

# PCB Material Selection for High-Speed Application

<sup>1</sup>Jayasekar Micheal,  
<sup>1</sup>PCB Design Engineer,  
 Intel Corporation,  
 Santa Clara, CA, USA-95054

<sup>2</sup>M. Amalanathan  
<sup>2</sup>Department of Physics, Nanjil Catholic  
 College of Arts and Science, Kaliyakkavilai,  
 Tamilnadu, India

**Abstract:-** Printed Circuit Boards (PCBs) have become integral to modern electronics. With the increasing demand for high-speed digital/Analog/RF Circuits, the need for PCBs with high-speed materials has grown significantly. High-speed materials have become necessary for PCBs in commercial, data centers, telecommunication, and aerospace applications. This article will discuss the basics of the high-speed materials selection process, its properties, the Lamination process, and the materials used in PCBs. PCBs are an essential component of electronic devices, and performance and reliability depend on the materials used in their construction. The PCB material selection process involves comprehensively understanding various materials and their properties. This article will explore the steps involved in the PCB material selection process.

**Keywords:-** PCB Fabrication, PCB lamination Process, Glass fabric, Flex PCB, Glass Weave Skew, Single Ply-Dual Ply, Dielectric Constant ( $dk$ ), Coefficient of Thermal Expansion (CTE).

## I. INTRODUCTION

High-speed Printed Circuit Board materials are characterized by their ability to transmit signals at high frequencies without losing signal integrity. These materials have low-loss tangent, low dielectric constant, and high thermal stability. They are made of high electrical conductivity and low dielectric constant materials. Copper is the most common conductor used in PCBs due to its high electrical conductivity. However, copper is not the best choice for high-speed circuits because it has a high loss tangent and dielectric constant. Therefore, high-speed PCB materials use specialized copper alloys, such as PTFE, ceramic-filled hydrocarbon, and liquid crystal polymer.

These materials are designed to transmit signals at high frequencies without losing signal integrity. PTFE, ceramic-filled hydrocarbon, and liquid crystal polymer are some of the popular high-speed materials used in PCBs. These materials have low-loss tangent, low dielectric constant, and high thermal stability. When selecting a high-speed material for a PCB, it is essential to consider the application, the frequency range, and the thermal requirements.

PCB material is an important factor in designing printed circuit boards. It regulates the overall signal attenuation of a channel as well as the integrity of measured data. Using the low-loss dielectric in combination with a fiber glass weave will improve signal performance better. There are a variety of glass styles one can choose from. There are also many variations in resin content versus lass.

The process involves identifying the application requirements, considering the material properties, manufacturing process, cost considerations, and reliability considerations. It is essential to choose the material that best suits the application requirements and not solely based on cost. With careful consideration and selection, PCB material can contribute significantly to electronic products' overall performance and longevity.

It's important to note that the specific considerations for selecting a PCB material will depend on the specific requirements of your design. Therefore, it's recommended to consult with a PCB designer or use simulation software to ensure the best material for your application.

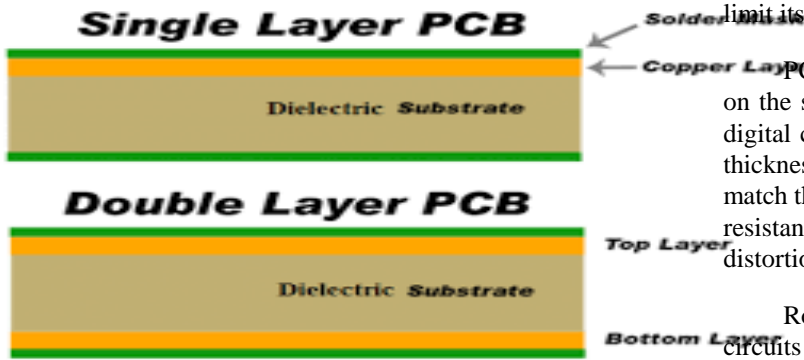
## II. PCB MATERIAL SELECTION PROCESS

The selection of PCB material is crucial in determining the functionality and reliability of electronic products. The PCB material selection process involves comprehensively understanding various materials and their properties. In this article, we will explore the steps involved in the PCB material selection process.

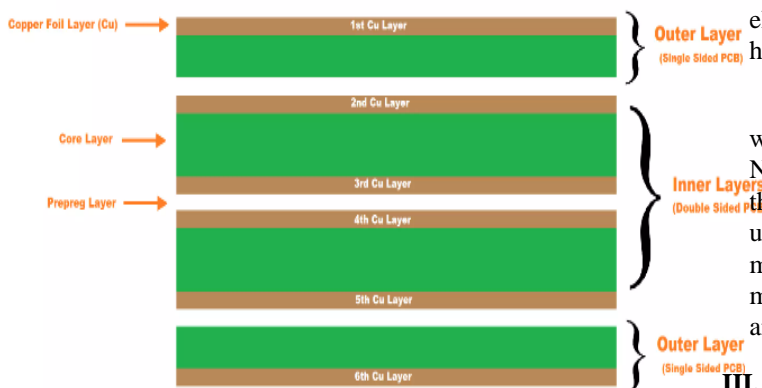
- **Step 1: Identify the Application Requirements** The first step in selecting PCB material is identifying the requirements. Factors such as frequency, power, operating temperature, and environmental conditions must be considered. For example, high-frequency applications require materials with low dielectric loss, whereas high-power applications require materials with good thermal conductivity.
- **Step 2: Consider Material Properties** The material properties are essential in selecting the PCB material. The most used PCB material is FR-4 (flame-retardant 4), which is a composite material made of woven fiberglass cloth and epoxy resin. However, other materials such as polyimide, ceramic, and PTFE (polytetrafluoroethylene) may be used in certain applications.
- Each material has unique properties, and the right material should be chosen based on the application requirements. For example, polyimide is suitable for applications that require high-temperature resistance, whereas ceramic materials are ideal for high-frequency and high-power applications.
- **Step 3: Consider the Manufacturing Process:** The PCB material selection process also considers the manufacturing process. The chosen material should be compatible with the manufacturing process, such as drilling, plating, and soldering. For example, materials with high glass transition temperatures ( $T_g$ ) are difficult to drill, and those with low  $T_g$  may not withstand the soldering process.

- **Step 4: Cost Considerations** The cost is also a critical factor in selecting PCB material. Materials such as PTFE and ceramic are expensive, whereas FR-4 is more affordable. However, the cost should not be the primary factor in selecting the PCB material. The right material should be chosen based on the application requirements, and cost considerations should come second.
- **Step 5: Reliability Considerations** The reliability of the PCB is critical in ensuring the long-term functionality of the electronic product. The chosen material should have good thermal stability, moisture resistance, and mechanical strength. For example, materials such as polyimide and ceramic are known for their high reliability, whereas FR-4 may have issues with delamination and moisture absorption.

The process of selecting a PCB material involves several factors (Picture 1 - Single Layer, Double Layer, and Picture 2 - Multi-layer), including the design's electrical, mechanical, thermal, and cost considerations. Here are some steps for selecting a PCB material.



Picture 1: PCB Structure



Side View of 6 Layer PCB Board

Picture 2: Multi-Layer PCB Structure

- **Electrical Parameters:** The first step is to determine the required electrical parameters of the design, including the dielectric constant (Dk), dissipation factor (Df), and loss tangent. These parameters will determine the speed at

which signals travel through the material, as well as the amount of energy lost as heat.

- **Determine the required mechanical properties:** The mechanical properties of the PCB material, such as its strength, stiffness, and thermal expansion coefficient, will impact its reliability and durability.
- **Determine the required thermal properties:** The thermal properties of the PCB material, such as its thermal conductivity and coefficient of thermal expansion, will impact its ability to dissipate heat and prevent damage to components.
- **Identify candidate materials:** Based on the required electrical, mechanical, and thermal properties, identify candidate materials that meet the design requirements. Common PCB materials include FR-4, Rogers RO4000, Isola FR408, and Arlon AD600.

There are several PCB materials options available for high-speed digital designs, including FR-4, Rogers, and Teflon. FR-4 is a widely used PCB material due to its low cost and good mechanical properties. However, it has a relatively high dielectric constant and dissipation factor, which may limit its use in Printed Circuit Boards.

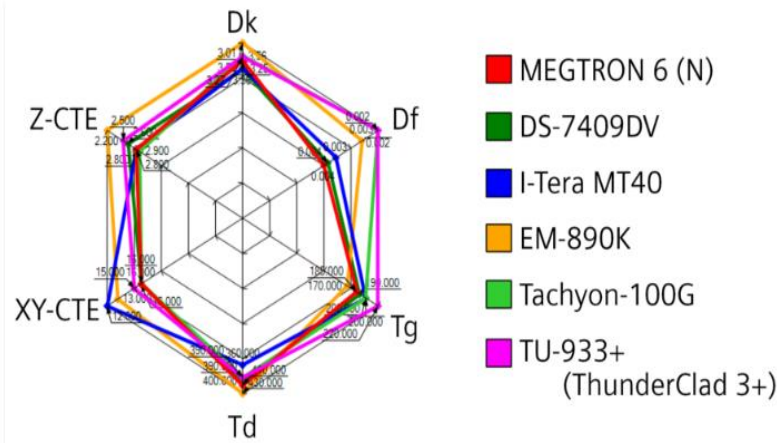
PCB material selection for digital circuits also depends on the signal frequency and the trace thickness. High-speed digital circuits operate at frequencies above 1 GHz, and the thickness of the copper traces on the PCB must be adjusted to match the signal frequency. Thicker copper traces have lower resistance and inductance, which reduces signal loss and distortion.

