*MPEC REPORT 7*

KiCad – Schematic Initialization, Electrical Rules Check, Assigning Footprints and Breadboard Mounting

WEEKLY PROGRESS:

**KiCad:**

The procedure is as follows:

Schematic initialization – in which you make the circuit on the software.

Electrical rules check – in which you check if your circuit has any problem in it.

Assign footprints to the components – in which we assign footprints according to the dimensions of our PCB.

Generate Netlist – To generate netlist, which helps us to connect the components via ratsnest in the .kicad\_pcb file.

Pcbnew – which shows us the software version of our actual PCB layout.

Generate gerber file – which allows us to create an image of all the connections we made in the .kicad\_pcb file.

In this week, we completed: schematic initialization, electrical rules check, and assigning footprints to the components.

We had also included the library for the IC LM3524.

Problems faced during electrical rules check:

1. We noticed that some of our ground points weren’t connected to the circuit, they were mistakenly placed to the side of the end terminals of components. So we, corrected them accordingly.

Problems faced during assigning footprints:

1. We didn’t know the dimensions of the components at the first instance, so we got the measured values and assigned the footprints to the respective components.

**Breadboard Mounting:**

While half of the group was working on KiCad, the other half was working on the Breadboard mounting.

We had mounted the first half of the circuit i.e. the left side of the IC.

Problems faced:

We were short of some components, like: 0.04 Ω resistor, 0.022 µF capacitor and a potentiometer.