ROHINI
COLLEGE OF ENGINEERING & TECHNOLOGY
(AUTONOMOUS)
Approved By AICTE & Affiliated to Anna University
NBA Accredited for All Eligible B.E/B.Tech Programs | NAAC Accredited with A+ Grade
Near Anjugramam - Kanyakumari Main Road, Palkulam, Variyoor P.O.-629401, Kanyakumari Dist, Tamil Nadu.

DEPARTMENT OF BIOMEDICAL ENGINEERING

VI Semester

24BM501 BIOMEDICAL DEVICE DESIGN

4.3 PCB Design and Fabrication

➢ 4.3.1 Printed Circuit Board (PCB)

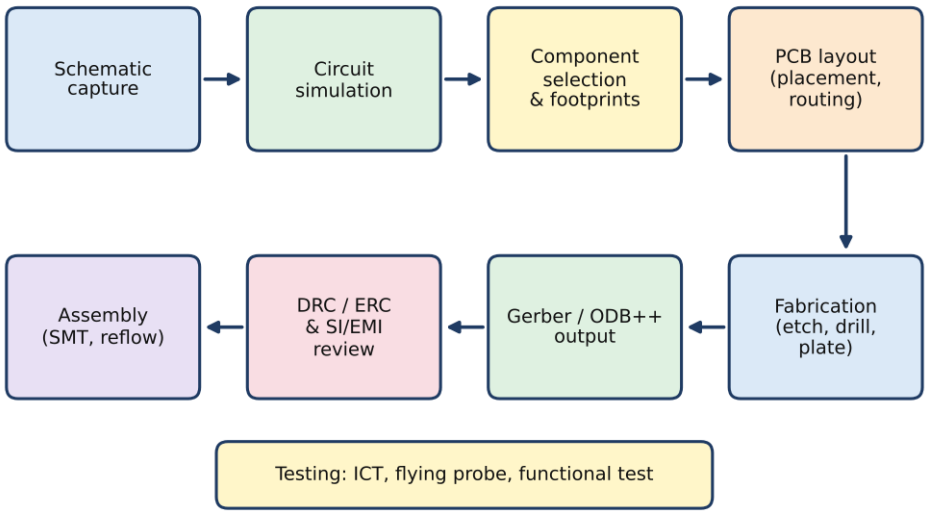
A **printed circuit board** is a rigid or flexible insulating board on which thin copper tracks connect electronic components. It replaces point-to-point wiring, giving a compact, reliable and reproducible circuit. Almost every biomedical device (ECG monitor, pulse oximeter, glucometer, infusion pump) contains one or more PCBs.

➢ 4.3.2 Types of PCBs

Type	Description	Biomedical use
Single-sided	Copper on one side only	Simple sensors, low-cost disposables
Double-sided	Copper on both sides with plated through-holes	Portable monitors, controllers
Multilayer (4, 6, 8+ layers)	Several copper layers with inner power and ground planes	Patient monitors, imaging electronics
Flexible (Flex)	Polyimide base that bends	Wearable patches, catheters, hearing aids
Rigid-flex	Rigid boards joined by flex sections	Compact implants, endoscopes
High-density interconnect (HDI)	Micro-vias and fine lines	Smartwatches, miniature implantables
Metal-core (MCPCB)	Aluminium base for heat removal	LED phototherapy, surgical lights

➢ 4.3.3 PCB Design Flow

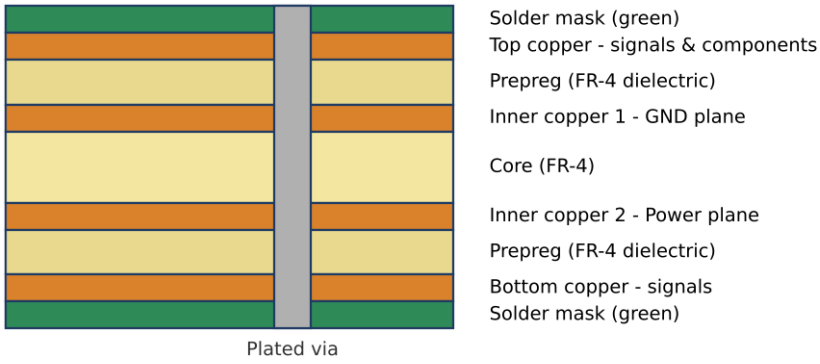
Fig. 4.7 PCB Design and Fabrication Flow



