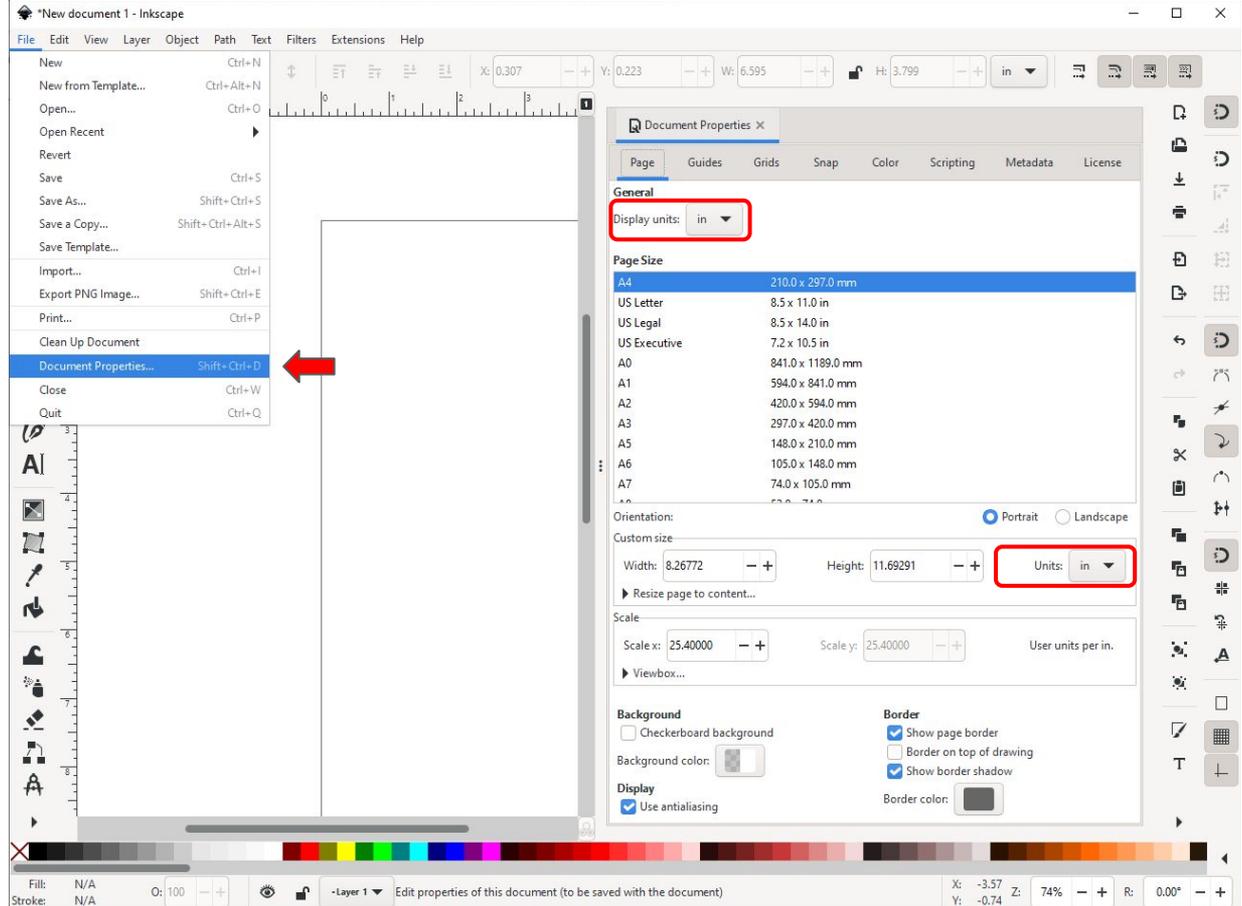


CNC Plasma Training

The Workshop by TBK Bank

Designing a simple sign in Inkscape

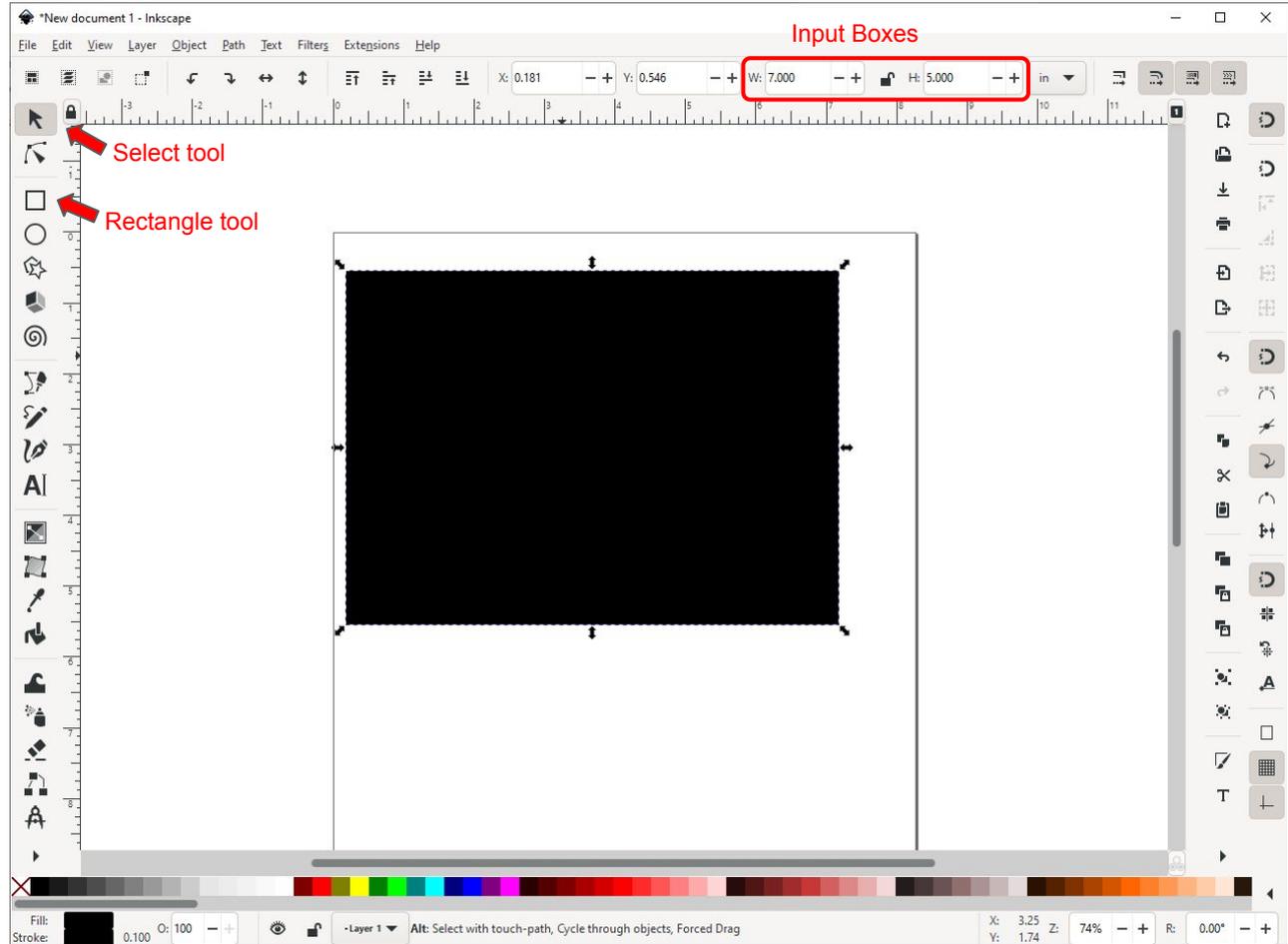
1. Launch Inkscape
2. Click on *File* → *Document Properties* and change “Display Units” and “Custom Size” units to Inches



