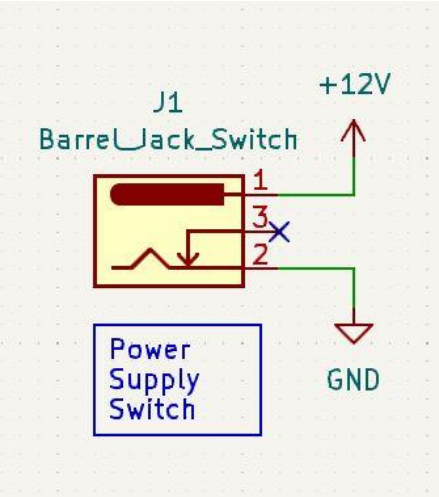
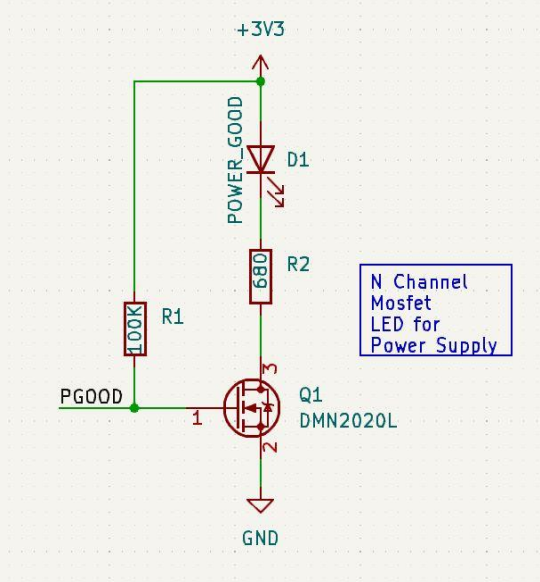
Schematic Design

Kicad 7.0 Schematic editor is used For designing the schematic of the calculator. The design mainly have five categories.

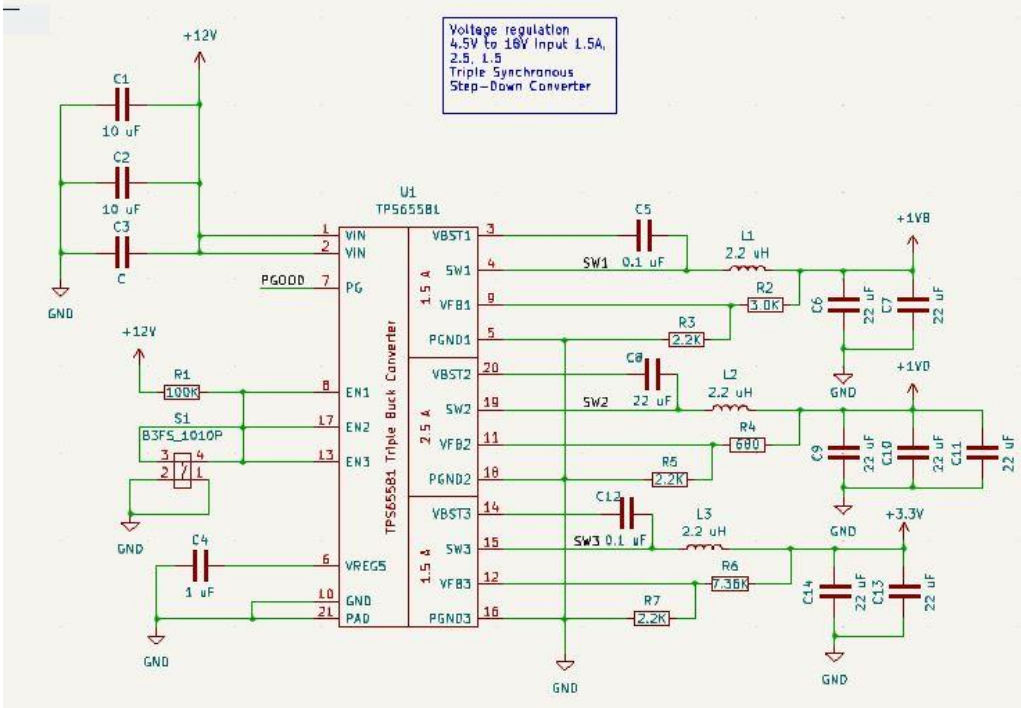
1. Power Schematic:
    - Voltage regulator which is able to give a voltage input from 4.5V to 18V and the current input varies from 1.5A,2.5A. It is a triple synchronous and a step-down Converter.



