THE PRINTED CIRCUIT DESIGNER’S GUIDE TO™

Design for Manufacturing (DFM)

David Marrakchi
Altium
This book has been technically reviewed by the following expert in the electronics industry:

Happy Holden

Happy Holden is the retired director of electronics and innovations for Gentex Corp. He is the former PCB technologist and CTO for Foxconn Advanced Technology of Taiwan and China. Happy also served as senior PCB technologist for Mentor Graphics’ System Design Division after holding senior consultant positions at TechLead, Merix and Westwood Associates.

In 1998, Happy retired from Hewlett-Packard after over 28 years where he managed the application organizations in Taiwan and Hong Kong. At HP, Happy also held positions in PCB manufacturing, software marketing and packaging Research and Development.

He holds a bachelor’s degree from Oregon State University in chemical engineering and studied for a master’s in computer science. He co-edited the new 7th Edition of Coomb’s *Printed Circuit Handbook* and authored I-Connect007’s *The HDI Handbook*. 


The Art of Quickturn

A quickturn time can only be as quick as the information provided and the technology you’re working with. If you wish to master the art of quickturn you must first learn how your PCB requirements impact the process steps, so make sure the information you provide is complete and consistent. Things like multiple laminations, via-in-pad and solder mask tenting are just a few examples of things that could potentially slow down the process and put your job on hold.

Ultimately, partnering with your PCB manufacturer during the design phase will eliminate delays and allow you to get things right the first time. Altium’s design guide is a wonderful launch point for any designer in the early stages of a project. The information in this book will help you avoid problems before they become problems.

— Amit Bahl
Sierra Circuits
## SECTION 1: Design Guidelines for Successful Manufacturing

<table>
<thead>
<tr>
<th>PAGE</th>
<th>CONTENT</th>
</tr>
</thead>
</table>
| 5    | Chapter 1  
      | A Brief Overview of the PCB Manufacturing Process |
| 8    | Chapter 2  
      | Selecting Your Materials |
| 12   | Chapter 3  
      | Strategizing Your PCB Layout |
| 18   | Chapter 4  
      | Placing and Orienting Your Components |
| 20   | Chapter 5  
      | Configuring Your Test Point Requirements |

## SECTION 2: Documentation Guidelines for Successful Fabrication and Assembly

<table>
<thead>
<tr>
<th>PAGE</th>
<th>CONTENT</th>
</tr>
</thead>
</table>
| 25   | Chapter 6  
      | Documenting Your PCB for Fabrication |
| 32   | Chapter 7  
      | Documenting Your Master Drawing |
| 46   | Chapter 8  
      | Documenting Your PCB for Assembly |

<table>
<thead>
<tr>
<th>PAGE</th>
<th>CONTENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>Conclusion</td>
</tr>
<tr>
<td>52</td>
<td>Glossary</td>
</tr>
<tr>
<td>56</td>
<td>Works Cited</td>
</tr>
<tr>
<td>58</td>
<td>About the Author</td>
</tr>
</tbody>
</table>
Design for Manufacturing

David Marrakchi
Senior Technical Marketing Engineer
INTRODUCTION

The Age of Information

In our digital age, information is only a search away, and the design problems that used to plague electrical engineers in the past have largely been ironed out by someone, somewhere. Never before in PCB design has information been so readily available, and problems documented so thoroughly. How does this affect you as an electronics designer? There is no need to continue reinventing the wheel and making the same cycle of mistakes as our predecessors did in years past.

Whether it’s seasoned electrical engineers or those fresh out of university, the same question is always asked: How do I design better? We are all dealing with the complexities of denser boards, higher clock rates, and smaller mechanical enclosures, and designing for those requirements alone can be a challenge. However, your designs exist beyond the digital domain, and to successfully produce a manufacturable board there are a number of additional guidelines to carefully consider throughout your entire design process.

The reality is, the process of designing better doesn’t end the minute you ship off your documentation to manufacturing; it ends when you get your board back in its physical form and it works as intended. This goal can be a challenge for most PCB designers, who commonly have to deal with a myriad of unique requirements that each manufacturer sets forth, only to get lost in the details as design projects run off their intended course.

Here’s the good news—there’s a way to design your PCB not just for the digital domain, but for the manufacturing world as well. And when you design it for manufacturability, you will start seeing your boards come back right the first time.
You can think of this guidebook as an accumulation of knowledge that has been handed down from those before you. The collective years of experience in the electronics industry have allowed us at Altium to soak up knowledge from PCB designers all around the world, and this knowledge we now pass on to you.

Would you like more information? Watch the on-demand webinar at Altium.com: *Tech Briefing: DFM - Maximize your PCB Production Yield.*

**What is Design for Manufacturing?**

The goal for this guidebook is simple - get a good board back, every time. And the applied methodology for doing this is design for manufacturing (DFM). You might have heard of DFM in the past, but what exactly does it mean?

*Design for manufacturing (DFM) is the process of designing a PCB that is manufacturable, functional, and reliable.*

With this definition in mind, we have several clear goals to reach by adopting the design practices within this guidebook:

1. Eliminate the need for multiple board re-spins due to manufacturing-specific details that were missed in a design process.
2. Design and produce boards that are both manufacturable and functional by following a set of best practices set forth by PCB design veterans.
3. Reduce the time spent on design revisions and ultimately meet time-to-market goals consistently by following a set of best practices for board layout and documentation.

To meet these goals, we've structured this guidebook to be read from start to finish to match up with your design workflow. As you read each section in the following chapters, you will be able to apply the knowledge to each stage of your PCB design process.
What You Will Find in This Guidebook

This guidebook is both theoretical and practical, and applies trusted and accepted design science that has resulted in consistently manufacturable boards.

The major sections in this guidebook include:

**Section 1: Design Guidelines for Successful Manufacturing**
In this section, we will be covering design practices that will produce a functional and manufacturable board layout. This section will include:

- **Chapter 1:** Understanding the typical PCB manufacturing process and its various stages.
- **Chapter 2:** Selecting the right materials for your PCB to meet your specific design requirements.
- **Chapter 3:** Strategizing your PCB layout including via/hole placement, solder mask layers, and silkscreen documentation.
- **Chapter 4:** Placing and orienting your components to ensure proper spacing and assembly.
- **Chapter 5:** Configuring test point requirements for successful board testing by your manufacturer.

**Section 2: Documentation Guidelines for Successful Fabrication and Assembly**
With your design complete and ready for manufacturing, we will then be moving on to properly documenting a PCB to provide crystal-clear design intent to your manufacturer. This section will include:

- **Chapter 6:** Understanding the main factor in the PCB documentation process and what needs to be sent to your manufacturer.
- **Chapter 7:** Assembling the master drawing of your PCB to accurately portray all of the fine details needed to manufacture a board.
- **Chapter 8:** Understanding what you need to include in your assembly documentation to have your bare board created with your selected components.

