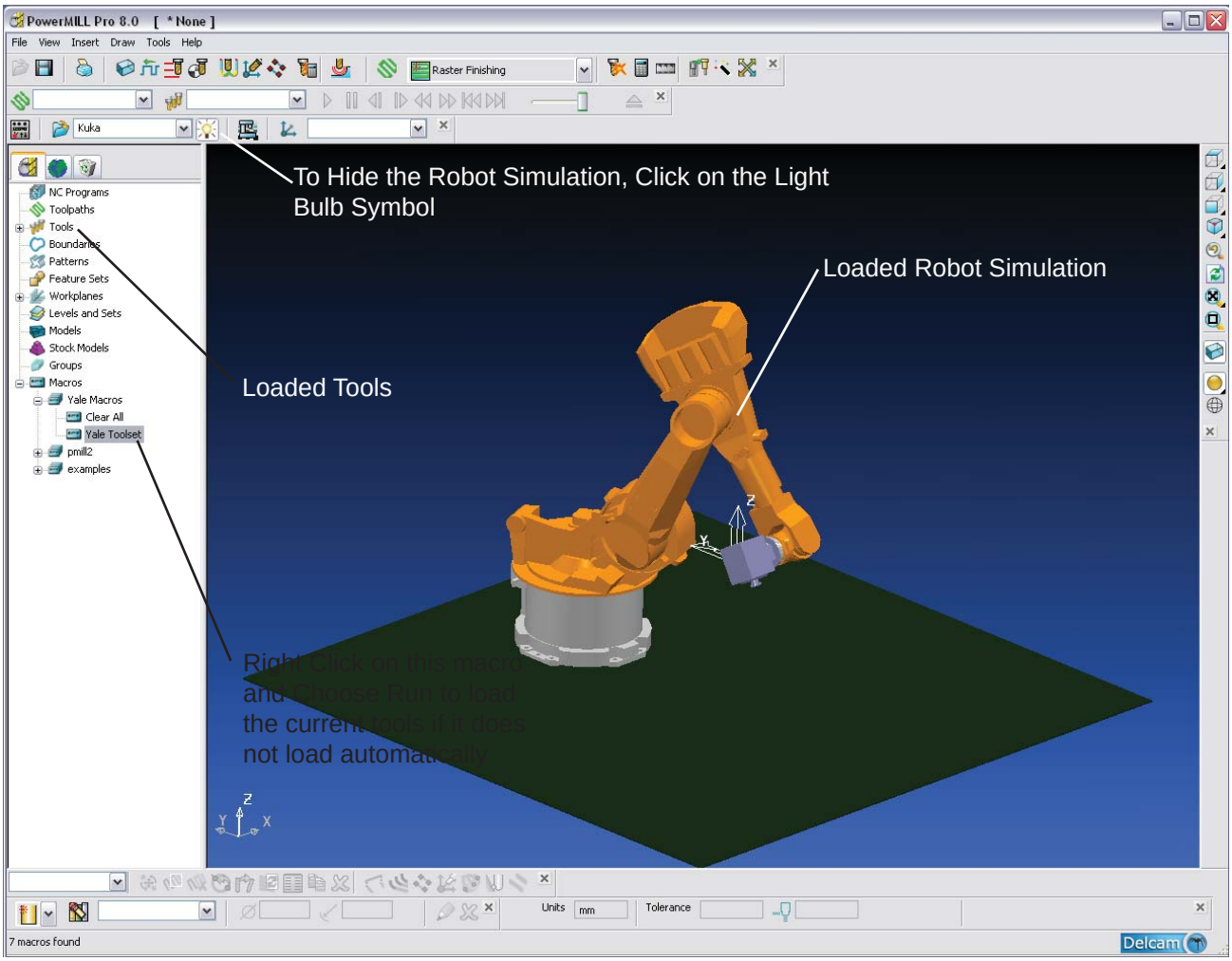
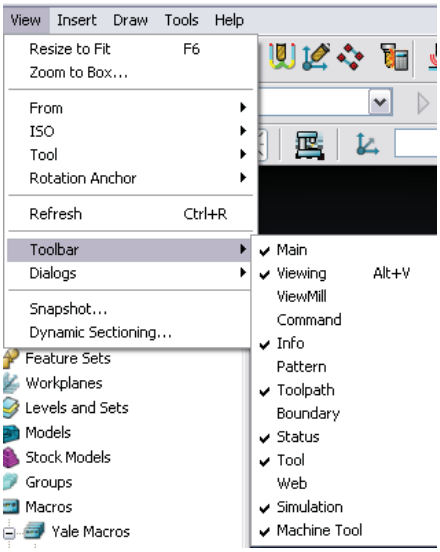
Using Delcam Powermill

Written by: John Eberhart
DM Lab Tutorial

Powermill is a sophistical tool path generating software. When powermill loads, make sure the macro that loads the current tools works. If there are no pre loaded tools, click on the Yale Toolset macro and the tools should load. If it still does not work, click on the DM online website under 5 Axis and the current tool dimensions will appear and you can manually create a tool.





