SYMBIOSIS INSTITUTE OF TECHNOLOGY (SIT)
Constituent of Symbiosis International (Deemed University), Pune
(Established under Section 3 of the UGC Act of 1956 vide notification number F-9-12/2001-U-3 of the Government of india)
Re-Accredited by NAAC with 'A++' Grade
॥वसुधैव कुटुम्बकम्॥

DEPARTMENT OF COMPUTER SCIENCE AND ENGINEERING

Printed Circuit Board Design CA-3 Report

Programme Name: B.Tech TYCS
Semester: VI

Under the Guidance of
Prof. **PARESH NASIKKAR**

Group Members:

Sr. No.	Name	PRN	Division
1	Abhishek Rajput	22070122007	CSE A
2	Omkar Darekar	22070122047	CSE A
3	Vasu Desai	22070122050	CSE A
4	Harsh Agrawal	22070122073	CSE A

Introduction

Printed Circuit Boards (PCBs) are essential components in electronic systems, allowing for reliable and compact circuit construction. This project focuses on the design, simulation, and layout of a transistor-based LED driver circuit using **KiCad**, an open-source PCB design tool. The final product involves converting a schematic into a PCB layout and visualizing it through 3D rendering.

Objectives

- Design a simple LED driver circuit using bipolar junction transistors (BJTs).
- Simulate and validate the schematic diagram.
- Generate the PCB layout with proper routing and grounding.
- Create a 3D view of the completed PCB for visual inspection.

