Instructions for machining components using the AXYZ CNC milling machine

CNC router: Room G17B (Grand Parade)

FABLAB BRIGHTON 2018

AXYZ Millennium Router Table Hand Console Operation

FILE KEY
Press to cycle through the list of jobs making the displayed job active.

START KEY
Press to start the active job. See below, RUNNING A JOB.

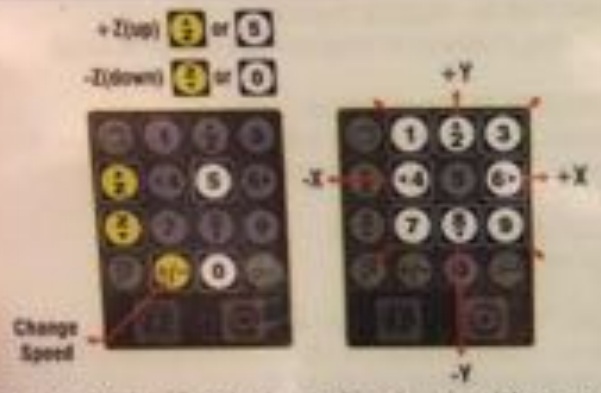
STOP KEY
Press to stop machine movement and pause the running job. Press START to resume job.

FUNCTION KEY
Access all of the machines functions. To access Function 13, press F,"1", "3" ENTER.

ENTER KEY
Pressing completes functions, or numeric value input. In job mode, holding ENTER displays job size. During the start sequence, press ENTER to select the starting shape.

TOGGLE KEY
In jog mode, toggles between FAST, MEDIUM and SLOW. Use it also to enter negative numbers and decimal points.

JOG MODE



The console is in JOG mode whenever it is not running a job or executing a function. All the numeric keys are also jog keys as shown. The key toggle between FAST, MEDIUM, SLOW and INCREMENT jog.

SETTING TOOL TIPS AND JOB PARAMETERS

- **Setting Tool Tips**

To set the tool tip for the tool at position 8
(i.e. tool 8 in the tool changer):
- Press F 25 8 ENTER
-When prompted, jog the head down so that the tool tip touches a common point such as the table surface
-Press ENTER to store the settings
Repeat the process for every tool bit that will be used.

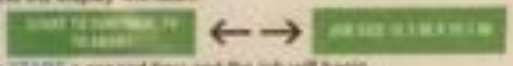
- **Set Surface & Lift Bottom/Top (Function84)**

Quick setup for 1" material, 1/16" waste cut, 1/4" lift top (Replace numbers in red with you actual measurements).
-Press F 84 ENTER. Lower the tool to the material surface.
-Press ENTER - F +/- 1 +/- 063 (-1.063)
-Press ENTER F 0 +/- 25 (0.25" lift top above material)
-Press ENTER to finish.

RUNNING A JOB

- **Starting a job**

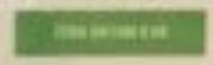
- Select the job File: Press FILE until the required job file appears on the hand conole display.
-The job is ready to go! Position the spindle at the lower left corner of the material using the jog keys.
- Press START. The head will lower to the "lift top" position, the spindle will start, and the display will flash:



- Press START a second time and the job will begin
- Press F 9 ENTER to abort the start sequence

- **Starting a 3D job**

-Similar to starting a 2D job (above), except after pressing START the first time, the display will show:



-Use the UP and DOWN keys to touch the surface of the material with the tip of the bit. If the material surface was set by Function 8 or Function 84, then simply press START to begin the job. Material surface has already been set.

- **Pausing and Aborting**

Press STOP to pause a running job. All machine movement will stop.
Restart the job by pressing START. To abort the job, press F 9 ENTER. The head will return to the start (origin) position.
- When the job is paused, all jog keys can be used to move teh head of the machine. Also, all functions marked with STOP key on the reverse can be used while the job is paused.

- **Restarting a job after replacing a bit**

-Pause and abort the job as described above. Note the shape number after pausing the job.
-Replace the bit, the jog keys can be used to positio the head accordingly.
-Reset the tool tip using Function 25
-Function 13 will return to the last job's origin
-Press START to begin the start sequence. The head will lower.
-Press ENTER to input shape number when prompted. This will start the job at the shape where it was previously paused.
For instance, press 1 2 ENTER to go to shape 12. Press +/- to step through the job one shape at a time.
-Press START and the job will begin at the selected shape.

AXYZ Millennium Quick Function Reference

FUNCTION 1 - *Set feed rate*
Set speed in in/min or mm/sec. Positive numbers 0.1 to max. speed.

FUNCTION 2 - *Spindle control*
Select:

Spindle AUTO
Spindle OFF
Spindle ON

When set to AUTO spindle will come on automatically when running a job. FUNC 6, or FUNC 17. It will always default to AUTO after these operations.

FUNCTION 3 - *Seek Z AXIS control*
Move Z axis up to seek the sensor, X and Y positions are set to 0.0.

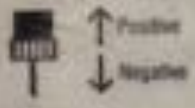
FUNCTION 5 - *Auxiliary Control*
This function controls the vacuum hold-down & chip extraction system automatically.
Control settings are: AUTO, OFF, ON

FUNCTION 6 - *Measured Move*

- Step 1: Enter X axis move distance ENTER
- Step 2: Enter Y axis move distance ENTER
- Step 3: Select HEAD UP or HEAD DOWN*
- Step 4: ENTER machine will move specified distance and direction.

* Ensure spindle is turned ON!

FUNCTION 7 - *Adjust Depth*
Adjust Lift Bottom by the specified measurement. Use inches or mm.



FUNCTION 9 - *Delete/Abort Job*
See Reverse.

FUNCTION 12 - *Seek X and Y Sensors*
Head will seek first the X axis sensor then the Y axis sensor. Origin will be set at the sensors. Soft stops will be enable.

FUNCTION 13 - *Seek Origin*
Head will return to origin. The origin will either be the start position of the last job run or origin set by FUNC 12 above.

FUNCTION 15 - *Plunge Speed*
Sets speed of vertical plunge into the material. Positive number 0.1 to max. Plunge speed. Use in/min or mm/sec.

FUNCTION 17- *Cut Rectangle*
Enter X and Y dimensions of a rectangle. As soon as the Y dimensions is entered the rectangle will be cut. Bit compensation must be set using FUNC 2.

FUNCTION 21 - *Repeat Job*
Set the number of job repetitions.

Step 1: Select job and press START
Step 2: Select FUNC 21
Step 3: Enter the number of job repetitions
Step 4: Press START to begin jobs.

FUNCTION 22 - *Digitizing Feature*
Digitizing of patterns or templates. See manual on usage.

FUNCTION 24 - *Disable Motors*
Disable and resets all motors and motor drives. All servo drive systems have this feature. Most steppers systems do not.

FUNCTION 25 - *Set Tool Tips*
See reverse.

FUNCTION 35 - *Store machine settings*
Stores following settings: F4, F8, F1, F15. This gives the flexibility to save different settings for different material thicknesses and other material properties. 16 F35 settings available. Ex. F35.1, F35.2.....F35.16

FUNCTION 36 - *Retrieve machine settings*
This function retrieves stored settings. 16 F36 settings available. F36.1, F36.2.....F36.16

FUNCTION 84 - *Quick Setup*
See Reverse.

FUNCTION 92 - *Clear job memory*
All jobs files will be erased from memory.

NOTE: ● indicates that the function can be adjusted any time during the job. Simply press the Stop buttom, and carry out the function.

"Functions shown in red represents Primary table functions"

AXYZ

1. Setup material and clamp

Make sure machine is off!

Clamp material, ensuring that sacrificial material is underneath your material, and make sure your material is square and lined up with the sacrificial front edge.

2. Setup machine

Twist Emergency Stop to check it's out and not activated.

Press the green button to turn on the machine.
Seek X/Y origin - YES - press enter to start.

Start spindle (may take time to warm up).

Timer counts down 10 mins whilst warming up spindle.

If you want to the stop the 10 min warmup, press the RED button on the keypad which has the spinning disc on it (NOT the emergency stop!).

Use ruler to measure the distance the clamp protrudes e.g. 20mm within the edges. Subtract this from the job size measurements (the clamps at the back don't matter).

Press 'F' and 3 and enter to set XY, then move the gantry manually using the 2 (towards back), 4 (left), 6 (right), 8 (towards front), 1,3,7,9 (diagonally).

Note that +- will speed up or slow down the movement of the gantry.

