

# 5 Axis Cutting Using Delcam Powermill

Written by: John Eberhart & Trevor Williams  
DM Lab Tutorial

This tutorial covers how to setup a job for a Multi-Axis Tool paths for the Robot.

**Note: You need to review the 3 Axis Powermill Tutorial before starting this tutorial. This tutorial starts where the previous tutorial left off.**

## What is Multi-Axis Milling?

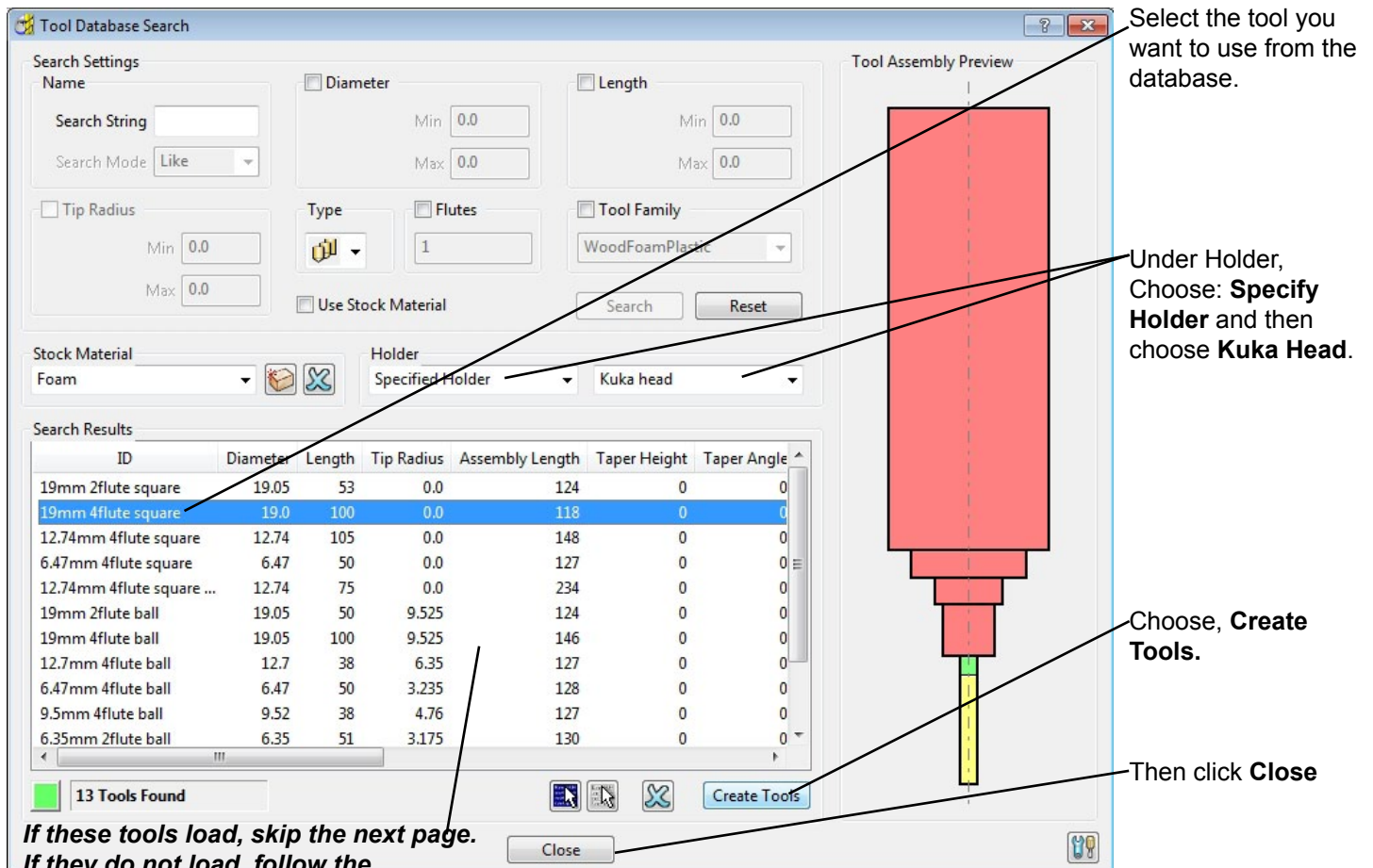
In standard 3-axis milling, which is what we do on the other mills, the tool moves in the X, Y and Z axes. 3-axis milling can also be done on the robot, where the tool stays aligned in the Z-axis and moves along the X, Y and Z axes.

In multi-axis milling (also known as 4 and 5-axis milling) using the robot, the tool does not stay aligned in the Z axis and can be moved in multiple direction relative to the Z axis. These are referred to as the A and B axes. Multi-axis tool paths can be trickier to create, but they give you the advantage of being able to do complex cuts without having to reorient your part, enabling you to do more focused finishing work and detailing that would be impossible in 3-axis milling.

## Using the Kuka Spindle Head Tool Holder:

When performing multi-directional cuts, it is a really good idea to be able to visualize the spindle and tool together. For the remaining roughing jobs, we are going to create tools with the Kuka Spindle.

To do this, right click on **Tools** and Choose: **Create Tool>From Database.**



Select the tool you want to use from the database.

Under Holder, Choose: **Specify Holder** and then choose **Kuka Head**.

Choose, **Create Tools**.

Then click **Close**.

ID	Diameter	Length	Tip Radius	Assembly Length	Taper Height	Taper Angle
19mm 2flute square	19.05	53	0.0	124	0	0
19mm 4flute square	19.0	100	0.0	118	0	0
12.74mm 4flute square	12.74	105	0.0	148	0	0
6.47mm 4flute square	6.47	50	0.0	127	0	0
12.74mm 4flute square ...	12.74	75	0.0	234	0	0
19mm 2flute ball	19.05	50	9.525	124	0	0
19mm 4flute ball	19.05	100	9.525	146	0	0
12.7mm 4flute ball	12.7	38	6.35	127	0	0
6.47mm 4flute ball	6.47	50	3.235	128	0	0
9.5mm 4flute ball	9.52	38	4.76	127	0	0
6.35mm 2flute ball	6.35	51	3.175	130	0	0

13 Tools Found

Create Tools

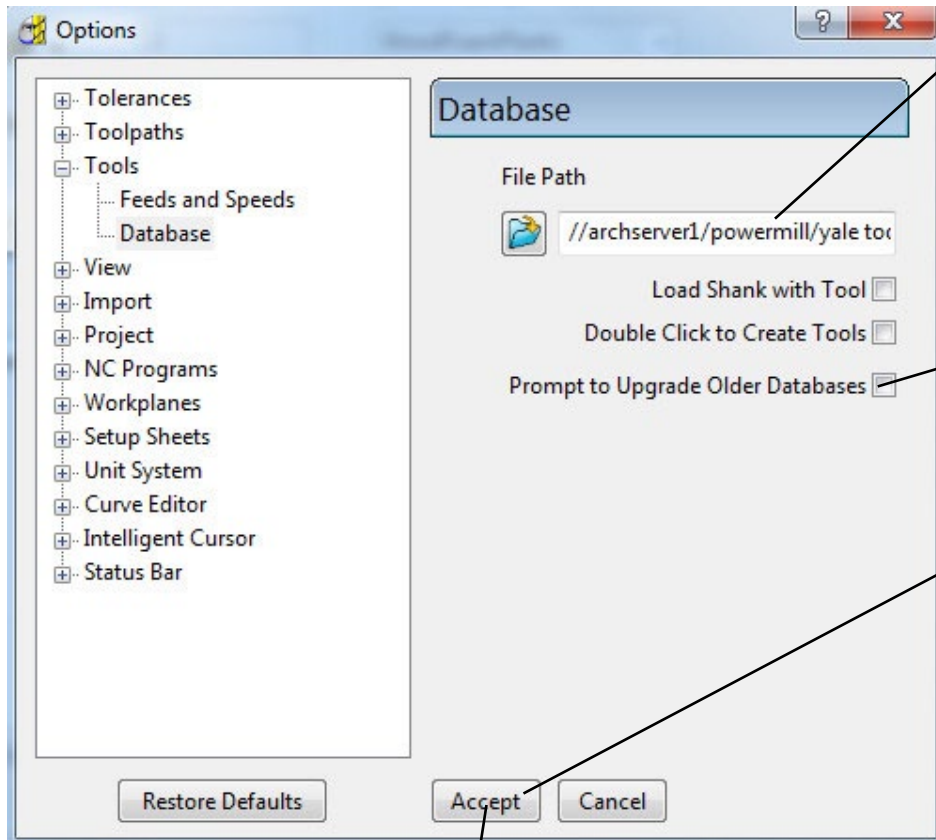
Close

**If these tools load, skip the next page.  
If they do not load, follow the instructions on the next page**

**If these tools load, skip this page. If they do not load, follow the instructions on this page to load the tool database.**



Click on this icon in the lower corner of the tool database search window to open the tool database options.



Delete the current pathway and type in the following pathway:

**//archserver1/powermill/yale tools/  
tool\_database.mdb**

Note this needs to be typed in exactly.

Uncheck the option:  
*Prompt to Upgrade Older Databases.*

Click Accept.