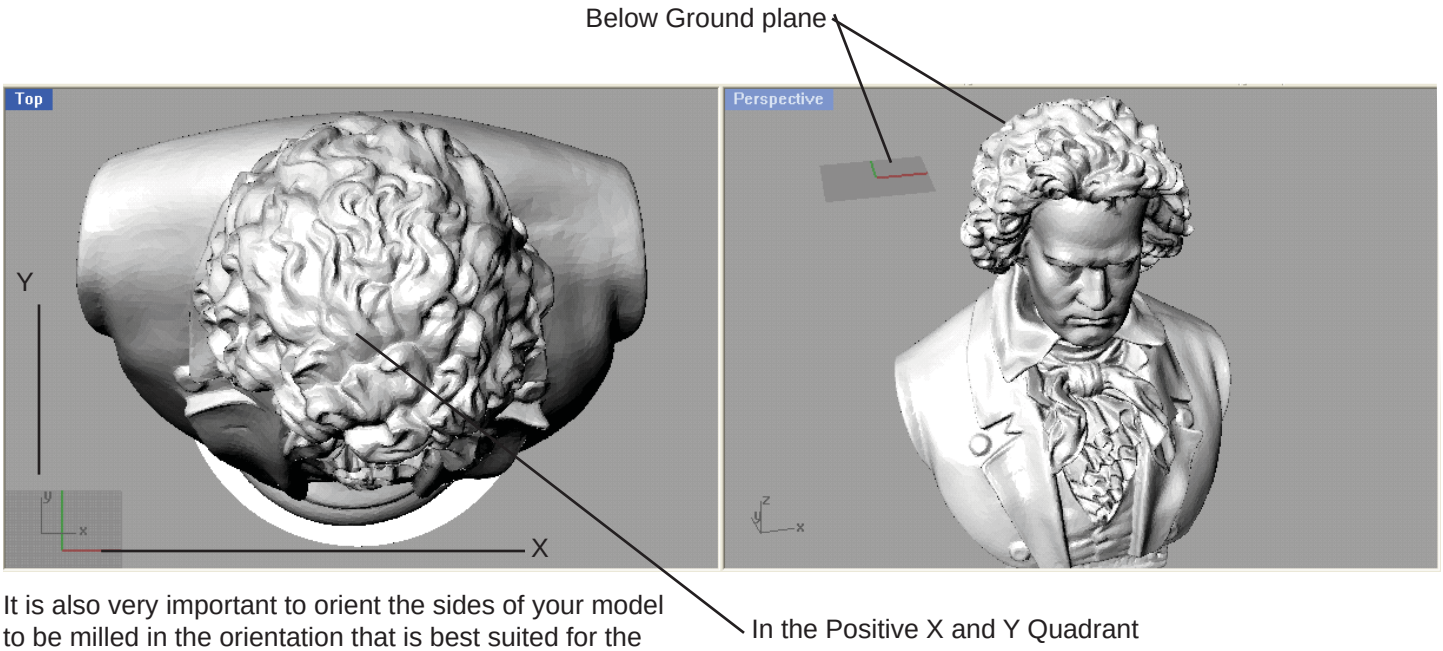
Verify that you have all off the toolbars you need loaded.

Click on View>Toolbar> and choose tyhe following Toolbars:

Main
Viewing
Info
Toolpath
Status
Tool
Simulation
Machine Tool

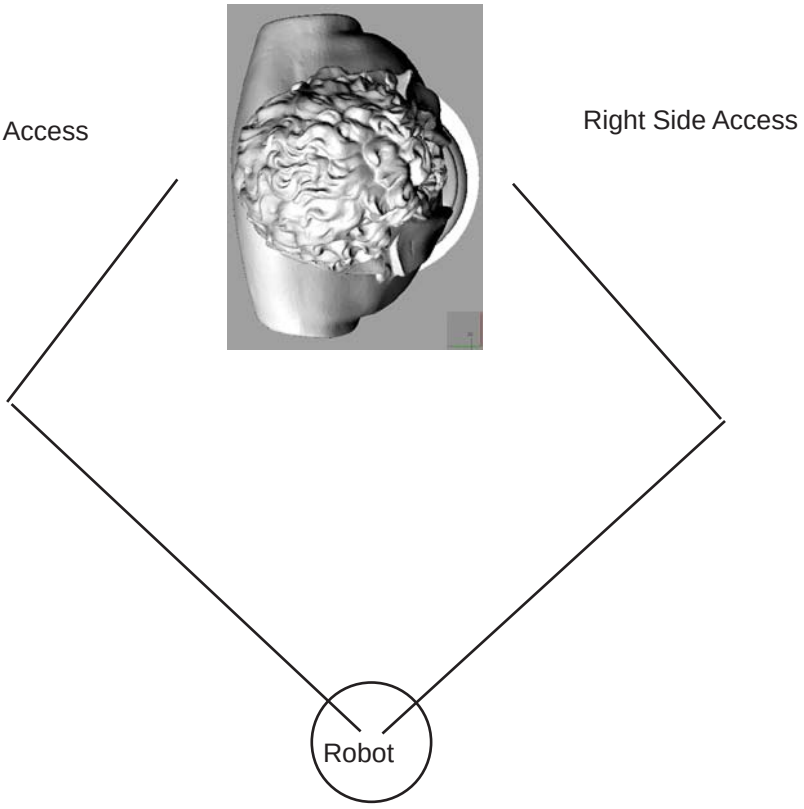
Setting up the correct origin for your model

It is important to setup the correct origin of your model. The robot will record a base point. That base point is the same point as the model origin. I recommend you located the model to be milled below the XY ground plane and in the positive XY quadrant.



It is also very important to orient the sides of your model to be milled in the orientation that is best suited for the robot.

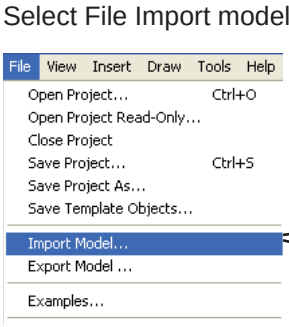
The robot has the best access to the top, left and right sides of the model. The back and front of the model is less accessible. It is best to orient your model with this in mind

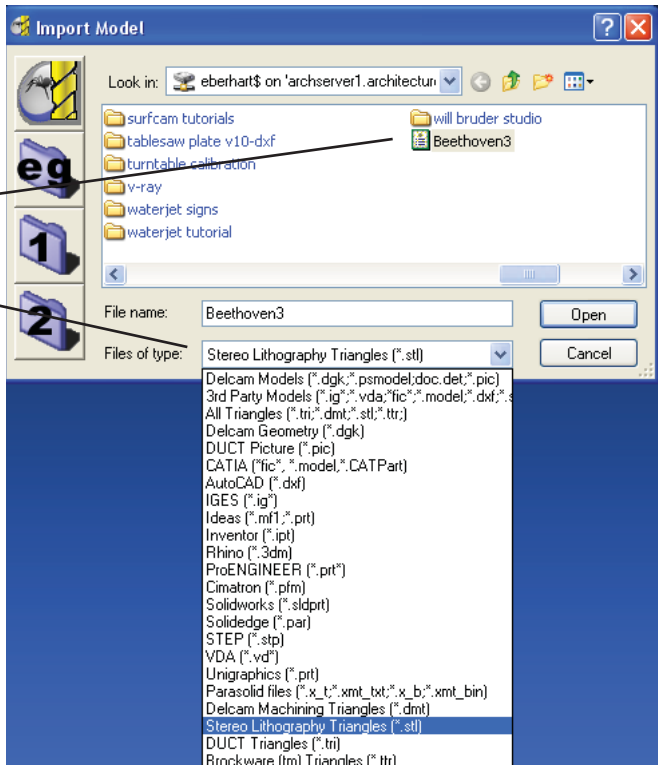


Importing Your Model

Powermill can import a wide range of models from Polygon models to Surface models. Powermill will import a number of native files such as Rhino and Inventor. However, if you your model is coming for other sources, then you can save it as either a STL or IGES. If it is a polygon type model save your model as STL. If it is a Nurbs type model save it as IGES.

