Daniel Mendez: Board 1 Report

Project Overview

Plan of Record

This project was the first foray into creating a schematic and a layout for a PCB. For this purpose, we utilized a simple circuit of a 555 Timer configured as an astable vibrator. Using this, we tested some basic design and measurement practices once the board was printed and brought up. The baseline features can be summarized below including metrics defined to specify whether they work or not.

- A 555-timer configured as an astable vibrator.
- The frequency of the 555 outputs to be 500 Hz with an error margin of +/- 200 Hz
- The duty cycle of the 555 outputs to be 50% with an error margin of +/- 18
- The 555 output would 4 red LED's with resistors of 50,300,1k and 10k.
- Indicator led's would be to show that the circuit is powered.
- A 5V supply for the entire circuit
- 1206 components to be used.
- The board would be brought up with no jumper switches connected, then the 555 is powered only, then the output LED's are connected.

A basic sketch of the initially planned schematic is shown below:

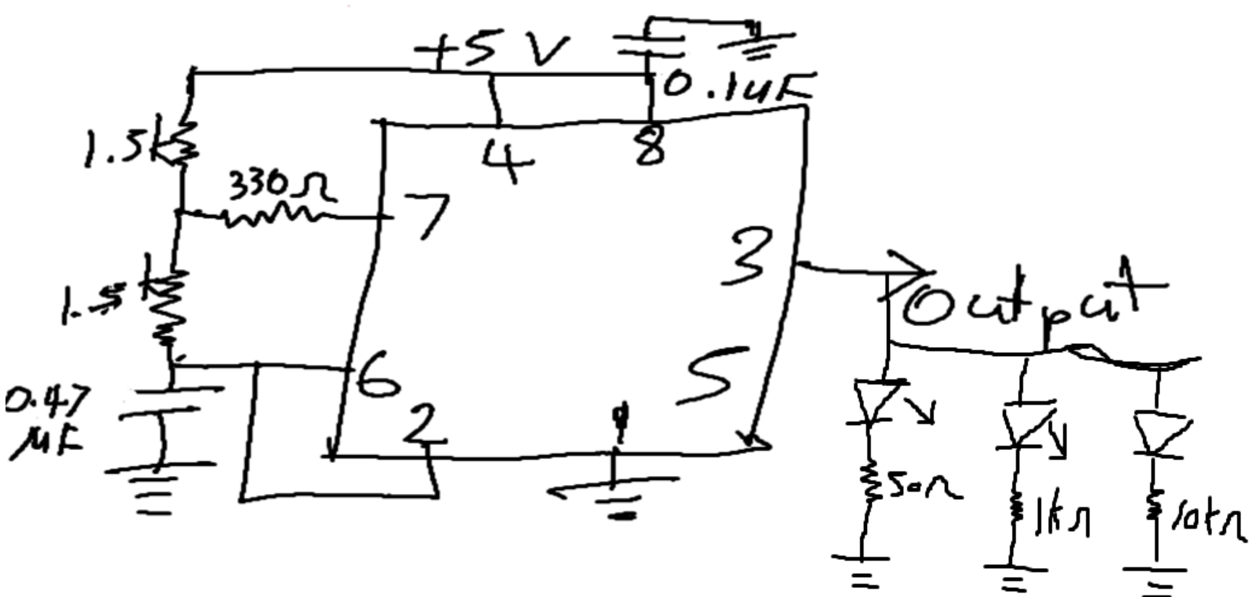


Figure 1 Basic sketch of the intended circuit.

The Altium schematic that was created based on this is shown below:

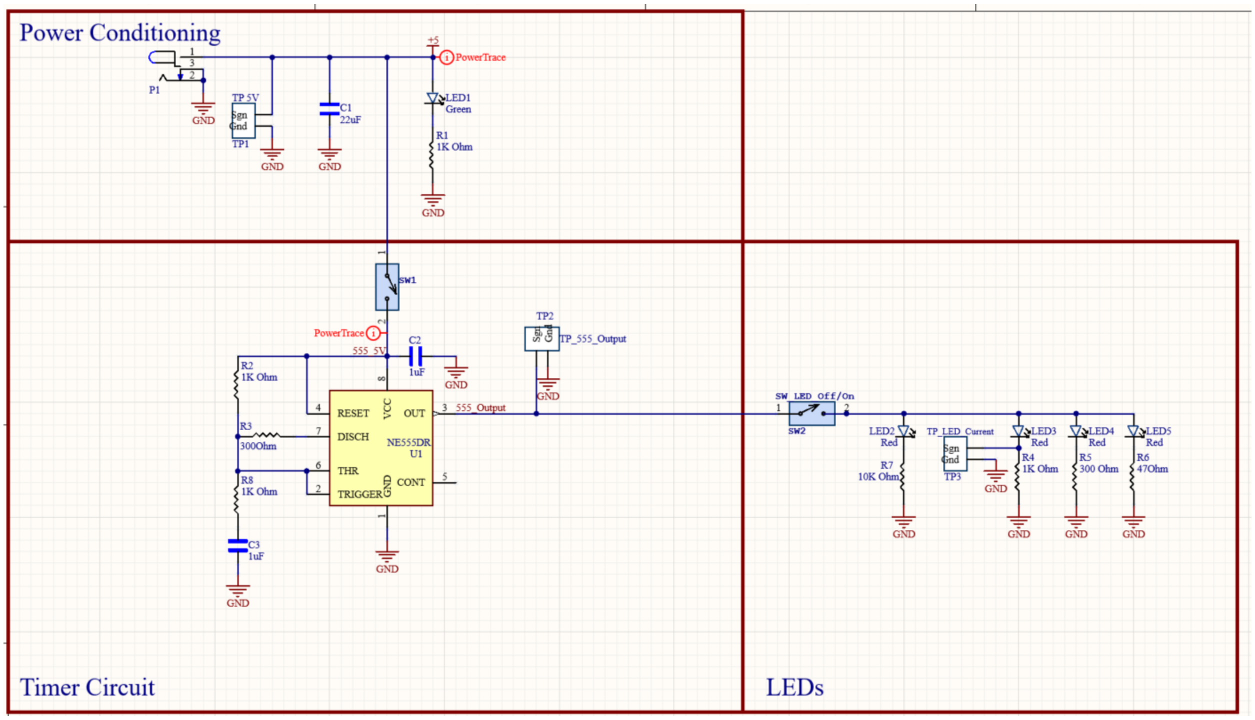


Figure 2 Board 1 Schematic

The board layout that was developed based on this is also shown below:

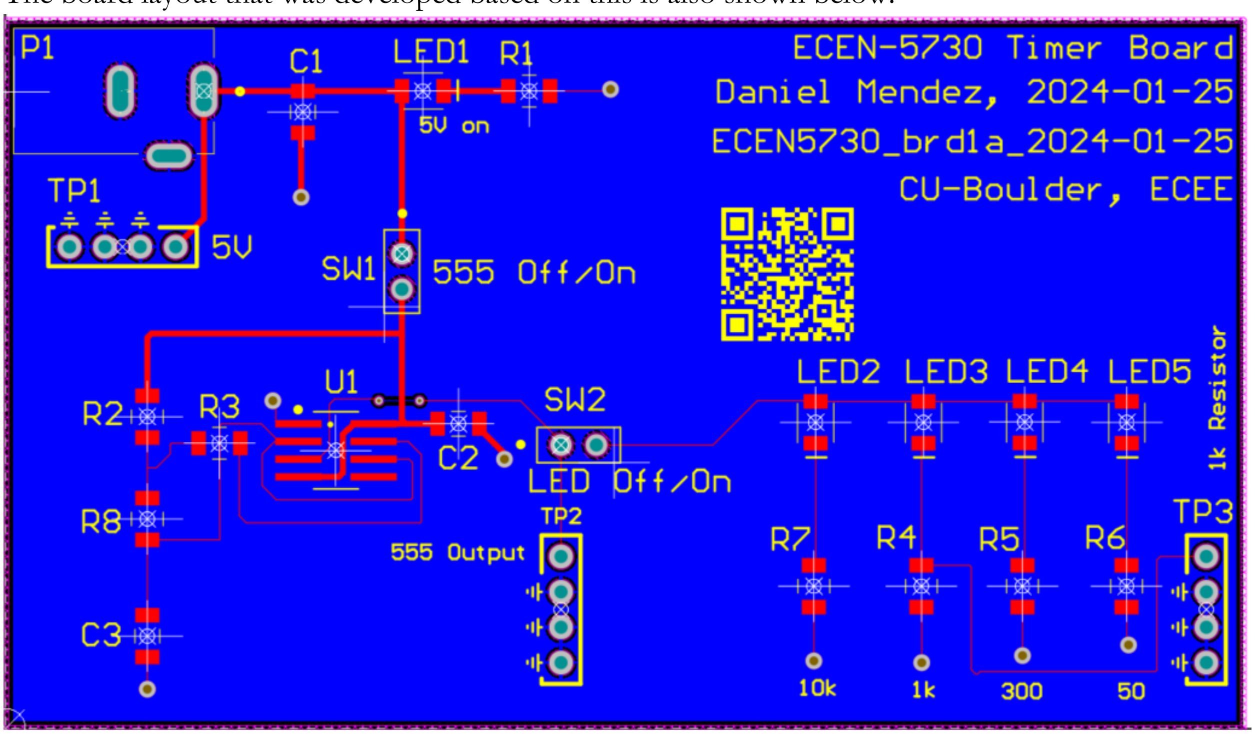


Figure 3 PCB Layout of Board 1

The printed circuit board is shown below as well as the assembled version.

