



PCB Design For Manufacturing Guidelines

Avoiding a premium for custom PCB is a primary concern, and understandably so. The overall manufacturing cost of PCBs encompasses various factors. Apart from fixed expenses like logistics, labor, and equipment, fluctuations are primarily driven by factors such as board dimensions, material selection, and process technology. These variables hinge upon the decisions made by engineers during the design phase.

To ensure seamless manufacturability of the circuit board design, minimize production expenses, enhance manufacturing efficiency, and uphold the ultimate product quality, the concept of **Design for Manufacturability** emerged as a pivotal force in the PCB domain. In this comprehensive discourse presented by FS Technology, we will delve profoundly into this subject, equipping you to augment project profitability. Let's commence this exploration.

What is Design for Manufacturing and What is it for?

Design for Manufacturing, an acronym for DFM, is an approach used to streamline the manufacturing process of circuit boards. Its main goal is to optimize dimensions, materials, tolerances, and functionality by employing the most efficient manufacturing methods available. This process should start in the early stages of conception, even before sketching begins. It's important for PCB design for manufacturing to align with the device concept's inception, with a primary focus on understanding how the customer will use it. Investing significant time, dedication, and effort into developing a robust DFM process will yield substantial benefits in later phases of PCB design.

Optimizing Component Costs

As the cornerstone of any circuit board, component costs are of significant concern. By partnering with FS Technology, a [PCBA company](#) with an autonomous engineering team, leveraging **DMFA** (Design for Manufacturing and Assembly) facilitates the identification of optimal component alternatives, simplifying design and manufacturing while curbing [component procurement](#) expenses.

Design Upgrade

Connecting with manufacturers, including printed circuit board manufacturers and casing suppliers, or seeking advice from industrial designers can prove highly effective. Continuously asking questions like ‘Can this be improved?’ and discussing how your electronic or PCB designers have addressed similar issues in previous designs can offer valuable insights. When initiating the DFM process, make an effort to anticipate potential challenges, establish a design sequence, outline potential obstacles at each juncture, and then explore similar products. Analyze how similar issues were resolved in these cases and strive to implement analogous solutions.

Fostering Collaboration

PCB design for manufacturability involves defining responsibilities for both manufacturers and customers. This allows manufacturers to address and resolve design-related issues, preventing flawed board production due to factors like delivery schedules and reducing conflicts between manufacturers and customers. To achieve optimal functionality, it’s crucial to engage all stakeholders in the design for manufacturability process. This includes electronic engineers, PCB designers, industrial designers, PCBA manufacturers, mold makers, and material suppliers. DFM check PCB requires a ‘cross-functional’ approach, which means having a comprehensive understanding of the design process at different levels: components, functional blocks, systems, and assembly. This approach ensures that the design is created without incurring unnecessary costs. Collaboration across departments, bringing together different segments to collectively discuss PCB production, guarantees timely identification and resolution of defects before major issues arise.

Cost-Efficient

Implementing an early **PCB DFM rule** significantly reduces expenses associated with requested alterations during the design phase. As the design evolves throughout the product development cycle, making changes becomes progressively more expensive and challenging due to their increasing complexity. DFM helps engineers identify materials that strike a balance between optimal performance and cost-effectiveness, thus avoiding unnecessary costs. One of DFM’s key features is integrating designs that maximize the utilization of raw materials while enhancing the product’s aesthetic intricacies. Importantly, DFM contributes to optimizing the manufacturing process, allowing for the re-evaluation of previous projects and the elimination of unnecessary steps in the production pipeline.

Restoration Design

Enlisting the collaboration of stakeholders in the product development process at the outset is simpler when developing a new product. Nevertheless, even when contemplating a subsequent iteration of an existing market product, a comprehensive PCB design for manufacturing checklist remains indispensable. Regrettably, design

errors often recur during replication of a previous design. Scrutinize and question every facet of your design during the DFM process.

PCB Design For Manufacturing Checklist

Usage Cycle

Thoroughly analyzing how users will interact with the product is crucial. This involves meticulous planning of the [PCB assembly process](#) to ensure efficient assembly steps and optimal tool utilization. Deciding on the appropriate soldering method (such as surface mount technology or through-hole soldering), determining the [optimal component placement orientation](#), refining routing, and ensuring both assembly speed and quality are all essential considerations. Incorporating automated equipment can enhance assembly speed, and strategic layout optimization can reduce assembly errors.

When building prototypes, there's no need to use materials with aerospace-level requirements or opt for fabrication with high environmental certifications. Similarly, complex board shapes aren't necessary if you're producing small quantities. Unless the design explicitly demands it, as it involves creating tools and dies for manufacturing low-volume parts, which increases the cost of [PCB prototyping](#).