Designing a simple sign in Inkscape - Draw the Outline

3. Select the “Rectangle” tool and click and drag to place a square or rectangle on the screen

4. Choose the “Select” tool, click on your rectangle and then use the input boxes at the top of the screen to set your rectangle’s size

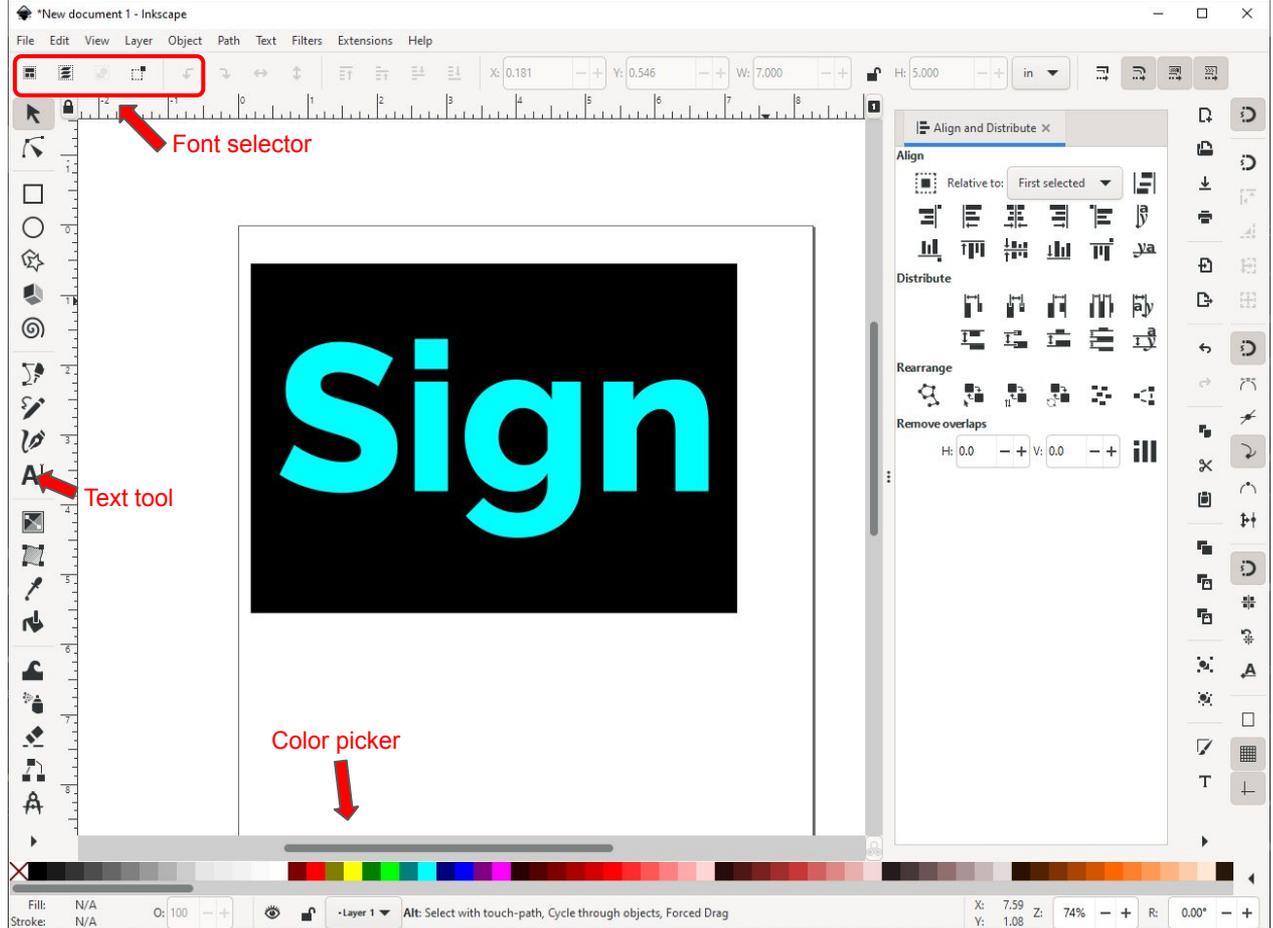


Designing a simple sign in Inkscape - Insert Text

5. Choose the “Text” tool, click anywhere on the screen, and type the text you wish to add to your sign. While you are still editing the text you can change the font using the drop down selector at the top of the screen.