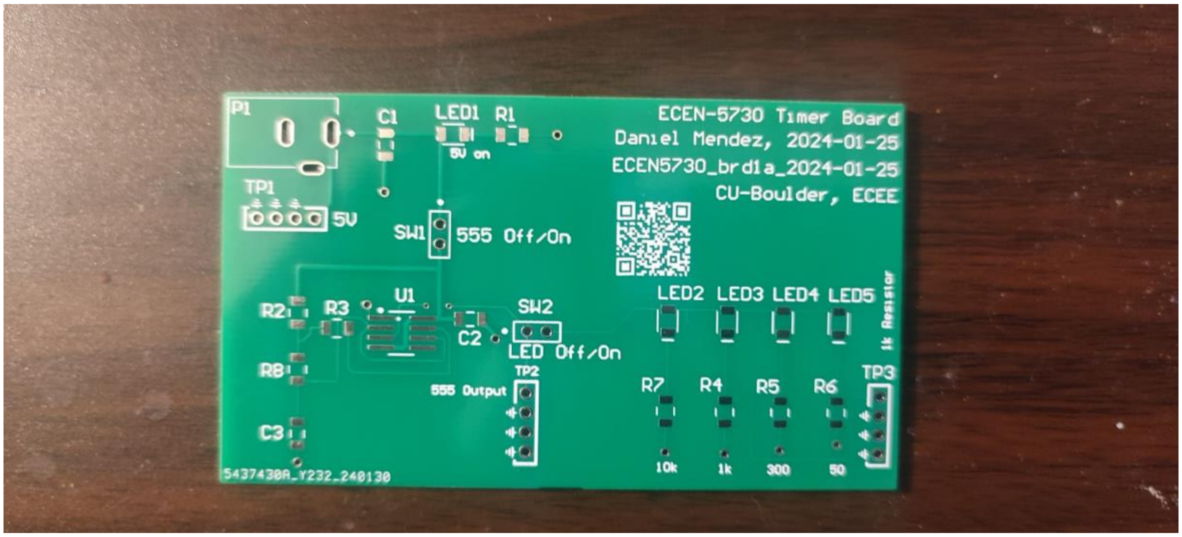


Figure 4 Printed PCB

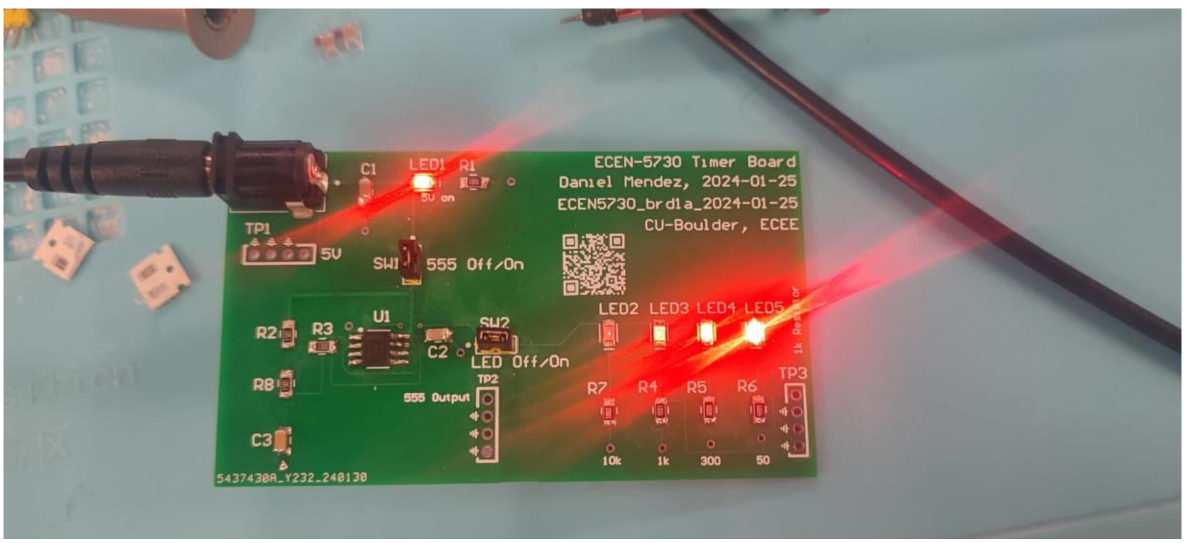


Figure 5 Assembled PCB powered on

What worked

Based on the plan of record, my definition of the circuit working included specifications for the duty cycle, the frequency as well as the indicator LED's. Clearly from the picture in figure 4 above, the indicator LED worked correctly as well as the 555 output LED's. as expected, the brightest LED was the one with the 50-ohm resistor and the weakest was the one with the 10k ohm resistor . The input voltage was measured with an oscilloscope to be exactly 5 volts. The waveform of the voltage input is shown below:

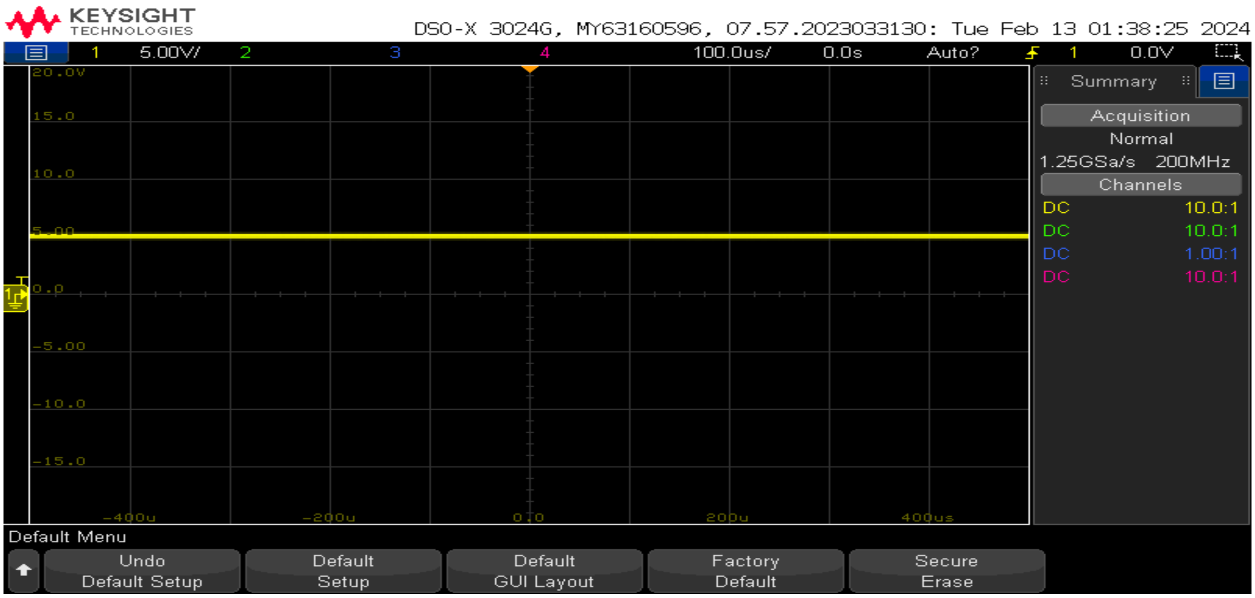


Figure 6 Input Voltage

A picture of the 555-output waveform is shown below when no load is connected. This shows us our Thevenin voltage.

