ECEN 3730-
PCB Design,
Connor Sorrell

PCB Design Board 1 Analysis & Report: 555 Timer Board

Introduction

In this lab, we designed, fabricated, assembled, and tested a 555 timer-driven LED demonstration PCB to practice best design practices, schematic capture, PCB layout, and surface-mount assembly. The board serves as my first schematic to fully functional PCB design process.

The board was designed using Altium Designer, manufactured by JLCPCB, and assembled using 1206-sized surface-mount components for ease of manual soldering.

Design Objectives

- Implement a 555 timer in astable mode to drive LEDs.
- Demonstrate varying brightness levels using different current-limiting resistors.
- Include test points for key measurements.
- Implement isolation switches for debugging.
- Minimize switching noise and power rail disturbances.
- Gain experience with manual SMT assembly.

Bill of Materials (BOM)

- 555 Timer IC (NE555)
- Power plug for 5V input
- 5 x 1206 load Resistors (10kΩ, 2 x 1kΩ, 300Ω, 50Ω)
- 5 x 1206 Indicator LEDs
- Test points
- 2 x 2-pin isolation switches
- 1206 passive components (resistors, capacitors)

---

1. Schematic Design

We designed the schematic in Altium Designer, ensuring proper component selection, signal routing, and test points. The main components include:

- 555 Timer in astable mode (configured for ~500 Hz, 60% duty cycle).
- LEDs with different resistors to observe brightness variations.
- Test points to verify circuit performance.
- Power LED for board operation status.
- Isolation switches to disconnect sections for debugging.

Here, we take special note on two important design practices:

- We will place our decoupling capacitor as close to our IC power pin as possible
    - The decoupling capacitor acts as a local charge reservoir, providing instantaneous current to the IC when needed.
    - If the capacitor is placed too far away, the parasitic inductance and resistance of the PCB traces will cause voltage drops and increased noise
        - Shorter trace lengths = lower loop inductance = lower noise

- We will understand why isolation switches can be placed anywhere in the circuit except between the IC power pin and its decoupling capacitor
    - If the isolation switch disconnects the capacitor, the IC is left without local charge storage, making it vulnerable to voltage fluctuations and causing the power trace to provide all transient current, compromising signal integrity
        - The correct location for an isolation switch is between the power rail and the IC's VCC pin, but before the decoupling capacitor.
        - This allows the decoupling capacitor to remain connected to the IC, ensuring it still provides local filtering and noise suppression even when the IC is isolated from power.

