**Steps to Make a PCB for Your GPS Tracker**

**1. Design the Circuit**

Before creating a PCB, you need a clear schematic that shows how the components are connected.

**Components in Your GPS Tracker:**

* **Microcontroller**: ESP32-PICO-D4, STM32, or ATmega328P.
* **GSM + GPS Module**: SIM808, SIM868, or Quectel MC60/MC65.
* **Power Management**:
  + Voltage regulator (e.g., AMS1117 or LM2596) if needed.
  + Li-Po battery (3.7V, slim) + charging module (TP4056).
* **Antennas**: Flexible or PCB-based GSM and GPS antennas.
* **Other Components**: Capacitors, resistors, and connectors for debugging.

You can use software like **Fritzing** to create a simple wiring diagram or **KiCad** and **Eagle** for professional schematic design.

This is the Hardware