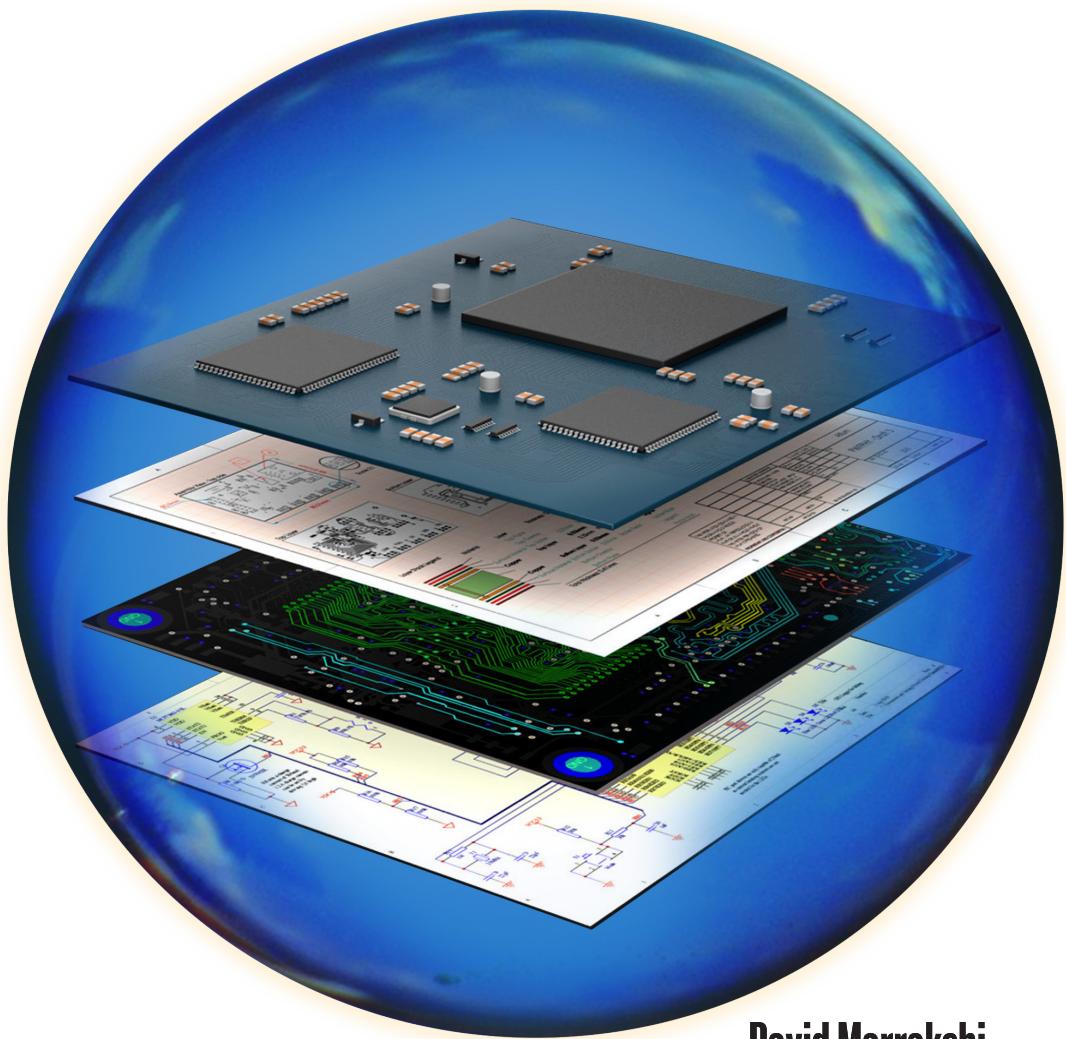


THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™

Design for Manufacturing (DFM)



David Marrakchi
Altium

- Peer Reviewer -

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***.

- Foreword -

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

The Printed Circuit Designer's Guide to...™ Design for Manufacturing (DFM)

By: David Marrakchi, Altium



© 2017 BR Publishing, Inc.
All rights reserved.

BR Publishing, Inc.
dba: I-Connect007
PO Box 50
Seaside, OR 97138-0050

Revised February 13, 2017

ISBN: 978-0-9796189-3-2



Visit I-007eBooks.com for more books in this series.



CONTENTS

THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™ DFM

PAGE

1 Introduction

◆ SECTION 1: Design Guidelines for Successful Manufacturing

PAGE

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

PAGE

25 Chapter 6

Documenting Your PCB for Fabrication

32 Chapter 7

Documenting Your Master Drawing

46 Chapter 8

Documenting Your PCB for Assembly

PAGE

50 Conclusion

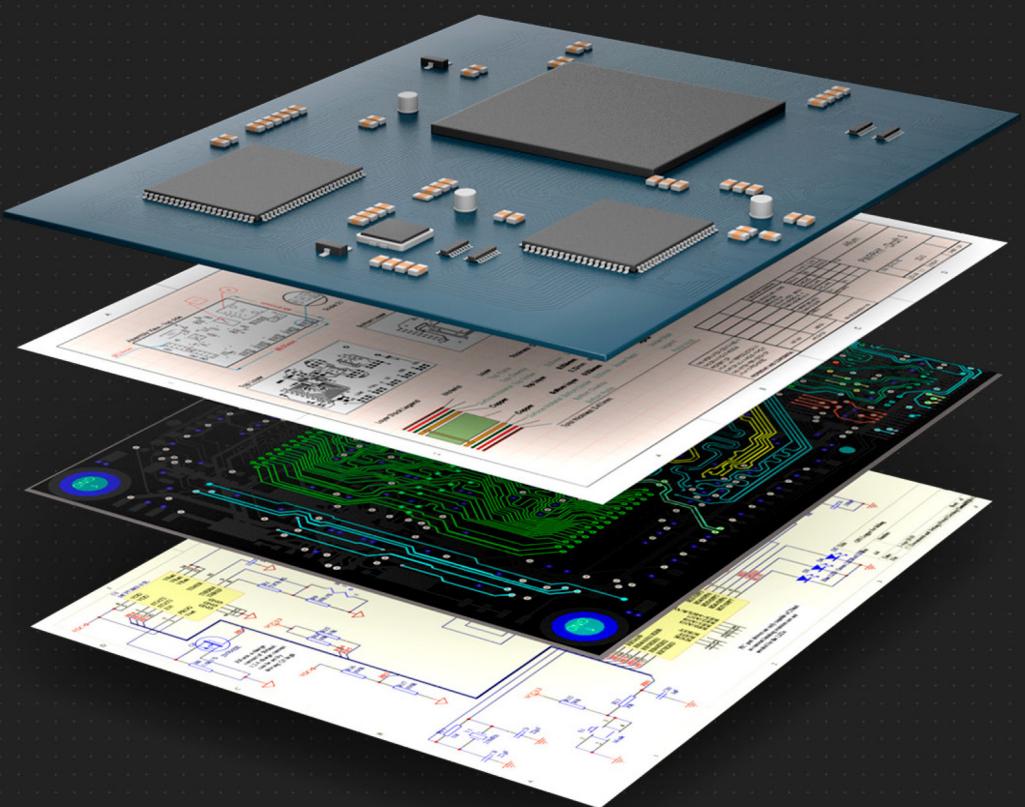
52 Glossary

56 Works Cited

58 About the Author

Altium

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.

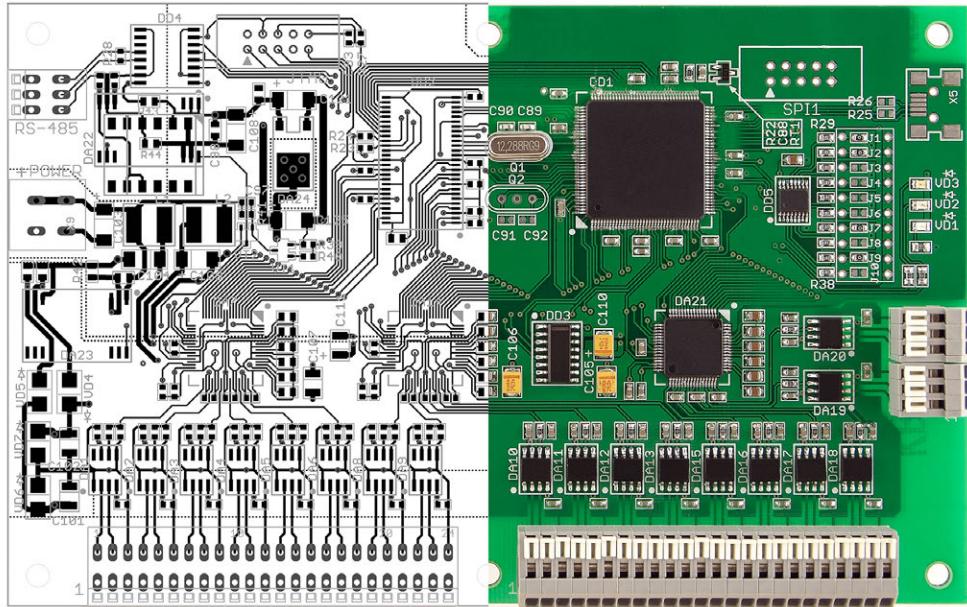
Chapter 7: Assembling the master drawing of your PCB to accurately portray all of the fine details needed to manufacture a board.

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



- Chapter 1 -

A Brief Overview of the

PCB Manufacturing Process

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.

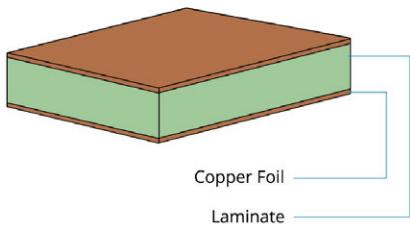
Making Manufacturing-Conscious Design Decisions

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.

