Design Mistakes

Common PCB Design Mistakes and How to Avoid Them

Overview

Learning from common mistakes can save time, money, and frustration. This guide covers the most frequent PCB design errors and their solutions.

Top 21 PCB Design Mistakes

Layout and Footprint Issues

1. Incorrect Landing Patterns

- Problem: Footprint doesn't match actual component dimensions
- Solution: Always verify with datasheet and manufacturer drawings
- Impact: Components won't fit or solder properly

2. Insufficient Annular Ring

- Problem: Not enough copper around drilled holes
- Solution: Maintain ≥0.15mm annular ring
- Impact: Reliable connections compromised

3. Inadequate Drill-to-Copper Distance

- **Problem**: Holes too close to copper features
- **Solution**: Follow manufacturer minimum spacing rules
- **Impact**: Short circuits and manufacturing failures

Trace and Spacing Problems

4. Inadequate Trace Width

- **Problem**: Traces too narrow for current requirements
- **Solution**: Calculate width based on current using IPC-2221
- **Impact**: Overheating, voltage drop, failure

5. Insufficient Trace Spacing

- **Problem**: Traces too close together
- **Solution**: Follow 3W rule for high-speed, maintain clearance
- **Impact**: Crosstalk, arcing, short circuits

6. Uneven Copper Shapes

- **Problem**: Unbalanced copper distribution
- **Solution**: Use copper pouring and thieving
- **Impact**: Warpage, plating issues

Component Placement Errors

7. Poor Component Placement

- **Problem**: Components too close or poorly oriented

- **Solution**: Follow placement guidelines, allow adequate spacing
- **Impact**: Assembly difficulties, routing issues

8. Sub-Optimal Antenna Layout

- **Problem**: Antenna placement affects performance
- **Solution**: Follow RF design guidelines, keep area clear
- **Impact**: Poor wireless performance

9. Decoupling Capacitor Location

- **Problem**: Capacitors placed too far from ICs
- **Solution**: Place decoupling caps as close as possible to power pins
- **Impact**: Power supply noise, erratic behavior

Copper and Plane Issues

10. Open Copper Areas

- **Problem**: Unconnected copper causing solderability issues
- **Solution**: Connect copper to net or remove
- **Impact**: Soldering problems

11. Unbalanced Stack-ups

- **Problem**: Layer stack-up not symmetrical
- **Solution**: Use symmetrical stack-up design
- **Impact**: Board warpage, reliability issues

12. Poor Reflow Considerations

- **Problem**: Thermal mass imbalances

- **Solution**: Balance copper distribution, consider thermal relief
- **Impact**: Soldering defects

Connectivity and Electrical Problems

13. Unconnected Pins (Floating Inputs)

- **Problem**: Left as no-connects when they shouldn't be
- **Solution**: Provide proper pull-up/pull-down or connection
- **Impact**: Erratic behavior, excess power consumption

14. Missing Net Connections

- **Problem**: Incomplete connections in schematic
- **Solution**: Run ERC, verify all connections
- **Impact**: Circuit doesn't work

15. Incorrectly Shorted Nets

- **Problem**: Accidental connections between nets
- **Solution**: Careful schematic review, use DRC
- **Impact**: Short circuits, damaged components

DFM-Specific Issues

16. EOL (End-of-Life) Parts

- **Problem**: Using discontinued components
- **Solution**: Check component lifecycle before design
- **Impact**: Cannot manufacture, redesign needed

17. Soldering Process Incompatibility

- **Problem**: Design not optimized for assembly method
- **Solution**: Design for chosen assembly process (reflow, wave)
- **Impact**: Assembly failures

18. Insufficient Design Rule Checking

- **Problem**: Not verifying against manufacturer capabilities
- **Solution**: Configure proper DRC rules, run checks frequently
- **Impact**: Manufacturing failures

High-Speed and Signal Integrity

19. Improper Impedance Control

- **Problem**: No impedance calculations or control
- **Solution**: Calculate impedance, use controlled stack-up
- **Impact**: Signal reflections, poor integrity

20. Ignoring Return Paths

- **Problem**: Not considering where return currents flow
- **Solution**: Provide continuous return planes
- **Impact**: EMI issues, signal integrity problems

21. 90-Degree Trace Angles

- **Problem**: Using right-angle bends in traces
- **Solution**: Use 45° or curved bends
- **Impact**: Impedance discontinuities, EMI

Prevention Strategies

Design Phase

- **Use DRC Early**: Configure rules before routing
- **Verify Footprints**: Double-check all dimensions
- **Calculate Requirements**: Trace width, impedance, etc.
- **Review Schematics**: ERC, visual inspection

Before Manufacturing

- **Run Full DRC**: Check all rules
- **Review Gerbers**: Visual inspection
- **Get Second Opinion**: Peer review
- **Consult Manufacturer**: Verify capabilities

Tools to Help Avoid Mistakes

- **DRC/ERC**: Automated rule checking
- **Footprint Validators**: Verify component dimensions
- **Impedance Calculators**: Ensure controlled impedance
- **Design Review Checklists**: Systematic verification

References: 21 Common PCB Mistakes, DFM Mistakes, Design Mistakes Guide