Plasma CNC Checklist and Instruction.

Our CNC Plasma table consists of the following parts:

A Plasma Source, the LGK-160IGBT, which produces the arc power that turns air into plasma.

A Yikuai XF-300H liquid cooled Plasma Torch.

A Transformer that converts local power to the correct voltage for the Plasma Source.

A Torch Cooler which cools and circulates coolant to the torch to help keep it cool.

A Plasma Table, 4' x 8' with an X and Y axis gantry and a Z axis holding a Plasma Torch.

A Control Console housing a computer, monitor, mouse, keyboard, USB ports, and other electronics to move the axes and control the Torch.

A Torch Height Controller in the Control Console.

The Plasma Source uses compressed air from our air compressor to run through the Torch to make plasma.

The Torch holder, on the Z axis, uses compressed air to hold the torch in place. Should the torch hit something, the seal is broken, the torch moves, and a loss of pressure signal is sent to the controller.

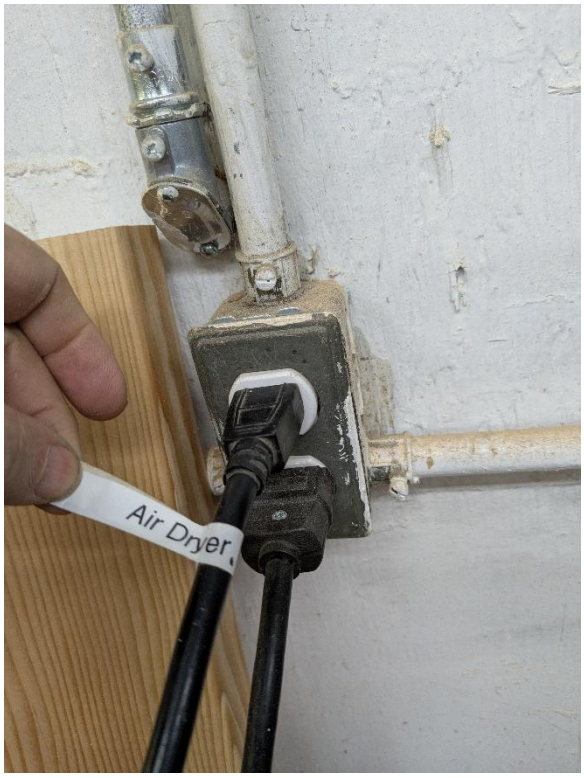
The Plasma Table can cut metal from 1mm thick up to 18mm thick with excellent cut quality. It can cut metal up to 35mm thick with less, but still very good, cut quality. Thicker metal requires a slower cut travel speed and a higher setting on the Plasma Source.

Metal is heavy. Get help moving large or heavy sheets.

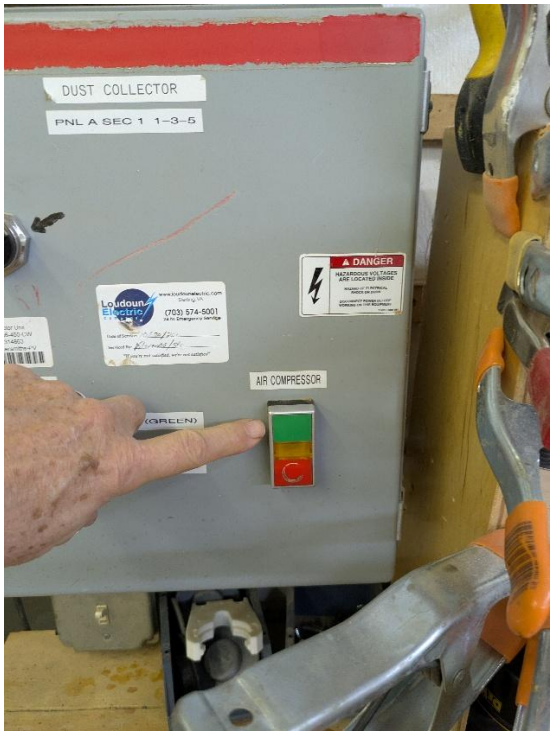
The metal will be **HOT** after cutting. Allow time for it to cool and be cautious when touching cut parts or the metal they were cut from.

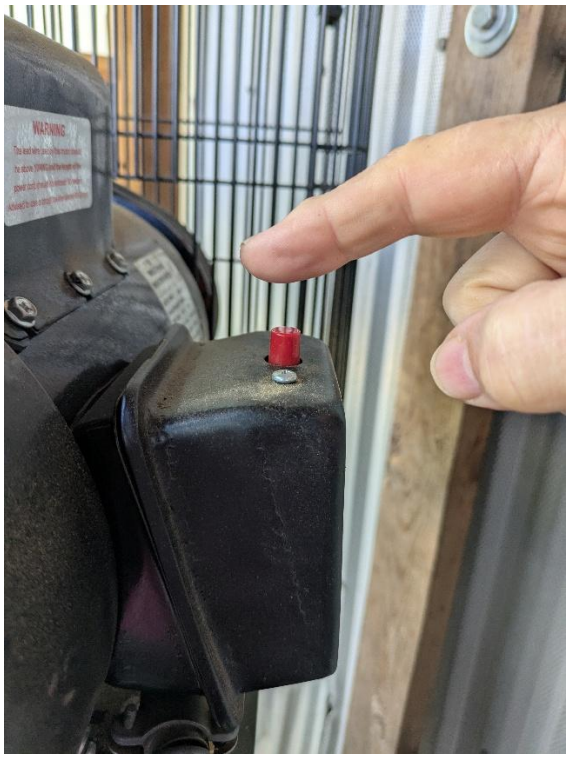
You absolutely must use the ground terminal. Torch will not stay struck without it.

Checklist
Plug in the Air Dryer in the Woodshop



Start the Compressor





You may have to reset the compressor overload switch. Press down the Red button.
You may also have to push the blue button on the Compressor power box.

In the back bay.
Turn on the main power box. You will hear a fan start running in the transformer in the corner.

Turn on the air valve.

Off

On

Note the pressures for the Torch Holder and the Plasma Source Supply

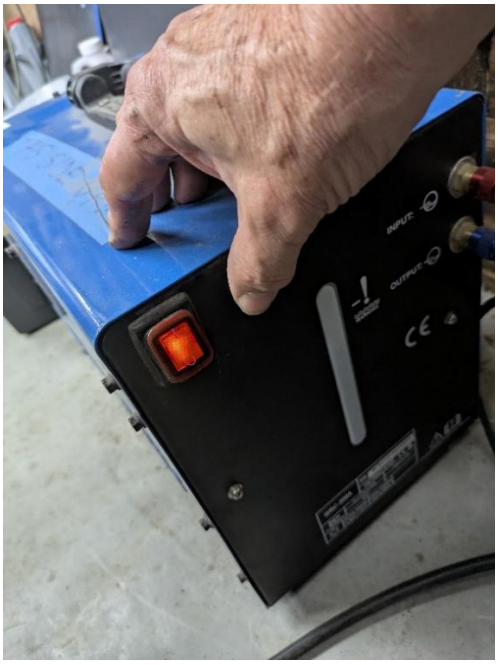
60 PSI Orange Plasma Air

35 PSI Grey Torch Holder

As soon as you turn on the Air, you will hear a hissing noise from the torch holder. The torch holder uses air pressure to hold the torch in the fixture.

You will need to pull (gently) twist, and otherwise adjust the torch until the holder seals and the air leakage stops.

Turn on the Plasma Source.
Flip the triple breaker up. Fans will run.



Turn on the Torch Cooler.
Coolant will begin to circulate.

You should see the following lights on the front of the Plasma Source.

Green Power light On

Green Air Pressure light On

Water Pressure light On

The Cutting light will be off but will come on when the cut starts

The Overload light will be Off

The Input Fault light will be Off

The Front panel switches should be set as follows

2-Step/4 Step – Up to the 2-Step Position

Water Cooled Torch selected (switch up)

Cutting/Check Air switch to the Cutting position (switch up)

Cutting Current knob – Set to show the Amperage needed for the material you want to cut. 100 Amps is good for 1/8 material

The Control Console uses 2 power supplies. A 220 VAC supply and a 110 VAC Supply. Both of these come from power cables plugged into wall outlets.

Turn on the small red switch on the lower left side of the Control Console.

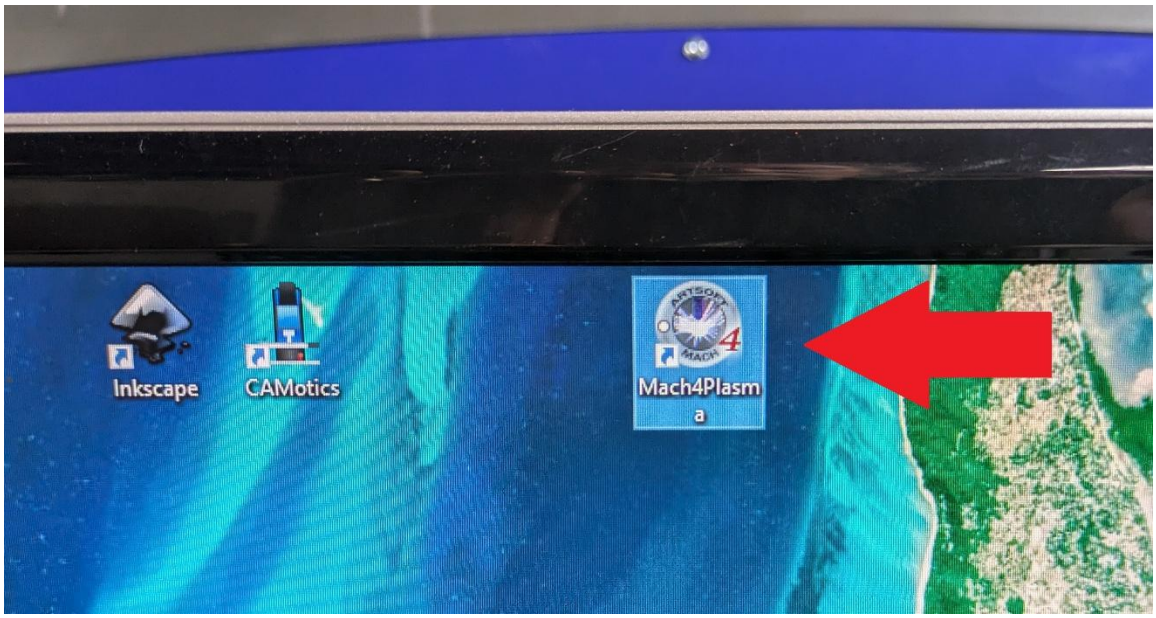
Press the Green power button on the upper right corner of the Control Console
A light below the Emergency Stop button will come on.

