Board 1: Practice Board — Final Board Report

Andrew Thibeault — 2023-20-02

Plan of Record

Board Function and Feature Set

The main purpose of building this board is to get experience with the entire board design flow, from simple sketch to layout to assembly and test, with a simple and functional design. The function of the board is to operate a 555 timer as an astable vibrator to drive four LEDs with different series resistors, which should result in varying brightnesses of the LEDs.

The following features are implemented on the board, and their design considerations given:

1. A power plug to power the board using a 5 V AC to DC converter.
2. A 555 timer chip and circuitry designed for about 500 Hz and a 60% duty cycle. The expected current load from the timer is about 61 mA. Both the fast timer (LMC555), with a current rating of 100 mA, and the slow timer (NE555), with a current rating of 225 mA, should be able to handle this. There is no concern about speed in this circuit, and the fast 555 is an extended part, so the slow 555 will be used (NE555 datasheet).
3. 1206 parts from the JLC library will be used, with a preference for basic parts due to their lower cost.
4. 4 green LEDs (LED datasheet), each in series with a different resistor of 10k, 1k, 300, and 47 ohms, will be driven by the 555 output. The 60% duty cycle will limit the average current passing through the diodes, so that the max current passing through the 47 ohm-connected LED will be 51mA at 500 Hz and 60% duty. The LED is rated for 60 mA at 1 kHz and 10% duty, so this current may be outside the rating. This will be examined in testing.
5. Indicator LEDs are used for indicating power being output from the power plug, as well as a signal being output from the 555.
6. Isolation switches separate the power circuitry from the 555, and the 555 from the LEDs, in order to incrementally test the circuit.
7. Test points are added to the 5V input rail and the 555 output to verify the correct voltage signals, and across the 1k ohm resistor to measure the current passing through the LEDs.

When fully assembled and powered, the four test LEDs should be lit, driven by a 5V pulsed waveform with approximately 60% duty cycle at 500 Hz from the 555 output.

Schedule

The board schematic and layout should be completed by 2023-01-26. Testing and final report should be completed by 2023-02-20.

