ECEN3730 Board 1

Christopher Sponza

September 26, 2024

1 Project Overview

1.0.1 Introduction

The purpose of this board was to understand and utilize best practices when designing PCBs. A simple astable oscillator with a 555 timer was chosen to demonstrate these practices. The following BON sketch was used to create the schematic and subsequent PCB for the board:

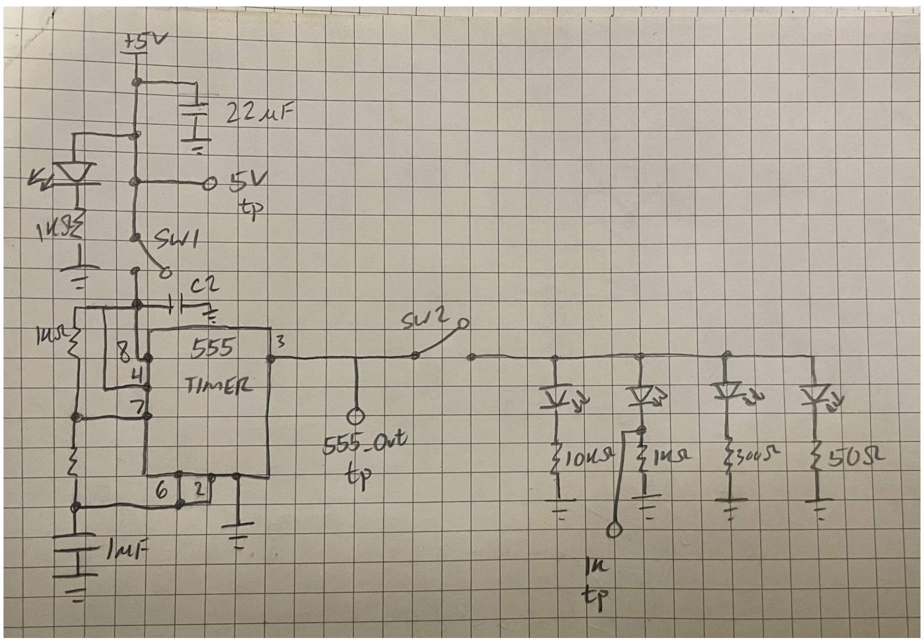

Figure 1: Back-Of-Napkin Sketch of the Circuit

The following were the specifications that were required to be met in order for the board to "work":

1.0.2 Engineering Specifications

- Powered by 5VDC
- Have switches to isolate 5V from the 555 timer and the 555 timer output from the LEDs
- Be able to measure the 5V power rail, the 555 timer output, and voltage across one LED.
- Achieve a 500Hz square wave from the 555 timer with a 70% duty cycle
- Use an LED to indicate 5V power and multiple LEDs connected in series with different resistors

- The current through one of the LEDs can be measured using a test point.
- Silkscreen text provides useful labels for components

1.0.3 Final Product

Using the BON sketch from Figure 1 and the engineering specifications, the following schematic and PCB were designed in Altium:

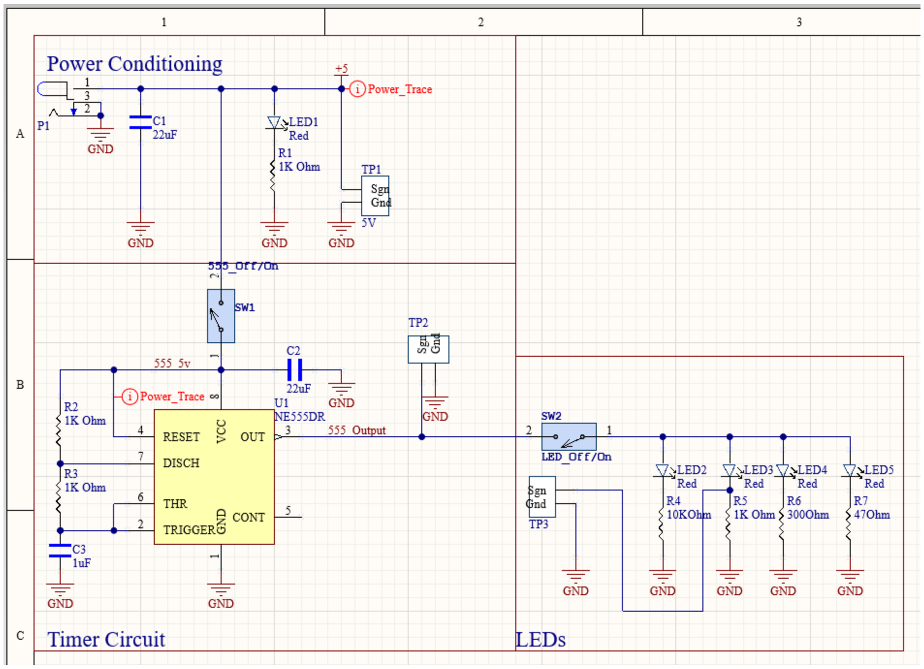

Figure 2: Schematic Designed in Altium

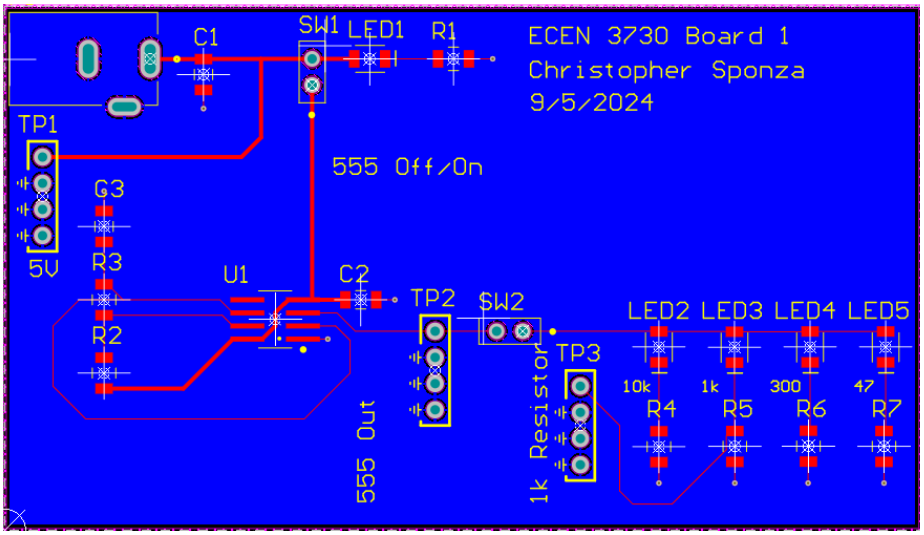

Figure 3: PCB Created in Altium

1.1 Bill of Materials

Part	Value	Quantity
NE555DR	-	1
Resistor	$1K\Omega$	4
-	$10K\Omega$	1
-	300Ω	1
-	47Ω	1
Capacitor	$1\mu F$	1
-	$22\mu F$	2
LED (Red)	-	5
10x Probe TP	-	3
Power Jack (Barrel)	-	1
2-Pin Header (Switch)	-	2

2 Project Analysis and Measurement

2.1 What Worked

The expectations I had for the board were the same as the engineering specifications. What I particularly liked were the silkscreen labels which made it easier to test and solder the board. The board came back from the fabrication vendor as expected:

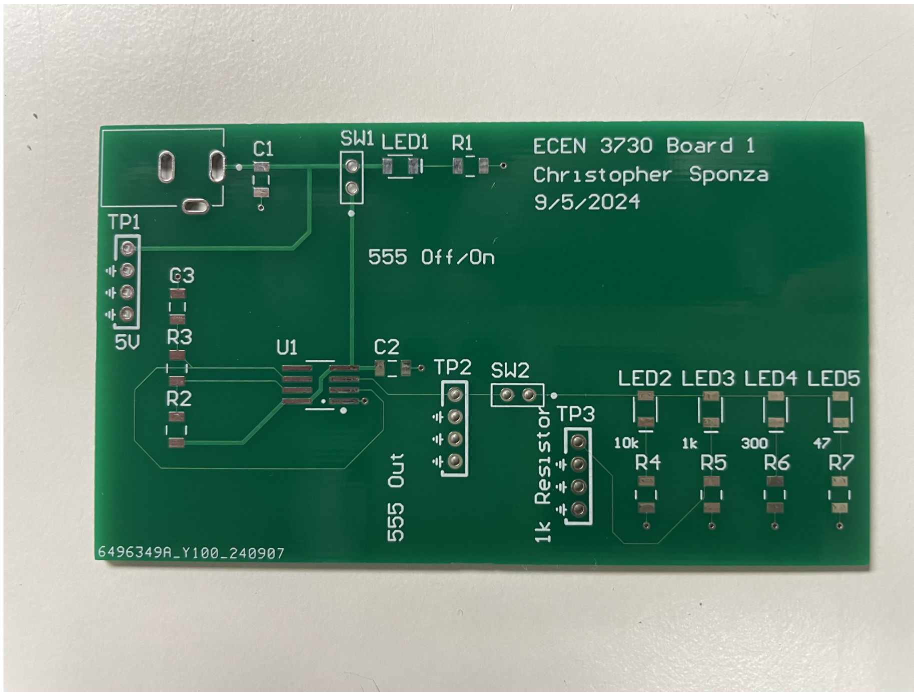

Figure 4: PCB Received from the Vendor

and the soldering of the surface mounted parts went relatively smoothly:

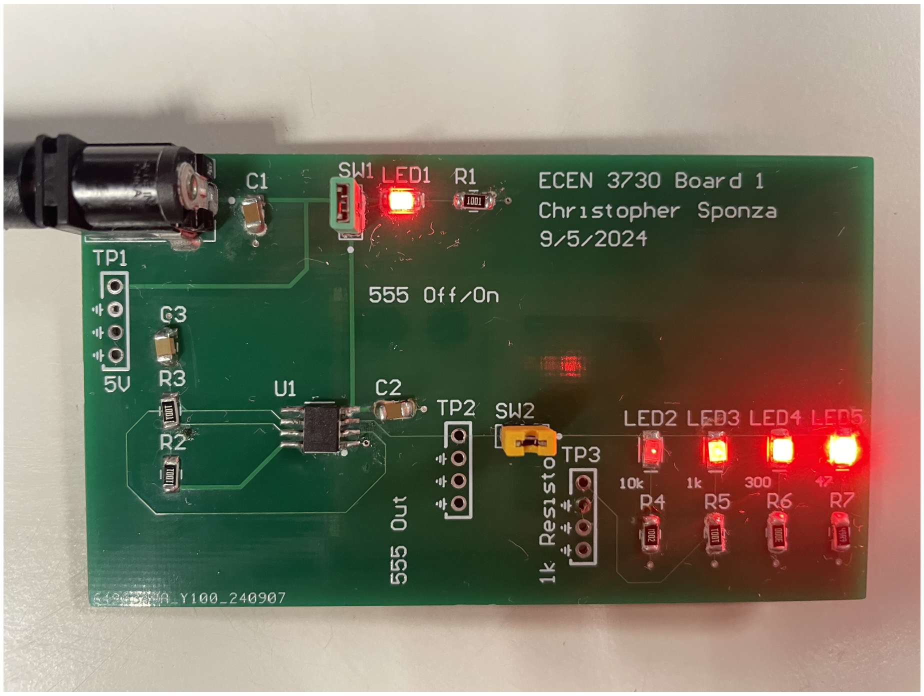

Figure 5: Constructed PCB Powered with 5V

The only thing that delayed construction of the board was the capacitors. I was unsure which capacitors were which values but this was quickly rectified with a multi-meter.

Based on the engineering specifications the board "worked". The 5V rail, output of the 555 timer, and voltage across a 1k resistor between an LED and ground was able to be measured. This enabled the switching noise on both the 5V rail and output of the 555 timer to be quantified and current through an LED to be measured.

