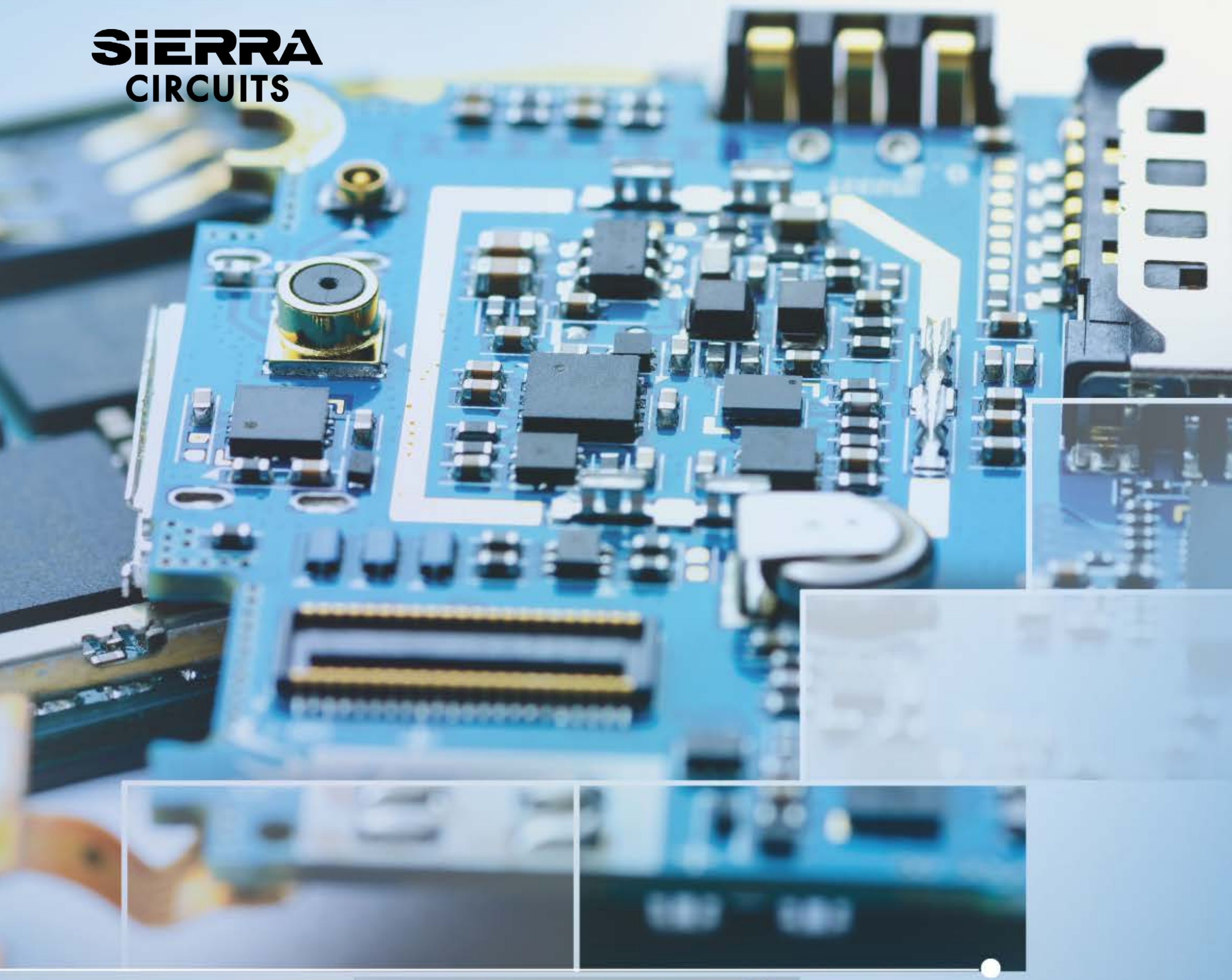
SIERRA CIRCUITS



PCB Material Design Guide

Table of Contents

Section 1. Printed circuit board materials.................................. 4

1.1 Prepreg... 4
1.2 Copper foil... 4
1.3 Copper clad laminates.. 5

Section 2. Basic properties of PCB materials' dielectric.............. 6

2.1 Thermal properties... 6

2.1.1 Glass transition temperature (T_g).. 6
2.1.2 Decomposition temperature (T_d).. 6
2.1.3 Coefficient of thermal expansion (CTE) 7
2.1.4 Thermal conductivity (k)... 8

2.2 Electrical properties... 8

2.2.1 Dielectric constant or relative permittivity (E_r or D_k)............ 8
2.2.2 Dielectric loss tangent or dissipation factor (Tan δ or D_f)..... 8

2.3 Mechanical properties.. 9

2.3.1 Tensile (Young's) modulus.. 9
2.3.2 Flexural strength or bend strength....................................... 9
2.3.3 Tensile strength.. 9

Section 3: Signal loss in PCB substrates.................................. 10

3.1 Dielectric loss... 10
3.2 Copper loss or skin effect loss.. 10
3.3. Correlation between signal loss and operating frequency.. 13

Section 4: Basic PCB material categories................................. 14

4.1 Normal speed and loss.. 14
4.2 Medium speed medium loss... 14
4.3 High speed low loss... 14
4.4 Very high speed very low loss (RF/microwave).................. 14
4.5 PCB material options.. 15

Section 5: Copper foil selection.. 17

5.1 Critical properties to be considered.................................. 17
5.2 Copper foil types... 17

Section 6: Choosing your PCB materials.................................... 19

6.1 PCB material selection best practices............................... 19
6.2 Sierra circuits preferred materials..................................... 20

Section 7: Choosing HDI materials - Key considerations............. 21

7.1 Quality of HDI materials.. 21

Section 8: The PCB Material Selector Tool.................................. 22

Section 9: PCB stack-up and its significance............................. 26

9.1 Significance of multi-layer PCB stack-up......................... 26
9.2 How to achieve best possible PCB stack-up design?........ 27

Section 1: Printed circuit board materials

Material selection is important for all PCB designs. The goal is always to select the right material for manufacturability, at the same time, meets your temperature and your electrical requirements. When dealing with high speed designs the material chosen significantly impacts the quality of the signals traversing through the traces. The type of materials you choose defines the quality of your end product.

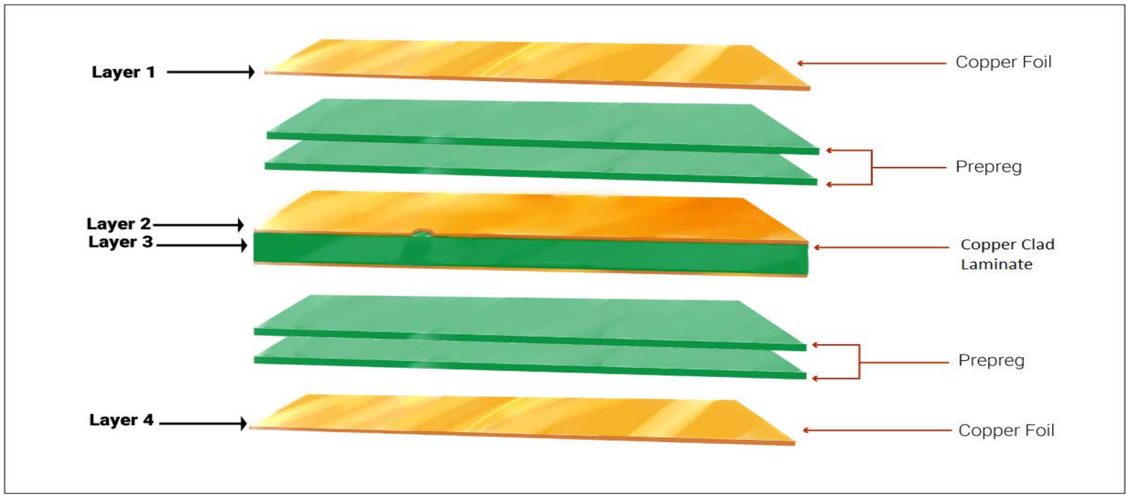


Figure 1: Materials used in a multi-layer PCB

A printed circuit board is manufactured using the following 3 items:

1.1 Prepreg

Prepreg is a sheet of woven-glass reinforcement impregnated with a resin that is not fully cured (B stage material). This is a tacky material and allows the bonding of different laminates or foils. Prepregs are available in a variety of glass weaves and epoxy compositions.

1.2 Copper foil

Copper foil on the outermost layer is the conductive medium through which electric current flows. Copper traces on a PCB are made by selectively etching copper foil. Traces (conductors or circuits) create the electrical connections on the PCB. In case of multilayer PCBs, the electrical connections between the layers are made through the holes that are drilled on the PCBs. Copper is deposited on the walls of these drilled hoes. These holes establish electrical connection between the inner and outer layers.

1.3 Copper clad laminates (Core)

Cores are basically made up of prepregs and copper foils that have been laminated and cured. A variety of materials with different thicknesses, epoxy properties, and types of glass weave are available. The designer must define the desired combination that is well suited for his/her application. The process of creating copper clad laminate begins with glass fiber interwoven to make glass cloth, this cloth is impregnated in epoxy resin and semi cured to make a ply of prepreg. Number of plies of prepreg are bonded together along with the copper foils on the outermost layers to make a laminate (copper cladded laminate). Some commonly used prepregs are 1080, 7628, 2116, 2113, and 2165.

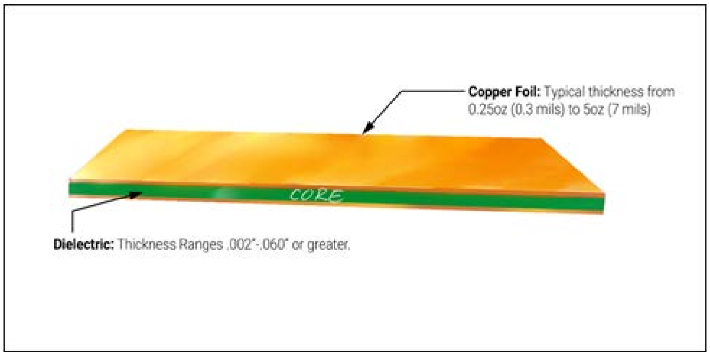


Figure 2: Copper clad laminate

Section 2: Basic properties of PCB materials' dielectric

Designers most often consider two basic properties- thermal and electrical when choosing materials for their PCB construction. In special applications, a designer might also consider other properties such as mechanical and chemical.

2.1 Thermal properties

A material's thermal properties govern its ability to withstand extreme temperatures while retaining its characteristics. Below are the thermal properties that need to be considered while selecting your PCB materials.

2.1.1 Glass transition temperature (T_g)

Glass transition temperature or T_g, is the temperature range in which a PCB substrate transitions from a glassy, rigid state to a softened, deformable state as polymer chains become more mobile. When the material cools back down, its properties return to their original states. T_g is expressed in units of degrees Celsius (°C).

Below the T_g the substrates are hard. The substrates attain rubbery state between the T_g and T_m (melting temperature) temperatures . At temperature greater than the melting point (melting temperature) the materials transform from solid to viscous liquid. During this phase the PCB material completely loses its shape.

2.1.2 Decomposition temperature (T_d)

Decomposition temperature or T_d, is the temperature at which a PCB material chemically decomposes (the material loses at least 5% of its mass). Like T_g, T_d is expressed in units of degrees Celsius (°C).

A material's T_d is an important parameter when assembling PCBs, because when a material reaches or surpasses its T_d, the resulting changes in its properties are not reversible. Whereas in case of T_g, the material properties return to their original states once the material cools below the T_g range.

So choose a material that can work in a temperature range that's higher than the T_g but well below the T_d. Most solder temperatures during PCB assembly are in the 200°C to 250°C range, so make sure T_d is higher than this (luckily, most materials have a T_d greater than 320°C).

2.1.3 Coefficient of thermal expansion (CTE)

The coefficient of thermal expansion or CTE, is the rate of expansion of a PCB material as it heats up. CTE is expressed in parts per million (ppm) expanded for every degree Celsius that it is heated.

As the material's temperature rises past Tg, the CTE will rise as well.

The CTE of a substrate is usually much higher than copper, this can cause interconnection issues when a PCB gets heated.

The CTE along the X and Y axes are generally low – around 10 to 20 ppm per degree Celsius. This is usually due to the woven glass that constrains the material in the X and Y directions, and the CTE doesn't change much even as the material's temperature increases above Tg.

So the material must expand in the Z direction. The CTE along the Z axis should be as low as possible.