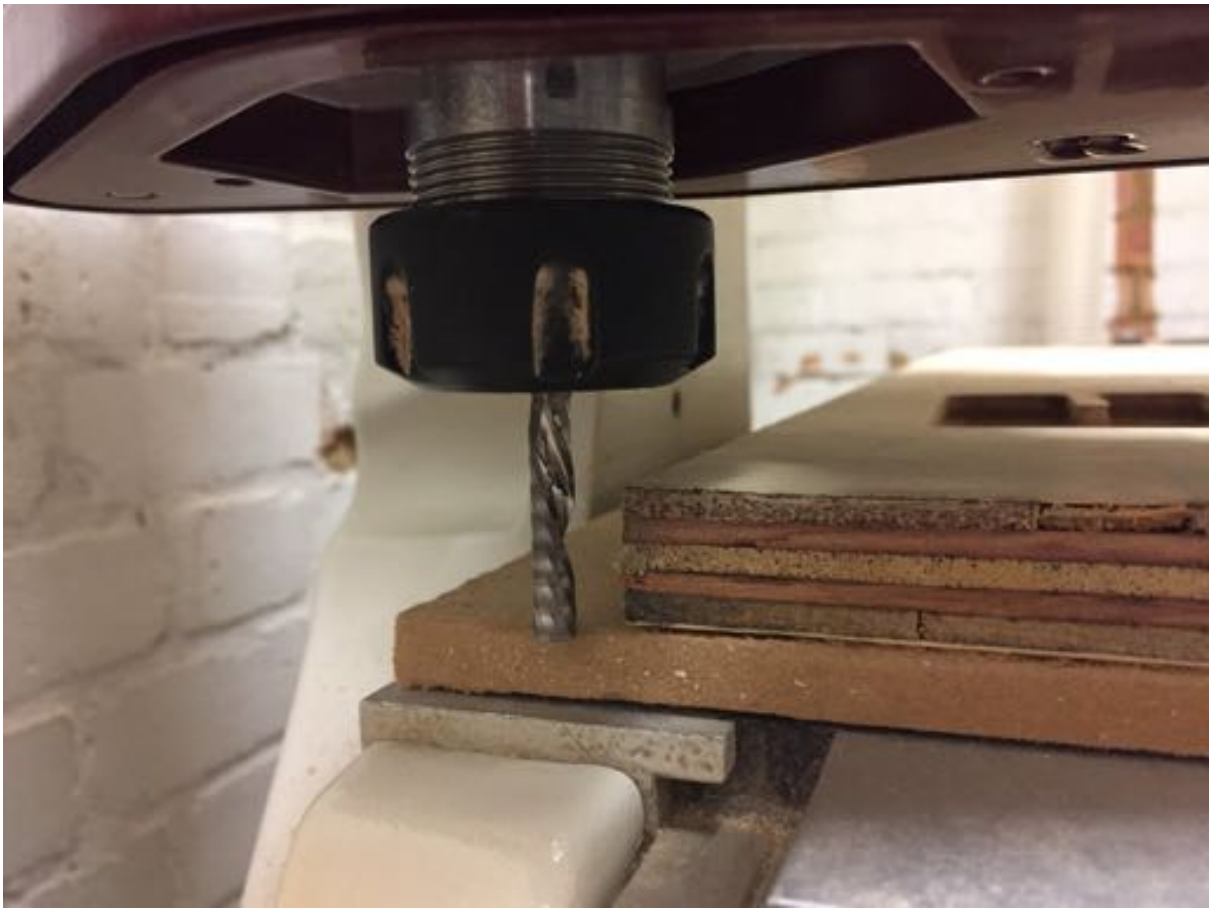
After you position XY to where you want 0 to be correctly, then Press 'F' + 3 and enter again to set the XY as zero. Check that the zero for X and Y is where it should be.

Note that 'F' and 13 will take the gantry back to XY 0.

Press 'F' and 84 and enter.

Set Surface: move the gantry down (use an appropriate speed! Using +-). Spin the spindle by hand as you move the gantry down using increments (very small!) until the cutting tool just starts to press into the surface. Then press enter.

Lift bottom: move the gantry up and over away from the edge, then down to your lower z limit to about half way through the sacrificial material. This will restrict the machine to that depth. Note that when you're there, z should be lower than our maximum cut depth -18.5mm we set earlier. Then press Enter to accept that value. Note that when prompted for the lift bottom, you can press 'F' and then enter the z-bottom limit manually (this might be needed if you can't actually move the gantry down because the workpiece is in the way).



Lift top: set the top limit well above the work surface (about 20mm) which is clearly above the 6mm safe z. Press Enter when done.

Press 'F' 13 and the machine will move to 0,0 some distance above the workpiece.

Then bring the guard down by turning the switch dial to the left to bring it down. The machine will hiss at you like it is tsking at your bad jokes.

3. VCarve Pro software (to generate tool path).

Start - Programs - V-Carve Pro.

File - Open (find .dxf file).

Job setup:

Width: X 1000 (if there is nothing blocking X)

Height: Y 980 (needs to be clear of the clamps at the front which are in the way)

Material (Z): 18mm (from our own measurements). Make sure that you zero the TOP option (the top radio button).

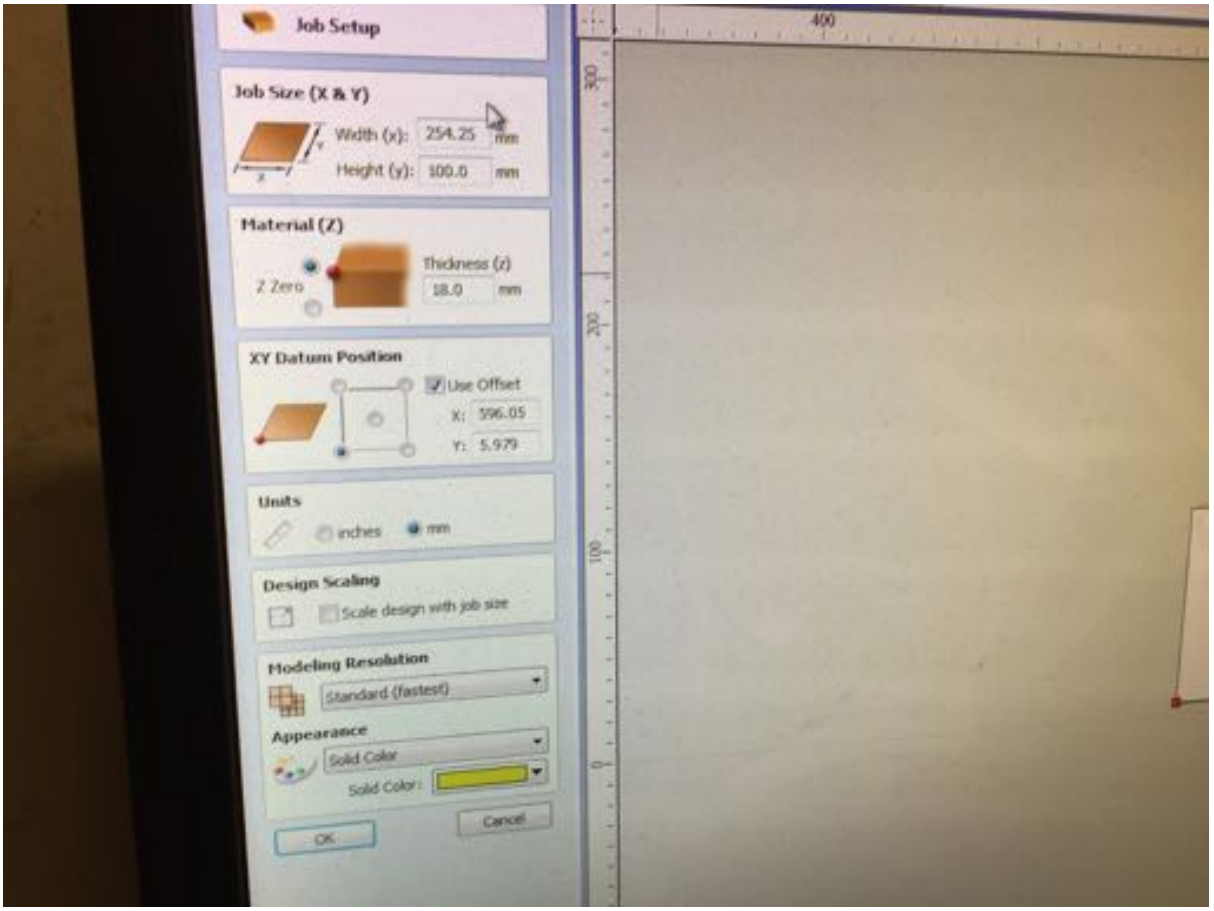
XY Datum Position: UNCHECK Use Offset (it will go to 0,0), and make sure you select the BOTTOM LEFT option.

Units: mm

Design Scaling: keep the option unchecked.

Modelling resolution: Set to STANDARD, and Appearance: SOLID COLOUR.

Click OK.



2D view control

Switch to toolpaths tab (it's the last icon on the right).

Toolpath Operations

First option/icon is Profile Toolpath.

Cutting depth

Start depth: 0mm

Cut depth: add 0.5mm to the depth of your material e.g. 18.5mm for material thickness of 18mm. This allows the cut to go beyond the material so it will go into the sacrificial material below.

Make sure the 'Show advanced toolpath options' option is selected.

Tool

Select - Metric - End Mills - 6mm

Keep the default settings but could play around with cutting parameters and slower speeds.

Click OK

Machine Vectors

Direction: Outside/right, Conventional

ALLOWANCE OFFSET: 0

Vector startpoint: leave unchecked

Layers

Need to consider outside vs inside cuts - for profile cuts and pocketing - TBC

Last pass
Don't tick separate last pass - would improve quality but requires changing tool.

Tabs
Constant number, start with about 6.

Tick add tabs to tool path
Length 6mm
Thickness 2mm

Select the drawing on your drawing view, then go to Edit Tabs, press Add Tabs and then move the tabs on the drawing to where you want them

SafeZ - is the same as Zjog in fabmodules - 6mm.