Step	Activity
1. Schematic capture	Draw the circuit in EDA tools (KiCad, Altium, Eagle, OrCAD) with symbols and net names
2. Simulation	Verify amplifier gain, filter response and power supply behaviour using SPICE
3. Component selection	Choose parts for performance, availability, medical-grade reliability; assign footprints
4. Board outline and stack-up	Define board shape, layers, thickness and material
5. Placement	Place components by function; keep analog, digital and power sections separate
6. Routing	Draw copper tracks; define width, spacing, vias and planes
7. Design rule check (DRC) and electrical rule check (ERC)	Detect shorts, clearance and connection errors
8. Output generation	Gerber files, drill file, bill of materials (BOM), pick-and-place file

➢ **4.3.4 Layer Stack-up of a Multilayer Board**

Fig. 4.8 Cross-section of a 4-Layer PCB Stack-up



- ✓ **Signal layers** carry tracks and components.
- ✓ **Ground plane** gives a low-impedance return path and shields noise.
- ✓ **Power plane** distributes supply voltage with low drop.
- ✓ **FR-4** (glass-epoxy) is the most common base material; **polyimide** is used for flexible boards.
- ✓ **Vias** connect layers (through, blind, buried).

➢ **4.3.5 Design Guidelines for Biomedical PCBs**

➢ **A. Signal integrity and noise**

- ✓ Keep **analog front-end** (ECG, EEG, EMG) away from switching regulators and digital clocks.
- ✓ Use a **solid ground plane** and **star or split grounding** with a single join point.
- ✓ Keep high-impedance input traces **short**, guarded and away from noisy tracks.
- ✓ Place **decoupling capacitors** (0.1 µF) close to every IC power pin.
- ✓ Route differential pairs together with equal length.

➢ **B. Patient safety and isolation**

- ✓ Maintain **creepage and clearance distances** between patient-connected circuits and mains, as given in IEC 60601-1 (for example, 2 × MOPP at the applied voltage).
- ✓ Use **isolation barriers** (opto-isolators, isolated DC-DC converters, digital isolators) for applied parts.
- ✓ Cut **slots** in the board under isolation barriers to increase creepage.
- ✓ Include **defibrillation and ESD protection** (TVS diodes, series resistors) on patient inputs.

➢ **C. Thermal, power and mechanical**

✓ Use sufficient **track width** for current (IPC-2221); add thermal vias under power devices.

✓ Keep **battery and charging** circuits away from sensitive sensors.

✓ Provide mounting holes, connectors at board edge and keep-out zones for enclosure fit.

➢ **D. Manufacturability (DFM) and testability (DFT)**

✓ Follow the fabricator's minimum track/space and drill limits.

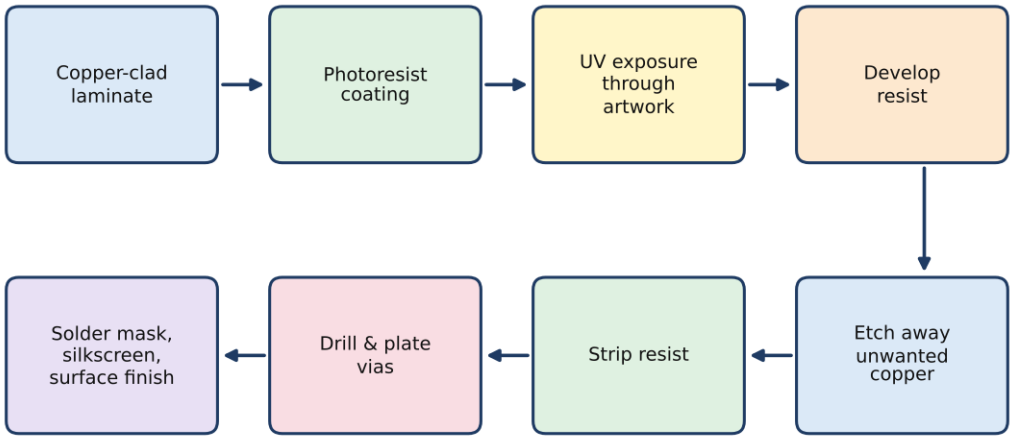
✓ Add **fiducials**, test points and programming headers.