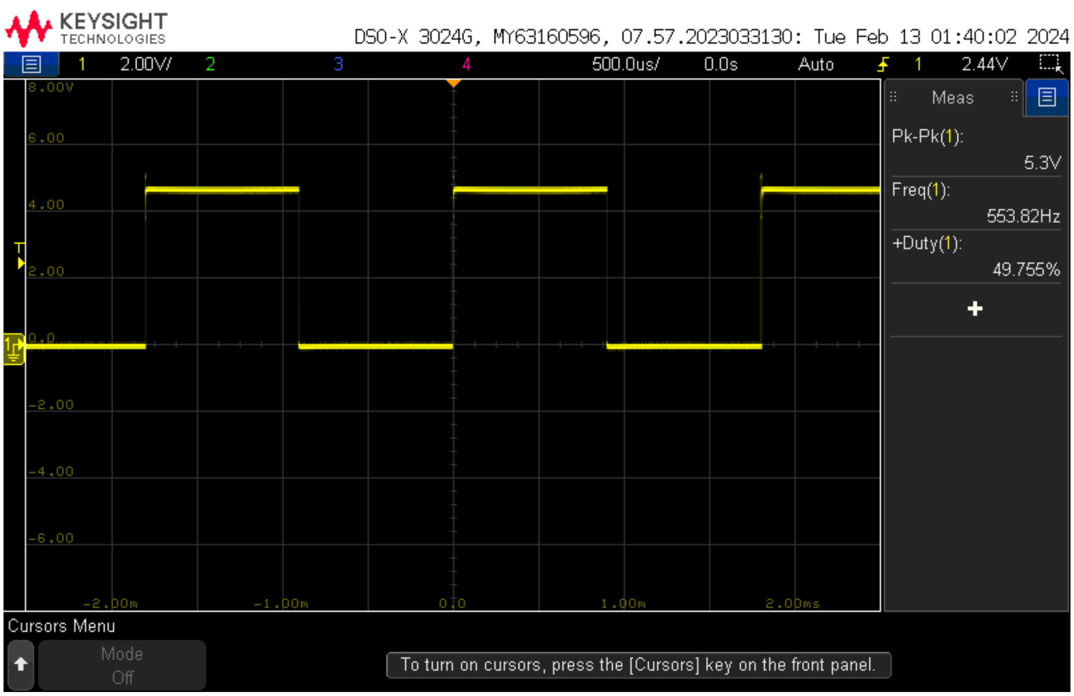


Figure 7 Open Circuit Voltage measured at 555-output

The following screenshot shows the load voltage when it is connected to the output of the 555 timer.

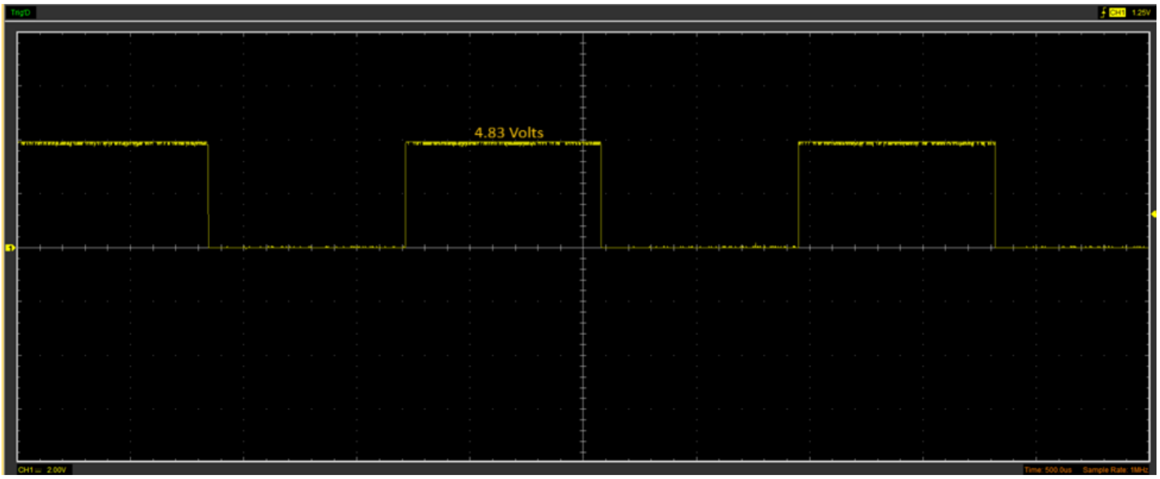


Figure 8 Load voltage waveform

Based on the voltage difference, we can now say that the voltage across the Thevenin resistor, V_{Rth}, is given by :

