1 PCB Design & Fabrication *(Altium, KiCad, Eagle) – 4 Weeks*

📅 Week 1: Introduction to PCB Design

- **Basics of PCB, Layers & Components**
- **Understanding Schematics**
- **Introduction to KiCad / Eagle / Altium Designer**

📅 Week 2: Schematic Design & Component Placement

- **Creating Circuit Schematics**
- **Footprints and Libraries**
- **Netlist & DRC checks**

📅 Week 3: PCB Layout & Routing

- **PCB Layout Techniques (1-layer, 2-layer)**
- **Manual & Auto Routing**
- **Ground Planes, Vias & Tracks**

📅 Week 4: Fabrication & Testing

- **Gerber File Generation**
- **BOM Creation**
- **Online Fabrication Services**
- **Soldering & Testing Demo**

🎯 **Mini Project: Design & simulate a PCB for a power supply or sensor module.**