**Wait a few seconds, while Powermill connects to and load the tools database. Note this can take 20-30 seconds to complete.**

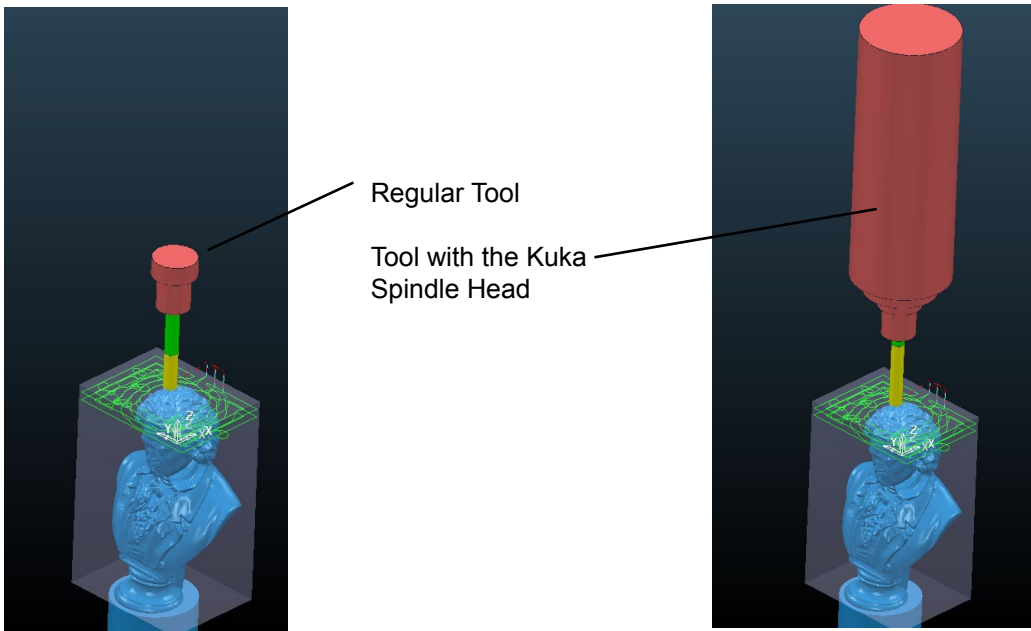
This will take a while to complete 20-30 seconds

ID	Diameter	Length	Tip Radius	Assembly Length	Taper Height	Taper Angle
19mm 2flute square	19.05	53	0.0	124	0	0
<b>19mm 4flute square</b>	<b>19.0</b>	<b>100</b>	<b>0.0</b>	<b>118</b>	<b>0</b>	<b>0</b>
12.74mm 4flute square	12.74	105	0.0	148	0	0
6.47mm 4flute square	6.47	50	0.0	127	0	0
12.74mm 4flute square ...	12.74	75	0.0	234	0	0
19mm 2flute ball	19.05	50	9.525	124	0	0
19mm 4flute ball	19.05	100	9.525	146	0	0
12.7mm 4flute ball	12.7	38	6.35	127	0	0
6.47mm 4flute ball	6.47	50	3.235	128	0	0
9.5mm 4flute ball	9.52	38	4.76	127	0	0
6.35mm 2flute ball	6.35	51	3.175	130	0	0

13 Tools Found

Create Tools

Your tools will now be loaded



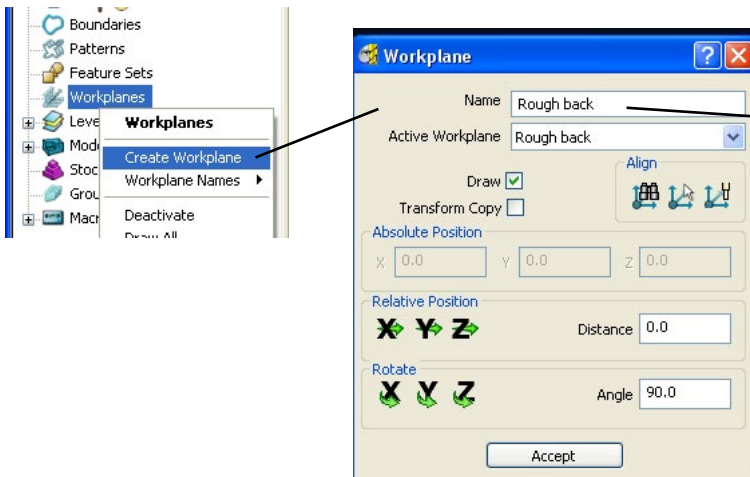
### Setting a new work plane for a sideways cut

The robot will always align its spindle axis along the Z axis and the tip of the spindle always points towards the origin.

In Powermill to rotate the spindle, you create a new reference plane and rotate the new reference plane so the Z axis and thus the robot arm will rotate to align with the new reference plane and corresponding Z Axis position.

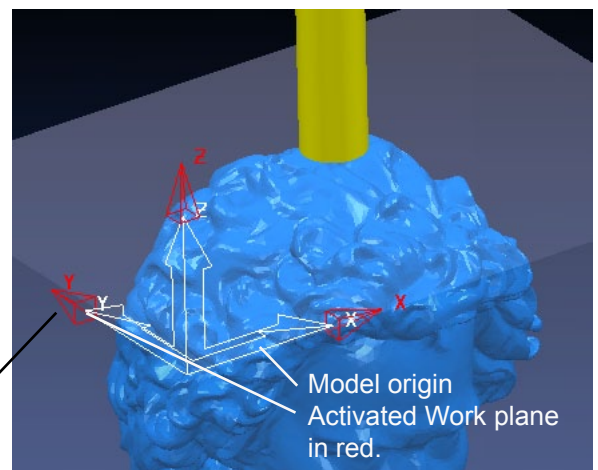
Creating a New Work plane:

Right Click on **Work Planes** and select **Create Work plane**. The following window will appear.



Name your work plane and then set it as active by choosing it in the active work plane drop down.

The new work plane will show up in red.



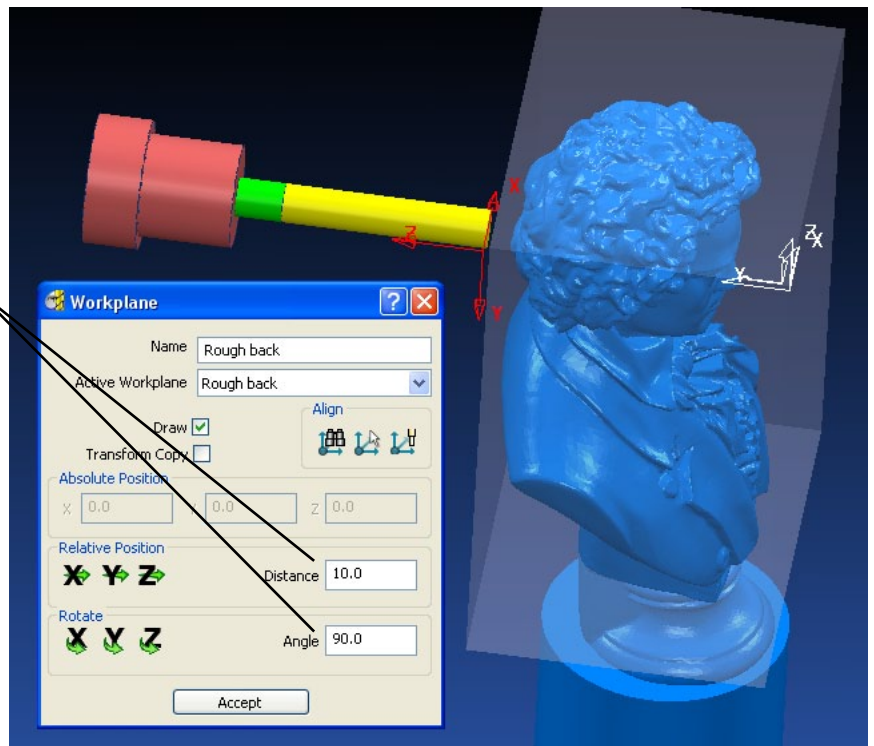
## Adjusting the work plane using the settings window.

Using the pop up window, you can move the position and rotation of the active work plane by inputting a value in millimeters and clicking on the appropriate translation or rotation buttons.

Using the input box, I rotated the work plane so the Z axis, and thus the tool is pointing towards the back. I also have the work plane's X and Y axis so they are on the surface and to the back of the box.

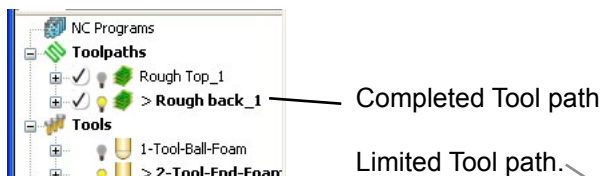
If these axes were in the middle, the tool would start at the middle of your piece.

*Note: If the block is moving with the tool, then you did not set the block's coordinate system to "Global Transform"  
(See the 3 Axis tutorial on how to do this)*

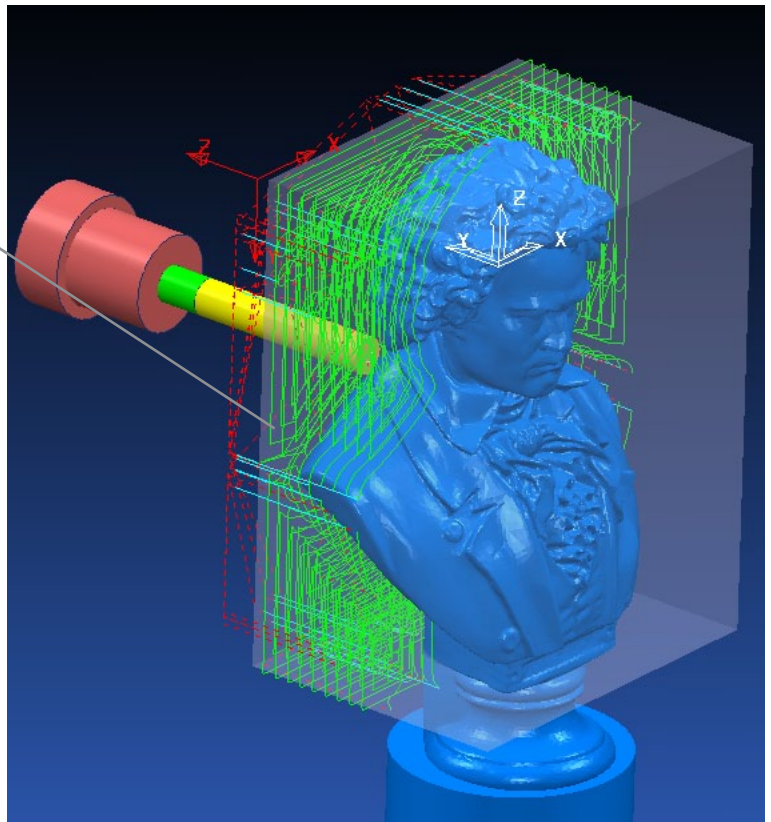
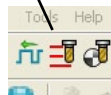


Perform a Rough Cut on the back of the piece.

Click on the machining strategies button, and using the same settings in the 3 Axis tutorial, I generated a rough cut on the back of the model. In addition, I limited the depth of the cut using the Limit Tool path method. To limit the tool path, right click on the tool path, and select Edit>Limit



Note: remember to reset your safe Z heights before calculating the tool path.