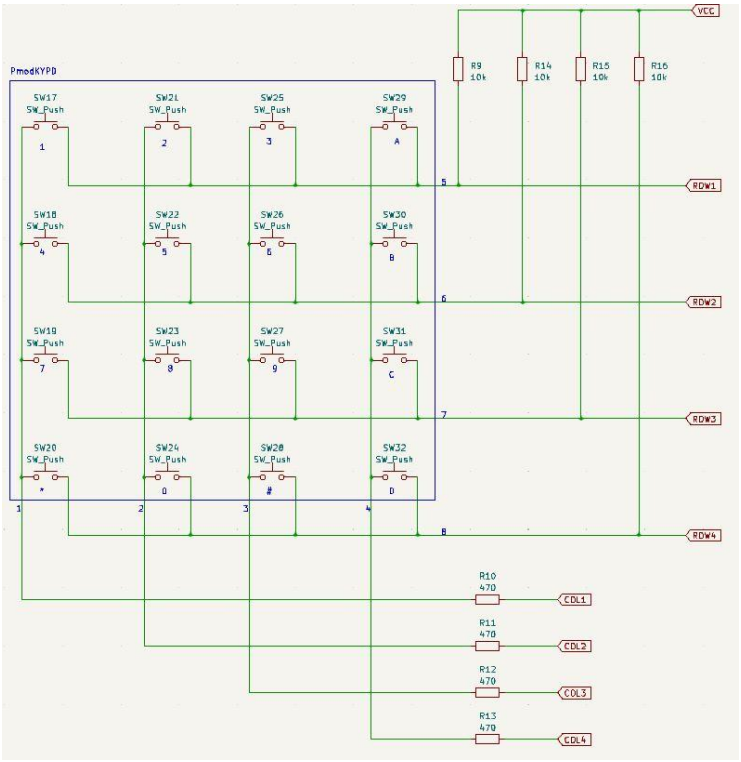
- Power Supply Switch: J1 Barrel_Jack_Switch is used.



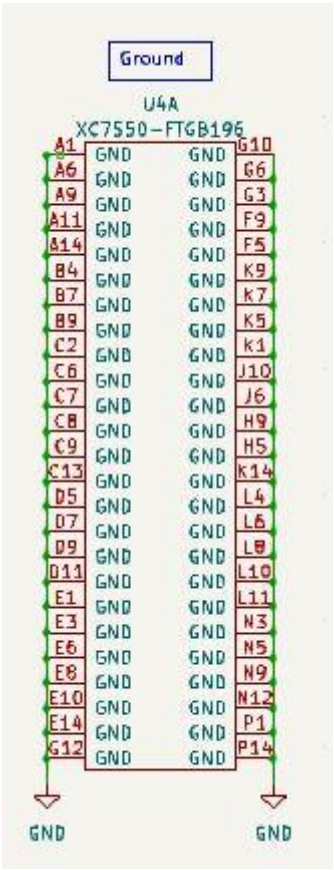
- LED: N channel MOSFET LED for Power supply is used it is included with Q1 DMN2020L N channel MOSFET, an LED and 100k and 680 ohm resistors have been used.