By the end of this guidebook you will be well equipped to implement the design and documentation practices into your own personal workflow to produce fabrication-ready PCBs.
- Section 1 -

Design Guidelines for Successful Manufacturing
Before undertaking a DFM process, it is important to understand the underlying process behind producing a physical PCB. Regardless of the various technologies present in each facility, a large majority of industry-leading manufacturers follow a specific set of steps to turn your design from digital bits to physical boards. The steps in this process are outlined in Figure 1.

With the final curing of your board complete, a manufacturer will then begin the electrical test process with the provided test points you established on your board layout. All boards that pass this verification process are considered complete and then make their way through shipping and transport.

**Typical Cost Drivers in the PCB Manufacturing Process**
The cost to have your board manufactured is largely determined by the specific materials and parts that you specify during your design phase.

**Making Manufacturing-Conscious Design Decisions**
By understanding the typical PCB manufacturing process, you will be well on your way towards making more informed choices for materials and part selection at design time. Now, let’s jump into the DFM process, starting with material selection.
Standard PCB Manufacturing Process

1 - Data transfer from customer
  2 - Data prep
  3 - Cores/laminate

4 - Dry film resist coating

5 - Place artwork
6 - Expose panels to ultraviolet light
7 - Develop panels (resist removal)

8 - Etch

9 - Strip resist

10 - Oxide coating

11 - Multilayer lamination
12 - Primary drilling

13 - Deburr and clean
14 - Desmear
15 - Copper deposition

Figure 1
16 - Dry film photoresist coat

17 - Expose and develop

18 - Copper pattern plate (electroplating)

19 - Strip resist

20 - Etch

21 - Solder mask and cure

22 - Hot air solder leveling (most common PCB surface finish)

23 - Surface finishes

24 - Legend and cure

25 - Fabrication and routing

26 - Electrical test/final inspection
- Chapter 2 -
Selecting Your Materials
Every design process begins with material selection, and this chapter focuses on selecting the right materials for your PCB design given the particular design requirements you outline in your specifications. We will be focusing largely on FR-4 as it is the most commonly used material for PCB design. If your specific material requirements are not listed in the following sections, please contact your manufacturer for further guidance.

**Basic Material Selection Process**

When designing a PCB, there are several material choices to consider based on your unique design needs. Before selecting a material, it is recommended to first define the functionality and reliability requirements that your board must meet. See Figure 2 for a visual on how to begin your material selection process. [2-1]

![Diagram: Designer/End User Materials Selection Map](image)

- **Better Thermal Performance Than FR-4**
  - YES: Greater Than 200°C Tg?
    - YES: PI, PI Blends - CE, CE Blends
    - NO: Modified Epoxy
  - NO: Modified Epoxy

- **Better Electricals? (High Speed, CTI)**
  - YES: High CTI?
    - YES: FR-6 / CRM-5
    - NO: Modified Epoxy, CE, PTFE, PI, PI Blends
  - NO: Aramid Based Systems, Quartz Based Systems, Metal Core

- **XY Dimensional Constraints (SMT)?**
  - YES: Leadless Or Very Complex Leaded?
    - YES: Multi-Epoxy, Modified Epoxy, CE
    - NO: Multi-Epoxy, Modified Epoxy, CE
  - NO: FR-4

- **Greater Than 10 Layers?**
  - YES: Z Axis Concerns?
    - YES: Multi-Epoxy, Modified Epoxy, CE, PI
    - NO: FR-4
  - NO: CEM-3, CRM-5, FR-6

- **Punched Plated Through-Holes**
  - YES: FR-6, CRM-5, CEM-1, CEM-3
  - NO: FR-2, XXP

- **Single Sided**
  - YES: FR-4 Type:
    - Bifunctional
    - Multifunctional
    - Tertiary
    - Modified Epoxy

Figure 2 - Designer/End-user materials selection map. [2-1]
Material Properties in Detail

Electrical Properties

The most critical properties to consider for electrical requirements are electrical strength, dielectric constant, and moisture resistance. Refer to Figure 3 for a list of some of the more common materials and their associated property values. Remember to consult with your manufacturer for more specific data on electrical properties.

<table>
<thead>
<tr>
<th>Property</th>
<th>FR-4 (Epoxy E-glass)</th>
<th>Multifunctional Epoxy</th>
<th>High Performance Epoxy</th>
<th>Bismaleimide Triazine/Epoxy</th>
<th>Polyimide</th>
<th>Cyanate Ester</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dielectric Constant (neat resin)</td>
<td>3.9</td>
<td>3.5</td>
<td>3.4</td>
<td>2.9</td>
<td>3.5-3.7</td>
<td>2.8</td>
</tr>
<tr>
<td>Dielectric Constant (reinforcement)</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>Electric Strength (V/mm)</td>
<td>39.4 x 10^3</td>
<td>51.2 x 10^3</td>
<td>70.9 x 10^3</td>
<td>47.2 x 10^3</td>
<td>70.9 x 10^3</td>
<td>65 x 10^3</td>
</tr>
<tr>
<td>Volume Resistivity (Ω·cm)</td>
<td>4.0 x 10^6</td>
<td>3.8 x 10^6</td>
<td>4.9 x 10^6</td>
<td>4 x 10^6</td>
<td>2.1 x 10^6</td>
<td>1.0 x 10^6</td>
</tr>
<tr>
<td>Water Absorption (wt%)</td>
<td>1.3</td>
<td>0.1</td>
<td>0.3</td>
<td>1.3</td>
<td>0.5</td>
<td>0.8</td>
</tr>
<tr>
<td>Dissipation Factor (DX)</td>
<td>0.022</td>
<td>0.019</td>
<td>0.012</td>
<td>0.015</td>
<td>0.01</td>
<td>0.004</td>
</tr>
</tbody>
</table>

Figure 3 - Typical properties of common dielectric materials. [2-2]

Copper Foil Types

Manufacturers will typically offer various types of foil for you to choose from, the most common being electro-deposited (ED) copper and rolled copper. Rigid boards will typically use electro-deposited copper foil whereas rigid-flex boards will use rolled copper foil.

Copper Resistance Values

As boards get denser and more complex, it becomes increasingly important to calculate your copper’s distributed resistance. You can use the following formula to easily compute the resistivity in your copper traces:

\[ R = \rho \frac{L}{A} \]

where:
- \( R \) is the end-to-end track resistance in Ohms
- \( \rho \) is the resistivity of the track material in Ohm Meters
- \( L \) is the track length in meters
- \( A \) is the track cross sectional area in square meters
Current Carrying Capacity of Copper

Figure 4 can be used as a reference to understand the current carrying capacity of internal layers for common copper thicknesses and temperature levels above ambient.