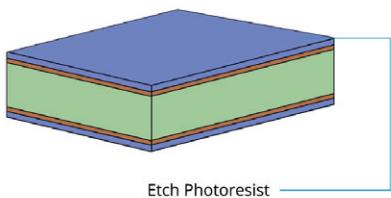
Figure 1

Standard PCB Manufacturing Process

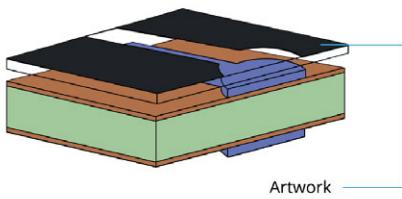
- 1 - Data transfer from customer
- 2 - Data prep
- 3 - Cores/laminate



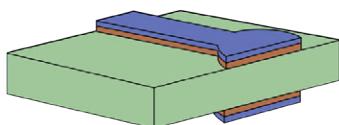
- 4 - Dry film resist coating



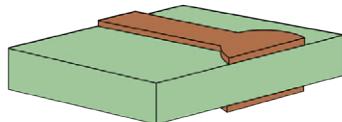
- 5 - Place artwork
- 6 - Expose panels to ultraviolet light
- 7 - Develop panels (resist removal)



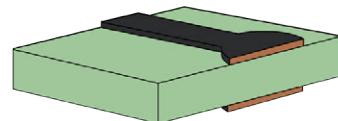
- 8 - Etch



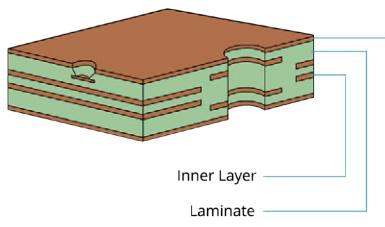
- 9 - Strip resist



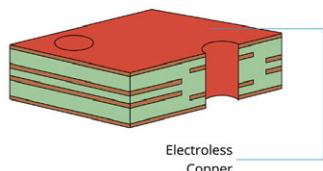
- 10 - Oxide coating



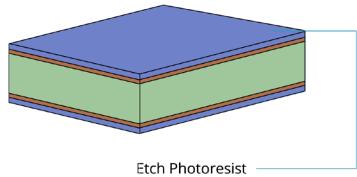
- 11 - Multilayer lamination
- 12 - Primary drilling



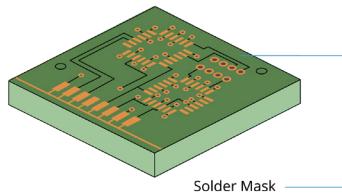
- 13 - Deburr and clean
- 14 - Desmear
- 15 - Copper deposition



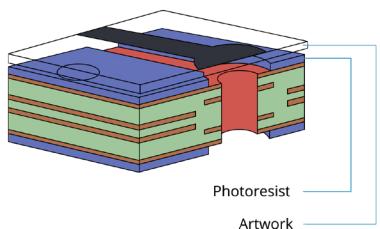
16 - Dry film photoresist coat



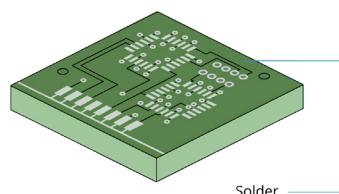
21 - Solder mask and cure



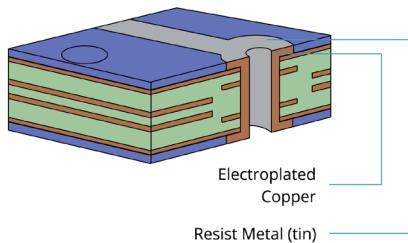
17 - Expose and develop



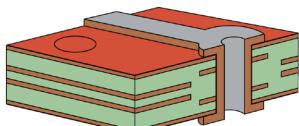
22 - Hot air solder leveling
(most common PCB surface finish)



18 - Copper pattern plate
(electroplating)

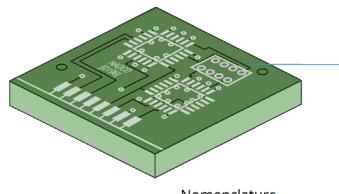


19 - Strip resist

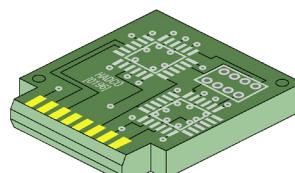


23 - Surface finishes

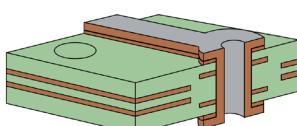
24 - Legend and cure



25 - Fabrication and routing



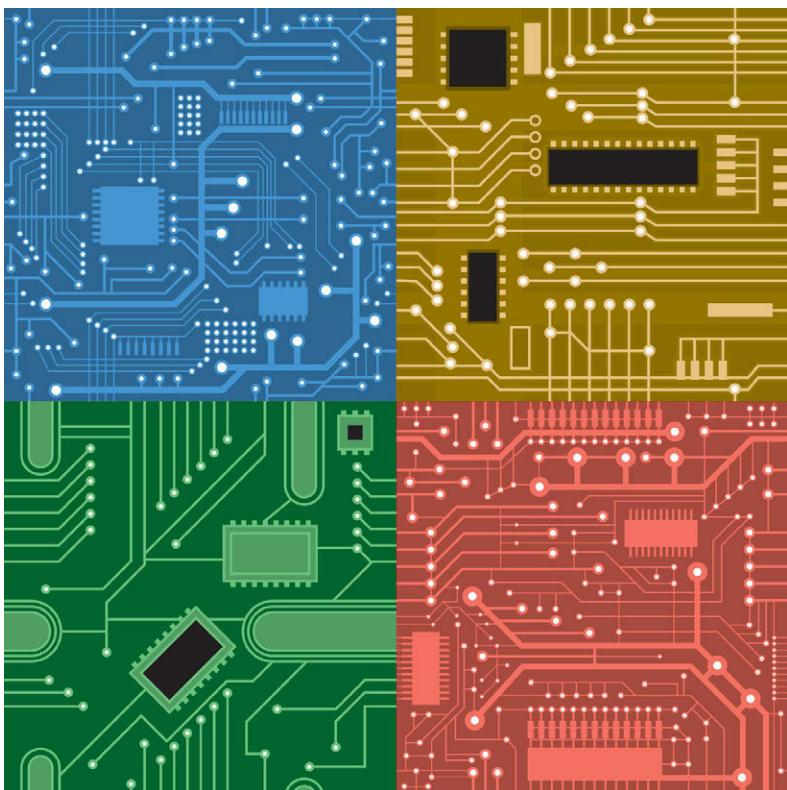
20 - Etch



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. [\[2-1\]](#)

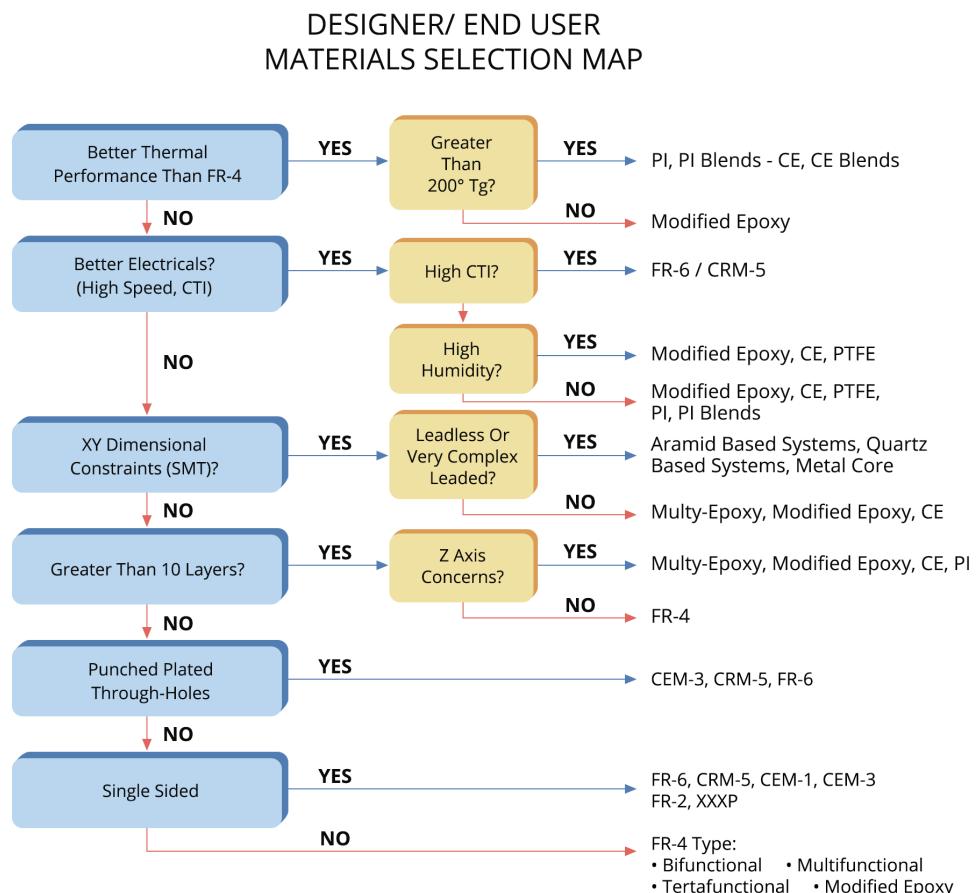


Figure 2 - Designer/End-user materials selection map. [\[2-1\]](#)

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.

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

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 [2-5] formula to easily compute the resistivity in your copper traces:

$$R = \rho * L / A$$

where:

R is the end-to-end track resistance in Ohms

ρ 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.