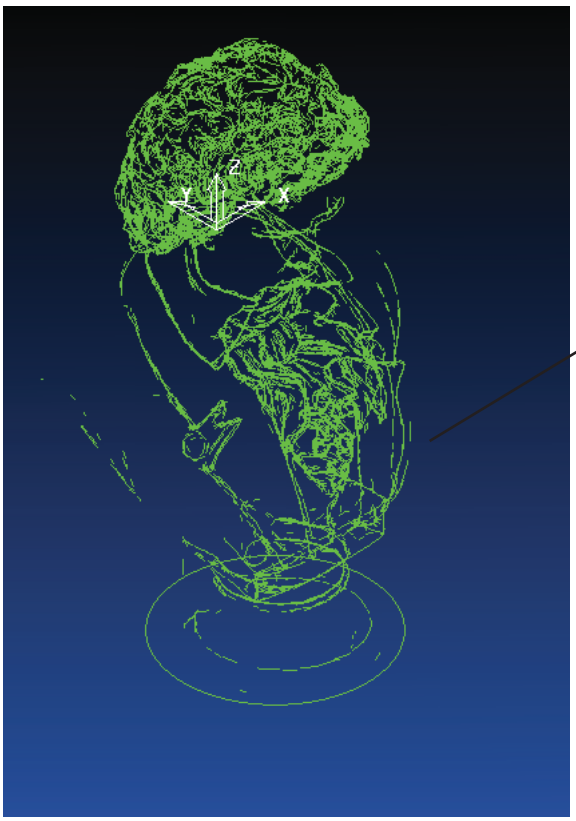
Select File Import model





Choose your Model and Select open

Your model will probably come in looking like a series of lines. To correct this, click on the wireframe icon and then the render shaded icon. You model will then look as it should.

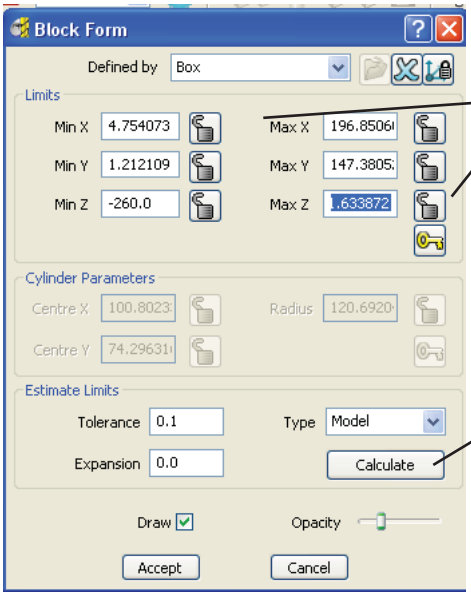


Render Shaded icon

Wireframe icon

Setting up a Stock Block

Click on the Block icon at the top of the powermill interface

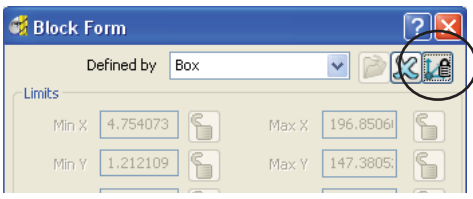


These values read off the extents of the model. You can change these numbers to adjust the size of the bounding box.

Click the Calculate button and a box will be generated around the model.

I adjusted the Min and Max XY and Z values to reflect the area I want to mill and to reflect the size of material I am going to mill

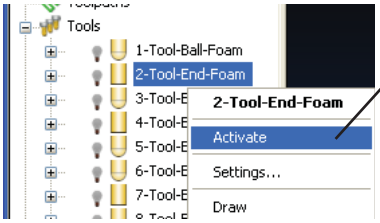
Note: ***You need to make the stock material the same size or a little bit smaller than your stock piece.***



Once your stock piece is set you need to lock the values. Click the lock Icon.

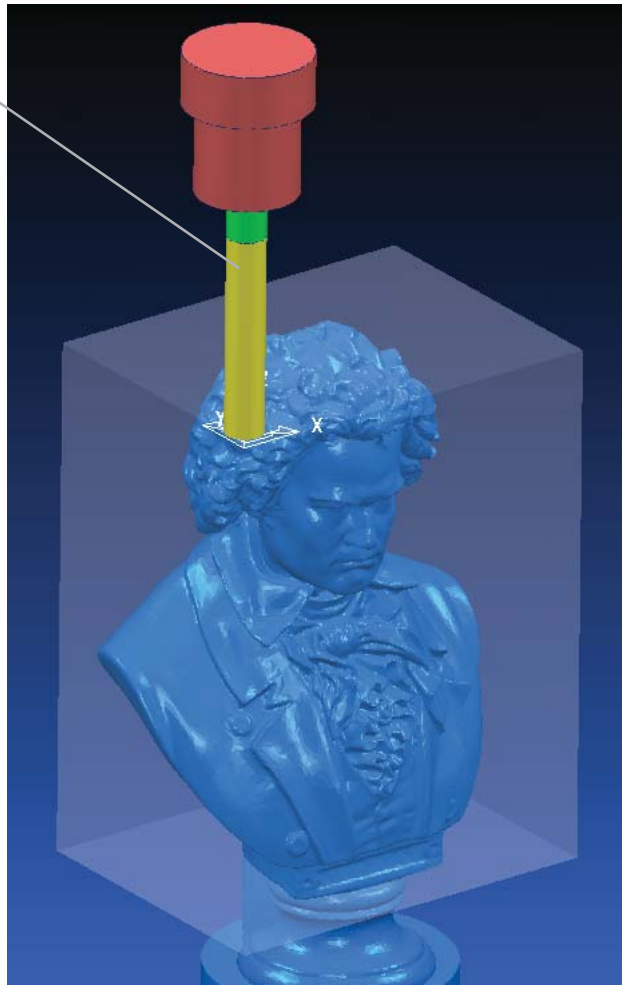
If you do not do this you will run into problems down the road.