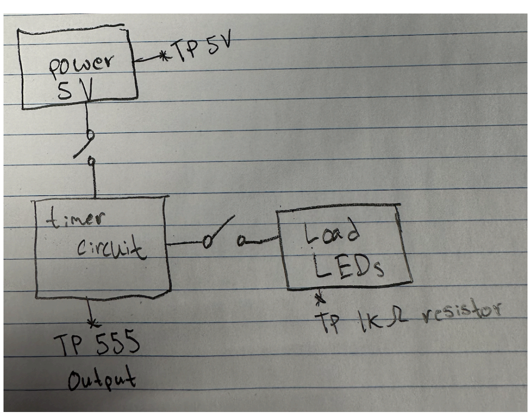

Figure 1: Back of the napkin schematic sketch

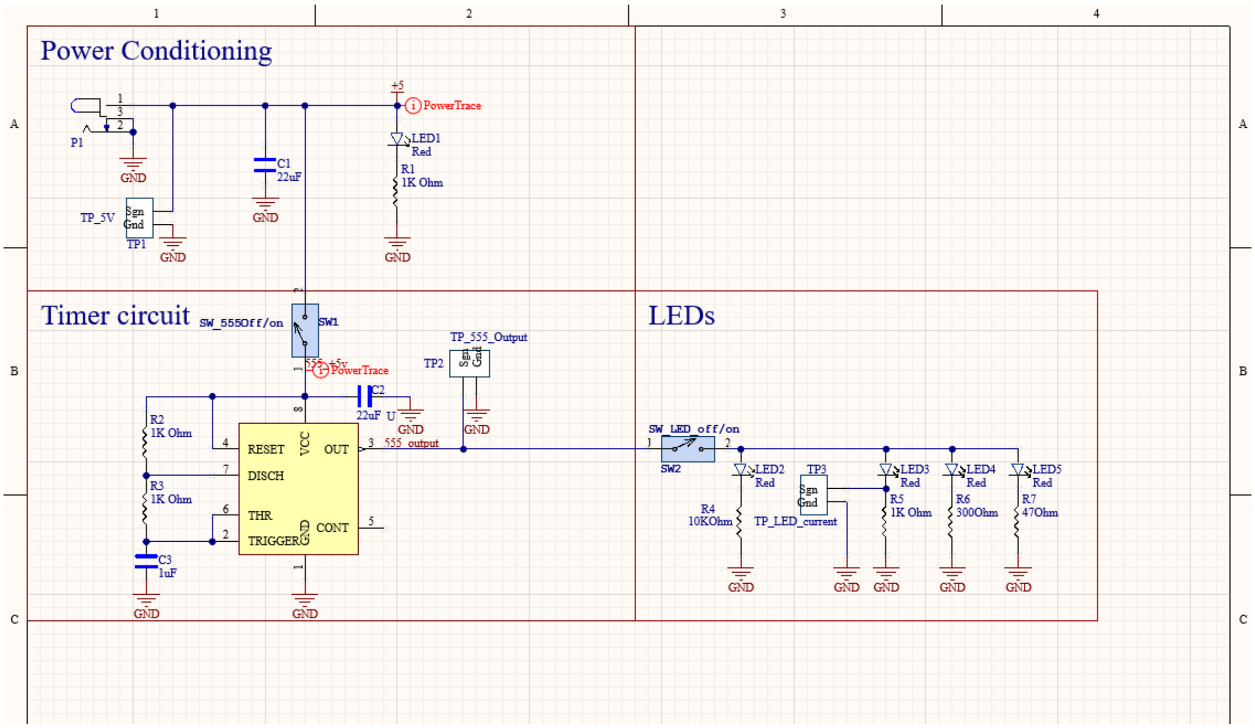

Figure 2: Full schematic sketch of the circuit on Altium, including the power conditioning, timer circuit, and load LEDs.

---

2. PCB Layout

Following best practices, we designed the PCB with:

- Solid ground plane to minimize noise.
- Short trace lengths to improve signal integrity.
- Clear test point labels for easy probing.
- On/off isolation switches to debug/measure certain aspects of the circuit
- LED indicator lights for ease of debugging

