

How do a Plasma CNC Table Project.

Where to get the CNC Plasma software.

Download the Machine Interface Controller from:

<https://software.openbuilds.com/>

This would go on your laptop. It is already loaded on the space laptops.

The OpenBuilds CAM-GCODE generator can be found at:

<https://cam.openbuilds.com/>

This will allow you to

- Import DXF/SVG/PNG/BMP/JPG/Gerber/Excellon files
- Import from a built-in Parts Library
- Setup toolpaths for CNC Routing, Laser cutting, Plasma Cutting and Dragknives
- Generate GCODE
- Send GCODE to OpenBuilds CONTROL

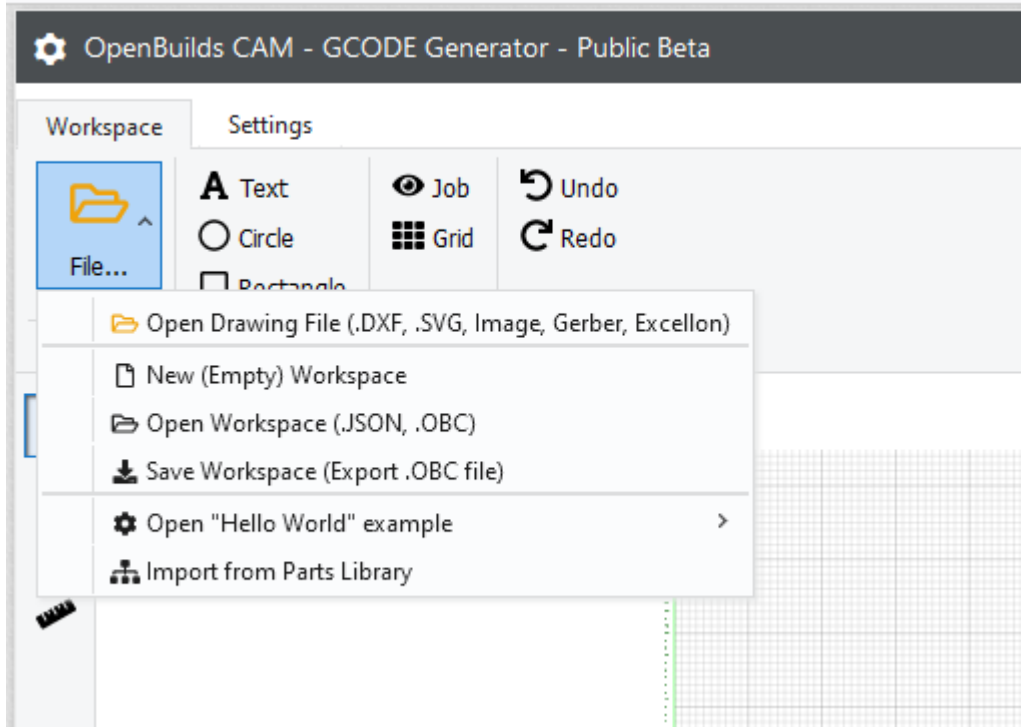
There are many places to get G-Code for the CNC Plasma Cutter. Most likely, unless the CAD/CAM software you are using has specific plasma cutter options, you will have to modify the G-Code by hand in order to get it to work and give you your desired results.

Following examples use the OpenBuilds CAM web site. (It has a plasma option.)

