13. Do NOT route any nodes that are assigned to GND or GNDPWR. This will be addressed later on. If you zoom in on a node, it should have white text which shows where the node connects to.


Do NOT route these

14. If you need some of your routes to cross over each other, you can make use of a via. A via is essentially just a hole that connects a route from the front of the board to the back. While placing a route, press the 'V' key and then place your route through a via. Press 'V' again and place a route to return to the other side of the board if necessary. This can make tracks a lot simpler.
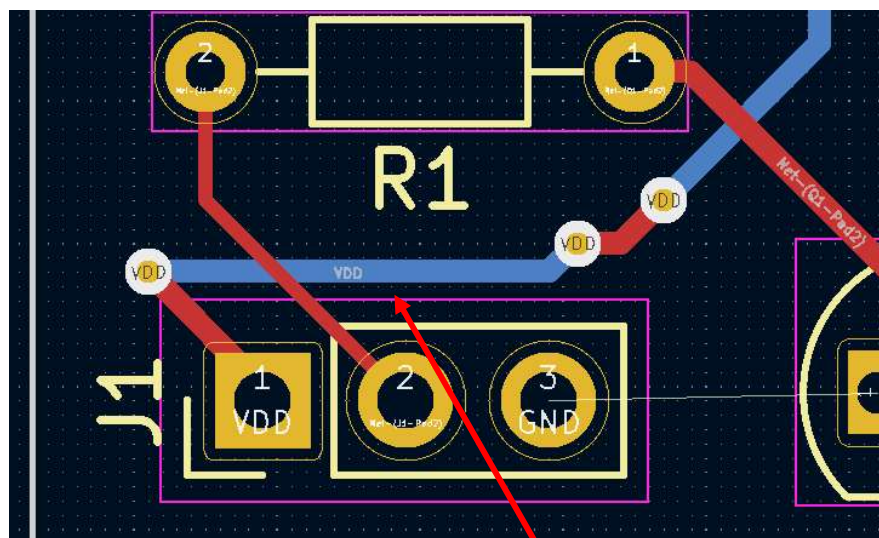

Blue routes indicate routes on the back, while red indicates routes on the front

15. Once all tracks have been routed (except for those going to GND), we can now start out polygon pour. A polygon pour is basically a layer of copper on the front and back of the board that we can use to assign to ground or GND. This is why we didn't route those nodes earlier, as the polygon pour will automatically connect to those nodes.

