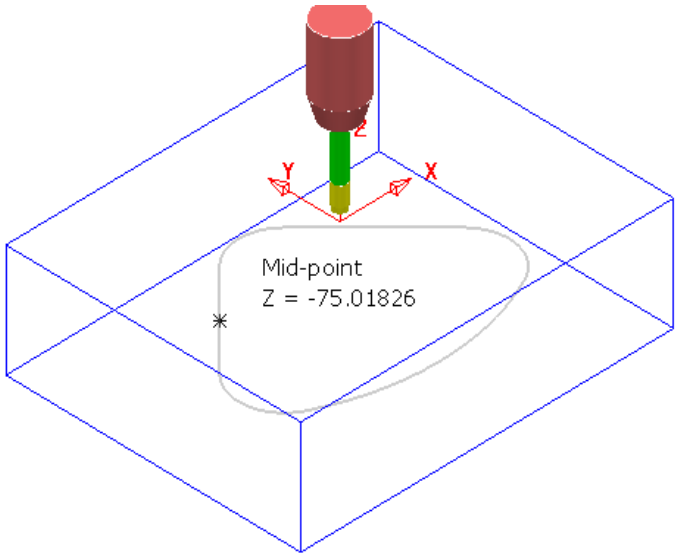


Copyright © Autodesk

26 In the **Limit** page ***untick*** both the **Z Limits**, ***Maximum*** and ***Minimum*** boxes.

27 Set the **Position** option to **The centre of the tool follows the pattern** .

28 Select the **Pick the lowest Z Height** option and snap at any point along the drive curve, **Pattern *segment***.

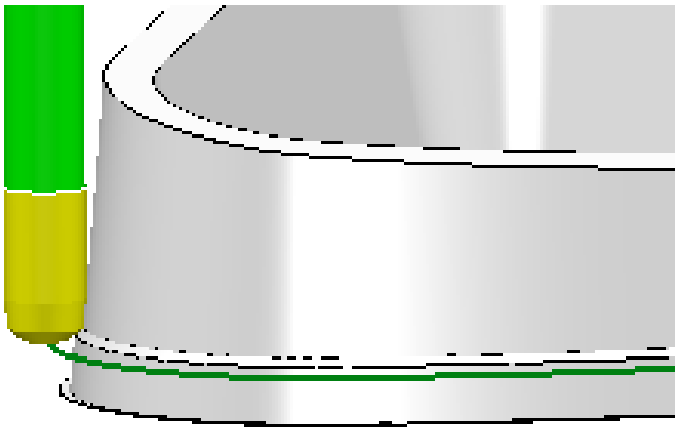



*The snapped **Z height value** is inserted into the **Lower limit** box.*

29 In the main **Leads and Links** dialog select **Lead in/out *1st choice*** set **Horizontal arc** with **Distance 0**, **Angle 90**, and **Radius 12**.

30 Accept the **Leads and Links** dialog.

31 Select **Calculate** in the main **Offset flat finishing** dialog to create the toolpath.



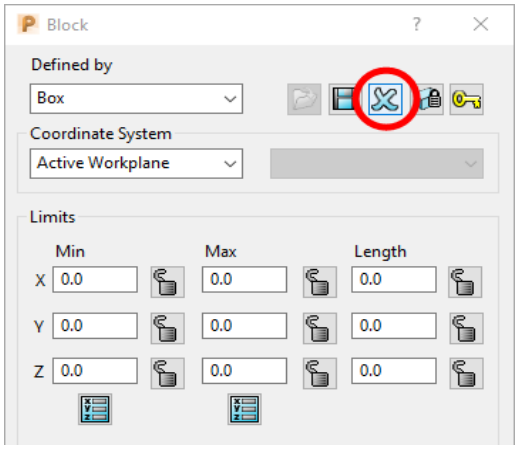
The Form tool is accurately aligned to the angle across the top of the step.

32 **Close** the **2D Curve Profile** dialog.

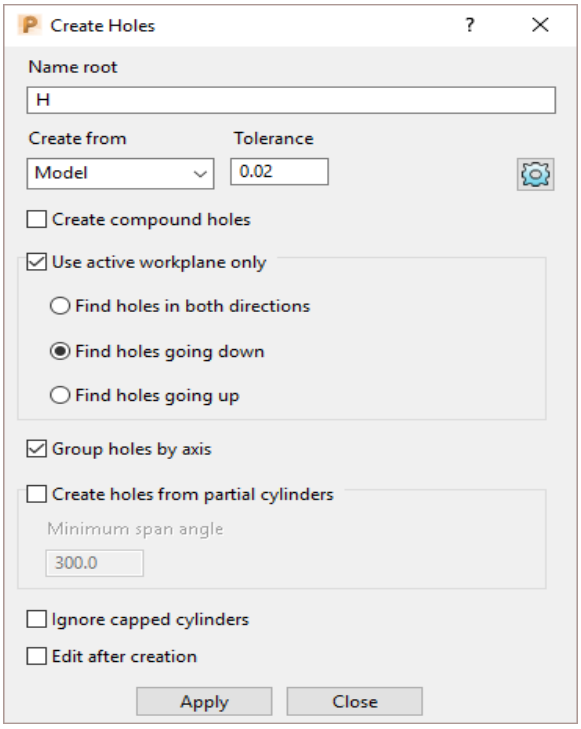
# Hole Feature – Drilling

**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 deleted.*

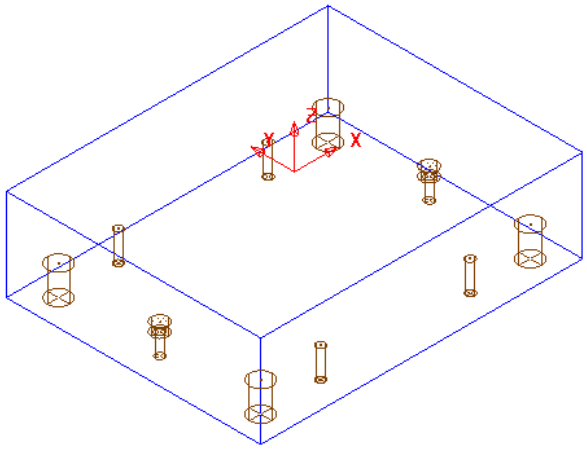



1. From the **main** toolbar select the **Block** dialog and click on the ***blue cross*** to delete the current **Block** definition.
2. Select the whole of the component **model**.
3. In the **PowerMill *explorer***, right mouse click on **Hole Feature Sets** and from the local menu select **Create Holes**.




4. Enter the data into the **Feature** dialog exactly as shown above before selecting **Apply**.


Copyright © Autodesk



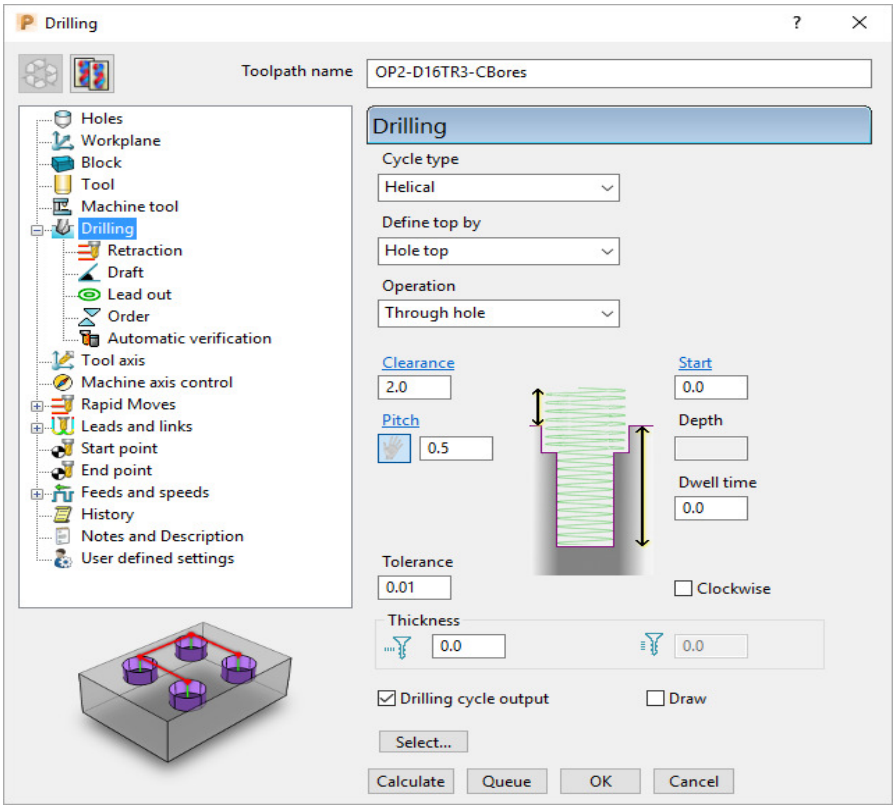
The **Feature Set** contains a total of **12** defined **Hole Features**.

*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.*

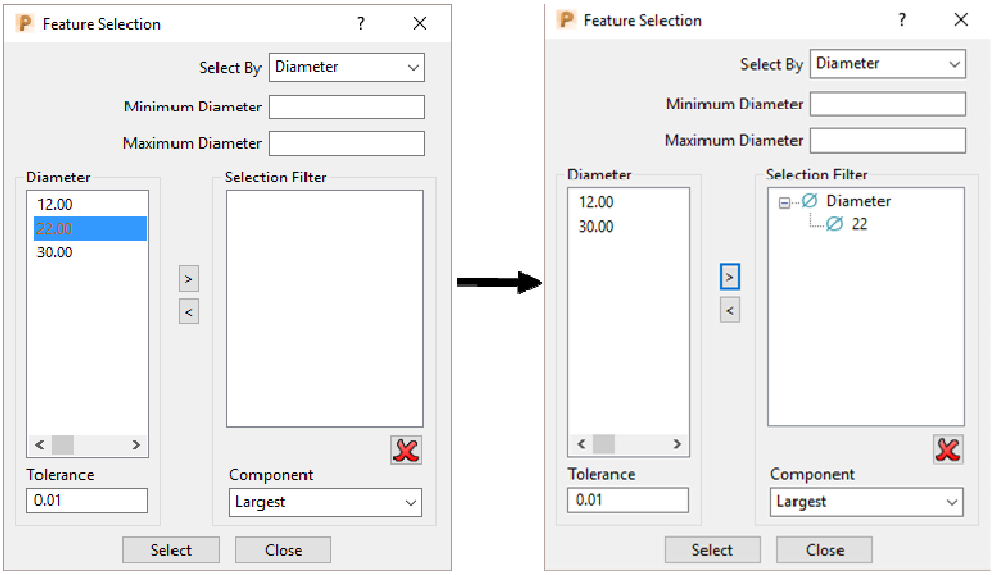
5 Rename the new **Hole Feature Set** as **OP2-Holes**.

