



Turn-Key PCB Assembly

## Rigid PCB Design For Manufacturability Guide

Updated: October 04, 2017

©Bittele Electronics Inc.

# Table of Contents

1.0 – Introduction .....	4
1.1 Scope .....	4
2.0 Required Documents .....	5
2.1 Gerber Files – RS-274X Format .....	5
2.2 ODB++ .....	6
2.3 Drill / Route File .....	6
2.4 Netlist File .....	7
2.5 Centroid File / Pick and Place File .....	7
2.6 Assembly Bill of Materials (BOM) .....	7
2.7 Drawings .....	8
2.8 Standard Manufacturing Specifications .....	9
3.0 Laminate Materials Selection .....	10
3.1 Material Selection & Properties .....	10
3.2 Laminate Material and Thickness .....	11
3.3 Prepreg Designation and Thickness .....	11
3.4 Copper Clad(Weight) for Materials .....	12
3.5 RF Substrates .....	12
3.6 Multi-layer Lay-Up .....	12
3.7 Multi-layer Lay-up Recommendation .....	13
3.8 Buried Capacitance .....	15
3.9 Material Substitutes (North America vs China) .....	16
4.0 Standard PCB Manufacturing Capabilities .....	17
4.1 PCB Technology Matrix .....	17
4.2 Etch Factor .....	18
4.3 Drill Selection .....	19
4.4 Aspect Ratio .....	20
4.5 Annular Ring .....	21
4.6 Tear Drop Pads .....	21
4.7 Hole Clearance .....	22
4.8 Conductor Clearance .....	22
4.9 Via Holes Treatment .....	23
4.10 Finished Board Thickness .....	23

4.11 Overall Finished Profile Tolerance .....	24
4.12 Board Outline .....	24
4.13 Edge Bevel .....	24
5.0 HDI Technology Capabilities .....	26
5.1 Tight trace width / space .....	26
5.2 BGA .....	26
5.3 Micro-via.....	27
5.4 Multiple Laminations/ Sequential Lamination .....	29
6.0 Surface Finish: Options and Requirements.....	30
6.1 HASL/LF-HASL .....	30
6.2 ENTEK/OSP .....	31
6.3 ENIG .....	32
6.4 Full Body Hard Gold .....	33
6.5 Selective Gold .....	33
6.6 Double Gold (Full-Body + Selective Gold) .....	34
6.7 Edge Connector Plating .....	34
6.8 Wire Bonding (Soft Gold).....	34
6.9 ENEPIG .....	35
6.10 Immersion Tin.....	35
6.11 Immersion Silver .....	36
6.12 Comparison Chart.....	37
7.0 Solder Mask: Options and Requirements .....	38
7.1 – What is a Solder Mask? .....	38
7.2 – Types of Solder Mask.....	38
7.3 – Solder Mask Colours .....	39
7.4 – Substitutes (North America vs China).....	40
7.5 – Solder Mask Tenting .....	40
7.6 – Solder Mask Plugging.....	40
7.7 – Peel-Able Solder Mask .....	40
8.0 Silkscreen: Options and Requirements.....	41
8.1 – What is a Silkscreen? .....	41
8.2 –Silkscreen Requirements .....	41
8.3 – Silkscreen Types.....	42
8.4 – Silkscreen Colours.....	42

8.5 – Substitute (North America vs China) .....	43
8.6 – Multiple Colour Silk on One PCB.....	43
8.7 – Serialization .....	43
9.0 Electrical Testing .....	44
9.1 – Electrical Testing Requirements .....	44
9.2 – Flying Probe Test .....	45
9.3 – Fixture (Bed of Nails) Test.....	45
9.4 – Cost .....	45
10.0 Controlled Impedance PCB .....	47
10.1 Impedance Calculators .....	49
10.2 Impedance Models .....	49
10.3 Impedance Affect Stack-up .....	49
10.4 Bittele TDR Calculations .....	50
10.5 TDR Coupons .....	50
11.0 Panelization .....	51
11.1 Fiducials .....	51
11.2 Tooling Holes .....	51
11.3 V-Score .....	52
11.4 Tab Routing .....	52
12.0 Report Types and Report Writing .....	53
12.1 Products Final Audit Report.....	53
12.2 Certificate of Conformance (C of C).....	53
12.3 Test Report .....	54
12.4 Solderability Report.....	54
12.5 Cross Section Report .....	54
12.6 Impedance Report (TDR) .....	54
13.0 RoHS Compliance.....	55
13.1 PCB Raw Material .....	55
14.0 Assembly Considerations.....	56
14.1 – Automated Optical Inspection (AOI) .....	56
14.2 – X-Ray Inspection.....	56
14.3 – Functional Testing (FCT).....	57

# 1.0 – Introduction

The purpose of this Design for Manufacturability (DFM) guide is to assist Bittele's 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 Bittele 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 Bittele's production facilities. At Bittele 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 titled "PCB Fabrication Specifications".

## 1.1 Scope

Consulting these DFM guidelines during your design process will allow you to plan a PCB that conforms to the capabilities of Bittele's facilities. A strong understanding of our manufacturing ability enables our clients to achieve the special features that their designs require, while maximizing efficiency with regard 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

In order 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 Bittele. Those sections are as follows:

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

## 2.0 Required Documents

Various PCB design files and other forms of documentation are required for Bittele 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 ODB++ format, all data required for PCB fabrication and assembly are included within the .TGZ compressed archive. Should you choose to use this format, you will just 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 silk screen*. 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 the 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 individual layer of the PCB. Normally, the CAD system will be used to output 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 Bittele will 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 will include 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

<b>PCB Manufacturing Drawing</b>	PCB Board fabrication information
<b>PCB Assembly Drawing</b>	PCB components on board assembly information
<b>Bill of Materials</b>	Table of information on all components to be placed on board
<b>Netlist</b>	component electrical connectivity information
<b>Gerber Files</b>	Group of files below
<b>Route files</b>	information on PCB copper design e.g. traces,pads
<b>Drill files</b>	information on PCB design drill holes
<b>Solder Mask Files</b>	Information on areas not to cover with solder mask e.g. pads,holes
<b>Board Outline File</b>	board size and shape information
<b>Silkscreen</b>	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 Bittele. 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 together 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 an XML version called ODB++(X), which was donated to the IPC organization by Valor Computerized Systems.