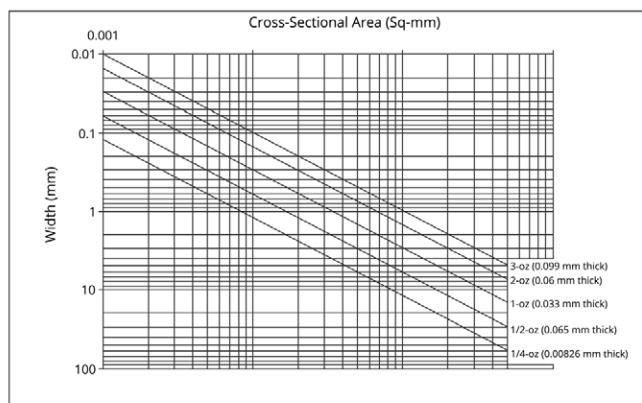
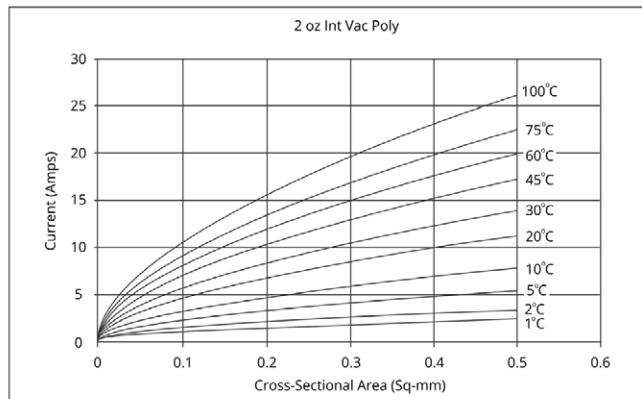


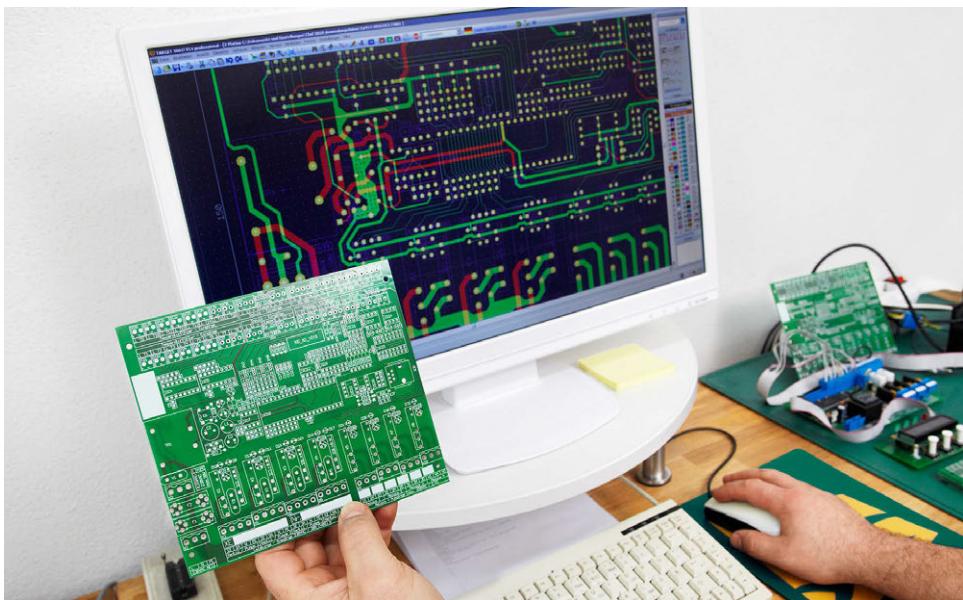
Figure 4 - Encapsulated conductor widths. [2-6]

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

Refer to [IPC-2612](#) [3-1] for a list of industry-standard 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:

Drill Number	Holes size	Finished Hole Size
70	.028"	.025"
65	.035"	.032"
58	.042"	.039"
55	.052"	.049"
53	.0595"	.056"
44	.086"	.083"
1/8"	.125"	.122"
24	.152"	.149"

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:

$$\text{Annular ring width} = (\text{diameter of the pad} - \text{diameter of the hole}) / 2$$

Exposed Vias

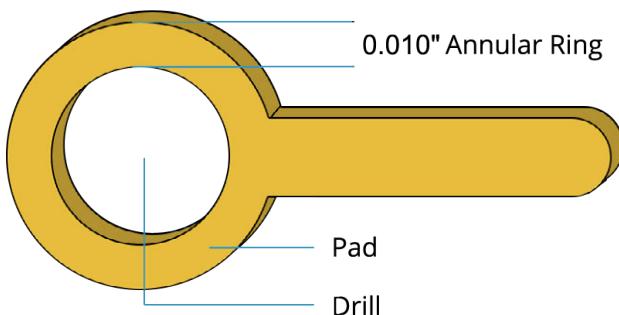


Figure 6 - Recommended annular ring width.

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:

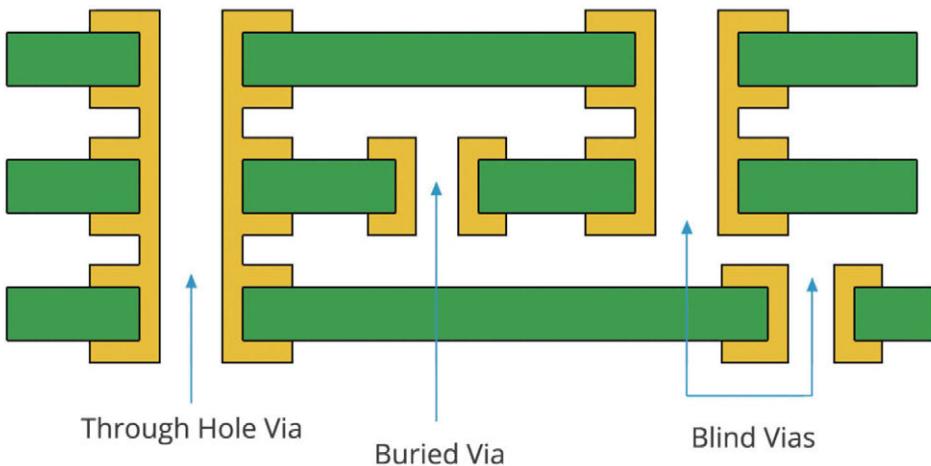


Figure 7 - Blind and buried vias.

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:

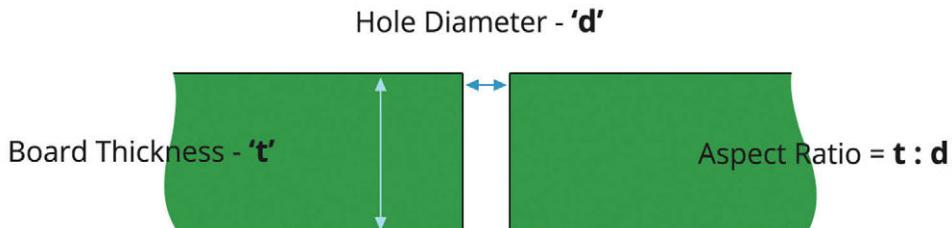


Figure 8 - Determining an aspect ratio for a PCB.

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:

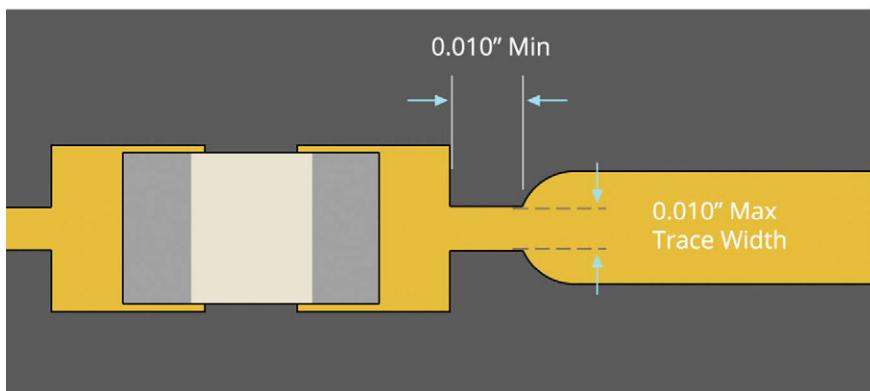


Figure 9a - Connecting large traces to component lands (good design).

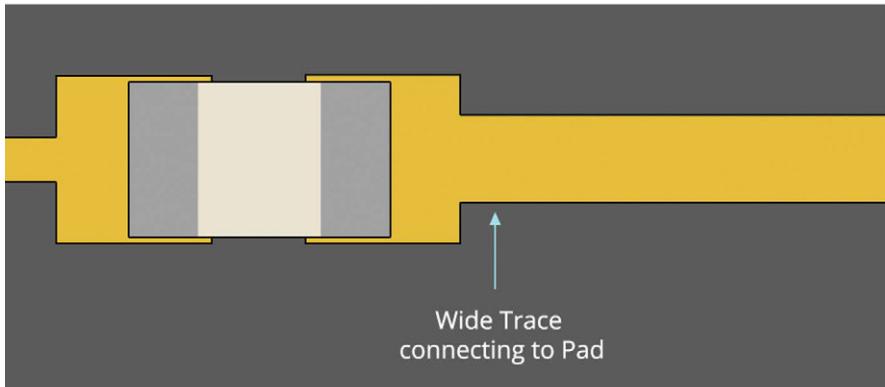


Figure 9b - Connecting large traces to component lands (bad design).

