TTM Technologies
Time-To-Market Interconnect Solutions™

5 Essential PCB Design Best Practices for Cost Savings

Julie Ellis
Field Applications Engineer

Glenn Ames
Director of Technology, TTM Logan (LG)

- A Publication of TTM Technologies, Inc. -

TABLE OF CONTENTS

A Letter from the Business Unit President

Look like a design genius by avoiding common pitfalls.

This eBook is the result of our experience building hundreds-of-thousands of printed circuit boards (PCBs) from prototype to volume production.

You will learn 5 best practices that will save your PCB project time and money, creating an effective, high-reliability PCB design.

The authors of this e-Book, Julie Ellis and Glenn Ames, are just two of our 200 + engineers devoted to our customer's success; Julie has been extensively involved in electric vehicles, autonomous driving, on-board advanced safety--including Lidar, 77GHz products, cameras, and Human-Machine Interface applications for our automotive customers. Her focus is on the seamless transition from prototype to volume production.

Glenn Ames is the Director of Technology at our Logan, Utah facility; Glenn has been the point man for hundreds of customers on thousands of jobs, helping improve PCB design for manufacturing efficiency, helping make products come to life for our customers.

We invite you to apply these best practices to your next PCB build

We hope you find the enclosed information useful; we invite you to apply these best practices to your next PCB build and reach out to us with the result. Did you get a higher yield? Did your costs shrink? Was it faster to go from prototype to volume production? Visit the last page of this eBook for more information on booking a consultation with one of our PCB engineering experts.

Good luck to you and we look forward to assisting you to meet your customers' needs.

Yours truly,

Jon Pereira

President - Automotive, Medical, Industrial & Instrumentation (AMI&I) Business Unit

CHAPTER ONE

Best Practice #1: Consider the Whole

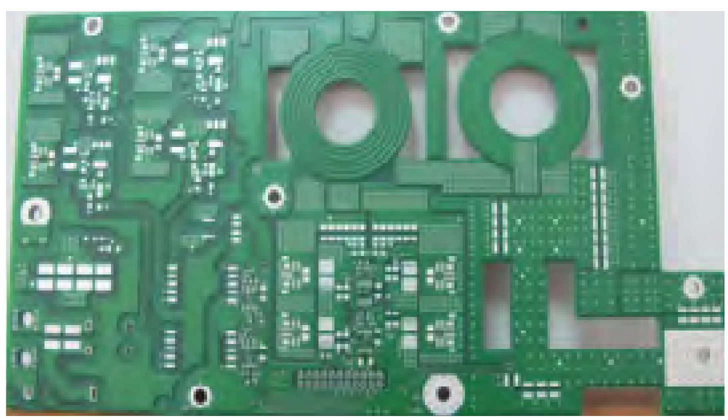
Consider the Whole

Design for Excellence by Considering the Entire Product.

Design for excellence--that's the goal. Build a high-reliability board that will stand up to the wear and tear of the manufacturing process and then, ultimately, the application that it will bring to life. Engage your whole team early, including your procurement team; your quality team; your hardware designers; your fabrication site's field applications engineers; and your process engineering team.

360 Degree Design: We recommended looking at the big picture when designing your PCB; context is everything when you begin the design process.

PCB-related design inputs can be outside the board; they are not just part of the PCB itself--they include environmental factors where it will function; size and space constraints of the enclosure; heat profiles of the components; and the longevity of the product. Factors within the board include layer count; overall thickness; copper thickness; the dimensions of the board; the arrays in which they will be assembled; hole sizes; line spaces; and materials.



4-Layer, 6 oz PCB: TTM Zhongshan (ZS) is dedicated to the Automotive industry. This is an example of a heavy copper board produced at that site.

Designing your board holistically, considering the path from prototype to volume production, considering the assembly processes, and application. Here are a few questions to get you started thinking about your design:

- How is this going to go down a pick-and-place assembly line? Will this require rails?
- How will assembly considerations put panel utilization at risk?
- Are there any overhanging parts? What will happen to them during assembly?
- Will reliability be compromized during any manufacturing, assembly, or use of the application?
- Can this be read by a vision system?
- Are there thermal management considerations?