We at Bittele 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 Bittele 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, and we will 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 Bittele: 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 Bittele 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 origin
- Rotation – Described part orientation in degrees (0, 45, 90, etc.)

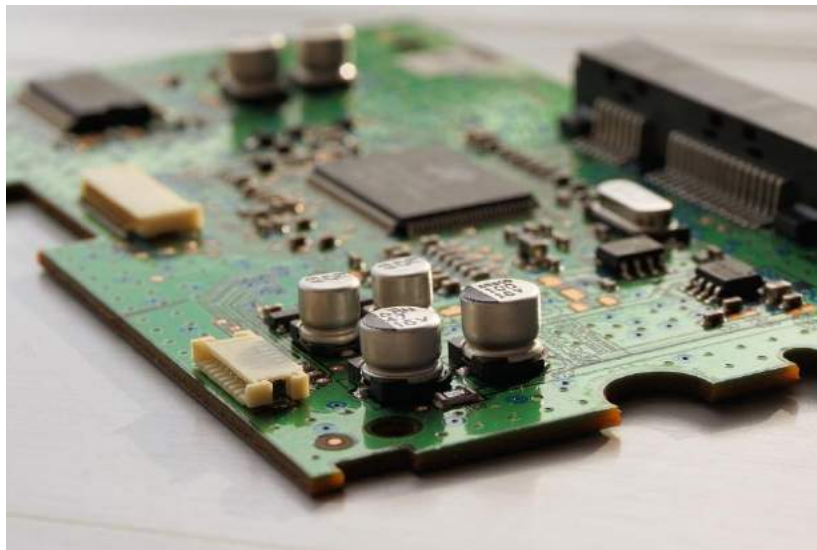
## 2.6 Assembly Bill of Materials (BOM)

Bill of Materials is a list of all the parts that are needed for 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 which 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 properly price your project.

- 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 a 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 board 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 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

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

**Dk** = Dielectric Constant (also referred to as  $\epsilon_r$  in electromagnetics) – Indicates the *relative permittivity* of an insulator material, which refers to its ability to store electrical energy in an electric field. For insulating purposes, a material with lower dielectric constant is better and in RF applications a higher dielectric constant may be desirable

**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 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
<b>Td (TGA @ 5% weight loss)</b>	N/A	300	N/A	300	N/A	300	350	340
<b>Dk (50% resin @ 2 GHz)</b>	4.2	3.93	4.38	3.93	4.28	3.93	4.3	4.04
<b>Df (50% resin @ 2 GHz)</b>	0.015	0.0167	0.015	0.0167	0.017	0.0167	0.015	0.21
<b>RoHS</b>	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 Bittele 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 Bittele representative for further details.

Table 4

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

## 3.4 Copper Clad(Weight) 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

Bittele 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.

## 3.5 RF Substrates

For radio frequency applications, Bittele Electronics 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 [here](#). Additional material types may be available upon special request with added lead time.

## 3.6 Multi-layer Lay-Up

Multi-layer boards have some physical properties which need to be considered by both the designer and manufacturer in order to ensure a quality construction. Multilayer board designs should have an **even** number of layers for 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 Bittele manufacturing team for what are suitable and achievable dimensions and tolerances.

It is recommended that multi-layer designs balance the lay-up relative to 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 specification 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  $\pm 10\%$  for the overall thickness which is thicker than 1mm, or  $\pm 0.1\text{mm}$  for 1mm thickness or thinner. Also, you need to always indicate where the thickness measurement is to be taken.

## 3.7 Multi-layer Lay-up Recommendation

We at Bittele Electronics 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 normal board thickness of 62mil. We at Bittele do offer **custom stack ups**, 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%				

## 3.8 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 2 (below), illustrates an 8 layer board with buried capacitance.

Layer			CU
1	F	0.0094	0.5
2	G		1
		0.0020	
3	P	0.0084	1
4	S		1
		0.0080	
5	S	0.0084	1
6	G		1
		0.0020	
7	P	0.0094	1
8	F		0.5
		Thickness	Thickness
		Nominal	Tolerance
		Material	Type
		0.062	0.006
		Standard	

**Figure 2: Buried Capacitance Stackup (Eight Layers)**



## 3.9 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 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
<b>Td (TGA @ 5% weight loss)</b>	N/A	300	N/A	300	N/A	300	350	340
<b>Dk (50% resin @ 2 GHz)</b>	4.2	3.93	4.38	3.93	4.28	3.93	4.3	4.04
<b>Df (50% resin @ 2 GHz)</b>	0.015	0.0167	0.015	0.0167	0.017	0.0167	0.015	0.21
<b>RoHS</b>	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 Bittele's PCB Fabrication and Assembly facilities are based in China the cost for importing North American laminates would be higher and increase 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 5**

Material Price Matrix				
	Cost in China	Lead Time	Cost in N.A.	Lead Time
<b>370HR</b>	130%	Longer	100%	Shorter
<b>FR408HR</b>	150%	Longer	120%	Shorter
<b>S1141</b>	70%	Shorter	120%	Longer
<b>S1000-H</b>	75%	Shorter	120%	Longer
<b>S1000-2</b>	70%	Shorter	120%	Longer
<b>IT180A</b>	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 manufactures 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 Bittelle are listed in Table 6 below. For a more comprehensive list of our capabilities please refer to our other document *PCB Fabrication Specifications*.