Activating a tool and Setting a safe height

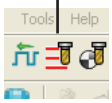


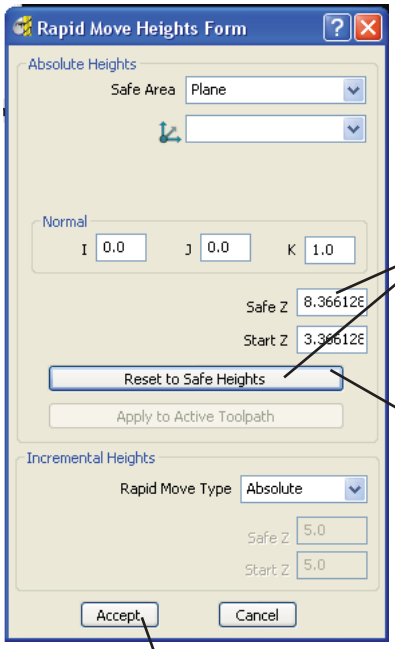
To make a tool active, right click on the tool you want to activate and select Activate. The tool will appear in the modeling window.

If you want to see the tool a solid (instead of wireframe), right click on tools and check Shaded.

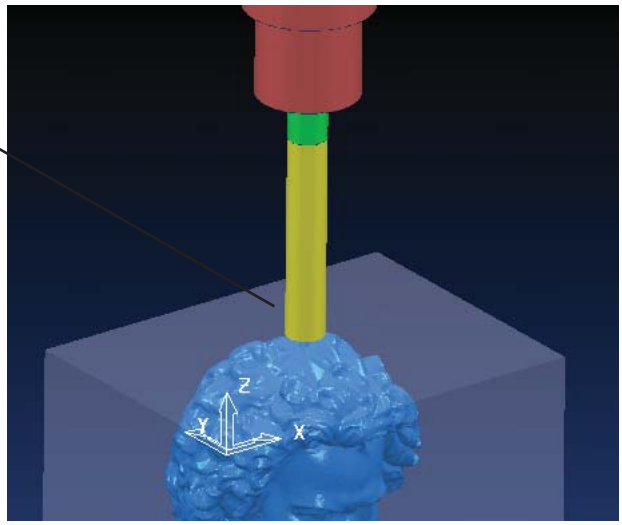


The tool should be pointing upward. However you need to set the safe heights. To do this click on the reset to safe height button.





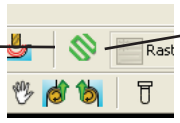
Click the Reset to Safe Heights, and the bit should move to the middle of the block and be above the block by the specified safe height.

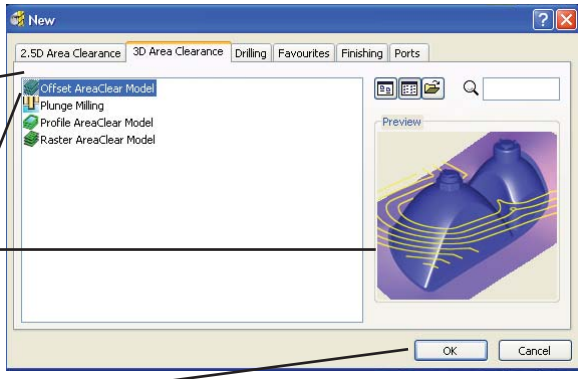


Click Accept when done

Creating a Rough Cut

Click on your machining Strategies button to open the machining library.





For Roughing a model, click the 3D Area clearance tab and then click on Offset Area Clear Model. A toolpath example will show up.

Click OK

Name your Toolpath (Rough Top)

Verify You are using the correct tool

Leave tolerance at 0.1

Thickness is how much the bit will stay away from the finished surface. Leave at 1 mm if unsure.

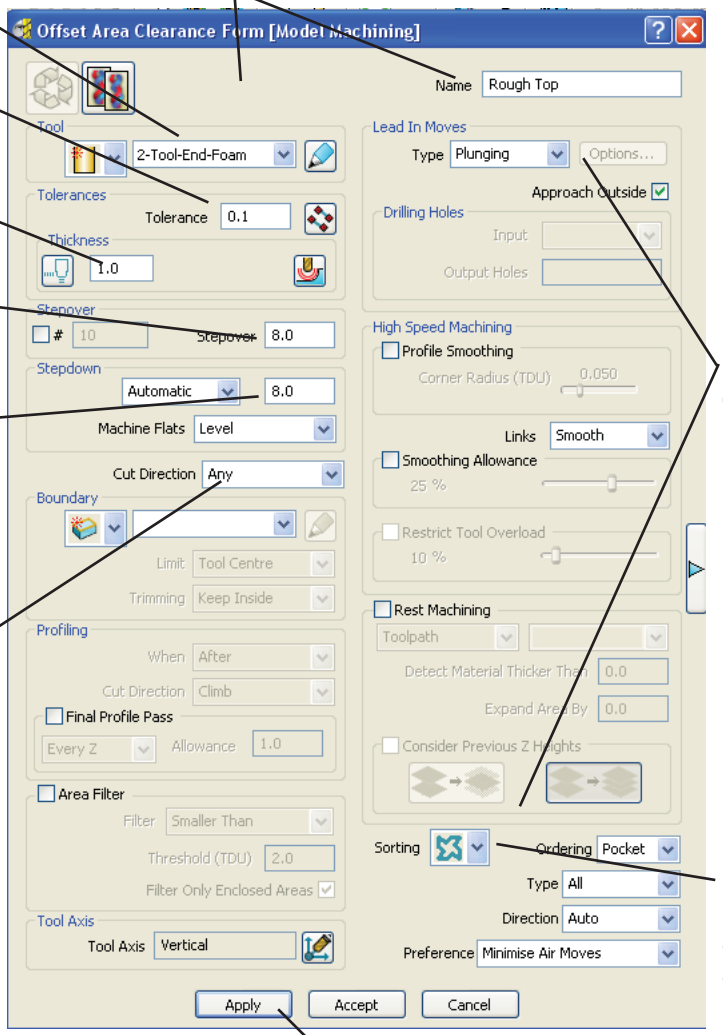
Stepover: How much the bit will shift over per pass. Typ. 2/3 the dia of the bit

Stepdown: How much the bit will cut per pass.