Board Design

Block Diagram and Schematic

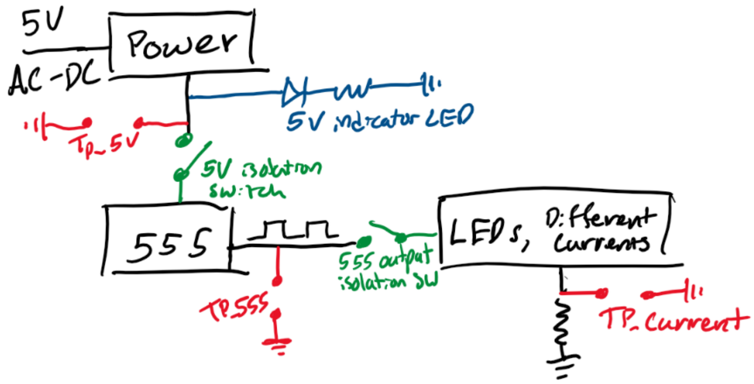

Figure 1: Block diagram

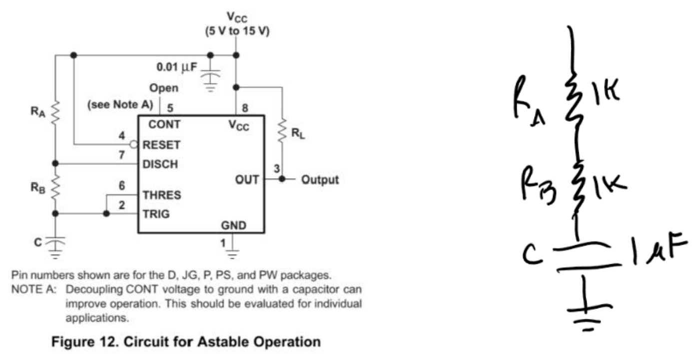

Figure 2: Rough sketch of 555 design

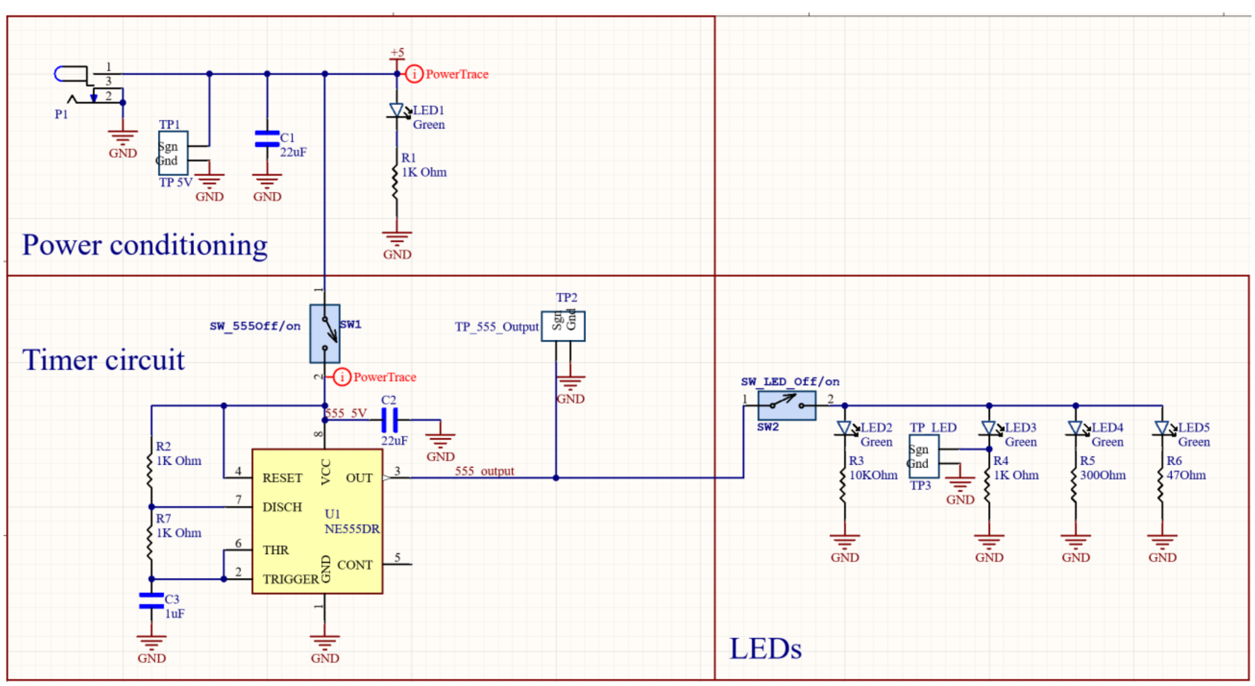

Figure 3: Circuit schematic

The schematic design in figure 3 gives light to additional details for how the board should function.

Decoupling capacitors are added close to the power input, to reduce noise from the power input, and to the 555, to reduce the inductive loop and thus switching noise.

The power rail current can be estimated from this schematic. In total, the power rail supplies an indicator LED as well as the 555 driving the test LEDs. The expected current draws are 5mA for the indicator LED, 3 mA to power the 555, and 61 mA to power the test LEDs; 69 mA in total, well within the current rating of the 20 mil trace that will be used for the power rail.

The 555 astable operation configuration can be seen in figure 2, with 1k resistors for R_A and R_B, which determine the duty cycle to be 66%. The 1 µF capacitor sets the frequency to 480 Hz. Using 1k resistors for the 555 as well as the indicator LED and one of the test LEDs reduces the number of unique parts, simplifying and reducing cost for self-assembly.

The cost of this board is expected to be at the minimum cost provided by the fab shop, less than $5 for 5 boards. A minimum of extended parts are used, and only basic design features provided by the PCB fab shop should be needed.

Assembly and Bring-up Plan

The unassembled board will be tested for shorts between power and ground, and across isolation switch pads.

The addition of isolation switches allows for assembly of the entire board at once as circuit blocks can be tested incrementally by adding switch connections one by one. The 555 SOIC will be soldered first to allow easy access to the leads, followed by the SMT parts, and finally the isolation switch pins and power jack, which can act as awkward obstacles for soldering other components. Solder joints will be visually inspected under a microscope for good joint quality.

