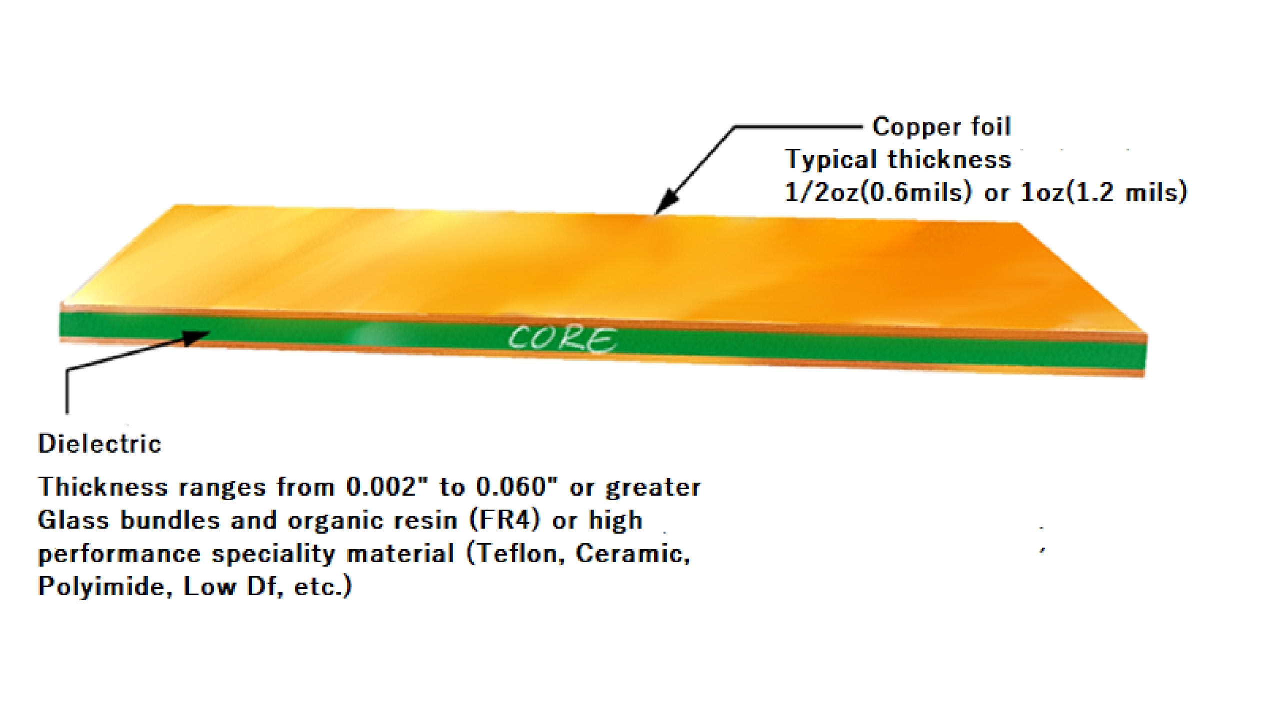
PCB DESIGN   PCB MANUFACTURING

PCB Substrates: Knowing Your Dielectric Material's Properties

A dielectric material conducts minimal electricity and provides an insulating layer between two conducting copper layers. The most common dielectric material is FR-4, but before selecting it for your board, you must carefully consider its properties.