Foam (Max cutting length of bit)

Wood(1/2 bit diameter)

Cut Direction-Set to Any If wood set to Climb



Leave all these settings as defaults

Sorting: Specifies how the bit will travel back and forth along the piece. Choose a sorting order. If unsure leave as default.

Hit **Apply** to calculate the toolpath.

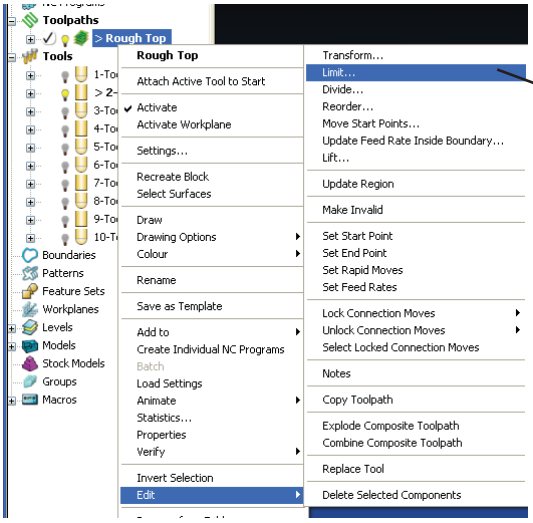
Your “Rough Top” toolpath is now listed under the toolpath tree.

If you hit Accept, the toolpath will not calculate. It will just record the toolpath settings.

Editing your Toolpath after it is created

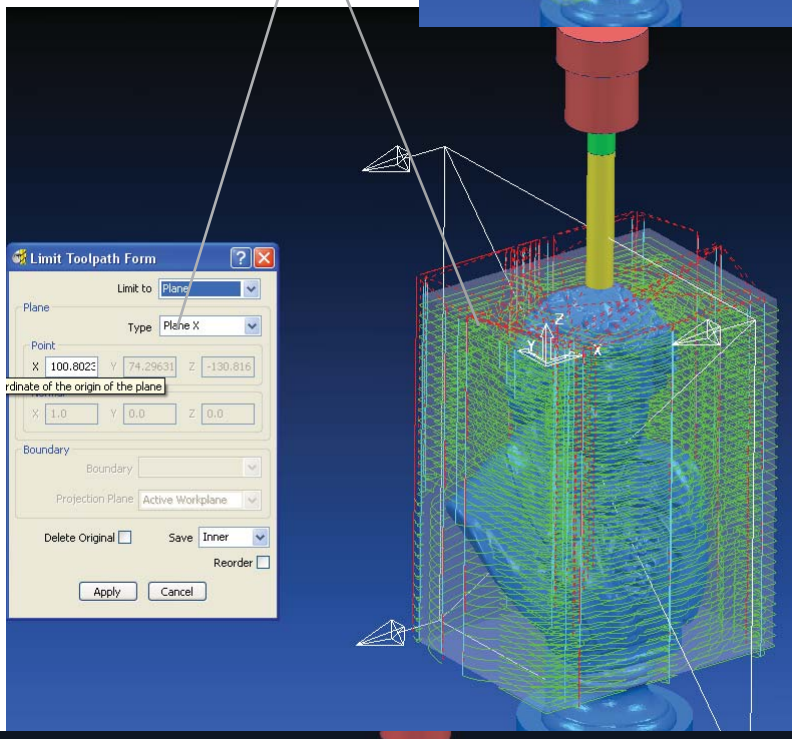
One of the biggest strengths of powermill is that you can edit the toolpath after it has been generated.

This toolpath goes the whole length of our model. I only want to cut 100mm down from the top. To do that I need to limit the depth of the toolpath.



Right click on Rough Top toolpath. Choose Edit>Limit.

