Lab report – ECE 1895 – Design Project1

Rayan Hassan – 4511021

Originally intended circuit: Wailing Siren Circuit

Originally I was going with a Wailing Siren Circuit, but all the designs I tried on my breadboard didn't work. My modification was simply adding an LED in parallel with the speaker.

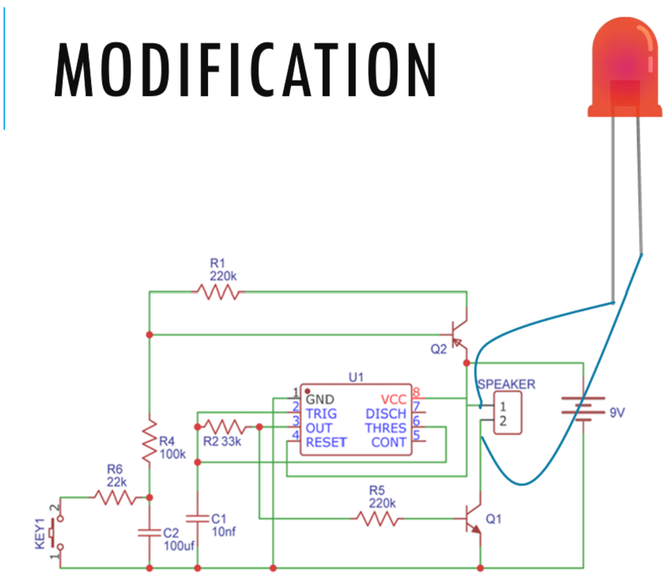


Figure A: Wailing Siren Circuit

Initially, when the key is not pressed, capacitor C2 is charged through R1 and R4. So the voltage across the base of transistor Q2 is positive, therefore it is OFF and speaker is OFF. When the button is pressed, C2 begins discharging through R6. So the voltage across Q2 is negative, therefore it is ON and we can hear the sound of the siren. The 555 timer is used in astable mode.

However, after I implemented it for the first time on my breadboard, it didn't work correctly. I used LTSpice to see what the problem was. Here are the schematics and output waveforms obtained:

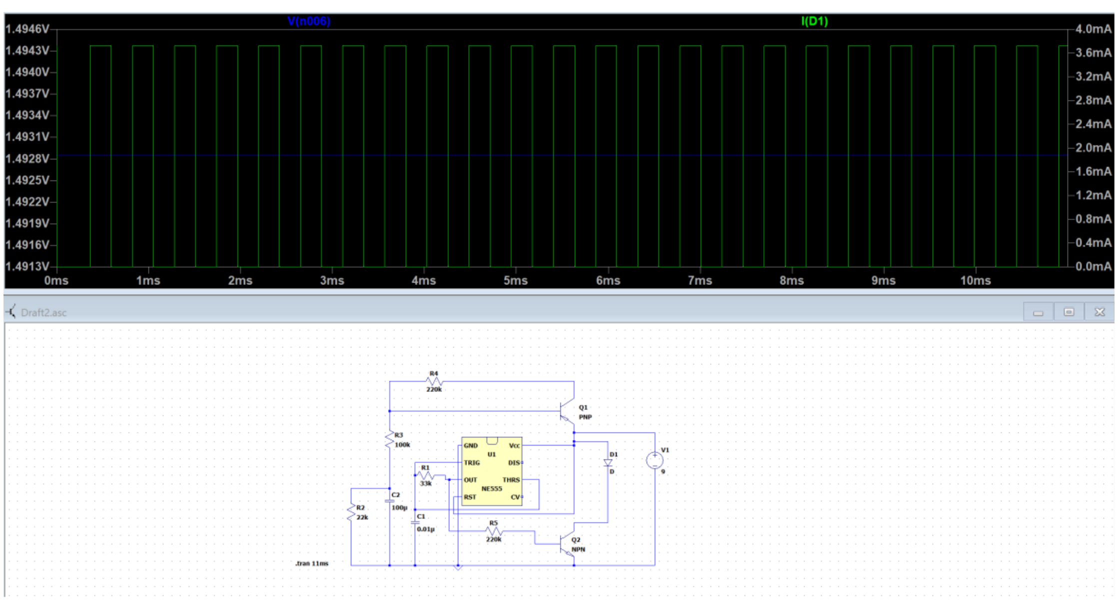


Figure B: Schematics & output waveform when Key is pressed

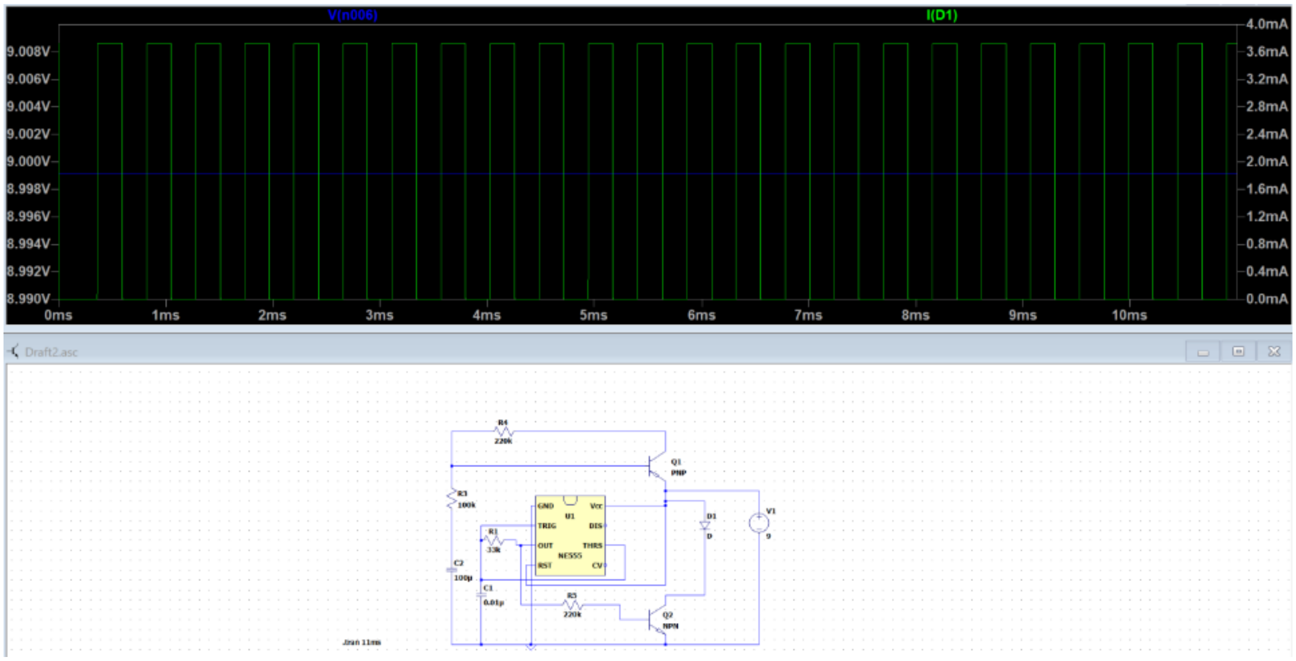


Figure C: Schematics & output waveform when Key is not pressed

The first figure is when the Key (push button) is pressed (R2=22k discharges the capacitor) and the second figure is when it is not (capacitor charging, no R2).

