Noe Sheridan

Beta Release Phase B

Embedded Systems Programming

Dr. Goncalo Martins

8th of June 2023

Project Requirements:

- Cost
    - Requirement not specified by customer. Efforts still made to reduce cost.
- Size
    - Enclosure maximum size 10 by 10 cm.
    - PCB less than the enclosure so that it can fit inside.
- Limitations of Timer
    - Due to size of 7-Segment Display, only four digits of time can be displayed. For clarity purposes this means the maximum time input for the timer is 59 minutes and 59 seconds. (59:59).
- LED Amount
    - Two LEDs used on PCB to communicate with user.
- Button Amount
    - Three buttons total, including the resent button which resets the entire timer, one to increment the timer, and one used to start and stop the timer.
- Type of Buzzer
    -
- Debounce for Buttons
    - To minimize hardware requirements and maximize space on the PCB, the debounce will be done through software.
- Shift Register
    - To minimize the number of Microprocessor pins used, a shift register will be used to communicate between the 7-segment display and the microprocessor.
- How Powered
    - Currently powered through USB connection with the Arduino Uno microprocessor. In the Future will be directly powered through the USB.

System Design

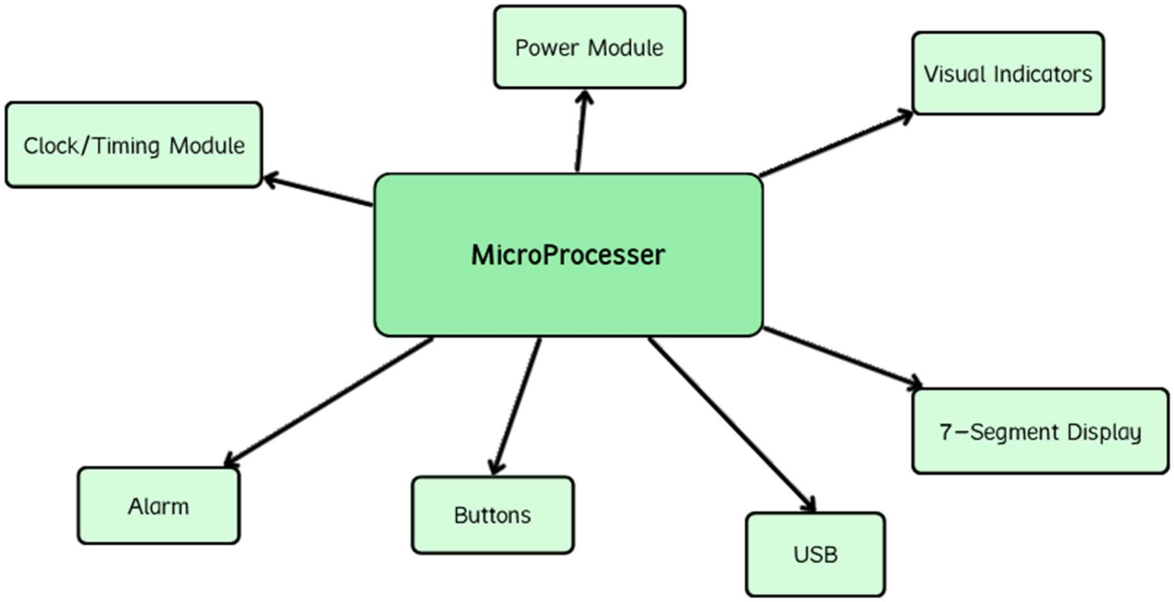

Figure 1: Block Diagram Version 1.0

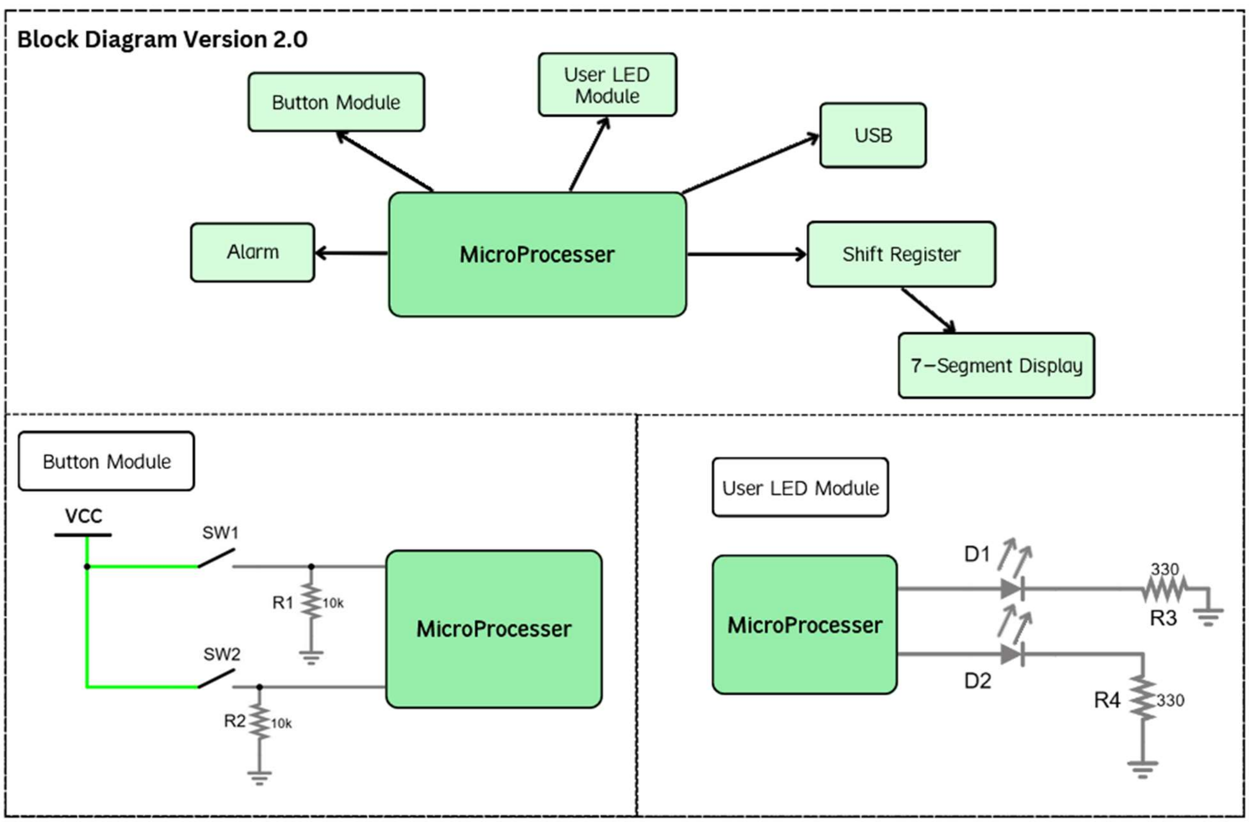

Figure 2: Block Diagram Version 2.0

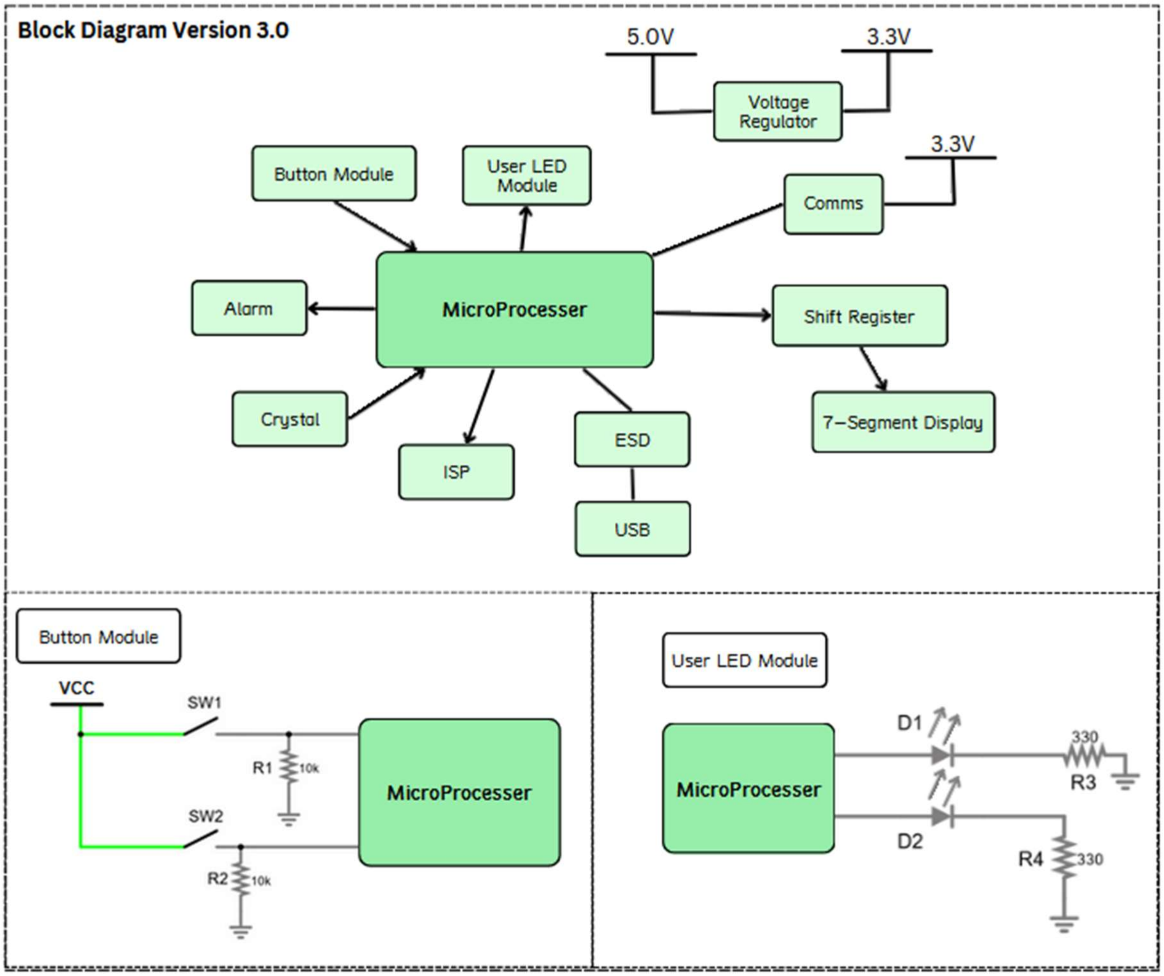

Figure 3: Block Diagram Version 3.0

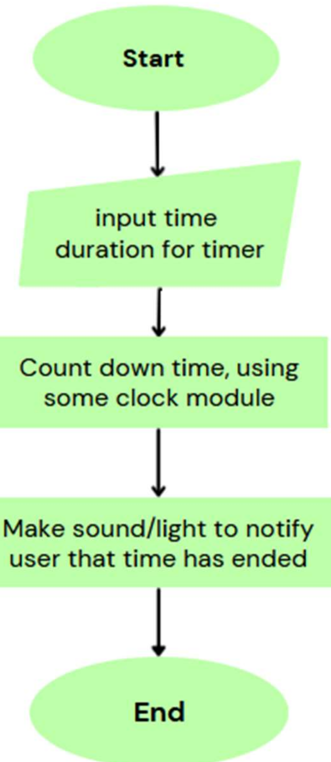

Figure 4: Signal Flow Diagram for System Ver 1.0

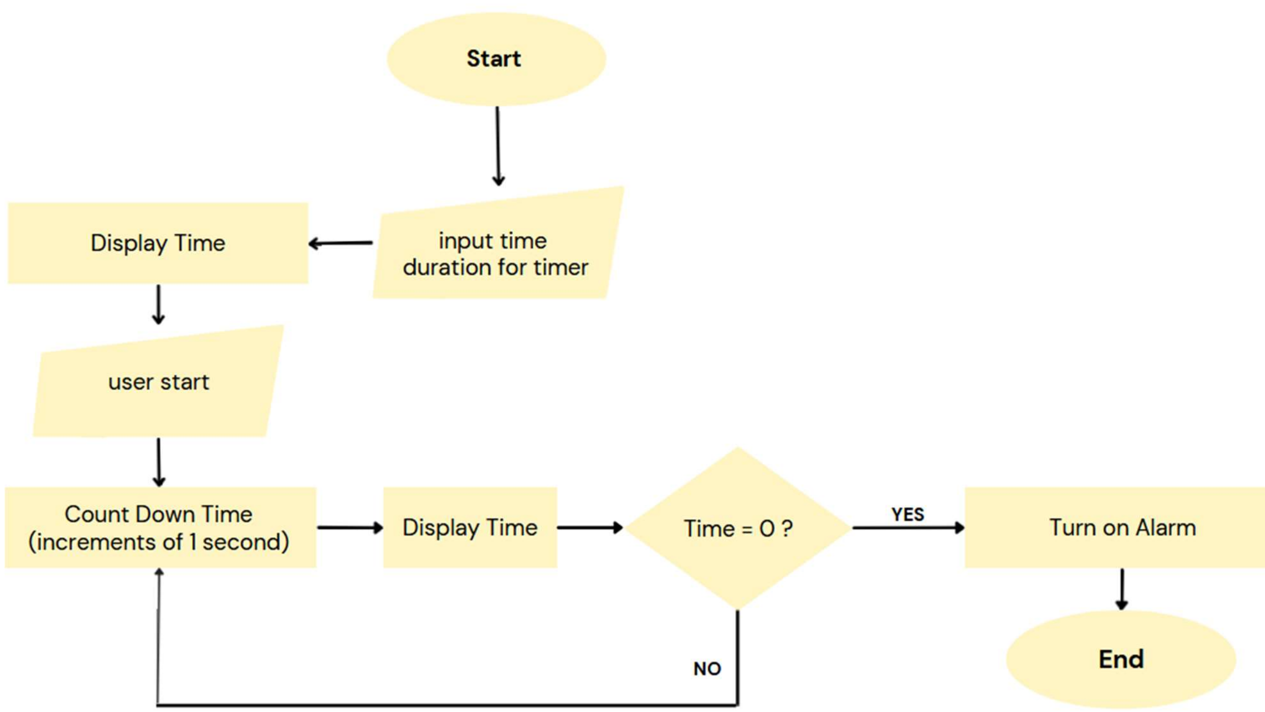

Figure 5: Signal Flow Diagram for System Ver 2.0

The figures above include Block Diagrams of the system design, and Signal Flow Diagrams for the same design, specifying the modules and some components to be used in the design and prototyping and the way in which the system will process signals for intended functionality as a user-programmable timer.

Components Selection

- Arduino Uno (selected for Phase A), Arduino Mega (preferred) Microcontroller
    - Arduino Uno selected for Phase A for accessibility, and already acquired prior to design stage. Used in first round of prototyping.
    - Price: $22.95 on Mouser Electronics.
- AT Mega 32 U4
    - Selected for Phase B thru Final Product development
    - 8-Bit Microcontroller, Arduino IDE compatible, selected as it is smaller than the Uno and more affordable at 24.44% of the Uno price.
