



PCB FABRICATION NOTES GUIDE

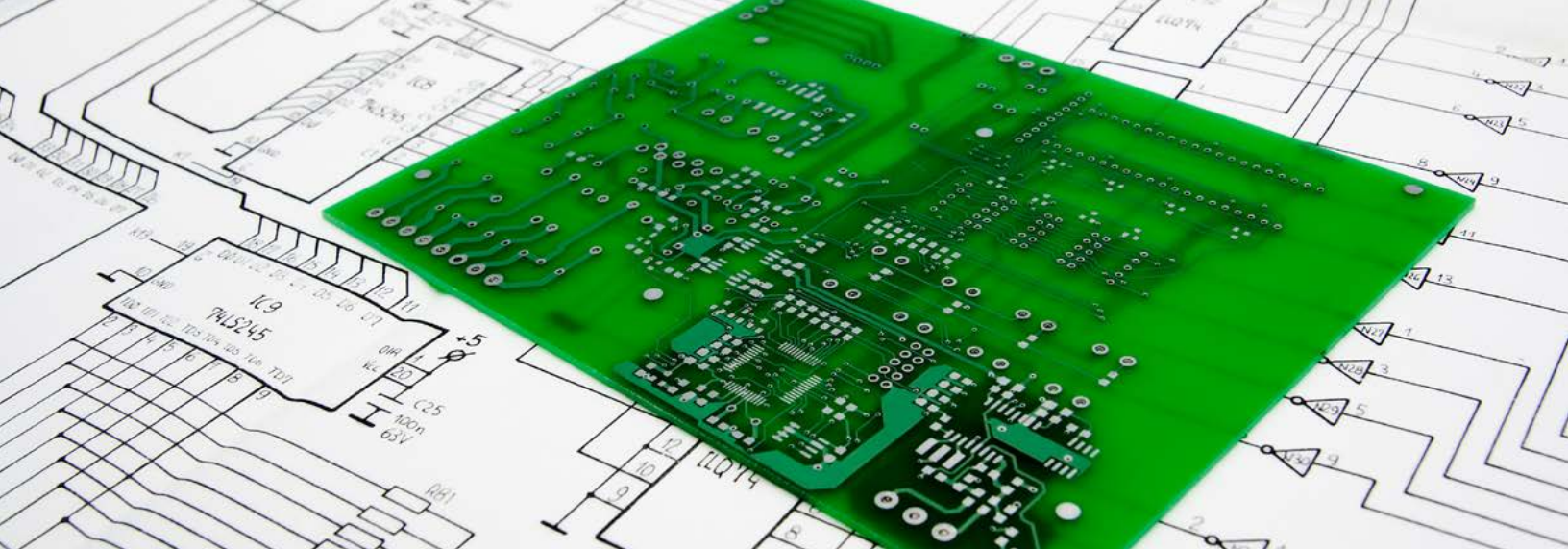


PCB Fabrication Notes Guide

When reviewing quotes for your printed circuit board (PCB) fabrication, remember that a lower price may not always represent the best option. Although it can be tempting to purchase based on budget alone, a thorough review of each of your quotes may reveal subpar materials and processes that do not meet your requirements or industry standards.

When the manufactured PCB is inconsistent with your expectations, you risk halting production while changes to the project are discussed, negotiated, and implemented into the design. This can result in much higher production costs as materials, processes, and components are added. Not only does it cost more in materials and labor, an insufficient quote will extend your PCB manufacturing time indefinitely while the fabrication company tries to work out the discrepancies from the information they originally received.

This is where detailed fabrication notes will help. When submitting a request for a quote (RFQ) to potential PCB manufacturers, you should include detailed diagrams and fabrication notes. With this information, a good PCB manufacturer will be able to provide you with an accurate and timely quotation that accounts for all relevant specifications and industry requirements. You can then compare bids from each PCB manufacturer with your own detailed specifications to ensure that you are getting exactly what you need.



What Are Fabrication Notes for PCB Designs?

To fabricate quality products, PCB manufacturers must be able to thoroughly understand the customer's requirements and specifications for each design. To this end, fabrication notes for PCB manufacturers have become a standard in the industry. When you order a circuit board, you should provide a diagram of the board along with notes that outline the construction guidelines of the PCB, including layers, circuits, connections, dimensions, drill sizes, and other details critical to the PCB's operation.