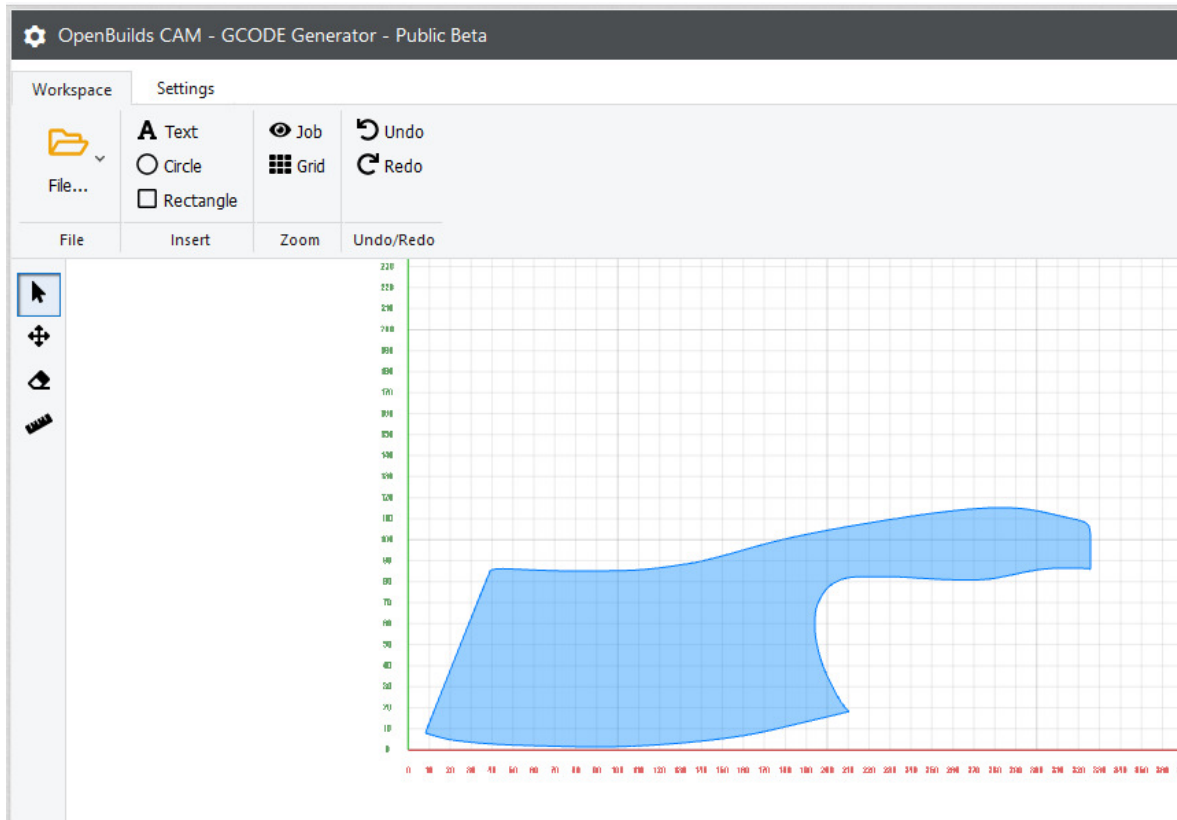
I start with .svg files, but there are other options.

Our Plasma CNC Table is a Makerspace build using BlackBox stepper drivers and other hardware that we made ourselves. (So, it's a hobby level machine. Advanced hobby to be sure. But not a commercial machine.) It can be a bit finicky and cantankerous to use. Things that can happen are; it loses it's zero location, the plasma does not start (trigger not pulled), the plasma stops for some unknown reason in the middle of a job, and some other strange behaviors. You are going to need patience, time, and some extra material. It is a 'work in progress'.