    - Price: $5.61 per unit on Mouser.
- 7-Segment Display
    - The display on which the clock time will be displayed for user.
    - LTC-5723HR selected.
    - Price: $2.97 per unit on Mouser
- 8-Bit Shift Register
    - Used to interface between the Uno and the 7-Segment display, reducing the pins on the microcontroller used to program the 7-segment display.
    - Texas Instruments SN74HC595PWRG4 shift register selected for further prototyping.
    - Price: $0.62 per unit on Mouser.
- Buzzer
    - Used as alarm module to notify user when the timer has run out.
    - CMI-1295 Buzzer selected for use.
    - Price: $1.18 per unit on Mouser
- Buttons
    - Phase A Prototype:
        - Only two buttons used for user input, one for Start/Stop commands, and the other to positively increment the timer duration in seconds. Using negative logic.
    - Phase B-Onward Prototypes:
        - Three user buttons used. The two specified in Phase A, and a third button to act as Reset for the timer module. Using negative logic.
        - Start/Stop and Increment Buttons are PTS125 Tactile Switch buttons, $0.43 per unit price on Mouser.
        - Reset Button is PTS526 Tactile Switch buttons, $0.13 per unit price on Mouser.

        - Total Buttons Price: $0.99
- USB Connector
    - Used for communication with programming computers and to connect to external power supply.
    - UJ4-MBH-4-SMT-TR selected for prototyping.
    - Price: $0.88 per unit on Mouser.
- LEDs
    - Used to communicate with user about connectivity and operation modes.
    - LED_0805_2012Metric selected for prototyping. One Green, one Red, and one Blue.
    - Green Price: $0.143 per unit when purchasing 100 on Mouser.
    - Blue Price: $0.131 per unit when purchasing 100 on Mouser
    - Red Price: $0.151 per unit when purchasing 100 on Mouser

Build Prototype (Phase A)

For Phase A of prototyping, a ‘shield’ was made to work with the Arduino Uno. It was called a shield as the design fit over top of the Arduino Uno controller board. As this first design used the Uno, internal timers, USB communications, and power supplies were used. As such the shield PCB only needed to include the two user buttons (Start/Stop and Increment), the shift-register, the Buzzer, the two user-LEDs, the 7-Segment display, and all the resistors required for optimal operation of these components. The schematic and a scan of the routed shield design are shown below (Figures 6 and 7).

