Thermistor Controlled LED Blinker PCB

Personal Project Report

Yil Verdeja

TABLE OF CONTENTS

LIST OF FIGURES

1 INTRODUCTION

In this project, I created a thermistor controlled light-emitting diode (LED) blinker printed circuit board (PCB) using the DipTrace schematic capture and PCB layout tool. Upon getting the manufactured PCBs, using SMT components ordered from DigiKey, I assembled the board by soldering each individual SMT component.

1.1 Purpose

The purpose of this project was to design, build and assemble a PCB and become more acquainted to schematic capture and PCB layout tools. Pursuing a simple project like this would not only give me satisfaction, but it would also allow me to complete more complicated hardware design projects in the future.

In order to complete the task, I decided to build a *simple* analog circuit that consisted of a thermistor, a 555 timer and an LED. By implementing a simple design, I could achieve my goals in the project.

2 METHODOLOGY

This section describes the process of designing, building and assembling the PCB.

2.1 Schematic Capture and Circuit Analysis

The process starts by designing the *simple* circuit. As mentioned, the circuit consists of a thermistor, a 555 timer and an LED. The 555 timer is an integrated circuit (IC) chip that is commonly used as an oscillator, a flip-flop or to provide time delays. For this circuit, the 555 timer was used to create a free running astable oscillator to continuously produce square wave pulses. It was desired to make the LED blink on and off with a changing duty cycle.

2.1.1 Main Concepts

In the figure below, a basic astable 555 oscillator circuit is shown. A more thorough explanation of the functionality of this circuit is provided in the following link.

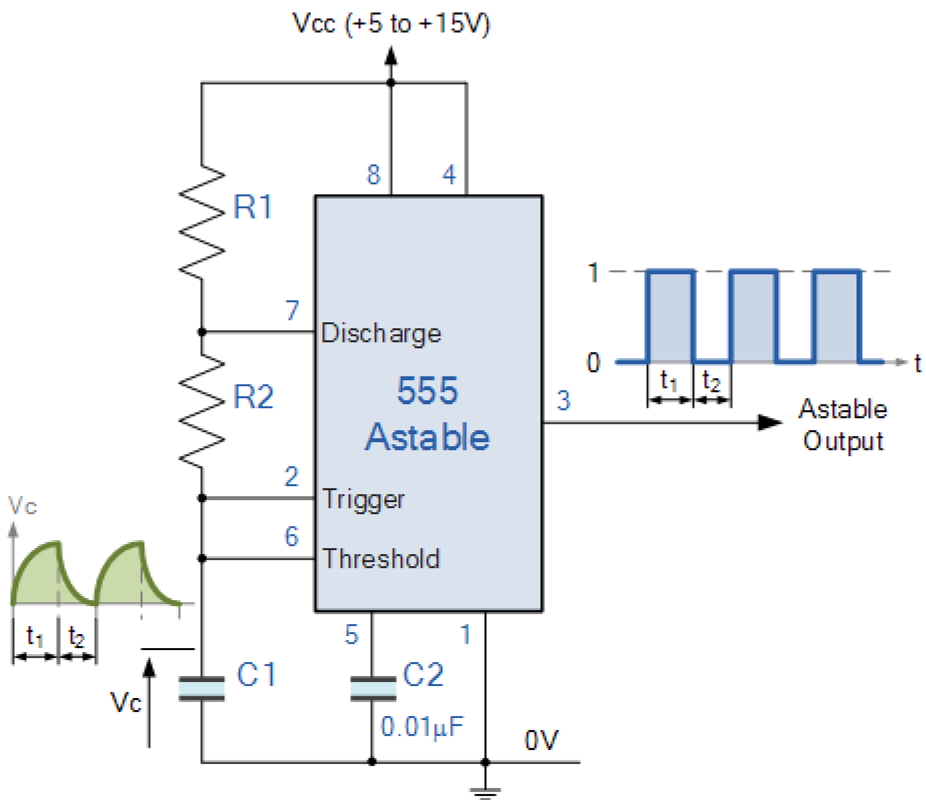

Figure 1. Basic astable 555 oscillator circuit[1]

The time ON t_1 and the time OFF t_2 are determined by the equations shown below:

$$t_1 = 0.693(R_1 + R_2) \cdot C \quad \text{Eq 1}$$

$$t_2 = 0.693 \cdot R_2 \cdot C \quad \text{Eq 2}$$

As shown, the ON time is dependent on both R_1 and R_2 while the OFF time is only dependent on R_2. In order to make the ON time only dependent on R_1 two bypassing diodes are introduced into the circuit. This controls the paths of the charging current and discharging current through the resistors and capacitors as shown in Figure 2.