6 Activate the toolpath **OP2-D16TR3-FIN1** to reinstate the **Block** and the tool **D16TR3**.

7 From the **Strategy Selector**, open the **Drilling** dialog and select the **Drilling** strategy.

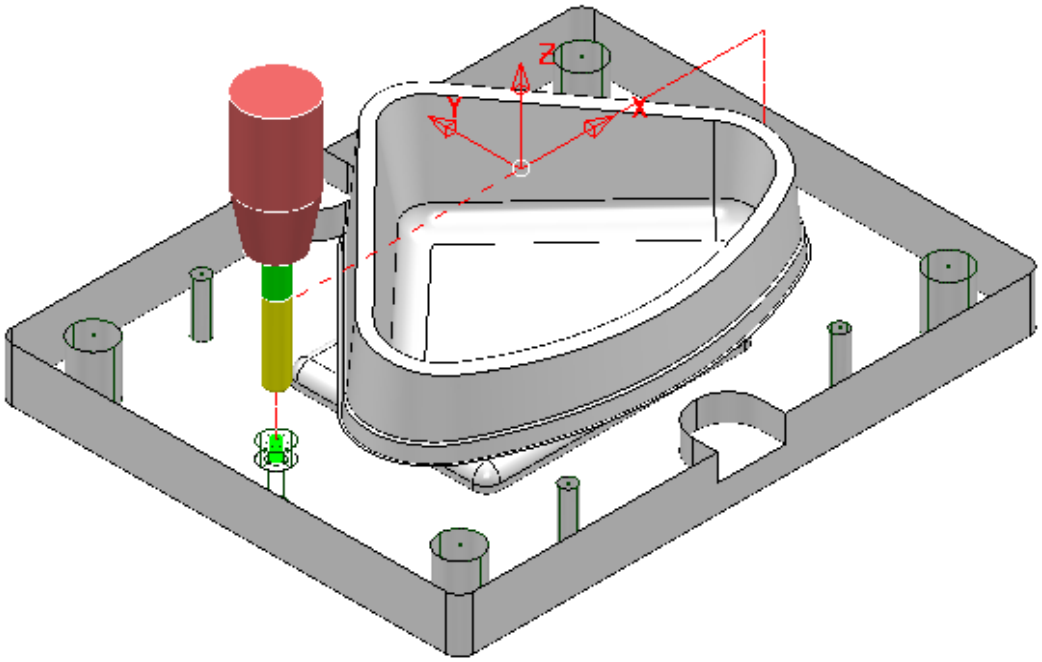



8 Select ***Cycle type* Helical** and input remaining data into the **Drilling** dialog exactly as shown above and then click on the **Select** tab.



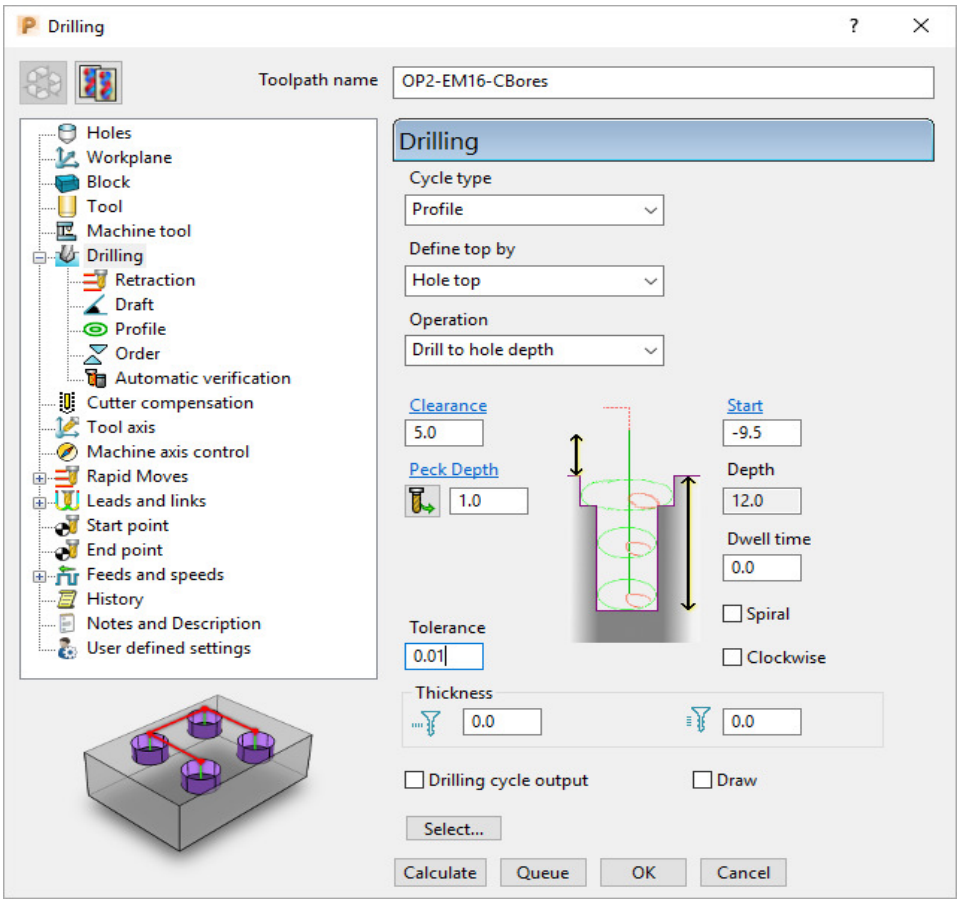

9 In the Diameter box select **22.00** and click on > to add these holes to the ***Selection Filter*** box.

10 Close the **Feature Selection** dialog and select **Calculate** in the main **Drilling** strategy dialog.

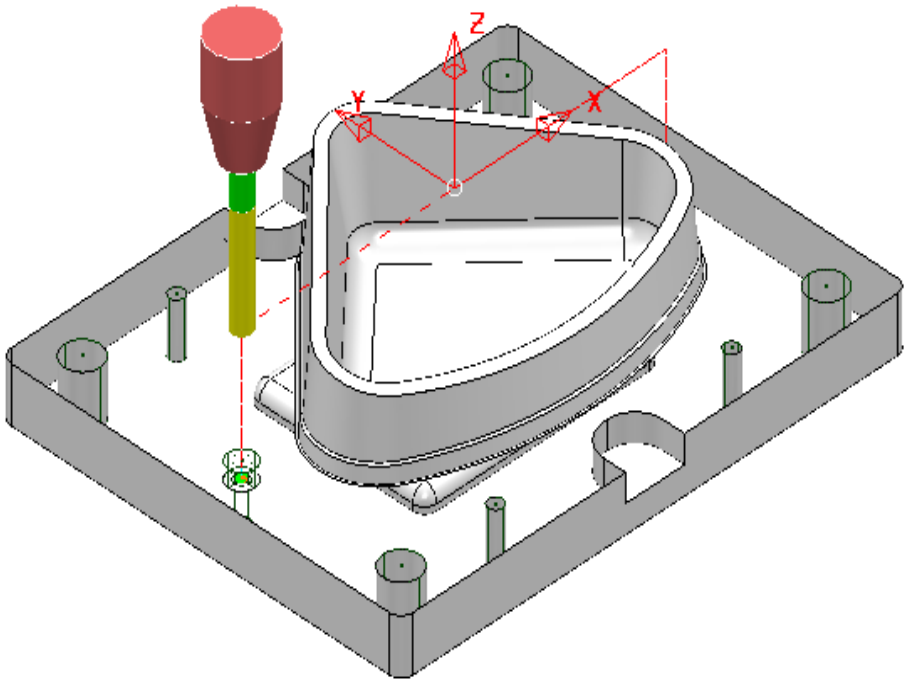



11 Make a **Copy** of the last **Drilling Toolpath** named **OP2-EM16-CBores** and this time use the ***Tool*** **EM16** and ***Cycle Type*** - **Profile** on both of the counterbore **Features**.

