coursera

REPORT ON BUILDING A 4-LAYER MIXED SIGNAL DESIGN
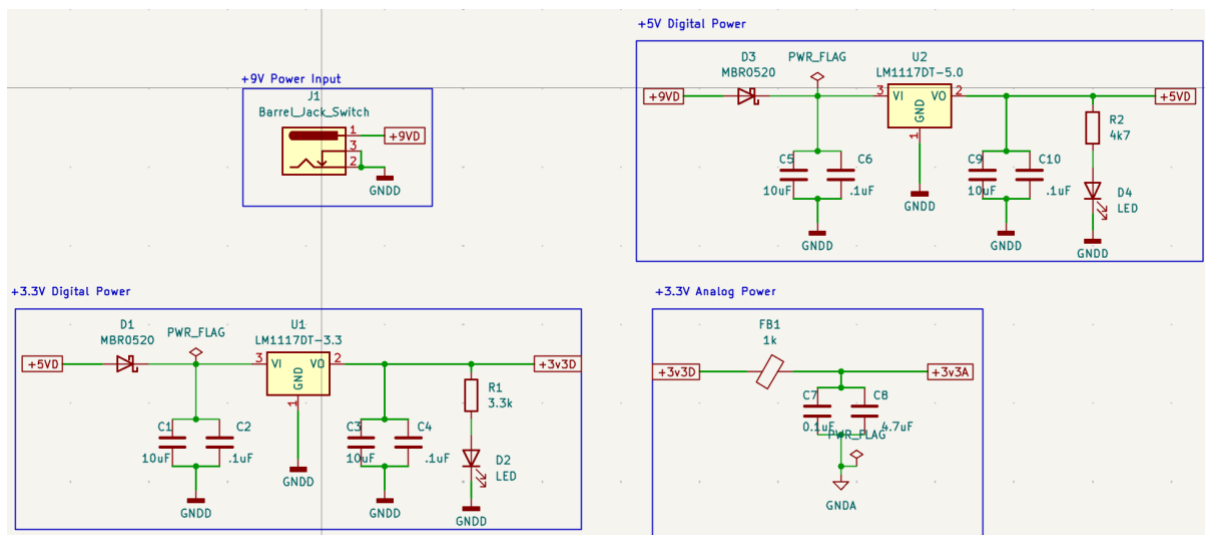