Rogers and Teflon are popular choices for high-speed circuits due to their low dielectric constant, low dissipation factor, and excellent thermal stability. Rogers materials are composed of woven glass and ceramic, while Teflon is a thermoplastic material. Both materials have excellent electrical properties that make them suitable for use in high-speed digital designs.

In the past, a limited number of high-speed laminates were available on the market, namely Nelco's N4000-13 or N4000-13 SI. Today, there are hundreds of choices – so many that we risk being overwhelmed with options. A general understanding of the nature of the materials can help to mitigate this risk, and a powerful PCB library comparing the material properties can help to alleviate the possibility of analysis paralysis.

### III. PCB MATERIAL SELECTION FOR HIGH-SPEED DIGITAL DESIGNS

High-speed digital circuits require a material with specific electrical properties that enable the transmission of high-frequency signals with minimal loss and distortion. The selection of the right PCB material for high-speed digital designs can improve the performance and reliability of electronic devices.



Picture 3: Material Performance Chart

One of the key considerations in the selection of PCB material for high-speed digital designs is the dielectric constant (dk) of the material. The dissipation factor (Df), and The coefficient of thermal expansion (CTE). A low dielectric constant material is preferred for high-speed digital designs as it reduces the capacitance between traces, minimizes signal loss, and preserves signal integrity.

Selecting PCB material for high-speed digital designs is based on the required electrical parameters of the design, including the dielectric constant, dissipation factor, and loss tangent. Here are some key considerations and formulas to help select the appropriate PCB material:

- **Dielectric constant (Dk):** The dielectric constant of the PCB material determines the speed at which electromagnetic signals travel through the material. The higher the dielectric constant, the slower the signal propagation. The formula for calculating the effective dielectric constant (Dkeff) of a PCB material is:

$$D_{keff} = (D_{k1} + D_{k2})/2 + (D_{k1} - D_{k2}) \times (T1/T2) \times (\ln(T1/T2)/\ln(e))$$

where Dk1 and Dk2 are the dielectric constants of the two adjacent layers, T1 and T2 are their respective thicknesses, and e is the base of the natural logarithm.

- **Dissipation factor (Df):** The dissipation factor is a measure of the amount of energy lost as heat in the PCB material. The formula for calculating the loss tangent (tan δ) of a PCB material is:

$$\tan \delta = Df/2\pi f Dk \text{ (where f is the frequency of the signal).}$$

Loss tangent (tan δ): The loss tangent is a measure of how much the signal degrades as it travels through the PCB material. The formula for calculating the characteristic impedance (Z0) of a PCB trace is:

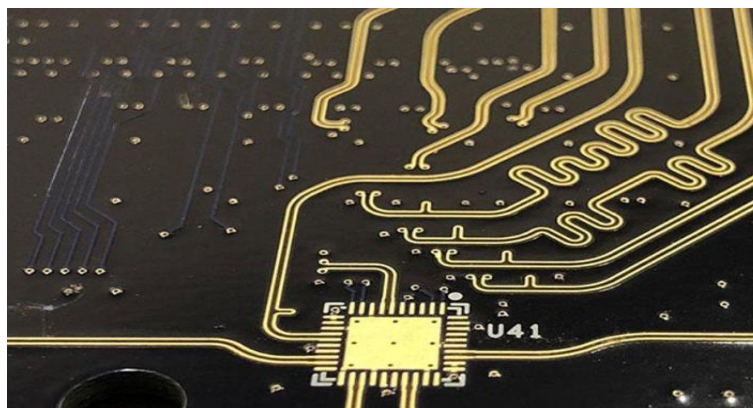
$$Z_0 = 87.45/(D_{keff}^{0.5}) \times \ln(5.98H/W + 1.7)$$

Where H is the height of the trace above the ground plane, and W is the width of the trace.

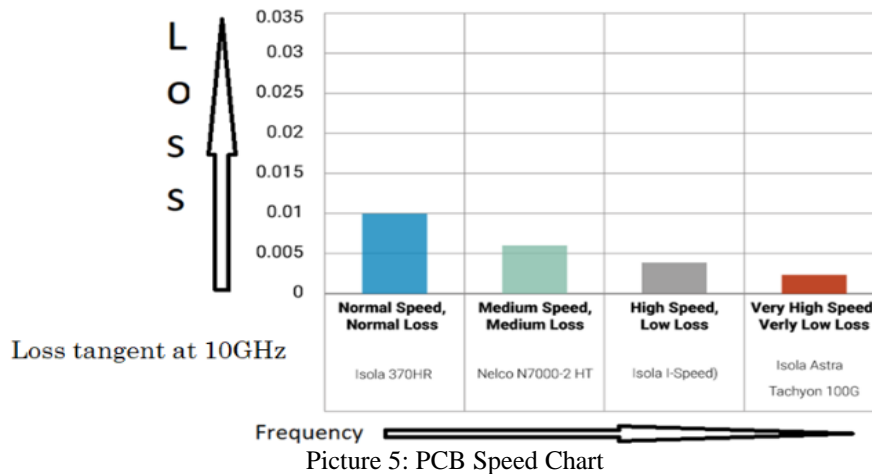
The coefficient of thermal expansion, or CTE, is the rate of expansion of a PCB material as it heats up. CTE is expressed in parts per million (ppm) expanded for every degree Celsius that it is heated. As a material's temperature rises past Tg, the CTE will rise as well.

#### IV. PCB HIGH-SPEED AND LOW-LOSS MULTI-LAYER MATERIALS

As technology advances, the demand for high-speed, low-loss PCBs has increased. Multi-layer PCBs, which consist of several layers of conductive materials and insulators, are commonly used in high-speed applications.



Picture 4: PCB High-Speed Routing



Picture 5: PCB Speed Chart

When selecting a multilayer material for a high-speed, low-loss PCB design, the key factors to consider include the dielectric constant (Dk), dissipation factor (Df), and thermal properties of the material. Properties of Low-Loss Multi-Layer PCB Materials The properties of high-speed, low-loss multi-layer PCB materials include:

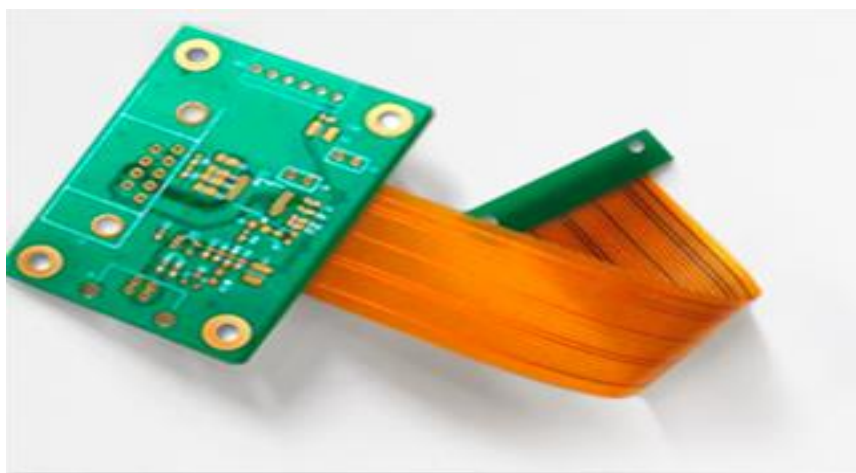
- Low dielectric constant is essential in high-speed PCBs, as it minimizes signal loss and attenuation. The low dissipation factor is important in high-speed PCBs, as it minimizes signal loss due to energy dissipation.
- High Tg is important in high-speed PCBs, as it ensures that the PCB material can withstand high temperatures during the manufacturing process and in operation. Low moisture absorption is essential in high-speed PCBs, as it minimizes the impact of moisture on the dielectric properties of the material.
- Calculate the insertion loss: It loss is a measure of the signal attenuation that occurs as a signal passes through the PCB material.  $Insertion\ loss = 20 \times \log_{10}(V1/V2)$
- Dielectric loss is a measure of the energy lost as heat due to the material's dielectric properties.  $Dielectric\ loss = Df \times \tan \delta$  (where Df is the dissipation factor and  $\tan \delta$  is the loss tangent.)
- Applications of High-speed, low-loss multi-layer PCB materials are commonly used in applications such as:

- High-Speed Digital: High-speed digital applications, such as servers and routers, require PCBs with low signal loss and attenuation to maintain signal integrity.
- Radio Frequency (RF) and Microwave: RF and microwave applications require PCBs with low dielectric constant and dissipation factor to minimize signal loss and attenuation.
- Aerospace and Defense: Aerospace and defense applications require PCBs that can withstand high temperatures, moisture, and other environmental stresses while maintaining signal integrity.
- Medical: Medical devices, such as imaging equipment, require PCBs with low signal loss and attenuation to ensure accurate signal transmission.

One of the most commonly used High Speed, low-loss materials is DS-740D with Glass 2116 8-Ply.