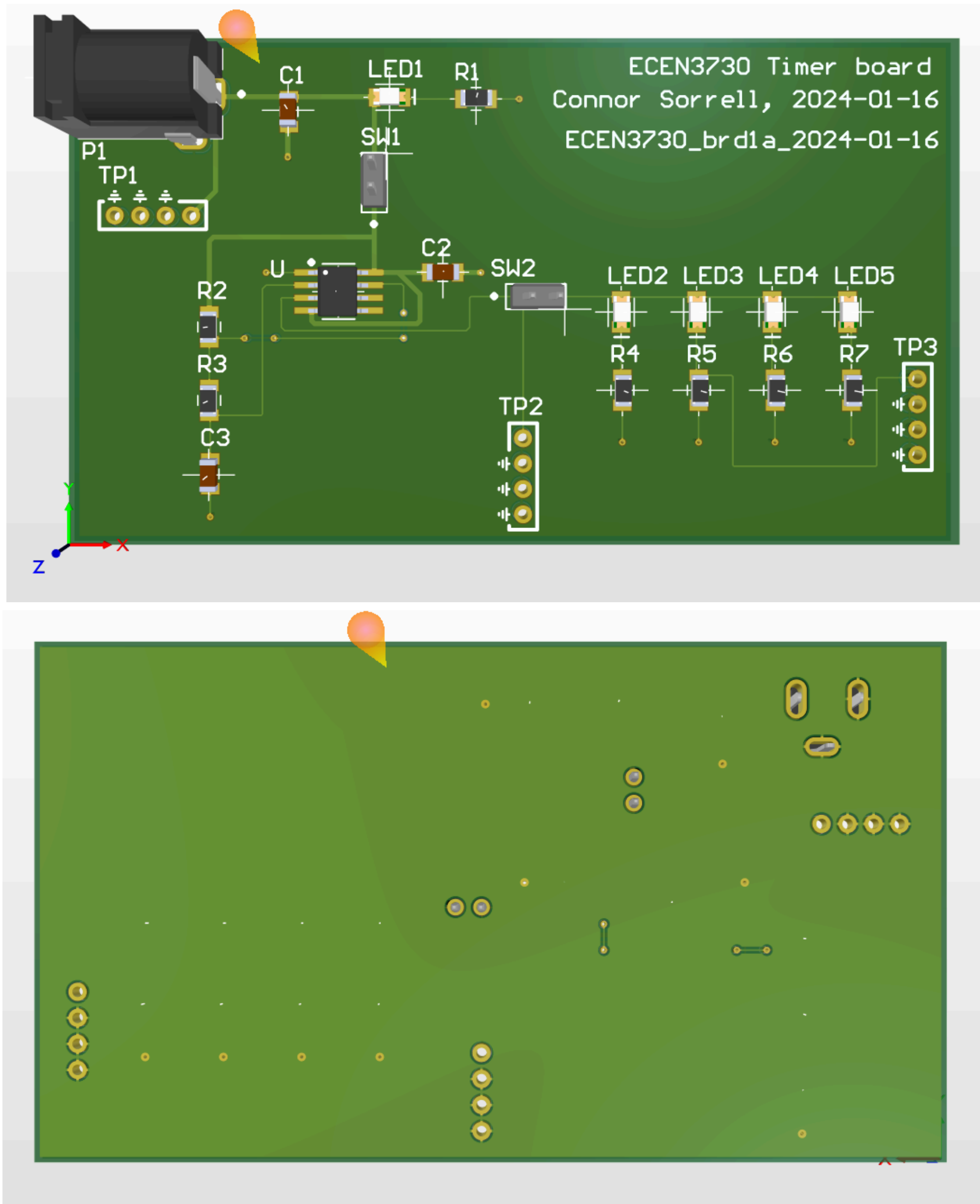

Figure 3: Shows the top and bottom of the completed board design in Altium

---

3. Assembly & Fabrication

- Boards fabricated by JLCPCB, unassembled
- Manually assembled boards using 1206 SMT components.
- Issues encountered: Soldered an LED facing the wrong direction which required almost an hour of rework because of my unfamiliarity with desoldering surface mount components.

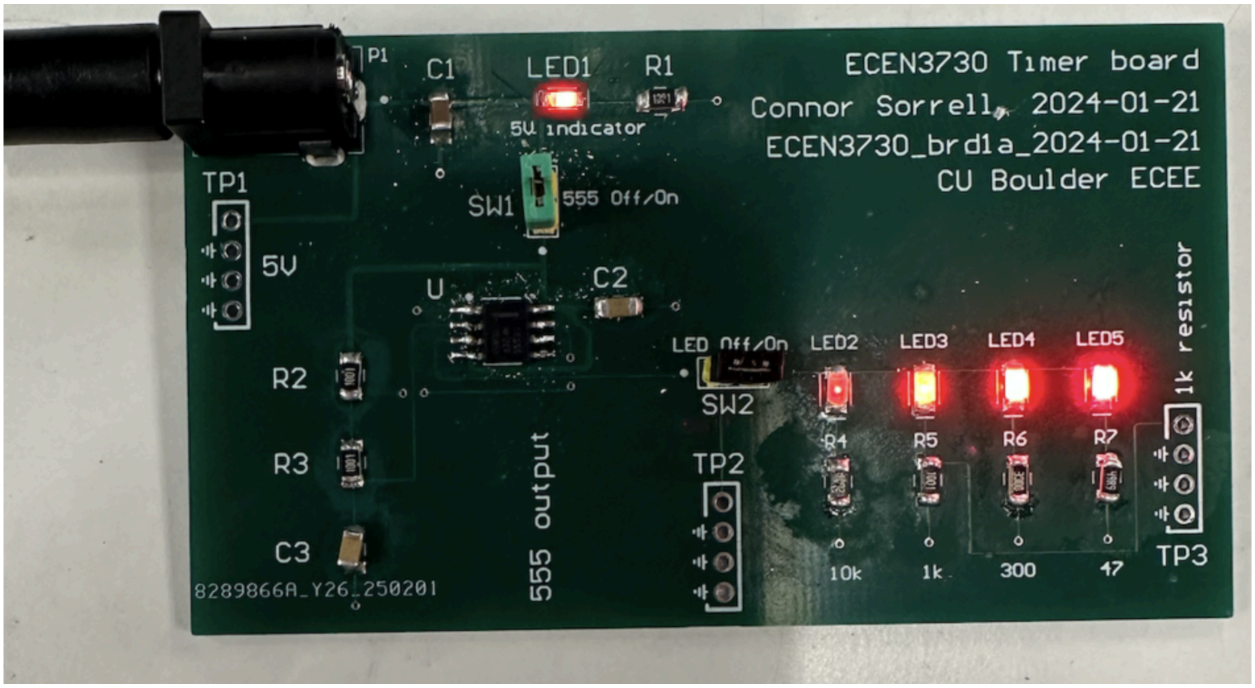

Figure 4: Picture of assembled board with lights on plugged into 5V power adapter

---

4. Testing & Measurements

- For the board to be considered **functional**, it must meet the following criteria:
- The 555 timer produces a 500 Hz square wave with ~60% duty cycle
- The LEDs blink at different intensities based on series resistors
- The circuit operates on 5V DC without excessive power consumption
- Isolation switches effectively disconnect parts of the circuit
- Power rail remains stable, with minimal switching noise

Using an oscilloscope, we measured key performance parameters:

4.1 555 Timer Output (No Load)

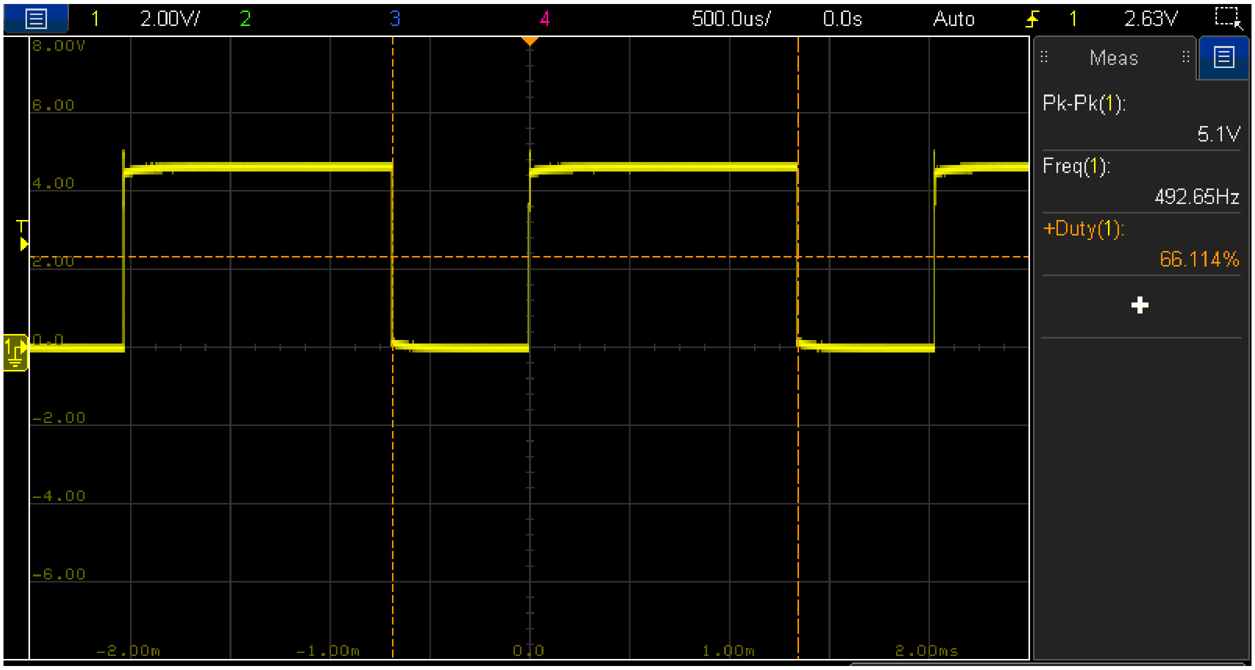

Figure 5: Output of the 555 timer under no load

- Peak-to-Peak Voltage: 4.9 V
- Frequency: ~500 Hz
- Duty Cycle: ~66%

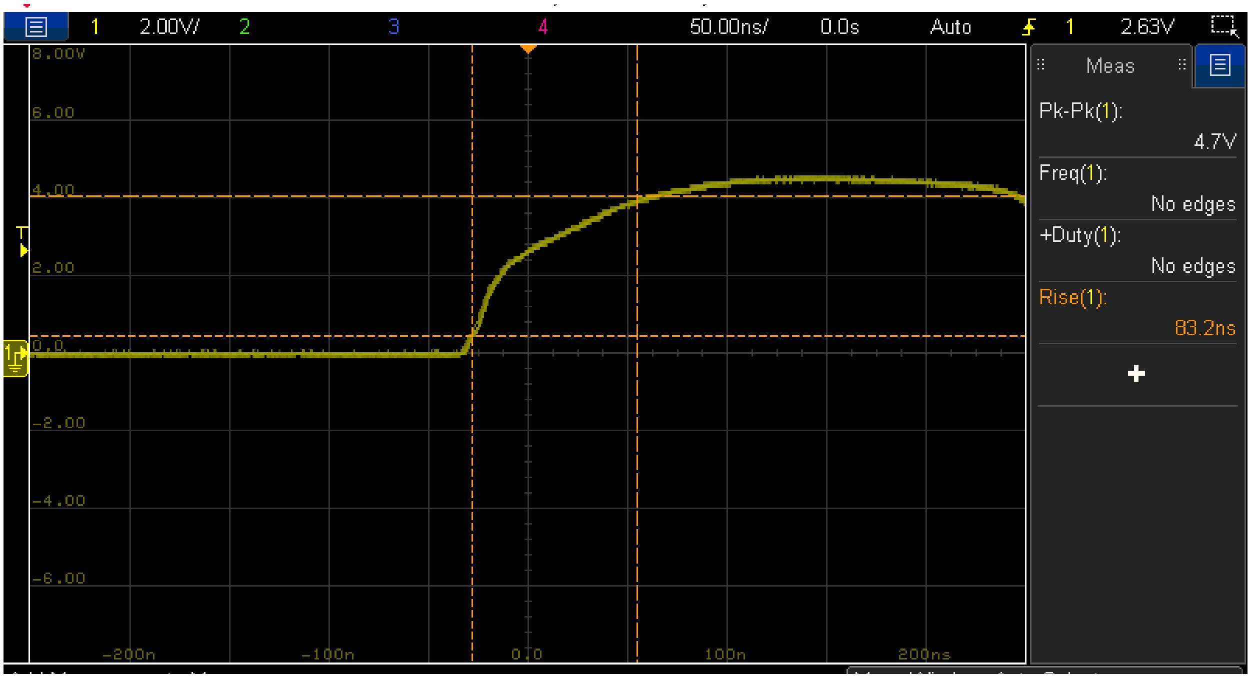

Figure 6: Shows rising edge of 555 Output with no load, rise time = ~80ns

- Oscilloscope shows a square wave output with peak-to-peak voltage of ~5V, frequency of 500Hz, and duty cycle of ~60%, confirming the timer circuit works properly
- The measured rise time is 80ns, which aligns with expected switching speed of the NE555.

Takeaways:

- The measured frequency and duty cycle match the design, confirming that the resistor and capacitor values in the astable circuit were correctly chosen.