Please note the Red Emergency Stop Button (Push for Emergency Stop, twist clockwise to release).

Reset the computer if it does not automatically start by pressing the power button.

(Picture Needed)

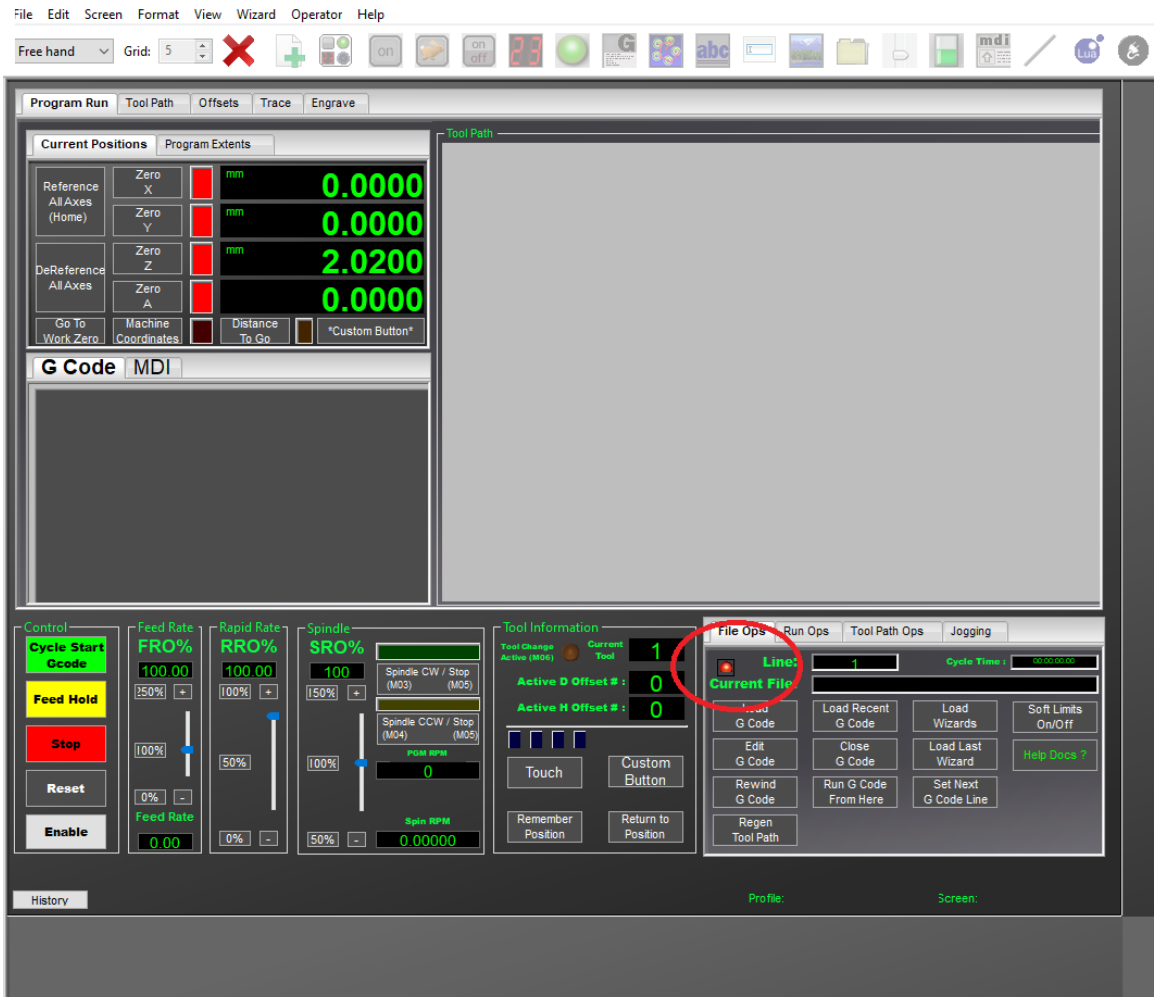
Once the system boots, select and double click the Mach4 Plasma Icon at the top center of the Screen.



Mach4 screen description

Do not Home the machine. This machine does not have a Home position. The work X-zero and Y-Zero is set manually.

Mach4 will look something like this.



Enable the Machine
First, enable the machine by clicking the Enable button in the lower left-hand corner.

Positioning your blank on the machine.
Place your material in a convenient place on the bed of the machine.
The material must lay absolutely flat. Use clamps as necessary. Put clamps in places where the torch will not run into them.

Attach the Ground Connection to your material in a place that will not be cut, or be in the way of the Torch path. It is a magnetic connection. Turn the knob clockwise to engage the magnet. Counter-clockwise to release. Shiny spots make good ground connections.
(Picture)

There is not a home location on this machine.

Select the Jogging tab, lower right window, 4th tab, says Jogging.

Move the Torch to where you want your X-zero Y-zero to be by using the X and Y jogging buttons.
X - moves the torch to the left.
X + moves the torch to the right.
Y – moves the gantry toward the front of the table
Y + moves the gantry toward the back of the table

When you have moved the torch to your desired X and Y zero location,
in Mach4, Zero all the axes (X, Y, and Z)
Upper left window, Zero X, Zero Y, and Zero Z.
The green numbers just to the right of the buttons should all go to 0.0000

The Torch Height Controller controls the Torch Height (Z axis in Mach4 is ignored.)

Height of Torch Nozzle above work.
2mm to 5mm (max). Closer is better as long as the Torch does not contact the work.
The Torch Height is adjusted using the Torch Height Adjustment, Torch Up and Torch Down switch. Start with the Torch tip 2mm above your work.

Move the Auto/Manual switch to Auto.

(Do not press the Zero Test button. The Arc Strike button will not do anything.)

Adjust the Torch Height Controller if necessary
Use these numbers as a starting point.

SetArc should be about 30. Bigger numbers for thicker materials. It is set using the Height knob.
Sensitivity should be about 20. It is set using the Sensitivity knob.
During the cut, the ARC number will show the arc voltage.

Load your Gcode
In Mach4, File – Load Gcode – Your file name

(Your Gcode file should have the file type extension, .tap, .nc, or .gcode)

When you are ready to cut, (Torch should be at your work X and Y zero and the Torch height should be set.

Click the Cycle Start Gcode button.

You can stop the machine by clicking the Stop button.
When stopped, you can send the torch back to X-zero Y-zero by clicking the Go To Work Zero button. You can restart your job by clicking the Reset button and then the Cycle Start Gcode button.

When your Gcode is complete the torch will shut off, move to the X and Y zero location, and rewind your code to the beginning.

Preparing Gcode
We currently have two ways to prepare Gcode for the Plasma Cutter.

Plasma cutting is essentially a 2D cutting process. X and Y movements. The Torch height remains the same throughout the cut (actually, the Torch Height Controller will keep the torch heigh at the correct level).

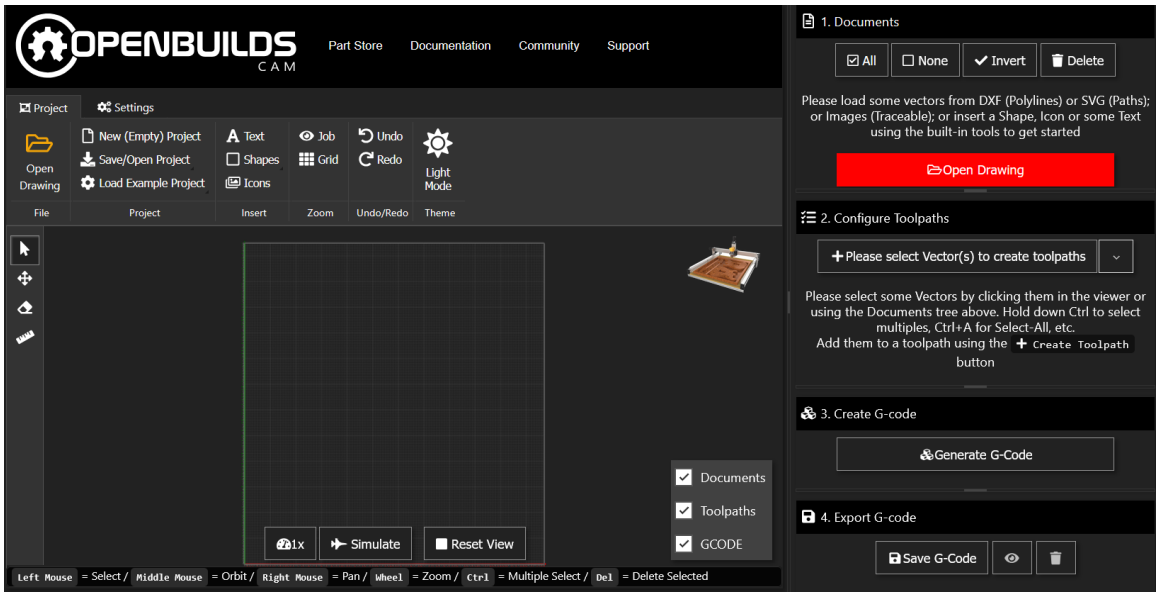
Prepare your design and save it as an .svg file.