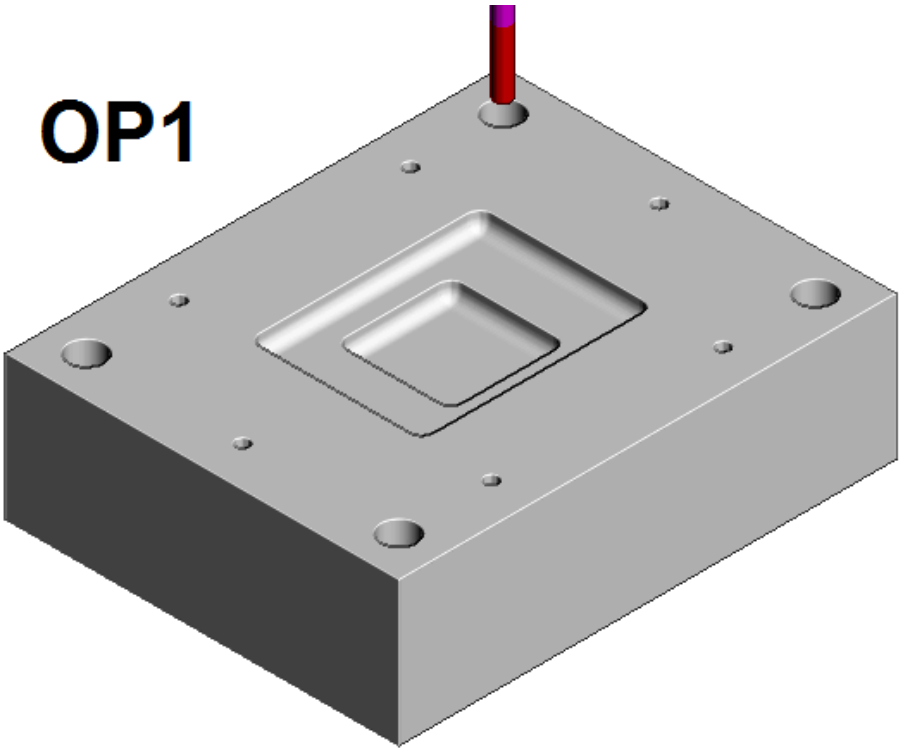

Copyright © Autodesk

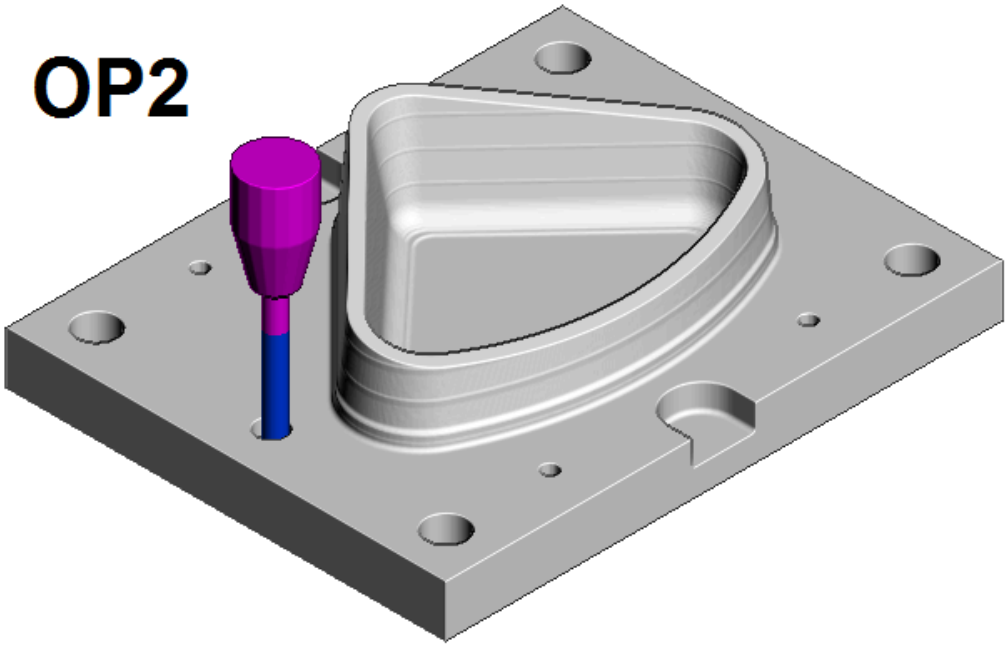



12 Input data into the **Drilling** dialog exactly as shown above and **Calculate**.




13 Perform **ViewMILL** simulations for both **OP1** and **OP2**.




14 **Save** the **Project** to update:-

**….\COURSEWORK\PowerMill_Projects\CornerBowl-Example**

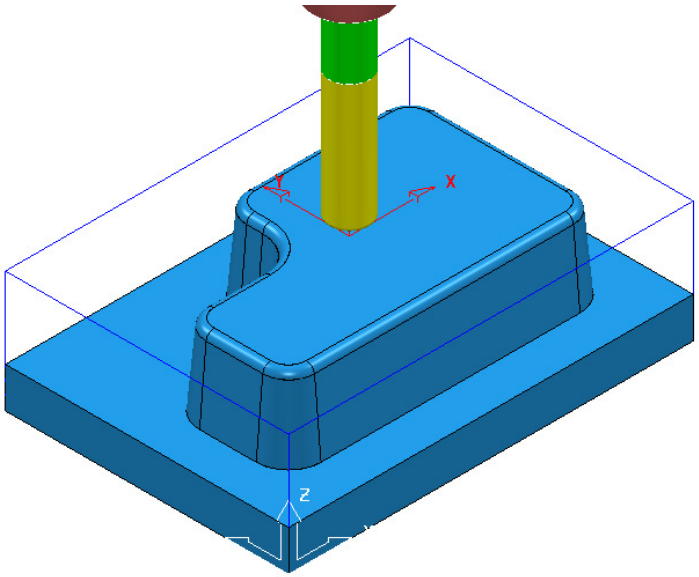

Copyright © Autodesk

# 2.5D Machining Example

1 Select **File** > **Delete All** and **Tools** > **Reset Forms**

2 **Open** the (read only) **Project**:-

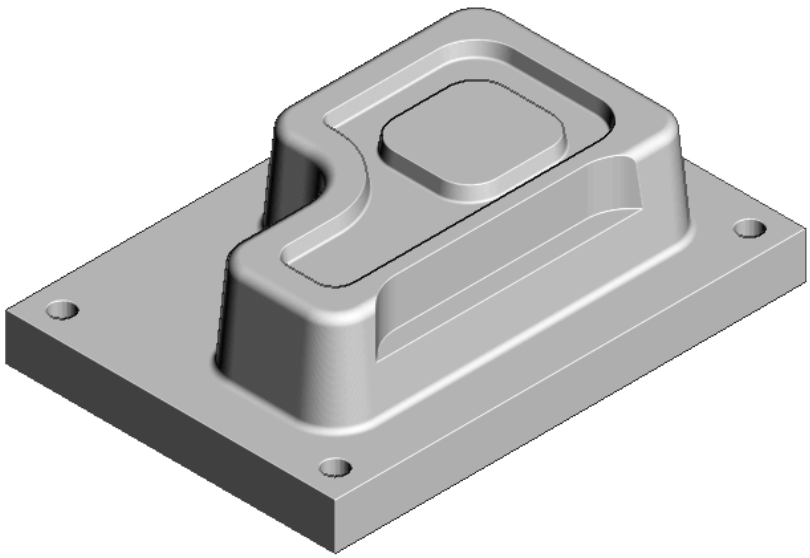
**….\PowerMill_Data\Projects\ 2DCurveMachining-2-Start**




3 **Save Project As**:

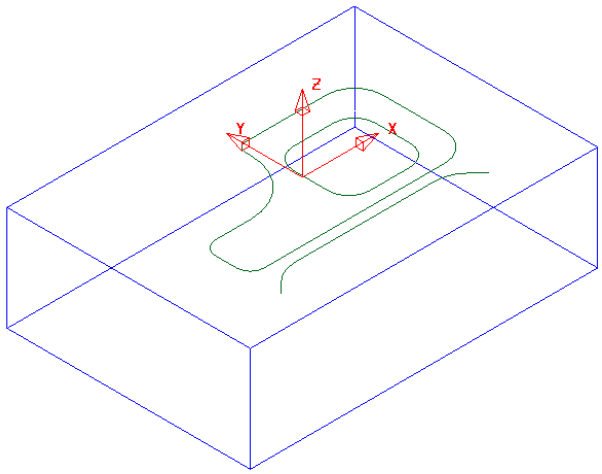
**….\COURSEWORK\PowerMill_Projects\2DCurveMachining-2**

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:-

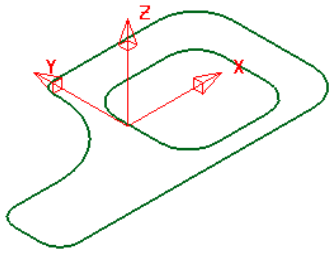
The **Model** also includes ***2D curves*** which can be made visible by switching on ***Wireframe*** in the **Views** toolbar.



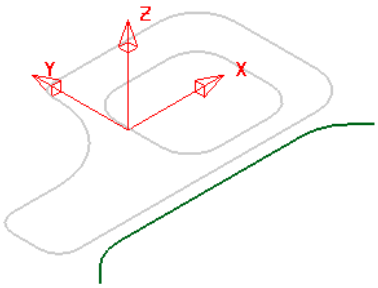

4 Create a new **Pattern** (1).

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.

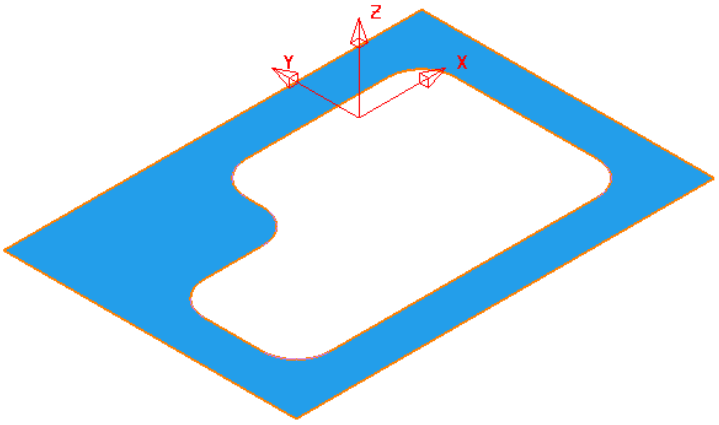




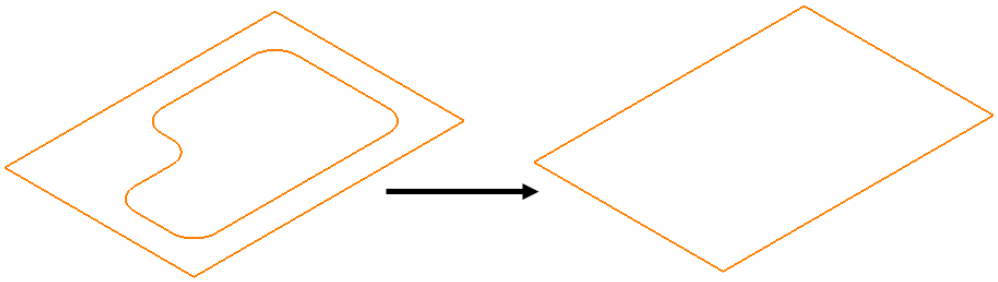
Copyright © Autodesk

8 Create another new **Pattern** (3).

9 Select the upper base surface on the **Shaded** model and **Insert** it into the new **Pattern**.

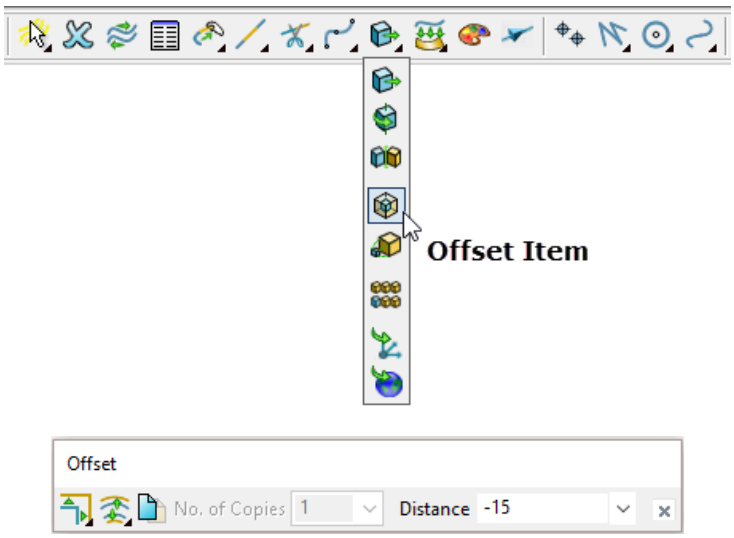



10 Select and **Delete** the inner **Pattern *segment***.



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**.



