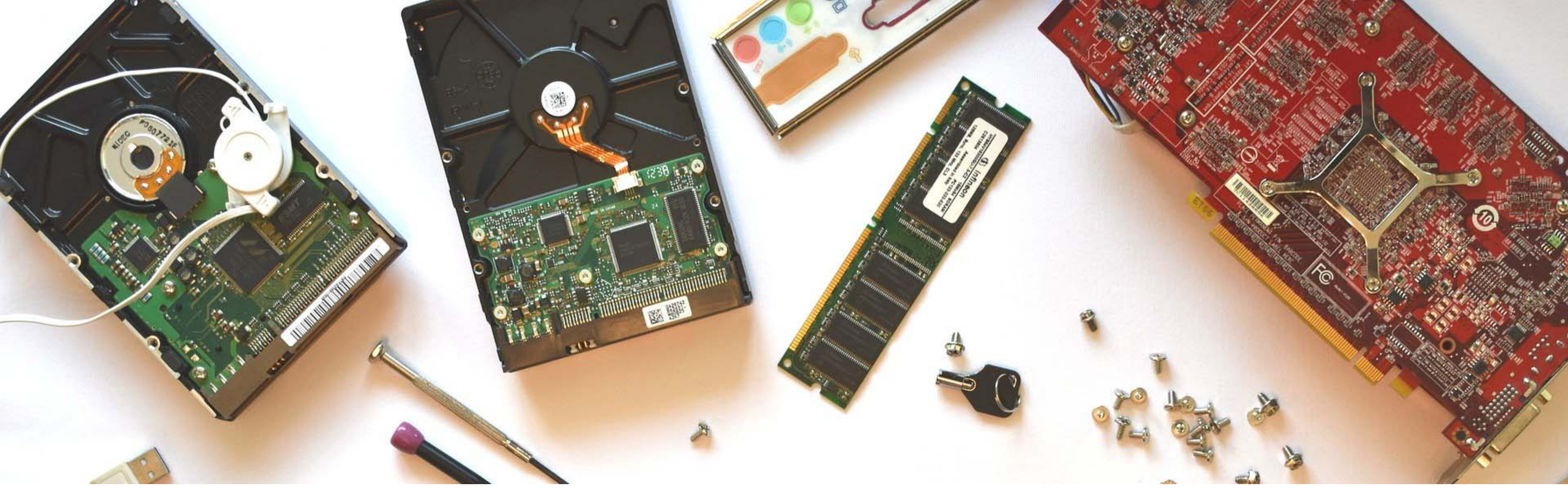

Shenzhen hopetime industry Co.,Limited

VS

Global Well Tech Limited

Turn-Key PCB Assembly

Rigid PCB Design For Manufacturability Guide

Updated: Mar 29th,2023

www.hopetimepcb.com

www.gwt-pcba.com

Table of Contents

1.0 – Introduction 5
1.1 Scope 5
2.0 Required Documents 6
2.1 Gerber Files – RS-274X Format 6
2.2 ODB++ 7
2.3 Drill / Route File 7
2.4 Netlist File 8
2.5 Centroid File / Pick and Place File 8
2.6 Assembly Bill of Materials (BOM) 8
2.7 Drawings 9
2.8 Standard Manufacturing Specifications 10
3.0 Laminate Materials Selection 11
3.1 Material Selection & Properties 11
3.2 Laminate Material and Thickness 12
3.3 Prepreg Designation and Thickness 12
3.4 Copper Weight & Heavy Copper for Materials 13
3.5 RF Substrates 14
3.6 Metal-Core PCBs 15
3.7 Multi-Layer Stack-Up 16
3.8 Multi-Layer Stack-Up Recommendation 16
3.9 Buried Capacitance 19
3.10 Material Substitutes (North America vs China) 20
4.0 Standard PCB Manufacturing Capabilities 21
4.1 PCB Technology Matrix 21
4.2 Etch Factor 23
4.3 Drill Selection 24
4.4 Aspect Ratio 25
4.5 Annular Ring 25
4.6 Tear Drop Pads 26
4.7 Hole Clearance 26
4.8 Conductor Clearance 27

4.9 Via Holes Treatment 27
4.10 Finished Board Thickness 28
4.11 Overall Finished Profile Tolerance 28
4.12 Board Outline 29
4.13 Edge Bevel 29
5.0 HDI Technology Capabilities 30
5.1 Tight Trace Width / Spacing (Trace/Space) 30
5.2 BGA 30
5.3 Micro-via 31
5.4 Multiple Laminations/ Sequential Lamination 33
6.0 Surface Finish: Options and Requirements 35
6.1 HASL/LF-HASL 35
6.2 ENTEK/OSP 36
6.3 ENIG 37
6.4 Full Body Hard Gold 38
6.5 Selective Gold 38
6.6 Double Gold (Full-Body + Selective Gold) 39
6.7 Edge Connector Plating 39
6.8 Wire Bonding (Soft Gold) 39
6.9 ENEPIG 40
6.10 Immersion Tin 40
6.11 Immersion Silver 41
6.12 Comparison Chart 42
7.0 Solder Mask: Options and Requirements 43
7.1 – What is a Solder Mask? 43
7.2 – Types of Solder Mask 43
7.3 – Solder Mask Design Rules 44
7.4 – Solder Mask Colours 45
7.5 – Substitutes (North America vs China) 46
7.6 – Solder Mask Tenting 46
7.7 – Solder Mask Plugging 46
7.8 – Peel-Able Solder Mask 46
8.0 Silkscreen: Options and Requirements 47

8.1 – What is a Silkscreen?....47
8.2 –Silkscreen Requirements....47
8.3 – Silkscreen Types....48
8.4 – Silkscreen Colours....48
8.5 – Substitute (North America vs China)....49
8.6 – Multiple Colour Silk on One PCB....49
8.7 – Serialization....49
9.0 Electrical Testing....50
9.1 – Electrical Testing Requirements....50
9.2 – Flying Probe Test....51
9.3 – Fixture (Bed of Nails) Test....51
9.4 – Cost....51
10.0 Controlled Impedance PCB....53
10.1 Impedance Calculators....55
10.2 Impedance Models....55
10.3 Impedance Affect Stack-up....55
10.4 Hopetimepcb TDR Calculations....56
10.5 TDR Coupons....56
11.0 – Flexible and Rigid-Flex PCBs....57
11.1 Flexible PCB Fabrication Capabilities....57
11.2 Flexible PCB Materials....58
11.3 Rigid-Flex PCBs....61
12.0 Panelization....62
12.1 Fiducials....62
12.2 Tooling Holes....62
12.3 V-Score....63
12.4 Tab Routing....63
13.0 Report Types and Report Writing....64
13.1 Products Final Audit Report....64
13.2 Certificate of Conformance (C of C)....64
13.3 Test Report....65
13.4 Solderability Report....65
13.5 Cross Section Report....65

13.6 Impedance Report (TDR)....65
14.0 RoHS Compliance....66
14.1 PCB Raw Material....66
15.0 Assembly Considerations....67
15.1 – Automated Optical Inspection (AOI)....67
15.2 – X-Ray Inspection....67
15.3 – Functional Testing (FCT)....68

1.0 – Introduction

The purpose of this Design for Manufacturability (DFM) guide is to assist customers in designing printed circuit boards (PCBs) that can be manufactured quickly and efficiently. These DFM guidelines define the various tolerances, rules, and testing procedures to which Hopetimepcb adheres during PCB manufacturing.

It is beneficial to all parties involved, in terms of both cost and efficiency, if these issues can be addressed during the design stage, rather than during production. By providing this guide, we hope to avoid the potential scenario where our client has finished designing a board, but must later revise their design due to facility limitations.

DFM guidelines are essential to promote a client's understanding of the various options that we offer, the reasons for each of these options, and the limitations of Hopetimepcb's production facilities. At Hopetimepcb, we specialize in turn-key production rigid multi-layer PCBs with part procurement and placement.

For a quick overview of our manufacturing specifications, please see our other document on our website www.hopetimepcb.com titled "Capability".

1.1 Scope

Consulting these DFM guidelines during your design process allows you to plan a PCB that conforms to the capabilities of hopetimepcb's facilities. A strong understanding of our manufacturing ability enables our clients to achieve the special features that their designs require while maximizing efficiency to both time and money.

Listed below are a few of the more notable benefits found in designing for manufacturability:

- Higher quality results by working within facility capabilities
- Reduced lead times by avoiding unnecessary delays
- Lower labor and material costs from correcting errors
- Higher first-pass yields
- Minimized environmental impact by avoiding waste from re-making boards

To fully reap the benefits listed above, our clients must understand our abilities concerning the specific type of PCB option(s) that they require. This guide is, therefore, divided into sections according to the different types of options and features that we offer at hopetimepcb. Those sections are as follows:

- Required Documents
- Laminate Materials Selection
- PCB Manufacturing Capabilities
- HDI Technology Capabilities
- Surface Finish: Options and Requirements
- Solder Mask: Options and Requirements
- Silkscreen: Options and Requirements
- Electrical Testing
- Controlled Impedance PCB
- Panelization
- Report Types and Report Writing
- RoHS Compliance

2.0 Required Documents

Various PCB design files and other forms of documentation are required for Hopetimepcb to fabricate a printed circuit board and populate the components onto that board. There are two file formats we can translate and accept for manufacturing: ODB++ and Gerber version RS-274X.

In the OBD++ format, all data required for PCB fabrication and assembly are included within the .TGZ compressed archive. Should you choose to use this format, you only need to provide the .TGZ archive, along with a *Bill of Materials (BOM)* for assembly.

If you choose to use the Gerber file format, please ensure that you are using the RS-274X version. This format specifies a set of files, where each individual file represents one type of design drawing, such as *top copper*, *bottom solder mask*, or *top silkscreen*. Please ensure that you provide a Gerber file for each design element in your PCB layout, as well as any other relevant documentation for fabrication and assembly. When using the Gerber format, additional files are required for board assembly, a centroid file for pick and place, as well as a Bill of Materials with part numbers.

2.1 Gerber Files – RS-274X Format

RS-274X, also known as Extended Gerber or X-Gerber, is one of three distinct formats for Gerber files and is the current industry standard. RS-274X format is an open ASCII vector format used by standard PCB design industry software for processing 2D binary images. All Gerber Files are also Computer Numerical Control (CNC) files, and so it is possible to drive a PCB fabricator using Gerbers since fabricators are CNC machines. Gerber files describe various board images, such as copper layers, solder mask, silkscreen, and paste mask. This data includes traces, vias, pads, component footprints, and planes, as well as drilling and milling data.

RS-274X is the “middle child” of the three Gerber file formats. The oldest format for Gerber files is RS-274D, or Standard Gerber, which has now been generally deprecated in favour of the Extended Gerber format. The data in RS-274X is much more comprehensive than RS-274D since the RS-274D format keeps many of its critical information separate from the main data file. The RS-274X format offers several benefits over the now-obsolete RS-274D, including high-level commands and controls, allowing for more precise machine plotting. Gerber X2, the third and latest Gerber format, was released in February of 2014. It is fully backwards-compatible with the RS-274X format but includes some extra metadata to avoid ambiguity. Gerber X2 has not yet seen widespread industry adoption, and so RS-274X remains the standard at present.

PCB layouts are created using a computer-aided design (CAD) system and saved in the RS-274X format, where the Gerber set contains the complete description for each layer of the PCB. Typically, the CAD system outputs one Gerber file for each relevant layer. These Gerber files can be loaded into a computer-aided manufacturing (CAM) system to provide data for each step of the PCB production process.

In keeping with industry standards, we at hopetimepcb accept the RS-274X Gerber format. This allows our clients to use the CAD design tool of their choice, provided that they can output their finished design files in Gerber format. The Gerber RS-274X format includes separate files for the different copper layers, silkscreen layers, solder mask layers, and milling/drilling locations included in a given design. Along with your Gerber files, you should send a *PCB Manufacturing Drawing* and a *PCB Assembly Drawing*, as well as a *Bill of Materials* for your order. A brief description for each of these file types is given in Table 1 below:

Table 1

PCB Manufacturing Drawing	PCB Board fabrication information
PCB Assembly Drawing	PCB components onboard assembly information
Bill of Materials	Table of information on all components to be placed on board
Netlist	component electrical connectivity information
Gerber Files	Group of files below
Route files	information on PCB copper design, e.g. traces, pads
Drill files	information on PCB design drill holes
Solder Mask Files	Information on areas not to cover with solder mask, e.g. pads, holes
Board Outline File	board size and shape information
Silkscreen	board surface markings information

2.2 ODB++

ODB++ is a printed circuit board (PCB) manufacturing database, with different data stored in a hierarchy of files and file folders. For convenient transmission of data, some common operating system commands can be used to create a single compressed file, while preserving the hierarchy information. That compressed file containing the printed circuit board (PCB) design information can be sent directly to a PCB Fabrication and Assembly company, such as hopetimepcb. ODB++ stands for *Open Data Base*, with the '++' suffix added in 1997 when component descriptions were enabled. ODB++ was developed and disseminated by Valor Computerized Systems, but was later acquired by Mentor Graphics in 2010.

The vast majority of electronic devices include a PCB, which acts to house the electronic components that power the device, and also to connect these components in a specific manner. Computer-aided design (CAD) software is often used to create the layouts for these PCBs, and that layout information must then be transferred to a photolithographic computer-aided manufacturing (CAM) system. These CAD and CAM systems are generally produced by different companies, and therefore must use an intermediary file format, such as ODB++, for successful data exchange. ODB++ has two versions: the original (owned by Mentor), as well as an XML version called ODB++(X), which was donated to the IPC organization by Valor Computerized Systems.

We at hopetimepcb can work with ODB++ files of revision 7.0 or lower; there are no restrictions on the design tool used for PCB layout, as long as the design can be translated to ODB++ format files. That being said, we hopetimepcb can **NOT** accept ODB++(X) files. If you find some trouble in outputting your ODB++ files , please forward your .CAM or .PCB files to us to attempt the conversion on your behalf.

2.3 Drill / Route File

Drill Files define the size and coordinates of any holes to be drilled in a PCB design. These files can be used to control a drilling machine, which creates holes for Vertical Interconnect Access (VIA), mounting, and THT component placement.

On a standard 2-sided PCB, drill files are needed for the CNC machine to make accurate through-vias. For more complex multi-layer boards, many of the holes will be micro-vias, which pass through only a few layers rather than the entire board. These types of holes are also known as blind and buried vias. We will need a separate drill file, with a distinct name, for each layer pair that will be included in your design.

For Example: Let us assume that you have a 4-layer board with most vias extending all the way from top to bottom, and some buried vias extending from layer 1 to layer 2. In this case, you should have two drill files, with one of them named "Drill_1-4" and another named "Drill_1-2".

Route Files are a type of file that define the electrically-conductive copper traces, known colloquially as "routing", on a PCB design. These files are used to control a fabrication machine in order to lay copper traces on a PCB board.

Regarding the submission of drill and route files to hopetimepcb: if you use the ODB++ format, drill files and route files will both be included in the .TGZ compression file. If you choose to use the Gerber RS-274X format, then separate drill files and route files will need to be provided in RS-274X format.

2.4 Netlist File

Netlist files contain the connectivity information for an electronic circuit. A netlist file is a collection of several related lists, one list for each group of electrically-connected pins. This file is generated from the circuit's schematic design, and it is used in PCB layout to determine which component pads should be connected by copper traces on the finished PCB

We at Hopetimepcb request a netlist file for the electrical testing of your bare PCBs; we ensure an exact match between the actual connectivity on your PCB and that of your netlist file for every board that leaves our facility. The ODB++ format contains your netlist file by default, and so there will be no need to provide one separately. If you use the Gerber RS-274X format, then you will need a separate netlist file in IPC356 format.

2.5 Centroid File / Pick and Place File

We require a centroid data file –also known as a *Pick and Place* file– in order to accurately assemble a PCB. The main purpose of a centroid data is to hold information on the position and orientation of all surface mount technology (SMT) devices on a circuit board design. Centroid files contain data regarding reference designators, XY locations, rotations, component packages, and placement on either the top or bottom side of the board.

- Reference Designator – Short alphanumeric code assigned to each part in the layout (R12, C17)
- Component Value/Package – Component part numbers, part values, and package size
- Layer – Either the top or the bottom side
- XY Location – Cartesian coordinates, beginning at the origin
- Rotation – Described part orientation in degrees (0, 45, 90, etc.)

2.6 Assembly Bill of Materials (BOM)

Bill of Materials is a list of all the parts that are needed for the PCB Assembly and matches those parts to the Reference Designators in the PCB layout. We require this list in a Microsoft Excel format. If multiple instances of the same part are used on one board, then they are all listed on one line of the BOM with their corresponding reference designators and total quantity. The following information comprises the BOM; incomplete BOMs may be acceptable, however, it is always best to include all of the following information in your BOM to minimize the risk of purchasing errors.

- Item # - Unique item number for each component (1, 2, 3)
- Ref Des – Matches BOM item to PCB layout location
- Quantity – Quantity of this component that will be needed for each board
- Manufacturer – The name of this component's manufacturer
- Manufacturer Part # - The part number assigned to a component by its manufacturer
- Description – Brief component description
- Package – Packaging size (ex. 0805) or type (ex. BGA, QFN)
- Type – SMT or THT
- Your Instructions – Any special requirements that you have for this part

We provide a sample Bill of Materials on our website that may be used as a basis for your Bill of Materials.

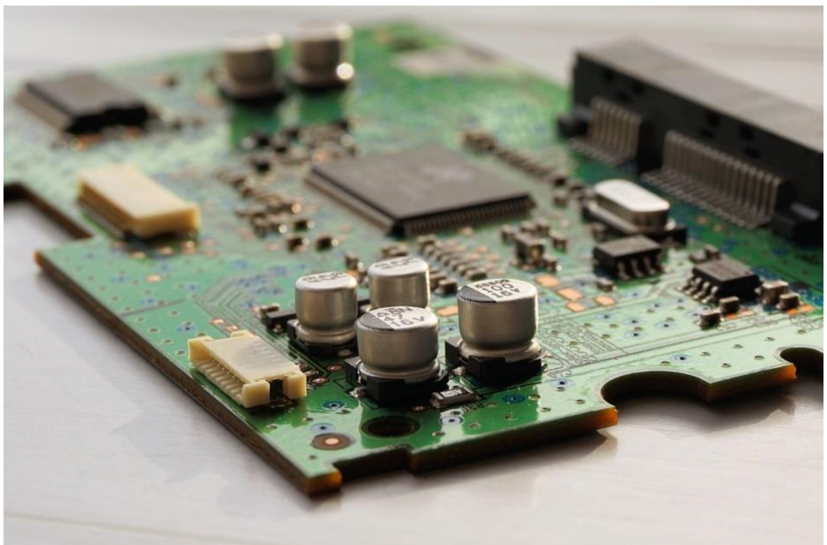

Figure 1: Assembled Printed Circuit Board

2.7 Drawings

Design drawings are not strictly required for all PCB fabrication and assembly projects, but they can help to make your intentions clearer to our production team, particularly regarding special requirements. We recommend you include design drawings for projects with a high number of layers or components.

2.7.1 PCB Manufacturing Drawing

PCB manufacturing drawings only contain that information which is needed for the fabrication of the PCB. This could be information such as the project name, board dimensions, board thickness, tolerances, material, copper weight, number of copper layers, or surface finish. These may be a list of notes beside the board's artwork or listed in a separate file. Other special information may also be included in the manufacturing drawing, such as controlled impedance requirements, via-in-pad tenting, gold-plated edge connectors, and so on.

2.7.2 PCB Assembly Drawing

PCB assembly drawings contain that information that is needed for assembly of the various components onto a PCB. Basic component information, such as position and orientation, are provided on this drawing, along with special requirements, such as height limit. Three-dimensional renderings could also be included to clarify special requirement areas but are not necessary for typical boards.

2.8 Standard Manufacturing Specifications

The following list outlines the parameters we require to manufacture your design. Please provide this information when requesting a quote as well, in order to price your project properly.

- Project Name – PCB name and Version #
- Dimensions [inches] – Board length and width
- Number of layers – 2, 4, 6, 8, 10, 12, 14, 16 layers
- Total thickness [inches] – Board thickness for one board with all layers included
- Surface Finish – Example: HASL, ENIG, Immersion Tin, Immersion Silver
- Solder mask – Indicates the desired solder mask colour for your PCB, and to which sides of the board that solder mask will be applied

Advanced parameters which may also be specified include

- Profiling – Indicate any panelization requirements for your project
- Impedance Control – Do you require a specific impedance for RF transmission lines
- Material (substrate) – Example: FR4 or IT-180A
- Silkscreen (Legend)– Indicates the desired silkscreen colour for your PCB, and to which sides of the board that silkscreen will be applied
- Testing – Electrical testing is standard with all our orders and cannot be excluded to ensure quality.
- Printed board handling and storage guidelines – Special information regarding safe handling if required

3.0 Laminate Materials Selection

Laminates are the primary material used in PCB fabrication, and different laminates have different properties, performances, and associated costs. This section outlines the various laminate options that may be selected for your PCB Design and gives a brief comparison and contrast between those options.

