PCB Design of a Simple Embedded System

By Victory Odhiambo

Introduction: Throughout the semester we dissected and analyzed the operation of a Printed Circuit Board (PCB) theoretically and the ideal designs of the board. This project finally provided an opportunity to experimentalize the vast concepts that had been taught. Prior to the implementation of the board, key rules and guidelines for components and trace placements had to be revisited. The primary objective of this project was to design an efficient and reliable PCB for an STM32F4 MCU with the aid of a reference video but tailored to meet specific size requirements. The STM32F4 series represents powerful ARM Cortex-M4 microcontrollers that belong to the ARM Cortex M4 family. These microcontrollers are known for their high performance and can be used in a wide range of applications. Alongside that, other objectives included generating Gerber files, and Bill of Materials (BOM) that would be useful in PCB manufacturing. The design task was to familiarize us with the KiCAD environment, the process of translating schematics into board designs, and the nuances of PCB layout considerations, as discussed in the lectures.

Method: Using Kicad, the STM32F4 MCU project schematic that consisted of different sections was first designed. The sections included the Power Circuitry which entailed components like diodes, capacitors, inductors, and a step-down converter, the STM32 Microcontroller, Connectors and USB, an Oscillator Circuit, some Status LEDs, and Communication Interfaces as shown in Figure 1. Snapeda and Ultralibrarian are platforms that offer a comprehensive collection of electronic components and were utilized for the components that could not be found within the Kicad library. Subsequently, a 4-layer PCB board comprising the listed components, vias, and traces between the different components was established as shown in Figure 2. The board size was restricted to at most 30mm by 25mm. Due to the limited PCB measurement, both the top and bottom surfaces of the board had to be utilized for the strategic placement of components. Several aspects of the board also had to be scaled down such as the trace width, diameter of the vias, and mounting holes. Ground was poured onto the whole second layer to counter potential electromagnetic interference with a few traces routed through it. Power was poured onto the third layer, and it similarly had some traces routed on it. Following that, Gerber files, and Bill of Materials were generated. For the Bill of Materials (BOM), JLCPCB was utilized to get the manufacturers' details, vendors' details, and prices for each component.

Results: On the project schematic, Design Rules Check (DRC) was run and initially some errors were encountered. After some research into the matter, the addition of power flags (PWR_FLAG) into the schematic by connecting them to the affected power supplies and ground pins, cleared the schematic of all errors. The Design Rules Check (DRC) was also run on the PCB board after its implementation and similarly, several errors were faced. Most of the errors stemmed from overlapping components, and components being too close to one another, traces, or to the vias. Other errors emanated from silkscreen constituents being clipped by solder masks. These errors were rectified by a close physical examination of the board while adjusting the position of the components. Eventually, a satisfactory 3-D PCB board was attained and observed as shown in Figure 3 and Figure 4. Gerber files were created and observed. From the JLCPCB a Bill of Materials was created and observed.

Conclusion: In conclusion, the successful design of STM32F MCU facilitated a deeper understanding of embedded systems and microcontrollers. This was a simple embedded system design that provided a stepping stone to the larger and more complex realms of embedded systems. Among the concepts that this project made clearer include but are not limited to the creation of footprints, signal, ground, and power distribution in a multi-layer PCB, and efficient use of the size provided for the design. Rectification of the errors that were observed when running the DRC further boosted the familiarization of Kicad. The experiments substantiated the theoretical concept studied in class providing a backbone to the abstract knowledge that was taught. Future sessions could include the manufacture and real-life testing of the PCB board design. Further, the characteristics and properties of the materials used for PCB manufacture should be put to the test to observe properties such as their resistance to temperature, adhesion, dielectric strength, and more.

