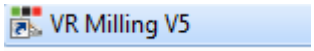
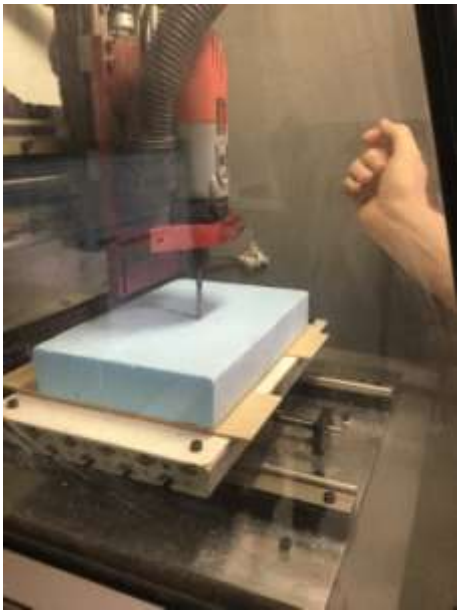
Using the Denford CNC for Composites

Working out tool bit sizes and ensuring the machine can cut the required model.

Because the material depth I would like to cut is 50mm I need to check what tool piece can do this and if it can clear the material and cut deep enough. To test this out we us the 'VR Milling V5' (shown below) to jog the tool piece to where it should go, with the material thickness and depth cut taken into account.

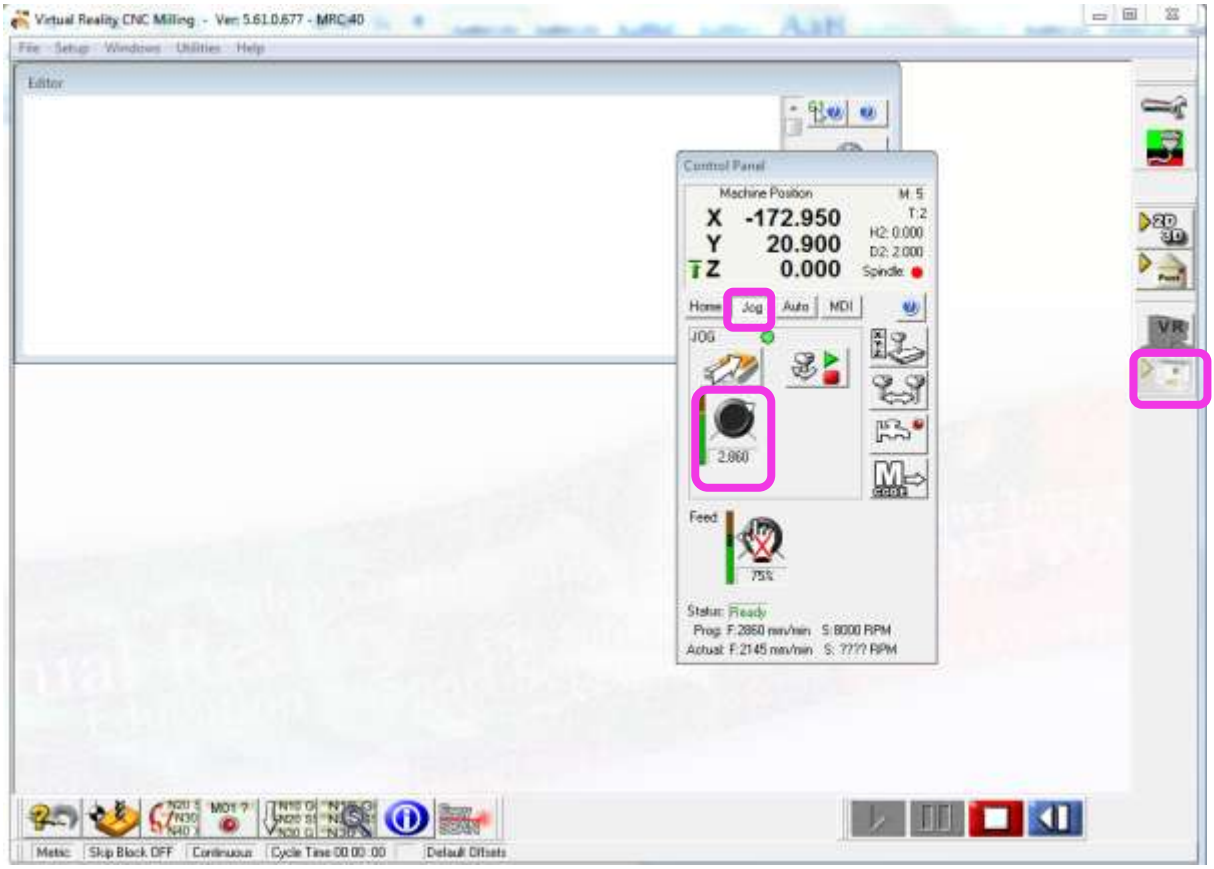


Clearance is a problem so we moved the holder of the drill up further to make sure the tool would clear the material and it can cut down to 50mm. Tool wouldn't go down far enough or clear the top. Now we need to be careful that the back board {photographs shown below} doesn't hit the material. To avoid this we'll have to stop the machine mid-way through the cutting. I only realised this was going to happen mid-way through cutting. Otherwise I would have set the it so it would clear the material from the start. Lower the tool piece and then zero the Z coordinate.



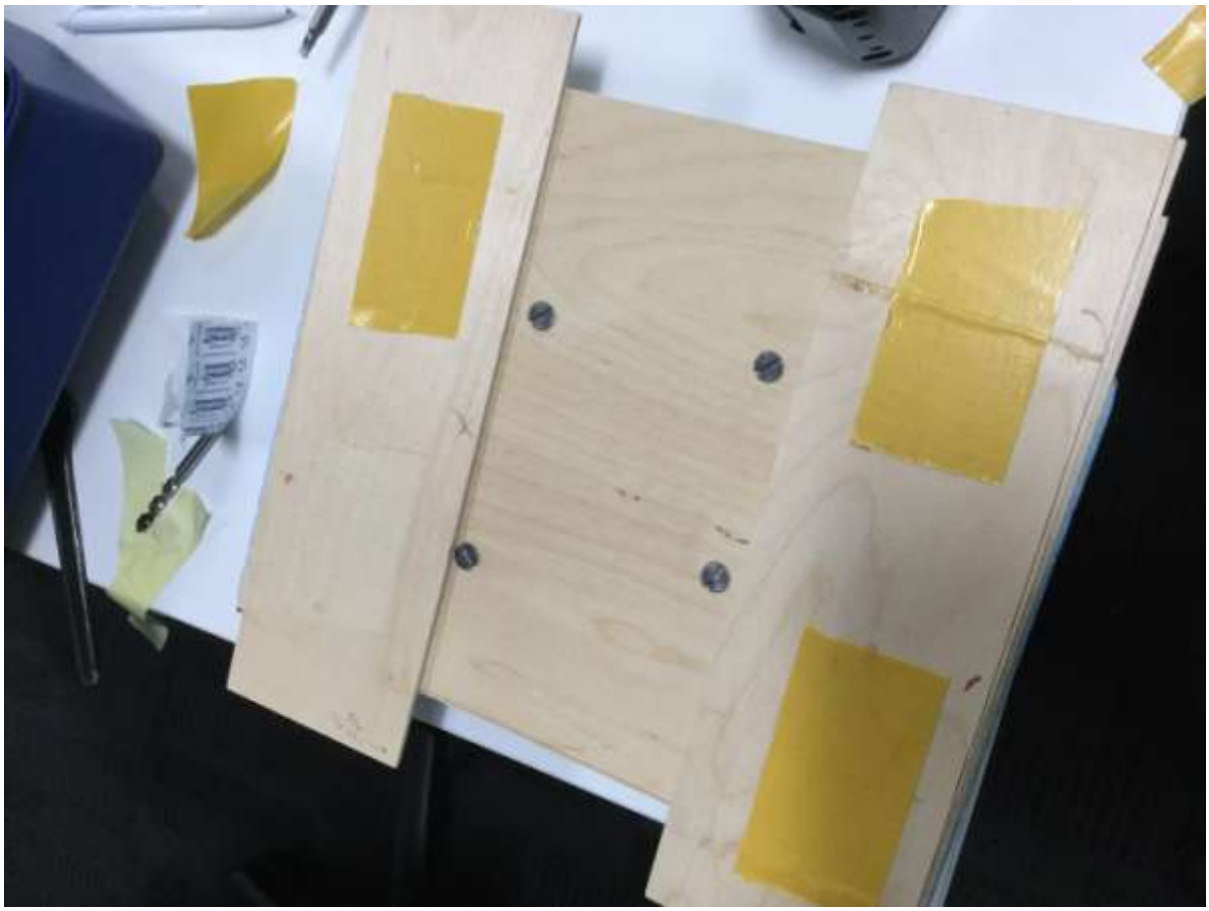
Then I moved the tool holder to hold the tool much higher so we could get the clearance (shown in the above photographs).

Opened up 'VR Milling V5' or 'Virtual Reality CNC Milling', then we need to open the control panel, shown below on the right of the screen. When the control panel opens then open up the Jog setting by clicking on 'Jog' shown below. How fast you can move the tool can be changed by moving the black dial. To move it quickly set it at 3000. To move it slowly set it at 500. You should move it slowly when it is close to the material or anything the tool might hit off. To move to tool up and down use the 'page up' and 'page down' buttons on the key board. To move from side to side and forward and backwards use the arrows keys. Note that when moving it backwards and forwards it is the bed of the machine that moves back and forward and not the tool.



Securing the Material

Once you have set up the cutting paths put the material piece into the machine using double sided tape (shown in the below photograph) The next step is to set the zero zero for the XYZ in relation to the billet position. I screwed down the foam to some wooden board to hold it in place.

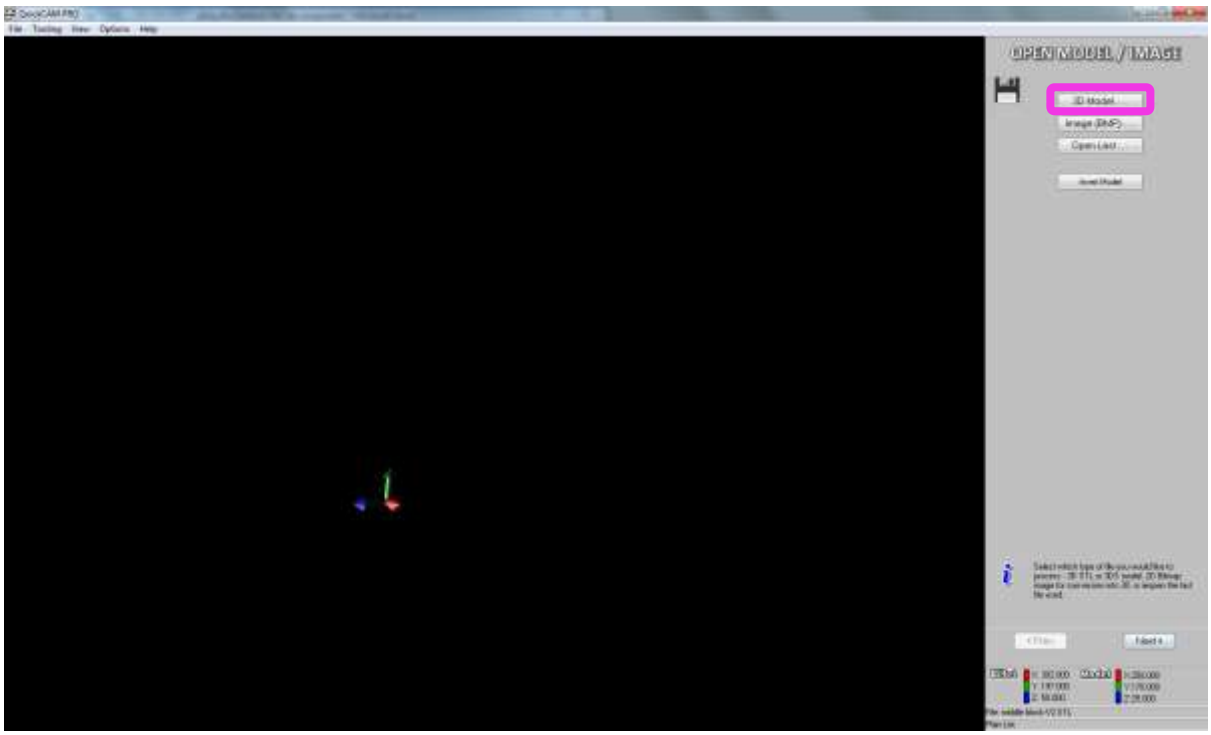


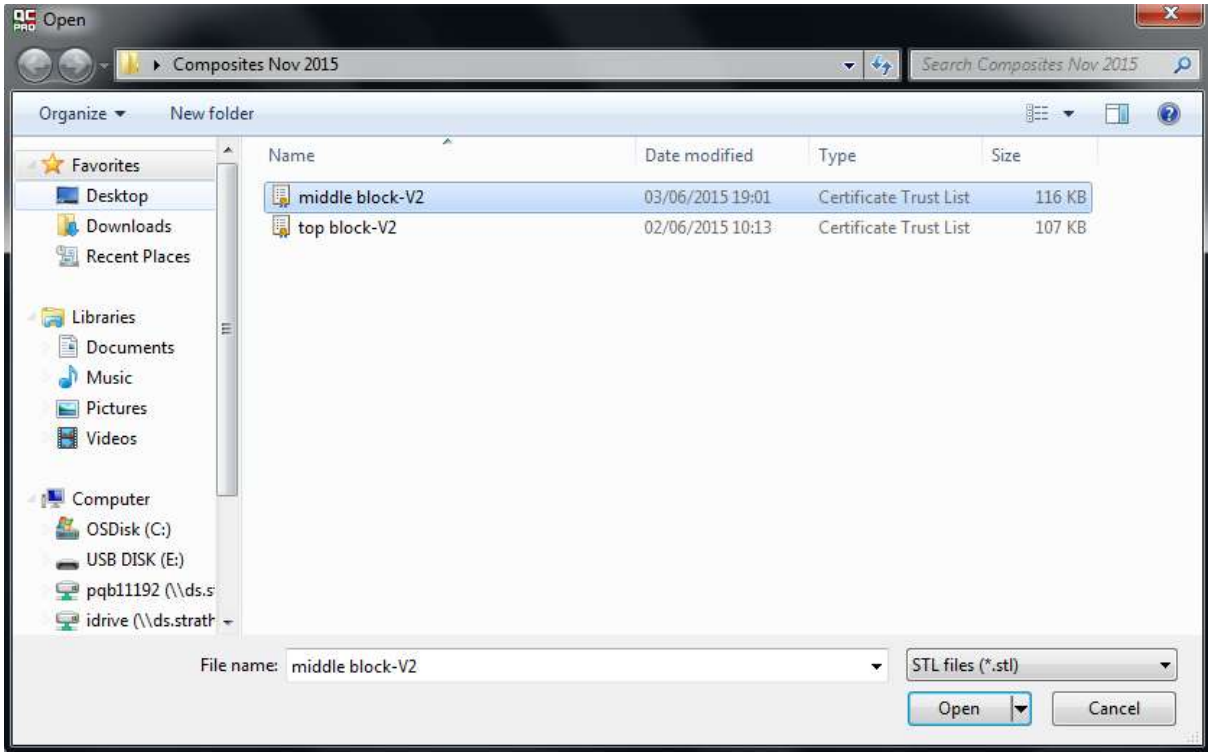
Import the Model.

The model was in an stl. format that was exported from a solidworks model.