3.1 Material Selection & Properties

As the first step in this DFM guideline, we will provide information to assist you in the selection of a suitable laminate material that meets your performance requirements while minimizing manufacturability issues. We start here because the cost of raw laminates is generally the single largest constituent of PCB fabrication cost.

The key factors to consider when selecting a laminate material for a PCB design are the cost, the quality, and the lead time. Due to the amount of material needed for PCB fabrication, it is essential to optimize the size of your designs; even a small difference in size can result in a significant difference in cost.

Different materials incur different costs and possess different characteristics, but higher quality laminates are typically also more expensive. The following are some of the main characteristics to take note of when comparing the properties of different laminates:

Tg = Glass Transition Temperature – Temperature at which a critical change of physical properties will occur. In the case of laminates, it transitions from a hard, glassy material into a soft, rubbery material. Hopetimepcb offers many High-TG options for high-temperature and high-power PCB applications.

Td = Decomposition Temperature - Temperature at which the laminate chemically decomposes.

Dk = Dielectric Constant (also referred to as ε_r in electromagnetics) – Indicates the *relative permittivity* of an insulator material, which refers to its ability to store electrical energy in an electric field.

Df = Dissipation Factor – Indicates the efficiency of an insulating material by showing the rate of energy loss for a certain mode of oscillation, such as mechanical, electrical, or electromechanical oscillation.

Our fabrication facilities are located in China, so it is advisable to choose high-quality local laminates in order to minimize the shipping cost and lead time. The Shengyi S1000-H (Tg 150) laminate is generally our default choice for a high-performance, mid-Tg laminate. Shengyi S1000-H is comparable to Isola FR406 (Tg 150), a standard North American laminate option. As outlined in Table 2 below, FR406 does slightly outmatch Shengyi S1000H in terms of Dielectric Constant and Dissipation Factor, but some clients may be willing to compromise on these factors for a lower cost and/or a faster lead time.

Table 2

	Tg 130			Tg 150			Tg 170			Tg 180	
	China	U.S.		China	U.S.		China	U.S.		China	U.S.
	S1141	FR406		S1000-H	FR406		S1000-2	FR406		IT180A	370HR
Td (TGA @ 5% weight loss)	N/A	300		N/A	300		N/A	300		350	340
Dk (50% resin @ 2 GHz)	4.2	3.93		4.38	3.93		4.28	3.93		4.3	4.04
Df (50% resin @ 2 GHz)	0.015	0.0167		0.015	0.0167		0.017	0.0167		0.015	0.21
RoHS	Y	Y		Y	Y		Y	Y		Y	Y

Shengyi S1141 (TG 130) is a good alternative to lower the cost of your project, at the sacrifice of some quality. In cases where higher quality is needed, we recommend Shengyi S1000-2M (TG 170), which is the closest in quality to Isola FR406 (Tg 170). Where quality is the highest priority, we would recommend utilizing ITEQ IT180A (TG 180), which is also RoHS compliant. ITEQ IT180A (TG 180) is comparable in quality and to Isola 370HR (TG 180). We at Hopetimepcb would suggest using Shengyi S1000H (Tg 130) for typical projects. We would recommend using one of the higher quality laminate materials if any of these three conditions occur:

1) If the PCB Design has **8 or more layers**
2) If Copper Board is heavy with a copper weight heavier than 3oz
3) If PCB Board is thin with a board thickness of less than 0.5mm

3.2 Laminate Material and Thickness

Table 3

Core material thickness, including copper (mm.)	Copper Weight (oz.)
0.145 mm.	H/H oz.
0.17 mm.	1/1 oz.
0.185 mm.	H/H oz.
0.2 mm.	1/1 oz. or H/H oz.
0.25 mm.	1/1 oz. or H/H oz.
0.3 mm.	1/1 oz. or H/H oz.
0.4 mm.	1/1 oz. or H/H oz.
0.5 mm.	1/1 oz. or H/H oz.
0.6 mm.	1/1 oz. or H/H oz.
0.7 mm.	1/1 oz. or H/H oz.
0.8 mm.	1/1 oz. or H/H oz.
0.9 mm.	1/1 oz. or H/H oz.
1.0 mm.	1/1 oz. or H/H oz.
1.1 mm.	1/1 oz. or H/H oz.
1.2 mm.	1/1 oz. or H/H oz.
1.5 mm.	1/1 oz. or H/H oz.
1.6 mm.	1/1 oz. or H/H oz.
2.0 mm.	1/1 oz. or H/H oz.
2.2 mm.	1/1 oz.
2.4 mm.	1/1 oz.
2.5 mm.	1/1 oz.
3.0 mm.	1/1 oz.

Table 3 lists the core material thickness with copper weight for normal FR4 Material

- 1/1 = 1 oz. copper per square foot on BOTH sides of the sheet
- 1/0 = 1 oz. copper per square foot, coated on only 1 ONE side of the sheet
- H/H = 0.5 oz. copper per square foot, coated on BOTH sides of the sheet
- 0/0 = UNCLAD (NO Copper).

3.3 Prepreg Designation and Thickness

Prepreg is a bonding material used in the fabrication of multi-layer PCB boards, which, after curing, has the same properties as the core /base layer materials. Prepregs have various glass styles 106, 1080, 3313, 2116 and 7628 used by board fabricators. Board fabricators use a variety of prepreg glass styles. These styles include 106, 1080, 3313, 2116 and 7628. Limitations may apply to the number and types of prepreg, so it is best to contact a Hopetimepcb representative for further details.

Table 4

Prepreg /Glass Styles	Pressed Thickness (mm)	Prepreg Resin content
106	0.05 mm.	Approx. 73%
1080	0.075 mm.	Approx. 65%
3313	0.09 mm.	Approx. 57%
2116	0.115 mm.	Approx. 55%
7628	0.185 mm.	Approx. 46%
7628H	0.195 mm.	Approx. 51%

3.4 Copper Weight & Heavy Copper for Materials

Copper clad FR-4 laminate materials are measured using ounce (oz.) weight per square foot.

- 0.25 oz. = 0.00035” (8.75µm)
- 0.5 oz. = 0.0007” (17.5 µm)
- 0.75 oz. =0.00105” (26.25µm)
- 1.0 oz. = 0.0014” (35 µm)
- 2.0 oz. = 0.0028” (70 µm)
- 3.0 oz. = 0.0042” (105 µm)
- 4.0 oz. or more = 0.0056” (140 µm) or more

Heavier copper weights allow for higher current carrying capability and heat dissipation characteristics in high power designs, making heavy copper a common choice for automotive, power distribution, and welding equipment industries . Hopetimepcb Electronics is capable of fabricating multi -layer PCB boards with a maximum copper weight of 10 oz. Copper weight of 4 oz. or higher will require an additional estimate, and may also affect lead times. The specified copper weight for a board will also impact the trace width and spacing requirements of the board, as shown in Table 5, below:

Table 5

Layer Type	Copper Weight	Width / Spacing
Internal Layer	1/2 oz.	3.0 / 4.0 mil (preferred) – 3.0 / 3.0 mil (minimum)
	1 oz.	4.0 / 5.0 mil (preferred) – 3.0 / 4.0 mil (minimum)
	2 oz.	5.0 / 7.0 mil (preferred) – 4.0 / 5.5 mil (minimum)
	3 oz.	6.0 / 10.0 mil (preferred) – 5.0 / 8.0 mil (minimum)
External Layer	1/2 oz.	4.0 / 4.0 mil (preferred) – 3.0 / 4.0 mil (minimum)
	1 oz.	4.0 / 6.0 mil (preferred) – 4.0 / 4.0 mil (minimum)
	2 oz.	5.0 / 8.0 mil (preferred) – 5.0 / 6.0 mil (minimum)
	3 oz.	6.0 / 10.0 mil (preferred) – 6.0 / 8.0 mil (minimum)

3.5 RF Substrates

For radiofrequency applications requiring very high fidelity, some clients find that standard FR4-type material does not meet signal loss requirements due to its relatively high dielectric constant of approximately 4.5. As such, specialized dielectric materials for RF applications offer lower dielectric constant to satisfy such design requirements. Hopetimepcb stocks Rogers RO4350B as our standard RF substrate. We offer core thicknesses of 10mil, 20mil or 30mil in stock, and the copper weight can be 0.5oz/0.5oz or 1oz/1oz. Other sizes available for special order with added lead-time are 6.6mil, 13.3mil, 16.6mil, 60mil. The dielectric constant of this material is 3.76 at 1GHz and averages 3.66 for 8GHz to 40GHz. Further details can be found on the material's datasheet. Additional RF materials may be available upon special request with added lead time.

The dielectric constant of a material describes the ratio of that material's permittivity to the permittivity of vacuum. Technically speaking, a material's permittivity describes the electric field strength between two charges separated by that material. The dielectric constant of a material affects its parasitic capacitance, which can become quite significant at high operational frequencies. Equation 1 and Equation 2 illustrate the effects of dielectric constant on signal loss. Equation 1 describes the capacitance of a parallel-plate capacitor—derived from Maxwell's Laws—where any two copper layers of a PCB can be viewed as parallel plates separated by the dielectric material. The k variable represents the dielectric constant in Equation 1, and clearly, the capacitance increases with k. Equation 2 described capacitive reactance, which decreases inversely to capacitance and frequency; this explains the increased significance of material dielectric constant for high-frequency operation.

$$\boxed{C = \frac{k\varepsilon_0 A}{d}} \quad Eqn. 1 \qquad\qquad \boxed{X_C = \frac{1}{2\pi f C}} \quad Eqn. 2$$

While standard rigid FR4-type materials are composed of glass-reinforced epoxy, RF materials primarily make use of Polytetrafluoroethylene, also known as PTFE, or by its trademark name: Teflon. In addition to its advantages in dielectric constant, PTFE materials also boast impressive thermal characteristics for high-temperature PCB applications. With high TG (glass-transition temperature) values, reaching up to 280°C, PTFE materials are suitable for both high frequency and high-power applications.

When preparing your project for PCB Fabrication using PTFE materials, it is important to reach out early in the process, if possible, for a quotation. Projects involving Rogers and other high-frequency PCB materials will require a special estimate, not covered by Hopetimepcb's online quoting tool. Extra care is necessary in PCB Fabrication for high-frequency materials since PTFE is relatively soft compared to FR4-type materials, making it more susceptible to damage while handling. The fabrication of plated through-holes and slots is also more difficult for PTFE materials, due to the famous non-stick properties of Teflon.

3.6 Metal-Core PCBs

For PCBs with very high thermal demands, Hopetimepcb can also provide professional metal-core PCB fabrication options, where a metal base is bonded to the copper signal layers, separated by a dielectric for insulation. Metal-Core PCBs are often used to help conduct heat away from vital components in high power designs, such as amplifiers, power supply modules, motor drives, commercial lighting, and many automotive applications. The standard Metal-Core PCB design consists of three main components:

Signal Layer: Contains copper pads and traces, subject to copper weight specifications defined in Section 3.4

Dielectric Layer: Offers high electrical insulation and high thermal conductivity

Metal Core: Most commonly aluminum for low-cost, but other options such as copper are available

Figure 2 shows a simple diagram of the standard Metal-Core PCB components described above:

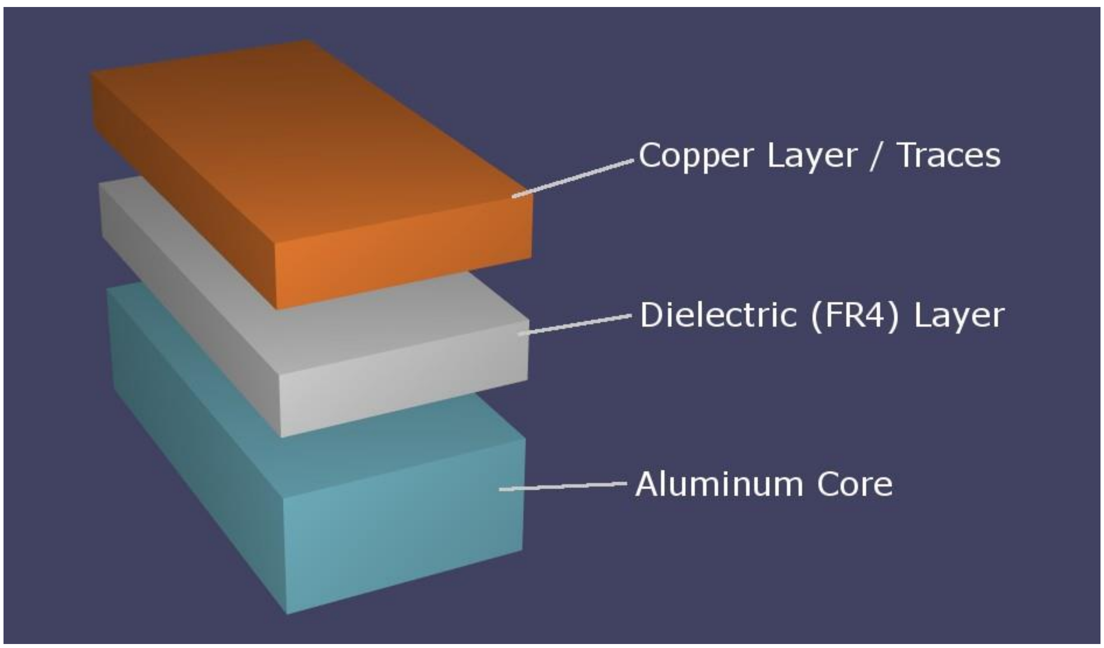

Figure 2: Simple Metal-Core PCB Visualization

Hopetimepcb generally provides metal-core PCB fabrication with core thicknesses ranging from **0.031” to 0.125”**, but we can also handle orders for other sizes with a special estimate. We also provide heavy copper options for the signal, power, and ground layers of your design, from **0.5 oz. to 10 oz.** or more.

3.7 Multi-Layer Stack-Up

Multi-layer boards have some physical properties which need to be considered by both the designer and manufacturer in order to ensure quality construction. Multilayer board designs should have an **even** number of layers for the best quality. Choose each layer's dielectric thickness from the provided core or prepreg thicknesses listed in Table 3 and Table 4. For possible combinations , please consult our Hopetimepcb manufacturing team for what are suitable and achievable dimensions and tolerances.

It is recommended that multi-layer designs balance the lay-up relative to the Z-axis median to minimize bow and twist. In other words, a layer stackup list should read the same from top to bottom as it does from bottom to top. Dielectric thickness, copper thickness and the location of layers, median, Z-axis need to be balanced. If the Multi-layer design rules are adhered to, the PCB should meet specifications for the maximum allowable bow and twist of 0.25mm per 25mm (1%) or better. It is also beneficial to balance the circuitry distribution between the front and back of the board as much as possible. This is more of a concern with thicker copper weights.

Thickness Tolerance increases as the overall thickness of a multi-layer board increases. You should specify a tolerance of **±10%** for the overall thickness, which is thicker than 1mm, or **+/-0.1mm** for 1mm thickness or thinner. Also, you need to always indicate where the thickness measurement is to be taken.

3.8 Multi-Layer Stack-Up Recommendation

We at Hopetimepcb can fabricate multi-layer PCB boards up to a maximum of 40 layers. PCB's with more than 20 layers will require an additional estimate before production can begin. Below are our recommended stackups for a normal board thickness of 62mil. We at Hopetimepcb do offer **custom stackups**, please let us know what you need, and our CAM engineer will review it for feasibility.

2 Layer Stackup				
Layer Order	Layer Name	Material Type	Thickness	Copper Weight
1	Top	Copper	1.4 mil	1 oz.
		Core	57.7 mil	
2	Bottom	Copper	1.4 mil	1 oz.
Finished board thickness: 62mil +/-10%				

4 Layer Stackup				
Layer Order	Layer Name	Material Type	Thickness	Copper Weight
1	Top	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
2	Inner 1	Copper	1.4 mil	1 oz.
		Core	36.6 mil	
3	Inner 2	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
4	Bottom	Copper	1.4 mil	1 oz.
Finished board thickness: 62mil +/-10%				

6 Layer Stackup				
Layer Order	Layer Name	Material Type	Thickness	Copper Weight
1	Top	Copper	1.4 mil	1 oz.
		Prepreg (2116)	4.5 mil	
2	Inner 1	Copper	1.4 mil	1 oz.
		Core	16.9 mil	
3	Inner 2	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
4	Inner 3	Copper	1.4 mil	1 oz.
		Core	16.9 mil	
5	Inner 4	Copper	1.4 mil	1 oz.
		Prepreg (2116)	4.5 mil	
6	Bottom	Copper	1.4 mil	1 oz.
Finished board thickness: 62mil +/-10%				
8 Layer Stackup				
Layer Order	Layer Name	Material Type	Thickness	Copper Weight
1	Top	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
2	Inner 1	Copper	1.4 mil	1 oz.
		Core	5.1 mil	
3	Inner 2	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
4	Inner 3	Copper	1.4 mil	1 oz.
		Core	5.1 mil	
5	Inner 4	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
6	Inner 5	Copper	1.4 mil	1 oz.
		Core	5.1 mil	
7	Inner 6	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
8	Bottom	Copper	1.4 mil	1 oz.
Finished board thickness: 62mil +/-10%				

10 Layer Stackup				
Layer Order	Layer Name	Material Type	Thickness	Copper Weight
1	Top	Copper	1.4 mil	1 oz.
		Prepreg (2116)	4.5 mil	
2	Inner 1	Copper	1.4 mil	1 oz.
		Core	3.9 mil	
3	Inner 2	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
4	Inner 3	Copper	1.4 mil	1 oz.
		Core	3.9 mil	
5	Inner 4	Copper	1.4 mil	1 oz.
		Prepreg (1080*2)	5.9 mil	
6	Inner 5	Copper	1.4 mil	1 oz.
		Core	3.9 mil	
7	Inner 6	Copper	1.4 mil	1 oz.
		Prepreg (2116*2)	9.1 mil	
8	Inner 7	Copper	1.4 mil	1 oz.
		Core	3.9 mil	
9	Inner 8	Copper	1.4 mil	1 oz.
		Prepreg (2116)	4.5 mil	
10	Bottom	Copper	1.4 mil	1 oz.
Finished board thickness: 62mil +/-10%				

3.9 Buried Capacitance

Buried Capacitance is a PCB production methodology where decoupled capacitance is gained from inserting a very thin dielectric layer inside a PCB. Adding buried capacitance to a board eliminates the need for decoupling capacitors, freeing up more PCB space as redundant pads and conductors can be removed. Having a lower number of components on a PCB will lower the cost and simplifies PCB assembly operations and also reduces the number of required steps. New technology, like this, grants greater freedom to designers, which allows them to design PCBs with greater performance, or equivalent performance in a smaller board size. Lastly, using buried capacitance can also reduce noise and high-frequency electro-magnetic interference to improve PCB quality. Figure 3 (below) illustrates an 8 layer board with buried capacitance.