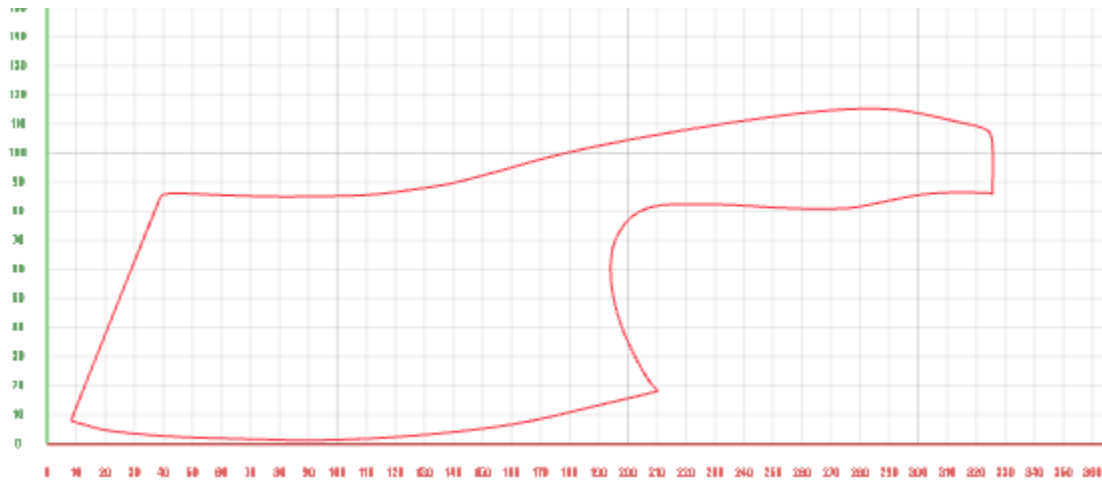
Generally, you load your file



Select your item on the screen:



Boarder of your item will turn Red

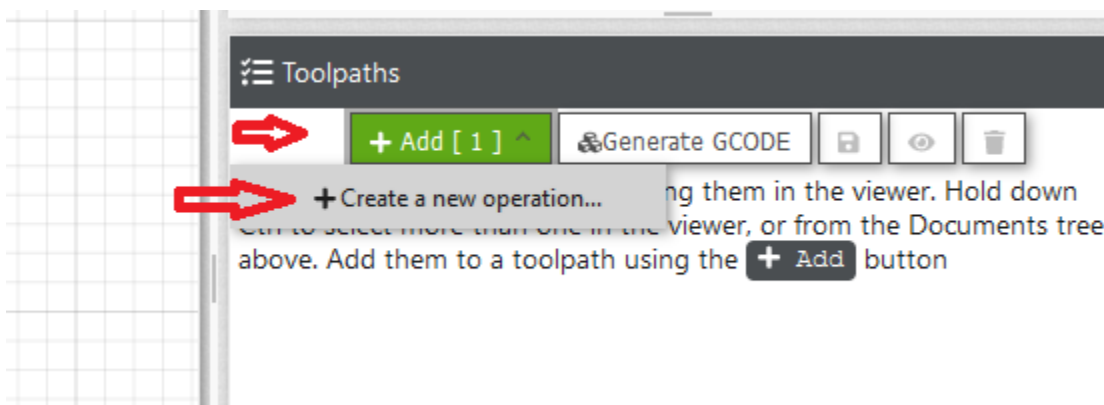


Under the Toolpaths section,

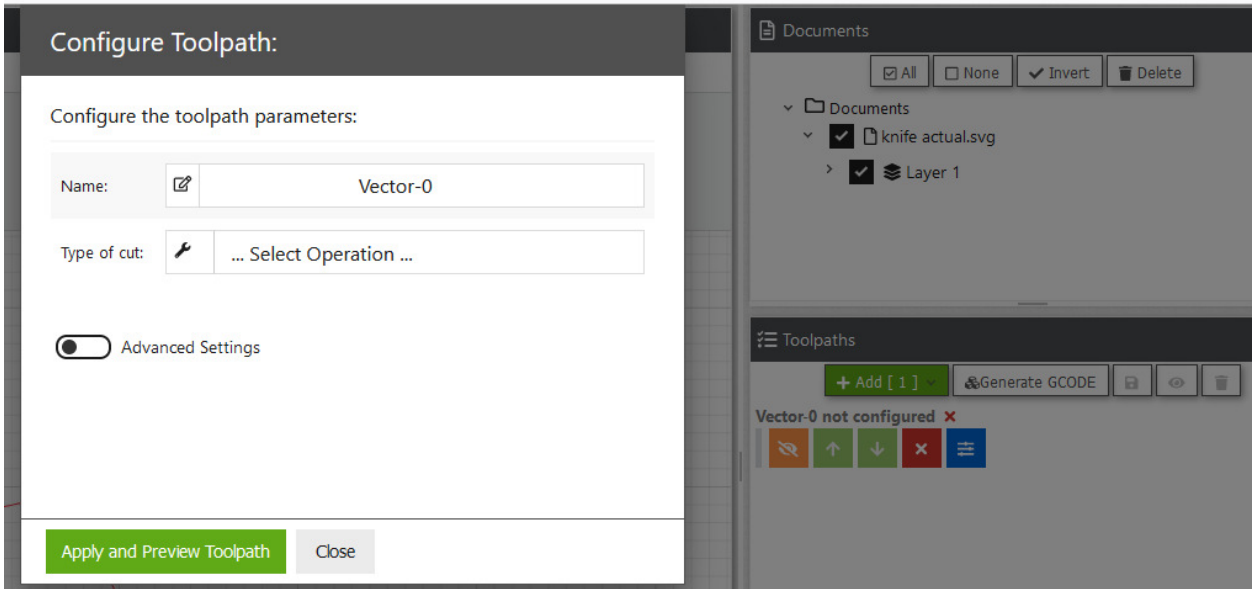
Select +Add [1]