Once fully assembled, the test points will be used to verify performance in each block, with all isolation switches initially removed. A 5V DC signal should be seen at TP_5V (referenced to figure 3), and the indicator LED for this block should light. Connecting the SW_555off/on switch, a pulsed waveform should be seen at TP_555_Output, with 5V peak-to-peak amplitude, 66% duty cycle, and 2.1 ms period. Connecting the SW_LED_off/on switch, the four test LEDs should light up to varying degrees. The highest resistance LEDs may be very faint or not visible at all due to low current. A pulsed waveform should be seen at TP_LED with around 2 V peak-to-peak amplitude and the same duty cycle and period as the 555 output.

The current though the 1k ohm resistor can now be measured using the voltage measured at TP_LED, and the current through the other resistors and series LEDs estimated from this. We can correlate these currents to the perceived brightness of the LEDs to get an idea of how these LEDs will perform in future circuits with a given current.

Synchronous switching noise on the power rail should be checked for, with the 555 loaded.

One potential risk site is that the LED connected in series with the 47 ohm resistor will be overdriven with current, and may heat up and operate inefficiently, decreasing its lifespan, or in worst case will fail. If a short is formed in this LED, the 47 ohm resistor still separates the 555 output from ground, so a short to ground will not be formed, though this resistor will be dissipating approximately .5 W, double its .25 W power rating. The risk of this happening is low, as LEDs can be safely overdriven. The reduction in LED lifespan is not a concern, as this board does not provide functionality outside of verification that it works as expected in a practice capacity, and a quick measure of how LED current relates to brightness.

Board Layout

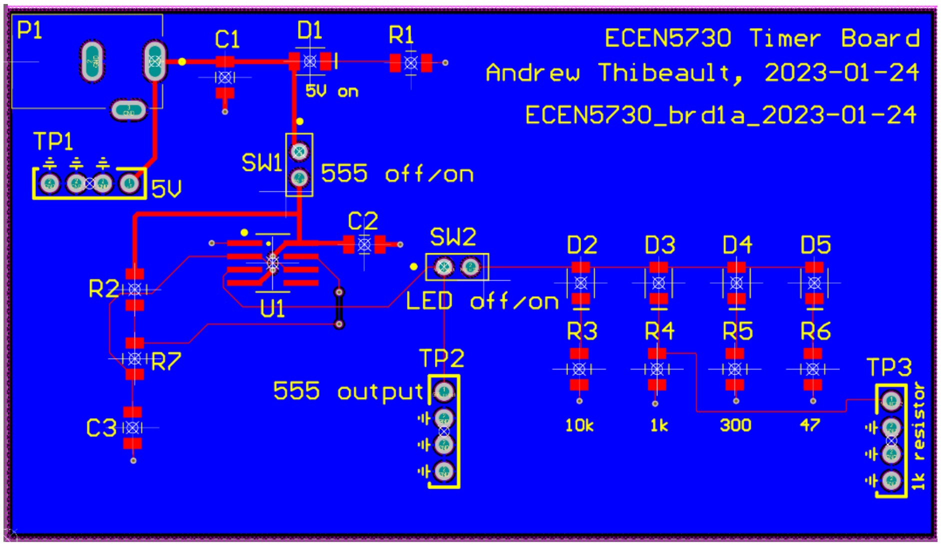

Figure 3: Final board layout

The board dimensions are 2”x3.5”. This was plenty of room to fit all of the components with space for easy soldering.

Decoupling capacitors are placed close to the power input and the 555 SOIC to minimize the inductive loop lengths.

20 mil routing widths are used for the power rail, mainly for ease of identification of the power rail, as the max expected current is well below the rated current of even a 6 mil line. 6 mil routing widths are used for the rest of the interconnects, to stay within the default design specifications of most fabrication shops.

Thermal relief vias are used for through-hole vias for ease of soldering.

A continuous copper pour ground plane is used to minimize inductive loop noise. Short cross-unders are used where minimally needed in order to minimize breaks in the ground plane.

Physical Board

Bare boards for self assembly and fully assembled boards were received from the fab shop.