By **Atar Mittal**
November 17, 2022 | 43 Comments

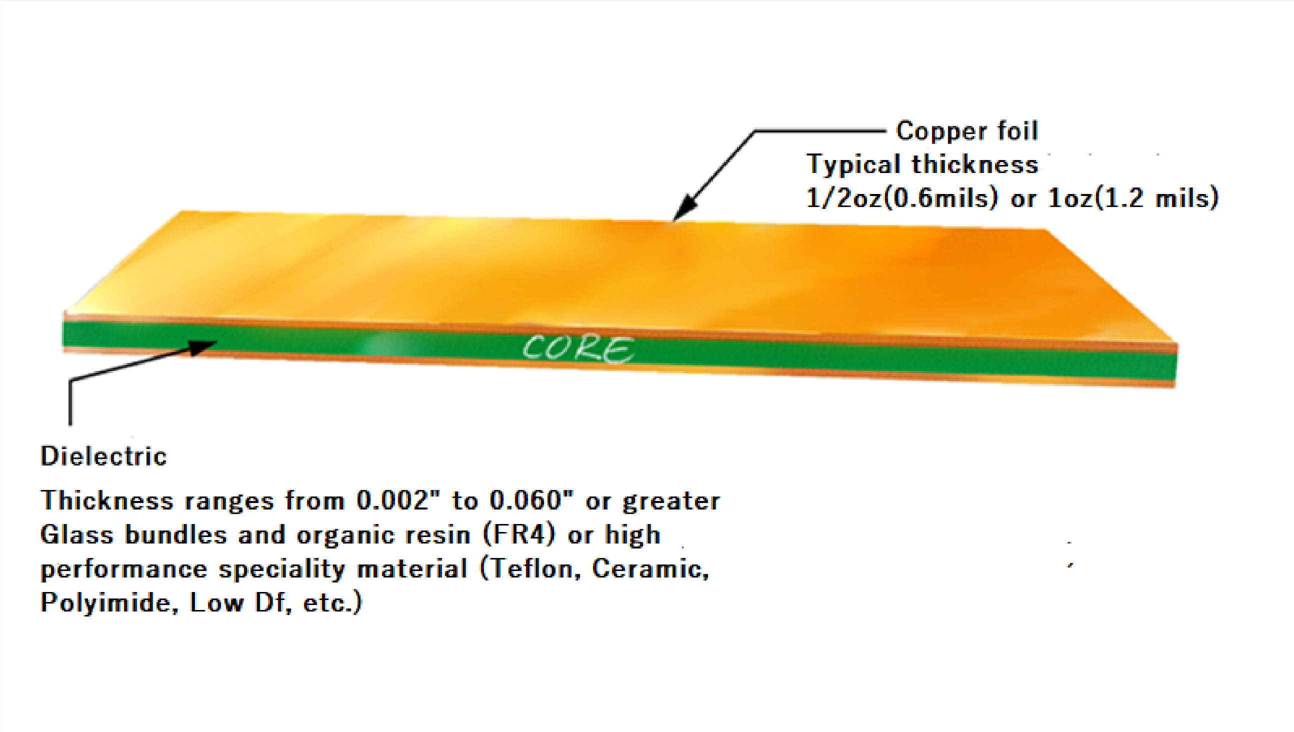


A PCB substrate is a dielectric layer that acts as an insulator between two copper layers in a stack-up. Based on its properties, it can alter the effects of signal attenuation, crosstalk, and electromagnetic radiation which directly impacts the signal integrity. For instance, a low Dk PCB material such as Rogers 4350B will exhibit lower losses even at high frequencies when compared to standard FR4.

If you’re working on aerospace or automotive designs, the board needs to sustain high temperatures, humidity, and stress while operating in a harsh environment. Hence, it needs to be mechanically robust and chemically stable. In short, understanding the characteristics of the dielectric is crucial for fabricating a quality printed circuit board.

In this article, we will discuss the electrical, mechanical, thermal, and chemical properties of a substrate which will help you choose the right material for your design.

Thermal properties of PCB Substrates



Dielectric material provides insulation between conducting layers (here shown as copper foil) in a PCB.

Glass transition temperature (Tg)

Glass transition temperature, or Tg, 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. Tg is expressed in units of degrees Celsius (°C).

Decomposition temperature (Td)

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

A substrate's Td is an important ceiling when assembling PCBs, because when a material reaches or surpasses its Td, changes to its properties are not reversible. Contrast this to Tg, glass transition temperature, where properties will return to their original states once the material cools below the Tg range.

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

Coefficient of thermal expansion (CTE)

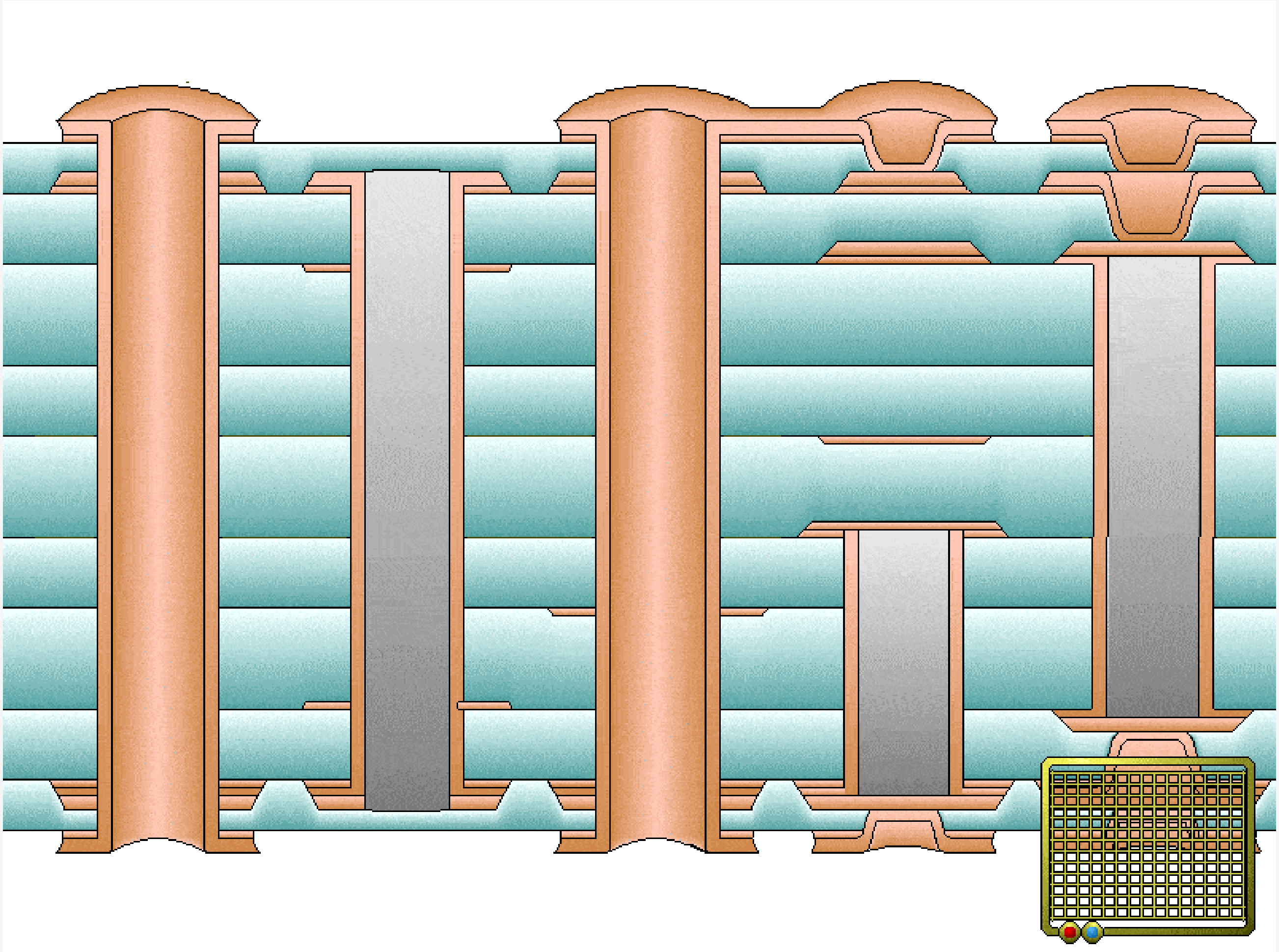
The coefficient of thermal expansion, or CTE, is the rate of expansion of a PCB material as it heats up. CTE is expressed in parts per million (ppm) expanded for every degree Celsius that it is heated.