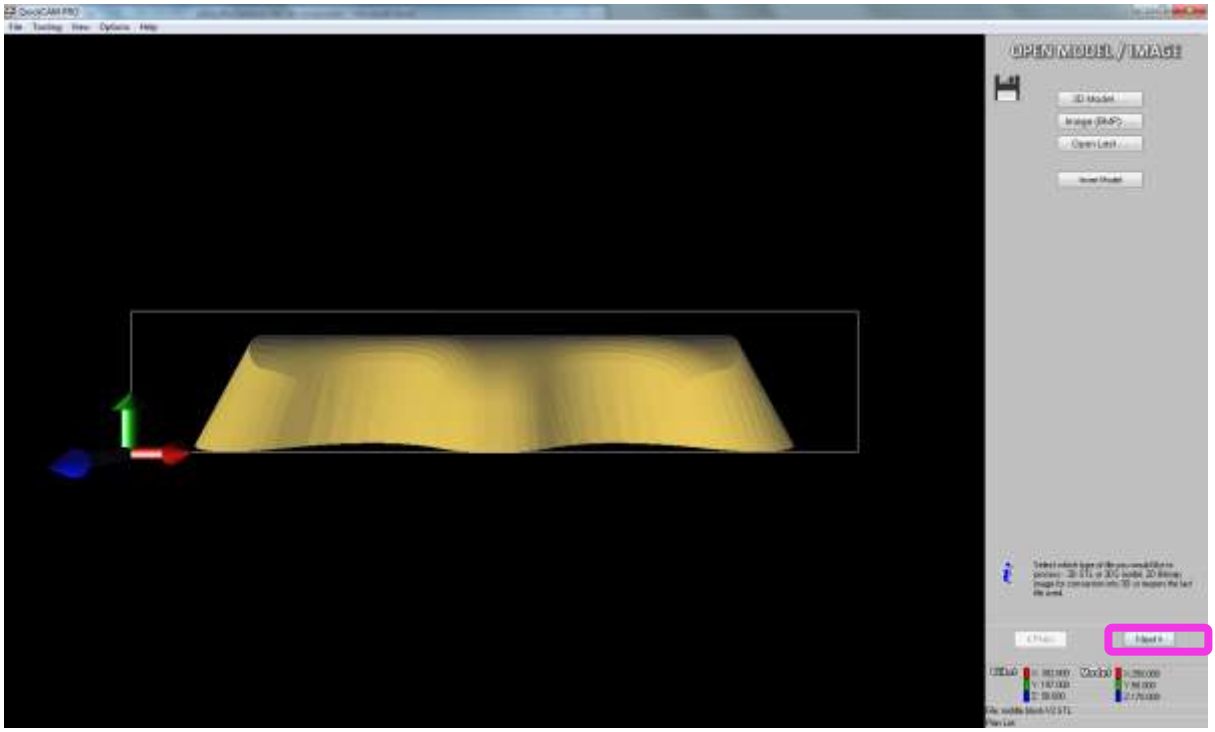
Open up the 'QuickCAM Pro' software. This software will set all the machine setting using your stl model and then convert it to gCode to communicate with the machine.

First, import the model.

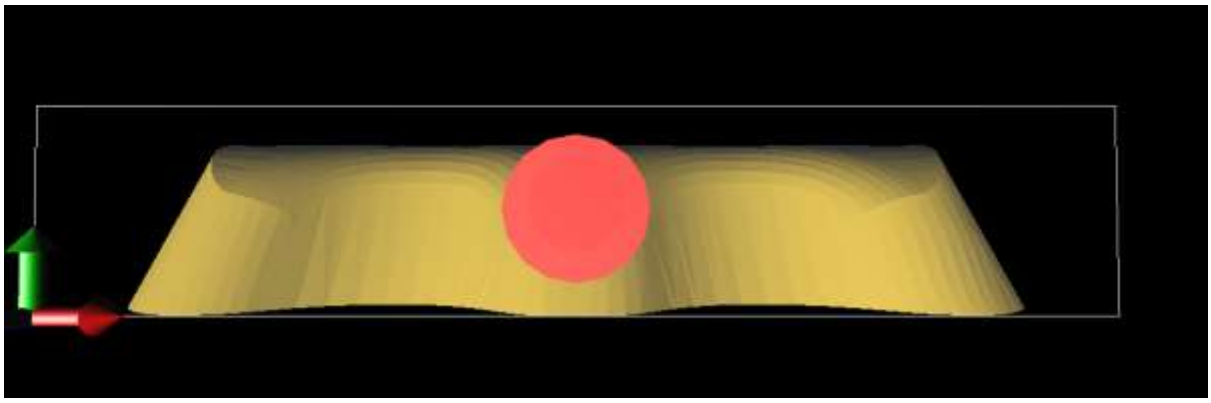




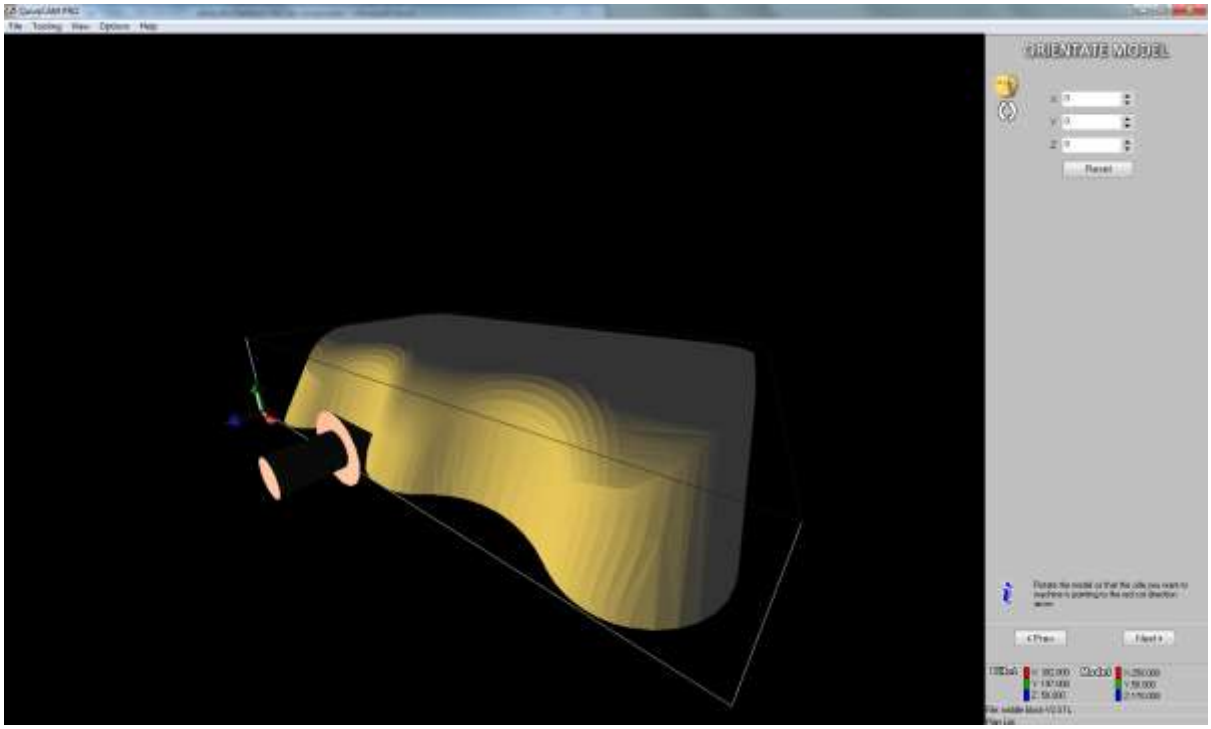
This will then show the imported model, shown below. If there was no issues here so, then select next.



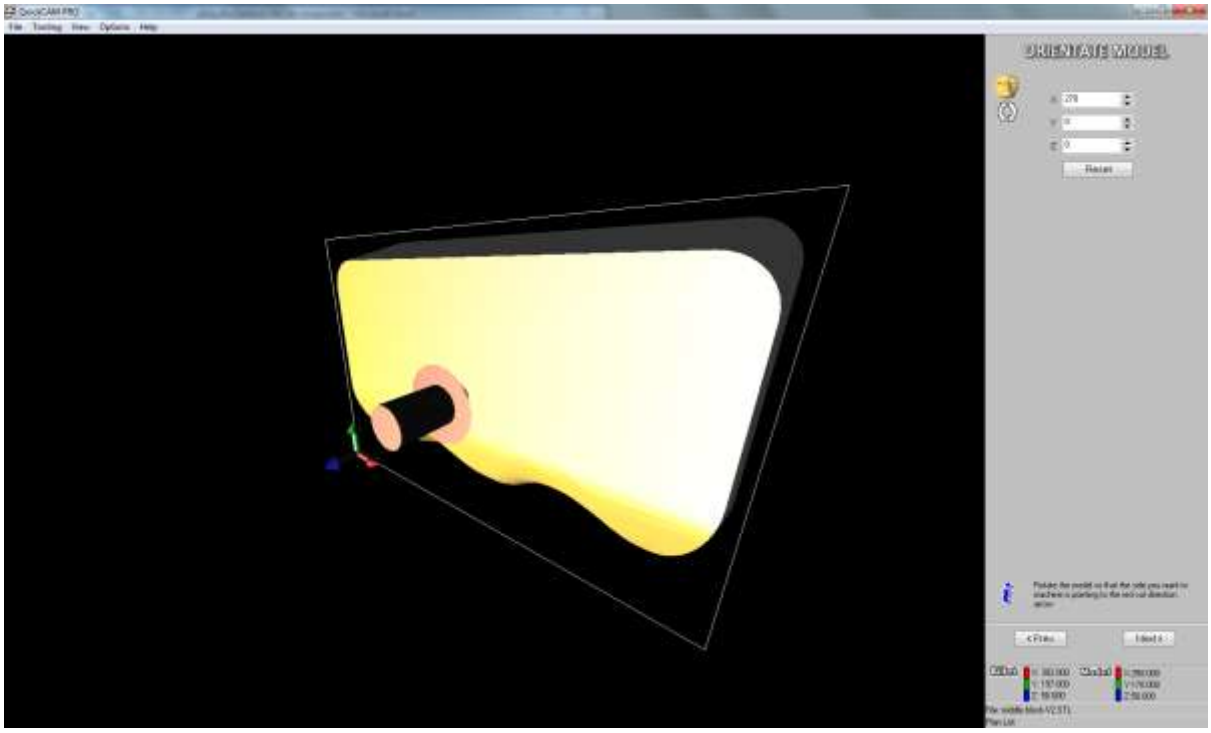
It will bring up a bit red dot in relation to the model, shown below. This indicates what direction the tool will cut from. You need to reset this.



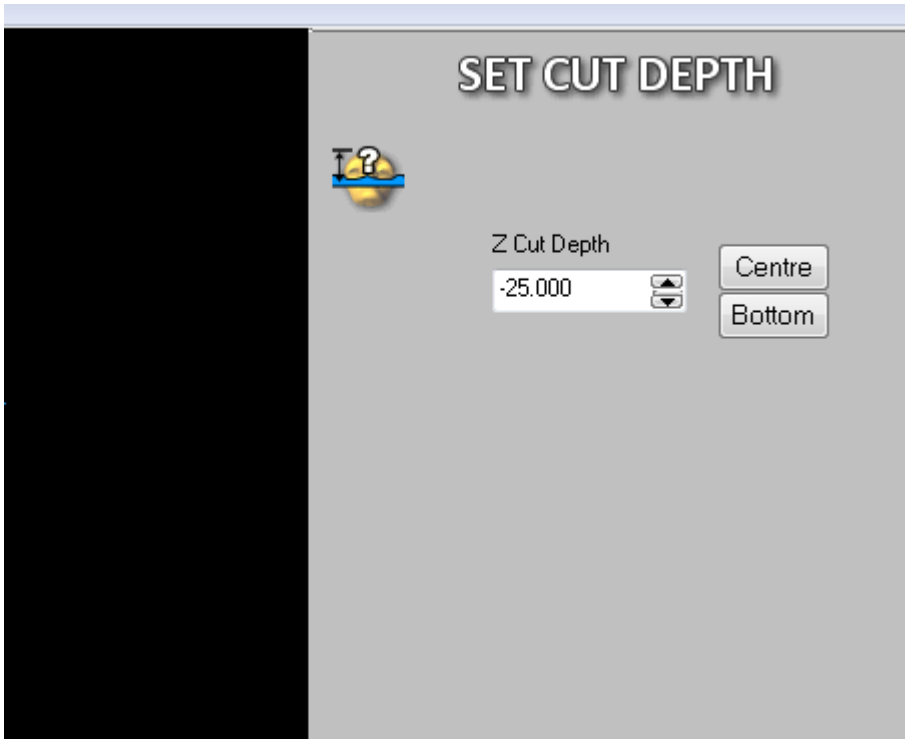
By clicking on the buttons at the side to 'Orientate Model' you can tell the software where the machine will position the tool to from a direction in relation to the model.



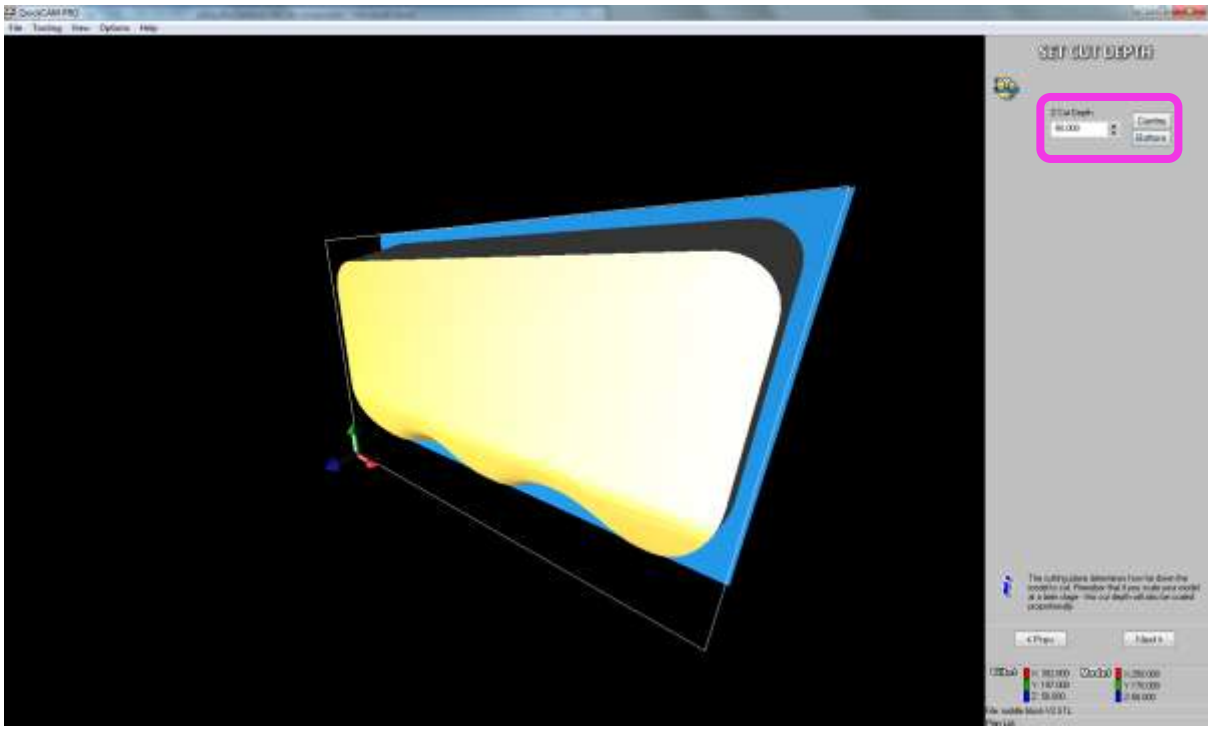
We want it to cut from above, shown below. Then press next.



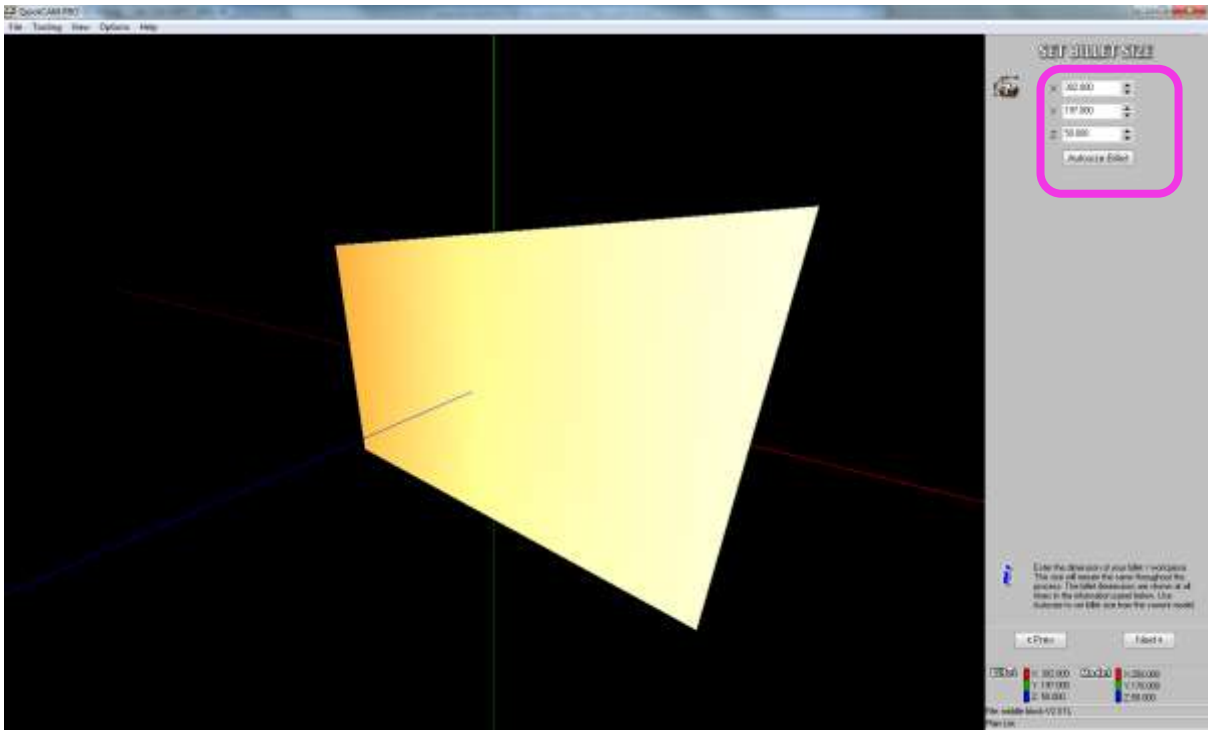
It will bring up the next step which is to 'set cut depth'. At the moment it's set to -25mm.



