Board1 Report

555 Timer Circuit

ECEN5730 Fall 2025

Hyounjun Chang

Project Overview

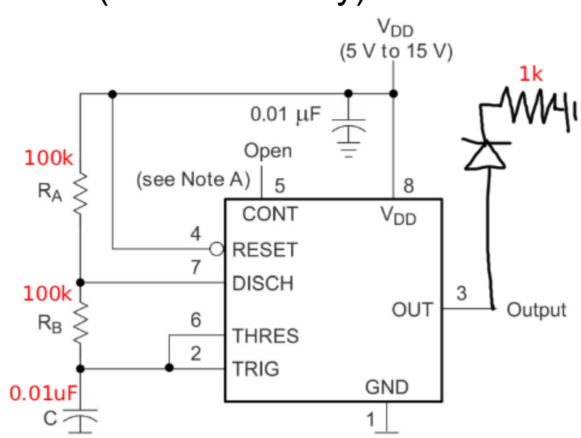
Plan of Record (POR)

- Create Schematic for 555 Timer Circuit with ~70% duty Cycle at ~500Hz **(Completed Week 2)**
- Select BOM by testing fast/slow 555 timers and different resistors/capacitors **(Completed Week 2)**
- Test the schematic on solderless breadboard (with BOM, selected by the class) **(Completed Week 2)**
- Finish PCB design of 555 Timer Circuit, adding on indicator LEDs, test points, and isolation switches for debugging **(Completed Week 3)**
- Do CDR of PCB Design **(Completed Week 3 with TA)**
- Bring-up the designed PCB board by assembling 1206 parts, and test/debug functionality of the assembled board **(Completed Monday of Week 7, delayed due to personal events)**

Project Goals (Working Board)

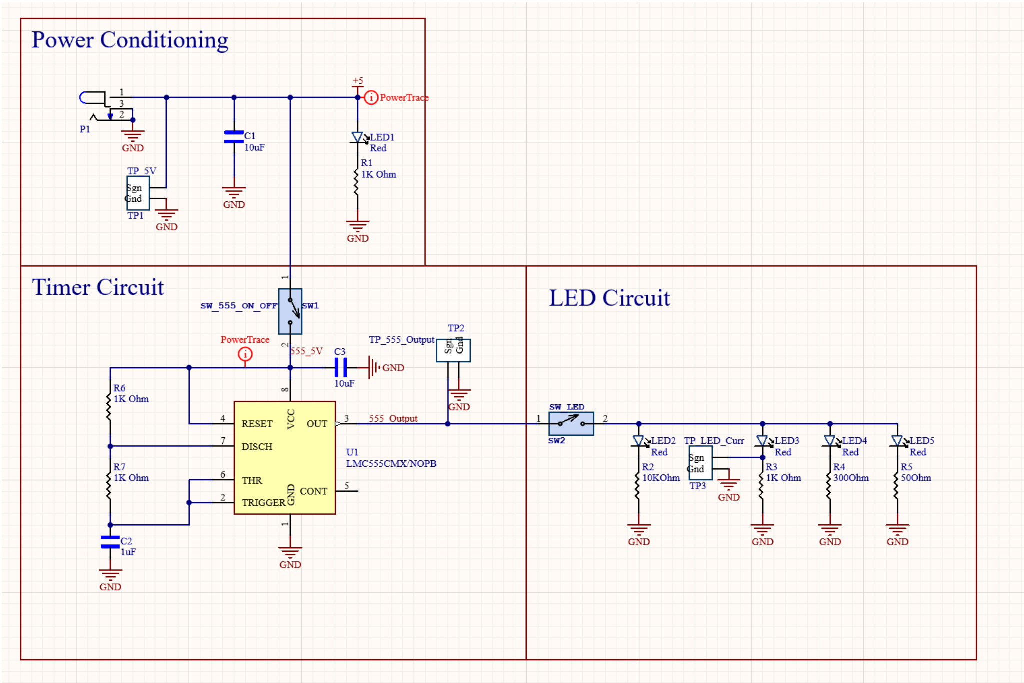
- Circuit takes 5V AC power jack input for power, with an indicator light for power
- Create a circuit outputting ~500Hz signal with ~70% duty cycle with the 555 Timer
- 555 Timer Circuit drives 4 LEDs with different resistors connected to them
- No major voltage spikes/drops/overshoots/undershoots
- Circuit is easily debuggable, with Test Points and LED indicators
- No issues with board assembly
- Being on Schedule!

