PowerMILL 2016

Getting Started

Release issue 1

PowerMILL

Copyright © 1996 - 2015 Delcam Ltd. All rights reserved.

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

The functionality and user interface in this manual is subject to change without notice in future revisions of the software.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence.

Delcam Ltd grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

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

The Vortex machining functionality is subject to patent applications.

Patent application: 1121277.6 Adaptive Clearance

The MachineDNA functionality is subject to patent applications.

Patent application: 1204908.6 Machine Testing

Licenses

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

PowerMILL 2016. Published on 13 May 2015

Contents

Introduction 1

Setting up your working directories 2
Setting up a Home folder in Windows 7 for user-defined macros 2
Specifying PowerMILL Default Directory paths 3
Specifying PowerMILL temporary file area 4

Start and close PowerMILL 5

Startup window 6
Explorer 9
Unit System 10
Accessing Help 12
Tooltips 12
Status bar help 12
Context-sensitive online help 12
Manuals 13

Cavity mold example 14

Loading the model 15
Defining the block around the die 17
Saving the project for the first time 19
Viewing the model and block 20
Predefined views 20
Zoom 22
Panning 24
NC program 24
Overview Template 24
Creating an NC program 26
Creating a roughing toolpath 28
Defining the roughing tool geometry 30
Defining the tolerances 34
Specifying rapid move heights 34
Specifying tool start point 35
Generating the roughing toolpath 36
Displaying the roughing toolpath 37
Saving project changes 39
Simulating the roughing toolpath 40
Creating the rest roughing toolpath 41
Defining the rest roughing tool geometry 42
Changing the Stepover and Stepdown values 44

Generating the rest roughing toolpath 44
Displaying the rest roughing toolpath 45
Simulating the rest roughing toolpath 46
Closing the roughing session 47
Re-opening the project 47
Checking which entities are active 48
Creating a finishing toolpath 49
Defining the finishing tool geometry 50
Creating the boundary for the selected surface 56
Completing and generating the finishing toolpath 59
Displaying the finishing toolpath 63
Simulating the NC program with generated toolpaths 64
Creating the Corner Finishing toolpath 66
Defining the Corner Finishing tool geometry 67
Completing and generating the Corner Finishing toolpath 69
Displaying the Corner Finishing toolpath 70
Simulating the Corner finishing toolpath 71
Writing NC programs 74
Writing each toolpath as a separate NC program file 74
Writing two NC program files 76

Index **81**

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 and how to 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 the examples in this guide 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:

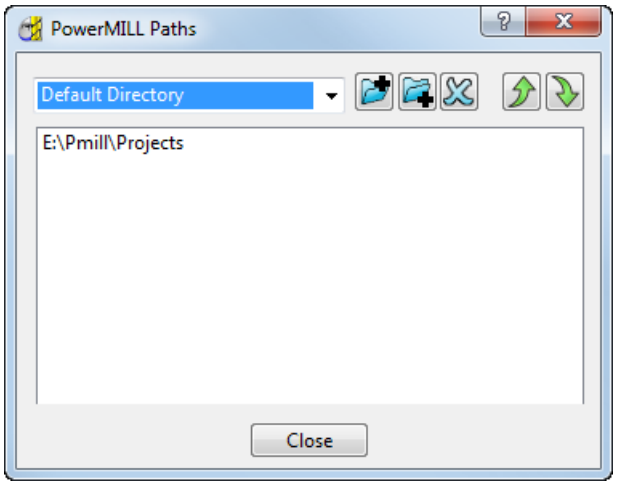
1. Open **Windows Control Panel** and click **System > Advanced system settings**. This displays the **System Properties** dialog.
2. Click the **Advanced** tab.
3. Click **Environment Variables**.
4. To add a new variable name and value, click **New** to display the **New User Variable** dialog.
   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 enables 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:

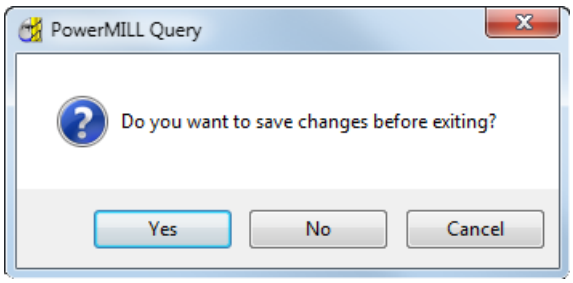
- Select **Start > All Programs > Delcam > PowerMILL > PowerMILL 2016**; or,
- Double-click the PowerMILL shortcut on your desktop:

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

To exit PowerMILL:

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

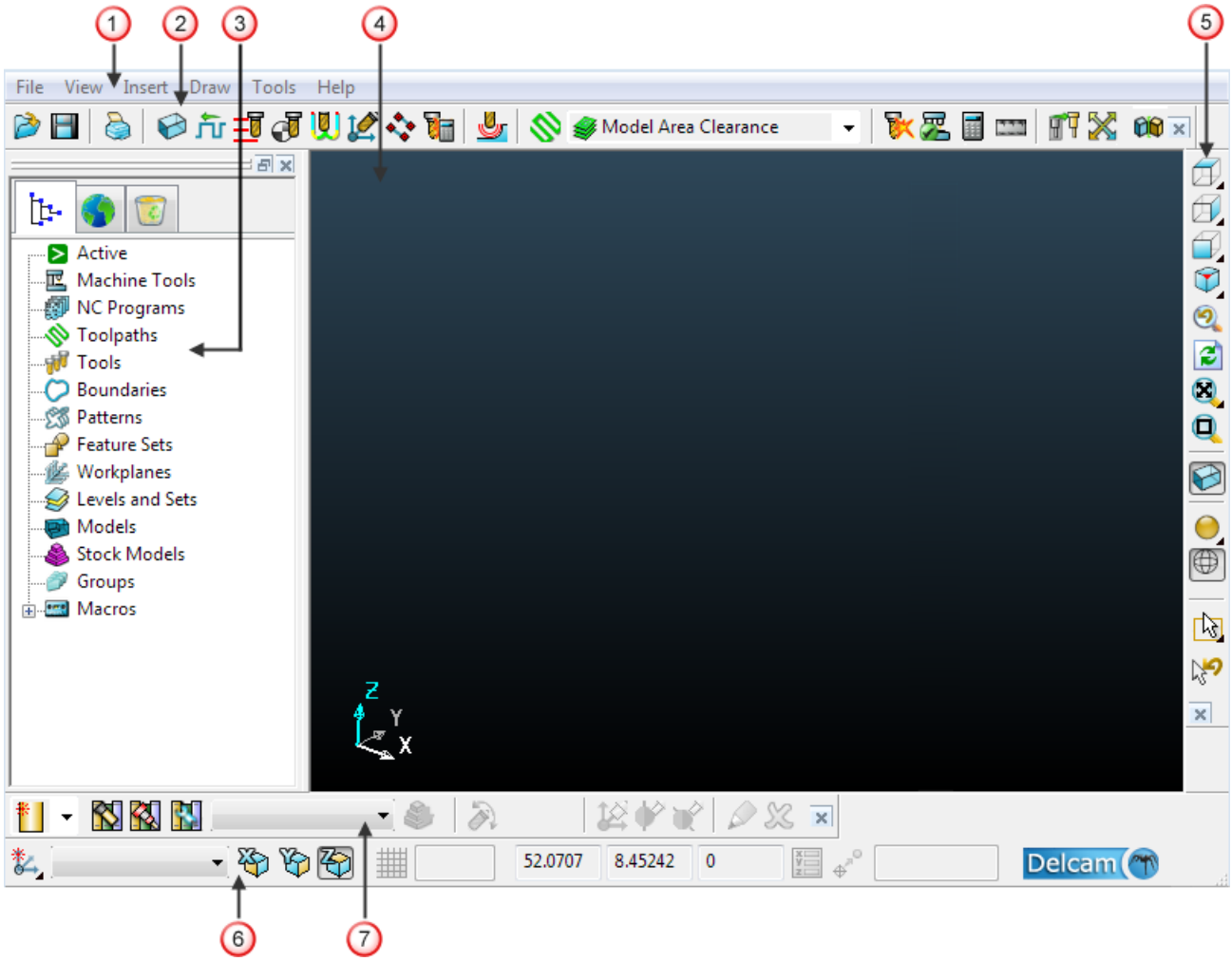
If you have not saved your project, the following message is displayed:



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

Startup window

The following window is displayed when you start 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 options 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.

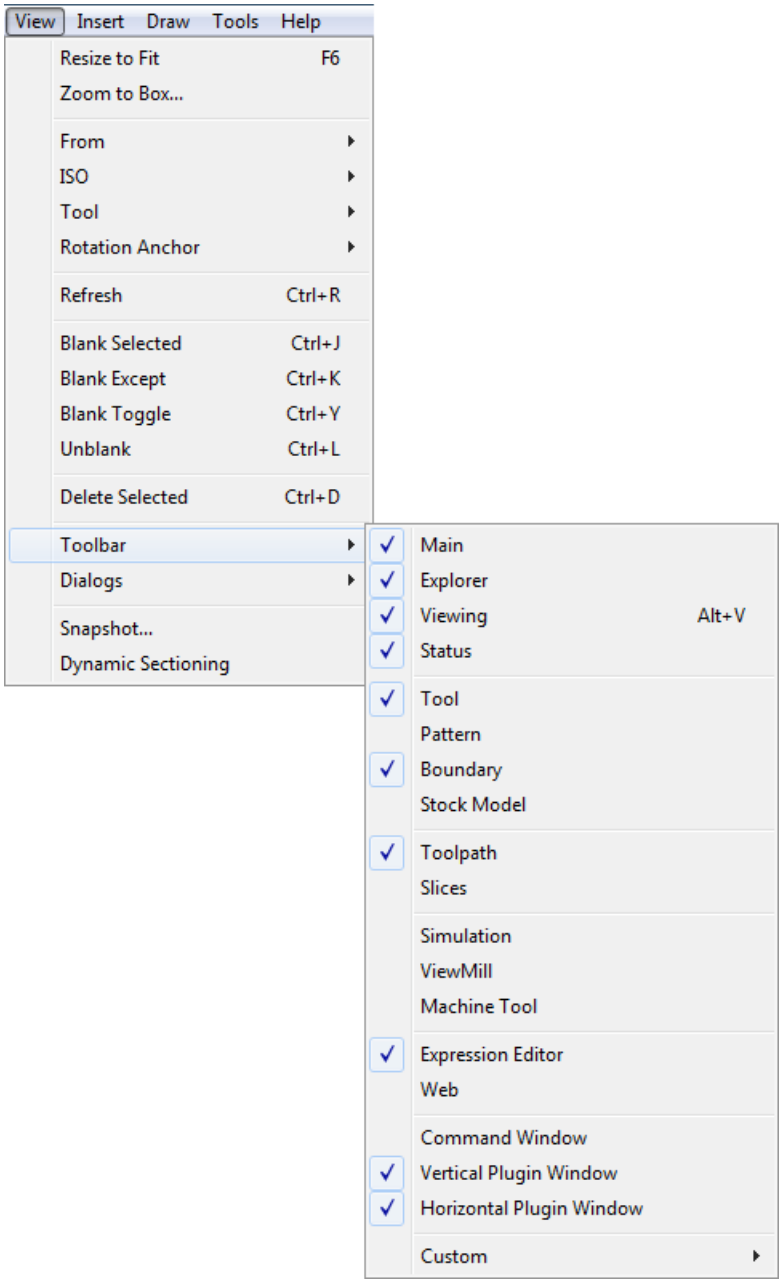
4 **Graphics window** is the working area of the screen.

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

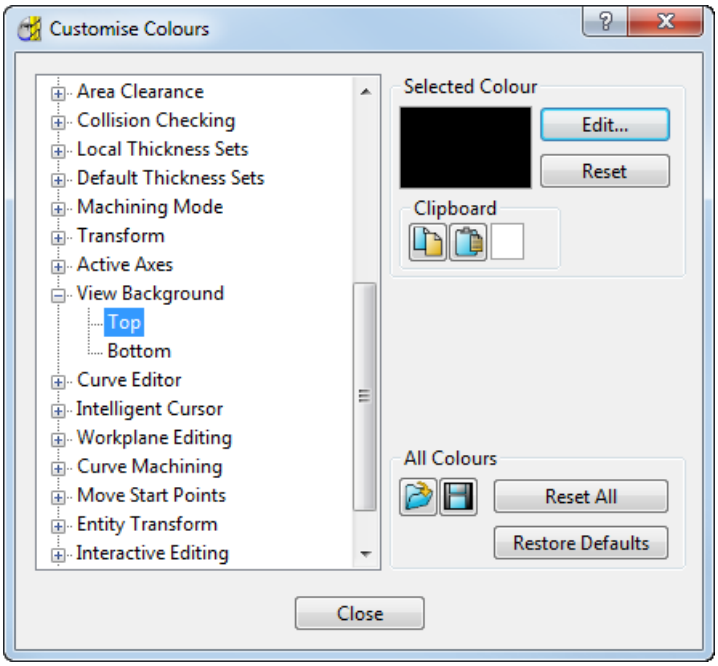
6 **Status and Information toolbar** enables you to create and activate workplanes, display various preset fields and display user defined fields. If you hover the cursor over a button, help is displayed rather than the information toolbar. The help can be, for example, a brief description of the item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

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

*PowerMILL features many toolbars which are not, by default, displayed at startup. To display one of PowerMILL's extra toolbars, select **View > Toolbar** and the toolbar you want to display, for example **Toolpath**. A ✓ next to a toolbar name indicates the toolbar is currently displayed.*



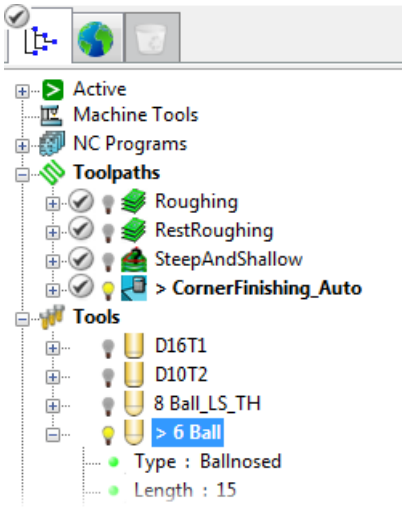
*If you want to change background colours, select **Tools > Customise Colours** and choose **View Background**. You can change the **Top** and/or **Bottom** colours and **Reset** them or **Restore Defaults** if you change your mind afterwards.*



*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 is displayed the next time you start PowerMILL.*

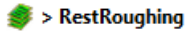
Explorer

The Explorer displays the entities used in a PowerMILL project. (The term *entity* broadly refers to all entries in the Explorer.) The entities in the Explorer are grouped together by type, for example, toolpaths are grouped in the **Toolpaths** branch and tools are grouped in the **Tools** branch. Because a PowerMILL project often contains a lot of entities, the Explorer uses an **Active** > system to indicate which entities have focus.



Making an entity active (shortcut)

Double-click an entity to make it active (you need to click the entity's icon and not its name). The new active entity is displayed in **bold** and proceeded by an angle bracket (**>**).



Each branch can only have one active entity at a time.

Drawing/unbarring an entity in the graphics area

You can draw or undraw entities in the graphics area. Click an entity's light bulb to specify its display setting:

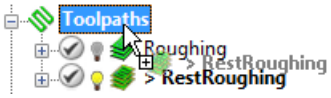
— Entity is always undrawn.

— Entity is drawn only when active.

— Entity is always drawn.

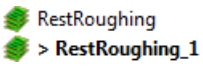
Duplicating an entity (shortcut)

1 Press **Ctrl** and drag the entity onto the title of its branch (when you drag the entity, position the mouse over the entity's icon).



2 Release the left mouse button when ⃠ changes to ⊞.

This creates a copy of the entity with the suffix _**1**. If you duplicate the entity again the suffix is _**2** and so on.



Displaying the context menus

There are two types of context menus available in the Explorer: general context menus and specific context menus. To display the context menus:

General — Right-click the title of an entity branch. The menu options are relevant to all the entities in the branch.

Specific — Right-click an individual entity. The menu options are relevant only to the entity.

Deleting an entity

Right-click the entity and select the delete menu option, for example, **Delete Toolpath**. The entity is placed in the **Recycle Bin**.

To permanently delete an entity:

- select the entity and press **Shift**+**Delete**; or
- right-click the entity and select the delete menu option while pressing **Shift**.

Permanently deleting entities reduces PowerMILL's memory requirements.

Repositioning the Explorer pane

Like all toolbars in PowerMILL, you can float and reposition the Explorer pane:

Double-click the top of the Explorer pane to float the pane and then drag it to a new position. To dock the pane on the opposite side of the window, drag the pane over the side of the window and release the left mouse button — the pane 'snaps' into position.

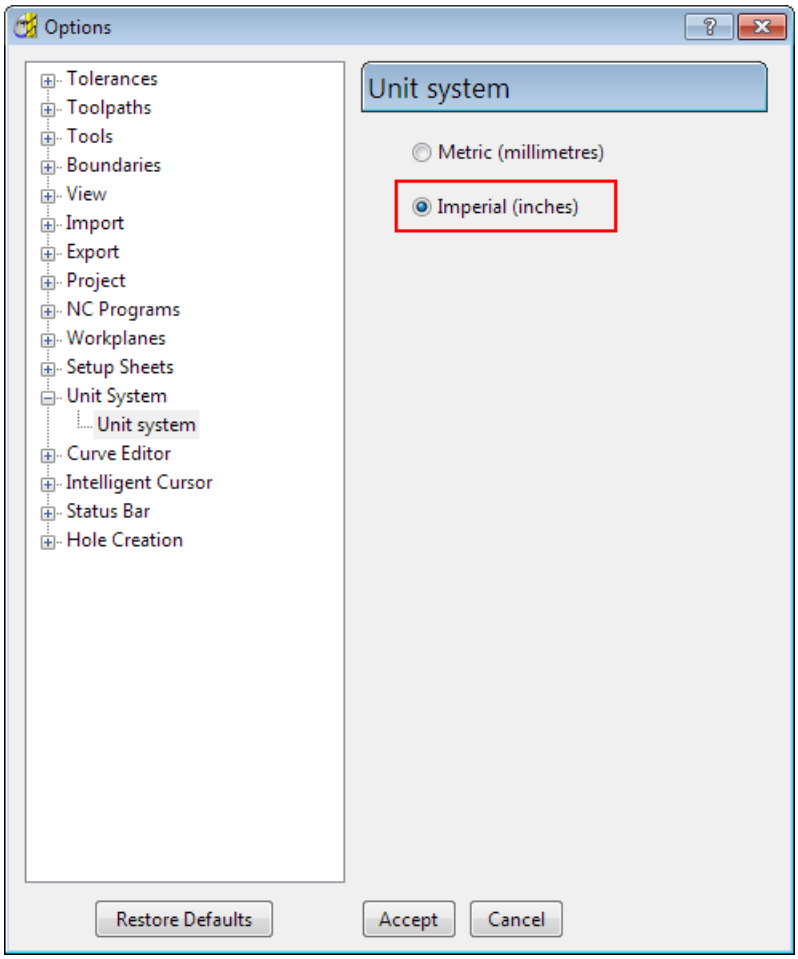
Unit System

By default, PowerMILL uses metric units. You can change the unit system before any project entities are 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:

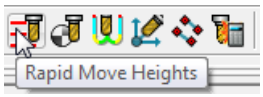
- Tooltips (see page 12)
- Status bar help (see page 12)
- Context-sensitive online help (see page 12)
- Manuals (see page 13)

For more information see Using the help window.

Tooltips

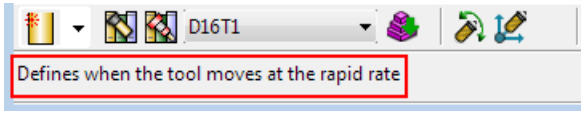
Tooltips are displayed by hovering the cursor over the required buttons.

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, which is located at the bottom of the screen. So, when you place the cursor over the **Rapid Move Heights** button, help is displayed in the **Status** bar as well as the tooltip.



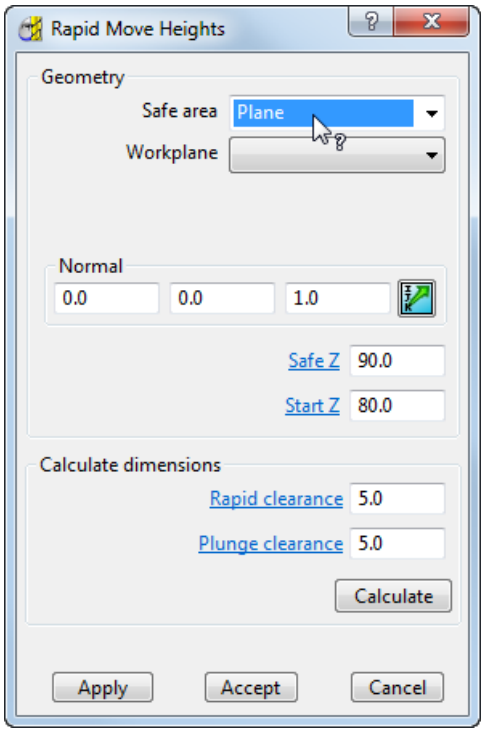
Context-sensitive online help

Pressing the **F1** key displays help for the area of the screen that has current focus, such as a display pane (Explorer or graphics window), a dialog, or a dialog tab. Where there is no focus, the contents page for the main **Help** is displayed, so 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 item 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** manual supplied with your PowerMILL DVD.
- **What's New** and **Getting Started** PDFs available for download.
- Online **Reference help**, **What's New** and **Getting Started**.

Cavity mold example

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

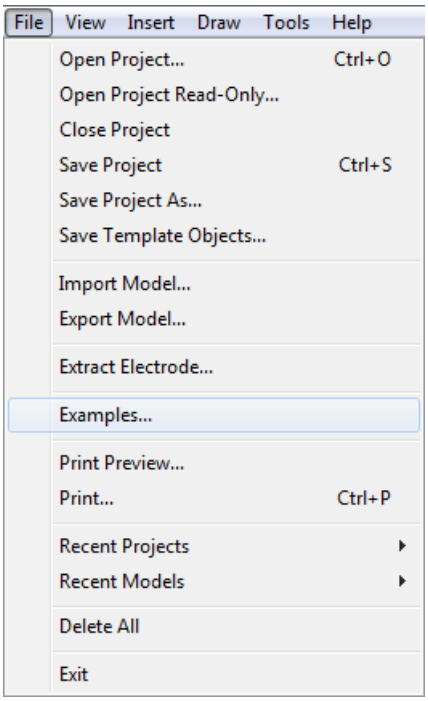
1. Start PowerMILL (see page 5).
2. Load the model (see page 15).
3. Save the project (see page 19).
4. View the model and block (see page 20).
5. Create an NC program (see page 24).
6. Create, simulate, and output each toolpath:
   - **Roughing** (see page 28) — Use this toolpath to rapidly remove the majority of excess material.
   - **Rest roughing** (see page 41) — Use this toolpath to remove material which the roughing toolpath is unable to remove. The toolpath uses the same strategy as the roughing toolpath but with a smaller tool to remove additional material.
   - **Finishing** (see page 49) — Use this toolpath to machine steep and shallow areas of the part.
   - **Corner Finishing** (see page 66) — Use this toolpath to smooth areas of the part which the previous toolpaths are unable to machine, for example corners between non-tangential surfaces.

