Grant Novota
Chris Choi
Board 1 Final Report
10/1/19

Final Report: LED 555 Timer

1. **Plan of Record**
    a. Purpose
        The purpose of this board is to use a 555 timer and hex inverter to drive current through red LEDs. The power for the board comes in through a 5V rail (power jack or USB input) and feeds a 555 timer with 50% duty cycle, 1 KHz frequency. The 555 timer output drives 4 red LEDs in parallel with series resistors of differing values ranging from 50 ohms to 10K ohms. The 5V rail feeds a 3.3V LDO, which then feeds a hex inverter. The hex inverter is connected to the 555 timer output and drives 3 red LEDs. The red LEDs will show different brightnesses as they will be drawing different currents. This will give us a general idea for the minimum drive current for the red LEDs and the relative brightness related to the current.
    b. Power Budget
        LEDs driven by the 555 output will consume 75 mA of output current, within the spec'd value of 100 mA drive current by the 555 timer. The red LEDs will have a forward voltage drop of about 1.8V. When the LEDs are being driven by the 5V 555 Timer, they will drive a maximum of 64 milliamps (assuming the smallest resistor value of 50 ohms). This value is calculated for a single red LED.
        When the LEDs are being driven by the 3.3V from the Hex inverter, they will drive a maximum of 30 milliamps (assuming the smallest resistor value of 50 ohms). This value is calculated for a single red LED.
        The LEDs driven by the hex inverter will consume 90 mA of output current, under the 100 mA maximum by the hex inverter. The worst case current consumed by the components is ~200 mA. Therefore the LDO should be sourced to drive 200 mA.
    c. Risk Management
        i. Potential Risks
            1. Receiving faulty components is a potential risk that will be mitigated by ordering extras.
            2. Miss-orienting the 555 timer could result in incorrect connections to the pins. We will check for the correct orientation before soldering it in. This will also be the case for the LEDs.
            3. Ensuring the right amount of voltage is applied to the circuit and the current limit is not too low. This can be mitigated by ensuring we apply a consistent 5V and 1A current limit.

4. When soldering there is the potential for cold solder joints. We can mitigate this by inspecting each connection after the joint is formed.

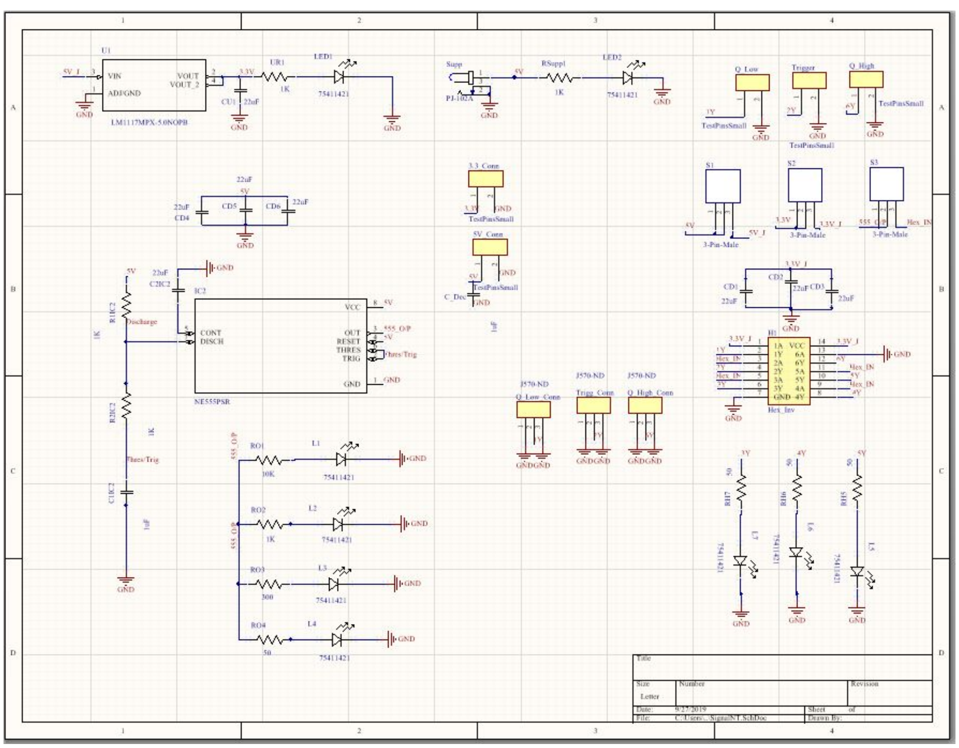
ii. Bring-Up Plan

We will consider this after we look at how we will measure noise and see some examples of features we can add to aid in the bring-up and test. Noise can be modified by adding a filtering capacitor.

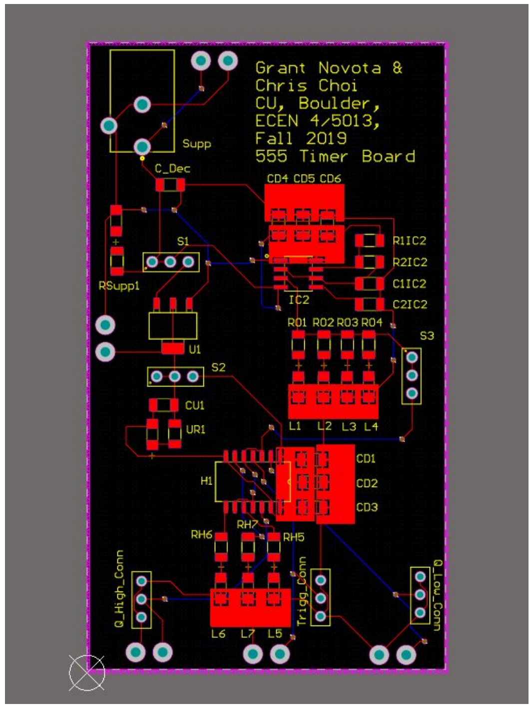
2. Bill of Materials

Index	Quantity	Part Number	Manufacturer Part Number	Description	Customer Reference	Available	Backorder	Unit Price	Extended Price USD
1	2	296-14635-1-ND	SA555DR	IC OSC SGL TIMER 100KHZ 8-SOIC		2	0	0.4	0.80
2	10	754-1142-1-ND	APT3216SRCPRV	LED RED CLEAR CHIP SMD		10	0	0.312	3.12
3	2	AZ1117CH-3.3TRG1DICT-ND	AZ1117CH-3.3TRG1	IC REG LINEAR 3.3V 800MA SOT223		2	0	0.38	0.76
4	2	296-28498-1-ND	SN74ALVC14NSR	IC INVERTER SCHMITT 6CH 14SOP		2	0	0.61	1.22
5	10	490-10476-1-ND	GRM188R61A226ME15D	CAP CER 22UF 10V X5R 0603		10	0	0.215	2.15
6	5	YAG1234CT-ND	RT0402FRE071KL	RES SMD 1K OHM 1% 1/16W 0402		5	0	0.1	0.50
7	7	SM6227FT50R0CT-ND	SM6227FT50R0	RES 50 OHM 1% 3W 6227		7	0	1.35	9.45
Total	**38**					**38**	**0**	**3.367**	**18.00**

3. Schematic Capture

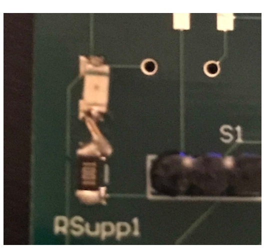


4. Board Layout



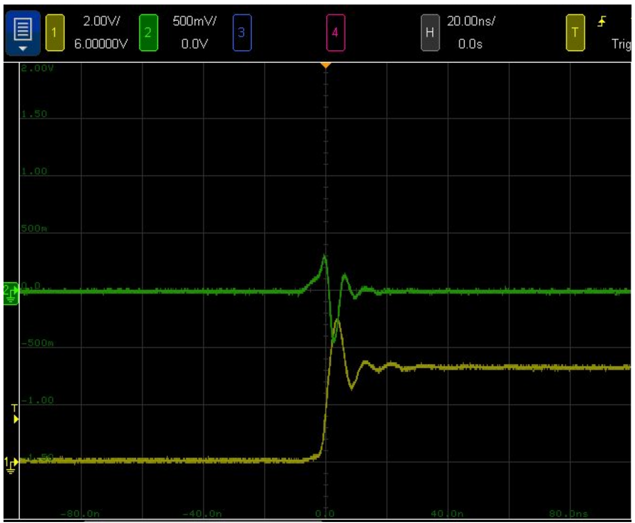
5. Documentation

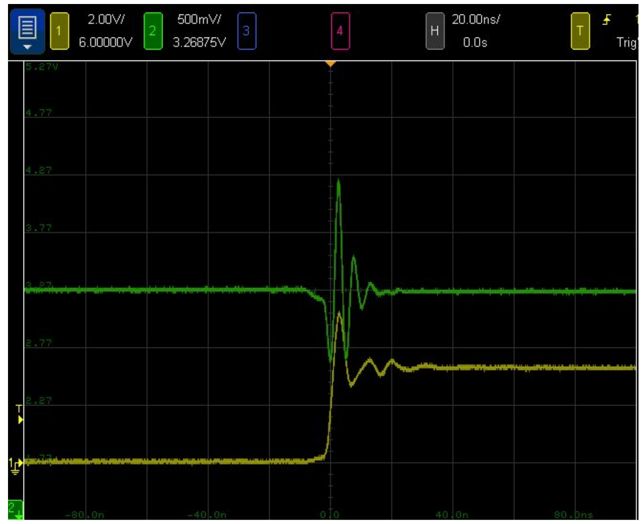
When assembling the board it is important to be methodical and test each subcircuit as soon as it is completed. We began by attaching power and the power indicator LED, then tested it to see if it would work as expected. The LED did not turn on, which indicated an open circuit. We found that a trace that was supposed to connect the LED to the subcircuit was missing. This was due to an error in the Altium schematic, where a ghost wire was not present to indicate that a trace needed to be placed to connect the LED. A jumper connection was made to solve this issue, which can be seen in the image below.



Next, we reconnected the board to the power supply and noticed the current quickly ramping up, which indicated a short circuit. After confirming the short with a multimeter, we investigated the issue by tracing the path around the circuit and viewing the pcb schematic. We found that a ground trace was touching the 5V pad; another error in the

Altium schematic. This was a difficult problem to fix that would require carving away the connections with a knife, so we were given a new board without these errors.
The subcircuit was reassembled and re-tested, resulting in the LED turning on to indicate that there were no errors.
Moving forward, we assembled the next subcircuit. After final assembly and taking scope shots, we compared the two results as shown below.





The right image to the left shows the trigger before adding the decoupling capacitors. We expected to see the switching noise drop upon adding the capacitors. The signal is being triggered on the rise of the 555 output, and the difference between the noise on the right image is significantly larger than the left image. The left image includes the use of decoupling capacitors. We are using the 100 kHz 555 calculated from the full on and off duty cycle of the chip. Fromi analyzing the images to the left, it can be estimated, the rise time is very small, approximately five nanoseconds