Using Openbuilds CAM Gcode Generator

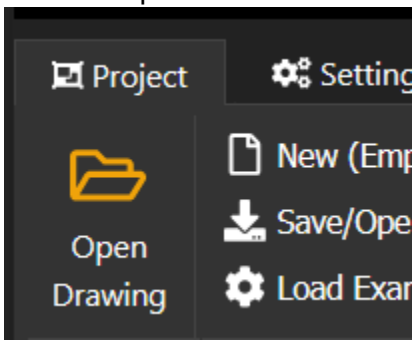
Online application

https://cam.openbuilds.com/

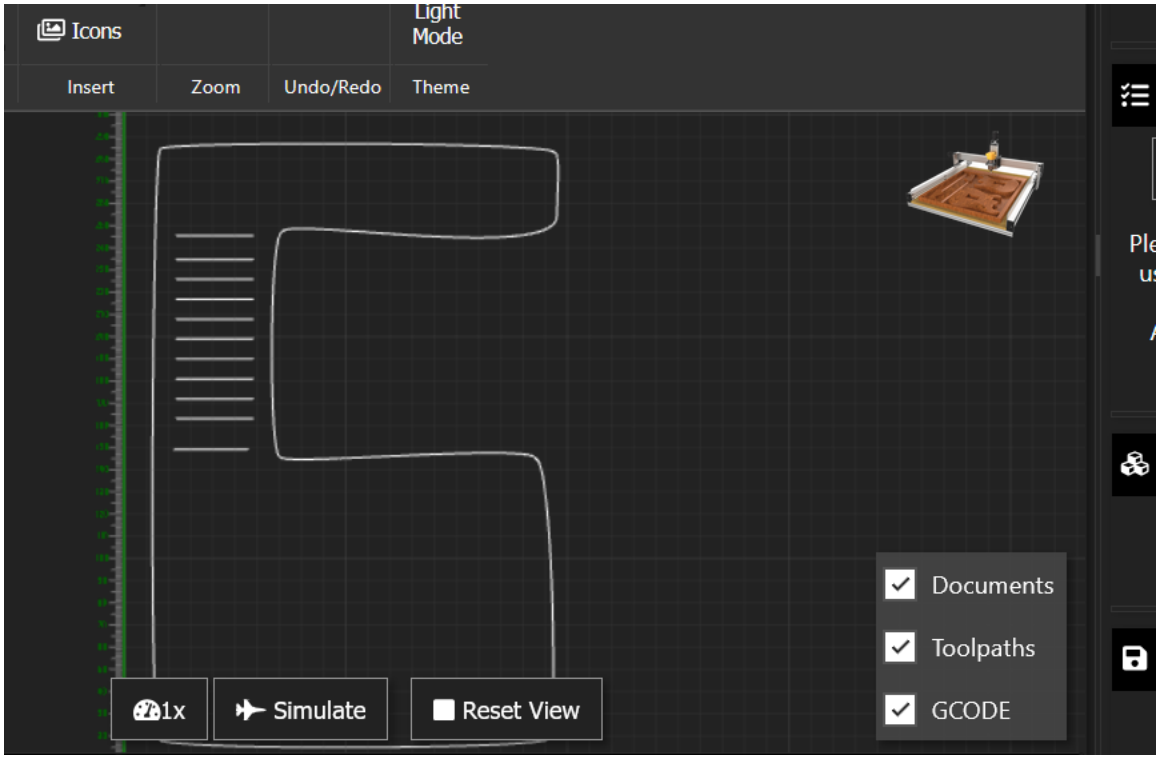
Launch the web site and give it a moment to complete loading.



Use the Open file icon to load your design

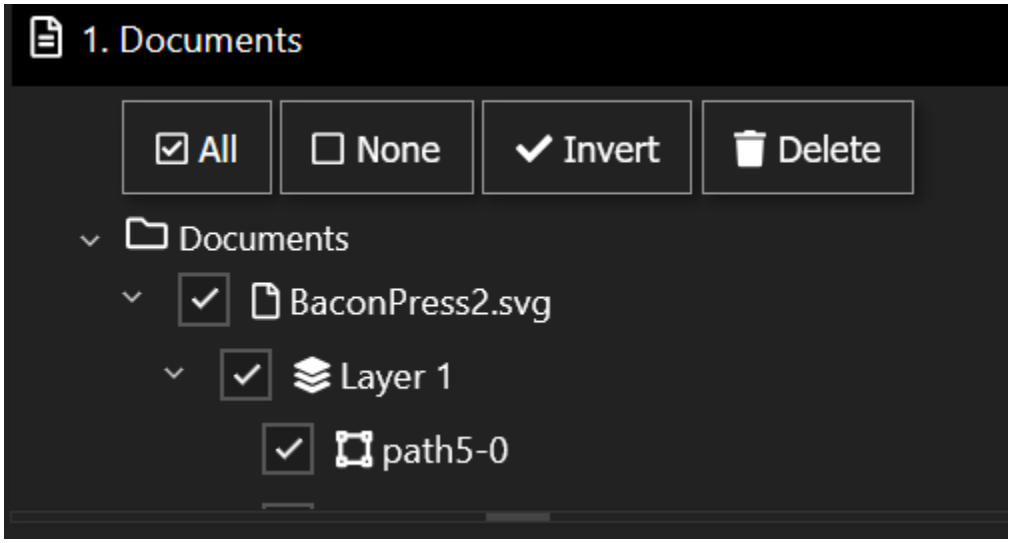


The Vectors for your design will show in the design window.



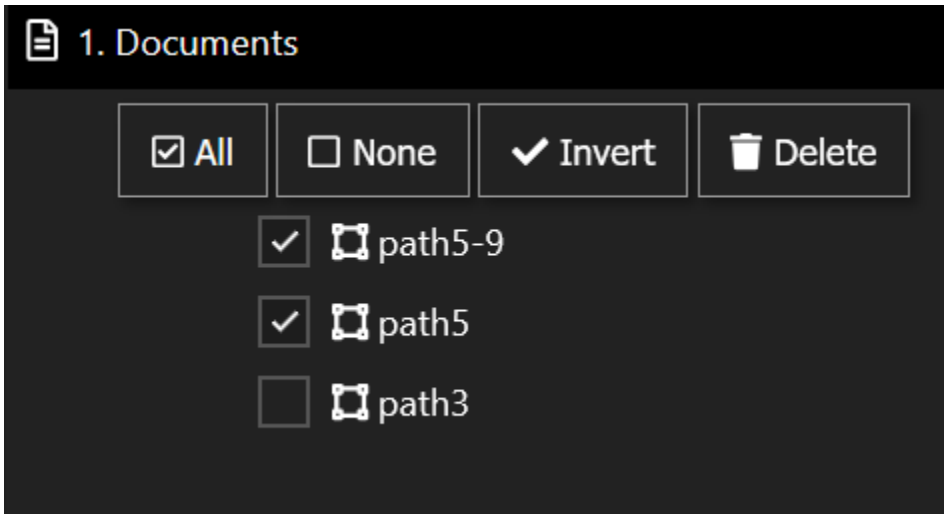
In this case, there are some internal features (short horizontal lines) and an outside outline cut.

In the Documenet window (upper right) click the open box next to your project file

