PCB Voltmeter Project

Report Summary

CPGR4233
Richard Homan
Reviewed by Juan Aguirre at the Writing Center
September 29, 2023

Introduction

The goal of the PCB Design project is to encourage the student understand the family of tools in KiCad. This is accomplished by having the student design a voltmeter, based on a ready made voltmeter design. To have the student focus only on design and process, a bill of materials is provided. Some of the most important items include a Xilinx XC9572XL series CPLD, an ECS 5032MV series oscillator, and two LP3869 family voltage regulators.

The PCB design project is designed to have students face a real world design problem, and solve it using KiCad as the design tool. In order to properly design the circuit, the student must educate themselves on the purpose, design, and layout of different components. This is done through the reading and understanding of datasheets, which contains information such as product specifications and recommended integrated designs.

Once the student understands how each component operates, the student creates the Schematic diagram. This is a theoretical circuit design which consists of simply connecting components together in their logical way. A notoriously time-consuming part of the project is the PCB layout design. The PCB layout uses the Schematic diagram as a reference for which real components are connected. However, the student must configure the layout such that no two traces (wires on a PCB) cross. Without the use of an auto-router, this often takes many trials to properly configure.

The most important aspect of any project is time management, and this project is no different. Reading datasheets, designing symbols and footprints, designing a neat Schematic, and carefully aligning traces and footprints in the PCB layout are time intensive. It is important that the student allocate his/her time wisely to the project. It is also important to start ahead in the expectance of problems and challenges. By the end of the project, students will be comfortable with the usage of KiCad.
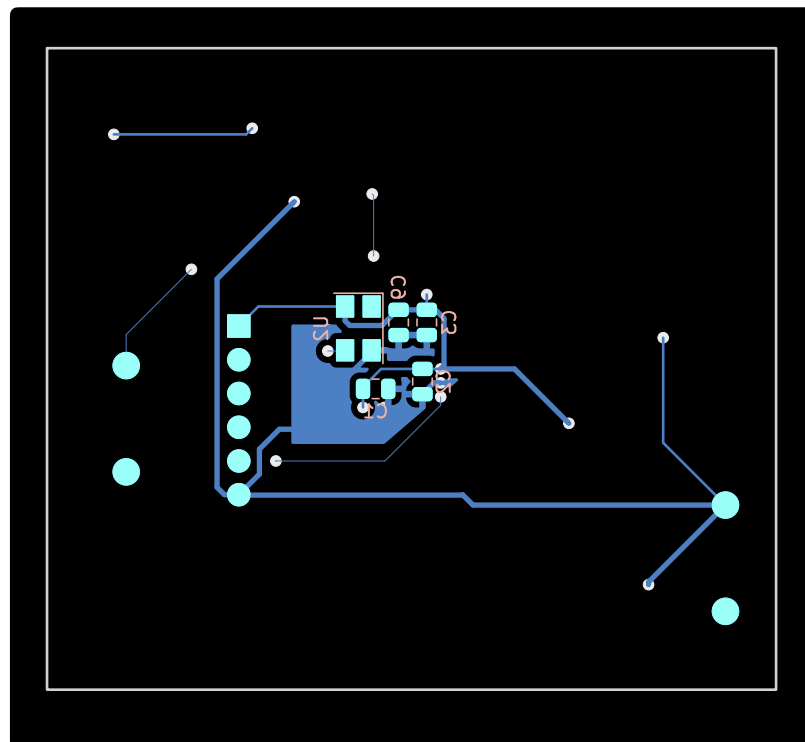
Design
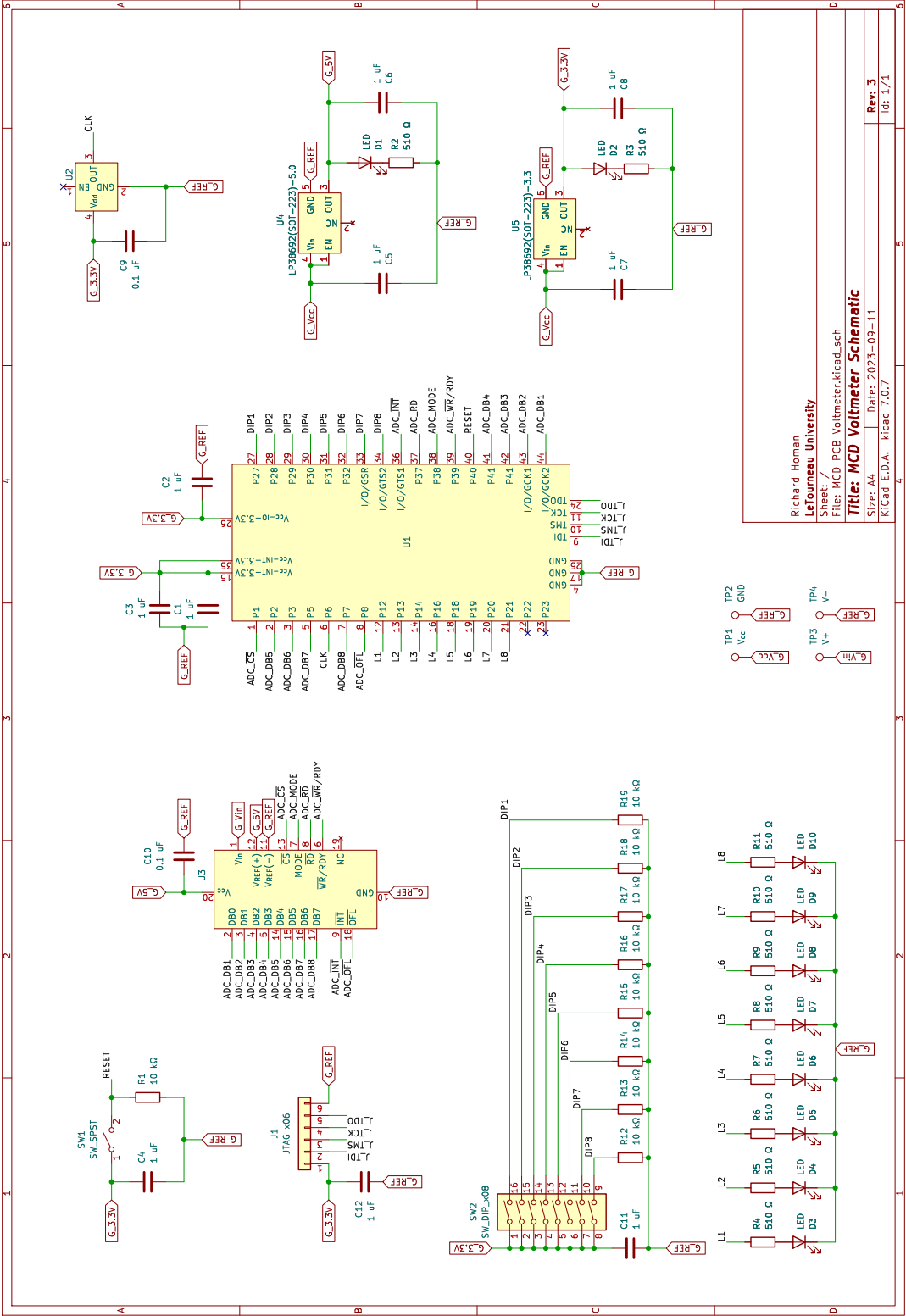
A. Schematic
B. Layers
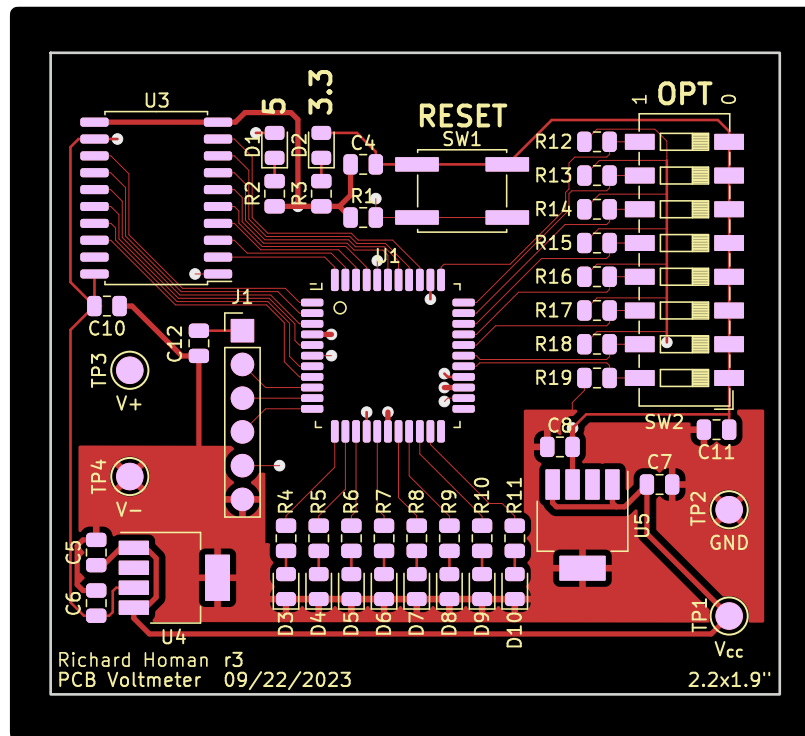
Top Layer