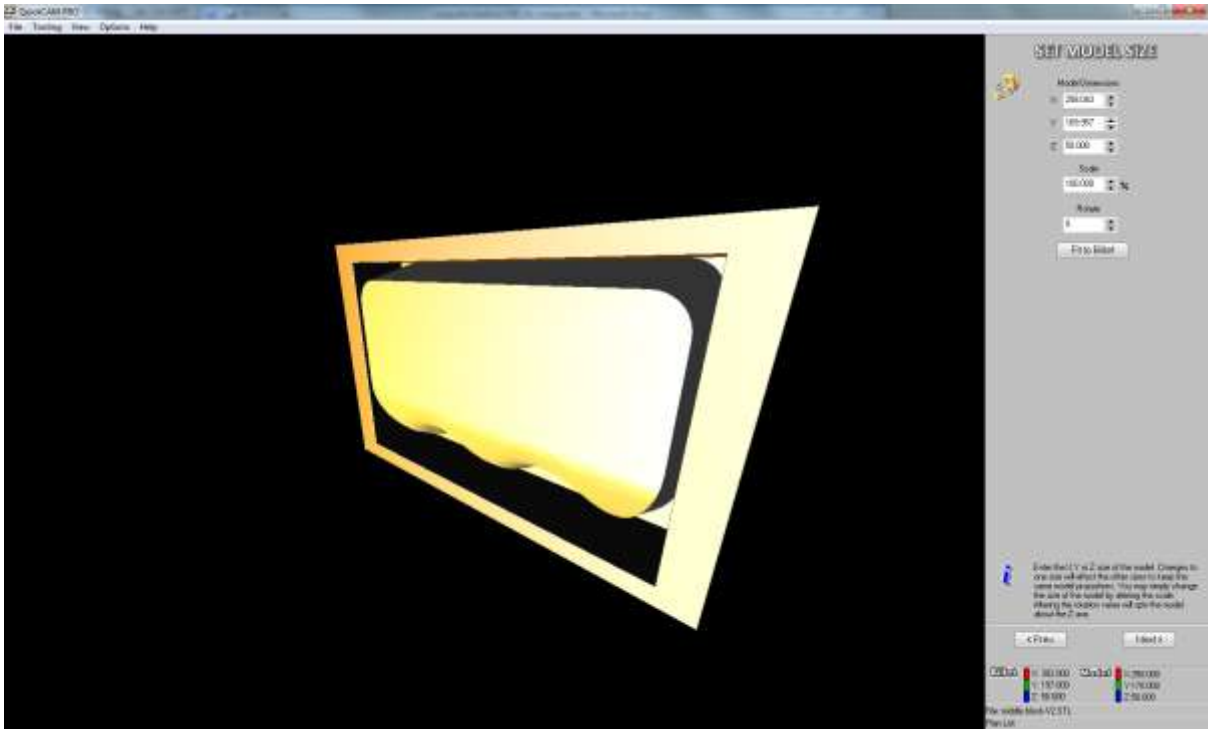
We want it to cut the whole way through the material to press the bottom button and it will set it to the depth of the material, which is 50mm. Then press next.



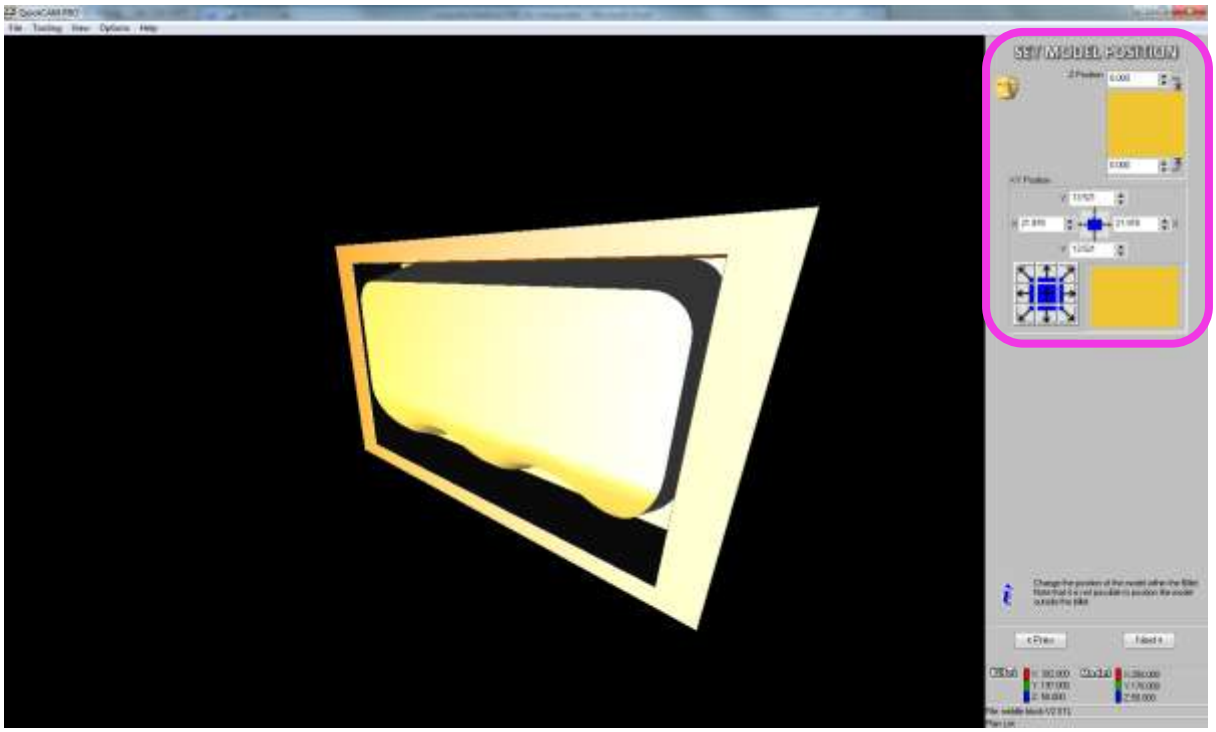
Then we need to set the size of the material piece we have to cut from, which is known as the Billet. Once, I've entered the dimensions then press next.



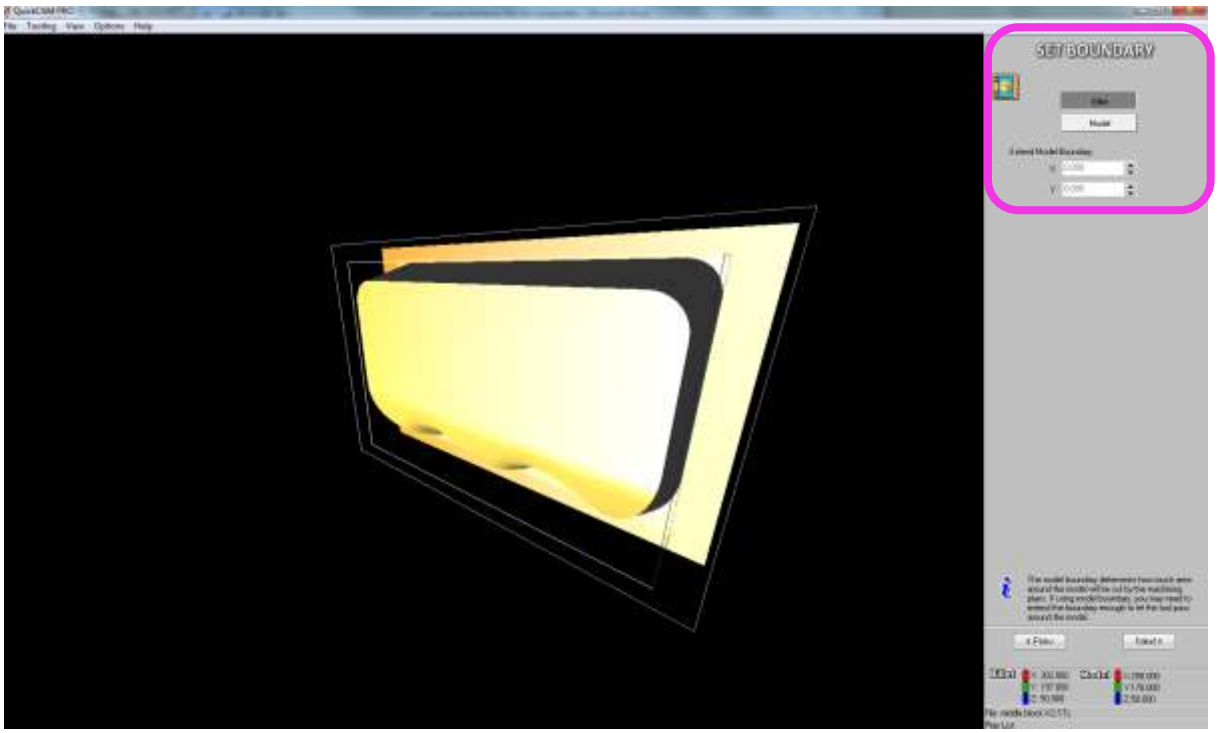
Next we have the options of adjusting the model size. We don't need to do this so just select next.



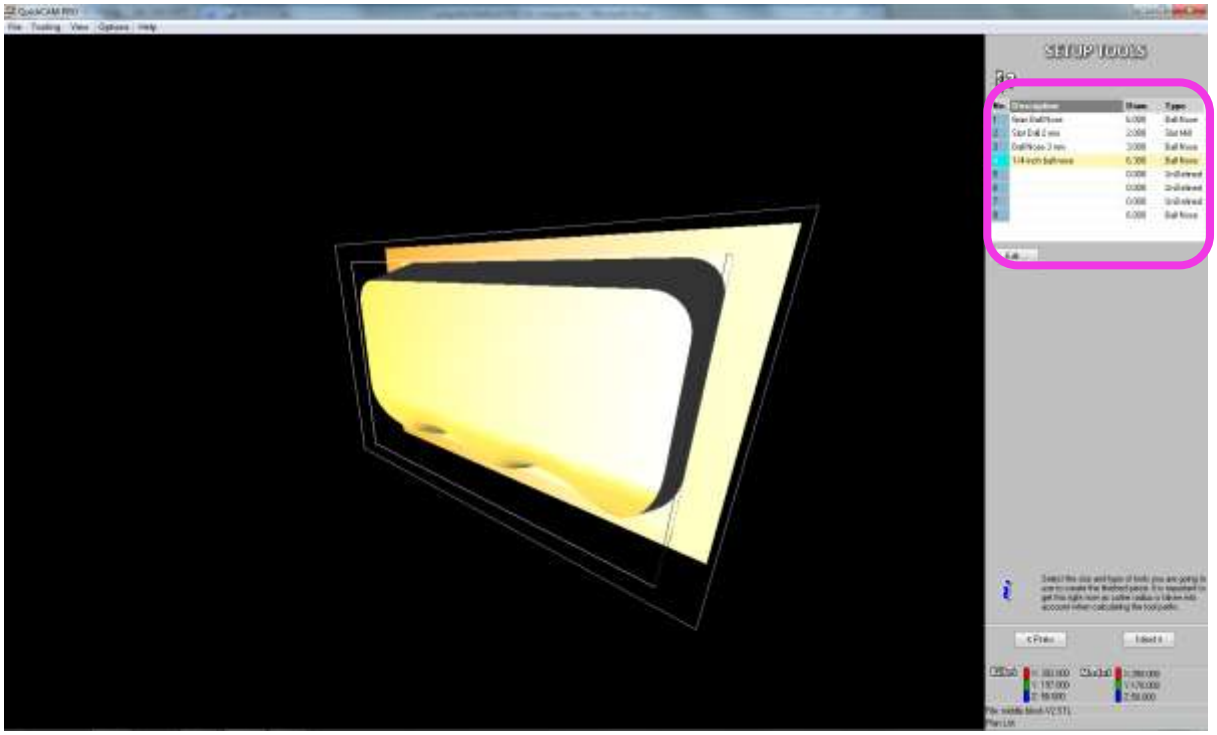
Next you can choose what part of the material the model will be cut out of. I left it in the centre as there wasn't much material to be left over. You can position it by using the control shown below.



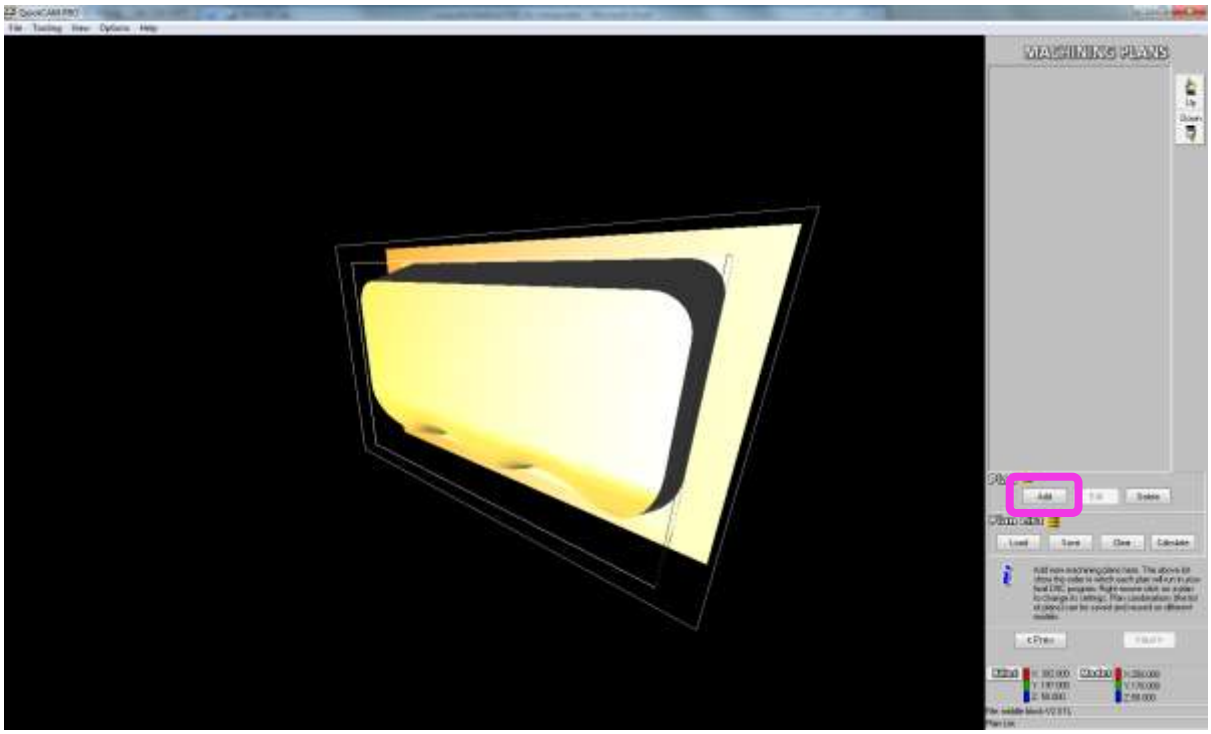
Next you can choose to cut out only the model or cut away all of the material in the Billet. I choose to cut away all of the material so I would have only one piece left when the cutting was finished. Then select next.



Next I chose the size of tool I used to cut. In this case it was quiet big because it was a big piece of material and I didn't need a perfect finish. Although a ball nose will still give a good finish. Once you've selected the right tool bit then select next button.