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

The CTE of a PCB substrate is usually much higher than copper, which can cause interconnection issues as a PCB is heated.

CTE along the X and Y axes are generally low – around 10 to 20 ppm per degree Celsius. This is usually thanks 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; aim for less than 70 ppm per degree Celsius, and this will increase as a material surpasses Tg.



A material's expansion is measured by the coefficient of thermal expansion (CTE). This image shows CTE in the Z direction.

The point at which the displacement is maximum on the CTE curve is referred to as glass transition temperature (Tg). If the temperature is higher than the Tg, the substrate softens and doesn't expand any further.

Thermal conductivity (k)

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 substrates 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.

Electrical properties of PCB Substrates

Dielectric constant or relative permittivity (E_r or Dk)

Considering the dielectric constant of a material is important for signal integrity and impedance considerations, which are critical factors for high-frequency electrical performance. The Er for most PCB materials is in the range of 2.5 to 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. Substrates 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.

LOW DK PCB MATERIALS

When a signal propagates through a substrate, its dielectric constant alters the signal behavior. This is because the EM wave emitted is not confined to the trace and passes through the substrate. This can cause severe signal integrity issues. This effect will be even more significant at high frequencies.

The speed of the propagating EM wave is inversely proportional to the dielectric constant. Hence, a low Dk PCB material will have the following advantages:

- High-speed signals propagate with lower losses
- Reduced crosstalk between closely spaced traces and vias
- Limits the need for capacitive coupling in the circuit
- Lowers the adverse effect of mismatched trace lengths in parallel nets

Even with all the advantages, the dispersion in the low Dk PCB material still exists, which can distort the digital signals. They are also more expensive than FR4. They may also need higher temperatures and pressure during the board fabrication process, which can drive the cost even higher. A few examples of high-speed materials include Rogers 4350B, RT Duroid 5880, and Isola-I speed.

HIGH DK PCB MATERIALS

Even though we associate power integrity with constant voltage input, the dielectric constant also affects the stability of the input power. If you place a high Dk material between the power and ground layers, the interplanar capacitance will become significant. The planes themselves act as a large decoupling capacitor which helps to maintain a constant input voltage. It also lowers the PDN impedance. However, a higher Dk will result in increased dielectric loss, crosstalk, and EMI issues. In order to balance power and signal integrity, it is advised to opt for a hybrid stack-up of high and low Dk PCB materials.

A higher dielectric constant substrate can be used for separating the power and ground layer whereas the lower Dk substrate supports the signal layer with less loss. Manufacturing hybrid stack-ups depends on manufacturer capabilities as they have to consider the difference in CTE of the materials used.

Dielectric loss tangent or dissipation factor (Tan δ or Df)

A material's loss tangent gives a measure of the power lost due to the material. The lower a material's loss tangent, the less power is lost. The Tan δ of most PCB materials ranges from 0.02 for most commonly used materials to 0.001 for very low-loss high-end materials. It also varies with frequency, increasing as frequency increases. Read the difference between microstrip and stripline in PCBs.

Loss tangent isn't usually a critical consideration for digital circuitry, except at very high frequencies above 1Ghz. However, it is a very important parameter for analog signals, as it determines the degree of signal attenuation and thus affects the signal-to-noise ratio at various points along signal traces.

The dielectric constant significantly impacts the controlled impedance and the width of the conductor. If the Dk is higher, then the trace width required will be lower to achieve the target impedance.

Volume resistivity (ρ)

Volume resistivity, or electrical resistivity (ρ), is one of the measures of the electrical or insulation resistance of a PCB material. The higher a material's resistivity, the less readily it allows the movement of electric charge and vice versa. Resistivity is expressed in ohm-meters (Ω-m) or ohm-centimeters (Ω-cm)

As dielectric insulators, circuit board substrates are required to have very high values of resistivity, in the order of 10^6 – 10^{10} Megaohm-centimeters. Resistivity is somewhat affected by moisture and temperature.