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:

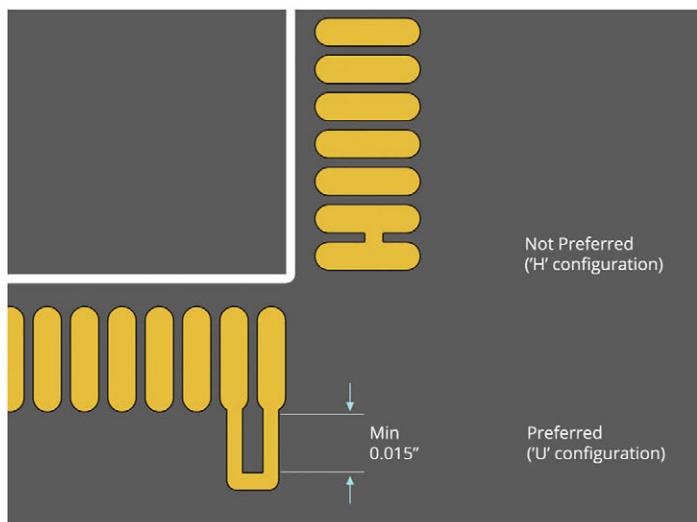


Figure 10 - "U" configuration for routing leaded SMT components.

- Chapter 4 -

Placing and Orienting Your Components



With your preferred component types established, it is now time to decide 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

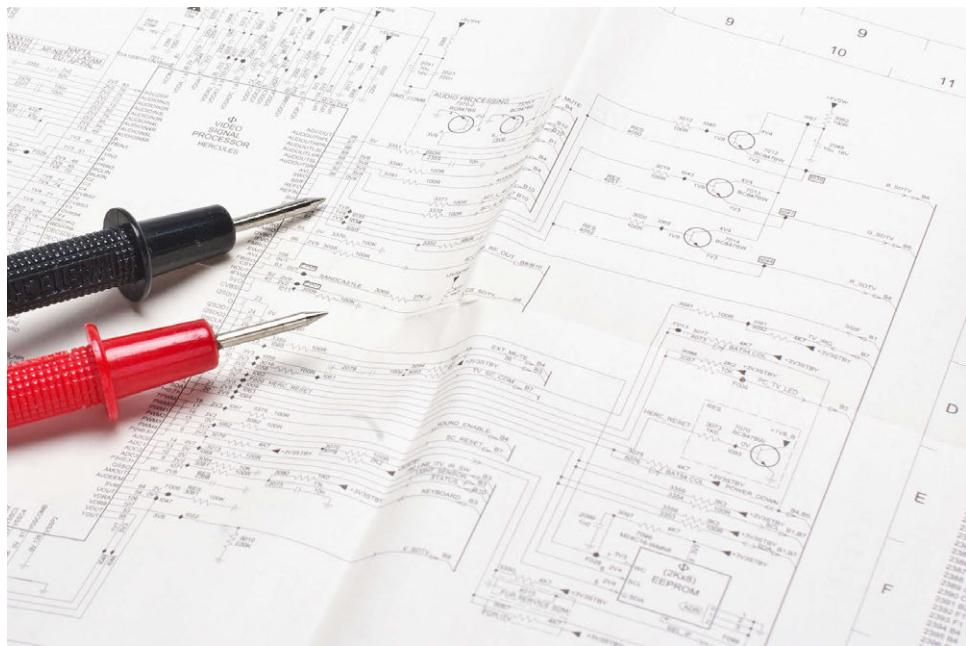
Before going into the specifics of component placement and orientation, there are several general guidelines to keep in mind:

- 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 assembly 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.

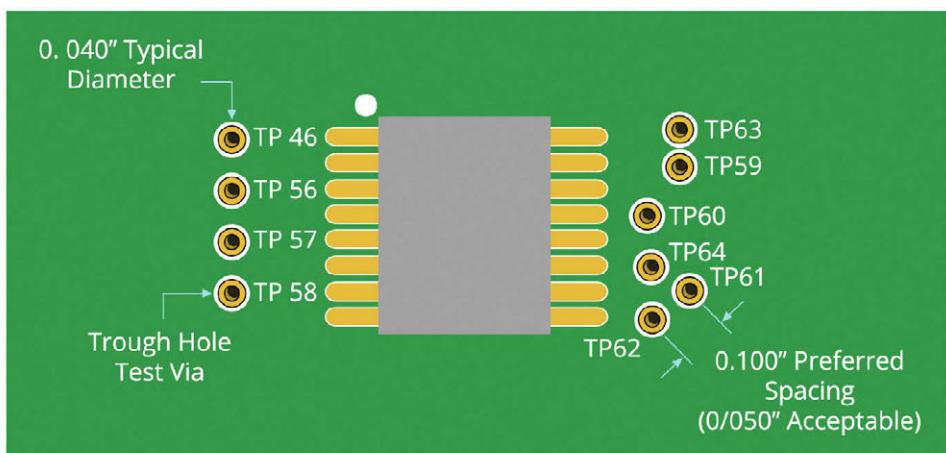


Figure 11 - Through-hole test via.

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.

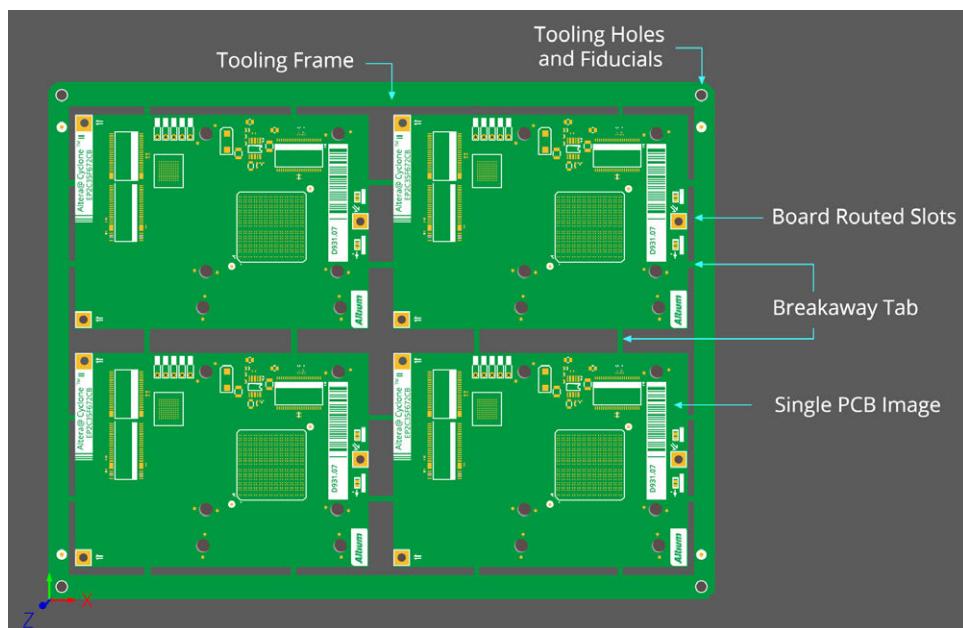


Figure 12 - Four rectangular circuits in a single panel with tooling holes & breakout tabs.

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



- Chapter 6 -

Documenting Your PCB for Fabrication

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.

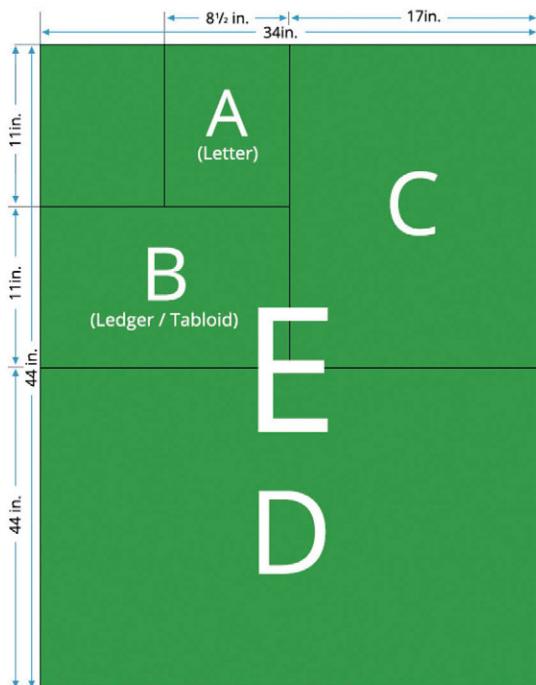


Figure 13 - Standard drawing sizes for PCB documentation. ^[6-2]

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.

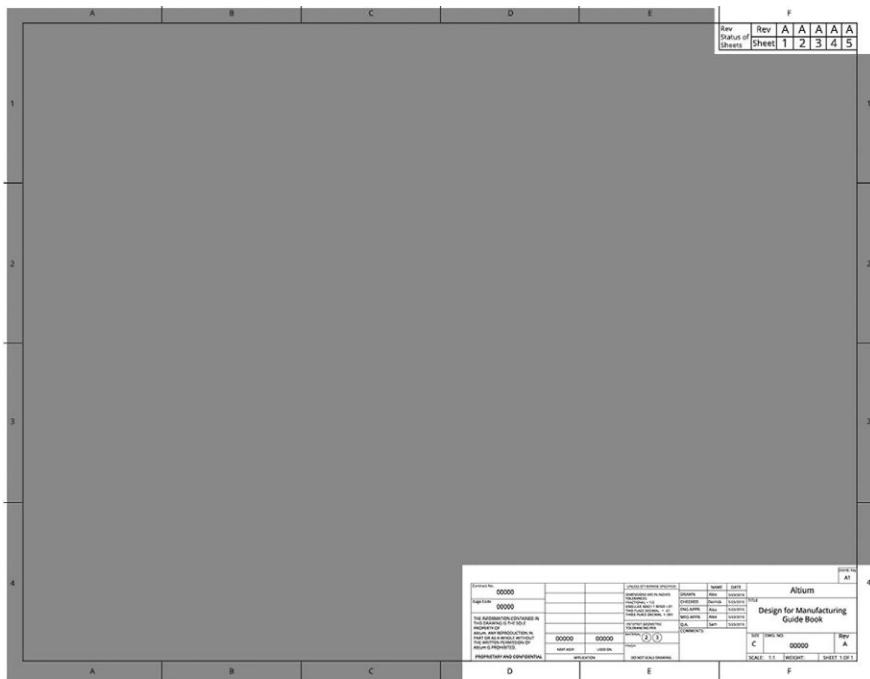


Figure 14 - Blank PCB drawing space with highlighted blocks.

