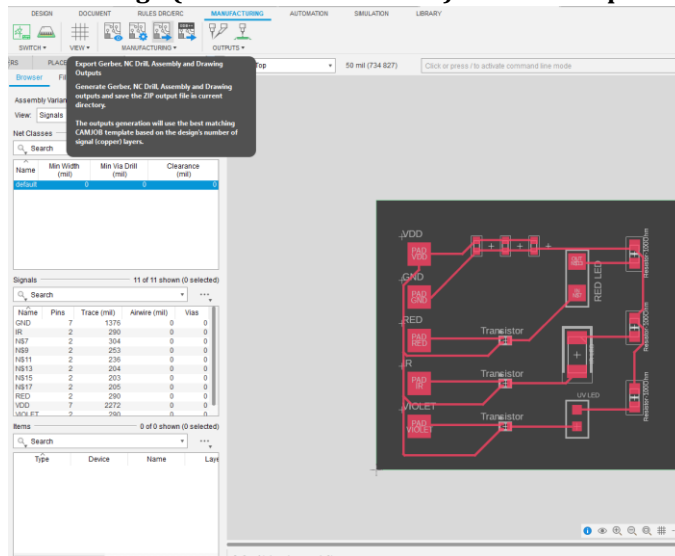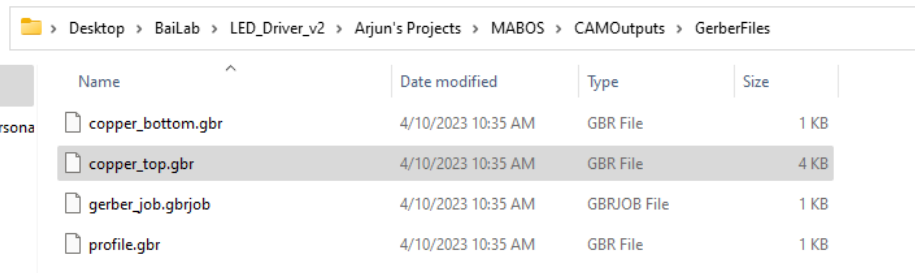**1.) Export circuit design (.brd file in Fusion360) as Gerber Zip File**
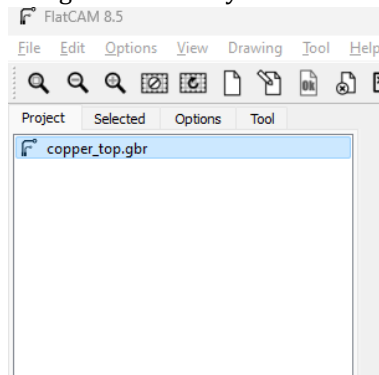

    a.

    b.   Select "**Export Gerber, NC Drill, Assembly, and Drawing Output**" output button, and export all files to a .zip file

   c.   You should see a file named *copper_top.gerber* in the *CAMOutputs* folder

       i.

2.) Open FlatCAM

    a.   Click '**File'** and '**Open Gerber'**

    b.   Navigate to the file you saved above (*copper_top.gbr)*, select said file


    c.

    d.   You should see the above file in the project list. Double Click it

    e.   Modify the parameters as listed below for **Total Diameter** and **Pass Overlap**, confirm the resulting design properly traces the counters of each trace be clicking **Generate Geometry** and visually confirming accuracy, then select **File** -> **Export SVG**
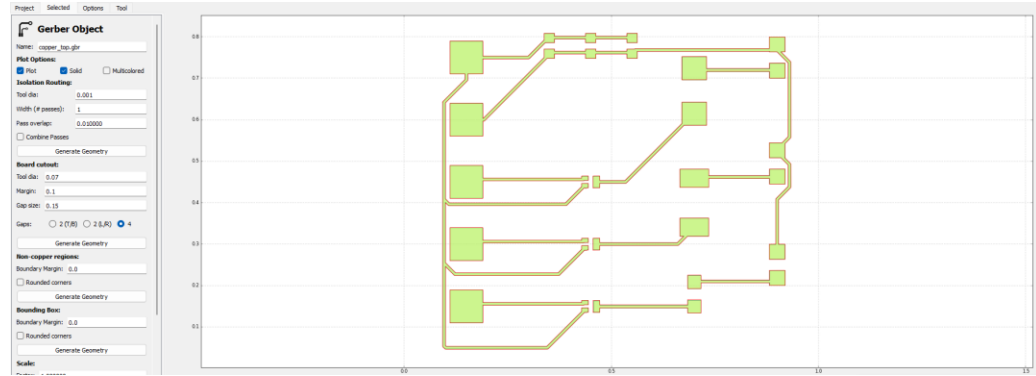
f.

