Delcam Advanced Manufacturing Solutions

PowerMILL 2012 R2

Getting Started

PowerMILL 2012 R2

Getting Started

Release issue 1

PowerMILL

Acknowledgements

Patents

The Raceline smoothing functionality is subject to patent applications.

Patent granted: GB 2374562 Improvements Relating to Machine Tools

Patent granted: US 6,832,876 Machine Tools

Some of the functionality of the ViewMill and Simulation modules of PowerMILL is subject to patent applications.

Patent granted: GB 2 423 592 Surface Finish Prediction

Licenses

Intelligent cursor licensed under U.S. patent numbers 5,123,087 and 5,371,845 (Ashlar Inc.)

PowerMILL 2012 R2. Published on 01 February 2012

Contents

Introduction

PowerMILL is the world's leading specialist NC CAM software for manufacturing complex shapes typically found in the toolmaking, automotive, and aerospace industries.

Key features:

- Wide range of milling strategies which include high-efficiency roughing, high-speed finishing, and 5-axis machining techniques.
- Fast calculation times for toolpath creation and postprocessing.
- Powerful editing tools to ensure optimum performance on the machine tool.

This **Getting Started** guide provides step-by-step instructions that highlight some of the features of this versatile software.

It is assumed that you know how to operate your machine tool, select suitable tools, and cutting conditions. ***If you are unsure about any aspect of operating your machine tool, consult an expert or seek advice from your machine tool supplier****.*

The machining parameters used in examples in this guide have been selected to illustrate the effects of different commands and options in PowerMILL. ***The values given are not necessarily suitable for cutting on a CNC machine. If you wish to machine any parts based on the examples given, carefully review and adjust the parameters to ensure safe cutting conditions****.*

Setting up your working directories

To make project management easier, it is recommended that you define a home folder and default directory paths before starting any work with PowerMILL:

- Setting up a Home folder in Windows 7 for user-defined macros (see page 2).
- Specifying PowerMILL default directory paths (see page 3).
- PowerMILL temporary file area (see page 4).

Setting up a Home folder in Windows 7 for user-defined macros

PowerMILL checks for the value defined in the Windows environment variable **Home** for directions to the user-defined macros folder. Decide where you want the **Home** environment variable to point to, for example, **E:\PowerMILL_Projects**, and set up a Windows environment variable:

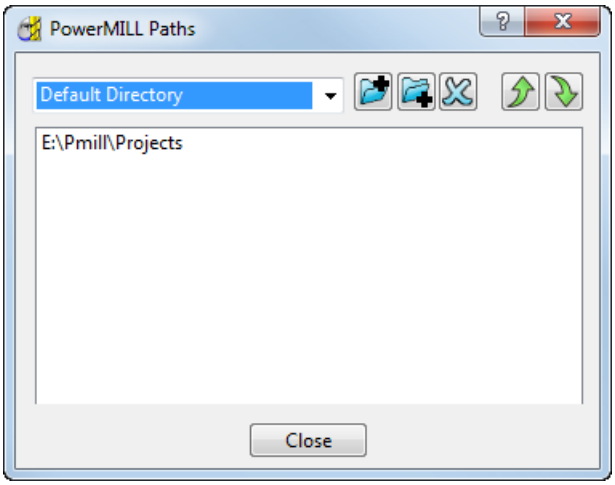
1. Open **Windows Control Panel** and click **System > Advanced system settings**. This displays the **System Properties** dialog.
2. Click the **Advanced** tab.
3. Click **Environment Variables**.
4. To add a new variable name and value, click **New** to display the **New User Variable** dialog.
   a. In the **Variable Name** field, enter **Home**.
   b. In the **Variable Value** field, enter the path where you want your home folder to be. For example, **E:\PowerMILL_Projects**.
5. Click **OK** in all open dialogs to save changes and close them.
6. Create a folder called **pmill** inside the **Home** directory. For example, **E:\PowerMILL_Projects\pmill**.

When user-defined macros are created or called, PowerMILL automatically locates the user-created macros in this folder.

Specifying PowerMILL Default Directory paths

To specify the **Default Directory** path in PowerMILL:

1. From the **Menu** bar, select **Tools > Customise Paths**. This displays the **PowerMILL Paths** dialog.
2. Select **Default Directory** from the drop-down list.



3. To add a **Default Directory** path, click  , and use the **Select Path** dialog to select the desired location. The path is added to the list. You can add multiple paths to the **Default Directory** list.

*You can have only one active **Default Directory** path. PowerMILL allows you to add multiple paths to the **Default Directory** list to help you efficiently organise multiple projects across different folders. To change the **Default Directory** load order, select the directory path you want to change, and use the  and  buttons to promote or demote the path.*

4. Click **Close**.

Your default working directory is changed to the directory path you selected and all your file operations (**Open Project**, **Save Project**, **Import Model**, **Export Model**) are changed to this location.

Specifying PowerMILL temporary file area

By default, PowerMILL projects use the default Windows **temp** folder defined in your **Windows System Properties** as the temporary area.

To define a custom PowerMILL temporary file area:

1. Right-click **My Computer**, and then click **Properties**. This displays the **System Properties** dialog.
2. Click the **Advanced** tab.
3. Click **Environment variables**.
4. Click **New** to add a new variable name and value.
   a. In the **Variable Name** field, enter **DELCAM_POWERMILL_USER_TEMPDIR**.
   b. In the **Variable Value** field, enter the path where you want your PowerMILL temporary folder to be. For example, **E:\PowerMILL_Projects\TemporaryFiles**.
5. Click **OK** in all open dialogs to save changes and close them.
6. Create the temporary folder in Windows to the path where you specified the variable. For example, **E:\PowerMILL_Projects\TemporaryFiles**.

If there are any problems with the variable you have specified, PowerMILL reverts to the default Windows temporary file area.

Start and close PowerMILL

To start PowerMILL:

- Select **Start > All Programs > Delcam > PowerMILL > PowerMILL2012 R2**

  or

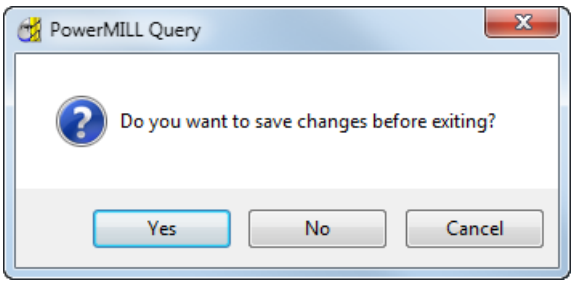
- Double-click the PowerMILL shortcut on your desktop:

This displays the **Startup** (see page 6) window.

To exit PowerMILL:

- Select **File > Exit** from the menu.
- Click the **Close** button in the top right-hand corner of the PowerMILL window.

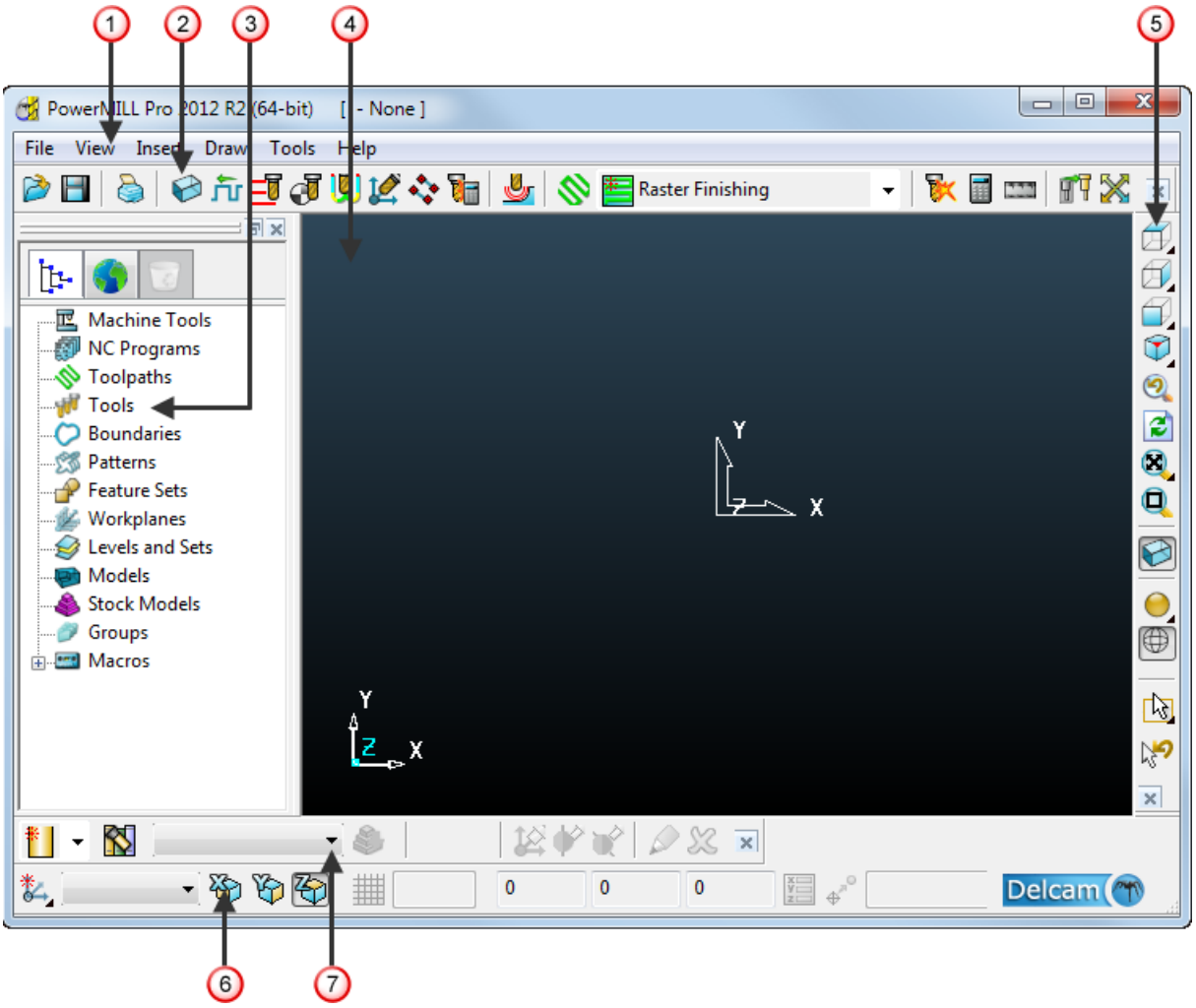
If you haven't saved your project, the following message is displayed:



Click **Yes** to save the project before exiting from PowerMILL.

Startup window

The following window is displayed on starting PowerMILL:



The screen is divided into the following main areas:

1 - **Menu bar** provides access to a number of menus. Selecting a menu, such as **File**, opens a list of associated commands and sub-menus. Sub-menus are indicated by a small arrow to the right of the text. For example, selecting **File > Recent Projects** ▸ displays a list of recently used projects.

2 - **Main toolbar** provides quick access to the most commonly used commands in PowerMILL.

3 - **Explorer** provides control over all PowerMILL entities.

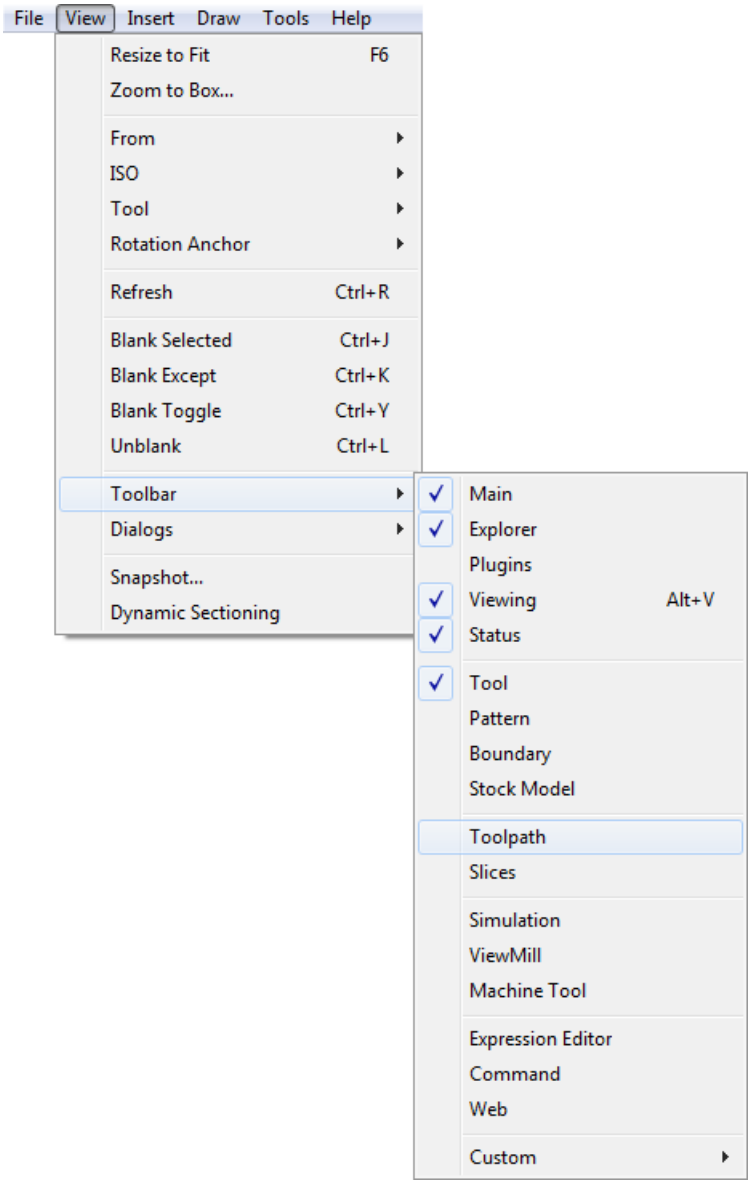
4 - **Graphics window** is the working area of the screen.

5 - **View toolbar** provides quick access to standard views and shading options in PowerMILL.

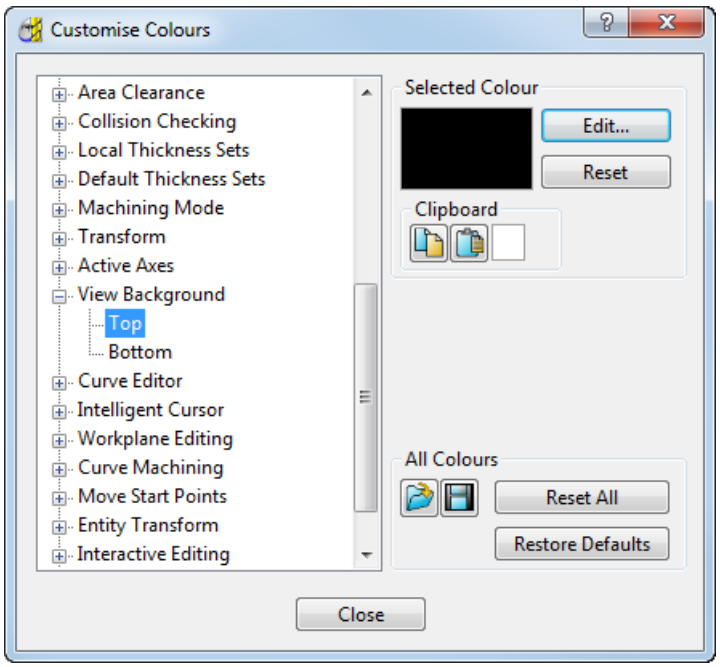
6 - **Status and Information toolbar** enables you to create and activate workplanes, display various preset fields and display user defined fields. If you hover the cursor over a button, help is displayed rather than the information toolbar. The help can be, for example, a brief description of the item beneath the cursor, or information about the calculation that is in progress.

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

*The other toolbars are not factory defaults, and may not be shown at startup. To display any of these, select them using the **View > Toolbar** option; for example, choose **View > Toolbar > Toolpath** to display the **Toolpath** toolbar.*



*If you want to change background colours, select **Tools > Customise Colours** and choose **View Background**. You can change the **Top** and/or **Bottom** colours and **Reset** them or **Restore Defaults** if you change your mind afterwards.*

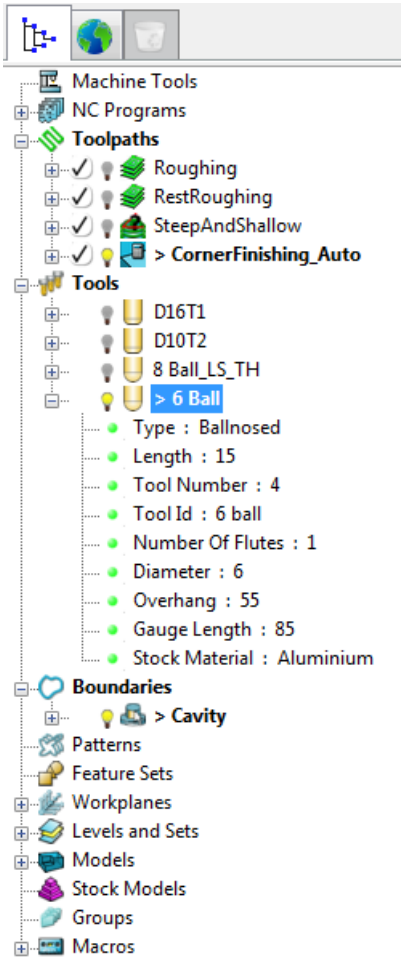


*PowerMILL remembers your toolbar and colour selections from one session to the next. For example, if you have the **Toolpath** toolbar displayed when you exit a session, it appears the next time you start PowerMILL.*

Explorer

The explorer  pane on the left of the graphics window controls the display and status of all PowerMILL entities, including NC programs, toolpaths, tools, and so on.

The following is an example of the explorer showing various entities and associated controls:



- Click ⊞ to expand or ⊟ to collapse the tree to view or hide subentries.
- Light bulb icons, , , work as a three-way switch for drawn entities, such as toolpaths and tools:
  - is the default setting when an entity is first created, indicating that it is currently **active**.
  - Clicking changes the icon to to indicate that the entity is **drawn**.

- Clicking changes the icon to to indicate that the entity is now hidden, or **undrawn**. These two icons, and , work together as a toggle. If an undrawn entity is made active again after being inactive, the icon reverts to the initial .

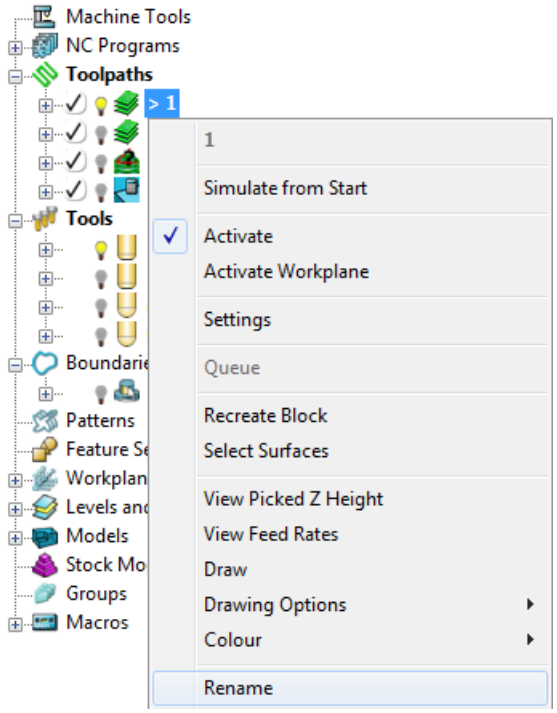
*An entity is always drawn when it is first activated or re-activated. When active, it is also displayed in **bold** and preceded by a ">" in the explorer.*

*In this example, **> 6 Ball** is the active undrawn tool, and **> CornerFinishing_Auto** is the active drawn toolpath.*

Renaming an entity

PowerMILL entities are given numeric default names on creation. So, for example, the first toolpath you create is called "**1**", the second "**2**", and so on.

You can rename an entity to give it a more meaningful name. To rename an entity, right-click an entity and select **Rename** from the context menu.



*To quickly rename an entity, select the entity in the explorer, and then single-click it. The name is now highlighted, and has a box around it: 1. Enter the new name, for example **RestRough**, and press **Enter** > RestRough.*

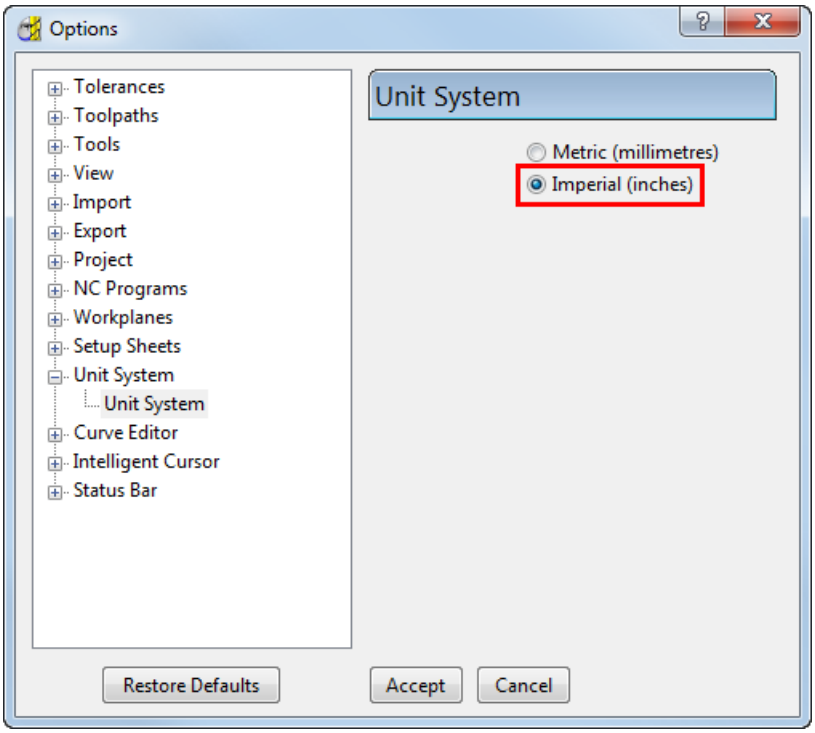
Unit System

By default, PowerMILL uses metric units. You can change the unit system before any project entities have been created.

This guide uses metric (mm) units.

To change the default units before you start work, for example, from **Metric** to **Imperial**:

1. From the **Tools** menu, select **Options**. This displays the **Options** dialog.
2. Select **Unit System** and click ⊞ to expand the tree to view available options.
3. Select the **Unit System** sub-item. This displays the **Unit System** page.
4. Select **Imperial (inches)**.



5. Click **Accept** to save changes.

Changing the unit system automatically resets all the default values on the dialogs to match the selected unit system. PowerMILL does not convert the values between the unit systems; instead, it chooses an appropriate set of default values (metric or imperial).

Accessing Help

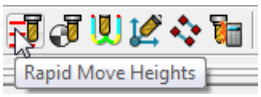
PowerMILL offers the following forms of help:

- Tooltips (see page 12)
- Status bar help (see page 12)
- Context-sensitive online help (see page 12)
- Manuals (see page 13)

Tooltips

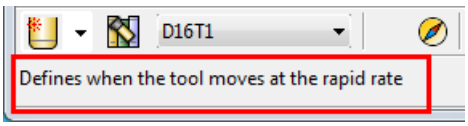
Tooltips are displayed by hovering the cursor over the required buttons.

Each tooltip gives a brief description of the function of the associated button, for example:



Status bar help

When you hover the cursor over a button, help is displayed in the **Status** bar at the bottom of the screen. So, when you place your cursor over the **Rapid Move Heights** button, you see the following help in the **Status** bar at the bottom of the screen as well as the tooltip:



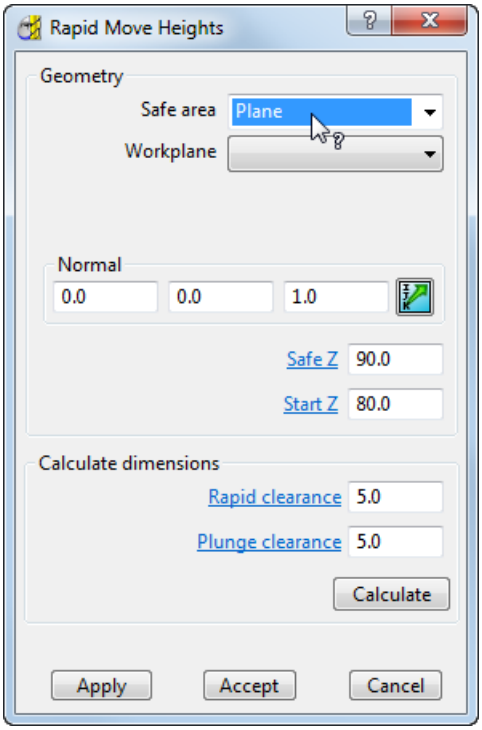
Context-sensitive online help

Pressing the **F1** key displays help for the area of the screen that has current focus, such as a display pane (explorer, graphics window), a dialog, or a dialog tab. Where there is no focus, the contents page for the main **Help** is displayed, so that you can navigate from there.

To display the help topic relating to an element on the main PowerMILL screen (a toolbar button, an explorer entity), hold down the **Shift** key and press **F1**. When the cursor changes to , click the item of interest to open the help topic: .

*If you want to cancel context-sensitive mode, press **Esc**.*

To display context-sensitive help for an element within a PowerMILL dialog (such as a field or a button), click  and when the cursor changes to  , click the item to open the associated help topic:



Manuals

The PowerMILL manuals offer additional help.

The documents are:

- **What's New** and **Getting Started** manuals.
- Online reference help.

Cavity mold example

This example shows you how to generate and output the basic toolpaths used to machine a forging die mold.