- The sharp edges of the waveform indicate the NE555 is switching correctly
  - This verifies that the 555 can generate a stable clock signal
  - If the duty cycle were incorrect, it could affect LED brightness or cause excessive power consumption, showing the importance of precise resistor/capacitor selection.
  - A longer rise/fall time might indicate higher output impedance, excessive loading, higher transient, which could greatly impact the switching performance in other designs

4.2 555 Timer Output (Under Load with LEDs Connected)

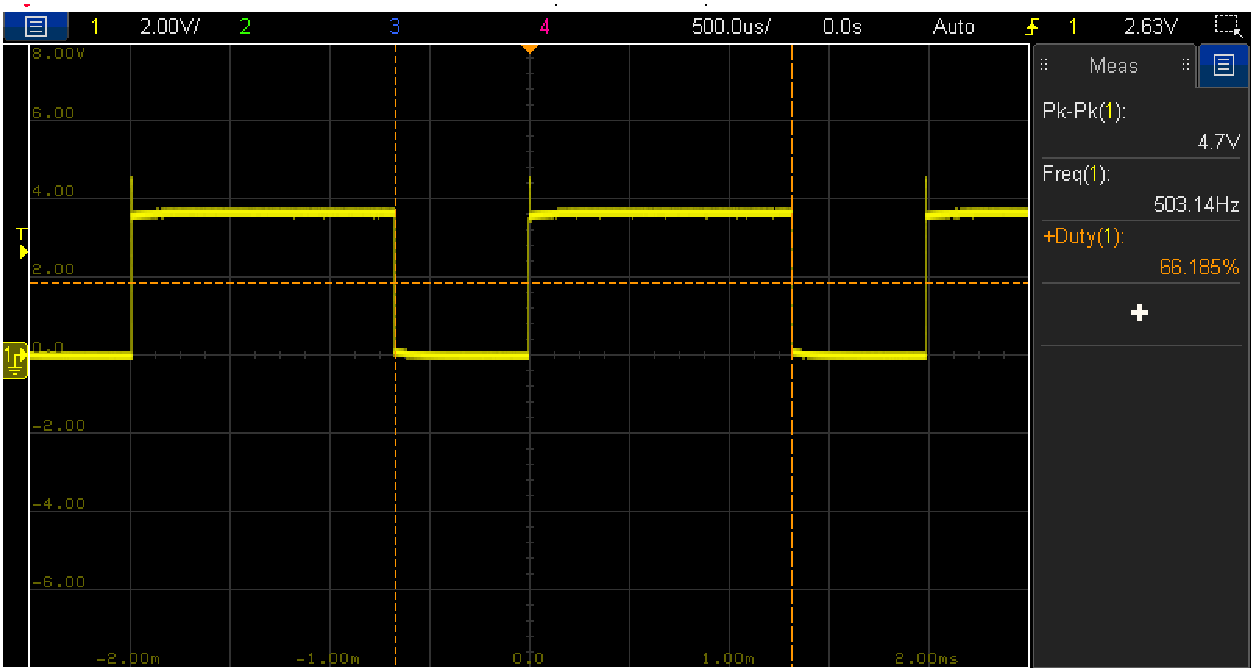

Figure 7: Shows output of 555 timer under load (All 4 LEDs connected), now at ~3.8 V