   To begin the pour, select the "Add a filled zone" button on the right-hand toolbar.
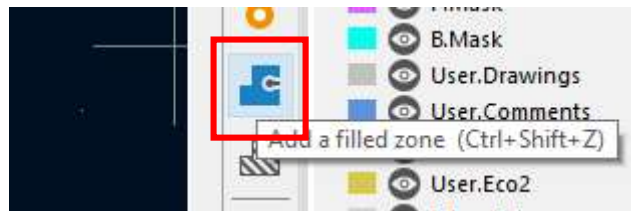
16. Click on one of the corners of the outline of your board. A properties page for the copper layer should pop up. Select GND or GNDPWR as the net for the layer and click OK. (Make sure only F.Cu OR B.Cu is selected, both layers need to be separate)
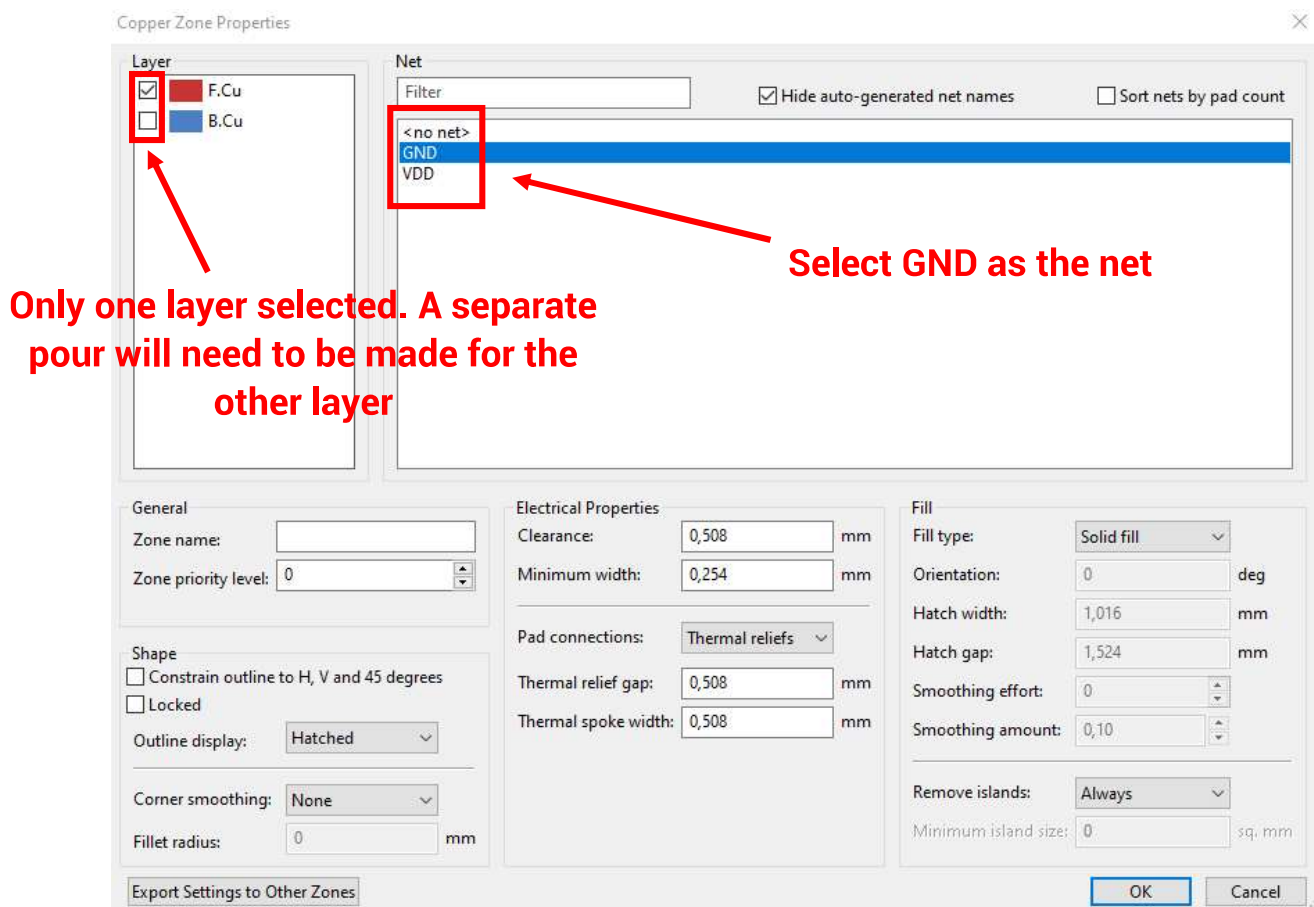
17. After GND is selected as the net, trace the outline of the board until it is complete. To view the poured zone, click on the "Show filled areas of zones" button on the left-hand toolbar. Then press the 'B' key to refresh the filled zone. (Notice how the pour does not intersect with any of the routes you placed. If you change routes later, but do not want to completely redo the polygon pour, simply press 'B' and it should correct itself!)

