Kitchen Timer Project

Phases A and B

Embedded Systems Programming

University of Denver

ENCE 3221

Dr. Martins

5/4/2023

Zach Bayler

Introduction:

The goal of this project is to design, build, and test an Internet of Things (IOT) kitchen timer. The first two phases of this development process include creating a prototype in the form of an Arduino shield to leverage the existing microprocessor and simplified coding environment to control a custom piece of hardware. This provided a first attempt at structuring a printed circuit board (PCB) as well as developing the basic capabilities of a timer. The second phase takes it a step further to design a completely standalone piece of hardware. Notably, the PCB included its own microprocessor which increased the hardware complexity. Additionally, the code was further developed to include some basic IOT functionality, allowing for a smartphone or computer to connect through Wi-Fi and control the timer. This report covers the requirements set out for the device, the developed design, component selection, the prototype (Phase A), the PCB design, the software development, and the enclosure design. Unless otherwise noted, the development in question refers to Phase B.

Project Requirements:

Create a kitchen timer system that:

- Displays time remaining in minutes and seconds
- Has a user button for incrementing the time remaining
- Has a user button for starting and pausing the countdown
- Provides a phone or computer with all the same functionality stated above
- Displays the current state with LEDs (counting, paused, and finished)
- Signals the completion of the countdown with a noticeable sound

System Design:

The components necessary to achieve the capabilities laid in the project requirements include a microprocessor to act as the brains of the system, 3 LEDs, a 4x7-segment display, a buzzer, and 3 buttons. More details are included in the hardware block diagram included in Appendix A.

Component Selection:

The components selected are shown in the Bill of Materials located in Appendix B. The components are mostly surface mounted so that they could be smaller and only interact with a single layer of the PCB. The PCB was selected to be a two-layer construction to save on cost. As for the specifics of the components, the details were provided by the professor.

Prototype (Phase A):

The Arduino Shield was developed to test the components and develop the basic code before moving on to more in-depth development. The circuit schematic and PCB layout and routing are included in Appendices F and G. However, this design was more practice as the physical board were provided by the professor and are his design (for the sake of time and to make sure it works).

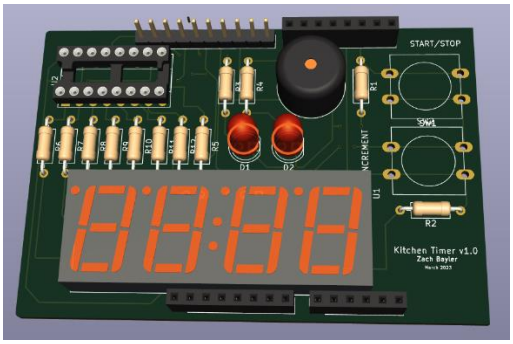
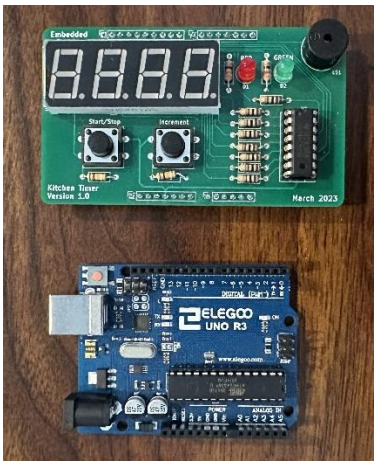
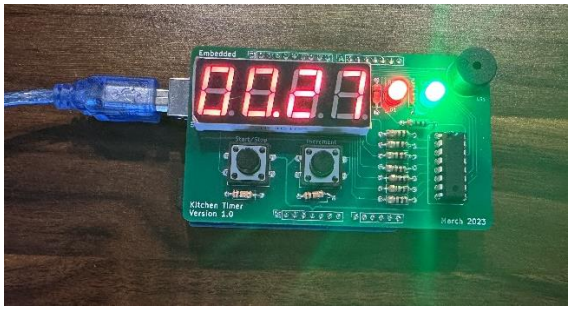


Fig. 1: Shows a 3D rendering of the PCB design.





Figures 2 and 3: Show the professors Arduino Shield running our code.

PCB Design:

Schematic (circuit) design:

The schematic is organized by modules so that the circuit components are not overwhelming. It is included in the Appendix A. The most notable components include:

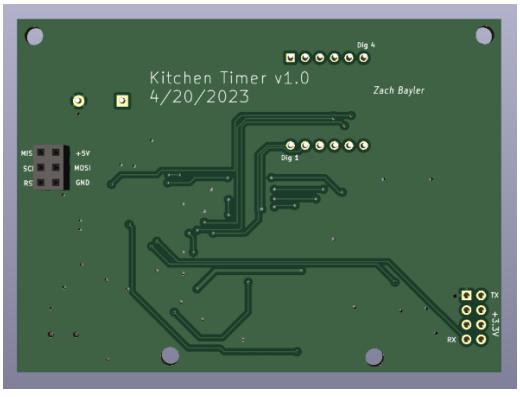
- Decoupling capacitors
- Voltage regulator
- Microprocessor
- USB connector (power supply)
- Status LEDs
- ESP Connector (Wi-Fi module)
- 7-segment display
- Crystal
- Reset button
- User functionality buttons
- Microprocessor programmer connector
- Buzzer