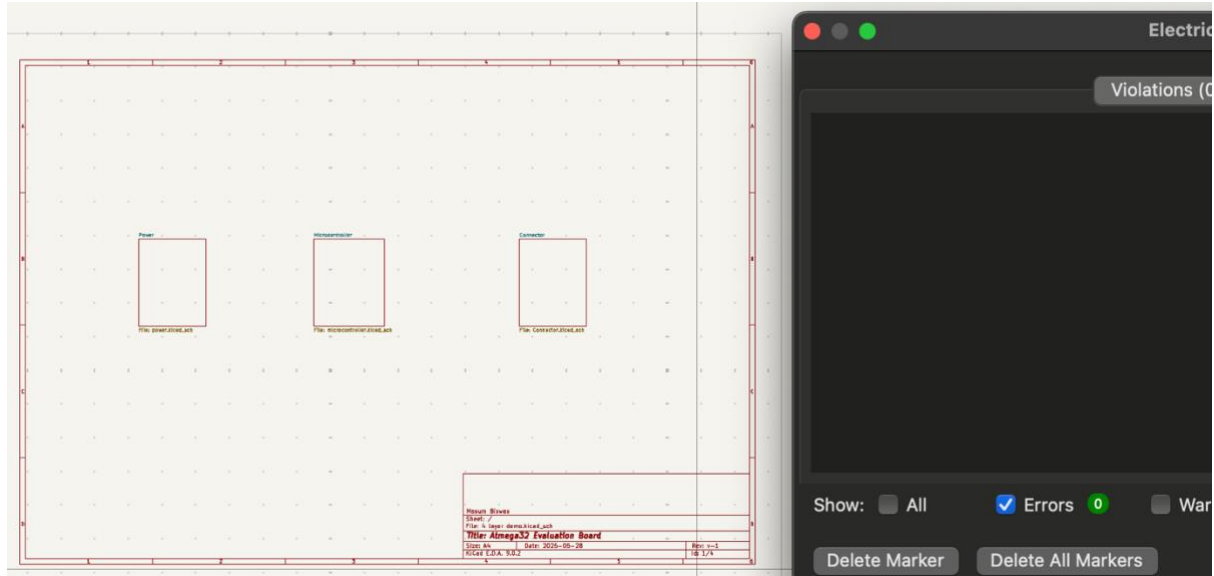
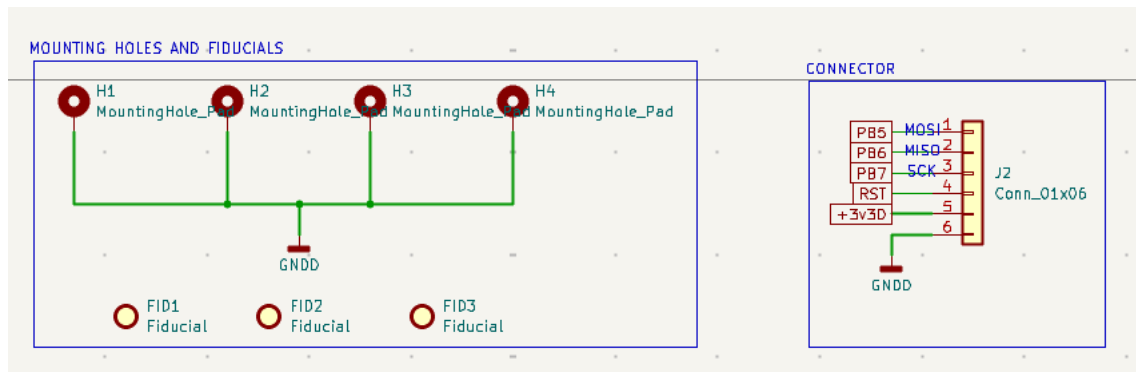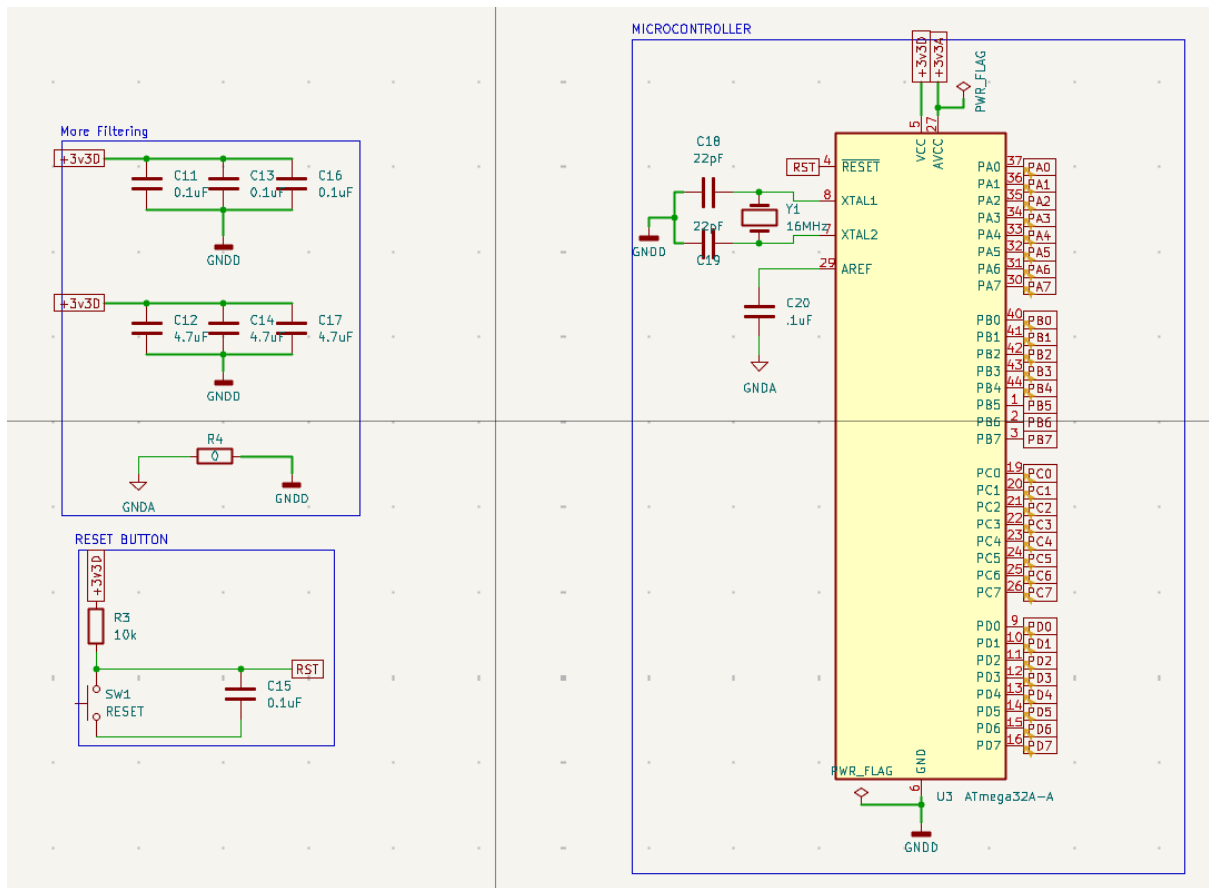## BY