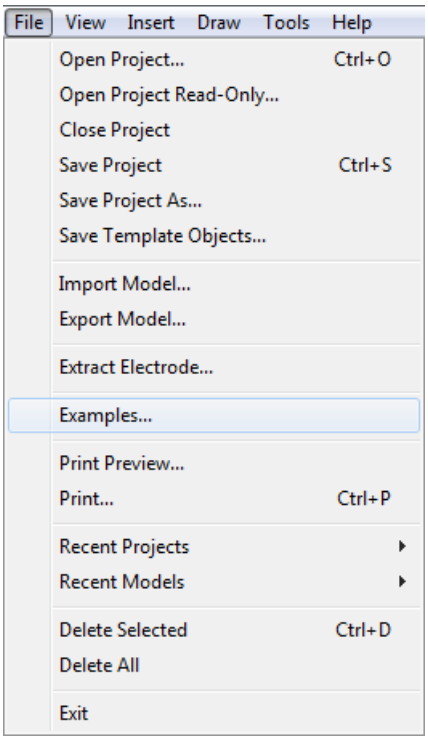
1. Start PowerMILL (see page 5).
2. Load the model (see page 15).
3. Save the project (see page 19).
4. View the model and block (see page 20).
5. Create an NC program (see page 24).
6. Create, simulate, and output each toolpath:
   - **Roughing** (see page 29) - to rapidly remove the majority of the excess material using the **Offset all** style in the **Model Area Clearance** strategy.
   - **Rest roughing** (see page 43) - to clear additional material using a smaller tool (the **Model Area Clearance** strategy is used again; this time only material not removed by the previous toolpath is machined).
   - **Finishing toolpath** (see page 52) - to generate steep and shallow moves within a specified boundary.
   - **Corner Finishing** (see page 69) - to clean up material that previous toolpaths were unable to finish, particularly in corners between non-tangential surfaces.

Loading the model

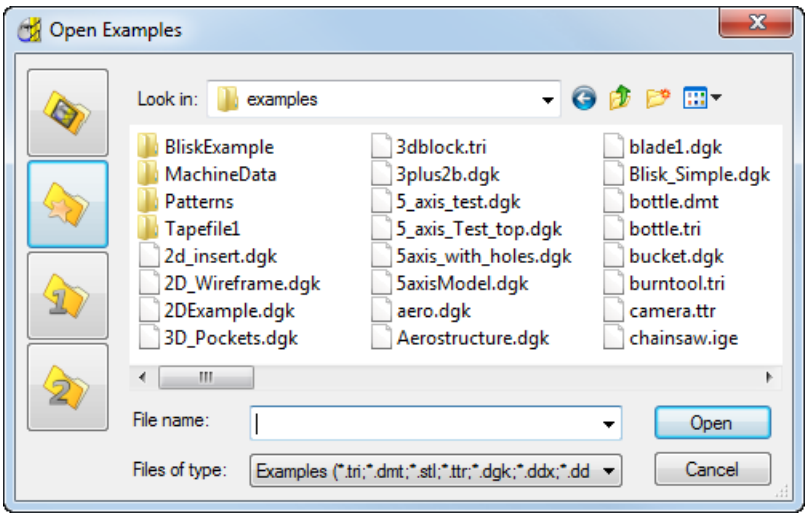
The model you will work on is a cavity mold in the **Examples** folder.

To load the model:

1 Select **File > Examples**.



The **Open Examples** dialog is displayed, automatically opening the drive and folder where the tutorial files are installed:

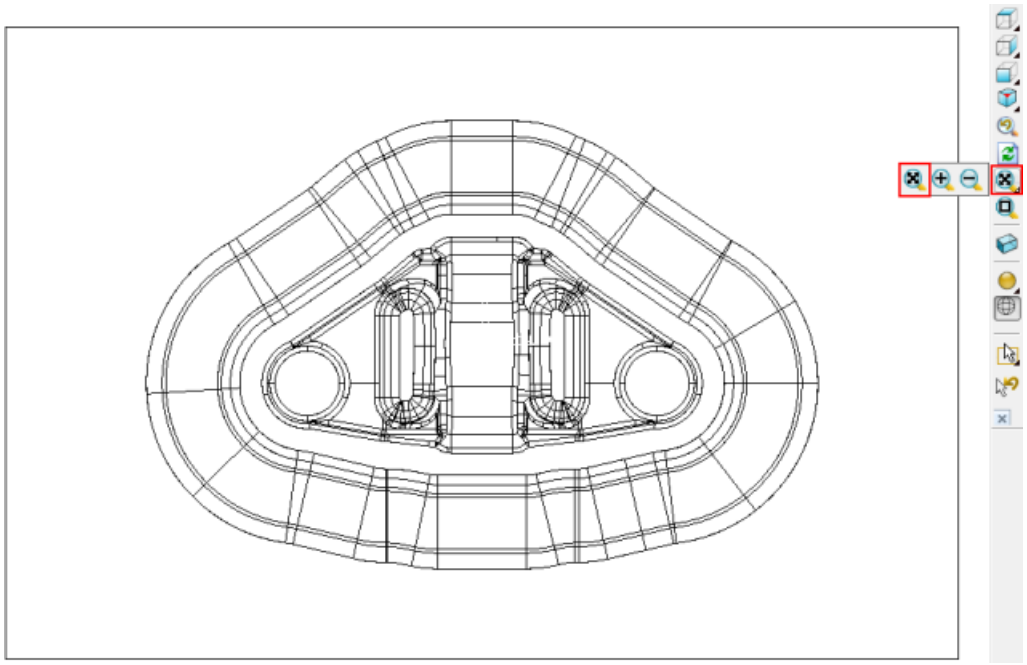


2 In the **Files of type** field, select **Delcam Geometry (*.dgk)** from the drop-down list.

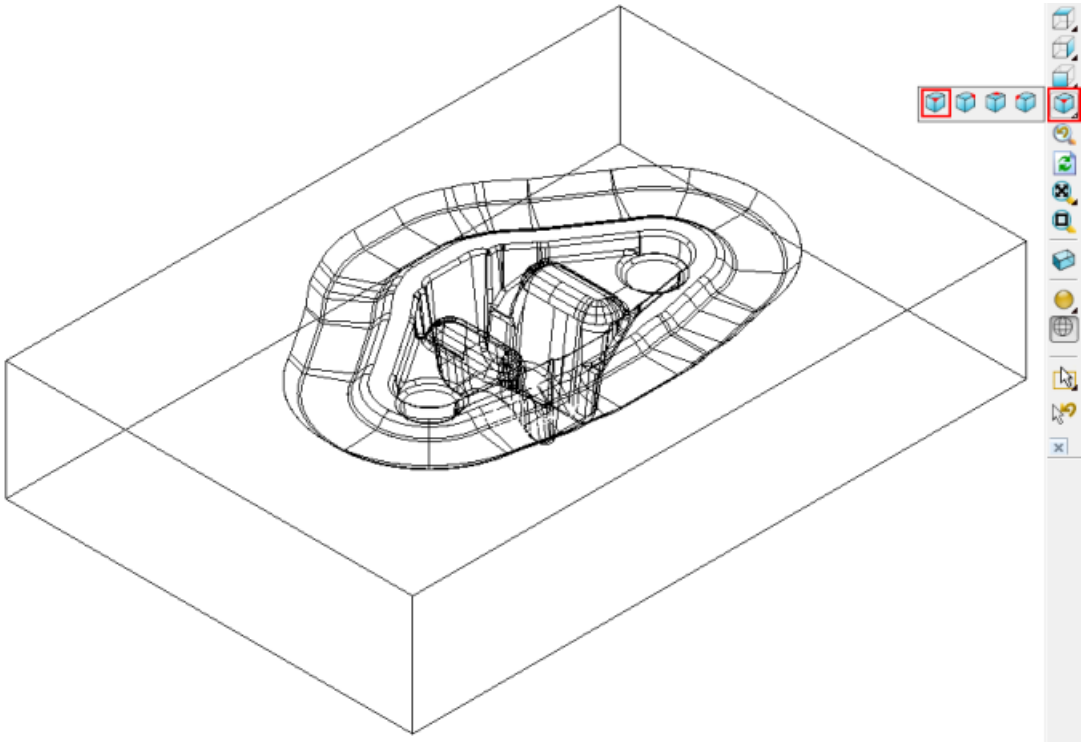
3 Select the cavity mold called **die.dgk,** and then click **Open**.

PowerMILL loads the file.

4 Click the **Resize to Fit** button on the **View** toolbar to fit the file within the display.

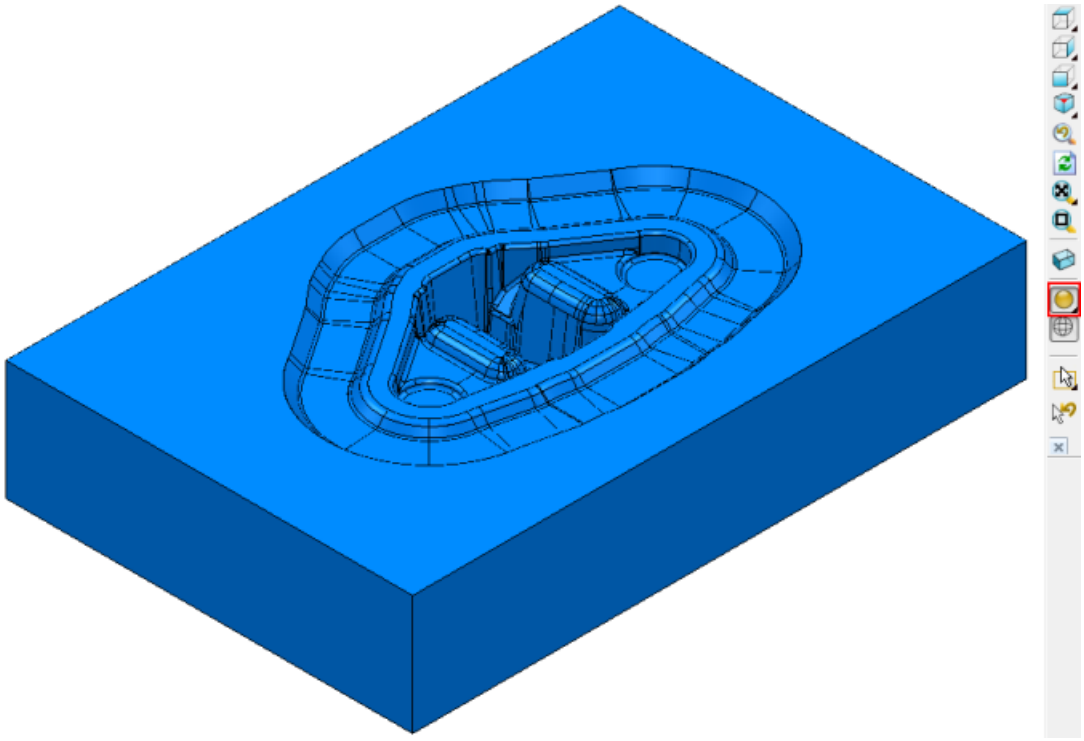


5 To change the display to an isometric view, select the **ISO 1** button on the same toolbar. The model looks like this:



More information on view manipulation is provided later *(see page 20).*

6 The die is currently shown with only the wireframe selected. To add shading, click the **Plain Shade** button on the **View** toolbar.



7 Remove the wireframe by clicking the **Wireframe** button near the bottom of the **View** toolbar.

*Click the **Wireframe** or **Plain Shade** buttons to toggle between **Wireframe**, **Plain Shade**, **Wireframe and Shaded**, and **No Model** views.*

For this exercise, keep the model shaded.

Defining the block around the die

The block defines the stock size. The part is then machined from the block. In this case, the block is a rectangular cuboid.

To define the block:

1 Click the **Block** button on the **Main** toolbar.

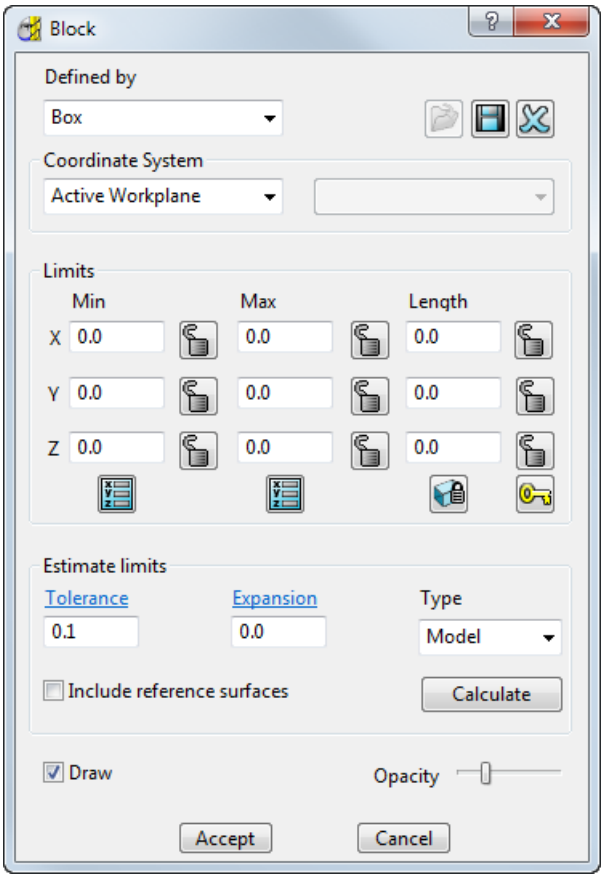
2 In the **Block** dialog, check that:

a **Defined by** is set to **Box**.

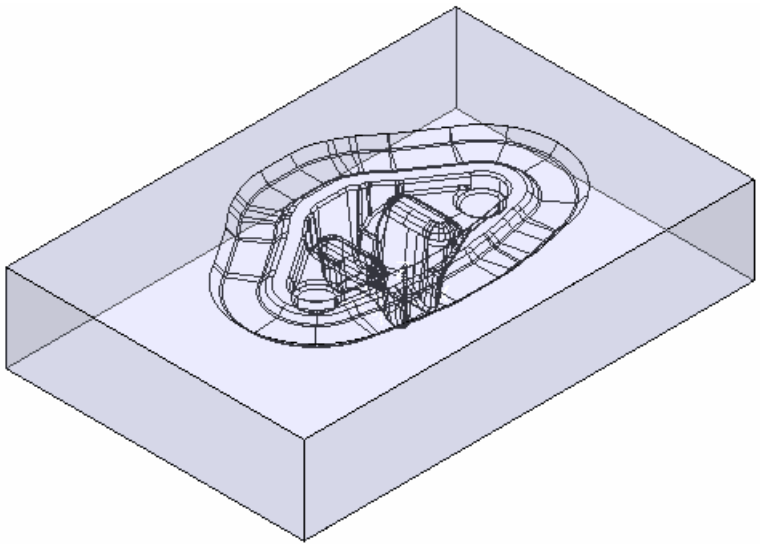
b **Coordinate System** is set to **Active Workplane**.

c In the **Estimate Limits** area, **Expansion** is set to **0** and **Type** to **Model**.

d **Draw** is selected.



3 Click **Calculate** to define a cuboid enclosing the die. Click **Accept** to close the dialog.



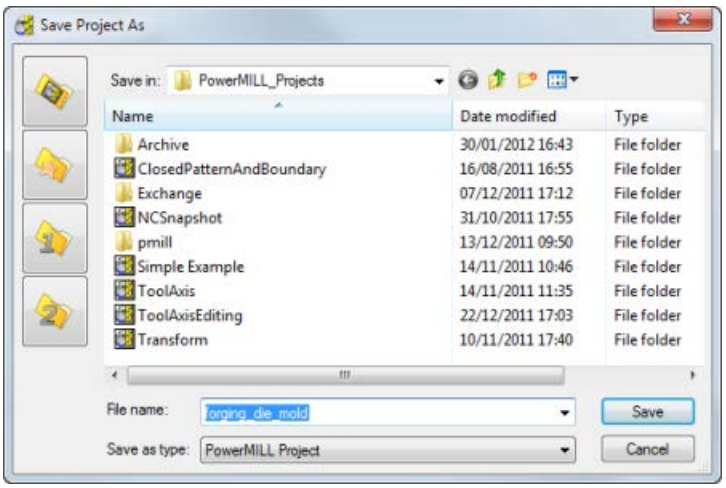
*To toggle the display of the block on and off, click the **Block** button on the **View** toolbar.*

Saving the project for the first time

PowerMILL saves all the entities, together with a copy of the model, as a single project.

To save a project:

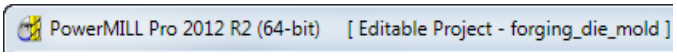
1 Click the **Save** button on the **Main** toolbar. Since you have not previously saved the project, the **Save Project As** dialog is displayed.



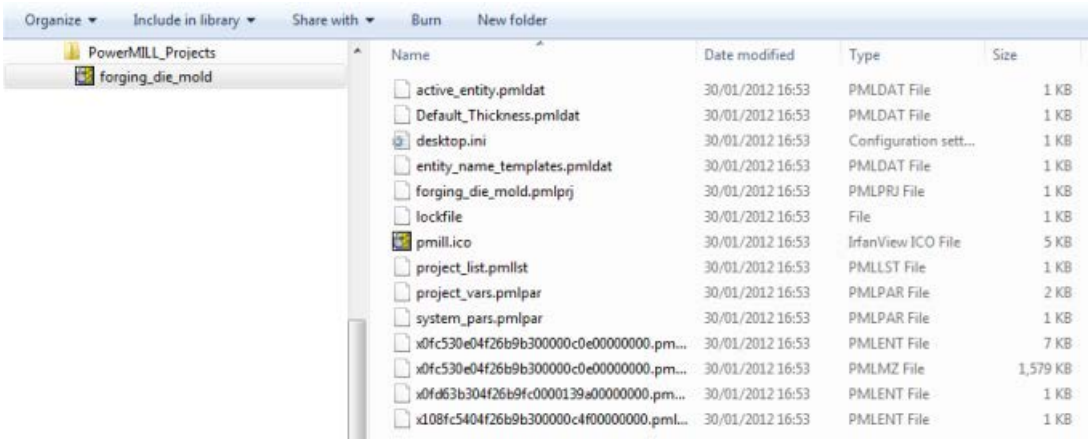
2 Move to the directory that you created for your projects (see page 2), and enter a name for your project in the **File name** field, for example **forging_die_mold**.

3 Click **Save**.

The PowerMILL window header now reflects the name of the project.



An associated file structure is created:



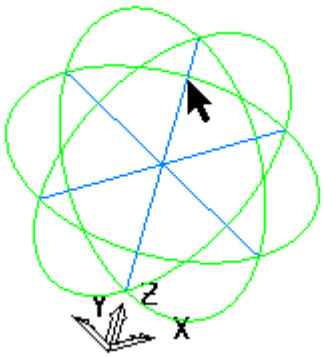
Do not manually edit these files *- you will corrupt your data.*

*From now on, click **Save** on the **Main** toolbar to update the saved version of the project.*

Viewing the model and block

To look at a specific area of the model and block you can either select one of the predefined views (see page 20) on the **View** toolbar, or use a mouse.

It is recommended that you use a three-button mouse. By holding down the middle button and moving the mouse pointer in the graphics area, you can control the view of the model. When you are moving the mouse (with the middle button pressed) the trackball is visible on the screen:



Start with the cursor in the middle of the window and press the middle mouse button, then drag the cursor up the screen. The image moves as you move the mouse.

Predefined views

The **View** toolbar is loaded by default when PowerMILL is started and is often the best starting point for dynamically manipulating model views. There are several fixed viewing directions available from the **View** toolbar.

*If the **View** toolbar is not currently displayed, select **View > Toolbar > Viewing** from the menu.*

View Along options allow you to orientate the view so that it is along the X, Y, or Z axis. Hover on any of these buttons to display a 2D view toolbar which has additional view options along the primary axes.

Use the **ISO View** option to change the view angle to any of the isometric views. Hover on the button to display other isometric view options.

Isometric views are often the best starting point for dynamically manipulating the view using the mouse.

Previous View option takes you back to the previous view of the entities.

Refresh option redraws all the currently displayed components, repairing any damage to the current view. This can be useful if the view loses definition as items are created, modified, or deleted on the screen.

Resize to Fit (see page 22) adjusts the view to fit the screen. Depending on the current view, PowerMILL zooms in or out to adjust the view. Hover on the button to view the **Zoom In** (see page 22) and **Zoom Out** (see page 22) buttons.

Use the **Zoom to Box** (see page 22) option to zoom in on a specific area of the model. Click and drag the mouse to create a box over a part of the model to zoom in. PowerMILL adjusts the view to zoom in on the selected boxed area.

Use the **Block** option to show or hide the block.

Use the **Plain Shade** option to show or hide the shaded representation of the model.

Hover on the **Plain Shade** button to show the other shading options available on the **Shading** toolbar.

Use the **Wireframe** option to show or hide the wireframe representation of the model.

Use the **Box Selection** mode to select entities with the mouse. Click and drag the mouse to create a box over the entities you want to select.

To deselect entities from a group of selected entities, hold down the **Ctrl** key and click the entities you want to deselect.

Use the **Drag Selection** button to select multiple entities with the mouse. Hover on the **Box Selection**  button to display the **Drag Selection**  button. Click the **Drag Selection**  button and click and select the relevant entities.

Use the **Last Selection** option to revert to the previous selection made with the **Box Selection** or **Drag Selection** options.

Zoom

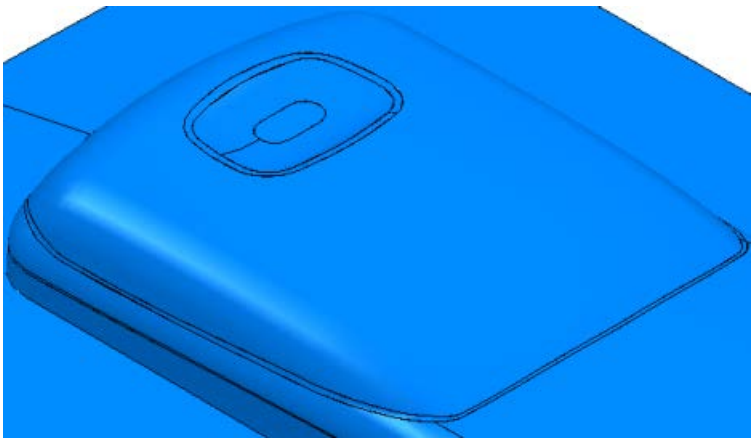
You can resize the model using the viewing options available on the **View** toolbar:

The **Zoom** options and control the zoom factor of the image on the screen.

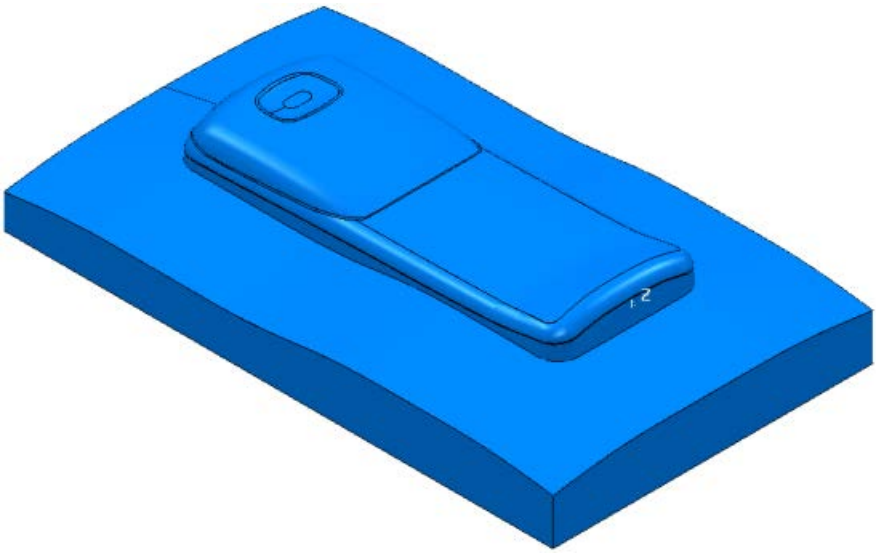
This is a pull-out toolbar. Hovering on the currently selected Zoom button (in this case, **Resize to Fit**) displays the **Zoom** toolbar.

Resize to Fit - zooms the image so that it just fits in the window.

It converts this:



into this:



This zooms in or out as required. So, it also zooms a microscopic image on the screen to one that just fits inside the window.

*If you can't see your graphics, click the **Resize to Fit** button on the **View** toolbar to fit the graphics to the middle of the screen.*

Zoom In - doubles the size of the image. Click this button repeatedly until the required zoom factor is reached. The image is zoomed about the centre of the graphics window.

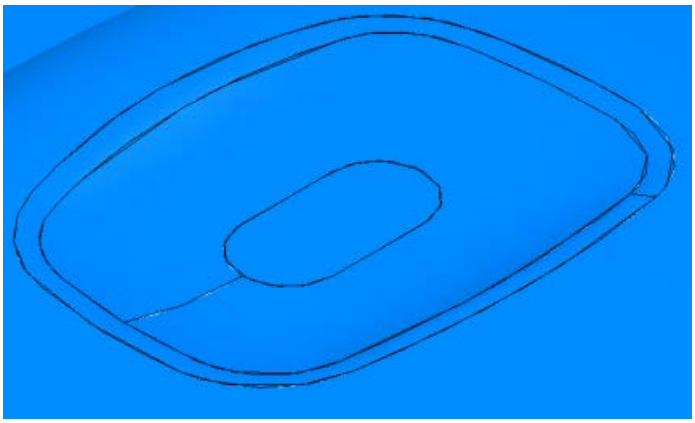
Zoom Out - halves the size of the image. Click this button repeatedly until the required zoom factor is reached. The image is zoomed about the centre of the graphics window.