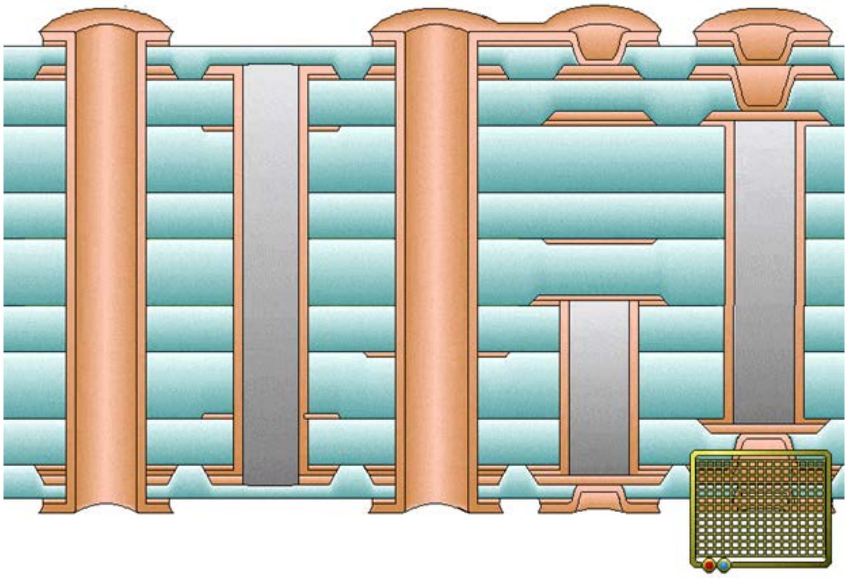


Figure 3: A material's expansion is measured by the coefficient of thermal expansion (CTE).

CTE is also useful to locate the Tg of a material by using a CTE curve. Plot a material's temperature versus displacement, then find the intercept of the two curves.

2.1.4 Thermal conductivity

Thermal conductivity, or k, is the property of a material to conduct heat; low thermal conductivity means low heat transfer while high conductivity means high heat transfer. The measure of the rate of heat transfer is expressed in watts per meter per degree Celsius (W/M °C).

Most PCB dielectric materials have a thermal conductivity in the range of 0.3 to 0.6 W/M °C, which is quite low compared to copper, whose k is 386 W/M °C. Therefore, more heat will be carried away quickly by copper plane layers in a PCB than by the dielectric material.

2.2 Electrical properties

2.2.1 Dielectric constant or relative permittivity (E_r or D_k)

It is the ratio of the electric permittivity of the material to the electric permittivity of free space (i.e., vacuum). It is also known as relative permittivity. The Er for most PCB materials is in the range between 2.5 and 4.5.

The dielectric constant varies with frequency and generally decreases as frequency increases; some materials have less of a change in relative permittivity than others. Materials suitable for high frequency applications are those whose dielectric constant remains relatively the same over a wide frequency range–from a few 100MHz to several GHz.

2.2.2 Dielectric loss tangent or dissipation factor (Tan δ or D_f)

The loss tangent is the phase angle between the resistive and reactive currents in the dielectric. The dielectric loss increases with increasing values of D_f. Low values of D_f result in a fast substrate while large values result in a slow substrate. D_f gives a measure of the power loss in a dielectric material. D_f is frequency dependent – higher the frequency, higher the D_f. Generally, lower the D_k of a PCB material, lower is the D_f, and flatter is the D_f vs Frequency curve. The values range from 0.001 to 0.030.

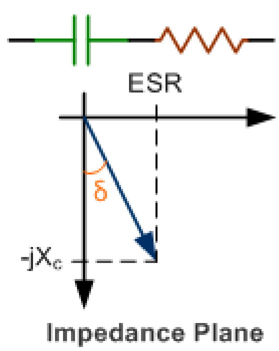


Figure 4: Dielectric loss tangent

2.3 Mechanical properties

A material's mechanical properties determine its ability to withstand external stresses and loads while retaining its characteristics. Below are some mechanical properties that can be considered if your board will be in a higher stress environment.

2.3.1 Tensile (Young's) modulus or elastic modulus

It is the ratio of the electric permittivity of the material to the electric permittivity of It is a measure of the stiffness of a solid material or the amount of force required to deform a material. It is defined as the ratio of the stress (force per unit area) along an axis to the strain (ratio of deformation over initial length) along that axis in the range of stress in which Hooke's law holds i.e., the stress-strain curve is linear. It is expressed as Pascals or pounds (force) per square-inch (psi).

2.3.2 Flexural strength or bend strength

It is a measure of a material's capability to withstand mechanical stress without fracturing when PCB is supported at the ends and is loaded in the center. IPC-4101 specifies the minimum flexural strength of various PCB materials. It is given in either Kg/ square-meter or Pounds/square-inch(psi).

2.3.3 Tensile strength

It is a measure of a material's maximum stress it can take before breaking, while being pulled from both sides. Like tensile modulus, it is expressed as Pascals or pounds/ square-inch (psi).

Section 3: Signal loss in PCB substrates

The PCB material can affect the signal integrity of your high-frequency circuits. You can minimize attenuation on your board by choosing the right PCB substrates and copper foil. These two materials play a very important role when it comes to signal loss in your PCB. Signal loss comprises dielectric loss and copper loss.

3.1 Dielectric loss

Dielectric materials are made up of polarized molecules. These molecules vibrate in the electric fields generated by the time varying signals on the signal traces. This heats up the dielectric and results in the dielectric loss part of signal losses. This signal loss increases with frequency. Signal loss can be minimized by using a material that has a lower dissipation factor. The higher the frequency, the more will be the loss in any given material. This is due to the changing electromagnetic field causing the molecules in the dielectric material to vibrate. The faster the molecules vibrate, the more loss there is.

3.2 Copper loss or skin effect loss:

Copper loss is essentially associated with the current that flows through the conductors. Electrons may not always flow through the centers of the conductors. If copper trace is finished with nickel, most of the current might flow through that nickel layer. The skin-effect loss gets larger as frequencies go up. This can be compensated by increasing the width of the traces which in turn creates larger surface area. Wider trace will always have lower skin effect loss. Copper foil-dielectric toothy interface profile increases the effective length and thus increases the copper loss.
Earlier, copper was made rough to make a strong bond between copper and substrate. This resulted in an increase in loss. The below images outlines the difference between smooth and rough copper.

Figure 5: Smooth copper (Source: Altium)

Figure 6: Rough copper (Source: Altium)

When an alternative current is passing through a conductor, the varying magnetic fields induce electric field (Eddy current). This induced electric field opposes the main current that is flowing through the center of the conductor but strengthens the current that is flowing outside on the outer surface of the conductor. As a result there is an increase in current density on the conductor surface. This phenomenon is called skin effect.

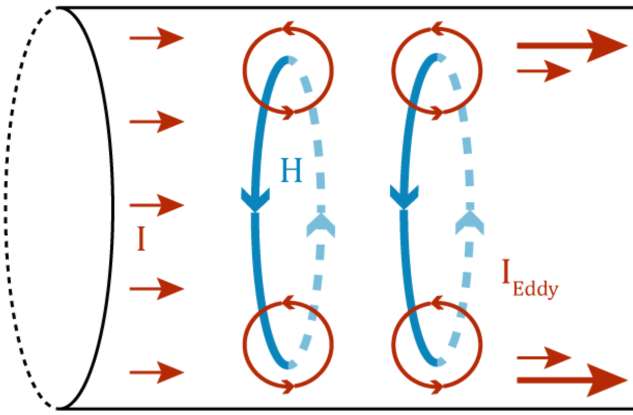


Figure 7: Formation of Eddy current due to alternating magnetic field

The effective section of the conductor where most of the current flows (skin depth) can be estimated with equation below:

$$\delta = \sqrt{\frac{1}{\pi \, . f . \, \mu . \sigma}}$$

Where,
δ is skin depth in meters
f is frequency in Hertz
μ is magnetic permeability
σ is electrical conductivity in Siemens per meter (S/m)
Skin depth in copper for different frequencies

Signal frequency	Skin depth in copper
1Hz	65.2 mm
1MHz	65 µm
1GHz	2.1µm
10GHz	0.65 µm
50GHz	0.29 µm

Skin depth effect reduces the effective cross section of the current carrying conductor. For conductors with a rough profile, a low skin depth causes the current flow through the contour of the material, this increases the length of the propagation path as illustrated in the below figure.

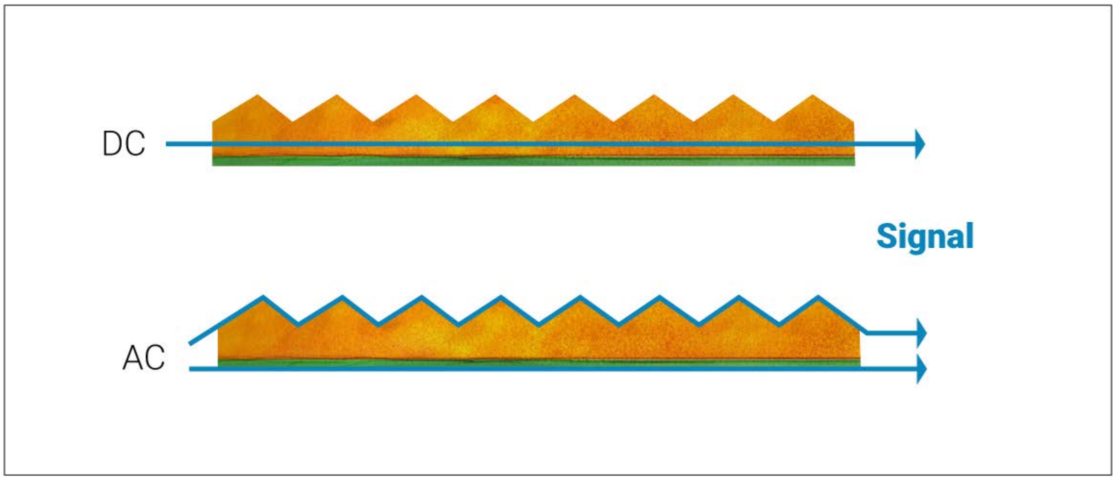
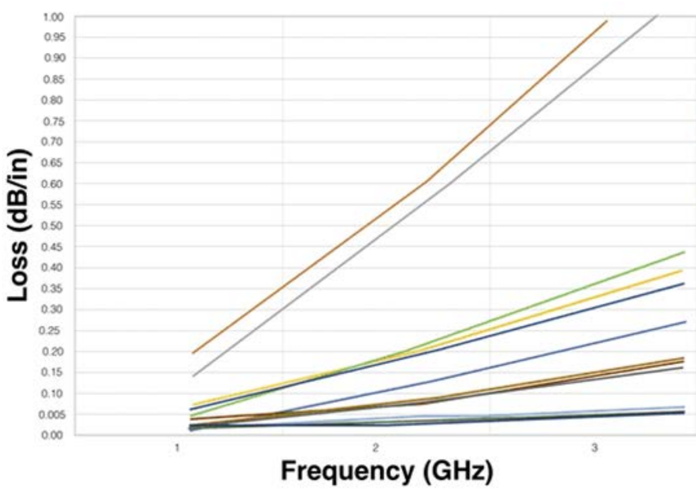


Figure 8: Comparison of current flow of DC and high frequency AC in rough copper foil

In order to mitigate this effect it is always recommended to use low profile copper with a good mechanical adhesion.

3.3 Correlation between signal loss and operating



Loss (dB/in)			
Freq (GHz)	2	5	10
370HR	0.19	0.6	1.1
NP175	0.13	0.55	1
N4800-20	0.067	0.2	0.37
N4800-20Si	0.0048	0.12	0.25
FR408HR	0.04	0.21	0.41
Ispeed	0.057	0.19	0.34
R4350	0.034	0.07	0.16
I-Tera	0.015	0.075	0.15
Meg-6	0.017	0.08	0.17
RF-35TC	0.018	0.019	0.047
R-3003	0.01	0.03	0.05
CLTE-XT	0.0095	0.04	0.047

Figure 9: Correlation between signal loss and operating frequency

As you can see from the graph above, there’s a direct correlation between signal loss and frequency. At the same time, we can also see that certain materials are less lossy than others.

Below chart classifies the essential materials into various buckets based on the signal loss properties.

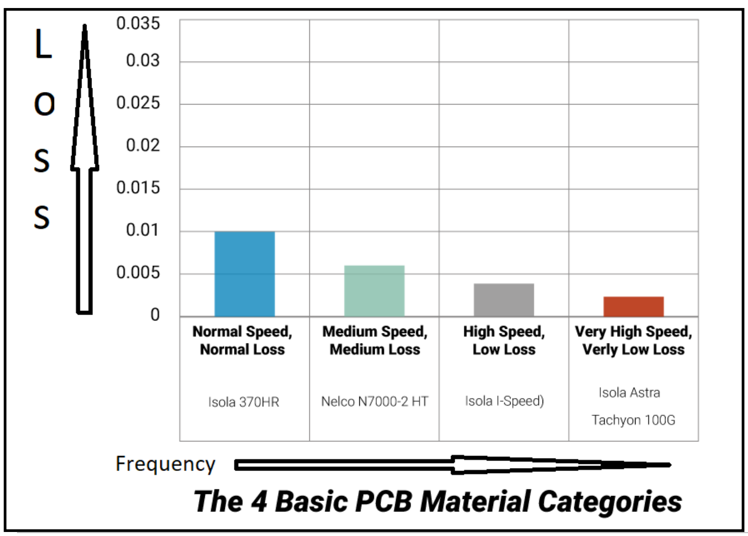


Figure 10: PCB material categories

On the left, we have materials like FR-4. These are your standard and simple to process, everyday materials that can be used in any application. But they are also the lossiest laminates. It can also have a plethora of other electrical and mechanical issues. Materials like Isola I-speed, Isola Astra and Tachyon exhibit low loss at high frequencies. We will be discussing more about the material categories in the next section.