![Graph showing current carrying capacity](image)

Figure 4 - Encapsulated conductor widths. [2-6]

Finished Board Thickness

As part of your final material selection process, you will want to calculate your finished board thickness. This measurement is made from copper to copper and will represent your maximum finished board thickness.
Chapter 3
Strategizing Your PCB Layout
With your material selections finalized, it is now time to dive into the specific details of your PCB layout. While individual engineering workflows might differ from one designer to the next, a number of primary design considerations demand precise DFM requirements to consider a board 100% ready for manufacturing. In the following sections, you will learn the specifics of strategizing your PCB layout including SMT and through-hole specifications, silkscreen documentation, solder mask applications, and more.

**Deciding Between Through-Hole or SMT**
Choosing plated through-hole (PTH) components or surface mount (SMT) will have a direct impact on your overall costs and manufacturing time. It is recommended to stick with SMT for professional board designs as this results in quicker board turnarounds and higher reliability.

**Silkscreen and Component IDs**
All component outlines on your silkscreen should be marked with a reference designator and polarity indicators (if applicable).

**Component Reference Designators**

**Solder Mask**
The solder mask is a thin, lacquer-like layer applied as a final coating to your PCB to protect various features, including copper traces and copper pour that should not be soldered.

**Vias and Holes**
Vias are a critical part of every PCB design and are responsible for transmitting electrical current between layers.

**Via Clearance Requirements**
Standard vias should maintain minimum clearances from adjacent conductors, and the clearance will largely depend on whether the via is tented or exposed.

**Via Size Guidelines**
When designing plated vias, it is recommended to maintain an aspect ratio of 8:1 between the hole diameter and the substrate thickness.
Figure 5 depicts typical standard drill sizes:

<table>
<thead>
<tr>
<th>Drill Number</th>
<th>Holes size</th>
<th>Finished Hole Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>70</td>
<td>.028”</td>
<td>.025”</td>
</tr>
<tr>
<td>65</td>
<td>.035”</td>
<td>.032”</td>
</tr>
<tr>
<td>58</td>
<td>.042”</td>
<td>.039”</td>
</tr>
<tr>
<td>55</td>
<td>.052”</td>
<td>.049”</td>
</tr>
<tr>
<td>53</td>
<td>.0595”</td>
<td>.056”</td>
</tr>
<tr>
<td>44</td>
<td>.086”</td>
<td>.083”</td>
</tr>
<tr>
<td>1/8”</td>
<td>.125”</td>
<td>.122”</td>
</tr>
<tr>
<td>24</td>
<td>.152”</td>
<td>.149”</td>
</tr>
</tbody>
</table>

Figure 5 - Standard drill sizes for vias and holes.

**Annular Rings**
The annular ring is the difference between the pad diameter and the corresponding drill diameter. Figure 6 shows how to easily calculate the width of an annular ring:

\[
\text{Annular ring width} = \frac{\text{diameter of the pad} - \text{diameter of the hole}}{2}
\]

**Exposed Vias**

Figure 6 - Recommended annular ring width.
Exposed vias are exposed electrical connections that are not covered with solder mask.

**Tented Vias**
Tenting a via covers the via and annular ring with solder mask, and should be set as the default method in your design workflow.

**Via-in-Pads and Microvias**
Via-in-pads allows for close placement of bypass capacitors and makes routing easier for any ball pitch BGAs, and assist with thermal management and grounding.

**Blind and Buried Vias**
Similar to through-holes, blind and/or buried vias (BBV) are holes that connect one or more layers. In this process, a blind via connects an outer layer to one or more inner layers but not to both outer layers, and a buried via connects one or more inner layers, but not to an outer layer.

See Figure 7 for an example of a blind and buried via application:

![Figure 7 - Blind and buried vias.](image-url)
**Aspect Ratio Plating**
Aspect ratio is the ratio between the thickness of the board and the size of the drilled hole (before plating). Figure 8 shows a visual example of how aspect ratios are determined on a PCB:

$\text{Aspect Ratio} = \frac{t}{d}$

**Trace Routing to Component Lands**
When you have a component’s termination that could generate heat and is connected to a large trace, the heat transfer produced can lead to a poor solder joint. In the following sections, you will learn how to mitigate these issues.

**Necking a Trace**
A general guideline for necking a trace is to keep it no wider than 0.010” where it connects to the pad and run it at least 0.010” before it connects to the large trace. Figure 9 shows an example of this process:
Connecting Pads to Traces

Every pad should be connected to its own trace, and it is recommended to have the routing from either outside the edges or inside the edges of the pads while keeping the routing symmetrical.

When routing leaded SMT components, it is recommended to route the trace over and then back in, forming a flipped “U” configuration, rather than forming an “H” by going directly between lands. See Figure 10 for an example of this “U” shaped configuration:

Figure 9b - Connecting large traces to component lands (bad design).

Figure 10 - “U” configuration for routing leaded SMT components.
Chapter 4
Placing and Orienting Your Components
With your preferred component types established, it is now time to de-
cide how to efficiently place and orient those parts on your board. This
process will have a large effect on how you utilize the available space
on your board layout, and can be one of those most challenging steps
in your design process. In the following sections, you will find specific
recommendations on how to optimize your component placement to
be both manufacturable and capable of meeting your specific design
requirements.

**General Component Placement and Spacing Guidelines**

Before going into the specifics of component placement and orienta-
tion, there are several general guidelines to keep in mind:

- Orient similar components in the same direction.
- Avoid placing components on the solder side of a board.
- Try to place all your SMT components on the same side
  of the board, and all the through-hole components (if
  mixed) on the top side of the board.
- When you have mixed-technology components (SMT and
  PTH), manufacturers might require an extra process to
  epoxy the bottom components.
- You should terminate all lands with only one trace.
- When you specify a chip under a device, this can make
  inspections, rework, and test more difficult.
- All components used on the wave solder sides of an as-
  sembly should first be approved by your manufacturer
  for immersion in a solder bath.

**Finalizing Your Component Placement and Board Orientation**

With the information presented in this chapter, you are now well
equipped to begin your component placement and orientation process
to meet fundamental manufacturability requirements. Now that your
design is well on its way to completion, it is time to finalize the board
layout process by configuring your test point requirements in the next
chapter.
Chapter 5 - Configuring Your Test Point Requirements
Defining proper test points on a board layout during your design process is critical for having your PCB tested and verified by your manufacturer. This chapter will cover general testing requirements for your PCB, and will then go into the specifics of test pad placement and panelization.

**General Test Point Requirements**
Before going into the specifics of test point and pad requirements, there are several general guidelines to keep in mind:

- Each node on your board should have at least one test probe point.
- It is not recommended to use component leads as test points.
- It is recommended to distribute your test points throughout your board.
- The spacing between test pads (center-to-center) should be maintained at 0.100”.

**Test Pads**
Test pads can be either vias/pads, a component pad (PTH), or a specified test point (TP) with its own reference designator.