$$V_{th} = V_{Rth} + V_L$$

Therefore, for this circuit, V_{Rth} would be: 5.3V – 4.83V = 0.47 Volts

To calculate the Thevenin resistance we would need to calculate the total load current which flows into each of the load resistors. The waveform below shows the waveform seen at the test point for the 1k ohm resistor:

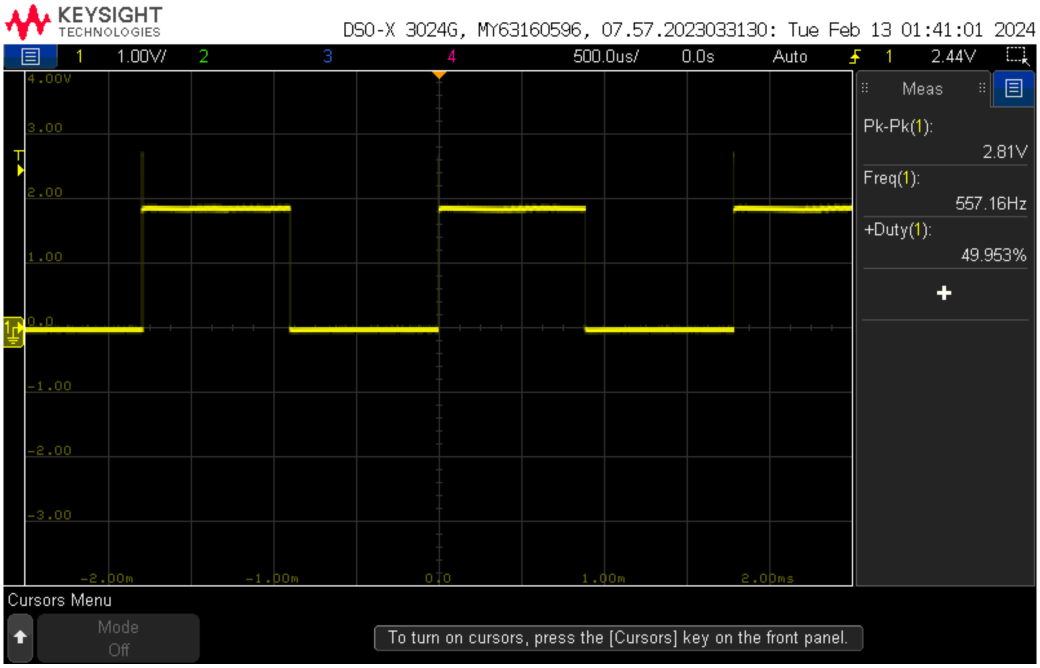


Figure 9 Voltage waveform across the 1k ohm resistor.

Recall that the output voltage with a load is 4.83V. Given this, we can deduce that the voltage drop across the RED LED's would be 4.83V- 2.81V which is 2.02 or approximately 2 volts. However, this value cannot necessarily be used for all the resistor branches due to the voltage drop being different based on the current in the branch and the size of the resistor. Therefore, I measured each resistors voltage drop separately using the spring tips.