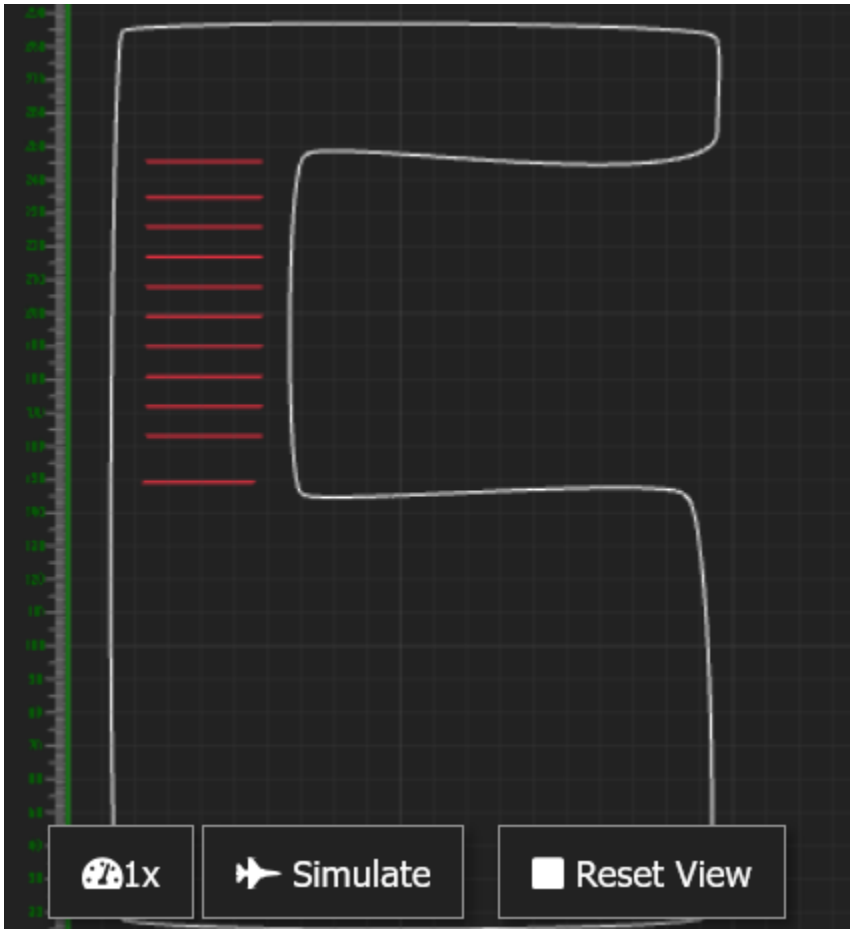


It will select all of the layers and paths in your design

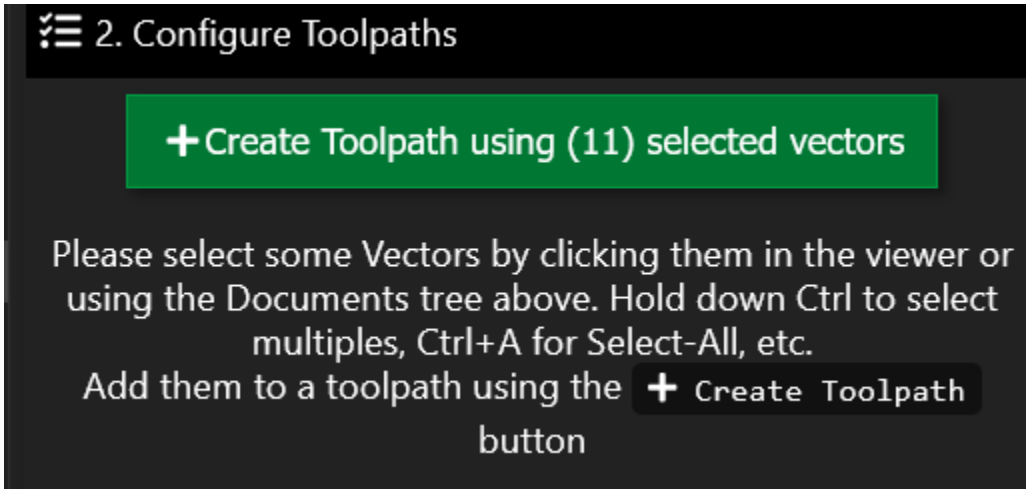
In this case, for my first toolpath, I am going to select the small internal lines. To remove the outline, I am deselecting path3. (Scroll down to see all layers and paths)



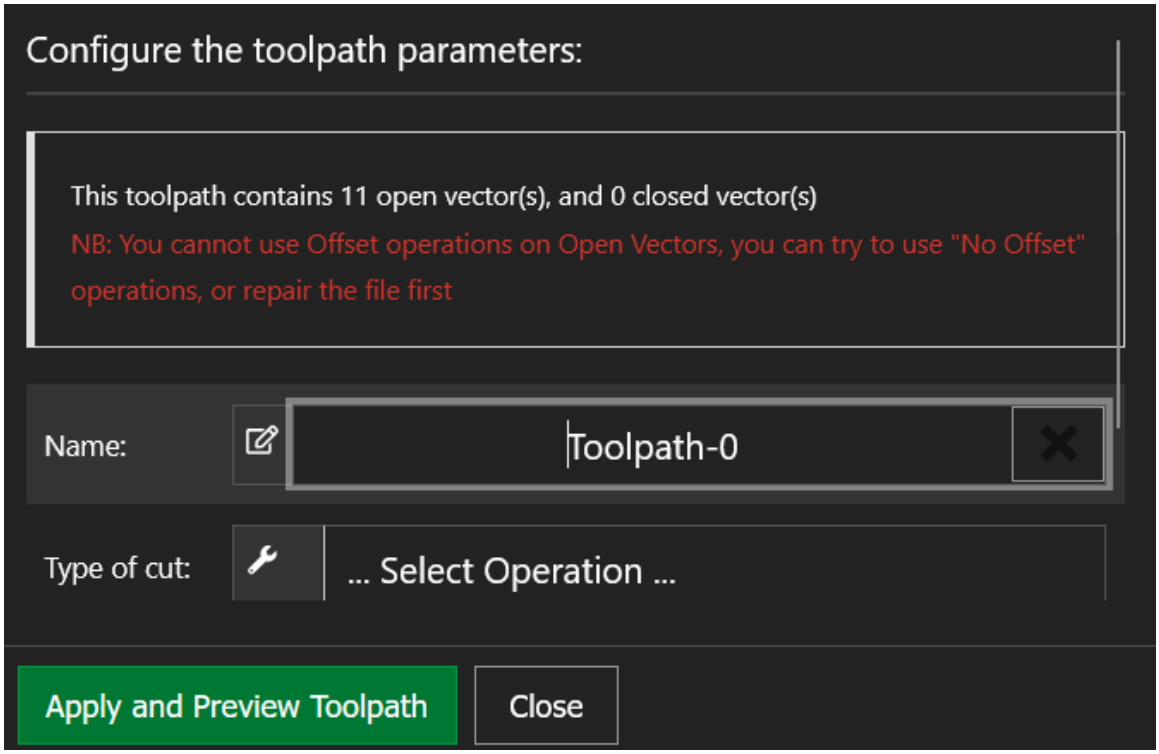
This will turn the outline from red to white. Only code for selected red outlines will be generated.



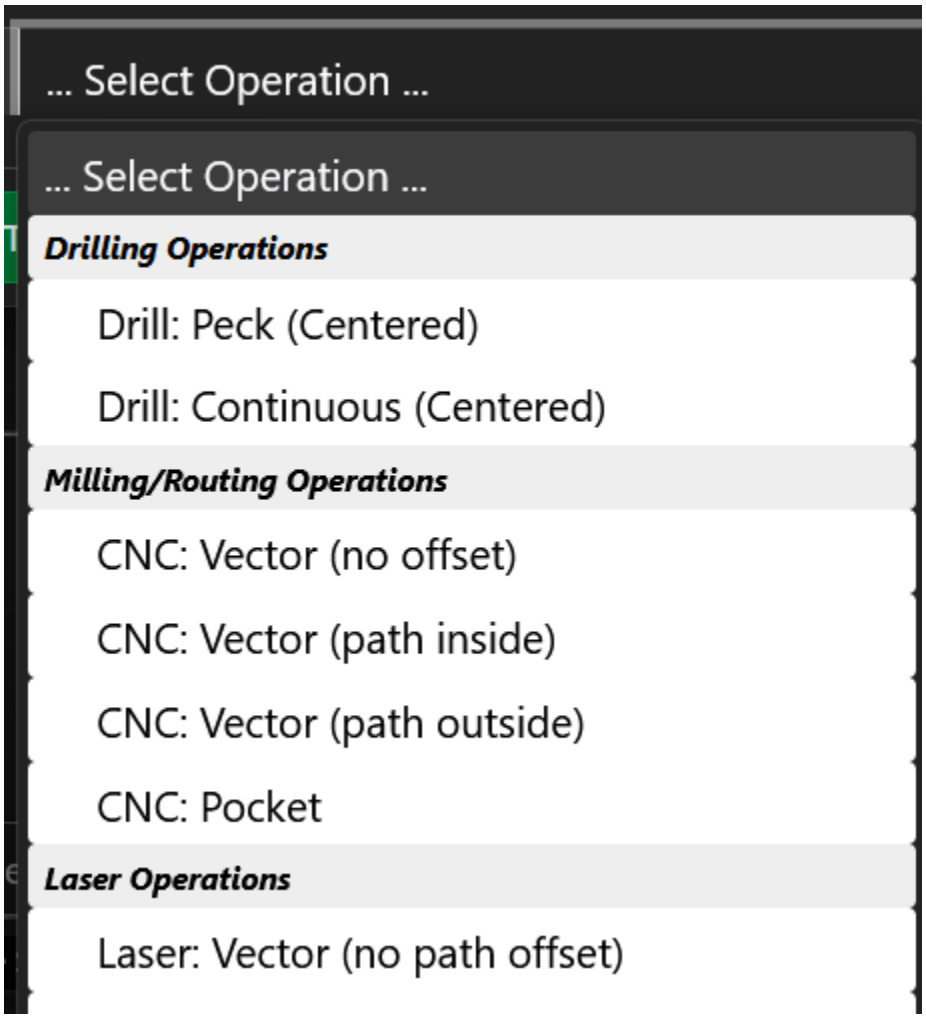
Click the green +Create Toolpath using (x) selected vectors button



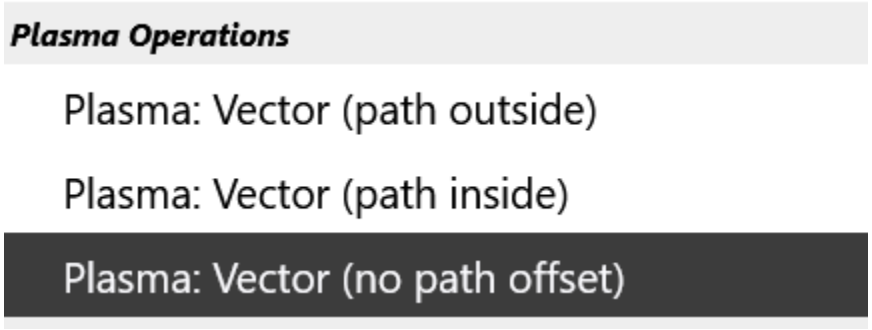
This will open a Configure the Toolpath parameters: window



Click the ...Select Operation.... Box next to type of cut.