g. Save the .svg file in the target directory

3.) **Launch Adobe Illustrator**
   a. Open .svg file in Adobe
   b. It may look blurry/oversized. Highlight the entirety of the drawing, and reduce the font size to 0.25 or 0.5
   c. It may also be slightly off center from the canvas, highlight the entire drawing and drag it onto the center of the canvas.
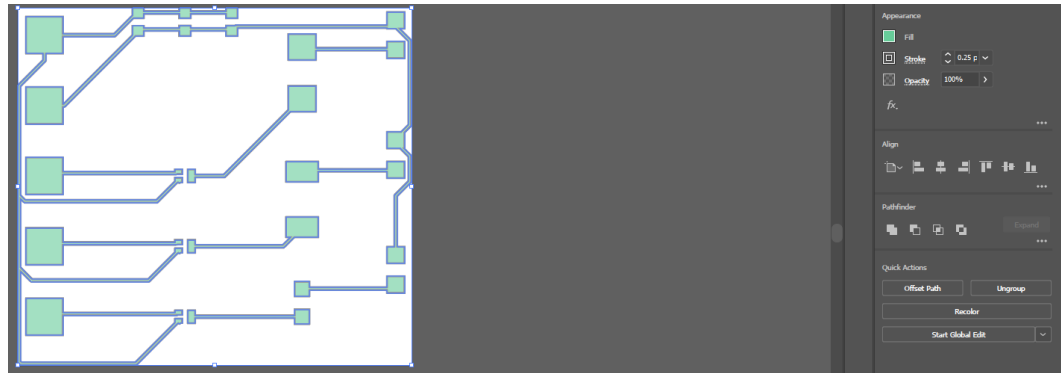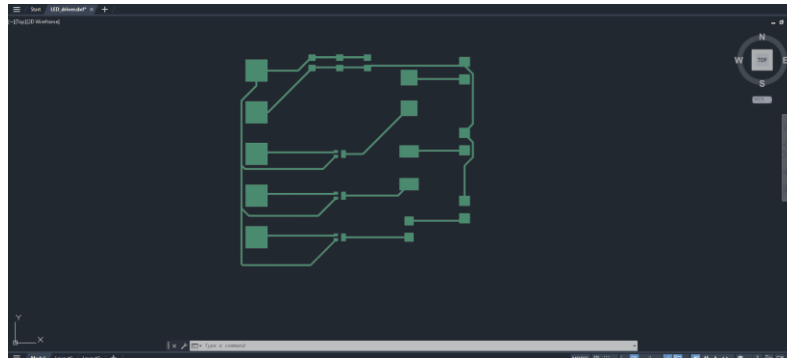

   d.
   e. Export as .dxf file
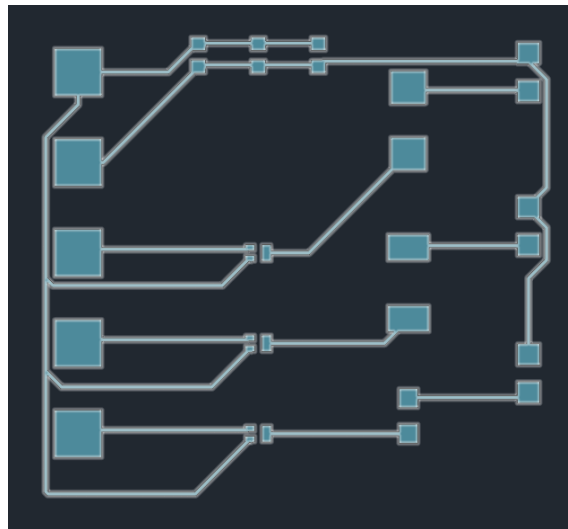   i. When exporting, take care to set the units to **millimeter** and the scale to **1**
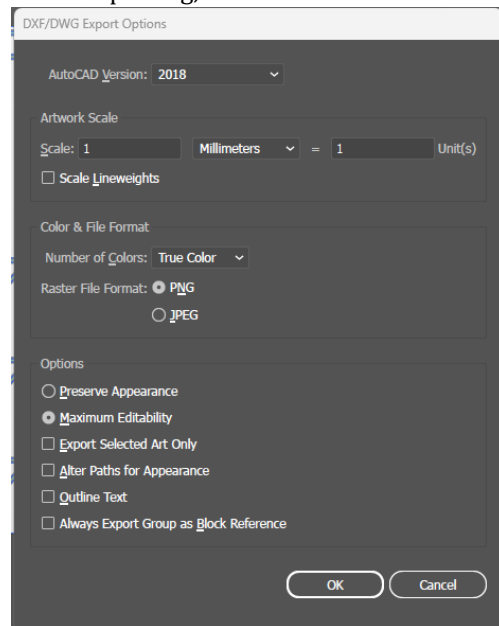
      ii.

