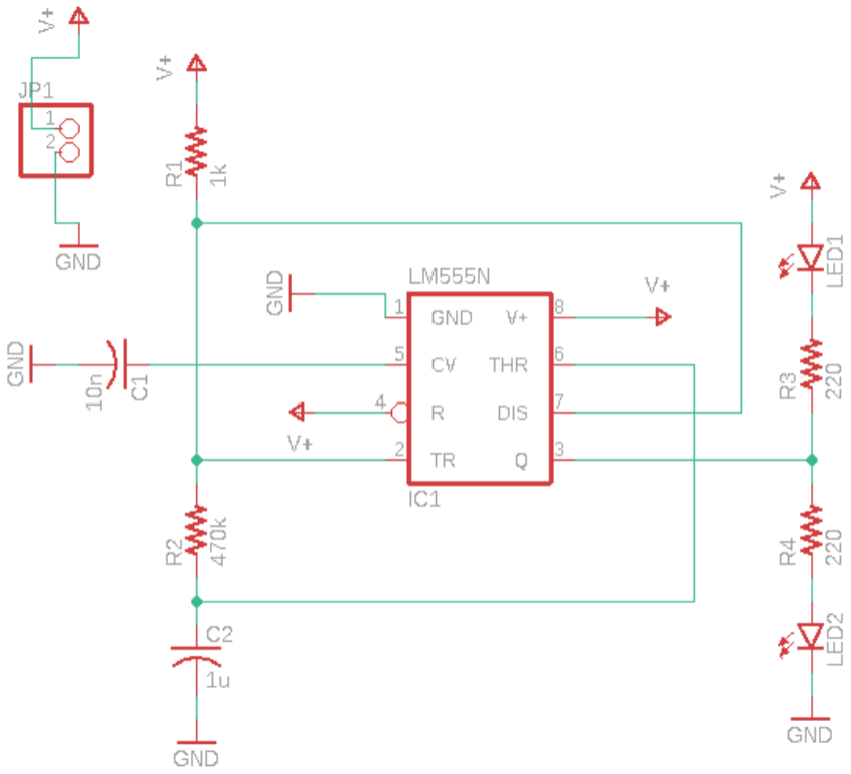
Course Name: Computer Engineering workshop (CEN 1006)		
LAB # 9: Introduction to Basic Circuit Designing		
Department	**Registration Number/Name**	**Semester/Section**
BS CEN	F24604018/Muhammad Hamzah Iqbal	1
Date	**Instructor's Name**	**Instructor's Signature**
8/12/2024	Maj Sheryar /Iqra Ashraf	

Objective:

- To familiarize students with Eagle.

Lab Task:

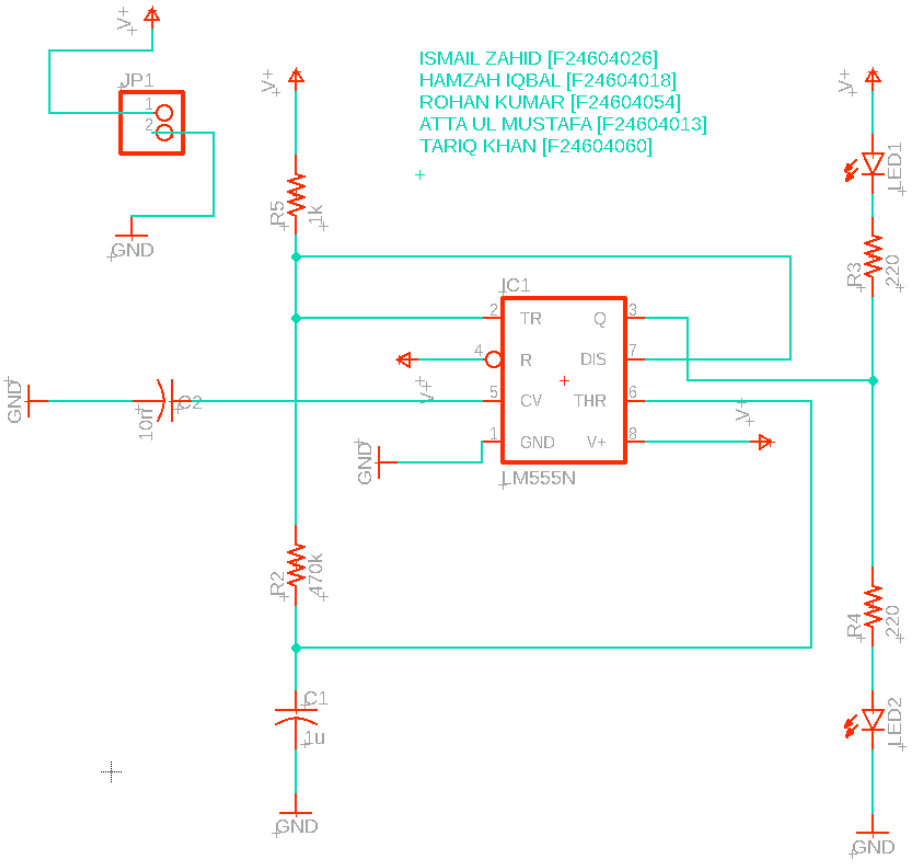
Draw the given circuits in Eagle:



Output:

Procedure: -

- Open Eagle, create a new project, and start a new schematic.
- Use the **"Add"** tool to place components (555 timer, resistors, capacitors, LEDs, diodes, and connecter known as pinhead) on the schematic.
- Search **"rcl"** for resistors and Capacitors, and search "**Pinhead"** for connector.
- For ground search **"supply"**
- Arrange the components logically with the 555 timer at the center.
- Use the **"Net"** tool to wire the components as per the schematic (connect V+, GND, and other pins as shown).
- Name the nets (e.g., V+, GND) and assign values to components (e.g., 10µF for C2, 220Ω for R3).
- Save the project.



Conclusion:

Eagle is a useful software for designing electronic circuits and PCBs. It simplifies the creation of high-quality schematics with precise component placement and interconnections. With its interactive interface, comprehensive component library, and error-checking tools, Eagle ensures efficient and accurate circuit design. This makes it an ideal choice for students and professionals to turn their electronic ideas into reality while reducing errors and saving time during the design process.