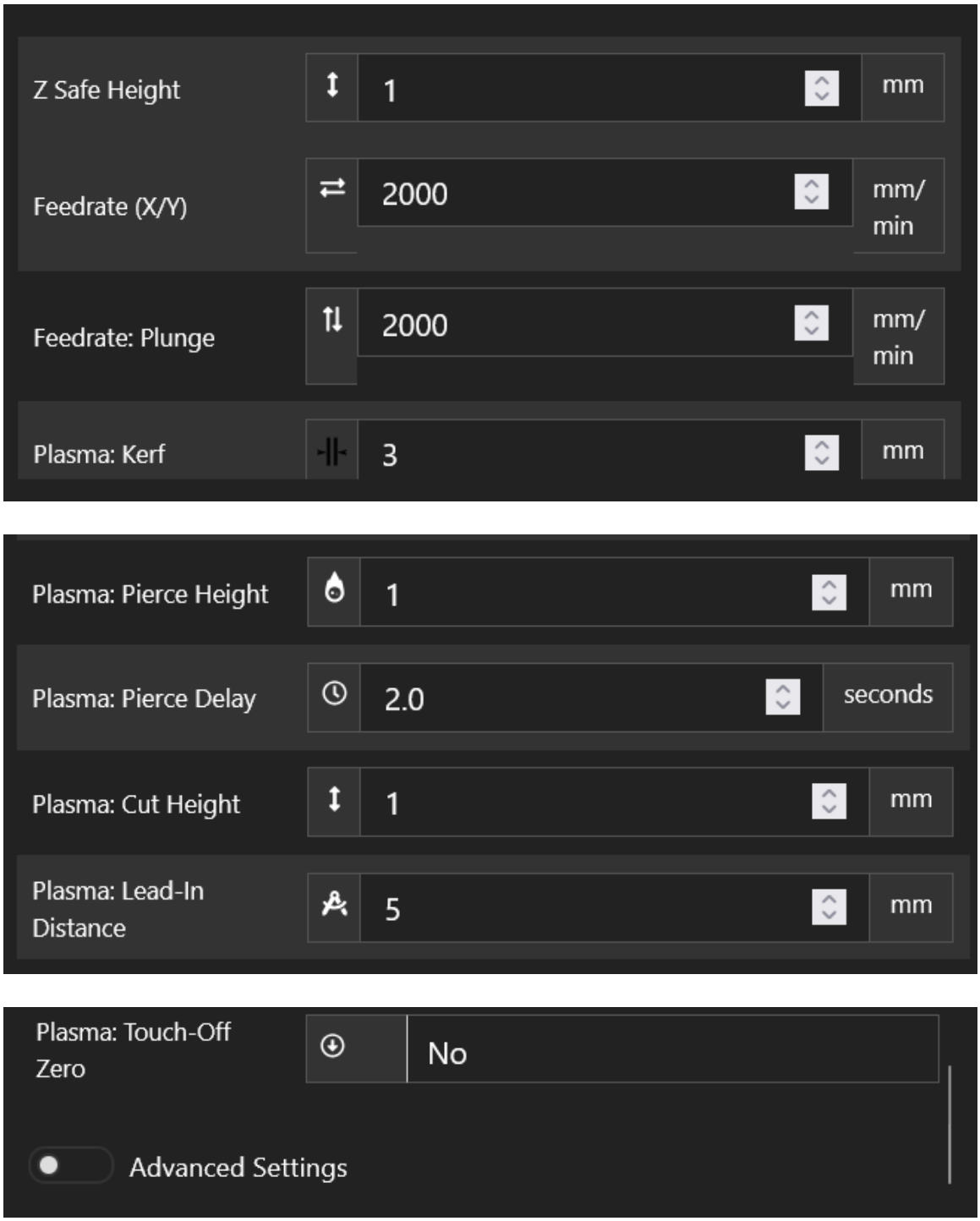


Scroll down the list to find *Plasma Operations*



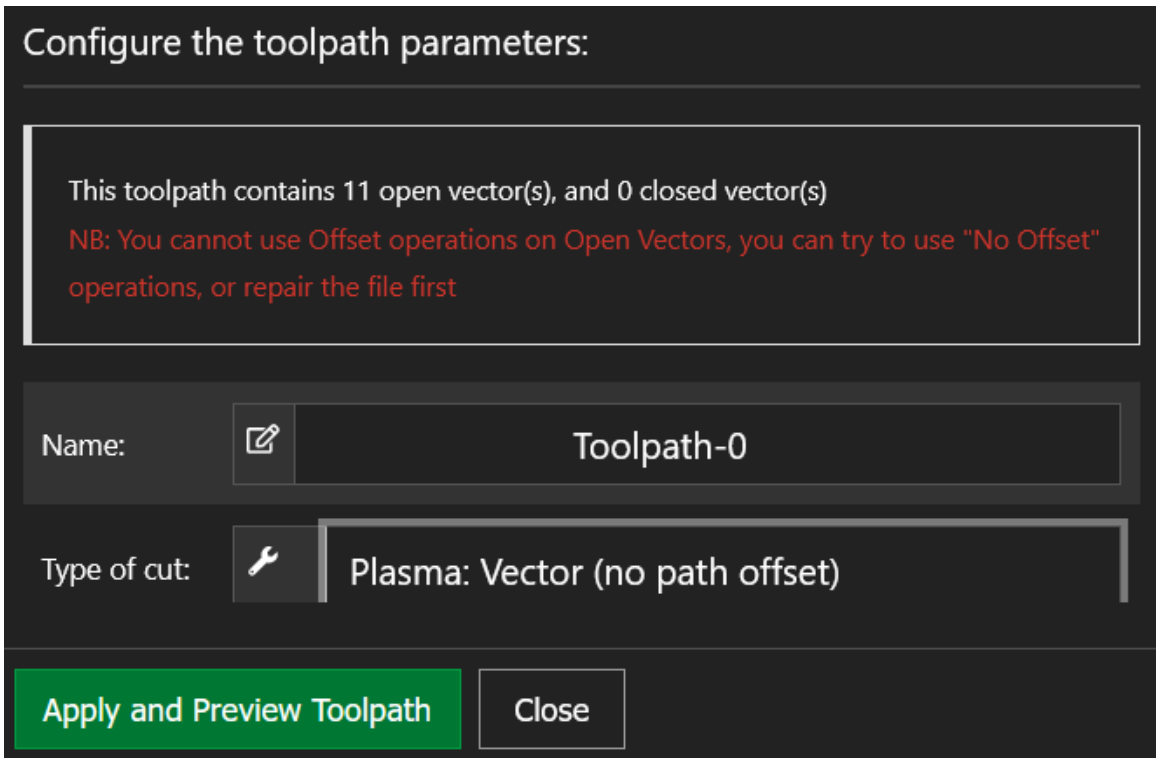
Since these are Open Vectors (just lines) we will have to use the Plasma: Vector(no path offset)

Then, Scroll down in the Configure Toolpath: Plasma: Vector (no path offset) window to set the following parameters.



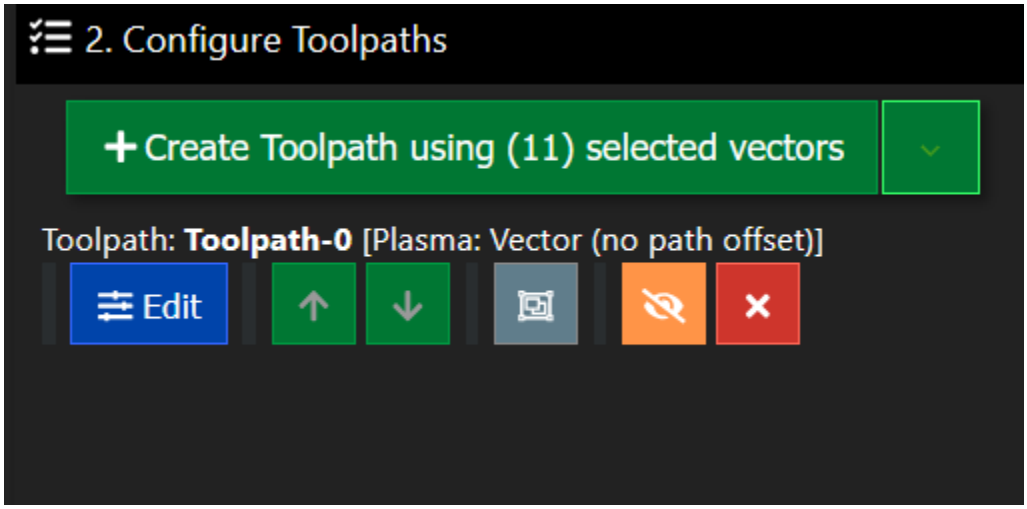
You can ignore the Advanced Settings

Z Safe Height	Set to 1 – but does not matter and this will be manually set and controlled by the Torch Height Controller
Feedrate (X/Y)	Torch speed. 2000 is a good start. Faster for thinner metal and slower for thicker metal.
Feedrate: Plunge	2000 (but will be controlled by the Torch Height Controller)
Plasma Kerf	3 (depends on nozzle, but this is a good starting value)
Plasma:Pierce Height	1 (but is controlled by the Torch Height Controller)
Plasm: Pierce Delay	This is the time it takes the plasma arc to establish and to pierce the material. 2.0 seconds is a good start value. Note: you **MUST** have the decimal point
Plasma: Cut Height	1 (but is controlled by the Torch Height Controller)
Plasma: Lead-In	This is how far off the actual edge of your work the Plasma torch will start cutting. Then it will move to the actual cut line. Prevents spoiling your edge.
Plasme: Touch-Off Zero	No