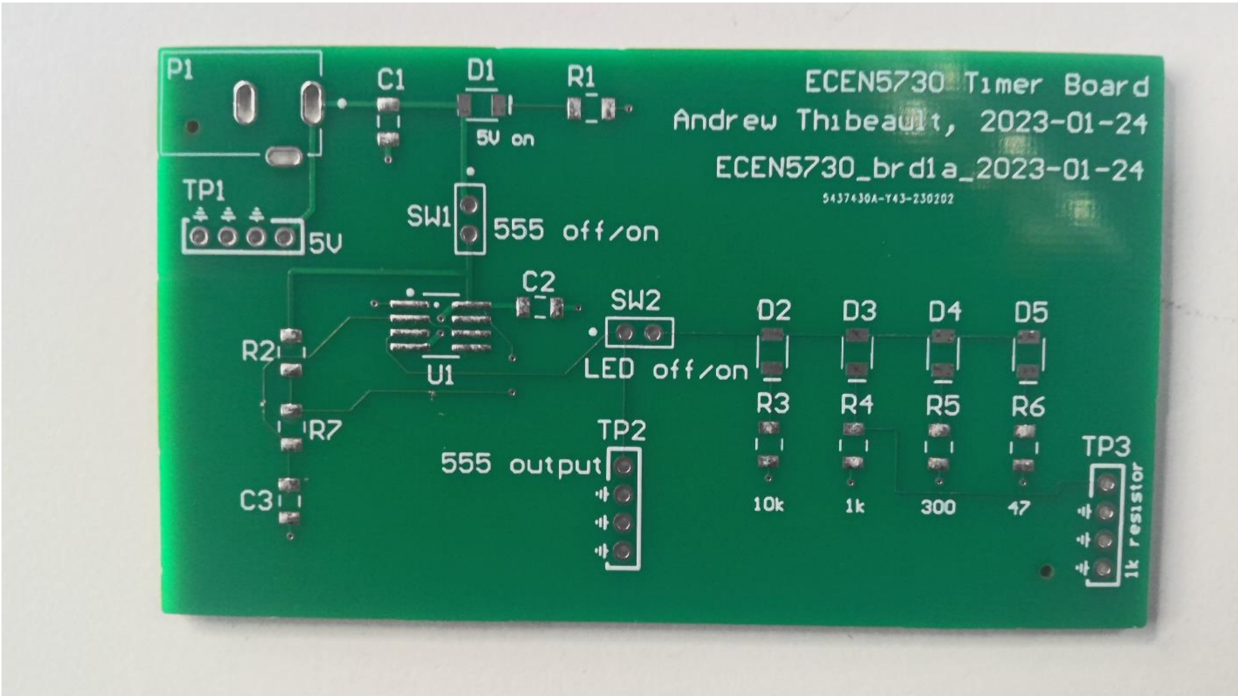

Figure 4: Bare board

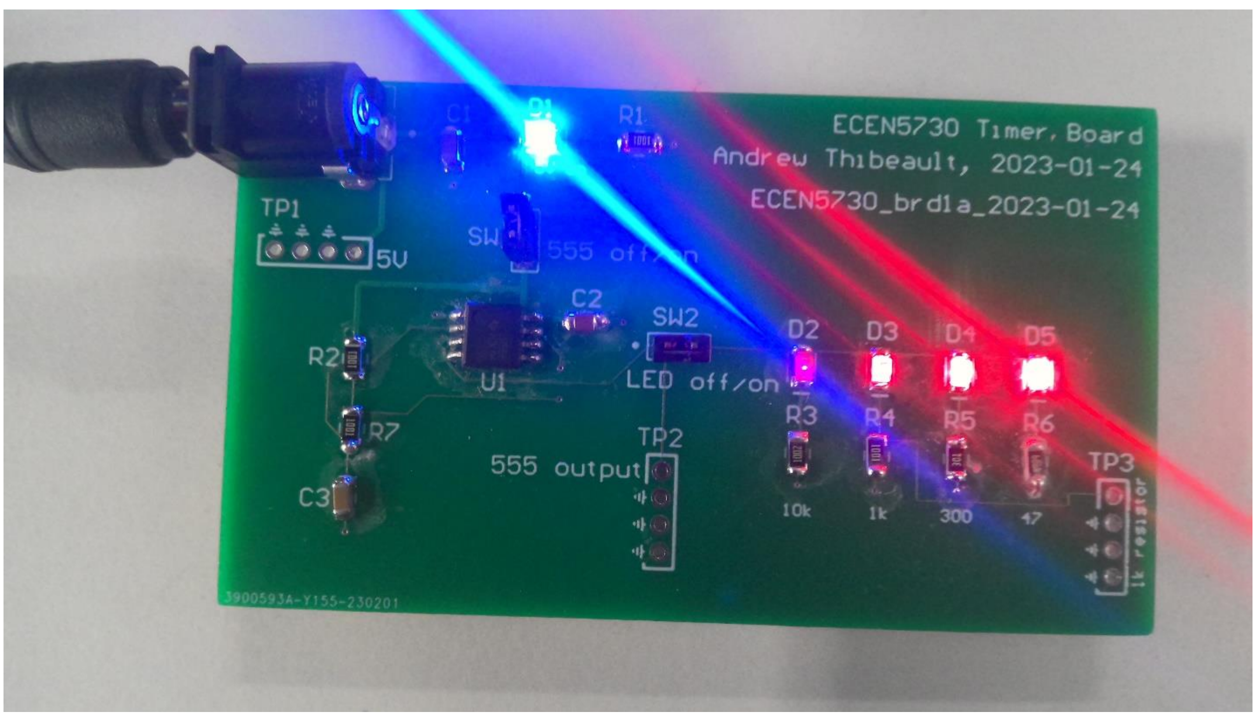

Figure 5: Self-assembled board, powered up

The components provided for the self-assembled board were slightly different than those designated in my schematic. Of note, the test LEDs were red instead of green, which will result in a different forward voltage drop, and