

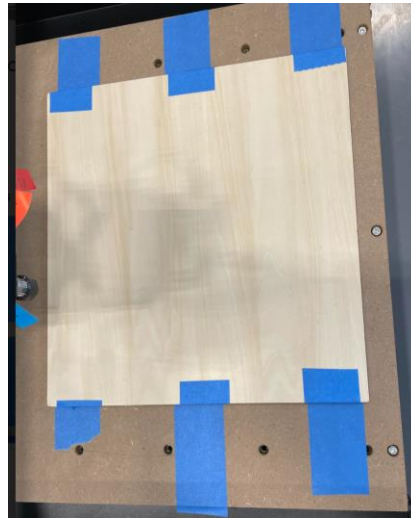
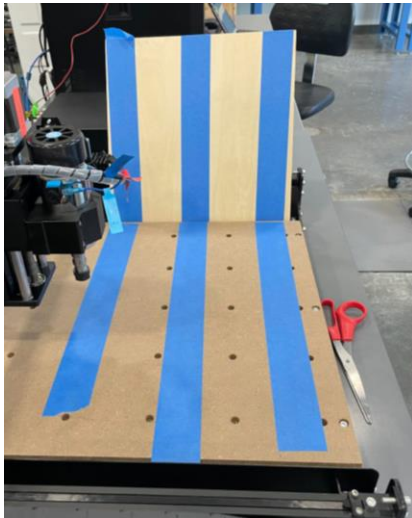
Operation Instructions

FoxAlien 4040-XE 300W Spindle 3-Axis

Braxton Brooks, Nhala Custodio, David Galeas
March 3rd, 2022

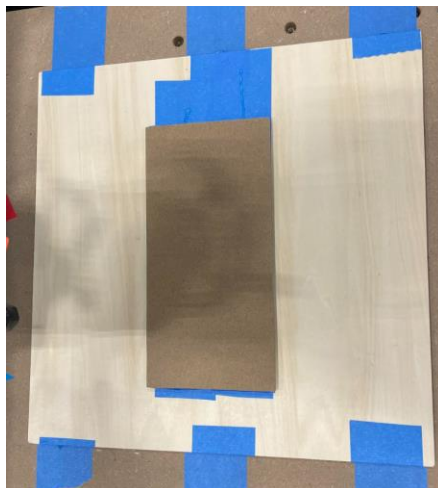
Preparing the work piece

1. Prepare a low profile scrap material that will serve as a buffer for the work piece and the wasteboard.
2. Using painter's tape, apply tape onto the scrap material. Then, apply tape on to the wasteboard. Make sure that the tape on the scrap material and the waste board are aligned




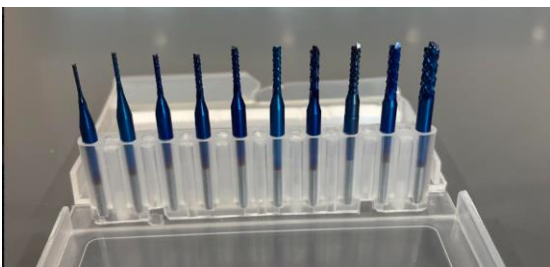
Note: Put the tape near the corners of the material to prevent it from lifting during router operation

3. Apply super glue on the surface of the painter's tape that is on the wasteboard. Then, while keeping the alignment of the tapes, fix the scrap material onto the wasteboard.
4. Measure the dimensions of the workpiece (length, width, thickness).
5. Using painter's tape, apply tape on the bottom of the workpiece. Then, apply tape onto the scrap material. Make sure that the tapes are aligned.
6. Apply super glue on the surface of the painter's tape that is on the scrap material. Then, while keeping the alignment of the tapes, fix the scrap material onto the scrap material.