It's important to consider factors like the quantity of parts to be manufactured, the choice of materials, required finishes, tolerances, and the need for secondary processes.

Design

The design phase forms the backbone of the product ideation process, requiring consideration of all necessary conditions. This stage encompasses electronic circuit design, [PCB layout](#), connector placement, and collaboration with industrial designers to strategically position indicators, control buttons, connectors, and cables associated with the printed circuit board. Components should not be oversized in either dimensions or electronic capability. Proper component selection should adhere to electronic design standards by choosing components capable of withstanding at least twice the capacity they will be used for.

Tracks should be appropriately sized to handle the expected current load. Signal tracks should be compact to avoid unnecessary space consumption. Precise dimensional tolerances should be defined.

For boards intended for automated assembly by pick-and-place machines, fiducials should be added to expedite the manufacturing process. Consult with your manufacturer to determine if tooling holes are necessary. It's essential to collaborate with your [contract manufacturer](#) to ensure your design aligns with sound manufacturing principles for PCBA fabrication.

Materials

Here you define if the material requires special conditions, this could include considerations for heavier weight copper (commonly 1oz or 2oz), PCB substrate material thickness (e.g., aluminum or [FR4 material](#)), and overall board thickness, which varies from 0.4 mm to 2 mm depending on the application. For instance, RF designs might necessitate thinner PCBs. FS Technology strongly recommends referring to resources like the [PCB copper thickness guide](#) and related documents during the developmental phase. The PCB DFM guidelines examine various material aspects:

How strong should the material be?

Should the [solder mask](#) have a specific color? The silkscreen generally needs to contrast with the mask.

How heat-resistant does it need to be?

Will the board carry significant current? Will there be high voltage? Track thickness should be considered for current calculations, and gap spacing for voltage calculations.

How flame-resistant should the material be?

What thickness is required? What substrate material is best? Is FR4 suitable, or is better heat dissipation needed, possibly using aluminum?

Once again, ensure you discuss the material with your [PCB manufacturer](#), exploring their compatible inventory materials, which can help secure lower material costs.

Environment

Your product's design must account for the environment it will be exposed to. Will the board be used in a harsh environment with vibrations, high temperatures, sun exposure, high humidity, or flammable atmospheres? This involves addressing potential issues like dust, humidity, and corrosion. Additionally, it's crucial to assess whether the circuit requires features like high-frequency operation or miniaturization.

The manufacturer's environmental certifications, including [ROHS certification](#) and electromagnetic interference resistance certification, should be thoroughly evaluated. Industries with specific attributes should also verify manufacturer qualifications, such as [ISO13485](#) (medical) or [IATF16949](#) (automotive) certifications. [Qualification Certification Download](#)

Compliance/Testing and Calibration:

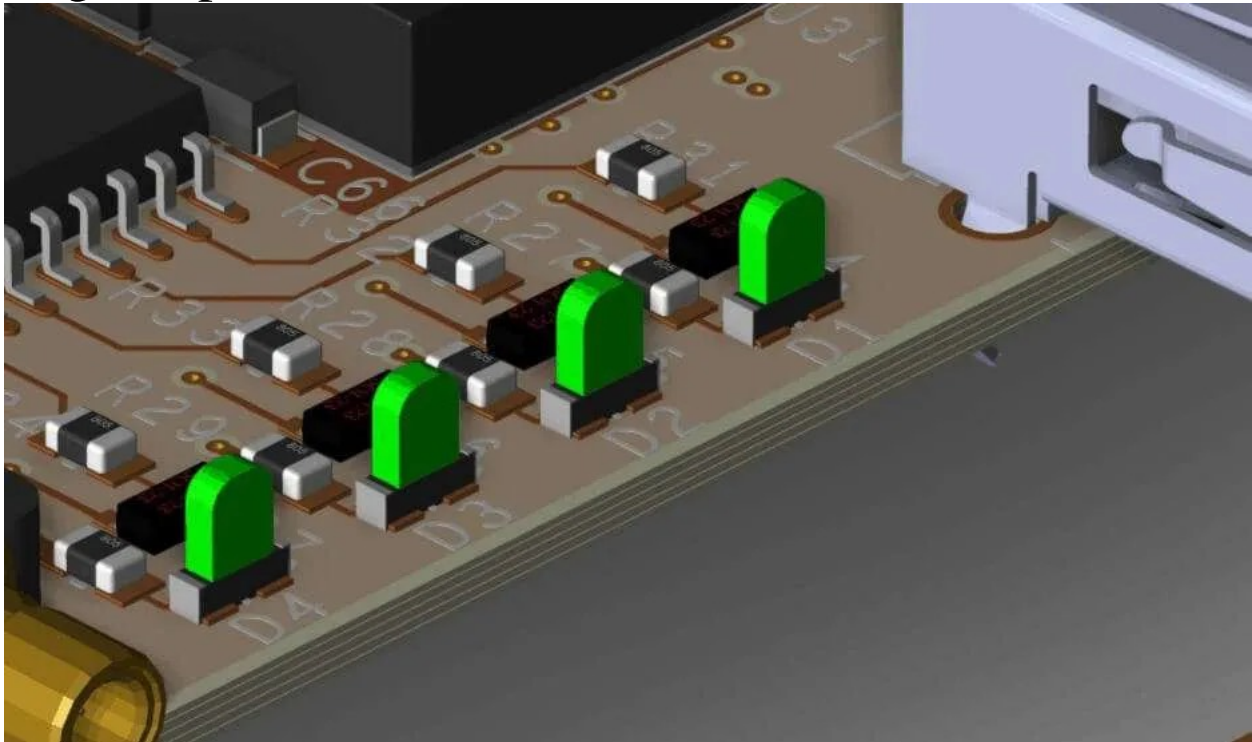
Defining the battery of tests or analyses that the printed circuit board will undergo before assembly is crucial. This is especially important for larger boards with numerous components or high manufacturing costs, as these [tests ensure that no electronic](#) flaws are overlooked. These tests can align with laboratory PCB tests aimed at securing certifications like the ones mentioned earlier.