GND nodes auto-connect to the pour
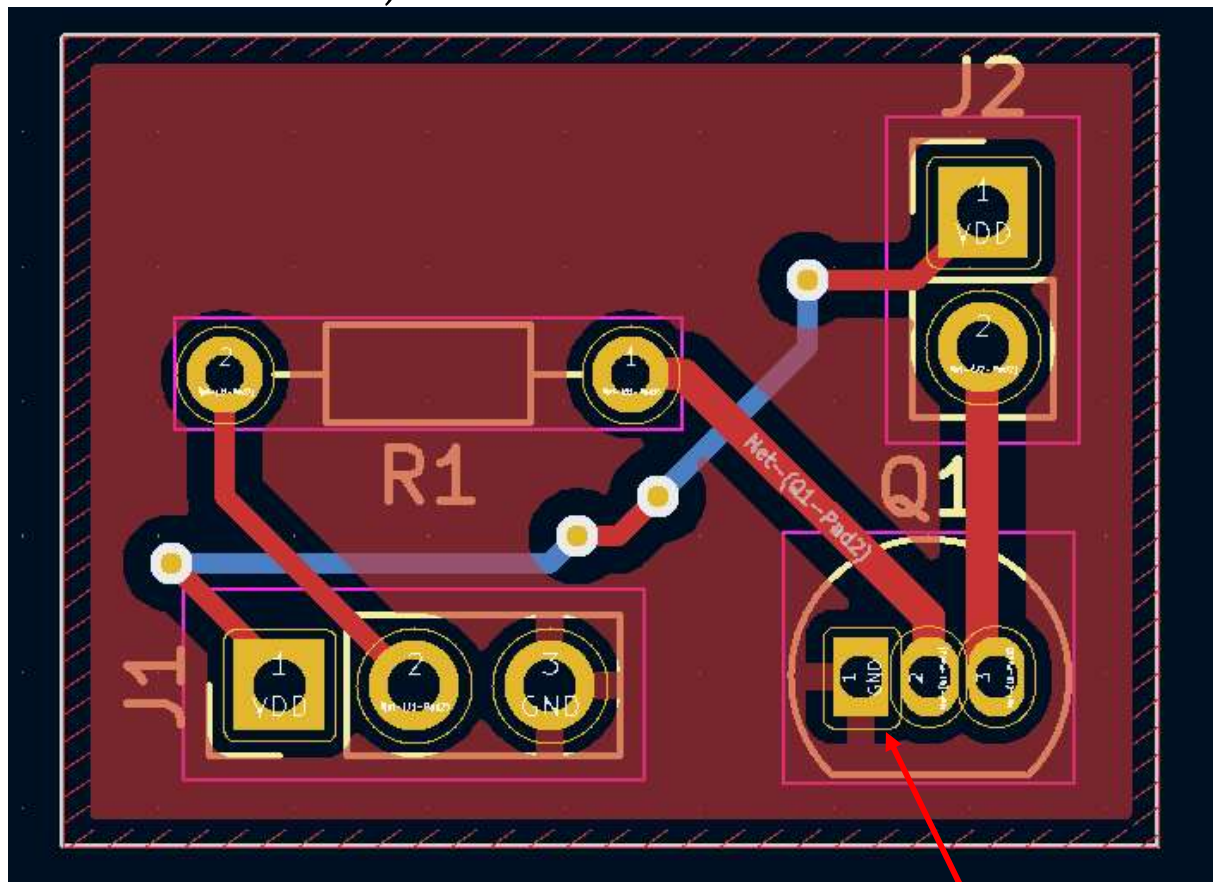
18. Do the same polygon pour for the B.Cu layer.
19. The PCB should finally be complete! You can even view it in the 3D Viewer under the 'View' tab on the top toolbar.