Type name of file

Click Calculate

Warning pop up - incase you don't have sacrificial layer warns it would cut into machine.
Click OK

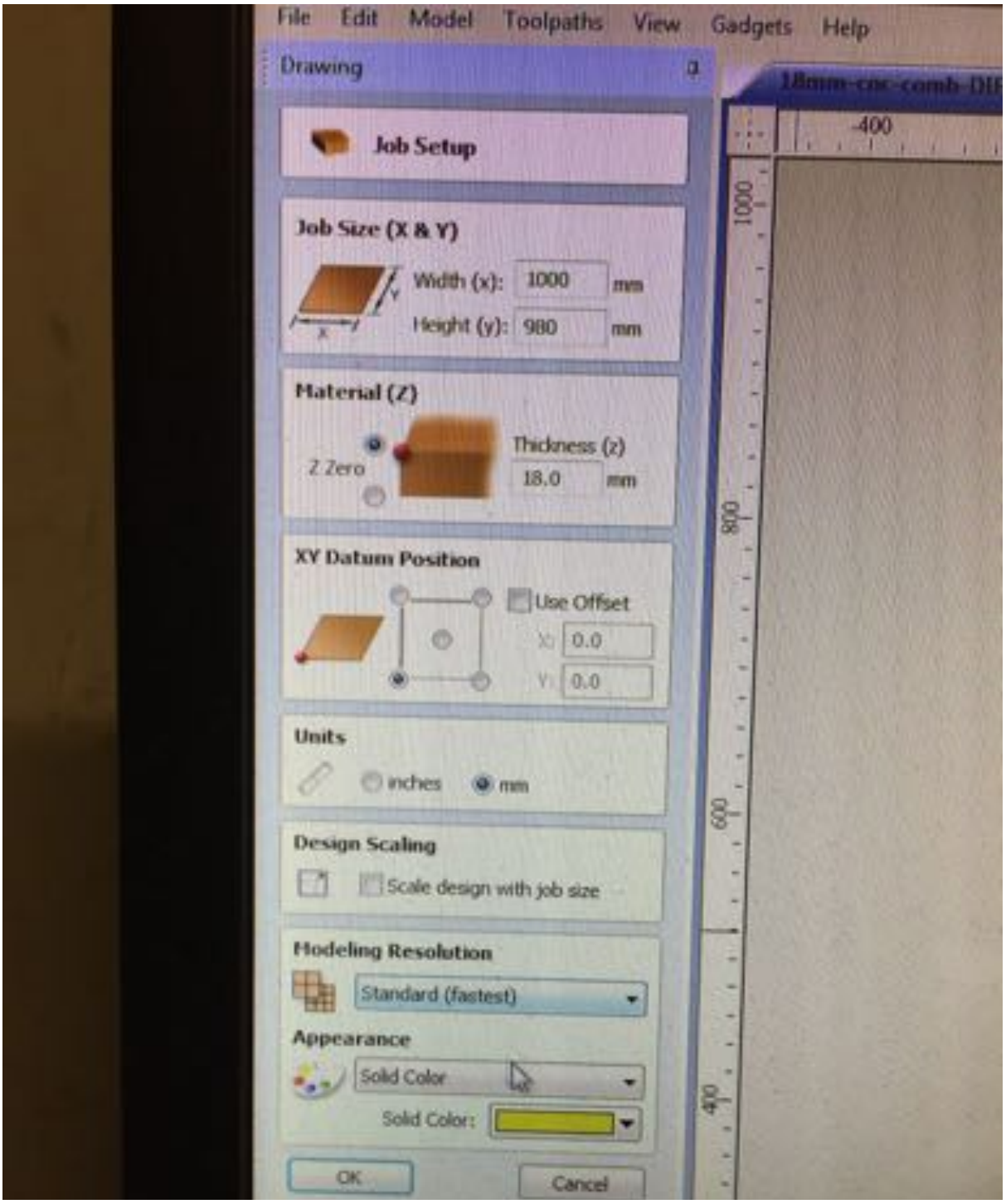
Then press Preview All Toolpaths and it gives you a rendered view and animation of the piece and you can see the toolpath.

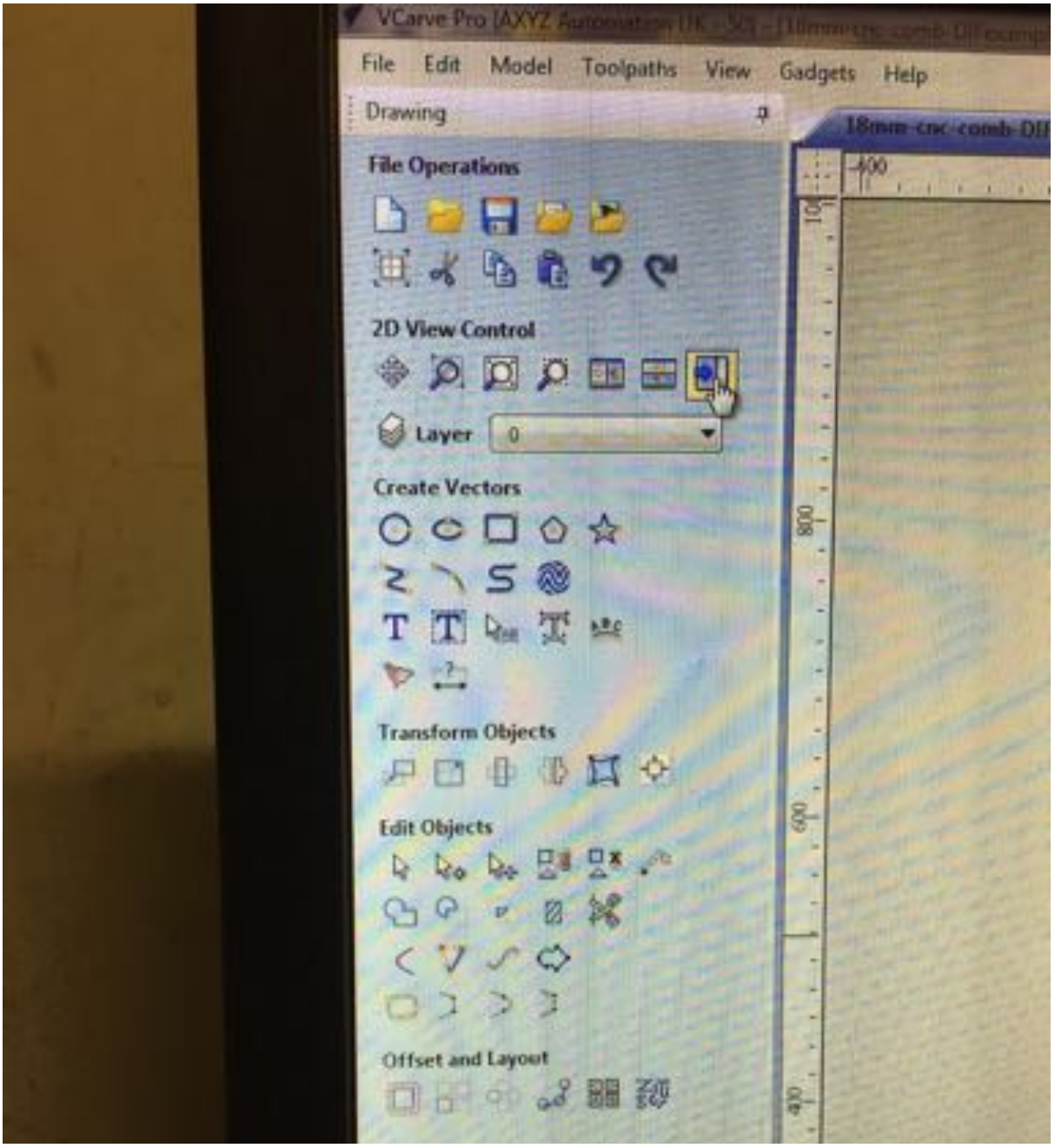
Press close.

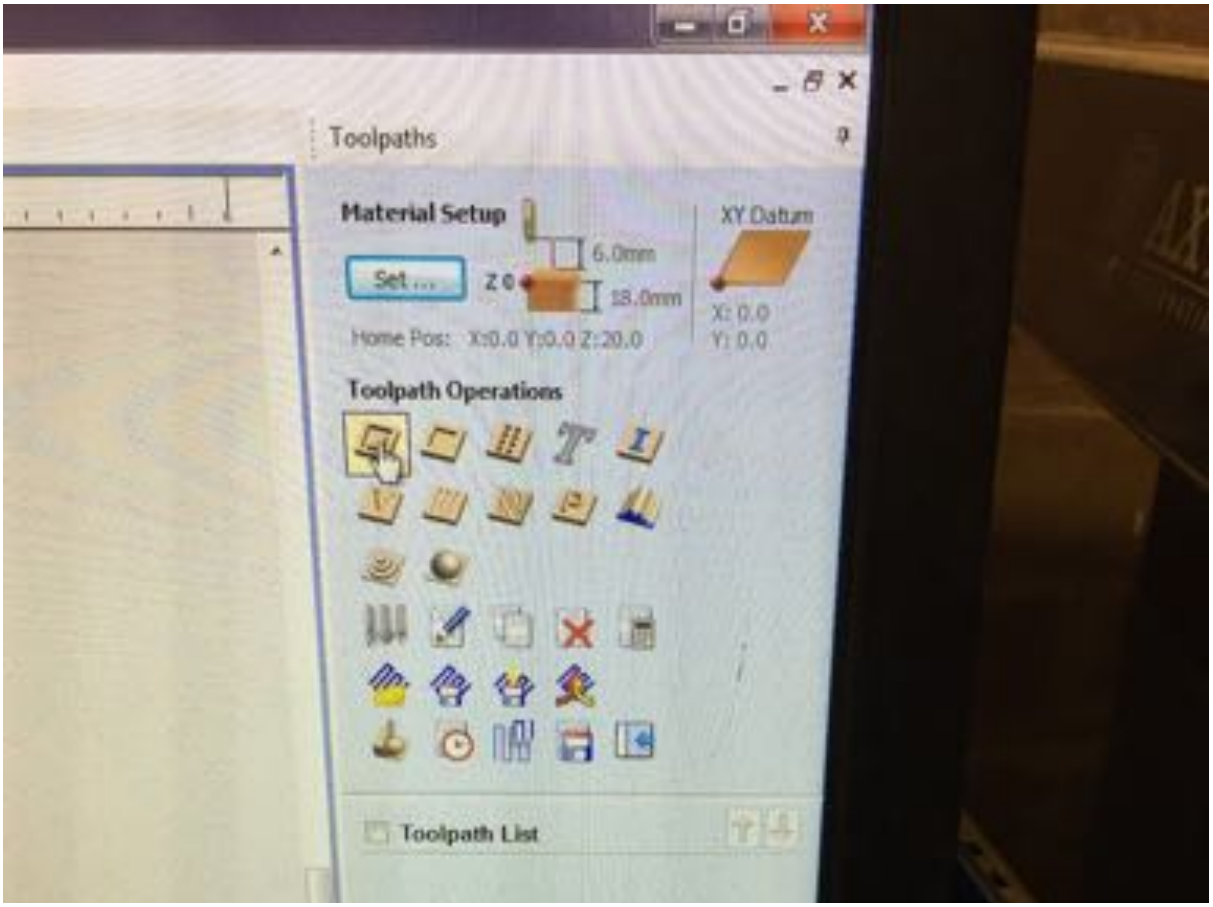
Select the profile (we called ours Profile 1).

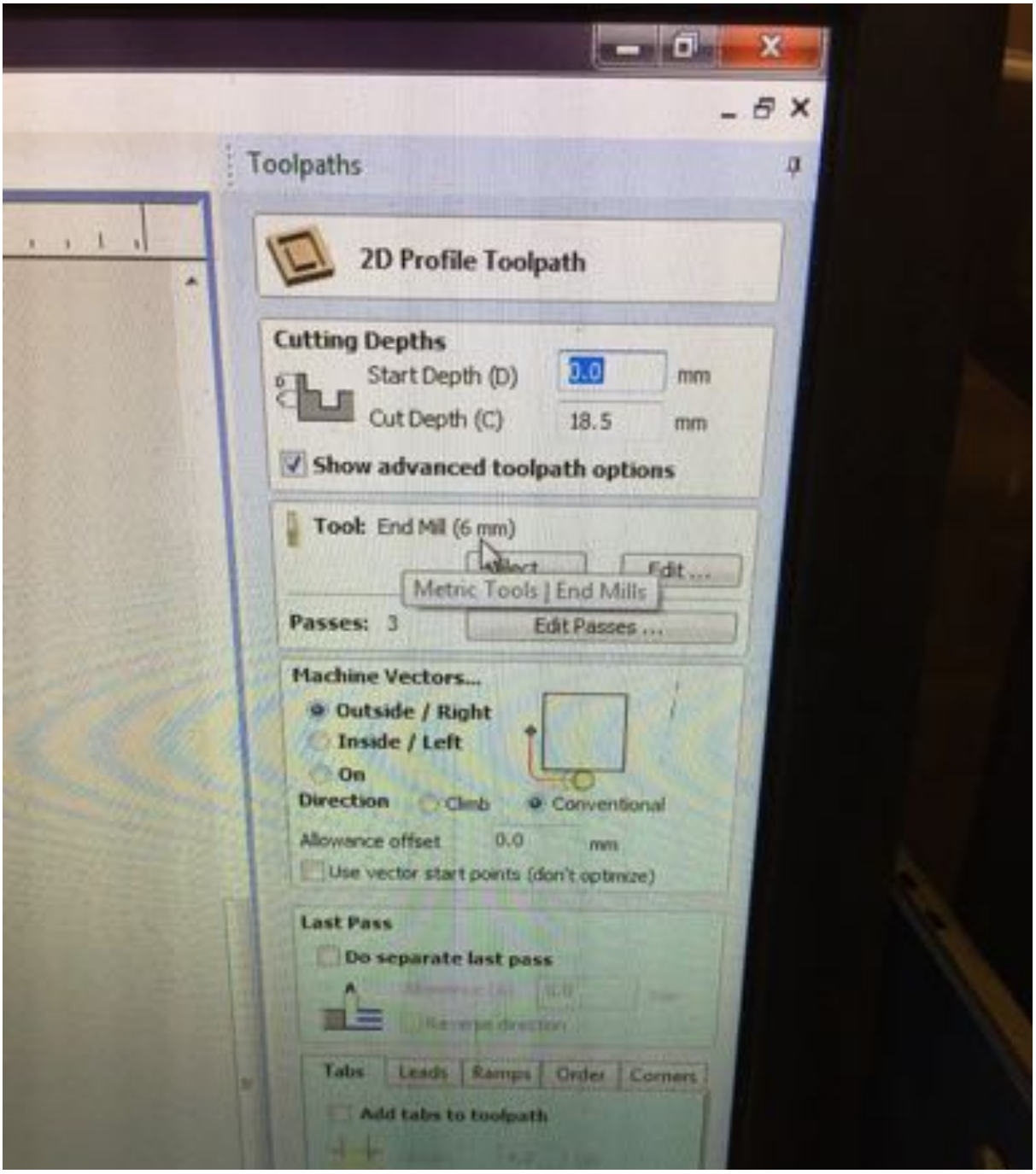
Click on Save Toolpath which is the 2nd from the bottom right.

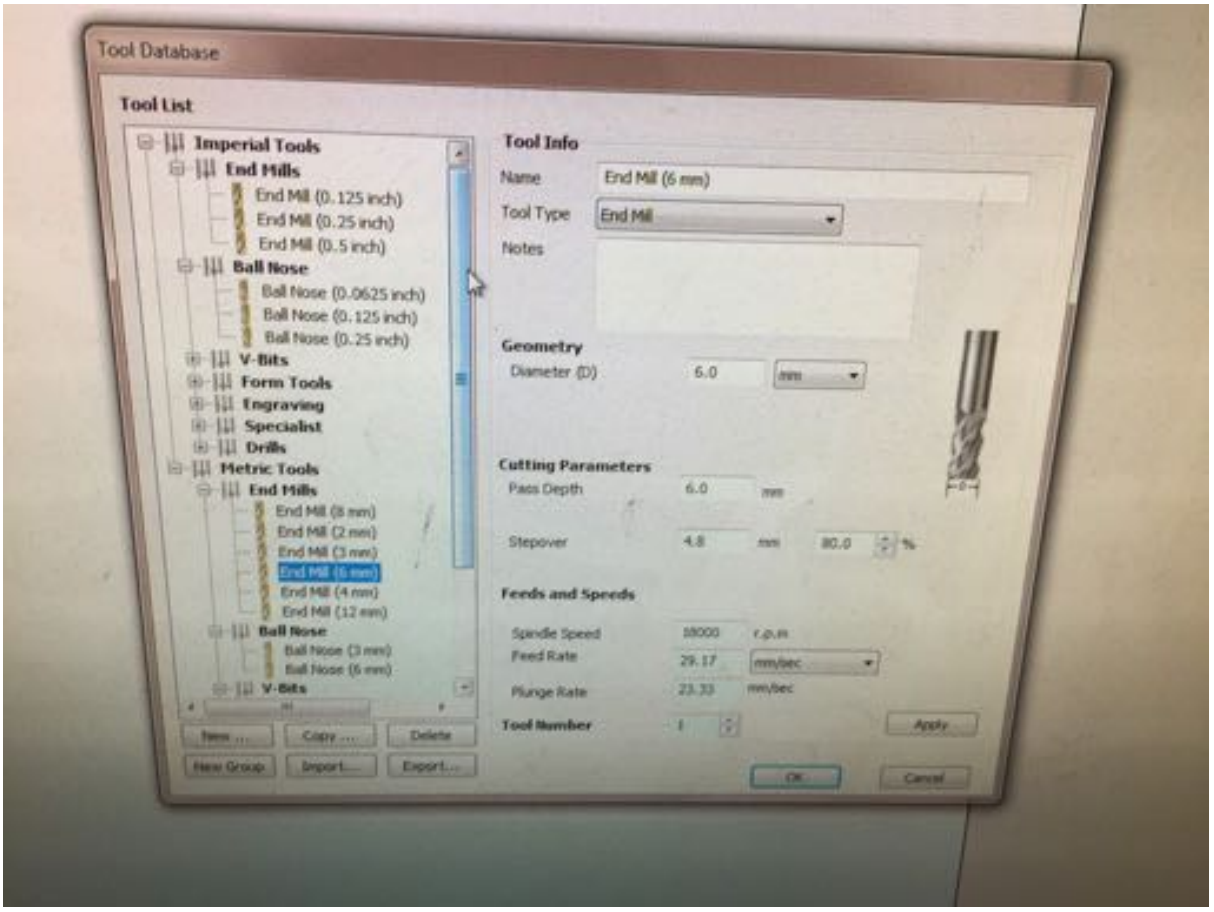
Keep the options unchecked (relating to output), then for the post processor chose the AXYZ Arcs (mm) (*.nc) and select Save Toolpath/s.

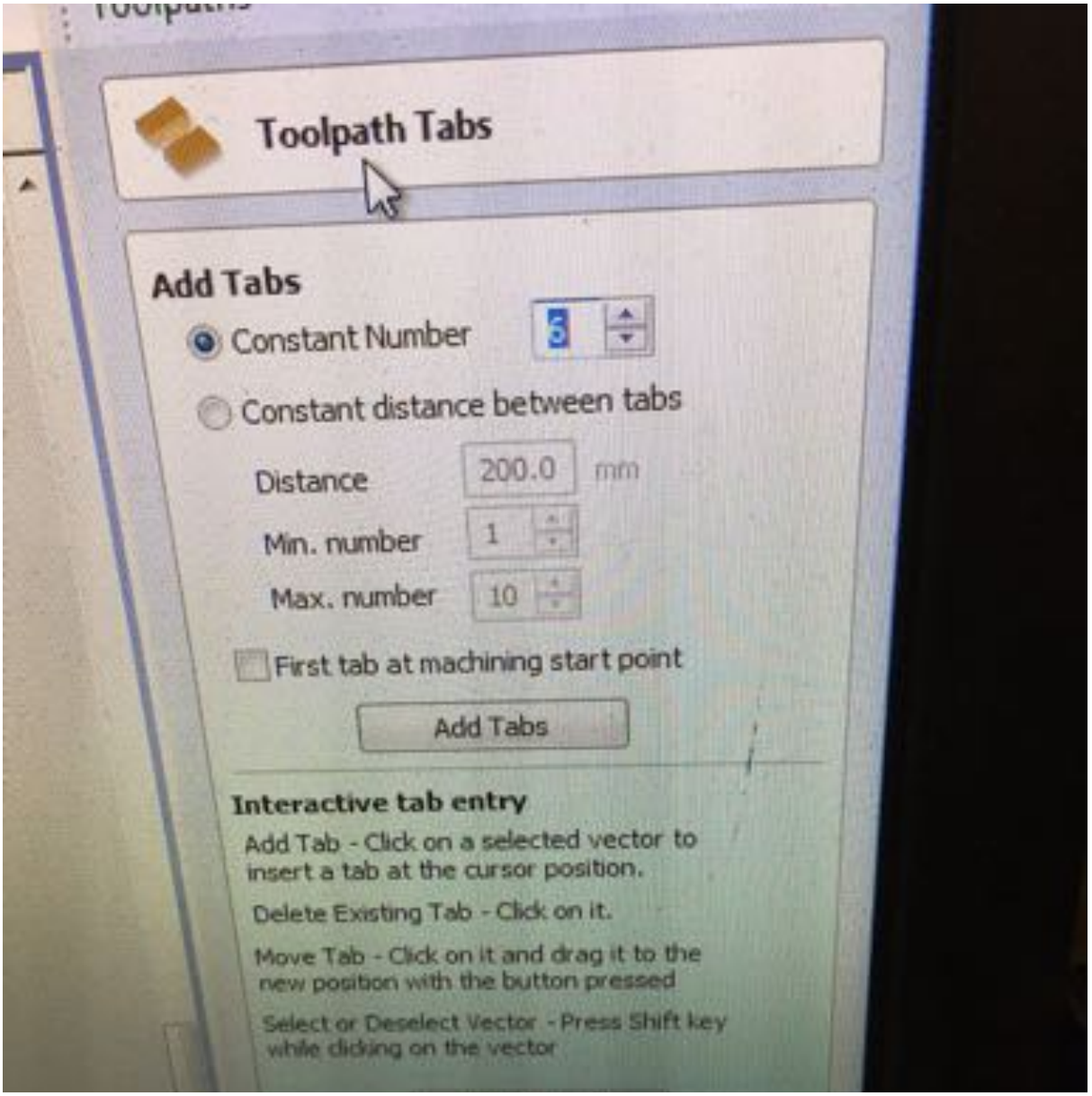

File Edit Model Toolpaths View Gadgets Help
Drawing
Job Setup
Job Size (X & Y)
Width (x): 1000 mm
Height (y): 980 mm
Material (Z)
Z Zero
Thickness (z)
18.0 mm
XY Datum Position
Use Offset
X: 0.0
Y: 0.0
Units
inches
mm
Design Scaling
Scale design with job size
Modeling Resolution
Standard (fastest)
Appearance
Solid Color
Solid Color:
OK
Cancel
400
1000
800
600
400

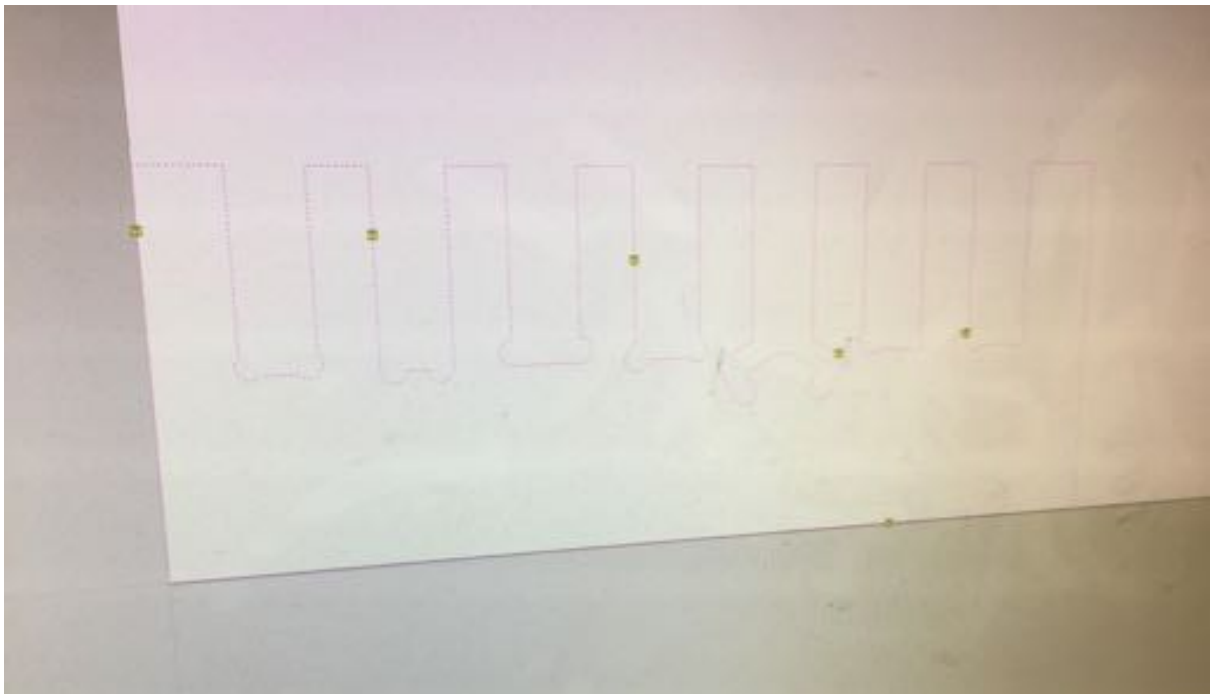

File Edit Model Toolpaths View Gadgets Help
Drawing
File Operations
2D View Control
Layer
0
Create Vectors
Transform Objects
Edit Objects
Offset and Layout

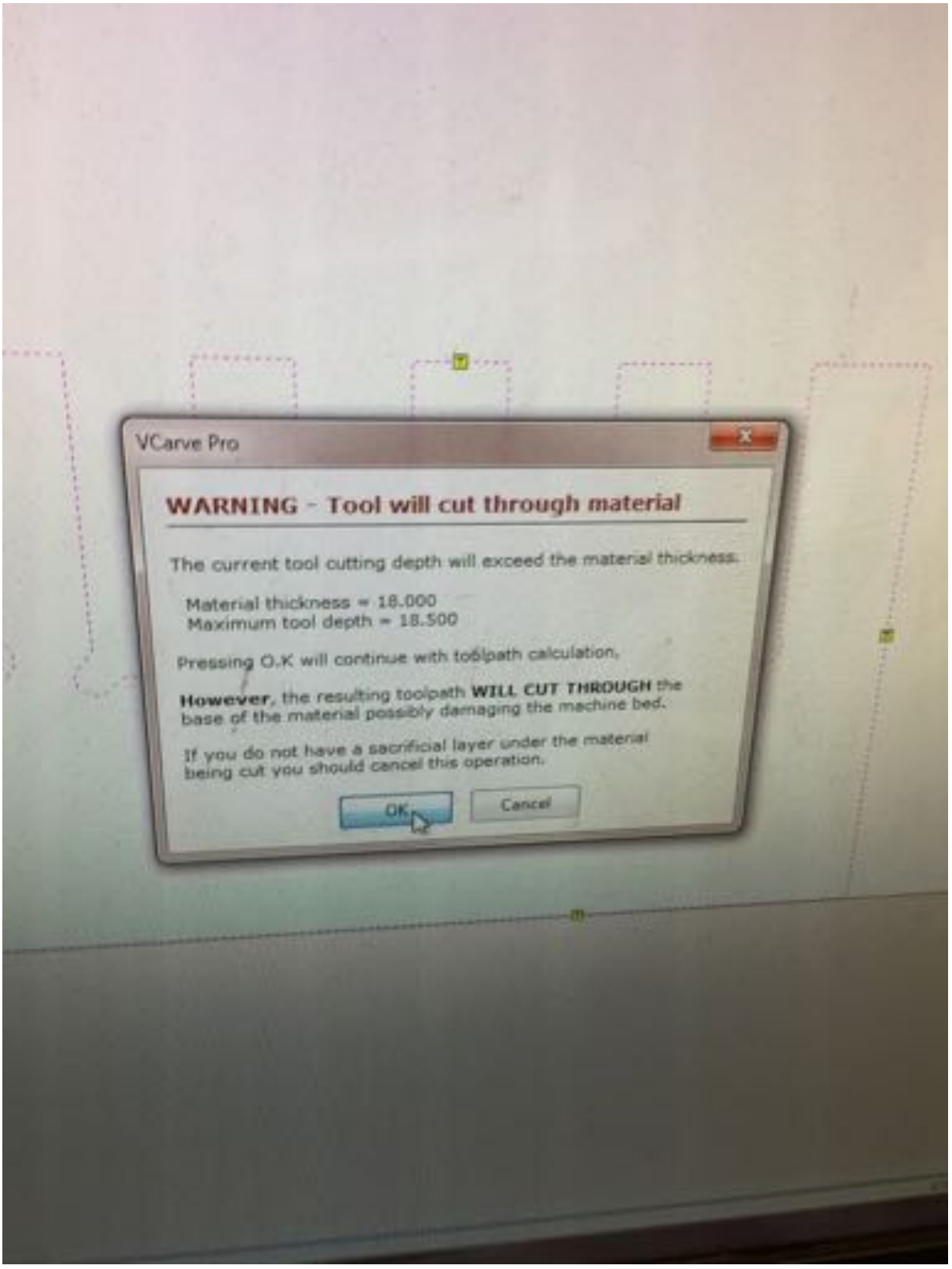

Toolpaths
Material Setup
Set ...
Z 0
6.0mm
18.0mm
Home Pos: X:0.0 Y:0.0 Z:20.0
XY Datum
X: 0.0
Y: 0.0
Toolpath Operations
Toolpath List

