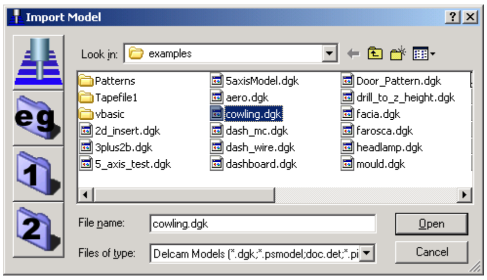
CADCAM using Powermill

In this exercise you will create the toolpaths necessary to machine the Cowling model.

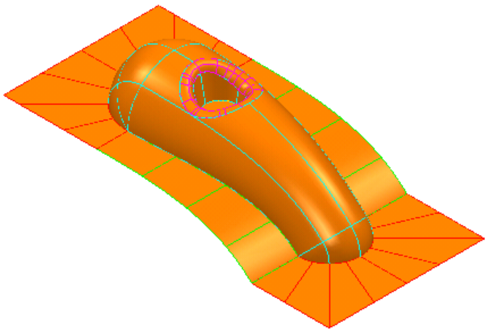
Create a folder on your h: called **Powermill**. Inside this create a folder called **cowling2009.** This will be used to save all work associated with this exercise.

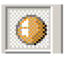
Now copy model from: **powermill\cowling.dgk** from my **S:** drive and paste it into the above **Cowling2009** folder on your H: drive.



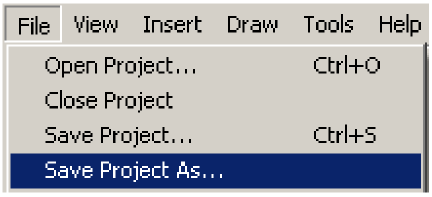
Next import model into Powermill.
File Import Model
Select: **H:\...\Cowling.dgk**

Select this  for the isometric view shown.



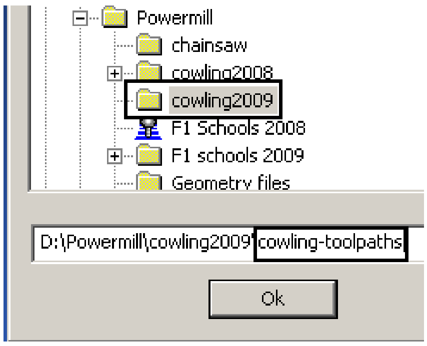
Press **F3** or the  icon to shade the part.
All the machining work we will do on this geometry will be saved independently in a 'project file'.

Although no toolpaths have yet been generated. Save the project anyway in order to define its name.



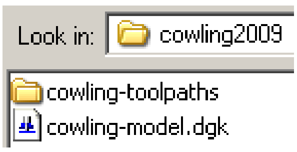
Select: **File Save Project As**:

For the location specify the **Cowling2009** folder and for the name enter **cowling-toolpaths** followed by **OK**.

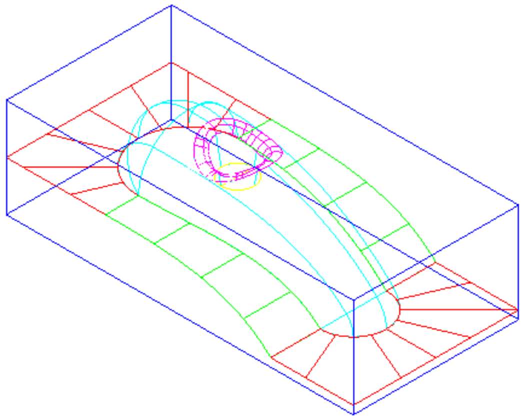


When opening the project in future the project will appear as show.

However examining using explorer you will find that it is, in fact, a folder containing several files with cryptic file names.



Block size



Next determine the block inside which we will carry out all machining operations.

Select block size .

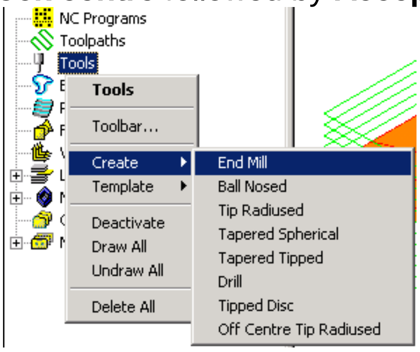
Choose **calculate** to determine the size automatically. An 'expansion allowance' can be applied if extra material is required.
Finally choose **accept** to close dialog.

Choose Feeds and speeds form.  Accept default values.

Rapid move heights  Choose **reset** followed by **Accept**.

Start point form  Choose **reset to block centre** followed by **Accept**.

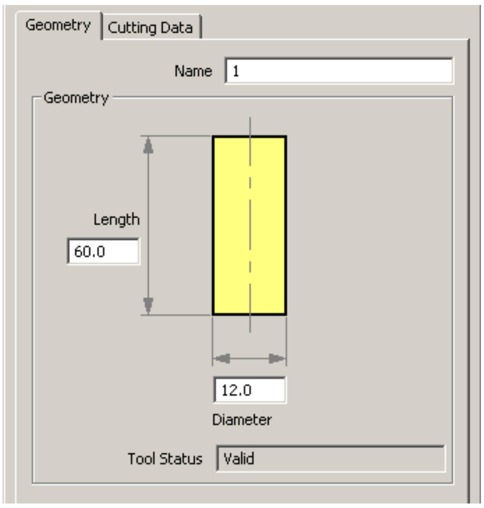
Tool definition



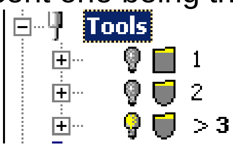
Before machining can take place tools must first be defined. For the purposes of this exercise, you will create 3 tools:

16mm Endmill
12mm Ball nose
6 mm Ball nose

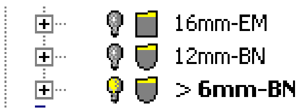
To create the first tool, right click on **tools** in the feature manager then select **create** and **endmill**.