6. Use the color picker at the bottom of the screen to make the text a different color than your rectangle.

7. Use the “Select” tool to move your text on top of your rectangle
***To get things perfectly aligned, you can find the “Align and Distribute” tools under “Object” in the menu*

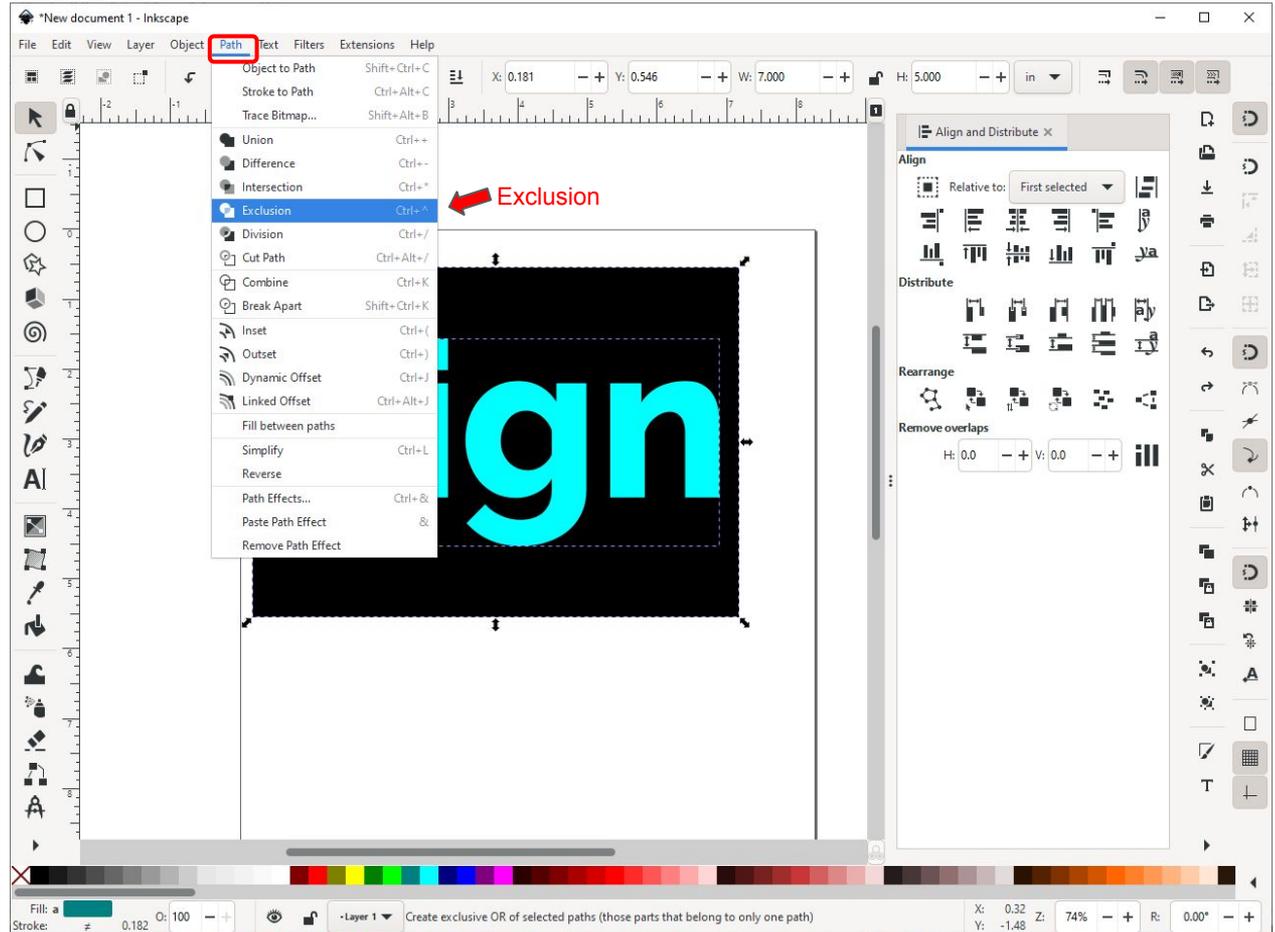


Designing a simple sign in Inkscape - Cut Out Text

8a. Using the “Select” tool, right click on your text and choose “Ungroup”

8b. Using the “Select” tool, draw a box around the rectangle and your text.

9. In the top menu, click on “Path”, then click on “Exclusion” to cut away the text.



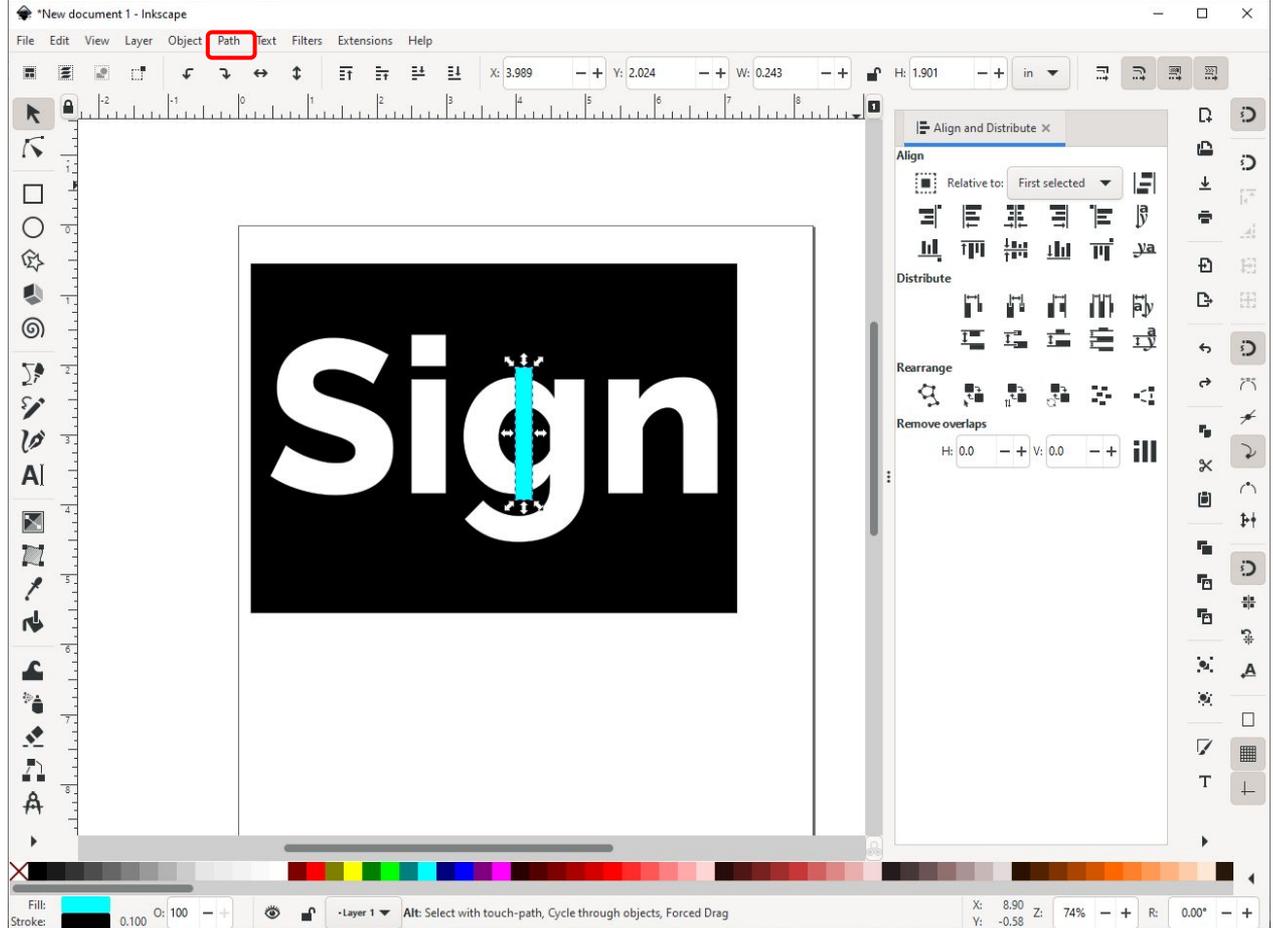
Designing a simple sign in Inkscape - Creating Bridges