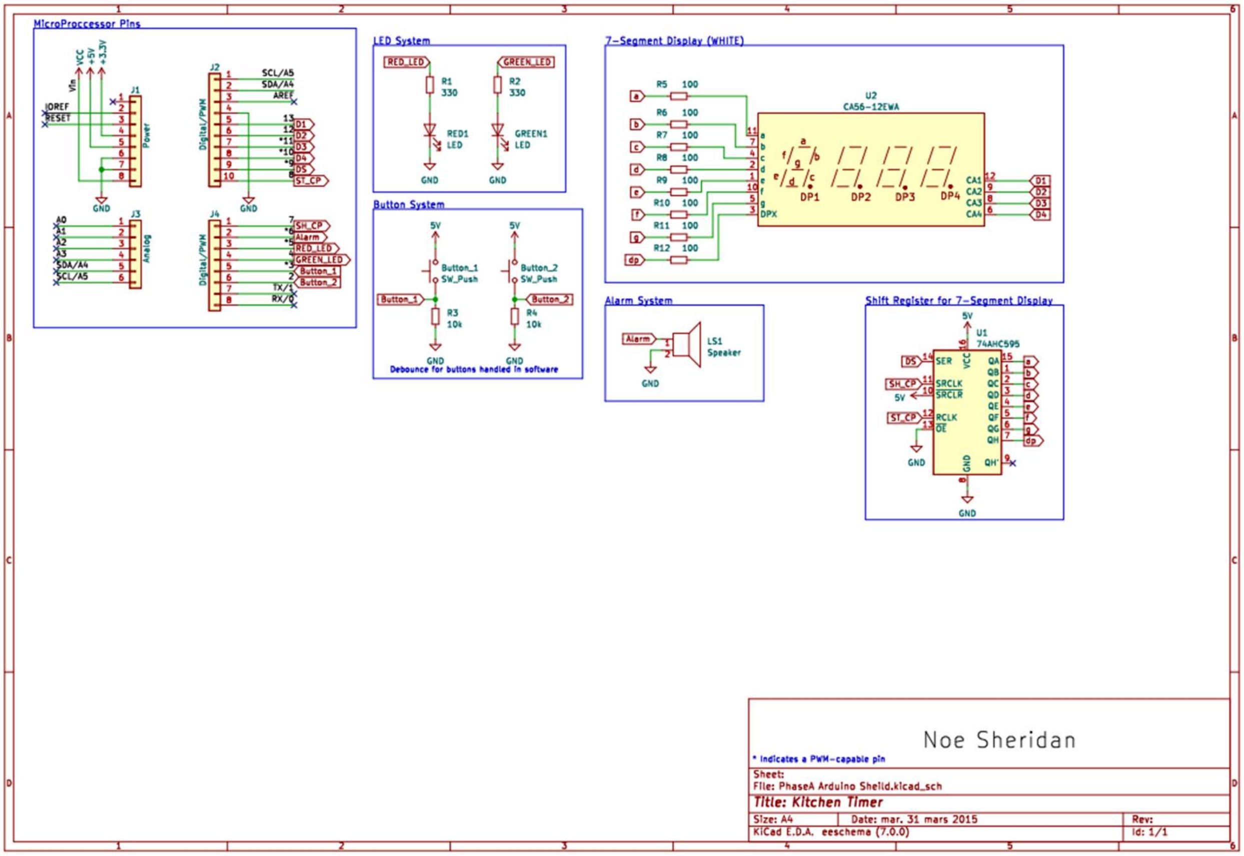

Figure 6: Arduino Uno Shield Prototype Schematic (Phase A)

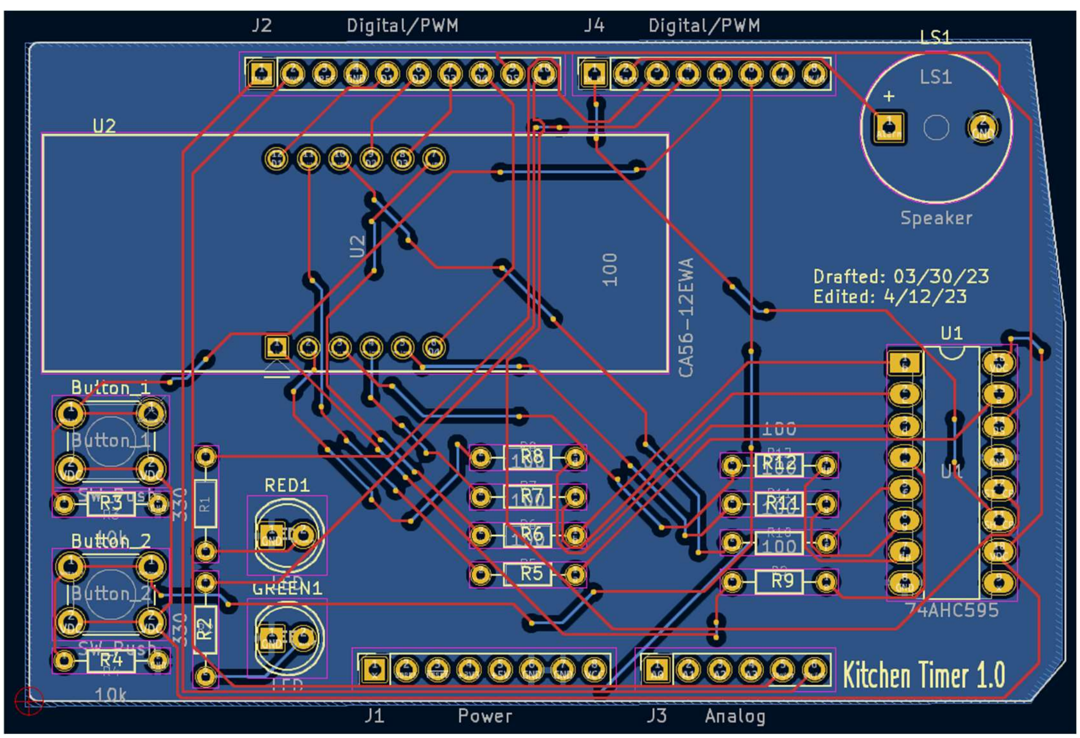

Figure 7: Arduino Uno Shield Routed PCB (Kitchen Timer Ver 1.0)

PCB Design

The figures below detail the current PCB design, which will utilize the AT-Mega microcontroller instead of the Arduino Uno. With this change in controller the inclusion of additional modules on the PCB are required for functionality. Including a voltage regulator, a crystal, many capacitors, a reset button, an ISP programmer, and a USB connector. Mounting holes were also added to mount the PCB within the enclosure. The PCB has a ground plane on the back side to reduce the amount routing. The schematic, images of the routed PCB front and backside, the 3-D view of front and back sides are shown below in Figures 8-12.

This design measures at 63.0 mm wide and 87.0 mm long with three mounting holes of diameter 3.0 mm.