See Figure 11 for an example of a through-hole test via.
**Test Pad Spacing and Tooling Requirements**

The spacing between test pads (center-to-center) should be maintained at 0.100”. This lets the board fabricator use larger probes, which are less expensive to setup and provide a more reliable reading.

**Test Pads for SMT Boards**

Components on SMT boards that are 0.35” high (or more) are difficult to probe, so it is recommended to keep the clearance at 0.100” between the test pads and the edge of these components.

**Test Tooling Requirements**

At a minimum, two tooling holes are required on the PCB. They should be as far apart as possible, diagonally placed, and have a 0.125” dia..

**Panelization**

Panelization, also known as step-and-repeat, is the method of placing two or more PCBs onto one panel, which allows boards to be secured during manufacturing, shipping, and assembly. Panelization can also save you time by processing multiple boards at once in bulk as shown in Figure 12.

![Figure 12 - Four rectangular circuits in a single panel with tooling holes & breakout tabs.](image)
General Guidelines for Panels
Common panel sizes are 12” x 18” and 18” x 24”. The following specifications should be included for a standard panelization:

- Breakaway strips should measure around 0.400”.
- Fiducials should be at least 0.125” away from a card edge or panel frame edge.
- Panel designs should have 0.125” unplated tooling holes located 0.2” from frame corners (or per your manufacturer’s guidelines.)

Tooling Holes
Tooling holes are required to accurately align and position the circuit board in machines and fixtures to be processed (e.g. routing fixtures, solder paste screen printing process, drill machines, test fixtures, etc.)

Depanelization Process
There are several depanelization methods outlined below, all having benefits for use depending on the physical constraints of your board shape and associated components. Your specific design requirements will determine what particular depanelization process to use, and it is recommended to consult with your manufacturer to select the ideal solution.

Some of the popular depanelization methods are:

- Breakaway Tabs
- Solid Breakaway
- V Grooving
- Irregularly Shaped PCBs

Finalizing Your Board Layout
By adding proper test points on a board, you will significantly increase the likelihood of detecting any manufacturing related errors during the post-production validation process. Given that every design has its limitations and unique physical constraints, it is always recommended to consult with your manufacturer to determine the ideal placement of test points.
Section 2 - Documentation Guidelines for Successful Fabrication and Assembly
Before you can send your design off to manufacturing, you will need to ensure that it is properly documented to clearly communicate your design intent. While electronic files such as Gerber and ODB++ provide enough basic information to make your board, they don't include all of the fine details about how you intend to have your board produced.

This chapter will focus on creating a standard PCB documentation template. It outlines all of the necessary details you will want to include in your documentation to help convey your design intent to your manufacturer. Chapter 7 will then go into the specifics of your master drawing. Chapters 6 and 7 pull information from the standard IPC-D-325A. [6-1]

**Drawing Sizes**

The first step in creating a master drawing is selecting an appropriate drawing area to contain all of your drawings. The dimensions of your drawing area are referred to as the drawing size and should comply with the ANSI-Y 14.1 [6-1] standard sizes as shown in Figure 13.

Figure 13 - Standard drawing sizes for PCB documentation. [6-2]
Primary Blocks of a PCB Template for Fabrication and Assembly
Several blocks need to be included on your PCB drawing. A block includes additional details and specifications that will help to clearly define your design requirements for manufacturing and should be fully detailed to avoid any potential production delays or errors.

Figure 14 shows a blank drawing space with the blocks highlighted.

![Figure 14 - Blank PCB drawing space with highlighted blocks.](image)

Title Block
The title block is an important part of your PCB design, as it communicates to your manufacturer basic information necessary for manufacturing your board. It should include:

- Title
- Scale
- Drawing number
- Cage code
- Approval block
The following figures show these sections in detail on the title block and provide additional details about what needs to be included:

**Title and Subtitle**

The title and subtitle provide a brief and accurate description of the PCB and should be written in capital letters.

**Drawing Number (DWG. NO.)**

The drawing number is used for filing and identification of the PCB project.

**Revision Block**

The revision block is used to keep track of the project revision and can be seen in Figure 17.
Material Block
The material block contains numbers corresponding to the appropriate notes, specifying the materials being used.

Revision Status Block
The revision status block contains information resides on the first page of the master drawing and shows the revision status for each individual sheet of the drawing. This block should be located at the top right corner of your PCB template.

Figure 18 - Material block.

Figure 19 - Revision status block.
Continuation Sheet Block
The continuation sheet block is used for sheets other than the first page. A continuation sheet block should be placed at the bottom right corner of the page as shown in Figure 20.

<table>
<thead>
<tr>
<th>DRAW</th>
<th>NAME</th>
<th>DATE</th>
<th>Rev.</th>
<th>Cage Code</th>
<th>DWG. NO.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Alex</td>
<td>5/23/2016</td>
<td></td>
<td>00000</td>
<td>00000</td>
</tr>
<tr>
<td>CHECKED</td>
<td>Sam</td>
<td>5/23/2016</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ENG APPR.</td>
<td>Dan</td>
<td>5/23/2016</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Q.A.</td>
<td>Jeff</td>
<td>5/23/2016</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Size

<table>
<thead>
<tr>
<th>Scale</th>
<th>Sheet</th>
</tr>
</thead>
<tbody>
<tr>
<td>1:1</td>
<td>6 of 9</td>
</tr>
</tbody>
</table>

Figure 20 - Continuation sheets.

Schematic Title Block
While a schematic title block shares much of the same information as its PCB counterpart, including drawing size, date, title and revision (see Figure 21), it also has a number of differences as outlined here:

![Figure 21 - Schematic title block.](image)

Reference Documents Block
The reference documents block lists the required project production documentation.

![Figure 22 - Reference documents block.](image)
Assembly Drawing Number
The assembly drawing is a detailed depiction of the entire board structure with all components placed.

Fab Drawing Number
The fabrication drawing depicts areas on the board that require construction, such as the layer stack and drill table.

BOM Document Number
The bill of materials (BOM) integrates all aspects of your design to produce your finished product. The BOM is discussed in greater detail later in this guidebook.

PCB Drawing Number
The PCB drawing number is the unique number assigned to the PCB drawing.

Project
This block is used to input the name or number of the main project.

---

File Name
The file name refers to the saved filename including extension.
Company Name and Address
This area is for your company's name and mailing address.