Now you need to create bridges so the middle of any letters don't fall out. In our example, after we exclude our text, the middle of the "g" will fall away if cut.

10. Select the "Rectangle" tool and draw a rectangle that bridges all the way across the part of the letter(s) that will fall out.

11. Once you've got the rectangle in position, use the "Select" tool to select it along with your sign.

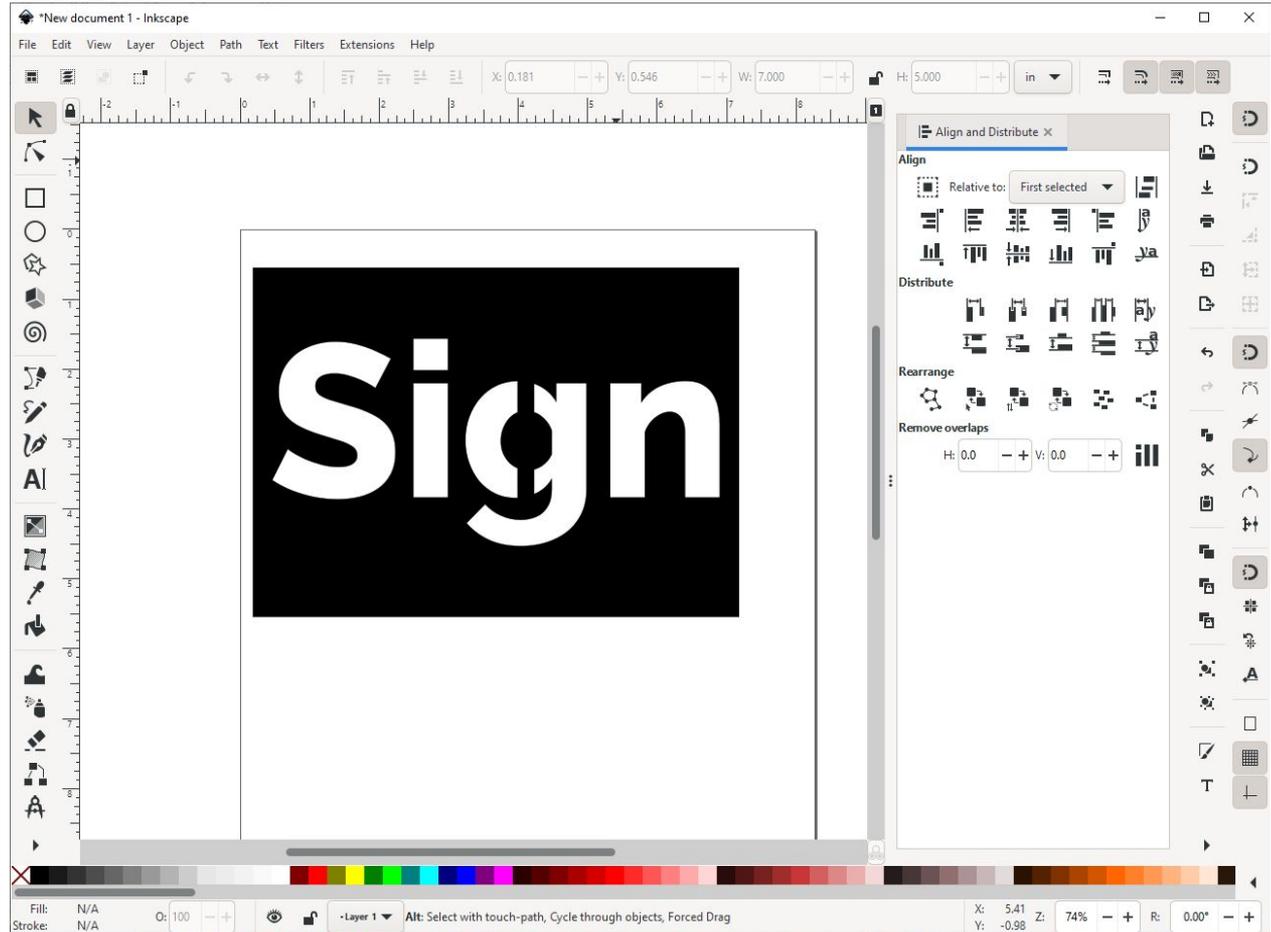
12. Click on "Path" in the top menu, and select "Union"



Designing a simple sign in Inkscape - Save your File

Make sure everything looks the way you want it to and then you're ready to save your file and move on to the next steps.

