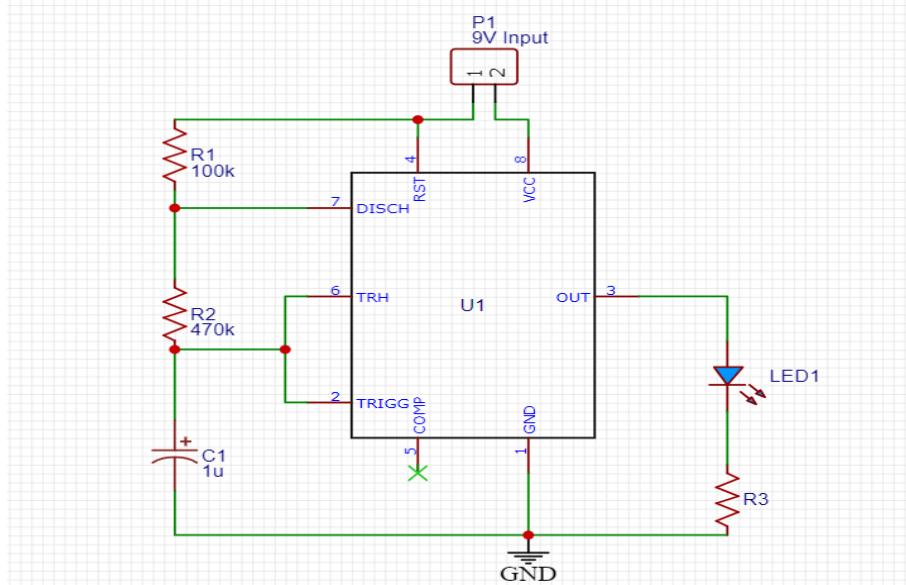


PCB Design, Development and Soldering

Name: -Devansh Modi ID: -202304009

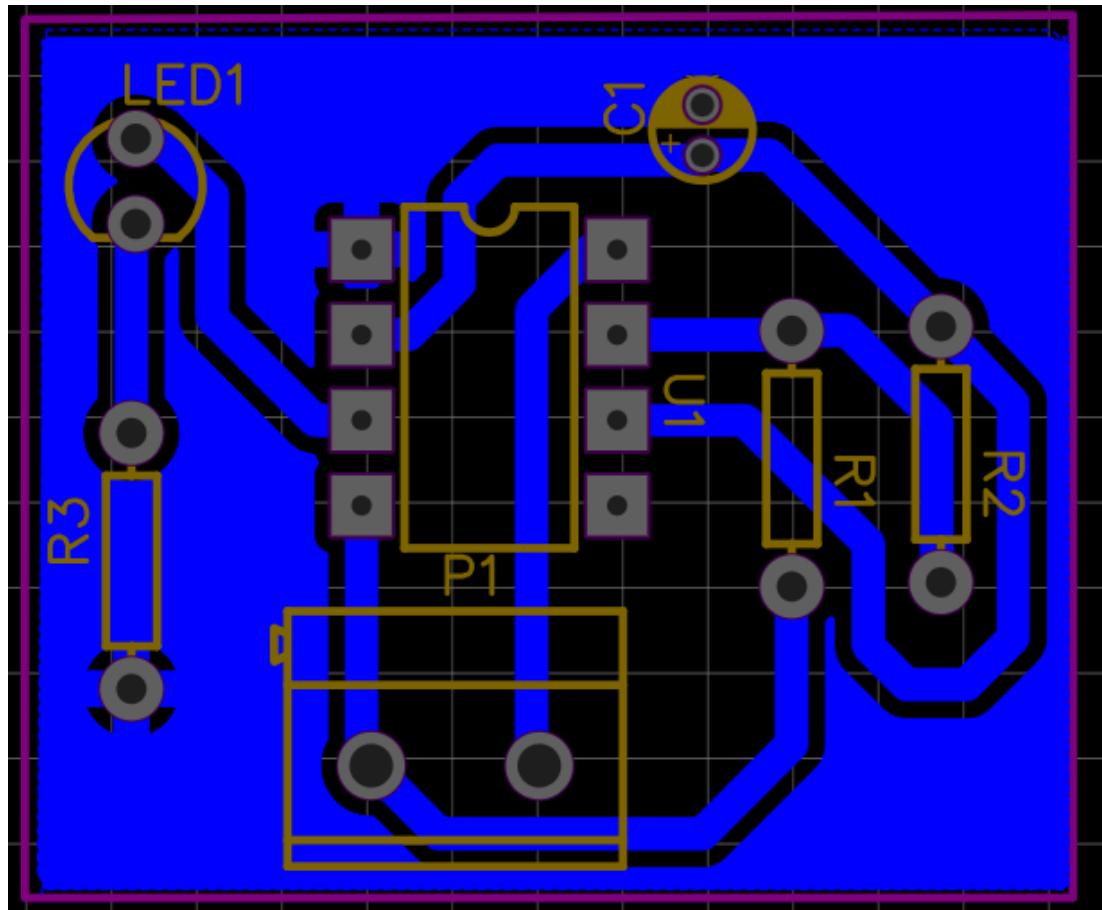
- ❖ **Aim:** - To design, develop, and test a printed circuit board (PCB)
- ❖ **Materials Required:**
 - Copper clad PCB board
 - PCB transfer paper
 - Laser printer
 - Etching solution (Ferric chloride)
 - Soldering iron, solder, flux
 - Electronic components (Resistors, Capacitor, 555 ic, LED, connecting wires, etc.)
 - PCB drill
 - Safety gloves
 - Cleaning materials (Cotton swabs)