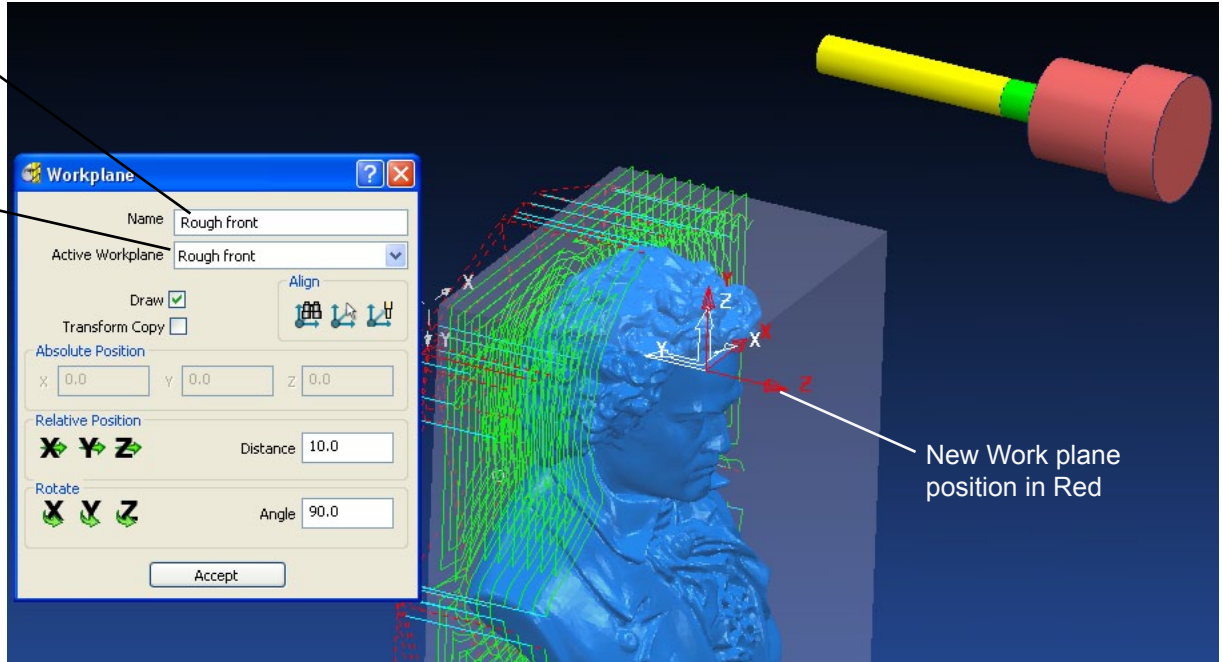
## Roughing the Front of the Piece:

To rough the front of the model, we are going to create a new Work plane, then move and rotate the work plane so that the tool is pointing to the front of the model. Make sure none of the work planes you already created are active. If they are, right click and uncheck activate.

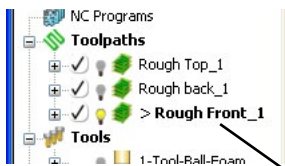
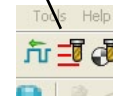
New Work plane  
(Rough Front)

Activate the new  
work plane

Using the move  
and rotate  
controls, position  
the new work  
plane to the front  
of the model and  
make sure the tool  
is pointing towards  
the model.

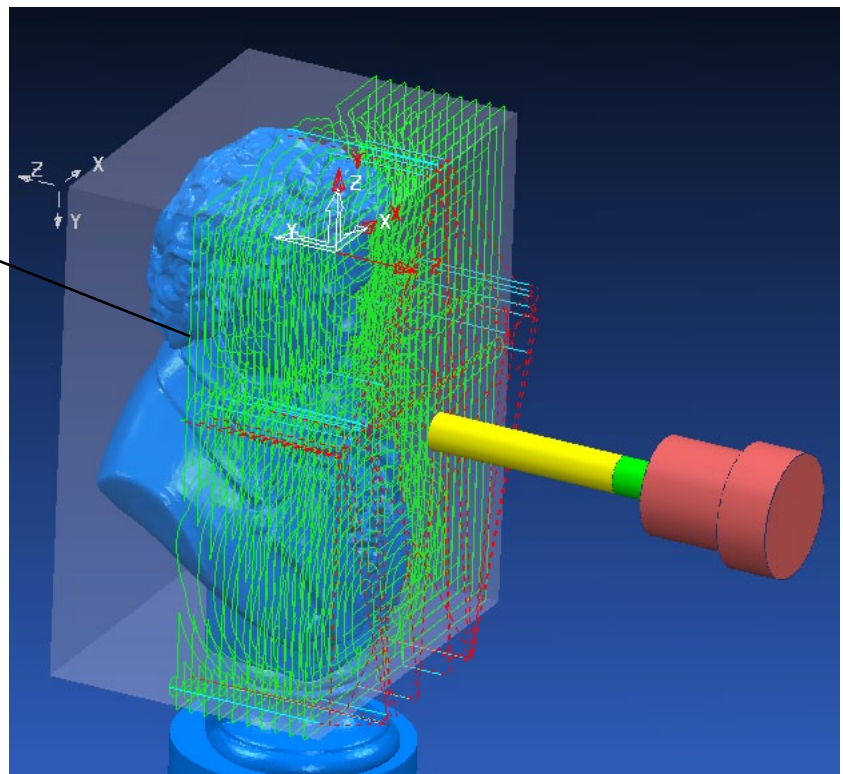


Note: remember to reset your safe Z heights before calculating the tool path.



Completed Tool path

Limited Tool path.



## Setting up tool moves to control the Robot Arm from the back to the front:

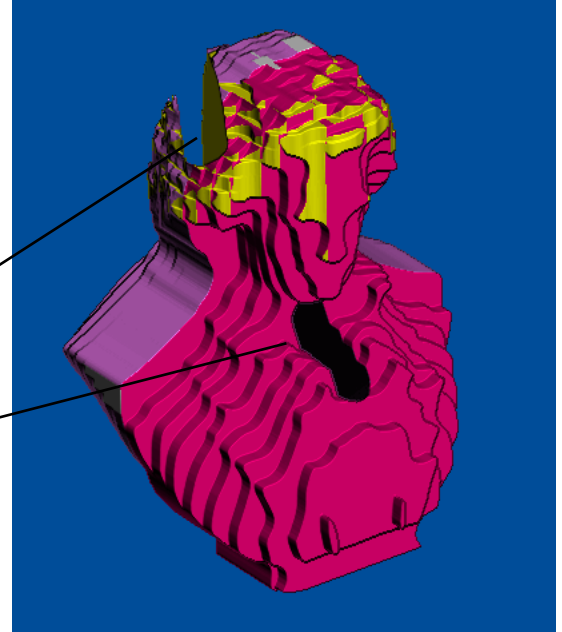
Robots by definition are stupid. Currently if we were to cut this job, the robot would finish the back and then take the shortest path to the front of the model and start cutting the front. That means the robot's arm and spindle would move through the part destroying it in the process.

If you simulate the tool paths, there will be a hole in the middle of the model where the bit passed.

**Warning:** Having holes through your project when it is simulated is not good and needs to be addressed. See Instructions on the next page on how to fix!

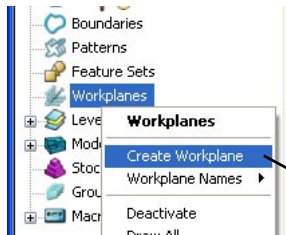
Cut in model when bit moved from top to back

Hole in model when bit passed through the model moving from the back to the front



In order to control how the arm moves around your model, you need to create a series of work planes and those work planes are then placed in the correct move order in the NC program.

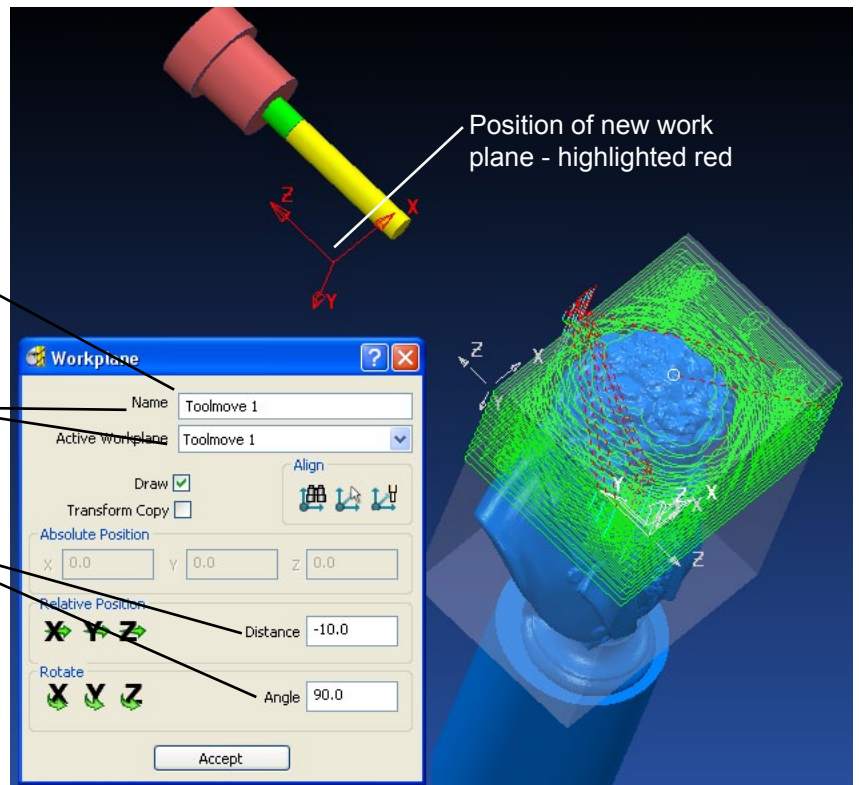
Create a new work plane. Right click on work planes and then right click and select **Create Work Plane**.



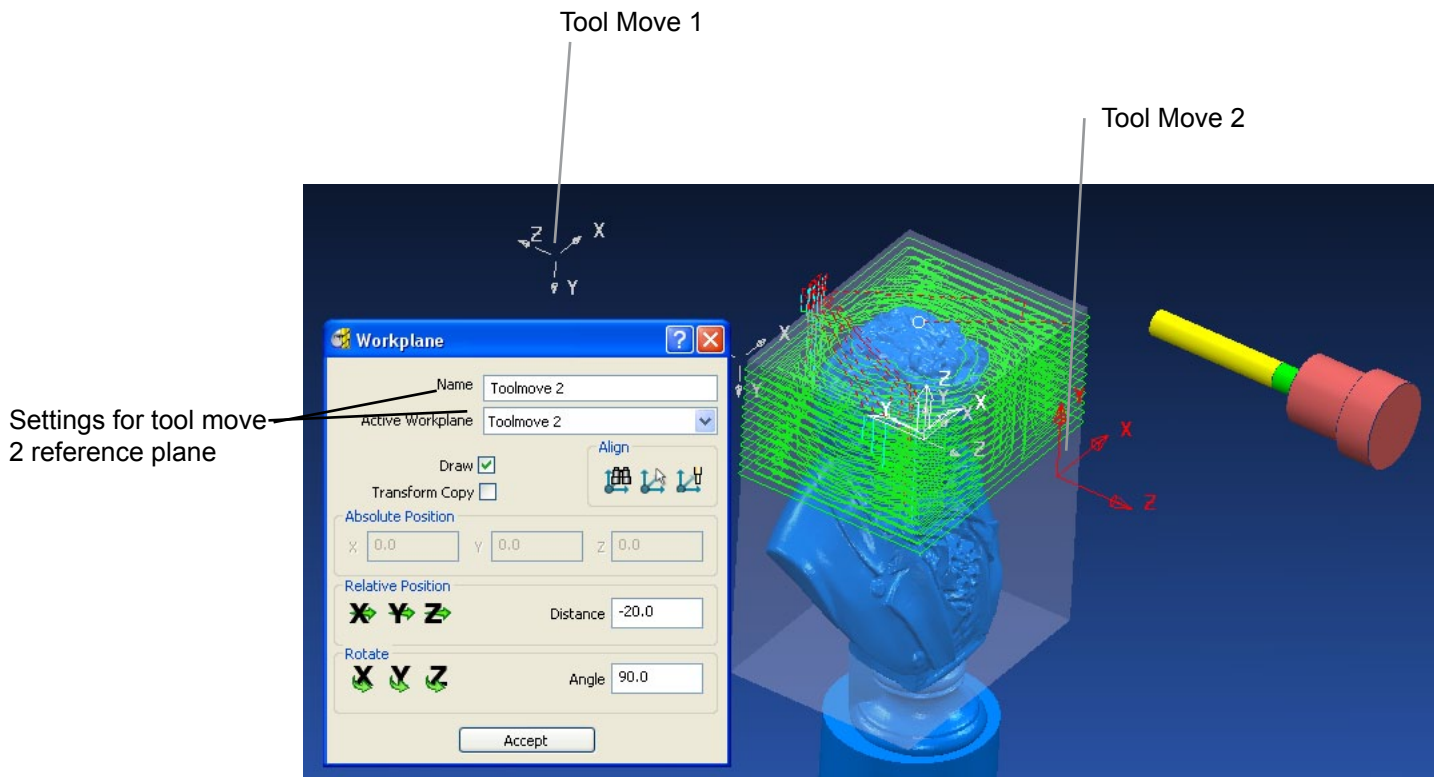
I named this work plane "Tool move 1" and made it active.

Using my Move and Rotate controls, I positioned the work plane to the top front and left side of the model.

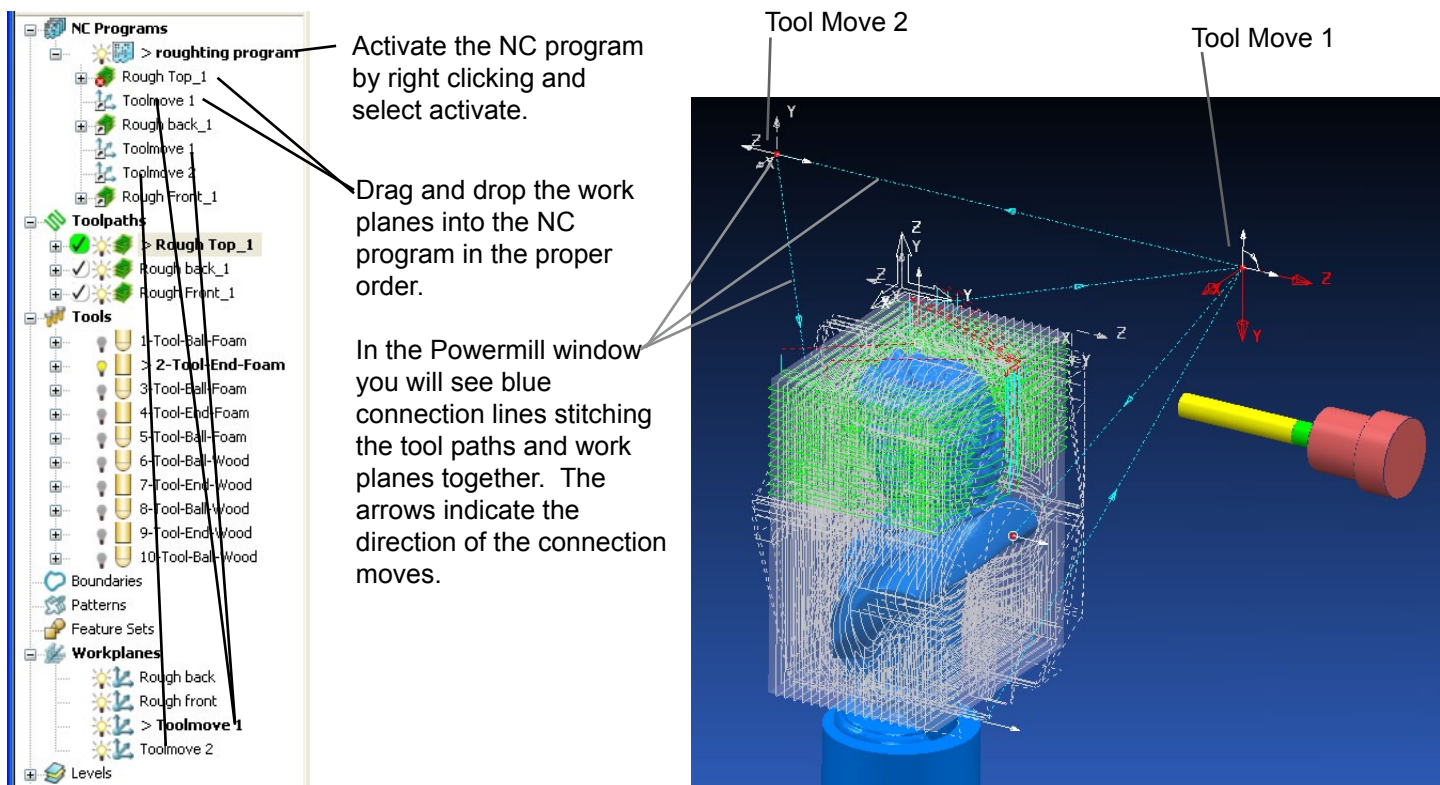
*Note: DO NOT create a work plane from an existing work plane, ALWAYS deactivate any work planes first. If you create a work plane from an existing work plane, the robot will not follow the connecting move correctly and could result in hitting your work piece, the floor, the table, or YOU!*



Repeat the process and create another tool move work plane for the front of the model



Loading the "tool Move" reference Planes into the NC program:

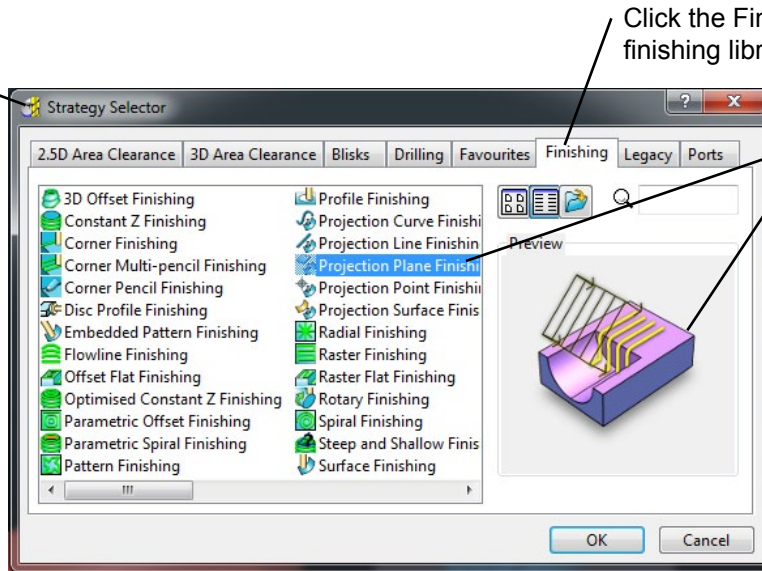
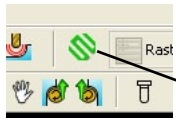


Simulate your job, verify that you have no collisions and post your Job!



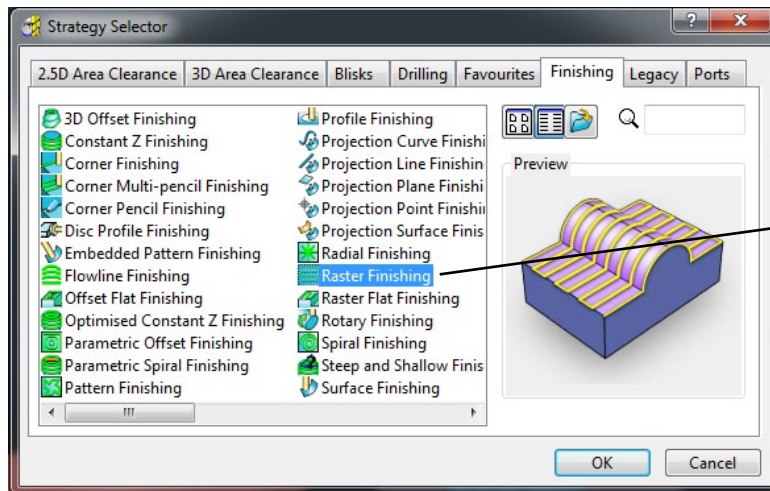
## 5 Axis Finish Cuts

Powermill Allows you to perform a number of different types of finish cuts. To perform a finish type cut, click on the tool path strategies icon and select a finish cut from the library of cuts.



Click the Finishing tab for the finishing library

Select a Finishing Strategy and a preview of how the cut will work appears

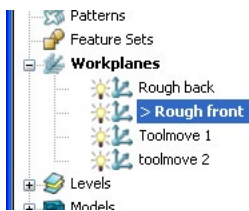


Most of the Tool path Strategies are for very specific uses. You should spend some time going through the various types of finish strategies.