<table>
<thead>
<tr>
<th>APPROVALS</th>
<th>DATE</th>
<th>PROJECT</th>
<th>TITLE</th>
</tr>
</thead>
<tbody>
<tr>
<td>ENG:-------</td>
<td>-----</td>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>DSN:-------</td>
<td>-----</td>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>CHK:-------</td>
<td>-----</td>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>REFERENCE DOCUMENTS</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>BOM: &lt;BOM DOG NO&gt;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ASSY DWG: &lt;ASSY DWG NO&gt;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FAB DWG: &lt;FAB DWG NO&gt;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PCB DWG: &lt;PCB DWG NO&gt;</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SIZE: B</td>
<td>SCALE: Scale</td>
<td>CAGE CODE</td>
<td>0ZL62</td>
</tr>
<tr>
<td>DWG NO.</td>
<td>FILE NAME</td>
<td>&lt;SCH DWG NO&gt;</td>
<td>Top Sheet, SchDoc</td>
</tr>
<tr>
<td>REV</td>
<td>SHEET: 1 OF 2</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 25 - Company name and address block.

Finalizing Your Basic Fabrication Documentation
Communicating basic information about your design to your manufacturer and stakeholders mitigates risks of design intent miscommunication. It is highly recommended to utilize the optional blocks that will best fit your particular project’s requirements to facilitate organization of your design documentation. Now that we have tackled the naming and organization of our documents, let's take a look at the content of the master drawing.
- Chapter 7 -
Documenting Your Master Drawing
The master drawing is the most critical part of your design documentation and will convey all of the fine details needed to manufacture your board. There are specific requirements that you should include in every master drawing.

Master Drawing Class Requirements

Documentation requirements for your master drawing will change depending on the class of the board you are designing. The class of the board can be thought as the level of care that you might give the board.

There are three class levels according to IPC-D-325A [6-11]: Class 1, Class 2, and Class 3. Refer to the standard for definitions.
Board Details
The board details defines board complexity and structure.

Board Type
The board type as mentioned in IPC-2221 specifies the complexity of your board. There are six primary board types that you will want to include on your master drawing:

- Type 1 - Single-sided
- Type 2 - Double-sided
- Type 3 - Multilayer board through-hole components only
- Type 4 - Multilayer board with through, blind and/or buried vias
- Type 5 - Multilayer metal-core board through-hole components only
- Type 6 - Multilayer metal-core board with through, blind and/or buried vias

Board Dimensioning
Board dimensioning is a large subject that warrants its own guidebook; this guide will touch on just a few key points. For a more detailed and complete look at dimensioning please refer to IPC-C-300 and ASME-Y-14.5.
• You should add a tolerance to each dimension you place.
• Avoid over-defining a drawing with unnecessary dimensions.
• Clearly dimension a drawing so that there is only one interpretation possible.
• Arrange your dimensions to maximize readability.
• Dimension without indicating manufacturing methods.
• Specify the origin.
• Linear dimensions should use a numerical value at its center with arrows.
Callouts

A callout connects an item with relevant detailed notes. An example of a callout can be seen in Figure 29 referencing the notes from the title block.

Notes:
1. Fabricate boards in accordance with IPC-RB-276, Class 3. Finished boards must meet quality conformance testing and inspection as specified.
2. Material in accordance with MIL-S-13949/4, laminated sheet, HTE copper-clad, type GF glass cloth base, flame resistant (meeting UL 94V-1 or better). Tg rating: 140 to 160°C.
3. Material in accordance with MIL-S-13949/12, plastic sheet, type GF base material, glass base preimpregnated (B-stage). Tg rating: 140 to 160°C.

Bow and Twist

The bow and twist notes tell you how flexible or durable the board is by testing how much a board can bend without breaking.
**Board Layer Stack**

The board layer stack legend includes details about each layer in your board. It is recommended to include the five columns (layer, material, thickness, type, and gerber) as shown in Figure 30 in every project to keep documentation consistent and streamlined across designs.

![Layer Stack Legend](image)

**Figure 30 - Layer stackup legend.**

**Materials**

The materials section defines what materials should be mentioned in the notes section of your master drawing and should specify:

- UL requirements
- Laminate type
- Copper foil

The material for the marking inks should also be mentioned. If the marking ink is conductive, then it needs to be properly isolated from the circuitry by spacing it away from other copper or given a coating.

**Hole Details**

The hole details provides information regarding each drill hole. A standard element required to manufacture a PCB is a drill legend, also known as a drill table. Each drill is represented by a symbol, a letter or the actual hole size. A properly constructed drill table should look like Figure 31.
Drill Pattern Drawing

The drill pattern drawing shows all drill locations and can be included as a pen plot, photoplot, or photographic composite copy. This must be produced at a 1:1 scale. Figure 32 shows a typical drill pattern documented on a PCB.
**Blind and Buried Vias**

Blind and buried vias, as mentioned in Chapter 3, are vias that connect layers, but do not make it all the way to either the top or bottom layer. When working with Type 4 or 6 boards (multilayer boards with blind and buried vias), it is important to specify the location of the vias for each layer pair on the drawing.

![Diagram of three via types](image)

**Example of 6 Layer BBV Board**

Figure 33 – Three via types.

**Test Points**

Test points are used to probe areas of your board post manufacturing to ensure quality. You should include test points as part of your drill table and drill pattern drawings.
### Markings

Markings are used to show the level of safety associated with your board. The appropriate markings should be placed or noted on the master drawing. In Figure 34 you will notice many countries have their respective markings, and there are even some standards that have a global reach, such as RoHS.

<table>
<thead>
<tr>
<th>Mark</th>
<th>Country</th>
</tr>
</thead>
<tbody>
<tr>
<td>☑️</td>
<td>Australia</td>
</tr>
<tr>
<td>☐️</td>
<td>Canada</td>
</tr>
<tr>
<td>☐️</td>
<td>China</td>
</tr>
<tr>
<td>☐️</td>
<td>Germany</td>
</tr>
<tr>
<td>☐️</td>
<td>Finland</td>
</tr>
<tr>
<td>🇮🇹</td>
<td>Italy</td>
</tr>
<tr>
<td>🇩🇪</td>
<td>Denmark</td>
</tr>
<tr>
<td>🇳🇴</td>
<td>Norway</td>
</tr>
<tr>
<td>🇦🇹</td>
<td>Austria</td>
</tr>
<tr>
<td>🇱🇵</td>
<td>Poland</td>
</tr>
<tr>
<td>☐️</td>
<td>Sweden</td>
</tr>
<tr>
<td>☐️</td>
<td>Switzerland</td>
</tr>
<tr>
<td>☐️</td>
<td>USA</td>
</tr>
<tr>
<td>☐️</td>
<td>Europe</td>
</tr>
<tr>
<td>☐️</td>
<td>Germany</td>
</tr>
<tr>
<td>☐️</td>
<td>Germany</td>
</tr>
<tr>
<td>☐️</td>
<td>Germany</td>
</tr>
<tr>
<td>☐️</td>
<td>USA</td>
</tr>
<tr>
<td>☐️</td>
<td>USA</td>
</tr>
<tr>
<td>☐️</td>
<td>USA</td>
</tr>
<tr>
<td>☐️</td>
<td>Japan</td>
</tr>
<tr>
<td>☐️</td>
<td>Japan</td>
</tr>
<tr>
<td>☐️</td>
<td>USA</td>
</tr>
<tr>
<td>☐️</td>
<td>USA</td>
</tr>
<tr>
<td>☐️</td>
<td>Saudi Arabia</td>
</tr>
<tr>
<td>🇪🇺</td>
<td>Holland</td>
</tr>
<tr>
<td>🇪🇸</td>
<td>The Globe</td>
</tr>
<tr>
<td>☐️</td>
<td>Germany</td>
</tr>
</tbody>
</table>
Restriction of Hazardous Substances (RoHS)
The Restriction of Hazardous Substances (RoHS) standard is used throughout the world. The RoHS standard restricts the use of substances such as lead and mercury in electrical and electronic equipment as they are hazardous to the environment and humans. Markings for RoHS should be placed or noted on the master drawing. Figure 51 shows some RoHS compliant markings.