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.

Contract No. 00000										Distrib. Key A1
Cage Code 00000										Altium
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF Altium. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF Altium IS PROHIBITED.										Design for Manufacturing Guide Book
PROPRIETARY AND CONFIDENTIAL										
UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL + 1/16 ANGULAR: MAC + 1 BEND +.01 TWO PLACE DECIMAL +.01 THREE PLACE DECIMAL +.001										
DRAWN Alex 5/23/2016										TITLE
CHECKED Derrick 5/23/2016										
ENG APPR. Alex 5/23/2016										
MFG APPR. Alex 5/23/2016										
Q.A. Sam 5/23/2016										
COMMENTS:										
SIZE DWG. NO. C 00000 Rev A										
SCALE: 1:1 WEIGHT: SHEET 1 OF 1										

Figure 15 - Title and subtitle block.

Drawing Number (DWG. NO.)

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

Contract No. 00000										Distrib. Key A1
Cage Code 00000										Altium
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF Altium. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF Altium IS PROHIBITED.										Design for Manufacturing Guide Book
PROPRIETARY AND CONFIDENTIAL										
UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL + 1/16 ANGULAR: MAC + 1 BEND +.01 TWO PLACE DECIMAL +.01 THREE PLACE DECIMAL +.001										
DRAWN Alex 5/23/2016										TITLE
CHECKED Derrick 5/23/2016										
ENG APPR. Alex 5/23/2016										
MFG APPR. Alex 5/23/2016										
Q.A. Sam 5/23/2016										
COMMENTS:										
SIZE DWG. NO. C 00000 Rev A										
SCALE: 1:1 WEIGHT: SHEET 1 OF 1										

Figure 16 - Drawing number block.

Revision Block

The revision block is used to keep track of the project revision and can be seen in Figure 17.

Contract No. 00000										Distrib. Key A1
Cage Code 00000										Altium
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF Altium. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF Altium IS PROHIBITED.										Design for Manufacturing Guide Book
PROPRIETARY AND CONFIDENTIAL										
UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL + 1/16 ANGULAR: MAC + 1 BEND +.01 TWO PLACE DECIMAL +.01 THREE PLACE DECIMAL +.001										
DRAWN Alex 5/23/2016										TITLE
CHECKED Derrick 5/23/2016										
ENG APPR. Alex 5/23/2016										
MFG APPR. Alex 5/23/2016										
Q.A. Sam 5/23/2016										
COMMENTS:										
SIZE DWG. NO. C 00000 Rev A										
SCALE: 1:1 WEIGHT: SHEET 1 OF 1										

Figure 17 - Revision block.

Material Block

The material block contains numbers corresponding to the appropriate notes, specifying the materials being used.

Contract No. 00000		UNLESS OTHERWISE SPECIFIED:		NAME DRAWN	DATE 5/23/2015	Altium Design for Manufacturing Guide Book		
Cage Code 00000		DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL: 1/12 ANGLE: M \pm 1° BEND: +0.01 TWO PLACE DECIMAL: +01 THREE PLACE DECIMAL: +001		CHECKED Derrick	5/23/2016			
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF Altium. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF Altium IS PROHIBITED.		INTERPRET GEOMETRIC TOLERANCING PER:		ENG APPR. Alex	5/23/2016			
00000		MATERIAL (2) (3)	FINISH	MFG APPR. Alex	5/23/2016			
NEXT ASSY		USED ON	DO NOT SCALE DRAWING	Q.A. Sam	5/23/2016			
PROPRIETARY AND CONFIDENTIAL		APPLICATION		COMMENTS:		SIZE C	DWG. NO. 00000	Rev A
						SCALE: 1:1	WEIGHT:	SHEET 1 OF 1

Figure 18 - Material block.

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.

	F					
Rev Status of Sheets	Rev	A	A	A	A	A
	Sheet	1	2	3	4	5

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.

DRAW	NAME	DATE	Rev.	Cage Code
CHECKED	Sam	5/23/2016	A	00000 DWG. NO.
ENG APPR.	Dan	5/23/2016		00000
Q.A.	Jeff	5/23/2016	Size	Scale
			C	Sheet 1:1 6 of 9

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:

APPROVALS	DATE	PROJECT	Altium		2175 SLK AVENUE SUITE 200 CARLSBAD, CA 92008 USA
ENG: -----	-----	TITLE	*		
DSN: -----	-----		*		
CHK: -----	-----		*		
REFERENCE DOCUMENTS		SIZE	CAGE CODE	DWG NO.	REV
BOM: <BOM DOC NO>		B	0ZL62	<SCH DWG NO>	A
ASSY DWG: <ASSY DWG NO>		SCALE:	Scale	FILE NAME	Top Sheet. SchDoc
FAB DWG: <FAB DWG NO>					SHEET 1 OF 2
<PCB DWG NO>					

Figure 21 - Schematic title block.

Reference Documents Block

The reference documents block lists the required project production documentation.

APPROVALS	DATE	PROJECT	Altium		2175 SALK AVENUE SUITE 200 CARLSBAD, CA 92008 USA
ENG: -----	-----	TITLE	*		
DSN: -----	-----		*		
CHK: -----	-----		*		
REFERECE DOCUMENTS		SIZE	CAGE CODE	DWG NO.	REV
BOM: <BOM DOG NO>		B	0ZL62	<SCH DWG NO>	A
ASSY DWG: <ASSY DWG NO>		SCALE:	Scale	FILE NAME	Top Sheet. SchDoc
FAB DWG: <FAB DWG NO>					SHEET 1 OF 2
PCB DWG: <PCB DWG NO>					

Figure 22 - Reference documents block.

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.

APPROVALS	DATE	PROJECT		2175 SALK AVENUE SUITE 200 CARLSBAD, CA 92008 Altium ™		
ENG: -----	--/--/--	*				
DSN: -----	--/--/--	TITLE *				
CHK: -----	--/--/--	*				
REFERENCE DOCUMENTS						
BOM: <BOM DOG NO>		SIZE	CAGE CODE	DWG NO.	<SCH DWG NO>	
ASSY DWG: <ASSY DWG NO>		B	0ZL62			
FAB DWG: <FAB DWG NO>		SCALE:	Scale	FILE NAME	Top Sheet. SchDoc	
PCB DWG: <PCB DWG NO>					SHEET	1 OF 2

Figure 23 - Project block.

File Name

The file name refers to the saved filename including extension.

APPROVALS	DATE	PROJECT		2175 SALK AVENUE SUITE 200 CARLSBAD, CA 92008 Altium ™		
ENG: -----	--/--/--	*				
DSN: -----	--/--/--	TITLE *				
CHK: -----	--/--/--	*				
REFERENCE DOCUMENTS						
BOM: <BOM DOG NO>		SIZE	CAGE CODE	DWG NO.	<SCH DWG NO>	
ASSY DWG: <ASSY DWG NO>		B	0ZL62			
FAB DWG: <FAB DWG NO>		SCALE:	Scale	FILE NAME	Top Sheet. SchDoc	
PCB DWG: <PCB DWG NO>					SHEET	1 OF 2

Figure 24 - File name block.

Company Name and Address

This area is for your company's name and mailing address.

APPROVALS	DATE	PROJECT *		2175 SALK AVENUE SUITE 200 CARLSBAD, CA 92008 USA	
ENG: -----	--/--/--	TITLE *		Altium TM	
DSN: -----	--/--/--	*			
CHK: -----	--/--/--	*			
REFERENCE DOCUMENTS					
BOM: <BOM DOG NO>					
ASSY DWG: <ASSY DWG NO>	SIZE B	CAGE CODE 0ZL62	DWG NO. <SCH DWG NO>	REV A	
FAB DWG: <FAB DWG NO>	SCALE: Scale	FILE NAME Top Sheet. SchDoc	SHEET 1 OF 2		
PCB DWG: <PCB DWG NO>					

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.