Zoom to Box - zooms in to a specific area of the image. Click and then use the left mouse button to drag a zoom box on the image. The view is then zoomed to fit the boxed area.

It converts this:

1 - Zoom box

into this:



*You can also resize an image by holding down the **Ctrl** key and the middle (or right) mouse button, and then dragging the mouse up to **Zoom In** or down to **Zoom Out**.*

Panning

You can **Pan** the image by holding down the **Shift** key and the middle (or right) mouse button, and then dragging the mouse in the direction in which you want to move the image.

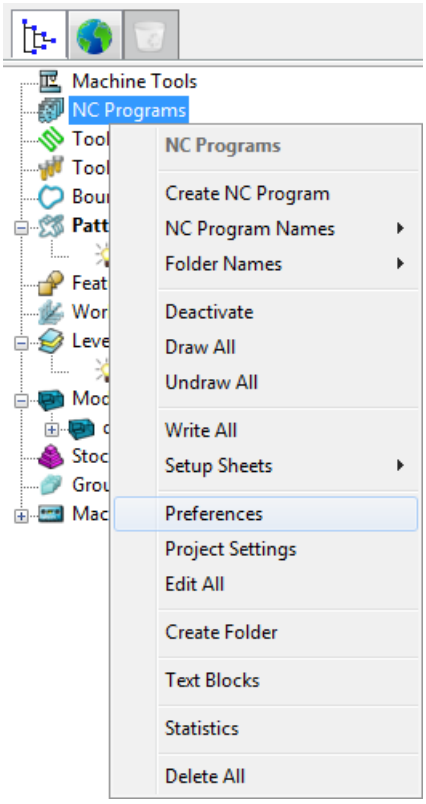
NC program

An NC program contains the commands and output settings that specify how the machine controller will machine the part.

It is advisable to create an NC program, and set the preferences for it before generating any toolpaths. However, it's just as easy to add generated toolpaths to the NC programs later (see page 79).

Overview Template

1 In explorer, right-click **NC Programs** and select **Preferences**.



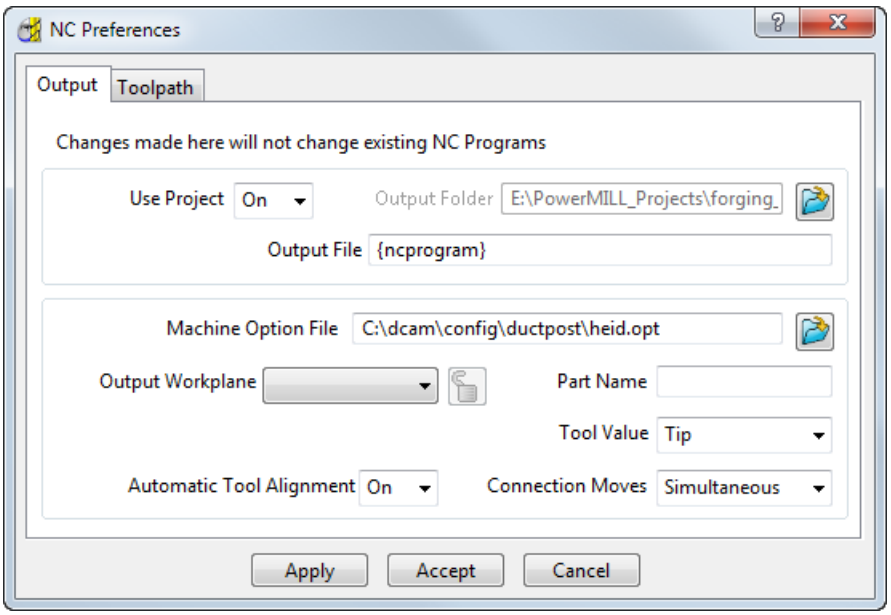
This displays the **NC Preferences** dialog.

2 On the **Output** tab of the **NC Preferences** dialog, select **Use Project - On** to write the NC program file into the PowerMILL project folder.

*Select **Use Project - Off** to write the NC program file to a different folder. You need to specify it in the **Output Folder** field.*

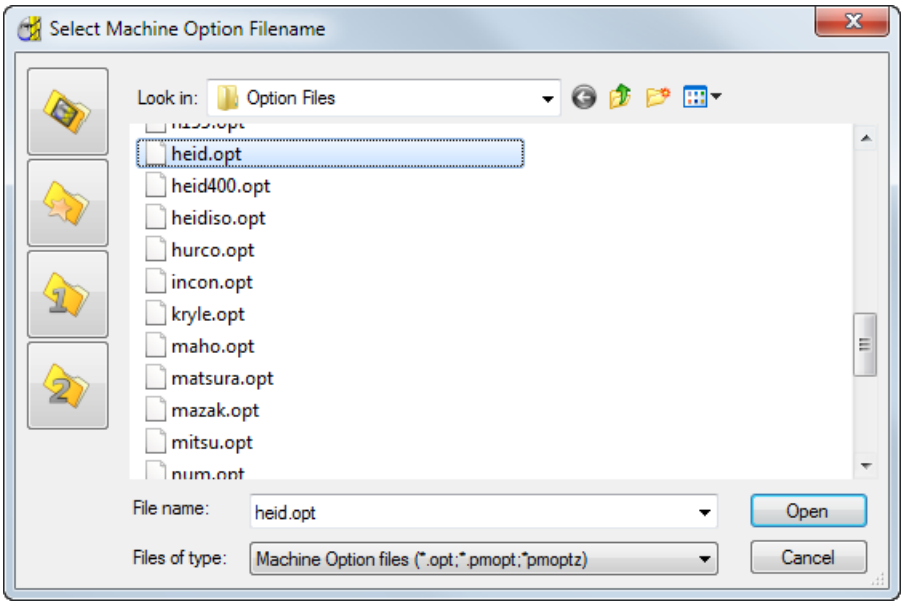
3 Enter the output file name to be used by default in the **Output File** field.

*The variable **%[ncprogram]** gives the output file the same name as the NC program. You can add your file name to this variable.*



4 Click  next to the **Machine Option File** field to open the **Select Machine Option Filename** dialog.

5 Browse to the folder where the option files are stored, and select the required machine option file (in this case **heid.opt**) and click **Open**.

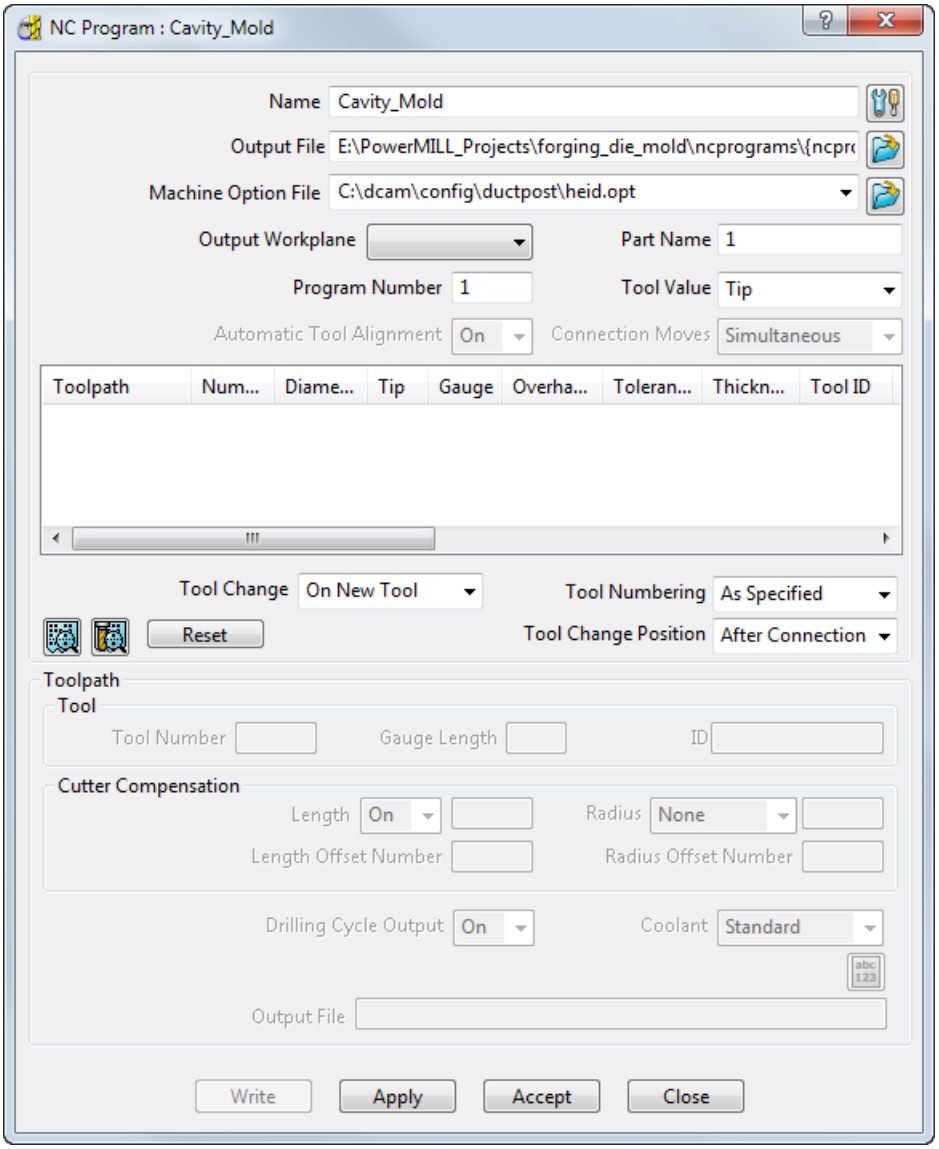


This closes the **Select Machine Option Filename** dialog and returns you to the **NC Preferences** dialog.

6 Click **Accept** to save your changes and close the dialog.

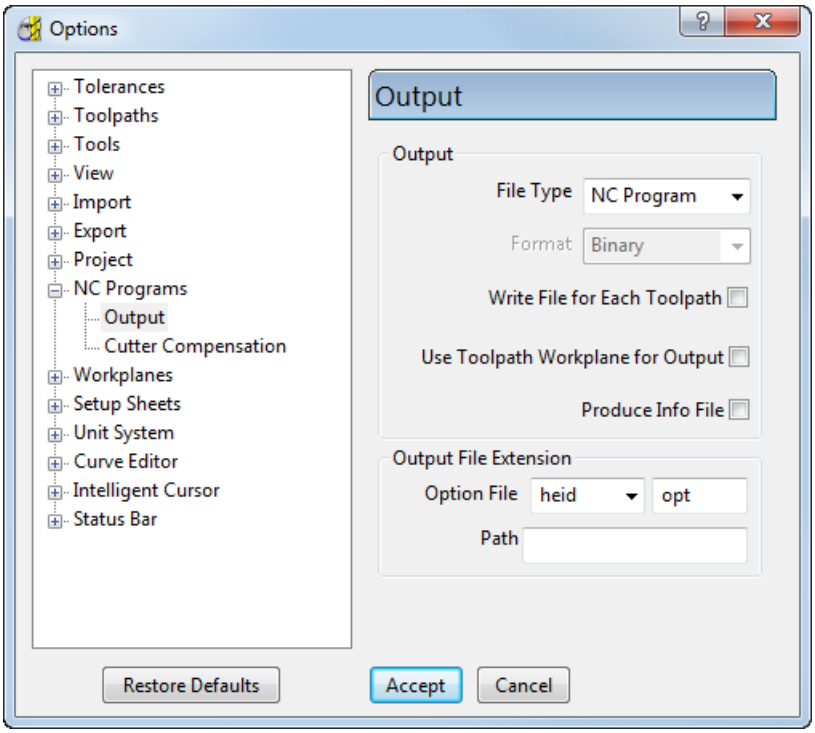
Creating an NC program

1. From the **NC Programs** context menu, select **Create NC Program**. This displays the **NC Program** dialog.
2. In the **Name** field, enter the name you want to call the NC program. For example **Cavity_Mold**.



3. In the **NC Program** dialog, click the **Options** button. The **Options** dialog is displayed.
4. On the **Output** tab:
   a. If selected, deselect the **Write File for Each Toolpath** option. This causes **Output File** to be displayed instead of **Root Name** at the top of the **NC Program** dialog.
   b. Select **Option File** as **heid.**
   c. Enter **Output File Extension** as **opt**.

d Click **Accept** to update and close the **Options** dialog.

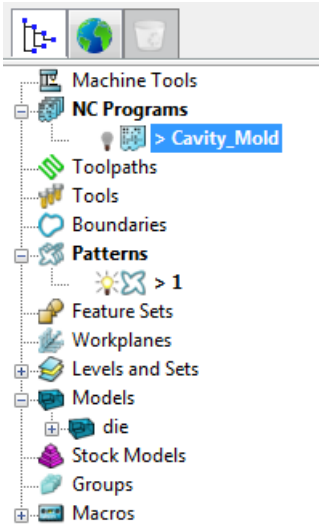


*If you want separate output files for each toolpath, select **Write File for Each Toolpath** option. This changes the **Output File** field to **Root Name** in the **NC Program** dialog and displays the default path and file name in the **Output File** field at the bottom of the dialog.*

The default path in the **Output File** field is based on the settings in the **NC Preferences** dialog. If you want to change this path, click the open folder button, browse to the appropriate folder, and enter the new file name (the file extension is determined by the settings you made in step 4b). If you used the **%[ncprogram]** variable in the **NC Preferences** dialog, hover on the displayed path to see how the file name will be resolved by PowerMILL.

5 Click **Accept** in the **NC Program** dialog to accept your selections and close the dialog.

6 To see the entity that you have just created, click ⊞ to expand the **NC Programs** node. The entity is automatically **active** (signified by **bold** text preceded by the **>** symbol).



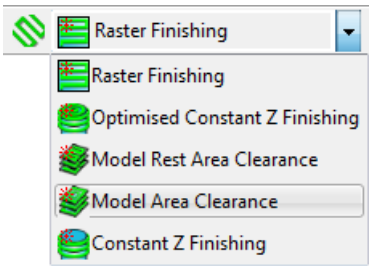
Any toolpaths you create are automatically added to the currently active NC program.

Creating a roughing toolpath

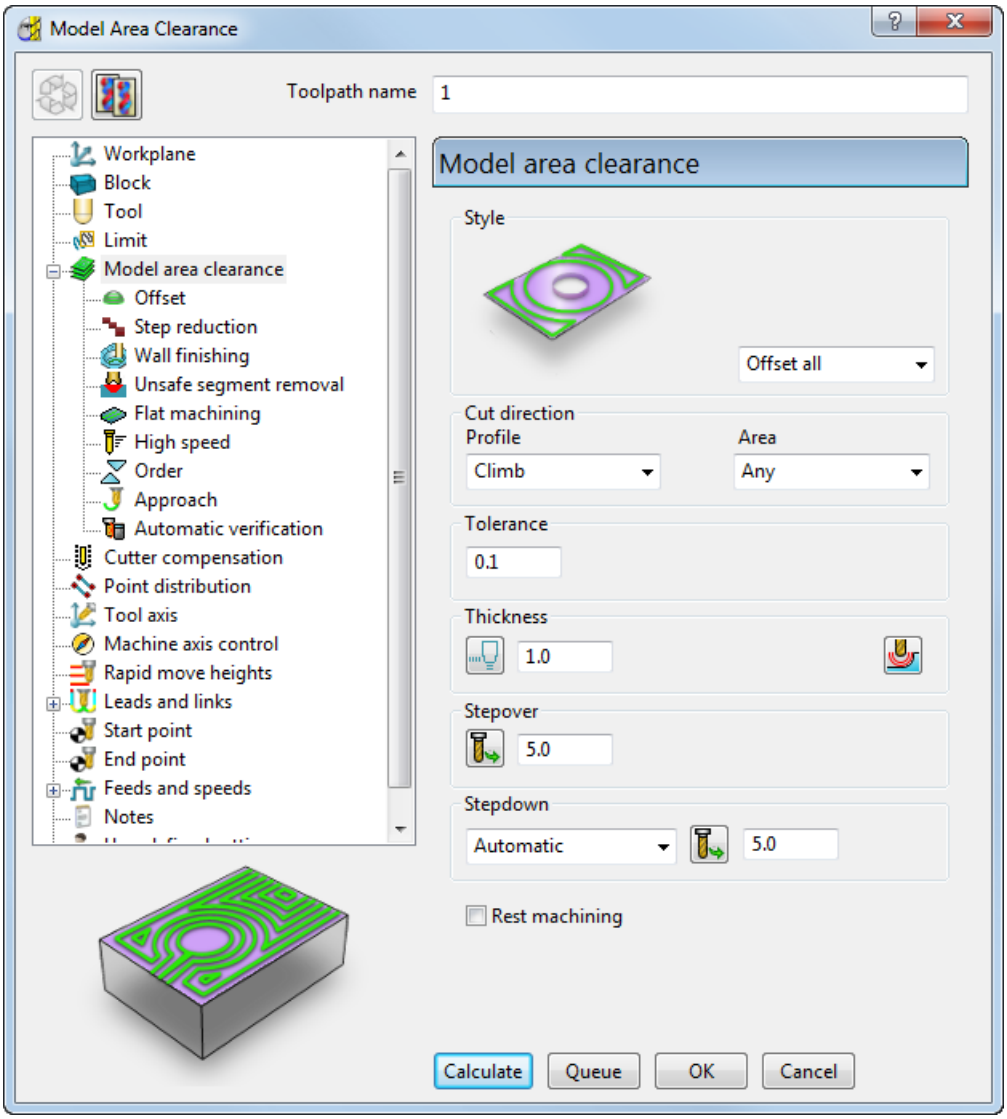
A **Model Area Clearance** strategy with the roughing **Style** set as **Offset all** is used to rapidly remove the majority of excess material on the model. This clears the area with contours that are generated by repeatedly offsetting the initial contour until no further offsetting is possible, and then steps down to the next level and repeats the offsetting until the bottom of the part is reached.

To create a roughing toolpath:

1 On the **Main** toolbar, from the **Create Toolpath** list, select **Model Area Clearance**.



This displays the **Model Area Clearance** dialog.



Use the various tabs to enter parameters required for the toolpath.

2 Give the toolpath an appropriate **Name**, for example **Roughing**.

3 Define the roughing tool geometry (see page 31).

4 Define the tolerances (see page 36).

5 Specify rapid move heights (see page 36).

6 Specify tool start points. (see page 37)

7 Generate the roughing toolpath (see page 38).

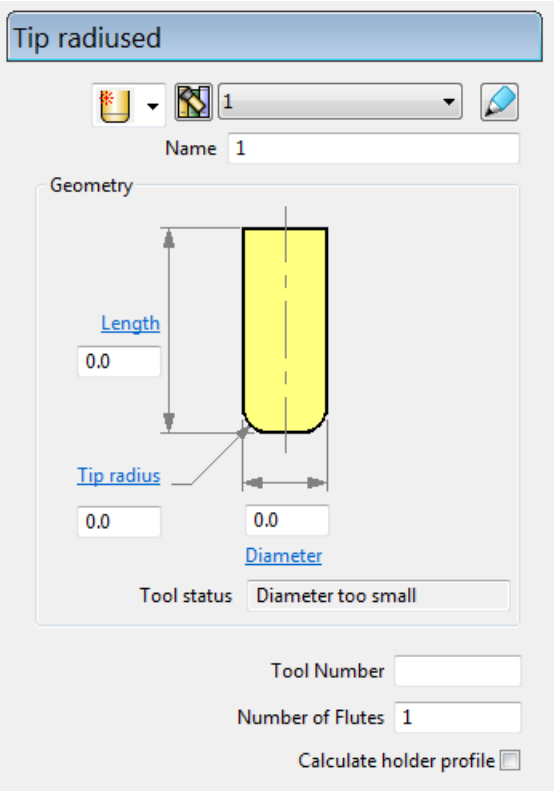
8 Display the roughing toolpath (see page 38).

9 Simulate the roughing toolpath (see page 41).

Defining the roughing tool geometry

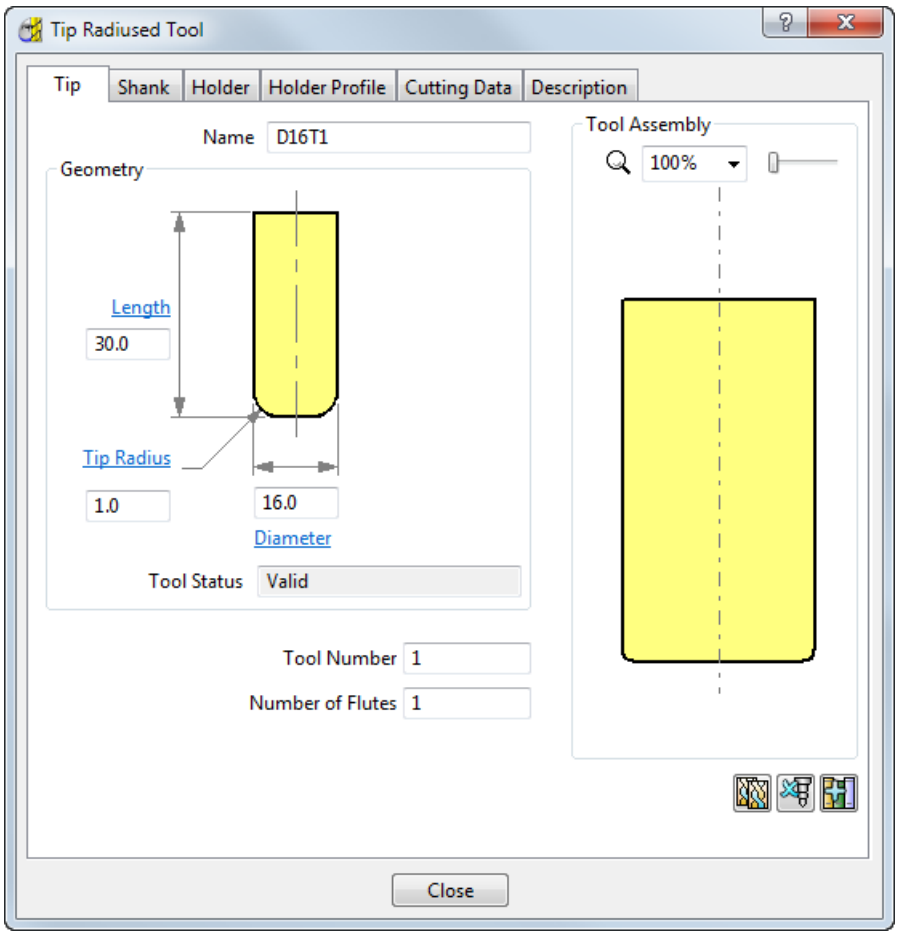
The next stage involves selecting a tool and defining its geometry. This example uses a **16** mm (**5/8** inch) tip-radiused tool.

1. In the **Model Area Clearance** strategy dialog, click Tool to select the **Tool** page.
2. On the **Tool** page:
   a. Click the arrow next to the **Create Tool** button in the **Tool** area.
   b. From the tool list, select to create a **Tip Radiused Tool**.
3. On the **Tool** page, click to display the **Tip Radiused Tool** dialog.