The current I(D1) through the diode represents the current through the speaker. The output is not really changing whether we press the button (with R2=22k) or we don't (without R2). In fact, when I did it on the breadboard, the speaker was ON all the time, which is not what we want. In fact, after we press the button, the capacitor is supposed to discharged and so the speaker should turn OFF gradually.

At that point, I figured that there was something wrong with the circuit itself, so I tried different implementations on my breadboard. However, none of them worked. The LED and the speaker did work, but incorrectly. Instead of decreasing in intensity, they kept going on the same one. Also, it didn't make a difference whether my switch was on or off, they would keep on emitting the same sound and light. Here are all the implementations I tried:

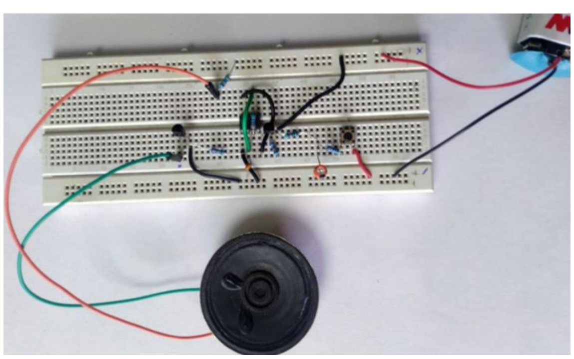


Figure D: Implementation from
https://circuitdigest.com/electronic-circuits/building-wailing-siren-circuit-using-555-timer-ic

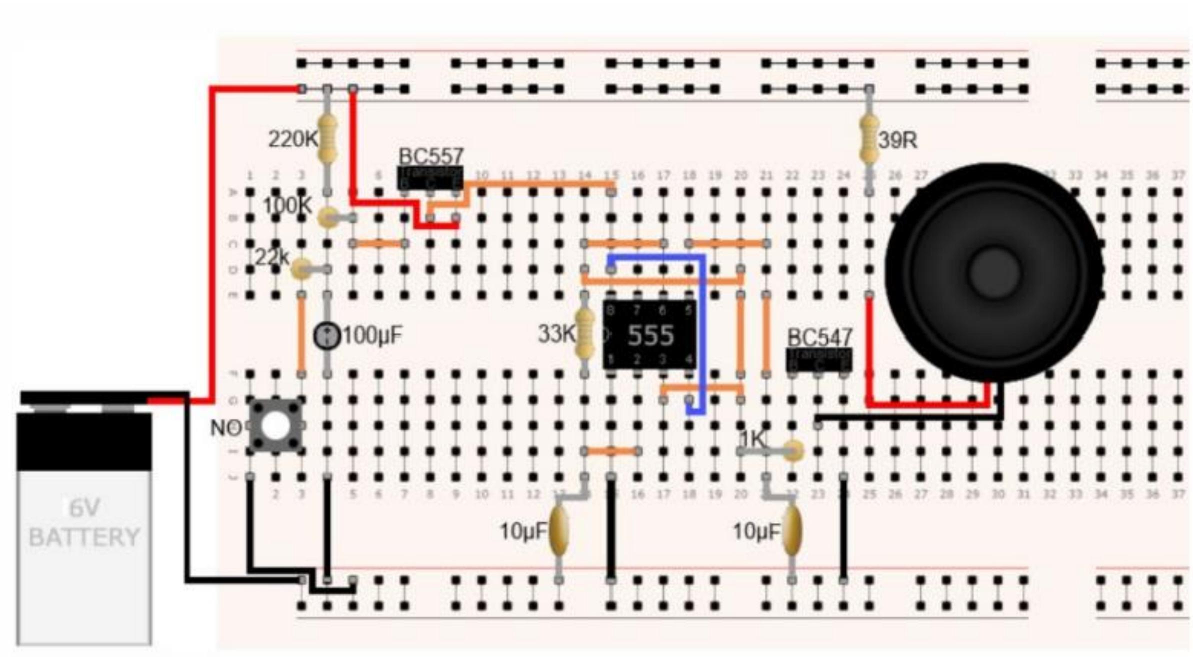


Figure E: Implementation from https://elonics.org/wailing-siren-circuit-using-555-ic/

NB: I didn't know I had to take pictures of the designs on my breadboard back then so I put the pictures of the designs I followed from the websites I visited. I implemented the exact designs on my breadboard.

None of these implementations worked so I changed my project to DingDong Generator circuit.

Actual circuit: DingDong generator circuit

Design overview

I chose a DingDong sound generator circuit. The circuit is shown below.

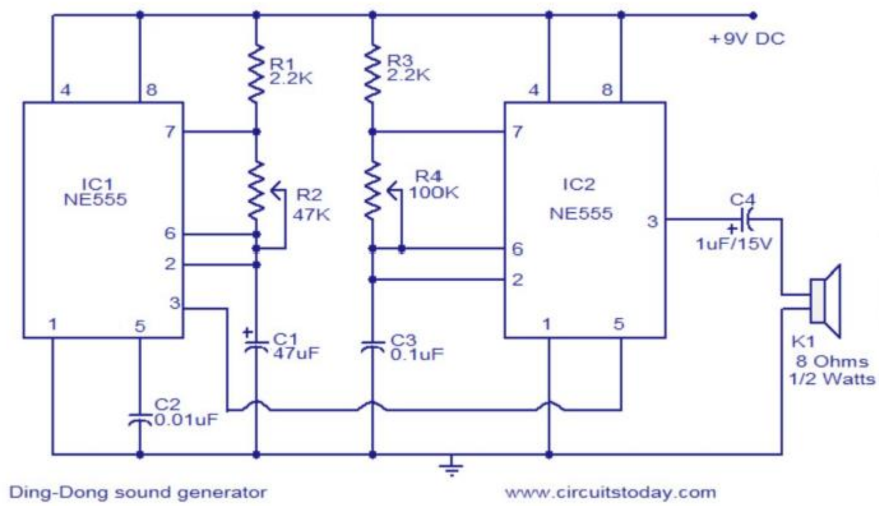


Figure F: DingDong sound generator circuit

https://www.circuitstoday.com/ding-dong-sound-generator

The circuit operates in the astable mode when it is ON. The capacitors C1 and C3 start charging via R1 and R3. After a certain time (after voltage across the timer reaches 2/3VCC), they start discharging through R2 and R4. Then, after the voltage across the 555 timer reaches 1/3VCC, the capacitors start charging again. This process is continuously repeated producing the pulse of 1Hz frequency which emits the ding dong sound. C2 is there to control the time delay with which the waveform is produced.