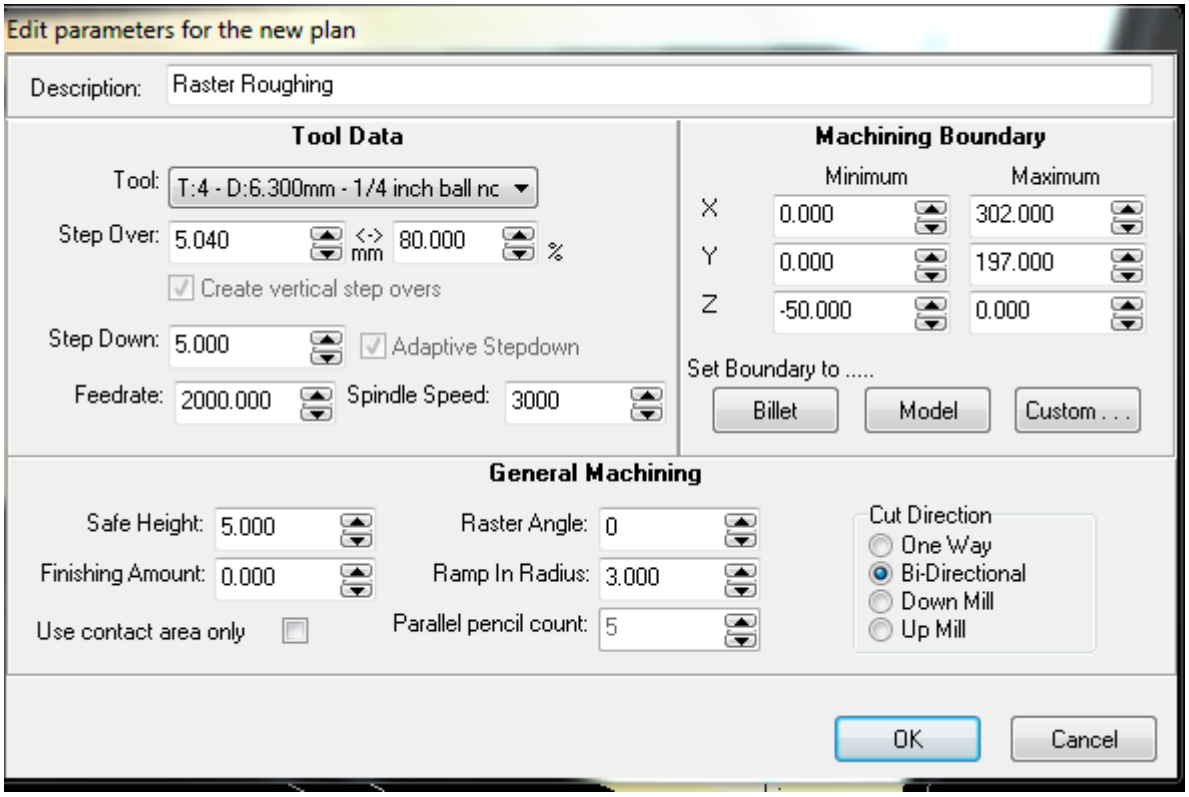


The next step is to calculate the machine paths. Similar to a lot of this type of software, select the add button to add a machine path.



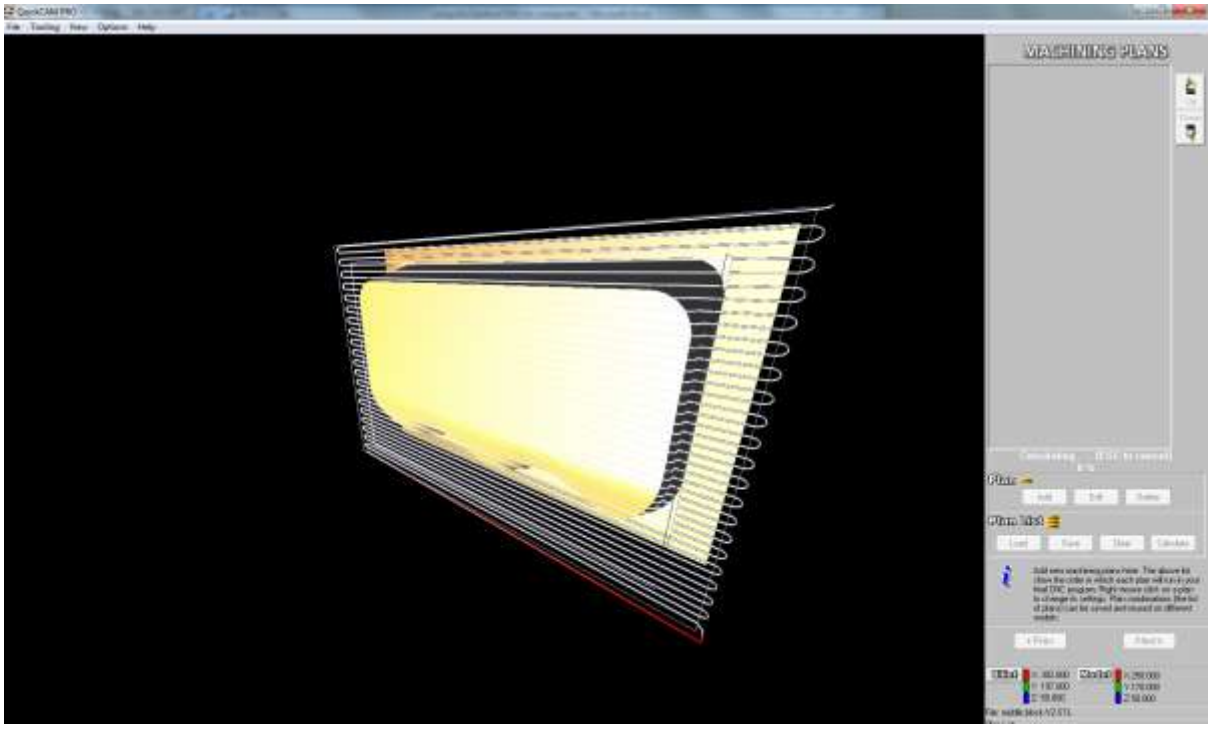
A pop up window will appear. The first pass of the tool we want to create a rough cut to take away the majority of the material. Afterwards we'll go over it again with a finishing pass to give it a smoother surface. Select 'Raster Roughing' option shown below.

A pop up window will appear. Adjust the settings to match what you are doing. In my case. The tool is a ¼ inch ball nose tool. I want a step over of 80%. This means only 20% on each pass will overlap with the last pass. It will give a rough finish but will get the job done much faster than less step over. Step down is how much each pass will go down by in relation to the last pass of the tool bit. Feed rate is how quickly it will move from one position to another. Spindle speed is how fast the tool piece will spin. In our case we don't ever adjust this because it always runs at one speed. We can adjust this manually as the machine is running on the front of the machine (shown in the photograph below). The safe height is the distance the tool will be above the material; while it's not cutting. Bi-directional is what I use because it means it cuts both ways and therefore reduces the time by reducing the amount of passes it takes to cut the material. If it can cut in both directions the it's faster than on direction. Then select ok.

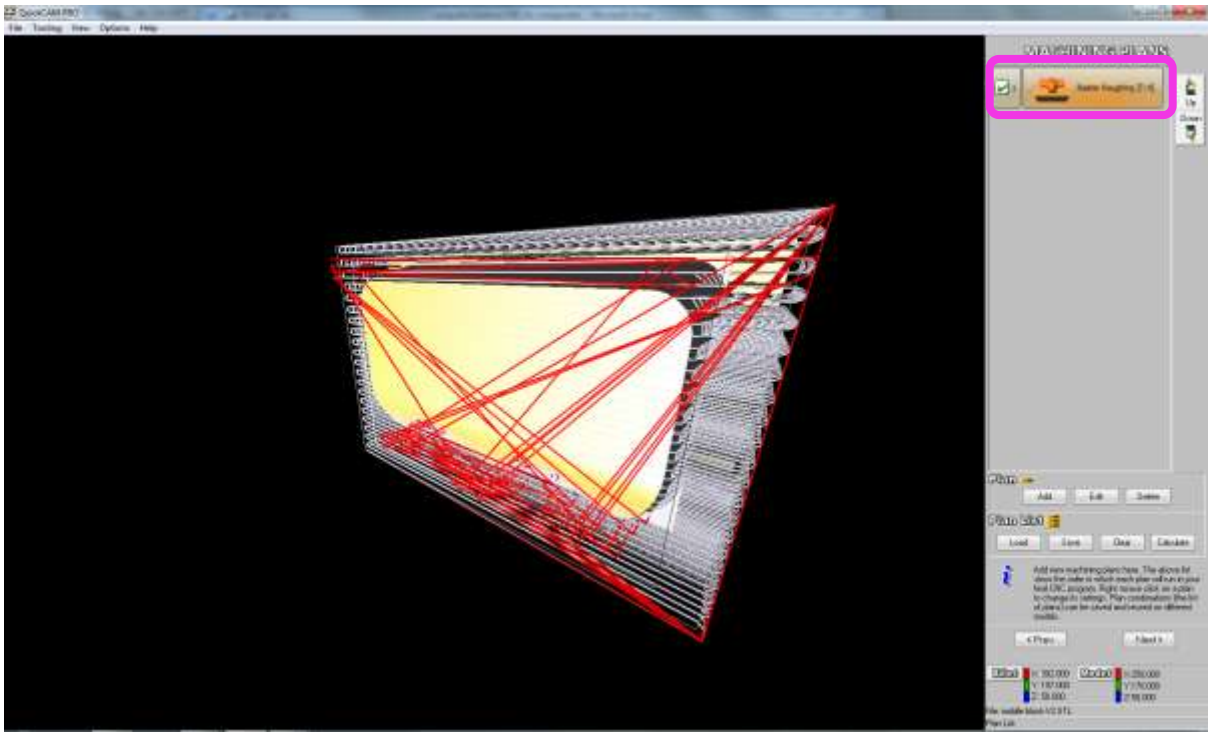



DENFORD
MRC 40
CNC PC

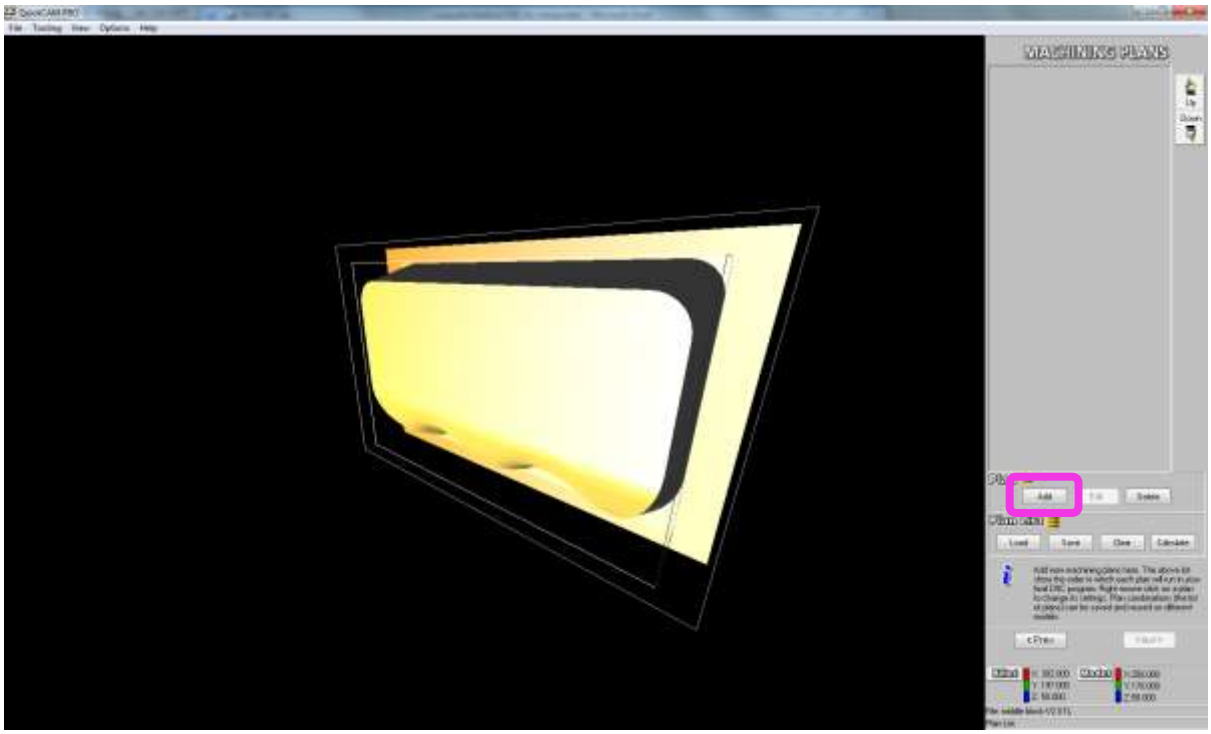
It will start calculating the tool pass for this, shown below.



Then the path will pop up in the window on the right and shown you all the paths it will take.

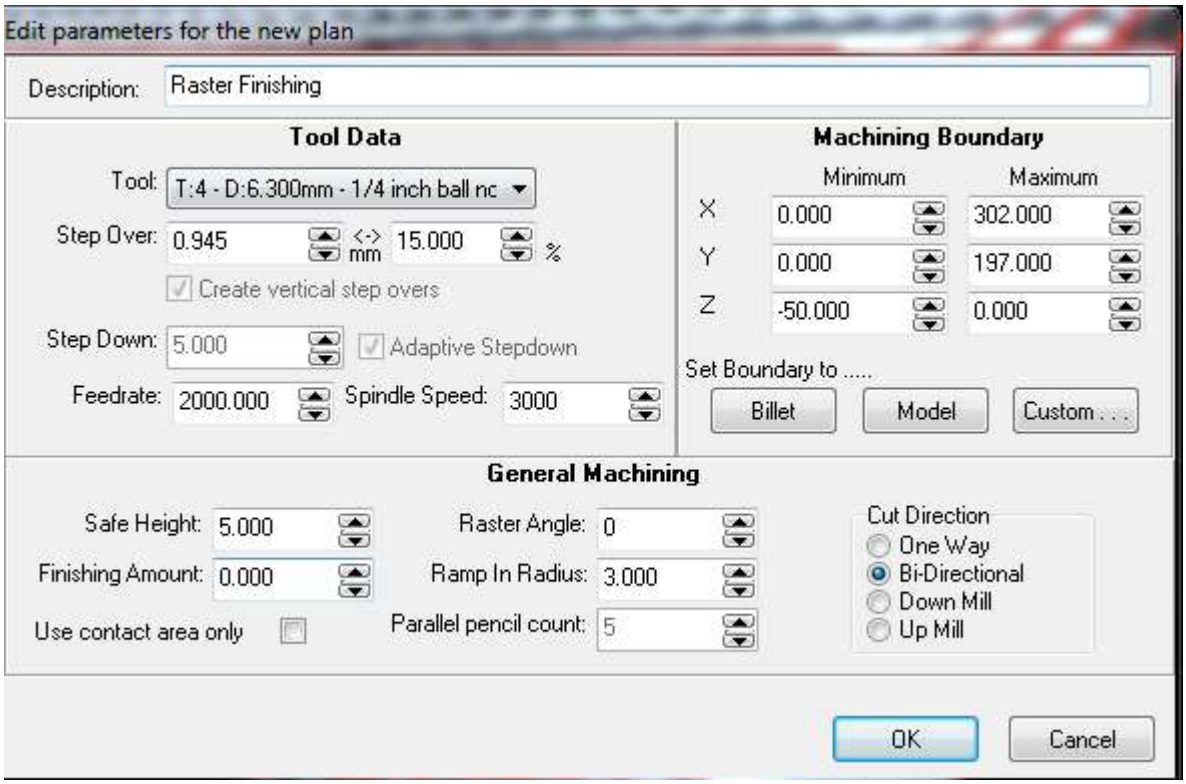


Next we need to make the finishing pass to make the final surface smooth. Select the add button to add a machine path.

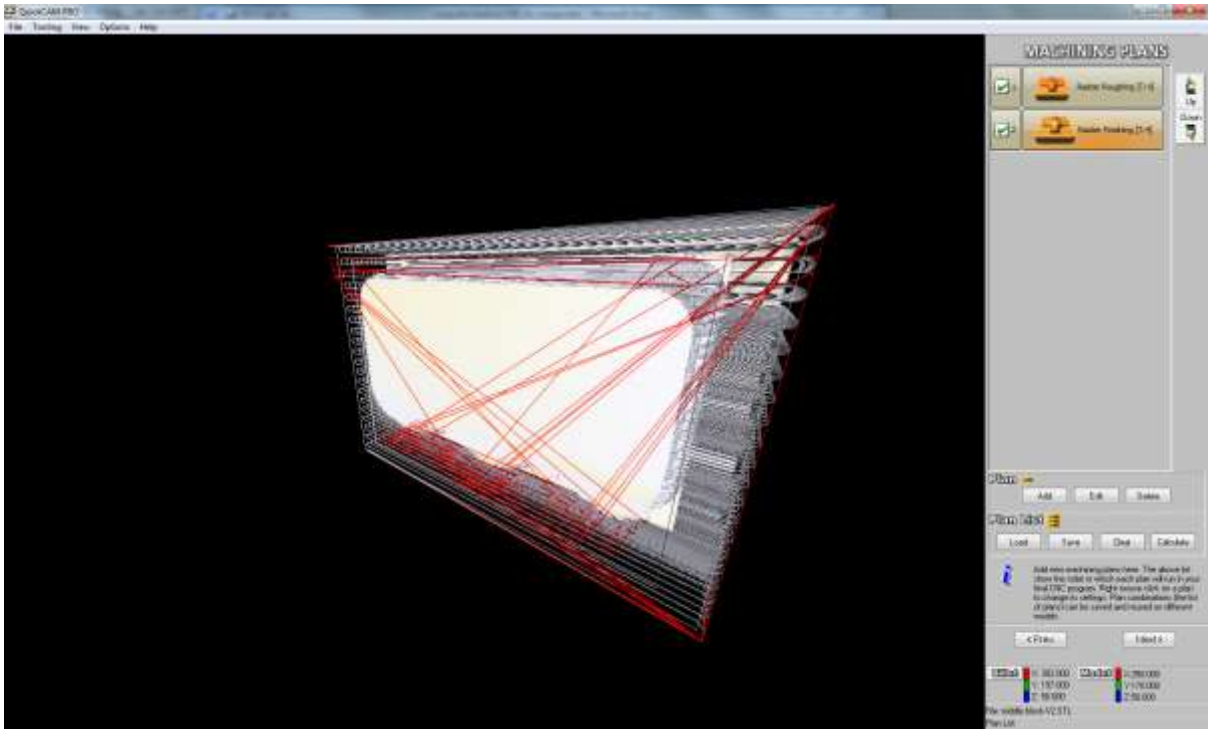


This time select 'Raster Finishing'.

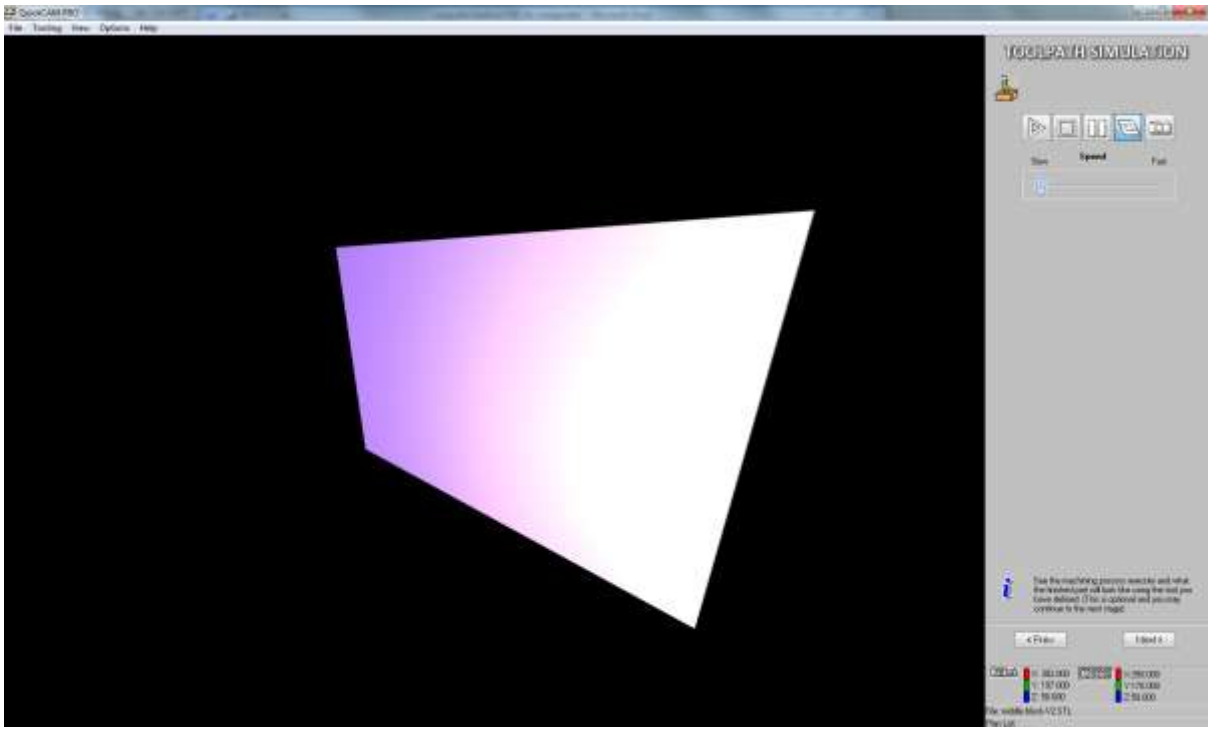
The parameters will be different from Raster Roughly. Make sure the step over is at 15% and the correct tool bit is selected. The rest of the setting should be as shown below. Then select ok.

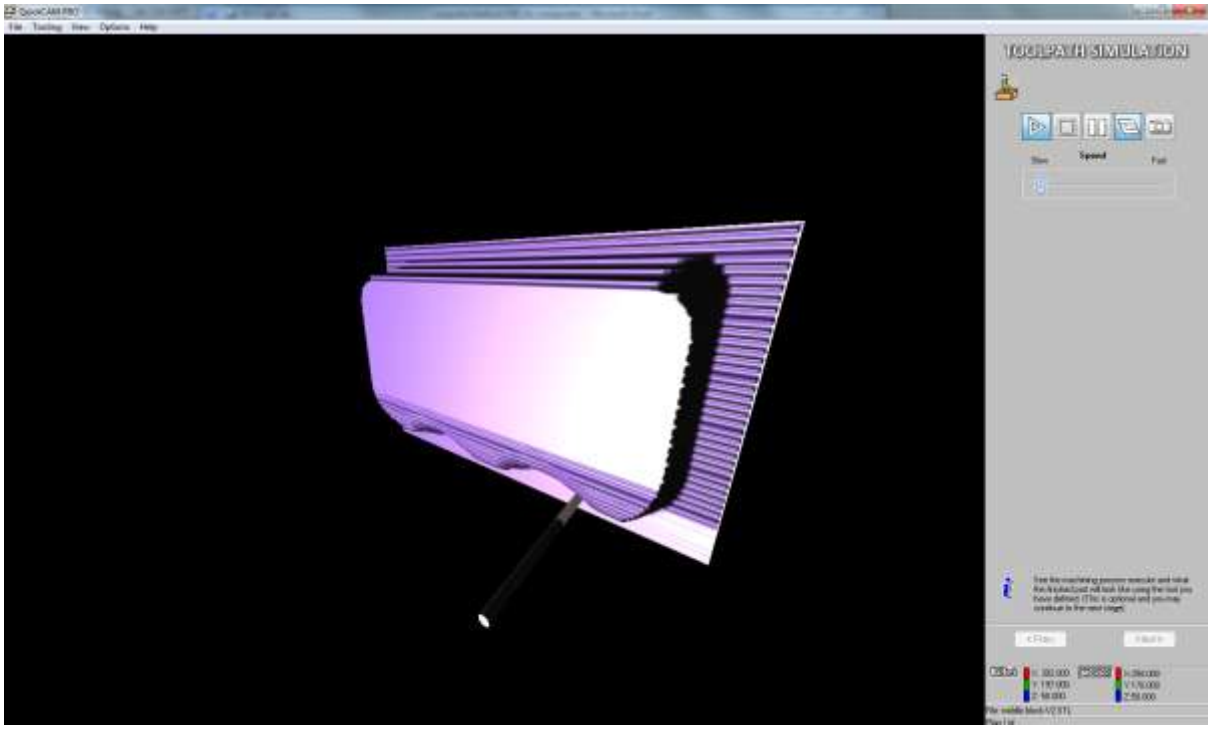


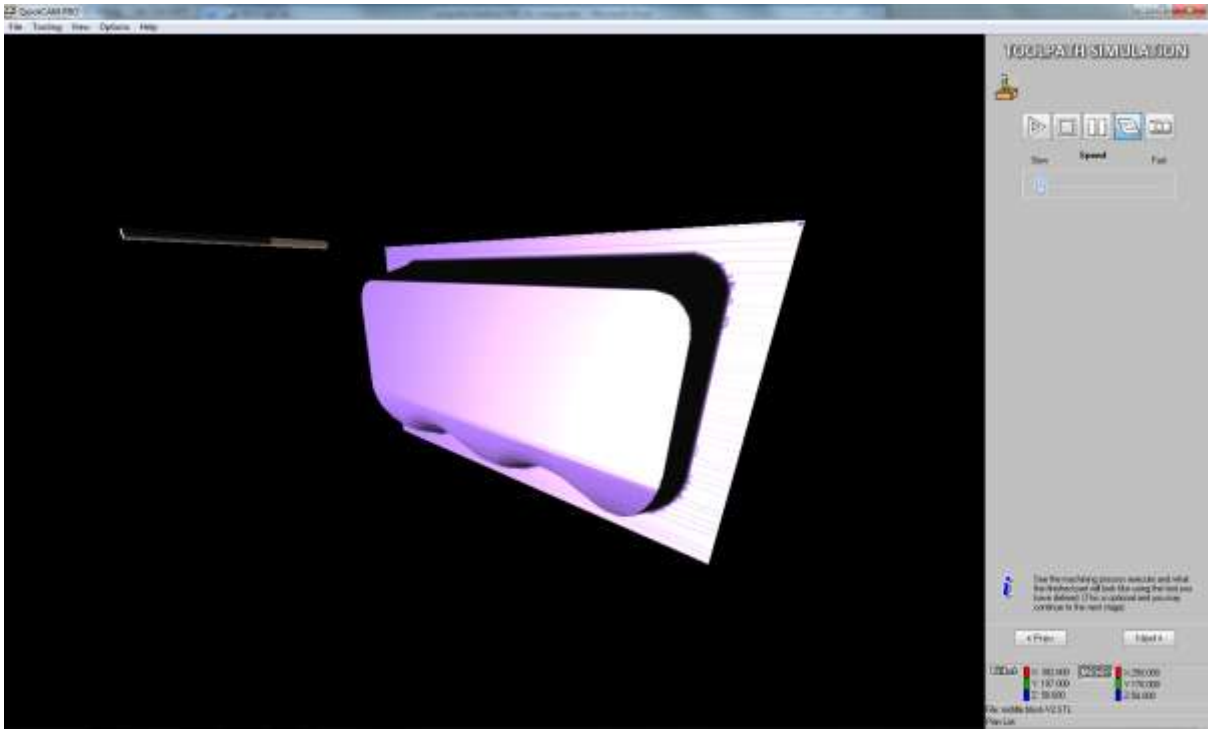
The software will calculate the paths for this machining step and add it to the list of 'Machining Plans' shown in the right column. If it all looks ok, then select next.



The next pop up window will be a 'Toolpath Simulator', shown below. Press Play to show a simulation of the paths the tool will cut and the final resulting piece.

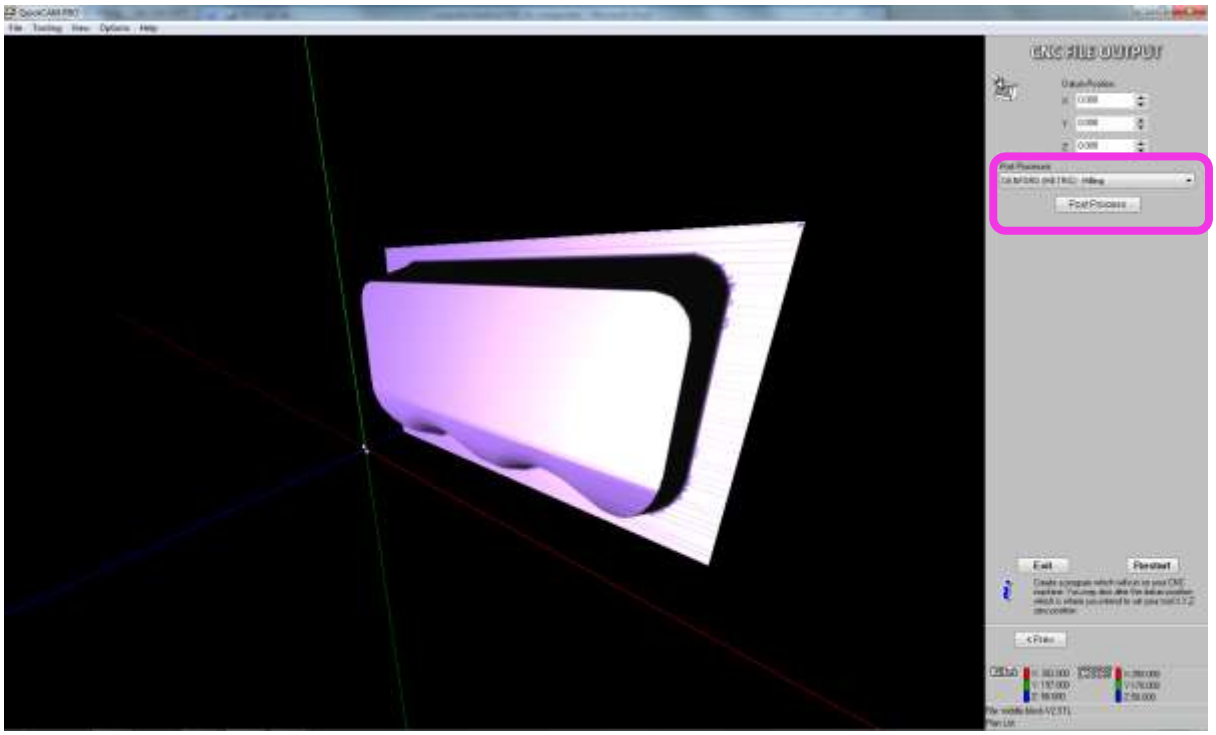




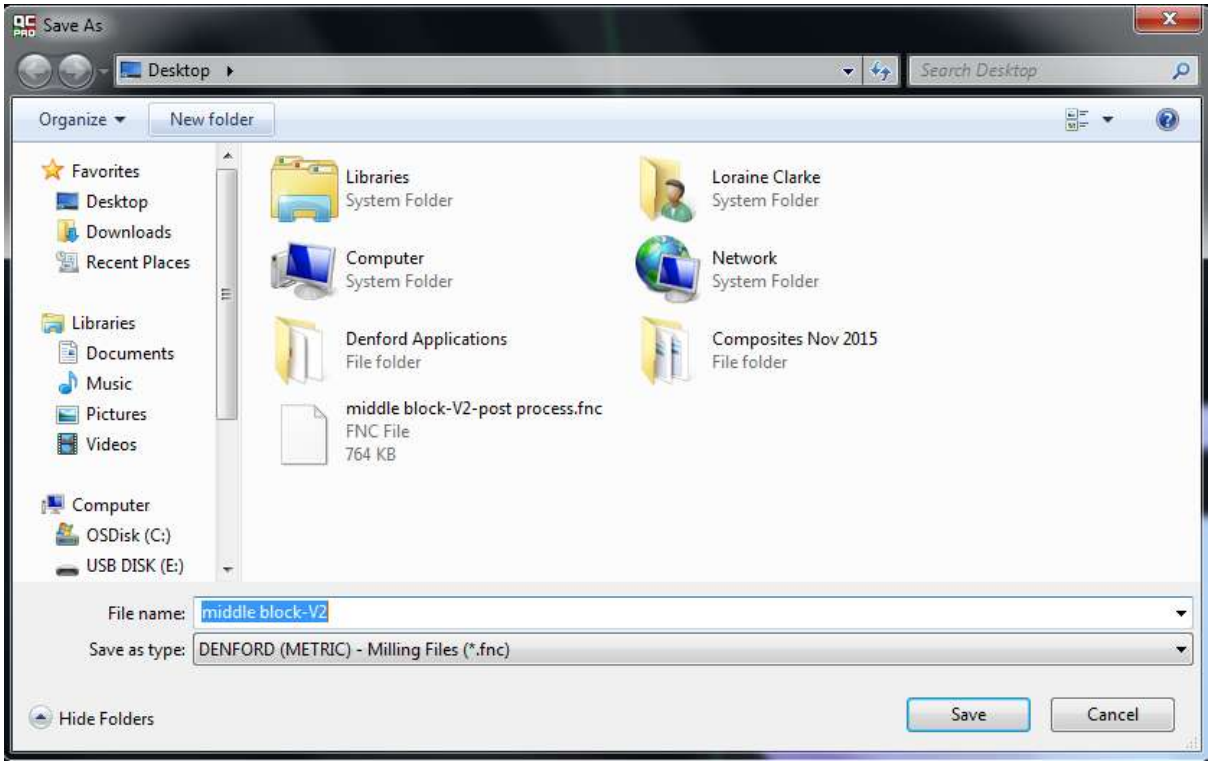


Then select next.