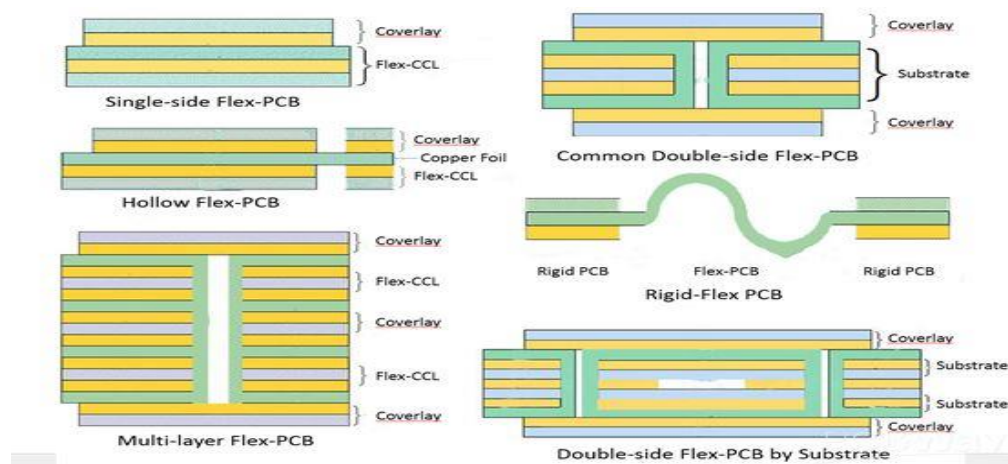
**V. FLEX PCB HIGH-SPEED MATERIALS**

Flex PCBs, or flexible printed circuit boards, are essential to modern electronic devices. They are widely used in applications where traditional rigid PCBs are unsuitable, such as in wearable devices, medical devices, and aerospace applications. The design of Flex PCBs requires careful consideration of various factors, including the choice of materials.



Picture 6: Flex PCB





Picture 7: Flex PCB Stackup's

Properties/Construction of Flex PCB High-Speed Materials include: (Temperature, flexibility, signal integrity) Flexibility is important in Flex PCBs, as it enables the PCB to bend and conform to the shape of the device. High Glass Transition Temperature ( $T_g$ ): A high  $T_g$  is important in high-speed Flex PCBs, as it ensures that the PCB material can withstand high temperatures during the manufacturing process and in operation.

Polyimide (PI) is the most widely used material for flex PCBs. It offers excellent thermal stability, high dielectric strength, and good mechanical flexibility. Polyester (PET) is another popular material for flexible circuits, particularly for low-cost and low-density applications. Polytetrafluoroethylene (PTFE) is a high-performance material known for its excellent electrical properties, low dielectric constant, and high thermal stability.

Copper Clad Laminate (CCL): CCL is a composite material consisting of a copper layer bonded to a substrate. The substrate material can be polyimide, polyester, or other flexible materials. CCL provides the necessary electrical conductivity and mechanical strength for flex PCBs.

Applications of Flex PCB High-Speed Materials Flex PCB high-speed materials are commonly used in applications such as:

- **Wearable Devices:** Wearable devices, such as smartwatches and fitness trackers, require Flex PCBs that can withstand repeated bending and flexing while maintaining signal integrity.
- **Medical Devices:** Medical devices, such as implantable sensors and pacemakers, require Flex PCBs that are biocompatible and can withstand sterilization processes.
- **Aerospace and Defense:** Aerospace and defense applications require Flex PCBs that can withstand harsh

environmental conditions, such as high temperatures, vibration, and shock, while maintaining signal integrity.

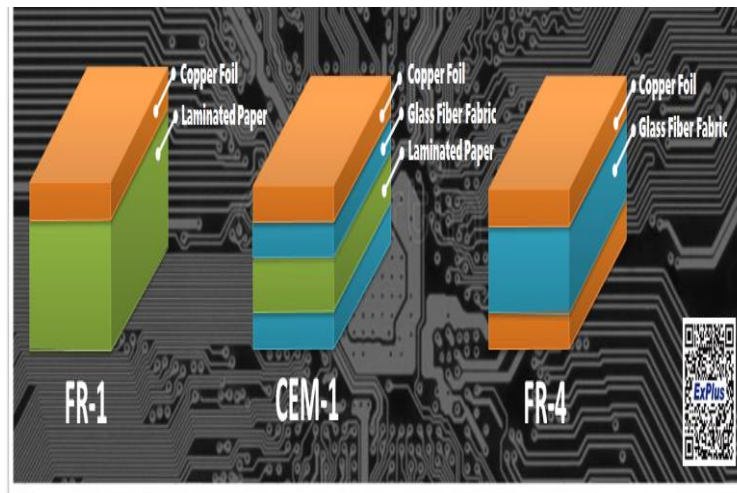
- **Consumer Electronics:** Consumer electronics, such as smartphones and tablets, require Flex PCBs designed to fit into tight spaces and withstand repeated bending and to flex while maintaining signal integrity.

## VI. PCB LAMINATE MATERIALS

Regarding basic PCB designing and production, there are various choices for PCB materials like FR and CEM series. Here, we will list the basic differences between the common materials.

- **FR series** – The most popular material. FR stands for “Flame Retardants,” which is a glass fiber epoxy laminate material that meets the UL94v-0 fire standard set by UL. The differences between material FR series (FR-1 to FR-4) are their thermal properties, heat resistance, and electrical strength. FR-4 is suitable and widely used for single-layer to multi-layered PCBs.
- **CEM series** – Low grade but widely used for single / double-layer PCBs CEM-1 to CEM-3. CEM-1 is a composite material composed of woven glass fabric and paper core combined with epoxy resin. Widely used for single-layer PCBs, and CEM-3 is made of glass cloth and glass mat composite substrate material, which was developed based on FR-4. For double-layer PCBs, it is a great alternative to FR-4. Comparison of prices for the above materials:  $FR-1 < FR-2 < CEM-1 < CEM-3 < FR-4$  (Picture 8).

So far, the material FR-4 is still the best choice for PCB production due to its cost-efficiency and good characteristics. And CEM-1 is the first choice of LED lighting manufacturers because it has achieved the best balance between heat dissipation performance and cost.



Picture 8: Material Comparison

PCBs provide a reliable, cost-effective means of interconnecting and mounting electronic components, making them an indispensable part of our daily lives. A crucial aspect of any PCB is the laminate material that it is made of.

PCB laminate materials comprise a substrate, a copper layer, and a solder mask. The substrate is typically made of fiberglass or some other composite material, while the copper layer is the conductive component that carries the electrical signals between components. The solder mask provides a protective layer that prevents accidental short circuits and damage to the PCB.

There are several types of PCB laminate materials available, each with its own unique set of properties that make it suitable for specific applications. The most common types of laminate materials are:

- **High-Temperature Laminates:** High-temperature laminates are made of materials that can withstand high temperatures without losing their mechanical or electrical properties. These materials are used in applications that require high-temperature resistance, such as aerospace, military, and industrial applications.
- **Metal Core PCBs:** Metal core PCBs are made of a metal core, usually aluminum or copper, which provides excellent heat dissipation. These types of PCBs are ideal for high-power applications, such as LED lighting and power electronics.
- **Ceramic PCBs:** Ceramic PCBs are made of ceramic materials, such as alumina or beryllia, which provide excellent thermal conductivity and electrical insulation. These types of PCBs are ideal for high-frequency applications, such as microwave and RF circuits.

PCB laminate materials are typically composed of a substrate, a copper layer, and a solder mask. The substrate is usually made of fiberglass or some other composite material, while the copper layer is the conductive component that carries the electrical signals between components. The solder mask provides a protective layer that prevents accidental short circuits and damage to the PCB.

The PCB laminate material will depend on the specific type of material being used. For example, FR-4, one of the most common PCB laminate materials, typically consists of woven fiberglass cloth infused with an epoxy resin. The epoxy resin provides the mechanical strength and stability of the substrate, while the fiberglass provides the reinforcement.

The copper layer is typically deposited onto the surface of the substrate using a process called electroplating. The thickness of the copper layer can vary depending on the application, but it is typically in the range of 1-2 ounces per square foot.

The solder mask is typically made of a polymer-based material, such as epoxy or polyimide, applied to the surface of the copper layer. The solder mask protects the copper layer from oxidation and prevents accidental short circuits between components.

#### *A. Properties of Laminate Material in Fabricate printed circuit boards*

- **Mechanical Properties:** There are many factors to think about while working with laminates.
- **Peel Strength:** Peel strength refers to the adhesive force between the laminate layers and the copper delamination.
- **Flexural Strength:** It has excellent flexural strength because it can repeatedly bend without breaking.
- **Delamination Time:** After subjecting to stress, how long it takes for a material to start peeling apart
- **Chemical Properties:** For different laminate materials, the following chemical characteristics are important.
- **Flame Resistance (FR):** If your PCBs are for heat application, the FR of the laminate should be considered.
- **Moisture Absorption:** Delamination occurs more quickly when the laminate absorbs moisture.
- **MCR:** PCB laminate with excellent resistance to methyl chlorine will have a wide chemical tolerance.
- **Thermal Properties:** As such, the following thermal qualities of laminate material must be considered. Higher Glass Transition Temperatures, Thermal Expansion, Thermal Conductivity, and Decomposition Temperature.

- **Dielectric Material Properties:** Avoid material losses on exposure to electric shocks within an acceptable range. PCB laminate material will depend on the specific type of material being used. However, in general, PCB laminate materials are composed of a substrate, a copper layer, and a solder mask, with each component's exact composition and properties varying depending on the application.