Raster Finishing is most like the finishing strategy in MadCAM. Corner Pencil Finishing is the same as the Pencil cut in MadCAM.

## Applying a 5 Axis Finish Cut.

Activate the front rough work plane and click on the tool path strategies button.





The Raster Finishing settings window appears.

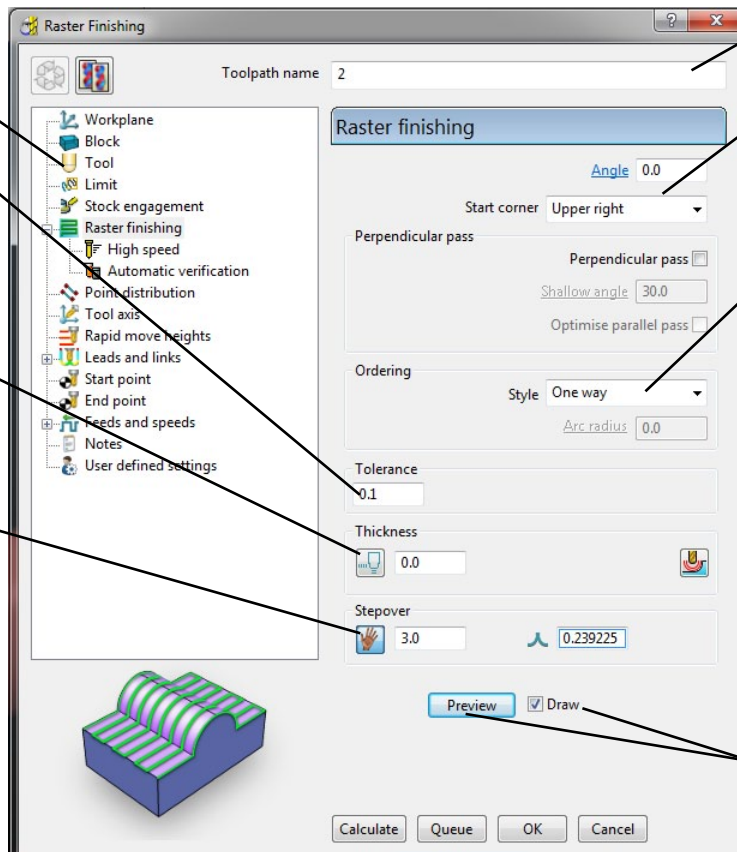
Choose the bit you want to use. In this case I am using a 1/2" dia ball nosed bit.

Leave our tolerance at 0.1

Your offset thickness should be at 0.0

Step over will be determined by the finish quality. Smaller the step over, the smoother the finish, but it takes longer.

I recommend you do a step over of 1-3mm

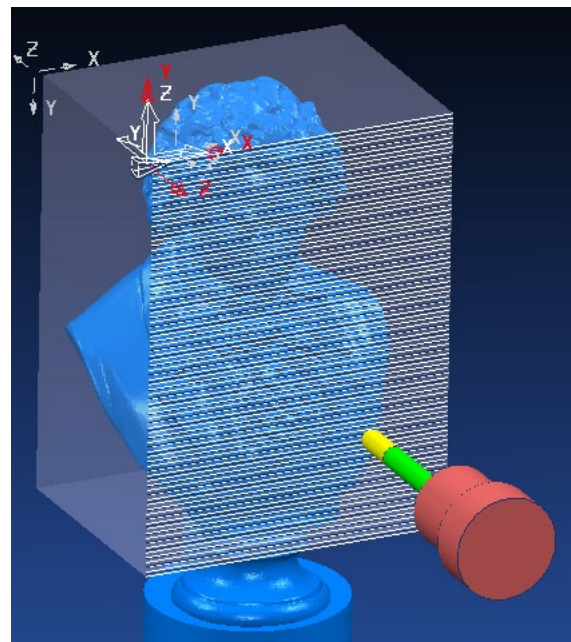


Name your tool path (finish planar)

Choose a starting corner.

You can adjust the ordering of the passes. If you are cutting wood, you want to choose one way. If you are cutting foam, both ways is fine.

Clicking draw and preview will show a rough idea of how the tool path will look.



Previewed Tool Path

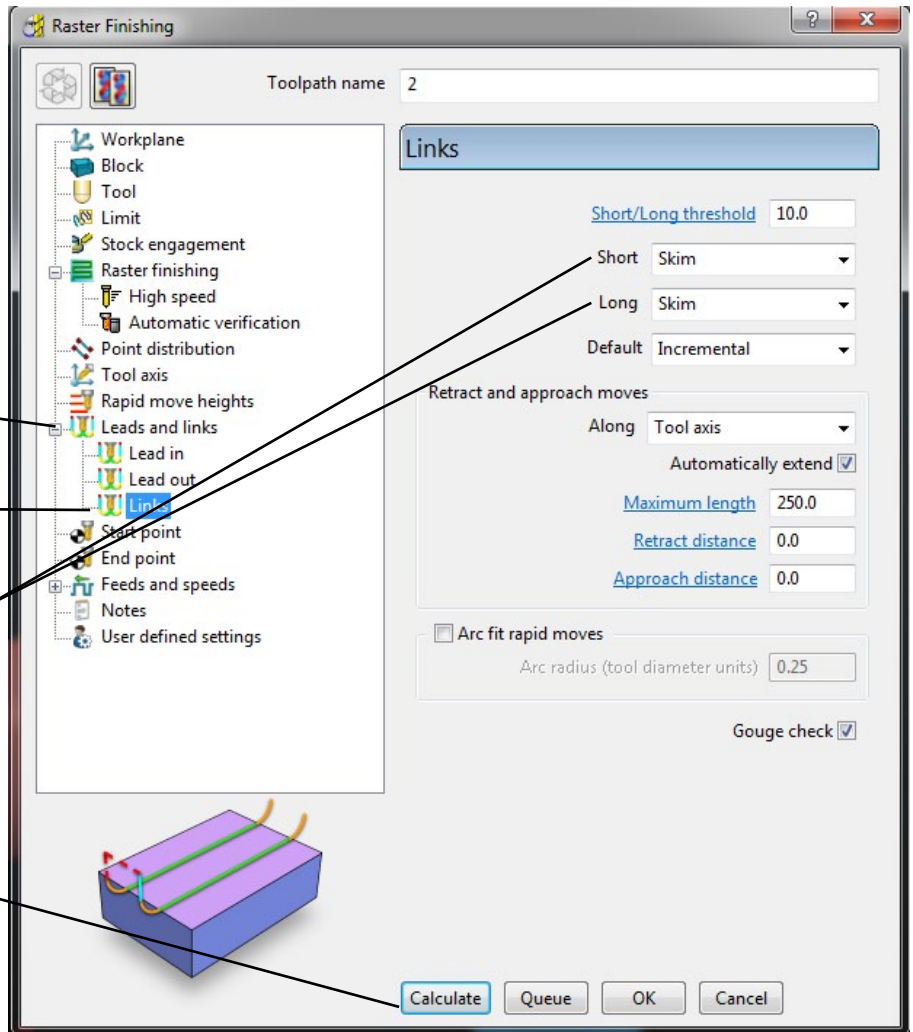
Expand the **Leads and Links** section by clicking the **[+]** button next to it.

Click on the links sub-section

Change the **Short** and **Long** items to **Skim**

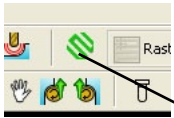
This will make the bit follow along the surface when it is moving from one cutting line to another. Without this option selected, the bit will move back to its starting position between strokes, and the cut will take significantly more time.

Finally click **Calculate** to calculate the tool path.

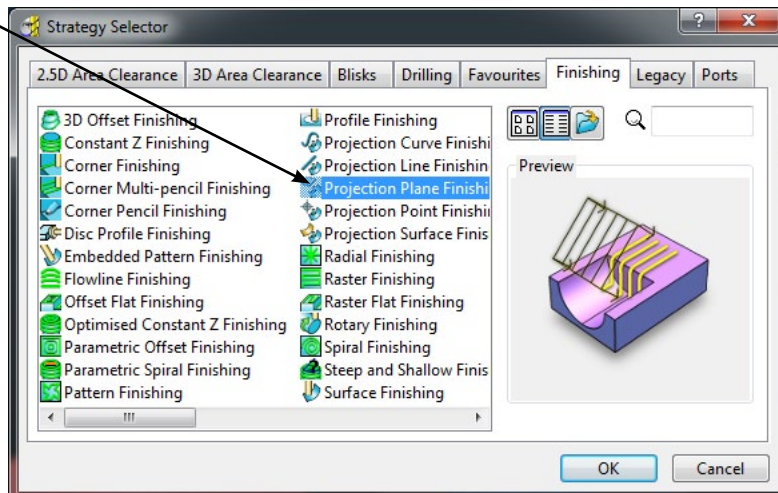


## Projected Point Full 5 Axis Finishing:

This tool path will point the bit at a specific point in space and then angle the bit as it moves back and forth across a surface. This is especially use full when you have a lot of detail where you need to angle the tool to get into.



Click on tool path strategies, then select **Projection Plane Finishing**



Set your workplane, tool, and toolpath name as usual. Set your step over to between 1-3mm.

Set your Lead in and Lead outs to Skim and on Surface like in the previous toolpath.

