*MPEC REPORT 8*

**KiCad – Generating netlist, connecting components in Pcbnew, generating gerber file and breadboard mounting and checking output**

WEEKLY PROGRESS:

**KiCad:**

In this week, we completed: generating netlist, connecting the components, and generating the gerber file.

Problems faced during connecting the components in Pcbnew:

1. We had to maximise the number of connections at the bottom of the PCB, so we had to edit the connections in such a way that we connect minimum number of components via physical wires.
2. We had to avoid cross connections which were happening while connecting the components.
3. We also had to make sure that the whole board got grounded when we filled the zone with GND.

**Breadboard Mounting:**

In this week, we connected the remaining half of the breadboard.

Problems faced:

1. After connecting the whole circuit on the breadboard, while checking for the output, we did not get the desired output of 13V. Instead, we got an output of +0.2V.

The final result:



