Basic PCB material electrical and thermal properties for design

Introduction:

In order to design PCBs intelligently it becomes important to understand, among other things, the electrical properties of the board material. This brief paper is an attempt to outline these key properties and offer some descriptions of these parameters.

Parameters:

The basic (and almost indispensable) parameters for PCB materials are listed below and further described in the treatment that follows.

dk, laminate dielectric constant
df, dissipation factor
Dielectric loss
Conductor loss
Thermal effects
Frequency performance

dk: Use the *design dk* value which is assumed to be more pertinent to design. Determines such things as impedances and the physical dimensions of microstrip circuits.

A reasonable accurate practical formula for the effective dielectric constant derived from the dielectric constant of the material is:

$$\varepsilon_{eff} = [(\varepsilon_r+1)/2] + [(\varepsilon_r-1)/2][1+(12.h/W)]^{-1/2}$$

Here
h = thickness of PCB material
W = width of the trace
ε_{eff} = effective dielectric constant
ε_r = dielectric constant of pcb material

df: The dissipation factor (df) is a measure of loss-rate of energy of a mode of oscillation in a dissipative system. It is the reciprocal of quality factor Q, which represents the

---

Signal Processing Group Inc., technical memorandum. Website: http://www.signalpro.biz. Signal Processing Gtroup Inc., designs, develops and manufactures analog and wireless ASICs and modules using state of the art semiconductor, PCB and packaging technologies. For a free no obligation quote on your product please send your requirements to us via email at spg@signalpro.biz or through the Contact item on the website.

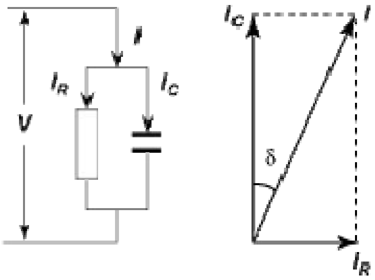
quality of oscillation. This allows judgments of frequency performance of the PCB material among other things.

Dielectric loss: At high frequencies, dielectric loss is dominant, and is dependent on the dissipation factor (loss tangent) for a given dielectric material.

The dielectric constant and loss factor are two of the most significant parameters that affect the performance of PCB circuits. Dielectric loss can be explained as follows:

Most capacitors, with dielectrics between the plates, lose a fraction of the energy when an AC current is applied. In other words, the dielectric is *less than perfect*. The simplest model for a capacitor with a lossy dielectric is as a capacitor with a perfect dielectric in parallel with a resistor giving the power dissipation. The current now leads the voltage by a very little less than 90°, where the difference δ (Greek letter delta) is termed the dielectric loss angle, as seen in Figure below.

Equivalent circuit for a lossy dielectric



Without bothering about the equations, the fraction of the maximum energy lost each cycle, divided by 2π is termed the 'loss factor' and its value is given by tan δ ('tan delta'): typically it is values of tan δ (loss tangent) that you will find quoted in reference material.

The table below shows some typical values of dielectric constant, loss factor, and dielectric strength. The AC values are measured at 1MHz.

Material	e_r	tan δ	(breakdown)MV/m
Air	1.0006	0	3.0
Polycarbonate	2.3	0.0012	275
FR-4	4.4	0.035	70
Alumina	8.8	0.00033	12

PCB materials and substrates have their own parameters. These are quoted by the manufacturer and should be consulted.

For dielectrics with small loss, δ is $\ll$ 1 and tan $\delta \approx \delta$. Power decays with propagation distance z as

$$P = P_o e^{-\delta k z}$$, where

P_o is the initial power,

$$k = \omega\sqrt{\mu\epsilon'} = \frac{2\pi}{\lambda},$$

ω is the angular frequency of the wave, and

$\varepsilon' = \varepsilon_0\, \varepsilon_r$.

µ = permeability of the material

ε_0 = is the permittivity of free space.

ε_r. = is the dielectric constant.

λ is the wavelength in the dielectric. (guide wavelength)

Conductor loss: PCB trace conductors all cause loss of the EM wave that travels on them. Therefore it is essential to know whether the conductors chosen will affect the performance of the circuit at the particular power levels and frequencies. The conductor loss can be described by the following relationships.