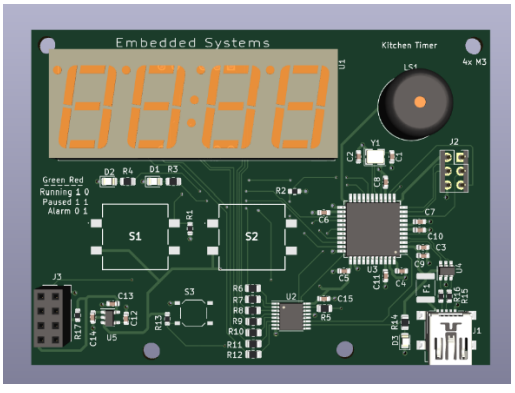
PCB layout:

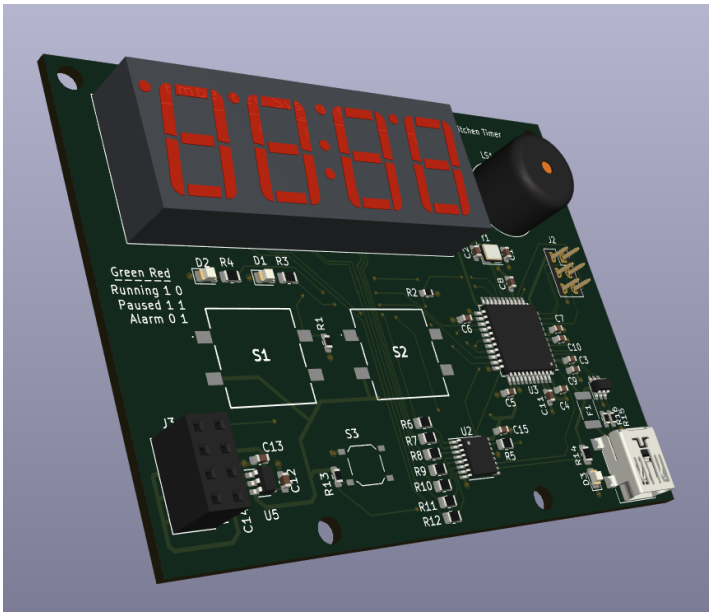
The components were laid out based on their modules to minimize routing complexity. Additionally, M3 mounting holes were included in the board so that it can attach into the enclosure.

PCB routing:

The routing was difficult, but mostly was kept to the top layer to minimize the disruption of the ground layer. Two different thickness were used: a larger one for the power connections and a smaller one for the signals.







Figures 4, 5, and 6: Three views of the 3D model view generated by KiCAD.

Assembling Stage:

*Currently not assembled yet as the purchased custom designed PCBs have not yet arrived.

Software Development:

The block diagram for Phase A (Appendix H) still remains mostly unchanged for this phase. The main inclusion is that of the Wi-Fi inputs to also control the timer. The Phase A code and test of the IOT software are both working, but the continuation of the code for the new microprocessor in Phase B will be completed when the hardware is available to test it on.

Enclosure Design:

The goal for the enclosure was to create a case for the PCB that protects it as well as allows it to lay flat on any of its sides all while maintaining its functionality. The design of the enclosure occurred after the PCB was ordered, therefore all the dimensions of the PCB and component placement were already set in stone. This provided some difficulty as all of the measurements had to be matched to the 3D-model and notably one of the mounting holes is blocked by the Wi-Fi module. Additionally, the buttons did not appear on the 3D model exported from the PCB design software, so measurements had to be taken from that software which is 2D only. To maintain the functionality, cutouts were created for the USB cable, buzzer, 7-segment display, and the buttons. Mounting bosses were included to hold the PCB at the correct height and hold it in place.