Current through the 10k Resistor is: 3.01V/10kOhms = 0.301mA

Current through the 1k Resistor is : 2.81V/1kOhms = 2.81mA

Current through the 300Ohm Resistor is given as : 2.76V/300Ohms = 9.2mA

Current through the 50 Ohm Resistor is given as : 2.45V/50 Ohms = 49mA

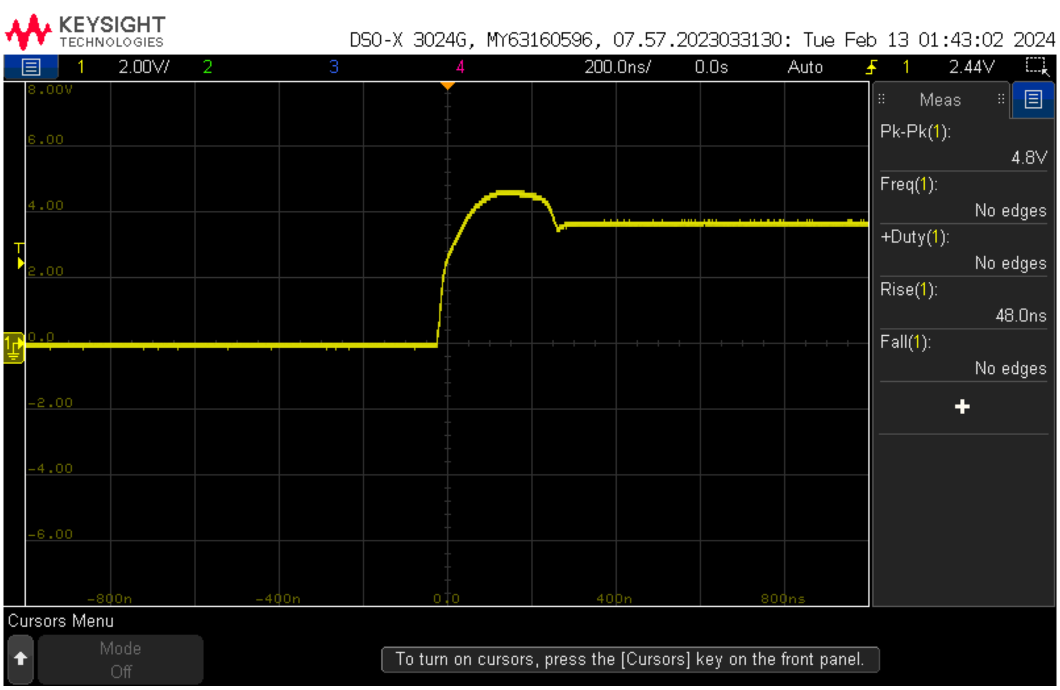
Therefore, the total load current is given as : 0.301 + 2.81 + 9.2 + 49 = 58.3mA

Using this current, we can now calculate the Thevenin resistance as:

$R_{TH} = V_{Rth} / I_L$ = 0.47V/58.3mA = 8.06 Ohms.

Clearly, the frequency is within 500Hz +/- 200Hz as designed and the duty cycle is very close to the 50% that we wanted. The reason in which the frequency is not exactly 500Hz is due to the value of the resistors which was used. Based on calculations, R1 and R2 needed to be 1531ohms and R3 needed to be 344 ohms when using a 0.47uF capacitor. The next closest values of 1.5k and 330 ohms cause the frequency and duty cycle to be a bit away from what's required but still acceptable.

The picture below illustrates the overshot seen due to using the slow version of the 555 Timer.



Note that using the faster CMOS version of the 555 Timer would have a shorter rise time but would provide more switching noise on the power rail as well as overshoot. This is due to the V=L di/dt where a shorter rise team leads to a higher induced voltage drop on mutually inductively coupled components/signals.

Fortunately for this board, everything worked as expected.

Analysis of Project

Things which worked well, and I would like to do in my future designs:

- Placing decoupling capacitors as close as possible to the IC as possible
- Moving very slowly when doing the schematic as well as the layout and double checking each connection as I made it. This led to no errors in my critical design reviews.
- Using my own resistor values as I wanted to use the 3-resistor circuit to get the duty cycle and frequency as close as possible to 50% and 500Hz respectively.
- Using jumper switched to isolate parts of the board during bring up. This was great so I can verify each subsection's functionality before connecting the rest of the board.
- Labelling the resistors at the 555 outputs made it very easy to tell which was which.

Everything worked as intended for this design fortunately and there were no hard errors.

Soft error wise, I believe there are some things I could improve.

- I should triple check the layout before exporting the files as I had typos in my silk screen labelling, and I had to redo it and regenerate the files.
- When soldering I believe I am using too much solder still, I need to let the flux do more of the work.
- I soldered LED's the wrong way. I learned how to verify based on the markings as well as using a multimeter in diode/connectivity mode.
- I could add more test points so I could see more of the circuit figures of merit and this would make measurements much easier.