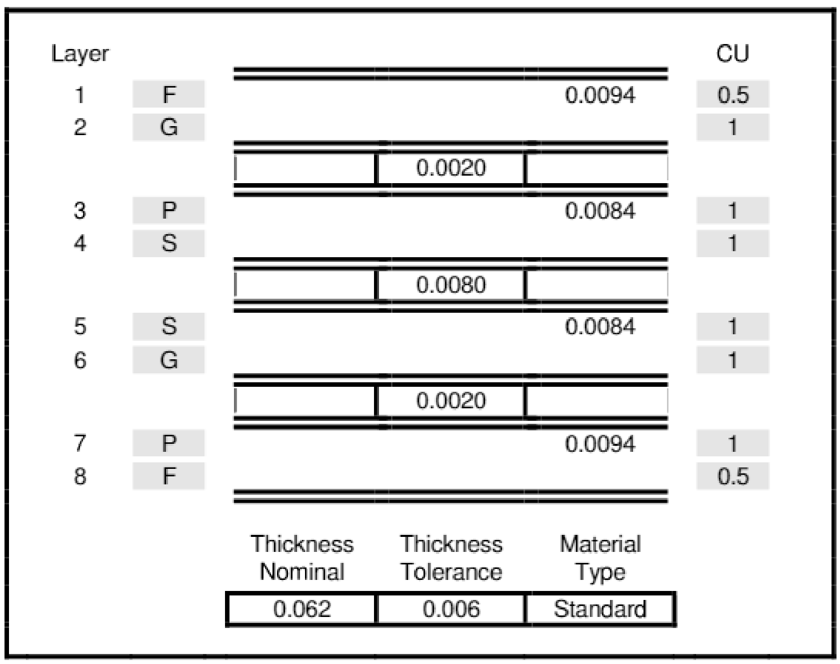

Figure 3: Buried Capacitance Stackup (Eight Layers)

3.10 Material Substitutes (North America vs China)

China Substitute Materials

- Shengyi S1141 (TG 140)
- Td 300, Dk 4.2, Df 0.015
- Material Datasheet Link

- Shengyi S1000H (Tg 150)
- Td 325, Dk 4.38, Df 0.015
- Material Datasheet Link

- Shengyi S1000-2M (TG 170)
- Td 340, Dk 4.28, Df 0.017
- Material Datasheet Link

- ITEQ IT180A (TG 180)
- Td 350, Dk 4.3, Df 0.015
- Material Datasheet Link

North American Laminates

- Isola FR406 (TG 170)
- Td 300, Dk 3.93, Df 0.0167
- Material Datasheet Link

- Isola 370HR (TG 180)
- Td 340, Dk 4.04, Df 0.021
- Material Datasheet Link

Table 6

	Tg 130		Tg 150		Tg 170		Tg 180	
	China	U.S.	China	U.S.	China	U.S.	China	U.S.
	S1141	FR406	S1000-H	FR406	S1000-2	FR406	IT180A	370HR
Td (TGA @ 5% weight loss)	N/A	300	N/A	300	N/A	300	350	340
Dk (50% resin @ 2 GHz)	4.2	3.93	4.38	3.93	4.28	3.93	4.3	4.04
Df (50% resin @ 2 GHz)	0.015	0.0167	0.015	0.0167	0.017	0.0167	0.015	0.21
RoHS	Y	Y	Y	Y	Y	Y	Y	Y

Finally, apart from the laminate quality, we must also take the available laminate's material cost and delivery lead time into account. Since Hopetimepcb's PCB Fabrication and Assembly facilities are based in China, the cost for importing North American laminates would be higher and increase the lead time, as illustrated in the chart above. Thus, from this comparison chart, we can conclude that the use of local substitute is preferable due to the lower material costs and shorter lead time.

Table 7

Material Price Matrix				
	Cost in China	**Lead Time**	**Cost in N.A.**	**Lead Time**
370HR	130%	Longer	100%	Shorter
FR408HR	150%	Longer	120%	Shorter
S1141	70%	Shorter	120%	Longer
S1000-H	75%	Shorter	120%	Longer
S1000-2	70%	Shorter	120%	Longer
IT180A	80%	Shorter	140%	Longer

4.0 Standard PCB Manufacturing Capabilities

PCB Manufacturing Capability is a summary of a PCB manufacturing facility's PCB fabrication and assembly options, limits and tolerances in regards to what circuit boards can be produced. This section introduces our facility's fabrication and assembly capabilities as well as cost and lead time requirements. Often clients will make PCB design choices based on the manufacturer's fabrication capabilities. With the knowledge of our capability, limits, and tolerances, mistakes can be reduced, and time can be saved. Since all PCBs should essentially be designed for manufacturing, these guidelines aim to aid the transition from design to fabrication and assembly.

4.1 PCB Technology Matrix

A brief overview of technological specifications that determine if a PCB can be fabricated by Hopetimepcb are listed in Table 8 below . For a more comprehensive list of our capabilities , please refer to our other document *PCB Fabrication Specifications*.

Table 8

Feature	Hopetimepcb Capability
Minimum trace width	4 mil (increases w/ copper weight – see Section 3.4)
Minimum hole size	0.1mm (laser) – 0.15mm(mechanical)
Maximum hole size	0.15mm (laser) – 6.0 mm (mechanical)
Minimum clearance	3 mil (but increases with higher copper weight)
Board thickness limits	0.2mm - 6mm for 2-layer board
Board thickness tolerance	±10% (>1.0 mm) – ±1.0 mm (<1.0 mm)
Maximum allowed copper weights	0.5oz (0.685mils) to 20oz (27.4mils)
Maximum & minimum board size	0.2"x0.2" to 14"x20" (or larger for bare boards)
Aspect Ratio	10:1 (for OSP finish, 8:1)
Layer count- The number of layers in a multilayer PCB	Up to 40 (below 20 recommended)
Bow & Twist	0.75%
Impedance Control	±5% (50 Ω) -- ±10% (>50 Ω)
Special features	Gold fingers, multiple surface finishes, jump score, blind/buried/micro vias, via filling, etc.

PCB key factors also affect each other; thus, they increase or decrease in relation to each other; for example:

- The trace width and the clearance affect each other; when trace width increases, the trace clearance will decrease, and its impedance will also decease.
- The minimum hole size and board thickness affect aspect ratio, so when minimum hole size decreases or board thickness increases, the aspect ratio will enlarge.
- Hole sizes will affect the annular ring and the space between hole to other features. When hole size enlarges, the annular ring and space will decrease.
- Copper weight affects impedance. When copper weight increases, the impedance will decrease.

4.1.1 Pricing

Hopetimepcb offers baseline pricing for PCB fabrication and assembly. These will start with default values and can be customized to meet your needs. The final cost will be affected by material choice and component cost. Material and component prices are constantly changing due to fluctuation in market prices. Some factors that affect PCB fabrication and assembly cost include:

- **Board Size**- While the PCB board is larger than 50x50mm, a smaller PCB can lower cost while a larger PCB board will cost more. Boards of less than 50x50mm are more complex to manufacture.
- **Number of layers**- Price will increase as the number of layers increases.
- **Laminate Materials**- A laminate material with Tg 140 would cost less while higher quality laminate with Tg 180 will cost more.
- **Copper Weight**- Copper Weight is separated into inner and outer layers. A higher copper weight, along with a larger number of layers, will increase production cost.
- **Gold Finger**- used to interconnect to PCBs with sockets on them like a PC motherboard. Gold fingers will increase price based on quantity and the number of sides requiring gold fingers.
- **Via in Pad**- Holes under the component pads, will increase the cost by adding an extra step to fabrication.
- **Blind, Buried, Micro-via**- Special vias require extra processing steps and multiple lamination steps.
- **Board Thickness**- The overall cost will vary depending on the board thickness required.
- **Surface finish**- Hopetimepcb offers the choice of Gold Immersion, HASL, Lead-free HASL, OSP and Silver Immersion at no additional cost. Hard Gold, Selective Gold Plating, etc., will affect pricing.
- **Impedance Control**- calculate electrical impedance and buried impedance. Tighter manufacturing tolerance to meet requirements adds to cost. TDR coupon & report are included.
- **V-scoring, Tab Routing, and Mouse holes**- Used to form multiple boards **into a panel** with no extra charge for services if multiple boards all have the same design. If multiple PCB designs are needed for panelization, additional fees for this service will apply.
- **Special Features**- countersink holes, etc.
- If the requirements of the PCB do not conform to the capacity of the factory; e.g. layer count, board thickness, board size, surface finish, and special features; then cost will be affected.

4.1.2 Lead Time

Lead time is an estimate of the amount of time needed to complete one operation or process from start to end. One type of lead time is the material delivery lead time which is the estimated time needed for board assembly materials to arrive at hopetimepcb facilities for non-standard materials. Another type of lead time is the manufacturing lead time, an estimate by our production facilities for the time needed to fabricate and assembly a PCB in question. Our minimum lead time for bare PCBs is **5 days**; however, lead time increases with quantity and number of layers. For turn-key orders, our standard lead time is **14 days;** however, you may ask one of our sales associates about rushed service. Factors beyond layers and quantity that affect lead time include:

- **Copper weight**- each extra ounce over 3oz. will add one extra day lead time
- **Oversized** or very small boards require an extra day lead time
- **Black solder masks** require an extra day lead time for bare boards only (not turnkey)

Table 7 shows the standard lead times for various numbers of PCB layers, which is the main factor in determining the manufacturing time required for a bare PCB. Please note that there are many other factors that can impact the overall project lead time, such as order quantity or special requirements like via in pad or oversized dimensions.

Table 9

Layers	5 days	6 days	7 days	8 days	9 days	10 days	11 days	12 days	13 days
1-2 Layers	X	X	X	X	X	X	X	X	X
4 layers		X	X	X	X	X	X	X	X
6 layers			X	X	X	X	X	X	X
8 layers				X	X	X	X	X	X
10 layers					X	X	X	X	X
12 layers						X	X	X	X
14 layers							X	X	X
16 layers								X	X
18 layers									X

4.2 Etch Factor

What is etching?

PCB etching is the process of selectively removing unwanted copper from a PCB's copper-clad substrates. The two primary methods of removing the copper cladding are mechanical etching and chemical etching. Mechanical etching uses a CNC machining tool with a special cutting tool to remove narrow strips of copper from the boundary of each pad and trace in order to electrically isolate them from the rest of the copper foil. Chemical etching uses a corrosive solution to dissolve away unwanted copper. A protective layer is cut to match the PCB's Gerber design files and then blocks the applied chemicals maintaining the pads and traces that need to remain intact . We at Hopetimepcb use chemical etching as a normal method due to its precision and efficiency.

What is etch factor?

To produce the copper layer features designed by our client, we use a corrosive solution to remove the unwanted copper area, thus leaving behind the desired pattern. But when etching, the corrosive solution will not only etch the unwanted copper, but also etch the walls of the copper features of the design (this means undercut). Figure 4 (below) illustrates the result. The ratio "D/C" is equal to the etching factor. Thus, the depth to which the etching occurs is proportional to copper layer thickness (i.e. thicker copper has deeper etching). This occurs because thicker copper requires more time exposed to the chemicals in order to be fully removed; thus, the corrosive chemicals have more time to etch the walls before being washed away.

The etch factor for PCBs manufactured by Hopetimepcb depends on whether the layer is inside the board or outside. Etch Factor for outer layers is **1.4,** and the Etch Factor for inner layers is **2.5.**

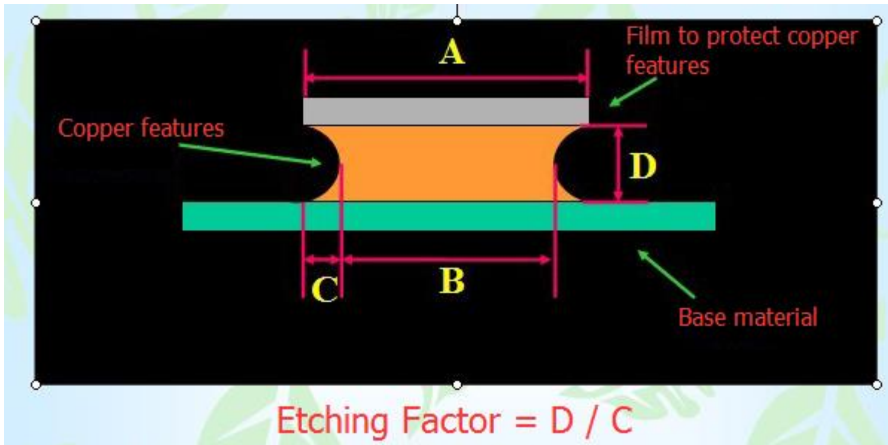

Figure 4: Etching Dimensions Profile View

4.3 Drill Selection

4.3.1 Hole Diameter

We Hopetimepcb can drill holes with sizes listed

below. Table 10

Available mechanical holes size	0.15mm - 6.0mm
Available laser holes size	4mil - 6mil (0.1mm-0.15mm)
Note: If the holes size is bigger than 6.0mm, we can also build them as cutouts (by milling), but the tolerance needs to be +/-0.15mm	

Holes drilled into printed circuit boards can either be plated in copper or left bare. The plated holes will be covered with copper on their walls after copper layers are etched, and they are used for forming electrical connections from one copper layer to another in a PCB. Non-plated holes are usually used for positioning or mounting, and thus do not require copper on their walls. Please indicate your choices in your Gerber files. Your circuit design program should be able to assist with this.

For multilayer boards, holes can be categorized based on their depth into through-holes, blind holes and buried holes. Through-holes will be drilled through the whole board from the top to bottom layer, blind holes will be drilled from one outer layer to an inner layer, and buried holes are drilled from an inner layer to another inner layer. See section *5.3 Micro-via* for further explanation of these hole types. A separate drill hole layer is necessary for each different depth of hole up to a maximum of 4 types. For example, a board with through-holes and buried via between layers 2 to 3 and 4 to 8 will require 3 different layers with hole locations and sizes.

4.3.2 Drill Tolerances

Hole size tolerance is the allowable range of variation for a PCB's drilled hole from the specified hole size from the PCB design.

Pressfit holes are plated holes with tighter tolerance, which are used for through-hole components that are not soldered to the board.

Hopetimepcb can control the various drill tolerances, as described in Table 11

below. **Table 11**

Plated hole size	+/-3mil
Pressfit hole size	+/-2mil
Non-plated hole size	+/-2mil
Hole location	+/- 3 mil

4.3.3 Slot Size Tolerance

Slots are special holes which have different length from their width. And usually, if their length is longer than twice their width, we will call them “long slots”; otherwise, we will call them “short slots”.

We at Hopetimepcb can control the slots size tolerance according to Table 12 below.

Table 12

Plated short slots	+/-0.15mm
Non-plated short slots	+/-0.1mm
Plated long slots	+/-0.1mm
Non-plated long slots	+/-0.075mm

4.4 Aspect Ratio

Aspect Ratio is calculated by dividing the maximum board thickness with the smallest drilled hole diameter size specified for that board design. Maximum board thickness is the PCB thickness without copper plating, solder or solder mask. The aspect will affect the difficulty of plating. The bigger the aspect ratio is, the harder the plating process will be.

We Hopetimepcb can build the board with aspect ratio listed in Table 13 below:

Table 13

Minimum plated holes size	Available aspect ratio	Maximum available board thickness
0.15mm	<=8 : 1	1.2mm
0.20mm	<=10 : 1	2.0mm
0.25mm	<=12 : 1 (20 : 1 is also available, but need an estimate at first)	3.0mm (If thicker, need an estimate at first)

4.5 Annular Ring

The annular ring is the width of the copper area around a via to connect it with the PCB trace network. Annular ring size is an important PCB design consideration since, during PCB fabrication, many conditions can cause holes to not be drilled perfectly centred. Therefore, it is necessary to design a sufficiently thick ring to allow for manufacturing tolerances and still produce a reliable electrical connection to the via. See Figure 5 below for an illustration. We at Hopetimepcb can fabricate the board with an annular ring listed in Table 14 below.

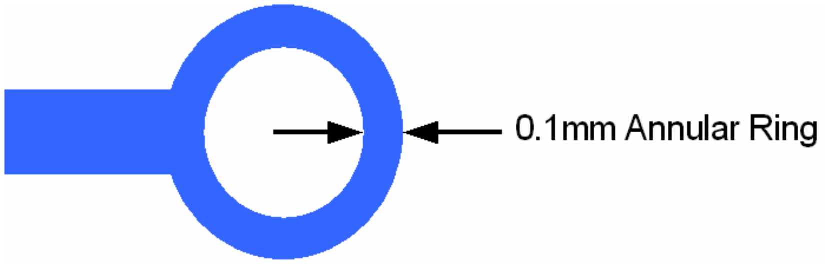

Figure 5: Illustration of Annular Ring

Table 14

Copper weight (oz.)	Minimum annular ring needed
0.5 oz. or 1.0 oz.	4.0mil (using 3.5mil in a few places is Acceptable)
2.0 oz.	6.0mil (using 5.0mil in a few places is Acceptable)
3.0 oz.	8.0mil (using 6.0mil in a few places is Acceptable)

4.6 Tear Drop Pads

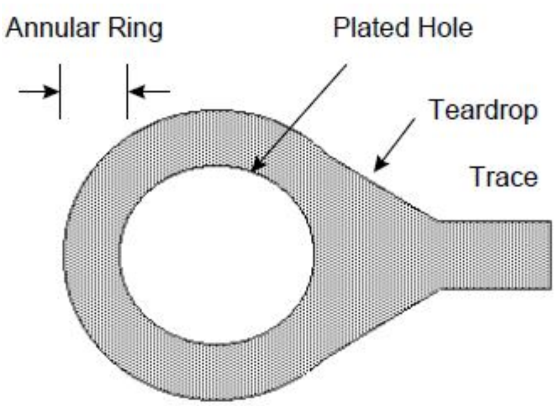

Figure 6: Teardrop Annular Ring

Tear Drop Pads are an extra feature that adds additional copper at the junction of annular rings with PCB traces. It is named this due to the teardrop like shape made with the ring. This teardrop shape design feature enhances structural integrity against possible thermal or mechanical stresses, while also compensating for small fabrication tolerances that may compromise structural integrity. A common type of fabrication error is hole misalignment, which removes too much copper from the junctions of a via pad and PCB trace, causing the possibility for a broken trace connection or too thin of a connection. Tear Drop Pads leave extra copper and lowers the chance of board functional problems occurring due to hole misalignment. We at Hopetimepcb typically add tear drop pads whenever possible if an annular ring is thinner than 7mil since it will increase tolerance limits for mistakes reducing the possibility of PCB problems occurring after fabrication and assembly.

Whether tear drop pads are needed or not depends on the annular ring thickness of the plated holes:

- For annular ring < 7mil tear drop pads are recommended to add, but not necessary
- For annular ring >= 7mil tear drop pads are not needed

4.7 Hole Clearance

Clearance means the space between two features. Usually, there will be slight misalignment between different copper layers, and we will locate holes according to the fiducial marks which we add on the working panels. So, if the space between the hole and other copper features is too narrow, the holes may be drilled too close to the copper features because of misalignment, potentially causing a short-circuit or damaged pads. The higher the layer count is, the bigger the misalignment could be, and the larger the clearance we will need.

The clearance to design between the holes to other copper feature will depend on the layer count, as shown below in Table 15:

Table 15

Layers count	Minimum clearance between the hole to other copper feature
<=6	8.0mil preferred (6.5mil minimum)
8	10.0mil preferred (7.0mil minimum)
>=10	12.0mil preferred (8.0mil minimum)

4.8 Conductor Clearance

Conductor clearance is the distance between traces or other copper elements. Sufficient trace clearance is essential to ensure manufacturing tolerances do not compromise the function of your board. In addition, to ensure the conductor width can match the Gerber, we must compensate (or enlarge) the conductor in Gerber to counteract the effect of undercut, as explained in Section 4.2 Etch Factor. The heavier the copper weight is, the deeper the etching, thus wider conductor clearance is needed to compensate.

The minimum conductor clearance depends on the copper weight, as shown in Table 16 below.

Table 16

Copper weight	minimum clearance (inner layers)	minimum clearance (outer layers)
0.5oz	4.0mil preferred (3.0mil minimum)	4.0mil preferred (3.0mil OK only in a few places)
1oz	5.0mil preferred (4.0mil minimum)	6.0mil preferred (5.0mil minimum)
2oz	7.0mil preferred (6.0mil minimum)	8.0mil preferred (7.0mil minimum)
3oz	10.0mil preferred (9.0mil minimum)	12.0mil preferred (10.0mil minimum)
>3oz	Ask our sales team to verify	Ask our sales team to verify

Copper traces and other features must also be kept at least **0.2 mm (8 mil)** from the board edge for standard tab routing, or 0.4 mm (16 mil) for V-Scoring. Feature-to-edge requirements are intended to protect against potential damage to copper features when individual boards are separated from the fabrication panel.

4.9 Via Holes Treatment

The plating thickness in vias will be determined by the parameters when doing copper plating for the board. Our copper plating can range from 20μm to 30μm. If you require thicker via plating, it will affect the copper weight on top and bottom layers, since the thicker the plating for the via holes, the thicker the top and bottom layers will be plated. If thermal conduction is your concern, we recommend filling the via holes with non-conductive epoxy and then plate over it using the "Via in pad" process. There is a cost to this process, however, it is less costly, and more reliable, than filling the hole with a conductive material.

Solder mask opening (also called solder mask clearance) indicates the area which should not be covered with solder mask oil. These openings are required to be on their own separate layer in the Gerber files for each outer side of your board. For the via holes in your board's Gerber files, if you have not designed a solder mask opening for them, we can treat them following ways outlined in Table 17.

Table 17

Via hole treatment	Limiting Condition	Extra cost
Plugged with solder mask	1. The hole diameter must be smaller than 0.5mm; 2. No solder mask opening on both sides; 3. Finished board thickness should be 0.5mm ~ 2.4mm.	No
Covered with solder mask	N/A	No
Via-in-pad	1. The hole diameter must be smaller than 0.6mm; 2. Finished board thickness is bigger than 0.5mm ~ 2.4mm. 3. The material cannot be PTFE material.	Yes

Plugged with solder mask: The holes will be built as normal through-holes first, and then be filled with solder mask oil. After plugging, light will not be able to go through the holes. When you are using a part with a BGA package, we advise that you plug the via holes under BGA area to avoid a short circuit when assembling.

Covered with solder mask: Solder mask oil will cover the top of the copper pads of the via holes, and solder mask oil may also flow into the holes, but the holes are not filled, and light is able to go through these holes.