and then

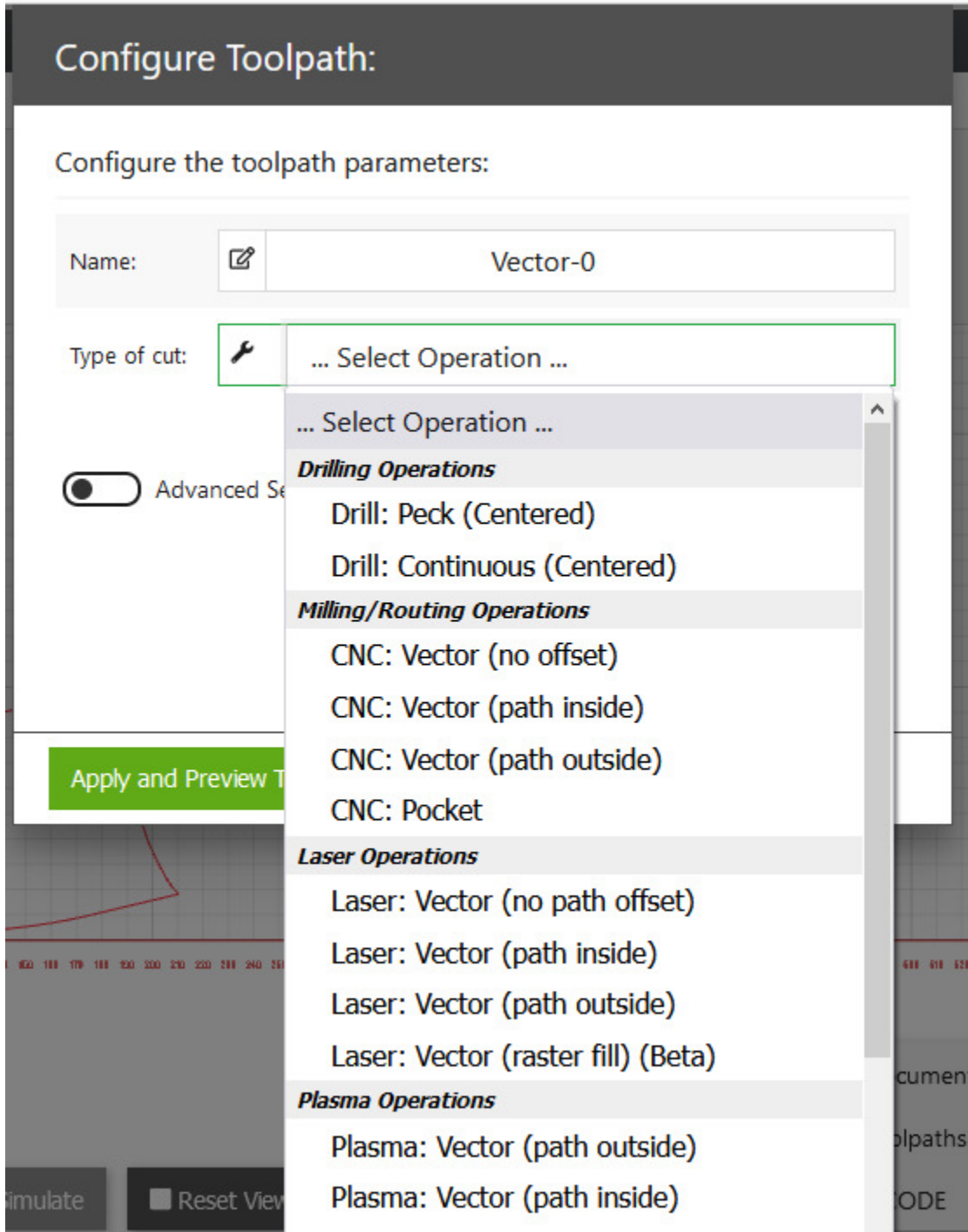
+ Create a new operation.....