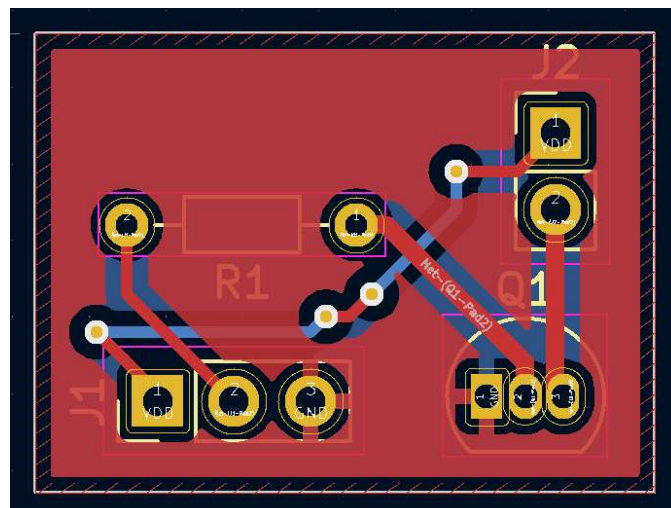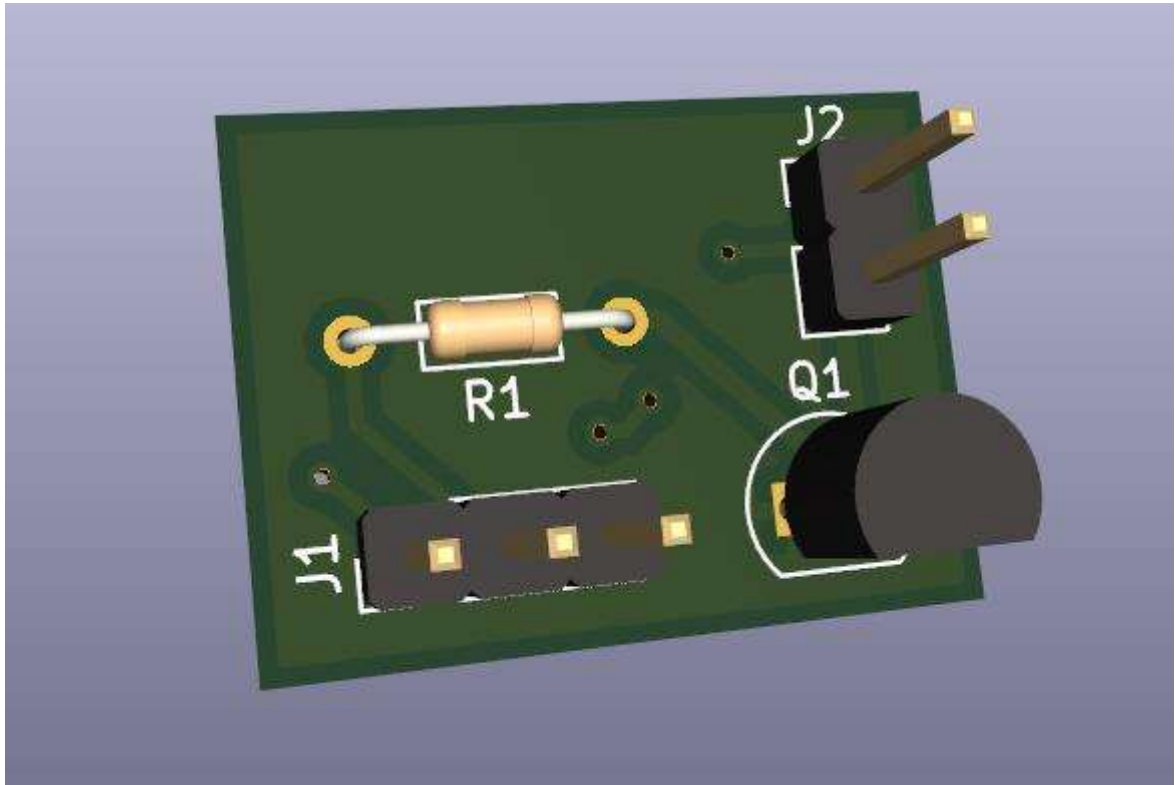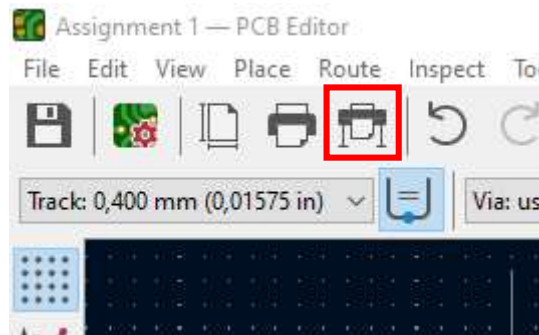
20. Unfortunately, manufacturers will not be able to print PCB's from KiCAD files. In order to convert it into a file format suitable for manufacturers, we need to generate gerber and drill files…

