PCB Design of a Simple Embedded System

Each student will undertake this project individually.

Project Overview:

In this project, you will use KiCAD (<u>preferably version 6 release</u>) to design a PCB for an STM32F4 MCU, following the steps and techniques provided in the reference video. 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.

The objective is to replicate and understand the schematic and then move on to the PCB layout phase. This design task will familiarize you with the KiCAD environment, the process of translating schematics into board designs, and the nuances of PCB layout considerations, as discussed in the lecture.

I suggest downloading and installing KiCad-v6 released on Jan 27 2023 (1.1 GB), from the link below.

https://downloads.kicad.org/kicad/windows/explore/stable

Reference Video:

Watch and follow each step in this video https://www.youtube.com/watch?v=C7-8nUU6e3E to complete the schematic of the project. Also, watch the PCB design part of the video, however, <u>you must reduce the board size as explained in PCB section below in page 3. You must apply the principles taught in class for the PCB design of high-speed embedded systems.</u>

<u>KiCad STM32 + USB + Buck Converter PCB Design and JLCPCB Assembly (Update) - Phil's Lab #11</u>

Note: While the tutorial video is a comprehensive guide, always strive to understand the reasoning behind each step and think critically about how you can improve or adapt the design to different scenarios or constraints.

Footprints and Symbols in KiCad:

Note: Should you be unable to find a component within the KiCad library, the Snapeda (https://www.snapeda.com/) or Ultralibrarian (https://www.ultralibrarian.com/) websites offer schematic symbols, PCB footprints, and 3D models for various components.

Ultralibrarian and Snapeda are platforms that offer a comprehensive collection of electronic component CAD models for engineers and PCB designers. By using one of these platforms, designers can easily find and download models for a wide range of electronic components, such as integrated circuits, passive components, and connectors, which can then be directly integrated into your design project.

As an example, the video below shows how to import footprints and symbols in KiCad:
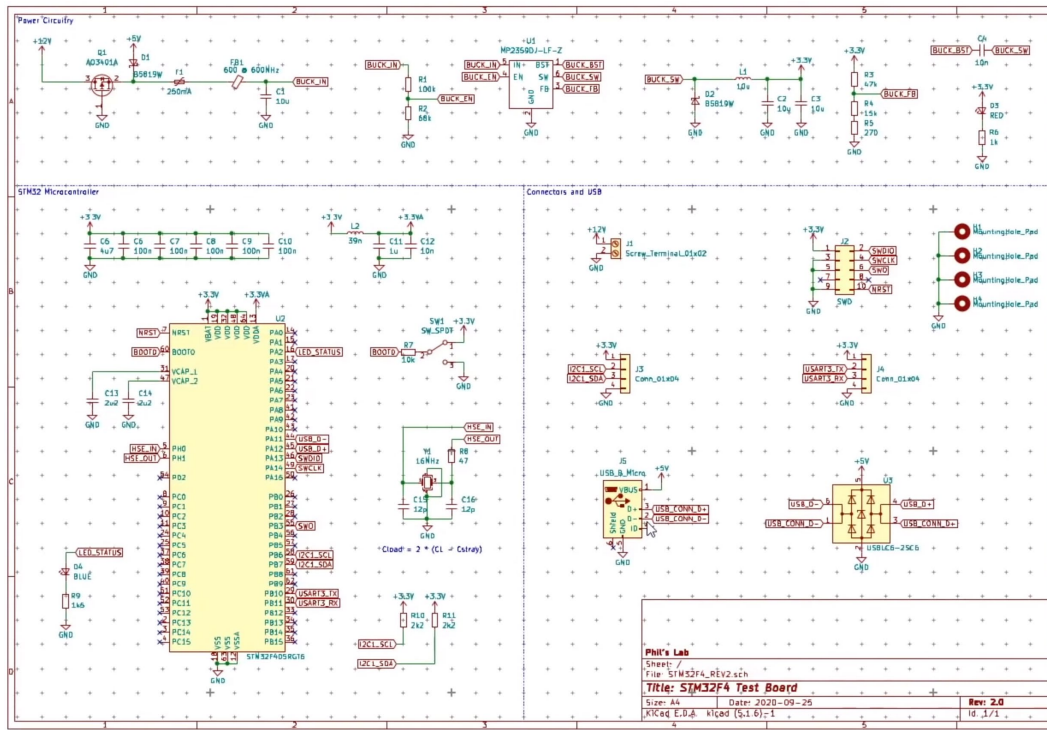
https://www.youtube.com/watch?v=W9cLnIjvybo&ab_channel=PlumPot

As an example, the video below shows how to design custom footprints and symbols in KiCad:

https://www.youtube.com/watch?v=pV-4ElYoXYU&ab_channel=Schematix

Schematic Explanation:

Based on the reference video, you will construct the design's schematic using KiCad, as shown below.

1. Power Circuitry:

- The power section consists of components like diodes, inductors, and capacitors to step down the input voltage to various voltage levels needed for the STM32F4 and other peripherals.