- ❖ **Procedure:**
 1. **Creating a schematic diagram:**



(Fig 1. Schematic of the circuit, ensuring all components and connections are correctly represented)

2. Converting the schematic to PCB Layout



(Fig. 2 PCB layout of the schematic)

- Track width is 1 mm and clearance in 0.5mm
- Wider tracks help in preventing overheating of tracks and have lower resistance.
- If the clearance is kept small there is higher risk of short circuits and interference between the two tracks.
- After checking for any errors and performing design rule check, we exported the Gerber files and other necessary documentation from the PCB design software.
- Then we printed our design on the PCB transfer paper (a thin piece of paper coated with wax and pigment) using Toner (A mixture of carbon powder, iron oxide, fumed silica and other chemicals).
- In our first version we made a mistake by printing the design in mirror image.

3. PCB Fabrication



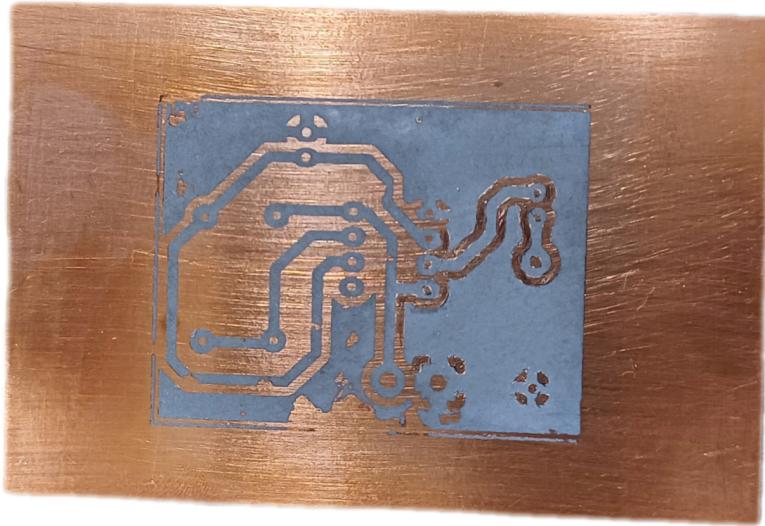
(Fig 3. Copper-clad PCB board)

- Removing the top protective layer on the copper-clad PCB board using iron wool.