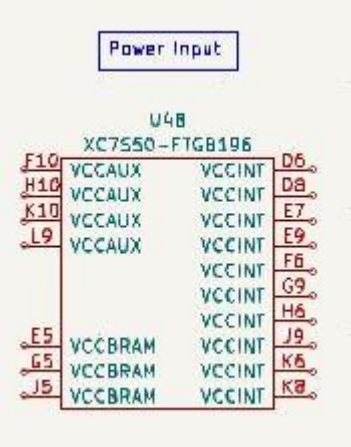
2. Keypad Schematic: This is an external input device have been used in the project. It is included with 4 columns and 4 rows together with 16 push buttons. It is included with the global labels such as ROW 1 to 4 and Col 1 to 4 in order to connect the schematic with the other hierarchical schematics.



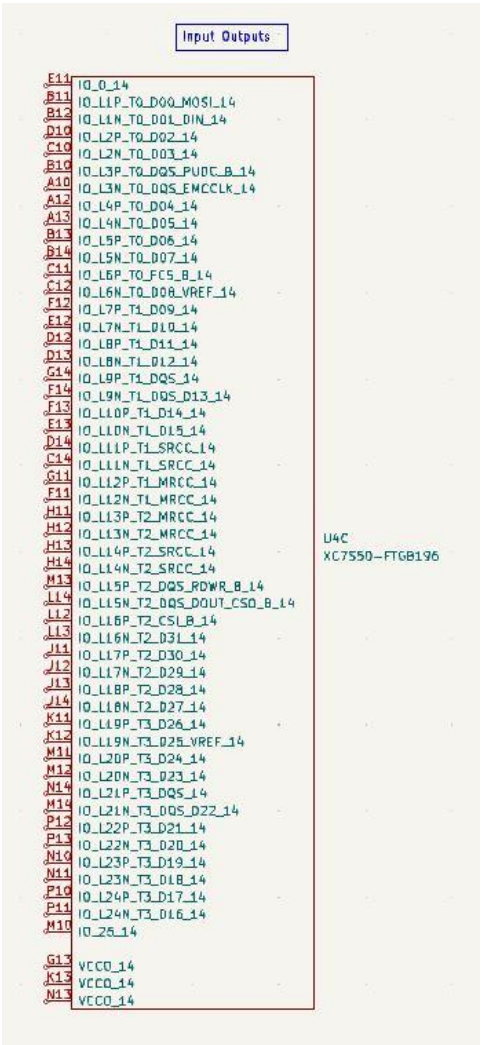
3. FPGA Schematic:
    - Ground: The schematic to represent Grounded pins



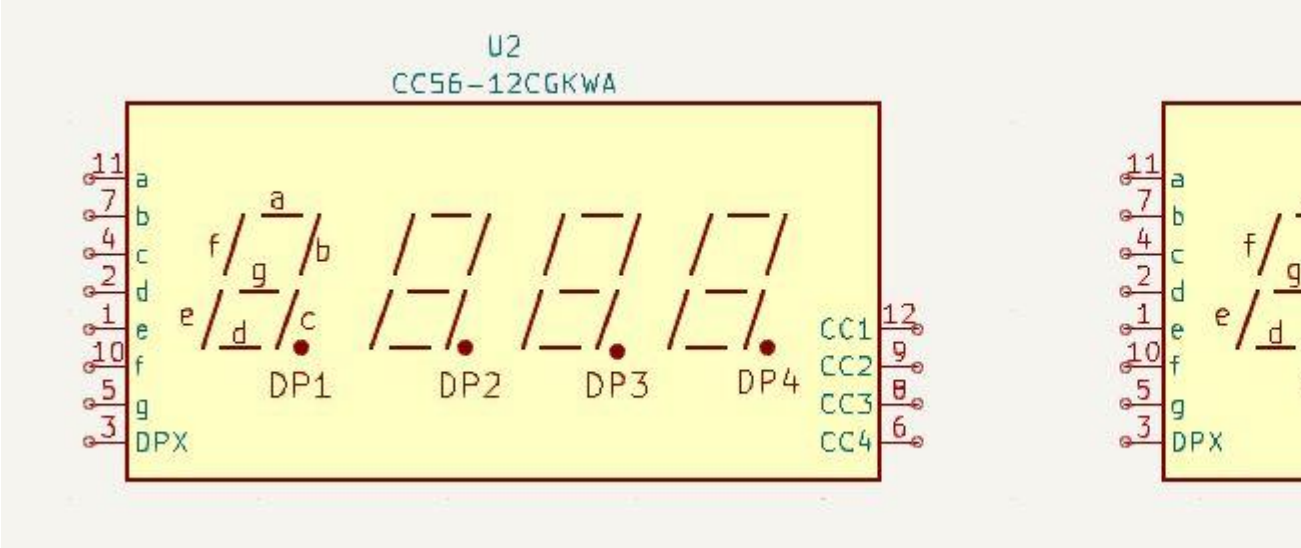
- Power Input: The schematic to represent Power inputs



- Input Outputs: The schematic to represent Input and Outputs

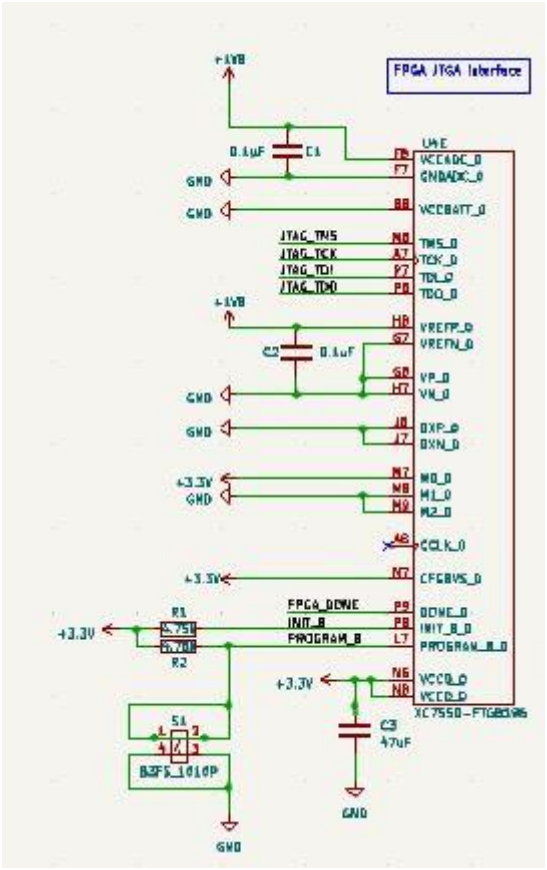


4. Seven Segment display Schematic: Two seven segments displays(U2 CC 56-12CGKWA) are used.

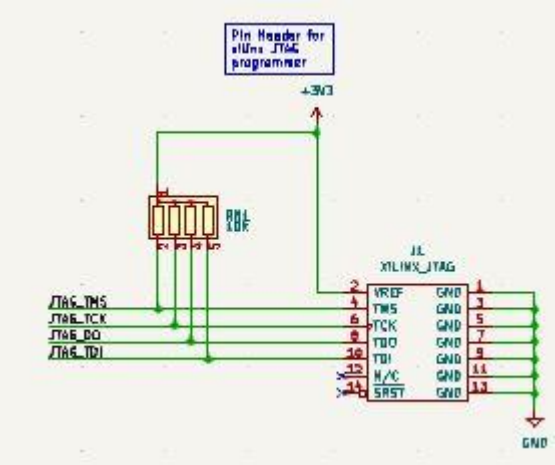


5. FPGA JTAG Interface Schematic:

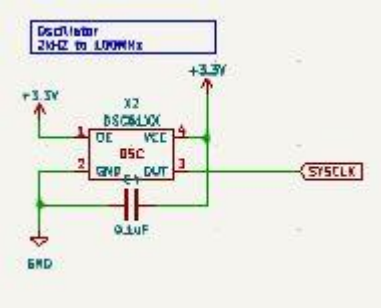
- FPGA JTAG Interface



- Pin Header for Xilinx JTAG programmer



- Clock: Oscillator 2KHz to 100MHz



- Tactile Switch: Used for the reset switch