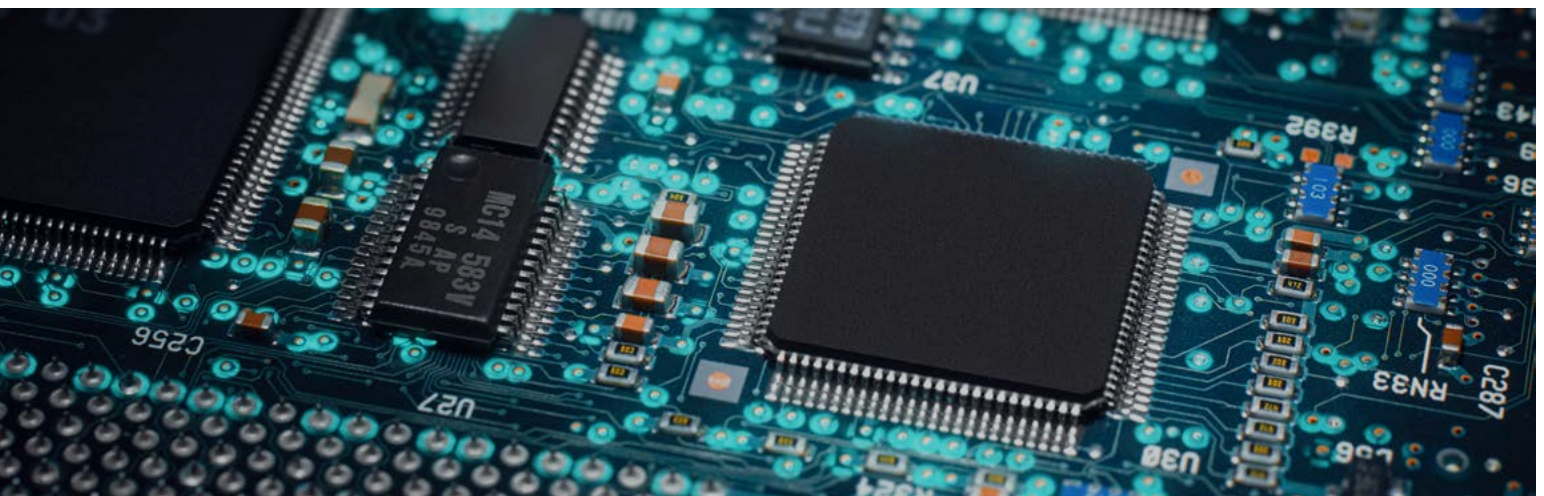
Be sure to clearly indicate all manufacturing requirements with a high degree of detail when you submit your request. Unfortunately, PCB manufacturers often receive insufficient fabrication notes, which results in incorrect or incomplete quotations. Since the manufacturer bases their pricing and delivery information on the customer request, insufficient detail can result in delayed production and inconvenient pricing adjustments after the quote has been accepted and the work has begun. In order to save yourself and your manufacturer time and money, it is easiest to submit detailed PCB assembly notes specific to your PCB design each time you order.

Defining Good Fabrication Notes for PCBs

When looking for guidelines for your PCB fabrication notes, the **Institute for Interconnecting and Packaging Electronic Circuits (IPC)** establishes reliable international standards and ground rules upon which to base your specifications. IPC-6012, for example, outlines the standards for rigid circuit boards, and IPC-6013 provides guidance for flexible circuits.

Good fabrication notes should include specifications for the following design and fabrication factors:

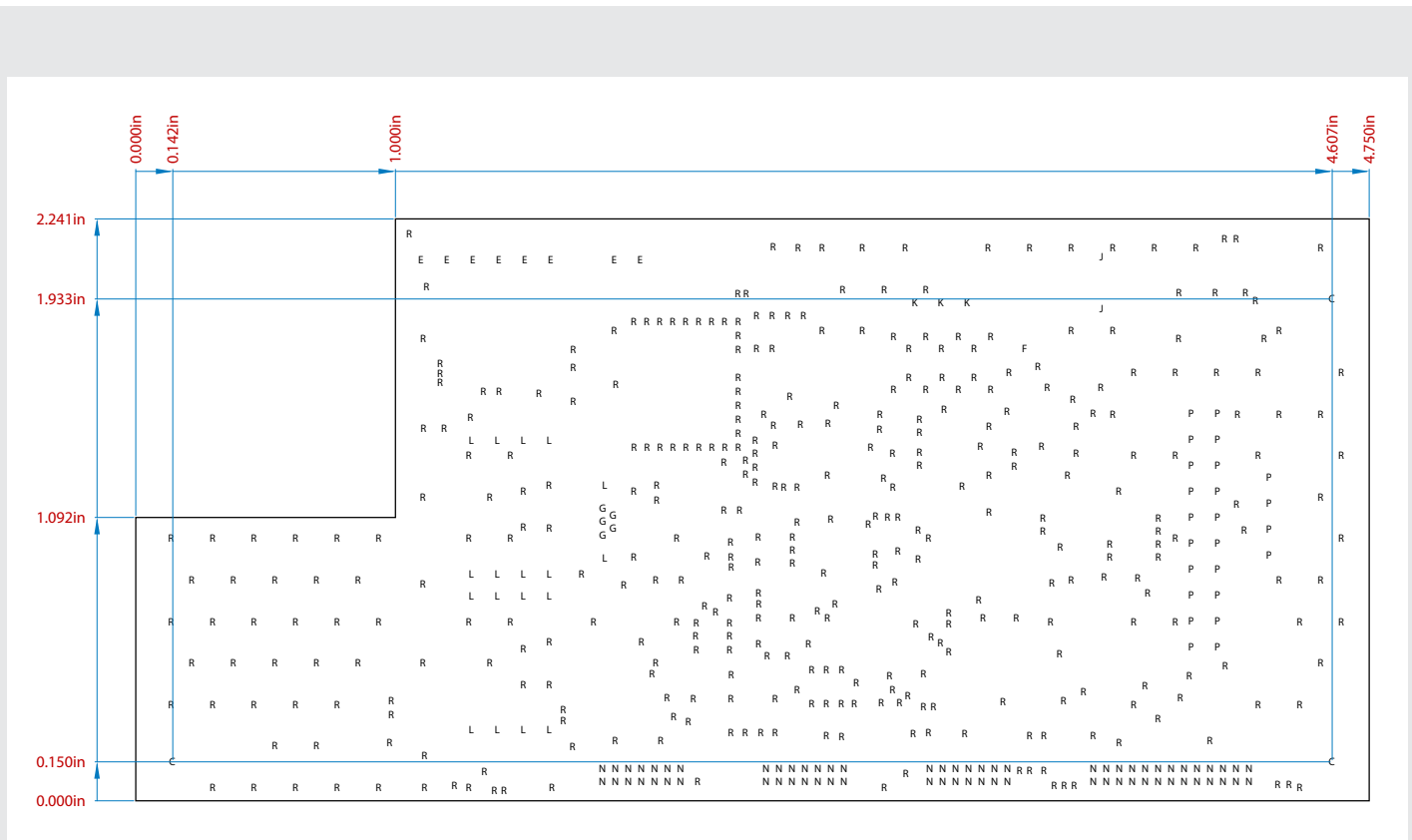
- Outline of board
- Standards
- Material
- Flatness
- Etch geometry
- Surface finish
- Stackup and impedance specifications/tolerances
- Holes
- Solder mask
- Silkscreen
- Drill chart
- Micro-sectioning and testing requirements
- X-outs
- Packaging requirements



The overall purpose of your fabrication notes is to **enhance Design for Manufacturability for optimal PCB fabrication and assembly** while **maintaining applicable industry standards and regulations**. Ideally, good fabrication notes will help your PCB manufacturer create your PCB to highly detailed specifications. As a general best practice, ensure that every note on your diagram clarifies something unique about the board. One of the fundamental definitions of your PCB should be its IPC class.

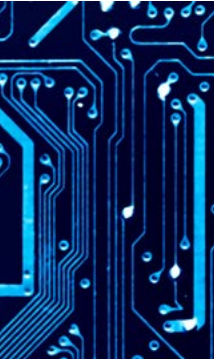
Outline

Having a board outline is critical to an efficient quote. The outline should identify the dimensions and locations of tooling holes, fiducials, and typically, a drill map. Here is an example:

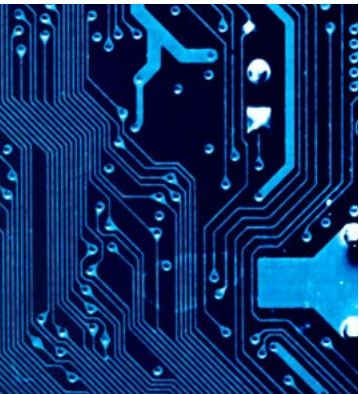


IPC Classes for Electronics and PCBs

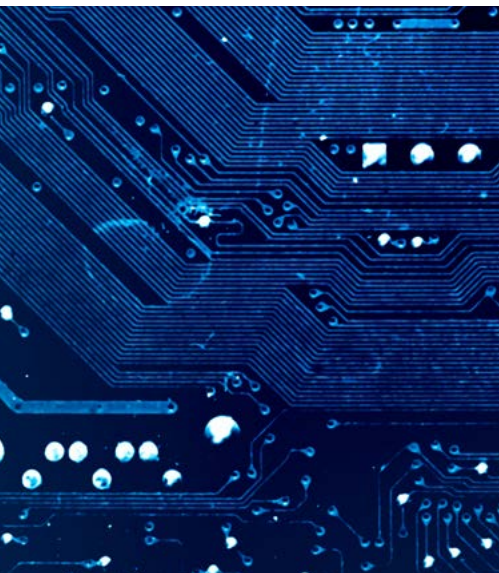
To assist PCB manufacturers with meeting standards for quality and functionality, the IPC developed a series of Classes—categories that define PCB requirements based on intended use. IPC Classes are ranked from Class 1, which is of the least concern, to Class 3 which has the most stringent requirements.



