



**Course Name:** Computer Engineering workshop (CEN 1006)

**LAB # 10:** Introduction to PCB Designing

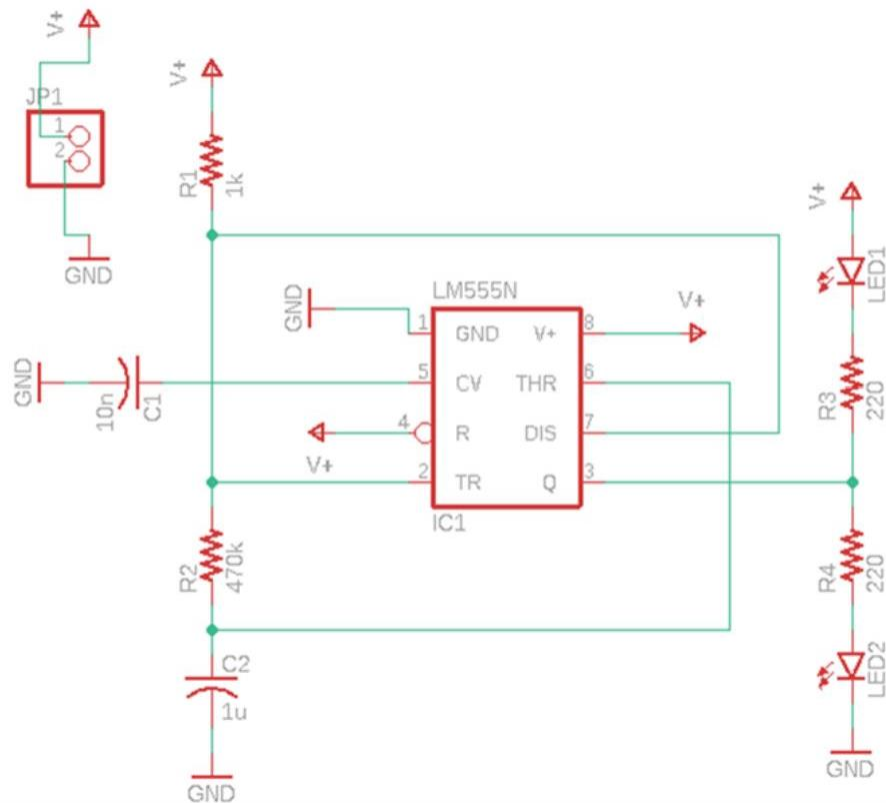
Department	Registration Number/Name	Semester/Section
BS CEN	F24604018/Muhammad Hamzah Iqbal	1
Date	Instructor's Name	Instructor's Signature
12/12/2024	Maj Sheryar /Iqra Ashraf	

**Objective:**

- To get familiar with PCB designing using Eagle Circuits.

### Lab Task:

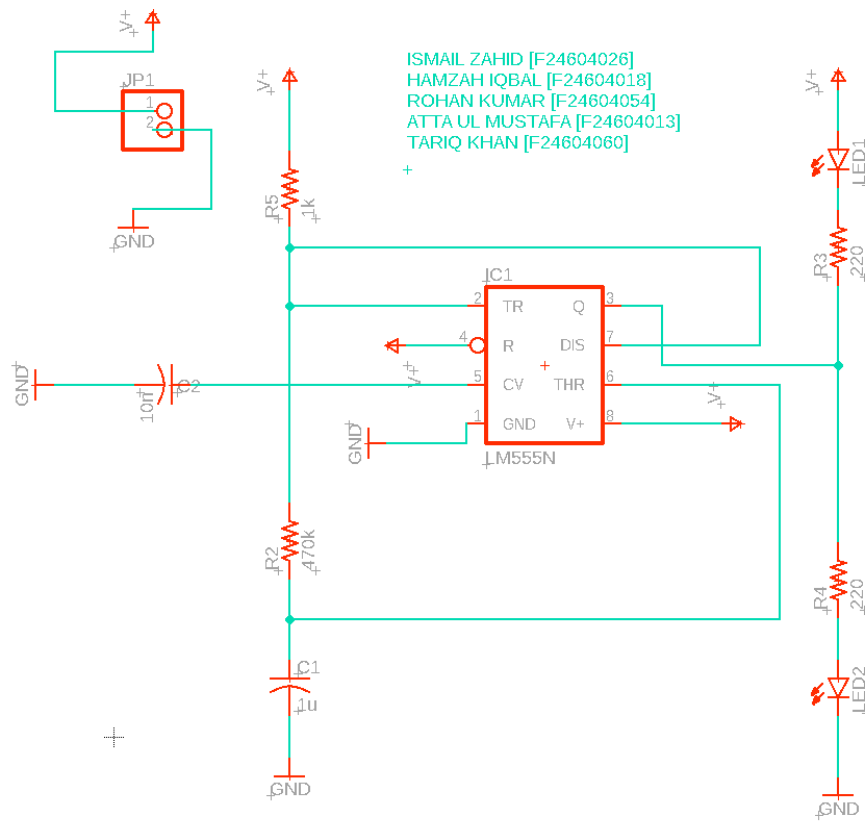
Make PCB of the following circuit. Also print your NUTECH ID and LAST NAME on the top right corner of your board.