My modification is a simple LED in parallel with the speaker, so that we can visually see how it lights up and changes in intensity as the speaker emits the ding dong sound.

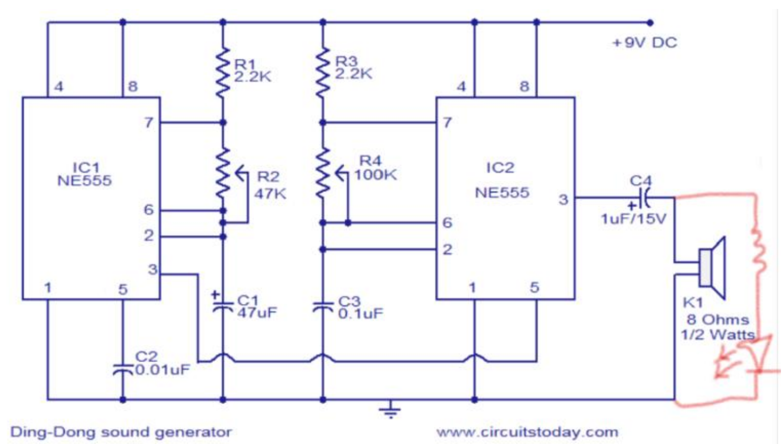


Figure G: DingDong generator sound circuit with modification

Design Verification

Here are the output waveforms and schematics on LTSpice. I(D1) is the current through the speaker and I(D2) is the current through the LED. V(n011) is the voltage across the speaker and V(n012) is the voltage across the LED

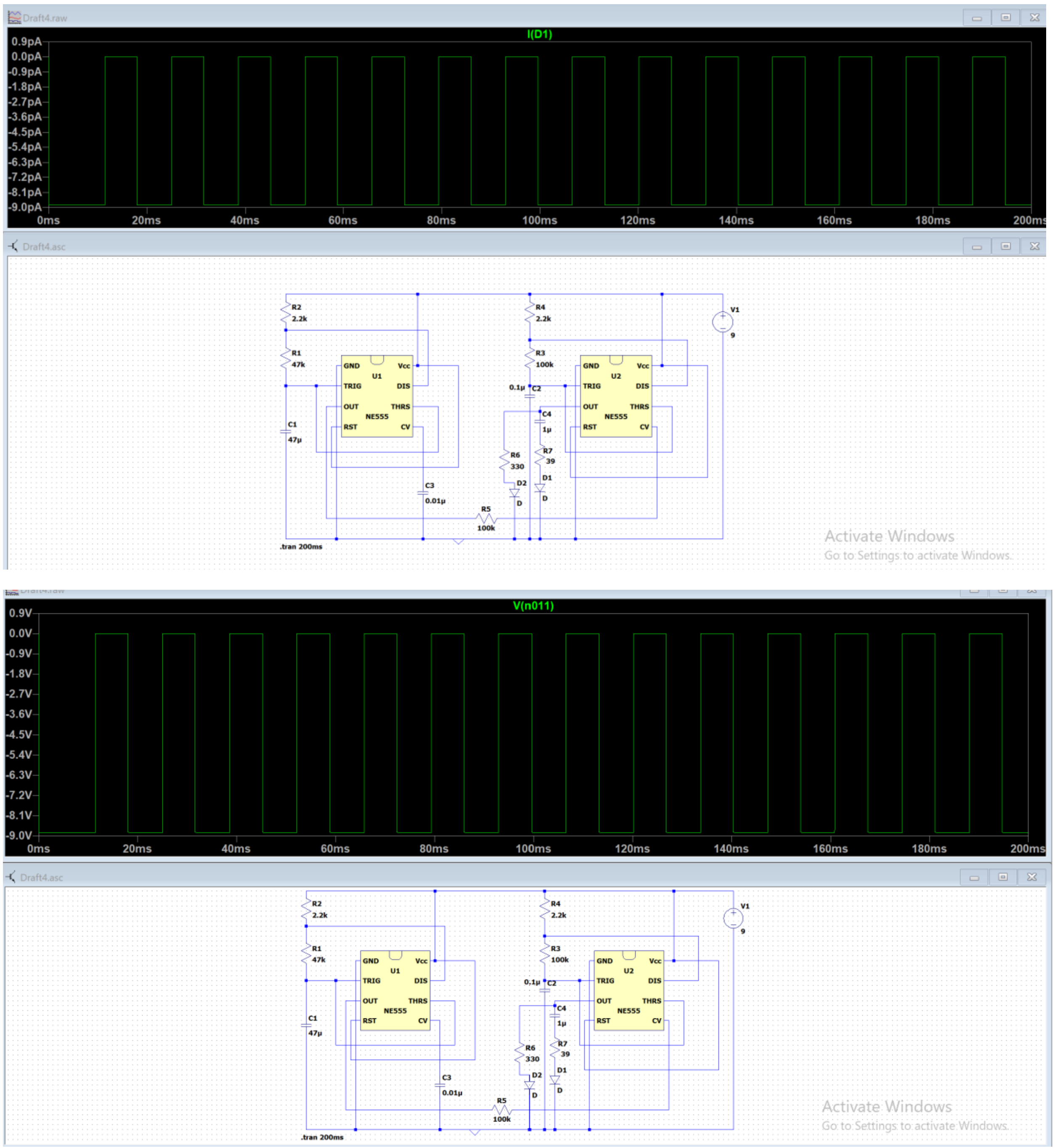


Figure H: Schematics and output current flowing in the speaker and the voltage across it