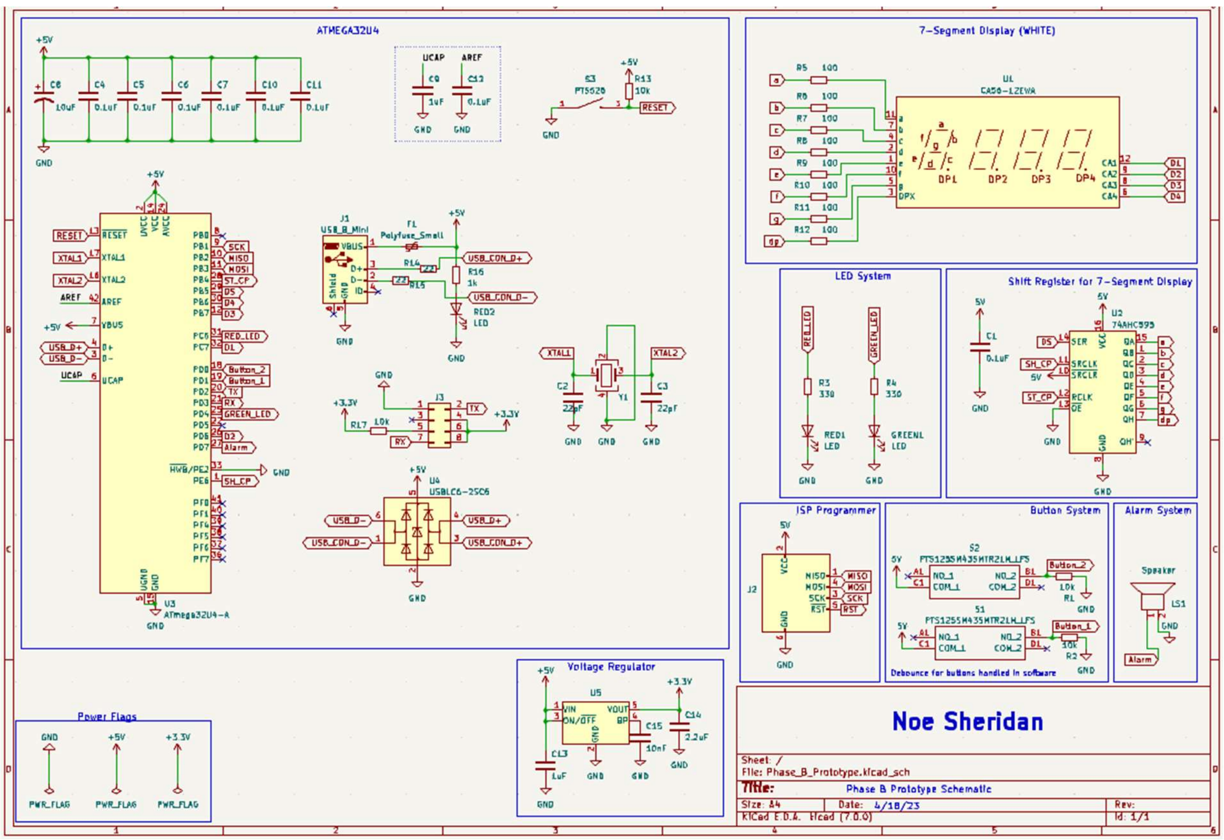

Figure 8: Schematic for the Phase B Prototype

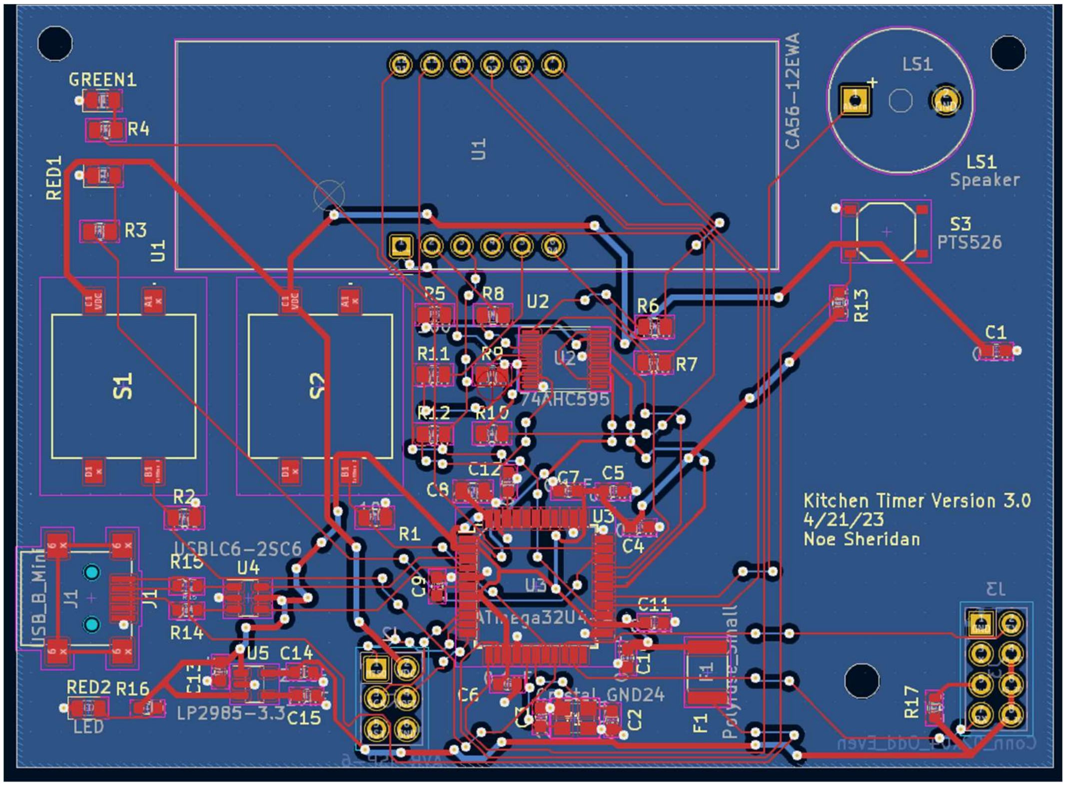

Figure 9: Routed Kitchen Timer PCB

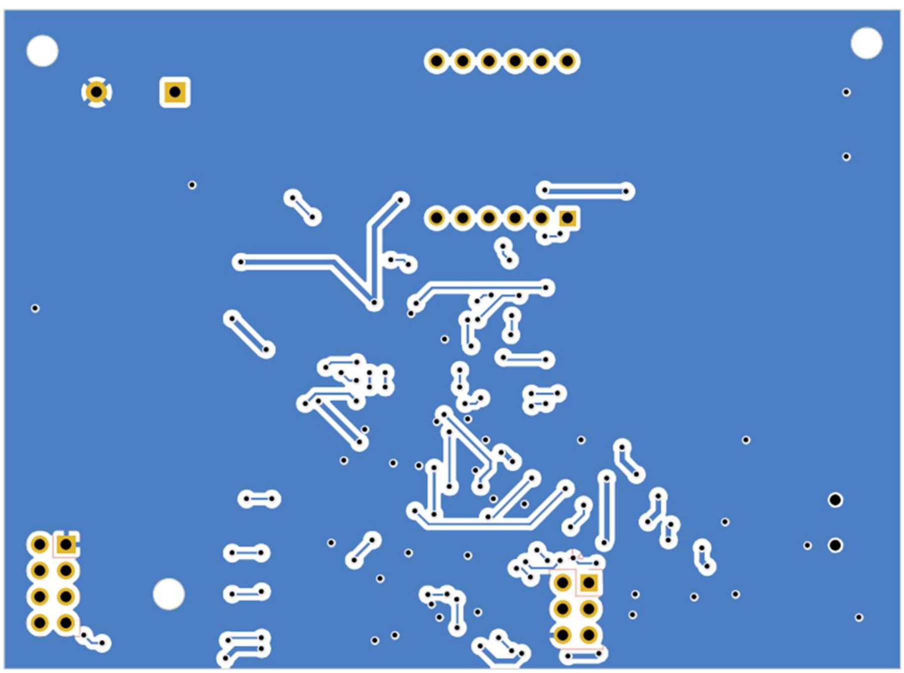

Figure 10: Routed Kitchen Timer PCB Backside (Ground Plane)

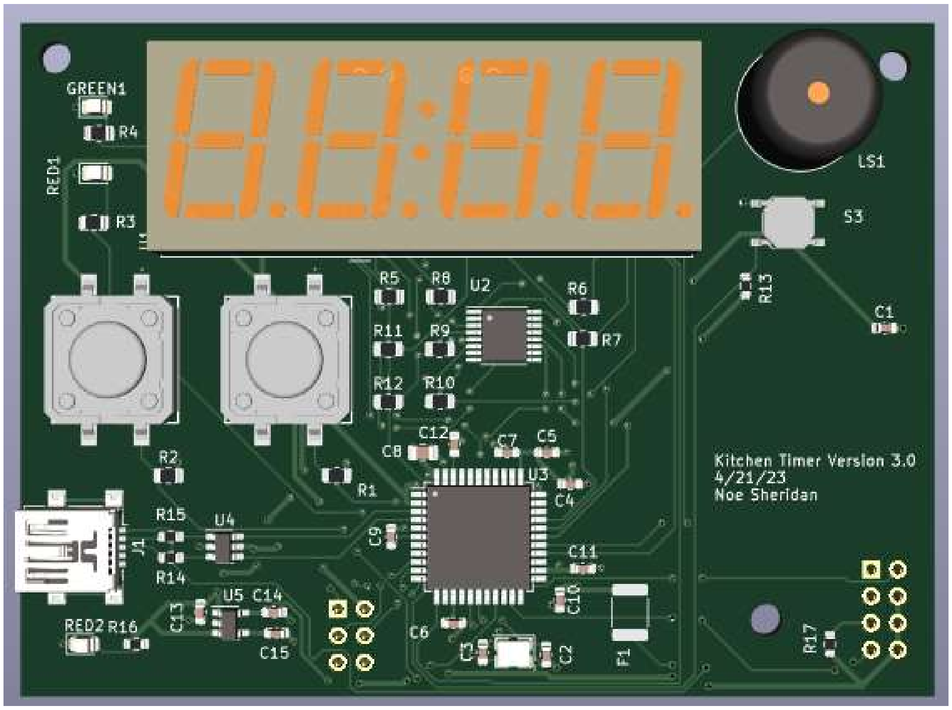

Figure 11: 3-D View of Routed PCB Front Side

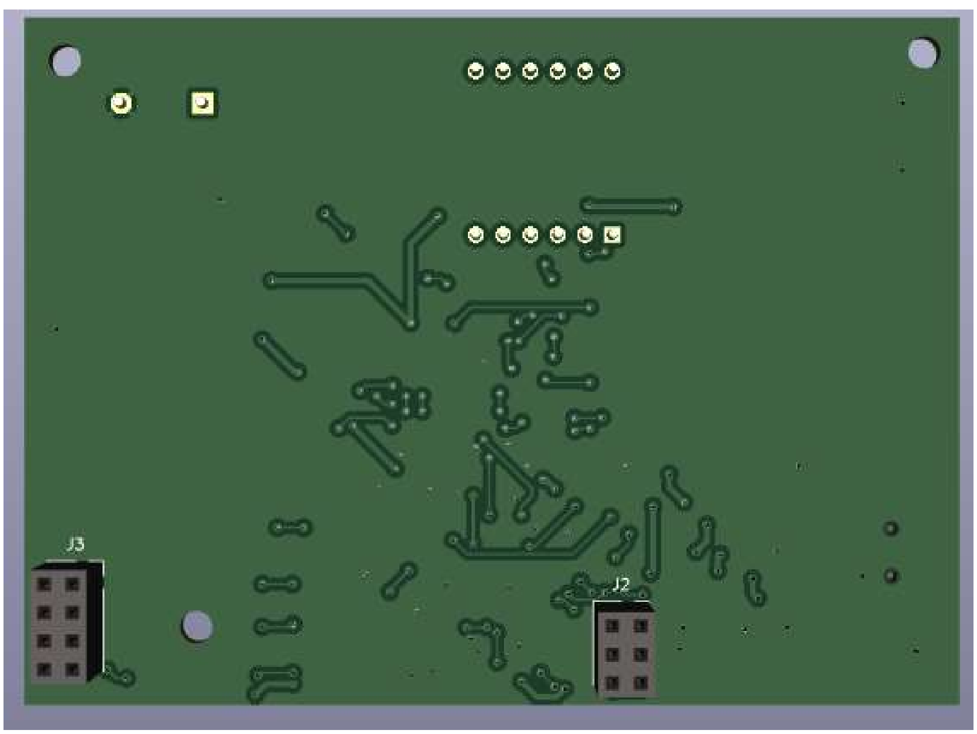

Figure 12: 3-D View of Routed PCB Backside

Assembly Stage

The PCB design detailed in the above section was sent to a Fab House and five copies were fabricated. To assemble the PCB shown bellow the surface mounting components were soldered first, starting with the AT-Mega 32 microprocessor (labeled U3), the 8-bit shift register (labeled U2), the USB-block connector (labeled U4), and the voltage regulator (labeled U5). Then the crystal (Y1), and the USB-port connector (J1) were soldered on by the instructor, Gonçalo Martins. Working outwards from the center, the surface-mounting capacitors, LEDs, and resistors were soldered to the board, leaving the through-hole components to be soldered last, as these components were much bulkier and harder to work around, once placed and secured on the PCB. This PCB was non-functional, specifically in that it was not recognized by the IDE set up to test the boards with an example program (which tests the LEDS and the Buzzer/Speaker). Many of the components were tested to determine if there were cold joins. Unfortunately, the source of this error is still unknown. Also, the USB LED is labeled incorrectly as RED2, when it should instead be BLUE1.