2.2 Measurements

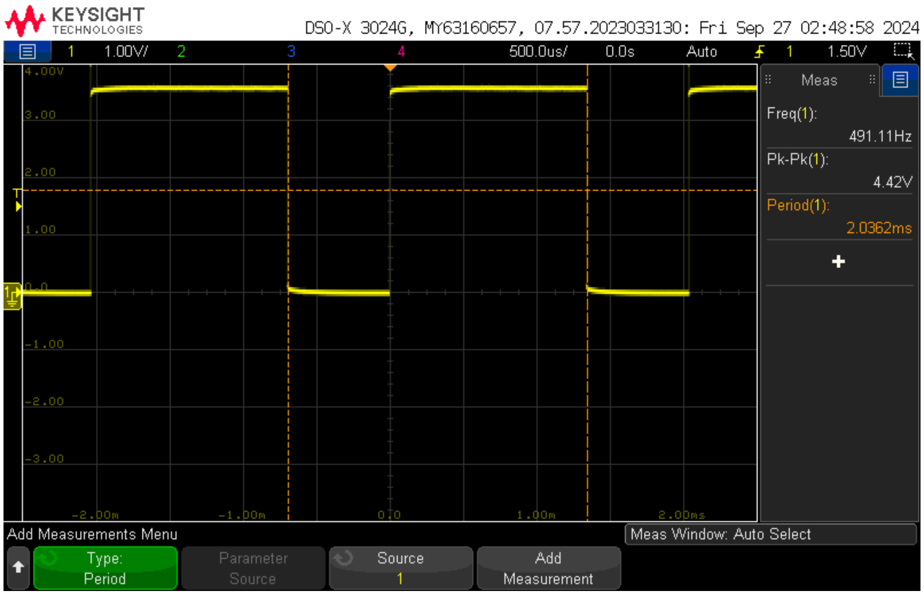

Figure 6: Output of the 555 Timer

Figure 6 shows the output of the 555 timer which resulted in a period of roughly 2.05 seconds corresponding to a frequency of roughly 490Hz. The signal is high for roughly 1.5ms corresponding to a duty cycle of roughly 73%. These two metrics are, for the purposes of the exercise, within close enough proximity of the desired specifications for the board to be considered "working".