13. Click on "File" in the top menu and save your design to a USB drive to take to the machine.

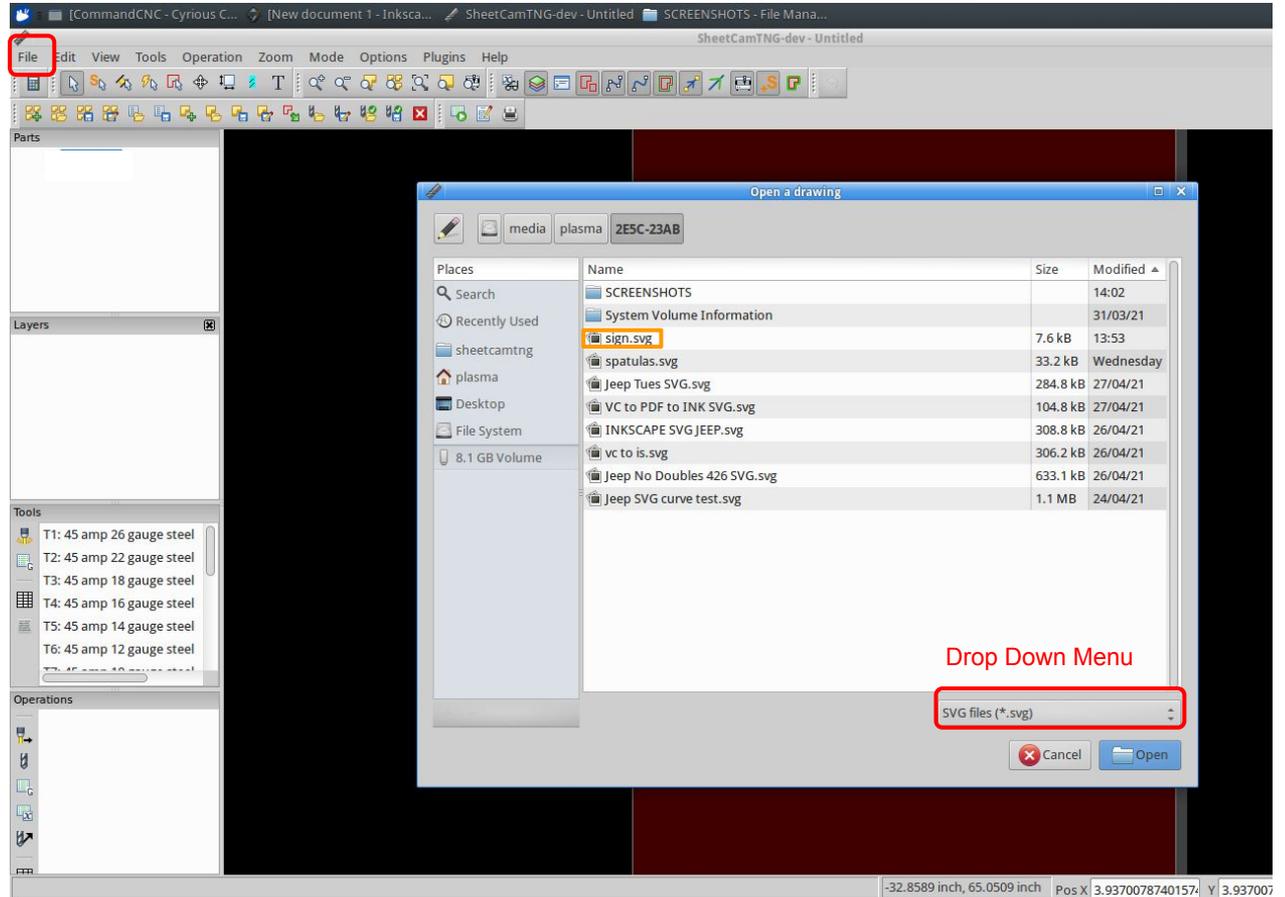


Creating your toolpaths in Sheetcam - Open your File

14. On the computer at the CNC Plasma Cutter, launch Sheetcam

15. In the top menu, click on "File" and then click "New Part".
Navigate to the USB drive, find your file, and open it.

***make sure SVG files (*.svg) is selected in the drop down menu*

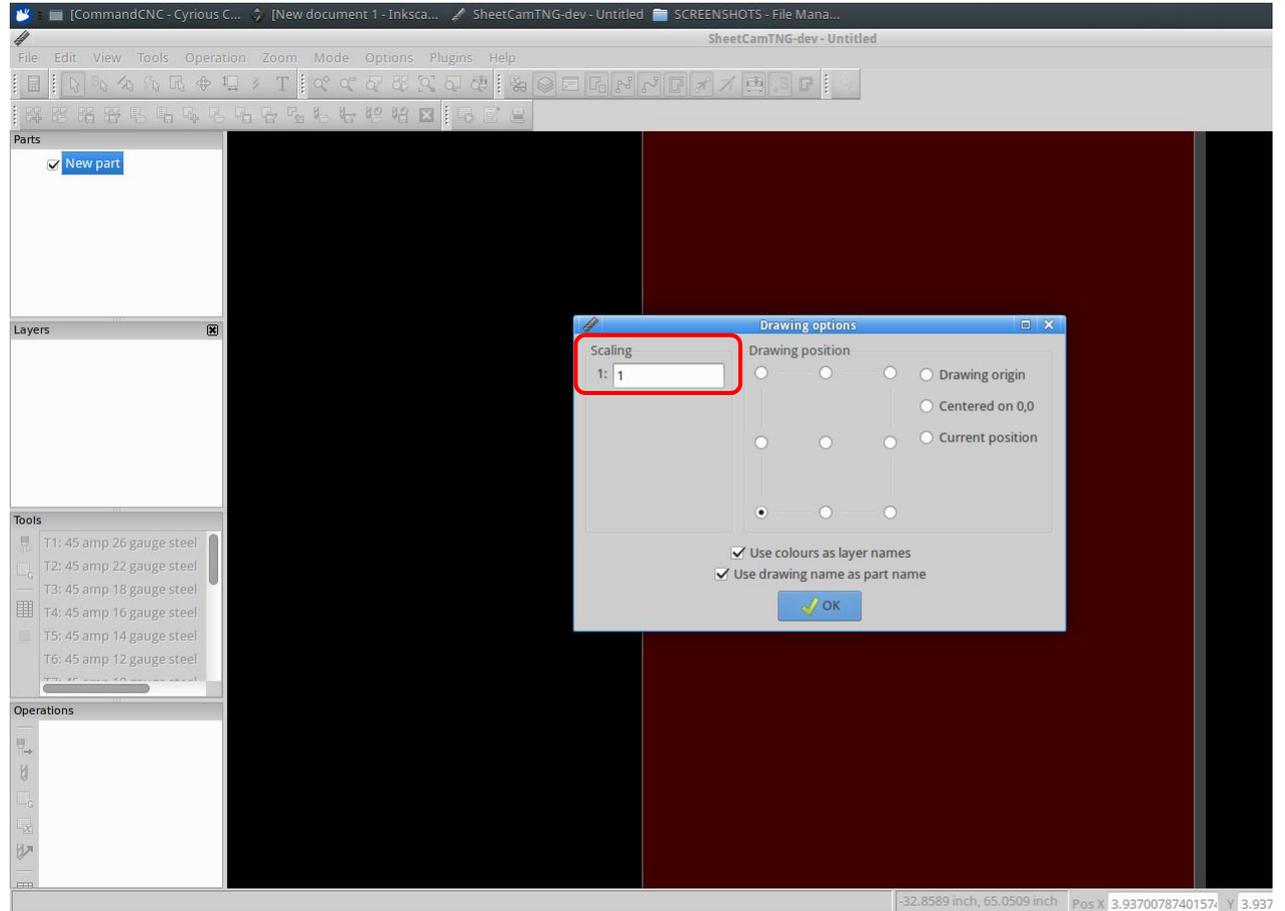


Creating your toolpaths in Sheetcam - Set your Scale

After clicking “Open”, you need to set your scale for your drawing

16. In the pop-up box, make sure that scaling is set to 1:1 and then click OK

