

Avoiding Common Missteps in PCB Design: A Manufacturer's Insights

You design PCBs. Cirexx manufactures them. This article covers seven common PCB design mistakes that manufacturers see and gives you tips on how to sidestep them.

Printed circuit boards (PCBs) are essential components in a broad spectrum of electronic devices, laying the groundwork for complex functionalities in our tech-driven world. However, the journey from PCB design to a fully operational board isn't always smooth. In fact, around 75% of the orders we see at the quoting stage contain errors in design. PCB design can be a complex task with many potential pitfalls, and even minor oversights can disrupt the entire process.

Here, we explore seven common—yet frequently overlooked—mistakes in PCB design, offering a unique perspective from the manufacturer's standpoint. Read on to learn more about the missteps that can significantly affect manufacturability and a PCB's performance and lifespan.

Incomplete Understanding of the End Product's Ambient Environment

One frequent oversight in PCB design entails not accounting for the operating environment. Several factors need to be considered:

- **Ambient temperature:** Like any electronic device, PCBs are sensitive to temperature. Will your device be expected to function in sudden temperature changes, like an outdoor device exposed to the hot sun during the day and freezing temperatures at night? Incorporating thermal-management strategies into the design can ensure the device is able to handle such temperature extremes without malfunctioning or shortening its lifespan.
- **Humidity:** Moisture can be a silent enemy to PCBs, leading to potential issues like oxidation and short-circuiting. Will your product be used in a humid environment or an area prone to rainfall? Can it withstand the condensation that could occur in high-humidity environments or during rapid temperature changes? Account-

ing for humidity during design can enhance the PCB's resilience to moisture-related damage.

- **Physical stress:** Just like a building designed to withstand earthquakes, PCBs may also need to endure various forms of physical stress. Will the device be subjected to heavy vibrations, as in an industrial setting or during transportation? Could it experience shocks, such as a handheld device that might be dropped? Ensuring that your PCB design will tolerate such physical stressors is crucial to its durability.

Holes Drilled Too Close to Copper Features

Designing a PCB often involves a delicate balance. One such aspect that's sometimes overlooked concerns the location of drilled holes. Improperly located drilled holes can result in:

- **Damage to copper features:** Copper features are integral parts of a PCB, carrying signals across the device. When drilled holes are placed too close to these features, there's a risk of potential damage or shorting during manufacturing or operational use. This could lead to signal-integrity issues or even component failure.
- **Compromised structural integrity:** The stability of the PCB may also be jeopardized by improper drilled-hole placement. Holes placed too close to the copper features can weaken the board's overall structure, making it more susceptible to damage from physical stress or handling.
- **Increased production time and costs:** Poorly positioned holes can complicate the manufacturing process. The increased difficulty may result in more time needed for production, causing delays in schedules and elevated costs. It might also contribute to a higher rate of manufacturing errors, potentially leading to wasted materials and additional costs for rework.

Inadequate Annular Ring Specifications

Annular rings, which comprise the area of copper pad around the drilled and finished hole in a PCB, play an important role in securing the component to the board and conducting current from the component pins. Ignoring annular ring requirements may cause:

- *Unreliable connections:* If the annular ring is too small, there may be inadequate surface area for the solder to form a reliable connection between the hole and the component pin, which may affect the board's performance.
- *Increased breakage risk:* An inadequately specified annular ring could also lead to a higher chance of breakage or damage, particularly in high-stress environments or during handling, installation, or operation.
- *Noncompliance with class requirements:* For IPC [Class 2 and stricter Class 3 requirements](#), especially in multilayer, rigid-flex, and flex designs, having an annular ring that falls outside the acceptable range could result in noncompliance. This could induce significant issues during the inspection or even require a redesign.
- *Manufacturing issues:* Insufficient annular rings can also lead to problems during the drilling process, potentially causing drill breakage and increasing manufacturing costs.

Gerber Files and Schematic Drawings Don't Match Up

Discrepancies between your Gerber file and schematic drawing can cause serious issues during manufacturing:

- *Incorrectly placed components:* A key outcome of inconsistent files is the potential misplacement of components. Components form the functional core of a PCB, and their positions significantly impact the PCB's performance. If the Gerber file and the schematic drawing offer conflicting information on component placement, it can lead to errors during assembly, affecting the board's function and potentially causing short circuits or signal interference.
- *Misrouted connections:* The connections between components—traced through the copper lines on the PCB—are the highways for electronic signals. Any discrepancy between the Gerber file and schematic drawing regarding these routes can lead to misrouting. This, in turn, can cause incorrect connections, crosstalk, or even circuit failure.
- *Production delays:* Discrepancies can also create confusion during the manufacturing process, leading to unnecessary delays. The manufacturer might have to halt production to resolve the conflicting information, costing valuable time. Moreover, significant revisions might need to be made to the design, leading to further delays and potential cost escalations.