With the exception of some prototypes produced in low quantities, all products must meet safety and quality standards. These standards can be [IPC PCB standards](#), regional standards, or internal standards specific to the company or to you as a customer.

Do you require any ISO certification? Who will provide UL testing, ETL, and others? Who will conduct and where will such testing take place?

Design Flaws for DFM Checks

Edge Gap

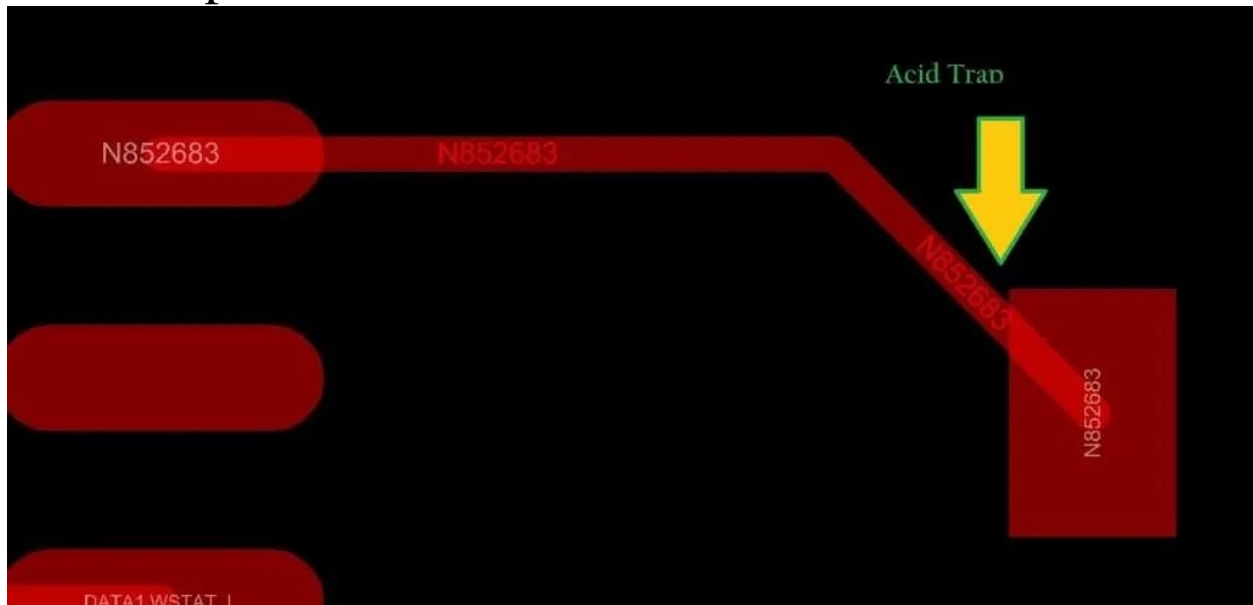


Pertains to the proximity of the PCB board's perimeter, encompassing the region adjacent to its outer edge. A prudent design mandates the allocation of a reserved portion within this space. Regrettably, designers frequently underestimate this consideration, leading to potential issues involving protective coatings during the PCB cutting process. This oversight may result in the removal of these coatings, rendering the copper layer vulnerable to corrosion due to inadequate protection. Nonetheless, this predicament can be rectified by incorporating the requisite edge clearance into the design.