A line of icons will be added in the Toolpaths window and the Configure Toolpath window will appear.



Select the kind of operation you want under Type of cut:











(There are more options, scroll down the window to see them.)

Select Z Safe Height, Feedrate, and Plasma Kerf (These numbers are good starting points)

Configure Toolpath: Plasma: Vector (no path offset)










Configure the toolpath parameters:

Name:		Vector-0		
Type of cut:		Plasma: Vector (no path offset)		
Z Safe Height		3		mm
Feedrate (X/Y)		300		mm/min
Plasma: Kerf		1.2		mm

[Apply and Preview Toolpath](#) [Close](#)

Set Pierce Height, Pierce Daley, Cut Height, Lead-in Distance, and Touch-Off
(These are good starting numbers)

Configure Toolpath: Plasma: Vector (no path offset)

Plasma: Pierce Height		0.1		mm
Plasma: Pierce Delay		2		seconds
Plasma: Cut Height		0.1		mm
Plasma: Lead-In Distance		5		mm
Plasma: Touch-Off Zero		No		

Advanced Settings

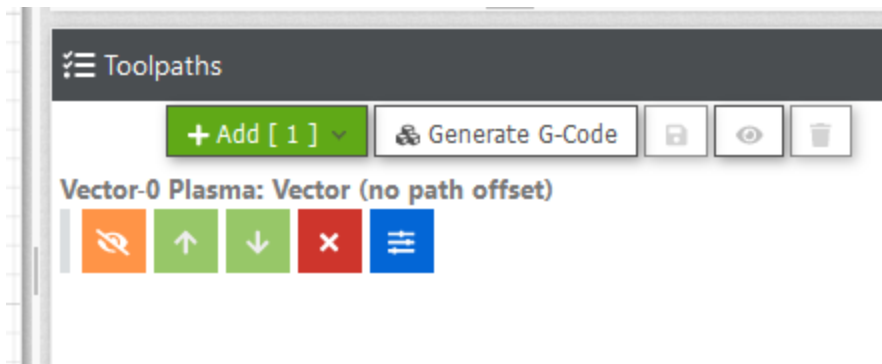
[Apply and Preview Toolpath](#) [Close](#)

Note: Pierce Height and Cut Height must be greater than zero

Click [Apply and Preview Toolpath](#).

The Toolpaths window will now show your selected toolpath, in this case

Vector-0 Plasma: Vector (no path offset)



The icons, left to right are:

Hide toolpath and exclude from gcode generation

Move this toolpath up in the order (if you have multiple toolpaths)

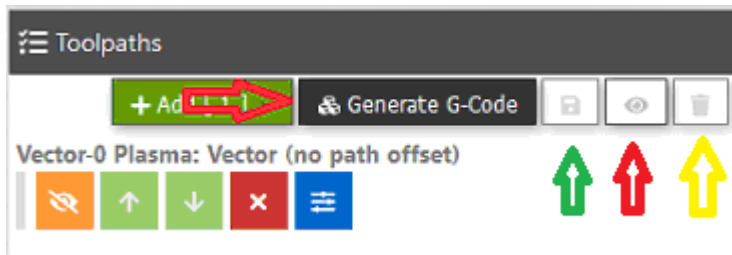
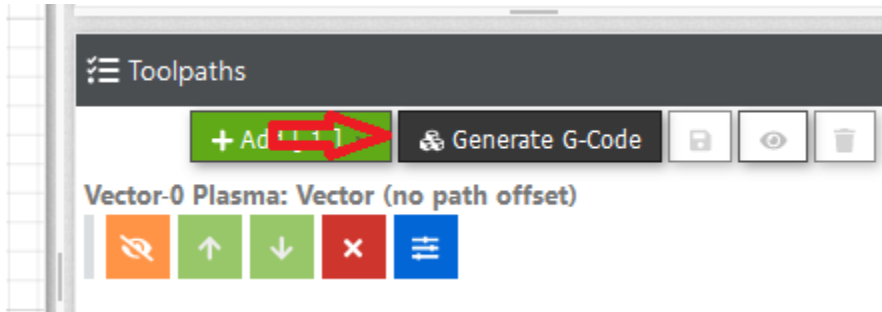
Move this toolpath down in the order (if you have multiple toolpaths)