## VII. PCB SEQUENTIAL LAMINATION

One of the popular techniques used in PCB manufacturing is Sequential Lamination, which involves the process of building PCBs in layers. Sequentially laminated Printed Circuit Board (PCB) goes through two to multi-lamination cycles. This build approach is the mainstay of High-density Interconnection (HDI) products and has been used in many forms.

### A. What is Sequential Lamination?

Sequential lamination is a process of manufacturing PCBs in which each layer of the board is fabricated separately and then combined into a single unit. The layers are then laminated together using a specialized machine that applies heat and pressure to fuse the layers into a single unit. The process is repeated for every layer, resulting in a complex multi-layer PCB.

- **HDI (2-6 Layers):** With 1 to 3 lamination cycles applied through this, outer layers with micro vias (Blind vias and inner layers with mechanical vias).
- **HDI (2-26 Layers):** With Multiple-Lamination, cycles applied through this based on Layer count and Stackup, outer layers with micro vias (Blind vias and inner layers with mechanical vias).

### B. Common Sequential Lamination Approach.

Advantages of Sequential Lamination Sequential lamination offers several advantages over other PCB manufacturing techniques, including:

- **High Density and Miniaturization** The sequential lamination process allows for the creation of high-density and miniaturized PCBs. By building the PCB in layers, designers can stack more components and traces in a smaller area, resulting in a more compact design.
- **Improved Signal Integrity** Sequential lamination also provides improved signal integrity. By building the PCB in layers, designers can control the impedance of the circuit, ensuring that signals are transmitted without distortion or interference.
- **Flexibility and Customization** The sequential lamination process is highly flexible and can be customized to meet specific application requirements. Designers can choose the number of layers, the thickness of each layer, and the materials used in each layer to create a customized PCB.
- **Reduced Cost** Sequential lamination can also be cost-effective compared to other PCB manufacturing techniques. By building the PCB in layers, designers can use less expensive materials for some of the layers, reducing the overall cost of the PCB.

Sequential lamination is commonly used in the manufacturing of complex PCBs, such as those used in aerospace, medical, and military applications. Here are some steps of sequential lamination:

- **Determine the number of layers:** The first step is determining the number of layers required for the design. This will depend on the complexity of the circuit and the required electrical performance.
- **Determine the layer stack up:** Based on the number of layers required, determine the layer stack up for the PCB. The layer stack-up will specify the order of the layers and the thickness of each layer.
- **Calculate the via sizes:** Vias are used to connect the different layers of the PCB. The size of the via will depend on the size of the trace and the thickness of the PCB material.
- **Determine the lamination sequence:** The lamination sequence specifies the order in which the layers will be laminated together. The lamination sequence must be carefully designed to ensure the vias are properly aligned and the impedance is maintained.
- **Calculate the final thickness:** The final thickness of the multilayer PCB can be calculated by adding up the thickness of each layer in the layer stack up.

It's important to note that the specific formulas and considerations for sequential lamination will depend on the specific requirements of your design. Therefore, it's recommended to consult with a PCB designer or use simulation software to ensure the best design for your application.

## VIII. PCB NON-HALOGENATED SUPER LAMINATE ADVANCE

One aspect of PCBs critical to their performance and safety is their composition, specifically the presence of halogenated materials. Halogens, such as chlorine and bromine, are used in the production of PCBs to improve their flame resistance. However, these materials can be harmful to the environment and human health. Non-halogenated super laminate advances are an innovative solution that offers the same benefits as halogenated materials without harmful effects.

Non-halogenated super laminate advances are made from a combination of materials, including epoxy resin, fiberglass, and other additives. These materials are carefully selected to provide excellent flame resistance and durability without the use of halogens. The non-halogenated super laminate advances are designed to meet or exceed industry standards, including Underwriters Laboratories (UL) and International Electrotechnical Commission (IEC).

One advantage of non-halogenated super laminate advances is that they are environmentally friendly. Halogens, such as bromine and chlorine, are persistent organic pollutants that can accumulate in the environment and pose a risk to human health. By using non-halogenated materials, PCB manufacturers can reduce their environmental impact and ensure that their products are safe for use.



Another advantage of non-halogenated super laminate advances is their improved thermal performance. These laminates have excellent thermal stability, which means they can withstand high temperatures without losing their mechanical or electrical properties. This makes them ideal for use in high-performance applications, such as aerospace and defense.

Non-halogenated super laminate advances are also more cost-effective than halogenated materials. While halogenated materials offer excellent flame resistance, they are more expensive to produce. They can be easier to handle and dispose of, which can reduce the overall cost of PCB production.

In conclusion, non-halogenated super laminate advances offer an innovative solution for PCB manufacturers who want to improve the safety and performance of their products while reducing their impact on the environment. These laminates provide excellent flame resistance, thermal stability, and durability without the use of harmful halogens. They are cost-effective and meet or exceed industry standards for safety and performance, making them an ideal choice for a wide range of applications. Here are some general considerations for selecting a non-halogenated super laminate for your PCB design:

- **Composition:** Non-halogenated super laminates are typically composed of a mixture of epoxy resin, filler materials, and reinforcing fibers. The specific composition will depend on the manufacturer and the desired properties of the material.
- **Environmental compliance:** non-halogenated super laminates are designed to be environmentally compliant, meaning they do not contain harmful halogens that can contribute to pollution or health risks. It's important to ensure that the selected laminate meets any applicable environmental regulations, such as RoHS.
- **Other properties:** non-halogenated super laminates can also have other properties that are important for specific applications, such as flame resistance, electrical conductivity, and chemical resistance.

It's important to note that selecting a non-halogenated super laminate for your PCB design will depend on the specific requirements of your design.

## IX. PCB MATERIAL GLASS WEAVE SKEW

Glass Weave Skew is a phenomenon that can occur during the lamination process. It refers to the misalignment or skewing of the glass weave in the PCB material, which can lead to issues with signal integrity, impedance, and other problems. In this article, we will discuss what Glass Weave Skew is, its causes, and its effects on PCB performance.

What is Glass Weave Skew? Glass Weave Skew occurs when the glass weave in the PCB material is not aligned correctly during the lamination process. The glass weave is the pattern of the glass fibers that run through the epoxy resin in the PCB material. When the glass weave is skewed, the

fibers are misaligned, which can cause a variety of issues with the PCB's performance.

What Causes Glass Weave Skew? Several factors can contribute to Glass Weave Skew, including:

- **Thermal Expansion:** During the lamination process, the PCB material is heated and then cooled rapidly, which can cause the material to expand and contract. If the expansion and contraction are not uniform, the glass weave can become skewed.
- **Uneven Pressure:** If the pressure is not distributed evenly across the PCB material during the lamination process, the glass weave can become skewed.
- **Manufacturing Defects:** The PCB material can have inherent manufacturing defects, such as uneven glass fiber distribution, which can cause Glass Weave Skew.

Effects of Glass Weave Skew on PCB Performance  
Glass Weave Skew can have several negative effects on PCB performance, including:

- **Impedance Variations:** The skewed glass weave can cause impedance variations in the PCB, which can lead to signal reflections, crosstalk, and other signal integrity issues.
- **Warping:** The skewed glass weave can cause the PCB to warp, which can lead to issues with component placement and soldering.
- **Solderability Issues:** The misaligned fibers can cause issues with the adhesion of the solder paste, leading to solderability issues.
- **Delamination:** The skewed glass weave can cause delamination of the PCB material, which can lead to PCB failure.

How to Prevent Glass Weave Skew? There are several ways to prevent Glass Weave Skew during the PCB manufacturing process, including:

- **Controlling the Lamination Process:** The lamination process must be controlled carefully to ensure that the pressure and temperature are applied evenly across the PCB material.
- **Choosing the Right Material:** Choosing a PCB material with a uniform glass weave can help prevent Glass Weave Skew.
- **Design Optimization:** Design optimization can help prevent Glass wave Skew by minimizing the thermal expansion and ensuring that the pressure is distributed evenly across the PCB material.

The glass weave skew is an important consideration when laminating glass weave PCB materials. The skew refers to the angular displacement of the glass weave pattern between adjacent layers. Here is an example of a formula for calculating glass weave skew:

- **Determine the glass weave pattern:** The first step is to determine the glass weave pattern of the PCB material being used. This pattern will determine the angle of the glass fibers in the material.
- **Determine the layer stack up:** Based on the required number of layers and the electrical requirements of the design, determine the layer stack up for the PCB.



- Calculate the skew angle: The skew angle is determined by the difference in angle between the glass weave pattern of adjacent layers. The formula for calculating the skew angle ( $\Delta\theta$ ) is:

$$\Delta\theta = \sin^{-1}(Dd/(2 \times T \times \sin(\theta)))$$

where  $Dd$  is the distance between adjacent glass fibers,  $T$  is the thickness of the PCB material, and  $\theta$  is the angle of the glass weave pattern.

- Determine the acceptable skew: The acceptable skew angle will depend on the specific requirements of the design. Typically, a skew angle of less than 5 degrees is acceptable for most designs.
- Adjust the layer stack up: If the calculated skew angle is outside the acceptable range, the layer stack up may need to be adjusted. This may involve changing the thickness of the PCB material or adjusting the layer order to minimize the skew angle.

It's important to note that the specific formulas and considerations for calculating glass weave skew will depend on the specific requirements of your design and the glass weave pattern of the PCB material being used. Therefore, it's recommended to consult with a PCB designer or use simulation software to ensure the best design for your application.