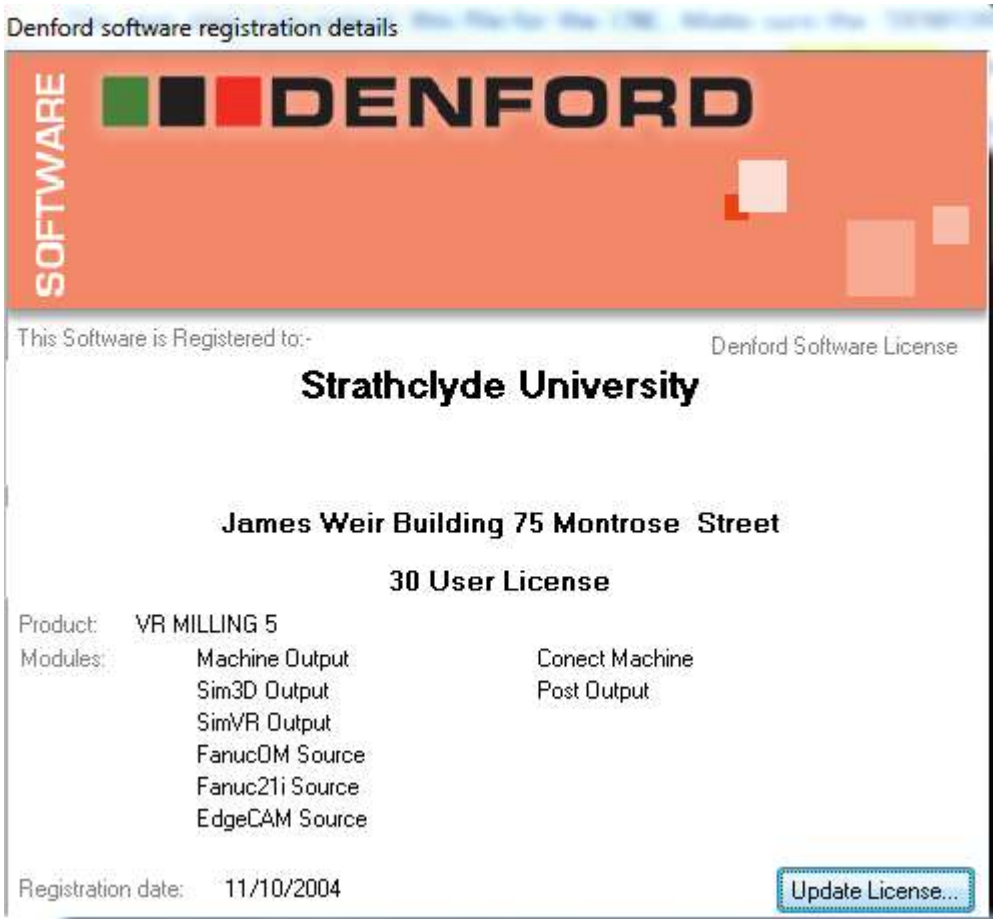
The next step is to output this file for the CNC. Make sure the 'DENFORD {METRIC) – Milling' option is selected from the drop down menu. Then Press the 'Post Process' button.



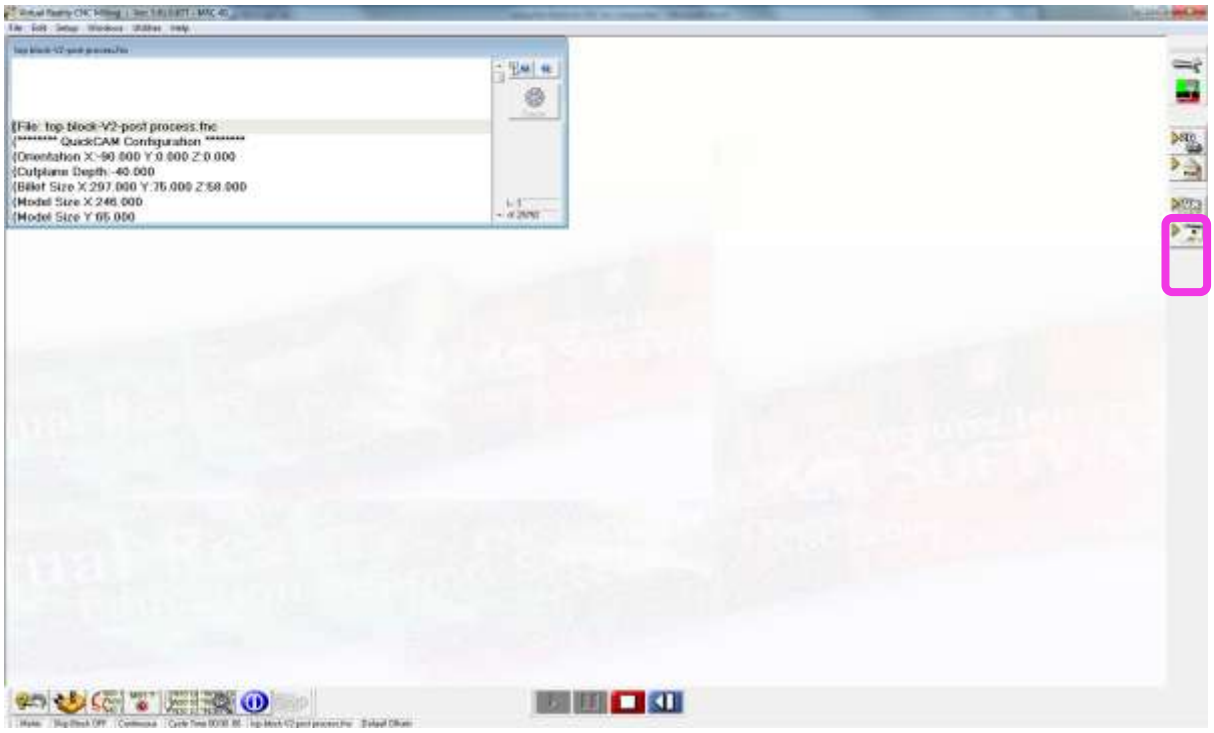
Next it will ask you where to save the file. In our case the machine works best with files saved on the desktop so best to do that.



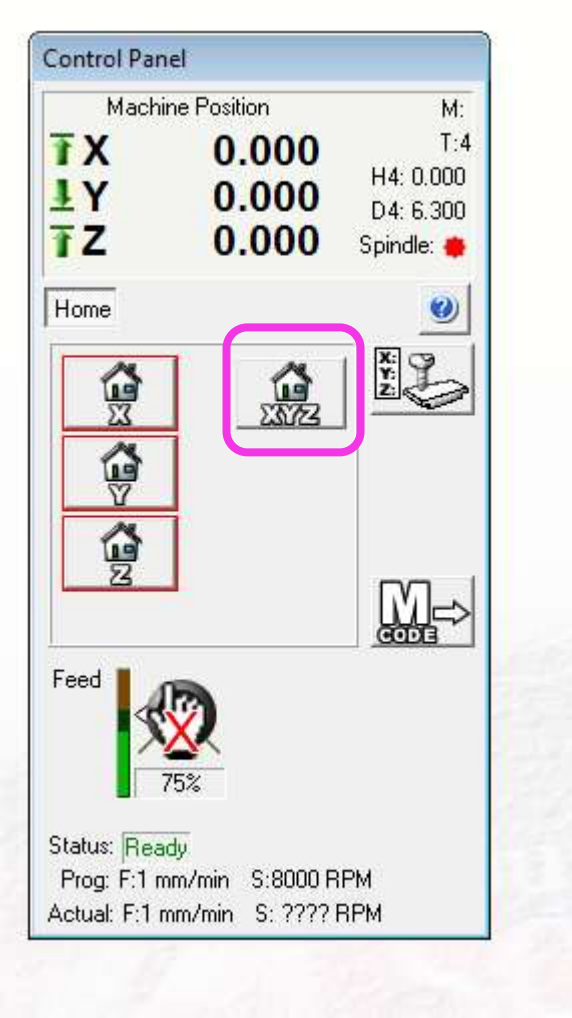
Once you have saved this it will automatically opn up the software to communicate with the CNC, 'Virtually Reality CNC Milling' shown below.



Select real machine to open up the control panel settings, shown below.



Then press the XYZ button. This will get the tool to travel to XYZ zero zero. Before it does this make sure the tool won't hit the material? Otherwise you won't get the other options you need like 'jog' menu to come up on the screen.

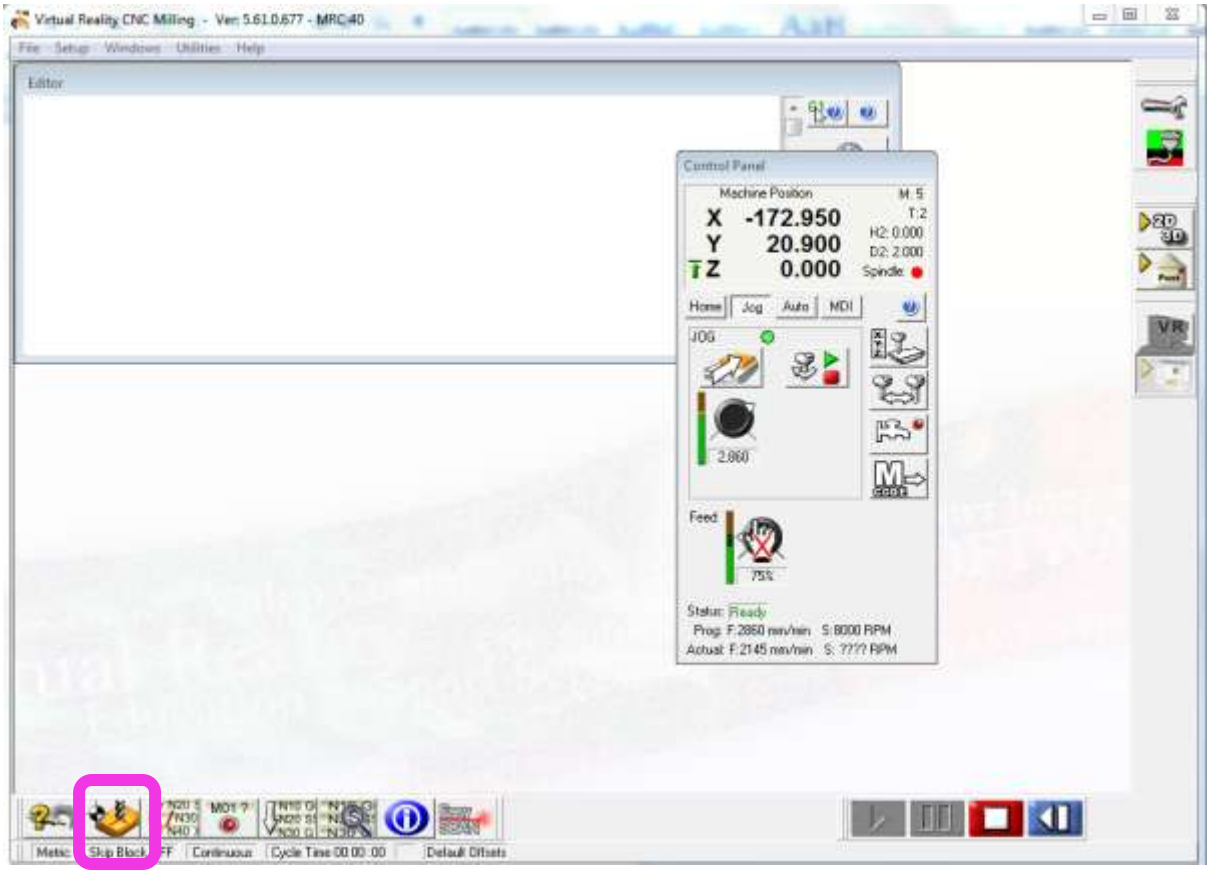


Next select the jog menu. Now we'll set the coordinates for the zero zero for XYZ.

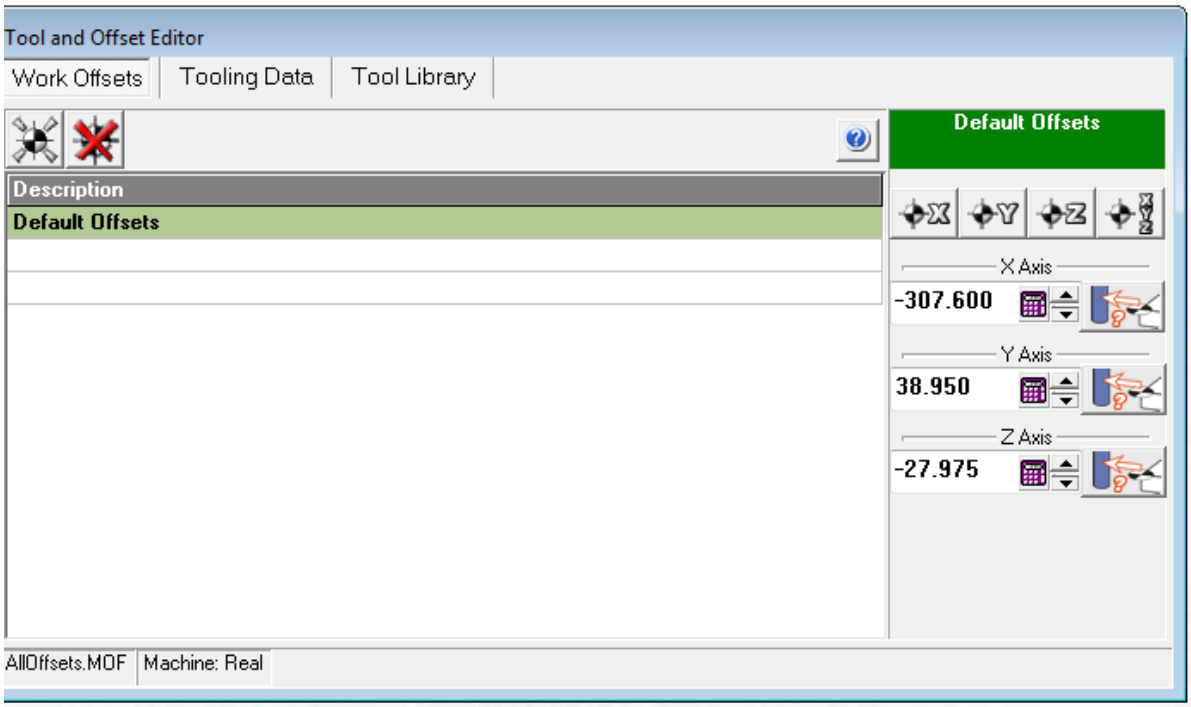
Setting the XYZ to Zero Zero

First we'll start with X.

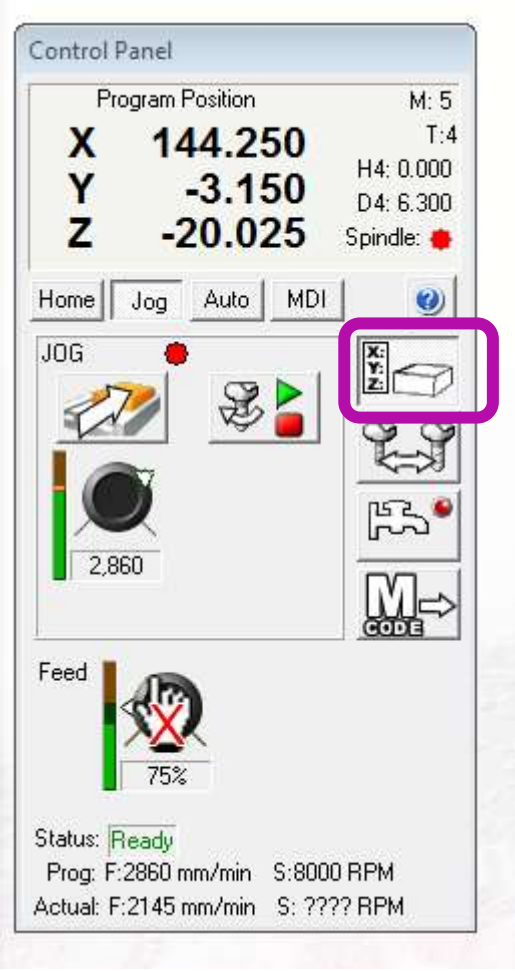
We need both the jog settings and the 'Tool and offset editor' at the bottom of the window, shown below.