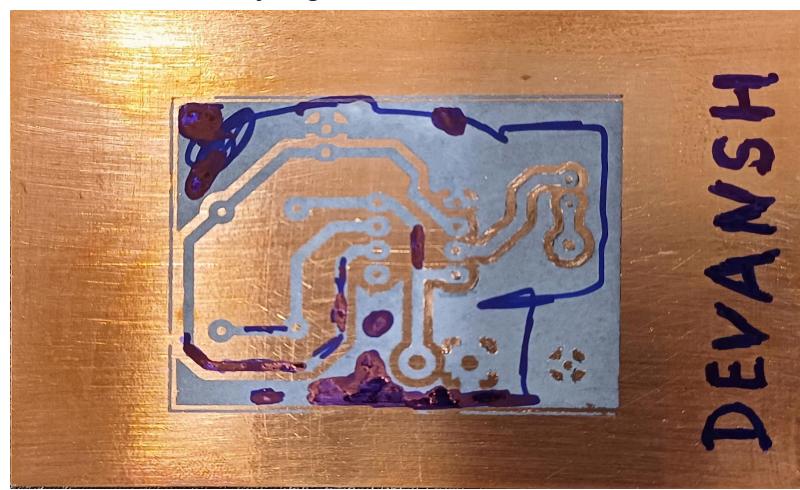
(Fig 4. PCB After removing the top protective layer and cleaning the copper surface)

- Transferring the toner (from PCB transfer paper) to the copper surface using an iron for 10 to 15 min).



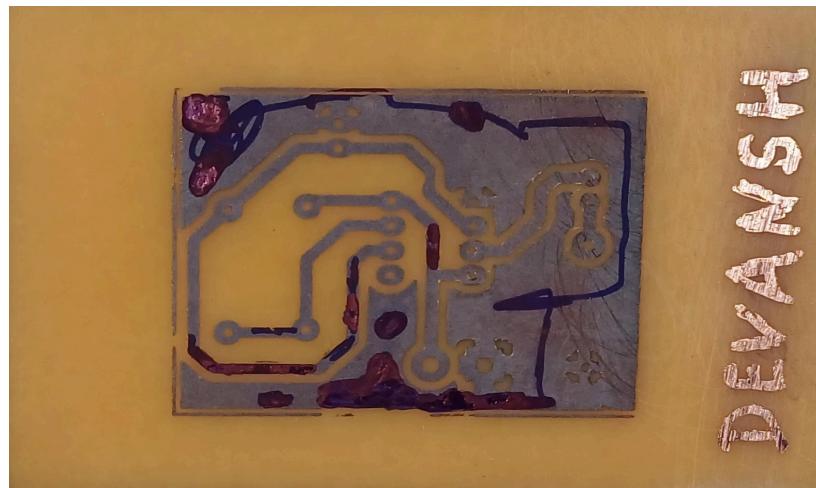
(Fig 5. Toner transferred to copper surface using an iron)

- The black toner on the PCB transfer paper has turned grayish because the paper sticks to the toner when heat is applied for a longer time. This color change won't affect the etching process later.
- Toner is generally made of carbon powder, iron oxide, polypropylene, fumed silica.
- In some areas shown in Fig 5., the toner is not transferred due to uneven pressure, leaving gaps in the design on the copper surface. To solve this, we can fill in the gaps using a marker, as it contains the necessary ingredients found in the toner.



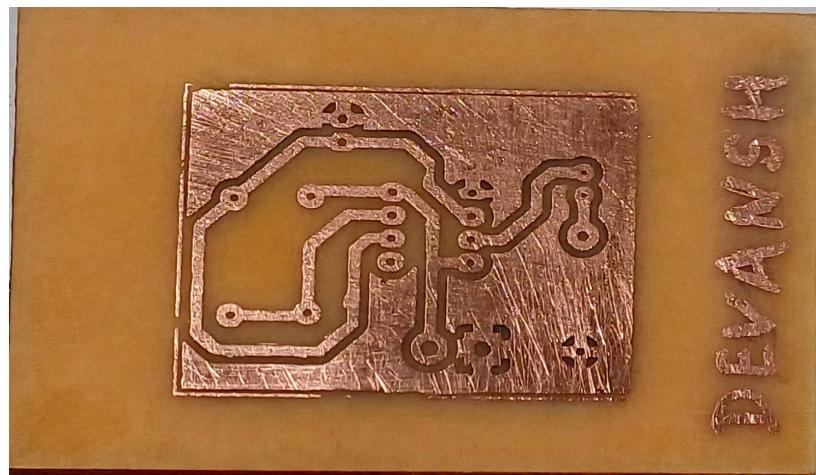
(Fig 6. Gaps filled up using a marker)