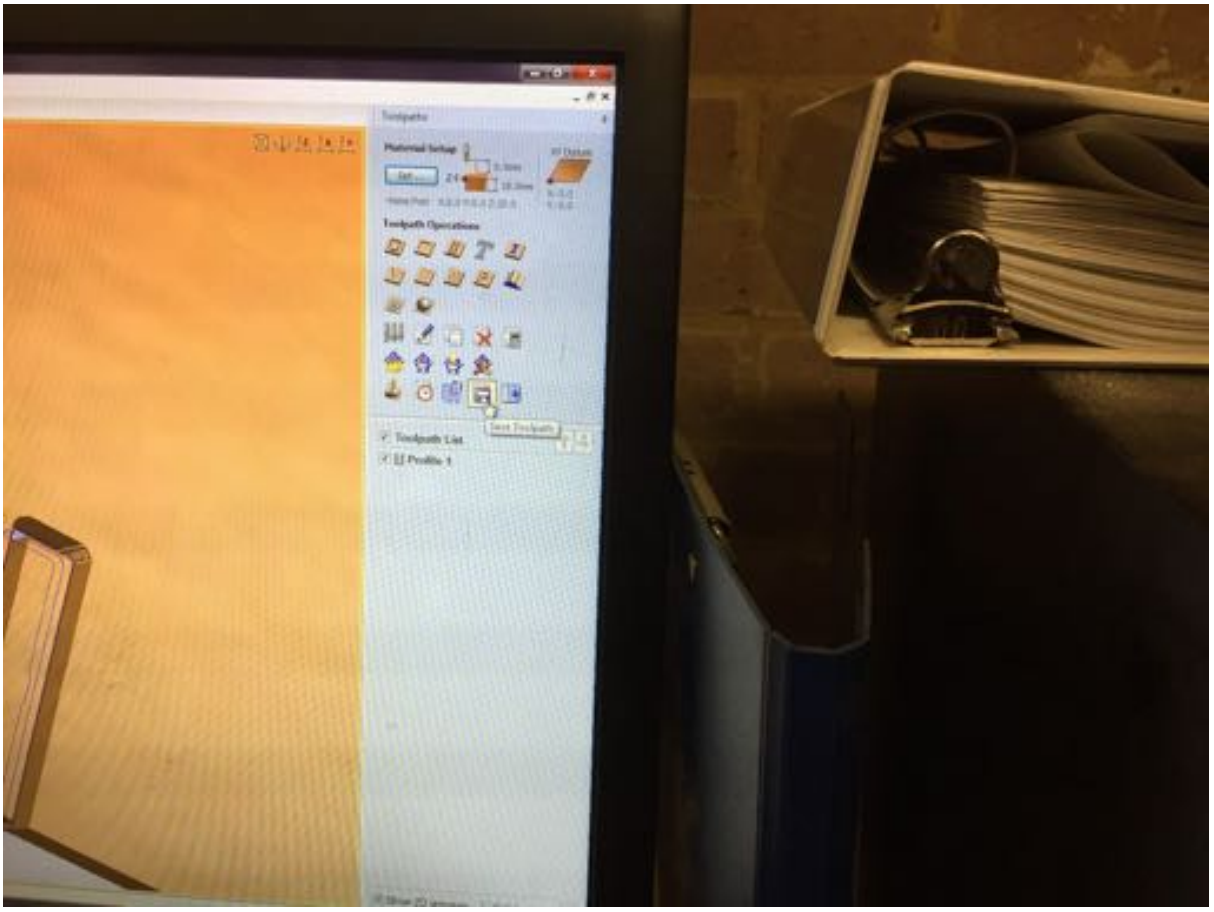

Toolpaths
2D Profile Toolpath
Cutting Depths
Start Depth (D)
0.0
mm
Cut Depth (C)
18.5
mm
Show advanced toolpath options
Tool: End Mill (6 mm)
Edit ...
Metric Tools | End Mills
Passes: 3
Edit Passes ...
Machine Vectors...
Outside / Right
Inside / Left
On
Direction
Climb
Conventional
Allowance offset
0.0
mm
Use vector start points (don't optimize)
Last Pass
Do separate last pass
Tabs
Leads
Ramps
Order
Corners
Add tabs to toolpath


Tool Database
Tool List
Imperial Tools
End Mills
End Mill (0.125 inch)
End Mill (0.25 inch)
End Mill (0.5 inch)
Ball Nose
Ball Nose (0.0625 inch)
Ball Nose (0.125 inch)
Ball Nose (0.25 inch)
V-Bits
Form Tools
Engraving
Specialist
Drills
Metric Tools
End Mills
End Mill (8 mm)
End Mill (2 mm)
End Mill (3 mm)
End Mill (6 mm)
End Mill (4 mm)
End Mill (12 mm)
Ball Nose
Ball Nose (3 mm)
Ball Nose (6 mm)
V-Bits
New ...
Copy ...
Delete
New Group
Import...
Export...
Tool Info
Name
End Mill (6 mm)
Tool Type
End Mill
Notes
Geometry
Diameter (D)
6.0
mm
Cutting Parameters
Pass Depth
6.0
mm
Stepover
4.8
mm
80.0
%
Feeds and Speeds
Spindle Speed
18000
r.p.m
Feed Rate
29.17
mm/sec
Plunge Rate
23.33
mm/sec
Tool Number
1
Apply
OK
Cancel


Toolpath Tabs
Add Tabs
Constant Number
5
Constant distance between tabs
Distance
200.0
mm
Min. number
1
Max. number
10
First tab at machining start point
Add Tabs
Interactive tab entry
Add Tab - Click on a selected vector to insert a tab at the cursor position.
Delete Existing Tab - Click on it.
Move Tab - Click on it and drag it to the new position with the button pressed
Select or Deselect Vector - Press Shift key while clicking on the vector




VCarve Pro
WARNING - Tool will cut through material
The current tool cutting depth will exceed the material thickness.
Material thickness = 18.000
Maximum tool depth = 18.500
Pressing O.K will continue with toolpath calculation.
However, the resulting toolpath WILL CUT THROUGH the base of the material possibly damaging the machine bed.
If you do not have a sacrificial layer under the material being cut you should cancel this operation.
OK
Cancel



4. AXYZ DNC software - sending toolpath to machine

Click on AMC File Mode and Select File (find your .nc file saved from step 3). Click Send File.

5. Cut

Go to the CNC machine and press the yellow folder button (top left) which will cycle through the files to be machined, and find your .nc file (but don't do anything yet).

Turn on the muck sucker extraction.

Then goto the CNC again and hit the Green IO button on the keypad (NOT the main green ON button). It will start! Then make sure you close the gates, put on your PPE and stand behind the gates.