4.) **Open AutoCAD**

a. Open the .dxf file

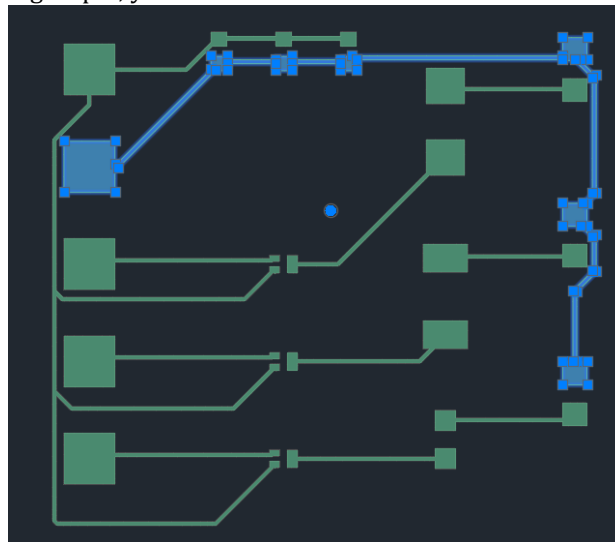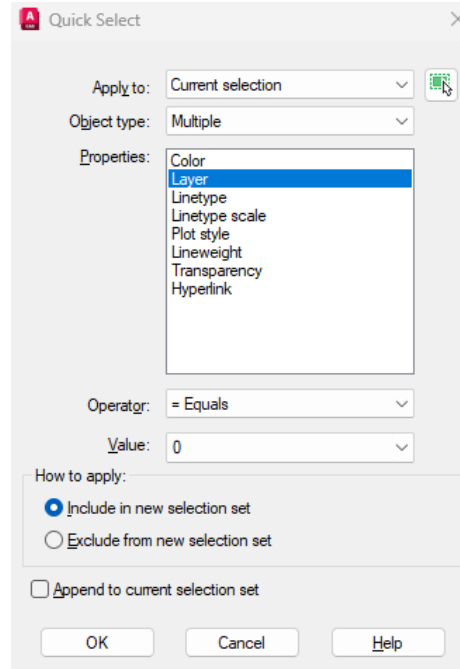   i.

b. Select the entire circuit, and type **X,** and you should see the word **EXPLODE** pop up. Click enter


    i.
    ii. Repeat this step once more

c. Upon doing step B, you should be able to select individual circuit traces as shown below


    i.

d. Highlight the entire circuit, type **QS** and the word **Quick Select** should pop up. Click enter, the following window will open up:

  i.
  ii. Set **Property** to **Layer**
 e. Save the resulting file as a .dxf file, copy this resulting file to a thumbdrive/flashdrive