[1] https://www.electronics-tutorials.ws/waveforms/555_oscillator.html

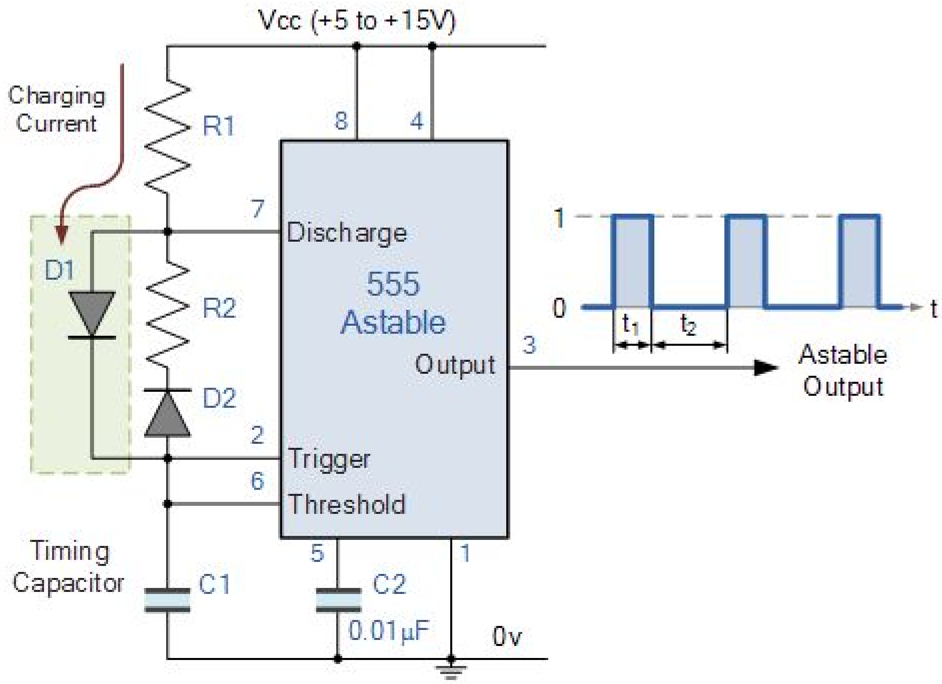

Figure 2. Using bypassing diodes to remove dependencies[2]

From this new circuit and from equation 1, the time ON t_1 is changed to the following equation:

$$t_1 = 0.693 \cdot R_1 \cdot C \qquad \text{Eq 3}$$

2.1.2 Schematic Analysis

R_2 was designed to be dependent on a thermistor in order to increase the OFF time on the 555 timer. A thermistor is a variable resistor whose resistance is dependent on temperature[3]. By keeping the ON time constant, I felt it was easier to see the change in duty cycle if the OFF time changed significantly.

[2] https://www.electronics-tutorials.ws/waveforms/555_oscillator.html
[3] https://www.omega.com/prodinfo/thermistor.html