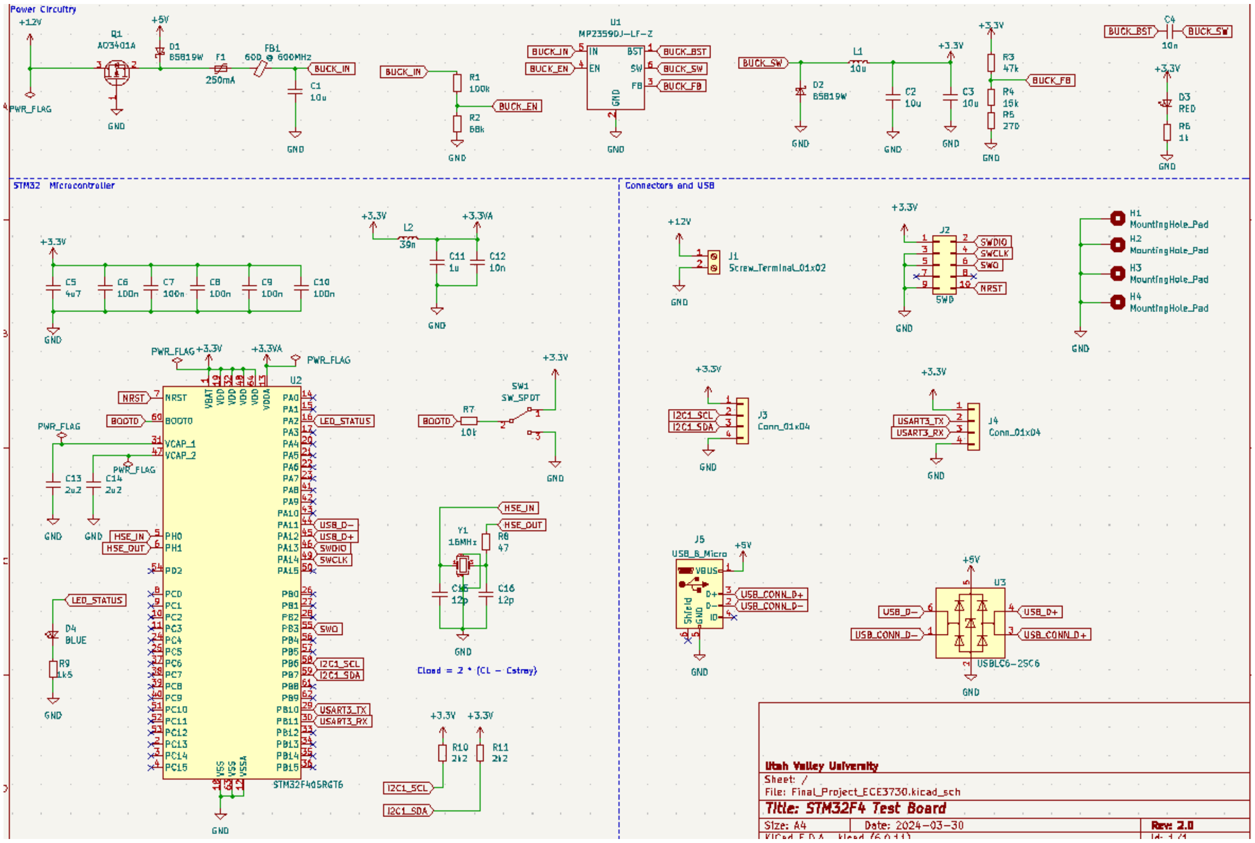

Figure 1