4. In the **Tip Radiused Tool** dialog, enter:
   a. **Name** - D16T1
   b. **Length** - 30 mm
   c. **Tip Radius** - 1 mm
   d. **Diameter** - 16 mm

e **Tool Number -** 1

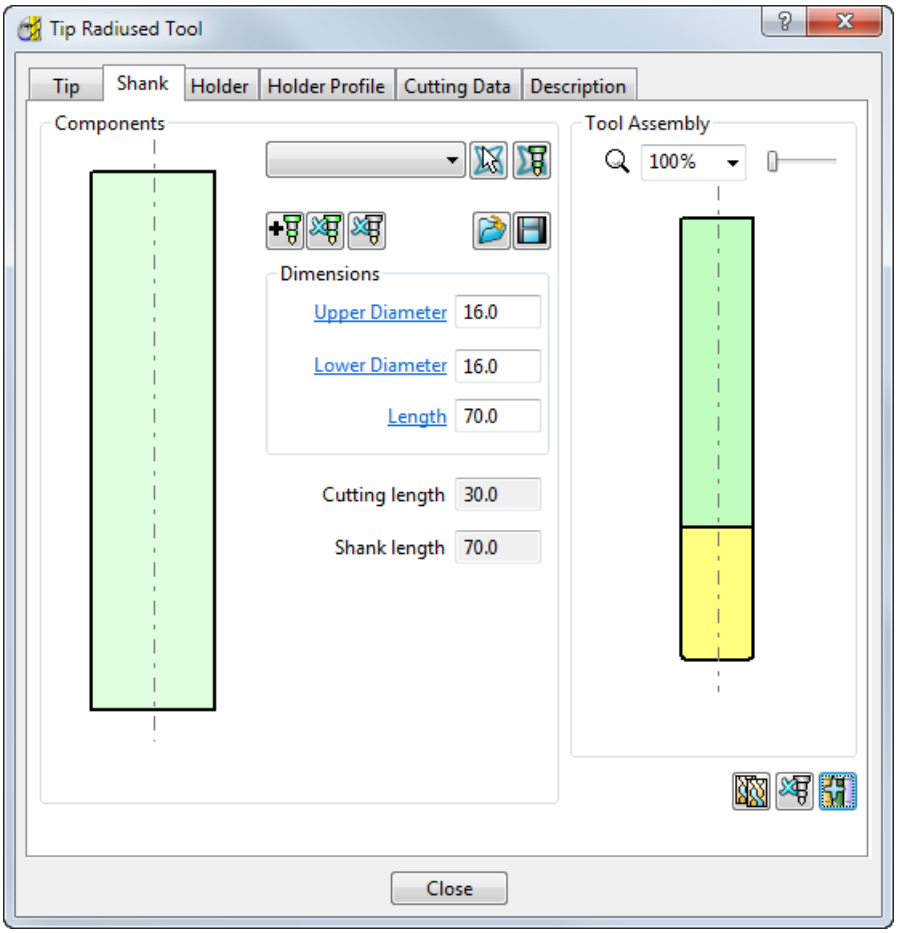


5 Select the **Shank** tab, click  to add a shank component. Enter:

a **Upper Diameter -** 16 mm

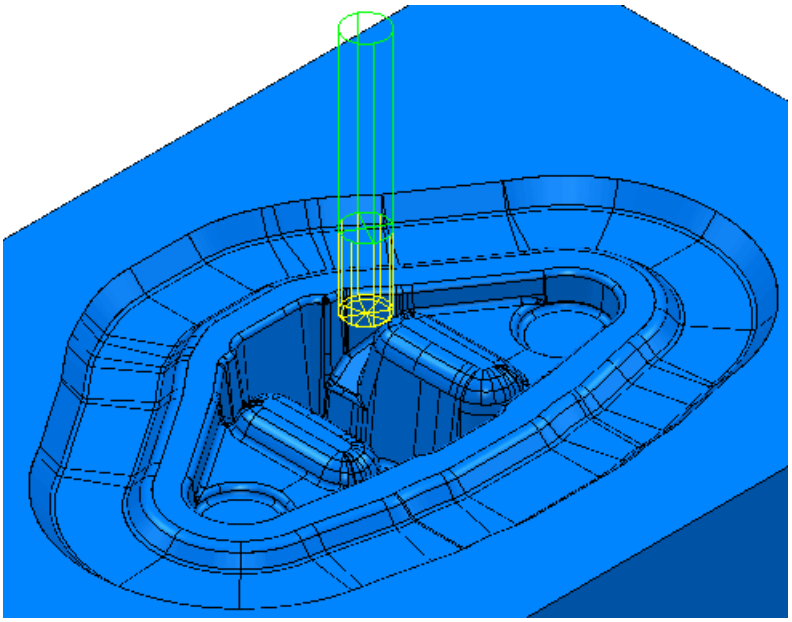
b **Length -** 70 mm

*The **Lower Diameter** automatically defaults to the **Upper Diameter**. This can be accepted for the current tool.*



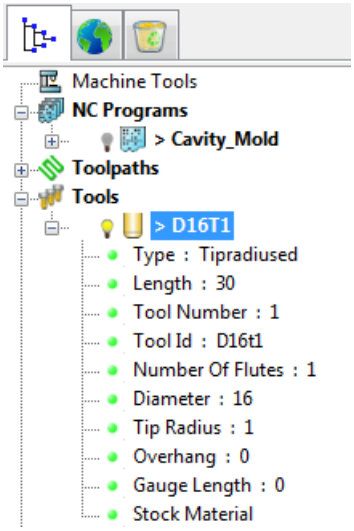
6 Click **Close**.

The tool is automatically aligned with the Z axis:

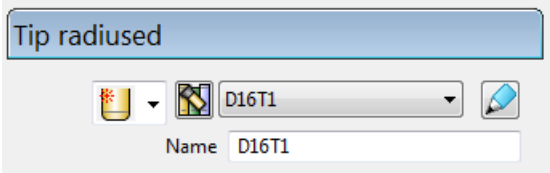


You can view and edit the created tool:

- In explorer, expand Tools to see the tool that you have created. Expand the tool node to see the specific tool's details.



- In the **Tool** tab on the **Model Area Clearance** dialog:

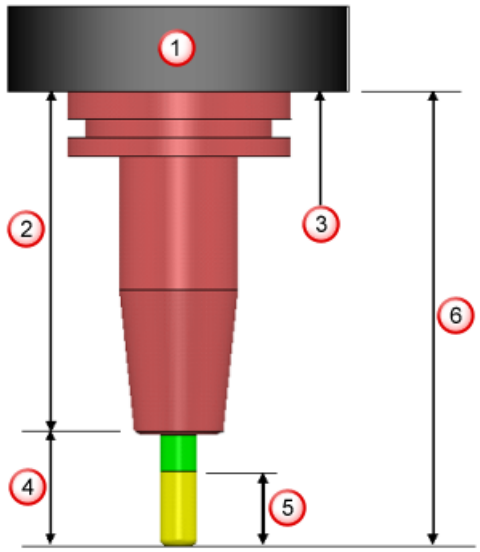


- On the **Tool** toolbar:

Tool length overview

The diagram below shows a cutting tool made up of a tip (yellow) and shank (green) mounted in a shrink-fit holder (red). The tool is mounted in the spindle of a machine (grey).

*The colours shown here correspond to those used in the PowerMILL **Tool** dialogs.*



1 - Spindle

2 - Holder length

3 - Gauge face

4 - Overhang

5 - Cutting length

6 - Gauge length

The **Cutting Length** represents the part of the cutter that removes material. This is set up in PowerMILL as the **Tip Length**.

The **Overhang** is the amount by which the cutter protrudes from the tool holder. This, typically, includes part of the **Shank Length**. The **Overhang** is fixed when the cutter is mounted in the holder.

*To obtain maximum tool life, the **Overhang** is typically kept to the minimum necessary to prevent the holder from hitting the part or unmachined stock.*

The **Holder Length** is the total length of all parts of the holder assembly that protrude from the spindle when the holder is mounted in the machine.

The **Gauge Length** is the total length of the cutter and holder assembly when it is mounted in the machine. It is measured from the tip of the tool to the **Gauge Face**, which is the ground face of the spindle.

Defining the tolerances

On the **Model area clearance** page of the strategy:

1. In **Tolerances**, enter **0.2** mm.
2. Enter a **Radial thickness**  of **0.5** mm.
3. Enter an **Axial thickness**  of **0.1** mm.
4. In **Stepover**, enter **7.0** mm.
5. In **Stepdown**, enter **4.0** mm.
6. In the stepdown list, select **Automatic**.

*The **Z Heights** are created automatically when the toolpath is calculated, and any existing **Z Heights** values are deleted.*

The accuracy of the machined part produced by PowerMILL is limited by the accuracy of the model read into the program. The original model must have been produced to an adequate tolerance.

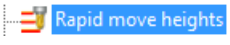
Specifying rapid move heights

The heights at which the tool can move safely without hitting the part or clamps are called rapid move heights.

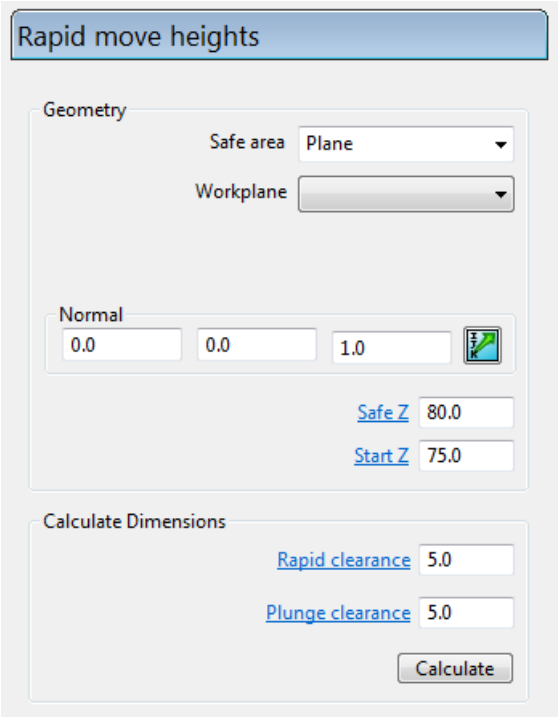
Use the **Rapid move heights** page on a strategy dialog to define the **Safe Z** and **Start Z** tool heights.

*To change rapid move heights after a toolpath is calculated, click the **Rapid Move Heights**  button on the **Main** toolbar to display the **Rapid Move Heights** dialog.*

To specify rapid move heights

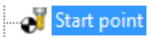
1. Select the **Rapid Move Heights** page  on the **Model Area Clearance** strategy dialog.
2. On the **Rapid Move Heights** page:
   a. Check that **Safe Area** is set to **Plane**.

b Click **Calculate**.



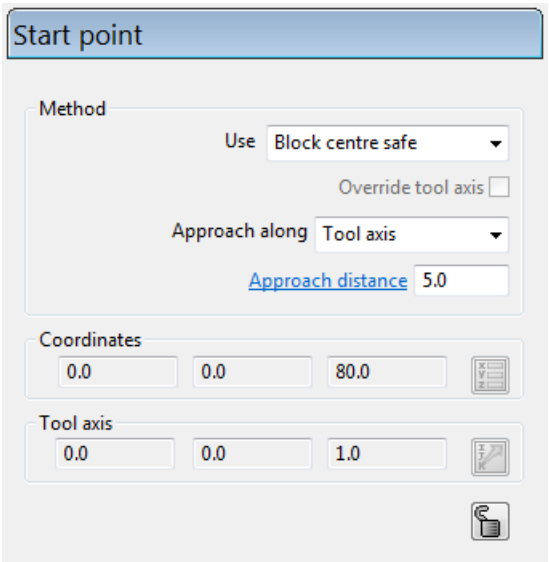
The use of absolute and incremental Z heights is controlled by the type of **Rapid Movement** selected.

Specifying tool start point

1 Select the **Start point** page  on the **Model Area Clearance** strategy dialog.

2 On the **Start Point** page, in the **Use** list, select **Block Centre Safe.**

This resets the tool X and Y values to the centre of the block model, with the Z coordinate at **Safe Z**.



Generating the roughing toolpath

On the **Model Area Clearance** strategy dialog:

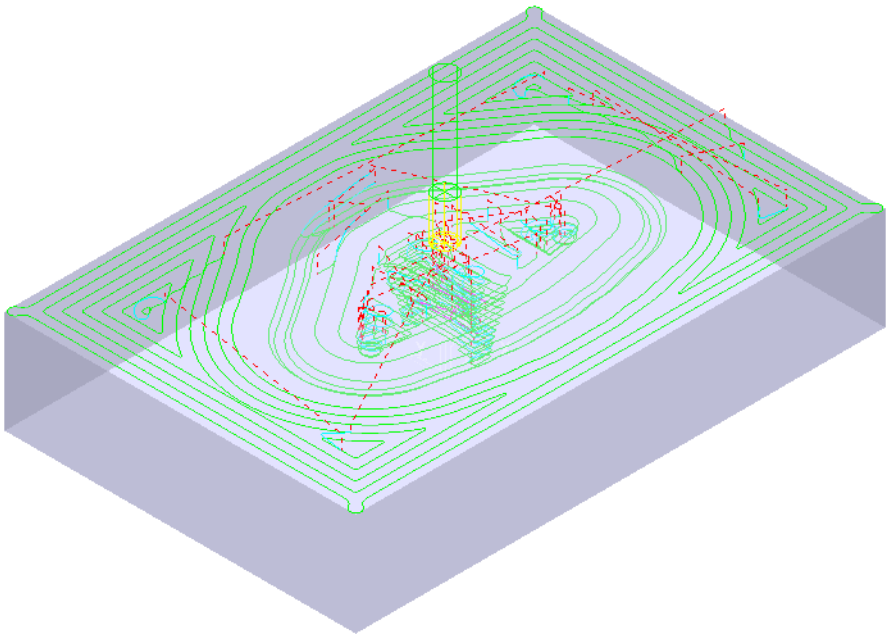
1. Select and expand the **Leads and links** page:
   a. Select the **Lead in** sub page.
   b. From the **1st choice** list, select **Ramp**.
2. Select the **High speed** page:
   a. Select the **Profile smoothing** option. Leave the default **Corner Radius** as **0.05**.
   b. Select **Raceline smoothing**, and adjust it to a small value, such as **5**%.
3. Click **Calculate** to generate the toolpath.

   Progress is shown on the **Status** bar at the bottom of the screen. The generation may take a minute or so, depending on the processing power of your PC.
4. After the toolpath is generated, **Close** the strategy dialog.

Displaying the roughing toolpath

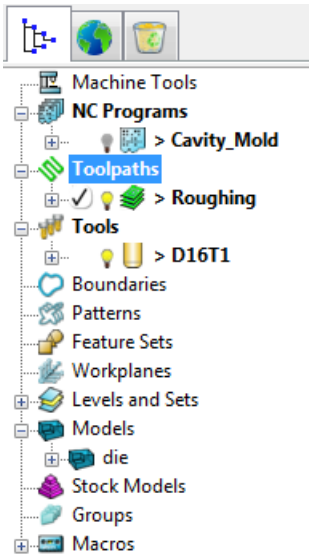
To improve visualisation of the toolpath, you can:

- Click the **Plain Shade** and **Wireframe** buttons to draw and undraw the model or wireframe.



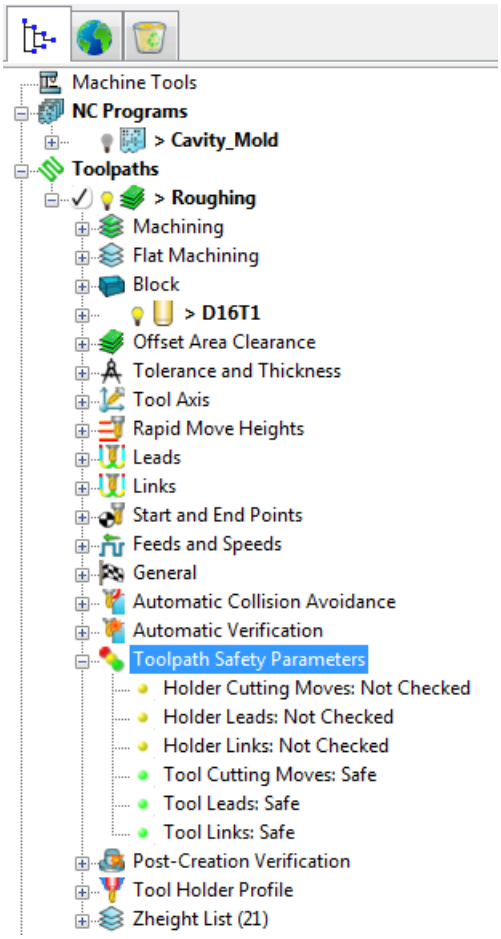
*To zoom in to the model, hold down the **Ctrl** key and the middle (or right) mouse button, and drag the mouse upwards.*

- Click the **Block** button on the **View** toolbar to undraw the block.
- Click ⊞ to expand the **Toolpaths** node in the explorer. The new toolpath is shown in **bold** and preceded by the **>** symbol to indicate it's active.



- Click the light bulb next to the toolpath to toggle toolpath display on or off .

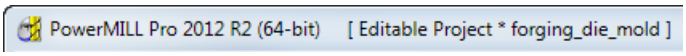
- Click ⊞ next to the toolpath to expand and view the parameters and their specific details used to create the toolpath.



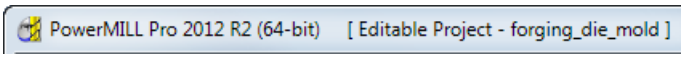
*The **Safety Status** icon ✓ at the top of the toolpath tree shows the toolpath is gouge-checked, but not checked for holder details. To see more information, expand the **Toolpath Safety Parameters** node in the toolpath tree.*

Saving project changes

When you have unsaved changes (in this case, the toolpath information) in your project, an asterisk (*) is displayed in the title bar.



Click **Save** on the **Main** toolbar to save the project changes and overwrite the previous file. The toolpath information is added to your project, and the asterisk disappears from the project header:

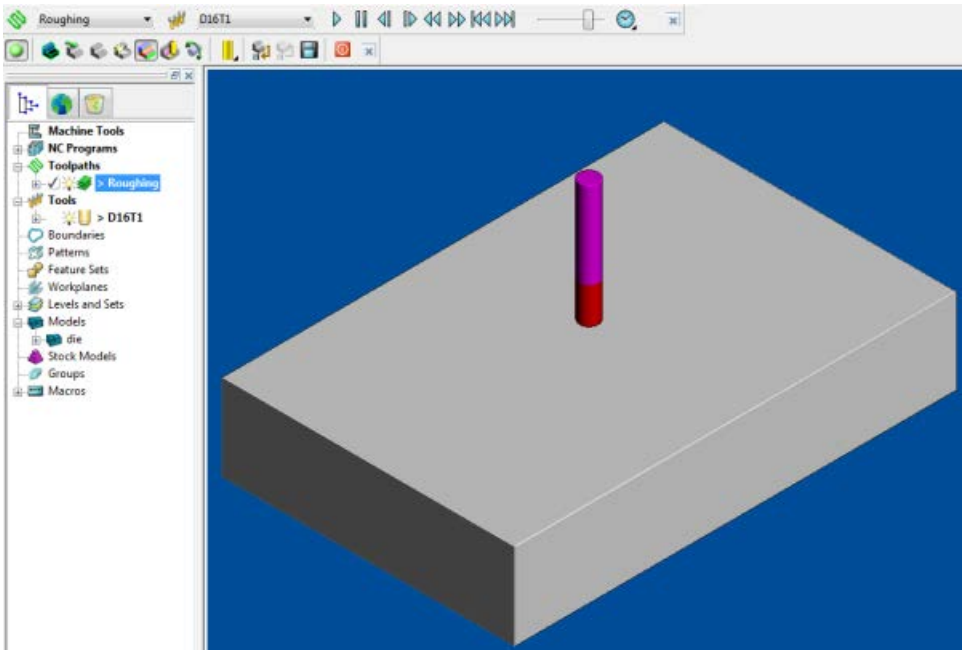


Simulating the roughing toolpath

To view a simulation of the toolpath:

1 Click the **ISO1** button on the **View** toolbar to reset the view.

2 On the **ViewMill** toolbar, click the **Toggle ViewMill Window** button. It turns green, and activates the simulation window, which initially shows a light grey block on the current background.

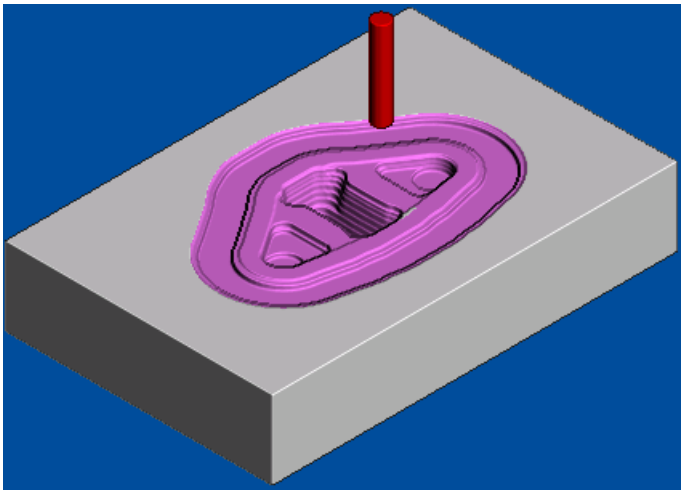


*If the **Simulation** toolbar is not already displayed, select **View > Toolbar > Simulation**.*

3 Select the current toolpath in the first of the two drop-down lists. The tool is selected automatically, and the **Play** buttons are highlighted:

4 The buttons on the **ViewMill** toolbar control the display of the simulation. Select the **Rainbow Shaded Image** option to visualise the material removed by different toolpaths, for example roughing and rest roughing.

5 To start the simulation, click the **Play** button. Allow the simulation to run through to the end.

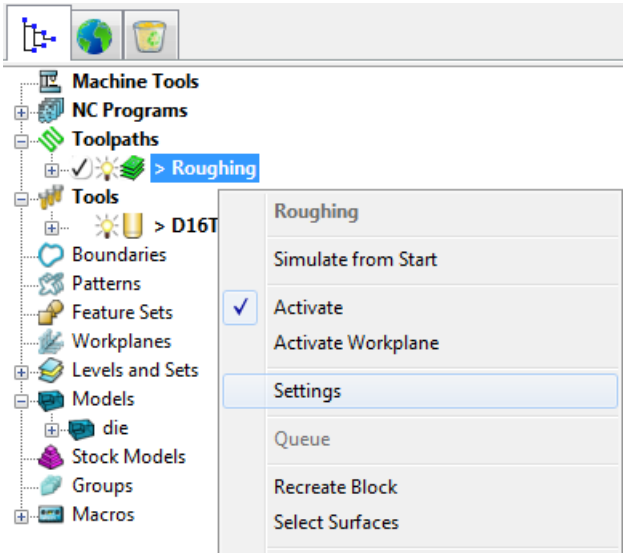


Creating the rest roughing toolpath

The rest roughing toolpath uses a smaller tool to eliminate the large terraces and to rough areas of the model that the large roughing tool couldn't reach, such as pockets and corners.

To create a rest roughing toolpath:

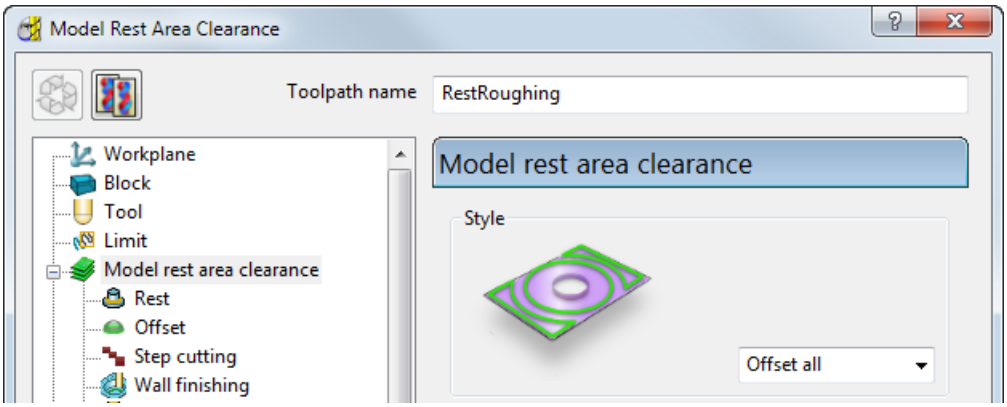
1 Display the **Settings** for the previous toolpath.



2 Click the **Create a new toolpath based on this one**  button.

3 A copy of the toolpath is created with a suffix of _**1**. Enter **RestRoughing** in the **Toolpath name** field to rename it.

4 Select the **Rest Machining** option. This switches the strategy to **Model rest area clearance strategy** and enables the **Rest** page.



5 Define the rest roughing tool geometry (see page 44).

6 Change the **Stepover** and **Stepdown** values (see page 46).

7 Complete the **Model rest area clearance** strategy dialog, and generate the rest roughing toolpath (see page 47).

8 Display the rest roughing toolpath (see page 47).

9 Simulate the rest roughing toolpath (see page 48).

Defining the rest roughing tool geometry

You can base the rest roughing tool on the existing roughing tool, although it needs a smaller diameter and larger tip radius.

This example uses a **10** mm (**3/8** inch) tip radiused tool.

1 On the **Tool** page of the **Model Rest Area Clearance** dialog of the **RestRoughing** toolpath click the **Edit** button.

2 In the **Tip Radiused Tool** dialog, click to create a new tool entity based on the existing roughing tool. It is given the default name of **D16T1_1**.