Surface resistivity (ρS)

Surface resistivity (ρS) is the measure of the electrical or insulation resistance of the surface of a circuit board substrate. Like volume resistivity, PCB materials are required to have very high values of surface resistivity, in the order of 10^6 — 10^{10} Megaohms per square. It is also somewhat affected by both moisture and temperature.

Electrical strength

Electrical strength measures a PCB substrate's ability to resist electrical breakdown in the PCB's Z direction (perpendicular to the PCB's plane). It is expressed in Volts/mil. Typical electrical strength values for PCB dielectrics are in the range of 800 V/mil to 1500 V/mil.

Electrical strength is determined by subjecting the PCB material to short high-voltage pulses at standard AC power frequencies.

Chemical properties of substrates

Flammability specs (UL94)

UL94, or the Standard for Safety of Flammability of Plastic Materials for Parts in Devices and Appliances testing, is a plastics flammability standard that classifies plastics from lowest (least flame-retardant) to highest (most flame-retardant).

The standards are defined by Underwriters Laboratories (UL). Most PCB substrates adhere to UL94 V-0; here are its requirements

1. The specimens may not burn with flaming combustion for more than 10 seconds after either application of the test flame.
2. The total flaming combustion time may not exceed 50 seconds for the 10 flame applications for each set of 5 specimens.
3. The specimens may not burn with flaming or glowing combustion up to the holding clamp.
4. The specimens may not drip flaming particles that ignite the dry absorbent surgical cotton located 300 mm below the test specimen.
5. The specimens may not have glowing combustion that persists for more than 30 seconds after the second removal of the test flame.

Moisture absorption

Moisture absorption is the ability of a board substrate to resist water absorption when immersed in water. It is given by a percentage increase in the weight of a PCB material due to water absorption under controlled conditions as per standard test methods. Most substrates have moisture absorption values in the range of 0.01% to 0.20%.

Moisture absorption affects the thermal and electrical properties of the substrate, as well as the ability of the material to resist conductive anode filament (CAF) formation when a PCB circuit is powered.

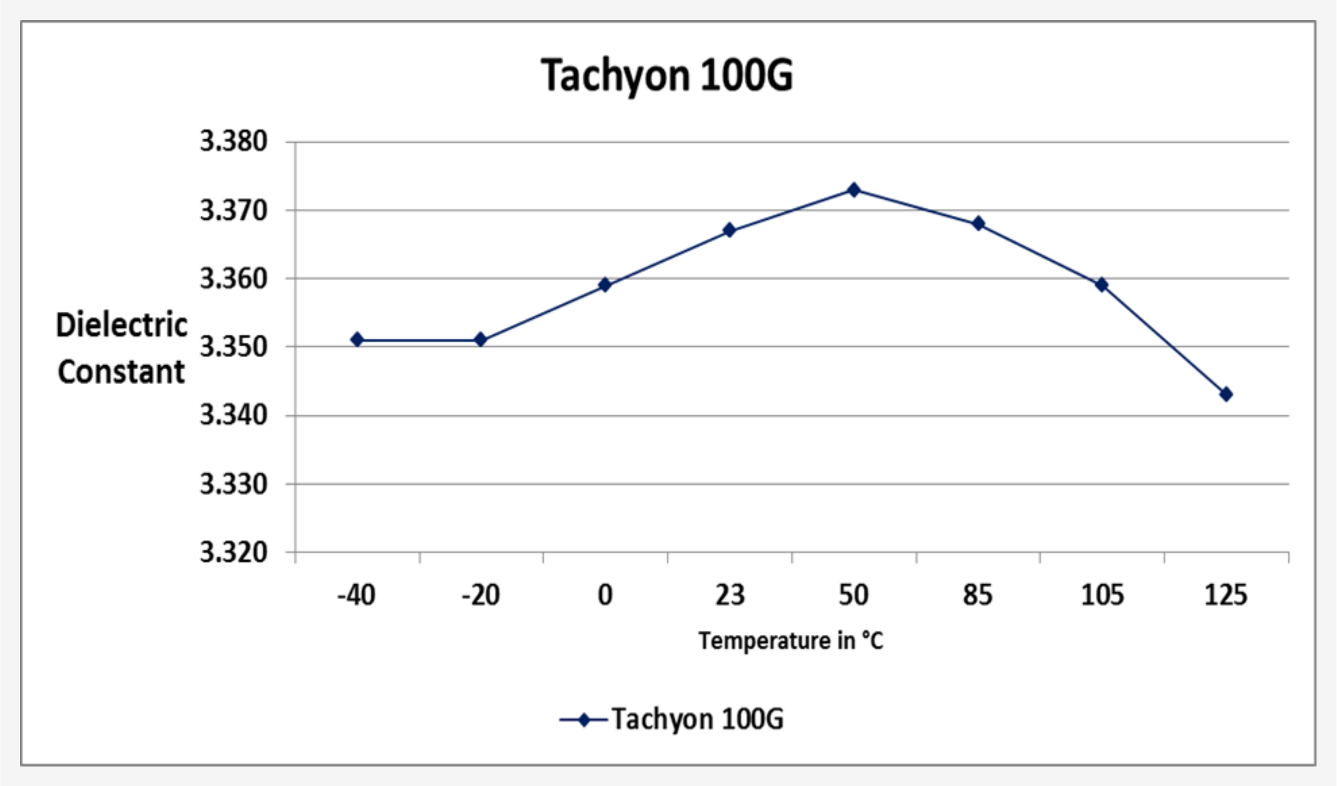
Methylene chloride resistance

Methylene chloride resistance is a measure of a material's chemical resistance; specifically, the ability of a PCB material to resist methylene chloride absorption.