Sketch of the Schematic (555 Timer only)



This design creates a circuit with 480Hz, 66.6% duty cycle timer. Initial capacitor sizes are small at 0.01uF. This schematic was tested in a solderless breadboard, which resulted in a circuit with similar frequency and duty cycle.

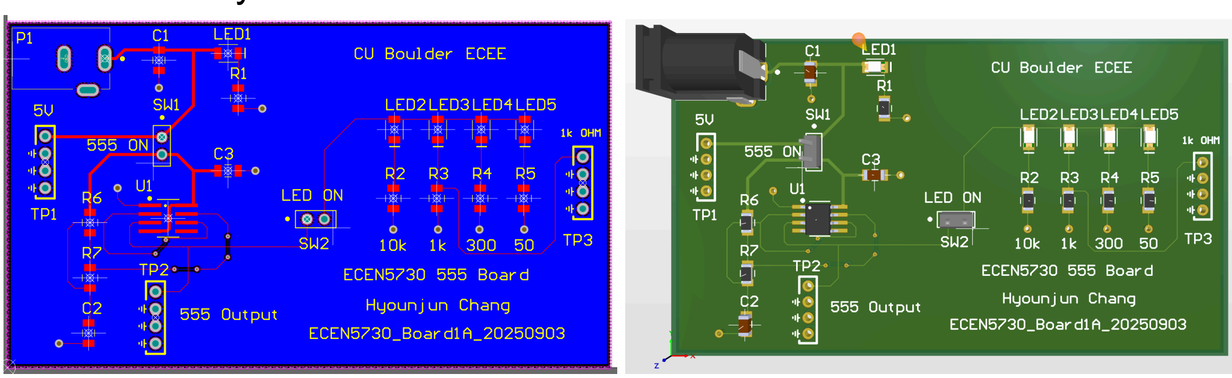
Altium Designer Schematic



Resistor values were changed from 100k to 1k ohms, and the 555 capacitor to 1uF, to match the available parts. Capacitors next to power lines were increased to 10uF to reduce power rail bounce.

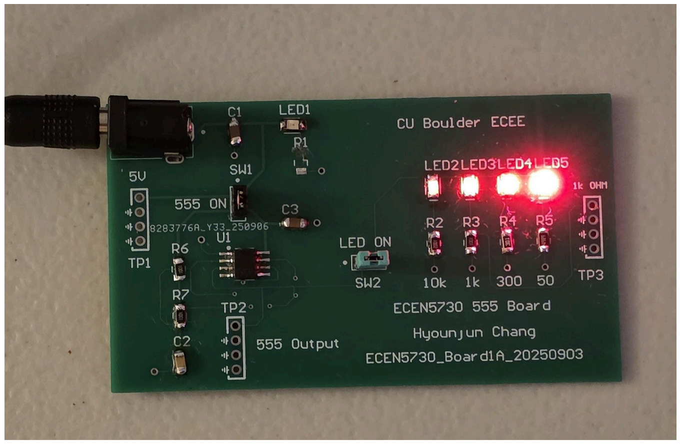
Additional parts, such as Power Conditioning circuit (for power supply), and LED indicators were added as well. Test points were added for future debugging, and 2 switches were added to isolate different parts of PCB.

PCB Board Layout



2D view (Left), 3D view (right), displayed on Altium Designer

Picture of Physical Board



Assembled 555 board with LED lights on. Note that LEDs get brighter as resistances drop.

Board Functionality:

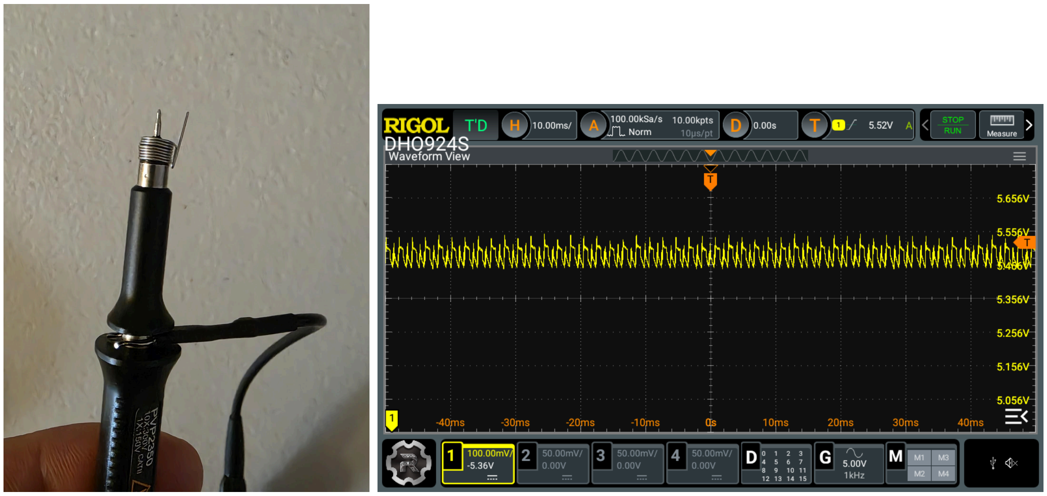
Test Points:
TP1: Power Rail Test Point
TP2: 555 Circuit Test Point
TP3: LED 1K Resistor Test Point

Expected Working Board:

- Stable 5V power-rail output on TP1, and LED1 on while power jack is connected
- TP2 outputs a 480Hz 67% duty cycle clock, with same Vpp (5V), when SW1 connected
- LED2 to LED5 shows different brightness levels (due to different current)
- TP3 (LED 1K Resistor) shows reduced voltage from Vpp
- TP2 (555 timer) output varies when SW2 is connected/disconnected (LEDs connected/disconnected), due to real-life characteristics

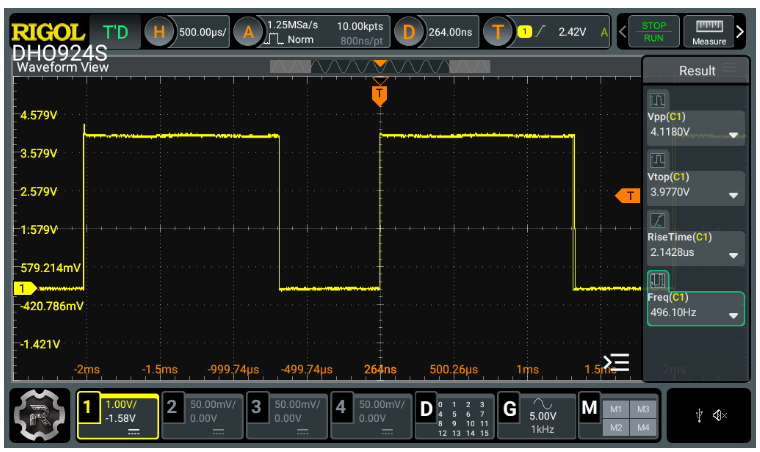
Actual Assembled Board:

Signals inside all test points (TP1, TP2, TP3) were reasonable, but had some surprising results. Oscilloscope probe and spring connector (for ground) was used to measure signals for all test points to minimize noise.



Spring Connector used in 10x Probe (left), TP1 power rail 5.5V output (right)

From a far glance, the 555 circuit seems to be working properly; LEDs which are connected turn on, with LED5, which has lower resistance, is the brightest. However, looking at the test points tells a slightly different story.



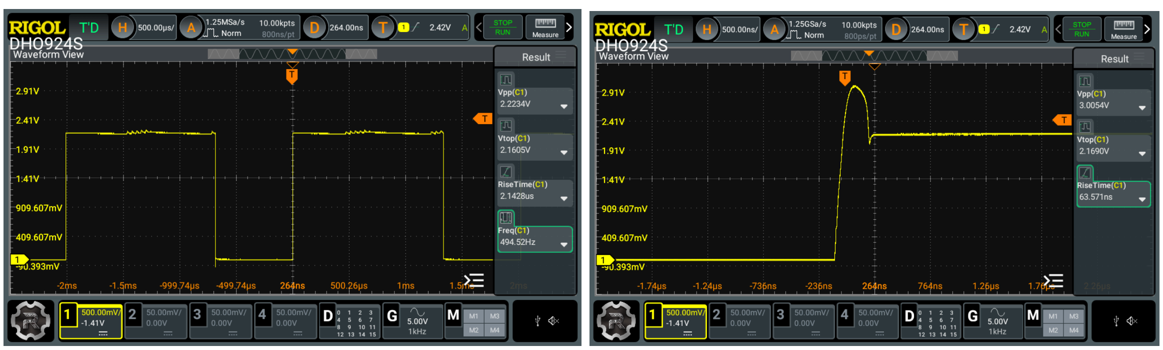
TP2 measures output signal of 496Hz, with 66% duty cycle, with 4 LEDS turned on.