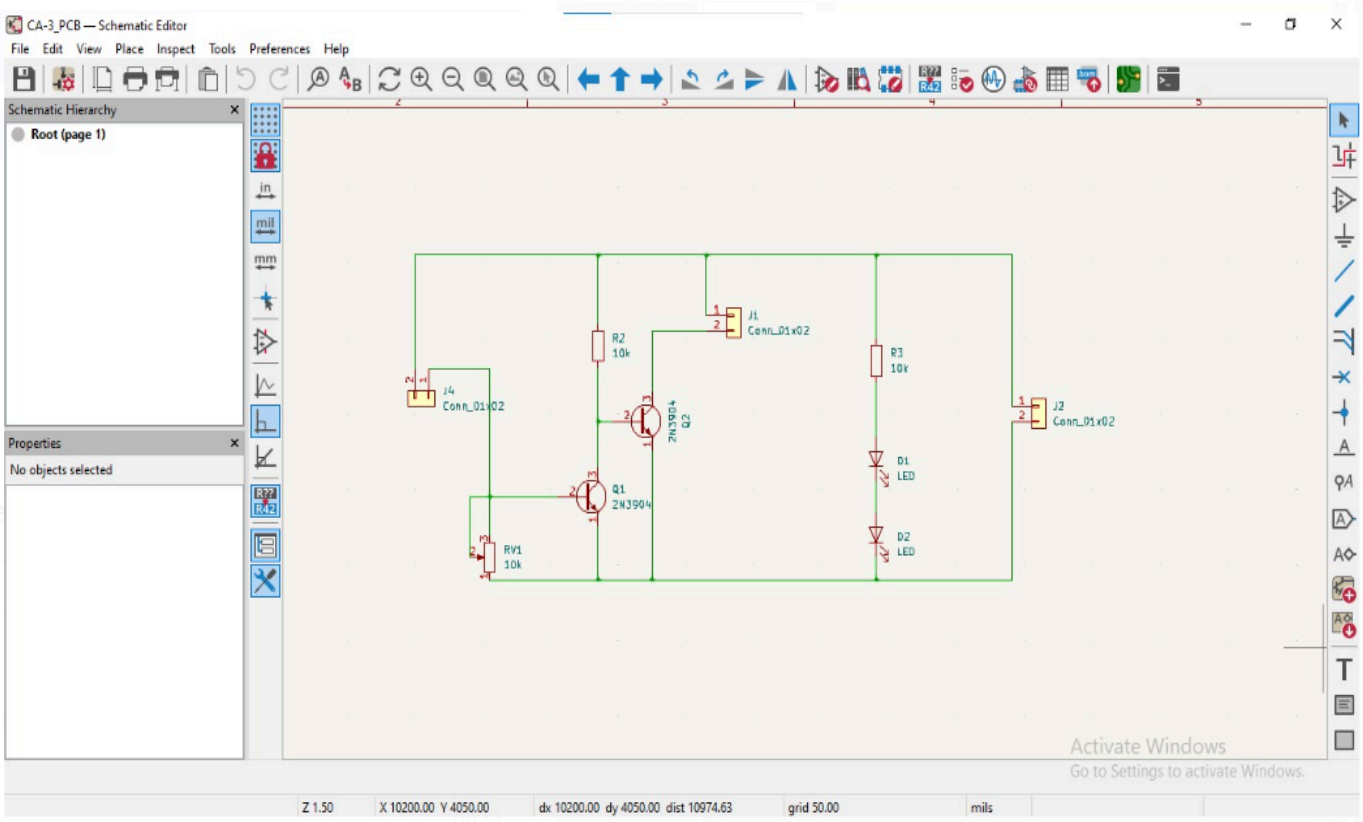
Components Used

- **2 x NPN Transistors (2N3904)**
- **3 x Resistors (10kΩ)**
- **2 x LEDs (Red)**
- **1 x Potentiometer (10kΩ, RV1)**
- **3 x Connectors (Conn_01x02)**
- **Power supply (via connectors J1 and J2)**

Schematic Design

- The schematic was created in KiCad's Schematic Editor.
  The circuit consists of:
- A potentiometer (RV1) used to vary the input voltage to the base of **Q1**.
- **Q1** and **Q2** act as switching transistors, driving the two LEDs.
- Resistors **R2** and **R3** limit the base and load current.
- LEDs **D1** and **D2** are connected in parallel but controlled individually.

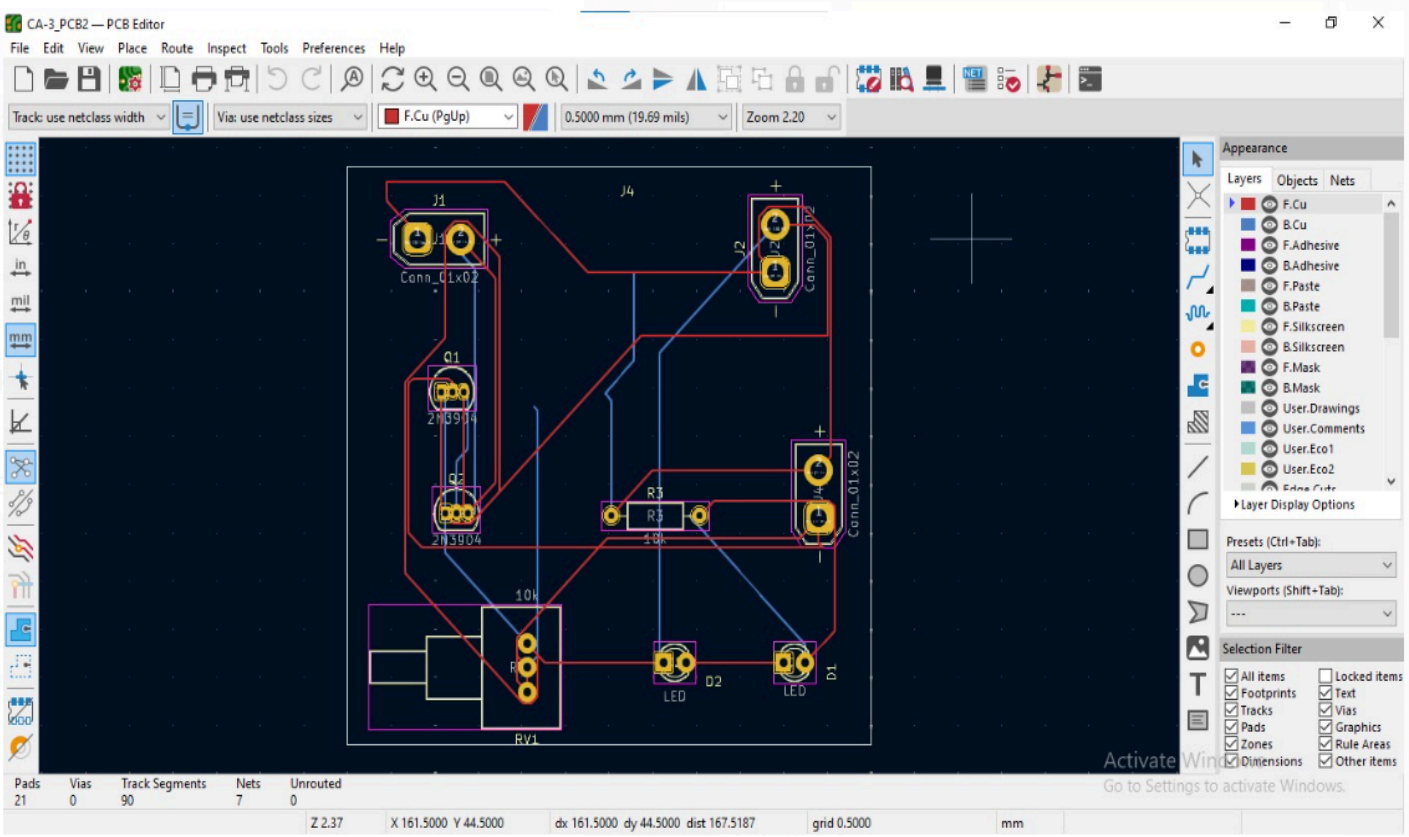
- Connectors **J1**, **J2**, and **J4** serve as power and control input terminals.

