Kitchen Timer Project

Phases A and B

Embedded Systems Programming

University of Denver

ENCE 3221

Dr. Martins

6/7/2023

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, the focus of this report, include creating an Arduino shield (Phase A) and then designing a standalone system (Phase B). Starting with Phase A, creating a prototype in the form of an Arduino shield leveraged 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 hardware and software capabilities of the timer. Then, Phase B, took it a step further by designing and building a standalone piece of hardware. Notably, the PCB in this phase 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:

To achieve the capabilities laid in the project requirements, a system including a microprocessor, 3 LEDs, a 4x7-segment display, a buzzer, a Wi-Fi module, and 3 buttons was chosen. The main design choices included the 7-segment display as the simplest and most efficient way to display the remaining time and a Wi-Fi module to enable IOT functionality through the connection to other Wi-Fi enabled devices. Included in Appendix A is a block diagram detailing the interaction between the various components.

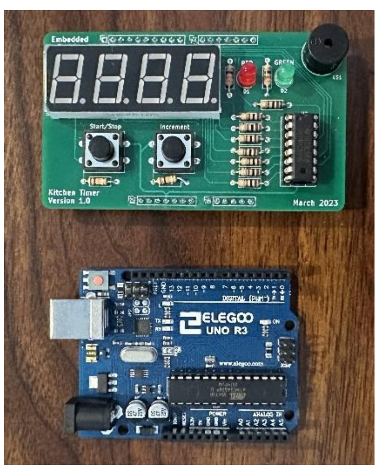
Component Selection:

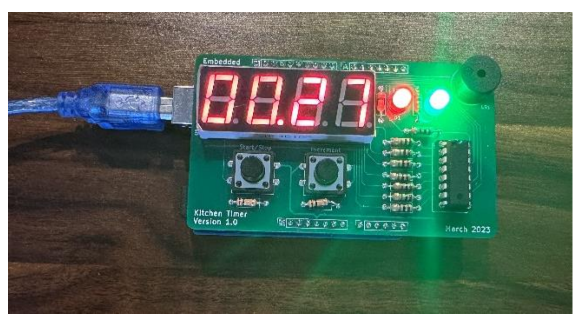
Specific components were selected for everything mentioned in System Design and a collection of resistors and capacitors are also included for interfacing the components correctly. The components are mostly surface mounted to minimize the footprint and so that they would only interact with one layer 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 during the design process and are shown in the Bill of Materials located in Appendix B.

Prototype (Phase A):

The Arduino Shield was developed to test the chosen components and develop the basic code before moving on to more in-depth development. The circuit schematic, PCB layout, and routing are included in Appendices C and D. However, this design was primarily an opportunity for practice as the physical board used for the software development and testing in this phase 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 my Arduino shield PCB design and Figures 2 and 3 show the professor's shield that we actually got to use. Additionally, a block diagram for the timer code shown in use in Fig. 3 is included in Appendix E.

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





Figures 2 (Left) and 3 (Right): 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 F. The most notable components include:

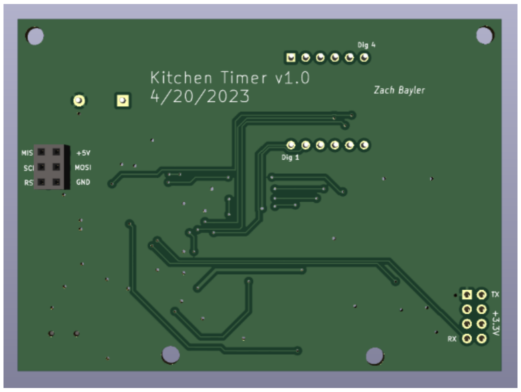
- Decoupling capacitors
- Voltage regulator
- Microprocessor
- USB connector (power supply)
- Status LEDs
- ESP Connector (Wi-Fi module)
- 7-segment display
- Crystal
- Reset button
- User functionality buttons
- Microprocessor programmer connector
- Buzzer

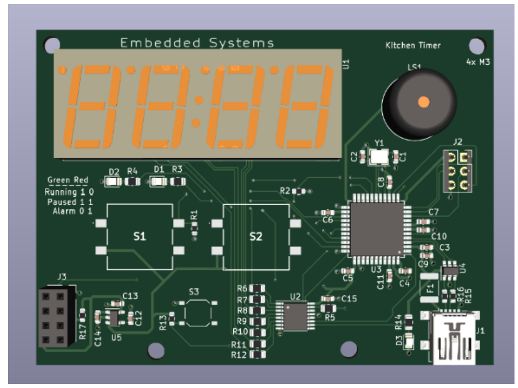
PCB layout:

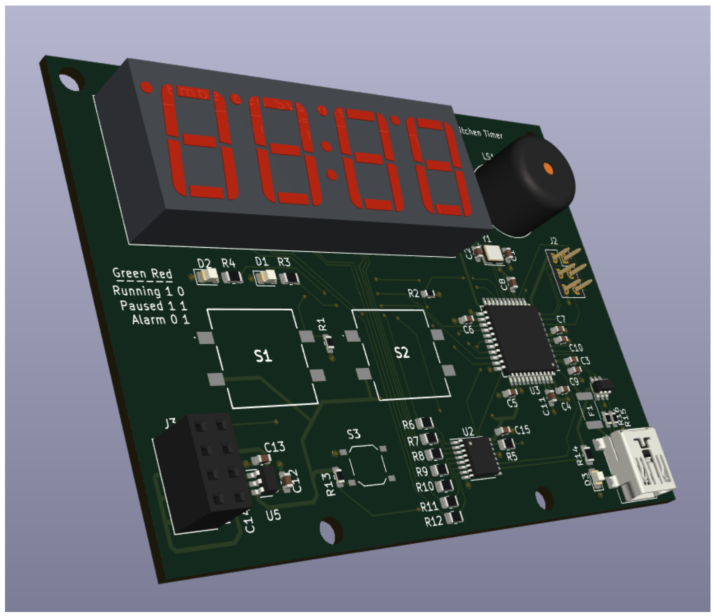
Designing the PCB layout started with placing the main components for user interfacing. The 7-segment display and buttons must be in useable positions especially when the enclosure is included. Then, the rest of the components were laid out based on their modules to minimize routing complexity. Additionally, M3 mounting holes were included in the board so that it can attach into the enclosure.

PCB routing:

The routing was difficult, as a second attempt ever designing a PCB, but was mostly kept to the top layer to minimize the disruption of the ground layer. Maintaining the ground layer is one technique used to maximize signal integrity. As for the traces, two different thicknesses were used: a larger one for the power connections and a smaller one for the signals. Other PCB design and routing concepts were not incorporated as this class only had the time to provide the very basics of a very complex field. Additionally, this circuit is not particularly complex nor intended for production and sale.



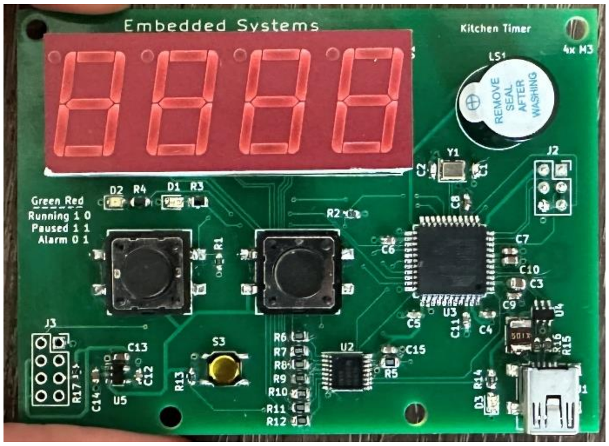


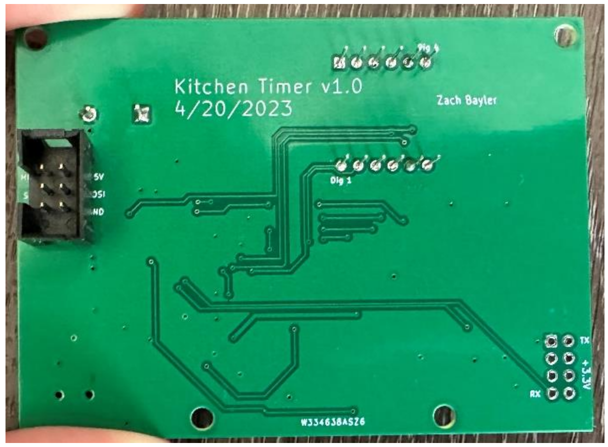


Figures 4, 5, and 6: Three views of the 3D model view generated by KiCAD.

Assembly Stage:

Once the order for the custom designed PCBs and all of the components arrived, the next task was soldering everything as designed. This was a cool but challenging experience and another brand-new skill introduced by this class. While I've soldered before, the added complexity of surface mounted components was a completely new experience. The precision required for the small pads and components required very steady hands that I do not possess.





Figures 7 (Left) and 8 (Right): Fully assembled Phase B independent PCBs.

Throughout the PCB assembly stage, various mistakes became apparent. First of all, the 7-segment display which was oriented for simplified routing, was upside down compared to the intended design. Additionally, the reset switch was much smaller than expected and thus not very accessible. When testing the hardware, the USB LED turned on, which indicated that power was received, but the computer didn't recognize the device. Attempts were made to try fixing the soldering connections (with the help of the professor) but to no avail. Two possibilities are that the microprocessor sustained too much heat on soldering and was damaged or that there is some mistake in the design of the PCB itself.