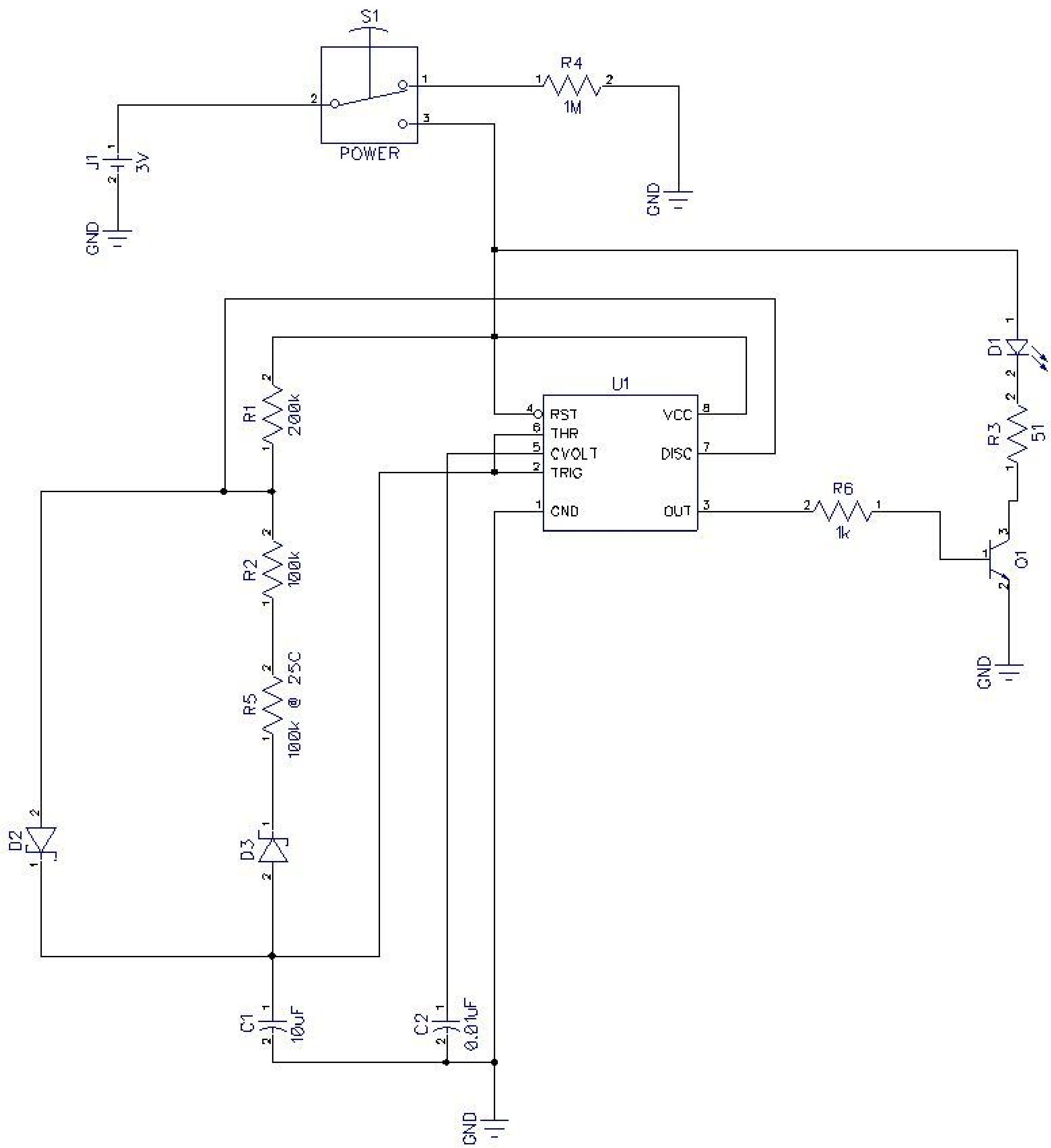

Figure 3. Schematic of Thermistor Controlled LED Blinker Circuit

The final schematic of the circuit is shown in Figure 3. It uses a switch to control the power to the circuit. It was a poor design choice to connect one of the terminals to a resistor and ground as that would very slowly drain the battery. It would have been much better to either keep the terminal connection open or to use a single-pole single-throw (SPST) switch as it has only one input and one output[4]. Nonetheless, this functionality can easily be removed by not connecting the 1 MΩ resistor on the PCB.

To power the LED, the current through the 555 timer output was sourced using a transistor. When the transistor switch is ON, it turns the LED ON. When the output of the 555 timer is low, the transistor switches OFF thus turning off the LED.

2.2 Prototyping

In order to verify functionality, the circuit was tested using components found in my home. In creating the schematic I used components that I already had available. As shown in the figure below, I connected the circuit using a breadboard and tested the circuit to see if it worked as expected.