- **Class 1.** Class 1 PCBs are built to the lowest standards, and are typically for general, short-term use. Since they are not held to a high standard, Class 1 electronics tend to be used in products such as children’s toys, in which failure after a period of use is expected and accepted.



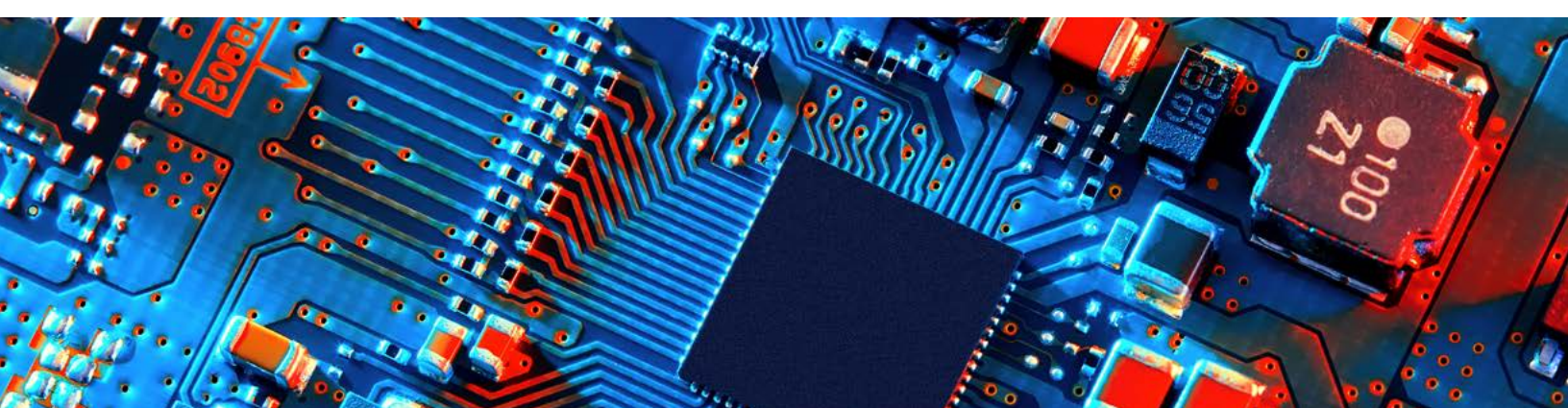
- **Class 2.** Class 2 PCBs are held to a higher standard than Class 1. They are expected to function reliably within their operating parameters for an extended period. They are used in products that have a longer service life, such as computers and televisions, but are not subject to the same quality or lifetime expectations as Class 3 electronics.



- **Class 3.** Class 3 PCBs are expected to meet the highest standards for quality and reliability. This class is reserved for critical electronics that must operate without interruption. You will find Class 3 electronics in medical equipment, aerospace applications, and military equipment where no-fail equipment is a must. Class 3 PCBs are subject to much more stringent testing requirements to ensure reliable operation.

You Get What You Pay For

While it can be tempting to select the least expensive PCB for your project, it is important to keep the IPC class designations in mind. With PCBs, you definitely get what you pay for, and **while a Class 1 design may seem ideal from a budget perspective, it could cost you in the long run.** Not only are you at risk of product failure, you may find that the type of product your PCB is meant to operate has more stringent requirements.