Just like moisture absorption, it is expressed by a percentage increase in the weight of a PCB material due to exposure to or soaking in methylene chloride under controlled conditions. Most PCB materials have methylene chloride resistance values in the range of 0.01% to 0.20%.

Effects of moisture absorption on dielectric constant (Dk)

Michael J. Gay, Director, High-Performance Products, Isola Group, stated that "Moisture is one of the sources of Dk variation in PCB materials. There are other possible sources that could potentially result in some minor variation, but moisture uptake over time is a fairly large potential factor." The below chart shows the moisture uptake in a 24-hour DI water soak at various temperatures for Tachyon 100G material.



Dielectric constant Vs temperature for a low Dk PCB material Tachyon 100 G. Image credit: Isola Group

Michael further explained, “As you can see, the Dk is about 0.02 higher after the material becomes saturated at 23 °C or room temperature. It is not a huge difference but might make you think of changing the line width requirement to center the impedance. Moisture uptake would happen to the clad laminate over many weeks or to the etched laminate over several hours. Uptake is dependent on the length of the path for the material to reach equilibrium. For example, a single etched signal layer core sitting in AOI for a few hours would pick up the maximum moisture content at RT (relative temperature). These areas are usually about 60% RH (relative humidity) and so moisture is readily available.”

In an actual PCB design, the moisture may not stay in the areas near hot components and be driven out of the board when temperatures reach 80-90 °C near hot chips.

TEST METHODOLOGY

- The sample was pulled from the water soak and then equilibrated for 4 hours before -40 °C reading.
- Each subsequent reading was taken after an equilibration of 150 minutes at a new temperature.

Key takeaways of the experiment:

- Dk response as expected: It increases as water transitions through phases and then decreases as moisture is driven off from the sample.
- Df response is also as expected: A slow increase in loss due to the same phase transitions in the retained moisture, followed by a decrease as the moisture is driven off.
- Additional testing could be supported with different test conditions.

Michael also likes to add, “For high-speed design, we recommend that materials are dry prior to lamination using a baking process. In materials like Tachyon 100G, moisture does not negatively influence curing, but as you can see, moisture does result in D_K shifting.”

For conveyorized oxide replacements, an efficient dryer at the end of a conveyorized oxide replacement line should remove all moisture from the inner layer surface. However, drying on layers for 30 minutes at 100°C or higher is considered a best practice, especially for the boards subjected to lead-free processes. Drying in racks is preferred.

For other materials, especially epoxy-based resin systems, moisture can also impede the cure reaction and result in lower thermal resistance and thermal properties. We recommend baking inner layers to avoid this problem.

On the finished board of all material, you will see a similar variation in D_k if the boards are not dry. All polymeric materials pick up moisture. The reason that moisture causes problems is that it is a polar molecule and interferes with the magnetic fields in the local vicinity. This is the very reason why some resins are higher loss and some are lower loss – polarity of the resin molecules. For instance, PTFE is a very low-polarity resin and is a low-loss material.

Michael concluded, “Moisture can easily find its way through the resin molecules and park between them because it is so small. The amount of water a plastic can absorb is a result of the amount of free space in the cured resin. Some have more free space than others.”

Mechanical properties PCB Substrates

Peel strength

Peel strength is a measure of the bond strength between the copper conductor and the dielectric material. It is expressed in pounds of force per linear inch (PLI, or average load per conductor width) required to separate bonded materials where the angle of separation is 180 degrees.

Peel strength tests are done on samples of copper traces of 1 OZ thick and ~ 32 to 125 millimeters wide after standard PCB manufacturing processes. It is completed under 3 conditions:

• After thermal stress: after a sample is floated on solder at 288 °C for 10 seconds
• At elevated temperatures: after a sample is exposed to hot air or fluid at 125 °C
• After exposure to process chemicals: after a sample is exposed to a specified series of steps in a chemical or thermal process

Flexural strength

Flexural strength is the measure of a material’s capability to withstand mechanical stress without fracturing. It is expressed in either Kg per square meter or pounds per square inch (KPSI).

Flexural strength is generally tested by supporting a PCB at its ends and loading it in the center. IPC-4101 is the Specification for Base Materials for Rigid and Multilayer Printed Boards, and it gives the minimum flexural strength of various PCB materials.

Young’s modulus

Young’s modulus, or tensile modulus, also measures the strength of a PCB material. It measures the stress/strain ratio in a particular direction, and some PCB laminate manufacturers give strength in terms of Young’s modulus instead of flexural strength. Like flexural strength, it is expressed in force per unit area.