MD MASUM BISWAS

INTRODUCITON

The assigned task was to design an astable multivibrator using KiCAD using SMD components and panelize the PCB, and finally upload it for the peer review. So, the task was done using the latest KiCAD version 9. And all the necessary files have been attached.

Methodology and Result

At first, the schematic was drawn using default grid settings according to the supplied circuit diagram. After that ERC was checked with zero errors.

More Filtering

+3v3D

C11 0.1uF   C13 0.1uF   C16 0.1uF

GNDD

+3v3D

C12 4.7uF   C14 4.7uF   C17 4.7uF

GNDD

R4 0

GNDA   GNDD

RESET BUTTON

+3v3D

R3 10k

SW1 RESET   C15 0.1uF   RST

MICROCONTROLLER

+3v3D +3v3A

PWR_FLAG

VCC AVCC

5 27

C18 22pF

RST RESET 4 RESET

PA0 37 PA0
PA1 36 PA1
PA2 35 PA2
PA3 34 PA3
PA4 33 PA4
PA5 32 PA5
PA6 31 PA6
PA7 30 PA7

8 XTAL1
Y1 16MHz
7 XTAL2

22pF
C19

GNDA

29 AREF

C20 .1uF

PB0 40 PB0
PB1 41 PB1
PB2 42 PB2
PB3 43 PB3
PB4 44 PB4
PB5 1 PB5
PB6 2 PB6
PB7 3 PB7

GNDA

PC0 19 PC0
PC1 20 PC1
PC2 21 PC2
PC3 22 PC3
PC4 23 PC4
PC5 24 PC5
PC6 25 PC6
PC7 26 PC7

PD0 9 PD0
PD1 10 PD1
PD2 11 PD2
PD3 12 PD3
PD4 13 PD4
PD5 14 PD5
PD6 15 PD6
PD7 16 PD7

PWR_FLAG GND

6

U3 ATmega32A-A

GNDD

MOUNTING HOLES AND FIDUCIALS

H1   H2   H3   H4
MountingHole_Pad MountingHole_Pad MountingHole_Pad MountingHole_Pad