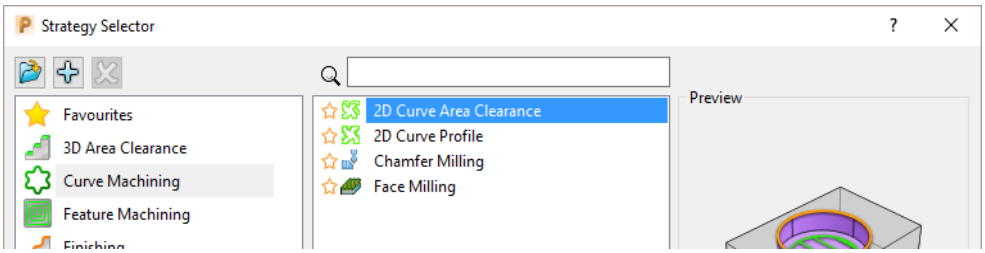

13 To accept the edits, select the ***green tick*** at the end on the **Curve Editor** toolbar.

# Machine the top recess with a 10 degree draft angle:-

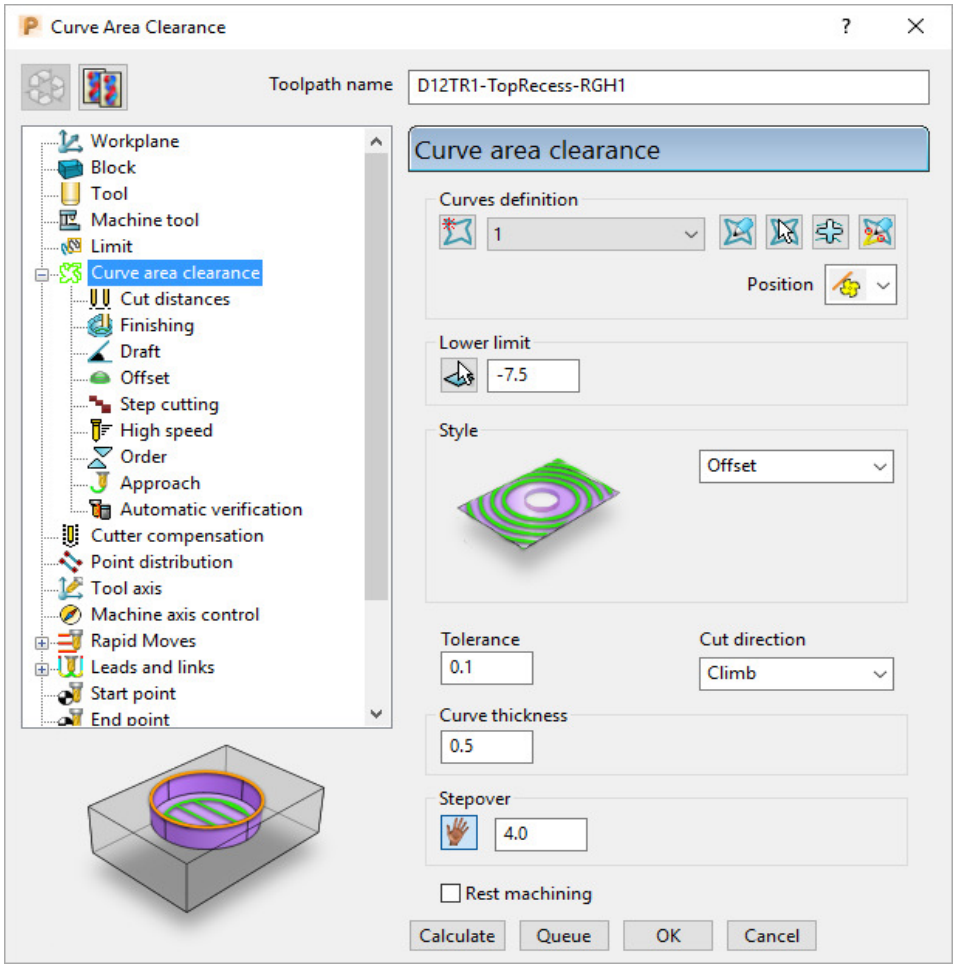
14 **Activate** one of the existing toolpaths to reinstate the settings (Workplane, Block, and Rapid Move Heights).

15 **Activate** the **D12TR1** tool.

16 From **Toolpath strategies** open the **2D Curve Area Clearance** dialog.

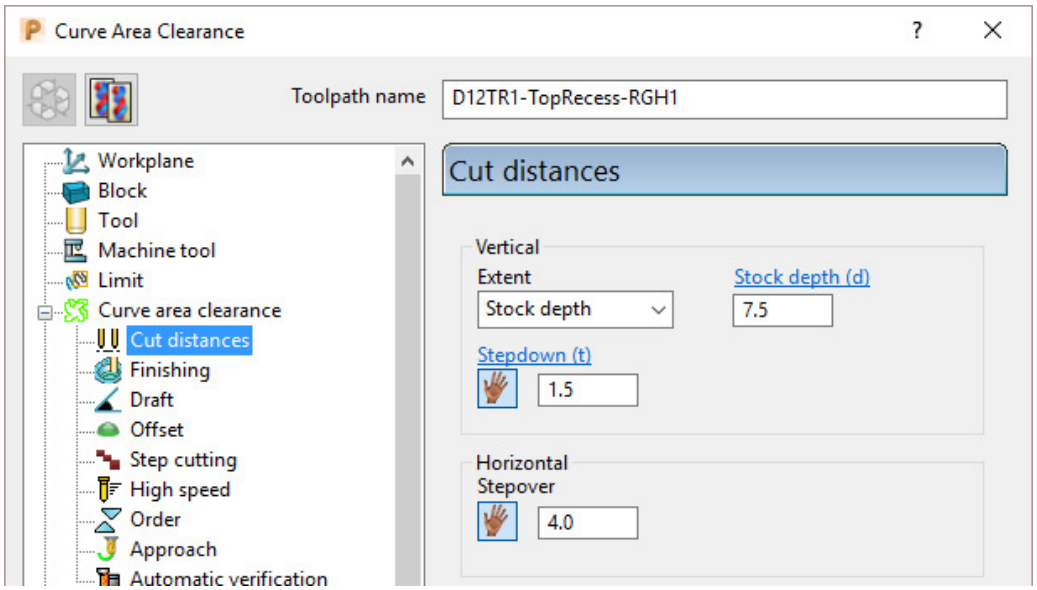


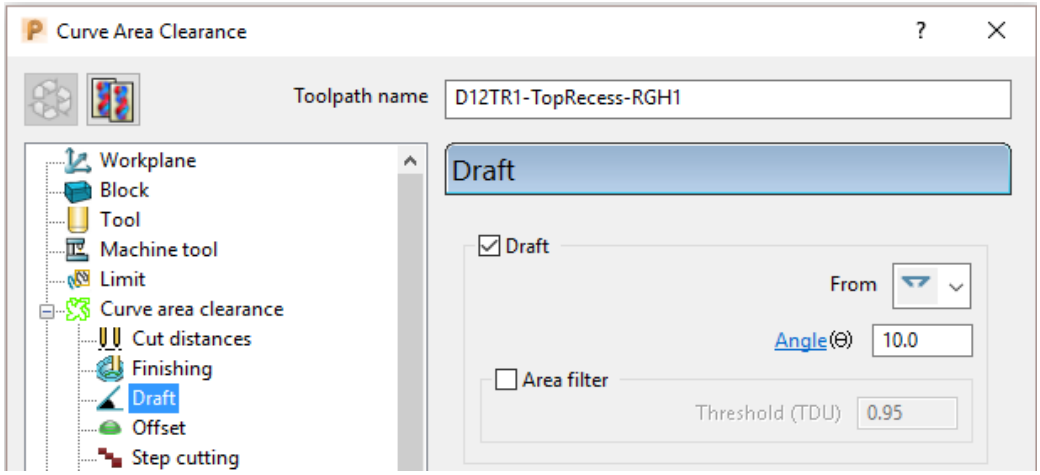

17 Input the data and settings exactly as shown in the following images.

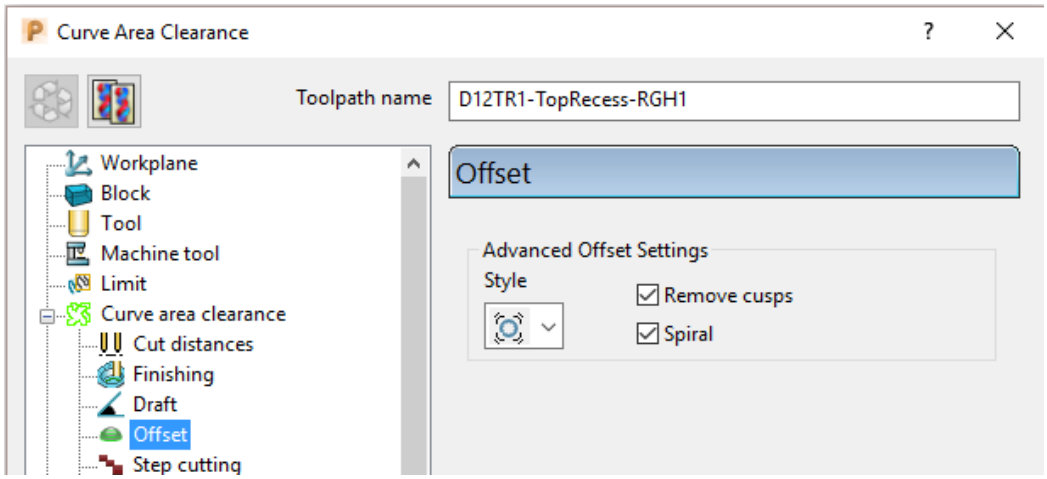



18 Select to preview and if required apply the ***Pattern Decoration*** options to modify the machining area and directions.