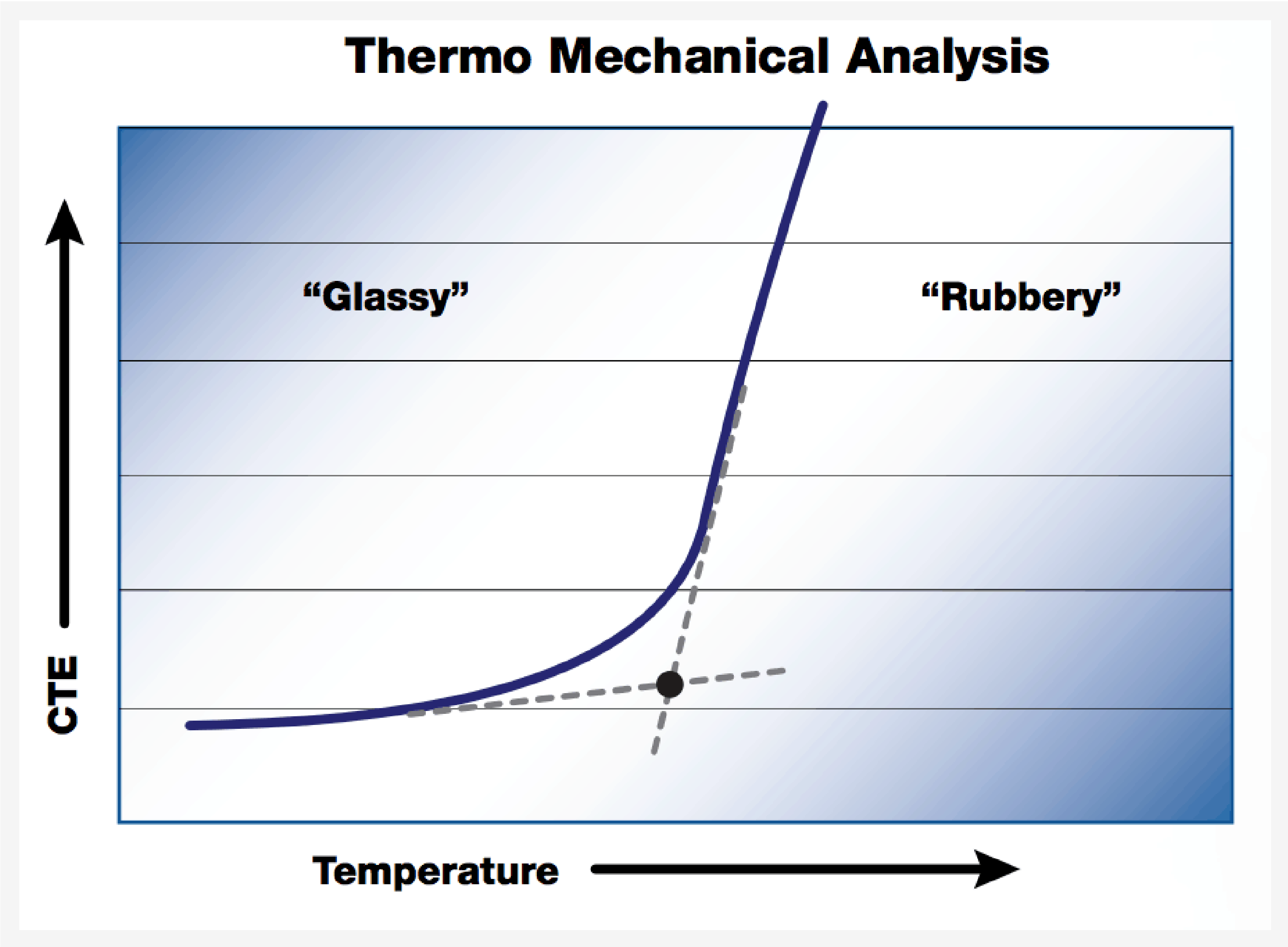
Some PCB laminate manufacturers give the PCB materials strength not in terms of Flexural Strength but in terms of Young's modulus which is a measure of Stress/strain ratio in a particular direction.

Density

The density of a PCB material is expressed in grams per cubic centimeter (g/cc) or pounds per cubic inch (lb/in^3).

Time to delamination

Time to delamination measures how long a material will resist delamination—the resin's separation from the laminate, foil, or fiberglass – at a specified temperature. Delamination can be caused by thermal shock, the wrong Tg in the material, moisture, and a poor lamination process.



Delamination is the separation of the resin from copper or reinforcement, and time to delamination measures how long this takes to occur at a specified temperature.

For more help selecting the right material, use the Sierra Circuits Material Selector tool. Input a specific material property or properties, and the tool will give you a list of high and low Dk PCB materials that meet your requirements. The tool also provides an easy-to-use comparison table that lets you compare the detailed properties of 50+ rigid and 10+ flex materials.

To learn more about PCB material selection, watch our webinar PCB Material Selection: Electrical and Manufacturing Considerations.

PCB Material Design Guide

9 Chapters - 30 Pages - 40 Minute Read

WHAT'S INSIDE:

- ✓ Basic properties of the dielectric material to be considered
- ✓ Signal loss in PCB substrates
- ✓ Copper foil selection
- ✓ Key considerations for choosing PCB materials

DOWNLOAD NOW

About the author: Atar Mittal is the Director and General Manager of the design and assembly division at Sierra Circuits. He is responsible for the design and development of strategies and process automation tools for complex printed circuit boards and assemblies. Atar is also currently engaged in the development of productivity tools for electronics designers that would have a tremendous impact on shortening the development time.