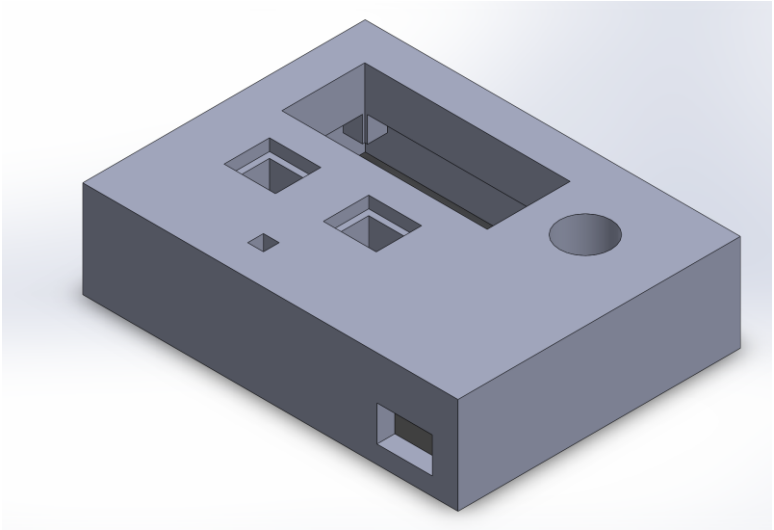
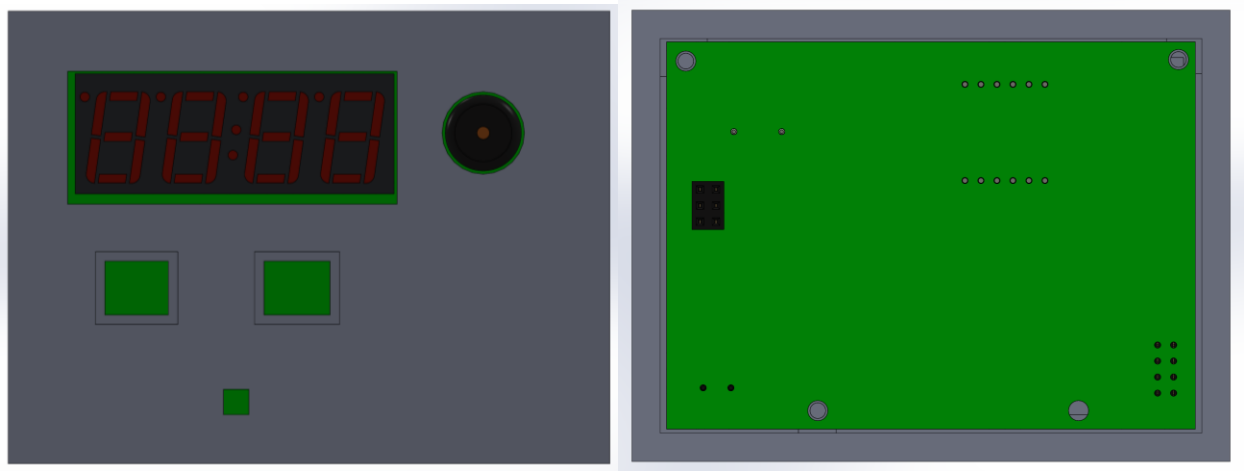


Fig. 7: 3D model of the PCB enclosure in SolidWorks. An engineering drawing is included in Appendix D.



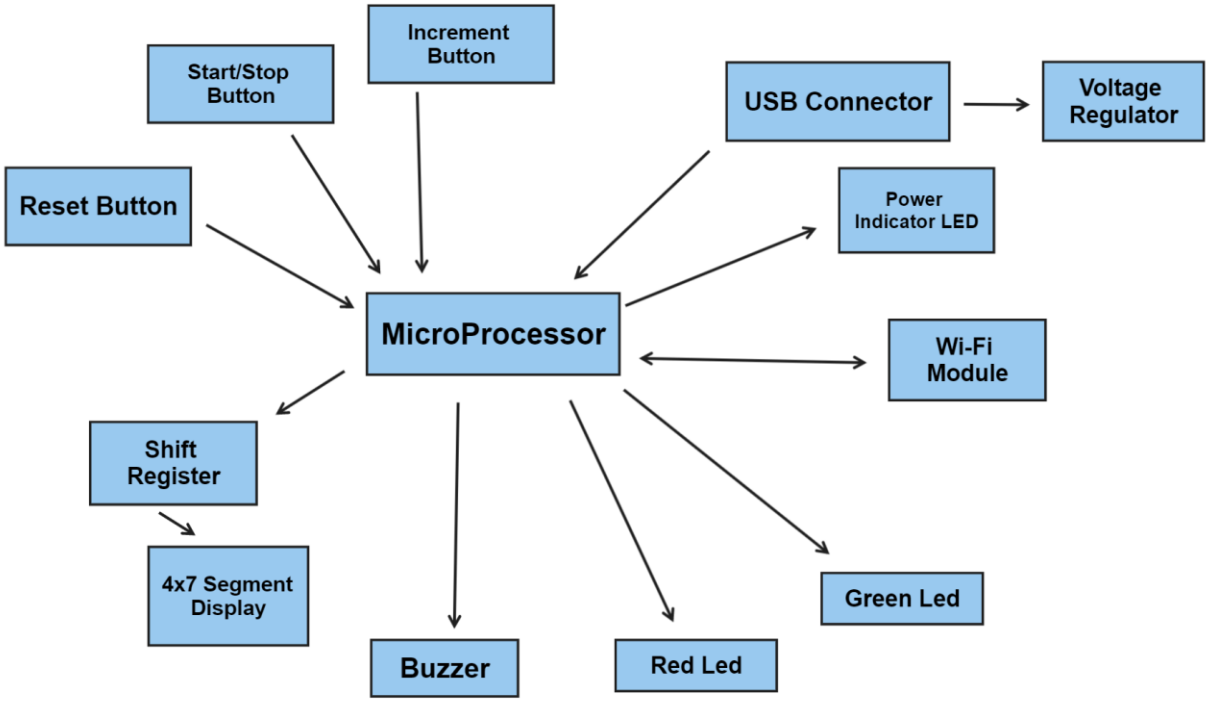
Figures 8 and 9: Front (on Left) and Back (on Right) view of the assembly with the PCB fitting in the enclosure.