GNDD

FID1   FID2   FID3
Fiducial   Fiducial   Fiducial

CONNECTOR

PB5 MOSI 1
PB6 MISO 2
PB7 SCK 3
RST 4
+3v3D 5
6

J2 Conn_01x06

GNDD

After completing ERC, the footprints were assigned to the schematic using the default KiCAD footprint library.
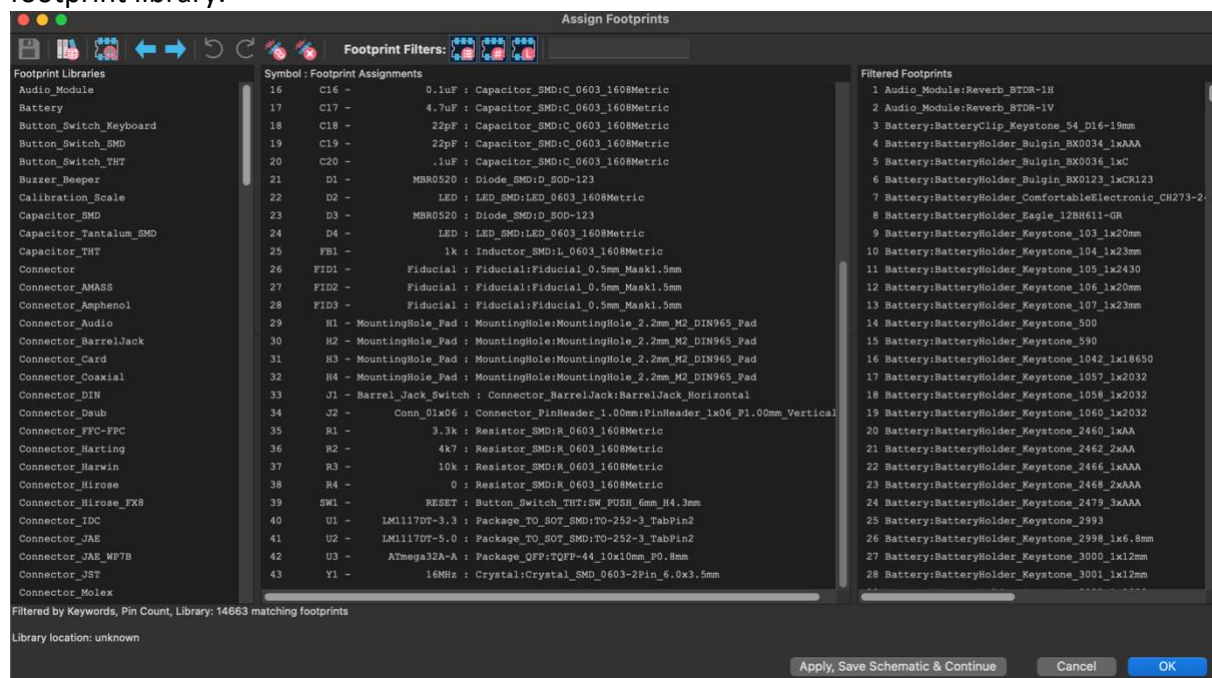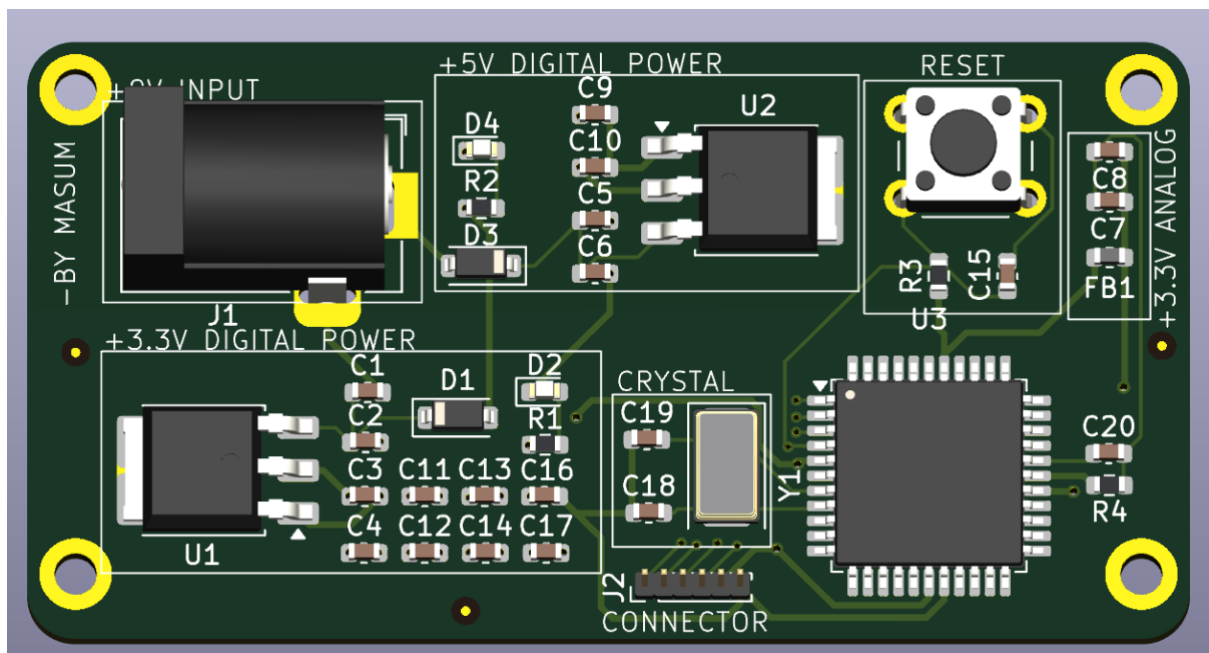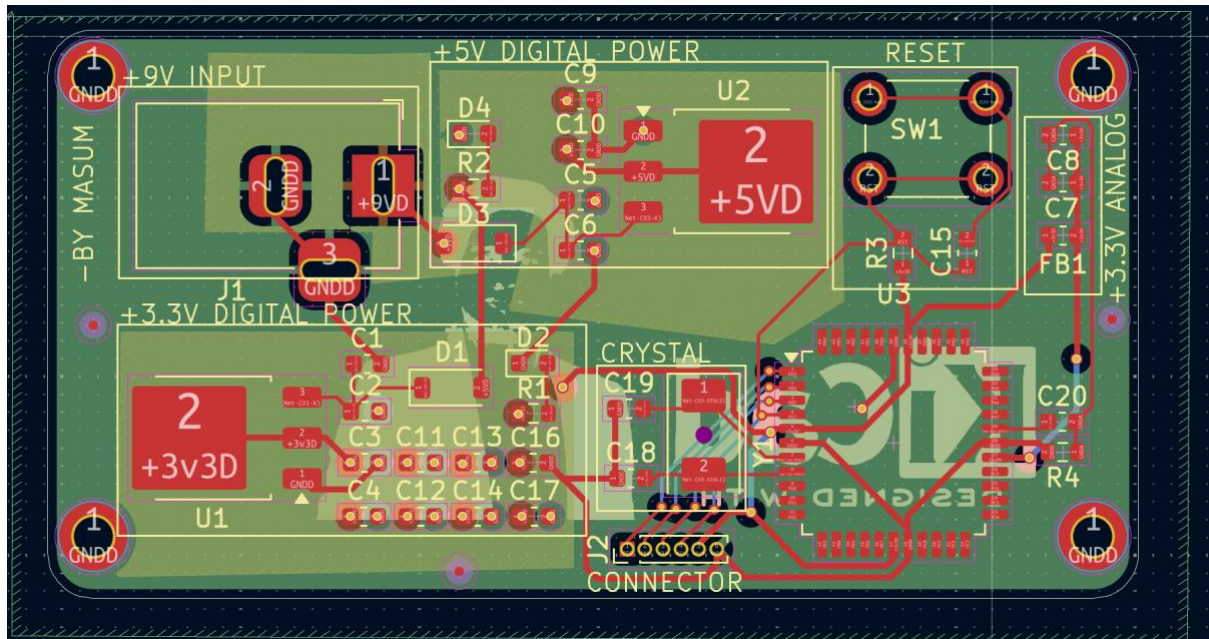