***When you import a DXF file that is designed in inches, you need to set scaling to 1:25.4*



Creating your toolpaths in Sheetcam - Create Toolpath

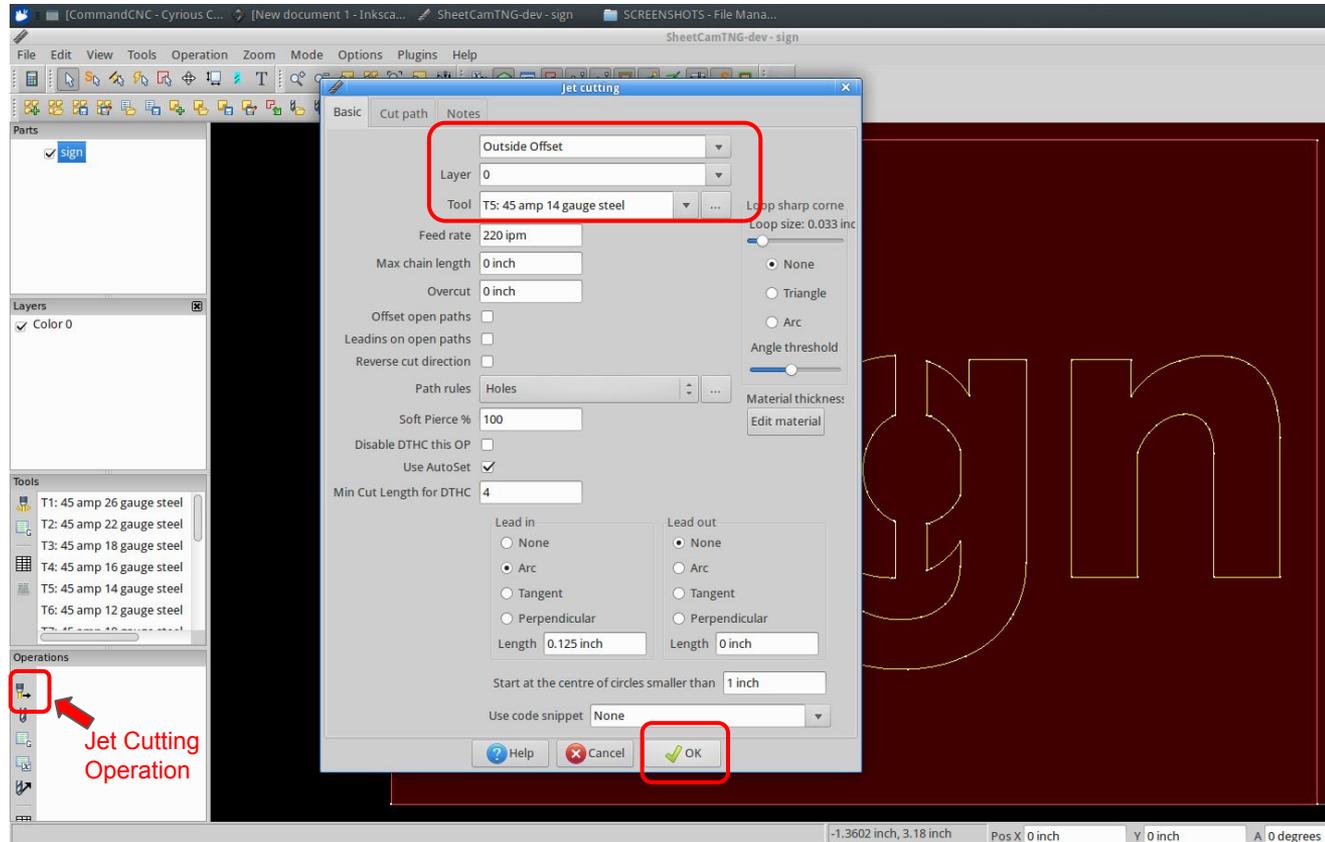
17. In the lower left portion of the screen under “Operations”, click on “Jet Cutting”

18. In the window that pops up, select Outside Offset in the first drop down menu.

19. Click the drop down next to layer and choose the active layer

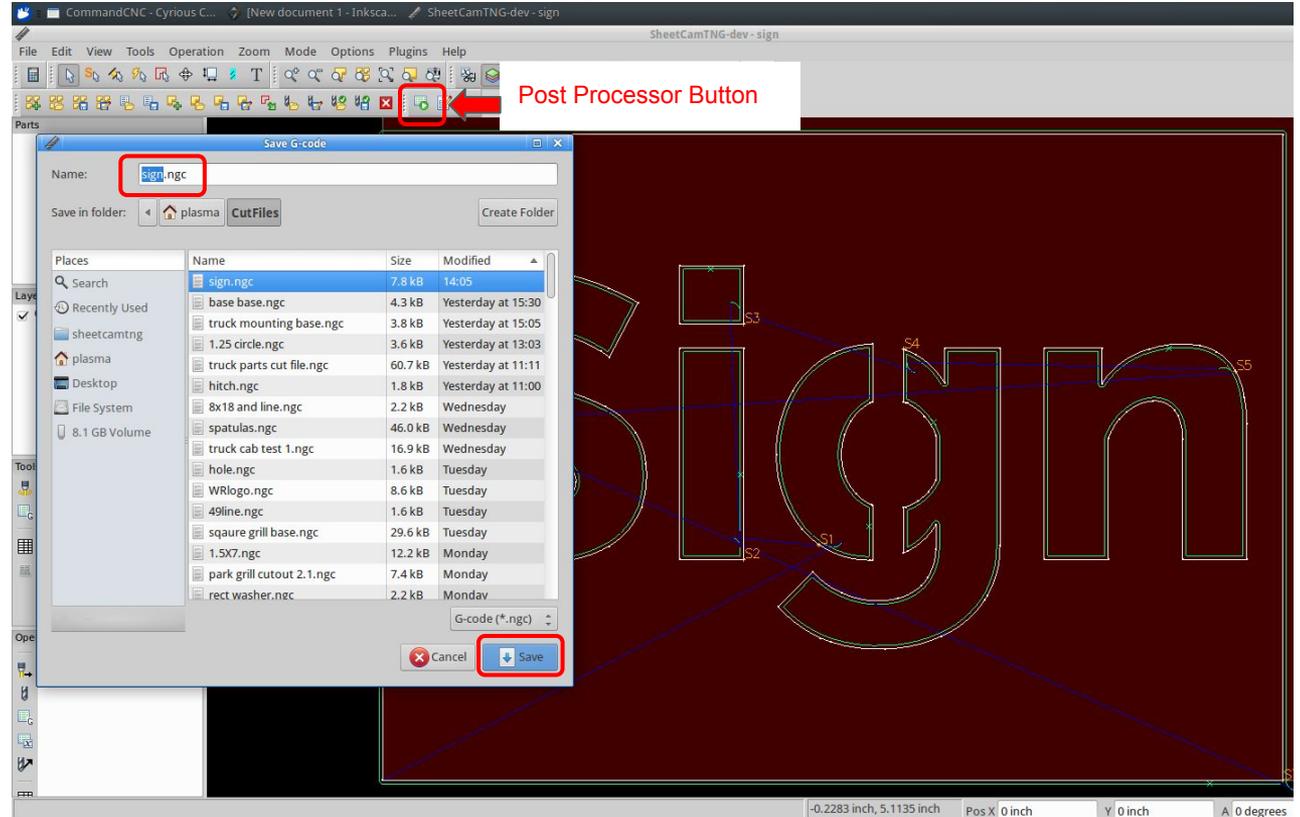
20. Select the tool according to the material you’re cutting (for this example, we’re cutting our sign from 14ga steel, so we’ll choose T5: 45 amp 14 gauge steel)

