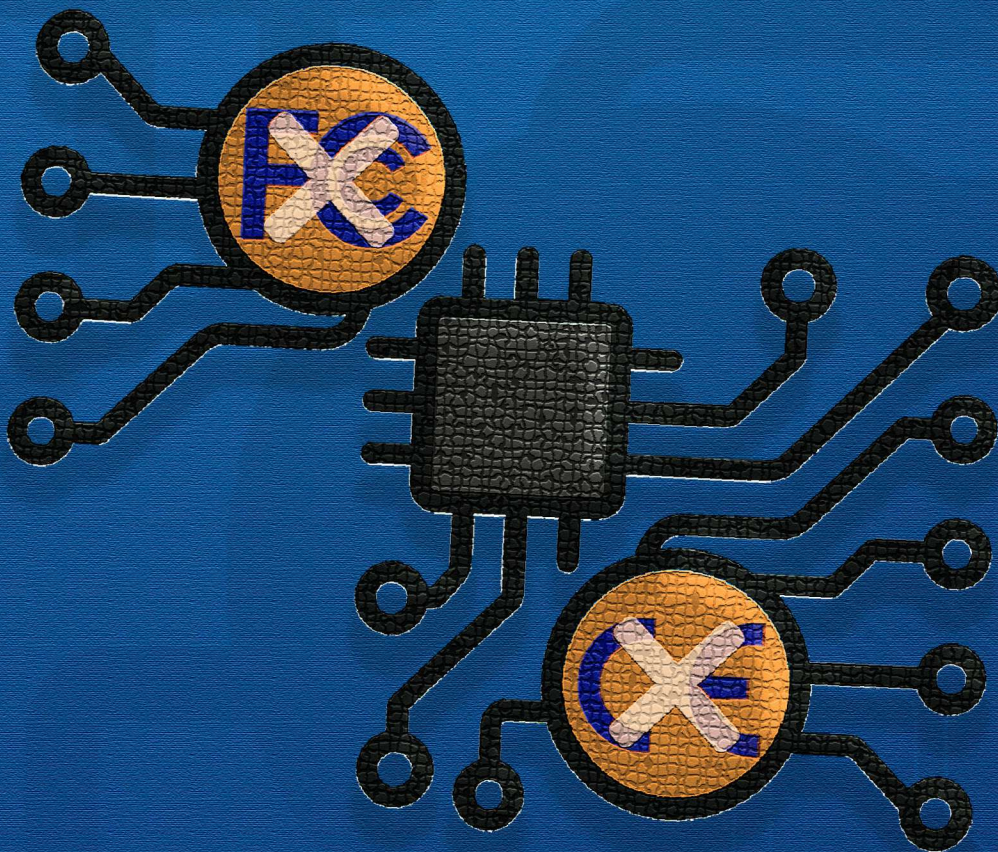


9 PCB DESIGN MISTAKES THAT FAIL FCC/CE CERTIFICATION



MISTAKE #1 Poor Ground Plane Strategy

- ✓ Use a solid, continuous ground plane whenever possible.
- ✓ Avoid splits, cutouts, and thin bridges that break return paths.
- ✓ Prefer 4-layer boards with dedicated ground and power planes for EMI-sensitive designs.
- ✓ Don't split analog and digital grounds on 2-layer boards unless absolutely necessary.
- ✓ Always provide low-inductance return paths for signals.

MISTAKE #2 Large High-Current Loop Areas

- ✓ Keep loop areas as small as possible.
- ✓ Place bypass capacitors close to IC power pins.
- ✓ Keep switch, diode, and capacitor close together in power stages.
- ✓ Avoid routing high-current traces in large loops around the board.

MISTAKE #3 Bad Connector Placement and Routing

- ✓ Treat all cables as potential antennas.
- ✓ Route high-speed signals with continuous ground reference.
- ✓ Avoid long or convoluted signal paths to connectors.
- ✓ Filter noisy signals before they reach I/O connectors.

MISTAKE #4 Unshielded Oscillators and Crystals Near Edges

- ✓ Place oscillators and crystals near the center of the PCB.
- ✓ Surround them with ground copper and add stitching vias.
- ✓ Keep them away from connectors and board edges.

MISTAKE #5 No Shielding or Guard Traces on Noisy Sections

- ✓ Use grounded metal shields on noisy sections like switching nodes or RF circuits.
- ✓ Add guard traces between noisy and sensitive signals.
- ✓ Contain noise at the source before it spreads.

MISTAKE #6 Improper High-Speed Trace Routing

- ✓ Maintain continuous reference planes under high-speed traces.
- ✓ Avoid crossing plane splits with high-speed signals.
- ✓ Route differential pairs symmetrically with consistent spacing.
- ✓ Minimize stubs and vias on critical traces.

MISTAKE #7 Switching Regulator Layout Mistakes

- ✓ Follow the manufacturer's reference layout closely.
- ✓ Keep switching paths short and tightly coupled.
- ✓ Pay special attention to grounding and capacitor placement.
- ✓ Remember switching regulators behave like RF circuits.

MISTAKE #8 Poor Layer Stackup Design

- ✓ Use stackups that tightly couple signal layers with ground.
- ✓ Avoid placing power and signal layers adjacent without a ground plane in between.
- ✓ Consider 4- or 6-layer stackups for better EMI control.
- ✓ Plan stackup early to prevent redesigns later.

MISTAKE #9 No Controlled Impedance for Critical Signals

- ✓ Use impedance-controlled traces for high-speed and RF signals.
- ✓ Define trace widths, spacing, and stackup in CAD tools before layout.
- ✓ Check impedance consistency across the entire routing path.
- ✓ Plan impedance control early, don't leave it for later.

For Entrepreneurs and Startups:

Develop and launch your electronic product **FASTER** without costly mistakes!

Doing it alone is slow and risky. Small missteps now can become big problems later. Skip the mistakes and launch faster.

→ Frustrated things aren't moving as fast as you want?

Feeling unsure what to do next? ←

Meet your guide: John Teel



Hey there, I'm a former microchip design engineer at Texas Instruments and founder of a hardware startup that sold products in hundreds of retail stores. My chip designs are in devices from Apple, Intel, and more.

Now, my full-time focus is helping people like you bring new electronic products to life, without wasting time, money, or risking everything.

**CLICK HERE
TO LEARN MORE!**