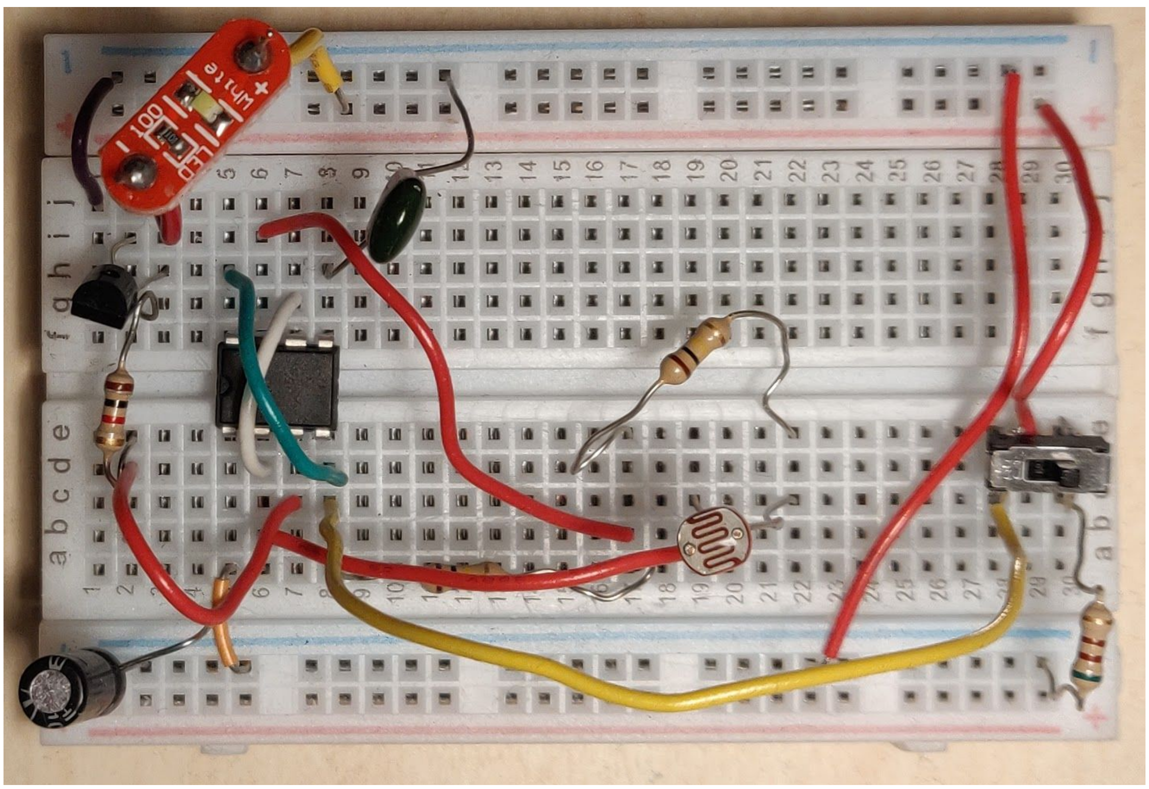

Figure 4. Prototyping circuit functionality[5]

Once I confirmed its functionality I went ahead and began designing the PCB layout.

[4] https://learn.sparkfun.com/tutorials/switch-basics/poles-and-throws-open-and-closed
[5] *Note: The power is not connected and I a photoresistor was used instead of a thermistor.*

2.3 PCB Design and Manufacture

This is the main aspect of the project: learning to create a PCB from the circuit designed. Similar to the schematic capture tool software, I used the DipTrace PCB Layout tool which easily exported the PCB layout to a Gerber file of all its layers.

In designing the PCB, I set a two main design specifications: make the thermistor easily accessible and design it in such a way that I can place it on a keychain. In order to meet the first specification, I placed the majority of the components in the center, and placed the thermistor close to the border of the board. This way, someone could easily press and warm up the thermistor with their body heat. To meet the second specification, I made the board small enough to fit in someone's pocket, I removed any sharp edges by making it round, and I added two key holes for the chain.

