

Lab 3

Buffer: Create the Layout with a Ground Plane

Brodrick Young

Spencer

10/4/24

ECEN 299

Objectives

The objective of the lab was to design a printed circuit board layout and make the gerber and drill files for manufacturing using Altium Designer.

Preliminary

Before starting the design, we duplicated the Lab 1 project folder and renaming it for Lab 3 and opened it in Altium Designer validating the schematic, making sure there were no errors in the design.

Procedures

Equipment and supplies

- Computer with Altium Designer installed

Procedure

We transferred the schematic into the PCB layout by updating the PCB document. After this, we defined the board shape by placing a rectangle on the origin. Changing the grid to 100 mils or smaller made the board easier to work with, especially when placing components. We also had to make changes to the constraint manager including setting the clearance and trace width for all nets to 10 mil, and configuring vias with a 24 mil diameter and 12 mil hole size. The polygon connect style and via connections were also adjusted, with the via connection style set to direct connect for the ground plane. These adjustments made sure the design would meet manufacturing requirements. Next, we placed the components on the board, like the header pins, resistors, and the op-amp. With all the components placed, we could route the electrical connections between them using the "Interactive Routing" tool. We started on the top layer of the board and used vias to create vertical connections between the ground pins on the top layer to the bottom layer which we made to be all ground. We also added text to the layout by placing our names on the top overlay layer, making sure it wasn't overlapping any components or wires. Once the layout was finished, which can be seen below as "Figure 1, PCB layout", we made the gerber and drill files. These files have all the information needed to manufacture the board. To verify that the files were correct, we uploaded them to JLCPCB's website, which showed us a preview of how the board would look when fabricated. This can be seen below as "Figure 2, PCB uploaded to JLCPCB".

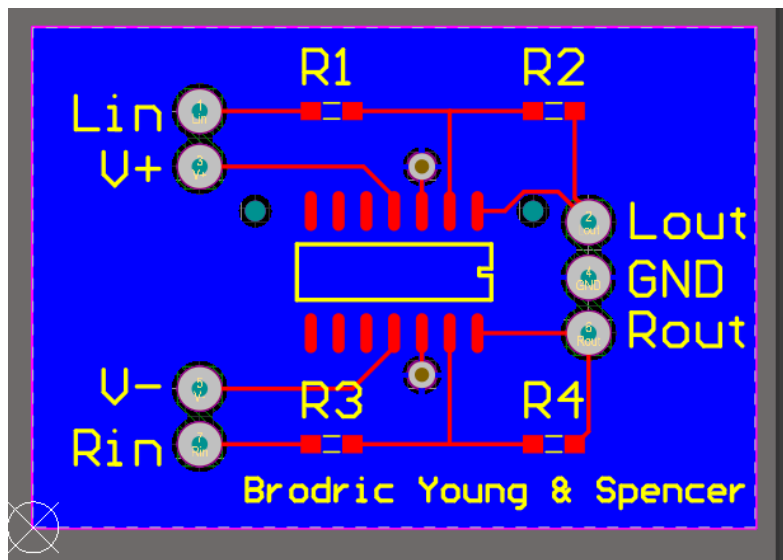


Figure 1, PCB layout

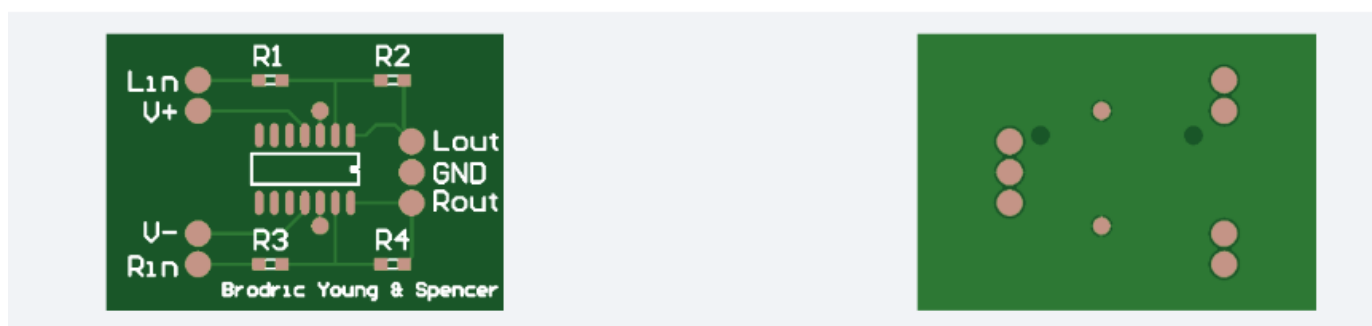


Figure 2, PCB uploaded to JLCPCB

Conclusion

Through this lab, we learned how to take a schematic and make it into a functional PCB layout. Working with Altium Designer taught us the importance of following design rules to avoid any manufacturing errors. The process of routing connections made it clear how important it is to organize and manage board space. Making the gerber and drill files and uploading them to JLCPCB showed us how PCB designs are prepared for fabrication. The lab provided a solid introduction to PCB design, and made it very clear for the need of precision in both layout and file preparation to make sure the manufacturing process would be successful.