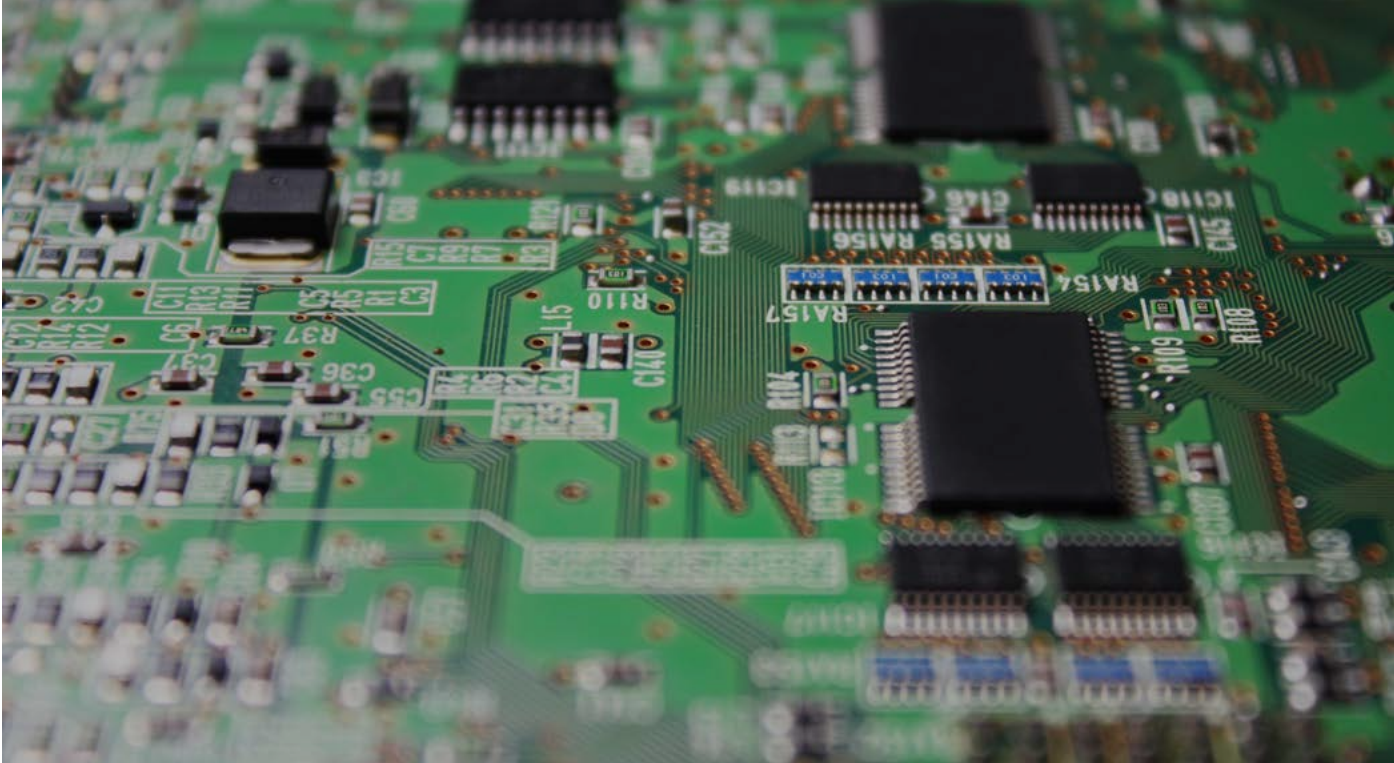


Material Specification Callout

You'll want to specify the material requirements for the board you have designed. You will get a quicker, more robust response if you use an IPC material classification instead of a brand name or use a brand name as an identifier with an "or equivalent" note. This will give you a better quote response rate from manufacturers, as well as a more accurate quote from each manufacturer called out in the material spec.

Here is an example of an appropriately specific material specification callout:

- Base Laminate shall meet or exceed requirements specified by IPC-4101/23. Material will meet min. 170Tg, UL 94V-0 flammability rating (use FR406 material or equivalent).



Identify Tolerances

Use the standard IPC tolerance call-out, or if preferred, note tolerances for each critical area. These notes are important so that your quote and the manufacturing is delivered accurately.

A standard tolerance call-out will look like this:

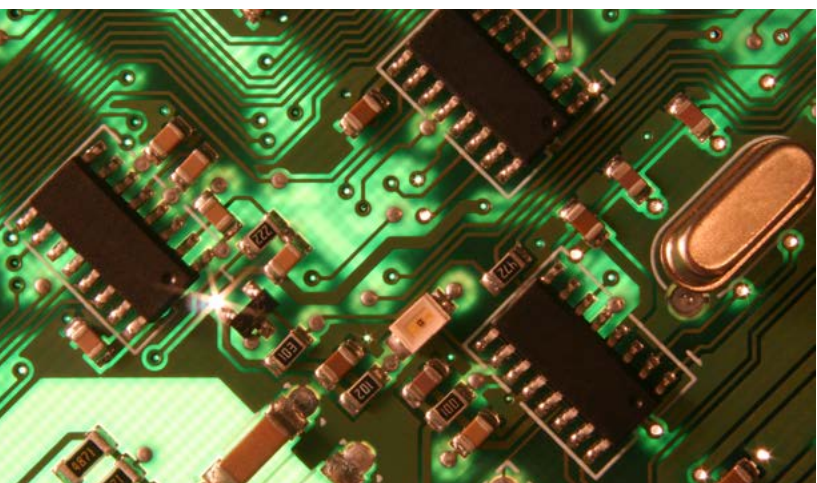
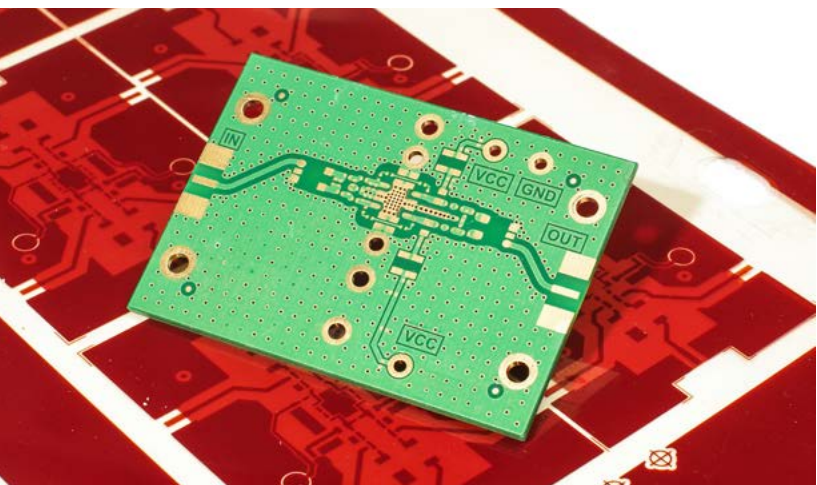
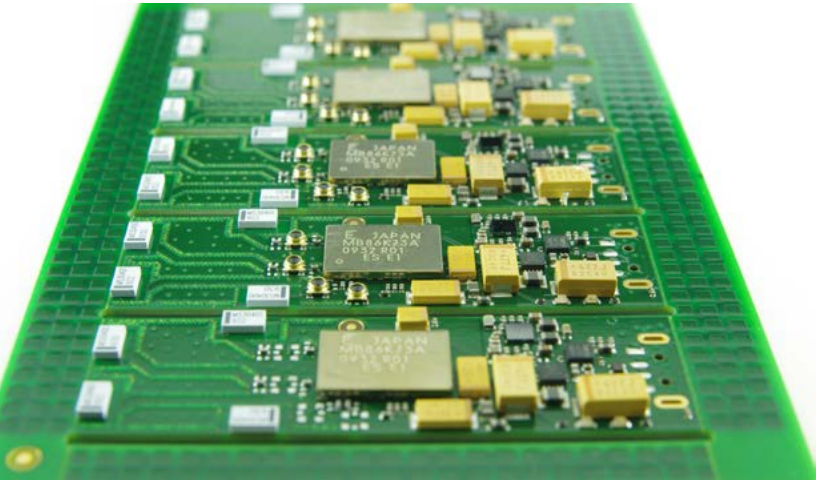
- Board to be built to IPC-Class 2 tolerances