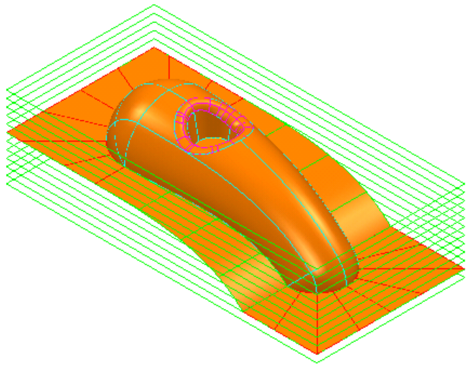
In the dialog box which appears enter 16mm for the diameter. Accept the default length.
Create the 2 ball nose cutters in a similar way.
This results in, 3 tools added to the feature manager under tools. The most recent one being the active tool.



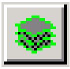
Right mouse click on each tool in the feature manager and select **rename** and give them familiar names as shown opposite. N.B. Names can be specified either at the time of creation or afterwards as above.



Area Clearance

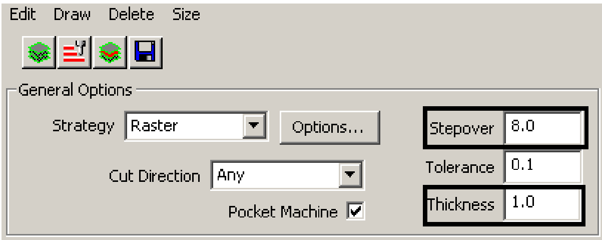


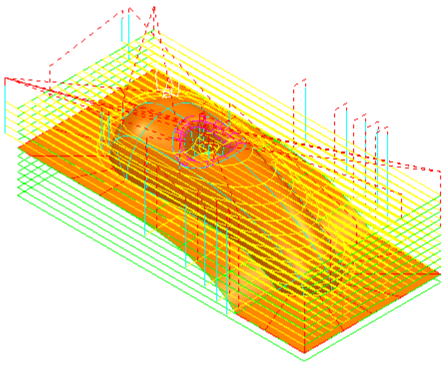
Activate the 16mm Endmill and then generate an area clearance cutter path for the cowling.

First define Z levels  using a stepdown of 5mm.
In ‘defined by’ section select **Stepdown**
For the distance specify **5** then choose **calculate** to work out the Z levels.

The Z levels will be represented graphically by green lines on the block. Finally choose **[close]** to accept these values.

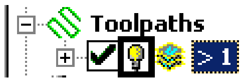
Next generate the area clearance cutter path using:
Step over 8mm (half tool dia)
Thickness 1mm (leave on 1mm for finish machining).





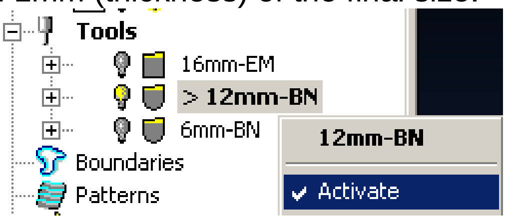
Finally choose [Apply] to generate the toolpath. This will result in the tool path shown opposite.
Under toolpaths in the feature manager will be a new item representing the toolpath just created.

In preparation for creating the next toolpath, hide the current toolpath by selection the bulb symbol until it turns grey. 



Semi Finish machining

Next carry out semi finish machining using Raster machining. Semi-finishing is used to remove the terraced appearance from the model to ensure a smoother finishing cut. If finish machining were attempted at this stage the uneven metal removal rates would result in changing cutting forces which might compromise the final surface finish. Instead semi-finishing is used to machine the block to within a uniform 1mm (thickness) of the final size.

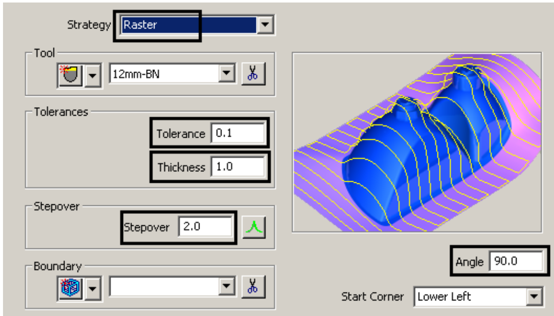


To generate the semi finishing toolpath activate tool number to (12mm-BN),

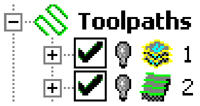
Next select finish machining .