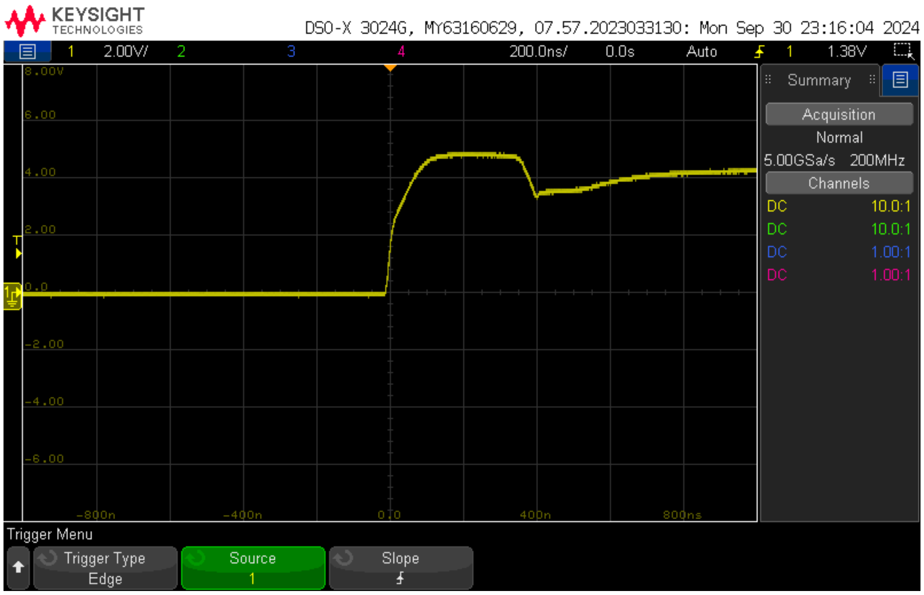

Figure 7: Rise Time of the 555 Timer

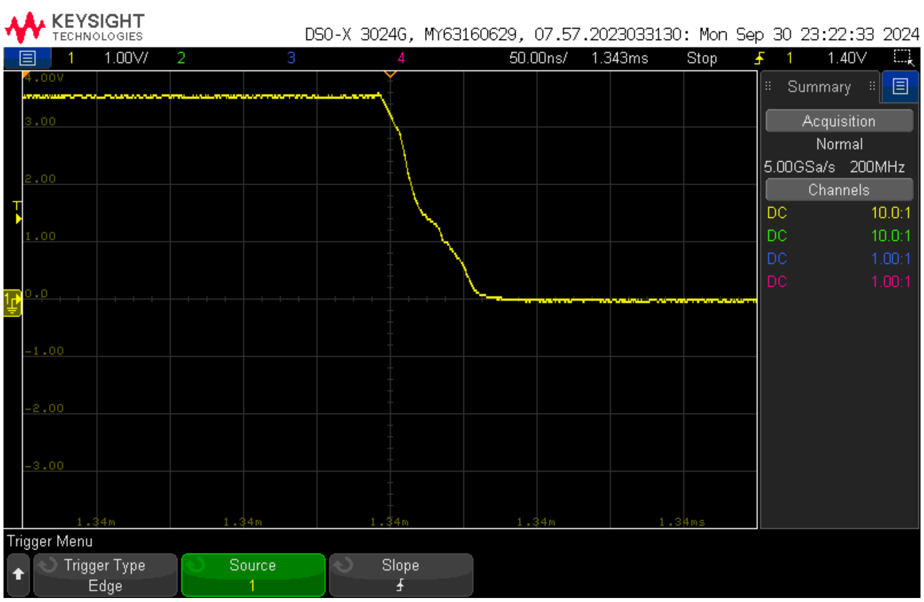

Figure 8: Fall time of the 555 Timer

Figures 7 and 8 show the rise ($\approx$ 40nS) and fall ($\approx$ 50nS) times of the 555 timer. The decoupling capacitor does a good job of mitigating the noise of the signal switching and the lack of noise on the falling edge can be attributed to the longer switching time. However, there is still a roughly 0.5V signal introduced on the rising edge.

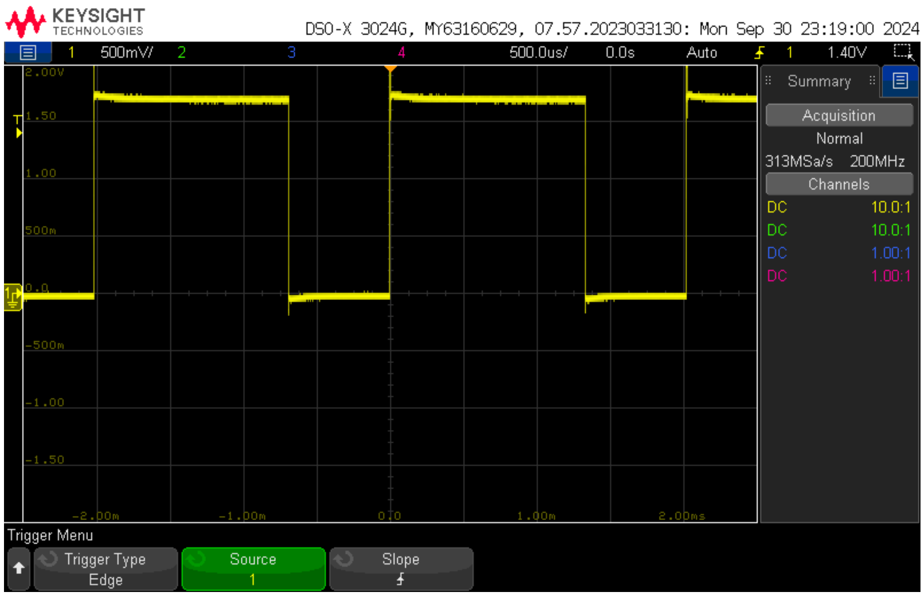

Figure 9: Signal Across the LED in Series with $1k\Omega$ Resistor

Figure 9 shows the signal output across the LED in series with a 1kΩ. I measured 1.75V across a 1kΩ resistor corresponds to a current of 1.75mA.

3 Conclusion

Overall I was happy with the design of the board. During its construction I learned best practices such as:

- Use a continuous ground plane and ground vias as signal return paths
- Use decoupling capacitors whenever practicable and place them close to components
- Test points, isolation switches, and indication LEDs are very useful in board bring-up and debug
- Minimize or avoid routing traces under one another by changing the orientation of components if possible

Perhaps the only thing I would do differently in the future is add small labels on the silkscreen under resistors and capacitors to indicate their values for more efficient construction. I would consider this to be the only "soft error" I made with this board and I would not say there were any "hard errors".