Section 4: Basic PCB material categories

4.1 Normal speed and loss

Normal-speed materials are the most common PCB materials—the FR-4 family. Their dielectric constant (Dk) versus frequency response is not very flat and they have higher dielectric loss. Therefore, their suitability is limited to a few GHz digital/analog applications. An example of this material is Isola 370HR.

4.2 Medium speed, medium loss

Medium-speed materials have a flatter Dk versus frequency response curve, and have a dielectric loss about half that for normal speed materials. These are suitable for up to ~10GHz. An example of this material is Nelco N7000-2 HT.

4.3 High speed, low loss

These materials also have flatter Dk versus frequency response curves and low dielectric loss. They also generate less unwanted electrical noise compared to other materials. They are suitable for ~10 - 30 GHz applications. An example of this material is Isola I-Speed.

4.4 Very high speed, very low loss (RF/microwave)

Materials for RF/microwave applications have the flattest Dk versus frequency response and the least dielectric loss. They are suitable for up to ~20GHz applications and beyond. An example of this material is Isola Tachyon 100G.

4.5 PCB material options

Normal/medium speed (0 - 10GHz)		
Manufacturer	**Material name**	**Application areas**
Isola	FR370HR	Medium speed, normal loss
Nelco	N7000-2 HT	Medium speed, medi-um loss

High speed (10 - 30GHz)		
Manufacturer	**Material name**	**Application areas**
Isola	FR408HR	High speed, low loss
Isola	I-Speed	High speed, low loss
Panasonic	Megtron6 R-5775	High speed, low loss
Isola	FR408HR	High speed, low loss

Very high speed/ microwave (20 - 60GHz)		
Manufacturer	**Material name**	**Application areas**
Isola	I-Tera MT40	Very high speed/frequency, very low loss
Rogers	RO3003	Very high speed/frequency, very low loss
Rogers	RO4350 B	Very high speed/frequency, very low loss
Isola	Tachyon-100G	Very high speed/frequency, very low loss
Isola	Astra MT77	Very high speed/frequency, very low loss
Isola	I-Tera MT40	Very high speed/frequency, very low loss

Section 5: Copper foil selection

5.1 Critical properties to be considered

Below are a few properties that we need to consider while selecting copper foil.

Copper thickness: Typical thickness varies from 0.25 oz (0.3 mils) to 5 oz (7 mils)

Copper purity: It is the percentage of copper found in the copper foil. Electronic grade copper foil has the purity of around 99.7%.

Copper-dielectric interface Profile: Low profile has lower signal copper losses at high frequencies.

5.2 Copper foil types

Electro-deposited copper: This type of copper has vertical grain structure and rougher surface. Electo-desposited copper is typically used in rigid PCBs.

Rolled copper: A type of copper, made very thin by processing between heavy rollers, extensively used to produce flexible PCBs. Rolled copper has horizontal grain structure and a smoother surface which makes them ideal for rigid-flex and flex PCBs.

Low profile copper:

The roughness in electro-deposited copper can be controlled by adding organic additives (also known as levelers)

The physical parameters of an ED copper foil can be controlled with the addition of organic additives into the plating bath. Certain additives – known as levelers – can decrease the roughness of the surface. During the electrodeposition process, the current density and electric field strength tend to be higher on the peaks of the substrate which therefore tend to grow faster. This yields a rough surface. However, some organic molecules are adsorbed preferentially on the peaks of a surface and the deposition is therefore favoured on the valleys, yielding a product with a much smoother profile.

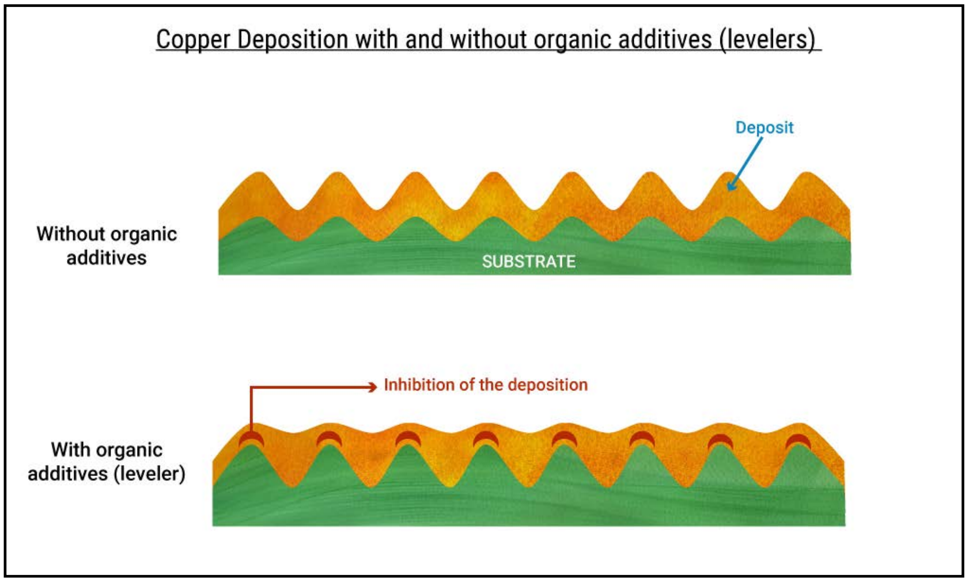


Figure 11: Copper deposition with and without organic additives (levelers)