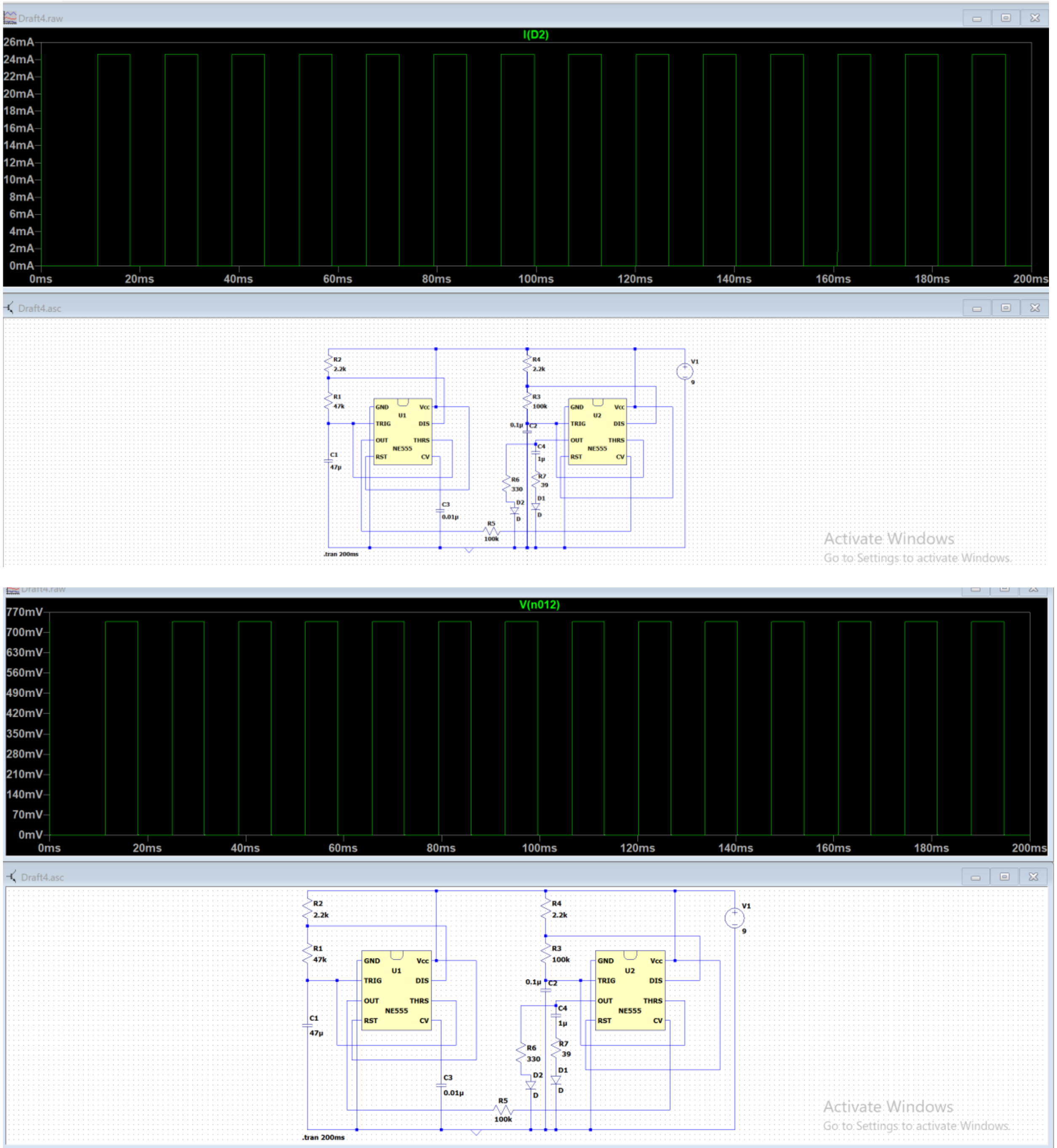


Figure I: Schematics and output current flowing through the LED and the voltage across it.

These output waveforms made perfect sense to me since as the capacitors charge and discharge, the current through the speaker and the LED, as well as their respective voltages, change according. This would result in the sound and light altering between high and low.

To represent my speaker and the LED on the schematics, I simply used diodes D1 and D2 respectively, of course in series with small resistors. To simulate it I used transient analysis because I want to monitor how the output currents change with time. I ran the simulation for 200ms which was enough to see the pattern and visualize the output. My voltage source was in DC mode as required by the circuit parameters.

I also tried it on my breadboard, and it worked. Here is an annotated picture:

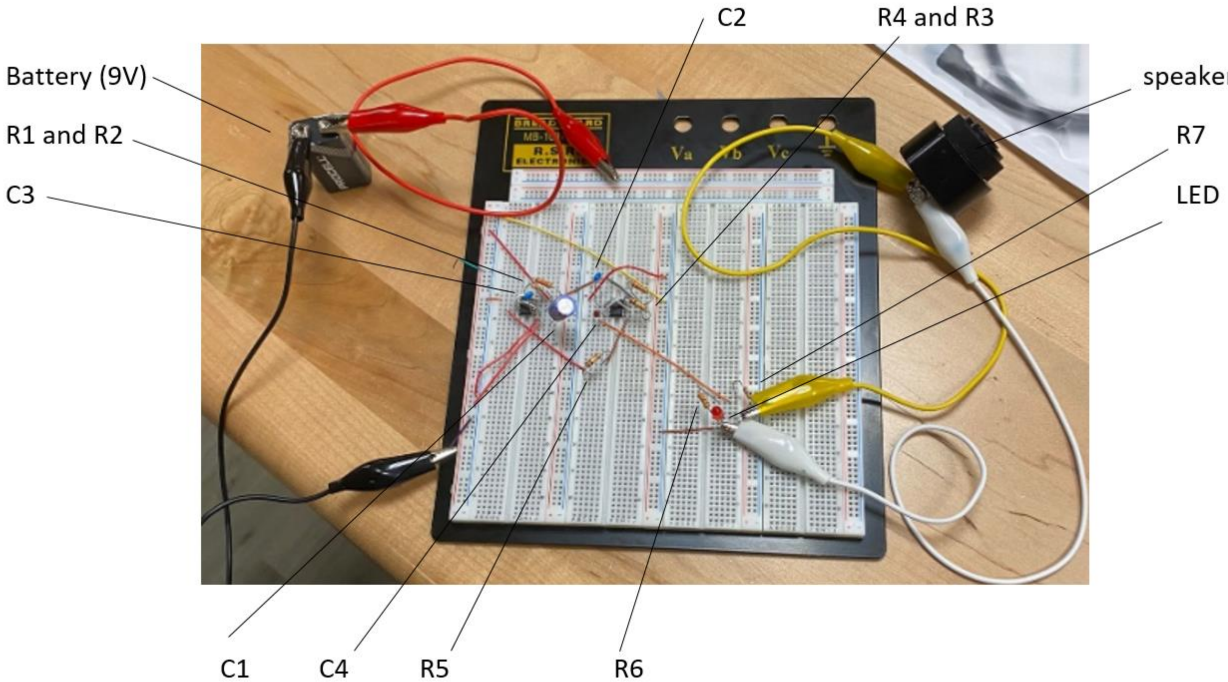


Figure J: Breadboard implementation of the DingDong Circuit

Design Implementation

My PCB schematics and layout on Altium are shown below.