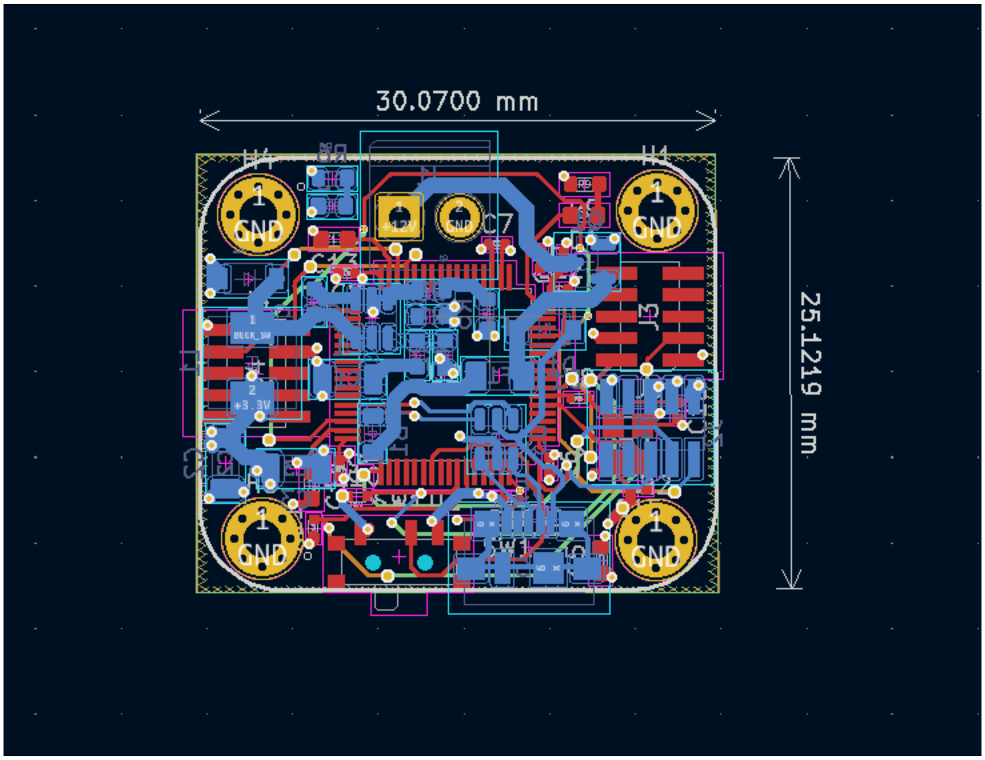

Figure 2

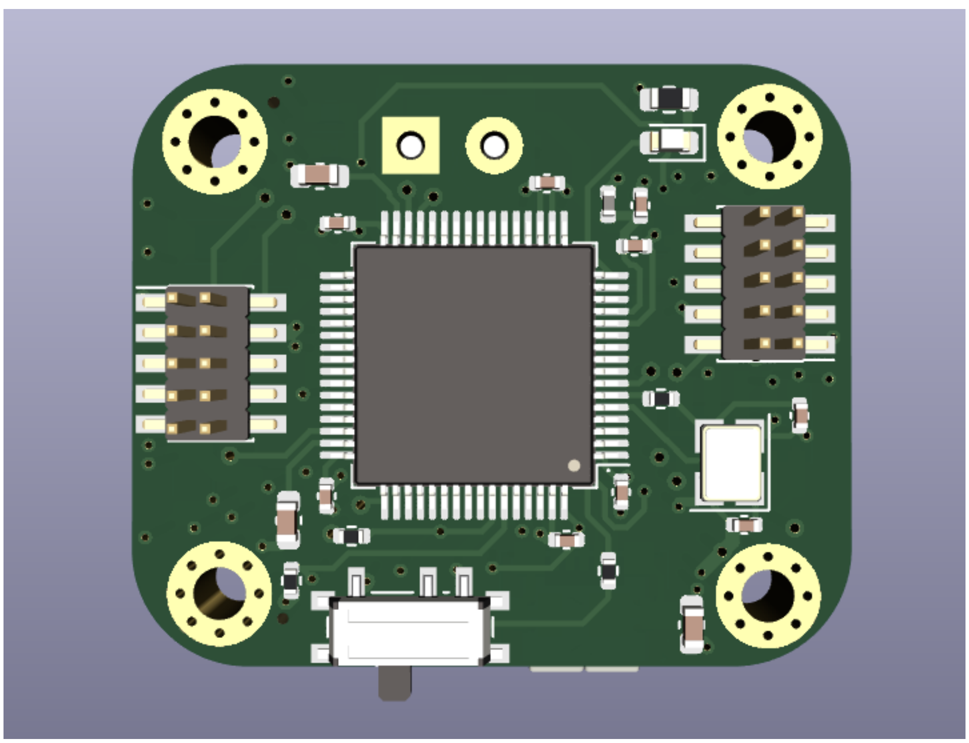