Spread glass and resin content plays a role in the Glass Weave Skew, and another factor is Low dk class/construction fare/ply (Single or dual Ply).

## X. PCB MATERIAL GLASS FABRIC

Glass fabric is a commonly used material in the manufacturing of printed circuit boards (PCBs). PCBs are an essential component of electronic devices, and their performance and reliability depend on the materials used in their construction. Glass fabric is a preferred material due to its excellent mechanical, electrical, and thermal properties.

Glass fabric is made by weaving together thin strands of glass fibers. The glass fibers are treated with epoxy resin, then cured to create a solid and stable material. The thickness of the glass fibers and the number of layers used determines the strength and thickness of the glass fabric.

One of the key benefits of glass fabric is its high tensile strength. Tensile strength refers to the amount of force a material can withstand before it breaks. Glass fabric has a high tensile strength, which makes it ideal for use in PCBs. PCBs need to withstand a range of mechanical stresses, and glass fabric's strength makes it resistant to bending and cracking.

Glass fabric is also a good electrical insulator. PCBs have copper traces that carry electrical signals between the electronic components on the board. The insulating material between the copper traces must be able to prevent the electrical signals from crossing over and causing a short circuit. Glass fabric's high dielectric strength makes it an excellent insulator for use in PCBs.

Another advantage of glass fabric is its thermal stability. Electronic devices generate heat during operation, and the PCB must be able to dissipate this heat to prevent damage to the components. Glass fabric's high thermal stability means that it can withstand high temperatures without degrading or losing its strength.

Glass fabric is also chemically stable, which means it is resistant to corrosion and degradation caused by chemical exposure. PCBs may be exposed to various chemicals during their operation and using glass fabric ensures that the PCB remains stable and does not deteriorate.

Glass fabric is an ideal material for use in the manufacturing of printed circuit boards. Its high tensile strength, excellent electrical insulation properties, thermal stability, and chemical resistance make it a preferred material for PCBs. The use of glass fabric in PCBs ensures that the board can withstand mechanical stresses, electrical signals, and high temperatures, improving the overall performance and reliability of electronic devices.

Glass fabric used in PCB materials refers to the weave pattern and weight of the fabric.

Glass fabric weave pattern: Woven E-glass Glass fabric weight:  $X \text{ oz/yd}^2$  (where  $X$  is a specific weight in ounces per square yard)

The E-glass weave pattern is used for PCB materials, as it provides good electrical properties, high strength, and dimensional stability.

The weight of the glass fabric used in PCB materials is typically specified in ounces per square yard. This weight determines the thickness and strength of the laminate. Common weights for glass fabric used in PCB materials include  $4 \text{ oz/yd}^2$ ,  $6 \text{ oz/yd}^2$ , and  $8 \text{ oz/yd}^2$ .

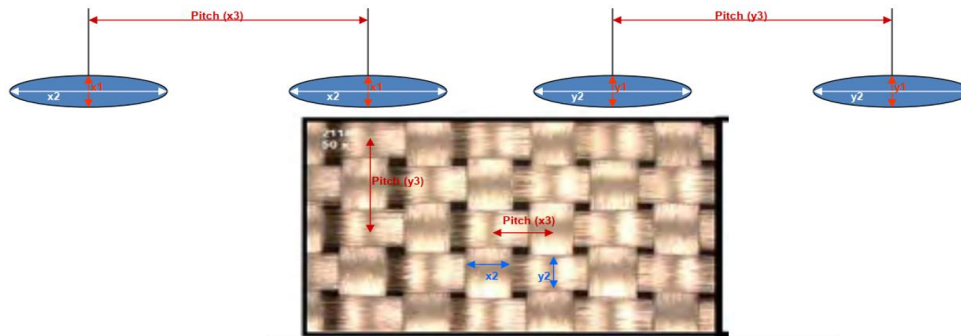
It's important to note that the glass fabric used in PCB materials will depend on the specific requirements of your design, such as the dielectric constant and loss tangent.

Class Style	Weave	Warp Count	Fill Count	Warp Yarn	Fill Yarn	Fabric Thickness Inches	Fabric Thickness mm	Fabric Nominal Weight OSY	Fabric Nominal Weight g/m2
1035	Plain	66	68	ECD 900-1/0	ECD 900-1/0	0.0011	0.030	0.88	30
1037	Plain	70	73	ECC 1200-1/0	ECC 1200-1/0	0.0011	0.030	0.68	23
1067	Plain	70	70	ECD 900-1/0	ECD 900-1/0	0.0014	0.035	0.91	31
106	Plain	56	56	ECD 900-1/0	ECD 900-1/0	0.0013	0.032	0.73	25
1078	Plain	54	54	ECD 450 1/0	ECD 450 1/0	0.0017	0.040	1.41	48
1086	Plain	60	60	ECD 450 1/0	ECD 450 1/0	0.0022	0.050	1.60	54
1080	Plain	60	47	ECD 450-1/0	ECD 450-1/0	0.0021	0.064	1.45	49
2113	Plain	60	56	ECE 225-1/0	ECD 450-1/0	0.0031	0.074	2.31	78
2313	Plain	60	64	ECE 225-1/0	ECD 450-1/0	0.0033	0.080	2.38	81
3313	Plain	61	62	ECDE 300-1/0	ECDE 300-1/0	0.0033	0.081	2.43	82
3070	Plain	70	70	ECDE 300-1/0	ECDE 300-1/0	0.0034	0.086	2.74	93
2116	Plain	60	58	ECE 225-1/0	ECE 225-1/0	0.0037	0.097	3.22	109
1506	Plain	46	45	ECE110-1/1	ECE 110-1/0	0.0056	0.140	4.89	165
1652	Plain	52	52	ECG 150-1/0	ECG 150-1/0	0.0045	0.114	4.09	142
7628	Plain	44	31	ECG 75-1/0	ECG 75-1/0	0.0068	0.173	6.00	203

**Fiberglass Yarn Nomenclature**

1st Letter E = E-glass ( electrical grade )  
 2nd Letter C = Continuous Filaments  
 3rd Letter Filament Diameter C, D, E, DE, G  
 1st number Yardage in one pound  
 2nd number Number of strands in a yarn/ strands plied or twisted

Picture 9: Different type of Glass Fabric Details

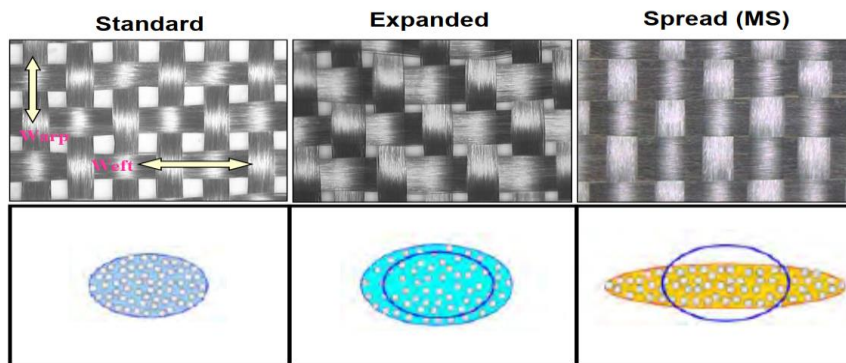


Glass Fabric	Measurements Results (mils)					
	X1	X2	X3	Y1	Y2	Y3
1035	1.12	10.20	16.10	0.53	10.90	15.50
106	1.00	4.80	18.50	0.60	10.20	20.60
1067	0.82	8.85	14.30	0.78	12.40	13.70
1080	1.60	8.20	17.00	1.10	12.10	22.40
1086	1.44	10.80	16.60	1.00	14.70	17.10
2113 / 2313	2.40	10.50	17.00	1.00	15.30	18.20
3313	1.90	13.10	16.20	1.50	11.00	16.30
3070	1.70	12.70	14.80	1.70	12.60	14.20
2116	2.20	14.10	17.20	2.00	14.50	17.30
1652	2.40	15.30	17.50	2.90	15.90	18.80

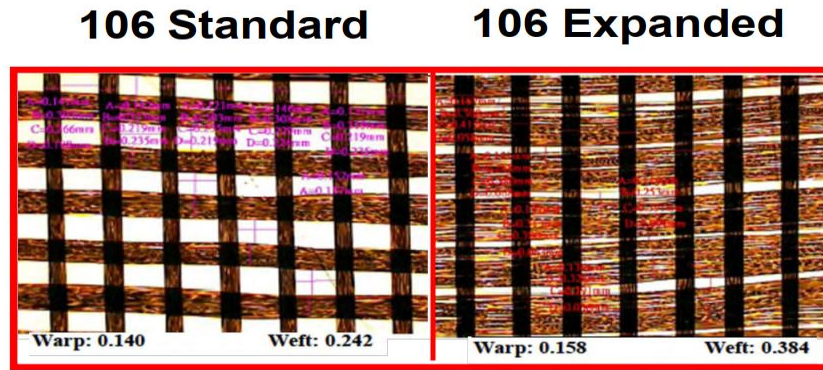
Picture 10: Glass Dimensions

A. Glass Definitions based on dimensions

- Expanded Weave – Glass Spread More than Standard in one direction
- Open Weave – Same as Expanded
- Open Filament – Same as Expanded
- Spread Glass – Glass is spread out in various processes
- Mechanically Spread (MS) Glass – Glass is Mechanically Spread in both the warp and fill directions.
- Square Weave – Glass that has a Balanced Density and/or Yarn count in Warp and Fill Directions.
- Flat Glass – Glass is made from fibers with little or zero twist