Software Development:

The block diagram for Phase A (Appendix E) remains the core for further development. However, the Wi-Fi interaction is the greatest thing missing from the current software. The Phase A code and test of the IOT software (partially provided by the professor) are both working, but the continuation of the code for the new microprocessor in Phase B was not completed as the hardware issues prevented testing.

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.

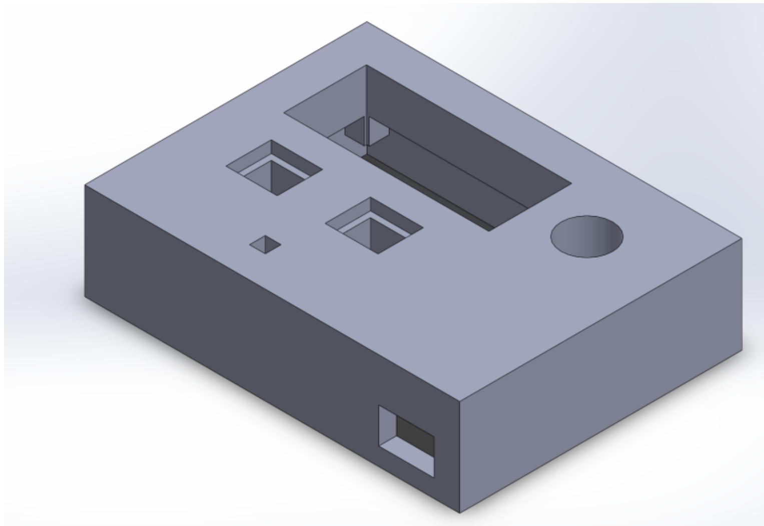
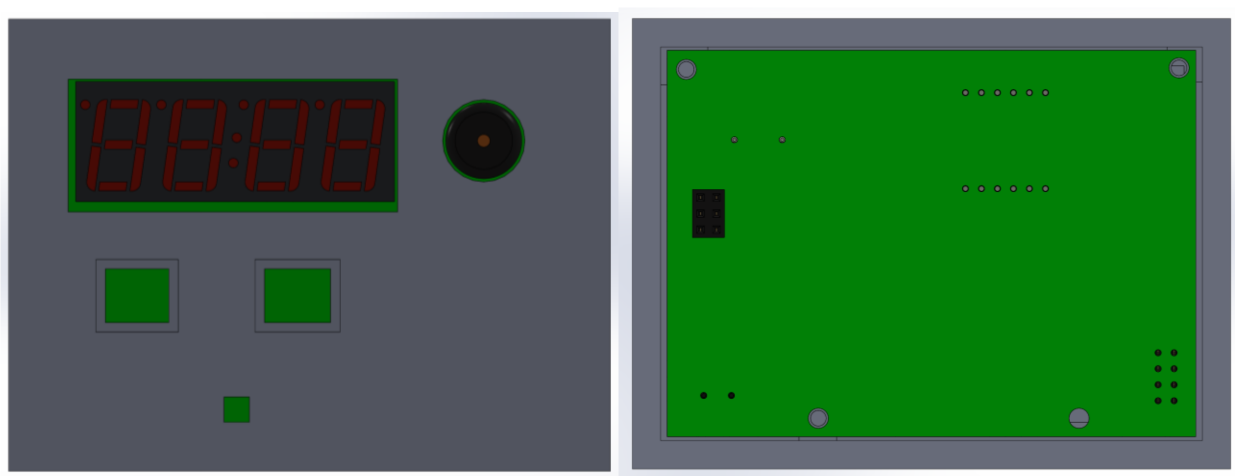


Fig. 9: 3D model of the PCB enclosure in SolidWorks. An engineering drawing is included in Appendix D.



Figures 10 and 11: Front (on Left) and Back (on Right) view of the assembly with the PCB fitting in the enclosure.

Figures 12 and 13: 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.