Delete this toolpath, and

Configure Toolpath (which will take you back to the Configure Toolpath windows as shown above)

When you are ready to generate your Gcode:

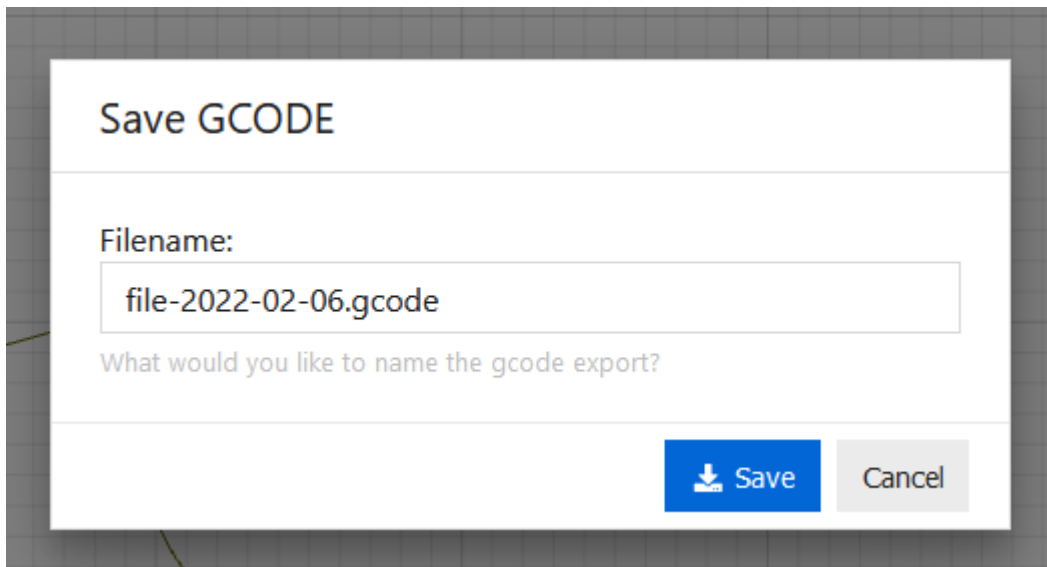
Select the Generate G-Code button



You can now Save (green arrow), view (red arrow), or delete (yellow arrow) your G-Code

If you select view the G-Code, you will also get an alternate option to save the G-Code

Save G-Code



You can rename the file at this point if you like.

View G-Code

Preview GCODE

```
; GCODE Generated by cam.openbuilds.com on 2022-02-
G21 ; mm-mode
G21; mm-mode
;
;
; Operation 0: Plasma: Vector (no path offset)
; Plasma Kerf: 1.2
;
G0 Z3; move to z-safe height
G0 F1000 X79.8547 Y1.3778
;
G0 Z0.01; Move to Pierce Height
```

Doc

At the CNC Plasma Table

Retrieve the Control Box (currently living on top of the lockers next to the white board in the Green Room). Also get the heavy duty extension cord (currently living under the lockers next to the white board in the Green Room).

You will also need a regular extension cord to power the Control Box (and perhaps as power for your laptop - take a power strip or outlet multiplier as needed).

Drag the Plasma cutter out to the CNC Plasma Table

The heavy duty extension cord plugs into the wall in the welding shop near the doors (wall between the welding shop and the paint booth).

Regular extension cord plugs into one of the outlets just a little farther down the wall (currently between the small blast cabinet and the gas welding cart).

Plug the cable from the plasma table into the control box. The plugs are arranged so that they only mate up with specific sockets. Also plug in the green ground banana plug.