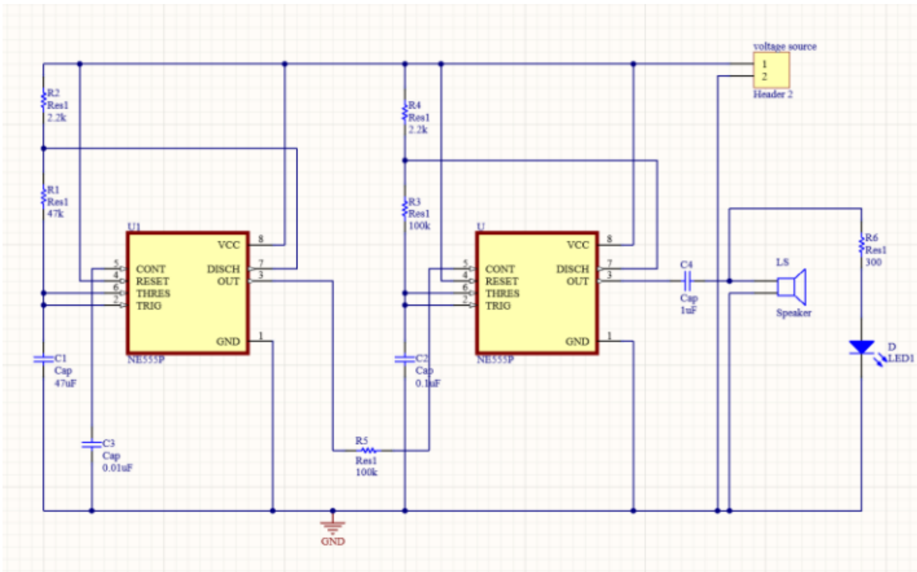


Figure K: PCB Schematics of DingDong circuit on Altium

In my schematics, I was able to find a loud speaker (LS) instead of adding one diode and a resistor for the speaker. Also, I used an LED with a 300 Ohms resistor. For my voltage source, I simply used a connector (Header 2).

The first time I tried verifying my board on oshpark, the preview of the board gave me questionable results.

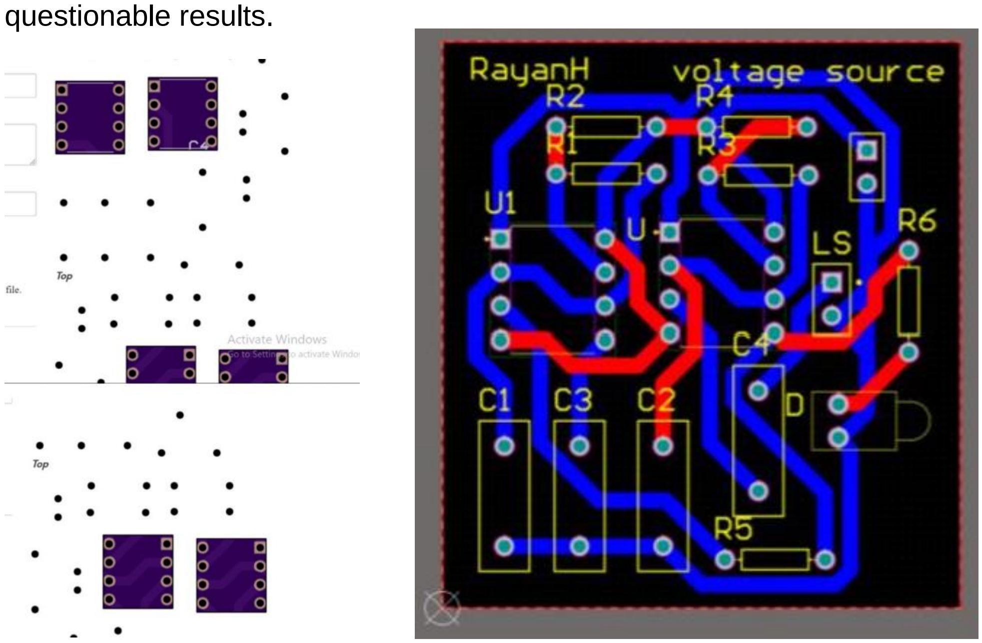


Figure L: Error in oshpark verification and corresponding PCB layout

After I spent hours trying to figure out what the problem was, I realised that my board outline was in the wrong layer. Actually, I had set it up correctly the first time but I ended up deleting my whole PCB layout and starting again since I didn't like my original design. That is why I forgot to draw my outline in the Keep-out layer the second time. Anyway, after I fixed it, I also notice that my name was written in the Top Overlay layer on the board (in yellow), and therefore wasn't going to appear on the actual one. So I changed that and I put in on the Top layer. My final, correct PCB layout is shown below.