It is well known that high-frequency current in a planar conductor decreases exponentially with penetration into the conductor, falling to 1/e of its surface value at one skin depth δ where

$$\delta = \sqrt{[2.0/\omega\,\mu\,\sigma]}$$

Here

ω = angular frequency of signal
μ = permeability of material
σ = conductivity of the material

Since the thickness of the material becomes small the resistance of the conductor increases causing loss due to skin effect alone. In addition to this the resistance and shunt conductance of the trace is a lossy system. The resistance ofcourse becomes dependent on the skin effect as well. Generally for a transmission line (of which the PCB trace is an example) the attenuation is given by:

$$\alpha = \tfrac{1}{2}\{ GZo + R/Zo\} \text{ Np/meter}$$

Here G is the per unit length shunt conductance and Zo is the characteristic impedance. The characteristic impedance is found from the relationship:

$$Zo = \sqrt{[L/C]}$$

Where L and C are the transmission line constants. (Provided by manufacturer or calculated from the construction of the PCB).

In many cases, to get a fairly close approximation , G can be neglected. The resistance is ofcourse also dependent on the skin depth. However, a calculation can be made without the effect of skin depth and then later modified with the inclusion of skin depth.

Going further, if a matched condition exists between the power generator and the PCB trace, the input power is simply,

$WT = 1/2[Vo^2/Zo]$, Vo being the amplitude of the voltage input.

The average power loss along the line per unit length is:

$$Ploss = \alpha(2WT)$$

Knowing the amplitude of the signal expected on the trace and the other parameters defined above conductor power loss can be assessed. Of course modern simulation programs can reduce the overload of calculations like magic, but the engineer must know *a priori* what to *expect*, at least to an order of magnitude. It is for this that the expressions above can come in handy.

(See also the section on *frequency performance* below)

Thermal effects: As with all other devices PCBs are subject to temperature effects. This section explores these effects and provides some parametric descriptions of the temperature effects.

What happens to a PCB at elevated temperatures? As with most materials, it will expand and contract with changes in temperature, expanding in three axes (length, width, and thickness) as the temperature increases. The amount of this expansion for the change in temperature is characterized by a PCB material's *coefficient of thermal expansion (CTE).* Because a PCB is typically formed of a dielectric laminated with copper (to form transmission lines and a ground plane), the material's linear CTE in the x and y directions is usually engineered to match the CTE of copper (about 17 ppm/°C). By doing this, the materials expand and contract together with changes in temperature, minimizing stress on the junction of the two materials.

The CTE in the z axis (the thickness) of the dielectric material is usually designed for a low value to minimize dimensional changes with temperature and maintain the integrity of plated through holes (PTHs). The PTHs provide paths from the top to the bottom of the circuit board as needed for ground connections, as well as for interconnecting multilayer circuit boards.

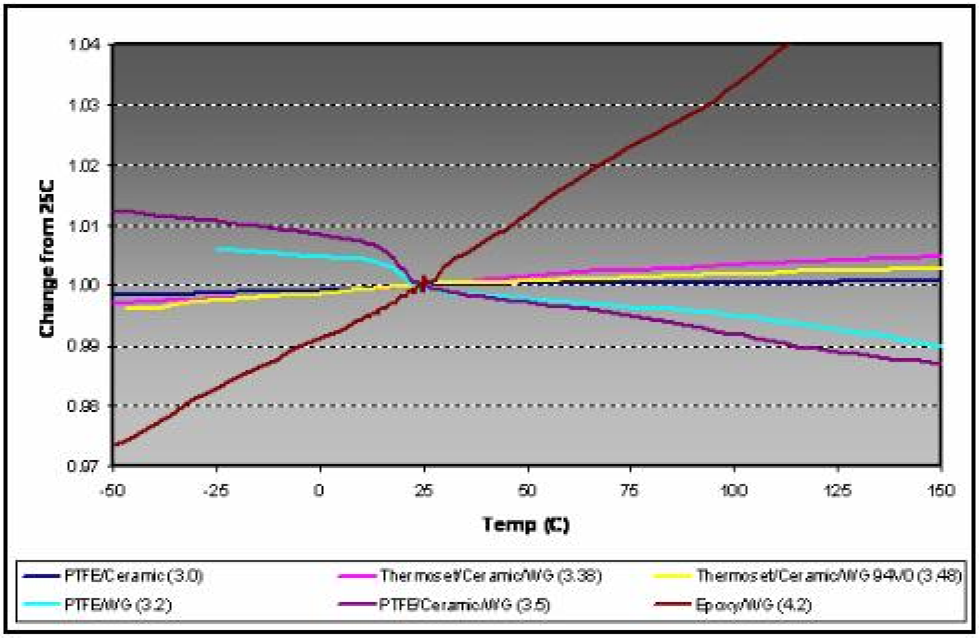
In addition to mechanical changes, temperature can also affect the electrical performance of a PCB. The relative dielectric constant of a PCB laminate, for example, varies as a function of temperature, as defined by a parameter known as the thermal coefficient of dielectric constant. The parameter describes changes in the dielectric constant (typically in ppm/°C). Because the impedance of high-frequency transmission lines is determined not only by the thickness of the substrate material but also by its dielectric constant, changes in z-axis CTE and dielectric constant as a function of temperature can significantly impact the impedance of microstrip and stripline transmission lines fabricated on that material.