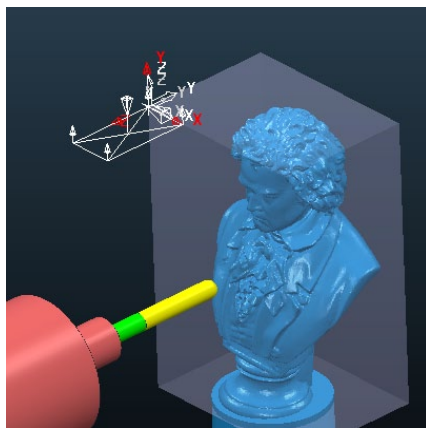
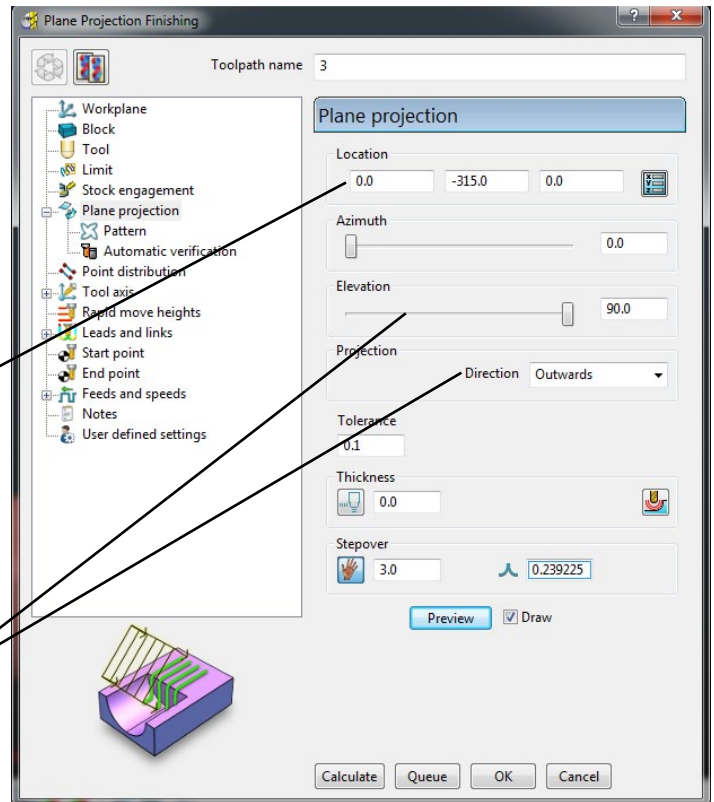
From the **Plane Projection** section, click on draw and preview to see a preview of the plane which will be projected onto the model. This is shown in the image below.

Press the **Preview** button at the bottom of the window.

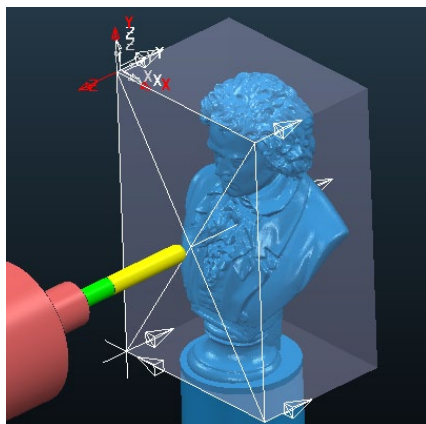
Set the **Location** values so that the corner of the plane moves from the origin to the bottom of the block. Although they're unlabeled, the data boxes are in the order of X, Y, and Z. Remember, that you are working in your *Front Workplane*, so these values should reflect that. In this case my plane has moved -315mm in the Workplane's Y-direction, which is now equivalent to the World Z-direction. This places it at the bottom of my 315mm tall block.

Change the **Direction** to **Outwards** and the **Elevation** rotation value to 90.

When that's done, go to the **Pattern** sub-section below **Plane Projection**.



Initial position of projection plane

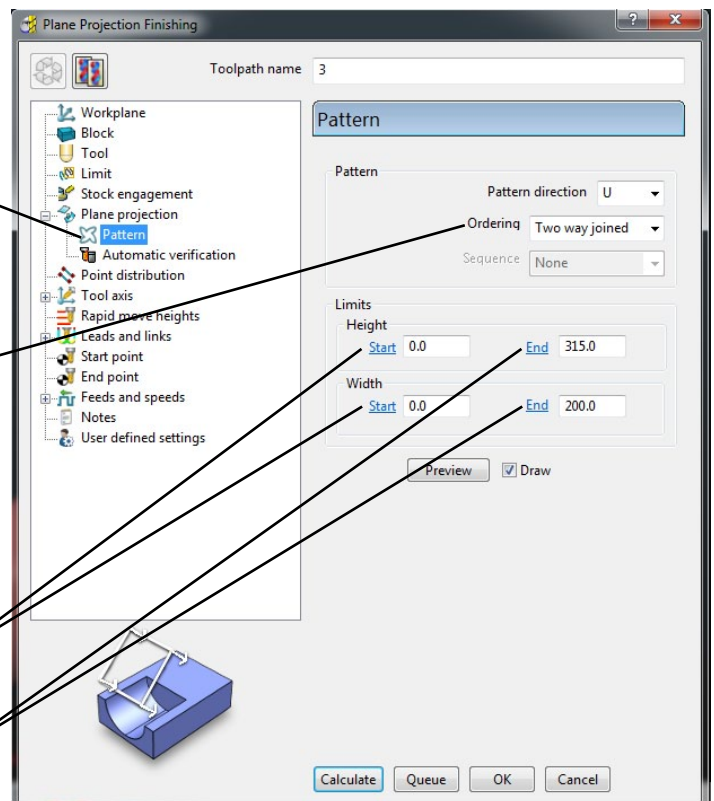


Modified position of projection plane

Set the **Ordering** value to **Two Way Joined**. This means that tool will move in a continuous zig-zag, cutting across the face of the material. This will be the fastest method of cutting.

Set the **Height** and **Width** Starts to 0.

Set the **Height** and **Width** Ends to your block limits.





Go to the **Tool Axis** section and change the **Tool Axis** value from **Vertical** to **Lead/Lean**.

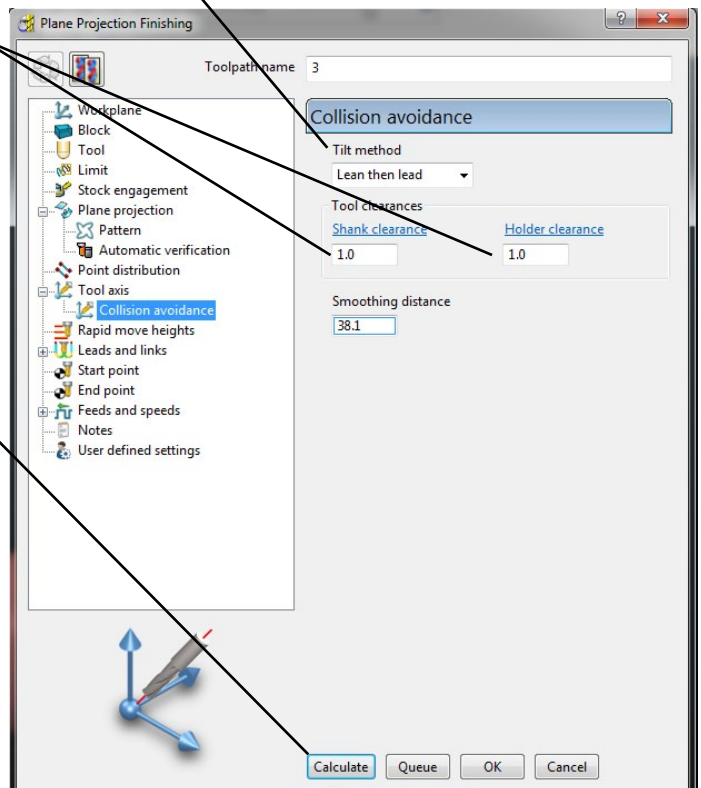
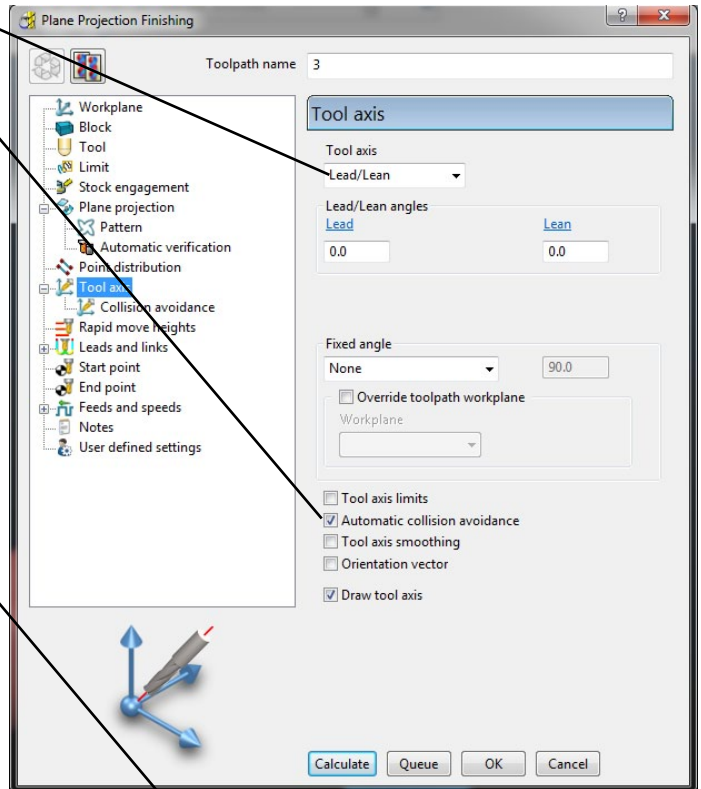
Then, check the box next to **Automatic collision avoidance** to enable that sub-section item.

Check the box next to **Draw Tool Axis** as well, so that in the preview window, you can see the axis that the tool will be using. By default, this will be vertical (in the workplane's Z-axis) but when the tool would reach a point on the surface that would cause it, the shank, or the holder to hit the model, it will automatically change the angle of the tool to avoid it. This enables the tool to reach into portions of the model otherwise inaccessible to standard 3-axis mills.

At this point, you can expand the **Tool Axis** section and go to the **Collision Avoidance** sub-section. Select the **Lead then Lean** Tilt method. This means that when the tool is about to hit an obstacle in the path it will first angle itself parallel to the toolpath (*Leading* ahead of the tool) then, if it's still about to approach an obstacle it will rotate the tool perpendicular to the toolpath (*Leaning* away from the obstacle).

The **Shank** and **Holder** clearances are the minimum number of millimeters that the tool will stay away from the surface. This should be no less than 1 mm greater than the remaining stock from your roughing pass. Whatever your tolerance was before, add 1 to that value and put it in here.

When you have your settings finished, press the **Calculate** button to begin calculating the toolpath.

