

Routing Rules

PCB Routing Rules and Best Practices

Overview

Proper PCB routing ensures signal integrity, manufacturability, and reliable operation. This guide covers essential routing rules.

Fundamental Routing Rules

1. Trace Width Rules

Current-Carrying Capacity:

- Signal traces: 6-10 mil (0.15-0.25mm)
- Power traces: Calculate based on current
- General formula: 1A per 10-20 mil width for outer layers

Impedance-Controlled Traces:

- Use calculator or field solver
- Depends on dielectric and stack-up
- Typical single-ended: 50-75 Ω
- Differential pairs: 90-100 Ω

2. Trace Spacing Rules

The 3W Rule:

- Distance between traces $\geq 3 \times$ trace width

- Critical for high-speed signals
- Prevents crosstalk

Clearance Requirements:

- Low voltage: 6-8 mil minimum
- High voltage: Calculate based on IPC-2221
- Manufacturing limits: Follow fabricator capabilities

3. Via Rules

Via Sizing:

- Drill diameter: 0.2-0.3mm standard
- Pad diameter: 0.4-0.6mm
- Aspect ratio < 10:1 for reliable plating

Via Placement:

- Minimize via count in signal paths
- Use vias in pads for fine-pitch BGAs
- Avoid vias under components unless filled

4. Differential Pair Routing

Requirements:

- Match lengths within 5-10 mil
- Maintain constant spacing
- No vias between pair members
- Symmetrical routing around obstacles

5. High-Speed Routing Rules

Critical Practices:

- Reference planes must be continuous
- Avoid 90° bends (use 45° or curves)
- Minimize layer transitions

- Don't route over plane splits
- Keep traces as short as possible

Design Rule Check (DRC)

Essential DRC Settings:

- Minimum trace width
- Minimum clearance
- Minimum via size
- Maximum via count
- Length matching tolerances

Routing Order Strategy

Recommended Sequence:

- Power and ground planes
- High-current power traces
- Controlled impedance signals
- Differential pairs
- Critical timing signals
- General signal routing
- Test points and connections

Common Routing Mistakes

- **Trace width too narrow** - Causes overheating
- **Insufficient spacing** - Creates crosstalk/arcing
- **90° angles** - Causes impedance discontinuities
- **Long stubs** - Creates signal reflections
- **Ignoring return paths** - Degrades signal integrity

Advanced Techniques

Length Matching

- Use serpentine for adjustment
- Match within timing budgets
- Consider propagation delay

Layer Transitions

- Use ground vias nearby
- Minimize discontinuities
- Maintain return path

Manufacturer Considerations

- Verify minimum capabilities
- Check impedance tolerances
- Confirm via types supported
- Review annular ring requirements

References: [Complete PCB Routing Guide](#), [DRC Guidelines](#), [Signal Integrity Basics](#)