Via-in-pad: The via holes will be filled with non-conductive epoxy, and then plated over them. While there are some via holes which are designed on SMD pads, we may also need to build via-in-pad to avoid the risk of leaking tin when assembling certain designs.

4.10 Finished Board Thickness

Finished overall board thickness can be measured from a PCB's top layer to its bottom layer, including the solder mask and copper layers. This dimension is used for designing enclosures for the board. Do not forget to account for component height in your overall design. The maximum board thickness will be, as shown in Table 18 below.

Table 18

Layers count	Maximum board thickness (inches)	Maximum board thickness (mm)
<=2 layers	0.149 preferred (0.177 possible)	3.8 preferred (4.5mm possible)
>2 layer	0.149 preferred (0.236 possible)	3.8 preferred (6.0mm possible)
Note: For 2 layer boards, if the board is thicker than 3.0mm, we may have to build it as a mock 4-layer board since we do not have the suitable substrate to match this thickness.		

4.11 Overall Finished Profile Tolerance

A surfaces profile is a 3-dimensional tolerance zone outline around a surface plane defined by using basic radii dimensions, coordinate dimensions, angular dimensions. The profile tolerance is a uniform boundary around a board surface where there are elements of the surface generated by offsetting each point forming two tolerance zones. Profile tolerances is what controls a feature's form, size, orientation, and sometimes location with profile elements that are curved lines, straight lines, and areas.

We at Hopetimepcb can fabricate a PCB with tolerances listed in Table 19.

Table 19

	imperial	metric
Hole to board edge	±6mil preferred (min. ±4mil)	±0.15mm preferred (min. ±0.1mm)
Board edge to board edge	±6mil preferred (min. ±4mil)	±0.15mm preferred (min. ±0.1mm)
V-score to V-score	±6mil preferred (min. ±4mil)	±0.15mm preferred (min. ±0.1mm)

4.12 Board Outline

As part of your design's Gerber files, we require an outline of your board's final shape. By doing this, you may control your final result and ensure conductors and components have a safe clearance from the edge. We at Hopetimepcb require a clearance of 0.2mm (8mil) for standard routing and 0.4mm(16mil) for V-score edges, between your copper and board edge, to allow for manufacturing tolerance. A larger clearance is preferred, if possible. Edge connectors do not require this clearance. We are capable of cutting complex shapes to your board at no additional cost. Our minimum cutter size 0.8mm, thus all cut-outs, must be larger than this. Otherwise, they may be a slot hole. Our outer board tolerance is +/-0.15mm.

4.13 Edge Bevel

Edge beveling is the process of making a transitional edge between two faces of a PCB usually performed on the outer edge of the PCB. Bevelling is commonly used on edge connectors to allow for smoother insertion into another circuit board socket. The edge connection pins are often called gold fingers because they are plated in gold, and there are several of them grouped together in parallel. Gold fingers are further discussed in Section 6.7 Edge Connector Plating. For gold fingers, we at Hopetimepcb have the options listed below:

- **Available beveling angle**: 20, 30, 45 and 60 degrees
- **Residual thickness after beveling**: >=0.3mm (0.2mm is okay, but ask for an estimate)

You can calculate the depth or residual thickness of the bevelling using the geometry shown below in Figure 7.

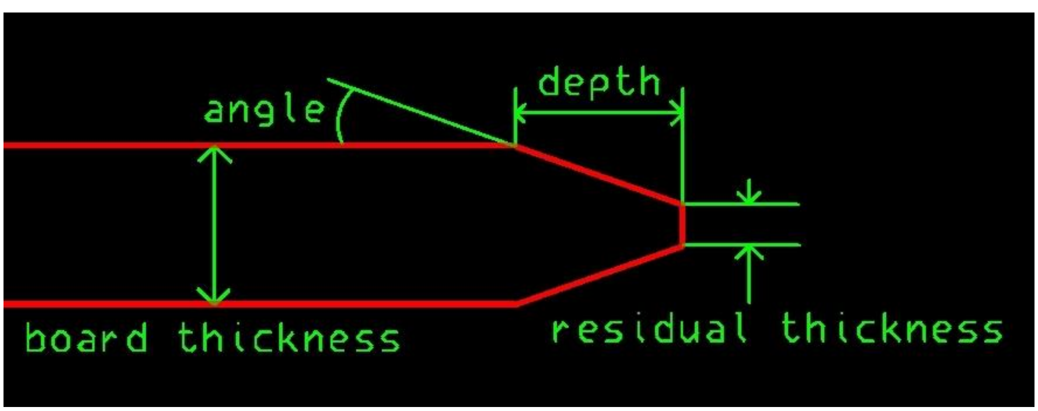

Figure 7: Edge Bevelling of Gold Finger Profile View

$$d = \frac{t - t_r}{2 * \sin a}$$

Where d is depth, a is the bevel angle, t is board thickness, and t_r is residual thickness.

5.0 HDI Technology Capabilities

In addition to our standard PCB options, we at Hopetimepcb can also build HDI boards, including features such as laser-drilled microvias or blind /buried vias. HDI boards commonly include even smaller trace width and spacing requirements than our usual minimums. Our capability for HDI PCB designs is specified in this section.

What are HDI boards?

High-Density Interconnect (HDI) Technology is used in the trace network of multi-layer PCB boards fabrication to interconnect different PCB layers. Hopetimepcb 's manufacturing facility maintains various limits or tolerances, which are recommended to follow for HDI PCB fabrication to avoid problems with your fabricated PCB. Various types of micro-via, such as blind via and buried via in the various layers, are used to make these complex HDI PCBs.

5.1 Tight Trace Width / Spacing (Trace/Space)

The traces of a printed circuit board are a continuous path of copper on which electricity travels. The clearance space on a PCB design refers to the space or gap used to separate traces from other elements on the copper layer. We must maintain a minimum clearance space for each trace width to prevent short-circuiting and allow for manufacturing tolerances. The minimum clearance trace width and space will be dependent on the copper weight of internal and external layers on the board. For our capability of trace width and space on HDI boards, please see Table 20 below.

Table 20

Trace Width / Spacing			
Internal Layer	1/2OZ:	3.0/4.0mil (preferred)	3.0/3.0mil(minimum)
	1OZ:	4.0/5.0mil (preferred)	3.0/4.0mil(minimum)
	2OZ:	5.0/7.0mil (preferred)	4.0/5.5mil(minimum)
	3OZ:	6.0/10.0mil (preferred)	5.0/8.0mil(minimum)
External Layer	1/2OZ:	4.0/4.0mil (preferred)	3.0/4.0mil(minimum)
	1OZ:	4.0/6.0mil (preferred)	4.0/4.0mil(minimum)
	2OZ:	5.0/8.0mil (preferred)	5.0/6.0mil(minimum)
	3OZ:	6.0/10.0mil (preferred)	6.0/8.0mil(minimum)

5.2 BGA

BGA is abbreviated from Ball Grid Array, a form of surface mount technology (SMT). BGA is now chosen more commonly in circuit design. BGA packages were developed due to the market demand for a more robust and convenient package to permanently mount integrated circuits with large numbers of pins. BGA allows for more interconnection pins per surface area than possible on a dual in-line or flat package. Some BGA components even mount integrated circuits with over 100 pins. To achieve this, the entire bottom of a BGA chip is largely filled with interconnection pins. By not limiting connections to the perimeter, connections under the SMD package will increase the efficiency of how space is utilized.

Over the years, Hopetimepcb has accumulated a vast amount of Ball Grid Array (BGA) assembly expertise and has developed a dependable process over time. Currently, our fabrication and assembly facilities use the most up to date BGA placement equipment, and we also utilize X-ray inspection equipment to verify the soldering. We have a proven record of producing BGA circuit boards with excellent yield rates and the highest quality in the electronics manufacturing industry. At Hopetimepcb, we can process BGA packages with the specifications listed in Table 21.

Our skilled workforce employs thermal profiles even for low volume prototype boards, as it is a key function in the BGA assembly process. We carefully review the circuit board files and BGA chip datasheets to make the most appropriate thermal profile for BGA assembly. Lead-free BGA circuit boards pass through a particular lead-free thermal profile to prevent ball issues, which may occur due to using a lower temperature. Alternatively, the costly leaded BGA boards are diverted through specific leaded processes to avoid high temperatures, which lead to pin shorts. We have effective quality inspection procedures in place to offer high-quality services.

We possess high-tech BGA placement equipment, precise BGA assembly processes, and automated x-ray inspection (AXI) system to provide better quality BGA circuit board assembly. AXI is used to identify assembly defects; our team uses 2D x-rays to render 3D images, to verify the issues such as a circuit board broken vias in inner layers and BGA ball’s cold solder break.

Table 21

Capability of BGA package	
Available Size	2mmx3mm ~ 45mmx45mm
Available material	Ceramics, plastics
Available pitch	Minimum 0.4mm (0.35mm is okay, but an estimate is needed.)

5.3 Micro-via

A standard micro-via consists of tiny copper-plated holes with a diameter of 6mil (0.15mm) or less and are made using a laser drill. Micro-vias can connect adjacent layers, allowing a single multilayer PCB to hold more circuit traces, which increases PCB circuit density. Although a single standard micro-via can only link two adjacent copper layers, there will still be significantly more available space for traces. Micro-vias consisting of various types such as blind via and buried via are used in High-Density Interconnect (HDI) PCB designs. Via-in-pad technology is also available, as described in Section 4.9, to further increase circuit density.

A standard micro-via is a highly reliable type of interconnection structure and should be used in a board design whenever possible. For an HDI PCB, the circuit routes need to interconnect several layers to connect components, but a standard micro-via can only be used to connect two adjacent layers. Thus, in order to use micro-via in HDI board design, we need to use the micro-vias in a compound design structure to connect more than two adjacent layers. There are two types of complex structures that use a standard micro-via: staggered and stacked structures. These structures are described below in sections 5.3.3 and 5.3.4.

We at Hopetimepcb can fabricate micro-via with a size ranging from **4mil (0.1mm)** to a maximum size of **6 mil (0.15mm)**. Fabrication of these vias will require separate drill files for each layer pair, with the via hole positions on all of the different layers of a multi-layer PCB; for example, one drill file for all top-to-bottom-layer holes, and another for all second-to-third layer holes, etc.

An HDI PCB board layer with buried and blind vias can be laminated on the same side a maximum of three times. This means that a board can be processed in a maximum of **3 steps** for blind or buried vias. For example, as boards are built from the center, we can have a buried via through a center layer, then a second process can add layers on either side with more buried vias and then a third pressing can add blind vias to the outer layers. Finally, through-holes can be drilled for a total of **4 drilling stages**. See Figure 8 below for an example.

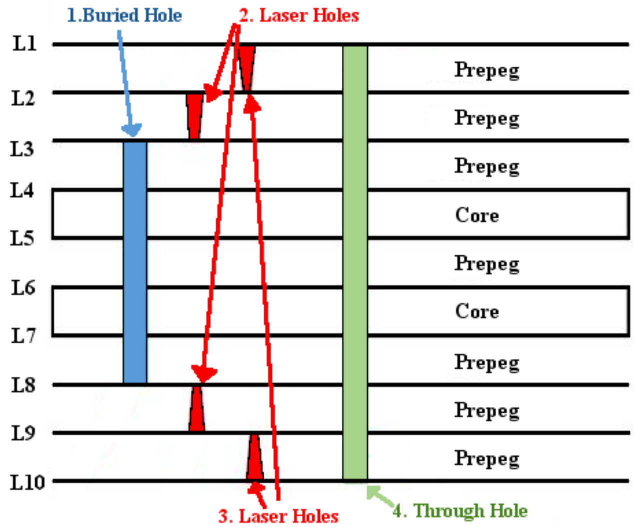

Figure 8: Example of 10-layer HDI board with maximum via steps

Table 22

Summary of HDI Capabilities	
Laser Drilling Diameter	4 mil (0.1 mm) – 6 mil (0.15 mm)
Maximum Blind/Buried Steps	3
Max. Through-Hole Drilling Stages	4
Max. Aspect Ratio for Laser Via Fill Plating	0.9:1 (preferred) – 1:1 (maximum)
Min. Gap Between Blind/Buried Hole Wall & Conductor	8 mil (laminate once) – 9 mil (laminate twice) – 10 mil (laminate 3 times) *preferred 7 mil (laminate once) – 8 mil (laminate twice) – 9 mil (laminate 3 times) *minimum
Min. Gap Between Laser-Drilled Hole Wall & Conductor	7 mil (1+N+1) – 8 mil (2+N+2)
Min Space Between Laser Holes & Conductor	6 mil (preferred) – 5 mil (min.)
Min Pad Size for Laser Holes	10 mil (4 mil hole diameter) – 11 mil (5 mil hole diameter)
Min BGA Pad Size	12 mil (LF-HASL) – 10 mil (Other Surface Finish) – (Preferred) 12 mil (LF-HASL) – 10 mil (HASL) – 7 mil (Other Surface Finish) – (Minimum) *See Section 6.0 for details on surface finish options
BGA Pad Size Tolerance	±1.5 mil (pad < 10 mil) – ±15% (pad > 10 mil) – (Preferred) ±1.2 mil (pad < 12 mil) – ±10% (pad > 12 mil) – (Minimum)

5.3.1 Blind via

A Blind via is a copper-plated hole on an HDI PCB that can connect one of the external layers with one or more internal layers, passing through two or more inner layers. Blind via can only be seen on one side of the board since it can only connect an outer layer to inner layers. However, a blind-via connection cannot pass through the entire board connecting directly to the other outer layer, though it must connect to one of the external layers.

5.3.2 Buried via

A Buried via is a copper-plated hole interconnecting two or more inner layers, but not connecting to an external layer. A buried via is hidden inside or buried within the board, so they are invisible from the outside as a buried via can only pass between the inner layers. Buried vias are only used to connect the various inner layers while not connecting to any outer layer. Thus, the drilling must be done **before the pressing process** of the PCB. For example, 1+n+1, 2+n+2, or 3+n+3, "n" represents a core layer with buried via on this multilayer board, and the numbers represent the couples of laser holes above the core layer. Also, laser holes on these inner layers can be plated closed with copper.

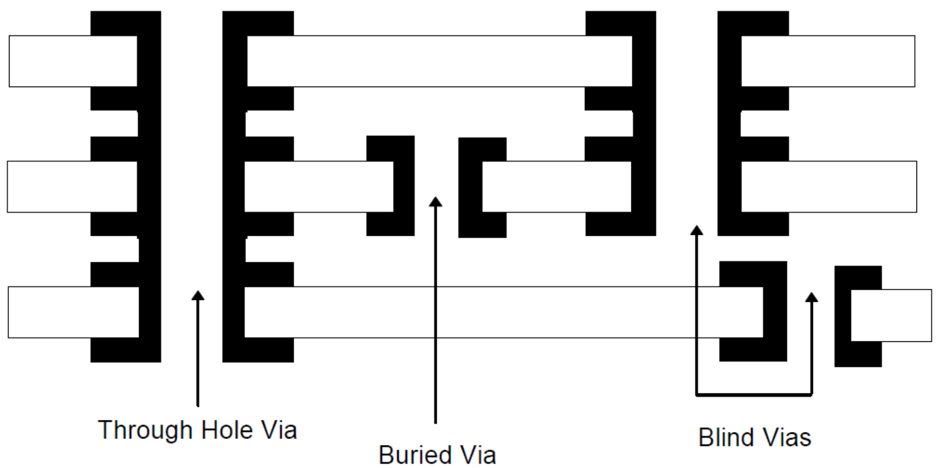

Figure 9: Illustration of via types

5.3.3 Stacked Micro-via

Stacked micro-via is a type of compound design structure which stacks micro-via on top of each other. Stacked micro-via use space efficiently, allowing you to achieve the highest possible circuit density and are easier to use than a staggered structure. Unfortunately, stacked micro-via are less reliable as the via experiences greater thermal stress during the solder reflow step. Consequently, they are considered to be less reliable than even a through-via.

5.3.4 Staggered Micro-via

Staggered micro-via is a type of compound design made by placing micro-via with small offsets from each other between layers. This is the most reliable complex design structure, but it requires slightly more space in the HDI PCB design.

5.4 Multiple Laminations/ Sequential Lamination

Lamination is a technique to manufacture a composite material with multiple layers, in this case, a multi-layer PCB. Laminating board materials into a multi-layer PCB involves fusing two or more different laminate boards using heat, pressure, welding, or adhesives. There are many different processes for lamination, and a variety of PCB materials allow for several lamination processes that may be applied to PCB design.

For non-HDI boards, a single lamination step fuses all layers. For HDI boards, our technique for creating a multilayer PCB is sequential lamination, a process starting on a core layer fused with a conductive and dielectric layer on both sides using multiple pressure passes. Sequential lamination allows both blind and buried via to be created during the build-up process allowing discrete or formed components to be embedded using High-Density Interconnect (HDI) technology for HDI PCB manufacturing. Due to the complexity of the sequential lamination process, it may add considerable cost to board fabrication as well as increasing lead time. Hopetimepcb suggest consulting with one of our representatives if your design for an HDI PCB requires to be constructed using sequential lamination. Although Hopetimepcb has the capability to fabricate multi-layer PCBs with up to 40 layers, it is recommended to design your PCBs with 20 layers at most. A PCB layer refers to a core, copper and prepreg layer in a PCB board, not including the silkscreen, solder mask, and other layers .

6.0 Surface Finish: Options and Requirements

Applying the surface finish is one of the most important yet least understood steps of PCB fabrication. Surface finishes are used to cover and protect a PCB's soldering pads. As a PCB designer, it is essential to understand the various finishes that are available to you, as well as the advantages and disadvantages of each finish.

Surface finish acts as a protective coating to shield copper not covered by a solder mask on a printed circuit board (PCB). A surface finish can be applied to a PCB using one of three primary ways: dipping, immersion, or electrolytic fusion. Dipping involves lowering and dipping parts of an unfinished PCB into a vat of liquid metal surface finish while restricting it to cover only the desired locations with the surface finish. Immersion is a method where a PCB is fully immersed in a bath of liquid metal surface finish, thereby fully galvanizing the PCB. Lastly, in the electrolytic plating process, the PCB is immersed in a solution containing dissolved metal ions known as an "electrolyte". An electric current is then passed through that solution so that metal ions deposit themselves onto the conductive surface of the PCB. Many different types of surface finishes exist, and we at Hopetimepcb offer a variety of the most popular finished as options in our PCB fabrication services.

6.1 HASL/LF-HASL

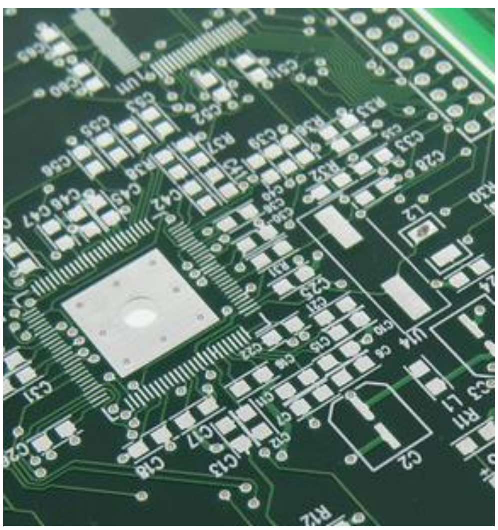

Figure 10: Example of a HASL Surface Finish

Hot Air Solder Leveling (HASL) is the most common type of PCB surface finish used in the industry. HASL finishes are composed of solder, with proportions of approximately 63% tin and 37% lead, commonly referred to as a 60/40 split. The process for applying this finish is begun by dipping the circuit board into a molten pot of the tin/lead alloy after the solder mask has been applied. Next, a Hot Air Leveler (HAL) removes the excess solder, using hot air knives to leave behind only the thinnest possible layer. This remaining layer of solder protects the traces underneath it from corrosion, while easing the task of soldering components to the board by pre-tinning the whole pad. HASL is a very cost-effective surface finish compared to other types of finishes and thus is considered a great choice for general-purpose boards.

Lead-Free Hot Air Solder Leveling (LF-HASL) is similar to HASL in appearance and usage; however, the solder, in this case, contains a mix of 99.3% Tin and 0.6% Copper. This alloy results in a higher melting point for lead-free solder, when compared with leaded solder. LF-HASL is a replacement for leaded solder, used when a lead-free or RoHS compliant PCB is required. Please note that a laminate with higher temperature tolerance is needed for applying this finish; otherwise, the process is identical.

In the past, HASL was one of the most popular surface finish choices due to its qualities as a low cost and robust solution. Recent fundamental changes in the PCB industry, such as new, more complex surface mount technology (SMT), have revealed HASL's shortcomings. HASL is not suitable for use with SMT due to uneven surfaces not being compatible with fine pitch components. Recently, lead-free LF-HASL became available, but now there are other lead-free options more suitable for a high-reliability product.