Figures 10 and 11: Two views of the first version of the printed product. Some of the features came out as smaller than expected as it was easy to zoom in and lose a sense of scale in the CAD software. Notably, the user button holes seem too tight to fit in a normal finger.

Conclusion:

At this current moment, the Phase B is not yet finished as the PCBs have not yet arrived to be assembled and tested. Since there isn't a platform to fully test the code, it hasn't been completed either. A lot of time went into designing the enclosure in the meantime and the first print version is completed. The prototype shield (Phase A) does work and an initial test of the Wi-Fi module was completed but not yet incorporated. Most of the work to complete Phase B is now waiting on the arrival and assembly of the PCB.

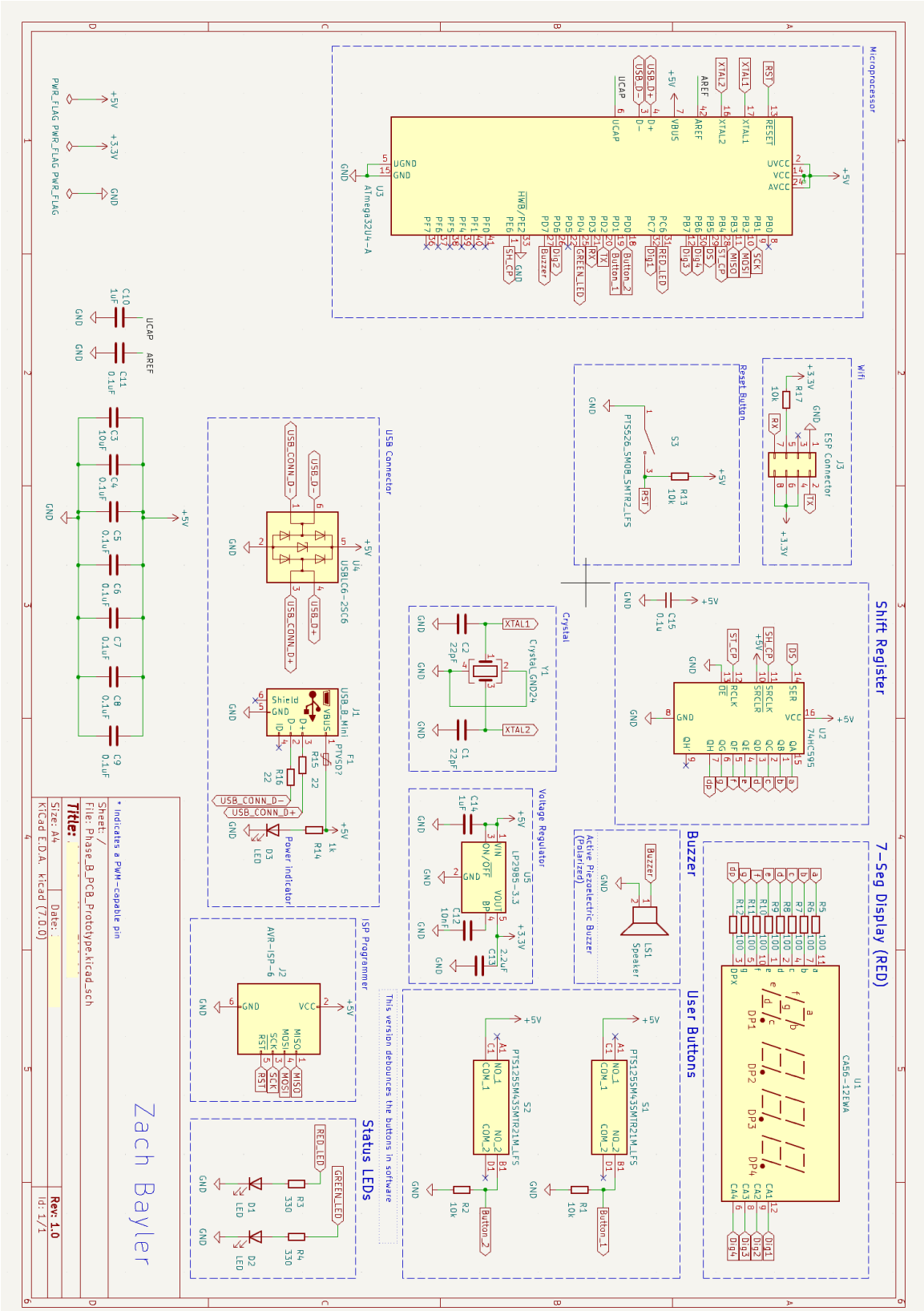
Appendix A: Hardware Block Diagram



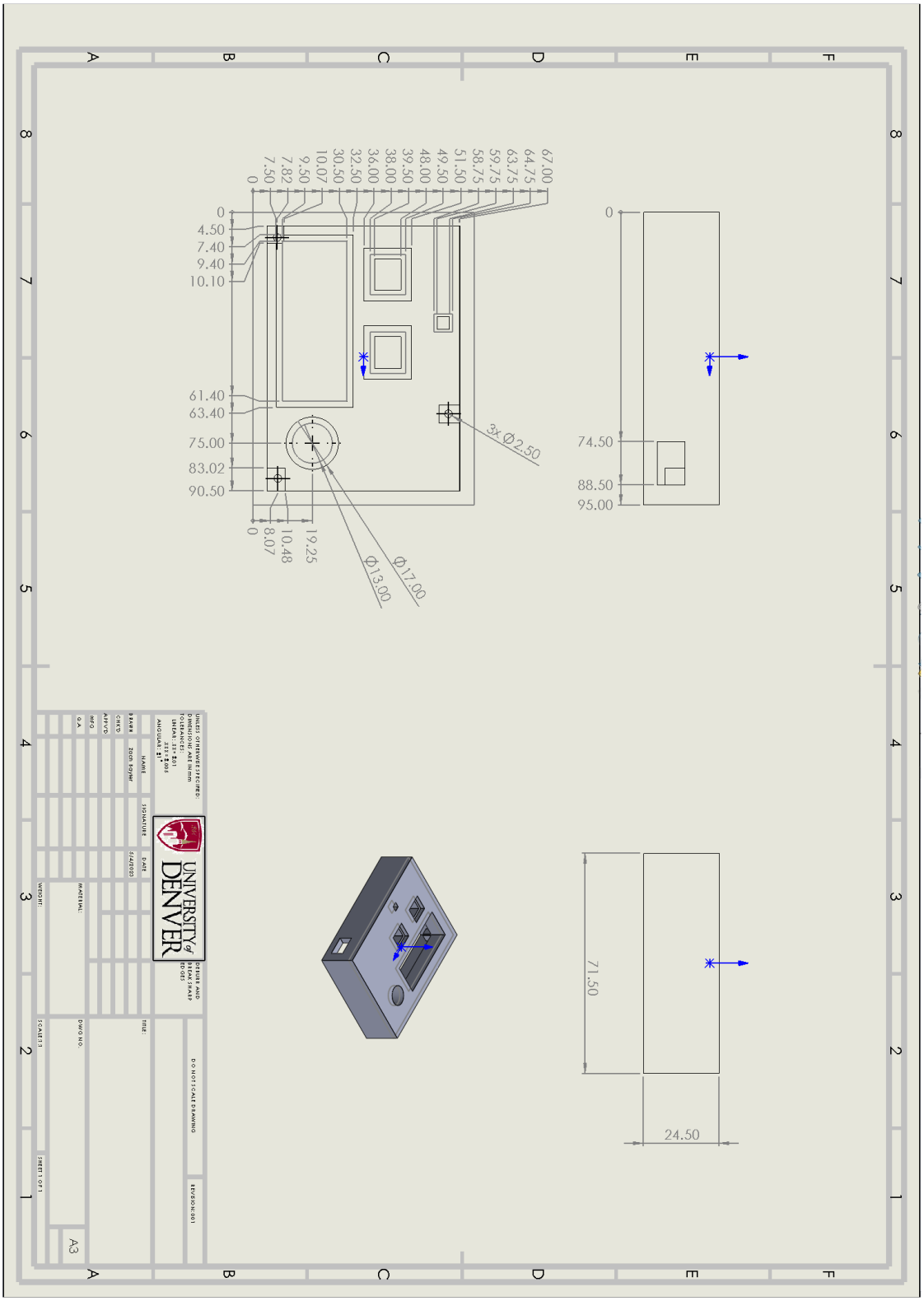
Appendix B: Bill of Materials

□□□	Sourced	Placed	References	Value	Footprint	Quantity
1	☐	☐	C4, C5, C6, C7, C8, C9, C11, C15	0.1uF	C_0603_1608Metric	8