a 50 ohm resistor was used in place of the 47 ohm resistor, which would result in more current draw at the same operating conditions.

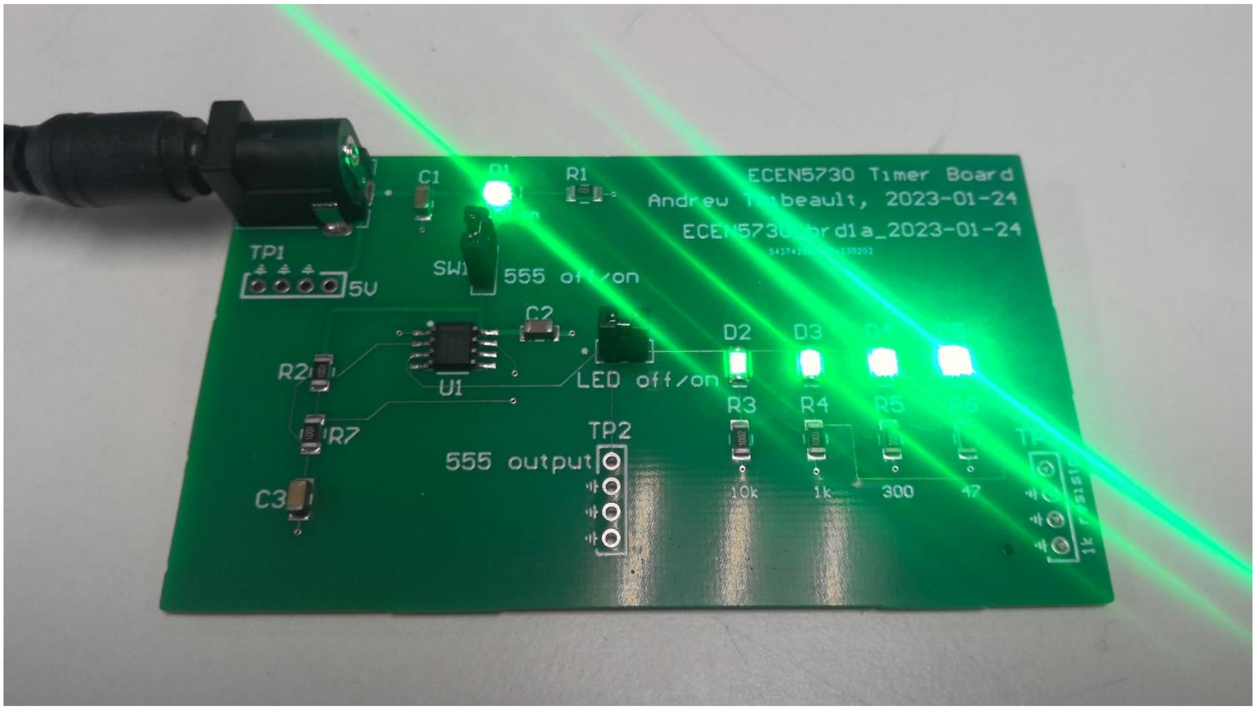

Figure 6: Fab shop-assembled board, powered up

Testing results

Fab shop assembled board:

With all switches off, the 5V indicator LED is lit, and a DC 5.50 V signal is seen at the TP1 (5V).