6. Aftermath

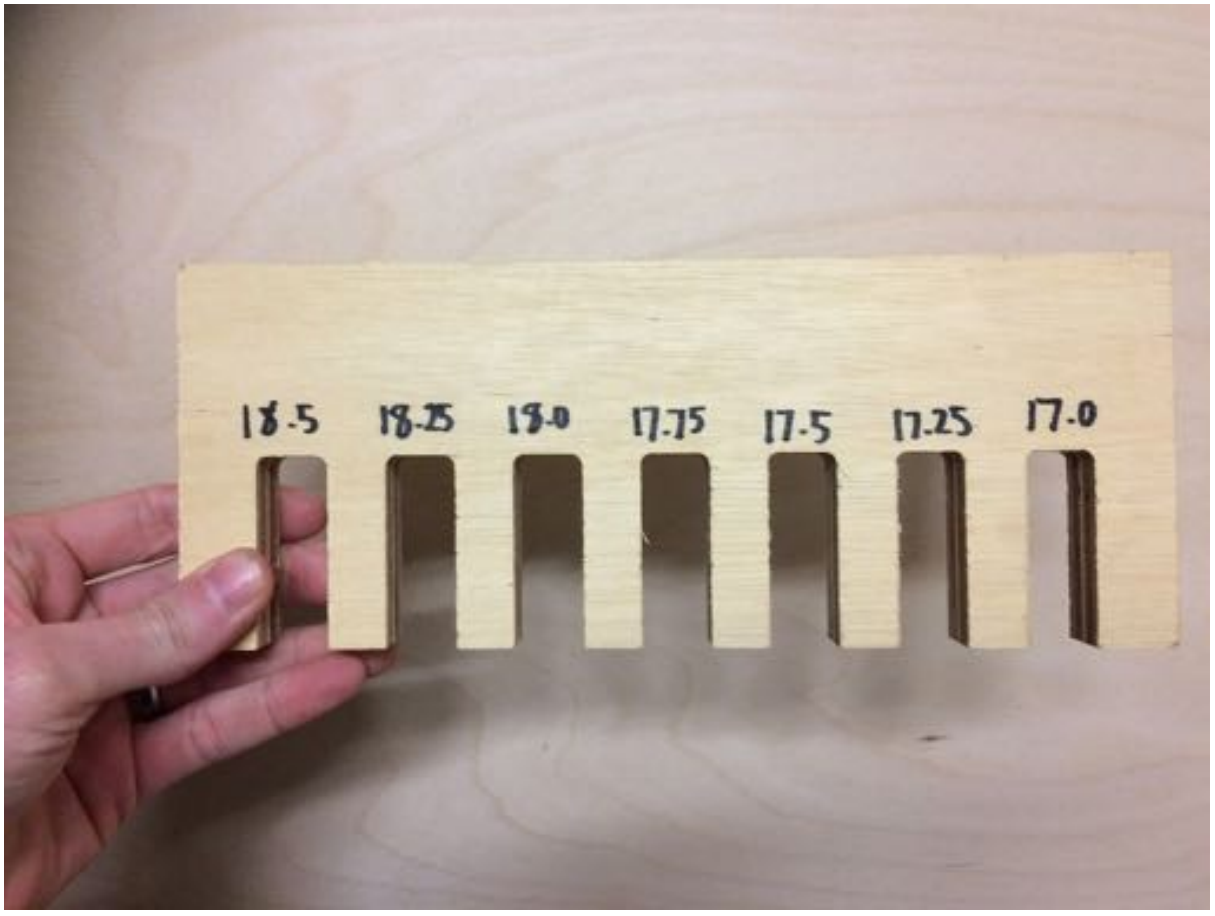
Once the cutting has finished, turn the machine off, gnaw at the part like a beaver with your two front teeth until you're able to take a full bite at the part like an apple, or you can cut it out with a chisel and hammer if that's easier.

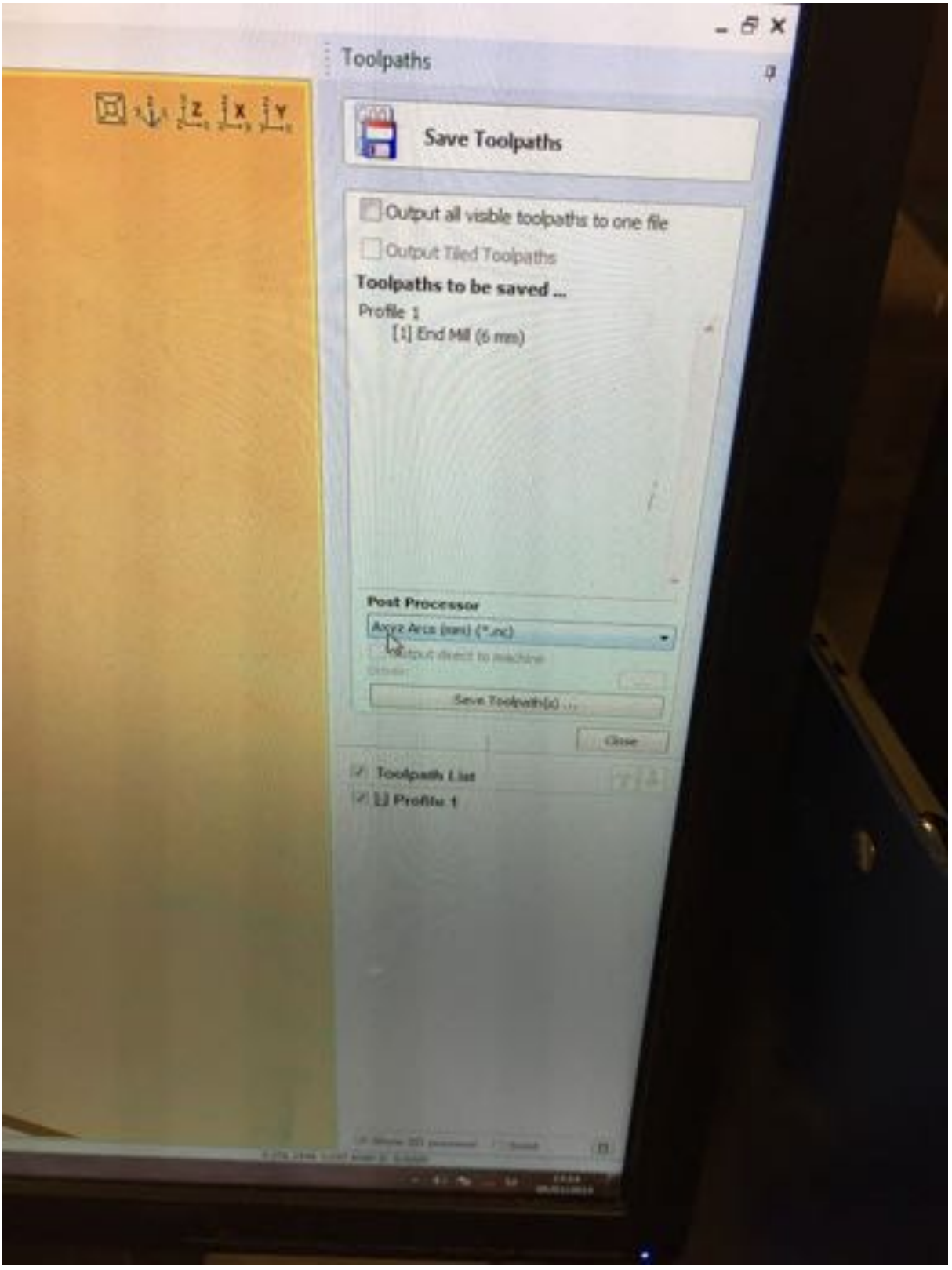
From our comb, here's a table of fits we achieved with the 18mm ply:

Comb slot width (as drawn mm)	Comb slot width (as measured mm)	Fit type	Notes
17.00		Forced fit	High interference shrink fit requiring considerable force to assemble with a mallet
17.25		Driving fit	Medium interference requiring light tapping to assemble with a mallet

17.50		Press fit	Light interference assembling can be done by pressing by hand
17.75		Location fit	Very close clearances for precise accuracy requirements, which can be assembled without force
18.00		Sliding	Minimal clearances for high accuracy requirements, which can be easily assembled and slide together freely
18.25		Easy running	Moderate clearances with a bit of play. Use when you have minimal requirements for accuracy
18.50		Loose running	Larger clearance which rattles, for use where accuracy is not essential

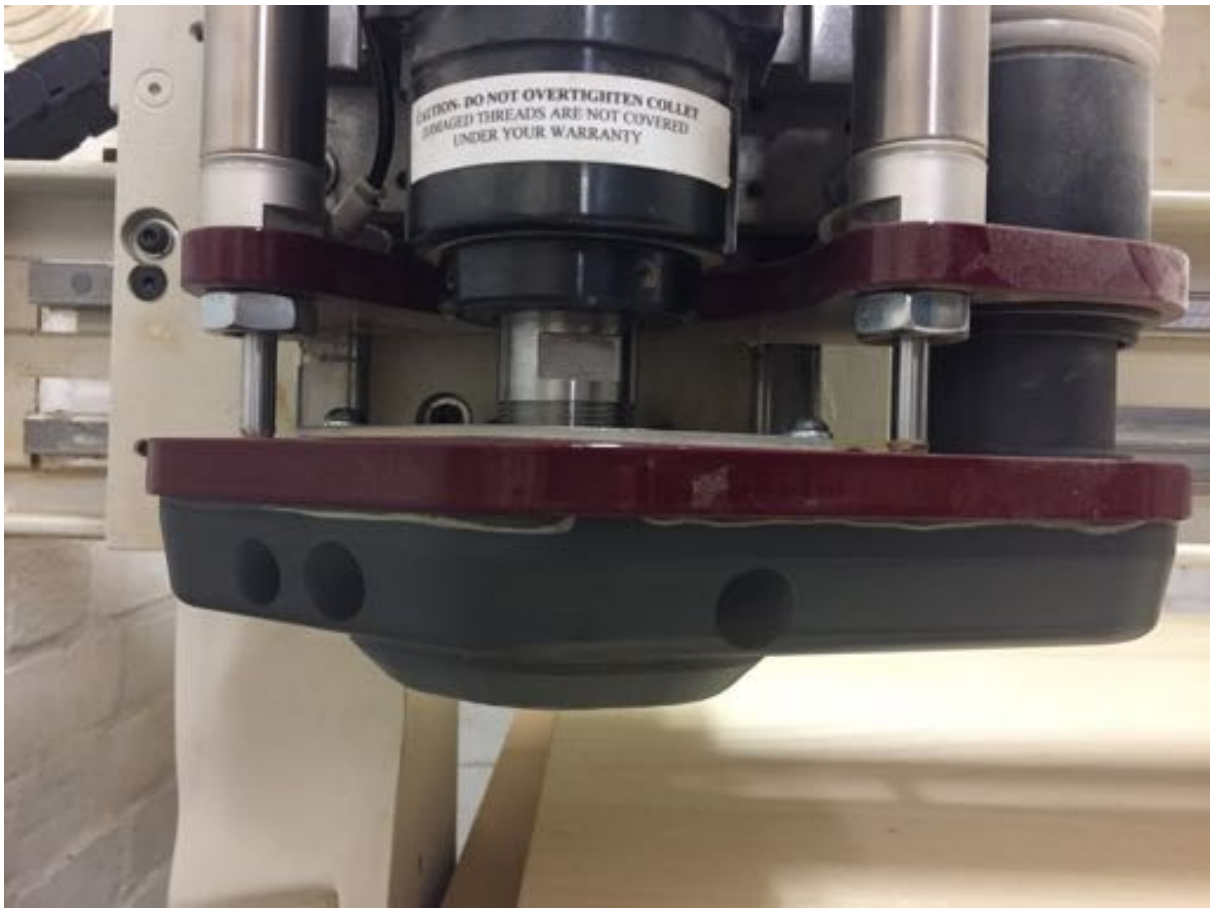
And here’s an image showing how the fit for one of the comb slots was tested with another piece of 18mm ply.