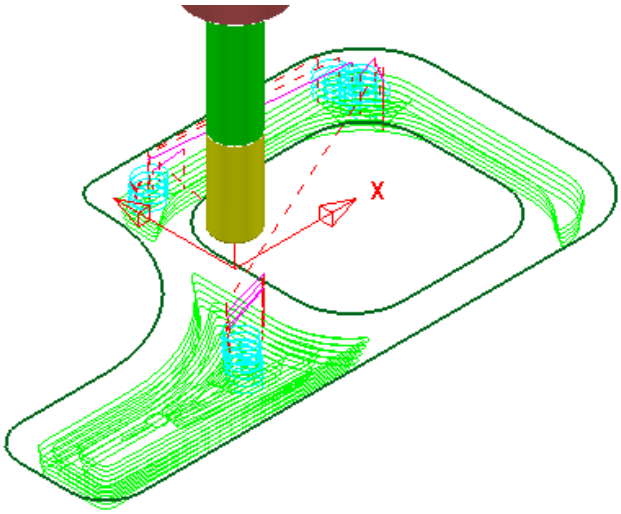

Copyright © Autodesk



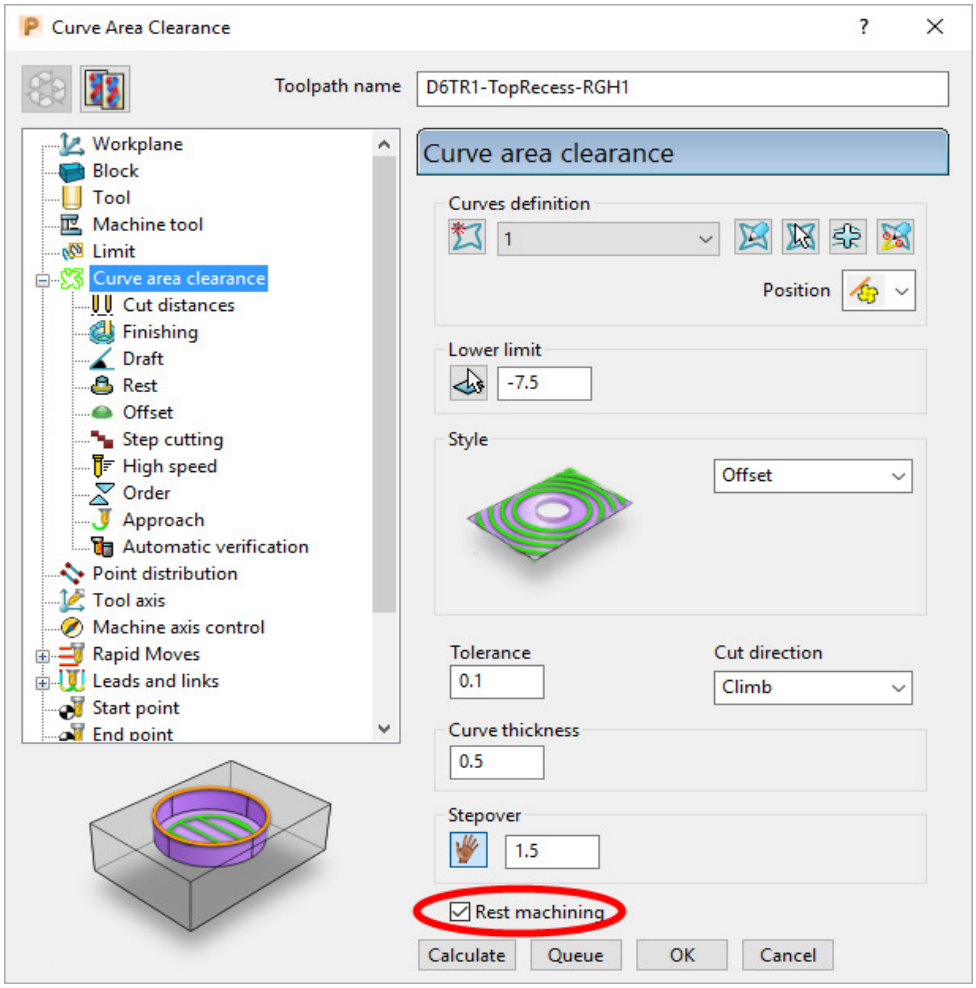

19 Select **Calculate** to process the toolpath and then **Close** the dialog.




20 Activate the **D6TR1** Tool.

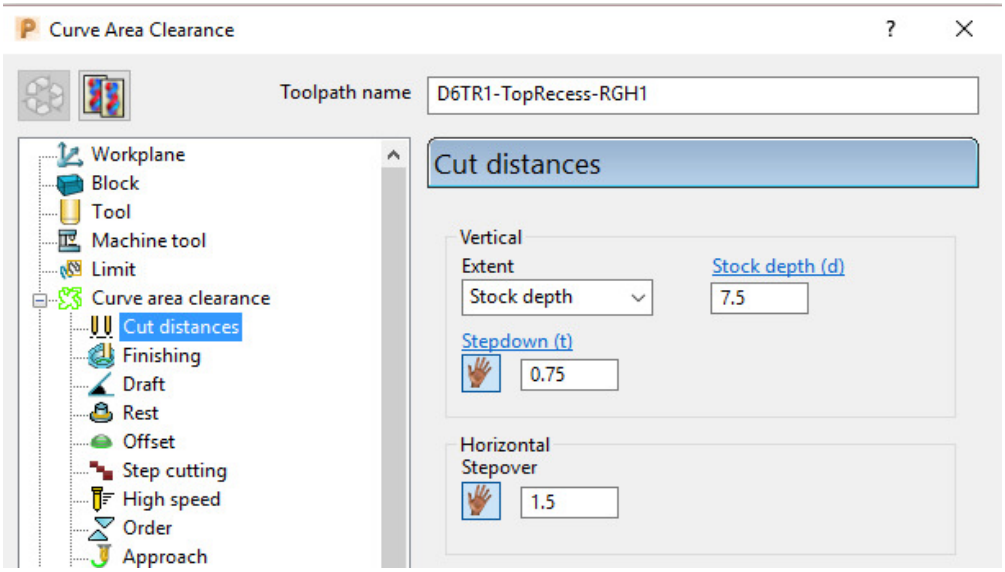
21 From Toolpath strategies open the **2D Curve Area Clearance** dialog.

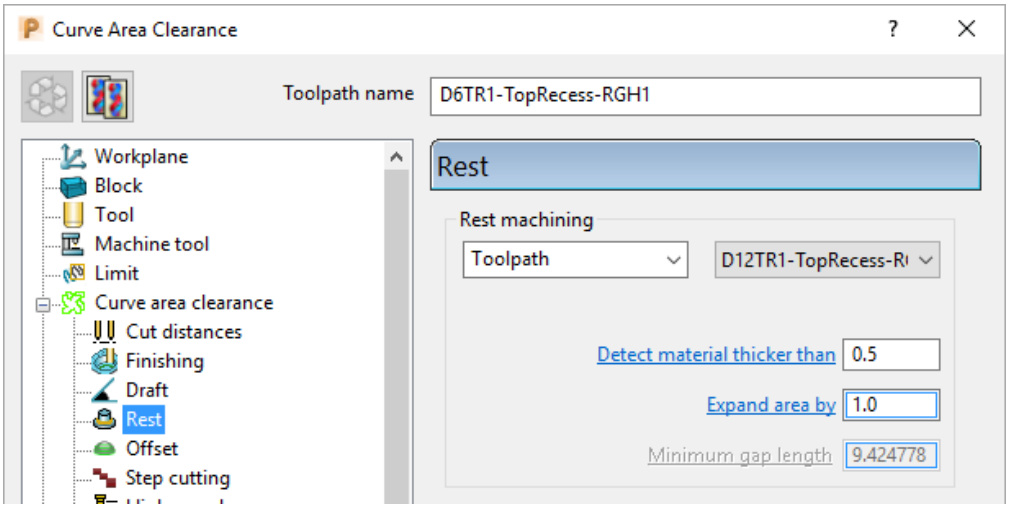
22 Input the data and settings exactly as shown in the following images.



