Matthew McCarthy

Wai Yung

ENGN 3055

5/16/19

Final Project Matthew McCarthy

The final project for ENGN 3055 had to be started with a course of action. In order to be able to complete the project on time and correctly, each requirement of the project must have been done in a chronological order. These steps were: First, building a MultiSim schematic and running a transient simulation, then building the bench measurement setup and collecting an oscilloscope capture, next designing a schematic and PCB of the circuit design from MultiSim and the bench measurement, the next step was creating a two-page problem statement, design method and calculation of the project, and lastly, uploading the two-page paper, MultiSim schematic, MultiSim transient simulation, bench measurement, and oscilloscope capture to Ulearn and uploading the the Eagle Schematic and the PCB to Github. The two-page paper was also uploaded to Github, as well as the bench measurement setup along with the oscilloscope capture.

When beginning the first step in the course of action for the project, the MultiSim schematic, I needed to ensure I had the proper VCC, resistors, capacitor and push button in order for the output results to be correct. My design included a 3.3V VCC input connecting to a 10kOhm resistor, which was then connected to a 1kOhm resistor, which was in parallel to a push button and a 1uF capacitor, and then lastly connected to an oscilloscope. The 10kOhm resistor was also connected to the push button 1uF capacitor, explaining why the 1kOhm resistor was in parallel. The last step in the MultiSim setup was to run a transient simulation waveform. After analyzing the waveform and screenshotting it along with the circuit, it was time to move to the next step in the project.

The bench measurement setup was next on the to-do list, and so was the oscilloscope capture of the setup. Using what I learned early on in the Circuit Theory Labs, I designed my circuit almost exactly like it was in MultiSim, on my breadboard. The only problem was that there were no 1kOhm or 10kOhm resistors in the closet. Therefore, I used a 1.1kOhm resistor and an 11kOhm resistor instead. The oscilloscope capture was a little difficult at first trying to get any sort of result. I later realized I made the simple mistake of connecting the probes incorrectly, so I made the change. It was now time to move on to the second-to last step in the process.

The Eagle schematic and PCB was next, and after using Eagle for Design 7, I felt more comfortable going into the software because it was only the second time I have ever used it. Unlike Design 7, it only took me about 15 minutes to build the schematic and then build the PCB using the schematic. I had some issues connected each component in my PCB board but after some troubleshooting I was able connect everything without any overlaps or other issues.

The last step in my process was to create a two-page paper and upload everything that I documented. The two-page paper was to discuss my process in building and completed the project. In order to upload everything correctly, I had to ensure they are all in the correct files. Some were in JPEG or PNG files and they needed to be PDF and BMP files. After submitting everything my project was complete.

The Final Project taught me more about circuit design as it reinforced my understanding of MultiSim, breadboard wiring, oscilloscope probing, Eagle and Github. In conclusion, this project was very difficult, but definitely taught me more about circuit design.