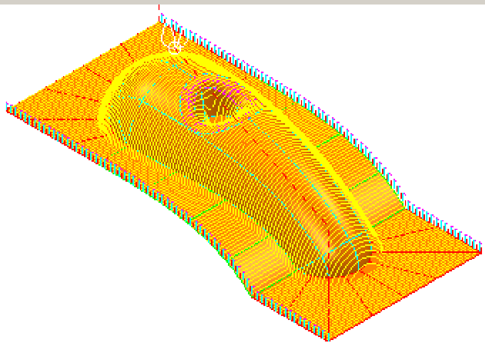
In the dialog which appears specify the following:-

Strategy: **Raster**
Tolerance: **0.1**,
Thickness: **1mm**,
Stepover: **2mm**.
Angle: **90 degrees**

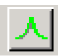


Then select **Apply** to generate machining paths across the part as shown opposite. Again a further item will appear under toolpaths in the feature manager.

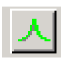




Finish machining

This time the part will be machined down to its final size. In this case it is required to achieve a specific surface finish. For this reason rather than specifying the step over directly the step over will be calculated to ensure that the cusp height is within the specified tolerance. This will be achieved by selecting the  icon.

Generate the finish machining path using the following parameters.

Tool: **tool 3 (6mm-BN)**
Strategy: **Raster**
Tolerance: **0.1**
Thickness: **0 (final geometry)**
Stepover: Calc based on tolerance using  icon.
Angle: **45 degrees**

Again select apply to generate the tool path.

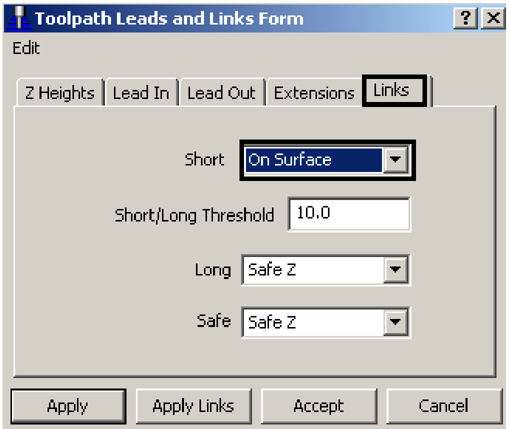
Leads and links

An efficient cutter paths in one in which the highest percentage of the time as possible is spent cutting metal. Therefore any non cutting moves are inefficient.

Currently there are wastefull vertical moves, at the end of each cutter pass, in the case of both the semi-finishing and finishing toolpaths. These can be modified using the leads and links option.

To modify the links select the toolpath of interest. Then select the lead and links option at the top of the screen.

In the dialog which appears.

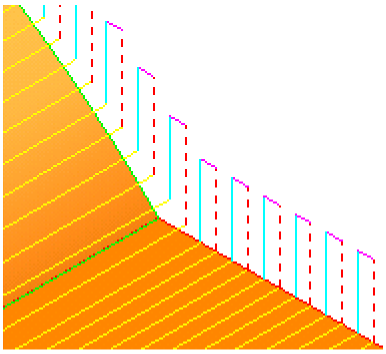


In the dialog which appears select ‘Links’ and specify on surface for the ‘Short’ links category.

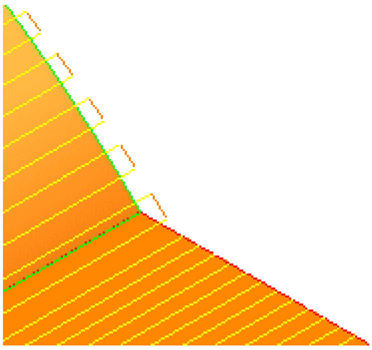
This operation may also be performed during toolpath creation using a similar icon on the finishing toolpath dialog box.

Perform this operation for both Semi-finishing and finishing toolpath.

Before leads and link modification



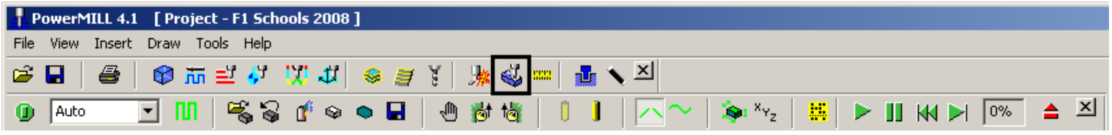
After leads and links modification



Simulation

Simulation allows the user to get a preview of the machining operation, enabling them to detect potential collisions before sending the program to the machine.

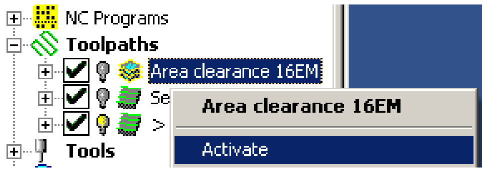
To activate the simulation toolbar select the  icon.

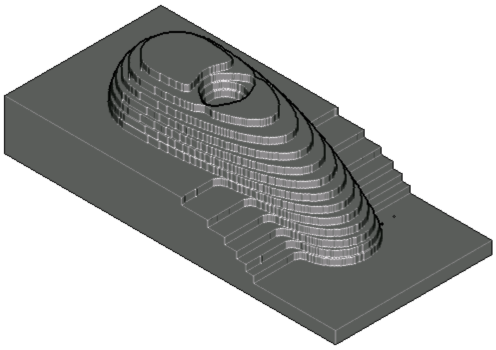


A B C D E

Select icon (A) to activate the simulation environment.
Select icon (B) to activate the higher resolution mode.
Select icon (C) to display a graphical representation of tool during simulation.
Select Icon (D) to play the simulation.