With SW1 (555 off/on) on, a pulsed voltage signal with 4.95 V amplitude, 65.8% duty cycle, 503 Hz frequency, and 93ns rise time is seen at TP2 (555 output). The frequency is higher than expected, but within the 10% tolerance of C3, the discharge capacitor on the 555. 525 mV of overvoltage is seen on the rise of the 555 pulse, likely due to the response of the BJTs used within the 555.

With SW2 (LED off/on) on, output voltage of the 555 is reduced to 4.08V, due to a Thevenin resistance internal to the 555, and the rise time was reduced to 52 ns. All 4 LEDs light up, increasing in brightness with lower resistance. The voltage across R4 (1k ohm) appears as a pulsed voltage signal with 1.61 V amplitude, and the same frequency as the 555 pulse. A 950 mV overvoltage is seen on the rise of the pulse, and 1.55 V on the fall of the pulse. No switching noise is seen on the power rail.

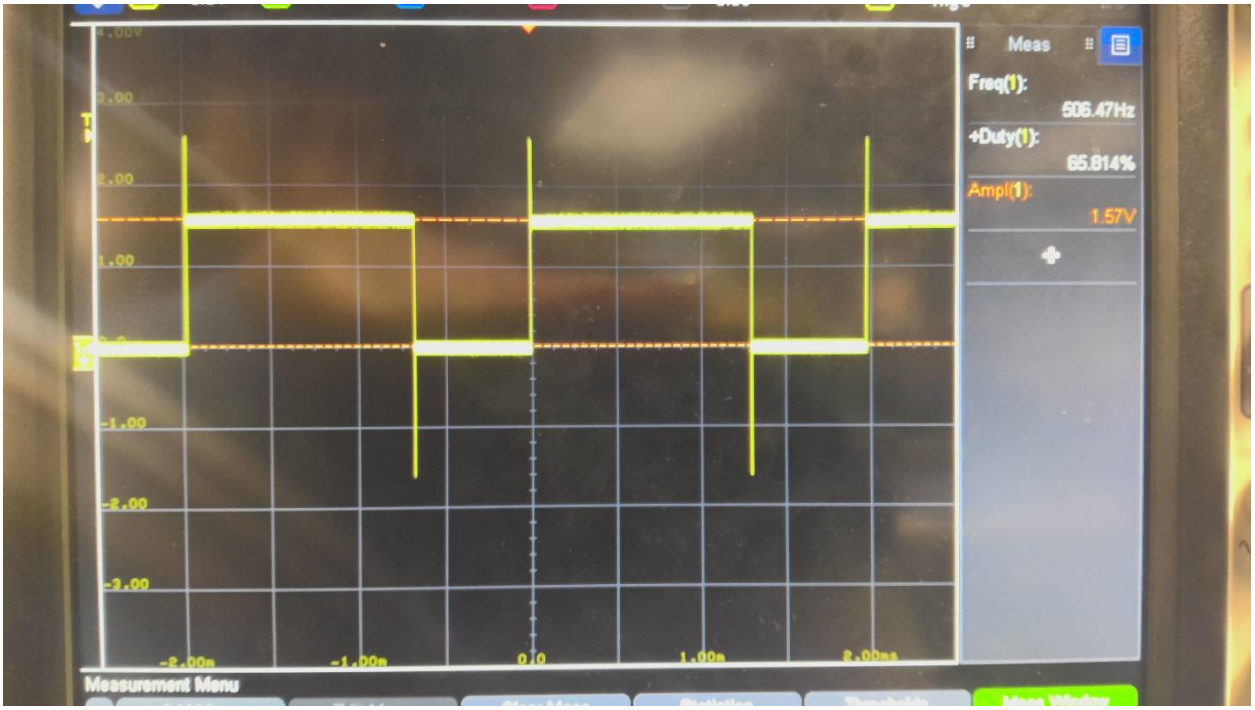

Figure 7: TP3 (1k resistor) voltage waveform

This voltage on the resistor indicates a 1.61 mA current through this 1 kohm resistor, and 2.47 forward voltage drop on the series connected LED. Estimating the same forward voltage drop on the other test LEDs, there should be 34 mA of current through the 47 ohm resistor, 5.37 mA through the 300 ohm resistor, and 0.161 mA through the 10 kohm resistor.