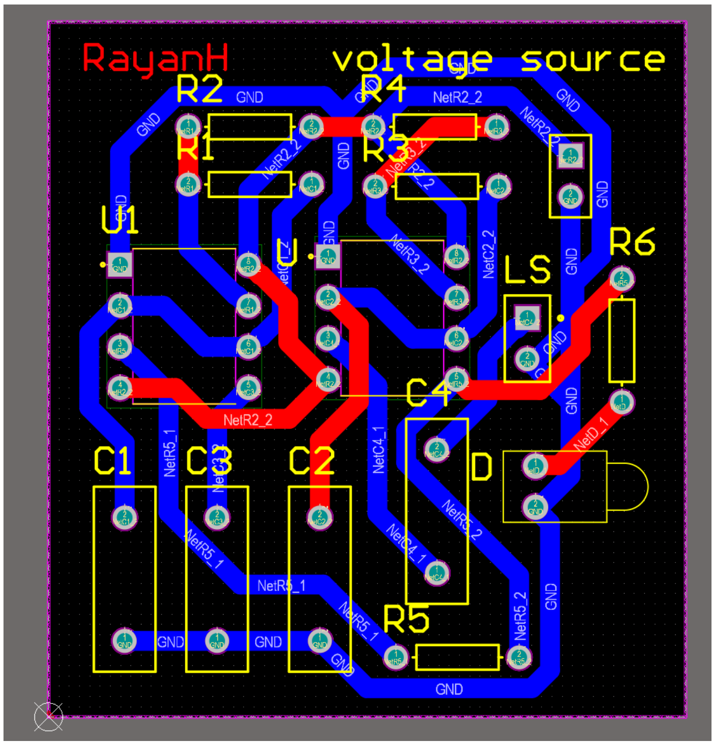


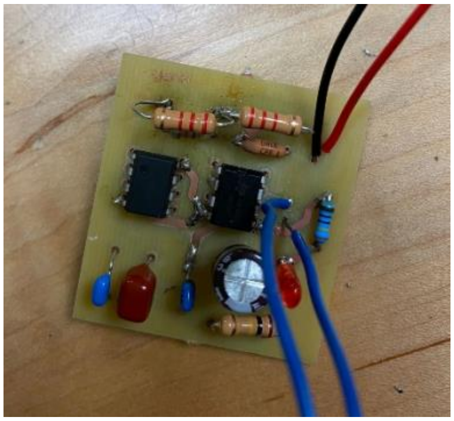
Figure M: Final PCB layout

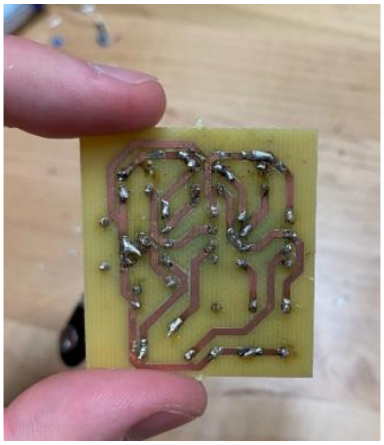
When it comes to my PCB design, I tried to keep the similar components in the same direction (expect for R6) that way it's easier to deal with. I also placed all edge components first (mainly the voltage source, the LED, speaker…) which just made it easy to add the traces later on. I didn't overlap any part. Also, I tried to keep most of the parts on one layer just to make it simpler for placing the components. It doesn't matter in my case since I didn't use any vias, but it's still easier to have most of them on one side for the soldering later on (specifically, on the bottom layer to avoid burning any components with the soldering iron while soldering on the top). Finally, I designed my PCB like my schematics (not exactly, but for instance R1,R2,R3 and R4 are on the top, the capacitors and R5 on the bottom, the LED, speaker, voltage source and C4 on the right…). It just made it easier to follow. I chose through hole components. I only got silk to solder mask clearance and minimum angular ring errors.

For my measurements I used 40mil for hole size and 60mil for my traces. That was enough for me to solder it later on. My board dimensions were 39.5mmX43mm.

Design manufacturing and Assembly

As mentioned previously, I had all through hole components. Soldering wasn't that complicated for me since my traces and holes were big enough. Here are a few pictures showing my soldered board.





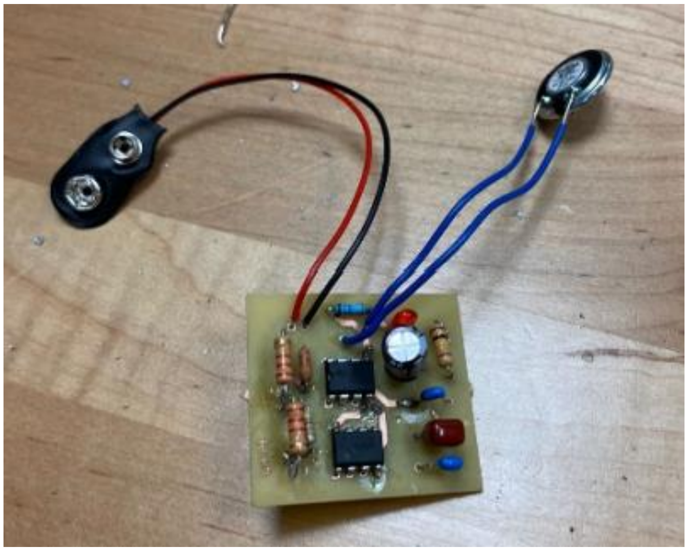


Figure N: Board from different views

(Top and Bottom layers)

I faced a few challenges when soldering my board though. Some of traces ripped off so I had to solder bridge them. That was a bit difficult in the beginning but I was able to do it successfully.

Also, it was hard for me to solder the 555 timer on, since the traces that connect pin 4 of U1 to pin 4 of U, and pin 2 of U to the positive terminal of the capacitor C2 were all on the top layer. I realised that was a mistake I made when designing my PCB layout since the tip of the soldering iron was too big to fit in between the pins. I actually ended up damaging one 555 timer and replaced by another one. At the end, I was able to solder the whole board, but I should have thought of these few details before.

Design testing

My board didn't work in the beginning. I wasn't getting any output (the LED wasn't lighting up and the speaker was OFF). The first thing I did was a connectivity test using a multimeter to see if all my components are connected. I realised that the 2.2k resistors weren't connected, so I made sure to correct that:

Figure O: Side view of PCB showing connection between 2.2k resistors

After that I tried it again, it worked, but incorrectly. It was giving one continuous sound and a fixed light form the LED at the same frequency (it wasn't changing like it is supposed to). I checked my 555 timers again using the oscilloscope and ran the connectivity check multiple times. My timers were working properly; the picture below shows the output of one of them.