Microwave circuits, of course, rely on tightly matched impedances between components and circuit junctions to minimize reflections that can result in signal losses and phase distortion. In a power amplifier, impedance-matching circuitry is used to make transitions from the typically low impedances of a power transistor to the (typically) 50-Ω characteristic impedance of an RF/microwave circuit or system. Changes in transmission-line impedance due to temperature effects from high-power signals can alter the frequency response of a high-frequency amplifier, so those effects should be minimized as much as possible by careful choice of PCB laminate.

In selecting PCB materials for multilayer designs in which the material might be expected to endure excessive short-term thermal steps, several material characteristics should be considered. These include the coefficient of thermal expansion (CTE), the glass transition temperature (Tg), and the decomposition temperature (Td). The CTE describes how a material changes dimension with temperature. For a given material, it will be specified in all three axes, with the z axis being through the thickness of the material. Ideally, a PCB material's CTE should be closely matched to copper, which is about 17 ppm°C. It should also be isotropic, with the same CTE in all three axes. For most PCB materials, however, the CTE is typically much higher in the z-axis (the thickness) than in the x or y axes. The z-axis CTE is a concern because as the PCB expands during heating, it can elongate plated via holes and cause fracturing. If the z-axis CTE is closely matched to copper, expansion of the PCB material and copper will be more uniform and the plated via holes will be more robust during thermal cycling.

The figure below shows the thermal effects on the dielectric constant of PCB materials. The dielectric constant is so important a design parameter that any change will effect the electrical performance of the PCB manufactured.

TcDk (thermal coefficient of dielectric constant) curve of some common PCB laminate materials



Frequency performance:

The frequency performance of the PCB materials are obviously of immense concern to the microwave or RF designer; perhaps not so to the designer who is primarily involved in low frequency design. Yet it is an issue that should be understood by PCB designers whether they are involved in high frequency or low frequency design.

The three most common transmission-line technologies used in microwave circuits are microstrip, coplanar and stripline circuits. Of the three, microstrip transmission lines are most often used in high-frequency PCBs, since they are relatively simple to fabricate and with fewer electrical variables to consider than the other two approaches.

Electromagnetic propagation in a microstrip circuit occurs by means of transverse-electromagnetic (TEM) plane waves. In an ideal microstrip circuit, signal energy propagates perpendicular to the electric (E) and magnetic (H) fields. In an actual microstrip circuit, because propagation also takes place in the dielectric material between the conductors, as well as in the air above the conductors, propagation occurs in a *quasi-TEM mode*.

There are four types of signal losses in a microstrip transmission line: conductor, dielectric, radiation, and leakage losses. Leakage losses usually are not a concern, due to the high volume resistivity (resistance) of PCB materials used for microwave circuits. At microwave frequencies, radiation losses tend
to be more of an issue for microstrip circuits than for coplanar or stripline circuits. Dielectric losses are a function of the PCB substrate material; in terms of loss performance, different materials can be compared by a parameter known as dissipation factor. *Lower values of dissipation factor signify laminates with lower dielectric losses.* Conductor losses are not quite as simple to size up because they are linked to a number of different variables in a microstrip circuit. (See conductor losses above).

Skin depth decreases with increasing frequency and the conductor loss will increase. *Higher frequencies translate into higher conductor losses.* As frequencies increase, the effects of copper conductor surface roughness also increase, to a point where they will reach a saturation point.