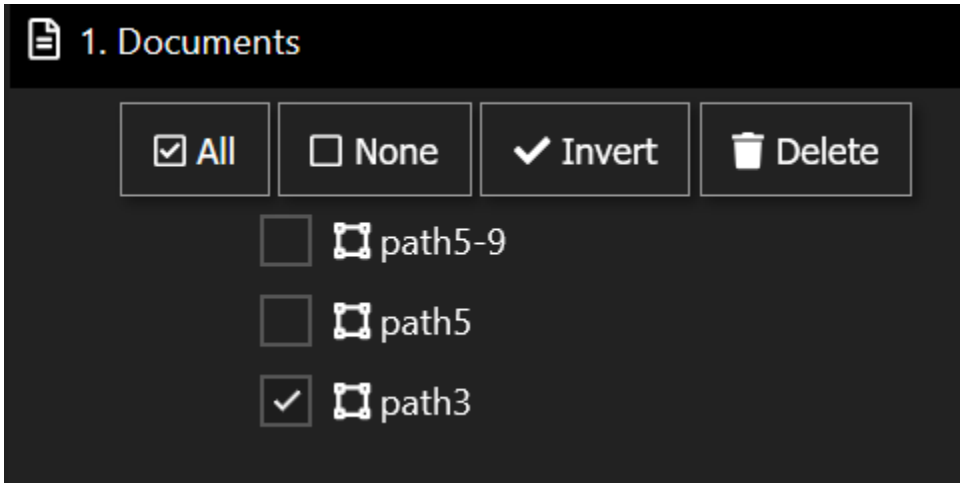
Then, click the Apply and Preview Toolpath button

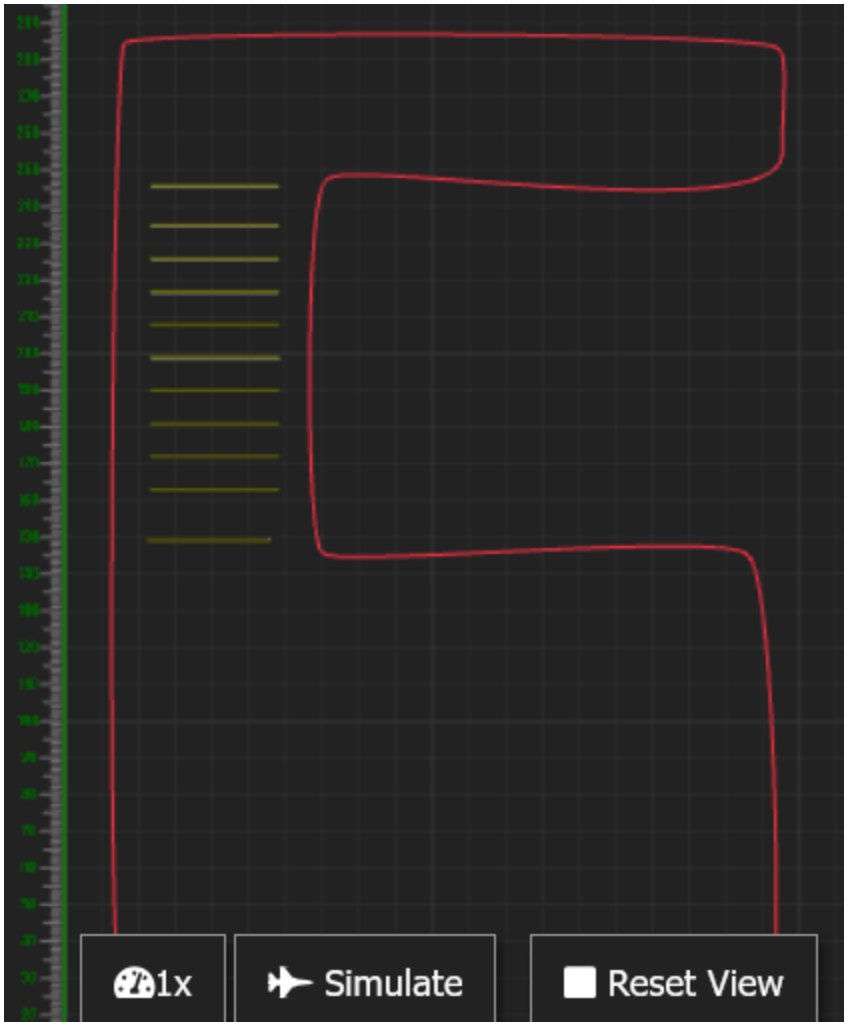
The Toolpath will be created and listed in the Configure Toolpaths window



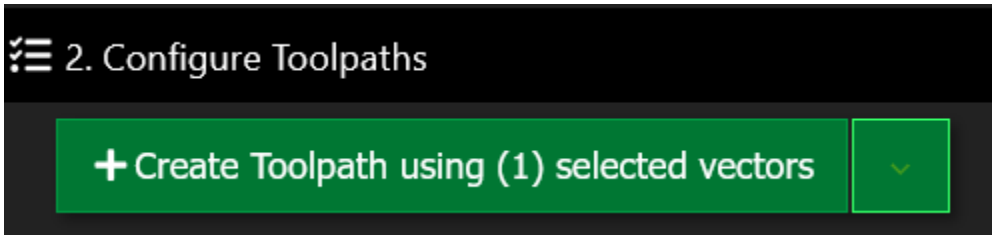
Next, lets get the outside cutout line.

In the Documents window, Unselect all of the paths except for path3

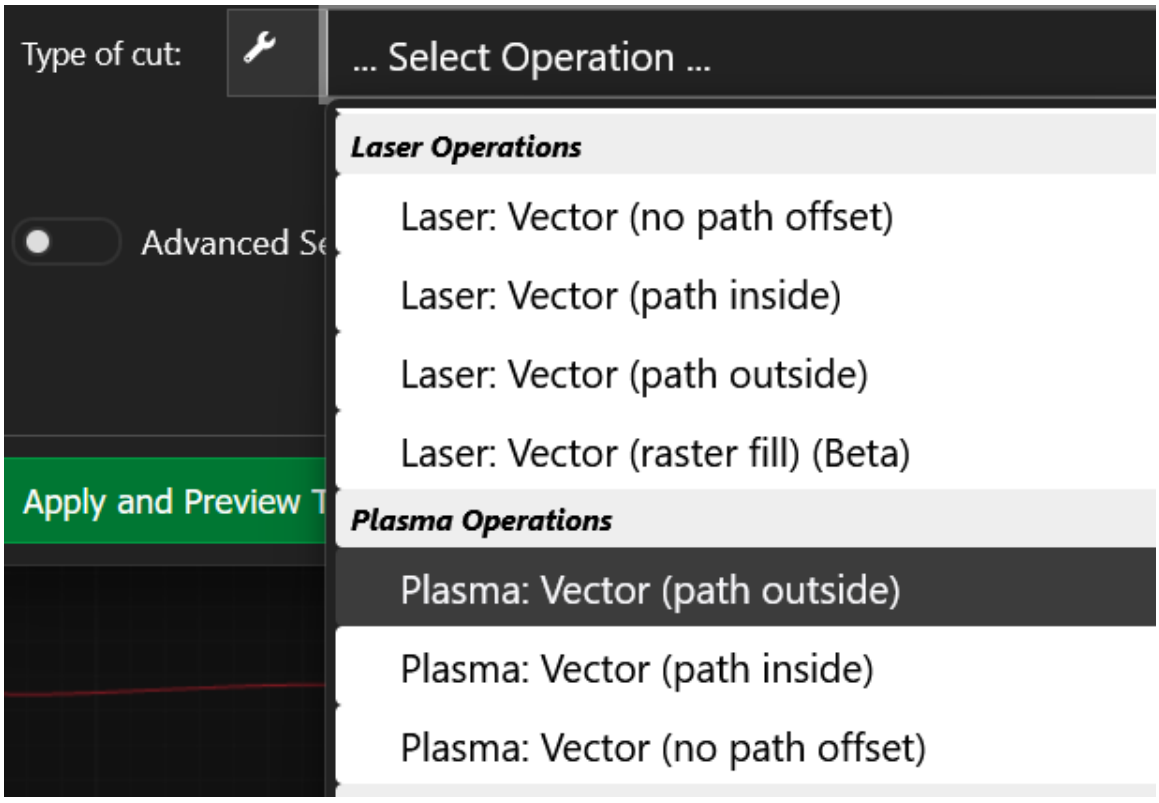




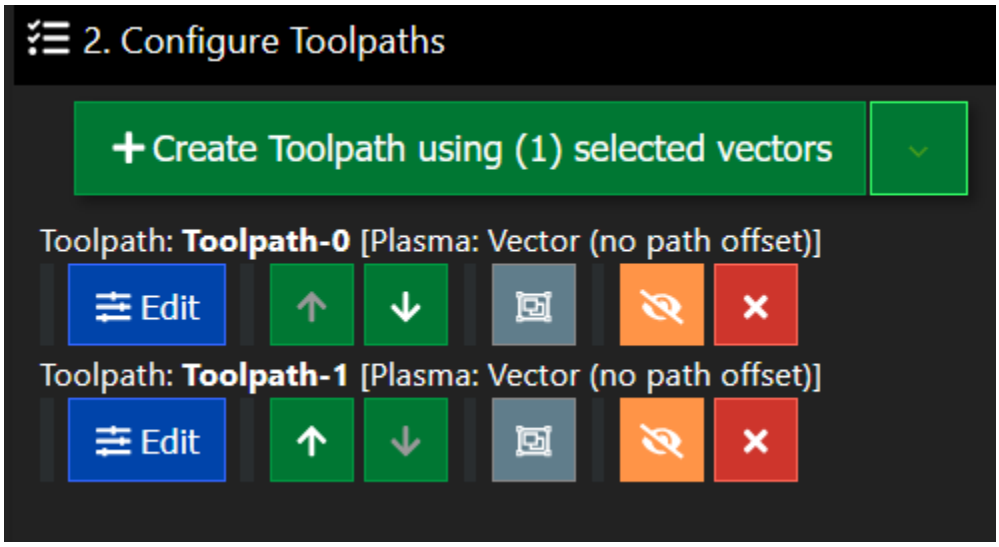
Then, click the +Create Toolpath using (1) selected vectors button