- We etched the PCB to remove unwanted copper. First, we prepared the etching solution using Ferric Chloride as the etchant. Then, we soaked the PCB for approximately 10 minutes while constantly stirring to expedite the reaction.



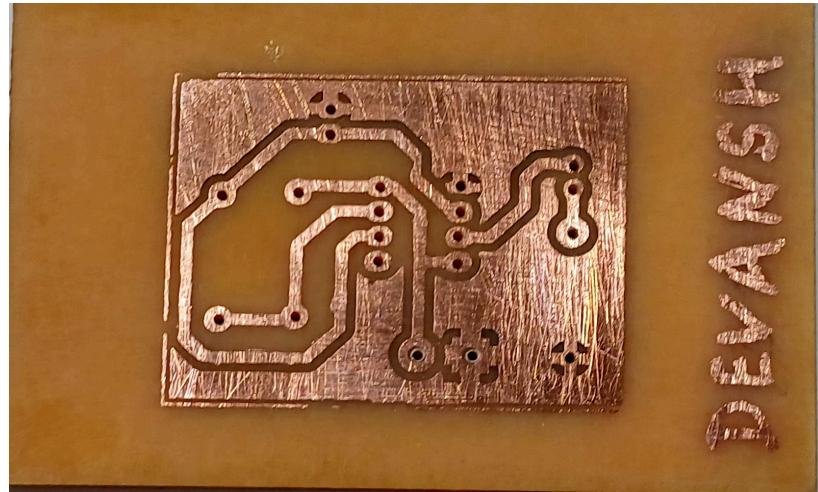
(Fig 7. PCB board after etching)

- Then we cleaned the remaining toner using sandpaper.



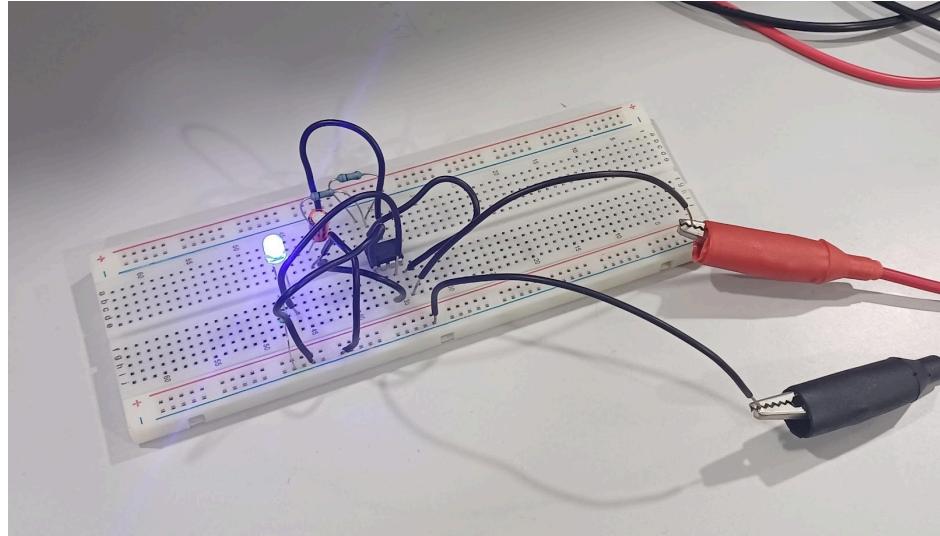
(Fig 8. PCB board after removing the toner)

- Then we drilled holes for component leads using a drill.