**Table 6**

Feature	Bittelle Capability
<b>Minimum trace width</b>	4 mil (but increases with higher copper weight)
<b>Minimum hole size</b>	0.1mm (laser) 0.15mm(mechanical)
<b>Minimum clearance</b>	3 mil (but increases with higher copper weight)
<b>Board thickness limits</b>	0.2mm - 6mm for 2 layer board
<b>Maximum allowed copper weights</b>	0.5oz (0.685mils) to 20oz (27.4mils)
<b>Maximum &amp; minimum board size</b>	0.2"x0.2" to 14"x20" (or larger for bare boards)
<b>Layer count- The number of layers in a multilayer PCB</b>	Up to 40 (below 20 recommended)
<b>Special features</b>	Gold fingers, multiple surface finishes, etc.

PCB key factors also affect each other thus they increase or decrease in relation to each other.

- The trace width and the clearance affects each other, when trace width increase, the trace clearance will decrease, and its impedance will also decrease.
- The minimum hole size and board thickness affects 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 the space will decrease.
- Copper weight affects impedance. When copper weight increases, the impedance will decrease.

#### 4.1.1 Pricing

Bittelle Electronics offers baseline pricing for PCB fabrication and assembly. These will start with default values, and can be customized to meet your needs. 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 PCB board is larger than 50x50mm, a smaller PCB can lower cost while a larger PCB board will cost more. Boards 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 cost by adding an extra step to fabrication.
- **Blind, Buried, Micro-via**- Special vias require extra processing steps and multiple lamination steps.
- **Board Thickness**- With a standard board thickness of 0.063 inch, overall cost will vary dependant on the board thickness required.
- **Surface finish**- Bittele electronics is proud to offer you 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 material delivery lead time which is the estimated time needed for board assembly materials to arrive at Bittele 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 with more layers and higher quantity lead time increases. Your project's lead time will be stated in your quote. For turn-key orders which include production, parts procurement and assembly standard lead time is **14 days** but 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)

## 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 Bittele 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 3 (below) illustrates the result. The ratio "D/C" is equal to the etching factor. Thus, the depth to which the etching occurs is proportional with 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 Bittele depend 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**.

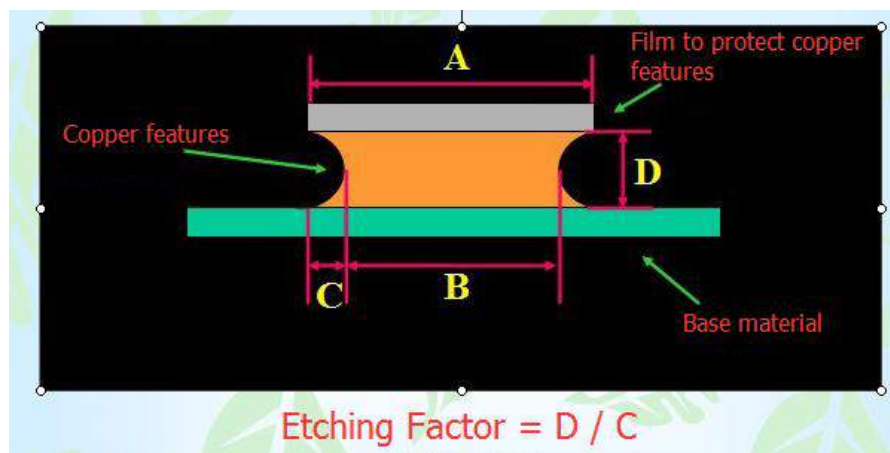


Figure 3: Etching Dimensions Profile View

## 4.3 Drill Selection

### 4.3.1 Hole Diameter

We Bittele can drill holes with sizes listed below.

Table 7

Available mechanical holes size	0.15mm - 6.0mm
Available laser holes size	4mil - 6mil (0.1mm-0.15mm)
<b>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</b>	

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 top and 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 on 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 Hole Size Tolerance

**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.

We at Bittele can control can control the holes size tolerance in Table 8 below.

**Table 8**

<b>Plated holes</b>	+/-3mil
<b>Pressfit holes</b>	+/-2mil
<b>Non-plated holes</b>	+/-2mil

### 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 of their width, we will call them as "long slots", otherwise, we will call them as "short slots".

We at Bittele can control can control the slots size tolerance according to Table 9 below.

**Table 9**

<b>Plated short slots</b>	+/-0.15mm
<b>Non-plated short slots</b>	+/-0.1mm
<b>Plated long slots</b>	+/-0.1mm
<b>Non-plated long slots</b>	+/-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 Bittele can build the board with aspect ratio listed in Table 10 below:

Table 10

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

## 4.5 Annular Ring

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 centered. 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 4 below for an illustration. We at Bittele can fabricate the board with annular ring listed in Table 11 below.