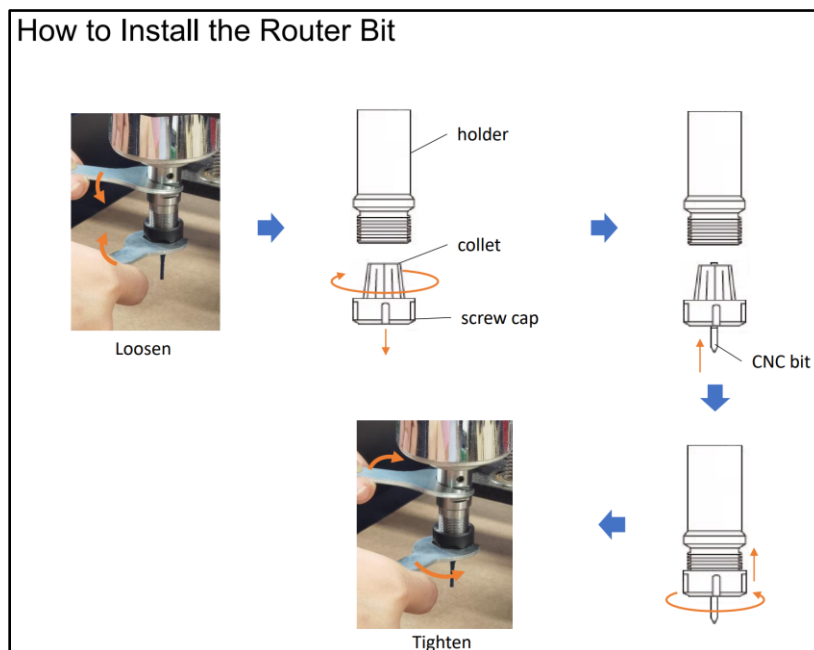


Preparing the router bit

1. Determine which type of router bit to use. There are two options that the CNC machine is able to do: engraving and cutting.

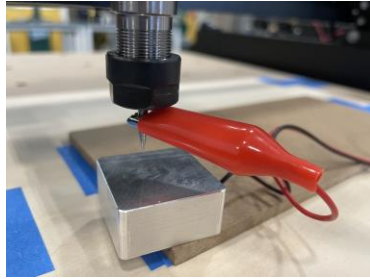
<p>For engraving,</p> <ul style="list-style-type: none">- The bits do not vary in size and blade shape- Make sure the blade is in good condition (sharp enough) to engrave onto the workpiece	
<p>For cutting.</p> <ul style="list-style-type: none">- The bits vary in sizes- Increase the bit size to be used when using thicker or denser materials	

2. Install the router bit by loosening the screw cap on the bottom of the spindle by holding the spindle with one wrench and using another wrench to unscrew the screw cap. Then, push in the handle part of the bit into the holder and screw the screw cap back in place. Figure below shows a step by step guide of the process.



Measuring the z-probe

1. The CNC router machine comes with a z-probe to set a safe z-height during operation. Measure the z-probe height using a caliper and note it down.



Exporting and preparing your own design

1. Start off by creating your design in AutoCAD (preferable in mm).
2. Export your design as a DXF file.
3. Go to OpenBuilds (<https://cam.openbuilds.com/>) to generate your gcode.
4. Set up your Controller as Generic GRBL and Tool initialization as Turn Spindle On and Off (M3/M5). Do not select a machine for setting 3 (these are OpenBuilds presets for their own machines.)

Application Settings

Please configure the application below

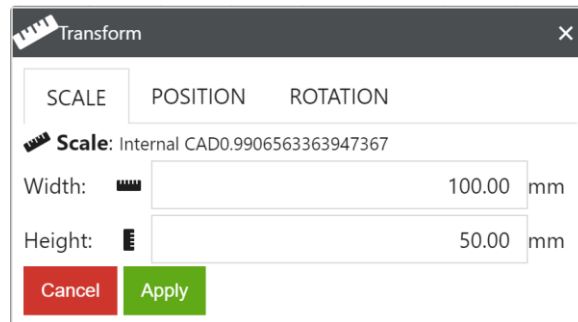
- 1 **Install CONTROL**
Used to Connect to and control you machine. If you already have it installed, please run the application, or
Windows Mac Linux
Windows CONTROL (EXE) v1.0.312
- 2 **Select your controller**
Sets approximate defaults below, which should suffice for most users
Generic GRBL
- 3 **Select your Machine**
Sets approximate defaults below, which should suffice for most users
Select Machine
- 4 **Select your Tool initialization**
Sets approximate defaults below, which should suffice for most users
Turn Spindle ...