Picture 11: Explanation Glass Spreading Terminology

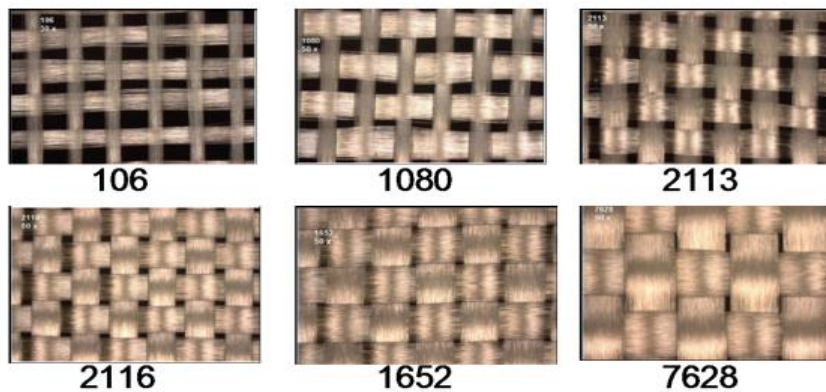


Picture 12: Standard Weave Versus Expanded Weave

**XI. PCB MATERIAL GLASS WEAVE SKEW SENSITIVITY**

Printed Circuit Boards (PCBs) are a critical component of modern electronics, and their performance and reliability depend on a variety of factors, including the choice of materials used in their construction. One important material property that can affect PCB performance is Glass Weave Skew Sensitivity. In this article, we will discuss what Glass Weave Skew Sensitivity is, how it affects PCB performance and ways to mitigate its effects.

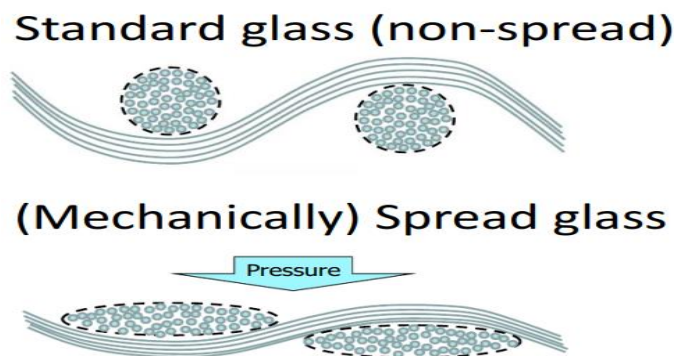
What is Glass Weave Skew Sensitivity? Glass Weave Skew Sensitivity is the degree to which a PCB is affected by Glass Weave Skew, which refers to the misalignment or skewing of the glass weave in the PCB material during the lamination process. The sensitivity of a PCB to Glass Weave Skew depends on several factors, including the type of PCB material used, the thickness of the material, and the number of layers in the PCB.



Picture 13: The Realities of PCB Construction with different material and design strategies

Signals see a spatially varying Dk and a spatially varying TD Local variation and global variation

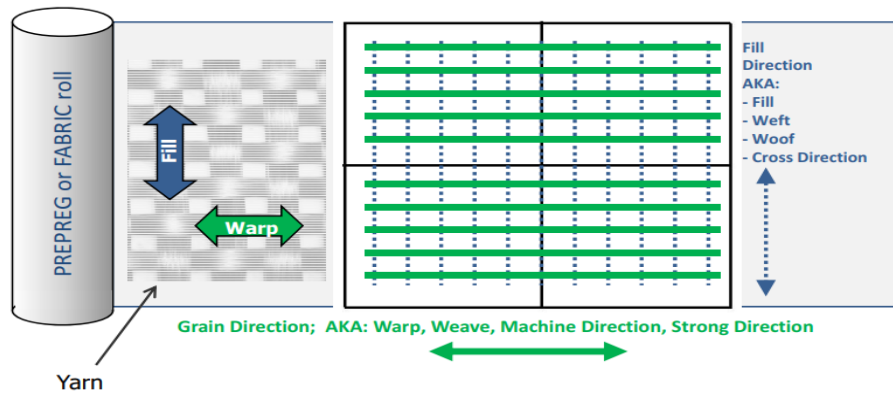
A. Material strategies  
 Standard glass Dk~6.8 (resin~3.3) non-spread, mechanically spread glass Dk~4.8 (resin~3.3) low-Dk glass.



Pictures 14: Standard glass Dk~6.8 (resin~3.3) non-spread, mechanically spread glass Dk~4.8 (resin~3.3) low-Dk glass.

Further Complication: Glass is Stretched (Straighter) along the Warp Direction





Pictures 15: Glass is Stretched (Straighter) along the Warp Direction

Typical spec for acceptable line to line skew ~ 20% UI

- @ 10 Gbps, UI = 100 psec, max acceptable skew ~ 20 psec total (end-to-end) (very conservative)
- @ 20 inch channel, 20 psec/20 inch = 1 psec/in as max acceptable glass weave skew (if it eats entire skew budget)

How Does Glass Weave Skew Sensitivity Affect PCB Performance? Glass Weave Skew Sensitivity can affect PCB performance in several ways, including:

- **Impedance Variations:** Glass Weave Skew can cause impedance variations in the PCB, leading to signal reflections, crosstalk, and other signal integrity issues. The degree of sensitivity to Glass Weave Skew determines how significant these issues are and how difficult they are to resolve.
- **Warping:** Glass Weave Skew can cause the PCB to warp, leading to issues with component placement and soldering. The degree of sensitivity to Glass Weave Skew determines how severe the warping is and how difficult it is to mitigate.
- **Solderability Issues:** Glass Weave Skew can cause issues with the adhesion of the solder paste, leading to solderability issues. The degree of sensitivity to Glass Weave Skew determines how significant these issues are and how difficult they are to address.
- **Delamination:** Glass Weave Skew can cause delamination of the PCB material, leading to PCB failure. The degree of sensitivity to Glass Weave Skew determines how likely delamination is to occur and how severe the consequences are.

How to Mitigate the Effects of Glass Weave Skew Sensitivity? There are several ways to mitigate the effects of Glass Weave Skew Sensitivity, including:

- **Choosing the Right PCB Material:** Choosing a PCB material with a low sensitivity to Glass Weave Skew can reduce the likelihood and severity of issues related to Glass Weave Skew.
- **Controlling the Lamination Process:** Controlling the lamination process carefully can reduce the degree of Glass Weave Skew and its effects on the PCB.
- **Design Optimization:** Optimizing the PCB design can reduce the impact of Glass Weave Skew by minimizing thermal expansion and ensuring that the pressure is distributed evenly across the PCB material.

- **Testing and Inspection:** Regular testing and inspection can help identify and address issues related to Glass Weave Skew before they cause significant problems.

Glass Weave Skew Sensitivity is an important consideration in PCB design and manufacturing. It can affect PCB performance in several ways, including impedance variations, warping, solderability issues, and delamination. Mitigating the effects of Glass Weave Skew Sensitivity requires careful material selection, lamination process control, design optimization, and regular testing and inspection. PCB manufacturers can ensure that their products perform reliably and consistently by taking these steps.

The sensitivity of glass weave skew refers to how much the skew angle changes due to changes in the PCB material thickness or the distance between the glass fibers. Here is an example of a formula for calculating glass weave skew sensitivity:

- **Determine the glass weave pattern:** The first step is to determine the glass weave pattern of the PCB material being used. This pattern will determine the angle of the glass fibers in the material.
- **Determine the layer stack up:** Based on the required number of layers and the electrical requirements of the design, determine the layer stack up for the PCB.
- **Calculate the skew angle:** The skew angle is determined by the difference in angle between the glass weave pattern of adjacent layers. The formula for calculating the skew angle ( $\Delta\theta$ ) is:  

$$\Delta\theta = \sin^{-1}(Dd/(2 \times T \times \sin(\theta)))$$

Where  $Dd$  is the distance between adjacent glass fibers,  $T$  is the thickness of the PCB material, and  $\theta$  is the angle of the glass weave pattern.

- **Calculate the sensitivity:** The sensitivity of the skew angle to changes in the PCB material thickness ( $\Delta T$ ) or the distance between the glass fibers ( $\Delta Dd$ ) is given by the following formula:  

$$\text{Sensitivity} = (\Delta\theta/\Delta T) \times (T/\theta) + (\Delta\theta/\Delta Dd) \times (Dd/\theta)$$
- **Determine the acceptable sensitivity:** The acceptable sensitivity will depend on the specific requirements of the design. A lower sensitivity indicates that the skew angle is less sensitive to changes in the PCB material thickness or the distance between the glass fibers.



It's important to note that the specific formulas and considerations for calculating glass weave skew sensitivity will depend on the specific requirements of your design and the glass weave pattern of the PCB material being used. Therefore, it's recommended to consult with a PCB designer or use simulation software to ensure the best design for your application.

## XII. PCB HIGH GLASS TRANSITION TEMPERATURE

The performance and reliability of a PCB depend on various factors, including the glass transition temperature (Tg) of the board. A high Tg is essential to ensure that the PCB can withstand high temperatures without losing its mechanical or electrical properties.