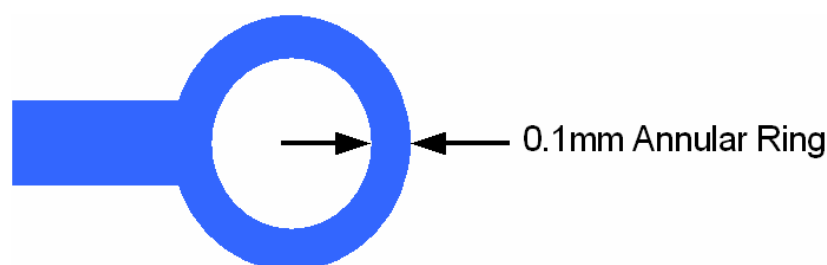


Figure 4: Illustration of Annular Ring

Table 11

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

## 4.6 Tear Drop Pads

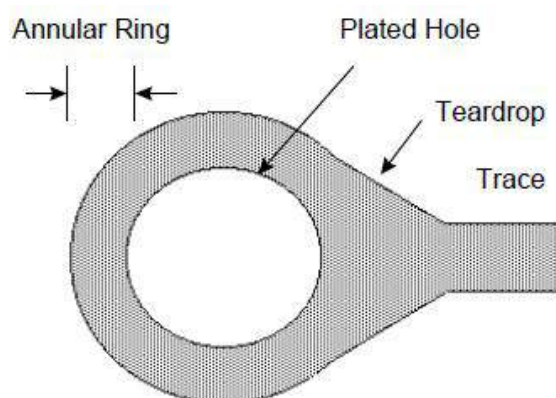


Figure 5: Teardrop Annular Ring

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 tear drop like shape made with the ring. This tear drop 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 Bittelle Electronics 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  $\geq$  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, and finally this can cause 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 12:

**Table 12**

Layers count	Minimum clearance between the hole to other copper feature
$\leq 6$	8.0mil preferred (6.5mil minimum)
8	10.0mil preferred (7.0mil minimum)
$\geq 10$	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 13 below.

**Table 13**

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

## 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 but it is less, and more reliable, than filling the hole with conductor.

**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 14.

**Table 14**

Via hole treatment	Limiting Condition	Extra cost
<b>Plugged with solder mask</b>	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
<b>Covered with solder mask</b>	N/A	No
<b>Via-in-pad</b>	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 15 below.



Table 15

Layers count	Maximum board thickness (inches)	Maximum board thickness (mm)
<b>&lt;=2 layers</b>	0.149 preferred (0.177 possible)	3.8 preferred (4.5mm possible)
<b>&gt;2 layer</b>	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 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 off setting 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 Bittele can fabricate a PCB with tolerances listed in Table 16.

Table 16

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

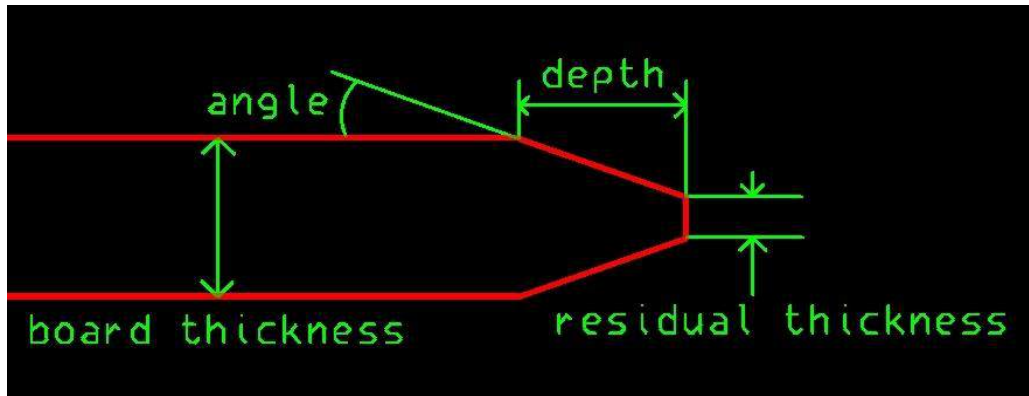


Figure 6: Edge Beveling of Gold Finger Profile View

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

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

## 5.0 HDI Technology Capabilities

Beside the standard PCB board, we at Bittelle can also build HDI boards (with laser holes, blind holes or buried holes and so on). And for such boards, we can build them with even smaller trace space. Our capability 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. Bittelle'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 / 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 17 below.

**Table 17**

Trace Width / Clearance Space			
<b>Internal Layer</b>	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)
<b>External Layer</b>	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 Bittele Electronics 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 Bittele we can process BGA packages with the specifications listed in Table 18.

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 18**

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

## 5.3 Micro-via

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-via 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.

A standard micro-via is a highly reliable type of interconnection structure, and should be used in a board design whenever possible. For a 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 Bittele Electronics can fabricate micro-via with a size ranging from 4mil (0.1mm) to a maximum size of 6mil (0.15mm). Fabrication of these vias will require separate drill files, with the via hole positions on all of the different layers of a multi-layer PCB. 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 7 below for an example.