Loading the model

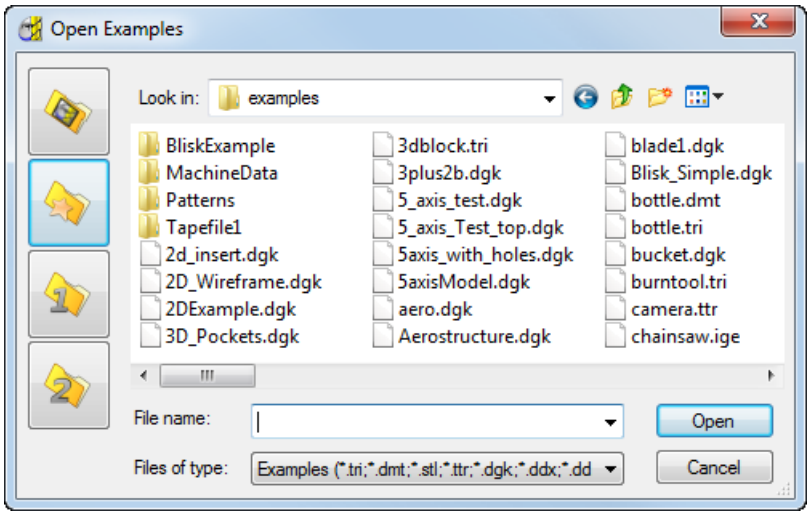
The example model is a cavity mold which is located in the Examples folder.

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:

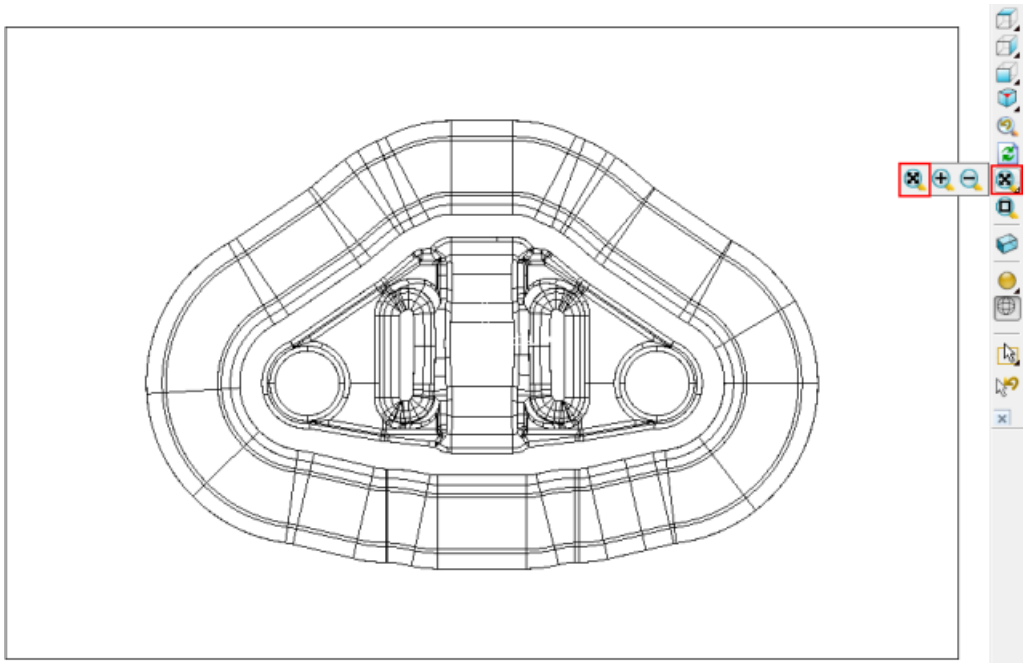


2 In the **Files of type** field, select **Delcam Geometry** (*.dgk) from the drop-down list.

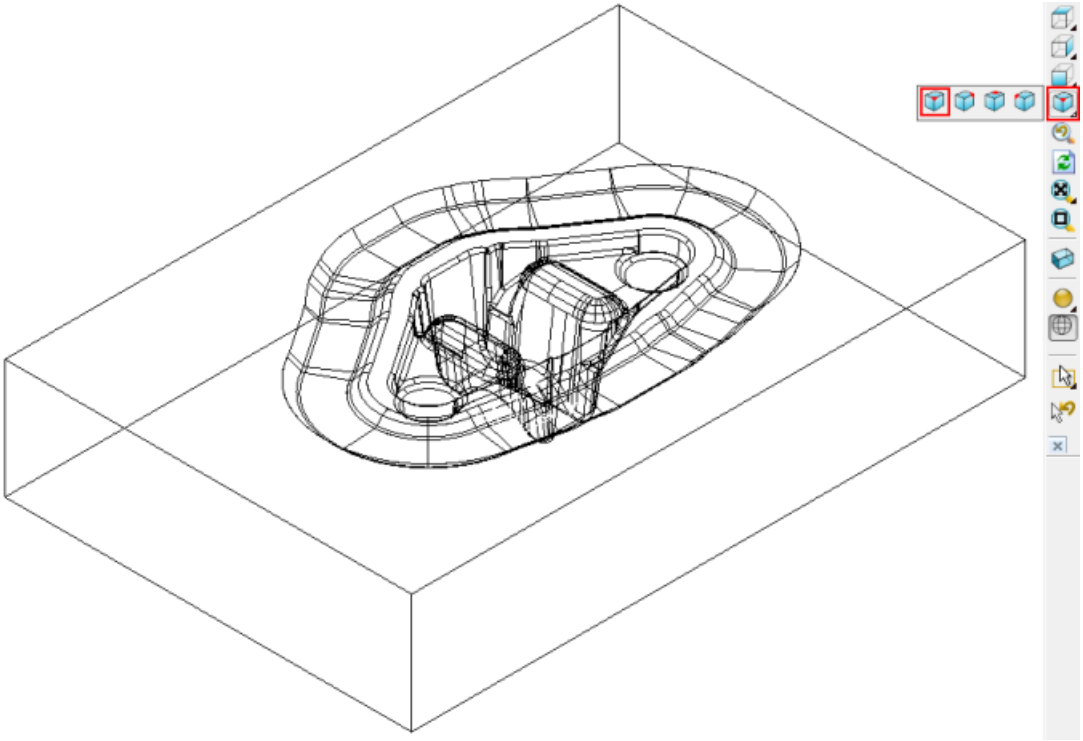
3 Select the cavity mold called die.dgk, and then click **Open**.

PowerMILL loads the file.

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

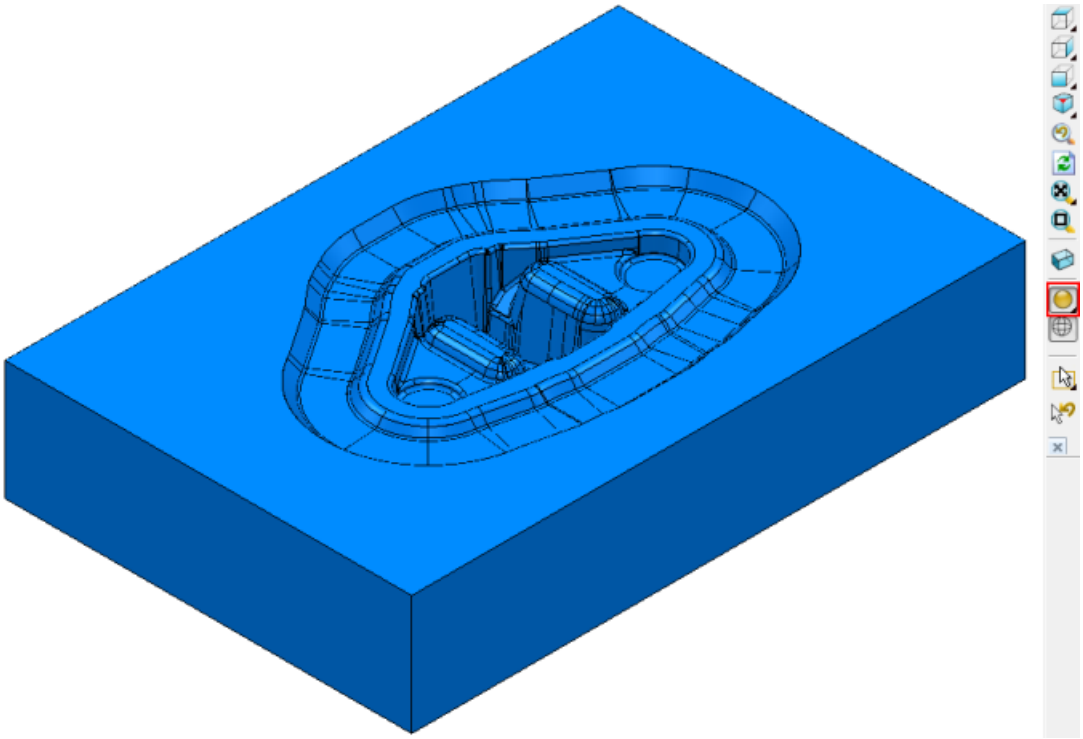


5 To change the display to an isometric view, select the **ISO 1** button on the same toolbar. The model looks like this:



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

6 The model is currently displayed 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 (located 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.

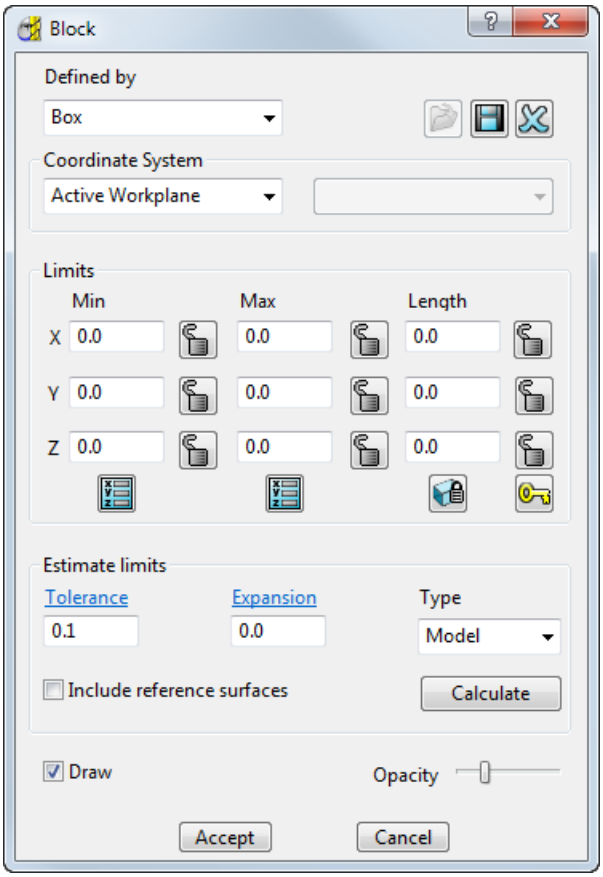
2 In the **Block** dialog, check that:

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

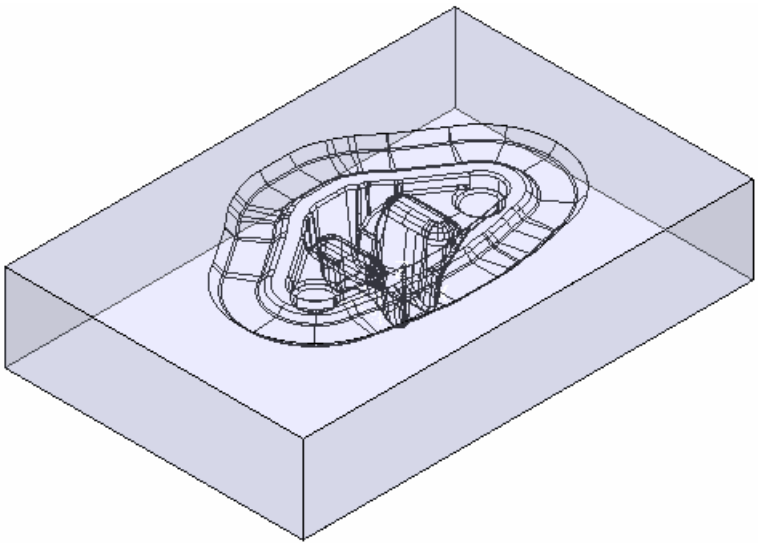
b **Coordinate System** is set to **Active Workplane**.

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.



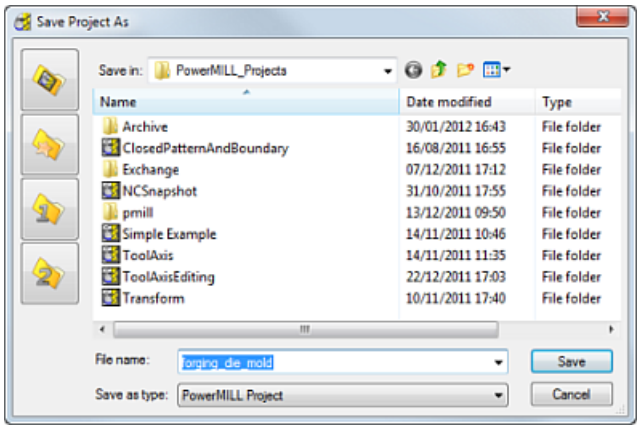
*To toggle the display of the block on and off, click the **Block** button on the **View** toolbar.*

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:

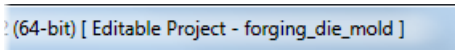
1 Click the **Save** button on the **Main** toolbar. Since you have not previously saved the project, the **Save Project As** dialog is displayed.



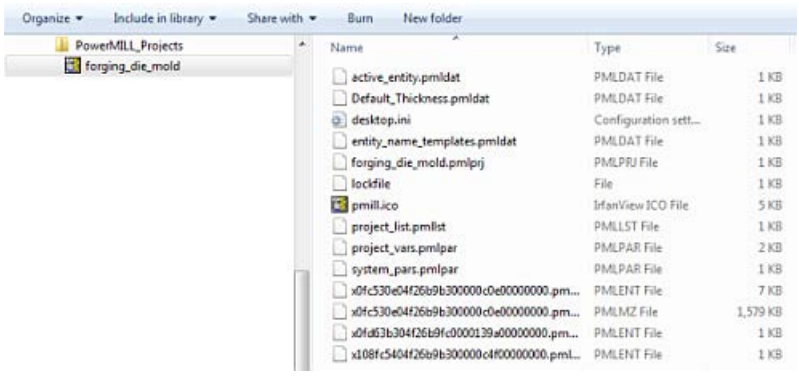
2 Open the directory that you created for your projects (see page 2), and enter a name for your project in the **File name** field, for example **forging_die_mold**.

3 Click **Save**.

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



An associated file structure is created:



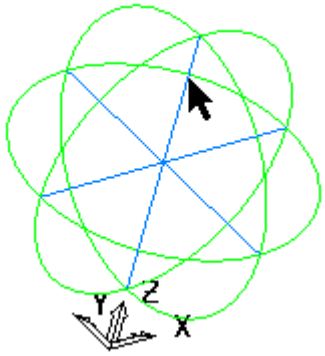
Do not manually edit these files *because you will corrupt your data.*

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 or block, you can either select one of the predefined views (see page 20) on the **View** toolbar, or use the mouse.

It is recommended that you use a three-button mouse. By holding down the middle button and moving the cursor in the graphics area, you can control the view of the model. When you move the mouse (with the middle button pressed), the trackball is displayed.



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.*

Use the **View Along** options to orientate the view so the model is viewed 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.

Use the **Previous View** option to view model as previously displayed.

Use the **Refresh** option to redraw all currently displayed components, repairing any damage to the current view. This is useful if the view loses definition as items are created, modified, or deleted.

Use the **Resize to Fit** (see page 22) option to adjust the view so the entire model can be seen. Depending on the current view, PowerMILL zooms in or out to achieve the new 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 you want to zoom in to. 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 **Previous Selection** option to revert to the previous selection made with the **Box Selection** or **Drag Selection** options.

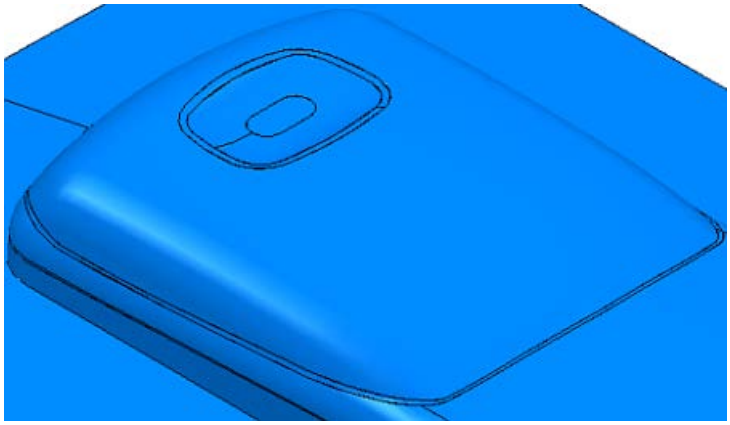
Zoom

Use the **Zoom** options on the **View** toolbar to resize the model.

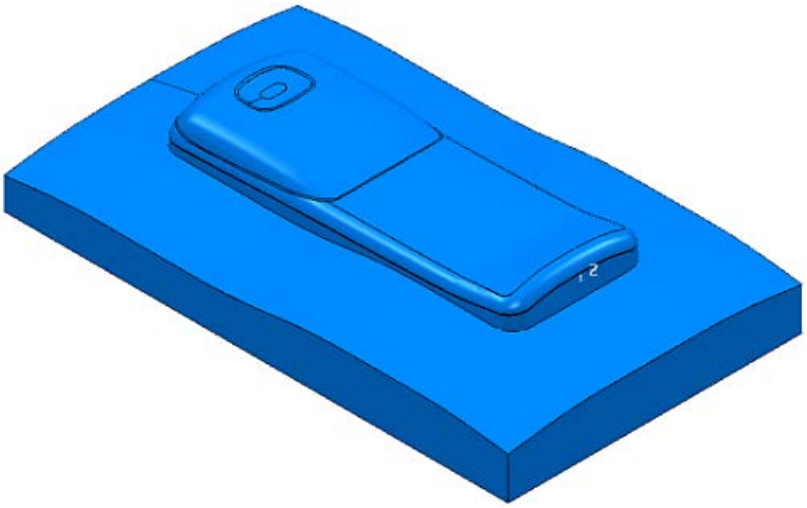
This is a pull-out toolbar. Hovering on the currently selected zoom button (in this case, **Resize to Fit**) displays the **Zoom** toolbar.

Resize to Fit — Use this option to adjust the view to see the entire model.

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.

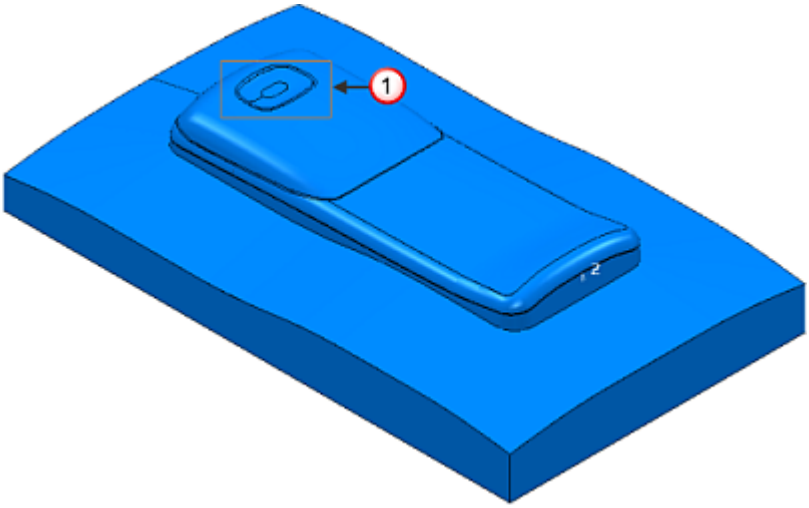
*If you cannot see your model, click the **Resize to Fit** button to centre the model at the middle of the screen.*

Zoom In — Use this option to zoom in to the model. Click repeatedly until desired view is achieved. This zooms the model about the centre of the graphics window.

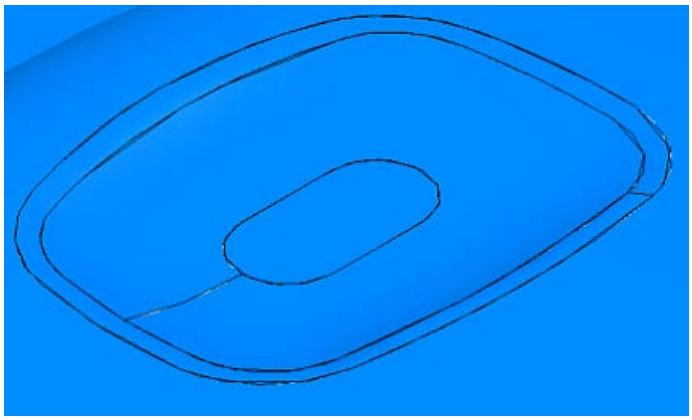
Zoom Out — Use this option to zoom out from the model. Click repeatedly until desired view is achieved. This zooms the model about the centre of the graphics window.

Zoom to Box — Use the option to zoom in to a specific area of the image. Click and then use the left mouse button to drag a zoom box over a section of the model. PowerMILL zooms in to the boxed area.

It converts this:



into this:



1 Zoom box

*You can also resize a model 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 model 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.

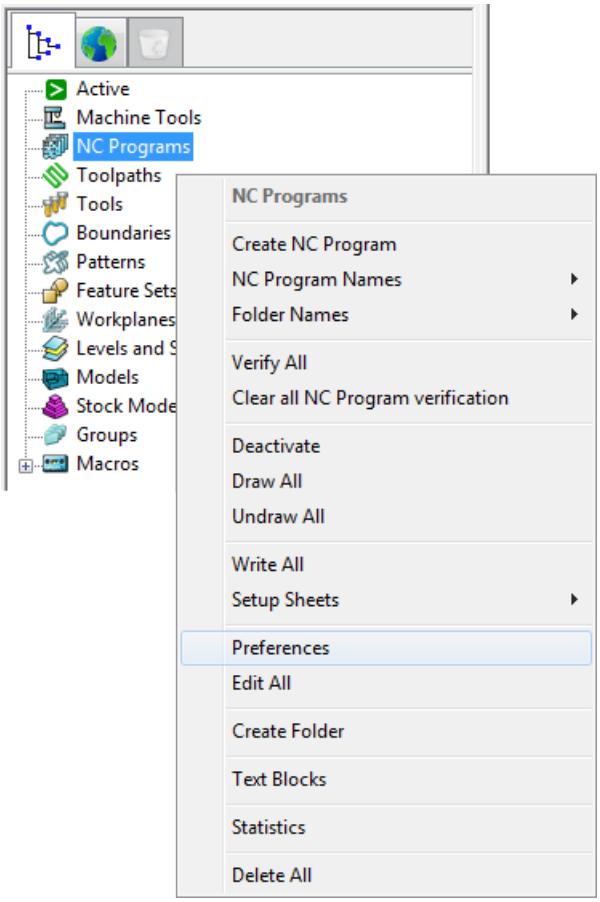
NC program

An NC program contains the commands and output settings that specify how the machine controller machines 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 76).