	A	B	C	D	E	F			
					Rev of Sheet	Rev of Sheet			
					1	2	3	4	5
Notes:									
1.	Fabricate boards in accordance with IPC-AB-214, Class 3. Printed boards must meet quality compliance testing and inspection as specified.								
2.	Material in accordance with MIL-STD-1344A, commercial sheet, Type QF glass cloth base, flame retardant (measuring UL 94-V1 or earlier), Tg rating: 140°C to 160°C.								
3.	Material in accordance with MIL-STD-1344A, 12 plies woven, type QF glass cloth base, flame retardant (measuring UL 94-V1 or earlier), Tg rating: 140°C to 160°C.								
4.	Boards fabricated must use standard resistor cage codes, ID and U marking its primary size and secondary size where indicated. Marking must be visible after soldering.								
5.	Conformal coating must be applied over entire copper (MIL-C-46052, using Type X000, environmental grade only). Min. Ag to primary and secondary sizes in accordance with IPC-9704D, class S, Class 3. Use appropriate solder mask thickness for each side of boards. Purchasing must be informed.								
6.	Solder mask margin/tolerance must not exceed ±0.020 in. Solder mask overlaps permitted on primary lands (min.), and shall not exceed ±0.020 in. No overlaps permitted on secondary lands.								
7.	Solder mask thickness: 0.020 in. min./0.030 in. max. Solder mask material to be light green in color and highly transparent.								
8.	Drill holes must be round, off-center, and not eccentric. Hole spacing may vary within 25A in. (±0.005 in.) max. hole-to-hole position.								
9.	All holes are printed/milled/etched unless otherwise. Minimum feature printing in printed holes to be 0.07 in. Copper plating in terms of hole and not plug hole.								
10.	Minimum annular ring: 0.020 in. minimum / 0.030 in. maximum.								
11.	A certificate of compliance with IPC-AB-214, Class 3, shall be submitted with each lot.								
12.	Through hole of holes as required. Converage and overspray shall meet the requirements of UL-D-021. Ternary holes are not applicable.								
13.	Buried lands and vias to be thick coated. Thread plated "T" holes may be reduced to the point of closure.								
14.	All exposed surface lands and vias to be thicker coated.								
15.	Printed the width tolerance ±0.020 in. in terms of pitch.								
16.	Minimum finished spacing ≥ 0.020 in.								
17.	Dimensions are off-centering and spacing, and are basic unless otherwise indicated.								
18.	Bore and hole shall not exceed ±0.020 in. per hole measured in accordance with IPC-TR-852, Method A, 2.2.2.								
19.	All through-holes (except via) may be rounded by the PCB fabricator to compensate for manufacturing process tolerances. Addition of fillets are not recommended.								
20.	Land/noise factor allowance ±0.020 in.								
21.	Showcase primary size, and secondary size (bottom) of board using white epoxy base per MIL-STD-3285, Type I, and appropriate artwork.								
22.	Test coupon thickness: 0.060 in., core/copper or pre-preg, unless otherwise specified. Any test coupons required shall be marked with the date and lot number. Each test coupon shall be tested and satisfied with each lot. A report of the test results shall be provided to the customer.								
23.	Test boards (except Test boards shall be electrostatic tested using ESD generated test for static. This information to be supplied in IPC-D-325A prime test report) shall follow the guidelines given in IPC-TR-852, guidance and reference for electrostatic testing of uncoated printed boards.								
Layer Stack Legend:									
	Material	Layer	Thickness	Dielectric Material	Type	Gerbler			
		Top Ply				Print Trace	GPO		
		Top Dielectric				Legend	GPO		
		Prepreg				Signal	G3		
		Midlayer 1	0.040mm			Dielectric			
		Core	0.25mm	Pb-A		Dielectric			
		Copper	0.030mm			Signal	G2		
		Midlayer 2	0.040mm			Dielectric			
		Prepreg	0.120mm	Pb-A		Dielectric			
		Copper	0.030mm			Internal Plane	GPO		
		Core	0.25mm	Pb-A		Dielectric			
		Copper	0.030mm			Signal	G3		
		Midlayer 3	0.040mm			Dielectric			
		Prepreg	0.120mm	Pb-A		Dielectric			
		Copper	0.030mm			Internal Plane	GPO		
		Core	0.25mm	Pb-A		Dielectric			
		Copper	0.030mm			Signal	G4		
		Midlayer 4	0.040mm			Dielectric			
		Prepreg	0.120mm	Pb-A		Dielectric			
		Copper	0.030mm			Bottom Layer	G4		
		Bottom Material	0.25mm			Bottom Dielectric			
						Bottom Trace	GPO		
						Total Thickness	1.90mm		
Dot Types:									
	Dot Type	Printed Hole Size	Pitch	Hole Diameter					
	1	0.2mm	0.4mm	0.2mm					
	2	0.2mm	0.4mm	0.2mm					
	3	0.2mm	0.4mm	0.2mm					
	4	0.2mm	0.4mm	0.2mm					
	5	0.2mm	0.4mm	0.2mm					
	6	0.2mm	0.4mm	0.2mm					
	7	0.2mm	0.4mm	0.2mm					
	8	0.2mm	0.4mm	0.2mm					
	9	0.2mm	0.4mm	0.2mm					
	10	0.2mm	0.4mm	0.2mm					
	11	0.2mm	0.4mm	0.2mm					
	12	0.2mm	0.4mm	0.2mm					
	13	0.2mm	0.4mm	0.2mm					
	14	0.2mm	0.4mm	0.2mm					
	15	0.2mm	0.4mm	0.2mm					
	16	0.2mm	0.4mm	0.2mm					
	17	0.2mm	0.4mm	0.2mm					
	18	0.2mm	0.4mm	0.2mm					
	19	0.2mm	0.4mm	0.2mm					
	20	0.2mm	0.4mm	0.2mm					
	21	0.2mm	0.4mm	0.2mm					
	22	0.2mm	0.4mm	0.2mm					
	23	0.2mm	0.4mm	0.2mm					
	24	0.2mm	0.4mm	0.2mm					
	25	0.2mm	0.4mm	0.2mm					
	26	0.2mm	0.4mm	0.2mm					
	27	0.2mm	0.4mm	0.2mm					
	28	0.2mm	0.4mm	0.2mm					
	29	0.2mm	0.4mm	0.2mm					
	30	0.2mm	0.4mm	0.2mm					
	31	0.2mm	0.4mm	0.2mm					
	32	0.2mm	0.4mm	0.2mm					
	33	0.2mm	0.4mm	0.2mm					
	34	0.2mm	0.4mm	0.2mm					
	35	0.2mm	0.4mm	0.2mm					
	36	0.2mm	0.4mm	0.2mm					
	37	0.2mm	0.4mm	0.2mm					
	38	0.2mm	0.4mm	0.2mm					
	39	0.2mm	0.4mm	0.2mm					
	40	0.2mm	0.4mm	0.2mm					
	41	0.2mm	0.4mm	0.2mm					
	42	0.2mm	0.4mm	0.2mm					
	43	0.2mm	0.4mm	0.2mm					
	44	0.2mm	0.4mm	0.2mm					
	45	0.2mm	0.4mm	0.2mm					
	46	0.2mm	0.4mm	0.2mm					
	47	0.2mm	0.4mm	0.2mm					
	48	0.2mm	0.4mm	0.2mm					
	49	0.2mm	0.4mm	0.2mm					
	50	0.2mm	0.4mm	0.2mm					
	51	0.2mm	0.4mm	0.2mm					
	52	0.2mm	0.4mm	0.2mm					
	53	0.2mm	0.4mm	0.2mm					
	54	0.2mm	0.4mm	0.2mm					
	55	0.2mm	0.4mm	0.2mm					
	56	0.2mm	0.4mm	0.2mm					
	57	0.2mm	0.4mm	0.2mm					
	58	0.2mm	0.4mm	0.2mm					
	59	0.2mm	0.4mm	0.2mm					
	60	0.2mm	0.4mm	0.2mm					
	61	0.2mm	0.4mm	0.2mm					
	62	0.2mm	0.4mm	0.2mm					
	63	0.2mm	0.4mm	0.2mm					
	64	0.2mm	0.4mm	0.2mm					
	65	0.2mm	0.4mm	0.2mm					
	66	0.2mm	0.4mm	0.2mm					
	67	0.2mm	0.4mm	0.2mm					
	68	0.2mm	0.4mm	0.2mm					
	69	0.2mm	0.4mm	0.2mm					
	70	0.2mm	0.4mm	0.2mm					
	71	0.2mm	0.4mm	0.2mm					
	72	0.2mm	0.4mm	0.2mm					
	73	0.2mm	0.4mm	0.2mm					
	74	0.2mm	0.4mm	0.2mm					
	75	0.2mm	0.4mm	0.2mm					
	76	0.2mm	0.4mm	0.2mm					
	77	0.2mm	0.4mm	0.2mm					
	78	0.2mm	0.4mm	0.2mm					
	79	0.2mm	0.4mm	0.2mm					
	80	0.2mm	0.4mm	0.2mm					
	81	0.2mm	0.4mm	0.2mm					
	82	0.2mm	0.4mm	0.2mm					
	83	0.2mm	0.4mm	0.2mm					
	84	0.2mm	0.4mm	0.2mm					
	85	0.2mm	0.4mm	0.2mm					
	86	0.2mm	0.4mm	0.2mm					
	87	0.2mm	0.4mm	0.2mm					
	88	0.2mm	0.4mm	0.2mm					
	89	0.2mm	0.4mm	0.2mm					
	90	0.2mm	0.4mm	0.2mm					
	91	0.2mm	0.4mm	0.2mm					
	92	0.2mm	0.4mm	0.2mm					
	93	0.2mm	0.4mm	0.2mm					
	94	0.2mm	0.4mm	0.2mm					
	95	0.2mm	0.4mm	0.2mm					
	96	0.2mm	0.4mm	0.2mm					
	97	0.2mm	0.4mm	0.2mm					
	98	0.2mm	0.4mm	0.2mm					
	99	0.2mm	0.4mm	0.2mm					
	100	0.2mm	0.4mm	0.2mm					
	101	0.2mm	0.4mm	0.2mm					
	102	0.2mm	0.4mm	0.2mm					
	103	0.2mm	0.4mm	0.2mm					
	104	0.2mm	0.4mm	0.2mm					
	105	0.2mm	0.4mm	0.2mm					
	106	0.2mm	0.4mm	0.2mm					
	107	0.2mm	0.4mm	0.2mm					
	108	0.2mm	0.4mm	0.2mm					
	109	0.2mm	0.4mm	0.2mm					
	110	0.2mm	0.4mm	0.2mm					
	111	0.2mm	0.4mm	0.2mm					
	112	0.2mm	0.4mm	0.2mm					
	113	0.2mm	0.4mm	0.2mm					
	114	0.2mm	0.4mm	0.2mm					
	115	0.2mm	0.4mm	0.2mm					
	116	0.2mm	0.4mm	0.2mm					
	117	0.2mm	0.4mm	0.2mm					
	118	0.2mm	0.4mm	0					

Board Details

The board details defines board complexity and structure.