The glass transition temperature is the temperature at which a material transitions from a hard, glassy state to a softer, rubbery state. In the case of PCBs, the Tg is the temperature at which the resin used in the board transitions from a rigid state to a more flexible one. The Tg is an important property of the resin used in the board since it affects the overall mechanical and electrical properties of the PCB.

To achieve a high Tg in a PCB, manufacturers use resins with a high Tg value. These resins are typically made of a combination of epoxy and other materials, such as polyimide. Polyimide is a type of plastic with excellent thermal stability and a high Tg value, making it an ideal material for use in PCBs.

Manufacturers may also use a combination of materials and techniques to increase the Tg of a PCB. For example, adding fillers to the resin can help to increase its Tg value. Fillers are materials that are added to the resin to increase its stiffness and strength. Some common fillers used in PCBs include glass fibers and ceramics.

Overall, a high Tg is an essential property of a PCB since it ensures that the board can withstand high temperatures without losing its mechanical or electrical properties. To achieve a high Tg, manufacturers use resins with a high Tg value, add fillers to the resin, and carefully design the PCB to reduce the amount of resin used. By doing so, they can create high-quality PCBs that are reliable and perform well in a wide range of applications.

The glass transition temperature (Tg) of a PCB material is a measure of its ability to maintain its mechanical properties over a range of temperatures. A high Tg is desirable in many applications, as it ensures that the material can withstand high temperatures without losing its strength or becoming brittle. Here is an example of a formula for selecting a PCB material with a high glass transition temperature:

- **Determine the required Tg:** The first step is to determine the required Tg for your application. This will depend on the specific requirements of your design, including the operating temperature range and the desired level of mechanical stability.

- **Calculate the Tg:** The Tg of a material can be measured using a variety of techniques, including differential scanning calorimetry (DSC) and dynamic mechanical analysis (DMA). The Tg is typically determined by measuring the temperature at which the material undergoes a transition from a glassy state to a rubbery state.
- **Evaluate the thermal properties:** The thermal properties of the material, such as thermal conductivity and coefficient of thermal expansion, will impact its ability to dissipate heat and prevent damage to components.
- **Compare and select:** Compare the properties and cost of each candidate material and select the material that best meets the design requirements and fits within the project budget.

It's important to note that the specific formulas and considerations for selecting a PCB material with a high glass transition temperature will depend on the specific requirements of your design. Additionally, it's important to consider the mechanical properties of the material, such as its strength, stiffness, and durability, when selecting a material for your design.

## XIII. PCB LAMINATE HIGH HEAT RESISTANCE

One aspect of PCBs that is essential for their performance is laminate, which refers to the insulating material that separates the individual layers of the board. High heat resistance is necessary for the laminate to ensure that the PCB can withstand high temperatures without losing its mechanical or electrical properties.

To achieve high heat resistance in the laminate, PCB manufacturers use materials with excellent thermal stability, such as high-temperature epoxy resins, polyimide, and ceramic. These materials can withstand high temperatures without breaking down or losing their properties, making them ideal for use in PCBs that are exposed to high temperatures.

Another technique used to achieve high heat resistance in the laminate is the use of fillers. Fillers are materials added to the laminate to increase its stiffness and thermal stability. Some common fillers used in PCBs include glass fibers and ceramics, which improve the laminate's ability to withstand high temperatures.

In addition to material selection and design, manufacturers may also subject the laminate to various tests to ensure that it meets the required specifications for heat resistance. These tests may include thermal cycling, which involves exposing the laminate to extreme temperatures repeatedly to simulate real-world conditions.

Overall, achieving high heat resistance in the laminate is essential to ensure the reliability and performance of PCBs, particularly in applications where the PCB is exposed to high temperatures. PCB manufacturers use a combination of materials, fillers, and design techniques to create laminates that can withstand high temperatures without losing their mechanical or electrical properties. By doing so, they can

produce high-quality PCBs that are reliable and perform well in a wide range of applications.

High heat resistance is a desirable property in PCB laminates for applications where the PCB is exposed to high temperatures, such as in automotive or aerospace electronics. Here are some considerations for selecting a PCB laminate with high heat resistance:

- **Material selection:** The choice of material used in the laminate can significantly impact the heat resistance of the PCB. High-temperature materials typically have high glass transition temperatures ( $T_g$ ) and can withstand exposure to high temperatures without degrading or losing their mechanical properties. Common materials used in high-heat-resistant PCB laminates include ceramic-filled epoxy, polyimide, and PTFE.
- **Glass transition temperature ( $T_g$ ):** The  $T_g$  of the material is a measure of its thermal stability. A high  $T_g$  material is able to withstand exposure to high temperatures without degrading. The formula for calculating the  $T_g$  of a material is:

$$T_g = 1 / [(1/T_{g1}) + (1/T_{g2}) + (1/T_{g3}) + \dots]$$

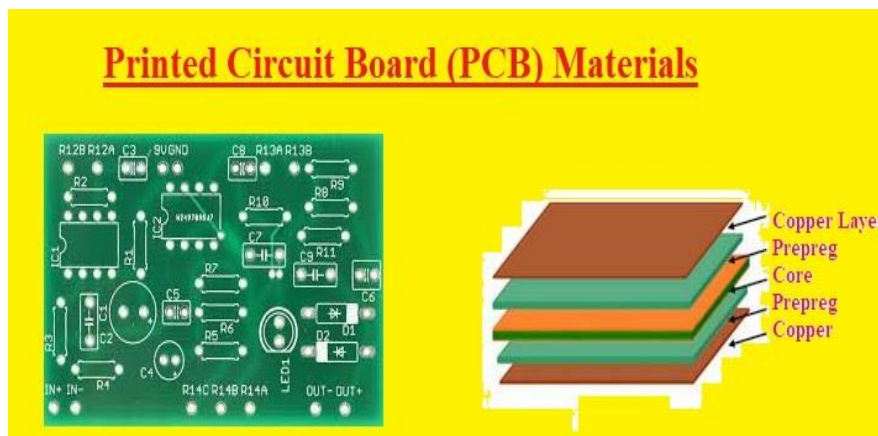
where  $T_{g1}$ ,  $T_{g2}$ ,  $T_{g3}$ , etc. are the individual  $T_g$  values of the components in the material.

- **Thermal conductivity:** The thermal conductivity of the material is also an important consideration for high-heat-resistant PCB laminates. A high thermal conductivity material can help to dissipate heat and prevent damage to components. The formula for calculating thermal conductivity is:

$$k = (Q \times L) / (A \times \Delta T).$$

where  $k$  is the thermal conductivity of the material,  $Q$  is the heat flow rate,  $L$  is the length of the material,  $A$  is the cross-sectional area of the material, and  $\Delta T$  is the temperature difference across the material.

#### XIV. PRINTED CIRCUIT BOARD MATERIALS



Pictures 16: PCB Material Stackup

PCB materials are generally made up of three elements that work together to meet the specific needs of the electronic system: Copper, Resin, and Glass and Surface Finish.

- **Copper:** Copper is the main metal in PCBs copper is used as an electric current conductor (paths on laminate and parts of electric connectors and electronic elements), and tin is used in solder for connections between elements and tracks on the board.
- **Resin:** Epoxy and polyurethane resins are excellent insulators against aggressive environments and contribute to the mechanical protection of PCBs. Epoxy resins and polyurethane are used for protection in thickness. These systems generally have two liquid components when mixed climb quickly in viscosity until fully cured. Epoxy resins have good mechanical and electrical characteristics, which is why they're the most used material in making printed circuit boards. Aside from having great features, epoxy resins are also inexpensive, and dissolve quickly and safely in cheap solvents, making them easy to use.

- **Glass:** The laminate most used in printed circuit boards (PCBs) is a glass fiber reinforced (fiberglass) epoxy resin with a copper foil bonded onto one or both sides. This glass fiber epoxy laminate is called as FR-4 Laminate or FR-4 CCL. The substrate, or base material, of a PCB, is typically fiberglass. The fiberglass that has historically been used for this purpose is designated as FR-4, a flame-resistant, woven fiberglass.
- **Surface Finish:** HASL / Lead-Free HASL is the predominant surface finish used in the industry. The process consists of immersing circuit boards in a molten pot of a tin/lead alloy and then removing the excess solder by using 'air knives', which blow hot air across the surface of the board.

The PCB core is a hard foundation material coated on one or two aspects with copper. The CORE is employed in the fabrication of single-sided and two-sided boards and in the manufacture of PCBs of sharing arrangements. The PCB center consists of the FR4 elements of titanium epoxy laminates and copper traces.

Some commonly used materials for high-speed PCBs list include below:

#### A. ISOLA - Tachyon 100G

Tachyon 100G laminate materials are designed for very high-speed digital applications up to and beyond speeds of 100 Gb/s. Tachyon 100G products use spread glass and reduced profile copper to mitigate skew and improve rise times, reduce jitter, and increase eye width and height. Tachyon 100G has a nominal dielectric constant (Dk) of 3.02 that is stable between -55°C and +125°C up to 40 GHz. In addition, Tachyon 100G offers a very low nominal dissipation factor (Df) of 0.0021.

#### B. ISOLA - FR408HR

FR408HR is a high-performance FR-4 resin system with a 230°C (DMA) Tg for multilayer PCB applications where maximum thermal performance and reliability are required. R408HR laminate and prepreg products are manufactured with Isola's patented high-performance multifunctional resin system, reinforced with electrical grade (E-glass) glass fabric. This system delivers a 30% improvement in Z-axis expansion and offers 25% more electrical bandwidth (lower loss) than competitive products at this pace.