Plug the Control Box into AC power

Power up your computer and plug the USB cable from the control box into your computer.

Plug the Plasma Cutter into the heavy duty extension cable. Do not turn on the Plasma Cutter yet.

Mount the Plasma Torch to the bracket on the CNC table.

This is done with some metal straps and thumb screws.

Be sure the trigger guard is outside the 'finger'

The torch just needs to be held snugly

Be sure the torch tip is vertical (not twisted to one side or the other - this would give you angled cuts in your kerf).

See picture on next page



Getting Ready to Cut

Launch the OpenBuilds Control on your computer (you may have to launch it twice in order to get the following screen to appear).

Click the **Connect** box (shows as **Disconnect** in the following image) and connect to whatever USB port the Controller is attached to. It will be a COMx port.

The screenshot displays the OpenBuilds CONTROL v1.0.170 software interface. The window title is "OpenBuilds CONTROL v1.0.170". The interface is divided into several sections:

- Top Toolbar:** Contains buttons for "Disconnect" (highlighted in red), "Reset", "Settings", "Open G-CODE", "Run Job", "Stop Job", "Check Size", "Tool On", "Tool Off", "Home All", "Wizards & Tools", "Unlock Alarm", "Stop Job", and "OpenBuilds.com".
- Middle Section:** Features "mm-mode" and "inch-mode" tabs. Below these are coordinate readouts for X, Y, Z, and XYZ axes, each with a "setzero" button. To the right is a jog control panel with "Incremental Jog" and "Continuous Jog" options, and jog rates of 0.1mm, 1mm, 10mm, and 100mm. Below the jog panel, it shows "Jog: 4999.9 mm/min". To the right of the jog panel are two "reset" buttons for "Feed: 100%" and "Tool: 100%".
- Bottom Section:** Includes a "3D View" tab, a "Serial Console" window showing log output, and a status bar at the bottom with "Port: Connected", "Controller: Idle", and "Job Queue: 0".

The Serial Console window shows the following log output:

```
[10:48:47] [ 3D viewer ] WebGL Support found! success: this application will work optimally on this device!  
[10:48:47] [ websocket ] Bidirectional websocket Interface started  
[10:48:48] [ update ] Checking for Updates  
[10:48:49] [ update ] You are already running OpenBuilds CONTROL 1.0.170  
[11:00:33] Detected a change in available ports: [COM3]  
[11:00:38] [ connect ] PORT INFO: Port is now open: COM3 - Attempting to detect Firmware  
[11:00:38] [ connect ] Checking for firmware on COM3  
[11:00:38] [ connect ] Detecting Firmware: Method 1 (Autoreset)  
[11:00:39] [ connect ] Detecting Firmware: Method 2 (Ctrl+X)  
[11:00:40] [ connect ] Detecting Firmware: Method 3 (Smoothieware)  
[11:00:40] [ ] Grbl 1.1g ['$' for help]  
[11:00:40] [ ] [MSG:'$H'! '$X' to unlock]  
[11:00:40] [ $$ ] $0=10 ;Step pulse time, microseconds  
[11:00:40] [ $$ ] $1=255 ;Step idle delay, milliseconds  
[11:00:41] [ $$ ] $2=0 ;Step pulse invert, mask  
[11:00:41] [ $$ ] $3=4 ;Step direction invert, mask  
[11:00:41] [ $$ ] $4=1 ;Invert step enable pin, boolean  
[11:00:41] [ $$ ] $5=0 ;Invert limit pins, boolean  
[11:00:41] [ $$ ] $6=0 ;Invert probe pin, boolean  
[11:00:41] [ $$ ] $10=1 ;Status report options, mask
```

Use the Jog Buttons (X, Y, and Z arrows) to move the torch to a reasonable starting place and raise the Z axis enough to fit your material under it. (You will likely want continuous jog. Slide the switch to the right. It will move as long as you hold the button down.)

Note: Each move must complete before you can initiate the next move. Just be patient. Jog the cutter, wait for the move to finish, and then make your next Jog as needed.

Jog the cutter to a suitable starting place on your material.