For type of cut, click ...Select Operation.... And choose Plasma Outside

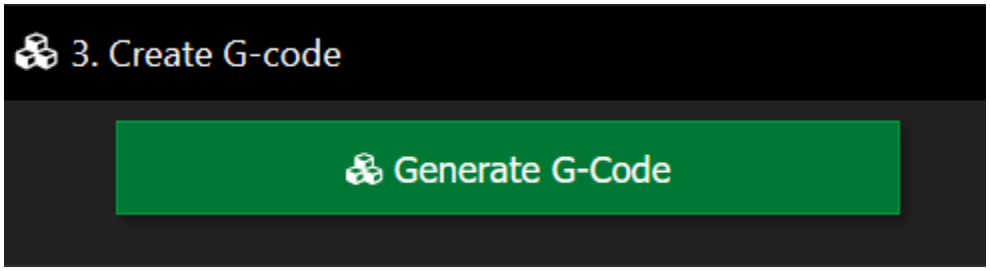


This will create a second toolpath with just the outside cut



Now, let's get the Gcode.

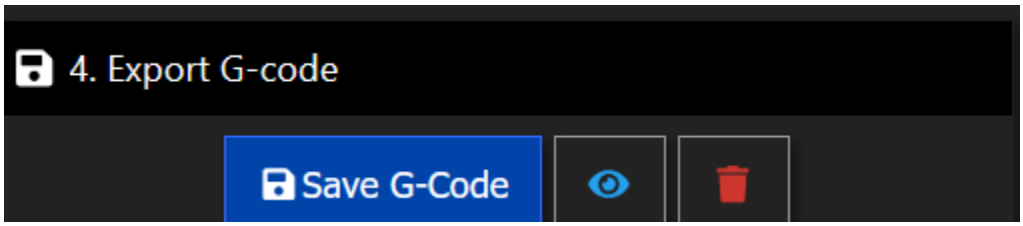
Click the Generate G-Code button



It will take only a moment to create the gcode.

Now, under the Export G-Code window, a blue Save G-Code button will appear.

Click it.



A Save GCODE window will appear.
You can rename the file of just click the Save button
File will be saved to your Downloads folder

Let's take a look at the resulting G-Code

; GCODE Generated by cam.openbuilds.com on 2025-10-29
G21 ; mm-mode
G21; mm-mode

; Operation 0: Plasma: Vector (no path offset)
; Plasma Kerf: 3 (this will cut the horizontal short lines)

G0 Z1; move to z-safe height (Z moves are ignored by the Plasma Cutter
G0 F1000 X23.2934 Y216.7788 (moves to the starting X and Y)

G0 Z1; Move to Pierce Height (again, ignored)
M3S1000; Tool On (turns on the Plasma Torch)
G4 P2.0; Pierce Delay (Pause, for arc to establish and cut through)
G1 F2000 Z1.0000; Direct Plunge (again, ignored)

G1 F2000 X58.2749 Y216.7788 Z1.0000 S1001000 (cutting move first line)
M5; Tool Off (torch off)

G0 Z1; move to z-safe height (ignored)
G0 X23.2934 Y225.7438 (more to start spot for next cut)

G0 Z1; Move to Pierce Height (ignored)
M3S1000; Tool On (turns on the Plasma Torch)
G4 P2.0; Pierce Delay (Pause, for arc to establish and cut through)
G1 F2000 Z1.0000; Direct Plunge (ignored)
G1 F2000 X58.2749 Y225.7438 Z1.0000 S1001000 (makes next cut)
M5; Tool Off (Torch off)

Etc. for the rest of the short line cuts

Then the outline

G0 Z1; move to z-safe height
G0 F1000 X15.7760 Y18.5196 (move to start spot)

G0 Z1; Move to Pierce Height (ignored)
M3S1000; Tool On (turns on the Plasma Torch)
G4 P2.0; Pierce Delay (Pause, for arc to establish and cut through)
G1 F2000 Z1.0000; Direct Plunge (ignored)
G1 F2000 X15.3157 Y19.3762 Z1.0000 S1001000
G1 F2000 X15.0999 Y20.3419 Z1.0000 S1001000 (then a whole bunch of X and Y
G1 F2000 X14.9382 Y21.3369 Z1.0000 S1001000 moves to cut the outline)
G1 F2000 X14.8071 Y22.3305 Z1.0000 S1001000
G1 F2000 X14.6953 Y23.3216 Z1.0000 S1001000
G1 F2000 X14.5966 Y24.3147 Z1.0000 S1001000
G1 F2000 X14.5073 Y25.3146 Z1.0000 S1001000

(and finally)

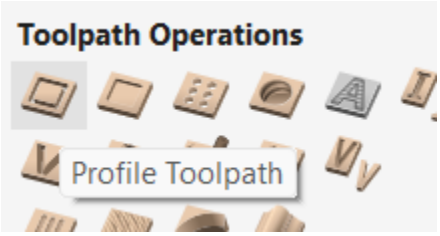
G1 F2000 X15.7760 Y18.5196 Z1.0000 S1001000 (last few X Y moves)
G1 F2000 X15.7760 Y18.5196 Z1.0000 S1001000
M5; Tool Off (Torch off)
; retracting back to z-safe
G0 Z1 (ignored)

M5 (End of Program)

Using Vcarve to make Gcode.

Launch Vcarve and set the size of your workpiece.
Design your vectors or import an .svg file

Select the vectors you want to cut in each operation, select Profile cutting.



Start Depth: 0
Cut Depth: Really doesn't matter but should be thiker than your material.

Tool: Select Plasma Cutter

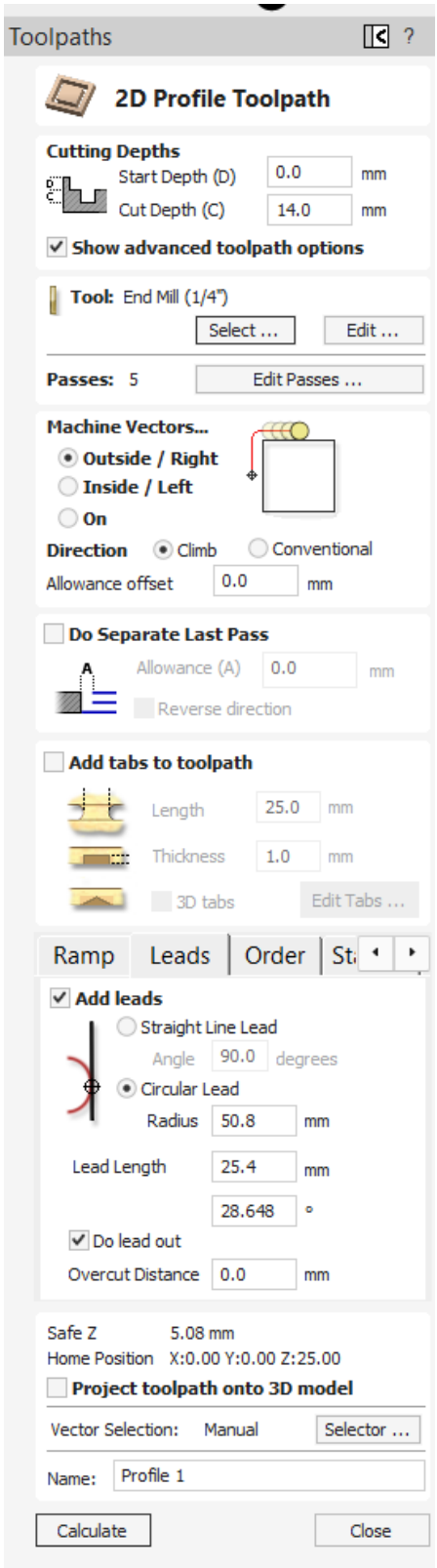
Passes: Edit to 1

Machine Vectors,
Select Outside, Inside, or On as needed.

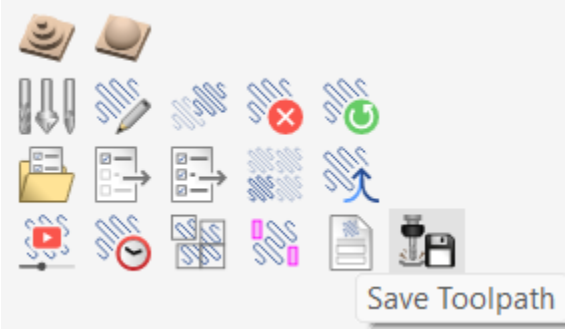
Leads
Checkbox Add leads
You can use straight or circular and set radius and length.
Checkbox Do Lead Out

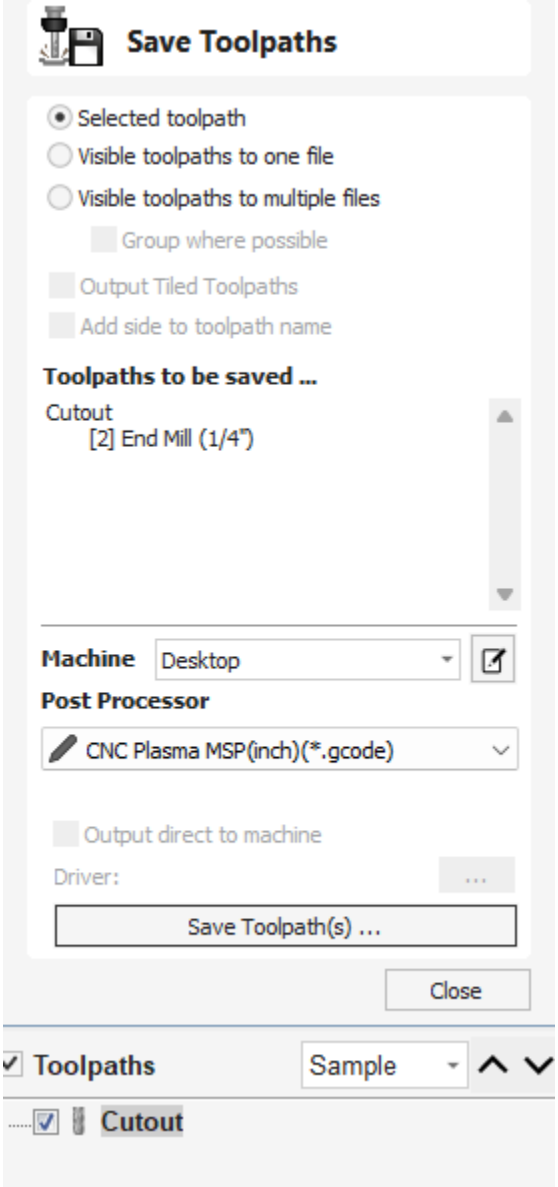
Name as needed.

Calculate.



Select Save Toolpaths





For Machine use:
Plasma Cutter

For Post Processor use:
CNC Plasma MSP(inch)(*gcode)

Save Toolpath(s)...

Let’s take a look at the Resultant Gcode

```
( Project Name )
( File created: Wednesday October 29 2025 - 02:18 PM)
( Created for BIG BLACK  CNC Plasma - Purcellville )
( Material Size )
( X= 23.622, Y= 23.622)
(Tools used in this file: )
(Plasma Torch)
G20 (inches)
G40 (Cancel cutter Radius Compensation)
G80 (Cancel canned cycle)
(Toolpath: Cutout Tool: Plasma Torch)
G94
(Rapid)
G0 X4.736Y8.833                 (moves to the start of the cut location)
(Rapid)
G0                              (Ignored)
(Plunge = Start Torch)
M03                             (Strikes the Torch)
G04 P2.0                        (Pause for arc to establish and pierce material)
G1X4.714Y8.844F80               (X and Y moves to cut out project)
X4.692Y8.855
X4.670Y8.866
X4.647Y8.877
X4.625Y8.887

.....

X0.626Y1.921
X0.614Y1.943
(Torch Off)
M05                             (Torch Off)
G0 X0 Y0                        (move to X0 and Y0)
M30                             (End of Program and Rewind)
```

General cutting information.

A definitive metal thickness and settings chart for the LGK-160IGBT is not publicly available

LGK-160IGBT general performance and settings

- **Cutting range:** For carbon steel, the LGK-160IGBT has a minimum severance of 1mm and maximum severance cut of 55mm and a high-quality cut of up to 35mm.
- **Output current range:** The amperage is adjustable from 40 to 160 amps.
- **Operating air pressure:** The recommended range for the compressed air is between 0.4 and 0.6 MPa (which is 4 to 6 bar or about 58 to 87 psi).
- **Cutting speed:** For CNC use, the operating speed is variable and depends on the metal thickness.

How to approximate the settings for your project
Because a universal chart is not provided, you must start with a conservative approach and fine-tune your settings based on the material and desired cut quality.

1. Determine the correct amperage and air pressure
Start with the minimum recommended settings for your metal thickness and increase them gradually.

- **Thin materials (1–5 mm):** Set the amperage and air pressure on the lower end of their ranges. Too much power can blow through thin metal too quickly or cause excessive warping.
- **Medium materials (6–20 mm):** Use mid-range settings for both amperage and air pressure. This should provide a good balance of speed and cut quality.
- **Thick materials (20–35 mm):** For a clean, high-quality cut, push the amperage and air pressure closer to their maximum rated values. This will ensure the plasma arc can cleanly and consistently penetrate the material. For the maximum severance cut of up to 55mm, you will need to be at the maximum settings and should expect a rougher cut.

2. Select the right consumables
The nozzle size directly impacts your cut. A smaller nozzle focuses the arc for finer detail on thinner materials, while a larger nozzle is necessary for more power and deeper penetration on thicker metals.

3. Test on scrap material
Before cutting your final piece, perform test cuts on similar scrap material. This is the most reliable way to find the perfect settings. Adjust your amperage, air pressure, and travel speed until you achieve the desired cut quality and minimize dross (melted material on the underside of the cut).

Cutting current and arc voltage (edited from the manual for some clarity)
The selection of cutting current should be according to the diameter of the nozzle, the relationship between the two should be as: I(current A)＝(70～100)×ϕ(mm).

Current = 70 to 100 x cutting speed in mm/min.

As the metal thickness increases, the influence of arc current to the cutting speed will become less. But increase of current may result in burning damage and will be detrimental to the electrode and the nozzle. So, when cutting a thick metal work-piece, the increase in cutting speed results in increasing arc voltage. The actual arc voltage is determined not only by the gas type but also by the air flux and nozzle shape. Working voltage increases with the increase of gas flux.

Gas flow Q

The arc voltage increases as the air flow increases, that is, the arc power, the cutting speed, as well as the cutting capacity and quality are improved accordingly. Because the arc compression level increases, the energy is more concentrated, the arc beam temperature, the arc spraying speed, as well as the arc current impulsion increases. But too much current may cause instability of the plasma arc. Usually, no change is made to air flow for one torch. But it can be adjusted a little when the cutting torch or cutting thickness is different.

Nozzle Selection and Amperage

Nozzle aperture diameter (mm)	1.0	1.2	1.4	1.6	1.8	2.0	2.2	2.4	2.6
Suitable current (A)	20~30	30~40	40~65	70~90	80~100	110~130	140~170	180~210	220~250

Cutting Speed

Typically, around 2000 mm/min. Faster (3000mm/min) for thinner material, slower (maybe down to 1000mm/min) for thicker material.

Testing of nozzle, air pressure, amperage, and cutting speed will be required to get the best result.

Torch Height Controller description

THC settings for various thickness metals

Creating design

CAD (Computer Aided Design). There are may programs in which drawings of parts may be made. Specific CAD instruction is beyond the scope of this checklist. Some tools are, Inkscape, Fusion360, SolidWorks, Correll, FreeCAD, TInkercad

Generating Gcode

There are many ways to create Gcode. Generally you would like your Gcode generator to create code for a plasma cutter. Not all Gcode (CAM) programs do this. You may need to manually edit your Gcode. Detail instruction is beyond the scope of this checklist.

Fixing Gcode

Depending on the CAM application that generates your gcode, you may need to manually edit.
Your Gcode will need to move the torch to the starting X and Y location
This is usually a G0 X(location) Y(location)
Any Z axis movements in Gcode will be Ignored.
An M3 code (on its own line) will strike the Torch.
Remove any Speed commands (These S commands refer to the RPM at which to spin a spindle. They are not F commands which are cutting speed (movement across the table.) They will begin with an S, like S1000. (This refers to spindle speed, of which there is none on the Torch. Torch power is controlled by the Amperage setting on the front of the Plasma Source.)
You will need a piercing pause immediately following the M3 command
A delay of 2 to maybe 5 seconds will be needed depending on the thickness of the material. This is accomplished with a G4 command and a Pause time.
Example: G4 P2.0 Causes a 2 second pause in the Gcode. Note: the decimal point is required by Mach4.

Cutting movements should all be G1 commands with an X, Y, and optional F (Feed speed). Z axis movements are ignored and Speed (S) commands should be removed (they slow down the code execution).
M05 will turn off the Torch
M30 will end the program and reset the code to the beginning.

G20 puts the machine into inch mode.
G21 puts the machine into millimeter mode

G00 is a rapid move to an X Y location
    G00 X10.0 Y10.0
G01 is a controlled move at the designated Feed speed
    G01 X20.0 Y20.0 F2000 (2000 mm per minute when in mm mode)

Feed speeds in mm/min for mild steel are around 2000. Faster (up to 3000) for thinner metals, slower (1000 of 1500) for thicker metals. Test.

Utilities

Inkscape – Vector drawing program

Camotics – Gcode Simulator.

Web Utilities

OpenBuilds Gcode Generator
https://cam.openbuilds.com/
Will take a .svg file and create Plasma Cutter Gcode