- The MP2359DJ-LF-Z is a buck converter, also known as a step-down converter. It is designed to take an input voltage and convert it to a lower output voltage while maintaining high efficiency. The buck converter topology is beneficial for reducing voltage from a higher source, such as a battery or power supply, to power low-voltage circuits or components. The MP2359DJ-LF-Z offers features like a built-in switching transistor, over-current protection, and under-voltage lockout, making it suitable for a variety of applications, including portable devices, distributed power systems, and other low-power applications where efficiency is crucial.

- Multiple filtering capacitors (`100nF` and others) are placed close to the power pins of the STM32F4 to filter out high-frequency noise.

2. STM32 Microcontroller:

- The main component in the design is the STM32F407ZGT6 microcontroller, which handles the main computation and control functions.

- It features various pins such as GPIOs (General Purpose Input Output), ADC (Analog to Digital Converter) pins, and communication interfaces like SPI, I2C, and UART.

- Decoupling capacitors are connected to their power pins to stabilize the voltage and filter high-frequency noise.

3. Connectors and USB:

- The design includes a USB Micro B connector, allowing the STM32F4 to communicate with a computer for programming or data transfer.

- Screw terminals and mounting holes are used for connecting external power and grounding, respectively.

- Several headers (like SWD for debugging and programming) are also present.

4. Oscillator Circuit:

- A 16MHz crystal oscillator is connected to the STM32F4, providing an external clock source for better accuracy and stability than the internal RC oscillator.

- Two capacitors (`22pF`) are connected to the oscillator to stabilize its operation.

5. Status LEDs:

- LEDs are connected to specific pins of the STM32F4 to show the operational status or for debugging purposes. They are driven through resistors to limit the current.

6. Communication Interfaces:

- The design also highlights the I2C communication interface with pull-up resistors (`4.7kΩ`) connected to the SDA and SCL lines.

PCB:

Based on the reference video, you are required to create the PCB design using KiCAD, as depicted below. Make sure to incorporate your unique logo in place of the "PHIL'S LAB" logo shown in the video.

In the instructional video, the PCB measures 50 mm by 55 mm with all components mounted on the upper side. For your design, the PCB dimensions should be reduced to 25 mm by 30 mm or less (preferably 25 mm by 25 mm), necessitating the strategic placement of components on both the top and bottom surfaces of the board. Additionally, consider proportionally reducing the size of the four mounting holes to match the scaled-down dimensions of your PCB. Apply your discretion to determine the appropriate dimensions for these mounting holes. In the video, the top layer is designated for routing traces, the second layer for power, the third for ground, and the fourth layer is utilized for a single signal trace. For your design, employ all four layers for a more comprehensive routing strategy that includes signals, ground, and power distribution.
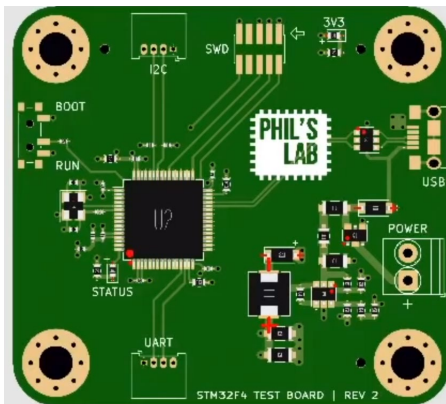
Objective:

Your main objective is to design the schematic and PCB using KiCad. You should:

1. Create a new project in KiCad and set up the schematic sheet.

2. Assign footprints to each component. Perform Electrical Rule Checks (ERC) to ensure no errors in the design.

3. Design the PCB layout while considering best practices like proper ground planes, minimizing trace lengths, especially for high-speed signals, and placing decoupling capacitors close to the IC power pins.

4. Run Design Rule Checks (DRC) to ensure no errors in the design.

5. A 3D view of the PCB that includes all the components.

6. Generate Gerber files, Bill of Materials (BOM), which are used for PCB manufacturing.

Refer to "Finalized_BOM" uploaded on Canvas as a sample to see how to build your BOM.

Evaluation Criteria:

1. Correctness of schematic and PCB layout as explained above, routing, your own logo, and 3D view of the PCB.

2. Adherence to best design practices and routing.

3. Absence of ERC and DRC errors.

4. Quality of silkscreen labels and design annotations.

5. Generated Gerber files and Bill of Materials (BOM) in a CSV format.

Submission:

1- You should present your work to your instructor, as explained in the evaluation criteria section, on <u>one of the following dates</u> during class hours:

   04/18/2024, 04/23/2024

   If you fail to meet the aforementioned presentation deadlines, your final opportunity will be on 04/25/2023 at 11:30 AM. Please note that this will result in a 40% reduction in your project grade.

2- Submit a one to two-page summary <u>technical report (follow instructions uploaded on Canvas)</u>. Please include details about the tasks you performed, the objectives of the project, the results you obtained, and any challenges you encountered. Include a screenshot of the PCB 3D view in your report. <u>Please ensure you submit your summary on Canvas in PDF format by</u> **5 PM, 04/25/2024**.

Good luck! Let's create an impressive PCB!