

# Design Basics

---

## PCB Design Basics

---

### Overview

---

PCB design transforms circuit concepts into physical boards. This guide covers the fundamental knowledge every designer needs.

### Step 1: Understanding Requirements

---

#### Circuit Analysis:

- Define circuit functionality
- Identify components needed
- Determine operating conditions
- Specify performance requirements

#### Design Constraints:

- Board size limitations
- Layer count requirements
- Budget considerations
- Manufacturing capabilities

### Step 2: Schematic Design

---

#### Creating the Schematic:

- Choose EDA software (KiCad, Eagle, Altium, etc.)

- Place component symbols
- Make electrical connections
- Add designators and values
- Verify net connectivity

#### Schematic Best Practices:

- Use standard symbols
- Label nets clearly
- Group related components
- Add notes and annotations
- Perform ERC (Electrical Rule Check)

## Step 3: Component Selection

---

#### Key Considerations:

- Package type (SMD, through-hole)
- Availability and lead time
- Cost considerations
- Datasheet specifications
- Footprint availability

## Step 4: PCB Layout

---

#### Board Setup:

- Define board outline
- Set layer stack-up
- Establish design rules
- Configure grid spacing

#### Component Placement:

- Place connectors on edges
- Group related components
- Consider signal flow
- Optimize for routing
- Allow for thermal management

#### Routing Guidelines:

- Power traces first (wide traces)
- Critical signals next (controlled impedance)
- General signals last
- Follow design rules (DRC)
- Minimize via count

## Step 5: Design Verification

---

#### Design Rule Check (DRC):

- Clearances and spacing
- Trace widths
- Via dimensions
- Unrouted nets

#### Electrical Rule Check (ERC):

- Short circuits
- Open connections
- Power/Ground issues

## Step 6: Output Generation

---

### Manufacturing Files:

- Gerber files (all layers)
- Drill files
- Fabrication drawings
- Assembly drawings
- Pick and place files

### Documentation:

- Bill of Materials (BOM)
- Assembly notes
- Test specifications

## Common Beginner Mistakes

---

- **Inadequate decoupling** - Place caps near ICs
- **Poor grounding** - Use solid ground planes
- **Ignoring trace width** - Calculate for current
- **Wrong footprints** - Verify datasheet dimensions
- **No design rules** - Set up DRC early

## Tools and Resources

---

### Popular EDA Software:

- KiCad (free, open-source)
- Eagle (free for small boards)
- Altium Designer (professional)
- EasyEDA (browser-based)

### Learning Resources:

- Manufacturer tutorials
- Online courses
- PCB design forums
- Application notes

---

References: *JLCPCB Beginner Guide*, *All About Circuits*, *ProtoExpress Schematic Guide*