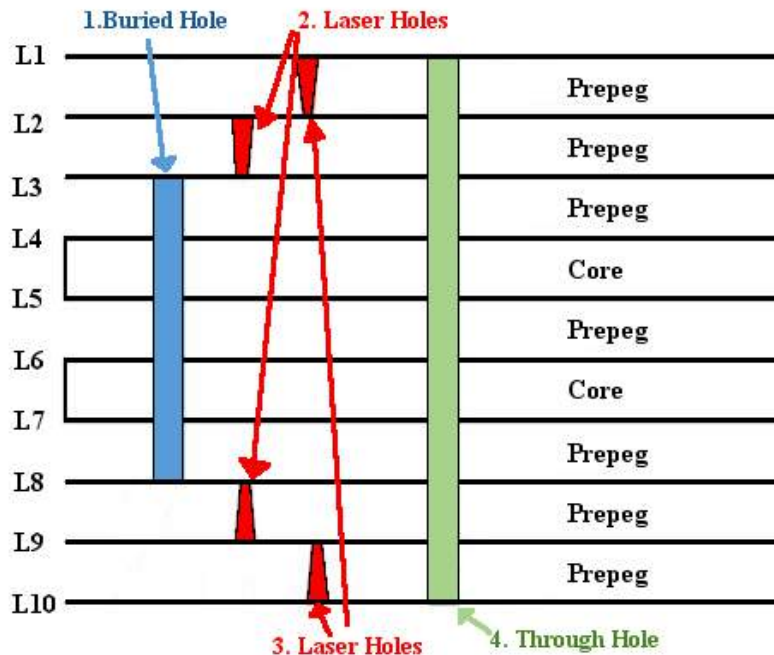


Figure 7: Example of 10 layer HDI board with maximum via steps

### 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.

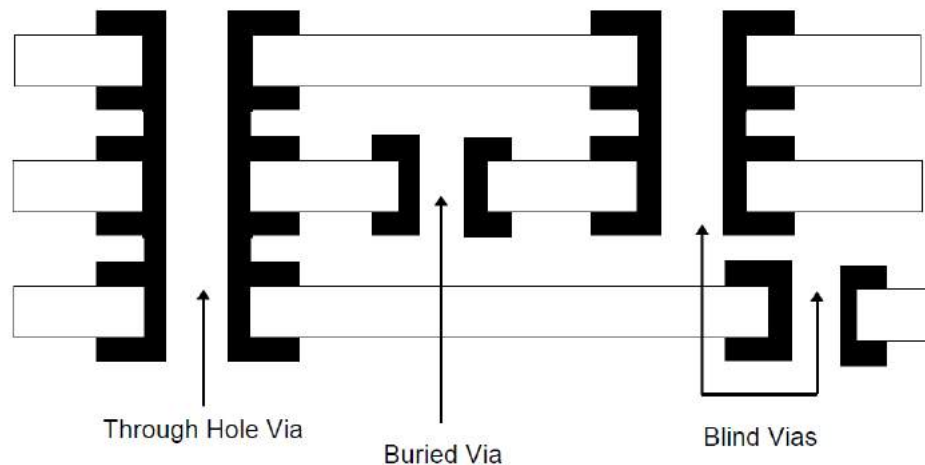


Figure 8: Illustration of via types

### 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 material allow for several lamination processes that may applied to PCB design.

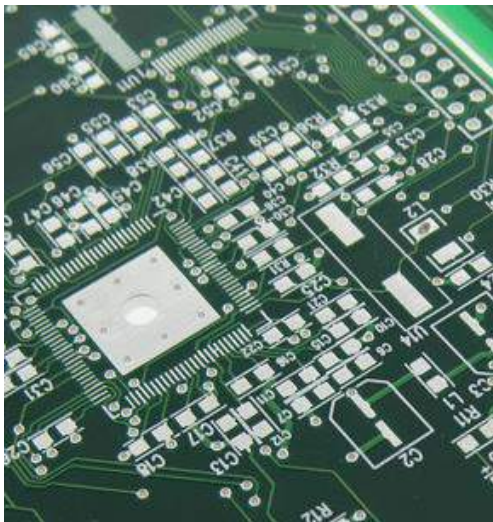
For non-HDI boards a single lamination step fuses all layers. For HDI boards our technique for creating a multilayer PCB is sequential lamination; a process starting on a core layer fused with a conductive and dielectric layer on both sides using multiple pressure passes. Sequential lamination allows 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 increase lead time. Bittele Electronics suggest consulting with one of our representatives if your design for an HDI PCB requires to be constructed using sequential lamination. Although Bittele Electronics 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 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 Bittele offer a variety of the most popular finished as options in our PCB fabrication services.

### 6.1 HASL/LF-HASL



**Figure 9: 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 using 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

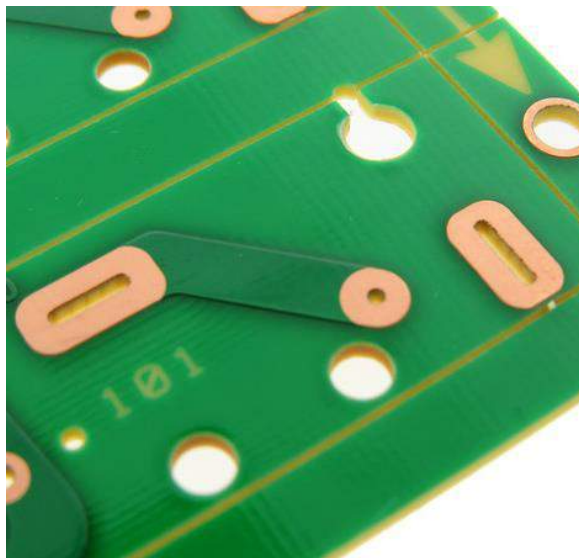
- ◆ Low-Cost Finish
- ◆ Widely Available
- ◆ Repairable Layer
- ◆ Excellent Shelf Life

### 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)

Bittele 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 10: 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

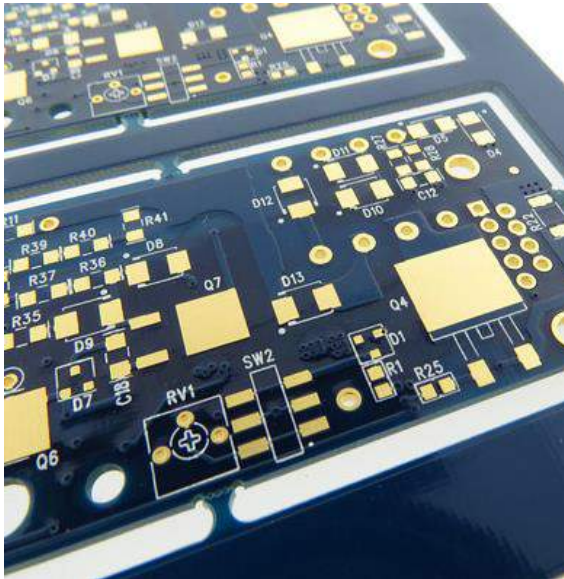
- ◆ Lead-free
- ◆ Flat surface
- ◆ Simple process
- ◆ Repairable
- ◆ Cost Effective

### 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 11: Example of an ENIG Surface Finish**

Electroless 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 increasing 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 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 its 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 Bittelle 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 its improved quality at the standard cost.