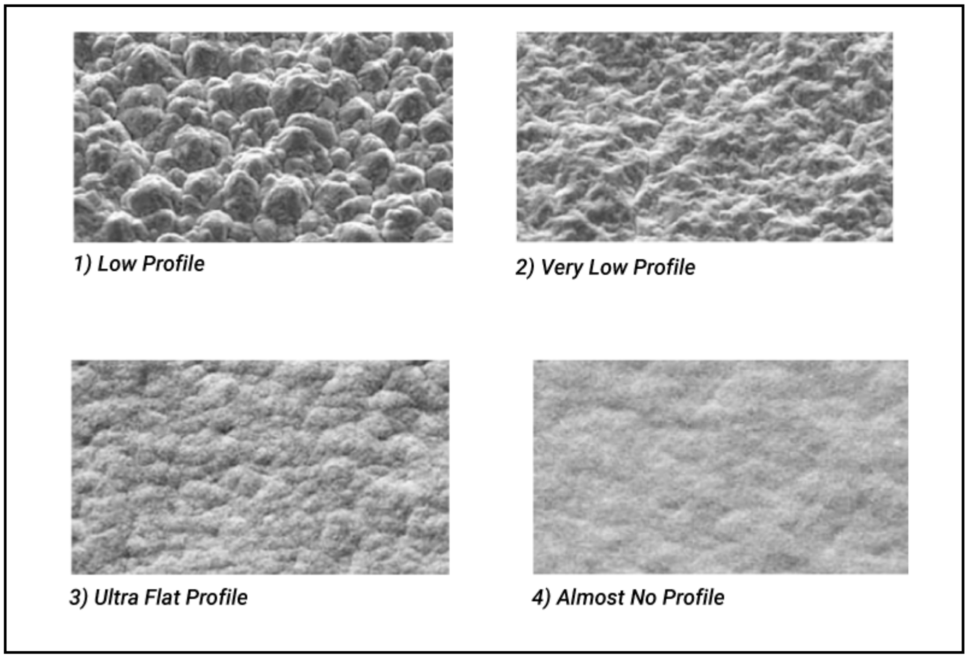


Figure 12: Copper profiles

Section 6: Choosing your PCB materials

6.1 PCB Material selection: Best practices

Match coefficient of thermal expansion (CTE): CTE is the most critical thermal characteristics for substrates. If the components of the substrates have different CTE, they may expand at different rates during the fabrication process.

Opt for tight substrate weave: The Dk distribution in tight substrate weave will be even.

Avoid FR4 for high-frequency applications: This is due to its high dielectric loss and steeper Dk versus frequency response curve. (For frequencies less than 1 GHz)

Use lower moisture absorption materials: Moisture absorption is the ability of a PCB material (copper in this case) to resist water absorption when immersed in water. It is given by percentage increase in weight of a PCB material due to water absorption under controlled conditions as per standard test methods. Most materials have moisture absorption values in the range of 0.01% to 0.20%.

Always use CAF-resistant materials: Conductive anodic filament (CAF) is a metallic filament that forms from an electrochemical migration process and is known to cause PCB failures. Using CAF-resistant materials is one of the most effective ways to prevent CAF formation and failure.

6.2 Sierra circuits preferred materials

Application type	Materials				
Standard FR-4 lead free boards	Isola 370 HR	Ventec VT47			
Ceramic reinforced boards	Rogers RO 4350 B	Rogers TTM	Rogers RO 4003	Rogers RO 4230	
High speed materials with processing similar to standard FR-4	Isola FR408 HR	Isola I-Speed	Isola I -Tera	Isola Astra MT77	Isola Tachyon 100G
Standard	Isola P95	Nelco			
Polyimide boards		N7000-2HT			
Advanced Teflon boards	Rogers RO 3000 series	Rogers RT/Duroid series	Rogers ULTRALAM 2000		
Standard flex boards	DuPont Pyralux AP	DuPont Pyralux LF	DuPont Pyralux FR		
Boards that require higher thermal conductivity	Thermagon 88	Laird IMP-CB			

Section 7: Choosing HDI materials: Key considerations

- Choose a PCB material whose Tg is more than or equal to the maximum withstanding temperature required for the PCB. For example, if operating temperature (Top) is 200 ºC, choose a PCB material whose Tg >= 200 ºC.
- The physical thickness of the PCB is important when considering the through hole aspect ratio.
- The physical thickness of a dielectric layer is important when considering the aspect ratio of the microvia to be plated. The current standard aspect ratio for a microvia is 0.75:1.
- For PCB material selection, another important consideration is the highest frequency content of the signals of the electrical circuit of the PCB.
- Along with these items, the dielectric constant of the material and the dissipation factor, will also play vital roles in HDI PCB material selection, as was discussed in earlier slides.

7.1 Quality of HDI materials

Dimensional stability: The material should be dimensionally stable; this also applies to the non-HDI PCBs as well. All materials shrink and stretch to some extent during manufacturing processes, and patterning must be scaled to compensate, which is not an issue provided the material movement is predictable.

Machinability: The material must be easily machinable. For HDI that means it can be laser drilled (Vaporization) without problems. Highly concentrated energy is directed in a focused beam on a specific area, which is absorbed by the material until it vaporizes.

Epoxy resin is the most commonly used thermosetting resin and is the backbone of the industry. Thanks to its relatively low cost, excellent adhesion (to the metal foils as well as to itself), and good thermal, mechanical, and electrical properties.

Make sure the chosen material is suitable for sequential laminations.

Sierra Circuits recommends Isola 370HR for normal-speed HDI PCBs and I-Speed and I-TeraMT40 materials for high-speed HDI PCBs.

Section 8: The PCB Material Selector Tool

In order to help you determine what sort of material will best suit your design needs, Sierra Circuits provides a list of materials with their most important properties.

TRY NOW

The operating frequencies:

Normal speed and normal loss - 0 to 3GHz

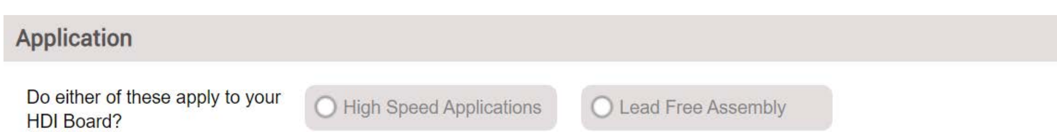
Medium speed, medium loss - 1 to 10GHz

High speed and low loss - 10 to 30GHz

Very high speed and very low loss (RF/microwave) - 20 to 60GHz

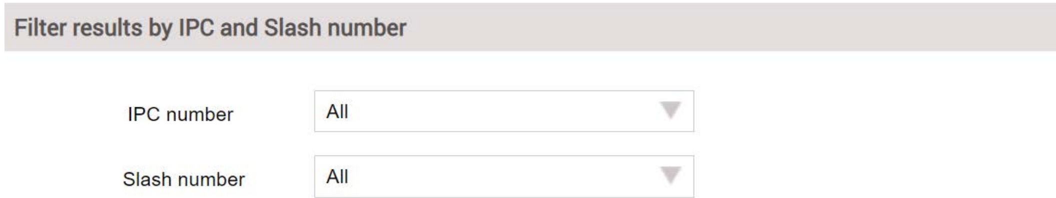
To view the material options you will have to key in the data into the various fields of our material selector tool.