The surface roughness of the copper conductor, at a circuit's copper-substrate interface, can impact the loss of a microstrip circuit. *A rougher conductor surface suffers higher losses.* Several methods have been developed to account for the copper roughness, and a simple model is the Morgan rule, which is a multiplier of the conductor losses α_c
Generalized conductor loss and the Morgan rule are given by Eqs. A and B, below

$$\alpha_c = 1/\delta \qquad \text{(Eq. A)}$$

where δ = skin depth.

$$\alpha_{c+\text{roughness}} = \alpha_c \{1.0+(2/\pi)[\tan^{-1}(\Delta/\delta)^2]\} \qquad \text{(Eq B)}$$

where $\alpha_{c+\text{roughness}}$ is the total conductor loss, including loss due to copper conductor roughness. Parameter Δ in Eq. 3 represents the root-mean-square (RMS) surface roughness of the copper conductor.

As with many models, the Morgan rule is limited at certain frequencies, and is typically More accurate at frequencies of *less than 10GHz.*

It is apparent that skin depth decreases with increasing frequency. In Eq. A, as the skin depth, δ , decreases, the conductor loss, α_c, will increase. *Higher frequencies translate into higher conductor losses*. As frequencies increase, the effects of copper conductor surface roughness also increase, to a point where Eq. B will reach a saturation point at its highest value.

Copper treatment:
Different types of PCB conductor finish can also provide different results in terms of conductor losses at higher frequencies. The manner in which copper is treated in the process of making a PCB's copper foil can also impact conductor losses. In general, a conductor composed of ore treated with a ferromagnetic material will exhibit degraded conductor losses in microwave transmission lines.

Numerous PCB variables influence the impedance of a microstrip transmission line, such as laminate dielectric constant (known as dk, ε_r, relative permittivity), thickness, copper weight, and control of circuit etching. For high-frequency applications, it is important that a PCB laminate have well controlled dk, as well as tightly controlled thickness, since variations in either will result in variations in transmission line impedance.

Dispersion:
A number of other factors can influence the impedance of a high-frequency PCB's transmission lines. Dispersion, for example, is often overlooked. Dispersion is a microstrip transmission-line property in which the propagation characteristics are different at lower frequencies than at higher frequencies. Dispersion can also be a concern in PCB materials where the dk value is considerably different at lower and higher frequencies. Dispersion typically plagues PCB laminates not nominally engineered for high-frequency applications.

Environmental conditions:
Environmental conditions can also play a role in how well a PCB material maintains impedance, especially at higher frequencies. Many traditional PCB materials may not have been formulated for stable dk performance in changing or hostile environments. All PCB materials are characterized by a parameter known as thermal coefficient of dielectric constant, or tCDk, in units of ppm/°C. This parameter describes how much the dielectric constant will change with changes in temperature. These changes in dk will also change the impedance of the microstrip transmission lines, so lower values of tCDk (resulting in minimal effects on impedance) are preferred.

Humidity:
Humidity can also affect PCB performance. If a PCB material is prone to absorb moisture, the water content can impact loss performance and impedance stability. Many standard PCB laminates have moisture absorption values of 2% or more, which means in a humid environment, the laminate can absorb moisture readily, and the electrical properties change. Compared to PCB materials, the dk of water is very high (about 70). In an environment with high humidity, excessive moisture absorption can raise a PCB material's dk and increase its dielectric loss. PCB materials formulated for high-frequency use typically exhibit low moisture absorption, with values

of 0.2% or less.

Radiation losses:

Microstrip radiation losses can be significant above certain frequencies and/or with certain circuit geometries. At very high frequencies, radiation losses can dominate the performance of a microstrip circuit and negate the benefits of using conductors with smooth copper or laminate material with low dissipation factor. One way to avoid the high radiation losses of microstrip at high frequencies is through the use of CBCPW transmission lines. When properly designed, CBCPW transmission lines can support quasi-TEM wave propagation at very high frequencies, *beyond the frequency limit of microstrip.* (CBCPW -> conductor backed coplanar waveguide)

Conclusions and discussions: A brief look at the thermal and electrical properties of PCB materials enables a PCB designer to get a better feel for his design. There are a number of parameters to be considered at the outset of the effort. There are many nth order parameters to consider but the parameters investigated above are perhaps the most important first order ones. If the PCB design does not fall within the limits of the first order parameters it will certainly not be workable at the nth order parameters.