Pros

- Low-Cost Finish
- Widely Available
- Repairable Layer
- Excellent Shelf Life

Cons

- Uneven Surfaces
- Not Good for Fine Pitch Components
- Thermal Shock
- Not Good for Plated Through-Hole (PTH)
- Poor Wetting
- New Call-to-action
- Solder Bridging
- May Contain Lead (HASL)

Hopetimepcb offers a variety of surface finish options, all for a standard price, including HASL. Please note before ordering that standard HASL finishes contain lead; therefore, boards with this finish will not meet RoHS standards. We suggest planning for the LF-HASL option if you want this type of surface finish.

6.2 ENTEK/OSP

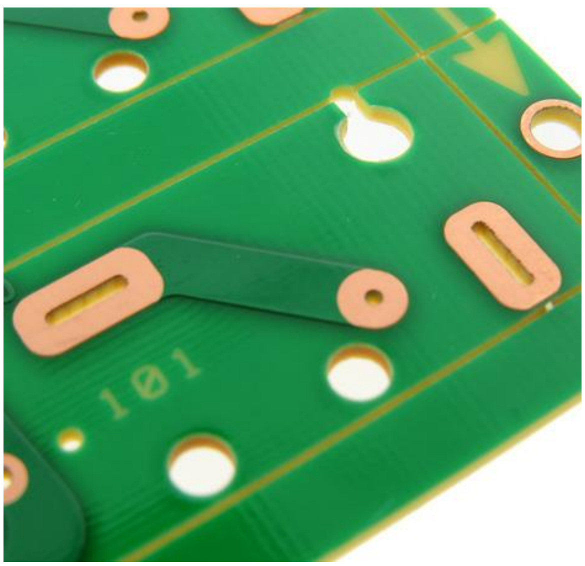

Figure 11: Example of an OSP Surface Finish

Organic Surface Protectant (OSP) is a type of water-based, organic surface finish that is typically applied to copper pads on a PCB. OSP is an organic chemical compound that will selectively bond to copper pads and provides an organometallic layer to protect that copper layer. However, OSP is not as robust as HASL and is very sensitive to small abrasions requiring gloves to avoid scratches. OSP is an environmentally-friendly compound, and extremely green in comparison with other lead-free finishes, which typically have more toxic substances or require substantially higher energy consumption. OSP is a good lead-free surface finish with very flat surfaces, but it has a very short shelf life. To apply this surface finish, you only need to dip the PCB into a chemical bath of the OSP compound, but you must note that this may only be done after all other processes are finished, including Electrical Test and Inspection. OSP is not a standard surface finish and incurs extra cost if chosen.

Pros

- Lead-free
- Flat surface
- Simple process
- Repairable
- Cost-Effective

Cons

- Not good for PTH (Plated Through-holes)
- No Way to Measure Thickness
- Short Shelf Life
- Can Cause ICT Issues
- Exposed Cu on Final Assembly
- Handling Sensitive

6.3 ENIG

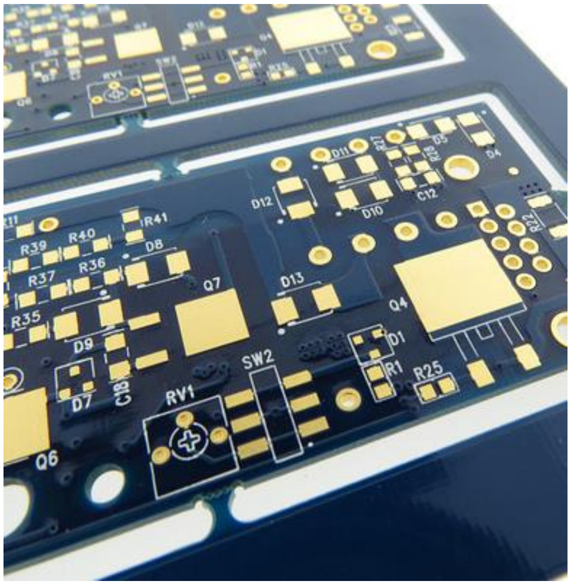

Figure 12: Example of an ENIG Surface Finish

Electro less Nickel/Immersion Gold (ENIG) is a double-layer metallic surface finish that is composed of a very thin layer of gold applied over a layer of nickel. A nickel layer is first plated onto the PCB copper pads using an electroless process, a controlled chemical reaction. Then a gold layer is applied on top of the nickel layer using immersion methods to cover the pads and traces.

Typically, an ENIG surface finish is only applied after a solder mask layer is applied; this is due to cost increases if all copper surfaces are plated with gold. After applying a solder mask layer, the layer of gold will be applied only to what is left exposed, reducing the total area plated with gold. The top layer of gold protects the nickel during the storage period, thus providing an excellent shelf life. This finish also provides a very flat surface, which is ideal for mounting parts such as Surface Mount Devices (SMDs) and Ball Grid Arrays (BGAs). ENIG is a finish preferred by many contract PCB assemblers due to the high electrical conductivity of gold.

As a result of the many advantages listed above, ENIG has become the most highly-used finish in the PCB industry since the growth and implementation of the RoHS regulation. Unfortunately, such great advantages cannot come without drawbacks, and for ENIG, the biggest drawback is it’s complex and sensitive application procedure. If this procedure is not properly controlled, quality issues such as "Black Pad" may occur. Black Pad is a buildup of phosphorous between the gold and nickel layers, which may result in fractured surfaces and faulty connections.

Pros

- Flat surfaces
- Strong
- Lead-Free
- Good for PTH (Plated Through-holes)
- Long Shelf Life

Cons

- Black Pad / Black Nickel
- Expensive
- Not Re-workable
- Damage from ET
- Signal Loss (RF)
- Limited availability
- Complicated Process (two-parts)

We at Hopetimepcb offer ENIG as a standard surface finish option , with a standard price , meaning no additional cost will be applied for selecting this finish . It is our most popular surface due to it’s improved quality at the standard cost.

6.4 Full Body Hard Gold

Figure 13: Example of a Full Body Hard Gold Surface Finish

Full Body Hard Gold also known as Hard Electrolytic Gold, is a layer of gold with hardeners for increased durability, plated over a barrier coat of nickel using an electrolytic process. Hard gold is extremely durable, and so this material is usually applied to high-wear areas, such as edge connector gold fingers and keypads, since its hardness can withstand repeated use; however, due to the high cost of hard gold, and its relatively poor solder-ability, it is rarely applied to solder-able areas.

Full Body Hard Gold is a rarely-chosen surface finish, where the full body of the PCB board is plated with hard gold. In order to apply a Full Body Hard Gold surface finish, an electrolytic process using an electric current or an immersion process is needed, depending on the PCB design. Due to the poor solder-ability of hard gold, a very active flux will be required to solder effectively to the hard-gold-plated pads.

Pros

- Hard, Durable Surface
- No Lead
- Long Shelf Life

Cons

- Very Expensive
- Extra Processing / Labor Intensive
- Use of Resist / Tape
- Plating / Bus Bars Required
- Demarcation
- Difficulty with Other Surface Finishes
- Etching Undercut Leads to Slivering / Flaking
- Not Solder-Able Above 17 µin
- Finish Does Not Fully Encapsulate Trace Sidewalls, Except in Finger Areas

We at Hopetimepcb also offer this finish option, but please note that the associated cost depends on the specific amount of gold plating area that is ordered. Please send all necessary documents and data to Hopetimepcb, and we will review your files before providing you with an estimate.

6.5 Selective Gold

Selective Gold involves using a gold surface finish to plate specific areas on a PCB; note that this does not include applying gold fingers. While we at Hopetimepcb offer this finish option, the associated cost depends on the specific area of gold plating ordered. Please send all necessary documents and data to Hopetimepcb, and we will review your files before providing you with an estimate.

6.6 Double Gold (Full-Body + Selective Gold)

Double Gold is a method that combines the use of both Full Body Hard Gold and Selective Gold methods. We atHopetimepcb offer this finish option, but please note that the associated cost depends on the specific area of gold plating ordered. Please send all necessary documents and data to Hopetimepcb, and we will review your files before providing you with an estimate.

6.7 Edge Connector Plating

Figure 14: Example of Edge Connector Plating

The Edge Connector is the part of a printed circuit board (PCB) with traces leading to the edge of the board, shaped to plug into a matching socket. Applying hard gold plating onto edge connectors as gold fingers is highly recommended. Hard gold is extremely durable; thus, when it is applied to high-use areas, such as edge connectors, they can withstand more wear and tear. We at Hopetimepcb also offer this finish option, but please note that the associated cost depends upon the specific amount of gold plating area that is ordered. Please send all necessary documents and data to Hopetimepcb, and we will review your files before providing you with an estimate. See Section 4.13 Edge Bevel for bevel angles available for gold fingers.

6.8 Wire Bonding (Soft Gold)

Wire Bonding, which uses soft gold, is another type of surface finish commonly referred to as “wire bondable gold”. The softer gold used in this type of surface finish can easily form strong metallic bonds with standard copper traces. The strong bond of gold and copper allows for more conductive connections when leads are soldered to the board. The process for applying a soft gold surface finish is similar to hard gold, using an electrolytic process to apply the finish, but the soft gold process requires that a solder mask first be applied. When soldered, soft gold remains in the alloy and produces a stronger welded joint at the point of soldering or wire bond. We at Hopetimepcb also offer this option, but please note that the associated cost depends on the specific amount of soft gold used in bonding. Please send all necessary documents and data to Hopetimepcb, and we will review your files before providing you with an estimate.

6.9 ENEPIG

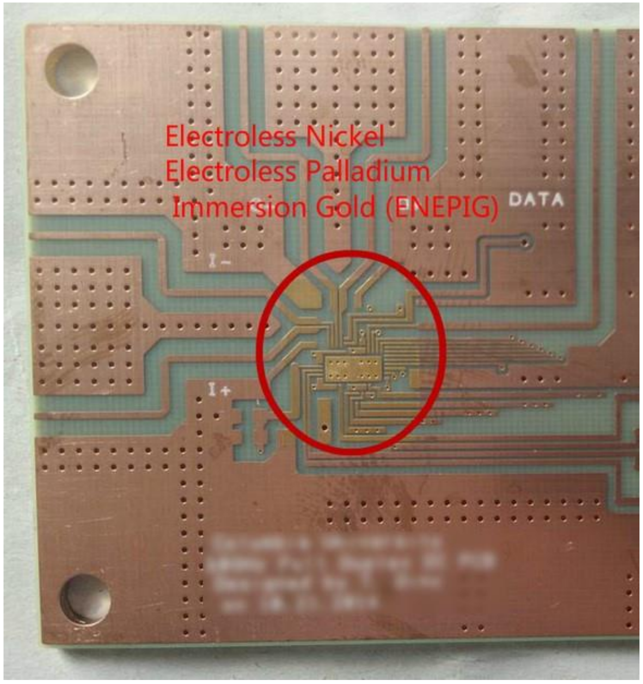

Figure 15: Example of an ENEPIG Surface Finish

Electroless Nickel / Electroless Palladium / Immersion Gold (ENEPIG) is an advanced and complicated surface finish. ENEPIG is similar to ENIG and was developed over a decade ago as a design improvement on the ENIG surface finish. ENEPIG recently became more popular due to a decrease in the price of palladium. The application process of ENEPIG is similar to the ENIG application process, with one extra step: the application of a palladium layer over the nickel layer, before adding the top gold layer. The palladium layer in the middle removes the possibility of “Black Pad”, caused by the nickel layer being corroded by the gold. The ENEPIG method forms a flat, coplanar, hard surface that is good for gold wire bonding, aluminum wire bonding, and provides excellent solder-ability.

Pros

- Flat surfaces
- Strong
- Lead-free
- Good for PTH (Plated Through-holes)
- Long Shelf Life

Cons

- Expensive
- Not Re-workable
- Very Limited Availability
- Complicated Process (Three Parts)

Please note that we at Hopetimepcb usually do not provide ENEPIG as one of our standard surface finish options. If a client is interested in this finish, we ask that they please contact one of our technical support specialists for further details about cost and lead time before finalizing their order.

6.10 Immersion Tin

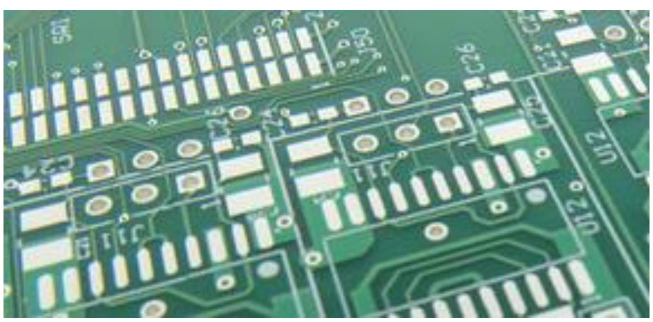

Figure 16: Example of an Immersion Tin Surface Finish

Immersion Tin is a method using a chemical process to apply a very thin tin layer over the copper layer. This method is a lead-free alternative that makes a consistently flat tin surface that solders well and is cost-efficient. This tin layer’s appearance is usually mostly white, so it is also commonly referred to as White Tin and is applied to the copper using an electroless chemical bath. This tin surface finish can protect the copper surface underneath from oxidation throughout the PCB's intended shelf life. However, one problematic aspect of this finish is the strong affinity of tin and copper for one another, which allows the diffusion of one metal into another. This process can cause the formation of "tin whiskers", which are small strands of diffuse tin that can cause shorts and reduce the quality of solder joints, negatively impacting the shelf life and the performance of the PCB.

Pros

- Lead-free
- High Reliability
- Flat Surface/Planar
- Cost-Effective
- Can Substitute for Reflowed Solder
- Top Choice for Press Fit Pin Insertion
- Re-workable

Cons

- Not Good for PTH
- Process Uses Thiourea, a Known Carcinogen
- Not Good for Multiple Reflow/Assembly Processes
- Tin Whiskers
- Could Damage Solder Mask
- Easy to Cause Handling Damage
- Difficult to Measure Thickness

We at Hopetimepcb offer Immersive Tin as a standard surface finish option, with a standard price, meaning no additional cost will be applied for selecting this finish.

6.11 Immersion Silver

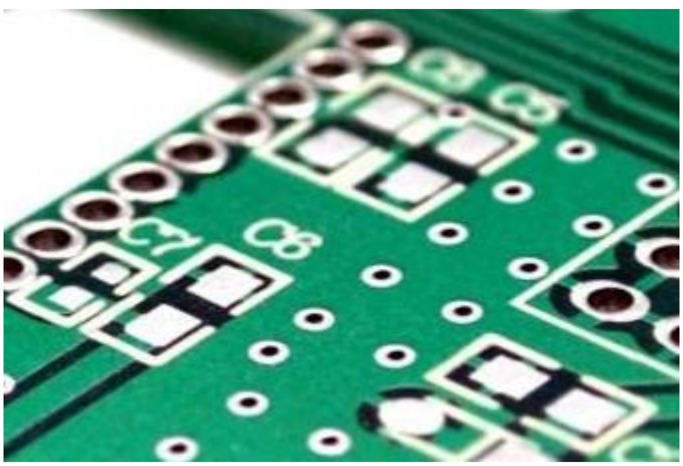

Figure 17: Example of Silver Immersion Surface Finish

Immersion Silver is a method that applies a lead-free layer of silver onto a PCB, to protect copper traces from corrosion. Silver surface finish has excellent solder-ability comparable to solder plating, as well as moderately long shelf life, though still less than some of the other finishes. It is a popular choice due to silver being the most electrically conductive metal available, and ideal for high-speed signals. Silver surface finish can be applied to copper traces with an electroless immersion reaction, displacing the copper layer. The application process for this surface finish forms a very flat surface, which is advantageous for SMD assembly. Silver immersion is a surface finish with benefits that far outweigh its costs, and thus it has gained widespread popularity since the RoHS and WEEE directives came into effect; however, silver immersion is not without its drawbacks. Silver is sensitive to contaminants, both in the air and on the board, and thus it should be packaged as soon as possible to prevent tarnishing.

Pros

- RoHS complaint
- Planar
- Fine pitch
- Cost-effective
- A good alternative to ENIG
- High stability

Cons

- Tarnishes
- Silver Whiskering
- Some Systems Cannot Throw into Micro-Via Aspect Ratios of > 1:1
- High Friction Coefficient/Not Suited to Compliant-Pin Insertion (Ni-Au Pins)

We at Hopetimepcb offer Immersion Silver as a standard surface finish option, with a standard price, meaning no additional cost will be applied for selecting this finish.

6.12 Comparison Chart

Hopetimepcb offers a variety of different surface finish choices, with some standard surface finishes like ENIG , HASL, LF-HASL, Tin Immersion , and Silver Immersion surface finishes for a standard price. Other offered surface finish choices are Hard Gold, Selective Gold Plating, and OSP, etc. at different prices. Please be advised that the associated cost of these non-standard options will be higher, and can depend upon the specific requirements of your project.

Considering both price and performance, the following options are suggested for typical projects:

- Electroless Nickel Immersion Gold (ENIG)
- Immersion Tin
- Immersion Silver

Table 23

	HASL	LF-HASL	OSP	ENIG	Tin	Silver	Hard Gold	Soft Gold	ENEPIG
Deposit	Dipped	Dipped	Dipped	Electro-less/ Immersion	Immersion	Immersion	Electrolytic	Electrolytic	Electro-less/ Immersion
Cost	$	$	$	$$	$$	$$	$$$	$$$	$$
RoHS	No	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Shelf Life	Long	Long	Medium	Long	Medium	Medium	Long	Long	Long

7.0 Solder Mask: Options and Requirements

The solder mask is a piece of the Printed Circuit Board (PCB) fabrication process that is often taken for granted, but this piece is absolutely vital in assuring the quality and functionality of a PCB. The striking difference in quality between boards with and without solder masks is the reason why we at Hopetimepcb include solder masks as a standard finish on all our boards. Having established the importance of solder masks, it follows that a PCB designer should understand the function of a solder mask in some detail and the types offered by their manufacturer.

7.1 – What is a Solder Mask?

A solder mask is a robust, permanent coating that is laminated over the copper traces of a PCB. This layer is sometimes called the "**solder stop mask**" or "**solder resist**". The main function of a solder mask is to prevent the formation of solder bridges during automated mass assembly. A solder bridge is formed when a small bead of solder creates an unintended electrical connection between two or more pads on a PCB. An additional purpose of a solder mask is to protect the copper traces against oxidation, which substantially improves the lifetime of the board.

The solder mask becomes even more essential in the mass assembly of PCBs, where a solder bath is used to tin each copper pad. During this process, the solder mask acts to ensure that no traces of solder are left in unintended areas as a result of the solder bath. Such trace amounts of solder could cause a short circuit between two points on the board, which should be unconnected.

7.2 – Types of Solder Mask

At Hopetimepcb , we offer Liquid Photo-Imageable solder mask, and peel-able solder mask (described in section 7.7). Liquid Photo-Imageable (LPI) solder masks are composed of an ink compound that can be silkscreened or sprayed onto the PCB. The LPI solder mask technique is commonly used with hot air surface levelling (HASL), and requires a clean environment, free of particles and contaminants, for application. After an LPI solder mask is applied, and the PCB is completely covered on both sides with the solder mask, the next stage in the process is curing.

Unlike some older solder masks, LPI inks are sensitive to UV light and can be cured after a short "tack cure cycle", making use of UV light exposure. This curing process cements the solder mask in place permanently. To ensure that the LPI mask is cured in the proper locations, negative film stencils of the top and bottom solder masks are printed using a contact printer and the original Gerber files. The film sheets are printed with black sections corresponding to any areas of the PCB that are to be left uncoated for soldering, or otherwise free from the solder mask. The entire PCB is then exposed to a UV light, which causes the solder mask to cure and harden in any exposed areas, but has no effect on those areas shielded by the black film. After curing is complete, the uncured mask can be washed off of the film-shielded sections, leaving the solder mask in only the desired areas.

Our high-quality solder mask is durable and long-lasting. Usually, we consider that a 1um solder mask layer can withstand 100 VDC. We can ensure the solder mask thickness will be minimum 5um on the conductor corner and minimum 10um on conductor surface. These thicknesses ensure that for the majority of designs, breakdown of the solder mask is not an issue.

7.3 – Solder Mask Design Rules

When starting a new PCB layout, or before submitting your design to Hopetimepcb for PCB Fabrication, it is worthwhile to check your solder mask spacing against the following criteria to ensure manufacturability:

Conductor Overlap: Refers to the relative size of the solder mask, compared to the copper feature. Hopetimepcb requires a **conductor overlap at least 4 mil larger than the feature size**.

Solder Mask Clearance: Related to conductor overlap, the solder mask clearance defines the actual space between the edge of the copper feature and the edge of the solder mask. Hopetimepcb requires a **minimum solder mask clearance of 2 mil**.

