Lab 4

Mixer & Mic Preamp: Part 2 Layout and Gerber and NC Drill Files

Brodric Young

Spencer Wyman

10/18/2024

ECEN 299

Objectives

Our objectives for this lab were to complete our schematic of the microphone preamp and mixer of our dual-channel stereo system, design the PCB, and verify we can upload it to JLCPCB ensuring it's ready to be fabricated.

Procedure

Equipment and supplies

- Altium Designer

Procedure

This lab was very similar to lab three, we basically did the same thing but for different schematics. We finished our schematic by adjusting the resistor footprints according to the resistors we have in our lab kits. Then we transferred our schematic to a PCB, defining the shape with a rectangle and setting the default via properties and constraints/clearances. After that it was up to us to place components on the PCB and route them together according to our schematic. This included to 10-header pin component and our 10 resistors which we had to route the proper way. We also created a ground plane and vias to it so that our ground pins could connect to the ground plane instead of routing wires for that. After repouring all polygon pours and making sure the designators were aligned correctly with our components and adding our names to the PCB, we created the NC Drill and Gerber files which we could upload to JLCPCB. The final product I uploaded to JLCPCB can be seen below as “Figure 1, My PCB Uploaded to JLCPCB”.

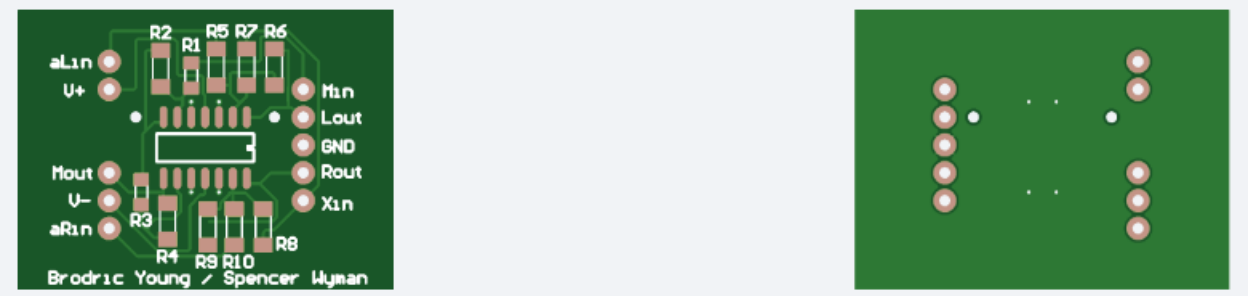

Figure 1, My PCB Uploaded to JLCPCB

Conclusion

In conclusion, this lab reinforced the process of designing and transferring a schematic to a PCB that we started doing in lab three. It focused on proper component placement and routing. This was especially difficult for me, I eventually gave up at trying to make it super pretty and ended up feeling like I had wires all over the place and bet that it could've been better somehow. But through this process and by creating a ground plane, defining constraints, and generating Gerber and NC Drill files, I gained experience with creating a PCB layout ready to be fabricated.