#### C. Panasonic - MEGTRON 6

Meg-6 Laminate R-5775/Prepreg R-5670. Megtron 6 is a popular, high-speed laminate material used in different types of PCB fabrication and manufacturing for many applications. This advanced basic material is designed for many different high-frequency measuring instruments, integrated circuit (IC) testers and mainframes.

Megtron 6 has a low transmission loss, thick layer count, and many different layers that enhance high-speed network equipment. These laminates have electrical properties that rival Polytetrafluoroethylene (PTFE) materials, thanks to their high processability and unique benefits. Using Megtron 6 laminates enhances your system performance in many different fields, including computing, telecommunications, high-speed transfer, and router applications. Megtron 6 sheets also have a low dielectric constant, high thermal resistance, high-density interconnect (HDI) and increased stability.

#### D. Rogers Corporation's RO4000 Series

This material has a low dielectric constant and loss, making it a good choice for high-frequency applications. Dk of 3.48 +/- 0.05, Dissipation factor of 0.0037 at 10 GHz, Low Z-axis coefficient of thermal expansion at 32 ppm/°C. RO4000 materials are reinforced hydrocarbon/ceramic laminates – not PTFE, and it is designed for performance-sensitive, high-volume applications. Low dielectric tolerance and low loss, Excellent electrical performance, allows applications with higher operating frequencies, and is Ideal for broadband

applications. Volume manufacturing process, RO4000 laminates can be fabricated using standard glass epoxy processes at Competitively priced.

SOME TYPICAL APPLICATIONS: Cellular Base Station Antennas/Power Amplifiers, RF Identification Tags, Automotive Radar, sensors, and LNAs for Direct Broadcast Satellites.

#### E. Arlon- 85N

This ceramic-filled laminate material has a low dielectric constant and high thermal conductivity, making it a good choice for high-speed designs with high power requirements. 85N is the ultimate pure polyimide laminate and prepreg system. Bromine-free chemistry provides best-in-class thermal stability for applications with sustained high in-use temperatures as well as for use in lead-free soldering applications.

Meets IPC4101/40 and /41 description and specification: Pure polyimide, no secondary resin, No epoxy added, blended or reacted.

Best-in-Class thermal properties: Decomposition temperature >407°C, T300>60 min.

Low Z-axis expansion: 1.2% between 50-260°C (vs. 2.5-4.0% for typical high-performance epoxies), Minimizing the risk of latent PTH defects caused during solder reflow and device attachment.

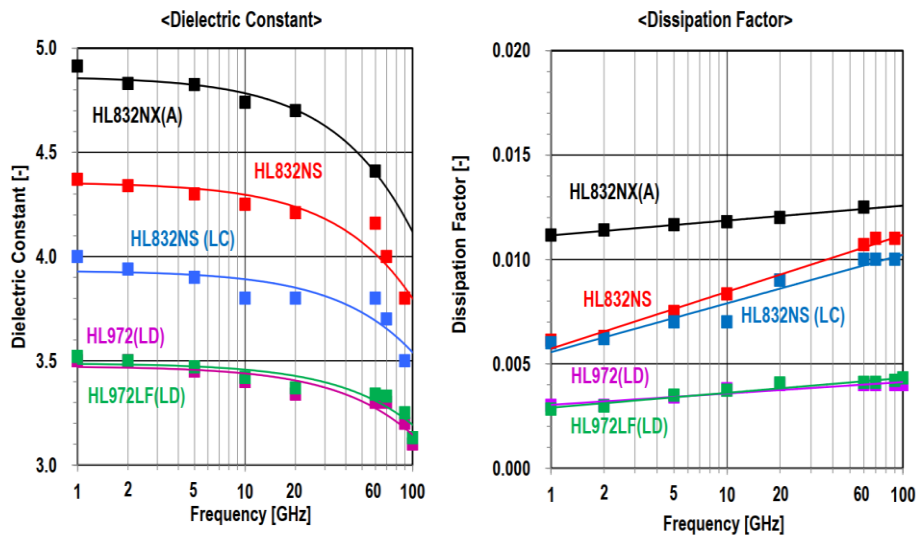
- Decomposition temperature of 407°C, compared with 300-360°C for typical high-performance epoxies, offering outstanding long-term high-temperature performance
- Toughened chemistry resists resin fracturing, Halogen-free chemistry, • Compatible with lead-free processing, RoHS/WEEE compliant

#### F. Mitsubishi Materials

- HL832NSA type LC: Nonhalogenated Super laminate Advance, Low coefficient of thermal expansion, High glass transition temperature, High stable modulus, High heat resistance, Low dielectric constant.
- CCL-HL972LF type LD and GHPL-970LF type LD, Low dielectric properties (Low Dk and Low Df), High Tg and low CTE, good for High-frequency IC packages, and RF application 2.4 to 60GHz

#### ➤ Dielectric Constant and Dissipation Factor (Fig-B)

HL832NX(A), HL832NS, HL832NS(LC), HL972(LD), HL972LF(LD), Thickness: 0.8mm (for 1-20GHz), 0.1mm (for 60-100GHz), Method: Cylindrical Cavity Resonator Method (for 1-20GHz), Fabry-Perot Resonator Method (for 60-100GHz).



Picture 17: Dielectric Constant and Dissipation Factor

*G.TUC (Taiwan Union Technology Corp.)*

**TU-862T:** TU-862T High Tg halogen-free materials are made of High Tg epoxy resin and E-glass fabric. Unlike conventional FR-4 material using brominated resin as flame retardant, TU-862T /TU-862P T achieves the flammability class of UL94V-0 by incorporating phosphorus and nitrogen compounds in the materials. Very low coefficient of thermal expansion, Higher Tg characteristics, Anti-CAF capability. Glass Styles: 106, 1080, 2113, 2116, 1506 and 7628, etc.

*H.EK (Elite Material c. Ltd)*

**EM-890K / EM-89BK:** High Tg / Ultra Low Loss / Halogen Free, Ultra-low Df, and excellent electrical performance, Excellent CAF resistance, Multiple laminations, and high thermal reliability applications, For high-speed Ethernet, network, 5G, and Antenna applications.

*I. DOOSAN - DS-7409DV*

High Speed, Low Loss Multi-Layer Materials, Superior Heat Resistance, Low Water Absorption, Good Dimensional Stability, High Tg (over 180°C), Superior Moisture Resistance, UV blocking & AOI compatibility.

## XV. CONCLUSION

High-speed materials have become an essential component of modern PCBs. PCB Material selection and Stackup design are crucial for modern high-speed digital world applications. There were several PCB materials and PCB manufacturers' entire world. We should do more study before finalizing the PCB material. There are several Simulation tools to simulate and select the correct material based on the requirement of the outcome product.

Get the datasheet of the material and find the right materials. These materials are designed to transmit signals at high frequencies without losing signal integrity. PTFE, ceramic-filled hydrocarbon, and liquid crystal polymer are some of the popular high-speed materials used in PCBs. These materials have low loss tangent, dielectric constant, and thermal stability. When selecting a high-speed material for a PCB, it is essential to consider the application, the frequency

range, and the thermal requirements. Using materials with a low CTE, layered construction methods, and careful design considerations, PCB manufacturers can create high-quality, reliable boards that can perform in various environments. A low CTE is necessary to ensure the board can withstand temperature changes without warping or cracking.

Another factor is based on the application and usability, like Communication, Consumer, Network, Storage, Consumer, Aerospace, Medical, Automobile, Military, etc. Select material based on the requirement specification of the product since there are huge cost variations based on the application and material selection. Example: no need to pick high-cost high-speed material for power supply/consumer products.

High-speed, low-loss multi-layer PCB materials with low dielectric constant and dissipation factor, high Tg, low moisture absorption, and uniformity make them ideal for applications such as high-speed digital, RF and Microwave, Aerospace and Defense, and medical devices. Also, consider Glass Weave Skew/ Glass Weave Skew Sensitivity for products to perform reliably and consistently. Use materials with fillers and design techniques to create laminates with a low dielectric constant. By doing so, they can produce reliable, reliable PCBs in a wide range of applications.

When going for miniature applications, based on the size and mechanical stability requirement, use Flex-PCB/Materials. Select the right Flex PCB material for the application, such as wearable and medical devices. Designers can ensure that their products perform reliably/flexibly and consistently. Designers must consider the minimum bend radius and follow best practices, such as avoiding sharp bends, using support structures, and testing the PCB to ensure that the PCB can withstand the required bending without damage.



**REFERENCES**

- [1.] IEEE 802.3ap standards,  
<http://standards.ieee.org/findstds/standard/802.3ap-2007.html>
- [2.] Dk & Df algebraic model,  
<http://www.ieee802.org/3/bj/public/tools.html>
- [3.] In-Situ De-embedding (ISD) by AtaiTec Corporation,  
[www.ataitec.com](http://www.ataitec.com)
- [4.] Acceptability of Printed Boards, IPC-A-600H-2010,  
<http://www.ipc.org>
- [5.] High-Frequency Structural Simulator (HFSS) by Ansys, Inc., [www.ansys.com](http://www.ansys.com)
- [6.] Material datasheets: ISOLA, Mitsubishi, Arlon, Panasonic, TUV, EK, and Doosan, Rogers, etc.
- [7.] IPC-4412A Amendment 1