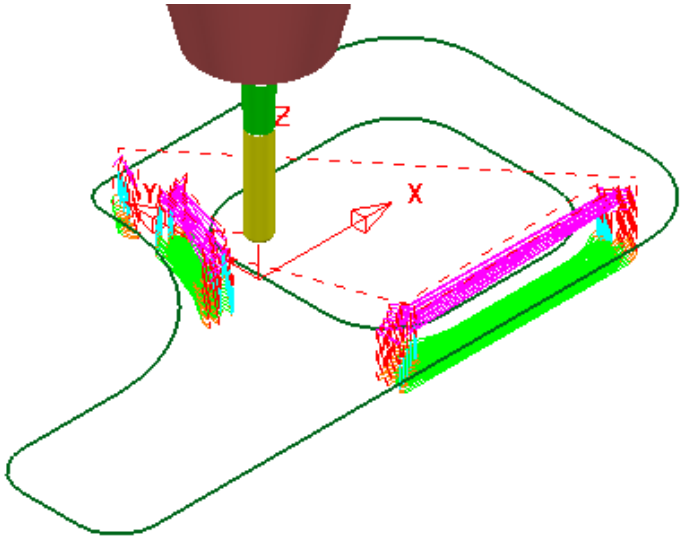


Copyright © Autodesk



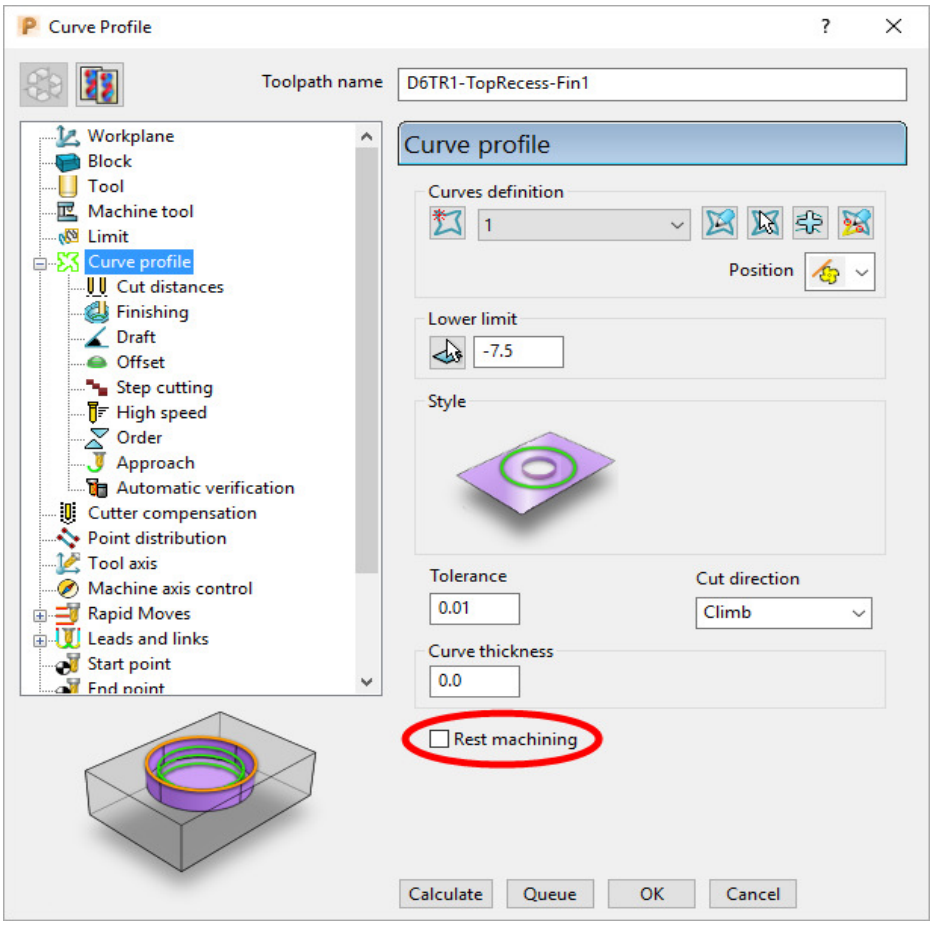



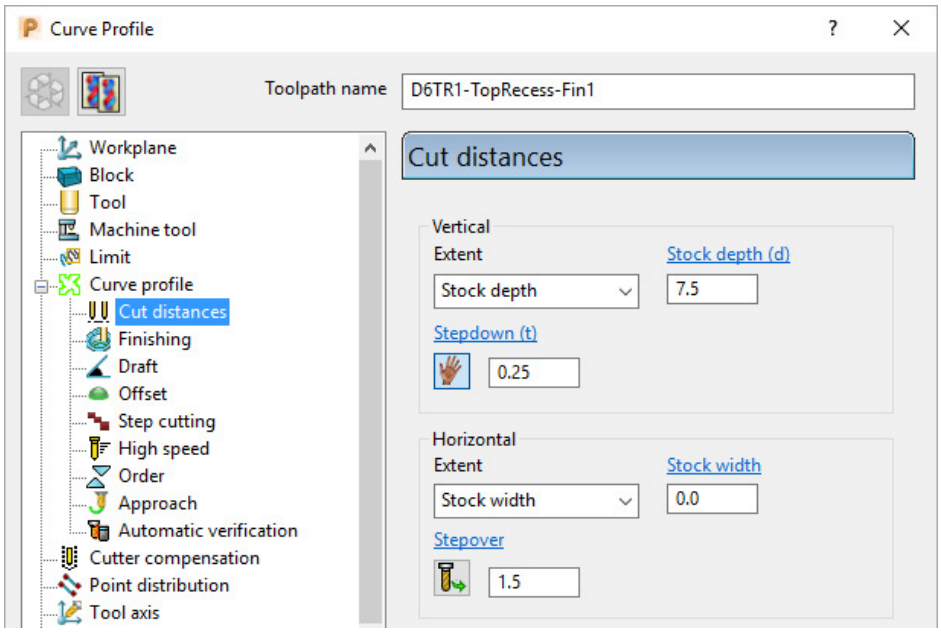

23 Select **Calculate** to process the toolpath and then **Close** the dialog.



24 From Toolpath strategies open the **2D Curve Profile** dialog.

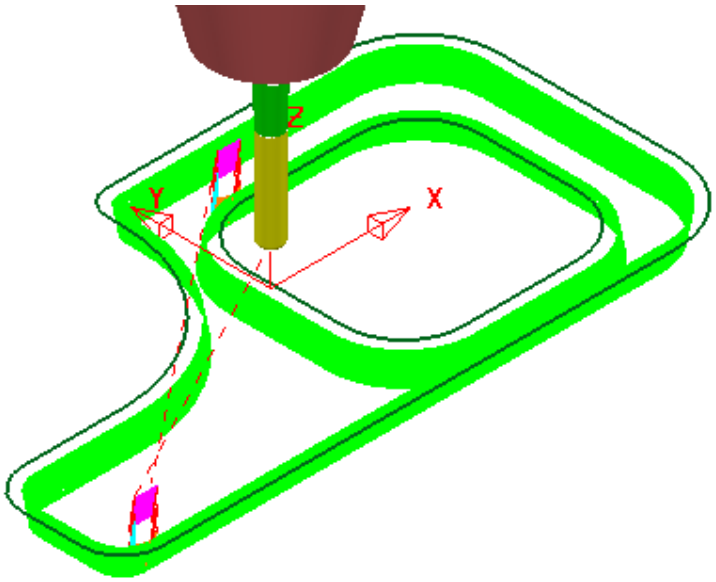
25 Input the data and settings exactly as shown in the following images.




26 Select **Calculate** to process the toolpath and then **Close** the dialog


Copyright © Autodesk

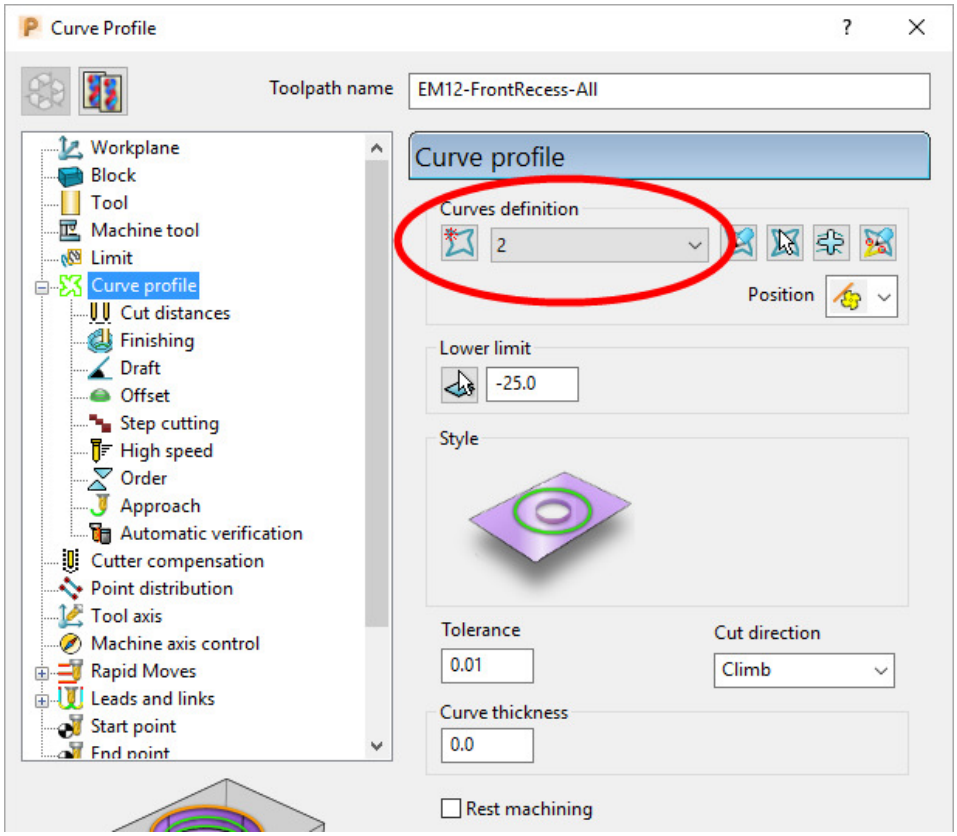


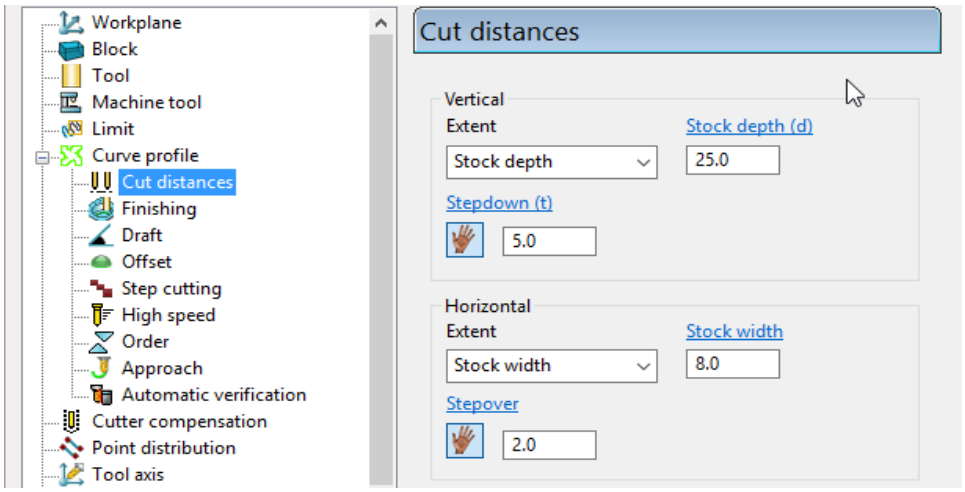

## The next stage is to machine the vertical, front recess:-

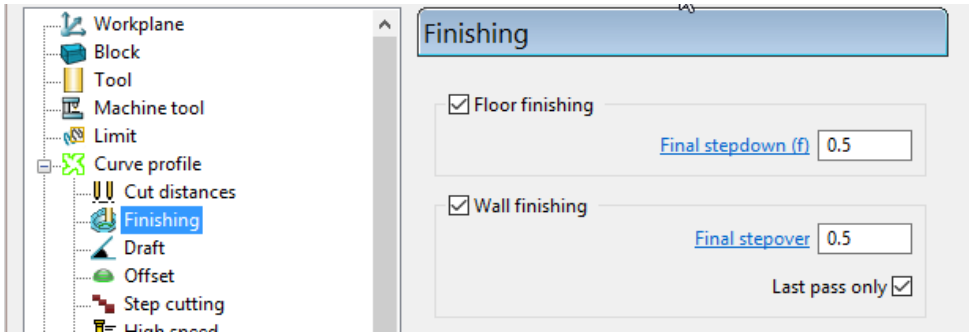
27 Activate the **EM12** Tool.