A plane will appear in your window and a pop up window



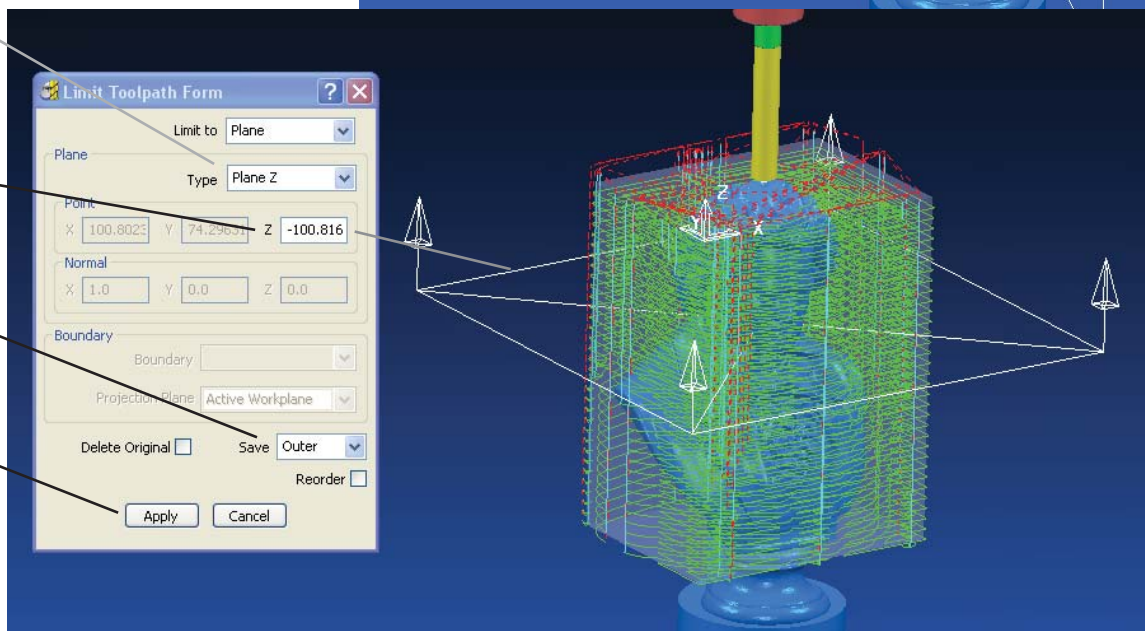
Set Type as: Plane Z

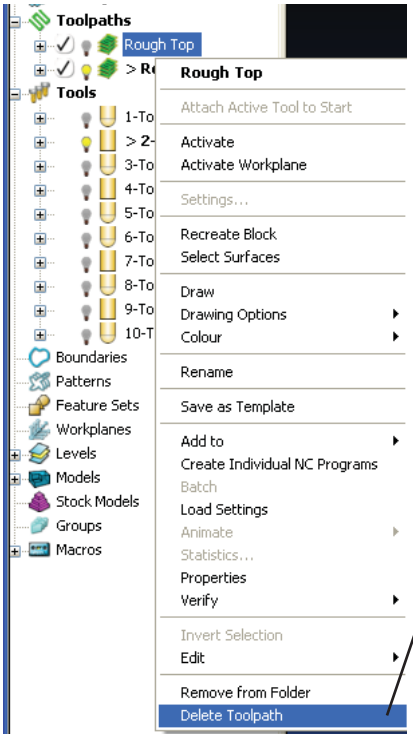
Adjust the Z height to set the depth of the limit plane.

Set the Save to Outer, so that the arrows are pointing upwards.

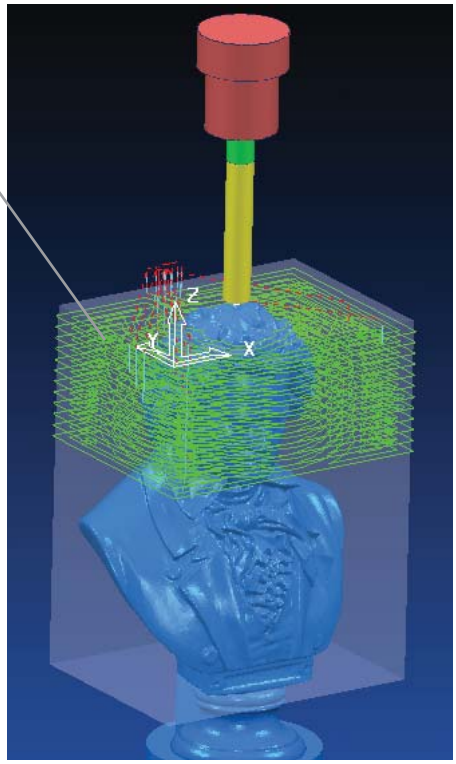
Click Apply

All Toolpaths below the Z plane will be deleted.





Toolpath now Limited

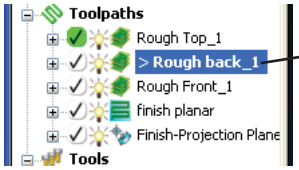


You will also see you now have two toolpaths. Simply delete the first toolpath.

Right Click on the toolpath, Select Delete Toolpath.

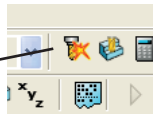
Collision Check your toolpath

When you complete a toolpath, you need to check for collisions. This looks at the tool and the object and makes sure you are not hitting anything.



Activate the Toolpath

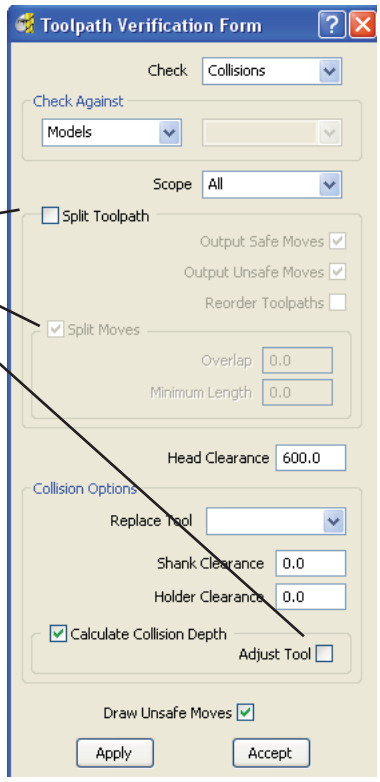
Click the Collision Check Icon



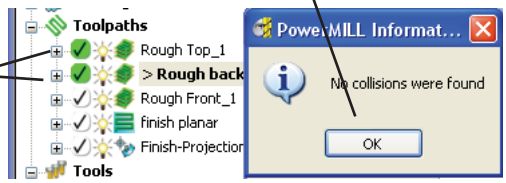
The following Window will appear.

Check off Split Toolpath, Split moves, and Adjust Tool.

Click Apply. The toolpath will be checked.



Green checkbox indicates toolpath is fine.



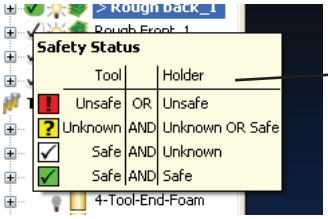
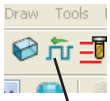


Chart Indicating the various collision check errors.

Adjusting the Feed and Speed rates for each toolpath:

The tools when loaded from the macro have some speed and feed setting assigned from the tool. However, it is always a good idea to adjust the speeds based on the type of material you are planning to cut. the tool feed and speed rates are set to each toolpath created, so you want to check each toolpath as you generate them.

To adjust the speeds, click on the Feedrates Icon at the top of the window:



This will bring up the feed rates form.

Click on the Load from active tool button

This will bring up the current feed and speed rates.

From here you can adjust the various rates.

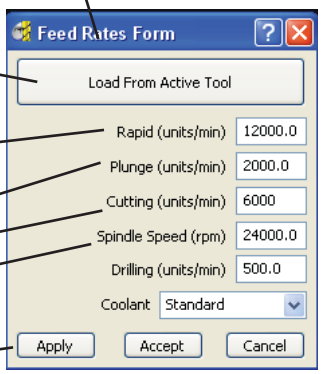
Rapid: How fast the bit travel when NOT cutting

Plunge: How fast the bit moves perpendicularly into the material

Cutting: How fast the Bit moves when cutting

Spindle: Always set to 24,000 rpm

When set, Click Apply.