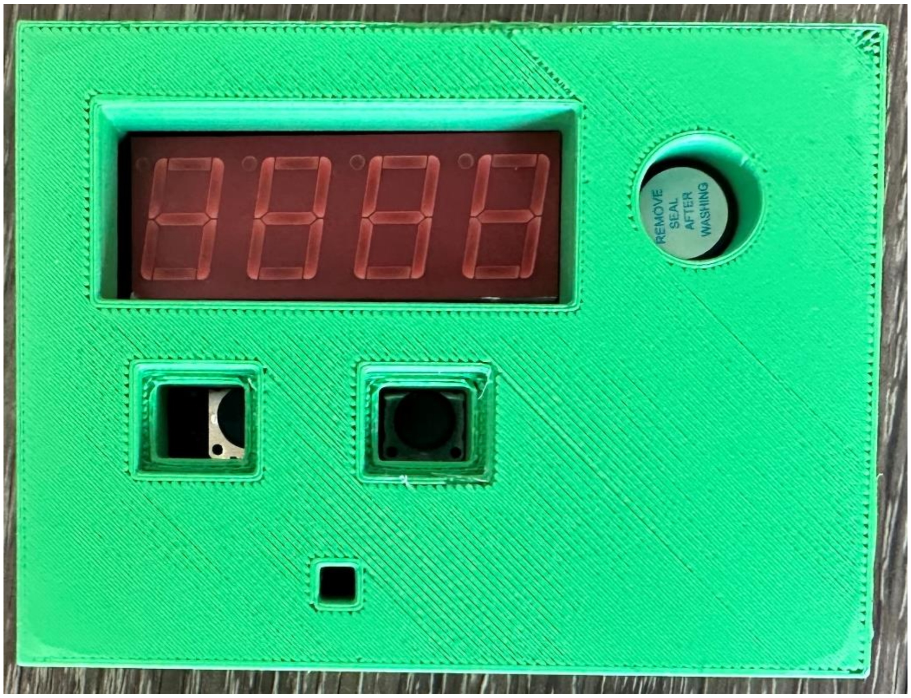


Figure 14: The assembled PCB board paired with its enclosure. Some of the measurements were off and will require an updated second version. The left button is visibly not lined up with the hole, which impacts intended insertion and fastening of the two components.

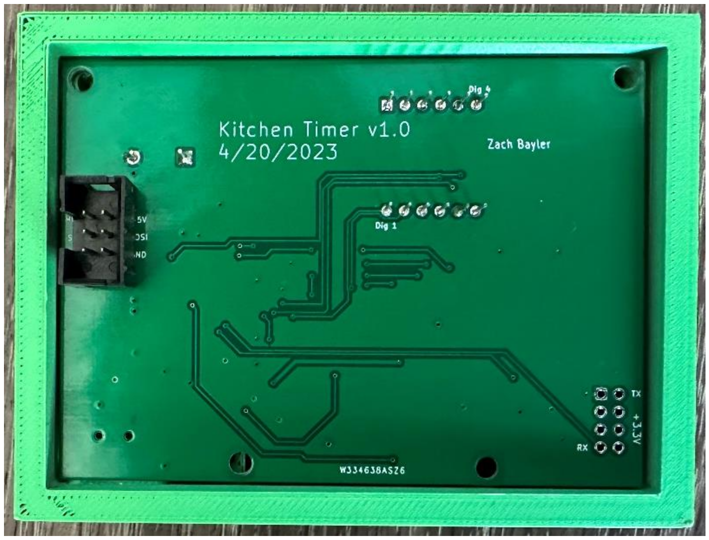


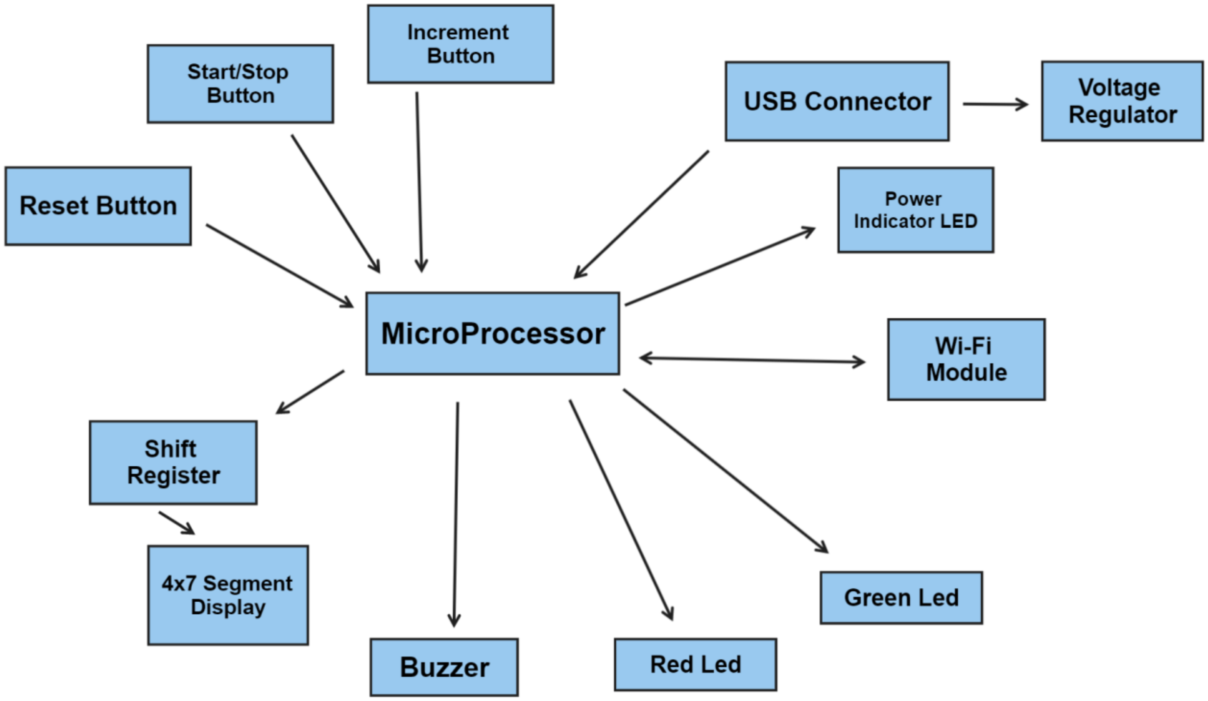
Figure 15: Back view of the assembled PCB board in its enclosure. Again, some of the measurements were off as seen by the bottom left mounting boss that doesn't align with the hole in the PCB.