## 6.4 Full Body Hard Gold



**Figure 12: 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  $\mu\text{in}$
- ♦ Finish Does Not Fully Encapsulate Trace Sidewalls, Except in Finger Areas

We at Bittele 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 Bittele, and we will review your files before providing you with an estimate.

## 6.5 Selective Gold

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

## 6.7 Edge Connector Plating



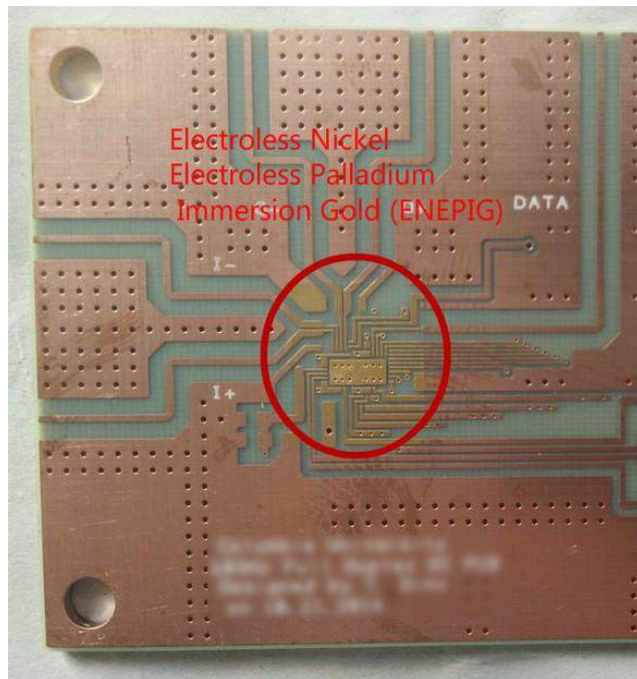
**Figure 13: Example of Edge Connector Plating**

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 Bittele 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 Bittele, 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 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 Bittele 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 Bittele, and we will review your files before providing you with an estimate.

## 6.9 ENEPIG



**Figure 14: 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

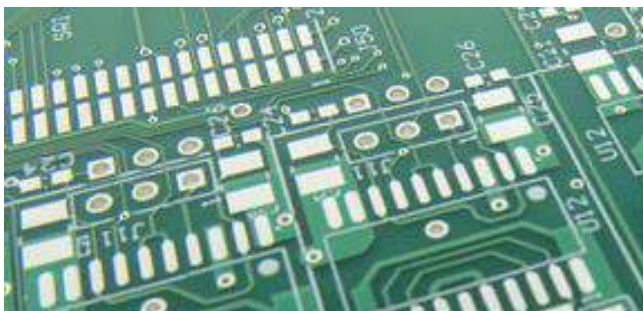
- ◆ Flat surfaces
- ◆ Strong
- ◆ Lead-free
- ◆ Good for PTH (Plated Through Holes)
- ◆ Long Shelf Life

### Cons

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

Please note that we at Bittele 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 15: 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 which 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

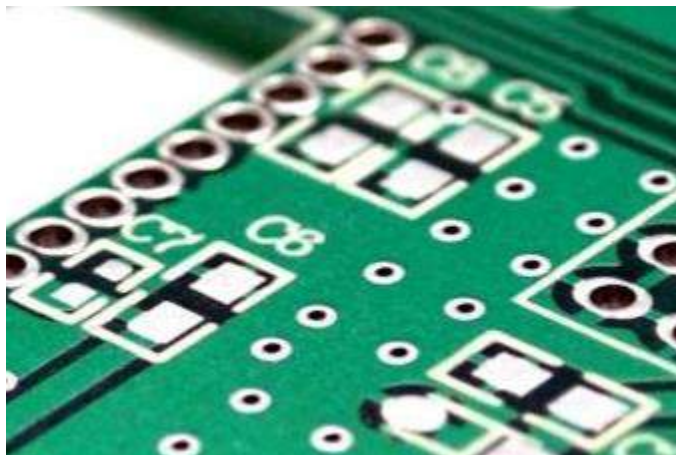
- ◆ Lead-free
- ◆ High Reliability
- ◆ Flat Surface/Planar
- ◆ Cost Effective
- ◆ Can Substitute for Reflowed Solder
- ◆ Top Choice for Press Fit Pin Insertion
- ◆ Re-workable

#### 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 Bittelle 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 16: 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 a 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
- ◆ 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 Bittelle 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

Bittelle electronics offers a variety of different surface finish choices, with some standard surface finishes like 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. with 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 19**

	HASL	LF-HASL	OSP	ENIG	Tin	Silver	Hard Gold	Soft Gold	ENEPIG
<b>Deposit</b>	Dipped	Dipped	Dipped	Electro-less/ Immersion	Immersion	Immersion	Electrolytic	Electrolytic	Electro-less/ Immersion
<b>Cost</b>	\$	\$	\$	\$\$	\$\$	\$\$	\$\$\$	\$\$\$	\$\$
<b>RoHS</b>	No	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
<b>Shelf Life</b>	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 Bittele 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