Figure 35 - RoHS marking labels.

Underwriters Laboratories Inc. (UL)
One marking not mentioned is the Underwriters Laboratories marking, UL. Having UL recognition for your board means that your base materials and design were manufactured through UL-approved processes. For more information on this standard, refer to the UL 796 standard for printed wiring boards. The UL marking should be indicated on the drawing or notes like all other markings.

Electrostatic Discharge (ESD)
These markings are electrostatic discharge markings and are placed on a static sensitive board. If there is enough space, markings may be added to the board via silkscreen. Make sure that these markings are visible on both the board (if applicable) and master drawing.

Marking Inks
Marking Inks are used to create drawing labels and markings. The master drawing should be marked with some fixed format information including part number, layer number, revision level, and orientation symbols.
Processing Conditions
The processing conditions include details about how your board will be produced during manufacturing and assist your manufacturer to efficiently optimize their equipment for your particular design requirements.

Quality Conformance Coupons
Quality conformance coupons, also known as test coupons, are small PCB sections used for testing and are made at the same time as the primary board. They are used for testing a variety of variables includes impedances and inter-plane capacitance. These quality conformance circuitries should appear on the master drawing as they appear in the PCB panel.

Process Specifications
Process specifications include the information a manufacturer needs when performing certain processes, such as cleaning and preparing your board. The process information should include information regarding tolerances.
Grid System
The grid system plays an important part in the creation of boards and is used for the location of features and items including components, plated through-holes, and surface mount land patterns. Features not residing on the grid need to be dimensioned with a given tolerance.

Manufacturing Features
The manufacturing features include additional information that will allow your manufacturer’s machinery to see how a board is oriented in space for efficient processing.

Datum
A datum is a reference point located on the printed board, usually a hole, that allows a machine to “see” how the board is oriented in space. For more detailed information, please refer to IPC-D-300. [7.3]

Fiducial Targets
Fiducials are a specific type of datum usually used for pick-and-place machines and allow the machine to know where the PCB is oriented in space. Fiducial targets must be shown on all surface mount artwork.

Figure 37
Datum placed at each of two corners of a BGA.

Figure 38 - Fiducials on an empty board.
Master Drawing Documentation
The master drawing documentation section includes all supporting details for your PCB including notes, callouts, and additional artwork to help clarify your manufacturing intent.

Artwork Configuration Control Chart
The artwork configuration control chart is a drawing that identifies and manages the revision levels of various artworks including silk screens, solder paste, and NC drill data.

<table>
<thead>
<tr>
<th>Artwork</th>
<th>DWG. No.</th>
<th>Rev Level</th>
<th>Cu Weight oz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Signal Layer 1</td>
<td>00001</td>
<td>A</td>
<td>.5</td>
</tr>
<tr>
<td>Signal Layer 2</td>
<td>00002</td>
<td>A</td>
<td>.5</td>
</tr>
<tr>
<td>Signal Layer 3</td>
<td>00003</td>
<td>A</td>
<td>.5</td>
</tr>
<tr>
<td>Signal Layer 4</td>
<td>00004</td>
<td>A</td>
<td>.5</td>
</tr>
<tr>
<td>Signal Layer 5</td>
<td>00005</td>
<td>A</td>
<td>.5</td>
</tr>
<tr>
<td>Signal Layer 6</td>
<td>00006</td>
<td>A</td>
<td>.5</td>
</tr>
<tr>
<td>Power Plane VCC</td>
<td>00007</td>
<td>A</td>
<td>1</td>
</tr>
<tr>
<td>Power Plane GND</td>
<td>00008</td>
<td>A</td>
<td>1</td>
</tr>
<tr>
<td>Solder Mask Top</td>
<td>00009</td>
<td>A</td>
<td></td>
</tr>
<tr>
<td>Solder Mask Bottom</td>
<td>00010</td>
<td>A</td>
<td></td>
</tr>
<tr>
<td>Silkscreen</td>
<td>00011</td>
<td>A</td>
<td></td>
</tr>
<tr>
<td>NC Drill</td>
<td>00012</td>
<td>A</td>
<td></td>
</tr>
</tbody>
</table>

Figure 39 - Artwork configuration control chart.
Notes
Notes accompany the fabrication drawings and are used to communicate your requirements and details for the fabrication process. The following is a list of possible notes that you might want to include with your drawings:

- Finished board specifications and class
- Material requirements
- B-stage material requirements (for multilayer boards)
- Board markings
- Construction
- Solder mask specifications
- Solder mask material & thickness
- Drilled hole requirements
- Copper plating thickness
- Etchback requirements
- Bow and twist requirements
- Silkscreen ink type
- Test coupon requirements
- Bare board electrical test requirements

Completing Your Fabrication Documentation
Your design documentation is arguably one of the most important aspects of your design process. Even the greatest PCB design will go to waste if you are not able to clearly communicate design intent to your manufacturer. In the next chapter, we will explore the last piece of the documentation puzzle for the final assembly of your PCB.
Chapter 8
Documenting Your PCB for Assembly
With your fabrication requirements documented, it is now time to move on to an equally important stage — documenting the instructions for component placement and final assembly. It is in the assembly stage where your bare board is brought to life with all the components you specify in your bill of materials. This chapter will cover what you need to know to have your board successfully assembled.