2	☐	☐	C1, C2	22pF	C_0603_1608Metric	2
3	☐	☐	C10, C14	1uF	C_0603_1608Metric	2
4	☐	☐	C3	10uF	C_0603_1608Metric	1
5	☐	☐	C12	10nF	C_0603_1608Metric	1
6	☐	☐	C13	2.2uF	C_0603_1608Metric	1
7	☐	☐	R5, R6, R7, R8, R9, R10, R11, R12	100	R_0805_2012Metric	8
8	☐	☐	R1, R2, R13, R17	10k	R_0603_1608Metric	4
9	☐	☐	R3, R4	330	R_0805_2012Metric	2
10	☐	☐	R15, R16	22	R_0603_1608Metric	2
11	☐	☐	R14	1k	R_0805_2012Metric	1
12	☐	☐	D1, D2, D3	LED	LED_0805_2012Metric	3
13	☐	☐	U1	CA56-12EWA	CA56-12EWA	1
14	☐	☐	U2	74HC595	TSSOP-16_4.4x5mm_P0.65mm	1
15	☐	☐	U3	ATmega32U4-A	TQFP-44_10x10mm_P0.8mm	1
16	☐	☐	U4	USBLC6-2SC6	SOT-23-6	1
17	☐	☐	U5	LP2985-3.3	SOT-23-5	1
18	☐	☐	Y1	Crystal_GND24	Crystal_SMD_Abracon_ABM8G-4Pin_3.2x2.5mm	1
19	☐	☐	F1	PTVSD?	Fuse_1812_4532Metric	1
20	☐	☐	S1, S2	PTS125SM43SMTR21M_LFS	PTS125_SMD_Button	2
21	☐	☐	LS1	Speaker	Buzzer_12x9.5RM7.6	1
22	☐	☐	S3	PTS526_SM08_SMTR2_LFS	PTS526_SMD_Button	1
23	☐	☐	J1	USB_B_Mini	USB_Mini-B_Lumberg_2486_01_Horizontal	1
24	☐	☐	J2	AVR-ISP-6	PinSocket_2x03_P2.54mm_Vertical	1
25	☐	☐	J3	ESP Connector	PinSocket_2x04_P2.54mm_Vertical	1