# Part 3: Gerbers and Drill Files

1. To generate our gerbers and drill files, select the "Plot" button on the top toolbar.


2. The assignment brief should specify which layers to generate gerber files for, but generally, these layers are:
   - F.Cu
   - B.Cu
   - F.Silkscreen
   - B.Silkscreen
   - F.Mask
   - B.Mask
   - Edge.Cuts

   Make sure ONLY these layers are selected for the gerber files.

Make sure ONLY these 7 layers are selected

Make sure this is NOT ticked
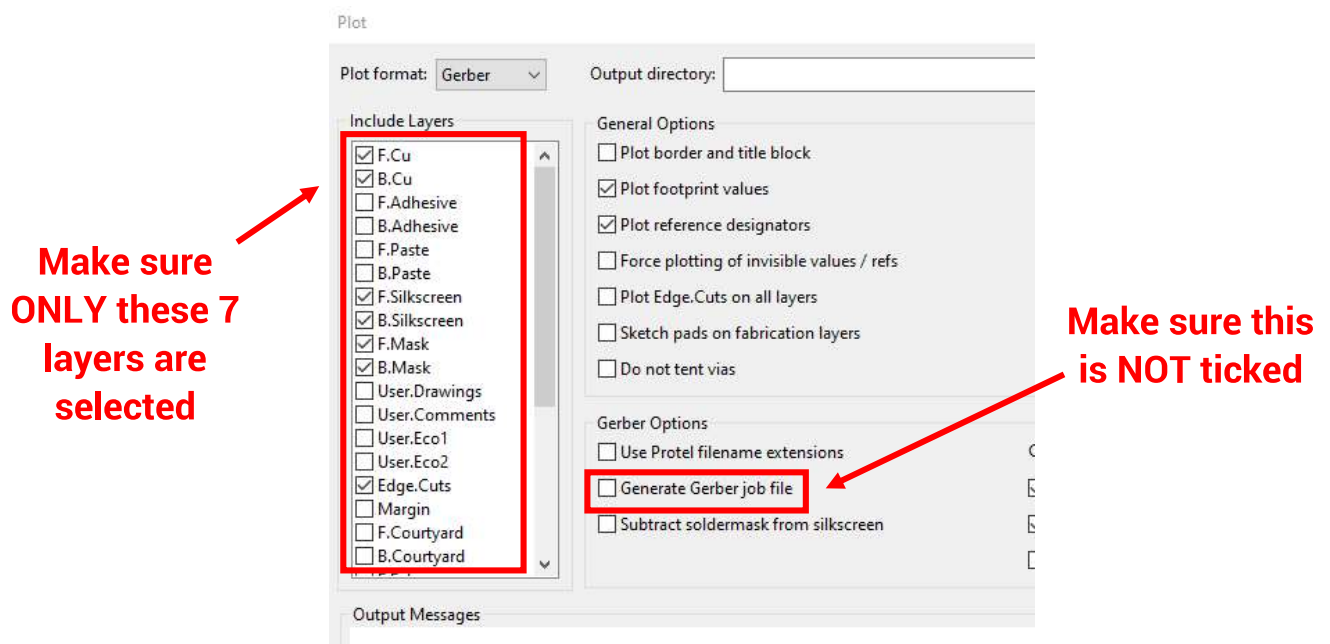
3. We need to assign a folder for all our gerber files to be saved in. Click on the Folder button next to the Output directory. Create a new folder called 'gerbers' within the existing project file. Select this folder.
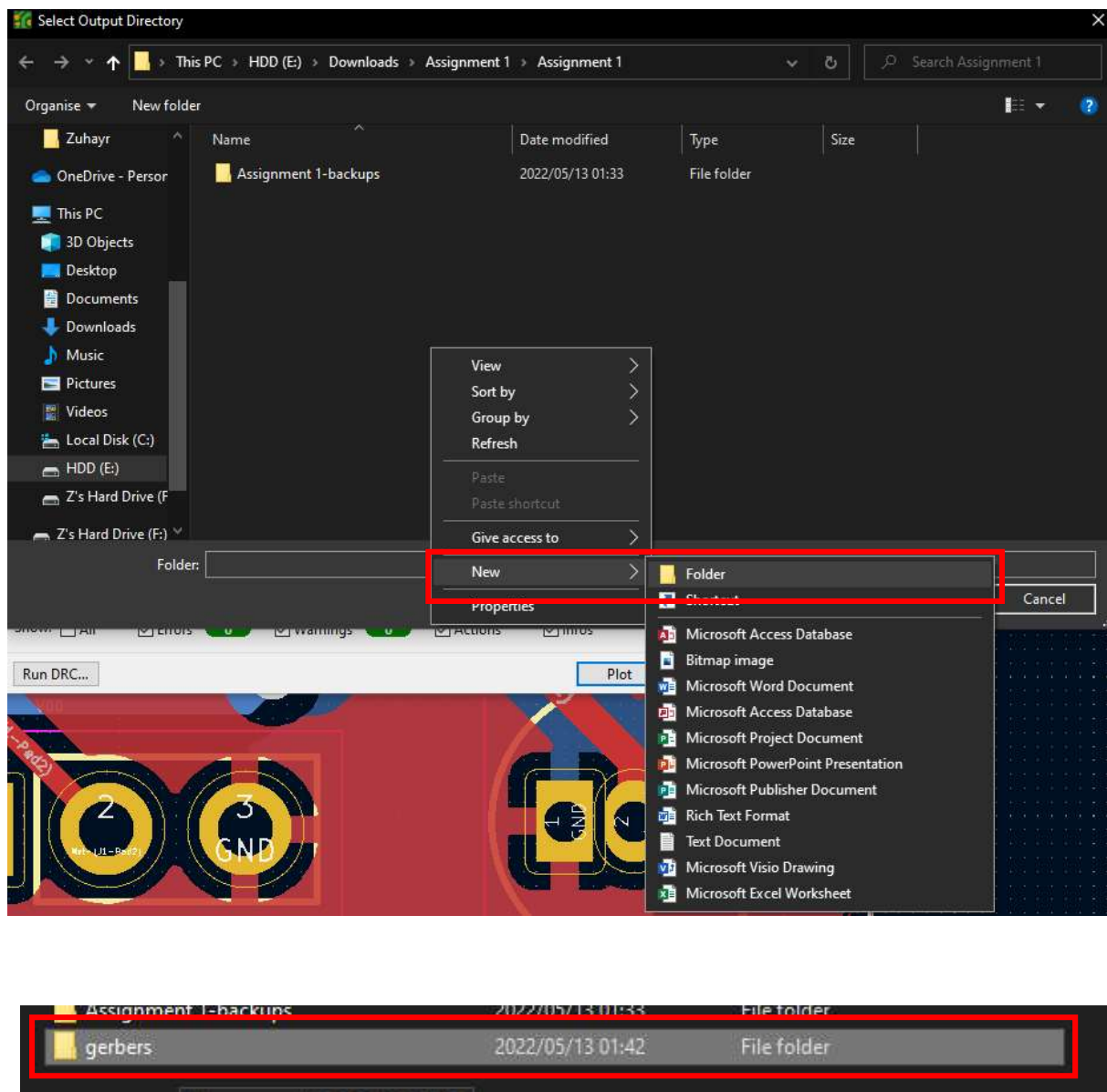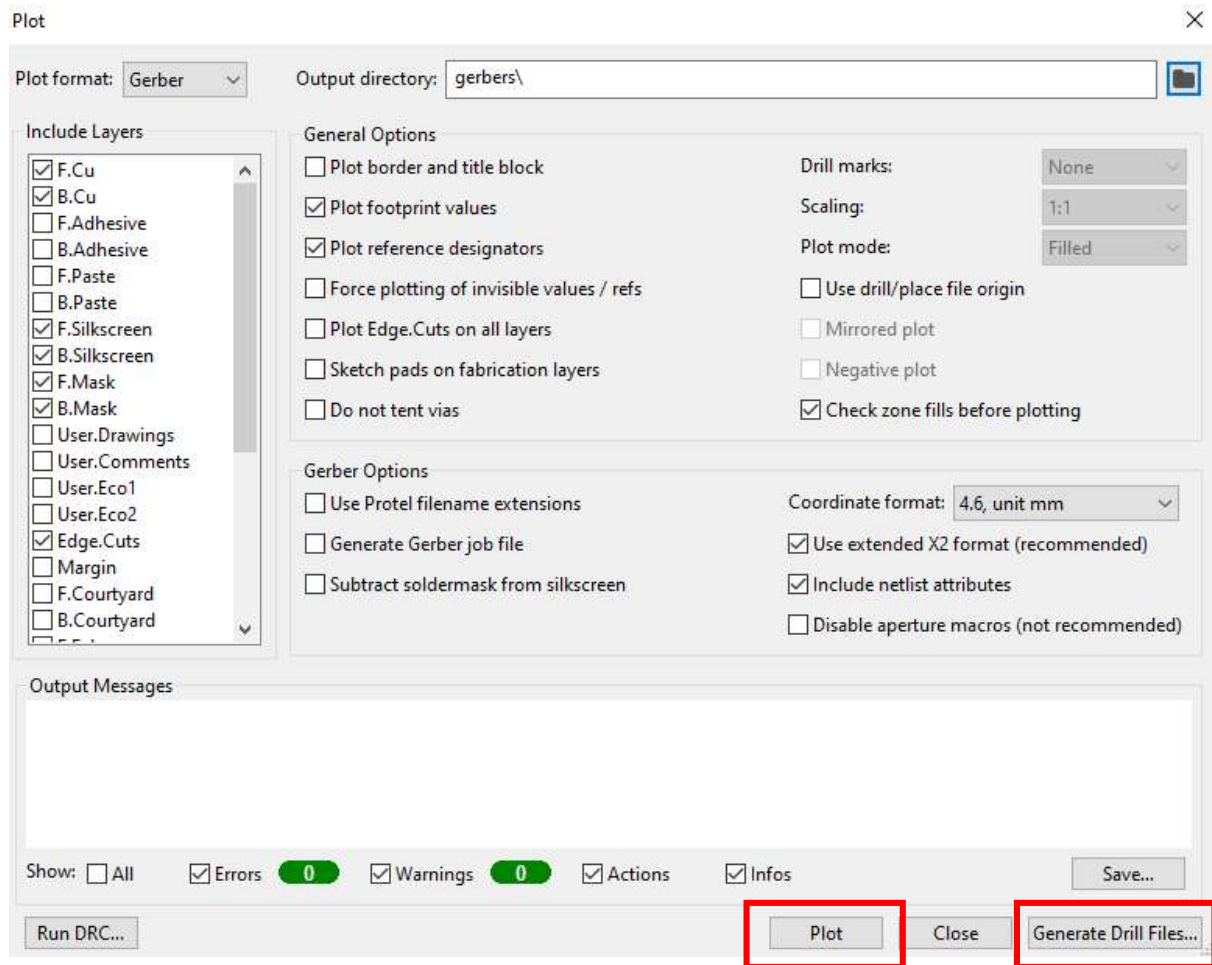