Overview Template

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



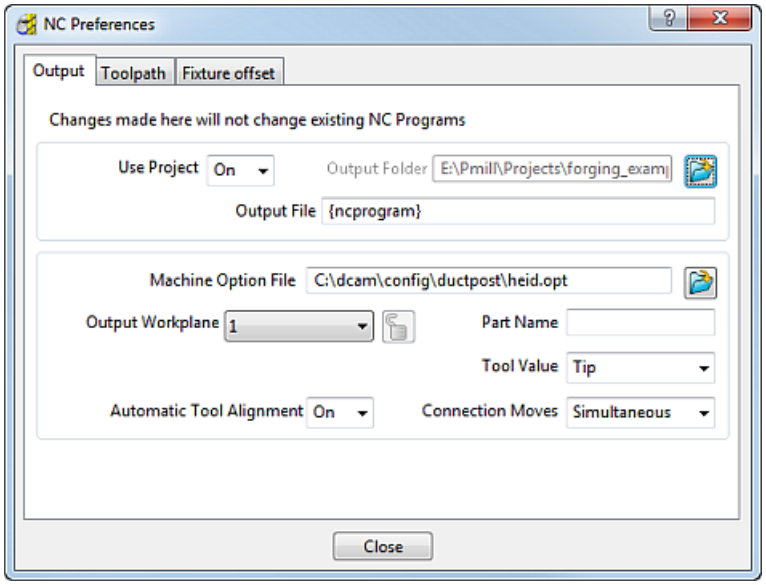
This displays the **NC Preferences** dialog.

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.*

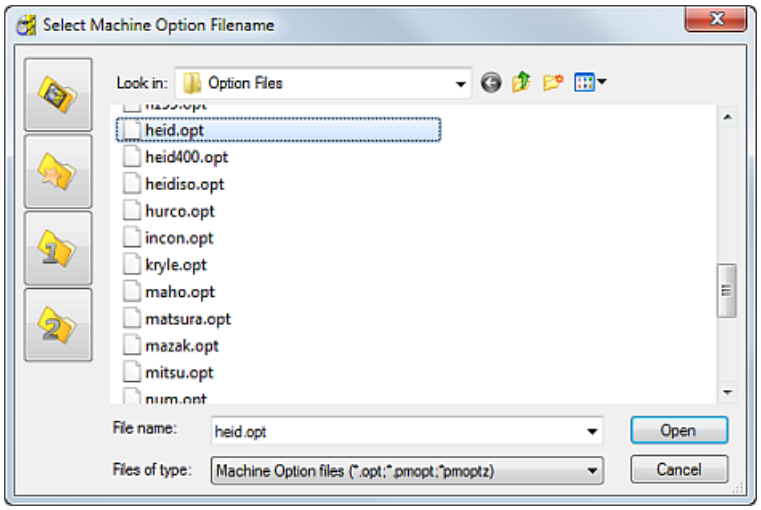
3 Enter the output file name to be used by default in the **Output File** field.

*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, select the machine option file (in this case **heid.opt**) and click **Open**.

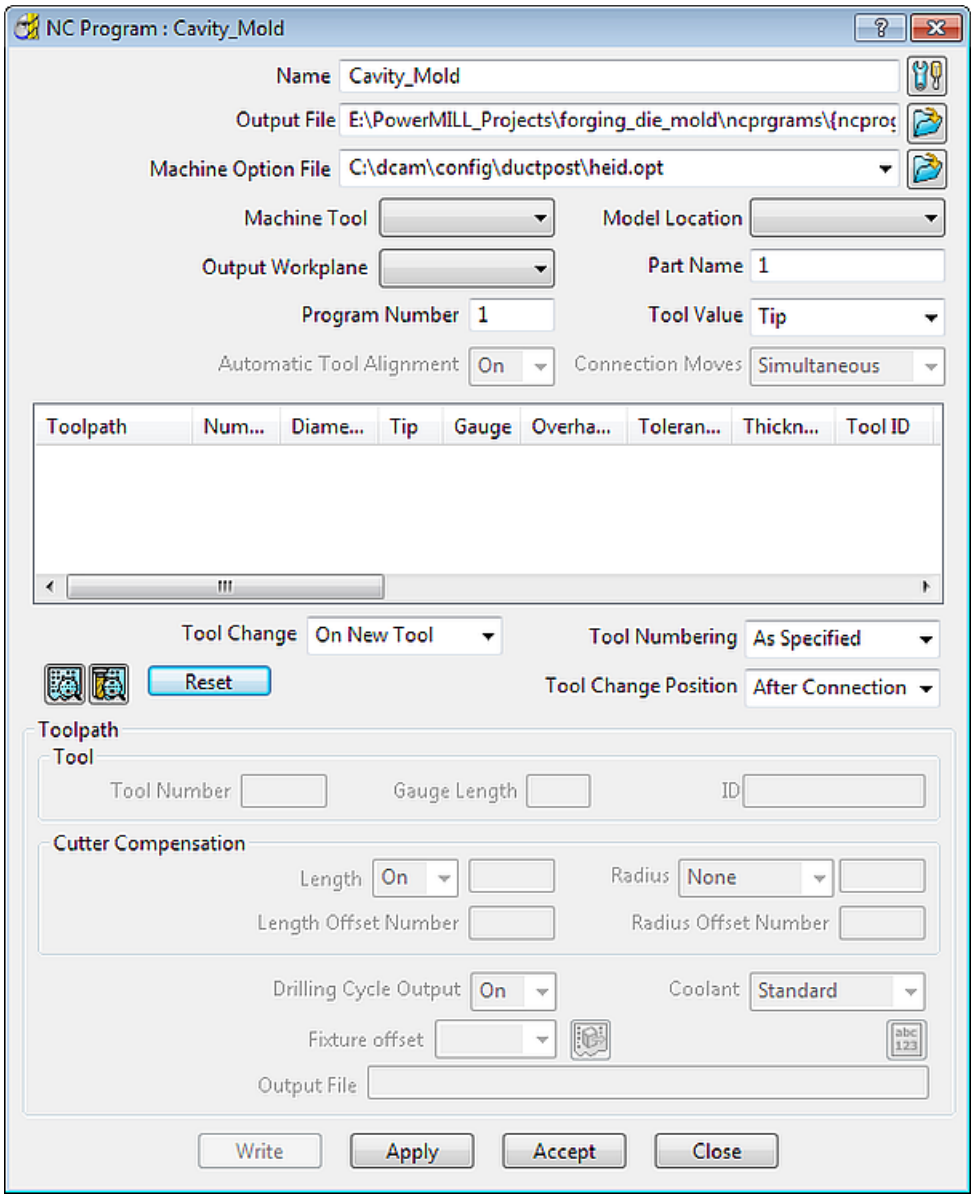


This closes the **Select Machine Option Filename** dialog and returns you to the **NC Preferences** dialog.

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

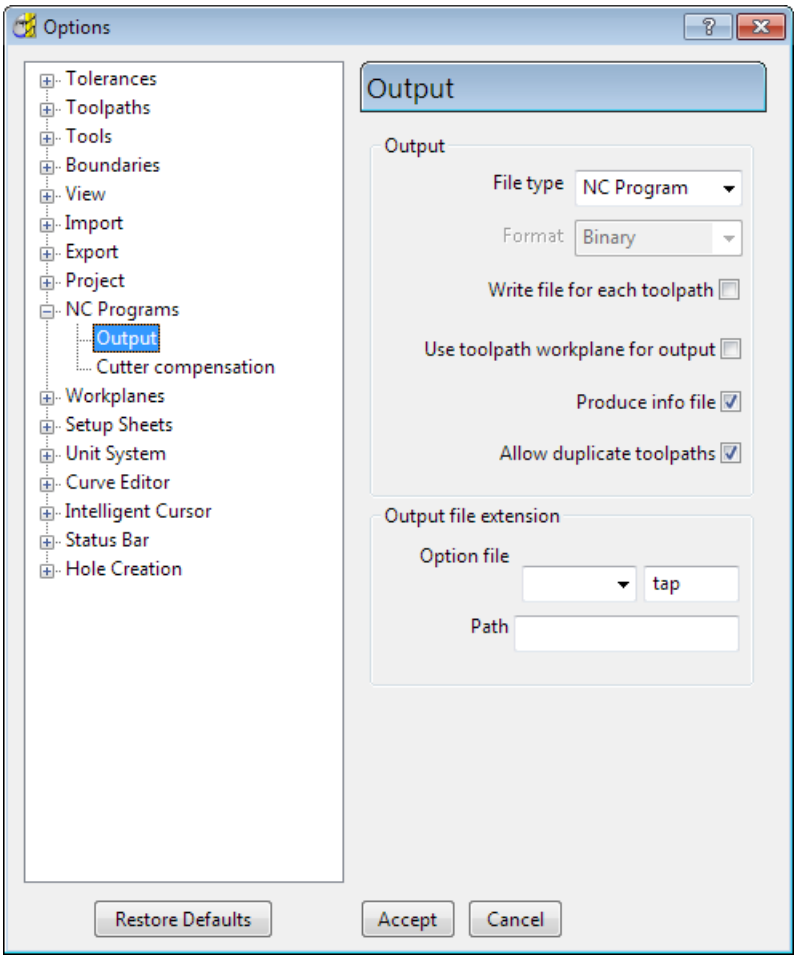
Creating an NC program

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. As a result, the **Output File** is displayed at the top of the **NC Program** dialog instead of the **Root Name**.
   b. Select **Option File** as **heid.**
   c. Enter **Output File Extension** as **opt**.

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

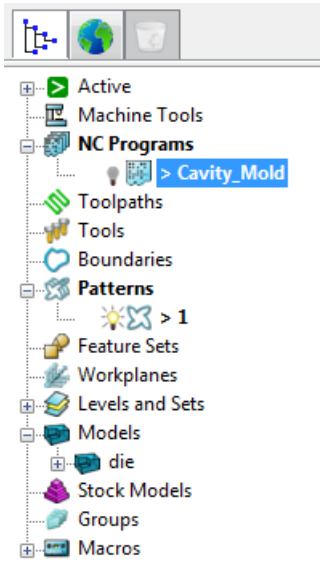


*If you want separate output files for each toolpath, select **Write File for Each Toolpath** option. This changes the **Output File** field to **Root Name** in the **NC Program** dialog and displays the default path and file name in the **Output File** field at the bottom of the dialog.*

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 over the path to see how the file name is displayed in 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).



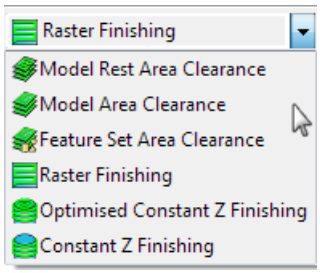
Any toolpaths you create are automatically added to the currently active NC program.

Creating a roughing toolpath

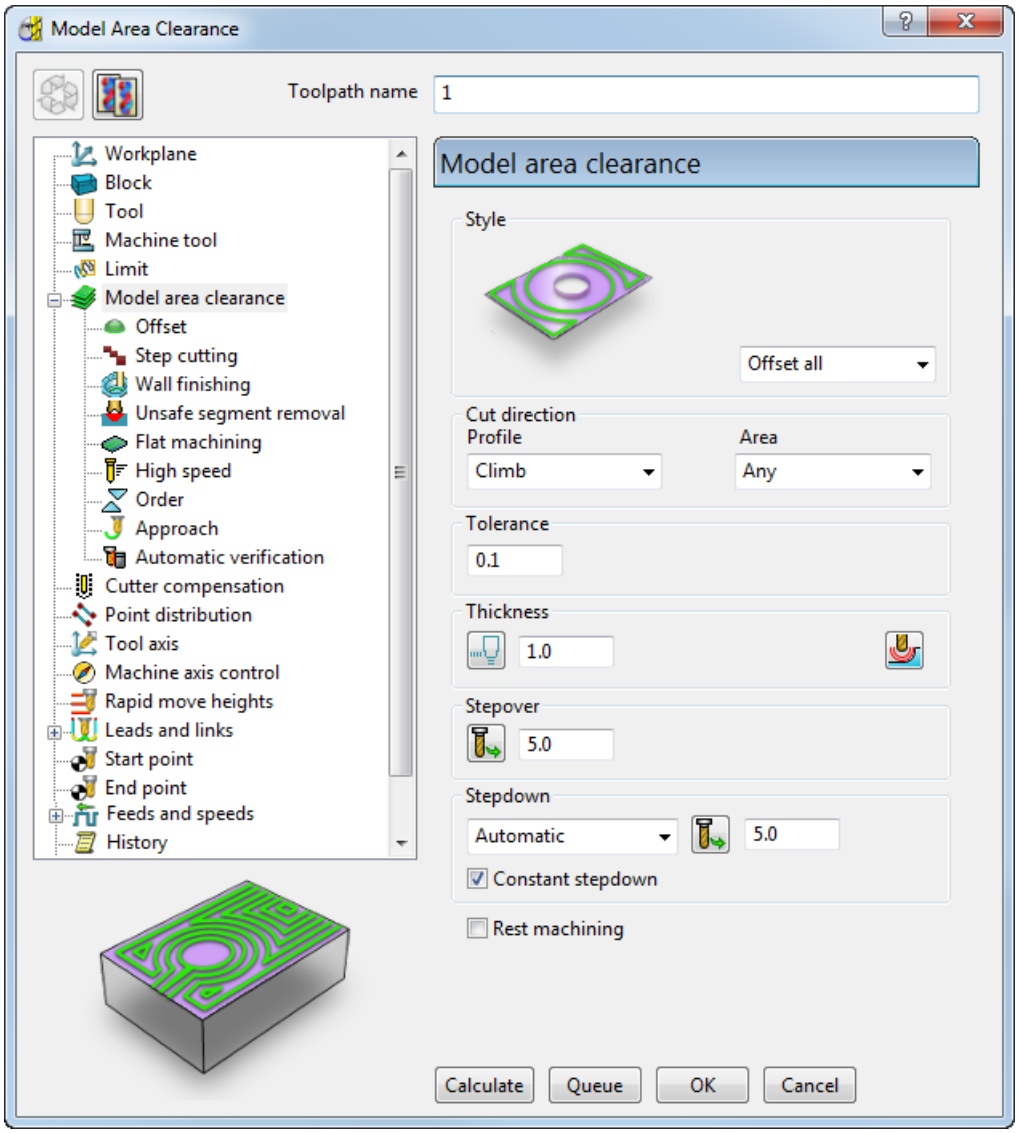
A **Model Area Clearance** strategy with the roughing **Style** set as **Offset all** is used to rapidly remove the majority of excess material on the model. This clears the area with contours that are generated by repeatedly offsetting the initial contour until no further offsetting is possible. It 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 for the toolpath.

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

3 Define the roughing tool geometry (see page 30).

4 Define the tolerances (see page 34).

5 Specify rapid move heights (see page 34).

6 Specify tool start points (see page 35).

7 Generate the roughing toolpath (see page 36).

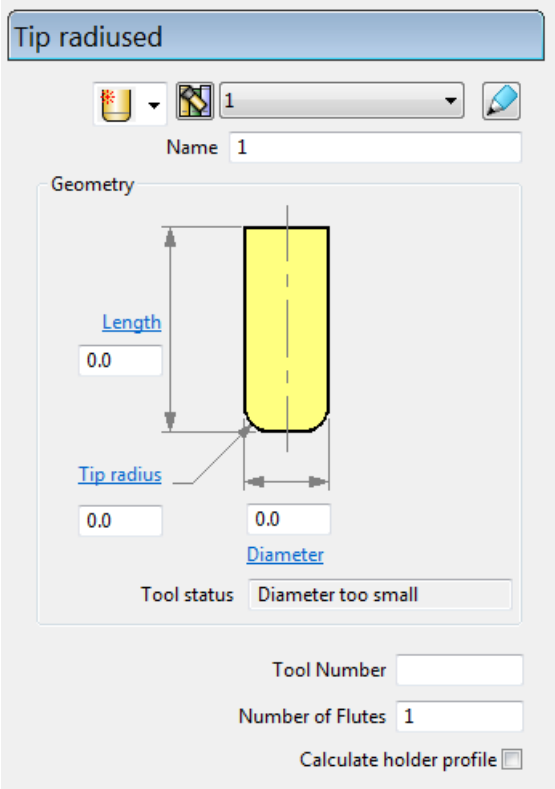
8 Display the roughing toolpath (see page 36).

9 Simulate the roughing toolpath (see page 40).

Defining the roughing tool geometry

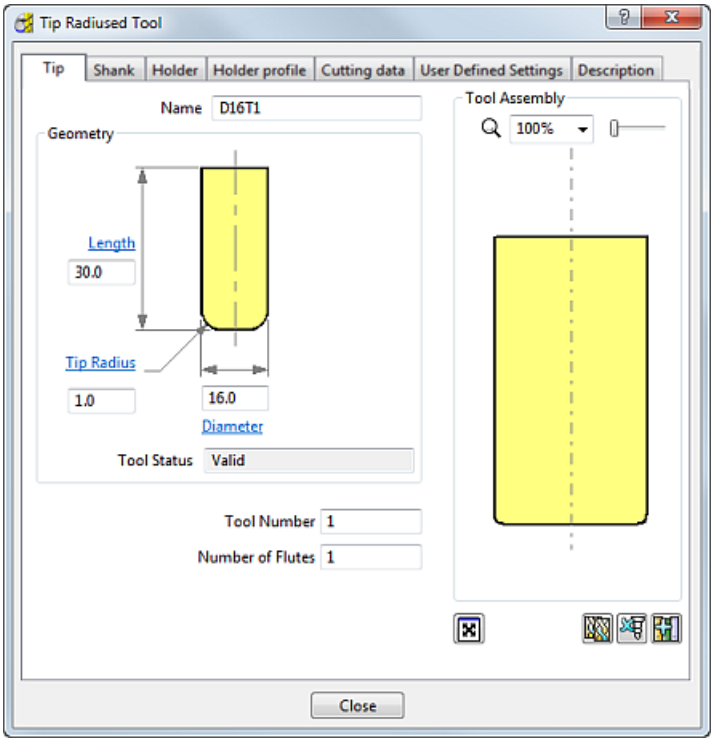
The next stage involves selecting a tool and defining its geometry. This example uses a **16** mm (**5/8** inch) tip-radiused tool.

1. In the **Model Area Clearance** strategy dialog, click Tool to display 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**

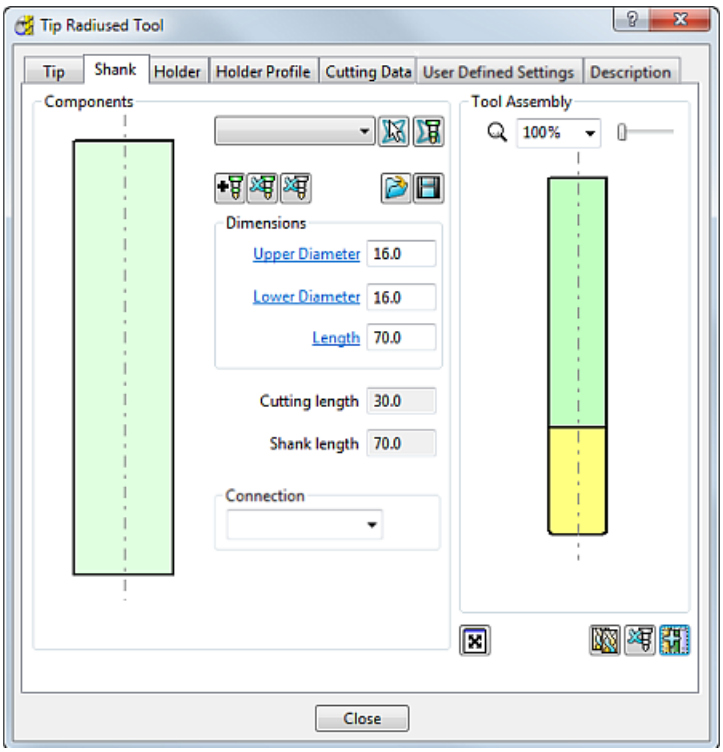


5 Select the **Shank** tab, click to add a shank component. Enter:

a **Upper Diameter : 16** mm

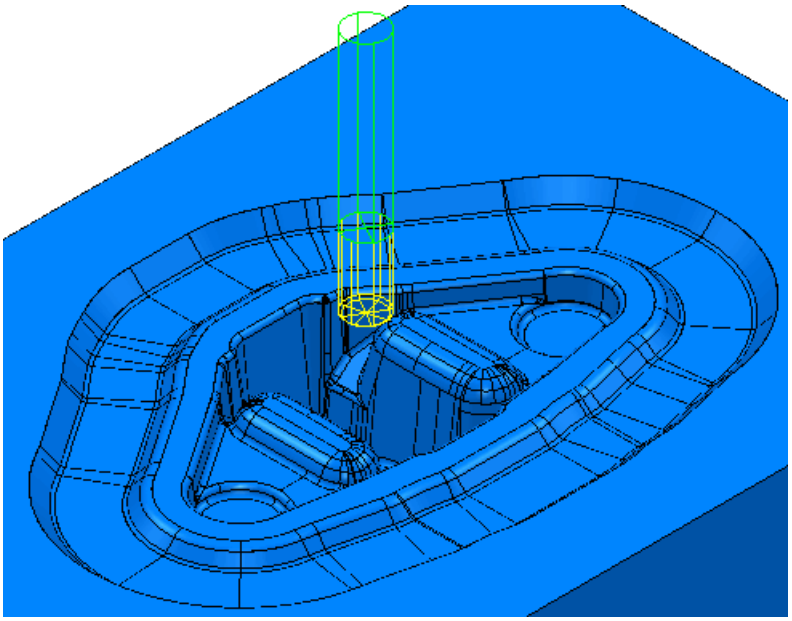
b **Length: 70** mm

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



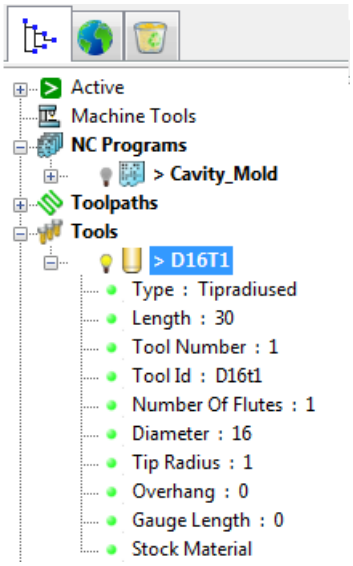
6 Click **Close**.