21. Click “OK”



Creating your toolpaths in Sheetcam - Create Cut File

22. Click on the Post Processor button in the top menu, give the file a name, and click save

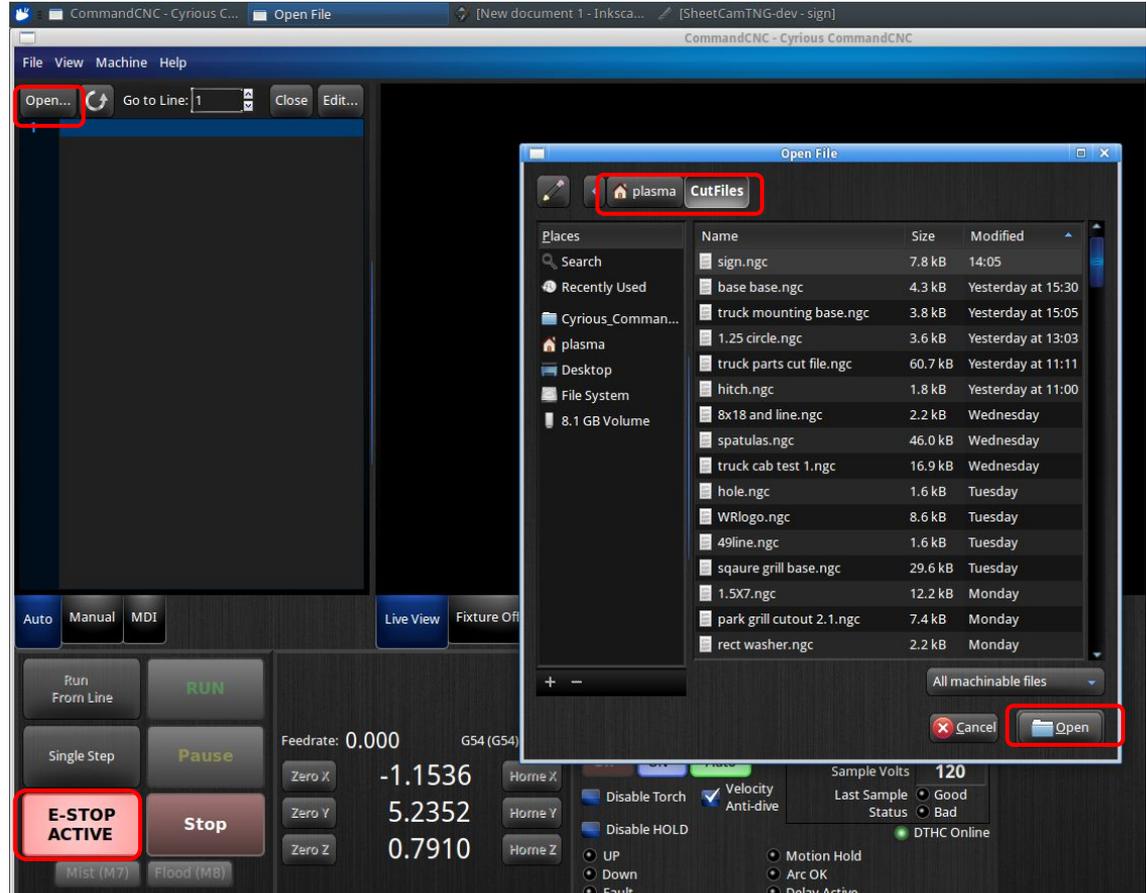


Cutting your sign (CommandCNC) - Open the File

Back on the computer's desktop,
launch CommandCNC

23. Click “Open”, then navigate
to Plasma→Cutfiles, find your file
and click “Open”

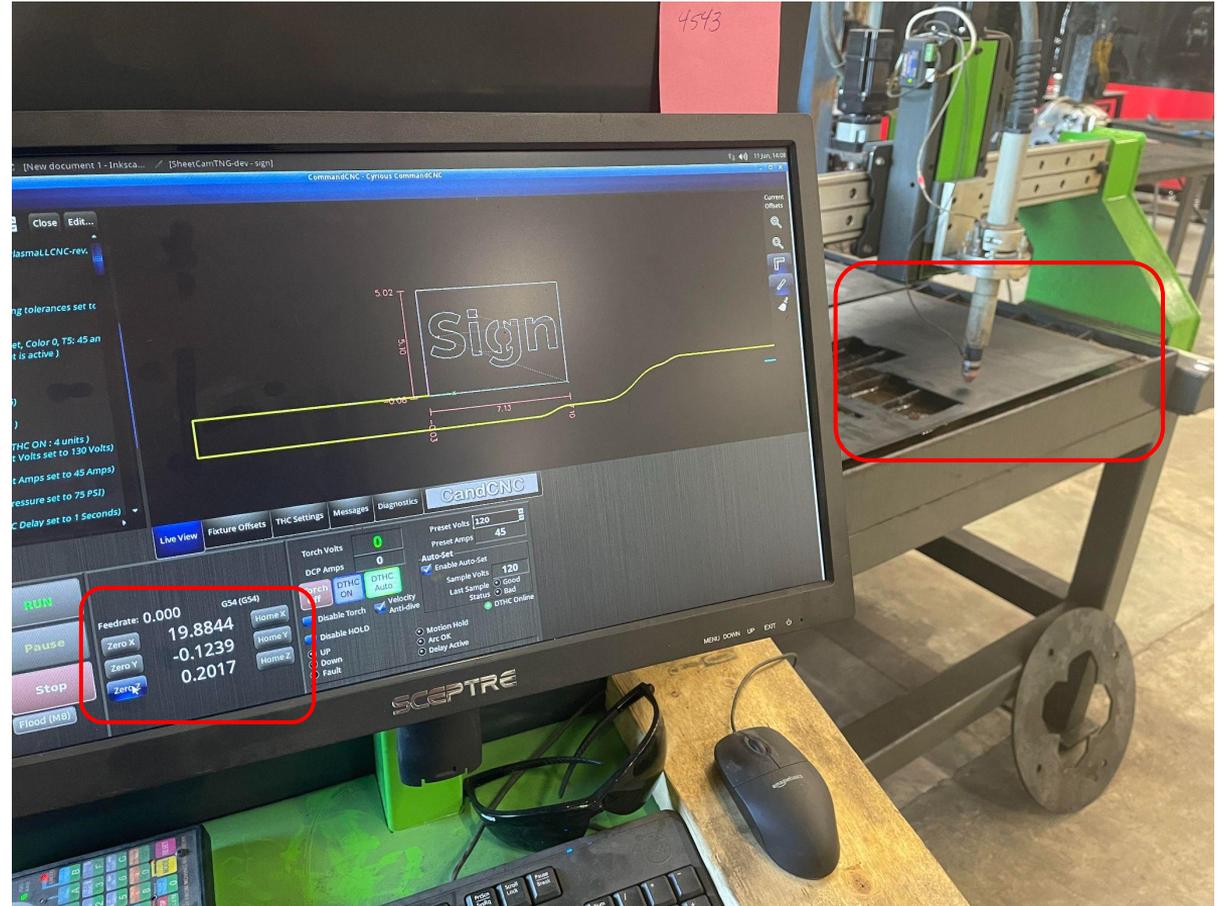
24. Click the red “E-STOP
ACTIVE” button to turn it off



Cutting your sign (CommandCNC) - Position Cut

25. Place your metal on the table, and using the arrow keys on the keyboard, position the torch where you want to cut the sign out

Notice how the X and Y coordinates change as you move the torch.



Cutting your sign (CommandCNC) - Zero the Torch

26. After positioning the torch, click “Zero X”, then click “Zero Y”

27. Click “Home Z”. This will lower the torch so it knows the height-position of the metal.

28. Finally, click “Zero Z”

The screenshot displays the CommandCNC software interface. The top menu bar includes File, View, Machine, and Help. Below the menu bar is a toolbar with buttons for Open, Go to Line (with a line number input field set to 1), Close, and Edit. The main window is divided into two panes. The left pane shows a CNC program with the following code:

```
1 (Filename: sign.ngc)
2 (Post processor: CandCNCPlasmaLLCNC-rev.
3 (Date: 11/06/21)
4 G20 (Units: Inches)
5 F10.0
6 G90 G40
7 G64 P0.010 Q0.001 (tracking tolerances set to
8 M68 E0 Q10
9 (Part: sign)
10 (Operation: Outside Offset, Color 0, TS: 45 an
11 M68 E0 Q80 (Auto Preset is active )
12 G4 P.1
13 (Preset Volts: 130)
14 (Preset AMPS: 45)
15 (Air Pressure Preset: 75)
16 (Soft pierce is off )
17 (Suggested Tip Size: 45 )
18 (DTHC Delay: 1 sec )
19 (Min Cut Length for DTHC ON : 4 units )
20 M68 E0 Q3130 (Preset Volts set to 130 Volts)
21 G4 P.1
22 M68 E0 Q445 (Preset Amps set to 45 Amps)
23 G4 P.1
24 M68 E0 Q875 (Air pressure set to 75 PSI)
25 G4 P.1
26 M68 E0 Q510 (DTHC Delay set to 1 Seconds)
```