It is my inference that the placement of the decoupling capacitor (C1) relatively far from the microprocessor may be introducing an error in the board, however I am not sure if that specific error caused the board's malfunction.

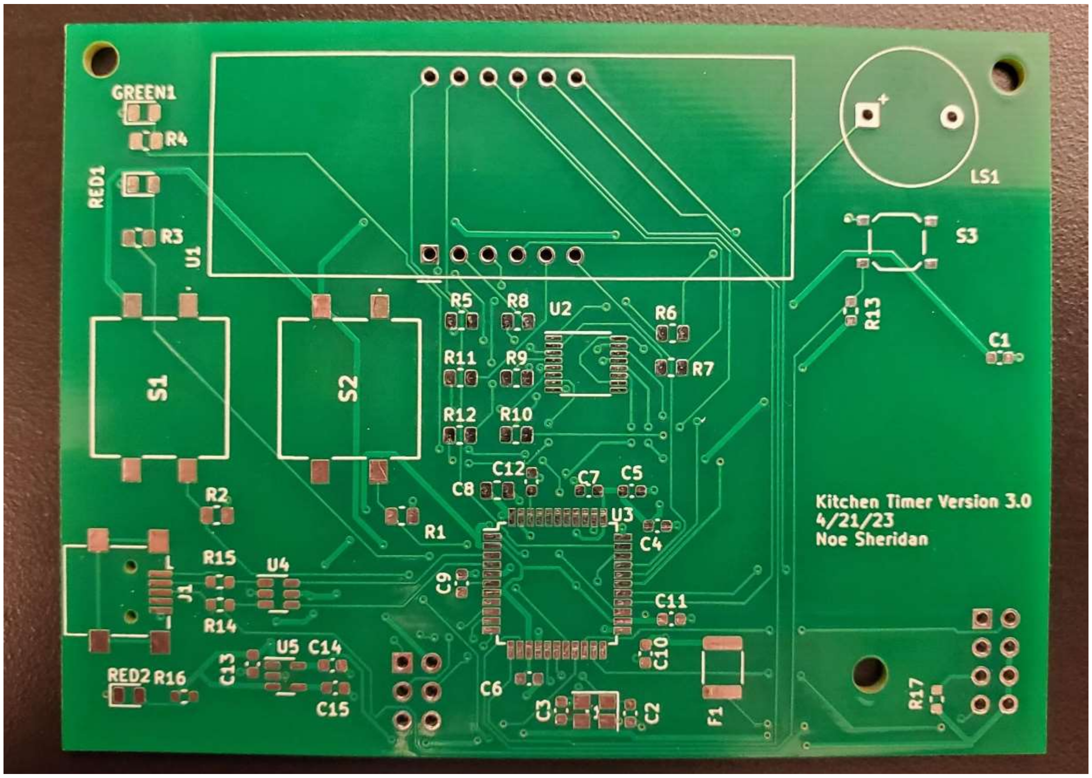

Figure 13: Physical PCB Prior to Assembly Process

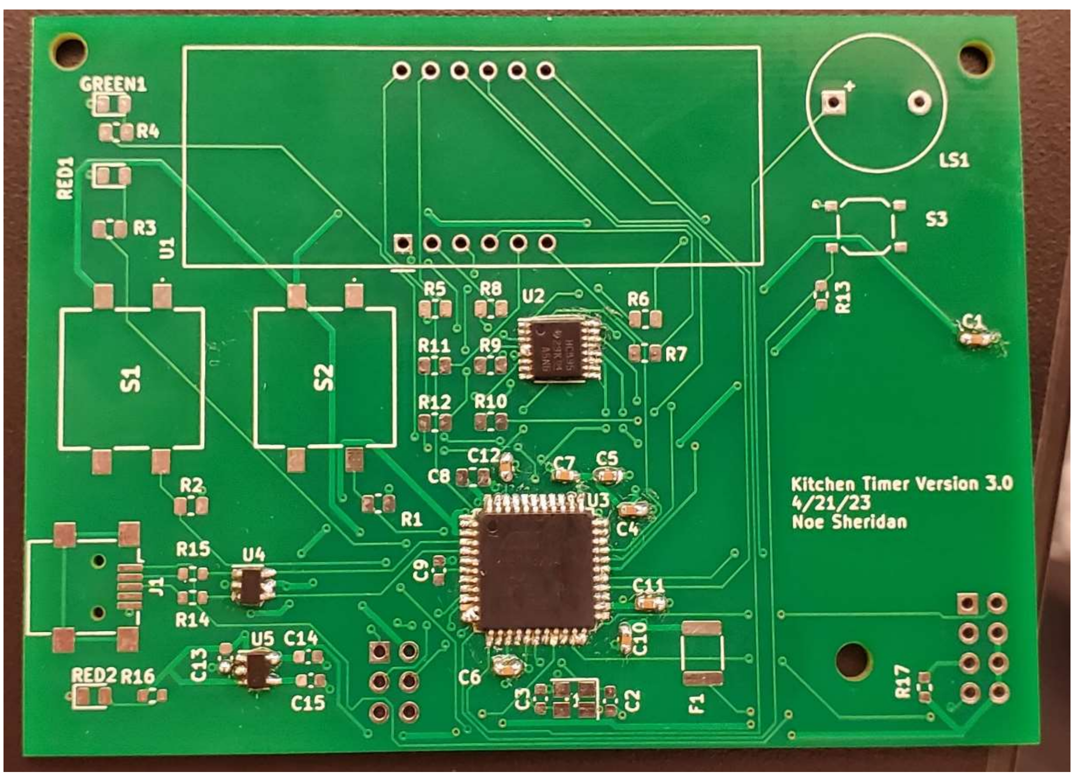

Figure 14: Physical PCB Part-way Through Assembly Process

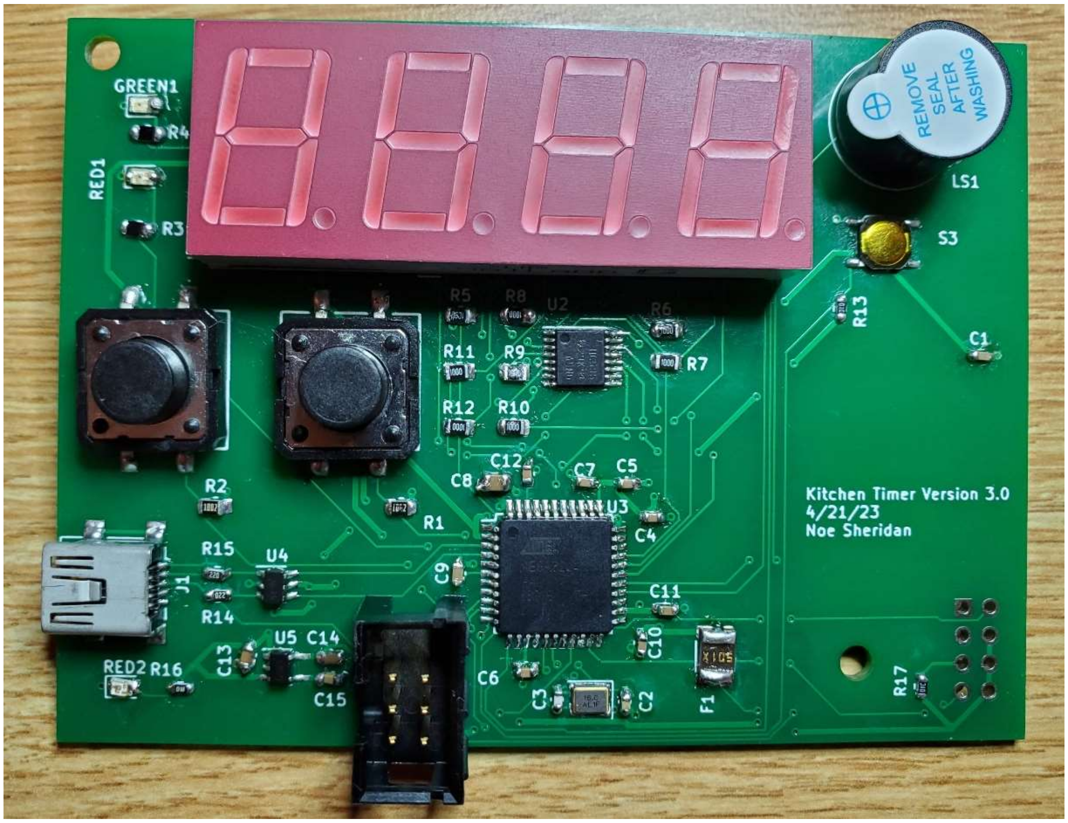

Figure 15: Assembled Physical PCB (Pin connector for Wi-Fi Module Skipped)

Software Design

This section was not included as the Physical PCB cannot be recognized by a computer, and thus the AT-Mega32 cannot be programmed. No further documentation or code development occurred because of this.

Enclosure Design

The first draft enclosure designed below is 94mm wide and 65mm tall, (9.4 x 6.5 cm). There are extruded cuts in the front to allow the user access to the buttons, LED, 7-segment display, and to allow room for the buzzer. On the user's left side there is an extruded cut to allow access to the USB connector. And on the back plate there are extruded cuts to give user access to the fuse connector and the connector for the Wi-Fi module, as well as circular extruded cuts which line up with the mounting holes on the PCB for mounting the PCB within the enclosure.

