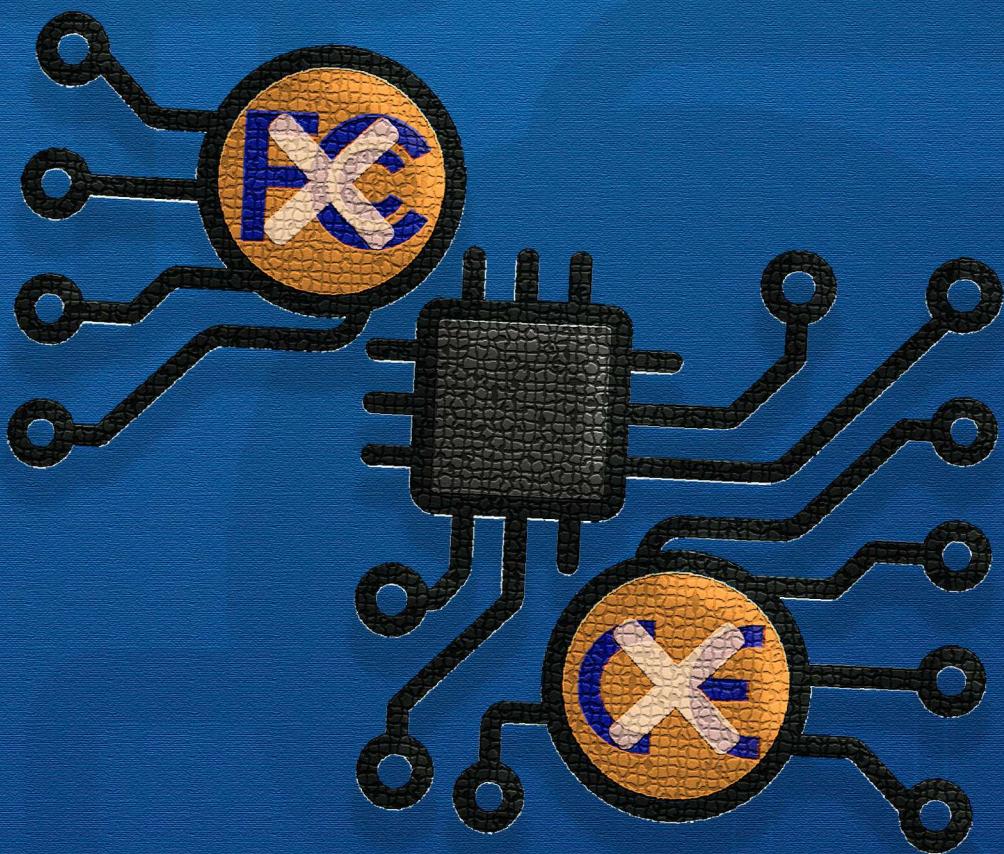


9 PCB DESIGN MISTAKES THAT FAIL FCC/CE CERTIFICATION



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!**