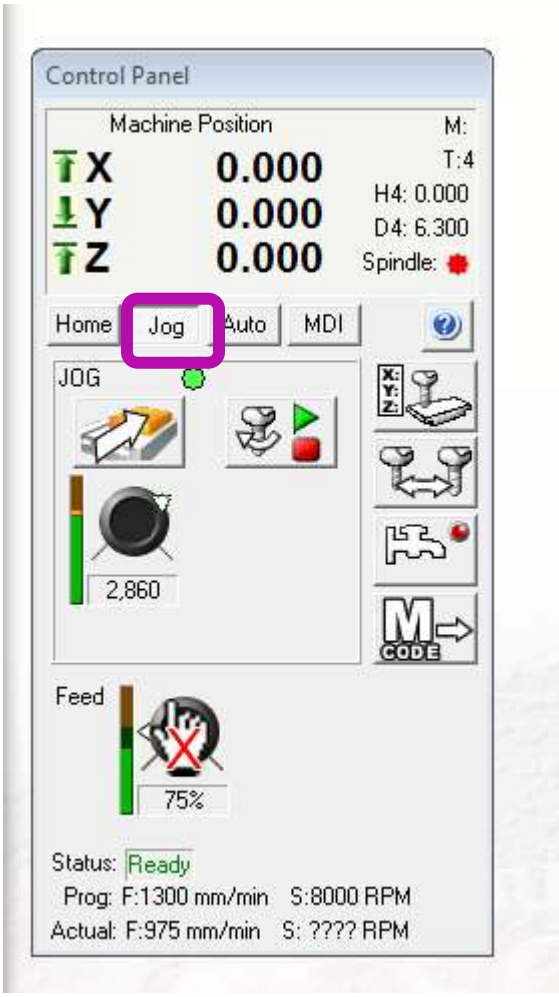
Which will bring up the window shown below.

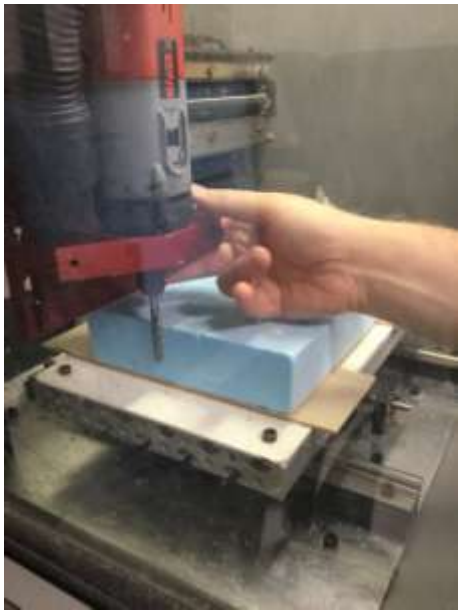

Tool and Offset Editor
Work Offsets
Tooling Data
Tool Library
Default Offsets
Description
Default Offsets
X Axis
-307.600
Y Axis
38.950
Z Axis
-27.975
AllOffsets.MOF
Machine: Real

Very Important. Make sure the 'Switch Between machine and program position display' is selected

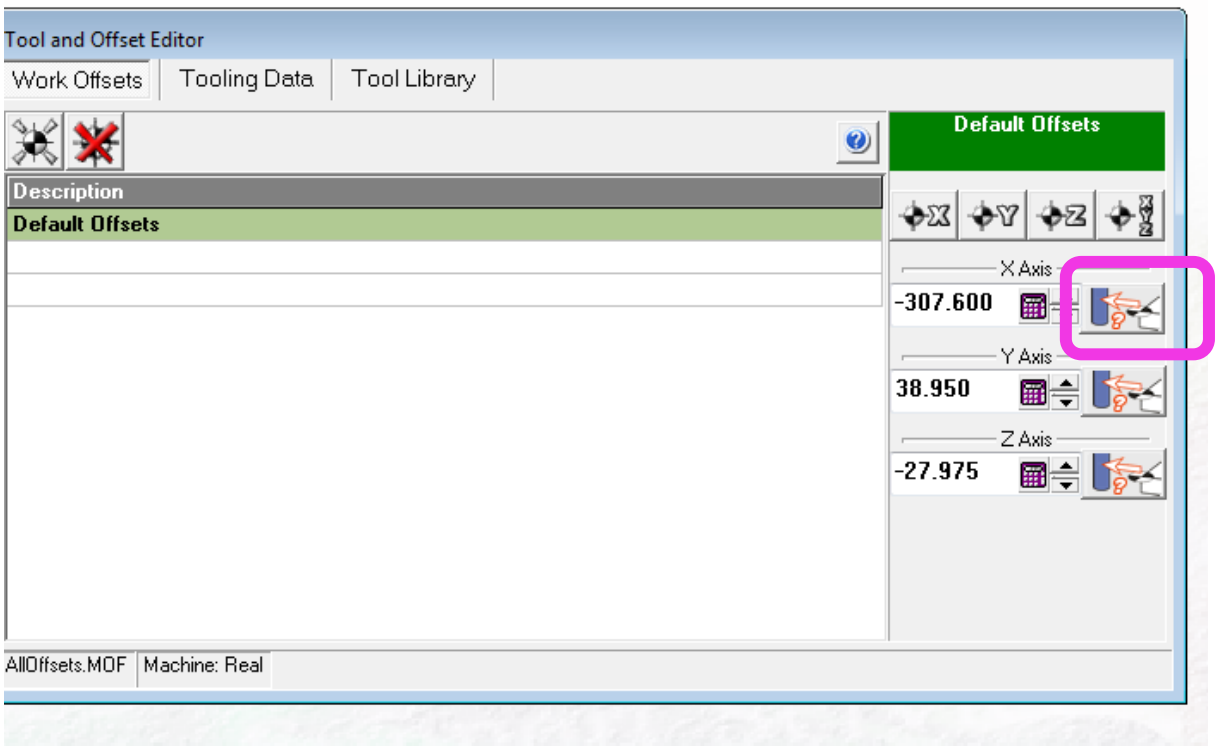


Using the jog settings to move the tool piece to the side of the material (shown in the below photograph) Be careful of not running the platform or the tool piece into the material.

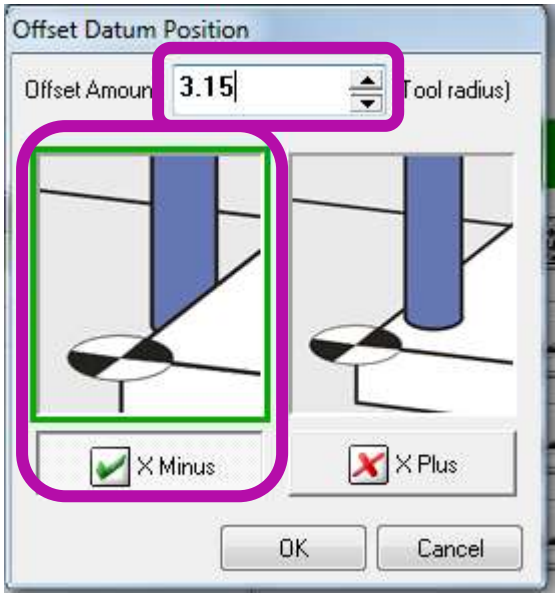




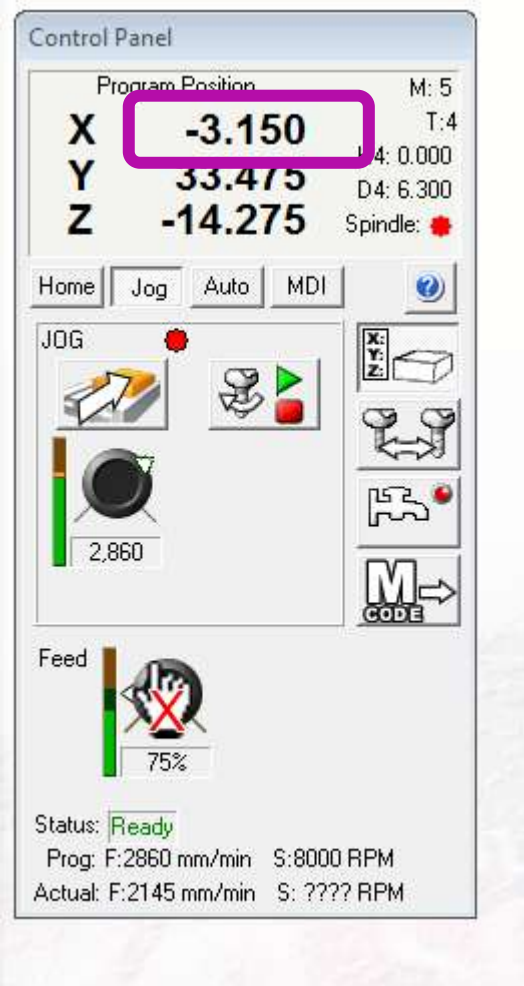
Next we want to set the position of where the tool is as 0 for X. Press the icon shown below related to the X coordinates. This will open up a pop up window where you can manually input the current position as the 0 for X.



Since we have positioned the tool at the side of the material (shown in the previous photograph) we need to tell the software that the tool is at the side of the material by selecting the left image, shown below. The tool offset amount is the half the thickness of the tool bit. This tell the machine to calculate where zero is for X taking into account that the tool is at the side of the material and the thickness of the tool bit itself is 6.3mm. Then press ok.

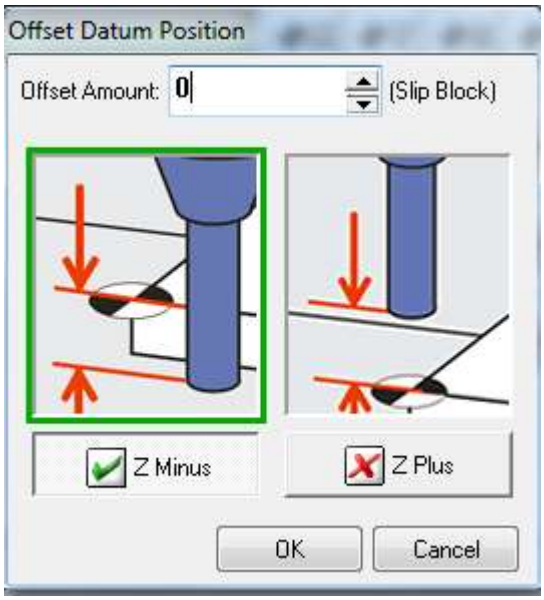


When you do this the X coordinate should show up as half the thickness of the tool bit, shown below.



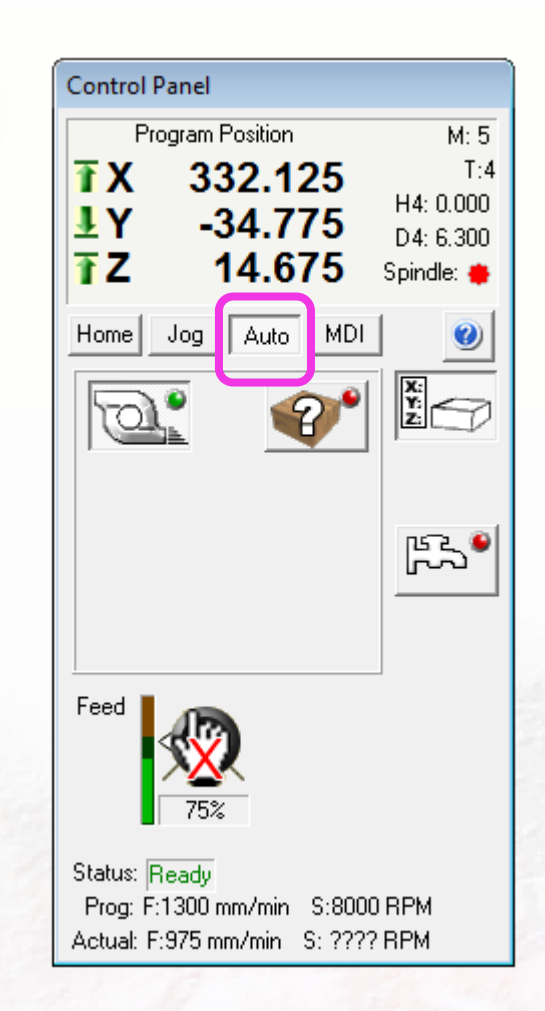
Do the same again for Y. Bring the tool bit to the front of the material and set the Y to zero accounting for the thickness of the tool and the position of the tool in relation to the material.

For setting Z, you bring the tool to touch the top of the material. This time because the tool isn't touching the side but the top of the material the offset is going to be set to 0, shown below.

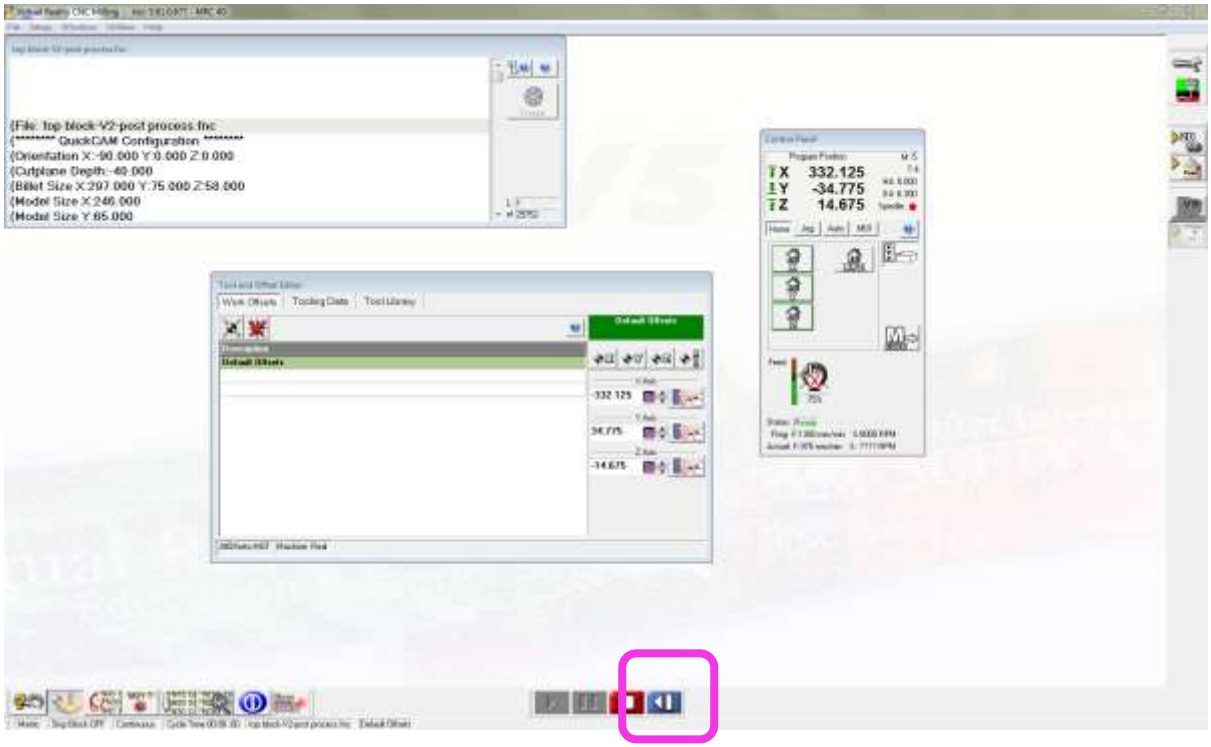


Starting milling

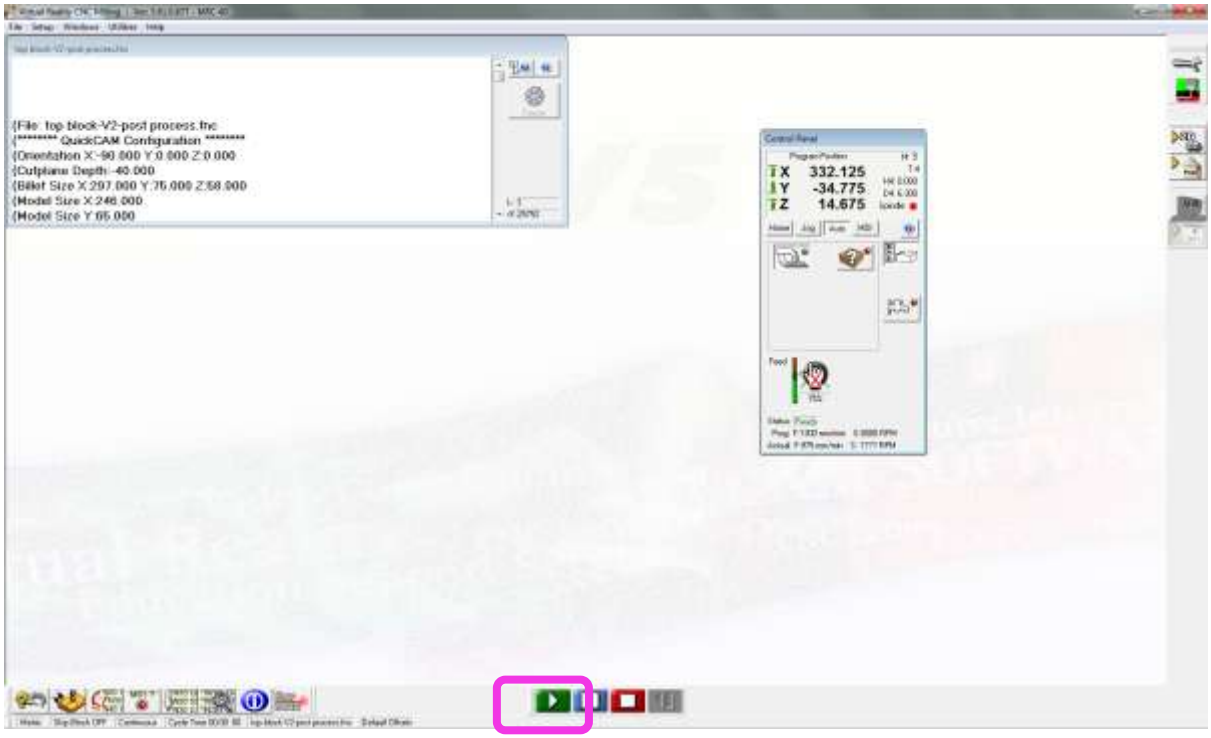
Go into the auto Mode in the control panel. You cannot mill without being in this mode.