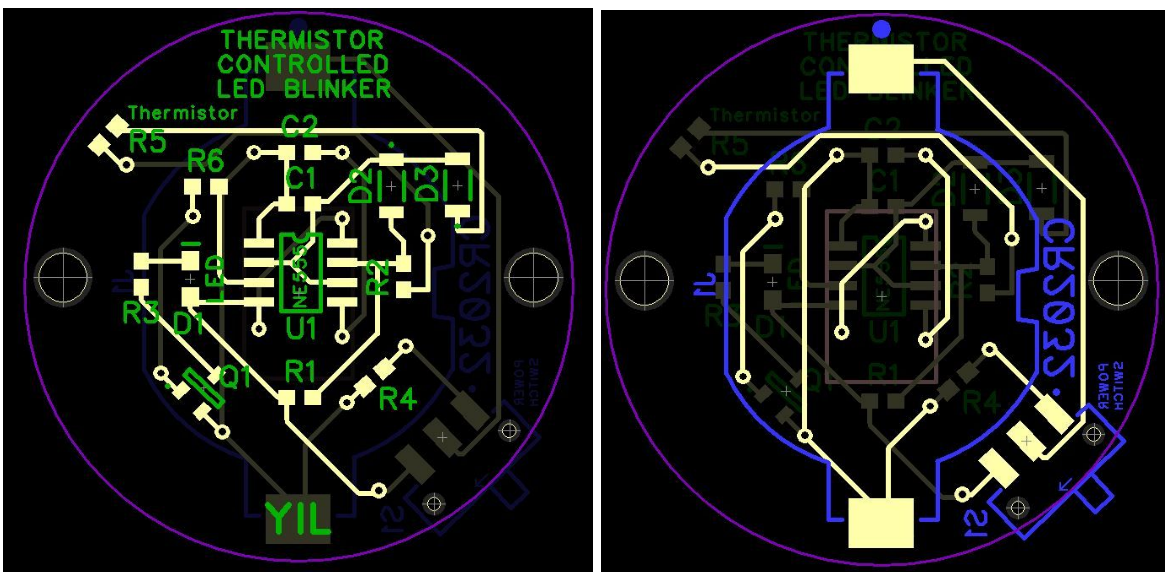

Figure 5. Top (left) and Bottom (right) views of the PCB Layout of the Circuit

The size was constrained to the length of the 3V CR2032 battery holder. The holder and the power switch were placed on the bottom side of the PCB, while the rest of the components were placed on the top.

The PCBs were manufactured using a chinese manufacture called JLC PCB[6].

[6]https://jlcpcb.com/e?gclid=CjwKCAiAu_LgBRBdEiwAkovNsM4tKCj8W8GYdN-DXSDrnuw3rzdzsbT9B2_3TbE0hC32aVz7UP4WchoC55AQAvD_BwE

2.4 Soldering

Another aspect of the project was to get more experience in soldering SMT components. Figure 6 shows the top and bottom of the unassembled PCB.