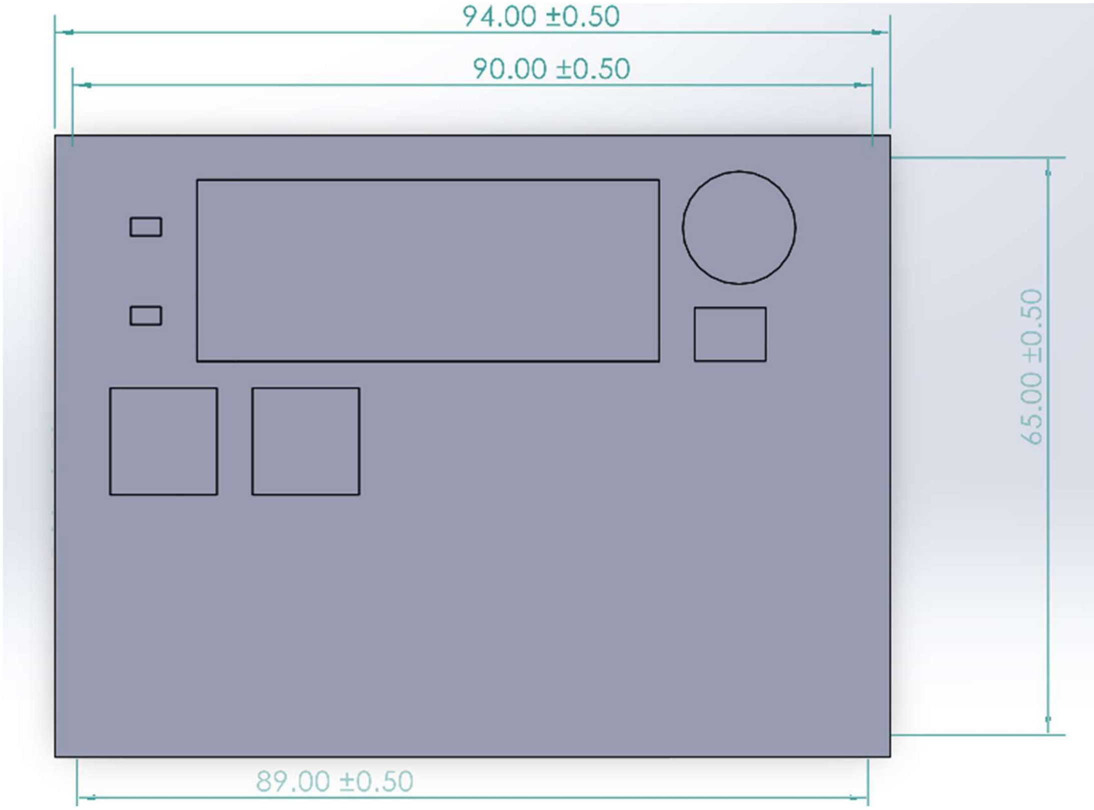

Figure 17: Front View of First Draft PCB Enclosure

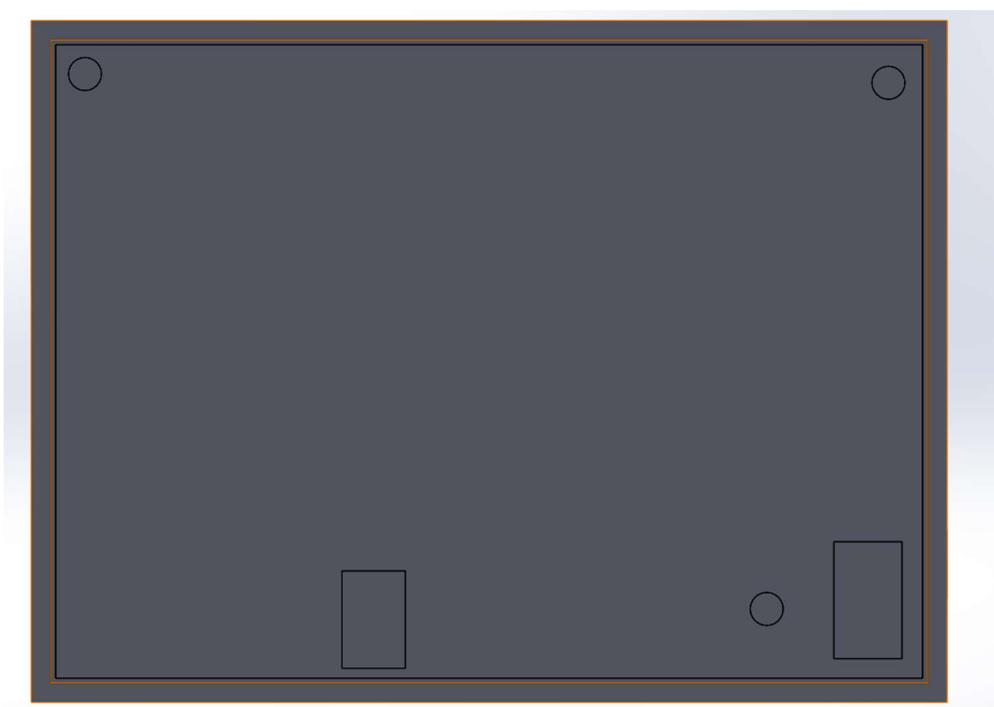
Figure 18: Back view of First Draft PCB Enclosure

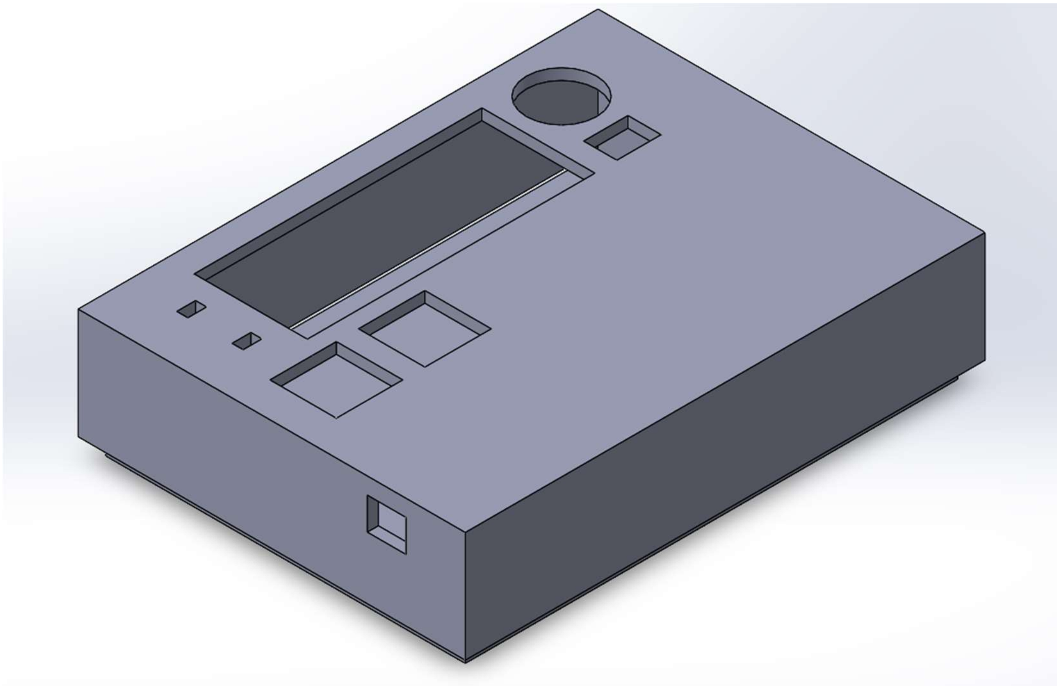

Figure 19: Isometric View of First Draft PCB Enclosure

After the assembly stage, this design was altered as the connection mounts J2 and J3 were configured for top/front side mounting even though the 3-D render of the PCB design shows otherwise, the first draft was also missing a hole through which to view the USB LED (labeled RED2). The size of the holes for the pin connectors J2 and J3 were also increased to create suitable clearance for the larger footprints of the physical components compared to those shown in KiCAD. This altered design is shown below in Figures 20-22. The engineering drawing for this design can be found in Figure 23.

The printed prototype images can be found below as Figure 24. With Figure 25 depicting the Assembled PCB inside the enclosure.