<u>Area clearance toolpath simulation.</u>
To simulate the area clearance or roughing cutter path.
Activate this cutter path on the side bar,

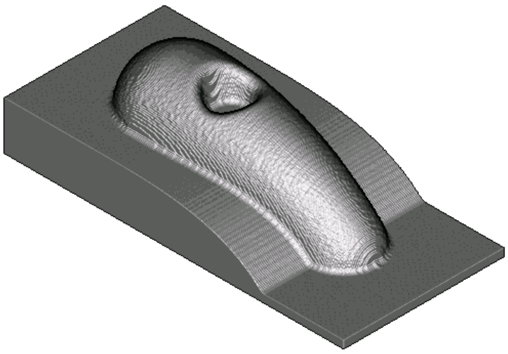




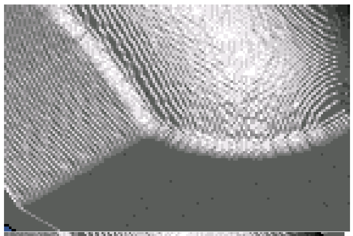
then select the simulate icon (D).

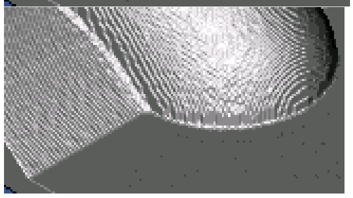
Repeat this process by activating the semi-finishing and finally the finishing toolpath in turn and then simulate.

Finally close the simulation window.



Cleanup machining



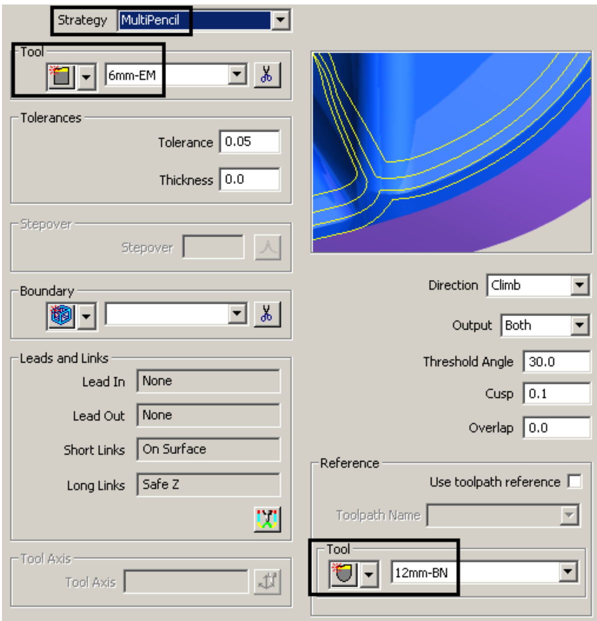


On close examination of the simulation result it will be clear that concave corners are still slightly round since the last cutter used was a ball nosed cutter. It will therefore be necessary to 'clean-up' these corners.

However as the bulk of the geometry has already been machined to its finals size it will only be necessary to machine those areas were inaccessible using the previous cutter.
There are a number of machining strategies available for this purpose. i.e. stitch, multipencil. etc

To do this exit simulation mode by selecting the icon and return to the machining environment.

In this case we will use the multi-pencil machining strategy to do some additional machining on the sharp corners using a 6mm end mill.
The software does not automatically know the size of the previous cutter used. Instead this is specified in the reference tool section.
It is not uncommon to 'pretend' that the reference tool was larger than that which was actually used. This ensures that the machining overlaps with the previously machined surface.



First of all create a new tool. This time create a 6mm-EM.

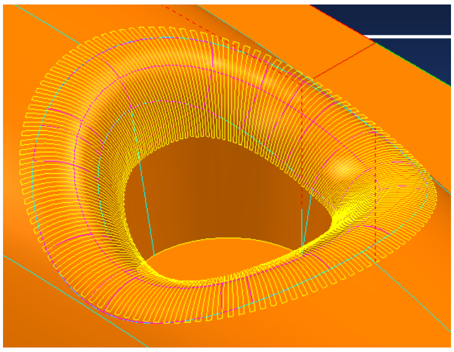
Next select the finish m/c dialog and enter the following information.

Strategy: **Multipincil**
Tool: **6mm-EM**
Ref tool: **12mm-BN**

Select apply to generate the toolpath.

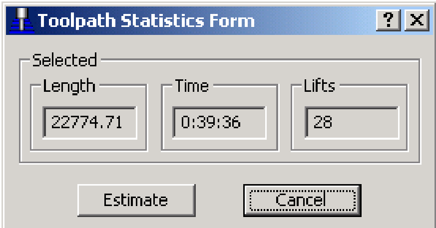
Finally return to the simulation environment and simulate all tool paths.