3 Rename the tool to **D10T2**.

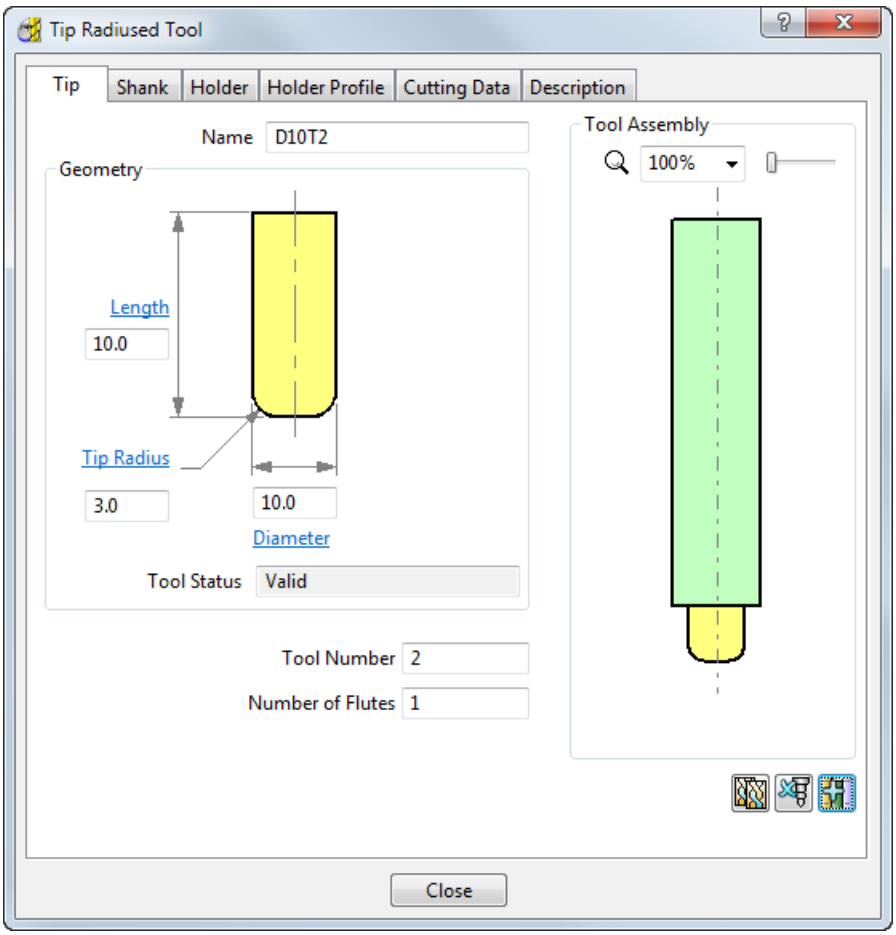
4 In the other fields, enter:

a **Length -** 10 mm

b **Tip Radius -** 3 mm

c **Diameter -** 10 mm

d **Tool Number - 2**

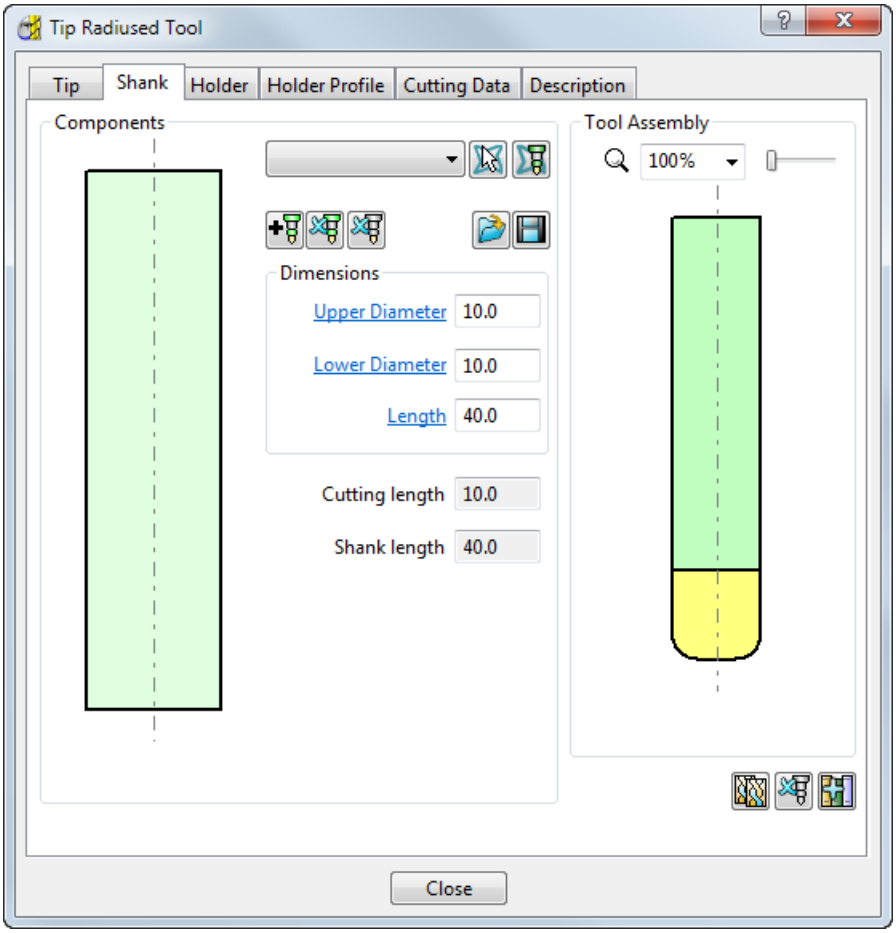


5 Select the **Shank** tab and change the values to:

a **Upper Diameter - 10**

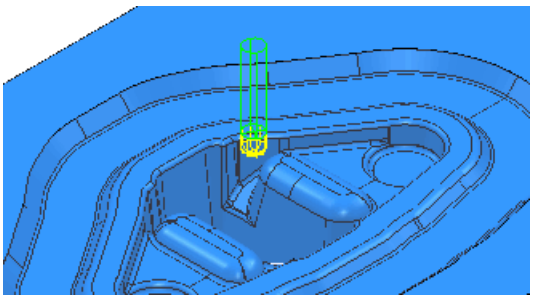
b **Length - 40**

The **Lower Diameter** automatically defaults to the **Upper Diameter**. This can be accepted for the current tool.



6 Click **Close**.

To see the rest roughing tool more clearly, click the light bulb icons for the roughing toolpath and tool, (toggle them to). This undraws (but doesn't delete) the entities on the screen.



Changing the Stepover and Stepdown values

On the **Model rest area clearance** page:

1 Enter a **Stepover** of 3.0 mm.

2 Enter a **Stepdown** of 1.5 mm.

Generating the rest roughing toolpath

On the **Model Rest Area Clearance** dialog:

1 Select the **Rest** page, and:

   a From **Rest Machining** type list, select **Toolpath**.

   b From the **Toolpath** list, select **Roughing** (the name of your roughing toolpath).

   c In the **Detect Material Thicker Than** field, enter **0.2** mm. The calculation ignores rest material thinner than 0.2 mm. This helps to avoid thin regions being rest roughed where the benefit of a second cut is negligible.

   d In the **Expand Area By** field, enter **0.2** mm. The rest areas are expanded by 0.2 mm measured along the surface. This can be used in conjunction with **Detect Material Thicker Than** to reduce the areas to be machined to the essentials (such as, corners), and then to offset these areas slightly to ensure that all details (for example, on the corners) are machined.

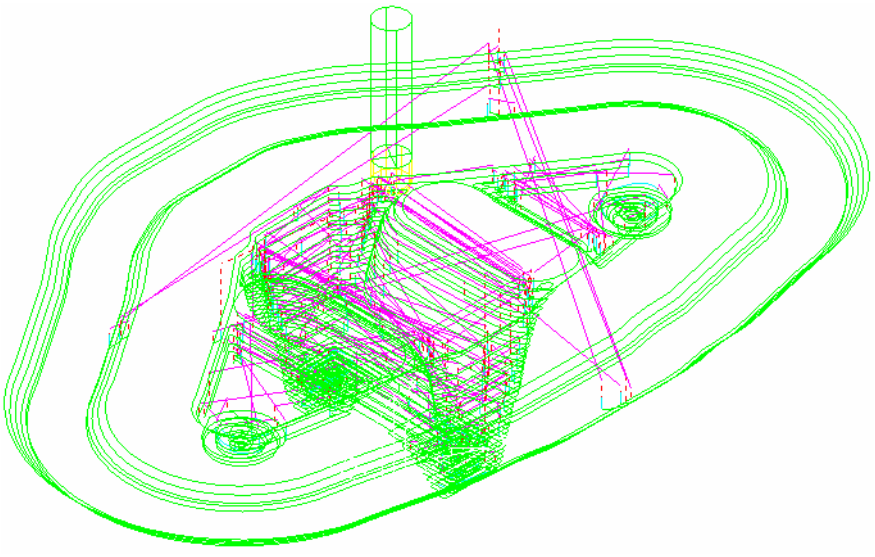
2 Click **Calculate** to generate the toolpath.

   Progress is shown on the **Status** bar at the bottom of the screen. The generation may take a minute or so, depending on the processing power of your PC.

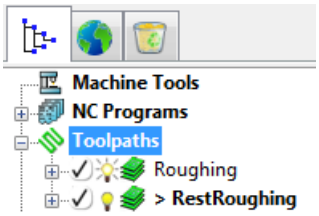
3 After the toolpath is generated, **Close** the strategy dialog.

Displaying the rest roughing toolpath

Use the **Plain Shade**  , **Wireframe**  , and **Block**  buttons to undraw the model and the block, and then zoom in to view the toolpath:



Click ⊞ to expand the **Toolpaths** node in the explorer. The new toolpath is shown in **bold** and preceded by the **>** symbol to indicate it's active.



*The **Safety Status** icon ✓ at the top of the toolpath tree shows the toolpath is gouge-checked, but not checked for holder details. To see more information, expand the **Toolpath Safety Parameters** node in the toolpath tree.*

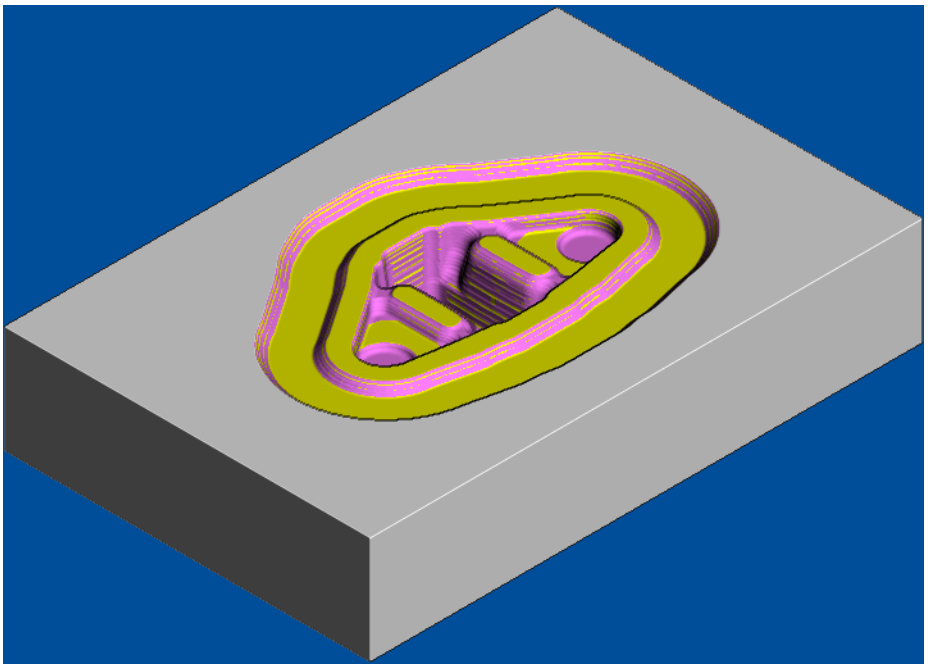
To save project changes, click on the **Main** toolbar.

Simulating the rest roughing toolpath

To view a simulation of the rest roughing toolpath:

1. On the **Simulation** toolbar, select the current toolpath in the first of the two drop-down lists. The tool is selected automatically and the **Play** buttons are highlighted:

2. To start the simulation, click the **Play** button. Allow the simulation to run through to the end.



If you have not started a different session since creating the roughing toolpath, the rest roughing toolpath is shown in a different colour, overlaid on the roughing simulation (see page 41).

3 Click the **Exit ViewMill** button, and select **Yes** to stop the simulation. The **Toggle ViewMill** button changes from green to red, and the standard PowerMILL window is displayed.

Closing the roughing session

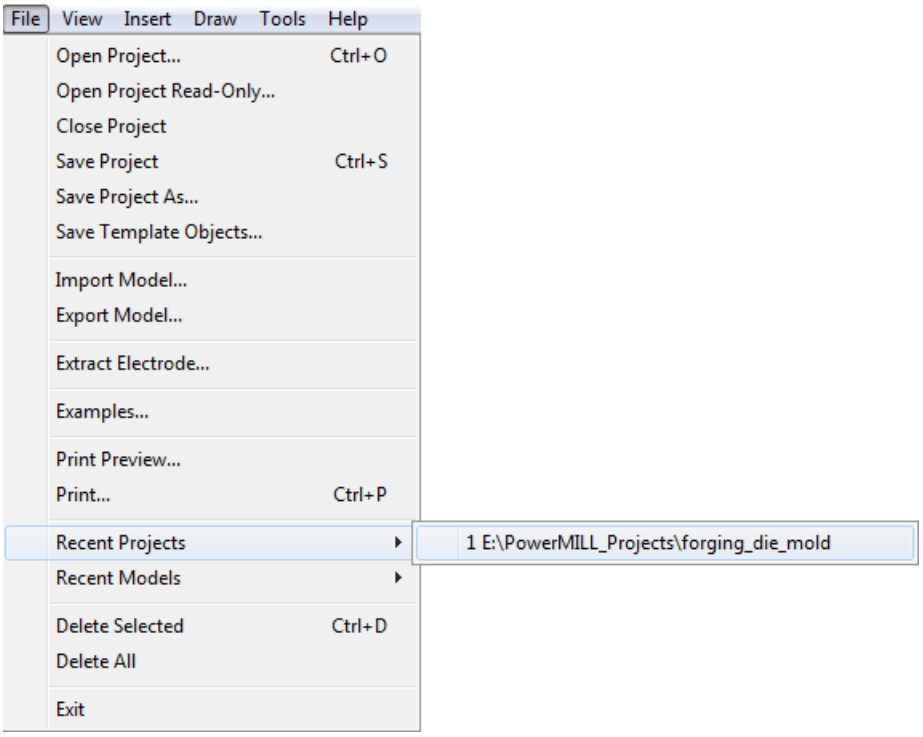
Now you have generated the roughing toolpath. Save the project and close PowerMILL until you are ready to create the finishing strategies.

To save project changes, click on the **Main** toolbar.

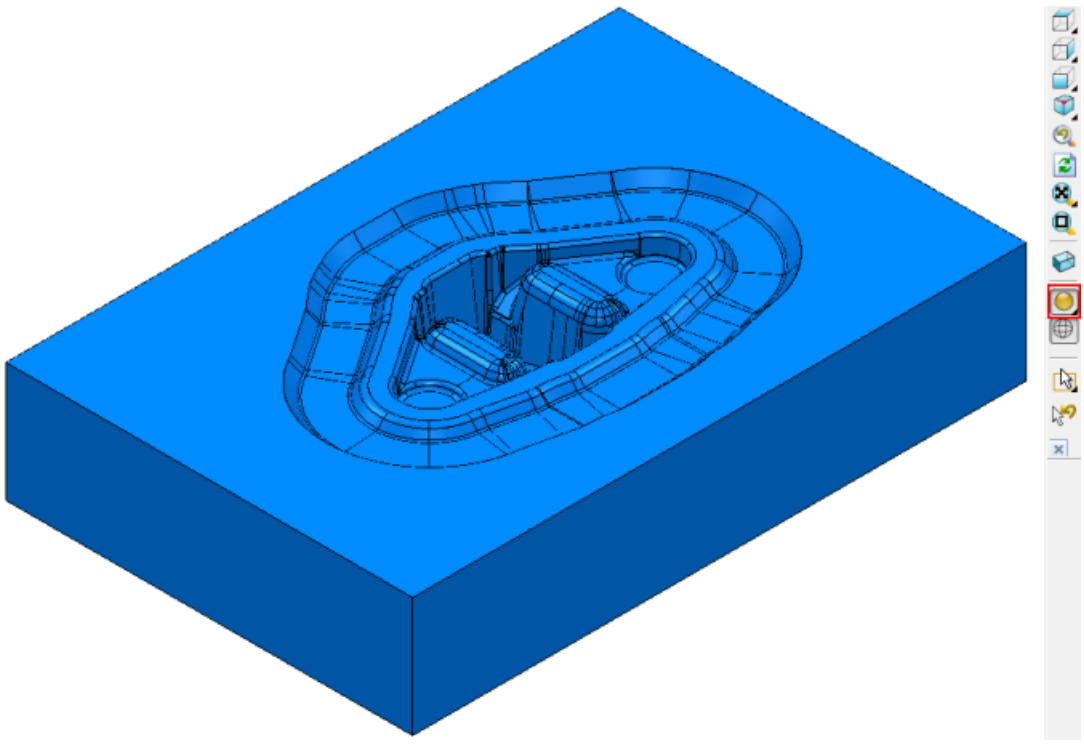
To exit PowerMILL, either select **File > Exit** from the menu, or click the **Close** button in the top right-hand corner of the PowerMILL window.

Re-opening the project

1. Restart PowerMILL; this automatically loads your toolbar and colour selections from the previous session.
2. Select **File > Recent Projects** from the menu, and then select the project containing the forging die example:



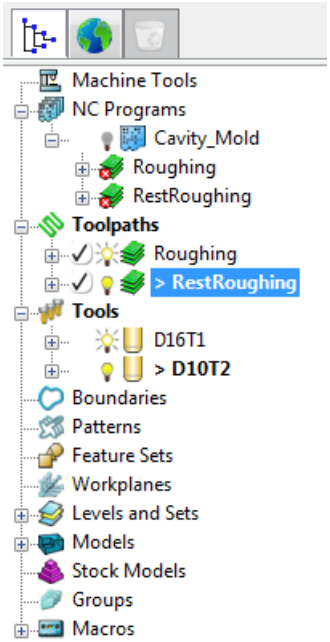
3. When the project is reloaded, adjust the model size and orientation as required.



Checking which entities are active

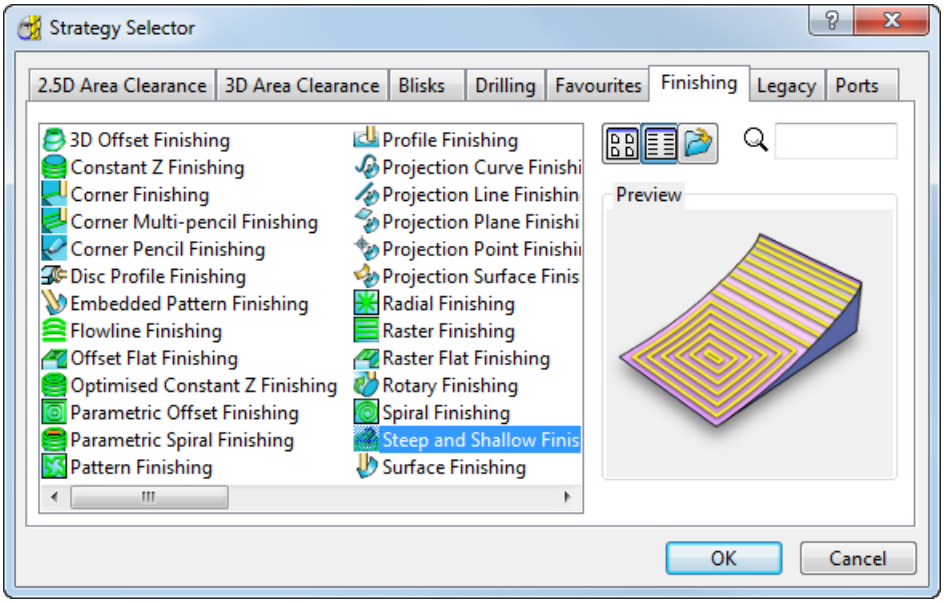
Before you create the finishing toolpaths, you can ensure they are automatically added to an NC program:

1. Check if the NC program in the explorer is shown in **bold** and preceded by the **>** symbol to indicate it's active. If it's not active, right-click the NC program name, and select the **Activate** option.
2. Expand the **Toolpaths** node, and undraw each of the roughing toolpaths by clicking the light bulb icon until it shows . Now you can easily see the new finishing toolpaths as you create them.



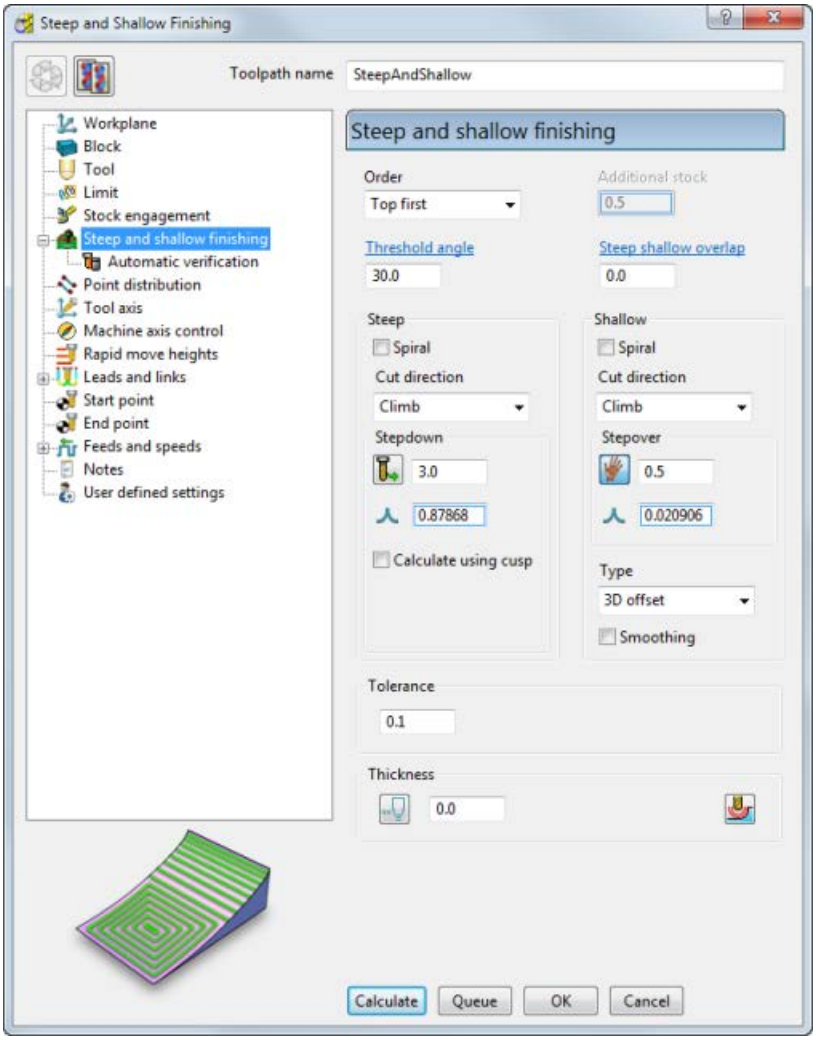
Creating a finishing toolpath

1. Click  on the **Main** toolbar to display the **Strategy Selector** dialog.
2. On the **Finishing** tab, select the **Steep and Shallow Finishing** strategy and click **OK**.



3. In the **Steep and Shallow Finishing** dialog:
   a. In the **Toolpath Name** field, enter **SteepAndShallow**.
   b. Select **Climb** from the **Cut direction** field.

c Change the **Stepover** value to **0.5**.



Defining the finishing tool geometry

The tool currently selected for the **Steep and Shallow Finishing** strategy dialog is the same tool that was used for the rest roughing toolpath. This tool is not appropriate for the current toolpath strategy, and must be changed. This example uses an **8** mm (**5/16** inch) ball nosed tool.

1 In the **Steep and Shallow Finishing** strategy dialog, click Tool to select the **Tool** page.

2 On the **Tool** page, from the tool list, select  to create a **Ball Nosed Tool**.

3 On the **Tool** page, click  to display the **Ball Nosed Tool** dialog.

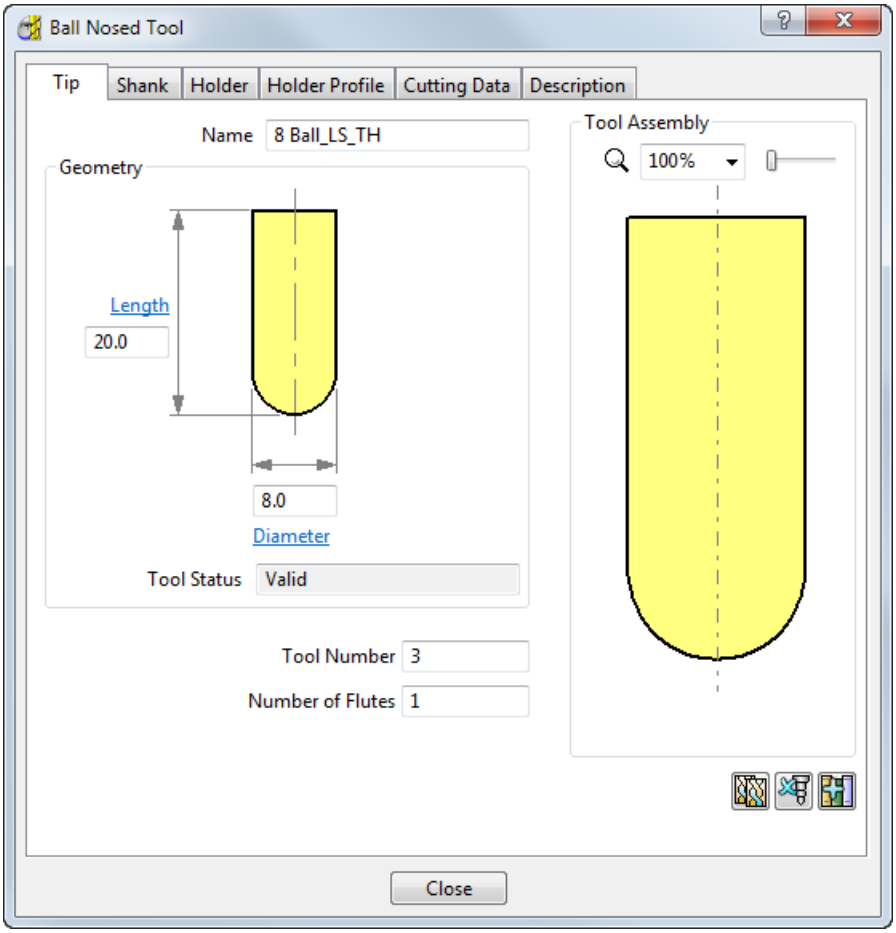
4 In the **Ball Nosed Tool** dialog, enter:

a **Name - 8 Ball_LS_TH**

b **Length - 20** mm

c **Diameter - 8** mm

d **Tool Number - 3**

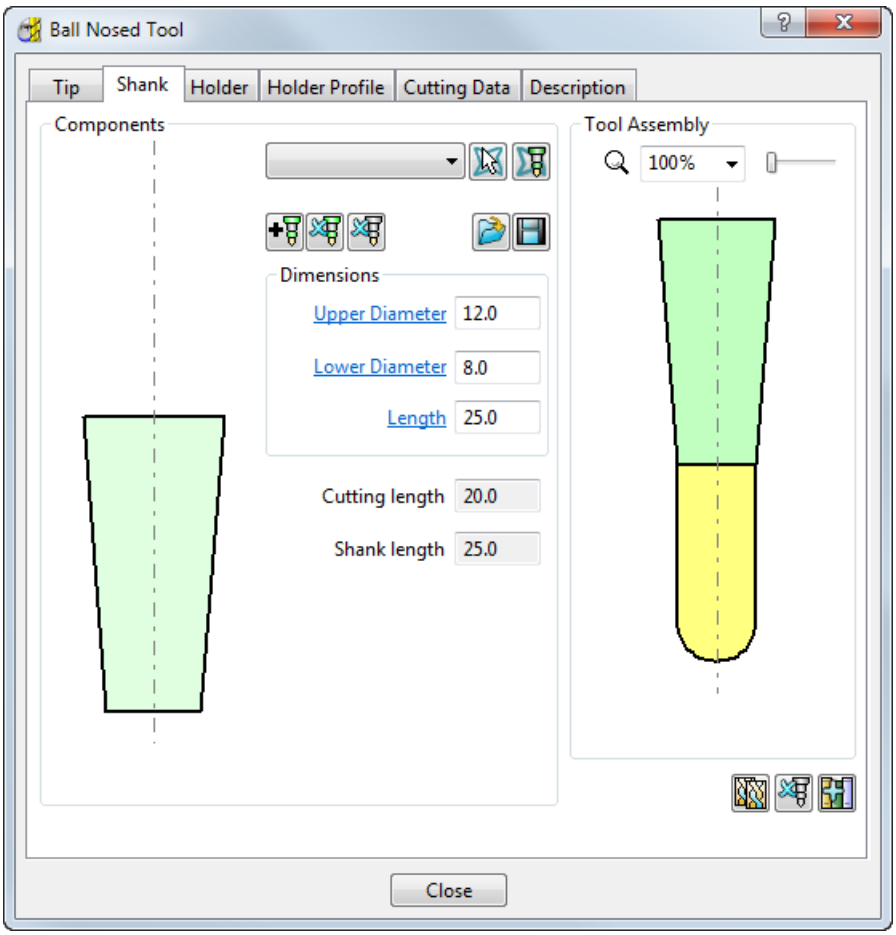


5 Select the **Shank** tab, click  to add a shank component. Enter:

a **Upper Diameter - 12**

b **Lower Diameter - 8**

c **Length - 25**

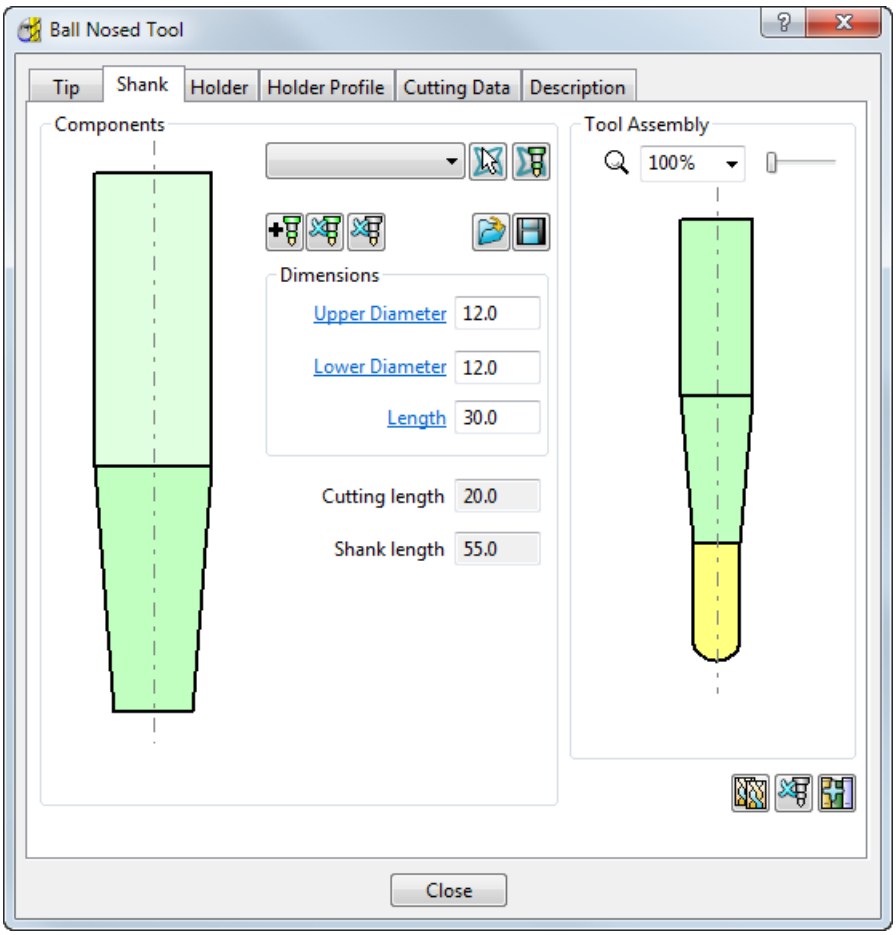


6 Click the button to add a second shank component. Enter:

a **Upper Diameter - 12**

b **Lower Diameter - 12**

c **Length - 30**



7 Select the **Holder** tab, and click the button to add a holder component. Enter:

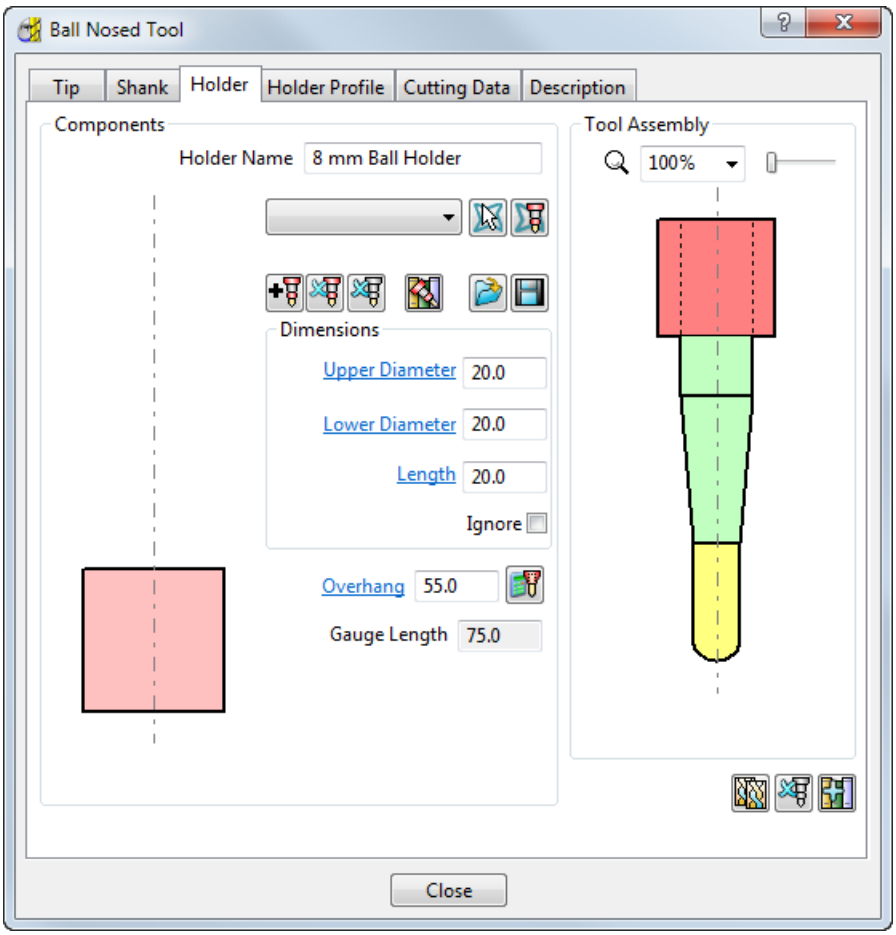
a **Name - 8 mm Ball Holder**

b **Upper Diameter - 20**

c **Lower Diameter - 20**

d **Length - 20**

e **Overhang - 55**



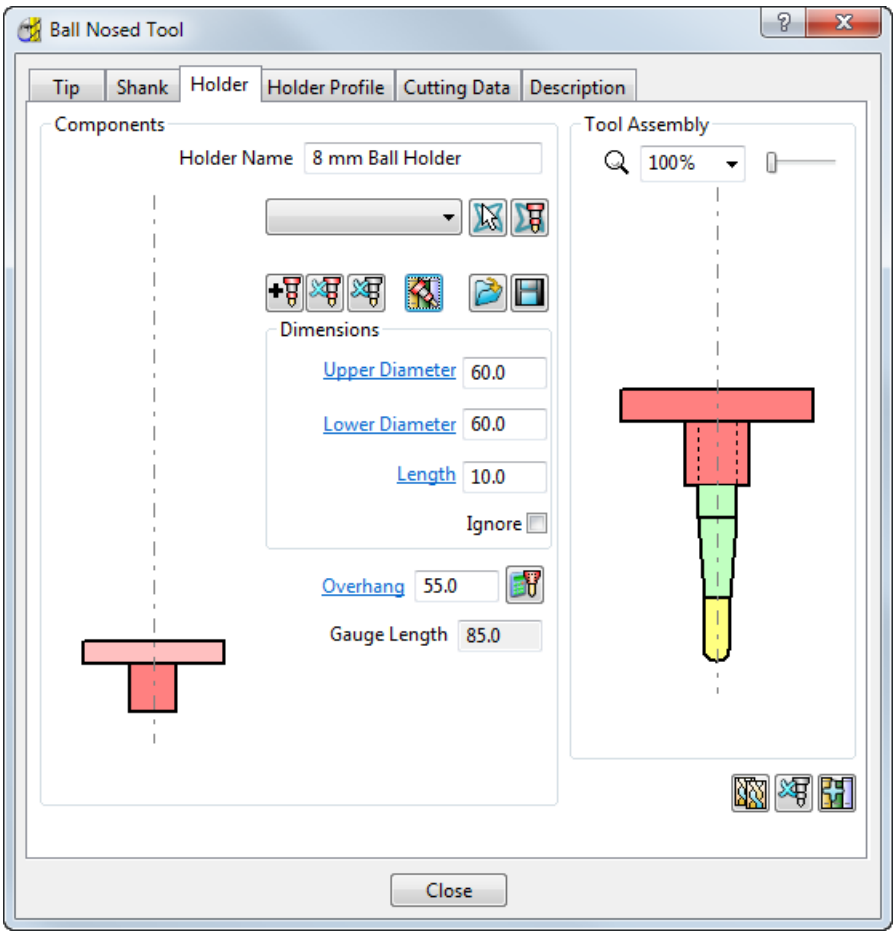
8 Click the button to add the upper part of the holder. Enter:

a **Upper Diameter - 60**

b **Lower Diameter - 60**

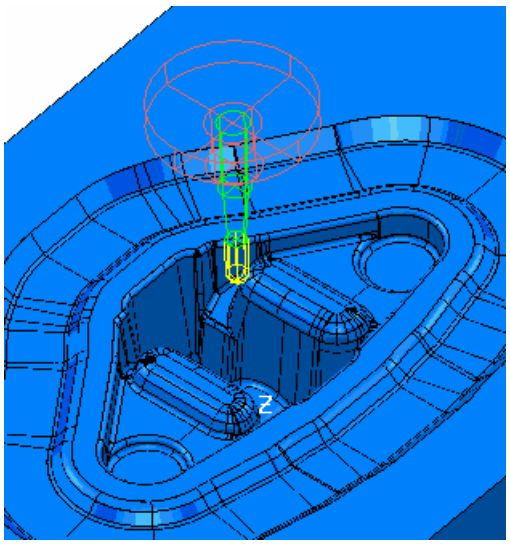
c **Length - 10**

d Overhang - 55



9 Click **Close** to update the **Tool** page with the new tool.

*The new tool is shown in the explorer, and on the **Tool** toolbar, and it's also drawn in the graphics window, where it is automatically aligned with the Z axis.*



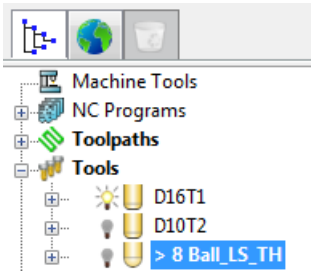
The toolpath is not yet complete as it needs to reference a boundary, but you can close the dialog without generating the toolpath.

10 Click **OK** to close the toolpath dialog.

Creating the boundary for the selected surface

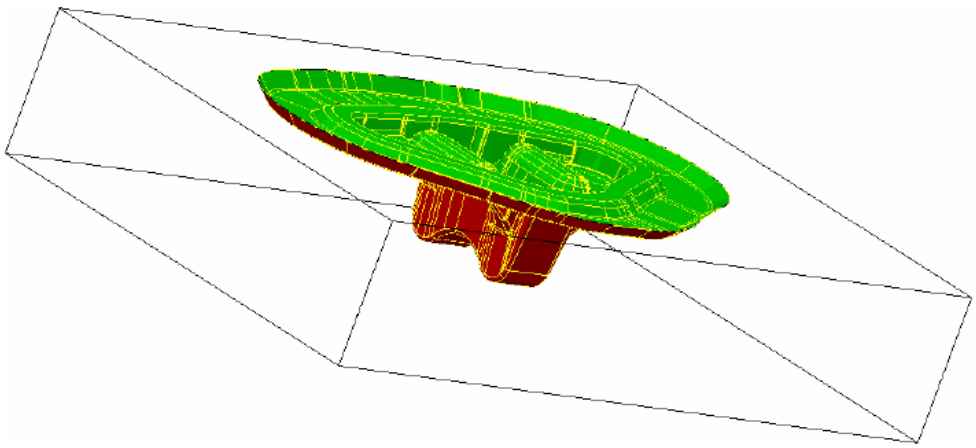
As it is only the cavity that needs to be finished by this toolpath, you can create a boundary for the cavity.

1 Undraw the **8 Ball_LS_TH** tool by toggling the light bulb icon to in the explorer.



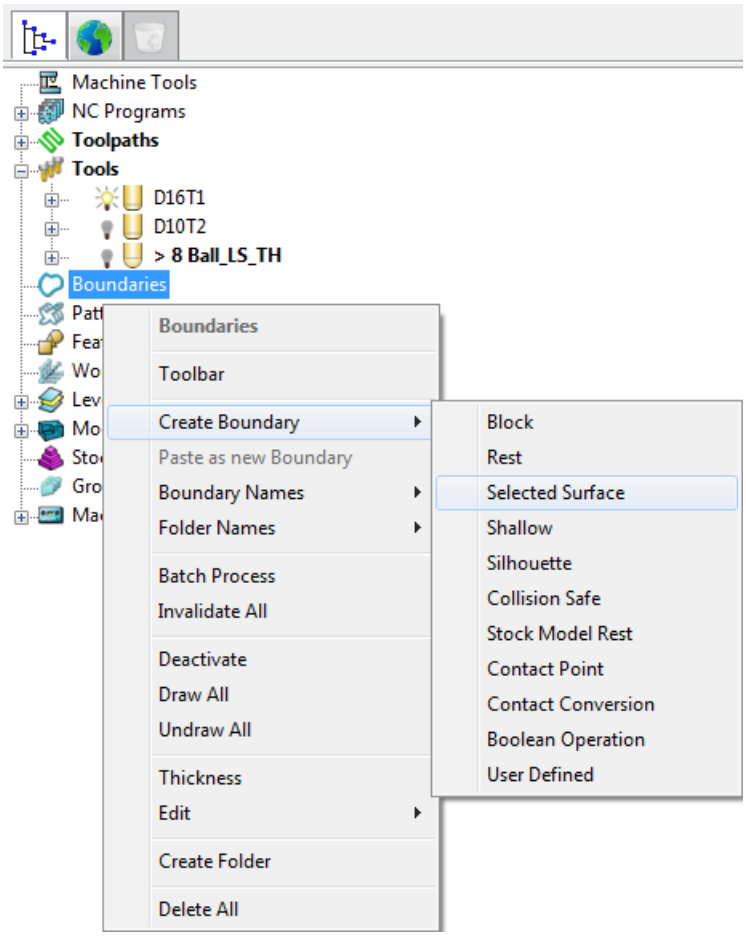
2 Click the **Block** button to undraw the block, and use the **Wireframe** button to undraw the wireframe.

3 Use the mouse to select only the cavity surfaces.



*You can use the **Drag Selection Mode** on the **View** toolbar to select multiple surfaces.*

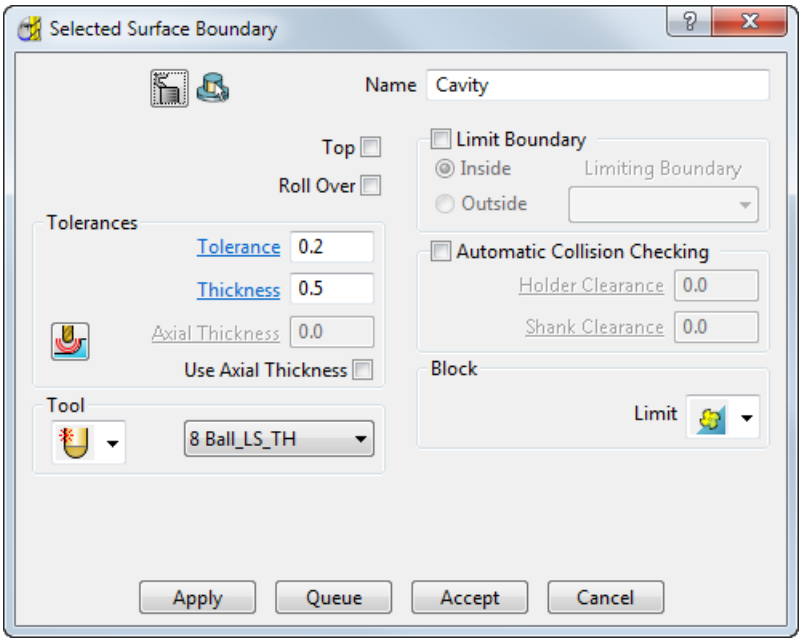
4 In the **Boundaries** context menu, select **Create Boundary > Selected Surface**.



5 In the **Selected Surface Boundary** dialog:

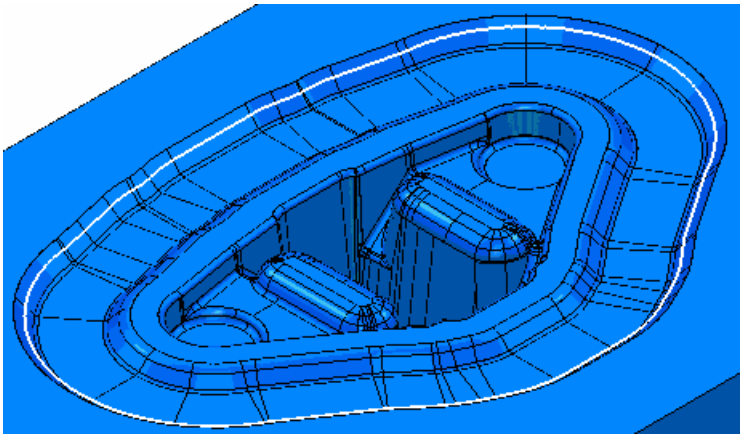
a In the **Name** field, enter **Cavity**.

b In the **Tool** list, select **8 Ball_LS_TH**.



6 Click **Apply**.

7 The boundary is calculated. It is shown in white by default:



Selected Surface Boundaries are recalculated if the selected surfaces change. This means that they behave like toolpaths in the way they take account of the selection state at the time of calculation.

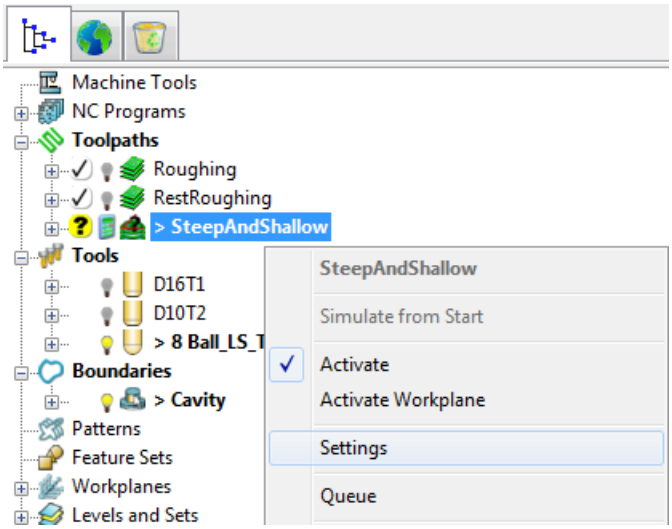
8 Click **Accept** to close the boundary dialog.

Completing and generating the finishing toolpath

You now need to add the created boundary to the **Steep and Shallow Finishing** toolpath together with the appropriate leads and links.

1 In explorer, expand **Toolpaths** and select the **SteepAndShallow** toolpath.

2 Right-click and select **Settings** from the context menu.

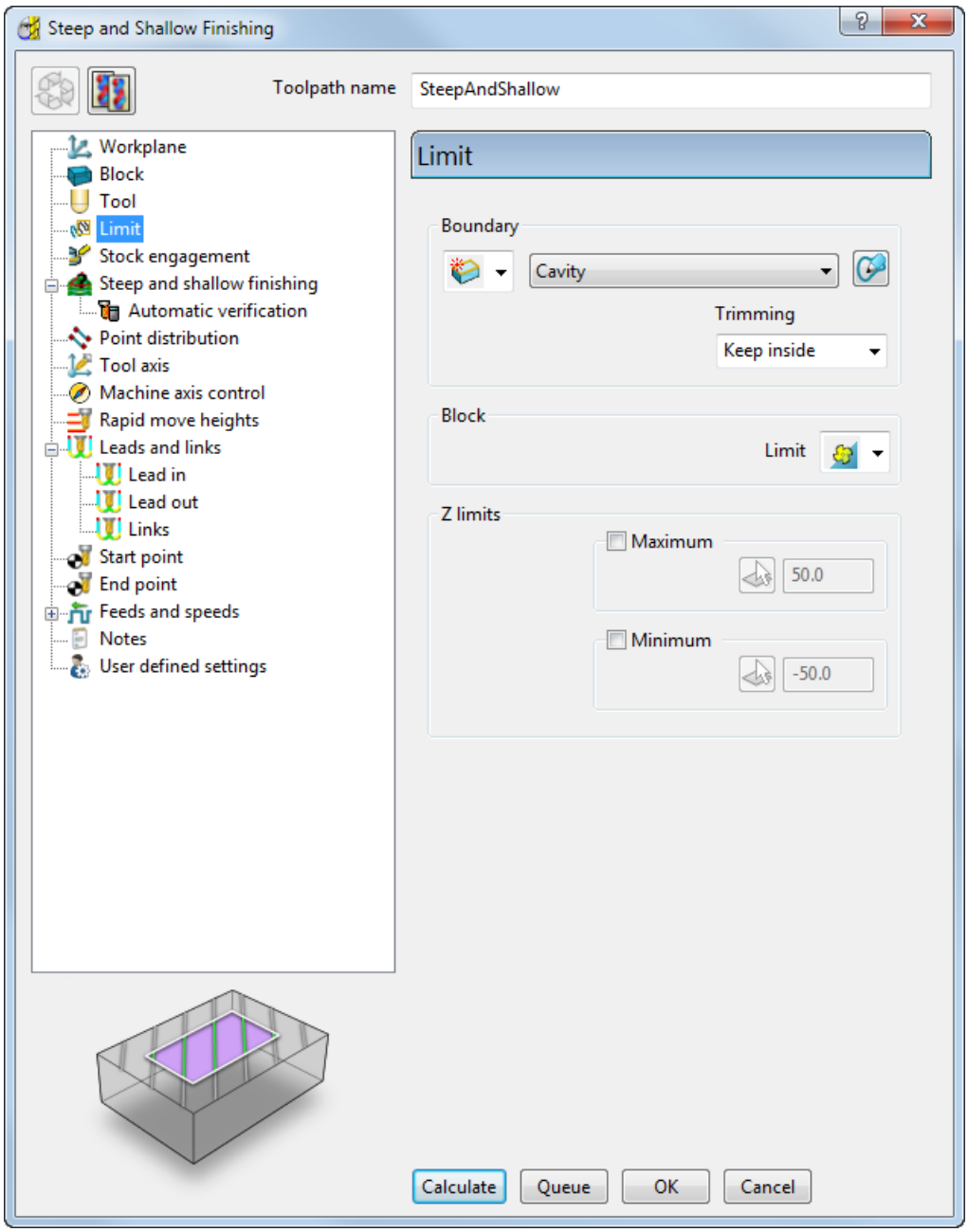


3 On the **Steep and Shallow Finishing** strategy dialog:

a Select the **Limit** page and:

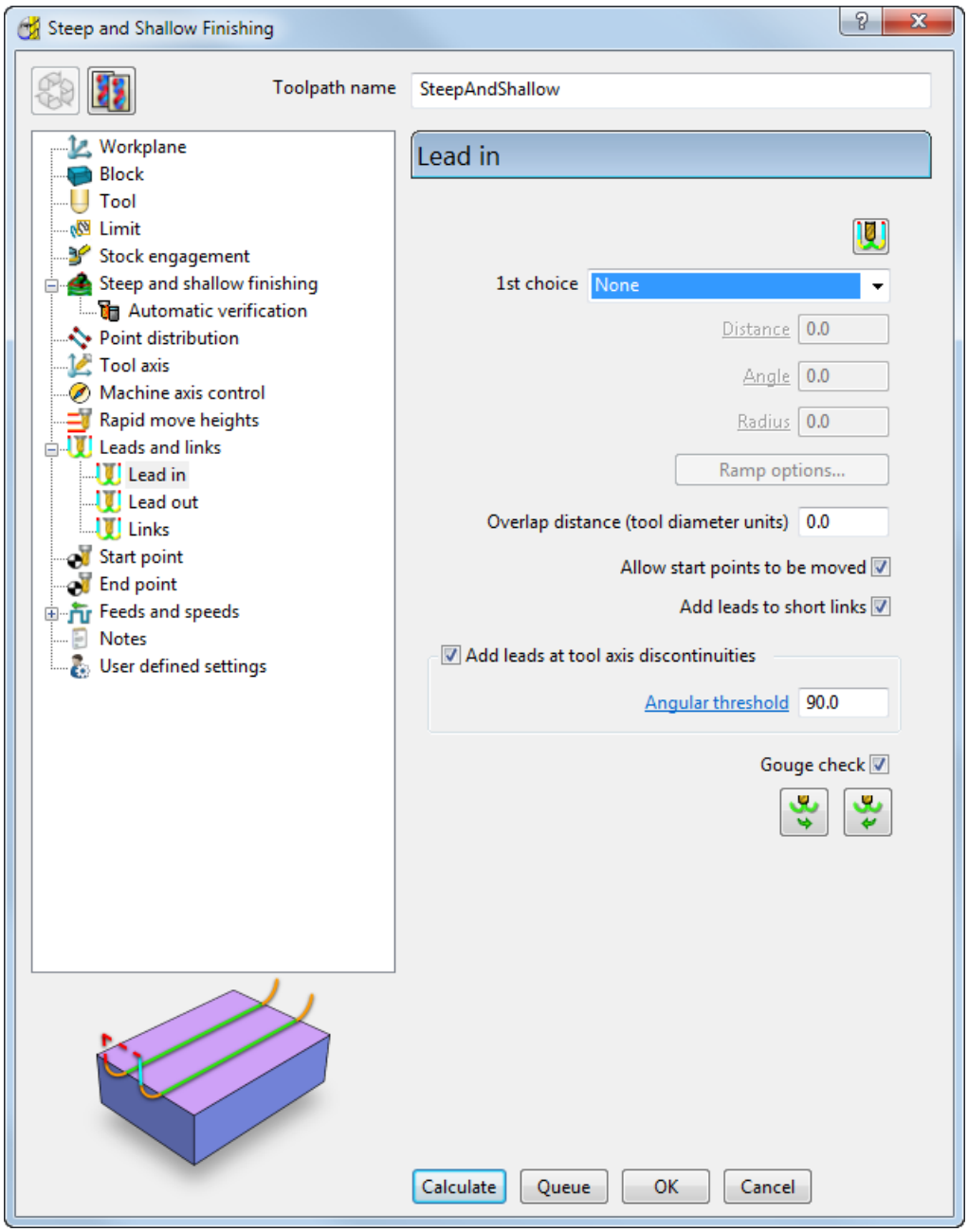
From the **Boundary** list, select **Cavity**.