The tool is automatically aligned with the Z axis:

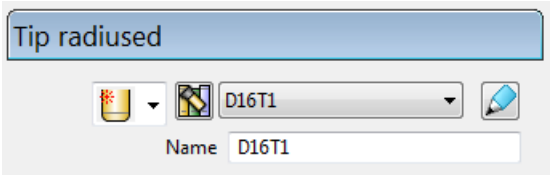


You can view and edit the created tool:

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



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

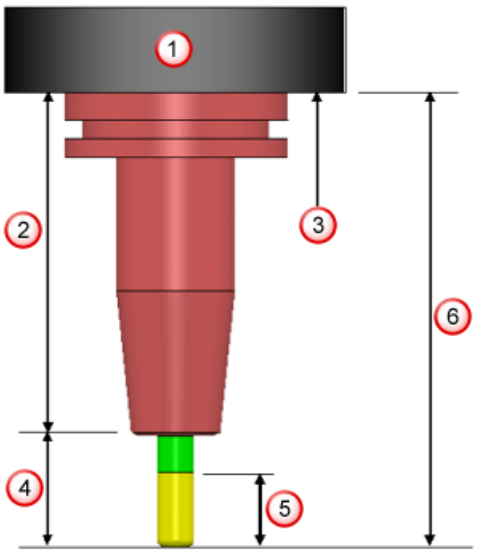


- On the **Tool** toolbar:

Tool length overview

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

*The colours used 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 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, keep the **Overhang** to the minimum length 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. Click on the **Thickness** button to enable the **Radial thickness** and **Axial thickness** fields.
3. Enter a **Radial thickness** of 0.5 mm.
4. Enter an **Axial thickness** of 0.1 mm.
5. In **Stepover**, enter: 7.0 mm.
6. In **Stepdown**, enter: 4.0 mm.
7. 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.

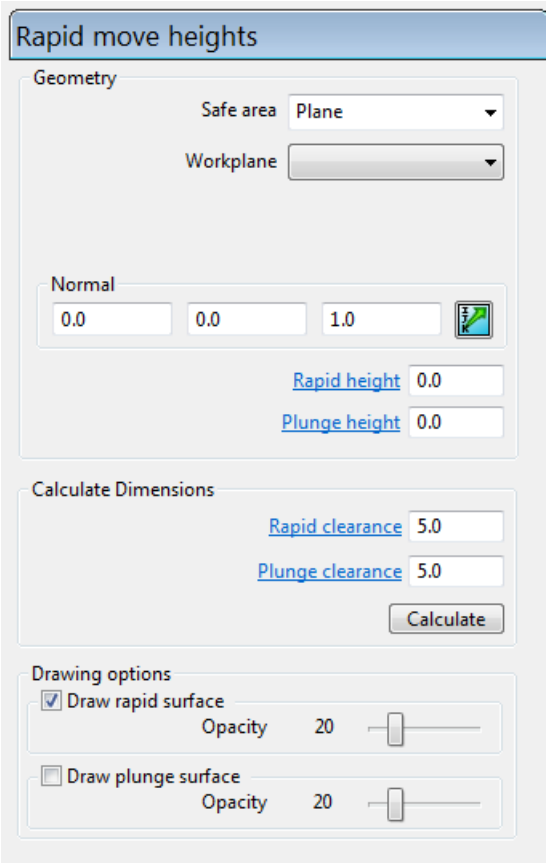
Use the **Rapid move heights** page on a strategy dialog to define the **Safe Z** and **Start Z** tool heights.

*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 Rapid move heights 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**.



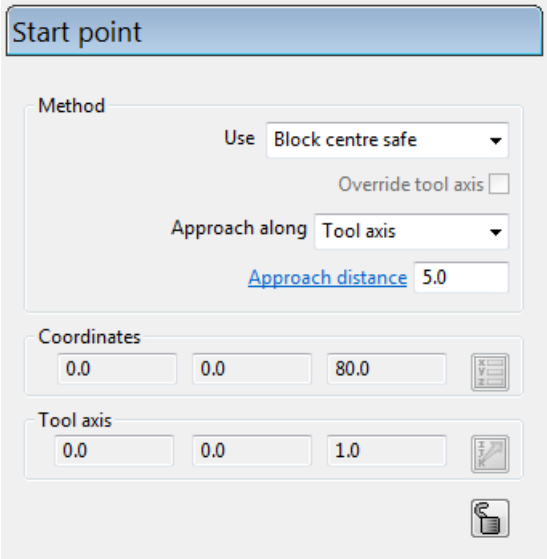
The use of absolute and incremental Z heights is controlled by the type of **Rapid Movement** selected.

Specifying tool start point

1 Select the **Start point** page Start point 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:

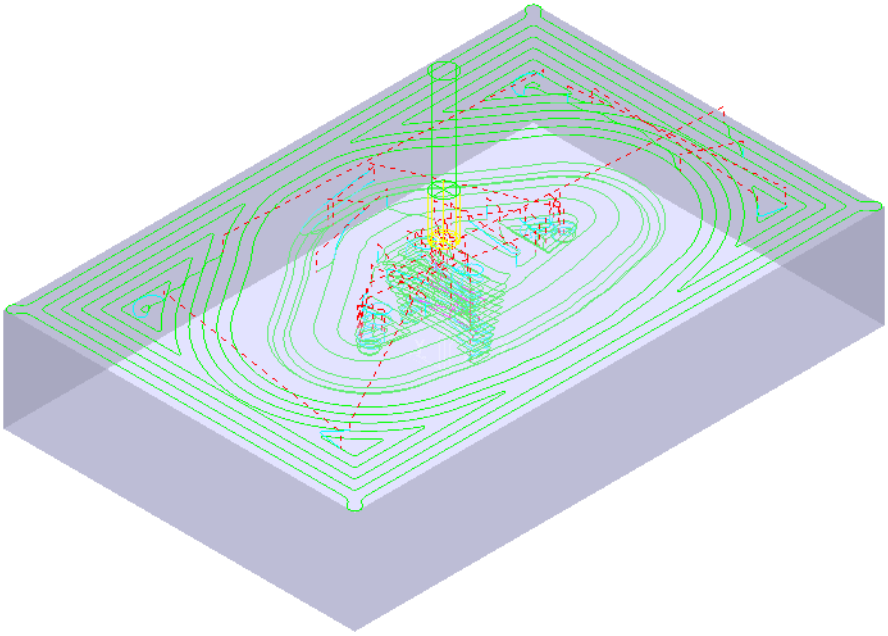
1. Select and expand the **Leads and links** page:
   a. Select the **Lead in** sub page.
   b. From the **1st choice** list, select **Ramp**.
2. Select the **High speed** page:
   a. Select the **Profile smoothing** option. Leave the default **Corner Radius** as **0.05**.
   b. Select **Raceline smoothing**, 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 the visual representation of the toolpath, you can:

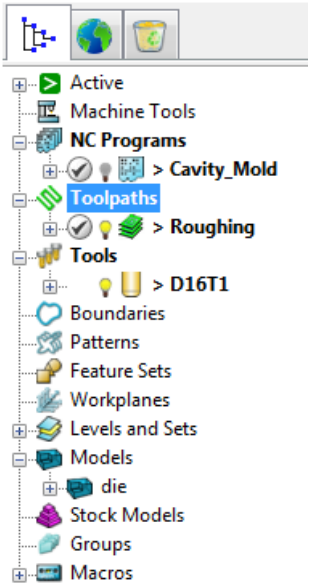
- Click the **Plain Shade** and **Wireframe** buttons to draw and undraw the model or wireframe.



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

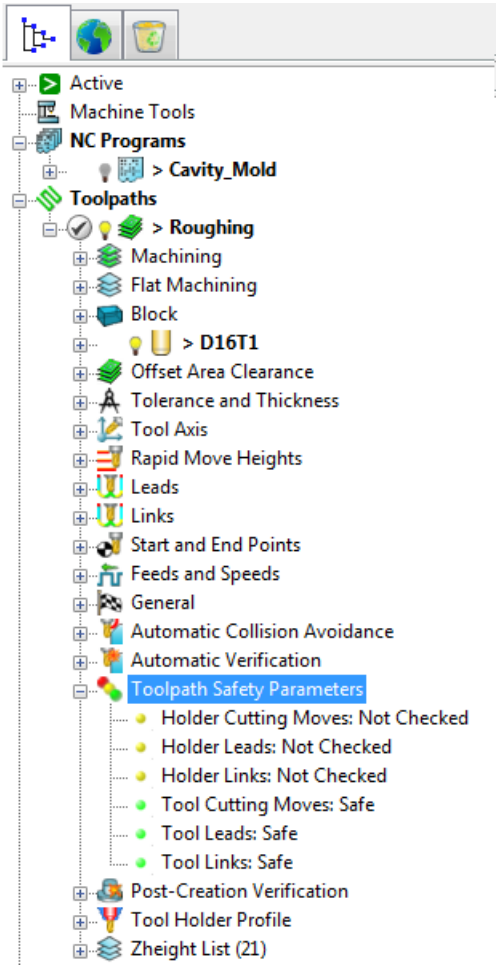
- Click the **Block** button on the **View** toolbar to undraw the block.

- Click ⊞ to expand the **Toolpaths** branch 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 .

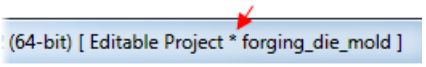
- Click ⊞ (next to the toolpath) to expand and view the parameters and their specific details used to create the toolpath.



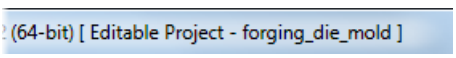
*The **Safety Status** icon ✓ at the top of the toolpath tree shows the toolpath is gouge-checked, but not checked for holder details. To see more information, expand the **Toolpath Safety Parameters** node in the toolpath tree.*

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:

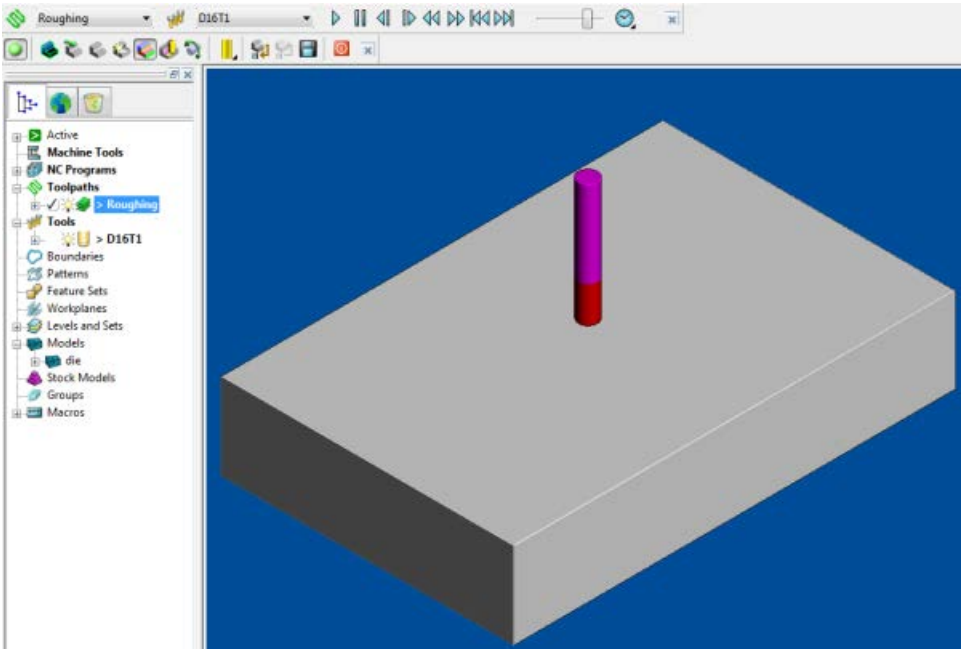


Simulating the roughing toolpath

To view a simulation of the toolpath:

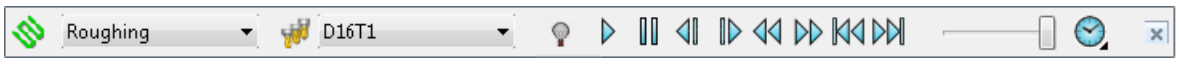
1 Click the **ISO1** button on the **View** toolbar to reset the view.

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.



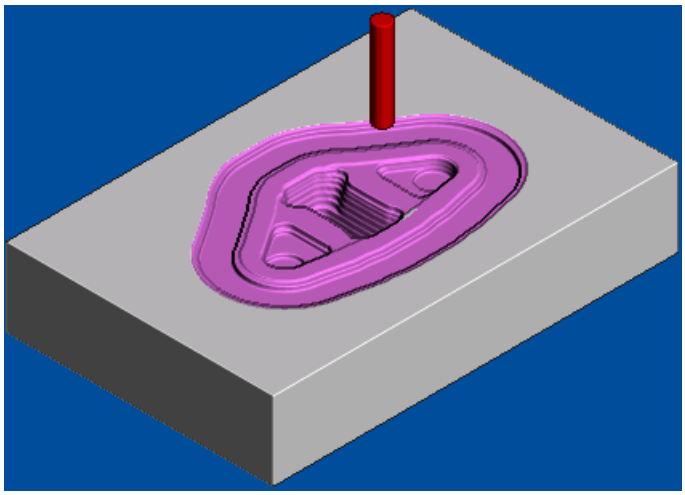
*If the **Simulation** toolbar is not already displayed, select **View > Toolbar > Simulation**.*

3 Select the current toolpath in the first of the two drop-down lists. The associated 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 for the best visual representation of material being 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.

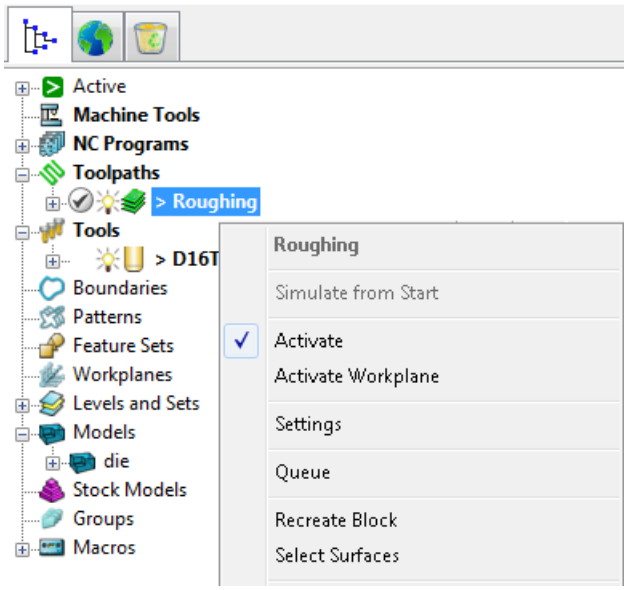


Creating the rest roughing toolpath

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 could not reach, such as pockets and corners.

To create a rest roughing toolpath:

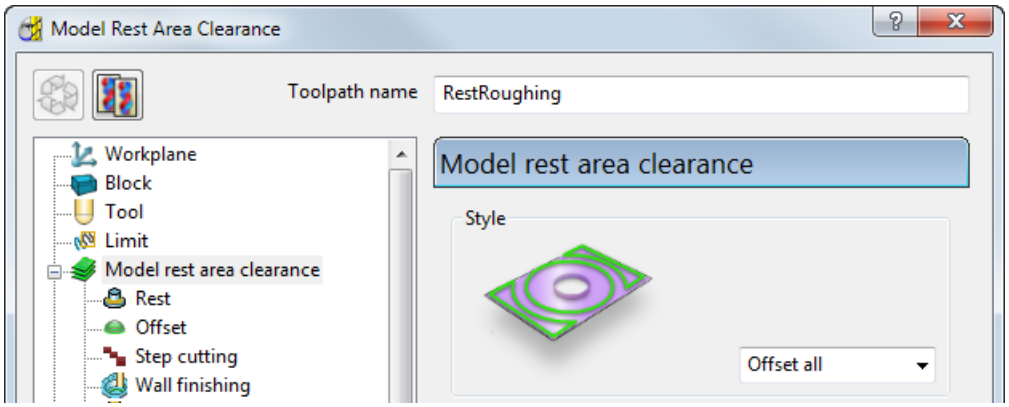
1 Display the **Settings** for the previous toolpath.



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

3 A copy of the toolpath is created with the suffix '_**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 42).

6 Change the **Stepover** and **Stepdown** values (see page 44).

7 Complete the **Model rest area clearance** strategy dialog, and generate the rest roughing toolpath (see page 44).

8 Display the rest roughing toolpath (see page 45).

9 Simulate the rest roughing toolpath (see page 46).

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**.

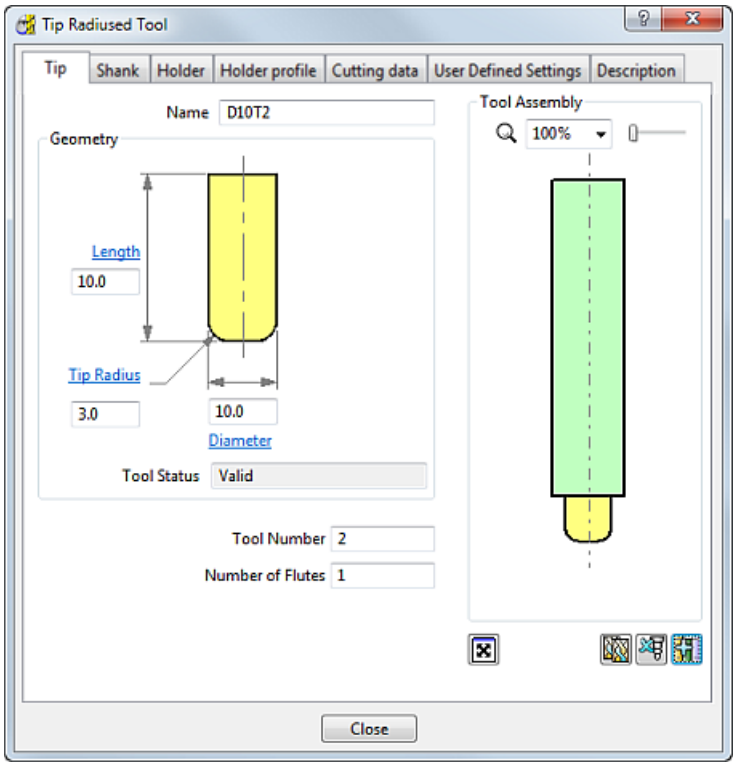
4 In the other fields, enter:

   a **Length:** 10 mm

   b **Tip Radius:** 3 mm

   c **Diameter:** 10 mm

d **Tool Number: 2**

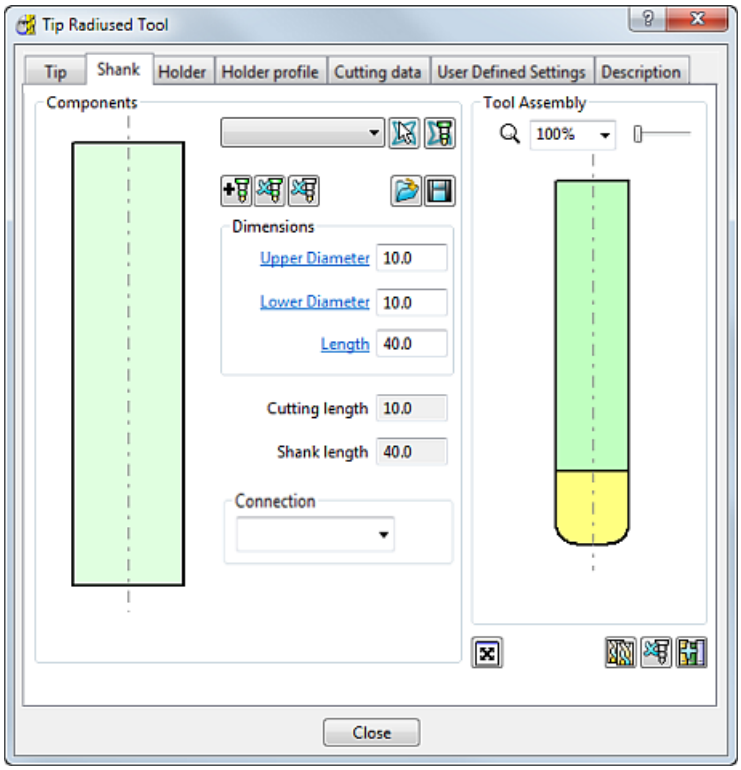


5 Select the **Shank** tab and change the values to:

a **Upper Diameter: 10**

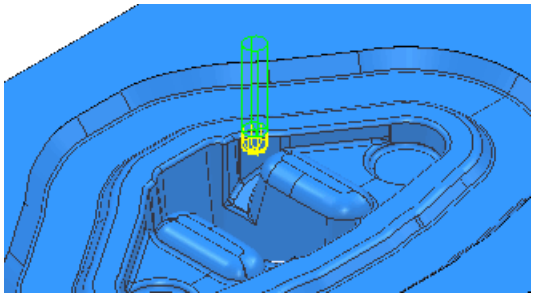
b **Length: 40**

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 does not delete) the entities from the project.



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.

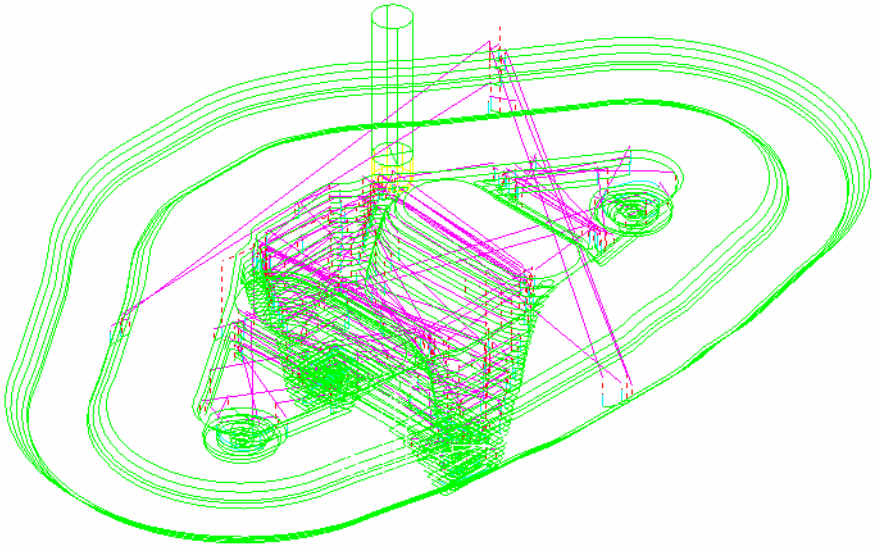
2 Click **Calculate** to generate the toolpath.