1.Application: High-speed application/lead free assembly



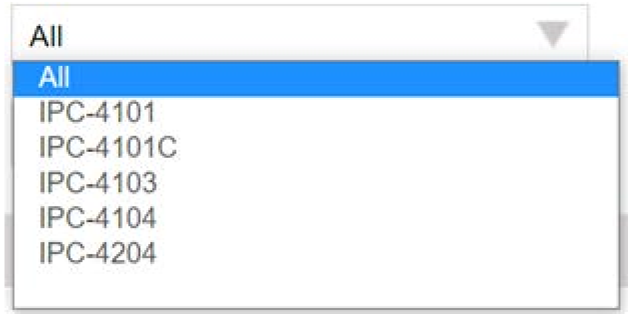
In this field, the users can select the type of application/assembly. The user is allowed to select both of these options.

2.Filter results by IPC and Slash number



The users can select the IPC standard applicable to their PCB in this section.

The drop-down **"IPC number"** lists the available IPC standards as shown in the below image.

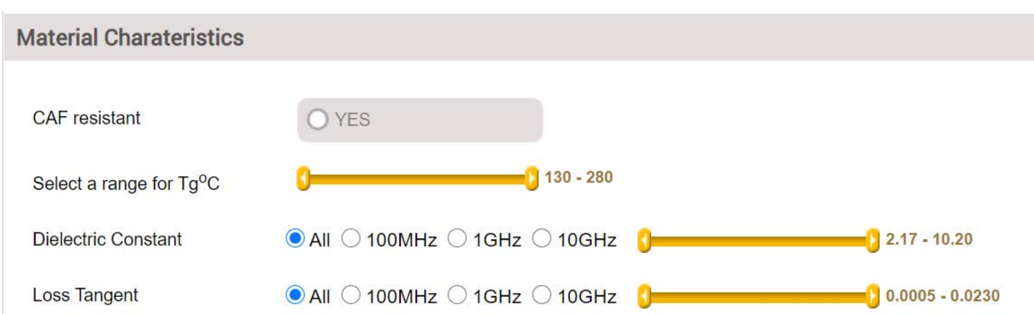


If the option **"All"** is selected from the drop-down, the user will get to see the complete range of materials that are available.

The significance of these IPC standards is explained at the end of this section.

Once the IPC standard is selected, the user can select the corresponding slash number (if any) of the selected IPC standard.

3.Material characteristics



In the material characteristics field, the user can define the following properties of the required material:

Whether CAF resistant: If yes, the user can click on the "YES" button.

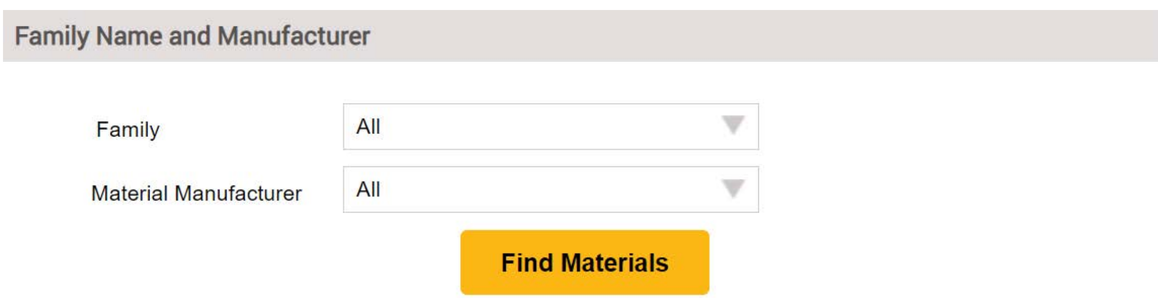
Conductive anodic filament, also called CAF, is a metallic filament that forms from an electrochemical migration process and is known to cause PCB failures.

Select range for Tg $^{\circ}$C: The range of the Tg (glass transition temperature) can be set by the available range 130 $^{\circ}$C to 280 $^{\circ}$C.

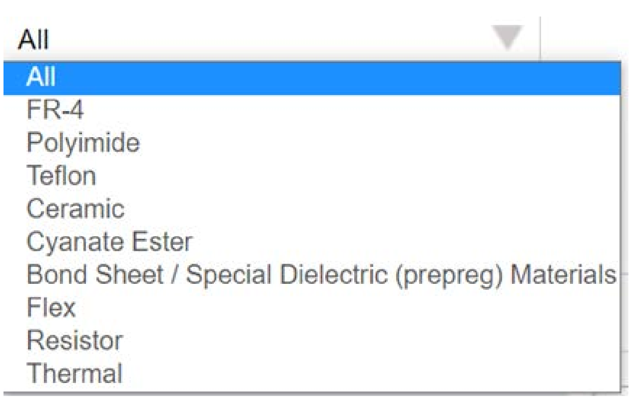
Dielectric constant and frequency: The frequency and the corresponding dielectric constant range can be defined in this field. The dielectric constant here ranges from 2.17 to 10.20.

Loss tangent: The frequency and the corresponding loss tangent range can be defined in this field. The loss tangent here ranges from 0.0005 to 0.0230.

4.Family name and manufacturer



The user can choose from the available family of the materials and manufacturers.



The next step is to hit the “Find Materials” button. The available materials for your requirement will be displayed on your screen as shown below:

20 Matching Materials Found

Compare	Manufacturer Name	Material Name	TG°C	Lead Free Assembly	DK (at 100MHz)	DK (at 1GHz)	DK (at 100GHz)	DF (at 100MHz)	DF (at 1GHz)	DF (at 100GHz)	More Details
☐	Isola	FR406	170	No		3.95	3.92		0.0161	0.0172	More
☑	Isola	FR 408	180	Yes	3.80	3.70	3.63	0.0100	0.0100	0.0130	More
☑	Isola	GETEK	175	No	3.80	3.60	3.50		0.0100		More
☐	Isola	IS620i	225	Yes		3.58	3.54		0.0059	0.0071	More
☐	Isola	370HR	180		4.30	4.17	4.03		0.0160	0.0230	More
☐	Nelco	N4000-13	210	No	3.80	3.70	3.60		0.0100	0.0140	More
☐	Nelco	N4000-13EP	200	Yes	3.80	3.70	3.60		0.0090	0.0080	More
☐	Nelco	N4000-13EP SI	200	Yes		3.40	3.20		0.0090	0.0080	More
☐	Nelco	N4000-13SI	210	No		3.50	3.30		0.0090	0.0090	More
☐	Nelco	N4000-29	175	Yes	4.50	4.30	4.20	0.0160			More

Compare

The users can shortlist the materials by selecting them and compare the properties of the shortlisted materials by hitting the compare button.