Bottom Layer

Personal Experience

While I am not new to KiCad, I would not consider myself an expert. In Fall of 2022, I experimented with KiCad on my own time and created a simple adjustable square wave generator. I learned a few critical components to designing in KiCad, which were how to design an intuitive Schematic and PCB layout. However, the PCB Voltmeter Design project is different than my personal project in that the PCB Design project is much more complex. Firstly, although we are given the Bill Of Materials, design was completely up to us. Many of the items in the BOM did not have built-in schematic symbols or PCB footprints, so each one either had to be downloaded from an online catalog or be manually created using KiCad's symbol and footprint editors. This alone took approximately 5 hours since I opted to create all of my own symbols and footprints. I decided to this because I did not trust symbols and footprints online, and I wanted my symbols and footprints to all share common design rules.

Symbol creation is relatively simple. This part of the project by far took the least amount of time as pins can arbitrarily be made and positioned on the KiCad symbol per the specification on the symbol's datasheet. On the other hand, footprint creation is more complex. Dimensions and positions are extremely important, and each pad must purposefully be placed and sized. This took the majority of my allotted symbol and footprint creation time.

Next, I worked on the Schematic digram, which is similar to symbol creation. This design is completely theoretical, so pins can be arbitrarily connected logically based on the purpose of the design. This stage, Schematic design and making the design readable, took less than 2 hours.

The difficult part comes in during PCB layout design. In this stage, going back and forward between Schematic and Layout is common when deciding which items go in which places. This is the greatest overall time and thought intensive part of design, as multiple considerations have to be made: trace width, footprint distances, planes, and trace overlap just to name a few. I spent approximately 8 hours, through trial and error, thinking through and placing footprints in various configurations around the PCB. Once I had found the best configuration (of my knowledge), I meticulously measured and placed footprints in certain positions based on a set of spacing and alignment rules I had decided upon. Then, I routed the traces, varying trace widths for different net purposes calculated with a trace width calculator online. Next, the ground planes were drawn, and the edges of the board trimmed.

Those inspecting the board may note that the design is on revision 3. Revisions 2 and 3 are quite similar, only differing in the position of a few LEDs and capacitors. Revision 1 was drastically different than the other two. The reason for this is because I had made my CPLD symbol based on the PC44 configuration rather than the VQ44 configuration. Upon discovery of this by the student assistant, Joao, I had to change my symbol. Because footprints rely on symbols, I also had to redesign my entire PCB layout. This error was totally based on my own failing to double check specifications given on the assignment details as well as on the CPLD spec sheet. The total time spent was approximately 4 hours.