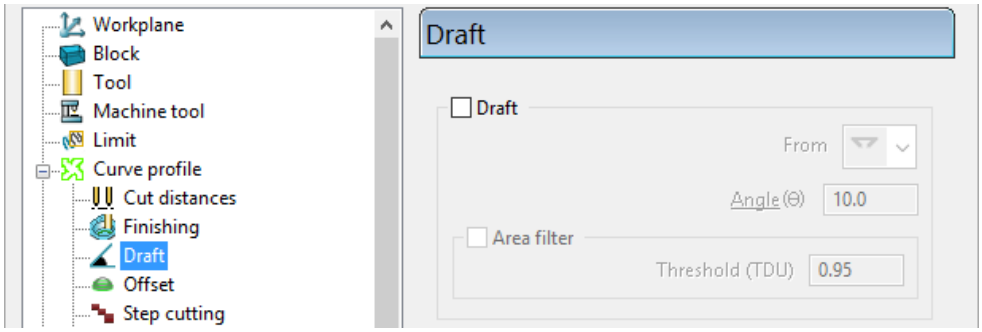
28 From **Toolpath strategies** open the **2D Curve Profile** dialog.

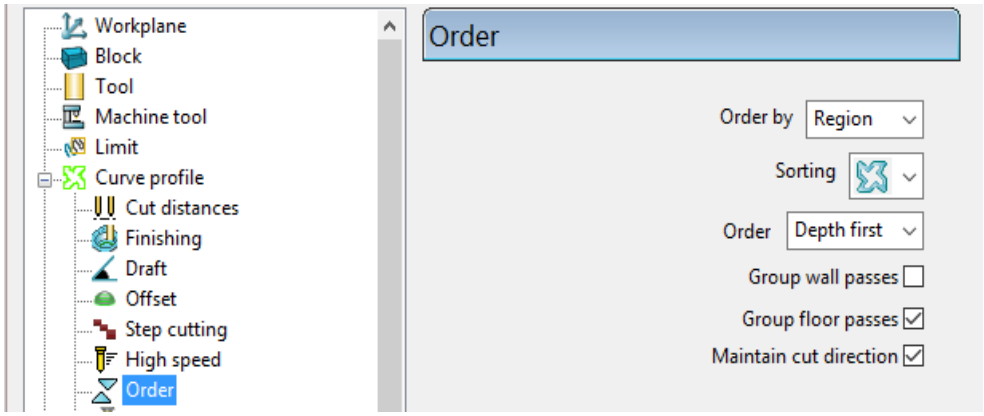
29 Input the data and settings exactly as shown in the following images.





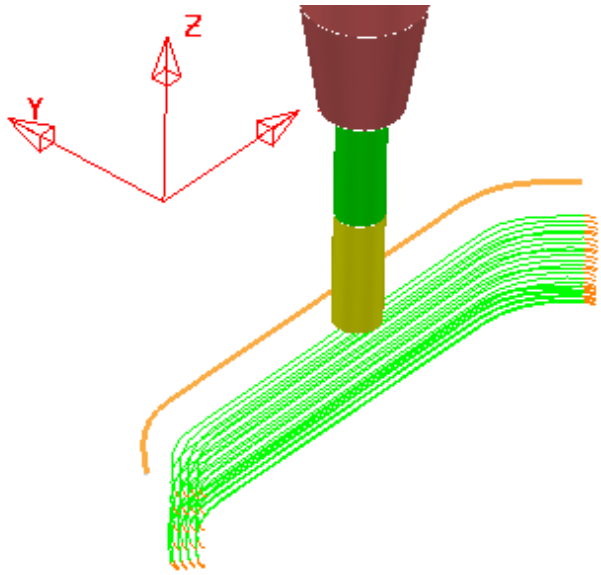



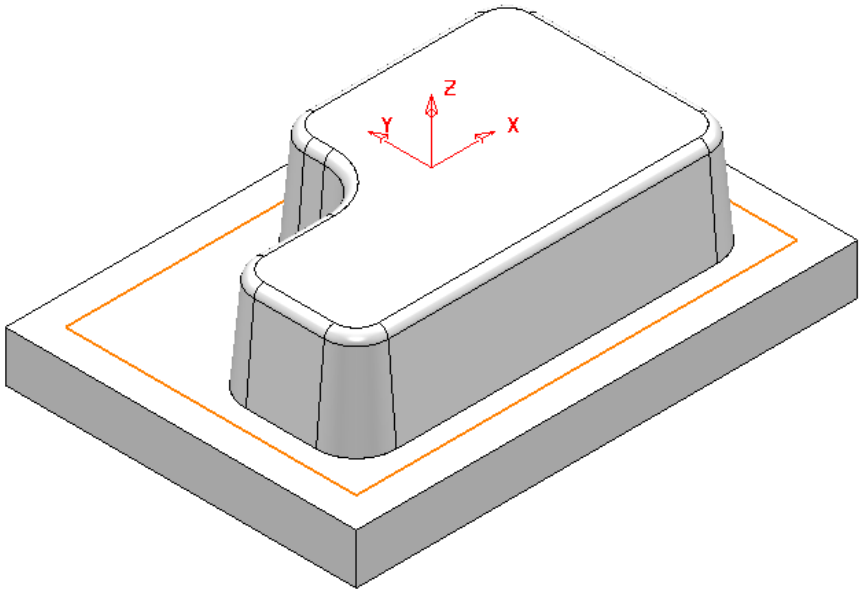

30 Select **Calculate** to process the toolpath and then **Close** the dialog.


Copyright © Autodesk




## Drill four Dia 12 holes offset 15mm from the base walls:-

The next stage is to create a **Hole Feature Set** defining four Dia 12 through holes down from the corners of **Pattern 3**.

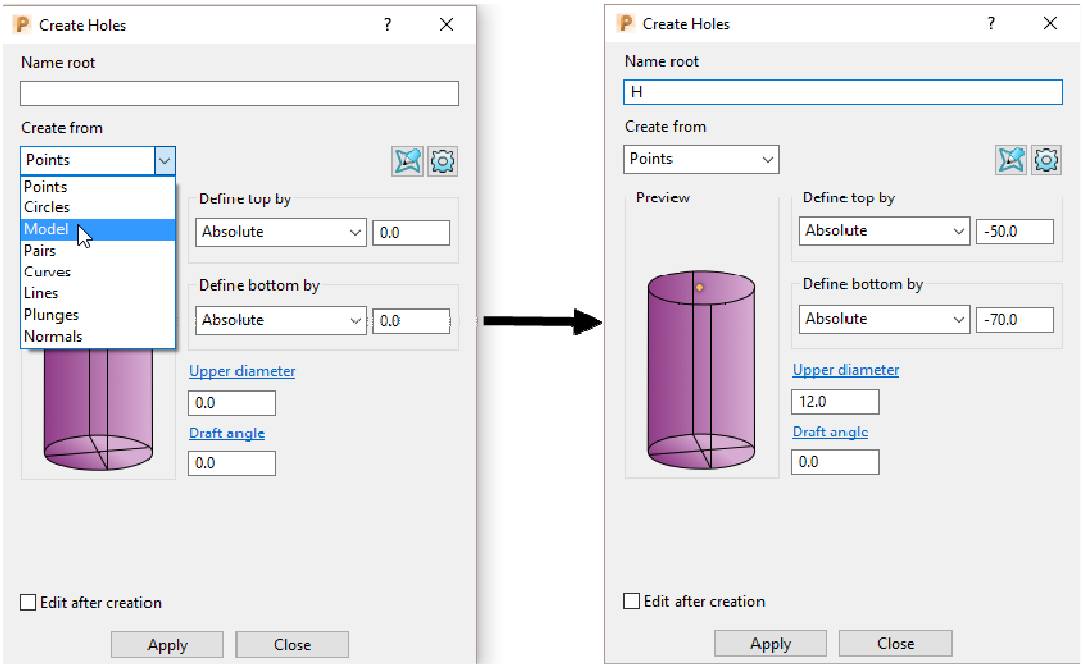



31 Select the rectangular **Pattern 3** located on top of the base surface.

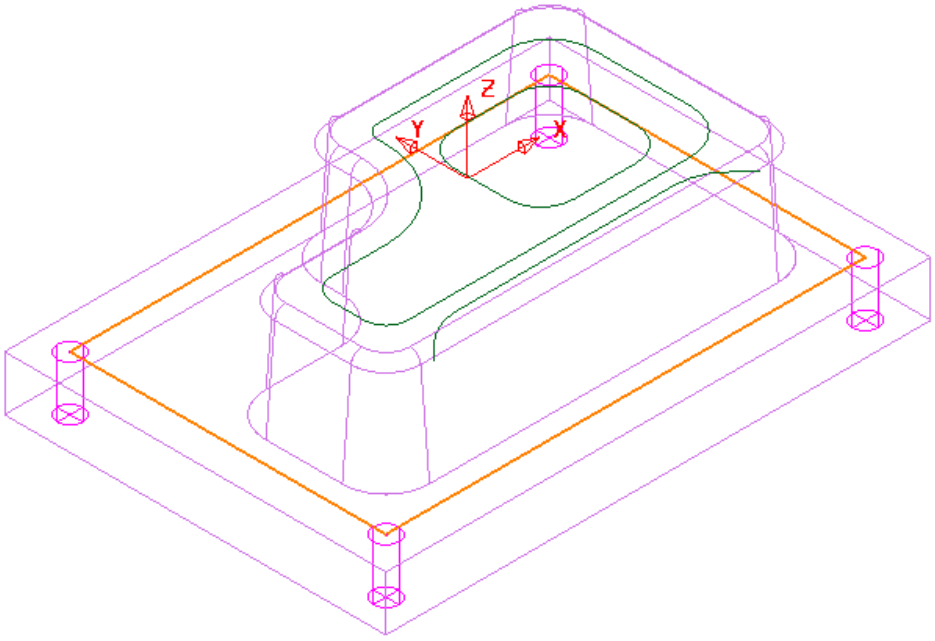
32 In the **PowerMill *explorer*** right mouse click on **Hole Feature Sets** and from the local menu select **Create Holes**.