Significance of the IPC standards explained

IPC-4101: Specification for base materials for rigid and multi-layer boards

This standard covers the prerequisites for laminate or prepreg utilized for rigid or multi-layer PCBs. The specification sheets are arranged by type of reinforcement, resin system and/or construction. These specification sheets are provided with a specification number for ordering purposes.

IPC-4101C is an update of the standard IPCB-4101.

IPC-4103: Specification for base materials for high-speed/high-frequency applications

This standard covers the prerequisites for the substrates that are used in the fabrication of high speed/high frequency PCBs. This specification applies to material thickness defined in the specification sheets as measured over the dielectric only.

IPC/JPCA-4104: This standard defines the specification for HDI and microvia materials.

IPC-4204: This standard covers the specifications for flexible metal-clad substrates that are used in fabrication of flexible PCBs.

Section 9: PCB material stack-up and its significance

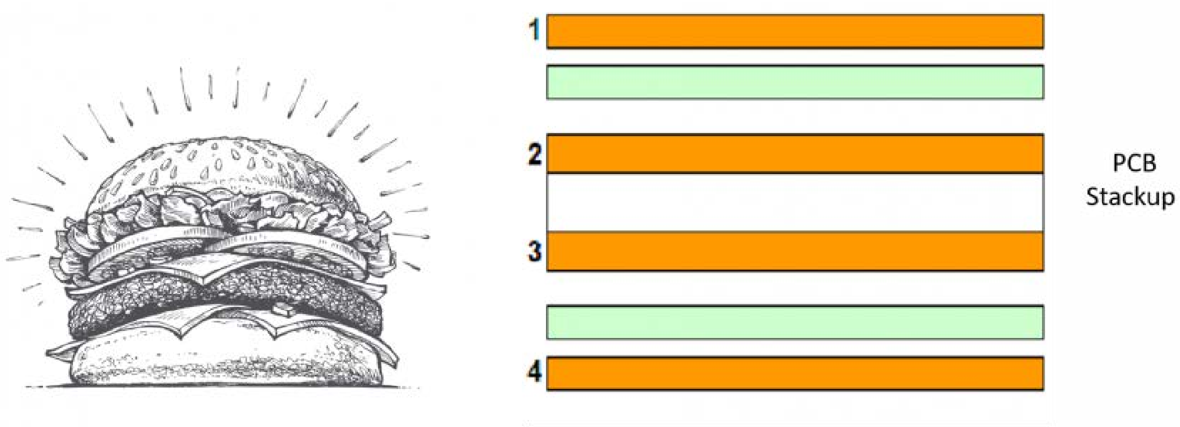


Figure 13: PCB stack-up

A stack-up is the construction of a multilayer PCB in a sequential order. A PCB stack-up layer consists of cores, prepregs, and copper foils. Generally, stack-ups are symmetrical. Majority of the products fall under 62 mil board thickness.

Standard stack-up design is dependent on the number of signal layers present. For standard PCB stack-up, key parameters include the number of layers, number of ground and power planes, frequency of the circuit, the sequence of the layers, and emission requirements. Some of the additional parameters include spacing between the layers and a shielded or unshielded enclosure.

9.1 Significance of multi-layer PCB stack-up

- An accurately stacked PCB will reduce electromagnetic emissions, crosstalk, and improve the signal integrity.
- It controls the impedance of traces.
- Reduces the size of the PCB.
- Reduces the routing density.
- Provides low noise ground and power planes.
- Reduces resistivity of the ground and power planes.

9.2 How to achieve best possible PCB stack-up design?

Whether it's an HDI or a standard stack-up board, following certain guidelines in accordance with the application and its requirements is the best possible way to implement the PCB stack-up design. Here are a few pointers for your PCB stack-up design:

The number of signal layers

PCB stack-up is dependent on the number of signal layers present in the design. Signal layers change in accordance with the application of the PCB. For instance, high-speed signal or high-power applications may require a greater number of layers compared to low-speed signal levels. The ground layers or planes are also considered before designing the PCB stack-up. PCB designers should avoid mixing different signals on inner layers.

The number of ground and power layers

Ground layer is the plane of copper in the PCB that is connected to the ground connection of the power supply. Power layer is a flat plane of copper in the PCB connected to the power supply and the ground. The purpose of using ground planes in your PCB is to provide voltage/signal return paths and to reduce noise and signal interference. Power planes are used to improve the decoupling ability of the circuits in the PCB. It also provides the shorter return paths which improves the EMC performance. Power planes have a larger current carrying capacity than the traces. This reduces the operating temperature of the PCB.

Controlled impedance traces

Controlled impedance is the characteristic impedance of a transmission line formed by a PCB trace and its associated reference planes. It is relevant when high frequency signals propagate on the PCB transmission lines. A uniform controlled impedance is important for achieving good signal integrity, which is the propagation of signals without significant distortion.

Sequential layer arrangement

Another key aspect of the stack-up design is the sequential layer arrangement. Arranging high-speed signal layers depending on the thickness of prepreg is one of the key parameters that a designer must consider before routing. Placing signal layers below the power plane will allow tight coupling.

For precise layer arrangement, keep a minimum distance between power and ground plane. Other key parameters include, avoid placement of two signal layers adjacent to each other, and build symmetric stack-up of the top and bottom layers.In sequential lamination, try to limit the number of lamination steps as it becomes more expensive and time consuming.

Determining layer material types

An important consideration for your PCB stack-up is the thickness of each signal layer. This should be established in conjunction with determining thicknesses for prepreg and core(s). There are standard thicknesses, as well as other properties, for different circuit board material types. Your process for selecting materials should include these electrical, mechanical and thermal properties.

For instance, layers that require higher voltage will require thicker copper layer.

Sequence of the stack-up:

It is recommended to have signal, ground/power, signal, ground/power and so on. As a good practice high-speed signals should be routed between two ground planes in the inner layers and on ground references for the outer layers.

Proper material selection is important since materials will affect the electrical performance of the signal traces. Following the guidelines provided in this design guide enables you to choose the best possible PCB materials for your design.

Sierra Circuits
1108 West Evelyn Avenue
Sunnyvale, CA 94086
+1 (408) 735-7137

www.protoexpress.com

SIERRA CIRCUITS