Figure 16: Side view of the assembled PCB board in its enclosure. The USB connector is accessible in the hole as intended.

Conclusion:

At present, the project is as finished as it will be. Everything is assembled, but the hardware is not working. A comprehensive troubleshooting and potential complete resoldering of the PCB is required to complete Phase B. As such, the code has not been developed as there would be no method for troubleshooting it on the board. That being said, the project was an amazing learning experience. So many issues and skills were touched upon from hardware-software compatibility, to software concurrency, to soldering surface mounted components, to end user considerations such as labeling buttons, and more. Overall, this greatly leveled up my skillset as an electrical engineer by teaching new concepts to consider, how to use new tools, and developing new skills.

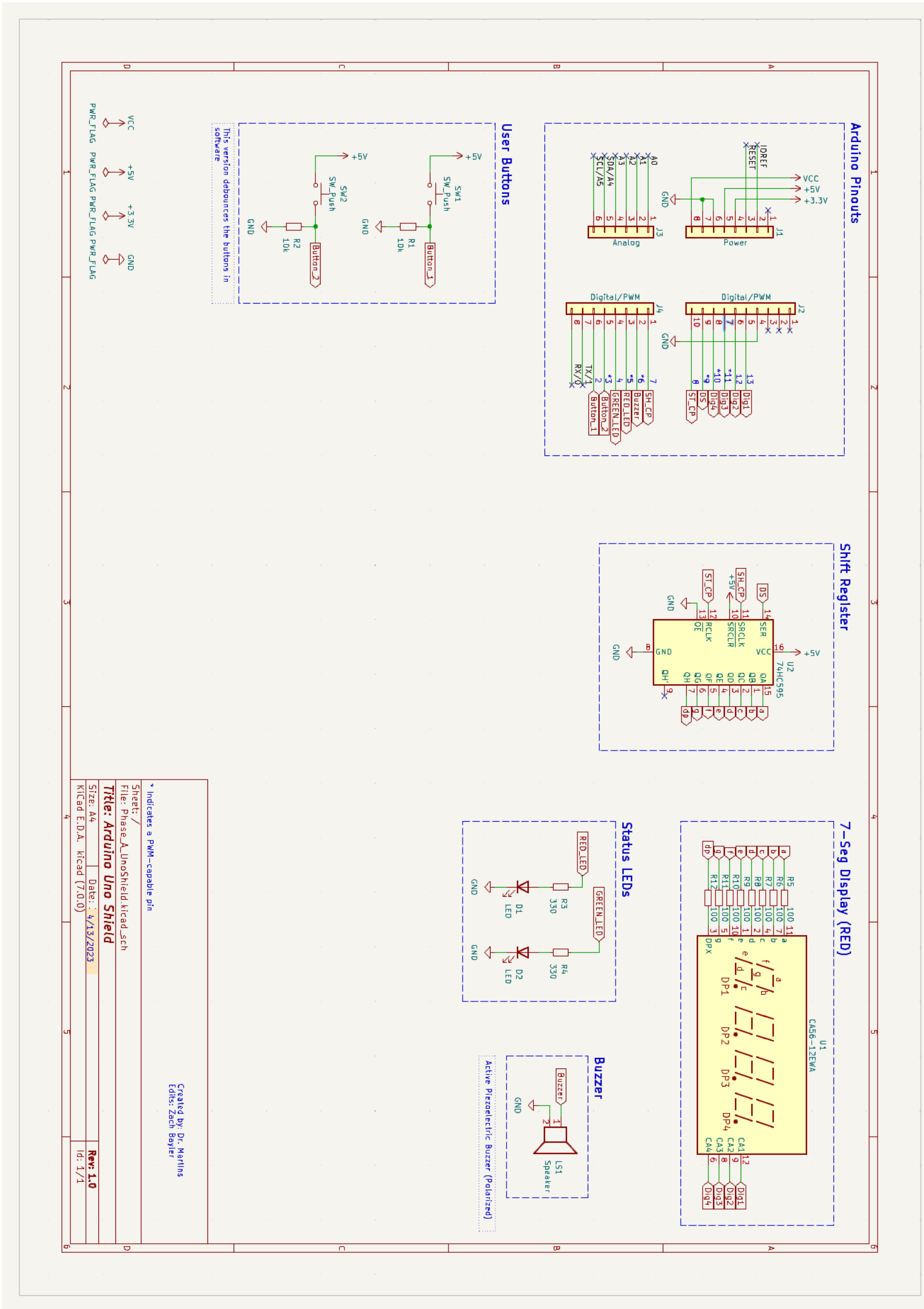
Appendix A: Hardware Block Diagram



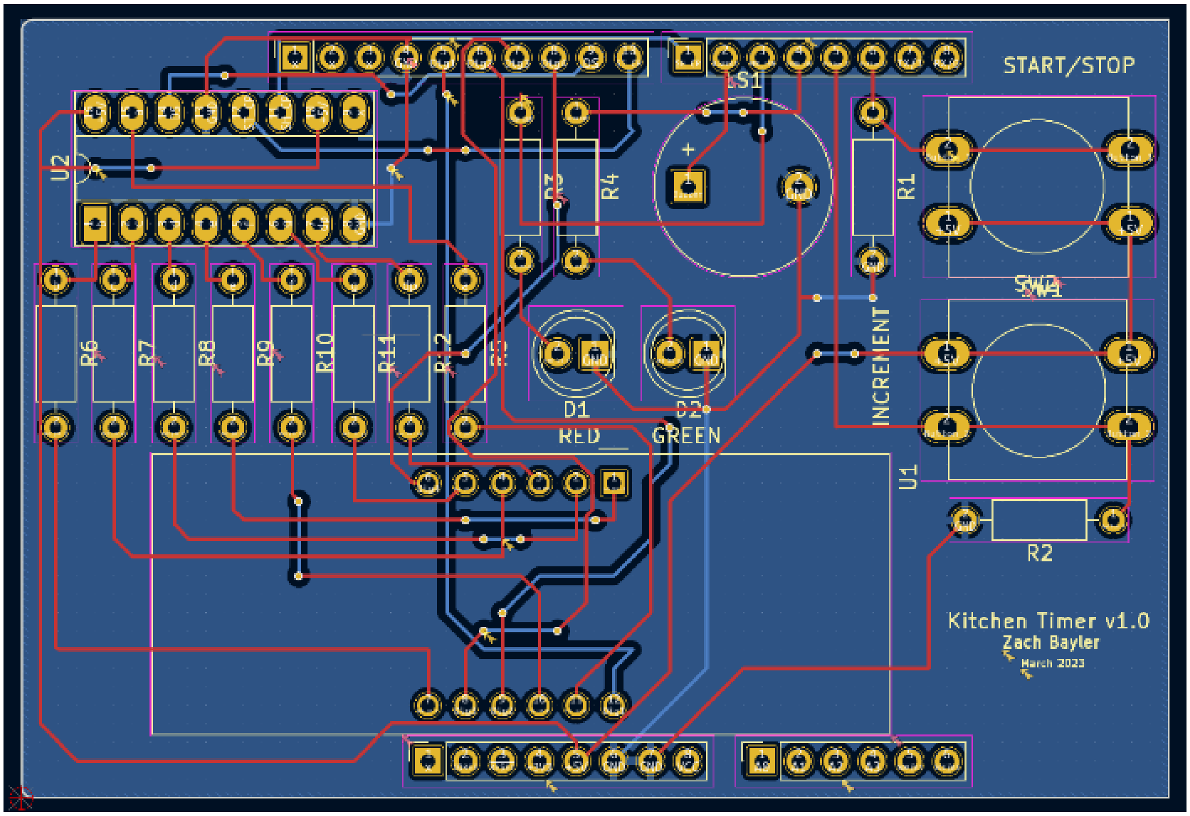
Appendix B: Bill of Materials

▯▯▯	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