Solder Mask Bridge: Refers to the width of the solder masked area that fills a gap between any two pads on the PCB. Hopetimepcb prefers a **minimum solder mask bridge of 4 mil**, with an absolute minimum of 3.5 mil.

Figure 17 shows a visual depiction of the spacing requirements described above for clarity.

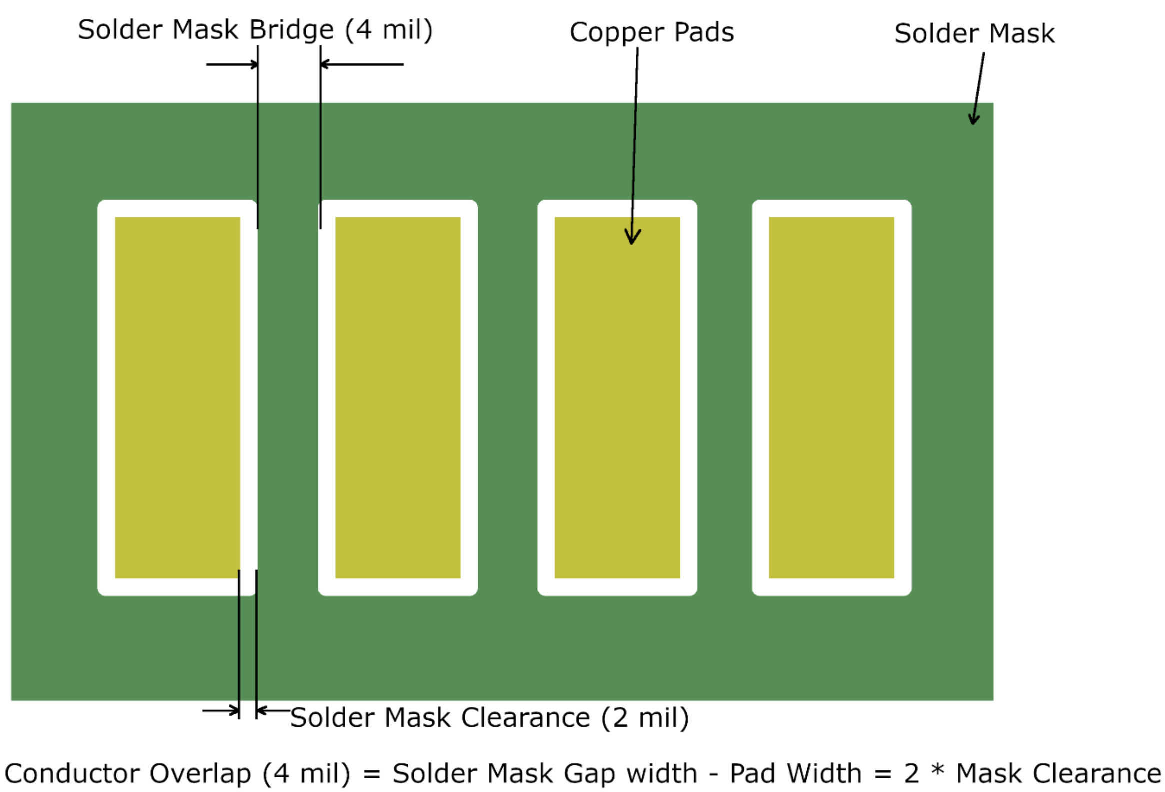

Figure 18: Visual Depiction of Solder Mask Spacing Requirements

7.4 – Solder Mask Colours

The colour of a solder mask has no effect on the electrical performance of a board, but there is an important aspect of contrast between traces, planes, and empty space to consider. Solder masks are available in a variety of colours, including some standard colours and a large variety of custom colours. The colour chosen can make quite a difference in the degree of difficulty for future troubleshooting of the PCB. We at Hopetimepcb have a number of standard solder mask colours available, but custom colours will need to be ordered in advance.

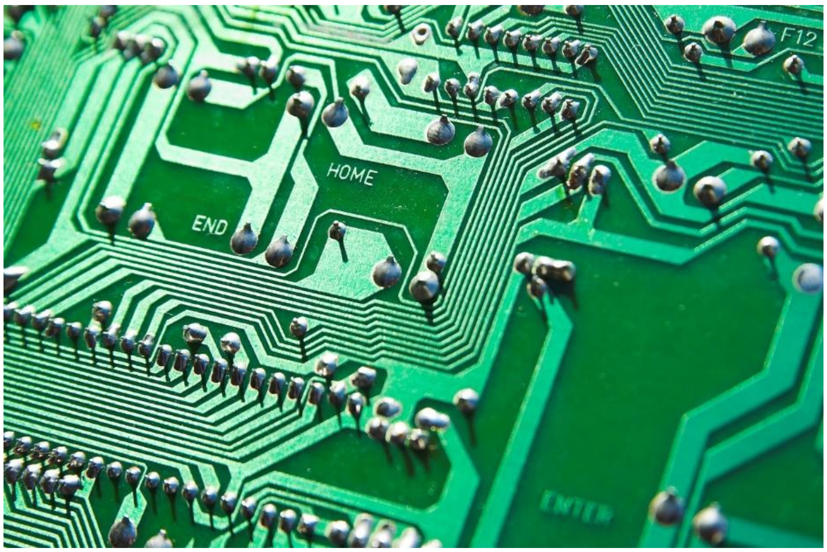

Figure 19: A Standard Green Solder Mask Protecting Copper Traces

7.4.1 – Standard Solder Mask Colours

We at Hopetimepcb offer the following standard solder mask colours: Green, Matte Green, Red, Blue, Matte Blue, Yellow, Black, Matte Black, and White.

Green

In terms of practical performance, green is the best choice for the solder mask colour. It is the industry standard because green is easy on the eyes, allowing for high contrast between traces, planes, and empty space on the PCB. High contrast allows for technicians to easily check, with the naked eye, for manufacturing defects.

Red

Red is a colour that looks professional, but the contrast between traces, planes, and empty space is lower than it is with a green solder mask. Some degree of magnification is suggested when inspecting fine traces on the board for defects with this colour.

Blue

Blue solder masks show a low contrast between traces, planes, and empty space; thus, magnification is mandatory when inspecting for manufacturing defects. These PCBs look aesthetically pleasing and very professional, making them a good choice for fully-developed finished products that will not require much trace visibility.

Yellow

Yellow solder masks show very high contrast between planes, traces, and empty spaces. In fact, the contract achieved by the use of a yellow mask is as high as green; however, the colour is unpopular.

Black

Black is a glossy colour that looks good but has almost no contrast between traces, planes and empty space. Black also absorbs heat, which increases the danger of overheating for sensitive components.

White

White has the lowest contrast and is also the hardest to clean. If possible, we avoid choosing white.

7.4.2 Custom colour

We at Hopetimepcb are happy to offer custom solder mask colours, such as purple and orange, but we do not keep these colours in stock. We will need to purchase these colours prior to production, so we employ a Minimum Order Quantity (MOQ) for custom colours. Expect a marginal increase in both price and turn time for your PCBs if you select a custom colour for the solder mask.

7.5 – Substitutes (North America vs China)

Hopetimepcb offers high-quality Taiyo solder mask oil, which has satisfied the requirements of most clients and is available in North America as well. Taiyo is the world's leading manufacturer of specialty inks and solder masks for printed circuit boards.

7.6 – Solder Mask Tenting

Solder masks can be used for covering the holes in a PCB by a process called tenting, the goal of which is to minimize the amount of exposed conductive material on the surface of the board in the interest of preventing shorts. We recommend tenting only for vias, and not THT pads, because some solder mask oil will flow into the holes during the tenting process, and this might affect the finished hole size.

7.7 – Solder Mask Plugging

We also offer solder mask plugging in which holes are plugged with solder mask oil on both sides. These should be indicated in your Gerber files. Hole sizes for plugging should be **smaller than 0.5mm (20mil)** in diameter.

7.8 – Peel-Able Solder Mask

A peel-able solder mask (PSM) is a type of temporary solder mask that is selectively applied to parts of a PCB. It is used to protect gold plated surfaces from being coated with solder before the Hot Air Solder Leveling (HASL) process, and then it is removed manually. If a peel-able solder mask is used to cover holes on a PCB, the mask's area should be 0.3mm bigger per side than the holes. Also, it should avoid any pads not required to be covered by at least 0.4mm.

8.0 Silkscreen: Options and Requirements

The silkscreen may seem straightforward at first glance, and indeed the basic duties of this PCB layer are relatively mundane when compared with some others; however, errors in the design of the silkscreen layer can severely impact the aesthetics of the finished PCB. Furthermore, a robust silkscreen layer will not only act to improve aesthetics but also it will assist in the troubleshooting and reworking of your PCB project.

A savvy PCB design engineer should understand the responsibilities of the silkscreen layer, as well as the design requirements that will allow for a clear and durable silkscreen. It is also useful to become familiar with the many options in colour and composition for the silkscreen on your PCB project.

This section describes the function and design of the silkscreen layer and proceeds with a discussion of the many silkscreen options that Hopetimepcb makes available to our clients.

8.1 – What is a Silkscreen?

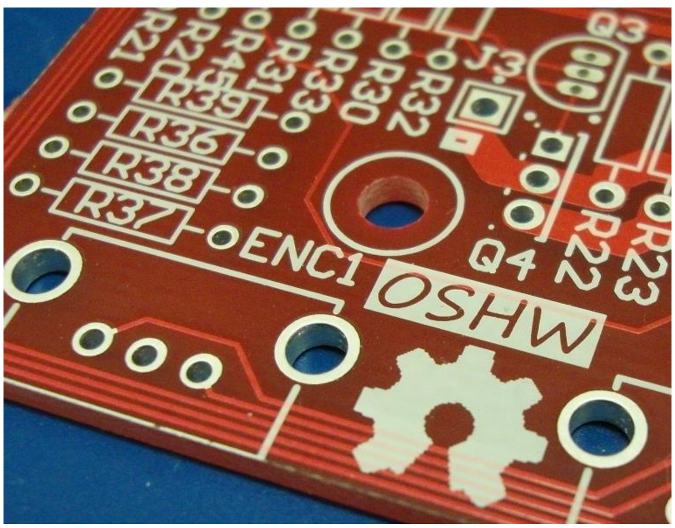

Figure 20: White Silkscreen Printing on Red PCB

The silkscreen is one of the many different layers that can be found within a PCB layout design. The silkscreen layer contains all of the human-readable text to be printed onto a PCB. Such information might include component reference designators, company logos, manufacturer marks, warning symbols, part numbers, version numbers, date codes, etc.

Printable space is quite limited on the surface of a PCB, and so it should be reserved for very useful or important information. For example, the silkscreen is most often used to print a component legend, which shows the locations of various components on the board. Such a legend will ease future troubleshooting and reworking of the board, allowing technicians to easily reference between the circuit schematic and the completed PCB.

Other common silkscreen applications are company logos and PCB design serial numbers.

8.2 –Silkscreen Requirements

In order to ensure that the silkscreen layer can be printed clearly, all silkscreen designs should adhere to the specific limits of their manufacturer’s printing equipment.

Hopetimepcb is able to print silkscreen layers with a ± 7 mil (0.007 inch) margin of error, and so we recommend that your silkscreen layers be designed with at least a 7-mil clearance from all component and board edges. In the case that this margin of error causes silkscreen to be printed over solder-able pads, our production process will automatically correct the mistake.

8.2.1 – Minimum Line Width & Text Height

Minimum line width refers to the lowest possible thickness of marking that a silkscreen printer can produce. We at Hopetimepcb have the capability to print PCB silkscreens with a minimum line width of 5 mil (0.125 mm).

Minimum text height refers to the smallest character that a silkscreen printer can produce, in terms of physical dimensions. Hopetimepcb can print PCB silkscreen text with a minimum height of 30 mil (0.762mm).

8.2.2 – Idents on Copper

Idents on Copper are permanent etchings on the PCB's copper layer that act as labels in addition to, or in place of, the silkscreen layer. Etched lines and letters are more resistant to wear and tear, but may compromise the protections that are in place on the PCB's copper layer.

At Hopetimepcb , we offer to apply etchings onto the PCB's copper layer, but only the ENIG or the HASL surface finish may be used in this case. Other surface finishes increase the risk of silkscreen discolouration and pollution of the exposed copper area.

8.2.3 – Logos

At Hopetimepcb , we certainly can print logos on silkscreen layers. However, keep in mind that all parts of your logo must also adhere to our limits in Minimum Line Width and Minimum Text Height, as outlined in Sections 8.2.1 and 8.2.2. Your CAD program should allow you to measure line width and text height once your logo is imported.

8.2.4 – Certificate Location

At Hopetimepcb, we offer to print certification logos onto your PCBs; simply indicate the location of these certificates in your design files.

8.3 – Silkscreen Types

At Hopetimepcb, we offer a robust heat-curable type of silkscreen ink. Solder mask oil may also be used for printing a silkscreen layer if a custom colour is chosen.

8.4 – Silkscreen Colours

Silkscreen lines may be printed in a wide variety of colours, and so it is useful to divide these colours into two distinct types, the standard colours and the custom colours, for the purposes of our discussion.

8.4.1 – Standard Colours

We at Hopetimepcb currently offer three standard silkscreen colours: **W**hite **Yellow** and **Black**.

8.4.2 – Custom Colours

For the other colour choices, we use custom-coloured solder mask oil for printing. This means that our selection of custom silkscreen colours is the same as that of custom solder mask colours in section 7.2.2.

8.5 – Substitute (North America vs China)

As mentioned in Section 8.3, we at Hopetimepcb offer robust heat-curable silkscreen materials. These materials are equivalent in performance between North American products and Chinese products. Our clients have been satisfied with the performance of our heat-curable standard silkscreen.

8.6 – Multiple Colour Silk on One PCB

At Hopetimepcb, we offer to print silkscreens with multiple colours on the same PCB; however, this process requires extra silkscreen film, which acts to increase the cost. Our capabilities limit the number of distinct colours on a single board to a maximum of three.

8.7 – Serialization

At Hopetimepcb, we are happy to offer silkscreen printing for unique serial numbers on each PCB. If serial numbers were not included in your original design, we could provide our own unique number on each of your PCBs, but we cannot guarantee a particular starting point for serial numbers that we supply.

9.0 Electrical Testing

Electrical testing can be considered the final stage in the PCB fabrication process. During this stage, electrical probes are used to test each unpopulated PCB for shorts, opens, resistance, capacitance, and other basic electrical parameters. Advanced electronic test equipment is used to verify that the net continuity for the PCB is within the expected range, which is provided by the NETLIST file of the PCB design. Testing the net continuity can protect against possible problems with the fabricated boards after the parts assembly step. Electrical tests are especially essential for multi-layer PCBs since the inner layers of the PCB will also require verification. All SMD pads and plated through-holes on the PCB need to be checked for open and short circuits.

9.1 – Electrical Testing Requirements

Hopetimepcb performs electrical tests according to the client's request when specified. If no testing requirements are specified, we will perform electrical testing according to IPC standard, depending on both the design and the cost of the PCB. All PCB electrical testing requires a NETLIST File from the original PCB layout design, which gives information about the manner of electrical connection for each pad on the board, with respect to every other pad.

9.1.1 – Testing File

Electrical testing is used to ensure that no error occurred during the production process that might have introduced unwanted short or open circuits into the PCB's copper layers. Comparing the PCB electrical test data with a NETLIST file ensures that the actual net relationships on the finished PCB match the net relationships in the original design.

If you choose to use the ODB++ format, the NETLIST file will be contained within your design files, and there is no need to supply one separately. If you use a different format, such as Gerber RS-274X, then you will need to supply a separate netlist file in IPC356 format.

CAD-Based Netlist File

At Hopetimepcb, we ask that you provide a CAD-based NETLIST file with your design files, if possible. A CAD-based NETLIST file is one that was generated before PCB routing was determined. This confers an extra level of protection by allowing us to detect errors that may have occurred during production *or* during Gerber file generation/conversion.

Gerber-Based Netlist File

If no CAD-based NETLIST file is available, we at Hopetimepcb are able to generate a reference NETLIST from the client's Gerber data. This type of NETLIST will allow us to detect errors that may have occurred during production, *but not* those that occurred during Gerber file generation/conversion.

9.1.2 – E.T. Stamping

At Hopetimepcb, we will print E.T. (Electrical Testing) stamps onto assembled boards at the client's behest.

9.1.3 – Capabilities

Our testing capabilities are listed below in Table 24:

Table 24

Testing Capabilities	
Minimum Continuity Resistance	0.1 Ohms
Maximum Test Voltage	1000 Volts
Maximum Isolated Resistance	25 M Ohm - 2 G Ohm
Electrical Test Pitch (Fixture)	0.020"
Electrical Test Pitch (Flying Probe)	0.004"

9.2 – Flying Probe Test

The Flying Probe test method makes use of electro-mechanically controlled probes to test points on a PCB one at a time. This electrical testing method is more suitable for testing low-to-mid-sized quantities of physically-smaller PCBs. The Flying Probe method boasts a lower associated cost, compared with the Fixture method, since no expensive programming or fixture setup is required. This meticulous method offers greater accuracy, which translates into increased fault coverage and defect detection over other testing techniques. The drawback to this method is an increased time required for each individual board test, which stems from the fact that the probe must move through the board in a specific sequence.

At Hopetimepcb , we consider the standard threshold for deciding the electrical test type to be 15 manufacturing panels. We will use Flying Probe if order quantity is less than 15 Manufacturing Panels or if the PCB order in question has an area smaller than 1 square meter

9.3 – Fixture (Bed of Nails) Test

The Fixture (Bed of Nails) test method dictates that a test template be created, with pins aligned to the test points in the circuitry of the PCB. Test templates, or *fixtures*, are made by first inserting pins into holes on an epoxy phenolic glass-cloth-laminated sheet. Each pin is aligned for an instant connection with a test point in the circuitry of the PCB, and all pins are fixed permanently. When a bare PCB is pressed down against the fixture, stable connections can be instantly and simultaneously formed with hundreds of test points within the circuitry of the PCB. This testing method requires an investment of time and material cost to create the initial fixture but allows for very rapid testing after that fixture is created.

At Hopetimepcb , we consider the standard threshold for deciding the electrical test type to be 15 manufacturing panels. We may choose the Fixture method if the order quantity is greater than 15 Manufacturing Panels, or if the PCB order in question has an area greater than 1 square meter.

9.4 – Cost

At Hopetimepcb, we perform two different types of electrical testing, and the associated cost differs between the two. These two types of tests are known as the *Fixture* method and the *Flying Probe* method.

The Fixture method, also known as Bed of Nails, requires that a template be created, which will then be used to test each PCB in turn. Building this template, or *Fixture* requires time and additional costs for the necessary materials, but can test large numbers of PCBs very rapidly. At Hopetimepcb, we offer a standard test price on the condition that your design incorporates less than 1,300 test points. In the case that your design boasts over 1300 test points, we will calculate the price depending upon the number of test points required on the fixture.

The Flying Probe method does not require expensive materials for the building of fixtures; instead, a machine sequentially connects electro-mechanically-controlled probes to individual test points on a PCB. This method is not well-suited to very high volume orders due to the time required to the probe to learn the testing sequence, and also to actually move through each board during testing. Since the time required for a Flying Probe test can vary so widely depending upon the design in questions, our cost for this testing method will be calculated depending on the PCB's physical size and layer count.

Hopetimepcb performs 100% electrical testing on all bare PCBs produced at its facilities, and this is included in all quotes unless otherwise specified.

10.0 Controlled Impedance PCB

Impedance Controlled PCBs are fabricated with tightly controlled dimensional tolerances to ensure the PCBs have signal transmission lines with accurate impedances. Upon request, Hopetimepcb offers free of charge design stack-up and impedance calculation assistance to help clients design Impedance Controlled PCBs. In the conceptual level of your PCB design, our team is willing to work with your engineering team to better control impedance by selecting the proper material and stack up.

Electrical impedance is the total amount of opposition given to the electrical current flow in an electrical circuit. Impedance can be calculated using the resistance and reactance of the current in a circuit when a voltage is applied measured and expressed in ohms. Resistance is the opposition to an electrical current flow present in all materials. Reactance is the opposition to an electrical current flow from inherent capacitance and inductance in the electrical circuit interacting with changes in the voltage and current.

Theoretically, for an ideal PCB performance situation, all output energy from a component's output pin would flow through the connected PCB routing into the load input pin on the other end. However, in reality, not all the energy is absorbed by the load, and any leftover energy would be reflected back into the PCB routing, flowing back toward the output source pin. Reflected energy is a concern for AC (Alternating Current) signals because reflected energy can cause noise that interferes with the original signal, changing the signal's waveform. In the worst-case scenario, reflected energy would affect the signal's integrity, resulting in unpredictable PCB behaviour. Impedance matching is not as much of a concern for Digital DC (Direct Current) signals as signals are either high or low, and devices' noise thresholds are usually able to compensate for the small amount of noise from reflected energy.