**Assembly Drawing Requirements**

The assembly drawing requirements define the final board assembly and provide specific instructions to your manufacturer on component placement, orientation, and identification. You will want to include the following details as part of your assembly drawing documentation:

<table>
<thead>
<tr>
<th>Location of components</th>
<th>Reference dimensions for envelopes</th>
<th>Conformal coating requirements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference designations for all parts</td>
<td>Component mounting and spacing installation requirements</td>
<td>Mechanical hardware including latches and mounting hardware</td>
</tr>
<tr>
<td>Orientation and polarity of components</td>
<td>Masking requirements</td>
<td>Cleanliness requirements</td>
</tr>
<tr>
<td>Required structural details for support and rigidity</td>
<td>Electrostatic discharge label</td>
<td>Workmanship specifications</td>
</tr>
<tr>
<td>Requirements for markings and lead forming</td>
<td>Electrical test requirements</td>
<td></td>
</tr>
<tr>
<td>Specific soldering requirements including solder paste</td>
<td>Eyelets and terminals</td>
<td></td>
</tr>
</tbody>
</table>

**Required Assembly Documentation**

The required assembly documentation consists of a number of assembly drawing templates that you will need to include with your final design, including schematic prints and a finalized BOM. In addition to your notes, these drawing templates will allow your manufacturer to clearly understand your design intent for final component placement and assembly.
Schematic Prints
The schematic prints outline your intended board component connections and are necessary to define and establish your required test points.

Bill of Materials
The bill of materials will include a detailed and sourceable part list that includes all necessary part supplier information. Providing a BOM to your manufacturer with included component designators and supplier information ensuring your design will be manufactured with the appropriate parts.

Cautionary Markings
Cautionary markings are very important for safe handling of your board. As part of your assembly documentation, you will need to include the electrostatic discharge markings shown in Figure 40 if your board requires special handling due to static sensitivity.

![Electrostatic discharge markings.](image)

Figure 40 - Electrostatic discharge markings.

Assembly drawings, like fabrication drawings, require their own set of notes. These notes include information about the merging of the board with its components, including assembly standards, handling instructions and solder specifications. If you think your manufacturer needs to know something about a specific component’s placement or assembly requirements, make a note of it.
In addition to the symbols above, the notes in the following table need to be placed above or as close to the title bar as possible. The notes used depends on the class of your board, and markings can be applied using copper etching or silk screening.  

Refer to [IPC-D-325](#) if you are unsure about which class your board is:

- **Class 1** - Devices sensitive to voltages of 2,000 or less
- **Class 2** - Devices sensitive to voltages between 2,001 and 4,000
- **Class 3** - Devices sensitive to voltages greater than 4,000

### Notes

Assembly drawings, like fabrication drawings, require their own set of notes. These notes include information about the merging of the board with its components, including assembly standards, handling instructions and solder specifications. If you think your manufacturer needs to know something about a specific component's placement or assembly requirements, make a note for it.

### Completing Your Design Documentation

This chapter concludes the final link needed to successfully document your PCB for fabrication and assembly. With a completed board layout in hand and all of the details documented to successfully communicate your manufacturing intent, you're now ready to ship your design files off to manufacturing to begin the production of your board.
DFM isn’t just about your design process; it is about being aware of what happens both before and after you complete your board layout, from the first component you place digitally to the last part a pick-and-place machine places on your PCB. At its core, DFM is as much an art as it is a science, requiring engineers to be aware not only of their own cares and concerns in the design process but every stakeholder’s needs as well. If there is one thing for certain in the world of electronics design, it is that no one part of this process exists in isolation, and everything is connected.

While this guidebook is extensive in scope, it is just the tip of the iceberg in the world of DFM. Standards will continue to change, processes will continue to be refined, and manufacturing will continue to get more efficient, but the fundamentals will remain the same. To design a successful PCB right the first time, you need to look through a wider lens and see the design you produce in the digital domain as one small piece of a greater puzzle. Shipping off your design and documentation to manufacturing isn’t the end, but merely the beginning of a much larger ecosystem.
Having reached the end of this guidebook, you should have a new, well-rounded perspective on which to base your future design decisions. The first section looked at the intricacies of the typical PCB design process and outlined specific guidelines to help you create manufacturable boards better, faster, and more reliably than ever. This process has many overarching elements, from the materials you select for each layer, to strategizing the placement of your components and test points.

From there, you moved beyond the design process to documentation, exploring what makes up a complete set of documentation required by every manufacturer. It started with the basic components of a PCB template, and then dived into the finer details, covering how to assemble your master drawing and prepare your manufacturing files.

Regardless of where your interests lead you after completing this guidebook, it is our hope that you’ve walked away with a clearer understanding of how to accomplish the goals that were set out at the beginning:

1. Eliminate the need for multiple board re-spins due to manufacturing-specific details that were missed in a design process.
2. Design and produce boards that are both manufacturable and functional by following a set of best practices set forth by industry-leading manufacturers.
3. Reduce the time spent on design revisions and ultimately meet time-to-market goals consistently by following a set of design practices for board layout and documentation.

And the most important goal of all: getting a good board back from manufacturing, right the first time, every time.
A
ANNULAR RINGS - The conductive material on the pad that surrounds the hole.
APERTURES - A defined space shape in a Gerber file used to create images of your layers.
ARC - Utilizes the current aperture to create circular segments.
ASPECT RATIO - The ratio between the thickness of the board and the size of the drilled hole before plating.
AUTO PLACEMENT INSERTION - A technology that automates the stuffing and populating of PCBs.

B
BALL GRID ARRAY (BGA) - The ball grid array is a package for integrated circuits. Instead of leads, it has a grid of pads to which are attached balls made from solder.
BARE BOARD - An unpopulated or unstuffed PCB without components.
BLIND VIA - A via which connects an outer layer to one or more inner layers but not to the other outer layer.
BOW & TWIST - A printed circuit board's characteristics that determine its flatness, flexibility, and durability.
BURIED VIA - A via which connects one or more inner layers, but not to an outer layer.

C
CAPTIVE PRESS-FIT STUDS - Threaded self-clinching studs.
COEFFICIENT OF THERMAL EXPANSION (CTE) - Measures the fractional change in size of an object relative to the change in temperature.
COLD SOLDER JOINTS - Unreliable and poor soldering areas where the solder did not melt completely.
CONDUCTOR - An electrical path between two component pads.
CONFORMAL COATING - A thin protective chemical coating that conforms to the topology of the PCB, protecting the circuitry.

D
DATUM - A reference point located on the printed board that allows a machine to "see" how the board is oriented in space.
DECOUPLING CAPACITORS - Also known as bypass capacitors, are used to suppress the high-frequency noise in power supply signals.
DESIGN FOR MANUFACTURING (DFM) - The process of designing a functional and reliable PCB that is easy to manufacture.
DIELECTRIC CONSTANT - The ratio of the permittivity of a substance to the permittivity of free space.
DRAW OBJECTS - Produce a straight line with thickness and endings dependent on the shape of the current aperture.
DRILL DRAWING TABLE - Lists the size and number of holes for each drill used on the board.
E
ELECTRO-DEPOSITED (ED) COPPER - A type of copper used to produce rigid PCBs.
ELECTROLESS COPPER - A widely used technique of depositing copper chemically to form plated-through holes.
ELECTROSTATIC DISCHARGE (ESD) - The sudden flow of electricity between two electrically charged objects caused by contact.
EPOXIED CHIP - Epoxy is usually applied to ICs with very thin wires to give them stronger mechanical bonding.
ETCHBACK - Chemical etching of plated through holes. There are two possible processes, positive etchback and negative etchback.
EXTENDED CODE COMMANDS - Two letter codes paired with the “%” sign for Gerber files.
EYELET - A hollow conductive tube that is used to create electrical connections from one side of a board to another or also as physical support.