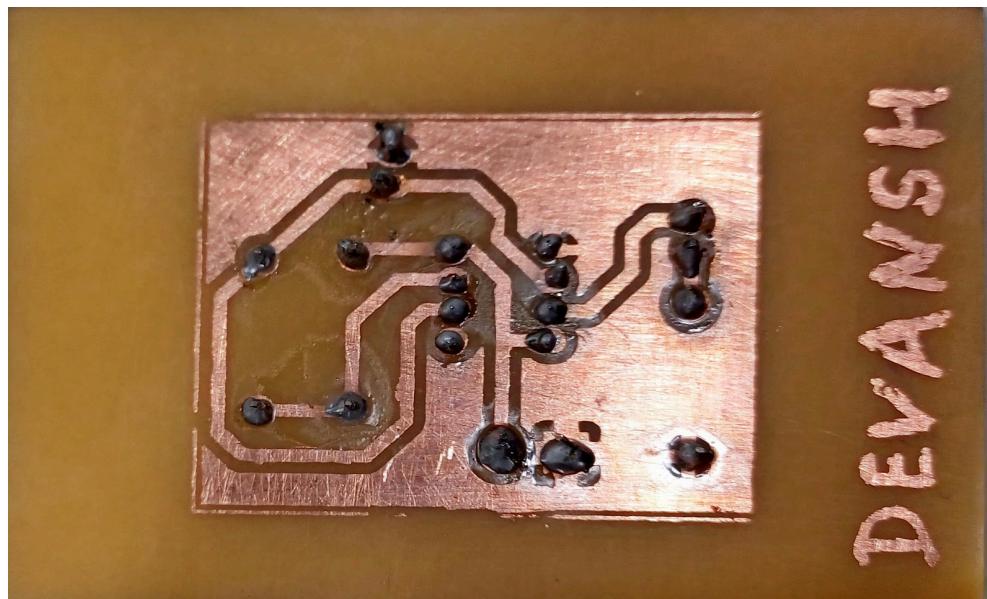
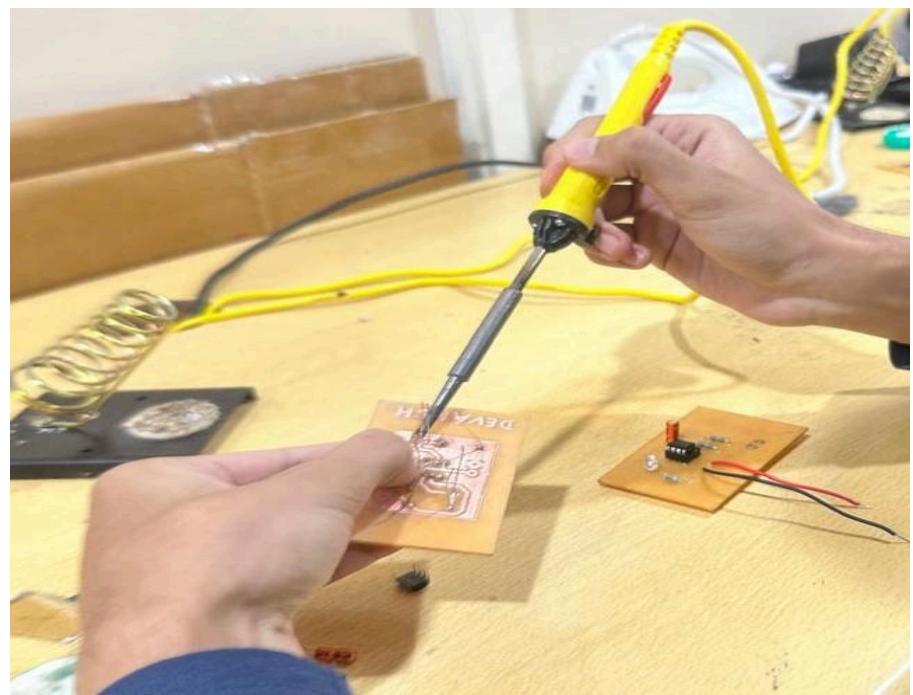
(Fig 9. Drill holes for component leads)

- Then we tested that all the components that are going to be soldered are functional and working by making the circuit on the breadboard.