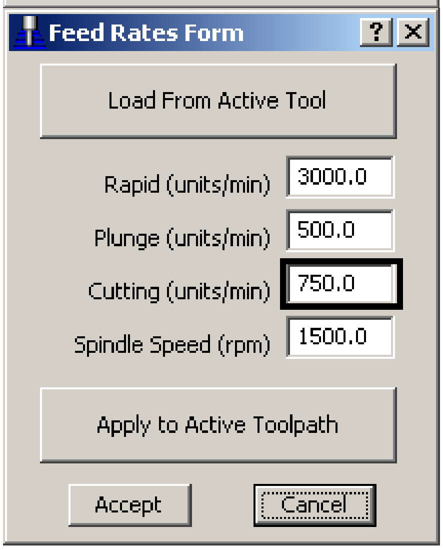
Radial Machining



Carry out radial machining around to fillet at the opt of the hole
Cutter: 6mm BN
Experiment with the parameters to get the effect shown opposite.

Processing time





To determine the machining time for a particular tool path right mouse click on the tool path and select **statistics**.
Total machining time can be determined by adding the total machining time of each tool path.

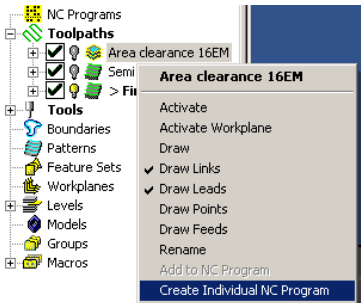
The time will of course be dependant on the feed rate.
The feedrate may be set in advance of creating the toolpath by selecting. 

If the toolpath has already been defined then the feedrate for that toolpath may be modified by activating the toolpath, activating the adjacent dialog box the selecting:

<u>Machining time.</u>
Edit the feedrates of the existing toolpaths according the attached recommendations then. Then evaluate the machining time for each toolpath and enter below.

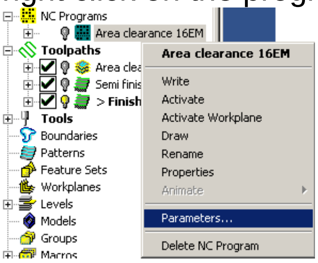
	Time:
Area clearance	
Simi-Finishing	
Finishing	
Cleanup	
Radial Machining	
Total time	

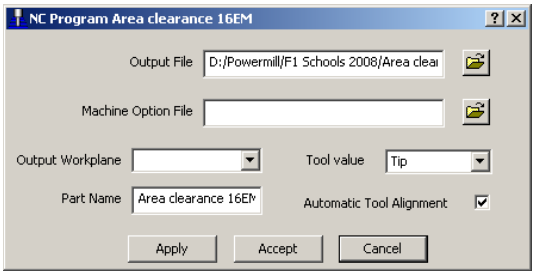
Post processing



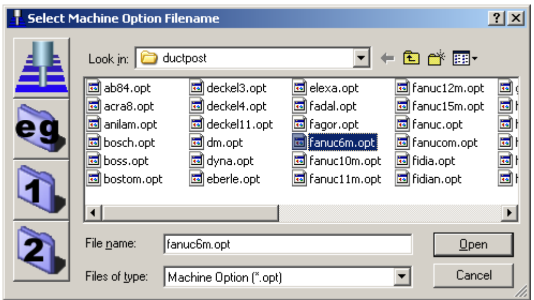
So far cutter paths are held in a neutral cutter location format called CL data.
Having created the toolpath it is then converted into CNC code. This is achieved using a process called post processing. This is a two stage process, first it is converted into a machine independent neutral NC program. This program is added to the NC programs at the top of the feature tree.

The cutter path is then ready for conversion into a Machine specific CNC program. To do this the type of machine must first be specified. To do this right click on the program and select parameters.

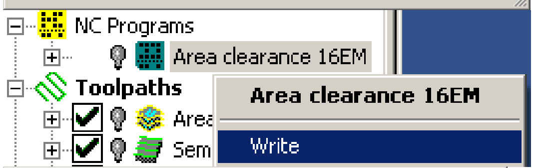




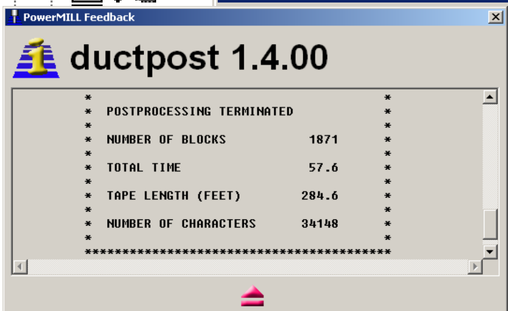
Select the folder opposite the machine option file.
Choose the required machine. E.g.
Fanuc6m.opt
Followed by **Open** then choose **apply**.
Also Specify Program name in Output file.



Finally right click on the program and choose **write** to create the actual NC program.



This writes a CNC program to disk using the format determined by the machine option file.
Shown opposite are some details such number of blocks (lines) estimated machining time, tape length (obsolete) and the number of characters.

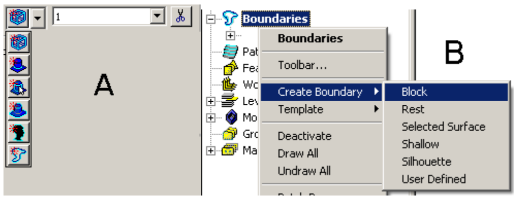


The CNC program is saved on disk with the extension *.tap. Its contents may be viewed by opening with a text editor or word processor. This program is subsequently transferred to the machine using a variety of methods included direct loading from disk to 'drip feeding'.