TP2 (555 circuit) output was degraded, due to differences between ideal circuit design and real-life components. The stable output voltage is around 4V, which is a lot lower than Vpp of 5.5V. There is an initial small overshoot which is not able to be seen in this timescale.

Zoomed in TP2 signal, LED on (left), LED off (right)

Further zooming into the signal shows that the initial “overshoot” is not an overshoot, but a parasitic component in the circuit. with a rise time of around 50n. It initially reaches 4.95V, close to supply voltage, then dips in voltage to 4V with LEDs on.

There was a difference in output when the LED was turned off - stable high voltage rose from 4.03V to 4.96V, and the “overshoot” reached 5.5V, which is equivalent to Vpp of power rail. This suggests that LED loads cause the 555 timer output voltage to drop, but the initial “overshoot” is not the result of LEDs by themselves.



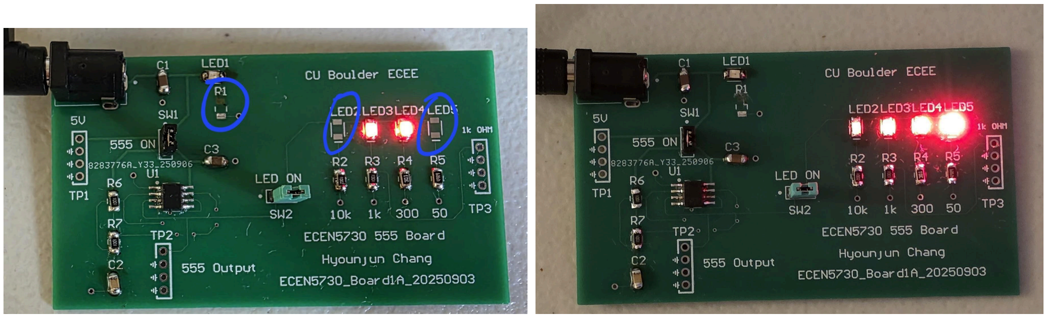
TP3 signal for 1K ohm resistor (left), zoomed in (right)

Frequency of signal at TP3 is around the same at 495Hz, and the same patterns remain visible, although with change in peak voltage compared to TP2. 4.03V stable output from TP2 is now at 2.16V in TP3, meaning that there is 1.8V drop along the LED. Current in the 1K ohm resistor is around 2mA, which will not burn the LED.

Same “initial overshoot” behavior is seen alongside the resistor as well. The voltage initially peaks to 3.3V, then falls down to 2.1V. Parasitic elements of the circuit are affecting this TP3 as well.

Project Analysis:

Failures in Board:



Initial assembly missing 2 LEDs (left), Final assembly with LED1 not functioning (right)

There were both hard and soft errors in the board during assembly.

Hard Errors:

- 1uF capacitor (C2) was initially placed in the wrong location (R1). Removing the 1uF capacitor caused the metal layer of R1 to fall off as well, preventing future solder connections
- As a result, LED1 did not turn on, as it was not connected to ground

Soft Errors:

- Only 3 LEDs were initially placed, as 2 LEDs were lost during assembly. Replacement parts were needed on Monday for the fix.
- Lack of capacitance values for C2 led to initial confusion. Since 2 different capacitor values are used, it would be helpful to write some values in the silkscreen.

What went well:

- Soldering of the 1206 parts was relatively easy, since there was enough space between any 2 parts.
- Silkscreen display matches the schematic, which is helpful for assembly
- Thermal relief of PCB made soldering much easier.

What didn't go well:

- Assembly of the board: many parts were lost due to disorganization of the workspace, causing further delays
- Lack of focus lead to parts being placed in wrong locations

Future Improvements:

- Adding values to capacitors that are not common reduces the likelihood of the part being placed incorrectly
- Reduce number of different parts, especially resistors and capacitors, as it is easy to misplace them during assembly

- Getting soldering correctly the first time reduces assembly issues, and correcting assembly mistakes is much harder! I need to learn proper techniques so parts do not swivel when soldering to make future assemblies more consistent
- Always keep spare parts to minimize delays if manually assembling!
- I need to be fully aware of schematic when assembling the equivalent circuit on the PCB