In total, there should be about 41.4 mA of current being drawn from the 555 through the LEDs and series resistors. This is below our expectation of 60 mA, due to the lower peak voltage of the 555 output when loaded. Using this current, the Thevenin resistance of the 555 is estimated to be 21 ohms.

The dimmest green LED is still plenty bright with approximately 0.161 mA of current, and even less could be used to still attain an easily visible LED, likely in the range of 0.01 to 0.1 mA.

Self-assembled board:

The board that I assembled by hand had slightly different results. The 555 output frequency was slightly higher at 520 Hz, and rise time faster at 72 ns. Loaded, the 555 output voltage was slightly lower at 4.02 V, with a rise time of 46 ns.

The red test LEDs were significantly dimmer, with the 10 kohm-connected LED barely perceptible. The voltage across the 1 kohm resistor was higher at 2.34 V, indicating a much smaller forward voltage drop through the diodes of 1.68 V due to the use of the red LEDs in this circuit. The current through the 1 kohm resistor is thus measured to be 2.34 mA. Assuming the same forward voltage drop across the other diodes, 46.8 mA is estimated through the 50 ohm resistor, 7.8 mA through the 300 ohm resistor, and 0.234 mA through the 10 kohm resistor. This results in a total current draw of 57.2 mA.

Finally, we can estimate the Thevenin resistance of the 555 to be 16 ohms. The 5 ohm discrepancy between the resulting Thevenin resistances in the fab-shop-assembled board and the self-assembled board can be attributed to resistor tolerances and differing forward voltage drops in the LEDs across a single board, giving error to the calculation. For more accuracy, the actual resistance of each resistor could be measured, along with the loaded voltage across each resistor.

For a minimum current for the red LEDs, I would use something in the range of the current through the 1 kohm resistor, on the order of 1 mA.

Discussion and Conclusion

This project allowed me to experience the entire PCB design process, from simple block diagram sketch to assembled and working board in hand. I gained experience with practical board planning, defining board features, what it means for the board to work, how to test the board, and identifying potential risk sites all before submitting the final design, in order to have a clear expectation of how my final board should operate, and how it may fail.

Through the use of Altium, I put into practice practical layout design. I learned how to reduce switching noise by pouring and manipulating a continuous ground plane, routing power and signal traces with short lengths and short cross-unders where necessary, and placing decoupling capacitors in close proximity to an IC. I put into practice practical considerations to reduce board cost from the fab vendor, such as using basic parts where possible, meeting the fab shop's basic design rules like minimum line widths, maximum board size, and including a keepout region. I also learned how to label a board effectively for ease of assembly and debugging. Proper labeling was integral to smooth and informed assembly and testing in this board. Finally, I learned how to check for errors in my layout and generate the proper design files for the fab vendor.

One error I made in the layout design was that I redefined the designators for the LEDs, which resulted in LEDs not being assembled by the shop on my assembled board. I was able to assemble these myself without issue.

Overall, both the fab-shop-assembled board and the self-assembled board worked as expected. The assembly of the bare board went smoothly. Using my predefined testing plan and working conditions, testing was a well defined process, and working conditions were clear to achieve.

One other discrepancy from my ideal planned board was that I was provided different LEDs than I had designed my board for. This resulted in a higher overall current output from the 555, but this did not present a problem, as it was well within the rated maximum output current. In the future, I will ensure to select the designed-for components for my self-assembled boards to avoid issues.

Switching noise was nonexistent on this board, likely due to using the slow 555 timer (a slow rise time means small dI/dt, thus less inductive noise), as well as implementing good layout practices as previously discussed.

All in all, I learned and began to solidify as habits good design practices that I will implement on future boards, and I made a few small errors that I know now to avoid in the future.