Board Type

The board type as mentioned in IPC-2221 [\[7-2\]](#) 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 [\[7-3\]](#) and ASME-Y-14.5. [\[6-2\]](#)

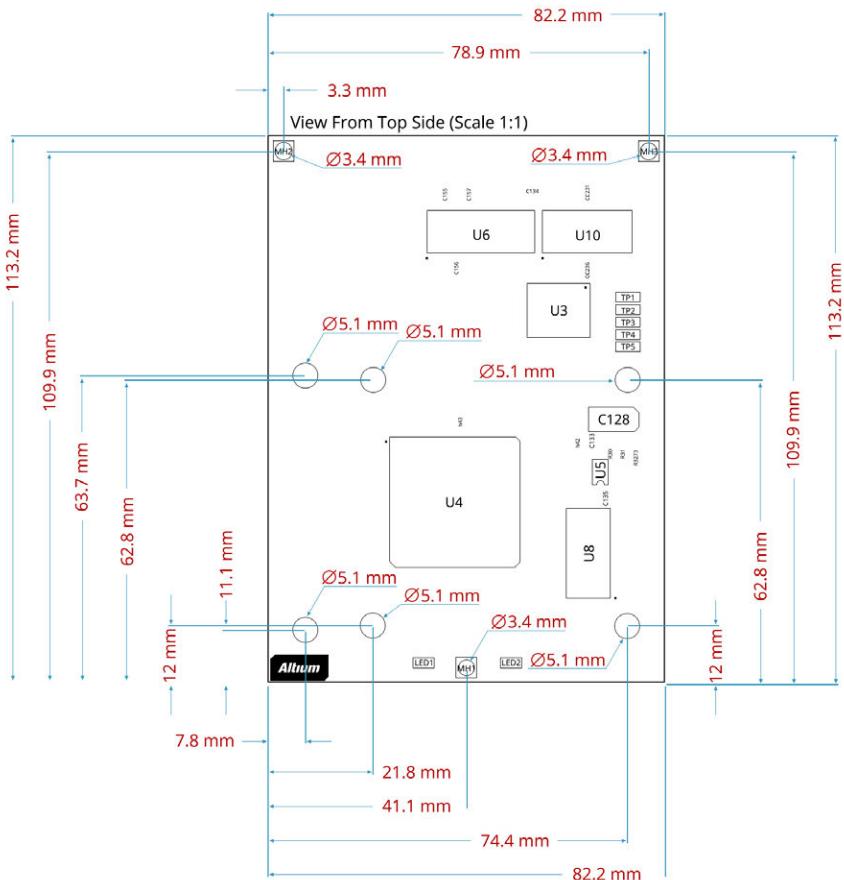


Figure 27 – A dimensioned PCB.

- 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.

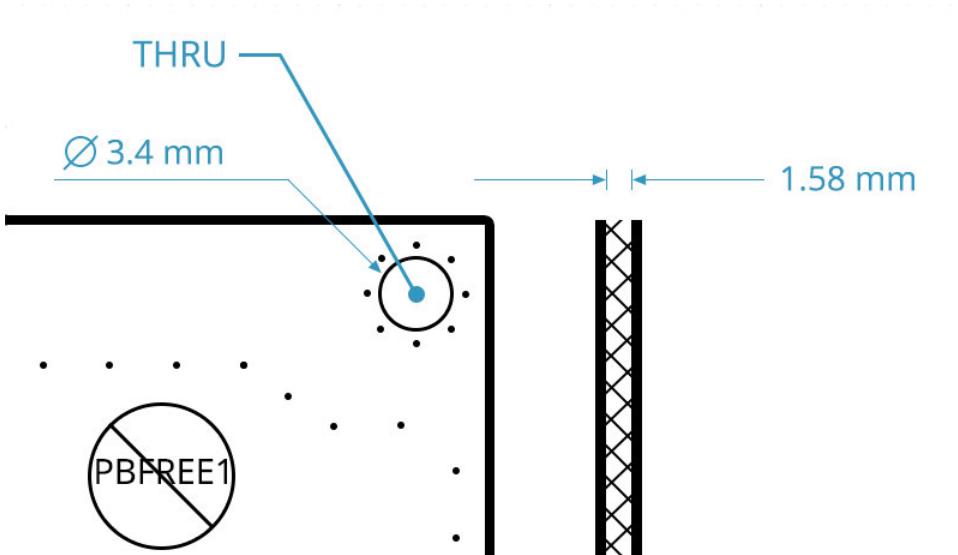


Figure 28 - Dimensioning holes.

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.									
Contract No. 00000 Cage Code 00000 THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF Altium. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF Altium IS PROHIBITED. PROPRIETARY AND CONFIDENTIAL									
UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL: +1/-0 ANGULAR: ±1° LENGTH: .001 INCH ±.01 TWO PLACE DECIMAL ±.01 THREE PLACE DECIMAL ±.001									
DRAWN Alex 5/23/2016 CHECKED Derrick 5/23/2016 ENG APPR. Alex 5/23/2016 MFG APPR. Alex 5/23/2016 Q.A. Sam 5/23/2016 COMMENTS:									
SIZE DWG. NO. Rev C 00000 A SCALE: 1:1 WEIGHT: SHEET 1 OF 1									
MATERIAL (2) (3) FINISH DO NOT SCALE DRAWING									
NEXT ASSY USED ON APPLICATION									

Figure 29 - Callout to notes from title block.

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				
Layer	Material	Thickness	Type	Gerber
Top Paste			Paste Mask	GTP
Top Overlay			Legend	GTO
Top Solder Mask	Solder Resist	0.01 mm	Solder Mask	GTS
Top Layer	Copper	0.04 mm	Signal	GTL
Dielectric 1	FR - 4	1.57 mm	Dielectric	
Bottom Layer	Copper	0.04 mm	Signal	GBL
Bottom Solder Mask	Solder Resist	0.01 mm	Solder Mask	GBS
Bottom Overlay			Legend	GBO
Bottom Paste			Paste Mask	GBP
Total thickness: 1.67 mm				

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 Table

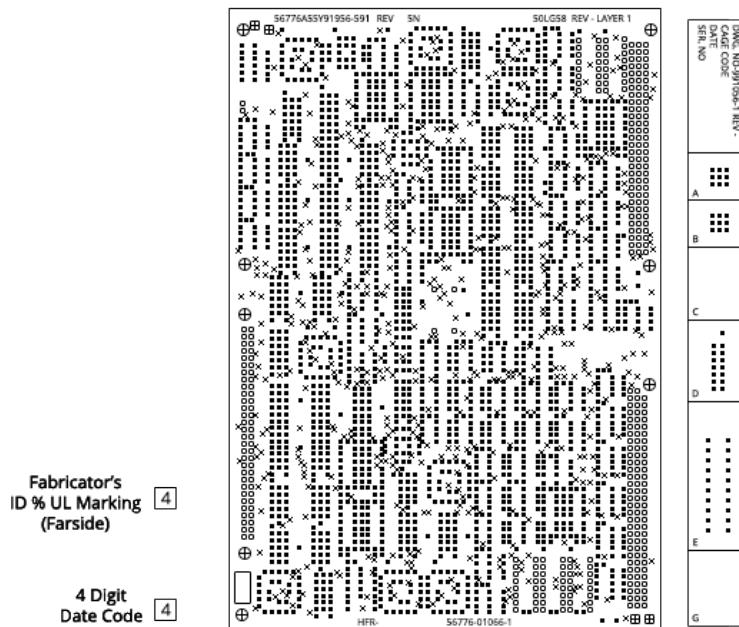
Symbol	Count	Hole Size	Plated	Hole Tolerance
□	3	1.1 mm	Plated	None
○	6	0.75 mm	Plated	None
◇	94	0.9 mm	Plated	None
✗	1	0.9 mm	Plated	None
+/-	58	0.8 mm	Plated	None
◆	3	1 mm	Plated	None
☆	4	3.3 mm	Non-Plated	None
☆	170	0.6 mm	Plated	None
✳	4	2.5 mm	Non-Plated	None
△	1	0.51 mm	Non-Plated	None
344 Total				

Figure 31 - Drill drawing table.

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.

22



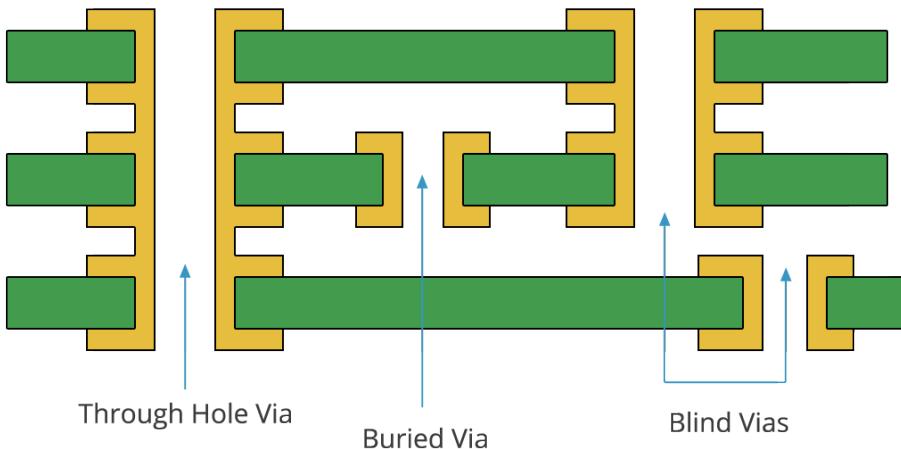
Example 93 32

Year
WeekFabricator
Cage Code

Figure 32 - Drill pattern 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.



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.