33 In the **Create Holes** dialog select the option ***create from* Points**.

34 Input the **Absolute** values **-50** (top) and **-75** (bottom).




35 Select **Apply** to create four Dia 12 Hole **Features**.

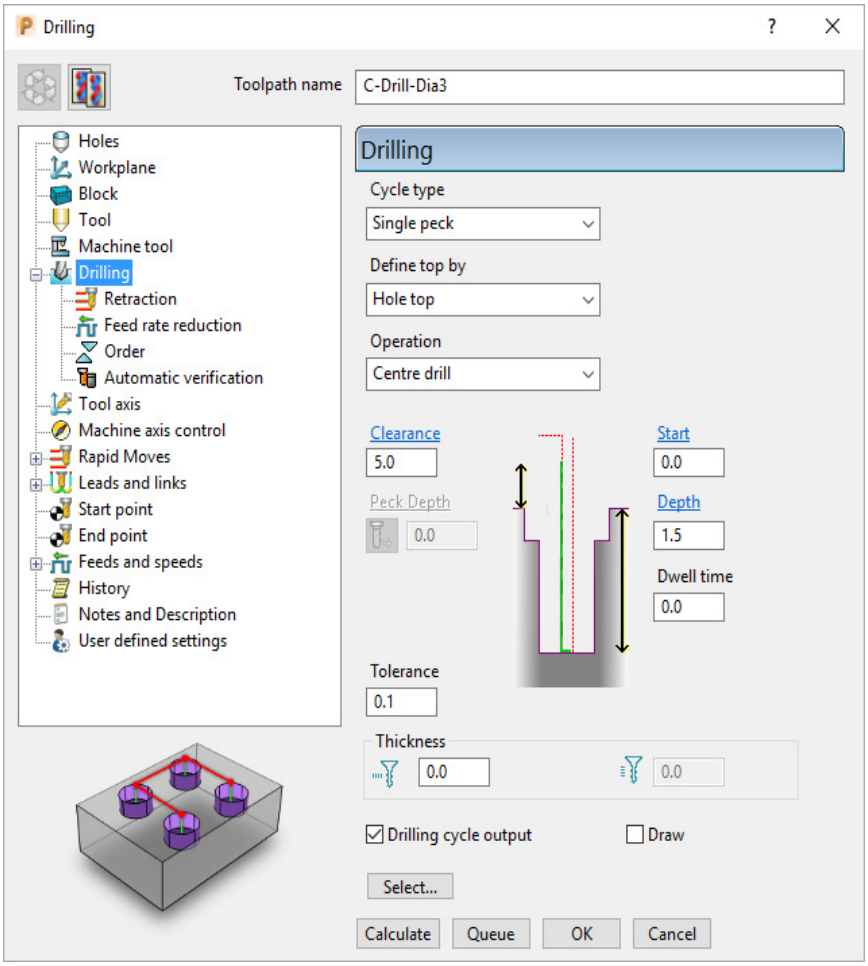



36 Activate the Dia 3 Drill named **Drill3**.

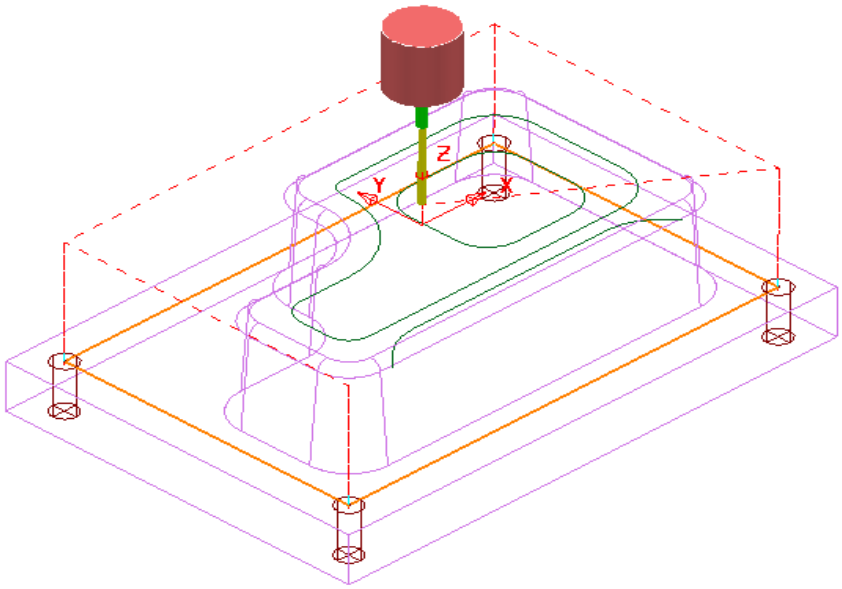
37 From Toolpath strategies select the **Drilling** – **Single Peck** option.

38 Input the data and settings exactly as shown in the following image.


Copyright © Autodesk

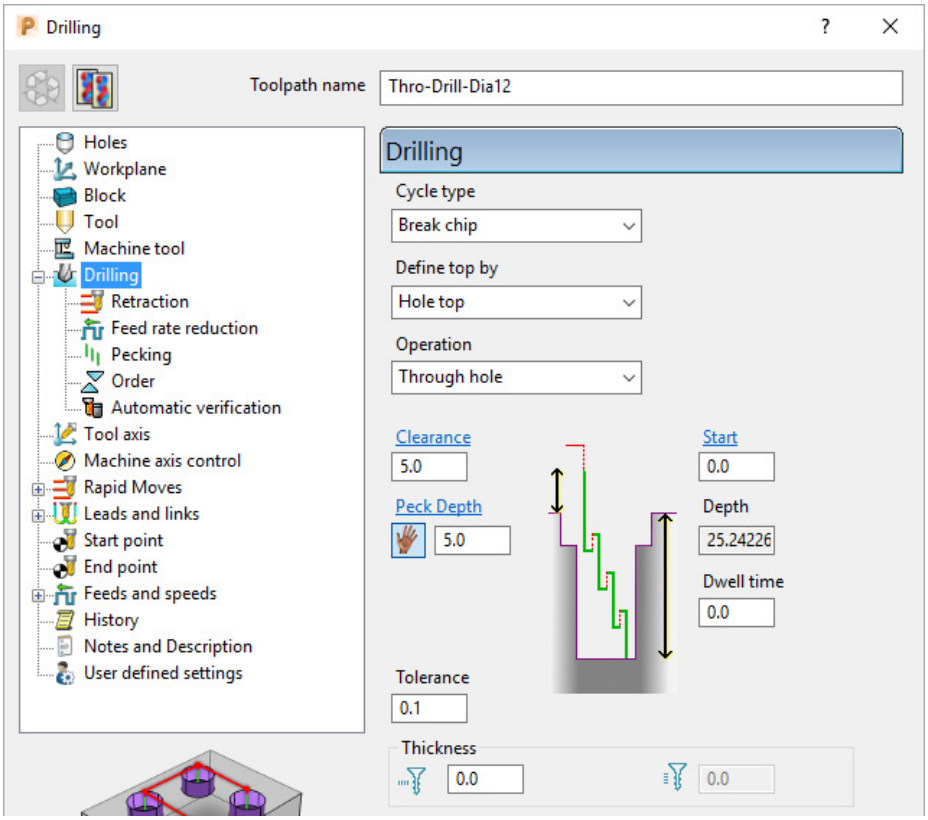



39 **Calculate** the Drilling toolpath and then select **Re-cycle** the strategy.

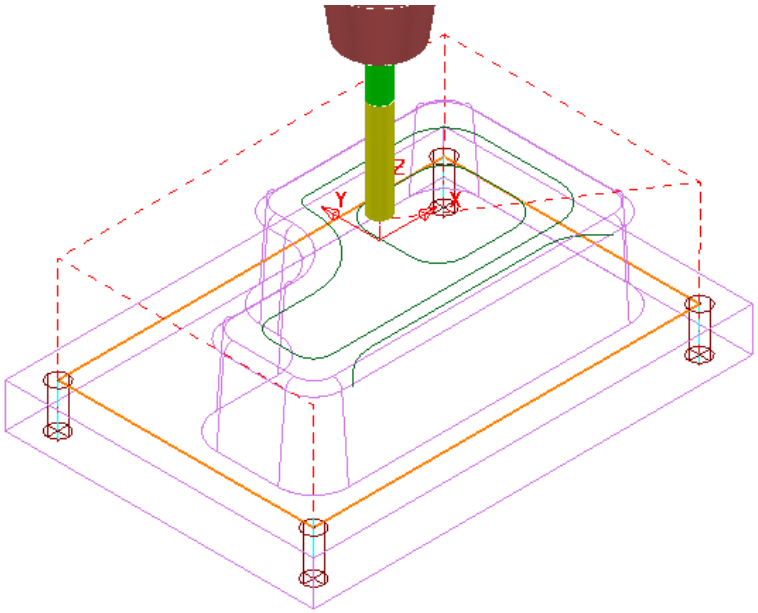


40 Activate the Dia 12 Drill named **Drill15**.

41 Input the data and settings exactly as shown in the following image.




42 **Calculate** the Drilling toolpath and then **Close** the above dialog.




**Save the Project** to update:-

**….\COURSEWORK\PowerMill_Projects\2DCurveMachining-2**


Copyright © Autodesk