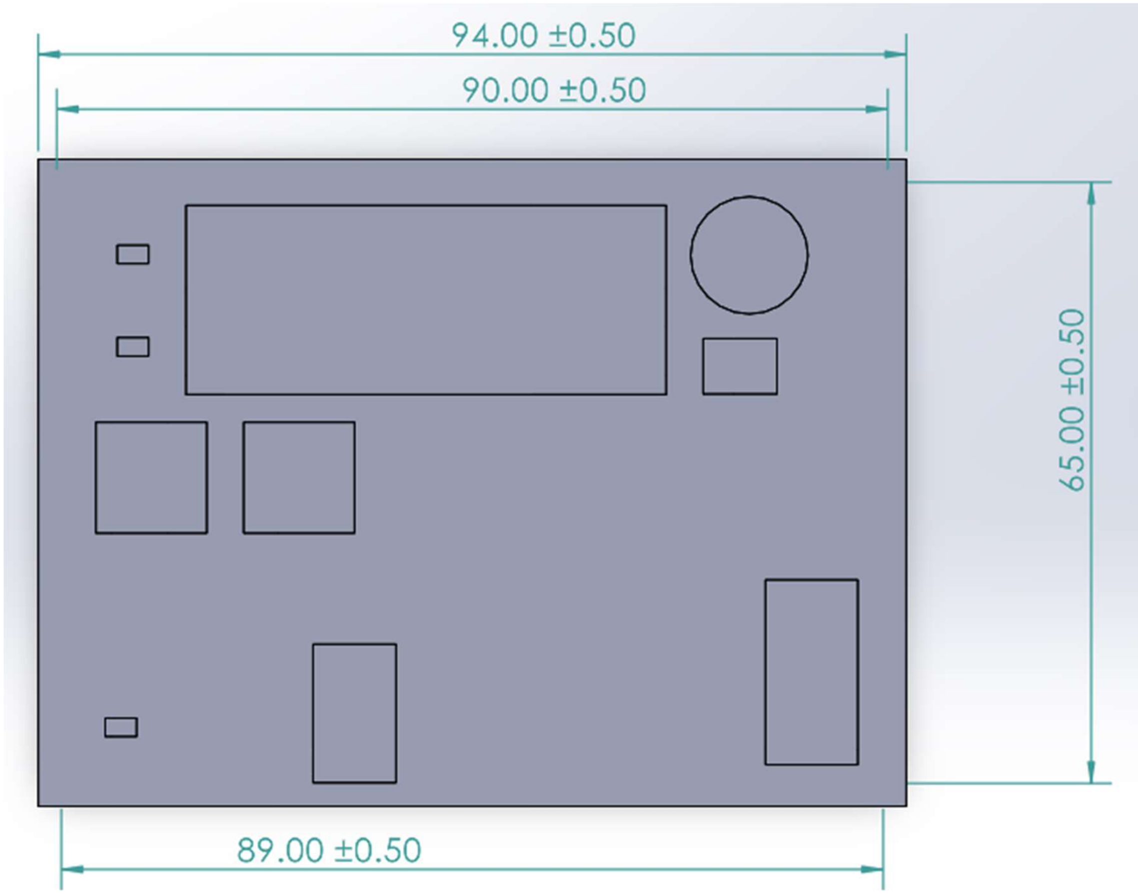

Figure 20: Front View of Final PCB Enclosure

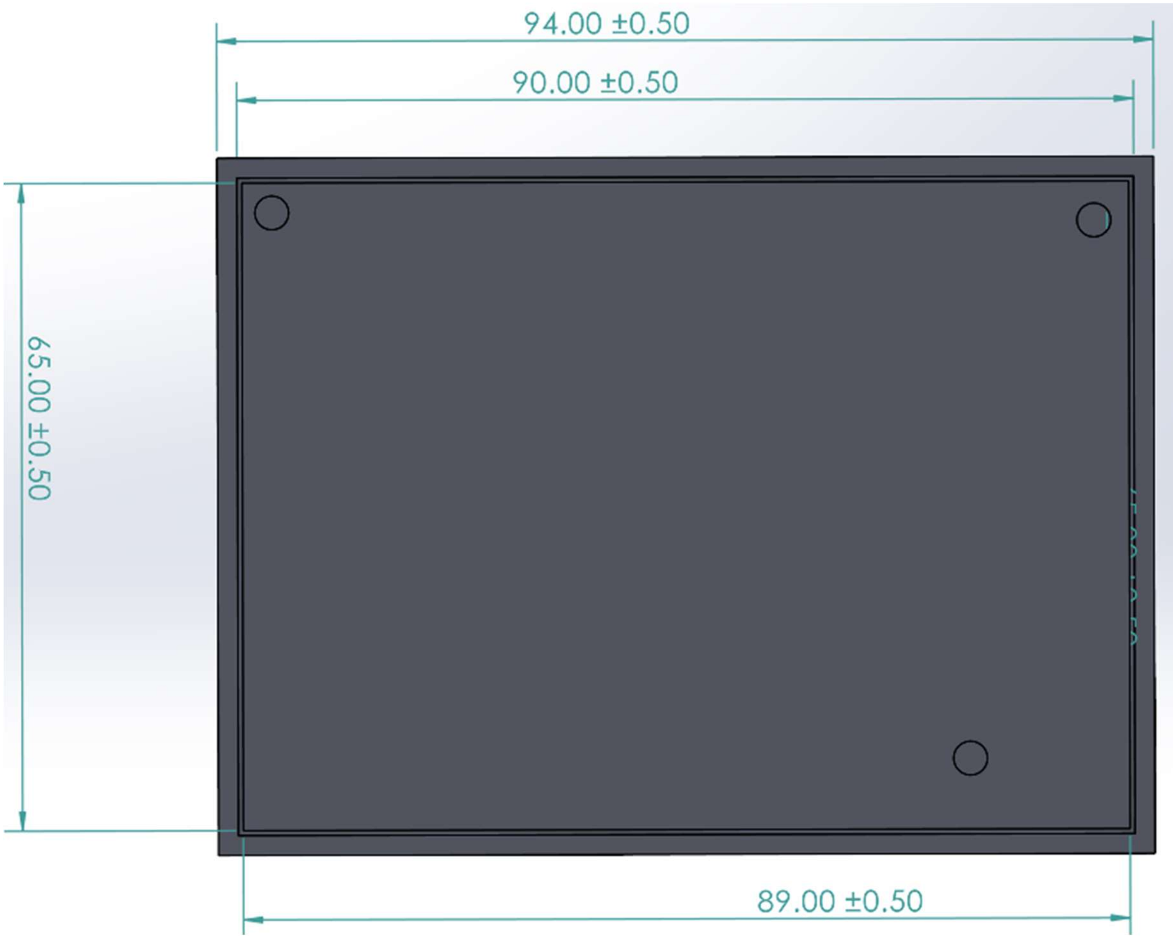

Figure 21: Back View of Final PCB Enclosure

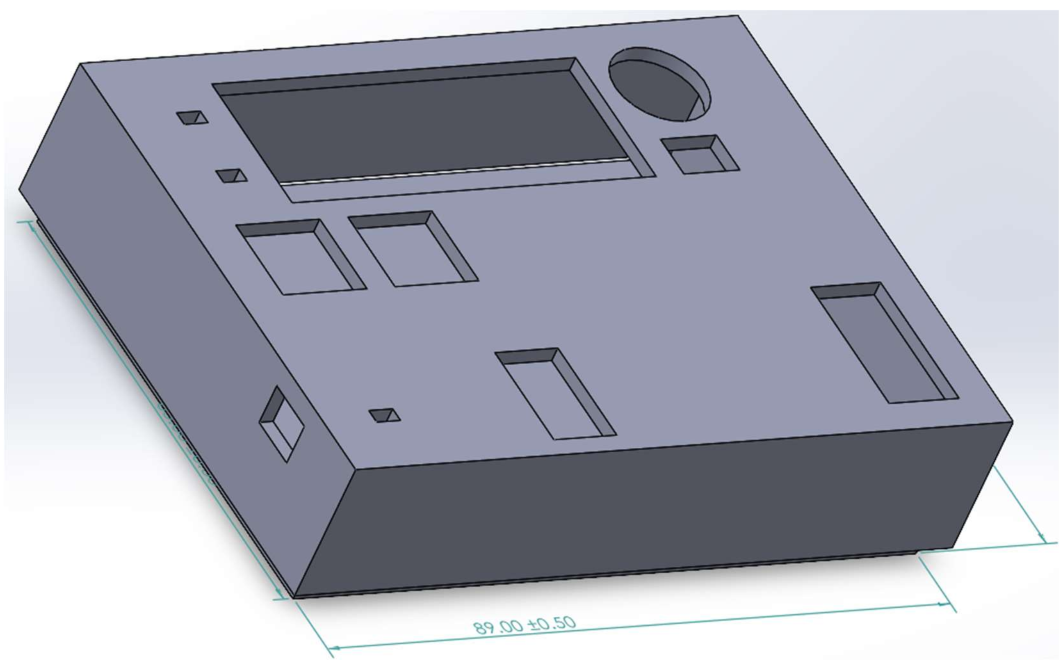

Figure 22: Isometric View of Final PCB Enclosure