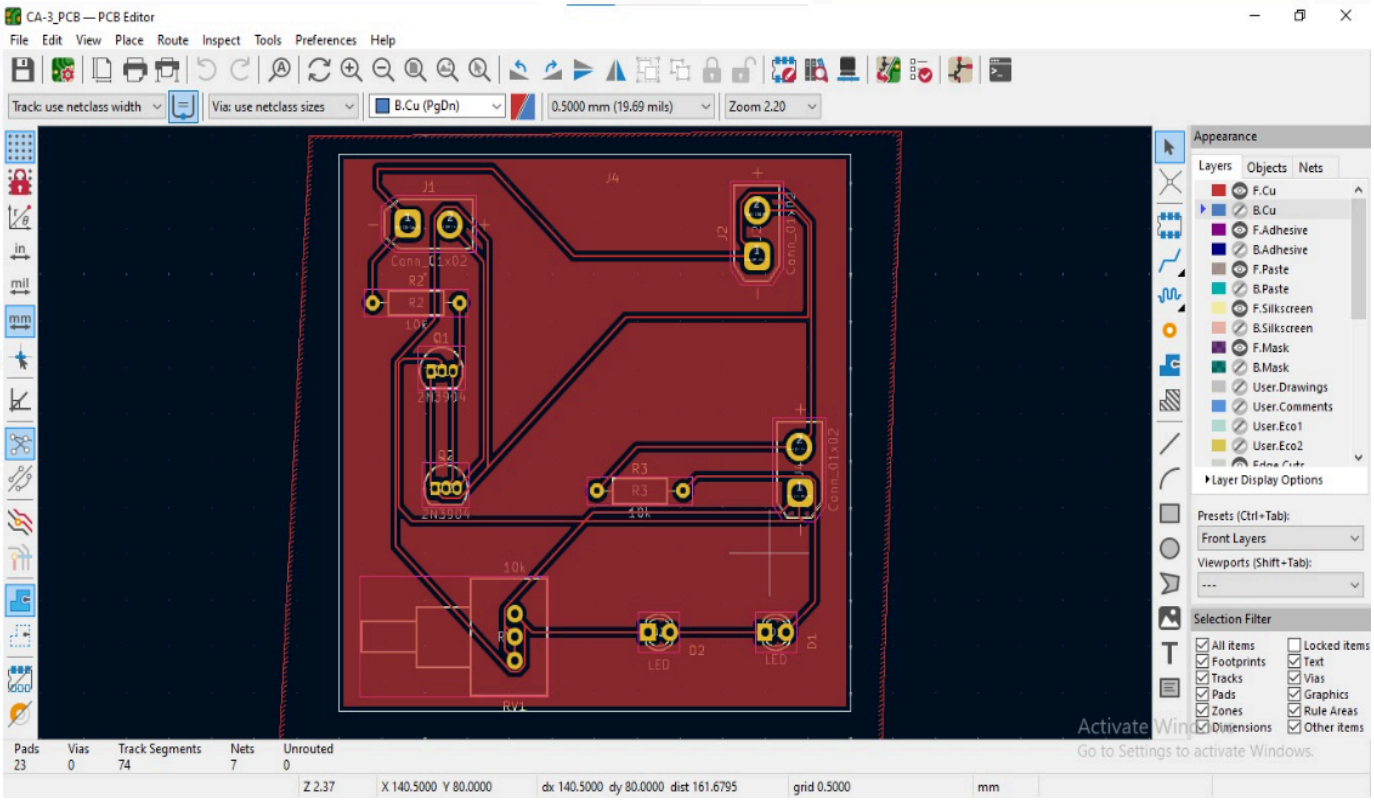


PCB Design

After validating the schematic, the PCB layout was created using the PCB Editor:

- The PCB is a two-layer board with top and bottom copper layers.
- Components were placed optimally to minimize the length of traces.
- Ground and VCC zones were created using fill zones (red areas).
- Routing was done manually for clarity and efficiency.
- Design rules were followed to ensure manufacturability.