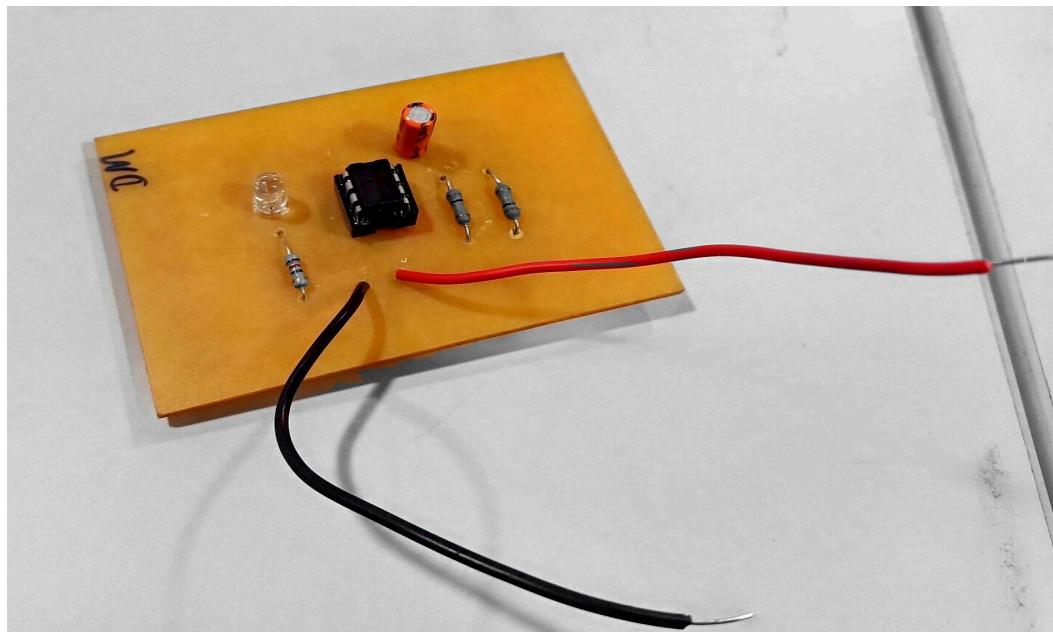


(Fig 10. Checking all the components are functional on the breadboard)

- Soldering the components to the PCB using Solder Iron, Solder wire and flux.