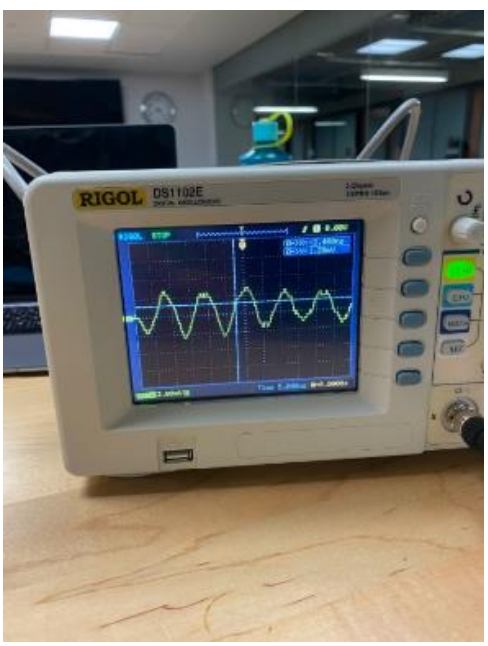


Figure P: Oscilloscope showing output waveform of one of the 555 timers (connection from pin3 to ground)

After that, I noticed that some of traces were not really connected so I had to solder on them just to make sure everything is working. I tried it again, and it still didn't work. So I thought that maybe there is something wrong with the components or their values. Specifically, I thought that the sound is not changing because the period is very small. In other words, I wasn't able to hear when the frequencies are changing because the output changed from high to low very quickly. To check that, I used LTSpice to simulate my circuit using different values of resistors and capacitors, and checking how the output changes (if the period gets larger).

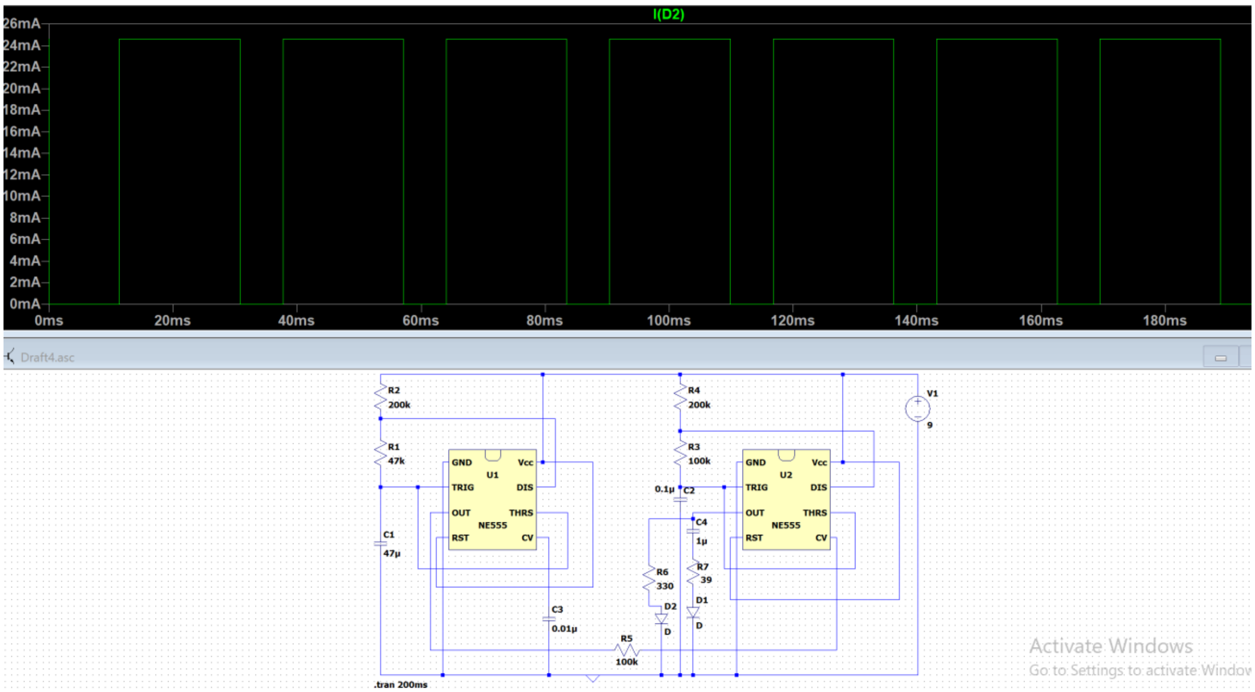


Figure Q: Schematics and output waveform if R2 and R4 values are increased

The figure above shows that if I increase the values of the 2.2k resistors, the period increases. So after I did that, I unsoldered my 2.2k resistors and replaced them by 100k resistors. I tried my board again and it still didn't work. Then, I figured that changing the values of these resistors actually reduces the duty cycle, which is bad because then I won't be able to clearly hear the difference in frequencies of the speaker and LED as they alter from high to low.

Therefore, I tried to think of a way to increase the period without changing the duty cycle. I figured that Pin 2 of the 555 timer (trigger) is responsible for the start time. So when the voltage across it drops to 1/3VCC the timer starts and therefore the output goes high. When the voltage reaches 2/3VCC, the timing cycle ends thanks to the threshold pin (pin 6) which is connected to pin 2 for that purpose. So I thought maybe if I changed the value of the capacitor C2, which is connected to pin 2 and pin 6, the period might change without impacting the duty cycle. So here is the output waveforms I got when I increased the value of C2.