Rewind button at the bottom of the page.



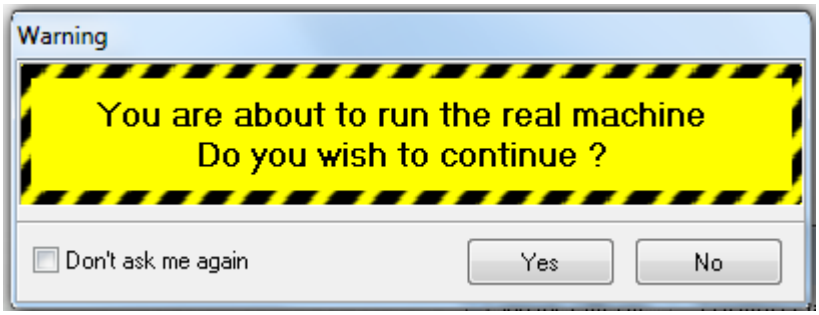
Play button shown below. Make sure the vacuum is on before the tool. Runs.



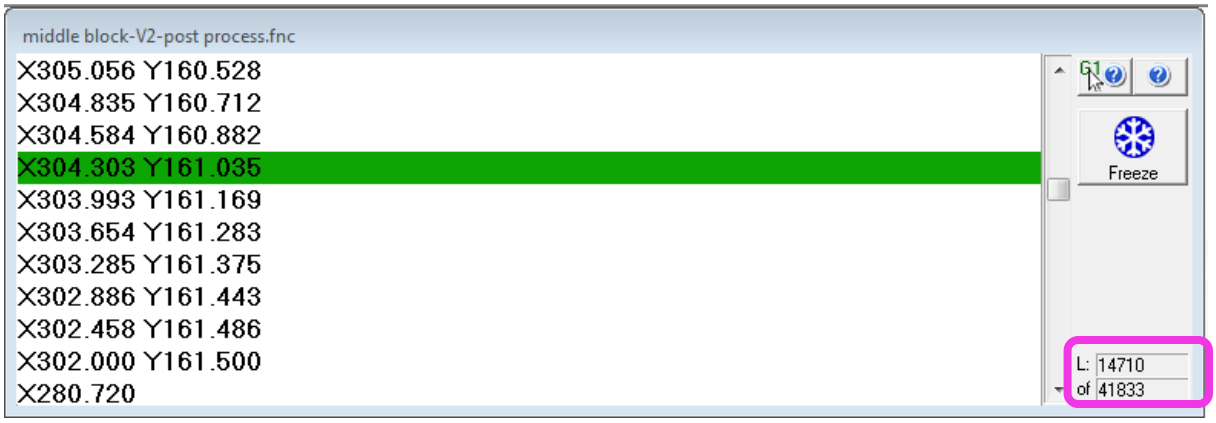
It may ask you to change the tool because it thinks there is another tool bit in the machine. It will run both Raster roughing and then Raster finish one after another.

It will ask if you want to run program now. Select ok.

It will ask again are you sure you want to run program now. Select Yes. This is the point to be sure the tool is securely in place the vacuum is on and the tool won't go into any material. Once you press yes keep your hand over the emergency stop button.



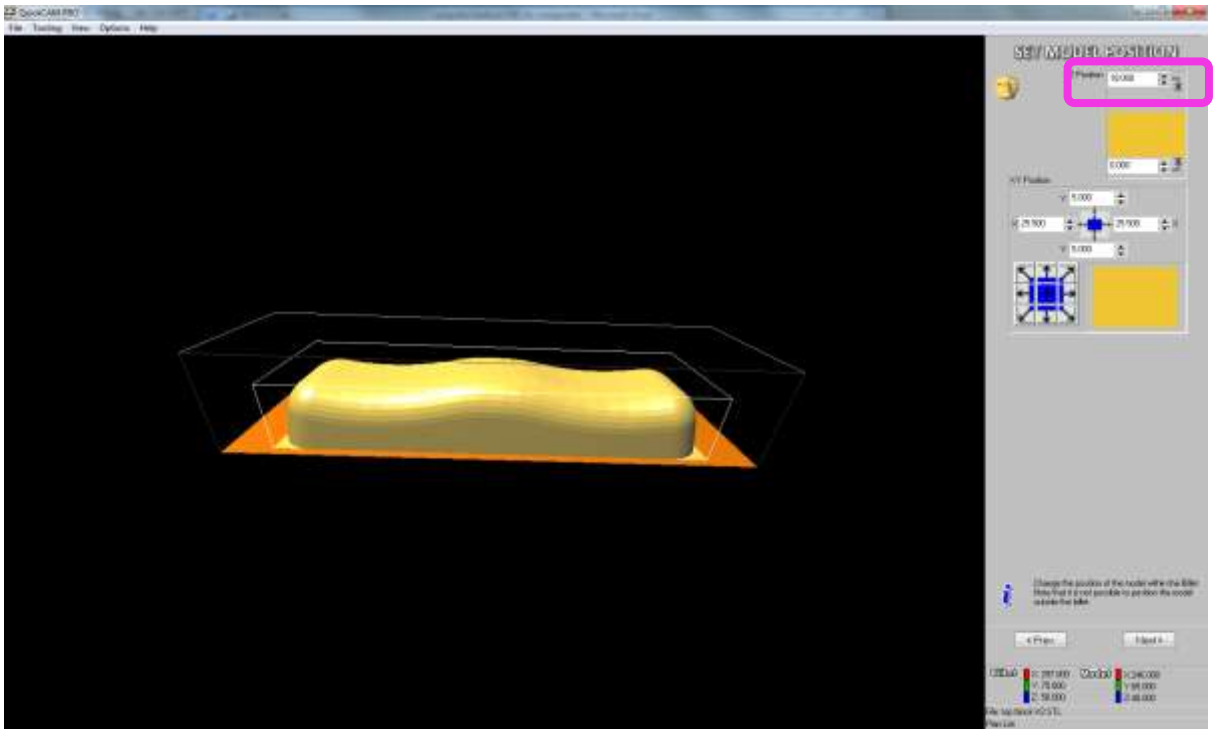
As it's running it will go through all the gcode line by line and show you what line it is currently on. This is important if you have to stop the machine. You can restart it where you stopped it. It also shows you how many lines of code there are.



To stop the machine there are 2 ways. The emergency stop button and the controls on the GUI, both shown below. Make sure you are always able to stop the machine quickly.

Exceptions to the other instructions

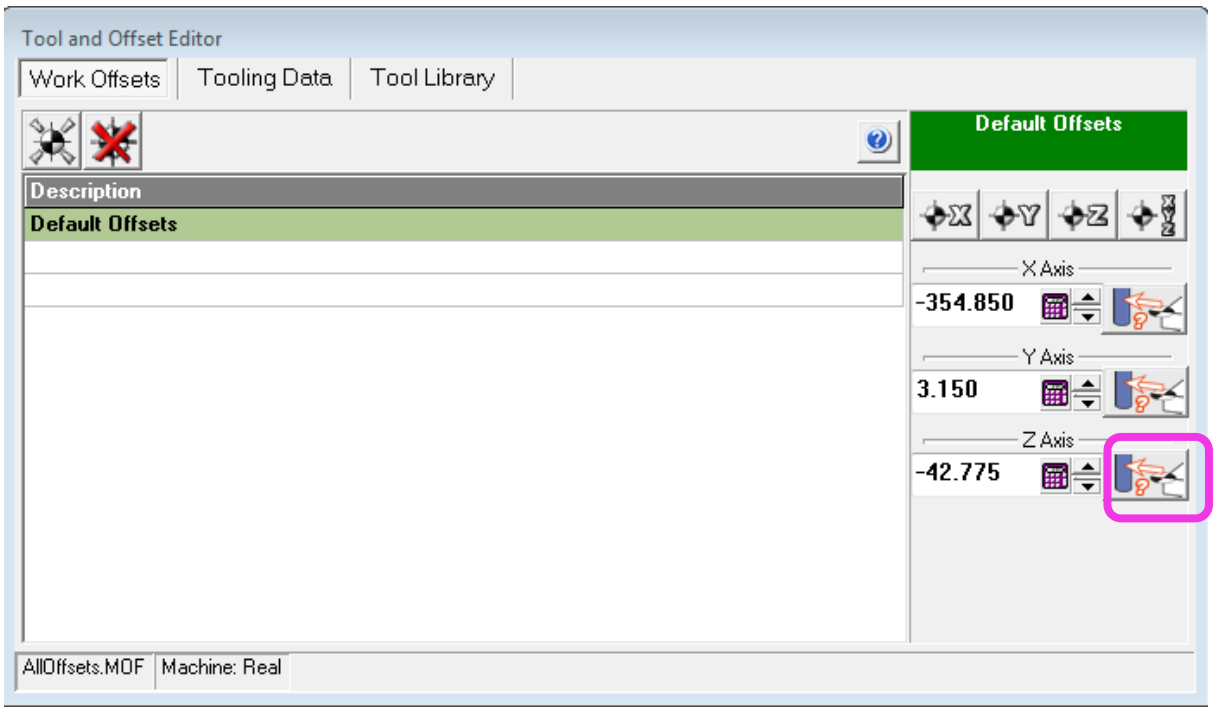
In one case the material thickness was greater than the model depth so I wanted to cut right down to the base of the material. To do this I adjusted the Z position by the difference between the material depth and model depth, which was 18mm.

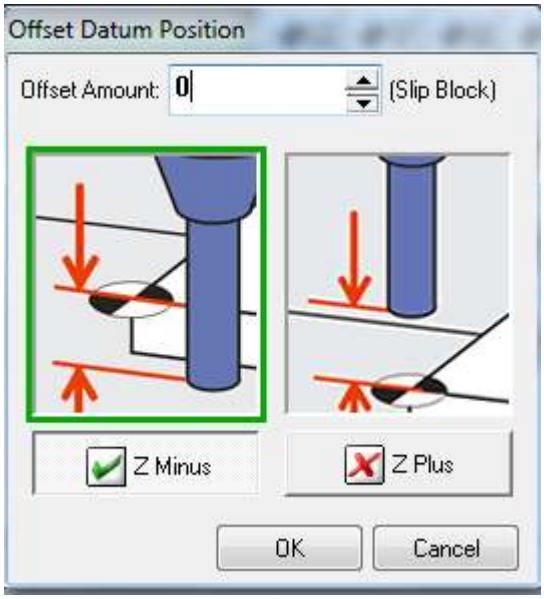


Stopping milling mid-way and starting again after resetting the tool Z depth

Took note of where I stopped the machine at @ 12905

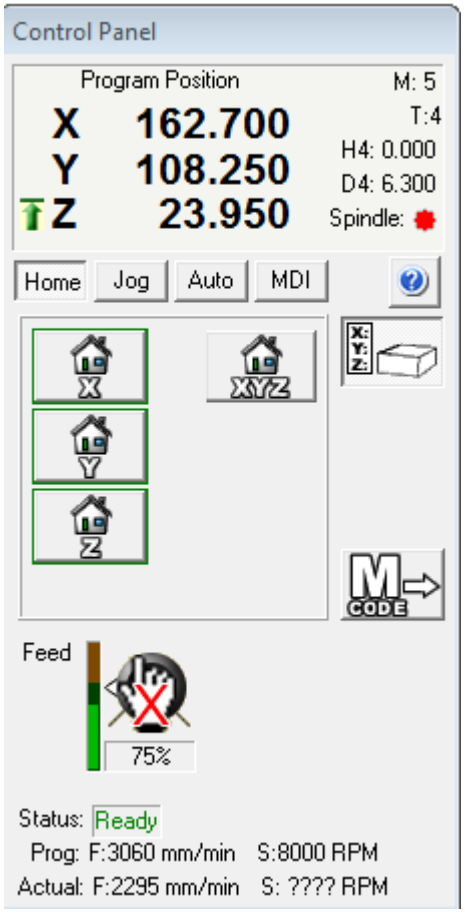
Lower the tool bit so there is more clearance. Then re-set the Z coordinate.





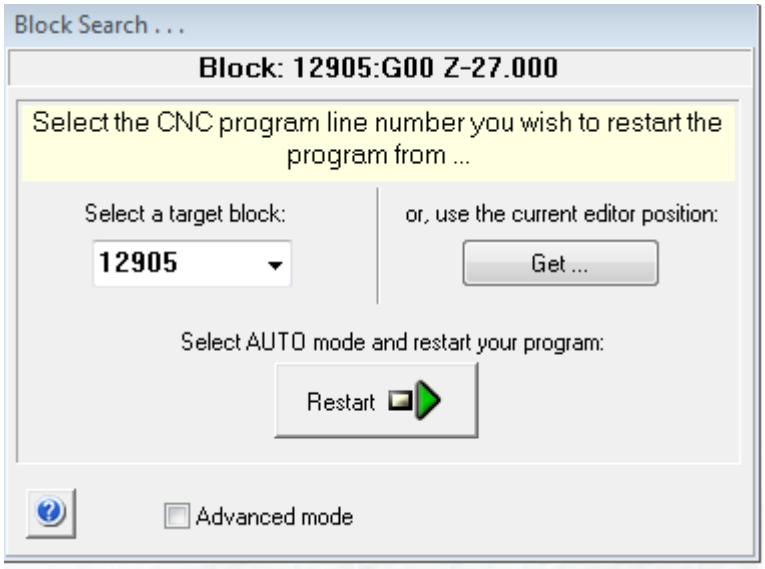
Then bring up zero as Z

Home to go to XYZ home.



Must be then restart. From the menu at the bottom of the page. Select the 'block search' icon.

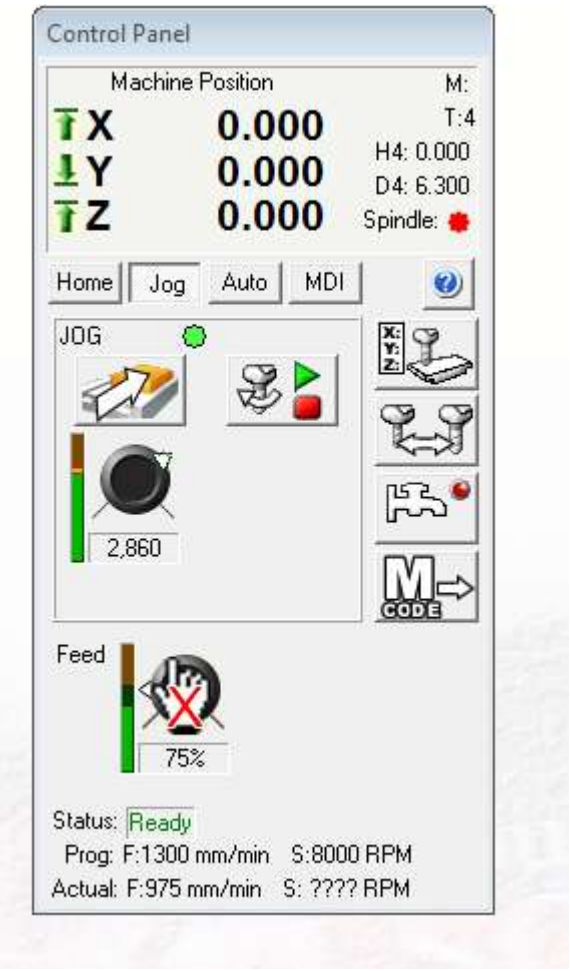
This opens up a pop up window shown below.



Must be in 'Auto' mode to be cutting.


Control Panel
Program Position
M: 5
X 354.850
T:4
Y -3.150
H4: 0.000
Z 23.950
D4: 6.300
Spindle:
Home
Jog
Auto
MDI
Feed
75%
Status: Ready
Prog: F:1300 mm/min S:8000 RPM
Actual: F:975 mm/min S: ???? RPM

Extra


Control Panel
Machine Position
M:
T:4
X 0.000
Y 0.000
Z 0.000
H4: 0.000
D4: 6.300
Spindle:
Home
Jog
Auto
MDI
JOG
2,860
CODE
Feed
75%
Status: Ready
Prog: F:1300 mm/min S:8000 RPM
Actual: F:975 mm/min S: ???? RPM