5. Set the working area to 400x400x55 mm. Check that the settings match the screenshot below.

5 **Customise Defaults**
 From your machine and controller choice above we have prepopulated the settings below. If you have any custom requirements, please customise the settings below

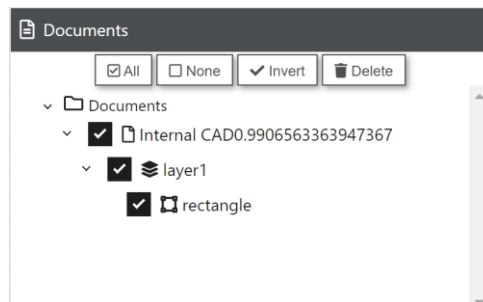
X-Axis Length	400	mm
Y-Axis Length	400	mm
Z-Axis Length	55	mm
Spindle / Laser / Plasma Command	S	
Power/Speed Scale	0 to 10000	
Power/Speed on new-line	<input type="checkbox"/>	
Rapid Move Command	G0	
Linear Move Command	G1	
Start G-Code	G54; Work Coordinates G21; mm-mode G90; Absolute Positioning	
End G-Code	M5 S0; Spindle Off	

6. Click Save then go to File. Click Open Drawing File and select your DXF file.
7. You will notice that your file will not be in the correct dimensions. You will have to fix the height or width of the piece, but the scale will still be accurate. You can also change the starting position of the piece and rotate it. (Starting position is important since you will manually set your zero XY coordinates)



Generating your Gcode

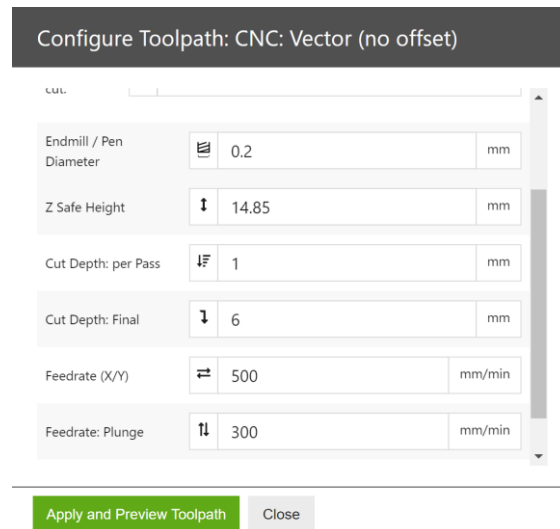
1. Under the Documents tab you will see your part and its different layers. You can pick specific parts to be cut or engrave a certain way.



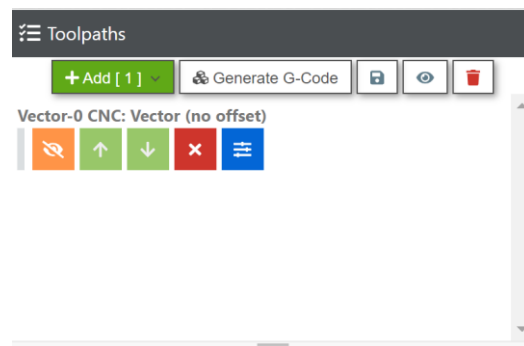
2. Once you decide what to engrave or cut you can create a new operation by adding it under Toolpaths. There are multiple types of cut but we will only use Milling/Routing Operations.
 - a. CNC: Vector (No Offset) Will create a path in the middle of the line

- b. CNC: Vector (Path Inside) Will create a path inside of the line
- c. CNC: Vector (Path Outside) Will create a path outside of the line
3. Define your Toolpath Parameters. Some of the key ones are Feedrate (X/Y and Plunge), Cut Depth (Final and Per Pass). Click Apply and Preview Toolpath.

Note: Visit <https://www.foxalien.com/blogs/news> to see different parameters that people have used to cut or engrave their material. As much as possible when cutting, do not use 1000 mm/min for feedrate as this will cause the bit to break.