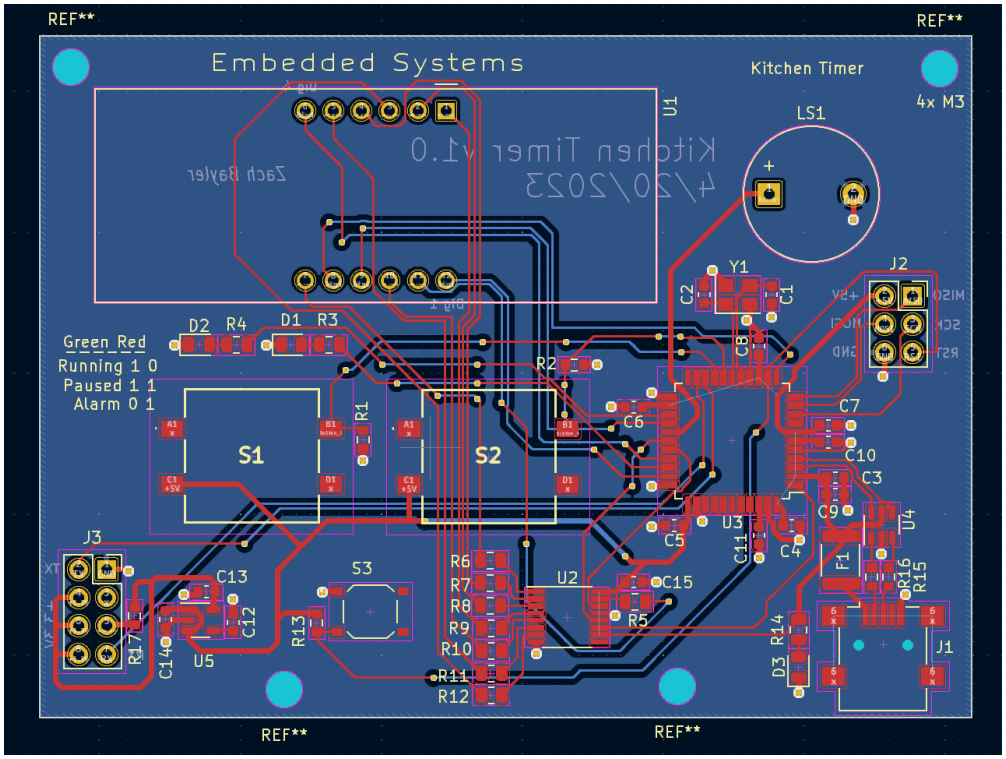
Appendix C: Phase B Schematic

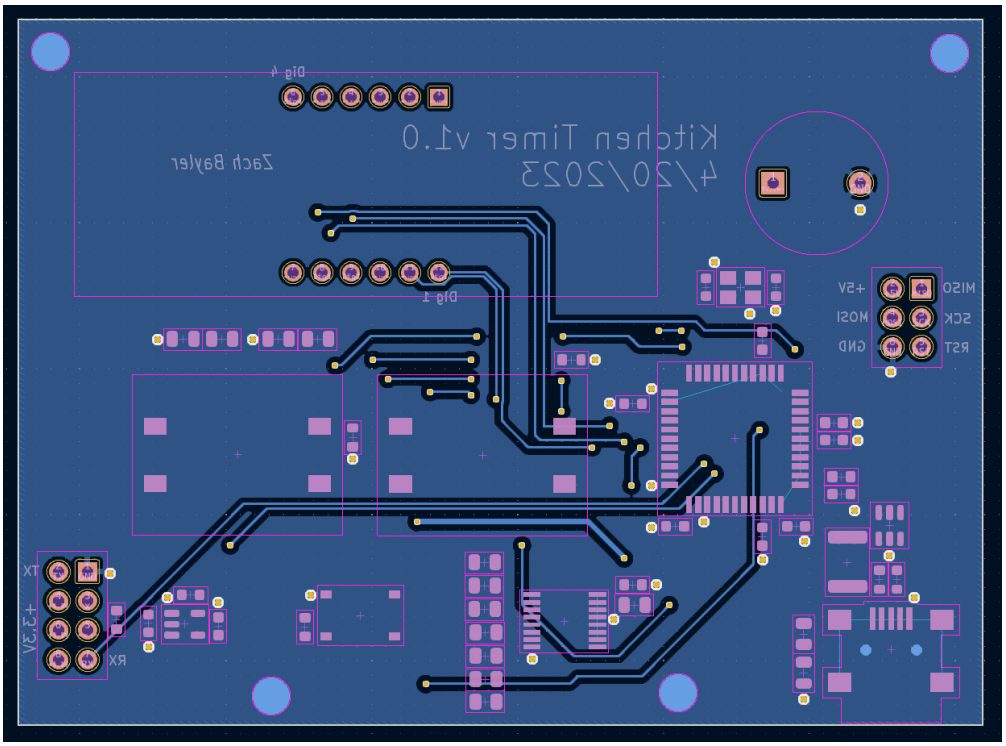


Appendix D: Engineering Drawing of the Enclosure

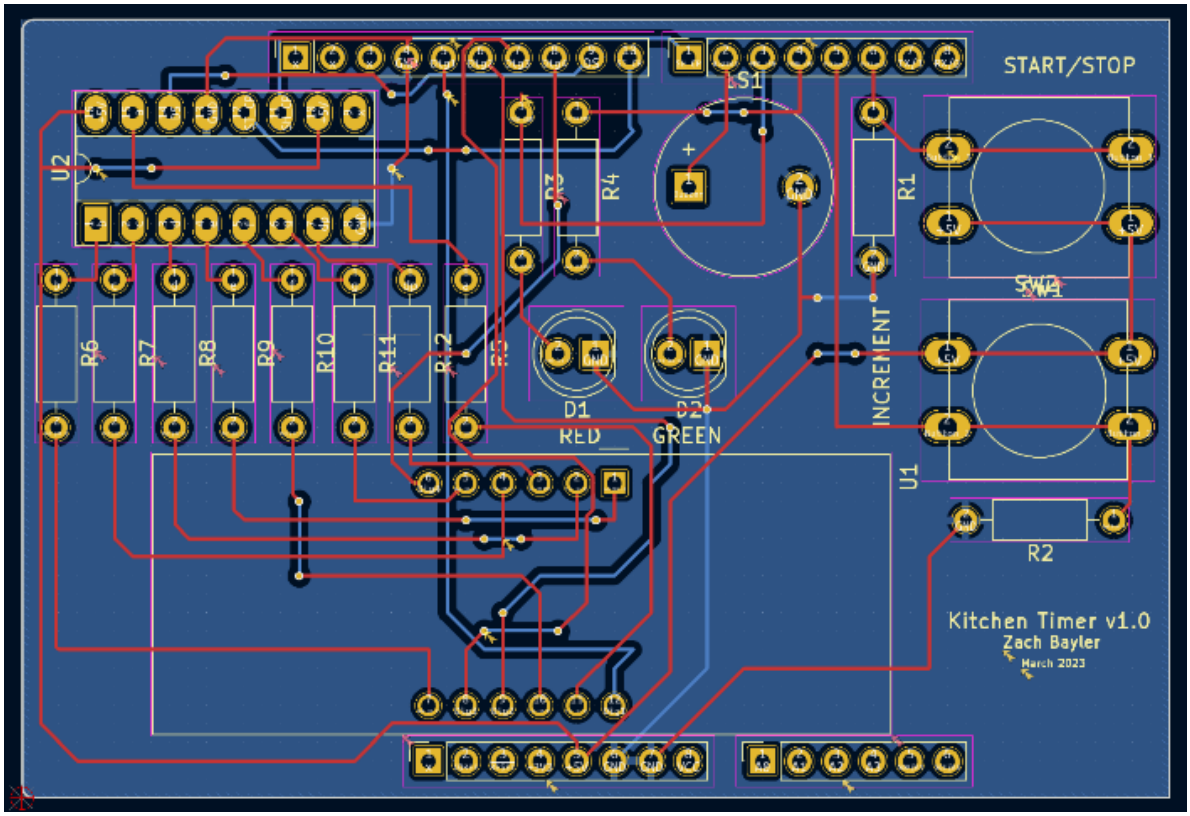


Appendix E: Phase B PCB Routing