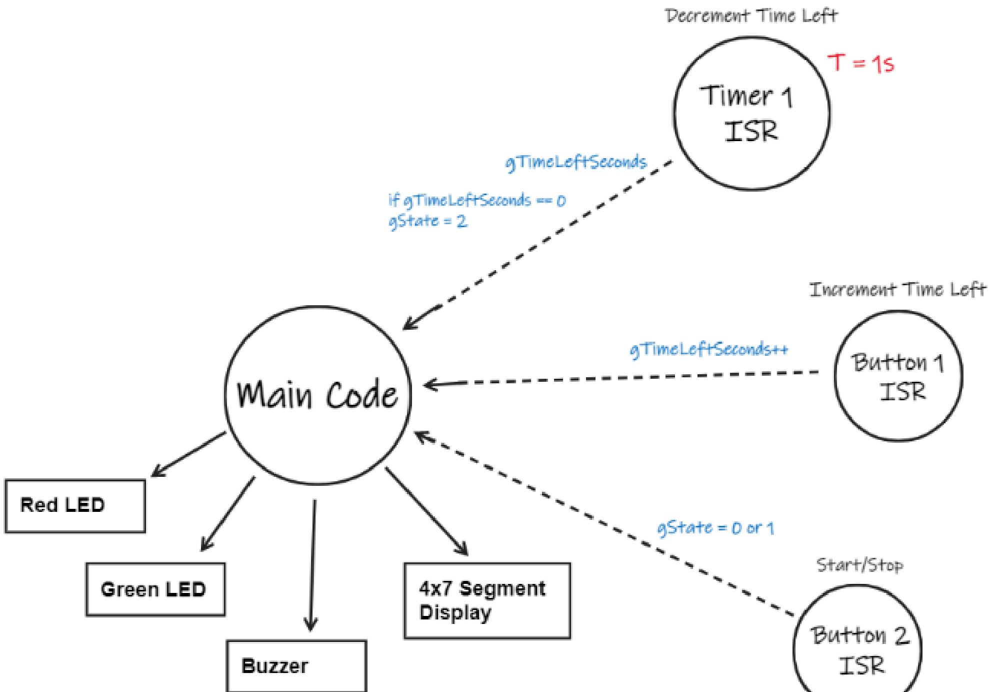
Appendix C: Phase A Schematic



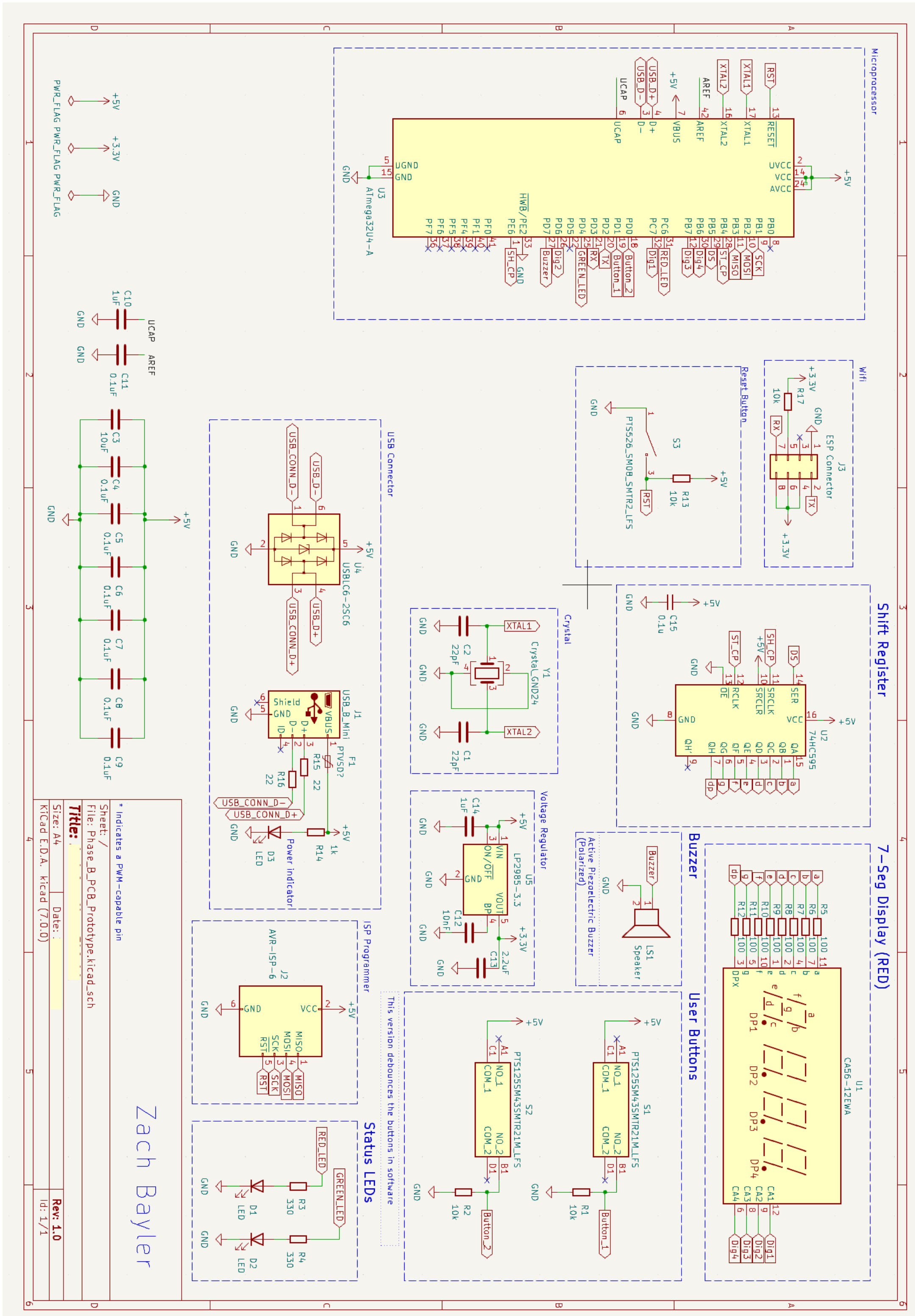
Appendix D: Phase A Routing



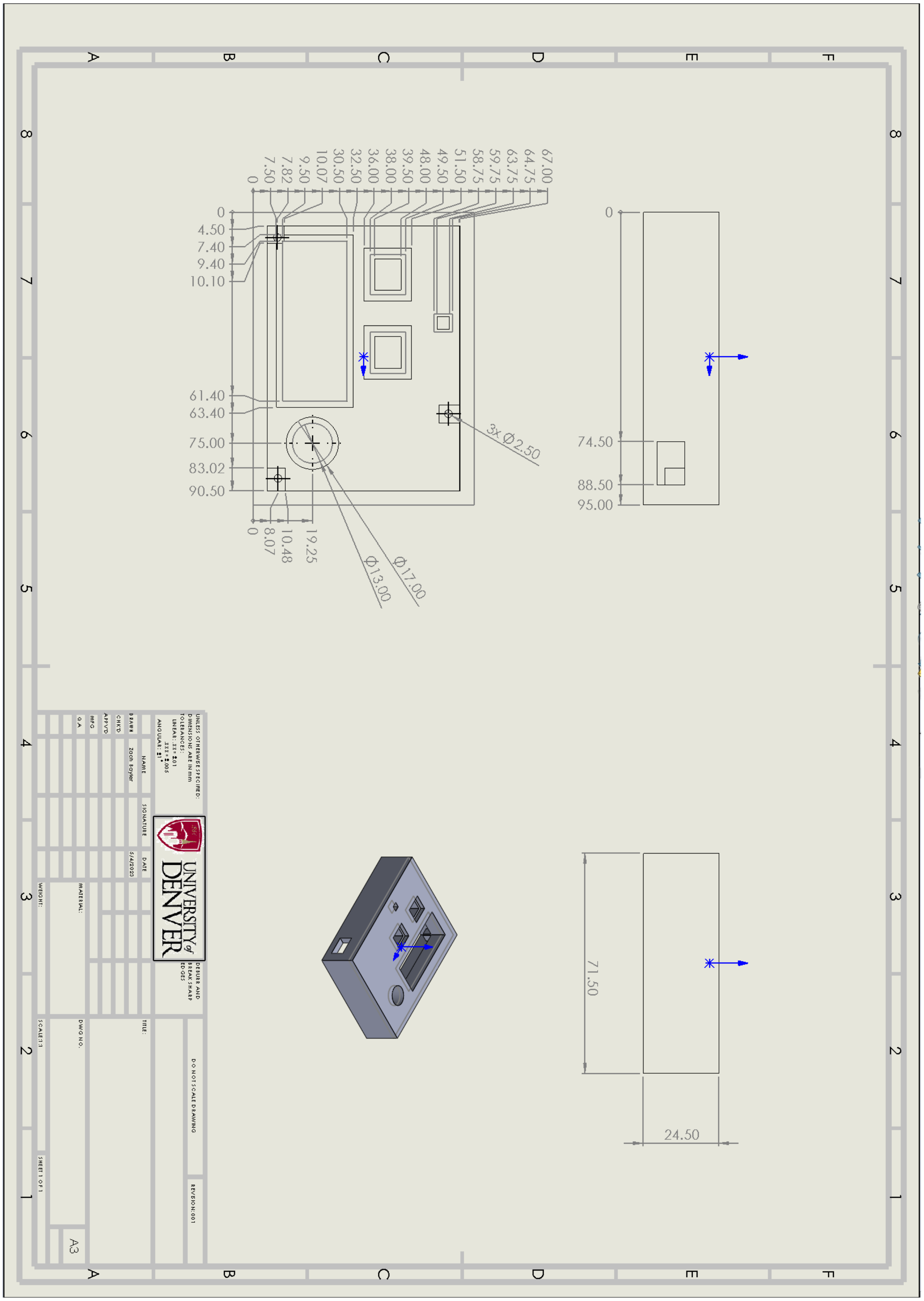
Appendix E: Phase A Code Block Diagram



Appendix F: Phase B Schematic



Appendix G: Engineering Drawing of the Enclosure



Appendix H: Phase B PCB Routing