Country	Australia	Canada	China	Germany
Country	Italy	Denmark	Norway	Austria
Country	Sweden	Switzerland	USA	Europe
Country	Germany	UK	USA	USA
Country	Japan	Japan	USA	Saudi Arabia
Country	Holland	The Globe	Germany	

Figure 34 - Safety level markings.

Restriction of Hazardous Substances (RoHS)

The Restriction of Hazardous Substances (RoHS) standard is used throughout the world [7-4]. 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 [7-5] 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. [6-1] 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.

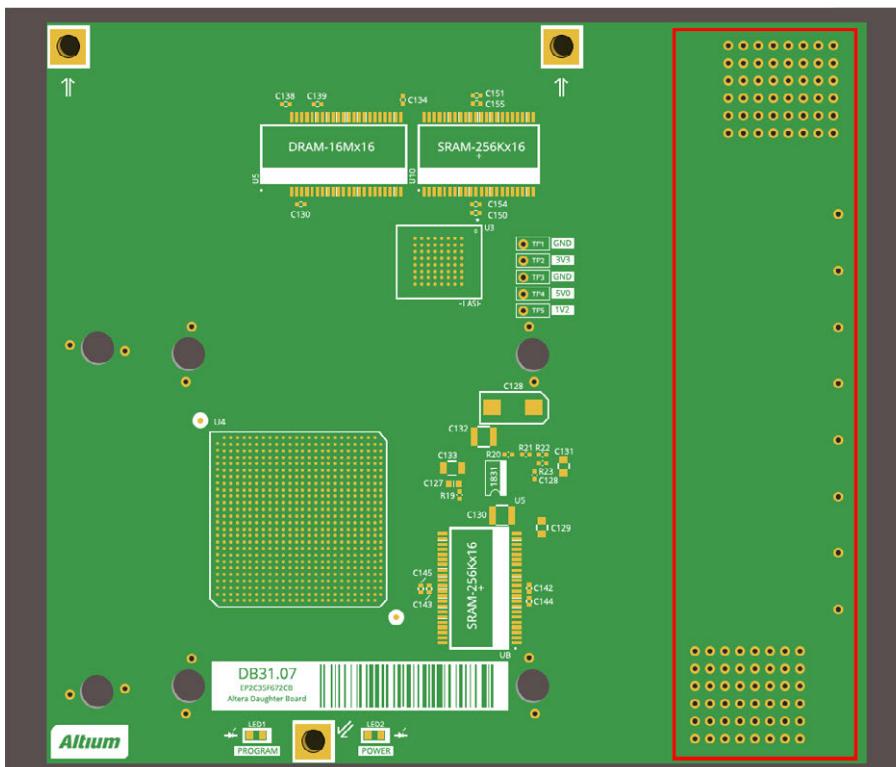


Figure 36 - Quality conformance coupons.

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. [\[7-3\]](#)

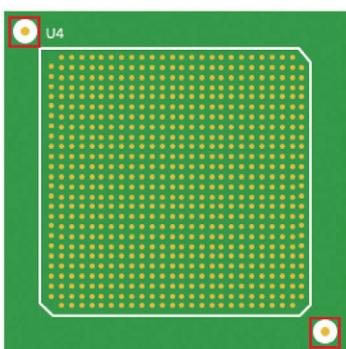


Figure 37
Datum placed at each of two corners of a BGA.

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 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.

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

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:

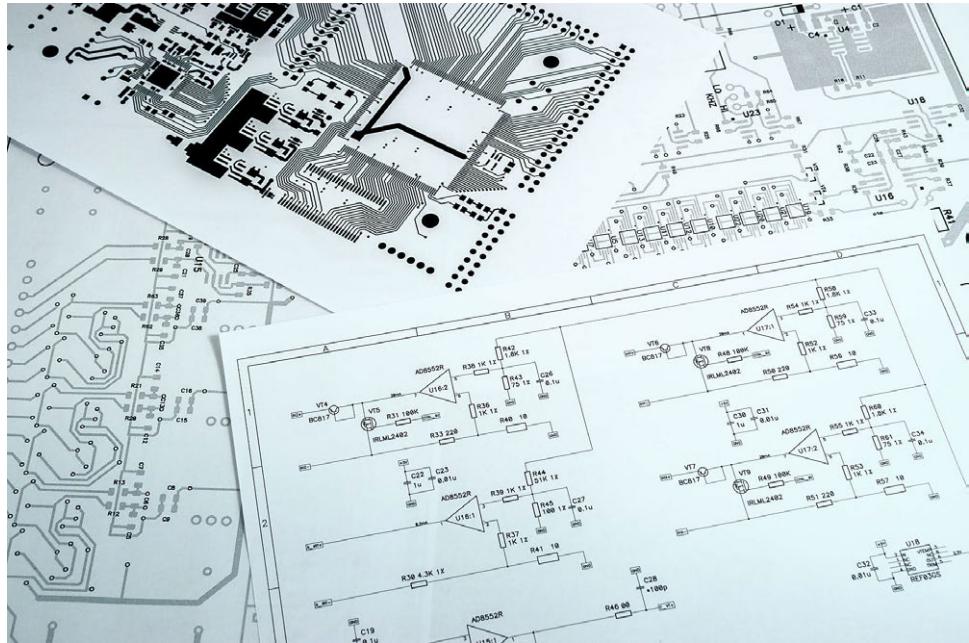
- Finished board specifications and class
- Material requirements
- B-stage material requirements (for multilayer boards)
- Board markings
- Construction
- Solder mask specifications
- Solder mask material & thickness
- Drilled hole requirements
- Copper plating thickness
- Etchback requirements
- Bow 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:

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

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.

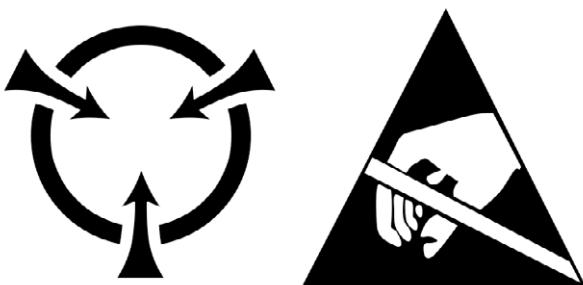


Figure 40 - Electrostatic discharge markings.

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. [6-1](#)

Class 1 & 2 Boards	"ELECTROSTATIC DISCHARGE CONTROL PROGRAM FOR PROTECTION OF ELECTRICAL AND ELECTRONIC PARTS, ASSEMBLIES AND EQUIPMENT SHALL BE IN ACCORDANCE WITH MIL-STD-1686, CLASS ____ AND MIL-HDBK-263."
Class 3 Boards	"ELECTROSTATIC DISCHARGE CONTROL PROGRAM FOR PROTECTION OF ELECTRICAL AND ELECTRONIC PARTS, ASSEMBLIES AND EQUIPMENT SHALL BE IN ACCORDANCE WITH MIL-STD-1686, CLASS 3, AND MIL-HDBK-263"

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.

Glossary

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.

P

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.

R

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.

S

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 a 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

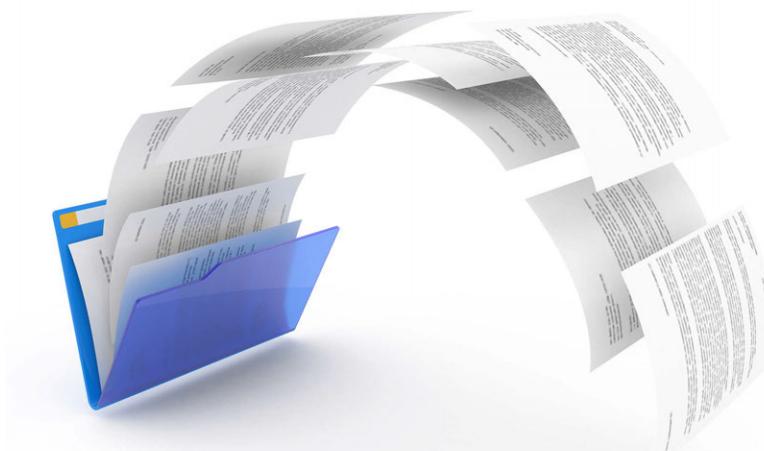
W

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



- [2-1] ["IPC-2222 - Material Selection."](#) 2012. Web.
- [2-2] "Selecting PCB Materials for High-frequency Applications." EDN. N.p., n.d. Web. [<https://goo.gl/C17I9y>]
- [2-5] ["PDN Analyzer."](#) PDN Analyzer. Altium, n.d. Web.
- [2-6] "[IPC-2152](#)", Standard for Determining Current-Carrying Capacity in Printed Board Design. Northbrook, IL: IPC, 2009. Web.
- [3-1] "[IPC-2612](#)." Sectional Requirements for Electronic Diagramming Documentation (Schematic and Logic Descriptions) (n.d.): n. pag. IPC. Web.
- [6-1] [Documentation Requirements for Printed Boards, Assemblies and Support Drawings](#). Lincolnwood, IL: Institute for Interconnecting and Packaging Electronic Circuits, 1995. Print & Web.
- [6-2] ASME Y14.5-2009: Dimensioning and Tolerancing: Engineering Drawing and Related Documentation Practices. New York: ASME, 2009. Print.
- [7-2] Generic Standard on Printed Board Design. Northbrook, IL: IPC, 1998. Print.
- [7-3] Standard Specification: Printed Wiring Board Dimensions and Tolerances, Single and Two Sided Rigid Boards. Evanston, IL: Institute, 1974. Print.
- [7-4] "[RoHS Guide](#)." RoHS Compliance Guide: Regulations, 6 Substances, Exemptions, WEEE. N.p., n.d. Web.
- [7-5] Printed-Wiring Boards. Northbrook, IL: Underwriters Laboratories, 1993. Print.

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.

Altium®



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.



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



Powered by Altium

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



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.