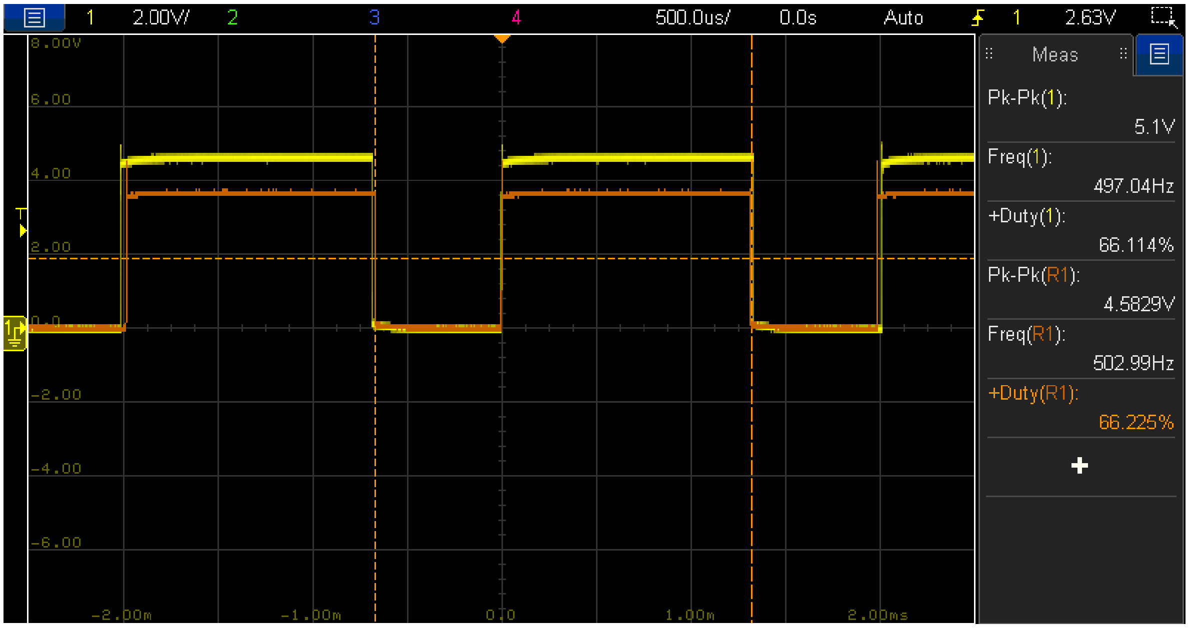

Figure 8: Shows yellow waveform (555 output w/ no load, 5V) vs. orange waveform (555 output with all 4 LEDs connected, 3.8 V)

- With LEDs connected, the measured peak-to-peak voltage drops from 5V to 3.8V
- Frequency remains stable at 500Hz, but rise time increases slightly, showing the 555 struggles more under load.

Takeaways:

- The voltage drop suggests that the 555 timer has a nonzero output resistance and that loading effects reduce the voltage at the output..
    - This confirms that the NE555 has limitations in sourcing current, reinforcing the importance of checking an IC's maximum output current capability.
    - If the voltage dropped too much, LEDs could dim inconsistently, demonstrating why understanding load effects is crucial in real-world circuits.