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.

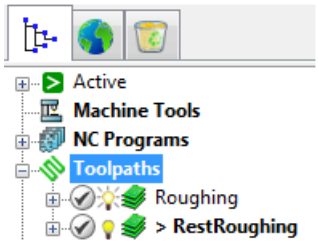
3 After the toolpath is generated, **Close** the strategy dialog.

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** branch 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.*

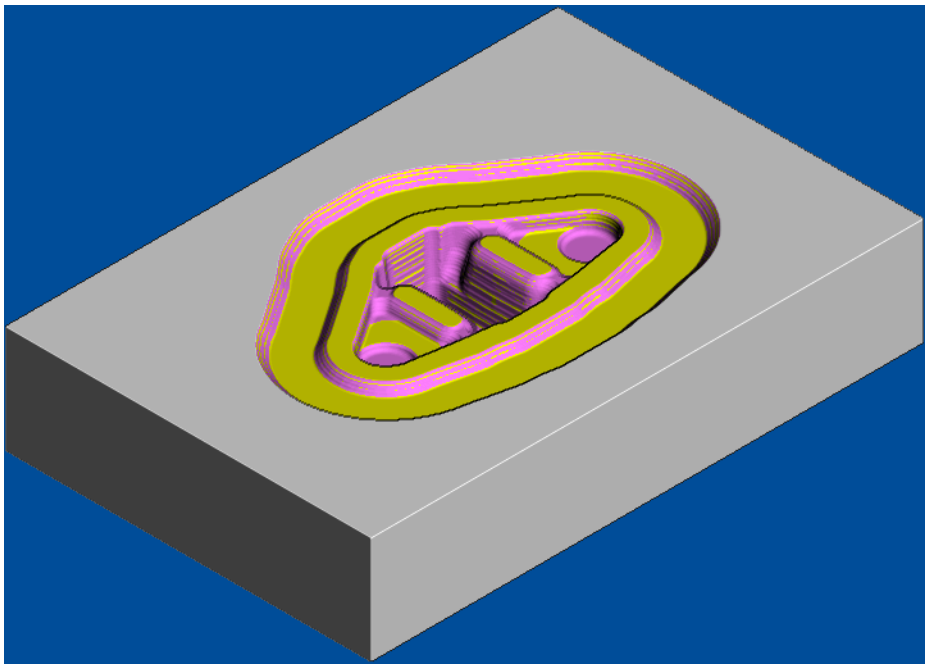
To save project changes, click on the **Main** toolbar.

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 another session since creating the roughing toolpath, the rest roughing toolpath is shown in a different colour, overlaid on the roughing simulation (see page 40).

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 graphics window is displayed.

Closing the roughing session

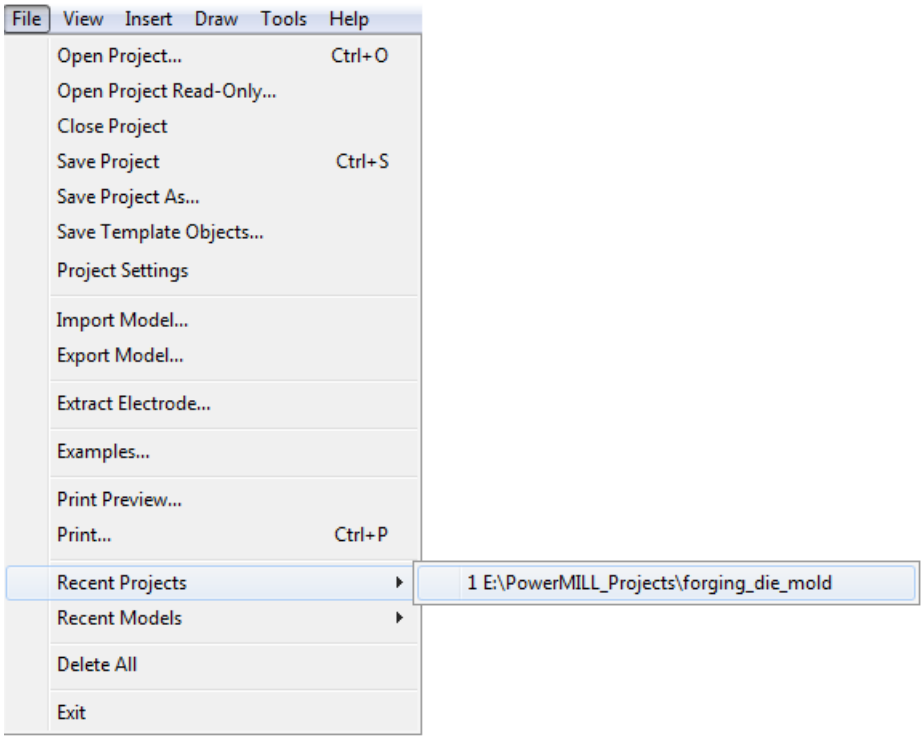
Now you have generated the roughing toolpath. Save the project and close PowerMILL until you are ready to create the finishing strategies.

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

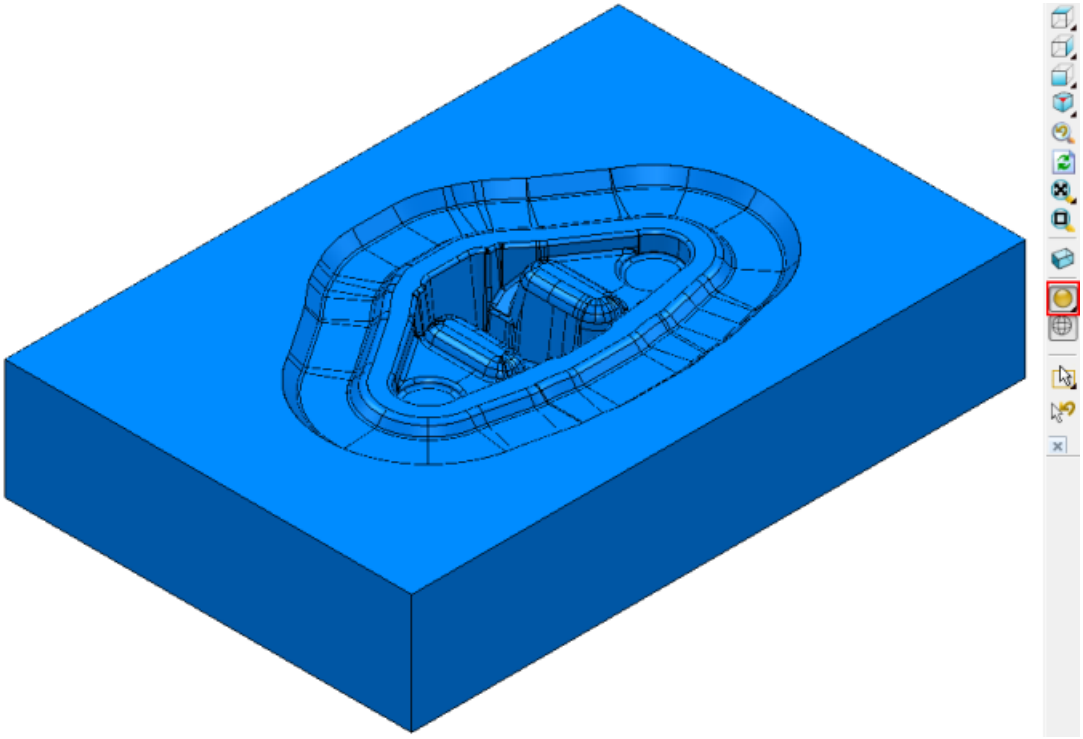
To exit PowerMILL, either select **File > Exit** from the menu, or click the **Close** button in the top right-hand corner of the PowerMILL window.

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.

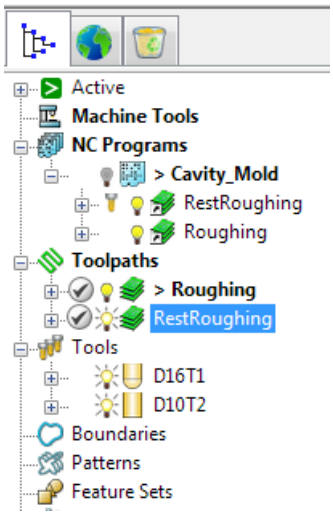


Checking which entities are active

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

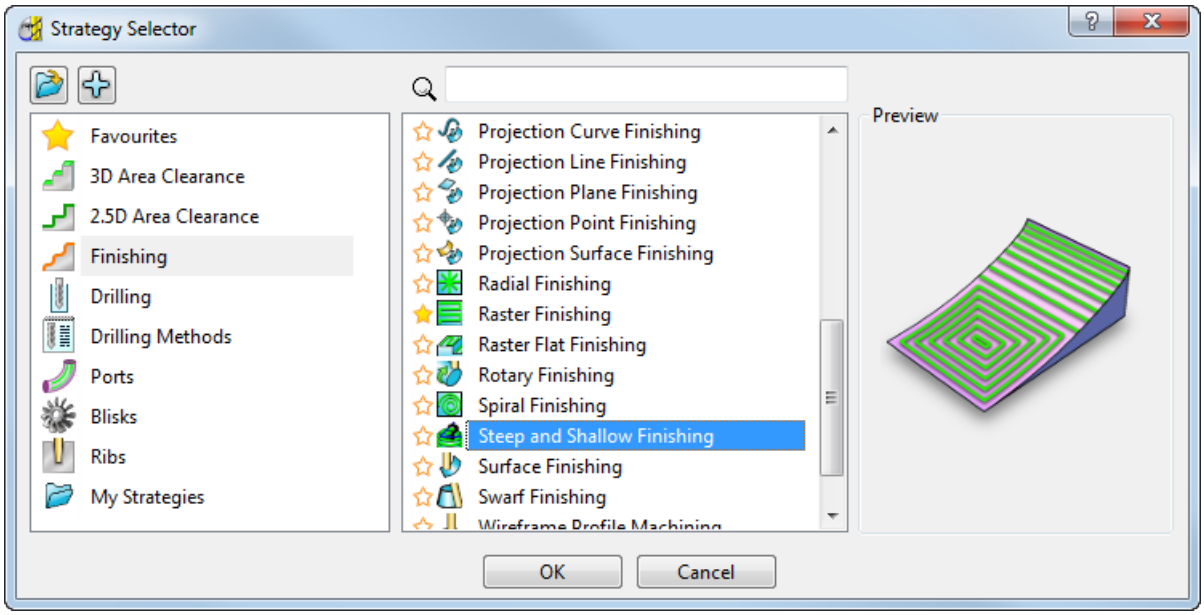
1 Check if the NC program in the Explorer is shown in **bold** and preceded by the **>** symbol to indicate it's active. If it's not active, right-click the NC program name, and select the **Activate** option.

2 Expand the **Toolpaths** node and click the light bulb icon (until it shows). This undraws each of the roughing toolpaths. Now you can easily see the new finishing toolpaths as you create them.



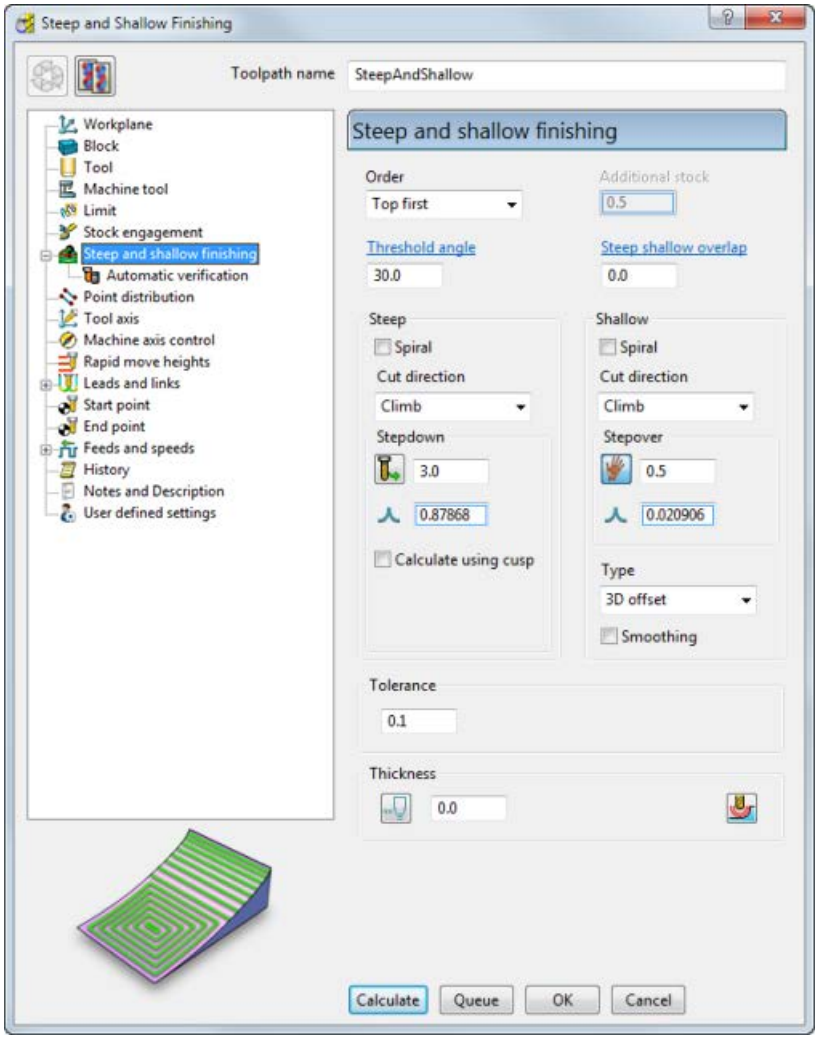
Creating a finishing toolpath

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.

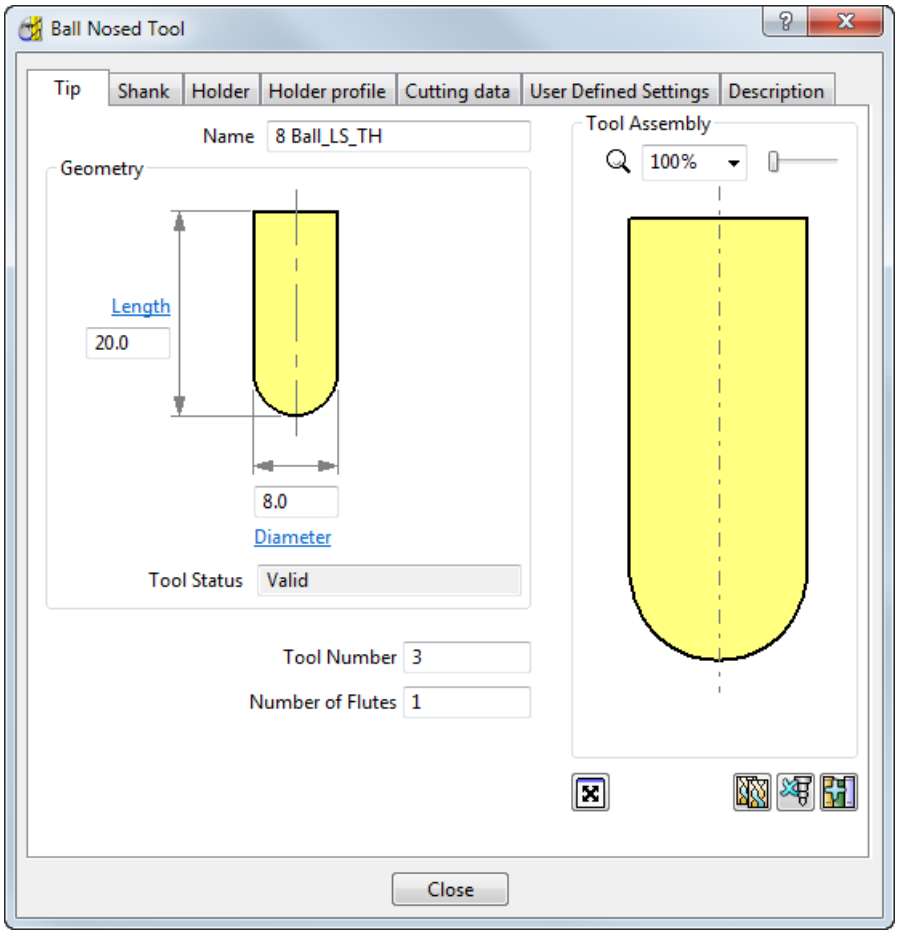
4 In the **Ball Nosed Tool** dialog, enter:

a **Name: 8 Ball_LS_TH**

b **Length: 20** mm

c **Diameter:** 8 mm

d **Tool Number:** 3

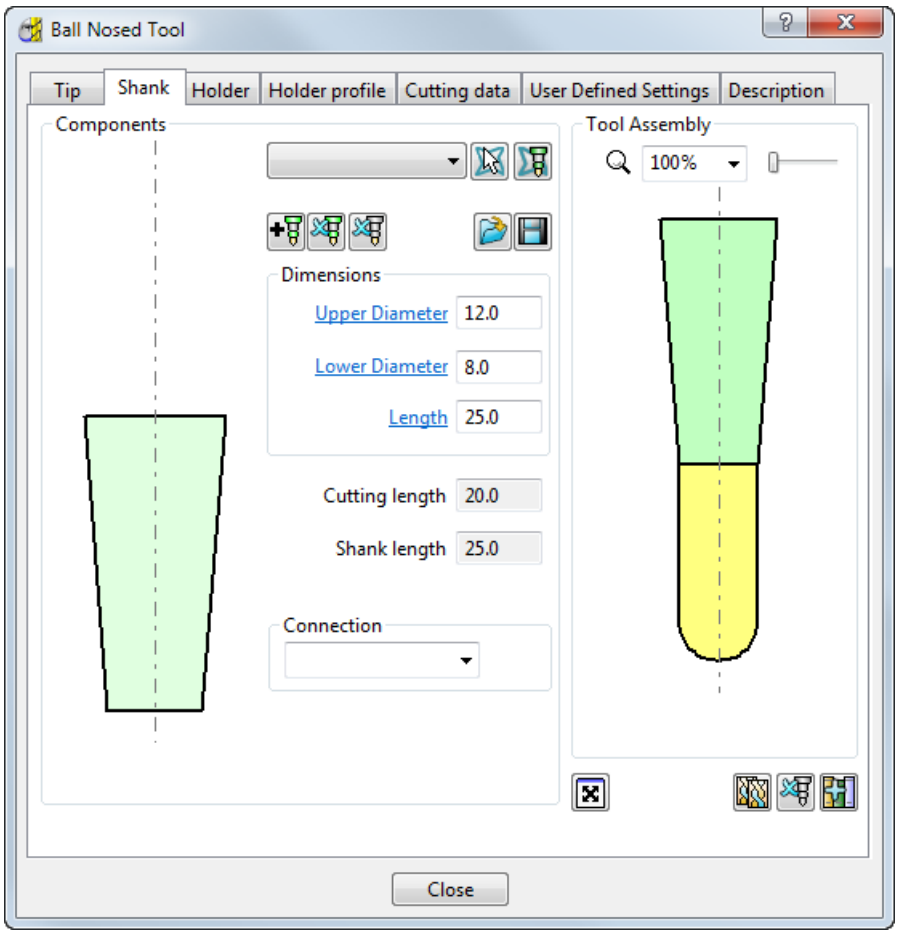


5 Select the **Shank** tab, click to add a shank component. Enter:

a **Upper Diameter:** 12

b **Lower Diameter:** 8

c **Length:** 25

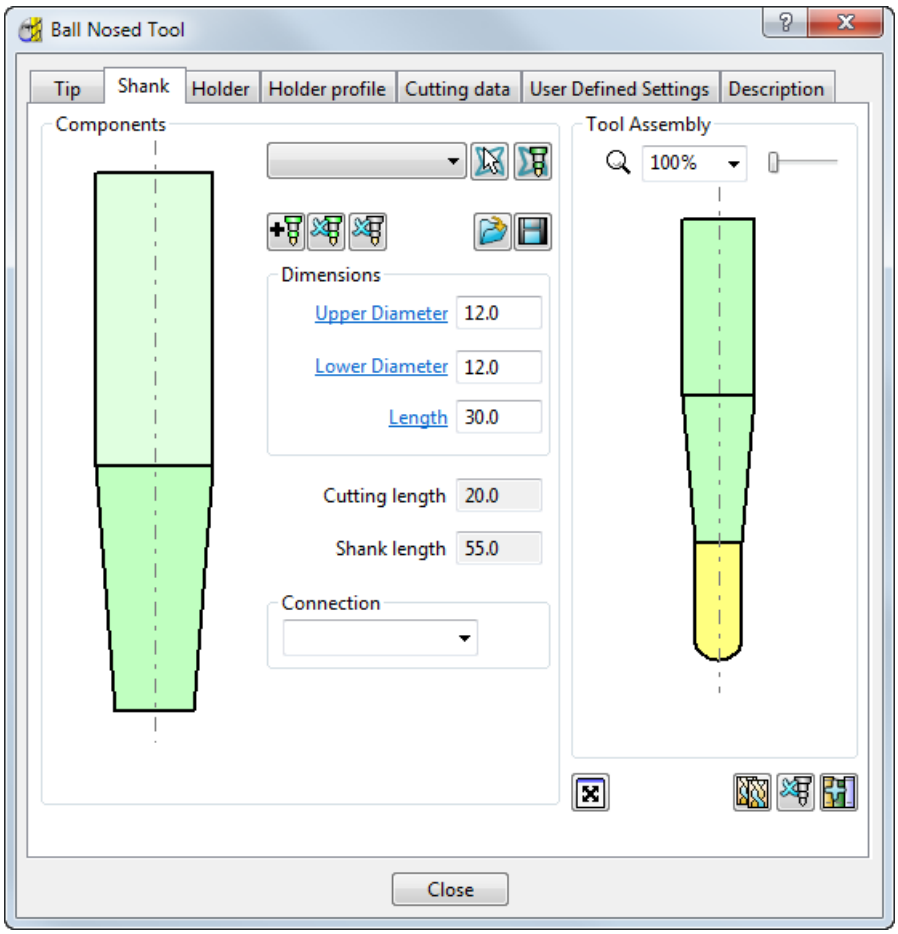


6 Click the button to add a second shank component. Enter:

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:

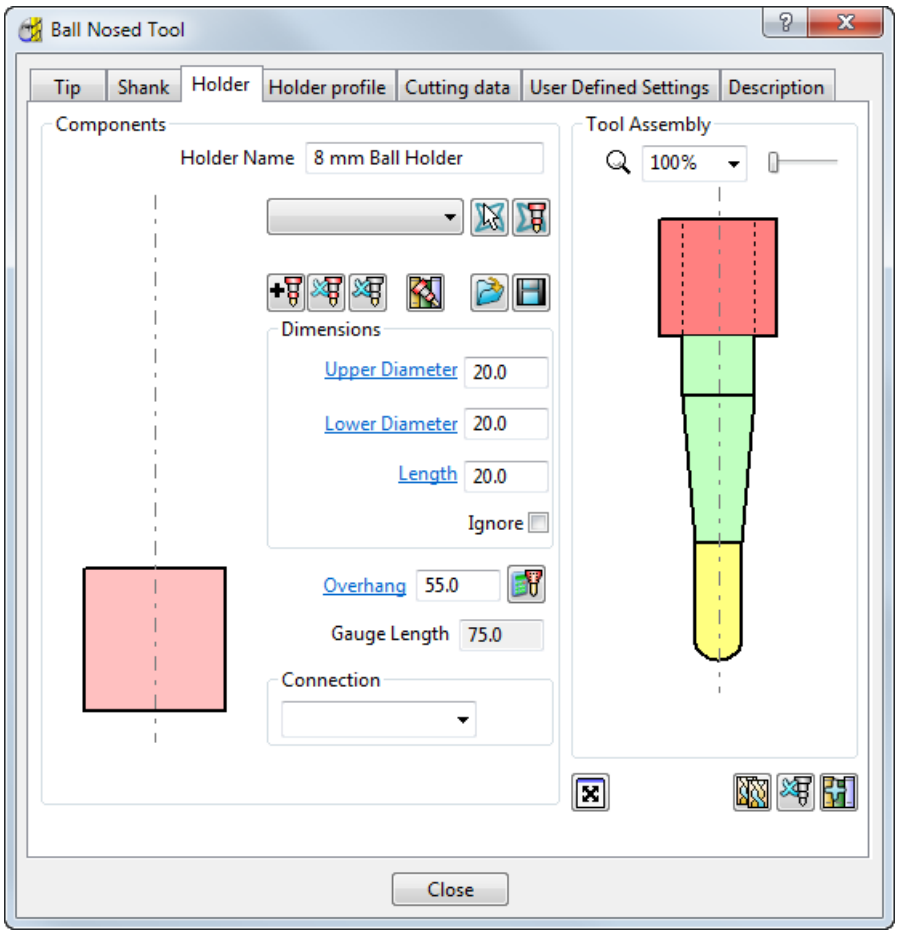
a **Name:** 8 mm Ball Holder

b **Upper Diameter:** 20

c **Lower Diameter:** 20

d **Length:** 20

e **Overhang:** 55



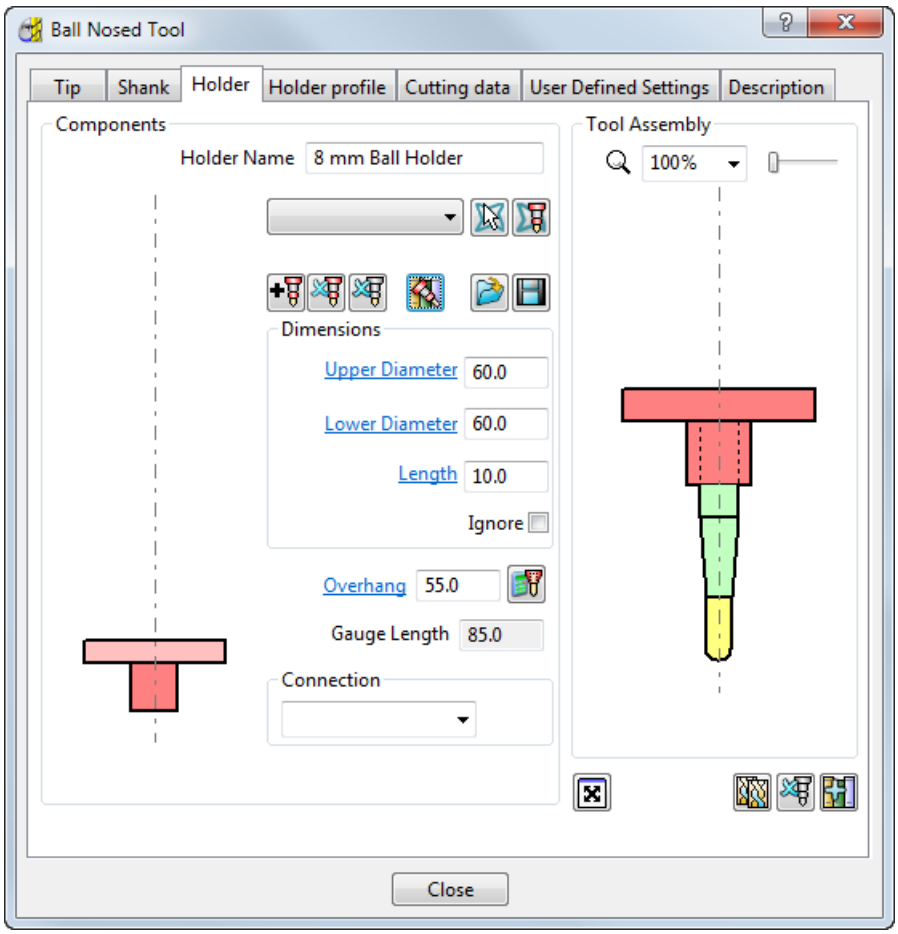
8 Click the button to add the upper part of the holder. Enter:

a **Upper Diameter:** 60

b **Lower Diameter:** 60

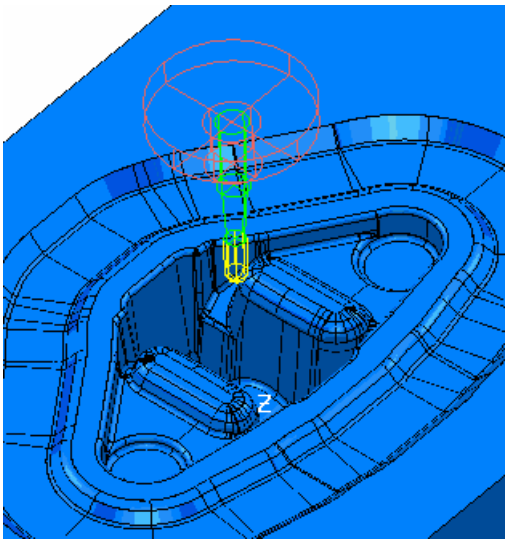
c **Length:** 10

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.*



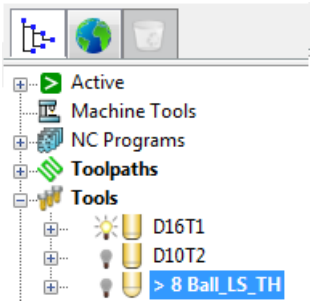
The toolpath is not yet complete as it needs to reference a boundary, but you can close the dialog without generating the toolpath.

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

Creating the boundary for the selected surface

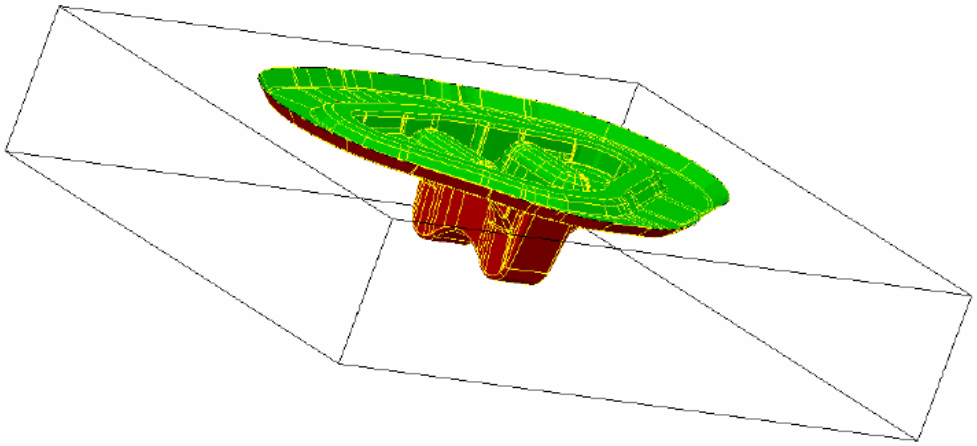
As it is only the cavity that needs to be finished by this toolpath, you can create a boundary for the cavity.

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



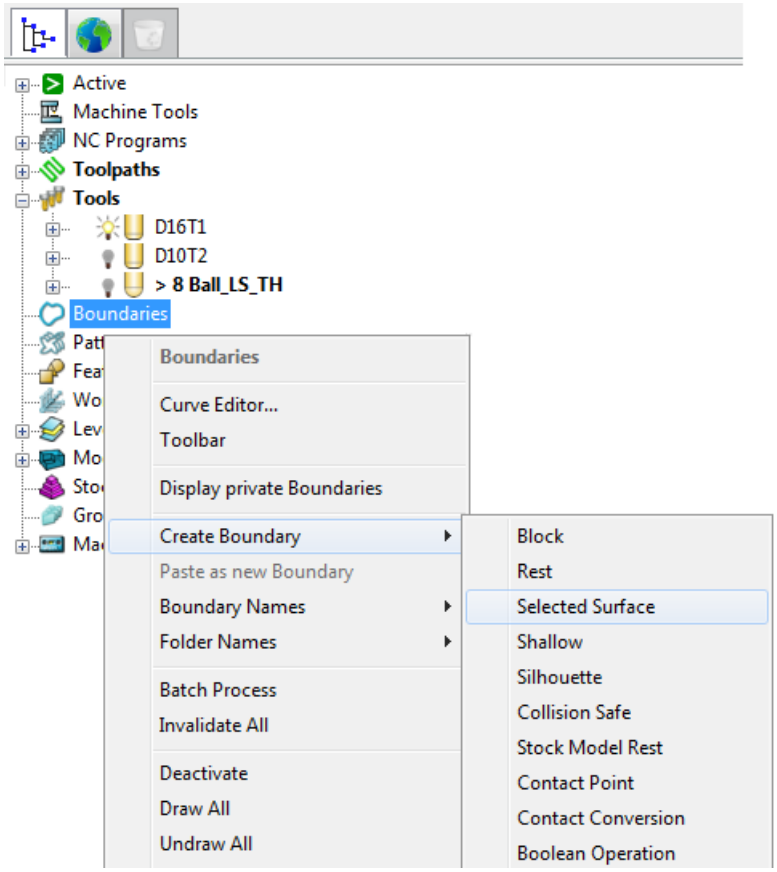
2 Click the **Block** button to undraw the block, and use the **Wireframe** button to undraw the wireframe.

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.*

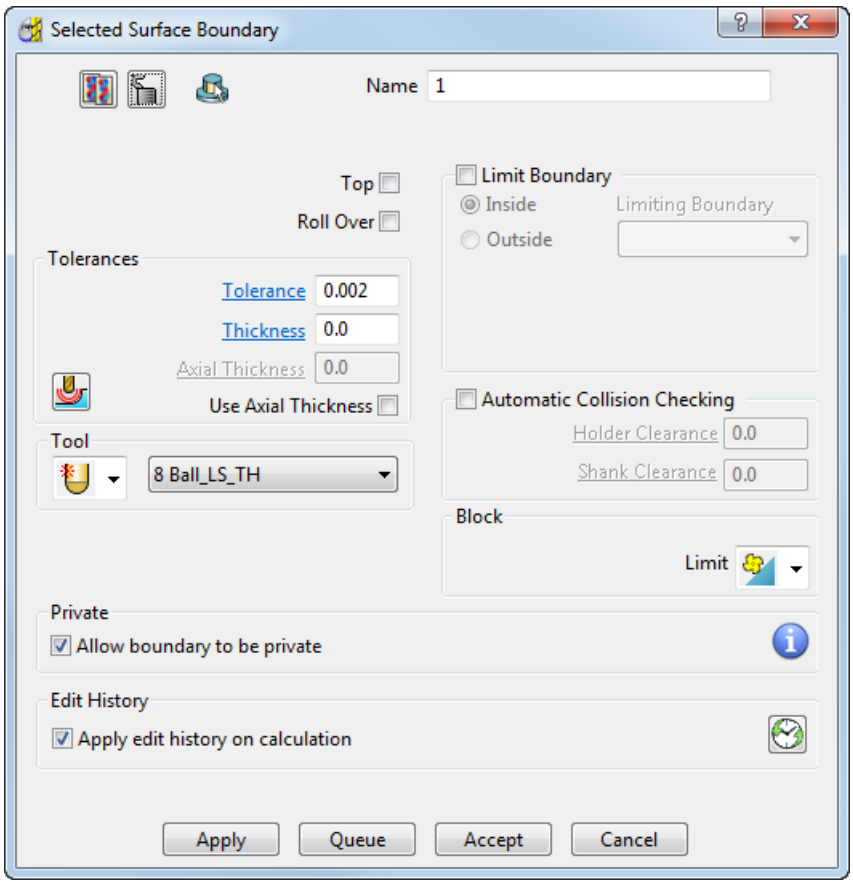
4 In the **Boundaries** context menu, select **Create Boundary > Selected Surface**.



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

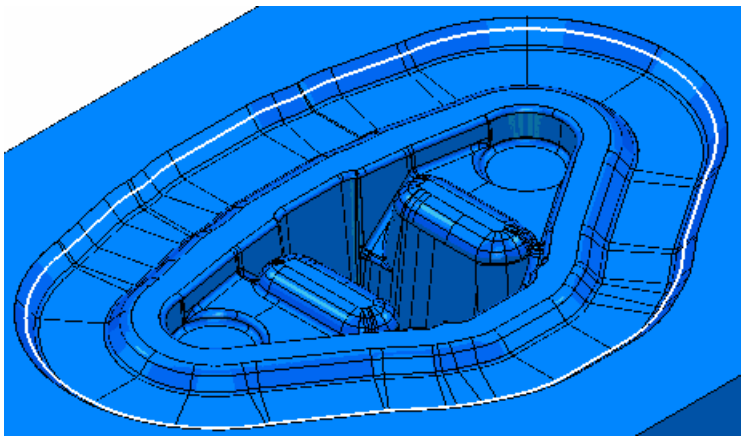
a In the **Name** field, enter **Cavity**.

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



6 Click **Apply**.

7 The boundary is calculated. By default, it's displayed in white.



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.*

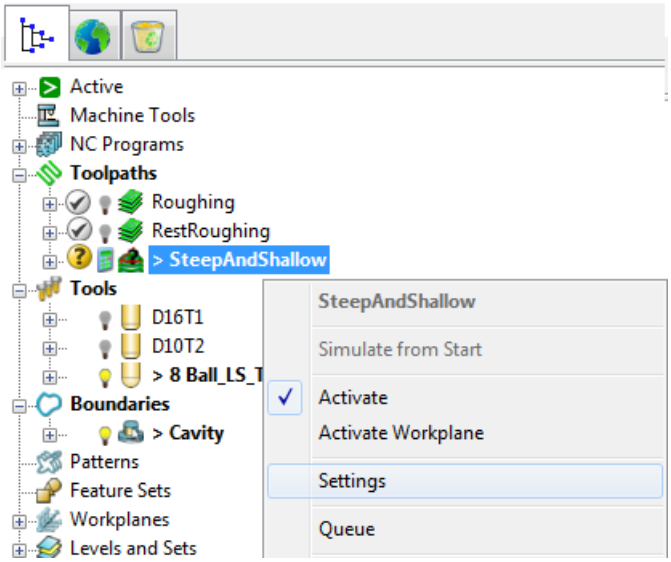
8 Click **Accept** to close the boundary dialog.

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 the Explorer, expand **Toolpaths** and select the **SteepAndShallow** toolpath.

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

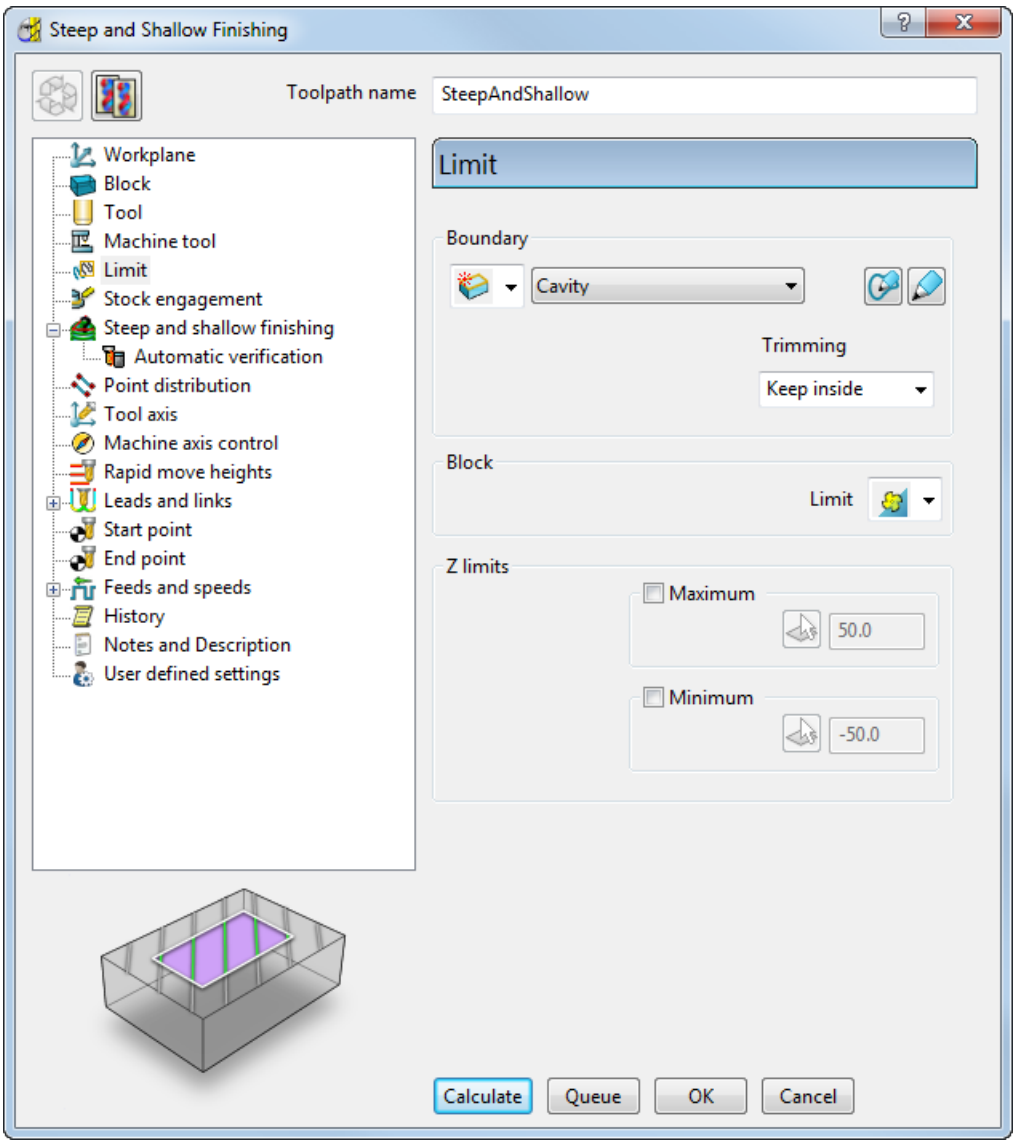


3 On the **Steep and Shallow Finishing** strategy dialog:

a Select the **Limit** page and:

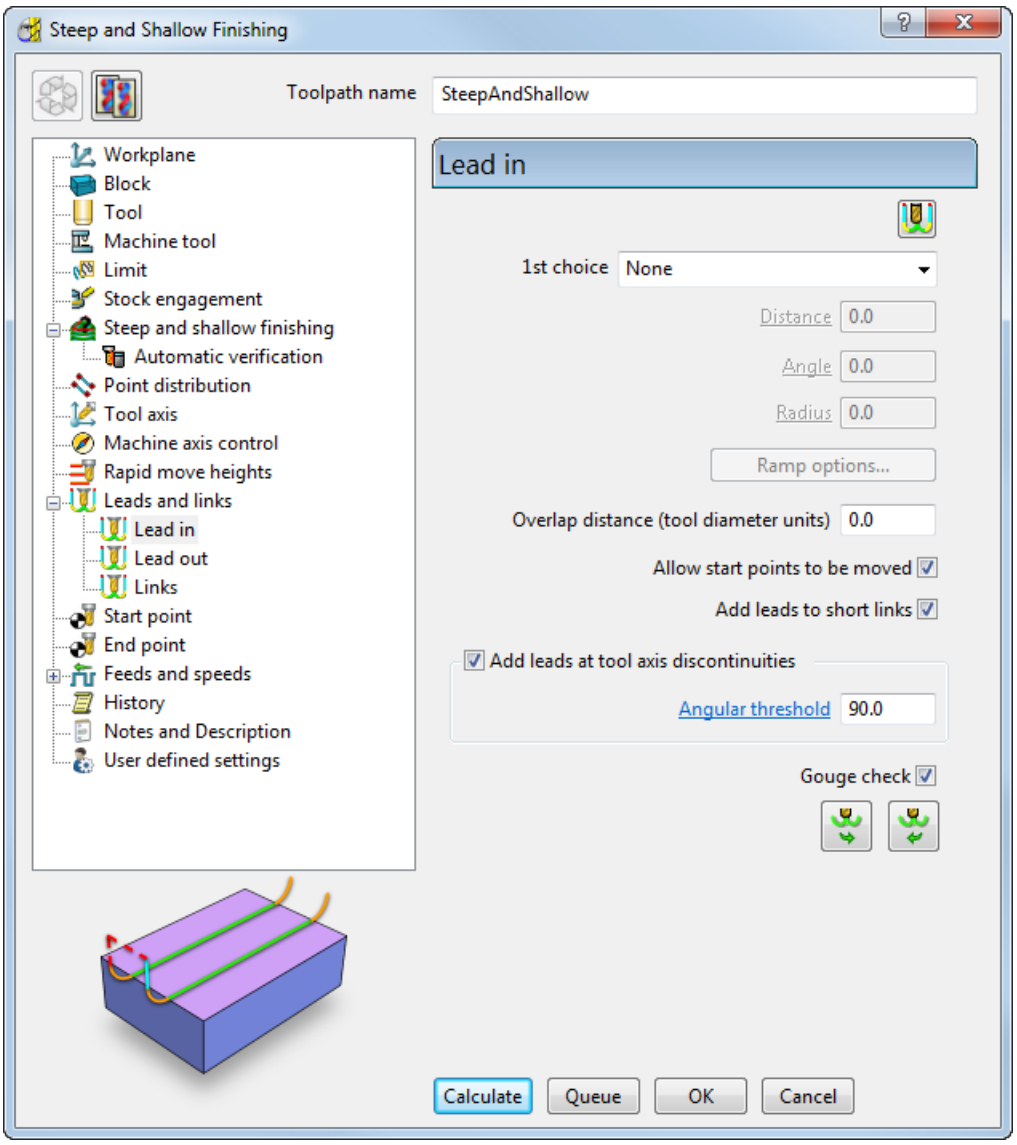
From the **Boundary** list, select **Cavity**.