Or you can identify each important tolerance area as below:

- Warp or twist shall not exceed 1% based on IPC-TM-650 2.4.22 standards
- Hole location shall not exceed 0.003" from drill data location
- Board outline tolerance is +/- 0.005"
- Finished line width shall not deviate from the 1-to-1 master pattern image by more than +/- 0.025 mm or 20%, whichever is less

Finish Features

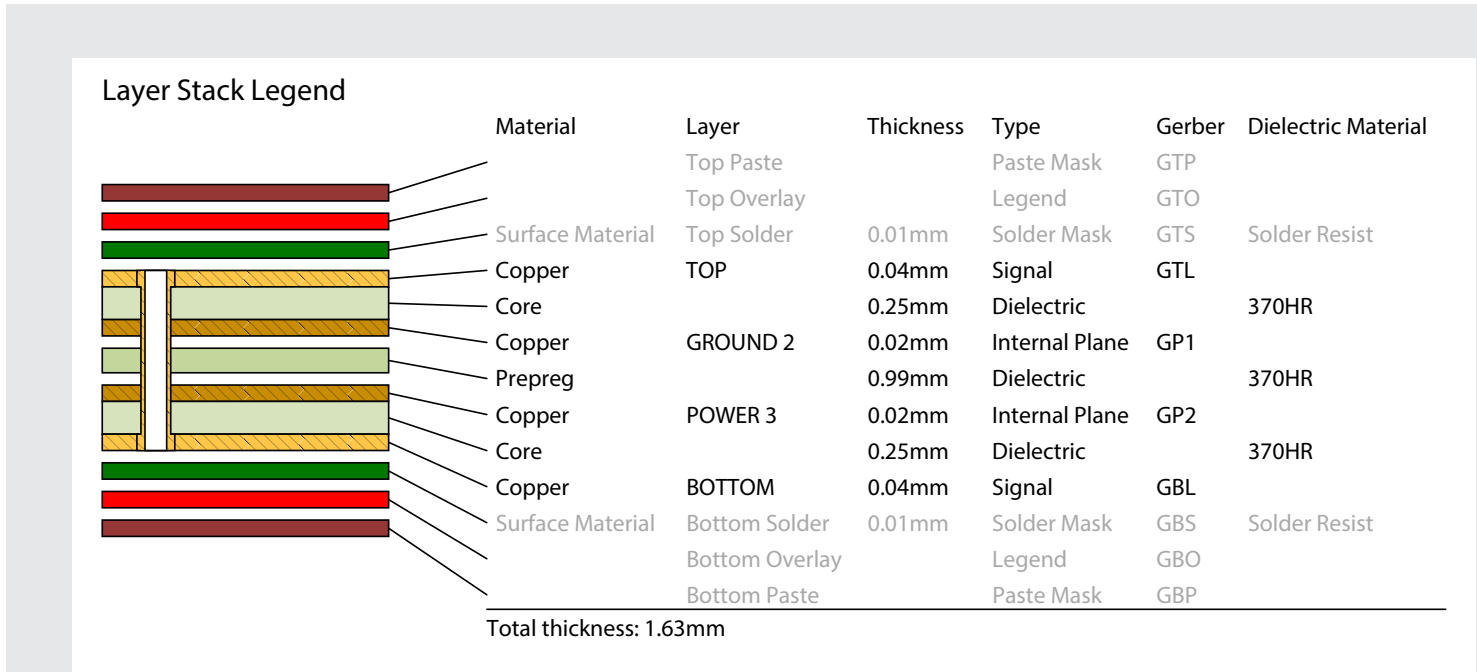
To ensure the finish features are delivered as you expect, make sure to call out at least the finish plating material, soldermask color, silkscreen color:



- **Finish:** Plate with electroless nickel immersion gold (ENIG), Per IPC 4552.
- **Solder Mask:** Apply liquid photo imageable solder mask (color red). Per IPC-M-840, to both sides of the board over bare copper.
- **Legend/Vendor Markings:** Silkscreen with white, permanent, non-conductive ink. Silkscreen must be trimmed or moved to avoid solderable component pads. Silkscreen must be legible and text may be a latered as required. Vendor identification, UL markings (94V-0 or better) and date code shall be located in ink, as indicated on page one of the assembly drawing.)

Identify Stackup and Impedance Requirements

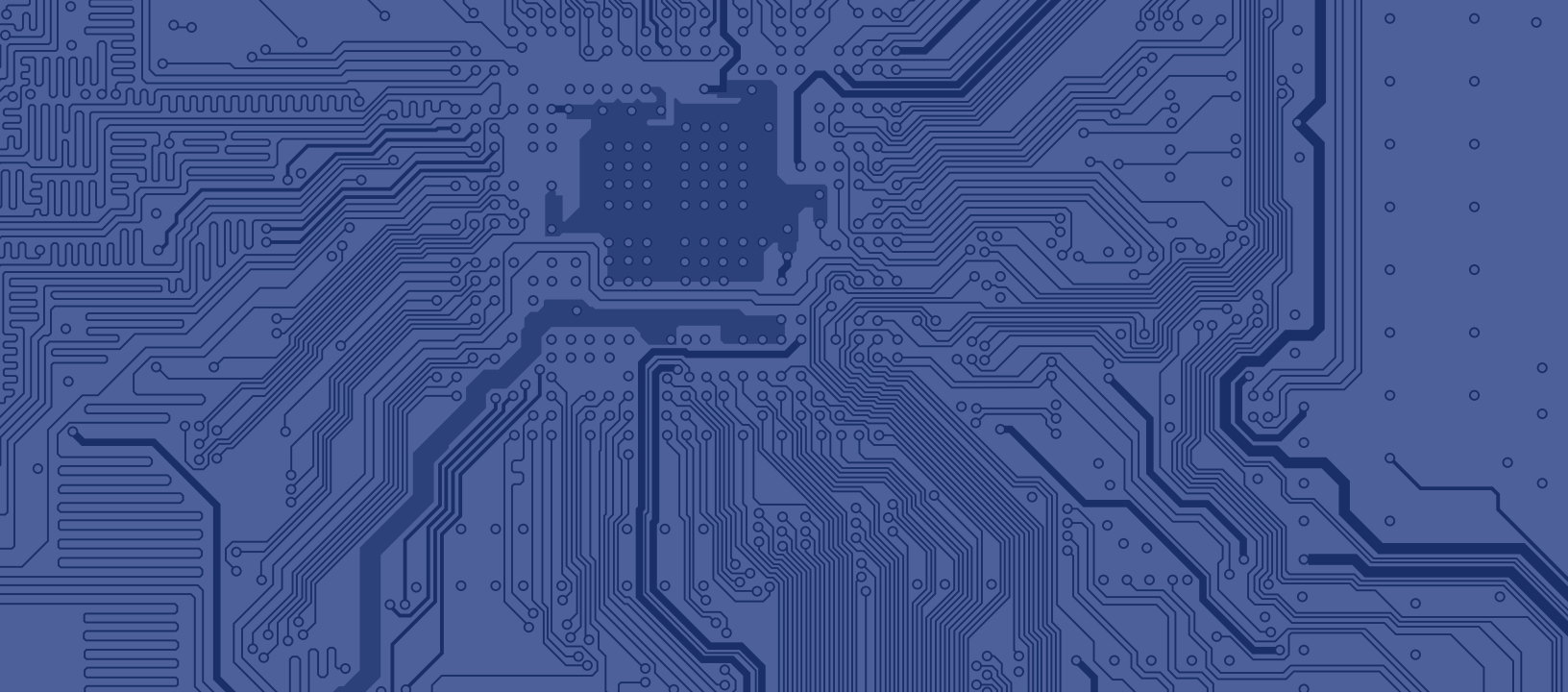
If there are multiple layers with impedance requirements, it is most efficient to include a stackup chart and note impedance requirements. For example:



See drill chart for finished hole size and tolerance.