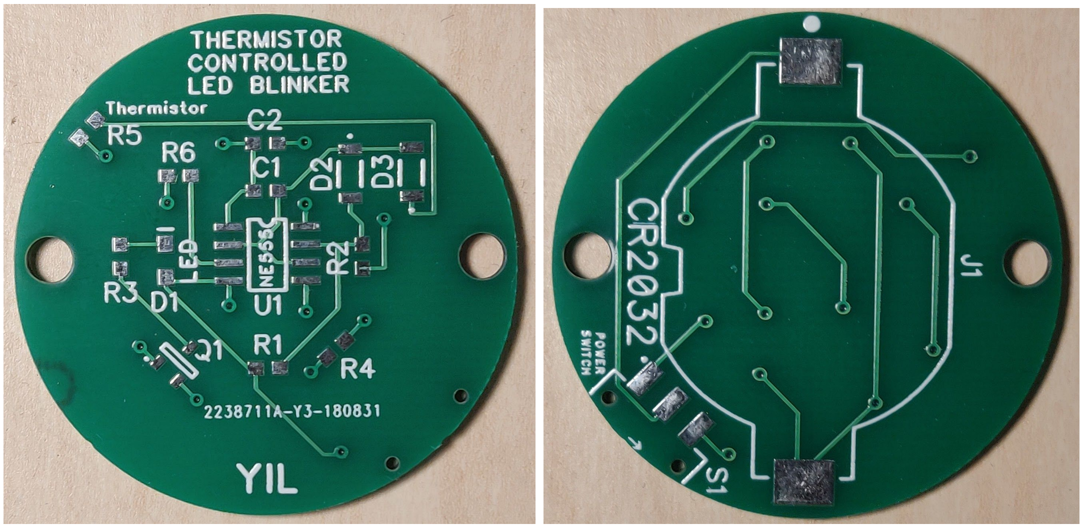

Figure 6. Unassembled PCBs

After soldering, I tested out each individual component to see if they were the correct values, and I tested the connectivity to see if the correct connections were applied. After that, I connected the 3V CR2032 battery, and waited for the LED to blink. If it did not blink, I would test each connection again to understand and fix where the problem existed.

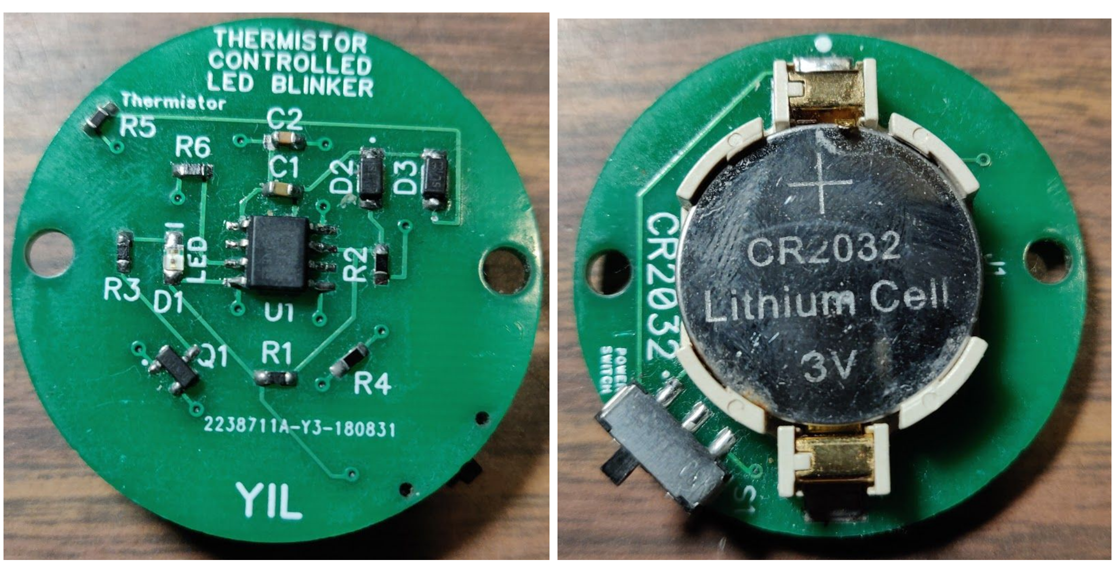

Figure 7. Assembled PCBs

3 RESULTS & DISCUSSIONS

While testing, the heat from a person touching the thermistor did not affect the blinking of the LED. This was expected since the thermistor has a non-linear characteristic curve. Changing the temperature of the thermistor by a few degrees affected the circuit, but it was very unnoticeable to the naked eye. Ideally the thermistor reading should have been more linear which could be achieved by either getting a more linear thermistor or by designing a circuit that would change its linearity.

3.1 Testing

Nonetheless, although not practical, the PCB was placed inside a freezer for two minutes in order to effectively cool down the thermistor. As expected, the LED would stay off for a much longer period than at room temperature. Heating the thermistor was not tested as there was no reliable heat source available.

4 CONCLUSION

All in all, I learned how to design a PCB, and became more proficient in designing a schematic and building a physical device from it. Although the thermistor compensation was not effectively designed, the goal of this project was not to create a successful and useful circuit.

4.1 Future Work

In the future, I would prefer to work on more complicated projects by combining both analog and digital circuits. My next task is to create a PCB that uses a microprocessor to do a certain task. I will be using the ATTiny85 microcontroller with the arduino IDE and an Inertial Measurement Unit (IMU) to design a small fidget device.

On top of that, it would be ideal to encase these products in a 3D printed container that I design to become more acquainted with Computer Aided Design (CAD) software and 3D printing.