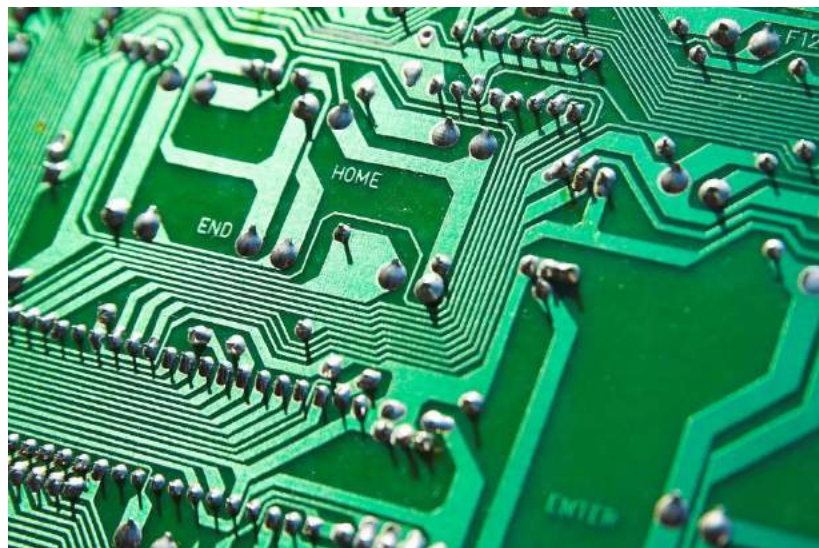
At Bittele, we offer Liquid Photo-Imageable solder mask, and peel-able solder mask (described in section 7.7). Liquid Photo-Imageable (LPI) solder masks are composed of an ink compound that can be silkscreened or sprayed onto the PCB. The LPI solder mask technique is commonly used with hot air surface leveling (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 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 Colours

The colour of a solder mask has no effect on the electrical performance of a board, but there is the 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 Bittele have a number of standard solder mask colours available, but custom colours will need to be ordered in advance.



**Figure 17: A Standard Green Solder Mask Protecting Copper Traces**

### 7.2.1 Standard colours

We at Bittele 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 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 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 contrast 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.

### 7.2.2 Custom colour

We at Bittele 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.

## 7.4 – Substitutes (North America vs China)

We at Bittele offer 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.

## 7.5 – 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.

## 7.6 – Solder Mask Plugging

We at Bittele 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.

## 7.7 – 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 Bittele Electronics makes available to our clients.

### 8.1 – What is a Silkscreen?

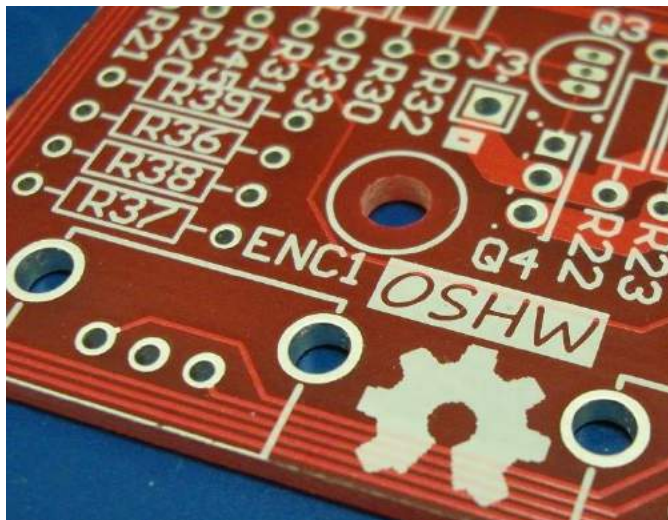


Figure 18: White Silkscreen Printing on Red PCB

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.

Bittele Electronics is able to print silkscreen layers with a  $\pm 4$  mil (0.004 inch) margin of error, and so we recommend that your silkscreen layers be designed with at least a 4-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 Bittele have the capability to print PCB silkscreens with a minimum line width of 5 mil (0.125mm).

Minimum text height refers to the smallest character that a silkscreen printer can produce, in terms of physical dimensions. We at Bittele 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 Bittele Electronics, 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 Bittele Electronics, 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 Bittele Electronics, 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 Bittele Electronics, 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 Bittele currently offer three standard silkscreen colours: **White**, **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 Bittele Electronics 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 Bittele Electronics, 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 Bittele Electronics, 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 can 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

Bittele Electronics 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 Bittele Electronics, 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 Bittele 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 Bittele, 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 20:

Table 20

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 requirement for each individual board test, which stems from the fact that the probe must move through the board in a specific sequence.

At Bittele Electronics, we consider the standard threshold for deciding the electrical test type to be 15 manufacturing panels. We would 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 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 Bittele Electronics, we consider the standard threshold for deciding the electrical test type to be 15 manufacturing panels. We may choose 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 Bittele Electronics, 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 Bittele, 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.

Bittele Electronics 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 Bittelle Electronics 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 interfere with the original signal, changing the signal's waveform. In a worst cases scenario, reflected energy would affect the signal's integrity, resulting in unpredictable PCB behavior. 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 21: 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.

Bittelle Electronics 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 22) 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 22: 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 labor costs are involved. Using cores with higher copper weight also adds to the material cost.

**Table 23: 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.

Bittele Electronics will provide free impedance calculations for PCB designs upon request. For performing impedance calculation, we at Bittele 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.