Figure 3

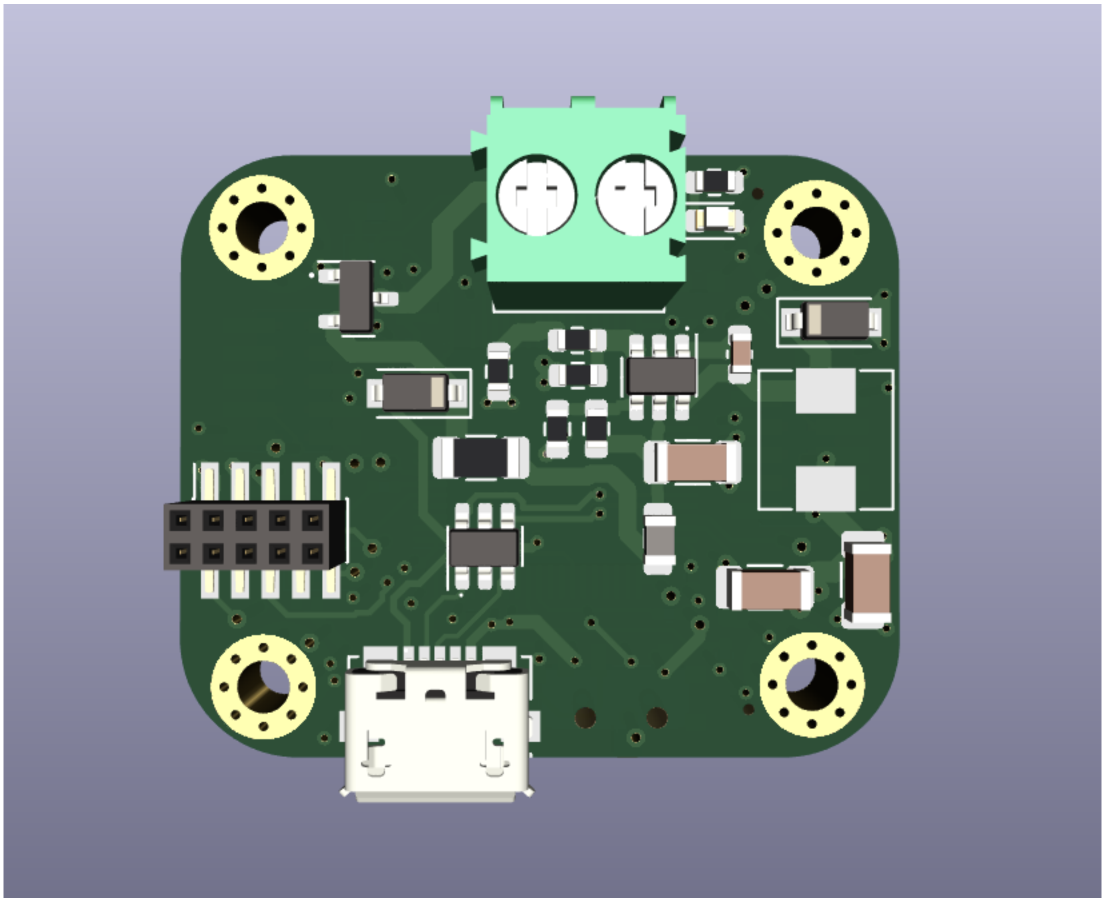

Figure 4