✓ Use standard components and footprints; avoid single-source parts.

✓ Provide silkscreen with polarity, pin-1 marks and reference designators.

➢ **4.3.6 PCB Fabrication Process**

Fig. 4.9 Subtractive Etching Process for PCB Fabrication



Stage	Description
Copper-clad laminate	FR-4 board with a copper foil bonded on one or both sides
Photoresist coating	Light-sensitive film applied on the copper
Exposure	UV light passes through the film artwork (Gerber); exposed resist hardens
Developing	Unexposed resist is washed away, leaving a protected track pattern
Etching	Ferric chloride or ammonium persulphate removes unprotected copper
Stripping	Remaining resist is removed to reveal copper tracks
Drilling and plating	Holes drilled by CNC; copper plated to connect layers
Solder mask and silkscreen	Green mask protects copper; text legend printed
Surface finish	HASL, ENIG (gold) or OSP to keep pads solderable
Electrical test and inspection	Flying-probe/bed-of-nails test and automated optical inspection (AOI)

➢ **4.3.7 Multilayer Fabrication**

- ✓ Inner layers are etched first, then stacked with **prepreg** (partially cured glass-epoxy) sheets.
- ✓ The stack is **laminated** under heat and pressure.
- ✓ Holes are drilled, **plated** and outer layers are patterned.
- ✓ Blind and buried vias use sequential lamination (HDI boards).

➢ **4.3.8 PCB Prototyping Methods**

Method	Description	When used
Breadboard / perfboard	No etching, hand wiring	Early proof of concept
Toner transfer / home etching	Laser-printed pattern transferred to copper	Low-cost simple boards
CNC isolation milling	Router mills copper to form tracks	In-lab quick-turn single/double sided
Quick-turn fabrication service	Professional board in 2 - 5 days	Works-like and engineering prototypes
Production fabrication	Volume process with strict quality	Pre-production and production

➢ **4.3.9 PCB Assembly (PCBA)**

- ✓ **Solder paste printing** through a stencil onto pads.
- ✓ **Pick-and-place** machines position surface-mount (SMT) components.
- ✓ **Reflow soldering** melts the paste in a controlled temperature profile.
- ✓ **Through-hole** parts are inserted and wave or hand soldered.
- ✓ **Inspection:** AOI, X-ray (BGA), manual inspection to IPC-A-610.
- ✓ **Testing:** in-circuit test (ICT), flying probe, functional test and burn-in.
- ✓ **Conformal coating** protects against moisture and sweat in wearable devices.

➢ **4.3.10 Standards Related to PCBs in Medical Devices**

- ✓ **IPC-2221 / IPC-2222:** generic PCB design.
- ✓ **IPC-6012 (Class 3):** high-reliability boards, used for medical and life-support devices.
- ✓ **IPC-A-610:** acceptability of electronic assemblies.
- ✓ **IEC 60601-1:** electrical safety, creepage and clearance.
- ✓ **IEC 60601-1-2:** electromagnetic compatibility (EMC).
- ✓ **RoHS / REACH:** restricted hazardous substances.

Example: Compact double-sided PCB for a wearable pulse monitor

ESP32-based board of about 30 × 20 mm with MAX30102 sensor, LDO regulator and Li-Po charger. Analog sensor section on one side, digital and wireless section on the other; ground pour on both layers; test points for power and I2C bus.