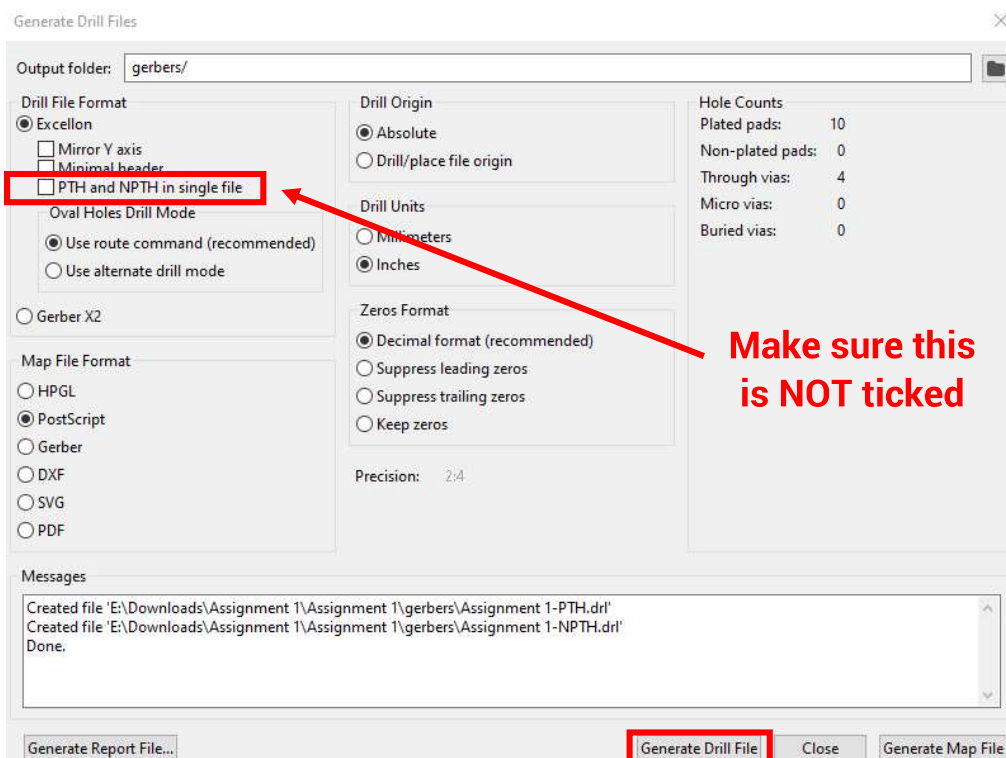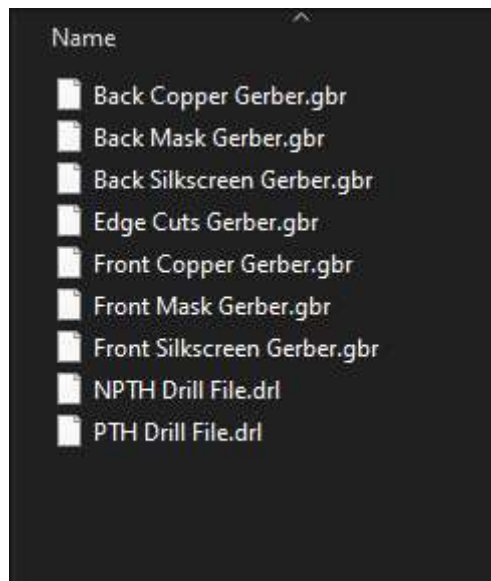
4. Click on the "Plot" button. The gerber files should be generated.

5. To get our drill files, click on "Generate Drill Files…". Make sure the "PTH and NPTH…" box is unticked, and generate your drill files.

6. You should now have 9 files in the "gerbers" folder: 7 gerber files and 2 drill files. Make sure to rename these files clearly and appropriately.

7. That should be all we need to do for the assignment! (If you want to verify that your gerbers are correct, you can open them in the Gerber Viewer, but it is not necessary)

This is all I know for the assignment, but they could always throw a curveball or two in there. I will send the links for 2 very useful videos on our EEE group.

Really hope this helps. Good luck everyone!