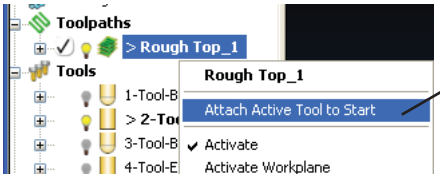


Recommended Rates:

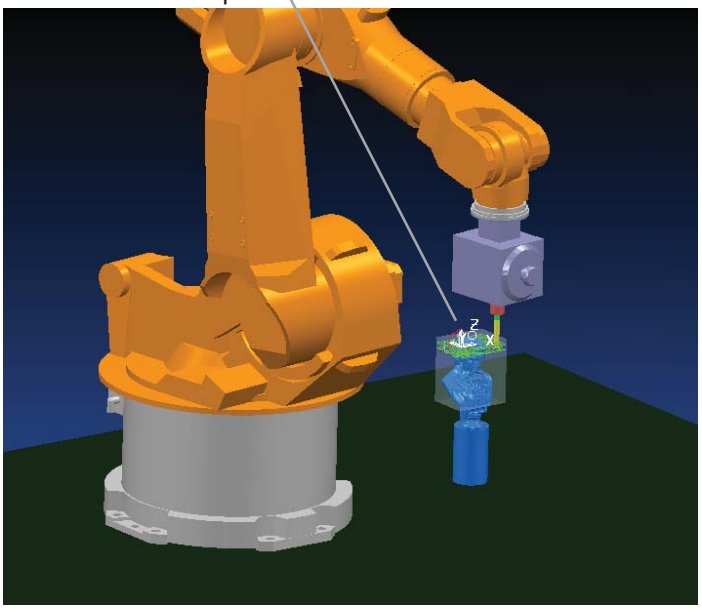
Foam:	Wood:
Rapid - 16,000 mm/min	Rapid - 16,000 mm/min
Plunge - 2,000 mm/min	Plunge - 2,000 mm/min
Cutting - 12,000 mm/min	Cutting - 8,000 mm/min

Previewing your toolpath

Powermill allows you to attach the tool to the toolpath and then play the tool.



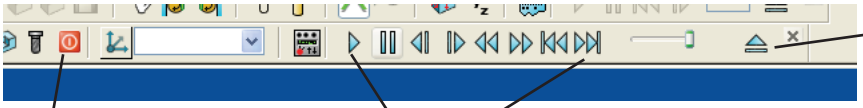
Right Click on the toolpath and choose Attach active tool to start. Your bit will jump to the starting point. If you click on the left or right arrows on your keyboard, the bit will follow the toolpath.



You can also play the bit along the toolpath, by clicking on the play stop and rewind buttons.

Running a Toolpath Simulation

Powermill will allow you to run a rendered simulation of your toolpath. To do this click on the Red Button to toggle into view mill. Then click on the Play button and you will see the tool cut the part.

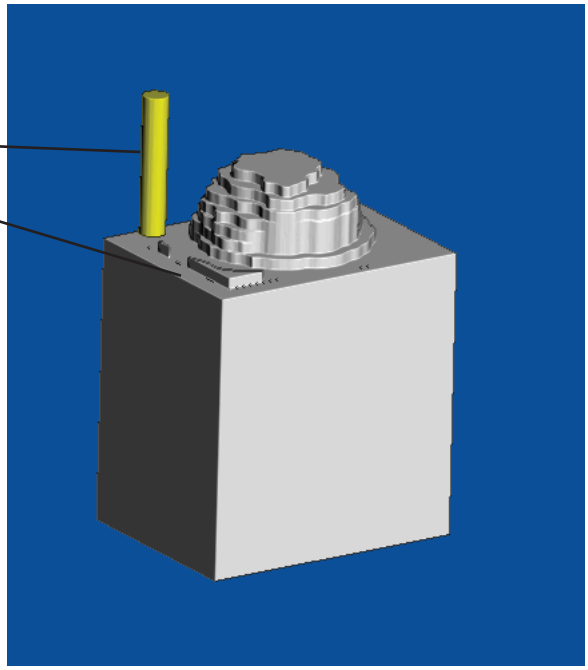


Click this button to exit out of View Mill and return to powermill.

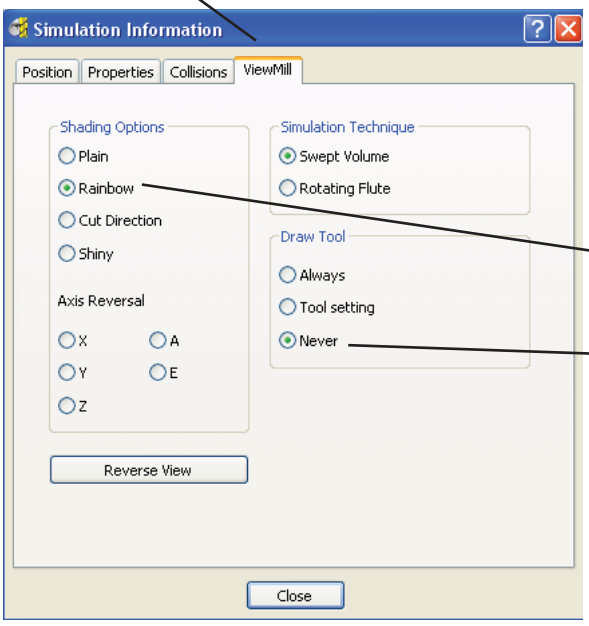
Toggle to View Mill

Click Play Stop and rewind to view the cutting.

View the Simulation



To change the simulation settings in ViewMill, click on the following Icon

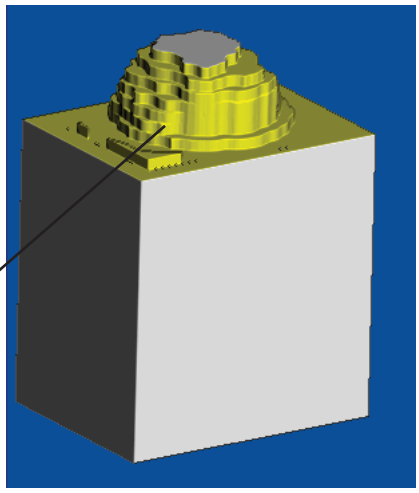


Click the Tab ViewMill, and change the following settings:

Shading Options>rainbow

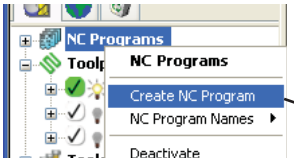
Draw Tool>Never

This will create a separate color per toolpath, and setting the tool to never will allow the simulation to run faster.