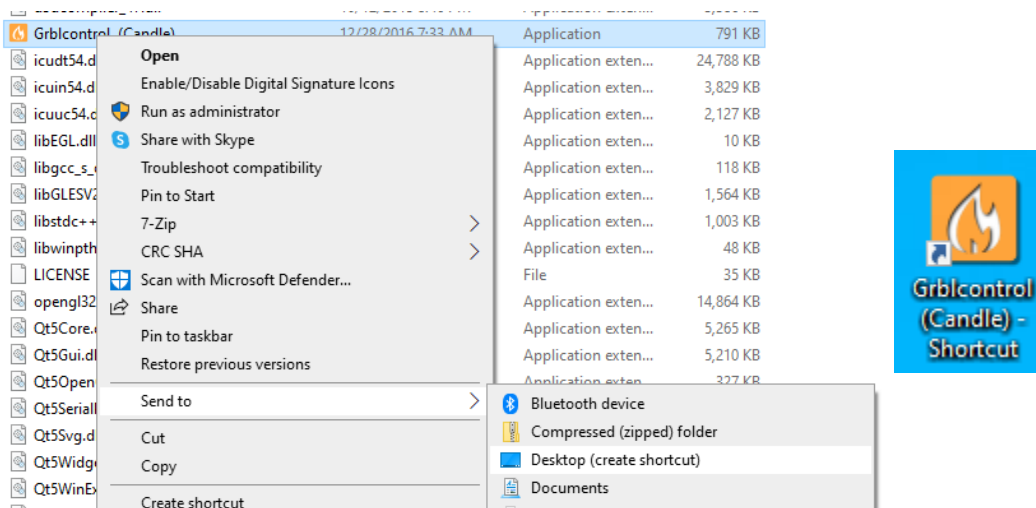
4. Once you have all your vectors, click Generate G-Code. (You can preview how the machine will cut/engrave your design by clicking Simulate on the work area).



5. Save the Gcode by clicking the save button right beside the Generate G-Code button.

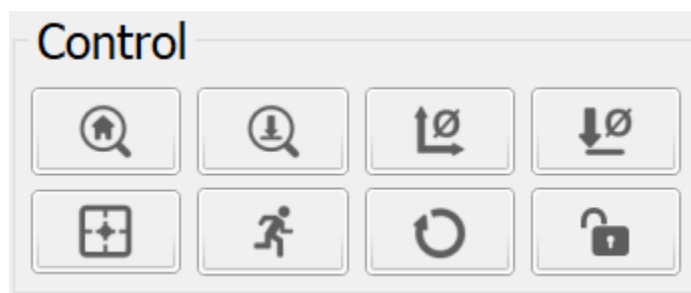
Downloading Candle (GrblControl) on your UTD account on the computer

1. Go to File Explorer > This PC > Windows (C:)
2. Click CNC R folder > FoxAlien4040-XE > for Window > Software > Grblcontrol(Candle 1.1.7)
3. Right click Grblcontrol (Candle), click Send To, then Desktop (create shortcut)

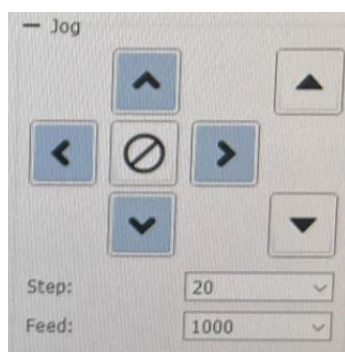


Cutting your design

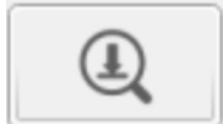
1. Open Candle (GrblControl).
2. Under the State tab, the status will display “Alarm”. Click the Unlock button on the Controls tab to disable the alarm.



3. Using the Jog controls, move your spindle to the desired zero XY coordinate. Click the zero XY button on the Controls tab.
4. Zero your Z coordinate by slowly going down (use step measure of 1). Use a piece of paper and stop once you can no longer slide the paper under the bit. Go up 0.1 mm step after that and click the zero Z axis button on the Controls tab.



5. Move the spindle by choosing 10mm steps. Move it by 10mm towards the right on the x-axis and up on the y-axis. Move it by 20mm upwards on the z axis.
6. Set your z-probe right under the bit. Clamp the alligator clip onto the bit. Then, click the z-probe command button on the Controls tab.



7. Make sure the spindle speed controller on the offline controller is on its max setting by turning the knob all the way up.
8. Open your gcode by clicking File and then Open.
9. Put on safety goggles (Face mask and gloves are also recommended).
10. When ready, click Send. (Make sure someone is always by the computer in case you need to pause your router due to something going wrong).

Note: The offline controller has an emergency stop button. Press it when something goes wrong.



Cleaning Up

1. Use the brushes and a trashcan to remove the residue on top of the part.
2. Pick your part up and check it out (See if adjustments need to be made to the Gcode).
3. Remove the remaining material.
4. Check Maintenance Instructions (Cleaning CNC Router and Drill Bits).