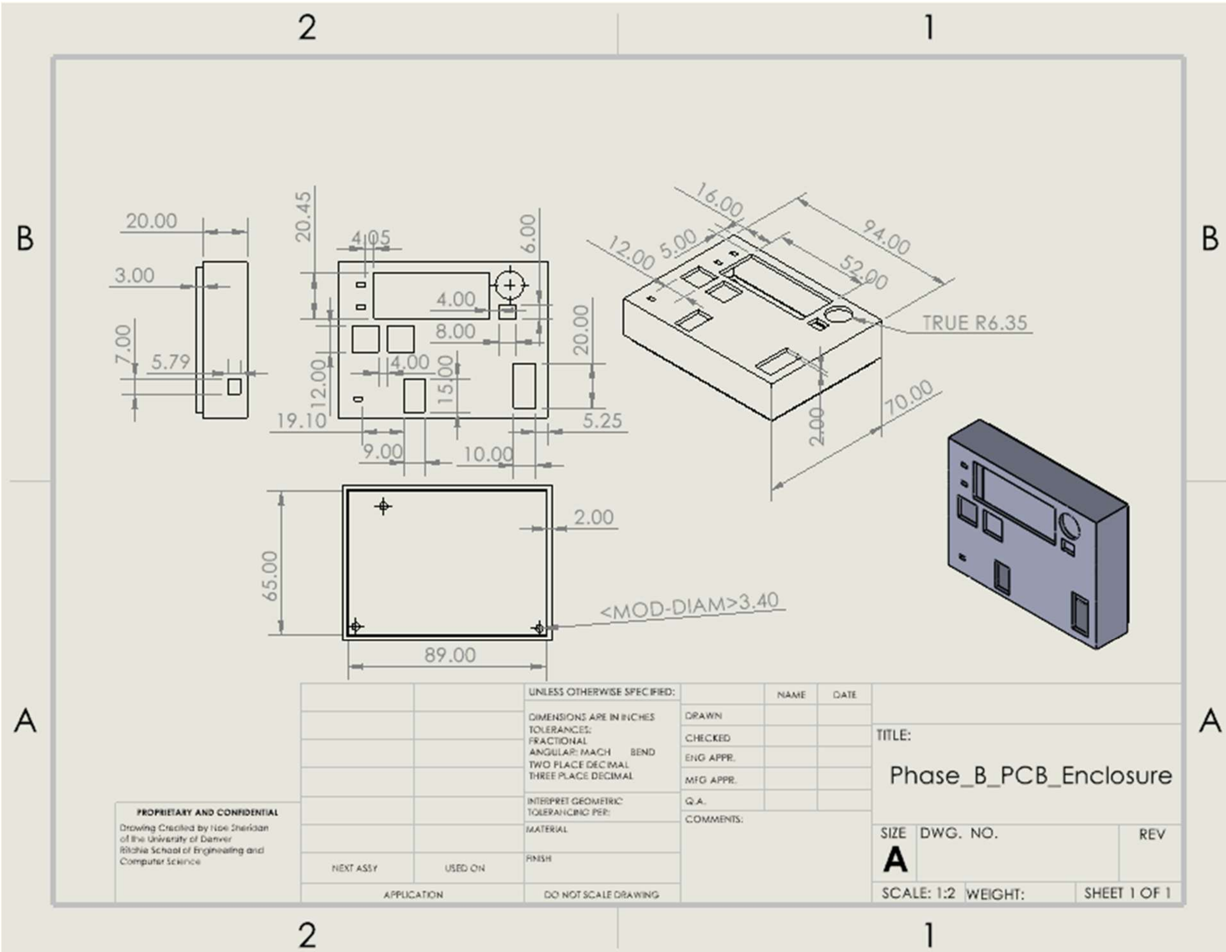

Figure 23: Engineering Drawing for Final PCB Enclosure

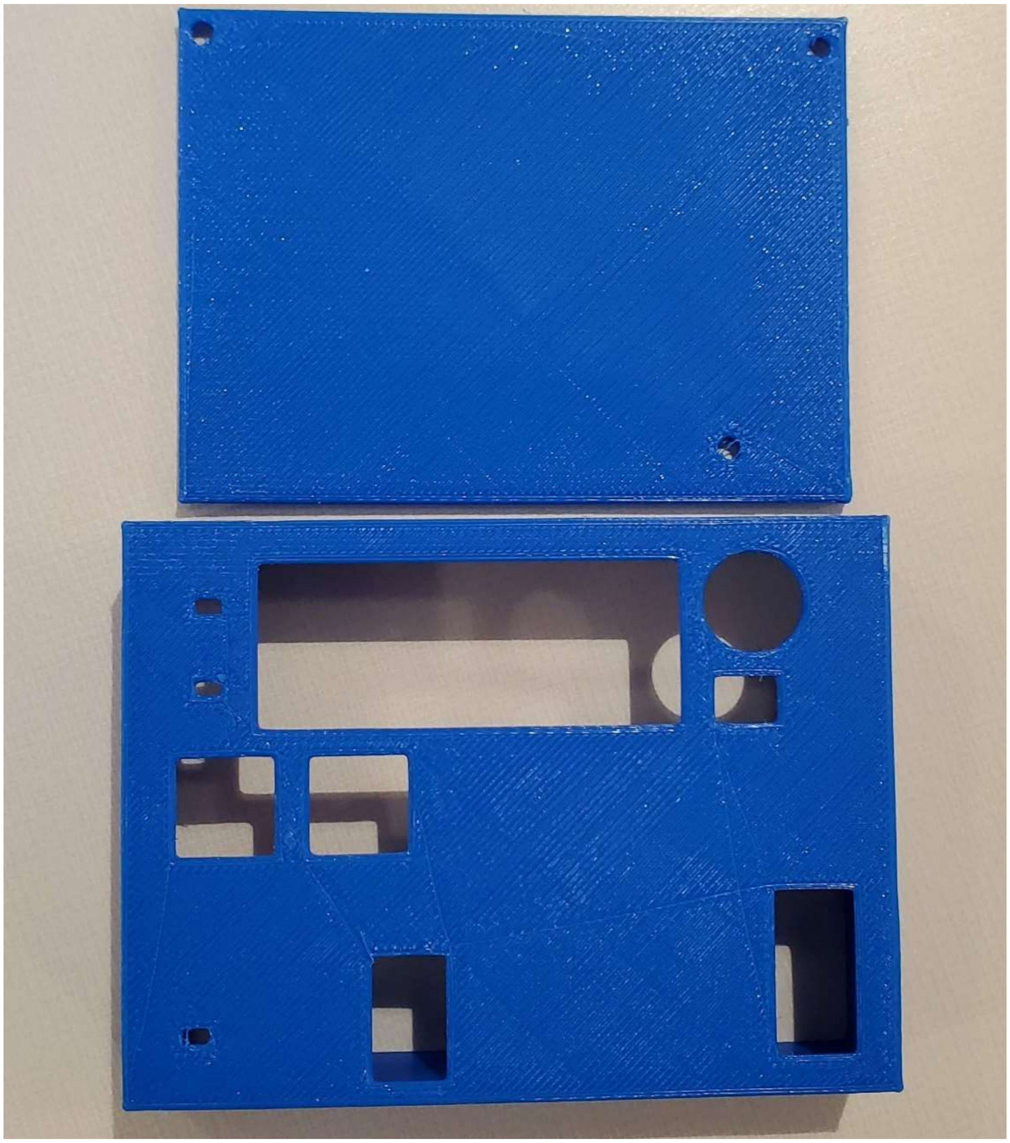

Figure 24: Physical PCB Enclosure

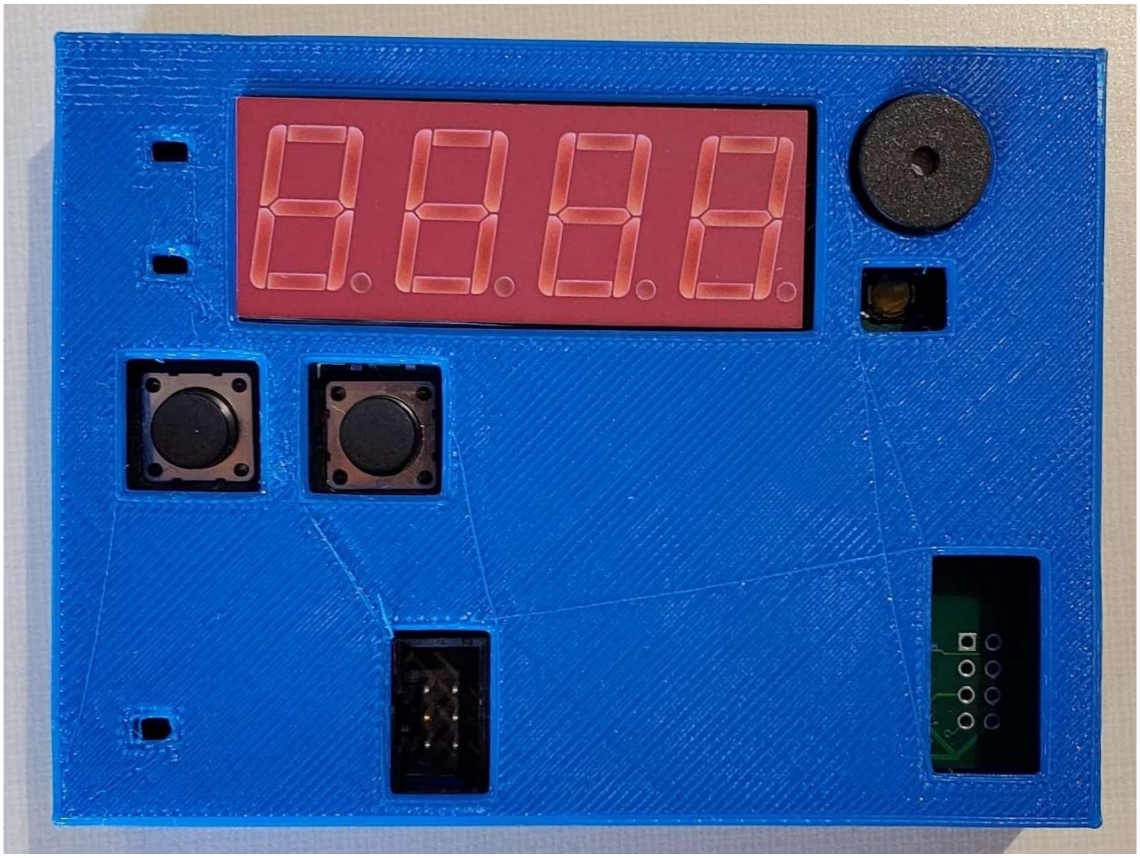

Figure 25: 3-D Printed PCB Enclosure with Assembled PCB inside