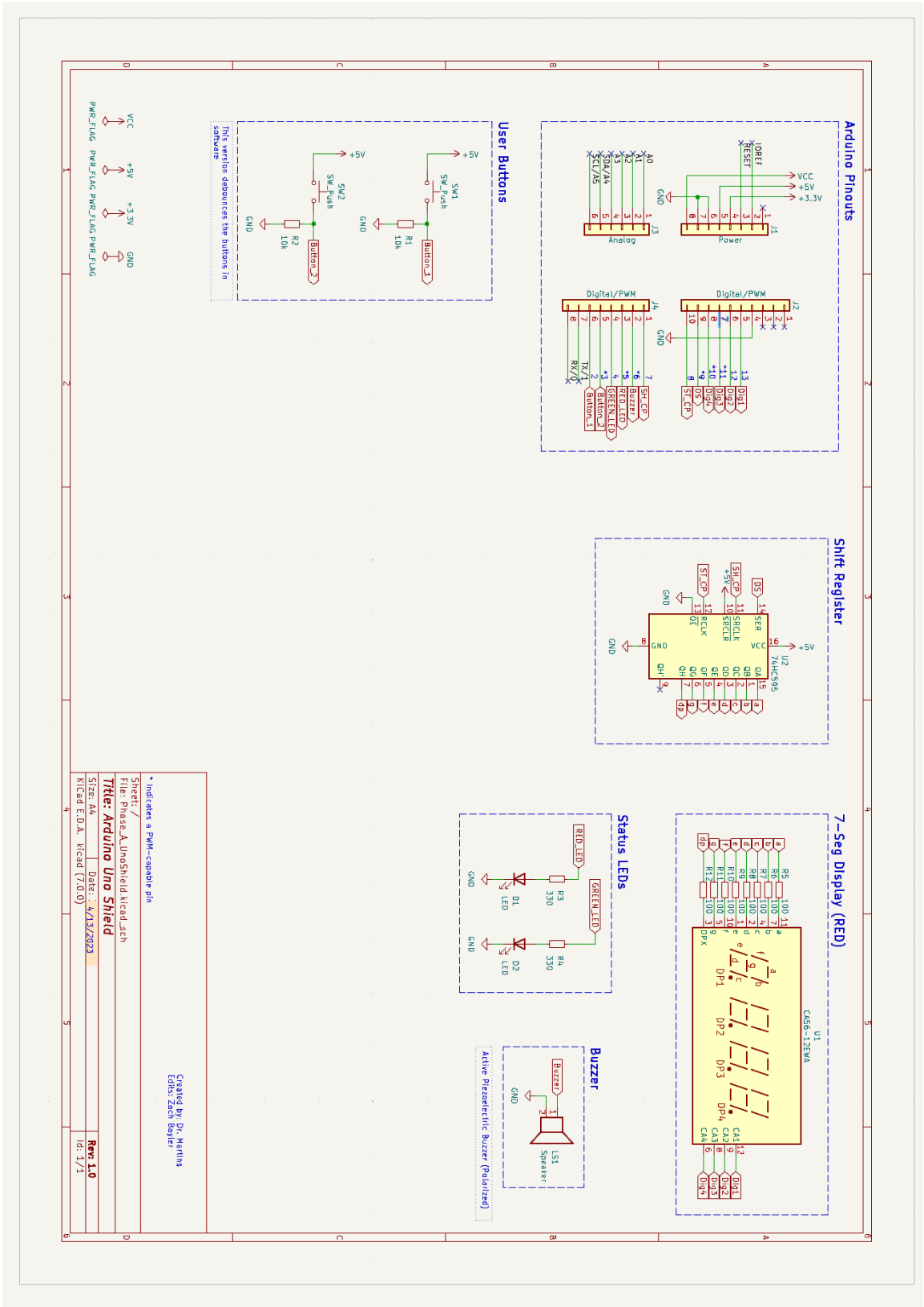




Appendix F: Phase A Routing



Appendix G: Phase A Schematic



Appendix H: Phase A Code Block Diagram