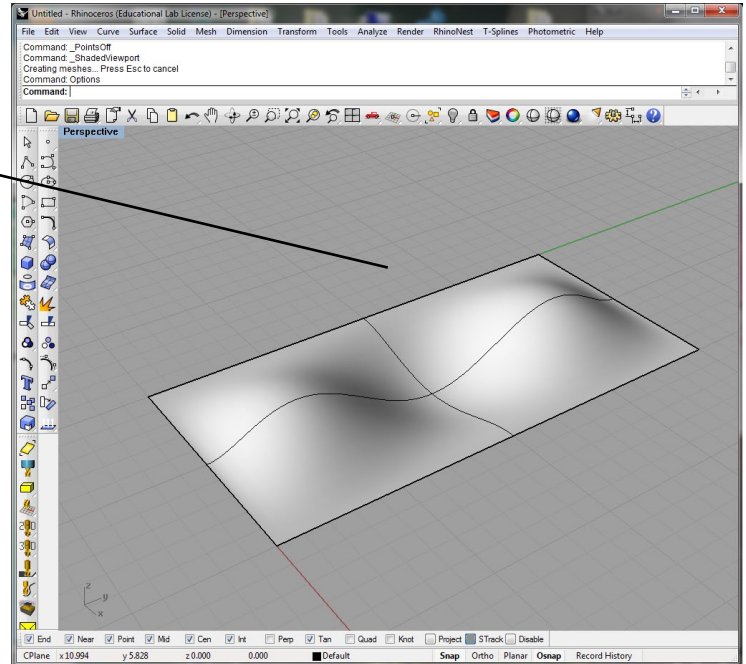


## Milling Along a Surface using the UV directional curves:

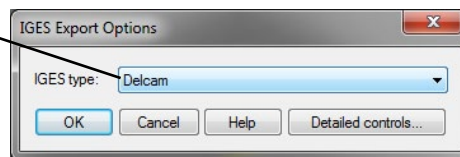
This finish tool path will follow the UV curves in the surface and use that to generate milling tool paths. This is a great way to have the bit follow the logic of the surface. However, this only works on individual surfaces and not across multiple surfaces.

Generate a surface or series of surfaces.

Select the model and export the model as an **IGES** file type.

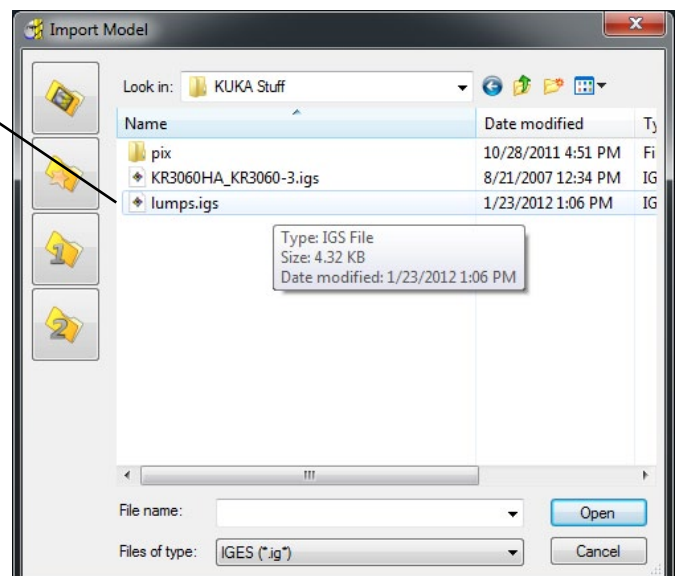
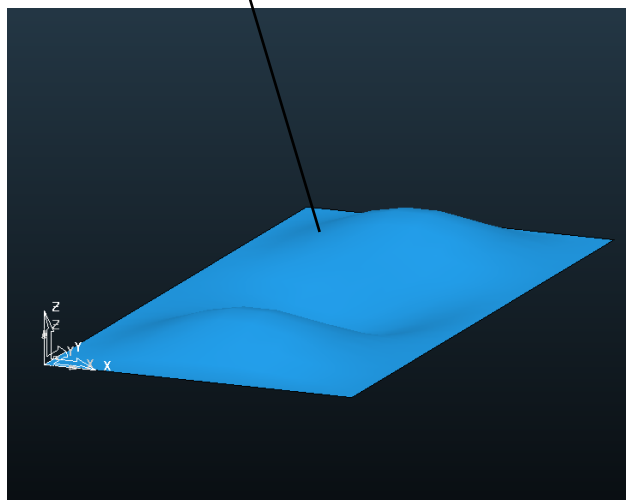


In Rhino, in the IGES export window, under IGES type, choose Delcam.

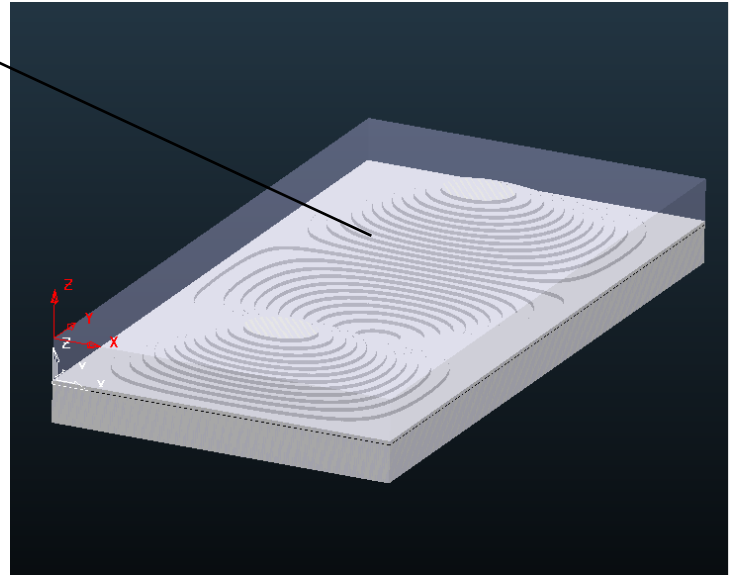


Open Powermill and Choose **File>Import Model...**  
Then select the model from the file browser.

Imported Model

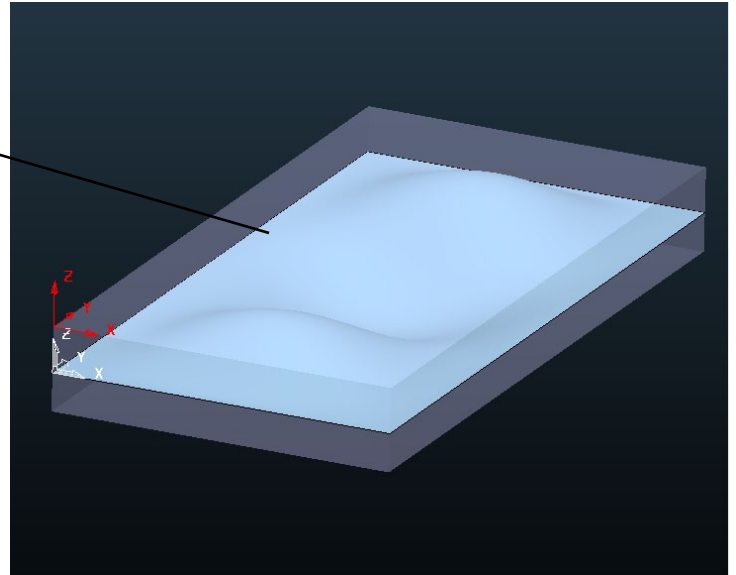


In Powermill, I have already created a bounding block and I ran a roughing path, using the same methods from earlier.

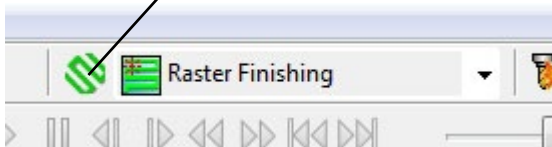


Make sure the surface you want to mill is selected in the model and then click on the tool path strategies button.

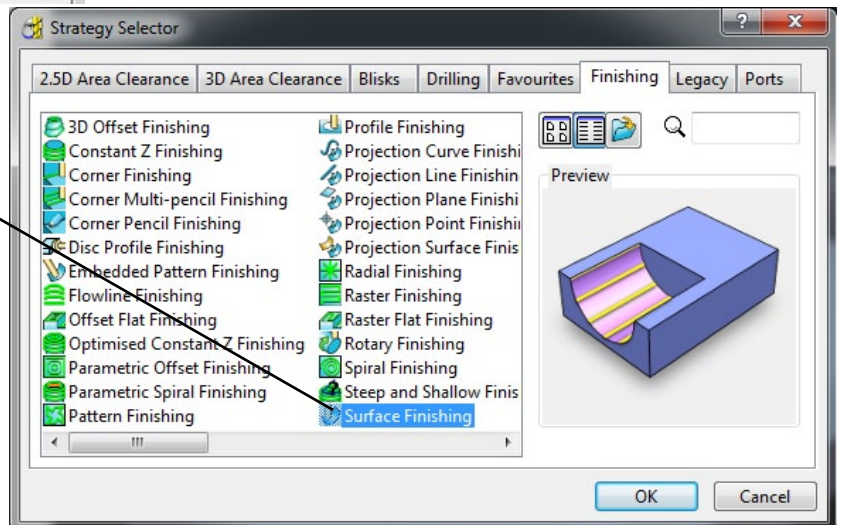
Select the individual surface you wish to mill first.