4.3 LED Current Measurements

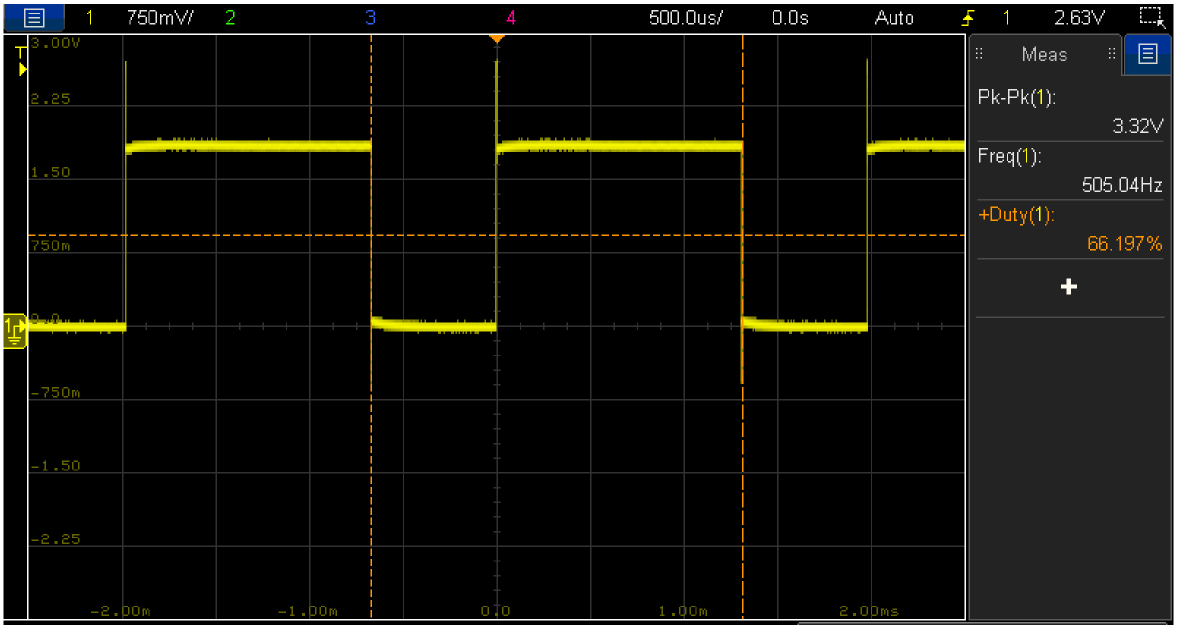

Figure 9: Shows the voltage drop over the 1k Ω resistor (~1.9V)

$$For\ the\ 1k\ \Omega\ Resistor:\ I_{LED} = \frac{V_{resistor}}{R} = \frac{1.9\ V}{1000\ \Omega} = 1.9\ mA$$

Resistor (Ω)	Voltage ($V_{resistor}$)	Current (I_{LED})
47	1.9	36 mA
300	1.9	6 mA
1k	1.9	1.9 mA
10k	1.9	190 µA

$$I_{load} = I_{LED1} + I_{LED2} + I_{LED3} + I_{LED4} = \sim 44mA$$

- We chose the NE555 timer, which can source or sink 200mA, which is more than enough to support driving the current through all the LEDs. Based on the components we chose, our total current draw through all the LEDs is ~44mA, which is well within our limits. As we move forward with future labs, it will be extremely important to consider current limitations when choosing ICs.

- The NE555 consumes 30mW of power at 5 V, and we can also estimate that our series resistors are consuming $P = IV = 44mA \cdot 1.9\,V = \sim 85mW$. Clearly, the majority of our power consumed is in the series resistors of the LED's. As we move forward, it will be vital to begin thinking about the power needs of our circuit and how to estimate it.

4.4 Thevenin Resistance of the 555 Timer

- Calculated using: $R_{th} = \frac{V_s - V_{load}}{I_{load}} = (5\,V - 3.8\,V)\,/\,44mA = \sim 30\,\Omega$

- $V_{oc} = 5\,V$ (Open Circuit Voltage) (Figure 5)
- $V_{load} = 3.8\,V$ (Loaded Voltage) (Figure 8)
- $I_{load} = 44mA$
- $R_{th} = 30\Omega$

4.5 Indicator LEDs

- The LED attached to the 10k resistor can still display a very dim, but visible light at ~0.18mA
- Since the LEDs are pulsed with a duty cycle of ~70%, the effective brightness is lower than in a continuous DC circuit.
- I would estimate that for an effective indicator light, an efficient trade-off between power consumption and LED brightness would be 3-5mA through the LEDs
  - Using $P = IV$, we can estimate $P = \sim 5mA \cdot \sim 2V = \sim 10mW$
  - Power conscious, yet good visibility.

Criteria for selecting the current limiting resistor for general indicator lights:

- Using my assumption from above, we can choose current limiting resistors for indicator lights in order to provide the LED with 3-5mA.
- We can estimate our LED forward voltage drop: We have 3.8V coming out of the 555 timer, and 1.9V over the 1k resistor, so we can assume our forward voltage drop is another 1.9 V across the LED.
- Example, with a red LED (~2 V forward voltage), 5V supply, and 5mA desired:

  $R_{limiting} = \frac{5\,V - 1.8\,V}{5mA} = \sim 650\Omega$

  - Based off these assumptions, a 1kΩ resistor would be smart for indicator LEDs because it would achieve low power consumption, enough visible light, and is a cheap, common part