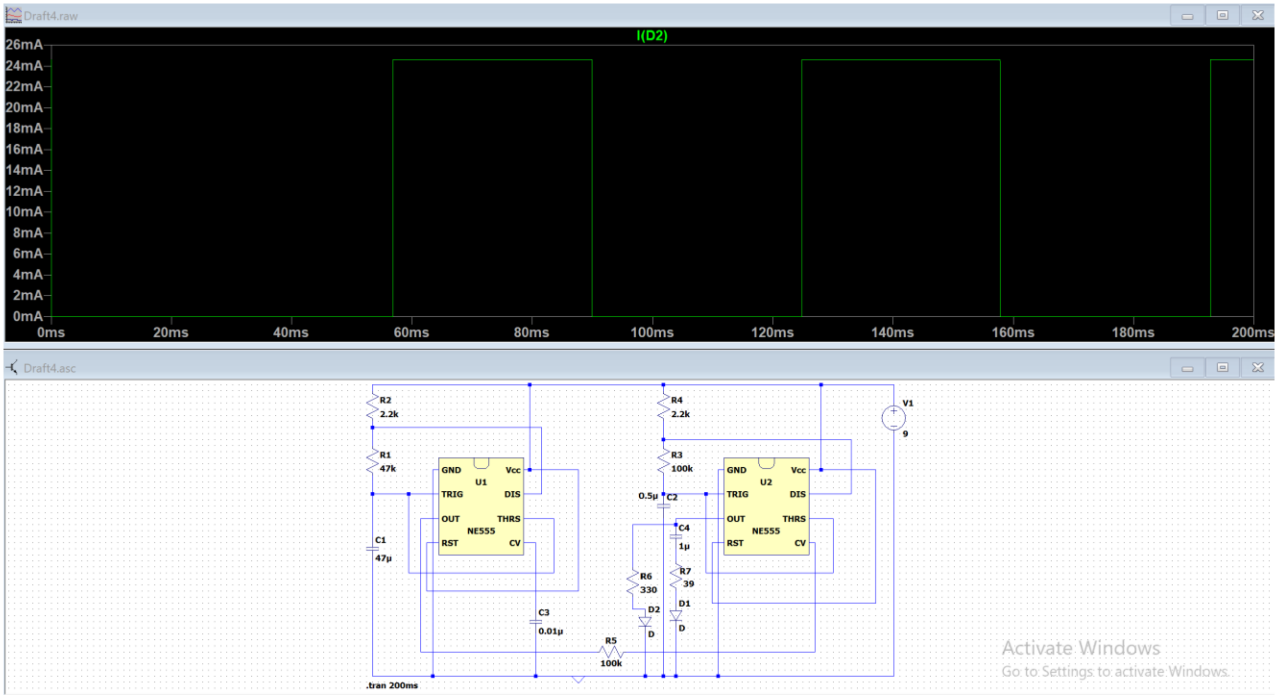


Figure R: Schematics and output waveform if C2 value is increased

This looked like it would work so I tried it on my board. I used a 0.47uF capacitor for C2, however; it still didn't work correctly and I was getting the same output as before. There was no change in the frequency, hence no change in the sound of the speaker nor the LED.

After I tried all these methods to remedy this issue, I thought that the problem might not be in the components and the design itself, but in the soldering. Therefore, I submitted my board to get printed again and soldered everything from scratch. Here is my newest board:

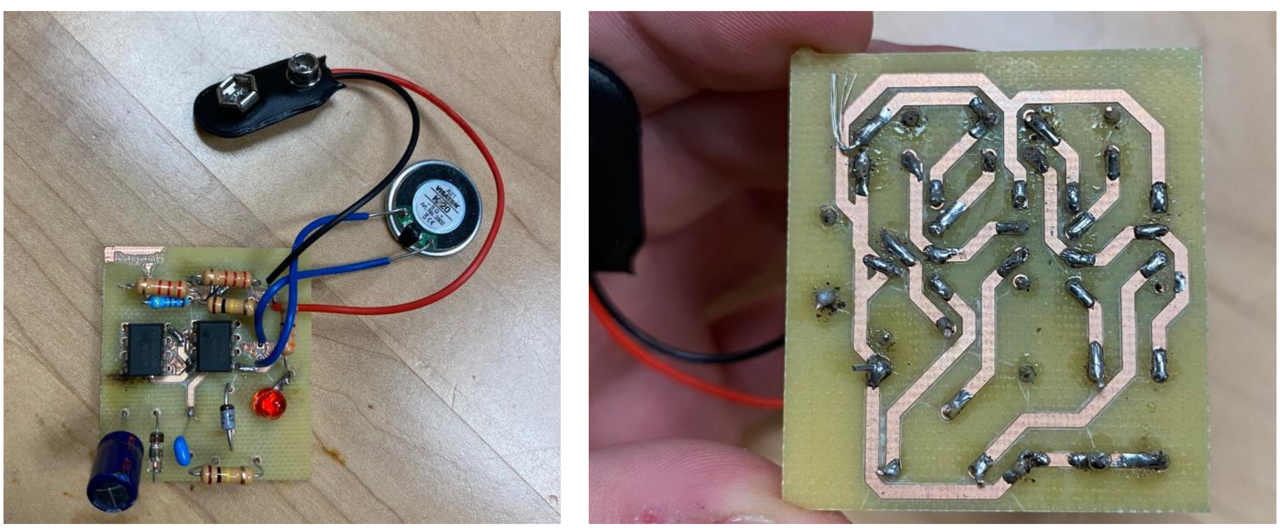


Figure S: New PCB soldered from scratch

Unfortunately, it still did not work. First, I checked if my timers were working properly. However, the second one gave me an output waveform that didn't seem right:

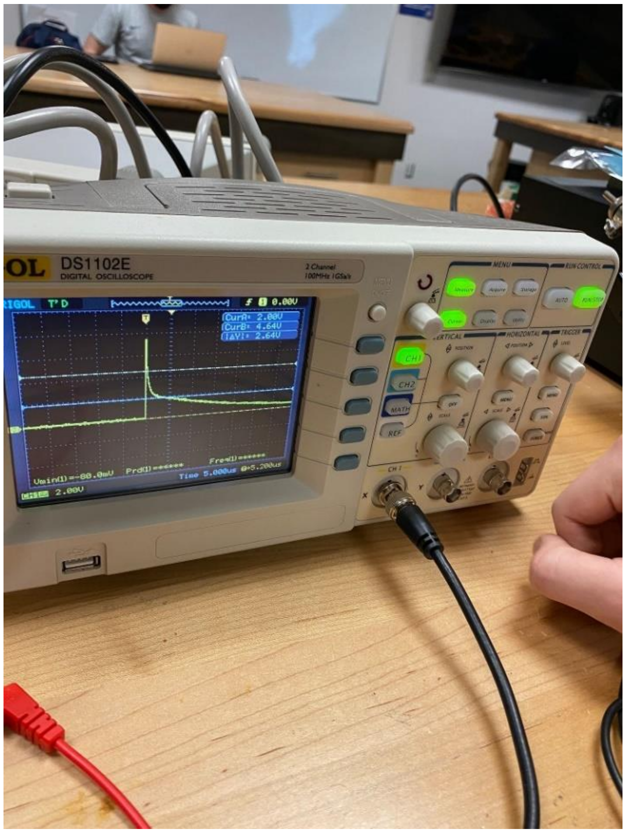


Figure T: Oscilloscope showing the output waveform of one of the 555 timers on the new PCB

I ran a connectivity check and realised that one of my 555 timers had one pin that was disconnected on the top layer. Therefore, I had to remove it and apply solder accordingly. Then, I checked again to see if the problem was solved, and indeed it was. However, I was still getting the same output. My only resort at this point was to change my components (use other capacitors, LED, resistors). I unsoldered my old components and replaced them by new ones. However, it still didn't work. I got the same response: a continuous noise and one fixed light from the LED. At that point I didn't have more time to figure it out, so I couldn't try more solutions to remedy that.

Here is a link that contains a video that shows how the DingDong sound generator circuit should work.

https://www.electronicshub.org/ding-dong-sound-generator-circuit/