From the **Trimming** list, select **Keep inside**.



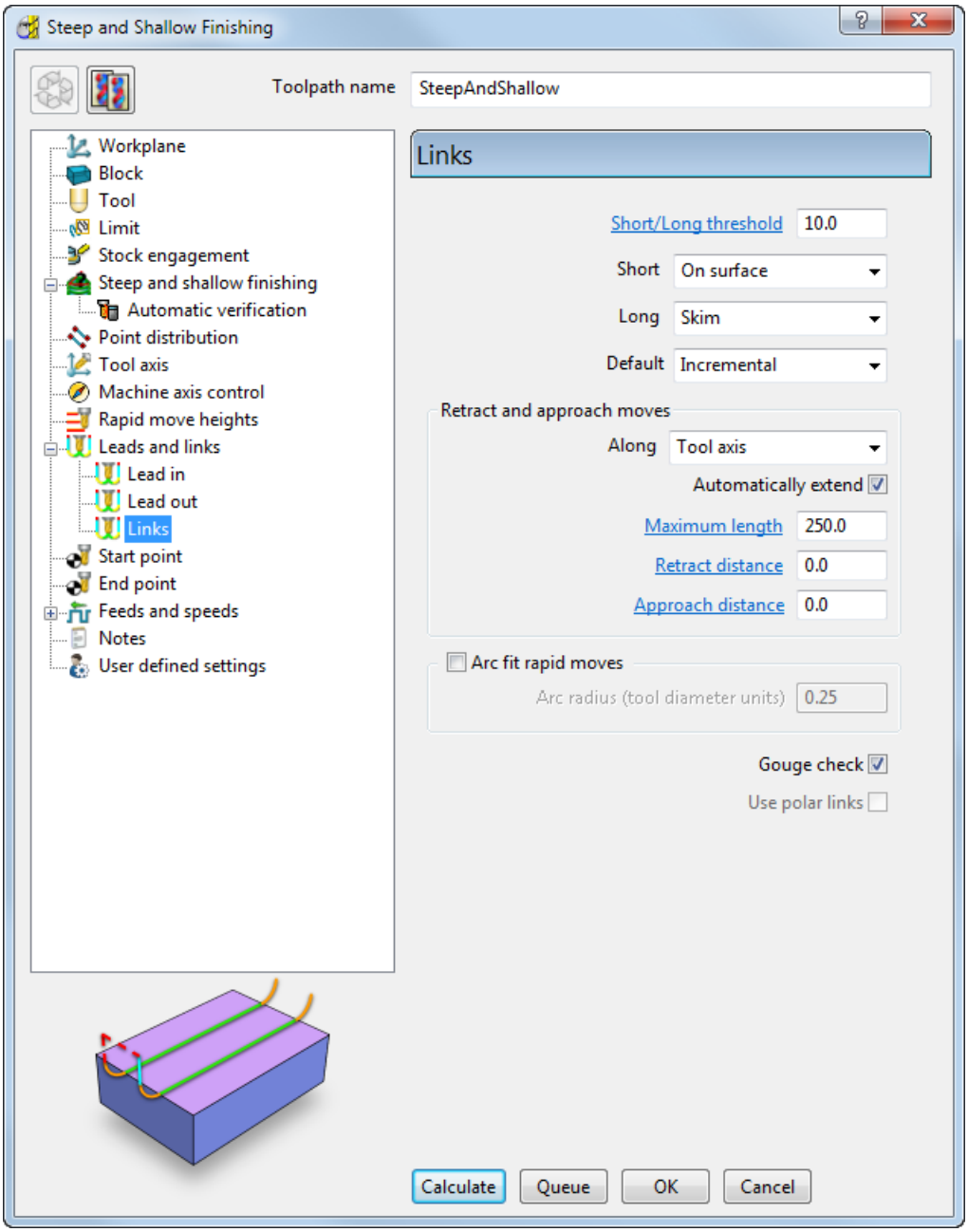
b Expand the **Leads and Links** page, select the **Lead in** page and:

In the **1st Choice** list, select **None**.



c Select the **Links** sub page to define link moves between the cutting moves in the toolpath and:

In the **Short** list, select **On surface** to define the type of link moves used to connect adjacent passes.



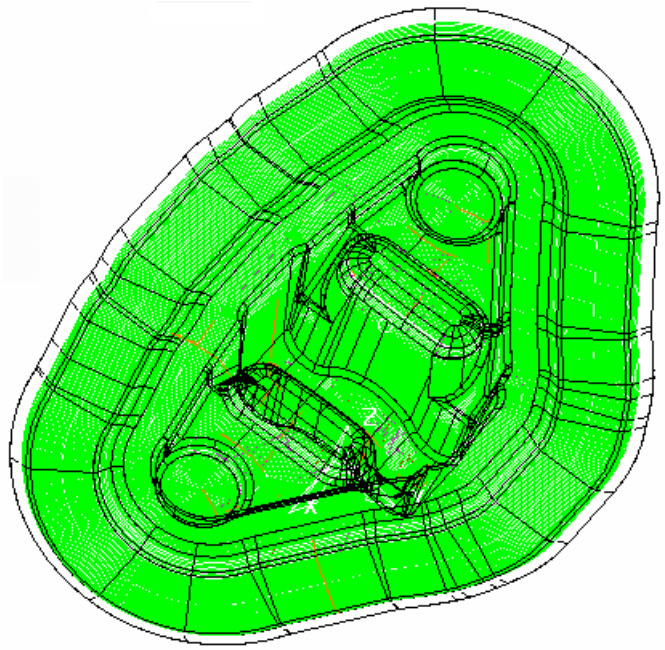
4 Click **Calculate** to generate the toolpath.

Progress is shown on the **Status** bar at the bottom of the screen. The generation may take a minute or so, depending on the processing power of your PC.

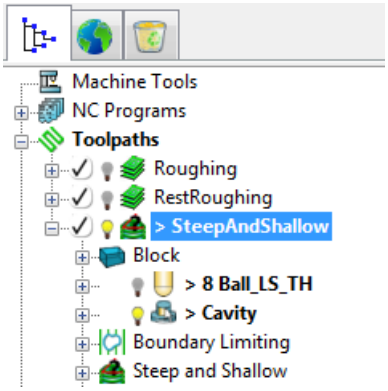
5 After the toolpath is generated, **Close** the strategy dialog.

Displaying the finishing toolpath

When the toolpath is generated, it is displayed on the screen:



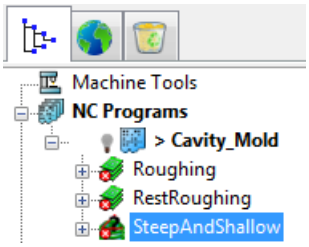
Click ⊞ to expand the **Toolpaths** node in the explorer. The new toolpath is shown in **bold** and preceded by the **>** symbol to indicate it's active.



To save project changes, click  on the **Main** toolbar.

Simulating the NC program with generated toolpaths

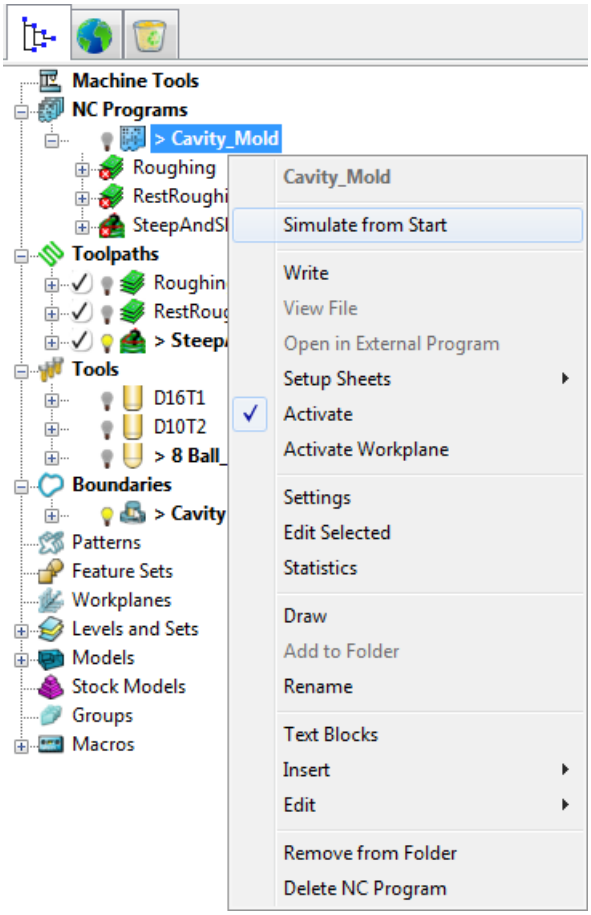
The new toolpath is automatically added to the active NC program. If for some reason, your toolpath is not there, use the mouse to drag the toolpath under the NC program manually.



To simulate the NC program:

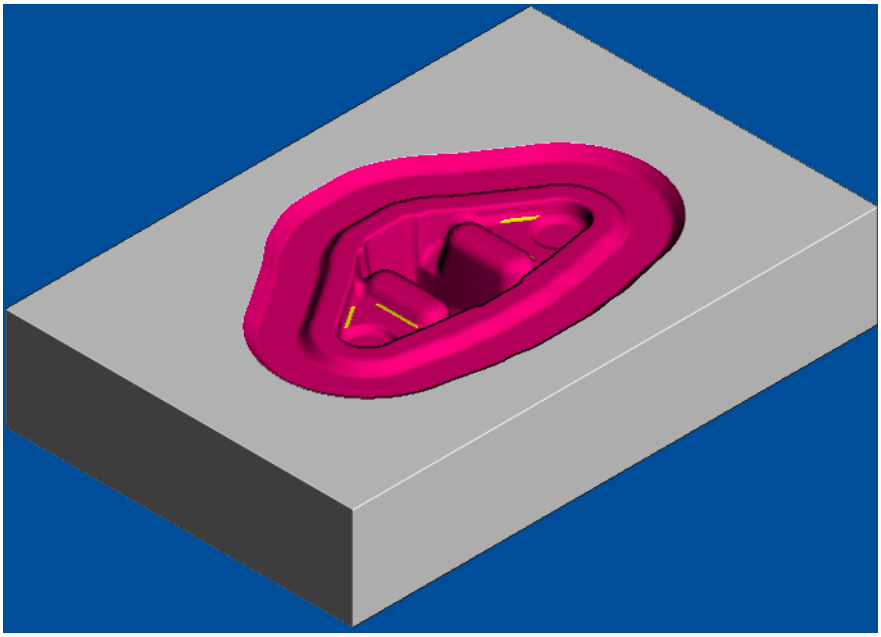
1. Click the **ISO1** button on the **View** toolbar to reset the view.
2. On the **ViewMill** toolbar, click the **Toggle ViewMill Window** button. It turns green, and activates the simulation window, which initially shows a light grey block on the current background.
3. From the **ViewMill Toolbar**, select the **Rainbow Shaded Image** option so you can visualise the differences between toolpaths.

4 In explorer, right-click the NC program **Cavity_Mold**, and select **Simulate from Start** from the context menu.

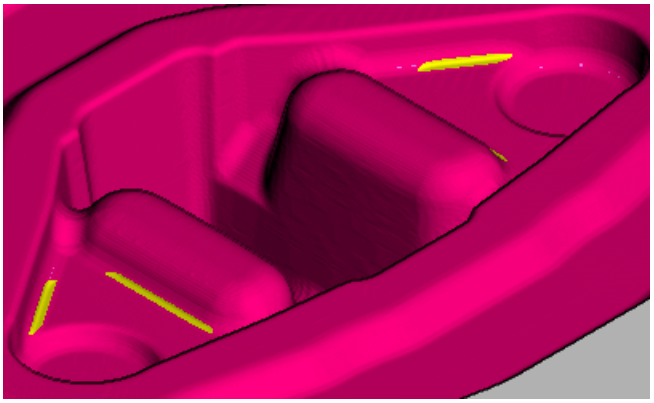


The NC program is automatically selected on the **Simulation Toolbar**, and the **Play** buttons enabled.

5 Click the **Play** button, and allow the simulation to run through to the end.



6 Zoom into the unmachined areas. You can see that some of the corners need cleaning up, particularly between non-tangential surfaces.



7 Click the **Exit ViewMill** button, and select **Yes** to stop the simulation. The **Toggle ViewMill** button changes from green to red, and the standard PowerMILL window is displayed.

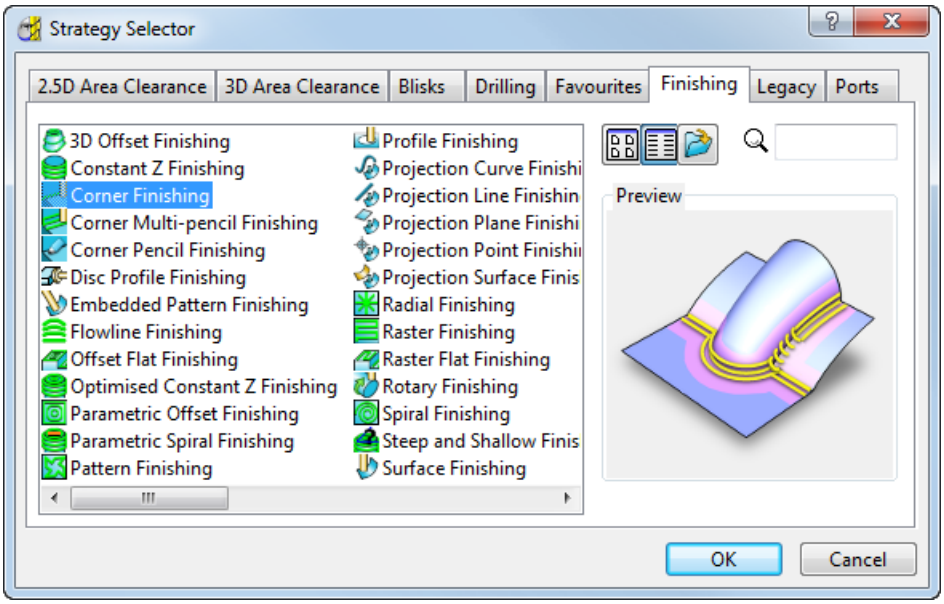
Creating the Corner Finishing toolpath

The **Corner Finishing** toolpath uses a smaller tool to machine the remaining corners, particularly between non-tangential surfaces.

To create the Corner Finishing toolpath:

1 Click  on the **Main** toolbar to bring up the **Strategy Selector** dialog.

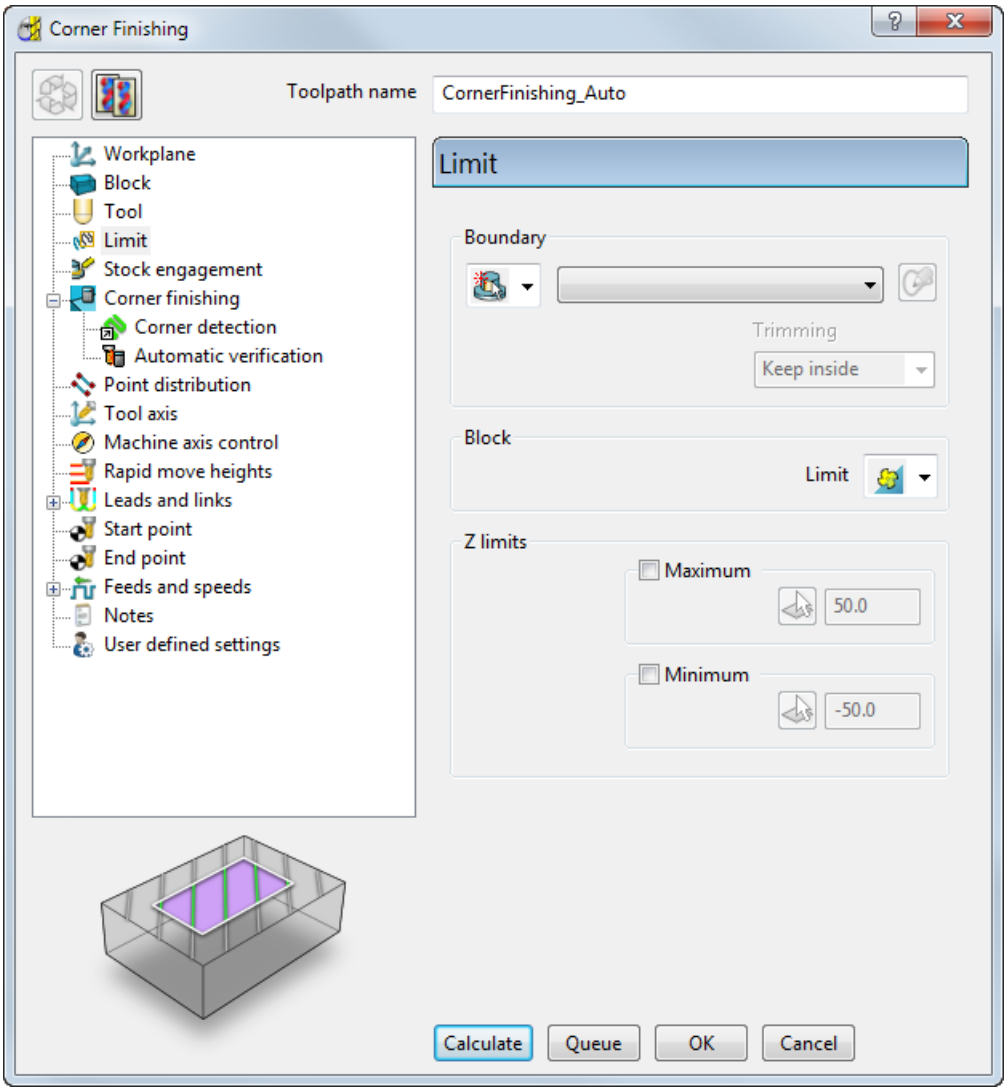
2 Select the **Corner Finishing** strategy on the **Finishing** tab, and click **OK**.



3 In the **Corner Finishing** dialog:

a In the **Toolpath name** field, enter **CornerFinishing_Auto**.

b Select the **Limit** page and choose **None** from the boundary list.



Defining the Corner Finishing tool geometry

You can base the **Corner Finishing** tool on the tool used for the **SteepAndShallow** strategy, although it needs a smaller diameter. This example uses a **6** mm (**1/8** inch) ball nosed tool.

1 Select the **Tool** page in the **Corner Finishing** dialog.

2 From the tool list, select **8 Ball_LS_TH**, and click the **Edit** button.

3 In the **Ball Nosed Tool** dialog which appears, on the **Tip** tab, click to create a new tool entity based on the existing tool.

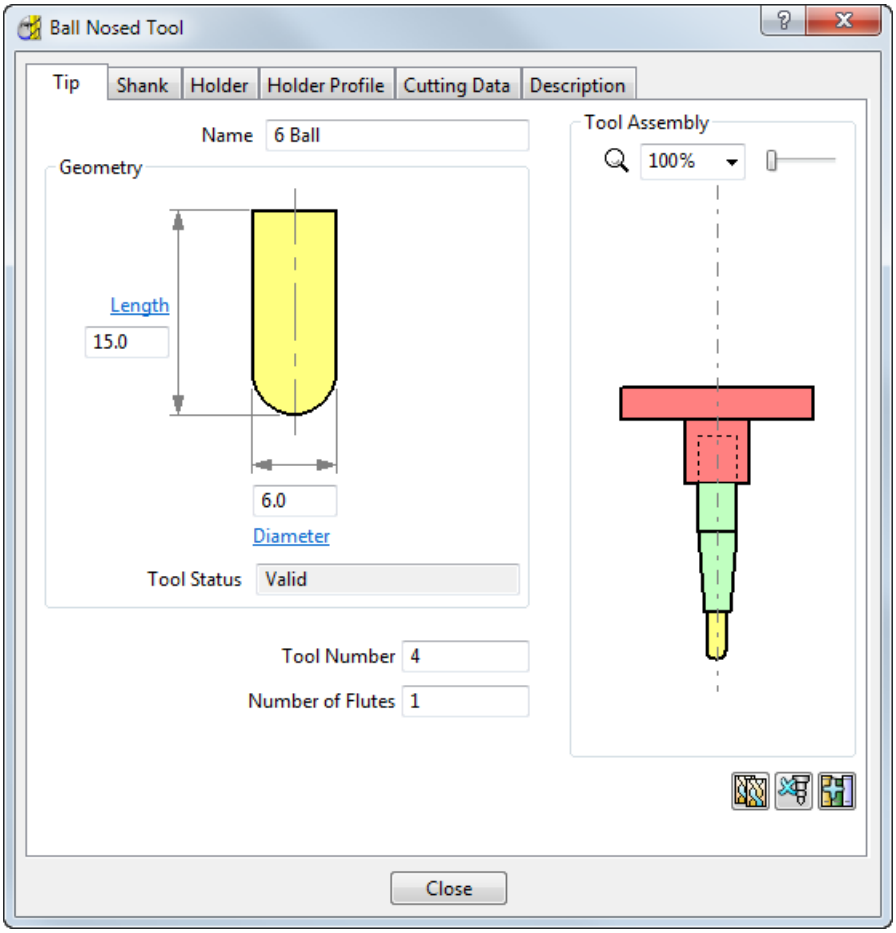
It is given the default name of **8 Ball_LS_TH_1**.

a Rename the tool to **6 Ball**.

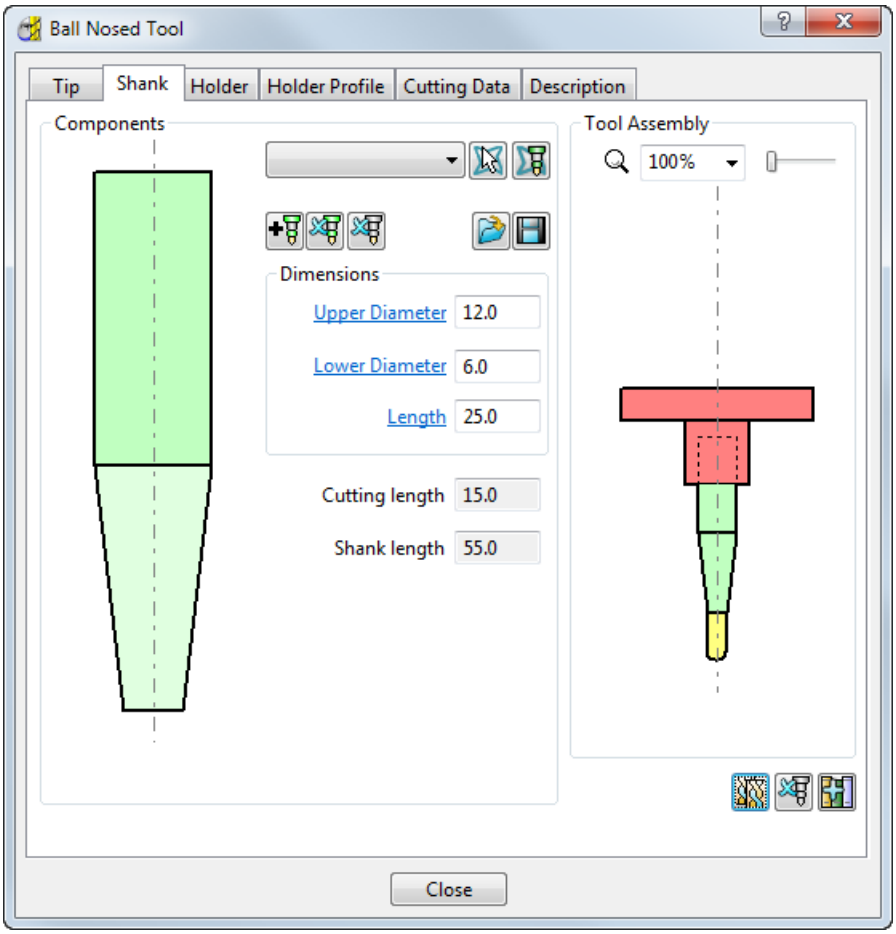
b Enter a **Length** of **15** mm.

c Enter a **Diameter** of 6 mm.

d Enter the **Tool Number** as 4.