Table "1" - Impedance Control Requirements

| Layer | Matched To | Line Width/ Gap | Impedance | Tolerance | Type |
|---------|------------|--------------------|-----------|-----------|--------------------|
| Layer 1 | Layer 3 | 18.0/10.0 | 50 OHMS | +/-10% | Coplanar Waveguide |
| Layer 1 | Layer 2 | 5.0 | 50 OHMS | +/-10% | Microstrip |
| Layer 1 | Layer 2 | 5.0/8.0 | 90 OHMS | +/-10% | Diff. Microstrip |
| Layer 1 | Layer 2 | 4.0/8.0 | 100 OHMS | +/-10% | Diff. Microstrip |
| Layer 3 | Layer 2, 5 | 5.0 | 50 OHMS | +/-10% | Stripline |
| Layer 3 | Layer 2, 5 | 5.0/8.0 | 90 OHMS | +/-10% | Diff. Stripline |
| Layer 4 | Layer 2, 5 | 5.0 | 50 OHMS | +/-10% | Stripline |
| Layer 4 | Layer 2, 5 | 4.0/8.0 | 100 OHMS | +/-10% | Diff. Stripline |
| Layer 6 | Layer 5 | 5.0 | 50 OHMS | +/-10% | Microstrip |
| Layer 6 | Layer 5 | 4.0/8.0 | 100 OHMS | +/-10% | Diff. Microstrip |

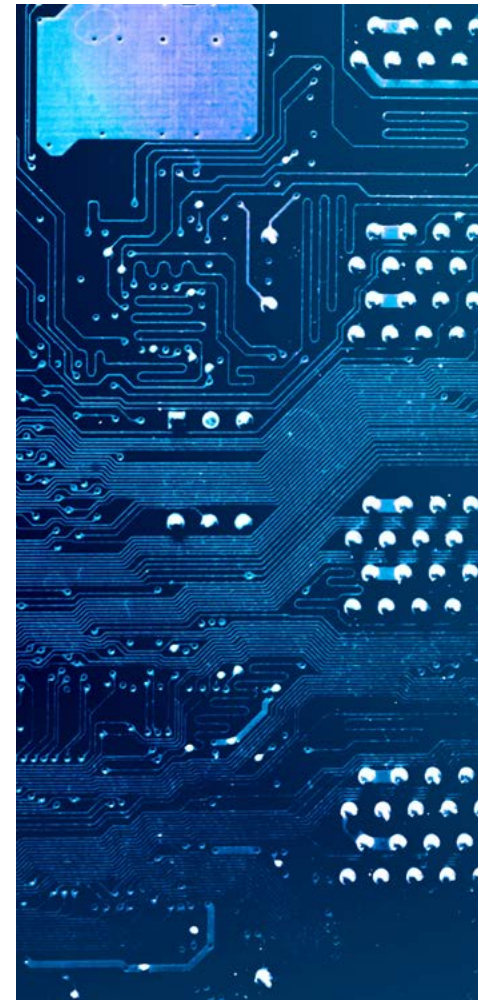


Micro-sectioning and Testing Requirements

Notes for required tests are very helpful since these can vary drastically depending on the need of the end customer. It may be as simple as calling for 100% electrical test or may require specific optical testing or Hi-Pot testing. Also, include a note if a micro-section and testing results are required with the shipment.

X-outs and Packaging

If you have an X-out maximum or packaging notations (# of boards per vacuum pack, etc.) you can also note these in the Fabrication Notes to help the manufacturer determine what yield they will need to meet when quoting quantities.





Working with RedSTAR Worldwide to Source Your PCBs

In PCB fabrication, details are critical to ensuring the accurate construction of the PCB. Vague fabrication notes can cause miscommunication regarding the board design, resulting in incorrect pricing and manufacturing time estimates. In order to avoid complications, it is important to have a clear understanding of what fabrication notes are, and what type of information should be included in these notes.

It is equally important to choose a PCB supplier who has extensive experience with meeting customer requirements and industry standards. A quality PCB provider will have a thorough understanding of various industry requirements and will be able to ask the right questions to ensure that they have all the information they need to provide an accurate quote and create a quality design.

Our attention to detail and personalized service is what sets RedSTAR apart. We have the knowledge and expertise to ensure a smooth ordering process, from initial design to delivery. Our seasoned engineers, technicians, and supply specialists work around the clock to handle any issues as they arise. For more information on RedSTAR's PCB supply and engineering services, **contact our experts today or request a quote.**

1827 S, Fremont Dr, Salt Lake City, UT84104

435-645-9270

www.redstarworldwide.com

sales@redstarworldwide.com

RFQ

Contact Us