Related Articles

Subscribe

Join the discussion

B I U S

43 COMMENTS

Oldest

tom bach 5 years ago

does the conductivity change at the glass transition temperature?

0 Reply

Nishidh Reply to tom bach 3 years ago

hi mark,
hole size depends on overall PCB thickness, the ratio of minimum drill size to PCB thick ness should be 1:10 means, suppose your PCB thickness is 63 mils so minimum drill size of via you can use is 7mils. drill bit breakage also depends on same thing suppose consider your pcb is 93mils and your drill size is 8 mils, so for thinner drill bit to cross thickness of 93 mils is tough so drill bit can break. i hope it will be useful to you.

0 Reply

Tim Wecreate 7 years ago

I'm in a PCB manufacturing company myself, and this was a good read for me and anyone trying to find the right PCB material. Look forward to read more of your posts Tim

-1 Reply

protoexpress Reply to Tim Wecreate 7 years ago

Glad you enjoyed the article, Tim. Let us know if you're looking for articles on a particular topic, and we'll add it to the agenda!

1 Reply

Imen Reply to protoexpress 5 years ago

How can you avoid the z axis CTE of the FR4 to avoid any issue during lamination.

0 Reply

protoexpress Reply to Imen 5 years ago

There shouldn't be any issues regarding the z axis CTE of FR4 during lamination.

Z axis CTE does not affect component assembly; however, it can affect the reliability of plated through-vias.

Note: Most of the PCB during normal operating use can withstand temps of 125 degree C. (The PCBs will also be good till the Tg of the material).

0 Reply

Imen Reply to protoexpress 5 years ago

and what about the in-plane CTE do you have please any document about the lamination of FR4 to copper.

0 Reply

JR Reply to protoexpress 2 years ago

My question is about how to substantiate flammability for FR-4 Printed Circuit Board material. I used to just say that it was UL 94-V0 rated. But I have heard rumors that the 94-V0 rating is no longer sufficient. Is that true, and if so how do I substantiate the flammability of FR-4 short of burning it.

0 Reply

bizjets100 . 6 years ago

I was wondering if you could help me sort out some critical requirements for a pub design ?
We need to build to M4093768 with the following exceptions:
GL and FR-4
Pyle H copper
Transition temp of 170C minimum.

The supplier is building to IPC-4101C
not sure if this spec is compatible given the exceptions

Thx Gene

0 Reply

bizjets100 . 6 years ago

I meant PWB.

0 Reply

bizjets100 . 6 years ago

TYPE H COPPER. Sorry about the auto correct.

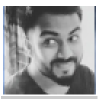
0 Reply

> **protoexpress** Reply to bizjets100 . 6 years ago
>
> Hi, Gene! This is a question right up our design team's alley. Feel free to contact Amreek, one of our designers, at amreeks@protoexpress.com. I've already informed him of your spec details.
>
> 0 Reply

Mark Eddy 4 years ago

Tg170; 6-Layer PWB; All inner layers .5oz Copper; Outer Layers 2oz Copper.
What challenges will the PWB Mfg have with this design?
Hole Sizes Criticality?
Drill Bit Breakage?

0 Reply



Joseph B. Pumphrey 4 years ago

Awesome analysis, Thank you

0 Reply

Stanton 3 years ago

Awesome things here. I am vesry glad to peer your post. Thank you a lot and I'm having a look forward to tuch you. Will you please drop me a e-mail?

0 Reply

Lilla 3 years ago

I like this site so much, saved to fav.

0 Reply

Arlen 3 years ago

That is a very good tip particularly to those fresh to
the blogosphere. Brief but very accurate information… Thank you for sharing this one.
A must read post!

0 Reply

Miquel Teetz 3 years ago

so much good info on here, : D.

0 Reply

Richard Puthota 2 years ago

Good basic insights on substrates. Very useful and well understood. Appreciate the way it is presented.

0 Reply

Homerenovates 2 years ago

Insertion loss : this is important in transmission lines and is related to the characteristic impedance, signal frequency, trace geometry, and effective dielectric constant.

0 Reply

venu 2 years ago

how to find elatisity in z drection

0 Reply

Milan Yogendrappa Editor Reply to venu 2 years ago

Hi, Venu

Thank you for posting your query!

Elasticity in the Z direction would depend on the flexural modulus of the material, as well as Poisson’s ratio. Both of which can normally be found on the datasheet of a material.

0 Reply

Corin 2 years ago

Is there any advantage to use a higher or lower dielectric constant? for a GHz application <4GHz.

0 Reply

Kanwarvir Saini Reply to Corin 2 years ago

A lower dielectric constant is generally better for high-frequency applications.

0 Reply

Kanwar Reply to Corin 2 years ago

A lower dielectric constant is generally better for high-frequency applications.

0 Reply

JR 2 years ago

My question is about how to substantiate flammability for FR-4 Printed Circuit Board material. I used to just say that it was UL 94-V0 rated. But I have heard rumors that the 94-V0 rating is no longer sufficient. Is that true, and if so how do I substantiate the flammability of FR-4 short of burning it.