To mitigate the aforementioned concerns, it is recommended to incorporate additional dimensions for protective coating on both the outer and inner layers of the PCB. For outer layers, a supplementary 0.010 inches of coating area is advised, while for inner layers, a greater allowance of 0.015 inches for coating area is suggested. Incorporating these specific edge gap considerations ensures that the PCB design remains resilient

against potential coating removal issues, safeguarding the integrity of the copper layer and minimizing the risk of corrosion.

Acid Trap



To resolve the PCB acid trap problem, it's important to understand wiring layout and the underlying cause of acid trap formation. Circuit boards develop traces through an etching process. During this process, acidic solutions or etchants can accumulate at trace corners, hindering smooth flow during immersion. This can lead to areas where the liquid becomes stagnant, creating what are known as 'acid traps.' Inadequate cleaning of residual acid or solution in these traps can result in issues after washing, such as over-etching, leading to open circuits or other damage. With modern manufacturing techniques advancing and demands for thinner traces increasing, the impact of acid trap problems has become more pronounced. To mitigate this issue and ensure the effectiveness of technology, consider the following design measures:

- Avoid acute angles in trace corners, aiming for angles of at least 90 degrees, as sharp angles are often responsible for acid trap formation.
- When traces need to be bent or connected to other areas, consider implementing chamfers or bevels to prevent acid traps.
- Utilize DFM tools to identify potential acid trap concerns and receive recommendations for improving your design.

Absence of Solder Mask or Overlay



In the process of circuit design, a crucial layer comes into play: the solder mask or overlay. Its role in preventing unwanted contact between pads and other metallic components cannot be overstated. The omission of this protective layer poses a significant risk, potentially leading to inadvertent contact between pads and resulting in the peril of circuit board short-circuit incidents. To avert such complications, integrating the protective layer into design protocols and weaving it into the production workflow becomes paramount. This strategic approach ensures that, whether dealing with expansive or compact circuit boards, the incorporation of the safeguarding layer remains steadfast, thereby mitigating potential setbacks and losses.

Optimizing Via Placement



In scenarios where spatial constraints are at play, a strategic solution involves the judicious addition of vias to liberate valuable room. Vias serve as conduits for establishing connections or pathways across diverse layers of the PCB. Yet, a word of caution is in order, as this approach could inadvertently lead to solder being drawn away from the board, thus jeopardizing the PCB's mountability and undermining soldering efficacy. The realm of [PCB vias](#) offers an array of options, necessitating a deliberate assessment of necessity and an astute choice of the most fitting type. The improper selection and excessive deployment of vias can prove counterproductive, eroding the overall board performance and reliability.

The spectrum of vias encompasses varied options, including microvias, blind vias, and [buried vias](#). The integration of these vias merits careful consideration, factoring in the unique exigencies of your design. While these advanced via types hold utility in specific contexts, a discerning evaluation of their applicability is indispensable to align vias with the design's requisites effectively.

Factors Affecting Design for Manufacturing

The objective of [PCBA DFM](#) is to [reduce manufacturing costs](#) without compromising performance. In addition to the points mentioned above, there are other factors that can impact design and assembly.

- **Reduce Parts:** This is a straightforward way to cut PCBA manufacturing costs. Using fewer materials reduces the need for engineer intervention, shortens production processes, lowers labor requirements, and even potentially reduces shipping costs.
- **Standardize Elements:** Custom parts take time and money. It's better to standardize PCB board dimensions and shapes that can be used across different devices, or design boards

that can serve multiple functions based on the installed components.

- **Implement Modular Assemblies:** Building upon commercial modules and non-customized designs allows for smoother and more cost-effective product modifications.
- **Easy to Connect:** Minimize the need for numerous **PCB connectors**, which can increase costs. If required, opt for commercial connectors. Use standard connectors whenever possible, employ self-tapping screws for quicker placement, and avoid overly long or thick screws, split washers, threaded holes, special heads, and those needing specific tools.
- **Connector Alternatives:** While the above points offer optimization solutions for connectors, it's recommended to consider alternatives such as **rigid flex PCB**. Though more expensive in manufacturing, they provide improved features and reduce overall project costs.

Tom Doslak Tech Driven EMS | 408 424 5804 \ Tomd@Techdrivenems.com