Output:

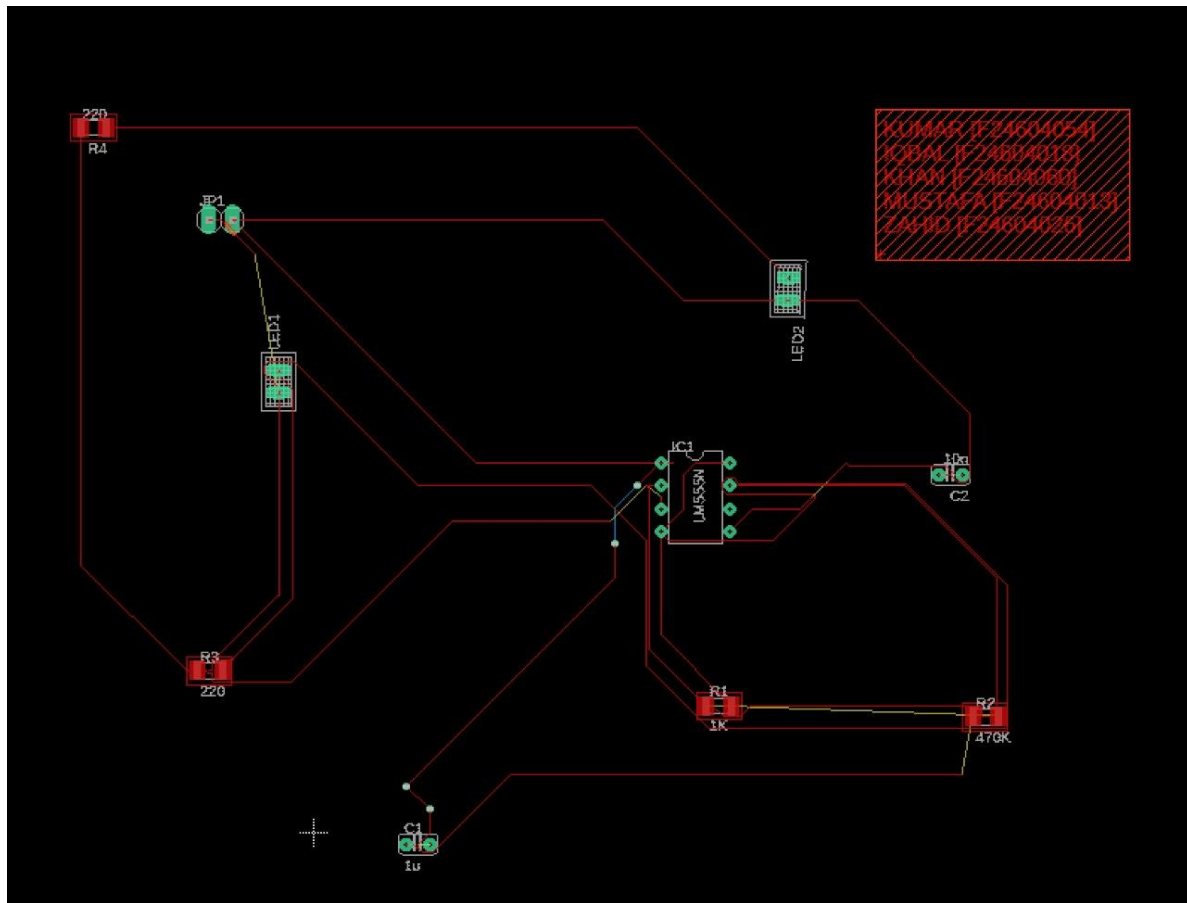
Procedure:

- Open the Eagle circuit board design software.
- Click on the project and then right click to select schematic.
- Click on 'Add part' to place components and design the desired circuit.
- This completes the schematic part of the circuit which was also designed in precious lab

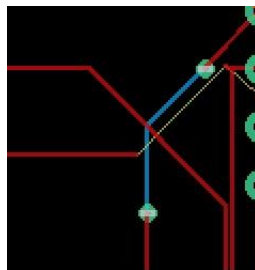


Activate Windows  
Go to Settings to activate Windows.

- Now Select the “switch to board option from the top tool bar”.
- A new window opens which is the PCB layout editor. The black space area, and all the components are at the outside bottom left of the board area. Now, we need to place the components into the editor.
- Now using the group option from the side tool bar, select all the components and using move all the components.
- Using the move option, place the components on the board as per the position you want the components to be on the board. You can see thin yellow wires running between the components. These wires are called air wires and are representations of connections between components. When we route the path between components, these air wires will disappear as an indication of successful connection.
- Now, make the connections or traces for the printed circuit board. For this, use the ‘Route Tool’ from the side tool bar.
- Make sure the lines don't cross each other.



Using Via the overlapping of wires was done



## Conclusion:

Eagle software is a powerful tool that can significantly streamline the PCB design process. It provides a user-friendly interface and a comprehensive set of features, including:

- Schematic capture: Design and simulate electronic circuits.
- PCB layout: Create professional-quality PCB layouts.
- Library creation: Build custom components for your designs.
- Routing automation: Automatically route traces on your PCB.
- Design rule checking: Ensure your design adheres to industry standards.

By using Eagle software, engineers can quickly and efficiently create high-quality PCB designs, reducing design time and increasing productivity.