Energy being reflected back and forth between the source and the load in a PCB can be avoided by impedance matching. In theory, matching impedance should ensure all of the energy emitted from the source flows into the load with little to no reflected energy.

Table 25: Sample Impedance Control Stackup

L1	FOIL		H oz
	Prepreg		
L2			1 oz
	Core		
L3			2 oz
	Prepreg		
L4			2 oz
	Core		
L5			1 oz
	Prepreg		
L6	FOIL		H oz

For a PBC to be considered to have controlled impedance, the routing of the traces must be designed in such a way that the impedance matches the specifications. In order to control impedance in a PCB, both the components and traces of the board must be matched correctly.

Hopetimepcb strives to keep standard PCB materials in stock at all times, including:

- Higher copper weights: 2 oz., 3 oz.; heavier weights (With Lead Time)
- Odd copper weights: H/1 oz., H/2 oz., 1/2 oz.
- Foil: 1/4 oz., H oz., 1 oz., 2 oz., 3 oz.

10.0.1 Impedance Controlled PCB Types

There are two types of Impedance Controlled PCBs, Foil-built, and Core-built. We offer Foil-built PCBs as a default choice since it is more economical and is slightly easier to process.

Foil-built PCB consists of one less core layer than core-built PCBs, as shown in the stack-up (Table 26) below, with **copper** foil layers on the outside. Foil-built boards require various types of foils with different copper weights. However, foils are much easier to acquire than different types of cores. Another factor to consider is that since Foil-built PCBs are covered in **copper** foil, the designer has more choice in regards to dielectric thickness for the outer layers due to using prepreg. Using prepreg boards is less expensive compared to cores, especially if the core is 5 Mils or thinner.

Table 26: Foil Build

	SOLDERMASK			0.5			
L1	FOIL		H oz	0.6	+	1.4	Plating
	Prepreg	2x1080		5.5			
L2			1 oz	1.2			
	Core	5		5			
L3			1 oz	1.2			
	Prepreg	2x1080		5.5			
L4	FOIL		H oz	0.6	+	1.4	Plating
	SOLDERMASK			0.5			
TOTAL BOARD THICKNESS CALCULATED				23.4			
TOTAL BOARD THICKNESS REQUIRED				23 +/- 10%			

A Core-built PCB would have core layers on the outside, so there is no need to use **copper** foil layers. However, material availability depends on the market, so cores with uneven copper weights may be difficult to acquire. In that case, PCB manufacturers need to order cores with higher copper weight then etch down the cores, which is costly since additional labour costs are involved. Using cores with higher copper weight also adds to the material cost.

Table 27: Core Built

	SOLDERMASK			0.5			
L1	FOIL		H oz	0.6	+	1.4	Plating
	Core	5.5		5.5			
L2			1 oz	1.2			
	Prepreg	2x1080		5.5			
L3			1 oz	1.2			
	Core	5.5		5.5			
L4	FOIL		H oz	0.6	+	1.4	Plating
	SOLDERMASK			0.5			
TOTAL BOARD THICKNESS CALCULATED				23.9			
TOTAL BOARD THICKNESS REQUIRED				23 +/- 10%			

10.1 Impedance Calculators

Impedance calculators are computer programs with many different sub-programs assembled together to simulate the expected performance of transmission lines and PCB materials. Impedance calculations have many complicated formulas, and the more complicated the PCB design is, the more complicated the formulas will be. Thus, impedance calculators require a skilled user who knows how to use these programs effectively and the underlying theory.

Hopetimepcb will provide free impedance calculations for PCB designs upon request . For performing impedance calculations, we at Hopetimepcb use the industry-standard Polar Impedance Calculator SI8000 or SI9000. We enter published prepreg values from datasheets and your specific design parameters onto the Impedance Calculator to calculate the impedance of PCB design.

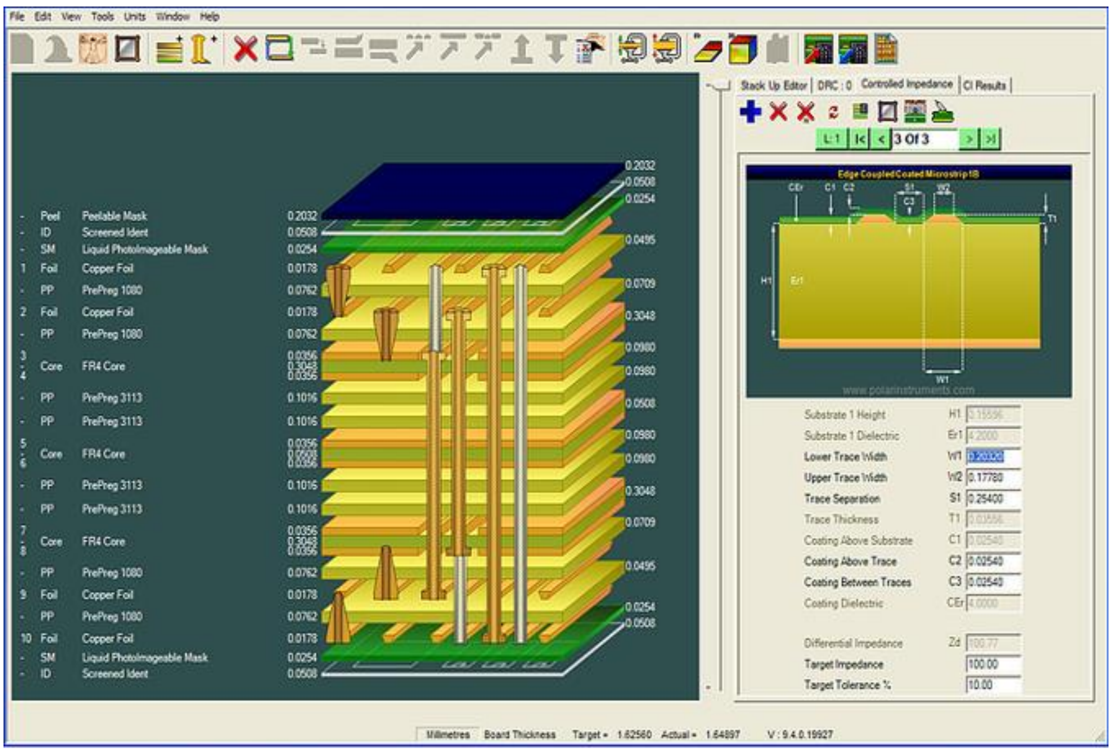

Figure 21: Impedance Calculator Software

10.2 Impedance Models

Impedance Models are diagrams with symbols labelled and showing the parameters for impedance calculation. We at Hopetimepcb offer to provide these Impedance Model diagrams with impedance parameters.

10.3 Impedance Affect Stack-up

Stack-Up is the number and arrangement of different layers in a PCB design. Stack-up design choices and parameters are affected by many factors (such as impedance, physical structure, blind holes and so on). Stack-up structure features affect the impedance on your signal lines. Changing the stack-up parameters can help to achieve your desired impedance, however, there may be other factors in your design that affect the stack-up to consider and balance. Some relationships to consider are listed below.

- The thicker the dielectric thickness is, the higher the impedance value will be.
- The smaller the dielectric constant is, the greater the impedance value will be.
- The thicker the copper weight is, the lower the impedance value will be.
- A thinner impedance trace width means a higher impedance value.
- Greater Inductance means higher impedance.
- Greater Capacitance means lower impedance.

For microstrip and stripline transmission lines, the largest factors that affect the impedance of a line are the dielectric constant of the substrate, the thickness of the copper and the width of the line. Our dielectric constants are listed in Section *3.1 Material Selection & Properties.* Please inquire about our available Rogers RF substrates if desired. Next, you may select your desired copper weight. With these two factors chosen, our impedance calculations can aid you in determining the width of your transmission lines.

10.4 Hopetimepcb TDR Calculations

TDR is "Time Domain Reflectometry" is a measurement technique for determining the characteristics of electrical lines in a PCB by observing reflected waveforms. TDR can also refer to a Time Domain Reflectometer, a type of electronic instrument needed to use time-domain reflectometry to analyze electrical or optical transmission media such as coaxial cable and optical fibre. For example, we can use TDR on a twisted pair wire or coaxial cable to locate faults and discontinuities in wire connection and other transmission media.

Hopetimepcb can use Time Domain Reflectometry (TDR) as a way to test whether impedance in a PCB is matched or not. However, our factory usually uses a different machine to test. A TDR test is performed by applying a very fast electrical step signal to the PCB using a controlled impedance cable and probe. The TDR testing equipment records and graphs the changes in impedance value using the part of the signal, which is reflected back. This graph data shows the impedance values for that PCB or the values simulated by the TDR test with average, standard deviation, minimum and maximum values.

10.5 TDR Coupons

A TDR coupon is a type of small test board on which we can perform impedance testing to verify if the board's impedance matches your request. Testing the impedance trace in a PCB after fabricating is difficult, and the board may be scrapped due to the fixture. Thus, to replace PCB impedance testing, we provide a TDR coupon for simulating the board traces to test the impedance of the PCB. On the TDR coupon, we will include specifications of the impedance value, trace width and expected dimensions of the PCB so that test results of the TDR coupon should match.

Hopetimepcb will provide serialized TDR coupons for each batch of PCB free of charge with our impedance report. If we have a PCB order with 2000 PCBs, of which we build one batch of 1000 pieces first, then build the rest a few days later, then we provide two coupons, one for each batch. The TDR coupon for each batch of PCB are fabricated at the same time as when fabricating the boards. Each coupon is built at the same time as PCB fabrication to ensure that they match your PCB, this way, we provide a TDR coupon to you for impedance testing of each batch.

11.0 – Flexible and Rigid-Flex PCBs

Flexible Printed Circuit Boards (FPCBs), also known simply as Flex PCBs, are a technology that allows for the manufacturing and assembly of printed circuits on a flexible dielectric substrate. FPCBs can be assembled using standard electronic components (i.e. resistors, capacitors, complex ICs), but the manufacturing process of the dielectric material and copper traces allow the board itself to take a non-standard or irregular shape. The option for flexible circuits is a valuable service in many different areas of the electronics industry, from common household devices to complex leading-edge technology; examples include:

- Communications
- Computer Hardware
- Automotive
- Medical
- Aerospace
- LED Lighting

Figure 22 - Flexible PCB

In addition to their convenience in terms of form factor adaptability, flexible PCBs also offer a number of other advantages that have contributed to their recent rise in both demand and accessibility. As size constraints on electronics projects become more stringent with each passing year, the reduction in weight, thickness, and size that comes with switching to an FPCB design becomes all the more appealing. In many small form factor designs, flexible PCBs are used simply to connect various rigid PCBs together in place of bulkier wires or cables. Hopetimepcb can also offer turnkey services for rigid-flex PCBs in the case where flexible sections are used to connect standard rigid PCBs, as described above.

FPCBs also offer advantages in terms of durability that might be surprising to those who are unfamiliar; notably, flexible PCBs are often able to withstand both sudden movement and sustained vibration better than their rigid counterparts. It is also straightforward to mount the FPCB on a heat sink to improve thermal performance.

The following subsections outline Hopetimepcb's PCB fabrication capabilities concerning flexible PCBs, providing various maximum and minimum manufacturing parameters to be used for reference when designing a flexible PCB project. Detailed information on various flexible PCB material is also provided, followed by a discussion of rigid-flex PCBs in terms of their advantages, applications, and the manufacturing thereof.

11.1 Flexible PCB Fabrication Capabilities

As an experienced long-time provider of flexible PCB services, Hopetimepcb offers leading-edge capabilities in FPCB fabrication. Table 28 shows Hopetimepcb's standard fabrication capabilities concerning FPCBs. Any information not included below or in the subsequent subsections should be treated the same as Hopetimepcb's rigid PCB capabilities and requirements. Please note that Hopetimepcb is currently working toward UL certification for FPCBs, but cannot provide this service for the time being.

Table 28

Flexible PCB Technology Matrix	
Available Layer Counts	1 to 4 layers
Maximum Finished PCB Dimensions	220 x 500 mm
Minimum Finished PCB Dimensions	10 x 10 mm
PCB Dimension Tolerance	±0.15 mm (preferred) – ±0.10 mm (minimum)
Minimum PCB Thickness	0.07 mm (single-layer), or 0.13 mm (2-layer)
Copper Weight Limits	0.5 oz. to 2.0 oz.
Surface Finish Options	ENIG (preferred)*
Coverlay Colour	Amber (yellow), Black
Bending Radius	Approx. 15-20 times board thickness
Lead time	14-days for 2-layer design at prototype quantities
Minimum Drill Size	6 mil
Controlled Impedance	Available – ±5 Ω (<50 Ω) or ±10% (>50 Ω)
Trace Width / Spacing	Same as Rigid PCB trace/space, given in Section 4.1

*Other surface finish options available, but require a special estimate and may not be feasible for all projects

11.2 Flexible PCB Materials

11.2.1 FPCB Material Options & Datasheets

Hopetimepcb In an effort to provide all clients with the best possible solutions for their unique and individual projects, Hopetimepcb offers a variety of adhesive-less flexible PCB substrates for your project. Hopetimepcb 's standard FPCB material stock includes the following options:

- Panasonic R-F775
- ShengYi SF305
- ShengYi SF201
- ShengYi SF202

Tables 29 through 32 show a summary of the material datasheet information for the four options listed above. To download the full datasheets, simply click on the options in the list above.

Table 29

Item	Unit	Condition	Typical value
Surface resistance	MΩ	C-24/23/50	1×10^{15}
Dielectric constant(Dk) (1MHz)	—	C-24/23/50	3.2
Dissipation factor(Df) (1MHz)	—	C-24/23/50	0.002
Solder heat resistance	—	E-1/135 288℃ solder float for 1min.	No abnormality
Moisture heat resistance	—	C-96/40/90 260℃ solder float for 1min.	No abnormality
Peel strength RA: 0.018mm(18μm)	N/mm	C-24/23/50	1.3
		260℃ solder float for 5sec.	1.3
Flammability(UL method)	—	A+E-168/70	94V-0
Tensile modulus	GPa	C-24/23/50	7.1
Chemical resistance	—	HCI 2mol/ℓ 23℃ 5min.	No abnormality
		NaOH 2mol/ℓ 23℃ 5min.	
		IPA 23℃ 5min.	
Dimensional stability	%	After etching MD direction	0.030
		After etching TD direction	0.037
		After E-0.5/150 MD direction	0.022
		After E-0.5/150 TD direction	0.027

Note: The sample thickness is RA copper foil 18μm, film 25μm.
Note: The above test methods are in accordance with JIS C 6481 other than the cases flammability is with UL 94.

Table 30

性能项目 Test Item	测试方法 Test Method	单位 Unit	IPC 标准值* Standard	典型值 Typical Value SF305 051813DR	SF305 051813SE
剥离强度 Peel Strength 90°	IPC-TM-650,No.2.4.9 Method A Method C	N/mm	≥0.7 ≥0.525	1.1 1.0	1.3 1.2
热应力 Thermal Stress	IPC-TM-650,No.2.4.13	-	Pass	Pass	
尺寸稳定性 Dimensional Stability	IPC-TM-650,No.2.2.4 Method B	%	±0.15	MD: -0.0684 TD: 0.0691	MD: -0.0418 TD: 0.0354
耐化学性 Chemical Resistance	IPC-TM-650,No.2.3.2	%	≥80	＞85	＞85
吸水率 Moisture Absorption	IPC-TM-650,No.2.6.2	%	≤4	0.04	0.04
体积电阻率 Volume Resistivity	IPC-TM-650,No.2.5.17	MΩ-cm	$\geq 10^6$	1.5×10^8	2.0×10^8
表面电阻 Surface Resistance	IPC-TM-650,No.2.5.17	MΩ	$\geq 10^5$	5.0×10^6	4.5×10^6
介电常数 1MHZ Dielectric Constant	IPC-TM-650,No.2.5.5.9	-	≤4.0	3.6	3.6
介电损耗 1MHZ Dissipation Factor	IPC-TM-650,No.2.5.5.9	-	≤0.04	0.031	0.03
电气强度 Dielectric Strength	IPC-TM-650,No.2.5.6.1	V/μm	≥80	134	140
耐折性 MIT	JIS C 6471 R0.8×4.9N	Times	-	＞600	＞350

注释 Explanations:* Certified to IPC-4204/2 Copper Clad Polyimide Dielectric with Epoxy Adhesive

Table 31

性能项目 Test Item		试验处理条件 Treatment Condition	单位 Unit	性能数据 Property Data IPC 标准值* Standard	典型值 Typical Value SF201 0512SE	SF201 0518SR
剥离强度 Peel Strength (90°)		A	N/mm	≥0.525	1.0	0.9
		288℃,5s		≥0.525	1.0	0.9
耐折性（MIT 法） Folding Endurance(MIT)		R0.8×4.9N	次 Times	-	＞10000	＞10000
热应力 Thermal Stress		288℃,20s	-	-	无分层、无起泡 No delamination	无分层、无起泡 No delamination
尺寸稳定性 Dimensional Stability	MD	E-0.5/150	%	±0.2	±0.1	±0.1
	TD				±0.1	±0.1
耐化学性（剥离强度保持率） Chemical Resistance		暴露化学品后 After Chemical Exposure	%	≥80	＞90	＞90
介电常数（1MHz） Dielectric Constant(1MHz)		C-24/23/50	-	≤4.0	3.2	3.2
介质损耗角正切（1MHz） Dissipation Factor (1MHz)		C-24/23/50	-	≤0.01	0.008	0.008
体积电阻率（湿热） Volume Resistivity		C-96/35/90	MΩ-cm	$\geq 10^6$	4.5×10^8	5.0×10^8
表面电阻（湿热） Surface Resistance		C-96/35/90	MΩ	$\geq 10^5$	3.0×10^6	3.5×10^6

注释 Explanations: C = 湿热处理条件 Humidity conditioning;
E = 恒温处理条件 Temperature conditioning.
* Certified to IPC-4204/11 Copper Clad Adhesiveless Polyimide

Table 32

性能项目 Test Item		试验处理条件 Treatment Condition	单位 Unit	性能数据 Property Data		
				IPC 标准值 Standard	典型值 Typical Value	
					SF202 0512DE	SF202 1012DE
剥离强度 Peel Strength (90°)		A	N/mm	≥0.5	1.2	1.4
		288℃,5s		≥0.5	1.2	1.4
耐折性（MIT 法） Folding Endurance(MIT)		R0.38×4.9N	次 Times	-	＞80	＞50
热应力 Thermal Stress		288℃, 20s	-	-	无分层、无起泡 No delamination	无分层、无起泡 No delamination
尺寸稳定性 Dimensional Stability	MD	E-0.5/150	%	±0.2	±0.1	±0.1
	TD				±0.1	±0.1
耐化学性（剥离强度保持率） Chemical Resistance		暴露化学品后 After Chemical Exposure	%	≥80	＞85	＞85
介电常数（1MHz） Dielectric Constant(1MHz)		C-24/23/50	-	≤4.0	3.2	3.3
介质损耗角正切（1MHz） Dissipation Factor (1MHz)		C-24/23/50	-	≤0.01	0.007	0.008
体积电阻率（湿热） Volume Resistivity		C-96/35/90	MΩ-cm	$\geq 10^{6}$	4.5×10^{8}	3.5×10^{8}
表面电阻（湿热） Surface Resistance		C-96/35/90	MΩ	$\geq 10^{5}$	1.5×10^{6}	2.0×10^{6}

注释 Explanations: C = 湿热处理条件 Humidity conditioning;
E = 恒温处理条件 Temperature conditioning.

11.2.2 Adhesive-Less vs. Adhesive-Based Materials

Two broad types of FPCB base materials are available for use in flexible PCB projects. Similar to their rigid counterparts, FPCBs are composed of copper cladding bonded to a dielectric material that separates the copper layers. Originally, the standard method for bonding copper foil to flexible dielectric material was to use flexible epoxy or acrylic-based adhesives. More recently, adhesive-less options have become available on the market, where the copper foil is attached directly to the dielectric core without the use of adhesives.