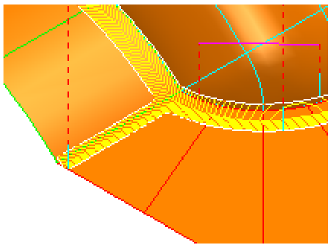
Powermill boundaries.

Boundaries may be use isolate specific areas for machining.

When a boundary is active machining is restricted to inside or outside of the boundary.
A boundary may be defined from the finish machining dialog box (A) or in the feature manager (B).

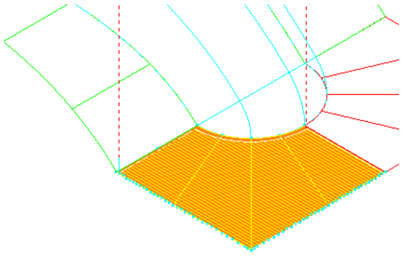


Rest boundary
Shown is a machining operation performed within a rest boundary.

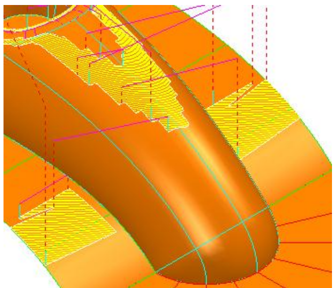


Rest defines a boundary which defines a region which is inaccessible to a cutter under a specified size.

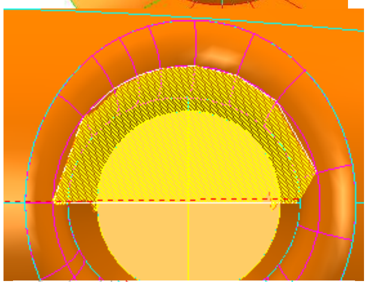
Selected surface
Boundaries can be created by selecting a surface and then using it to create a boundary.



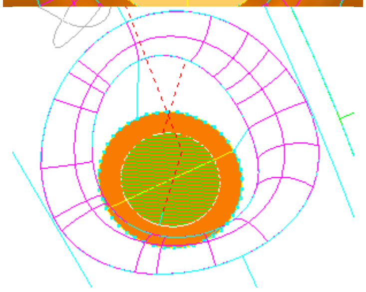
Shallows region boundaries.
Boundaries can be created to define regions of a model while lie between two specified angles to the horizontal.



User defined boundary
Shown is a machining operation performed within a user defined boundary.
A user defined boundary is defined by manually selecting points around a specified region of the model.



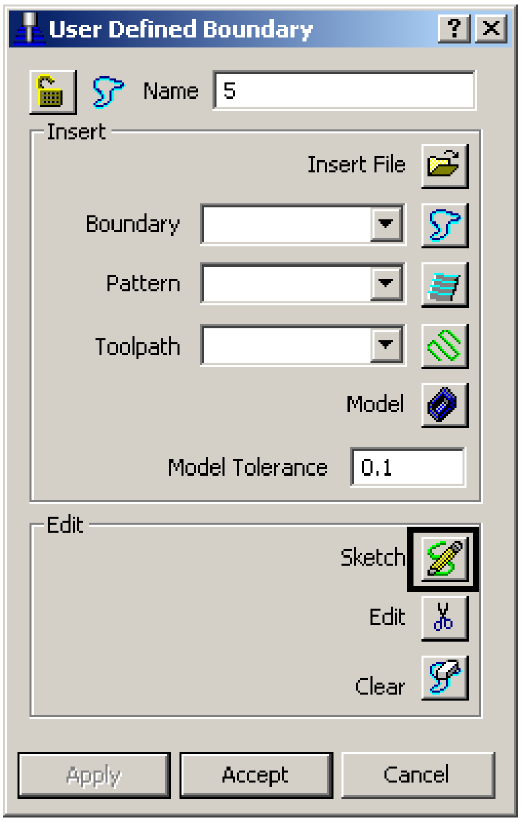
The image opposite illustrates how the tool radius is taken into account when generating a toolpath within a specified boundary.
Not also that adjacent surfaces are always taken into account.



The create a user defined boundary, right click on boundaries the select

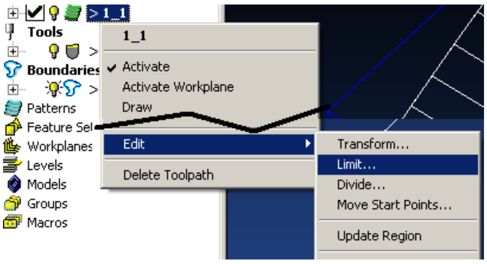
- Create boundaries
- User defined.

When the dialog box shown opposite appears select sketch and pick one by one points in the model to define the boundary.



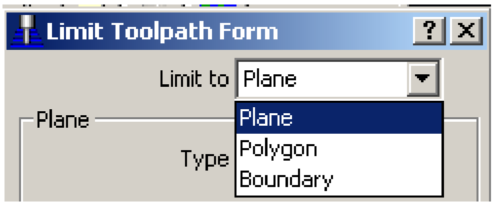
Limiting to a boundary

Once a cutter paths has been create it may be limited for a variety of reasons.
If for example a tool is not long enough to machine to the full depth of the component you may limit to a certain minimum value.

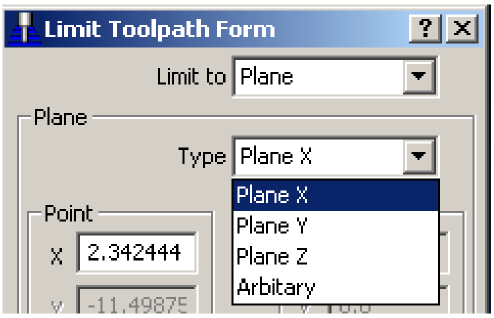


To limit a toolpath right click on the toolpath and select **Edit** and **Limit**. This presents the user with the dialog box shown opposite.
In this dialog the use has 3 options:

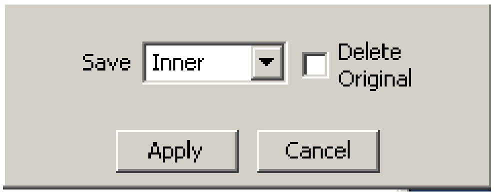
- Plane
- Polygon
- and boundary



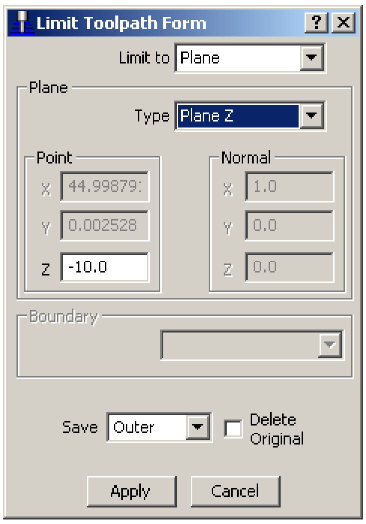
The 'plane' option allows the cutter path to be limited to specified horizontal or vertical plane.
The polygon option allows the user to draw a polygon 'on the fly' with which to limit the cutter path.
Finally the boundary option allows the user to trim to a pre-defined boundary.

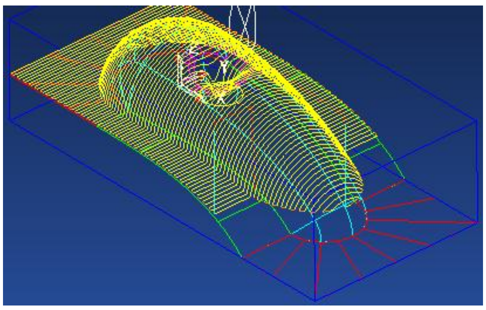


In each case the user may choose to keep the inside or outside or both and may choose to delete or keep the original as required.

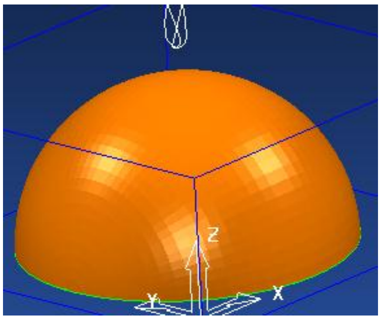


You could for example limit an existing tool by removing anything below Z-10.

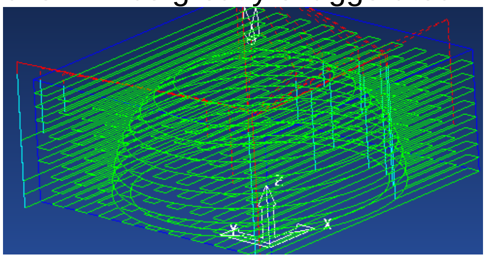


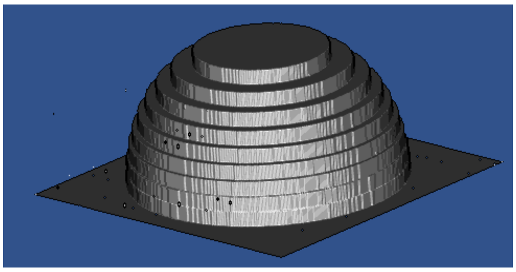


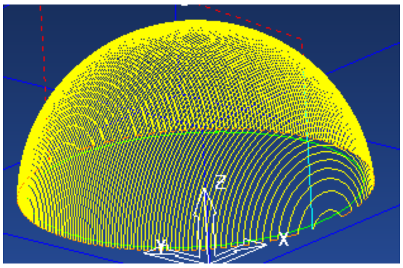
Boundaries / Const Z machining

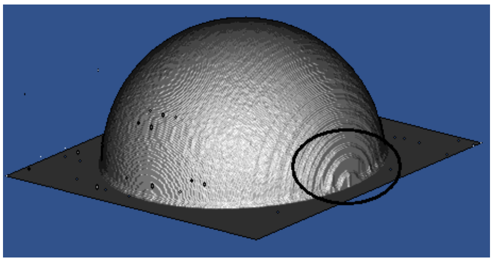


Shown opposite is a hemisphere.
Below it are typical area clearance and raster finish toolpaths.
Using raster machining this would result in adequate surface finish in the middle section but when traversing along steep faces the step over will be greatly exaggerated .