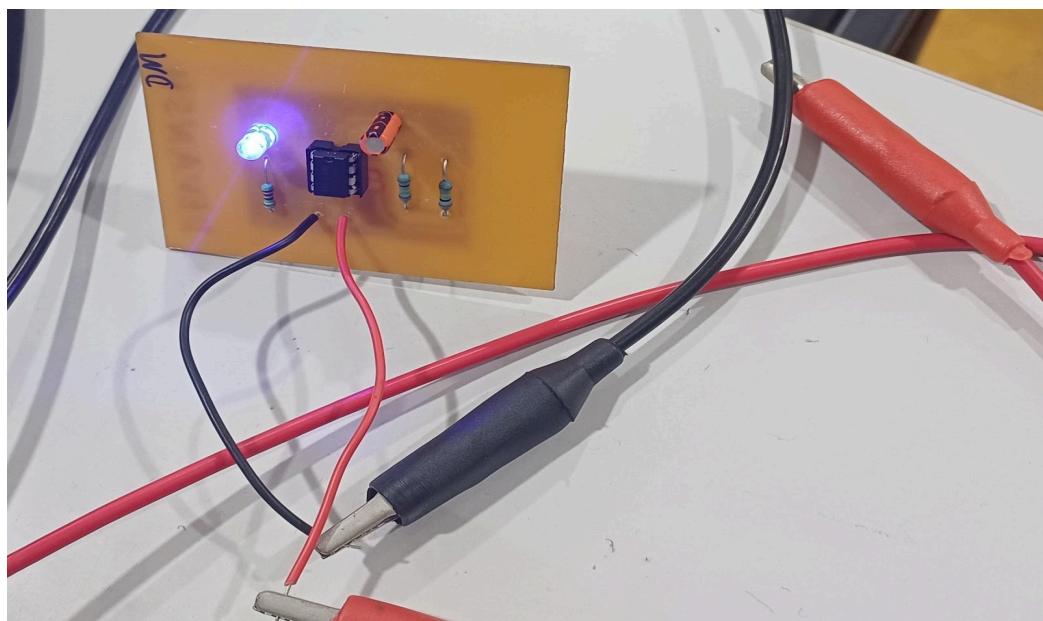


(Fig 11. Components soldered to the PCB)



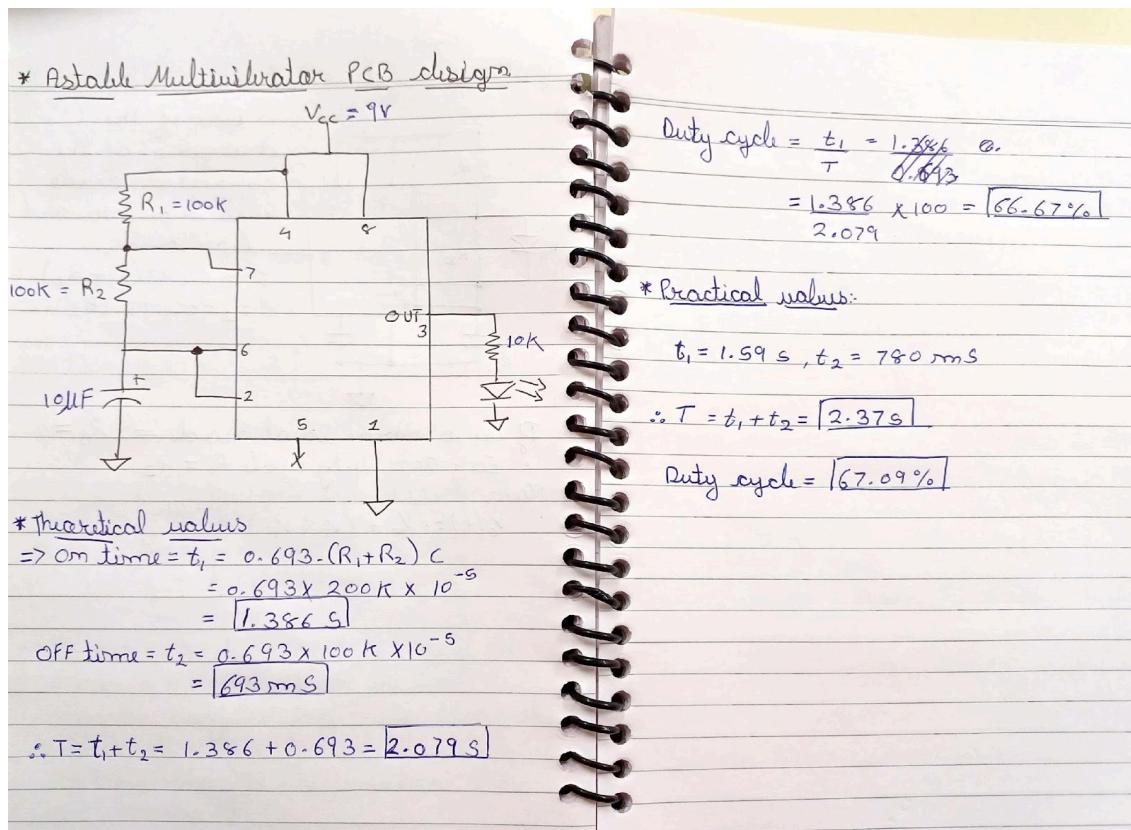
(Fig 12. Front view of the PCB after soldering)

- Inspecting the PCB visually for any soldering issues. Using the multimeter for checking any short circuit.