All the chosen components were SMD type, except for the connector pin, and 9V Barrel Connector. After assigning the footprint, it was time for the PCB layout design. Ever since that board setup was set to default values, mostly. A double-layer board was selected.
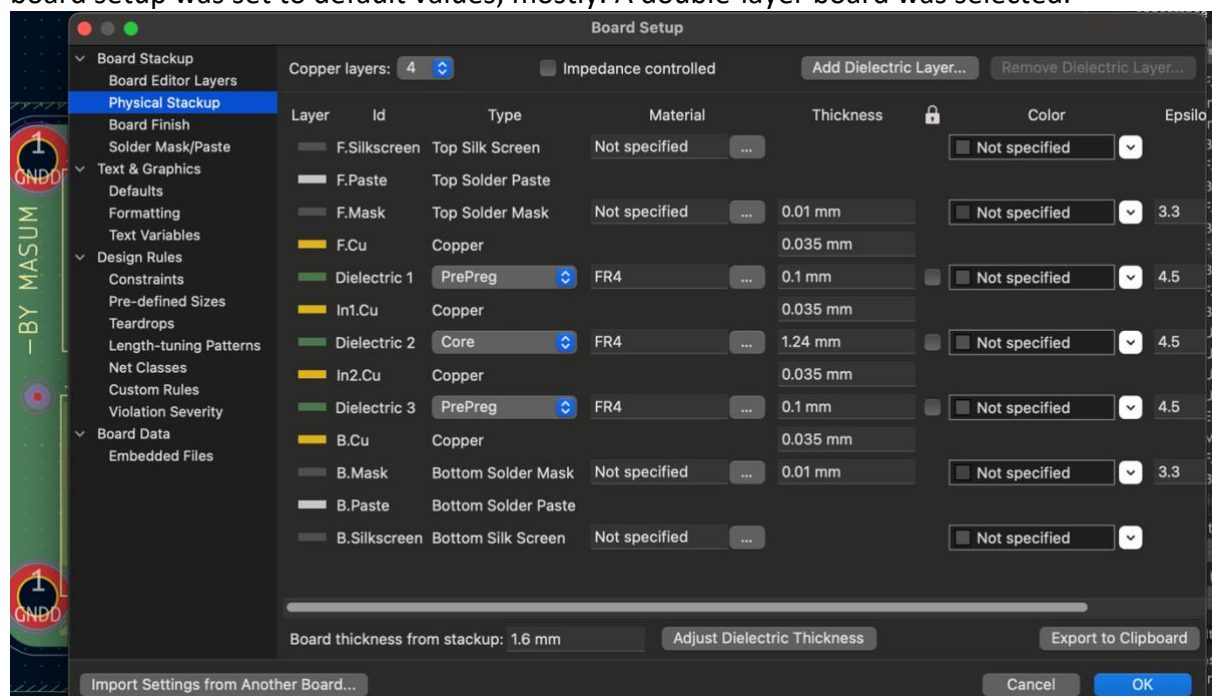
Components were updated to the PCB from the schematic page. Then all of them were placed accordingly.

Following that, the tracing was completed. I used a lot of vias for the ease of tracing the copper. And for heat distribution, Copper fill was used with the GND net at the 2nd layer.

Back side of the PCB