> *GOOD DESIGN DOESN'T COST, BUT IT PAYS.*
>
> **Richard Driehaus**

Designing for excellence means considering the unit as a whole from the outset of your design process. Approach each design from fabrication, assembly, and test perspectives and involve your entire team at each of these stages. While this approach may seem time-consuming, it will ultimately lead to a better design and long-term cost savings.

CHAPTER TWO

Best Practice #2: Keep It Simple

TTM Technologies

Keep It Simple

Simplified designs ensure cost savings and improve reliability.

Design your PCB as simply as possible to meet your requirements. Keeping processes as simple as possible not only promises better yields, but is an effective way to ensure cost savings and, in some cases, improve reliability. Once you have determined the most complex aspect of your design, simplify wherever possible. Making other features as large as possible will get higher yields--making complex choices across the board, just because one feature has that requirement, will cost money and time.

> *A DESIGNER KNOWS THEY HAVE ACHIEVED PERFECTION NOT WHEN THERE IS NOTHING LEFT TO ADD, BUT WHEN THERE IS NOTHING LEFT TO TAKE AWAY.*
>
> **ANTOINE DE SAINT-EXUPERY**

Proven track record with emerging technologies:
TTM is the automotive industry's trusted high-reliability PCB supplier. We are engaged from prototype to volume for e-mobility solutions, including 48V powertrain, electric & hybrid vehicles, charging stations, and battery test equipment.

Confirm the most complex design attributes, using the checklist below:

Checklist: PCB Design Attributes

- ☐ 1. Top Level Specification (IPC Class, MIL standard, etc.)
- ☐ 2. Overall dimensions (L X W X T)
- ☐ 3. Layer count
- ☐ 4. Laminate material type (High frequency, high-speed material, etc.)
- ☐ 5. Copper layer thickness on plated and unplated layers (know max voltage if high voltage)
- ☐ 6. Surface finish
- ☐ 7. Soldermask colour
- ☐ 8. Legend Ink colour
- ☐ 9. Drill chart (smallest via and pad critical)
  - a. Incude blind, buried, microvias, and back-drills
- ☐ 10. Smallest linewidth and spaces
- ☐ 11. Stack-up
  - a. Number of lamination cycles
  - b. Controlled impedance requiremest
  - c. Via-In-Pad adds copper wrap plating to outer layers(increases required etch lines/spaces)
- ☐ 12. Via protection (VIPPO, Soldermask plugs, or tents, UV plugs)
  - a. For example we can't put SM plugs on top of tin or immersion silver; the baking process would discolour those surface-finishes.

CHAPTER THREE

Best Practice #3: Know Where You're Going

Know Where You're Going

Fabricator Capabilities and Design Should Be Cohesive; Not at Odds.

Once you have determined the main design attributes, there are some other things you should consider to determine the right fabrication site with the right technology capabilities, including heavy copper for high-voltage or high-current applications, radio frequencies with specific surface and plating requirements, or small features and high-density interconnect (HDI) requiring laser-drilled microvias. These three different technologies may need to go to three different sites because of the equipment and expertise required for these builds. We explore heavy copper for high-voltage and RF requirements in the following sections.

High Voltage or High-Current

We need to look at the copper thickness parameters to consider if thicker copper plating is required in the holes than what is standard for a normal class 2 or class 3 board. For example, a standard class 2 board requires .787 mil copper plating in the hole walls, whereas a class 3 board requires 1 mil of copper plating. But sometimes for heavy copper and also a lot of press-fit pin connector manufacturers require a minimum of 1.18 mil in the holes. Most fabricators can easily accommodate 1.18 mil or 1.2

Spotlight:

TTM Toronto (TOR)

Key Capabilities

- **Up to 42 layers**
- **0.010" – 0.250" thick**
- **IPC 6012 Class 2 and 3**
- **HDI / Microvia**
- **20:1 Aspect Ratio**
- **I/L & O/L LWS 0.002"/ 0.002"**
- **Via in Pad & Via fill**
- **Max Panel size 21" x 24"**
- **Buried & Blind vias**

Surface Finishes

HASL ∘ ENEPIG ∘ ENIG
Immersion Silver & Tin
Hard Gold ∘ OSP

mil in-hole, but you need to check their process capabilities to see if they can go higher than that in copper thickness in the hole wall. A lot of other high volume PCB shops have their lines set on a timer, so it's very difficult to adjust the time and leave these parts longer in the tank when the lines are continuously conveying multiple panels and multiple part numbers.