(Fig 13. The circuit is functional and operates as intended)

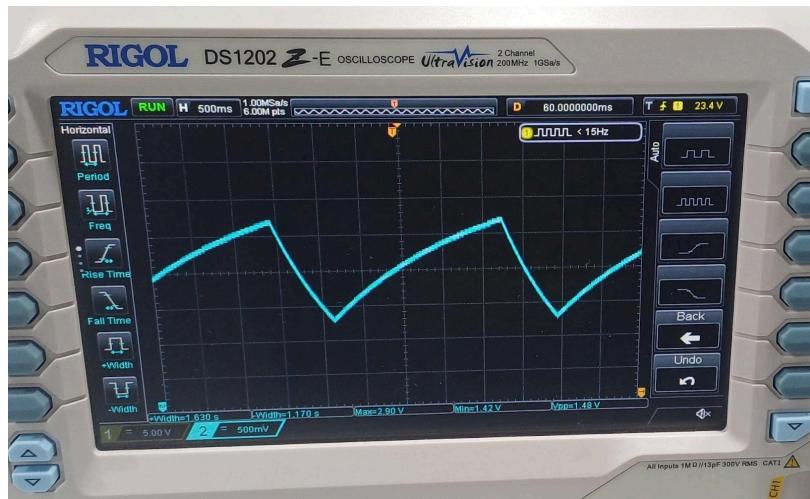
Circuit Analysis:



- We can get the desired value of on and off time by adjusting the values of R₁, R₂ & Capacitor.
- Measured the output on the Oscilloscope

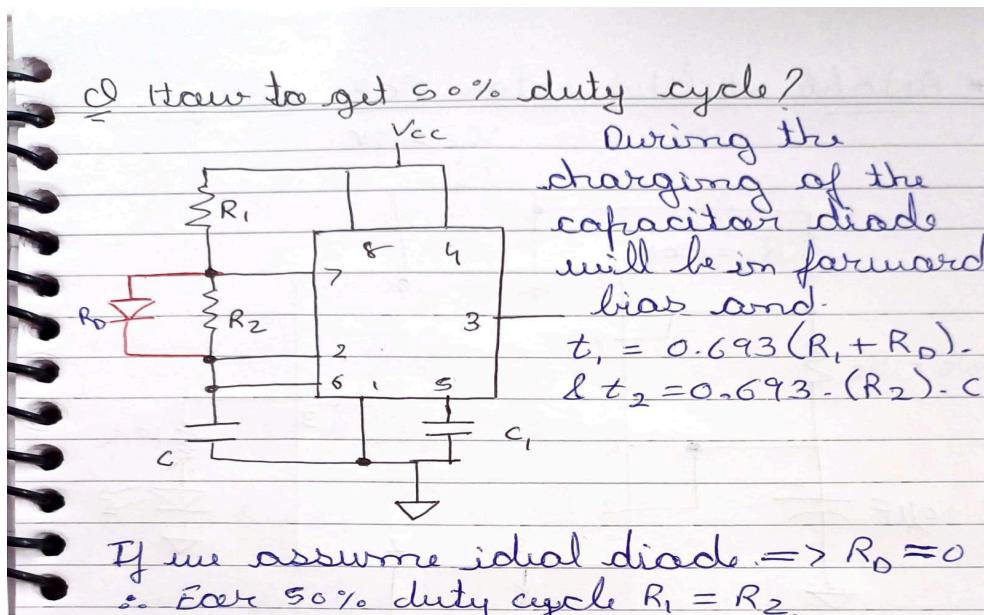


(Fig 14. Voltage across LED measured using Oscilloscope)



(Fig 15. Voltage across Capacitor measured using Oscilloscope)

- In this configuration $t_1 > t_2$ always, and hence, duty cycle will be always greater than 50%.
- To get a 50% duty cycle we need to make $R_1 = 0$. But if we do that then the collector region of the internal transistor will be directly exposed to supply voltage, and the transistor may get damaged.
- We can add a diode across R_2 to achieve this.



- In fact, we can also achieve a duty cycle of less than 50% by adjusting the value of R_1 in the above circuit.

Conclusion: In this project we learned how to design and fabricate a basic PCB, what are the errors and mistakes that can happen during the making and how to resolve them.