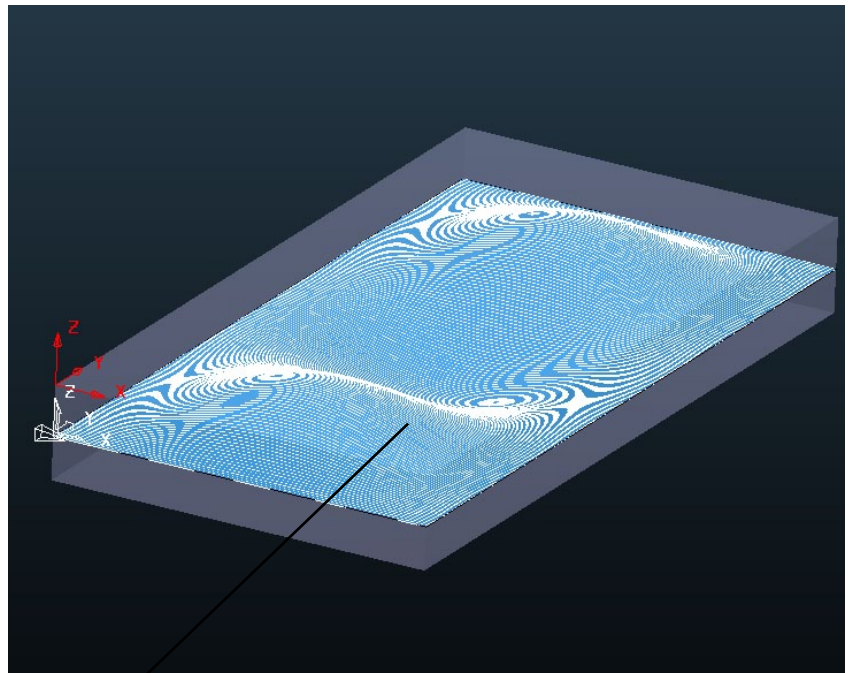
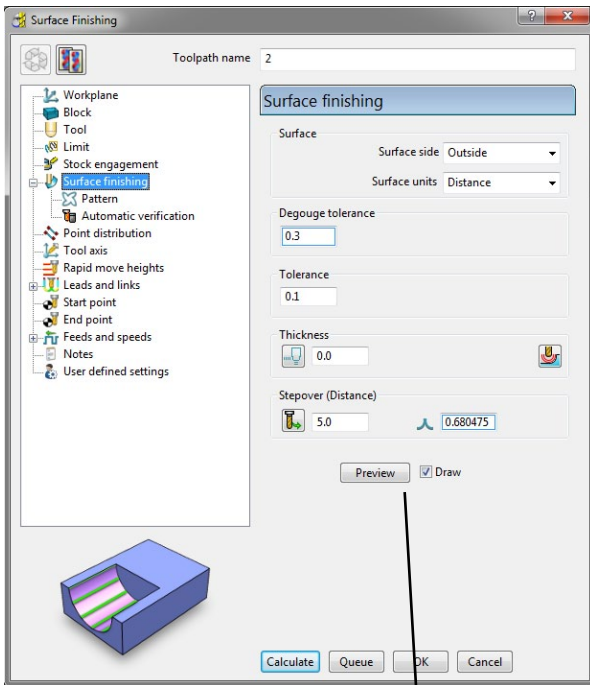
Click the Tool Path Strategies button



Click on the **Finishing** tab and choose **Surface Finishing**.





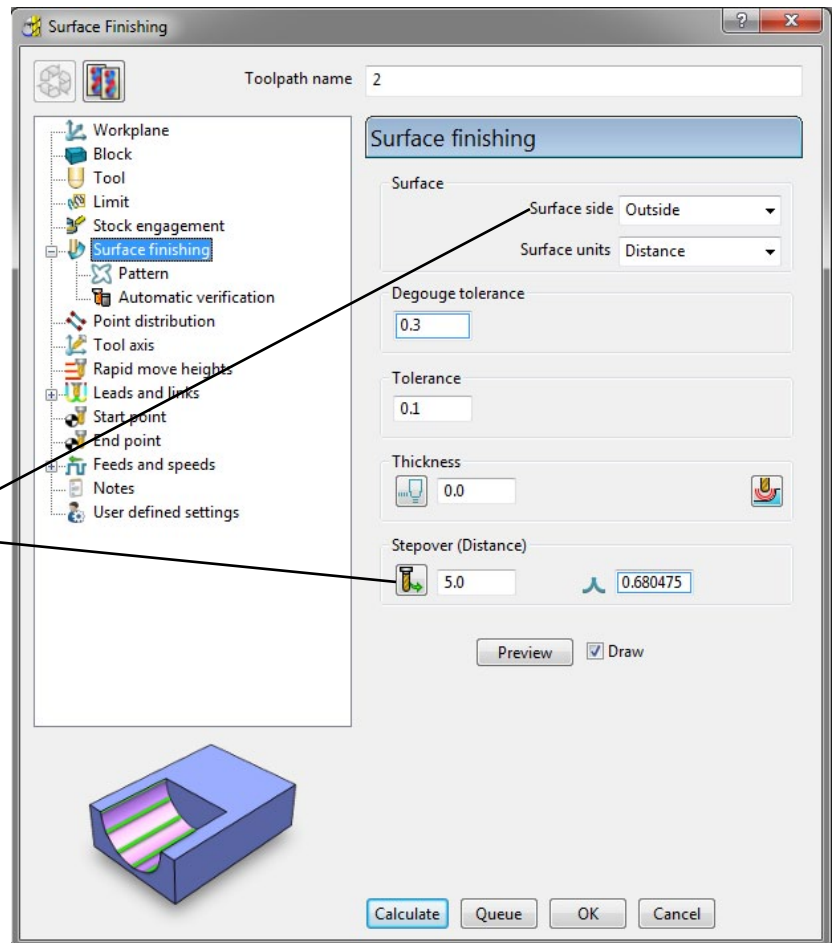


Before adjusting settings, check on the **Draw** check box and click on **Preview**.

You should see the pathway lines appear. Once you see these lines then you can adjust the settings.

When you adjust the settings, you can keep hitting preview. This allows you to see how your settings are changing on the model.

For these settings, I have adjusted the step over size. It's also important to make sure **Surface Side** is set to **Outside**, otherwise, the robot will attempt to mill from the inside surface of your model.

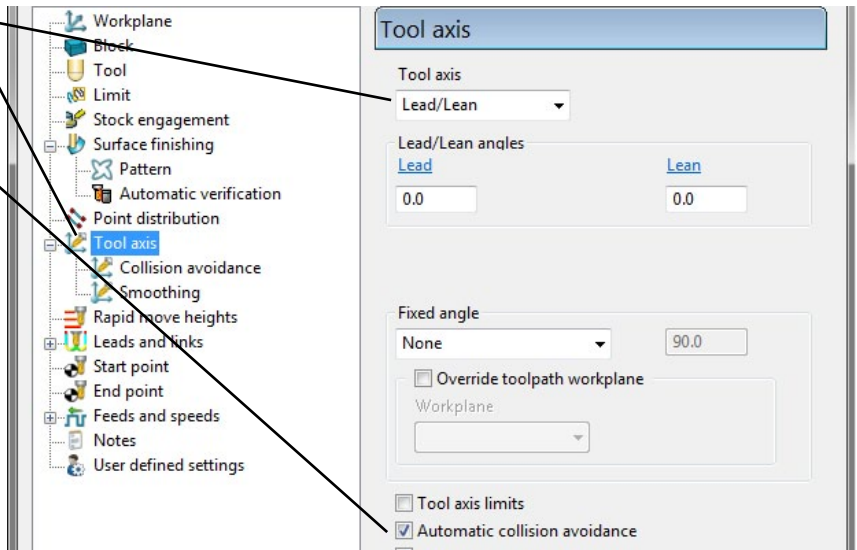


Under the **Tool Axis** section set the **Tool Axis** to **Lead/Lean**.

Make sure to enable **Automatic Collision Avoidance**.

Like before, go into the **Collision Avoidance** subsection and change the **Tilt Method** to **Lead then Lean**.

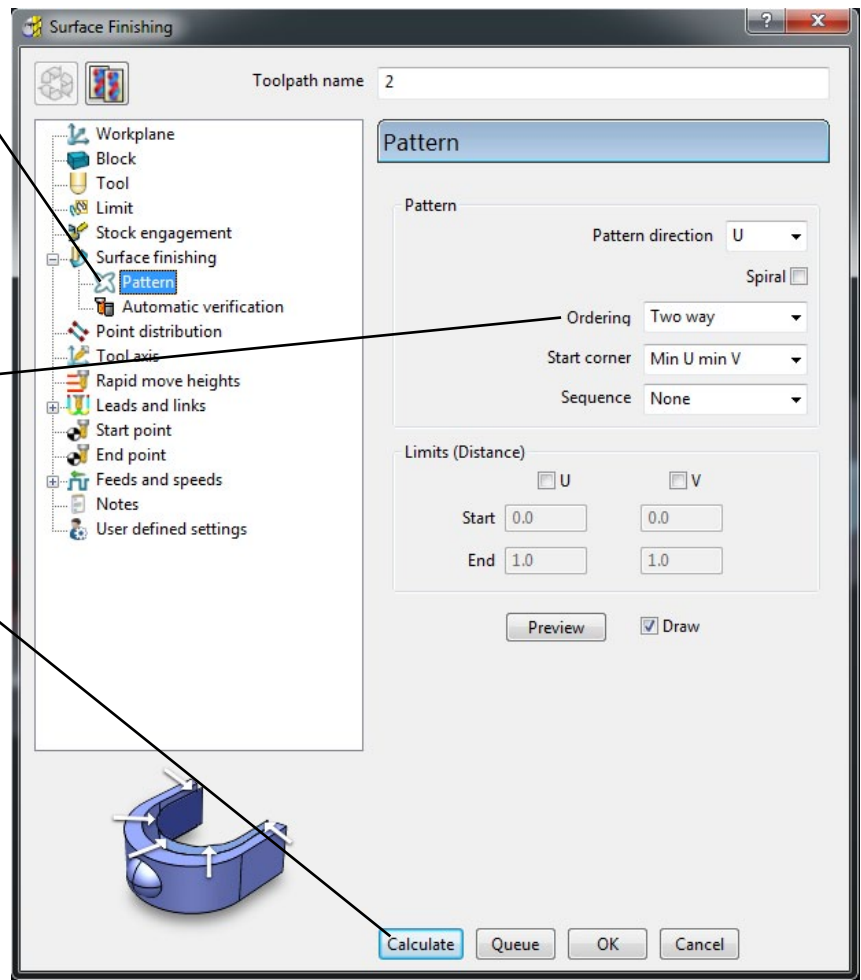
Make sure that your clearances are set to a minimum of 1 mm larger than your roughing pass's tolerance.



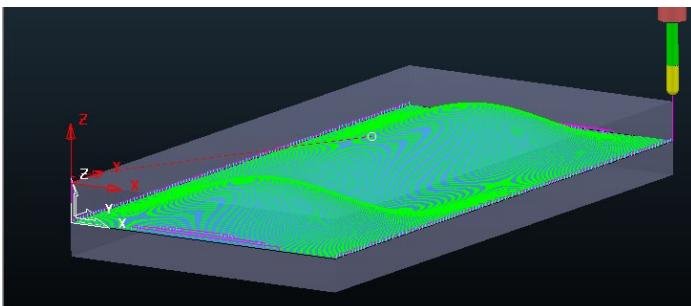
Under the **Pattern** sub-section of **Surface Finishing** you'll find the **Ordering** settings.

Make sure this is set to **Two Way** to give your model the smoothest finish quality.

When your settings are finalized, hit the **Calculate** button.



Generated Tool Path



Milled Model