F
FIDUCIAL MARK - A round pad or other mark on the surface of a PCB used for optically aligning automatic insertion equipment to the component footprints on the board.
FLASHES - Reproduction of apertures that are often reproduced many times and are commonly used to create pads for Gerber files.
FR-4 - A flame retardant woven glass fabric with epoxy resin.

G
GLASS TRANSITION TEMPERATURE (Tg) - The temperature region at which epoxy transitions from a hard glassy material to a soft rubbery material.

H
HYBRID PCB - Mixed component technology, with both surface mount components and through-hole components.

I
IN-CIRCUIT TEST (ICT) - A powerful test technique that uses a bed of nails, or flying probes, to gain access to all the nodes of a populated PCB.

L
LAMINATE - A dielectric material, usually infused with glass.
LAMINATION - The process of bonding (pressing) together two or more layers of material.
LAND - A land is the remaining conductive material that remains after etching.
LAND PATTERN (LANDS OR PADS) - A combination of lands intended for the mounting and interconnection of a particular component.

N
NC (Numerical Control) Drill File - A PCB fabrication file that defines the tools, locations (X & Y coordinates), and hole sizes that are to be drilled.
PAD - A pad is the remaining conductive material after etching. See also “Land” and “Trace”.

PANELIZATION - The method of placing two or more PCBs onto one panel, which allows multiple boards to be made at the same time, reducing cost.

PART NUMBER - A unique number used to identify a part design within a corporation for consistent and easy reference.

PLANES - Planes are special solid copper internal layers.

PLASTIC LEADED CHIP CARRIER (PLCC) - A square component package commonly having J-leads on all four sides.

POLARITY INDICATORS - Indicate components that can be connected to a circuit only in one direction (e.g., polarized capacitors, diodes, & LEDs).

PREPREG - An abbreviation of pre-impregnated, which is fiberglass impregnated with a resin bonding agent.

PTH (PLATED THROUGH-HOLE) - A hole in which electrical connection is made between external or internal layers or both, by the plating of metal on the wall of the hole.

REFERENCE DESIGNATOR - Reference designators identify components on a electrical schematic or on a PCB (e.g., R253, TP12)

REFLOW SOLDERING - A process of soldering surface mount components to a PCB by mass heating of the entire assembly. The heating process causes solder paste, pre-applied to component land patterns, to melt and form solder fillets between the component leads and land patterns on the board.

REGIONS - Sections defined by linear and circular segments and are commonly used for copper pours for Gerber files.

REGISTRATION - The process of aligning layers of a PCB with holes that have been precisely drilled in specific locations.

RESIN - A high-temperature thermoplastic used with glass to manufacture multilayered printed circuit laminates.

RESISTOR PACK - Resistors that come in pre-wired packs.

RESTRICTION OF HAZARDOUS SUBSTANCES (RoHS) - A European directive, though widely adopted worldwide, that aims to reduce the use of hazardous substances in electrical and electronics equipment.

ROLLED COPPER - A type of copper, made very thin by processing between heavy rollers, extensively used to produce flexible PCBs.

SHADOWING - The blocking of the solder wave from small components by larger components or through hole component pins.

SIP-TYPE PACKAGES - IC packages that have one row of connecting pins.

SMALL OUTLINE TRANSISTORS (SOT) - A discrete semiconductor package having two gull wing leads on one side and one on the other side.
SOLDER BRIDGING - The merging of two solder joints that form an unintended connection between the two.

SOLDER FILLET - A general term used to describe the contour of the solder joints formed between the component termination and the PCB land pattern after soldering.

SOLDER MASK - A coating of material used to protect or mask conductive traces or areas of a PCB against solder bridging.

SOLDER PASTE - A combination of minute spherical solder particles, flux, solvent and a suspension agent which is used in reflow soldering.

SOLDER SIDE (BOTTOM) - A term used to describe the soldered side of a PCB using through hole technology.

SURFACE MOUNT DEVICE (SMD) - A device that is designed for placement and soldering onto pads on the surface of a substrate.

SURFACE MOUNT TECHNOLOGY (SMT) - The technology of assembling PCBs and hybrid circuits where components are mounted onto pads on the surface of the substrate.

SMALL OUTLINE INTEGRATED CIRCUIT – (SOIC) - An integrated circuit package having two parallel rows of gull wing leads.

T

TEARDROP PADS - Added copper/metal to a pad in order to reduce the mechanical and thermal stresses.

TEST POINT - A via or a pad with its own reference designator for probing and testing the nodes on a PCB.

THERMAL RELIEF - A technique used with vias and holes to maintain process temperature to prevent poor hole filling and cold solder joints.

TOMBSTONE (DRAWBRIDGE) - The condition which exists when one end of a chip component is pulled off the solder pad resulting in an open circuit.

TOOLING HOLES - A general term used for holes or slots in PCBs or blank material to aid in the manufacturing process.

TRACE - A conductive path or line. See also “Land” and “Pad.”

V

VIA - A plated through-hole used as a through connection for conductors from the component side to solder side of the board or an outer layer to an inner layer.

W

WAVE SOLDER - The soldering of an assembly by passing the surface mount components, mounted on the solder side of the board, over an adhesive and then over a molten wave of solder.

Z

ZIF SOCKETS - A zero-insertion-force socket for mounting electronic devices that is designed not to stress or damage them during insertion.
Works Cited


ABOUT THE AUTHOR

David Marrakchi
Altium
Sr. Technical Marketing Engineer
San Diego, CA

David currently serves as a Senior Technical Marketing Engineer at Altium and is responsible for managing the development of technical marketing materials for all Altium products. He also works closely with the Altium marketing, sales, and customer support teams to define product strategies including branding, positioning, and messaging.

David brings over 15 years of experience in the EDA industry to the Altium team, and he holds an MBA from Colorado State University and a B.S. in Electronics Engineering from DeVry Technical Institute.
Get Your Designs Right the First Time, Every Time

Altium has been serving the ECAD community for over 30 years, with 90k+ users worldwide, offering various solutions for different user requirements.

**CircuitStudio**
Affordable PCB design tool with intuitive schematic, routing, and simulation capability.

**SOLIDWORKS PCB**
Mechanical and PCB technology integrated within SolidWorks to support multidiscipline design and mechatronics.

**Altium Designer**
Advanced, all-in-one PCB design environment offering integration with the mechanical worlds of MCAD and PDM.

For more information on Altium solutions, visit [www.altium.com](http://www.altium.com).