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



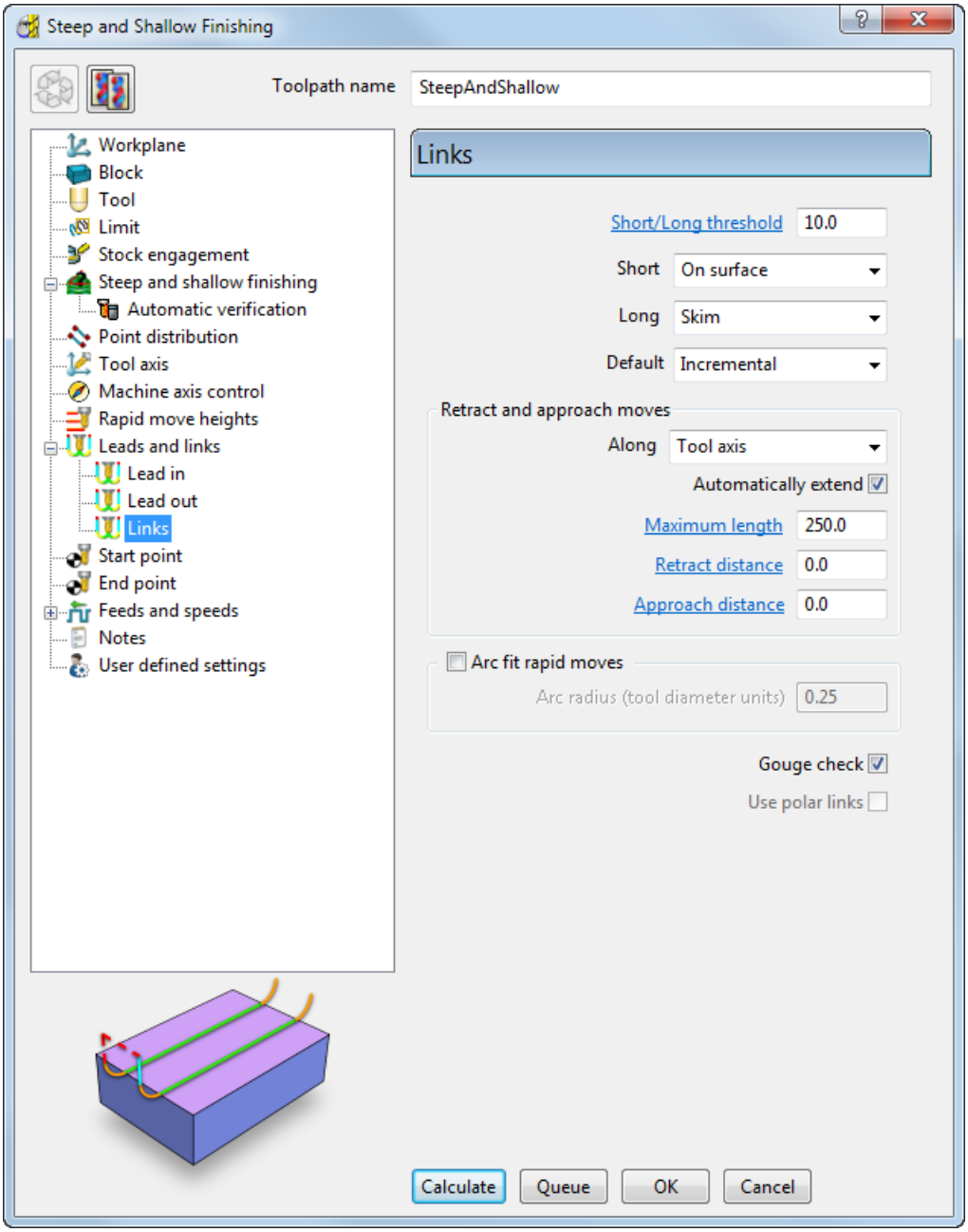
b Expand the **Leads and Links** page, select the **Lead in** page and:

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.



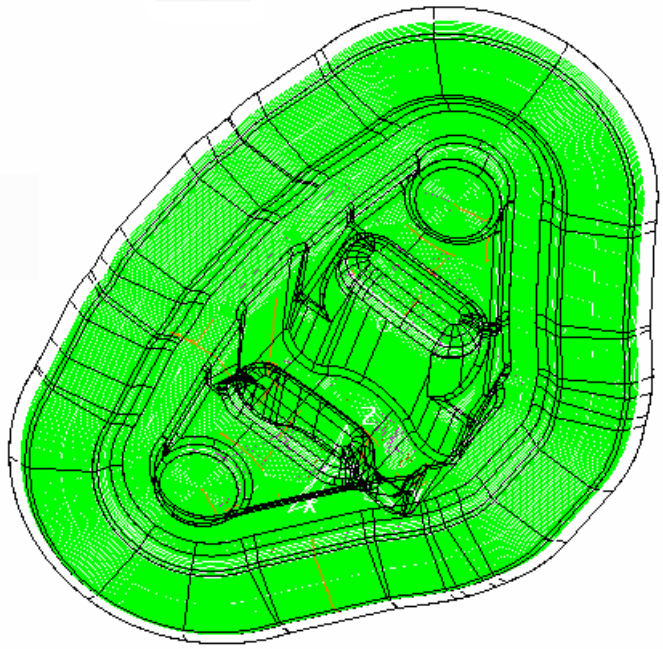
4 Click **Calculate** to generate the toolpath.

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.

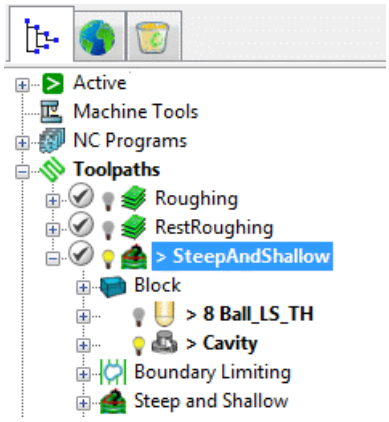
5 After the toolpath is generated, **Close** the strategy dialog.

Displaying the finishing toolpath

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



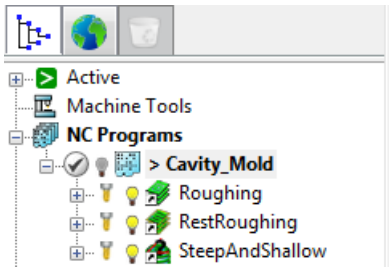
Click ⊞ to expand the **Toolpaths** branch in the Explorer. The new toolpath is shown in **bold** and preceded by the **>** symbol to indicate it's active.



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

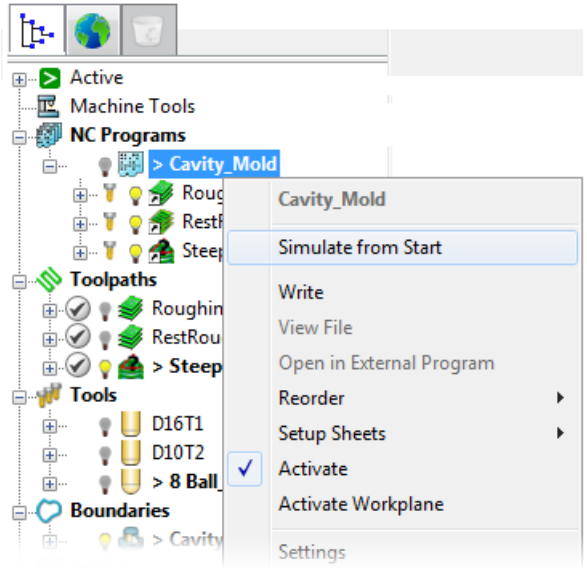
Simulating the NC program with generated toolpaths

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.



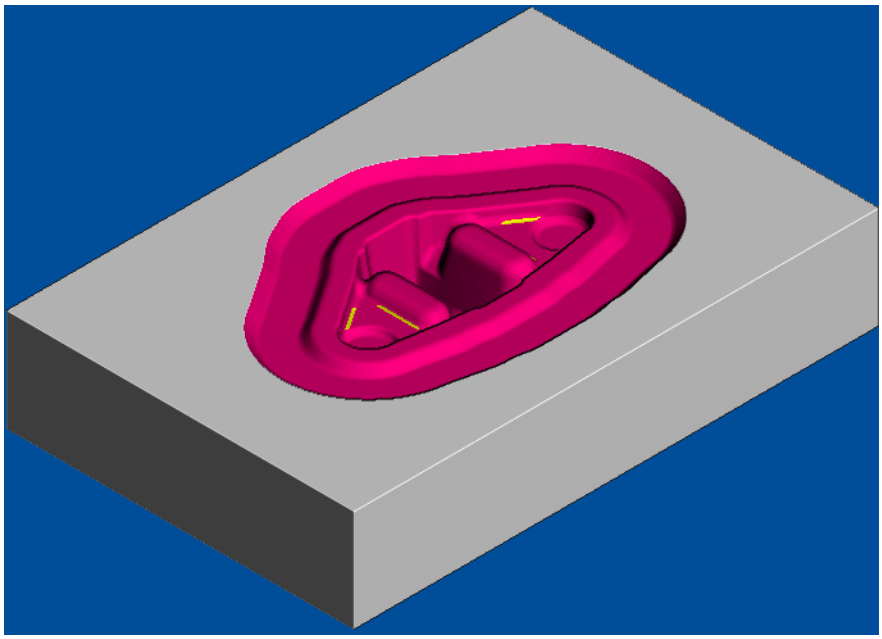
To simulate the NC program:

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 the differences between toolpaths are seen more easily.
4. In the Explorer, right-click the NC program **Cavity_Mold**, and select **Simulate from Start** from the context menu.

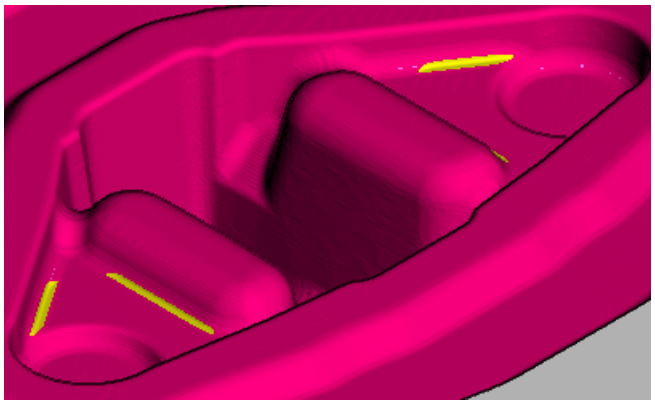


The NC program is automatically selected on the **Simulation Toolbar**, and the **Play** buttons enabled.

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.



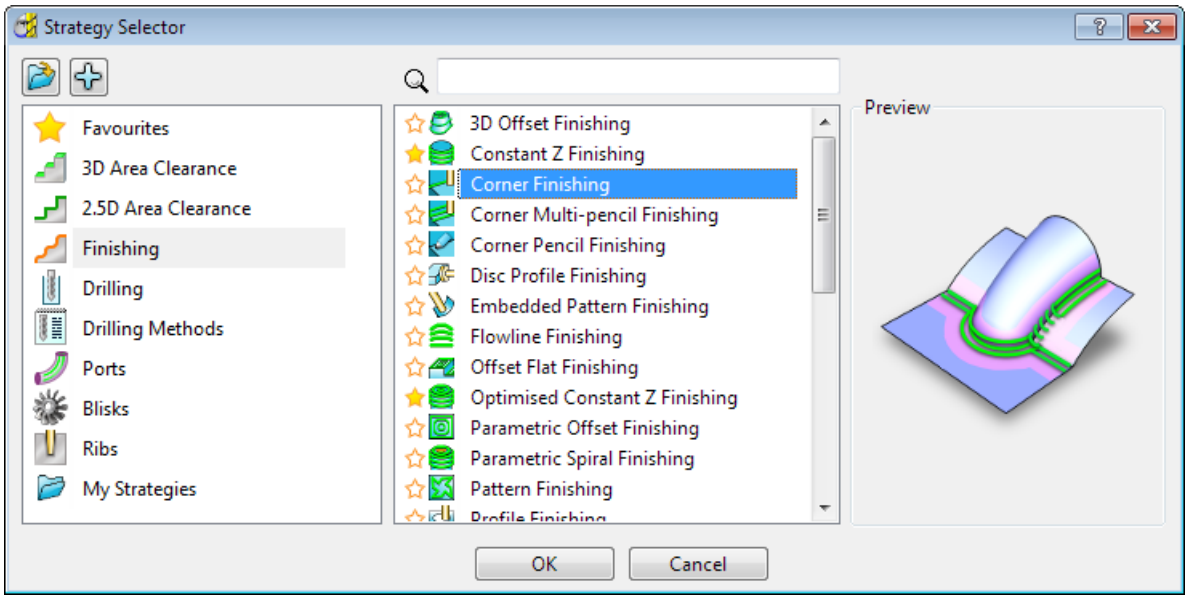
7 Click the **Exit ViewMill** button, and select **Yes** to stop the simulation. The **Toggle ViewMill** button changes from green to red, and the standard PowerMILL graphics window is displayed.

Creating the Corner Finishing toolpath

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

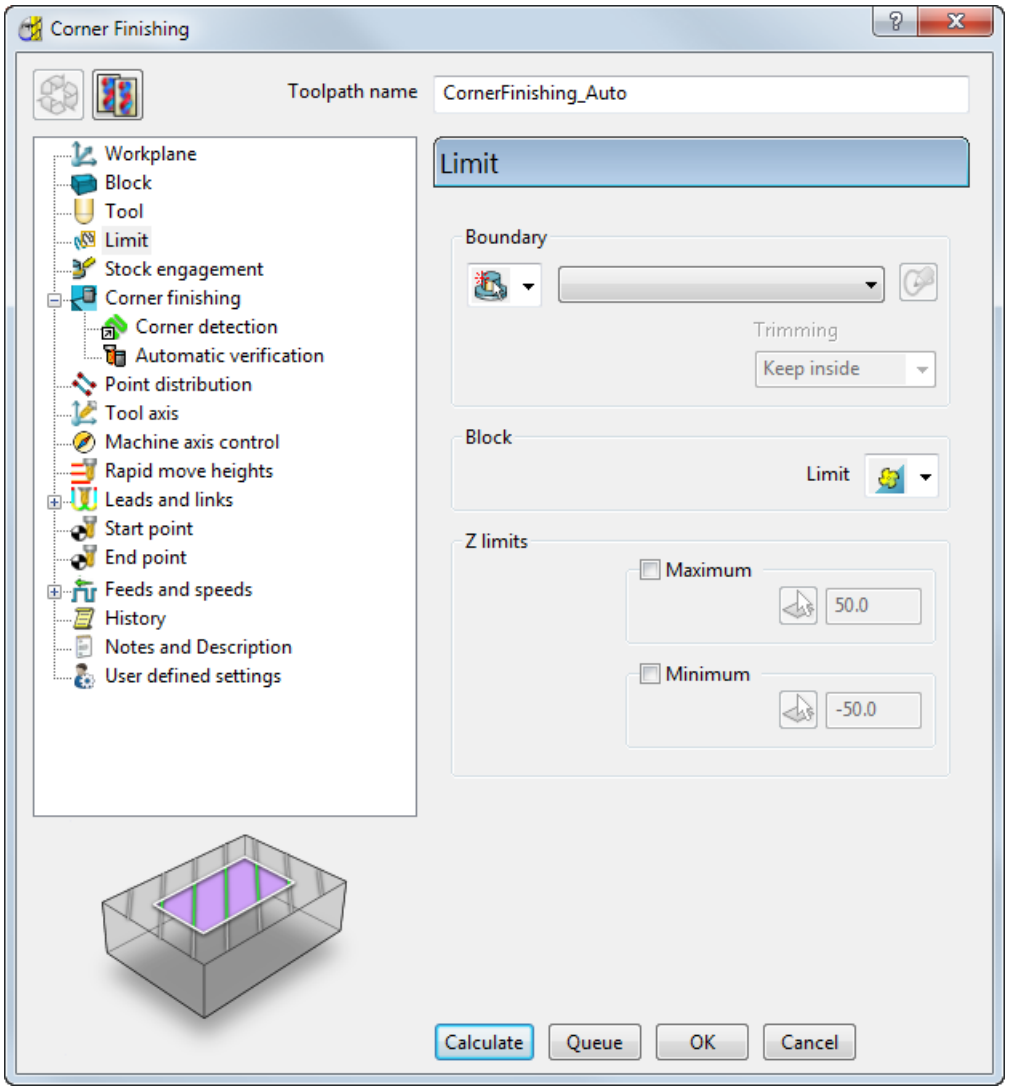
To create the Corner Finishing toolpath:

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.

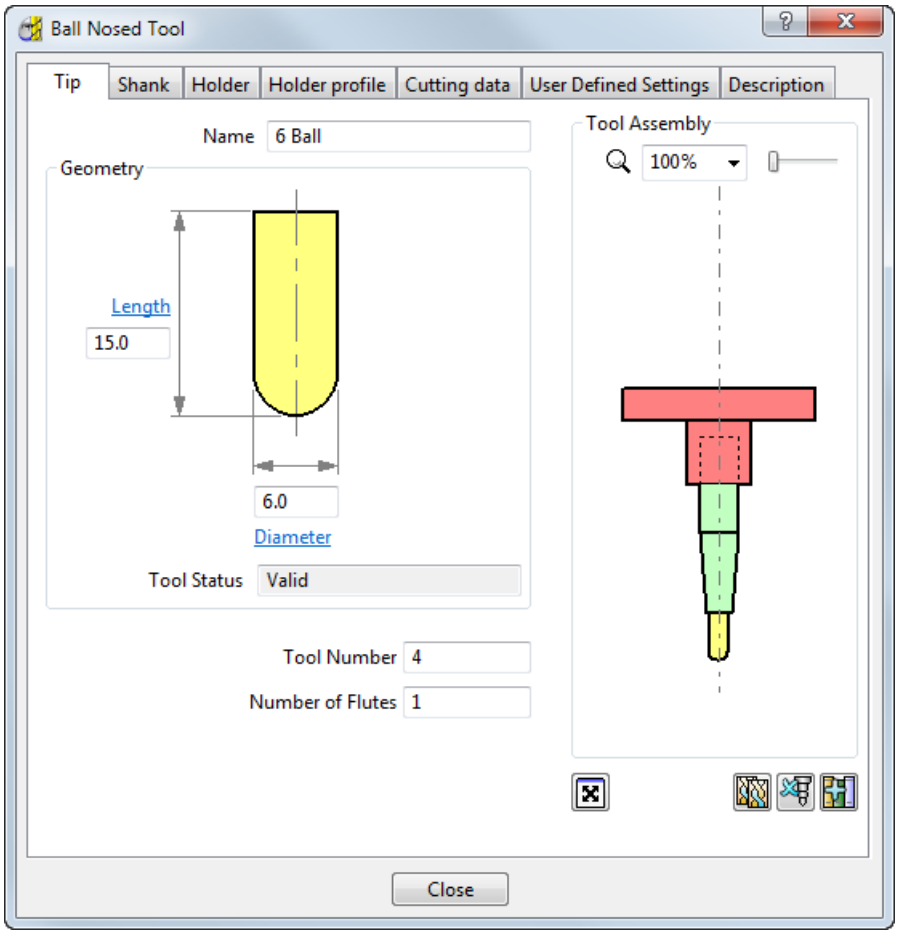
It is given the default name of **8 Ball_LS_TH_1**.

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

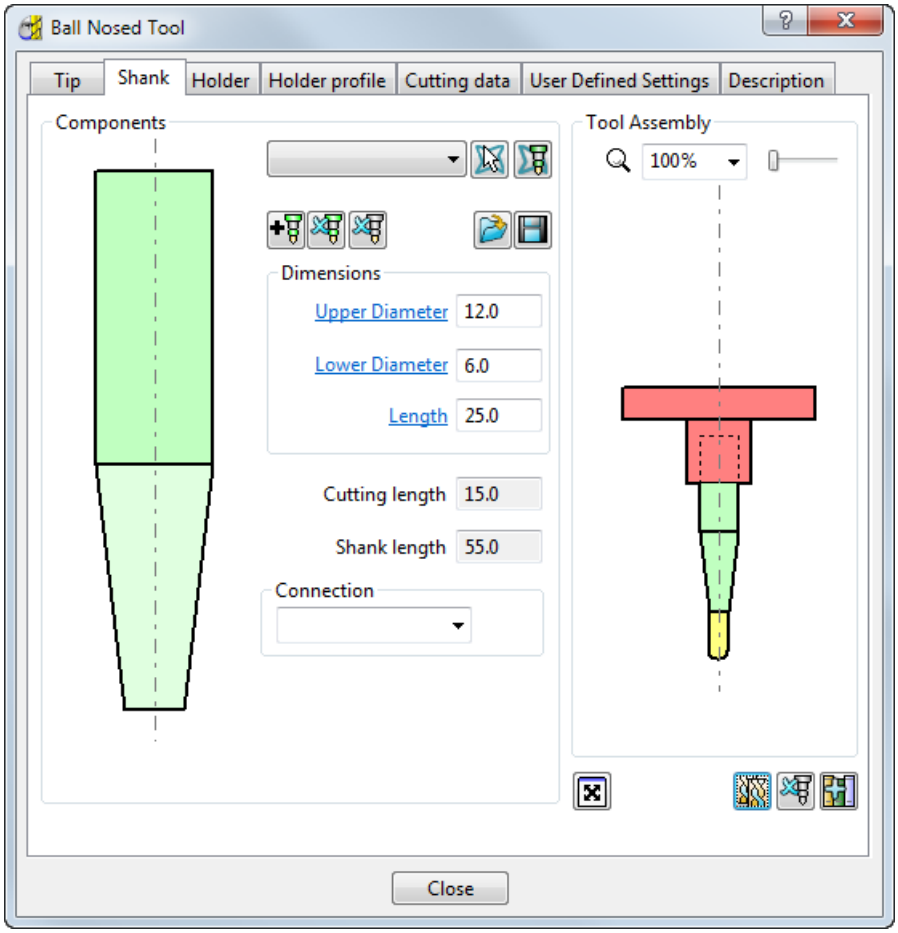
b Enter a **Length** of **15** mm.

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 **Toolpath > Tool** from the 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 unmachined 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.