5.) **Launch EZCAD** on the desktop associated with the laser printer, plug in the thumbdrive and import the .dxf file

 a. **File -> Import Vector File -> File_Path.dxf**
 b. Draw a Rectangle around the entire circuit by selecting **Draw Rectangle** (yellow square icon on the left vertical bar of the application window).
 c. Select the circuit and rectangle elements in the **Object List** panel, and then click **Hatch** (the top horizontal bar of the application window, icon looks like an **H**)
 d. Set **Hatch Type** to the concentric square mode, as shown below, and the **Line Space** to 0.015, or however small needed to get a good output
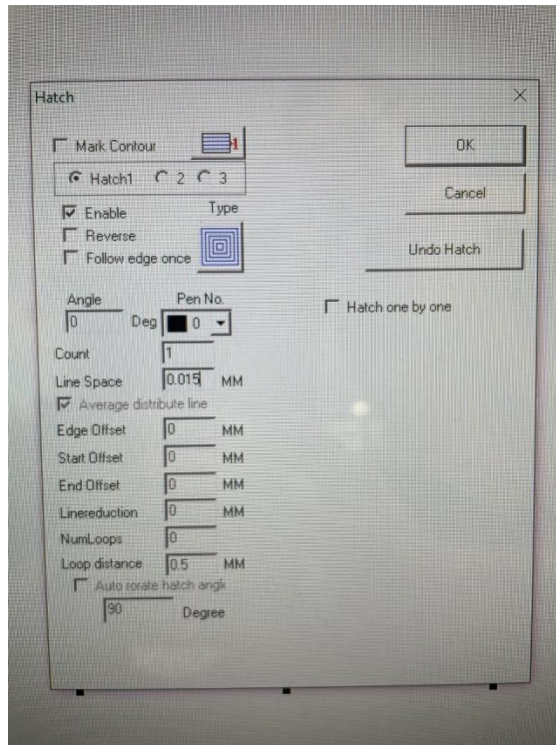
e.
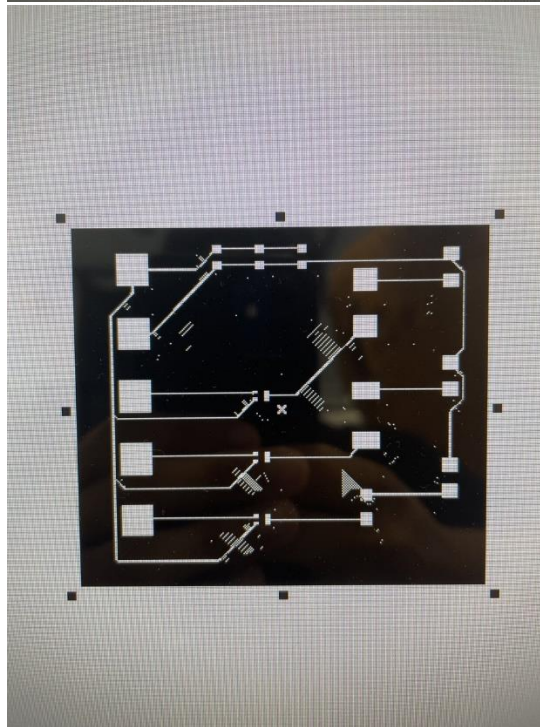
f.
   i.   Good output ^
   ii. Random 'shading' is normal

6.) **Laser Cutting Process**
   a. Take sheet of Copper-FR4, align under the laser.
      i.   Spray the copper sheet with a thin film of soldermask, dry with pressurized air carefully
   b. Make sure the 'dot' and 'line' of the laser align by adjusting the height of the laser.

    c. Select **'Red'** or '**Light'** on the bottom of the screen to highlight the area to be cut. Align the copper board appropriately. If you cannot see outline (UV Laser has this issue), place a white tissue above the copper plate. You should now see the outline.

    d. Test the settings of the laser

        i. For the UV Laser, I used:

            1. **Loop Count**: 1

            2. **Speed:** 1200

            3. **Current**: 0.08

            4. **Everything else**: default

        ii. For IR Laser, I Used:

            1. Power: 30%

            2. Loop Count: 1

        iii. Cut

7.) **Acid Etching**

    a. Wear PPE

    b. Place acid etchant into a rectangular tray, enough to submerge pcb

    c. Place pcb within acid using *plastic tweezers*

    d. Start the rocker, leave within for 16 minutes at speed 40

8.) **Acetone Wash**

    a. After acid etching, take pcb out of acid bath using *plastic tweezers*

    b. In a separate, empty, container, use acetone the clean the PCB (hold the pcb with metal tweezers now)

    c. Do so until all solder mask has washed off

    d. Once done, dispose of all chemicals in chemical waste bin, and clean pcb with water

    e. DONE!