Ask your supplier if they can do more than 1.2 mil copper-plating on the walls. One of the things people do for high voltage or high power boards that get hot is design thermal vias under the component to sink heat through the hole wall copper to the opposite side of the board. One way of helping transfer that heat is to increase the copper thickness in those holes.

When we discuss those thermal vias, we want to make sure we don't violate minimum drill hole walls and drill hole wall distances. There are other issues at play; when we drill holes very close together the drills can heat up the material and the drill bits can drag the glass fibres inside the cloth and cause other problems inside the cloth. We don't want holes to be too close together. Our normal minimum design guideline is 16 mil for same-net circuits and larger (depending on material) for uncommon nets.

We also need to determine if those thermal vias are under a big component and which side needs to be soldered to the body of the component and whether those via holes need to be protected via fill or plug material to prevent solder from flowing down the hole and starving the pad of solder. This is a key consideration during the PCB design stage, even though it affects the assembly process.

Are any other thermal management solutions, such as metal core PCBs,

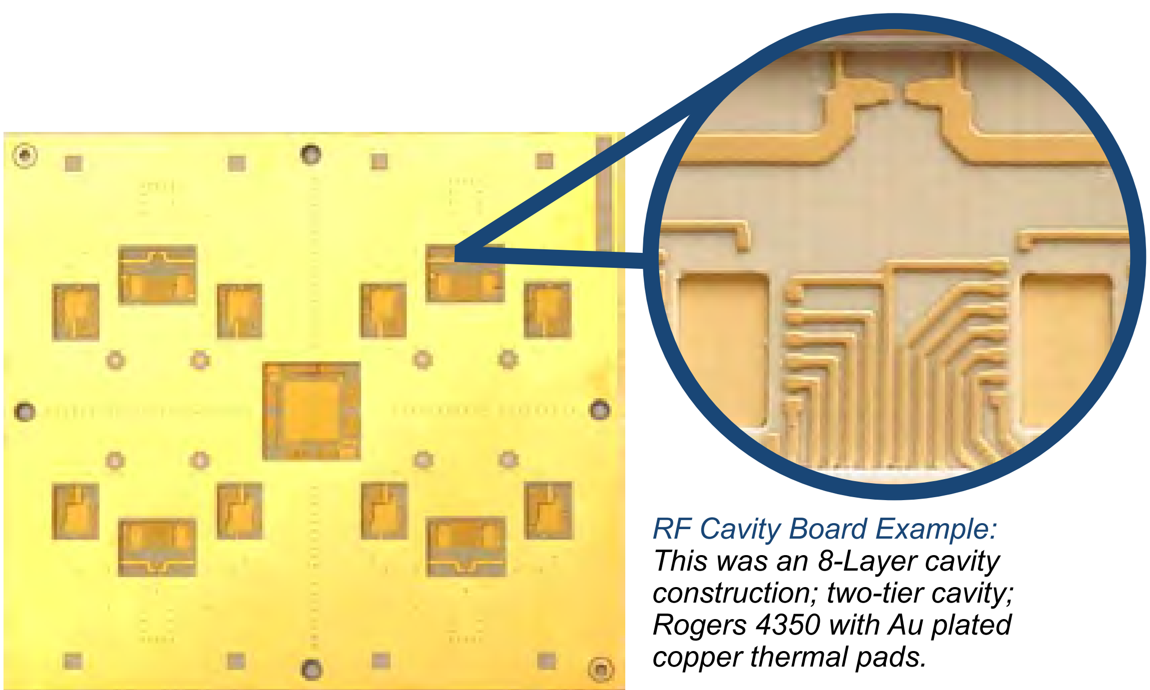
> *A GOOD DESIGNER MAY NOT HAVE ALL THE ANSWERS, BUT THEY KNOW WHICH QUESTIONS TO ASK.*
>
> **Rudy Duke**

coins, plugs, or other things like that which could be useful in the design? Lastly, for our automotive customers, you need to consider whether conductive anodic filament (CAF) is a concern. If so, the stack-up needs to be created keeping the reduction of CAF opportunities in-mind and you must also consider CAF design rules for minimum distances between hole walls.