18.5
18.25
18.0
17.75
17.5
17.25
17.0

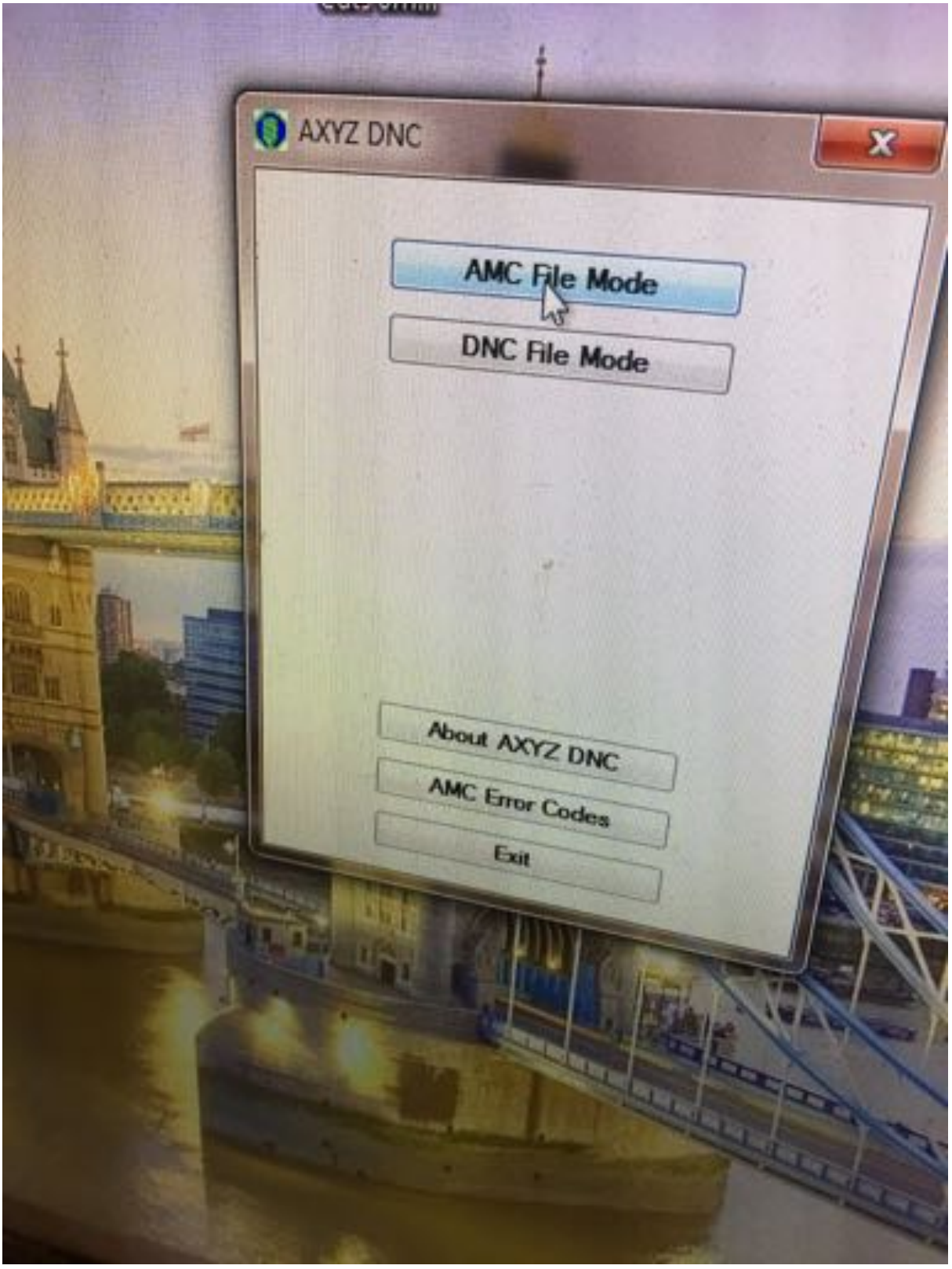

Toolpaths
Save Toolpaths
Output all visible toolpaths to one file
Output Tiled Toolpaths
Toolpaths to be saved ...
Profile 1
[1] End Mill (6 mm)
Post Processor
Close
Toolpath List
Profile 1

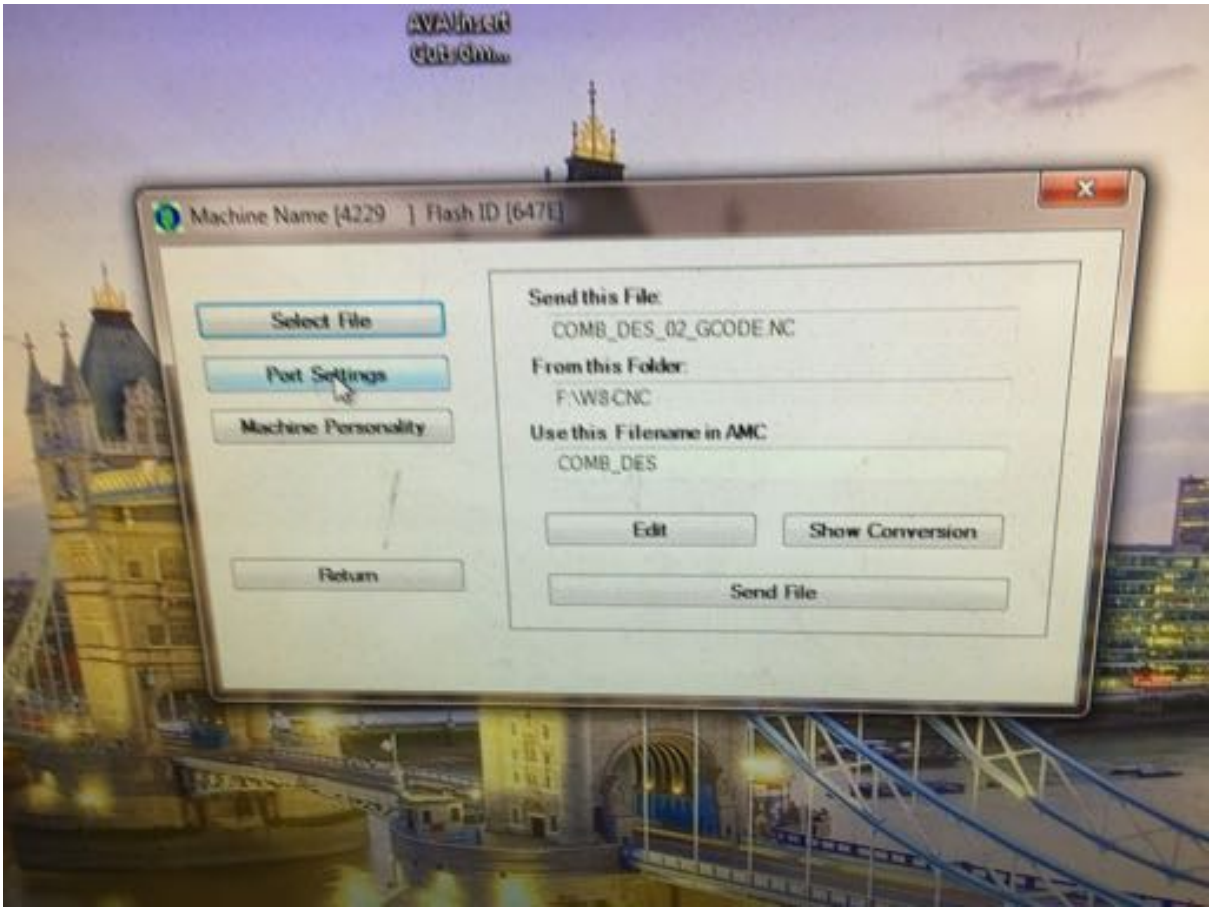

AIR SUPPLY
PRESSURE
GUARD/PRESSURE
FOOT UP/DOWN


220 V
AIR SUPPLY
PRESSURE


CAUTION: DO NOT OVERTIGHTEN COLLET
DAMAGED THREADS ARE NOT COVERED
UNDER YOUR WARRANTY


TEL: 01525 860255
Mucksucker
UNICORN GROUP UK
Est. 1965
FAX: 01525 861635
FAILURE TO
OPERATE


AXYZ DNC
AMC File Mode
DNC File Mode
About AXYZ DNC
AMC Error Codes
Exit



On machine:
Centre X/Y: Function 3
Rate: +/- button
2,4,6,8 to move (1,3,7,9 move diagonally)
Function 3 set

Set Z: Function 84
0 and 5 up and down
Function 84 set

Open toolpath software??

Drawing:
Import DXF file
Export as R12 Lines and Arcs
Make sure your drawing avoids the clamps.

Switch to toolpath tab:

Start cut: 0
Cut depth: 18.5mm (material thickness + half mm)
Metric: 6mm end mill

References and useful links

1. CNC design guide https://www.engineeringclicks.com/design-guide-cnc-milling/
2. Online design feedback for CNC parts https://www.plethora.com/how-it-works