Setting Up an NC Program and Posting the Job

The last step is to create an NC program and post that program out to the CNC Mills. The NC program is where you order the various toolpaths. To create an NC program, Right Click on NC Programs, and Click on Create NC Program.



Right Click and Select Create NC Program

Give your program a Name

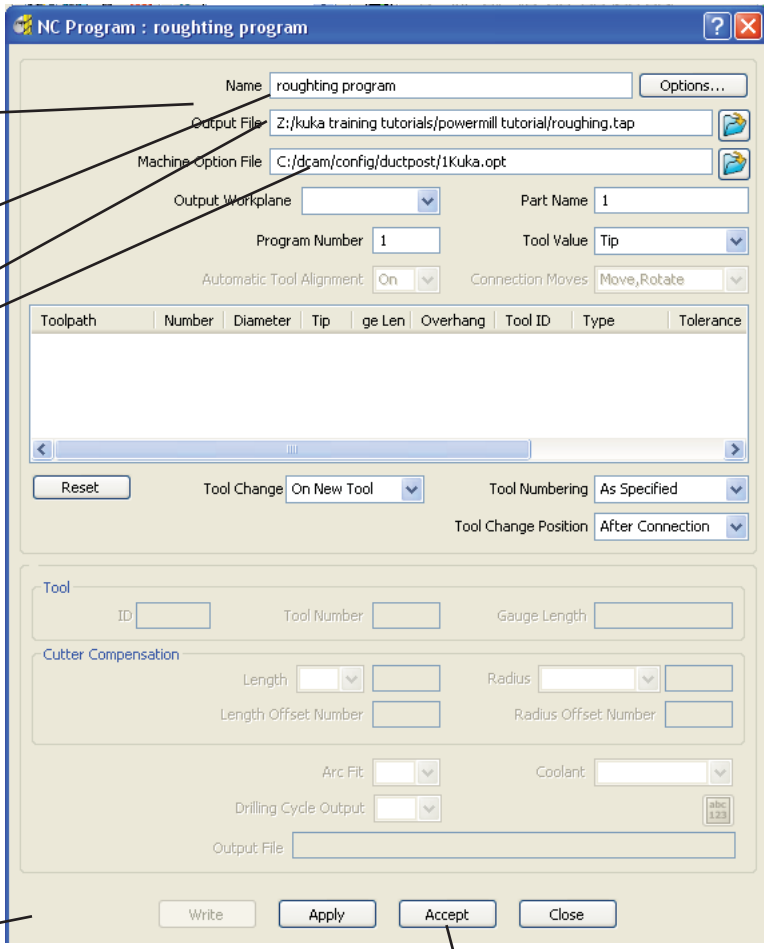
Specify where the job will post

Choose the Post type:

1 Kuka for the Robot

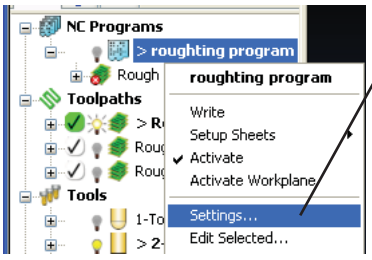
Roland for the Medium Mill

Precix for the Large Mill



Click Accept to Continue

Your NC Program will have your name. To add various toolpaths to your program, drag and drop the toolpath into the NC Program.



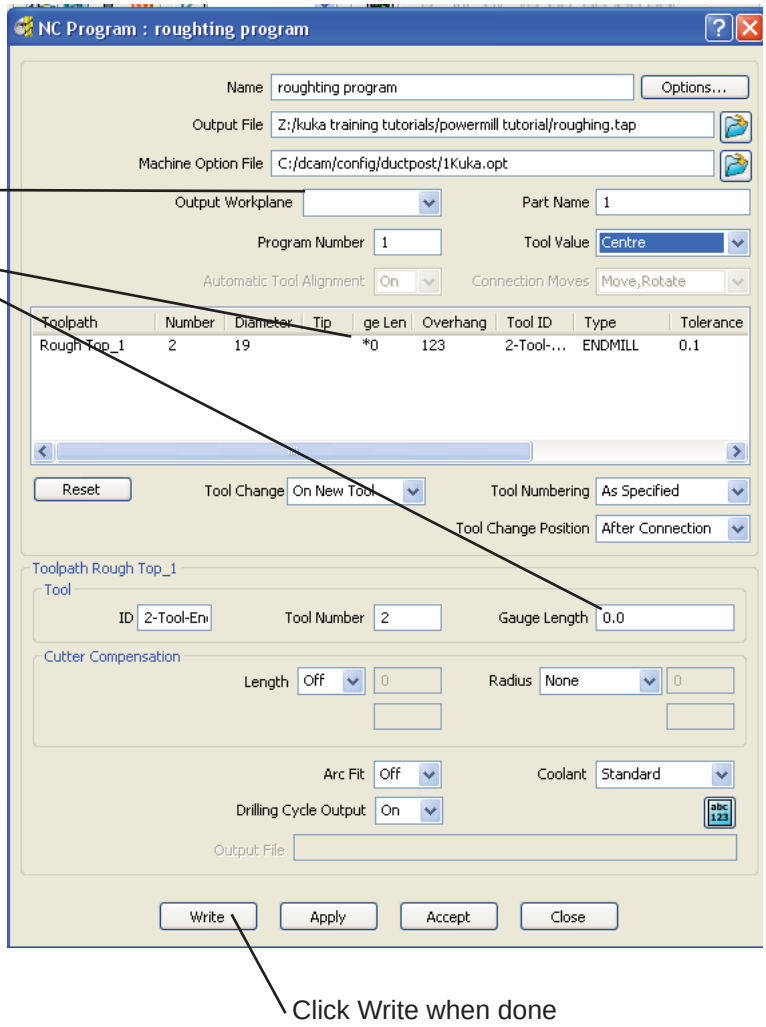
After you are finished placing your toolpaths into your program, right click on the NC program and choose Settings.

For outputting to the Robot, check the following settings:

Output Workplane: Blank

Highlight your various toolpaths and set Gauge length to: 0

When done, Click Write to create your outputted Toolpath.



Follow instruction posted at the robot for converting the posted file and loading into the robot.