To prevent such problems, double-check your files for consistency before sending them to your PCB manufacturer.

Poor Communication Between the PCB Provider and Customer

Effective communication between the PCB manufacturer and the customer is crucial to avoid:

- *Misunderstood design requirements:* If the customer's design requirements aren't accurately conveyed or comprehended, it can result in a final product that misses the mark. This could manifest as inadequate performance, component mismatches, or functionality errors.
- *Production delays:* Unclear instructions, conflicting notes on the drawings, or unanswered queries can create bottlenecks in the manufacturing process. Production may need to be paused to resolve queries or clarify instructions, thereby lengthening timelines.
- *Assumption-based errors:* In the absence of clear communication, there might be instances where assumptions are made to fill in the gaps. These assumptions can lead to design or manufacturing errors, causing performance issues, or even necessitating rework.

Clear and prompt communication can streamline the production process and result in a high-quality product that meets the customer's needs.

Stubs in the Design

Stubs, or portions of unused transmission lines, can lead to signal reflection and degradation. Consequences include:

- *Increased signal noise:* Stubs can cause reflections that elevate signal noise. This can degrade the overall signal quality, which might affect the PCB's functionality and performance.
- *Signal interference:* The signal reflection caused by stubs can lead to interference with other signals on the PCB. This may cause crosstalk or other signal-integrity issues.
- *Reduced performance:* The overall effect of stubs can lead to reduced performance of the PCB, causing slower operation or even failure of the board.

Avoiding stubs in the design can significantly enhance signal quality and integrity.

Incorrect Trace and Space vs. Copper Weight

An improper balance between trace, space, and copper weight can lead to various problems:

- *Inadequate power delivery:* A mismatch in the trace-space-copper weight equation can lead to inadequate power delivery, affecting the performance of components and the overall PCB.
- *Overheating risk:* If the copper weight is too high relative to the trace and space, it can lead to excessive current flow, which could cause overheating.

- *Signal-integrity issues:* The improper balance can also cause crosstalk, leading to signal-integrity issues, thereby affecting the quality and reliability of the PCB.

Careful planning and adherence to PCB design rules helps maintain the correct balance and ensures optimal PCB performance.

Inexperienced PCB Designer

Designing a PCB requires a thorough understanding of the manufacturing process. A designer unfamiliar with this can lead to:

- *Manufacturing incompatibilities:* A design might look perfect on paper, but it could be difficult or even impossible to manufacture. This, in turn, may result in a significant waste of time and resources.
- *Increased production time and costs:* Designs that don't consider manufacturing constraints can lead to complex, time-consuming manufacturing processes, thereby escalating production time and costs.
- *Suboptimal designs:* Without the knowledge of manufacturing, a designer might not fully optimize the plan. This could foster a PCB that doesn't meet performance expectations or isn't cost-effective.

Gaining a thorough understanding of the [PCB design process and its key steps](#) is crucial for creating optimal designs with improved performance and reliability. Ensure your PCB designer is experienced and knowledgeable about the manufacturing process to build practical, efficient designs.

Final Thoughts: Leveraging the Lessons Learned

Recognizing and avoiding these common PCB design missteps can drastically improve your design process's efficiency and the final product's quality. A keen eye for detail, thorough understanding, and consistent communication are your best tools against these pitfalls.

Steering clear of these prevalent PCB design pitfalls is more than just a preventive measure—it's an investment in your design process's effectiveness and the eventual quality of your product. The significance of these errors extends beyond isolated issues, contributing to the larger narrative of technological progress and innovation.

Ultimately, the path to proficient PCB design is paved with knowledge, precision, and collaboration. As designers and engineers, your capacity to learn from past mistakes, adapt, and innovate will continue to shape the landscape of electronic design. You have the power to engineer high-quality, efficient, and innovative PCBs.



Don Kaufman, business development manager at Cirexx International, has more than 33 years of technical experience in the printed-circuit-board industry. He possesses extensive knowledge of integrated circuits, materials, specialty laminates, chemicals, and equipment. He also prides himself on staying up-to-date with the latest technologies and procedures in PCB layout and manufacturing.