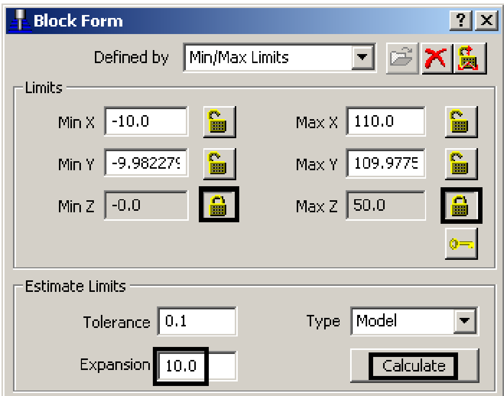




An alternative is to use constant Z machining for the steep faces.
This can be used most effectively with boundaries. By using boundaries we can isolate the 'shallow' and 'steep' areas and single these out for separate machining strategies.

<u>Dome assingment</u>

In your powermill folder create a new folder called **Dome2009**.
Copy the files **Dome-model.stl** from my **S:** drive and insert it into this folder.
Start Powermill in import the model Dome-model.stl.
Finally save the project **dome-toolpaths** in the folder called **Dome2009**



Again we will determine the block size as before however in this case we will expand it horizontally be 10 mm in the Z and Y directions.

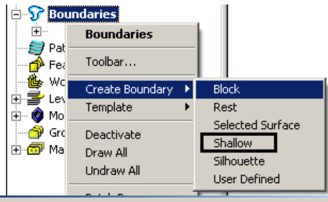
This is achieved as follows.
Activate the block size tool and select calculate in the normal.
Next lock the min Z and max Z by selecting the lock symbol adjacent to these numbers.
Finally enter 10 for Expansion and select calculate again.

Creating boundaries

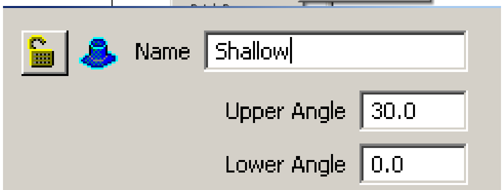
In this step you will create boundaries to isolate shallow regions (regions between horizontal and 30 degrees to the horizontal) and steep regions (areas between 30 degrees to horizontal and almost vertical).

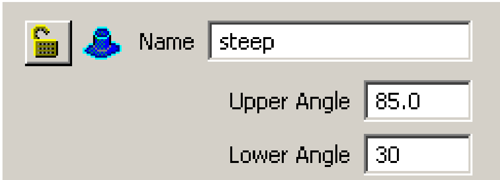
To create a boundary select right click on boundaries and select

- Create boundary
- Shallow.



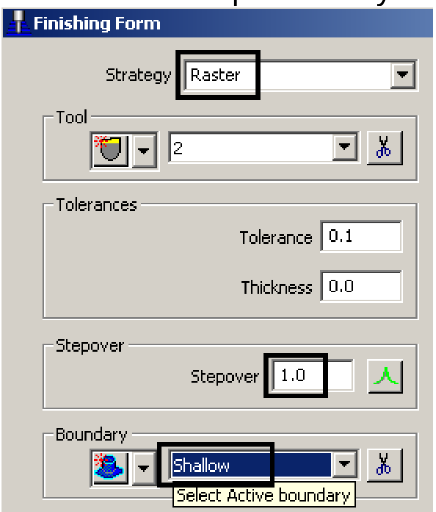
In the dialog box which appears specify the angles
Lower angle: **0.0**
Upper angle: **30.0**
Give it the name: **Shallow**
Choose **Apply** followed by **Accept**
Repeat for steep boundary
Lower angle: **30**
Upper angle: **85**
Give it the name: **Steep**
Choose **Apply** followed by **Accept**

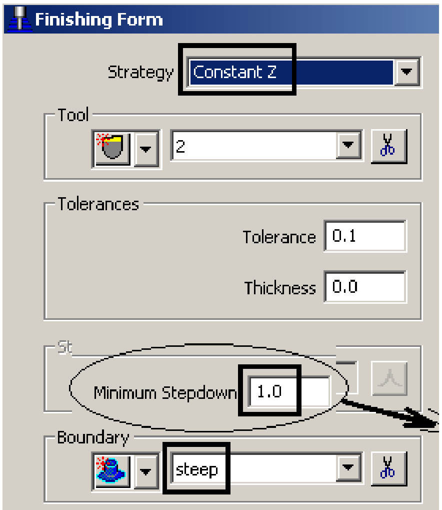


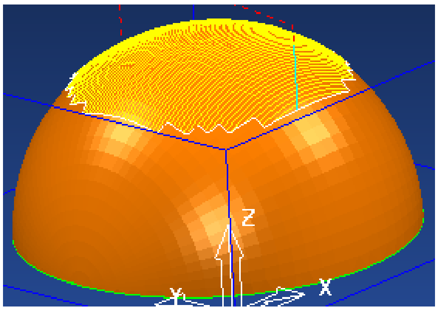


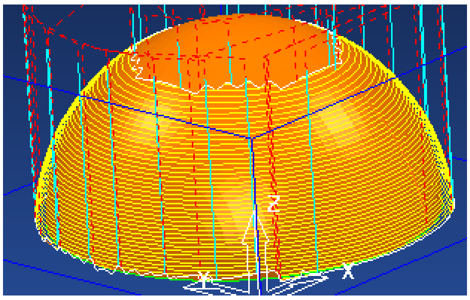
Finally perform regular raster machining in ‘shallow’ regions and constant z machining in steep regions. To do this:

- Activate the shallow boundary and raster machine the top of the domes.
- Activate the steep boundary and constZ machine the sides of the dome.









Finally Save the project.