High Frequency, RF Applications

Another family of products is the high-frequency, RF applications group. These require special materials and because the antennas are so critical with regard to dimensioning, you have very tight etching tolerances while defining the traces which may require additional process adjustments.

If you have an RF antenna, ranging from 24 – 77 GHz for automotive radar, you are looking at a different type of material and the stack-up needs to be calculated accordingly. In these cases electroless nickel immersion gold, or ENIG finish, which is often used for fine pitch BGA assemblies because it solders so well may be tempting. However, it is important to consider that the gold is processed with phosphorous and phosphorus's high magnetic properties slow high-speed signals down. When we see these types of antennas we usually recommend silver or immersion silver or immersion tin finishes, occasionally ENEPIG.



RF Cavity Board Example:
This was an 8-Layer cavity construction; two-tier cavity; Rogers 4350 with Au plated copper thermal pads.

Seamless Prototype to Volume Manufacturing

Transitioning from a North American prototype to volume in Asia Pacific can be challenging. The best approach is to design with volume in mind at the prototype stage. Using common tooling, materials, and process definitions that can be shared between your prototype shop and your volume fabricator will benefit your project in the following ways:

1. Automotive customers can prototype and build to design validation in North America
2. Faster development cycle
3. No need to redesign because of material electrical parameters
4. No need for multiple qualifications

If your transfer from prototype to volume is not seamless, reliability and efficiency may be at stake: With 25 global manufacturing and assembly sites, it just made sense for TTM to develop a process for our customers to get to market more efficiently and streamline the transition from North American prototype to Asia Pacific volume.

CHAPTER FOUR

Best Practice #4: Room to Move

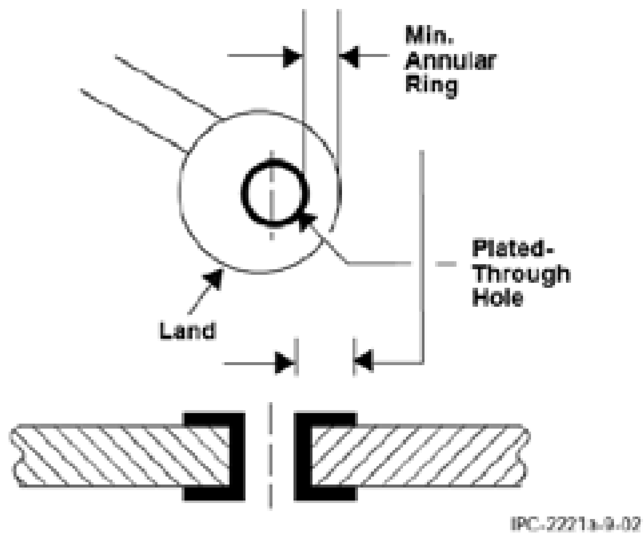
TTM Technologies

Room to Move

Allow for Material Shifts Within Your Design.

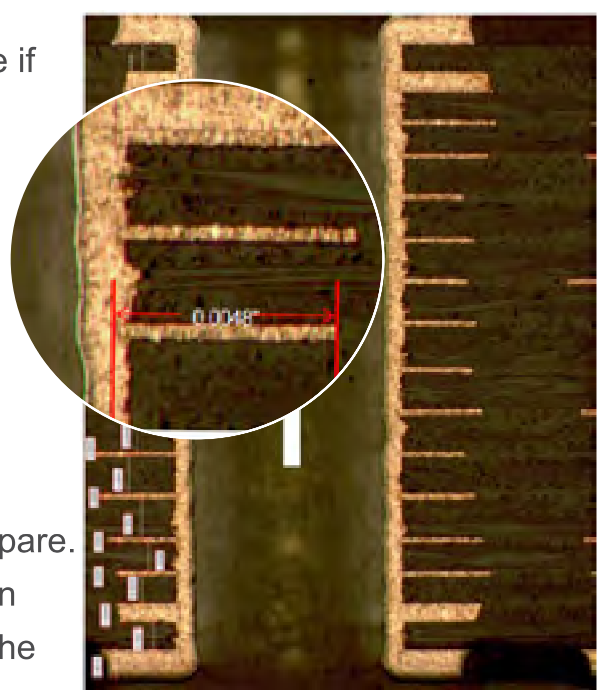
One of the common design issues fabricators are challenged by is insufficient spacing. Many designs could do with more space and the yields would be higher as a result. One of the most-common areas where space is crucial to success is annular ring.