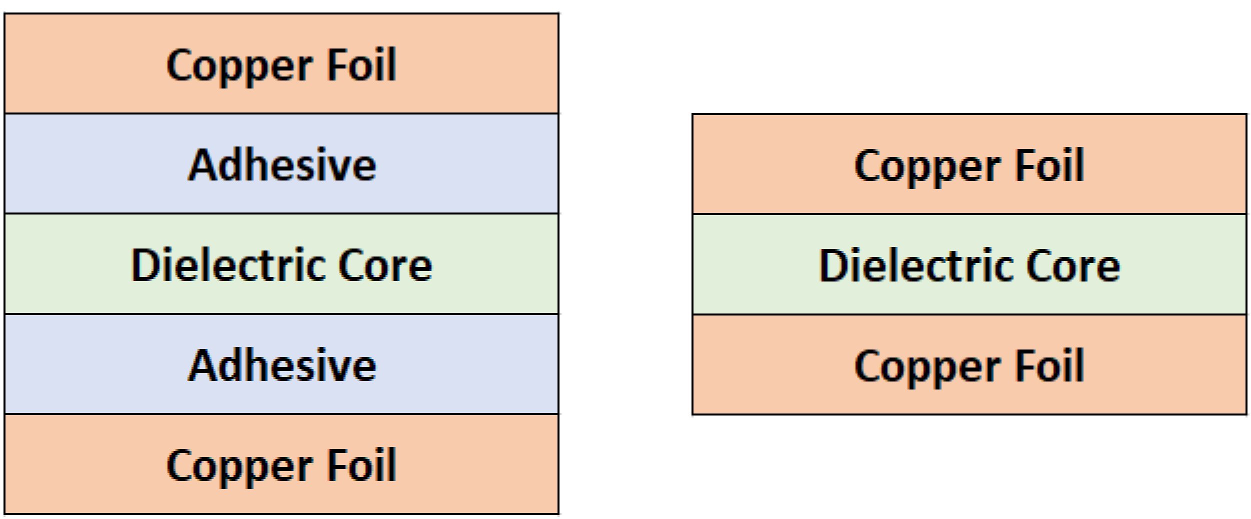

Figure 23 - Simple Stackups for Adhesive-Based FPCB Material (Left) and Adhesive-Less Material (Right)

Adhesive-less flex core materials are generally favoured over adhesive-based options for the superiority they provide in many areas of finished PCB characteristics. Though adhesive-based materials can sometimes offer a stronger bond between the copper foil and the dielectric, this is highly dependent upon the specific adhesive used. Meanwhile, adhesive-less materials generally offer higher bend radius, lower minimum finished board thickness, stronger plated hole integrity, more accurate controlled impedance characteristics, and improved heat performance. In order to meet IPC 2223C guidelines regarding FPCBs, adhesive-less materials are often required . That being said, Hopetimepcb can source specific adhesive -based materials if required for a particular project , but there will likely be some additional cost and lead time associated with the procurement.

11.3 Rigid-Flex PCBs

Rigid-Flex PCBs are hybrid devices, normally consisting of multiple standard PCBs made of rigid FR4 material, where each rigid PCB is connected by a length of flexible PCB, as shown in the image below. This arrangement is superior to simple wire or cable connections in many cases due to its adaptable form-factor as well as its improved mechanical connection strength.

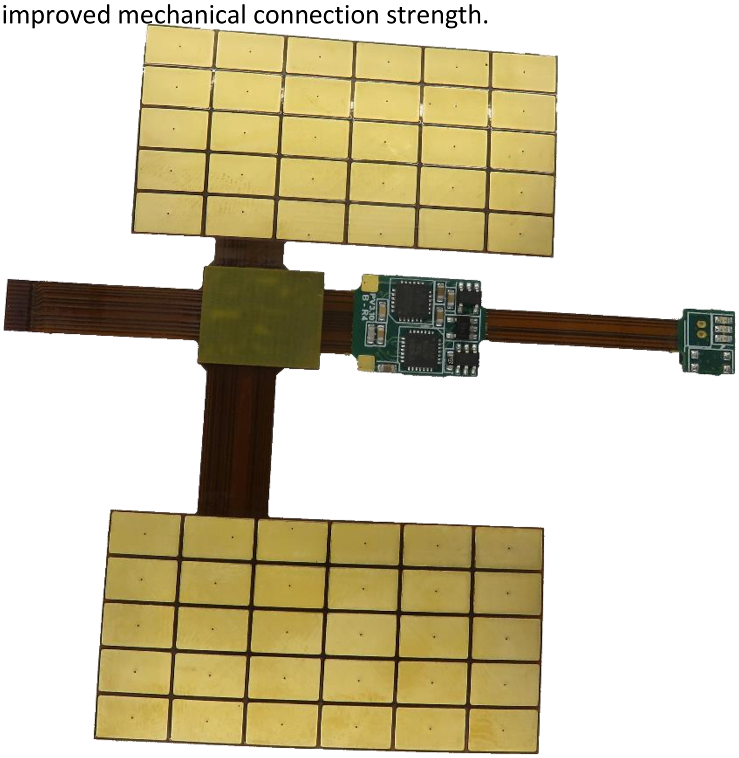

Figure 24 - Example of a Rigid-Flex PCB

Rigid-flex PCBs allow the complex pieces of a design to be laid out using standard rigid PCB practices while still offering many of the advantages in adaptability and reduced form factor that come with FPCBs. That being said, there are a few key requirements to keep in mind when designing a rigid-flex board for fabrication:

- The number of copper layers for the rigid and flexible portions must be exactly equal
- A minimum space of **1.0 mm (40 mil)** from all board edges must be kept entirely free of holes and components to allow for proper bonding of the rigid sections to the flexible sections

12.0 Panelization

Panels consist of multiple circuit board designs combined together in order to form a single large board called a “panel”. Some designs strive to fit multiple circuit boards onto a single panel, as more circuit boards on one panel can be more cost-efficient. There are two main types of panelization, simple panels (also called arrays), and complex panels. Simple panels have the same circuit board repeated on one panel, while complex panels are composed of different circuit boards on the same panel.

Panelization is a way to safely manufacture multiple PCBs simultaneously while keeping the PCB’s separation process afterwards as smooth as possible. In terms of large volume PCB manufacturing, the cost will be lower based on how many boards fit onto a single panel as the process will be more efficient. Higher panelization efficiency requires carefully setting PCB design sizes for effective panel space usage. For the highest efficiency, circuit board length and/or width should be integer divisors of default panel size when margin and spacing are taken into account.

We at Hopetimepcb offer panelization services to clients for PCB fabrication, but we do not offer any set default panel sizes. We only require the PCB design to be panelized to have a circuit board size within a range of 50mm*50mm to 500*500mm. Hopetimepcb primarily produces low and medium volumes of PCB. Therefore when fabricating, we typically will automatically adjust for the most appropriate panel size for better material utilization. With these smaller volumes of PCB fabrication, the cost-saving from higher production efficiency is not significant; therefore, Hopetimepcb does not offer cost-saving for using panelization. However, producing multiple designs simultaneously does save a portion of the shipping costs. You are not required to layout the panels yourself as our technicians will do this to optimize our process.

12.1 Fiducials

Fiducial markers, also known simply as fiducials, are marks meant to be seen on images produced by an imaging system. Fiducials, also known as circuit pattern recognition marks, are a point of reference for SMT placement equipment to accurately locate and place parts onto the printed circuit boards. By measuring the fiducials locations relative to the board layout, the machine can compute how much each part must be moved relative to the layout to ensure accurate part placement.

We at Hopetimepcb can apply these fiducial marks onto our fabricated panelized boards if specified. Multiple fiducial marks are needed as common measurable points to precisely determine a board's orientation. At least three fiducial marks placed asymmetrically are needed to allow machines to determine the offset of both the X and Y axis and determine if the PCB has rotated during clamping. If a clamped board has been rotated, the SMT placement machine will automatically rotate parts to match. Additional fiducial marks are required to further fine-tune the targeting for placement of parts such as ball grid array packages. The lower end boards do not require as much precision and may only have two fiducials or use screen printed fiducials.

12.2 Tooling Holes

Tooling holes on PCBs are added for a variety of reasons depending on the requirements of the equipment being used. They are mainly used to aid in PCB alignment and orientation for drilling and routing during assembly. We at Hopetimepcb apply tooling holes to panelized boards when specified, and our hole size must be in the range of 0.8mm to 4.0mm. For typical tooling holes, Hopetimepcb suggests to drill tooling holes at the standard size of 2.0mm.

12.3 V-Score

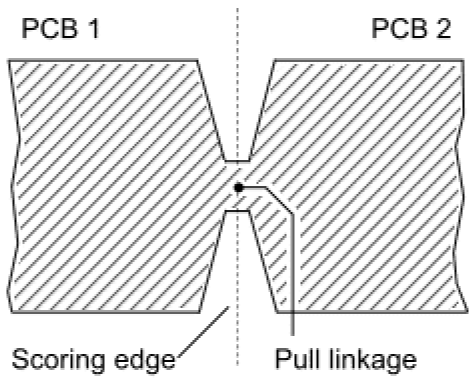

Figure 25: V Score Profile View

Hopetimepcb can apply scores to our fabricated panelized boards when requested. Scores are essentially V-shaped grooves made on a panel with multiple boards so the PCBs can be easily separated. Typically, scores are V-shaped grooves with 1/3rd removed on top and 1/3rd cutaway on the bottom, leaving 1/3rd of the material remaining in the middle to hold the PCBs together in a panel. For thin panels, **thinner than 0.8mm/31mil** (down to min. 0.6mm/24mil), one-sided scoring is recommended. V-scores must be applied as straight lines onto PCB boards, so various restrictions exist for use. Scores must be straight V-shaped lines to pre-separate the circuit boards and are formed with precision cutting tools; therefore, scores can only be used for square or rectangular PCBs.

12.4 Tab Routing

Figure 27a: Photo of Tab Routing Holes

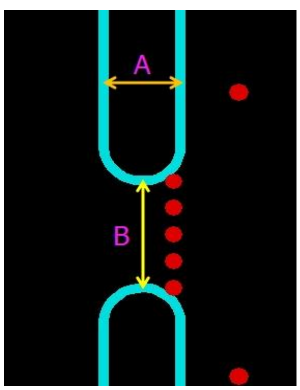

Figure 27b: Tab Routing Dimensions

Hopetimepcb can also apply tab routing to our fabricated panelized boards if required. Tab routing is a popular PCB panelization approach that uses small tabs on all four sides of a PCB to attach to the other boards or rails. Tab routing can include tabs with or without perforations. Of these, the perforated type allows PCBs to be separated manually. A key advantage of using tab routing is that non-rectangular boards can be produced, all panelized circular or irregular shaped PCB use tab routing. However, a disadvantage of tab-routing is the additional board material that is required, which increases fabrication costs. PCBs can be removed from panels before or after assembly because panels allow for easier assembly, the usual approach is to remove PCBs after assembly. When removing PCBs from panels after assembly, extra care is taken so that the parts are not damaged. We at Hopetimepcb can panelize your design for you to optimize our production. However, if you require your own tab design, we require that (as seen in Figure 27b) dimension "A" be a minimum of 0.8mm but recommend 1.6mm. Dimension "B" depends on the thickness and number of layers of your board. Our customer service team can assist you with this step to suit your design. Tab routing is our recommended routing method.

13.0 Report Types and Report Writing

The standard final report for a PCB includes a variety of sections. Hopetimepcb will provide PCB reports with the following sections. These sections are described in detail below.

- Products final audit report
- Certificate of Compliance
- Test Report
- Solderability Test Report
- Cross-section Record
- Impedance Report

13.1 Products Final Audit Report

The Products Final Audit Report section of the PCB report contains information on the fabrication and assembly for that PCB. The PCB order data is recorded here, such as the Bill of Materials with parts and part quantity needed for each PCB. The fabricated PCB specifies physical data such as material used, board thickness, copper thickness , hole size, board dimension , bow and twist, line width & space. Hopetimepcb includes all the materials for PCB fabrication and parts for assembly with the PCB order data in this report.

13.2 Certificate of Conformance (C of C)

The Certificate of Conformance is the section of the PCB report with material data and test records that certify how the used materials meet specifications for various directives or guidelines. Firstly, the material composition data provides proof of the PCB being RoHS Compliant. Secondly, material property data will prove that this PCB complies with the UL Standards needed for UL certification and UL flammability. Hopetimepcb will include these official certifications with our final PCB report and provide necessary PCB data in order to prove conformance with the certification date and a date code included.

13.2.1 RoHS Compliance

Hopetimepcb will provide data with confirmation of RoHS Compliance in our C of C (Certificate of Conformance) section of the PCB Report when RoHS Compliance is required. When an order is required to be RoHS Compliant , we will carefully regulate the use of high concern substances like cadmium , lead, and mercury during PCB fabrication and assembly. (See section 14.0 RoHS Compliance for more information)

13.2.2 U.L. Certificate

Hopetimepcb can include UL Certificates and related verification data in our "Certificate of Compliance " section of the final PCB Report. To obtain a UL Certificate, it is required that the Underwriters Laboratories test representative samples of the PCB and that these samples comply to requirements given on UL Standards. Underwriters Laboratories Inc. test a PCB for thermal shock, bond strength, and plating adhesion and investigates the fabrication and assembly process before approving UL recognition.

13.3 Test Report

Hopetimepcb will perform a variety of tests over the course of PCB fabrication and assembly with the reports for these tests included in this section. An Electrical Test Report with the result from a Flying Probe Test or a Fixture (Bed Board) Test is always included in this section (See section 9.0 Electrical Testing for an overview of these methods). Also, when required, Hopetimepcb can perform a HI pot Test and record results in this report section. Lastly, we can perform functional tests when test parameters are specified by the client, providing a functional test report afterward.

13.4 Solderability Report

The Solderability Test Report is a description of our solderability test results, including test type, factors, and results. This report shows our testing methods, the test conditions, and how we measured and compared the final test results to reach a conclusion on the PCB's solderability.

13.4.1 Peel Strength Report

Peel strength is a way to measure the bond strength of a material, typically an adhesive measured with average load per unit width of bond line. If required, we can perform a peel strength test and provide a report on this PCB's peel strength.

13.5 Cross Section Report

The Cross-Section Report includes a variety of cross-sections, micro-sections, and X-sections showing the physical design data for this PCB. In this section of the PCB report, physical data such as copper thickness, hole wall thickness, and solder mask thickness will be included. Hopetimepcb will provide cross-sections and various other diagrams showing the PCB's various layers and micro-via.

13.6 Impedance Report (TDR)

Impedance Report is an optional report that Hopetimepcb will provide if impedance data is required by the client. An impedance test is performed using a Time Domain Reflectometry (TDR), the test results are recorded in this report along with the PCB's stack-up. A TDR test is performed by applying a very fast electrical step signal to the PCB using a controlled impedance cable and probe. The TDR testing equipment records and graphs the changes in impedance value using the part of the signal, which is reflected back. This graph data shows the impedance values for that PCB or the values simulated by the TDR test with average, standard deviation, minimum and maximum values. (See section 10.0 Controlled Impedance PCB for more information)

14.0 RoHS Compliance

RoHS (Restriction of Hazardous Substances) is a mandate for restricting the use of certain hazardous substances in electrical and electronic equipment. RoHS aims to restrict the use of hazardous substances such as cadmium, lead, and mercury in the manufacture of electronics and electronic devices. RoHS Compliance essentially refers to acting in full accordance with RoHS regulations while keeping documentation on the testing of RoHS controlled substances.

14.1 PCB Raw Material

For a PCB to be considered RoHS compliant, it cannot have restricted raw materials over a certain limit in its materials. Hopetimepcb strives to ensure that any of our fabricated PCB boards that have been requested to be RoHS Compliant conform to the RoHS given maximum limits. Requesting RoHS compliance will limit your choice of dielectric board material. The following are restricted materials in the PCBs:

- Lead (Pb): < 0.1%
- Mercury (Hg): <0.1%
- Cadmium (Cd): <0.01%
- Hexavalent Chromium(Cr6+): <0.1%
- Polybrominated biphenyls (PBB's): <0.1%
- Polybrominated diphenyl ethers (PBDE's): <0.1%
- Bis(2-Ethylhexyl) phthalate (DEHP)
- Butyl benzyl phthalate (BBP)
- Dibutyl phthalate (DBP)
- Diisobutyl phthalate (DIBP)

14.1.1 Solder

Solder is a fusible metal alloy that can be melted and used to make electrical connections between electronic components. Solders can be separated into two main types: lead-free solder, which is RoHS compliant, and non-RoHS compliant leaded solder. Leaded solder alloys commonly used for electrical soldering are 63/37 Sn-Pb, which has the lowest melting point (183 °C or 361 °F) of all the tin-lead alloys. However, due to the use of lead, this type of solder is not RoHS compliant.

For the assembly of our turn-key orders, we at Hopetimepcb use lead-free solder that fully complies to RoHS guidelines. Our solder used contains only Tin (Sn), Silver (Ag), and Copper (Cu) as raw materials. Various lead-free solders were developed to provide RoHS compliant solder that can be used for commercial PCB assembly. Currently, the most popular commercial lead-free solder is an alloy of Tin-Silver-Copper due to its reduced melting point of 217 ˚C.

15.0 Assembly Considerations

The last parameter a circuit designer should consider is the population of their PCB with electrical components. Modern PCB manufacturing automates the process of soldering components to boards, which allows for faster and lower-cost production for boards with many components. For a small quantity of boards with few components of manageable size, hand soldering is more efficient as it does not incur machine setup costs. Therefore, to provide the best possible price for our service, not all orders use automated assembly.

Pitch is the distance between SMD pads measured center to center. At Hopetimepcb, the smallest component size we can place is **0201** imperial (0603 metric) as our component pads cannot have a finer pitch than **0.2mm (8mil)** to prevent shorts in the soldering stage. Moreover, your component outlines and pads should also have a clearance of at minimum **0.2mm (8mil)** between components, and we recommend more if possible. Fine pitch assembly requires stricter tolerances and more precise manufacturing thus we charge an additional cost for component assembly with pitches of **0.5mm (20mil)** or less. Another cost consideration is placing components on both sides of the board. While we are fully capable of two-sided assembly, the extra step adds extra cost, therefore should only be designed when necessary.

For assembly service, please ensure you include a “pick and place” file with your order as described in *Section 2.5 Centroid File / Pick and Place File* as well as a Bill of Materials as described in *Section 2.6 Assembly Bill of Materials (BOM)*.

15.1 – Automated Optical Inspection (AOI)

Automated Optical Inspection (AOI) is a tool in circuit board assembly to efficiently and accurately detect production errors before boards leave the facility. AOI uses cameras and image processing software to identify assembly errors such as missing or misplaced components, soldering short circuits and disconnected components.

At Hopetimepcb, AOI is one tool we use to provide the best product quality to our turn-key customers. We do not conduct AOI on bare PCBs as our other testing methods are effective, and AOI is generally more used for component related issues. Due to the added labour of setting up the AOI equipment, it is more economical for our clients if we conduct this type of inspection on larger (>=50 pcs) or complex orders (>=50 components). All of our work is visually inspected by our quality assurance technicians. There is no additional cost when AOI is used, as it is included in our standard assembly services.

15.2 – X-Ray Inspection

For Ball Gate Array (BGA) and Quad Flat No Lead (QFN) packed components, the solder pads are placed below the component. This saves space on a board, allowing for more density among components and a smaller overall board size. The disadvantage of these components is that their solder joints cannot be visually inspected, whether by a technician or automated inspection. Therefore, in order to ensure the proper installation of these components, x-ray equipment is utilized to see through the components and observe the solder joints. The use of this additional equipment and labour adds to the cost of the order when BGA or QFN components are used.

15.3 – Functional Testing (FCT)

Functional Testing (FCT) is the final testing method to be performed after a PCB is fully assembled. The purpose of FCT is to find component failures, assembly defects, or potential design issues. FCT results allow for troubleshooting to occur as soon as possible and confirm that components function as specified. The purpose of FCT lies mainly in avoiding assembly issues like shorts, opens, missing components, or the installation of incorrect parts. This testing provides additional assurance above our AOI and visual inspections.

Functional testers typically work on a computer that runs advanced testing software, which in turn operates the various testing instruments such as digital multimeters, input/output PCBs, and communication ports. The FCT procedure determines whether boards pass or fail based on whether or not the test results satisfy a set of predefined requirements. Often, the FCT procedure includes multiple cycles of testing, each with different loading conditions or operating modes, according to the specific function of the PCB under test.

At Hopetimepcb, we offer Functional Testing (FCT) in addition to our turn-key assembly services. If you desire FCT for your project, we simply ask that you specify detailed instructions for the FCT of your particular PCB. Of course, we at Hopetimepcb are happy to assist you in developing a proper FCT procedure if you are unsure. Our functional testing engineer will preview your requirements, such as test scope and test instruction, design the test jig if necessary, set up the instruments, and prepare the test report form and design testing workflow.

At Hopetimepcb, we maintain state-of-the-art functional test equipment, such as adjustable DC power supplies, a 200MHz digital oscilloscope, a signal generator, an LRC multi-meter, and a universal programmer. In the case that a board does not pass FCT, a troubleshooting program will be launched to find the root cause of the failure. A Design for Assembly (DFA) form will be issued by our process engineer, and an 8D quality assurance report will be created by our quality engineer. This DFA report will be sent to the customer as a design reference, and another as a sample in our quality improvement guideline.

If you have any further questions about your order, please consult our sales team during the quotation stage of your order. Design issues that arise during production may delay your order.

Thank you for working with Hopetimepcb to make your project a success!