Lower the Z axis until the tip of the Plasma Torch is just above your material. Very close. Less than 1mm. So you can just barely see light between your material and the torch tip.

Click the Set Zero XYZ button to zero all axes. (yep, the plural of axis is axes.)

Load your G-Code by using the Open G-Code button and selecting your file.

When you are ready, turn on the Plasma Cutter (switch on the back), set it for internal air pump, and adjust the power knob for the material you are cutting.

Click the Run Job icon.

Notes:

I **Strongly** suggest that the first time you run a job you **'cut air**. Set the Z axis high and, with the plasma cutter turned **off**, run your job. This will send the machine through all the motions and you can confirm that it will be cutting where you want it to and not hitting any limit switches or anything else on the table. (I've been using plasma cnc cutters for years and I still *always* cut air the first time.)

The Big Red Emergency Stop Button at the front right of the machine will stop execution.

You can restart by releasing the button, clearing any errors, and resuming the cut. You can also use the Stop Job button on the screen.

Most metal will lay on the table and not need any hold downs to keep it from moving. If your metal is warped, you may need weights of some sort to hold it flat. If it is bowed up, the plasma torch could drag on it and cause your material to be dragged across the table. Not good.

The Jog rate and feed speed are not particularly fast. In fact, they are quite slow. Be patient.

The CNC Plasma Table assumes that your material is absolutely flat. Shim your piece on the table as necessary to make it flat. Use metal (conductive) shims.)

If you start your cut off the side of your material, you may encounter a problem. The Plasma Cutter senses a return path of the current. If the Torch is blasting into air, this return path is not activated. When the Plasma Machine senses this, it will shut off the plasma after a very short interval (likely before the cutter gets to the edge of your material).

If you need to do this, place a piece of sacrificial metal in the area next to your material. The Plasma Cutter will sense the return current through the sacrificial piece and keep the plasma stream going until it reaches your material.

Once the cutter is into your material, the return path is continuous (unless, of course, there is an opening in your material - if so, place some thin sacrificial material under the hole. Shim your entire piece of material as necessary.

Our Plasma CNC Table is a Makerspace build using BlackBox stepper drivers and other hardware that we made ourselves. (So, it's a hobby level machine. Advanced hobby to be sure. But not a commercial machine.) It can be a bit finicky and cantankerous to use. Things that can happen are; it loses it's zero location, the plasma does not start (trigger not pulled), the plasma stops for some unknown reason in the middle of a job, and some other strange behaviors. You are going to need patience, time, and some extra material. It is a 'work in progress'.

Modifying G-Code from other Sources.

If you have G-Code from other sources, or from a CAM that does not support plasma cutters, you can modify it by hand.

Have the CAM software prepare G-Code, cutting outside or with no offset as needed, using a 1mm diameter tool. Set the feed speed to 300 mm (per minute).

Once you have the G-Code, find each cut segment and add the following code.

G0 F1000 X53.3798 Y54.2541

Find the G-Code move to the starting location

G0 Z0.0000 F300

M3S1000

G4 P2

through

Move to Pierce Height and set Z speed

Plasma On with Spindle Speed 1000

Pierce Delay - wait 2 sec for plasma to cut

G1 X54.4274 Y52.7283 F300

.....

G1 X53.3798 Y54.2541

G-Code to make the cut

M5

G0 Z5.0000

Plasma Off

Move to z-safe height

(These same 2 last lines also at the end of the job.)

Piercing:

When the Plasma Torch is above the sheet of metal and the plasma comes on, it starts to drill a hole through the material by blasting it with hot plasma. The metal that is being melted needs someplace to go and is sprayed out. Sometimes, a small channel is formed where the molten metal exits. This is quite common. It will leave a little nick on the surface of your metal. Like between 1 and 3 mm long. Looks like this



or this



You may be able to avoid this with a lead in - starting your cut away from the actual line you want and cutting to your desired line.

Thicker material may need longer pierce times !

You can contact Jim on Slack if you would like some assistance, someone to look over your shoulder, review your G-Code, or just for some company when you are cutting (and to commiserate with when something goes wrong :-)).

This document is a work in progress and will likely be updated and added to on an irregular basis.