4 To adjust the shank to match the tip, select the **Shank** tab, click the bottom shank component (it turns pale green), and enter **6** in the **Lower Diameter** field.



5 You can leave the **Holder** as it is. Click **Close** to update the toolpath dialog with the new tool.

The new tool is shown in the explorer, and on the **Tool** toolbar, and it's also drawn in the graphics window, where it is automatically aligned with the Z axis.

Completing and generating the Corner Finishing toolpath

In the **Corner Finishing** strategy dialog:

1 Select the **Corner finishing** page and:

a From the **Output** list, select **Both**. This creates two separate toolpaths for steep and shallow regions.

b Enter a **Threshold Angle** of **65**. This specifies the angle, measured from the horizontal, that determines the split between steep and shallow portions of the surface slope.

c Enter a **Cusp** of **0.01**. This defines the maximum allowable cusp height. The stepover between tool passes is automatically calculated from this value and the geometry of the tool and part.

d From the **Cut direction** list, select **Any**. This uses both the **Climb** and **Conventional** methods of milling.

2 Select the **Corner detection** page and:

a Select **8 Ball_LS_TH** from the toolpath reference list. The **Corner Finishing** toolpath makes a comparison between the current and previous tools, and automatically machines the corners that the previous tool was unable to access.

b Enter an **Overlap** of **0.5**. This indicates how far the toolpath is extended beyond the borders of the un-machined region. It is also used as the overlap value between the steep and shallow portions of the toolpath.

c Enter a **Detection limit** of **165**. This specifies the angle at which PowerMILL finds corners. Only corners *less* than the specified angle are machined.

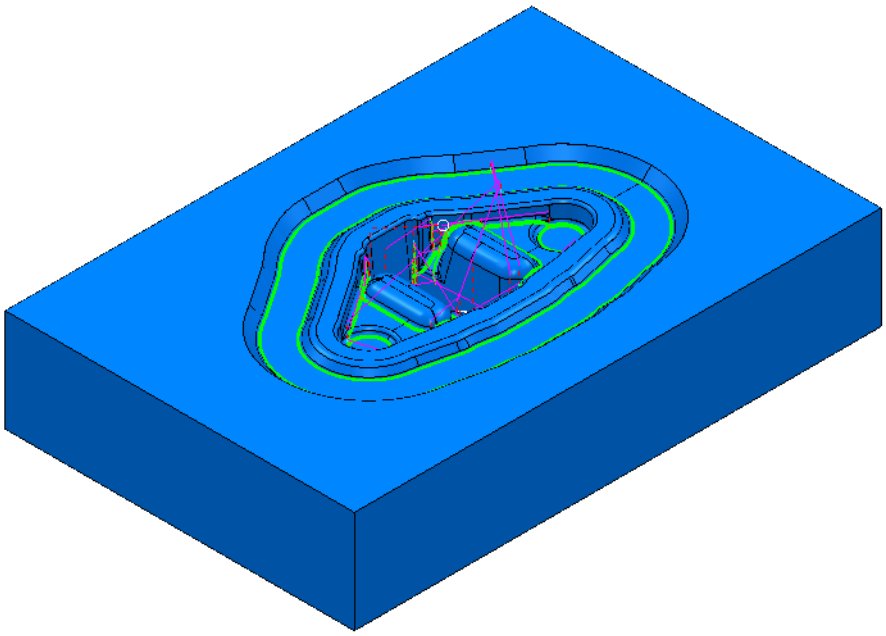
3 Click **Calculate** to generate the toolpath.

Progress is shown on the **Status** bar at the bottom of the screen. The generation may take a minute or so, depending on the processing power of your PC.

4 After the toolpath is generated, **Close** the strategy dialog.

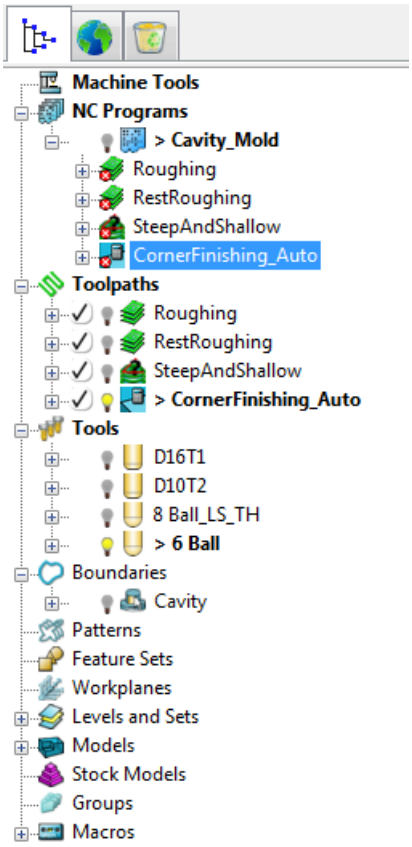
Displaying the Corner Finishing toolpath

When the toolpath is generated, it is displayed on the screen:



Click ⊞ to expand the **Toolpaths** node in the explorer. The new toolpath is shown in **bold** and preceded by the **>** symbol to indicate it's active.

If the NC program **Cavity_Mold** remains active, the toolpath is automatically added to it:

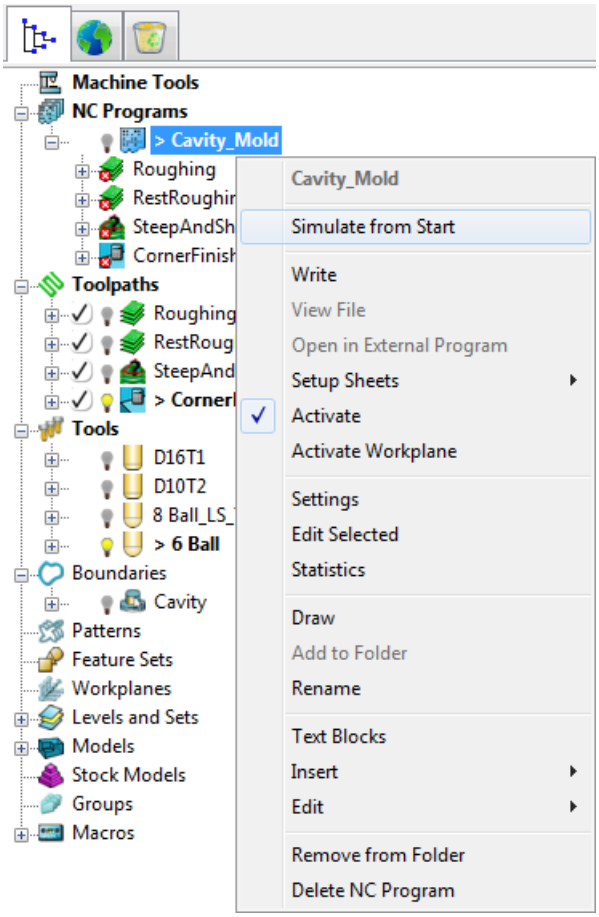


To save project changes, click on the **Main** toolbar.

Simulating the Corner finishing toolpath

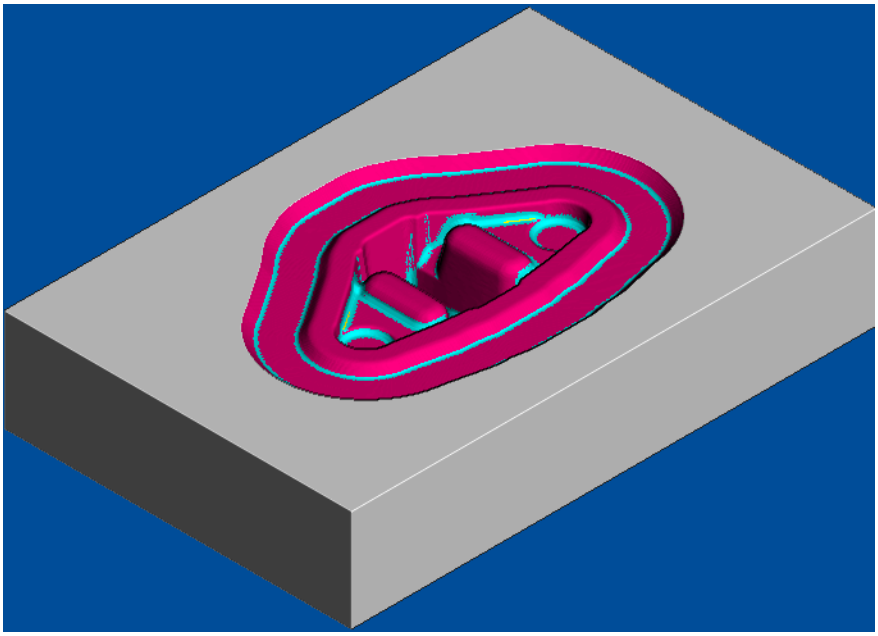
1. Click the **ISO1** button on the **View** toolbar to reset the view.
2. On the **ViewMill** toolbar, click the **Toggle ViewMill Window** button. It turns green, and activates the simulation window, which initially shows a light grey block on the current background.
3. From the **ViewMill Toolbar**, select the **Rainbow Shaded Image** option so you can visualise the differences between toolpaths.

4 In explorer, right-click the NC program **Cavity_Mold**, and select **Simulate from Start** from the context menu.

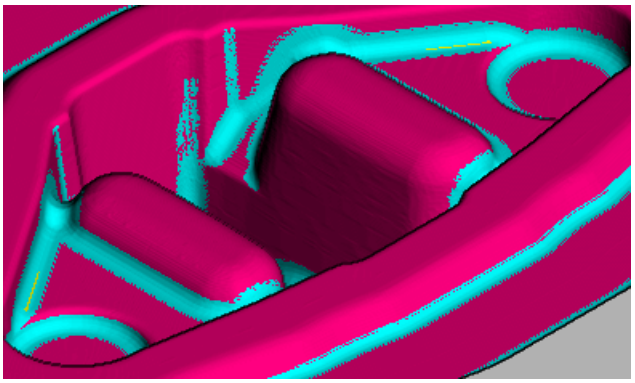


The NC program is automatically selected on the **Simulation** toolbar, and the **Play** buttons enabled.

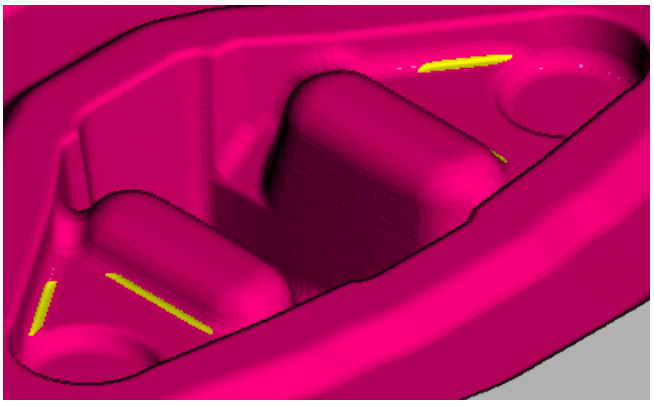
5 Click the **Play** button, and allow the simulation to run through to the end.



6 Zoom into the machined areas to view the cleaning up of the non-tangential surfaces.



This compares with the previous finishing toolpath which left some material in the corners.



7 Click the **Exit ViewMill** button, and select **Yes** to stop the simulation. The **Toggle ViewMill** button changes from green to red, and the standard PowerMILL window is displayed.

Writing NC programs

When the toolpaths are generated, you can add them to an NC program to be postprocessed as an output file for a specific NC machine controller. Any number of toolpaths can be included, and reordered as required depending on the limitations of the particular NC machine and the postprocessor.

By default, all toolpaths in an NC program are saved to a single NC program file. The following examples show you how to:

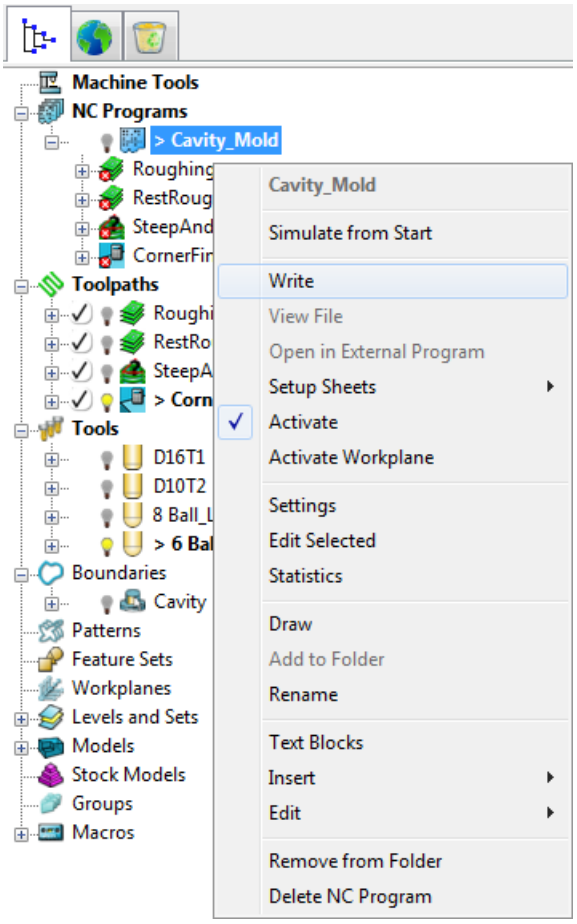
- Write each toolpath as a separate NC program file (see page 77).
- Write two NC program files (see page 79) with the toolpaths grouped by their functionality.

Writing each toolpath as a separate NC program file

This procedure shows how to generate separate NC program files for each toolpath in an NC program.

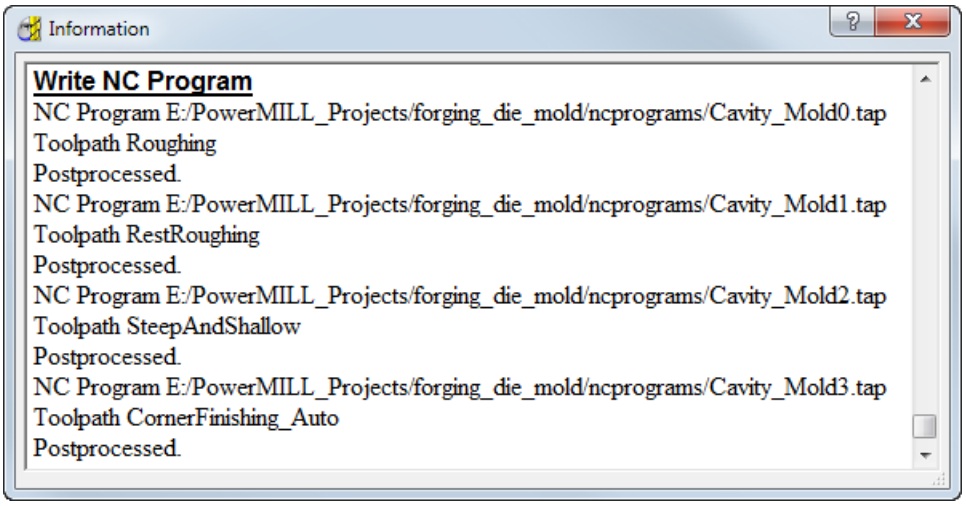
*To have separate NC program files for each of your toolpaths, the **Write File for Each Toolpath** option must be selected on the **Output** tab available from **Tools > Options > NC Programs**.*

1 In explorer, right-click the NC program, **Cavity_Mold**, and select **Write** from the context menu.

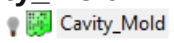


The symbol against a toolpath indicates a tool change. It is always displayed for the first tool in the sequence. It is also displayed when a different tool is used.

2 PowerMILL postprocesses the toolpaths using the specified parameters, and displays a confirmation window showing where the files are saved.



3 Click to close the **Information** window.

4 The colour of the NC program **Cavity_Mold** in the explorer changes to a bright green colour, 

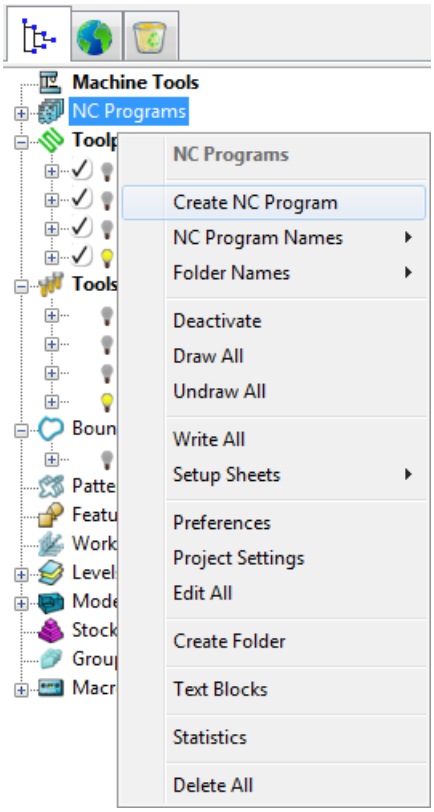
, to show it has been calculated correctly.

5 To save project changes, click  on the **Main** toolbar.

Writing two NC program files

This procedure shows how to generate two NC program files, one with two roughing toolpaths, and another with the two finishing toolpaths.

1 From the **NC Programs** context menu, select **Create NC Program**.



This displays the **NC Program** dialog.

2 In the **Name** field, enter **Cavity_Roughing**.

3 In the **NC Program** dialog, click the **Options**  button. The **Options** dialog is displayed.

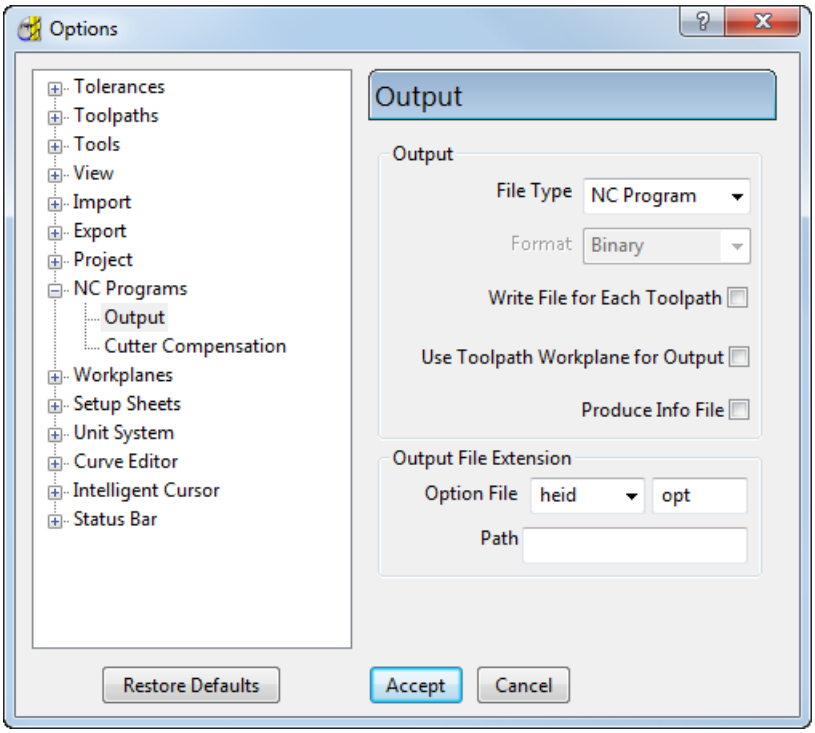
4 On the **Output** tab:

a If selected, deselect the **Write File for Each Toolpath** option. This causes **Output File** to be displayed instead of **Root Name** at the top of the **NC Program** dialog.

b Select **Option File** as **heid.**

c Enter **Output File Extension** as **opt**.

d Click **Accept** to update and close the **Options** dialog.



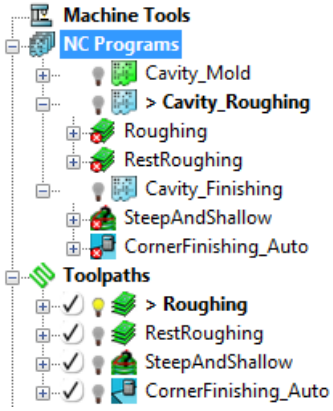
5 Click the **Accept** button at the bottom of the **NC Program** dialog to create a new roughing NC program.

6 Right-click the individual NC program **Cavity_Roughing**, and select **Edit > Copy NC Program**.

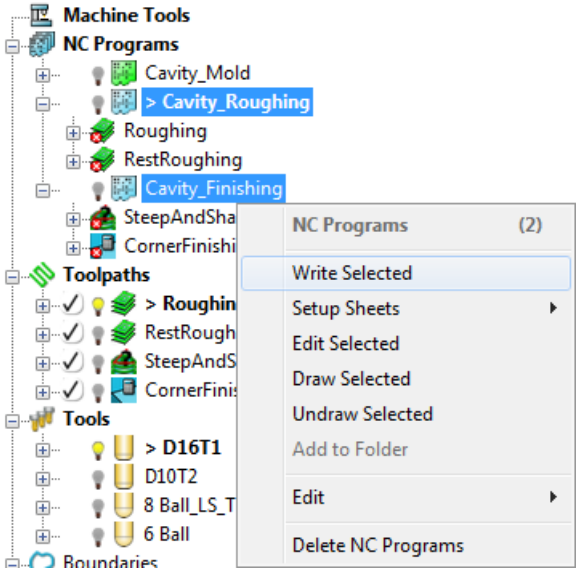
7 A new entity is added to the **NC Programs** list with the default name of **Cavity_Roughing_1**. Right-click it and **Rename** it as **Cavity_Finishing**.

*To identify which NC program is currently active, expand the **NC Programs** node in the explorer; the active program is shown in **bold** and preceded by the **>** symbol.*

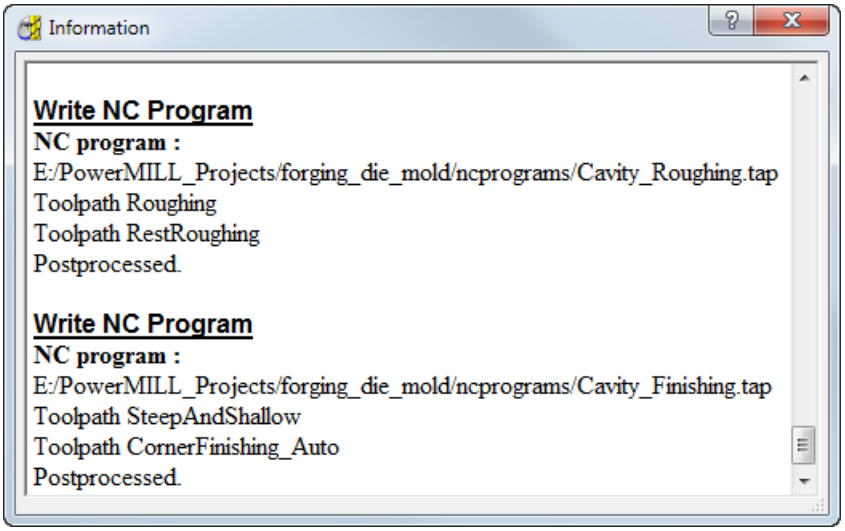
8 In explorer, move the two roughing toolpaths under **Cavity_Roughing**, and the two finishing toolpaths under **Cavity_Finishing**.



9 Select the **Cavity_Roughing** and **Cavity_Finishing** NC programs. Right-click, and select **Write Selected**.



10 PowerMILL post-processes the NC programs using the specified parameters, and displays a confirmation window showing where the programs are saved.



*Two NC programs are written, **Cavity_Roughing.tap** containing both roughing toolpaths, and **Cavity_Finishing.tap** containing both finishing toolpaths.*

11 Click to close the **Information** window.

12 To save project changes, click on the **Main** toolbar.

Index

P

R

S

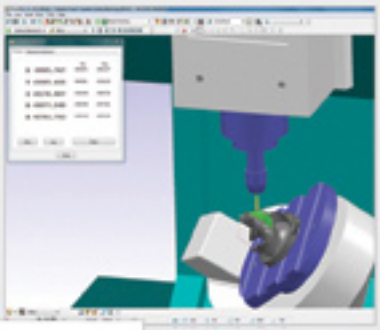
T

V

W

PowerMILL 2012 R2

PowerMILL 2012



new features

Delcam TV

www.delcam.tv

PowerMILL Learning Zone

www.delcam.tv/lz

PowerMILL Website

www.powermill.com

THE QUEEN'S AWARDS FOR ENTERPRISE: INNOVATION 2010

THE QUEEN'S AWARDS FOR ENTERPRISE: INTERNATIONAL TRADE 2011

Powering your productivity

Delcam Headquarters | Small Heath Business Park | Birmingham | B10 0HJ | UK
+44 (0)121 766 5544 | marketing@delcam.com | www.delcam.com

To contact your local reseller, visit **www.delcam.com/resellers**