UQ MARS Individual Project:
RGB LED Controller PCB v1

Project Overview

Subsystem: Electronics
Authors: Oscar Lloyd (2025), Binara Wasala (2025)
Mentor(s): Binara Wasala, Ganesh Srinivasa.
Discord Help: Projects Channel
Time Estimate: 1-3 Weeks

Project Difficulty:

Mechanical - ☆☆☆☆☆
Electrical - ★★☆☆☆
Software - ★☆☆☆☆

Project Context

The goal of this project is to design a simple but effective PCB to manually control an RGB LED. This is a great way to get hands-on experience with PCB design, circuit fundamentals, and soldering.

Getting Started Resources

- **KiCad Download** – Our recommended PCB design software.
- **KiCad Setup Tutorial** – A step-by-step guide to installing KiCad.
- **Introduction to PCBs** – A beginner-friendly UQ MARS workshop on designing PCBs in KiCad.
- **Introduction to KiCad** – A detailed tutorial series on KiCad PCB design.
- **JLC PCB Capabilities** – Ensure your design follows their manufacturing constraints (rigid PCB).
- **UQ MARS Materials List** – A list of UQ MARS components available for use in this project. Anything not included in this list must be ordered yourself.

Have you followed these resources and you're still stuck? Jump into the **UQ MARS Discord** and post in the **#Projects Channel** for help from mentors and fellow members!

UQ M.A.R.S
Mechatronics and Robotics Society

Project Objective

By the end of this project, you will:

- Design your own basic PCBs.
- Gain confidence using KiCad for PCB design (or another preferred PCB Design software).
- Understand key electronic design principles, including the role of resistors and capacitors.
- Take home a fully functional RGB LED controller you built yourself!
- Learn and apply soldering techniques to assemble your PCB.

Project Requirements

In Scope:

- The LED or LED strip should be able to display all RGB colours (0-255 for red, green, blue).
- Three control knobs will adjust the red, green, and blue light levels.

Out of Scope:

- No automatic LED colour transitions – full manual control is fine.
- The circuit will be powered via USB (no batteries needed).

Functional Requirements and Constraints

Functional Requirements:

- The circuit must work with a single **RGB LED** or a **non-addressable LED strip**.
- Colour intensity of the LED will be adjusted using **potentiometers**.
- **MOSFETs** should control the LED terminals rather than direct potentiometer connections.
- The power source must be **5V USB**, soldered to the PCB's positive and negative pads.
- The PCB design should be limited to **2 layers**.
- Sufficient supporting circuitry (e.g., pull-up resistors, bulking capacitors) should be included.

Specifications/Constraints:

- No **microcontrollers** should be used.
- **ICs are optional**—check with mentors if you're considering one.
- The entire system must operate on **5V USB** without additional power management.
- Components should be **affordable and easy to source**.
- The design doesn't need to be ultra-durable—just functional and manufacturable.
- Avoid using SMD (Surface-Mount Devices).

Project Phases and Timeline

1. **Phase 1: Understanding the Problem**
    - Research and plan how the circuit will function.
    - Sketch rough schematics of your design.
2. **Phase 2: Design and Planning**
    - Create detailed schematics with **component values**.
    - Assemble the circuit on a **breadboard** to test functionality.
    - Reach out for parts if needed!
3. **Phase 3: Implementation**
    - Fully design the PCB in **KiCad** and get a mentor to review it.
    - Once approved, we'll **order your PCB** and parts.
    - Assemble and solder your PCB when it arrives!
4. **Phase 4: Testing and Refinement**
    - Use a **power supply unit (PSU)** in the lab to test your PCB.
    - If everything works, congratulations—you're done!

Additional Considerations

- **Cost Efficiency:** Keep it budget-friendly while meeting the requirements.
- **Manufacturability:** Ensure the design aligns with JLC PCB's manufacturing capabilities and can be soldered by hand (you).

Deliverables

At the end of the project, you should be able to show:

- **A schematic** of your design (hand-drawn or from KiCad).
- **Your PCB**—either in its raw form or fully soldered and assembled.

Mentor Notes

- Before routing your PCB in KiCad, make sure you check JLCPCB's 2-layer manufacturing constraints. If you're unsure, ask for help! - Binara 2025