Annular Ring



*External Annular Ring: Measurements **do** include through-hole plating.*

Annular ring is the pad around the drilled hole in the circuit board. We talk about annular ring dimensions two different ways: over the drill size, for example if you have a 10 mil drilled hole and a 20 mil pad, then you have 10 mil over the drill size. We could also say it's a 5 mil annular ring, meaning there should be 5 mil per side. IPC or customer specifications will dictate the annular ring that is required. However, in order to account for the shifts in material during manufacturing processes, give your fabricator as much annular ring as you can spare. To meet the minimum, you will need to design your board with a greater tolerance to meet the desired outcome.



Internal Annular Ring: Measured from drilled-edge to etched-edge of foil.

CHAPTER FIVE

Best Practice #5: Make the Most of It

TTM Technologies

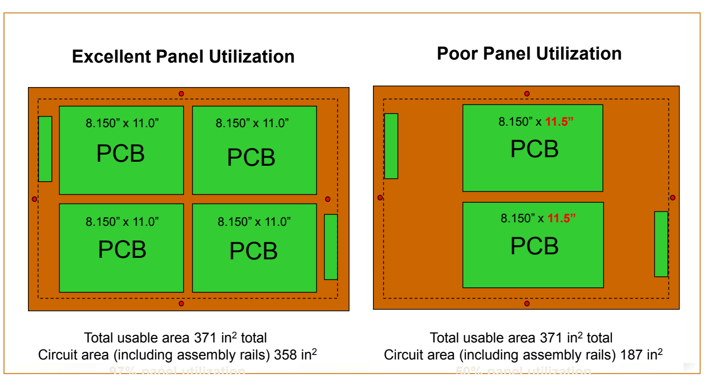
Make the Most of It

Creating an array that maximizes panel utilization saves mega bucks.

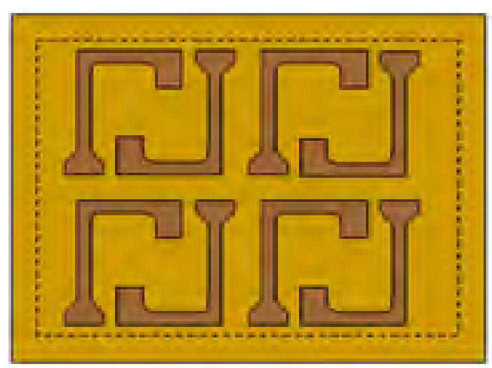
PCB fabricators manufacture panels, not individual boards. A standard panel is 18" x 24" all vendors have this size capability. However, there are other sizes and many vendors will have options for you.

Considering panel utilization in the very early phases of product design, not at PCB layout, fabrication, or assembly stages--your assembler requires keep-out rails for conveyance of PCBs on their assembly lines--is critical to ensuring you get the most out of your materials costs. Work with your assembler and fabricator to plan single PCBs with rails, or multiple-up "arrays" with rails to ensure you you can fit as many assembly-ready arrays on a production panel as possible.

You pay for the entire panel, regardless of the wasted space. Get the most of each panel by optimizing your layout! If the design allows, you may also consider nesting your circuits to optimize even further.



Cost Savings Made Easy: Panel utilization is just one way you and your team can save costs. Working with your team to ensure the best array for manufacture and cost savings is critical to effective PCB design.



Nested Panels: If these were mocked up side-by-side, the yield would be six parts; nesting has increased the number of parts by two.