0 Reply

> **Milan Yogendrappa** Editor Reply to JR 2 years ago
>
> Beside UL 94-V0 standard, properties of PCBs materials like glass transition temperature (typical value: 180 degrees) and decomposition temperature guides you to understand the thermal behaviour of the material. To know more about it, read our article "How to choose PCB materials and laminates for fabrication."
>
> 0 Reply

Eugene 2 years ago

thanks for the article ! Does dielectric constant depend on dielectric layer thickness ? Also any suggestion on PCB materials for 100GHz applications ?

0 Reply

> **Milan Yogendrappa** Editor Reply to Eugene 2 years ago
>
> Dielectric constant usually depends on the frequency and temperature at which the material is operating. It doesn't depend on dielectric layer thickness.
> Tachyon 100G/ Astra MT77 materials are suitable for this frequency range.
>
> 0 Reply

> **Milan Yogendrappa** Editor Reply to Eugene 2 years ago
>
> Dielectric constant usually depends on the frequency and temperature at which the material is operating. It doesn't depend on dielectric layer thickness.

Tachyon 100G/ Astra MT77 materials are suitable for this frequency range.

0 Reply

tom Reply to Milan Yogendrappa 1 year ago

Yup.. and it's a complex quantity, both real and imaginary are functions of frequency

0 Reply

tom 1 year ago

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:
==================

This makes no sense… what two curves ? you are plotting temperature vs displacement… what is the other curve ?

0 Reply

Milan Yogendrappa Editor Reply to tom 1 year ago

We have elaborated on section after looking into your query. Here's the explanation:
As you increase the temperature the displacement of the substrate increases. At the Tg point, the PCB material becomes plastic and transitions to a deformed state, beyond this it doesn't deform further. We can conclude that the point at which the displacement is maximum on the CTE curve is the glass transition temperature (Tg).

0 Reply

Vassilis 1 year ago

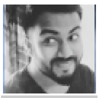
Is there a formula to calculate the change of the dk of the material after it has absorbed all the moisture? For example, assuming a pcb material has dk of 3.5 and moisture absorption of 0.06%. So the material will have 3.5 nominal dk without moisture (after baking at 120°C for a few hours). What will be the dk of the material after it stays in water for 24 hours and then dried?

0 Reply

Milan Yogendrappa Editor Reply to Vassilis 1 year ago

Hi Vassilis,
There isn't any specific formula to calculate the dielectric constant (Dk) in terms of moisture absorption. An experiment was carried out to analyze the Dk variation based on the presence of moisture on the PCB material. The sample was pulled from the water soak and then equilibrated for 4 hours before -40°C. Each subsequent reading was taken after equilibration of 150 minutes at a new temperature(graph shown under moisture absorption section). We can conclude that Dk increases as the PCB material absorb moisture. The Dk then decreases as the moisture is driven off from the material.

0 Reply



Joseph B. Pumphrey 4 years ago

Awesome analysis, Thank you

0 Reply

Richard Puthota 2 years ago

Good basic insights on substrates. Very useful and well understood. Appreciate the way it is presented.

0 Reply

Homerenovates 2 years ago

Insertion loss : this is important in transmission lines and is related to the characteristic impedance, signal frequency, trace geometry, and effective dielectric constant.

0 Reply

Corin 2 years ago

Is there any advantage to use a higher or lower dielectric constant? for a GHz application <4GHz.

0 Reply

shot dead 1 year ago

Good way of explaining, and good post to get information about my presentation focus, which i am going to present in school.

0 Reply

Turgut 1 year ago

As a RF Design Engineer, this post is really insightful. It describes a lot of information given on material datasheets in a clear way. A must read for any new engineers.

0 Reply

Lucy Iantosca Admin Reply to Turgut 1 year ago

Thank you, we really appreciate!

0 Reply

Reduce Re-Spins with our Zero Defect Guarantee.

Our platform sources the whole package, creating a thread from design to delivery. Fabricating PCBs. Procuring parts. Assembling your boards.

Once you upload your BOM, Our best-in-class software automatically checks for errors and cleans your file for you. After you place an order, your dedicated CAM engineer will run a full DFM and DFA check to ensure your prototype achieves Zero Defect every time—all at no additional cost to you.

Get to market faster and spend less time troubleshooting.

Start Quote →

Talk to a Sierra Circuits PCB Expert today

24 hours a day, 7 days a week.

Call us: +1 (800) 763-7503
Email us: through our Customer Care form

AS9100D – Military & Aerospace | IPC-A-600 Inspectors | IPC 6012 CLASS 2 |

IPC 6012 CLASS 3/ES | UL Certification - Flex & Rigid Flex PCBs | UL Certification - Rigid PCBs |

RoHS / REACH Compliance | ISO 9001:2015 Certificate | ISO 13485:2016 |

MIL-PRF-31032 / 3 Flex Certificate | ITAR Registered | JCP Certification Approval Letter

Sierra Circuits is a minority-owned business, certified by the National Minority Supplier Development Council.

SOLUTIONS BY INDUSTRY

Aerospace & Defense

Automotive

Consumer & Wearables

Healthcare & Medical Devices

Healthcare & Medical Devices

Universities & Research Institutions

Industrial Manufacturing

Sierra Circuits, Inc.
1108 West Evelyn Avenue
Sunnyvale, CA 94086, USA

Get PCB design tips and prototyping advice to build better boards - delivered to your inbox, every week.

email@address.com

Submit