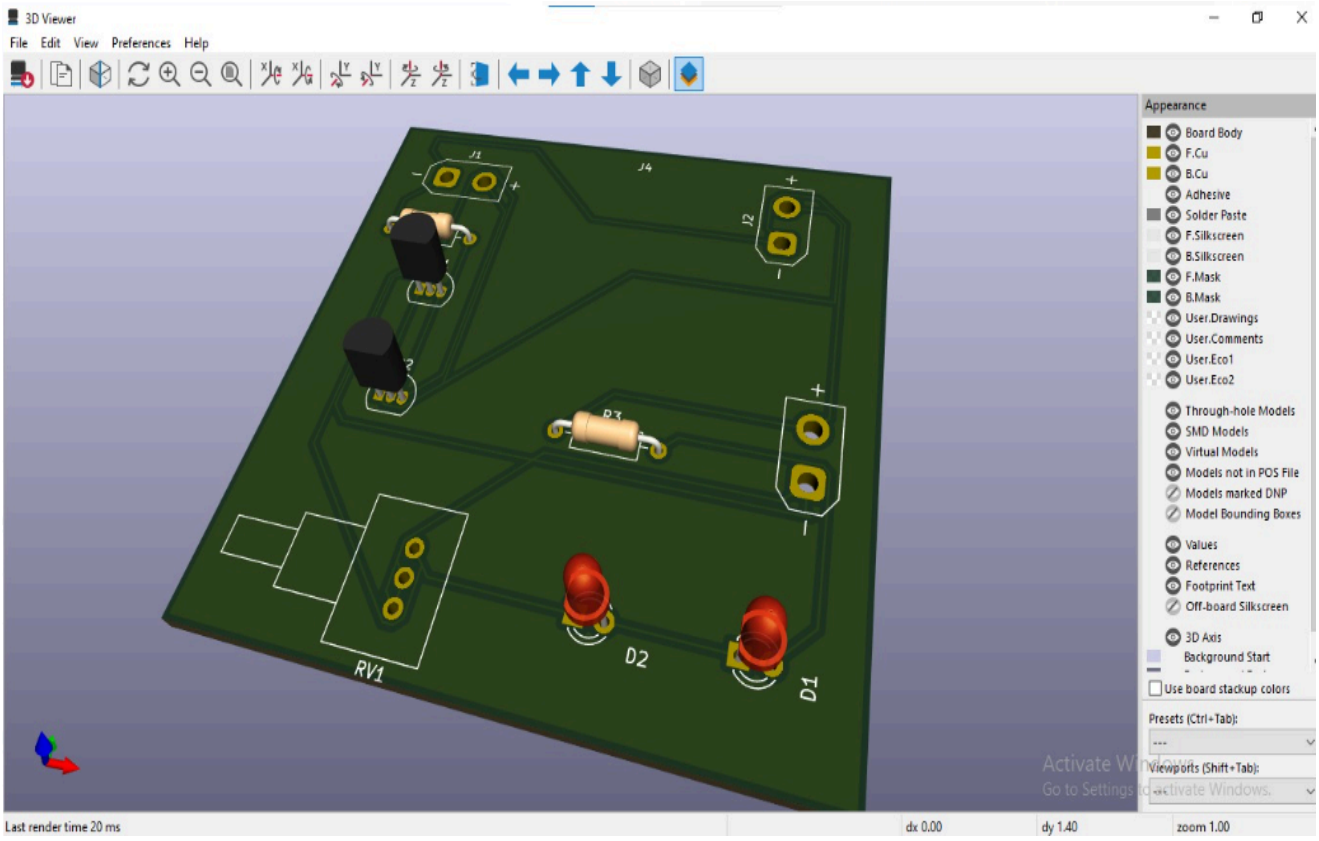

CA-3_PCB2 — PCB Editor
File Edit View Place Route Inspect Tools Preferences Help
Track: use netclass width
Via: use netclass sizes
F.Cu (PgUp)
0.5000 mm (19.69 mils)
Zoom 2.20
Appearance
Layers Objects Nets
F.Cu
B.Cu
F.Adhesive
B.Adhesive
F.Paste
B.Paste
F.Silkscreen
B.Silkscreen
F.Mask
B.Mask
User.Drawings
User.Comments
User.Eco1
User.Eco2
Layer Display Options
Presets (Ctrl+Tab):
All Layers
Viewports (Shift+Tab):
Selection Filter
All items
Footprints
Tracks
Pads
Zones
Dimensions
Locked items
Text
Vias
Graphics
Rule Areas
Other items
J1
J4
J2
Conn_01x02
Q1
2N3904
Q2
R3
10k
RV1
D2
D1
LED
Pads 21
Vias 0
Track Segments 90
Nets 7
Unrouted 0
Z 2.37
X 161.5000 Y 44.5000
dx 161.5000 dy 44.5000 dist 167.5187
grid 0.5000
mm


CA-3_PCB — PCB Editor
File Edit View Place Route Inspect Tools Preferences Help
Track: use netclass width
Via: use netclass sizes
B.Cu (PgDn)
0.5000 mm (19.69 mils)
Zoom 2.20
Appearance
Layers Objects Nets
F.Cu
B.Cu
F.Adhesive
B.Adhesive
F.Paste
B.Paste
F.Silkscreen
B.Silkscreen
F.Mask
B.Mask
User.Drawings
User.Comments
User.Eco1
User.Eco2
Layer Display Options
Presets (Ctrl+Tab):
Front Layers
Viewports (Shift+Tab):
Selection Filter
All items
Footprints
Tracks
Pads
Zones
Dimensions
Locked items
Text
Vias
Graphics
Rule Areas
Other items
J1
J4
J2
Conn_01x02
R2
10k
Q1
2N3904
Q2
R3
RV1
D2
D1
LED
Pads 23
Vias 0
Track Segments 74
Nets 7
Unrouted 0
Z 2.37
X 140.5000 Y 80.0000
dx 140.5000 dy 80.0000 dist 161.6795
grid 0.5000
mm

3D Visualization

The final board was rendered in the 3D Viewer, showing proper placement and orientation of all through-hole components:

- Transistors, resistors, potentiometer, and LEDs are correctly positioned.
- PCB traces are visible, with clear routing paths.
- Component footprints match the schematic and PCB layout.



Practical Implementation (Fabricated Board)

The physical fabrication of the PCB was completed and components were soldered as per the design:

- The copper etching and drilling were manually done or CNC-fabricated.
- Components were placed as per the 3D model.
- The board was tested and verified for correct functionality.

Conclusion

This project provided hands-on experience in circuit design, PCB layout, and 3D visualization using KiCad. The transition from schematic to a real-world PCB highlighted the importance of proper design techniques, layout optimization, and component placement. The LED driver circuit worked as expected, showcasing the effective use of BJTs in controlling load circuits.