The right pane shows a 2D plot of a sign. The plot is a square with a dashed outline and the word "Sign" cut into it. The plot axes are labeled with dimensions: 5.02, 5.10, 7.10, 7.13, 0.03, and -0.05. The plot is centered at the origin (0,0).

The bottom of the interface features a control panel with several buttons and settings. The buttons include Run From Line, Single Step, PRESS FOR E-STOP, Stop, Mist (M7), Flood, Zero X, Zero Y, Zero Z, Home X, Home Y, Home Z, Torch Off, DTHC ON, and DTHC Auto. The settings include Feedrate: 0.000, G54 (G54), Torch Volts: 0, DCP Amps: 0, Preset Volts: 120, Preset Amps: 45, Auto-Set (checked), Sample Volts: 120, Last Sample Status: Good, and DTHC Online (checked). The buttons Zero X, Zero Y, Zero Z, and Home Z are highlighted with red boxes and numbered 1, 2, 3, and 2 respectively.

Cutting your sign (CommandCNC) - Cut your Sign!

29. Click "Run"
30. Check that all settings look correct - preset volts should match book settings, DTHC should be set to Auto, etc.
31. Click "OK" if everything looks good
32. Click "Resume" to start cutting!

The screenshot displays the CommandCNC software interface. On the left, the G-code editor shows a program for cutting a sign, with line 20 highlighted. The main window shows a 3D model of a sign with dimensions. An "Operator Message" dialog box is open, displaying an information icon and text: "The PAUSE after this message is to allow the Operator to check the DTHC settings to confirm they are loaded from the G-Code and are correct. If you need to do anything do it before you hit the RESUME button." The dialog has an "OK" button. The control panel at the bottom features several buttons: "Run From Line" (with a red box and callout 1), "Resume" (with a yellow box and callout 4), "Stop", "Mist (M7)", "Flood (M8)", "Torch off", "DTHC OFF", "DTHC Auto", "Auto-Set", "Enable Auto-Set", "Sample Volts 130", "Last Sample Status", "DTHC Online", "UP", "Down", "Fault", "Motion Hold", "Arc OK", and "Delay Active". The "Pre-set Volts" field is set to 130 (with a red box and callout 2) and "Pre-set Amps" is 45. The "DTHC" is set to "Auto".