4.6 Power Rail Noise (Switching Noise on 5V Rail)

- Observed noise amplitude: 62 mV
- Synchronous with 555 switching

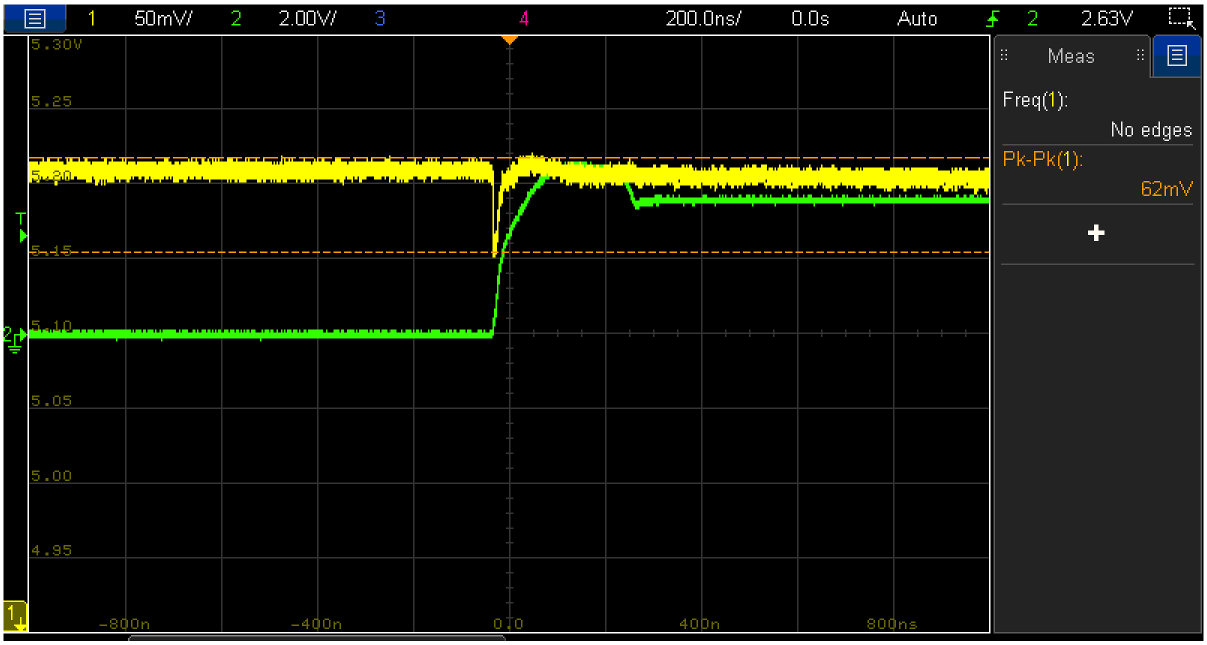

Figure 10: The switching noise voltage on the 5 V power rail (yellow waveform), synchronous with the 555 switching signal (green waveform)

- 67mV noise is relatively low, meaning the power integrity is well-maintained.
    - The decoupling capacitor selection and PCB layout (ground plane) helped mitigate excessive noise
- We still see IR drop and L dI/dt effects on the power rail due to parasitic resistances and inductances.

---

5. Analysis & Discussion

Measured Performance vs. Expectations

- The oscilloscope verified the expected 555 timer signal (~500 Hz with ~60% duty cycle)
- LED brightness correlated with resistor values as predicted, verifying current-limiting resistors function correctly.
- Current measurements aligned with expected values
- Switching noise was minimal, confirming a well-designed power layout with decoupling capacitor close to the IC and a ground plane

Design & Measurement Practices Used

- Ground planes minimized noise
- Short trace routing reduced parasitics.
- Test points and isolation switches improved debugging and allowed precise measurements.
- Oscilloscope was properly triggered to reduce artifacts.
- 1206 components were *relatively* easy to solder.

What Could Be Improved:

- Soldering surface mount parts. Initially, I accidentally soldered the 5v indicator LED facing the wrong way, causing it not to turn on. I can safely say that Removing surface-mount components proved to be challenging. The process of desoldering and reworking the component took significantly longer than expected, reinforcing the importance of not just double checking, but triple checking before placing components.
  - Key Takeaway: Be absolutely sure of component orientation prior to soldering, which can prevent costly mistakes and save hours of rework in future projects.
  - I need to improve my SMT skills, particularly desoldering.
- Pay closer attention to measurement values when I take scope screenshots moving forward. My screenshots appear confusing because they include irrelevant values (peak-to-peak measurements) and would be more informative if I included the "maximum" value instead.

---

6. Conclusion

This lab provided hands-on experience in the entire PCB design flow, from the back of the napkin sketch to schematic & board design to assembly and testing. Key takeaways include:

- The importance of good PCB layout for signal integrity.
  - Keeping power traces short to minimize loop inductance
  - Place decoupling capacitors as close to IC as possible
  - Utilize a ground plane for return current
- The real-world behavior of the 555 timer's output resistance.
  - The effect of the thevenin resistance on the circuit
  - Current-drawing capabilities
- Best measurement practices to verify circuit operation.
  - Debug with spring tips
  - Place test points all around circuit for easy debugging