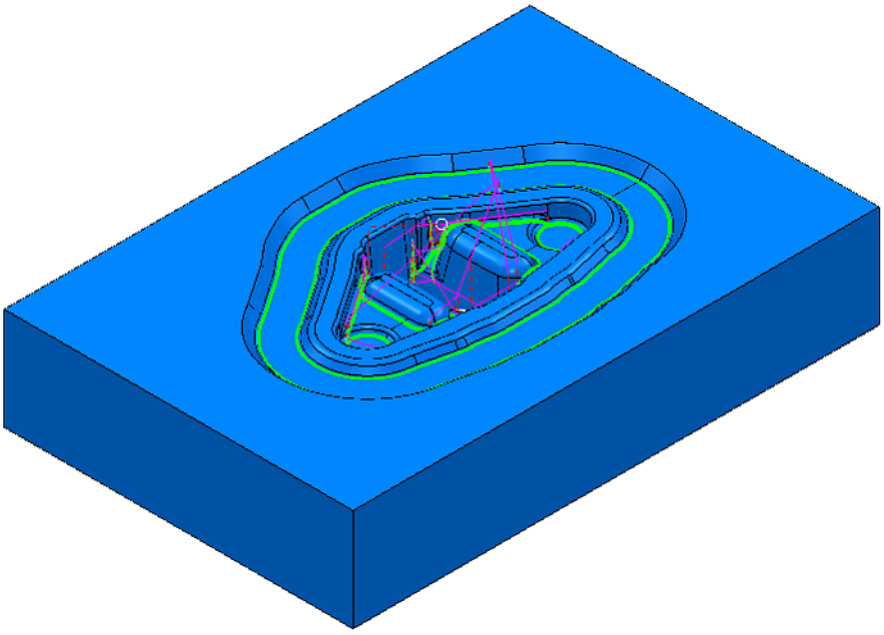
3 Click **Calculate** to generate the toolpath.

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.

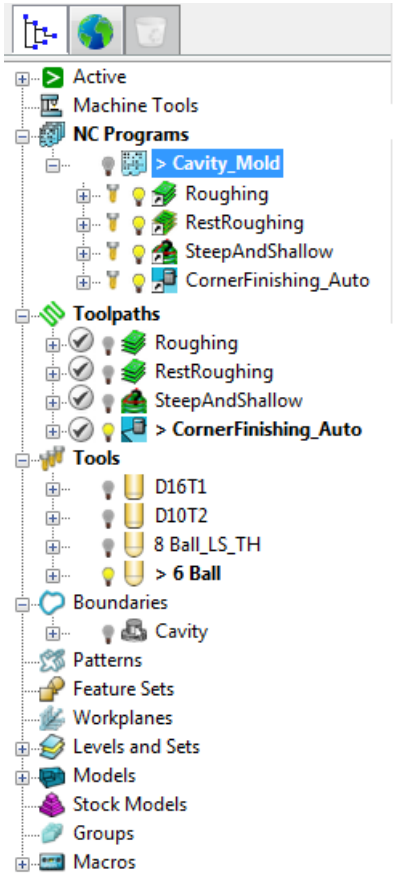
Displaying the Corner Finishing toolpath

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



Click ⊞ to expand the **Toolpaths** branch 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:

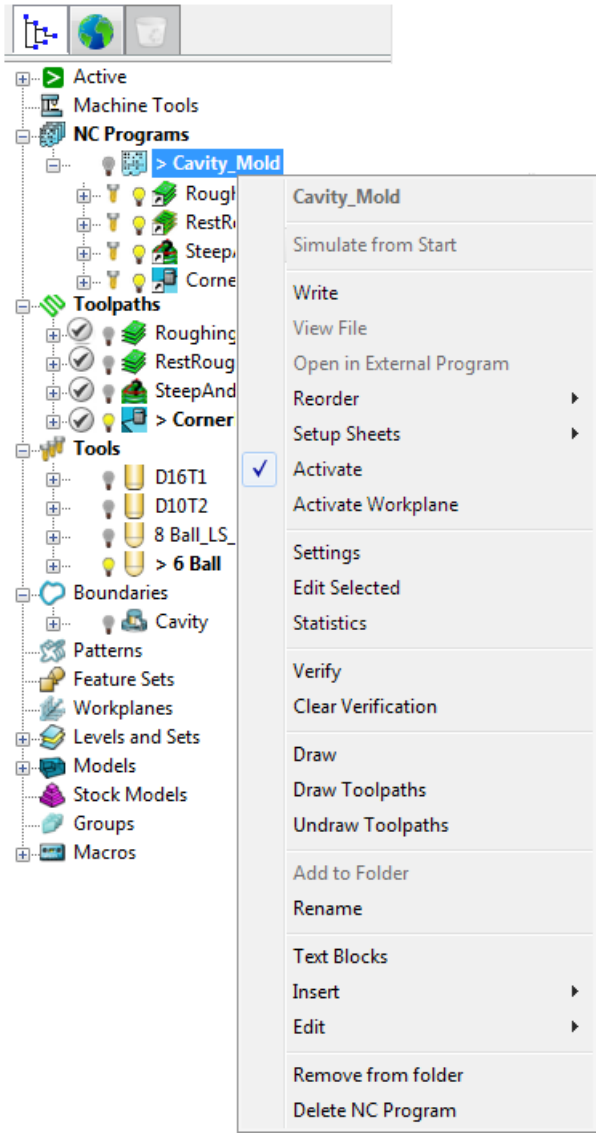


To save project changes, click on the **Main** toolbar.

Simulating the Corner finishing toolpath

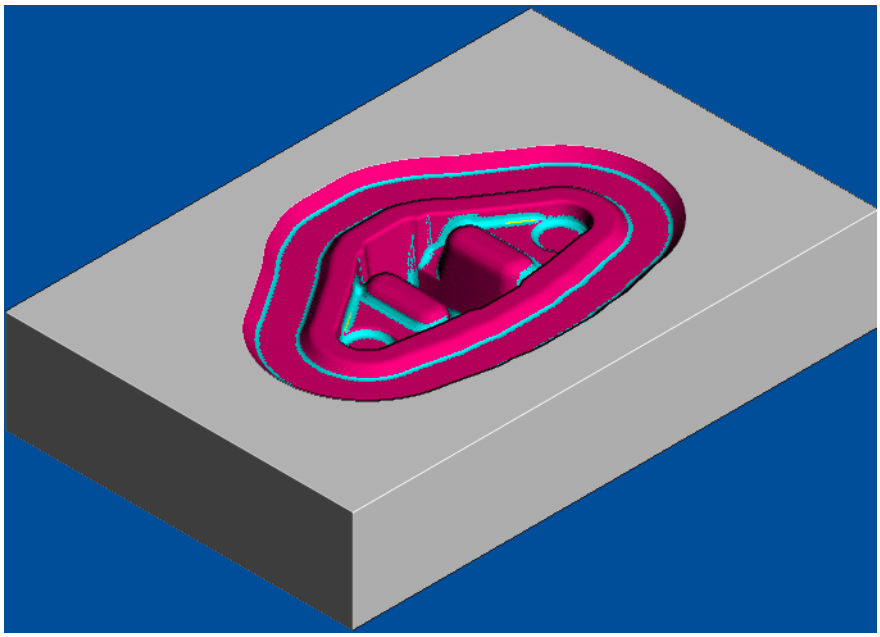
1. Click the **ISO1** button on the **View** toolbar to reset the view.
2. On the **ViewMill** toolbar, click the **Toggle ViewMill Window** button. It turns green, and activates the simulation window, which initially shows a light grey block on the current background.
3. From the **ViewMill Toolbar**,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

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

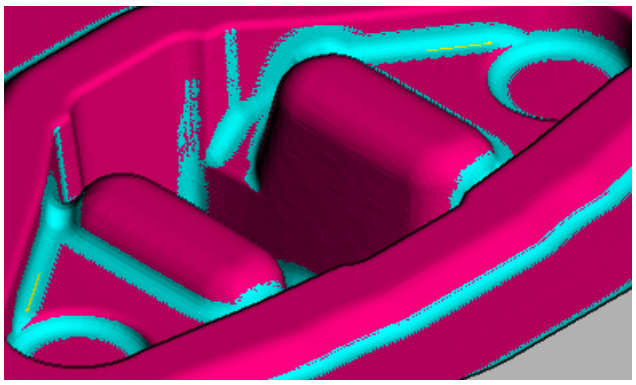


The NC program is automatically selected on the **Simulation** toolbar, and the **Play** buttons enabled.

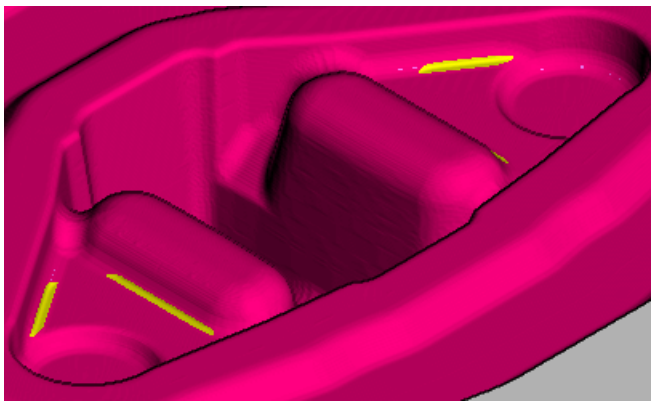
5 Click the **Play** button, and allow the simulation to run through to the end.



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



This compares 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 graphics 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:

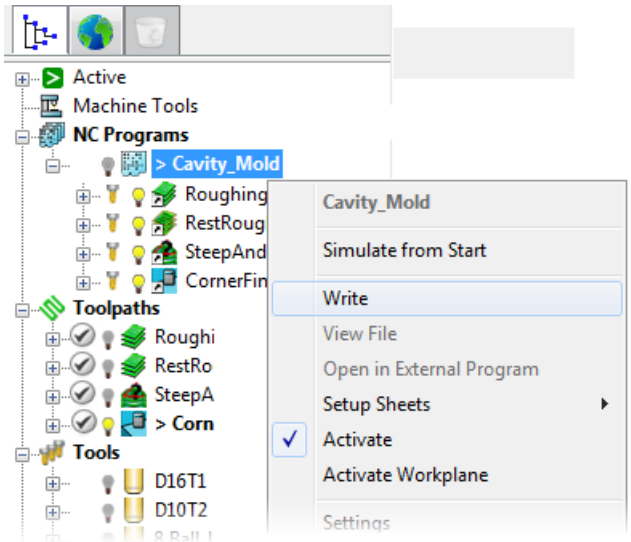
- Write each toolpath as a separate NC program file (see page 74).
- Write two NC program files (see page 76) with the toolpaths grouped by their functionality.

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.

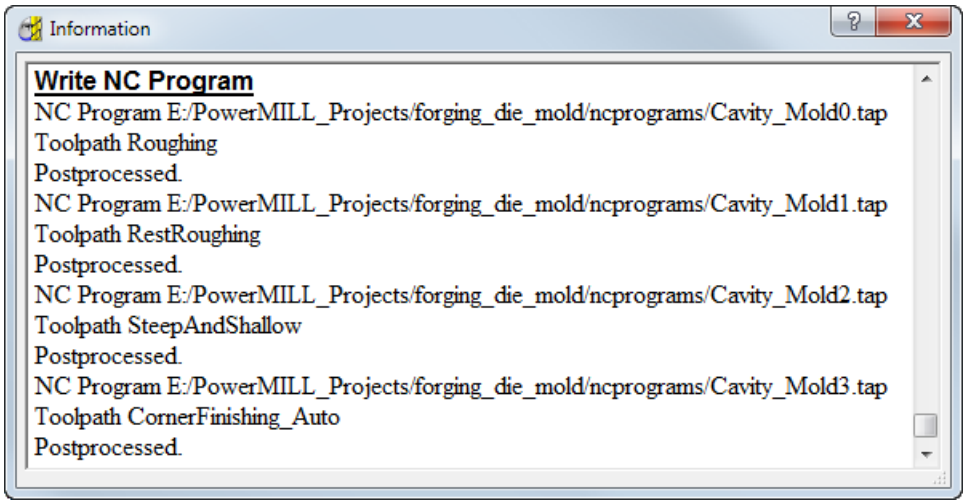
*To have separate NC program files for each of your toolpaths, the **Write File for Each Toolpath** option must be selected on the **Output** tab available from **Tools > Options > NC Programs**.*

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



The tool symbol next to 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.

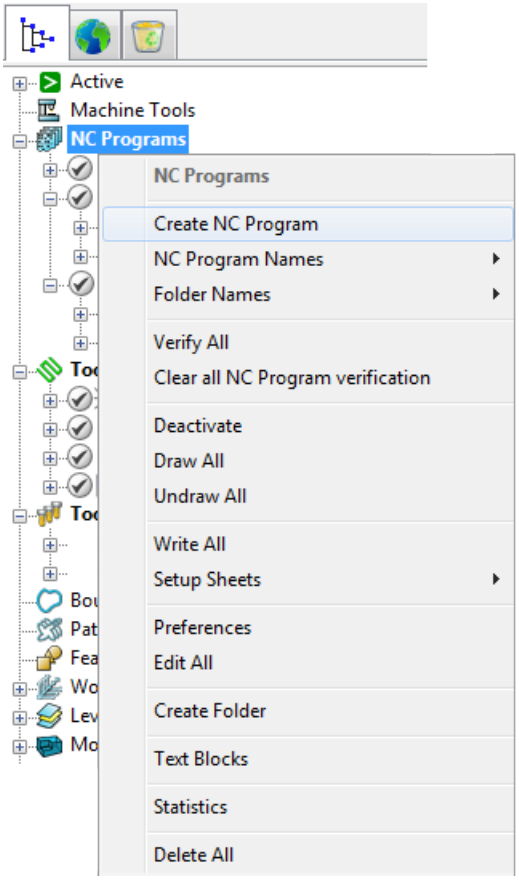
4 The colour of the NC program **Cavity_Mold** in the Explorer changes to a bright green colour, Cavity_Mold , to show it has been calculated correctly.

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

Writing two NC program files

This example 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.

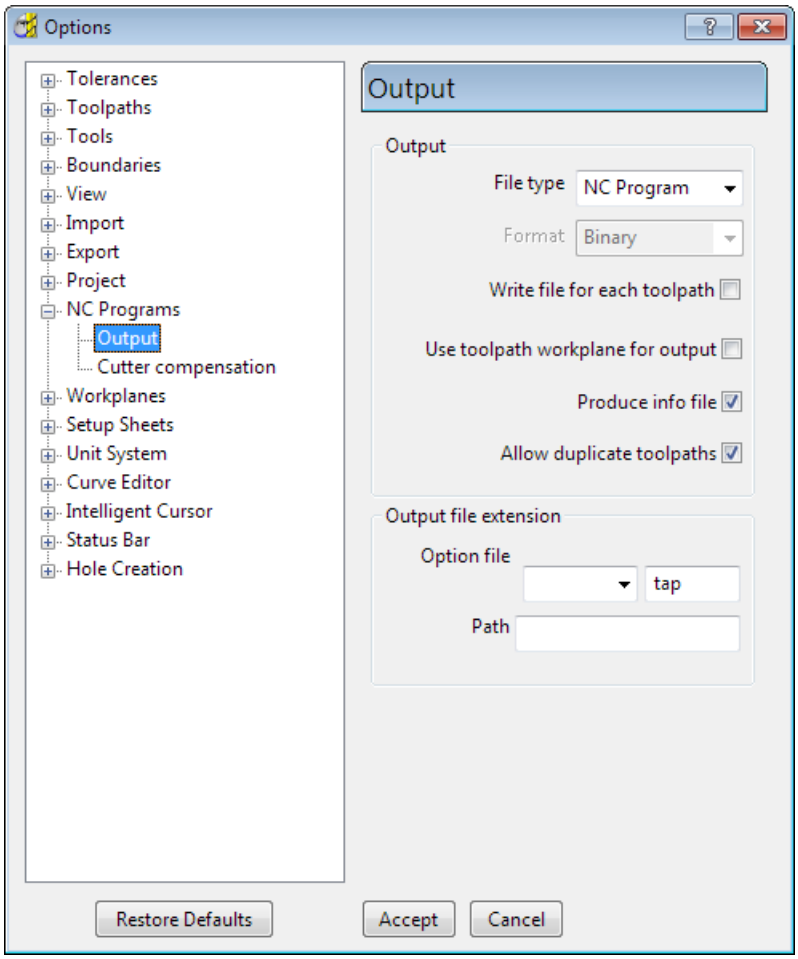
4 On the **Output** tab:

a If selected, deselect the **Write File for Each Toolpath** option. As a result, the **Output File** is displayed at the top of the **NC Program** dialog instead of the **Root Name**.

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

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

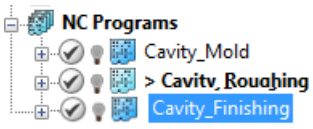
d Click **Accept** to update and close the **Options** dialog.



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

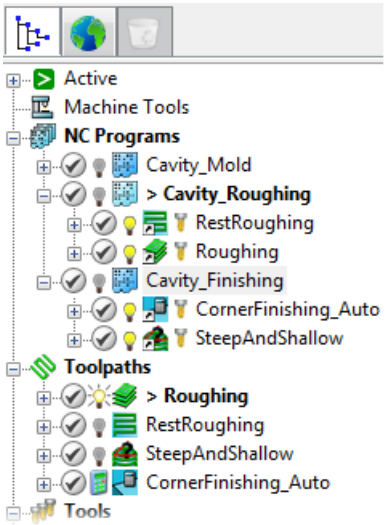
6 From the individual NC Program, **Cavity_Roughing,** context menu, and select **Edit > Copy NC Program**.

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**.

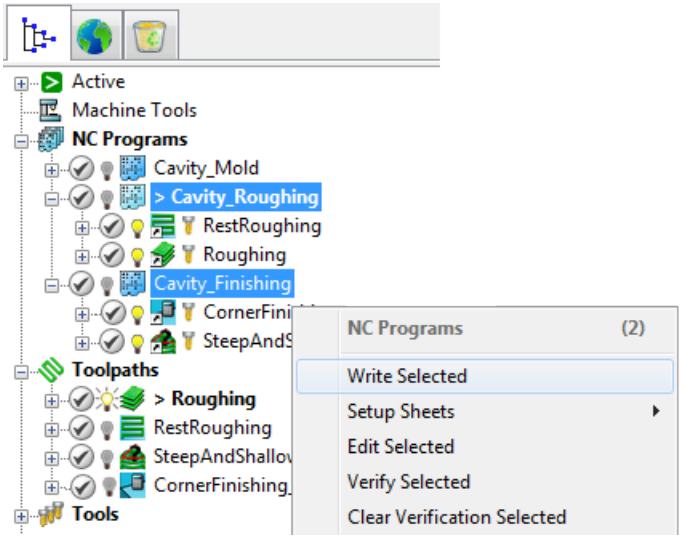


*To identify which NC program is currently active, expand the **NC Programs** node in the Explorer; the active program is shown in **bold** and preceded by the **>** symbol.*

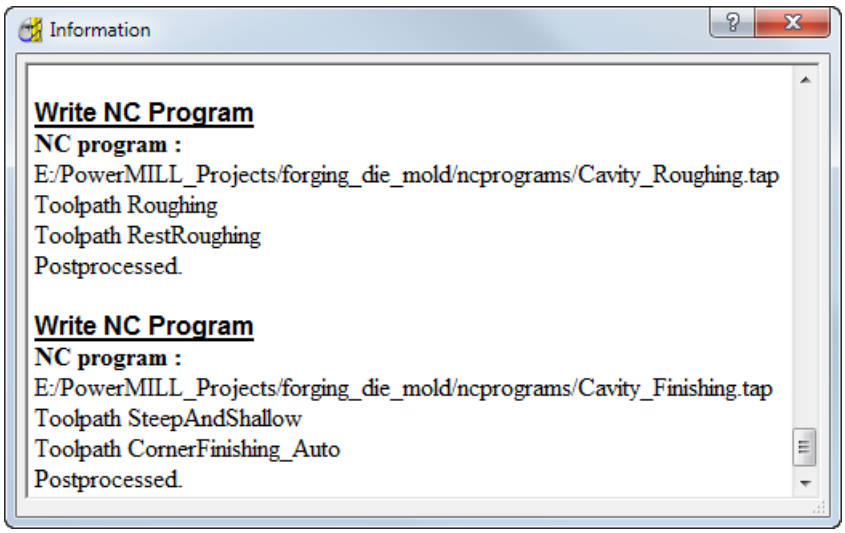
8 In the Explorer, move the two roughing toolpaths under **Cavity_Roughing**, and the two finishing toolpaths under **Cavity_Finishing**.



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



10 PowerMILL postprocesses 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

3

3D offset moves • 14

A

Along moves • 65
Area clearance
  Offset • 28, 40
  Tool geometry • 30, 41

C

Cavity mold example • 14
Constant Z moves • 14
Contour offsetting • 14, 45
Corner finishing • 65, 70
Creating tree objects • See Entities
Cutting length • 33

D

Defining block • 17
Displaying toolpaths • 36, 44, 62, 69

E

Entities
  Verifying active • 47

F

Finishing toolpaths
  Interleaved constant Z • 48
  Tool geometry • 49, 66
Forging die mold • See Cavity mold example

G

Gauge length • 33
Graphics window • 6

H

Holder length • 33

I

Imperial units • 5, 10
Isometric vews • 20

M

Metric units • 5, 10
Models
  Opening • 15
  Resizing to fit • 20, 22
  Shading • 20
  Viewing • 20

N

NC programs
  Creating • 24
  Setting preferences • 24
  Writing • 73, 75

O

Opening
  Models • 15
  Projects • 46
Overhang • 33

P

PowerMILL
  Closing • 5
  Context help • 12
  Entities • 9, 10
  Explorer • 9, 47
  Manuals • 13
  Projects
    Opening • 46
    Saving • 19, 38
  Starting • 5
  Startup window • 6
  Status bar • 12
  Toolbars
    Main • 6, 28
  Tooltips • 11, 12
  Units
    About • 5, 10
    Changing to imperial • 10

R

Rapid move heights • 12, 34
Resizing to fit • 20, 22
Roughing toolpaths
  Rest roughing • 40
  Roughing • 28
  Tool geometry • 30, 41

S

Screen layout • 6
Selected surface boundary • 55
Shank length • 33, 41
Stepdown • 34, 43
Stepover • 34, 43
Stitch moves • 65
Surfaces
  Non-tangential, machining • 65
  Selecting • 55

T

Temporary file area • 4
Tip length • 33, 41, 66
Toggle Simulation window • 39, 45, 63, 70
Tolerance • 34
Tool geometry • 30, 41, 49, 66
Toolbars
  Main • 6, 28
Toolpath
  Displaying • 36, 44, 62, 69
  Finishing • 48, 58, 65
  Roughing • 28, 40
  Simulating • 39, 45, 63, 70
  Writing • 73, 75
Tools
  Ball nosed tool • 49, 66
  Length definitions • 33
  Start point • 35
  Tip radiused • 30, 41
Tooltips • 11, 12

V

ViewMill • 39, 45, 63, 70
Views
  Isometric • 20
  Locating • 22
  Panning • 24
  Predefined • 20
  Zoom in and out • 22

W

Wireframe • 20, 36, 44, 55, 62, 69
Working directories • 2
  Setting up a Home environment in Windows • 2
  Setting up your working directories • 2, 4
  Specifying PowerMILL Default Directory paths • 3
  Specifying PowerMILL temporary file area • 4