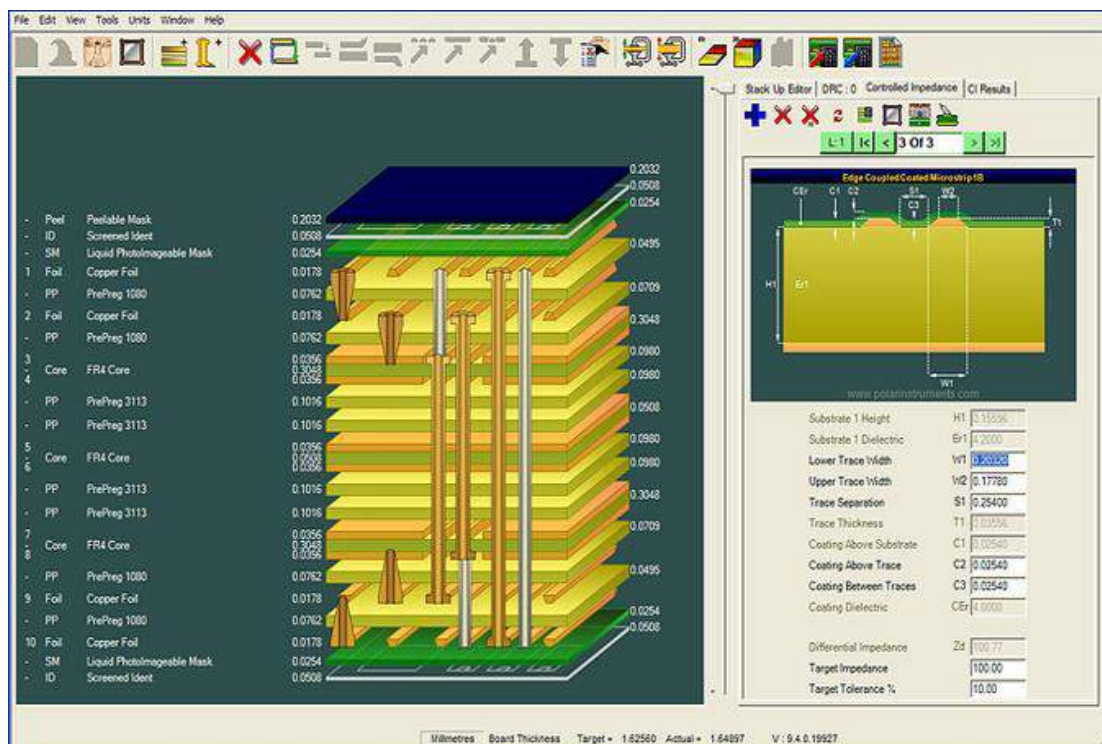


Figure 19: Impedance Calculator Software

## 10.2 Impedance Models

Impedance Models are diagrams with symbols labeled and showing the parameters for impedance calculation. We at Bittele 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 higher impedance value.
- Greater Inductance means higher impedance.
- Greater Capacitance means lower impedance.

For microstrip and stripline transmission lines, the largest factors that affect 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 Bittele 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 fiber. 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.

Bittele Electronics 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 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 TDR coupon should match.

Bittele Electronics 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 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, 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 Bittele Electronics 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. Bittele 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 Bittele 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 lay out the panels yourself as our technicians will do this to optimize our process.

### 11.1 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 the how much each part must be moved relative to the layout to ensure accurate part placement.

We at Bittele 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 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.

### 11.2 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 Bittele 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, Bittele suggests to drill tooling holes at the standard size of 2.0mm.

## 11.3 V-Score

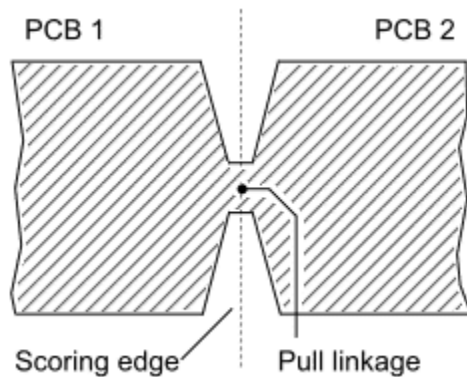


Figure 20: V Score Profile View

Bittele Electronics 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 cut away 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.

## 11.4 Tab Routing

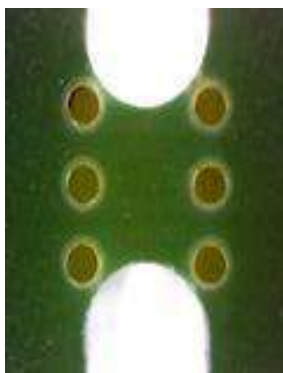


Figure 22a: Photo of Tab Routing Holes

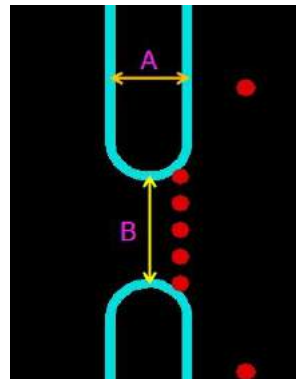


Figure 22b: Tab Routing Dimensions

Bittele 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 Bittele 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 22b) 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.

## 12.0 Report Types and Report Writing

The standard final report for a PCB includes a variety of sections. Bittele 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

### 12.1 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. Bittele includes all the materials for PCB fabrication and parts for assembly with the PCB order data in this report.

### 12.2 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. Bittele 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.

#### 12.2.1 RoHS Compliance

Bittele Electronics 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 13.0 RoHS Compliance for more information)

#### 12.2.2 U.L. Certificate

Bittele can include UL Certificates and related verification data in our "Certificate of Compliance" section of 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.

## 12.3 Test Report

Bittele 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, Bittele 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 afterwards.

## 12.4 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 PCB's solderability.

### 12.4.1 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.

## 12.5 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. Bittele will provide cross-sections and various other diagrams showing the PCB's various layers and micro-via.

## 12.6 Impedance Report (TDR)

Impedance Report is an optional report that Bittele 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)



## 13.0 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.

### 13.1 PCB Raw Material

For a PCB to be considered RoHS compliant, it cannot have restricted raw materials over a certain limit in its materials. Bittele Electronics 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)

#### 13.1.1 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 Bittele 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 its reduced melting point of 217 °C.



## 14.0 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 Bittele 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 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)*.

### 14.1 – 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 Bittele Electronics 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 labor of setting up the AOI equipment, it is more economical for our clients if we conduct this type of inspection on larger ( $\geq 50$  pcs) or complex orders ( $\geq 50$  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.

### 14.2 – 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 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.

## 14.3 – 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 Bittele Electronics, 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 Bittele 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 Bittele Electronics, 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 Bittele Electronics to make your project a success!**